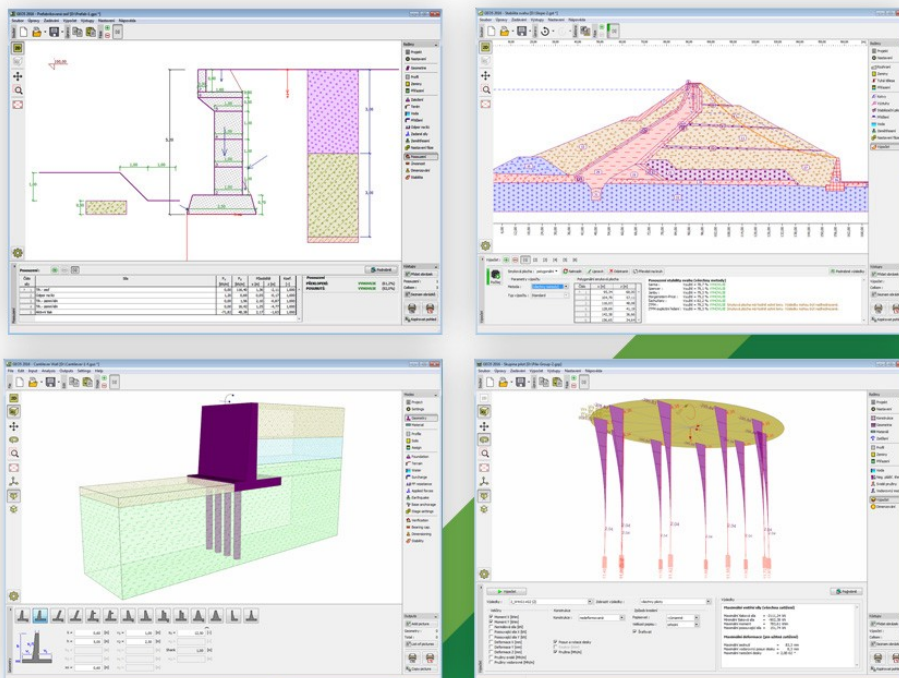


# GEO5

## User's Guide

---

Edition 2016



## Content

Using the Help.....	36
Using the Search Function.....	37
User Interface.....	38
Application Window.....	39
Control Menu.....	40
Tool and Control Bars.....	41
File.....	42
Edit.....	43
Template.....	44
Construction Stages.....	44
Visualization.....	45
Frames.....	47
Outputs.....	47
3D Visualization.....	48
Visualization Settings.....	49
Drawing Styles Administrator.....	50
Frames.....	51
Tables.....	52
Dialog Windows.....	54
Active Dimensions and Objects.....	55
Mouse Functions.....	56
Mouse Context Menu.....	58
Units - Metric / Imperial.....	60
Copy to Clipboard.....	60
GeoClipboard™.....	61
Copying and Pasting Project Data.....	61
Copying and Pasting 2D Interfaces.....	62
Copying and Pasting Soils and Rigid Bodies.....	63
Copying and Pasting 2D Assignment.....	64
Options.....	65
Options - Input.....	65
Options - Copy to Clipboard.....	66
Options - Print and Pictures.....	67
Common Input.....	68
Input and Edit of Soils.....	69



---

Classification of Soils.....	70
Soil and Rock Symbols.....	72
Manual Classification of Soils.....	73
Interfaces in 2D Environment.....	74
Adding Interface.....	75
Editing Interface Points.....	76
Editing and Removing Interface.....	78
Input Interface Corrector.....	80
World Coordinates.....	81
Input of Objects and Data.....	84
Add.....	84
Add Graphically.....	85
Assigning Soils.....	86
Design Coefficients.....	87
Running Several Analyses / Verifications.....	88
Program Connection.....	89
Selection and Store of Views.....	90
Visualization Settings of Results.....	91
Setting a Color Range.....	92
Scale Color Definition.....	93
DXF Import and Export.....	94
Loading Data into Template.....	95
Loading Data into Interface.....	96
Loading Data into GeoClipboard.....	97
Data Input using Templates.....	97
Modification of Template During Data Input.....	98
DXF Export.....	99
DXF Import.....	100
Table Data Import.....	101
(1) Input File.....	104
(2) Input File Preview.....	105
(3) Parameters for Input File Splitting into Columns.....	106
(4) Input File Split into Columns.....	107
(5) Assign Columns to Imported Data.....	107
(6) Result of Import Preview.....	108
Import LandXML.....	108
Import gINT.....	110

---

---

Import of Terrain Points.....	111
Import of Soils and Profiles.....	112
Heredity - Construction Stage.....	112
Standards and Analysis Methods.....	113
Administrator.....	113
Import and Export of Settings.....	114
Settings List.....	115
Analysis Settings.....	116
Materials and Standards.....	117
Wall Analysis.....	118
Excavations.....	120
Stability Analysis.....	121
Settlement.....	122
Spread Footing.....	123
Pile.....	124
Pile CPT.....	125
Micropiles.....	125
Pile Group.....	126
Adding New Settings.....	127
Import of Older Data.....	128
Basic Changes in Settings Between Version 15 and Older Versions.....	128
Verification Methodology.....	129
Analysis According to the Safety Factor (ASD).....	130
Analysis According to the Theory of Limit States (LSD).....	130
Verification According to EN 1997.....	131
Partial Factors.....	131
Design Approaches.....	132
Design Approach 1.....	133
Design Approach 2.....	134
Design Approach 3.....	134
National Annex (NA).....	135
Partial Factors on Water.....	135
Analysis of Walls (Support Structures).....	137
Analysis of Sheet-pile Structures.....	138
Analysis of Foundations (Spread Footing, Piles).....	138
Slope Stability Analysis.....	139
Load Combinations.....	140

---

---

Analysis According to LRFD.....	141
LRFD 2003 - Analysis of Retaining Walls (Support Structures).....	142
LRFD 2003 - Analysis of Spread Foundations.....	143
LRFD 2012 - Design Situations.....	144
LRFD 2012 - Analysis of Retaining Walls (Support Structures).....	145
LRFD 2012 - Analysis of Spread Foundations.....	146
LRFD 2012 - Slope Stability Analysis.....	147
Analysis According to Chinese Standards.....	148
Design Situations.....	148
Individual Programs.....	149
Program Earth Pressure.....	150
Project.....	150
Settings.....	151
Geometry.....	152
Profile.....	153
Soils.....	153
Basic Data.....	154
Assign.....	155
Terrain.....	156
Water.....	157
Surcharge.....	158
Earthquake.....	159
Stage Settings.....	160
Analysis.....	161
Program Cantilever Wall.....	162
Project.....	162
Settings.....	163
Geometry.....	164
Material.....	165
Profile.....	165
Soils.....	166
Basic Data.....	167
Assign.....	168
Foundation.....	169
Terrain.....	170
Water.....	171
Surcharge.....	172

---

Front Face Resistance.....	173
Applied Forces.....	174
Earthquake.....	175
Base Anchorage.....	176
Stage Settings.....	177
Verification.....	178
Bearing Capacity.....	179
Dimensioning.....	180
Stability.....	182
Program Gravity Wall.....	183
Project.....	184
Settings.....	184
Geometry.....	185
General Wall Shape.....	187
Material.....	190
Profile.....	192
Soils.....	193
Basic Data.....	194
Assign.....	195
Foundation.....	196
Terrain.....	197
Water.....	198
Surcharge.....	199
Front Face Resistance.....	200
Applied Forces.....	201
Earthquake.....	202
Stage Settings.....	203
Verification.....	203
Bearing Capacity.....	204
Dimensioning.....	206
Stability.....	207
Program Prefab Wall.....	208
Project.....	209
Settings.....	209
Geometry.....	210
Profile.....	211
Soils.....	212

Basic Data.....	213
Assign.....	214
Foundation.....	215
Terrain.....	216
Water.....	217
Surcharge.....	218
Front Face Resistance.....	219
Applied Forces.....	220
Earthquake.....	221
Stage Settings.....	222
Verification.....	222
Bearing Capacity.....	223
Dimensioning.....	225
Slip on Georeinforcement.....	226
Stability.....	227
Program Masonry Wall.....	228
Project.....	229
Settings.....	229
Types of Blocks.....	230
Geometry.....	231
Material.....	232
Profile.....	233
Soils.....	234
Basic Data.....	235
Assign.....	236
Foundation.....	237
Terrain.....	238
Water.....	239
Surcharge.....	240
Front Face Resistance.....	241
Applied Forces.....	242
Earthquake.....	243
Base Anchorage.....	244
Stage Settings.....	245
Verification.....	246
Bearing Capacity.....	247
Dimensioning.....	248

---

Stability.....	249
Program Gabion.....	250
Project.....	251
Settings.....	251
Material.....	252
Geometry.....	253
Profile.....	254
Soils.....	255
Basic Data.....	256
Assign.....	257
Foundation.....	258
Terrain.....	259
Water.....	260
Surcharge.....	261
Front Face Resistance.....	262
Applied Forces.....	263
Earthquake.....	264
Stage Settings.....	265
Verification.....	265
Bearing Capacity.....	266
Dimensioning.....	268
Stability.....	269
Program Abutment.....	269
Project.....	270
Settings.....	270
Geometric Section.....	271
Wings.....	272
Geometry Plane View.....	273
Footing Steps.....	274
Material.....	275
Profile.....	276
Soils.....	277
Basic Data.....	278
Load - LC.....	279
Assign.....	280
Foundation.....	281
Terrain.....	282

---

---

Water.....	283
Surcharge.....	284
Front Face Resistance.....	285
Applied Forces.....	286
Earthquake.....	287
Stage Settings.....	288
Verification.....	289
Bearing Capacity.....	290
Dimensioning.....	291
Stability.....	293
Program Nailed Slope.....	293
Project.....	294
Settings.....	294
Geometry.....	295
Types of Nails.....	296
Geometry of Nails.....	297
Material.....	298
Profile.....	299
Soils.....	300
Basic Data.....	301
Assign.....	302
Terrain.....	303
Water.....	304
Surcharge.....	305
Earthquake.....	306
Stage Settings.....	307
Internal Stability.....	307
Verification.....	308
Bearing Capacity.....	309
Dimensioning.....	310
External Stability.....	311
Program Sheet Design.....	312
Project.....	313
Settings.....	313
Profile.....	314
Soils.....	315
Basic Data.....	316

---

---

Assign.....	317
Geometry.....	318
Anchors.....	319
Props.....	320
Supports.....	321
Pressure Determination.....	322
Terrain.....	323
Water.....	324
Surcharge.....	325
Applied Forces.....	326
Earthquake.....	327
Stage Settings.....	328
Analysis.....	328
Stability.....	330
Program Sheeting Check.....	332
Project.....	333
Settings.....	334
Profile.....	334
Modulus Kh.....	335
Pressiometric Tests (PMT).....	336
Dilatometric Tests (DMT).....	338
Soils.....	339
Basic Data.....	340
Geometry.....	341
Adding and Editing Section.....	342
User's Catalog.....	343
Material.....	345
Pressure Determination.....	345
Assign.....	346
Excavation.....	347
Terrain.....	348
Water.....	349
Surcharge.....	350
Applied Forces.....	351
Anchors.....	352
Props.....	353
Supports.....	354

---



---

Earthquake.....	355
Stage Settings.....	356
Analysis.....	357
Internal Stability.....	360
External Stability.....	361
Heave Failure.....	362
Dimensioning.....	363
Program Anti-Slide Pile.....	364
Project.....	365
Settings.....	366
Profile.....	367
Modulus Kh.....	368
Pressiometric Tests (PMT).....	369
Dilatometric Tests (DMT).....	370
Soils.....	371
Basic Data.....	372
Geometry.....	373
Adding and Editing a Section.....	374
User's Catalog.....	375
Material.....	377
Pressure Determination.....	377
Rock.....	378
Assign.....	379
Front Face .....	380
Terrain.....	381
Water.....	382
Surcharge.....	383
Applied Forces.....	384
Anchors.....	385
Supports.....	386
Earthquake.....	387
Stage Settings.....	388
Analysis.....	389
Dimensioning.....	392
Program Shaft.....	393
Project.....	394
Settings.....	394

---

---

Geometry.....	395
Profile.....	396
Soils.....	397
Basic data.....	398
Assign.....	399
Water.....	400
Surcharge.....	401
Stage settings.....	403
Load Analysis.....	403
Dimensioning.....	404
Program Slope Stability.....	406
Project.....	407
Settings.....	407
Interface.....	408
Embankment.....	409
Earth Cut.....	410
Soils.....	411
Basic Data.....	412
Rigid Body.....	413
Assign.....	414
Anchors.....	415
Reinforcements.....	416
Anti-Slide Piles.....	417
Surcharge.....	418
Water.....	419
Earthquake.....	420
Stage Settings.....	421
Analysis.....	421
Input of Slip Surface.....	424
Restrictions on the Optimization Procedure.....	425
Height Multiplier.....	426
Program Rock Stability.....	427
Project.....	428
Settings.....	428
Terrain.....	429
Rock.....	430
Slip Surface - Plane.....	431

---

---

Slip Surface - Polygonal.....	432
Parameters - Polygonal Slip Surface.....	433
Water - Plane Slip Surface.....	435
Water - Polygonal Slip Surface.....	435
Surcharge - Plane and Polygonal Slip Surface.....	436
Anchors - Plane and Polygonal Slip Surface.....	437
Earthquake.....	438
Stage Settings.....	439
Analysis - Plane Slip Surface.....	440
Analysis - Polygonal Slip Surface.....	440
Geometry.....	441
3D View.....	442
Slip Surface - Rock Wedge.....	443
Parameters - Rock Wedge.....	444
Water - Rock Wedge.....	445
Surcharge - Rock Wedge.....	446
Anchors - Rock Wedge.....	447
Analysis - Rock Wedge.....	448
Program MSE Wall.....	449
Project.....	450
Settings.....	450
Geometry.....	451
Material.....	452
Types of Reinforcements.....	453
Adding and Editing Type of Reinforcement.....	454
User's Catalog.....	455
Reinforcement.....	457
Reinforcement.....	458
Profile.....	459
Soils.....	460
Basic Data.....	461
Assign.....	462
Terrain.....	463
Water.....	464
Water.....	465
Surcharge.....	466
Front Face Resistance.....	467

---

---

Applied Forces.....	468
Earthquake.....	469
Stage Settings.....	470
Verification.....	471
Dimensioning.....	472
Bearing Capacity.....	473
Slip on Georeinforcement.....	474
Internal Stability.....	475
Global Stability.....	476
Stability.....	477
Program Spread Footing.....	478
Project.....	479
Settings.....	479
Profile.....	480
Dilatometric Tests (DMT).....	481
Soils.....	482
Basic Data.....	483
Assign.....	484
Foundation.....	485
Load.....	486
Geometry.....	487
Footing Bottom.....	489
Sand-Gravel Cushion.....	489
Material.....	490
Surcharge.....	491
Water, Incompressible Subsoil.....	492
Stage Settings.....	493
Bearing Capacity.....	493
Settlement and Rotation.....	495
Dimensioning.....	496
Program Pile.....	496
Project.....	497
Settings.....	498
Profile.....	499
Modulus of Subsoil Reaction.....	500
Soils.....	500
Basic Data.....	501

---

---

Assign.....	502
Load.....	503
Geometry.....	504
Material.....	505
Water, Incompressible Subsoil.....	506
Negative Skin Friction.....	507
Stage Settings.....	508
Vertical Bearing Capacity - Analytical Solution.....	508
Vertical Bearing Capacity - Spring Method.....	509
Settlement - Linear Load-Settlement Curve (Poulos).....	511
Settlement - Non-Linear Load-Settlement Curve (Masopust).....	511
Horizontal Bearing Capacity - Elastic Subsoil (p-y Method).....	512
Horizontal Bearing Capacity - Brom's Method.....	513
Program Pile CPT.....	515
Project.....	516
Settings.....	516
CPT.....	517
Import CPT.....	519
GWT + NSF.....	520
Soil Classification.....	521
Profile.....	522
Soils.....	523
Basic Data.....	524
Assign.....	525
Construction.....	526
Group of Piles.....	528
Geometry.....	529
Bearing Capacity.....	530
Settlement.....	531
Program Pile Group.....	533
Project.....	533
Settings.....	534
Structure.....	535
General Pile Group Shape.....	536
Geometry.....	540
Material.....	541
Load.....	542

---

---

Load Acting on a Pile Group.....	543
Profile.....	545
Soils.....	545
Basic Data.....	546
Assign.....	547
Water.....	548
Negative Skin Friction.....	549
Vertical Springs.....	550
Horizontal Modulus.....	551
Stage Settings.....	552
Vertical Bearing Capacity - Analytical Solution.....	552
Settlement - Cohesive Soil.....	553
Settlement - Cohesionless Soil (Load-Settlement Curve).....	554
Analysis - Spring Method.....	555
Dimensioning.....	556
Program Micropile.....	557
Project.....	558
Settings.....	558
Profile.....	559
Soils.....	560
Basic Data.....	561
Geometry.....	562
Material.....	563
Assign.....	564
Load.....	565
Water.....	566
Standard Penetration Tests (SPT).....	567
Pressiometric Tests.....	569
Verification of Cross-Section.....	570
Root Verification.....	571
Program Slab.....	572
Project.....	573
Settings.....	573
Joints.....	574
Lines.....	575
Macroelements.....	576
Openings.....	577

---

---

Joint Refinements.....	578
Line Refinements.....	580
Macroelement Refinements.....	581
Mesh Generation.....	583
Mesh Generator Warning.....	585
Joint Supports.....	587
Line Supports.....	589
Beams.....	590
Catalog of Materials.....	592
Editor of Materials.....	593
Types of Cross-Section.....	595
Catalog of Profiles.....	595
Cross-Section Editor.....	596
Internal Hinges.....	597
Macroelement Subsoils.....	599
Winkler-Pasternak Parameters C1 a C2.....	600
Calculation of Winkler-Pasternak Constants from Deformation Parameters of Soils. .	601
Load Cases.....	601
Load Case Parameters.....	603
Joint Loads.....	604
Line Loads.....	605
Temperature Load.....	607
Macroelement Loads.....	607
Free Point Loads.....	608
Free Line Loads.....	609
Free Area Loads.....	611
Combination ULS.....	612
Parameters of ULS Combinations.....	613
Generator of ULS Combinations.....	614
Combination SLS.....	617
Parameters of SLS Combinations.....	618
Generator of SLS Combinations.....	619
Dimensioning Parameters.....	619
Macroelement Dimensioning.....	620
Analysis.....	621
Analysis Procedure.....	622
Results.....	623

---

---

Tool Bar - Results.....	624
Results Visualization Settings.....	625
List of Variables.....	625
List of Variables of Dimensioning.....	626
Values.....	627
Distributions.....	628
Coordinate System (Sign Convention).....	629
Program Beam.....	631
Project.....	632
Settings.....	632
Winkler-Pasternak Parameters C1 a C2.....	633
Calculation of Winkler-Pasternak Parameters C1 and C2 from Geological Profile.....	634
Calculation of Winkler-Pasternak Constants from Deformation Parameters of Soils. .	634
Geometry.....	635
Subsoil.....	635
Interface.....	636
Location.....	637
Soils.....	638
Basic Data.....	639
Assign.....	640
Water.....	641
Supports.....	642
Load Cases.....	643
Load Case parameters.....	644
Load.....	646
Combination ULS.....	647
Parameters of ULS Combinations.....	648
Generator of Combinations.....	649
Combination SLS.....	652
Parameters of SLS Combinations.....	653
Generator of Combinations.....	654
Analysis.....	654
Program Settlement.....	655
Project.....	656
Settings.....	656
Interface.....	657
Embankment.....	658



---

Earth Cut.....	659
Incompressible Subsoil.....	660
Soils.....	661
Basic Data.....	662
Assign.....	663
Surcharge.....	664
Water.....	665
Stage Settings.....	666
Analysis.....	667
Consolidation Parameters.....	668
Program Ground Loss.....	670
Project.....	671
Settings.....	671
Buildings.....	672
Profile.....	673
Soils.....	673
Assign.....	674
Geometry.....	675
Measurement.....	676
Stage settings.....	677
Analysis.....	678
Damage.....	679
Program Terrain.....	681
Project.....	681
Basic Data.....	682
Global Coordinate System.....	683
Soils.....	684
Assign.....	685
Points.....	686
Import of Points.....	687
Automatic Calculation of Height.....	688
Edges.....	690
Water.....	691
Bore Holes.....	692
Earth Grading.....	694
Generate.....	696
Modeling Terrain on Edges.....	696

---

---

Point Constructions.....	697
Line Constructions.....	699
Launching.....	700
Program FEM.....	702
Topology.....	702
Coordinate Systems.....	703
Project.....	705
Settings.....	706
Stability Analysis.....	707
Plane Strain Analysis.....	707
Axial Symmetry.....	708
Tunnels.....	711
Consolidation.....	712
Principle of Numerical Solution of Consolidation.....	712
Advanced Input.....	716
Ko Procedure.....	717
Water Flow.....	718
Flow Analysis.....	719
Interface.....	720
Soils.....	721
Materials Models.....	722
Linear Models.....	723
Elastic Model.....	724
Modified Elastic Model.....	724
Nonlinear Models.....	725
Mohr-Coulomb (MC).....	727
Mohr-Coulomb Model with Tension Cut Off.....	727
Modified Mohr-Coulomb (MCM).....	728
Drucker-Prager.....	728
Softening and Hardening.....	729
Angle of Dilation.....	730
Influence of Material Model.....	731
Modified Cam-Clay Model (MCC).....	732
Generalized Cam-Clay Model (GCC).....	734
Numerical Implementation of MCC and GCC Models.....	736
Hypoplastic Clay.....	739
Material Models in Flow Analysis.....	745

---

---

Coefficient of Permeability.....	749
Basic Data.....	751
Geostatic Stress, Uplift Pressure.....	753
Rigid Bodies.....	754
Assign.....	756
Contact Types.....	756
Contact Elements.....	758
Lining.....	759
Module Lining - FEM.....	761
Free Points.....	761
Free Lines.....	762
Line Refinement.....	763
Settings.....	764
Generator of Lining Shape.....	765
Generator of Anchored Regions.....	767
Stages of Construction.....	768
Free Points.....	769
Free Lines.....	771
Point Refinement.....	772
Line Refinement.....	773
Mesh Generation.....	775
Mesh Generator Warning.....	777
Adjusting Original Geometry.....	779
Standard Boundary Conditions.....	779
Construction Stages.....	780
Activation.....	781
Activity of Regions Below GWT.....	783
Assign.....	784
Lining.....	785
Beams.....	786
Anchors.....	787
Beam Loads.....	788
Generator of Anchors on Free Line.....	789
Beams.....	790
Types of Cross-Section.....	792
Beam End-Points Connection.....	793
Degradation and Strengthening of Beams.....	794

---

---

Catalog of Profiles.....	795
Cross-Section Editor.....	796
Catalog of Materials.....	796
Editor of Materials.....	797
Contacts.....	798
Contacts and Beams (Water Flow).....	799
Point Supports.....	801
Point Flow.....	802
Line Supports.....	803
Line Flow.....	804
Anchors.....	805
Anchor End Points.....	807
Anchors in the Stability Analysis.....	808
Props.....	809
Reinforcements.....	811
Anchoring Geo-Reinforcements.....	813
Axial Stiffness of Geosynthetics.....	815
Surcharge.....	816
Beam Loads.....	818
Water.....	820
Analysis.....	821
Transient Flow Analysis.....	822
Recommended Modeling Procedure.....	828
Loss of Convergence of Nonlinear Analysis.....	829
Settings and Analysis Description.....	830
Solution Method.....	831
Change of Stiffness Matrix.....	831
Initial Solution Step.....	832
Maximum Number of Iterations.....	833
Convergence Criterion.....	833
Setting Newton-Raphson Method.....	833
Setting Arc-Length Method.....	834
Setting Arc Length.....	835
Automatic Arc Length Control.....	836
Line Search Method.....	837
Plasticity.....	838
Course of Analysis.....	839

---

Results.....	840
Results Tool Bar.....	841
Results Visualization Settings.....	841
List of Variables.....	842
Monitors.....	844
Monitors Settings.....	845
Graphs.....	846
Stability.....	847
Setting Basic Parameters of Slope Stability Analysis.....	848
Setting Driving Parameters of Relaxation of Reduction Factor.....	849
Outputs.....	850
Adding Pictures.....	851
List of Pictures.....	852
Print and Export Document.....	853
Print and Export Picture.....	854
Control Menu - Print and Export.....	855
Tool bar - Print and Export.....	856
Setting Header and Footer.....	857
Page Properties.....	859
Page Numbering.....	859
About the Company.....	860
Theory.....	862
Stress in Soil Body.....	862
Geostatic Stress, Uplift Pressure.....	862
Effective/Total Stress in a Soil.....	863
Increment of Earth Pressure due to Surcharge.....	865
Increment of Earth Pressure under Footing.....	866
Earth Pressures.....	867
Sign Convention.....	867
Active Earth Pressure.....	868
Active Earth Pressure - The Mazindrani Theory (Rankine).....	869
Active Earth Pressure - The Coulomb Theory.....	870
Active Earth Pressure - The Müller-Breslau Theory.....	871
Active Earth Pressure - The Caquot Theory.....	872
Active Earth Pressure - The Absi Theory.....	873
Active Earth Pressure - Total Stress.....	874
Passive Earth Pressure.....	874

---

---

Passive Earth Pressure - The Rankine and Mazindrani Theory.....	875
Passive Earth Pressure - The Coulomb Theory.....	876
Passive Earth Pressure - The Caquot - Kérisel Theory.....	877
Coefficient of Passive Earth Pressure $K_p$ .....	877
Reduction Coefficient of Passive Earth Pressure.....	880
Passive Earth Pressure - The Müller - Breslau Theory.....	880
Passive Earth Pressure - The Absi Theory.....	881
Passive Earth Pressure - The Sokolovski Theory.....	882
Passive Earth Pressure - Total Stress.....	884
Earth Pressure at Rest.....	885
Earth Pressure at Rest for an Inclined Ground Surface or Inclined Back of the Structure .....	886
Alternate Angle of Internal Friction of Soil.....	886
Distribution of Earth Pressures in case of Broken Terrain.....	887
Influence of Water.....	888
Without Ground Water, Water is not Considered.....	889
Hydrostatic Pressure, Ground Water behind the Structure.....	889
Hydrostatic Pressure, Ground Water behind and in front of the Structure.....	890
Hydrodynamic Pressure.....	891
Special Distribution of Water Pressure.....	892
Uplift Pressure in Footing Bottom.....	893
Influence of Tensile Cracks.....	894
Minimum Dimensioning Pressure.....	894
Earth - Pressure Wedge.....	895
Surcharge.....	897
Surface Surcharge - Active Earth Pressure.....	897
Strip Surcharge - Active Earth Pressure.....	898
Trapezoidal Surcharge - Active Earth Pressure.....	899
Concentrated Surcharge - Active Earth Pressure.....	899
Line Surcharge - Active Earth Pressure.....	900
Surcharge in Non-Homogeneous Soil.....	902
Surface Surcharge - Earth Pressure at Rest.....	902
Strip Surcharge - Earth Pressure at Rest.....	902
Trapezoidal Surcharge - Earth Pressure at Rest.....	903
Concentrated Surcharge - Earth Pressure at Rest.....	903
Surface Surcharge - Passive Earth Pressure.....	904
Influence of Earthquake.....	904

---

Mononobe-Okabe Theory.....	907
Arrango Theory.....	908
Influence of Water.....	909
EN 1998-5 Seismic Effects.....	911
Forces from Earth Pressure at Rest Acting on the Rigid Structure.....	911
Influence of Earthquake according to Chinese Standards.....	912
Influence of Earthquake according to JTJ 004-89.....	912
Influence of Earthquake according to JTS 146-2012.....	913
Influence of Earthquake according to SL 203-97.....	916
Seismic Fortification Intensity according to Chinese Standards.....	917
Water Influence according to Chinese Standards.....	918
Importance Coefficient for Seismic Design $C_i$ .....	920
Adjusting Coefficient for Seismic Bearing Capacity $\xi_a$ .....	921
Influence of Friction between Soil and back of the Structure.....	921
Table of Ultimate Friction Factors for Dissimilar Materials.....	923
Adhesion of Soil.....	924
Parameters of Rocks.....	924
Analysis of Walls.....	926
Evaluation of Forces in the Footing Bottom.....	926
Verification - Limit States.....	927
Verification - Safety Factor.....	928
Internal Sliding.....	929
Reinforcements.....	930
Base Anchorage.....	931
Accounting for Wall Jump.....	932
Dimensioning of Masonry Wall According to AS 3700.....	933
Dimensioning of Masonry Wall According to EN 1996-1-1.....	935
Dimensioning of Gravity Wall - Masonry According to EN 1996-1-1.....	936
Dimensioning of Gravity Wall - Masonry According to GB 50003-2011.....	937
Bearing Capacity of Foundation Soil.....	939
Wall Dimensioning.....	940
Internal Stability of a Gabion.....	941
Internal Stability of a Gabion Wall - Safety Factor.....	944
Internal stability of a Gabion Wall - Limit States.....	945
Calculating Abutment Forces.....	946
Reduced Passive Earth Pressure.....	947
Nailed Slope.....	948

---

Analysis of Internal Stability.....	948
Analysis of Bearing Capacity of the Nails.....	949
Estimated Bond Strength.....	951
Total Bearing Capacity of a Nail.....	953
Verification - Factor of Safety.....	953
Verification - Theory of Limit States.....	954
Nail Force.....	955
Dimensioning of Concrete Cover.....	955
Sheeting Design.....	956
Analysis of Pile Sheeting Wall.....	956
Analysis of Anchored Wall Fixed in Heel.....	957
Analysis of Anchored Wall Simply Supported at Heel.....	958
Sheeting Check.....	960
Method of Dependent Pressures.....	961
Spring Method According to JGJ 120-2012.....	962
Modulus of Subsoil Reaction.....	963
Modulus of Subsoil Reaction According to Schmitt.....	964
Modulus of Subsoil Reaction According to Chadeisson.....	964
Modulus of Subsoil Reaction According to CUR 166.....	965
Modulus of Subsoil Reaction Determined from Iteration.....	966
Modulus of Subsoil Reaction According to Menard.....	968
Modulus of Subsoil Reaction According to NF P 94-282.....	969
Modulus of Subsoil Reaction Specified by Dilatometric Test (DMT).....	970
Modulus of Subsoil Reaction According to Chinese standards.....	971
Nonlinear Modulus of Subsoil Reaction.....	972
Braced Sheeting.....	973
Automatic Calculation of the Coefficient of Pressure Reduction Below Ditch Bottom.....	974
Strengthening of the Soil.....	974
Internal Stability of Anchors.....	975
Failure by Heave.....	977
Anti-Slide Pile.....	978
Determination of Forces Acting on an Anti-Slide Pile.....	979
Distribution of Pressures Above the Slip Surface.....	981
Shaft.....	981
Calculation of Load Acting on a Shaft.....	981
Flexible Shaft Structure.....	983
Semirigid Shaft Structure.....	984

---



---

Rigid Shaft Structure .....	985
Calculation of Internal Forces on a Shaft (Dimensioning).....	986
Slope Stability.....	987
Soil Body.....	987
Influence of Water.....	987
Surcharge.....	990
Anchors.....	991
Reinforcements.....	991
Reinforcement End.....	993
Anti-Slide Piles.....	994
Influence of an Earthquake.....	995
Earthquake Effect - Standard Analysis.....	996
Earthquake Analysis According to JTJ 004-89.....	997
Earthquake Analysis According to SL 203-97.....	997
Verification According to EN 1997.....	998
Analysis According to the Theory of Limit States / Safety Factor.....	999
Polygonal Slip Surface.....	999
Sarma.....	1001
Spencer.....	1003
Janbu.....	1005
Morgenstern-Price.....	1009
Shahunyants.....	1012
ITF Method (Imbalance Thrust Force Method).....	1015
Optimization of Polygonal Slip Surface.....	1019
Changing the Inclination of Dividing Planes.....	1019
Circular Slip Surface.....	1020
Fellenius / Petterson.....	1021
Bishop.....	1021
Spencer.....	1022
Janbu.....	1022
Morgenstern-Price.....	1022
Shahunyants.....	1023
ITF Method (Imbalance Thrust Force Method).....	1023
Optimization of Circular Slip Surface.....	1023
Foliation.....	1023
Influence of Tensile Cracks.....	1023
Rock Stability.....	1024

---

---

Plane Slip Surface.....	1025
Stepped Slip Surface.....	1025
Tensile Strength of Rock.....	1026
Undulated Slip Surface.....	1027
Anchorage of Rock Slope.....	1028
Surcharge of Rock Slope.....	1028
Influence of Water Acting on Slip Surface.....	1029
GWT Above Slope Toe.....	1030
GWT on Tension Crack.....	1030
GWT on Tension Crack, Max. Tens. Crack.....	1032
Water Acting Only on Tension Crack.....	1033
Own Water Force Acting Only on Slip Surface.....	1034
Own Water Force Behavior.....	1034
Polygonal Slip Surface.....	1035
Geometry of Rock Block.....	1036
Anchor Forces, Surcharge.....	1036
Influence of Water.....	1037
Solution Procedure.....	1038
Cone Friction Concept.....	1040
Rock Wedge.....	1041
Geometry of Rock Wedge.....	1042
Stereographic Projection.....	1043
Influence of Ground Water.....	1045
Resolution of Acting Forces.....	1046
Verification.....	1047
Verification According to the Factor of Safety.....	1047
Verification According to the Theory of Limit States.....	1047
Rock - Shear Resistance Criteria.....	1048
Mohr - Coulomb.....	1048
Mohr - Coulomb Parameters.....	1048
Hoek - Brown.....	1049
Parameters Hoek - Brown.....	1050
Calculation of Hoek-Brown Parameters.....	1053
Barton - Bandis.....	1055
Barton - Bandis Parameters.....	1055
Unit Weight of Rocks.....	1058
Influence of Seismic Effects.....	1059

---

---

MSE Wall.....	1061
Internal Stability.....	1061
Verification - Safety Factor.....	1063
Verification - Limit States.....	1063
Shapes of Slip Surfaces.....	1064
Extensible Reinforcements - Active Earth Pressure.....	1065
Inextensible Reinforcements - Combination of Earth Pressures.....	1066
Analysis of Foundation Bearing Capacity.....	1067
Bearing Capacity on Drained Subsoil.....	1068
Standard Analysis.....	1069
Bearing Capacity on Undrained Subsoil.....	1070
Standard Analysis.....	1071
Bearing Capacity of Foundation on Bedrock.....	1071
Standard Analysis.....	1072
Solution According to CSN 73 1001.....	1072
Analysis According to EC 7-1 (EN 1997-1:2003).....	1073
Parameters to Compute Foundation Bearing Capacity.....	1073
Horizontal Bearing Capacity of Foundation.....	1077
Homogenization of Layered Subsoil.....	1078
Effective Area.....	1080
Determination of Cross-Sectional Internal Forces.....	1081
Verification of Foundation Eccentricity.....	1082
Analysis of Uplift.....	1083
Standard Approach.....	1083
Cone Method.....	1084
DL/T 5219 - 2005.....	1085
Pile Analysis.....	1087
Vertical Bearing Capacity.....	1087
Analytical Solution.....	1087
NAVFAC DM 7.2.....	1088
Pile Base Resistance.....	1088
Pile Shaft Resistance.....	1088
Bearing Capacity Factor $N_q$ .....	1089
Coefficient of Lateral Earth Pressure $K$ .....	1089
Friction Angle on Pile Skin.....	1090
Adhesion Coefficient.....	1091
Critical Depth.....	1091

---

Tomlinson.....	1092
Adhesion Coefficient.....	1093
Effective Length.....	1093
Effective Stress Method.....	1094
Coefficients of Pile Bearing Capacity.....	1094
CSN 73 1002.....	1095
Verification.....	1096
Verification According to the Theory of Limit States.....	1096
Design Coefficients.....	1097
Verification According to the Safety Factors.....	1097
Vertical Bearing Capacity - Spring Method.....	1098
Load-Settlement Curve.....	1099
Shear Strength of Skin.....	1099
Coefficient of Increase of Limit Skin Friction.....	1100
Depth of Deformation Zone.....	1100
Incompressible Subsoil.....	1103
Negative Skin Friction.....	1103
Influence of Technology.....	1104
Shear Resistance on Skin.....	1104
Stiffness of Subsoil Below the Pile Heel.....	1106
Distributions of Forces Acting on a Pile.....	1106
Dependence of Shear on Deformation.....	1107
Pile Settlement.....	1107
Nonlinear Theory (Masopust).....	1107
Approach According to Masopust.....	1108
Regression Coefficients.....	1110
Coefficients $m_1$ , $m_2$ .....	1111
Secant Modulus of Soil $E_s$ .....	1112
Settlement-Influence Factor $I_s$ .....	1114
Linear Theory (Poulos).....	1115
Settlement of Piles According to Poulos.....	1115
Secant Modulus of Soil $E_s$ .....	1117
Correction Factor for Soil Poisson's Ratio $R_v$ .....	1118
Correction Factor for Stiffness of Bearing Stratum $R_b$ .....	1119
Base-Load Proportion for Incompressible Pile $BETA_o$ .....	1120
Correction Factor for Pile Compressibility $C_k$ .....	1121
Correction Factor for Poisson's Ratio of Soil $C_v$ .....	1122

Correction Factor for Stiffness of Bearing Stratum $C_b$ .....	1123
Pile-Stiffness Factor $K$ .....	1124
Basic Settlement-Influence Factor $I_o$ .....	1124
Correction Factor for Pile Compressibility $R_k$ .....	1125
Correction Factor for Finite Depth of Layer on a Rigid Base $R_h$ .....	1126
Horizontal Bearing Capacity - Elastic Subsoil (p-y Method).....	1127
Constant Distribution of Modulus of Subsoil Reaction.....	1128
Linear Modulus of Subsoil Reaction.....	1128
Modulus of Subsoil Reaction According to CSN 73 1004.....	1129
Modulus of Subsoil Reaction According to Matlock and Reese.....	1130
Modulus of Subsoil Reaction According to Vesic.....	1131
Pile Horizontal Bearing Capacity - Brom's Method .....	1132
Pile CPT.....	1134
Classification of Soils According to Robertson.....	1134
Coefficient of Penetrometer (Net Area Ratio).....	1138
Bearing Capacity.....	1139
EN 1997-2.....	1139
NEN 6743.....	1140
LCPC (Bustamante).....	1141
Determination of Equivalent Average Cone Tip Resistance.....	1141
Schmertmann.....	1142
Determination of Average Cone Tip Resistance.....	1143
Correlation Coefficient $K$ .....	1144
Negative Skin Friction.....	1145
Shaft Friction Coefficient ALFAs.....	1146
Influence of Overconsolidation (OCR).....	1149
Coefficient of Influence of Pile Shape $s$ .....	1150
Coefficient of Influence of Pile Widened Base BETA.....	1150
Coefficient of Reduction of a Pile Base Bearing Capacity ALFA p.....	1151
Pile Group.....	1152
Calculation of Pile Toe Settlement.....	1153
Graphs to Calculate Settlement.....	1153
Calculation of Load-Settlement Curve.....	1155
Verification.....	1155
Verification According to EN 1997-2.....	1155
Correlation Coefficients for Evaluating Standard Values of Bearing Capacity.....	1157
Verification According to the Safety Factor.....	1157

Verification According to Limit States.....	1158
Pile Group.....	1158
Analytical Solution.....	1158
Cohesionless Soil (Analysis for Drained Conditions).....	1159
Efficiency of a Pile Group.....	1159
Cohesive Soil (Analysis for Undrained Conditions).....	1160
Analysis According to the Safety Factor.....	1160
Analysis According to the Theory of Limit States.....	1161
Pile Group Settlement.....	1161
Spring Method.....	1163
Calculation of Stiffness of Vertical Springs.....	1164
Micropile.....	1164
Verification Based on Safety Factor.....	1165
Verification Based on Limit States.....	1165
Verification of the Micropile Tube.....	1166
Coupled Section Bearing Capacity.....	1167
Micropile Lifetime.....	1167
Coefficient $F_{ut}$ .....	1167
Coefficient of the Influence of Corrosion.....	1168
Bearing Capacity of Cross-Section Loaded by Normal Force.....	1168
Bearing Capacity of Cross-Section Loaded by Combination of Bending Moment and Normal Force.....	1169
Influence of Buckling.....	1170
Internal Stability of Section.....	1171
Geometric Method (Euler).....	1172
Salas Theory.....	1174
Constant A Reflecting the Type of Support in the Micropile Head.....	1175
Coefficient $f$ .....	1175
Véas-Souche Theory.....	1175
Modulus of Horizontal Reaction of Subsoil.....	1176
Calculation of the Modulus of Horizontal Reaction of Subsoil $E_r$ .....	1177
Values of the Modulus of Subsoil Reaction $E_p$ .....	1178
Bearing Capacity of the Micropile Root Section.....	1179
Lizzi Theory.....	1179
Skin Friction of the Micropile Root.....	1179
Littlejohn Theory.....	1181
Zweck Theory.....	1181

---

Bowles Theory.....	1182
Véas Theory.....	1182
Coefficients of Type of Application of Micropile.....	1184
Bearing Capacity of the Root in Rock.....	1184
Skin Friction and Bearing Capacity of the Micropile Root in Rock.....	1184
Bustamante (SPT, Pressiometer PMT).....	1185
Skin Friction of the Micropile Root - Graphs.....	1185
Field Testing.....	1188
Cone Penetration Tests (CPT).....	1188
Standard Penetration Tests (SPT).....	1188
Pressiometric Tests (PMT).....	1190
Dilatometric Test (DMT).....	1190
Settlement Analysis.....	1191
Stress in the Footing Bottom.....	1192
Overall Settlement and Rotation of Foundation.....	1193
Influence of Foundation Depth and Incompressible Subsoil.....	1194
Influence of Sand-Gravel Cushion.....	1195
Analysis Using the Oedometric Modulus.....	1196
Analysis Using the Compression Constant.....	1197
Analysis Using the Compression Index.....	1197
Analysis According to NEN (Buismann, Ladd).....	1197
Analysis Using the Soft Soil Model.....	1199
Analysis According to the Janbu Theory.....	1200
Analysis for Cohesionless Soils.....	1200
Analysis for Coarse-Grained Soils.....	1200
Analysis for Sands and Silts.....	1201
Analysis for Overconsolidated Sands and Silts.....	1201
Analysis for Cohesive Soils.....	1202
Analysis for Overconsolidated Cohesive Soils.....	1202
Settlement Analysis Using DMT (Constrained Soil Modulus).....	1203
Theory of Settlement.....	1203
Primary Settlement.....	1205
Secondary Settlement.....	1207
Consolidation Analysis.....	1208
Determination of the Influence Zone Depth.....	1210
Theory of Structural Strength.....	1210
Method of Restriction of the Primary Stress Magnitude.....	1211

---

Characteristics of Settlement Analyses.....	1212
Compression Index.....	1212
Oedometric Modulus.....	1215
Compression Constant.....	1216
Compression Constant 10.....	1217
Void Ratio.....	1218
Recompression Index.....	1218
Janbu Characteristics.....	1219
Influence of Load History.....	1220
Coefficient m.....	1221
Modified Compression Index.....	1221
Index of Secondary Compression.....	1222
Overconsolidation Index of Secondary Compression.....	1224
Ground Loss.....	1224
Analysis of Subsidence Trough.....	1224
Volume Loss.....	1224
Recommended Values of Parameters for Volume Loss Analysis.....	1225
Classic Theory.....	1227
Analysis for Layered Subsoil.....	1228
Shape of Subsidence Trough.....	1230
Coefficient of Calculation of Inflection Point.....	1230
Subsidence Trough with Several Excavations.....	1231
Analysis of Subsidence Trough in Depth.....	1231
Calculation of Other Variables.....	1232
Analysis of Failure of Buildings.....	1232
Tensile Cracks.....	1233
Gradient Damage.....	1233
Relative Deflection.....	1234
Failure of a Section of a Building.....	1234
Dimensioning of Concrete Structures.....	1235
EN 1992-1-1 (EC2) or EN 1992-2.....	1236
Materials, Coefficients, Notation.....	1236
Standard Values of Coefficients.....	1237
Verification of Rectangular Cross-Section Made of Plain Concrete.....	1239
Verification of Rectangular RC Cross-Section Under M, V.....	1240
Verification of Rectangular RC Cross-Section Under N, M, V.....	1241
Verification of Circular RC Cross-Section.....	1242



Verification of Spread Footing for Punching Shear.....	1244
Design of Longitudinal Reinforcement for Slabs.....	1245
Design of Shear Reinforcement for Slabs.....	1246
Verification of Crack Width.....	1246
CSN 73 1201 R.....	1247
Materials, Coefficients, Notation.....	1247
Verification of Rectangular Cross-Section Made of Plain Concrete.....	1248
Verification of Rectangular RC Cross-Section Under M, V.....	1249
Verification of Rectangular RC Cross-Section Under N, M, V.....	1251
Verification of Circular RC Cross-Section.....	1252
Verification of Spread Footing for Punching Shear.....	1253
Design of Longitudinal Reinforcement for Slabs.....	1254
Design of Shear Reinforcement for Slabs.....	1255
CSN 73 6206.....	1255
PN-B-03264:2002.....	1256
Materials, Coefficients, Notation.....	1257
Verification of Rectangular Cross-Section Made of Plain Concrete.....	1257
Verification of Rectangular RC Cross-Section Under M, V.....	1259
Verification of Rectangular RC Cross-Section Under N, M, V.....	1260
Verification of Circular RC Cross-Section.....	1261
Verification of Spread Footing for Punching Shear.....	1261
Design of Longitudinal Reinforcement for Slabs.....	1262
Design of Shear Reinforcement for Slabs.....	1263
BS 8110:1997.....	1264
Materials, Coefficients, Notation.....	1264
Verification of Rectangular Cross-Sections Made from Plain Concrete.....	1265
Verification of Rectangular RC Cross-Section Under M, V.....	1266
Verification of Rectangular RC Cross-Section Under N, M, V.....	1267
Verification of Circular RC Cross-Section.....	1268
Verification of Spread Footing for Punching Shear.....	1268
Design of Longitudinal Reinforcement for Slabs.....	1270
Design of Shear Reinforcement for Slabs.....	1271
IS 456.....	1271
Materials, Coefficients, Notation.....	1272
Verification of Rectangular Cross-Sections Made from Plain Concrete.....	1272
Verification of Rectangular RC Cross-Section Under M, V.....	1273
Verification of Rectangular RC Cross-Section Under N, M, V.....	1274

---

Verification of Circular RC Cross-Section.....	1275
Verification of Spread Footing for Punching Shear.....	1276
Design of Longitudinal Reinforcement for Slabs.....	1277
Design of Shear Reinforcement for Slabs.....	1278
IS Road Bridges.....	1279
ACI 318-11.....	1279
Materials, Coefficients, Notation.....	1279
Verification of Rectangular Cross-Section Made of Plain Concrete.....	1279
Verification of Rectangular RC Cross-Section Under M, V.....	1280
Verification of Rectangular RC Cross-Section Under N, M, V.....	1281
Verification of Circular RC Cross-Section.....	1282
Verification of Spread Footing for Punching Shear.....	1283
Design of Longitudinal Reinforcement for Slabs.....	1284
Design of Shear Reinforcement for Slabs.....	1285
AS 3600-2001.....	1286
Materials, Coefficients, Notation.....	1286
Verification of Rectangular Cross-Sections Made from Plain Concrete.....	1286
Verification of Rectangular RC Cross-Section Under M, V.....	1287
Verification of Rectangular RC Cross-Section Under N, M, V.....	1288
Verification of Circular RC Cross-Section.....	1289
Verification of Spread Footing for Punching Shear.....	1289
Design of Longitudinal Reinforcement for Slabs.....	1290
Design of Shear Reinforcement for Slabs.....	1291
SNIP 52-101-2003.....	1292
Materials, Coefficients, Notation.....	1292
Verification of Rectangular Cross-Sections Made from Plain Concrete.....	1293
Verification of Rectangular RC Cross-Section Under M, V.....	1294
Verification of Rectangular RC Cross-Section Under N, M, V.....	1295
Verification of Circular RC Cross-Section.....	1295
Verification of Spread Footing for Punching Shear.....	1296
Design of Longitudinal Reinforcement for Slabs.....	1297
Design of Shear Reinforcement for Slabs.....	1297
GB 50010-2010.....	1298
Materials, Coefficients, Notation.....	1298
Verification of Rectangular Cross-Sections Made from Plain Concrete.....	1299
Verification of Rectangular RC Cross-Section Under M, V.....	1300
Verification of Rectangular RC Cross-Section Under N, M, V.....	1301

---

Verification of Circular RC Cross-Section.....	1302
Verification of Spread Footing for Punching Shear.....	1303
Design of Longitudinal Reinforcement for Slabs.....	1305
Design of Shear Reinforcement for Slabs.....	1306
NZS 3101-2006.....	1307
Materials, Coefficients, Notation.....	1307
Verification of Rectangular Cross-Sections Made from Plain Concrete.....	1307
Verification of Rectangular RC Cross-Section Under M, V.....	1309
Verification of Rectangular RC Cross-Section Under N, M, V.....	1310
Verification of Circular RC Cross-Section.....	1311
Verification of Spread Footing for Punching Shear.....	1312
Design of Longitudinal Reinforcement for Slabs.....	1314
Design of Shear Reinforcement for Slabs.....	1314
Dimensioning of Steel Cross-Sections.....	1315
Verification According to EN 1993-1-1 (EC3).....	1317
Verification According to CSN 731401.....	1318
Verification According to the Safety Factor.....	1319
Verification According to the Theory of Limit States.....	1320
Verification According to GB 50017-2003.....	1321

## Using the Help

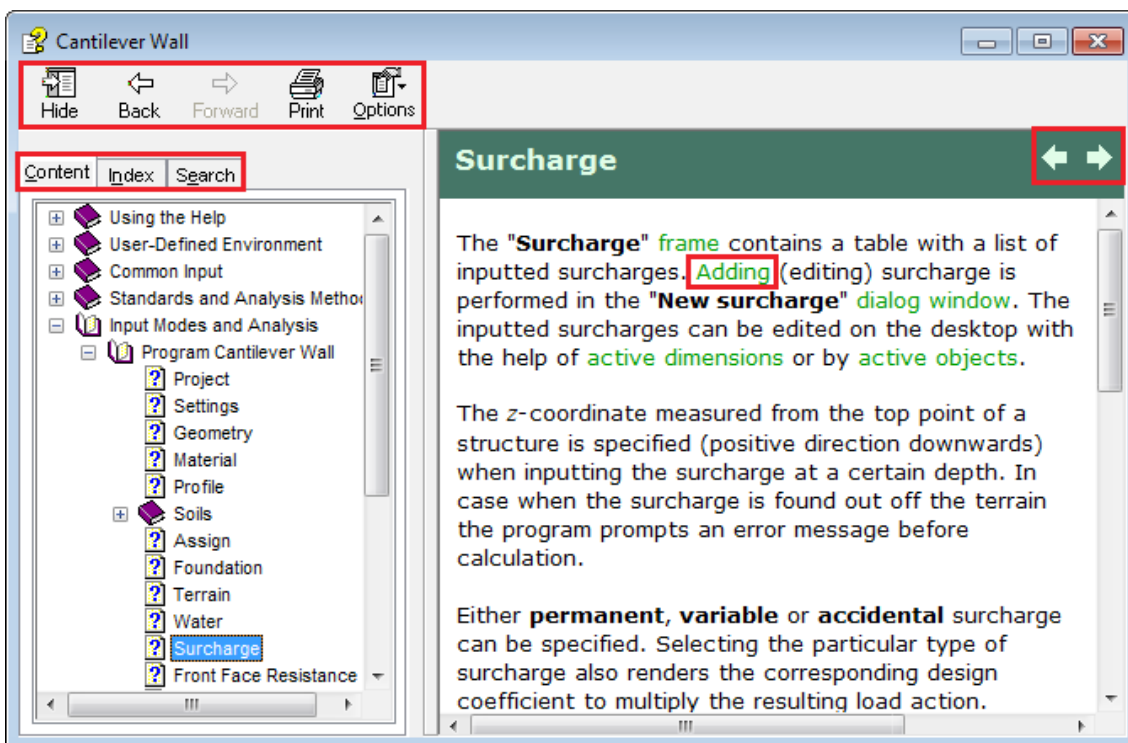
The text Help for all our programs is displayed in standard Windows Explorer window. Help can either be launched by directing through the program menu (items **"Help"**, **"Content"**), or by pressing **"F1"** button anywhere in the program.

Some dialog windows (e.g., **"Add new soils"**) allow opening a corresponding chapter of Help by pressing the help button **"?"**.

The dialog window contains:

- A tool bar with basic control buttons. The **"Hide (Show)"** button hides (shows) tree with the list of help topics. The **"Back/Forward"** buttons allow listing through pages, which have been recently opened. The **"Print"** button opens the print dialog window and the **"Options"** button opens a menu to set the EXPLORER window properties.
- Bar with three tabs - **"Contents"** (shows a tree with individual topics), **"Index"** and **"Search"**.
- **"Tree"**, which contains the list of topics - individual items in the tree are **opened/closed** by clicking the symbols **"+"**/**"-"** in front of the name.
- Window for displaying the help itself - the window header contains the name of currently shown topic and **"Back/Forward"** buttons that function in the same way as the already described buttons on the tool bar.

The text of each help may contain further cross references to other items. The text of these references is colored in green.

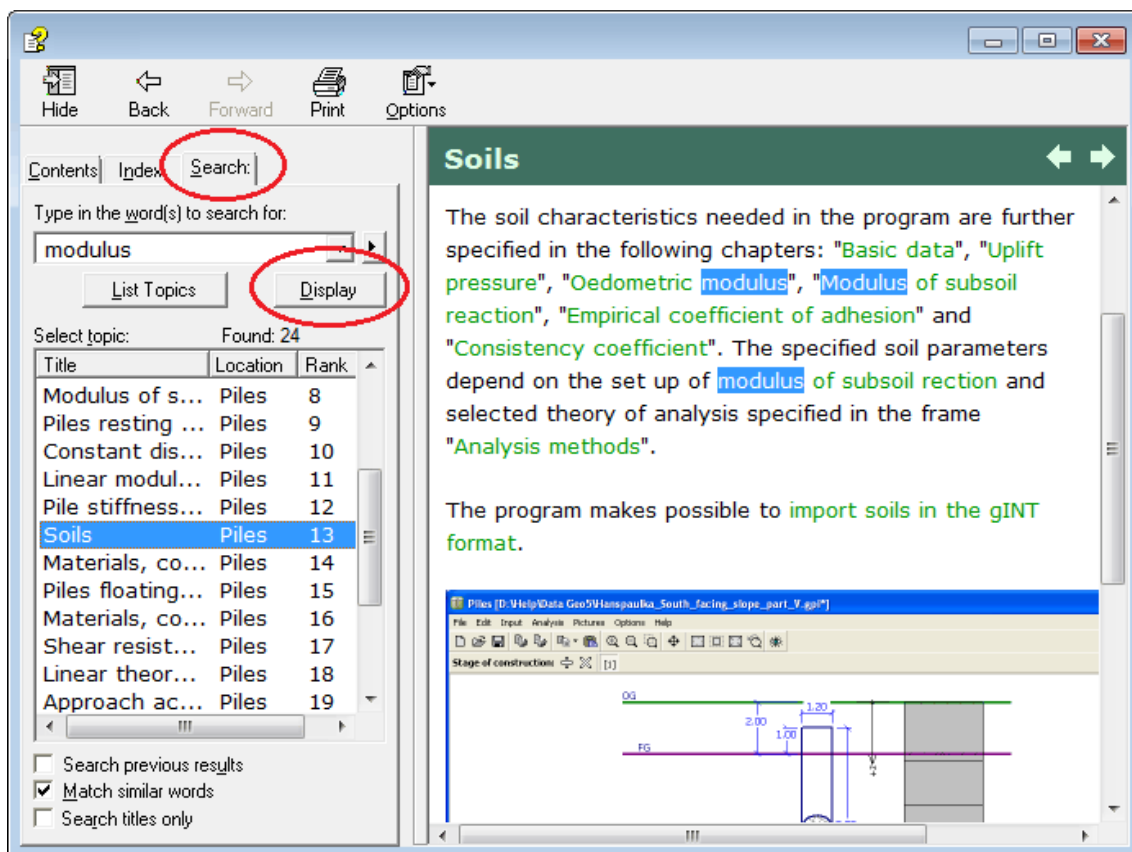


*"Help" dialog window - "Content" tab*

## Using the Search Function

The **"Search"** function allows you to find an arbitrary text in the help subjects. Write the search texts into the **"Input searched text"** field and the **"List of topics"** button launches the search. The list of found topics containing the searched text is displayed under the buttons in a column. Highlight the topic of your interest by clicking on it and the **"Display"** button opens the corresponding topic in the right part of the dialog windows (double-clicking option is also available.)

The searched text is highlighted in blue. Switching back to the **"Contents"** tab shows the topic location in the main tree (contents of the help).



Dialog window "Help" - "Search" tab

## User Interface

GEO5 programs are standard Windows applications and respect the standard properties of Windows interface.

**User environment** is described on pages:

- Application Window
- Control Menu
- Tools and Control Bars
- 3D Visualization
- Visualization Settings

- Frames
- Tables
- Dialog Windows

**Mouse functions** are described on pages:

- Active Dimensions and Objects
- Mouse Functions
- Mouse Context Menu

All GEO5 programs support two **sets of units** (**Metric / Imperial**).

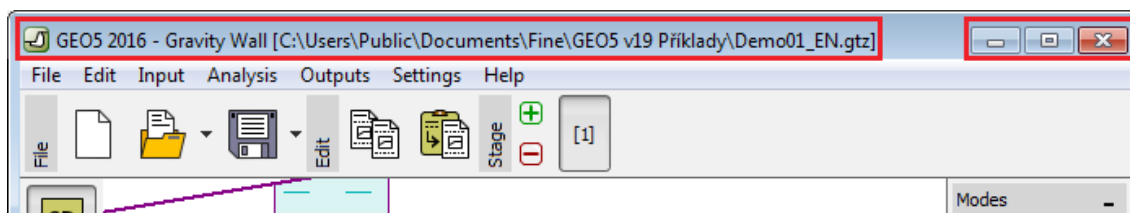
**Clipboard functions** are described on pages:

- Copy to Clipboard
- Geoclipboard

Programs allows to set other individual settings for **Print parameters**, **Copy to clipboard** and **Input parameters** (**undo - redo** function, **snap to grid**, horizontal and vertical **rulers**) in dialog window "Options".

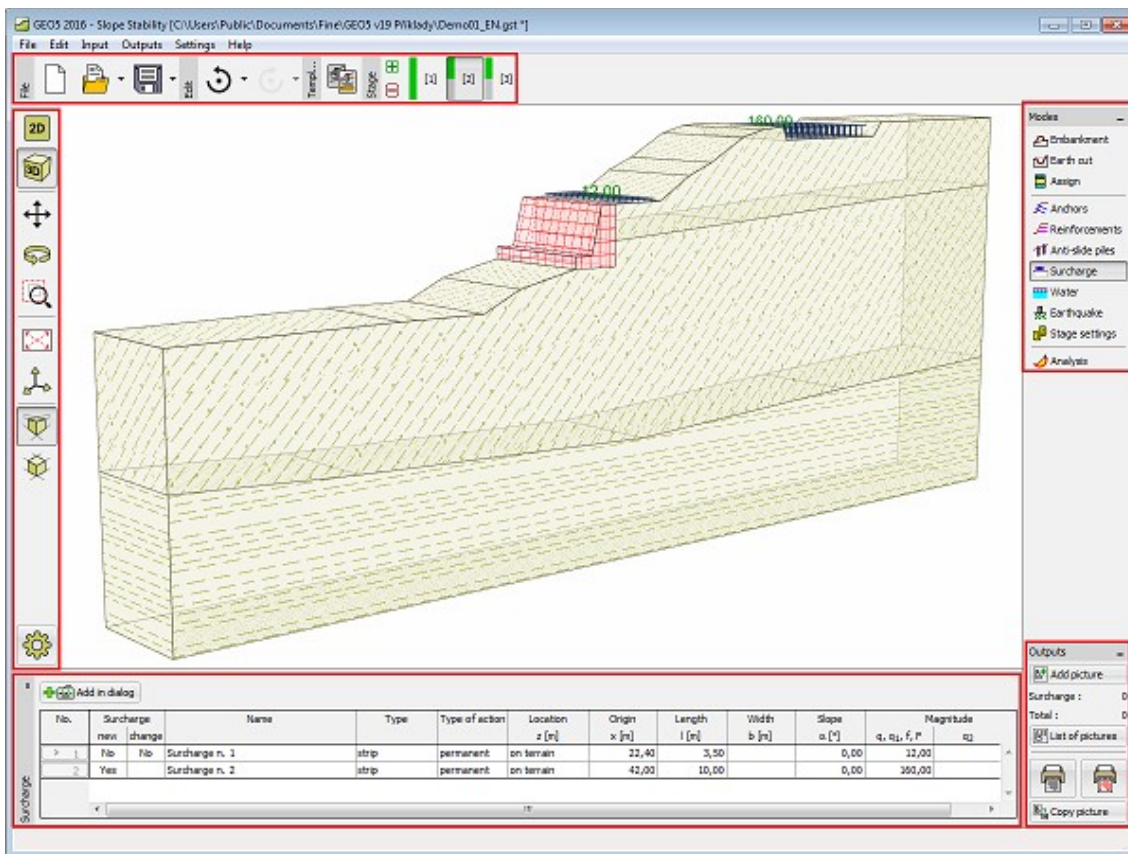
## Application Window

The application is launched in a standard dialog window containing all the managing tools typical for a Windows environment (minimizing, maximizing and closing the program window). The window header displays information about currently executed task (file name and location) - see picture:



*Control tools of the application window*

Program window consists of **control menu**, **tool and control bars** and desktop, which visualizes the executed task. The bottom part of the desktop displays **frames** that allows the user to introduce various input parameters into the task. Location of the individual elements on the desktop is evident from the following figure:



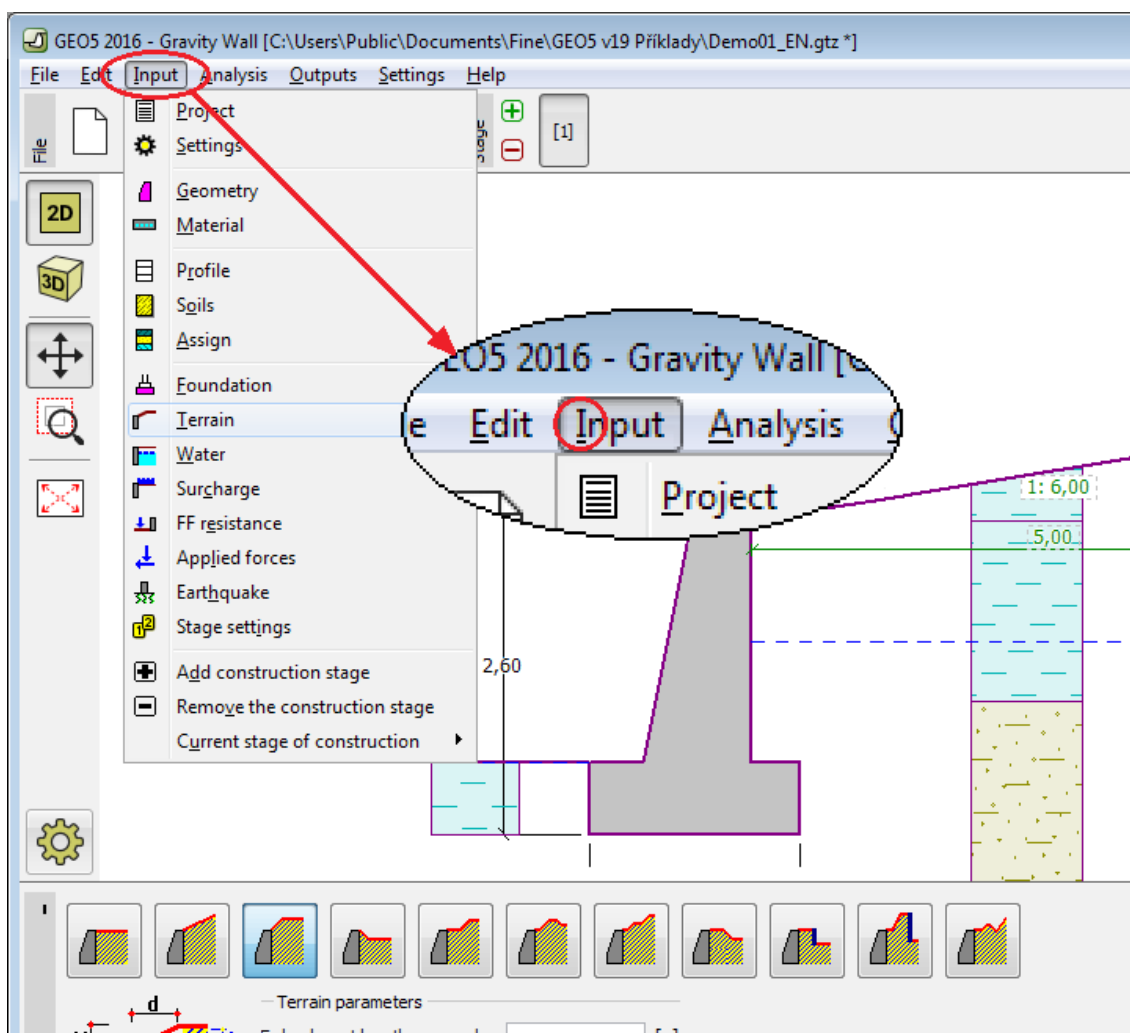
Program tools arrangement

## Control Menu

Item in the menu can be selected by clicking the **left mouse button** over it, or alternatively by using the keyboard by pressing **ALT + underlined letter** in the selected item menu.

As typical for the Windows environment, some options in the menu can be replaced by buttons on individual toolbars, or with keyboard shortcuts (existing keyboard shortcut is displayed next to the item in the menu - for example, **Save - CTRL + S**).

Some options in the program can only be set using the menu - e.g., "Options".



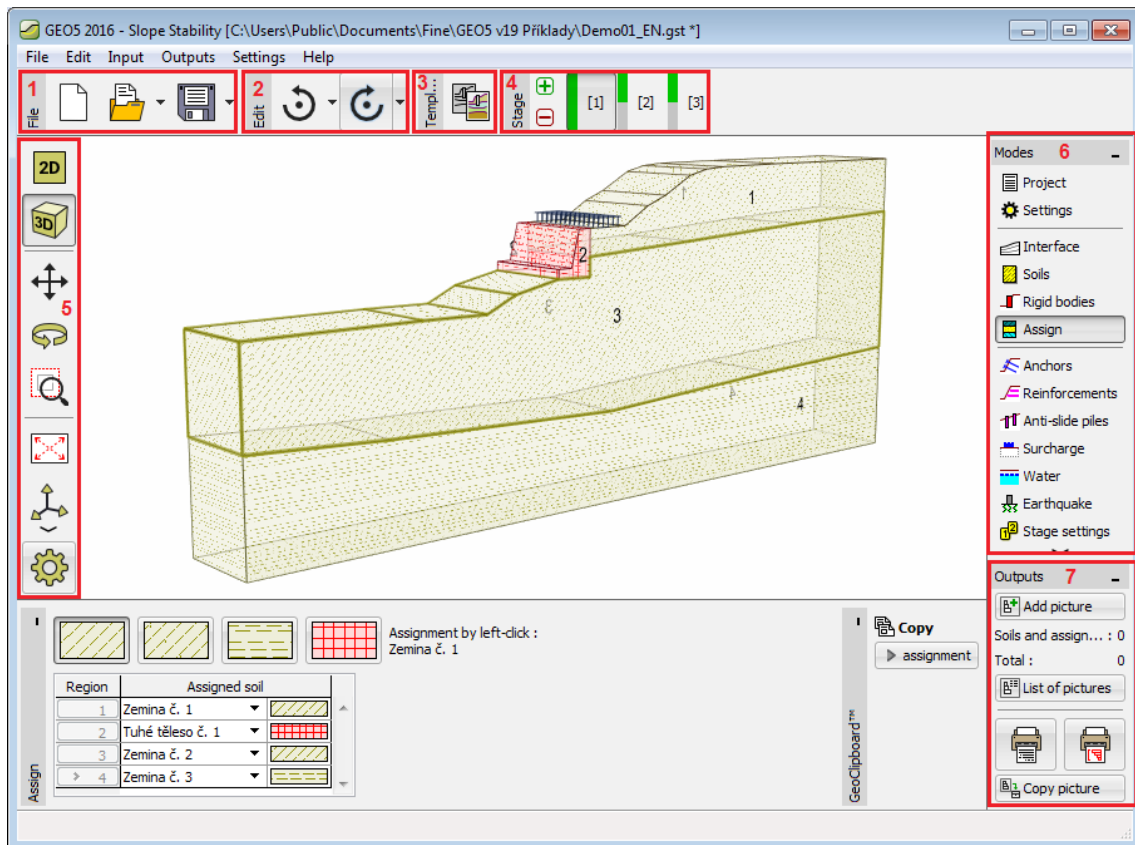
*Control menu of a GEO5 program*

## Tool and Control Bars

Each program contains the following tool and control bars:

- (1) File
- (2) Edit
- (3) Template
- (4) Stage
- (5) Visualization
- (6) Modes
- (7) Outputs





Tool and Control Bar Location on the Desktop

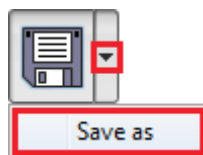
## File

Buttons on the **tool bar** are for working with files. The tool bar contains the following buttons:



Tool bar "File"

Several buttons are divided into two parts and the button can control more functions (right part with the arrow).







Using the button for more functions

Individual buttons functions are following:



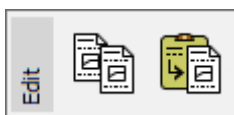
### New file

- Opens a new file - if there is an existing task opened in the same window, the program prompts the user to save unsaved data.

- 
**Open file**
  - Opens an existing file - if there is an existing task opened in the same window, the program prompts the user to save unsaved data.
- 
**Open recent files**
  - Opens a list of recently edited files.
- 
**Save data into file**
  - Saves data of currently opened task - if no name is assigned to the task, the program opens the "**Save as**" dialog window.
- 
**Save as**
  - Opens the "**Save as**" dialog window - currently running task can be saved under a different name or to a different location.

## Edit

Buttons on the **tool bar** are used for controls of data in a running task. The tool bar has a different appearance in 1D and 2D programs. The tool bar contains the following buttons:

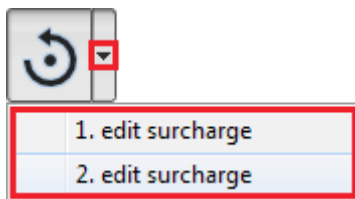


*Tool bar "Edit" - 1D programs*




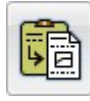

*Tool bar "Edit" - 2D programs*




Several buttons are divided into two parts and the button can control more functions (right part with the arrow).



*Using the button for more functions*

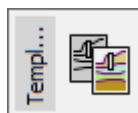
Individual buttons functions are following:

- 
**Copy data to clipboard**
  - **Copies** the data from the current task to clipboard
- 
**Paste**
  - Opens a dialog window and **pastes** selected data from a different GEO5 program - for example from the "**Earth Pressures**" program to "**Gravity Wall**" program.
- 
**Undo**
  - Returns the last performed step (the function is available only in programs with 2D environment and must be allowed in "**Options**").

- 
**Undo (more steps)**
  - Opens a list of steps, that can be undone.
- 
**Redo**
  - Restores one returned step (the function is available only in programs with 2D environment and must be allowed in "Options").
- 
**Redo (more steps)**
  - Opens a list of steps, that can be redone.

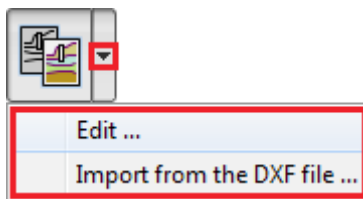
## Template

Buttons on the **tool bar** are used to work with DXF templates. The tool bar contains the following buttons:






*Tool bar "Template"*

Several buttons are divided into two parts and the button can control more functions (right part with the arrow). The button gets this function only after loading the template.



*Using the button for more functions*

Individual buttons functions are following:



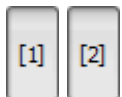
- 
**Template**
  - Imports **template from a DXF file**. If the template is loaded, turns **on / off** its visualization on the desktop.
- 
**Edit**
  - Opens a dialog window for **template layer editing**.
- 
**Import from the DXF file**
  - Opens a dialog window for **import** of a new template from a DXF file.

## Construction Stages

This **tool bar** manages the construction stages. The following picture shows the location of individual buttons:



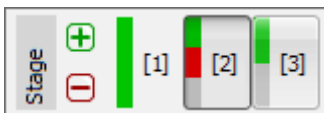
*Tool bar "Stages"*

- 
**Adds construction stage**
  - adds new construction stage at the end of list
- 
**Removes construction stage**
  - removes the last construction stage from the list
- 
**Construction stage 1,2 ...**
  - switches between individual stages of construction - selection is performed using the left mouse button

This bar allows to define stages of construction. Construction stages serve to model gradual building of the construction (**essential for programs Sheeting check, Settlement, FEM**). This function can also be used for parametric studies and in each construction stage assume different soil assignment or different design coefficients. It is rather advantageous to model earthquake effects on a structure in a separate stage of construction as it is then possible to assume different factors of safety or different design coefficients.




For individual types of input (soil assignment, anchors, supports...) there always exists relationship over construction stages ([Heredity](#)).

Some programs show construction stage analysis status using a color stripe.



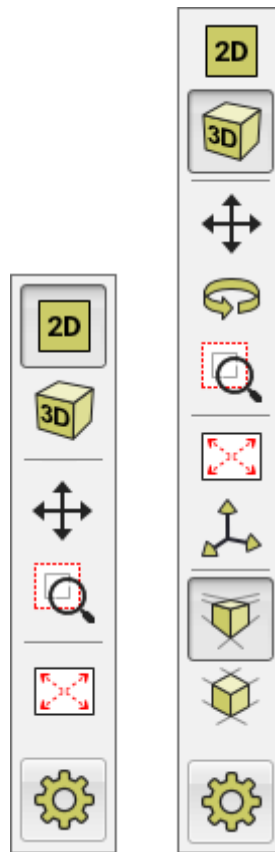
*"Stages" tool bar with analysis status color stripes*

The colors have the following meaning:

- 
  - green - there is an analysis in the construction stage which IS SATISFACTORY
- 
  - red - there is an analysis in the construction stage which IS NOT SATISFACTORY
- 
  - grey - there is an analysis in the construction stage which was not performed yet








## Visualization




Buttons on the tool bar allow the user to change setting of visualization on the desktop. The tool bar has different appearance in 2D and 3D mode. The tool bar contains the following buttons:



*Tool bar "Visualization" - tool bar appearance in 2D and 3D mode*

Functions of individual buttons functions are following:

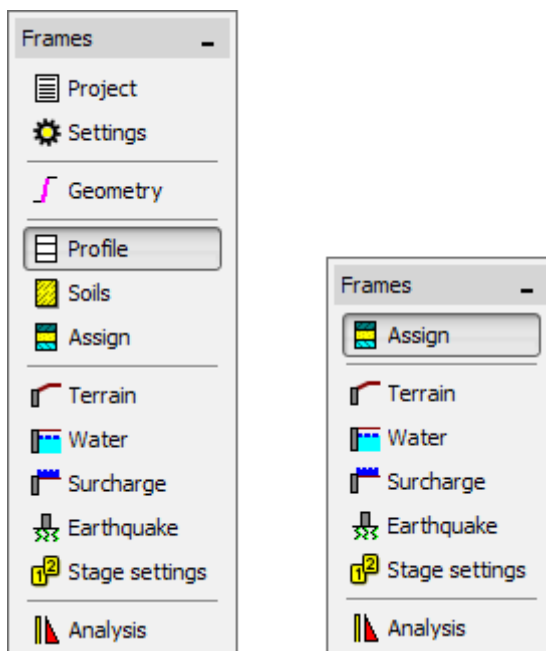
- 
**2D view**
  - Applies the 2D visualization mode.
- 
**3D view**
  - Applies the 3D visualization mode.
- 
**Move displayed area**
  - Moves the current view in an arbitrary direction - to proceed move mouse in the desired location while keeping the left mouse button pressed.
- 
**Rotates the scene**
  - Rotates the displayed drawing in an arbitrary direction (3D view) - to move the drawing slide the mouse while pressing the left mouse button.
- 
**Shows marked area**
  - Shows and scales up the marked region - the region is selected using the left mouse button.
- 
**Modify scale**
  - Scales the view such that all objects are visible (pressing the left mouse button).
- 
**Pre-defined 3D view**
  - Sets the predefined 3D view of drawing (3D view).

- 
**Perspective view**
  - Sets the perspective view of drawing (3D view).
- 
**Axonometric view**
  - Sets the axonometric view of drawing (3D view).
- 
**Visualization Settings**
  - The button opens the frame "Visualization Settings". The parameters for picture drawing can then be changed in the frame.

## Frames

The vertical **tool bars** let the user select the desired mode of inputting data (Project, Geometry, Profile etc.) including analysis type and verification. Selection of the mode from this bar displays in the bottom part of the desktop the corresponding **frame** for data input.

The tool bar only contains those frames, where the input of data makes sense. This means, that if a task has more **construction stages**, the tool bar is complete, however some items are missing in further construction stages. Data cannot be changed in the missing frames.



Control bar "Frames" for switching between input data modes

## Outputs

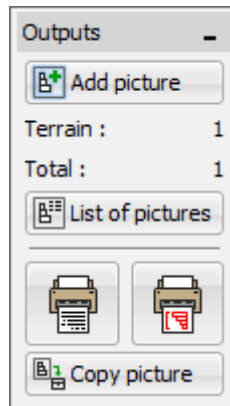
A standalone **tool bar** serves to manage pictures and output document.

The "**Add picture**" button opens the "New picture" dialog window. The next line in the bar provides the number of stored pictures in the given mode of data input. The "**Total**" line shows the total number of stored pictures for this file. The "**Picture list**" button opens the **list of pictures**.

The two other buttons open the dialog windows "**Print and Export Document**" and "**Print and**

Export Picture".

"Copy picture" buttons saves the current view from the desktop to clipboard.

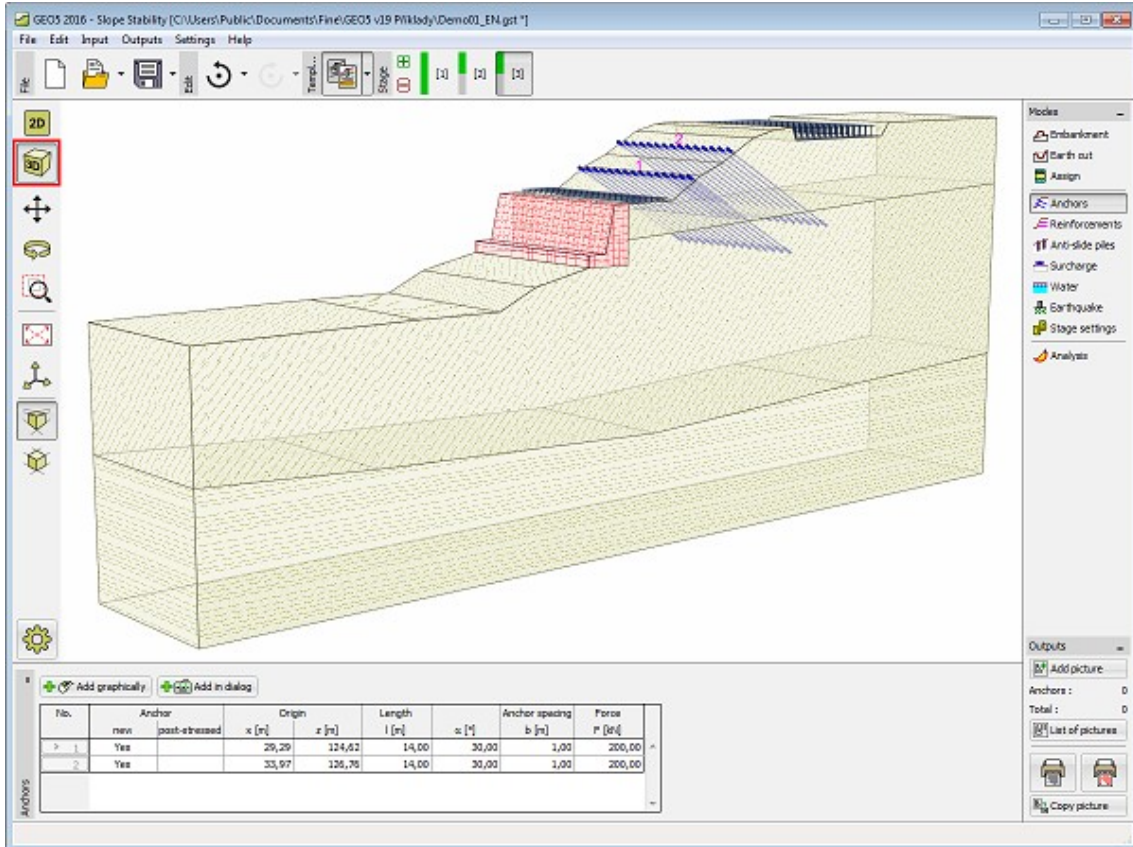


Control bar "Outputs"

### 3D Visualization

Programs (1D and 2D) enable 3D visualization on the desktop. 3D visualization is informative and serves for better orientation in the structure (for example dislocation of objects) and for results presentations.

3D visualization is set on the "Visualization" tool bar, which also contains tools for working with the visualization.





3D visualization

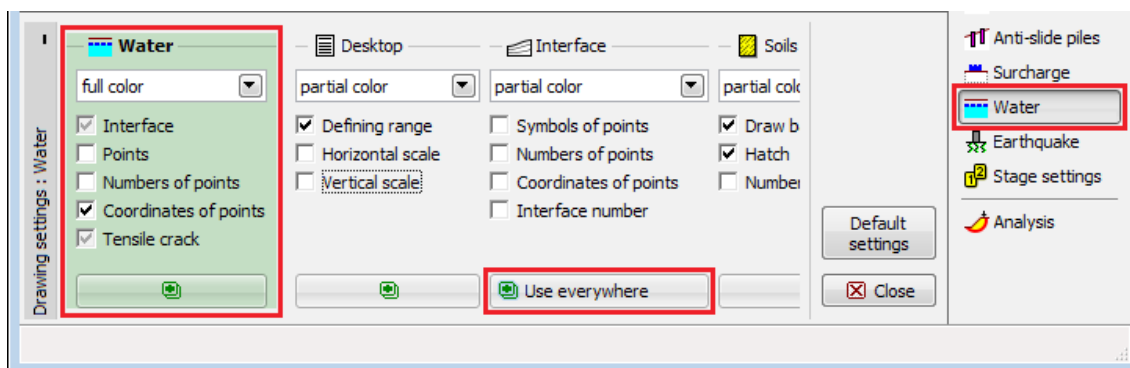
## Visualization Settings

Using the frame "**Visualization Settings**", it is possible to set the parameters of the visualization (what is showed on the desktop). In any frame, it is possible to switch to the "**Visualization Settings**" mode, just by clicking the the button on the "**Vizualization**" tool bar.



Button of visualization settings

Individual columns in the window correspond with the individual frames. On the left is always the settings for the current frame (frame "**Water**" on the figure). In other columns, visualization of ther objects is defined. It is possible to set visualization only of the objects, which are currently viewed.



Frame "Visualization Settings"

Button "**Use everywhere**" in the bottom part of each column sets the defined parameters of visualization to **all frames**.

Button "**Default settings**" sets all the parameters to default.

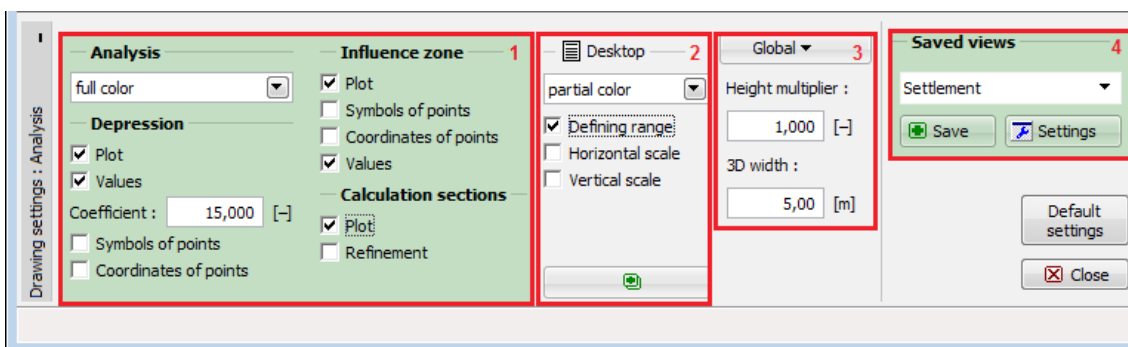
**Settings in this frame mainly serves for defining parameters of the visualization on the desktop - the settings of parameters of drawing for outputs (print) are defined in the "Add Picture" mode.**

The frame can also contain columns with special settings, which are only displayed in several cases in some programs:

- Visualization settings of **analysis results** (1).
- Desktop (2D programs) - defines several special settings (defining range, scale) and are global settings for all **frames** (2).
- Global - Height multiplier (2D programs) - enables change of scale in vertical direction (*z*) (3).
- Global - Stretch Drawing (1D programs) - enables change of scale in horizontal direction (*x*) (3).
- Global - 3D width - defines the width of the visualized structure in **3D visualization** mode (3).
- **Store views** (programs Settlement, FEM) (4).

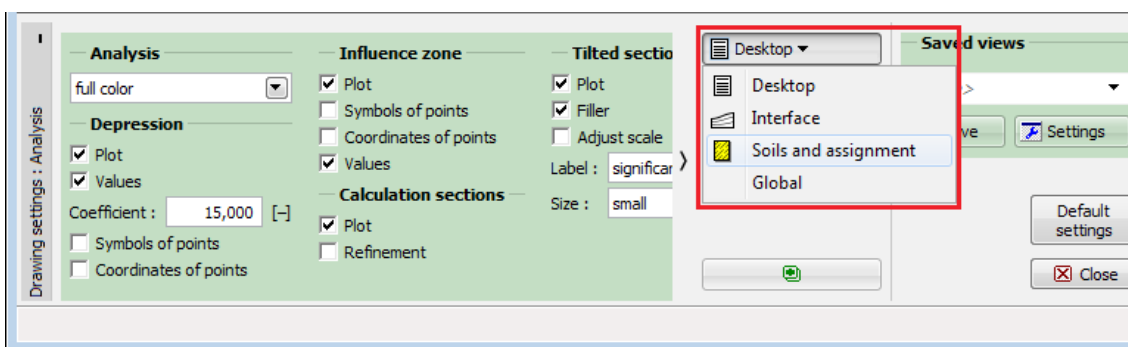


Settings of background colors, styles of lines and fills is defined for all programs in the "Drawing Styles Administrator".



Frame "Visualization Settings" - special settings

First column in the frame is sometimes divided into several sections (eg. visualization results settings). In this case, the columns are merged into one in other frames. It is possible to select the frame, of which the visualization is to be set using the button in the header of the column.



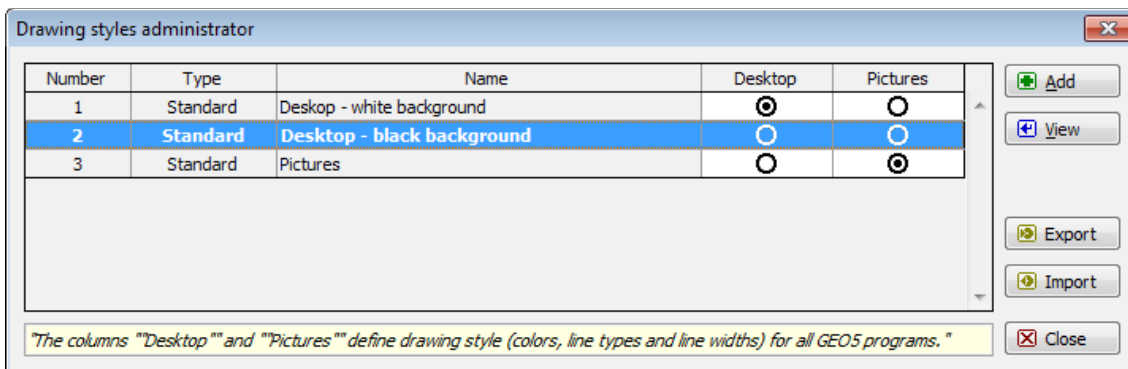
Frame "Visualization Settings" - merge multiple sections (columns)

## Drawing Styles Administrator

**Drawing styles administrator** enables to globally set the picture styles (colors, line styles, filling styles, selections colors). The dialog window is available from the menu (items "Settings", "Visualization style"). The user can define the style of visualization for pictures on the desktop and in the outputs for all GEO5 programs. The current program version only enables to select from basic pre-defined styles (white background, black background).

Possibility of creation of own visualization styles (define the colors, styles etc.) will be implemented in the next program version.

The style of the structure visualization is defined in the frame "Visualization Settings".



Dialog window "Drawing styles administrator"

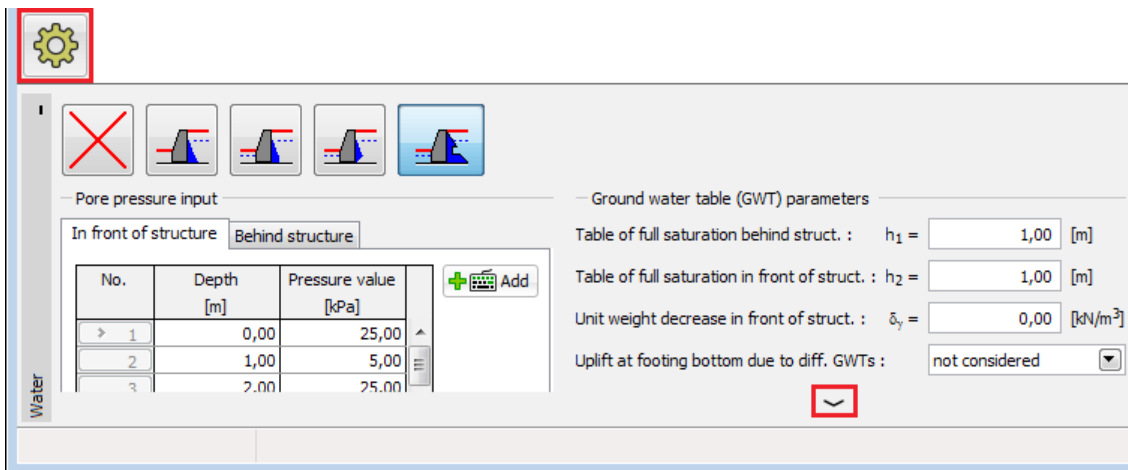
## Frames

Frame is a permanently opened window in the bottom part of the application window. Frames are changed depending on the selected input data mode of a given task selected from the control bar "Frames" and using the button on the control bar "Visualization Settings". Frame may contain the following items: table, combo list, fields for data input ( $h_1, h_2, \dots$ ) and command buttons.

When selecting data using keyboard, use the "Tab" function key together with cursor arrows for moving within the selected element (for example combo list). Selection of the checkboxes is done by the space button. When selecting using the mouse, we use the left mouse button.

The buttons that open dialog windows ("Add", "Add in dialog" and "Add graphically") can only be controlled by the left mouse button.

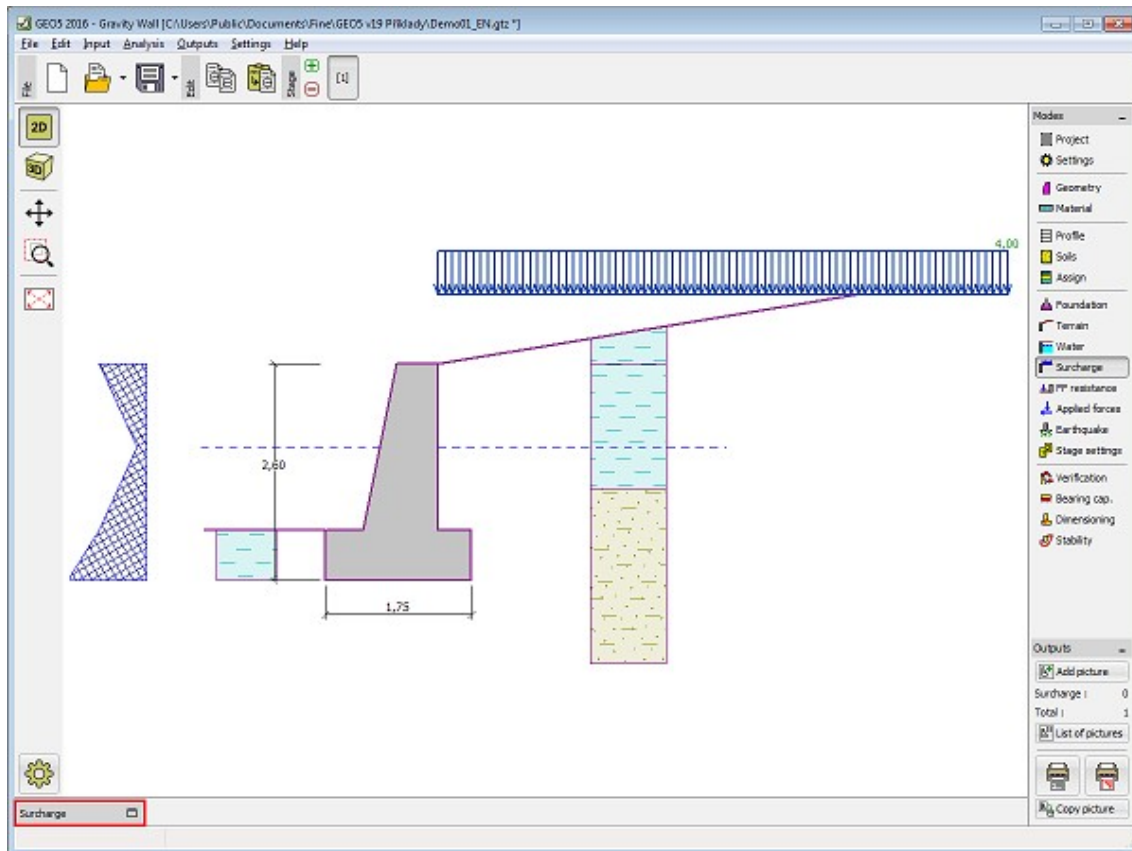
If the frame window is not enough wide (or high) so all elements could be visualized, it is possible to move the frame in vertical or horizontal direction using the arrows.



Frame control elements

Frame can be minimized using the button in the upper left corner. In this case the frame space is taken by the drawing space. The height of the frame can also be changed by clicking the left mouse button on the upper frame edge and dragging upwards or downwards. In some cases it is more advantageous to exploit the frame space for increasing the drawing space area, which is possible owing to the fact that the program uses the system of active dimensions and active objects, which means frames doesn't need to be displayed all the time.

To maximise the frame back, pres the button in the left bottom corner of the desktop showing the frame name. Providing the frame is minimized, e.g. in the mode "**Water**" it remains hidden even when switching to other input data modes.



Frame control elements

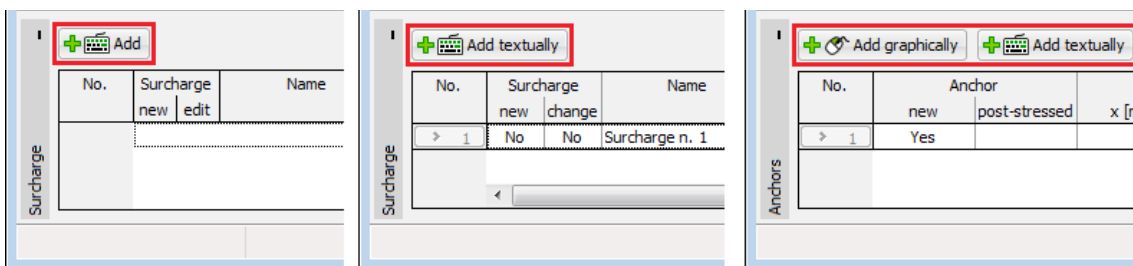
## Tables

Table is a list of input data (for example a list of surcharges, soils, profile interfac etc.). The table header contains a list of items (surcharge, name, width, size...).

### Adding new items to the table

The form of the table depends on the selected **frame**. Graphical bonds between the items in the table and on the desktop are important. Some data (objects) can be only **input** textually, others also graphically using the mouse. If the program enables both, textual and graphical input, both buttons are shown in the table.

If the table **does not contain any items** yet, or **no item is selected in the table**, only buttons "**Add**", "**Add textually**" or "**Add graphically**" are visible above the table. Using these buttons, new items are added into the table.



Adding a new item into the table

**Table item editing**

If a row is selected (highlighted green), buttons **"Edit"** and **"Remove"** are activated above the table. The **number of the item** is given in the brackets. Using these buttons, individual rows can be edited.

Selection of items is made by pressing the **left mouse button**. Clicking the right mouse button opens the **context menu**.

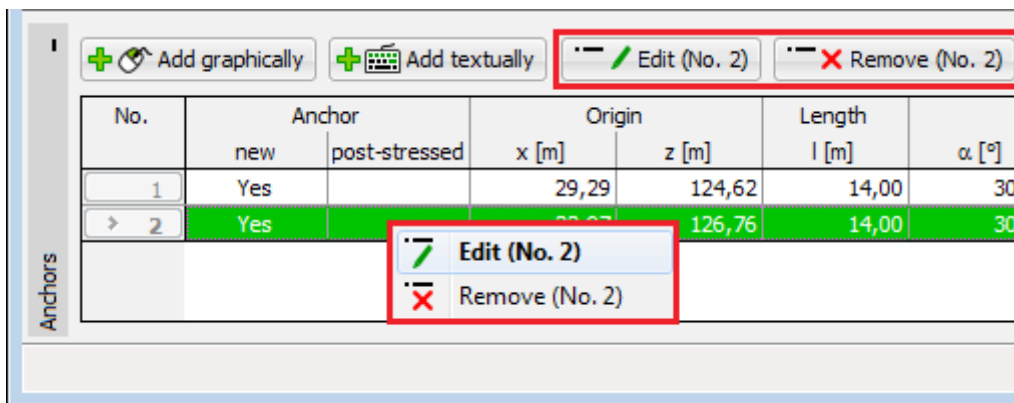
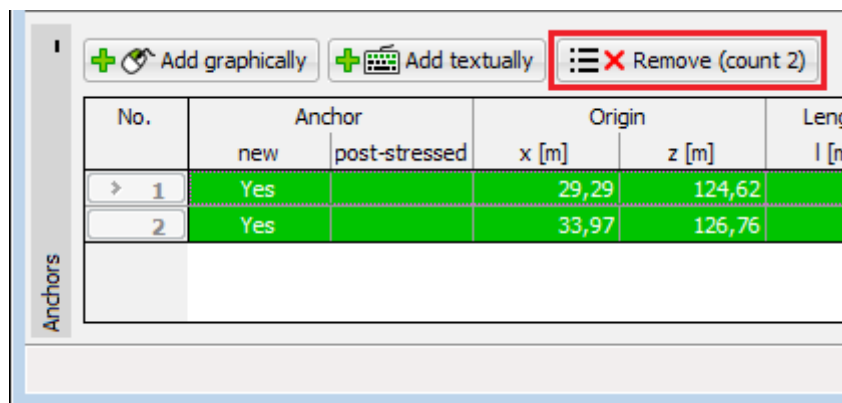


Table item editing

If the operation **enables** it (eg. **removal of an item** from table), it is possible to edit more items (rows) at a time. There are two selected rows on the figure below. From the picture, we can see that the only option in this case is removal of both items from the table, and therefore, the **"Edit"** button is not available.

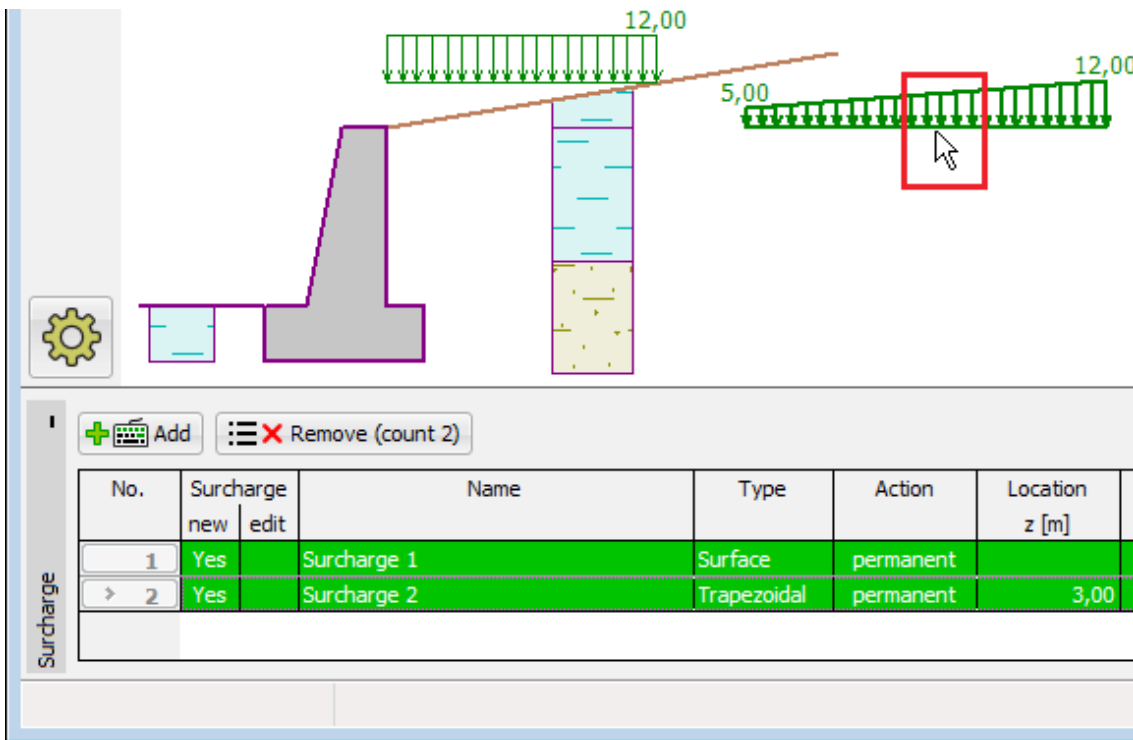
**Selection of more items (rows) in a table is done by holding CTRL and clicking the left mouse button.** By holding **SHIFT** and clicking the left mouse button, all rows above or below the selected rows are selected.

By clicking outside of the selected table rows, the selection is cancelled.



*Editing more table items*

The state of selection of rows in the table corresponds with the states of visualization of the objects on the desktop (and reverse). If a row or multiple rows are selected in the table, the relevant objects on the desktop are highlighted with the same colour. If the mouse marker is over one of the objects, the objects is shown **bold**. After clicking on "**Remove**", the objects and table rows are shown **red**.



*Visualization of selected objects*

Marking objects using these colors is implicitly set. This setting, however, can be modified in the dialog window "**Drawing styles administrator**".

## Dialog Windows

A dialog window is one of the elements that allows to input data into the program. In all GEO5 programs, dialog windows apply to conventional windows management typical for the Windows environment. A left mouse button is used when selecting objects in the window or alternatively the "**Tab**" function key when using the keyboard. When moving inside an object (for example input field) use the arrow buttons and the "**ENTER**" key.

A dialog window can contain the following items: **table**, combo list, fields for inputting data (number, text) and command buttons. The "**OK**" command button confirms the selection, while the "**Cancel**" button leaves the input mode.

Providing the window contains a certain non-typical control element (or this element has some other than typical effect) its function is described in the corresponding data input regime.

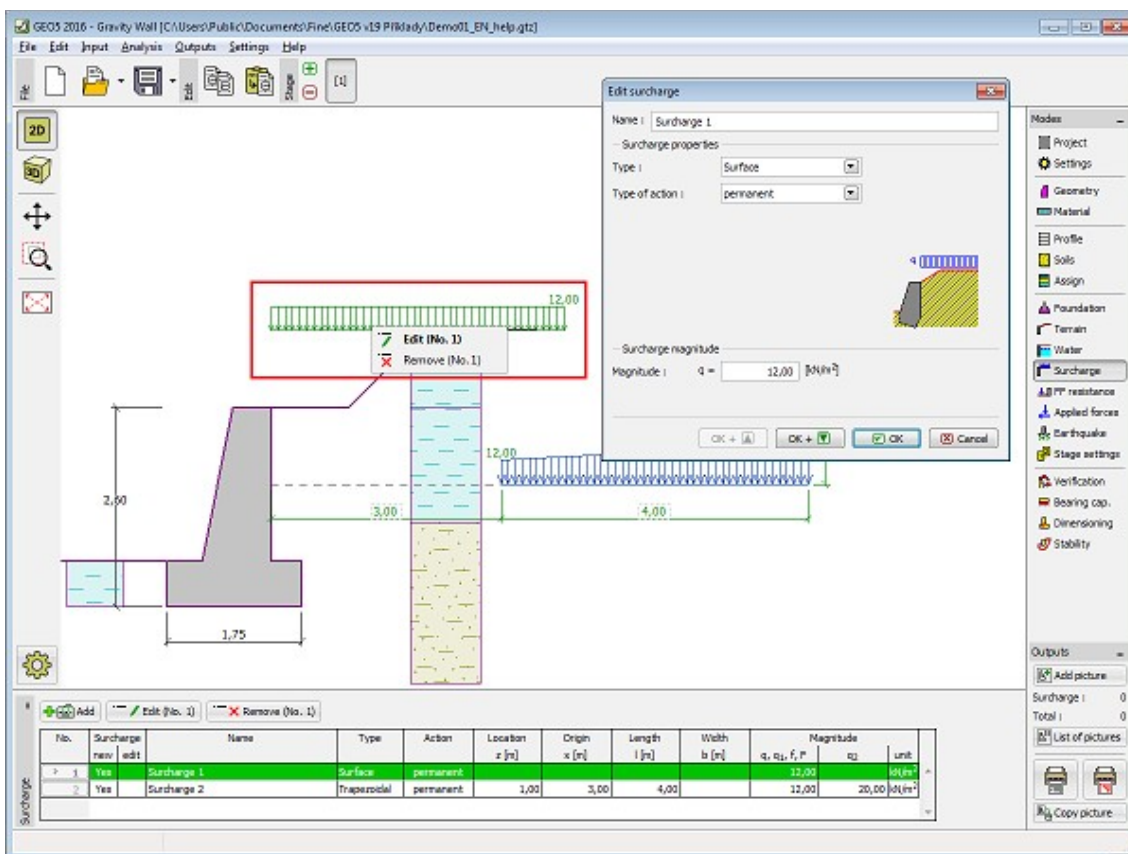
As an example consider the following picture showing the "**Edit surcharge**" dialog window that contains the "**OK+▲**" and "**OK+▼**" buttons. These buttons allow the user to move within the list of input surcharges and at the same time to confirm changes made in the window. Pressing this button results in the same action as if closing the window with the "**OK**" button and opening it again for the next element in the list.

*Dialog window example*

## Active Dimensions and Objects

The system of active dimensions and objects allows faster editing of input data.

- **Active dimension** is a dimension that can be edited directly on the desktop. The value of active dimension is labeled by a frame (dashed line). Clicking the **left mouse button** on the value then changes the frame view (is plotted by a solid line), the cursor starts to blink and the dimension can be edited. The "**Enter**" button closes the editing mode. The change is immediately displayed on the desktop.
- **Active object** functions in a similar way. Clicking the object (double click) then activates the editing mode. In this case, however, the values are not edited directly on the desktop, but rather in the dialog window originally used to create the object. The picture shows an example of an active object ( surcharge), when clicking on the desktop opens the "**Edit surcharge**" dialog window. With active objects, it is also possible to use the **context menu** option.



Example of using active dimensions and active objects

## Mouse Functions

As well as other Windows application, GEO5 programs use the mouse for controls.

### Utilization

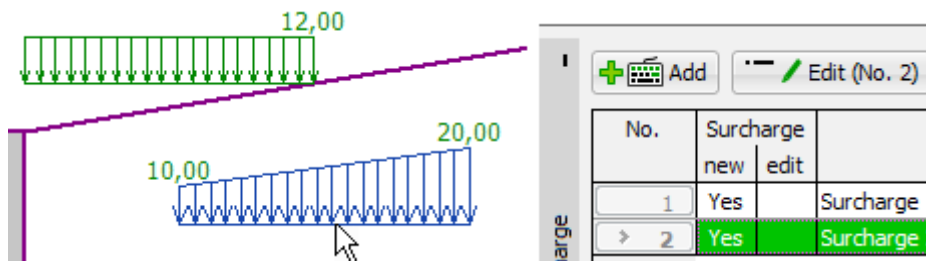
Mouse Icon	Mouse Button	Function	Description
	left	Selection	Active object or table row is selected by clicking the left mouse button.
	CTRL + left, SHIFT + left	Multiselection	Arbitrary number of active objects or table rows is added (or removed) to selection by holding CTRL and clicking the left mouse button. All rows of a table below or above the selected row are selected by holding SHIFT and clicking the left mouse

#### Selection



left

Active object or table row is selected by clicking the left mouse button.



Sample of utilization

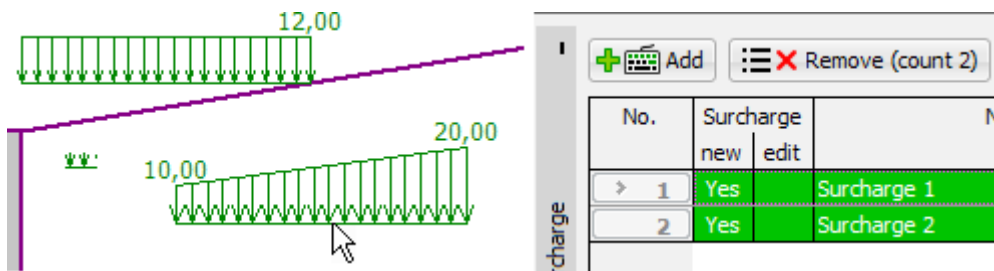
#### Multiselection



CTRL + left, SHIFT + left

Arbitrary number of active objects or table rows is added (or removed) to selection by holding CTRL and clicking the left mouse button. All rows of a table below or above the selected row are selected by holding SHIFT and clicking the left mouse

button.



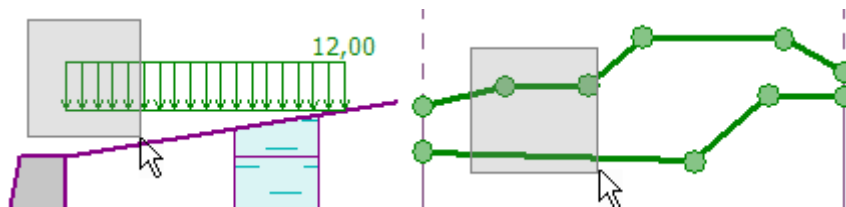
Sample of utilization

**Graphical Selection**



left

By holding the left mouse button and dragging the mouse, a highlighted area is created. Active objects in the area (even partially selected) are added to the selection.



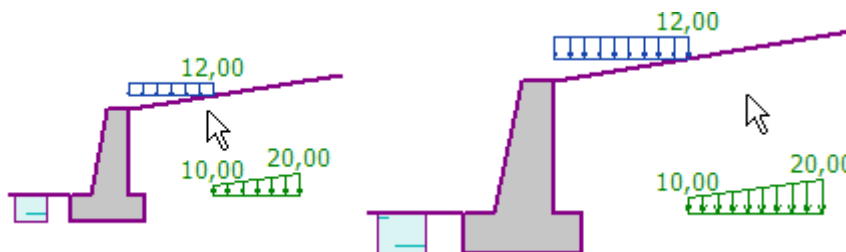
Sample of utilization

**Scale Change**



Middle (mouse wheel)

Rotating the mouse wheel zooms in or out leaving the center of the frame.



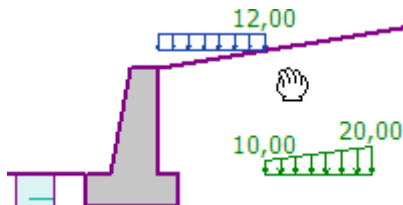
Sample of utilization

**Scale Change**



Middle (mouse wheel)

Double clicking the mouse wheel scales the visualization to the maximum zoom, while all objects are still visible on the desktop.



Sample of utilization

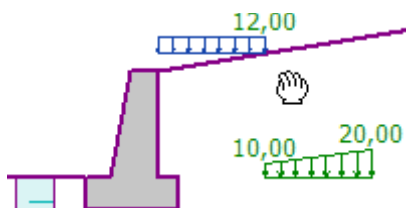
**Shift**



Middle (mouse wheel)

By pressing and holding the middle mouse button (wheel), and moving the mouse, we can shift the view in any direction.

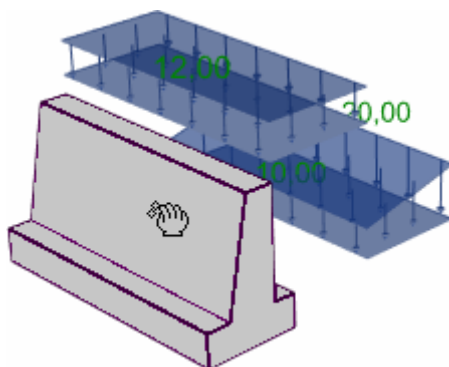




Sample of utilization



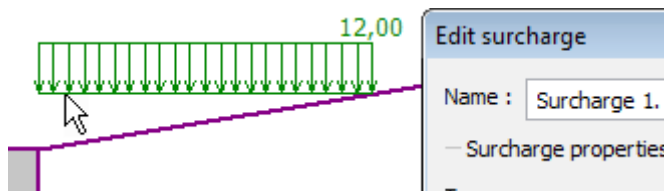
Middle (mouse wheel) Holding **CTRL** button and pressing the middle mouse button (wheel) will rotate **3D visualization scene**.



Sample of utilization



left Double clicking the left mouse button on an **active object** or **table row** opens the relevant dialog window.



Sample of utilization



right Clicking the right mouse button on an **active object** or **table row** opens the **context menu**.



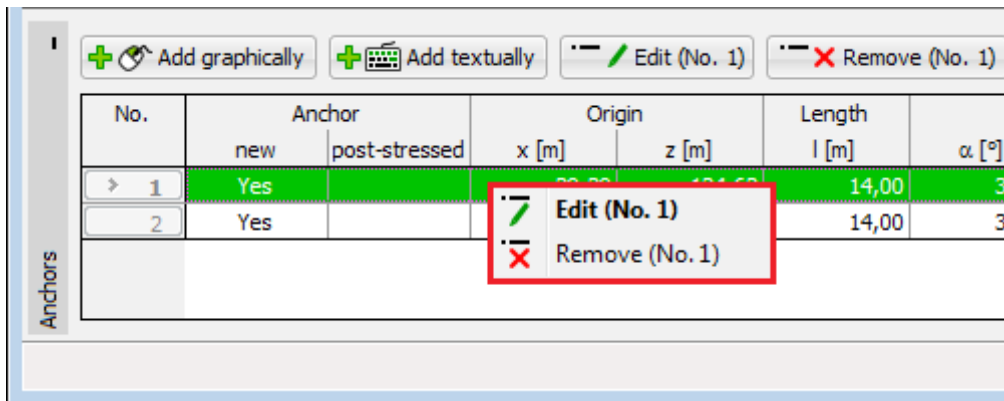
Sample of utilization

## Mouse Context Menu

Programs have an implemented context menu of the **right mouse button**. Context Menu opens by clicking the right mouse button on an **object** or **table row**.

**Context menu when editing tables**

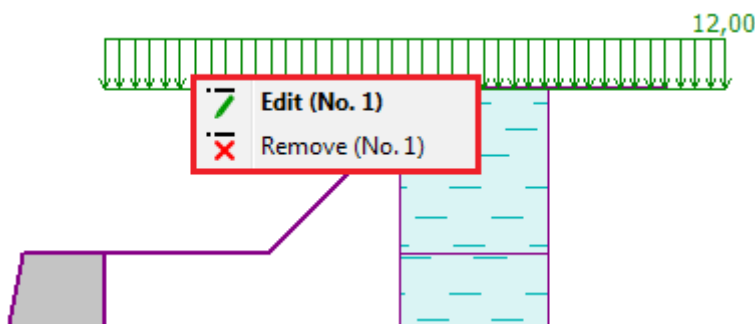
If a row in a table is selected by the right mouse button, the context menu will appear. The number in the brackets next to individual items shows the number of the edited object. The required item in the context menu can be selected by either left or right mouse button.



*Context menu when editing tables*

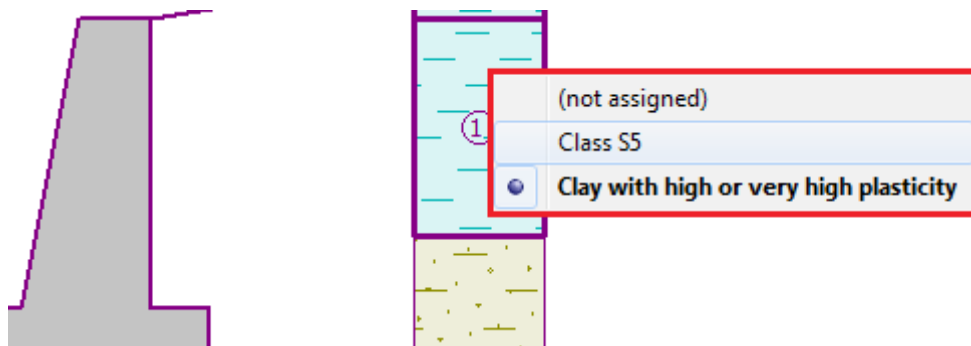
**Context menu when editing objects**

If an object is selected by the right mouse button, the context menu will appear. The number in the brackets next to individual items shows the number of the edited object (corresponds with the number of the table item). The required item in the context menu can be selected by either left or right mouse button.



*Context menu when editing objects*

Context menu can be also used for assignment of soils.

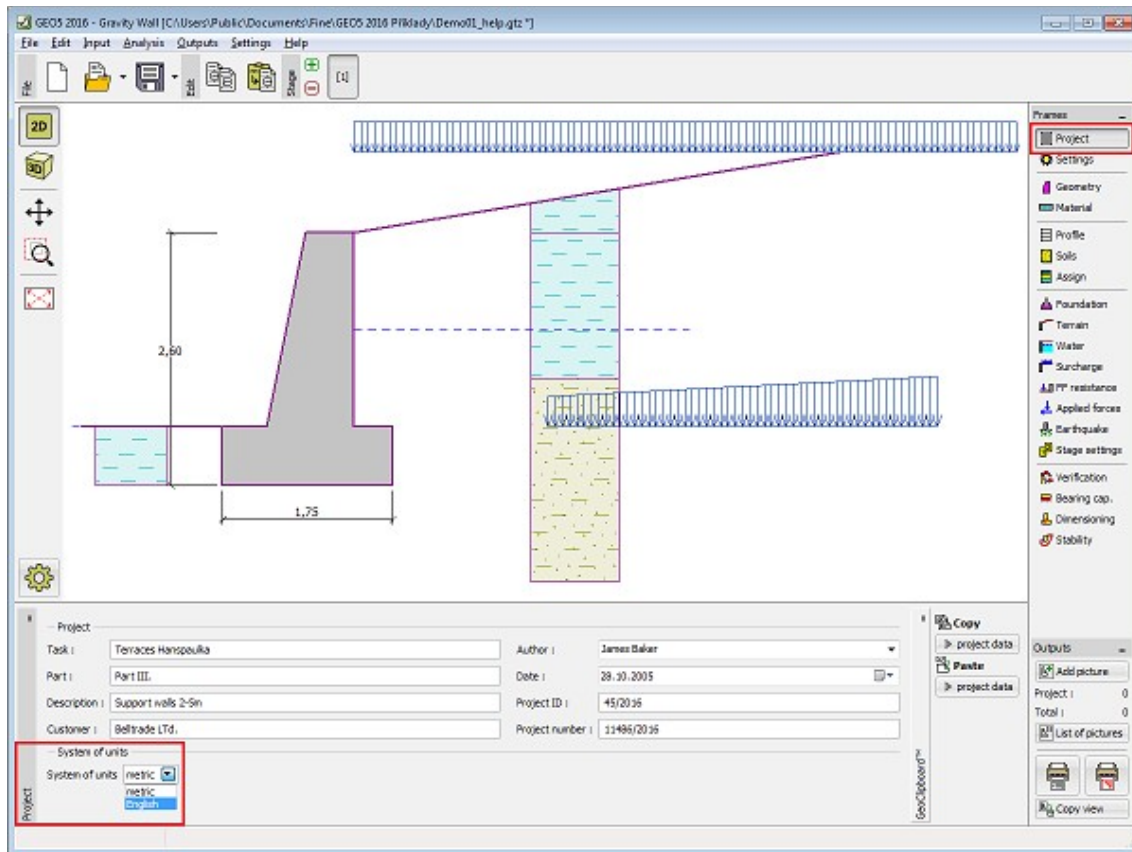


*Context menu - assignment of soils*

Alternatively, it is possible to use active dimensions and objects.

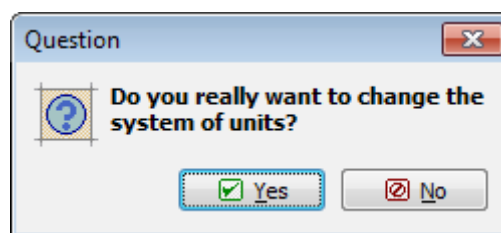
## Units - Metric / Imperial

The program allows you to select either **metric or imperial** units in the frame "Project".



*Change of units*

Use combo list to select the desired type of units. A prompt message appears requesting to confirm the selection.



*Dialog window to confirm the change of units*

## Copy to Clipboard

The program allows using the Windows clipboard in two different ways:

- It is possible to copy the current desktop view. The picture can be then inserted into an arbitrary editor (MS Word, Paintbrush, Adobe Photoshop, etc.). Copying the pictures to clipboard can either be done using the **control menu** (items "**Outputs**", "**Copy picture**") or by the button on the control bar "**Outputs**". The settings of parameters is defined in the dialog window "**Options**", tab "**Copy to clipboard**".

- It is possible to copy the program input data (soil parameters, profile and interfaces, surcharges, water impact, terrain, etc.). The copied data can be then inserted into another GEO5 program. Copy to clipboard can be either done using the control menu (items "**Edit**", "**Copy data**", "**Paste data**") or using the button on the tool bar "**Edit**".

It is also possible to use a special **GeoClipboard™** of the GEO5 software, which enables data transfer between input modes or construction stages of one or more programs.

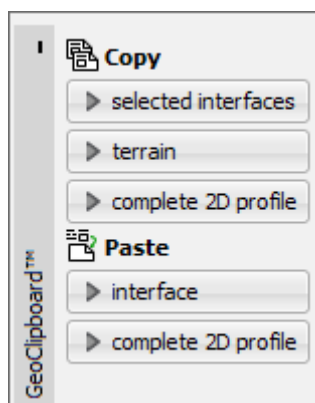
## GeoClipboard™

**GeoClipboard** is a special clipboard used in GEO5 software. It allows to copy and paste data between modes and stages of the same or another program.

Basic characteristics of the **GeoClipboard** are:

- GeoClipboard can simultaneously contain different data, e.g. copying soils after interfaces doesn't remove interface data
- data are saved after program exit and computer restart till they are replaced by other data from same data category
- every computer user has its own GeoClipboard
- during pasting data form GeoClipboard the preview of changes is always shown and the user can change pasting parameters

**GeoClipboard** controls are always placed into the relevant frame. It looks like this:



*GeoClipboard*

**GeoClipboard** is implemented for the following data:

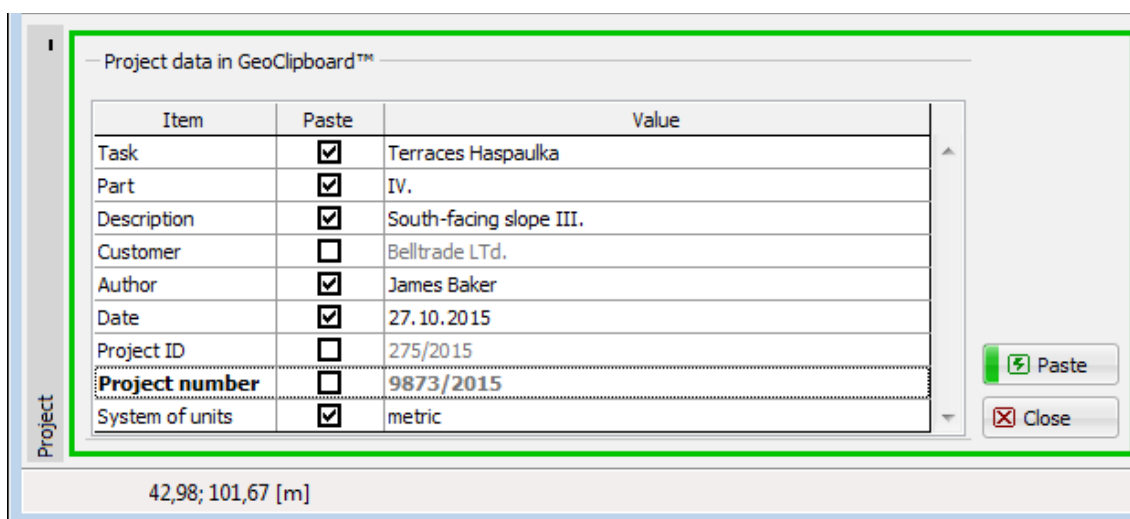
- **project data**
- **2D interfaces**, including copying the analysed GWT and topology of stages in FEM
- **soils and rigid bodies**
- **2D assignment**

## Copying and Pasting Project Data

GeoClipboard allows to copy and paste project data.

While copying all entered non-empty data is copied.

Pasting of project data is done in the following window:



*Pasting of project data from GeoClipboard*

In this window it can be specified which project data is pasted ("Paste" column). Pasting is done and data is changed when the "Paste" button is pressed.

## Copying and Pasting 2D Interfaces

GeoClipboard allows to copy and paste 2D interfaces between the following modes:

- Interface
- Embankment and Earth cut
- Water
- Incompressible Subsoil

It also allows to copy data from FEM program:

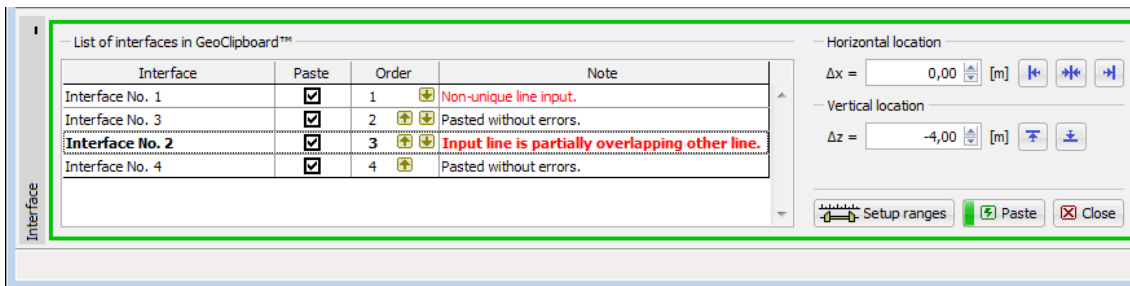
- in "**Analysis**" regime it allows to copy analyzed GWT, especially after water flow analysis
- in "**Activity**" regime it allows to copy interfaces with respect to active and inactive areas

It is possible to copy the following items:

- current interface
- selected interfaces
- terrain of current stage
- complete 2D profile

### Pasting of interfaces

Pasting interfaces from GeoClipboard is the same process as if the user enters interfaces step by step. The pasting parameters can be entered in the following window:



*Pasting of 2D interfaces from GeoClipboard*

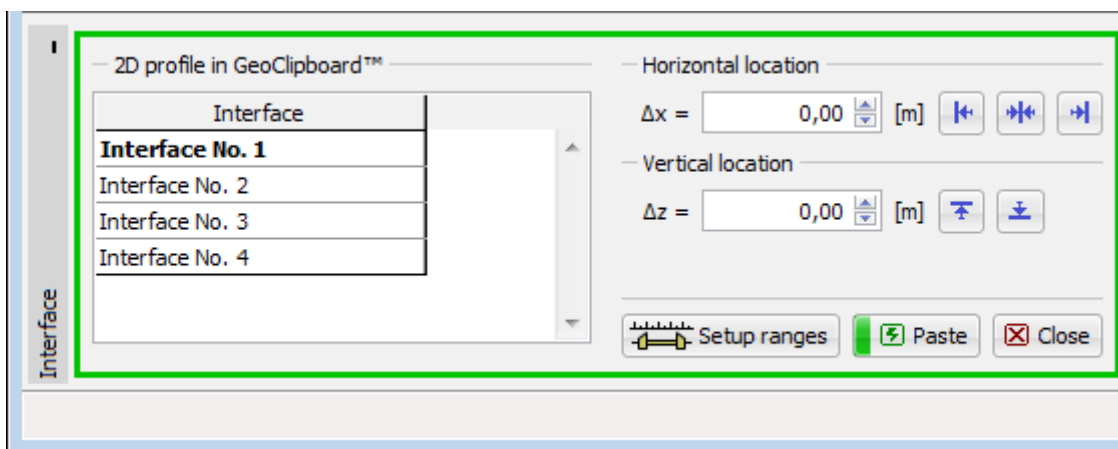
In this window it can be specified, which interfaces are pasted ("**Paste**" column), the order of pasting (change by mouse click on or by keys Ctrl+Shift+up or down arrow) and locate it to the desired place. In the same mode there is a possibility to paste only standalone interface, thereafter the "**Paste**" column acts like a radio button and the "**Order**" column is hidden.

Program always shows preview of changes caused by pasting and the result of pasting is described in "**Note**" column. Data are changed when "**Paste**" button is pressed.

There is a possibility to repeatedly paste one interface with changed location to enter skew-layered profile.

**Pasting of complete 2D profile**

This mode allows pasting of complete 2D profile from GeoClipboard to the current 2D profile. The interfaces to be pasted cannot be specified, only locating to the desired place can be done. The program shows the preview of the resulting 2D profile. The profile is pasted and the data is changed when the "**Paste**" button is pressed.



*Pasting of complete 2D profile from GeoClipboard*

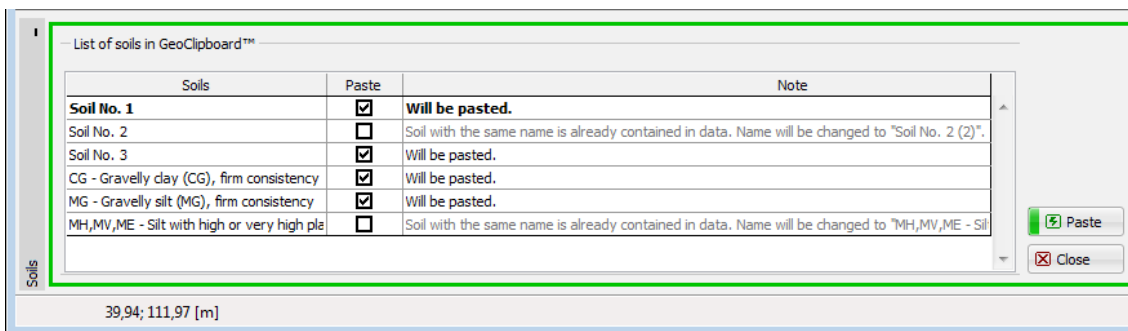
**Copying and Pasting Soils and Rigid Bodies**

GeoClipboard allows to copy and paste soils and rigid bodies.

It is possible to copy the following items:

- current soil (rigid bodies)
- selected soils (rigid bodies)
- all soils (rigid bodies)

Pasting of soils (rigid bodies) is done in the following window:



Pasting of soils from GeoClipboard

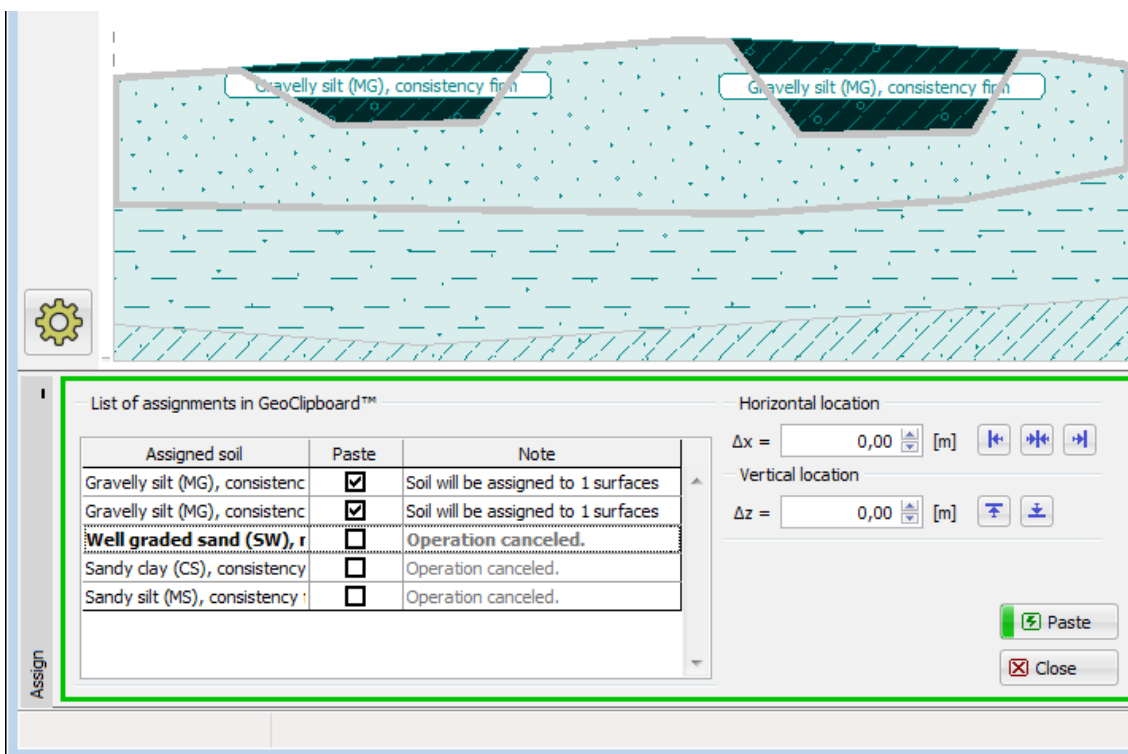
In this window it can be specified which soils (rigid bodies) are pasted ("Paste" column). If the name of the pasted soil (rigid body) is identical to the name of the soil (rigid body) already contained in the data, the program changes the name to be unique. Pasting is done and the data is changed when the "Paste" button is pressed.

## Copying and Pasting 2D Assignment

GeoClipboard allows to copy and paste assignments.

While copying the surfaces as polygons with assigned soils are copied to the GeoClipboard.

Pasting of assignments is done in the following window:



Pasting of assignments from GeoClipboard

In this window it can be specified which 2D assignments of soils are pasted ("Paste" column). The location of the pasted surfaces can be changed. The resulting state is available as a preview, with marked surfaces and information about pasted soil assignments. All surfaces in the data completely contained in each pasted surface are affected.

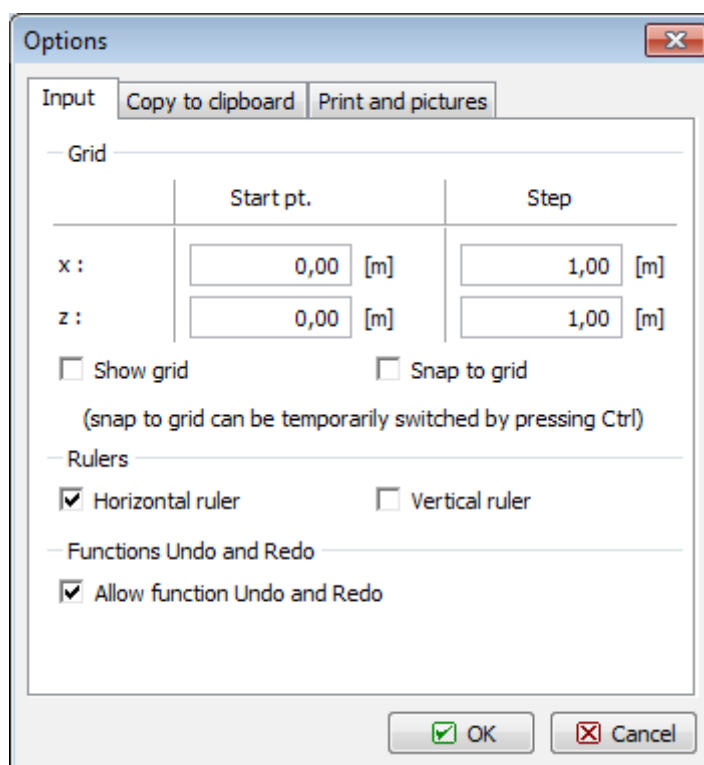
The assigned soils, which are not already found in the data, are added to the list of soils. Pasting is done and data is changed when the **"Paste"** button is pressed.

## Options

**"Options"** dialog window allows to set some of the program's special functions (**Input**, **Copy to clipboard**, **print view**, etc.).

The **"Options"** dialog window is opened from the control menu (items **"Settings"**, **"Options"**).

The window contains individual tabs (number and content may vary depending on the program), which allow to specify corresponding settings.



*Dialog window "Options"*

## Options - Input

The **"Input"** tab allows **setting** the **"Grid"** parameters and parameters of functions **"Undo"** and **"Redo"**.

This tab is implemented **only in 2D programs** (FEM, Slope stability, Settlement, Beam, etc.).

### Grid

- sets the grid origin and step in the *x* and *z* directions

### Show grid

- shows / hides grid on the desktop

### Snap to grid

- turns on / off the snap to grid option using the mouse (when shifting the mouse the cursor jumps over the defined grid - a point off the grid can be specified by holding the **"CTRL"** key)

### Horizontal rule

- shows / hides horizontal rule with a scale of distances on the



desktop

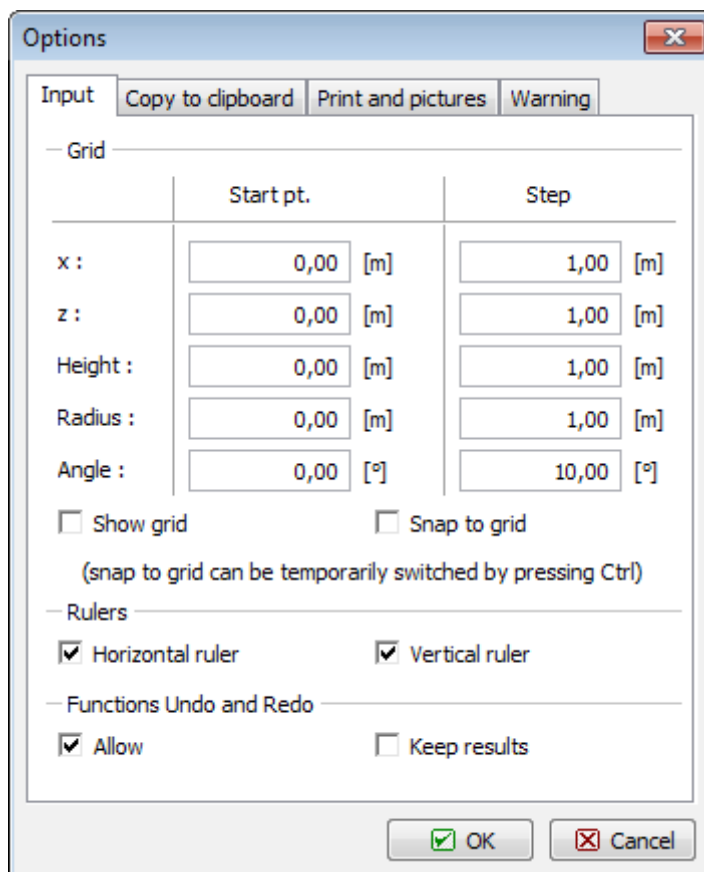
**Vertical rule**

- shows / hides vertical rule with a scale of distances on the desktop

**"Undo and Redo" functions**

- turns on / off the possibility of using these functions in the program (on **tool bar** these buttons are "foggy")

In some modules it is possible to specify if the results are stored with undo data by checking **"Keep results"**. Storing results with undo can be very time consuming. If the results are not stored, it is necessary to make the calculation again after pressing Undo.



*"Options" dialog window - tab "Input"*

## Options - Copy to Clipboard

The **"Copy to clipboard"** tab allows setting the controlling parameters:

**Picture size**

- This setting defines the picture size. Enter the picture width and the height is calculated according to the picture contents automatically.

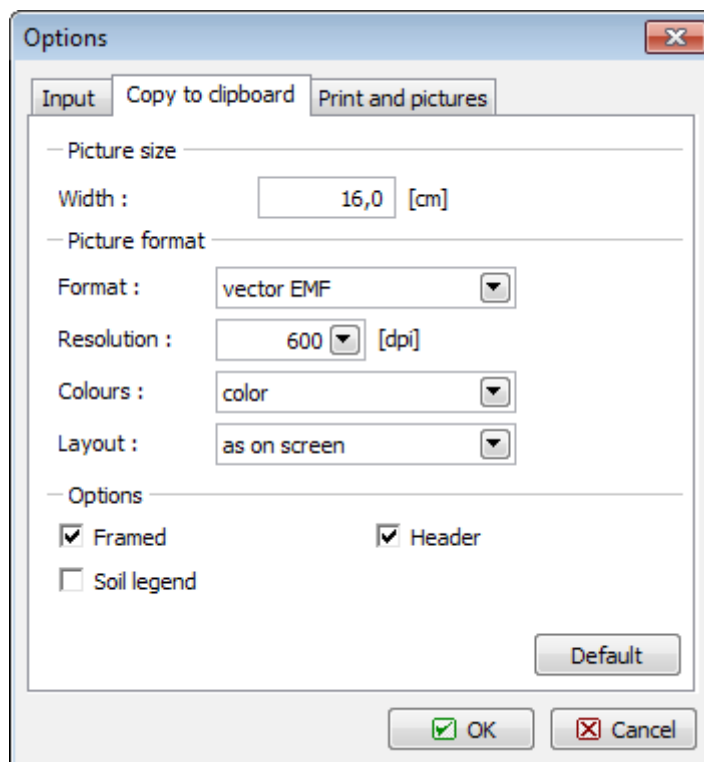
**Picture format**

- This setting defines the picture format (\*.EMF, \*.WMF, \*.BMP), its resolution, color and orientation. Recommended setting is displayed in the figure (format: \*.EMF, resolution: 600 dpi, color).

**Options**

- This setting defines the picture frame and header. If both options are checked, the picture contains both the frame and header.
- Option "Soils legend" adds legend of used soils into the picture.

The "**Default**" button in the window sets original implicit values.



*"Options" dialog window - tab "Copy to clipboard"*

## Options - Print and Pictures

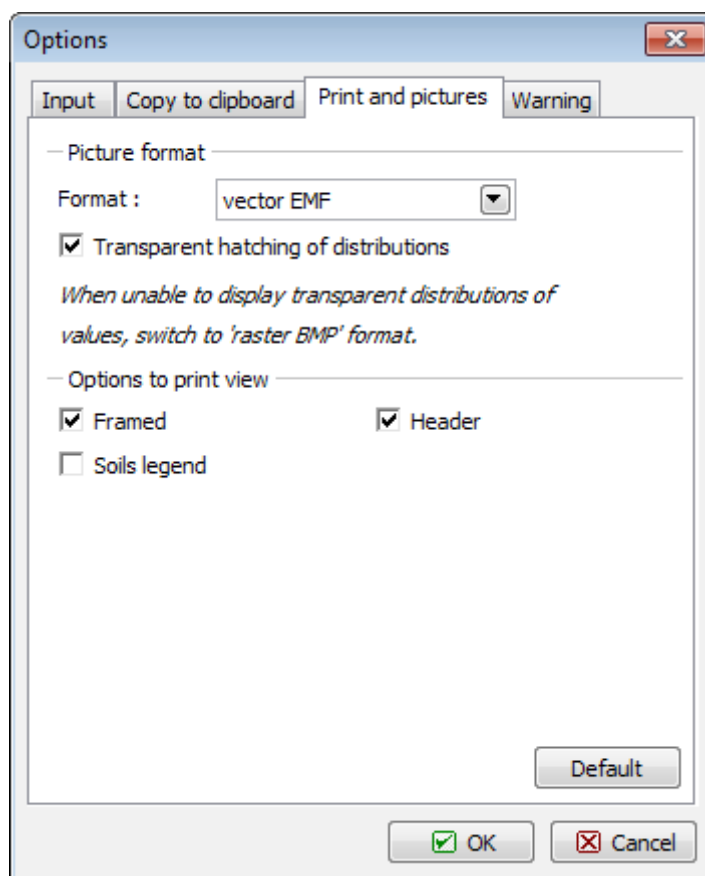
The "**Print and pictures**" tab allows setting the picture parameters assumed for "**Print and export picture**" and "**Print and export document**" dialog window.

**Picture format** This setting defines the picture format (\*.EMF, \*.WMF, \*.BMP).

**Options** This setting defines the picture frame and header. If both options are checked, the picture contains both the frame and header.

Option "**Soils legend**" adds legend of used soils into picture. This is valid only for "**Print and export picture**".

The "**Default**" button in the window sets original implicit values.



"Options" dialog window - tab "Print and pictures"

## Common Input

This chapter contains explanations common to more GEO5 programs:

- [Input and Edit of Soils](#)
- [Interface in 2D Environment](#)
- [Input of Objects and Data](#)
- [Assigning Soils](#)
- [Design Coefficient](#)
- [Running Several Analyses / Verifications](#)

Very important function in all GEO5 programs is possibility to define [Construction Stages](#).

Some GEO5 programs allow to [run another GEO5 programs](#) with automatical data transfer.

Basic functions for work with **graphical outputs** in programs FEM, Settlement are described on pages:

- [Selection and Store of Views](#)

- [Setting a Color Range](#)

Following pages describe **Data Import**:

- [DXF Import and Export](#)
- [Table Data Import](#)
- [Import LandXML](#)
- [Import gINT](#)


**Clipboard functions** are described on pages:

- [Copy to Clipboard](#)
- [Geoclipboard](#)

## Input and Edit of Soils

In the "**Add new soil**" dialog window, we input a name of a soil and parameters that should be obtained from laboratory measurements or geological survey.

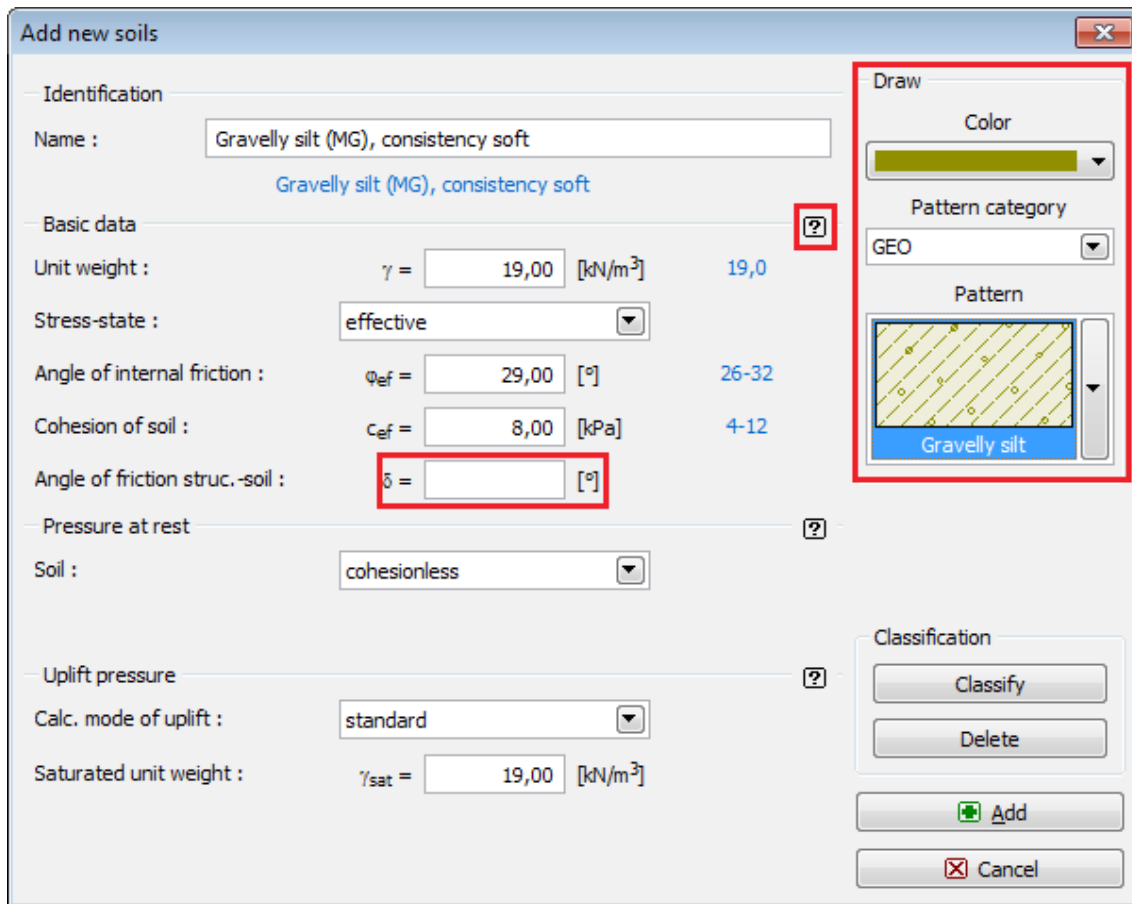
All input fields in the window are mandatory. The only exception is the value of  $\gamma_{sat}$  (unit weight of saturated soil) in the section "**Uplift**". If this field remains empty, the program automatically adds the value of  $\gamma$  (unit weight of soil).

The Hint button "" provides information about the theory of analyses linked to individual values put in.

[Color and pattern category](#) of a soil can be set in the combo lists in the right part of the dialog window.

If no geological survey or laboratory experiments are available, the soil can be specified with help of the soil database containing approximate values of basic characteristics. The "**Classify**" button opens the "[Classification of soils](#)" dialog window with values offered to insert into the window. The "**Delete**" button allows to remove information about classified soil from the catalog. Soil parameters that do not appear in the catalog ("Friction angle struc-soil" in the picture) must be assigned manually in any case. The characteristics of rocks is not listed in the built database, these parameters must be also defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help (for [solution of earth pressures](#), for [calculation of rock stability](#) or for [analysis of bearing capacity of foundation on bedrock](#)).

The specified soil is inserted into the list of soils by pressing the "**Add**" button.

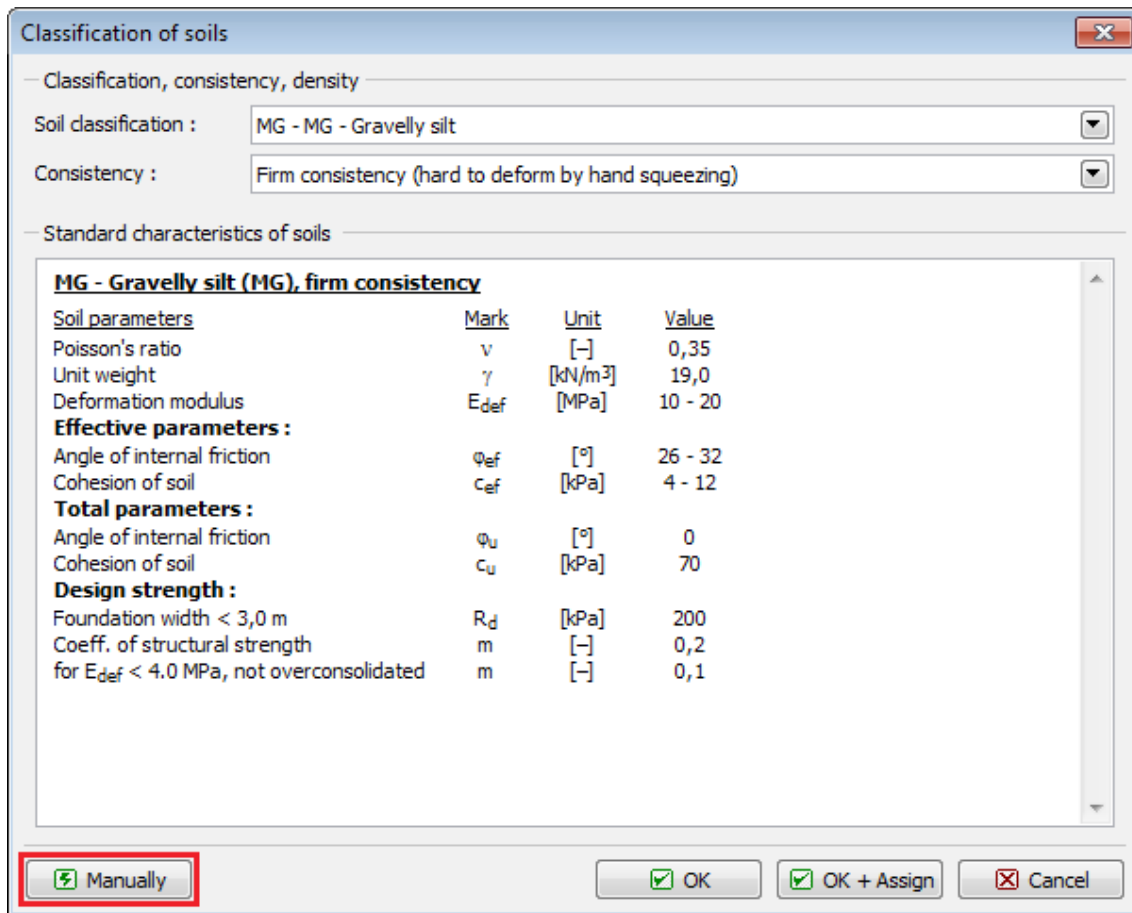


"Add new soils" dialog window

## Classification of Soils

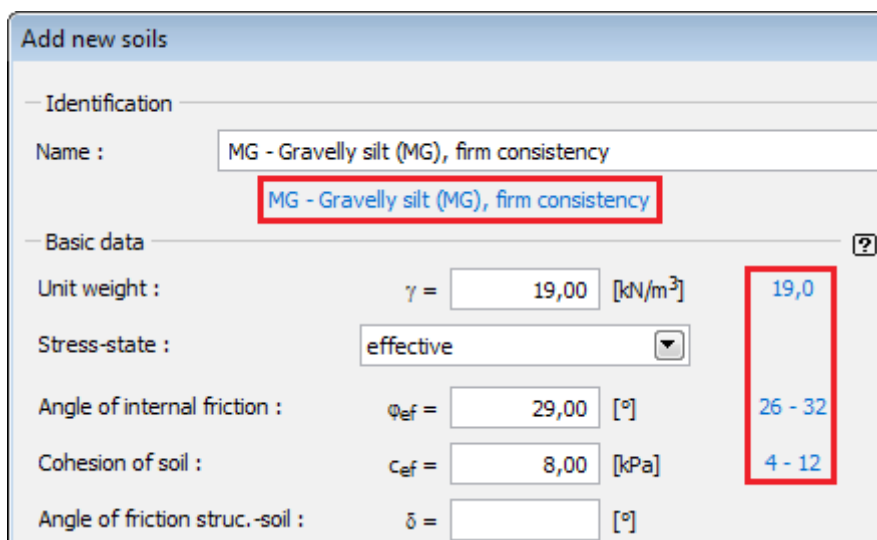
Approximate values for a specific soil can be obtained from the catalog of soils. Select the desired soil in the combo list and specify its consistency or compactness, respectively. The soil parameters obtained from the catalog appears in the window.

The "**Manually**" button opens the "**Manual classification of soils**" dialog window that allows to classify the soil if its parameters are known, e.g., from laboratory measurements (grading, moisture, compactness....).



"Classification of soils" dialog window

Pressing the "OK" button shows recommended values next to corresponding input fields (see picture) in the "Add new soils" dialog window. Pressing the "OK+Assign" button then assigns the average values of soil parameters into individual input fields. The "Cancel" button leaves the window with no action.



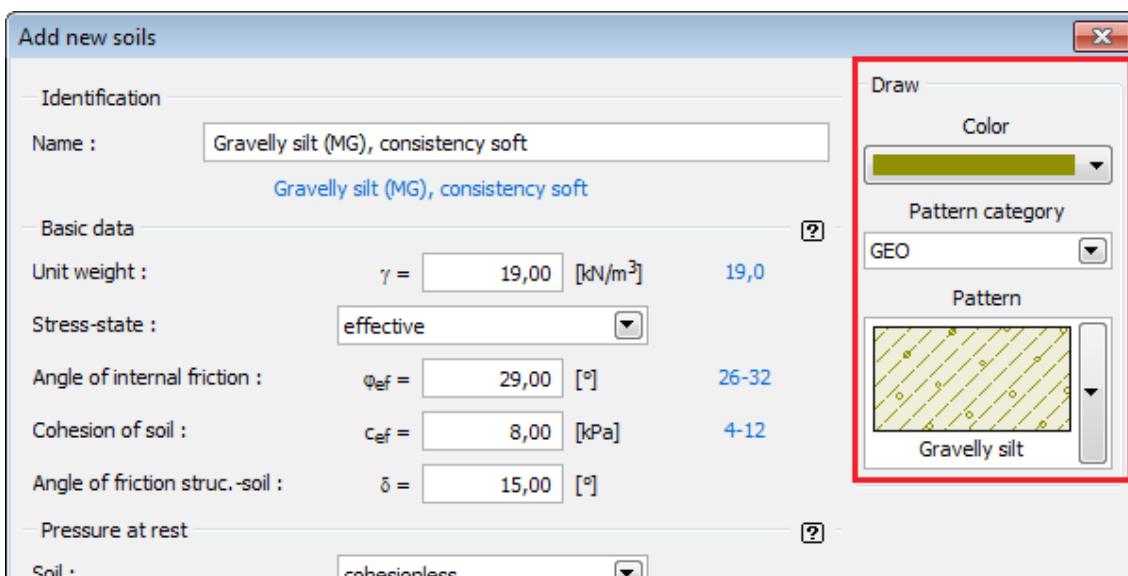
Soil classification - recommended range of values

## Soil and Rock Symbols

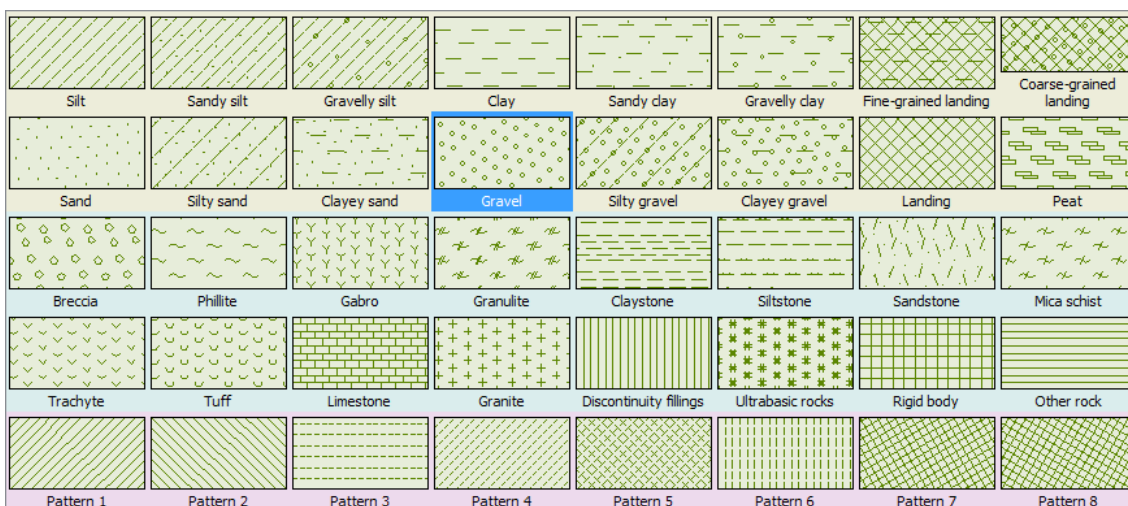
It is possible to select a pattern category from a combo list for each input soil (patterns of GEO5 software, full color, gINT patterns or patterns according to the YS 5204 Chinese standard 1) and the color of the pattern. These are displayed in the input profile

The color you choose in the combo list is used to plot soils or rocks on the desktop and to pictures, which are either stored in the "Picture list" or printed using "Print and export pictures" (to visualize the same (full) colors in pictures, the option "full color" in the "Visualization Settings" must be set).

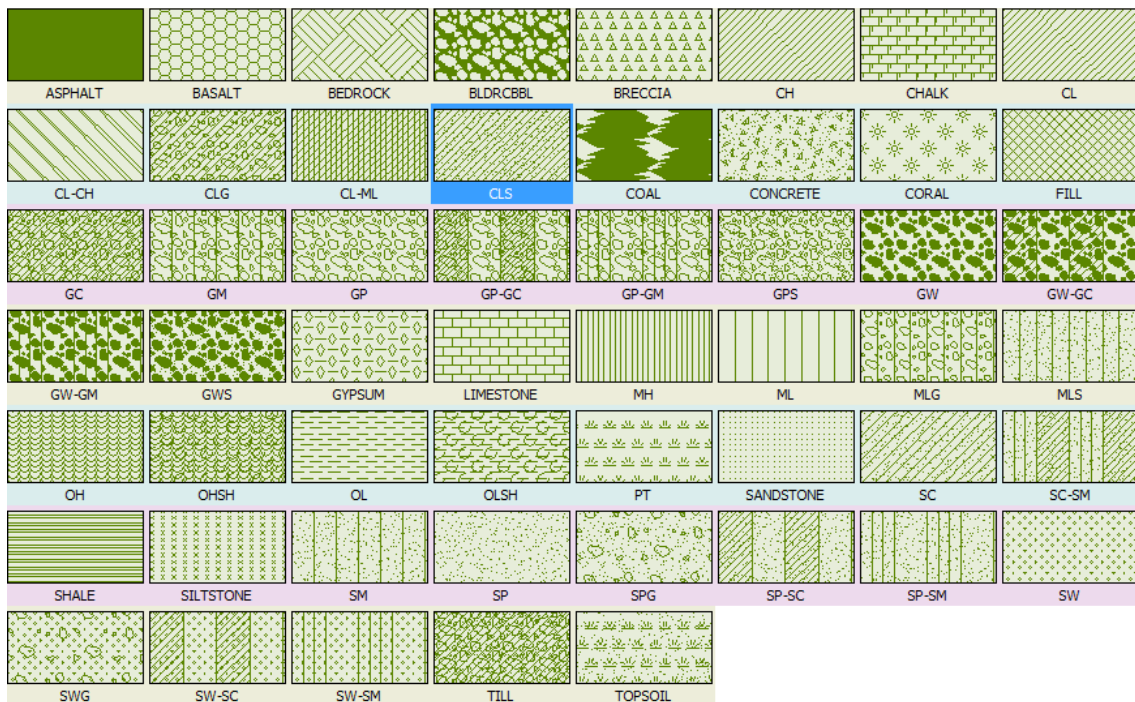
The pattern color should be chosen with respect to the desktop background or printout paper, to be sure it is sufficiently visible.



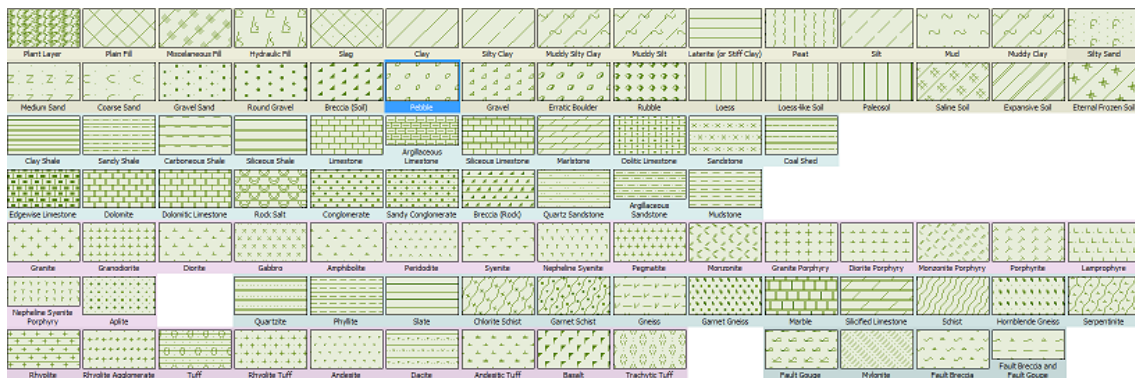
"Add new soils" dialog window - selection of color and pattern category



Patterns of soils- GEO5



Patterns of soils - gINT



Patterns of soils - YS 5204 - 2000

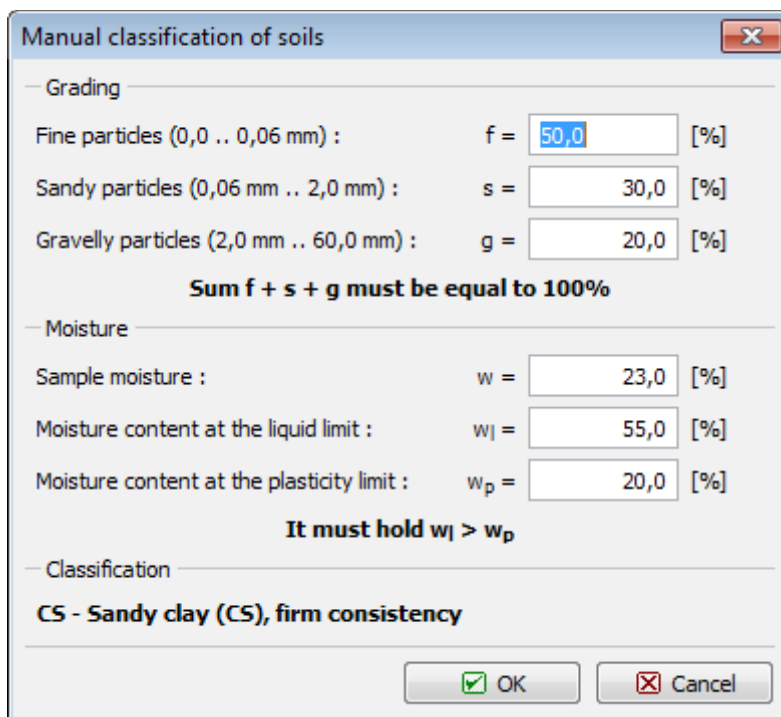
Literature:

<sup>1</sup> - YS 5204-2000 - Specification for mapping symbol of geotechnical investigation report

## Manual Classification of Soils

This dialog window allows to specify the soil parameters, which are then added into the catalogue of soils. The "OK" button returns back to the "Classification of soils" dialog window with the setting and classified soil.





"Manual classification of soils" dialog window

## Interfaces in 2D Environment

The left part of the **frame** contains a **table** with a list of interfaces. Above the table are two basic buttons, which are required for interface input:

**Setup Ranges** Opens the "World coordinates" dialog window that allows to set the world dimensions (left and right edge).

**Add interface** Turns on the mode for **inputting a new interface** - individual interfaces can be added in an arbitrary order. Each interface is automatically stored in the list of interfaces when leaving the input mode.

Interfaces are ordered in the table downwards and **it is possible to edit and delete them**.

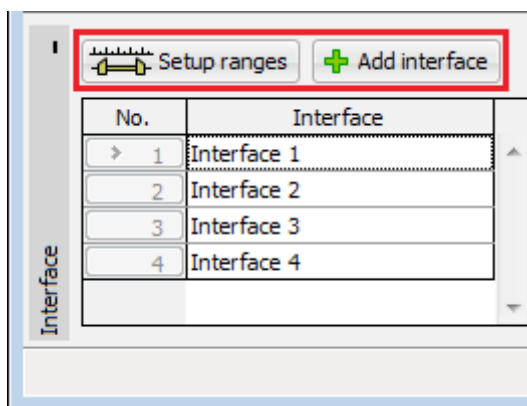
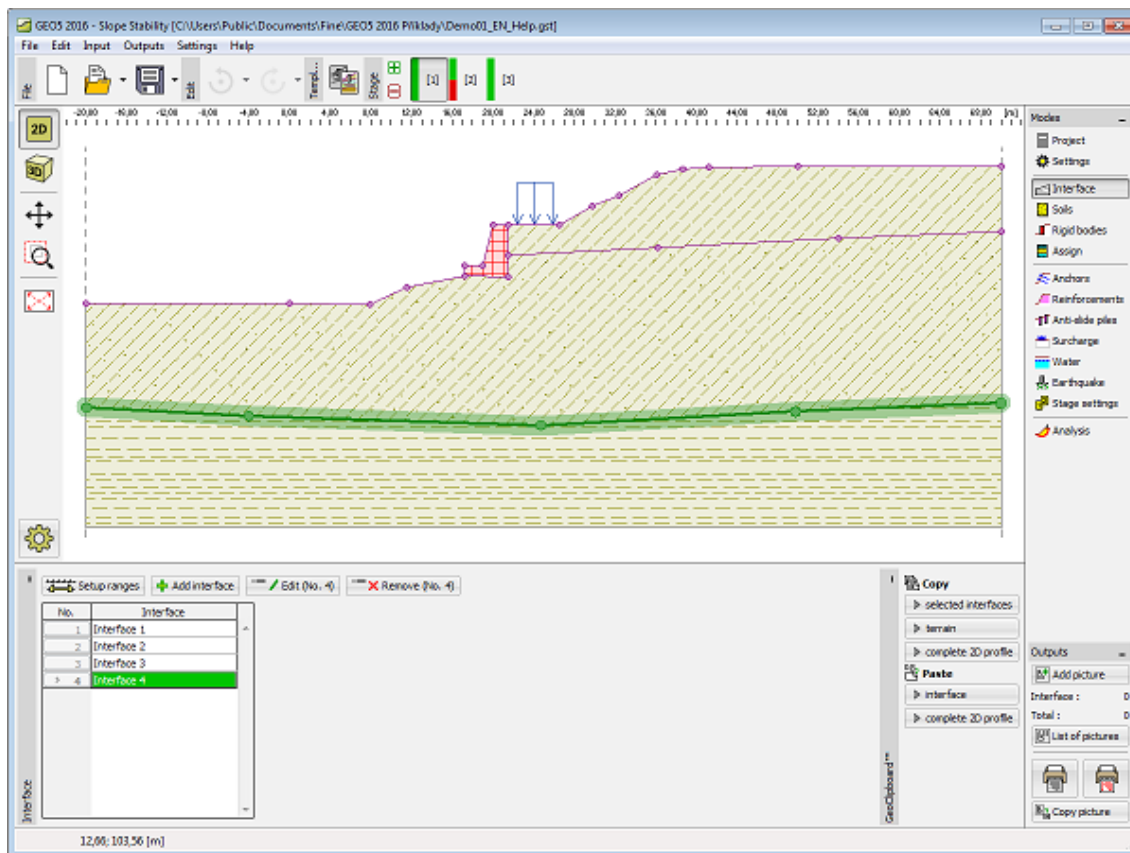


Table with the list of interfaces

Every change made to a given interface can be reverted using the "UNDO and REDO" buttons on the tool bar.

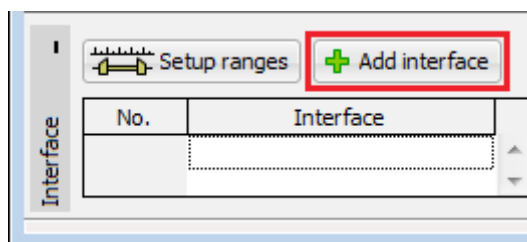
It is possible to copy and paste interfaces using **GeoClipboard**.



Frame "Interface"

## Adding Interface

The **"Add interface"** button starts the mode of inputting points of a new interface. The **mouse mask** is changed to an axes cross and the visualization of the frame changes.



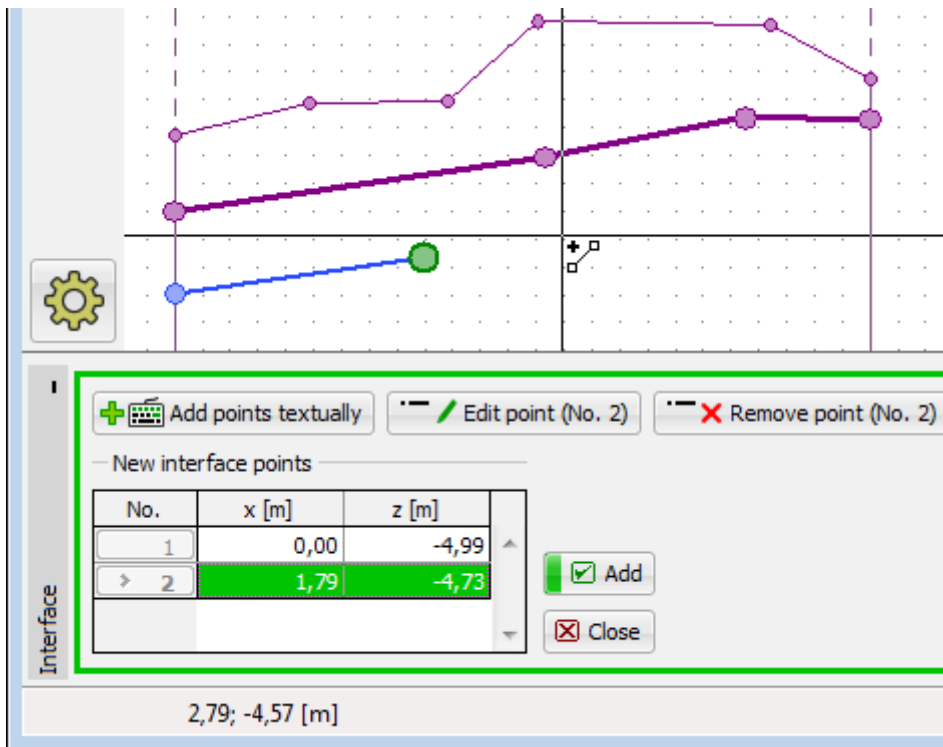
Frame "Interface"

By pressing the left mouse button on the desktop, it is possible to input points of the interface. Waipoints ( $x, z$ ) of every input point are added into the **"New interface points"** table. Alternatively, it is possible to add points in a dialog window by pressing the button **"Add point in dialog"**. Input point is always rounded to four significant figures (two decimal places) - input by mouse and keyboard is then completely equivalent.

During the input, it is possible to **edit and delete** individual points.

Input is terminated by pressing button **"Add"** (adds the input interface into the **interface list**), or by pressing button **"Close"** (input interface is discarded).

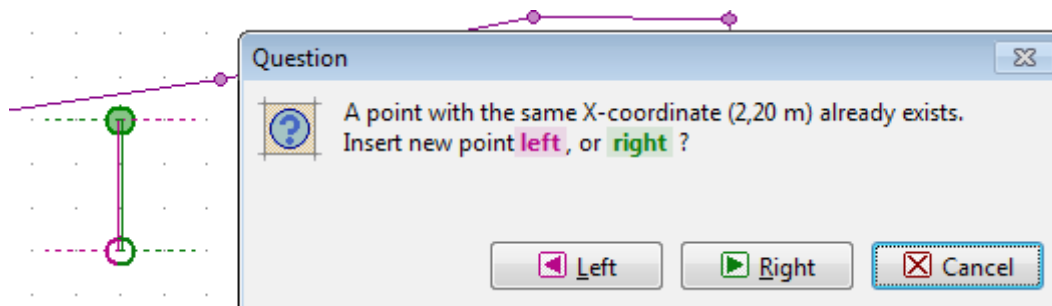
When inputting, it is possible to use the function of the **grid**.



Mode of interface points input

When inputting points, it is possible to use templates obtained from **DXF import**.

The program also allows to introduce vertical interfaces - in such a case the program requests to insert the point to the **left** or **right**. The buttons for confirming the action are colored - the same color is also used to visualize both input variants on the desktop.



Vertical interfaces

The program also contains an automatic **corrector of input interface** that determines the interface end points and then adds the interface to the list of interfaces.

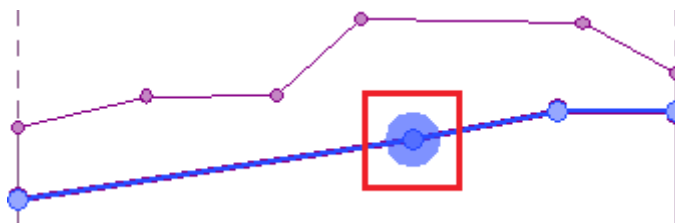
## Editing Interface Points

When inputting or editing an interface, it is possible to edit or remove individual points, graphically and textually in a dialog window.

### Editing interface point

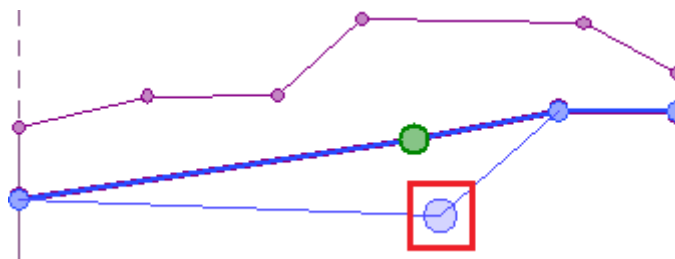
Move the mouse cursor on the point, which is to be edited. The area around the point is

highlighted.



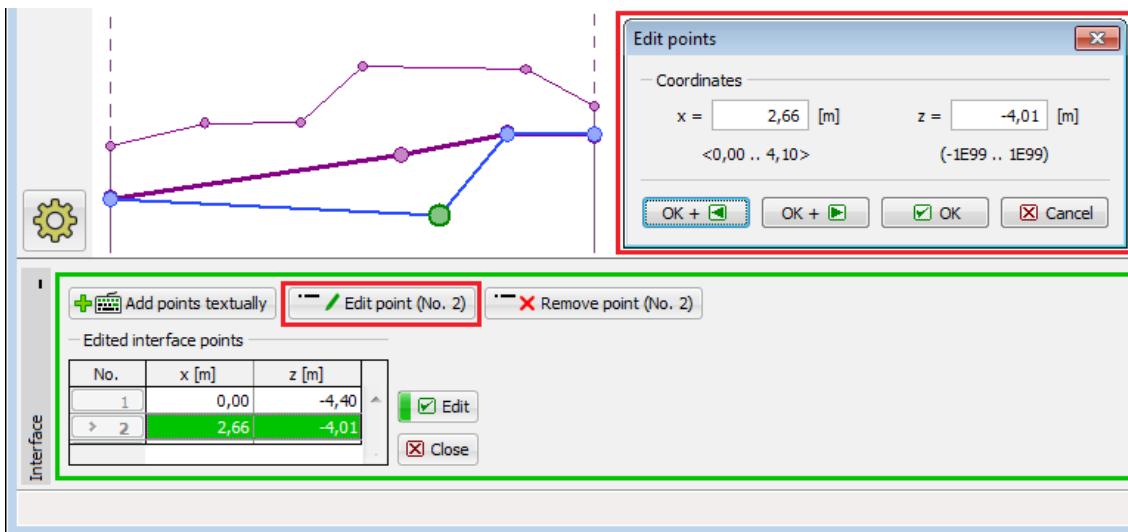
*Editing interface points*

The point is moved by holding the left mouse button and dragging. By releasing the left mouse button, the point is changed.

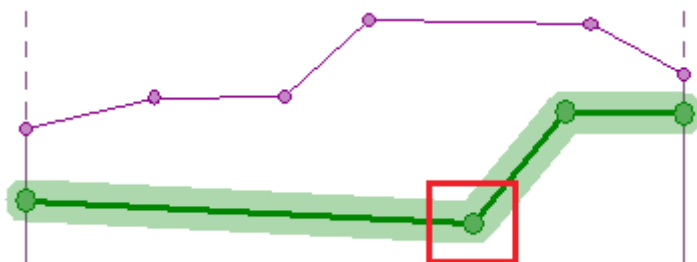


*Editing interface points*

Alternatively, it is possible to press "Edit point" in the table, and change the coordinates in the dialog window. Editing is terminated by clicking the button "OK".



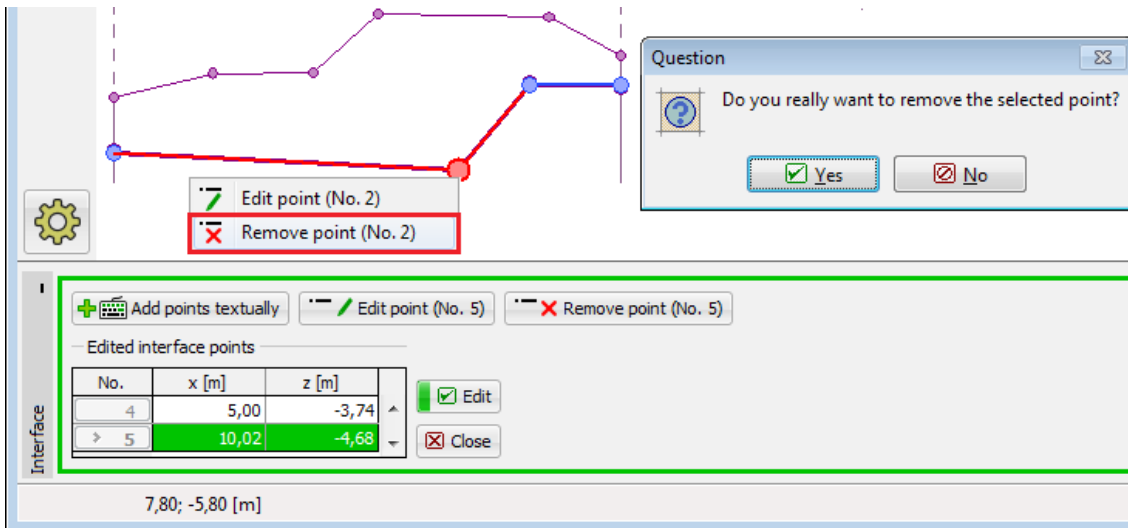
*Editing interface points*



*Editing interface points - result of coordinates change*

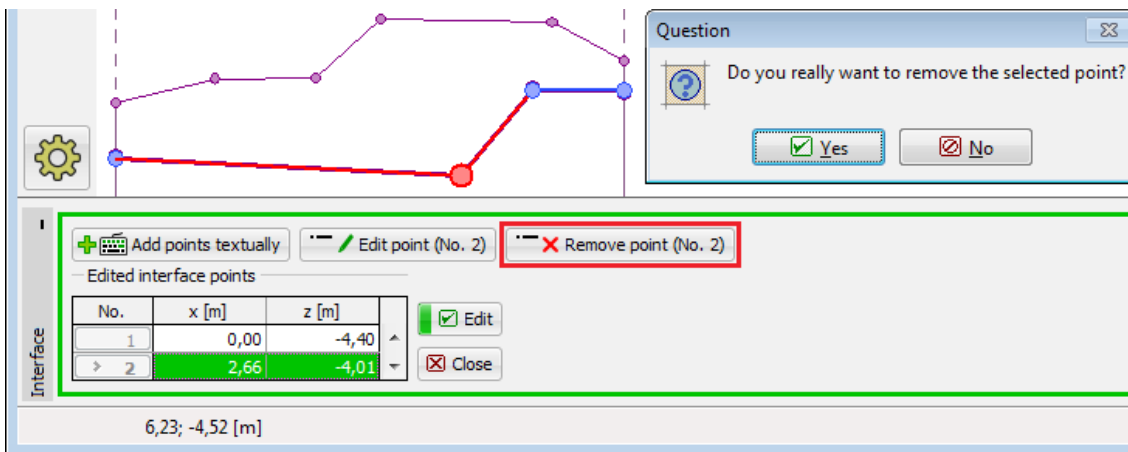
### Removing interface point

Move the mouse cursor on the point, which is to be deleted. The area around the point is highlighted. After clicking the right mouse button, the context menu appears. Select the "Remove point" item. The program highlights the point and the lines which are affected by deleting the point in red. After confirmation by the user, the program deletes the point and adjusts the interface.



Removal of interface point - graphically

Alternatively, it is possible to select the point in the table and click on "Remove point". Further process is identical to the one described earlier.

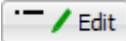


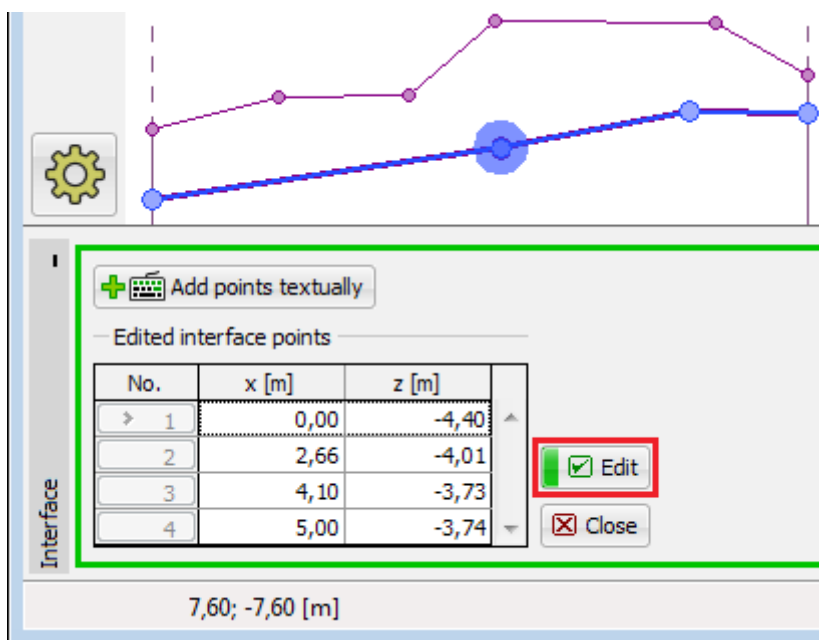
Removal of interface point - textually

## Editing and Removing Interface

### Editing Interface

An interface is selected in the table or on the desktop by pressing the left mouse button.

By double-clicking the mouse button on the interface, or by pressing the button , the interface editing mode is turned on. The interface being edited is highlighted in blue and it is possible to change its shape.

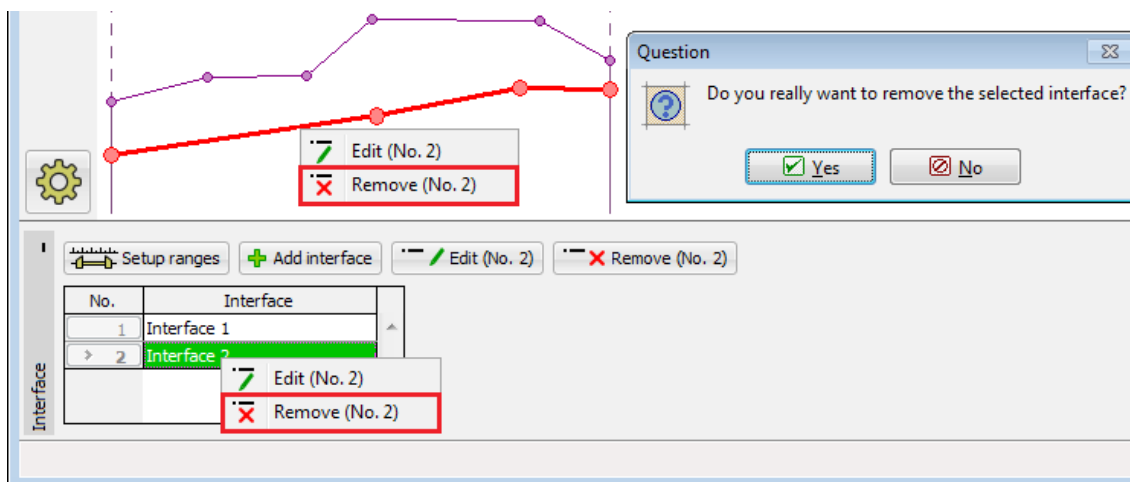


*Editing Interface*

Input is terminated by pressing the button  Edit (adds the input interface into the **interface list**), or by pressing the button  Close (changes are discarded).

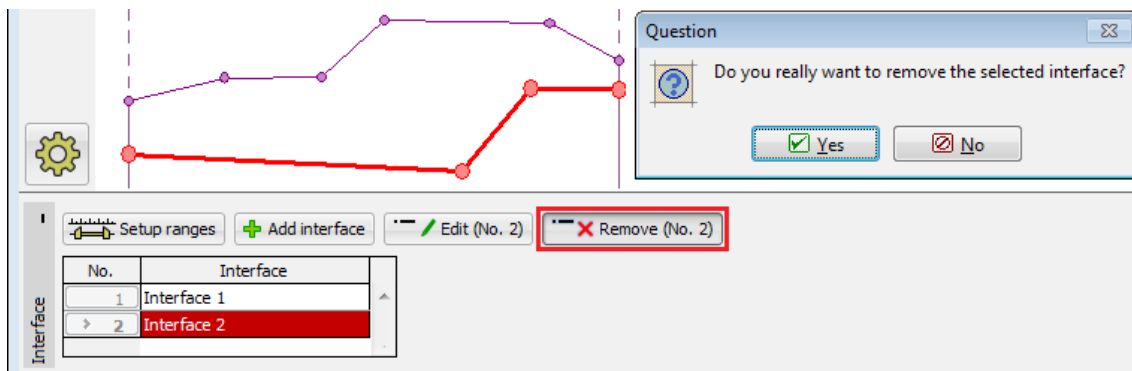
**Deleting Interface**

In the table list, or on the desktop select an interface. Pressing the right mouse button on the interface (table row) opens the **context menu**. After pressing "**Remove**" the program highlights the interface and the table row in red. After confirmation by the user, the interface is removed.



*Removing Interface*

Alternatively, it is possible to select interface in a table and press "**Remove**". Further process is the same as described earlier.



*Removing Interface*

After interface edit, automatic **interface corrector** (as after adding interface), which checks the interface shape, and corrects the end points if required.

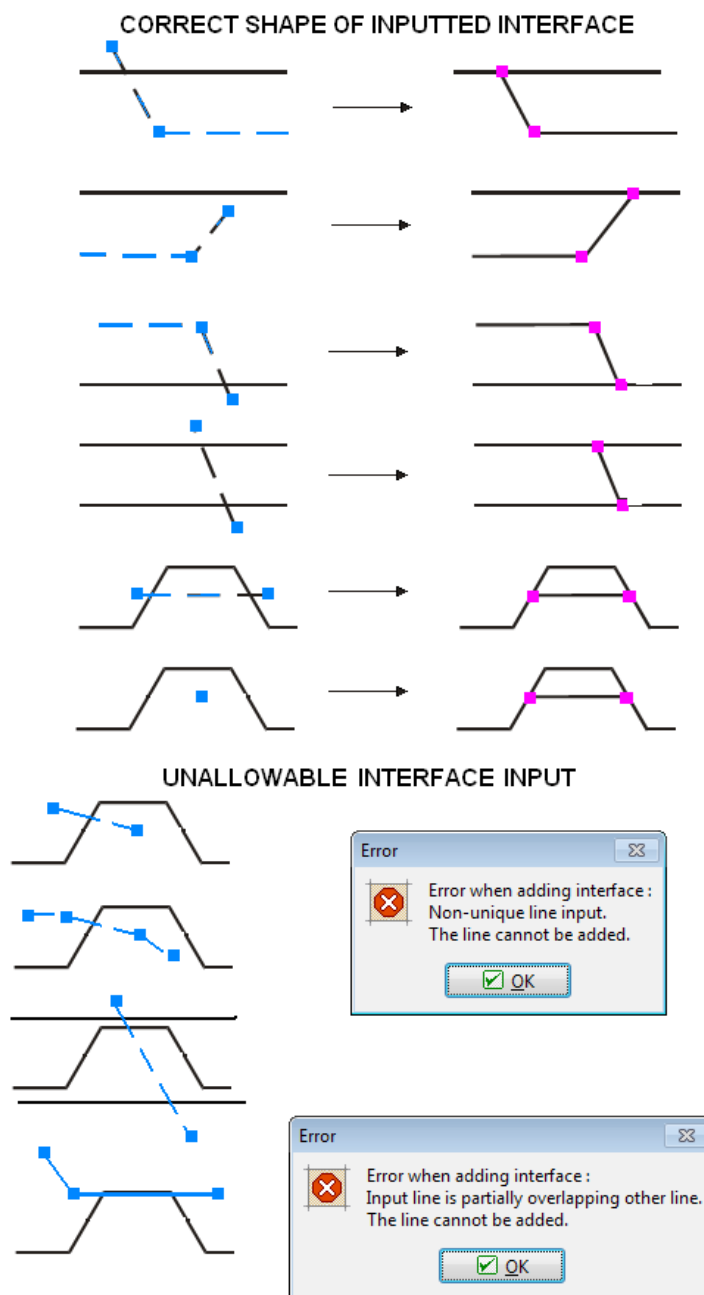
## Input Interface Corrector

When the inputting or editing process is completed the program automatically modifies the input interface to comply with the software requirements, i.e. the end points touch the world edges or other interfaces. The automatic corrector can be further used to simplify the input process - for example if only one point is used to specify an interface, the program automatically creates a horizontal interface containing the defined point.

If two interfaces collide, the corrector creates new end points of the current interface. These points then also become the points of the interface being touched. All lines of individual interfaces thus start and end in a point.

In case of an incorrect input (see the picture below) the interface cannot be stored. In this case the interface must be modified or the inputting process must be stopped using the "**Cancel**" button.

Here are some examples of interface corrector functions (correct and incorrect input):



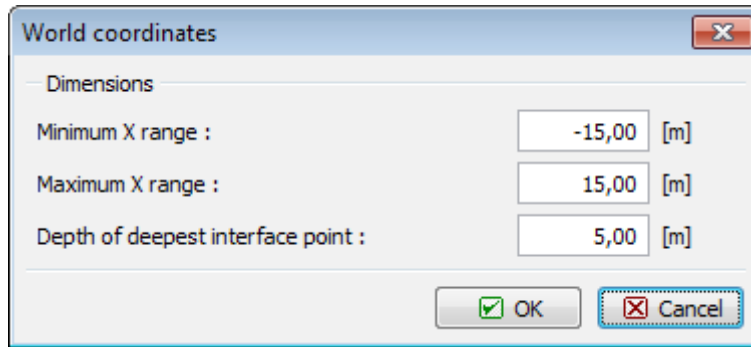
*Correct and incorrect interface shapes*

## World Coordinates

This dialog window is used to specify the world coordinates (dimensions) for a given task - left and right edges. The depth from the lowest point of the interface is for most of the programs auxiliary input - does not have any effect on the analysis itself. In the FEM program, determination of correct world dimensions is very important and can effect the analysis results drastically.

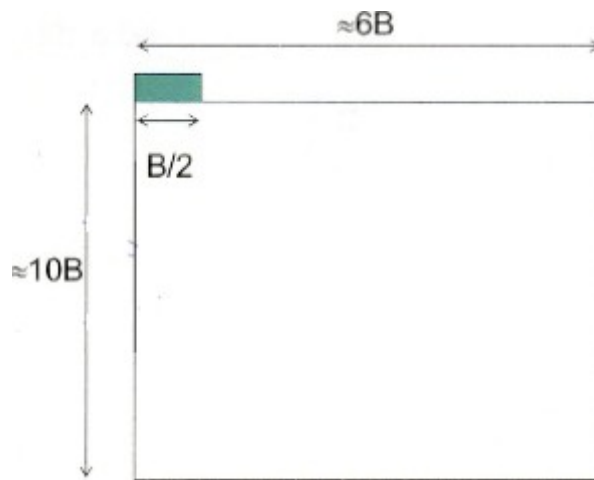
The world coordinates can be changed at any time - when increasing dimensions, all input interfaces are automatically prolonged, when reducing dimensions all points falling off the new world coordinates are automatically removed.



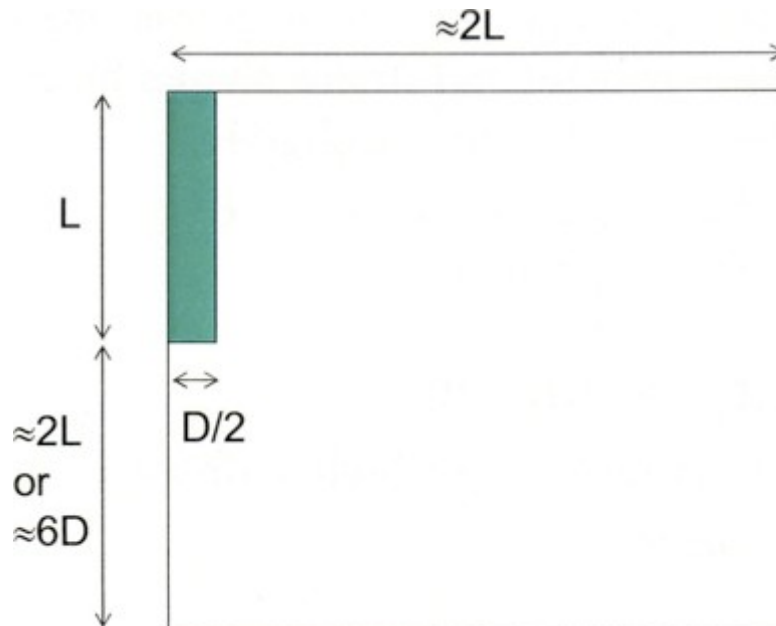


*"World coordinates" dialog window*

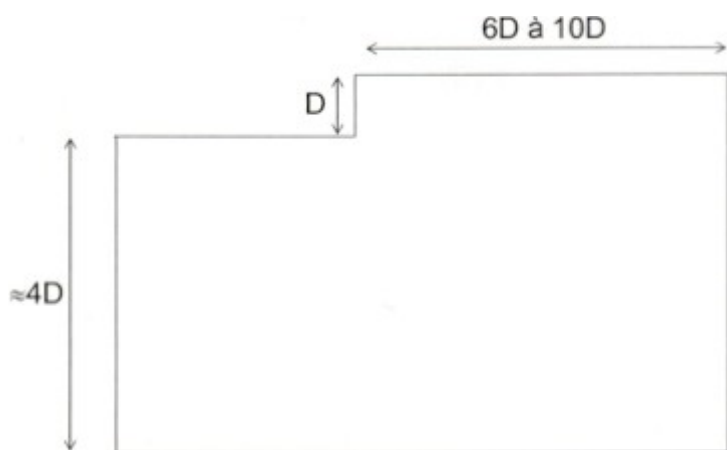
Recommended world coordinates for the FEM program are evident from the pictures.



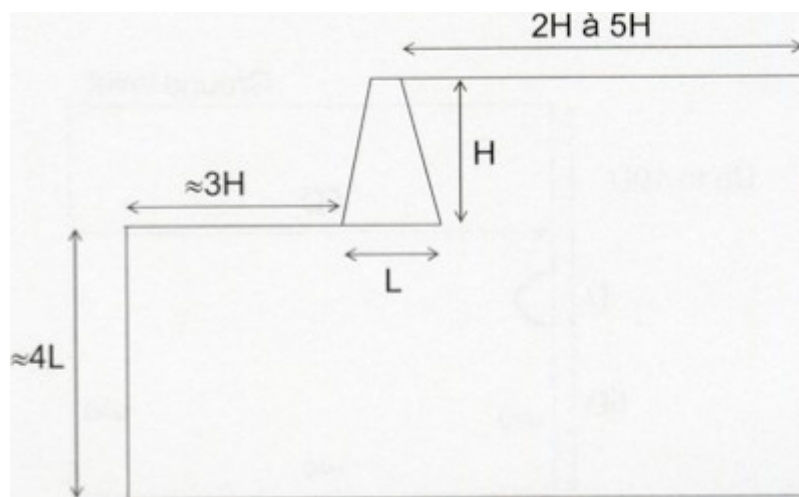
*Recommended world coordinates (boundaries) - Spread Footing (Shallow foundations)*



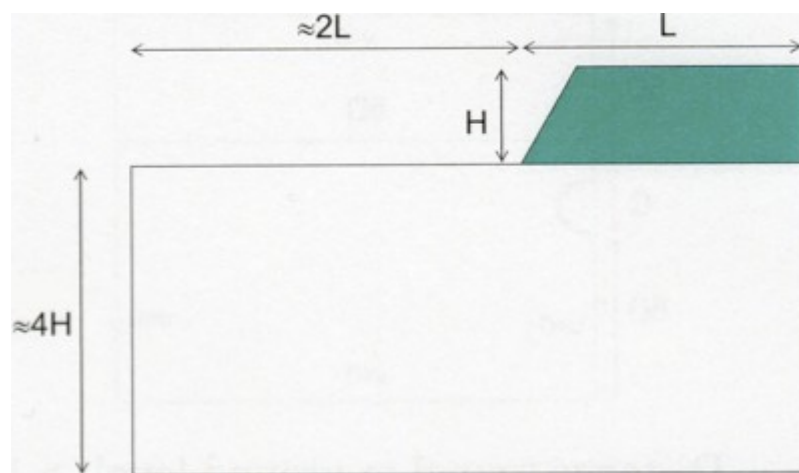
*Recommended world coordinates (boundaries) - Deep and pile foundations*



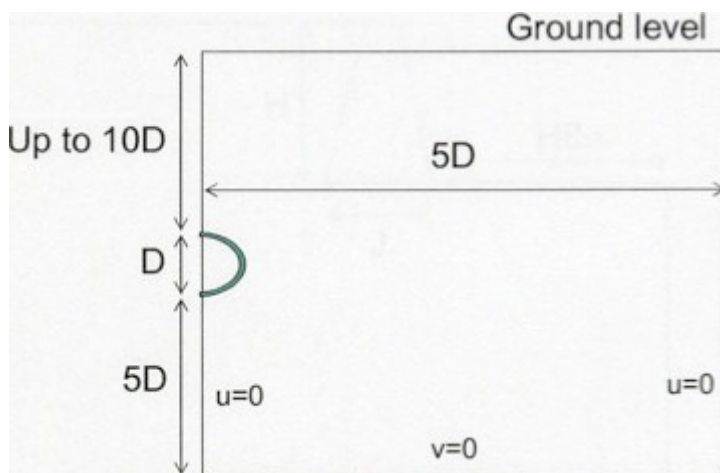
*Recommended world coordinates (boundaries) - Excavations*



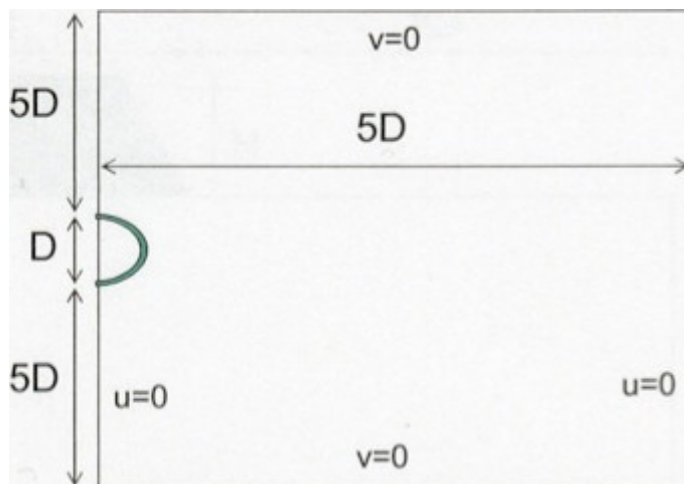
*Recommended world coordinates (boundaries) - Supporting structures and Walls*



*Recommended world coordinates (boundaries) - Embankments and Slopes*



*Recommended world coordinates (boundaries) - Shallow tunnels saved*



*Recommended world coordinates (boundaries) - Tunnels with high overburden*

## Input of Objects and Data

Input data can be entered to the GEO5 programs in several ways. Data is input in **frames** and **dialog windows**. Access to individual frames (input modes) is provided by the "Frames" tool bar. Apart from direct input in the frames, it is also possible to add data (coordinates of points, profile depths...) and objects (surcharge, anchors, georeinforcements etc.) using the buttons "**Add**", "**Add in dialog**" and "**Add graphically**".

### Add

#### Add (Add in dialog)

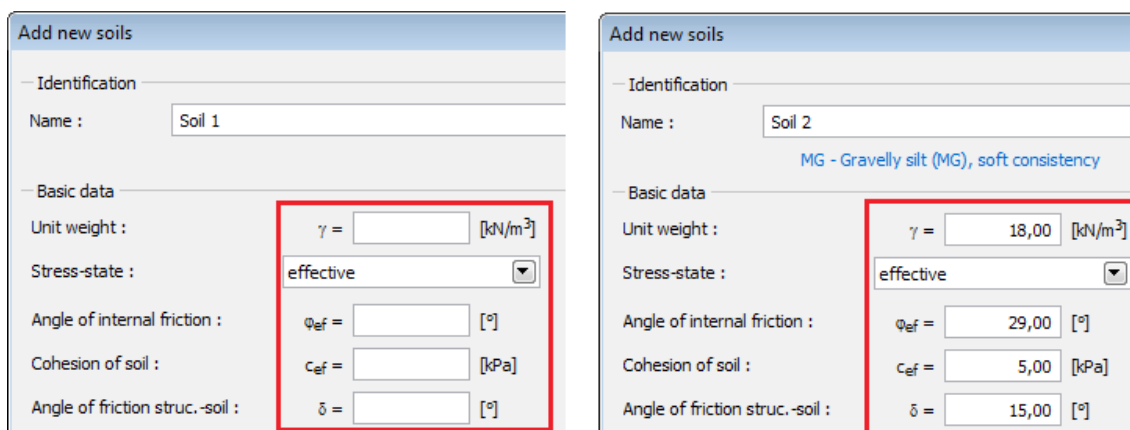
The button "**Add**" ("**Add in dialog**") opens a **dialog window**, where required data is input (for example parameters of soils, surcharge, forces etc.). After confirmation, the data are saved and the item is added into the **table**.



*Visualization of the buttons*

If the frame is used the first time (the table with the list of items is empty), or the table contains items, but none of those are selected, the programs opens an empty dialog window, and all the data needs to be input. If there is a selected item in the table and then the dialog window is opened, it will use the data from the selected item, which can be further modified and saved as a new item in the table list.

The figure below shows an easy example. The frame was used the first time, all fields in the dialog window are empty (left picture). The parameters are input, and added into the table. The table contains selected item "Soil 1". Then, the "Add in dialog" button is used again, and the dialog window contains pre-defined parameters from "Soil 1". After required revision of the parameters, it is possible to save the new data as another item of the table list.

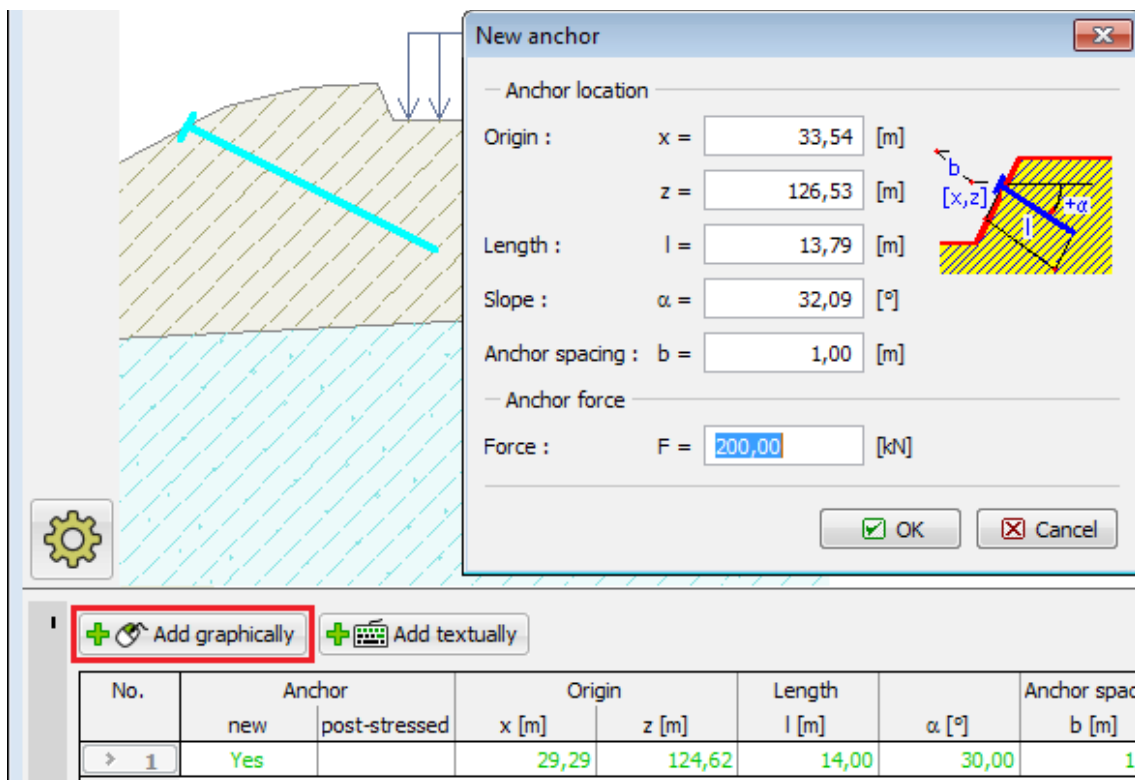


Input modes - using existing data

## Add Graphically

### Add Graphically

The button "Add graphically" turns on the mode of graphical input. The mouse mask changes to an axes cross, and using the left mouse button, it is possible to add the required object. For example when adding anchors, by clicking on the desktop, the initial and end point of the anchor are input. After inputting the second point, the program opens a dialog window. From now on, the process is the same as describe in the chapter "Add"



Graphical input mode

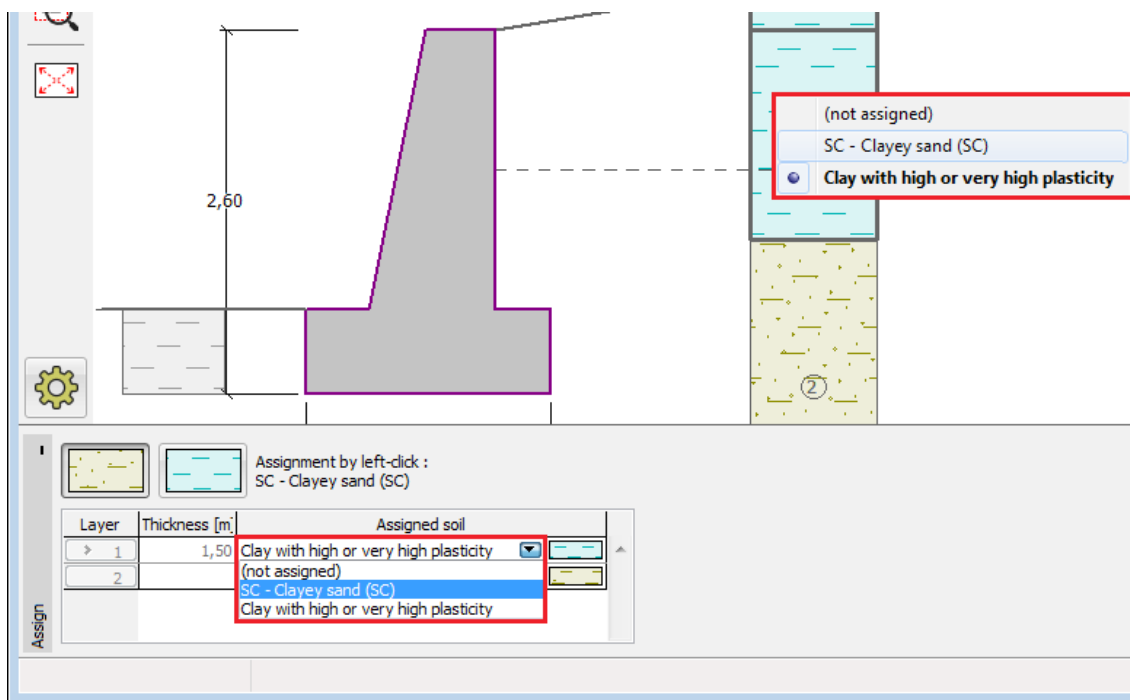
## Assigning Soils

Three options are available to assign soils into individual profile layers.

Clicking the left mouse button on the tool bar button above the table selects the desired soil (positioning the mouse cursors in the bar above the soil button displays a bubble hint with the soil name). The soil is inserted by moving the mouse cursor (the **cursor mask** changes into a "hand") first into a specific layer and then by pressing the left mouse button.

The second option requires to open a combo list of a specific interface and then select the desired soil to be assigned. All changes in the soil assignment are automatically displayed on the desktop.

Last option is to use the mouse **context menu**.



Frame "Assign"

## Design Coefficients

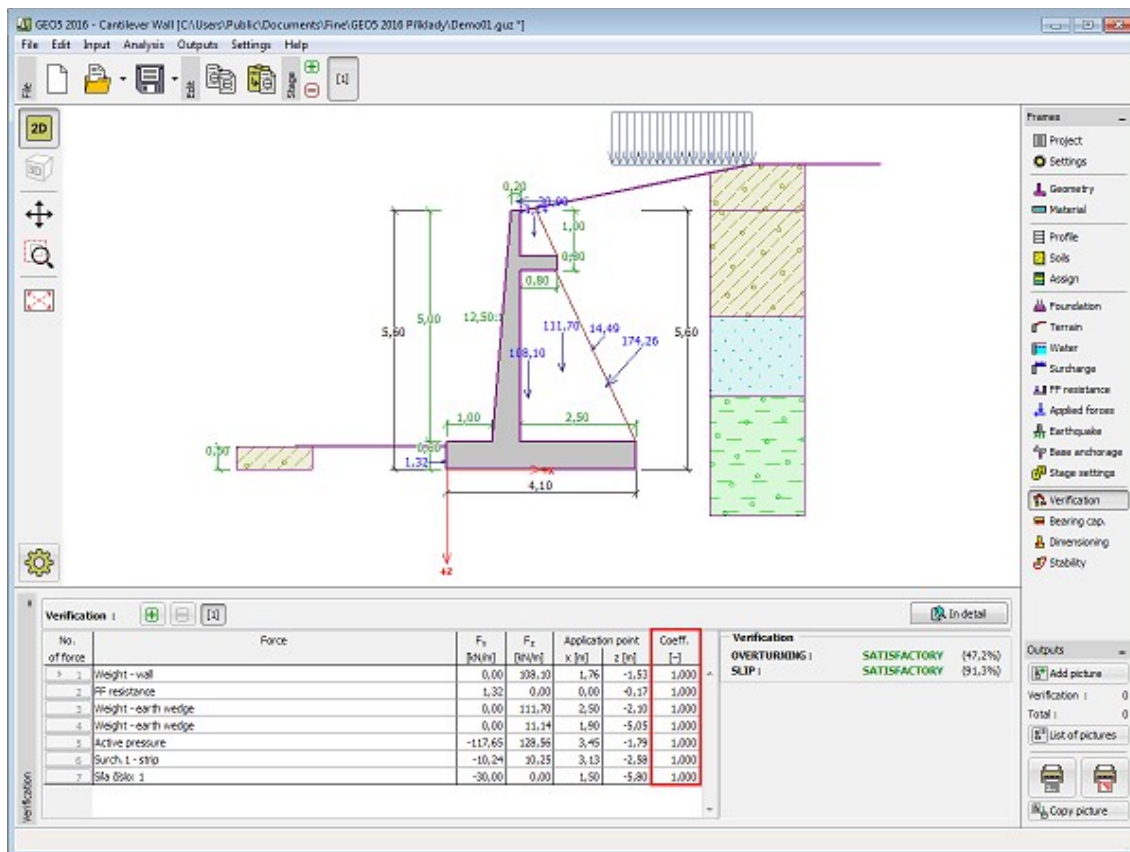
The "Analysis" or "Verification" (for verification methodology - classical way) frames that display the list of computed forces allow to specify design coefficients. A design coefficient multiplies the corresponding force. When inputting the coefficient the results are automatically recomputed and the desktop shows modified forces.

Design coefficients are advantageous for example for:

- Structure testing when a structure response to an increase of force specified directly in the analysis window can be visualized.
- Excluding several forces from verification or their reduction.
- Specifying design combinations - e.g., different coefficients can be assigned in the sense of EC to main load variables and side variable loads.

The following combinations can be specified, for example when performing the wall verification:

	Analysis 1	Analysis 2	Analysis 3
• Wall	1.0	1.0	1.0
• Active pressure	1.0	1.0	1.0
• Surcharge 1	1.0	0.5	0.5
• Surcharge 2	0.5	1.0	0.5
• Surcharge 3	0.5	0.5	1.0



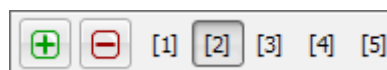
Frame "Verification" - application of design coefficients

## Running Several Analyses / Verifications

Most frames that display the analysis results allow to define more than one analysis to be run. Several analyses within one construction stage are carried out for example for:

- Dimensioning structure in more locations
- Analyses of various slip surfaces
- Verification with various design coefficients

The tool bar in the top part of the frame lets you to manage individual analyses.



Frame "Analysis" - "Running more analyses / verification" tool bar



**Add**

- adds additional analysis onto the tool bar



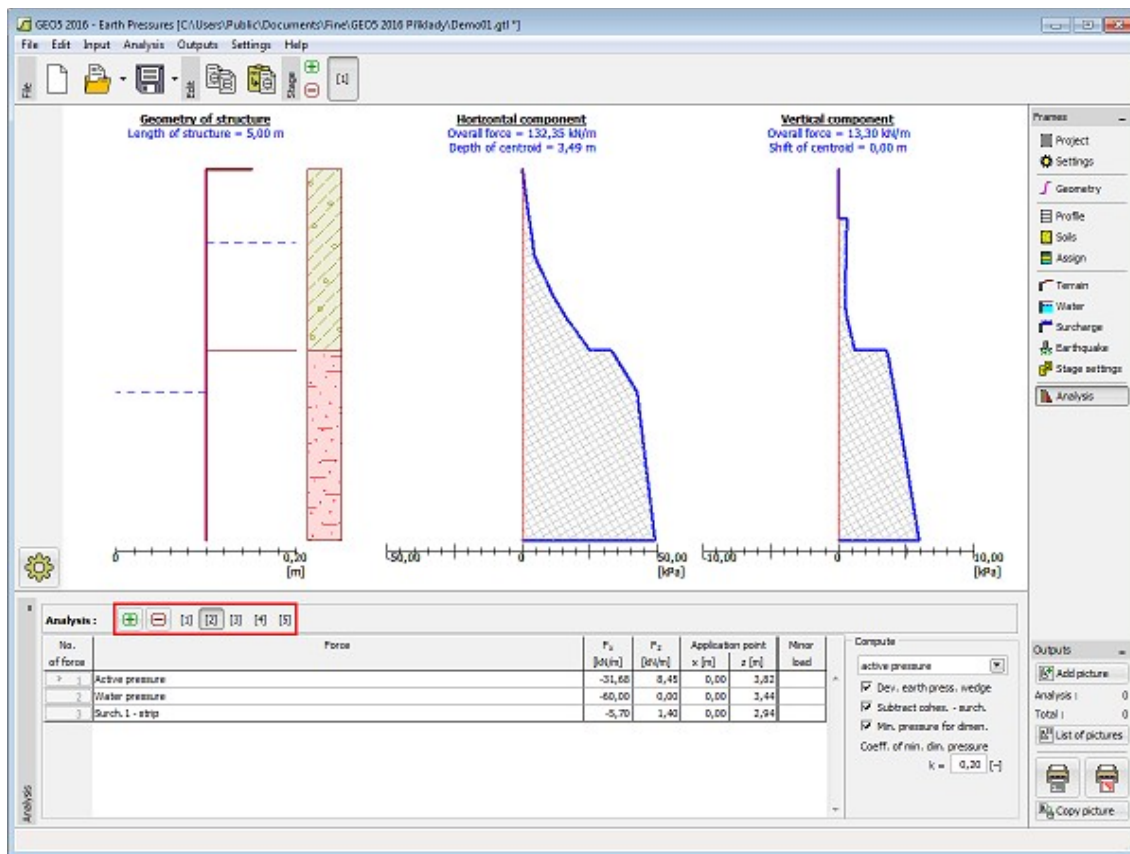
**Remove**

- removes the **currently selected** analysis



**Analysis 1,2,3 ...**

- switches between individual analyses



Frame "Analysis" - "running more analyses / verification"

## Program Connection

In some cases, it is possible to launch another program from a currently running program. For example, the "Cantilever wall" program allows to run the "Slope stability" program to verify the external stability of a structure, or the "Spread footing" program to verify the bearing capacity of a footing of a structure.

The new program loads the data of the structure and then it behaves as a standalone program - closing the program, however, is different. Pressing the "OK" button (on the right below the tool bars) closes the program and the analysis data are passed to the original running program. This is not the case if closing the program by pressing the "Cancel" button.

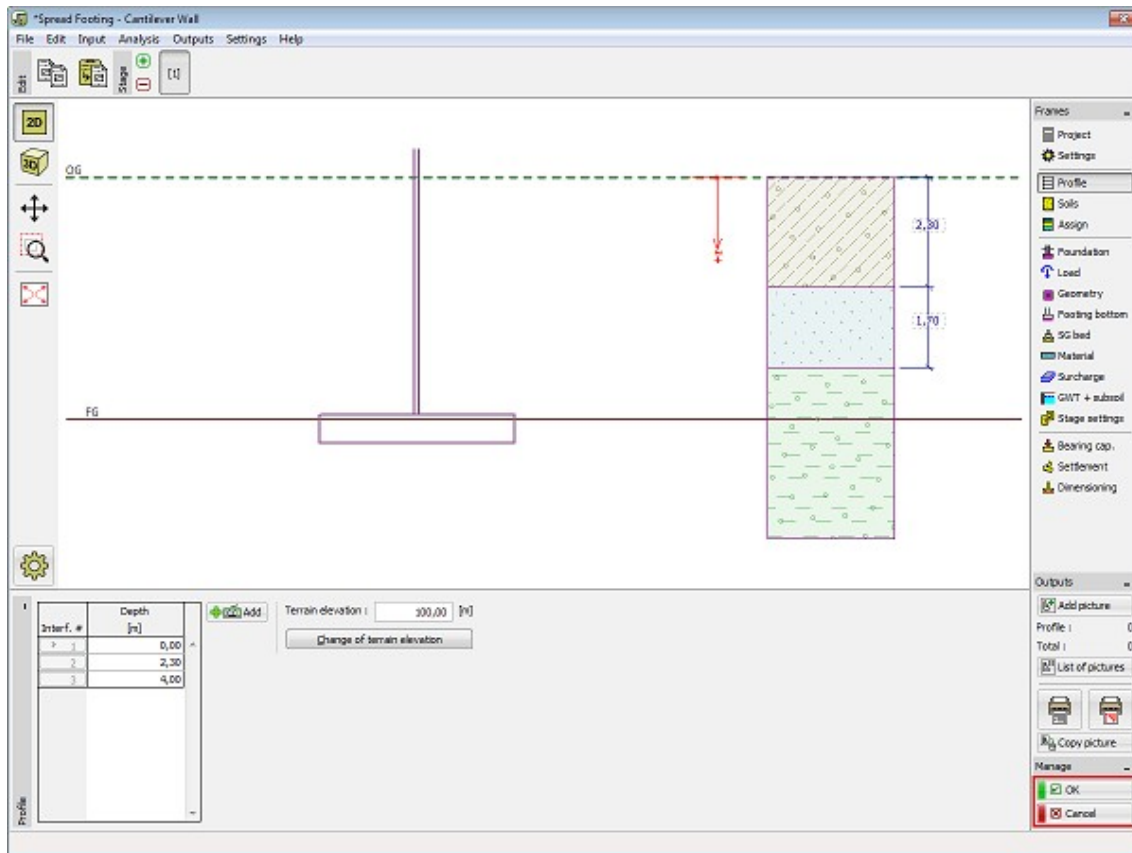
**When running it for the first time**, the program creates data of a structure and passes on the structure dimensions, geology, loads, surcharges and other data. The program then requires you to **input some additional data**, e.g. the analysis method, analysis setting, slip surfaces, stages of construction, etc.

**When running it again** (always necessary if some changes were made in the original program) the program regenerates the data to be passed on, but **keeps the data already input to this program**. For example, when connecting the original program with the "Spread footing" program the new program keeps the additionally input sand-gravel cushion together with input soil - the footing dimensions, foundation geometry, and geological profile are, however, regenerated.

Some actions are not allowed in the new program - e.g. to change the basic setting of the project, unit, etc. The generated task, however, can be saved into new data using the "Save



as" button and work with it as with any other independent task.



"Spread footing" runned from "Cantilever wall" program

## Selection and Store of Views

The programs offer a number of ways of displaying results. A specific option can be selected from the "Visualization settings of results" frame. Quite often, it is necessary to go through a complex and tedious setting of views - for example, if we are interested in the distribution of internal forces developed in beams using FEM, it is necessary to turn off the color range, draw only undeformed structure, select a variable to be displayed, select a suitable magnification, etc.

To simplify the way of managing individual views the programs allows us to **store the current view** using the "Selecting and storing views" bar, and also to **go from one view to another** in a relatively simple way.

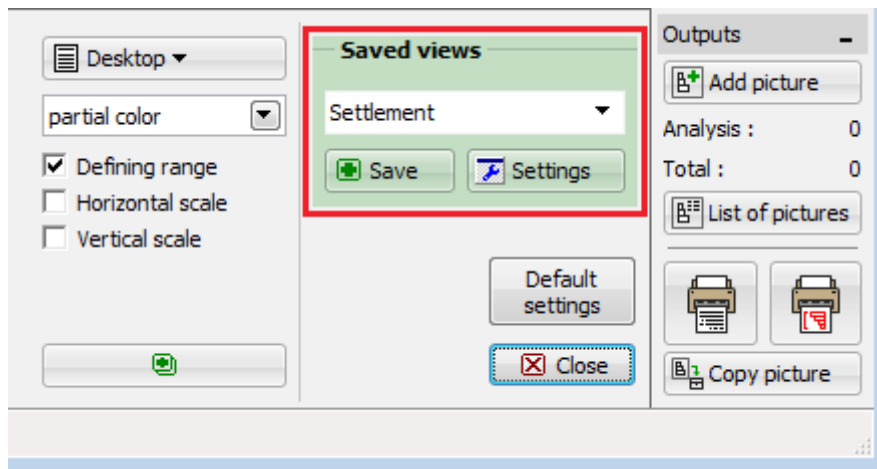
### The stored view keeps:

- All settings from the "Visualization settings of results".
- Drawn variables
- Color range
- Picture zoom

The view is stored for all **stages of construction** - if it is not possible in a certain construction stage to perform such a setting (e.g. in the first construction stage the settlement and depression are not defined) the programs displays the closest possible setting and the defined

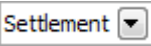
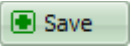

view is switched to **<none>**.

Control elements are shown in the "Analysis" frame in the "Visualization Settings" mode



*"Saved views" - control elements*

The following control units are available to **manage views**:

- |   |                           |   |
|---|---------------------------|---|
|    | <b>Select view</b>        | • Combo list allows for selecting an already specified and stored view. |
|  | <b>Store current view</b> | • Opens the "New view" dialog window to store a new view.               |
|  | <b>Open view manager</b>  | • Opens the window "Picture manager" with a list of views.              |

## Visualization Settings of Results

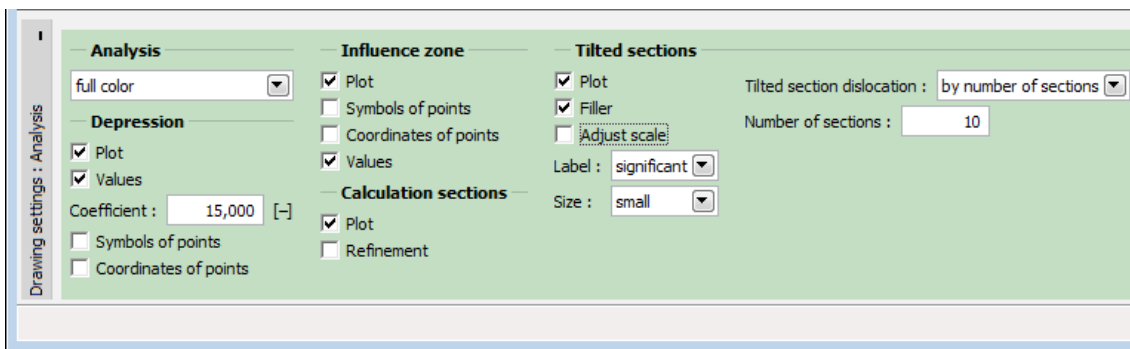
The "Visualization Settings of Results" frame provides tools for a lucid way of displaying the results both on the screen and in the printed document:

- Parameters to draw **depression line** and **influence zone**
- Setting surface views and **color scale** drawing
- Setting and drawing **tilted sections**

The programs based on the **finite element method** further allows to set:

- Parameters to draw the finite element **mesh**
- Parameters to draw construction - **deformed / undeformed** (note that undeformed option must be selected when displaying beam internal forces)
- **Distribution of internal forces** along interfaces and on beam elements

All information specified in this frame (including the setting of current magnification) can be stored using the **selecting and storing views** bar.

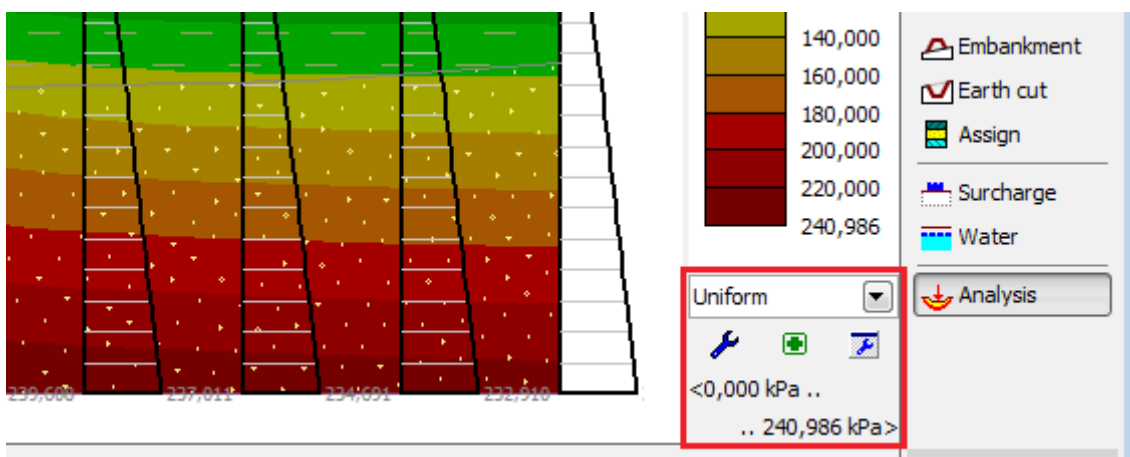


Frame "Visualization Settings" - results visualization settings

## Setting a Color Range




The color range is an important tool providing a lucid way of visualization of results. The program offers two predefined types of color ranges - "**Uniform**" and "**Across zero**". Both ranges have a moving minimum and maximum value and predefined colors. The minimum and maximum values are **automatically regenerated** whenever the variable or a stage of construction is changed. The "**Uniform**" range means that colors are uniformly spread from the minimum to the maximum value. The "**Across zero**" range draws the positive values using warm colors (yellow, red), while cold colors (green, blue) are used to represent negative values.

The program allows to introduce **user-defined ranges** with both the **fixed** minimum and maximum and the **moving** minimum and maximum. A user-defined range is specified in the "**Scale color definition**" dialog window. The range is always defined for the current unit (e.g. *kPa*, *m*) - when switching the units the program always adjusts the range particular for a given unit.



Control units of a tool bar "Ranges"

The following control units are available to **manage ranges**:

- |   |                            |  |
|---|----------------------------|--|
|  | <b>Select range</b>        | <ul style="list-style-type: none"> <li>a combo list for selecting an already specified and stored range</li> </ul>                       |
|  | <b>Define color ranges</b> | <ul style="list-style-type: none"> <li>opens the "<b>Scale color definition</b>" dialog window to create a user-defined range</li> </ul> |
|  | <b>Store current range</b> | <ul style="list-style-type: none"> <li>opens the "<b>New range</b>" dialog window to</li> </ul>  |



### Open range manager

- store a new range
- opens the window with a list of automatic and user-defined ranges

## Scale Color Definition

The "**Scale color definition**" dialog window lets you to create a **user-defined color scale**.

The "**Floating minimum and maximum**" check box determines the basic type of a scale. If checked, the minimum and maximum values of a scale are automatically adjusted whenever the corresponding variable or a construction stage is changed. In such a case it possible to adjust the following:

- **Scale refinement** (the minimum number of levels is 4 and the maximum is 100)
- **Scale color**
- **Uniform scale / across zero**

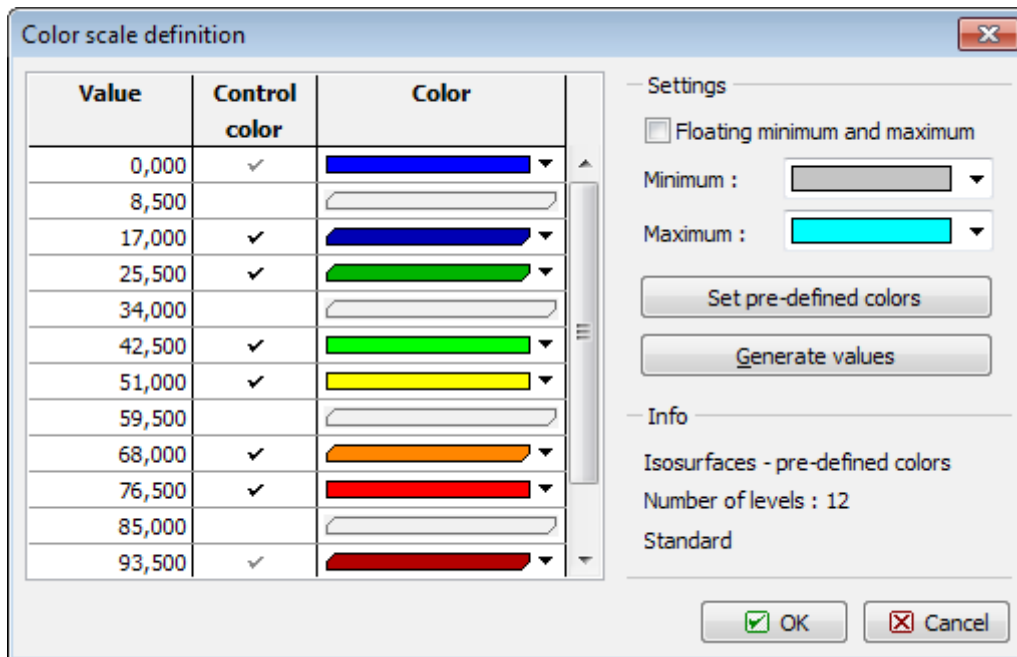
The **number of scale levels and scale type** are specified in the "**Scale generation**" dialog window, which opens after pressing the "**Generate values**" button. It is possible to adjust both **values** and **colors** in the table in the left part of the dialog window. The range values can be easily changed in the table. If the box in the "**Control color**" column is checked, it is possible to choose an arbitrary color from the combo box. The color on intermediate not checked rows are automatically blended from the input colors in checked rows. The default values can be recalled anytime after pressing the "**Predefined colors**" button.

An important property of the program is a definition of ranges with the **fixed minimum and maximum**. If the "**Floating minimum and maximum**" check box is not checked, the color range is fixed and its minimum and maximum values are input. As oppose to the moving range it is further possible to specify:

- **Range end points** (in the "**Scale generation**" dialog window)
- **Colors** to display **values out of the range**

When changing a variable or a construction stage the **color range remains still the same**, keeping the same end ranges. The values found outside the range (below the minimum or above the maximum value) are drawn using colors specified in the right part of the window. **The minimum and maximum range values** are input in the "**Scale generation**" dialog window. The input minimum and maximum values **are linked to the same unit** - e.g. when specifying the rage of 0 - 200 *kPa*, this range is kept the same for all variables being specified in *kPa* - when changing the currently displayed variable to the variable settlement, the current range is switched to that corresponding to the unit of settlement.

For both the fixed and floating scale it is possible to choose whether the colors in the ranges are distributed **uniformly** or **across zero**. The "**Uniform**" scale means that colors are smoothly spread from the minimum to the maximum value of the scale. The "**Across zero**" scale draws the colors above the selected value using warm colors (yellow, red), while cold colors (green, blue) are used to represent the colors below the selected value.



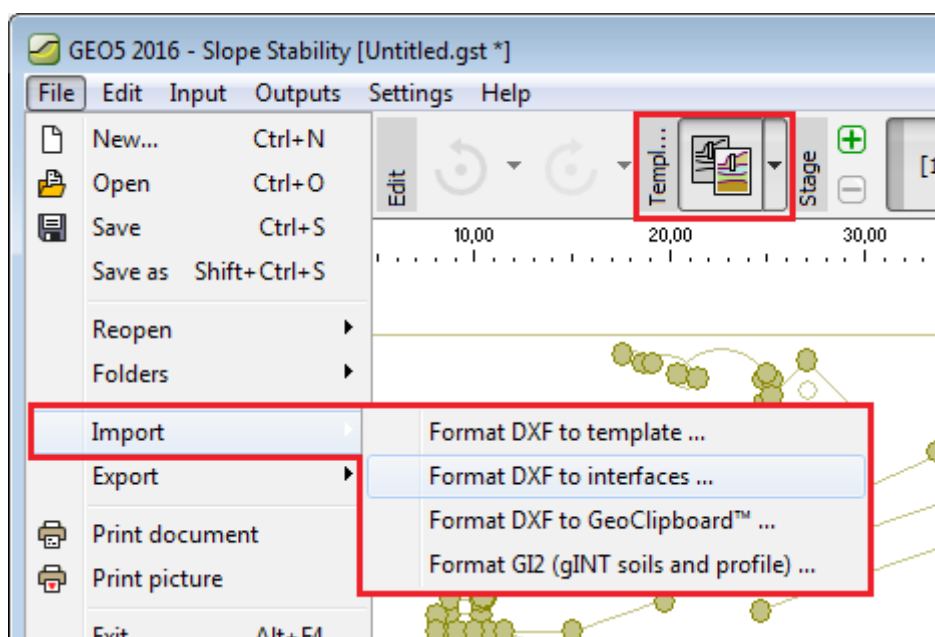
"Scale color definition" dialog window

## DXF Import and Export

2D programs (Slope stability, Settlement, FEM, Beam) allow to import and export data in DXF format.

1D programs (Cantilever wall, Gabion, Spread footing, Pile...) only allow to export data in the \*.DXF format.

The program main menu (item **"File"**) contains **"Import"** - **"Format DXF to template"**, **"Format DXF to interface"**, **"Format DXF to GeoClipboard"** and **"Export"** - **"Format DXF"** items.



### *Menu and tool bar - Export - Import DFX"*

Data import proceeds in several steps:

- Loading data into template
- Loading data into interface
- Inputting data using template
- Modifying template during data input

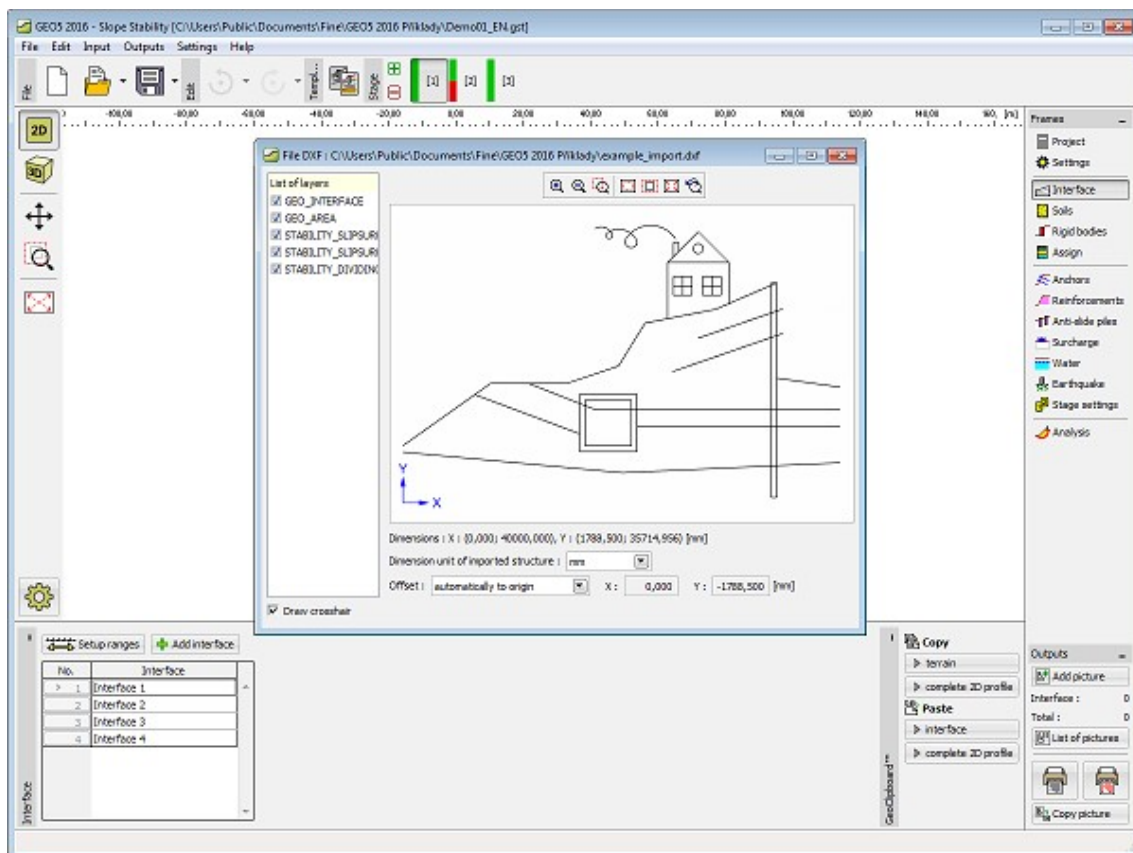
The data input to the program can be **exported in DXF format** anytime.

## Loading Data into Template

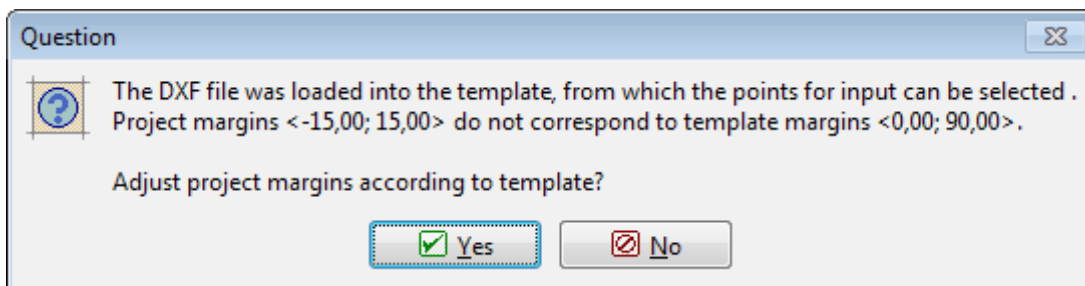
Select the **"Import"**, **"Format DXF into template"** item in the program menu (**"File"**) and then select the file to be imported. The loaded data is displayed in the dialog window **DXF Import** and then is read into the template. The layer selection can be **modified anytime**.

When importing data, it is possible to adjust the world margins based on the imported data - this is particularly useful when defining a new task.

Imported data is not transferred directly into the program. Instead, it is read into a template, which is **used to transfer data into program** later on. When the data is loaded, the template is displayed on the desktop and the buttons on a horizontal tool bar, which are used to manage the template, are made available.



*Loading data into a template*



*Modifying world margins*

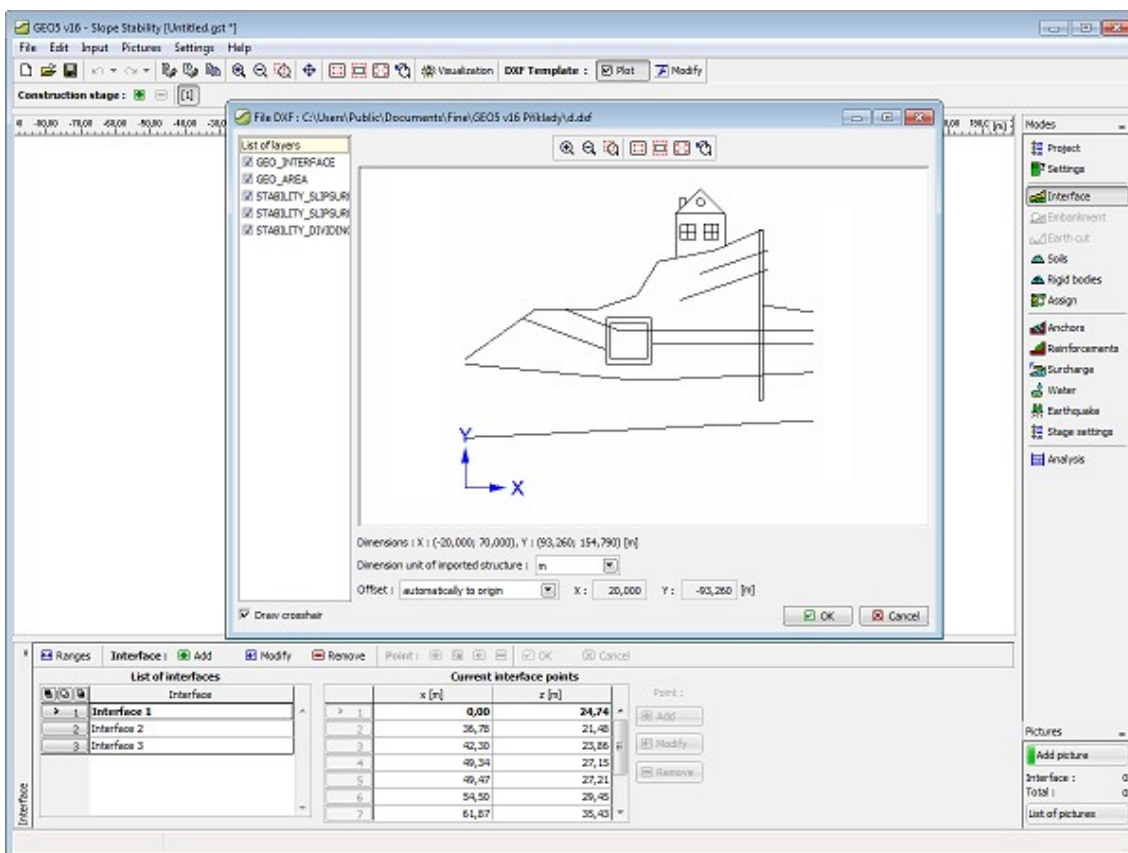
## Loading Data into Interface

Select the **"Import"**, **"Format DXF into interface"** item in the program menu (**"File"**) and then select the file to be imported.

The loaded data is displayed in the dialog window **DXF Import**, which allows to select individual layers and specify other parameters. Program automatically adjusts the world margins based on the loaded data.

After pressing the **"OK"** button a new file is created and DXF data is loaded into interfaces.

If not all selected layers of the structure are successfully loaded, the program allows to use the loaded DXF file **as a template**.



*Loading data into an interface*

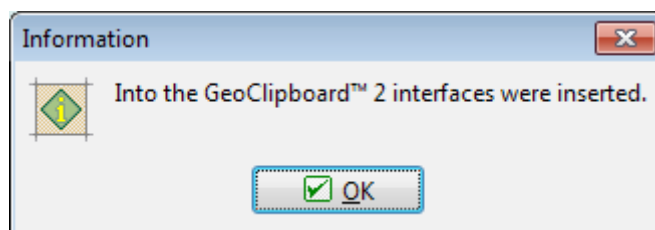


## Loading Data into GeoClipboard

Select the "**Import**", "**Format DXF into GeoClipboard**" item in the program menu ("**File**") and then select the file to be imported.

The loaded data is displayed in the dialog window **DXF Import**, which allows to select individual layers and to specify other parameters.

After pressing the OK button the selected layers of DXF file are converted into interfaces and copied into **GeoClipboard**. Data from GeoClipboard can be afterwards pasted into various places in the program.

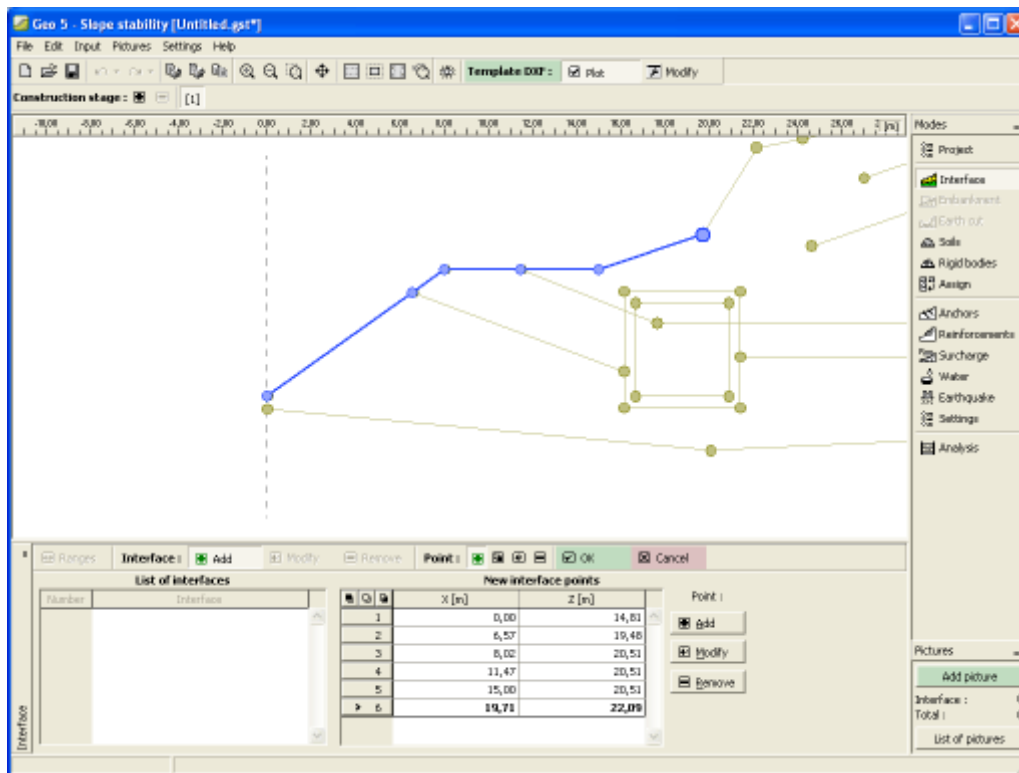


*Information about succesfull copying into GeoClipboard*

## Data Input using Templates

Inputting data using a template is essentially the same as standard input of data in the program. The main difference appears in the possibility of adding a point from a template into the data being input. During the input the mouse cursor appears as an axial cross - when approaching the template it turns into a small cross and long axes disappear. When a point is now input (using the left mouse button) the point from the template is inserted (the input point has now the same coordinates as the point in the template). To accelerate the input of individual lines it appears useful to employ the zooming tools. After interfaces are put in, the procedure can be applied to input other entities. During input it is possible to **modify the template anytime**.

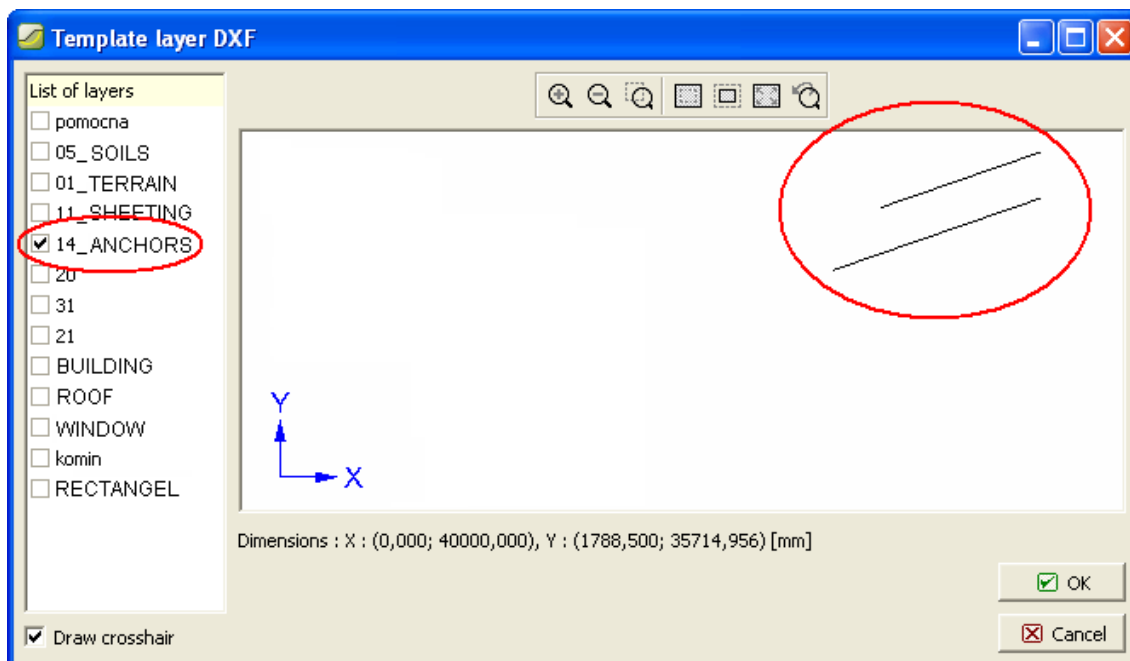




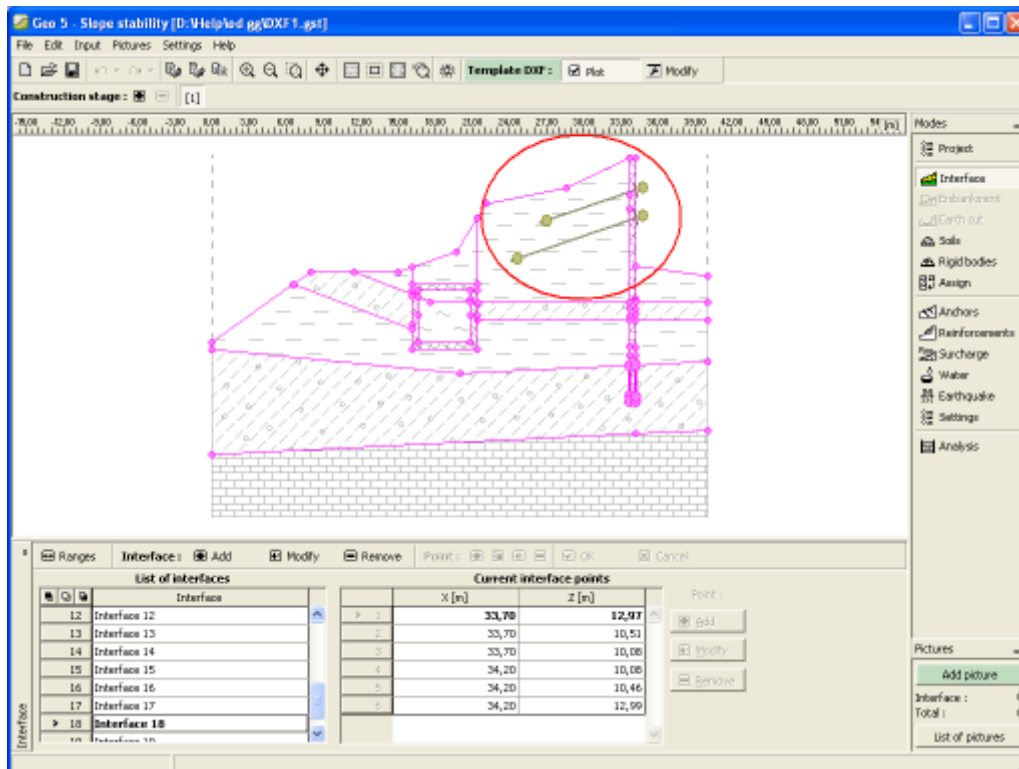
Inputting data using a template

## Modification of Template During Data Input

When inputting data, the template can be modified anytime by pressing the "Modify" button on the "Template DXF" tool bar. This opens a dialog window with individual layers of the template. For example, when inputting anchors, it is possible to turn off all layers except anchors - inputting anchors then becomes simple and lucid.



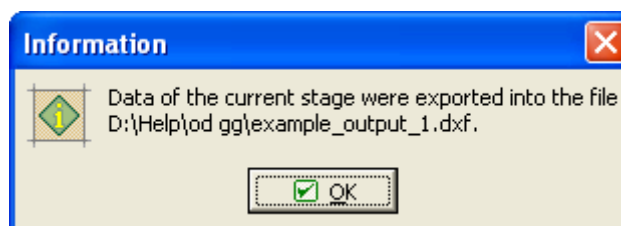
*Turning on/off layers in a template*



*Display after modifying layers in a template*

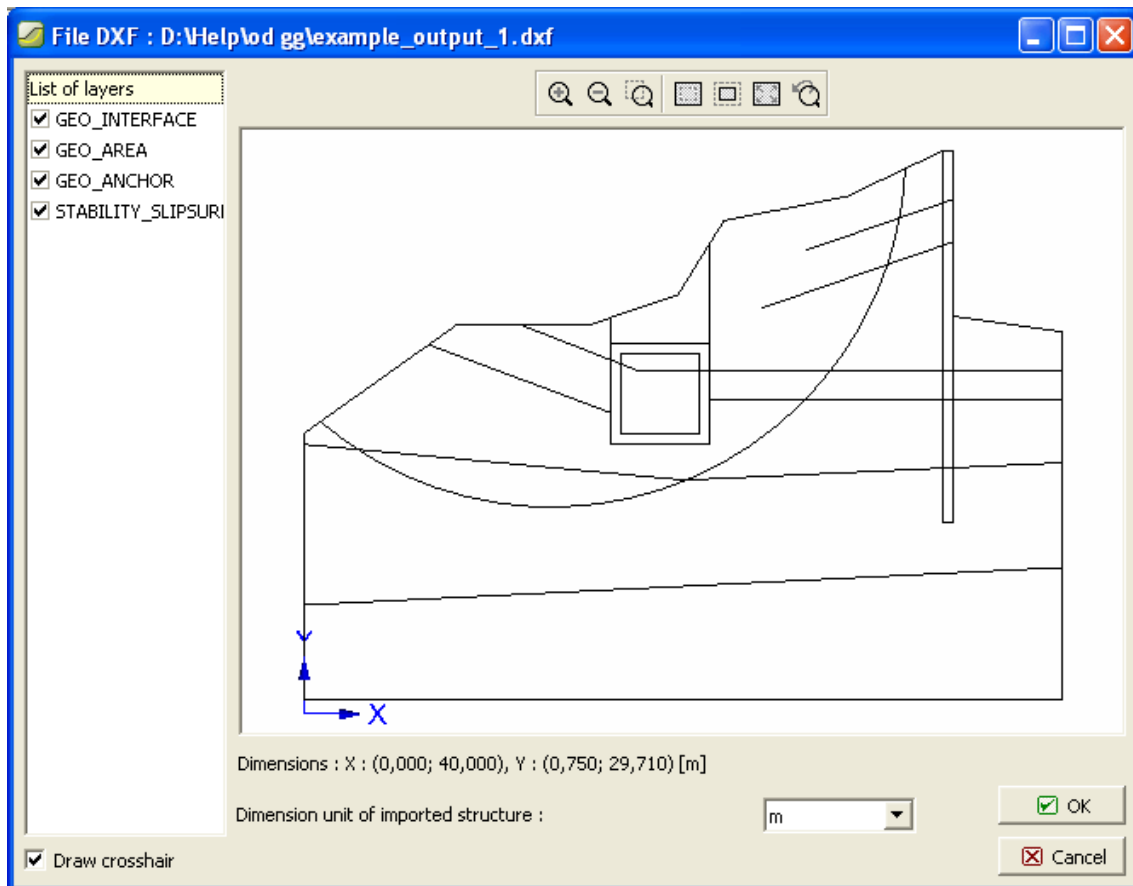
## DXF Export

In the program menu (item "File") select the item "Export", "Format DXF". Next, select the file name intended for export. Using a dialog window the program then provides information regarding the performed data export.



*Information regarding the performed data export*

The exported data can be verified by importing them back into the program.



*Check of exported data*

## DXF Import

In the dialog window DXF Import the parameters of the DXF import are specified.

In the upper part there is the preview of the imported data.

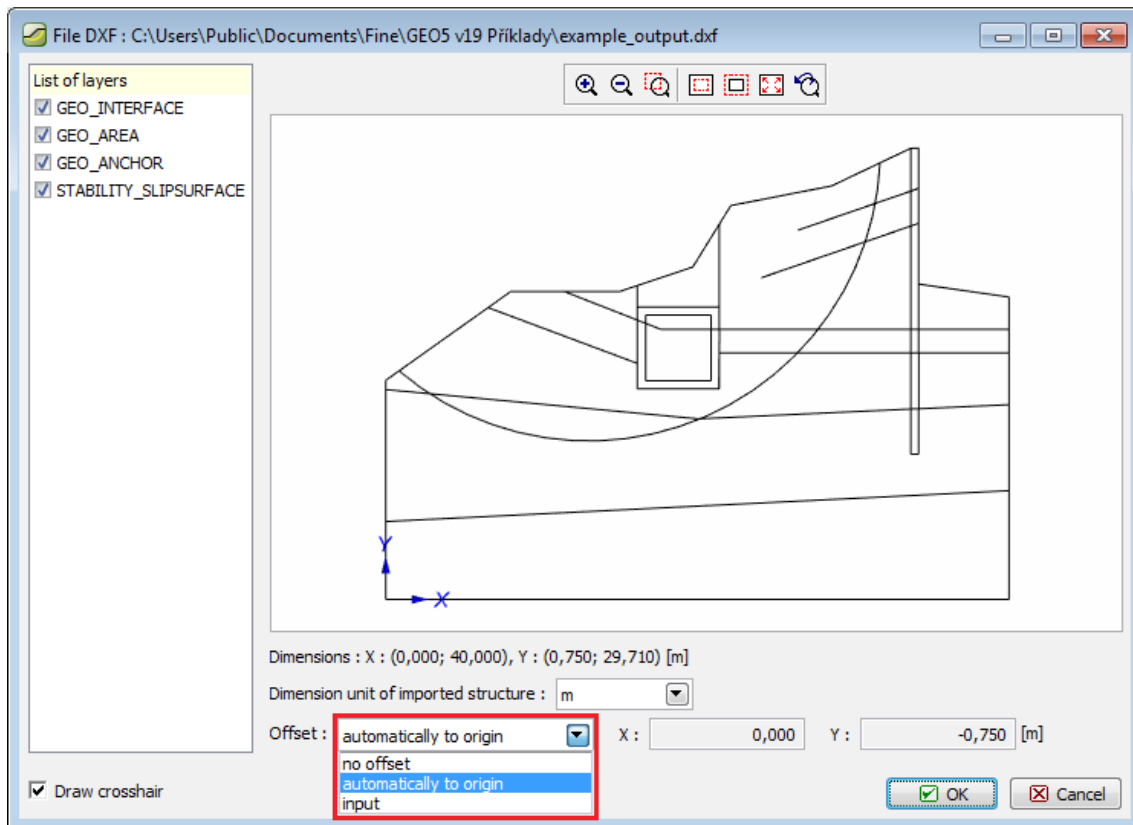
In the left part the layers to be read can be selected.

In the lower part the unit used when creating the DXF file can be specified. The program attempts to estimate it but it is **always necessary to check** whether the unit is specified correctly.

The structure can also be moved. The program offers the following options:

- **do not move** - this option reads the data in the same way as they were entered
- **automatically from zero** - this option moves the structure left bottom corner to the coordinate system origin
- **input** - this option allows to define the shift manually

If the program does not allow to input arcs and circles, the way of splitting it into line segments can be specified.



*Dialog for DXF Import and its parameters*

## Table Data Import

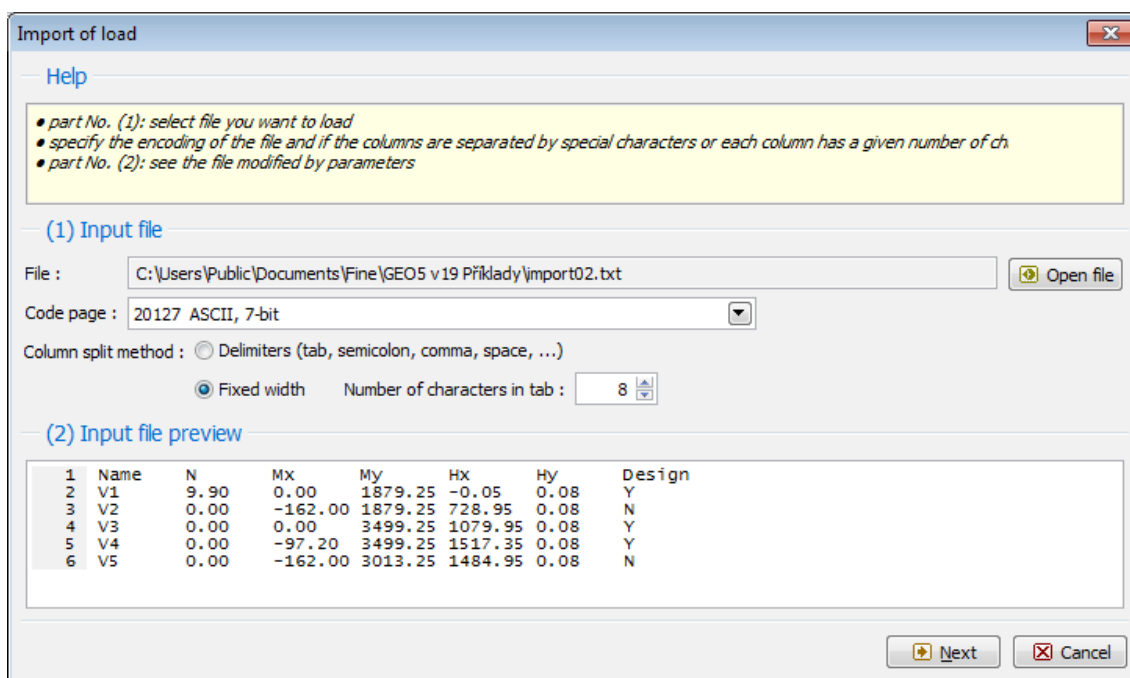
This tool is made for importing table data (i.e. data organized in columns) into tables in the program, e.g. load, coordinates, etc. The tool can read the following formats:

- text file separated by **delimiters**, e.g. commas, semicolons, tabs, file extension CSV
- generic text file with **fixed width of columns**, data is organized into columns using spaces or tabs, usual file extension TXT
- Microsoft Office Excel **tables** (Office Open XML) - file extension XLSX
- OpenOffice system **tables** (OpenDocument) - file extension ODS

Import is organized gradually in three steps, which moreover varies according to the loaded format. At the top of the window brief help is displayed. The program attempts to analyze the file content and to suggest the best parameters of the transfer. If the user changes some parameters, the program tries to remember these changes and use them appropriately for other files.

### **1st step: select a file, determine the file type and view its contents**

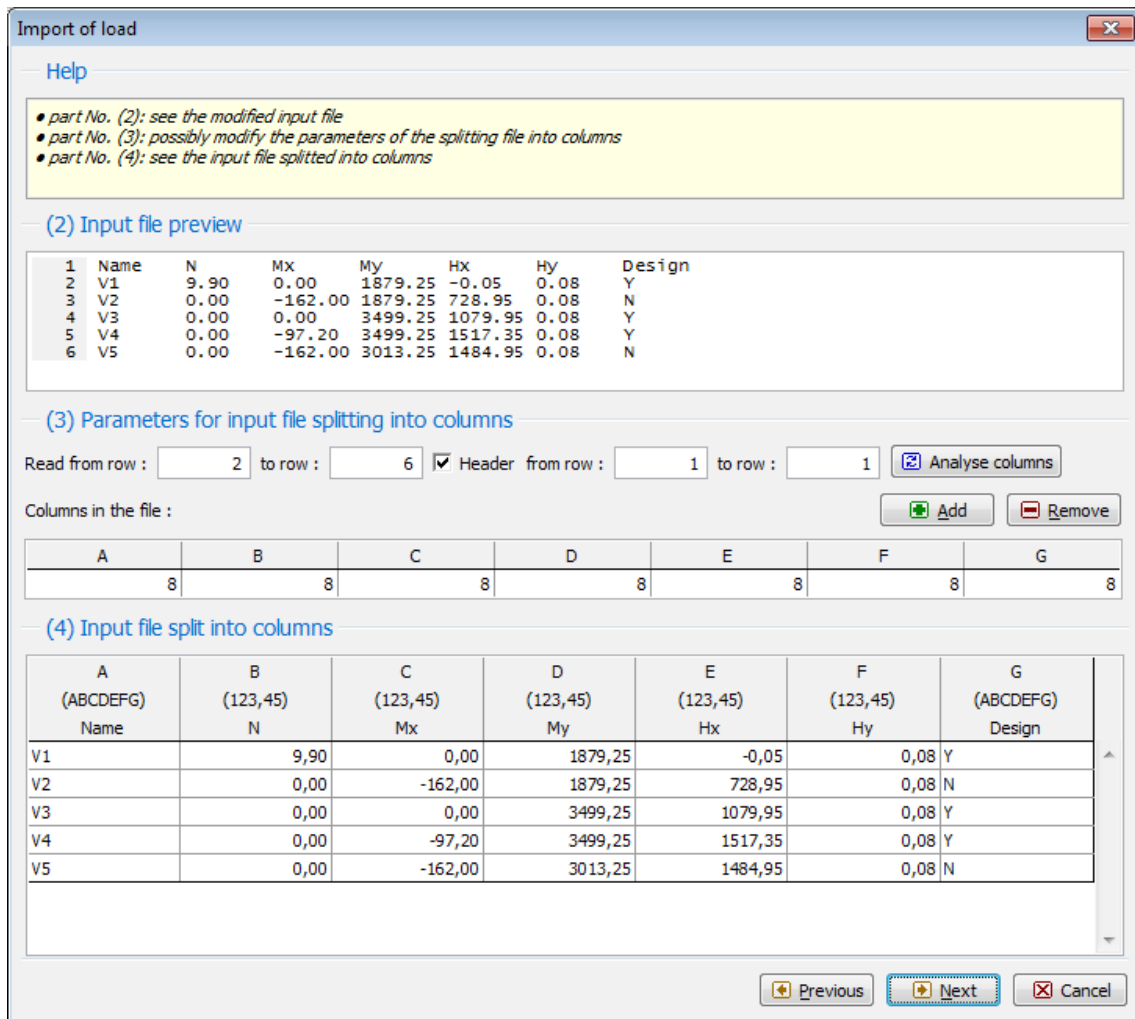
In this step the visible parts are (1) **Input file** and (2) **Input file preview**.



*1st step*

## 2nd step: splitting the input file into columns

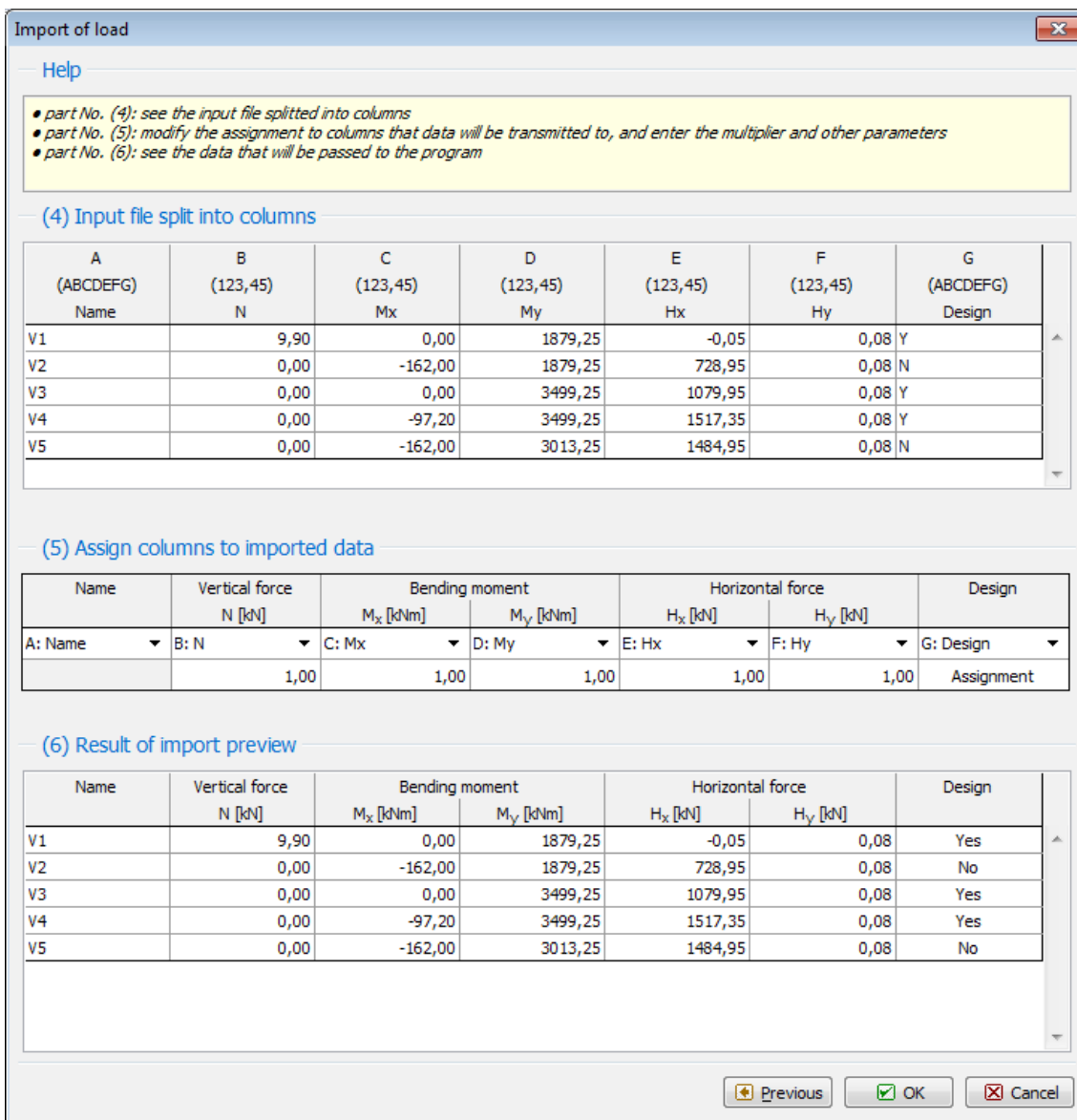
In this step the visible parts are (2) Input file preview, (3) Parameters for input file splitting into columns and (4) Input file split into columns.



2nd step

### 3rd step: Assigning columns to data

In this step the visible parts are (4) Input file split into columns, (5) Assign columns to imported data and (6) Result of import preview.



3rd step

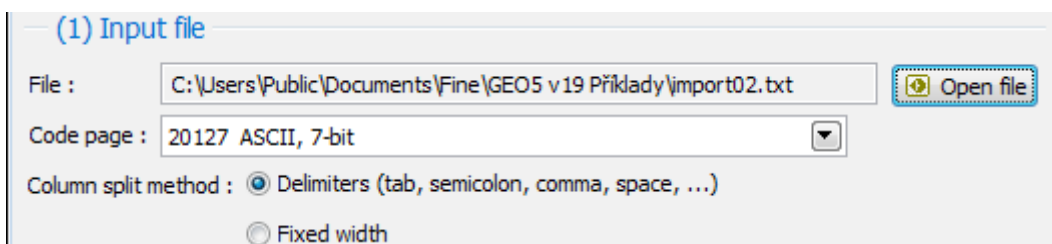
After pressing the "OK" button the data is transferred into program.

## (1) Input File

In this section the input file and its basic parameters are specified. The file is opened in the standard way by pressing the "Open file" button. The program analyzes the input file and fills in the data in this section.

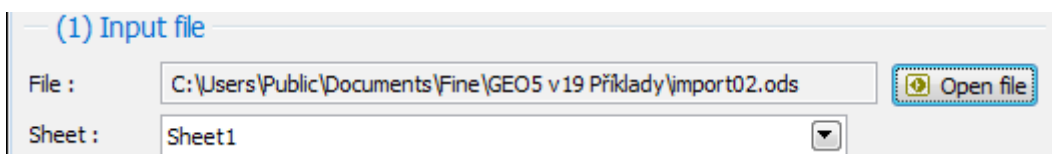
If the **text file** is imported, the following parameters are determined:

- Encoding - encoding (language) in which the file is written can be changed
- The style of separating columns - it is specified whether the file is separated by special characters (which are then entered [here](#)) or whether the columns have fixed width
- The number of characters in a tab - program replaces the tabs with spaces for further processing, this parameter can affect how



Text file

If the **spreadsheet file** is imported (e.g. Excel), here it is possible to determine which sheet is imported.

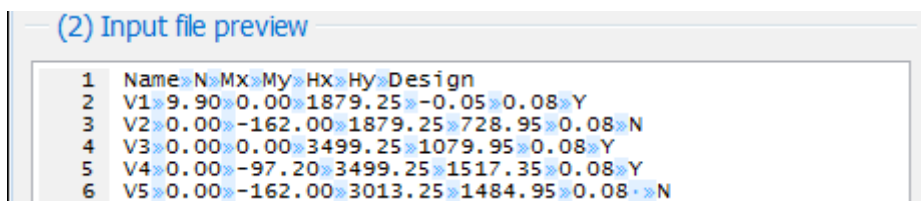


Spreadsheet file

In both cases it is possible to check the result in (2) **Input file preview**. If everything is OK, go further by clicking "**Next**".

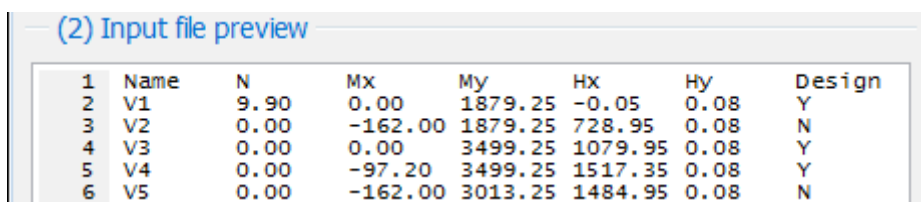
## (2) Input File Preview

If the **text file with delimiters** is being imported, commonly used delimiters are highlighted in the preview .



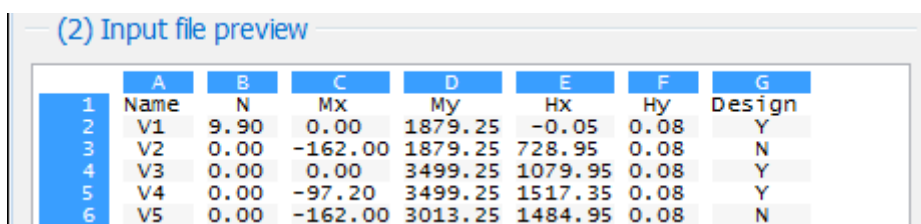
Text file with delimiters

If the **text file with fixed width** is being imported, the preview looks like this.



Text file with fixed width

If the **spreadsheet file** is being imported, the preview contains cell addresses.





*Spreadsheet file*

### (3) Parameters for Input File Splitting into Columns

The program analyzes the input file and fills this part with the parameters obtained.

If the **text file** with **delimiters** is being imported, the following parameters are specified:

*Text file with delimiters*

- the first and the last row to be loaded is determined, if any row contains a header and possibly the first and the last row of the header
- determine the column separators by checking the switches of each type, or check "**Other**" and add another separator into the input line
- "**Treat consecutive delimiters as one**" switch determines how the program will handle immediately following delimiters (even various types of delimiter)
- text qualifiers specify whether the text columns are marked left and right with a character
- comment qualifier specifies the character from which the contents of the file is ignored to the end of the row

The "**Analyse delimiters**" button reanalyses the parameters after changing the row range.

If the **text file** with **fixed width** is being imported, column count and width of each column is specified.

A	B	C	D	E	F	G
8	8	8	8	8	8	8

*Text file with fixed width*

- the first and the last row to be loaded is determined, if any row contains a header and possibly the first and the last row of the header
- the "**Add**" button inserts a column at the end of the list, the "**Remove**" button removes the last column
- in the "**Columns in the file**" table the width of each column is specified

The "**Analyse columns**" button reanalyses the parameters after changing the row range.

If the **spreadsheet file** is being imported, only the first and the last row to be loaded is determined if a row contains a header and possibly the first and the last row of the header

*Spreadsheet file*

In all cases it is possible to check the result in the part (4) **Input file split into columns**. If everything is OK, go further by clicking "**Next**".

## (4) Input File Split into Columns

In this part the input file splitted by **parameters** is displayed. The first header row contains the letters A and further, the second row specifies the type of data in the column, the third header row possibly contains the retrieved header from the imported file. The data type can be:

- (ABCDEFG) is a general text
- (123,45) is a number with the decimal point
- (123) is a number without the decimal point

(4) Input file split into columns

A (ABCDEFG)	B (123,45)	C (123,45)	D (123,45)	E (123,45)	F (123,45)	G (ABCDEFG)
Name	N	Mx	My	Hx	Hy	Design
V1	9,90	0,00	1879,25	-0,05	0,08	Y
V2	0,00	-162,00	1879,25	728,95	0,08	N
V3	0,00	0,00	3499,25	1079,95	0,08	Y
V4	0,00	-97,20	3499,25	1517,35	0,08	Y
V5	0,00	-162,00	3013,25	1484,95	0,08	N

*Processed input file*

## (5) Assign Columns to Imported Data

The program prepares the initial assignment of columns. It is then possible to change the assignment manually. The system remembers the user changes and uses it preferably in the same cases.

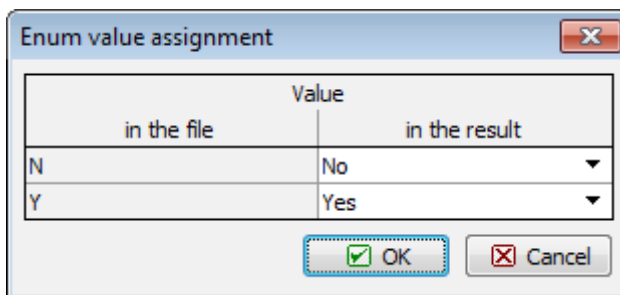
Assignment of columns is carried out in the following table:

(5) Assign columns to imported data

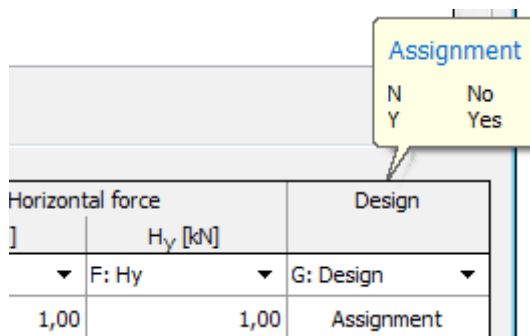
Name	Vertical force N [kN]	Bending moment M <sub>x</sub> [kNm]    M <sub>y</sub> [kNm]		Horizontal force H <sub>x</sub> [kN]    H <sub>y</sub> [kN]		Design
A: Name ▼	B: N ▼	C: Mx ▼	D: My ▼	E: Hx ▼	F: Hy ▼	G: Design ▼
	1,00	1,00	1,00	1,00	1,00	Assignment

*Column assignment*

- The table header contains the columns that are required by the program mode to which the import is done.
- In the first table row the input file column is assigned to be forwarded to the appropriate column of data. The columns can be used repeatedly, but of course you can only assign a compatible data type, i.e. you cannot for example use a text column for number.
- In the second table row the multiplier for numbers can be entered, pressing the "**Assign**" button shows the window in which it is specified how the values with a **Yes/No** type and similar are treated. The current assignments and other information is displayed in the hint-bubble if the focus is in the appropriate column.



Value assignment



Hint bubble

In all cases it is possible to check the result in the part (6) Result of import preview. If any problems arise, an error message is displayed in the window.

If everything is OK, you can complete the import by pressing the "OK" button.

## (6) Result of Import Preview

This section shows the data that will be passed into the program.

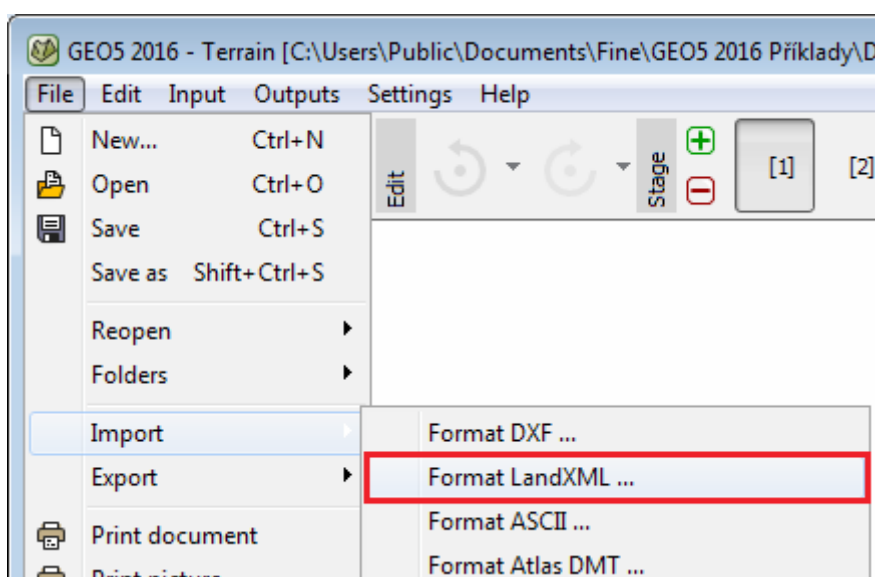
(6) Result of import preview

Name	Vertical force N [kN]	Bending moment		Horizontal force		Design
		M <sub>x</sub> [kNm]	M <sub>y</sub> [kNm]	H <sub>x</sub> [kN]	H <sub>y</sub> [kN]	
V1	9,90	0,00	1879,25	-0,05	0,08	Yes
V2	0,00	-162,00	1879,25	728,95	0,08	No
V3	0,00	0,00	3499,25	1079,95	0,08	Yes
V4	0,00	-97,20	3499,25	1517,35	0,08	Yes
V5	0,00	-162,00	3013,25	1484,95	0,08	No

Result of the import

## Import LandXML

Programs "Terrain" allow you to import data in LandXML format. Select the "Import", "Format LandXML" item in the program menu ("File") and then select the file to be imported in the standard way.



### *Import LandXML*

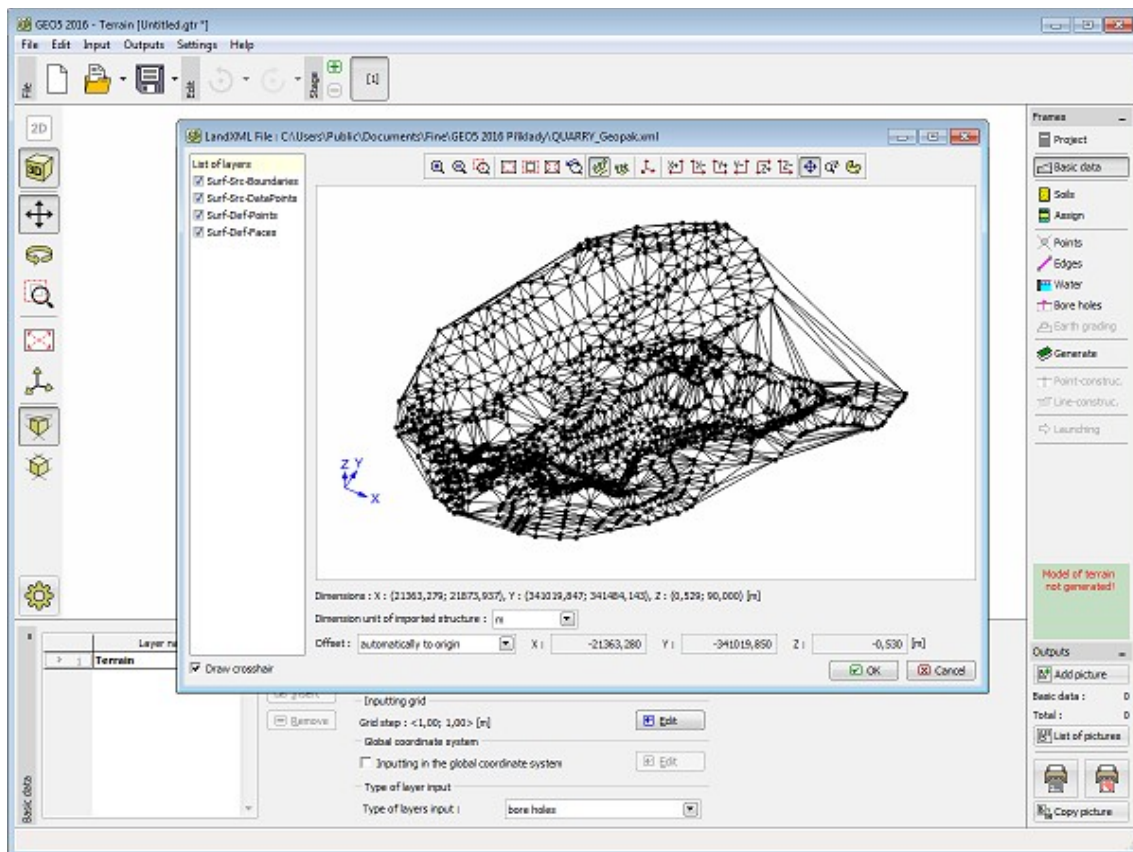
The loaded data is displayed in the dialog window, which allows to select individual layers to be loaded as points and interfaces. The same dialog window also allows to modify the unit used when creating the LandXML file. The structure can also be moved. The program offers the following options:

- **do not move** - this option reads data in the same way as they were input
- **automatically from zero** - this option moves the structure left bottom corner to the coordinate system origin
- **input** - this option allows to define the shift manually

Up on import the program automatically adjusts the world margins based on the loaded data.

Supported LandXML elements: Units, Alignments, CgPoints, Parcels, PlanFeatures, Roadways, Surfaces, Survey.

Not supported LandXML elements: GradeModel, Spiral curves except clothoids.



Loading data

## Import gINT

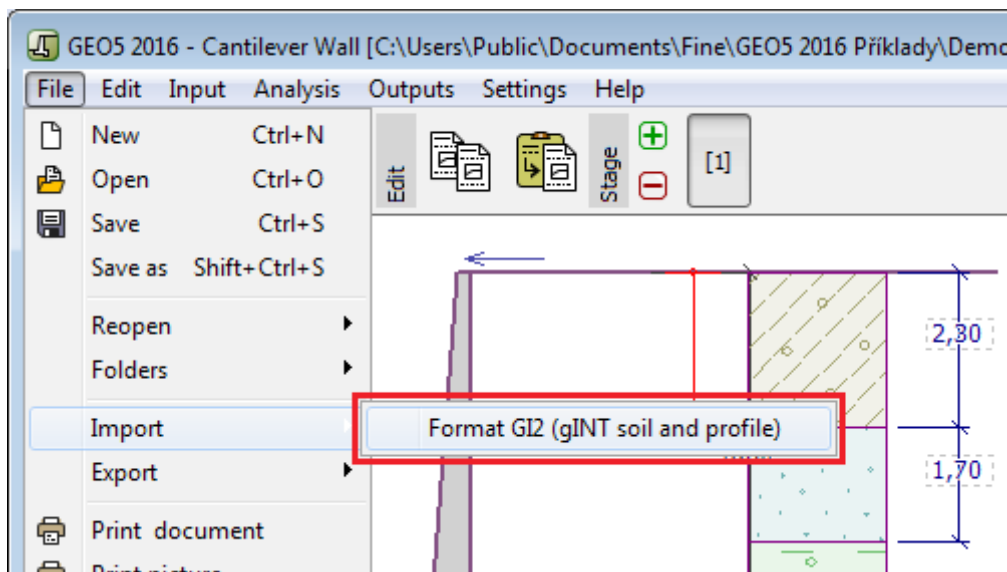
The program is able to import data in the gINT (\*.gi\*) format.

The gINT (\*.gi\*) format is used to process and present geotechnical, geological and hydrological data. Detailed information regarding this company can be found on [www.gintsoftware.com](http://www.gintsoftware.com).

The main program menu (item **"File"**) contains the **"Import"** - **"Format GI\*"** items.

The program allows to open three types of files from gINT programs:

- **\*.GI1** - points of terrain in program **"Terrain"**
- **\*.GI2** - soils and geological profile in all GEO5 programs
- **\*.GI3** - CPT data in program **"Pile CPT"**



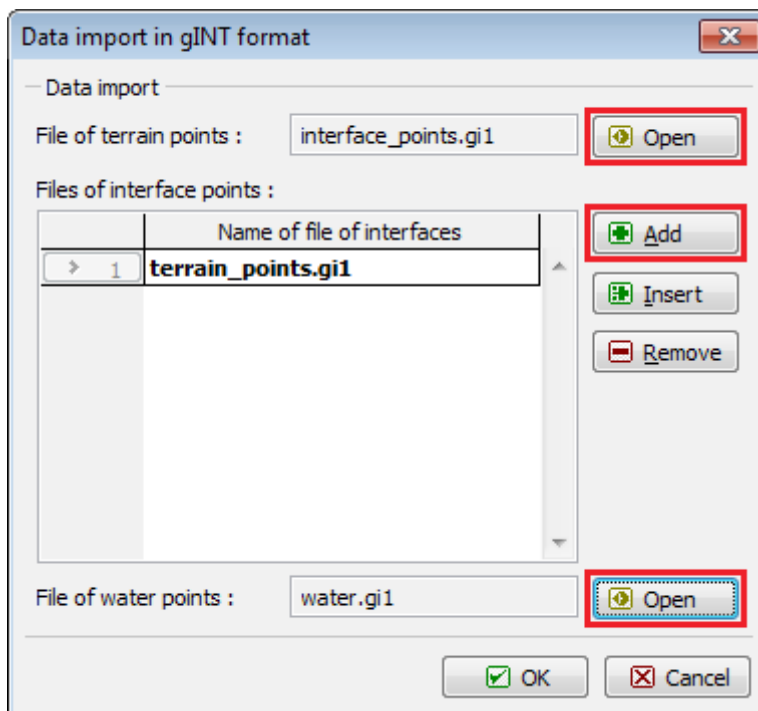
Menu "Import gINT"

## Import of Terrain Points

The program "Terrain" allows to import points of terrain surface. The gINT file is in this case written as \*.gi1. The main program menu (item "File") contains the "Import" - "Format GI1 (gINT interface points)" items. Using the "Open" button in the "Import of data in gINT format" dialog window loads the gINT file - import a set of points of terrain surface. The "Add" button lets you import arbitrary number of interfaces.

The "Open" button in the bottom part of the dialog window loads the "File of water points"

While importing, the program automatically transforms the format of units of the imported data into the one used in the GEO5 program.



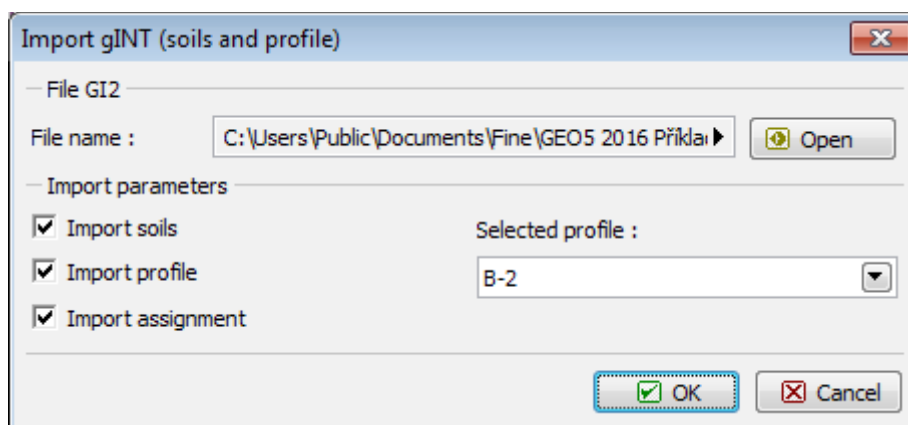
*"Import of data in gINT format" dialog window*

## Import of Soils and Profiles

The **"Open"** button in the **"Import gINT (soils and profile)"** dialog window loads the gINT file. The gINT file is in this case written as \*.gi2. The **"Selected profile"** combo list lets you to select the profile. The type of import (Import soils, Import profile, Import assignment) is then determined in the **"Import parameters"** sheet.

While importing, the program automatically transforms the format of the imported data into the one used in the program.

The program **"Terrain"** allows to import more profiles (boreholes) together (option **"All profiles"** in the combo list **"Selected profile"**). All imported profiles must have same count of layers. If profiles contain different order of assigned soils, then assignment is made according to the last profile.



*"Import gINT (soils and profile)" dialog window*

## Heredity - Construction Stage

Construction stages ([Tool bar Construction Stages](#)) allow to create the construction step by step and check all construction stages (this is necessary in programs **Sheeting Check**, **Settlement**, **FEM**).

For individual input types (soil assignment, anchors, supports...) there always exists relationship over [construction stages](#).

There are two types:

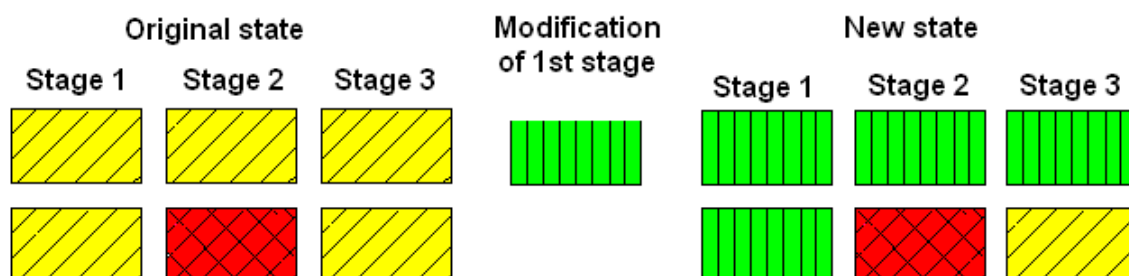
**Defined heredity** - (anchors, supports, surcharges...) - these objects always remember the stage, in which they were created. This is also the stage where these objects can be edited. In all subsequent stages these objects can be either removed or it is possible to change some of their properties (post-stressing anchor, change of surcharge magnitude, translating support...). When defining a new construction stage these objects are automatically carried over to that stage.

**Automatic heredity** - (assigning of soils, terrain profile, influence of water, analysis setting...) - for such types of inputs the properties from the previous stage are carried over to a new one if created. When changing properties in the current construction stage the program proceeds as follows:

- If the property in the next stage remains the same as in the previous stage it also receives

the tag new - this change also applies to all subsequent stages.

- If the property in the next stage differs from the one in the previous stage (this means that this property has been in the next stage already changed) then this change is not carried over to subsequent stages.



*Changes within stages of construction - automatic heredity*

## Standards and Analysis Methods

GEO5 allows setting of [standards and analysis methods](#) centrally for all GEO5 programs.

In all GEO5 programs, these parameters can be specified in the **frame "Settings"**, which enables the user to:

- select analysis parameters in the [Settings list](#)
- store and manage settings in the [Settings Administrator](#)
- create and manage new [user settings](#)

The program allows to perform the structure verification according to five methodologies:

- [Verification according to factor of safety](#)
- [Verification according to limit states](#)
- [Verification according to EN 1997](#)
- [Verification according to LRFD](#)
- [Verification according to the Chinese standards](#)

The programs allows to define [design situations](#) (for different [construction stages](#)), which may differ by the partial factors.

## Administrator

The Settings Administrator is the main tool for managing individual ["Settings"](#). In particular, it enables the user to:

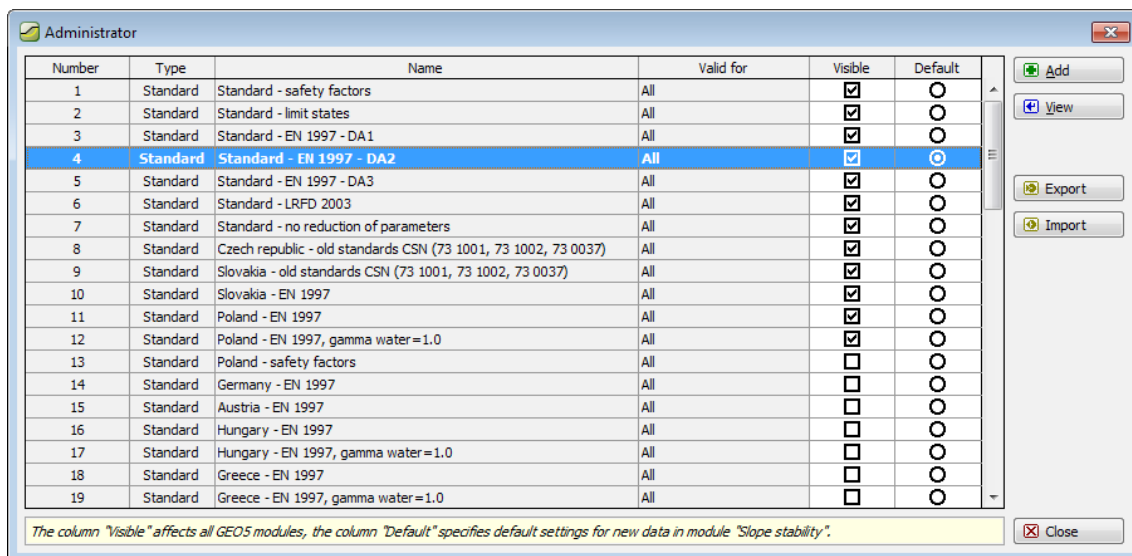
- **determine the visibility of Settings** in the ["Settings list"](#) (it is determined by checking the box in the **"Visible"** column)
- **specify the default Settings** for new data files of the current program (the **"Default"** column)



- **view basic Settings** which are currently selected (available by pressing the **"View"** button)
- **add user Settings** (pressing the **"Add"** button opens **"New settings"** dialog window with a copy of the currently selected Setting)
- **edit input user Settings** (by pressing the **"Edit"** button)
- **delete user Settings** (by pressing the **"Remove"** button)

In addition, the Settings Administrator enables **exporting and importing** Settings stored on the disk.

Visibility and default settings are switched using the mouse or pressing the Space key (visibility) or Shift+Space (default settings).

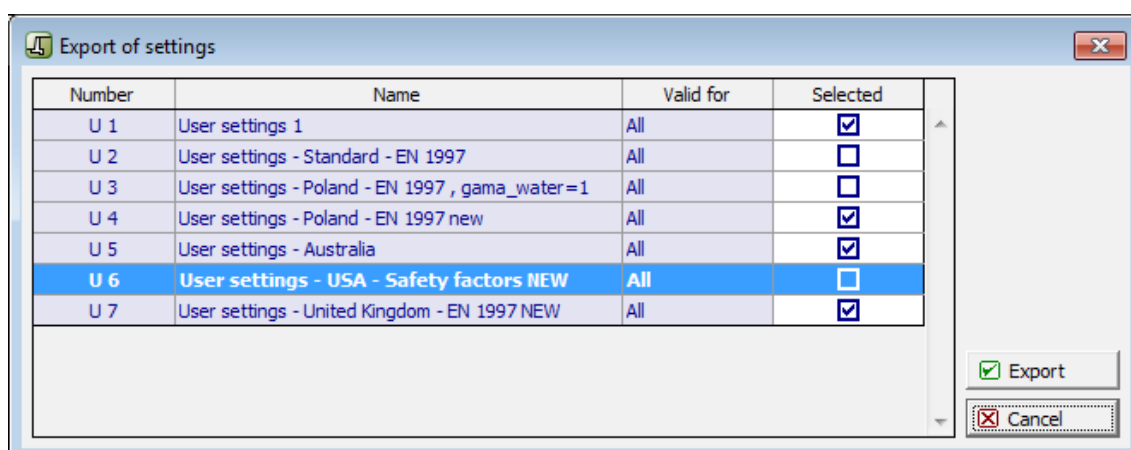


"Settings Administrator" dialog window

## Import and Export of Settings

The selected user settings in the **"Settings Administrator"** can be saved into a file (**"Export"**) in formats **\*.g5c** and **\*.xlm** and subsequently loaded (**"Import"**) on a different computer that has GEO5 programs installed. It enables sharing analysis settings between several users, for example in companies that share more licenses distributed among several subdivisions.

These formats may be particularly useful when solving various problems with our hotline.



*Export (Import) of the selected "Settings" of analysis parameters*

## Settings List

The "**Settings list**" dialog window allows to choose the current "**Settings**", which will then drive both the calculation and verification analysis of the given task.

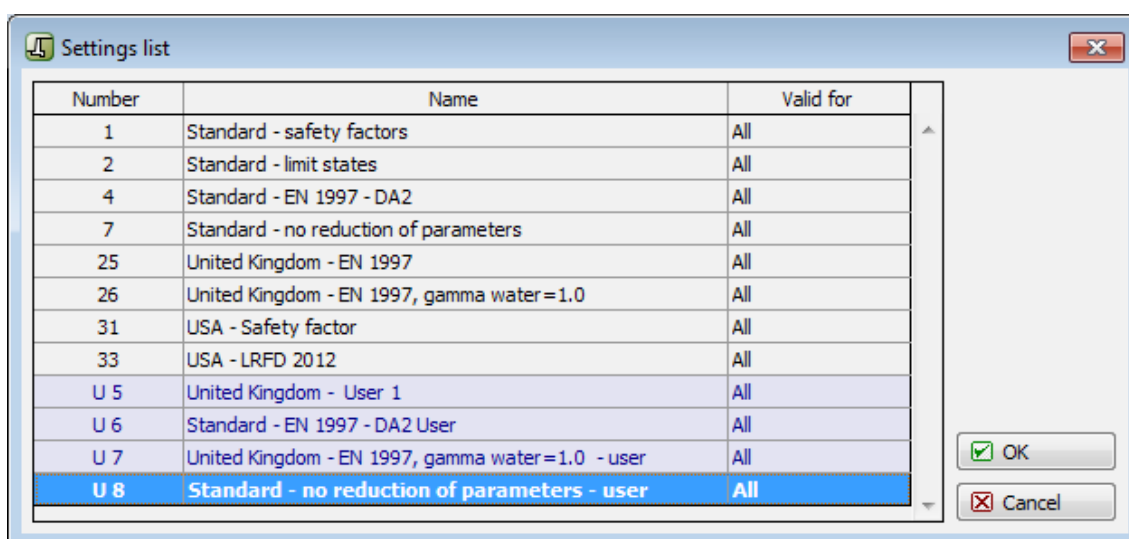
This list contains two types of settings:

- **basic**, which accompanies the software distribution and cannot be edited or deleted
- **user**, which is defined by the user

The list applies to all GEO5 programs, only some of the Settings can be restricted to a specific program.

For lucidity, only Settings, which are checked in the "**Settings Administrator**" as visible, are displayed. When running the program the first time, the Settings visibility is determined according to the country of destination. Subsequently, the program remembers the changes made by the user.

To work efficiently with the GEO5 programs, it is for most countries sufficient to create one or several specific "**Settings**". Then, for the **solution of individual tasks** the user just selects the **particular Setting**. The analysis methods, values of coefficients and the verification methodology then do not need to be specified. This results in a lucid and simpler work with the given program.



*"Settings list" dialog window*

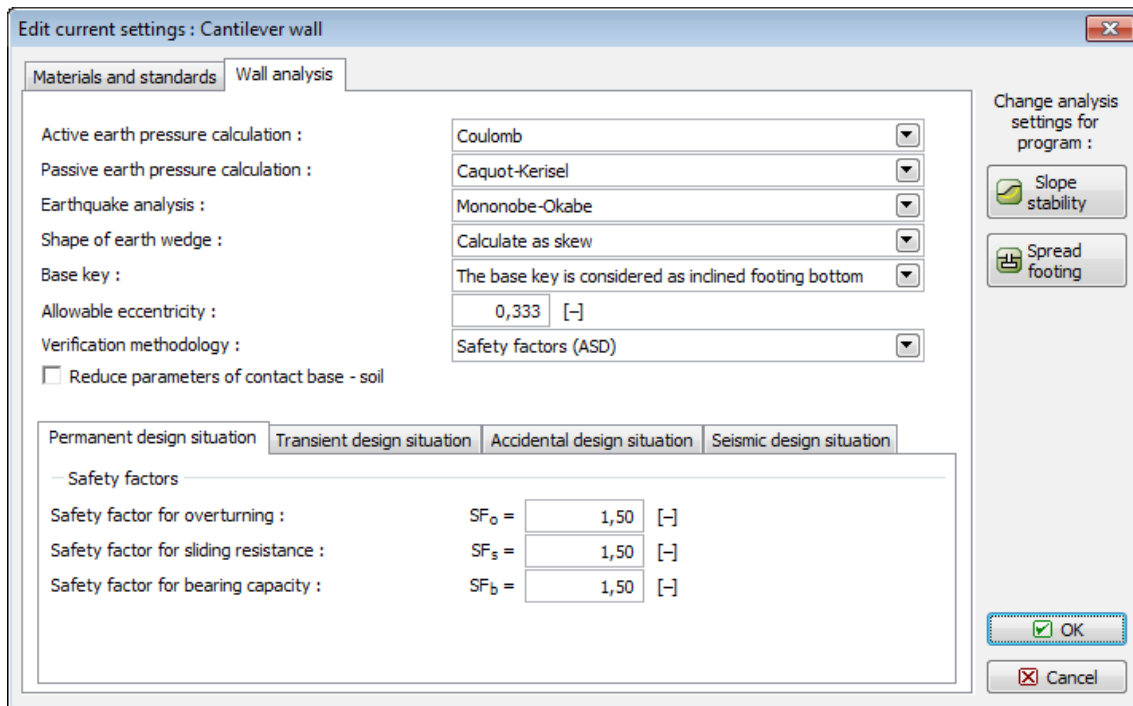
## Analysis Settings

An analysis setting is a set of data, which is a key for performing various calculations in the program. These include, in particular:

- methods and theories of the analyses
- **verification methodology**; the way of proving safety of the structure (**factor of safety, limit states, EN 1997, LRFD, chinese standards**)
- actual values of partial factors and degrees of safety for individual **design situations**

An analysis setting is typically the same for a large number of tasks - owing to this, the program enables creating a "**Settings list**". Individual settings can be edited, **exported and imported** in the "**Settings Administrator**".

A setting can be **valid** for **all GEO5 programs** or for **one selected program** only.

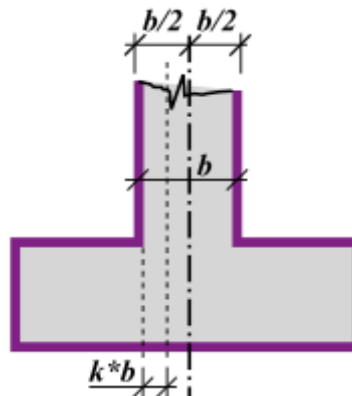


"Edit current settings" dialog window

## Materials and Standards

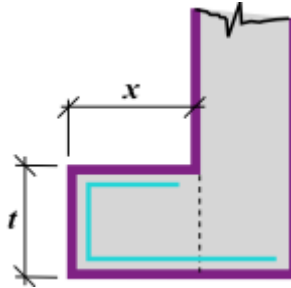
This tab allows to input materials and analysis standards for:

- **Concrete structures**
- Concrete structures of bridge abutments
- **EC 2 coefficients** are specified for the analysis of Concrete structures according to **EN 1992-1-1**. Both default and user-defined values can be adopted.
- **Parameter of cross-section determination** - this coefficient determines the location of the critical section for the calculation of bending moment within the wall foundation. Default value for concrete and steel-reinforced concrete columns is  $k = 0$ , i.e. the critical section is found at an exposed column face. For masonry structures, it is recommended to choose the value of this coefficient equal to 0,25.



Critical section for the maximum bending moment within the wall foundation

- Parameter of wall jump  $k$**  - this parameter determines whether the load-bearing reinforcement of the wall front jump is verified or not. For jump length  $x \leq k \cdot t$  the reinforcement verification is not performed. When choosing the option "**according to standard**" the wall short jump parameter is automatically back calculated according to the selected standard for **dimensioning of RC structures**. When choosing the option "**input parameter**" the value of parameter  $k$  is input by the user. The parameter of short jump of wall is considered for the **analysis of walls** and for the **analysis of spread footings**.

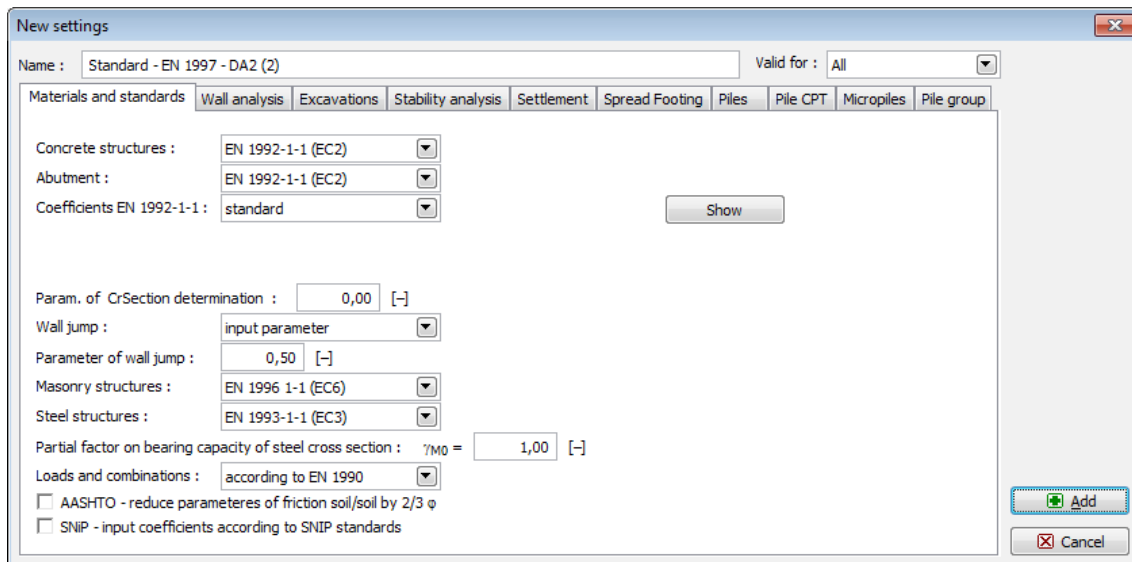


*Scheme for determining the parameter of short jump of wall*

- Masonry structures (defined in the "**Masonry Wall**" program only)
- Steel structures** (defined in "**Slab**", "**Sheeting Check**" and "**Anti-Slide Pile**" programs)
- Loads and combinations (defined in "**Slab**" and "**Beam**" programs only)
- AASHTO - enables the reduction of friction between the soil-soil interface **in the wall analysis** to the value of:

$$\delta = \frac{2}{3} \varphi$$

- SNiP - enables inputting design coefficients in the sense of Russian standards SNiP.



*"New settings" dialog window - "Materials and standards" tab*

## Wall Analysis

This tab allows to input parameters for the **wall analysis**:

- 
- Active earth pressure calculation (Caquot, Coulomb (ČSN 730037), Müller-Breslau (DIN 4085), Mazindrani (Rankine), Absi).
  - Passive earth pressure calculation (Caquot-Kerisel (ČSN 730037), Coulomb, Müller-Breslau, Sokolovski (DIN 4085), Mazindrani (Rankine), Absi).
  - Earthquake analysis (Mononobe-Okabe, Arango, JTJ 004-89, JTS 146-2012, SL 203-97).
  - Shape of earth wedge (Calculate as skew, Consider always vertical).
  - Base key (The base key is considered as front face resistance; the base key is considered as inclined footing bottom).
  - **Allowable eccentricity** - for assesment of contact stress at footing bottom is the value assumed of maximum allowable eccentricity in range of 0.1 to 0.4.
  - Internal stability - this way of calculation is adopted in "**MSE Wall**". Slip surface has a different shape (straight, broken) according to the selected standard of calculation.
  - Hinge Height Concept - represents the way of analysis of precast walls according to AASHTO, in which the favorably acting gravity force of a part of the structure is reduced. It is used only in "**Redi Rock Wall**" program. This program also allows for inputting the "**Coefficient of reduction of first block-base**".
  - In the case of verification methodology according to the limit states and factor of safety, it is possible to reduce the parameters of the foundation - soil interface. The coefficient of reduction of structure - soil interface  $\mu$  represents the amount of wall resistance against slip resp. against translation when in contact with the soil.
  - When running the verification analysis according to the theory of limit states, the program enables the reduction of the tangent of the angle of internal friction  $\varphi$  employing the coefficient  $\gamma_{m\varphi}$ .
  - Verification methodology (factor of safety, limit states, analysis according to EN 1997, analysis according to LRFD, analysis according to the chinese standards).
  - Design situations are specified for all verification methodologies.

The screenshot shows the 'New settings' dialog window with the 'Wall analysis' tab selected. The 'Name' field is 'Standard - EN 1997 - DA2 (2)' and 'Valid for' is 'All'. The 'Excavations' tab is active, showing parameters for earth pressure calculations and safety factors. The 'Reduction coeff. of contact base - soil' parameter is highlighted with a red box, showing a value of 1,00 [-].

Parameter	Value	Unit
Active earth pressure calculation :	Coulomb	
Passive earth pressure calculation :	Caquot-Kerisel	
Earthquake analysis :	Mononobe-Okabe	
Shape of earth wedge :	Calculate as skew	
Base key :	The base key is considered as inclined footing bottom	
Allowable eccentricity :	0,333	[-]
Internal stability :	Standard - straight slip surface	
<input type="checkbox"/> Hinge Height Concept		
Reduction coeff. of contact first block - base :	1,00	[-]
Verification methodology :	Safety factors (ASD)	
<input checked="" type="checkbox"/> Reduce parameters of contact base - soil		
<b>Safety factors</b>		
Safety factor for overturning :	SF <sub>o</sub> = 1,50	[-]
Safety factor for sliding resistance :	SF <sub>s</sub> = 1,50	[-]
Safety factor for bearing capacity :	SF <sub>b</sub> = 1,50	[-]
Safety factor for mesh strength :	SF <sub>n</sub> = 1,50	[-]
Safety factor for sliding along geo-reinforcement :	SF <sub>sr</sub> = 1,50	[-]
Safety factor for geo-reinforcement strength :	SF <sub>st</sub> = 1,50	[-]
Safety factor for pull out resistance of geo-reinf. :	SF <sub>po</sub> = 1,50	[-]
Safety factor for connection strength :	SF <sub>con</sub> = 1,50	[-]
<b>Reduction coefficients</b>		
Reduction coeff. of friction between blocks :	$\gamma_f = 1,50$	[-]
Reduction coeff. of contact base - soil :	$\mu = 1,00$	[-]

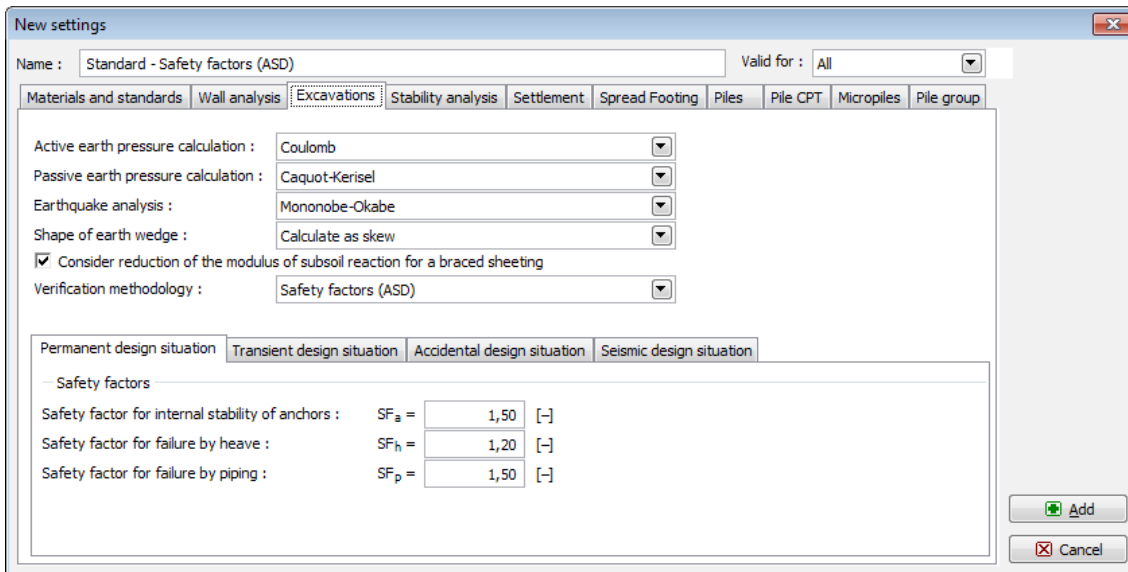
"New settings" dialog window - "Wall analysis" tab

## Excavations

This tab allows to input parameters for the analysis of excavations and earth pressures:

- Active earth pressure calculation (Caquot, Coulomb (ČSN 730037), Müller-Breslau (DIN 4085), Mazindrani (Rankine), Absi)
- Passive earth pressure calculation (Caquot-Kerisel (ČSN 730037), Coulomb, Müller-Breslau, Sokolovski (DIN 4085), Mazindrani (Rankine), Absi)
- Method of calculation (dependent pressures, JGJ 120-2012)
- Earthquake analysis (Mononobe-Okabe, Arango, JTJ 004-89, JTS 146-2012, SL 203-97)
- Modulus of subsoil reaction (standard, input, pressiometer PMT, dilatometer DMT, CUR 166, chinese standards). Standard settings contains recommended international methods for calculation of modulus of subsoil reaction (for **Sheeting Check** program in the frame "Modulus Kh") - other methods described herein are used only in specific countries.
- Shape of earth wedge (Calculate as skew, Consider always vertical)
- Consider reduction of the modulus of subsoil reaction for a braced sheeting - this option is used only in "**Sheeting Check**" and "**Anti-Slide Pile**" programs, when program during the analysis reduces values of the modulus of subsoil reaction automatically.
- Verification methodology (factor of safety, limit states, analysis according to EN 1997, analysis according to LRFD)

- **Design situations** are specified for all verification methodologies.

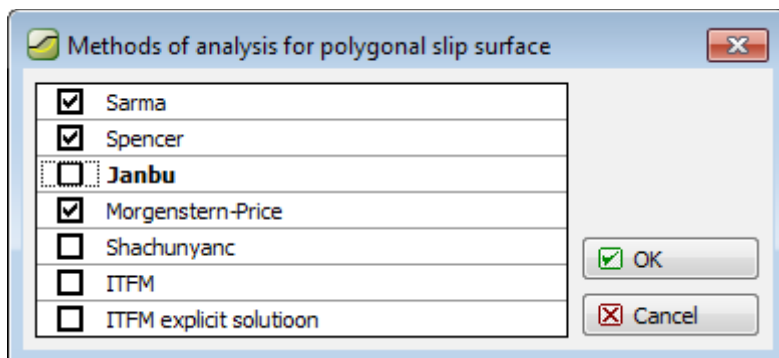


"New settings" dialog window - "Earth pressures" tab

## Stability Analysis

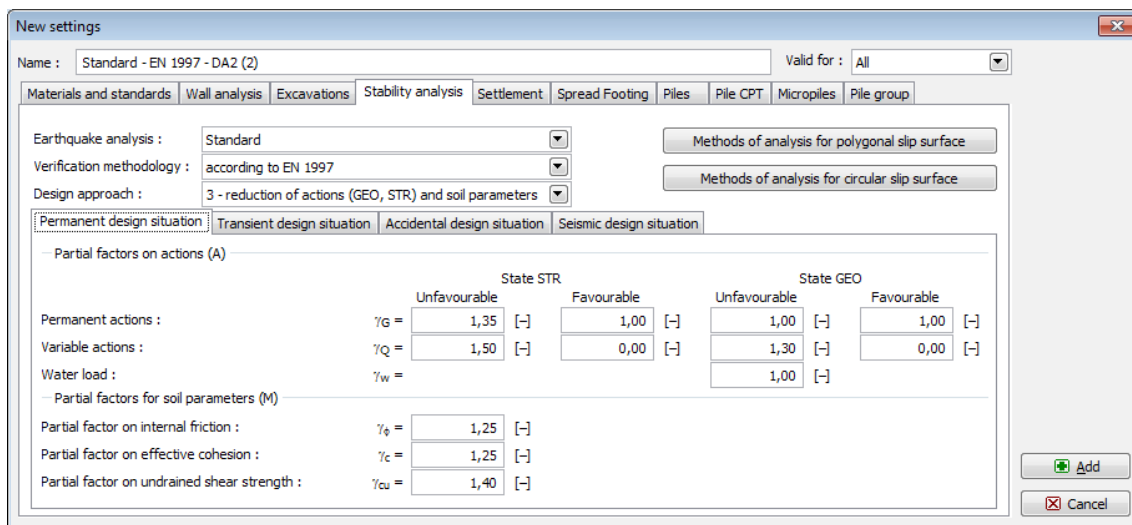
The tab allows to input parameters for **stability analysis**:

- Earthquake analysis (**Standard**, **JTJ 004-89**, **SL 203-97**)
- **Verification methodology** (factor of safety, limit states, analysis according to **EN 1997**, analysis according to **LRFD 2012**, analysis according to Chinese standards)
- **Design situations** are specified for all verification methodologies
- The "**Methods of analysis for polygonal slip surface**" and "**Methods of analysis for circular slip surface**" buttons opens a dialog window that allows to select the analysis method. The program allows to calculate for the selected slip surface (**polygonal**, **circular**) all analysis methods. However, some of them are very exotic and known only at the country of their origin. Thus the methods, the user is not interceded in, can be turned off.



Dialog window - "Analysis methods for polygonal slip surface" - selecting the method of analysis





"New settings" dialog window - "Stability analyses" tab

## Settlement

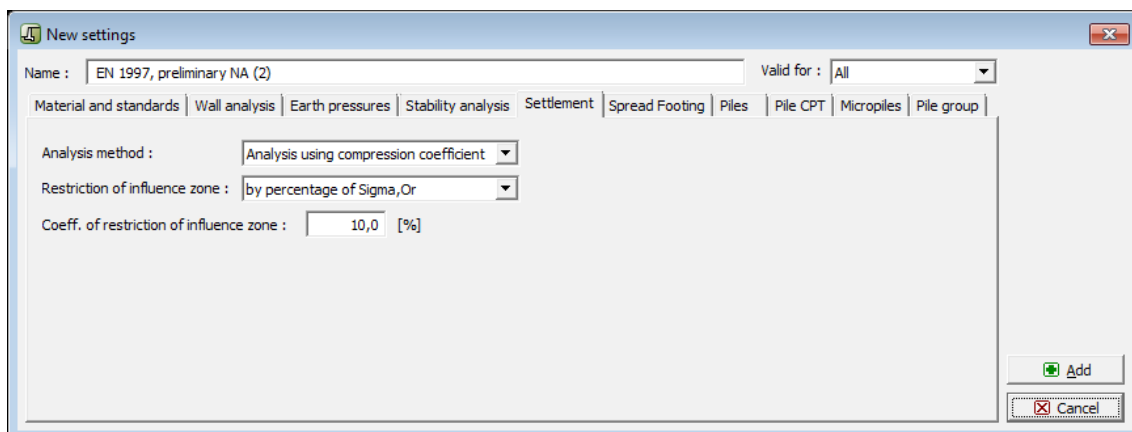
This tab allows to input parameters for the **settlement analysis**:

Analysis methods:

- CSN 73 1001 (Analysis using oedometric modulus)
- Analysis using compression coefficient
- Analysis using compression index
- NEN (Buismann, Ladd)
- Soft soil model
- Janbu's theory
- Analysis using constrained modulus

Restriction of the influence zone:

- based on structural strength
- by percentage of  $\sigma_{or}$  (The coefficient to bound the influence zone is input in [%])



"New settings" dialog window - "Settlement" tab

## Spread Footing

This tab allows to input parameters for the [analysis of bearing capacity of foundation](#):

### Analysis for drained conditions:

- [standard approach](#)
- CSN 73 1001
- PN-81B-03020
- IS:6403-1981
- EC 7-1 (EN 1997-1:2003)
- NCMA
- GB 50007-2002
- SNiP 2.02.01-83
- DS/EN 1997-1 DK NA:2013

### Analysis for undrained conditions:

- [standard approach](#)
- CSN 73 1001
- IS:6403-1981
- EC 7-1 (EN 1997-1:2003)
- DS/EN 1997-1 DK NA:2013

### Analysis of spread footing on rock subsoil:

- [standard approach](#)
- CSN 73 1001
- EC 7-1 (EN 1997-1:2003)

### Analysis of uplift

- [standard approach](#)
- [cone method](#)
- DL/T 5219-2005

**Allowable eccentricity** - for [assessment of the eccentricity of foundation](#) is assumed the value of maximum allowable eccentricity in range of 0.1 to 0.4.

[Verification methodology](#) ([factor of safety](#), [limit states](#), [analysis according to EN 1997](#), [analysis according to LRFD](#), [analysis according to Chinese standards](#)).

[Design situations](#) are specified for all verification methodologies.

The screenshot shows the 'New settings' dialog window with the 'Spread Footing' tab selected. The 'Name' field is 'Standard - EN 1997 - DA2 (2)' and 'Valid for' is 'All'. The 'Materials and standards' tab is active, showing the following settings:

- Analysis for drained conditions : EC 7-1 (EN 1997-1:2003)
- Analysis for undrained conditions : EC 7-1 (EN 1997-1:2003)
- Analysis of spread footing on rock subgrade : EC 7-1 (EN 1997-1:2003)
- Analysis of uplift : Standard
- Allowable eccentricity : 0,333 [-]
- Verification methodology : according to EN 1997
- Design approach : 1 - reduction of actions and soil parameters

The 'Permanent design situation' tab is selected, showing the following parameters:

Permanent actions :	Combination 1		Combination 2	
	Unfavourable	Favourable	Unfavourable	Favourable
Partial factors on actions (A)	$\gamma_G = 1,35$ [-]	$1,00$ [-]	$1,00$ [-]	$1,00$ [-]
Partial factors for soil parameters (M)				
Partial factor on internal friction :	$\gamma_\phi = 1,00$ [-]		$1,25$ [-]	
Partial factor on effective cohesion :	$\gamma_c = 1,00$ [-]		$1,25$ [-]	
Partial factor on undrained shear strength :	$\gamma_{cu} = 1,00$ [-]		$1,40$ [-]	
Partial factor on unconfined strength :	$\gamma_v = 1,00$ [-]		$1,40$ [-]	

Buttons for 'Add' and 'Cancel' are visible at the bottom right.

"New settings" dialog window - "Spread Footing" tab

## Pile

This tab allows to input parameters for the **analysis of pile**:

### Analysis for drained conditions:

- CSN 73 1002
- Effective stress
- NAVFAC DM 7.2

### Analysis for undrained conditions:

- Tomlinson
- NAVFAC DM 7.2

### Load-settlement curve:

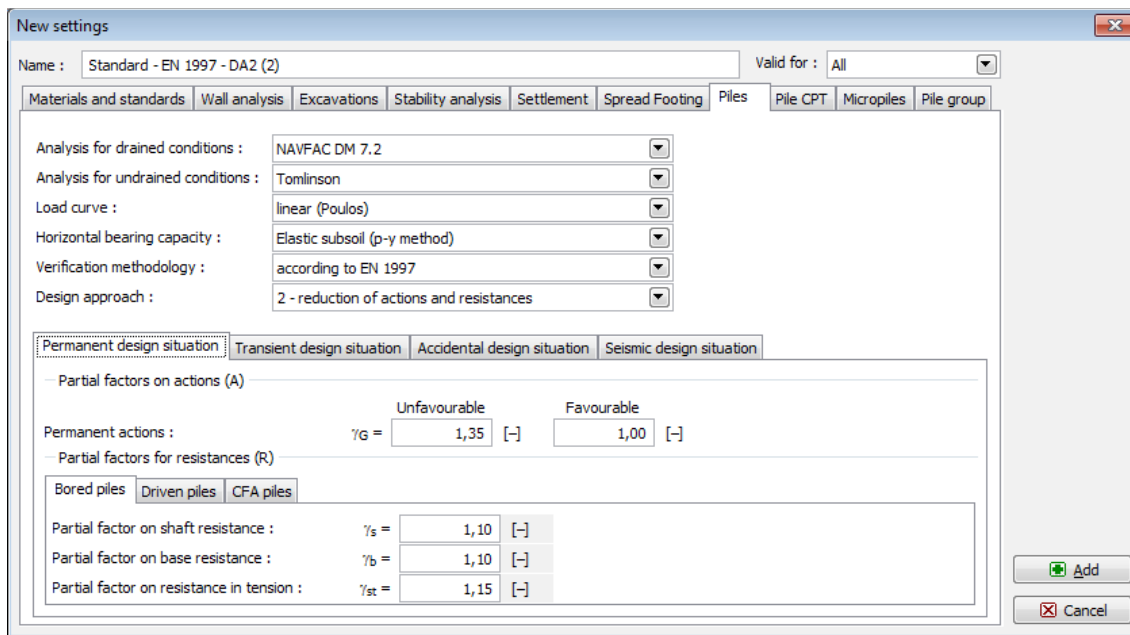
- nonlinear (Masopust)
- linear (Poulos)

### Horizontal bearing capacity:

- Elastic subsoil (p-y method)
- Broms method

Verification methodology (factor of safety, limit states, analysis according to EN 1997).

Design situations are specified for all verification methodologies.



"New settings" dialog window - "Pile" tab

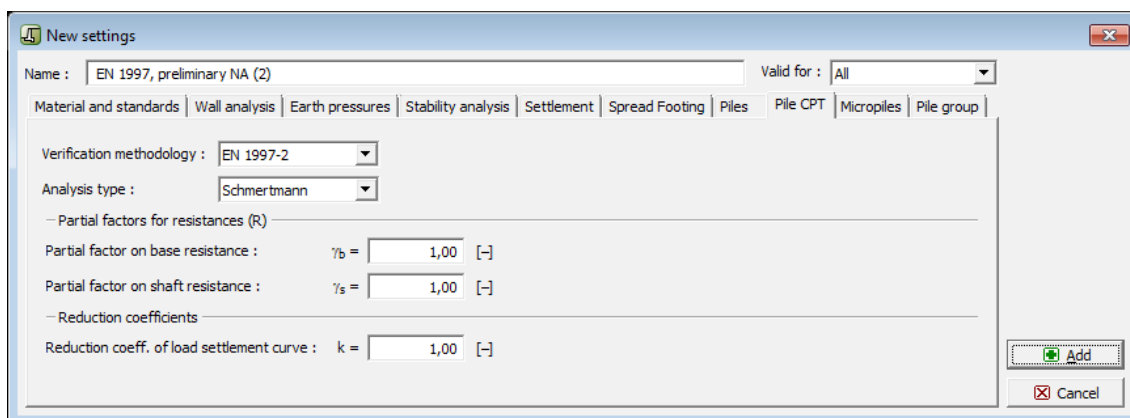
## Pile CPT

This tab allows to input parameters for the analysis of pile CPT:

**Verification methodology** (factor of safety, limit states, NEN 6743, EN 1997-2).

**Analysis type:**

- EN 1997-2
- NEN 6743
- LCPC (Bustamante)
- Schmertmann



"New settings" dialog window - "Pile CPT" tab

## Micropiles

This tab allows to input parameters for the analysis of micropiles:

**Calculation of stem bearing capacity:**

- geometric method (Euler)
- Salas theory
- Véas-Souche theory

**Calculation of root bearing capacity:**

- Lizzi theory
- Littlejohn theory
- Bowles theory
- Zweck theory
- Véas theory
- root in the rock
- Bustamante (SPT, Pressiometer PMT)

Verification methodology (factor of safety, limit states).

Design situations are specified for all verification methodologies.

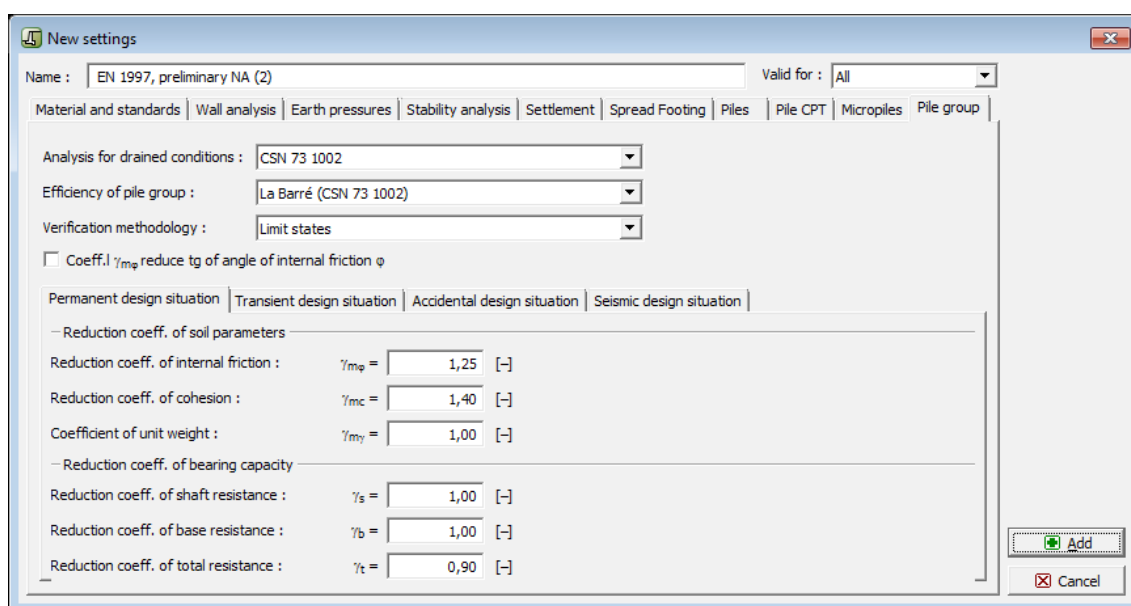
The screenshot shows the 'New settings' dialog window with the 'Micropiles' tab selected. The 'Name' field contains 'EN 1997, preliminary NA (2)' and 'Valid for' is set to 'All'. The 'Verification of stem bearing capacity' is set to 'geometric method (Euler)', 'Verification of root bearing capacity' is set to 'Lizzi theory', and 'Verification methodology' is set to 'Limit states'. Under 'Reduction coeff. of soil parameters', the following values are entered:  $\gamma_{mp} = 1,25$  [-],  $\gamma_{mc} = 1,40$  [-],  $\gamma_{mf} = 1,00$  [-],  $\gamma_{sc} = 1,50$  [-],  $\gamma_{ss} = 1,50$  [-], and  $\gamma_r = 1,50$  [-]. The 'Add' and 'Cancel' buttons are visible at the bottom right.

*"New settings" dialog window - "Micropiles" tab*

## Pile Group

This tab allows to input parameters for the analysis of group of piles:

- Analysis for drained conditions: CSN 73 1002, Effective stress, NAVFAC DM 7.2
- Efficiency of pile group: UFC 3-220-01A, La Barré (CSN 73 1002), Seiler-Keeney, input efficiency
- Verification methodology (factor of safety, limit states, analysis according to EN 1997).
- Design situations are specified for all verification methodologies.



"New settings" dialog window - "Pile group" tab

## Adding New Settings

The program contains a relatively large number of **basic Settings** applicable to individual countries and theoretical approaches. Despite that, it is quite probable that most users will require to modify it and create their own **user Settings**.

A setting can be valid for all GEO5 programs or for the current program only (this can be specified in the right upper corner of the dialog window).

Pressing the **"Add"** button opens a dialog window, which displays the current **setting of the program**:

- if the **"Input for the current task"** is the current setting, the window is opened in the regime pertinent to the current program
- if the current setting is selected from the **Settings list**, a copy of this Setting with the same validity is opened

After editing and specifying the name of the **new Setting**, this Setting is saved into the **"Settings Administrator"** by pressing the **"Add"** button so it can be subsequently selected from the **"Settings list"**.

User-defined Settings is reasonable to create for example:

### 1) based on countries and standards

- settings for Boguto
- settings for Borito
- settings for pro Borito, bridge structures

### 2) based on investor

- settings for highways
- settings for railways
- settings for buildings

### 3) based on analysis methods

- analysis based on Mazindrani
- analysis based on Coulomb

### 4) individually

- my way
- Peter's way

The goal is to create a "Settings list" so the user does not need to care for the way of inputting various types of analysis and accompanied coefficients. The created settings can be "Exported" and made available to other users. Providing such Settings have a broader validity the company FINE will implement them into the pre-defined Settings so that they become available to all users of the GEO5 software.

## Import of Older Data

Older data of GEO5 programs are automatically transformed into the new format of Settings, after import to version 15. The import is performed in the following steps:

- Old data with their program settings are loaded.
- The loaded data is compared with the Settings visible in the "Settings list". If the data is compatible with one of the Setting in the list, this Setting is adopted as current and the import is completed.
- The loaded data is compared with all Settings in the "Settings Administrator". If the data is compatible with one of the Setting in the list, this Setting is adopted as current and made visible.
- If the program does not find any compatible setting, the setting of loaded data is stored as "Input for current task". In such a case we recommend to create a new user-defined Setting from these data. This ensures that the setting will be assigned to other imported files.

In most cases the program loads the data such that the results are identical with the results of the original version. Nevertheless, several exceptions exist for which the compatibility could not be achieved.

## Basic Changes in Settings Between Version 15 and Older Versions

Upon load a file from an older version, the settings remain almost always the same as specified in the previous versions. The settings are not compatible in the following cases:

1. In the analysis according to the factor of safety or the theory of limit states when different coefficients were specified in different stages of construction. In such case, the program adopts settings from the first stage only and assigns them to the permanent and transient design situations. Settings from the other stages are not loaded and thus it is necessary to input them into corresponding design situations manually.
2. In the program "Nailed Slope", when performing the verification analysis according to the theory of limit states, the coefficient reducing the self-weight of soil behind the structure is different from one. This coefficient was removed in version 15. To maintain the same results, it is necessary to input a modified weight of soil in the frame "Soils".

3. Programs "**Earth Pressures**", "**Sheeting Design**" and "**Sheeting Check**", analysis based on EN 1997, **design approach 2**, we introduced a new factor, **Partial factor reducing the soil resistance** ( $\gamma_{Re}$ ), which reduces the magnitude of the passive earth pressure. The default value of this coefficient is equal to 1.4. To maintain the same results, it is necessary to input its value in the "**Excavations**" tab equal to 1.0.

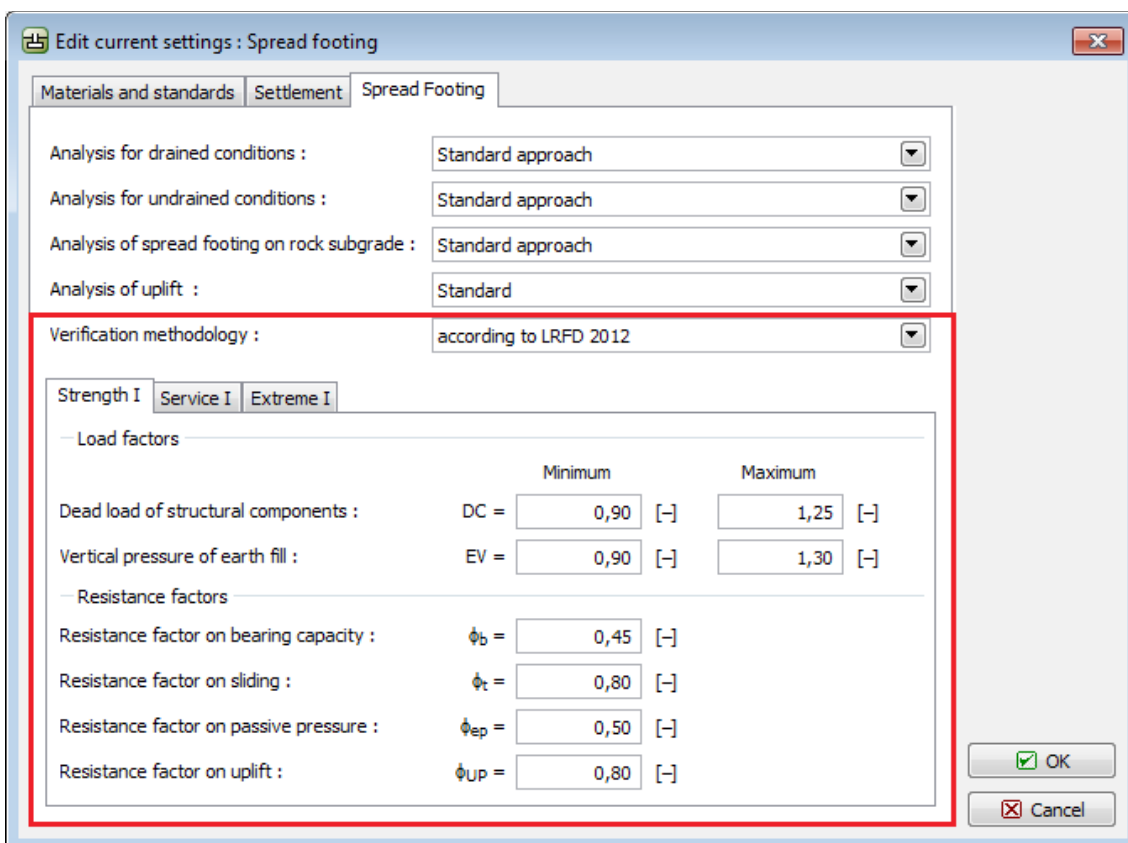
## Verification Methodology

The program allows to perform the structure verification according to these methodologies:

- Verification according to factor of safety
- Verification according to limit states
- Verification according to EN 1997
- Verification according to LRFD
- Verification according to the Chinese standards

Specific calculations (e.g. pressure calculation, determination of bearing capacity of foundation soil) are the same for all verification methodologies - they differ only in the way of introducing the design coefficients, combinations and the procedure for verifying the structure safety.

Verification methodology can be selected in dialog wondow "Settings".



Selection of verification methodology



## Analysis According to the Safety Factor (ASD)

The verification methodology based on the "**Safety factor**" is historically the oldest and most widely used approach for structure safety verification. The principal advantage is its simplicity and lucidity.

In general, the safety is proved using the safety factor:

$$FS = \frac{X_{pas}}{X_{act}} > FS_{req}$$

where:  $FS$  - Computed safety factor  
 $X_{pas}$  - A variable resisting the failure (resisting force, strength, capacity)  
 $X_{act}$  - A variable the causing failure (sliding force, stress)  
 $FS_{req}$  - Required factor of safety  
 $q$

When performing the analysis using the "**Safety factor**", neither the load nor the soil parameters are reduced by any of the design coefficients.

Detailed description for individual programs and types of structures can be found in the following chapters ([Walls and retaining structures](#), [Slope Stability](#), [Spread Footing](#), [Pile](#), [Rock Stability](#), [Micropile](#), [Pile CPT](#), [Pile Group](#)).

## Analysis According to the Theory of Limit States (LSD)

The verification methodology based on the theory of "**Limit states**" proves the safety by comparing a resisting variable (resisting force, strength, bearing capacity) and a variable causing failure (sliding force, stress).

$$X_{pas} > X_{act}$$

where:  $X_{pas}$  - A variable resisting the failure (resisting force, strength, capacity)  
 $X_{act}$  - A variable causing the failure (sliding force, stress)

$X_{act}$  is in general determined from the design parameters of soils and load:

- soil parameters are reduced by corresponding coefficients
- load (its action) is increased by corresponding coefficients

$X_{pas}$  is determined based on the following assumptions:

- soil parameters are reduced by corresponding coefficients
- the calculated structure resistance is reduced by a corresponding coefficient

In general, it can be stated that the verification based on "**Limit states**" is more modern and apt approach in comparing to the "[Safety factor](#)". However, it is less lucid.

Modern standars used for verification of structure safety ([EN 1997](#), [LRFD](#)) arise from the concept of limit states. In addition, they introduce various values for the coefficients of partial factors for favorably and unfavorably acting loads.

Detailed description for individual programs and types of structures can be found in the following chapters ([Walls and retaining structures](#), [Slope Stability](#), [Spread Footing](#), [Pile](#), [Rock Stability](#), [Micropile](#), [Pile CPT](#), [Pile Group](#)).

## Verification According to EN 1997

Designing a structure according to EN 1997-1 essentially follows the analysis of [limit states](#).

[Partial factors](#), adjusting the characteristic values of load, material and resistance, are introduced into the analysis depending on the selected "[Design approach](#)".

Partial factors are identical for all analyses in a given program. However, a "[Design situation](#)" can be selected for individual stages.

The programs can be grouped into several categories based on the selected approach:

- [Analysis of walls, supporting structures](#) (Walls, Abutment, Nailed Slope)
- [Analysis of sheeting structures](#) (Sheeting Design, Sheetting Check, Earth Pressures)
- [Foundation analysis](#) (Spread Footing, Pile)
- [Slope stability analysis](#)

GEO5 programs support following **National Annexes**:

Finland, France, Poland, Germany, Slovakia, Austria, Singapore, Denmark, Belgium, Netherland, United Kingdom, Greece, Hungary, Bulgaria, Slovenia, Italy, Portugal.

Desired National Annex can be selected in [Settings List](#).

## Partial Factors

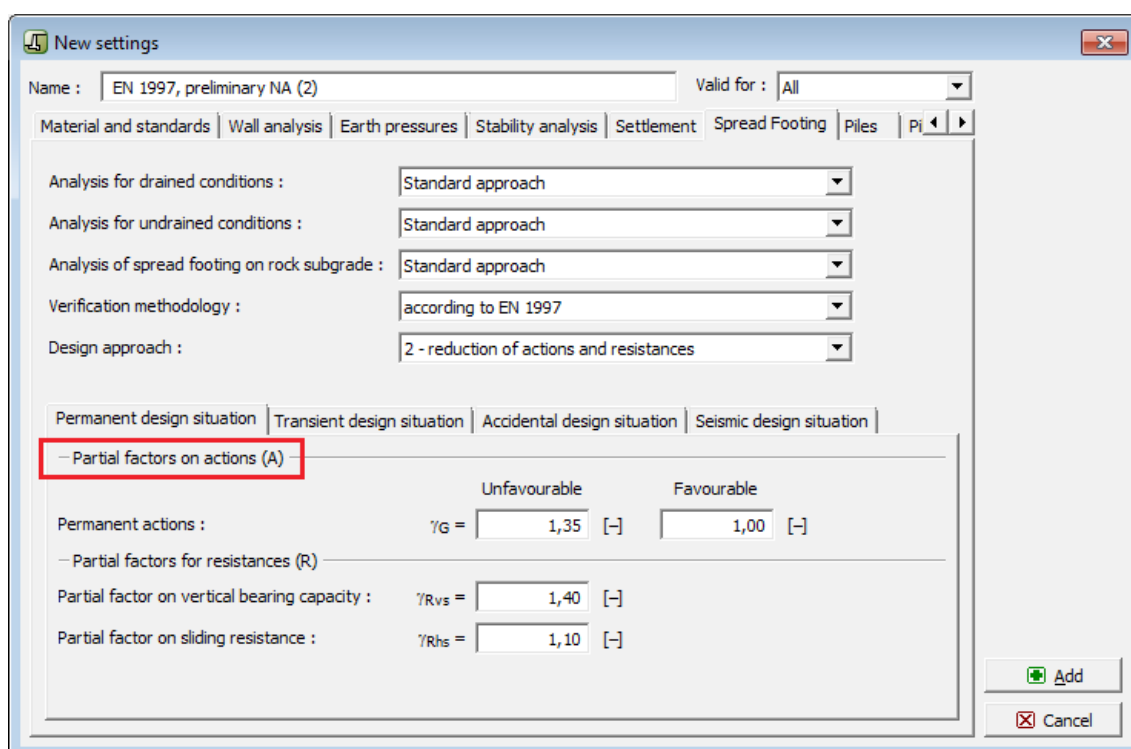
The "**Settings**" dialog window allows to input the partial factors for the analysis based on EN 1997.

The "**Design approach**" combo list allows to select one of the three "[Design approaches](#)". Depending on the selected design approach, the dialog window displays the **partial factors on actions, material or resistance** and [coefficients of combination](#) for variable load actions.

The section for inputting partial factors on actions also enables to input partial factors reducing the [action of water](#).

The "[Settings administrator](#)" and the "[Settings list](#)" contain a large number of pre-defined settings for individual **countries EU - settings EN 1997** according to selected [national annexes](#) (NA). In most countries, only one Design approach is then specified depending on NAD and used program (type of geotechnical task) - several pre-defined settings are available only for some countries.

The program enables to input each set of parameters four times - for individual [design situations](#). The program then adopts the coefficients based on the design situation set in the frame "**Stage settings**".



"New settings" dialog window - input of partial factors for the analysis based on EN 1997

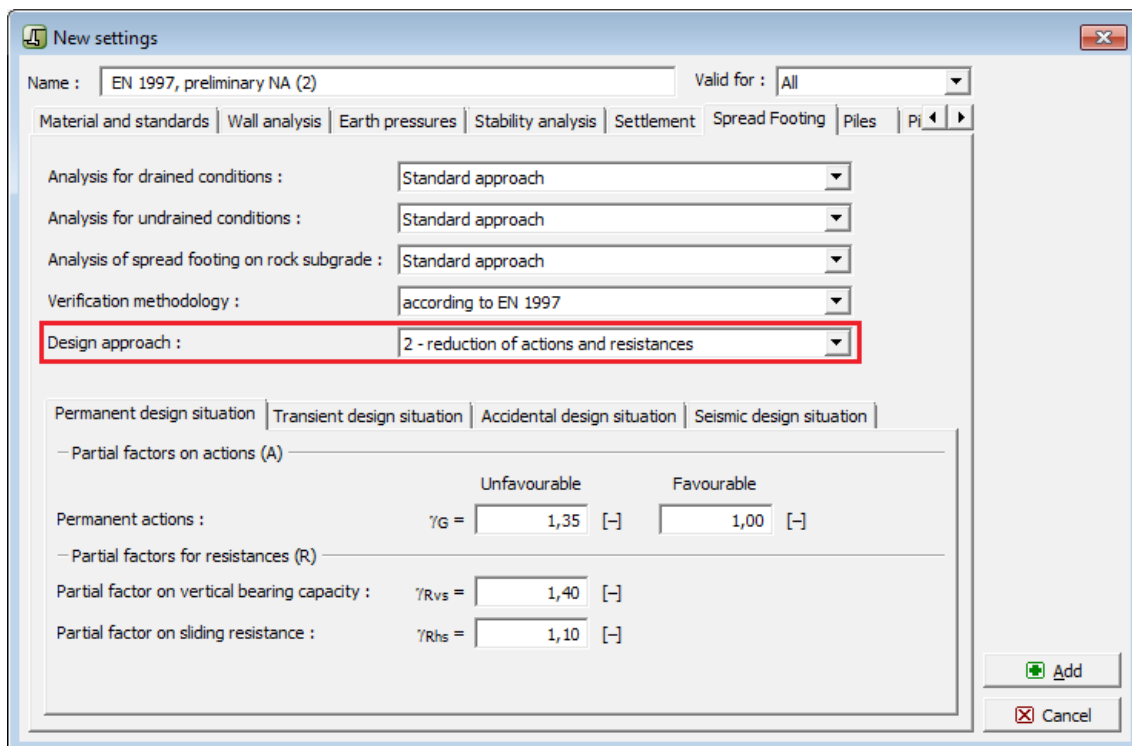
## Design Approaches

EN 1997-1 introduces three **design approaches** into the analysis; they differ by the application of partial factors.

According to EN 1997-1 the partial factors are generally applied to load actions, their impact on properties of foundation soil  $M$ , resistance  $R$  or both. The values of partial factors not only differ by the assumed **design approach**, but also by the type of the analyzed geotechnical task (support structures, piles, etc.). The values of partial factors are in general specified by the Eurocode in **Annexes A**; the national choice of values of partial factors specifies **NA**. The program automatically displays the required coefficients depending on the selected design approach or on the selection of other parameters in the **setting**.

Regarding the fact that individual **Design approaches** introduce the partial factors into the analysis in a different way (e.g., partial factors on actions on a structure and the resulting structure resistance or actions and soil parameters) it is logical that the results attributed to these design approaches may also considerably differ. If the **National Annex** does not recommend a **Design approach** for a given geotechnical task, it is up to the designer to select it (and therefore also to evaluate whether the results correspond to the analyzed situation).

- **Design approach 1** - Verification is performed for two sets of coefficients (**Combination 1 and Combination 2**) used in two separate analyses. Coefficients are applied to **load actions and to material parameters**.
- **Design approach 2** - Applies partial factors to **load actions and material resistance** (bearing capacity).
- **Design approach 3** - Applies partial factors to **load actions** and at the same time to **material** (material parameters of soil).



"New settings" dialog window, analysis based on EN1997 - Selection of design approach

## Design Approach 1

The verification analysis is performed for two sets of coefficients (**Combination 1** and **Combination 2**) used in two separate analyses. For **combination 1**, the partial factors are applied to **load actions only**, the remaining coefficients are set equal to 1.0. For **combination 2** the partial factors are applied to **material parameters (material parameters of soil) and to variable load actions**, the remain coefficients are set equal to 1.0.

In programs **analyzing walls** and performing **stability analyses** the analysis is carried out for **both combinations automatically** and the results are presented for the **most severe situation**. Detailed description of the results for both combinations is available in the output protocol.

This approach is not applicable for the "**Sheeting Check**" program. The combination, for which the analysis should be carried out, must be selected in the "**Excavations**" tab.

In "**Spread Footing**" and "**Pile**" programs, it is necessary to specify service load even for the bearing capacity analysis. The **design load** is adopted with combination 1, the **service load** then with combination 2.

Verification methodology :

Design approach :

Permanent design situation | Transient design situation | Accidental design situation | Seismic design situation

– Partial factors on actions (A) –

	Combination 1		Combination 2	
	Unfavourable	Favourable	Unfavourable	Favourable
Permanent actions : $\gamma_G =$	<input type="text" value="1,35"/> [-]	<input type="text" value="1,00"/> [-]	<input type="text" value="1,00"/> [-]	<input type="text" value="1,00"/> [-]

– Partial factors for soil parameters (M) –

	Combination 1	Combination 2
Partial factor on internal friction : $\gamma_\phi =$	<input type="text" value="1,00"/> [-]	<input type="text" value="1,25"/> [-]
Partial factor on effective cohesion : $\gamma_c =$	<input type="text" value="1,00"/> [-]	<input type="text" value="1,25"/> [-]
Partial factor on undrained shear strength : $\gamma_{cu} =$	<input type="text" value="1,00"/> [-]	<input type="text" value="1,40"/> [-]
Partial factor on unconfined strength : $\gamma_v =$	<input type="text" value="1,00"/> [-]	<input type="text" value="1,40"/> [-]

*Input of partial factors for design approach 1*

## Design Approach 2

Design approach 2 applies the partial factors to **load actions** and to **material resistance** (bearing capacity).

Verification methodology :

Design approach :

Permanent design situation | Transient design situation | Accidental design situation | Seismic design situation

– Partial factors on actions (A) –

	Unfavourable	Favourable
Permanent actions : $\gamma_G =$	<input type="text" value="1,35"/> [-]	<input type="text" value="1,00"/> [-]

– Partial factors for resistances (R) –

Partial factor on vertical bearing capacity : $\gamma_{Rvs} =$	<input type="text" value="1,40"/> [-]
Partial factor on sliding resistance : $\gamma_{Rhs} =$	<input type="text" value="1,10"/> [-]

*Input of partial factors for design approach 2*

## Design Approach 3

Design approach 3 applies the partial factors to **load actions** and at the same type to **material** (material parameters of soil).

Contrary to other design approaches, it distinguishes **geotechnical loads - State GEO** (load actions caused by soils - e.g. earth pressures, pressures due to surcharge, water action) and **loads applied to structures - State STR** (the program considers the self weight of a structure, input forces acting on the structure, anchors, geo-reinforcements, mesh overhangs). A different set of coefficients, specified in the "Partial factors" dialog window, is used for each type of load. Partial factors applied to geotechnical loads are mostly smaller than those applied to structure loads.

Verification methodology :

Design approach :

Permanent design situation | Transient design situation | Accidental design situation | Seismic design situation

– Partial factors on actions (A) –

	State STR				State GEO				
	Unfavourable		Favourable		Unfavourable		Favourable		
Permanent actions :	$\gamma_G =$	<input type="text" value="1,35"/>	<input type="text" value="[-]"/>	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>

– Partial factors for soil parameters (M) –

Partial factor on internal friction :  $\gamma_\phi =$   [-]

Partial factor on effective cohesion :  $\gamma_c =$   [-]

Partial factor on undrained shear strength :  $\gamma_{cu} =$   [-]

Partial factor on unconfined strength :  $\gamma_v =$   [-]

*Input of partial factors for design approach 3*

## National Annex (NA)

**National annex (NA)** offers details on the method of application of the Eurocode at a national level (in individual EU countries) and it was usually issued together with ENV of the given country.

The National Annex therefore determines the choice of **partial factors** at a national level and the application of **design approaches** for individual geotechnical tasks. Owing to the fact that the content of NA remains open in some member countries, national annexes are not implemented into individual programs for all member countries.

Individual national annexes can be selected from the pre-defined settings available in the **settings administrator** and **settings list**.

New **settings** can be **created** by users from the existing ones and in that way to define **own National annexes**.

## Partial Factors on Water

Partial factor on water action adjusts the force magnitude due to water action; the magnitude of pore pressure respectively.

Verification methodology :

Design approach :

Permanent design situation | **Transient design situation** | Accidental design situation | Seismic design situation

– Partial factors on actions (A) –

	Combination 1				Combination 2			
	Unfavourable		Favourable		Unfavourable		Favourable	
Permanent actions :	$\gamma_G =$	<input type="text" value="1,35"/>	<input type="text" value="[-]"/>	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value=""/>
Variable actions :	$\gamma_Q =$	<input type="text" value="1,50"/>	<input type="text" value="[-]"/>	<input type="text" value="0,00"/>	<input type="text" value="[-]"/>	<input type="text" value="1,30"/>	<input type="text" value="[-]"/>	<input type="text" value=""/>
<b>Water load :</b>	$\gamma_w =$	<input type="text" value="1,35"/>	<input type="text" value="[-]"/>	<input type="text" value=""/>	<input type="text" value=""/>	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value=""/>

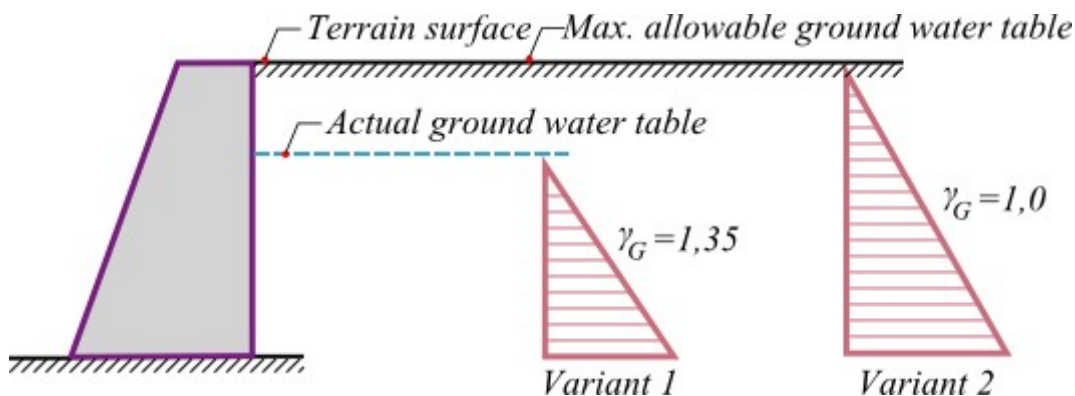
– Partial factors for soil parameters (M) –

	Combination 1		Combination 2		
Partial factor on internal friction :	$\gamma_\phi =$	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value="1,25"/>	<input type="text" value="[-]"/>
Partial factor on effective cohesion :	$\gamma_c =$	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value="1,25"/>	<input type="text" value="[-]"/>
Partial factor on undrained shear strength :	$\gamma_u =$	<input type="text" value="1,00"/>	<input type="text" value="[-]"/>	<input type="text" value="1,25"/>	<input type="text" value="[-]"/>

*Partial factors applied to action of water*

The partial factor on water action can be input, because EN 1997 offers several ways how to account for the influence of water. The two basic approaches are:

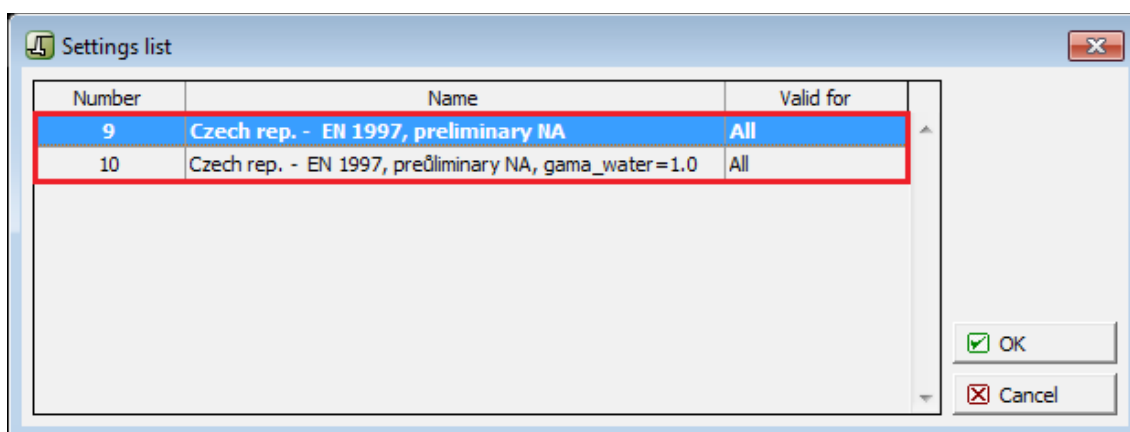
- **Variant 1** - the coefficient of water action is set to 1.3 or 1.35, respectively (some NA). In this case the actual ground water table is considered and its influence is multiplied by the input partial factor.
- **Variant 2** - the coefficient of water action is set to 1.0 or in other words, the action of water is not considered in the analysis. In this case the maximum allowable ground water table must be considered.



*Partial factors on water*

Selection of a particular option for the verification remains upon the user.

Providing the user adopts both options, we recommend introducing two settings in the "Settings administrator", which differ by the magnitude of coefficient  $\gamma_w$ .



Settings list - pre-setting for both variants of partial factors on water action

## Analysis of Walls (Support Structures)

Analysis based on EN 1997 introduces several **partial factors** according to selected **Design approach** (DA).

Designing a structure according to EN 1997-1 essentially follows the analysis of **limit states**.

### Load reduction (DA1, DA2, DA3):

All **design approaches** consider partial factors reducing load. These are used to multiply all forces entering the analysis. For actual verification of individual modes of failure the program determines, whether the **force or pressure acts favorably or unfavorably**. Depending on that these actions are then multiplied by the corresponding partial factor. Information regarding the applied partial factors is stored in the analysis protocol.

#### Forces acting on construction

Name	F <sub>hor</sub> [lbf/ft]	App.Pt. Z [ft]	F <sub>vert</sub> [lbf/ft]	App.Pt. X [ft]	Coeff. overtur.	Coeff. sliding	Coeff. stress
Weight - wall	0.0	-6.10	10005.0	6.01	1.000	1.000	1.350
FF resistance	-124.6	-0.67	0.0	0.00	1.000	1.000	1.000
Weight - earth wedge	0.0	-7.00	8243.4	8.64	1.000	1.000	1.350
Active pressure	9031.7	-6.49	12610.5	11.10	1.000	1.350	1.350
Force No. 1	2000.0	-23.00	0.0	5.10	1.350	1.350	1.350

Analysis protocol

The frame analysis allows to define "**Secondary variable actions**" - corresponding partial factors are then multiplied by the **combination coefficients** of load.

When analyzing supporting structures, the water actions and so the determination of the corresponding **partial factor for water** become very important.

### Reduction of material (DA1, DA3):

Soil parameters are automatically reduced by corresponding partial factors.

### Reduction of resistance (DA2):

Corresponding magnitudes of resistant forces, moments and bearing capacities are reduced.

When performing analysis according to the **Design approach 1**, all verifications are carried out twice for both combinations of load. For a given limit state the highest stressed design is displayed on the desktop.



## Analysis of Sheet Pile Structures

Analysis based on EN 1997 introduces several **partial factors** according to selected **Design approach** (DP).

Designing a structure according to EN 1997-1 essentially follows the analysis of **limit states**.

### Load reduction (DA1, DA2, DA3):

In programs that consider the overall earth pressure in the analysis (**Earth Pressures, Sheet Pile Design, Sheet Pile Check**) the partial factors are used to multiply individual components of pressure acting on a structure.

The basic assumption of the analysis is that the **active earth pressure acts unfavorably** whereas the **passive earth pressure** is considered as **favorable**. Individual pressure diagrams are therefore multiplied by the corresponding partial factor.

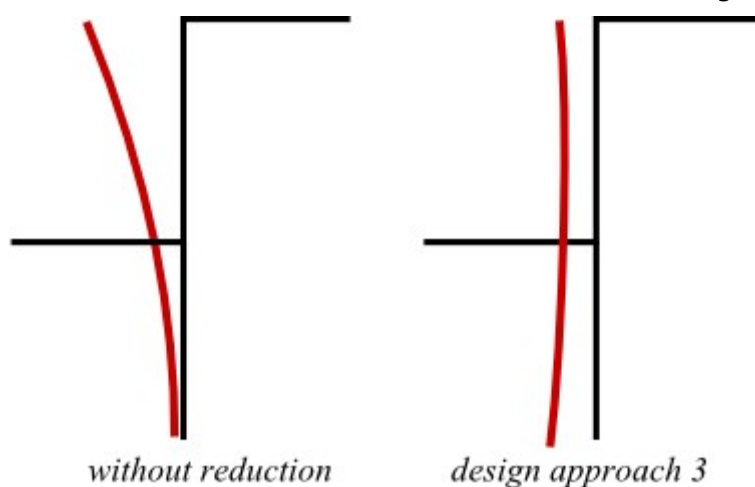
### Reduction of material (DA1, DA3):

Parameters of soils are automatically reduced by the corresponding partial factors.

**Reduction of resistance (DA2):** is considered. Partial factor on resistance reduce the passive earth pressure in front of structure.

In **simple words**, DA1 - Combination 2, DA2 and DA3 **increase the magnitude of active pressure** and **reduce the magnitude of passive pressure**, while DA1 - combination 1 only **increases the magnitude of active pressure**.

This approach may therefore **change in some cases the structure behavior** and deliver **misleading results**. Caution must therefore be exercised when reducing input parameters.



*Response of sheeting structure after excavation of soil*

## Analysis of Foundations (Spread Footing, Piles)

Analysis based on EN 1997 introduces several **partial factors** according to selected **Design approach** (DP).

Designing a structure according to EN 1997-1 essentially follows the analysis of **limit states**.

### Load reduction (DA1, DA2, DA3):

Load of foundation is taken as a result of analysis of the upper structure.

- load cases are determined according to rules provided by EN 1990:2002

- combinations of load cases are calculated according to EN 1991

The results of calculated combinations then serve as an input to "**Spread footing**" and "**Pile**" programs.

Either **design** (bearing capacity analysis, dimensioning of foundation) or **service** (analysis of settlement) **load** is considered. In **Design approach 1**, the analysis is performed for both the input design load (combination 1) and input service load (combination 2).

Only the **structure self-weight** or the **weight of soil above footing** is multiplied by the partial factors in the program. The specified design load must be determined in accord with the **EN 1990** and **EN 1991** standards - individual components of load must be **multiplied** by the corresponding partial factors - **the program does not change the input load any further**.

#### Reduction of material (DA1, DA3):

Parameters of soils are automatically reduced by the corresponding partial factors.

#### Reduction of load (DA2), for piles (DA1, DA2, DA3):

The program "**Pile**" assumes partial factors being dependent on the type of pile (**bored, driven, CFA**). The window allows to define all partial factors. The analysis then adopts partial factors depending on the type of pile selected in the frame "**Geometry**". Verification of the **tensile pile** always considers the pile self weight. For the **compressive pile** the pile self-weight can be neglected depending on the settings in the frame "**Load**". The **actual verification** analysis is performed according to the theory of limit states.

Vertical and horizontal bearing capacity of foundation is reduced in the "**Spread Footing**" program.

## Slope Stability Analysis

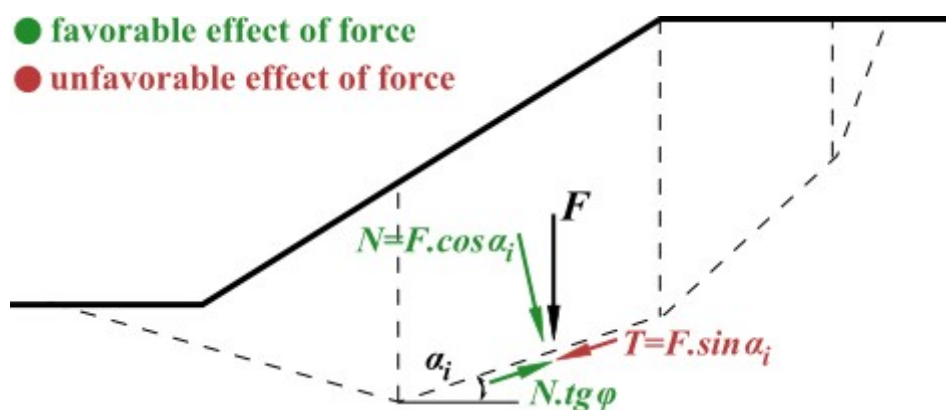
**Analysis based on EN 1997** introduces several **partial factors** according to selected **Design approach** (DP).

Designing a structure according to EN 1997-1 essentially follows the analysis of **limit states**.

#### Load reduction (DA1, DA2, DA3):

Loads acting on a strip are reduced in the analysis by partial factors. Depending on the inclination of the slip surface the program evaluates whether the **gravity force** acting on a given block is favorable or not. If the favorable action of force is greater than the unfavorable one, the program adopts the favorable coefficient. Based on that the weight of block  $W$  is pre-multiplied by the partial factor for the permanent load.

**Action of water** is reduced by the **partial factor**, which multiplies the resulting **pore pressure** and forces due to unconfined water above terrain.



*Determining whether the force action is favorable or not*

For input surcharges, the program first evaluates whether these act favorably or not, and then pre-multiplies the **overall load** by the corresponding partial factor.

**Reduction of material (DA1, DA3):**

Parameters of soils are automatically reduced by the corresponding partial factors.

**Reduction of resistance (DA2):**

Resistance on a slip surface is reduced.

**WARNING ! Calculation of slope stability according to DA2 or DA1 (comb. 1) using !! total parameters gives very unrealistic results.**

These are caused by a different reduction of the self-weight of massive (favorable and unfavorable). If adopting the above mentioned approaches **we recommend to adjust the partial factors manually** (i.e. increase the partial factor on resistance on the slip surface and decrease partial factors on load actions).

## Load Combinations

Actions of loads that act simultaneously are introduced into the analysis with the help of load combinations defined in **EN 1990 Basis of Structural Design**. Most of the loads are considered as permanent. Surcharges and input forces can be specified as variable load. The program automatically determines the values of individual partial factors depending on whether a given load acts in favor or unfavorably.

By default the variable loads are considered as **primary**. Nevertheless, the "**Verification**" and "**Dimensioning**" frames allow to specify the variable loads as **secondary** - such a load is then pre-multiplied by the corresponding coefficient reducing its magnitude. Providing that all loads are considered in the basic combination as secondary the program prompts a warning and the verification is not accepted.

Four types of combinations can be specified in the frame "**Stage settings**":

**Persistent and transient design situation:**

$$\sum_{j \geq 1} \gamma_{G,j} \cdot G_{k,j} + \gamma_{Q,1} \cdot Q_{k,1} + \sum_{i > 1} \gamma_{G,i} \cdot \psi_{0,i} \cdot Q_{k,i}$$

- where:
- $G_{k,j}$  - characteristic value of  $j^{th}$  permanent load
  - $\gamma_{G,j}$  - partial factor of  $j^{th}$  permanent load
  - $Q_{k,i}$  - characteristic value of secondary  $i^{th}$  variable load
  - $Q_{k,1}$  - characteristic value of primary variable load
  - $\gamma_{Q,i}$  - partial factor of  $i^{th}$  variable load
  - $\psi_0$  - factor for quasi-permanent value of variable load

**Accidental design situation:**

$$\sum_{j \geq 1} G_{k,j} + A_d + \psi_{1,i} \cdot Q_{k,1} + \sum_{i > 1} \psi_{2,i} \cdot Q_{k,i}$$

- where:  $G_{k,j}$  - characteristic value of  $j^{th}$  permanent load

- $Q_{k,1}$  - characteristic value of primary variable load
- $\psi_{1,i}$  - factor for frequent value of variable load
- $\psi_{2,i}$  - factor for combination value of variable load
- $A_{Ed}$  - design value of extreme load

**Seismic design situation:**

$$\sum_{j \geq 1} G_{k,j} + A_{Ed} + \sum_{i > 1} \psi_{2,i} \cdot Q_{k,i}$$

- where:
- $G_{k,j}$  - characteristic value of  $j^{th}$  permanent load
  - $Q_{k,i}$  - characteristic value of secondary  $j^{th}$  variable load
  - $\psi_{2,i}$  - factor for quasi-permanent value of variable load
  - $A_{Ed}$  - design value of seismic load

**Load partial factors** and **combination coefficients** are introduced in the "Partial factors" dialog window.

No. of force	Force	Fx [lb/ft]	Fz [lb/ft]	Application point x [ft]	z [ft]	Minor load
1	Weight - wall	0.0	10005.0	6.01	-6.10	
2	FF resistance	124.6	0.0	0.00	-0.67	
3	Weight - earth wedge	0.0	8243.4	8.64	-7.00	
4	Active pressure	-9031.7	12610.5	11.10	-6.49	
5	Force No. 1	-2000.0	0.0	5.10	-23.00	<input checked="" type="checkbox"/>

*Input of secondary loads*

## Analysis According to LRFD

When performing analysis according to LRFD (**Load Resistance Design Factor**) we follow the theory of **limit states**.

The analysis according to LRFD introduces two types of design coefficients:

- coefficients modifying the load magnitude (**Load factors**)
- coefficients reducing the soil resistance (**Resistance factors**)

The program allows verification according to two methods of the analysis, LRFD 2003 or LRFD 2012.

**LRFD 2003** is implemented in the program to perform:

- Analysis of retaining walls (support structures)
- Analysis of spread foundations

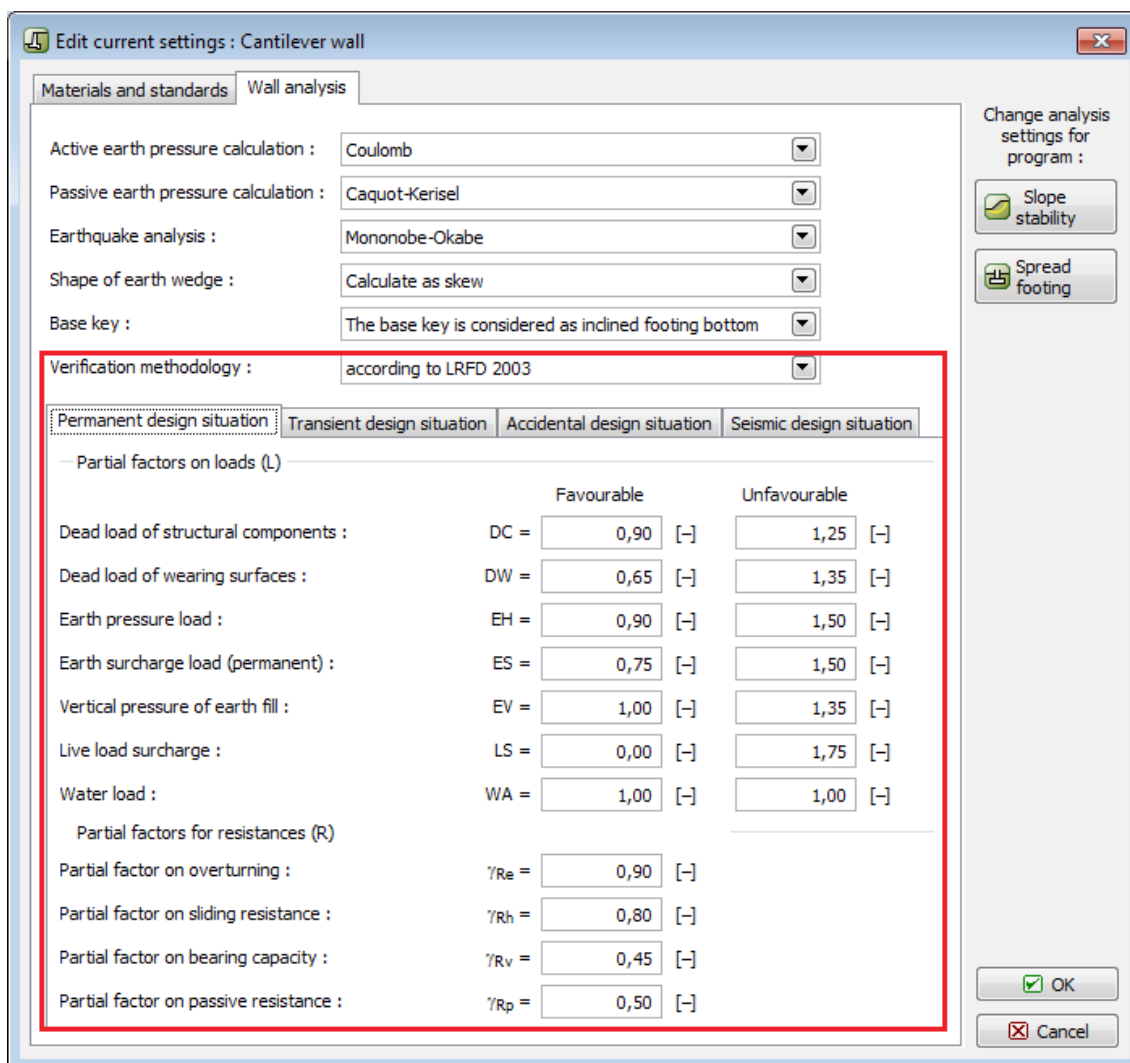
**LRFD 2012** is implemented in the program to perform:

- Analysis of retaining walls (support structures)
- Analysis of spread foundations
- Slope stability analysis

LRFD 2012 introduces new types of **design situations** (Strength I, Service I, Extreme I).

## LRFD 2003 - Analysis of Retaining Walls (Support Structures)

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).



*Analysis based on LRFD 2003 - Input of partial coefficients for wall analysis*

When checking individual cases of failure the program determines, whether the **force or pressure acts favorably or not** and then pre-multiplies the force by corresponding partial factor. The overall resistance of a structure against failure is in pre-multiplied by the corresponding resistance factor in the final verification.

In case of supporting structures, information about the applied design factors is listed in the analysis protocol.

**Forces acting on construction**

Name	F <sub>hor</sub> [lb/ft]	App.Pt. Z [ft]	F <sub>vert</sub> [lb/ft]	App.Pt. X [ft]	Coeff. overtur.	Coeff. sliding	Coeff. stress
Weight - wall	0.0	-6.10	10005.0	6.01	0.900	0.900	1.250
FF resistance	-124.6	-0.67	0.0	0.00	1.000	1.000	1.000
Weight - earth wedge	0.0	-7.00	8243.4	8.64	1.000	1.000	1.350
Active pressure	9031.7	-6.49	12610.5	11.10	1.000	1.500	1.500
Surcharge n.1	67.8	-8.95	96.4	10.05	0.000	0.000	1.500
Force No. 1	2000.0	-23.00	0.0	5.10	1.350	1.350	1.350

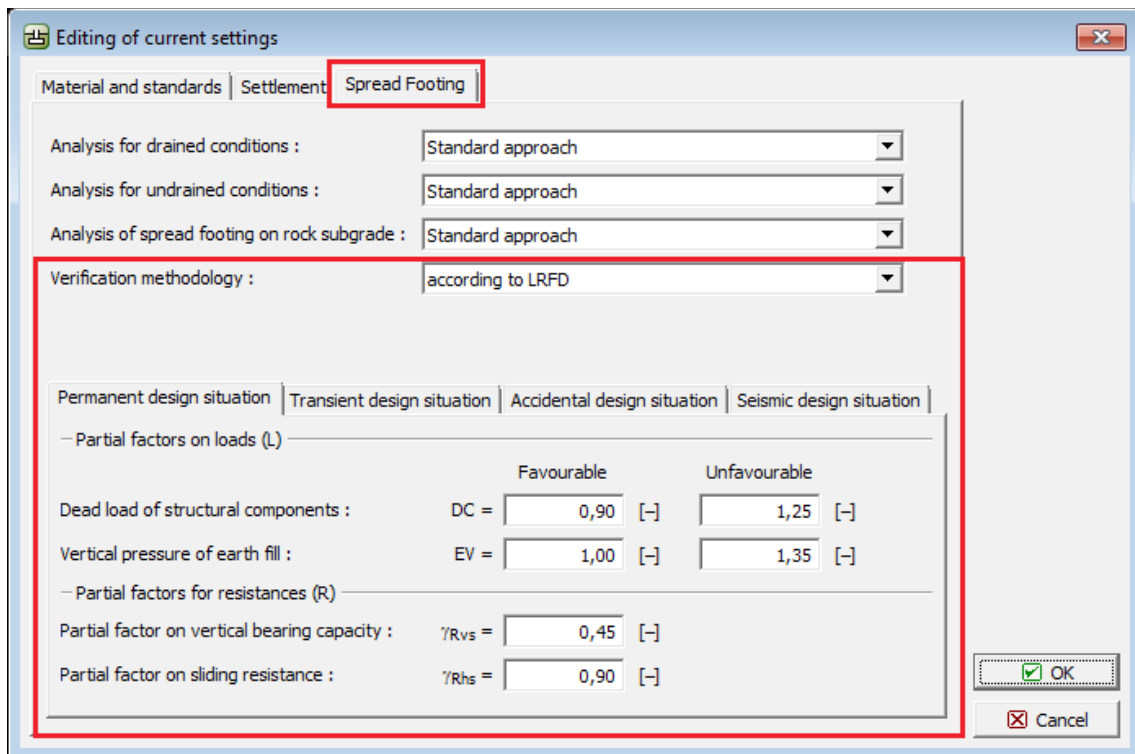
*Analysis protocol*

## LRFD 2003 - Analysis of Spread Foundations

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).

Only partial factors are used to multiply the **self weight of spread foundation** and the **weight of soil above the footing (overburden)**. Individual components of load must be **pre-multiplied** by the corresponding partial factors - the program does not further modify the input load.

When performing the final verification the overall resistance of the structure against failure is multiplied by the corresponding resistance factor.



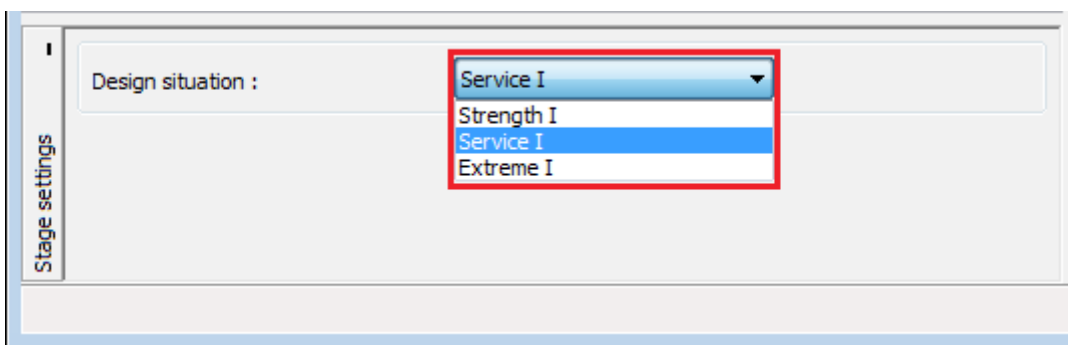
*Analysis based on LRFD 2003 - input of partial factors for foundations*

## LRFD 2012 - Design Situations

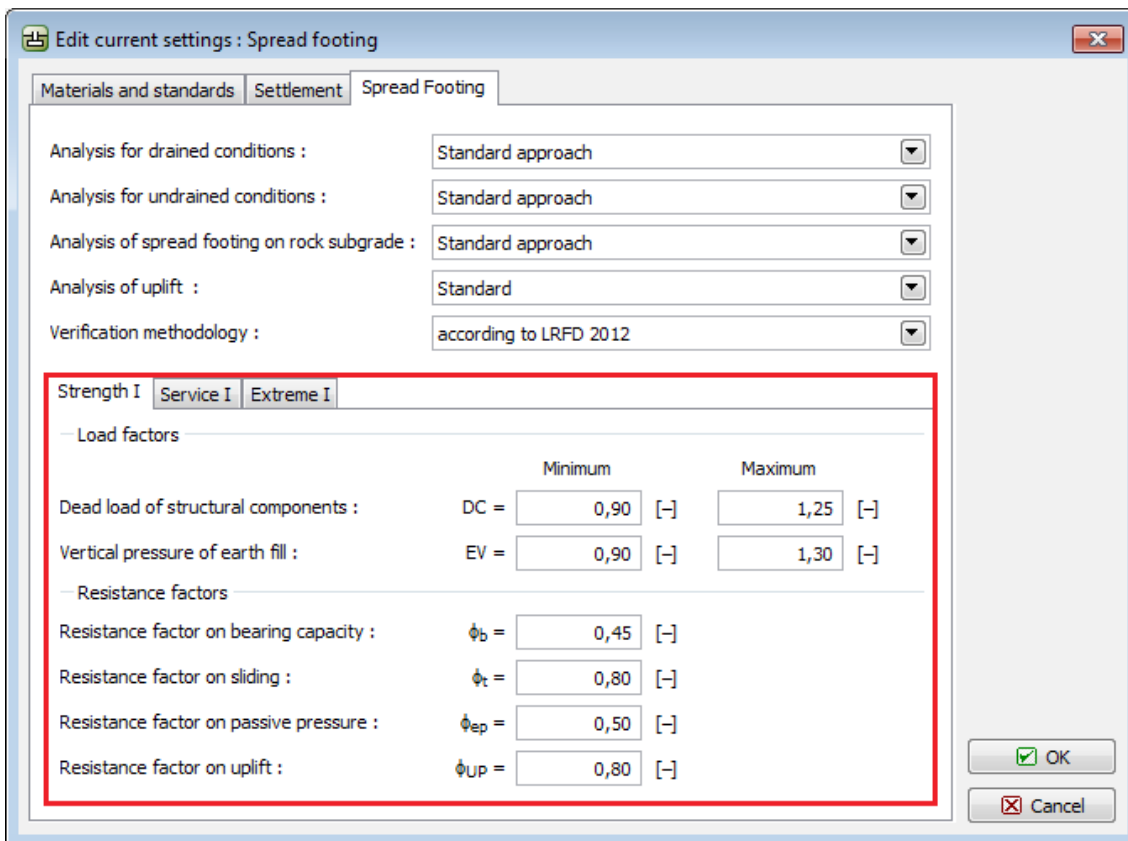
LRFD 2012 introduces the following design situations for the analysis of **support structures (retaining walls), foundations, and slope stability**:

- **Strength I**: the basic design situation that reduces the structure resistance and the magnitude of load.
- **Service I**: this design situation assumes for most cases the partial factors (**load, resistance reduction**) equal to 1.0.
- **Extreme I**: this design situation assumes for most cases the partial factors of **resistance reduction** equal to 1.0.

The type of design situation is selected in the "**Stage settings**" frame. The values of partial factors (**load, resistance reduction**) can be modified in the "**Settings**" frame.



LRFD 2012 - Selection of design situation



*Analysis based on LRFD 2012 - input of partial factors*

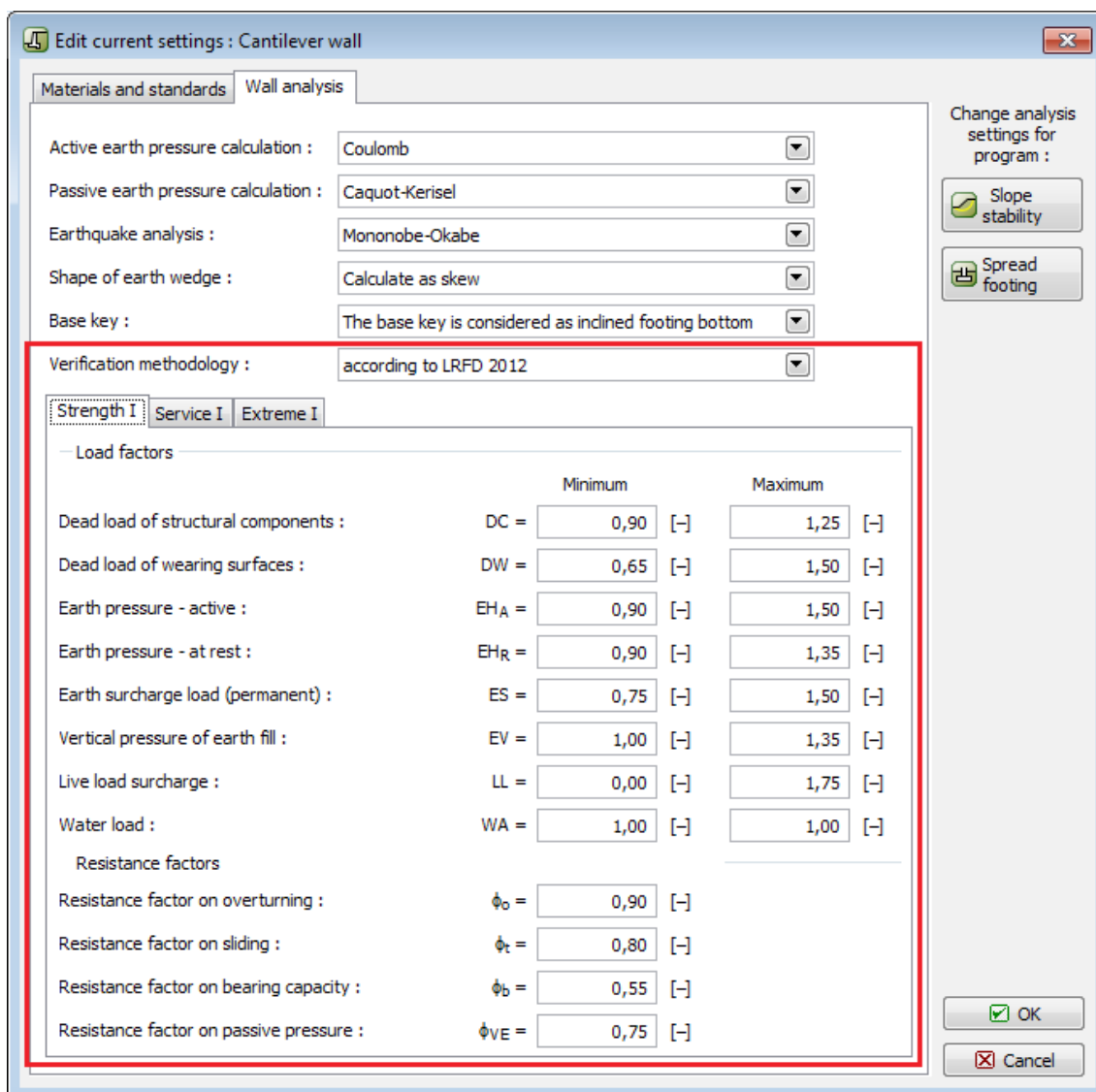
## LRFD 2012 - Analysis of Retaining Walls (Support Structures)

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).

These coefficients enter the analysis according to the selected design situation.

When evaluating individual cases of failure, the program determines, whether the **force or pressure acts favorably or unfavorably**. It is then multiplied by the corresponding partial factor.

When performing the final verification the overall resistance of the structure against failure is multiplied by the corresponding resistance factor.



*Analysis based on LRFD 2012 - input of partial factors for wall analysis*



For support structures (walls), the information about the applied design factors are provided in the analysis protocol.

#### Forces acting on construction

Name	F <sub>hor</sub> [lb/ft]	App.Pt. Z [ft]	F <sub>vert</sub> [lb/ft]	App.Pt. X [ft]	Coeff. overtur.	Coeff. sliding	Coeff. stress
Weight - wall	0.0	-6.10	10005.0	6.01	0.900	0.900	1.250
FF resistance	-124.6	-0.67	0.0	0.00	1.000	1.000	1.000
Weight - earth wedge	0.0	-7.00	8243.4	8.64	1.000	1.000	1.350
Active pressure	9031.7	-6.49	12610.5	11.10	1.000	1.500	1.500
Surcharge n.1	67.8	-8.95	96.4	10.05	0.000	0.000	1.500
Force No. 1	2000.0	-23.00	0.0	5.10	1.350	1.350	1.350

*Analysis protocol*

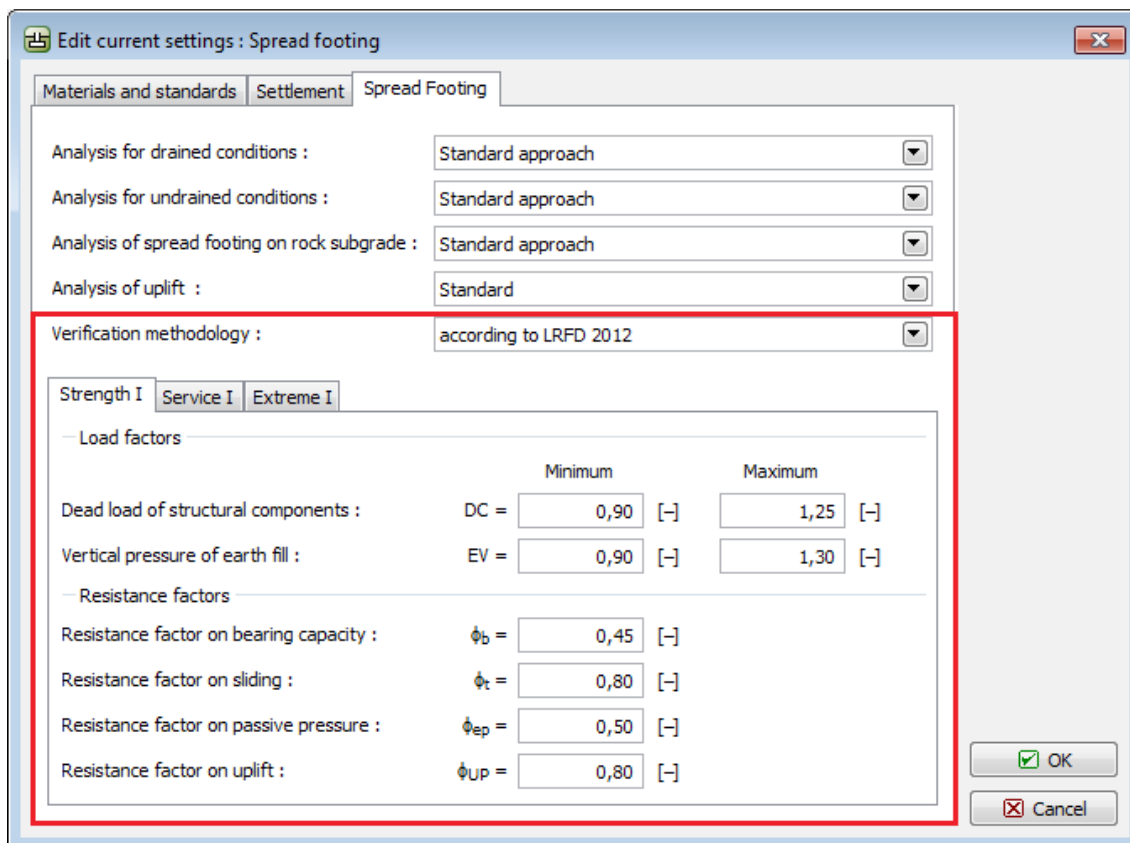
## LRFD 2012 - Analysis of Spread Foundations

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).

These coefficients enter the analysis according to the selected design situation.

Partial factors for load are used to multiply the **self weight of spread foundation** and the **weight of soil above foundation (overburden)** only. Individual components of load must be **multiplied** by corresponding partial factors - the input load is not adjusted by the program in any way.

When performing the final verification, the overall resistance of the structure against failure is multiplied by the corresponding resistance factor.



Analysis based on LRFD 2012 - input of partial factors for foundations

## LRFD 2012 - Slope Stability Analysis

Analysis according to LRFD introduces two types of design coefficients - coefficients modifying the load magnitude (**Load factors**) and coefficients reducing the soil resistance (**Resistance factors**).

These coefficients enter the analysis according to the selected design situation.

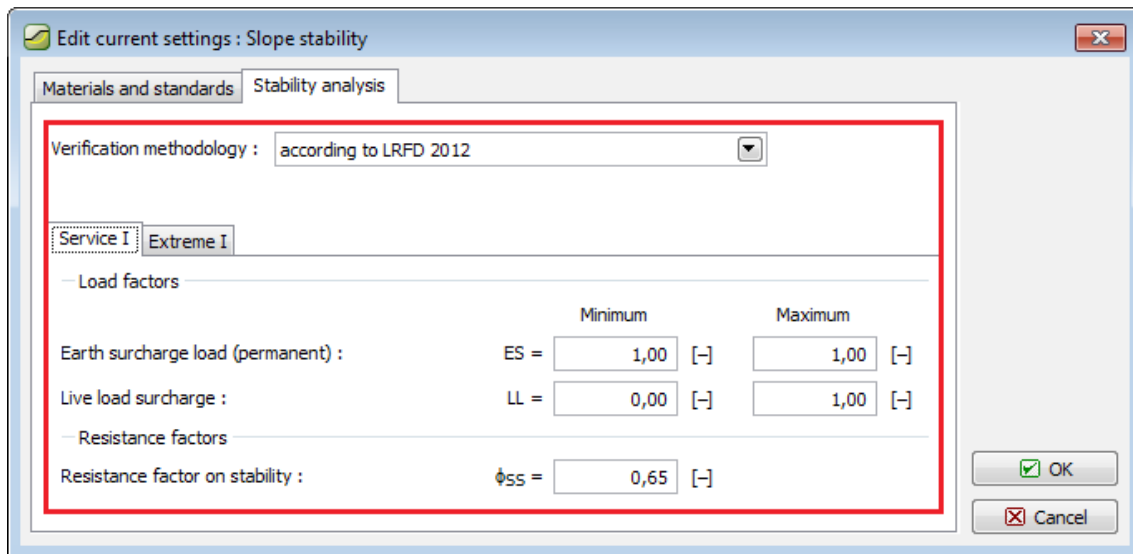
The input loads are verified, whether they act favorably or unfavorably. The **surcharge magnitude** is then multiplied by the corresponding partial factor for load (**ES** or **LL**, respectively).

### Resistance reduction (Resistance factors):

The overall resistance on slip surface is reduced by the  $\phi_{SS}$  partial factor. When evaluating safety, the following condition on the slip surface must be satisfied:

$$\phi_{SS} F_{pas} \geq F_{act}$$

- where:
- $\phi_{SS}$  - resistance factor on stability
  - $F_{pas}$  - resisting (passive) forces acting on slip surface
  - $F_{act}$  - active forces acting on slip surface



*Analysis based on LRFD 2012 - input of partial factors for stability analyses*

## Analysis According to Chinese Standards

GEO5 programs allow to perform various analyses based on the methodologies provided by Chinese standards.

**Geotechnical analyses** are verified using the **safety factor**. Neither calculated forces or soil parameters are reduced by any coefficient.

**Dimensioning of steel-reinforced concrete and masonry structures** follows the GB 50153-2008 or JTS D30-2004 standards. In this case, each force entering a combinations is pre-multiplied by the corresponding coefficient.

Another coefficient influencing the dimensioning is the **Coefficient of structure importance** (GB 50153-2008, 8.2.2-1) to be specified in the "**Settings**" frame when performing the structure verification according to GB 50010-2010.

**Earthquake analysis** and seismic combination analysis according to GB 50010-2010 further exploit **Seismic coefficients of strength** (GB 50011-2010), which increase the calculated bearing capacity of a cross-section. These coefficients are specified in the "**Settings**" frame in the "**Materials and standards**" tab.

Analysis of **sheeting structures** follows to **JGJ 120-2012** standard (Technical specification for retaining and protection of building foundation excavations). This is eg. a **determination of modulus of subsoil reaction**.

## Design Situations

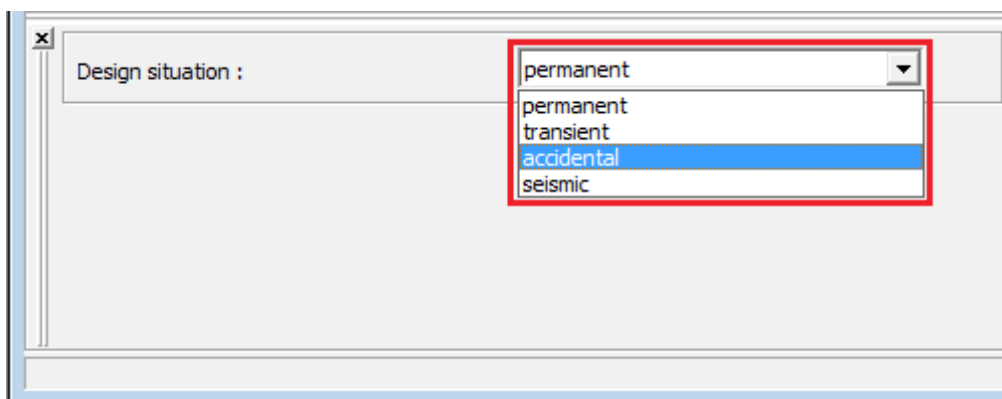
The programs allows to define four design situations, which may differ by the analysis coefficients. These are:

- **Permanent design situation** - most common situation and type of verification, adopted when proving the safe design of a structure for the assumed lifetime.
- **Transient design situation** - can be used for temporary structures (construction stages). Typically, lower safety is required in comparison to the permanent design situation.

- **Accidental design situation** - adopted for extraordinary loads (e.g. blast, vehicle impact, flood, fire, etc.). The values of partial factors are typically equal to one.
- **Seismic design situation** - applied to the analysis of earthquake. It might seem similar to the accidental design situation, but for earthquake higher safety is sometimes required. In some countries, the required safety is even the same as for the permanent design situation.

Safety coefficients and partial factors are specified in the [analysis settings](#).

The corresponding design situation for a given construction stage is selected in the frame "**Stage settings**".



*Selection of design situation*

## Individual Programs

This chapter contains basic description of individual ways of inputting data into the program:

- [Earth Pressure](#)
- [Cantilever Wall](#)
- [Gravity Wall](#)
- [Prefab Wall](#)
- [Masonry Wall](#)
- [Gabion](#)
- [Abutment](#)
- [Nailed Slope](#)
- [Redi-Rock Wall](#)
- [Sheeting Design](#)
- [Sheeting Check](#)
- [Anti-Slide Pile](#)
- [Shaft](#)
- [Slope Stability](#)
- [Rock Stability](#)
- [MSE Wall](#)

- Spread Footing
- Pile
- Pile CPT
- Pile Group
- Micropile
- Slab
- Beam
- Settlement
- Ground Loss
- Terrain
- FEM (and modules Consolidation, Water Flow, Tunnel)

## Program Earth Pressure

This program computes basic earth pressures (active pressure, passive pressure, and pressure at rest) acting upon an arbitrary shaped structure.

**The help in the program "Earth Pressure" includes the following topics:**

- Input of data into individual frames:
 

Project	Settings	Geometry	Profile	Soils	Assign
Terrain	Water	Surcharge	Earthquake	Stage Settings	Analysis
- Standards and analysis methods
- Theory for analysis in the program "**Earth Pressure**":
 

Stress in Soil Body	Earth Pressures
---------------------	-----------------
- Outputs
- General information about the work in the User Environment of GEO5 programs
- Common input for all programs

## Project

The frame "**Project**" is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in

text and graphical outputs.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "GeoClipboard".

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows to choose an already created setting from the "**Settings list**".

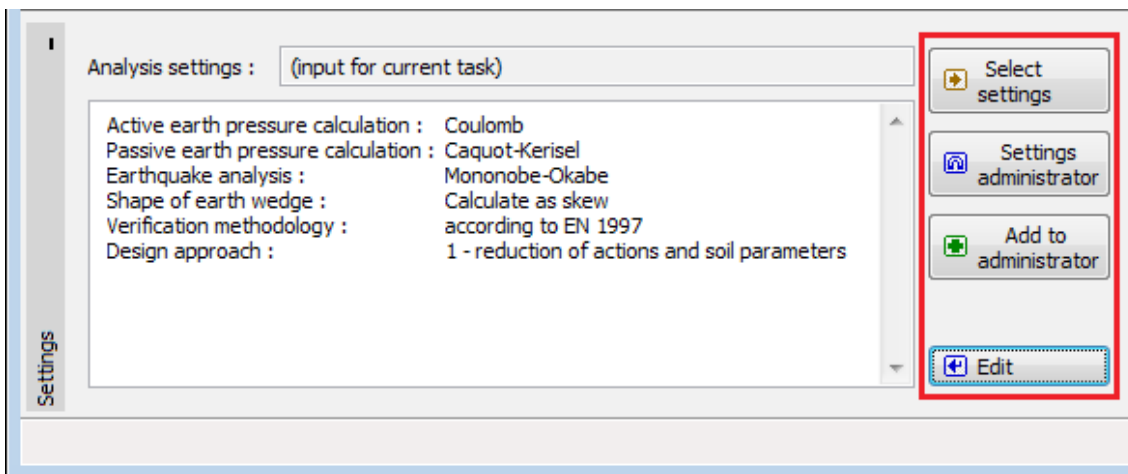
The "**Settings Administrator**" button opens the "**Administrator**" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Materials and standards**" and the "**Excavations**" tabs.



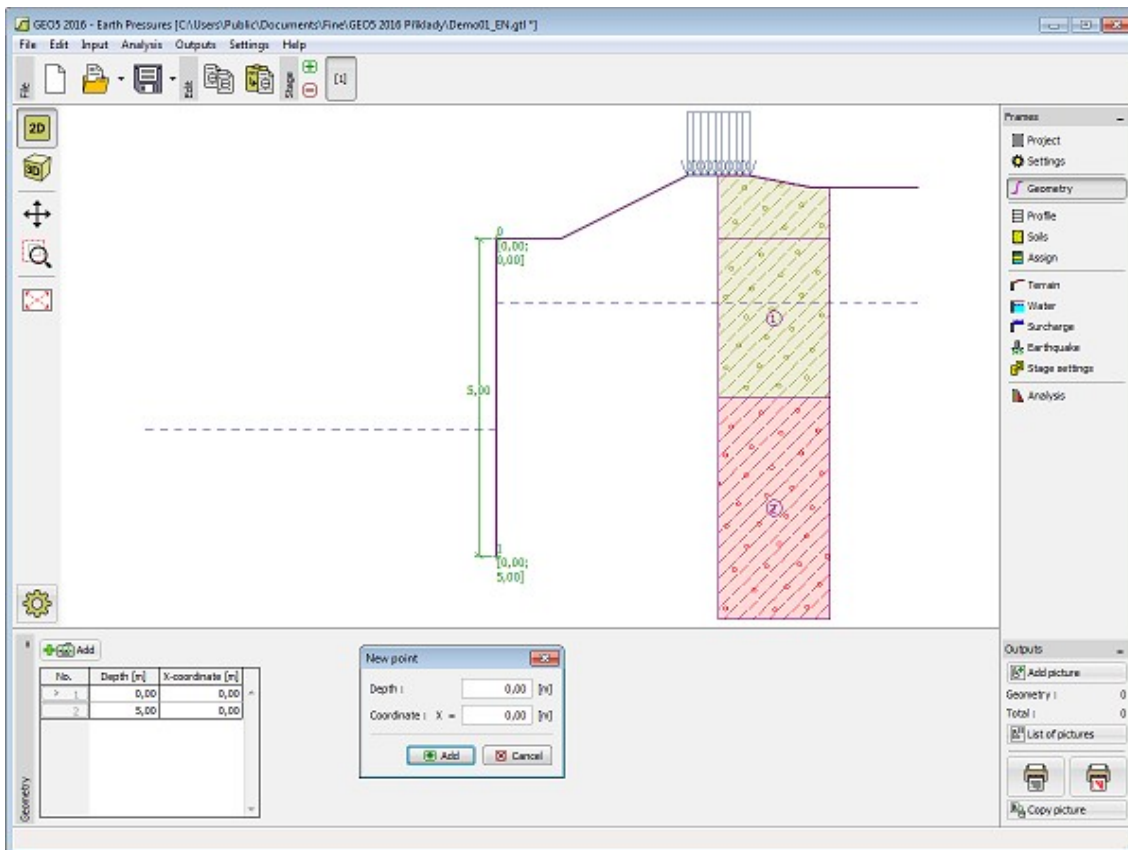
Frame "Settings"

## Geometry

The "**Geometry**" frame contains table listing the points of a structure. Adding (editing) points is performed in the "**New point**" dialog window.

The existing geometry points can be further edited on the desktop with the help of **active objects** - double clicking on a selected point opens a dialog window to edit the point.

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Geometry"

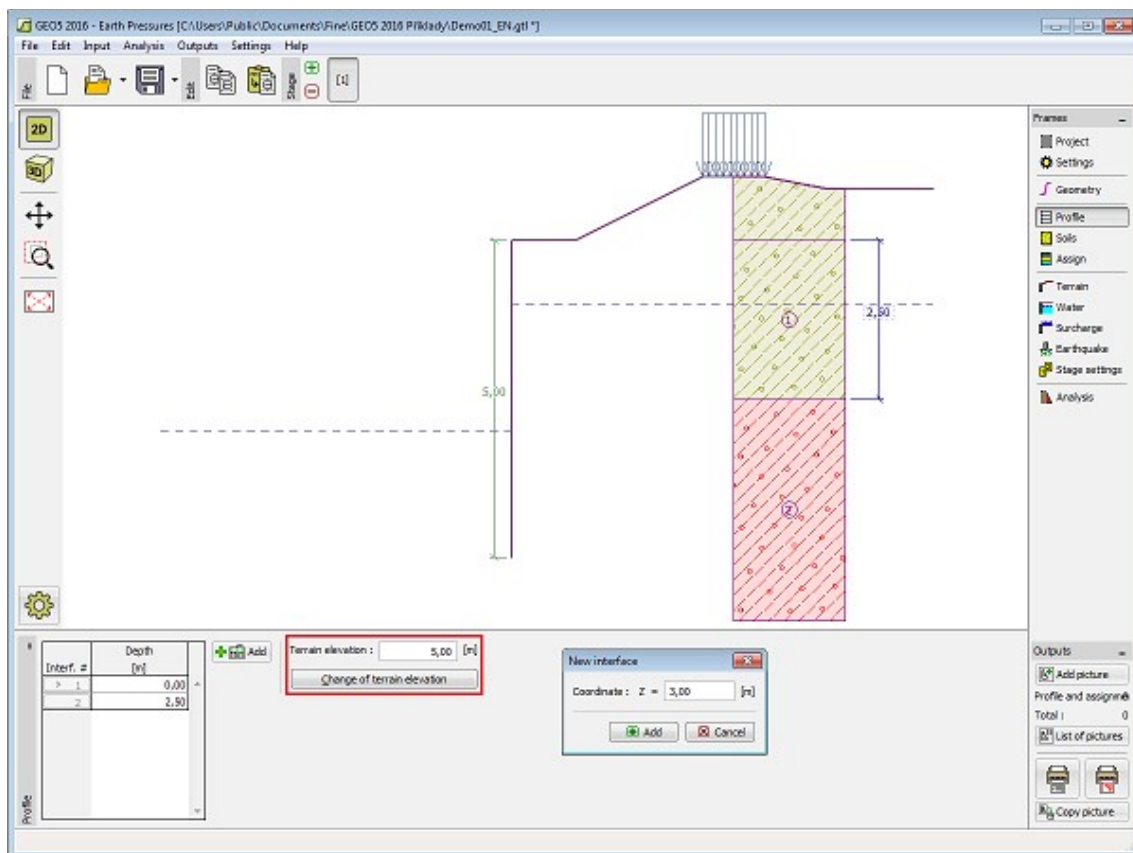
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of **active dimensions**.

**Adding** (editing) layer is performed in the "**New interface**" dialog window. The  $z$ -coordinate measured from the top point of a structure is specified ( $z$ -axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

## Soils

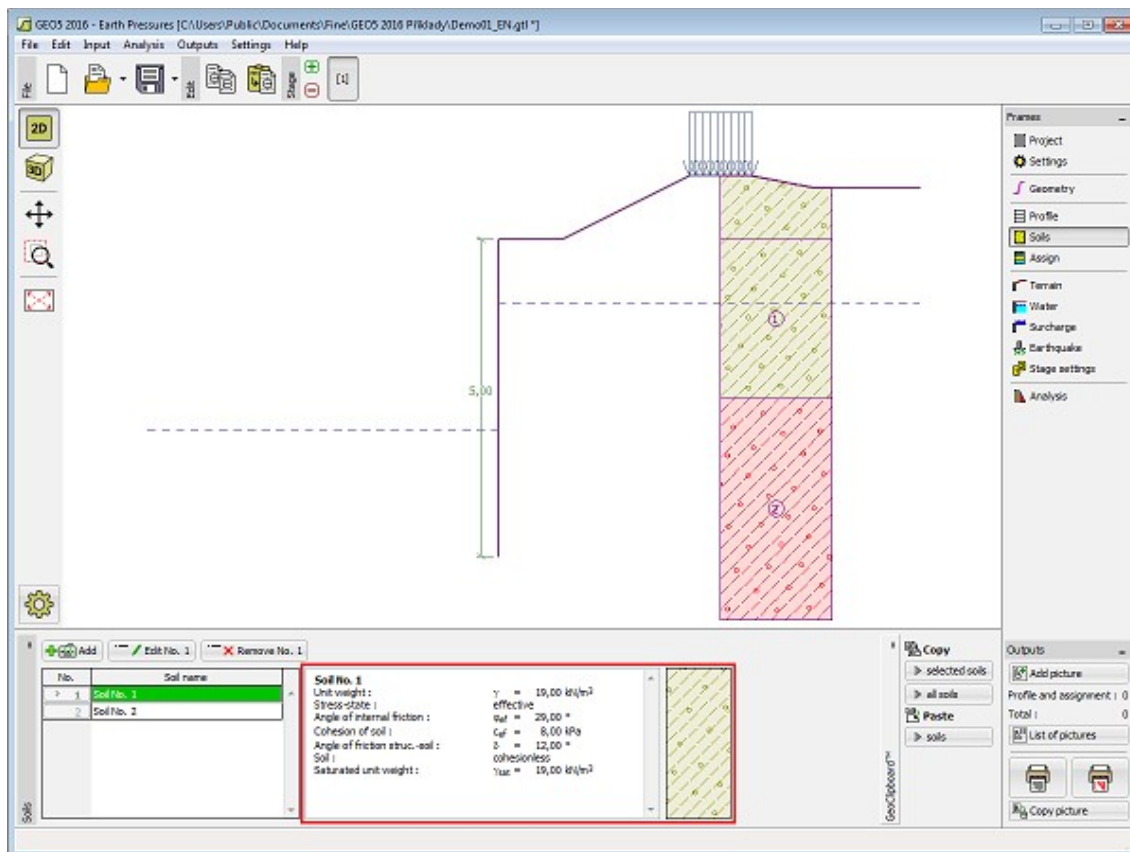
The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

**Adding** (editing) a soil is performed in the "**Add new soils**" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "**Basic data**", "**Earth pressure at rest**" and "**Uplift pressure**".

The program makes it possible **import soils in the gINT format**. Data of input soils can be copied within all GEO5 programs using "**GeoClipboard**".





Frame "Soils"

## Basic Data

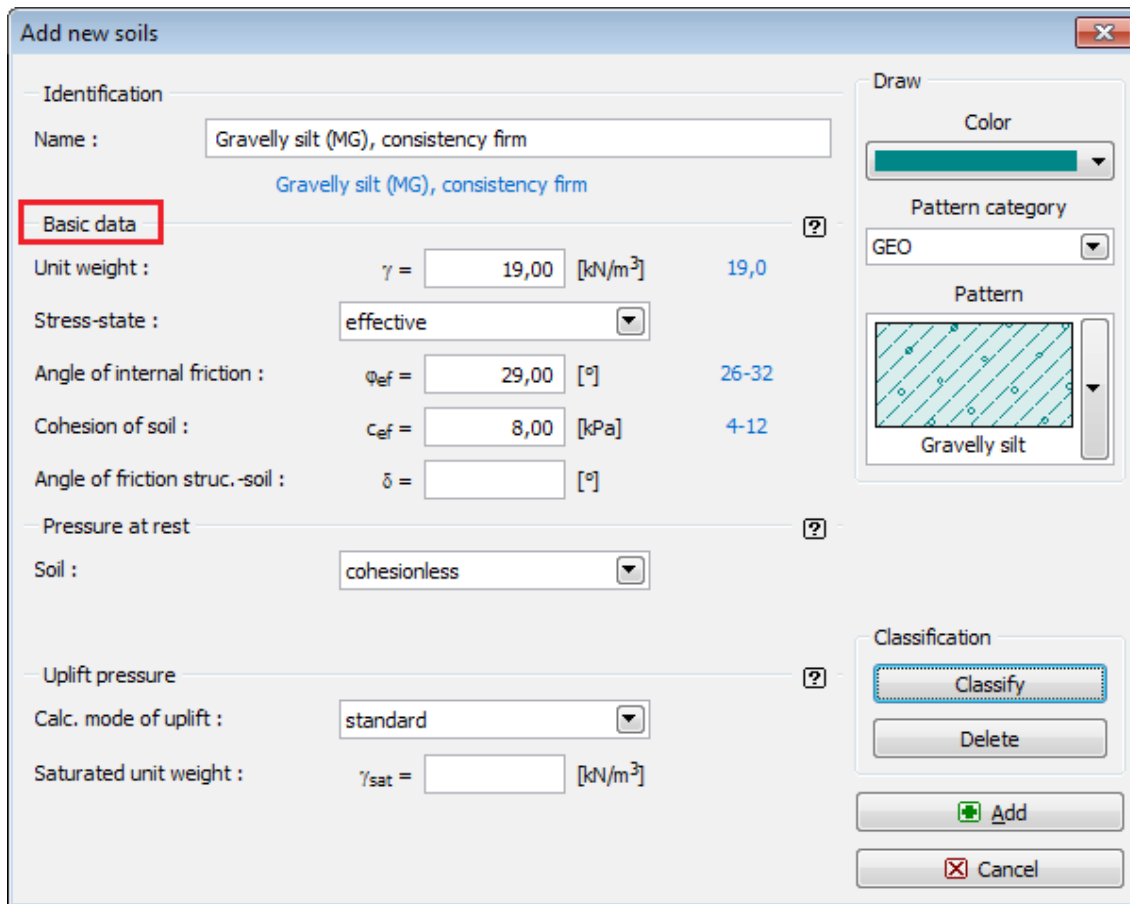
This part of the window allows to introduce basic parameters of the soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **soils database**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually. Approximate parameters of rocks are presented in the **theoretical section**.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the settings in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil and load, structure duration and water conditions.

For **effective stress**, it is further needed to specify the **angle of internal friction between the soil and the structure**, which depends on the structure material and type of soil. Possible values of this parameter are listed in the **table of recommended values**.

For **total stress**, it is further needed to specify the **adhesion of soil to the structure face  $a$** .

The associated theory is described in detail in the chapter "**Earth pressures**".



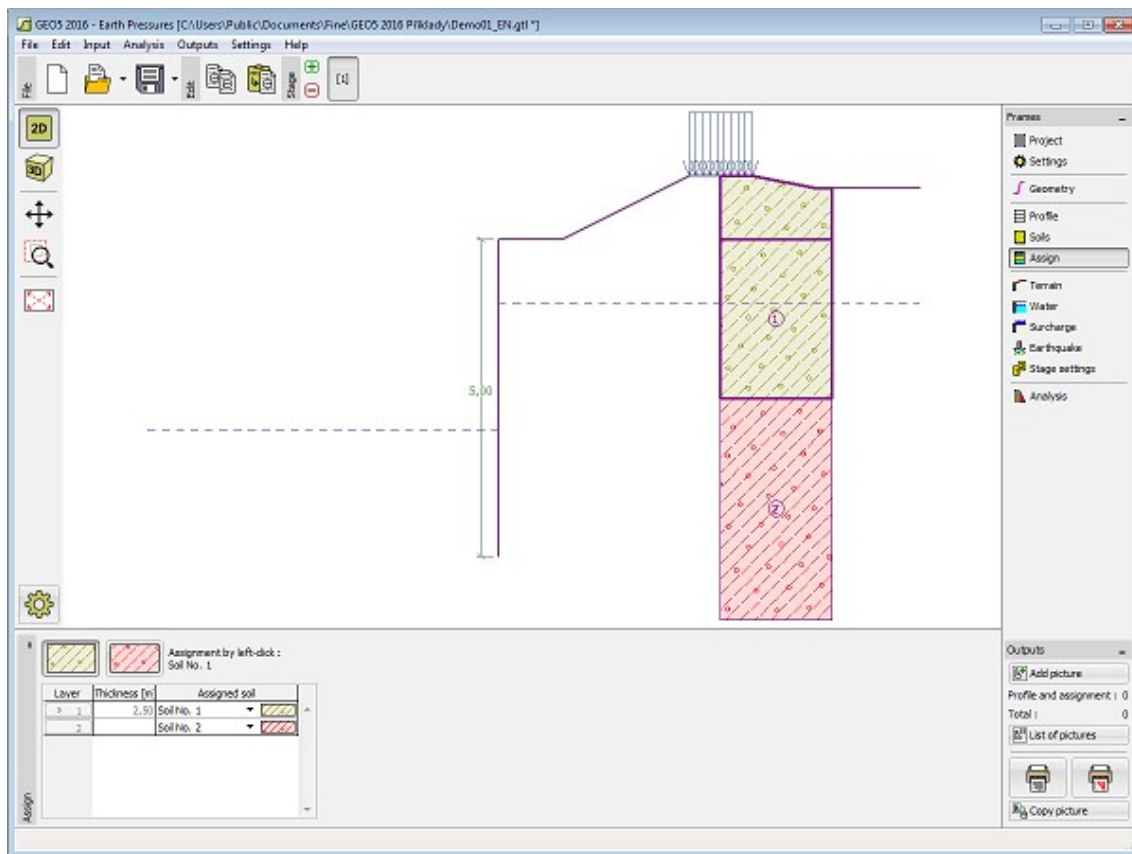
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



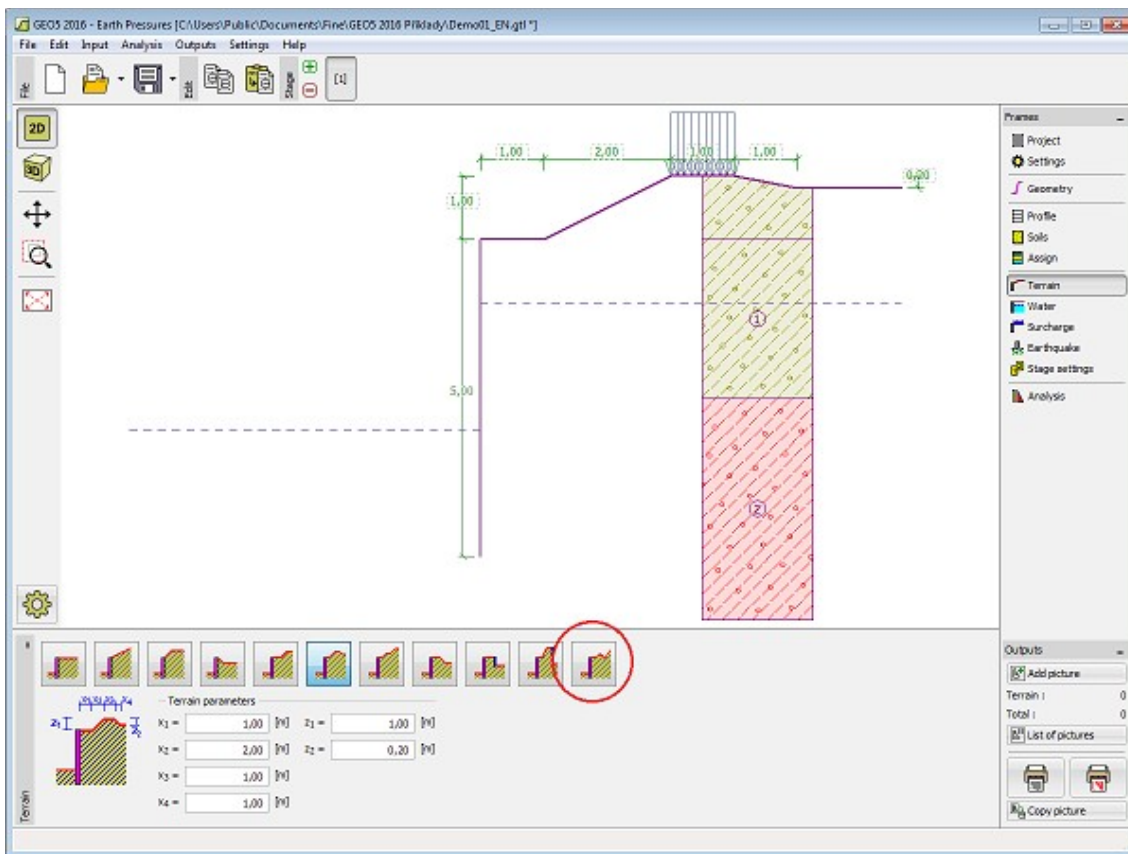
Frame "Assign"

## Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

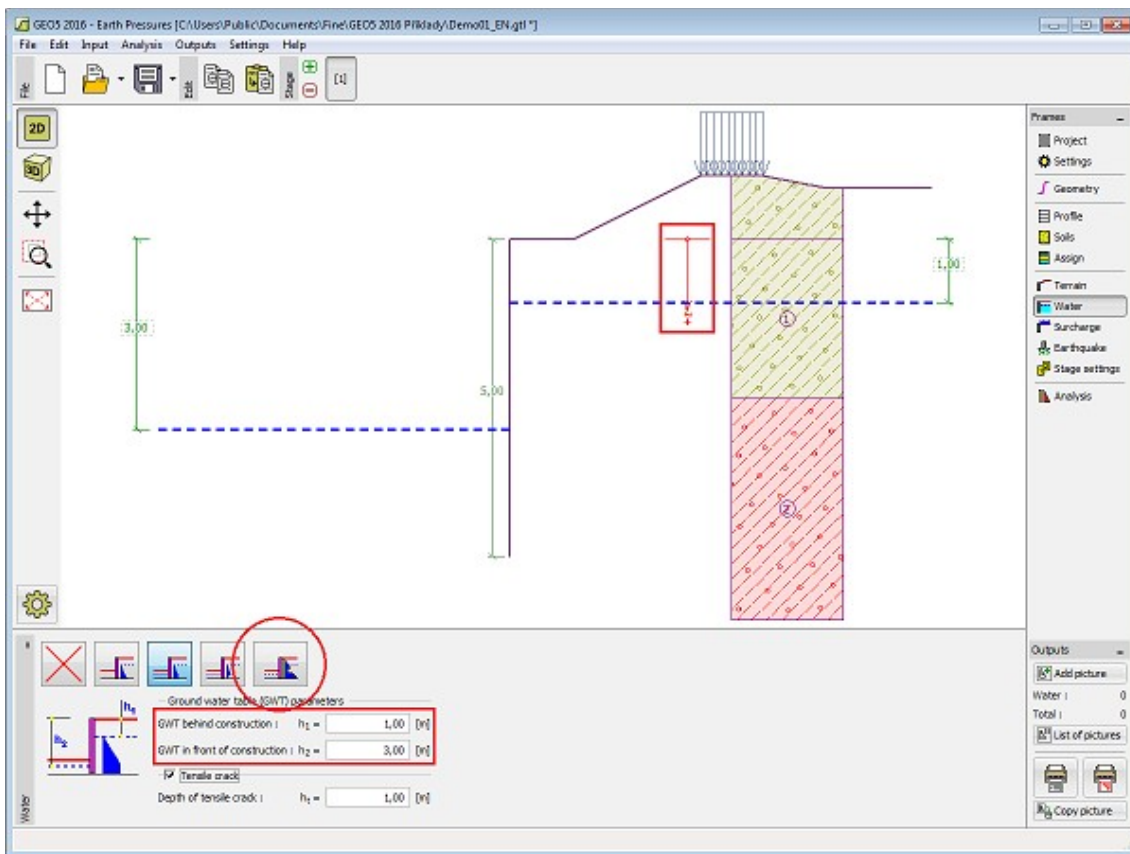
The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".

The program further allows for specifying a depth of **tensile cracks** filled with water.



Frame "Water"

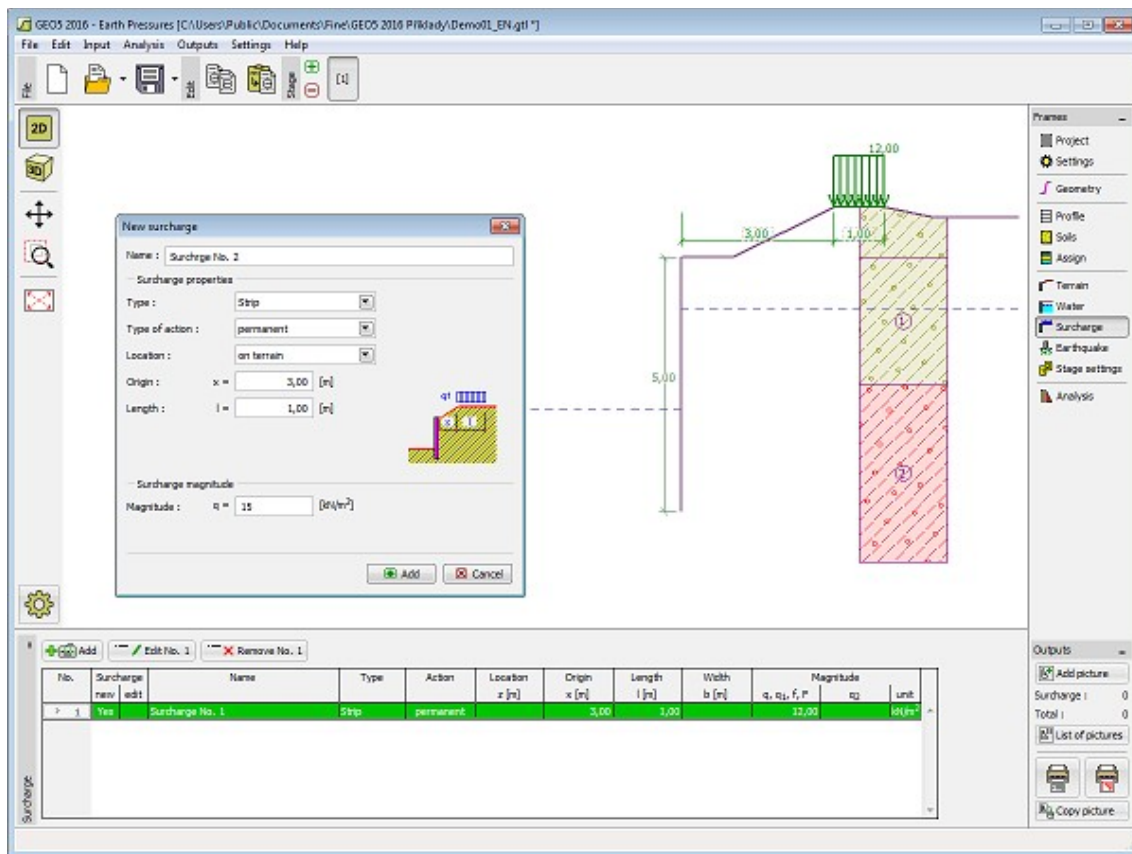
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".



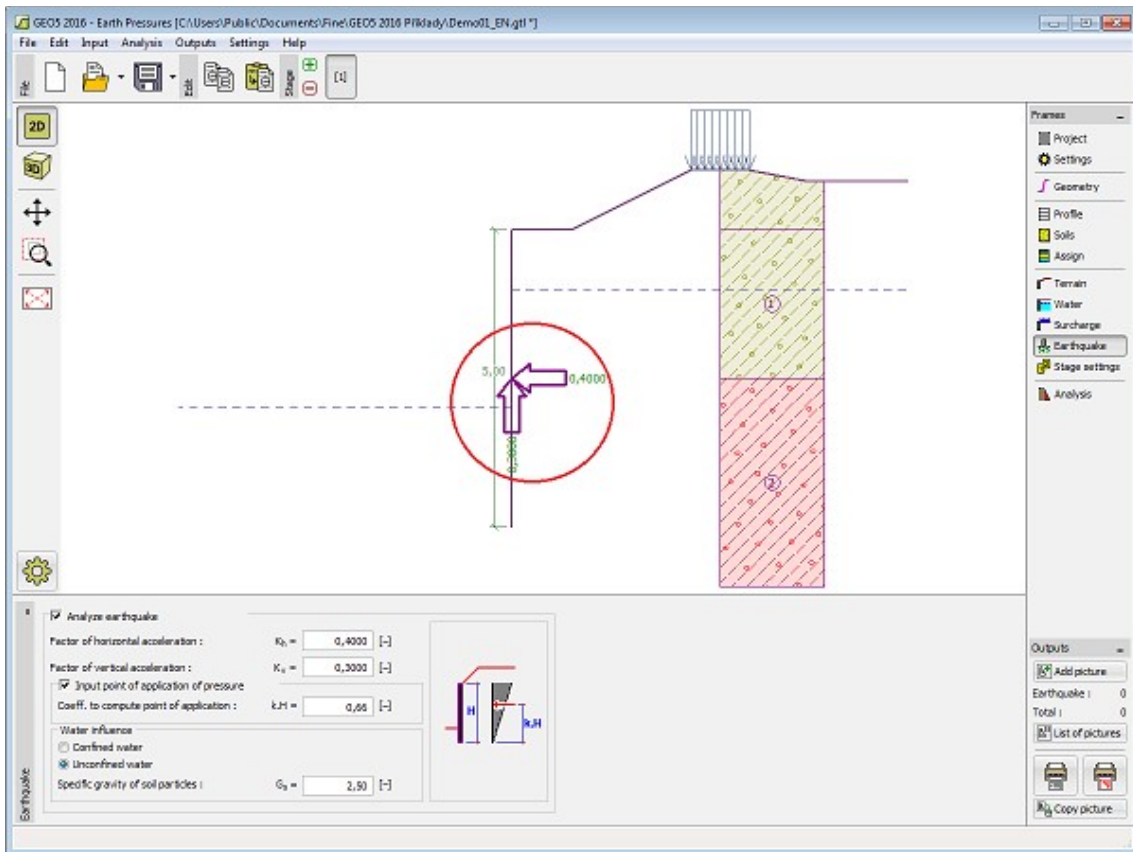
Frame "Surcharge"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



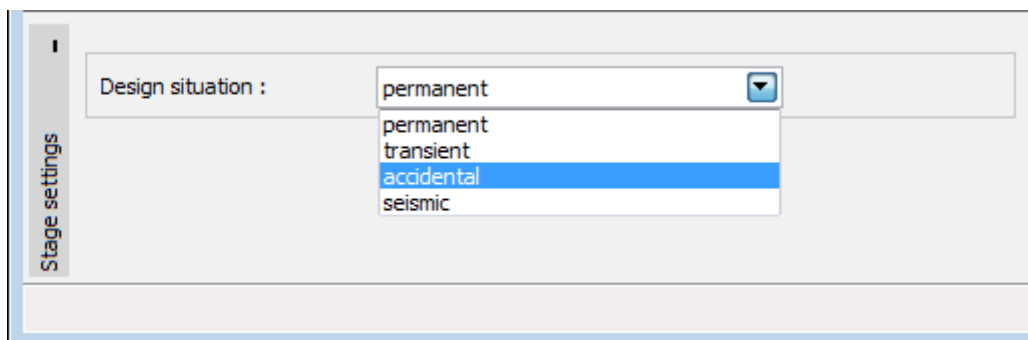
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"



## Analysis

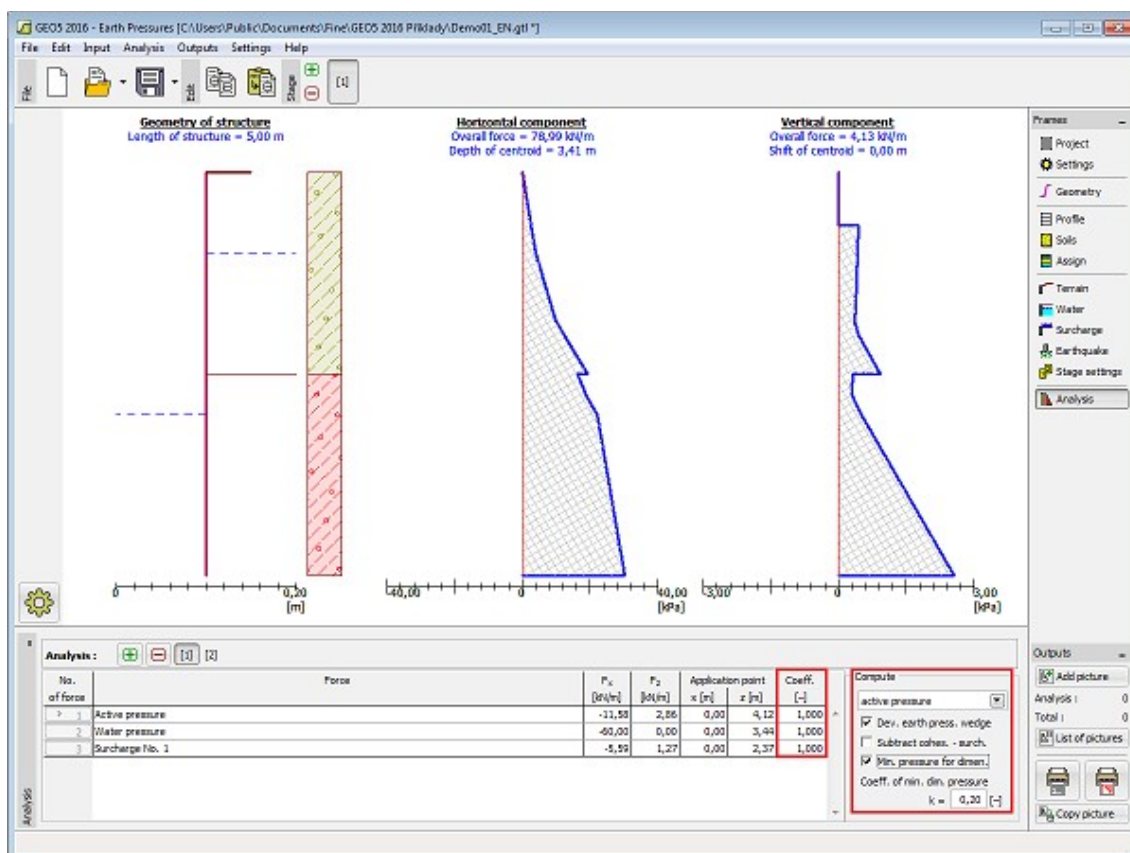
The frame "Analysis" shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the factor of safety, or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations". Providing the analysis is carried out according to "Design approach 1", it is necessary to enter the combination number in the right part of the window.
- Analysis according to LRFD - in the case the last column is not displayed.

The "Analysis" frame displays the analysis results. The frame serves to select type of computed earth pressure (active pressure, pressure at rest, passive pressure). Two options "Create soil wedge" and "Minimum dimensioning pressure" are available when computing the active earth pressure.

The analysis results are displayed on the desktop and are updated immediately for an arbitrary change in input data or setting. Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Analysis"



## Program Cantilever Wall

This program is used to verify cantilever wall design. It offers a number of wall shapes and analyzes reinforced concrete cross-sections.

The help in the program "Cantilever Wall" includes the following topics:

- Input of data into individual frames:

Project	Settings	Geometry	Material	Profile	Soils	Assign
Foundation	Terrain	Water	Surcharge	Front Face Resistance	Applied Forces	Earthquake
Base Anchorage	Stage Settings	Verification	Bearing Capacity	Dimensioning	Stability	

- Standards and analysis methods

- Theory for analysis in the program "Cantilever Wall":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity	Dimensioning of Concrete Structures
---------------------	-----------------	-------------------	---	-------------------------------------

- Outputs

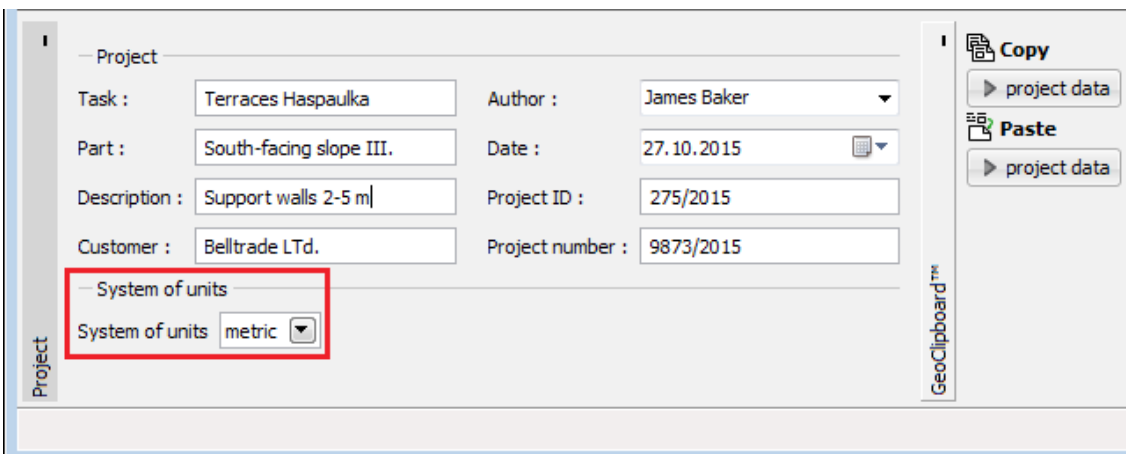
- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

## Project

The frame "Project" is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows user to switch analysis units (metric / imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".



Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

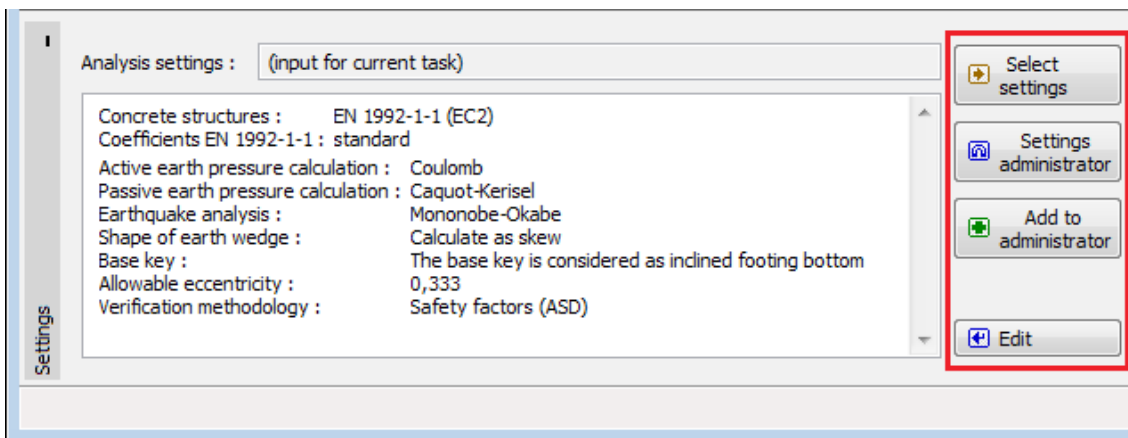
The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Materials and standards"** and **"Wall analysis"** tabs.



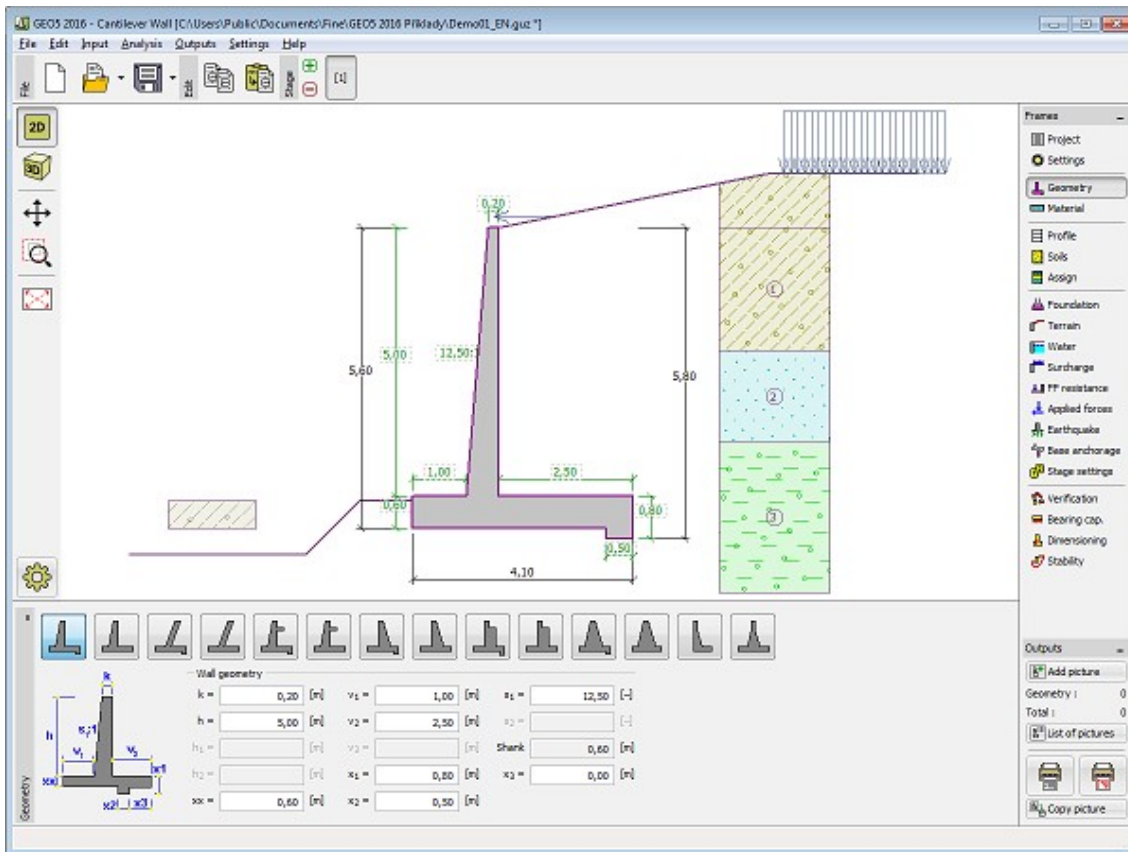
Frame "Settings"

## Geometry

The "Geometry" frame allows by pressing the button for selecting the wall shape. The selected shape with a graphic hint "Chart of wall geometry" appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

In case the structure is composed of inclined segments it is required to enter the ratio of sides of an inclined segment 1:x. **The straight structure** is specified by entering the value zero.

The program makes it possible to export the geometry of a structure in the \*.DXF format.



Frame "Geometry"

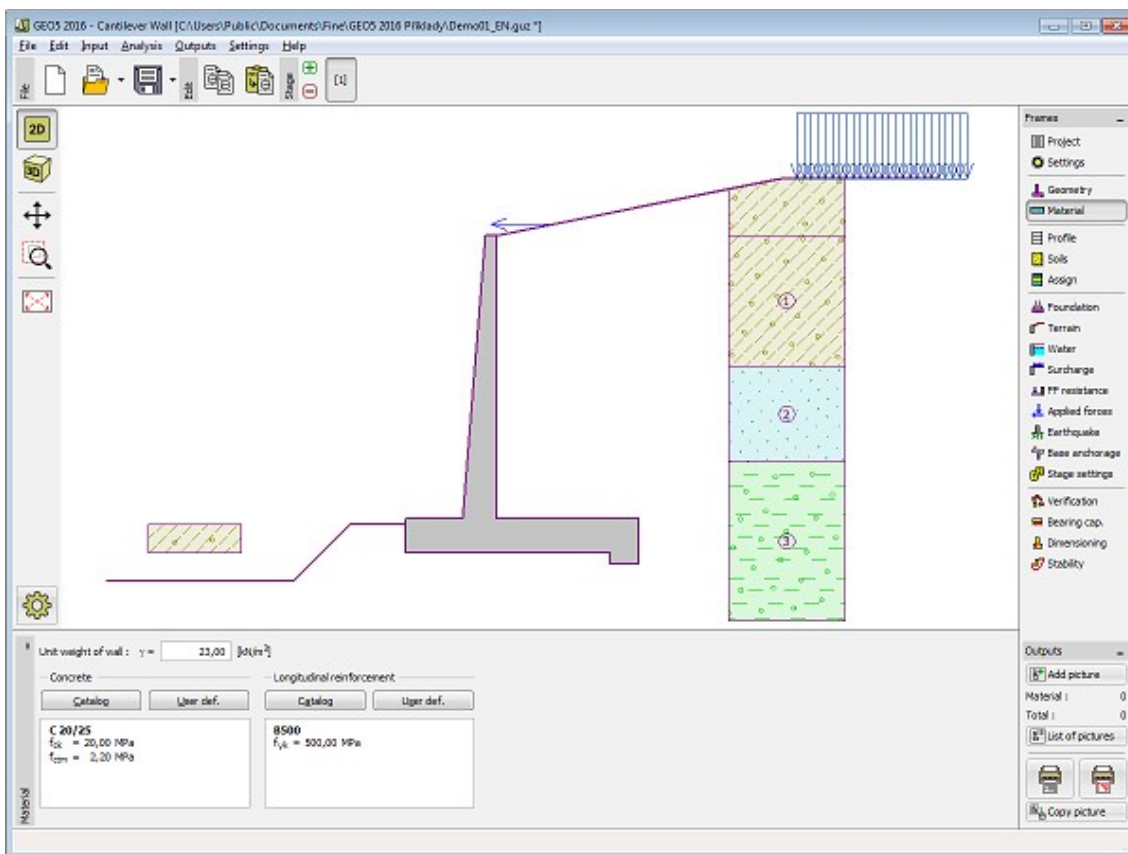
## Material

The "Material" frame allows to select material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- The "Catalog" button opens the "Catalog of materials" dialog window (for concrete or steel reinforcements), the list of materials then serves to select the desired material.
- The "Own" button opens the "Editor of material - Concrete" dialog window (for concrete) or the "Editor of material - Reinforcing steel bars" dialog window (for longitudinal steel reinforcements), which allows for manual specification of material parameters.

The catalogs content depends on the selection of standard for the design of concrete structures set in the "Materials and standards" tab. The input field in the upper part of the frame serves to specify the wall unit weight.



Frame "Material"

## Profile

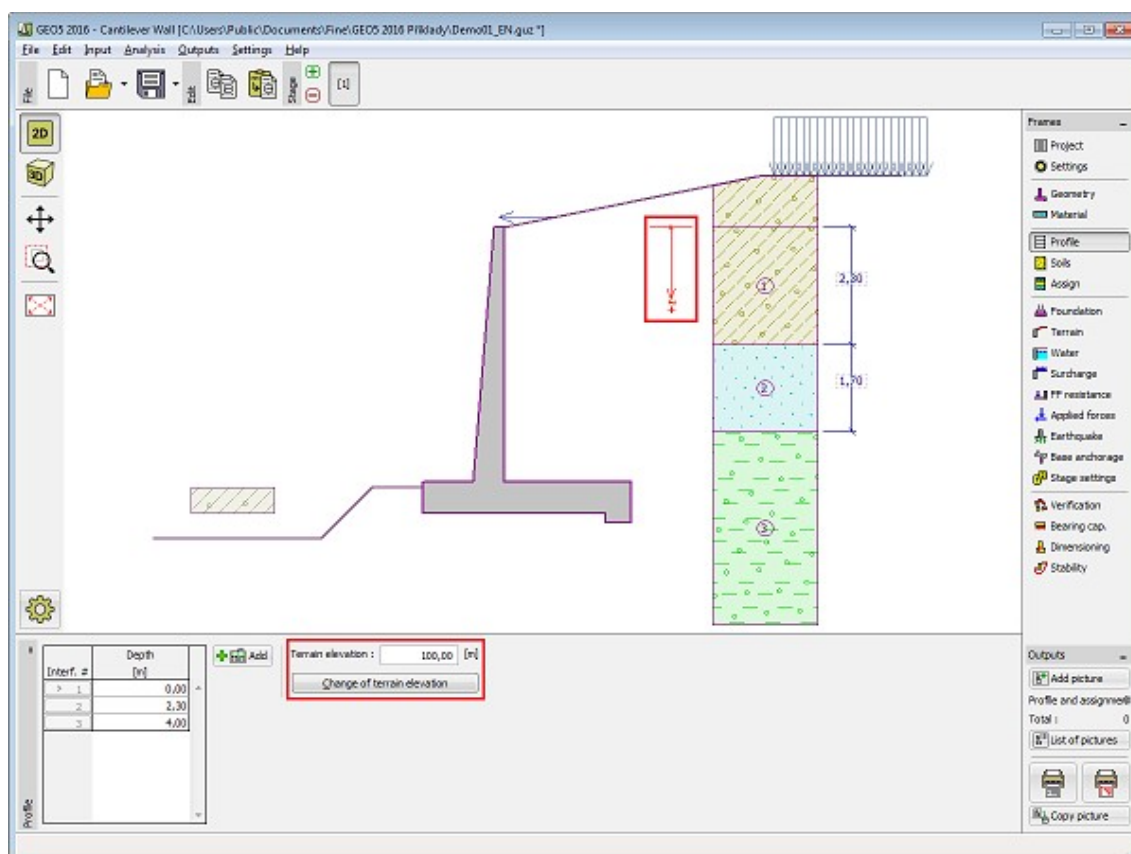
The "Profile" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "New interface" dialog window. The z-coordinate

measured from the top point of a structure is specified ( $z$ -axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

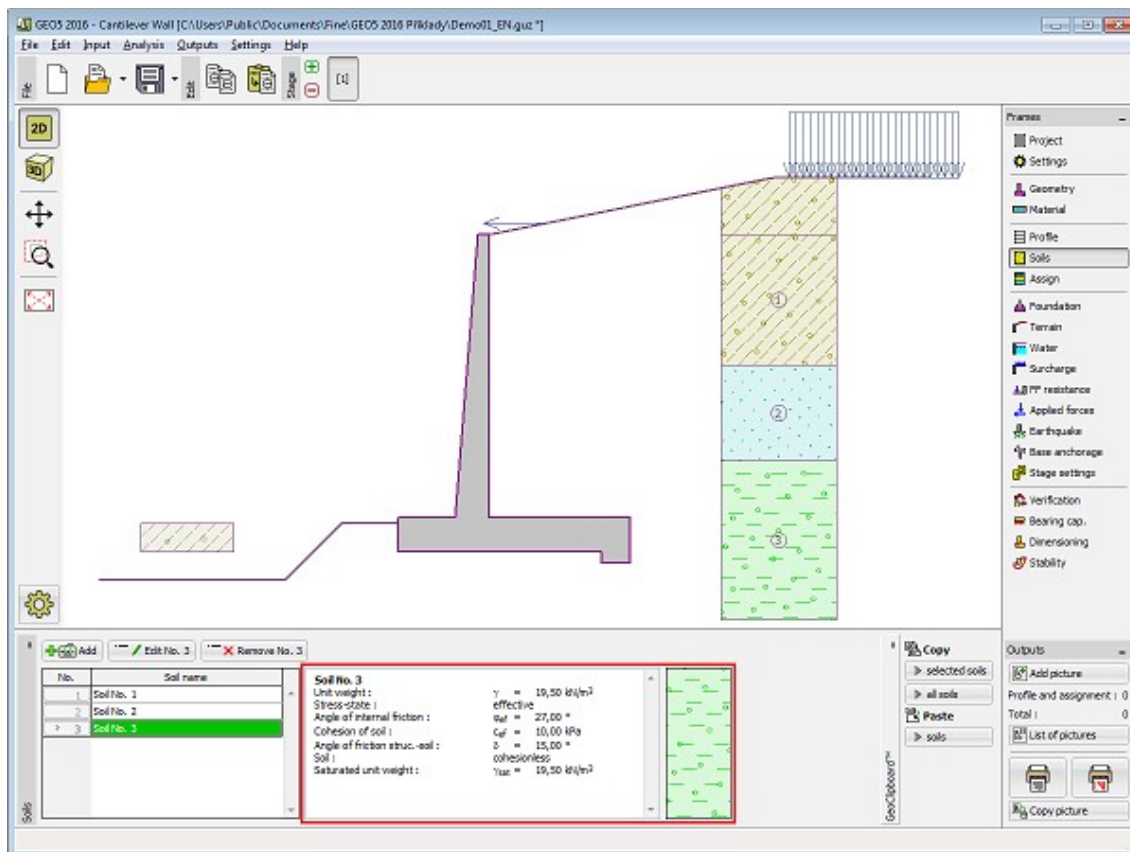
## Soils

The "**Soils**" frame contains a **table** with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

**Adding** (editing) a soil is performed in the "**Add new soils**" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "**Basic data**", "**Earth pressure at rest**" and "**Uplift pressure**".

The program makes it possible to **import soils in the gINT format**. Data of input soils can be copied within all GEO5 programs using "**GeoClipboard**".



Frame "Soils"

## Basic Data

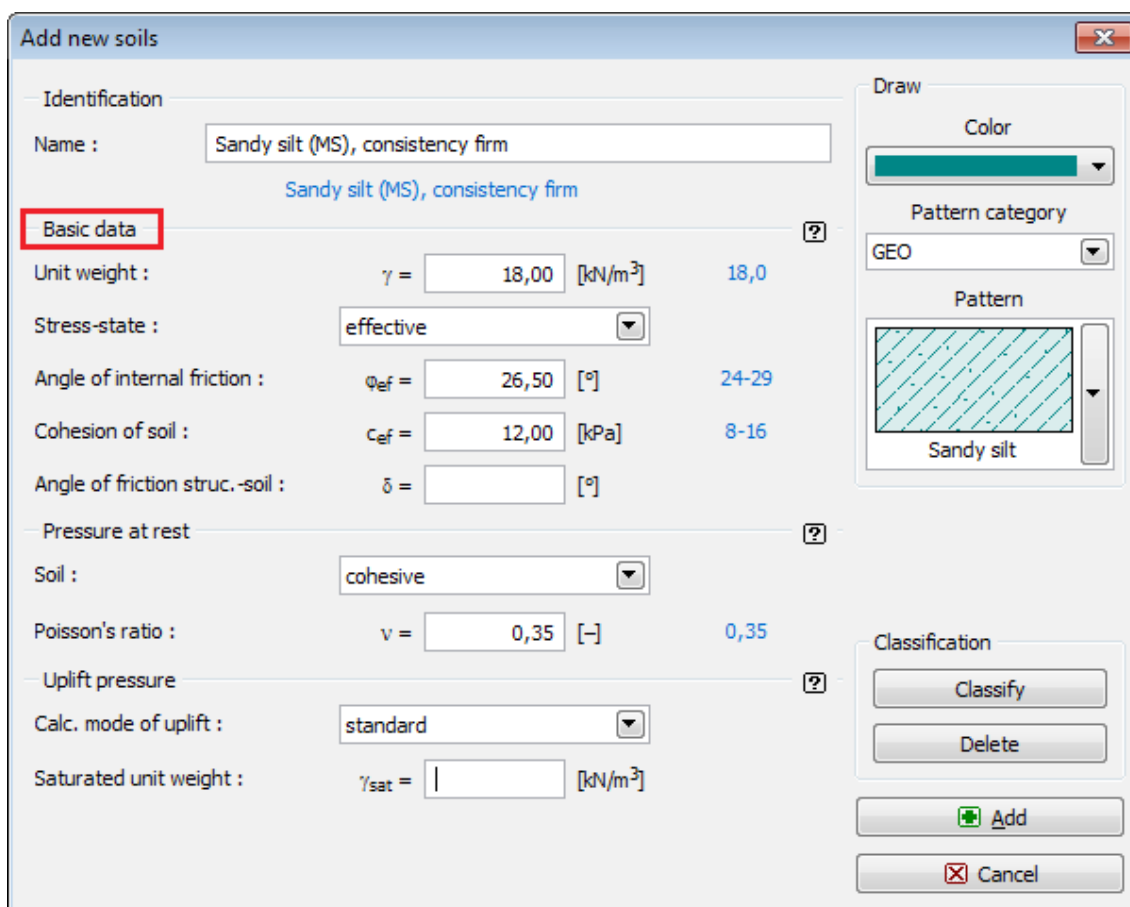
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil, type of load, structure duration and water conditions.

For **effective stress** further needs to specify the **angle of internal friction between the soil and structure**, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the [table of recommended values](#).

For **total stress** further needs to specify the **adhesion of soil to the structure face  $a$** .

The associated theory is described in detail in chapter "[Earth pressures](#)".



Dialog window "Add new soils" - "Basic data"

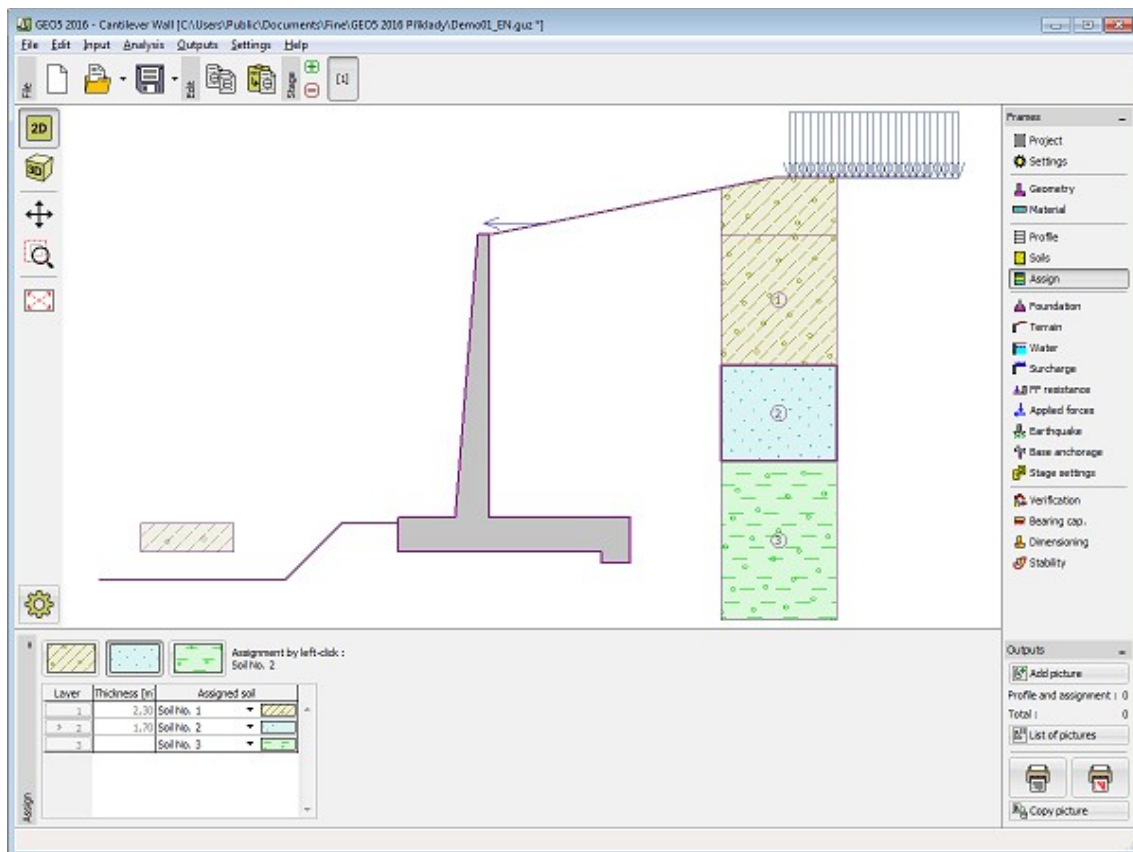
## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).





Frame "Assign"

## Foundation

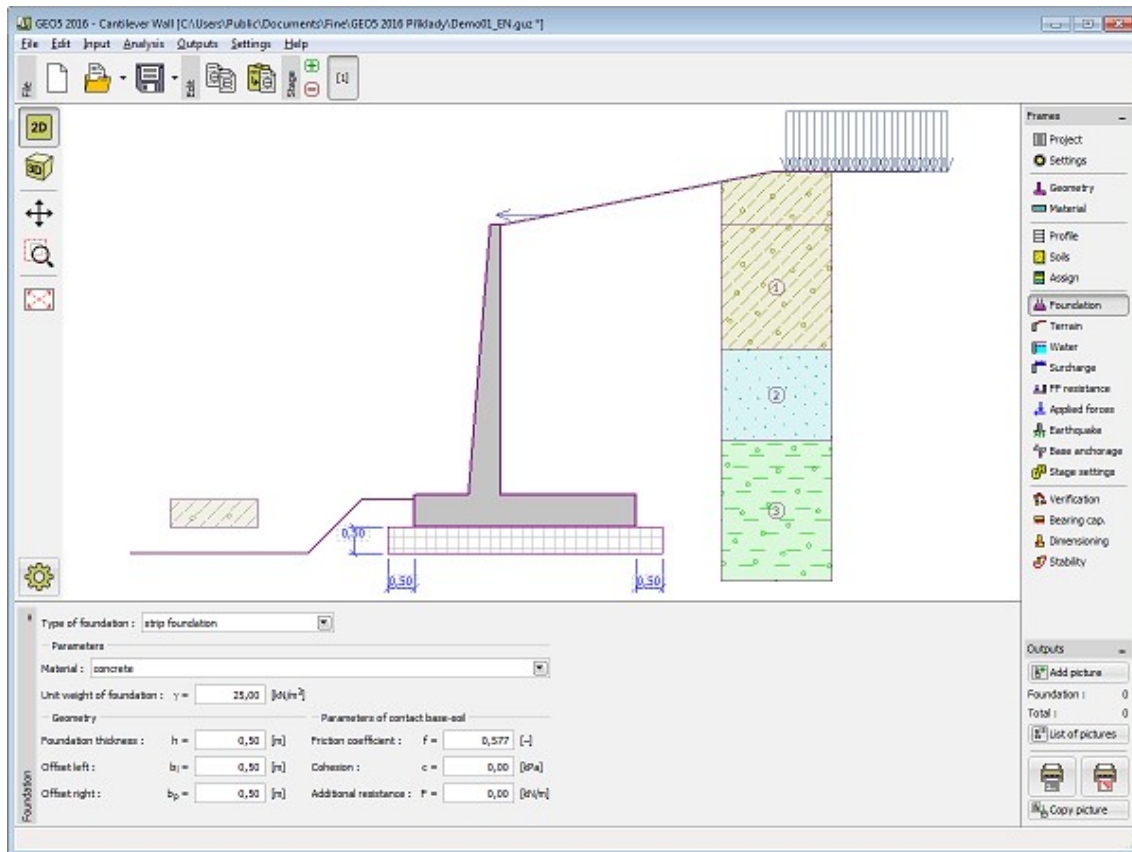
The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle  $\psi$  [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient  $\mu$  [-]. Both options require inputting the cohesion  $a$  [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material**  $\gamma$  and **parameters of contact base-soil** (friction coefficient  $f$ , cohesion  $c$ , additional resistance  $F$ ).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

**Strip foundation** and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of **active dimensions**.



The input data introduced in this frame influence the actual **wall analysis** (check for slip) and further the **bearing capacity of foundation soil**.



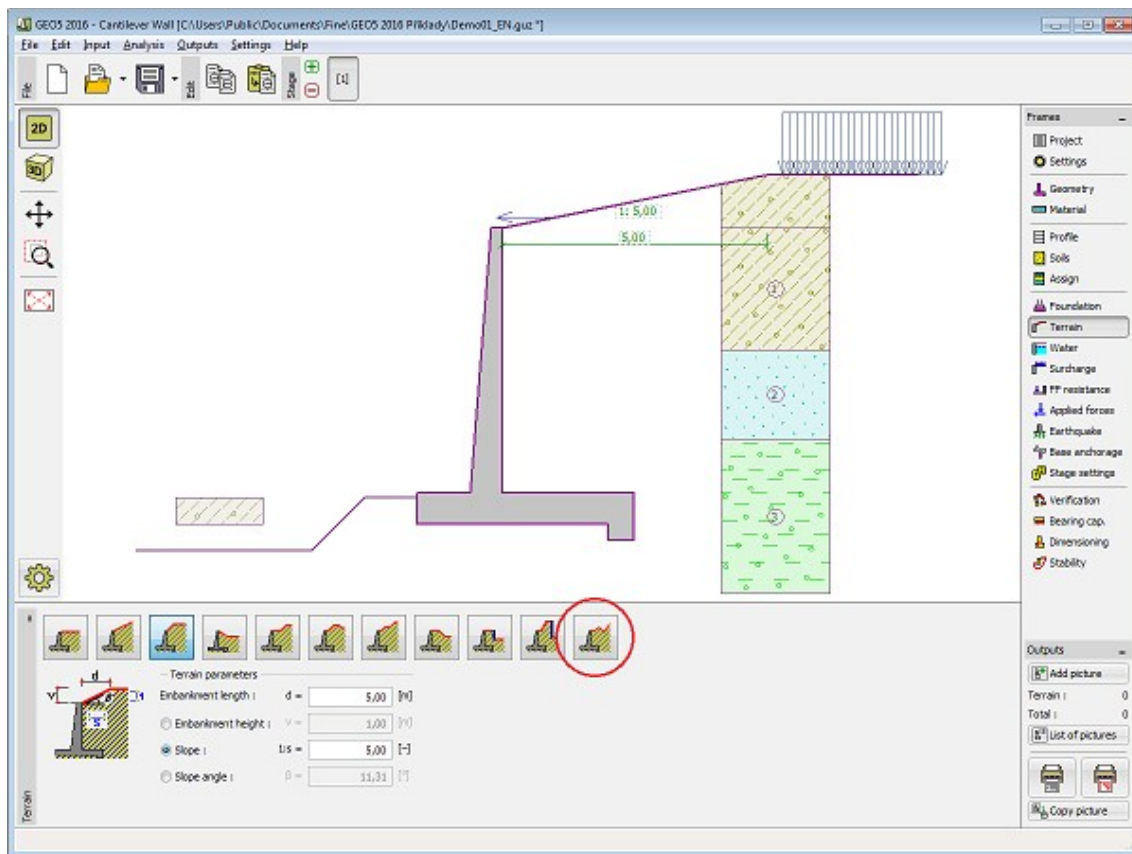
Frame "Foundation"

## Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

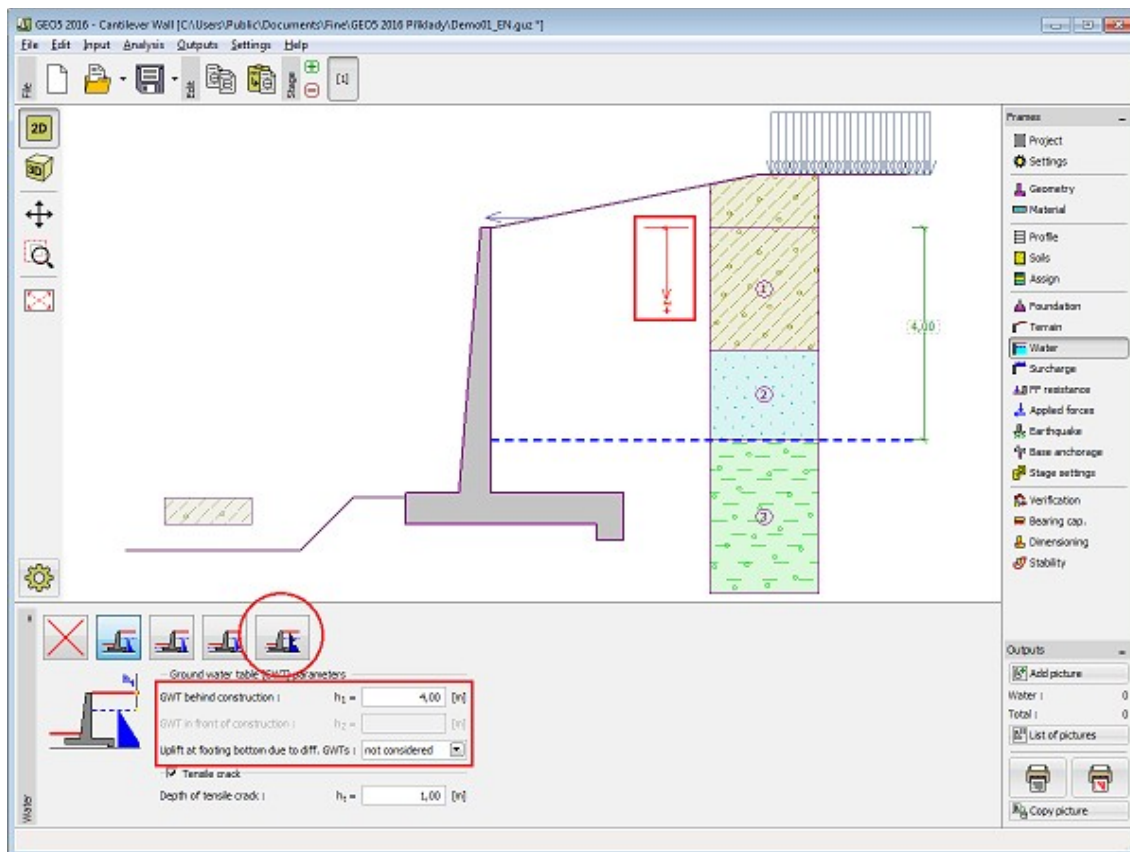
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be **linear, parabolic** or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".

The program further allows for specifying a depth of **tensile cracks** filled with water.



Frame "Water"

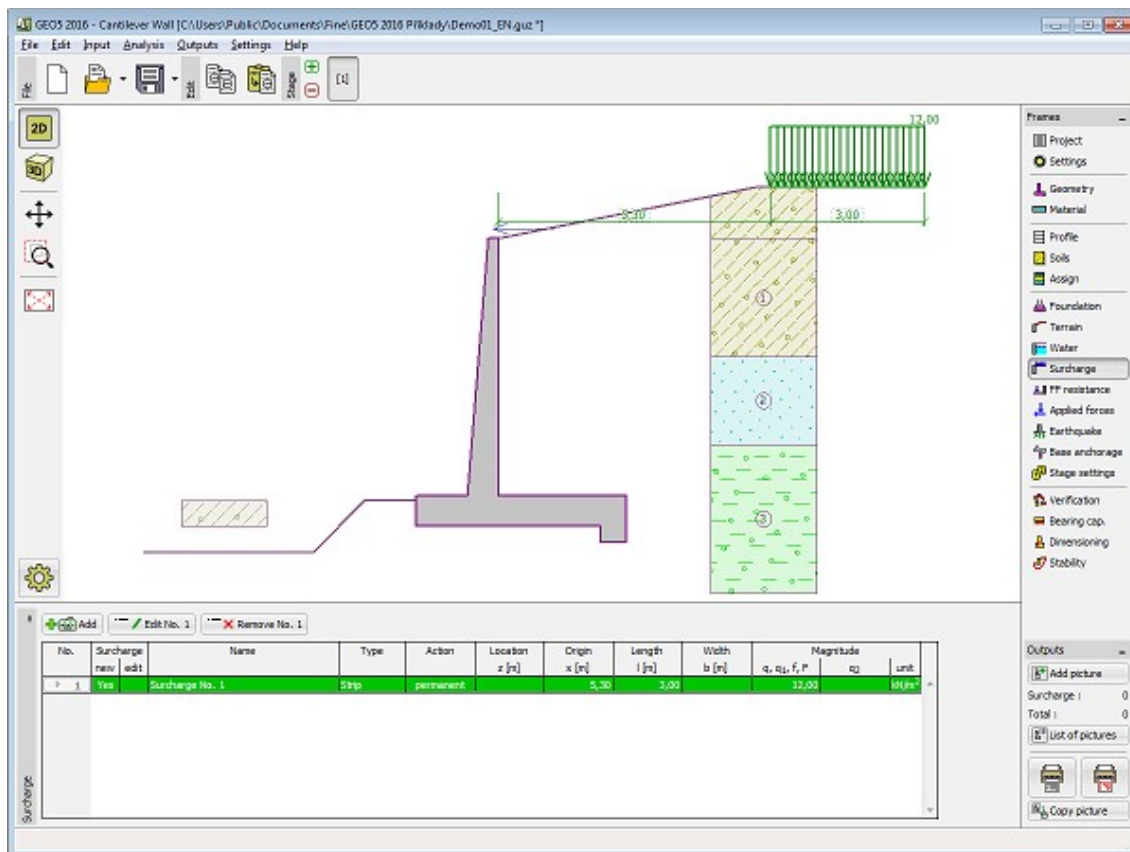
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".



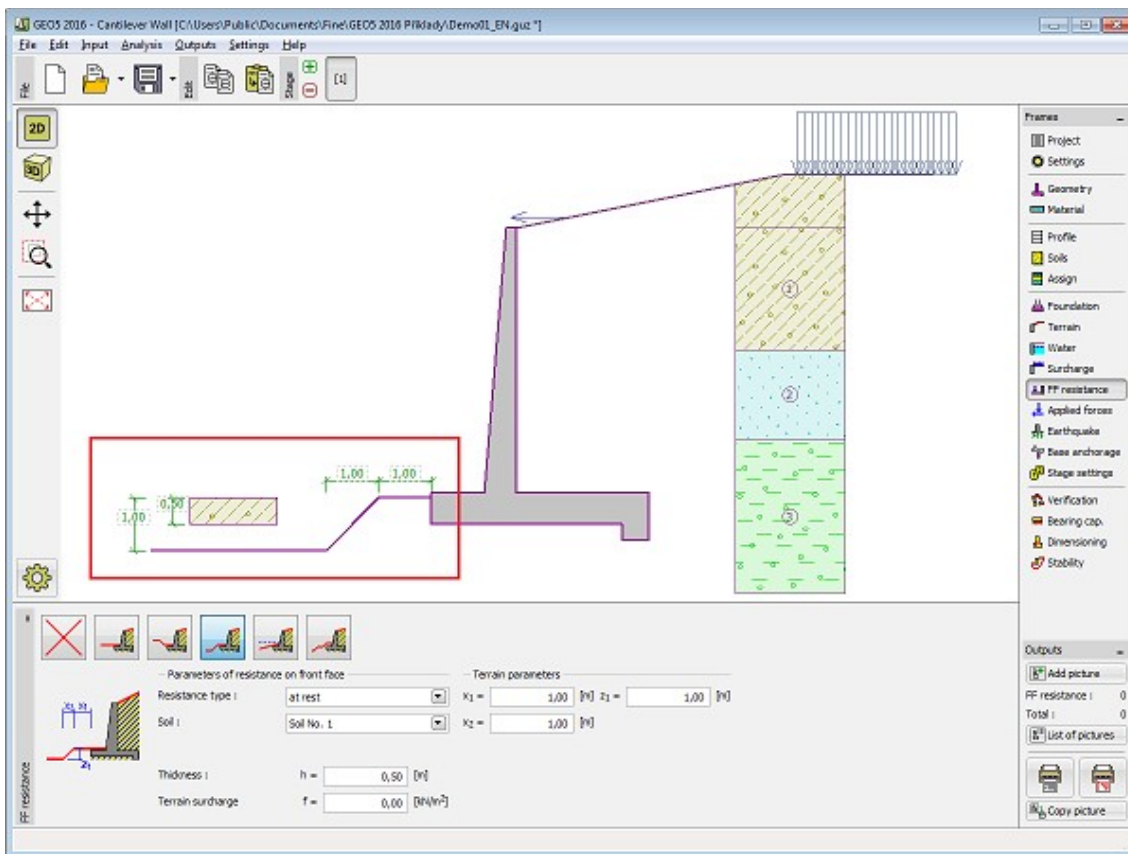
Frame "Surcharge"

## Front Face Resistance

The "**Front face resistance**" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the frame "**Soils**"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

The resistance on a structure front face can be specified as a **pressure at rest**, **passive pressure** or **reduced passive earth pressure**. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the input type of reduced passive pressure.



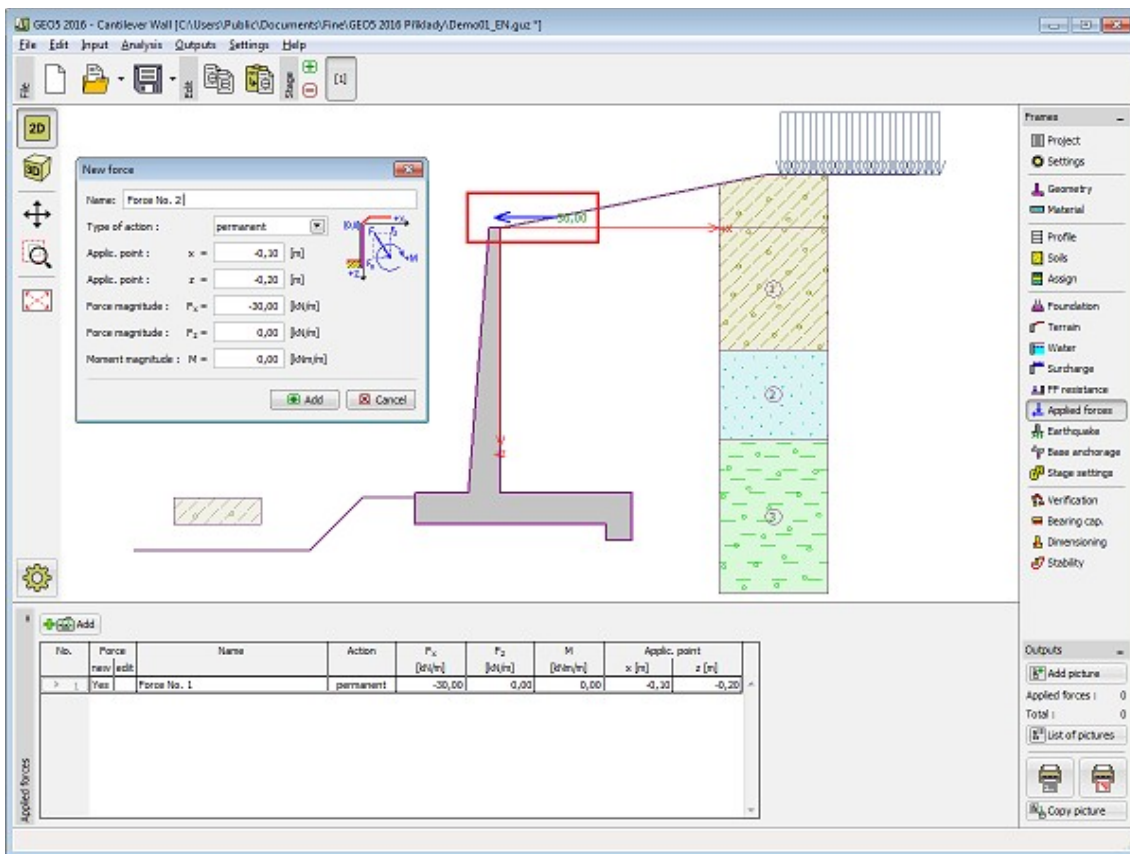
Frame "Front face resistance"

## Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

**Applied forces** represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.



Frame "Applied forces"

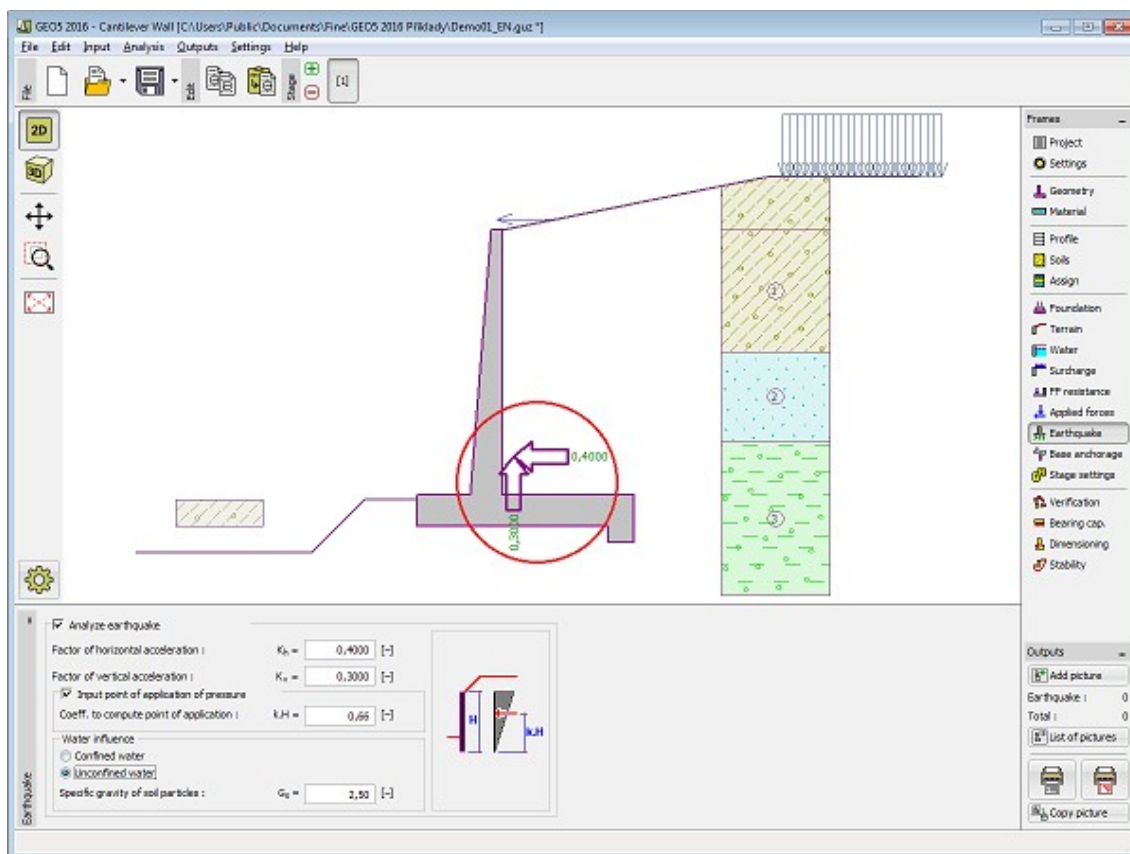
## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

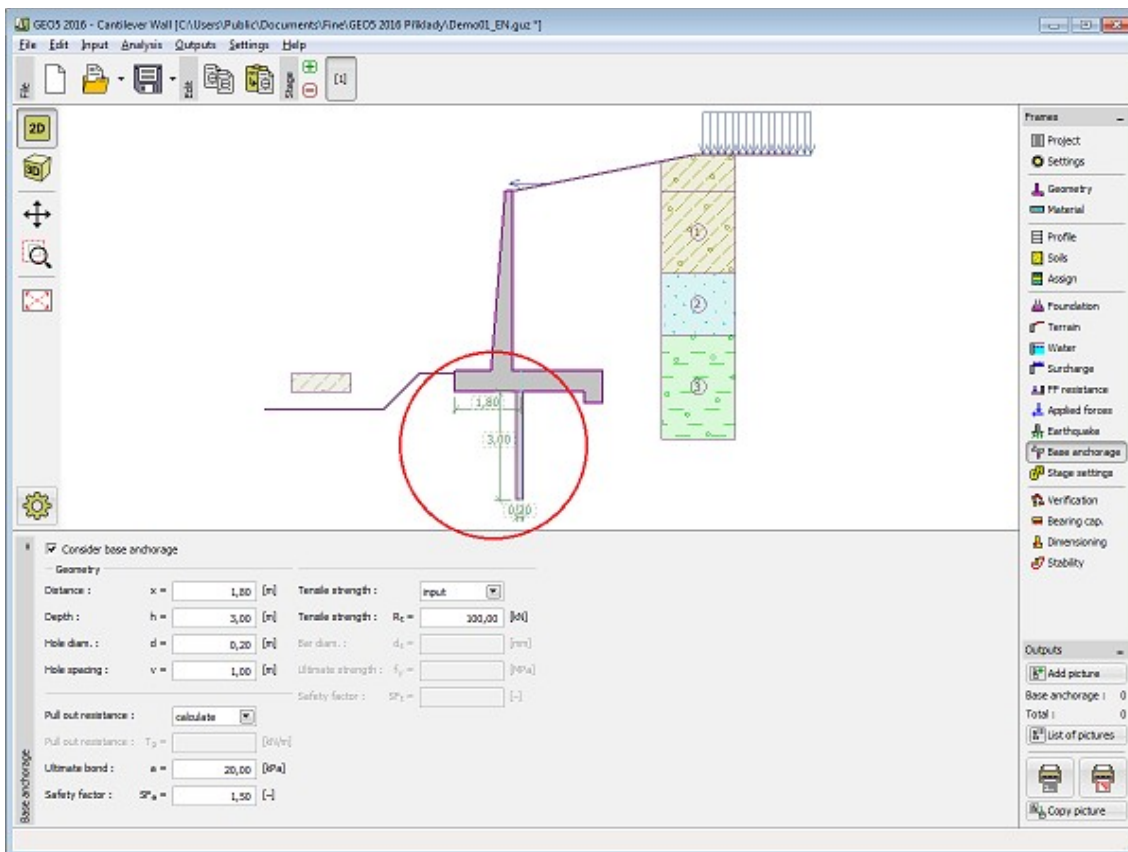




Frame "Earthquake"

## Base Anchorage

The frame "**Base anchorage**" serves to input parameters (anchorage geometry, bearing capacity against pulling-out and pulling-apart) specifying an anchorage of the wall foundation. Geometry of footing anchorage can be edited either in the frame by inserting values in the inputting boxes or on the desktop with the help of **active dimensions**. The bearing capacity values can be either input or **computed by the program** from the input parameters.



Frame "Base anchorage"

## Stage Settings

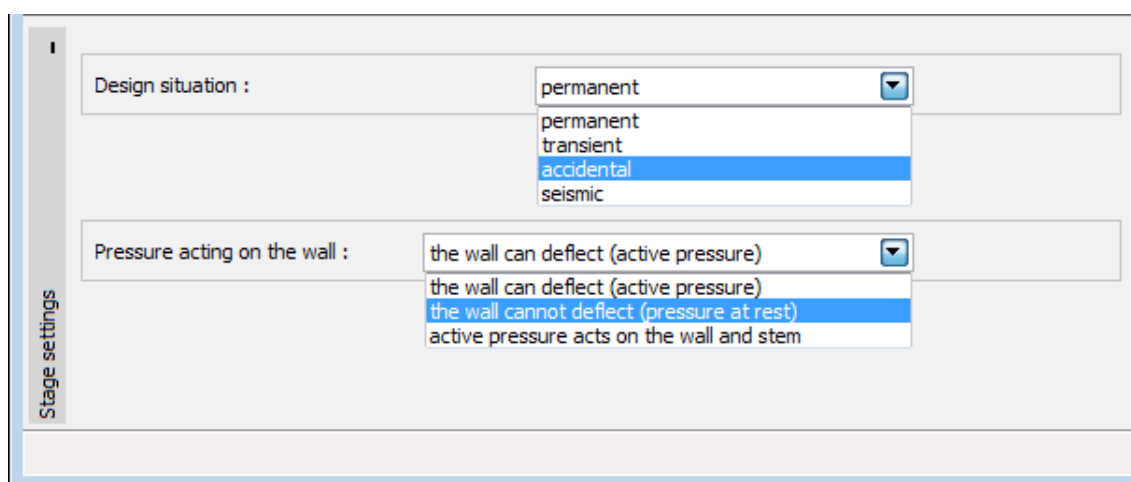
The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).

Next, the frame serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move, an active pressure is assumed, otherwise, a pressure at rest is used. The third option enables to load both the wall and stem by an active pressure.





Frame "Stage settings"

## Verification

The frame "**Verification**" shows the analysis results. **Several computations** can be carried out for a single task.

The frame appearance is adjusted based on the selected **verification methodology**.

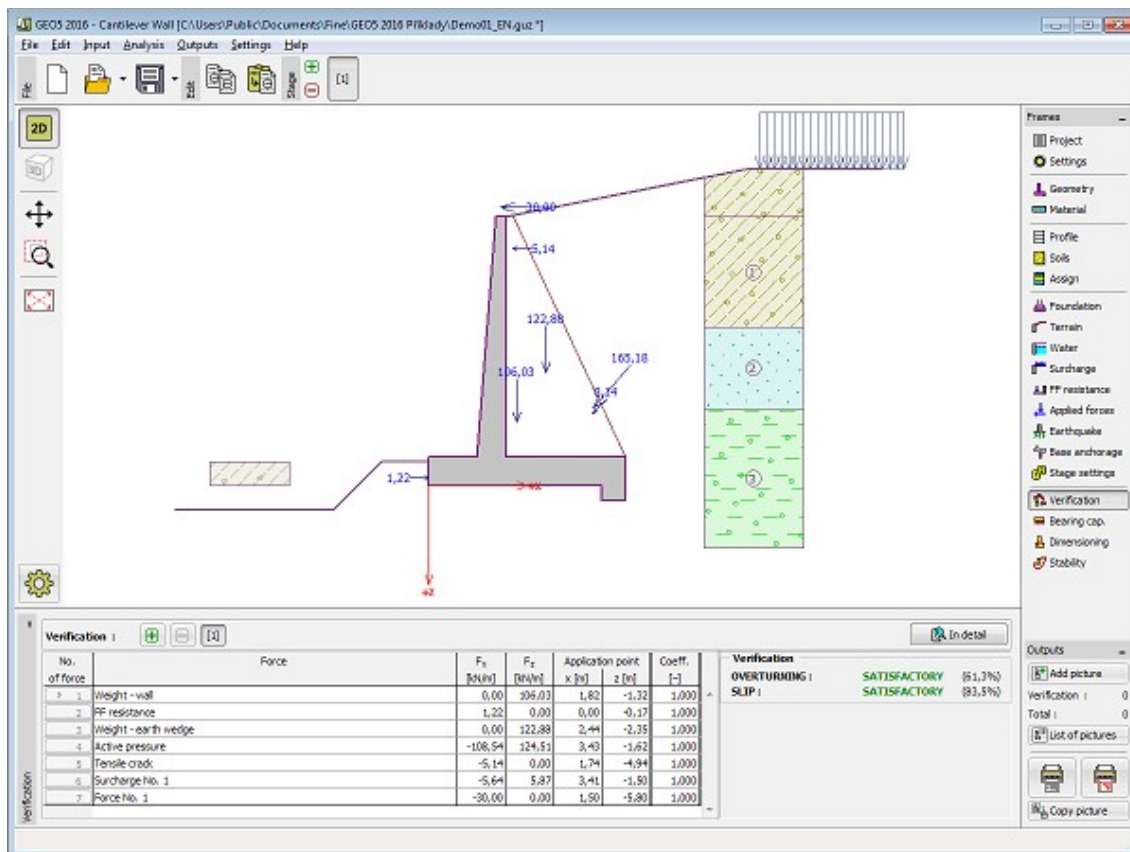
- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

The wall is loaded either by **active pressure** or **pressure at rest** depending on input in the frame "**Stage settings**".

The procedure for **wall verification** is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning** and **translation**. The "**In detail**" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Verification"

## Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "**Verification**". The program "**Spread footing**" then considers all verifications as load cases.

The frame contains following analysis options:

- Insert bearing capacity of foundation soil** The foundation soil bearing capacity is input. The **eccentricity** and **bearing capacity** analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays detailed listing of the results.
- Analyse bearing capacity by program "Spread footing"** Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**Cantilever wall**". The "**Spread footing**" program must be installed for the button to be active. Overall length of wall foundation is input.
- Analyse bearing capacity by program "Pile"** The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run**"

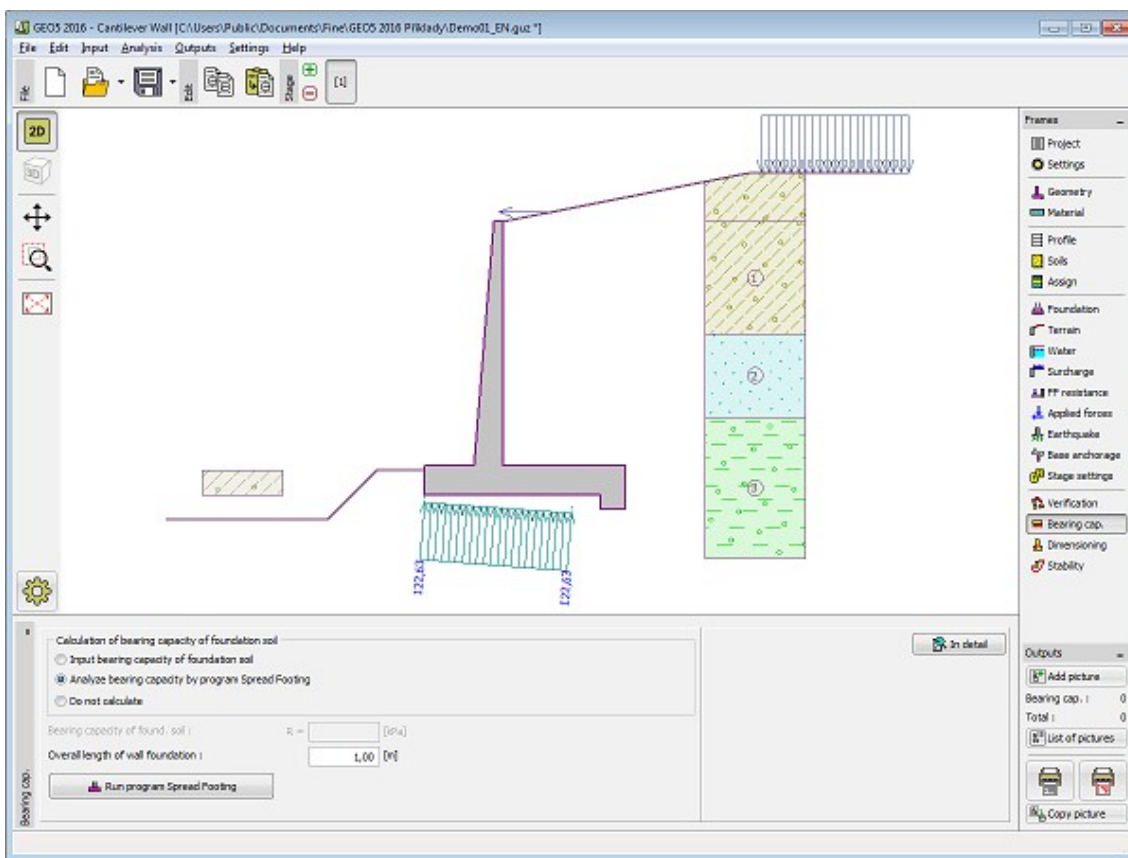
**program Pile**" is available if the wall has pile foundation (frame "Foundation"). Pile spacing  $s$  is input.

- Analyse bearing capacity by program "Pile group"**

The procedure is identical as if calculating soil bearing capacity by the "Spread footing" program. The "Run program Pile group" is available if the wall has pile foundation with more then one pile (frame "Foundation"). Pile spacing  $s$ , overall number of pile rows  $n$  and loading length  $l$  are input.
- Do not calculate (pile footing)**

The foundation soil bearing capacity is not calculated.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Bearing capacity"

## Dimensioning

The "Dimensioning" frame serves to design and verify the reinforcement of wall cross-section - the cross-section subjected to dimensioning is selected in the combo list.

- Wall stem verification**
- Construction joint verification**

**depth** of construction joint from construction top edge is specified
- Wall jump verification**

- **Verification of heel of wall**

The frame appearance is adjusted based on the selected [verification methodology](#).

- Verification according to the [factor of safety](#), or the theory of [limit states](#) - the last column in the table allows for inputting the [design coefficients](#), which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- [Analysis according to EN 1997](#) - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "[Load combinations](#)".
- [Analysis according to LRFD](#) - in the case the last column is not displayed.

Calculation of forces and their action on the analyzed cross-section is described [here](#).

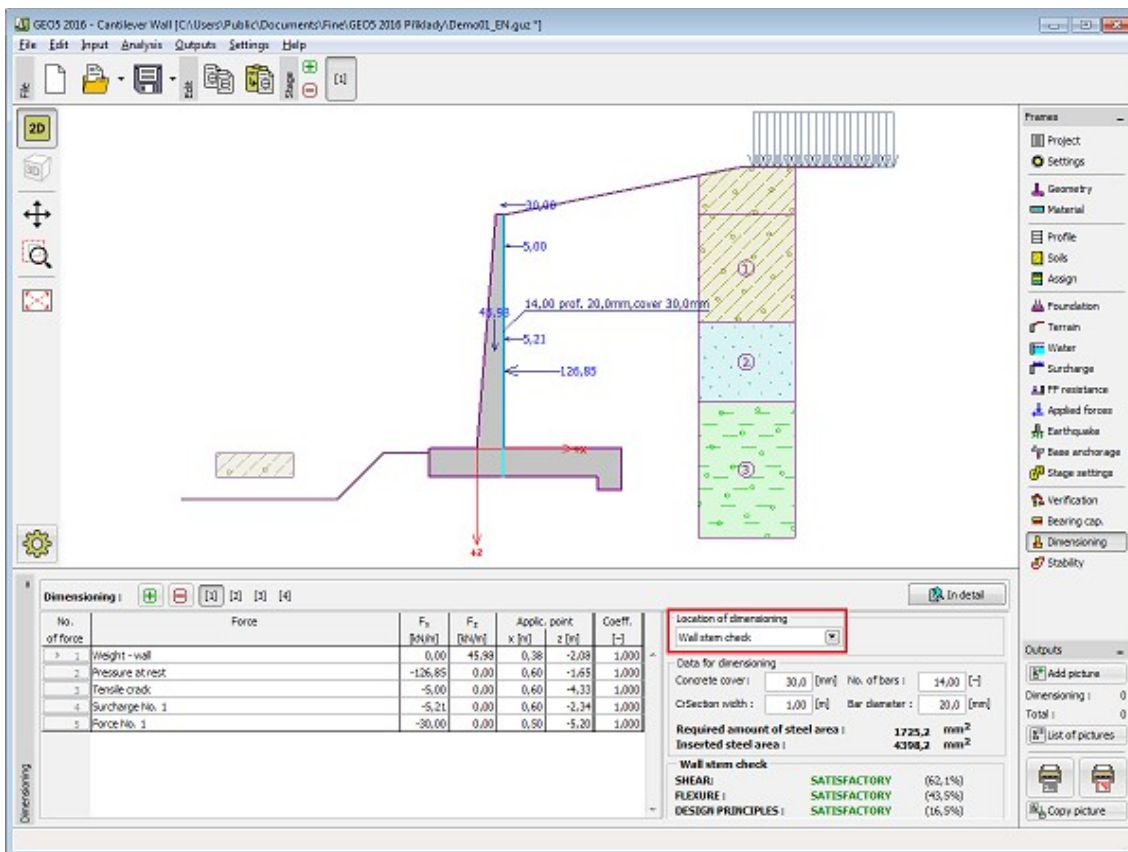
The wall stem and construction joint are always loaded by the [pressure at rest](#). When verifying the front wall jump the wall is loaded either by the [active pressure](#) or the [pressure at rest](#) depending on input specified in the frame "[Stage settings](#)".

The [procedure to derive distribution of internal forces](#) in individual cross-sections is described in the theoretical part of this documentation. [In addition, force from earth pressure at rest](#) is taken into account when considering earthquake analysis.

[Dimensioning of the reinforced concrete](#) is performed according to the standard set in the "[Materials and standards](#)" tab.

[Several computations](#) for various cross-sections can be carried out. Various [design coefficients](#) of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "[Visualization settings](#)".

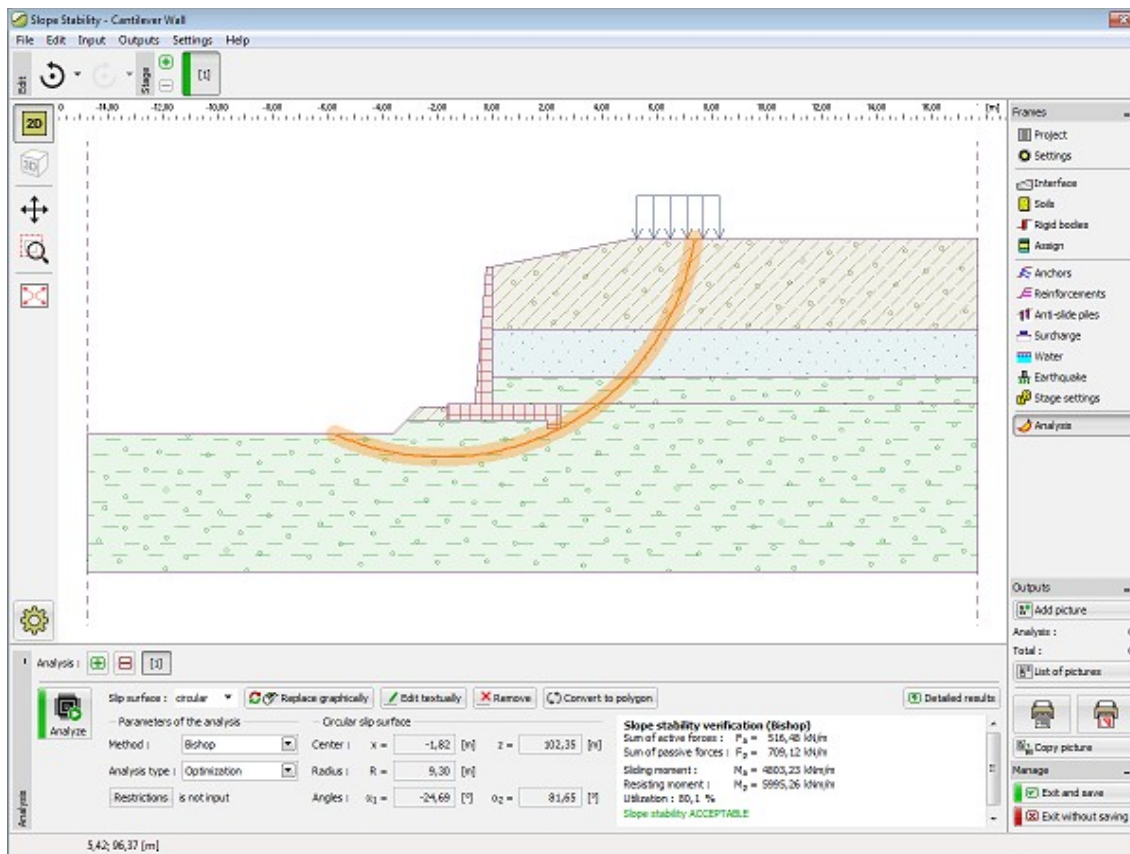


Frame "Dimensioning"

## Stability

Pressing the **"Stability"** button launches the **"Slope stability"** program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program **"Slope stability"** is installed.

After completing all analyses press the **"OK"** button to leave the program - all data are then carried over to the analysis protocol of the **"Cantilever wall"** program.



Frame "Stability"

## Program Gravity Wall

The program is used for analysis of gravity walls. It offers a range of wall shapes and verifies mass concrete cross-sections.

The help in the program "Gravity Wall" includes the following topics:

- Input of data into individual frames:

Project	Settings	Geometry	Material	Profile	Soils	Assign
Foundation	Terrain	Water	Surcharge	Front Face Resistance	Applied Forces	Earthquake
Stage Settings	Verification	Bearing Capacity	Dimensionin g	Stability		

- Standards and analysis methods

- Theory for analysis in the program "Gravity wall":

Stress in Soil	Earth Pressures	Analysis of Walls	Analysis of	Dimensioning of
----------------	-----------------	-------------------	-------------	-----------------



Body

Foundation  
Bearing Capacity Concrete  
Structures

- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.

The screenshot shows the 'Project' frame with the following data:

Task :	Terraces Hanspaulka	Author :	James Baker
Part :	South-facing slope III.	Date :	28.10.2005
Description :	Support walls 2-5m	Project ID :	845/2014
Customer :	Belltrade LTd.	Project number :	11486/2014
System of units	metric		

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in

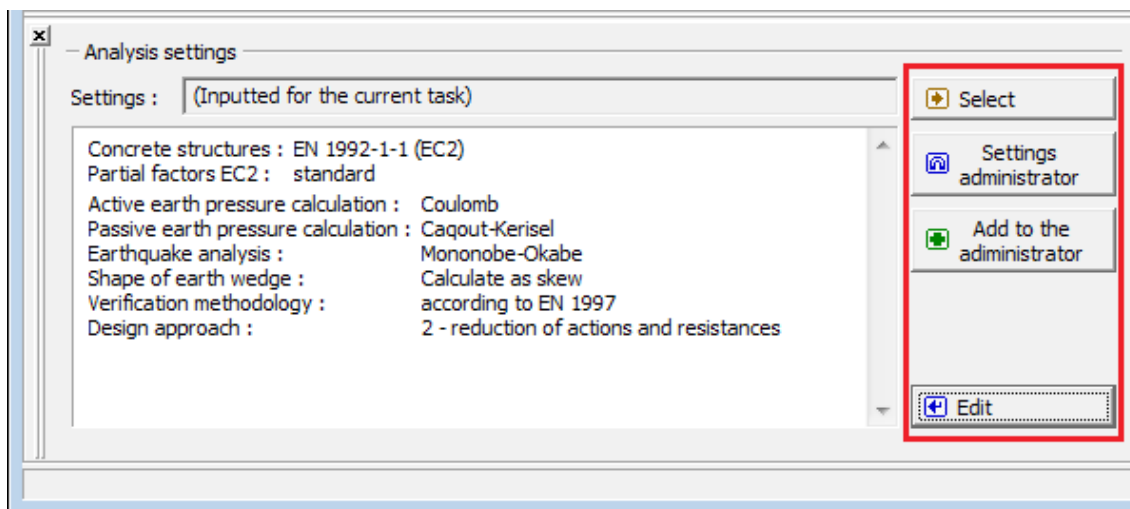
the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Materials and standards"** and **"Wall analysis"** tabs.



Frame "Settings"

## Geometry

The **"Geometry" frame** allows to select the desired wall shape. The selected shape with a graphic hint (**"Chart of wall geometry"**) appears in the left part of the frame. The shape of the wall can be edited either in the frame by inserting values into the input fields, or on the desktop using **active dimensions**.

In case the structure is composed of inclined segments, it is required to enter the ratio of sides of an inclined segment  $l:x$ . A **straight structure** is specified by entering the value zero.



Defined wall shapes

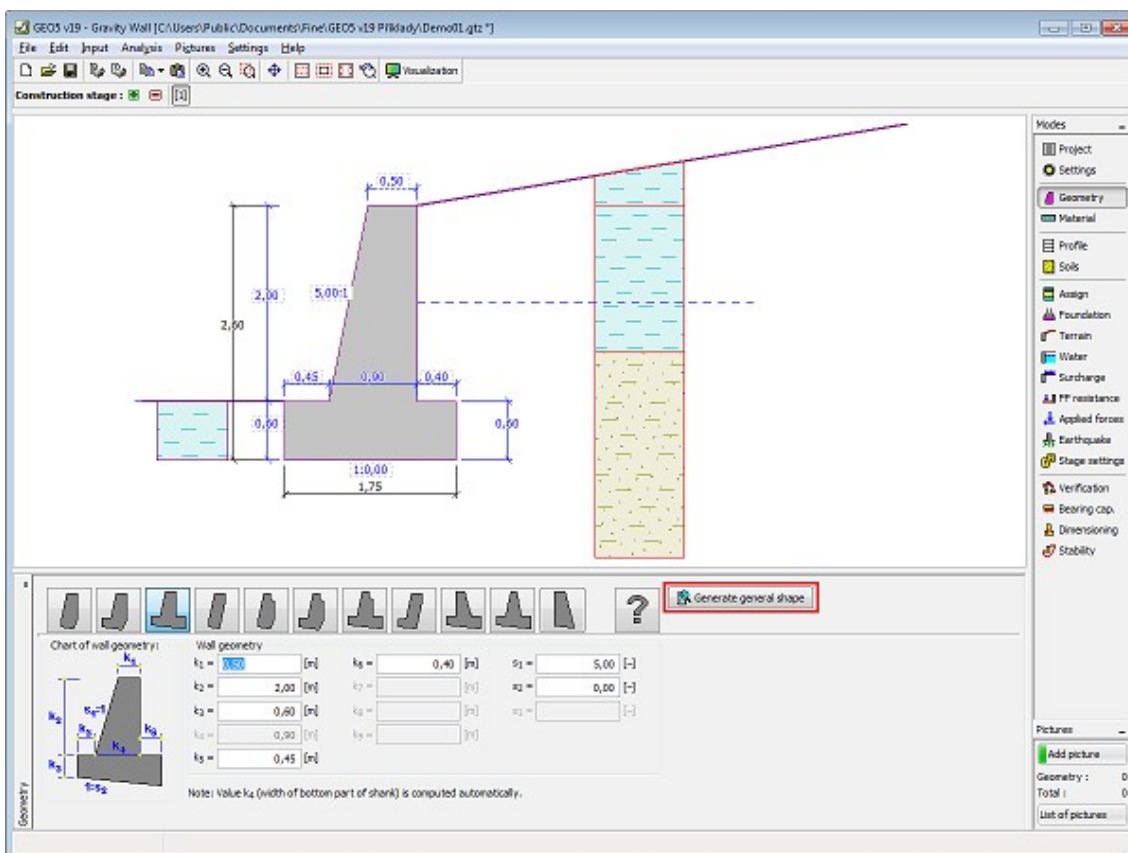
If defined wall shapes are not satisfactory for the wall geometry input, program allows to enter **general shape of gravity wall**. General wall shape is entered by coordinates of points, but it is also possible (by pressing the button **"Generate general shape"**) to generate coordinates of structure from already input predefined wall.



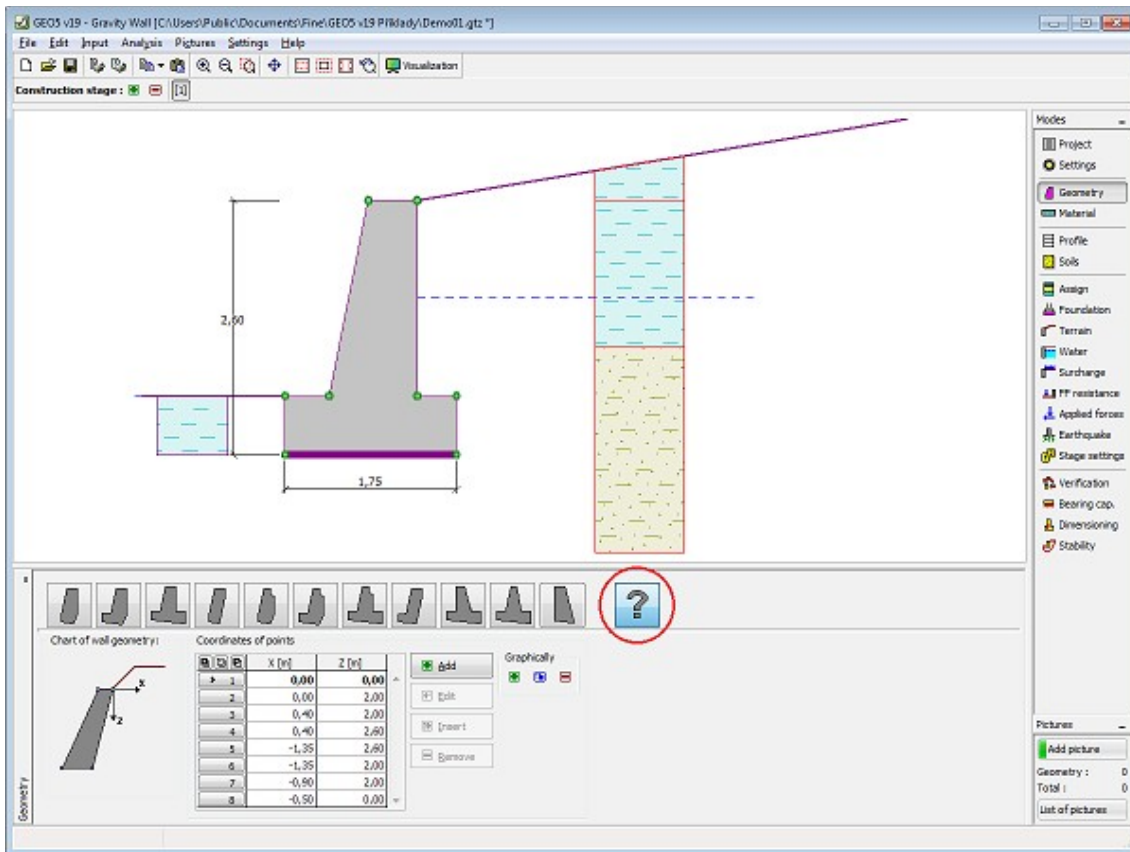


*General wall shape input*

The program allows to **export** the geometry of the structure in \*.DXF format.



Frame "Geometry"



Frame "Geometry" - general wall shape

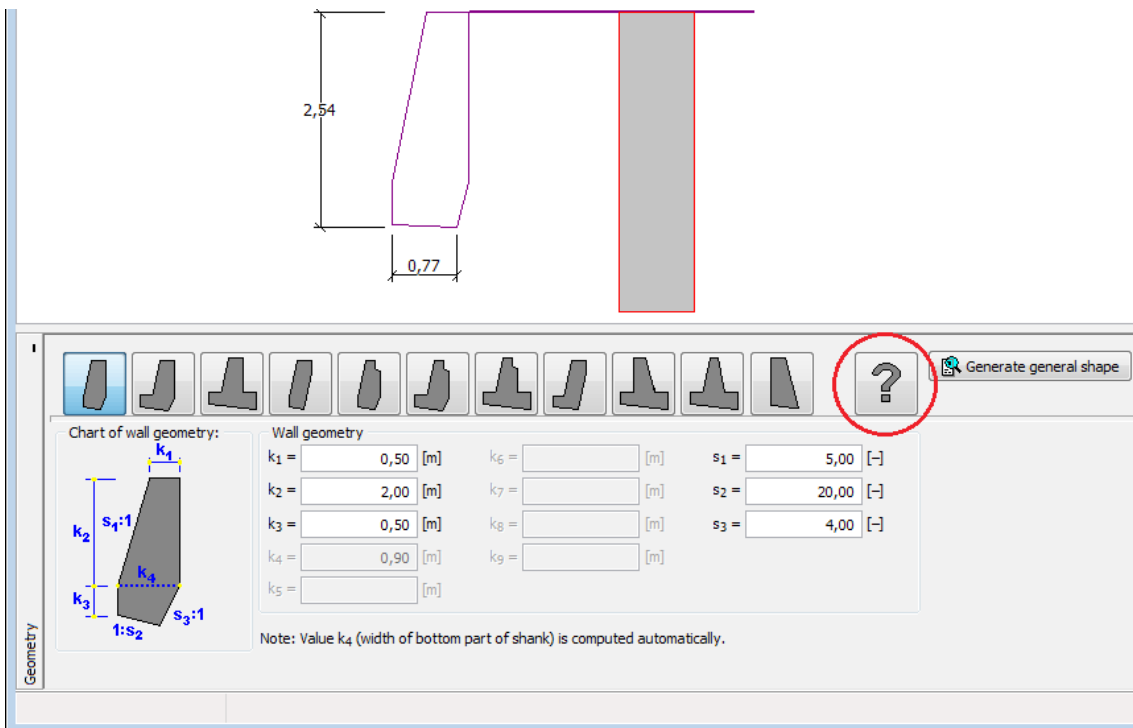
## General Wall Shape

### Input of general wall shape in a new task

The program allows to input general wall shape in two ways:

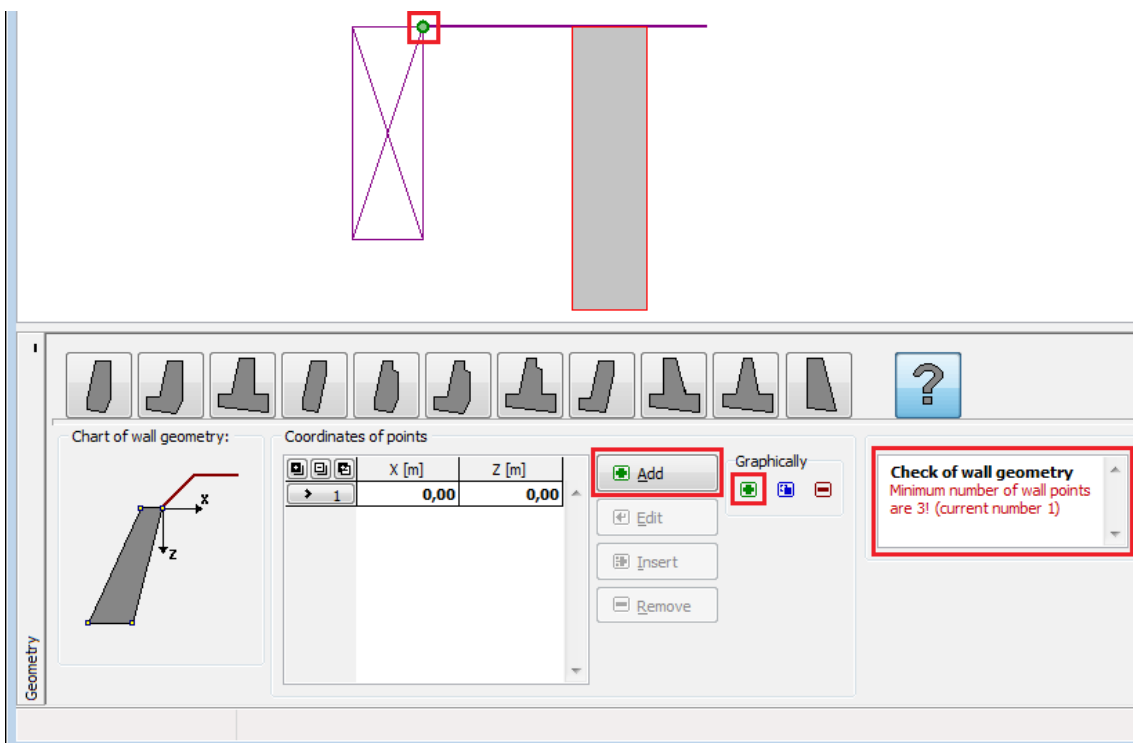
#### 1. Input of general wall shape using points

Pressing the icon with question mark on the **"Wall geometry scheme"** tool bar, will create a rectangular shape and a first structure point [0,0]. Minimum input number of points is 3 (in case less number is input, the program will show an error message).




Frame "Geometry" - new task

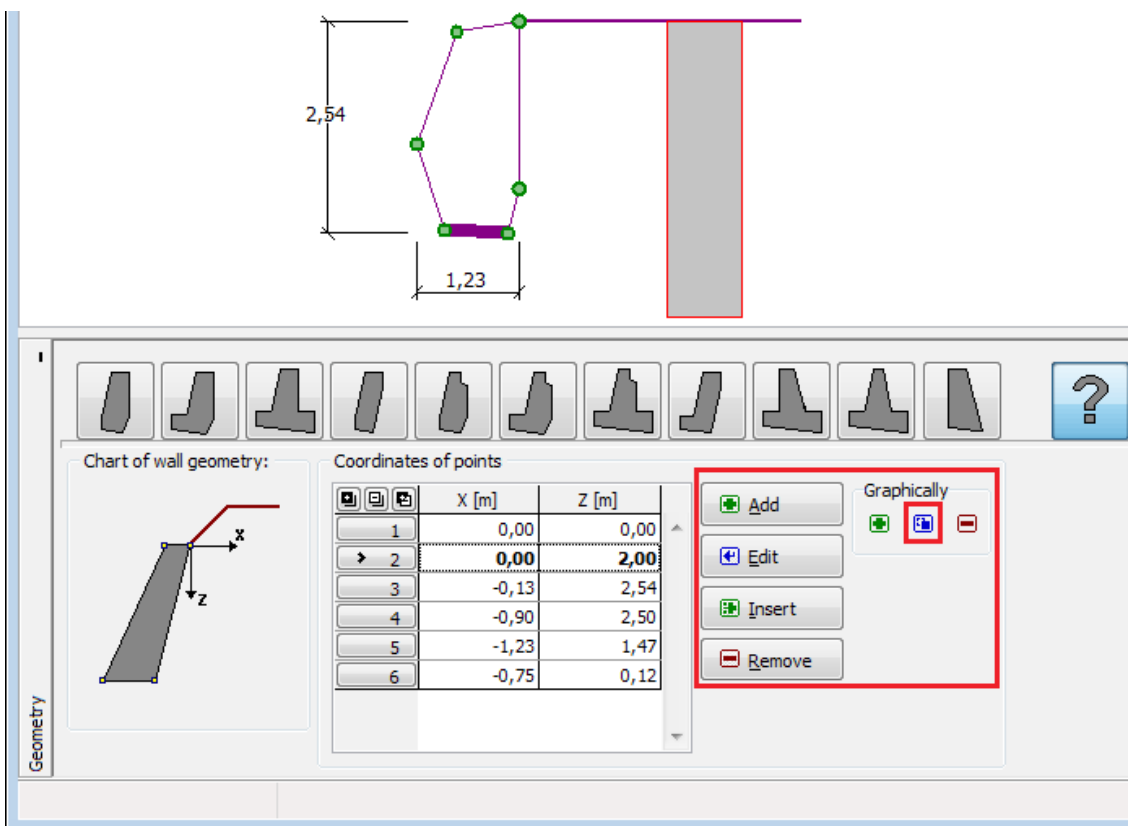
Using the **"Add"** button, which opens a dialog window **"New point"**, more structure points are input (it is possible to input points by clicking on the desktop).



Frame "Geometry" - general wall shape input using points

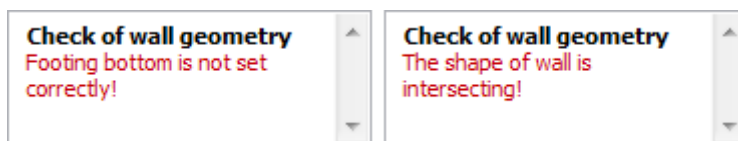
Input points are being added to the table, and it is then possible to edit them, insert more in between and delete them using the desired buttons **"Edit"**, **"Insert"** and **"Remove"** or by

clicking the points on the desktop in the corresponding mode. Points can be moved right on the desktop by mouse after clicking on the special icon .



Frame "Geometry" - edit points

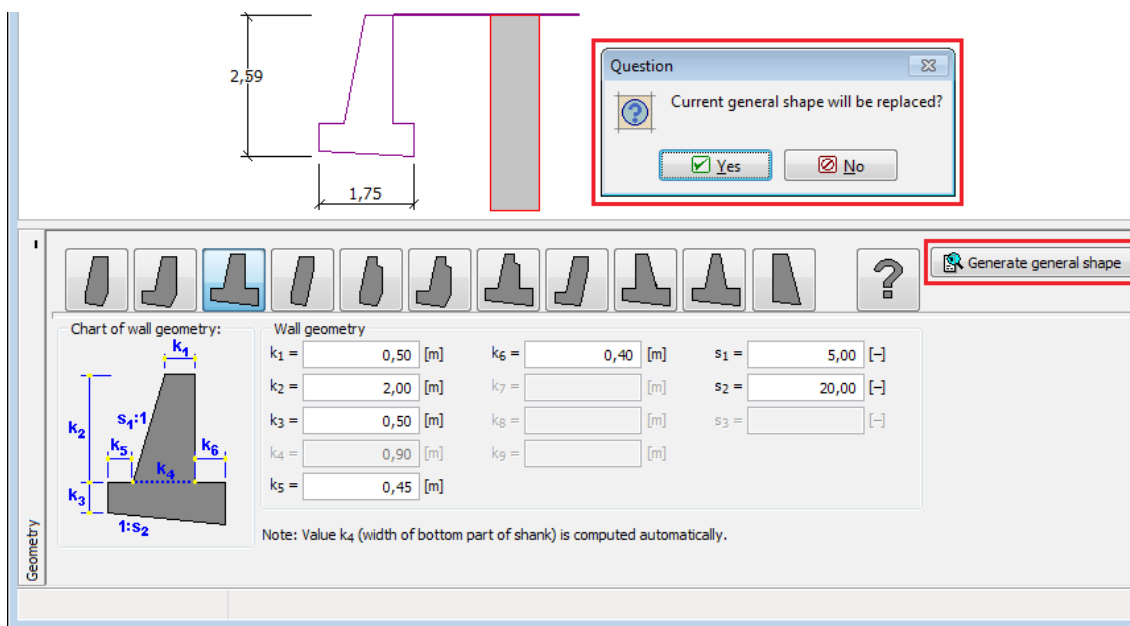
The maximum inclination of footing bottom is considered 45°. In case of wrong inclination of footing bottom input, or if the wall shape is intersecting, the program checks the general geometry of the wall and warns the user of an error. In that case, the wall geometry has to be changed.



Frame "Geometry" - point input error message

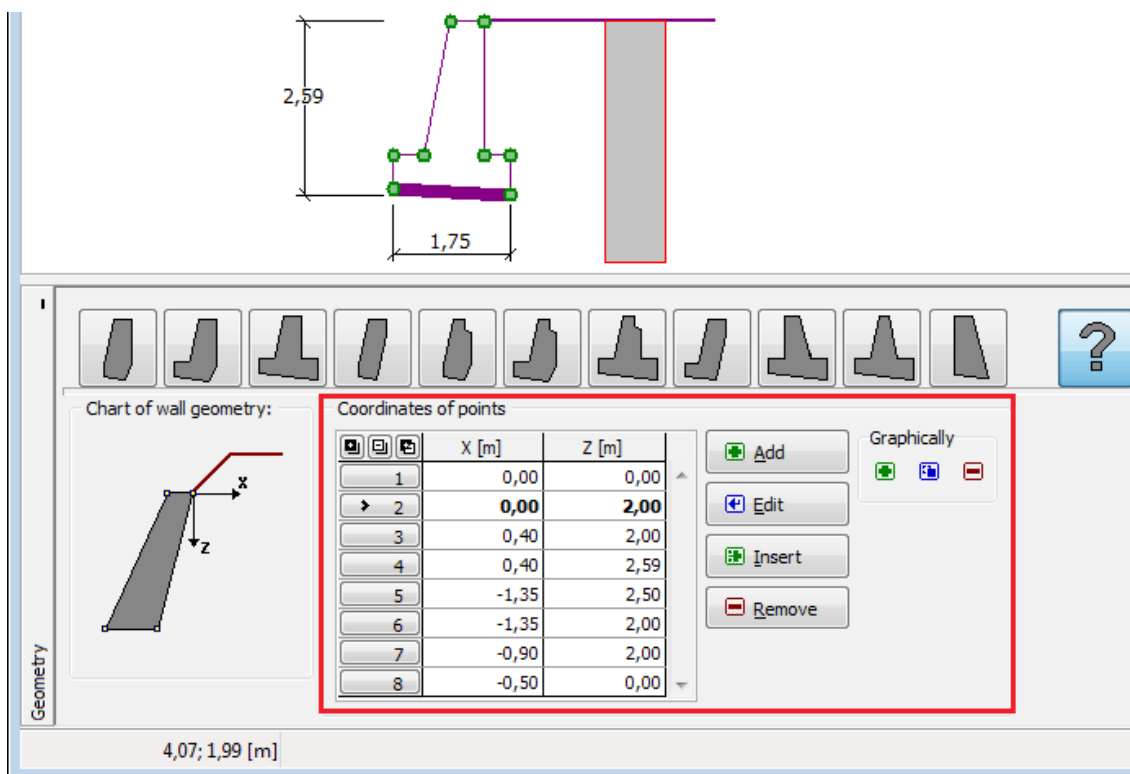
## 2. Input of general wall shape using general shape generator

The structure defined by the scheme of construction and its dimensions can be taken to the general wall shape input by pressing the "**Generate general shape**" button. It is possible to work with the newly generated points and edit the generated wall shape.



Frame "Geometry" - input of general wall shape using general shape generator

Frame appearance is then changed as in the first case of general wall shape input. It is possible to work with the picture of the structure as already described.



Frame "Geometry" - frame appearance after point input

## Material

The "Material" frame allows to select the material parameters (concrete, stone masonry). The

wall input weight is always input into the field in the upper part of the frame.

If the wall is made from concrete, the program offers two options:

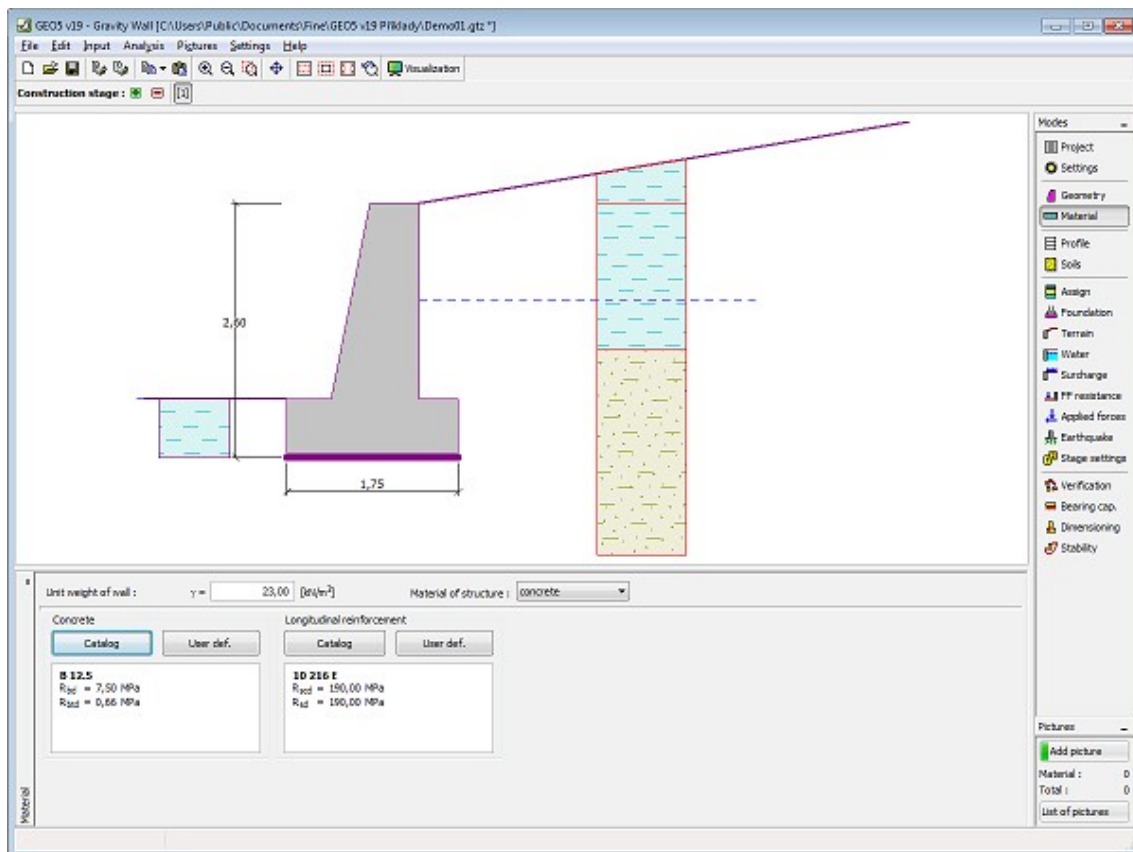
- The "**Catalog**" button opens the "**Catalog of materials**" dialog window (for concrete or steel reinforcements), the desired material is then selected from the list.
- The "**User defined**" button opens the "**Editor of material - Concrete**" dialog window (for concrete) or the "**Editor of material - Reinforcing steel bars**" dialog window (for longitudinal steel reinforcements), which allows to specify the material parameters manually.

The content of catalogs depends on the selection of standard for the **design of concrete structures** set in the "**Materials and standards**" tab.

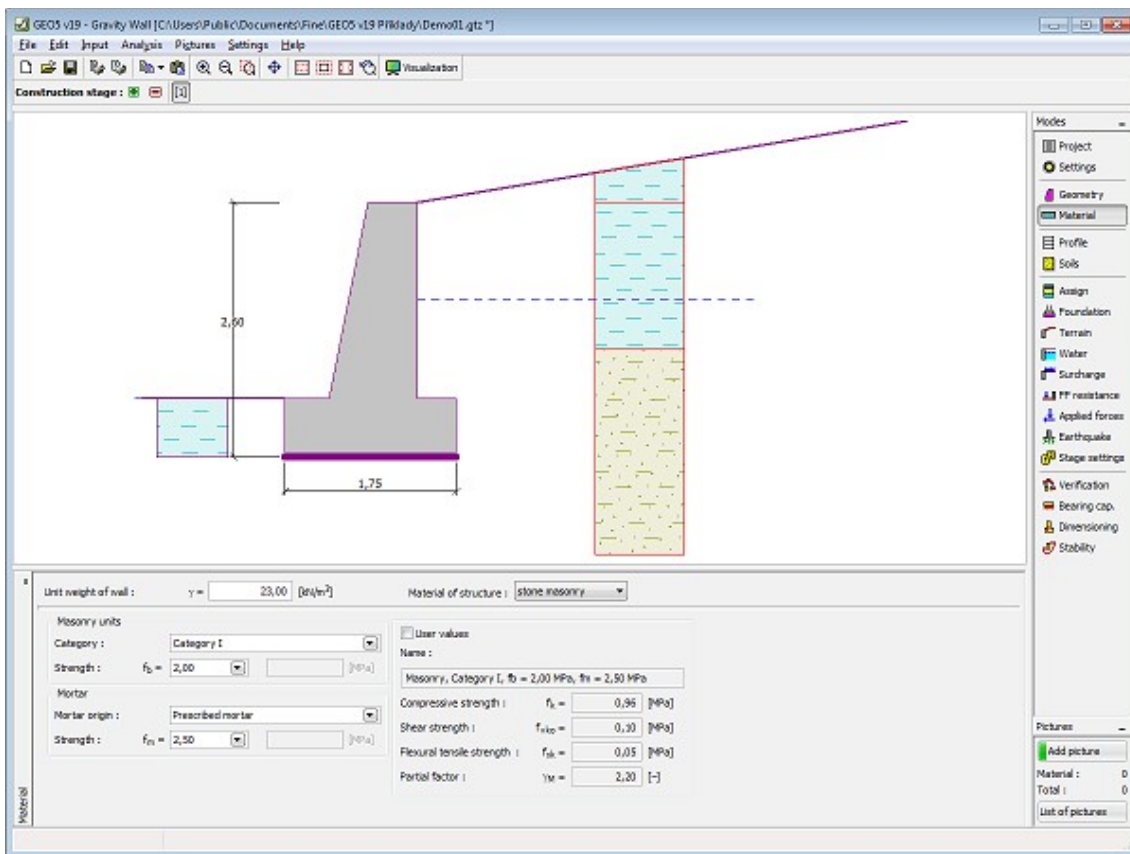
If the wall is made from stone masonry, material characteristics of masonry units according to selected standard are specified in the "**Materials and standards**" tab:

- **EN 1996-1-1** - strength of masonry units  $f_b$ , mortar origin and strength of mortar  $f_m$
- **GB 50003-2011** - type of masonry units, strength grade of stone, strength grade of mortar

If the "**User values**" button is checked, it is possible to input user defined material characteristics.



Frame "Material" - concrete



Frame "Material" - stone masonry

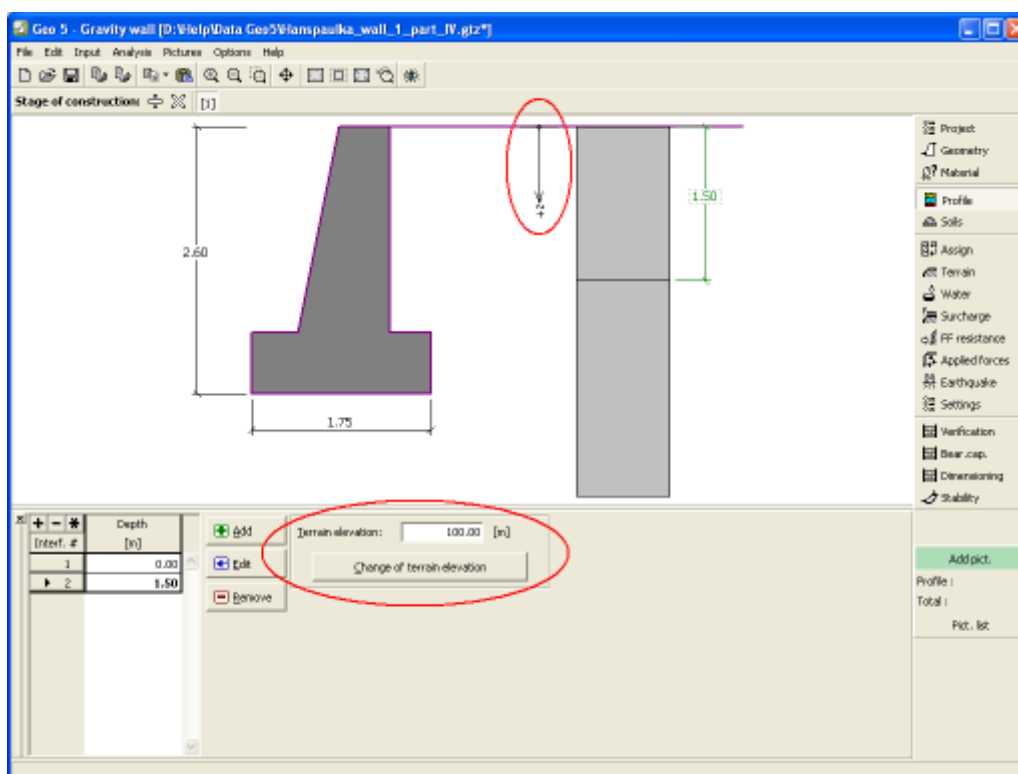
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "**New interface**" dialog window. The  $z$ -coordinate measured from the top point of a structure is specified ( $z$ -axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to import a profile in the gINT format.



The frame "Profile"

## Soils

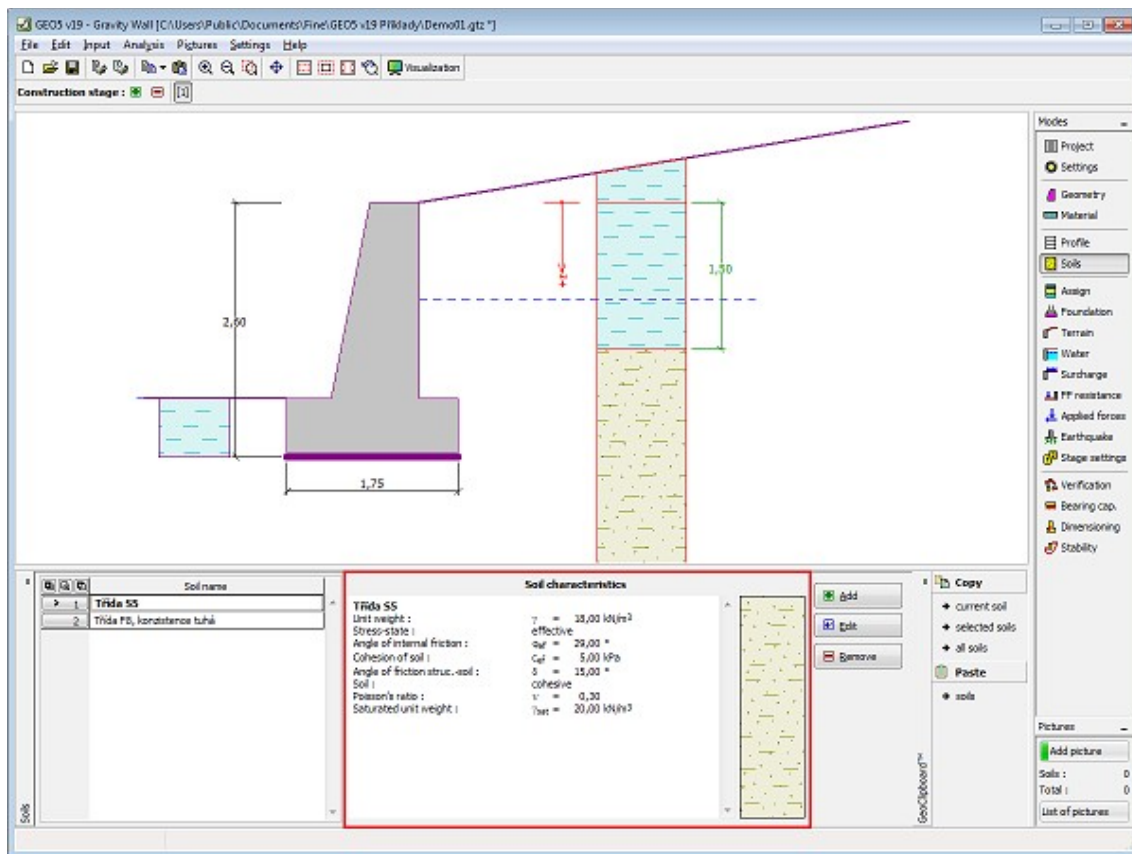
The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".





Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil, type of load, structure duration and water conditions.

For **effective stress** further needs to specify the **angle of internal friction between the soil and structure**, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the [table of recommended values](#).

For **total stress** further needs to specify the **adhesion of soil to the structure face  $a$** .

The associated theory is described in detail in chapter "**Earth pressures**".

**Add new soils**

Identification

Name :   
Sandy silt (MS), consistency firm

Basic data

Unit weight :  $\gamma =$   [kN/m<sup>3</sup>] 18,0

Stress-state :

Angle of internal friction :  $\phi_{ef} =$   [°] 24-29

Cohesion of soil :  $c_{ef} =$   [kPa] 8-16

Angle of friction struc.-soil :  $\delta =$   [°]

Pressure at rest

Soil :

Poisson's ratio :  $\nu =$   [-] 0,35

Uplift pressure

Calc. mode of uplift :

Saturated unit weight :  $\gamma_{sat} =$   [kN/m<sup>3</sup>]

Draw

Color

Pattern category

Pattern

Classification

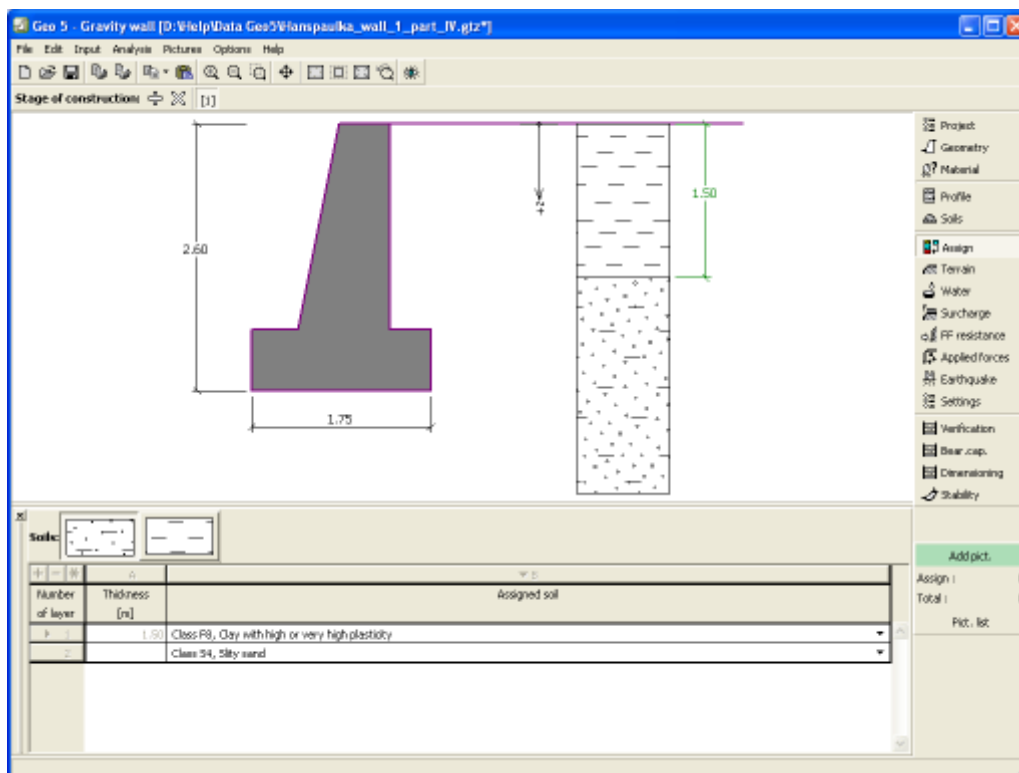
*Dialog window "Add new soils" - "Basic data"*

## Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

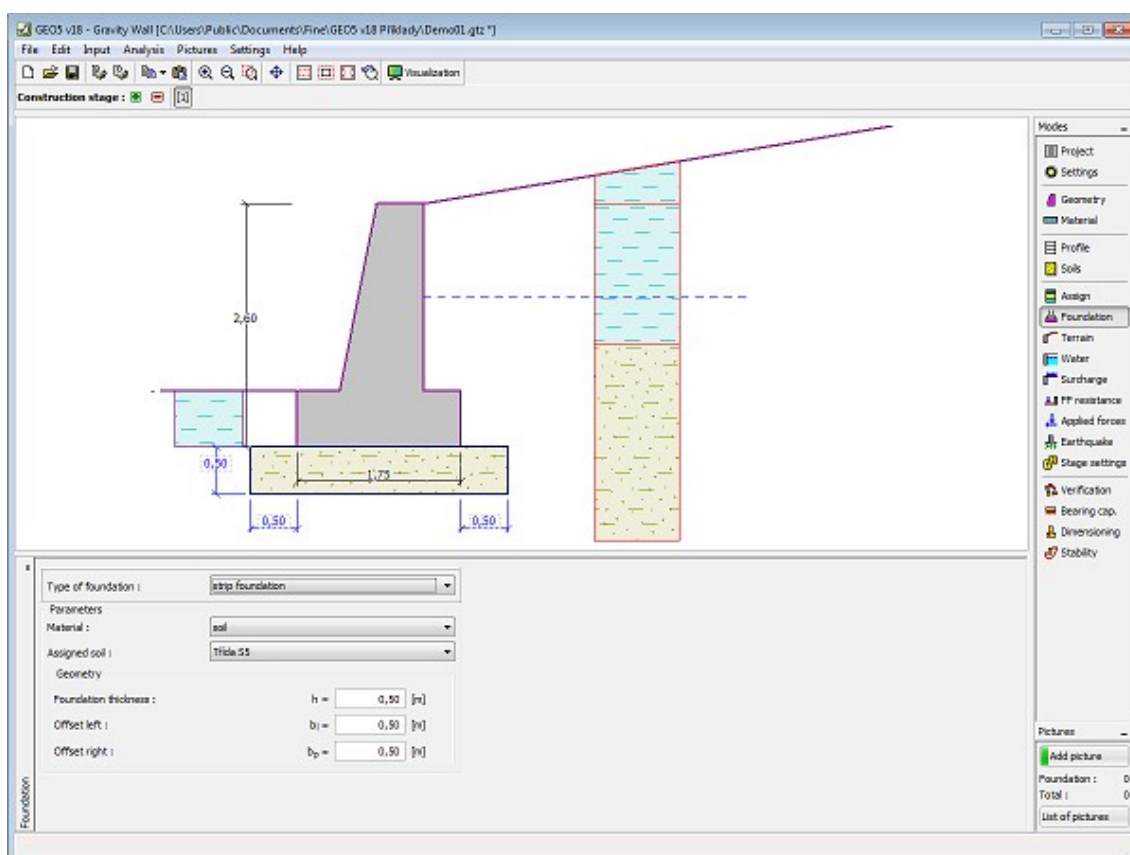
## Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle  $\psi$  [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient  $\mu$  [-]. Both options require inputting the cohesion  $a$  [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material**  $\gamma$  and **parameters of contact base-soil** (friction coefficient  $f$ , cohesion  $c$ , additional resistance  $F$ ).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

**Strip foundation** and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of **active dimensions**.

The input data introduced in this frame influence the actual **wall analysis** (check for slip) and further the **bearing capacity of foundation soil**.



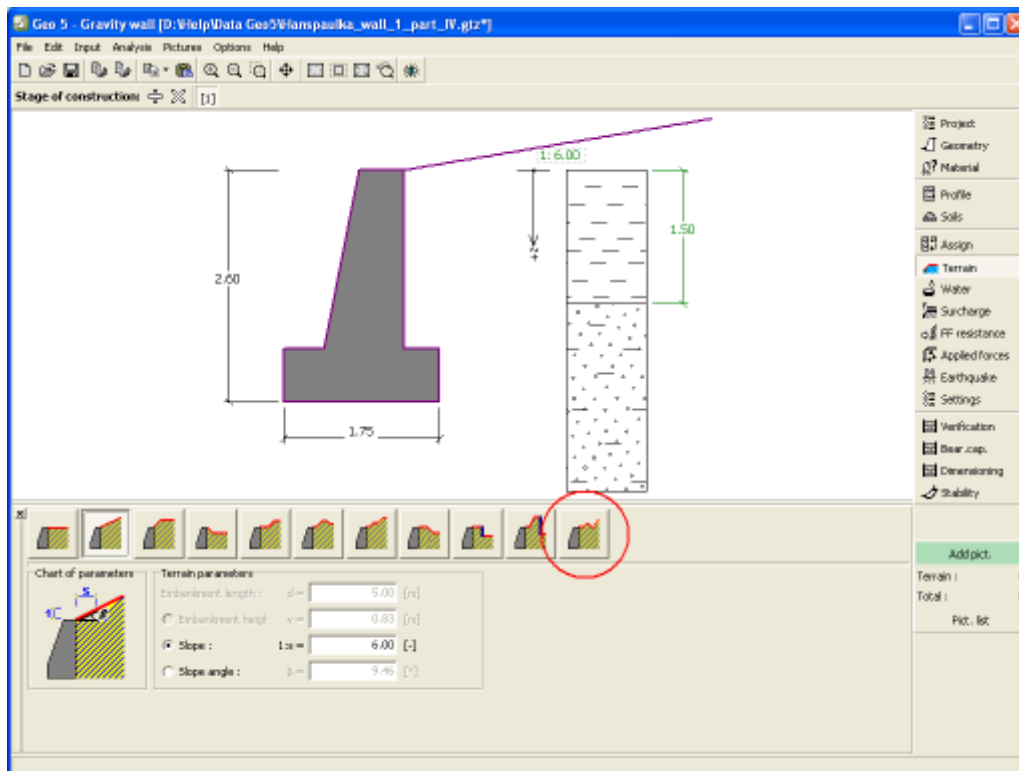
Frame "Foundation"

## Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

The **"Water"** frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint (**"Chart of parameters"**) of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

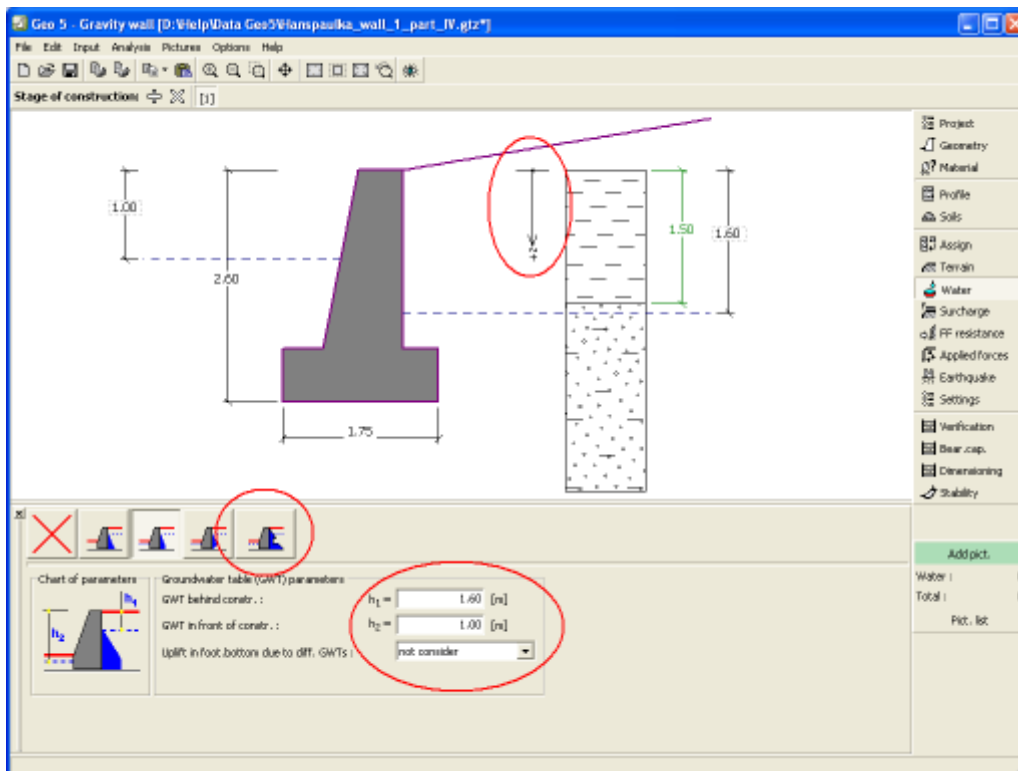
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be **linear, parabolic** or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs **"In front of structure"** and **"Behind structure"** appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter **"Influence of water"**.

The program further allows for specifying a depth of **tensile cracks** filled with water.



Frame "Water"

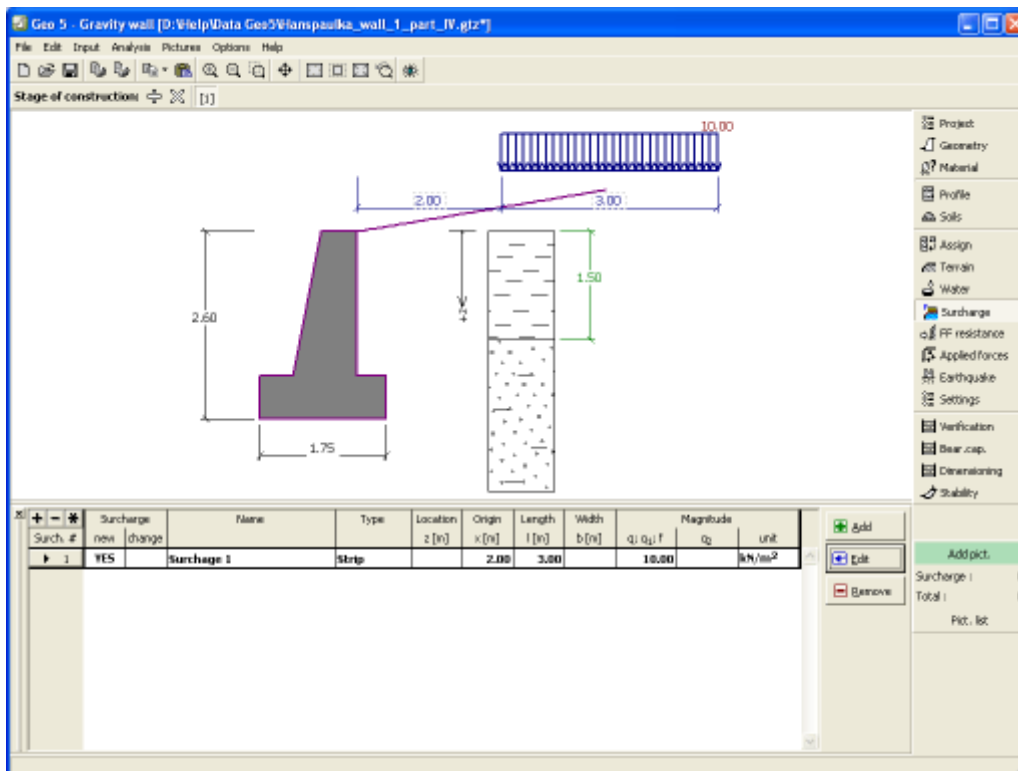
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".



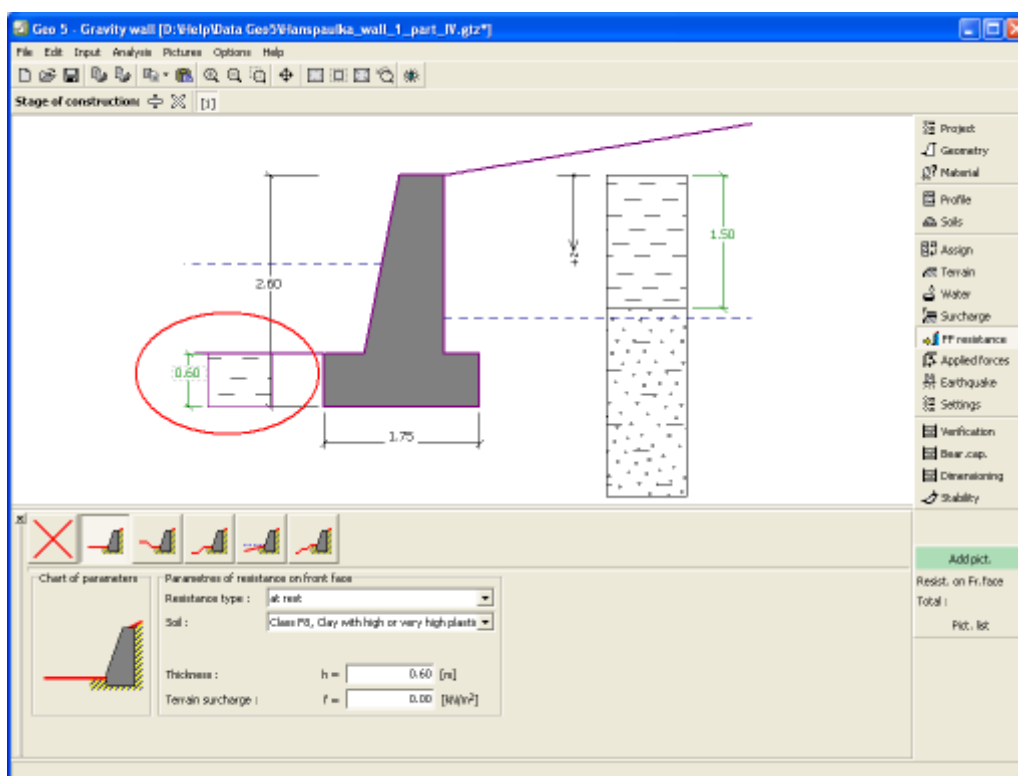
Frame "Surcharge"

## Front Face Resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the frame "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive earth pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the input type of reduced passive pressure.



Frame "Front face resistance"

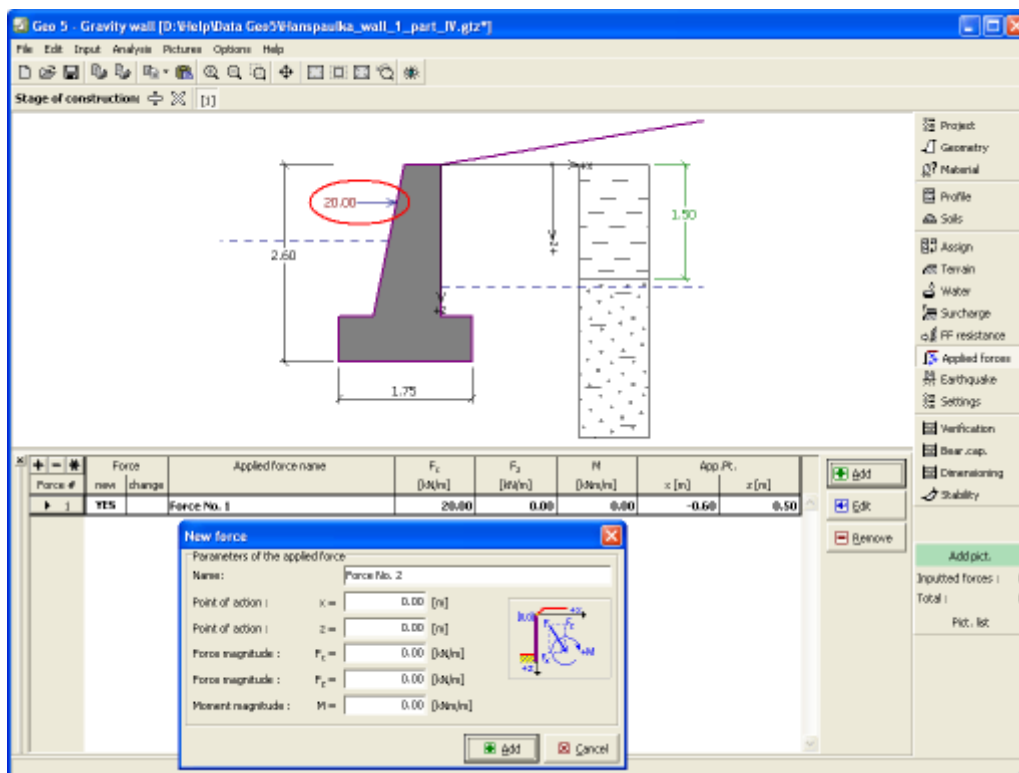
## Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

**Applied forces** represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.





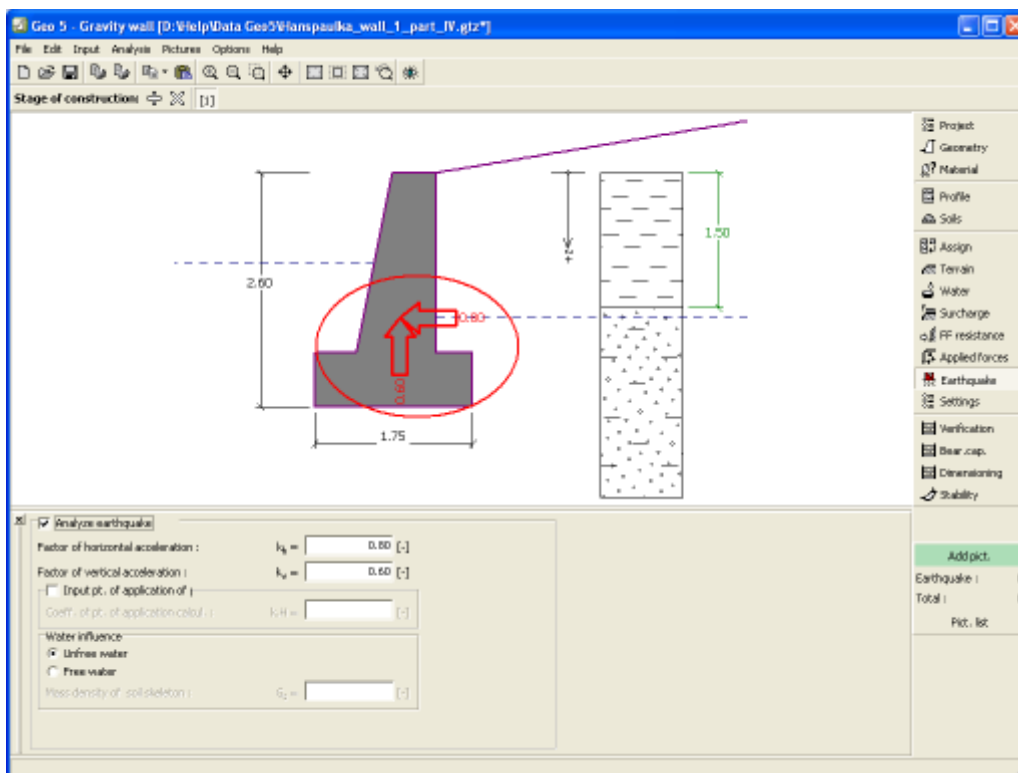
Frame "Applied forces"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

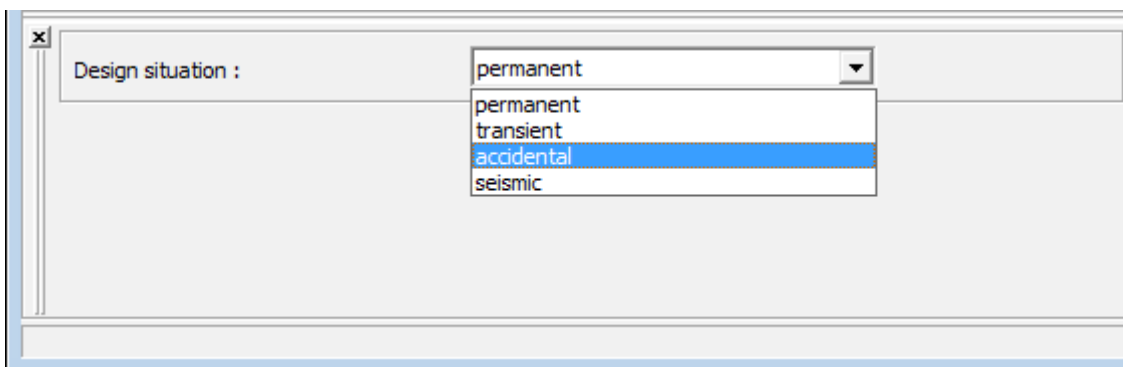


Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage. Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Verification

The frame "Verification" shows the analysis results. Several computations can be carried out for a single task.

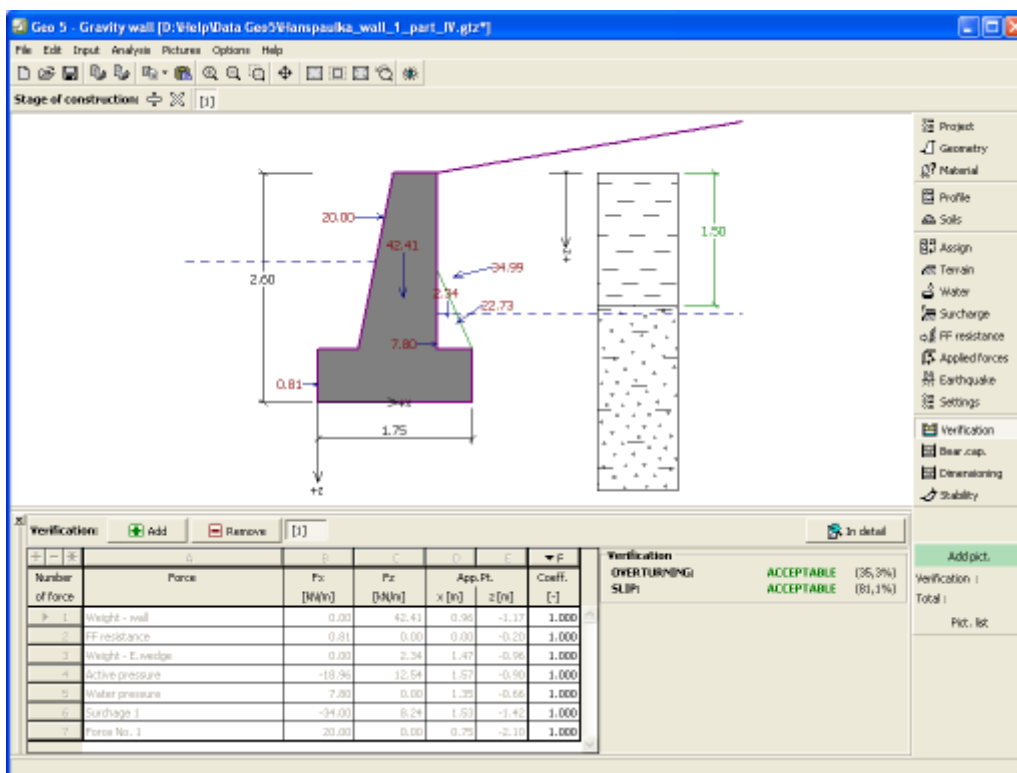
The frame appearance is adjusted based on the selected **verification methodology**.

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in this case the last column is not displayed.

The procedure for **wall verification** is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The "**In detail**" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Verification"

## Bearing Capacity

The "**Bearing capacity**" frame displays the analysis of foundation soil bearing capacity result. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "**Verification**". The program "**Spread footing**" then considers all verifications as load cases.

The frame contains following analysis options:

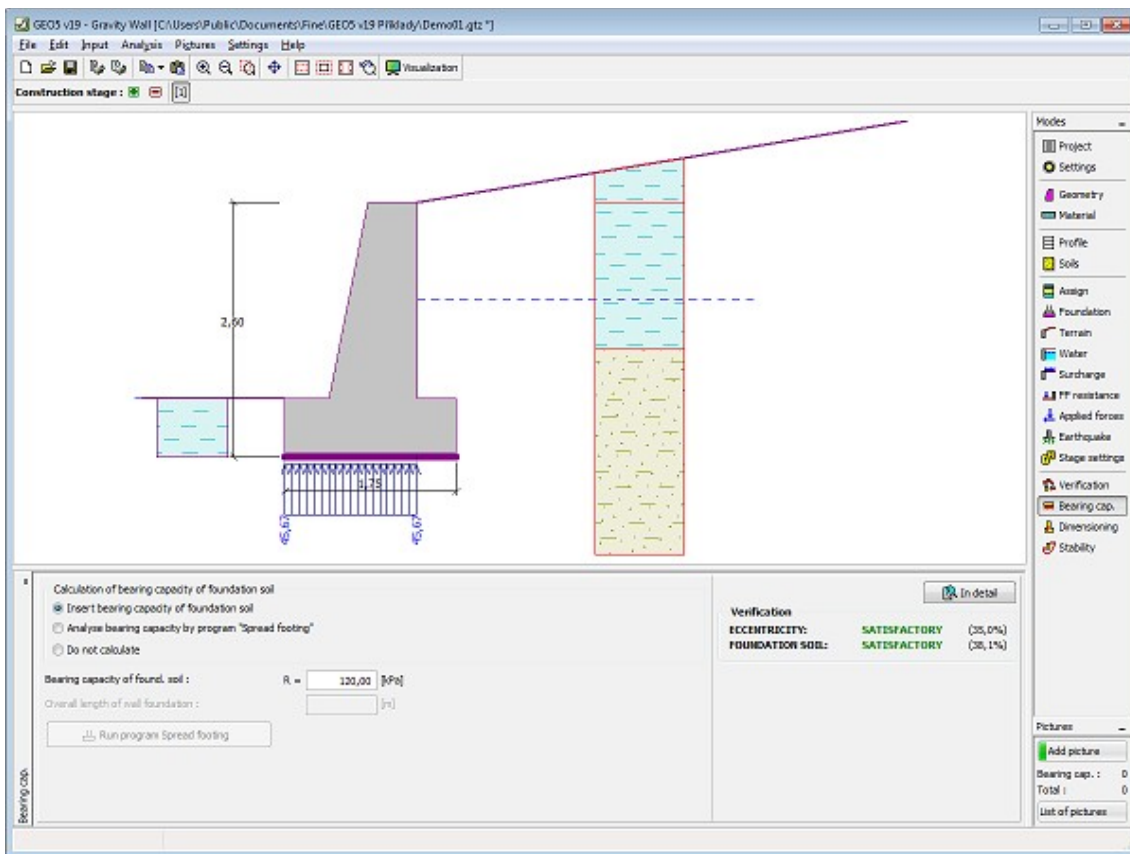
- **Insert bearing capacity of** The foundation soil bearing capacity is input. The

**foundation soil**

**eccentricity** and **bearing capacity** analysis results are displayed in the right part of the frame. The **"In detail"** button opens a dialog window that displays detailed listing of the results.

- **Analyse bearing capacity by program "Spread footing"** Pressing the **"Run program Spread footing"** button opens the program **"Spread footing"** which allows to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the **"OK"** button leaves the analysis mode - the results and all plots are transferred into the program **"Gravity wall"**. The **"Spread footing"** program must be installed for the button to be active. Overall length of wall foundation is input.
- **Analyse bearing capacity by program "Pile"** The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile"** is available if the wall has pile foundation (frame **"Foundation"**). Pile spacing  $s$  is input.
- **Analyse bearing capacity by program "Pile group"** The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile group"** is available if the wall has pile foundation with more than one pile (frame **"Foundation"**). Pile spacing  $s$ , overall number of pile rows  $n$  and loading length  $l$  are input.
- **Do not calculate (pile footing)** The foundation soil bearing capacity is not calculated.

Visualization of results can be adjusted in the frame **"Visualization settings"**.



Frame "Bearing capacity"

## Dimensioning

The "**Dimensioning**" frame serves to design and verify the reinforcement of wall cross-section - the cross-section subjected to dimensioning is selected in the combo list.

- **Wall stem verification**
- **Construction joint verification - depth** of construction joint from construction top edge is specified
- **Wall jump verification**

The frame appearance is adjusted based on the selected **verification methodology**.

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

Calculation of forces and their action on the analyzed cross-section is described [here](#). The wall stem and construction joint are always loaded by the **pressure at rest**.

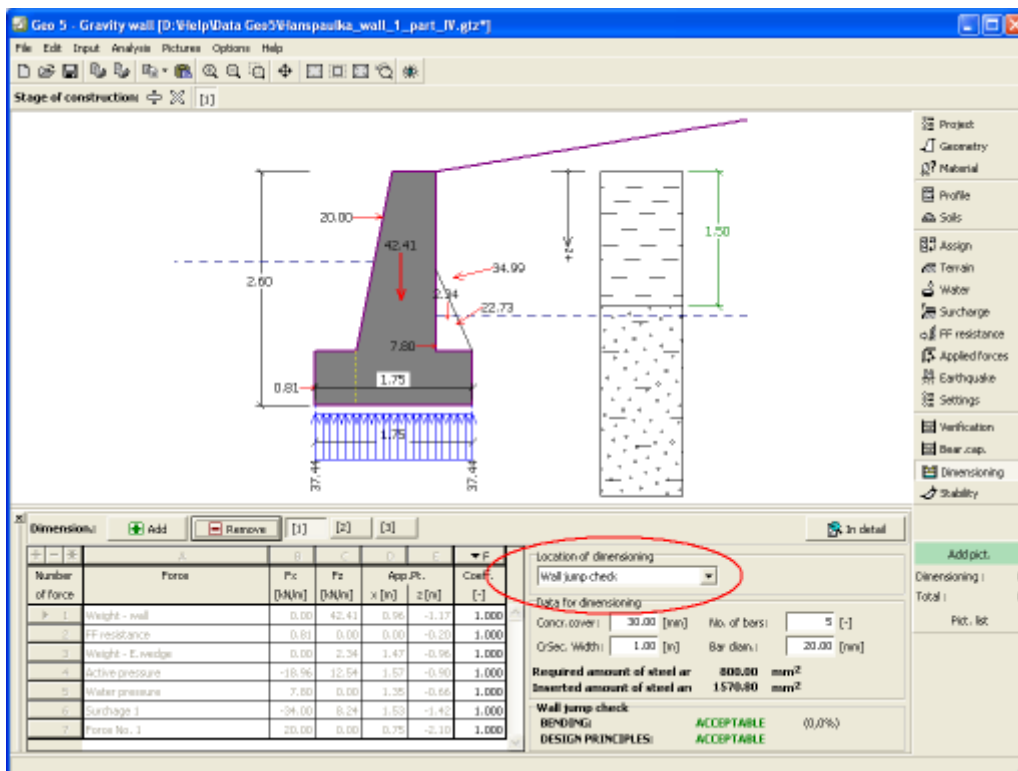
The **procedure to derive distribution of internal forces** in individual cross-sections is described

in the theoretical part of the help.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Visualization settings".

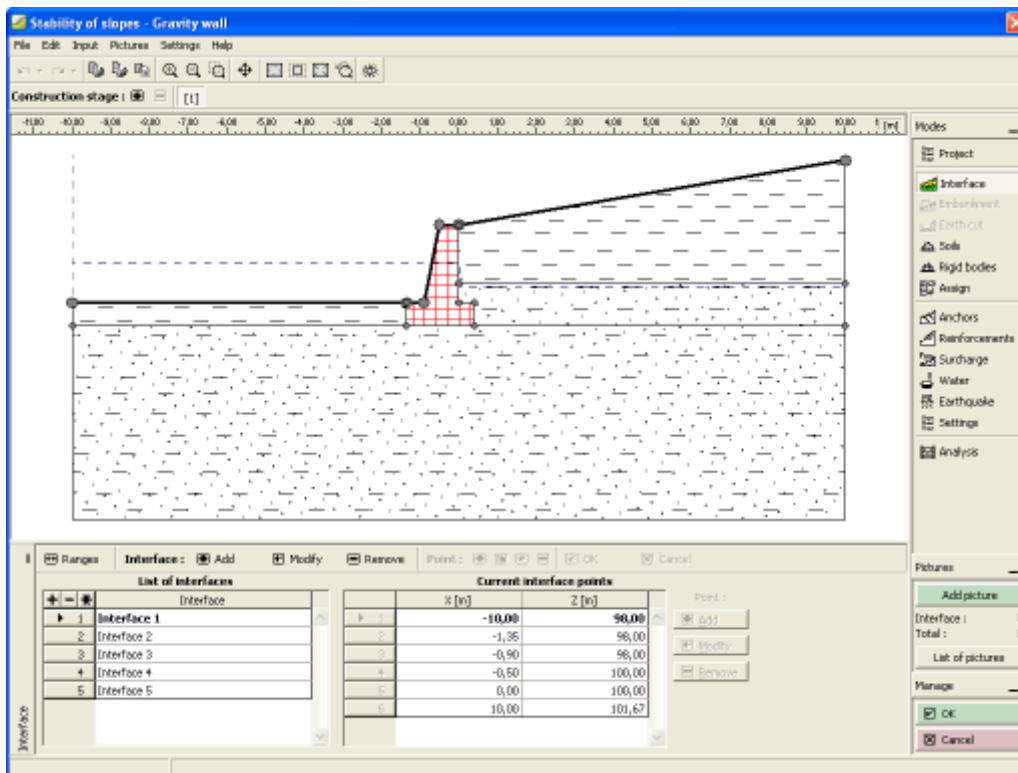


Frame "Dimensioning"

## Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program - all data are then carried over to the analysis protocol of the "Gravity wall" program.



Frame "Stability"

## Program Prefab Wall

This program is used to verify retaining walls made of prefabricated blocks.

The help in the program "Prefab Wall" includes the following topics:

- Input of data into individual frames:

Project	Settings	Geometry	Profile	Soils	Assign	Foundation
Terrain	Water	Surcharge	Front Face Resistance	Applied Forces	Earthquake	Stage Settings
Verification	Bearing Capacity	Dimensioning	Slip on Georeinforcement	Stability		

- Standards and analysis methods

- Theory for analysis in the program "Prefab Wall":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity
---------------------	-----------------	-------------------	---

- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.

*Frame "Project"*

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

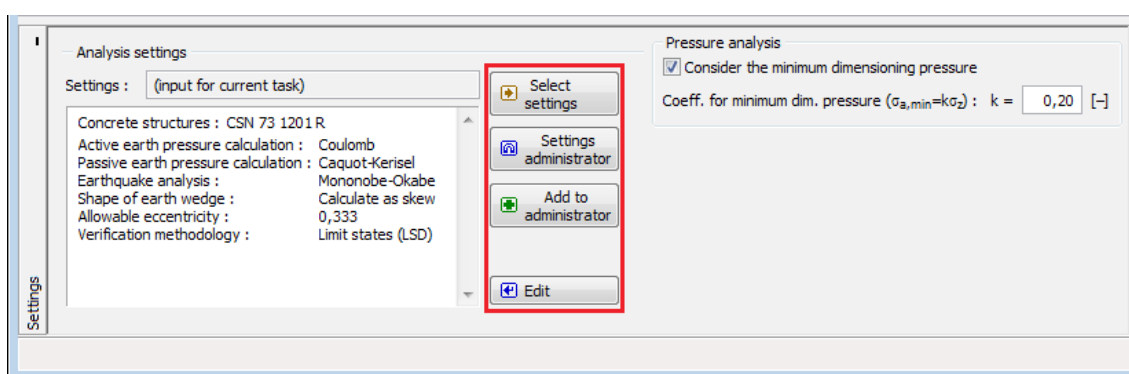


The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

The program allows to specify a value of the **minimum dimensioning pressure**  $k$  (by checking the option "**Consider the minimum dimensioning pressure**"). For real structures showed that when considering cohesive soil behind the wall of the upper blocks of structure behaves problematically. For this reason, it is recommended to implement a value of  $k$  into the calculation for the case of backfill consisting of cohesive soils.

Settings of analysis parameters are performed in the "**Materials and standards**" and "**Wall analysis**" tabs.



Frame "Settings"

## Geometry

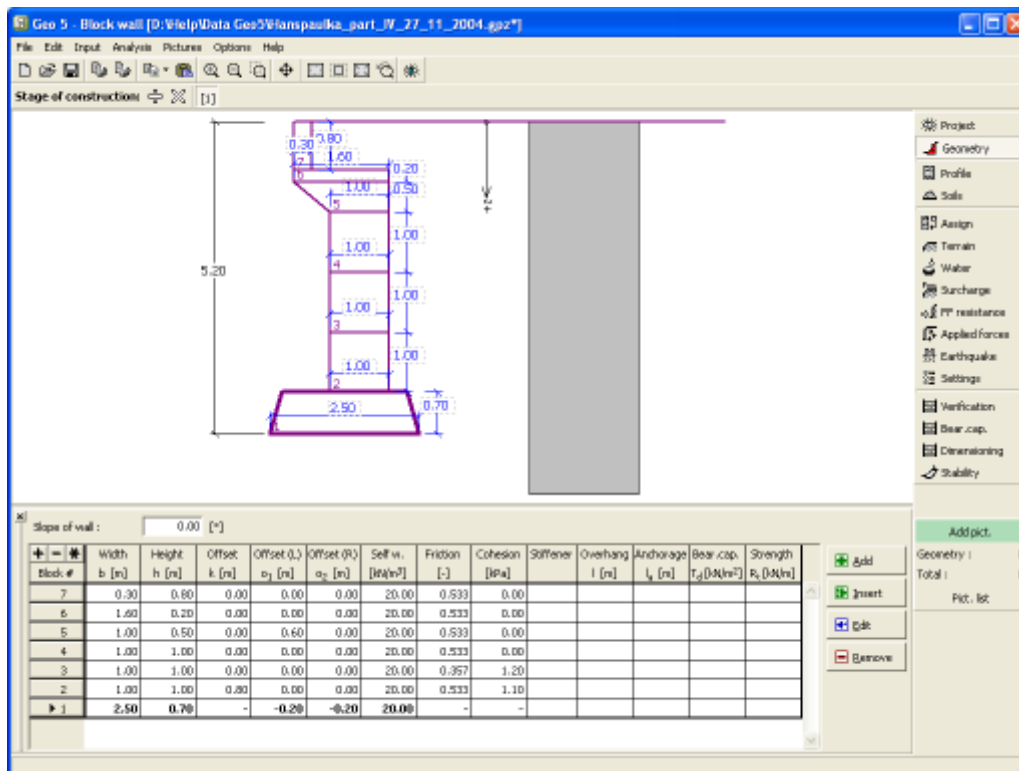
The "**Geometry**" frame contains a table with a list of input structural precast units (blocks) of a wall (the lowest block is labeled as No. 1). **Adding** (editing) blocks is performed in the "**New block**" dialog window.

This dialog window serves to define **the geometry of a block**, **parameters of reinforcement** (length, anchorage length, tensile strength and pull out resistance) and **material characteristics** (self weight, shear resistance between two blocks, cohesion).

The program allows for adding (inserting) another block in between two already existing blocks of a structure. Inserting a new block is performed in the "**Insert block**" dialog window that complies with the "**New block**" dialog window. The inserted block is ordered such to proceed the currently selected block of a structure.

The input blocks can be further edited on the desktop with the help of **active dimensions** or **active objects** - double clicking on a structure opens a dialog window with a given block. **When using the regime of active objects the visualization of detailed dimensions must be turned off in the "Visualization style settings" dialog window.**

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Geometry"

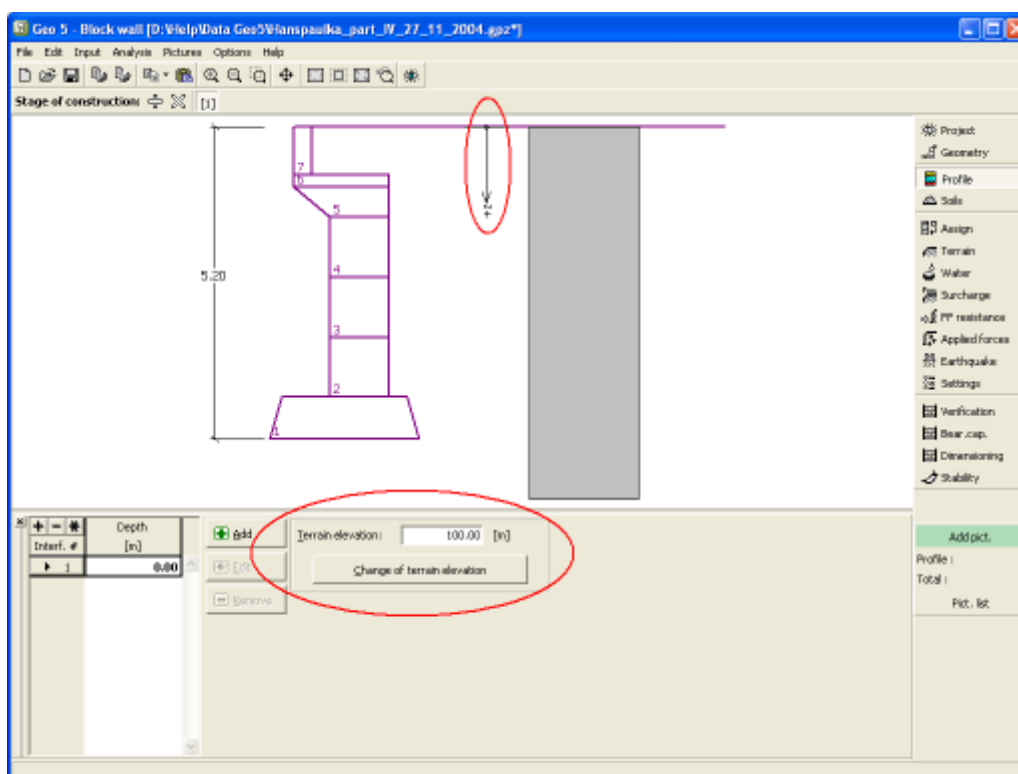
## Profile

The "Profile" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "New interface" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "Change of terrain elevation" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".

The program makes it possible to import a profile in the gINT format.



Frame "Profile"

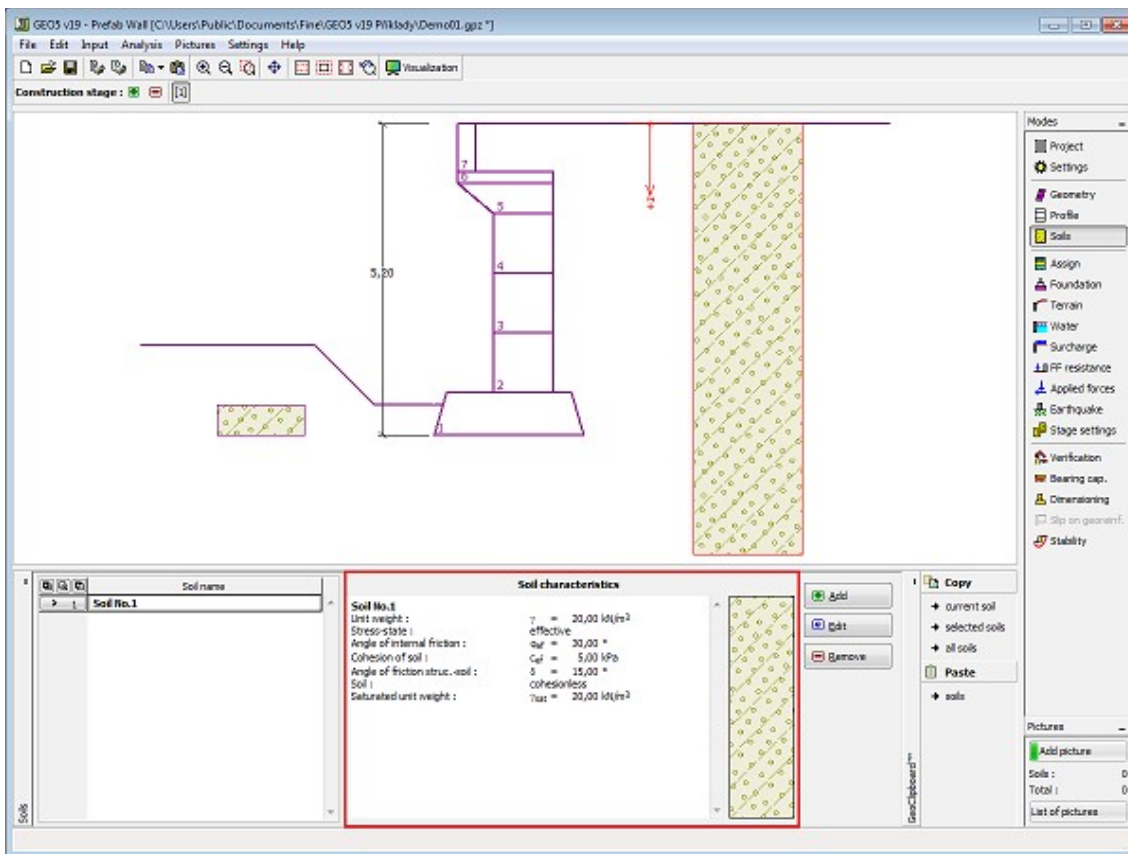
## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil, type of load, structure duration and water conditions.

For **effective stress** further needs to specify the **angle of internal friction between the soil and structure**, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the **table of recommended values**.

For **total stress** further needs to specify the **adhesion of soil to the structure face  $a$** .

The associated theory is described in detail in chapter "**Earth pressures**".

**Add new soils**

Identification

Name : Gravelly clay (CG), consistency firm

Gravelly clay (CG), consistency firm

**Basic data**

Unit weight :  $\gamma =$  19,50 [kN/m<sup>3</sup>] 19,5

Stress-state : effective

Angle of internal friction :  $\phi_{ef} =$  27,00 [°] 24-30

Cohesion of soil :  $c_{ef} =$  10,00 [kPa] 6-14

Angle of friction struc.-soil :  $\delta =$  [°]

Pressure at rest

Soil : cohesive

Poisson's ratio :  $\nu =$  0,35 [-] 0,35

Uplift pressure

Calc. mode of uplift : standard

Saturated unit weight :  $\gamma_{sat} =$  [kN/m<sup>3</sup>]

Draw

Color

Pattern category

GEO

Pattern

Gravelly clay

Classification

Classify

Delete

Add

Cancel

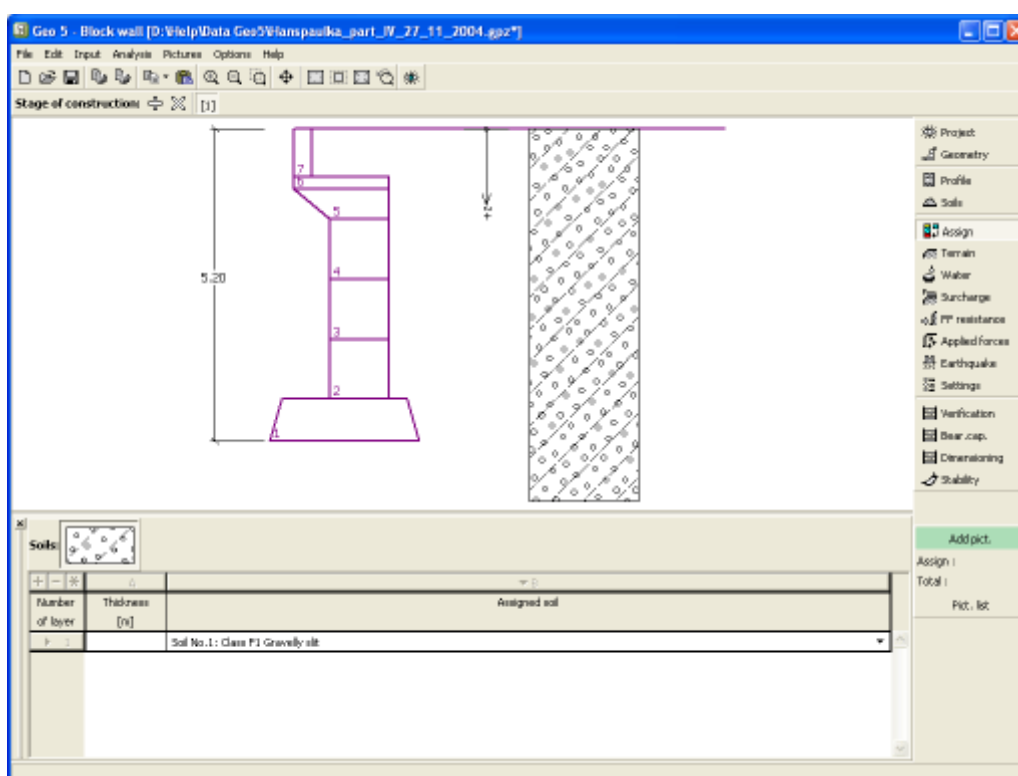
*Dialog window "Add new soils" - "Basic data"*

## Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

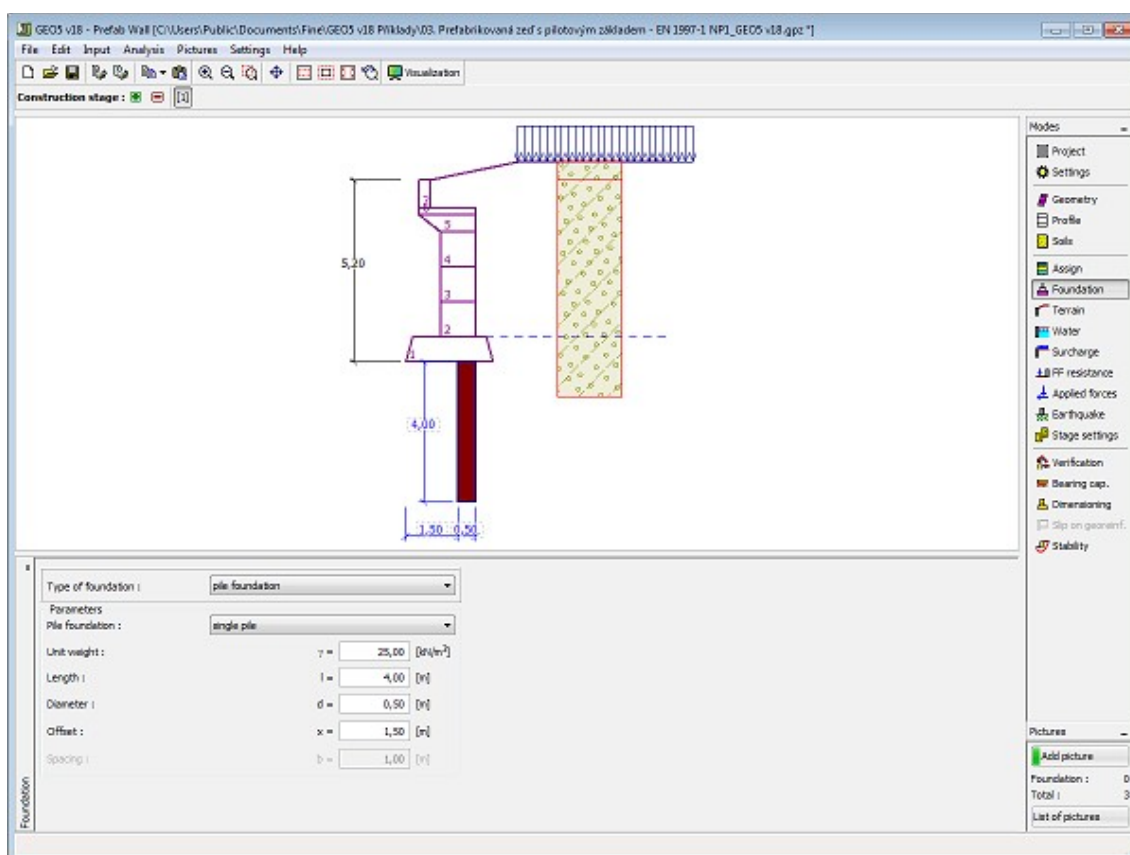
## Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle  $\psi$  [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient  $\mu$  [-]. Both options require inputting the cohesion  $a$  [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material**  $\gamma$  and **parameters of contact base-soil** (friction coefficient  $f$ , cohesion  $c$ , additional resistance  $F$ ).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

**Strip foundation** and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of **active dimensions**.

The input data introduced in this frame influence the actual **wall analysis** (check for slip) and further the **bearing capacity of foundation soil**.



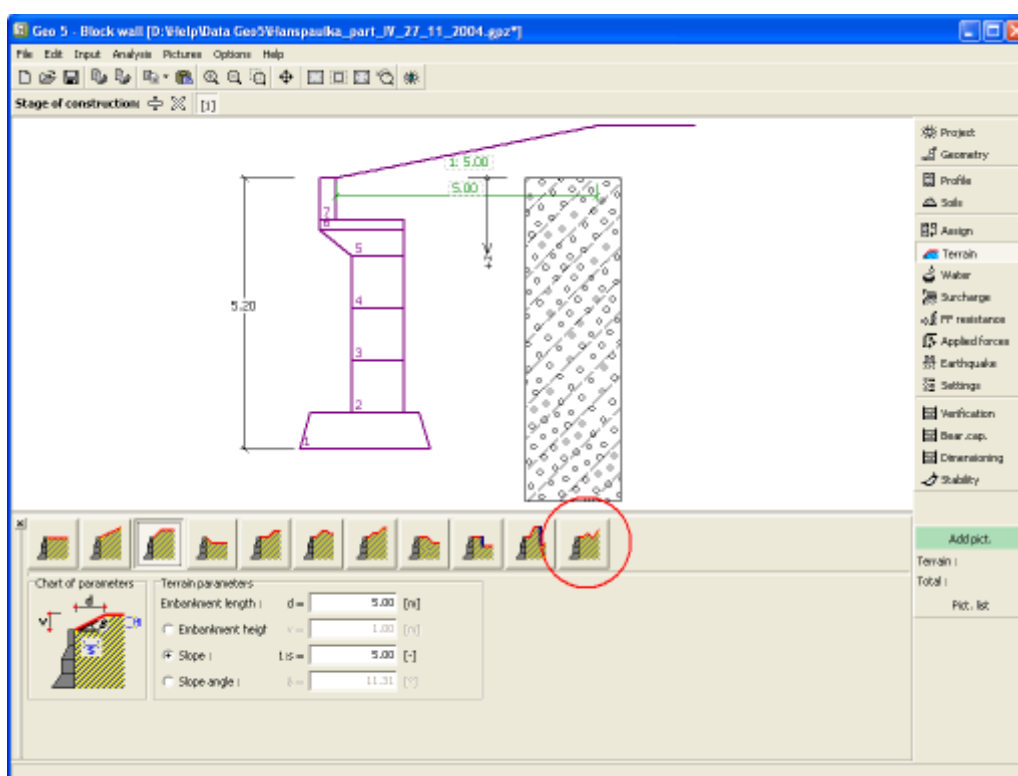
Frame "Foundation"

## Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Water parameters ( $h_1$ ,  $h_2$ ...) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be **linear**, **parabolic** or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

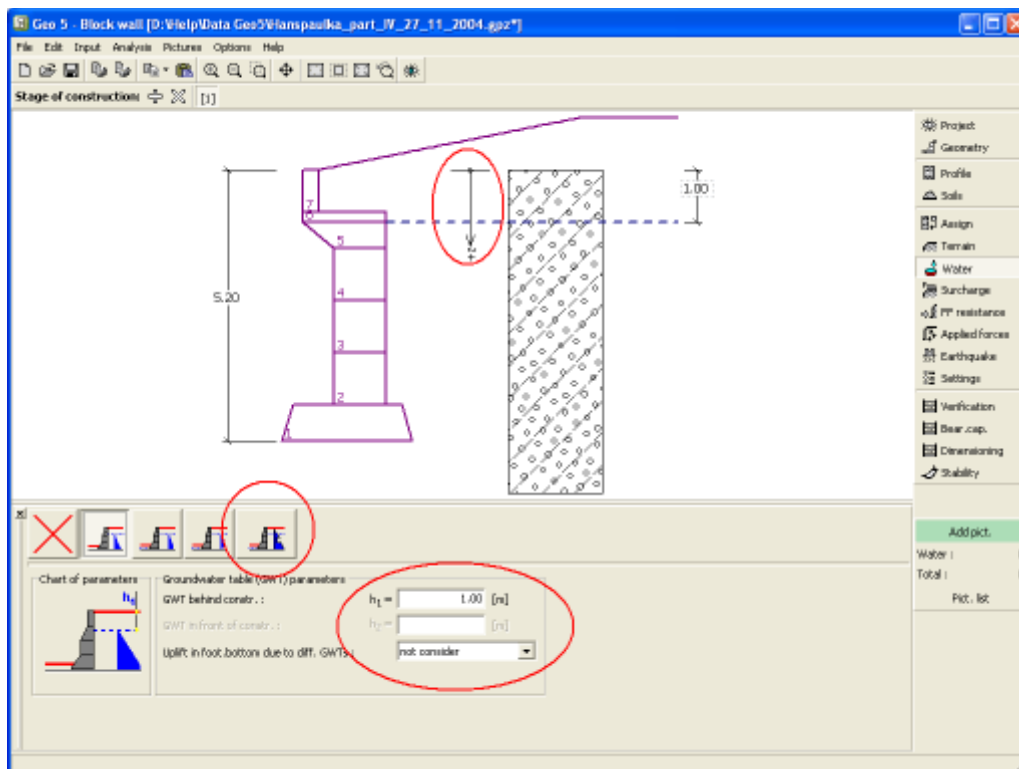
The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".

The program further allows for specifying a depth of **tensile cracks** filled with water.





Frame "Water"

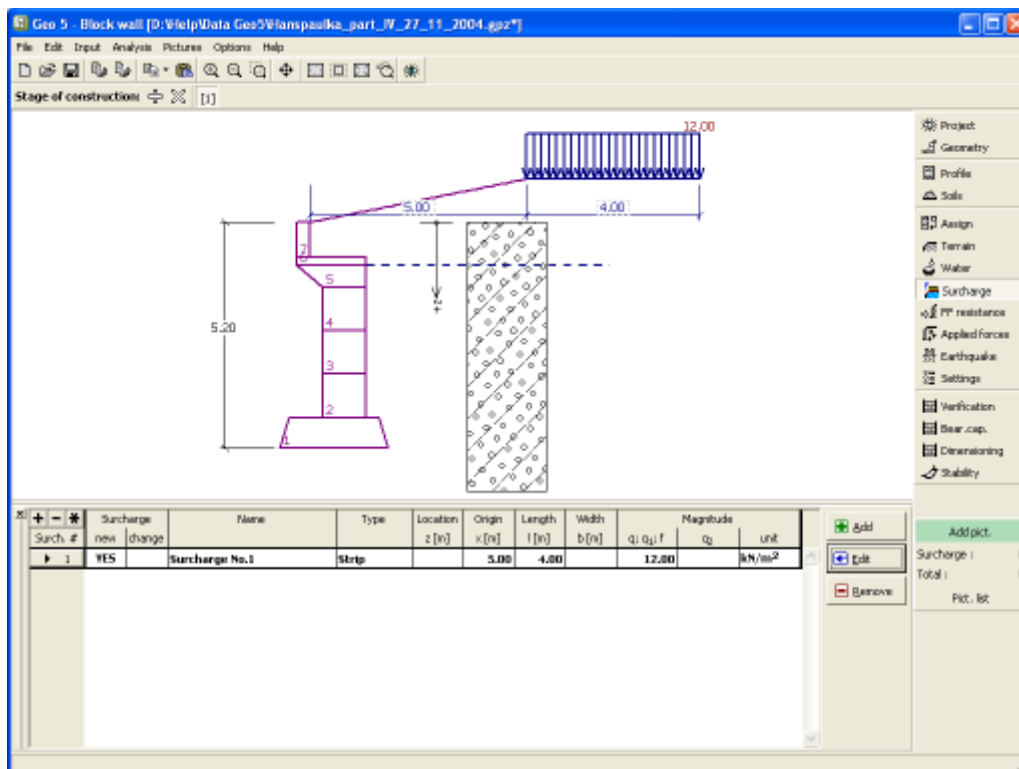
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".



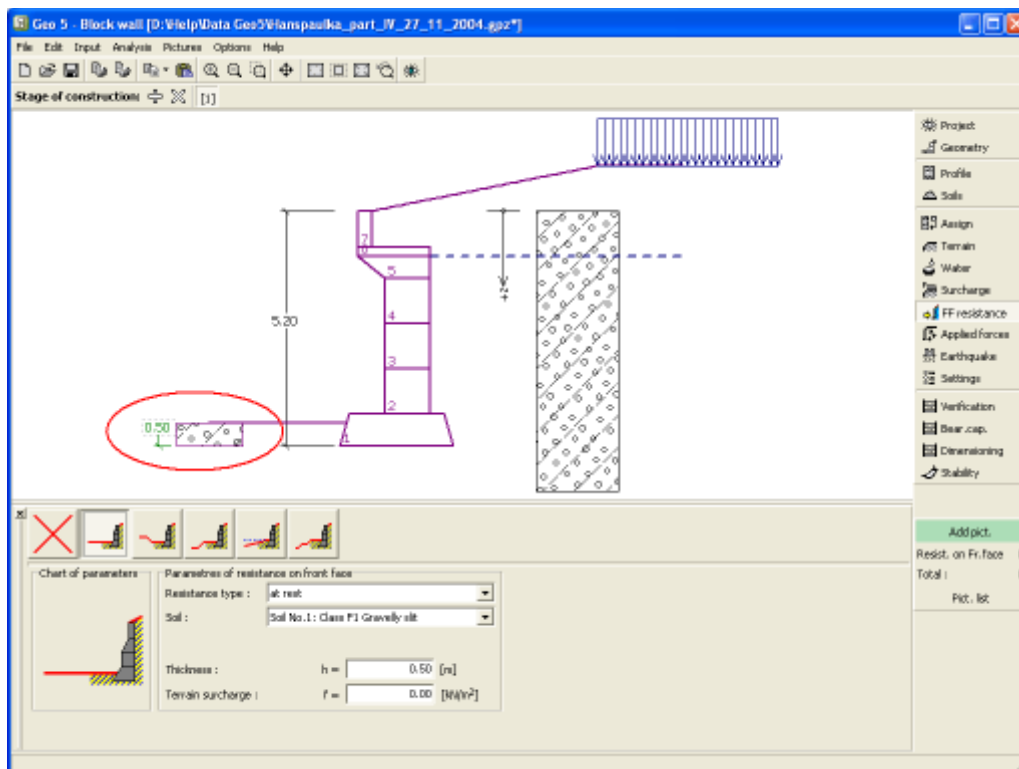
Frame "Surcharge"

## Front Face Resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint ("Chart of parameters") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the frame "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive earth pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the input type of reduced passive pressure.



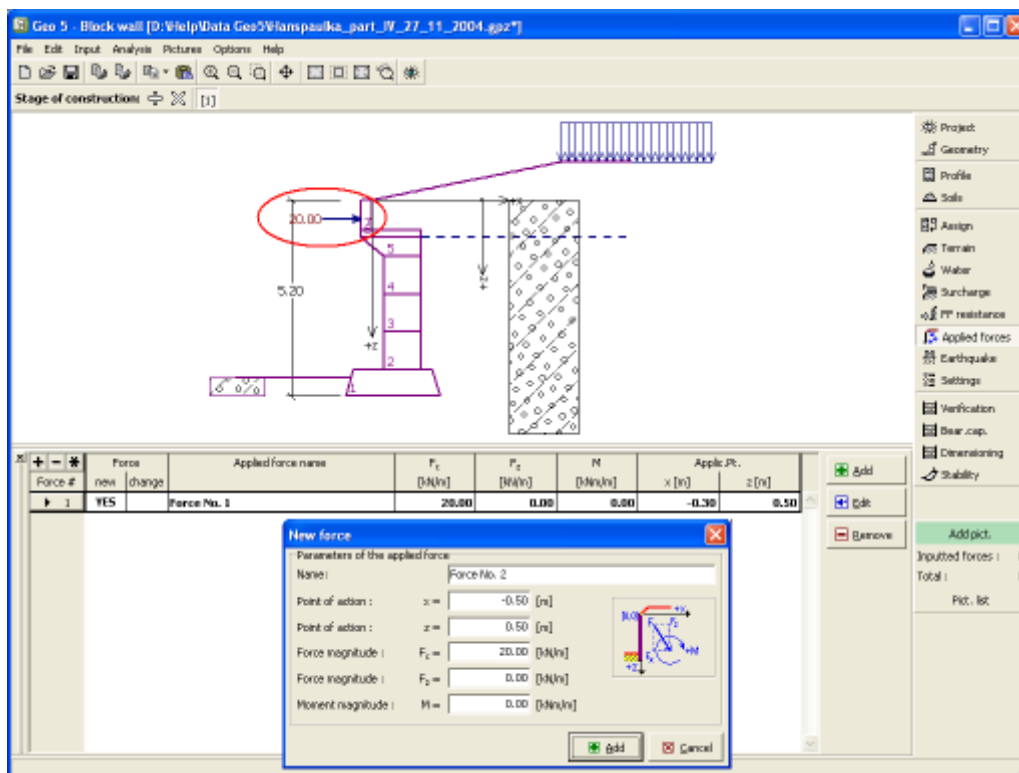
Frame "Front face resistance"

## Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

**Applied forces** represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.



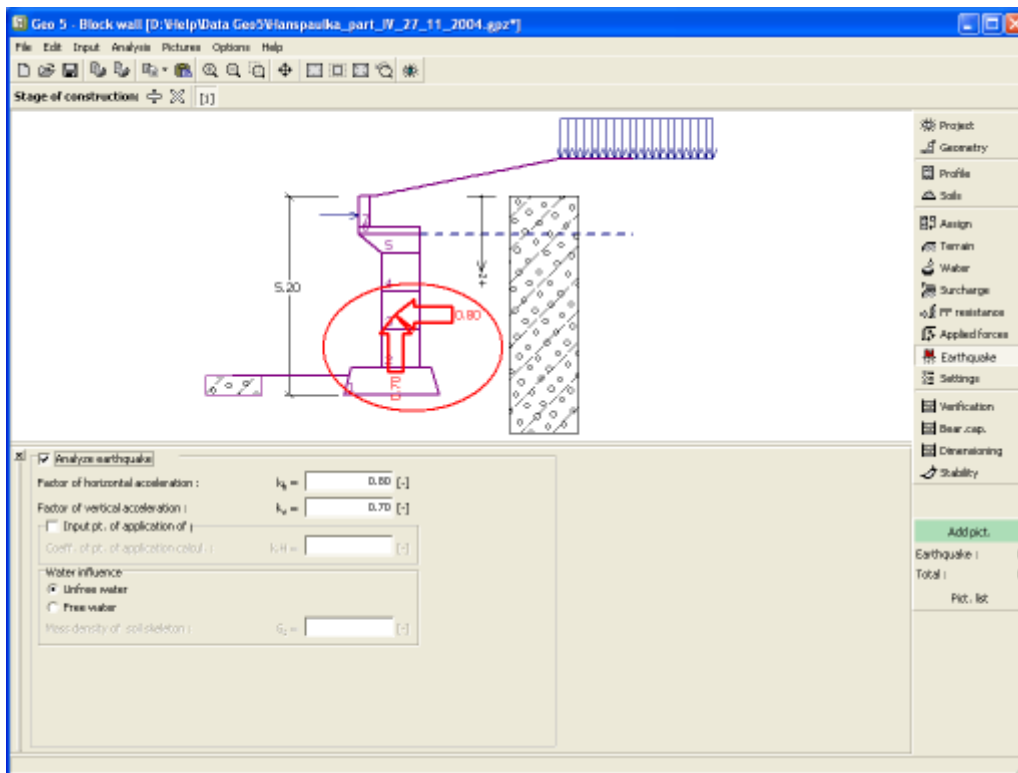
Frame "Applied forces"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



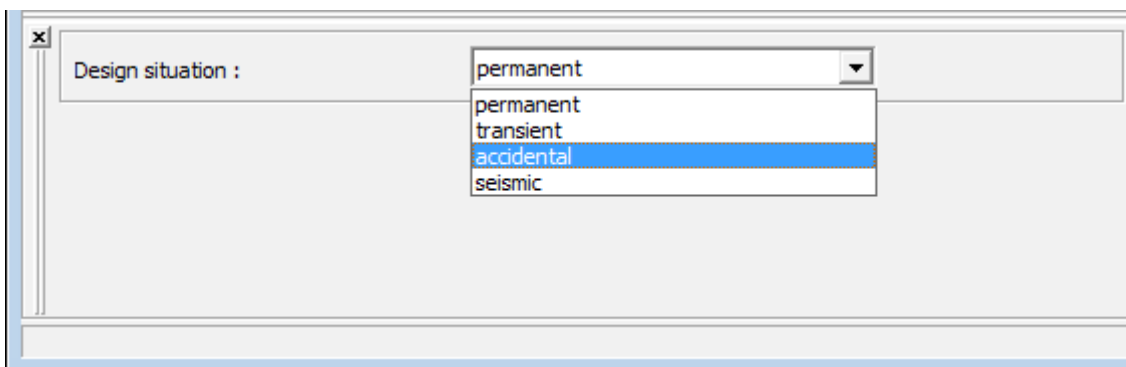
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Verification

The frame "Verification" shows the analysis results. Several computations can be carried out for a single task.

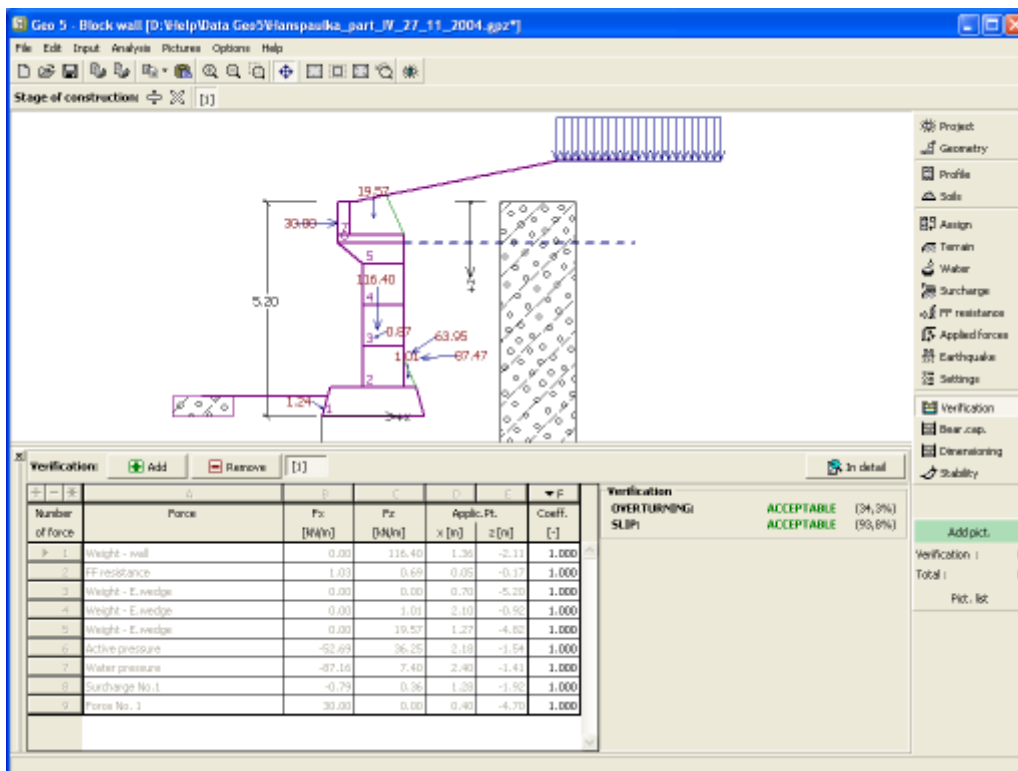
The frame appearance is adjusted based on the selected **verification methodology**.

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

The procedure for **wall verification** is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The "**In detail**" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Verification"

## Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "**Verification**". The program "**Spread footing**" then considers all verifications as load cases.

The frame contains following analysis options:

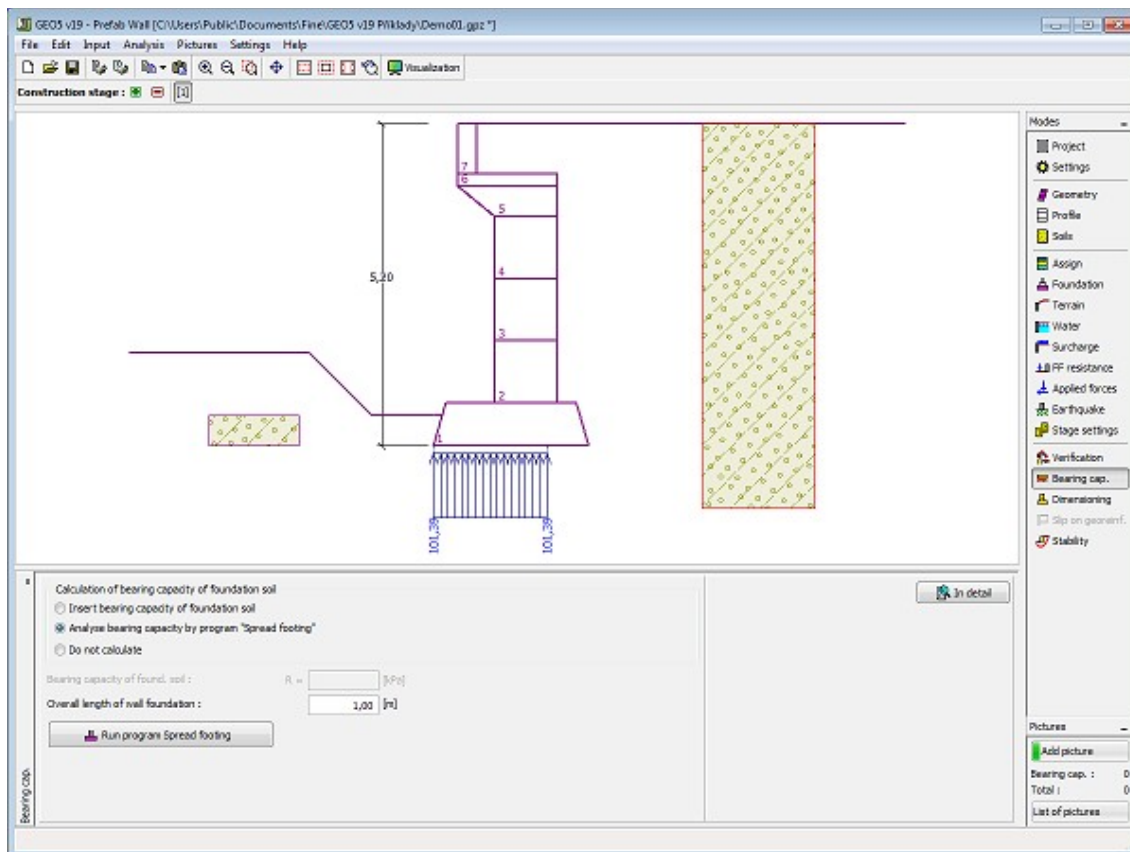
- **Insert bearing capacity of** The foundation soil bearing capacity is input. The

**foundation soil**

**eccentricity** and **bearing capacity** analysis results are displayed in the right part of the frame. The **"In detail"** button opens a dialog window that displays detailed listing of the results.

- **Analyse bearing capacity by program "Spread footing"** Pressing the **"Run program Spread footing"** button opens the program **"Spread footing"** which allows to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the **"OK"** button leaves the analysis mode - the results and all plots are transferred into the program **"Prefab wall"**. The **"Spread footing"** program must be installed for the button to be active. Overall length of wall foundation is input.
- **Analyse bearing capacity by program "Pile"** The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile"** is available if the wall has pile foundation (frame **"Foundation"**). Pile spacing  $s$  is input.
- **Analyse bearing capacity by program "Pile group"** The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile group"** is available if the wall has pile foundation with more than one pile (frame **"Foundation"**). Pile spacing  $s$ , overall number of pile rows  $n$  and loading length  $l$  are input.
- **Do not calculate (pile footing)** The foundation soil bearing capacity is not calculated.

Visualization of results can be adjusted in the frame **"Visualization settings"**.



Frame "Bearing capacity"

## Dimensioning

The "**Dimensioning**" frame allows to verify joints between individual blocks of a wall. The "**Joint above block No.**" field serves to select the desired joint subjected to verification analysis. The verification against **overturning** and **translation** is performed in the same way as for the **entire wall** - friction between blocks and cohesion of a block material are input in the frame "**Geometry**".

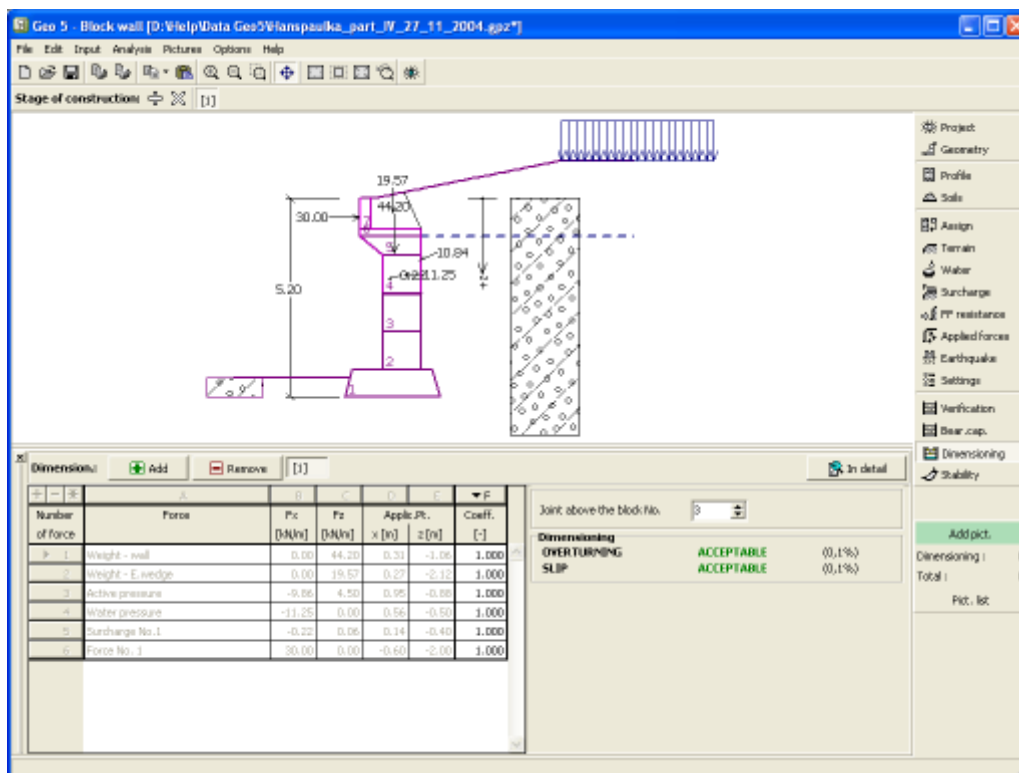
The frame appearance is adjusted based on the selected **verification methodology**.

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

**Several computations** for various cross-sections can be carried out. Various **design coefficients** of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "**Visualization settings**".





Frame "Dimensioning"

## Slip on Georeinforcement

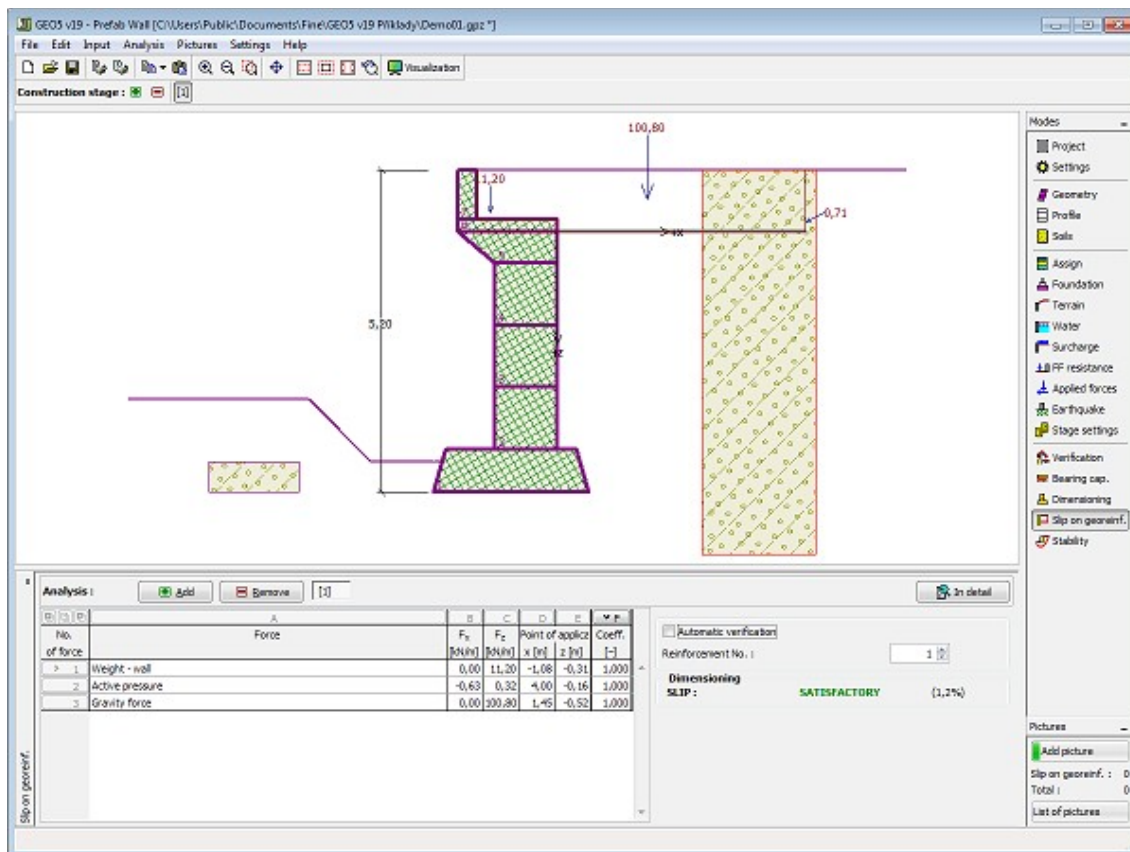
The **frame** serves to verify the limit state for slip along reinforcement - the frame is therefore accessible only in **stages**, where the reinforcements are defined.

The window requires inputting the reinforcement number - the forces entering verification analysis together with the shape of sliding block are then displayed. The calculated forces are stored in the **table**.

**Several calculations** for various reinforcements can be carried out. Various **design coefficients** of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The **"In detail"** button opens the dialog window that contains detailed listing of the dimensioning results.

The verification procedure depends on settings in the **"Wall analysis"** tab - either based on **factors of safety** or according to the **theory of limit states**. The solution procedure is described **herein**.

Visualization of results can be adjusted in the frame **"Visualization settings"**.

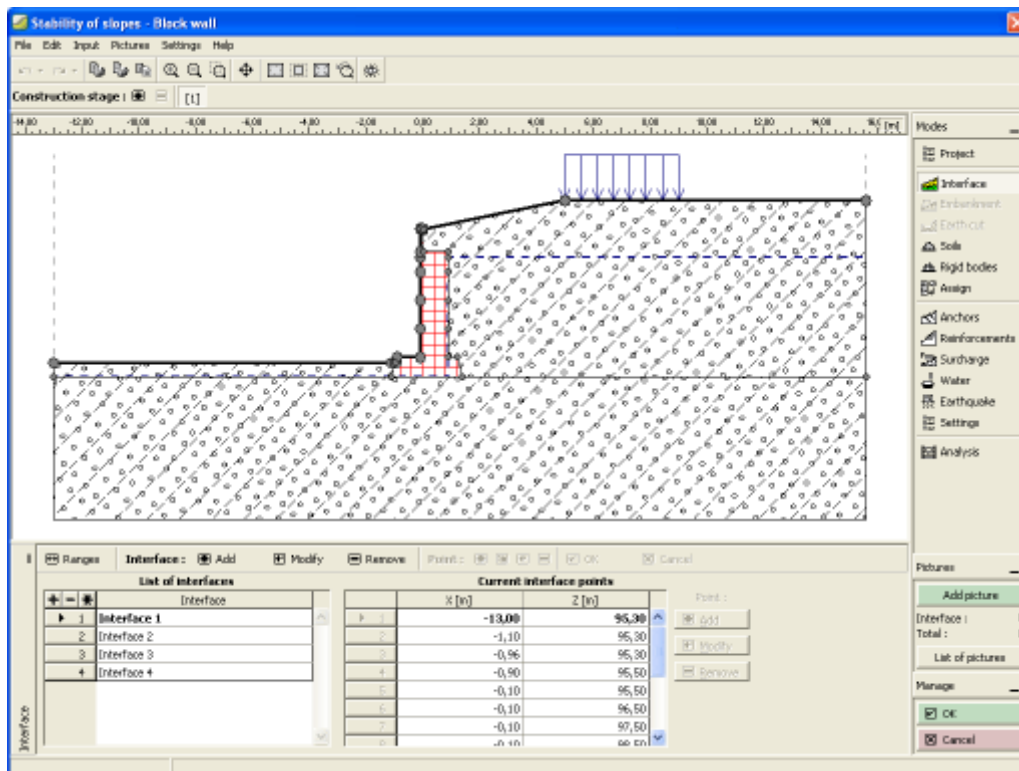


Frame "Slip on georeinforcement"

## Stability

Pressing the **"Stability"** button launches the **"Slope stability"** program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program **"Slope stability"** is installed.

After completing all analyses press the **"OK"** button to leave the program - all data are then carried over to the analysis protocol of the **"Prefab wall"** program.



Frame "Stability"

## Program Masonry Wall

This program is used for design and analysis of reinforced masonry block wall according to various standards.

The help in the program "Masonry Wall" includes the following topics:

- Input of data into individual frames:

Project	Settings	Types of Blocks	Geometry	Material	Profile	Soils
Assign	Foundation	Terrain	Water	Surcharge	Front Face Resistance	Applied Forces
Earthquake	Base Anchorage	Stage Settings	Verification	Bearing Capacity	Dimensionin g	Stability

- Standards and analysis methods

- Theory for analysis in the program "Masonry Wall":

Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation	Dimensioning of Concrete
---------------------	-----------------	-------------------	------------------------	--------------------------

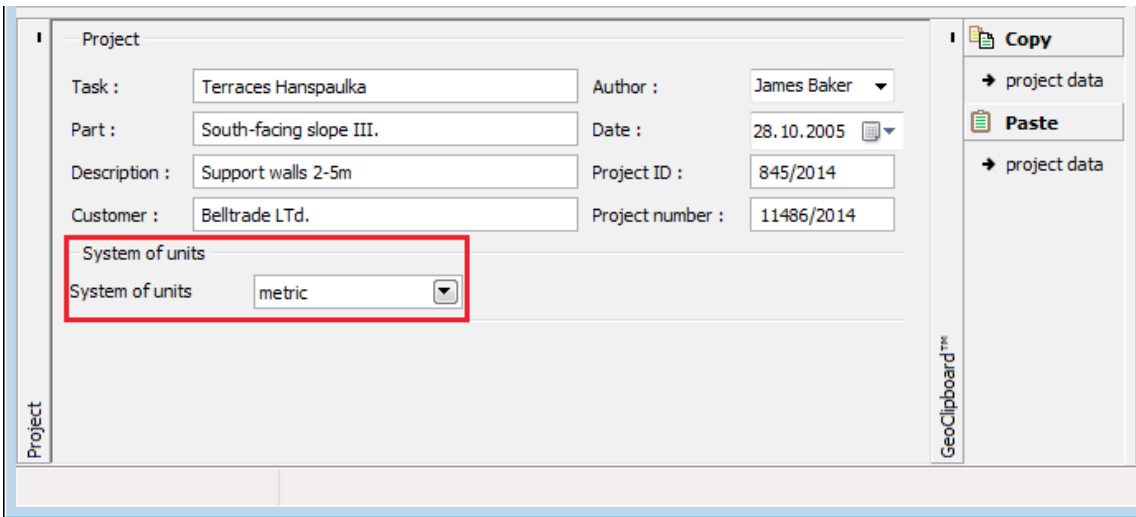
## Bearing Capacity Structures

- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.



Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

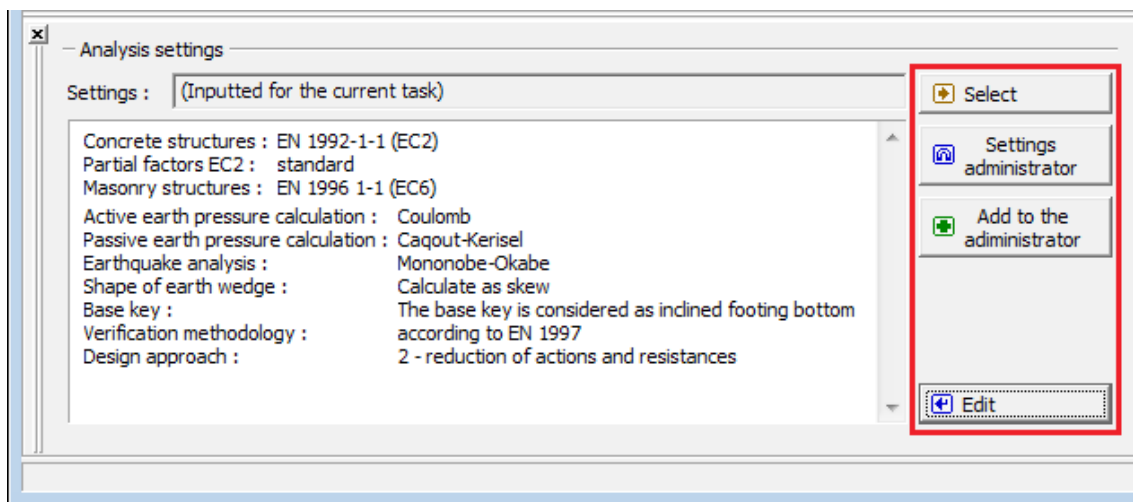
The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Materials and standards**" and "**Wall analysis**" tabs.



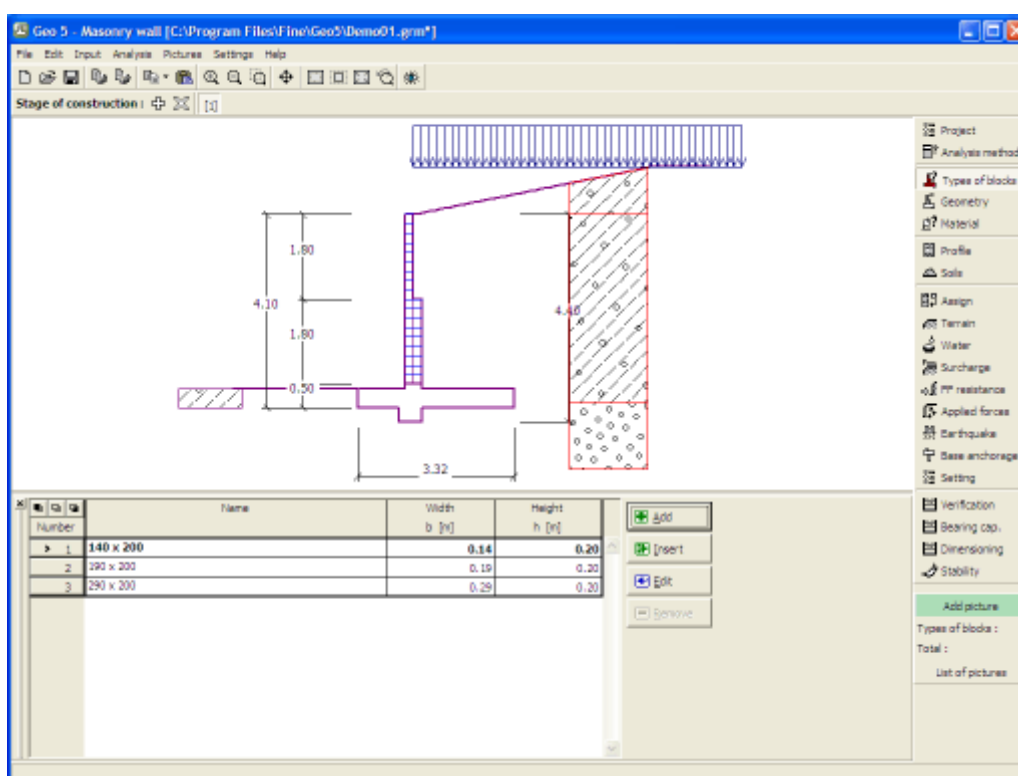
Frame "Settings"

## Types of Blocks

The "**Types of blocks**" frame contains a table with a list of input blocks. Adding (editing) blocks is performed in the "**New type of block (Edit type of block)**" dialog window.

This dialog window serves to define **the geometry of a block** (width and height).

The program allows for adding (inserting) another block in between two already existing blocks of a structure. Inserting a new block is performed in the "**Inserted type of block**" dialog window that complies with the "**New type of block**" dialog window. The inserted block is ordered such to proceed the currently selected block of a structure.



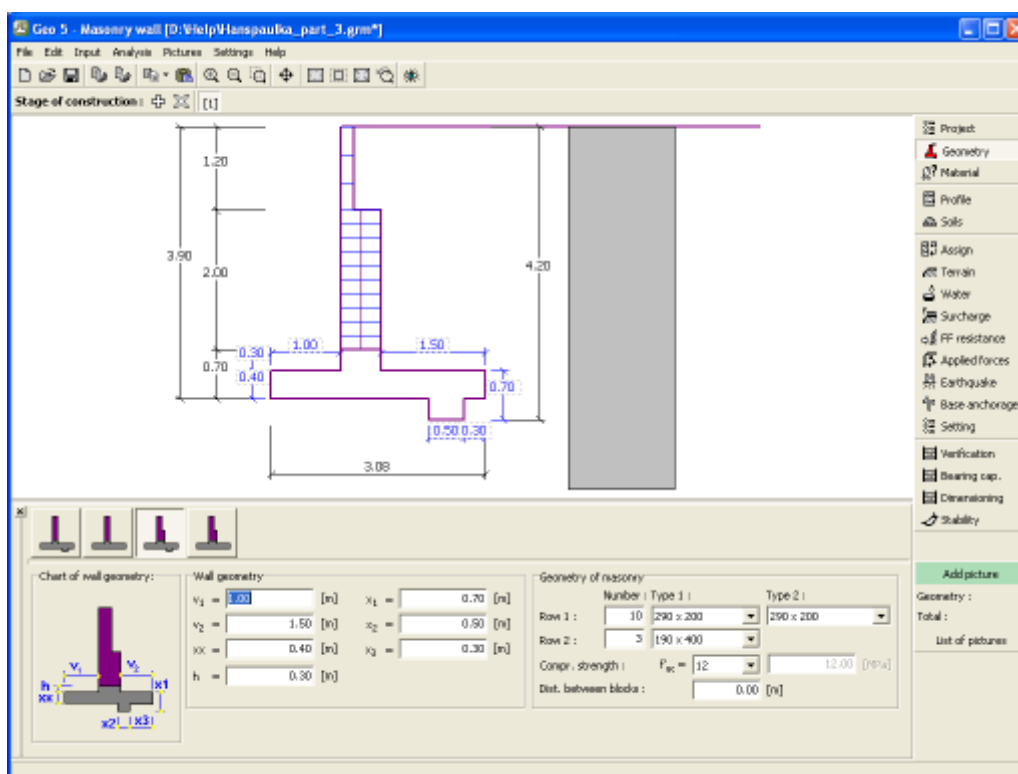
Frame "Types of blocks"

## Geometry

The "**Geometry**" frame allows by pressing the button for selecting the wall shape. The selected shape with a graphic hint "**Chart of wall geometry**" appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

Based on the selected shape of a wall, you specify in the frame "**Geometry and masonry material**" the number and dimensions of masonry blocks in individual columns, or if applicable also the thickness of vertical joint between blocks. In addition it is necessary to input compressive strength of masonry, which serves as the basic input parameter for the bearing capacity verification of reinforced masonry (according to **EN 1996-1-1** or **AS 3700**).

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Geometry"

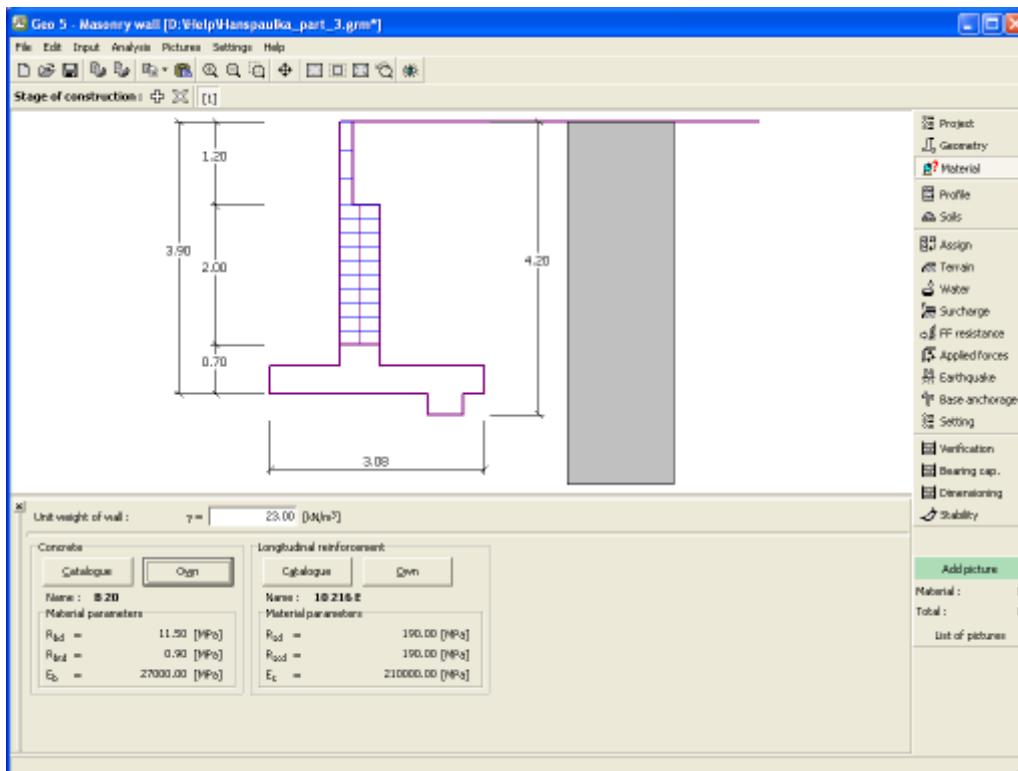
## Material

The "**Material**" frame allows to select material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- The "**Catalog**" button opens the "**Catalog of materials**" dialog window (for concrete or steel reinforcements), the list of materials then serves to select the desired material.
- The "**Own**" button opens the "**Editor of material - Concrete**" dialog window (for concrete) or the "**Editor of material - Reinforcing steel bars**" dialog window (for longitudinal steel reinforcements), which allows for manual specification of material parameters.

The catalogs content depends on the selection of standard for the design of concrete structures set in the "**Materials and standards**" tab. The input field in the upper part of the frame serves to specify the **wall unit weight**.



Frame "Material"

## Profile

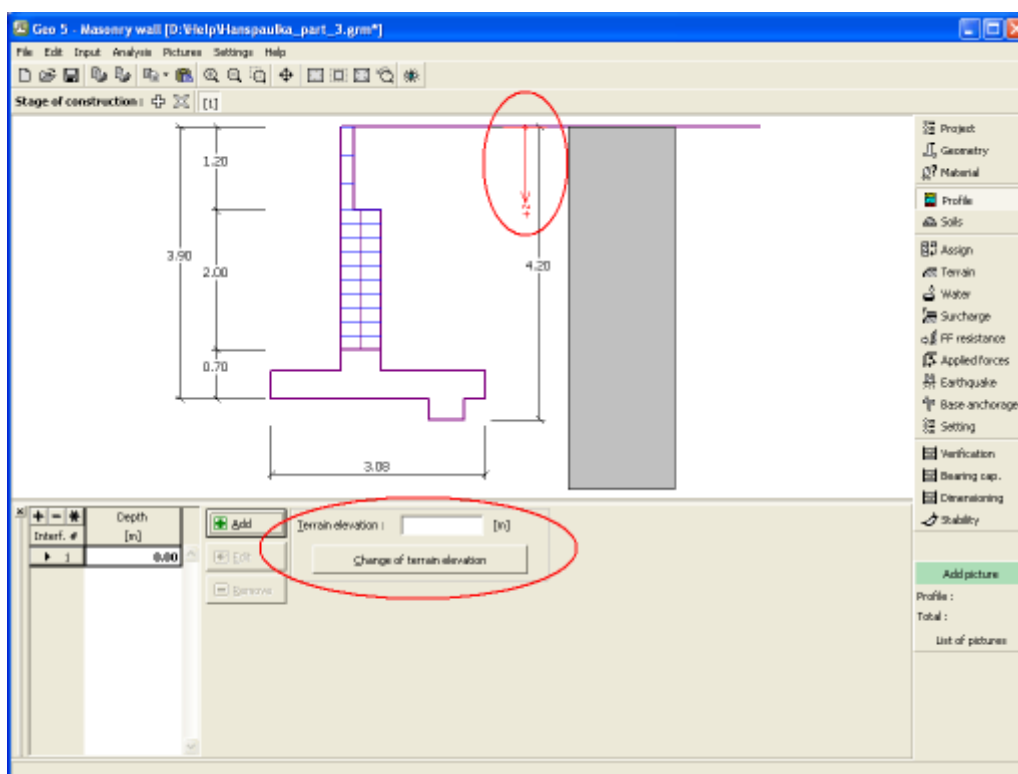
The "Profile" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "New interface" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "Change of terrain elevation" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".

The program makes it possible to import a profile in the gINT format.





Frame "Profile"

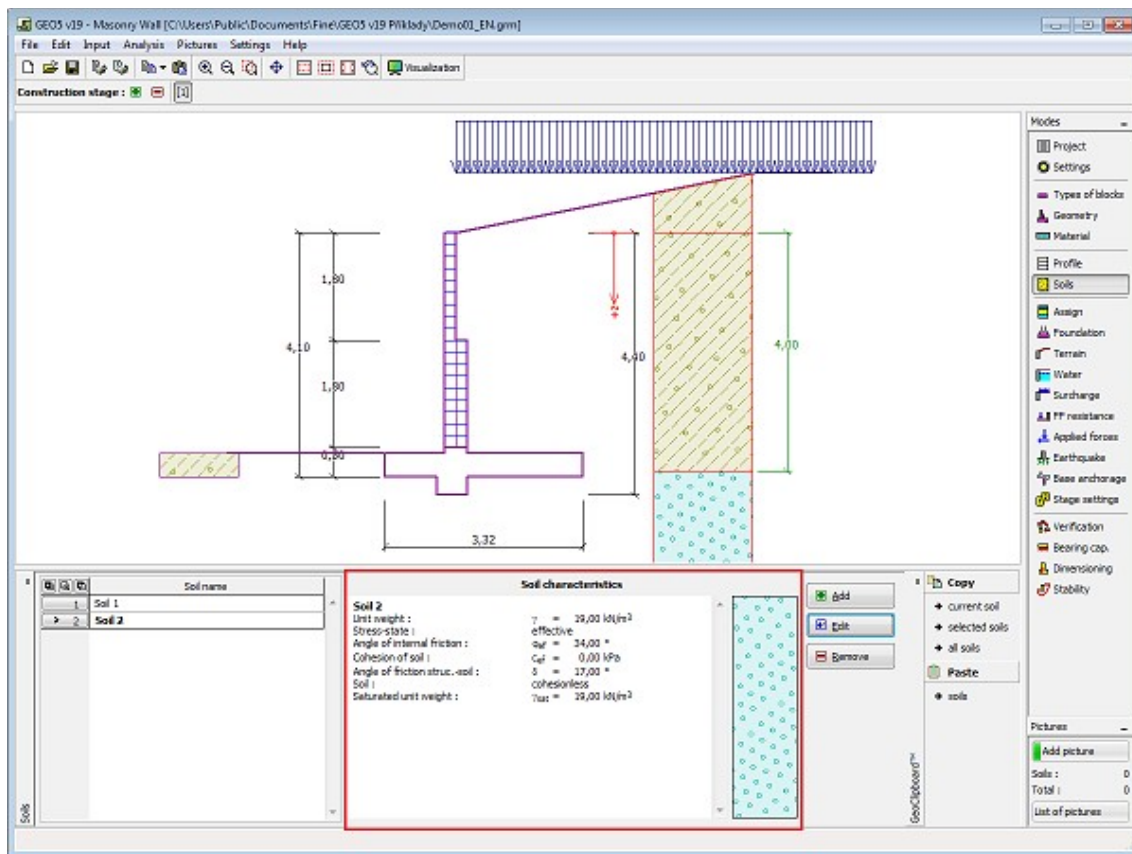
## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

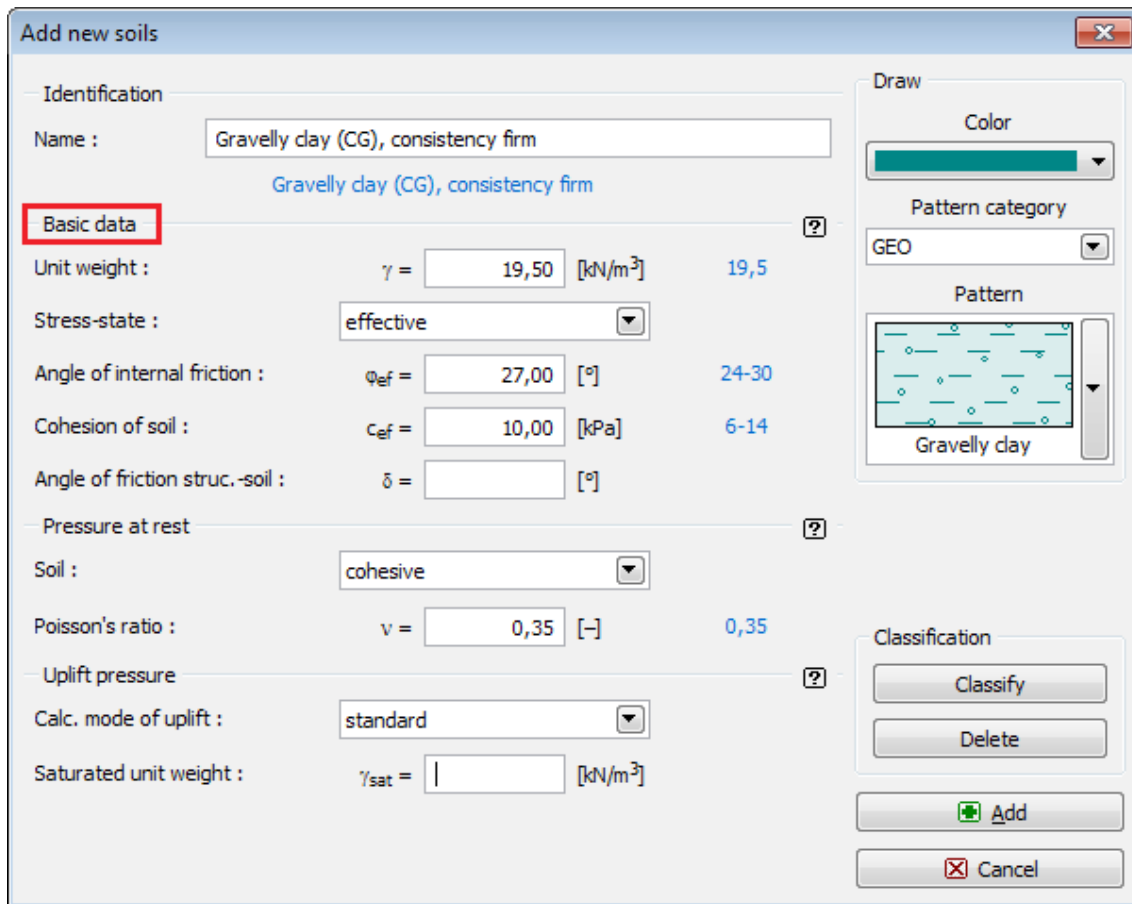
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil, type of load, structure duration and water conditions.

For **effective stress** further needs to specify the **angle of internal friction between the soil and structure**, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the [table of recommended values](#).

For **total stress** further needs to specify the **adhesion of soil to the structure face  $a$** .

The associated theory is described in detail in chapter "**Earth pressures**".



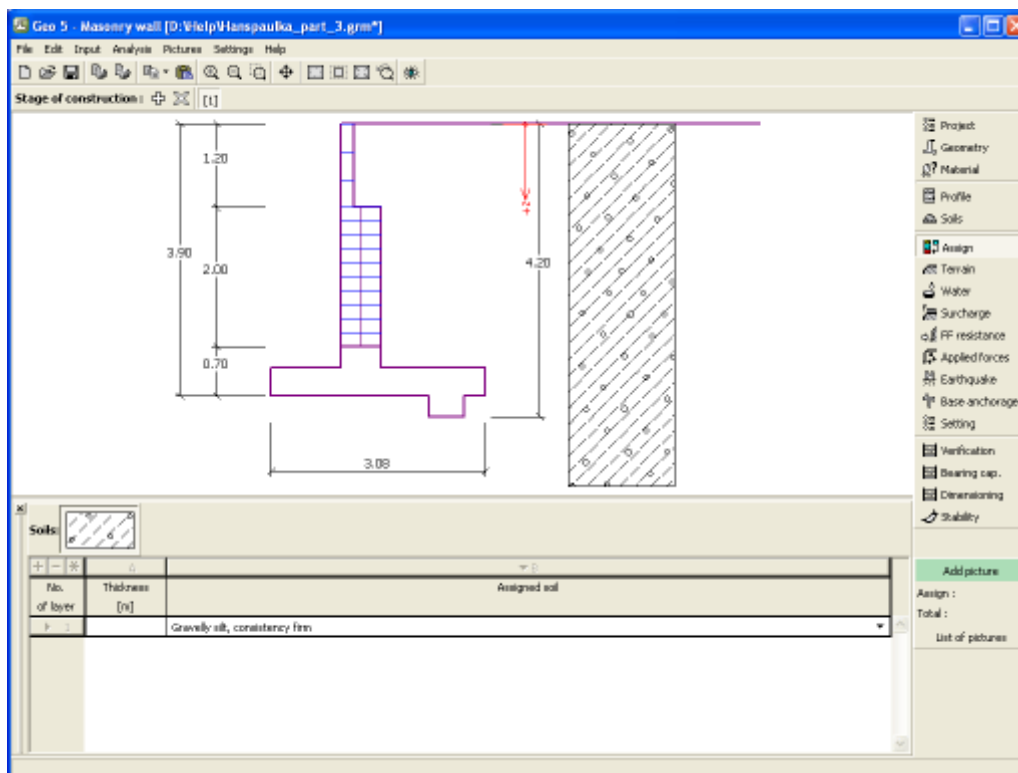
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

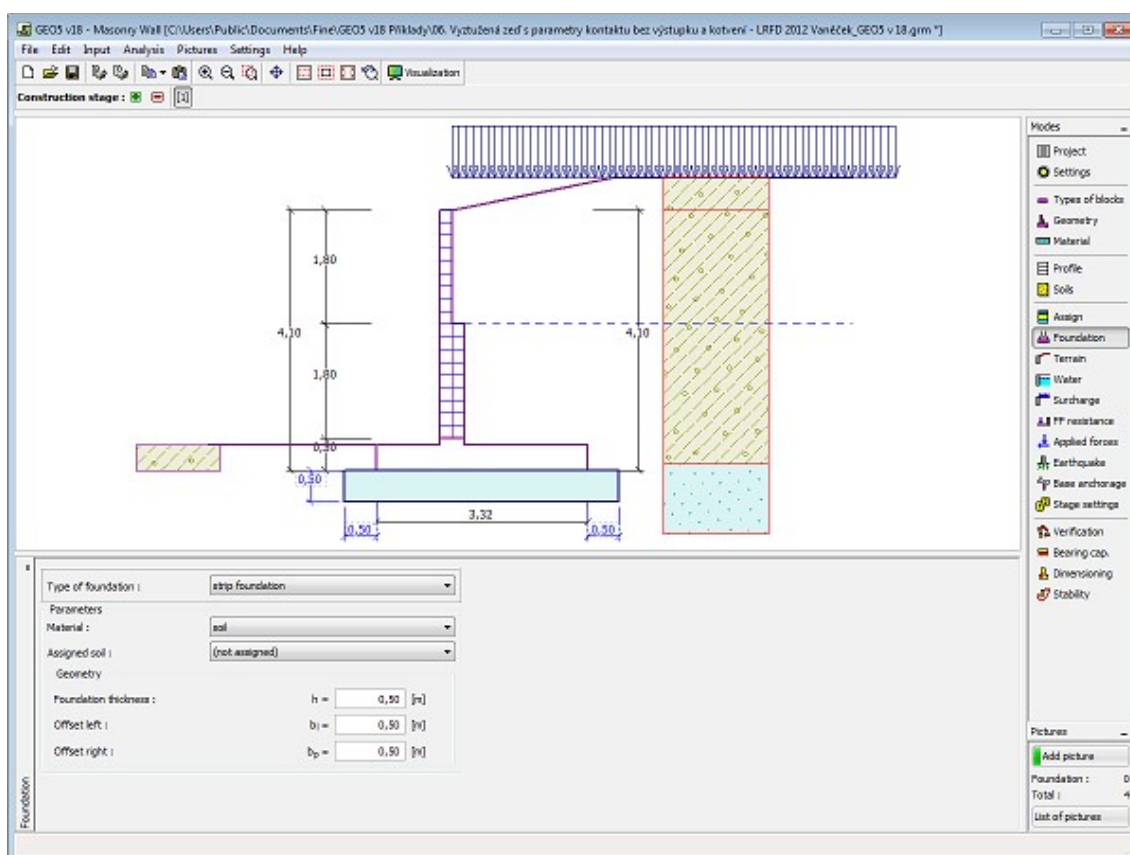
## Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle  $\psi$  [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient  $\mu$  [-]. Both options require inputting the cohesion  $a$  [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material**  $\gamma$  and **parameters of contact base-soil** (friction coefficient  $f$ , cohesion  $c$ , additional resistance  $F$ ).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

**Strip foundation** and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of **active dimensions**.

The input data introduced in this frame influence the actual **wall analysis** (check for slip) and further the **bearing capacity of foundation soil**.



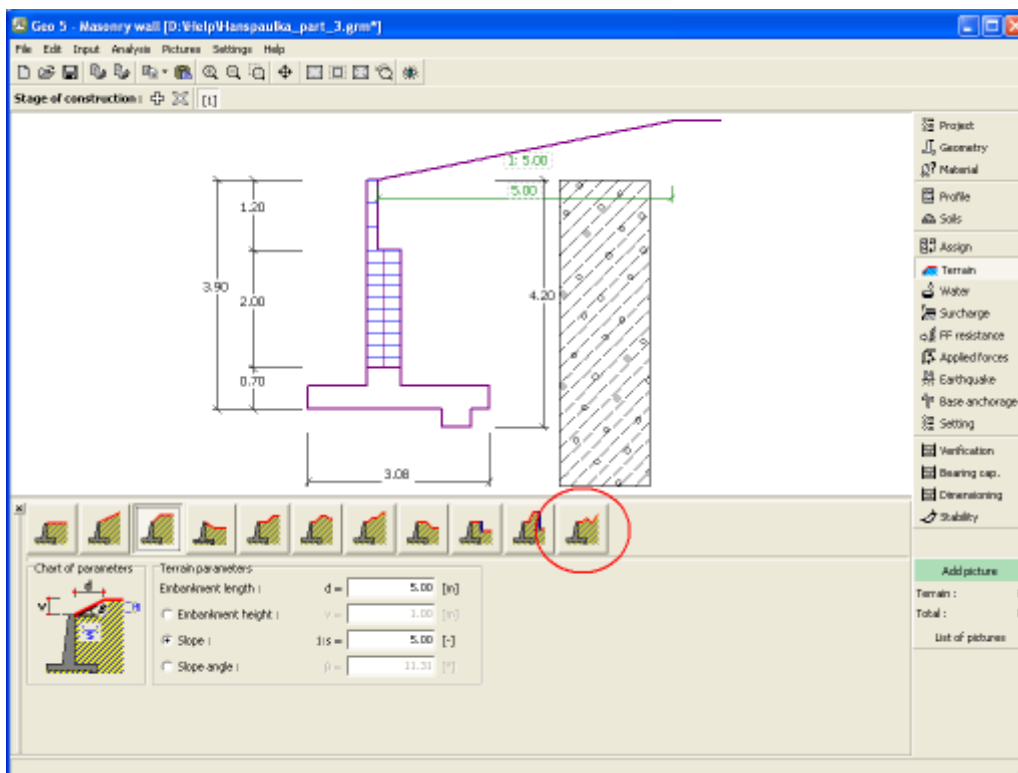
Frame "Foundation"

## Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

The "Water" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("Chart of parameters") of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

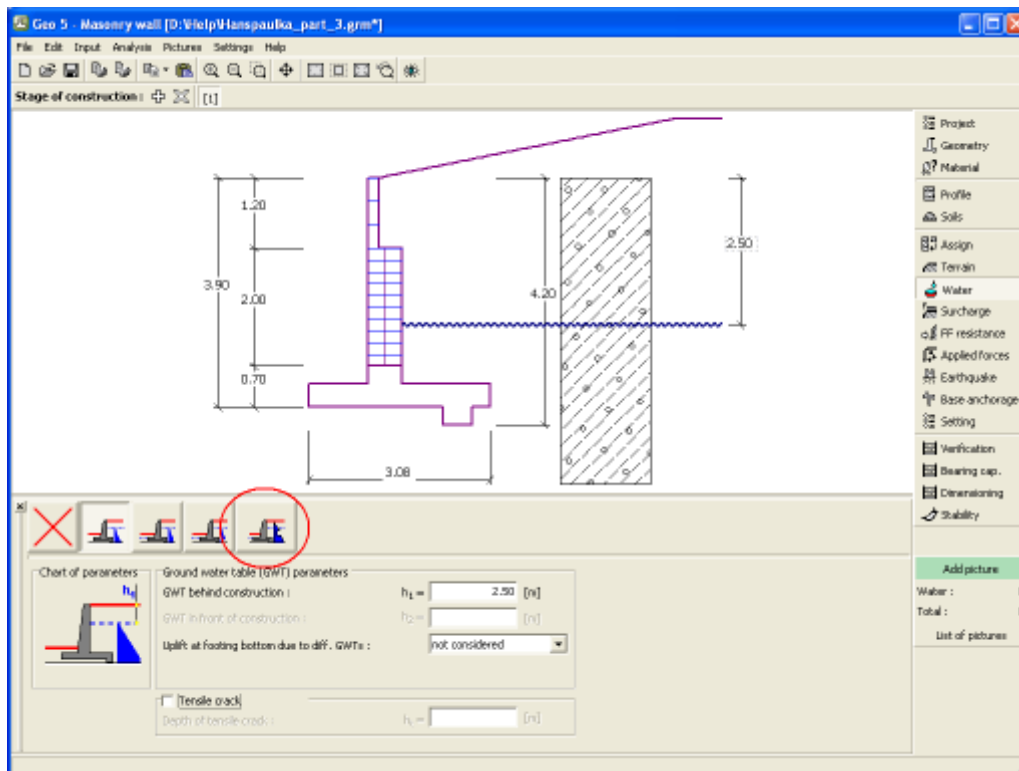
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be linear, parabolic or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "In front of structure" and "Behind structure" appear with tables. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified above the structure or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "Influence of water".

The program further allows for specifying a depth of tensile cracks filled with water.



Frame "Water"

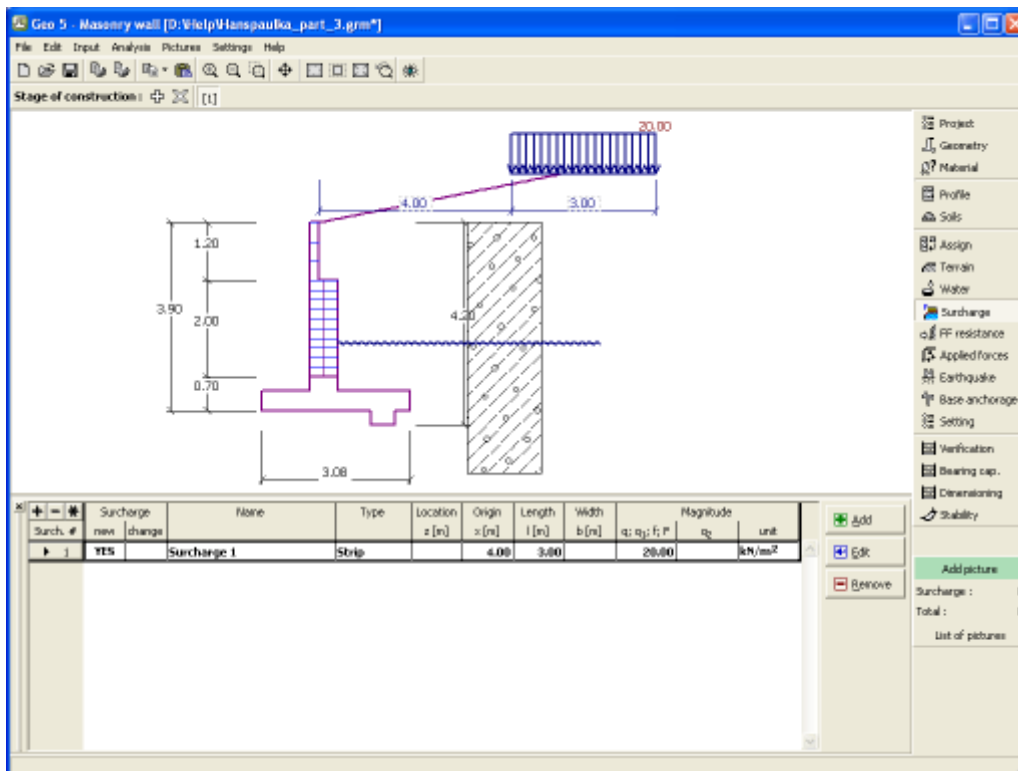
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The  $z$ -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".



Frame "Surcharge"

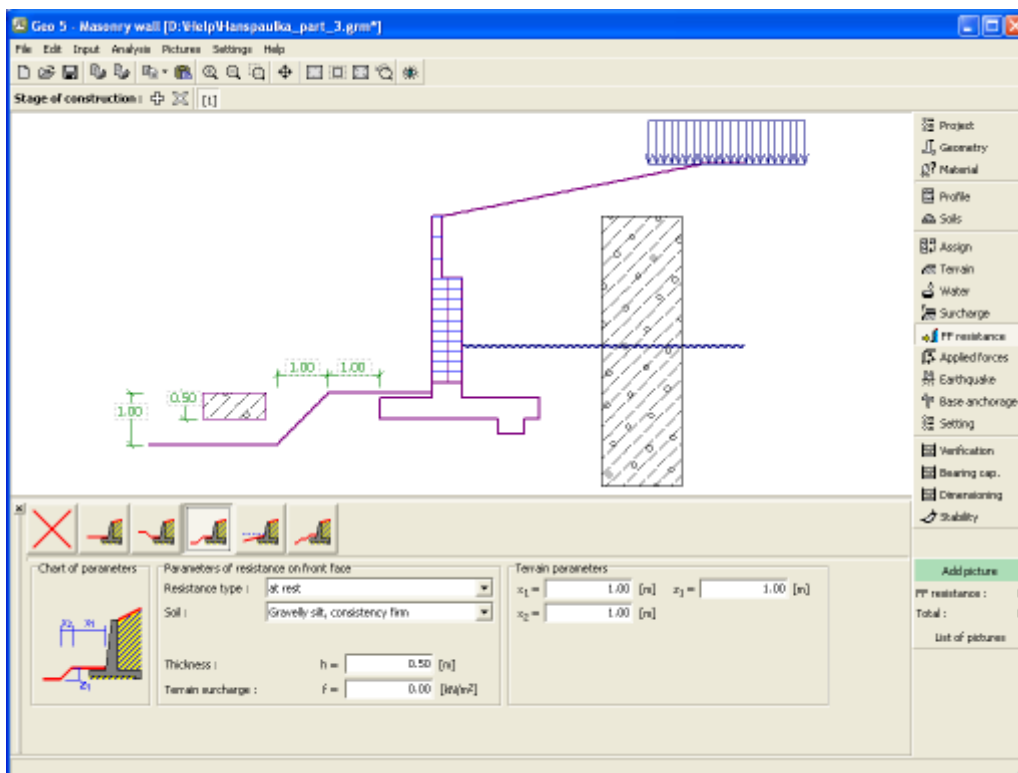
## Front Face Resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the frame "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive earth pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the input type of reduced passive pressure.





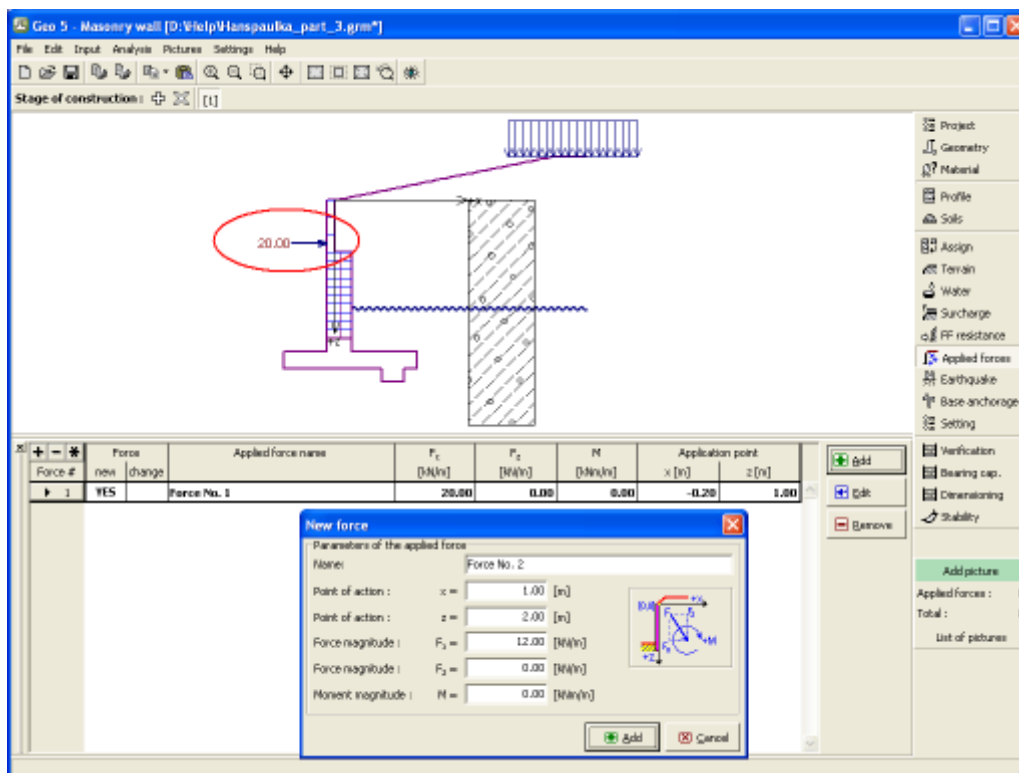
Frame "Front face resistance"

## Applied Forces

The "Applied forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.



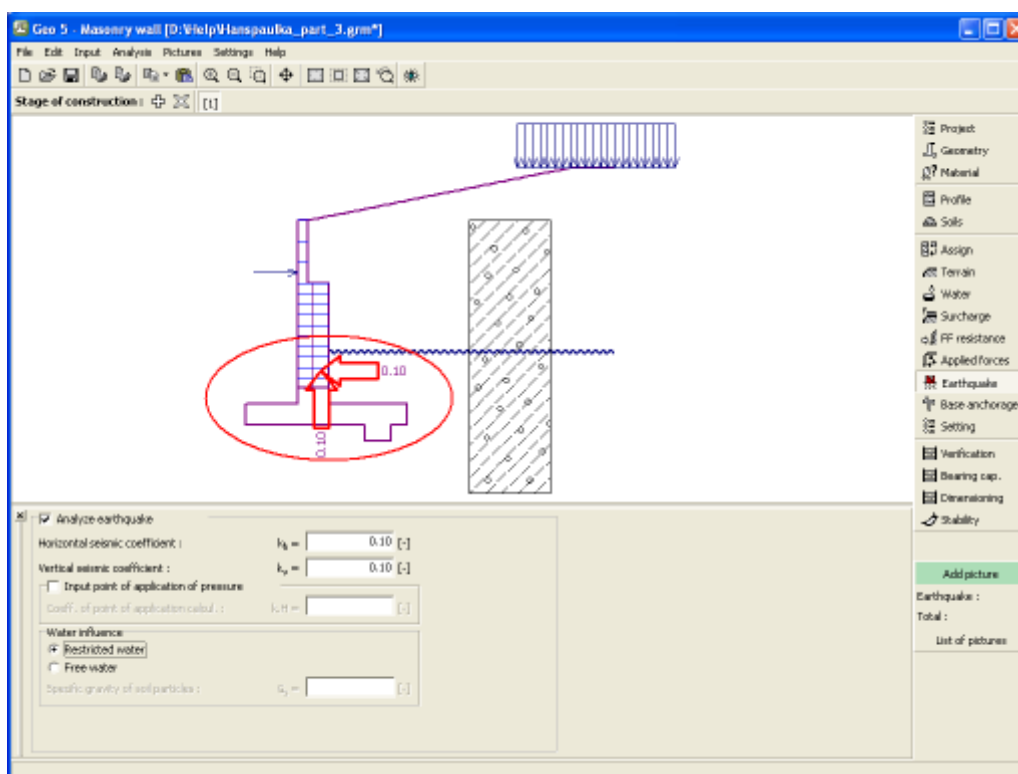
Frame "Applied forces"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

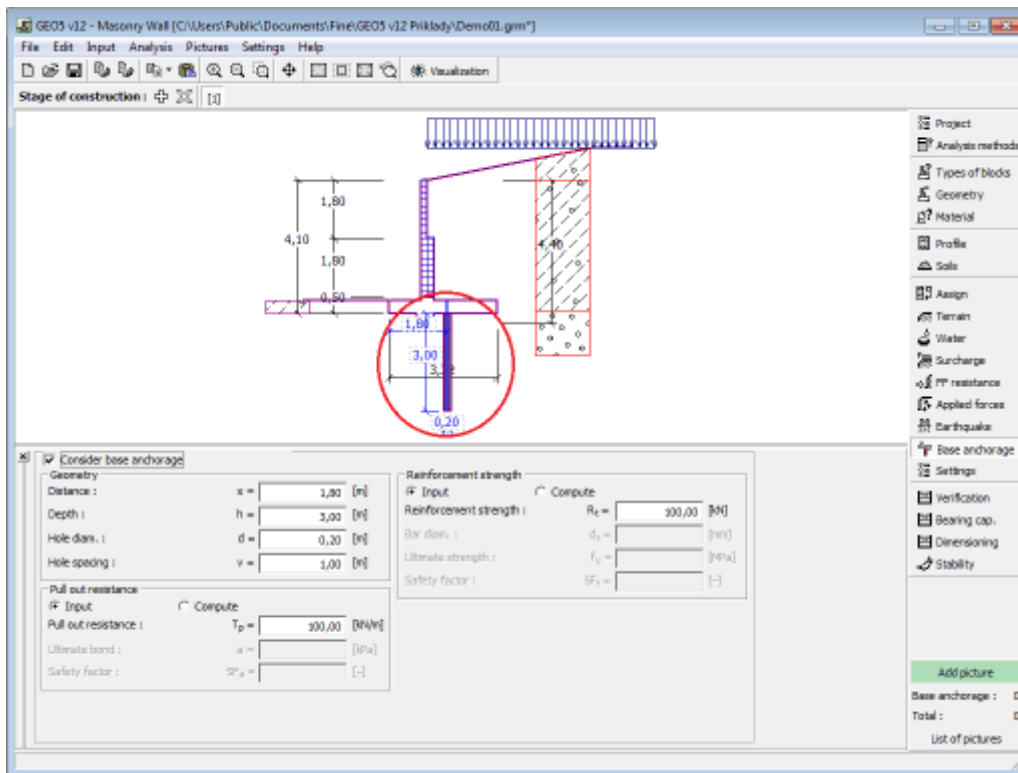
Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



Frame "Earthquake"

## Base Anchorage

The frame "**Base anchorage**" serves to input parameters (anchorage geometry, bearing capacity against pulling-out and pulling-apart) specifying an anchorage of the wall foundation. Geometry of footing anchorage can be edited either in the frame by inserting values in the inputting boxes or on the desktop with the help of **active dimensions**. The bearing capacity values can be either input or **computed by the program** from the input parameters.



Frame "Base anchorage"

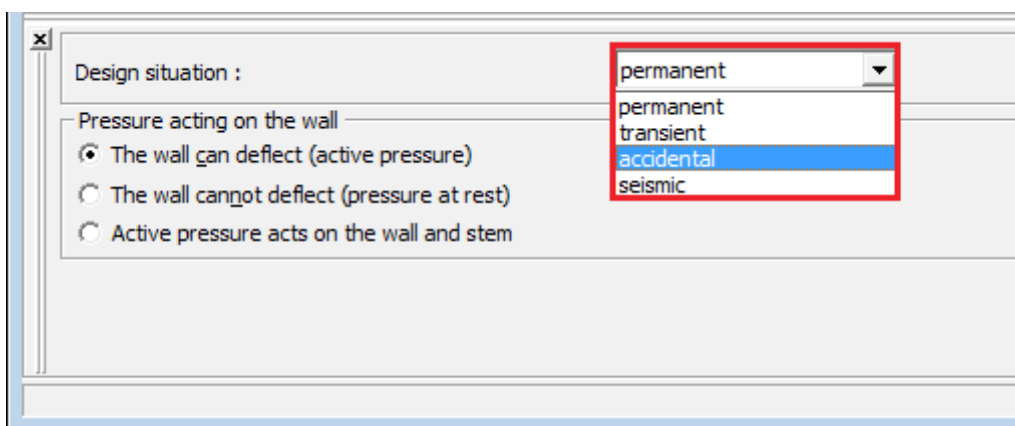
## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).

Next, the frame serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move, an active pressure is assumed, otherwise, a pressure at rest is used. The third option enables to load both the wall and stem by an active pressure.



Frame "Stage settings"

## Verification

The frame "Verification" shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology.

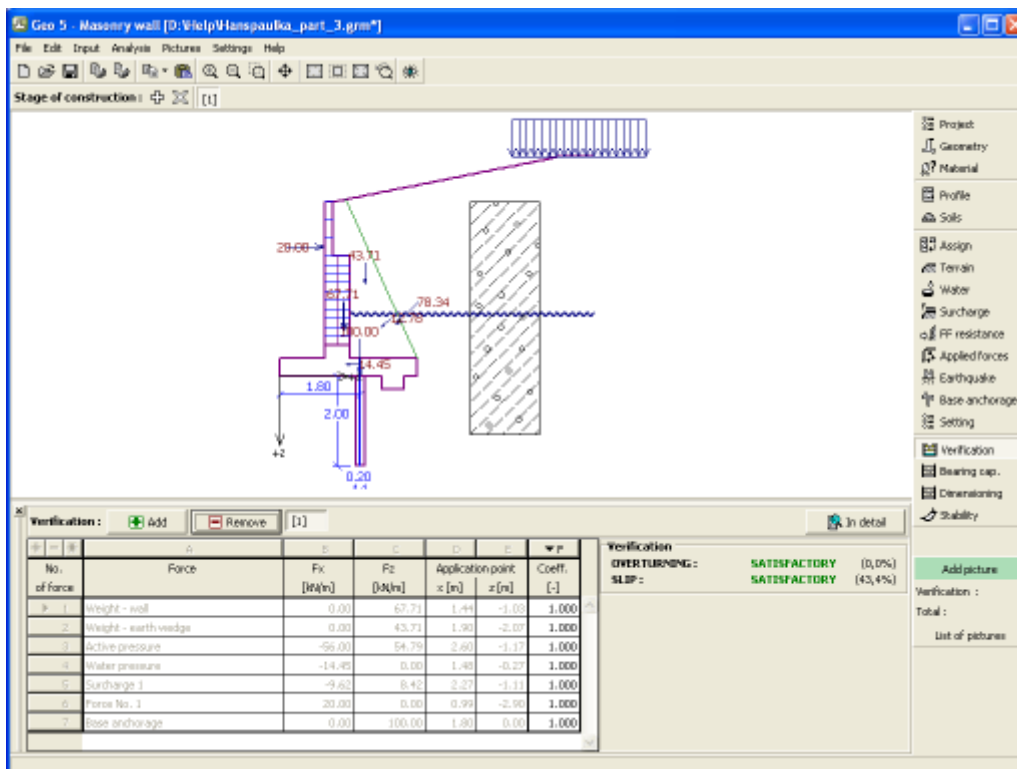
- Verification according to the factor of safety, or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in the case the last column is not displayed.

The wall is loaded either by active pressure or pressure at rest depending on input in the frame "Stage settings".

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Verification"

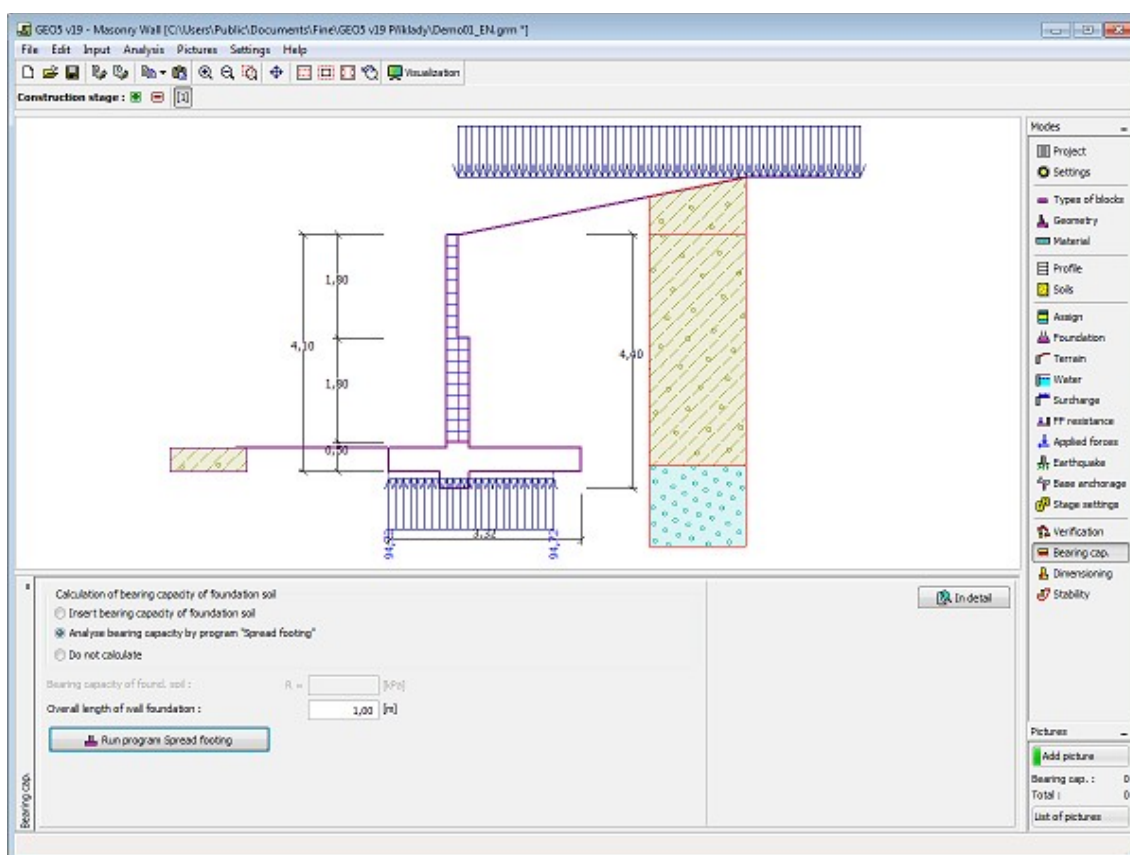
## Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "**Verification**". The program "**Spread footing**" then considers all verifications as load cases.

The frame contains following analysis options:

- **Insert bearing capacity of foundation soil** The foundation soil bearing capacity is input. The **eccentricity** and **bearing capacity** analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays detailed listing of the results.
- **Analyse bearing capacity by program "Spread footing"** Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**Masonry wall**". The "**Spread footing**" program must be installed for the button to be active. Overall length of wall foundation is input.
- **Analyse bearing capacity by program "Pile"** The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile**" is available if the wall has pile foundation (frame "**Foundation**"). Pile spacing  $s$  is input.
- **Analyse bearing capacity by program "Pile group"** The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile group**" is available if the wall has pile foundation with more than one pile (frame "**Foundation**"). Pile spacing  $s$ , overall number of pile rows  $n$  and loading length  $l$  are input.
- **Do not calculate (pile footing)** The foundation soil bearing capacity is not calculated.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Bearing capacity"

## Dimensioning

The "**Dimensioning**" frame serves to design and verify the reinforcement of wall cross-section - the cross-section subjected to dimensioning is selected in the combo list.

- **Construction joint verification** - the number of a joint between masonry blocks is input
- **Wall jump verification**
- **Verification of heel of wall**

The frame appearance is adjusted based on the selected **verification methodology**.

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

Calculation of forces and their action on the analyzed cross-section is described [here](#).

The wall is loaded either by the active earth pressure or by the pressure at rest depending on the setting in the frame "**Stage settings**". [The procedure to derive distribution of internal forces](#) in individual cross-sections is described in the theoretical part of this documentation. [In](#)

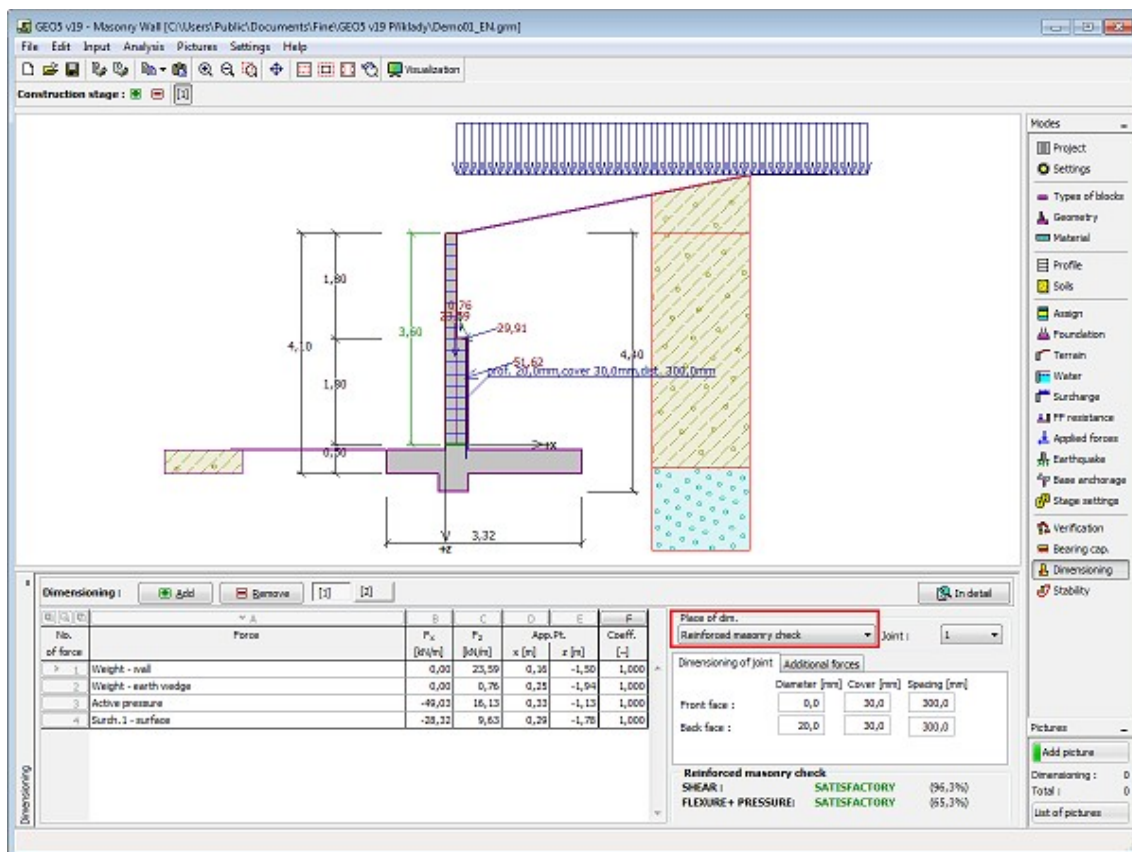
addition, force from earth pressure at rest is taken into account when considering earthquake analysis.

Joints between masonry blocks are verified according to AS 3700 or EN 1996-1-1 depending on the setting in the "Materials and standards" tab. The program verifies the bearing capacity for bending, shear and combination of compression and bending. Reinforcement can be specified on both front and back sides of a structure. An additional load applied to a cross-section (bending moment, compressive normal force and shear force) can also be specified. These additional forces are added to the computed ones.

Dimensioning of reinforced concrete is performed according to the standard set in the "Materials and standards" tab.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Dimensioning"

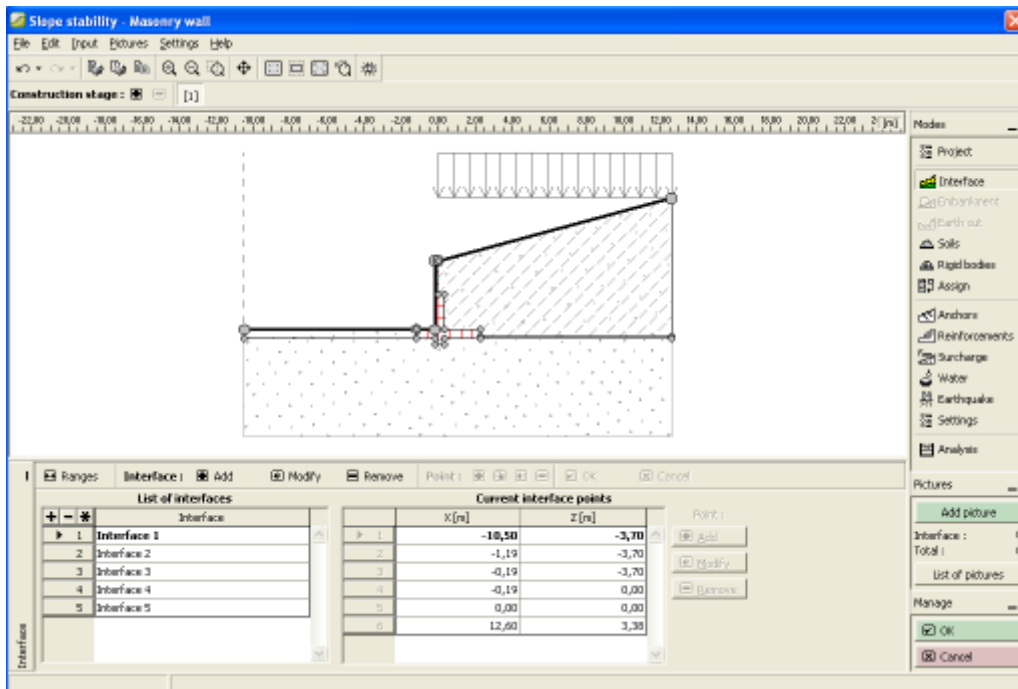
## Stability

Pressing the "Stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program - all data are then



carried over to the analysis protocol of the "Masonry wall" program.



Frame "Stability"

## Program Gabion

This program is used for analysis of gabions. It allows analysis of any structure shapes including overhangs requiring anchoring.

The help in the program "Gabion" includes the following topics:

- Input of data into individual frames:

Project	Settings	Material	Geometry	Profile	Soils	Assign
Foundation	Terrain	Water	Surcharge	Front Face Resistance	Applied Forces	Earthquake
Stage Settings	Verification	Bearing Capacity	Dimensionin g	Stability		

- Standards and analysis methods

- Theory for analysis in the program "Gabion":

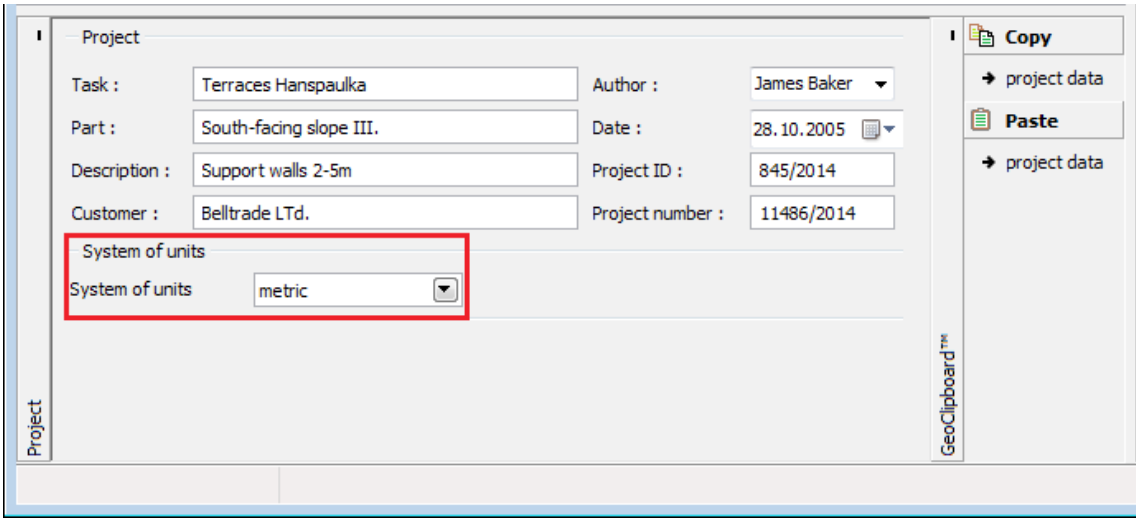
Stress in Soil Body	Earth Pressures	Analysis of Walls	Analysis of Foundation Bearing Capacity
---------------------	-----------------	-------------------	---

- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.



The screenshot shows the 'Project' frame with the following data:

Field	Value
Task	Terraces Hanspaulka
Part	South-facing slope III.
Description	Support walls 2-5m
Customer	Belltrade LTd.
Author	James Baker
Date	28.10.2005
Project ID	845/2014
Project number	11486/2014
System of units	metric

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

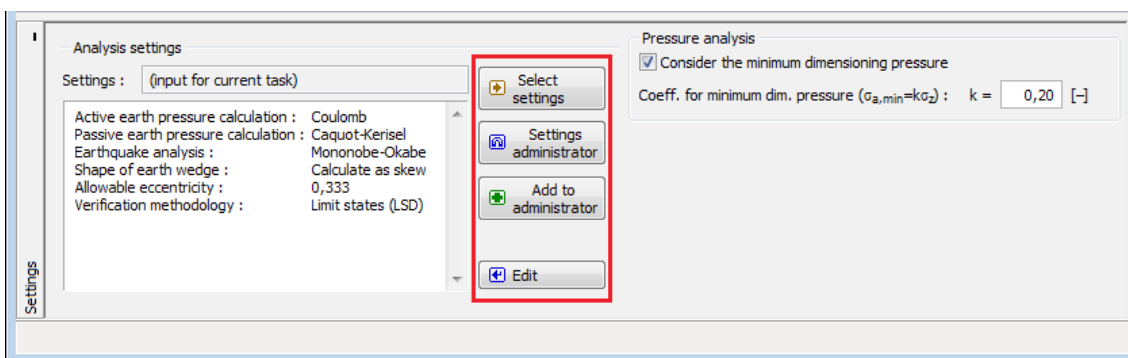
The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Program allows to specify a value of the **minimum dimensioning pressure  $k$**  (by checking the option "**Consider the minimum dimensioning pressure**"). For real structures showed that when considering cohesive soil behind the wall of the upper blocks of structure behaves problematically. For this reason, it is recommended to implement a value of  $k$  into the calculation for the case of backfill consisting of cohesive soils.

Settings of analysis parameters are performed in the "**Materials and standards**" and "**Wall analysis**" tabs.



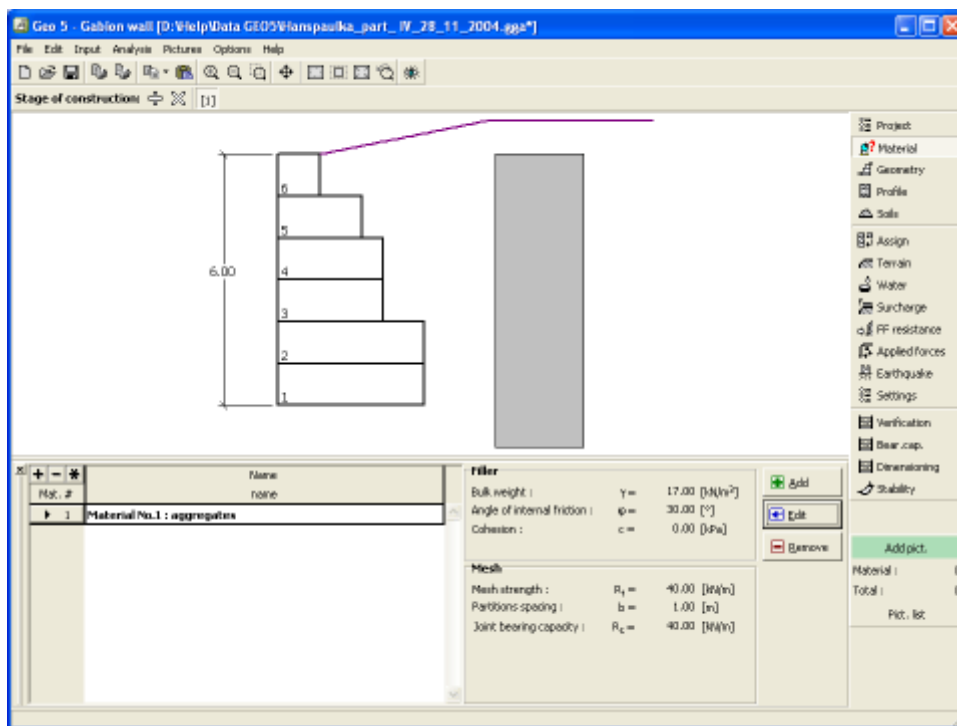
Frame "Settings"

## Material

The "**Material**" frame contains a **table** with a list of input filling (aggregates) and material parameters of applied gabion wire netting. Adding (Editing) material and netting is performed in the "**New material (Edit material)**" **dialog window**.

The material parameters of filling and netting of currently selected gabion block are displayed in the right part of the frame.

An approximate value of the angle of internal friction of the material of gabion filler is for a well graded gravel in the range of 35 - 40, for a quarry masonry it can be larger.



Frame "Material"

## Geometry

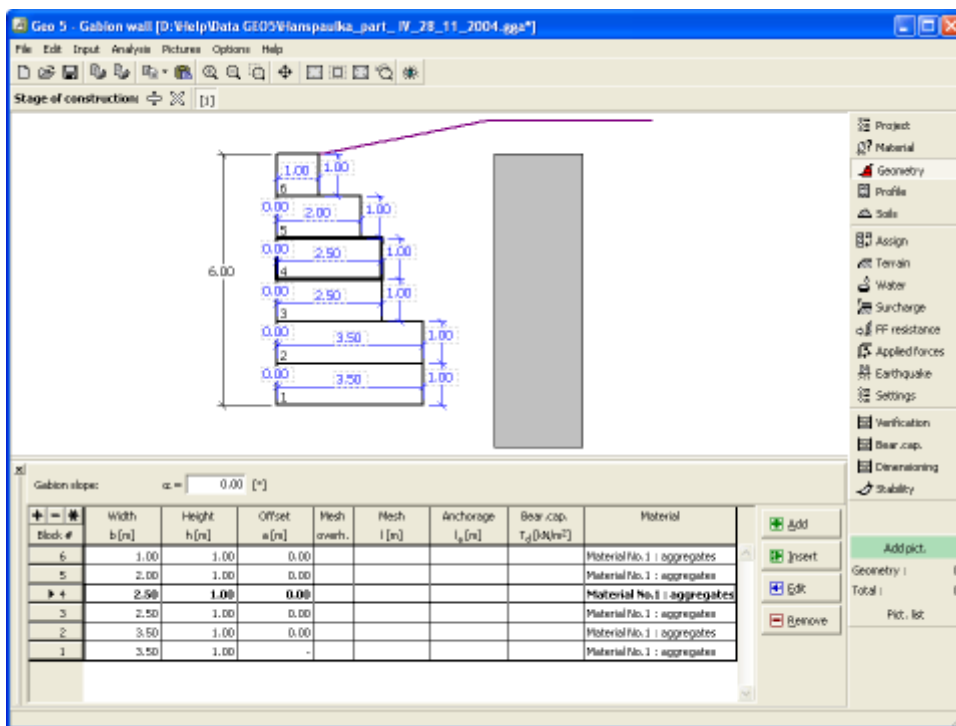
The "Geometry" frame contains a table with a list of input blocks of a wall (the lowest block is labeled as No. 1). Adding (editing) blocks is performed in the "New block (Edit block)" dialog window.

This dialog window serves to define the geometry of a block, and parameters of mesh overhang (overhang length, overhang anchorage, bearing capacity against pull out).

The program allows for adding (inserting) another block in between two already existing blocks of a structure. Inserting a new block is performed in the "Insert block" dialog window that complies with the "New block" dialog window. The inserted block is ordered such to proceed the currently selected block of a structure.

The input blocks can be further edited on the desktop with the help of active dimensions or active objects - double clicking on a structure opens a dialog window with a given block. When using the regime of active objects the visualization of detailed dimensions must be turned off in the "Visualization style settings" dialog window.

The program makes it possible to export the geometry of a structure in the \*.DXF format.



Frame "Geometry"

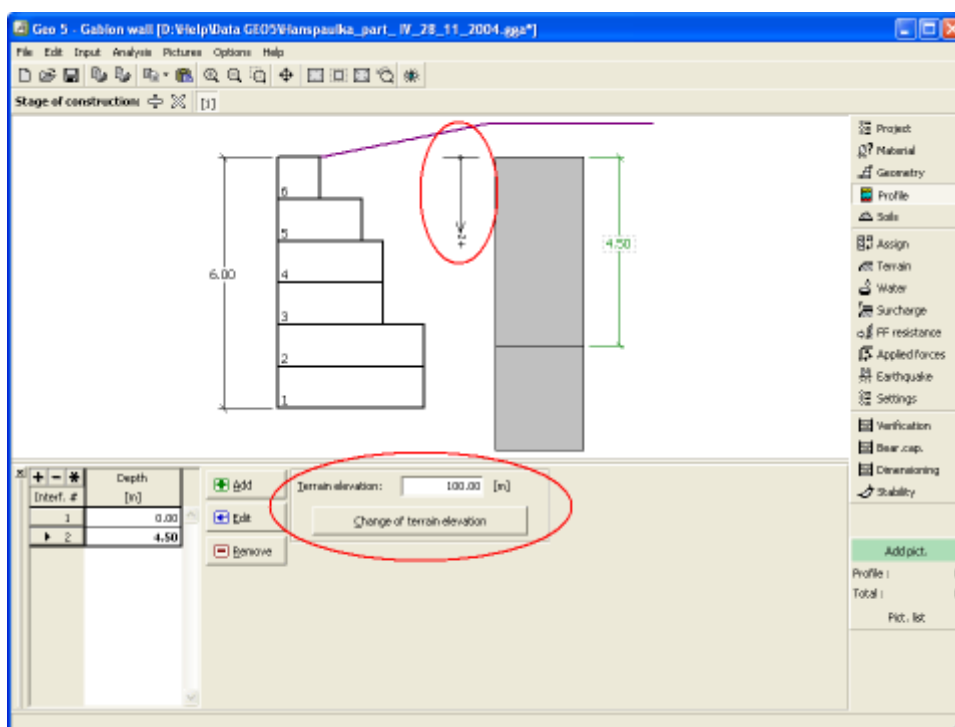
## Profile

The "Profile" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "New interface" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "Change of terrain elevation" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".

The program makes it possible to import a profile in the gINT format.



Frame "Profile"

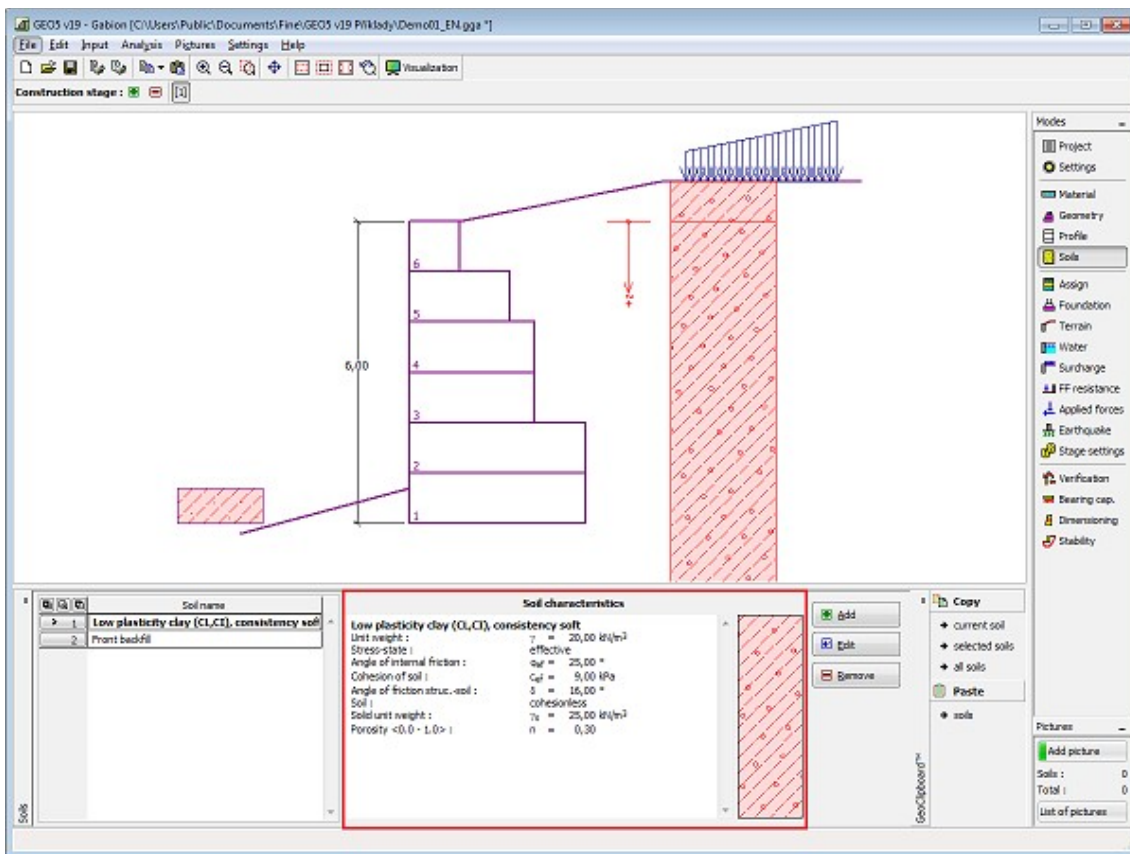
## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil, type of load, structure duration and water conditions.

For **effective stress** further needs to specify the **angle of internal friction between the soil and structure**, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the [table of recommended values](#).

For **total stress** further needs to specify the **adhesion of soil to the structure face**  $a$ .

The associated theory is described in detail in chapter "**Earth pressures**".

The screenshot shows the 'Add new soils' dialog window with the 'Basic data' tab selected. The 'Name' field contains 'Sandy clay (CS), consistency firm'. The 'Basic data' section includes the following fields:

- Unit weight:  $\gamma = 18,50$  [kN/m<sup>3</sup>] (value: 18,5)
- Stress-state: effective
- Angle of internal friction:  $\phi_{ef} = 24,50$  [°] (range: 22-27)
- Cohesion of soil:  $c_{ef} = 14,00$  [kPa] (range: 10-18)
- Angle of friction struc.-soil:  $\delta =$  [°]
- Soil: cohesive
- Poisson's ratio:  $\nu = 0,35$  [-] (value: 0,35)
- Calc. mode of uplift: standard
- Saturated unit weight:  $\gamma_{sat} =$  [kN/m<sup>3</sup>]

The 'Draw' section includes a color selector (green), a pattern category dropdown (GEO), and a pattern preview for 'Sandy clay'. The 'Classification' section contains buttons for 'Classify', 'Delete', 'Add', and 'Cancel'.

Dialog window "Add new soils" - "Basic data"

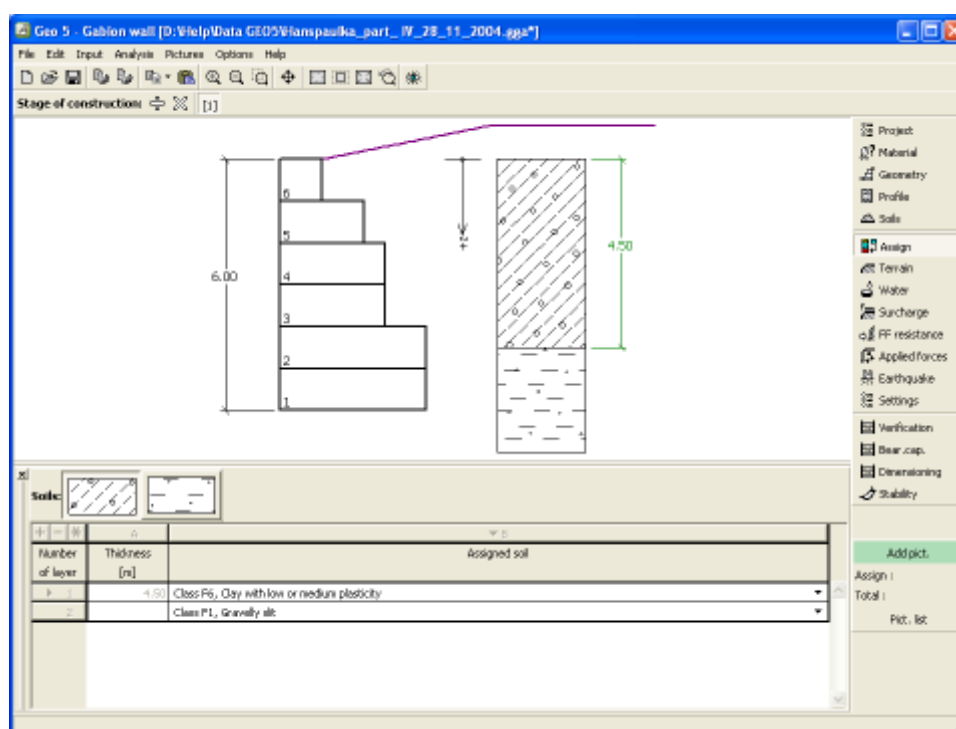
## Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).





Frame "Assign"

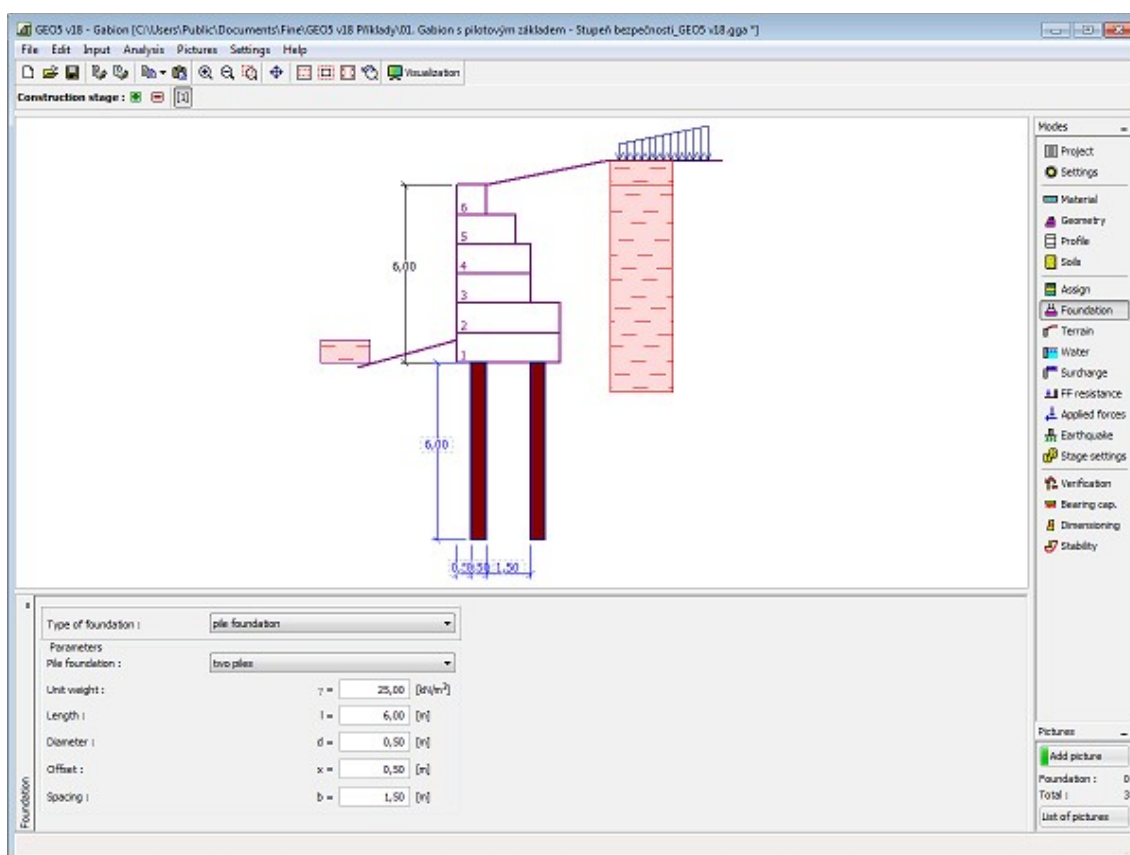
## Foundation

The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle  $\psi$  [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient  $\mu$  [-]. Both options require inputting the cohesion  $a$  [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material**  $\gamma$  and **parameters of contact base-soil** (friction coefficient  $f$ , cohesion  $c$ , additional resistance  $F$ ).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

**Strip foundation** and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of wall foundation (**strip foundation, pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of **active dimensions**.

The input data introduced in this frame influence the actual **wall analysis** (check for slip) and further the **bearing capacity of foundation soil**.



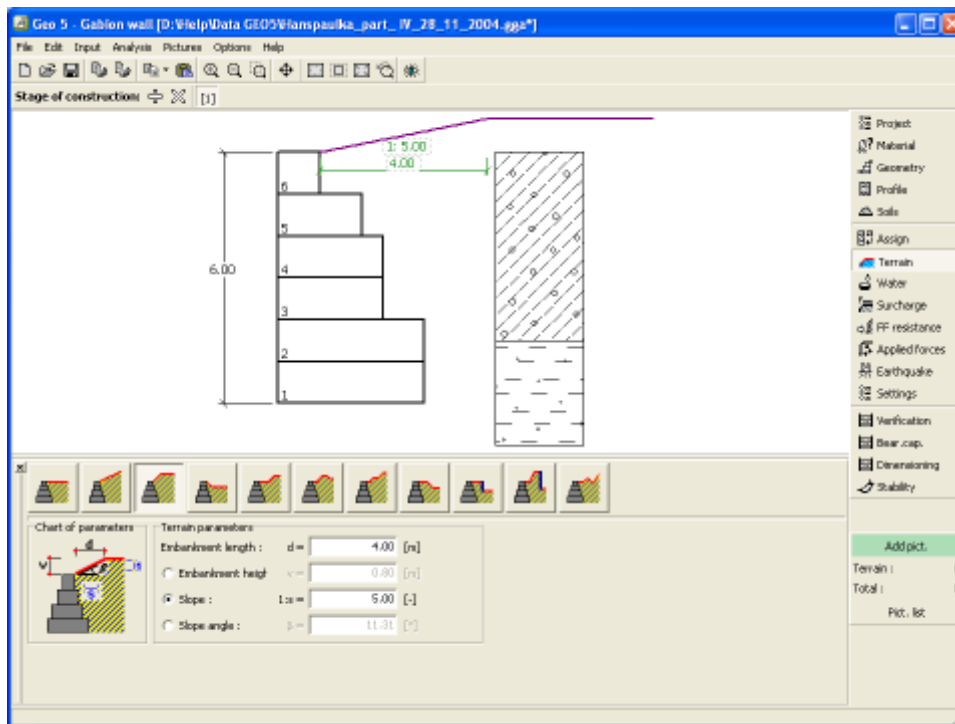
Frame "Foundation"

## Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

The **"Water"** frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint (**"Chart of parameters"**) of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

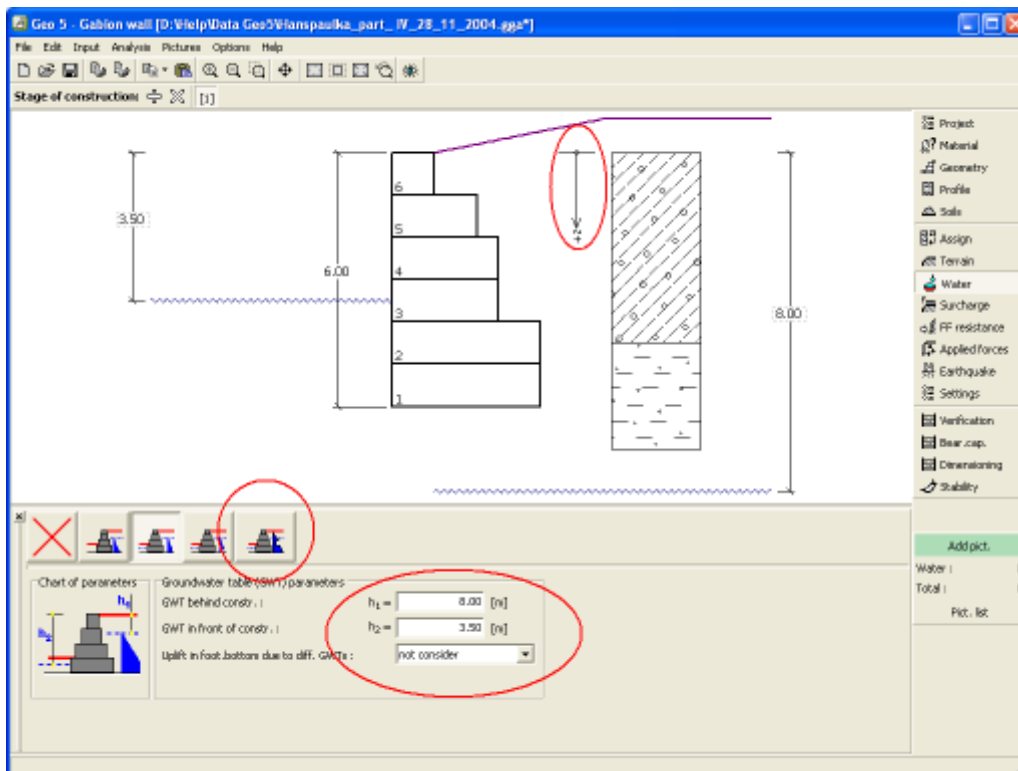
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be **linear, parabolic** or it may not be considered at all. When verifying the wall, the uplift pressure in foundation joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs **"In front of structure"** and **"Behind structure"** appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter **"Influence of water"**.

The program further allows for specifying a depth of **tensile cracks** filled with water.



Frame "Water"

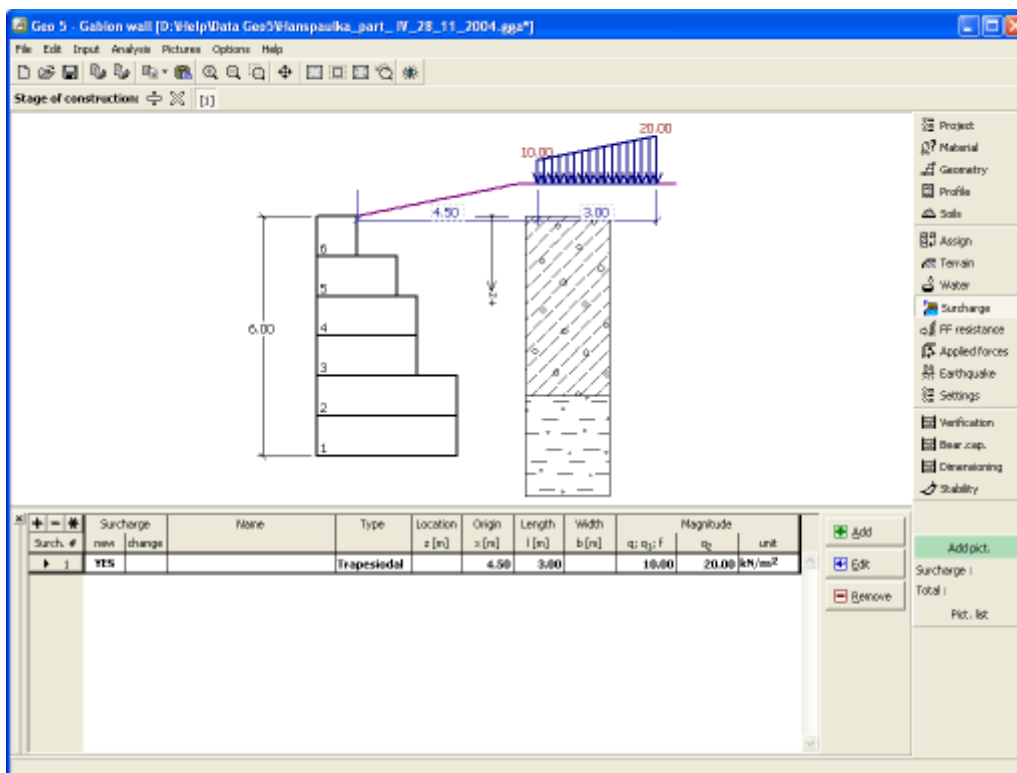
## Surcharge

The "Surcharge" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "New surcharge" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either permanent, variable or accidental surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



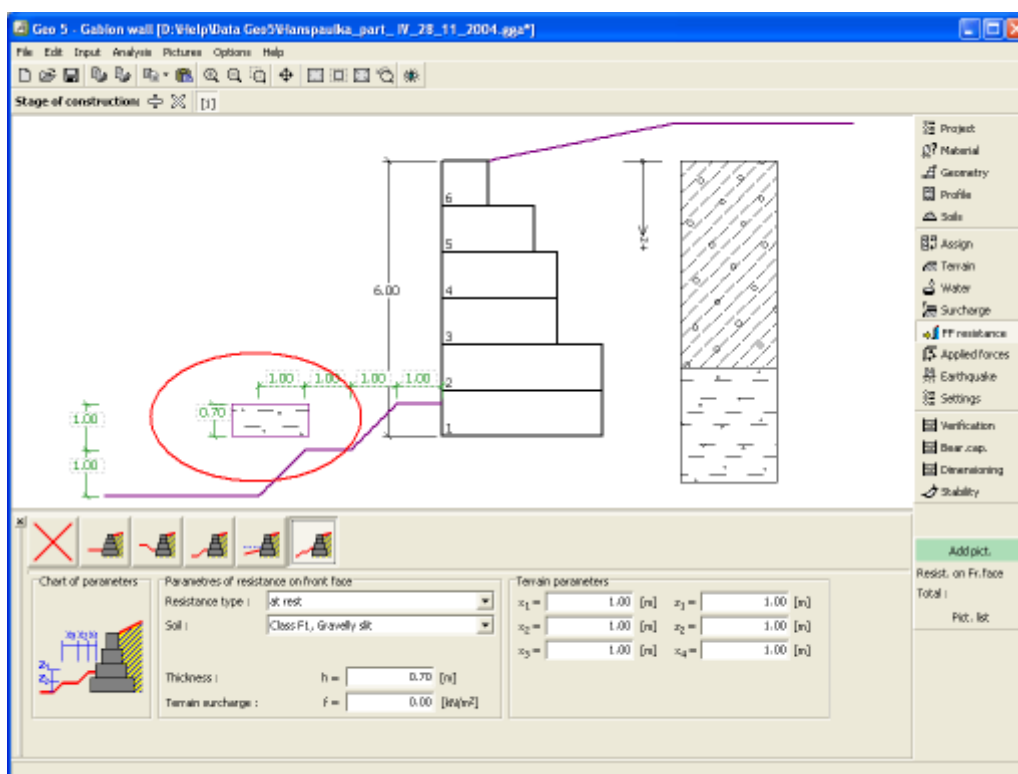
Frame "Surcharge"

## Front Face Resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint ("Chart of parameters") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the frame "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive earth pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the input type of reduced passive pressure.



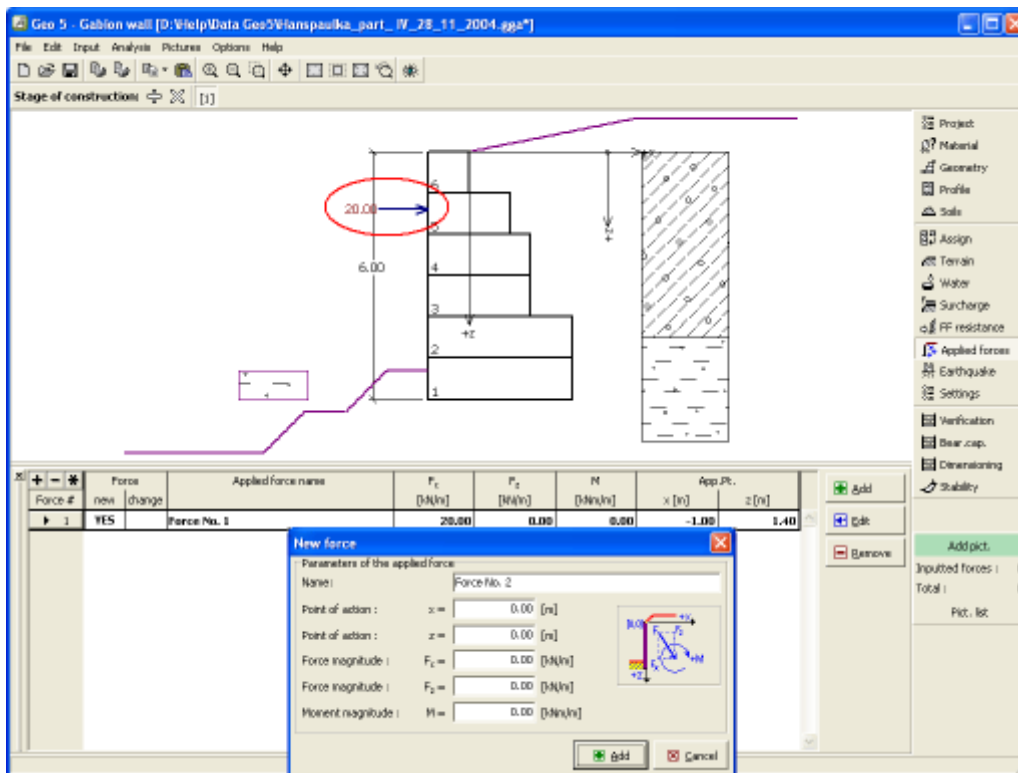
Frame "Front face resistance"

## Applied Forces

The "Applied forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.



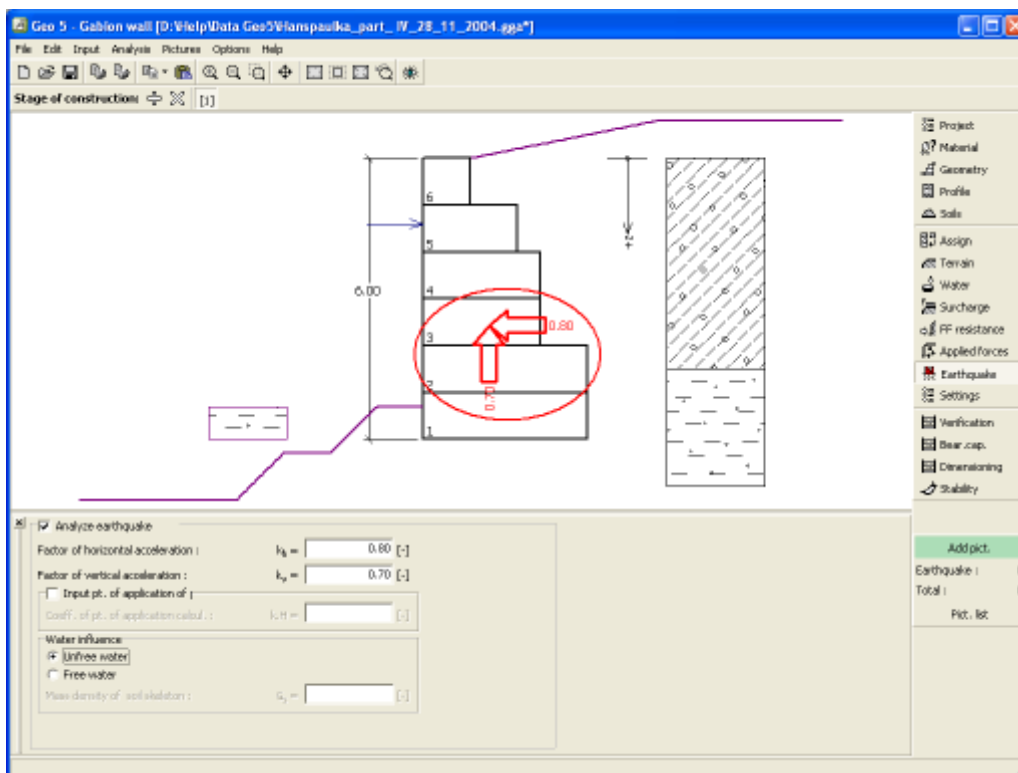
Frame "Applied forces"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".

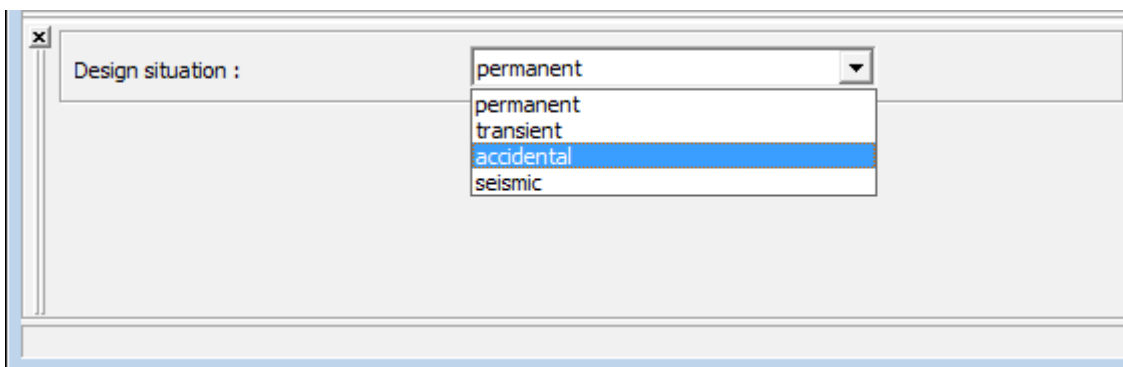


Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage. Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Verification

The frame "Verification" shows the analysis results. Several computations can be carried out for a single task.



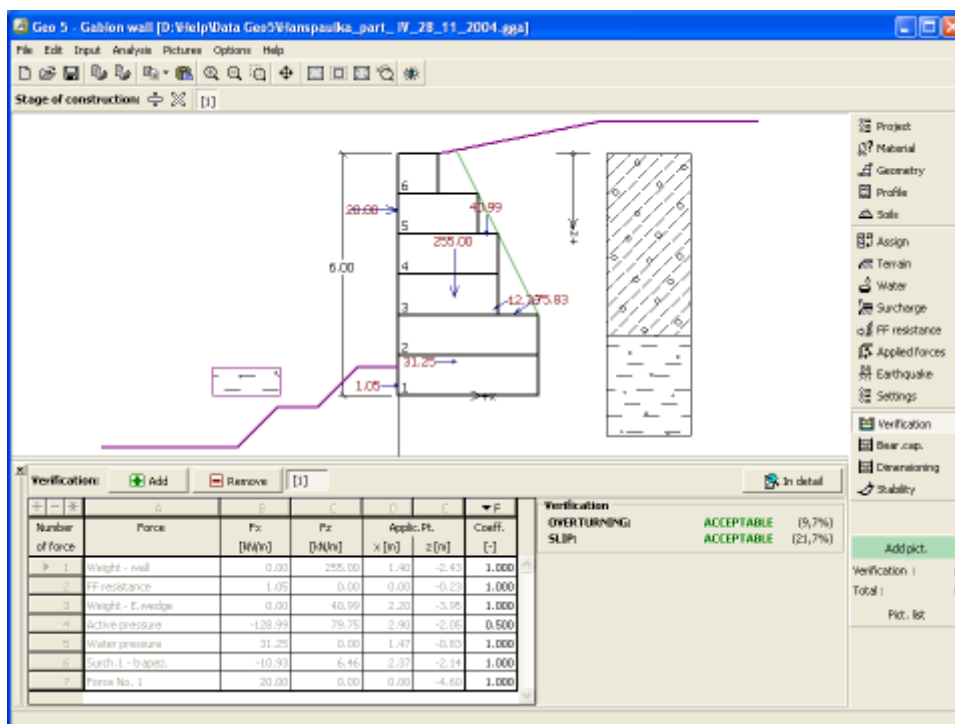
The frame appearance is adjusted based on the selected **verification methodology**.

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

The procedure for **wall verification** is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The **"In detail"** button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Verification"

## Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "**Verification**". The program "**Spread footing**" then considers all verifications as load cases.

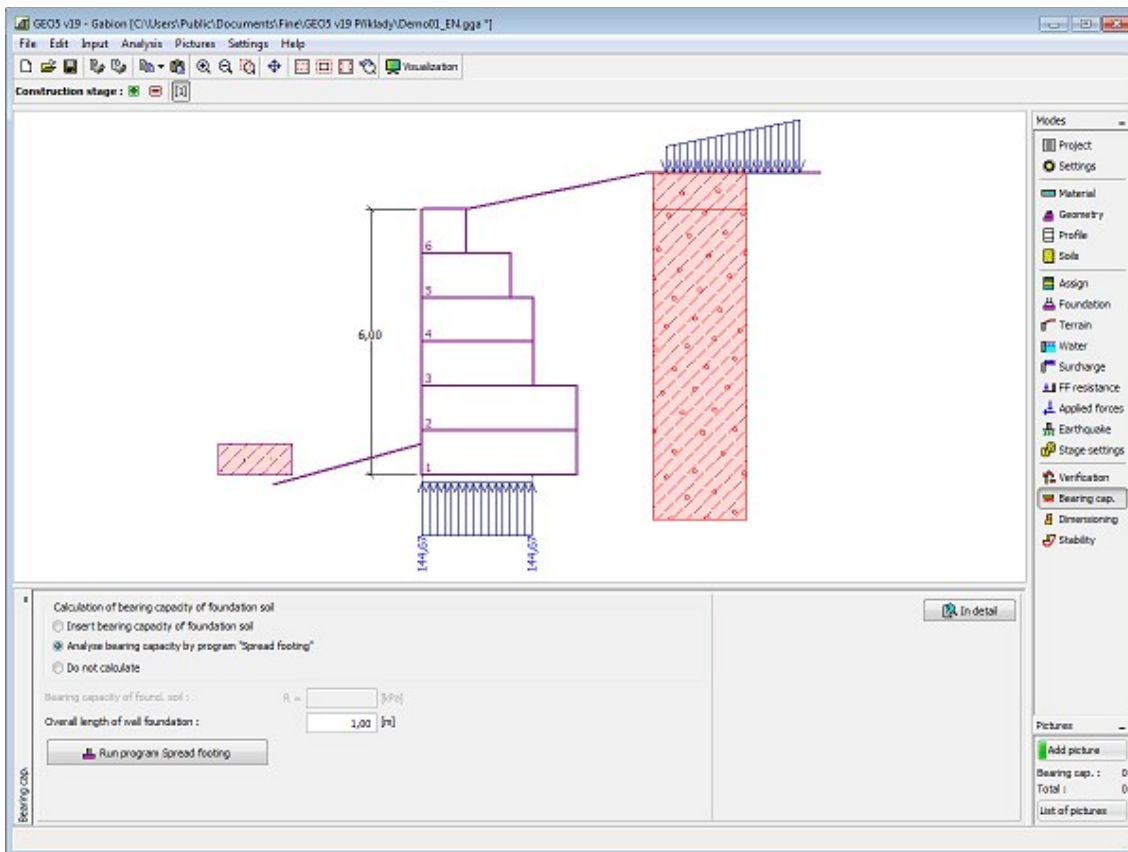
The frame contains following analysis options:

- **Insert bearing capacity of foundation soil** The foundation soil bearing capacity is input. The **eccentricity** and **bearing capacity** analysis results are

displayed in the right part of the frame. The **"In detail"** button opens a dialog window that displays detailed listing of the results.

- Analyse bearing capacity by program "Spread footing"** Pressing the **"Run program Spread footing"** button opens the program **"Spread footing"** which allows to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the **"OK"** button leaves the analysis mode - the results and all plots are transferred into the program **"Gabion"**. The **"Spread footing"** program must be installed for the button to be active. Overall length of wall foundation is input.
- Analyse bearing capacity by program "Pile"** The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile"** is available if the wall has pile foundation (frame **"Foundation"**). Pile spacing  $s$  is input.
- Analyse bearing capacity by program "Pile group"** The procedure is identical as if calculating soil bearing capacity by the **"Spread footing"** program. The **"Run program Pile group"** is available if the wall has pile foundation with more then one pile (frame **"Foundation"**). Pile spacing  $s$ , overall number of pile rows  $n$  and loading length  $l$  are input.
- Do not calculate (pile footing)** The foundation soil bearing capacity is not calculated.

Visualization of results can be adjusted in the frame **"Visualization settings"**.



Frame "Bearing capacity"

## Dimensioning

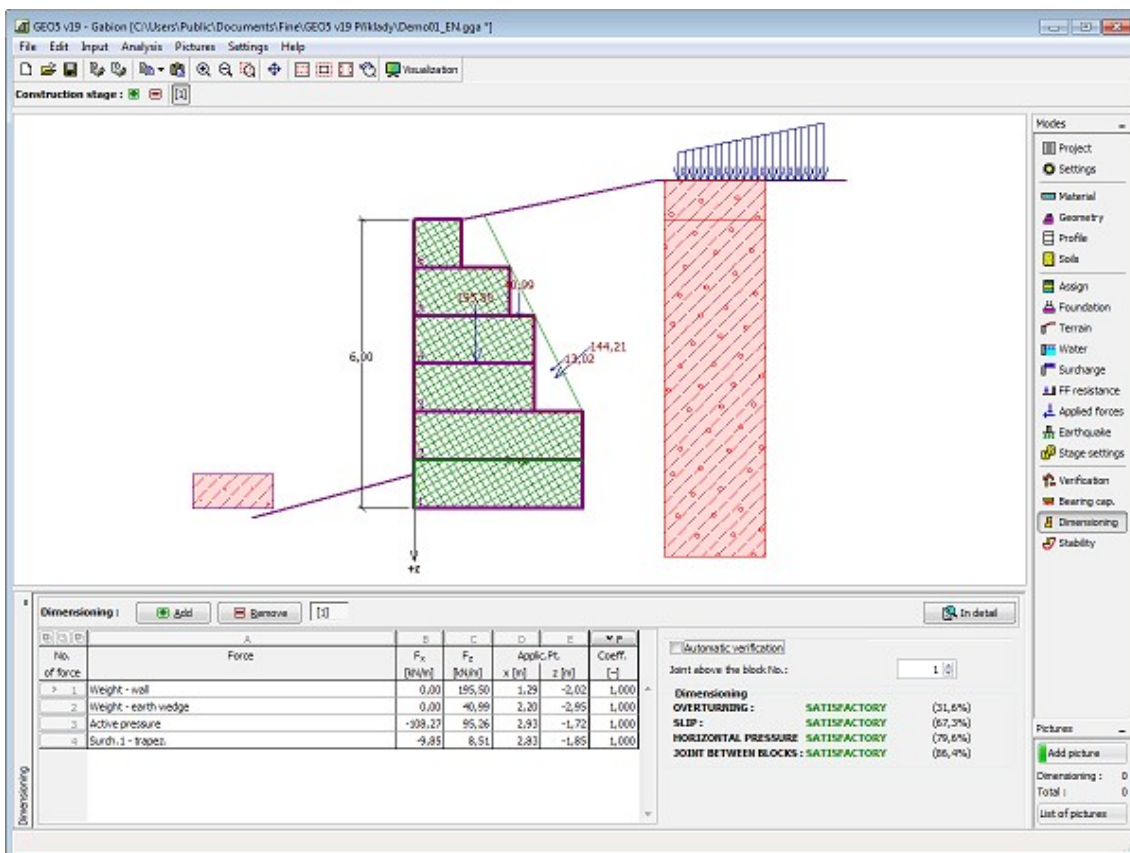
The "Dimensioning" frame allows to verify individual joints of gabion blocks. The "Joint above block No." field serves to select the desired joint subjected to verification analysis. The verification against overturning, translation, for side pressure and joint between blocks is performed.

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety, or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in the case the last column is not displayed.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Visualization settings".

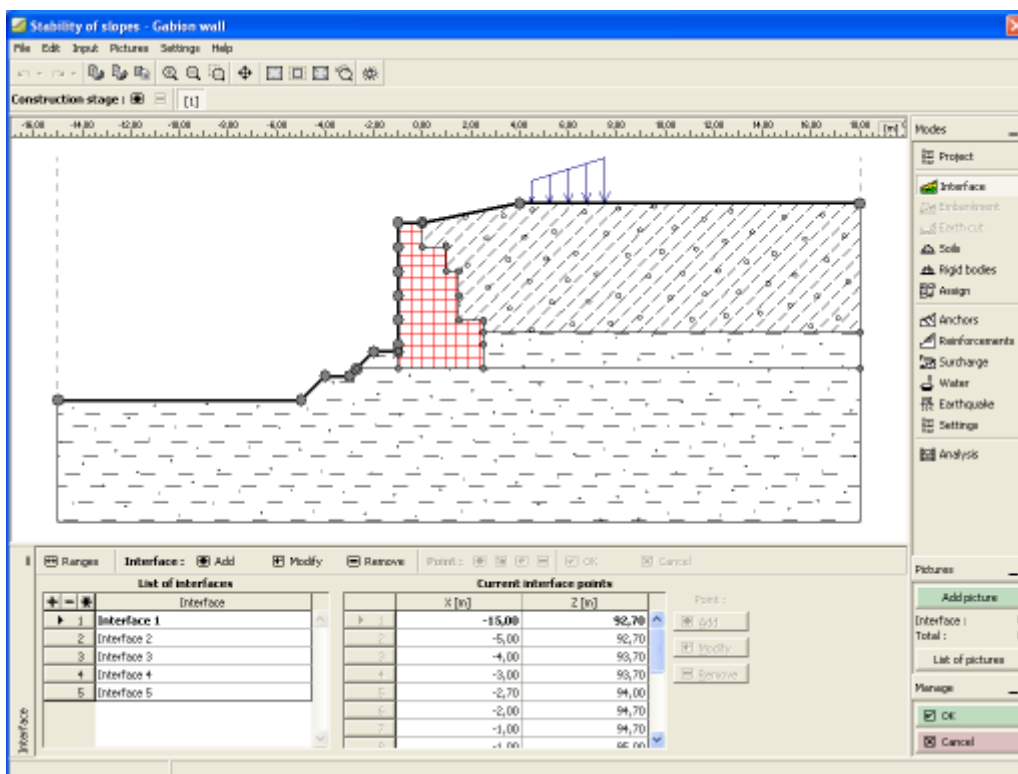


Frame "Dimensioning"

## Stability

Pressing the **"Stability"** button launches the **"Slope stability"** program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program **"Slope stability"** is installed.

After completing all analyses press the **"OK"** button to leave the program - all data are then carried over to the analysis protocol of the **"Gabion"** program.



Frame "Stability"

## Program Abutment

This program is used to design abutments including wing walls. It allows to check the abutment for overturning, translation, bearing capacity of foundation soil and dimensioning of decisive or reinforced concrete sections (including wings).

The help in the program **"Abutment"** includes the following topics:

- Input of data into individual frames:

Project	Settings	Geometric Section	Wings	Geometry Plane View	Footing Steps	Material
Profile	Soils	Load - LC	Assign	Foundation	Terrain	Water
Surcharge	Front Face Resistance	Applied Forces	Earthquake	Stage Settings	Verification	Bearing Capacity
Dimensioning	Stability					

- Standards and analysis methods
- Theory for analysis in the program "**Abutment**":
  - Stress in Soil Body
  - Earth Pressures
  - Analysis of Walls
  - Analysis of Foundation Bearing Capacity
  - Dimensioning of Concrete Structures
- Outputs
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but

also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows to choose an already created setting from the "**Settings list**".

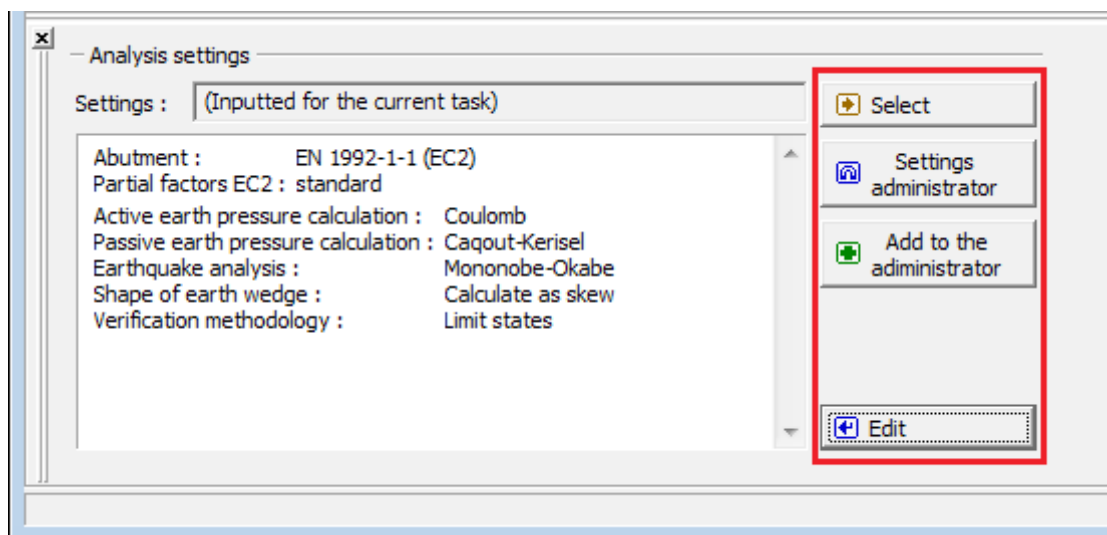
The "**Settings Administrator**" button opens the "**Administrator**" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Materials and standards**" and "**Wall analysis**" tabs.



Frame "Settings"

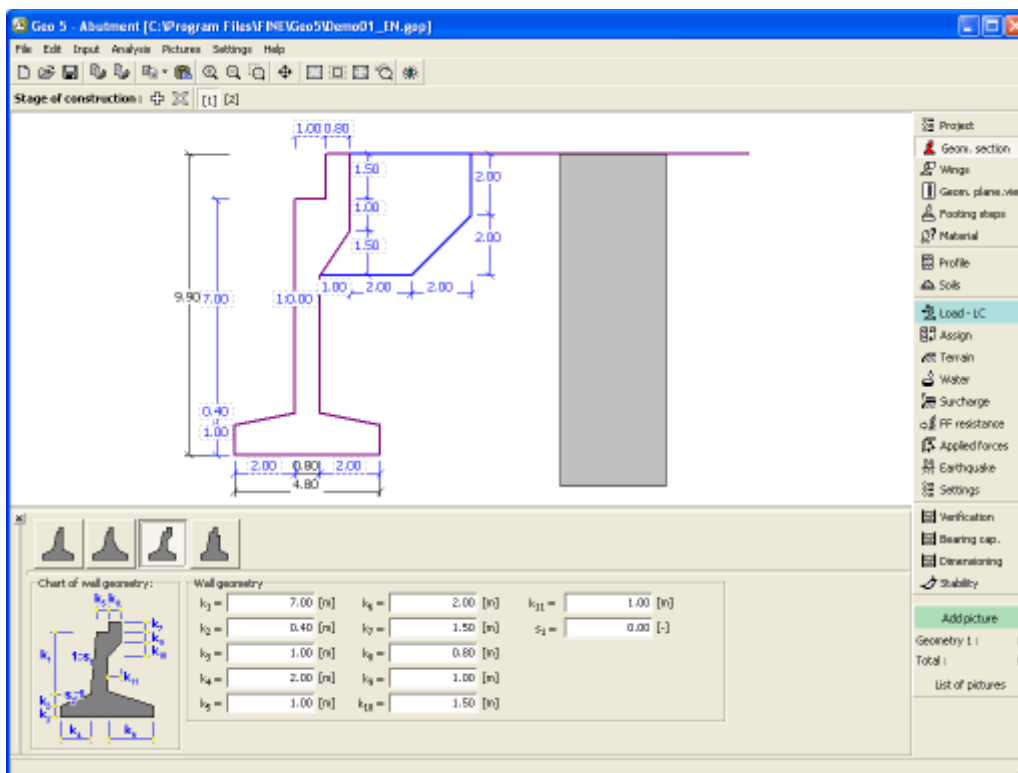
## Geometric Section

The **frame "Geometric section"** allows for selecting the shape of bridge abutment. The selected shape with a graphic hint "**Chart of wall geometry**" appears in the left part of the frame. The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

In case the structure is composed of inclined segments it is required to enter the ratio of sides of an inclined segment 1:*x*. **The straight structure** is specified by entering the value zero.

The frame serves to specify the final shape of abutment including the closure wall. The abutment can be verified also for the construction state (without the closure wall) based on the choice in the frame "**Load - LC**". The abutment length and the length of abutment foundation is specified in the frame "**Geometry plane view**".

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Geometric section"

## Wings

The frame "**Wings**" allows for inputting the bridge wings dimensions. The wings can be either symmetrical or unsymmetrical. Assuming unsymmetrical wings requires inputting the right and left wing dimensions separately. The screen always displays the currently input wing - only the left wing is then visualized in the remaining frames.

The frame "**Geometry plane view**" can also be used to input or edit the wing thicknesses and lengths.

The Wing-abutment joint cross-section can also be verified in the frame "**Dimensioning**". The load due to moment is considered. The whole wing is loaded by **active pressure** developed behind the wall. The "**Dimensioning**" dialog window serves to input the magnitude of **surface surcharge** to determine the wing pressure. The surcharge specified in the frame "**Surcharge**" is then not taken into account and the terrain behind the wing is considered as flat. The resulting moment applied to the joint is obtained by multiplying the overall magnitude of soil pressure acting on the wall surface and by the difference of centroids of the pressure resultant and the joint.

The length of cross-section used for dimensioning is considered by default as the wing height - a different length of wing-abutment joint can also be specified after selecting the option "**Reduce for dimensioning**".

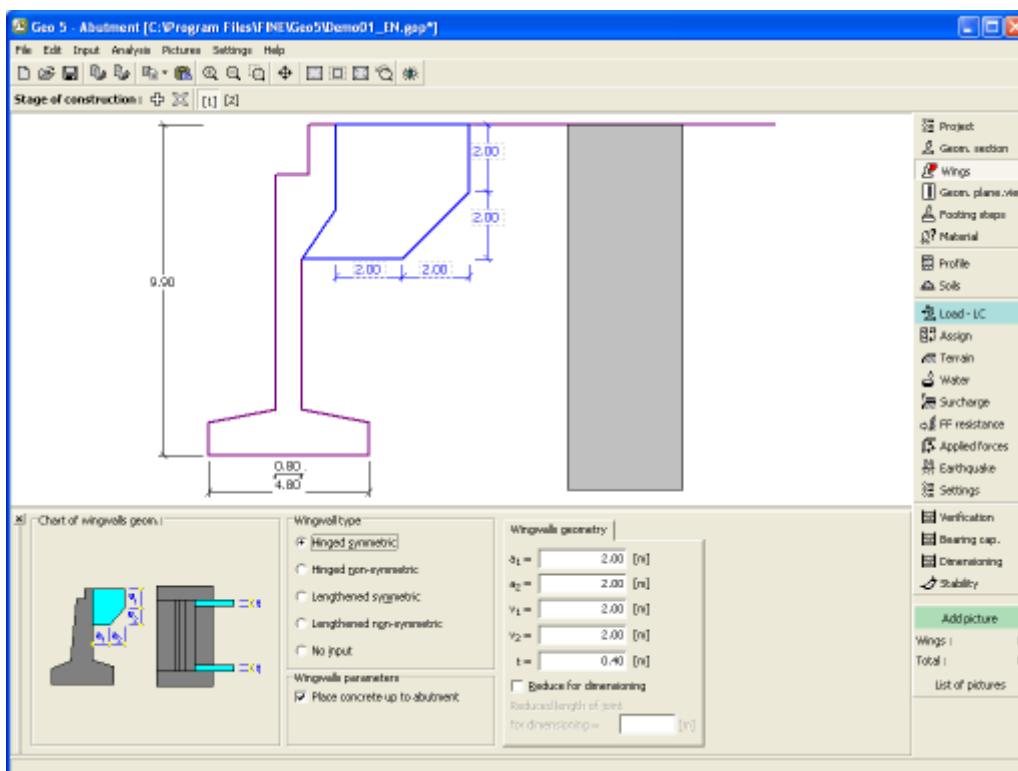
When using prolonged wing walls it is possible to input dimensions of the foundation below the wall. Such foundation jumps are reflected in the analysis by computing a fictitious width of the foundation as:



$$d_{fict} = \frac{A_{tot}}{S}$$

where:  $A_{tot}$  overall area of foundation including all jumps  
 $S$  length of abutment foundation  
 $d_{fict}$  fictitious width of foundation for verification analysis

The foundation is then considered as being rectangular, which is simplified but rather conservative assumption.



Frame "Wings"

## Geometry Plane View

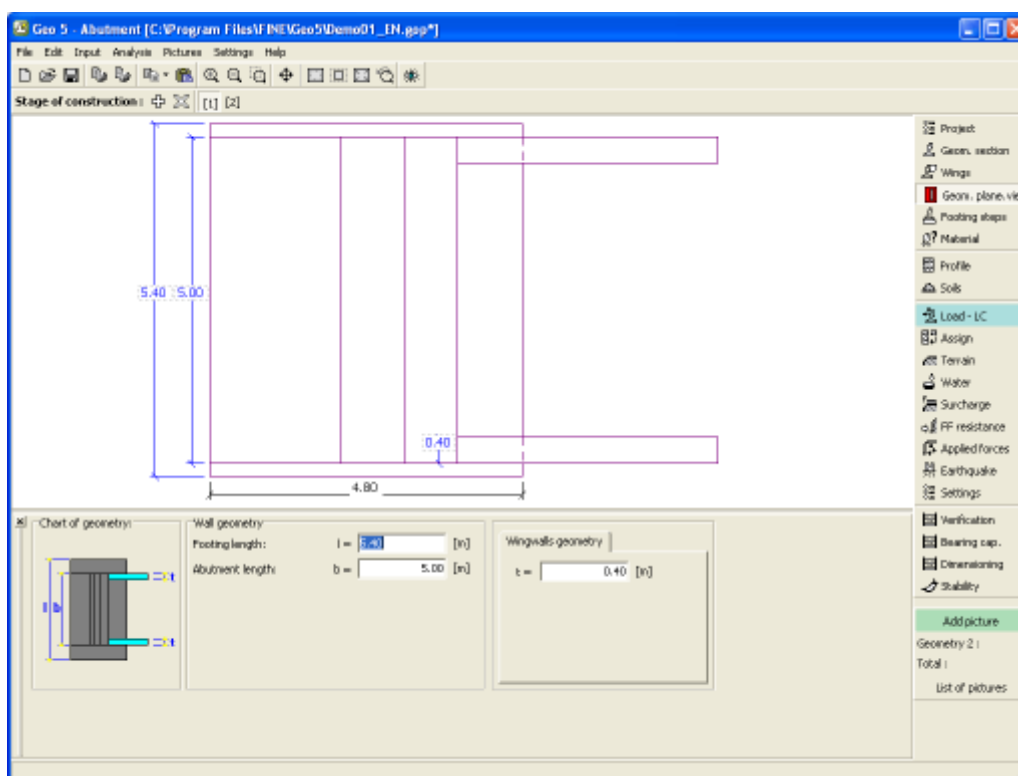
The frame "Geometry plane view" allows for inputting the abutment length, length of abutment foundation and also dimensions of abutment wings.

Dimensions can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

For details on **the effect of abutment dimensions** on verification analysis we refer the reader to section "Calculating of abutment forces".

The program makes it possible to export the geometry of a structure in the \*.DXF format.



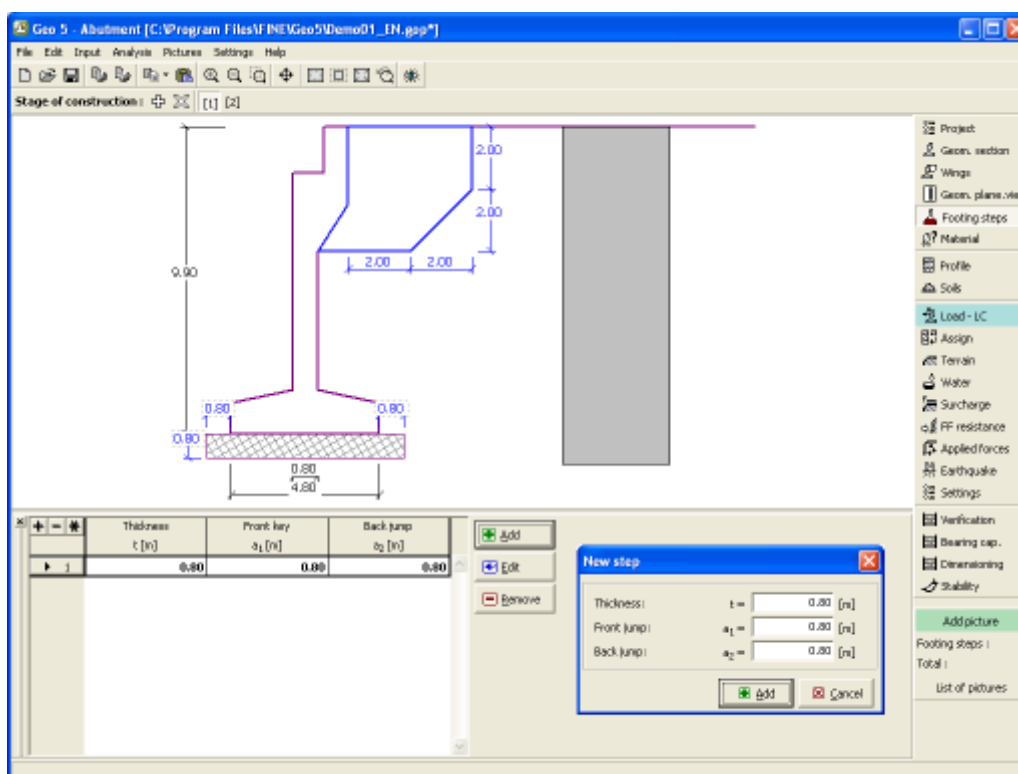


Frame "Geometry plane view"

## Footing Steps

The frame "Footing steps" serves to input the steps of foundation below abutment. This option thus allows for specifying additional shapes of bridge abutment.

Adding (editing) foundation step is performed in the "New step" dialog window. Input footing steps can be edited on the desktop with the help of active dimensions or active objects, respectively.



Frame "Footing steps"

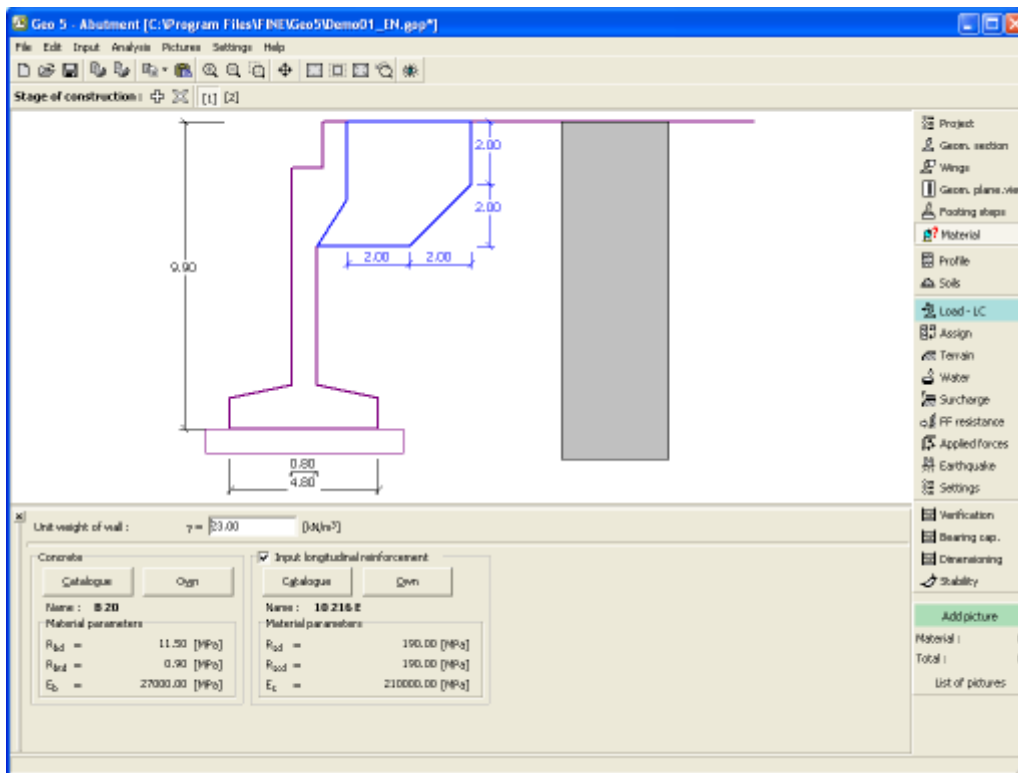
## Material

The "**Material**" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- The "**Catalog**" button opens the "**Material catalog**" dialog window (for concrete or steel reinforcements), the list of materials then serves to select the desired material.
- The "**Own**" button opens the "**Concrete**" dialog window (for concrete) or the "**Reinforcing steel bars**" dialog window (for longitudinal steel reinforcements), which allows for manual specification of material parameters.

The catalogs content depends on the selection of standard for the design of concrete structures set in the "**Materials and standards**" tab. The input field in the upper part of the frame serves to specify the **abutment unit weight**.



Frame "Material"

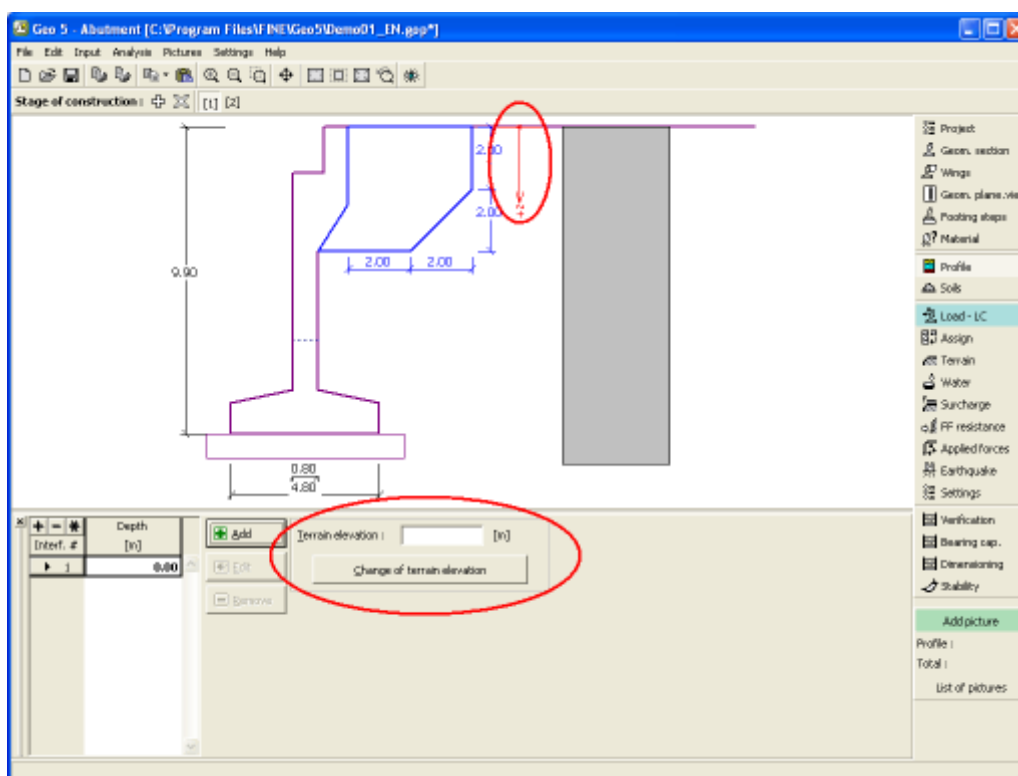
## Profile

The "Profile" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "New interface" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "Change of terrain elevation" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".

The program makes it possible to import a profile in the gINT format.



Frame "Profile"

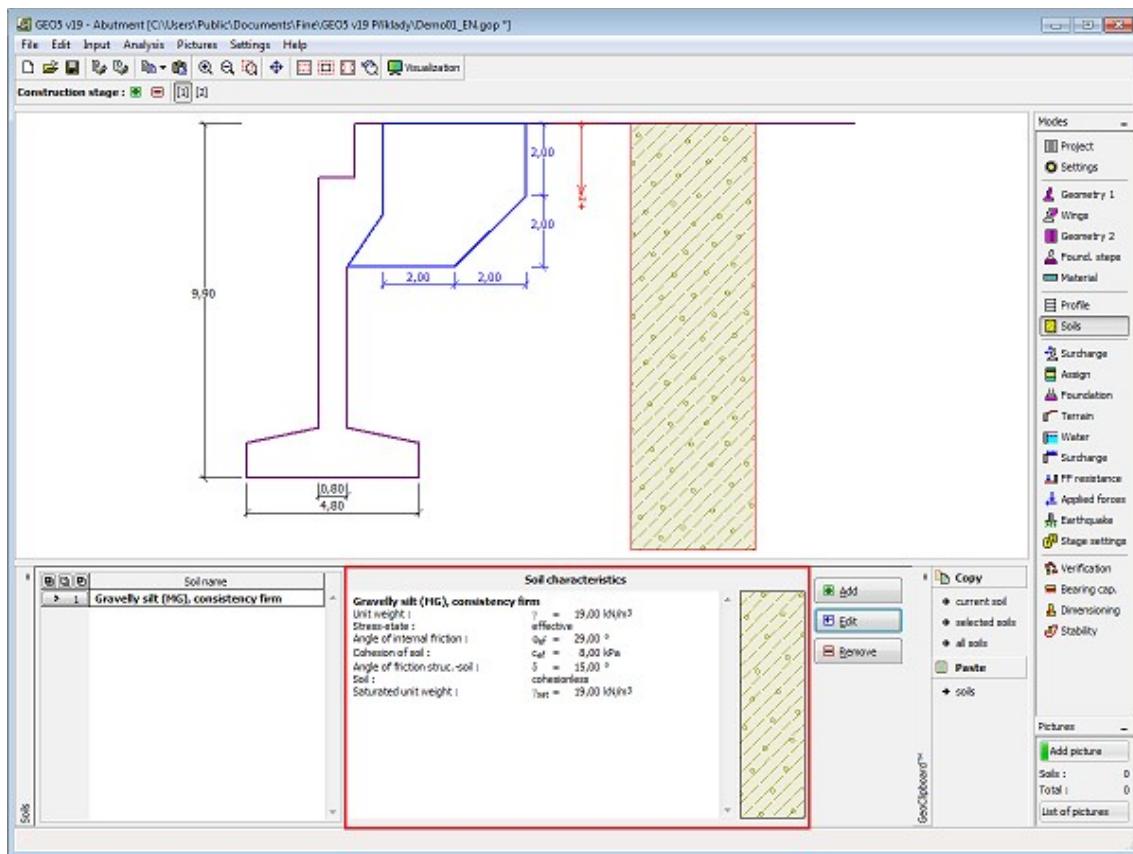
## Soils

The **"Soils"** frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The program makes it possible import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil, type of load, structure duration and water conditions.

For **effective stress** further needs to specify the **angle of internal friction between the soil and structure**, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the [table of recommended values](#).

For **total stress** further needs to specify the **adhesion of soil to the structure face**  $a$ .

The associated theory is described in detail in chapter "**Earth pressures**".

Dialog window "Add new soils" - "Basic data"

## Load - LC

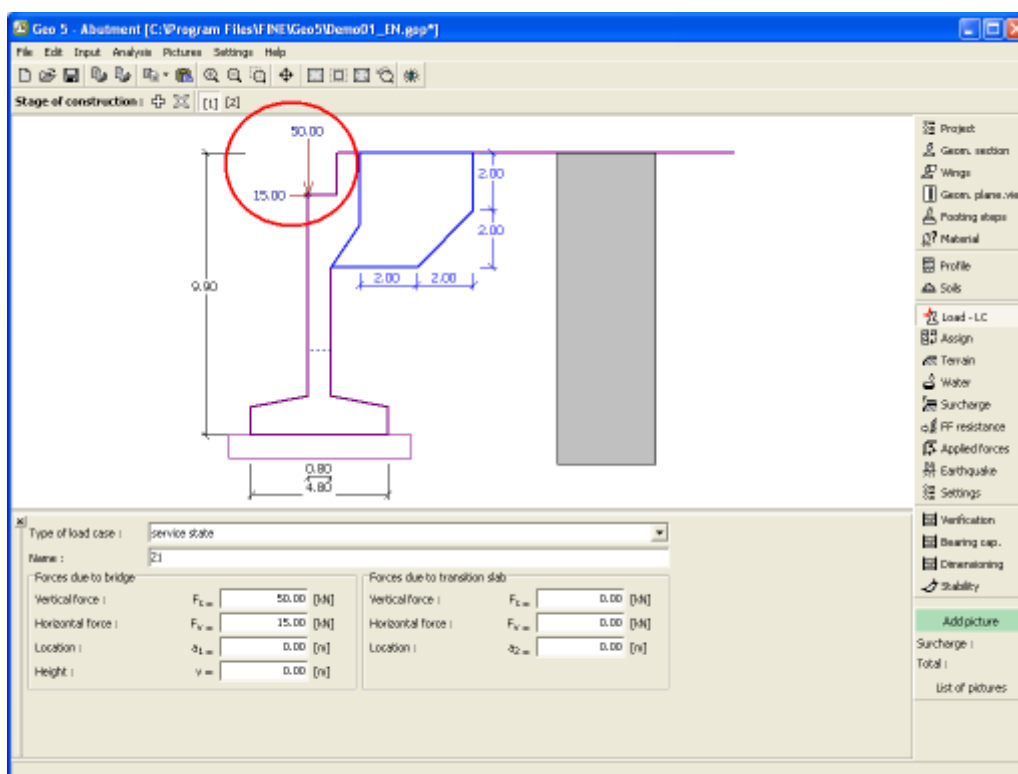
The frame "Load - LC" serves to specify individual **load cases** (construction, service) and the load caused by the bridge and transition slab. Verification and dimensioning analyses of the whole bridge abutment or only its part are carried out according to the specified type of LC.

When performing the analysis according to EN 1997 or LRFD, the input load due to **bridge and transition slab is not INCREASED** by any **partial factors**. The input forces must be determined **in accordance** with the corresponding standards (EN 1990, EN 1991).

No load specified in the case of **construction state** and the abutment is verified in a given stage of construction without a closure wall and bridge wings.

In the case of **service state** the abutment is loaded by the **bridge and transition slab**, the whole abutment is verified.

For abutment verification it appears advantageous to exploit the **stage of construction** and specify in individual stages different load cases (e.g. construction state, service state without live load, service state with all loads). Individual stages then allow inputting different loads, surcharges, terrain shapes, type of pressure analysis (active, at rest), design coefficients, etc.



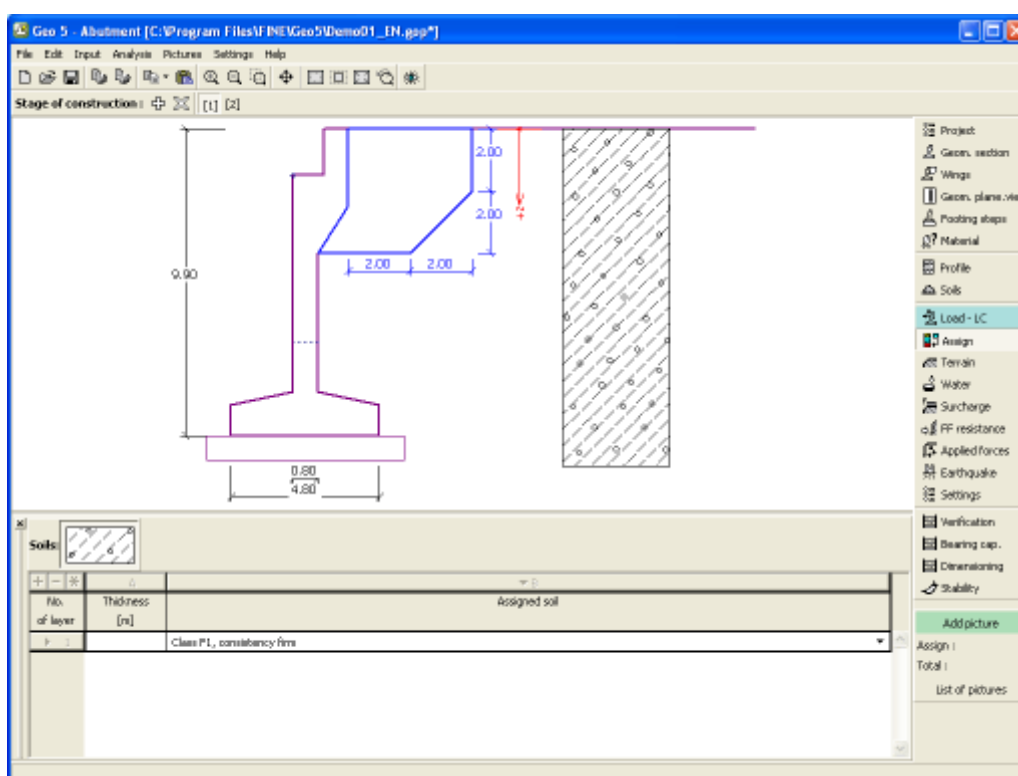
Frame "Load - LC"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

## Foundation

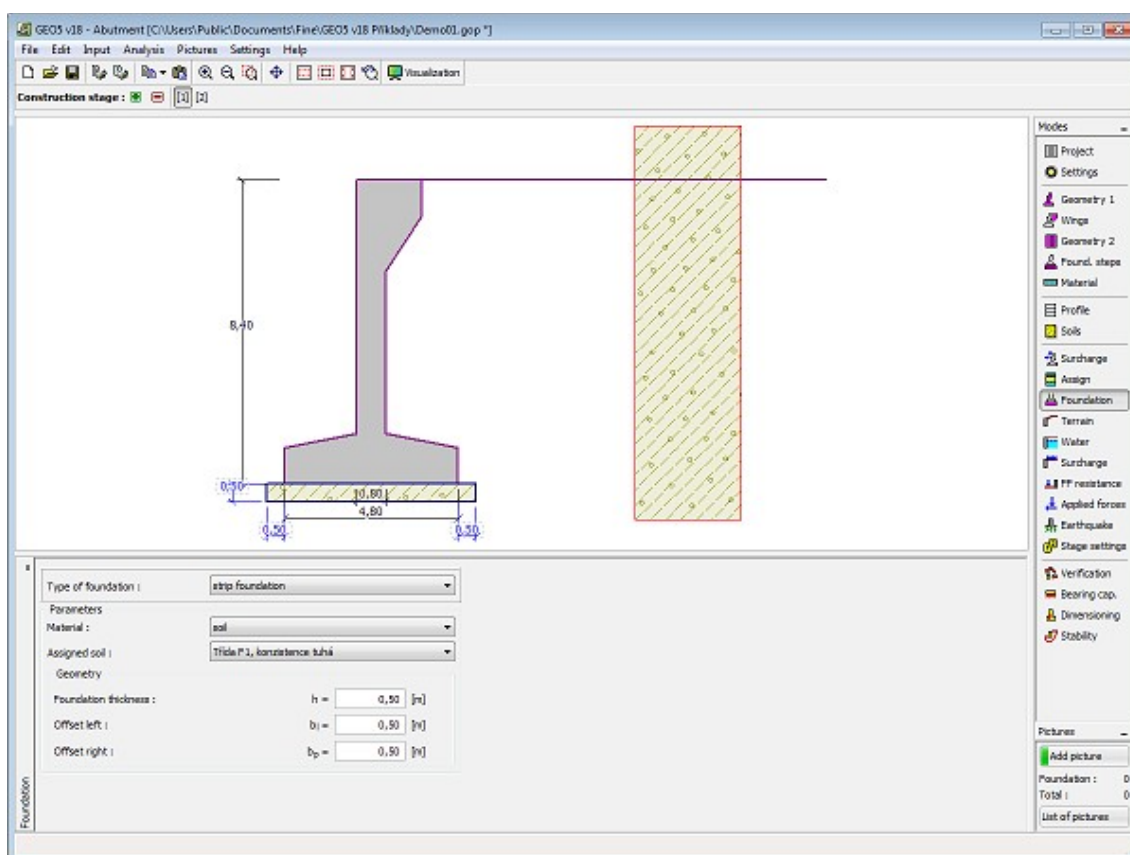
The "**Foundation**" frame serves to specify the type of wall foundation. The following types of wall foundation are available:

- **soil from geological profile** - the wall is founded on the soil assigned from the geological profile specified in the "Profile" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and structure are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle  $\psi$  [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient  $\mu$  [-]. Both options require inputting the cohesion  $a$  [kPa] between foundation (base) and soil.
- **strip foundation** - strip foundation material is represented either by **soil** (input in "Soils" frame), or concrete - requires inputting the **unit weight of foundation material**  $\gamma$  and **parameters of contact base-soil** (friction coefficient  $f$ , cohesion  $c$ , additional resistance  $F$ ).
- **pile foundation** - the wall can be founded on one row of piles or two rows of piles, respectively.

**Strip foundation** and **pile foundation** can be adopted for the wall foundation only if the type wall with **straight footing bottom without jump** is selected in the "Geometry" frame. The geometry of wall foundation (**strip foundation**, **pile foundation**) can be modified either in the frame by entering specific values into the inputting fields or on the desktop with the help of **active dimensions**.

The input data introduced in this frame influence the actual **wall analysis** (check for slip) and further the **bearing capacity of foundation soil**.





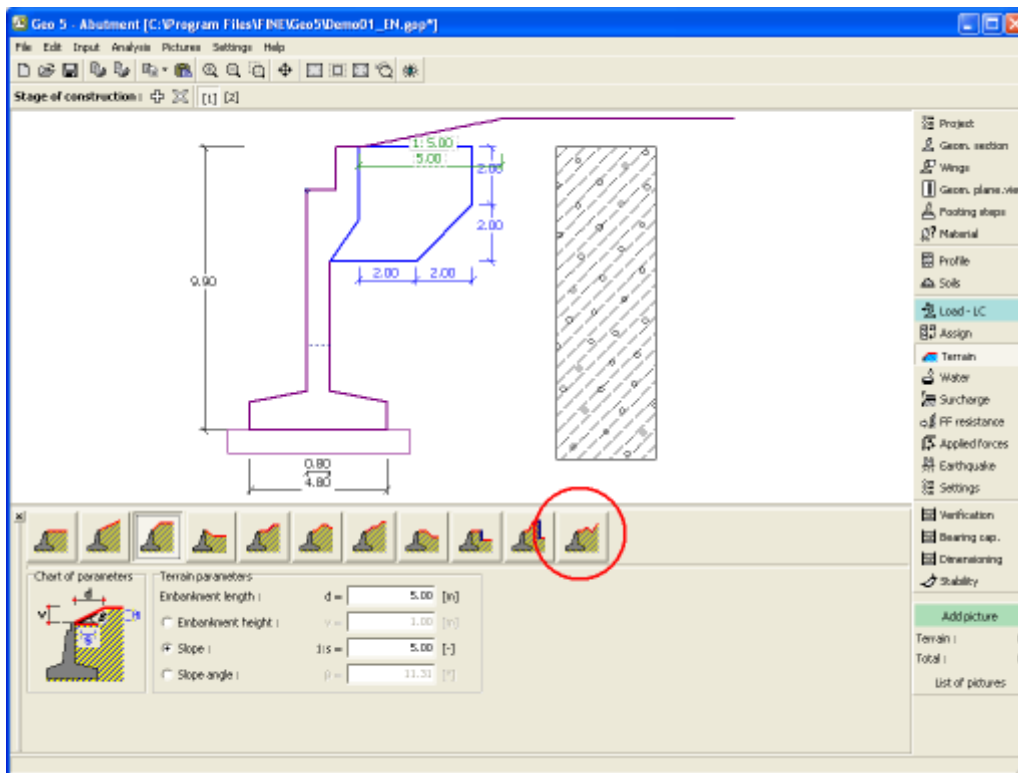
Frame "Foundation"

## Terrain

The "**Terrain**" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

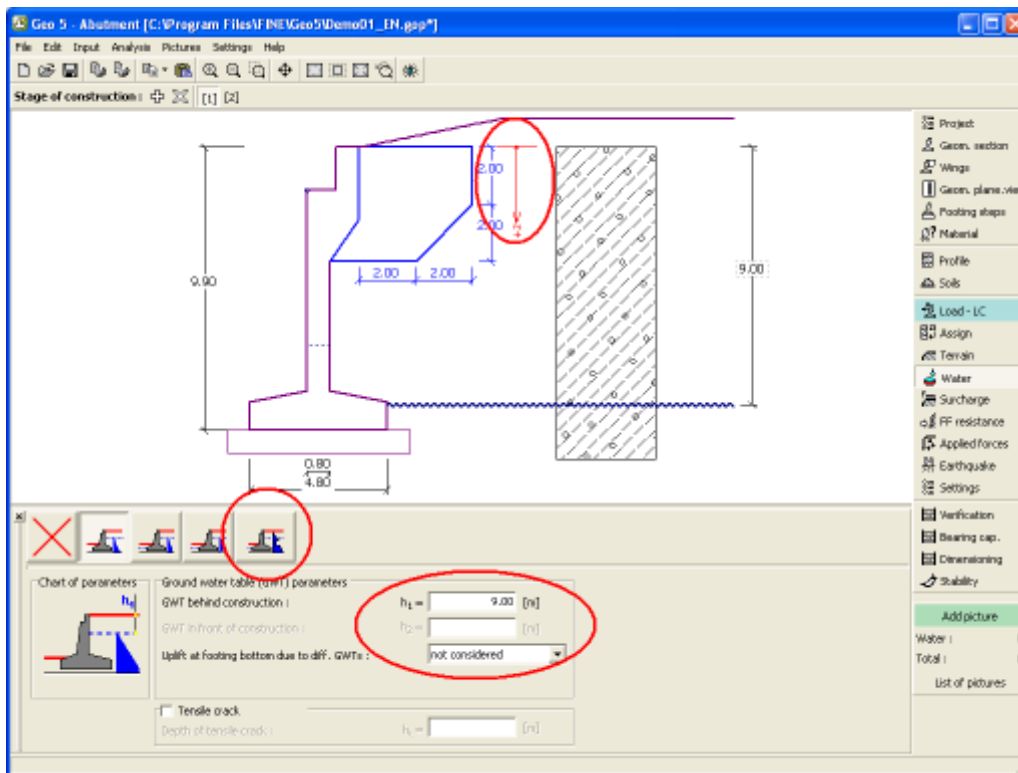
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be **linear, parabolic** or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".

The program further allows for specifying a depth of **tensile cracks** filled with water.



Frame "Water"

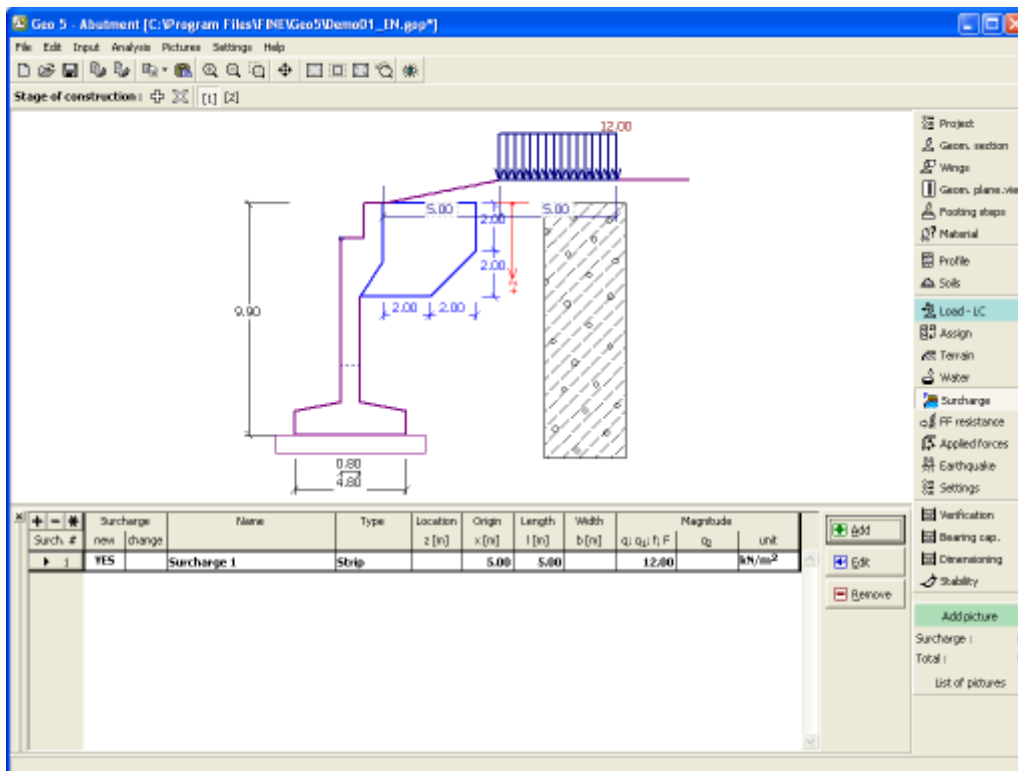
## Surcharge

The "Surcharge" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "New surcharge" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either permanent, variable or accidental surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "Influence of surcharge".



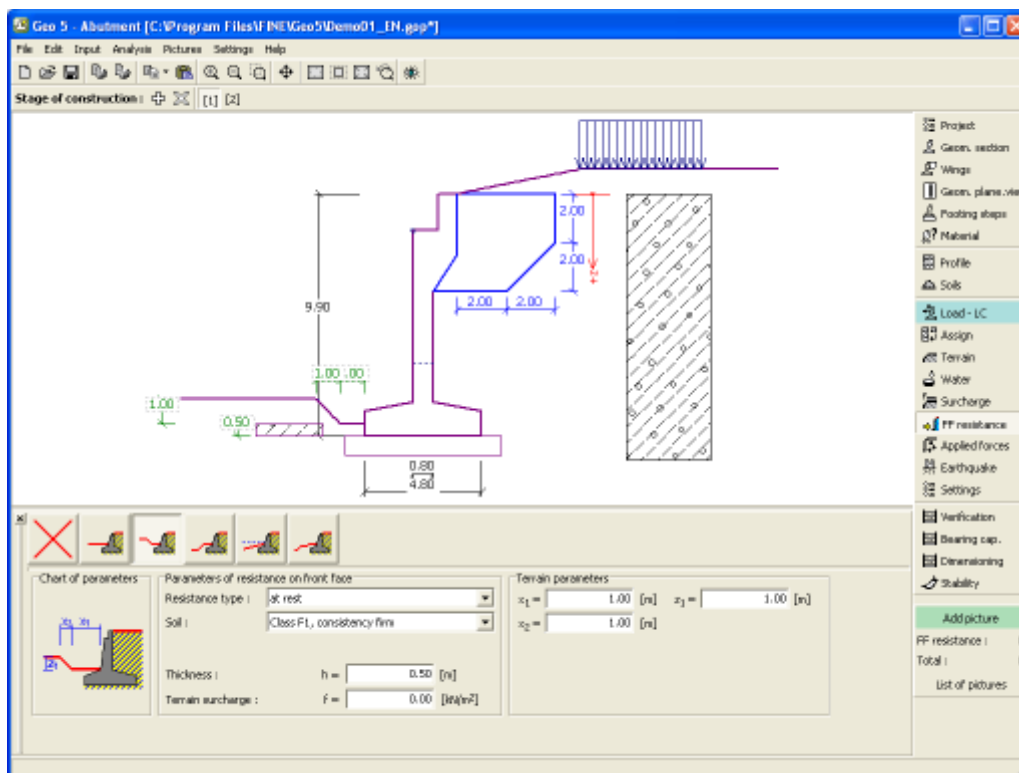
Frame "Surcharge"

## Front Face Resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the frame "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive earth pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the input type of reduced passive pressure.



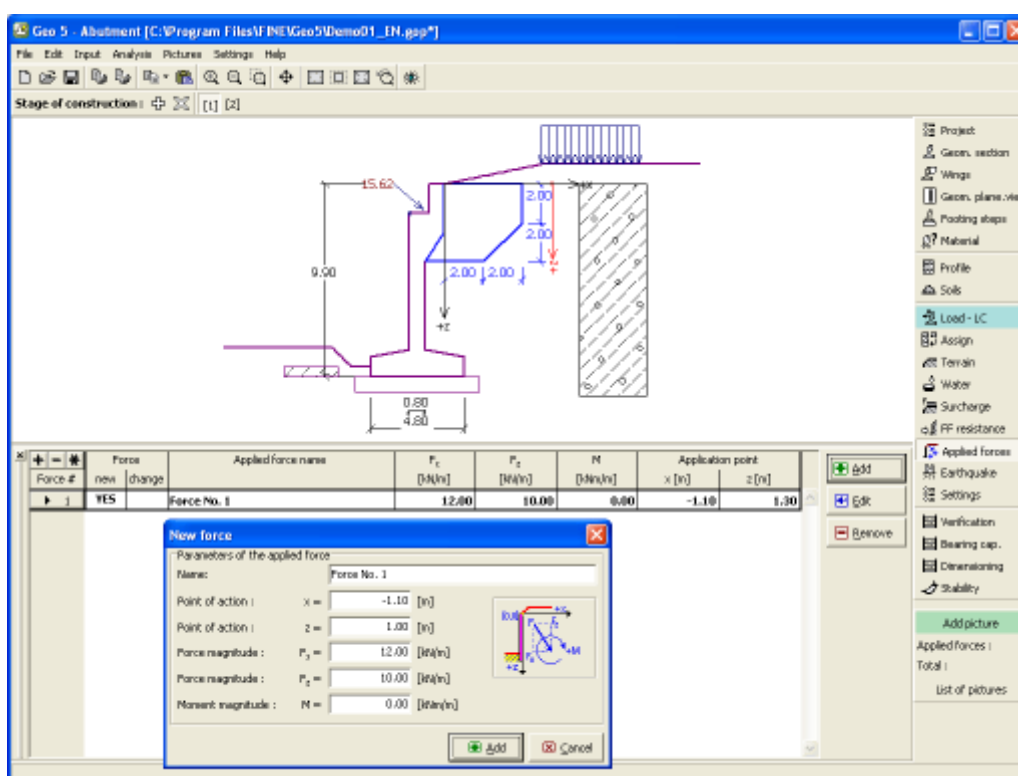
Frame "Front face resistance"

## Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

**Applied forces** represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.



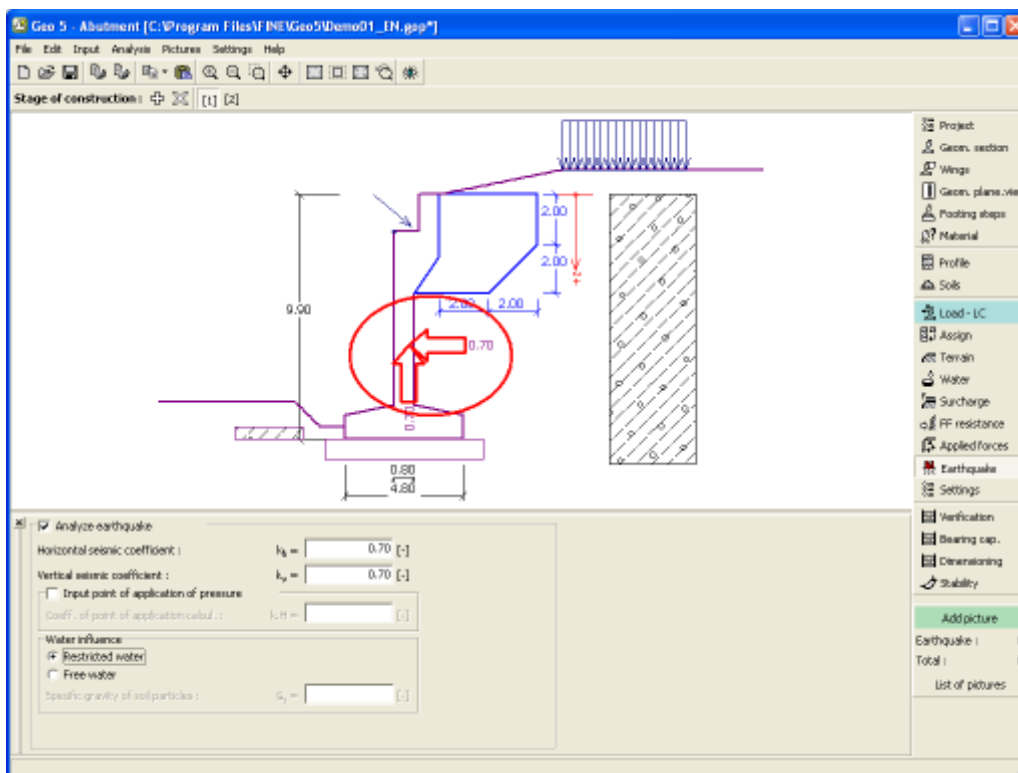
Frame "Applied forces"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



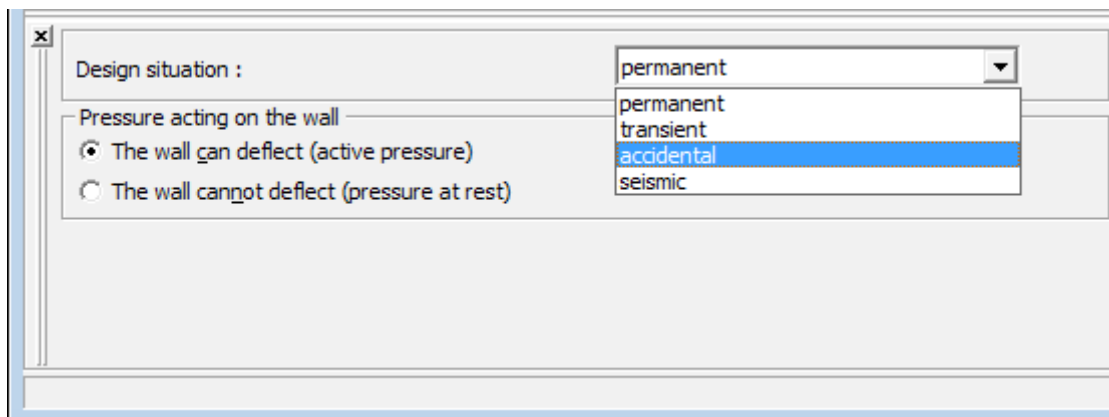
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage. Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).

Next, the frame serves to specify the type of pressure acting on a wall based on the allowable wall deformation. Providing the wall is free to move, an active pressure is assumed, otherwise, a pressure at rest is used.



Frame "Stage settings"

## Verification

The frame "Verification" shows the analysis results. Several computations can be carried out for a single task.

The frame appearance is adjusted based on the selected verification methodology.

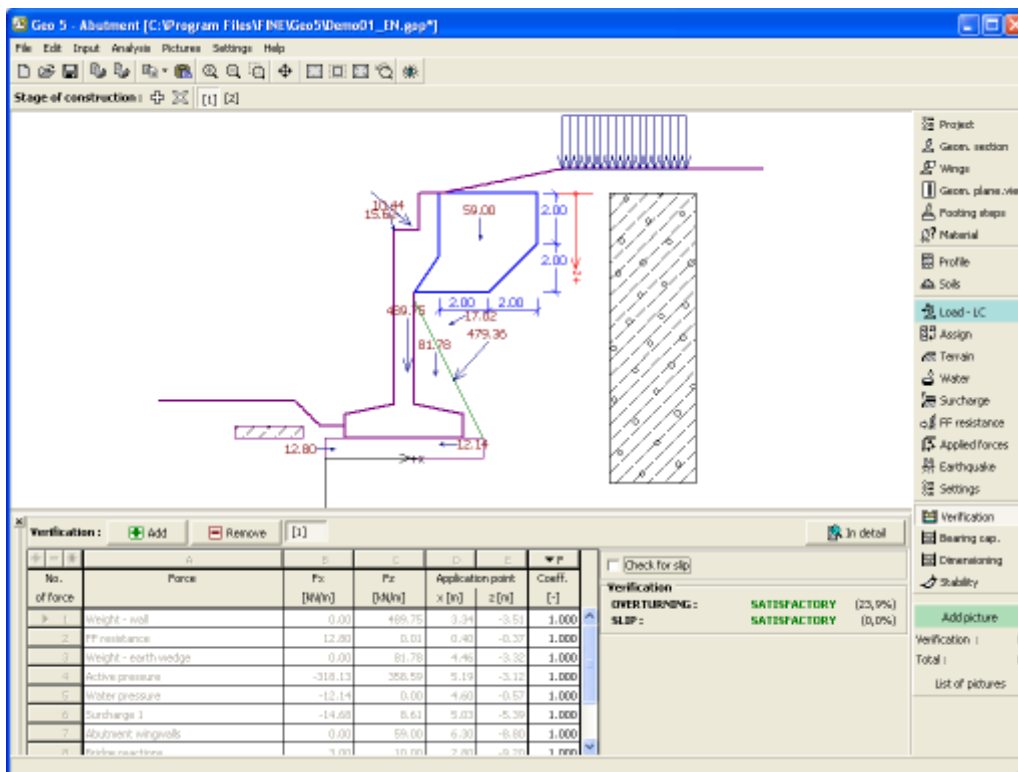
- Verification according to the factor of safety, or the theory of limit states - the last column **F** in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations".
- Analysis according to LRFD - in the case the last column is not displayed.

The wall is loaded either by active pressure or pressure at rest depending on input in the frame "Stage settings".

The procedure for wall verification is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against overturning and translation. The "In detail" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Verification"



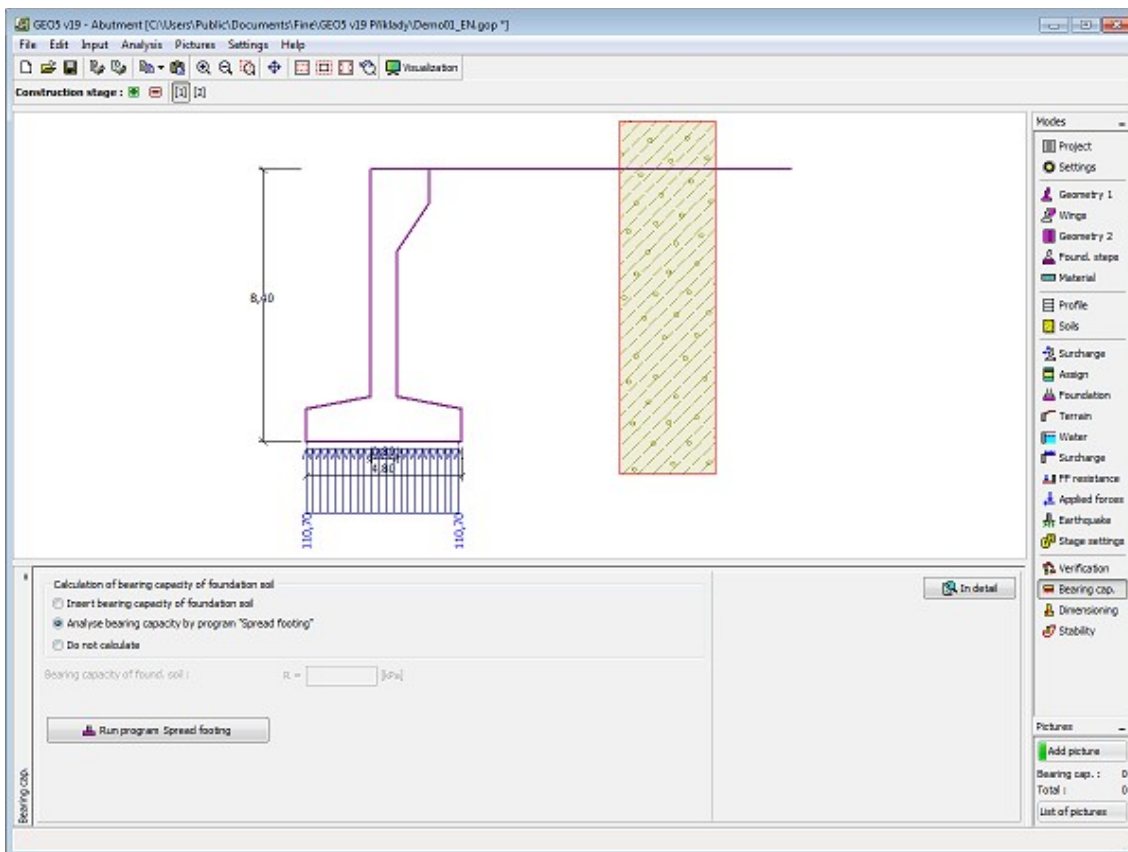
## Bearing Capacity

The "**Bearing capacity**" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "**Verification**". The program "**Spread footing**" then considers all verifications as load cases.

The frame contains following analysis options:

- **Insert bearing capacity of foundation soil** The foundation soil bearing capacity is input. The **eccentricity** and **bearing capacity** analysis results are displayed in the right part of the frame. The "**In detail**" button opens a dialog window that displays detailed listing of the results.
- **Analyse bearing capacity by program "Spread footing"** Pressing the "**Run program Spread footing**" button opens the program "**Spread footing**" which allows to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "**OK**" button leaves the analysis mode - the results and all plots are transferred into the program "**Abutment**". The "**Spread footing**" program must be installed for the button to be active. Overall length of wall foundation is put in.
- **Analyse bearing capacity by program "Pile"** The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile**" is available if the wall has pile foundation (frame "**Foundation**"). Pile spacing  $s$  is input.
- **Analyse bearing capacity by program "Pile group"** The procedure is identical as if calculating soil bearing capacity by the "**Spread footing**" program. The "**Run program Pile group**" is available if the wall has pile foundation with more than one pile (frame "**Foundation**"). Pile spacing  $s$ , overall number of pile rows  $n$  and loading length  $l$  are input.
- **Do not calculate (pile footing)** The foundation soil bearing capacity is not calculated.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Bearing capacity"

## Dimensioning

The "Dimensioning" frame serves to design and verify the reinforcement of abutment cross-section - the cross-section subjected to dimensioning is selected in the combo list. The table shows the abutment forces.

Offer of cross-sections that can be verified depends on the selected load case (construction, service). The following cross-sections are available for both the construction and service state:

- **Wall stem verification**
- **Construction joint verification**      **depth** of construction joint from construction top edge is specified
- **Wall jump verification**

The service state makes also possible to verify:

- **Verification of closure wall**
- **Verification wing - abutment**      the surface surcharge due to terrain is input, for actual analysis we refer to section "Wings"

The frame appearance is adjusted based on the selected verification methodology.

- Verification according to the factor of safety, or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of

data and setting in the frame.

- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations".
- **Analysis according to LRFD** - in the case the last column is not displayed.

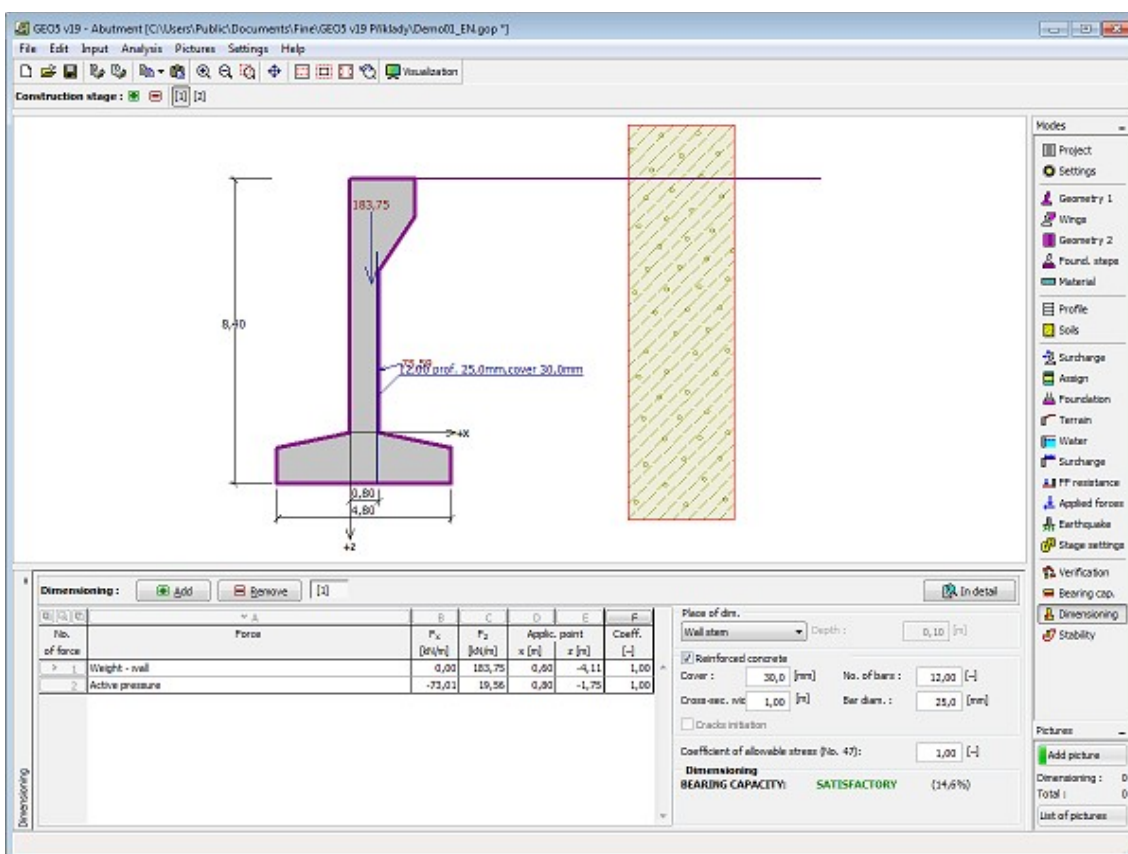
The abutment is loaded either by **active pressure** or **pressure at rest** depending on the input specified in the frame "Stage settings", an **active earth pressure** is used when analyzing wing walls.

The procedure to derive distribution of internal forces in individual cross-sections is described in the theoretical part of the help.

Dimensioning of the reinforced concrete is performed according to the standard set in the "Materials and standards" tab. Verification analysis based on the standard **CSN 73 6206 "Design of concrete and steel reinforced concrete bridge structures"** is described herein.

Several computations for various cross-sections can be carried out. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The **"In detail"** button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "Visualization settings".

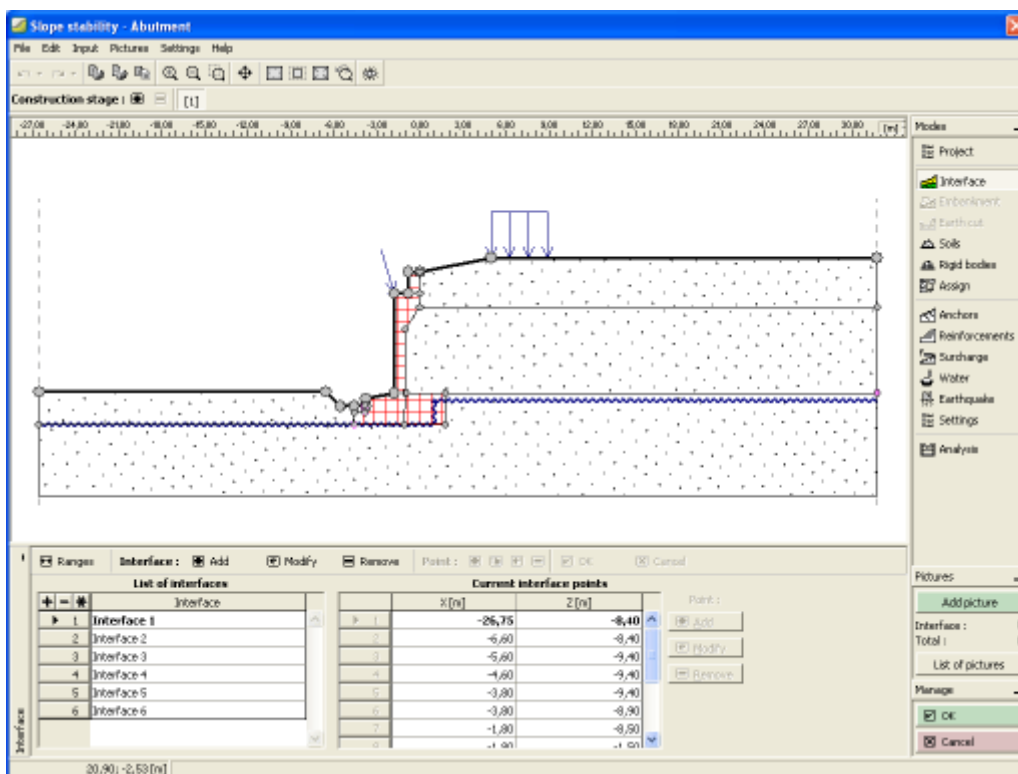


Frame "Dimensioning"

# Stability

Pressing the **"Stability"** button launches the **"Slope stability"** program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program **"Slope stability"** is installed.

After completing all analyses press the **"OK"** button to leave the program - all data are then carried over to the analysis protocol of the **"Abutment"** program.



Frame "Stability"

## Program Nailed Slope

The program analyses nailed walls and slopes of various shapes.

The help in the program "Nailed Slope" includes the following topics:

- Input of data into individual frames:

Project	Settings	Geometry	Types of Nails	Geometry of Material Nails	Profile
Soils	Assign	Terrain	Water	Surcharge	Earthquake
Internal Stability	Verification	Bearing Capacity	Dimensioning	External Stability	Stage Settings

- Standards and analysis methods
- Theory for analysis in the program "**Nailed Slope**":
  - Stress in Soil Body
  - Earth Pressures
  - Analysis of Walls
  - Nailed Slope
  - Dimensioning of Concrete Structures
- Outputs
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but

also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows to choose an already created setting from the "**Settings list**".

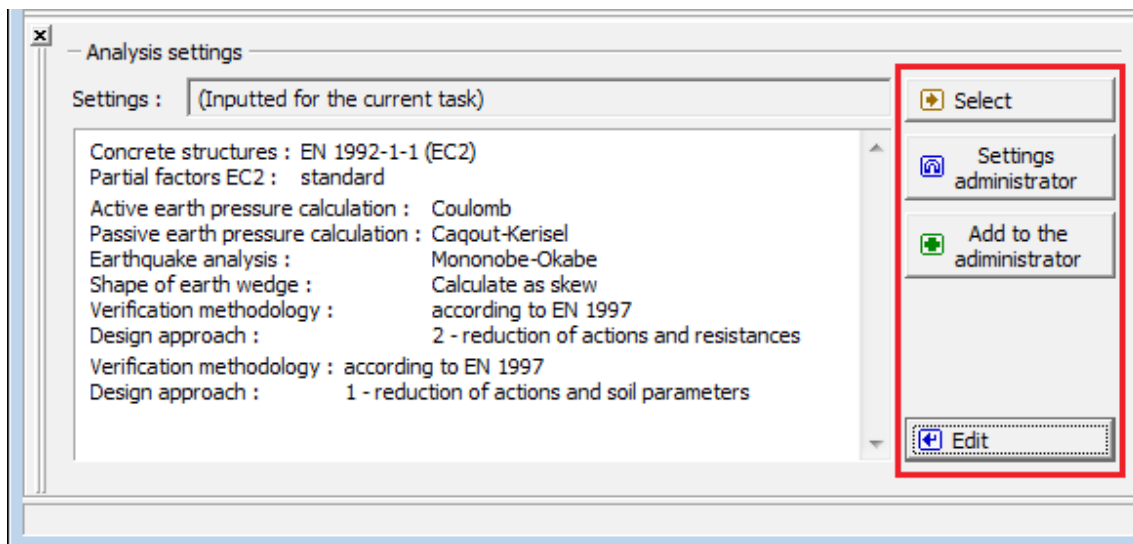
The "**Settings Administrator**" button opens the "**Administrator**" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Materials and standards**", "**Wall analysis**" and "**Stability analysis**" tabs.



*Frame "Settings"*

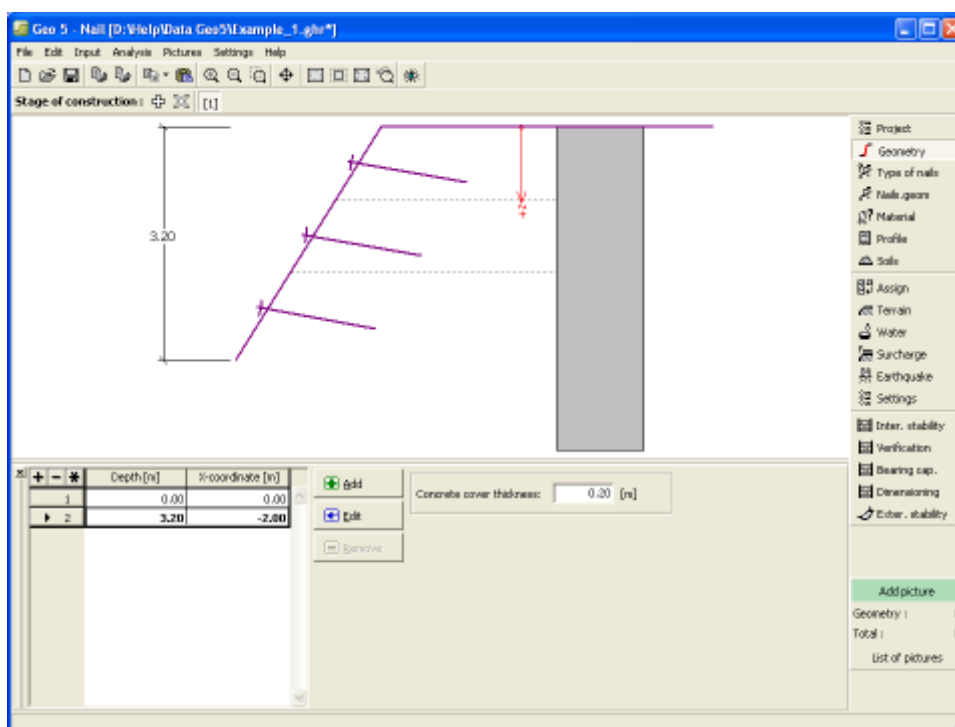
## Geometry

The **frame "Geometry"** contains a **table** with a list of input points of the structure front face. **Adding** (editing) points is performed in the "**New point**" dialog window.

The input points can also be edited on the desktop with the help of **active objects** - double-clicking an already input point then opens a dialog window for its editing.

One needs to specify depth ( $z$ -coordinate from the top point of a structure - positive direction is assumed downwards) and an  $x$ -coordinate (negative direction is assumed to the left, no overhang of a structure is allowed).

The program makes it possible to **export** the structure geometry in the \*.DXF format.

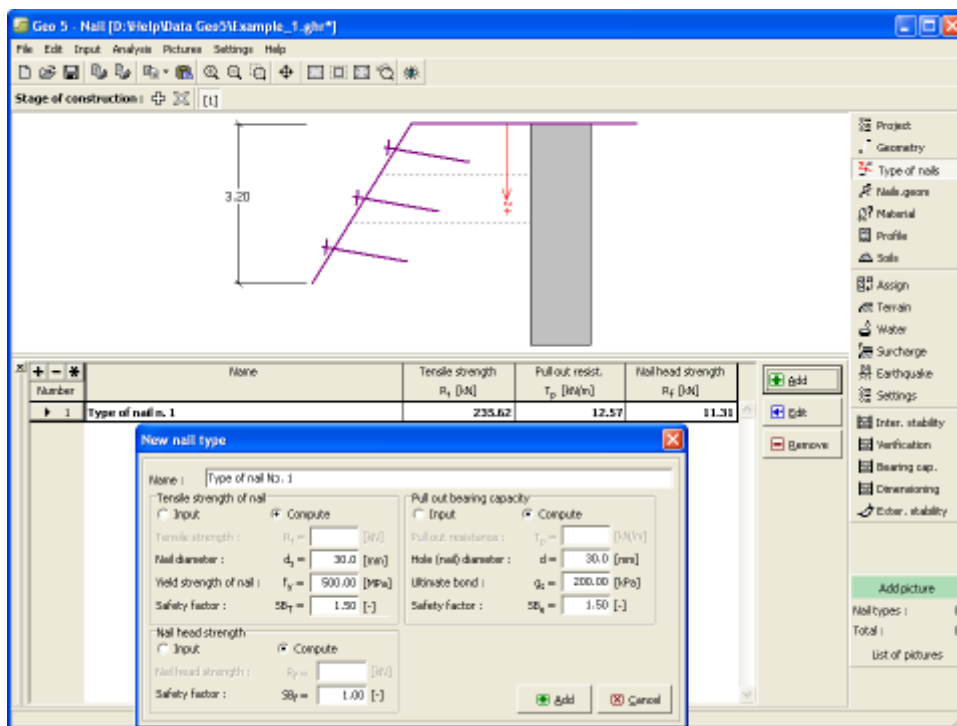


Frame "Geometry"

## Types of Nails

The frame "Types of nails" serves to specify a nail type in a specific table. The strength parameters of nails can be either **input** by the user or directly **determined by the program** depending on the input data.

The table lists the following input data; either input or computed - **nail head strength, nail tensile strength** and its **pull - out resistance** per 1 m (1 ft).



Frame "Types of nails"

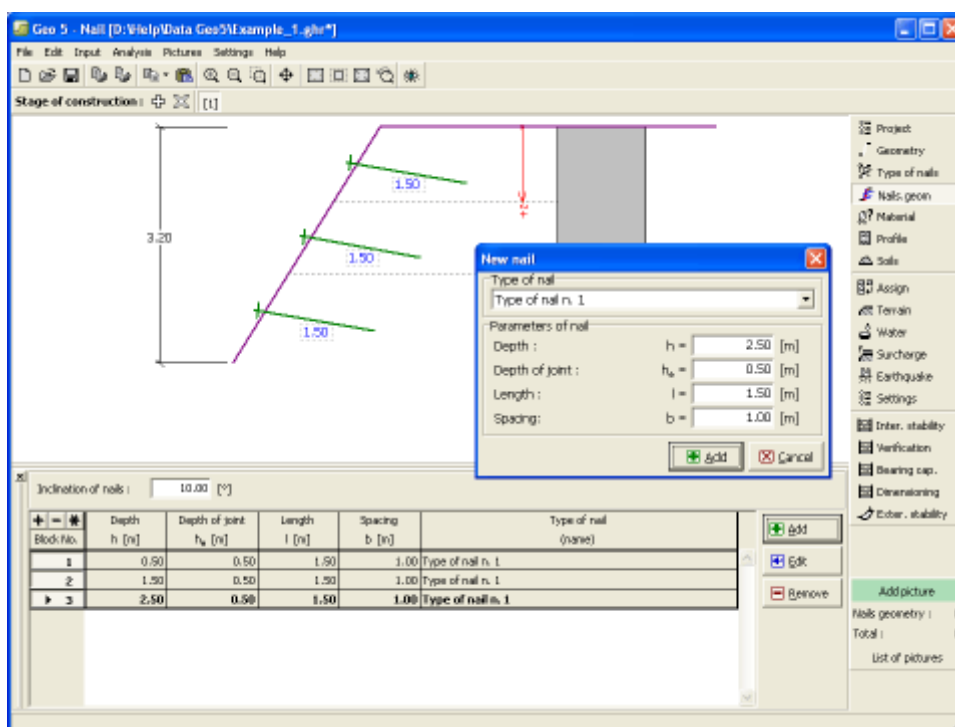
## Geometry of Nails

The frame "Geometry of nails" contains a table with a list of input nails. Adding (editing) nails is performed in the "New nail (Edit nail)" dialog window. The input nails can also be edited on the desktop with the help of active objects.

The user is required to specify the nail depth, **depth of a bench from a given nail** (the next nail must be introduced as deep as to be located below the bench of the upper nail), nail length, its diameter and distance.

Inclination of nails is regarded from the horizontal line in clockwise direction and is constant for all nails.





Frame "Geometry of nails"

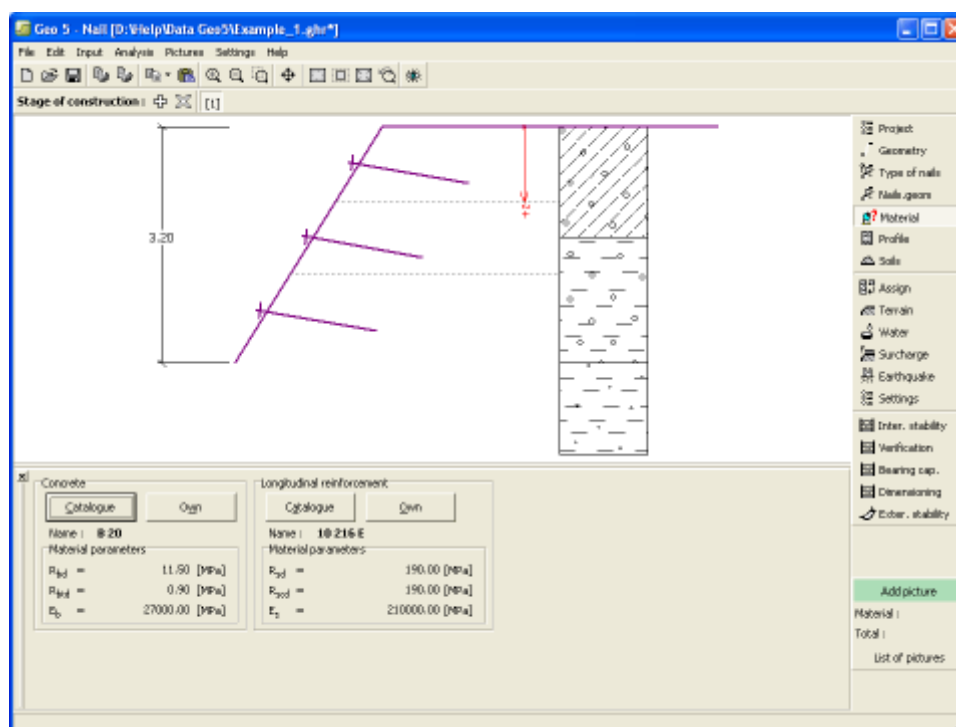
## Material

The "Material" frame allows for the selection of material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- The "Catalog" button opens the "Material catalog" dialog window (for concrete or steel reinforcements), the list of materials then serves to select the desired material.
- The "Own" button opens the "Concrete" dialog window (for concrete) or the "Reinforcing steel bars" dialog window (for longitudinal steel reinforcements), which allows for manual specification of material parameters.

The catalogs content depends on the selection of standard for the design of concrete structures set in the "Materials and standards" tab.



Frame "Material"

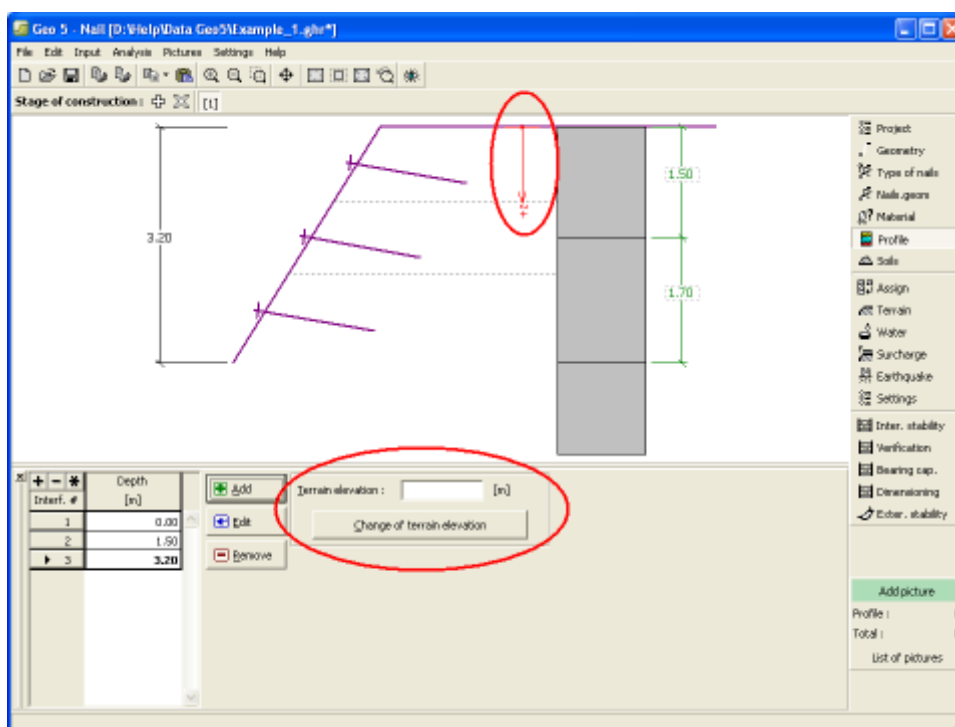
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "**New interface**" dialog window. The  $z$ -coordinate measured from the top point of a structure is specified ( $z$ -axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to import a profile in the gINT format.



Frame "Profile"

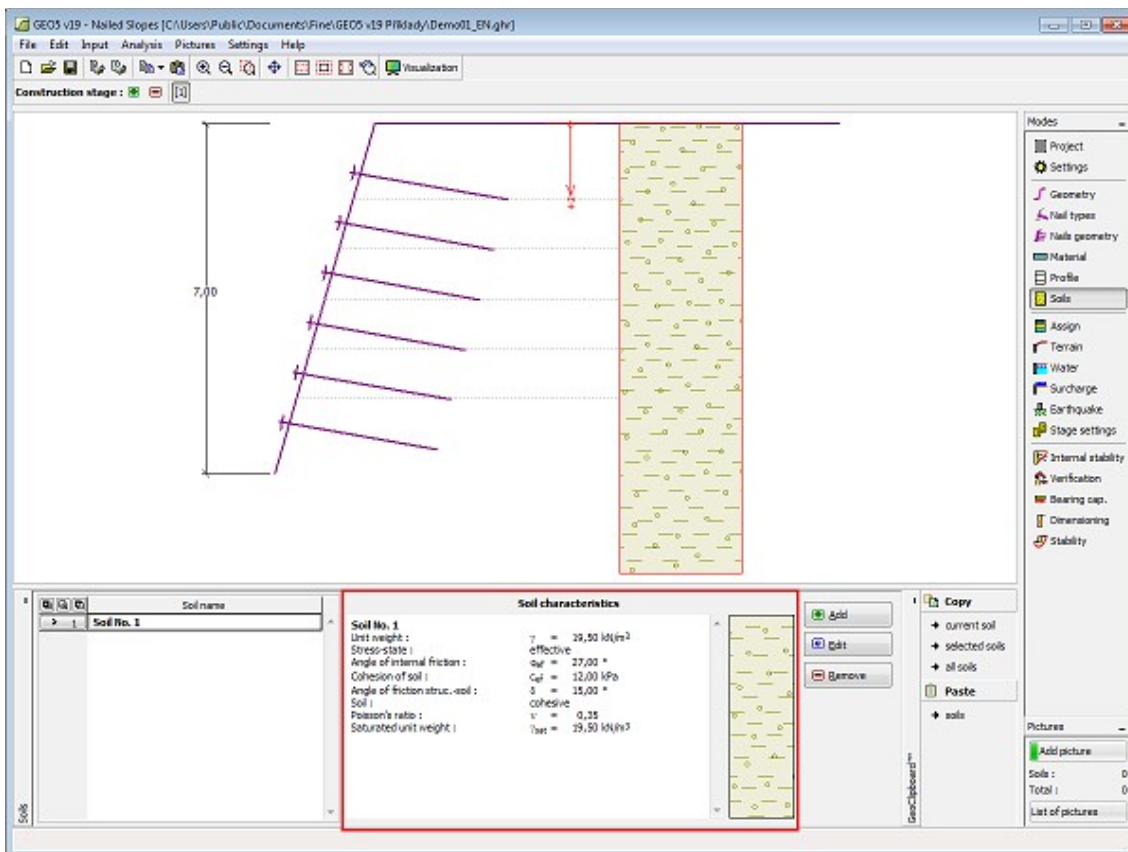
## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest" and "Uplift pressure".

The program makes it possible import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

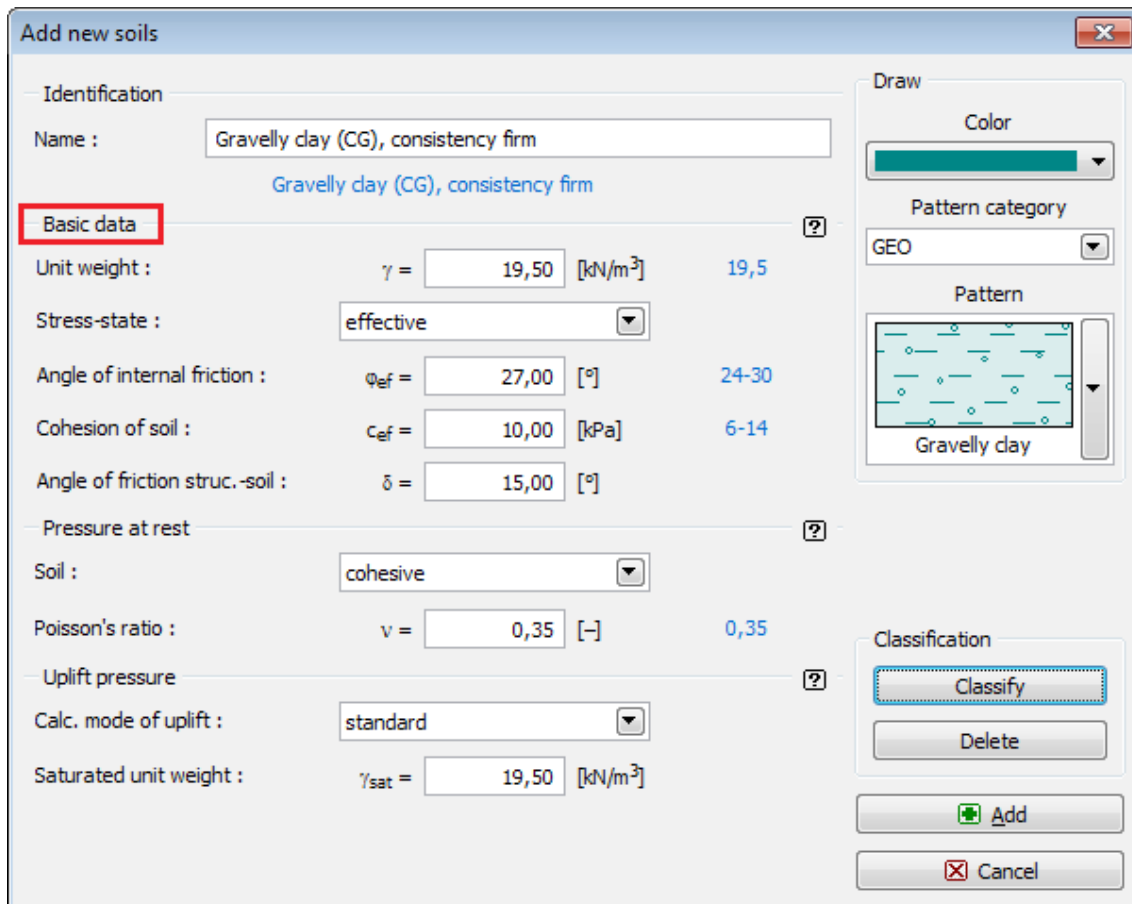
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the setting in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil, type of load, structure duration and water conditions.

For **effective stress** further needs to specify the **angle of internal friction between the soil and structure**, which depends on the structure material and the type of soil. Possible values of this parameter are listed in the **table of recommended values**.

For **total stress** further needs to specify the **adhesion of soil to the structure face**  $a$ .

The associated theory is described in detail in chapter "**Earth pressures**".



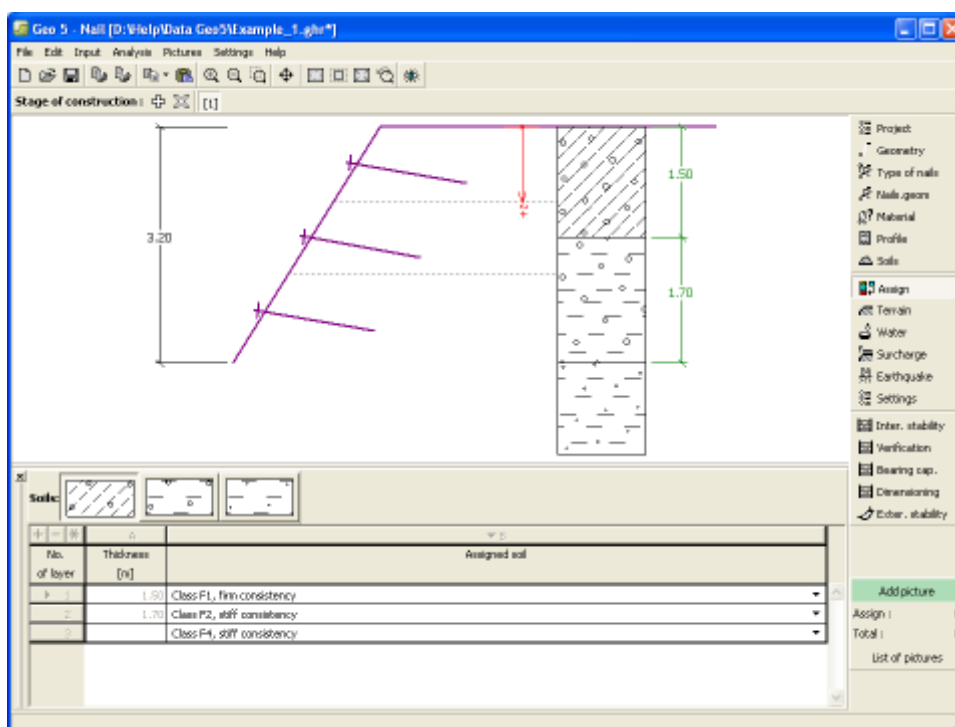
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



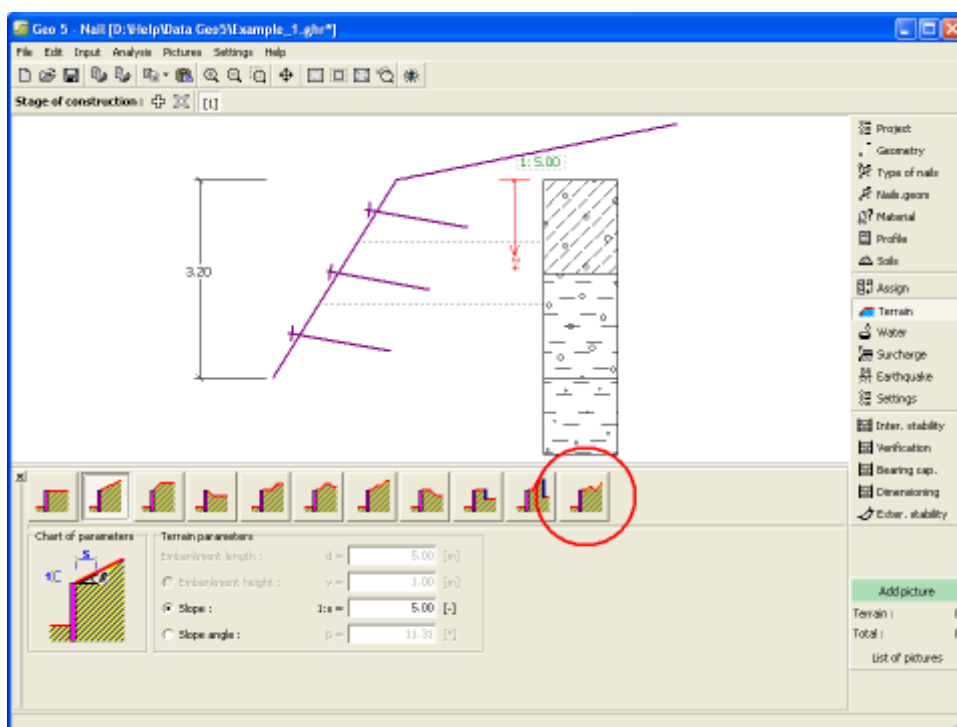
Frame "Assign"

## Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("Chart of parameters") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

## Water

The **"Water"** frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint (**"Chart of parameters"**) of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

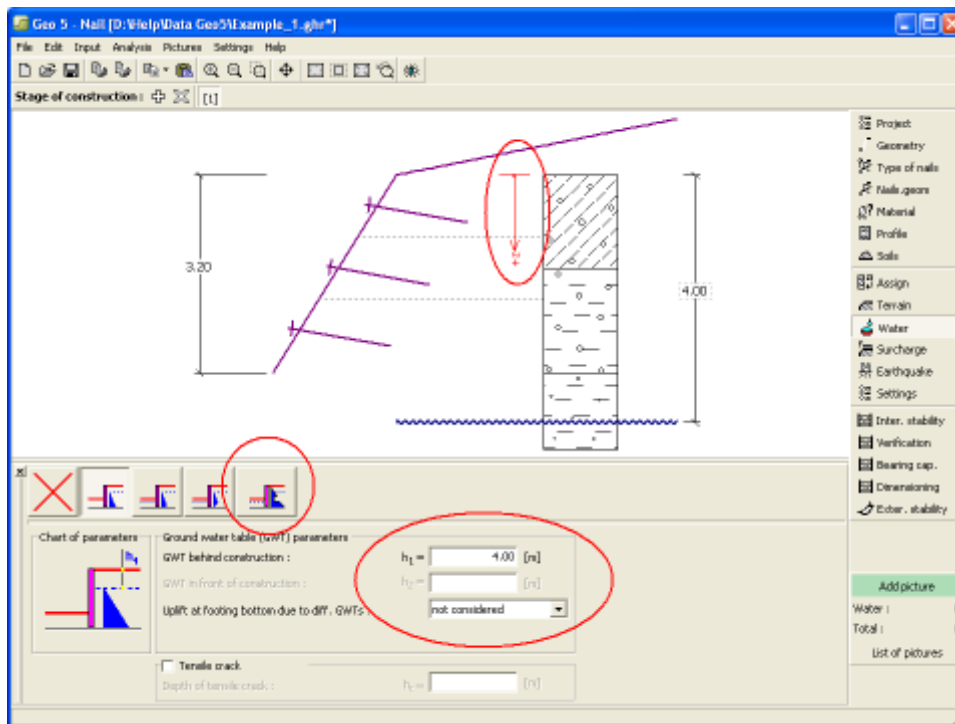
The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be **linear, parabolic** or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

The last option is a manual input of pore pressure both in front and behind the structure. Two tabs **"In front of structure"** and **"Behind structure"** appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter **"Influence of water"**.

The program further allows for specifying a depth of **tensile cracks** filled with water.



Frame "Water"

## Surcharge

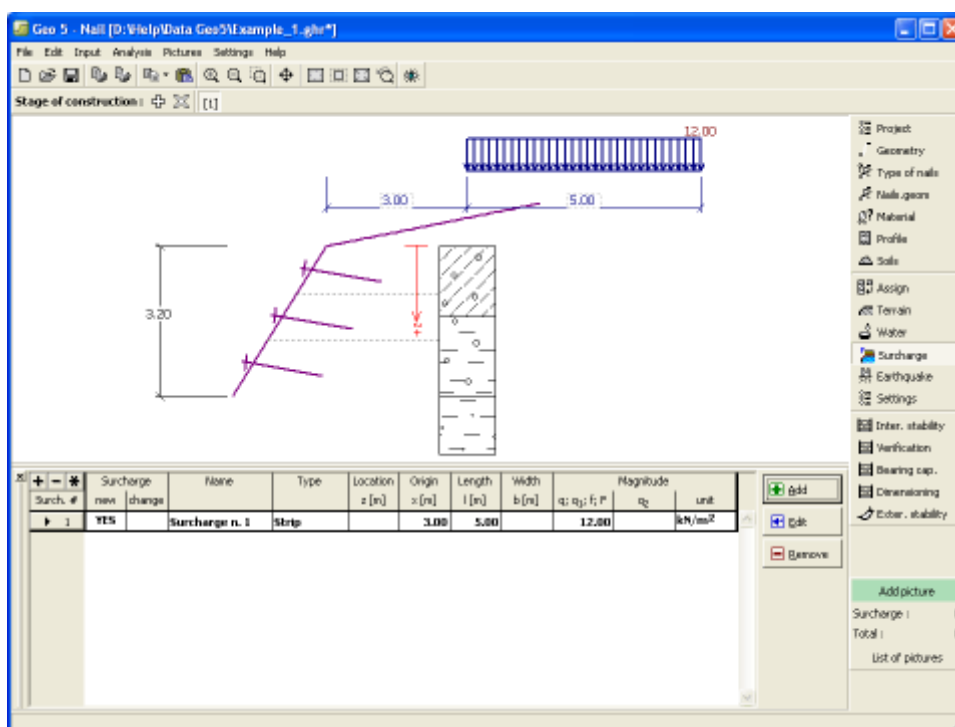
The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The  $z$ -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".





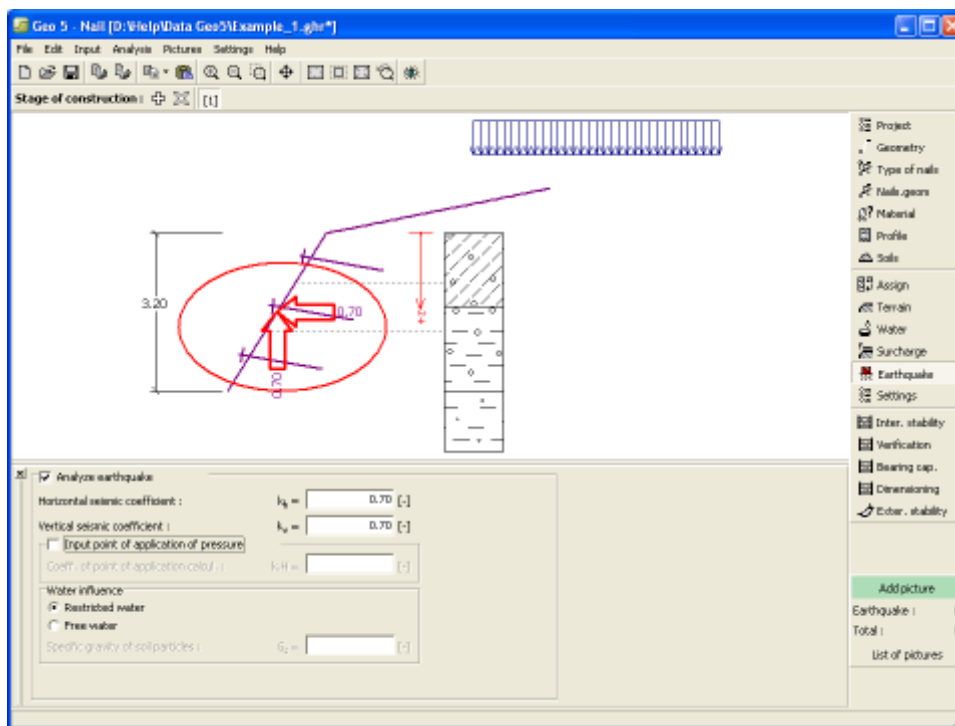
Frame "Surcharge"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



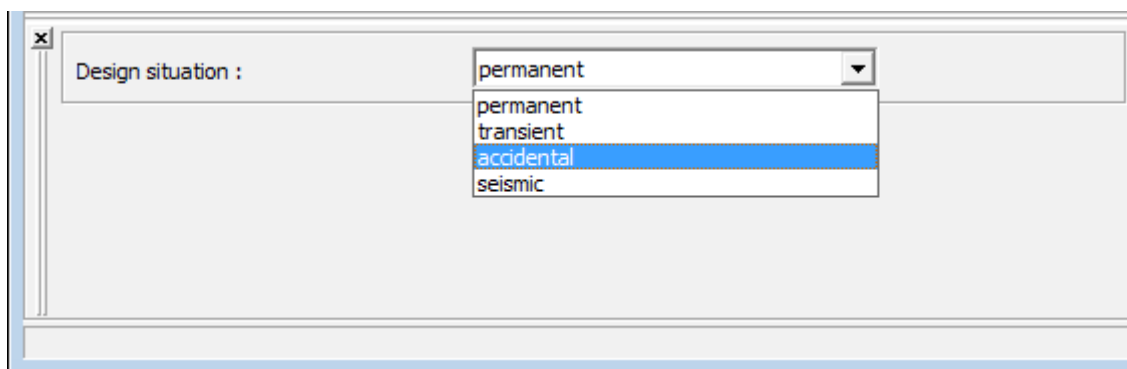
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Internal Stability

The frame allows for the verification of internal stability of a structure assuming either **plane** or **broken slip surface**.

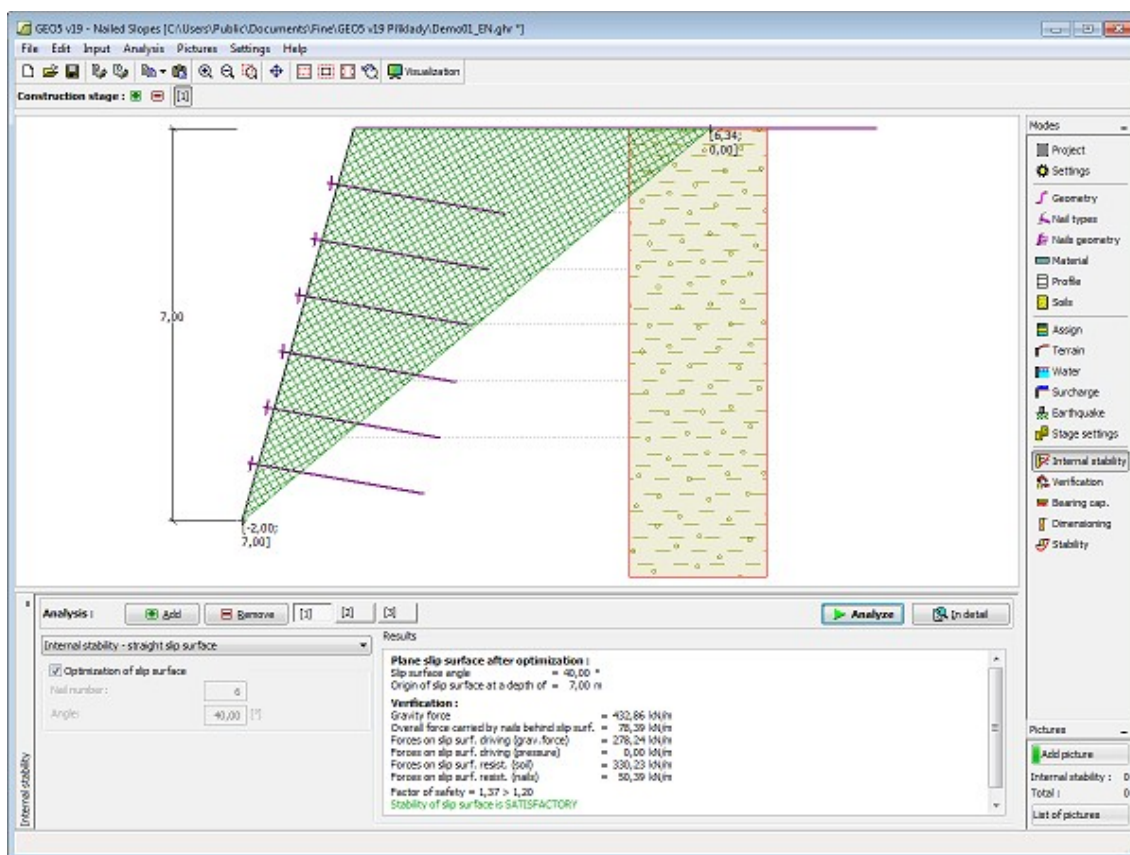
The verification of internal stability is performed:

- According to **EN 1997** (the actual verification is performed according to the theory of **limit states**).
- According to **LRFD** (the actual verification is performed according to the theory of **limit states**).
- Verification according to the **factor of safety**, or the theory of **limit states** (verification is performed depending on the setting in the "Stability analysis" tab)

Individual steps of the verification procedure are described [herein](#).

This frame also allows for the verification of **nails bearing capacity**.

To determine the **nail force**, in this frame is defined reduction coefficient of active earth pressure  $k_N$ .



Frame "Internal stability"

## Verification

The **frame "Verification"** shows the analysis results. **Several computations** can be carried out for a single task.

The frame appearance is adjusted based on the selected **verification methodology**.

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether

the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations".

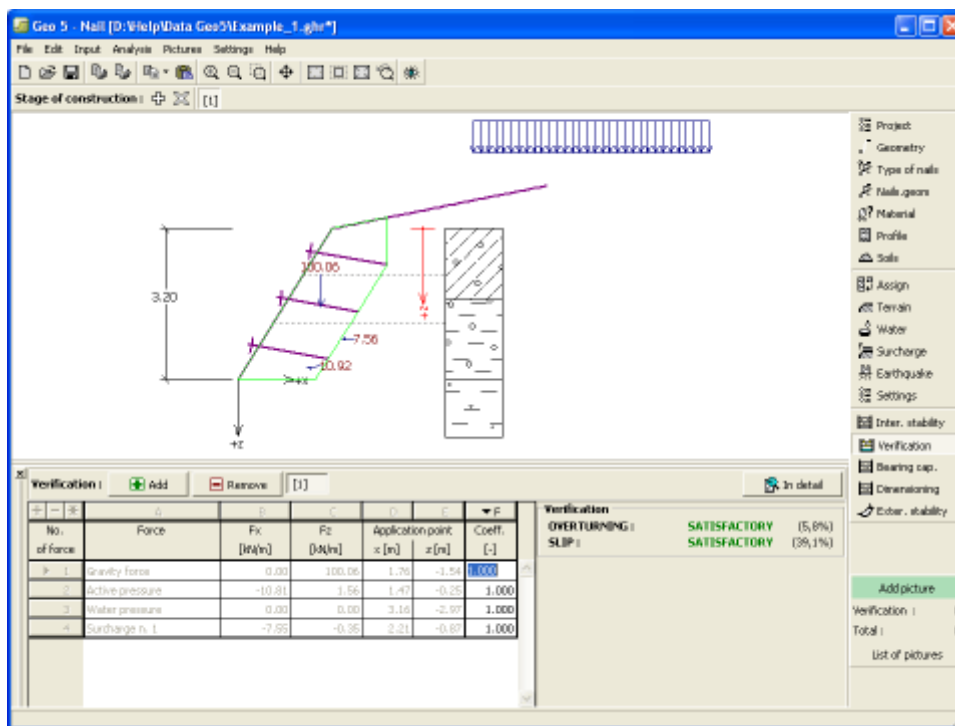
- **Analysis according to LRFD** - in the case the last column is not displayed.

To verify the external stability a **fictitious structure** (wall) is created and further subjected to the verification analysis. A fictitious wall consists of the structure front face, a line connecting end points of individual nails, a vertical line constructed from the end point of the first nail up to the terrain depth and from the end point of the last nail up to the structure depth (thus the bottom edge of a fictitious structure is always horizontal). The wall points that cause a concave curvature of the structure back face are automatically excluded by the program. The structure is loaded by an **active earth pressure**.

The procedure for **wall verification** is described in the theoretical part of the help.

The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and translation**. The "In detail" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Verification"

## Bearing Capacity

The "Bearing capacity" frame displays the results from the analysis of foundation soil bearing capacity. The stress in the footing bottom (assumed constant) is derived from all verifications performed in the frame "Verification". The program "Spread footing" then considers all verifications as load cases.

Three basic analysis options are available in the frame:

- **Input the foundation soil** The input field serves to specify the foundation soil bearing

**bearing capacity**

capacity. The results of verification analysis of a soil for **eccentricity** and **bearing capacity** are displayed in the right part of the frame. The **"In detail"** button opens the dialog window that displays detailed listing of the results of verification analysis of foundation soil bearing capacity.

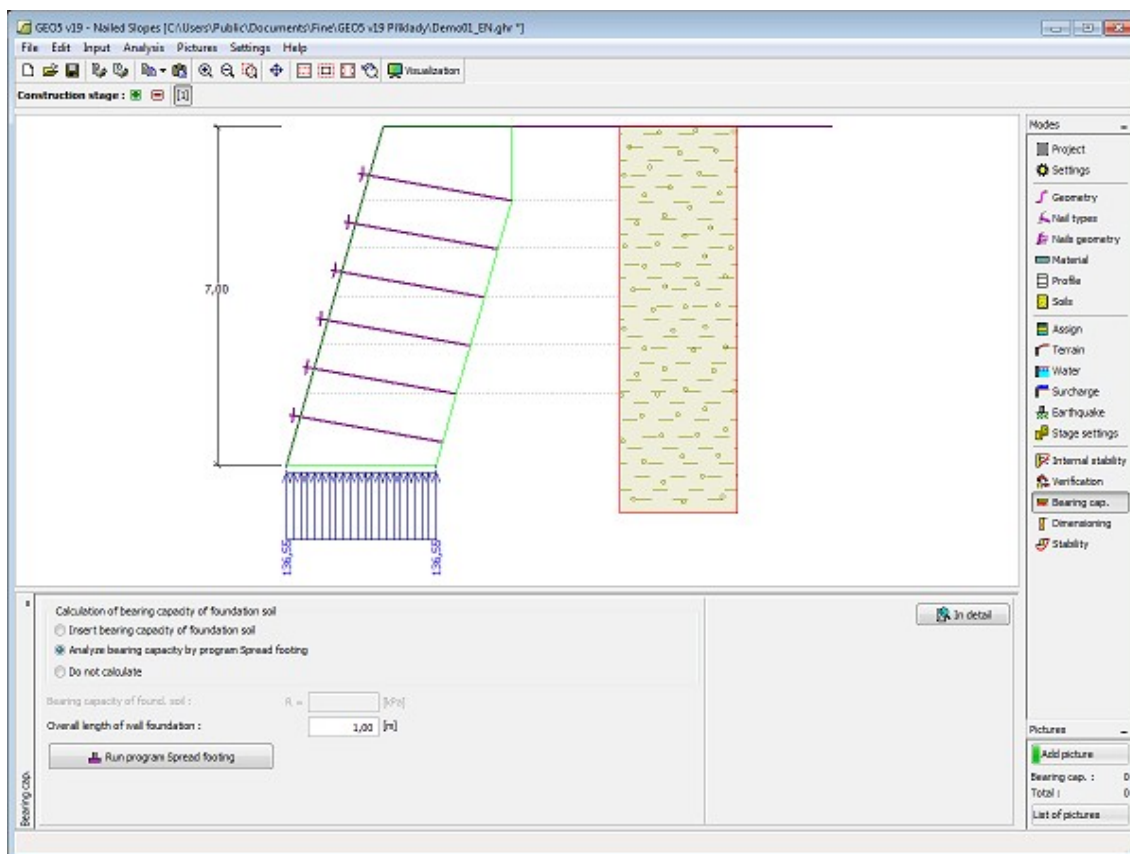
- **Compute the foundation soil bearing capacity using the program "Spread footing"**

Pressing the **"Run program Spread footing"** button starts the program **"Spread footing"** that allows for computing the soil bearing capacity or settlement and rotation of a footing. Pressing the **"OK"** button leaves the analysis regime - the results and all plots are copied to the program **"Nailed Slope"**. The program **"Spread footing"** must be installed for the button to be active. Overall length of wall foundation is input.

- **Do not compute (pile footing)**

The foundation soil bearing capacity is not computed.

Visualization of results can be adjusted in the frame **"Visualization settings"**.



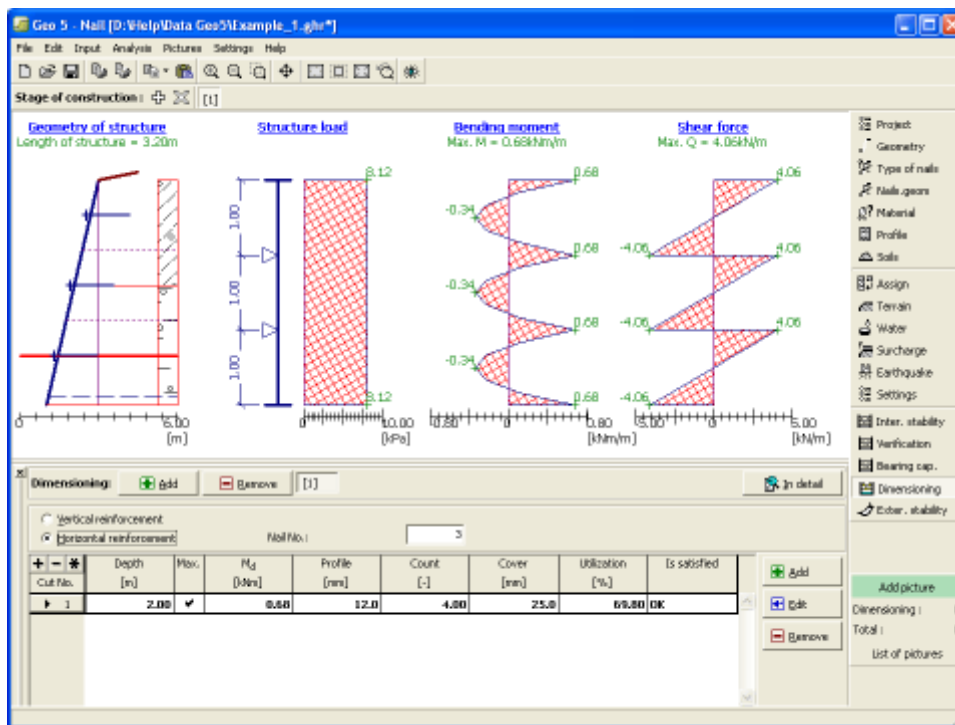
Frame "Bearing capacity"

## Dimensioning

The frame **"Dimensioning"** allows for the design and verification of the **reinforcement of the structure concrete cover**. The upper part of the frame serves to choose whether the **vertical or horizontal reinforcement** and its location will be verified. The program then **determines internal forces** developed on the selected section.

The **table** in the bottom part of the frame serves to specify locations for the verification of the designed reinforcement depending on the input standard for **dimensioning of reinforced concrete** (the standard is specified in the "Materials and standards" tab). A cross-section is loaded by the bending moment in a given point. An amount of the **tensile reinforcement** in the cross-section is input. If the moment is positive, the designed reinforcement is placed at the structure front face and if it is negative, then at the structure back face.

Visualization of results can be adjusted in the frame "Visualization settings".

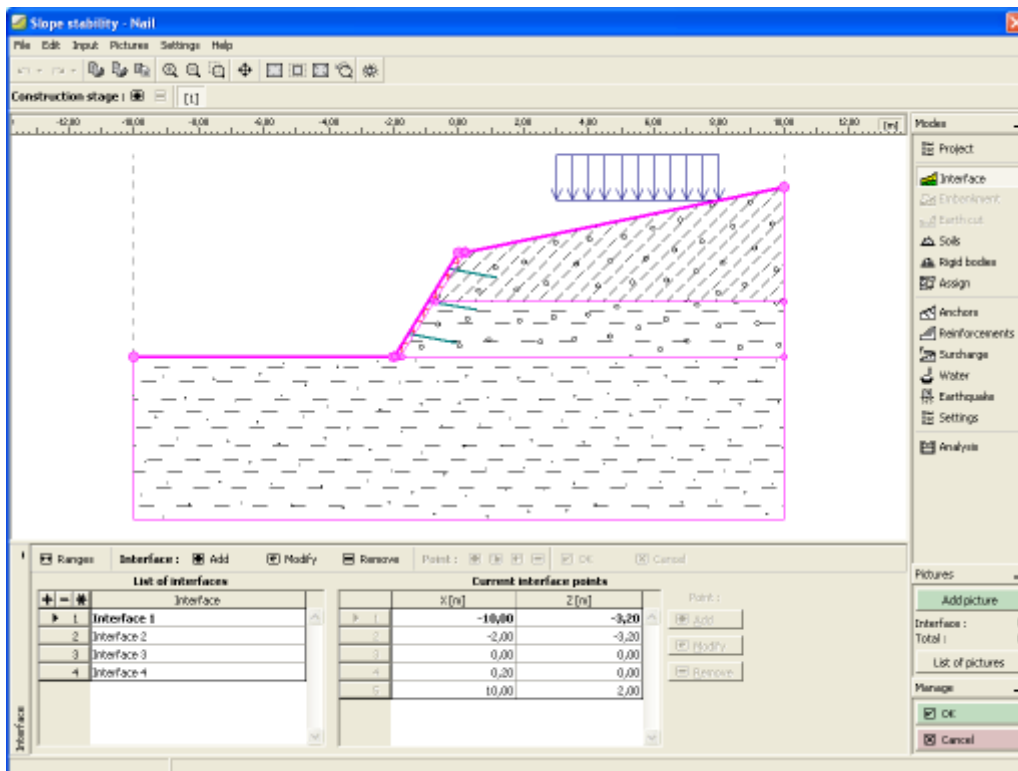


Frame "Dimensioning"

## External Stability

Pressing the "External stability" button launches the "Slope stability" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "Slope stability" is installed.

After completing all analyses press the "OK" button to leave the program - all data are then carried over to the analysis protocol of the "Nailed Slope" program.



Frame "External stability"

## Program Sheeting Design

This program is used for fast design of non-anchored and preliminary design of anchored retaining walls. The results are required embedment lengths, internal forces on the structure and forces in anchors. **"Sheeting Design"** provides only preliminary analysis.

Final analysis of multiplied anchored walls should be provided by **Sheeting Check** program (elasto-plastic nonlinear method).

**The help in the program "Sheeting Design" includes the following topics:**

- Input of data into individual frames:

Project	Settings	Profile	Soils	Assign	Geometry	Anchors
Props	Supports	Pressure Determination	Terrain	Water	Surcharge	Applied Forces
Earthquake	Stage Settings	Analysis	Stability			

- Standards and analysis methods
- Theory for analysis in the program **"Sheeting Design"**:

[Stress in Soil Body](#)   
 [Earth Pressures](#)   
 [Sheeting Design](#)   
 [Braced Sheeting](#)

- [Outputs](#)
- General information about the work in the [User Environment](#) of GEO5 programs
- [Common input](#) for all programs

## Project

The [frame "Project"](#) is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in [text and graphical outputs](#).

The frame also allows user to switch analysis units ([metric / imperial](#)). Project data can be copied within all GEO5 programs using ["GeoClipboard"](#).

The screenshot shows the 'Project' frame with the following data:

Task :	Terraces Hanspaulka	Author :	James Baker
Part :	South-facing slope III.	Date :	28.10.2005
Description :	Support walls 2-5m	Project ID :	845/2014
Customer :	Belltrade LTd.	Project number :	11486/2014
System of units	metric		

*Frame "Project"*

## Settings

The [frame "Settings"](#) allows to introduce the basic [settings](#) of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the ["Settings list"](#).

The **"Settings Administrator"** button opens the ["Administrator"](#) dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

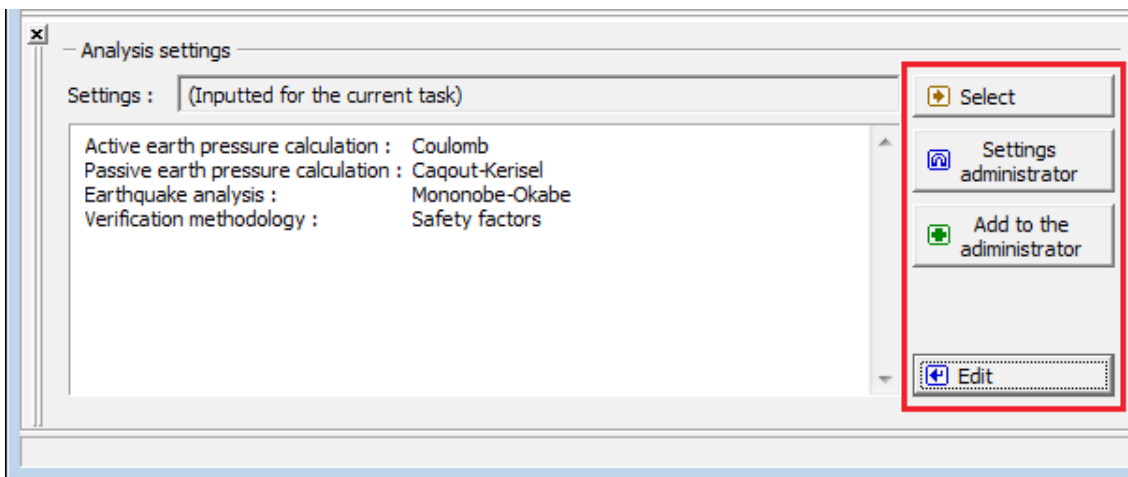


The "**Add to the administrator**" button allows to create user-defined Settings, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when importing older data.

Settings of analysis parameters are performed in the "**Excavations**" tab.



Frame "Settings"

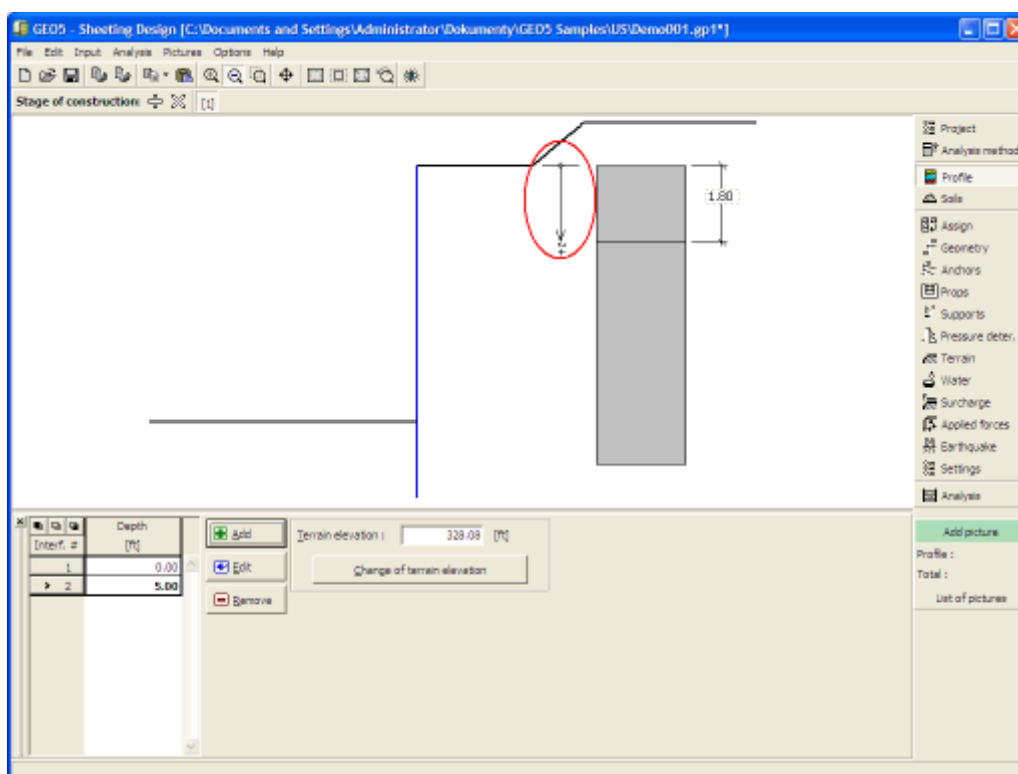
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "**New interface**" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to import a profile in the gINT format.



Frame "Profile"

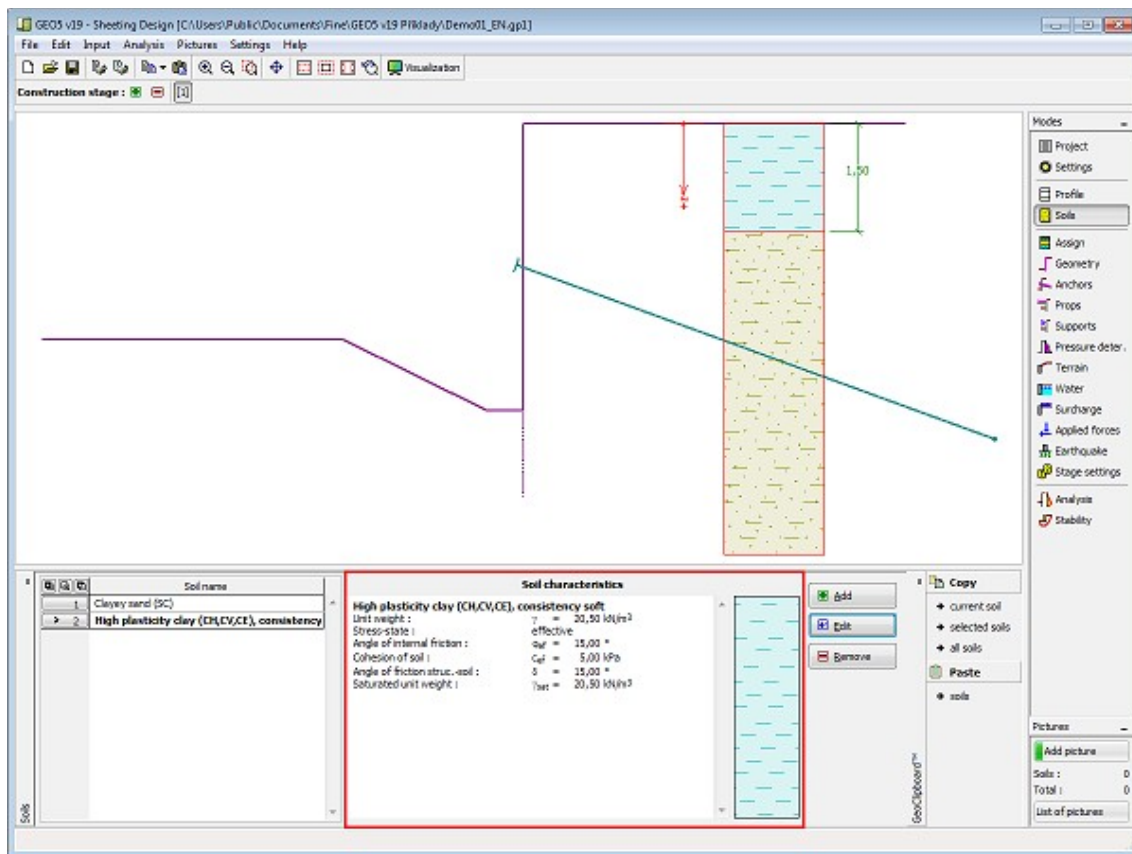
## Soils

The **"Soils"** frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data" and "Uplift pressure".

The program makes it possible import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

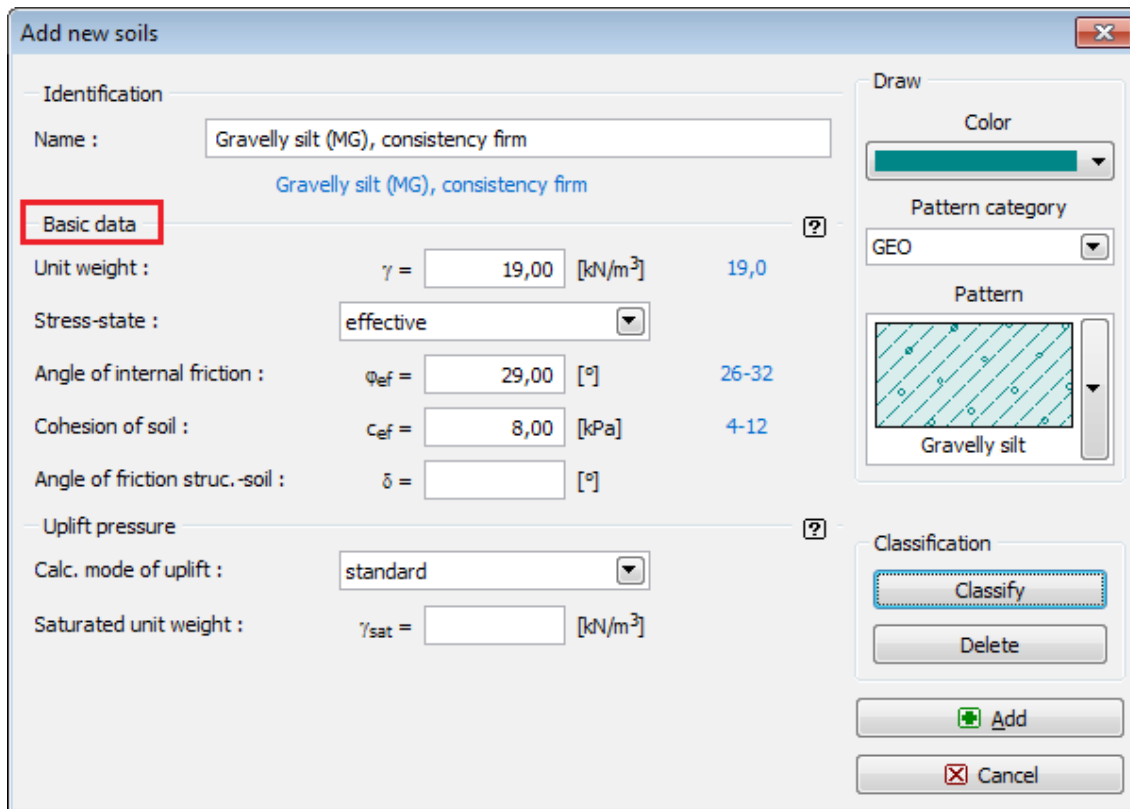
This part of the window allows to introduce basic parameters of the soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **soils database**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually. Approximate parameters of rocks are presented in the **theoretical section**.

Either **effective or total** parameters of the angle of internal friction and cohesion are specified depending on the settings in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil and load, structure duration and water conditions.

For **effective stress**, it is further needed to specify the **angle of internal friction between the soil and the structure**, which depends on the structure material and type of soil. Possible values of this parameter are listed in the **table of recommended values**.

For **total stress**, it is further needed to specify the **adhesion of soil to the structure face  $a$** .

The associated theory is described in detail in the chapter "**Earth pressures**".



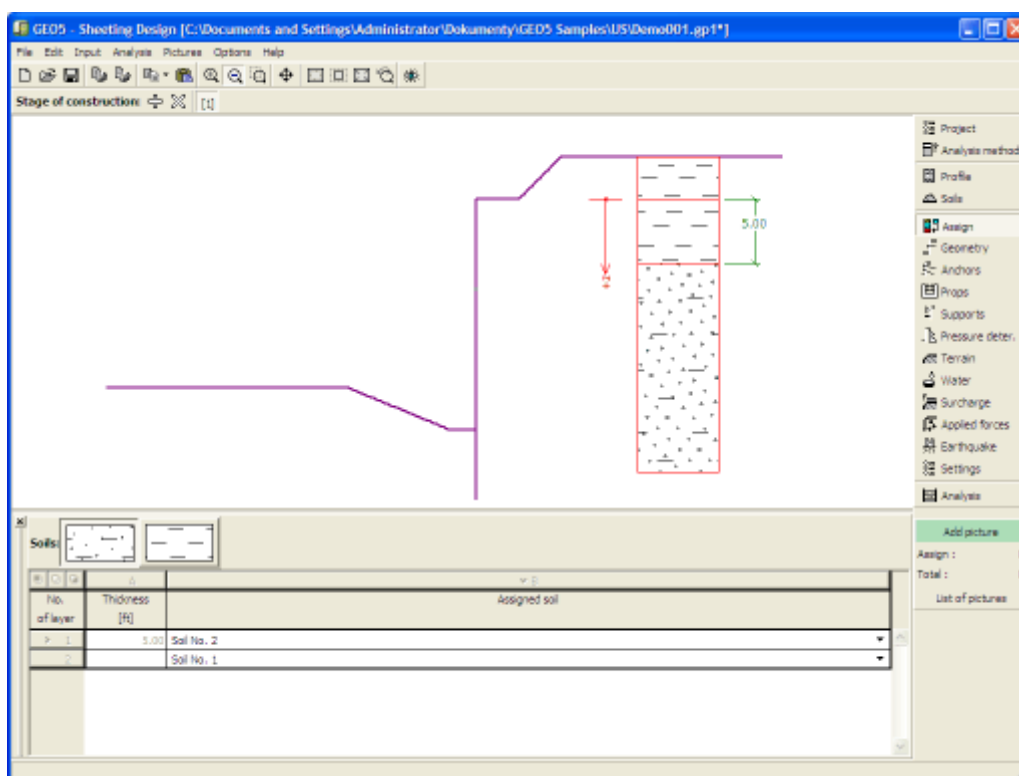
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



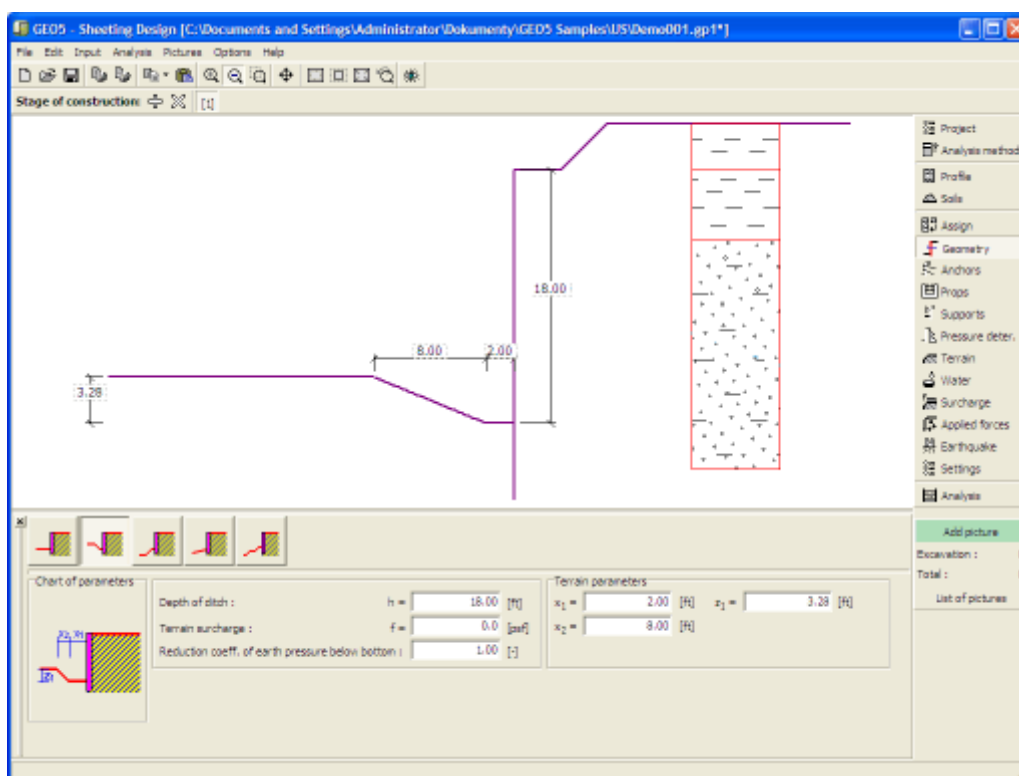
Frame "Assign"

## Geometry

The "Geometry" frame is used to specify the depth of a construction ditch and by pressing the button to choose the shape of a bottom. The selected shape with a graphic hint ("Parameter chart") appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The frame can be further used to input surcharge of a construction ditch bottom and coefficient of reduction of earth pressure below the ditch bottom (this coefficient serves to analyze braced sheeting).

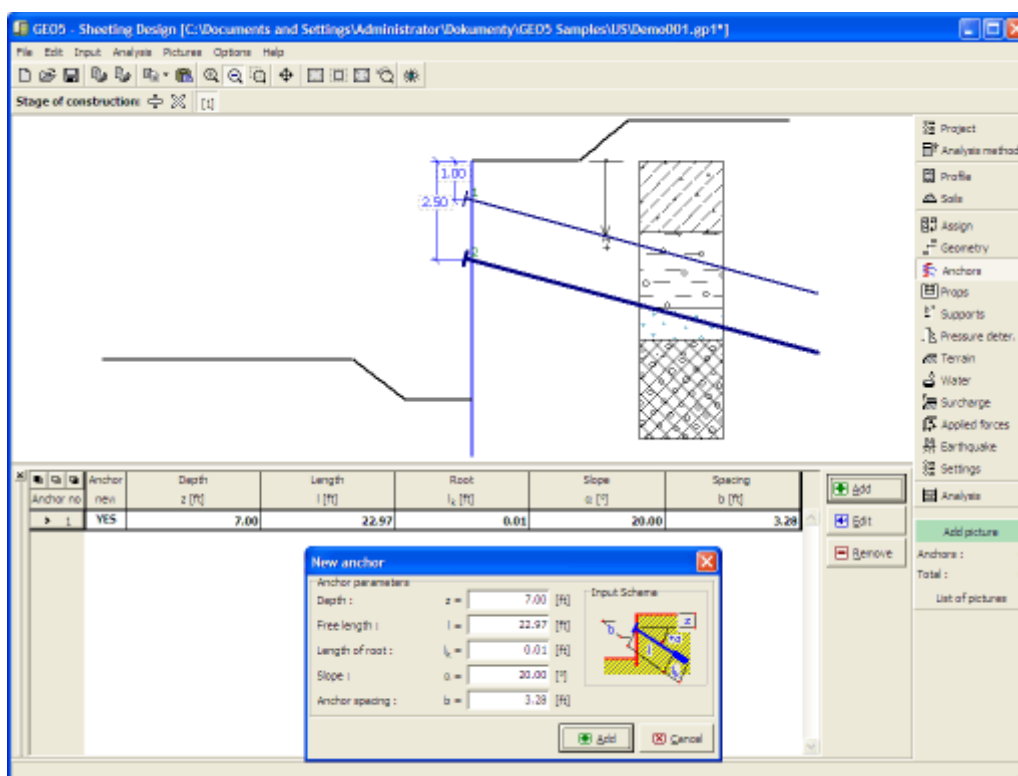
The program makes it possible to export the geometry of a structure in the \*.DXF format.



Frame "Geometry"

## Anchors

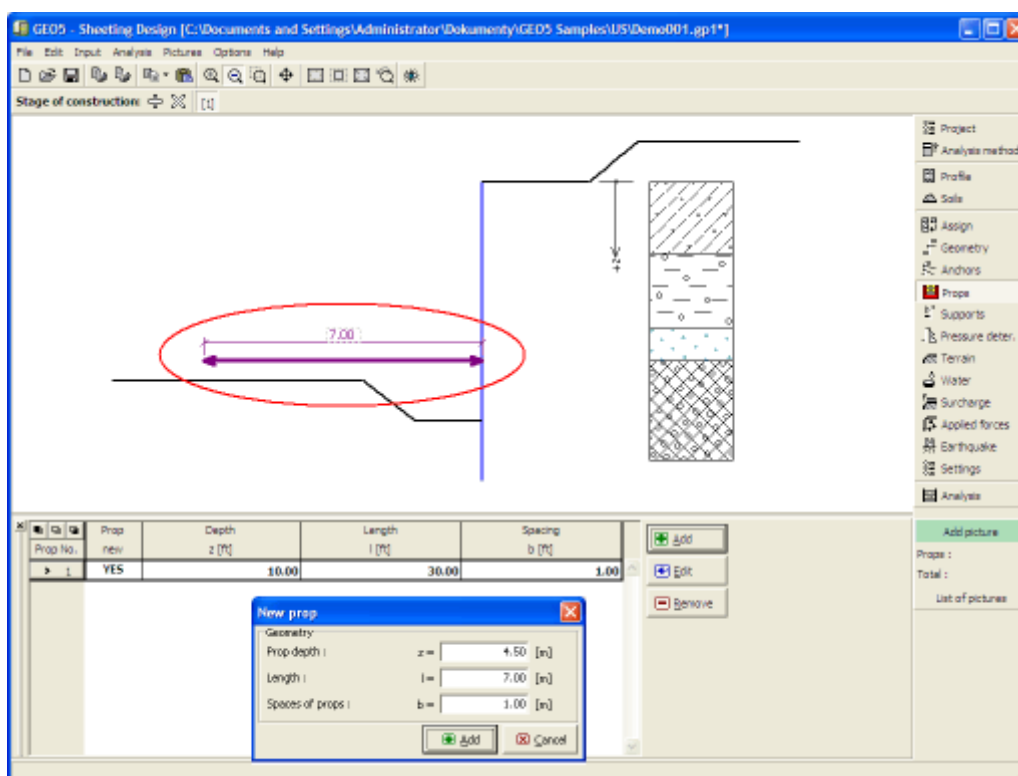
The "Anchors" frame contains a table with a list of input anchors. Adding (editing) anchors is performed in the "New anchor (Edit anchor)" dialog window. The input anchors can be edited on the desktop with the help of active objects.



Frame "Anchors"

## Props

The "Props" frame contains a table with a list input props. Adding (editing) props is performed in the "New prop (Edit prop)" dialog window. The input props can also be edited on the desktop with the help of active dimensions or active objects, respectively.

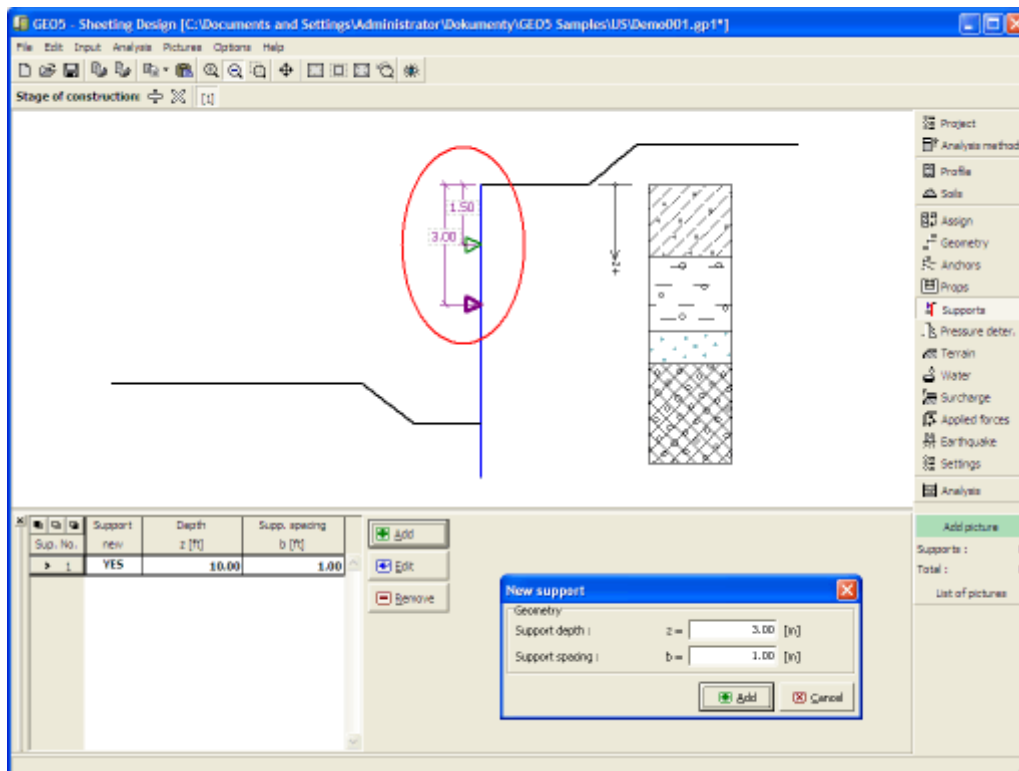


Frame "Props"

## Supports

The "Supports" frame contains a table with a list of input supports. Adding (editing) supports is performed in the "New support (Edit support)" dialog window. The input supports can also be edited on the desktop with the help of active dimensions or active objects, respectively.



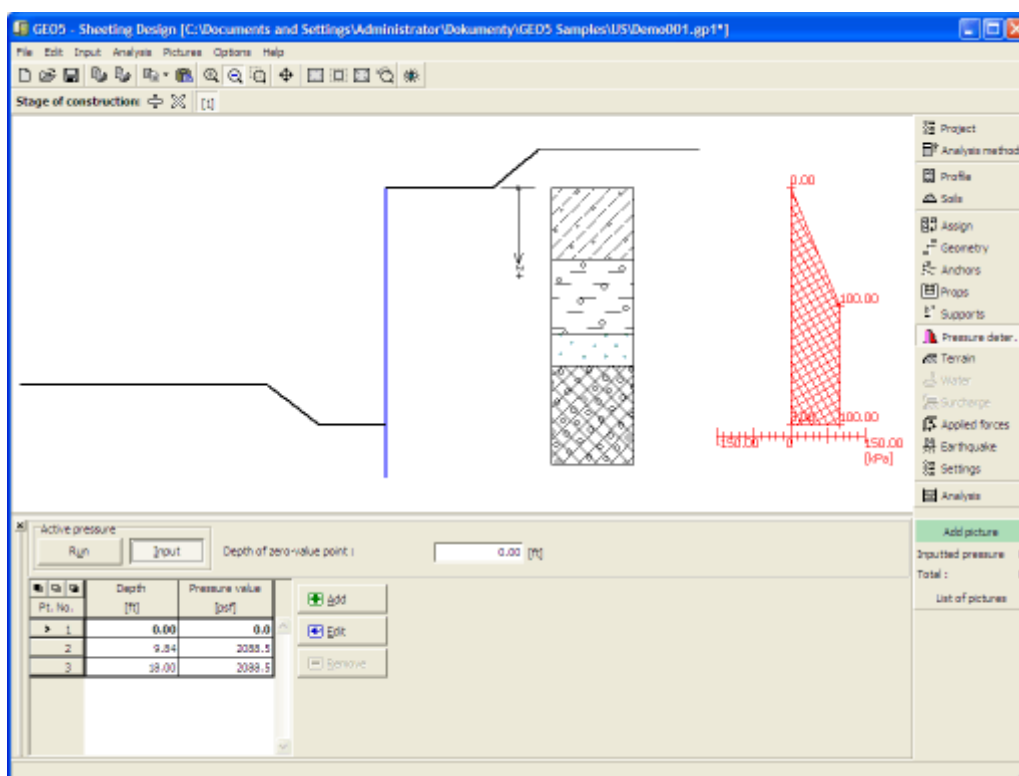


Frame "Supports"

## Pressure Determination

The "**Pressure specification**" frame allows by pressing the button "**Analyze**" ("**Input**", respectively) for selecting a method for the calculation of active earth pressure. Choose option "**Analyze**" if you wish the active earth pressure to be computed automatically based on specified earth profile.

In some special cases (redistribution of earth pressures due to presence of anchors, nonstandard rotation of a structure) it is advisable to specify the distribution of earth pressure on a structure manually. Selecting the option "**Input**" opens a table in the frame with a list of input points and the corresponding pressure value. The pressure is specified up to the depth of structure increased by the depth of zero point (the depth of zero point is introduced in the top part of the frame). The depth of zero point equal to zero is selected if we wish to specify the pressure values only up to the depth of construction ditch. Below the ditch the program computes the pressure values based on the specified geological profile. Providing the earth pressure is specified manually the program does not account for the influence of terrain profile, surcharge and water.



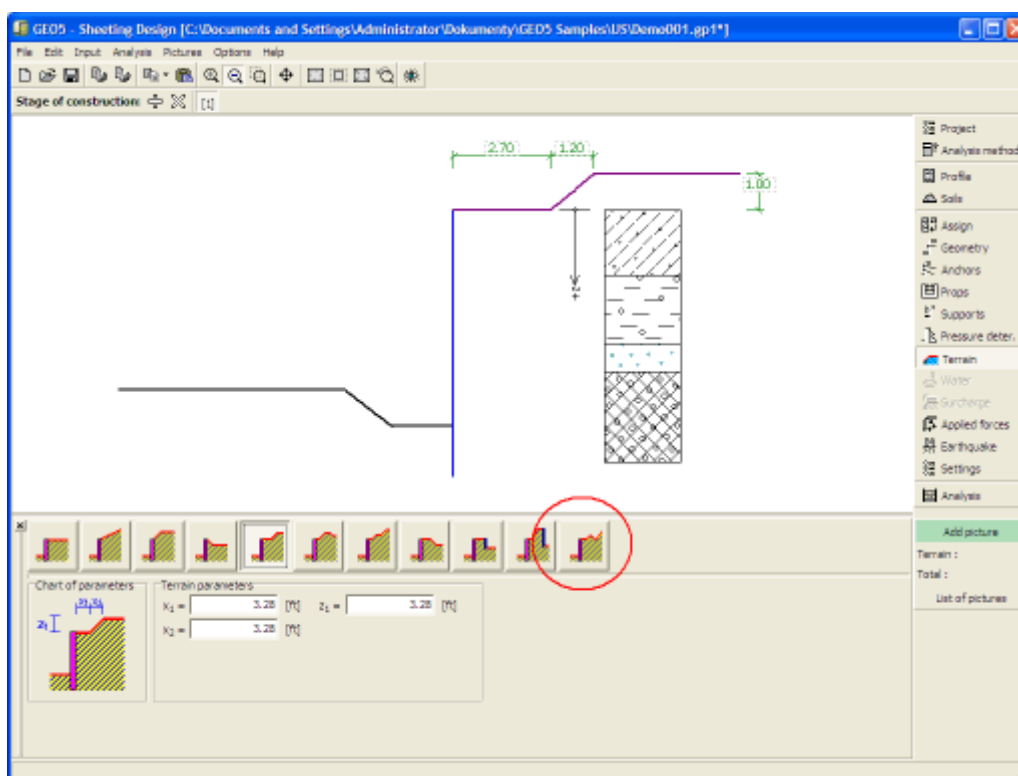
Frame "Pressure determination"

## Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

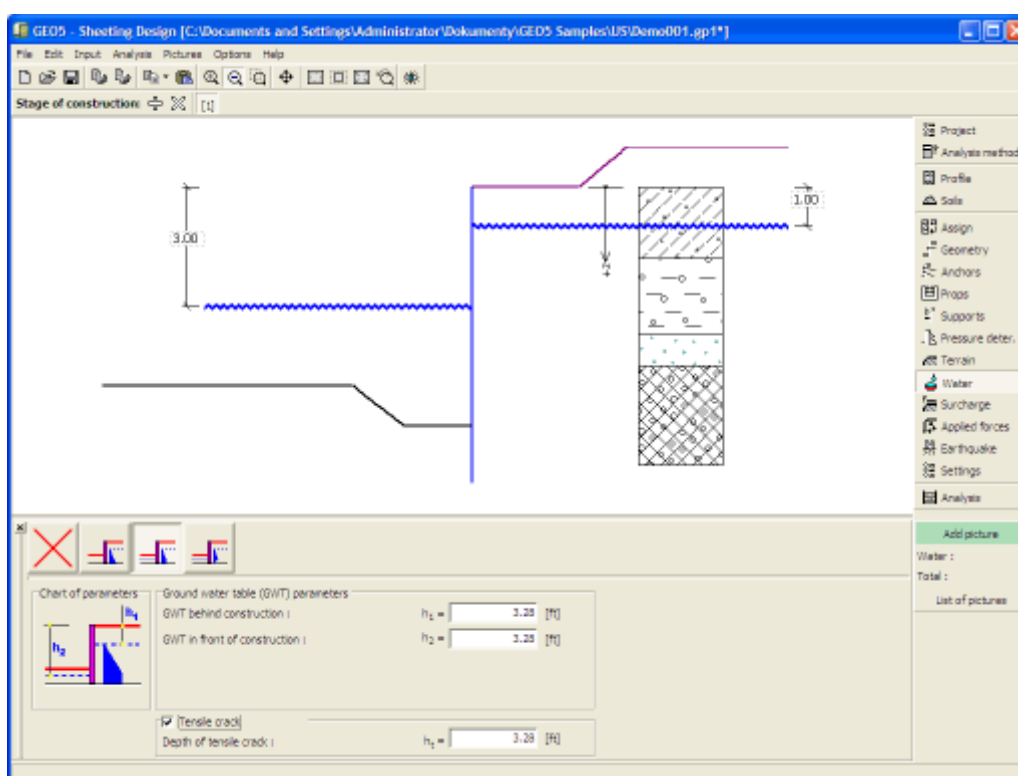
## Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".

The program further allows for specifying a depth of **tensile cracks** filled with water.



Frame "Water"

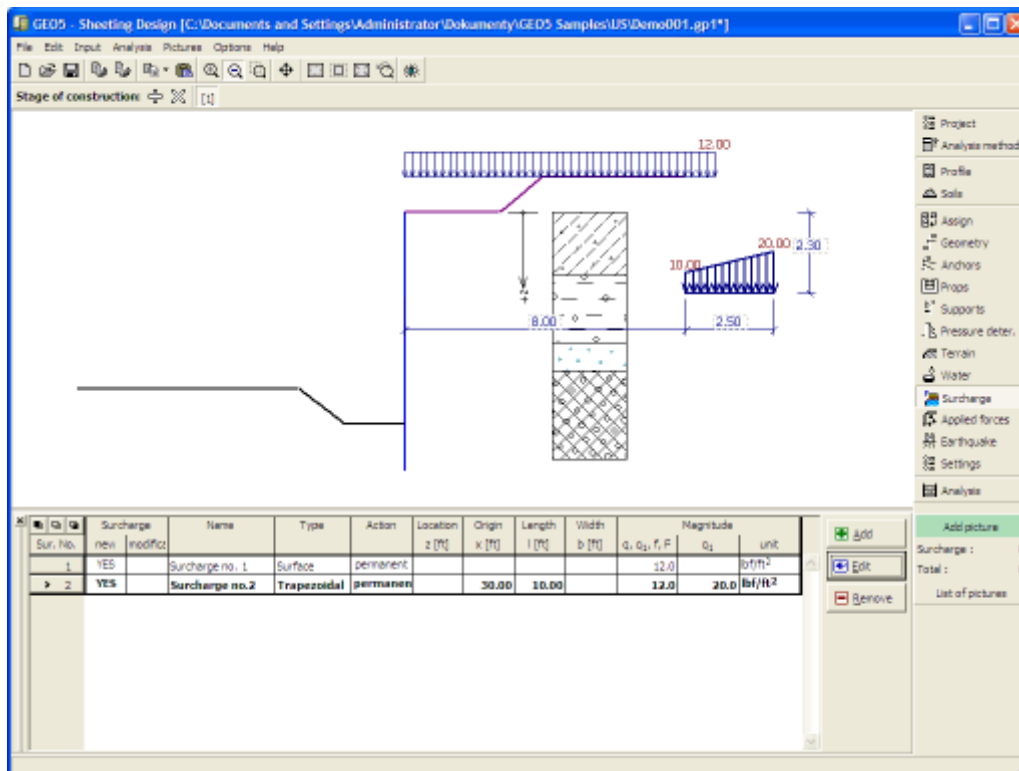
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The  $z$ -coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharge is described in the theoretical part of the help, chapter "**Influence of surcharge**".



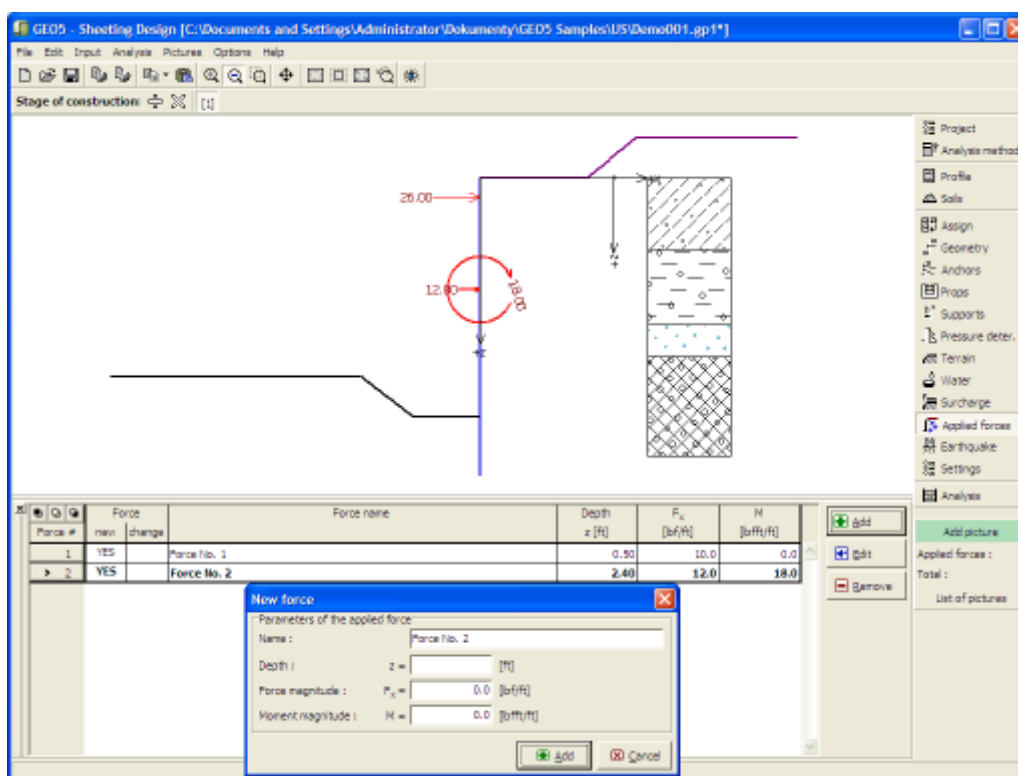
Frame "Surcharge"

## Applied Forces

The "Applied forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.



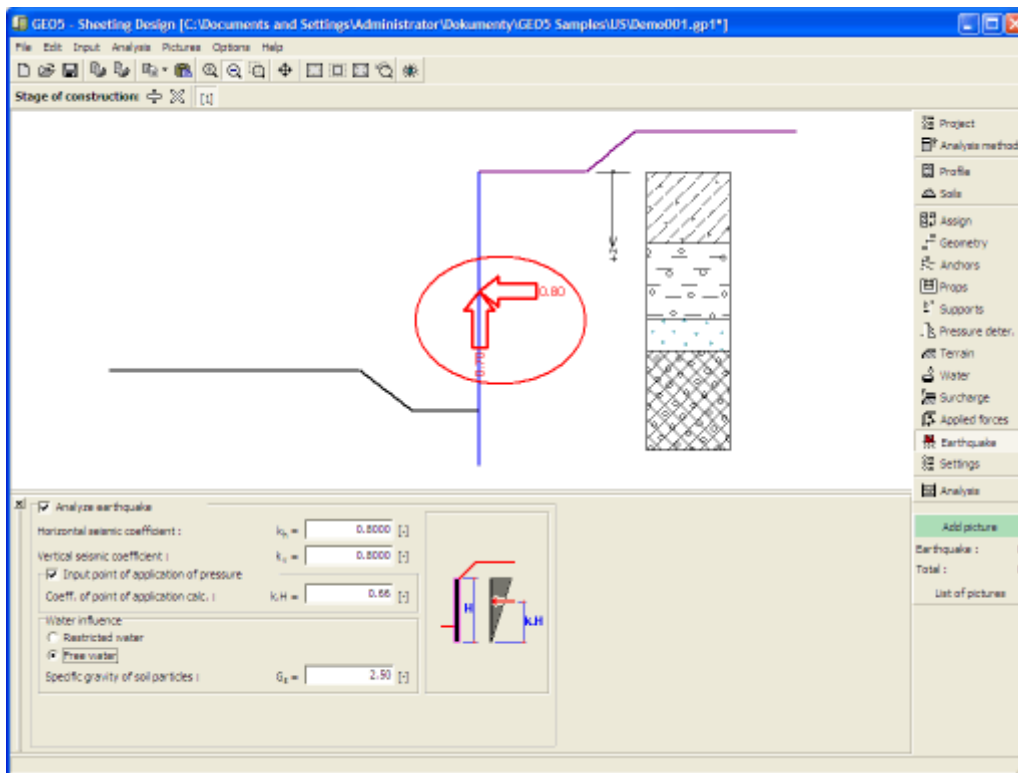
Frame "Applied forces"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



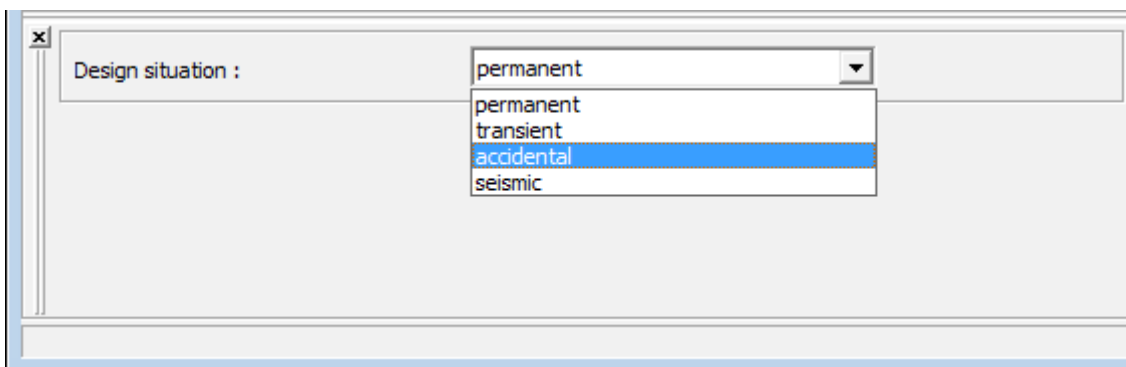
Frame "Earthquake"

## Stage Settings

The **frame "Stage settings"** serves to input settings valid for a given construction stage.

Selected **design situation** determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected **verification methodology**. LRFD 2012 introduces new types of **design situations (Strength I, Service I, Extreme I)**.



Frame "Stage settings"

## Analysis

The **"Analysis" frame** displays the analysis results. The analysis is carried out by pressing the **"Analyze"** button in the right part of the frame. The frame has two variants. The first variant

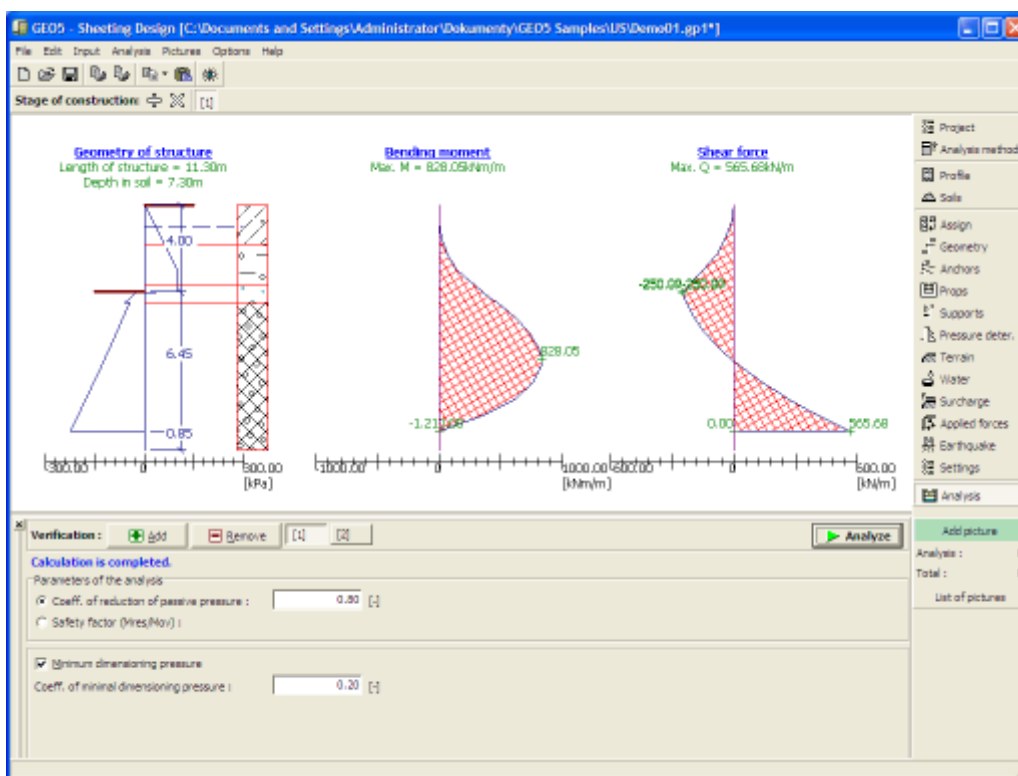
applies to a wall free of anchors (sheet pile) and the second one to an anchored (strutted) wall.

A coefficient of reduction of passive earth pressure (or factor of safety) together with a choice whether to consider a **minimal dimensioning pressure** behind the structure is specified for a **non-anchored** wall.

A type of heel support (**fixed, free**) and analysis parameters (coefficient of reduction of passive pressure, minimum dimensioning pressure) are specified for an **anchored** wall.

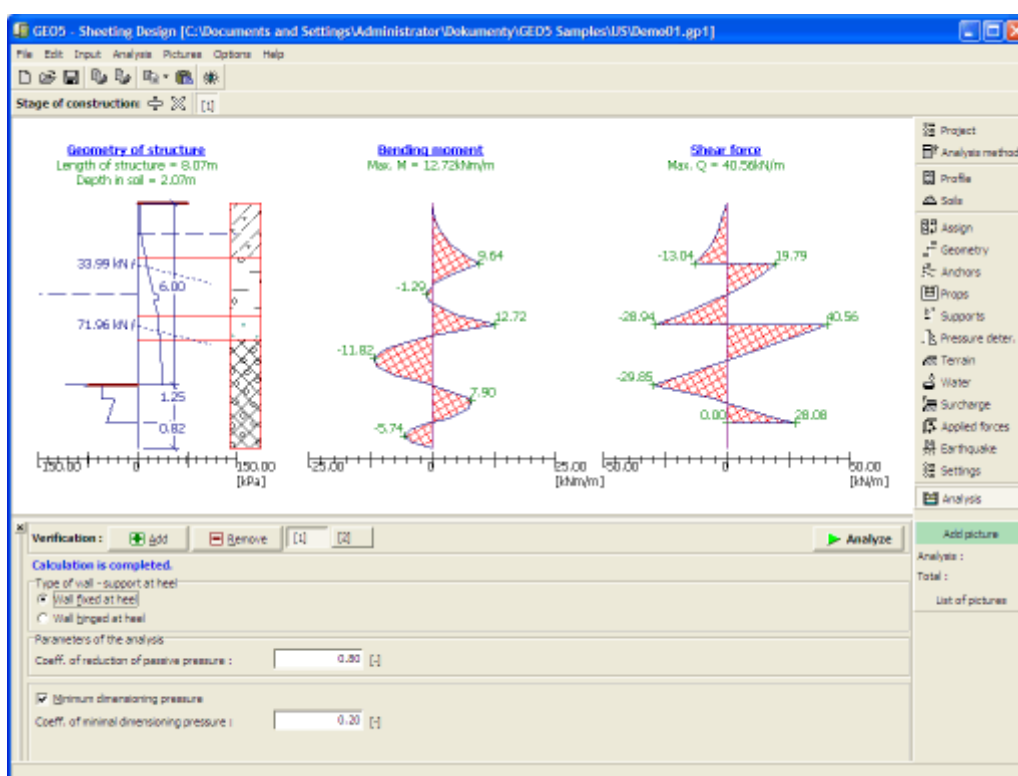
When performing the analysis according to **EN 1997** or **LRFD** the design coefficients are introduced in the **"Excavations"** tab. Providing the analysis is carried out according to **Design approach 1**, it is necessary to also enter the combination number.

The analysis results are displayed on the desktop. Visualization of results can be adjusted in the frame **"Visualization settings"**.



Frame "Analysis" - non-anchored wall





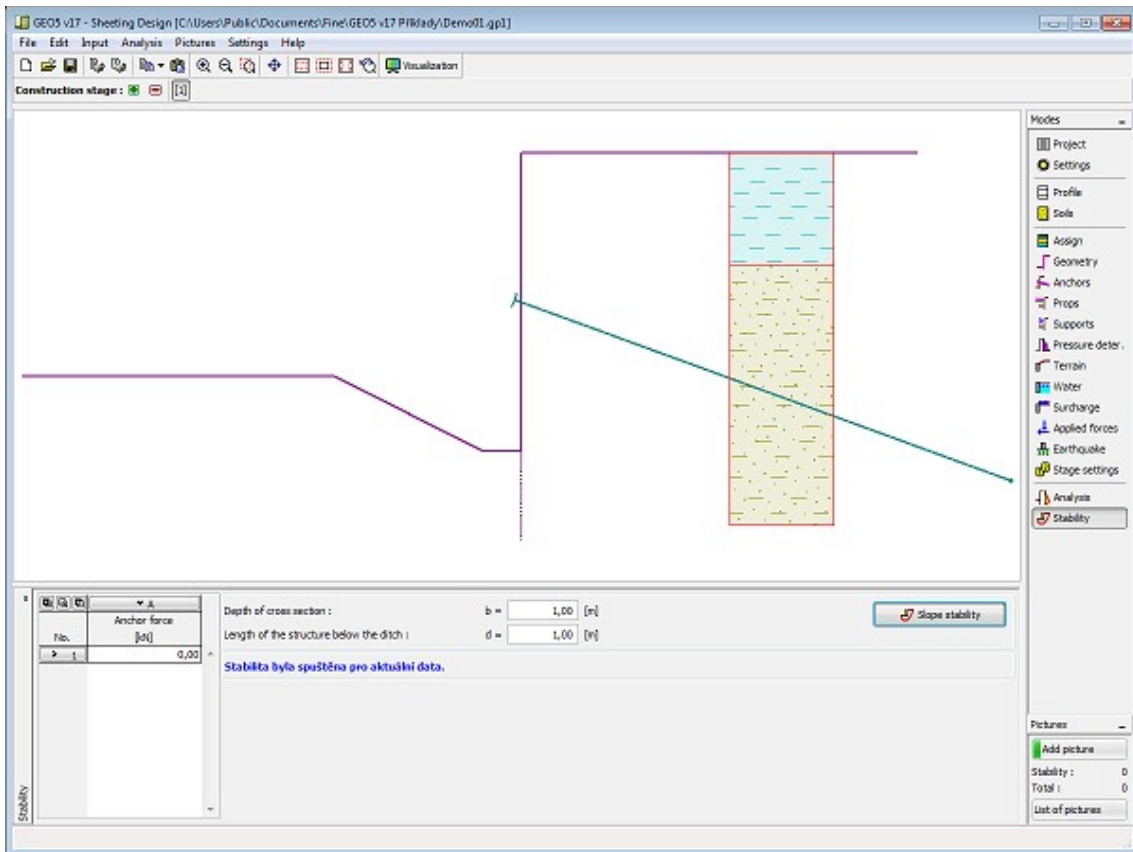
Frame "Analysis" - anchored wall

## Stability

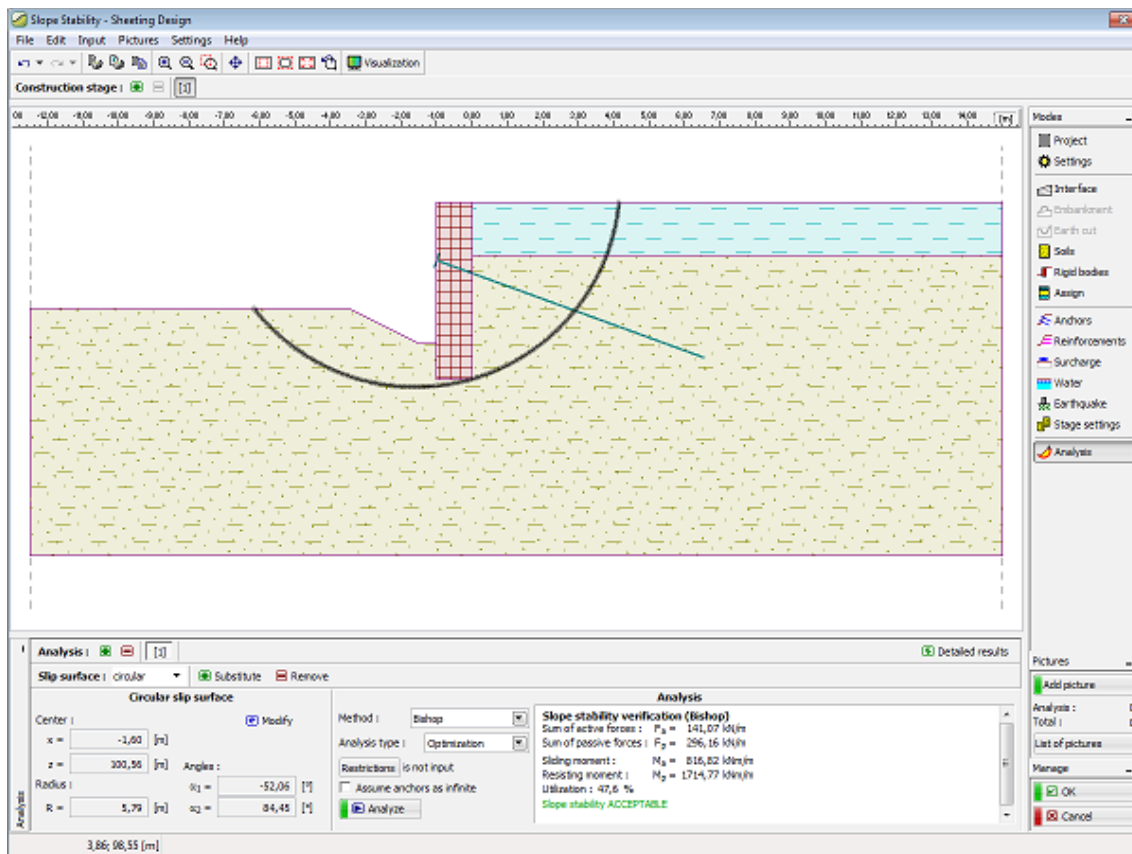
The "**Stability**" frame serves to evaluate the external stability of a structure employing the "**Slope stability**" program. This frame is used to introduce the geometry of a structure - "**Depth of cross section**" and "**Length of the structure below the ditch**", for anchored walls the corresponding "**Anchor force**" should also be specified.

Pressing the "**Stability**" button **launches** the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Sheeting design**" program.



Frame "Stability"



Program "Slope stability"

## Program Sheeting Check

This program is used to analyze deep excavations and retaining structures by the method of elasto-plastic non-linear analysis i.e. the magnitude of pressures acting upon a structure depend on its deformation. It models real behavior of the structure during construction process and determinates internal forces and deformations.

Preliminary design of wall dimensions, internal forces and anchor loads can be performed by [Sheeting Design](#) program.

**The help in the program "Sheeting Check" includes the folowing topics:**

- Input of data into individual frames:

Project	Settings	Profile	Modulus Kh	Pressiometri c Tests (PMT)	Dilatometric Tests (DMT)	Soils
Geometry	Material	Pressure Determination	Assign	Excavation	Terrain	Water

Surcharge	Applied Forces	Anchors	Props	Supports	Earthquake	Stage Settings
Analysis	Internal Stability	External Stability	Heave Failure	Dimensioning		

- Standards and analysis methods

- Theory for analysis in the program "**Sheeting Check**":

Stress in Soil Body	Earth Pressures	Sheeting Check	Dimensioning of Concrete Structures	Dimensioning of Steel Cross-Sections
---------------------	-----------------	----------------	-------------------------------------	--------------------------------------

- Outputs

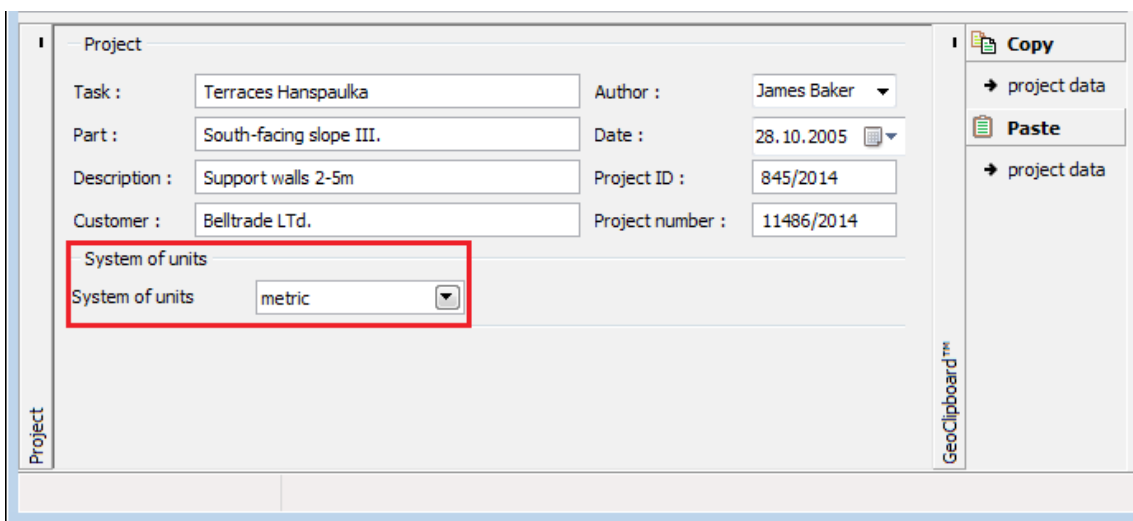
- General information about the work in the **User Environment** of GEO5 programs

- Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".



Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings**, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

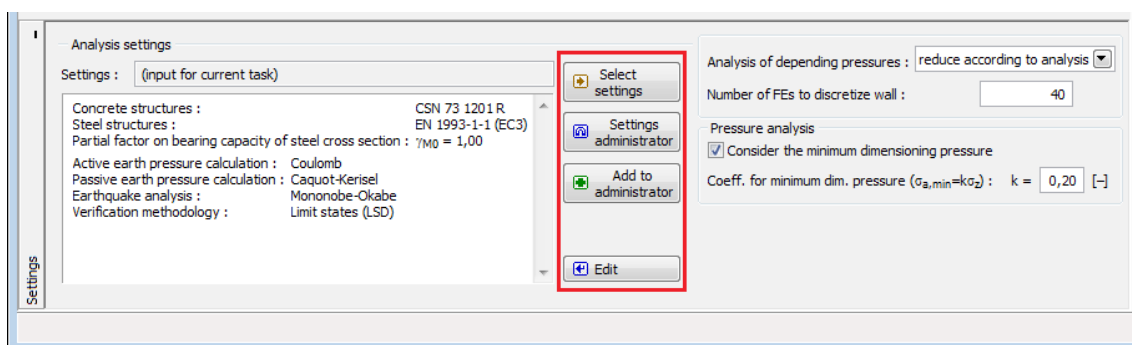
The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Materials and standards"** and the **"Excavations"** tabs.

When performing analysis according to **EN 1997** or according to the theory of **limit states**, the program enables to set whether to reduce the soil parameters for the calculation of limit pressures. When modeling a real behavior of the structure we recommend not to reduce these pressures.

The frame allows the user to specify subdivision of a wall in to finite elements (by default, the number of elements equals *40*) and the specify whether the structure is loaded by the **minimum dimensioning pressure**.



Frame "Settings"

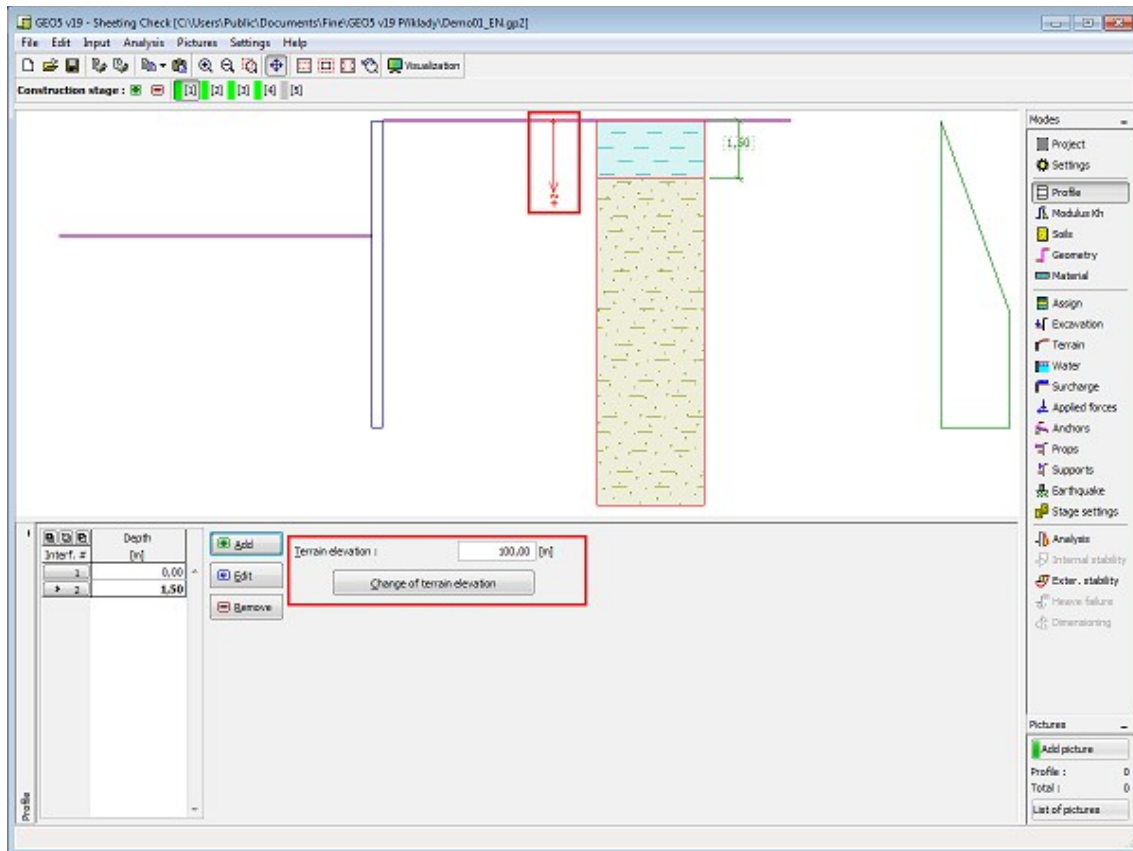
## Profile

The **"Profile"** frame contains a **table** with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of **active dimensions**.

**Adding** (editing) layer is performed in the **"New interface"** dialog window. The *z*-coordinate measured from the top point of a structure is specified (*z*-axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

## Modulus Kh

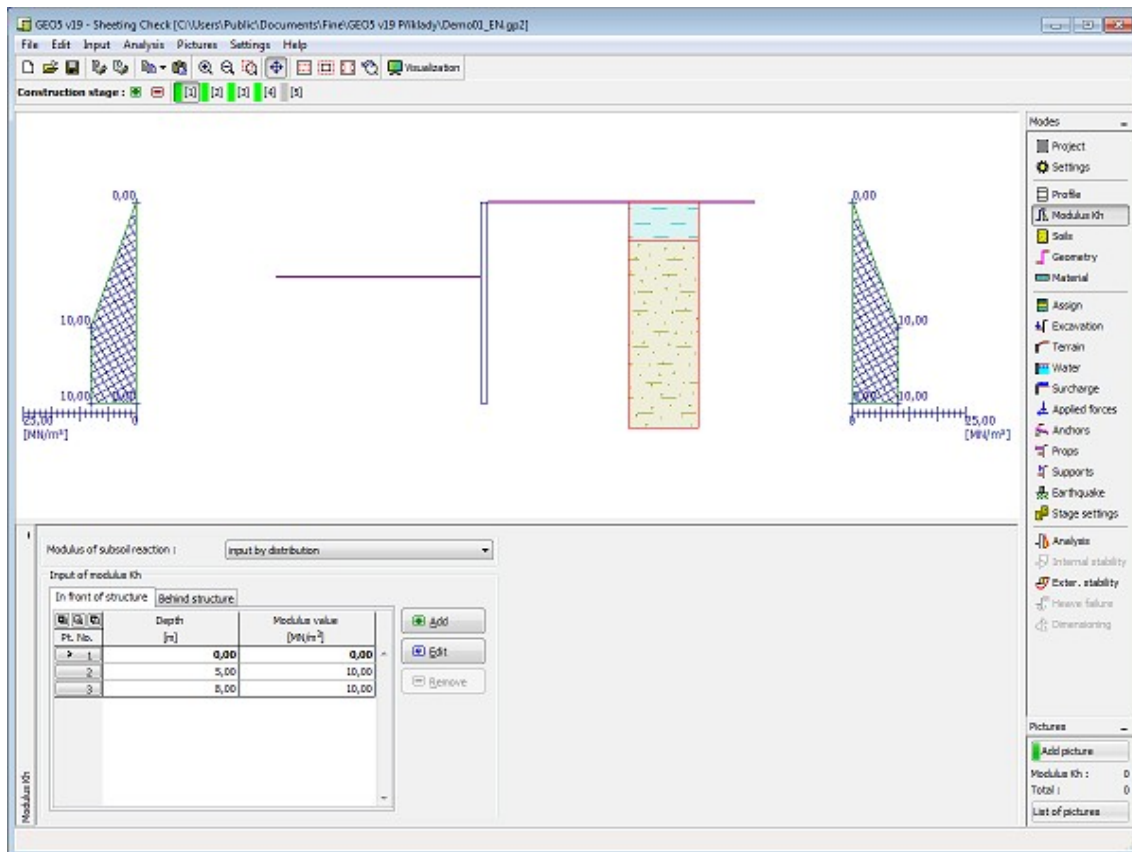
This **frame** serves to specify a type of analysis for calculation of the **modulus of subsoil reaction**, which is an important input parameter when analyzing a **sheeting structure** by using the **method of dependent pressures**.

The way of calculation of the **modulus of subsoil reaction**  $k_h$  is selected in the "**Settings**" frame (in the "**Edit current settings**" dialog window in the "**Excavations**" tab).

The frame can take different forms depending on the selected method of calculation:

- **standard** (option "analyze – **Schmitt**", "analyze – **Chadeisson**", "manual **iteration**" or "automatic **iteration**")
- **input** (selecting the option "**Input by distribution**" opens a table in the frame that allows to **input** the values of the modulus of subsoil reaction  $k_h$  both in front of and behind the structure. For option "**Input as a soil parameter**" the modulus  $k_h$  is specified in the "**Soils**" frame, where the modulus of subsoil reaction is considered either as **linear**, or as **nonlinear - curve**)

- **pressiometer PMT** (modulus of subsoil reaction  $k_h$  is input either by **pressiometric test**, or as a parameter of soil in the "Soils" frame. Then there is specified method of calculation - according to **NF P 94-282** or according to **Menard**)
- **chinese standards** (for "m" method is defined **horizontal displacement** at the ditch bottom  $v_b$  [mm] and magnitude of modulus  $A$  [MN/m<sup>3</sup>], or option **input** as a parameter of soil – "c" method, "k" or "m" method)



Frame "Modulus  $k_h$ "

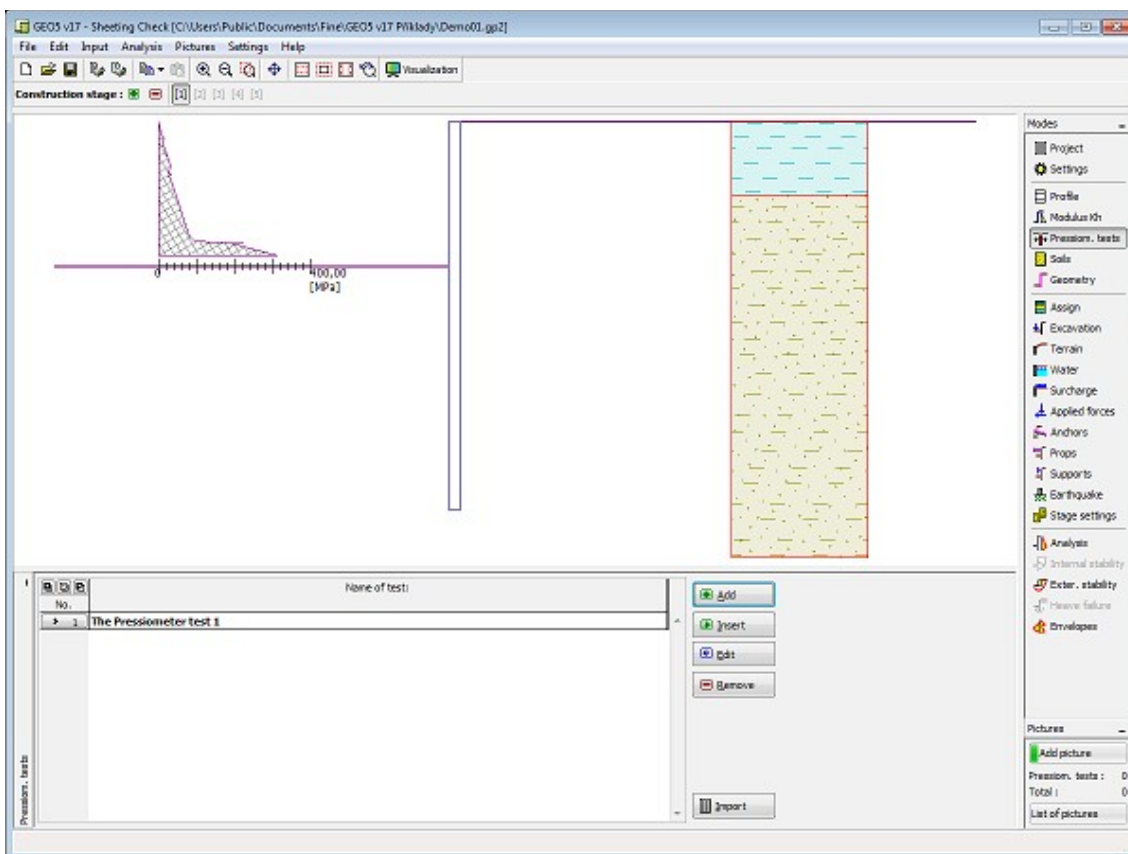
## Pressiometric Tests (PMT)

The **frame "Pressiometric tests"** serves to input the name of **pressiometric test** and depth of the first point of PMT  $h$  [m] from the finished grade.

The results of **pressiometric test (PMT)** are imported into the program by pressing the **"Import"** button. The procedure of an import of table data is more described **herein**.

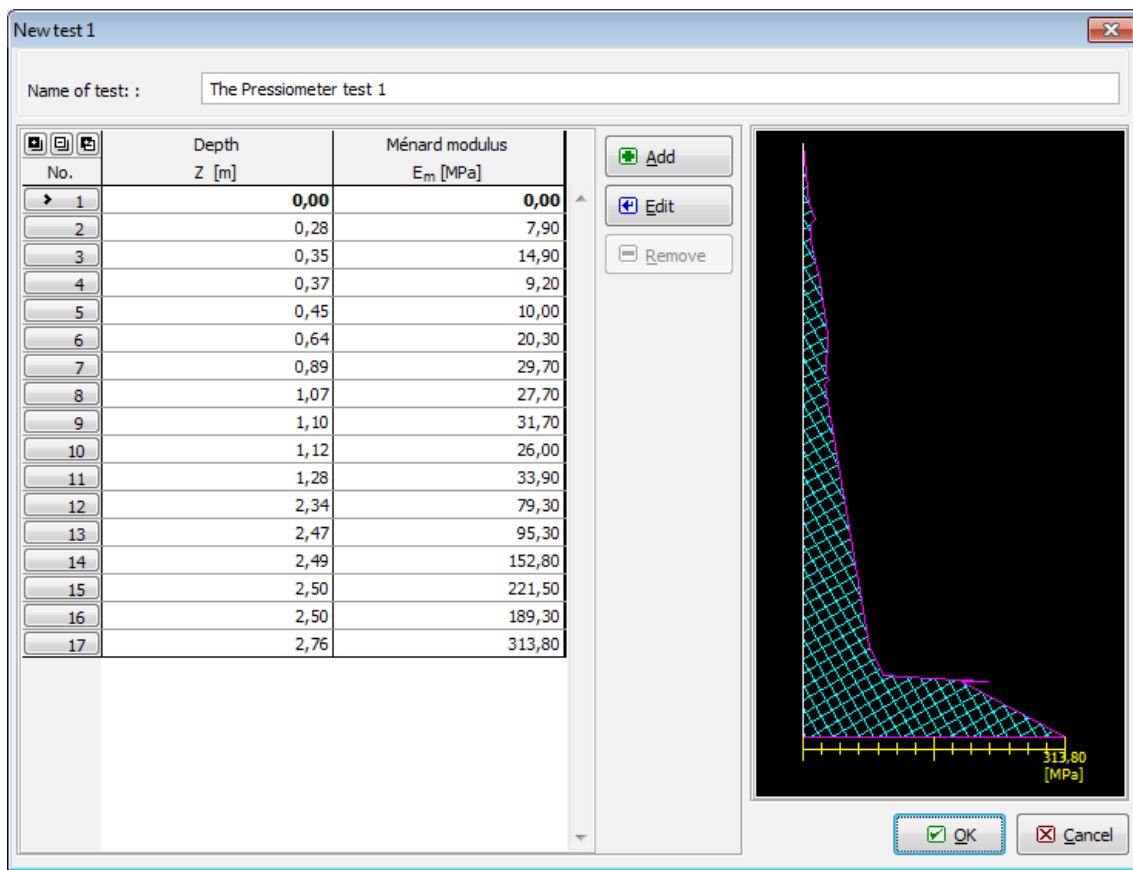
This frame contains a **table** with list of the input values of **pressiometric test (PMT)**. **Adding (editing)** of values is performed in the **"New value of test, or Edit the value of test" dialog window**, where is specified the depth  $z$  [m] (in the  $x, y$  coordinate system) and the measured values of **pressiometric (Menard) modulus  $E_m$  [MPa]**.

**Note:** The frame is accessible only in the case, when in the "Settings" frame is selected an option **"pressiometer PMT"** (the "Excavations" tab) and together for the **modulus of subsoil reaction** is chosen the option **"input pressiometer test"** in the "Modulus  $k_h$ " frame.



Frame "Pressiometric tests"





Dialog window "New test"

## Dilatometric Tests (DMT)

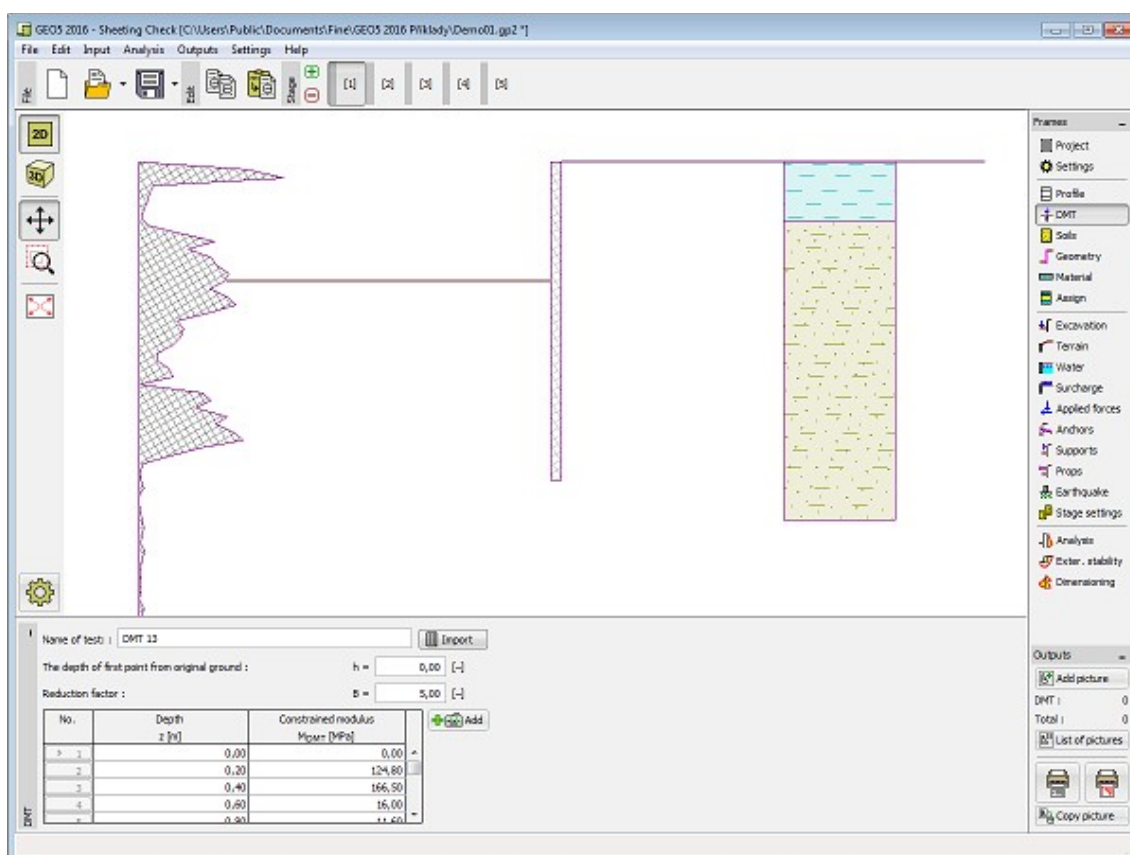
The frame "DMT" serves to input the name of dilatometric test, depth of the first point of DMT  $h$  [m] from the finished grade and characteristic length of sheeting structure (coefficient of reduction)  $B$ .

The results of dilatometric test (DMT) are imported into the program by inserting the file in format **UNI** (\*.uni). It is a **standardized and universal format** for import of the measured data obtained from dilatometric tests, which is used in the world.

This frame contains a table with list of the input values of dilatometric test (DMT). Adding (editing) of values is performed in the "New value of test, or Edit the value of test" dialog window, where is specified the depth  $z$  [m] and the measured values of constrained soil modulus  $M_{DMT}$  [MPa].

If during the evaluation of dilatometric test is measured the zero value of constrained soil modulus  $M_{DMT}$ , then program allows the automatic correction of measurement errors - instead of zero value is considered the arithmetic average of the next upper and lower non-zero value of  $M_{DMT}$  in the calculation.

**Note:** The frame is accessible only in the case, when in the "Settings" frame is selected an option "dilatometer DMT" (the "Excavations" tab) for calculation of the modulus of subsoil reaction.



Frame "Dilatometric tests (DMT)"

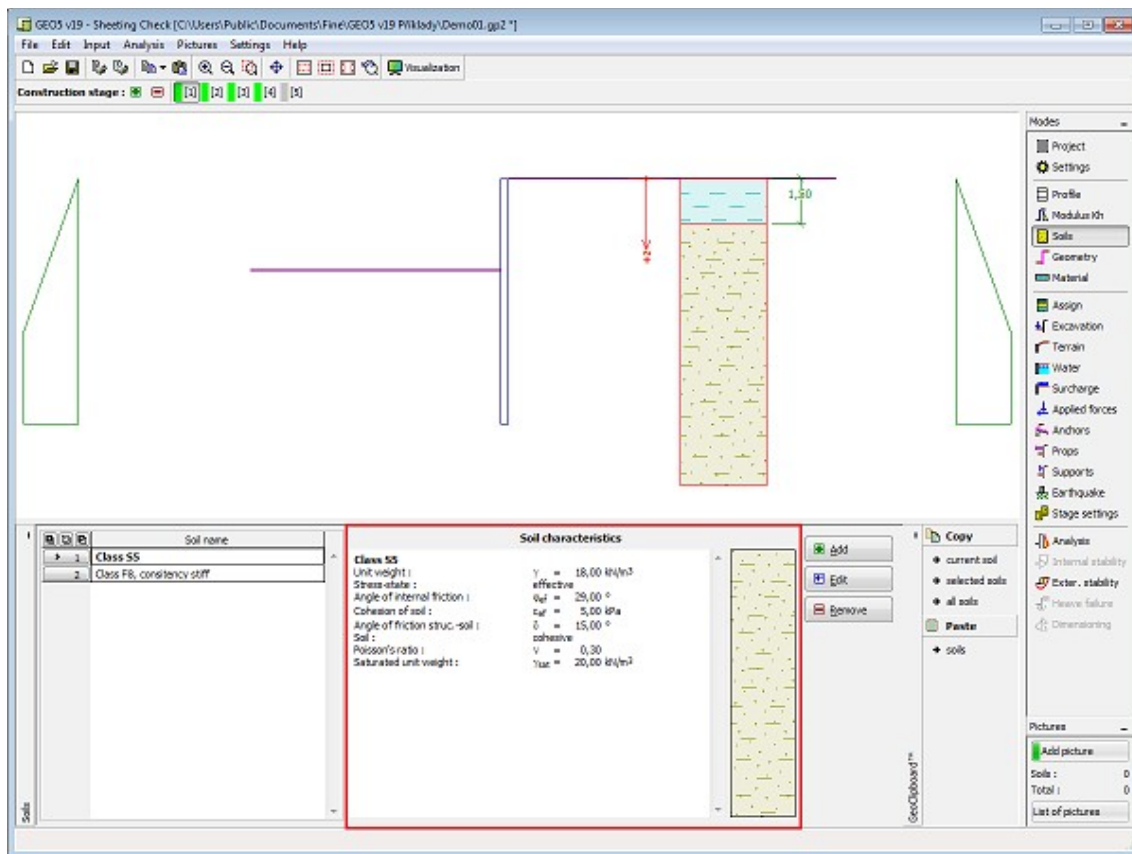
## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest", "Uplift pressure" and "Modulus of subsoil reaction".

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

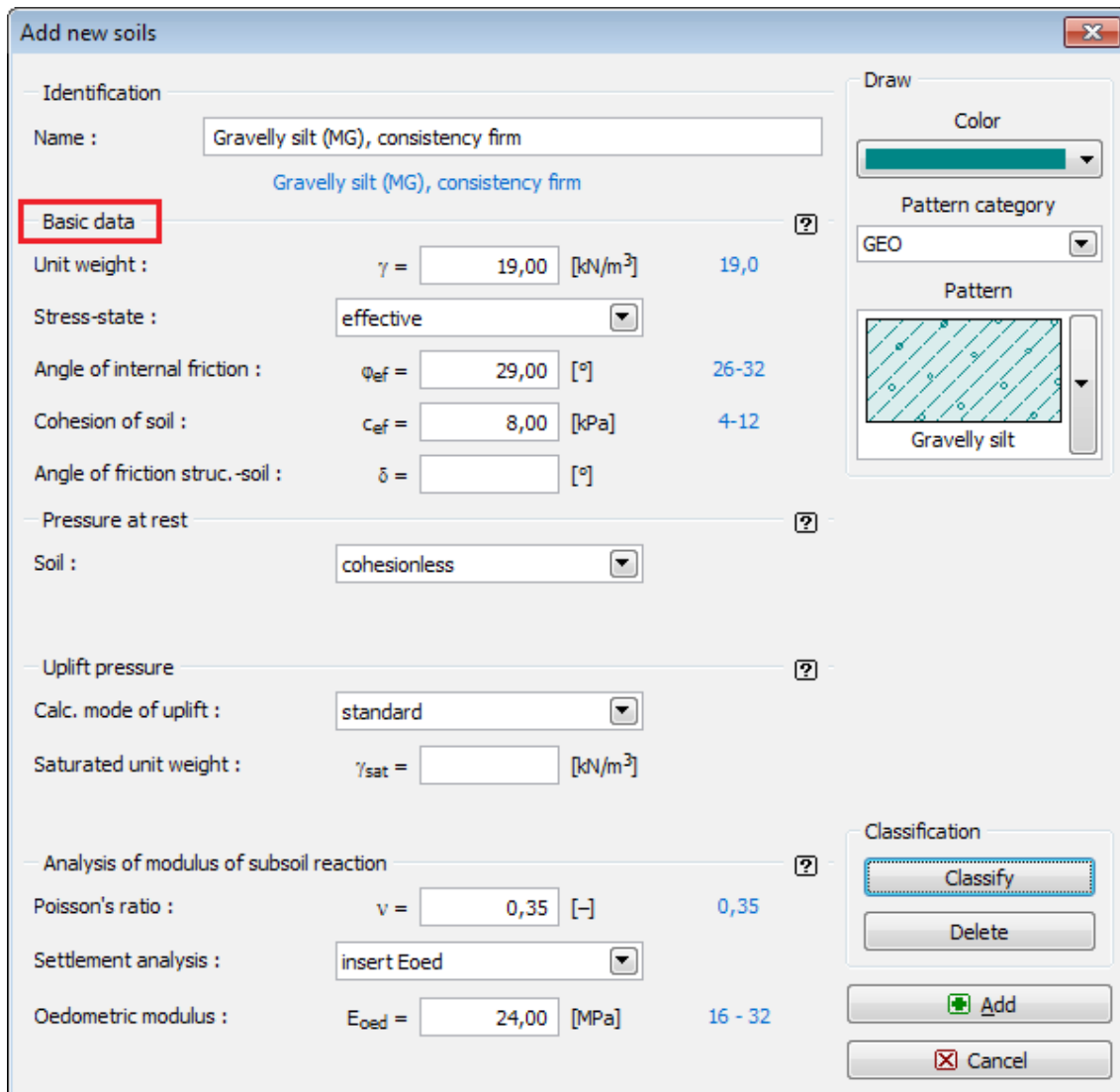
This part of the window allows to introduce basic parameters of the soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **soils database**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually. Approximate parameters of rocks are presented in the **theoretical section**.

Either **effective** or **total** parameters of the angle of internal friction and cohesion are specified depending on the settings in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil and load, structure duration and water conditions.

For **effective stress**, it is further needed to specify the **angle of internal friction between the soil and the structure**, which depends on the structure material and type of soil. Possible values of this parameter are listed in the **table of recommended values**.

For **total stress**, it is further needed to specify the **adhesion of soil to the structure face**  $a$ .

The associated theory is described in detail in the chapter "**Earth pressures**".



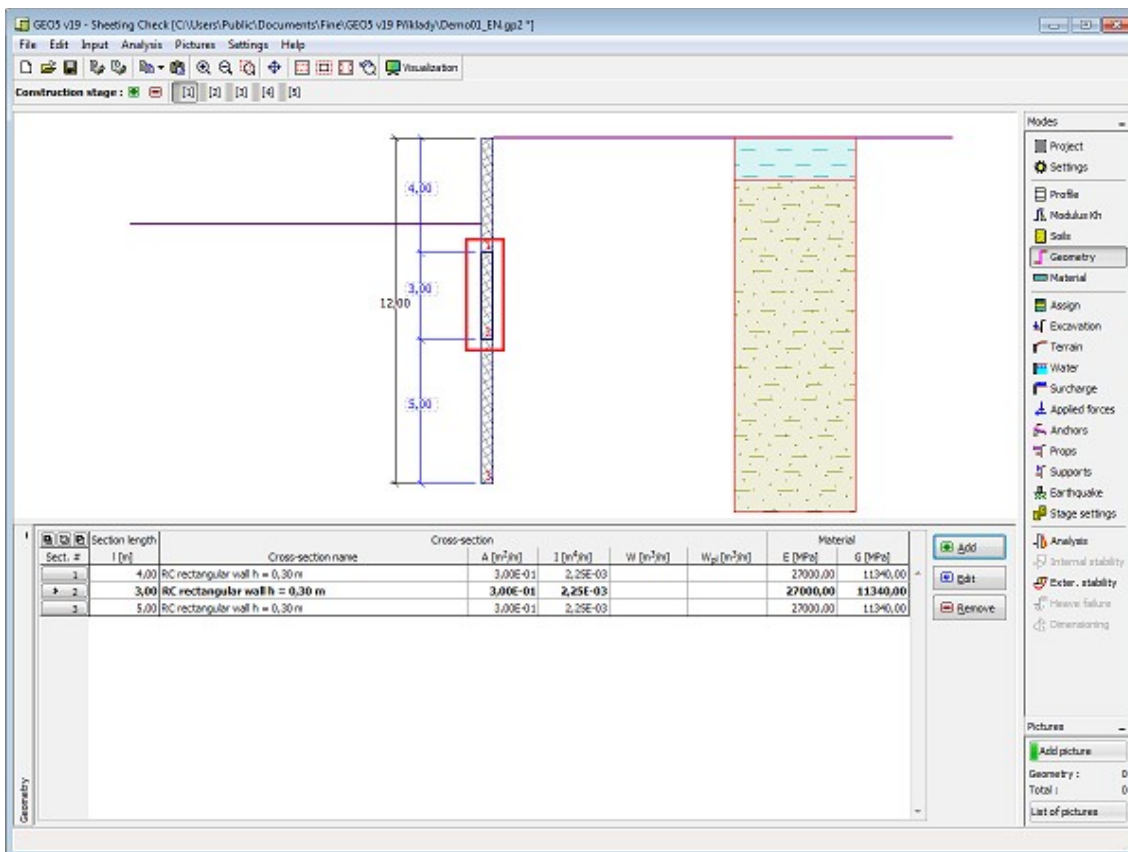
Dialog window "Add new soils" - "Basic data"

## Geometry

The "Geometry" frame contains a table with a list of input structural sections forming the sheeting structure. For each section the table stores its cross-sectional characteristics ( $A$  - area,  $I$  - Moment of inertia - these variables are always expressed with respect to 1 m run of structure length) and material characteristics ( $E$  - Modulus of elasticity,  $G$  - Shear modulus).

Adding (editing) sections is performed in the "New section (Edit section)" dialog window.

The input sections can be further edited on the desktop with the help of active objects - double-click on a structure opens a dialog window with a given section.



Frame "Geometry"

## Adding and Editing Section

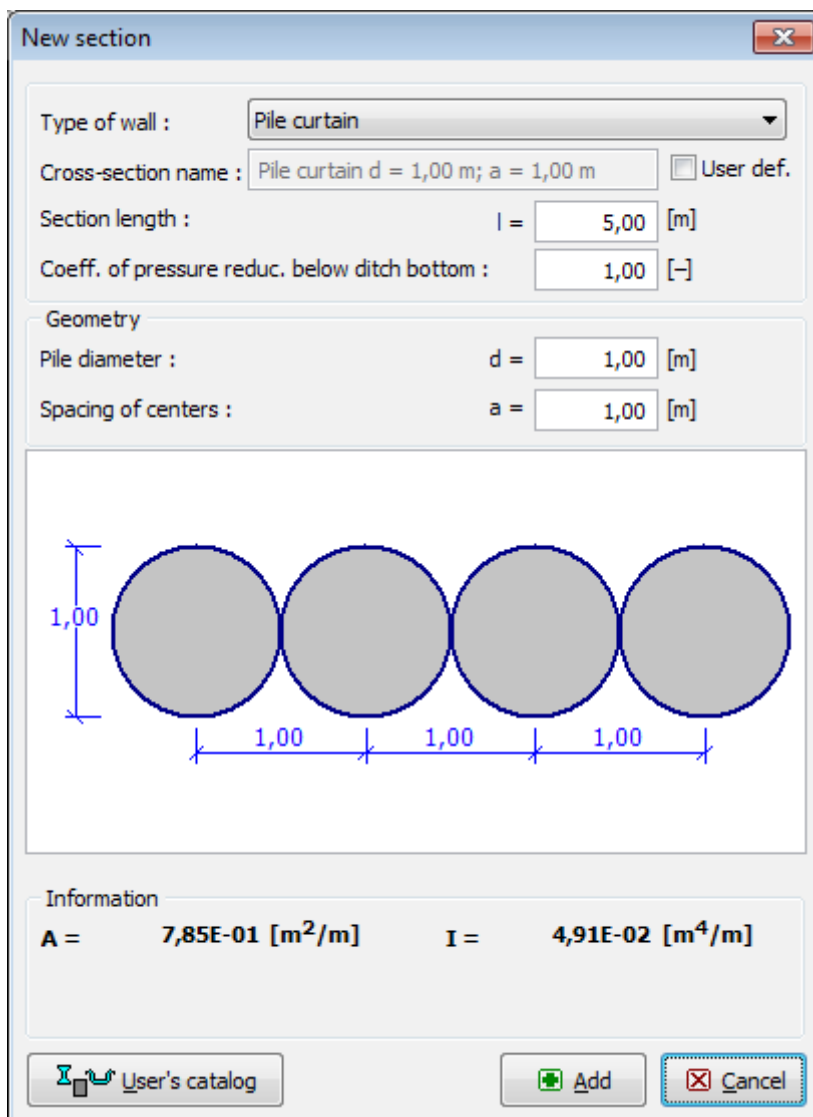
The "New section or Edit section" dialog window contains the following items:

- Type of wall**
  - combo list contains individual structural types of **sheeting walls** (pile curtain, reinforced concrete rectangular wall, sheet pile wall, steel I cross-section, or user input of cross-sectional characteristics)
- Section length l [m]**
  - use input field to specify length of a given section of sheeting structure
- Coefficient of pressure reduction below ditch bottom**
  - this **coefficient** is possible to **input** or **calculate automatically** for pile curtain, steel I cross-section or for user input of **A, I, E, G** (for **braced sheeting** this coefficient equals to 1.0)
- Geometry**
  - contains information about geometry for selected structural variant. For **pile curtain** is defined diameter of piles **d [m]**, spacing of piles **a [m]** and type of cross-section (square, circle). For **reinforced concrete rectangular wall** is defined its thickness **h [m]**. For **steel I cross-section** is defined spacing of profiles **a [m]**. For **user input of A, I, E, G** and option "**Dimension of steel cross-section**" is moreover defined cross-sectional modulus **W**
- Profile**
  - contains information about profile for selected structural variant "**Steel I-section**" (button "**Catalog**" opens the "**Catalog of**

**Information**

- profiles"**, which contains a list with profile classes and individual profiles)
- contains overview of cross-sectional characteristics of the input cross-section ( $A, I, E, G$ ). Cross-section area  $A$ , moment of inertia  $I$  and cross-sectional modulus  $W$  are always evaluated for 1 m run of the sheeting structure length

The "**User catalog**" button in the bottom part of the window opens the "**User catalog**" dialog window.



*Dialog window "New section"*

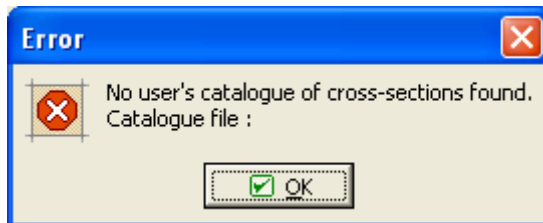
**User's Catalog**

The user catalog allows the user to define and store own cross-sections and their characteristics that appear in a **sheeting structure**. At first use of the catalog (has not been yet created) the program prompts a warning message that no catalog was found. Then, pressing the button "**OK**" opens the "**Save as**" **dialog window** that allows to enter the catalog name and

saving it into a specified location by pressing the "Save" button (by default a folder used for saving the project data is assumed).

The program allows the user to create more than one catalog. The next catalog is created by pressing the "New" button - the program asks, whether the current catalog should be replaced (**the currently loaded catalog is not deleted!**) and saves the new catalog under a new name. The "Open" button allows for load an arbitrary user catalog and by pressing the "Save as" button for saving it under a different name.

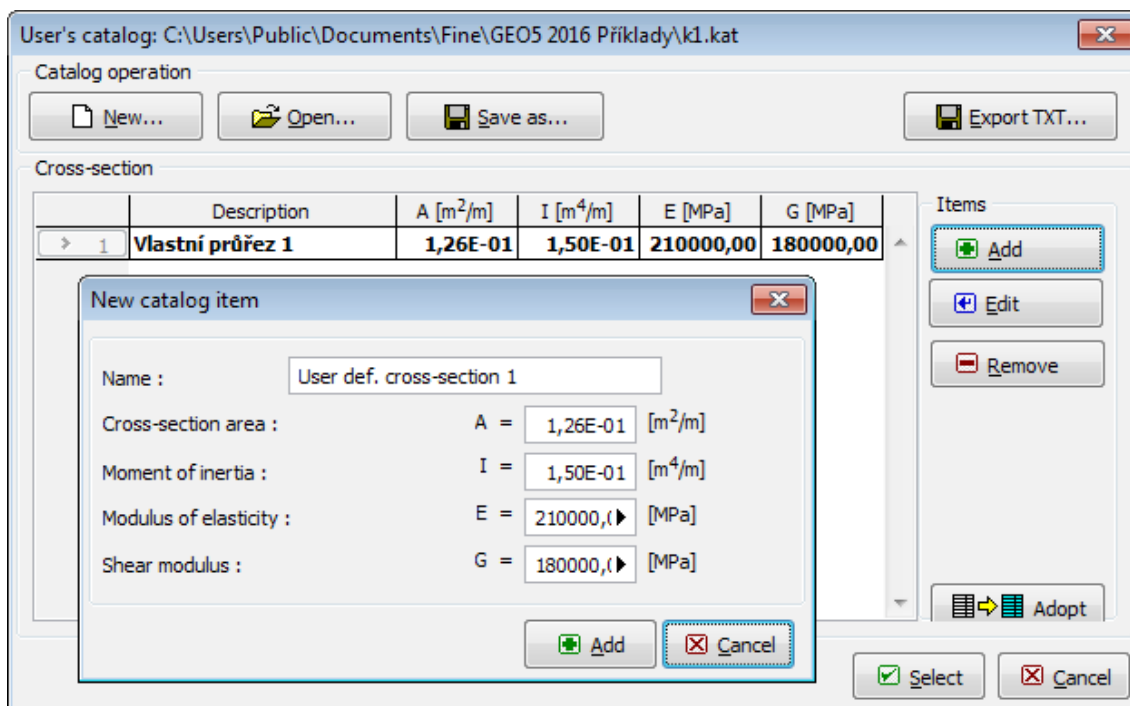
"Export TXT" button allows for exporting of currently loaded user catalog to text file.



Dialog window at first use - user catalog of cross-sections

The "User's catalog" dialog window contains a table listing the user defined cross-sections. The "Add item" button opens the "New catalog item" dialog window that allows for specifying and subsequent saving of characteristics of a new cross-section into the catalog. Buttons "Edit item" and "Remove item" serve to edit individual items in the table.

The "Accept current" button accepts the current cross-sectional characteristics of a cross-section specified in the "New section" dialog window and opens the "New catalog item" dialog window that allows for modifying and saving the current cross-section.



Dialog windows "User's catalog" and "New catalog item"



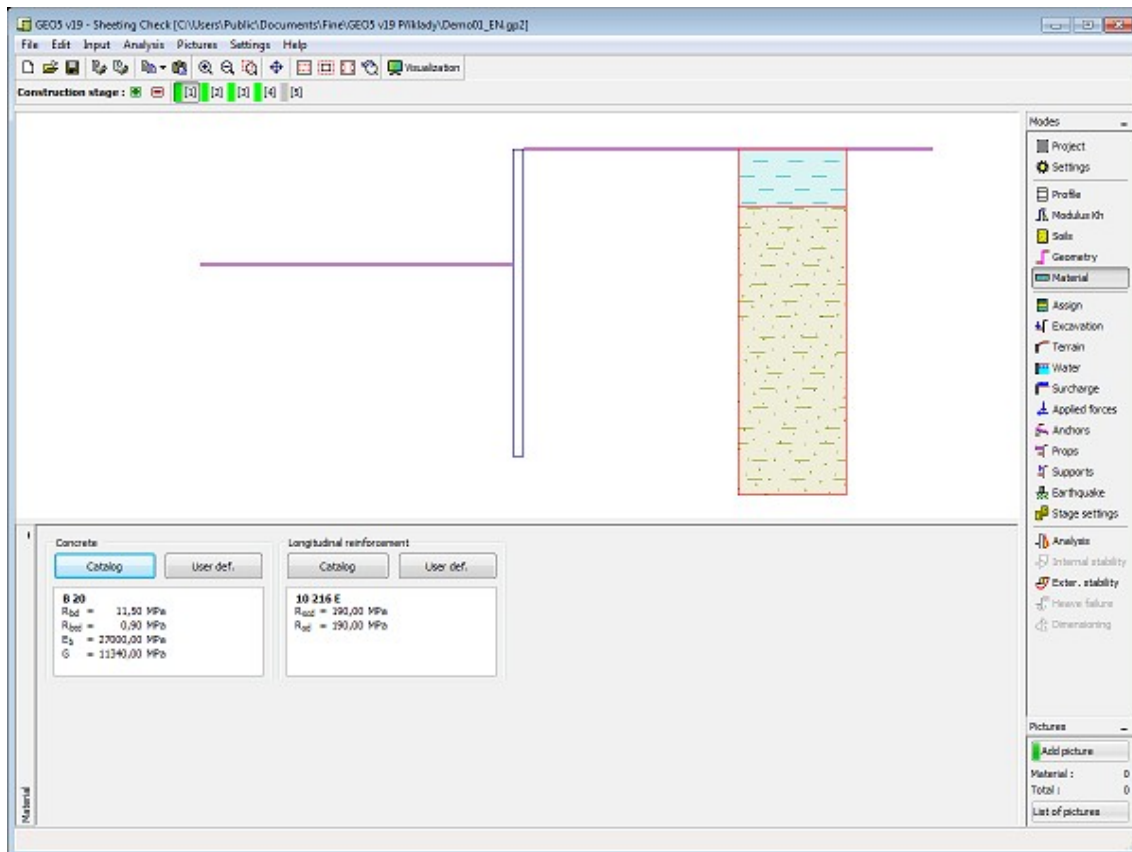
## Material

The frame **"Material"** allows to select material parameters of used concrete, longitudinal reinforcement or structural steel.

Two options are available when selecting the material type:

- The **"Catalog"** button opens the **"Catalog of materials"** dialog window (for concrete or steel reinforcement), the list of materials then allows to select the required material.
- The **"User defined"** button opens the **"Editor of material - Concrete"** dialog window (for concrete) or the **"Editor of material - Reinforcing steel bars"** dialog window (for longitudinal reinforcement) or the **"Editor of material - Structural steel"** dialog window (for structural steel), which allows to input the specification of material parameters manually by user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of **concrete** or **steel** structures set in the **"Materials and standards"** tab.



Frame "Material"

## Pressure Determination

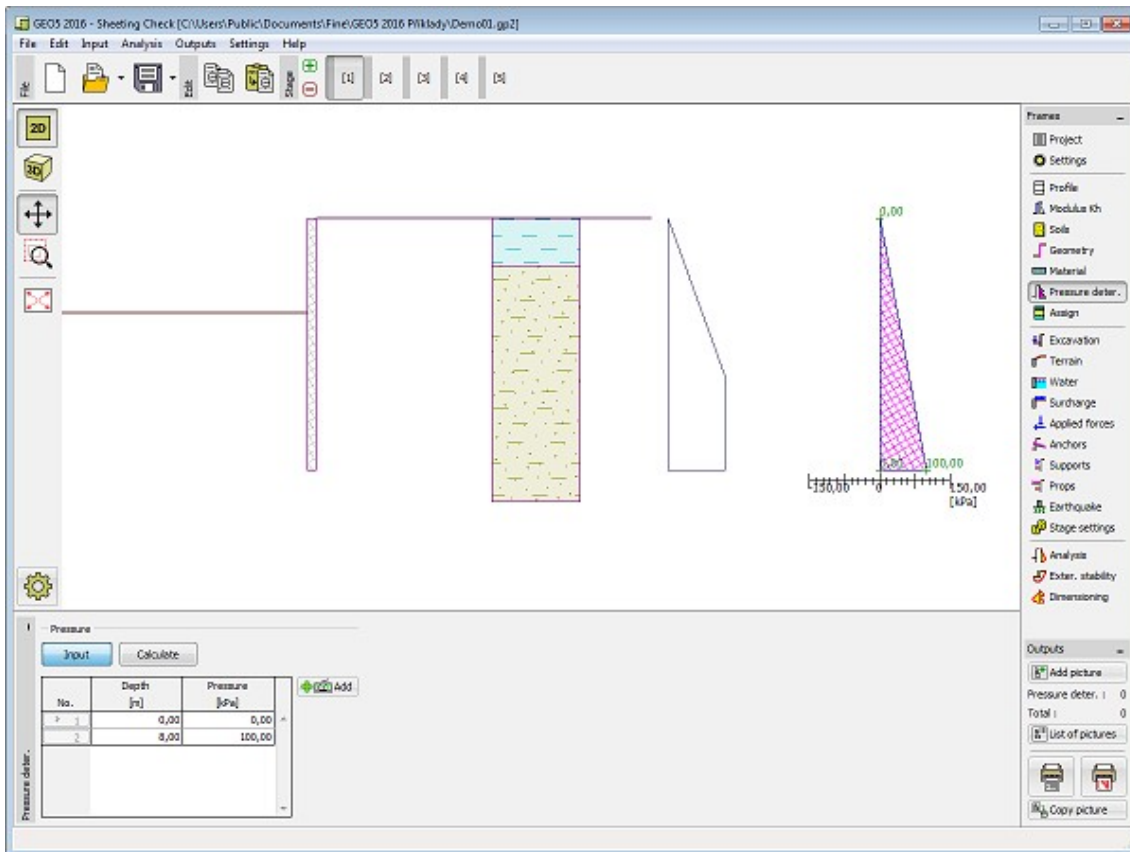
The frame **"Pressure determination"** allows to input the values of earth pressure behind **sheeting structure** manually (by pressing the **"Input"** button). The frame is accessible only in the case, when in the **"Settings"** frame is selected an option **"JGJ 120-2012"** (the **"Excavation"** tab).

This frame contains a **table** with list of the input values of **earth pressure**. **Adding** (editing) of



these values is performed in the **"New point, or Edit point"** dialog window by pressing the **"Add (Edit)"** button, where is specified the depth of point [m] and the value of earth pressure [kPa].

It is possible to calculate the values of **active earth pressure**, or **earth pressure at rest** automatically (by pressing the **"Calculate"** button).



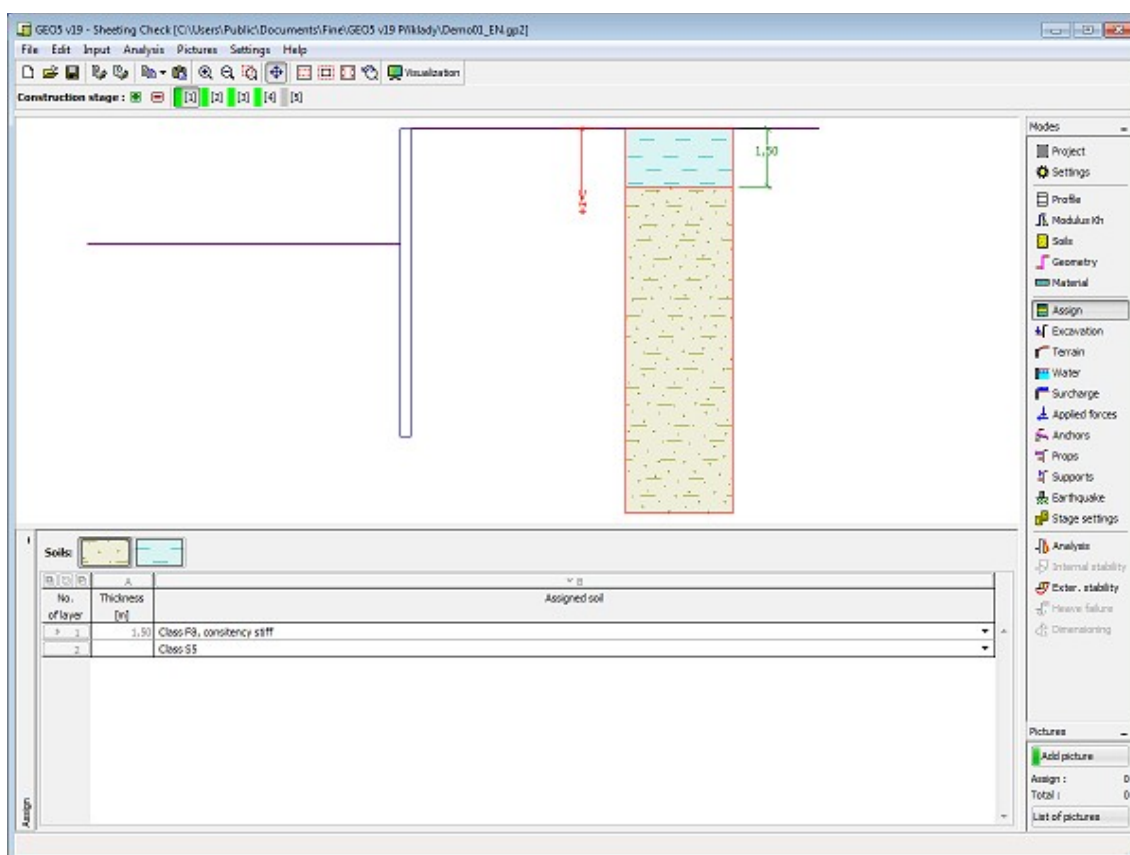
Frame "Pressure determiaterial"

## Assign

The **"Assign"** frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to **import soil assignment in the gINT format**.



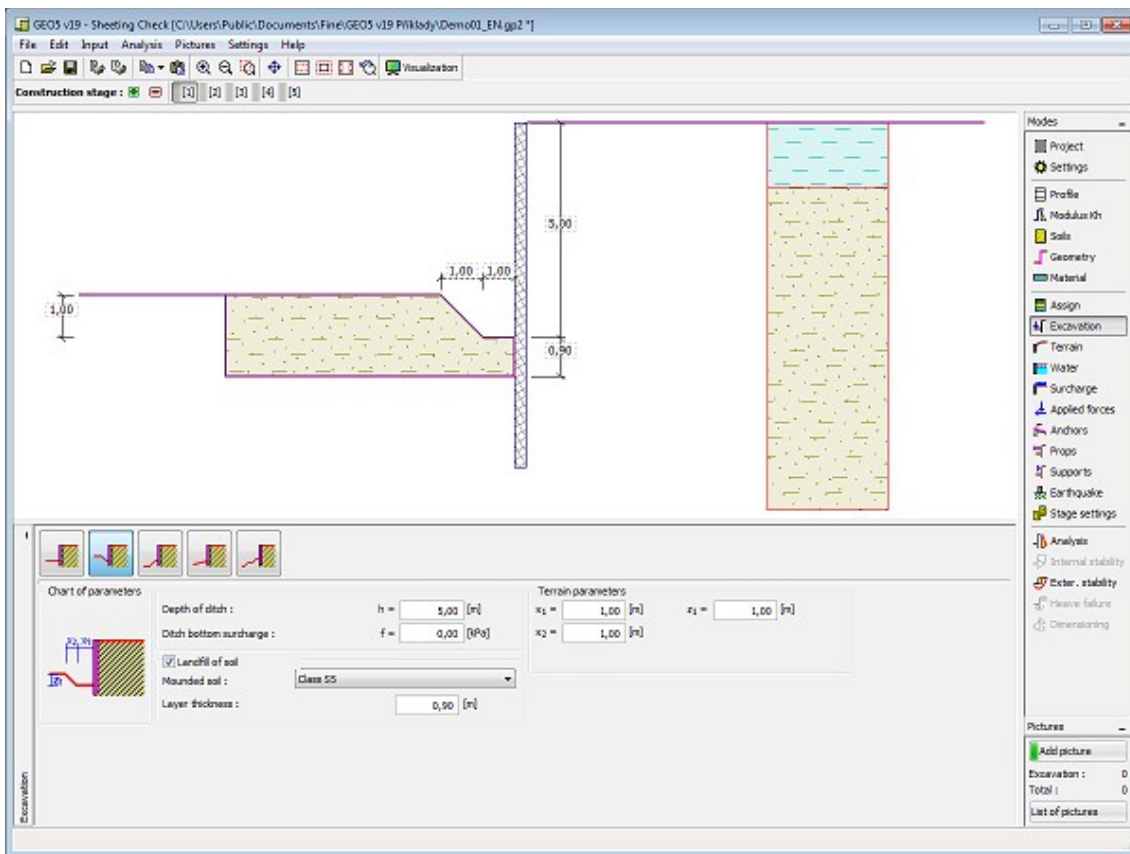
Frame "Assign"

## Excavation

The "**Excavation**" frame serves to input the depth of a construction ditch  $h$  [m] and by pressing the button to select the shape of the ditch base in front of **sheeting structure**. The selected shape with a graphic hint "**Chart of parameters**" appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The frame also allows to specify ditch bottom **surcharge** or a thickness of layer of landfill of soil below the ditch bottom (the soil can be selected from a combo list containing soils input in the frame "**Soils**"). When introducing the landfill of soil with **braced sheeting** it is assumed that there is a sheeted structure in the location of landfill of soil, i.e., all pressures are acting on the entire width of a structure as above the construction ditch base.

In this frame it's possible to input **strengthening of the soil** at the heel of sheeting structure. The principle of calculation is described in more detail [herein](#).



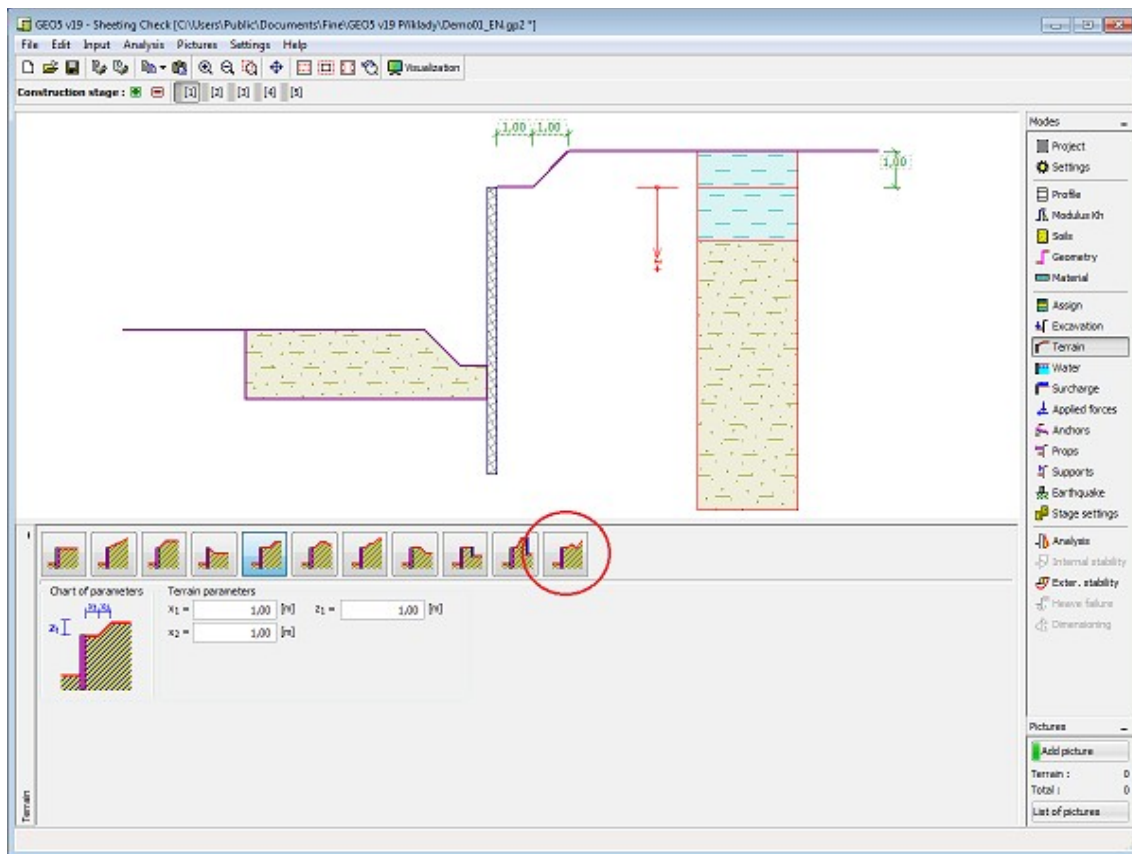
Frame "Excavation"

## Terrain

The "Terrain" frame allows to specify shape of the terrain by pressing the button. The selected shape with graphic hint ("Chart of parameters") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates  $[0,0]$  coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "Distribution of earth pressures for broken terrain".



Frame "Terrain"

## Water

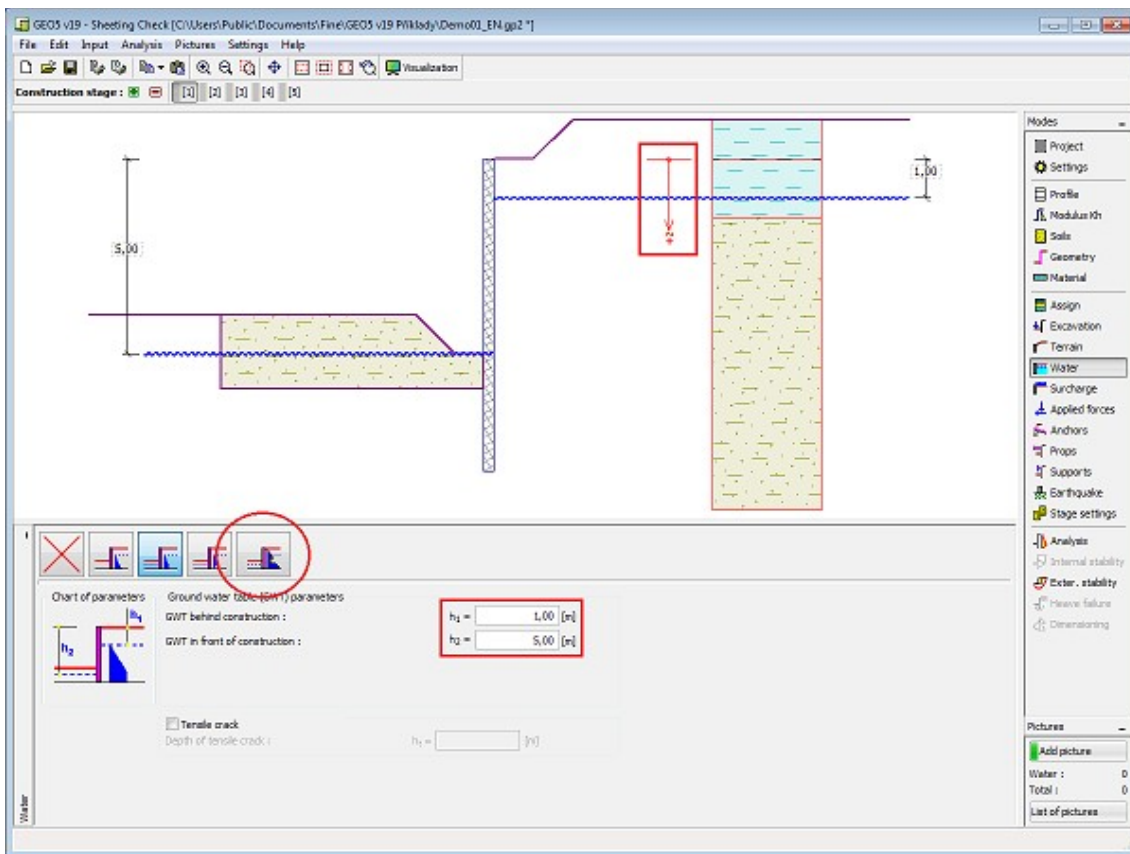
The "**Water**" frame allows to select the type of water by pressing the button. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Parameters of water ( $h_1$ ,  $h_2$  etc.) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option is a manual input of pore pressure both in front of and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with **tables**. The table is filled with values of pore pressure in front of, or behind the structure at a depth of "z" (z-axis).

The ground water table (GWT) can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".

The program further allows to specify a depth of **tensile crack**  $h_t$  [m] filled with water.



Frame "Water"

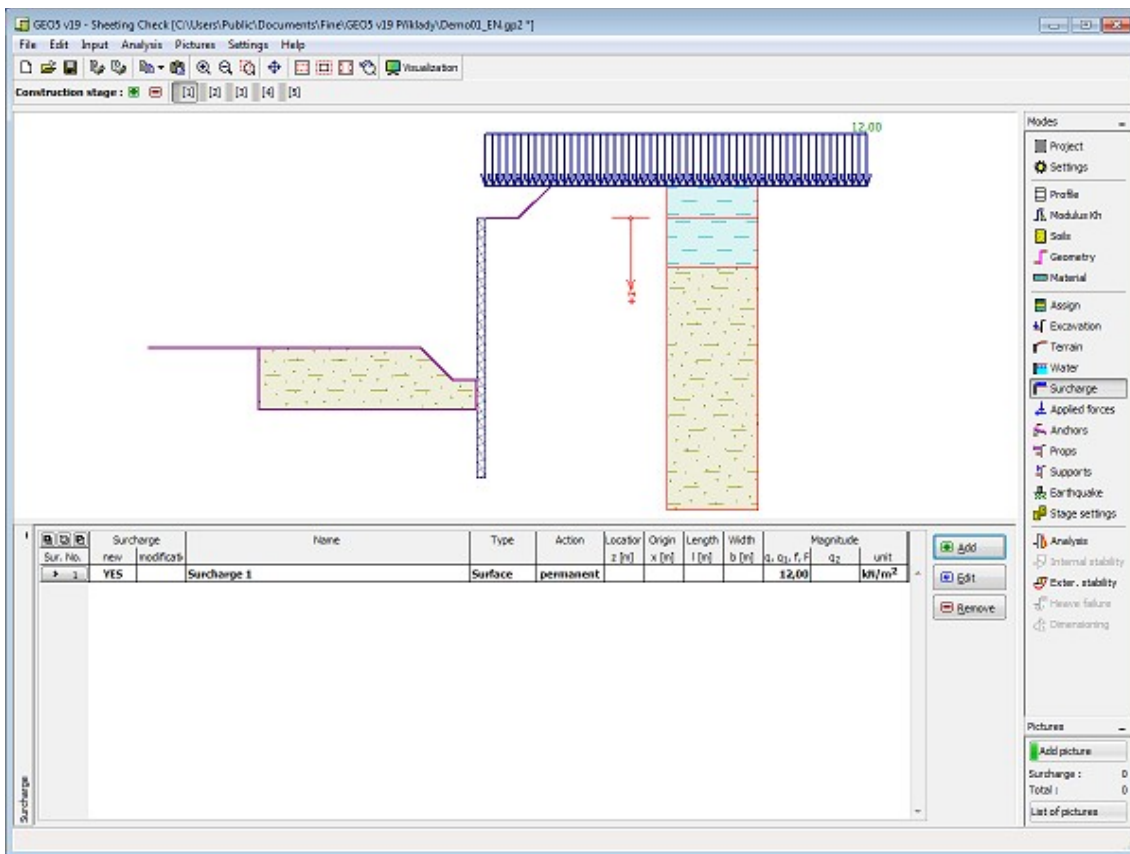
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action.

Analysis of earth pressures due to surcharge is described in the theoretical part of the help, chapter "**Influence of surcharge**".



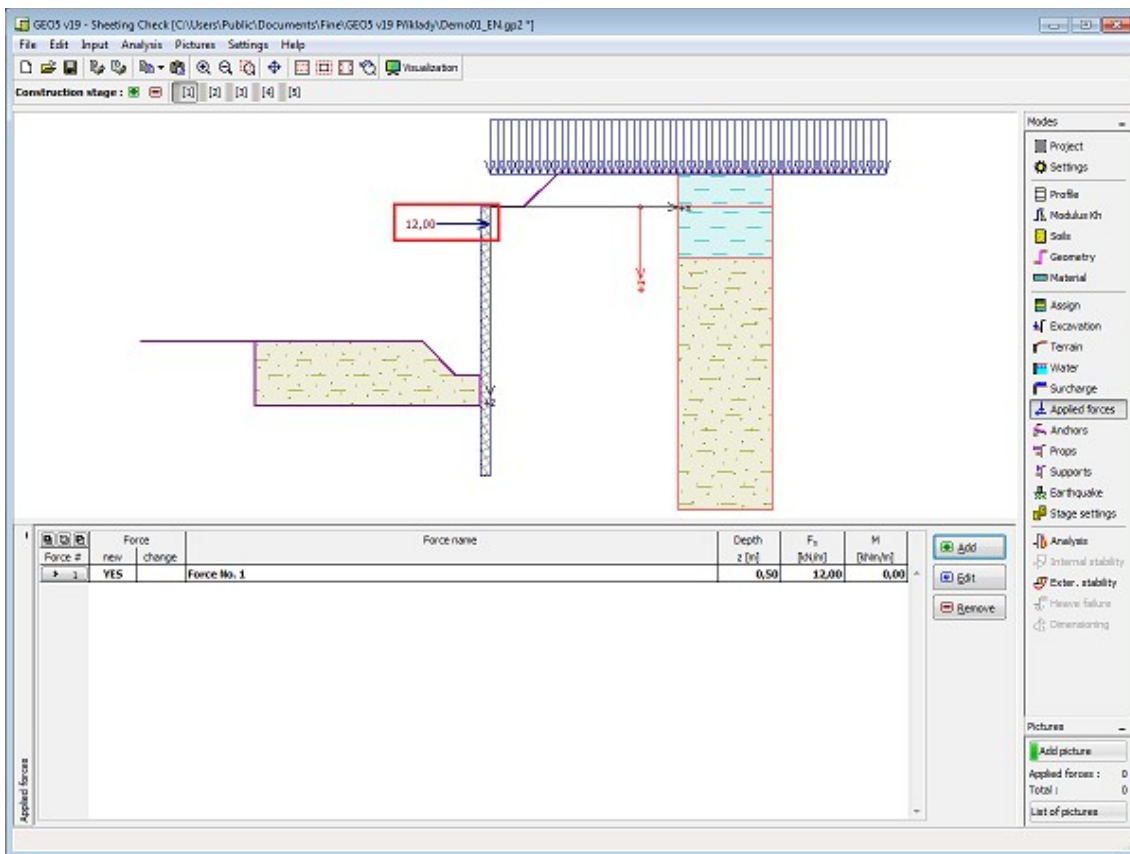
Frame "Surcharges"

## Applied Forces

The "Applied forces" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "New force" dialog window. The input forces can also be edited on the desktop with the help of active objects.

Applied forces represent an additional load on the structure of the wall, braced sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as a surcharge.



Frame "Applied forces"

## Anchors

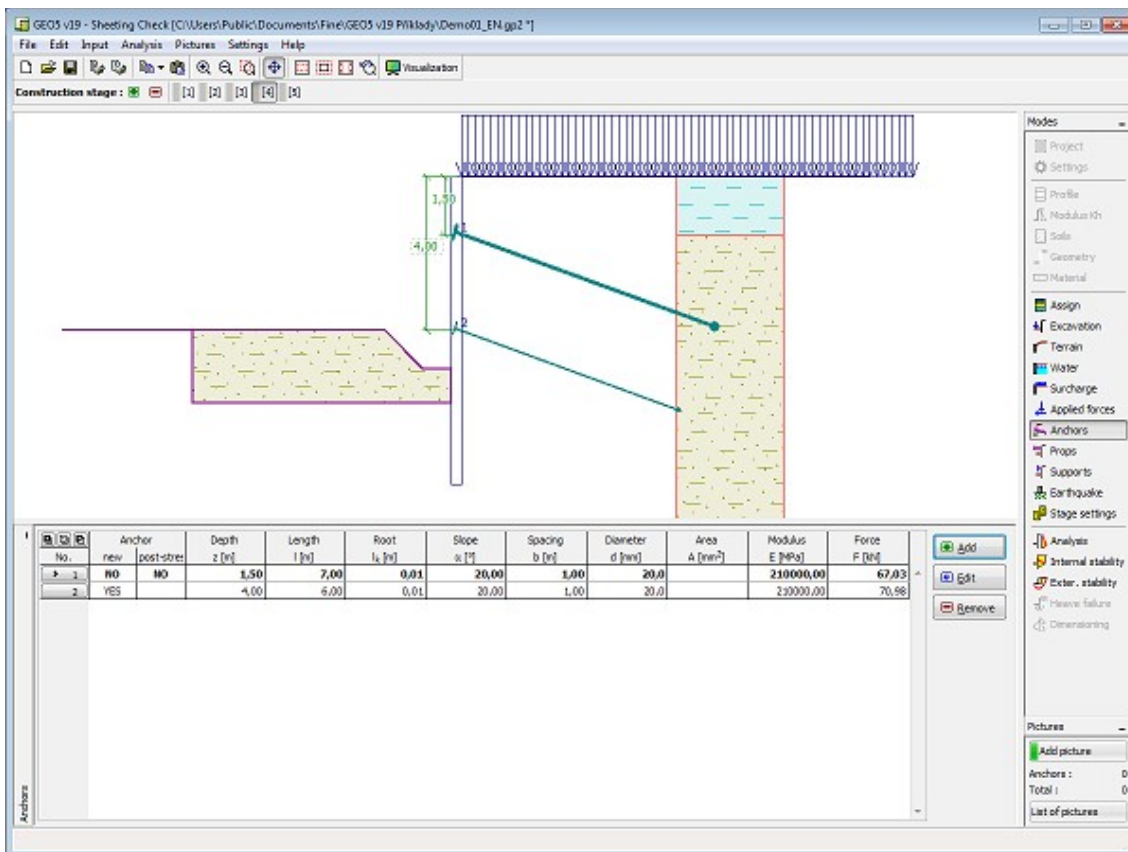
The "Anchors" frame contains a table with a list of input anchors. Adding (editing) anchors is performed in the "New anchor (Edit anchor)" dialog window by pressing the "Add" ("Edit", "Remove") button. The input anchors can be edited on the desktop with the help of active objects.

The support is automatically placed on already deformed structure (displacement is obtained from the previous construction stage).

The anchor stiffness becomes effective in subsequent stages of construction. Due to the displacement of the sheeting structure the forces in anchors are changing. In subsequent stages the anchor can no longer be edited, it's only possible to change post-stress normal force of anchors.

**Note:** The program doesn't check the anchor bearing capacity against tearing.





Frame "Anchors"

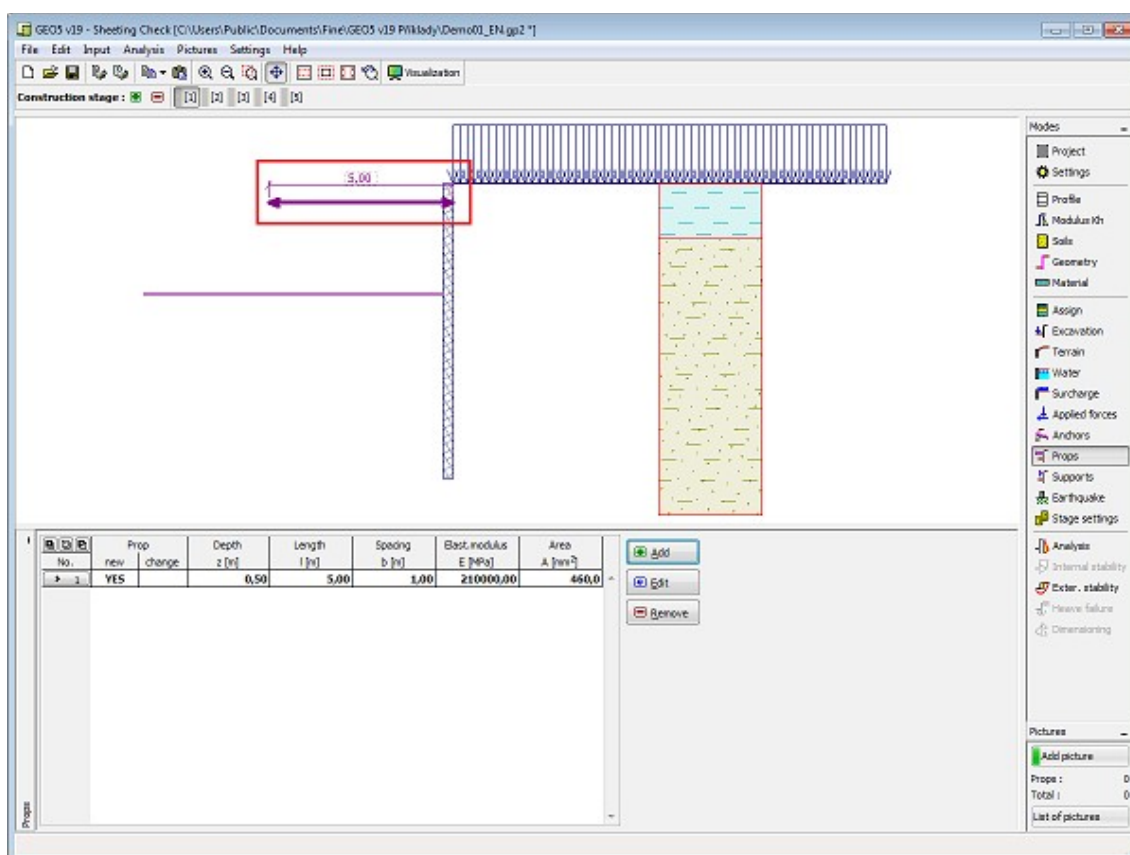
## Props

The "**Props**" frame contains a table with a list of input props. Adding (editing) props is performed in the "**New prop (Edit prop)**" dialog window by pressing the "Add" ("Edit", "Remove") button. The input props can also be edited on the desktop with the help of active dimensions or by active objects.

The prop is introduced on already **deformed structure** automatically (obtained from the previous stage of construction). In subsequent stages the props can no longer be edited, it's only possible to change stiffness of props. In the analysis props are modeled in the same way as anchors but with the initial force equal to zero.

**Note:** The program doesn't check the prop bearing capacity neither for compression or for buckling.



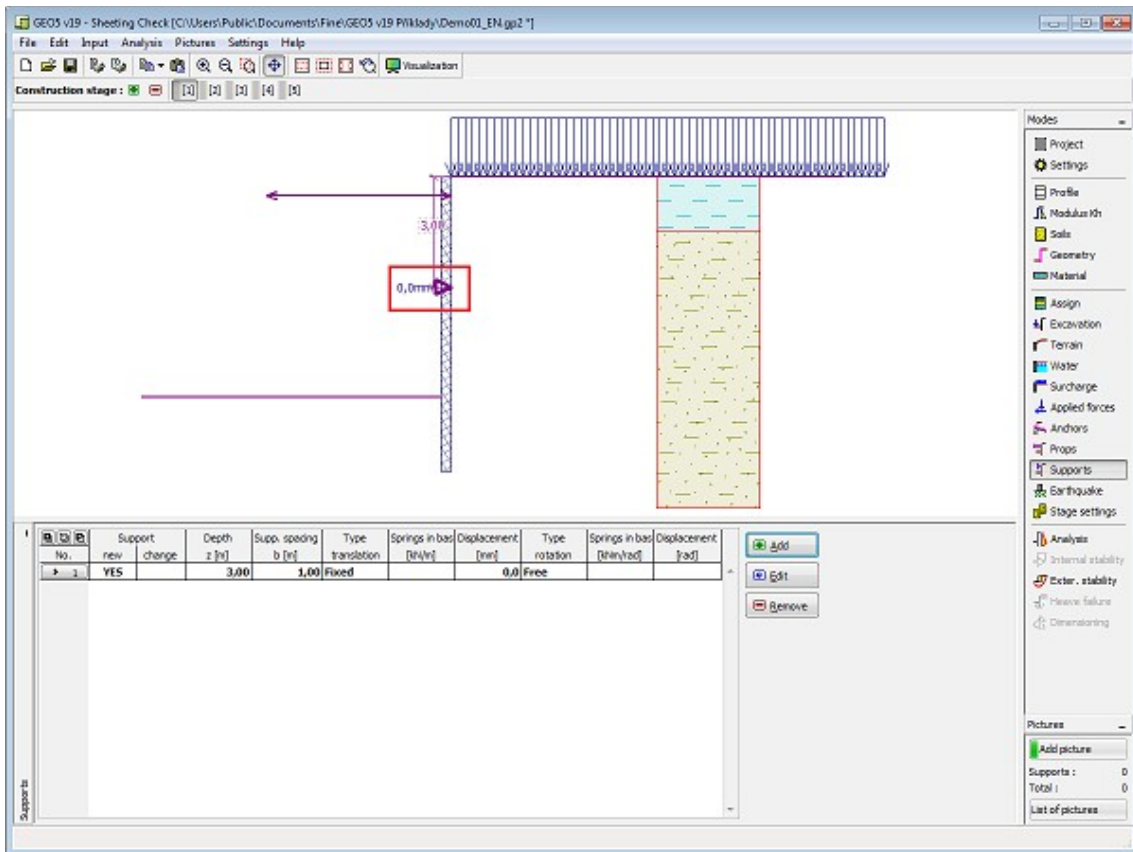


Frame "Props"

## Supports

The "**Supports**" frame contains a table with a list of input supports. Adding (editing) supports is performed in the "**New support (Edit support)**" dialog window. The input supports can also be edited on the desktop with the help of **active dimensions** or **active objects**, respectively.

The support is input on already **deformed structure** automatically (obtained from the previous **stage of construction**). In subsequent stages the supports can no longer be edited, it is only possible to input forced displacement of supports.



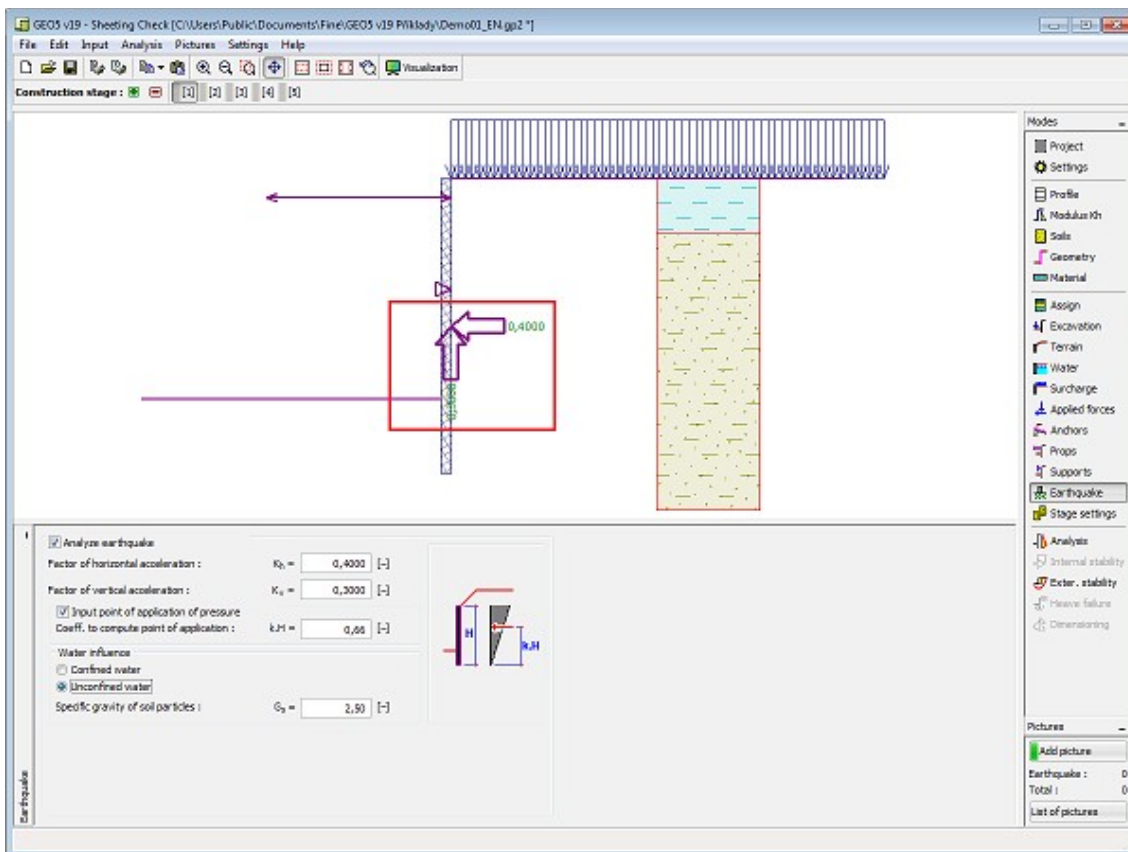
Frame "Supports"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help in chapter "Influence earthquake".



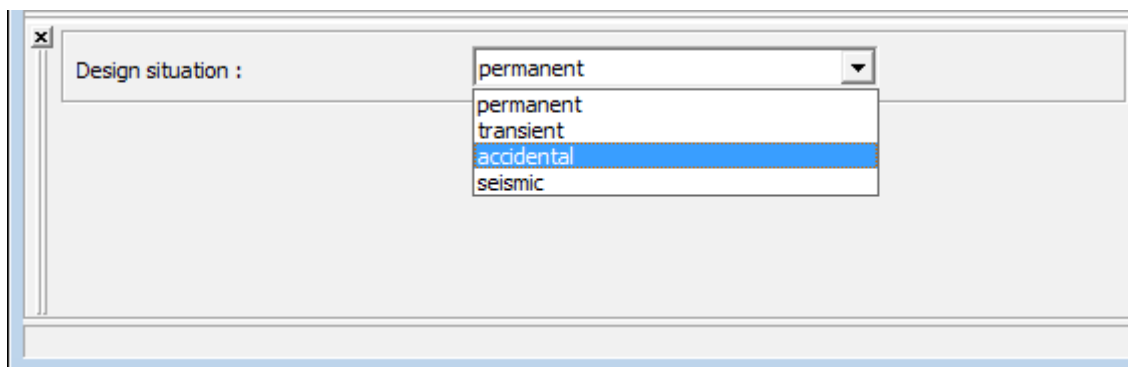
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Analysis

The frame "**Analysis**" displays the analysis results. Switching to this regime automatically runs the analysis. The frame contains three buttons to show the analysis results:

- **$K_h$  + earth pressures**

Variation of the **modulus of subsoil reaction** is displayed in the left part of the desktop (by default a blue color with hatching) is assumed. Referring to the **method of depending pressures** some of the springs (values of modules of subsoil reaction) are removed (spring stiffness set equal to zero) from the analysis. The analysis **may fail to converge** providing the critical (limit) state developed both in front and behind the structure and there is not enough constrains available (anchors, supports). The program exists without finding a solution. An error message appears in the bottom part of the frame - such a case calls for **modification in problem input** - e.g., add an anchor, change a depth of excavation, improve soil parameters, etc.

Some construction stages display (by default a yellow dotted line is assumed) deformation at the onset of mobilization of the **earth pressure at rest** - this is a complementary information showing plastic deformation of a structure.

Distributions of limiting pressures (by default a green dashed line is assumed) are presented in the right part of the window (**passive pressure**, **pressure at rest** and **active pressure**). The **actual pressure acting on a structure** is plotted in a solid blue line.

Both **deformed** (by default a solid red color is assumed) and undeformed structure appears in the right part of the desktop. Forces and displacements developed in anchors, supports and props are also shown.

- **Internal forces**

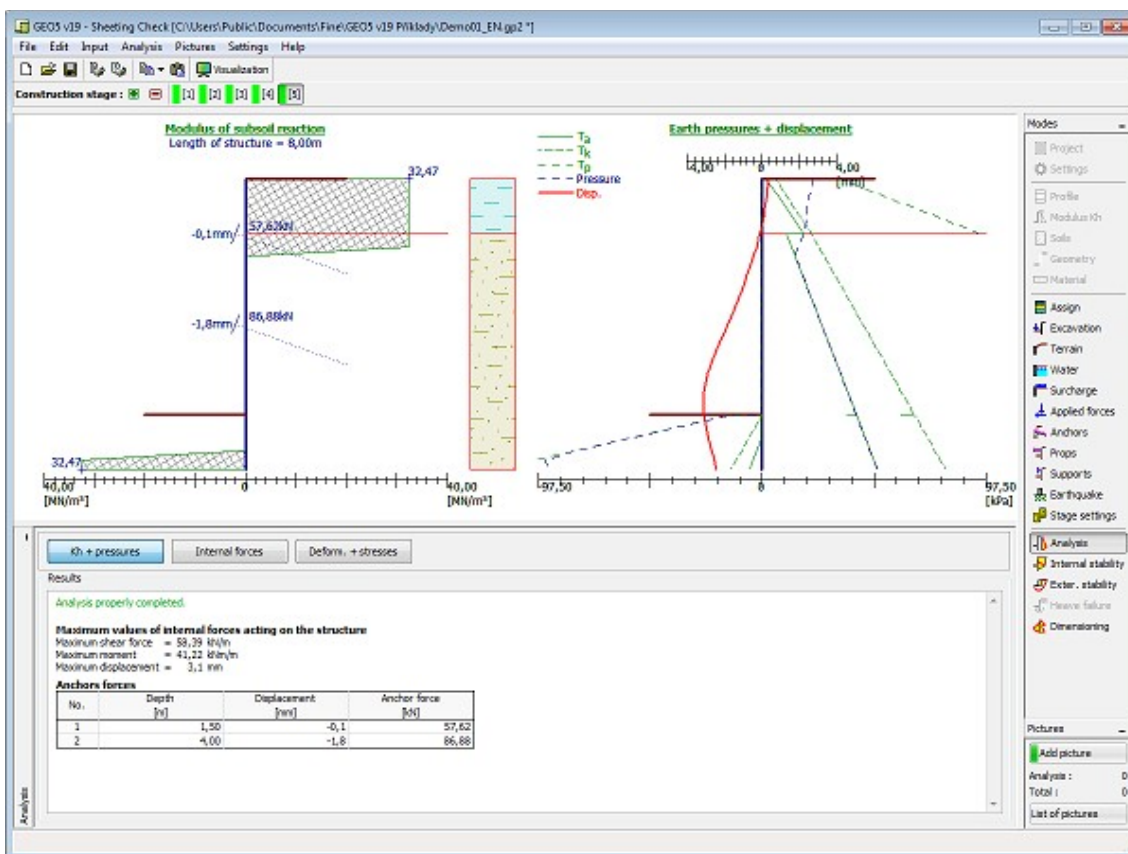
Plot of a structure together with forces acting in anchors, reactions and deformations of supports and props appear in the left part of the desktop. Distributions of bending moment and shear force are then plotted on the right.

- **Displacement + Stress**

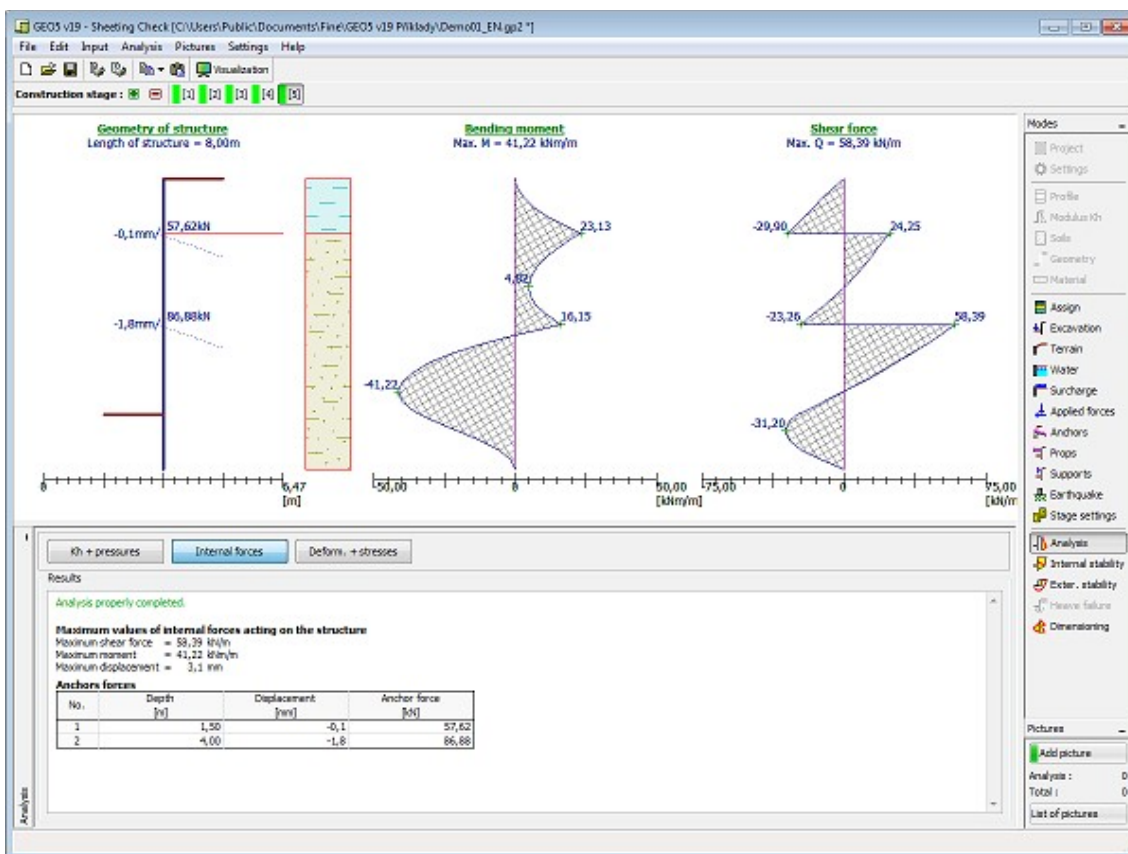
Plot of a structure together with forces acting in anchors, reactions and deformations of supports and props appear in the left part of the desktop. The deformed shape of a structure together with overall pressure acting on a **sheeting structure** is then plotted on the right.

Providing the modulus of subsoil reaction is found by iteration it is necessary to check the **course of manual iteration** in the dialog window "**Iteration**". Details are provided in the theoretical part of the help, chapter "**Modulus of subsoil reaction determined by iteration**".

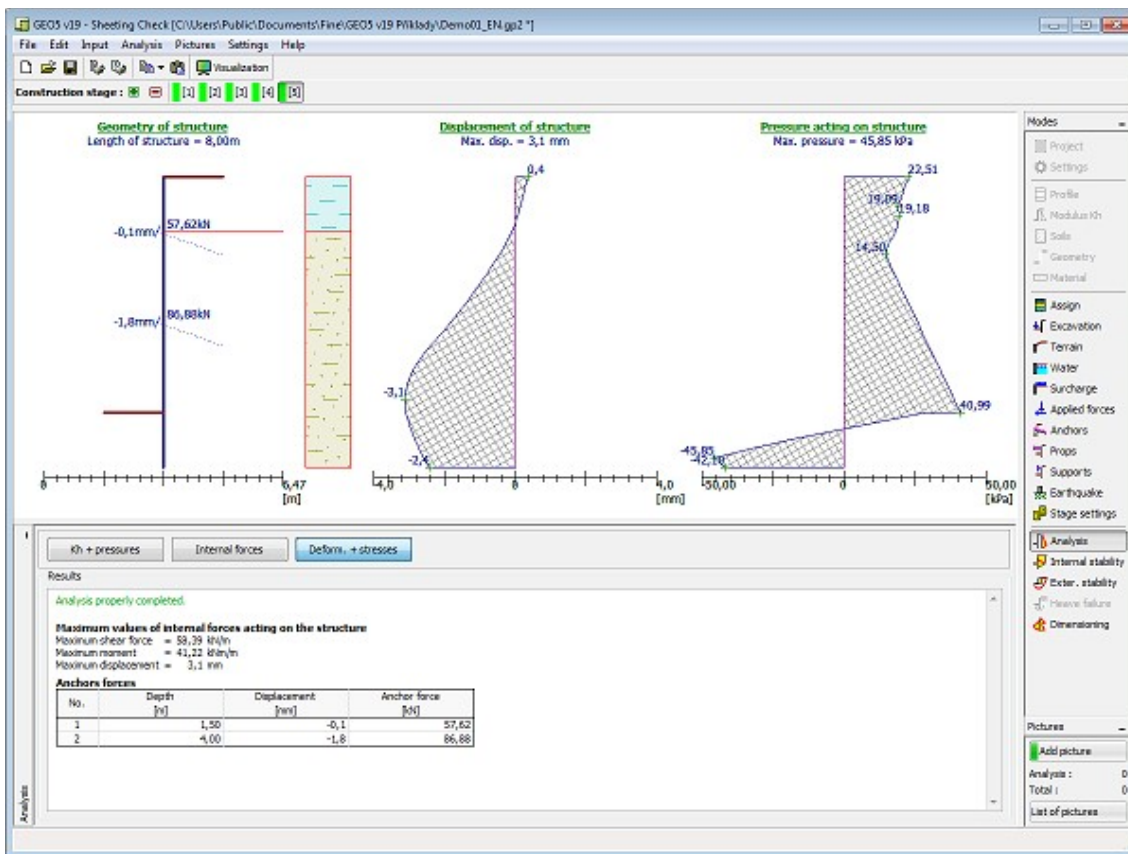
Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Analysis" - modulus of subsoil reaction, earth pressures and displacement



Frame "Analysis" - bending moment and shear force

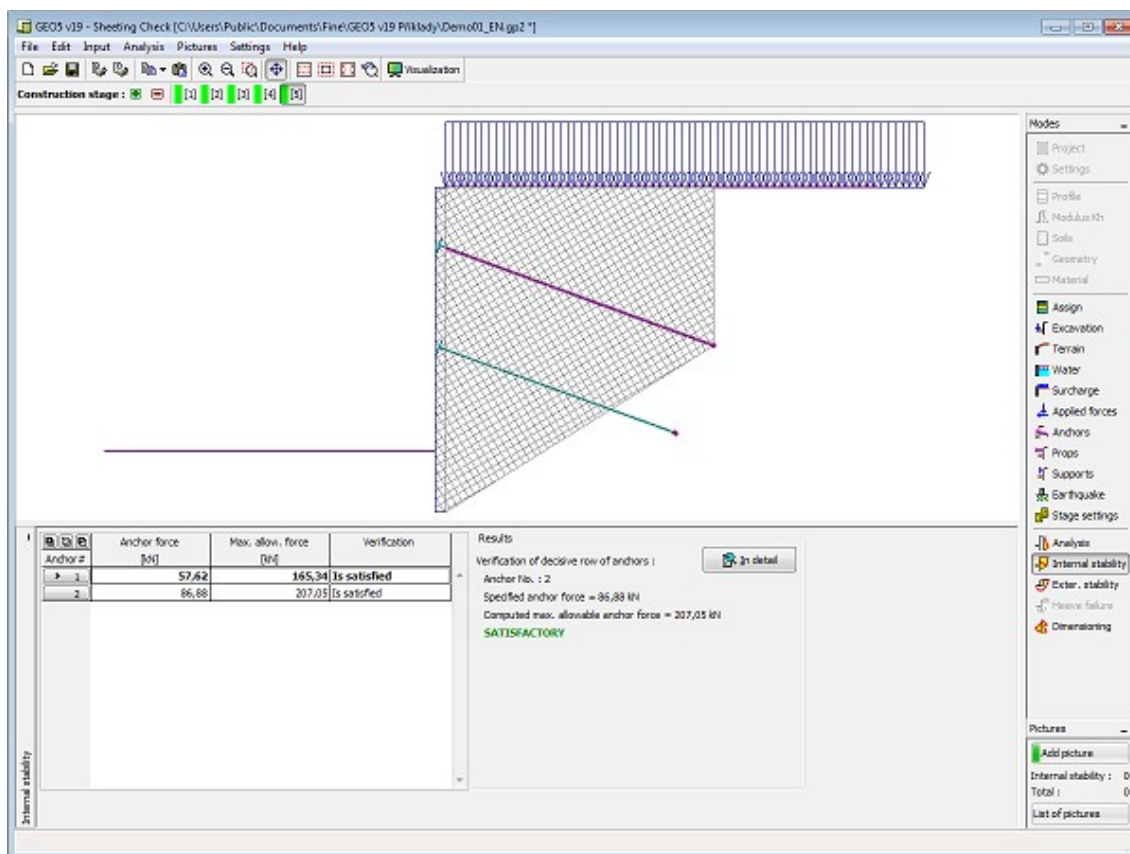


Frame "Analysis" - displacement and earth pressure acting on structure

## Internal Stability

This frame serves to check the internal stability of anchors - the frame is therefore accessible only in stages, in which the anchors are introduced. For each row of anchors the table shows input anchor forces and the maximum allowable forces in each anchor. Overall check for the most stressed row of anchors is displayed in the right part of the frame.





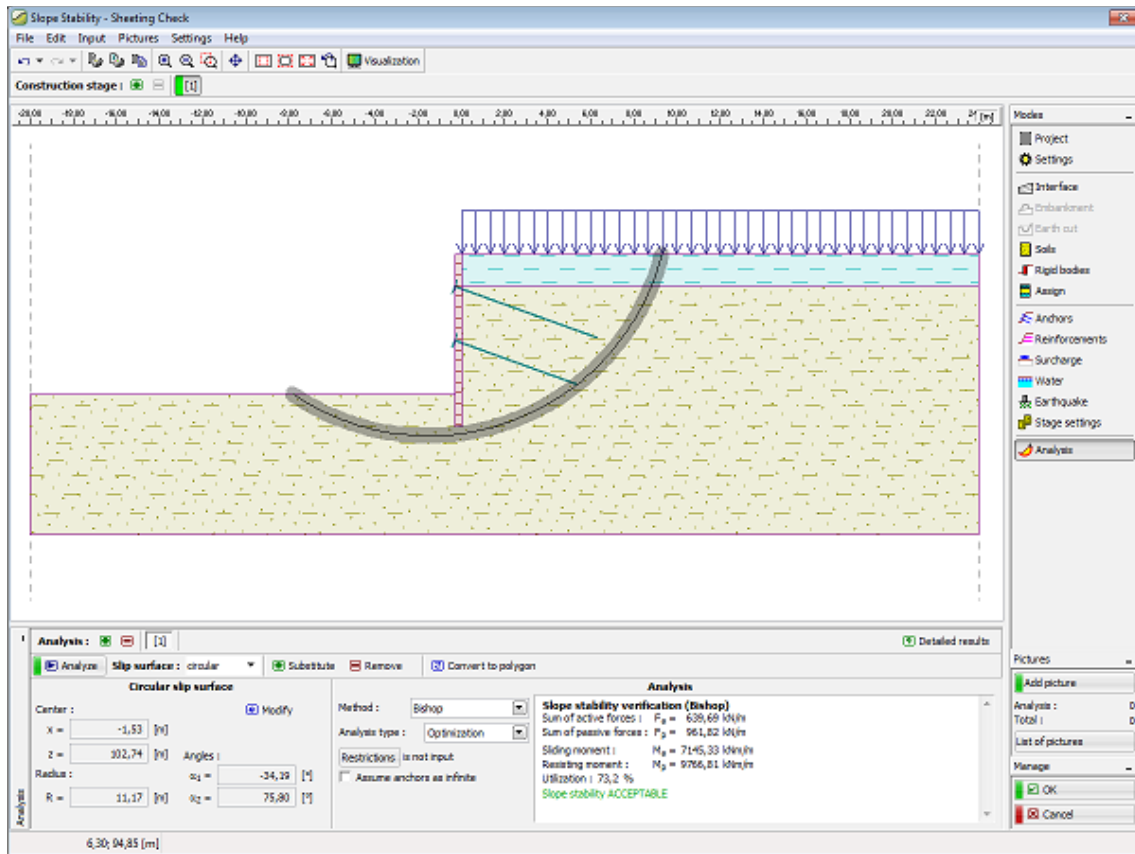
Frame "Internal stability"

## External Stability

By pressing the "**External stability**" button launches the "**Slope stability**" program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program "**Slope stability**" is installed.

After completing all analyses press the "**OK**" button to leave the program - all data are then carried over to the analysis protocol of the "**Sheeting check**" program.

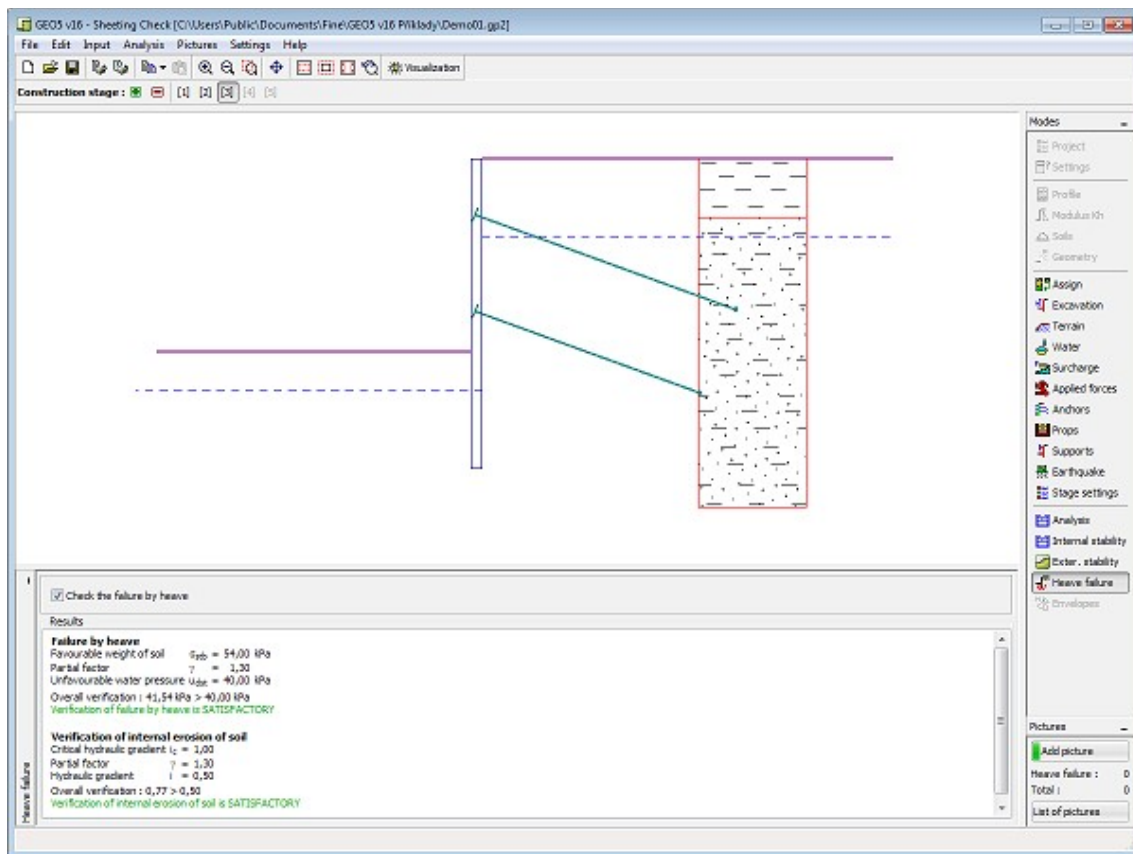




Frame "External stability"

## Heave Failure

This frame "**Heave failure**" serves to check the failure by heave and failure by piping. The frame is accessible only in the case, where the influence of water is considered as "Hydrodynamic pressure" (the base of a structure is sunk into permeable subsoil, which allows free water flow below the structure).



Frame "Heave failure"

## Dimensioning

In the frame "Dimensioning", it is possible to display an envelope of internal forces and displacements from all analyses (stages of constructions). Normally, the envelope is constructed from the results of all construction stages, however, it can only be created from the **selected stages**. The "Modify" button opens the dialog window "Construction stage selection", where it is possible to select the constructions stages that are used to generate the current envelope (by pressing corresponding buttons).

The maximum values of calculated internal forces (bending moments and shear forces) and the magnitude of displacement are displayed at the bottom part of the frame.

The program allows to dimension **steel-reinforced concrete** and **steel cross-sections** (by checking the option "Check cross-section").

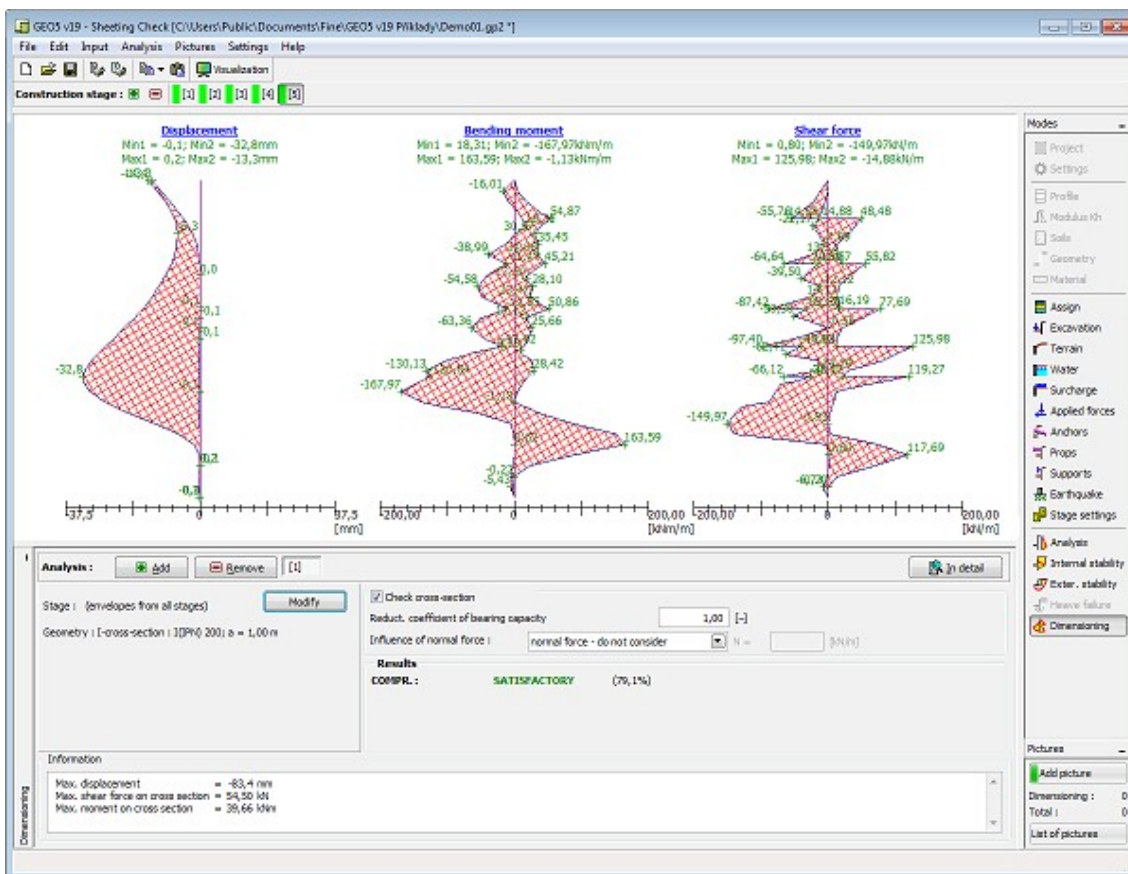
When checking the cross-section, it is possible to input the **reduction coefficient of bearing capacity**, which reduces the overall bearing capacity of a cross-section. When performing the analysis **with the reduction of earth pressures** this coefficient should be considered 1.0. For analysis **without earth pressures reduction** (to ensure a realistic behavior of a **sheeting structure**), it is necessary to increase the calculated forces, by adopting a coefficient greater than 1.0 (For EN 1997 is the value in interval 1.35-1.5).

For dimensioning of **steel cross-sections**, it is possible to assume **influence of normal force** in these ways:

- **normal forces - do not consider:** program doesn't consider influence of normal force.

- **normal forces - from nearest anchor:** program assumes maximum value of local normal force near the anchor as  $N = F \cdot \sin \alpha$  [kN/m], where  $\alpha$  represents slope of the anchor
- **normal forces - sum of all anchors:** program adds influence of normal force from input anchors as the sum of influence of all anchors.
- **normal forces - input:** user-defined value of normal force  $N$  [kN/m]

The frame allows to perform a larger number of analyses pro dimensioning of a cross-section. The "In detail" button at the right part of the frame opens the "Dimensioning" dialog window to show detailed results.



Frame "Dimensioning"

## Program Anti-Slide Pile

This program is used for design of pile walls stabilizing slope movement or increasing safety factor of the slope. The first analysis should be done in the Slope Stability program, where the active and passive forces acting on pile wall are computed. Next, the load and slip surface position are transferred to the Anti-Slide Pile program, where other analyses are performed (determination of internal forces on pile, pile deformation and dimensioning of pile reinforcement).

The help in the program "Anti-Slide Pile" includes the following topics:

- Input of data into individual frames:

Project	Settings	Profile	Modulus Kh	Pressiometri c Tests (PMT)	Dilatometric Tests (DMT)	Soils
Geometry	Material	Pressure Determinati on	Rock	Assign	Front Face	Terrain
Water	Surcharge	Applied Forces	Anchors	Supports	Earthquake	Stage Settings
Analysis	Dimensionin g					

- Standards and analysis methods

- Theory for analysis in the program "**Anti-Slide Pile**":

Stress in Soil Body	Earth Pressures	Sheeting Check	Anti-Slide Pile	Dimensioning of Concrete Structures	Dimensioning of Steel Cross- Sections
------------------------	--------------------	-------------------	-----------------	---	---

- Outputs
- General information about the work in the [User Environment](#) of GEO5 programs
- [Common input](#) for all programs

## Project

The [frame "Project"](#) is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in [text and graphical outputs](#).

The frame also allows user to switch analysis units ([metric / imperial](#)). Project data can be copied within all GEO5 programs using "[GeoClipboard](#)".

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings**, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

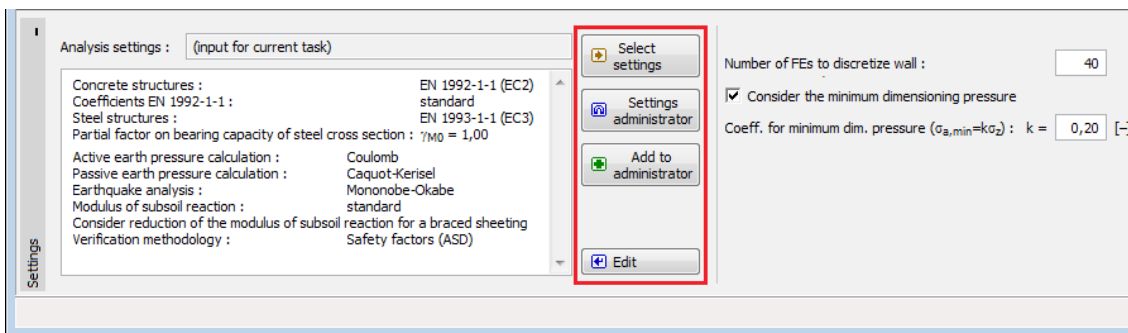
The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Materials and standards"** and the **"Excavations"** tabs.

When performing analysis according to **EN 1997** or according to the theory of **limit states**, the program enables to set whether to reduce the soil parameters for the calculation of limit pressures. When modeling a real behavior of the structure we recommend not to reduce these pressures.

The frame allows the user to specify subdivision of a wall in to finite elements (by default, the number of elements equals *40*) and the specify whether the structure is loaded by the **minimum dimensioning pressure**.



Frame "Settings"

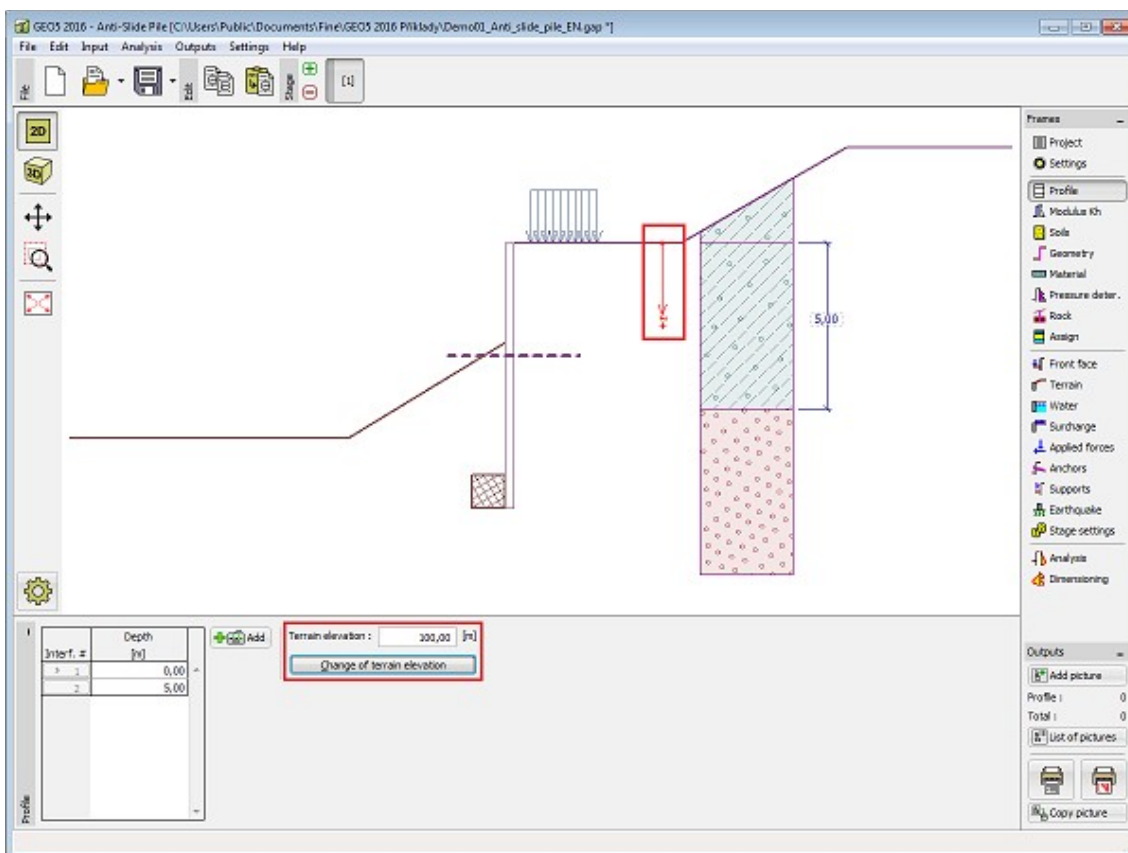
## Profile

The "Profile" frame contains a table with a list of input interfaces. After specifying interfaces it's possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "New interface" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "Change of terrain elevation" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".

The program makes it possible to import a profile in the gINT format.





Frame "Profile"

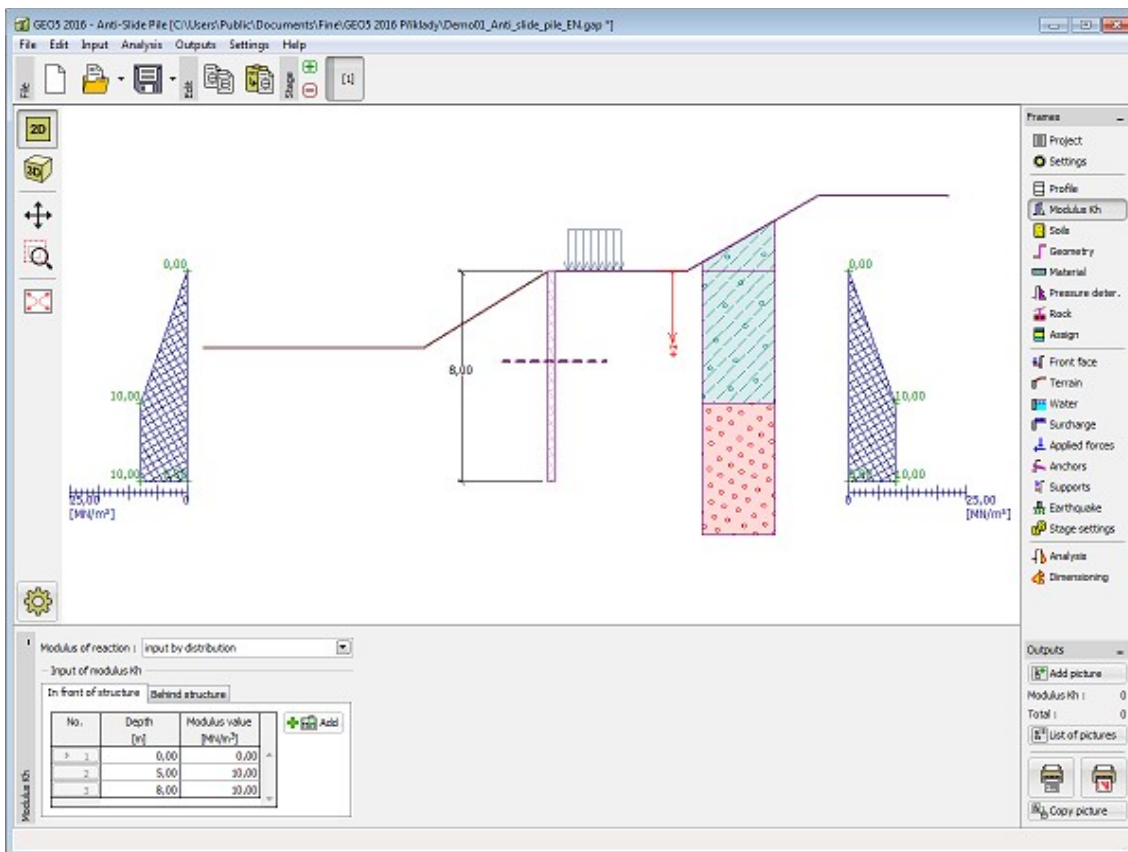
## Modulus Kh

This frame serves to specify a type of analysis for calculation of the modulus of subsoil reaction, which is an important input parameter when analyzing a sheeting structure by using the method of dependent pressures.

The way of calculation of the modulus of subsoil reaction  $k_h$  is selected in the "Settings" frame (in the "Edit current settings" dialog window in the "Excavations" tab).

The frame can take different forms depending on the selected method of calculation:

- **standard** (option "analyze – Schmitt", "analyze – Chadeisson", "manual iteration" or "automatic iteration")
- **input** (selecting the option "Input by distribution" opens a table in the frame that allows to input the values of the modulus of subsoil reaction  $k_h$  both in front of and behind the structure. For option "Input as a soil parameter" the modulus  $k_h$  is specified in the "Soils" frame, where the modulus of subsoil reaction is considered either as **linear**, or as **nonlinear - curve**)
- **pressiometer PMT** (modulus of subsoil reaction  $k_h$  is input either by **pressiometric test**, or as a parameter of soil in the "Soils" frame. Then there is specified method of calculation - according to NF P 94-282 or according to **Menard**)
- **chinese standards** (for "m" method is defined **horizontal displacement** at the ditch bottom  $v_b$  [mm] and magnitude of modulus  $A$  [MN/m<sup>3</sup>], or option **input** as a parameter of soil – "c" method, "k" or "m" method)



Frame "Modulus  $k_h$ " - volba "zadat průběhem"

Modulus  $K_h$

Parameters

Analysis method : Ménard

Ménard modulus : input pressiometer test

Characteristic length : a = 1,00 [m]

$l_v$

$a = \frac{2}{3}l$

Frame "Modulus  $k_h$ " - selection "pressiometer Ménard"

Modulus  $K_h$

Method of calculation of subsoil modulus : method "m"

Displacement at the ditch :  $v_b = 0,00$  [mm]

Magnitude of modulus at the ditch : A = 0,00 [MN/m<sup>3</sup>]

Frame "Modulus  $k_h$ " - selection "Chinese standards"

## Pressiometric Tests (PMT)

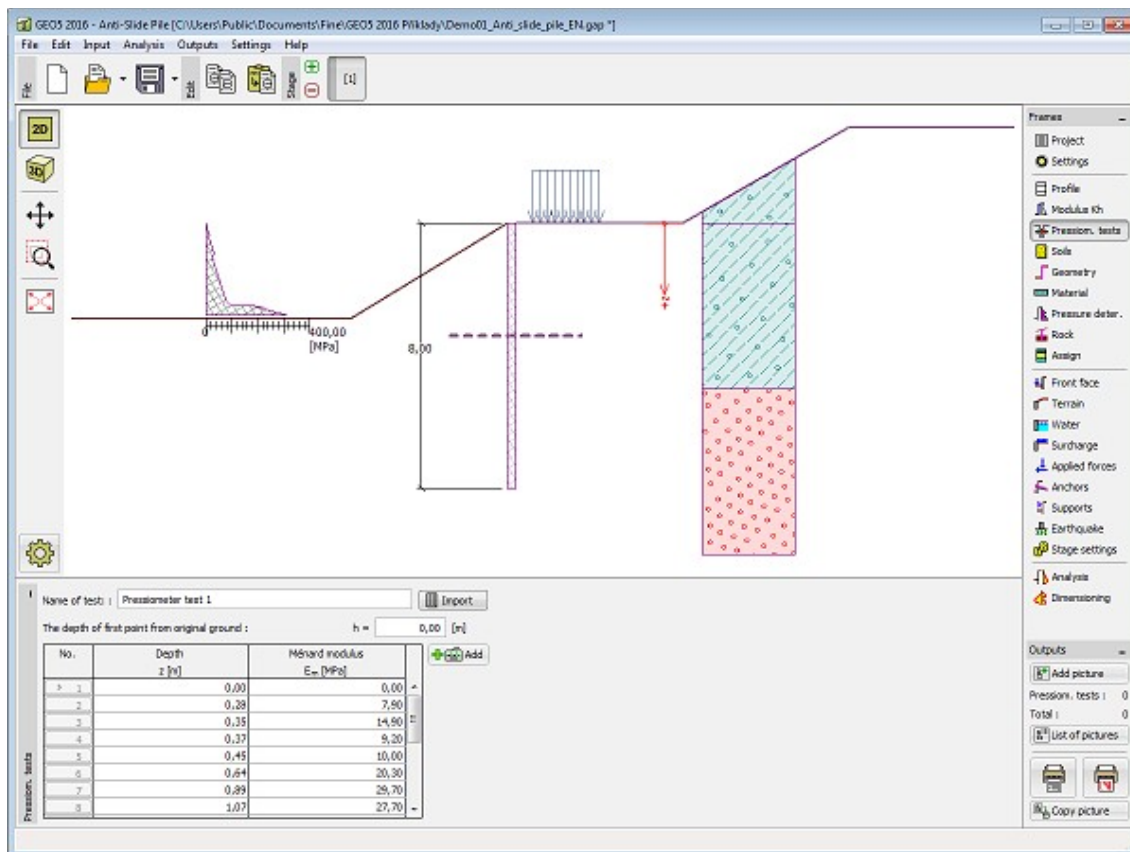
The frame "**Pressiometric tests**" serves to input the name of a **pressiometric test** and depth of the first point of PMT  $h$  [m] from the finished grade.

The results of **pressiometric test (PMT)** are imported into the program by pressing the "**Import**" button. The procedure of an import of table data is more described [herein](#).

This frame contains a **table** with list of the input values of **pressiometric test (PMT)**. Adding (editing) of values is performed in the "**New value of test**, or **Edit the value of test**" dialog **window**, where is specified the depth  $z$  [m] (in the  $x, y$  coordinate system) and the measured values of **pressiometric (Menard) modulus  $E_m$  [MPa]**.

**Note:** The frame is accessible only in the case, when in the "**Settings**" frame is selected an option "**pressiometer PMT**" (the "**Excavations**" tab) and together for the **modulus of subsoil reaction** is chosen the option "**input pressiometer test**" in the "**Modulus  $K_h$** " frame.





Frame "Pressiometric tests"

## Dilatometric Tests (DMT)

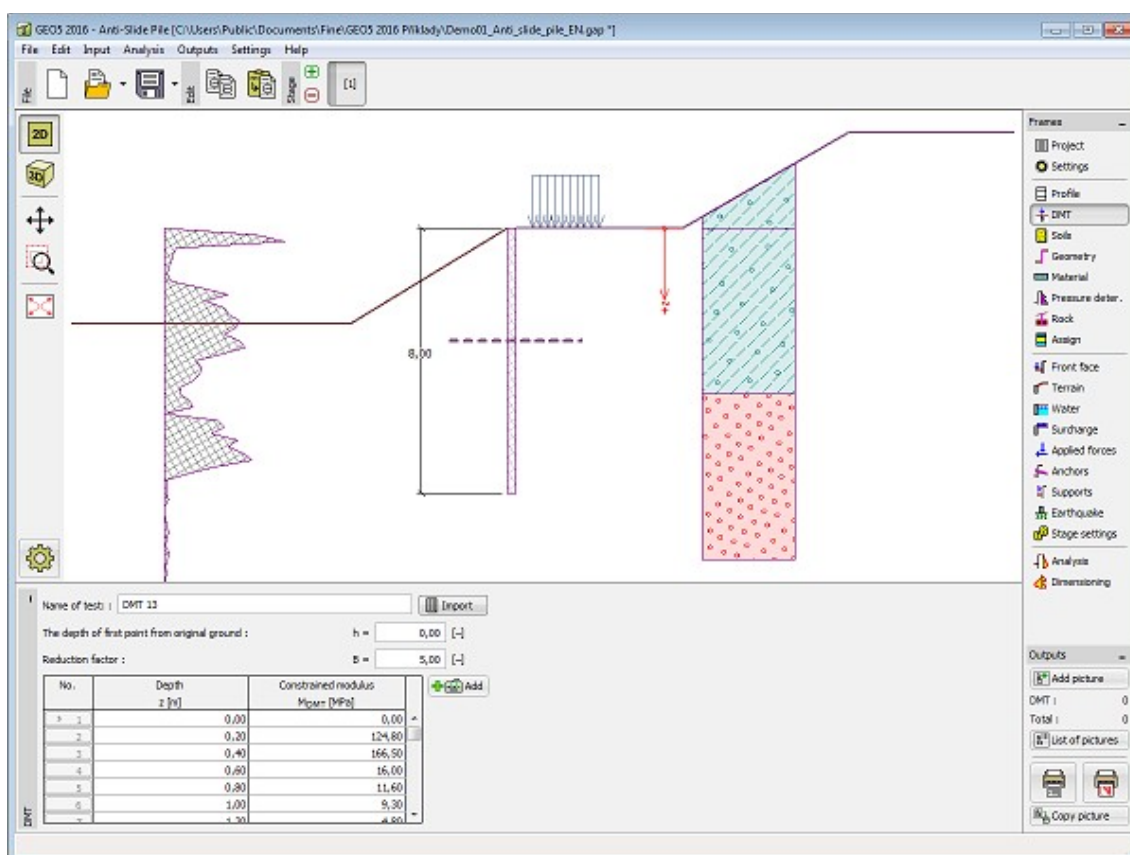
The frame "DMT" serves to input the name of dilatometric test, depth of the first point of DMT  $h$  [m] from the finished grade and characteristic length of sheeting structure (coefficient of reduction)  $B$ .

The results of dilatometric test (DMT) are imported into the program by inserting the file in format **UNI** (\*.uni). It's a **standardized and universal format** for import of the measured data obtained from dilatometric tests, which is used in the world.

This frame contains a table with list of the input values of dilatometric test (DMT). Adding (editing) of values is performed in the "New value of test, or Edit the value of test" dialog window, where is specified the depth  $z$  [m] and the measured values of constrained soil modulus  $M_{DMT}$  [MPa].

If during the evaluation of dilatometric test is measured the zero value of constrained soil modulus  $M_{DMT}$ , then program allows the automatic correction of measurement errors - instead of zero value is considered the arithmetic average of the next upper and lower non-zero value of  $M_{DMT}$  in the calculation.

**Note:** The frame is accessible only in the case, when in the "Settings" frame is selected an option "dilatometer DMT" (the "Excavations" tab) for calculation of the modulus of subsoil reaction.



Frame "Dilatometric tests (DMT)"

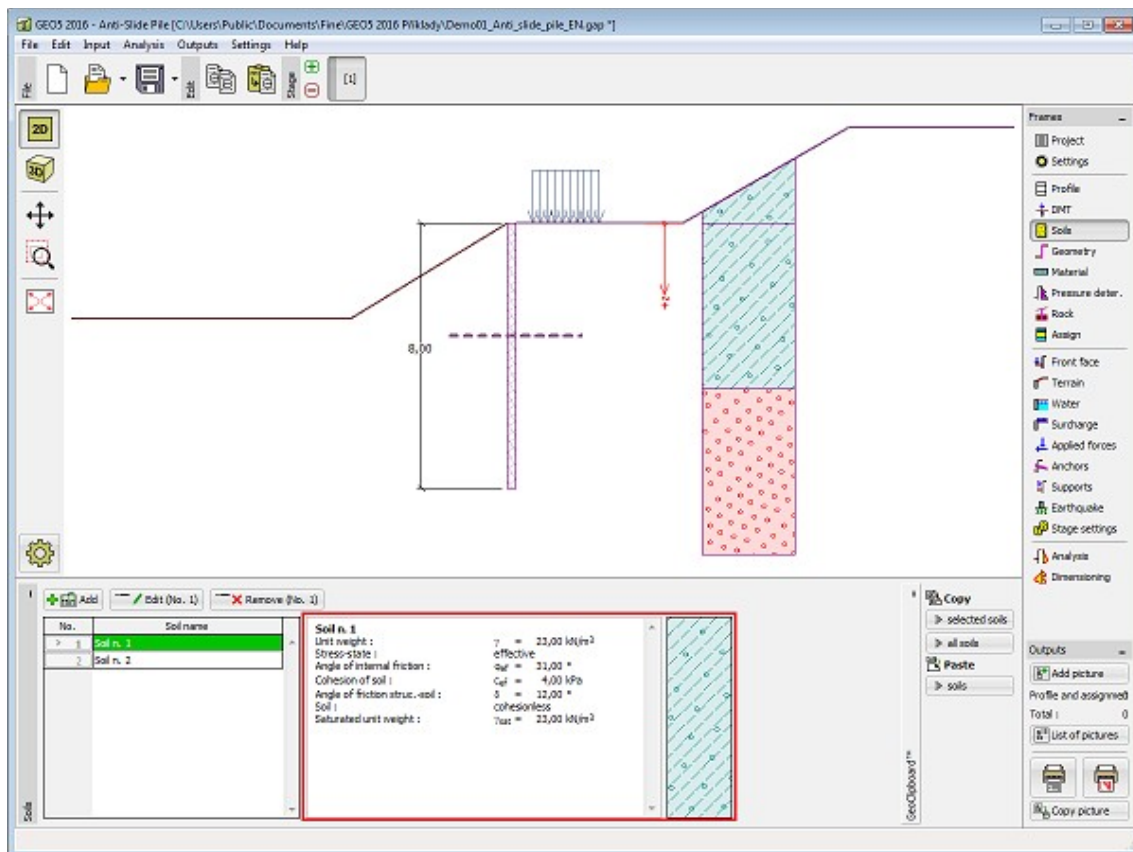
## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Earth pressure at rest", "Uplift pressure" and "Modulus of subsoil reaction".

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

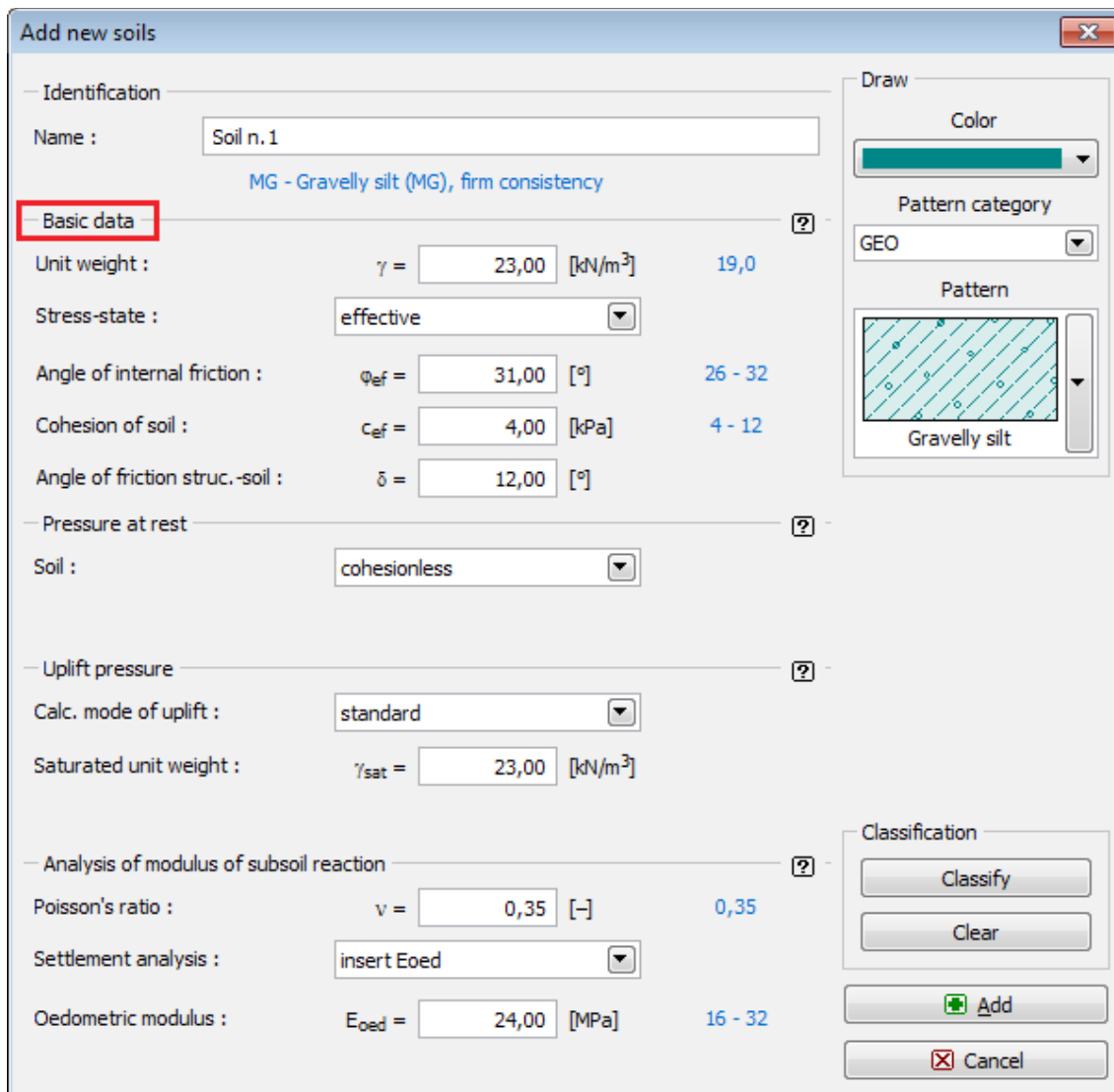
This part of the window allows to introduce basic parameters of the soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **soils database**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the built-in database, these parameters must be defined manually. Approximate parameters of rocks are presented in the **theoretical section**.

Either **effective** or **total** parameters of the angle of internal friction and cohesion are specified depending on the settings in the "**Stress analysis**" combo list. Whether to use **effective or total parameters** depends primarily on the type of soil and load, structure duration and water conditions.

For **effective stress**, it is further needed to specify the **angle of internal friction between the soil and the structure**, which depends on the structure material and type of soil. Possible values of this parameter are listed in the **table of recommended values**.

For **total stress**, it is further needed to specify the **adhesion of soil to the structure face**  $a$ .

The associated theory is described in detail in the chapter "**Earth pressures**".



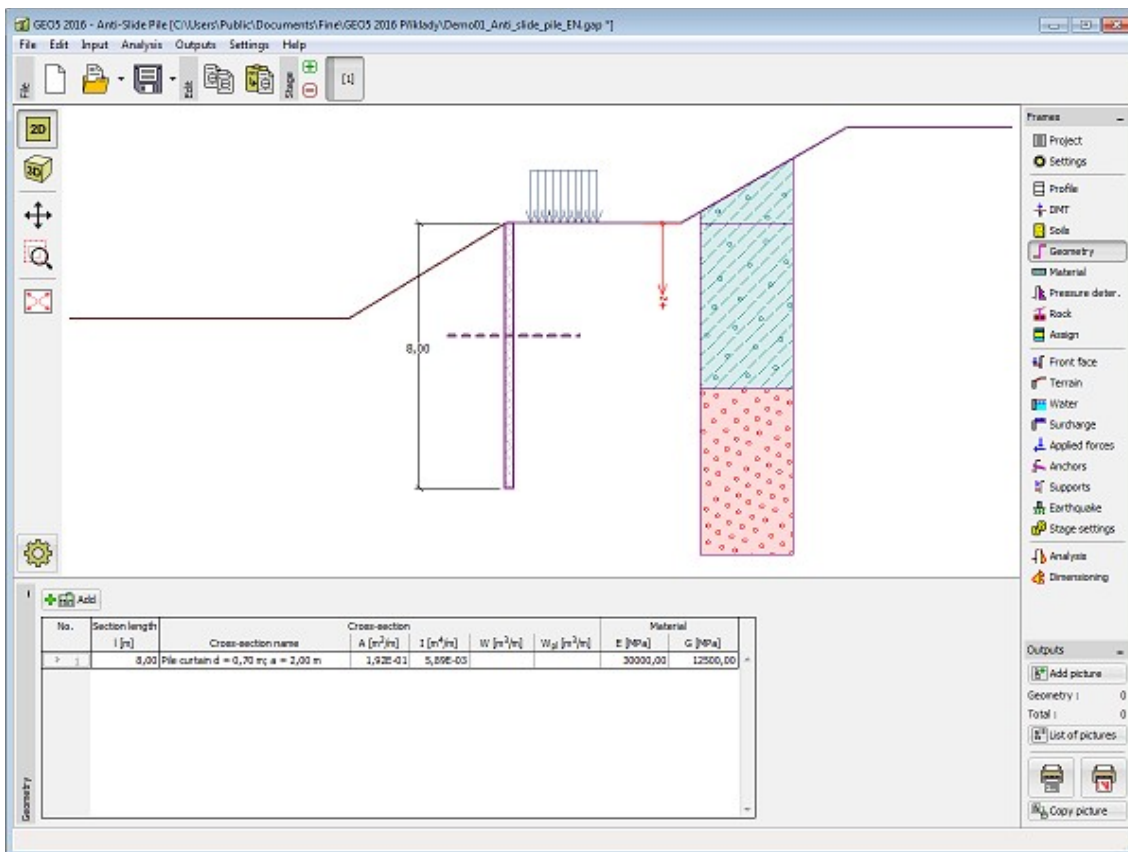
Dialog window "Add new soils" - "Basic data"

## Geometry

The "Geometry" frame contains a table with a list of input structural sections forming the anti-slide pile. For each section the table stores its cross-sectional characteristics ( $A$  - area,  $I$  - Moment of inertia - these variables are always expressed with respect to 1 m run of structure length) and material characteristics ( $E$  - Modulus of elasticity,  $G$  - Shear modulus ).

Adding (editing) sections is performed in the "New section" dialog window.

The input sections can be further edited on the desktop with the help of active objects - double-click on a structure opens a dialog window with a given section.



Frame "Geometry"

## Adding and Editing a Section

The "New section" or "Edit section" dialog window contains the following items:

### Type of wall

- the combo list contains individual structural types of **shetting walls** (pile curtain, reinforced concrete rectangular wall, sheet pile wall, steel I cross-section, or user input of cross-sectional characteristics)

### Section length $l$ [m]

- use input field to specify length of a given section of sheeting structure

### Coefficient of pressure reduction below ditch bottom

- this **coefficient** is possible to **input** or **calculate automatically** for pile curtain, steel I cross-section or for user input of  $A$ ,  $I$ ,  $E$ ,  $G$  (for **braced sheeting** this coefficient equals to 1.0)

### Geometry

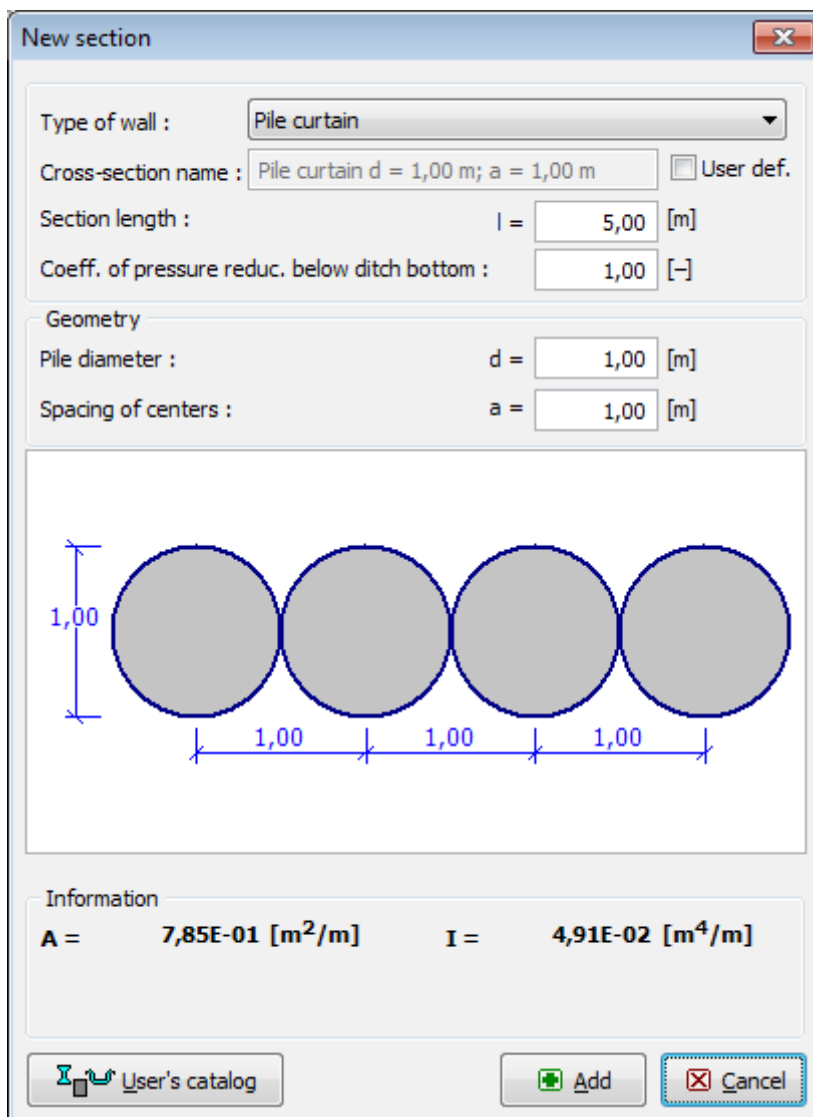
- contains information about geometry for selected structural variant. For **pile curtain** is defined diameter of piles  $d$  [m], spacing of piles  $a$  [m] and type of cross-section (square, circle). For **reinforced concrete rectangular wall** is defined its thickness  $h$  [m]. For **steel I cross-section** is defined spacing of profiles  $a$  [m]. For **user input of  $A$ ,  $I$ ,  $E$ ,  $G$**  and option "**Dimension of steel cross-section**" is moreover defined cross-sectional modulus  $W$

### Profile

- contains information about profile for selected structural variant "**Steel I-section**" (button "**Catalog**" opens the "**Catalog of**

- Information**
- profiles**", which contains a list with profile classes and individual profiles)
  - contains overview of cross-sectional characteristics of the input cross-section ( $A, I, E, G$ ). Cross-section area  $A$ , moment of inertia  $I$  and cross-sectional modulus  $W$  are always evaluated for 1 m run of the sheeting structure length

The "**User's Catalog**" button in the bottom part of the window opens the "**User's catalog**" dialog window.



Dialog window "New section"

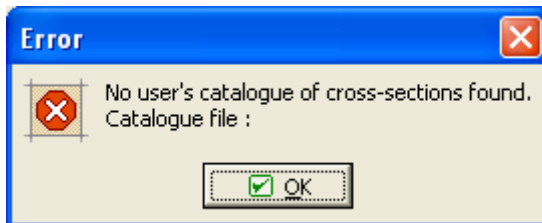
## User's Catalog

The User Catalog allows to define and store own cross-sections and their characteristics that appear in a **sheeting structure**. At first use of the catalog (has not been yet created) the program prompts a warning message that no catalog was found. Then, pressing the button "**OK**" opens the "**Save as**" dialog window that allows to enter the catalog name and saving it

into a specified location by pressing the "Save" button (by default a folder used for saving the project data is assumed).

The program allows the user to create more than one catalog. The next catalog is created by pressing the "New" button - the program asks, whether the current catalog should be replaced (**the currently loaded catalog is not deleted!**) and saves the new catalog under a new name. The "Open" button allows for load an arbitrary user catalog and by pressing the "Save as" button for saving it under a different name.

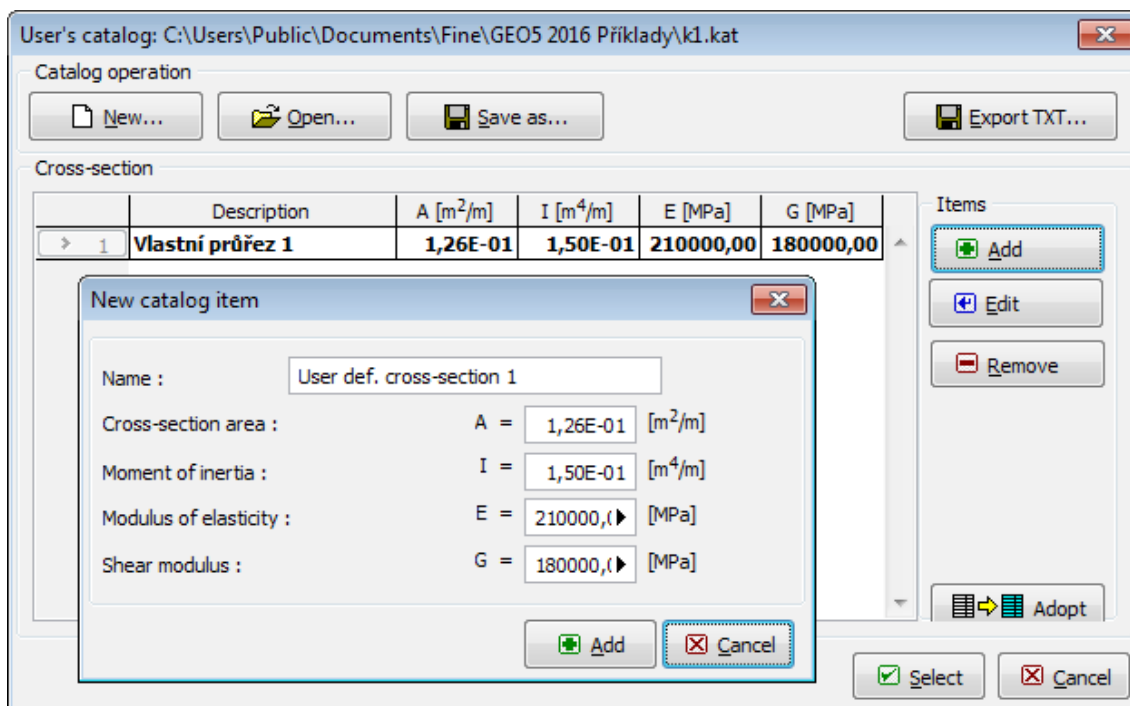
"Export TXT" button allows for exporting of currently loaded user catalog to text file.



Dialog window at first use - user catalog of cross-sections

The "User's catalog" dialog window contains a table listing the user defined cross-sections. The "Add item" button opens the "New catalog item" dialog window that allows for specifying and subsequent saving of characteristics of a new cross-section into the catalog. Buttons "Edit item" and "Remove item" serve to edit individual items in the table.

The "Accept current" button accepts the current cross-sectional characteristics of a cross-section specified in the "New section" dialog window and opens the "New catalog item" dialog window that allows for modifying and saving the current cross-section.



Dialog windows "User's catalog" and "New catalog item"



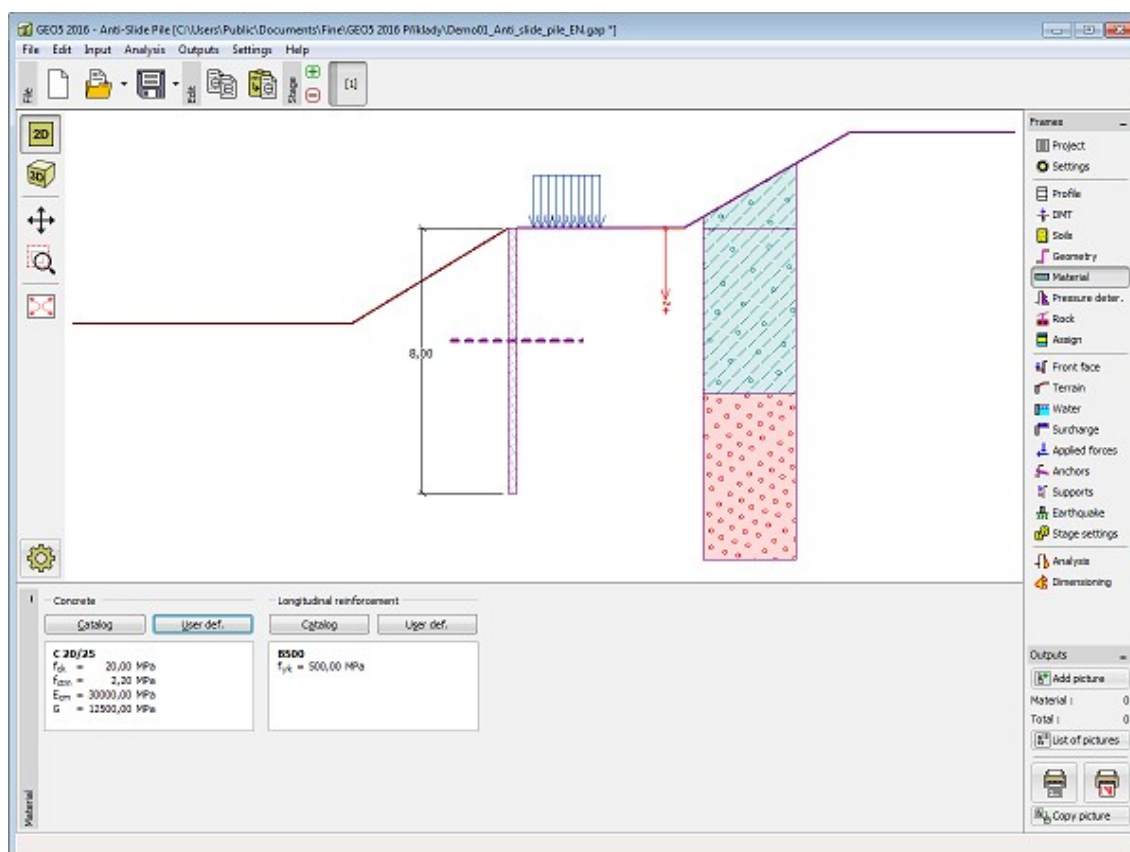
## Material

The **frame "Material"** allows to select material parameters of used concrete, longitudinal reinforcement or structural steel.

Two options are available when selecting the material type:

- The **"Catalog"** button opens the **"Catalog of materials"** dialog window (for concrete or steel reinforcement), the list of materials then allows to select the required material.
- The **"User defined"** button opens the **"Editor of material - Concrete"** dialog window (for concrete) or the **"Editor of material - Reinforcing steel bars"** dialog window (for longitudinal reinforcement) or the **"Editor of material - Structural steel"** dialog window (for structural steel), which allows to input the specification of material parameters manually by user.

The content of catalogs depends on the selection of relevant standard for the dimensioning of **concrete** or **steel** structures set in the **"Materials and standards"** tab.



Frame "Material"

## Pressure Determination

The **frame "Pressure determination"** allows to input the **depth of the slip surface** and **load on the anti-slide pile** above the slip surface.

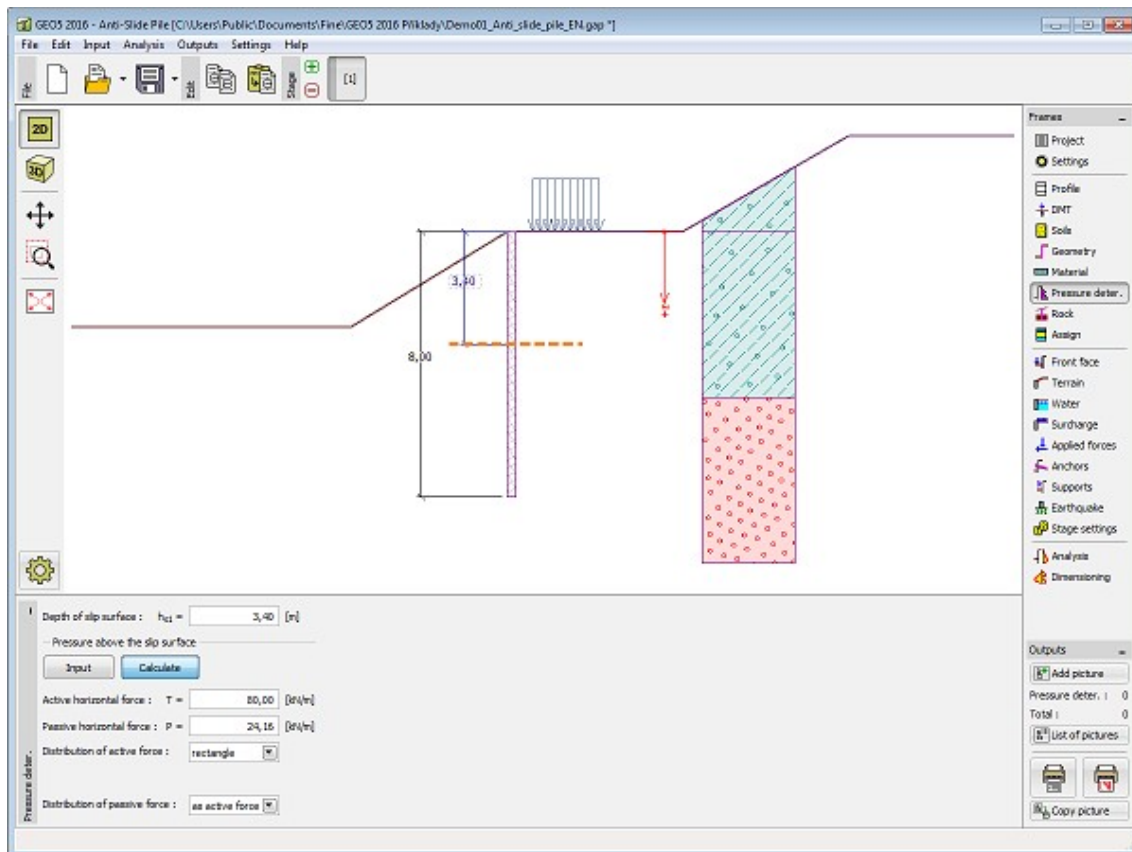
If the standard selection **"Calculate"** is checked, the load is specified by active horizontal force and passive horizontal force that acts on the pile above the input slip surface. **Active and passive forces** can be calculated in the **Slope stability** program and transferred into the



program Anti-Slide Pile. Next, the **shape of the pressure distribution** must be specified behind (active force) and in front of the pile (passive force). Program does **NOT INCREASE** the input passive and active force by any partial factor during the analysis - it is necessary to determine them according to required standards and rules.

If the forces are obtained from the **Slope stability** program, they correspond with the way of analysis set in the program. For example, when analysis is done according to Eurocode **EN 1997-1**, the received values are already design values of forces.

The values of the resulting pressure acting on the structure above the slip surface can be input in the table (option **"Input"**) too.



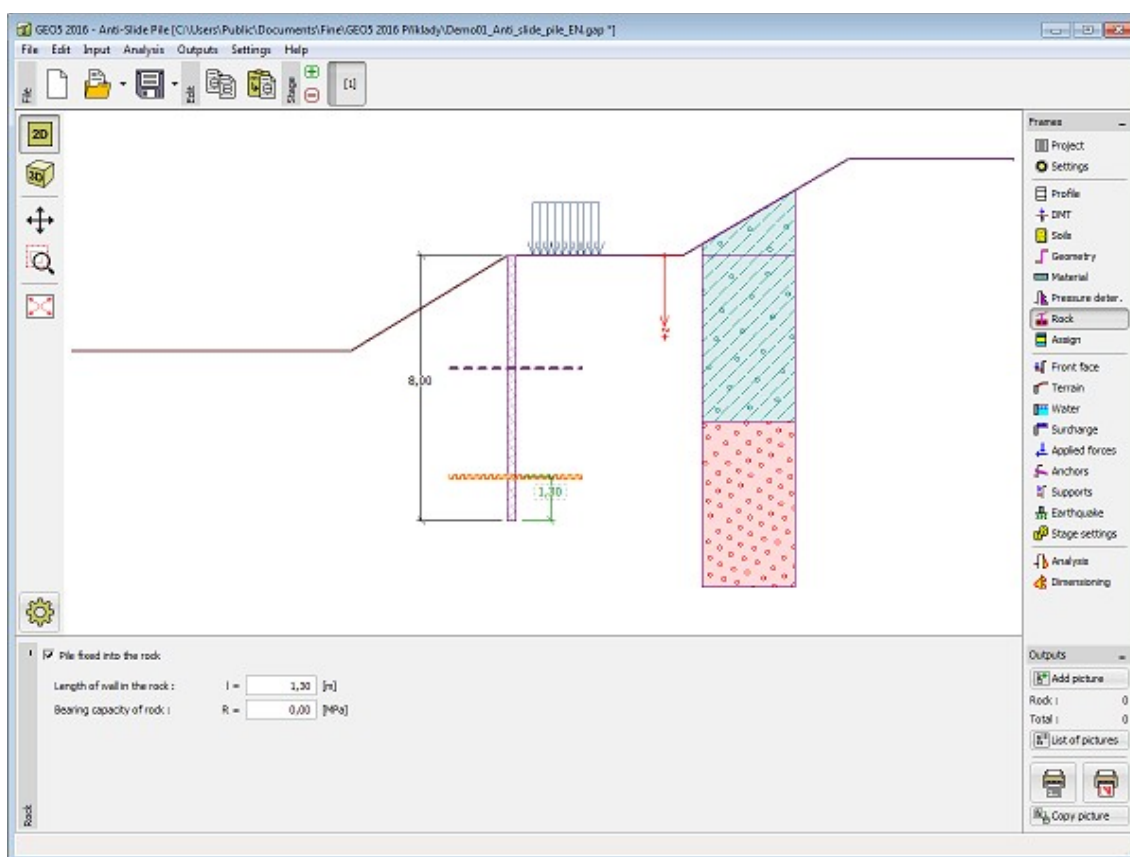
Frame "Pressure Determination"

## Rock

The **frame "Rock"** allows to input the length of the pile fixed into rock  $l$  [m] and **rock bearing capacity**  $R$  [MPa].

The rock is only modelled as elastic material and no limit passive pressure is considered - contact stress can reach any values. In the frame **"Analysis"** the program checks, if **maximum stress** does not exceed **design rock bearing capacity**. Active pressure of the rock is not considered as null.

Note: In many cases, the slip surface follows the rock subsoil, hence this case should be always investigated.

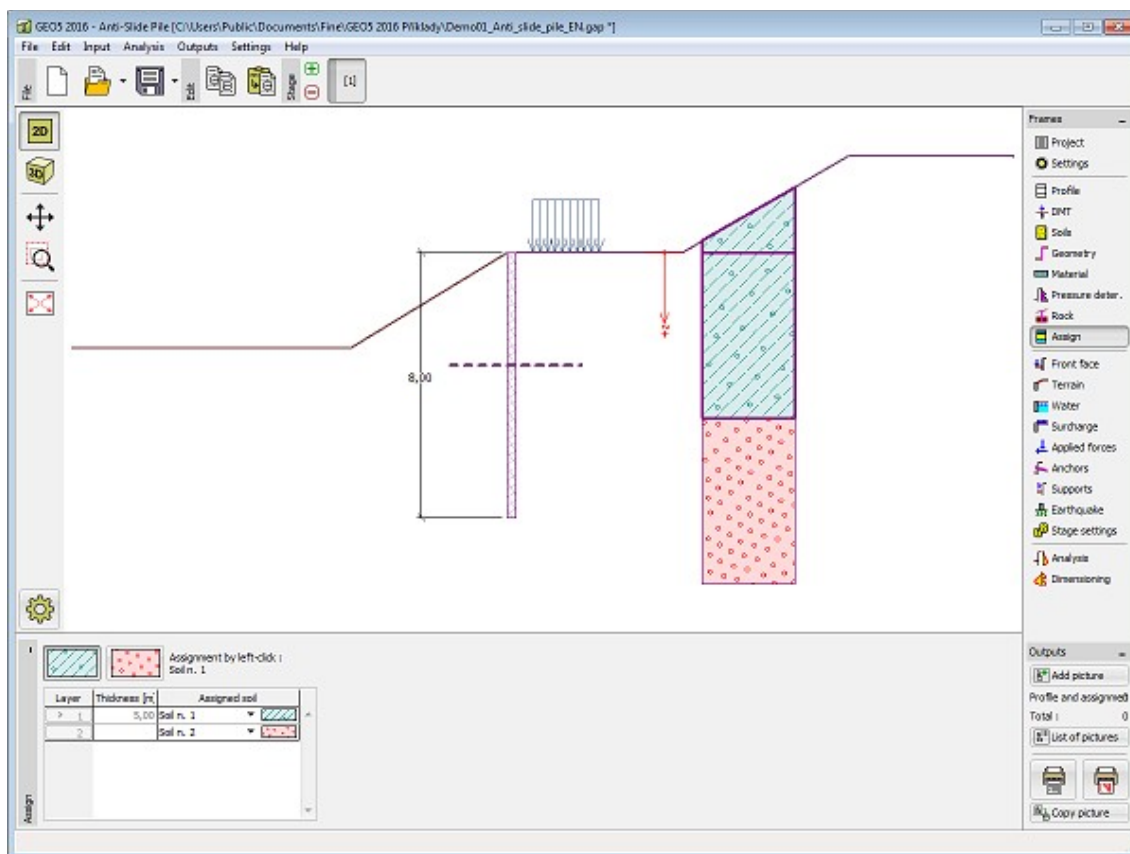


Frame "Rock"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

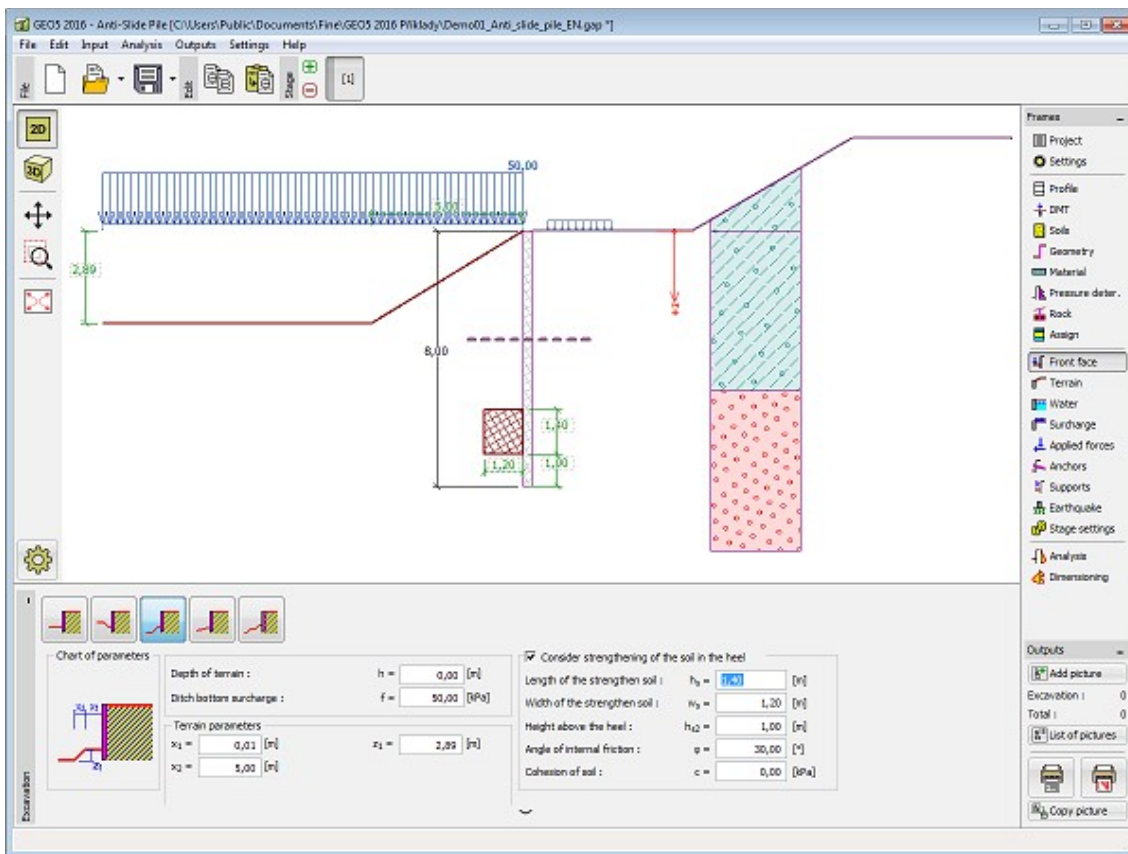


Frame "Assign"

## Front Face

The "**Front Face**" frame serves to input the shape of the terrain in front of the structure. The selected shape with a graphical hint appears in the left part of the frame. The dimensions of a structure can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

In this frame, it is possible to input **strengthening of the soil** at heel of the piles. The principle of calculation is described in more detail [herein](#).



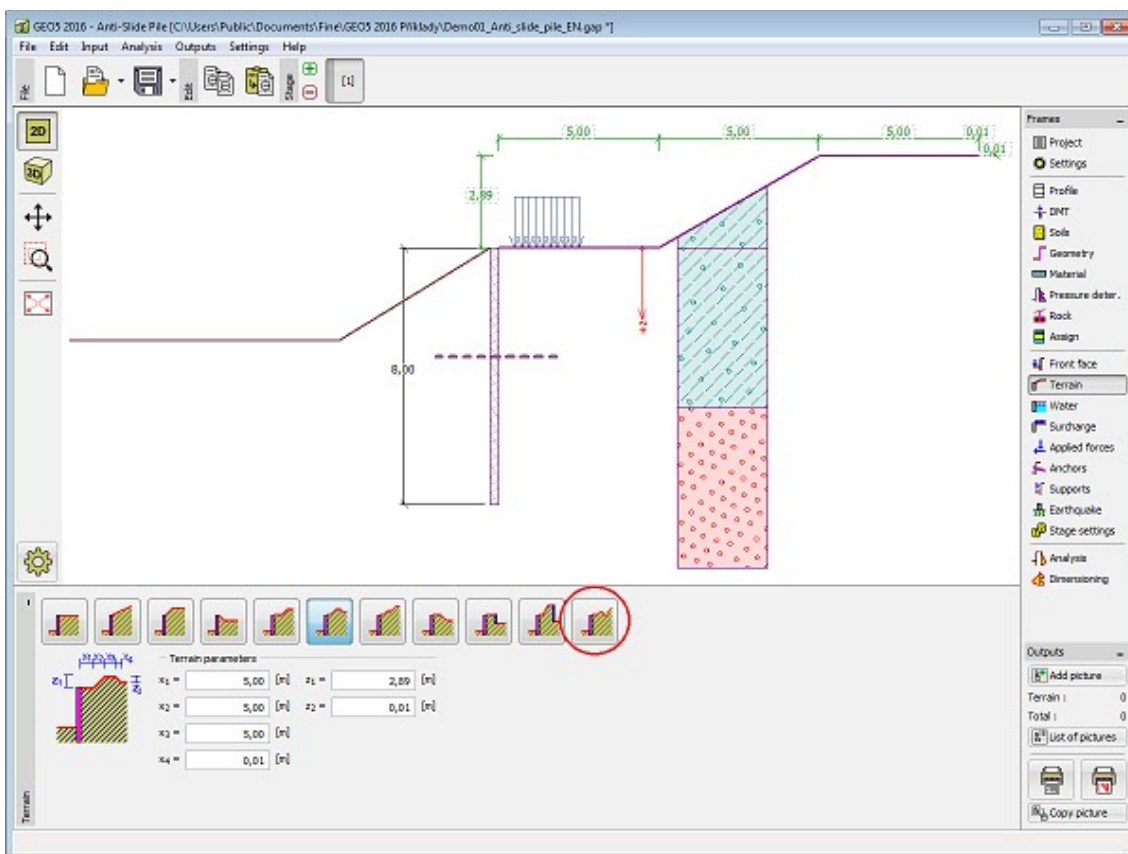
Frame "Front Face"

## Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0,0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".

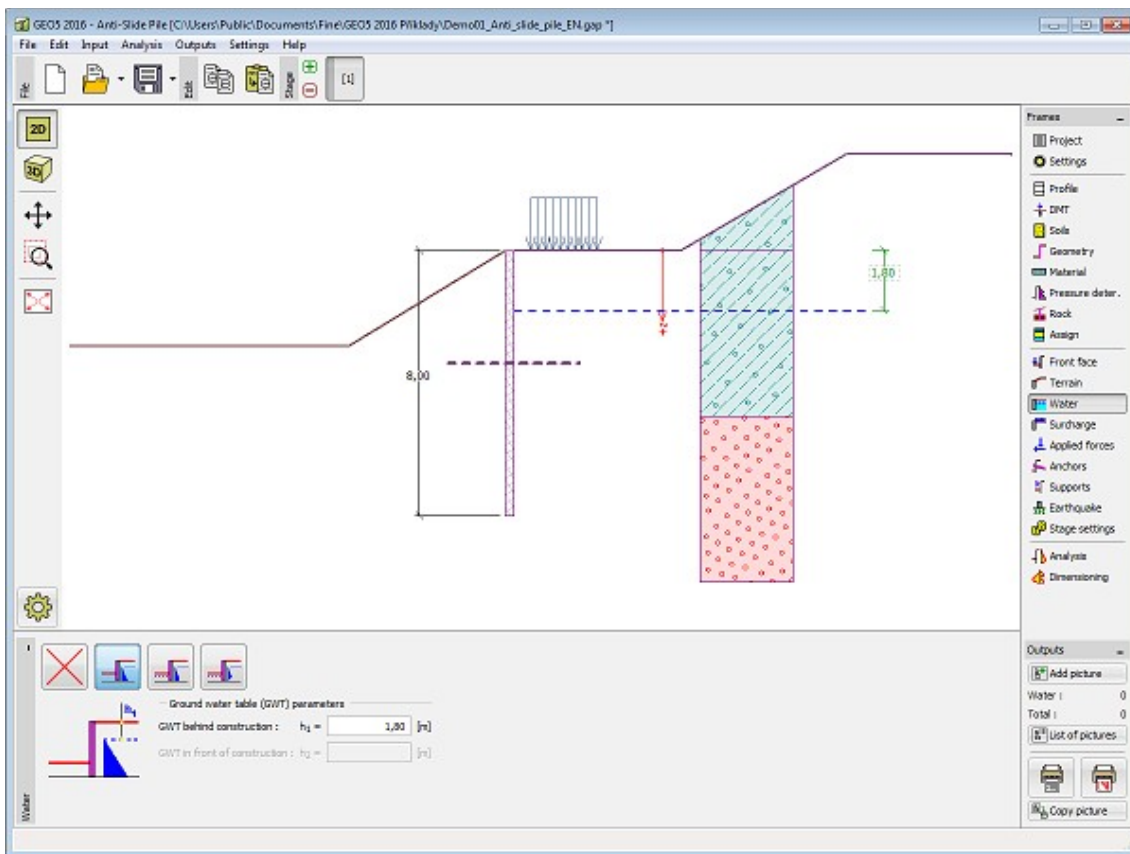


Frame "Terrain"

## Water

The "**Water**" frame allows to select the type of water by pressing the appropriate button. Parameters of water can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".



Frame "Water"

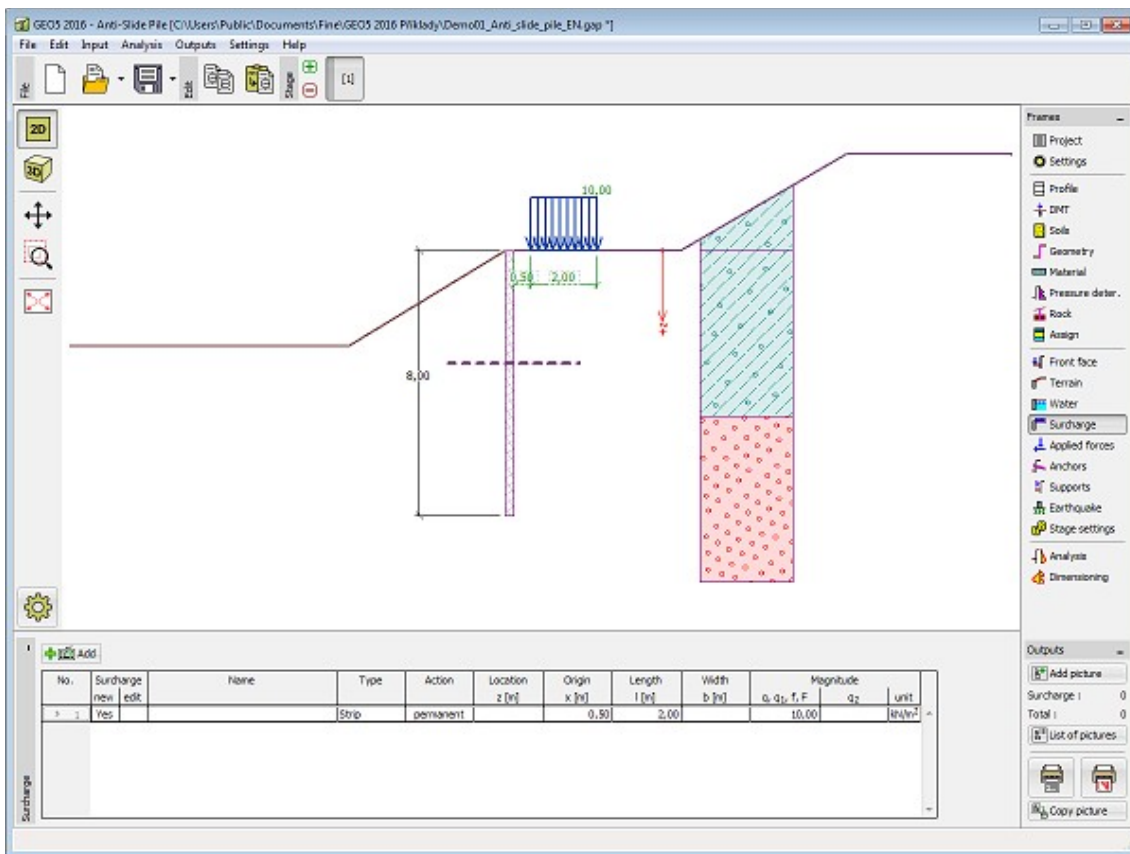
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".



Frame "Surcharge"

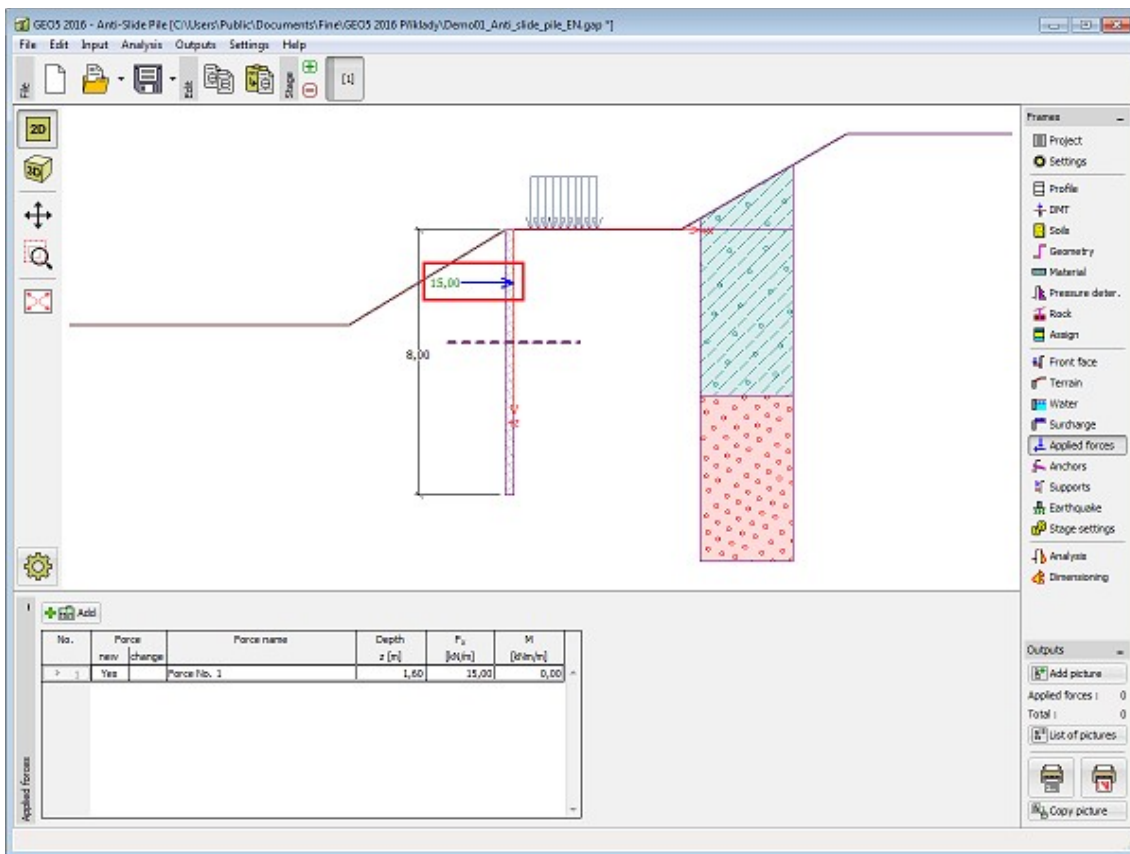
## Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "**New force**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

**Applied forces** represent an additional load on the structure of the wall, braced sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.





Frame "Applied forces"

## Anchors

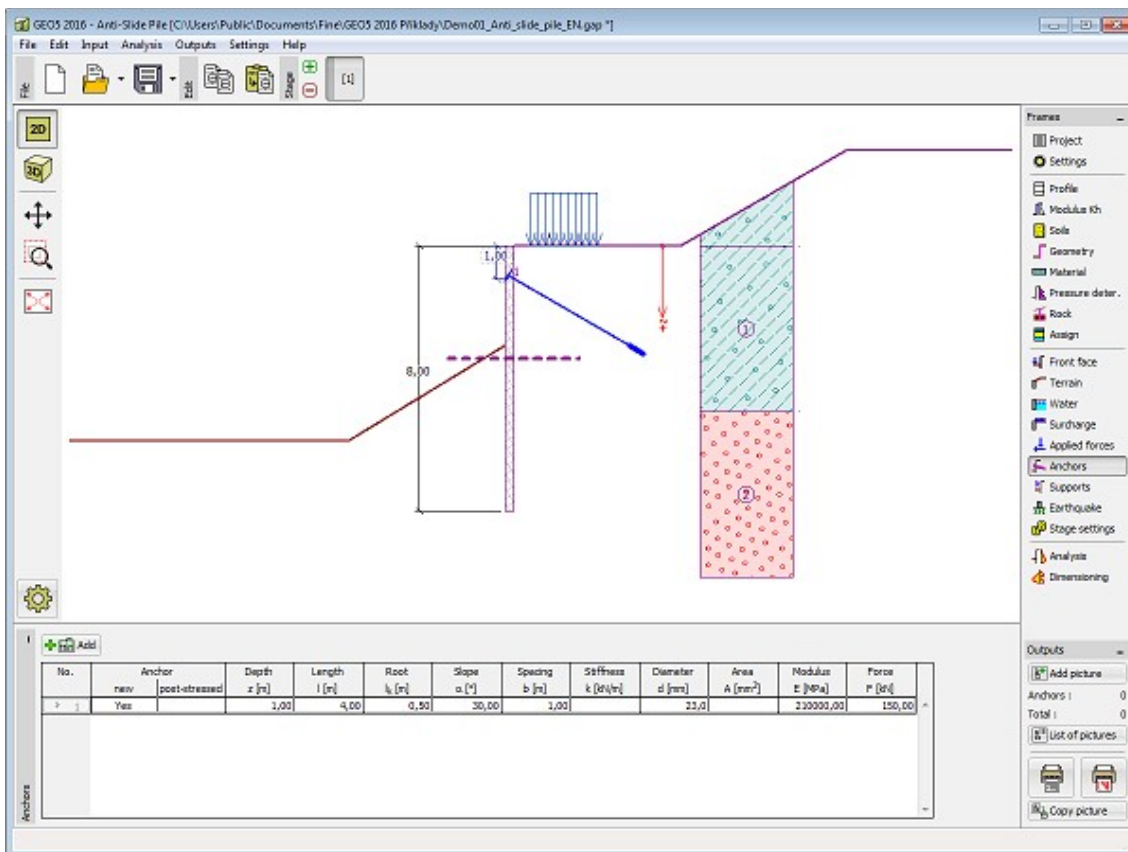
The "**Anchors**" frame contains a table with a list of input anchors. Adding (editing) anchors is performed in the "**New anchor**" dialog window. The input anchors can be edited on the desktop with the help of active objects.

The anchor is automatically placed on already **deformed structure** (displacement is obtained from the previous construction stage).

The **anchor stiffness becomes effective** in subsequent stages of construction.

Due to the displacement of the sheeting structure the forces in anchors are changing. In subsequent stages the anchor can no longer be edited, it is only possible to change the force in the anchor.





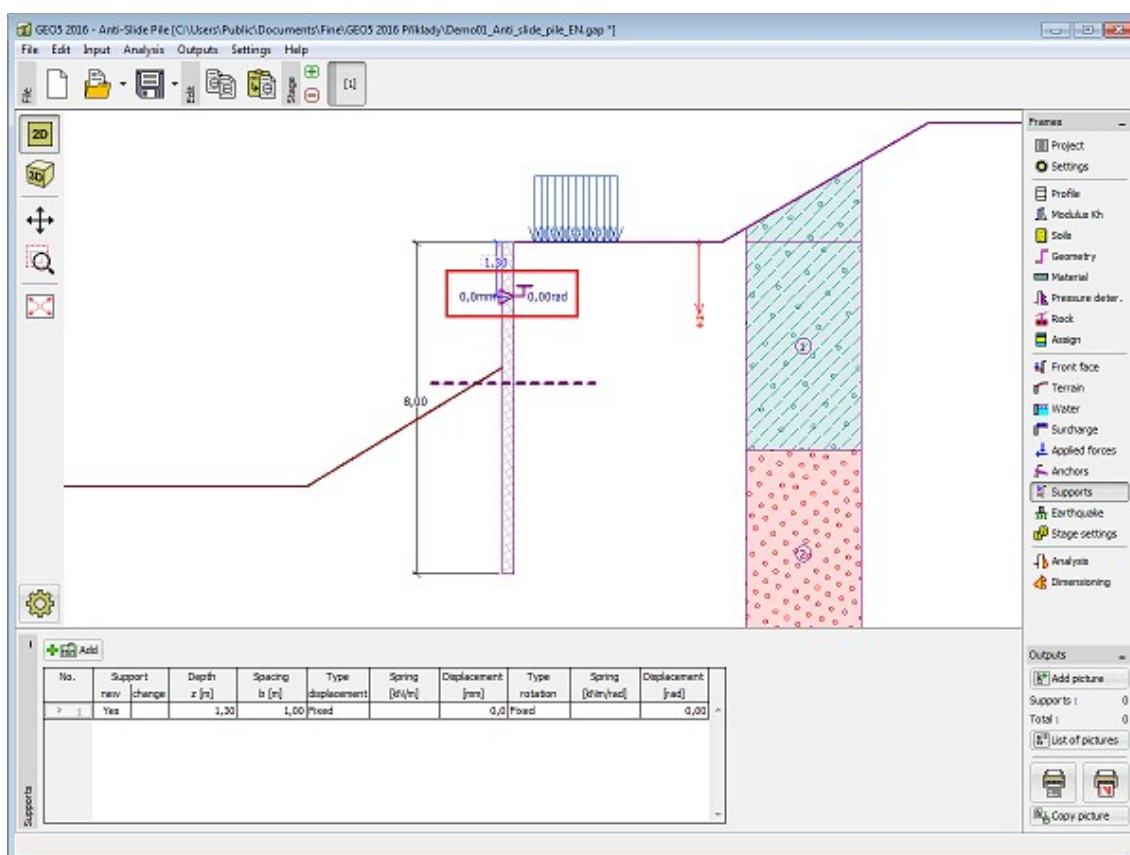
Frame "Anchors"

## Supports

The "**Supports**" frame contains a table with a list of input supports. Adding (editing) supports is performed in the "**New support (Edit support)**" dialog window. The input supports can also be edited on the desktop with the help of active dimensions or by active objects.

The support is automatically placed on already **deformed structure** (displacement is obtained from the previous construction stage).

In subsequent stages the supports can no longer be edited, it is only possible to input forced displacement of supports.



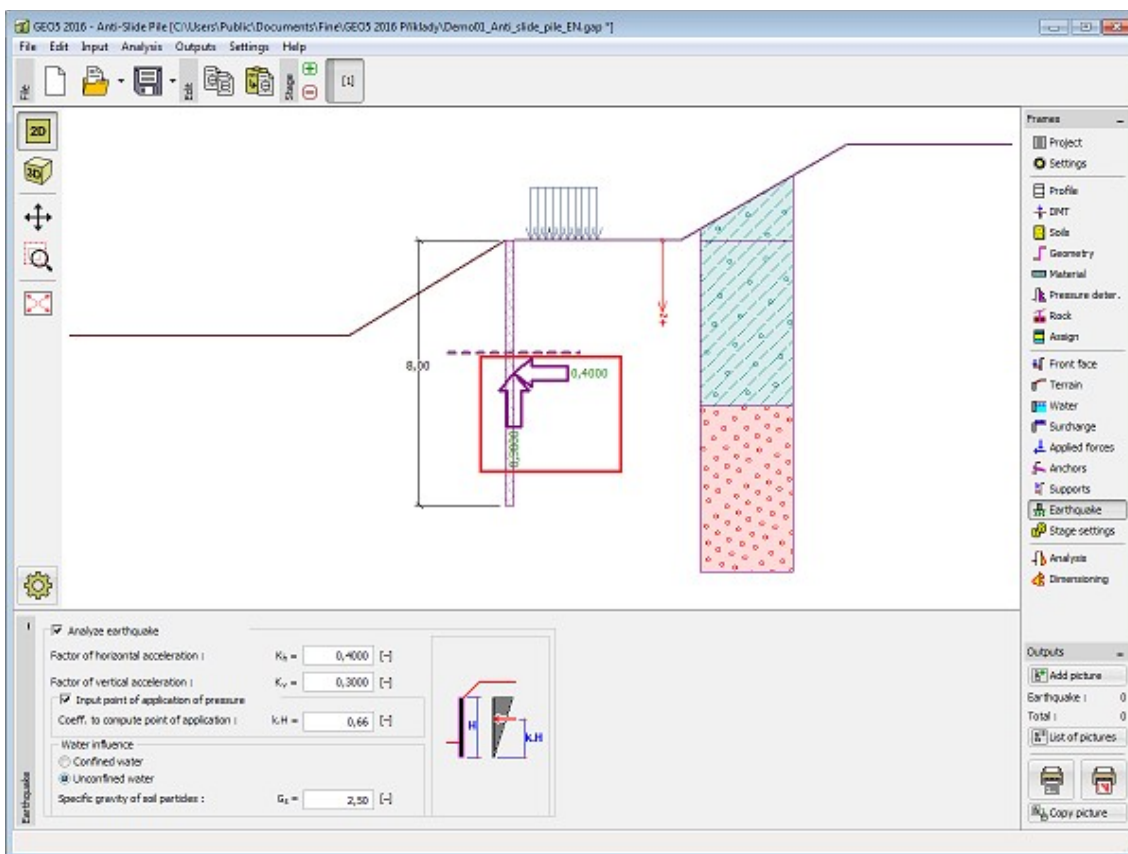
Frame "Supports"

## Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help in chapter "**Influence earthquake**".



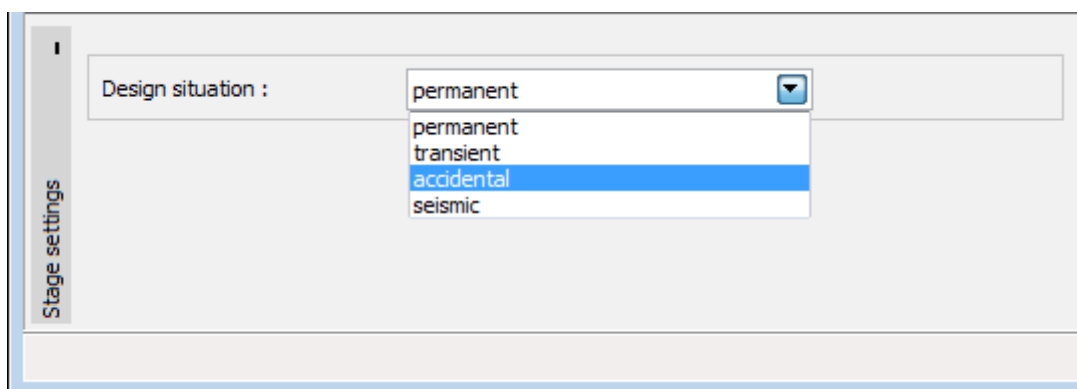
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Analysis

The frame "**Analysis**" displays the results of [analysis of the anti-slide pile](#). Switching to this mode automatically runs the analysis.

The maximum values of **internal forces and displacement** are shown in the bottom window. If the pile is [fixed in the rock](#), the check of the **rock bearing capacity** is performed.

The frame contains three options to show the analysis results, that can be changed by buttons in the right part of the frame.

- **Modulus of subgrade reaction + earth pressures**

Variation of the [modulus of subsoil reaction](#) is displayed in the left part of the desktop (by default a blue color with hatching) is assumed. Referring to the [method of depending pressures](#) some of the springs (values of modules of subsoil reaction) are removed (spring stiffness set equal to zero) from the analysis. The analysis **may fail to converge** providing the critical (limit) state developed both in front and behind the structure and there is not enough constrains available (anchors, supports). The program exists without finding a solution. An error message appears in the bottom part of the frame - such a case calls for **modification in problem input** - e.g., add an anchor, change a depth of excavation, improve soil parameters, etc.

Distributions of limiting pressures (by default a green dashed line is assumed) are presented in the right part of the window ([passive pressure](#), [pressure at rest](#) and [active pressure](#)). The **actual pressure acting on a structure** is plotted in a solid blue line.

Both **deformed** (by default a solid red color is assumed) and undeformed structure appears in the right part of the desktop. Forces and displacements developed in anchors, supports and props are also shown.

- **Moment + Shear force**

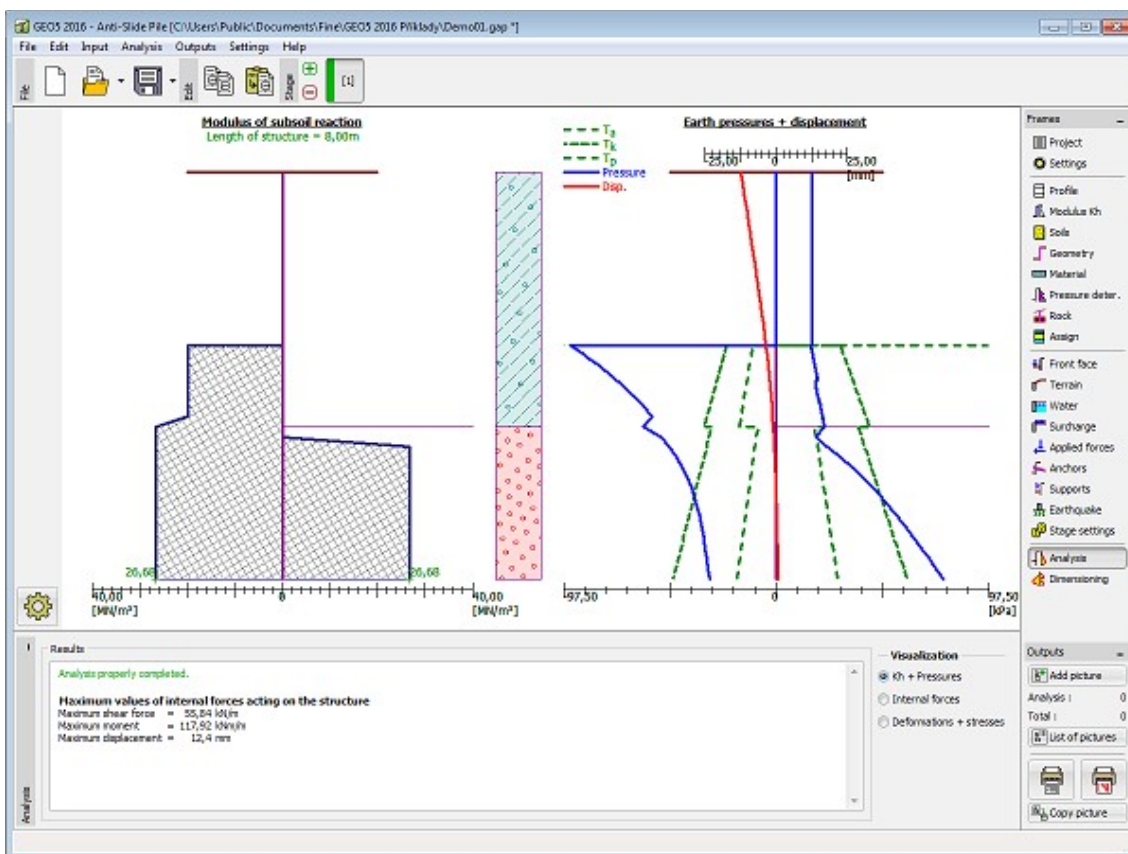
Plot of a structure together with forces acting in anchors, reactions and deformations of supports and props appear in the left part of the desktop. Distributions of bending moment and shear force are then plotted on the right.

- **Displacement + Stress**

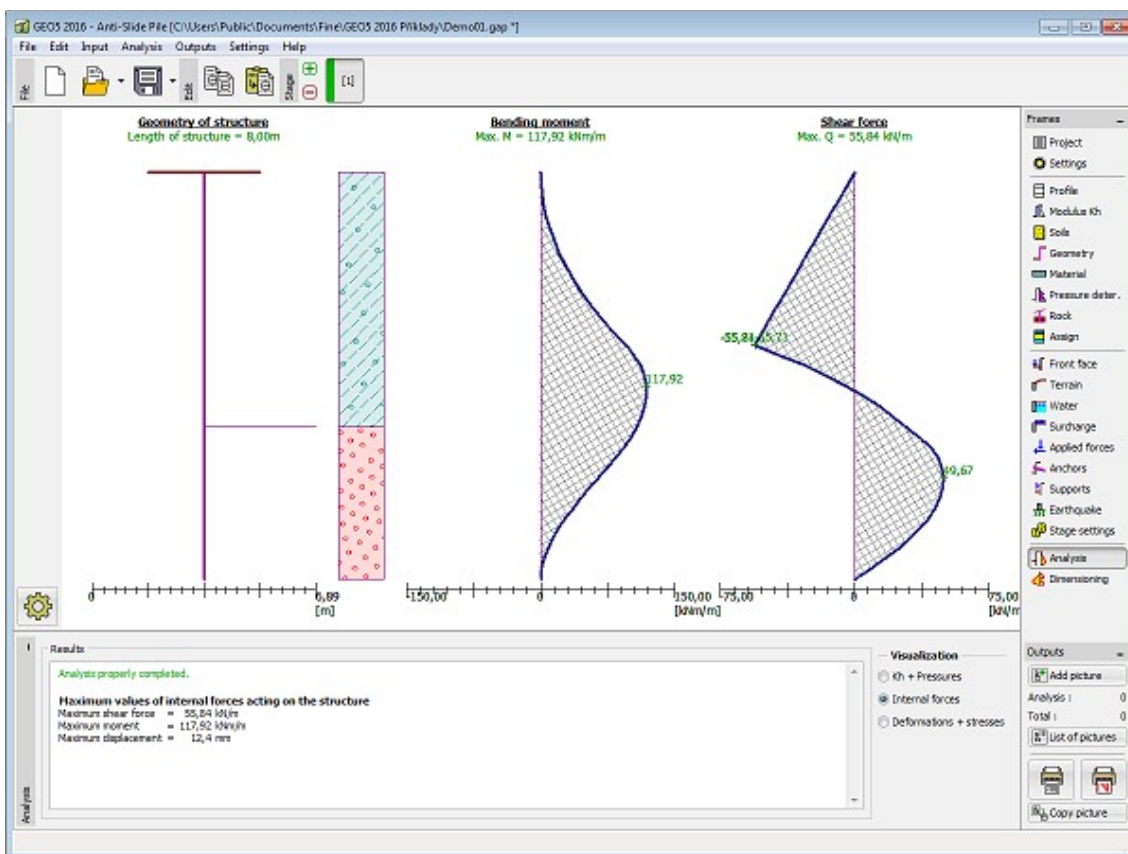
Plot of a structure together with forces acting in anchors, reactions and deformations of supports and props appear in the left part of the desktop. The deformed shape of a structure together with overall pressure acting on a [sheeting structure](#) is then plotted on the right.

Providing the modulus of subsoil reaction is found by iteration it is necessary to check the **course of manual iteration** in the dialog window "**Iteration**". Details are provided in the theoretical part of the help, chapter "[Modulus of subsoil reaction determined by iteration](#)".

Visualization of results can be adjusted in the frame "[Visualization settings](#)".

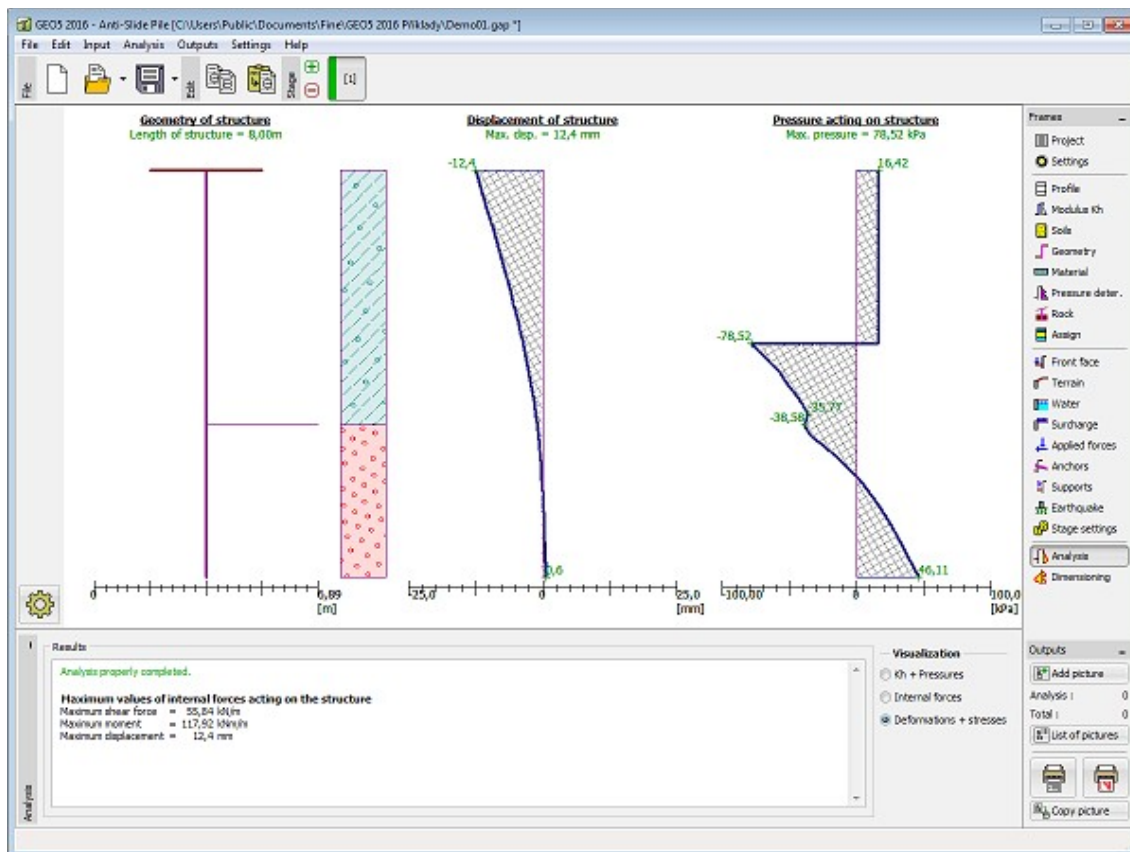


Frame "Analysis" - modulus of subsoil reaction, earth pressures and displacement



Frame "Analysis" - bending moment and shear force





Frame "Analysis" - displacement and earth pressure acting on structure

## Dimensioning

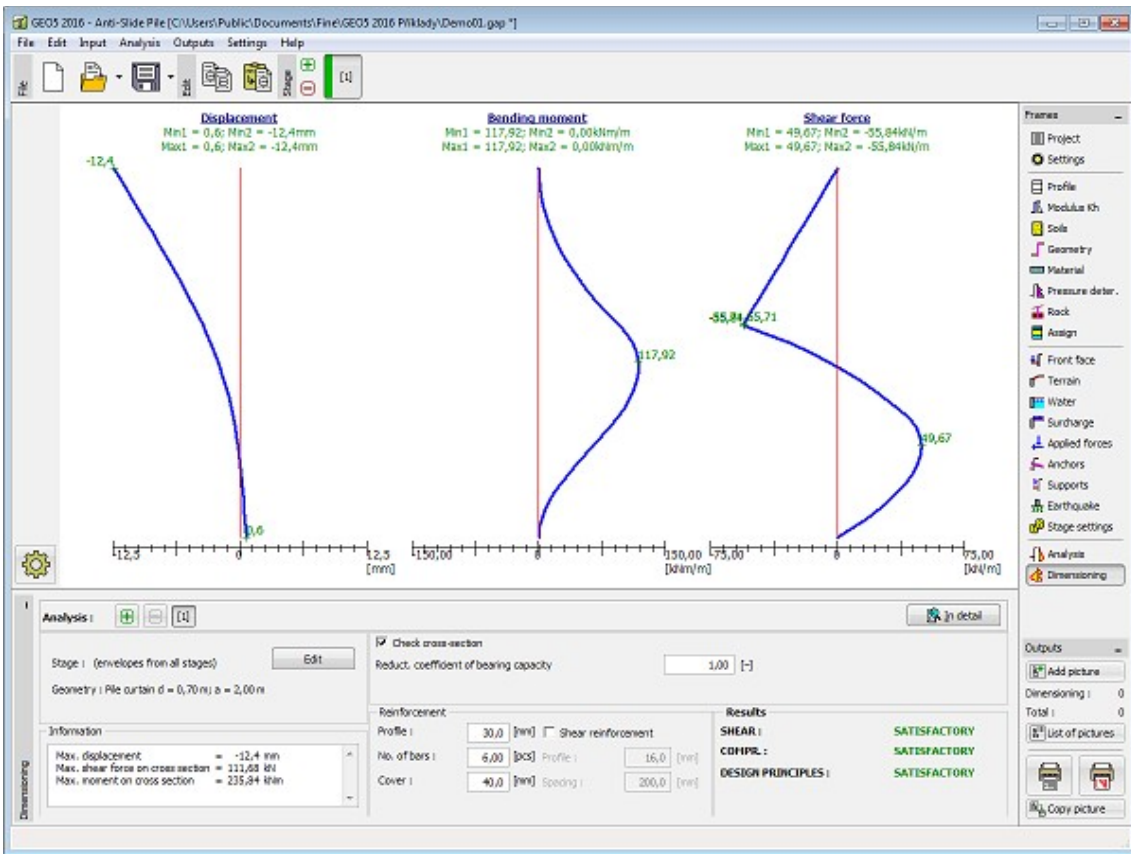
In the frame "Dimensioning", it is possible to display an envelope of internal forces and displacements from all analyses (stages of constructions). Normally, the envelope is constructed from the results of all construction stages, however, it can only be created from the **selected stages**. The "Modify" button opens the dialog window "Construction stage selection", where it is possible to select the constructions stages that are used to generate the current envelope (by pressing corresponding buttons).

The maximum values of calculated internal forces (bending moments and shear forces) and the magnitude of displacement are displayed at the bottom part of the frame.

The program allows to dimension steel-reinforced concrete and steel cross-sections (by checking the option "Check cross-section"). When checking the cross-section, it is possible to introduce the **reduction coefficient of bearing capacity**, which reduces the overall bearing capacity of a cross-section. The magnitude of this coefficient depends on the way, how active and passive forces (frame "Pressure determination") were computed. If these values are design values (already increased by partial factors), this coefficient should be 1.0 - if not, this coefficient should be higher then 1.0. (For EN 1997 is this value in interval 1.35 - 1.5).

The frame allows to perform a larger number of analyses pro dimensioning of a cross-section. The "In detail" button at the right part of the frame opens the "Dimensioning" dialog window to show detailed results.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Dimensioning"

## Program Shaft

This program is used to analyze spatial earth pressures on circular shaft and determination of internal forces on the structure.

**The help in the program "Shaft" includes the following topics:**

- Input of data into individual frames:

Project      Settings      Geometry      Profile      Soils      Assign  
 Water      Surcharge      Stage Settings Load Analysis      Dimensioning

- Standards and analysis methods

- Theory for analysis in the program "**Shaft**":

Geostatic Stress, Uplift Pressure      Strip Surcharge - Active Earth Pressure      Concentrated Surcharge - Active Earth Pressure      Line Surcharge - Active Earth Pressure      Concentrated Shaft Surcharge - Earth Pressure at Rest

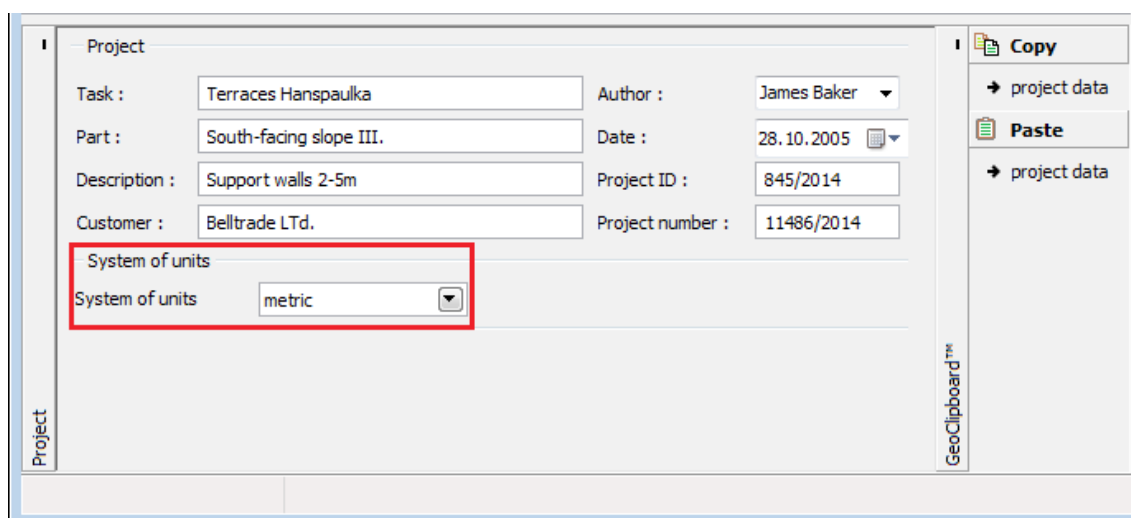


- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.



The screenshot shows the 'Project' frame with the following data:

Field	Value
Task	Terraces Hanspaulka
Author	James Baker
Part	South-facing slope III.
Date	28.10.2005
Description	Support walls 2-5m
Project ID	845/2014
Customer	Belltrade LTd.
Project number	11486/2014
System of units	metric

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

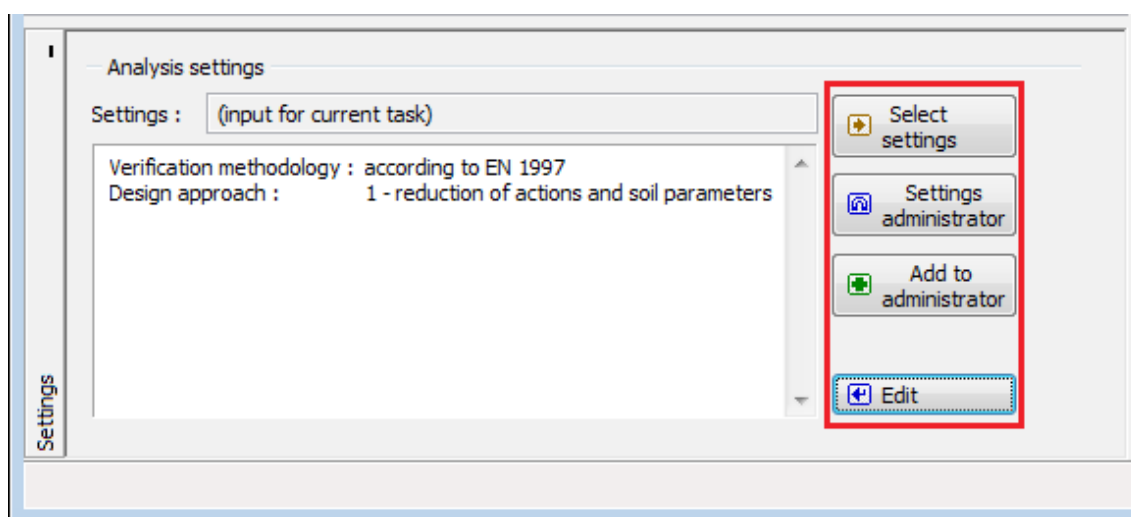
The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Excavations**" tab.



Frame "Settings"

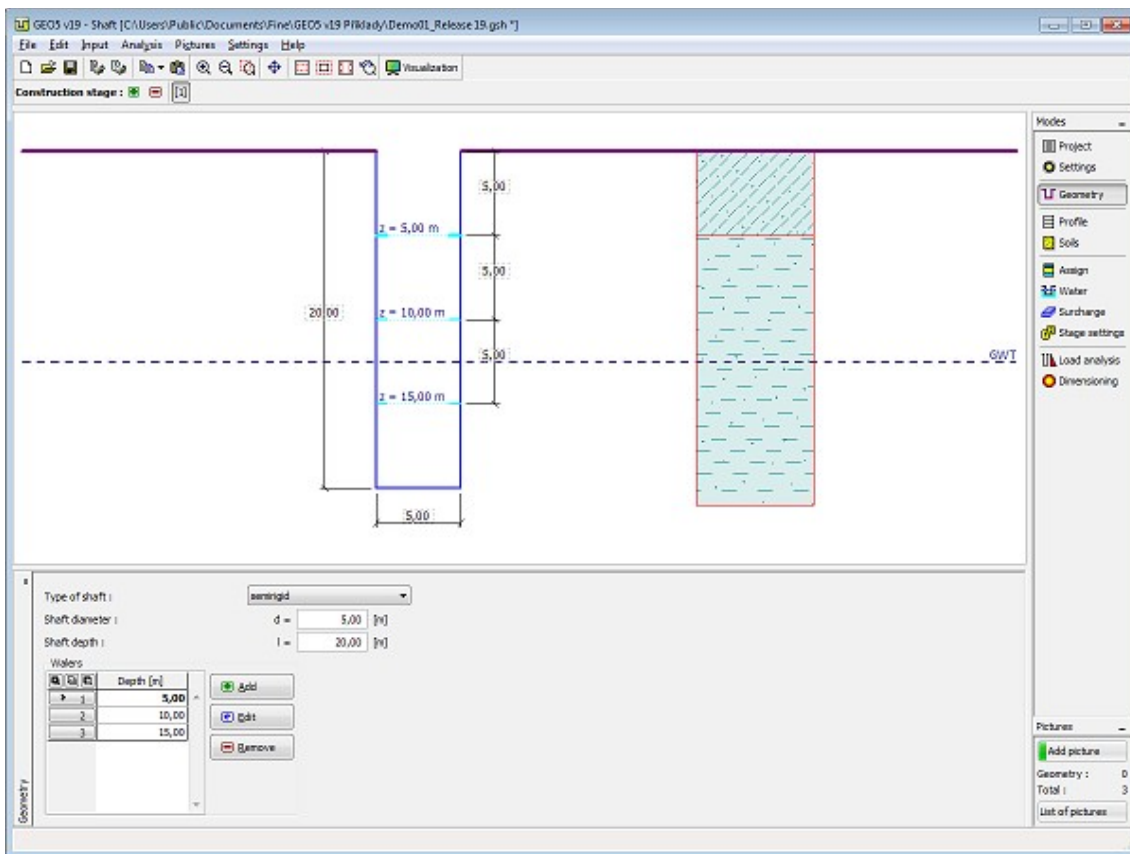
## Geometry

In the **frame "Geometry"**, the **shaft type** (**flexible**, **semirigid**, **rigid**) is selected and **shaft diameter**  $d$ , **shaft depth**  $l$  and **depth of walers**.

The frame contains a **table** with a list of waler depths. **Adding** (modification) of segments is made in "**New waler**" or "**Edit waler**" **dialog window**.

Dimensions of the structure and depth of walers can be modified in the frame by editing values in the input fields or on the desktop by using **active dimension**.

The program allows to **export** the geometry of a structure in \*.DXF format.



Frame "Geometry"

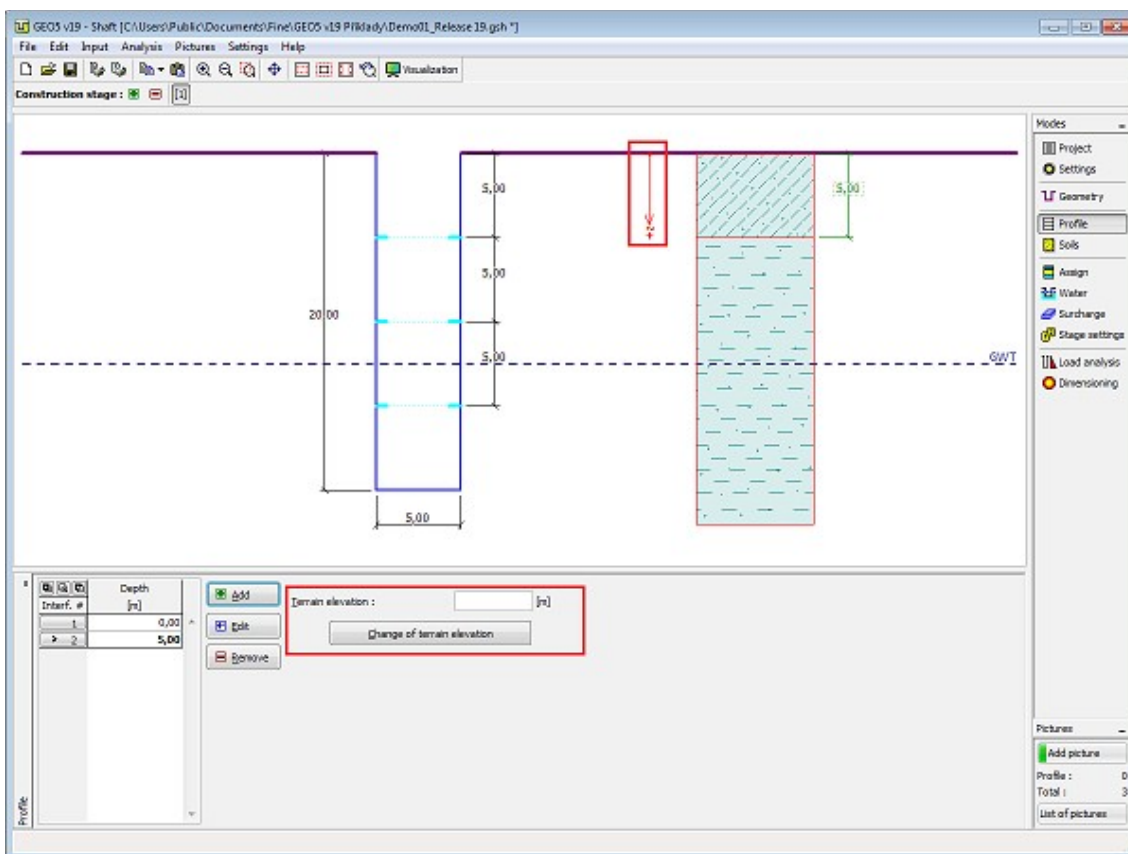
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "**New interface**" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to import a profile in the gINT format.



Frame "Profile"

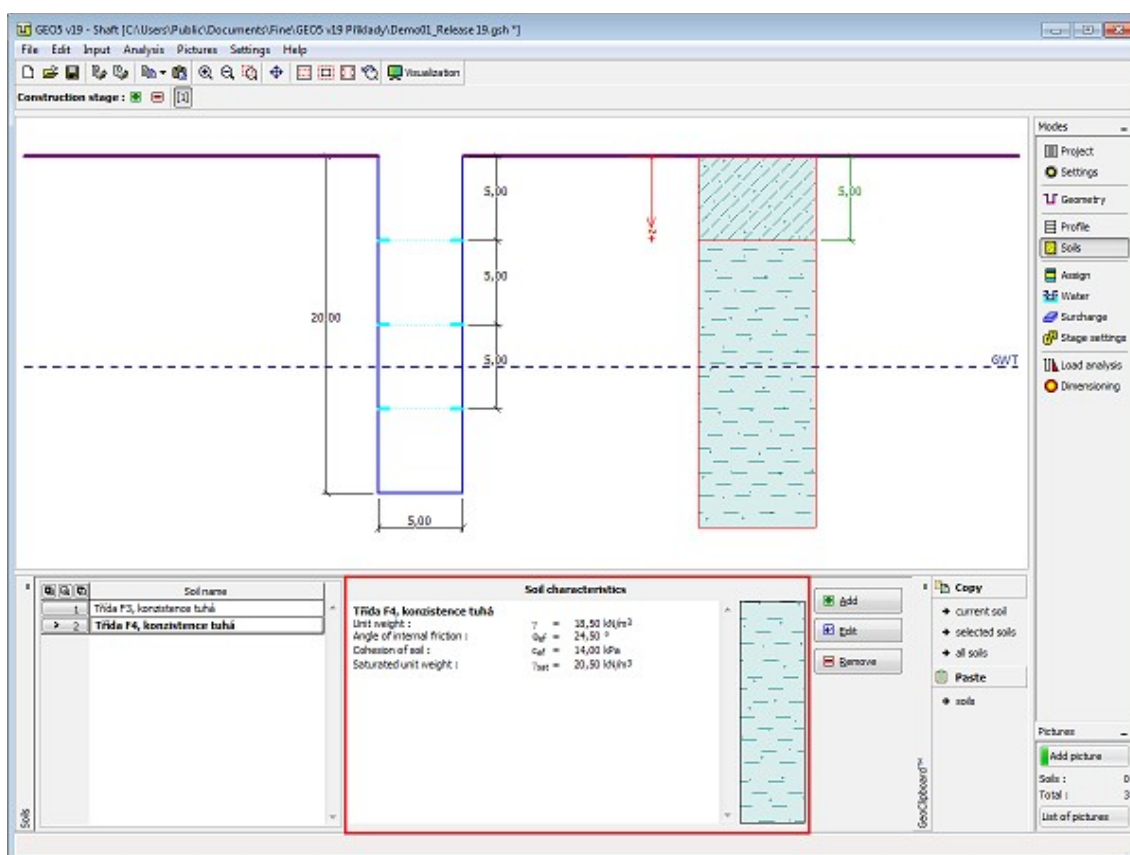
## Soils

The **"Soils"** frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the **"Add new soils"** dialog window.

The soil characteristics needed in the program are further specified in the following chapters: **"Basic data"** and **"Uplift pressure"**.

The program makes it possible **import soils in the gINT format**. Data of input soils can be copied within all GEO5 programs using **"GeoClipboard"**.

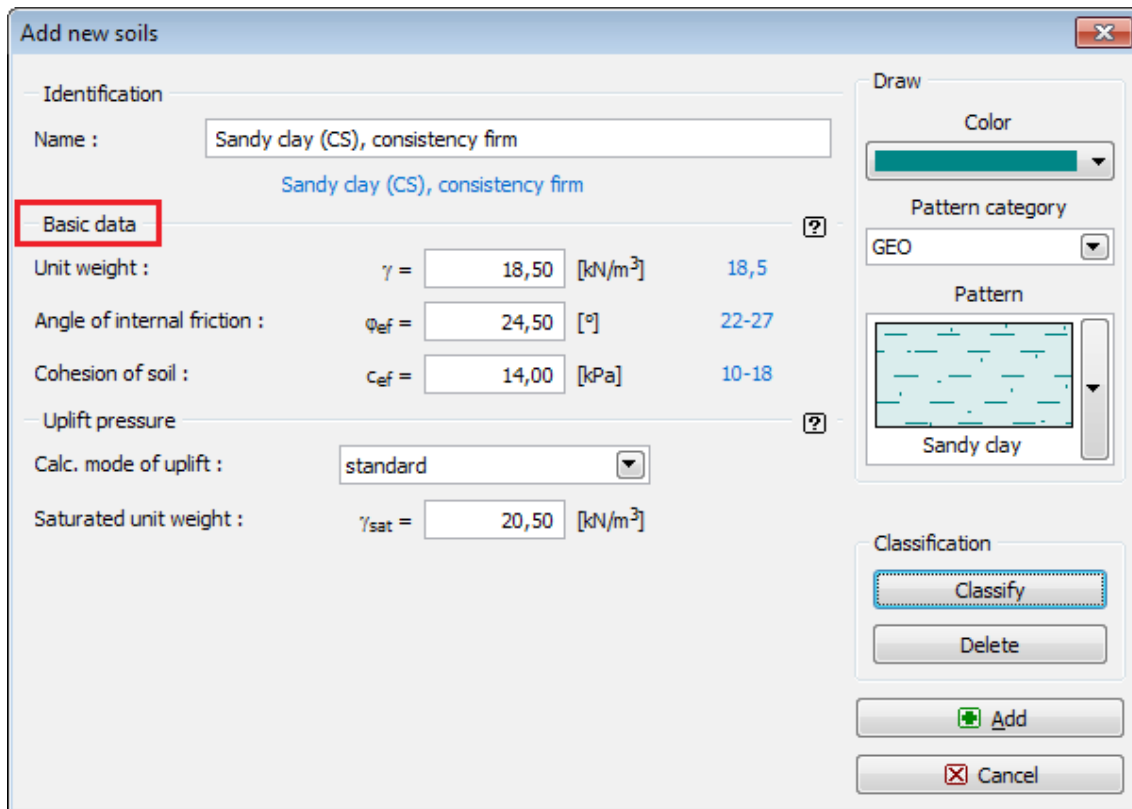


Frame "Soils"

## Basic data

This part of the window introduces basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from a geotechnical survey or laboratory experiments. If these data are not available, it is possible to exploit built-in [database of soils](#), which contains the values of selected soil characteristics. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

The associated theory is described in detail in chapter "[Earth pressures](#)".



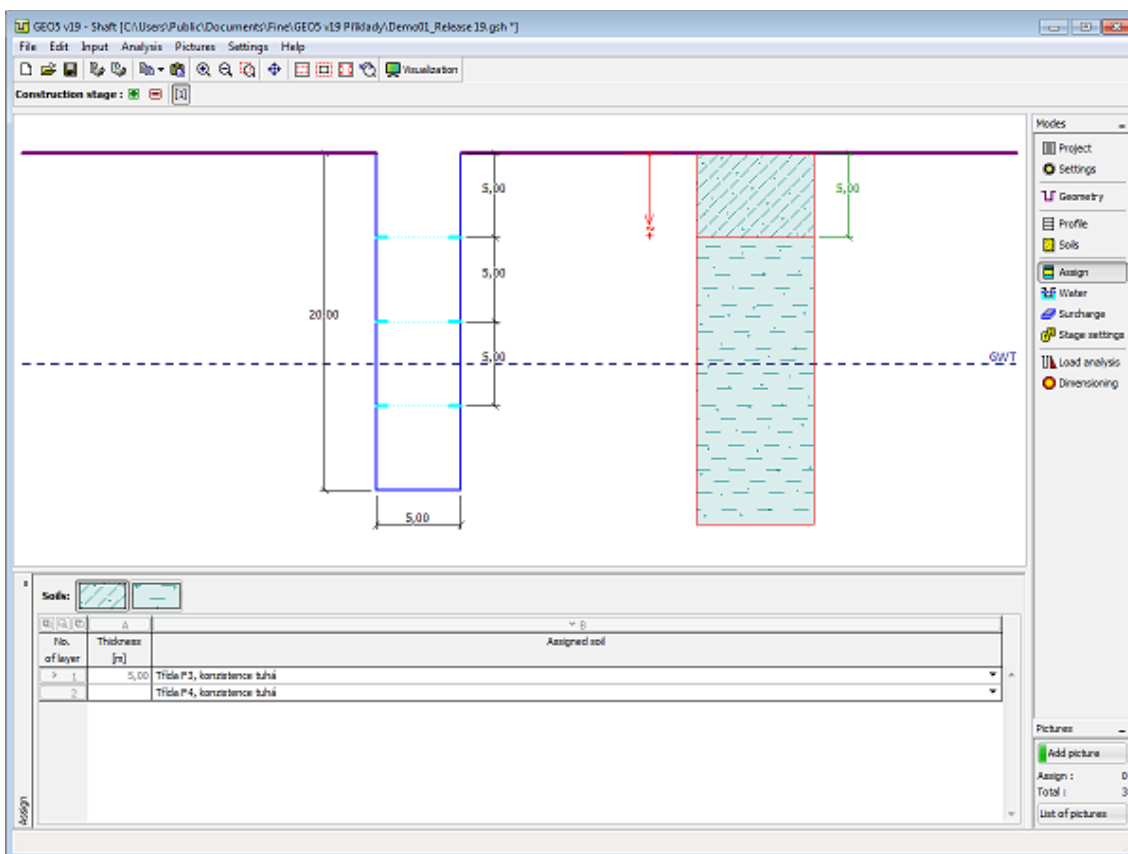
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



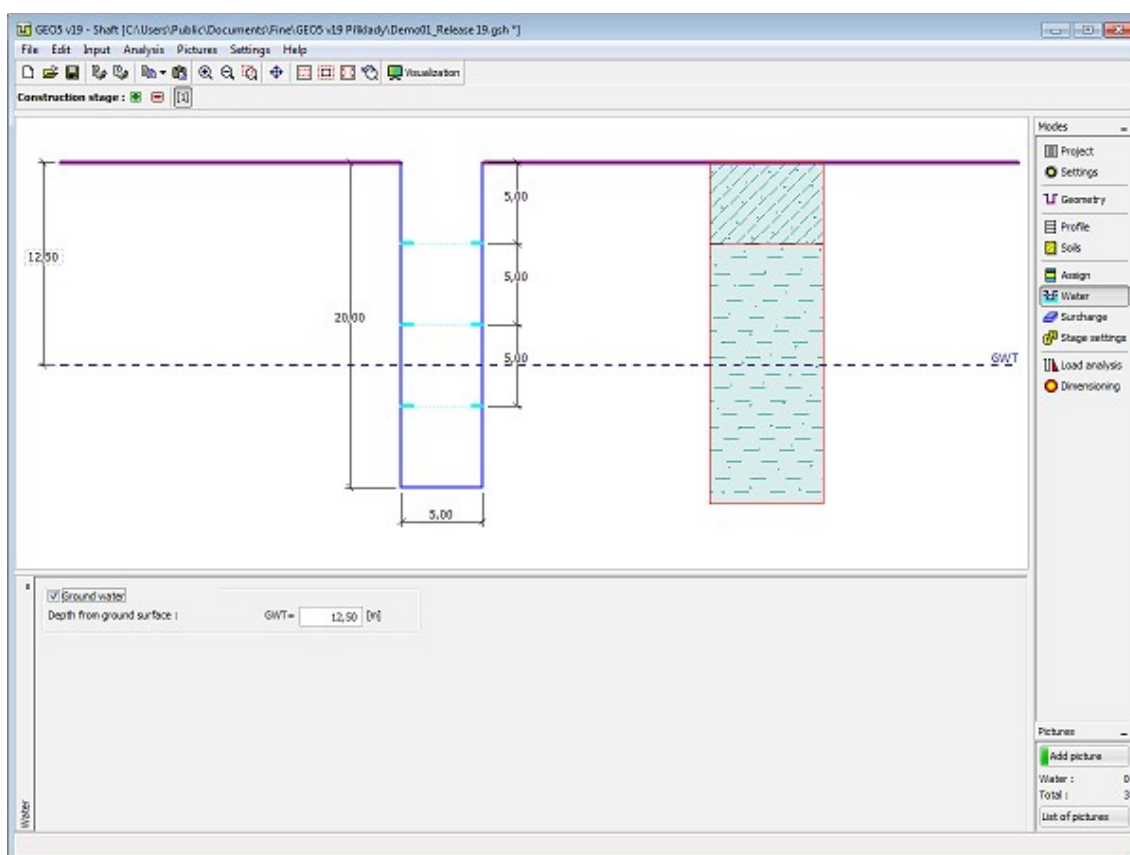
Frame "Assign"

## Water

The frame "**Water**" serves to enter a **depth of ground water table**.

The values can be edited either in the frame by entering values into particular fields, or on the desktop with the help of **active dimensions**.

The **GWT** changes the **geostatic stress** in the soil profile.



Frame "Water"

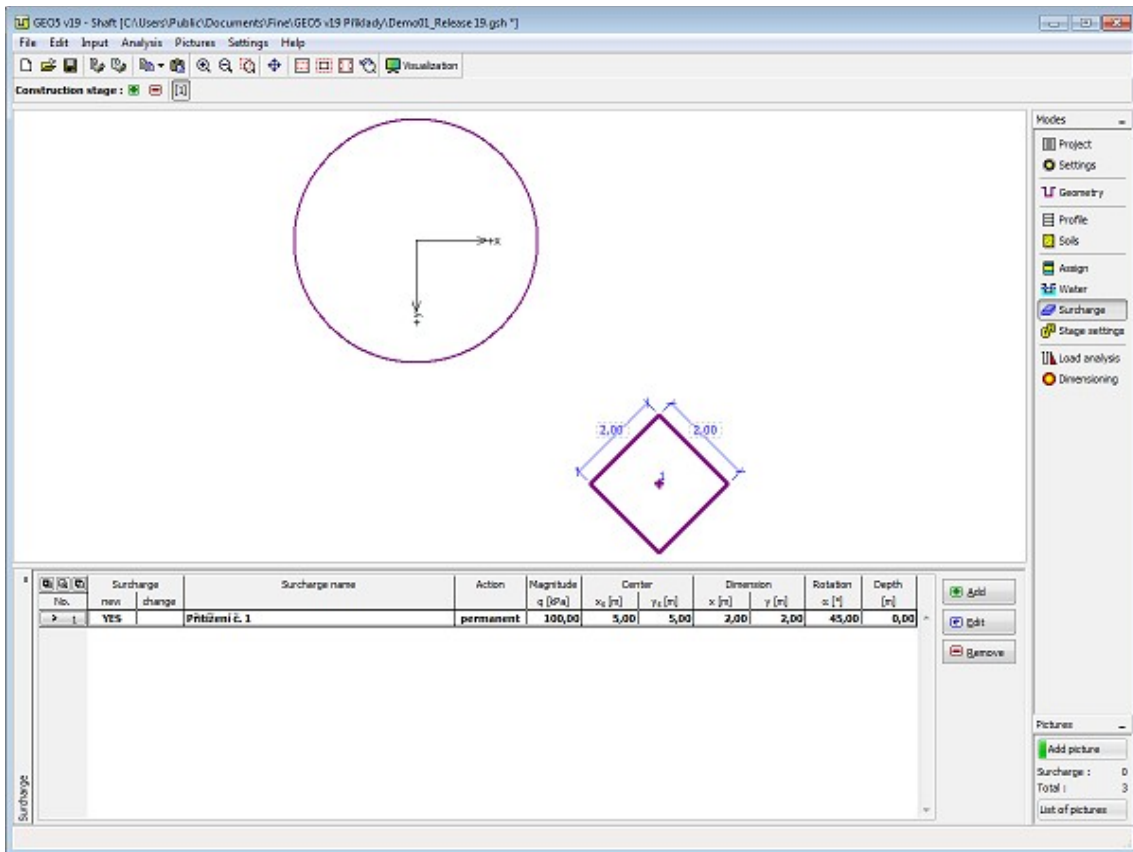
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can also be edited on the desktop with the help of active objects.

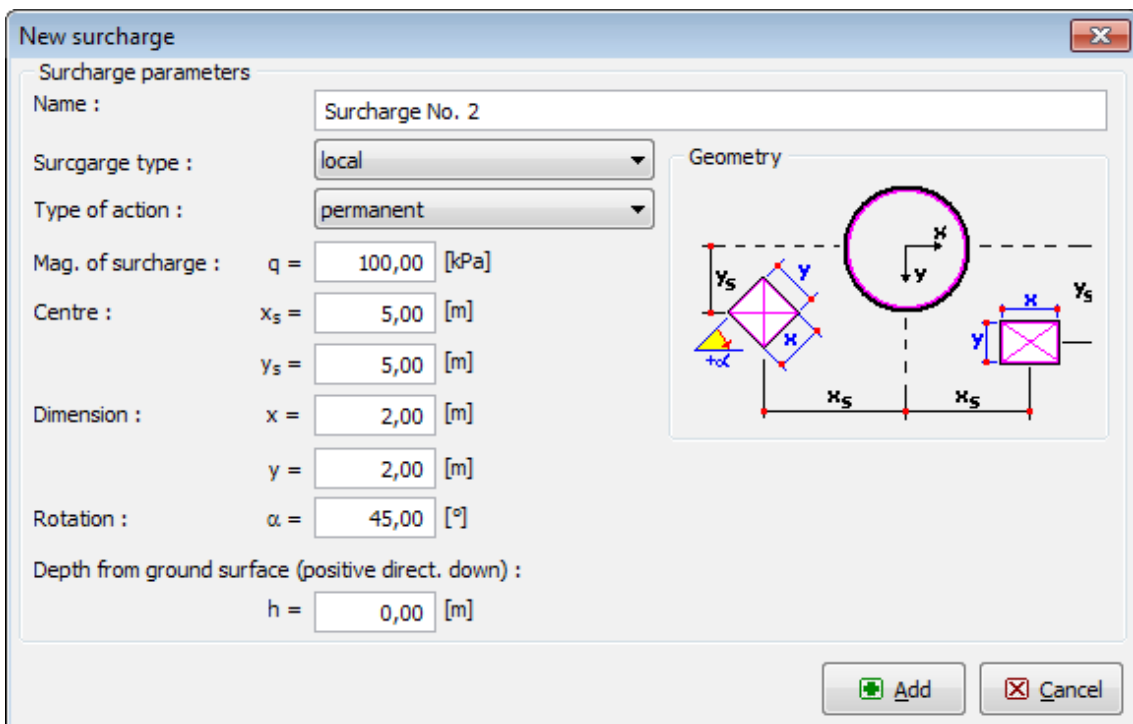
It is possible to input surcharge as **permanent**, **variable** or **accidental**. Type of surcharge is either **surface** or **local**. The final effect is multiplied by corresponding verification coefficient according to the type of surcharge.

In case of considering surcharge in a different depth than on surface (for example foundation of surrounding buildings), depth  $h$  under the surface is input (positive direction downwards).





Frame "Surcharge"



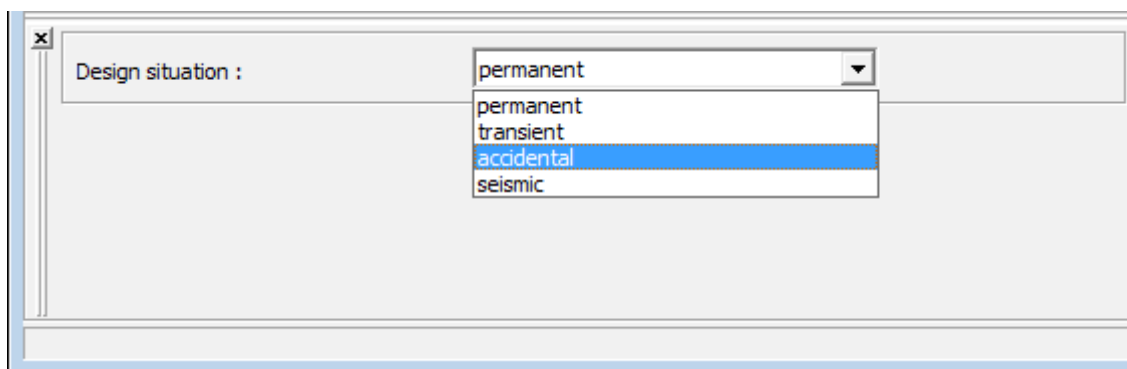
Dialog window "New surcharge"

## Stage settings

The frame "**Stage settings**" serves to input settings valid for a given construction stage.

Selected **design situation** determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected **verification methodology**. LRFD 2012 introduces new types of **design situations** (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Load Analysis

The frame "**Load analysis**" allows to determine the final load in **vertical direction**, final load on the **walers** or in the **input depth** (for shafts without walers). Program computes all partial loads, multiplies them by corresponding partial factors and shows the final load on the screen.

Calculated uniform earth pressure can be modified (in the compliance with DIN or SNIP standards) by reduction coefficient, so the former "**circular**" load is changes to "**elliptical**". It is possible to specify the way of modification (increase and decrease the load, only decrease the load) and the value of the **reduction coefficient** (recommended value is 25 %).

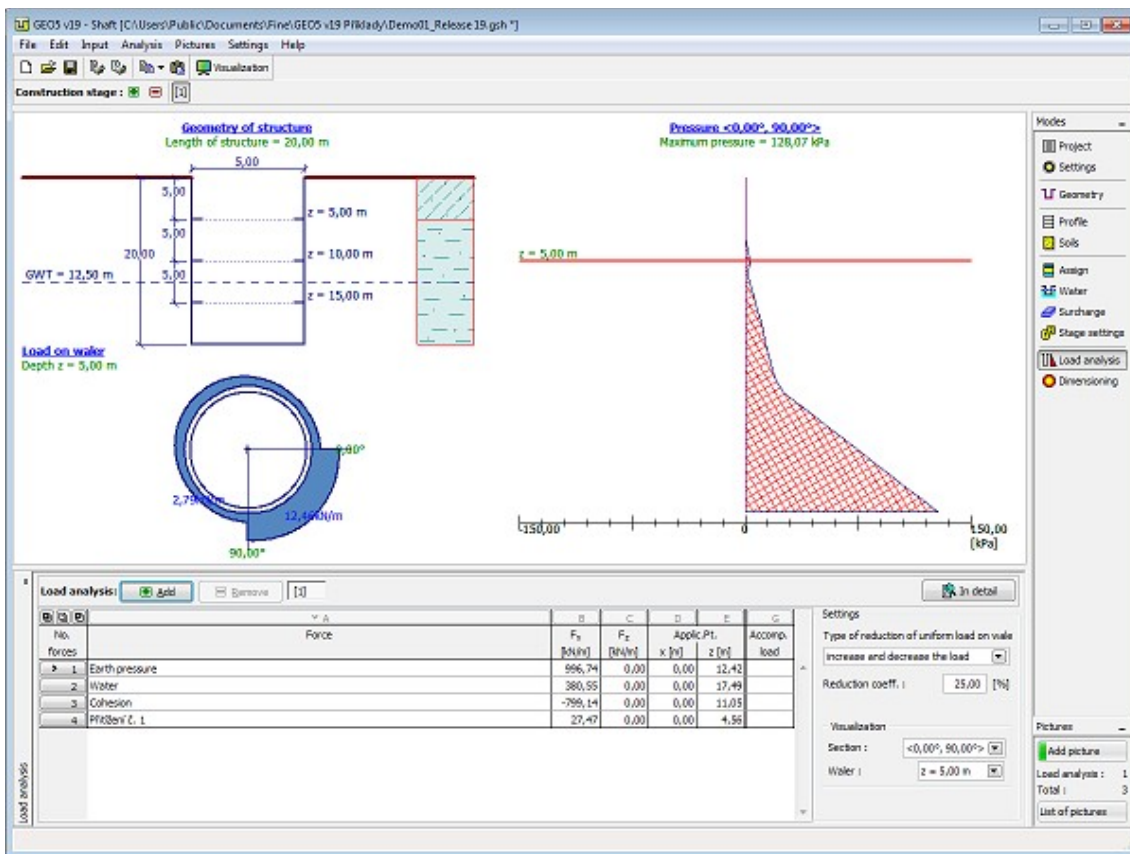
Computed **load** is the input for analysis of **internal forces** in the frame "**Dimensioning**". Program automatically computes the load on all walers or in input depth (shaft without walers).

**Several computations** can be carried out for a single task. This is very useful for determination of combinations of load cases - in the frame "**Dimensioning**" you can work with all here computed combinations.

The frame appearance is adjusted based on the selected **verification methodology**:

- Verification according to the **safety factor**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiplies the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

The **analysis results** are displayed on the desktop and are updated immediately for an arbitrary change in input data or setting. Visualization of results can be adjusted in the "**Visualization style settings**" dialog window.



Frame "Load analysis"

## Dimensioning

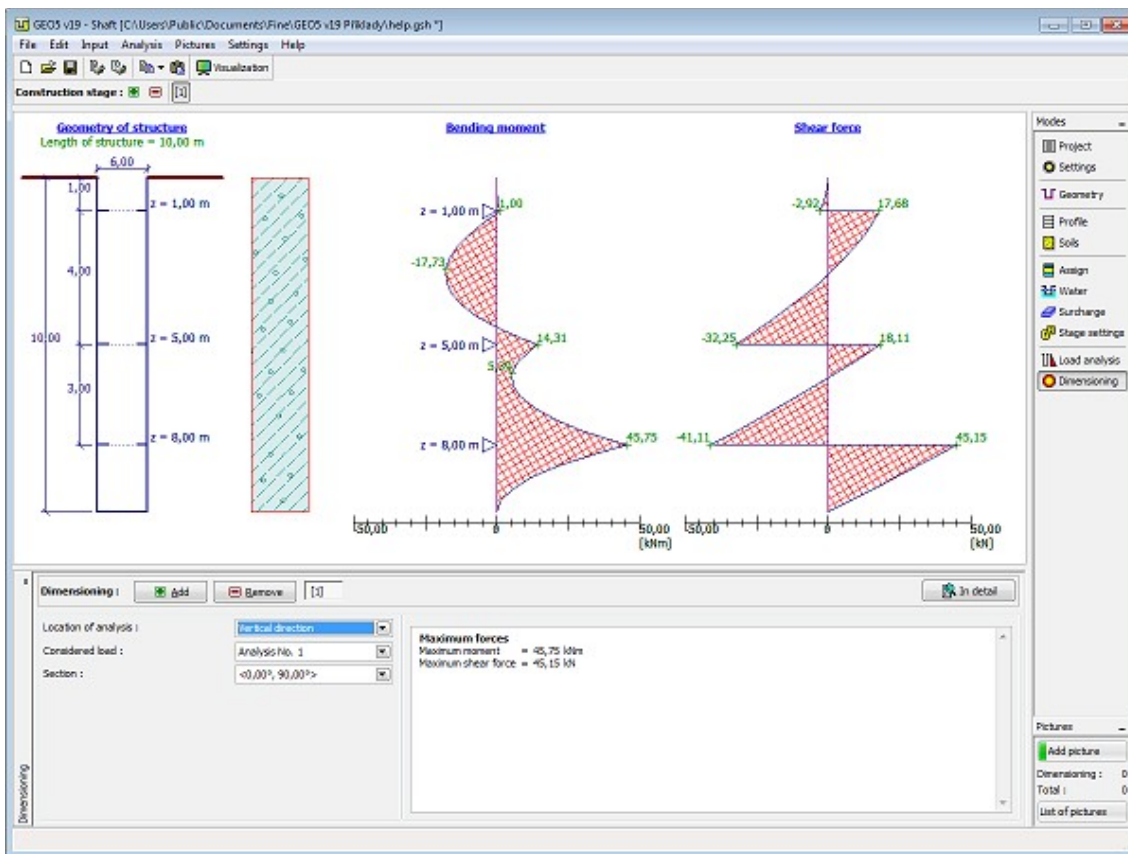
In the frame "**Dimensioning**", the program computes the internal forces on the shaft for the load, already defined in the frame "**Analysis**". Two types of analysis can be performed:

- internal forces in vertical direction
- internal forces on walers or in specified depth (for shafts without walers)

The analysis is performed for the selected load or the envelopes of internal forces computed for all specified loads (load cases).

Several computations can be carried out for a single task. Maximum values of internal forces are displayed in the output window. The "**In detail**" button opens a dialog window that contains detailed listing of dimensioning results.

When analysing in vertical direction, the program allows to select specific section (considering local surcharge is input, so more sections are available) or compute envelope of internal forces for all sections.



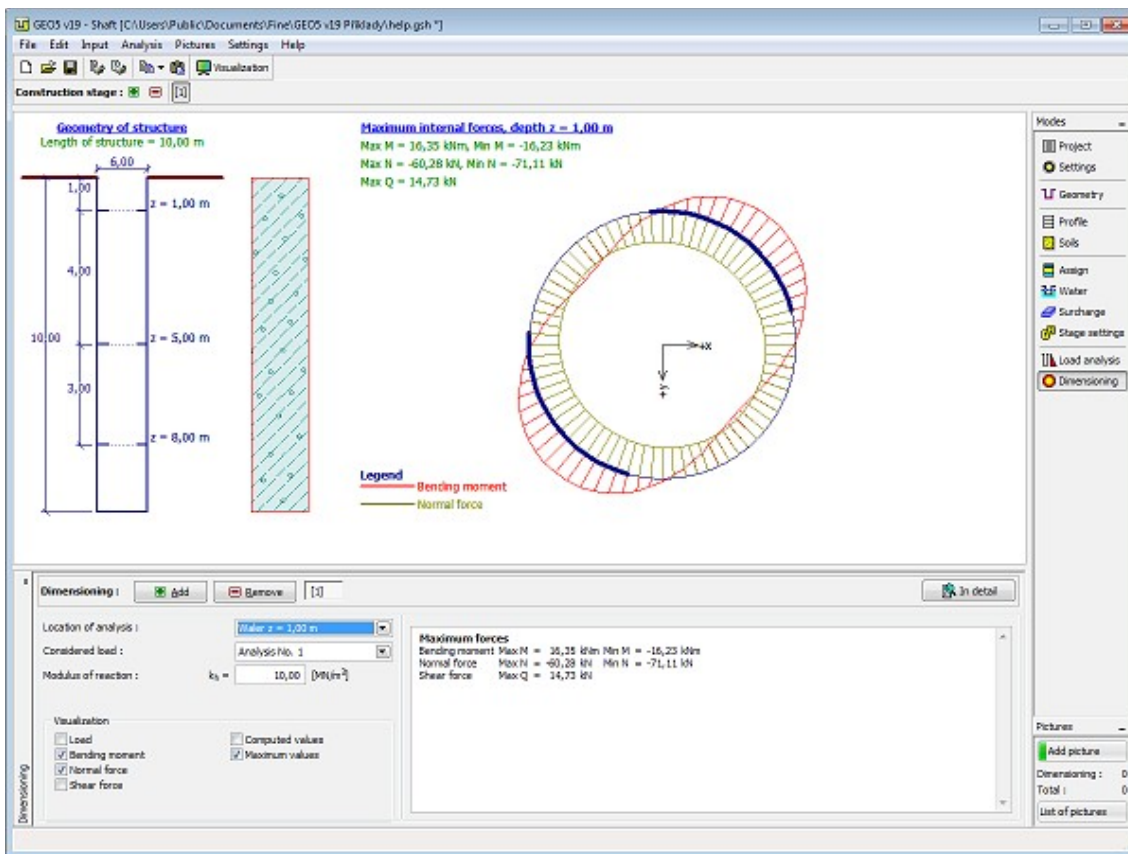
Frame "Dimensioning" - vertical direction

When analysing in horizontal direction (Internal forces on the water or in input depth), the **modulus of subsoil reaction** has to be inputed for the soil in a specified depth.

The program shows the load, bending moment and shear force - the rendering form is defined in the part "**Visualization**". The "**Maximum values**" button hides all values except the maximum values.

Part of the picture is marked with a bold line - this the the part of the shaft, that deforms into soil. In this part, the subsoil springs are considered in the analysis by polygonal method.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Dimensioning" - waler

## Program Slope Stability

This program is used to perform slope stability analysis (embankments, earth cuts, anchored retaining structures, MSE walls, etc.). The slip surface is considered as circular (Bishop, Fellenius/Petterson, Janbu, Morgenstern-Price or Spencer methods) or polygonal (Sarma, Janbu, Morgenstern-Price or Spencer methods).

**The help in the program "Slope Stability" includes the following topics:**

- Input of data into individual frames:

Project	Settings	Interface	Embankmen	Earth Cut	Soils	Rigid Body
Assign	Anchors	Reinforceme	Anti-Slide	Surcharge	Water	Earthquake
Stage	Analysis	nts	Piles			
Settings						

- Standards and analysis methods

- Theory for analysis in the program "**Slope Stability**":
  - Stress in Soil Body   Parameters of Rocks   Anti-Slide Pile   Slope Stability
- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".

*Frame "Project"*

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows to choose an already created setting from the "**Settings list**".

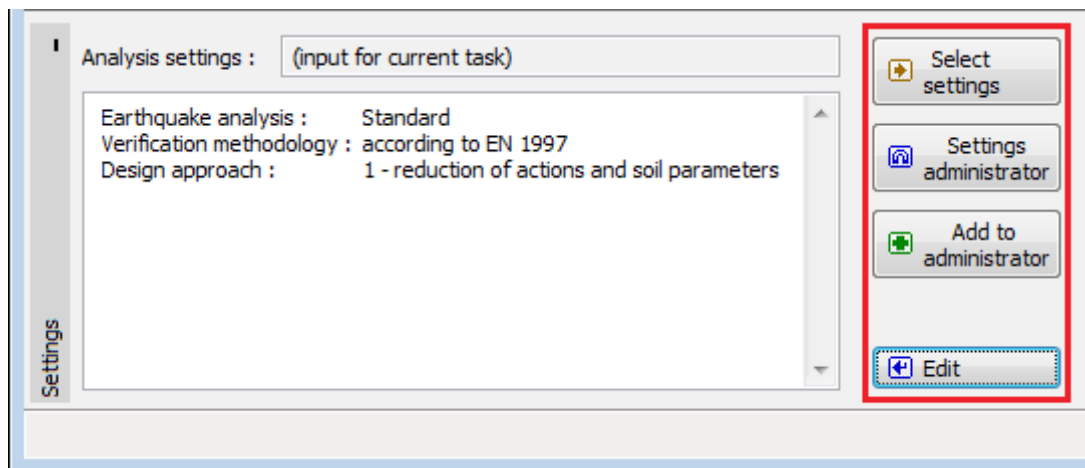
The "**Settings Administrator**" button opens the "**Administrator**" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Materials and standards**" and "**Stability analysis**" tabs.



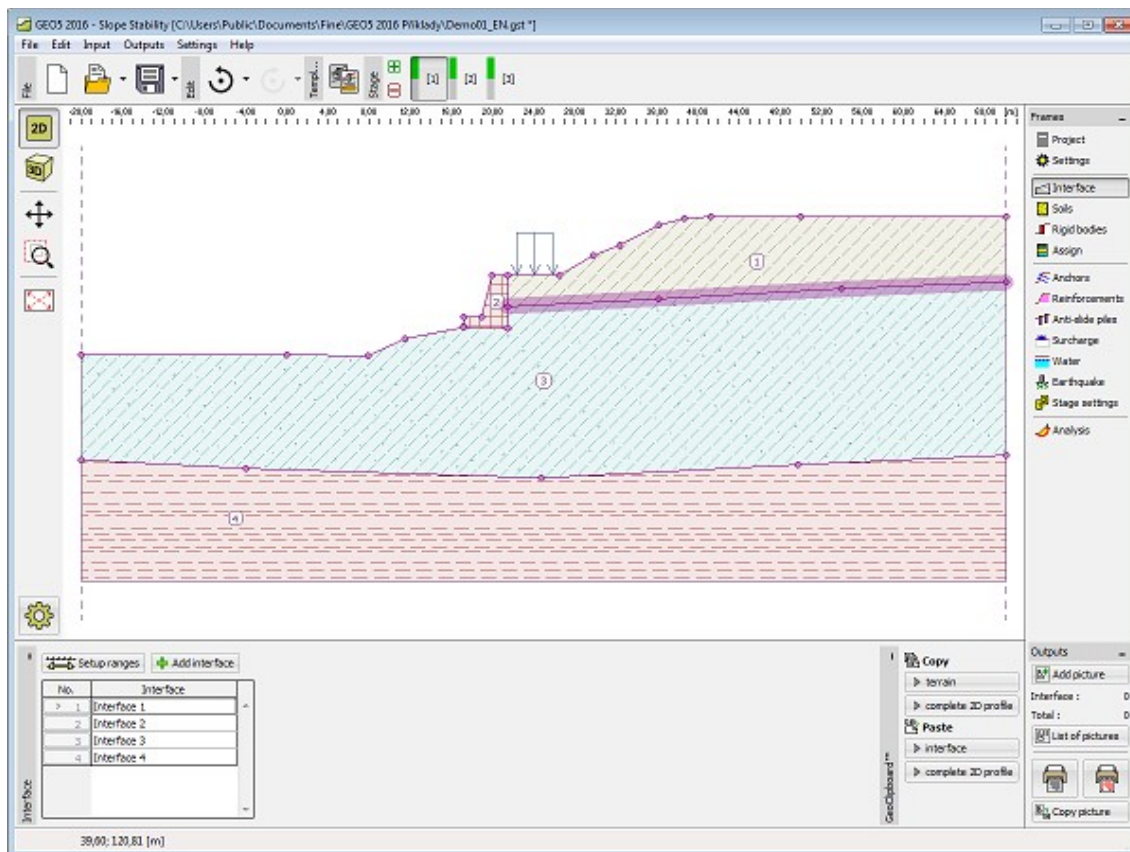
Frame "Settings"

## Interface

The "**Interface**" **frame** serves to introduce individual soil interfaces into the soil body. Detailed description how to deal with interfaces is described **herein**.

The program makes it possible to **import or export** interfaces in the \*.DXF format. They can also be **imported in the gINT format**. Input interfaces can be copied within all 2D GEO5 programs using "**GeoClipboard**".





Frame "Interface"

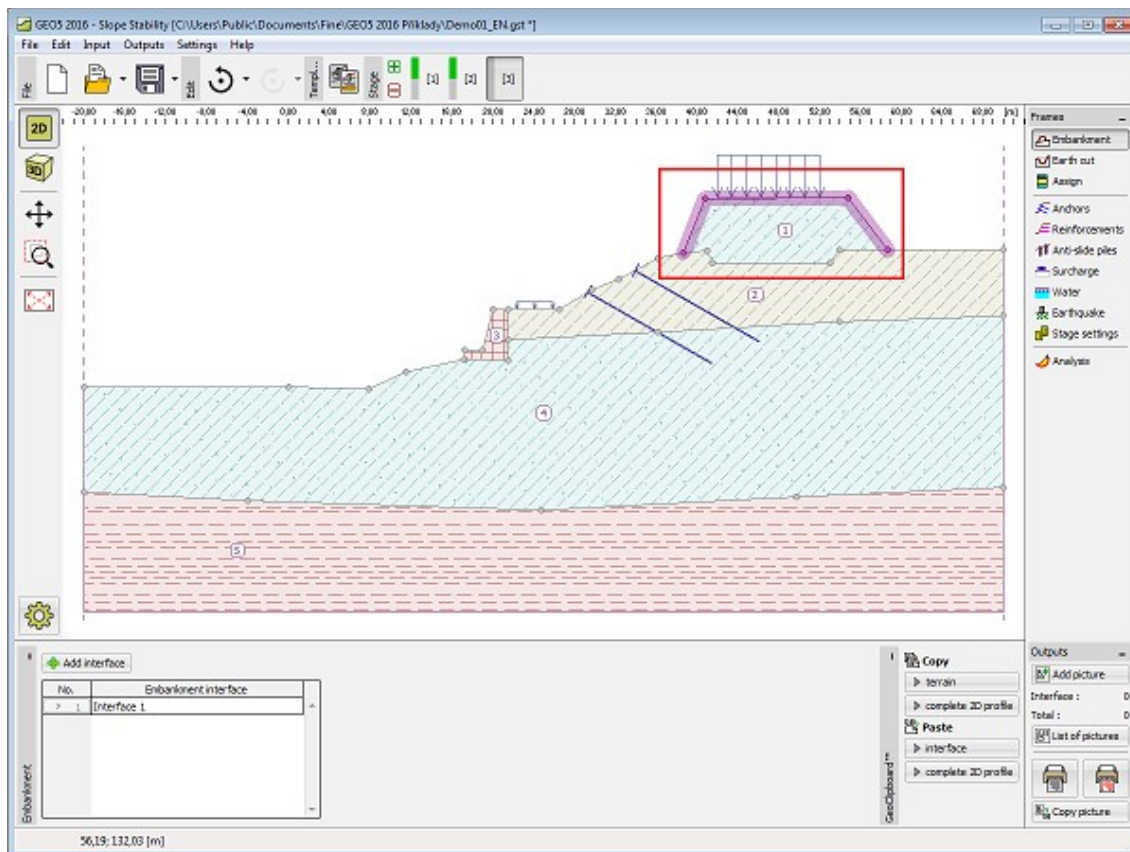
## Embankment

The "**Embankment**" frame allows to **input interfaces** to create an embankment above the current terrain. The frame contains a **table** with a list of interfaces forming the embankment. A table listing the points of currently selected interface of the embankment is displayed in the mid section of the frame. Inputting an embankment interface follows the same steps as used for **standard interfaces**.

An embankment cannot be specified in the first **stage of construction**. An embankment cannot be built if there is an **earth cut** already specified in a given stage - in such a case either a new stage of construction must be introduced for embankment input or the existing open cut must be first removed.

Input interfaces of an embankment can be copied within all 2D GEO5 programs using "**GeoClipboard**".





Frame "Embankment"

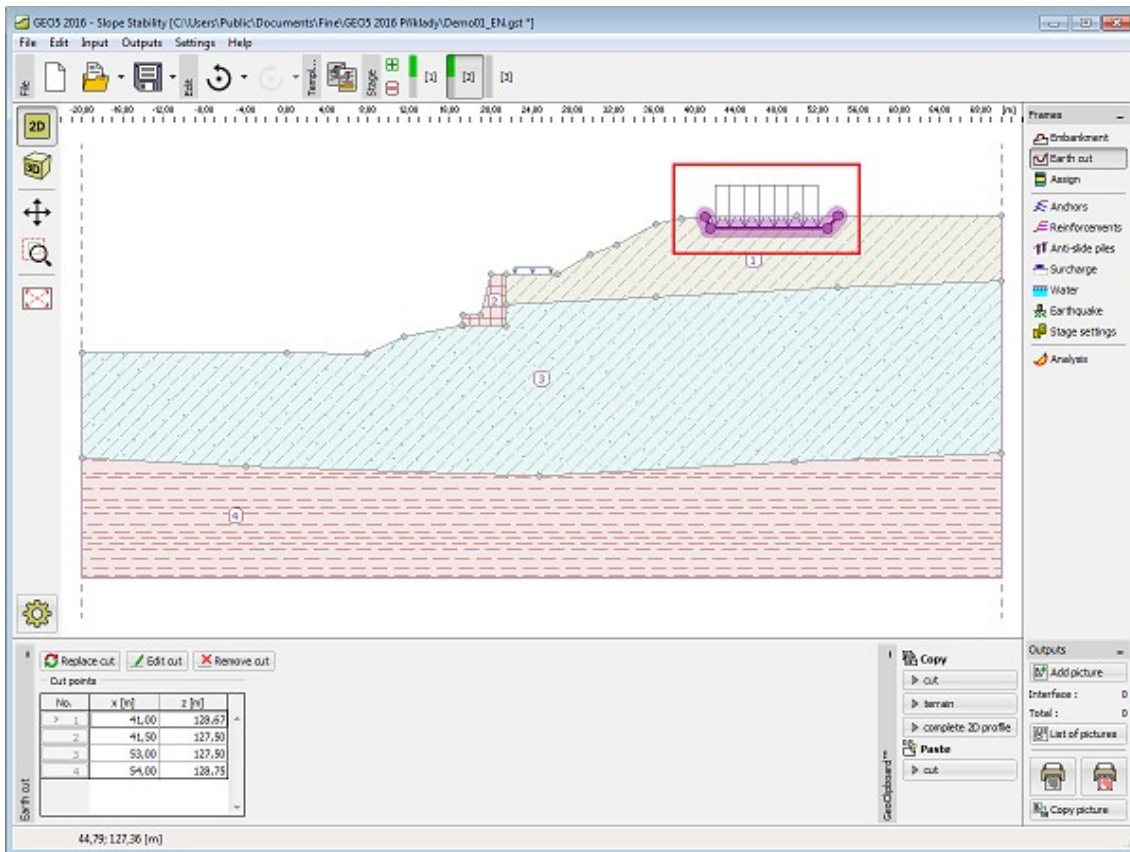
## Earth Cut

The "**Earth cut**" frame serves to specify the shape of an open cut. This function allows to modify the terrain profile within a given **stage of construction**. Several **earth cuts** can be introduced at the same time. In such a case some of the lines in the cut appear partially above the terrain.

A **table** listing individual interface points is displayed in the left part of the frame. Inputting an earth cut interface follows the same steps as used for **standard interfaces**.

An open cut cannot be specified in the first **stage of construction**. An earth cut cannot be built if there is an **embankment** already specified in a given stage - in such a case either a new stage of construction must be introduced for earth cut input or the existing embankment must be first removed.

Input interfaces of an earth cut can be copied within all 2D GEO5 programs using "**GeoClipboard**".



Frame "Earth cut"

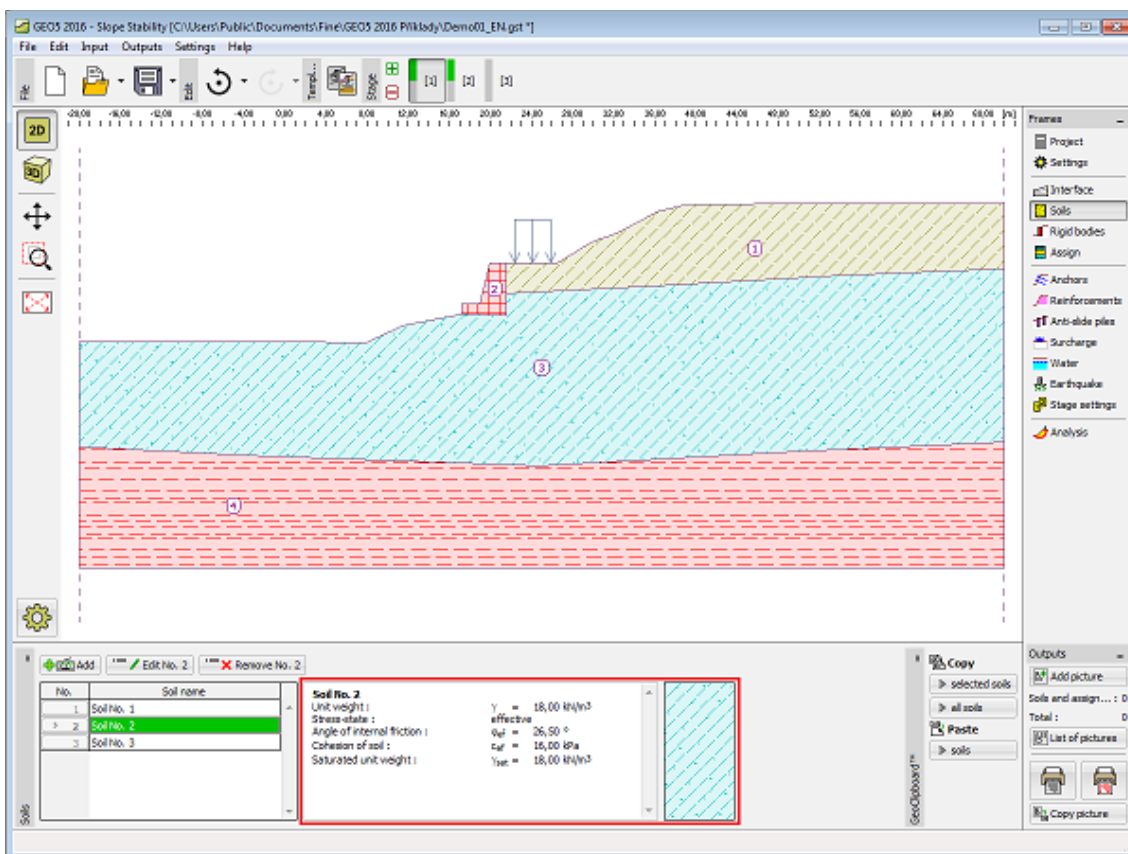
## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Uplift pressure", "Foliation" and "Parameters for rapid draw down". An input of parameters further depends on the selected type of analysis (effective / total stress state), which is set in the combo list.

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

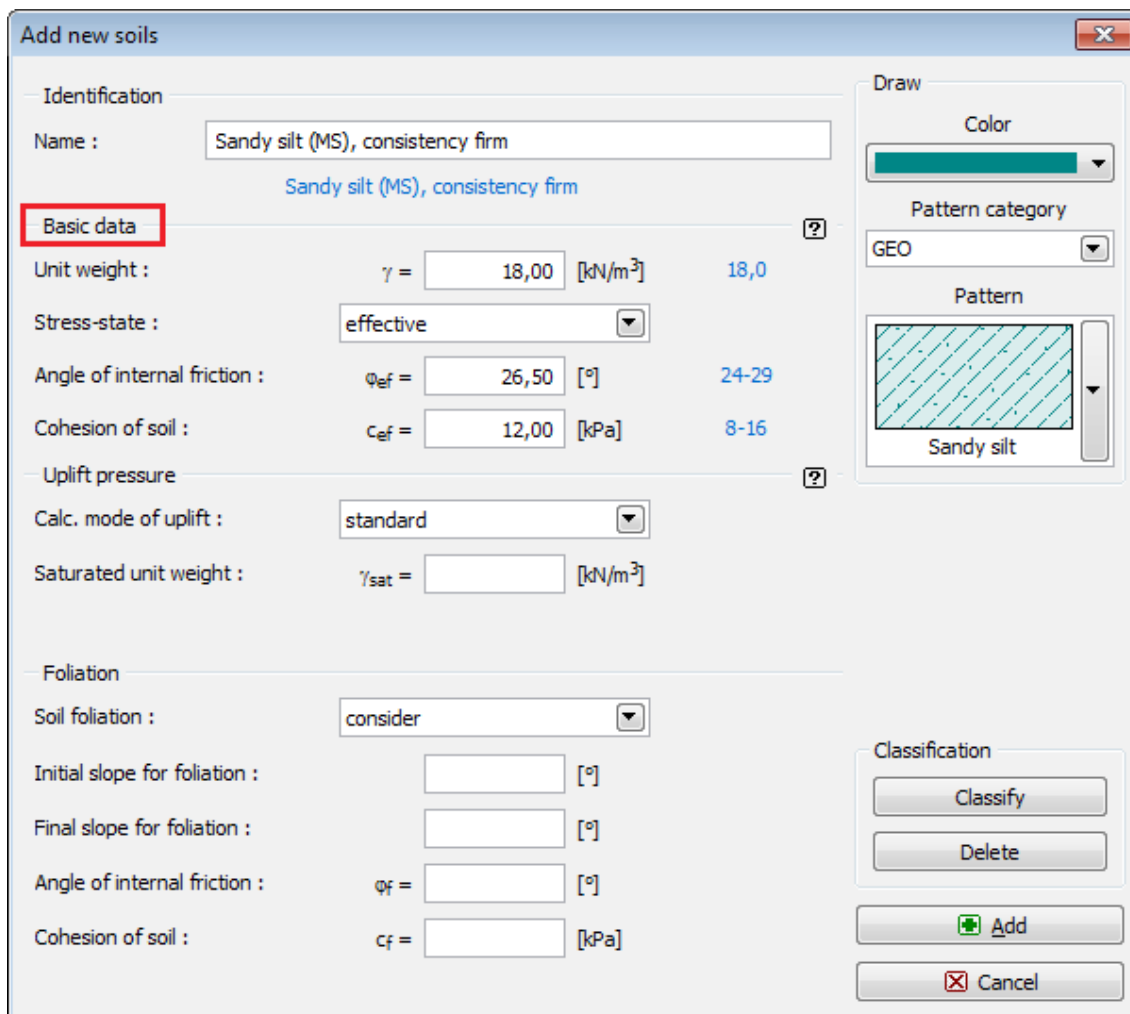
This part of the window serves to introduce basic parameters of soils - **unit weight**, **angle of internal friction** and **cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

The analysis method of slope stability differs for:

- **drained conditions:** for the slope stability calculations to determine equilibrium conditions on the slip surface (**circular**, **polygonal**) is considered **effective stress** according to the equation  $N * tg\varphi_{ef} + c_{ef} * l$ .
- **undrained conditions:** in case of **total stress** for the calculation of passive forces on the slip surface (**circular**, **polygonal**) is considered according to the equation  $c_u * l$ .

In some countries it is customary to specify both shear strength parameters  $\varphi_u, c_u$  for **total stress**. In this case it is necessary to specify the task as an **effective stress** using parameters  $\varphi_{ef}, c_{ef}$  in program "Slope Stability".

The associated theory is described in detail in chapter "Slope stability analysis".



Dialog window "Add new soils" - "Basic data"

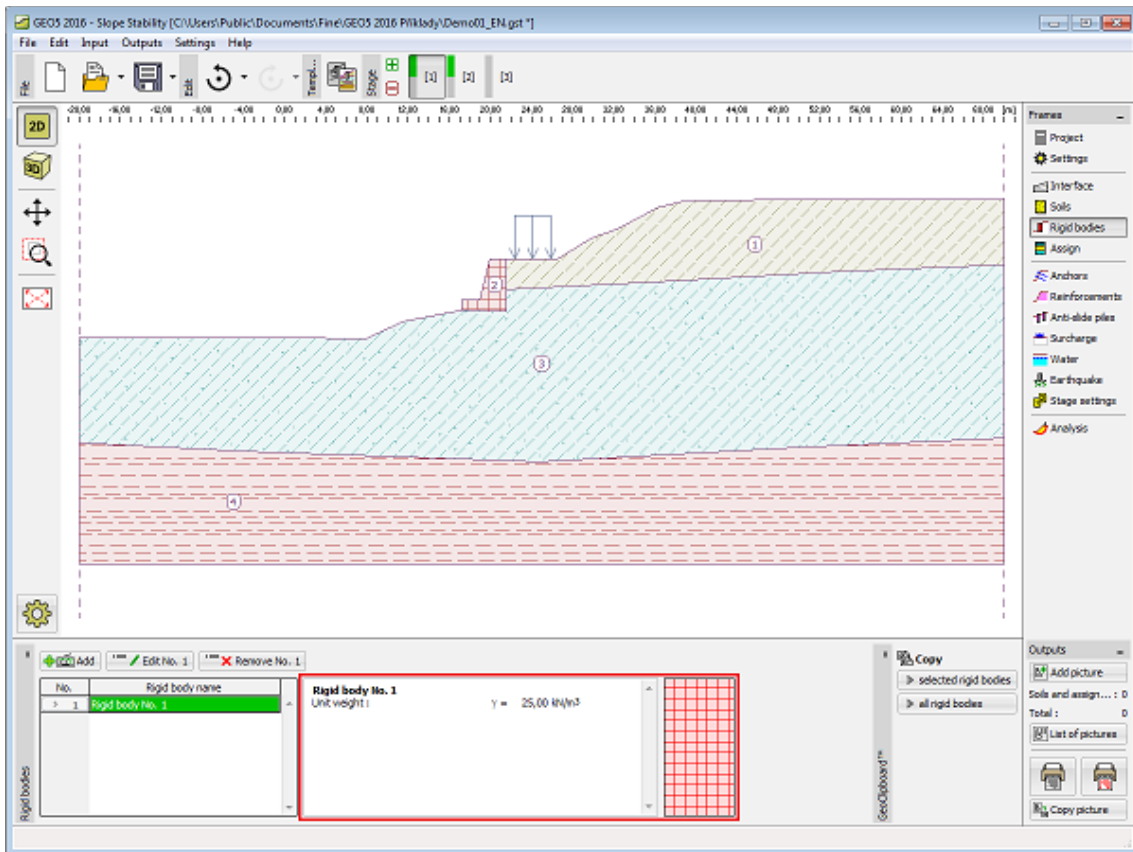
## Rigid Body

The "**Rigid bodies**" frame contains a table with a list of input rigid bodies. The rigid bodies serve to model regions with a high stiffness - e.g., **sheeting structures** or **rock subgrade**. This table also provides information about the currently selected rigid body displayed in the right part of the frame.

Adding (editing) rigid bodies is performed in the "**Add new rigid body**" dialog window. This window serves to input the unit weight of the rigid body material and to select color and pattern. The rigid bodies are in the frame "Assign" ordered after input soils.

**Rigid bodies** are introduced in the program as regions with high strength so they are **not intersected by a potential slip surface**. Providing we wish the slip surface to cross a rigid body (e.g., pile wall) it is recommended to model the rigid body as a soil with a cohesion corresponding to pile bearing capacity against slip.

Input rigid bodies can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Rigid bodies"

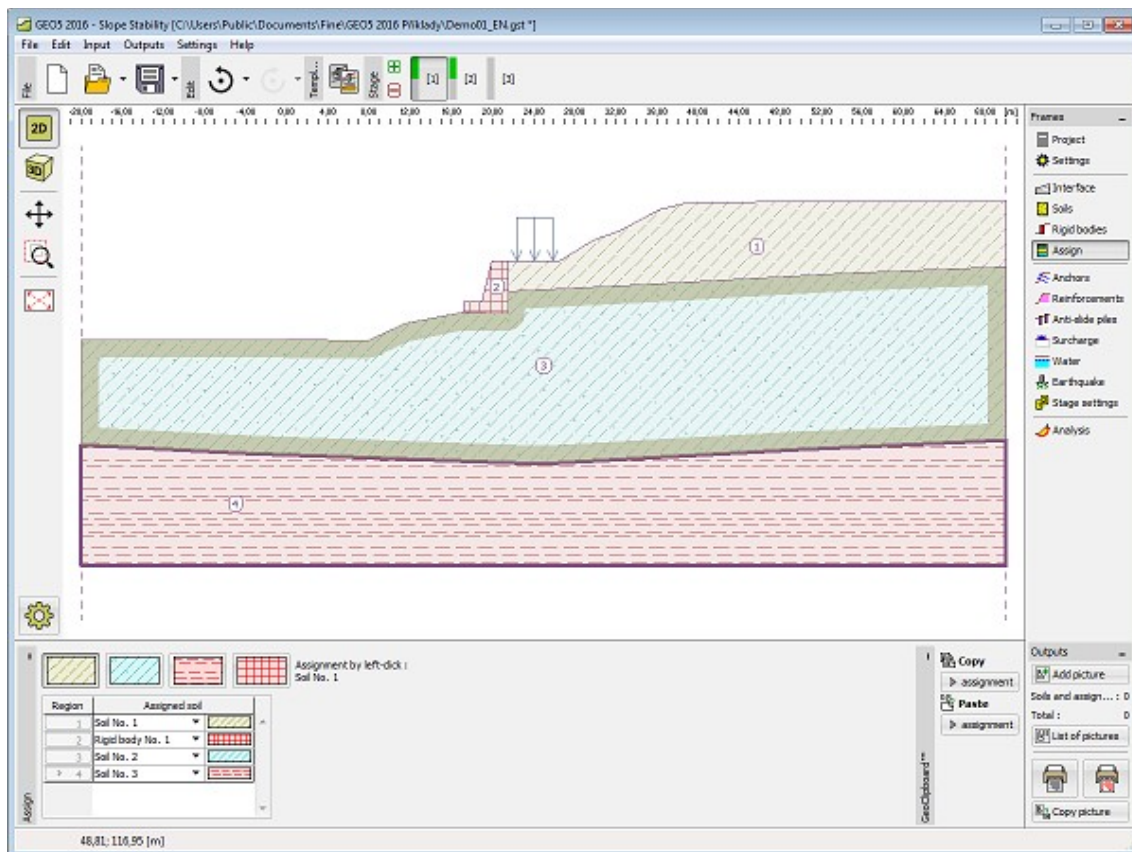
## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [here](#).

The program makes it possible to [import soil assignment in the gINT format](#). Assign of soils can be copied within all 2D GEO5 programs using "GeoClipboard".





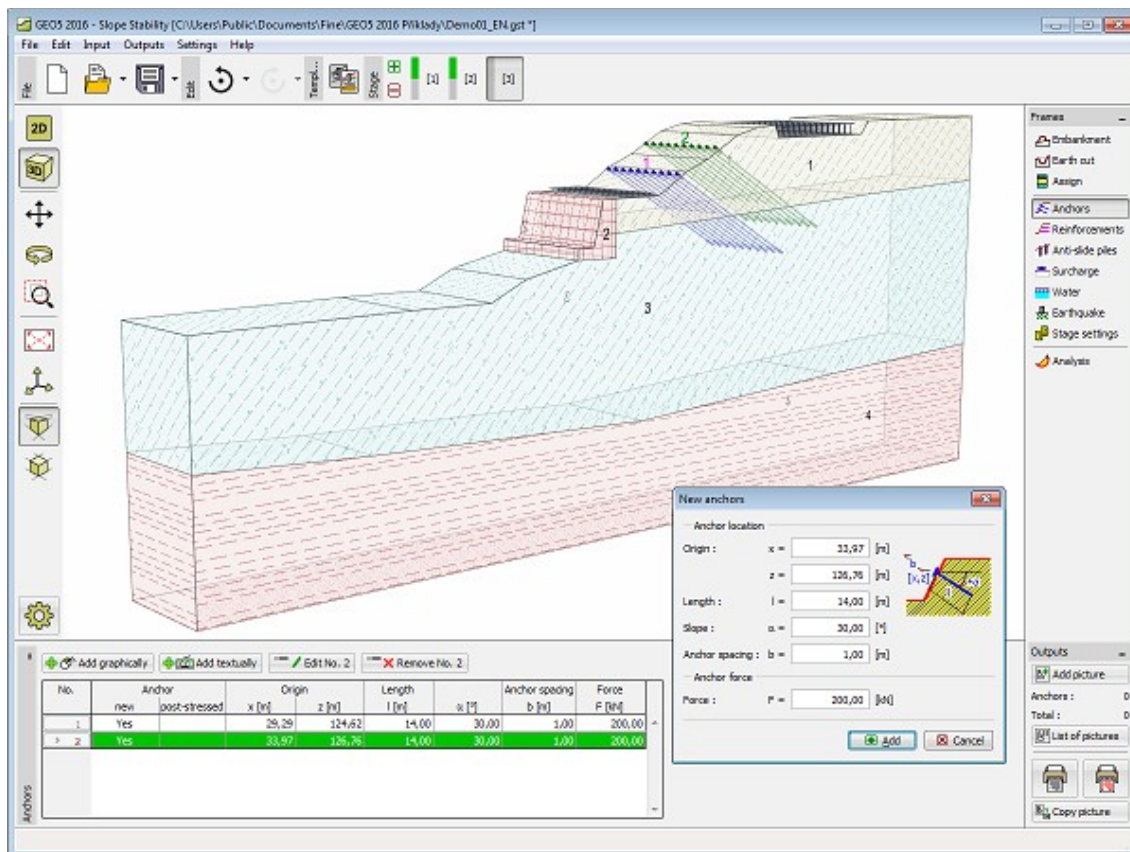
Frame "Assign"

## Anchors

The "**Anchors**" frame contains a table with a list of input anchors. Anchors are input one of two ways (the "**Add in dialog**" or "**Add graphically**" button).

The "**New anchor**", or "**Modify anchor properties**" dialog window serves to input the location of anchor (starting point  $x, z$ ), its length  $l$  and inclination  $\alpha$ , spacing between anchors  $b$ , shift of anchor row  $b_s$  and pre-stress force  $F$ . Starting point of the anchor is always **attached to the terrain (ground surface)**. All input parameters can be modified in the **stage of construction**, in which the anchor was introduced. In subsequent stages the program allows to modify the magnitude of an anchor pre-stress force (by checking the "**Anchor post-stressing**" option).

**Influence of anchors** on the analysis is described in more detail in the theoretical part of the help.



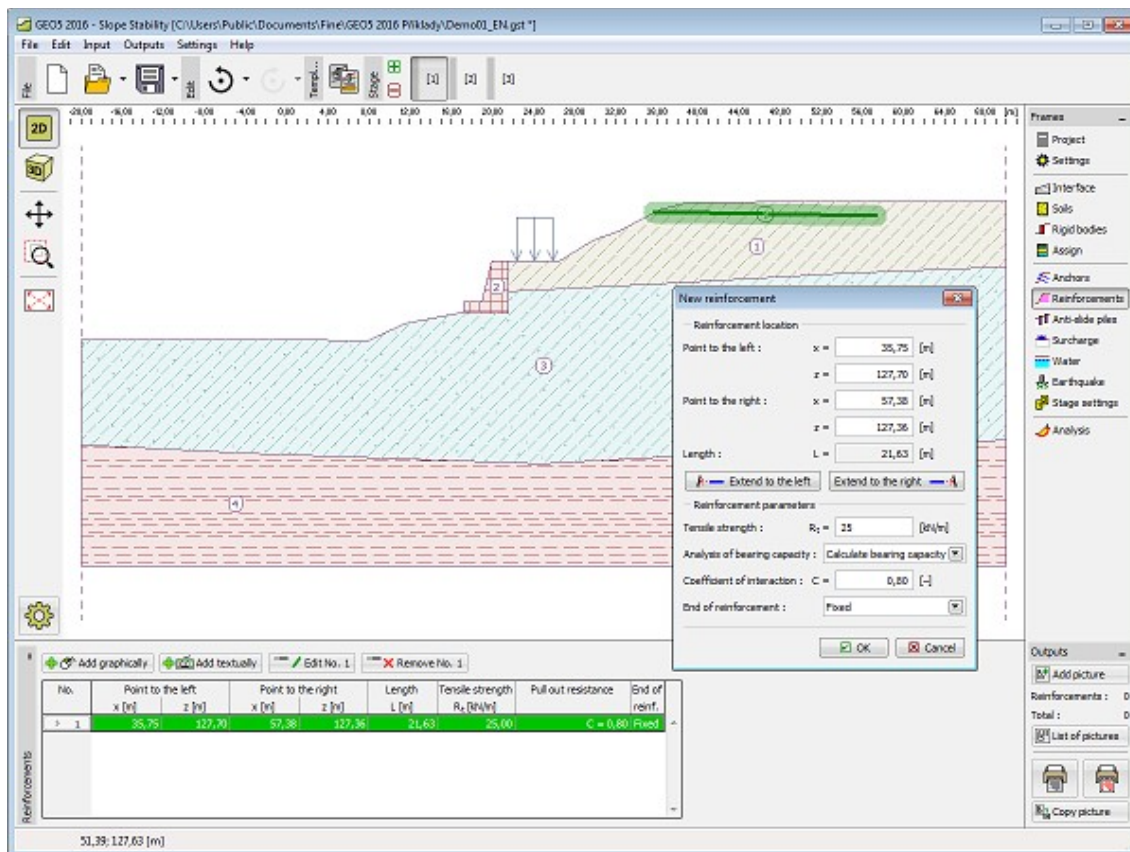
Frame "Anchors"

## Reinforcements

The **"Reinforcements"** frame contains a table with a list of input reinforcements. Reinforcements are input one of two ways (the **"Add in dialog"** or **"Add graphically"** button).

The **"New reinforcement"**, or **"Modify reinforcement parameters"** dialog window serves to input the location of reinforcement, anchorage length (from both left and right end), tensile strength of reinforcement  $R_t$  and end of reinforcement (fixed or free). For the calculation of bearing capacity is selected one of three options in a combo list: **"Calculate bearing capacity"** (coefficient of interaction  $C$  is defined), **"Input anchorage length"**  $l_k$ , or **"Input bearing capacity"** (Pull out resistance  $T_p$  is defined). All input parameters can be modified only in the stage of construction, in which the reinforcement was introduced. In subsequent stages the geo-reinforcement can only be removed.

Influence of reinforcements in the analysis is described in more detail in the theoretical part of the help.



Frame "Reinforcements"

## Anti-Slide Piles

The "Anti-slide piles" frame contains a table with a list of input piles. Piles are input one of two ways (the "Add in dialog" or "Add graphically" button).

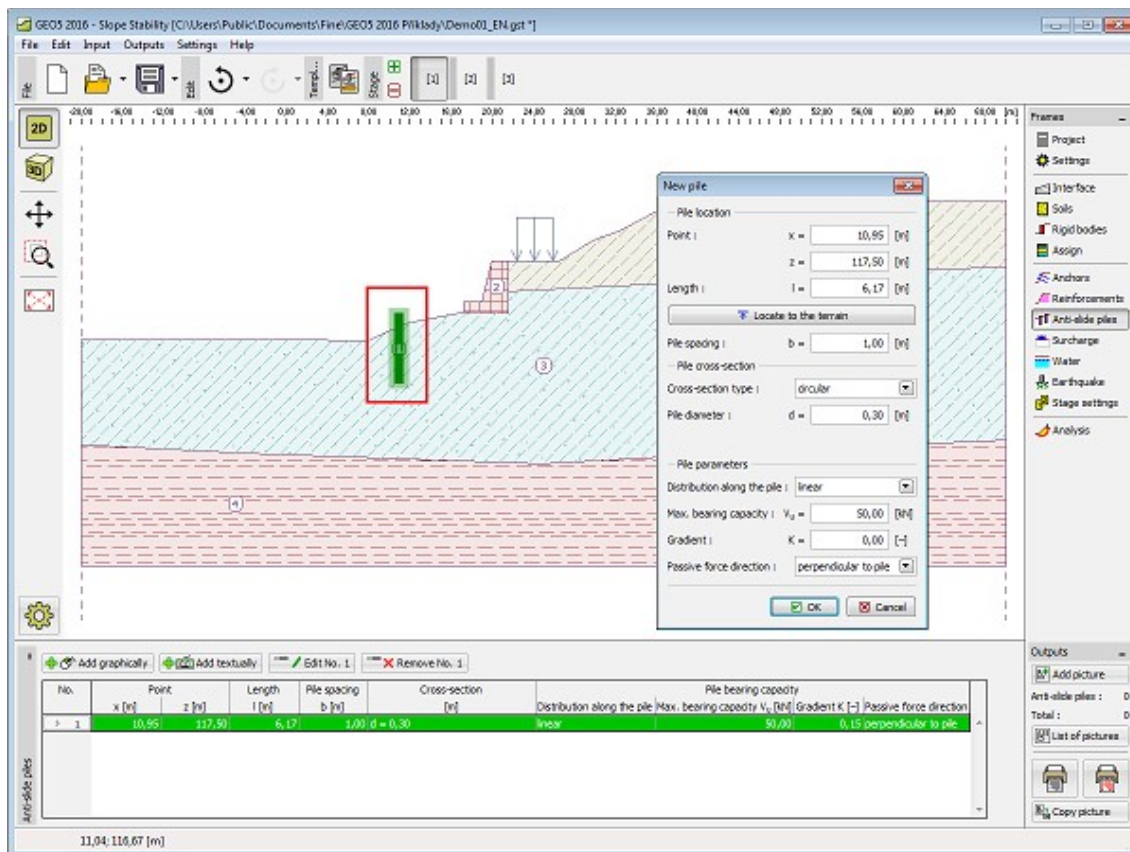
The "New pile", or "Modify pile properties" dialog window serves to input the location of pile (coordinates  $x$ ,  $z$ , length of pile  $l$  and spacing between piles  $b$ ).

The "Locate to the terrain" button places the starting point of pile head on the ground surface. Sometimes, an anti-slide pile can be located in mass directly (in this case it's possible to analyze slope stability, but it's not possible to run the "Anti-Slide Pile" program).

Then it's specified the cross-section of pile (circle - diameter of pile  $d$ , rectangle - dimensions  $s_x$ ,  $s_y$ ) and the parameters of pile - distribution of bearing capacity along the pile length (linear, constant), maximum bearing capacity  $V_u$ , gradient  $K$  and direction of passive force (perpendicular to the pile, parallel to slip surface). All input parameters can be modified in the stage of construction, in which the anti-slide pile was introduced. In subsequent stages the anti-slide pile can only be removed.

Influence of anti-slide piles on the assessment of slope stability is described in more detail in the theoretical part of the help. Other calculations of the anti-slide piles (analysis of an internal forces, dimensioning of reinforcement of piles) are based on the analysis of active and passive forces in the "Anti-Slide Pile" program.





Frame "Anti-slide piles"

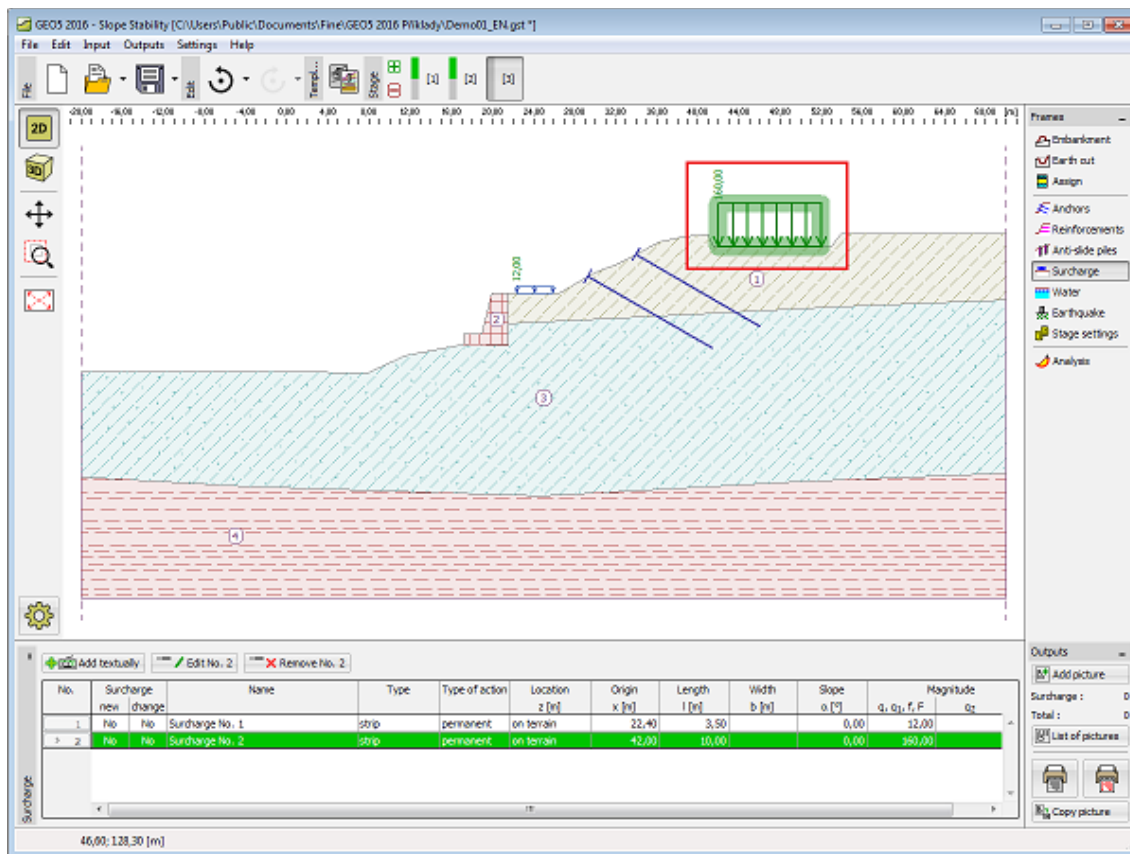
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active objects.

All input parameters of a surcharge can be modified in the construction stage where the surcharge was specified. Only the surcharge magnitude can be modified in all subsequent construction stages (option "**Adjust surcharge**").

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient. Accidental surcharge with favorable effect is not considered in the analysis.

Influence of surcharge on stability analysis of slopes is described in the theoretical part of the help.



Frame "Surcharge"

## Water

The "**Water**" frame serves to set the type of ground water table. Six options to specify the **type of water** are available from the combo list.

Inputting the ground water table or isolines, respectively, is identical with the standard **input of interfaces**.

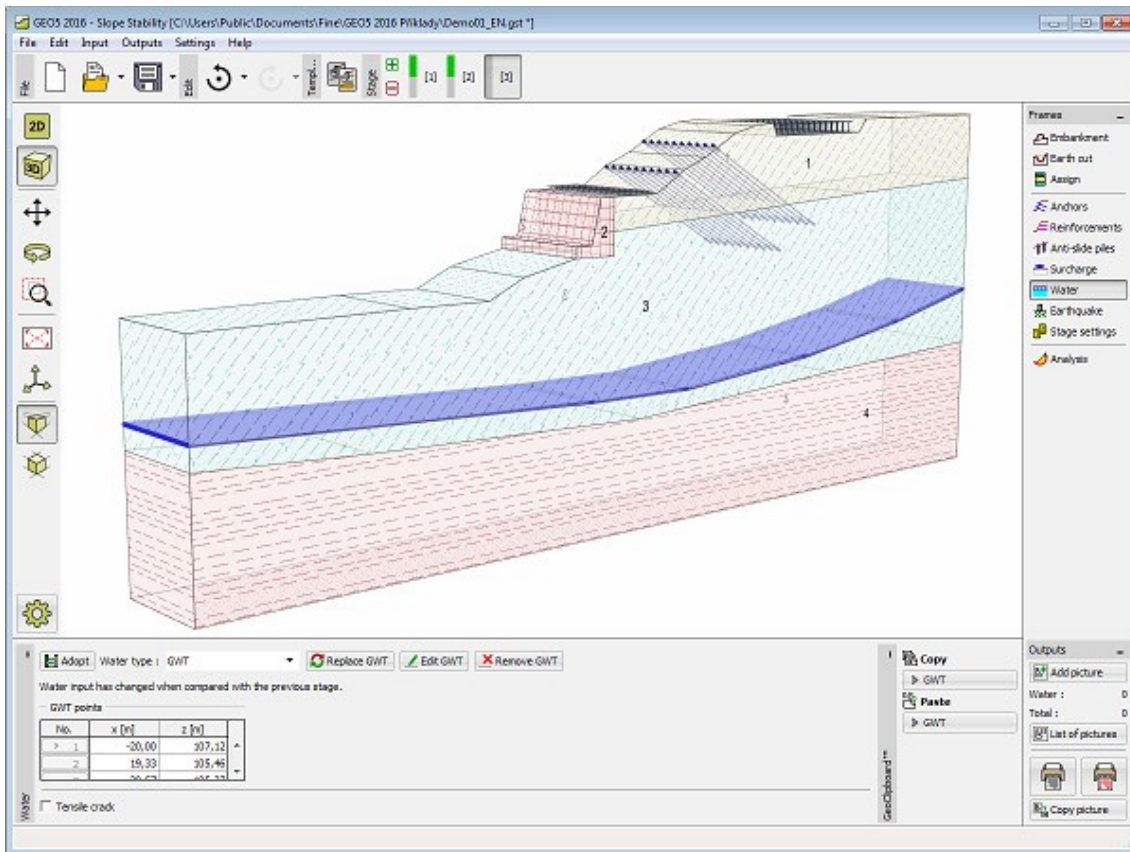
A field for specifying a value of coefficient  $R_u$  or pore pressure appears next to the table if introducing water using isolines of  **$R_u$ -interfaces** or **pore pressure**, respectively. Pressing the button with a blue arrow next to the input field opens the "**Coefficient  $R_u$** " or "**Pore pressure**" dialog window to enter the desired value. It is advantageous to input all values at once using the "**OK+↑**" and "**OK+↓**". The value of a given quantity found in a specific point between two isolines is approximated by **linear interpolation** of values pertinent to given isolines. For option "**Coefficient  $R_u$** " the first (the most top one) is always **identical with terrain** - it therefore cannot be deleted.

The **ground water table** (resp. **table of suction** or **original GWT**) is specified as continuous interfaces, which can be located even above the terrain.

If the input data in individual stages are different, the program then allows for accepting the data from the previous stage of construction by pressing the "**Adopt**" button.

The program further allows for specifying a depth of **tensile cracks** filled with water.

Input interfaces of water can be copied within all 2D GEO5 programs using "**GeoClipboard**".



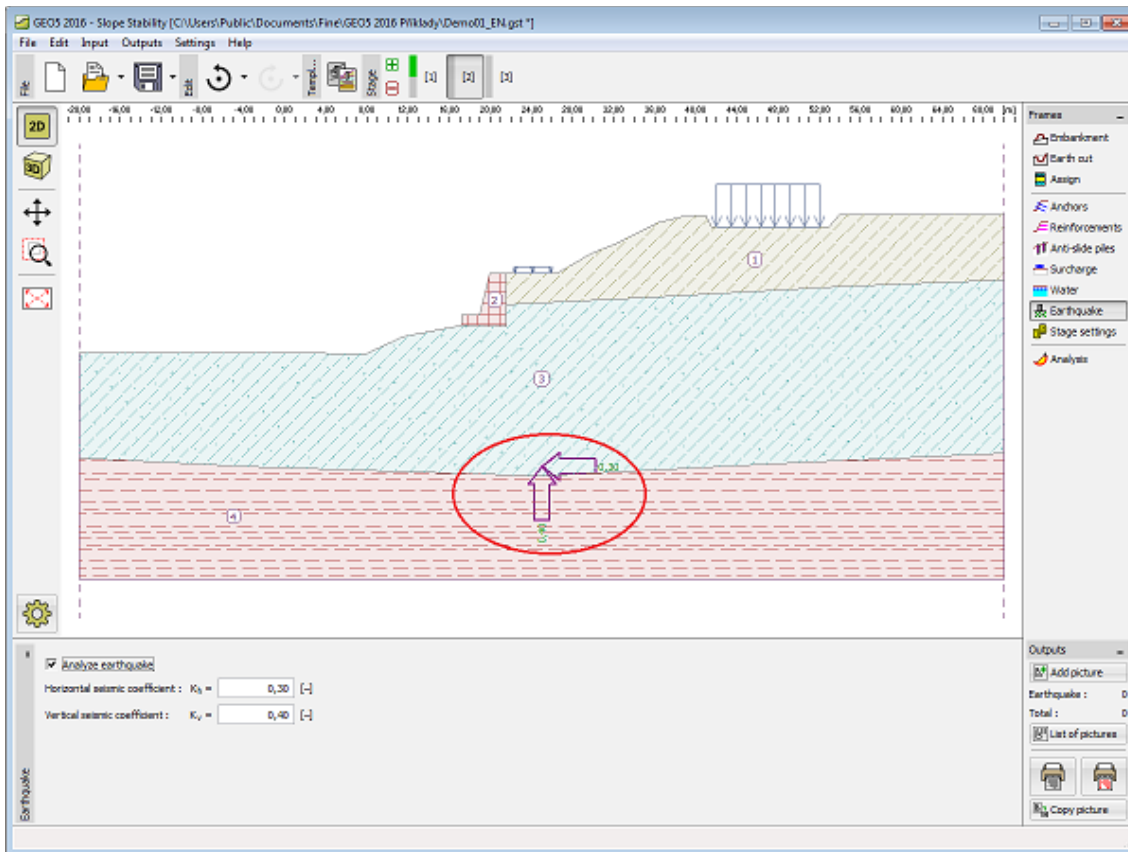
Frame "Water"

## Earthquake

The "**Earthquake**" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $K_h$  and  $K_v$  can be calculated following the approach adopted from EN 1998-5.

Slope stability analysis while accounting for earthquake is described in the theoretical part of the help, chapter "**Influence of earthquake**".



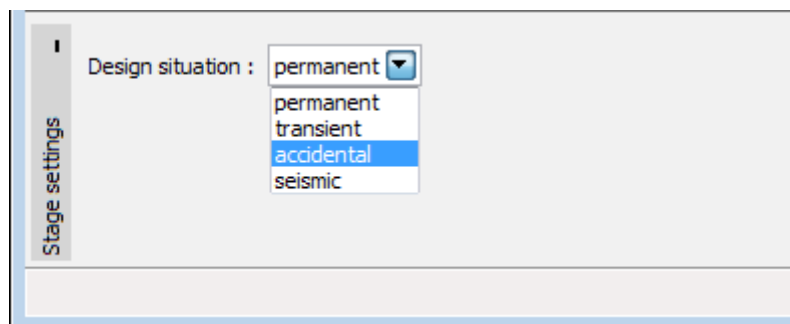
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Service I, Extreme I**).



Frame "Stage settings"

## Analysis

The "Analysis" frame displays the analysis results. Several analyses can be performed for a

single task.

The starting point in the **slope stability analysis** is the selection of the type of slip surface. The input is available from a combo list in the left top part of the frame containing two options - **circular slip surface** and **polygonal slip surface**. After introducing the slip surface the analysis is started using the "**Analyze**" button. The analysis results appear in the right part of the frame.

**The type of analysis** is selected in the mid section of the frame - seven methods are available for the **circular slip surface** (Fellenius/Petterson, Bishop, Spencer, Janbu, Morgenstern-Price, Shahunyants or ITFM), and six methods are available for the **polygonal slip surface** (Sarma, Spencer, Janbu, Morgenstern-Price, Shahunyants or ITFM). For both cases of the assumed slip surfaces it is possible to perform the analysis employing all methods at once (in such a case, however, the slip surface cannot be optimized).

The actual verification of slope stability can be performed, depending on the settings in the "**Stability analysis**" tab:

- **Verification according to EN 1997**, where the **load is reduced** by the analysis partial factors and the verification is performed according to the **theory of limit states**.
- **Verification according to the factor of safety**
- **Verification according to the theory of limit states**

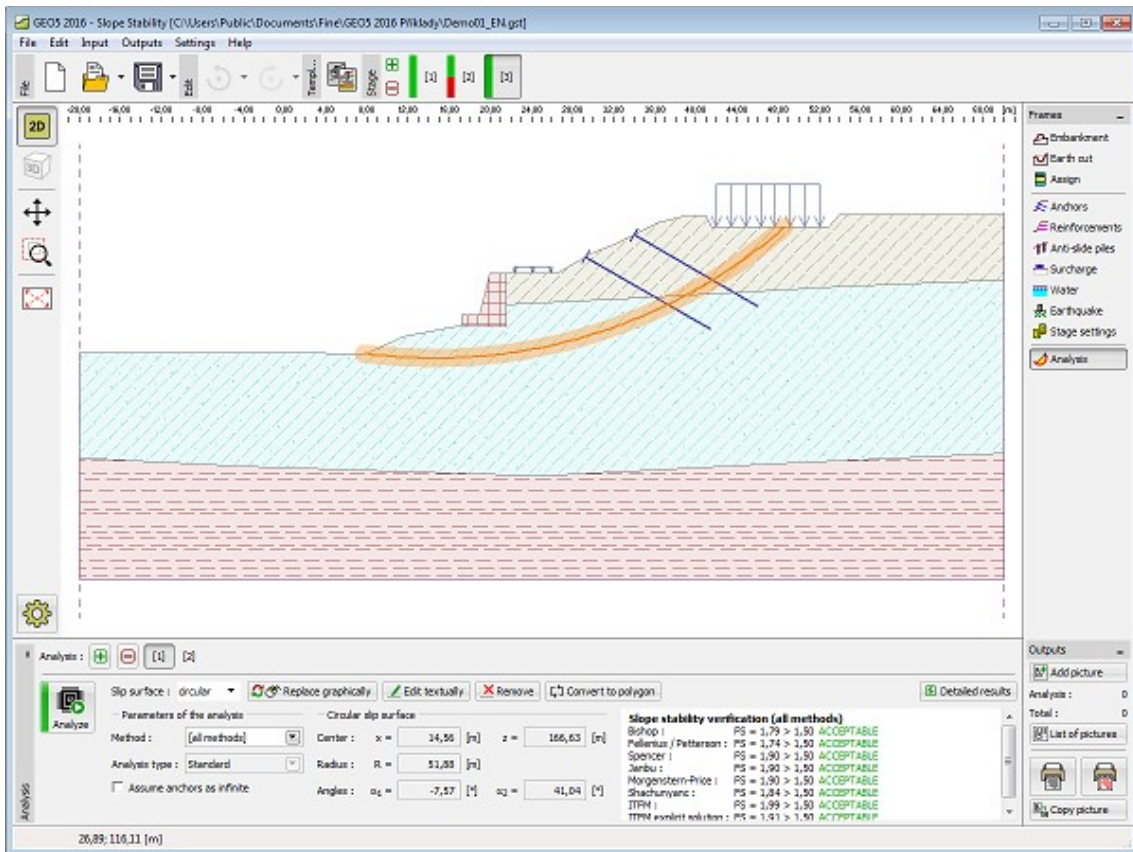
The combo list (items "**Standard**" and "**Optimization**") allows for optimizing either the **circular** or **polygonal** slip surface. Choosing the "**Optimization**" option activates the "**Restrictions**" button - pressing this button changes the frame appearance and makes it possible to introduce **restrictions on the optimization procedure**.

It is also possible to specify **how to deal with anchors in the analysis** (box "**Assume anchors as infinite**").

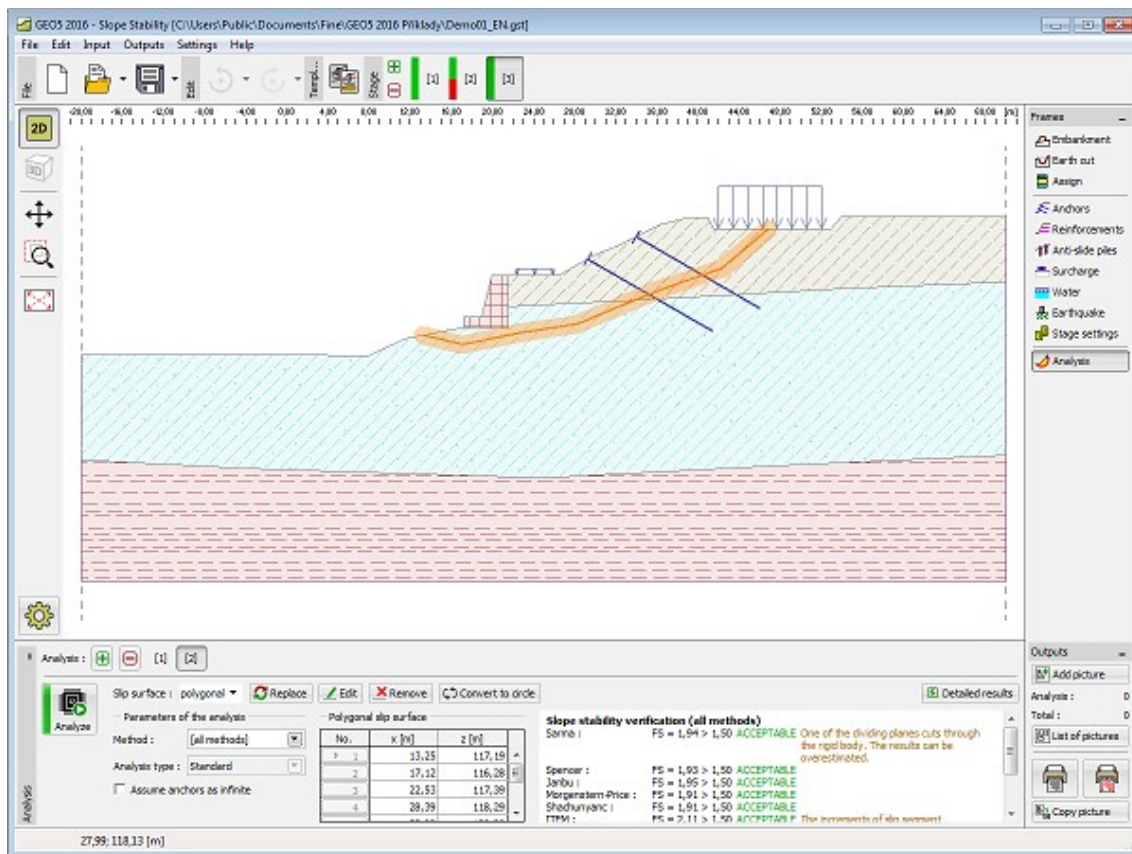
The slip surface, even the optimized one, must be **introduced** in the frame.

The analysis results appear in the left part of the frame and the optimized slip surface on the desktop. Visualization of results can be adjusted in the frame "**Visualization settings**".





Frame "Analysis" - circular slip surface

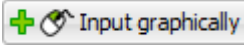


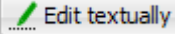
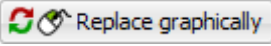
Frame "Analysis" - polygonal slip surface

## Input of Slip Surface

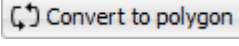
Select the slip surface type from the list (circular, polygonal). It is possible to input the slip surface according to its type in several ways:

- **Circular slip surface**

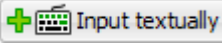
**Graphically** - after pressing the button  , input three points that define the circular slip surface using the left mouse button.

It is possible to change the input slip surface in the dialog window "**Circular slip surface**" after pressing the button  , or replace the slip surface using the mouse button by pressing .

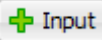
The button  cancels the slip surface input.

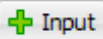

The button  converts the circular slip surface to polygonal.

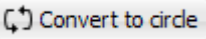
The button  removes the slip surface.

**Textually** - by pressing the button  the dialog window "**Circular slip surface**" is opened, and  $x$ ,  $y$  coordinates, and the diameter is input.

- **Polygonal slip surface**

**Graphically** - by pressing the button  on the tool bar, the slip surface input mode is turned on - while adding, the process is the same as with **interface input**.

**Textually** - press the button  and using the button above the table  a dialog window "New points" is opened. In the dialog window, the points of the slip surface are added using the  $x, y$  coordinates.

The button  converts polygonal slip surface into circle slip surface. Functions of the rest of the buttons are the same as with the circular slip surface.

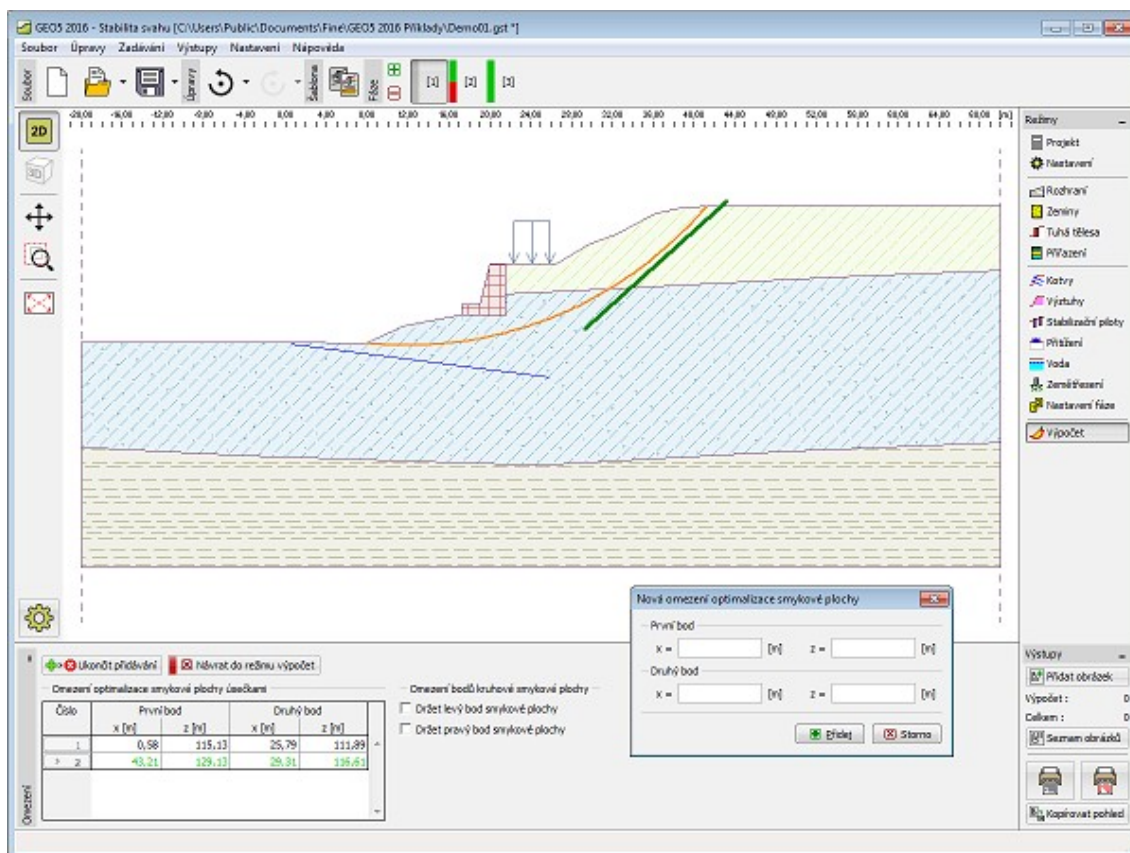
## Restrictions on the Optimization Procedure

The "Analysis" frame allows (after pressing the "Restrictions") for specifying restrictions on the optimization process.

**Regardless of the assumed type of slip surface (circular, polygonal)** it is possible to introduce into the soil body (with the help of mouse) segments, which should not be crossed by the optimized slip surface. These segments also appear in the **table** in the left part of the frame.

**Polygonal slip surface** also allows for excluding some points from optimization, either entirely or partially only in specified direction. "Keeping the point fixed" during optimization process is achieved by checking the box in the table with corresponding point.

This input mode is quitted by pressing the red button "Return to analysis".



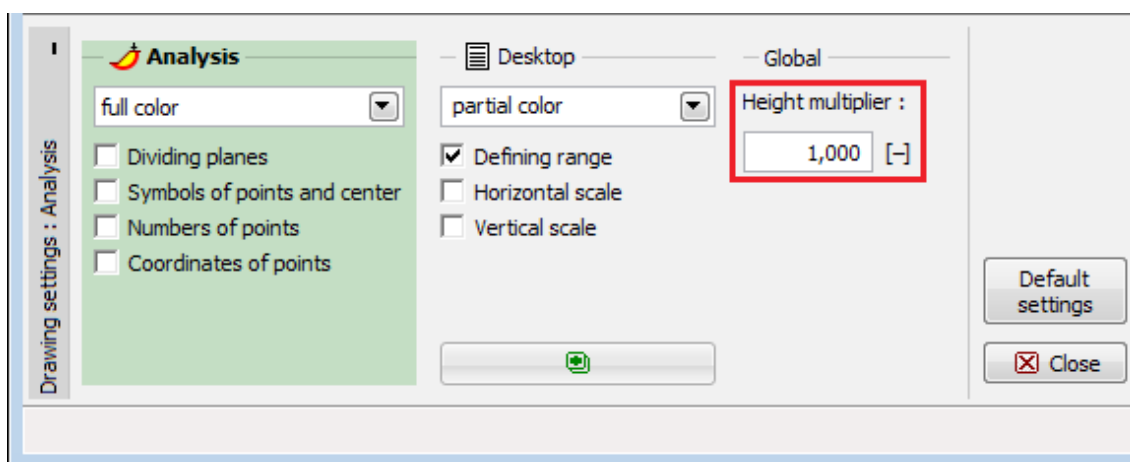


*Frame "Analysis" - restrictions on slip surface optimization by segments*

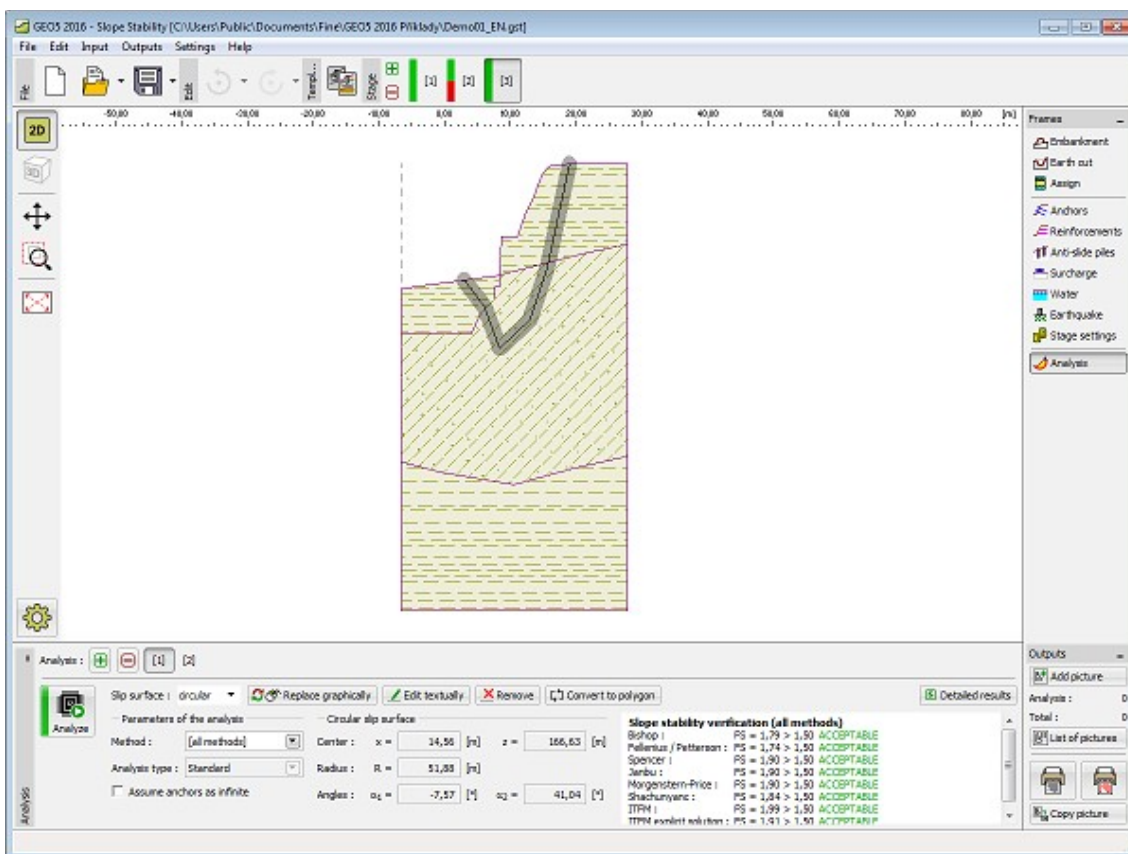
## Height Multiplier

Providing the analyzed slope is too long or has small height the plotted slip surface might not be sufficiently visible. This problem can be solved by selected courser scale in the vertical direction with the help of height multiplier. The value of this multiplier is set in the frame "Visualization settings" dialog window, tab "Global 2D". Using standard setting ("Height multiplier" equal to one) plots undistorted structure proportional to its dimensions.

Only polygonal slip surface can be input graphically when exploiting the height multiplier option. The circular slip surface must be in such a case input manually in the "Circular slip surface" dialog window using the "Input" button.



*Setting height multiplier*



Visualization of the resulting slip surface when using height multiplier

## Program Rock Stability

This program is used to analyze the stability of rock slopes and walls for a specified type of failure, including a planar or polygonal slip surface or rock wedge.

The help in the program "Rock Stability" includes the following topics:

- Input of data into individual frames: Project, Settings

Plane and polygonal slip surface:	Terrain	Rock	Slip Surface (Plane)	Slip Surface (Polygonal)	Water (Plane Slip Surface)	Water (Polygonal Slip Surface)
-----------------------------------	---------	------	----------------------	--------------------------	----------------------------	--------------------------------

Surcharge	Anchors	Earthquake	Stage Settings	Analysis (Plane Slip Surface)	Analysis (Polygonal Slip Surface)
-----------	---------	------------	----------------	-------------------------------	-----------------------------------

Rock Wedge:	Geometry	Slip Surface Parameters	Water	Surcharge	Anchors
-------------	----------	-------------------------	-------	-----------	---------

Earthquake	Stage Settings	Analysis
------------	----------------	----------

- Standards and analysis methods
- Theory for analysis in the program "**Rock Stability**":
  - Parameters of Rocks
  - Rock Stability
- Outputs
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".

Task :	Terraces Hanspaulka	Author :	James Baker
Part :	South-facing slope III.	Date :	28.10.2005
Description :	Support walls 2-5m	Project ID :	845/2014
Customer :	Belltrade LTd.	Project number :	11486/2014
System of units	metric		

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows to choose an already created setting from the "**Settings list**".

The "**Settings Administrator**" button opens the "**Administrator**" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

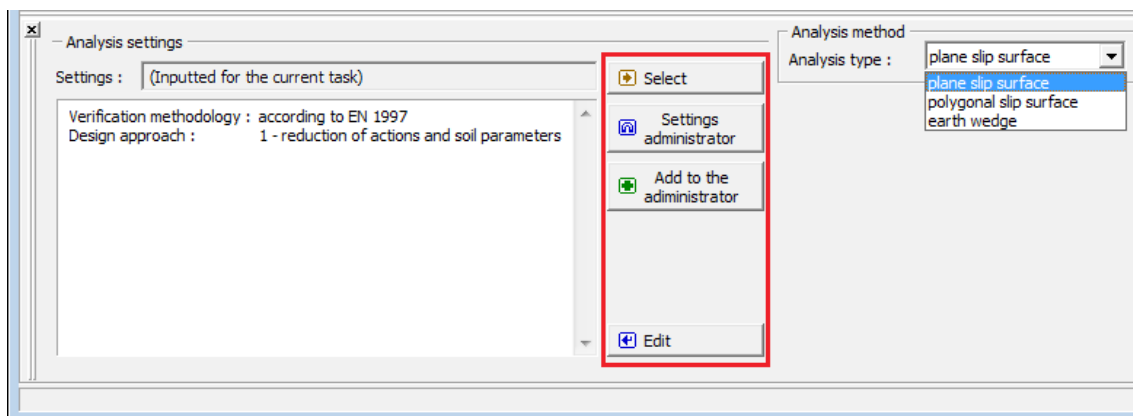
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Stability analysis**" tab.

The frame also allows to select the type of slip surface:

- **Plane slip surface**
- **Polygonal slip surface**
- **Earth wedge**



Frame "Settings"

## Terrain

The **frame "Terrain"** contains a **table** with a list of defined sections of a rock slope.

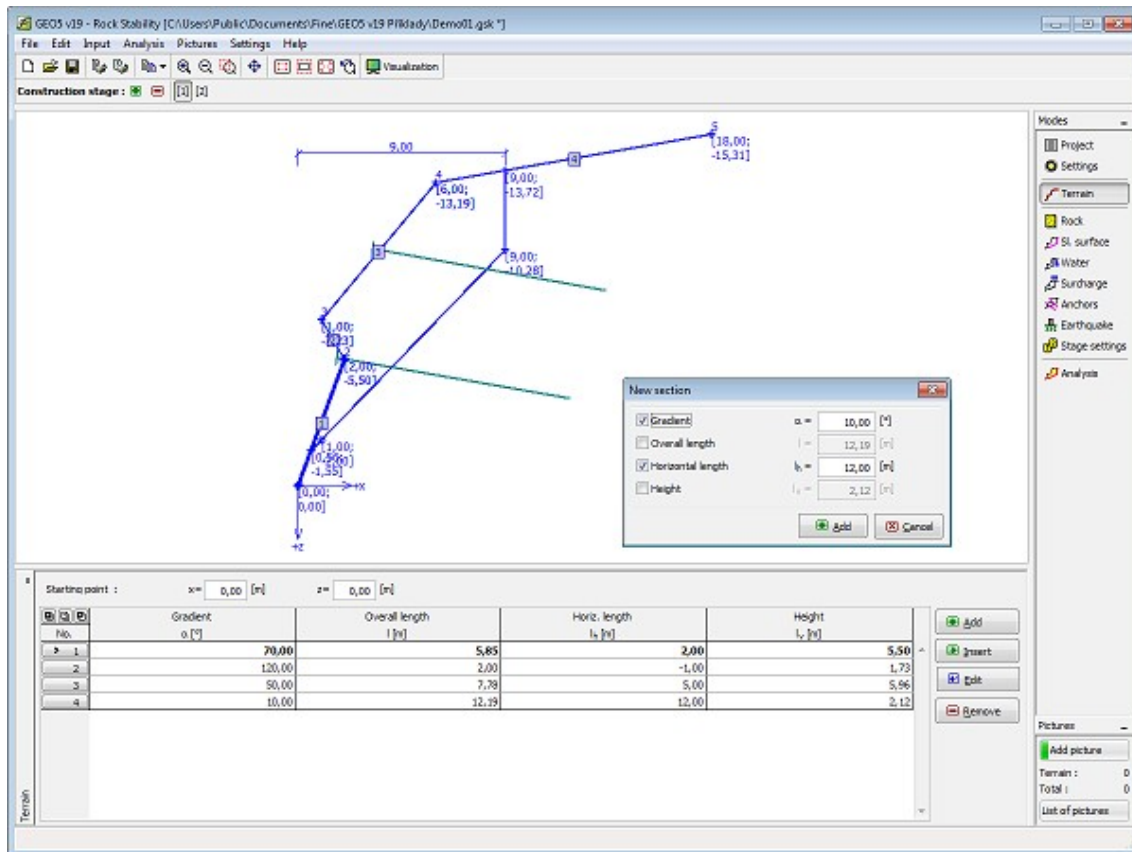
The **coordinates of the origin** - the first point of terrain followed by defined sections - are entered in the upper part of the frame. In the program the slope is always oriented from **the left to the right**.

**Adding** (editing) section is performed in the "**New section (Edit section)**" **dialog window**. These sections can also be edited on desktop with the help of **active objects**.

Each section can be defined by its dip, by the overall length of section, by the horizontal length and height of section of a rock slope. Only **two selected values** are used while the others are determined by the program automatically (if more than two entry fields are checked than the input and computation are not carried out). Both vertical and horizontal sections as well as overhangs can be represented.

In case of a proper input the **program automatically plots the defined section** on desktop

using dashed line, so that before accepting the defined section by pressing the "Add" button it is possible to check, whether the section is correctly defined.



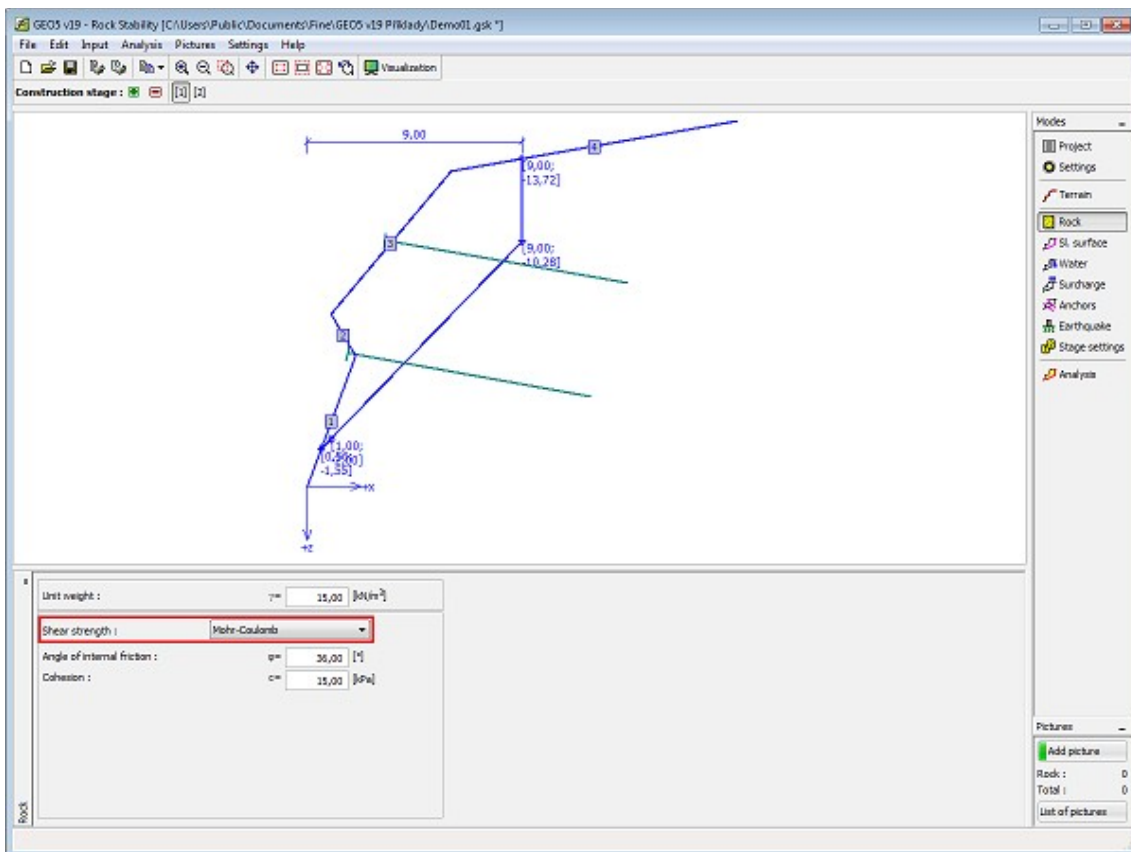
Frame "Terrain"

## Rock

The frame "Rock" allows for entering the material parameters (approximate values) of a rock slope (depending on the type of shear strength) including the unit weight of a rock. Three types of shear strengths on a slip surface are available in the program:

- Mohr - Coulomb
- Barton - Bandis
- Hoek - Brown

Material parameters of rock are then entered based on the selected method.



Frame "Rock"

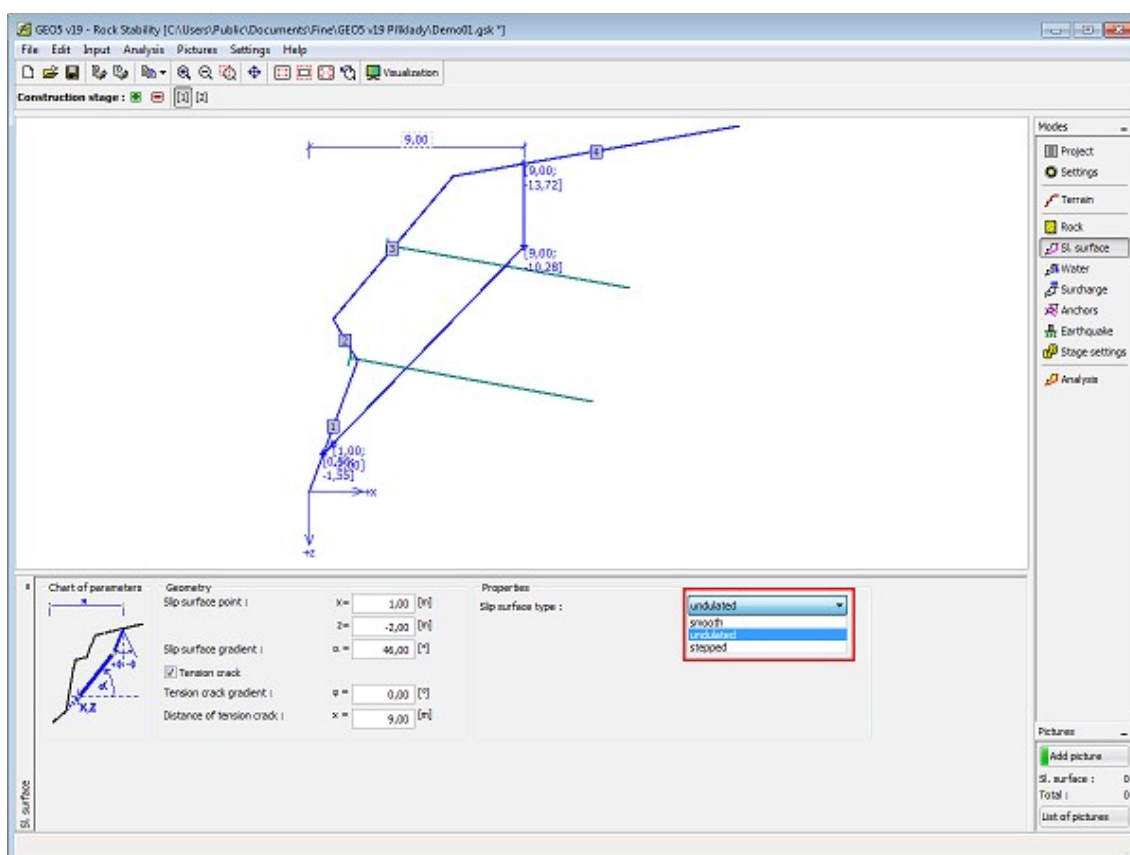
## Slip Surface - Plane

The **frame "Slip surface"** serves to specify the shape and parameters of a plane slip surface. The slip surface is defined by a point in the rock body and by its gradient. The program automatically determines intersections of the slip surface with terrain.

The program also allows for defining a **tension crack** with an arbitrary gradient (not available for stepped slip surface). The crack is defined by a horizontal distance from the **origin** and by its gradient.

The **plane slip surface** can further be labeled as smooth, **undulated** or **stepped**.

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Slip surface - plane"

## Slip Surface - Polygonal

The **frame "Slip surface"** contains a **table** with a list of defined sections of a slip surface. Adding (editing) section is performed in the **"New section (Edit section)" dialog window**. These sections can also be edited on desktop with the help of **active objects**.

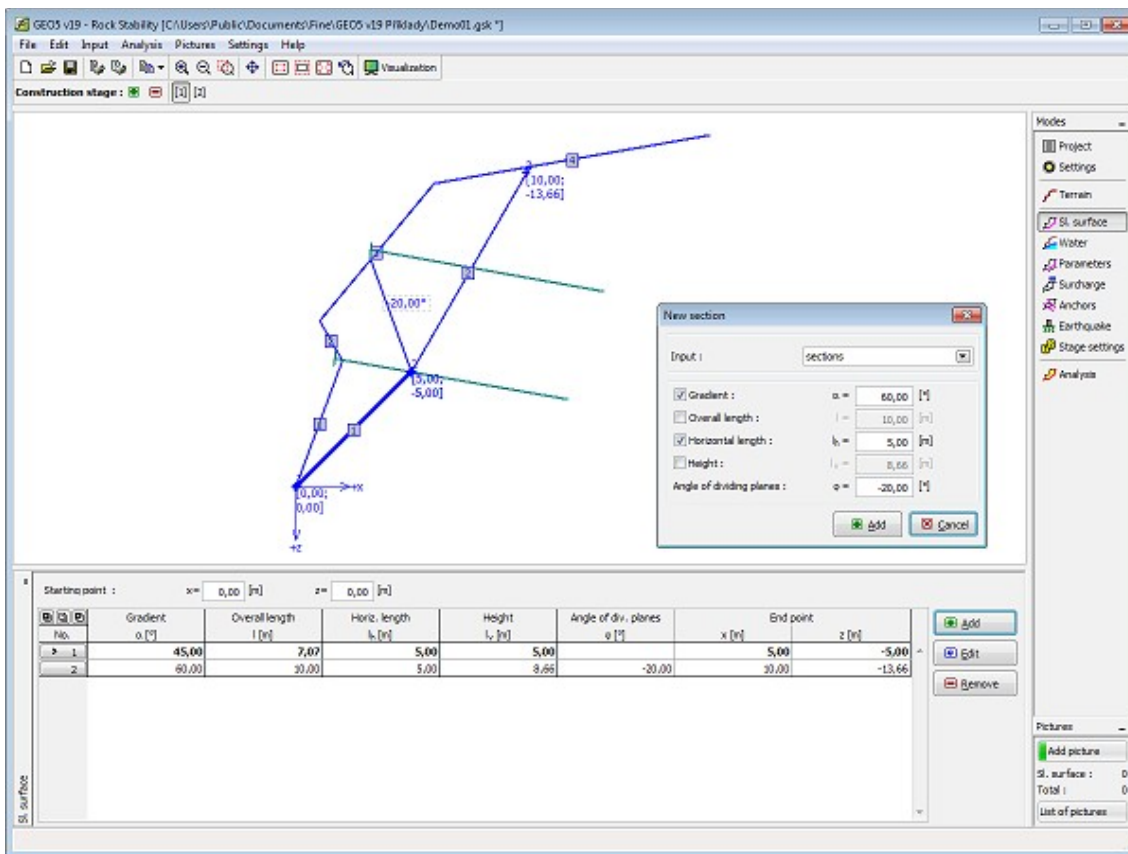
The coordinates of the slip surface **origin** - a point on the slip surface followed by other sections - are entered in the upper part of the frame. This point can be found even out of the soil body - the program then automatically calculates the intersection of slip surface with terrain.

Individual sections of the slip surface can be defined by their dip, by the overall length of section, by the horizontal length and height of section of a rock slope. Only **two selected values** are used while the others are determined by the program automatically (if more than two entry fields are checked than the input and computation are not carried out). Both vertical and horizontal sections as well as overhangs can be represented.

In case of a proper input the **program automatically plots the defined section** on desktop using dashed line, so that before accepting the defined section by pressing the **"Add"** button it is possible to check, whether the section is correctly defined.

General assumptions for the calculation of **polygonal slip surface** are listed **here**.

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Slip surface - polygonal"

## Parameters - Polygonal Slip Surface

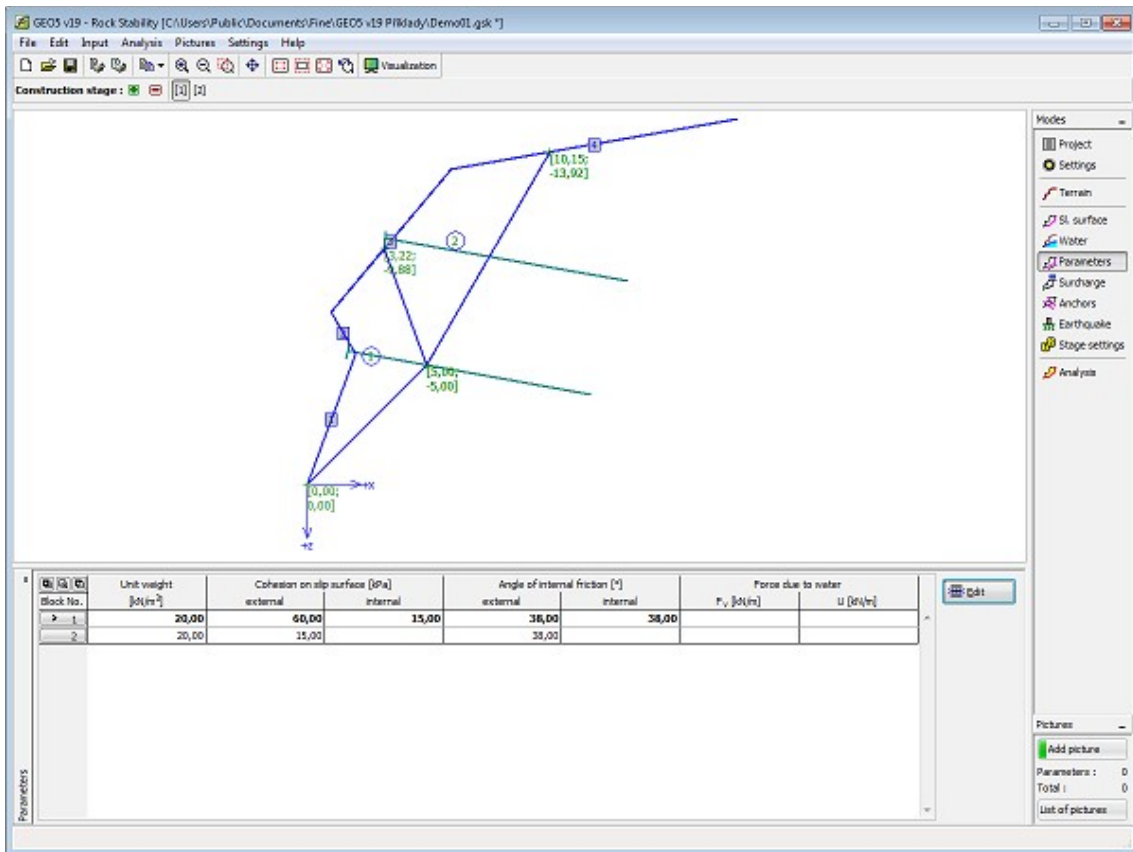
The frame "Parameters" contains a table with a list of blocks, which are created by entering a polygonal slip surface. Parameters of individual blocks are edited in the "Edit block" dialog window. Blocks can also be edited on desktop with the help of active objects.

The Mohr-Coulomb strength parameters on a slip surface and in the joints separating individual blocks including the unit weight of a rock are specified here.

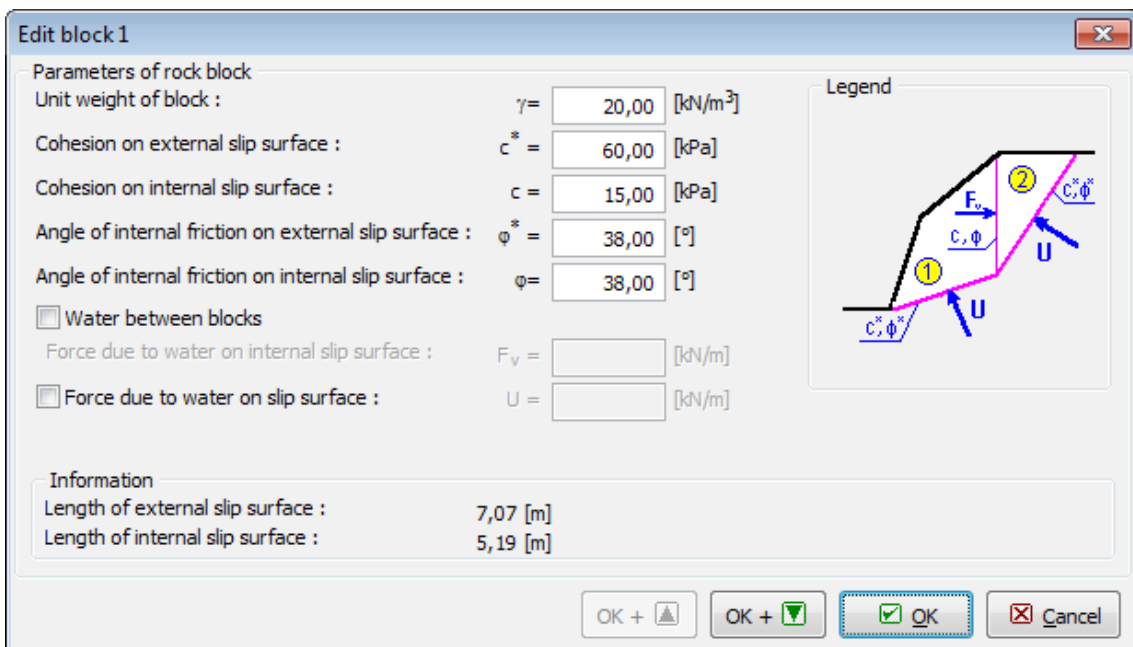
This window also serves to introduce forces due to water in rock blocks.

General assumptions for the calculation of polygonal slip surface are listed here.





Frame "Parameters" - polygonal slip surface

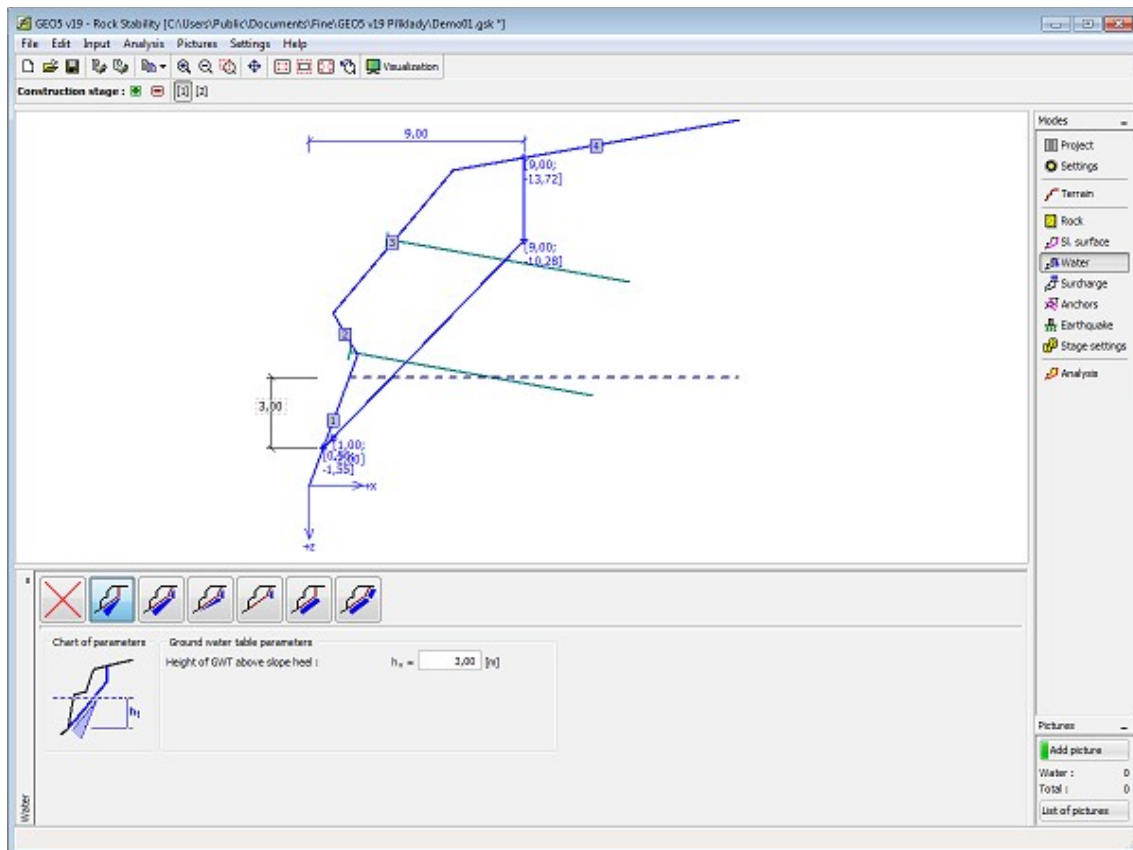


Dialog window "Edit block"

## Water - Plane Slip Surface

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Water parameters can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

Solution procedure when accounting for water is described in the theoretical part of the help "**Influence of water on slip surface**".



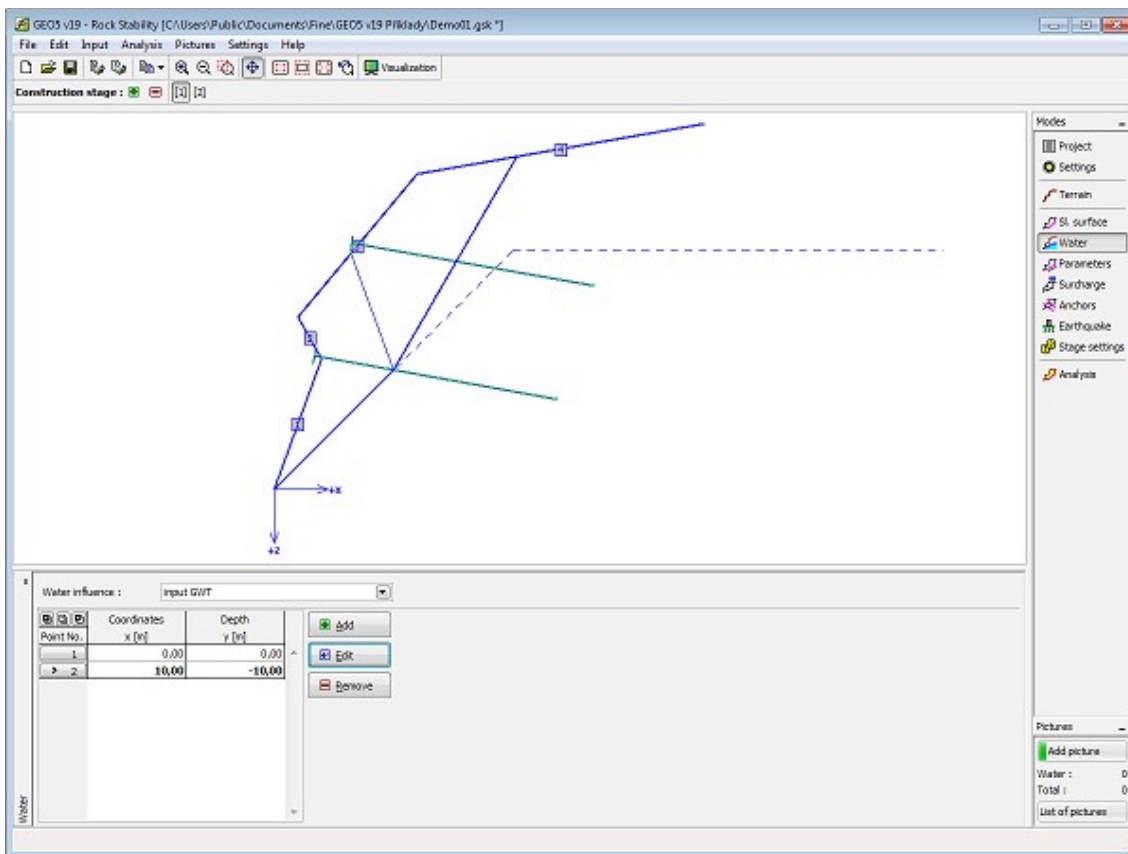
Frame "Water" - plane slip surface

## Water - Polygonal Slip Surface

The "**Water**" frame serves to input **influence of water** (not considered, input forces on blocks, input horizontal water level, input GWT). Water parameters can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

Solution procedure when accounting for water is described in the theoretical part of the help "**Influence of water on polygonal slip surface**".

For option "**input forces on blocks**" the forces of water acting on the slip surface  $F_V$  or forces from the water acting on the inner sliding surface  $U$  are entered in the frame "**Parameters**" (by pressing the button "**Edit**").



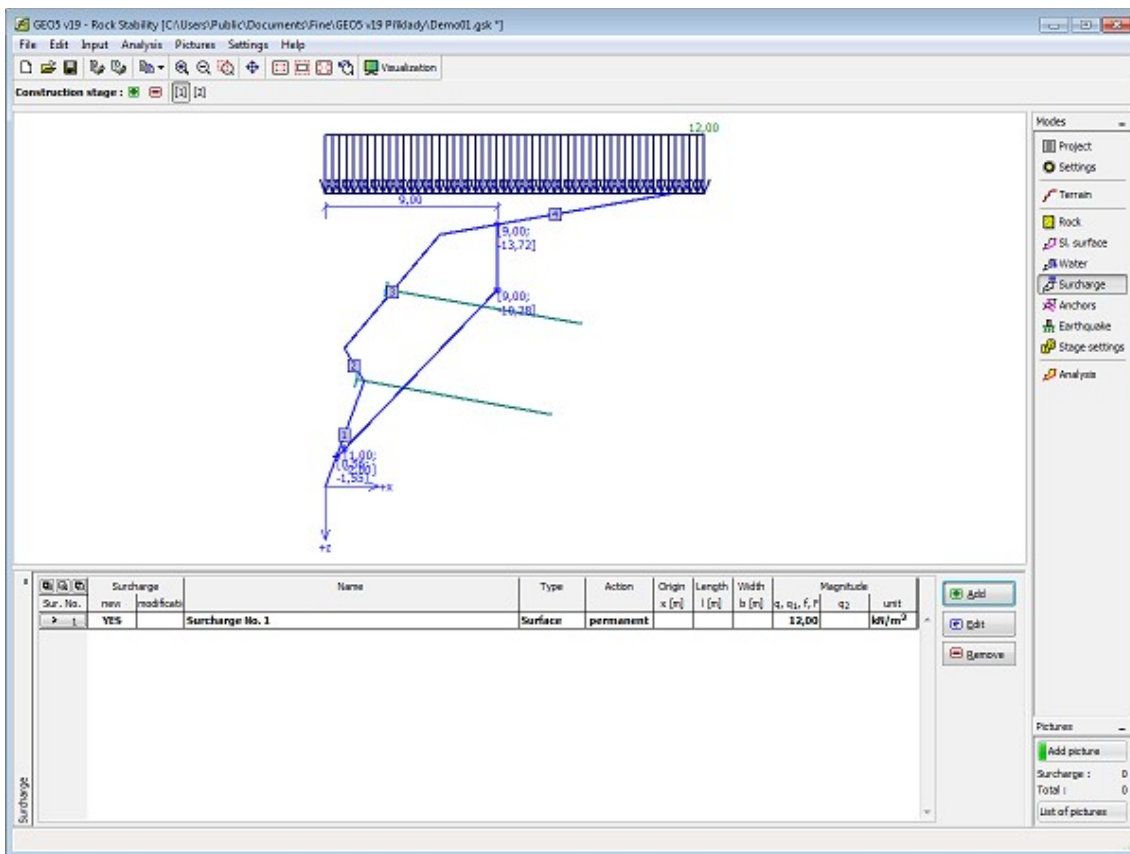
Frame "Water" - polygonal slip surface

## Surcharge - Plane and Polygonal Slip Surface

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or active objects, respectively.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Introducing surcharge forces into the analysis differs for a plane and a polygonal slip surface.



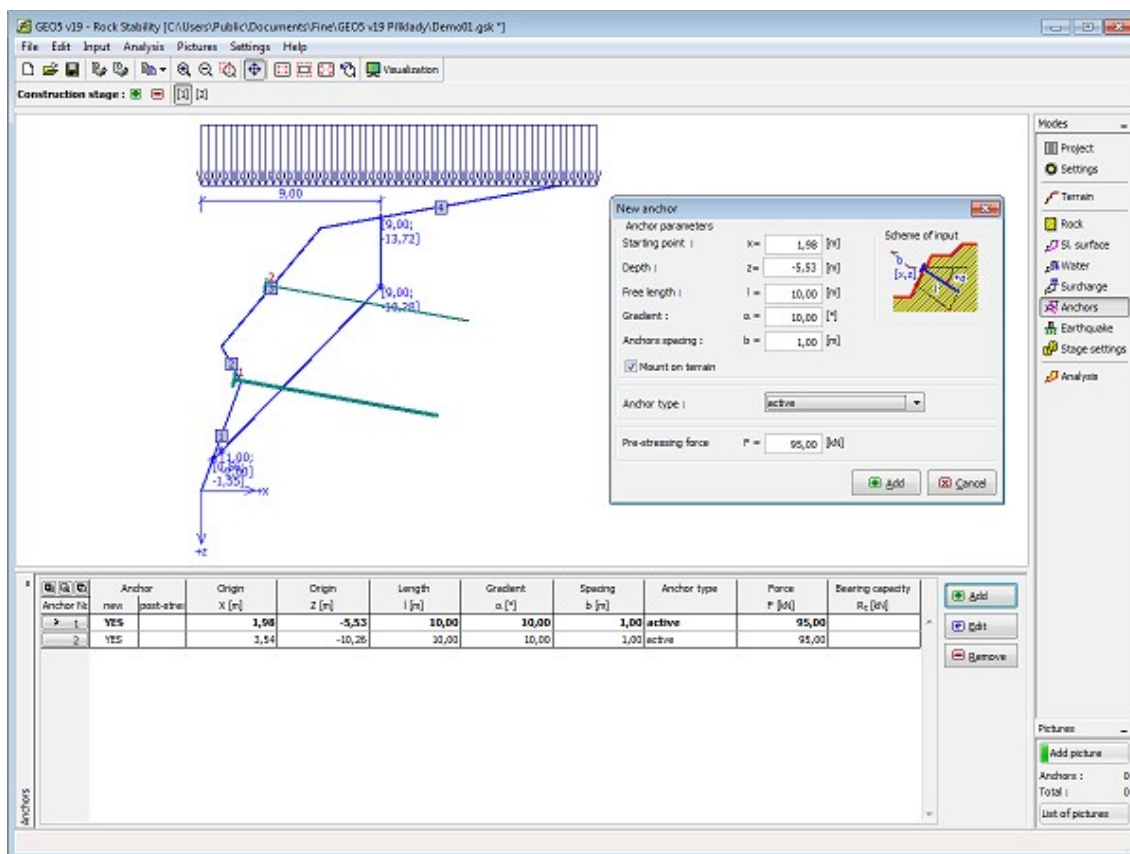
Frame "Surcharge" - plane and polygonal slip surface

## Anchors - Plane and Polygonal Slip Surface

The "Anchors" frame contains a table with a list of input anchors. Adding (editing) anchors is performed in the "New anchor (Modify anchor parameters)" dialog window. The input anchors can be edited on the desktop with the help of active objects.

The following is specified - location (origin), depth, free length, anchor slope, spacing between anchors and anchor force. The anchor origin can automatically be positioned on terrain (by checking the particular entry field). All anchor parameters can be modified only in the construction stage, where it was introduced. The subsequent stages allow only for adjusting the anchor force (option "Post-stressing anchor").

The plane slip surface allows for defining active and passive anchors. Only active anchors are allowed with the polygonal slip surface.

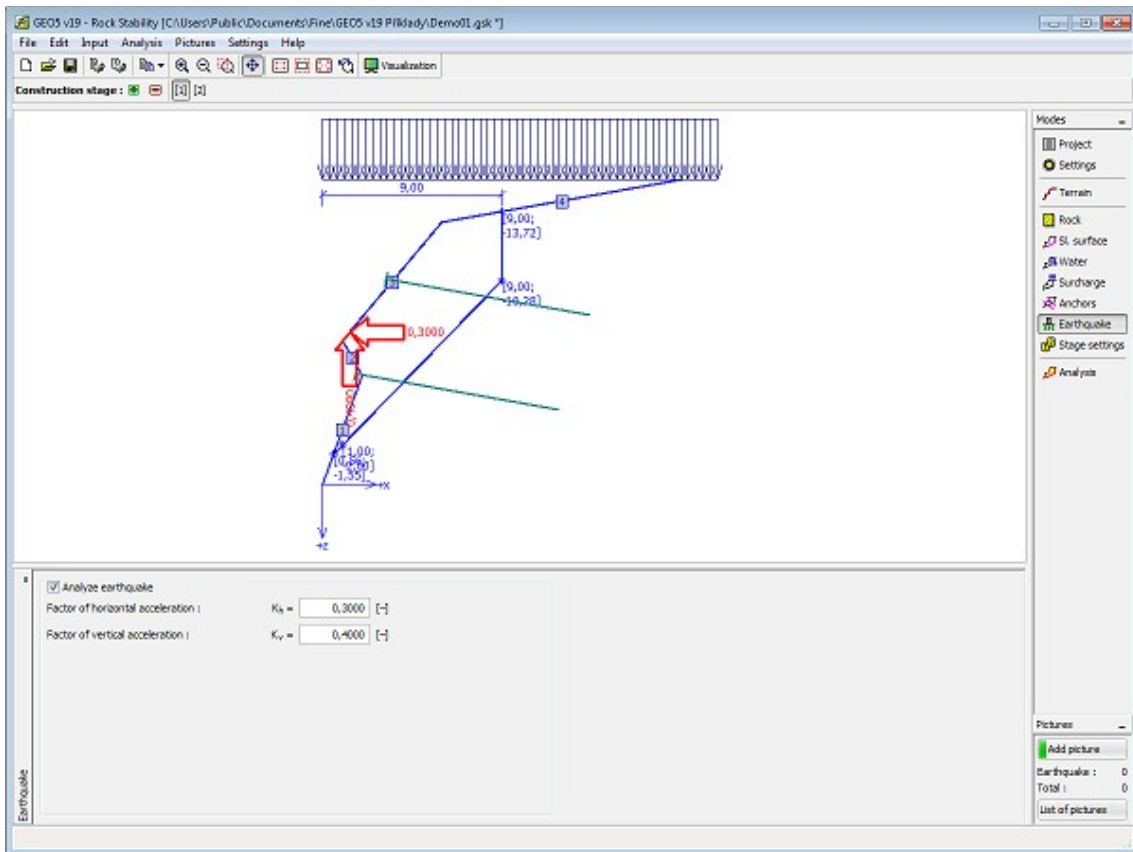


Frame "Anchors" - plane and polygonal slip surface

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

Rock slope analysis while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



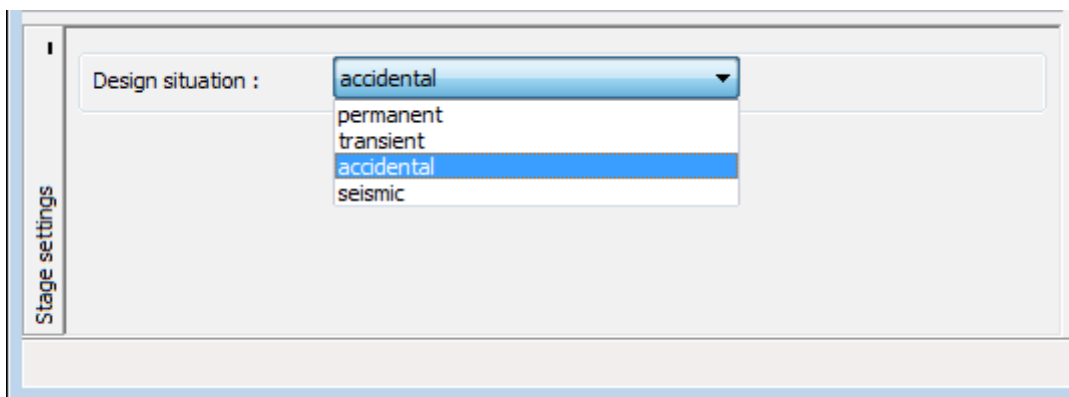
Frame "Earthquake"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Service I**, **Extreme I**).



Frame "Stage settings"

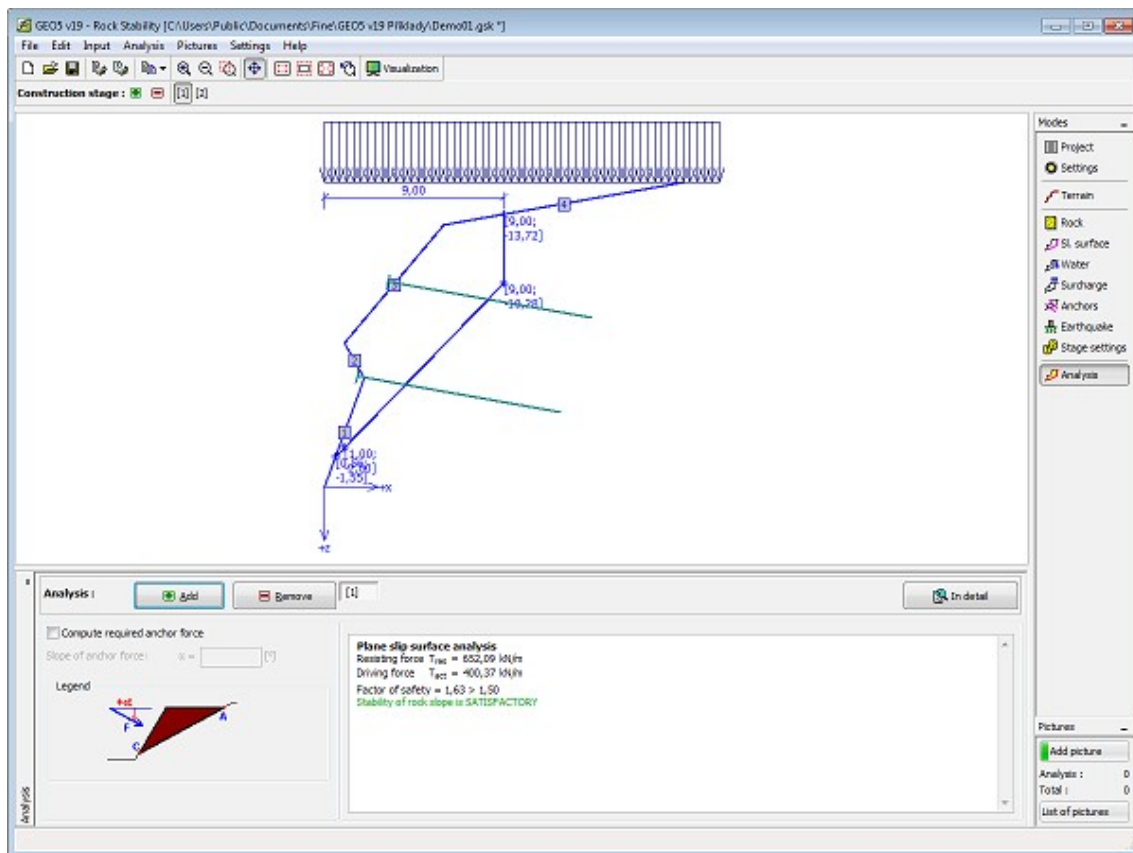
## Analysis - Plane Slip Surface

The "**Analysis**" frame displays the analysis results. Several analyses can be performed for a single task.

Assessment of the rock slope for **plane slip surface** can be carried out according to the selected **verification methodology** based on the input in the "**Settings**" frame. The **analysis** results are displayed in the frame in the bottom part of desktop.

In this frame the program makes it possible to determine the **anchor force needed** for obtaining the required safety factor. In such a case the "**Compute required anchor force**" entry field must be checked and the slope of anchor force from horizontal must be entered.

Visualization of results can be adjusted in the "**Visualization style settings**" dialog window.



Frame "Analysis" - plane slip surface

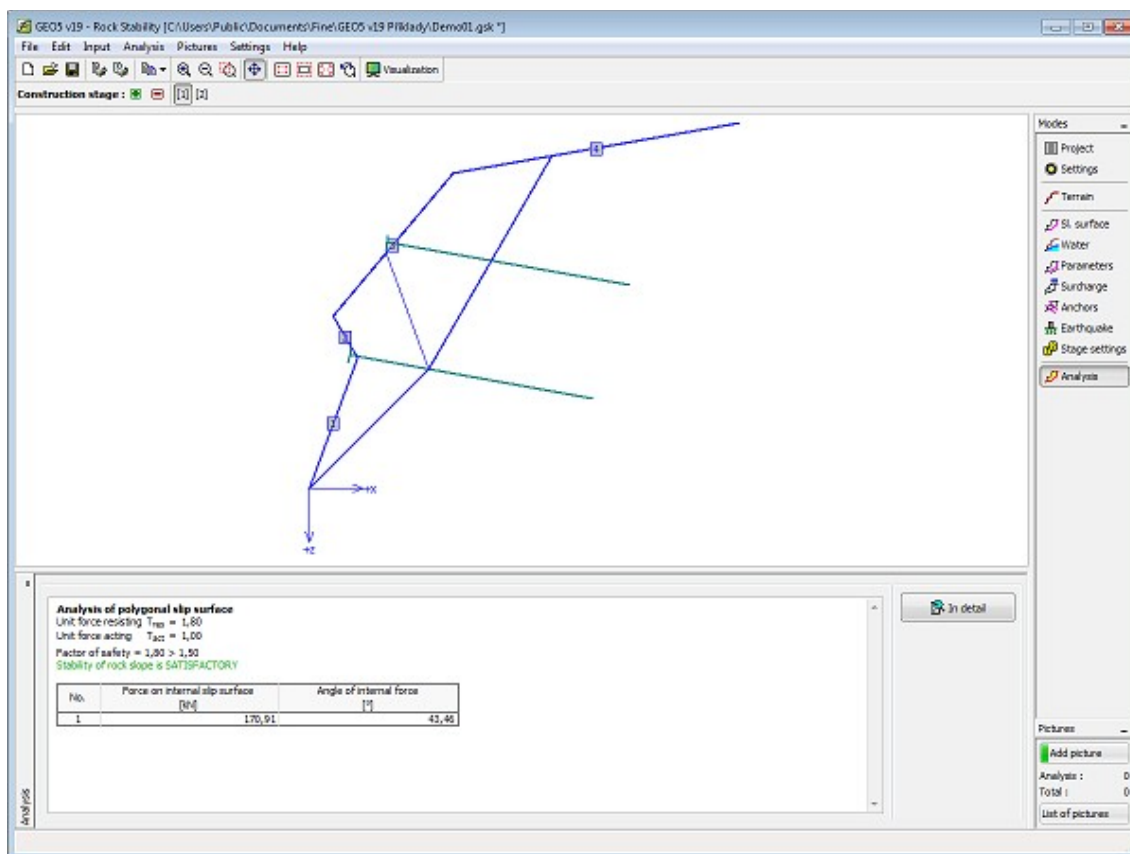
## Analysis - Polygonal Slip Surface

The "**Analysis**" frame displays the analysis results. Several analyses can be performed for a single task.

Assessment of the rock slope for **polygonal slip surface** can be carried out according to the selected **verification methodology** based on the input in the "**Settings**" frame. The **analysis** results are displayed in the frame in the bottom part of desktop.

Visualization of results can be adjusted in the frame "**Visualization settings**" dialog window.





Frame "Analysis" - polygonal slip surface

## Geometry

The **frame "Geometry"** allows for entering the shape of a rock slope (earth wedge).

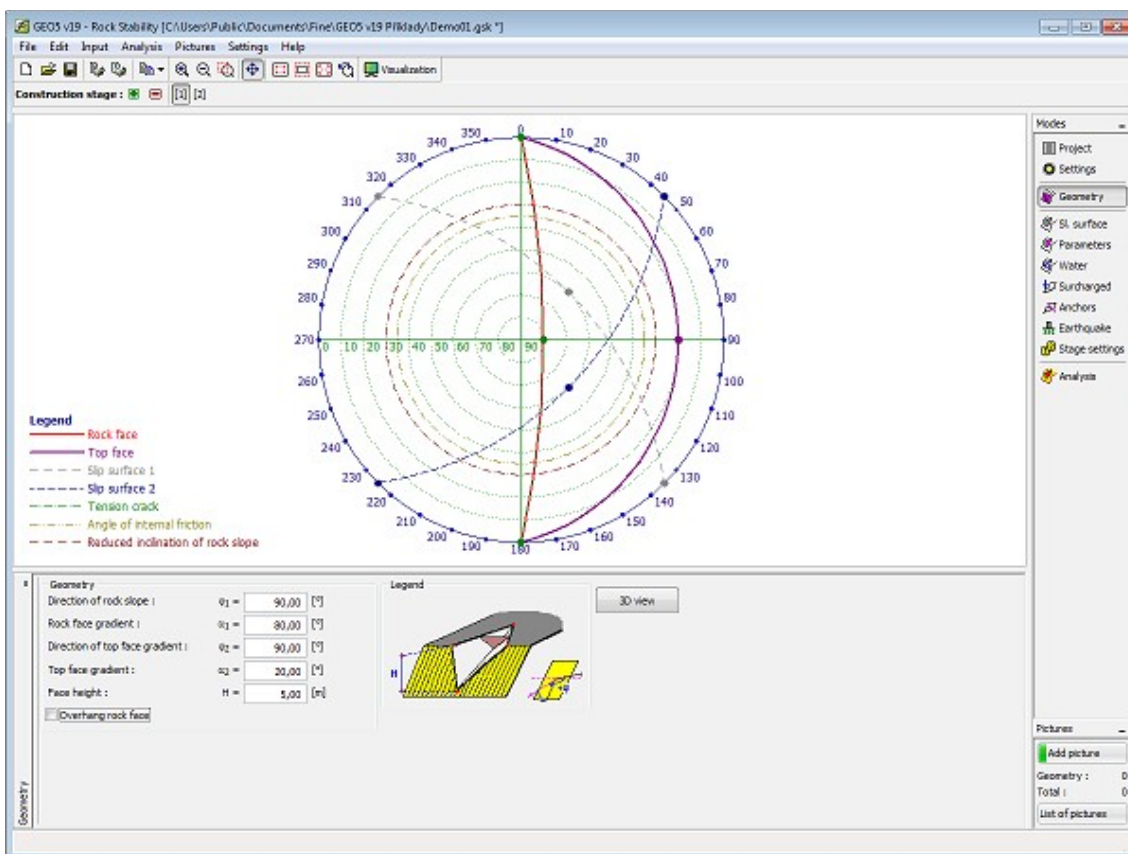
Geometry of earth wedge is defined by **directions and gradients of fall lines of faces** forming the wedge. Geometry of earth wedge is displayed on desktop using a **stereographic projection**.

The **"3D view"** button opens the dialog window for viewing an earth wedge in space.

Pressing the button **"Overhang rock face"** can be modeled **overhang rock faces**.

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.

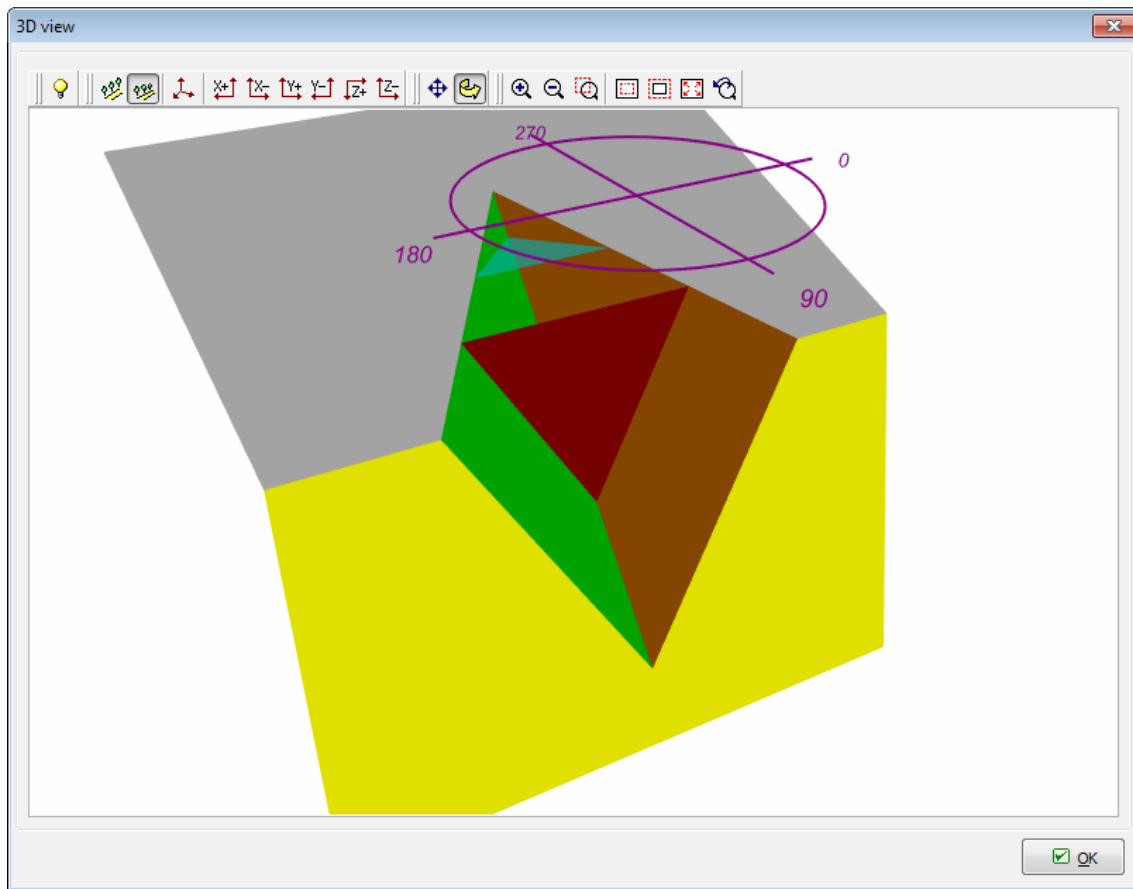




Frame "Geometry" - input using directions and gradients of fall lines of faces

### 3D View

3D view allows for **graphical check of defined values**. The picture can be rotated, translated, zoomed in and out and highlighted in a standard way.

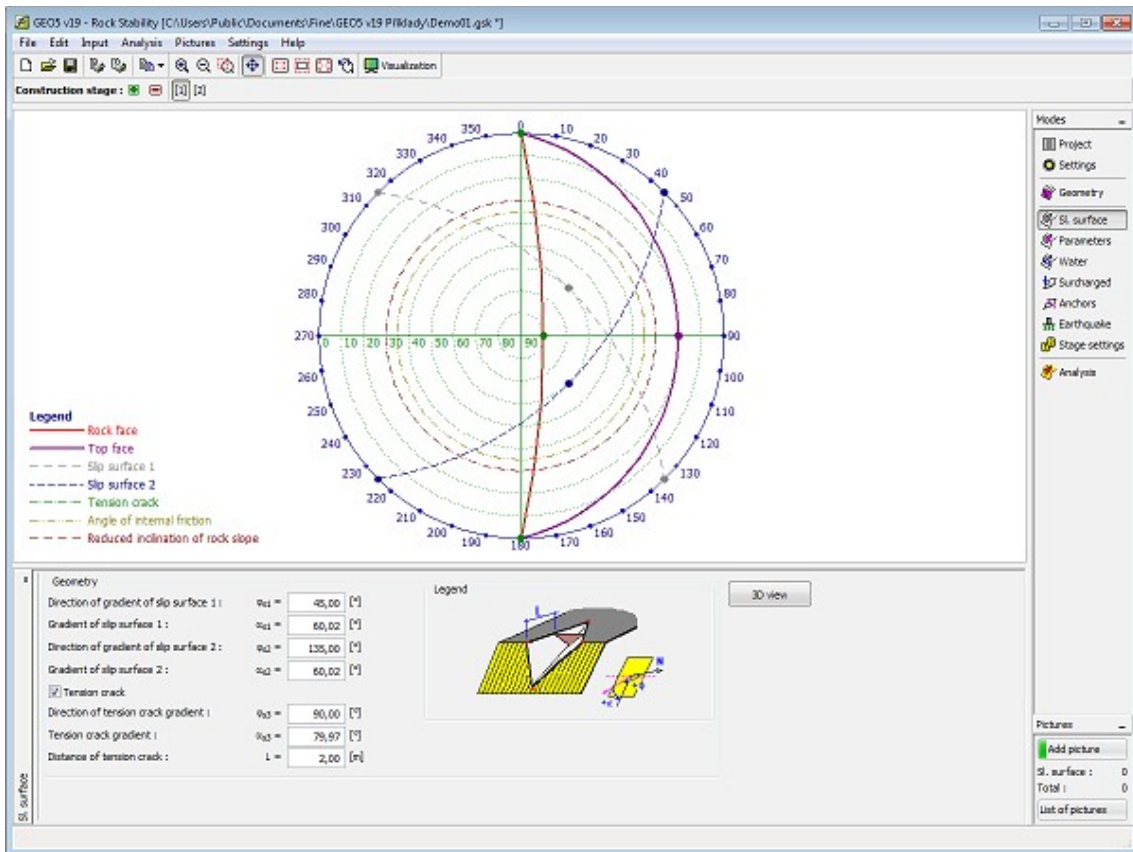


*Dialog window "3D view"*

## Slip Surface - Rock Wedge

The frame "**Slip surface**" serves to enter the shape of a slip surface using **directions and gradients of fall lines of faces** forming the wedge. A tension crack can also be defined. Geometry of earth wedge is displayed on desktop using a **stereographic projection**.

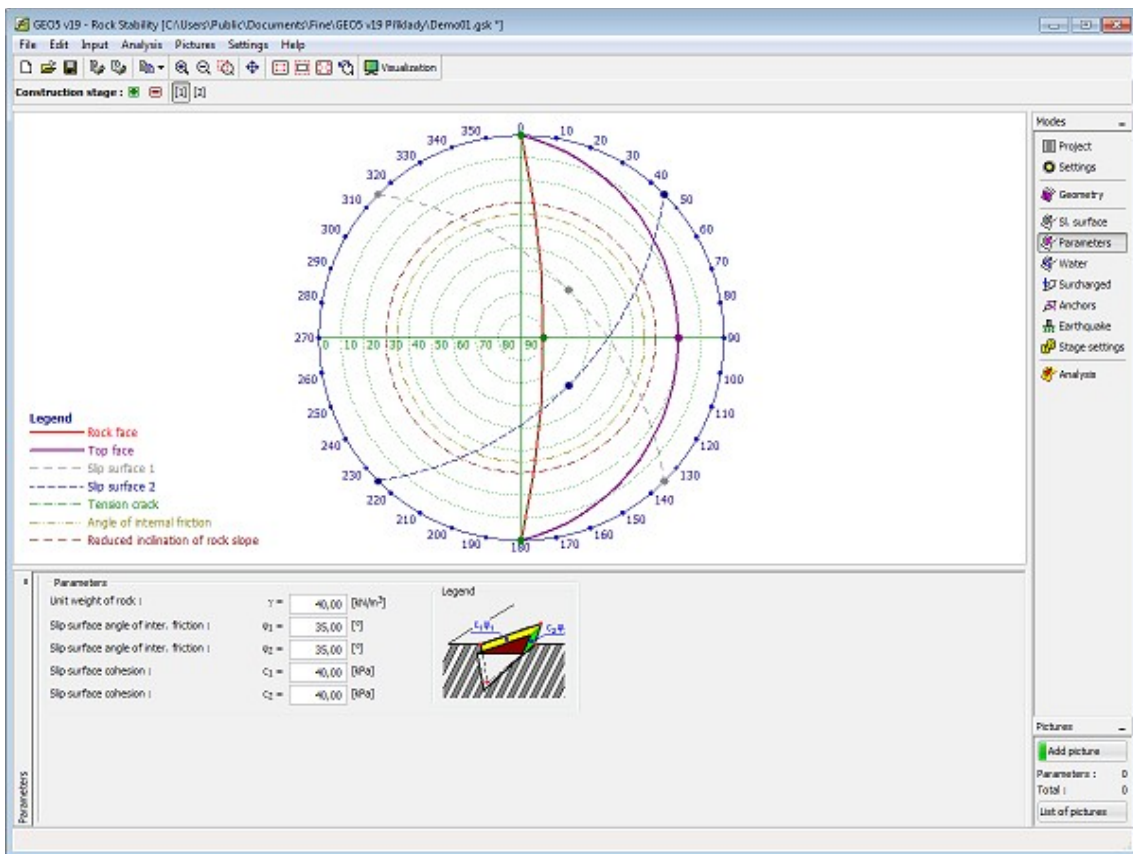
The "**3D view**" button opens the dialog window for viewing an earth wedge in space.



Frame "Slip surface" - rock wedge

## Parameters - Rock Wedge

The frame "Parameters" serves to enter parameters of an earth wedge. The unit weight of a rock and the Mohr-Coulomb strength parameters of slip surfaces must be specified.

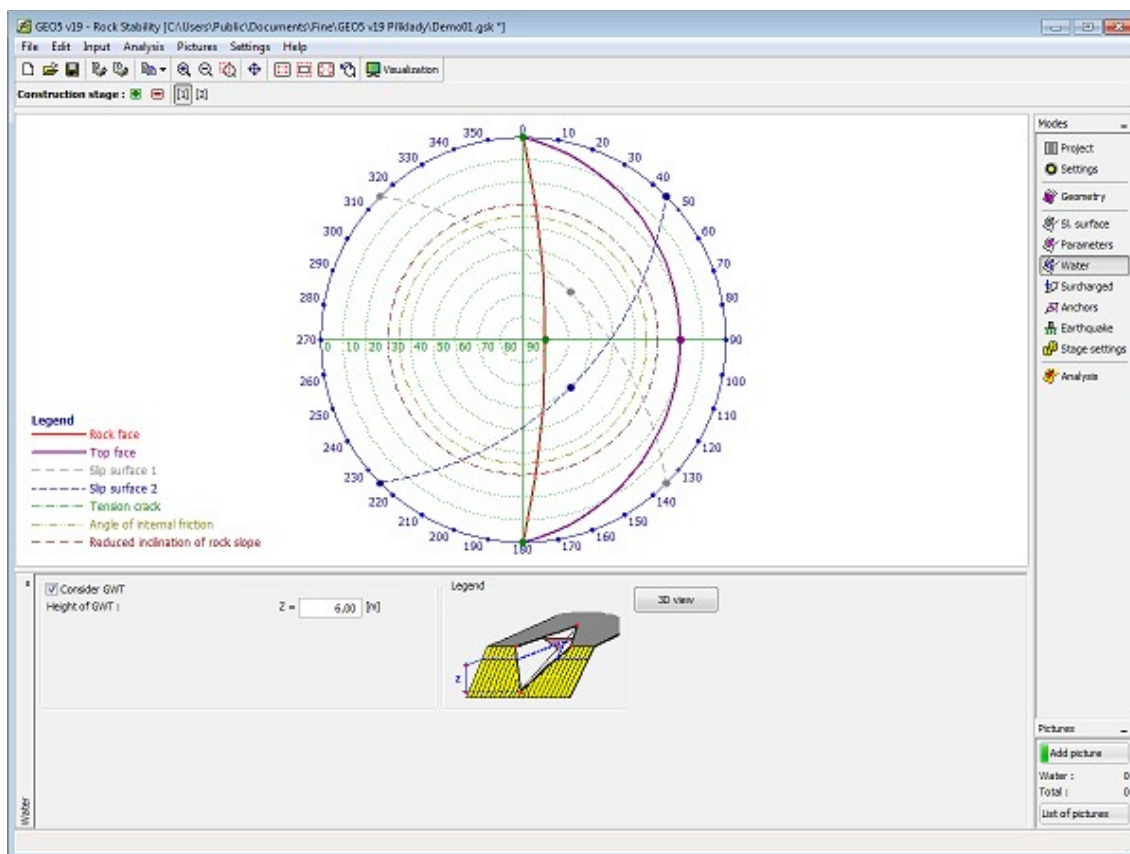


Frame "Parameters" - rock wedge

## Water - Rock Wedge

The frame **"Water"** allows for introducing water into analysis. If the influence of water is taken into account then checking the respective entry field opens the field for entering the height of GWT above the lowest point of an earth wedge.

Solution procedure when accounting for water is described in the theoretical part of the help **"Influence of ground water"**. The **"3D view"** button opens the dialog window for viewing an earth wedge in space.

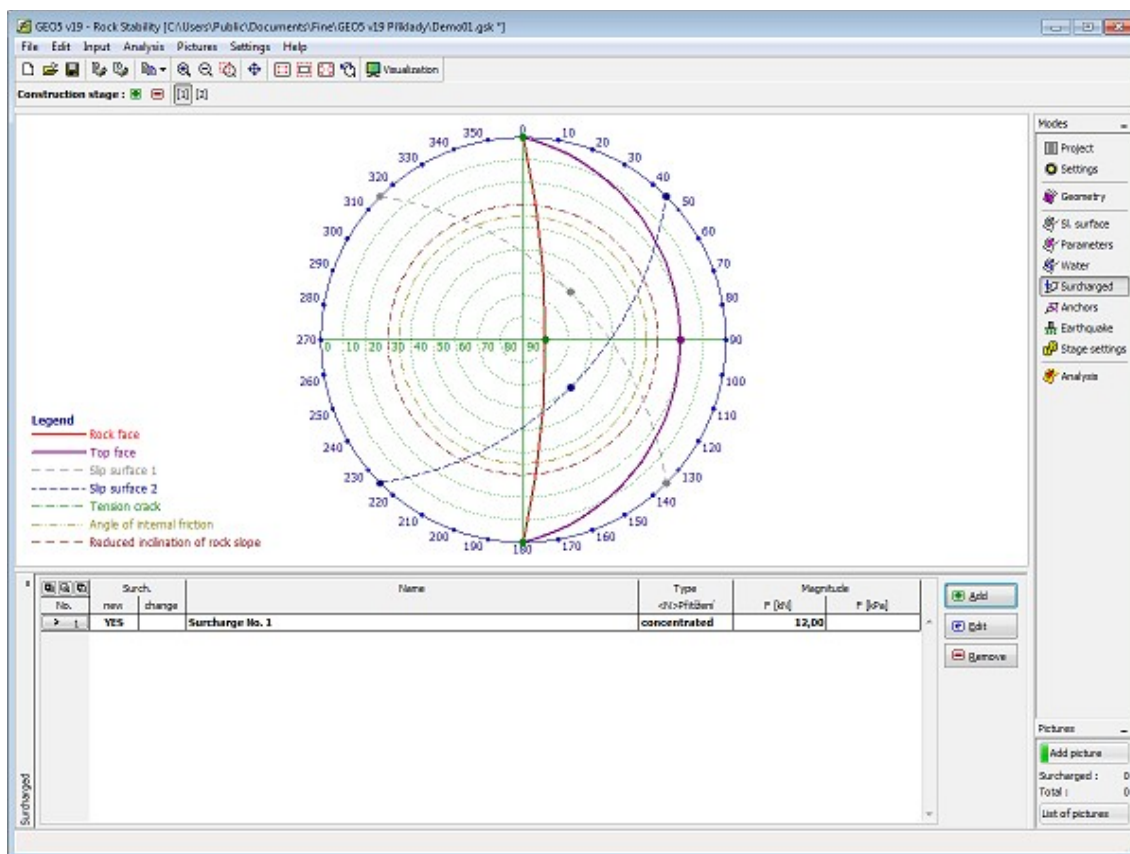


Frame "Water" - rock wedge

## Surcharge - Rock Wedge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New (edit) surcharge**" dialog window.

Surcharge forces are introduced into the stability analysis of earth wedge using resolution of forces.

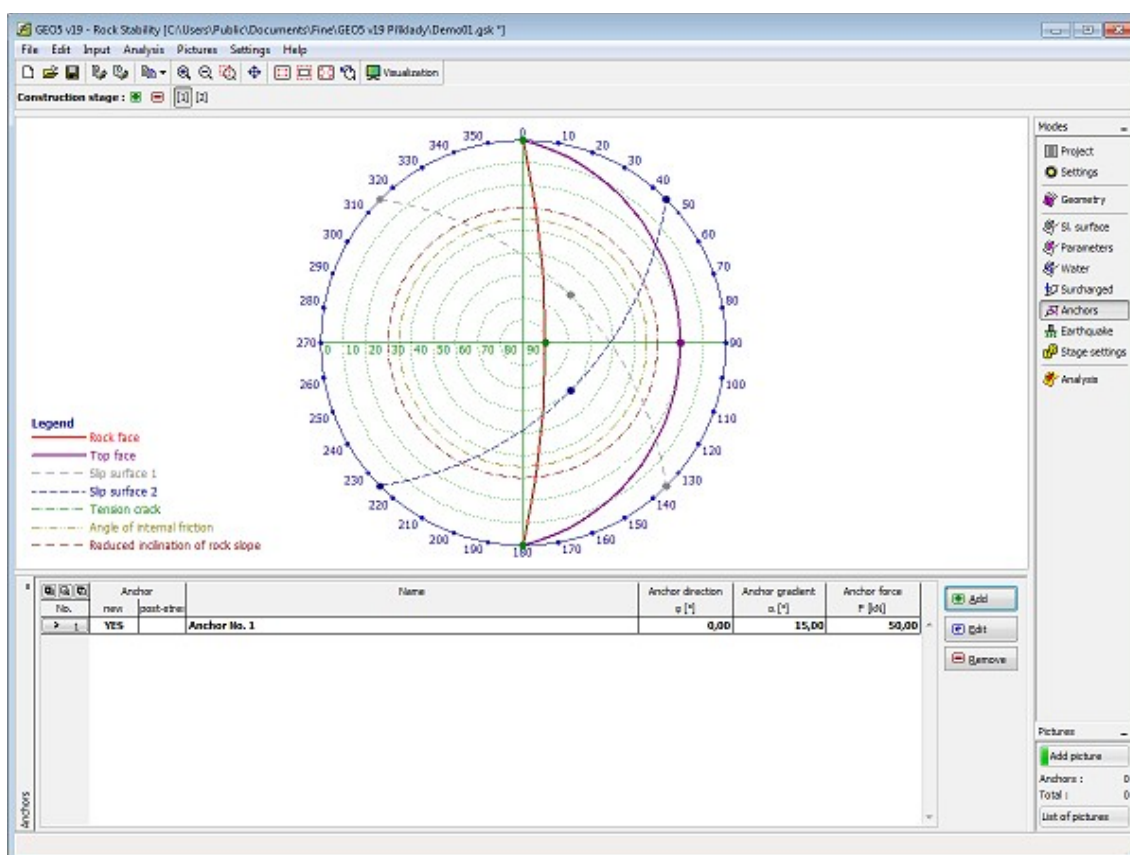


Frame "Surcharge" - rock wedge

## anchors - Rock Wedge

The "anchors" frame contains a table with a list of input anchors. Adding (editing) anchors is performed in the "New anchor (Modify anchor parameters)" dialog window.

Anchor forces are introduced into the stability analysis of earth wedge using resolution acting of forces.



Frame "Anchors" - rock wedge

## Analysis - Rock Wedge

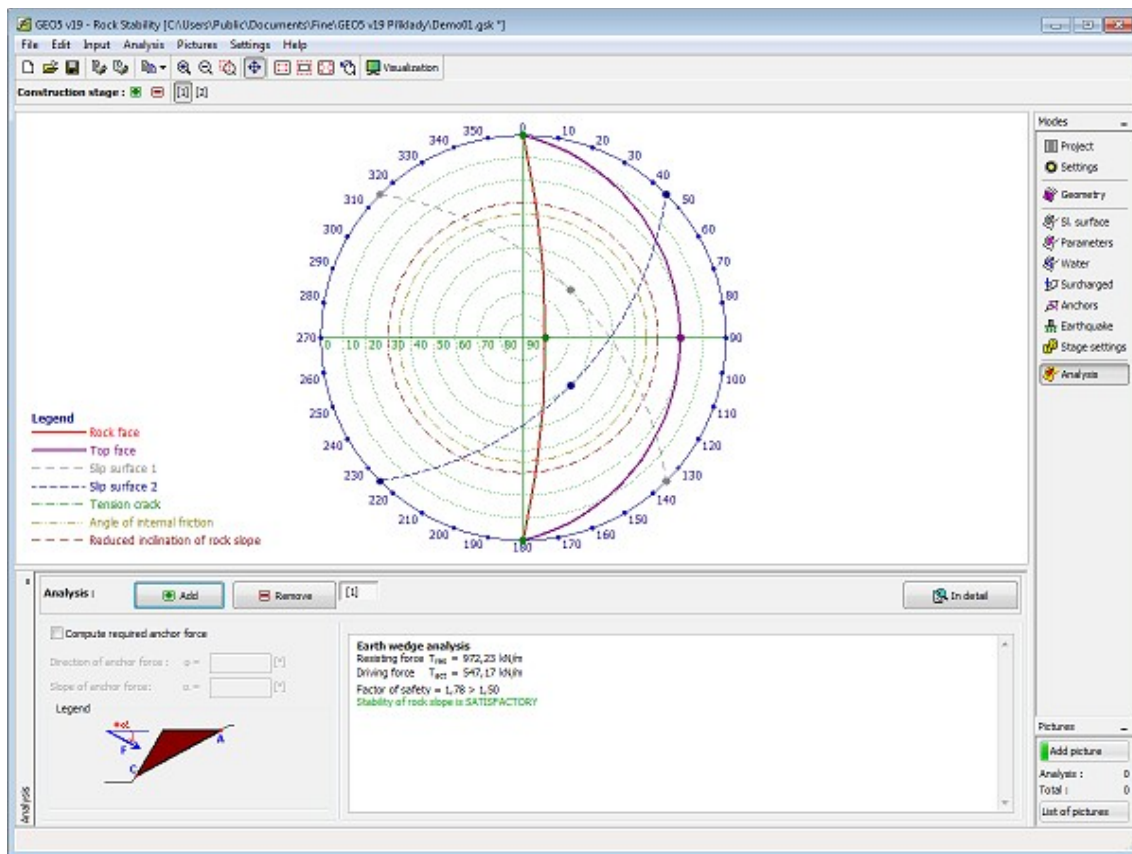
The "**Analysis**" frame displays the analysis results. Several analyses can be performed for a single task.

Assessment of the rock slope, that is considered as rock wedge can be carried out according to the selected verification methodology based on the input in the "Settings" frame. The analysis results are displayed in the frame in the bottom part of desktop.

In this frame the program makes it possible to determine the **anchor force needed** for obtaining the required safety factor. In such a case the "**Compute required anchor force**" entry field must be checked and the slope of anchor force from horizontal and its direction must be entered.

Visualization of results can be adjusted in the frame "Visualization settings" dialog window.





Frame "Analysis" - rock wedge

## Program MSE Wall

This program is used for verification of mechanically stabilized earth walls and segmental retaining walls reinforced by geogrids (georeinforcements).

The help in the program "MSE Wall" includes the following topics:

- Input of data into individual frames:

Project	Settings	Geometry	Material	Types of Reinforcement	Reinforcement (Blocks)	Reinforcement
Profile	Soils	Assign	Terrain	Water (Blocks)	Water	Surcharge
Front Face Resistance	Applied Forces	Earthquake	Stage Settings	Verification	Dimensioning	Bearing Capacity
Slip on Georeinforcement	Internal Stability	Global Stability	Stability			



- Standards and analysis methods
- Theory for analysis in the program "**MSE Wall**":
  - Stress in Soil
  - Earth Pressures
  - Analysis of Walls
  - Slope Stability
  - MSE Wall Body
- Outputs
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

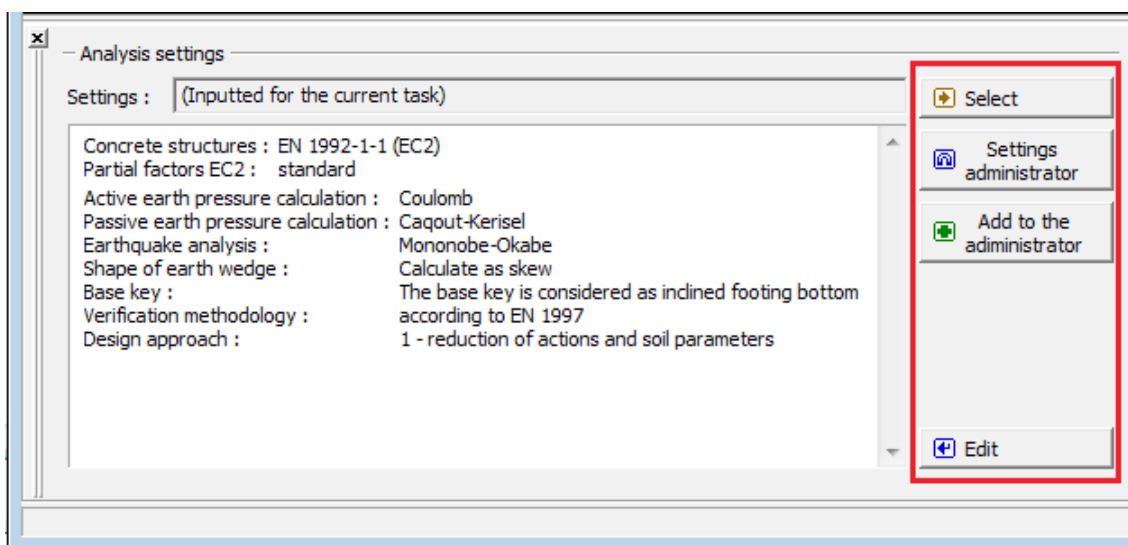
The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Materials and standards"**, **"Wall analysis"** and **"Stability analysis"** tabs.



Frame "Settings"

## Geometry

The **"Geometry"** frame allows by pressing the button for selecting the wall shape.

The shape of a wall can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The first type of geometry (wall) enables to define **foundation**, for other types (embankments) the program allows for inputting the **cover**. The selected type of geometry influences other frames and their inputting modes (water, surcharge, reinforcement). The following **verification** options are available for individual types of geometries:

### Geometry type

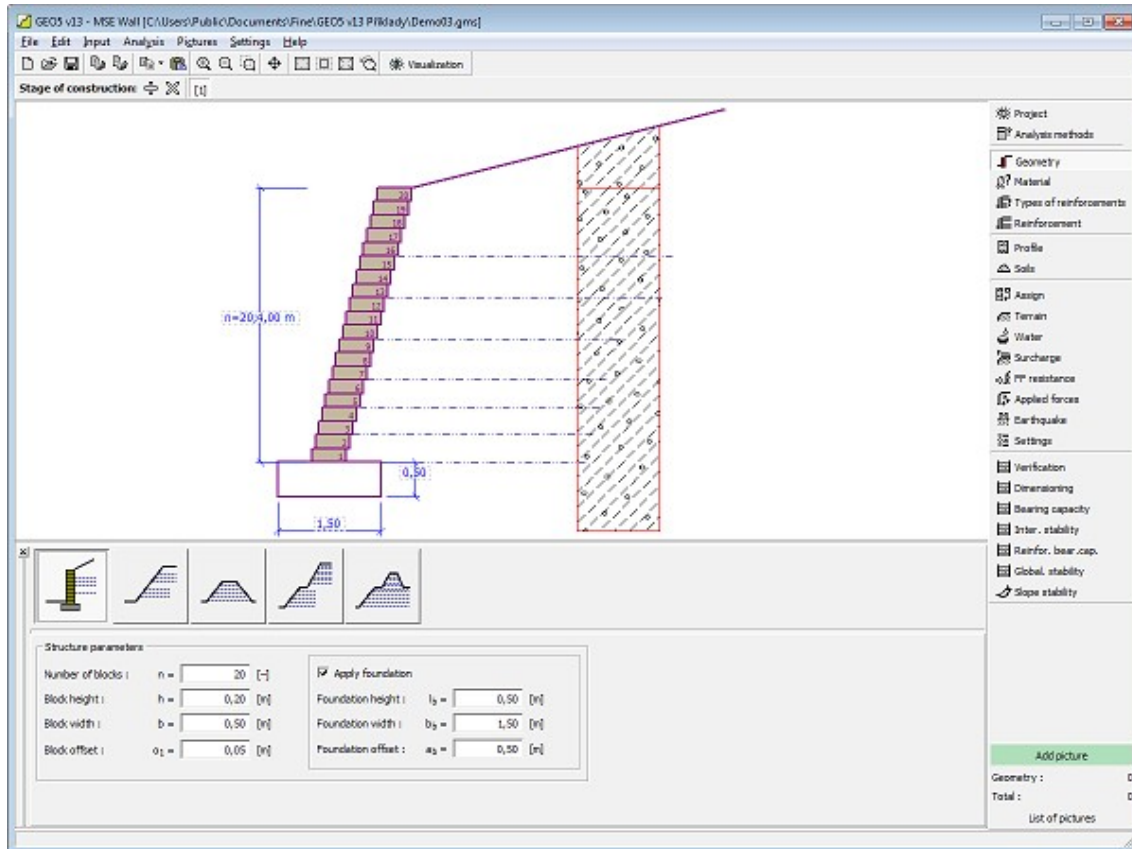
- 1 Wall with the option to define foundation
- 2 One-sided slope

### Verification

- Verification, dimensioning, bearing capacity, internal stability, reinforcement bearing capacity, global stability, slope stability
- Verification, bearing capacity, internal stability, global stability, slope stability

- 3 Two-sided slope Global stability, slope stability
- 4 One-sided zoned by benches Verification, bearing capacity, internal stability, global stability, slope stability
- 5 Two-sided zoned by benches Global stability, slope stability

The program makes it possible to export the geometry of a structure in the \*.DXF format.



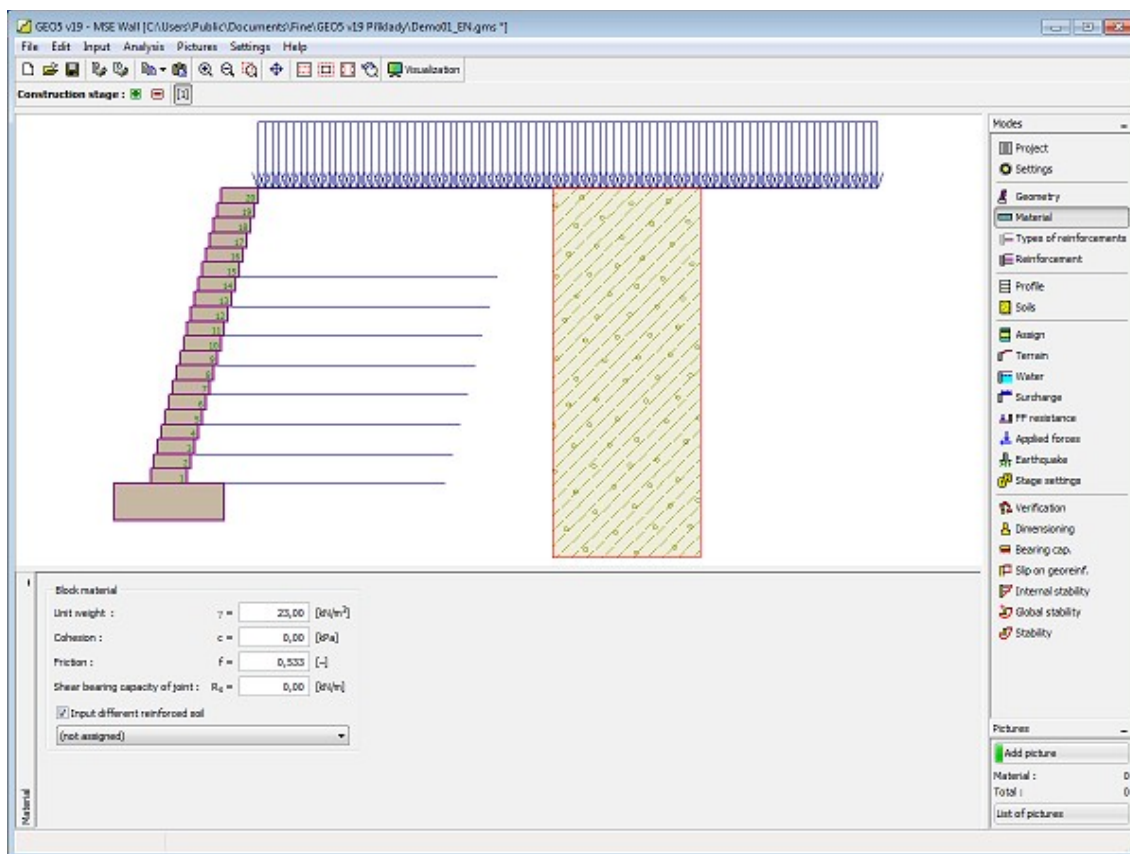
Frame "Geometry"

## Material

The "Material" frame serves to choose parameters of the material adopted for blocks or cover. Defining materials depends on the selected type of "Geometry". The first type of geometry (structure with blocks) requires inputting the unit weight of blocks  $\gamma$ , cohesion  $c$ , friction  $f$  and shear bearing capacity of joint  $R_s$  [kN/m].

The other types of geometry (structure without blocks) enable to consider a cover, which requires inputting the unit weight  $\gamma$  and shear resistance  $R_s$  [kPa].

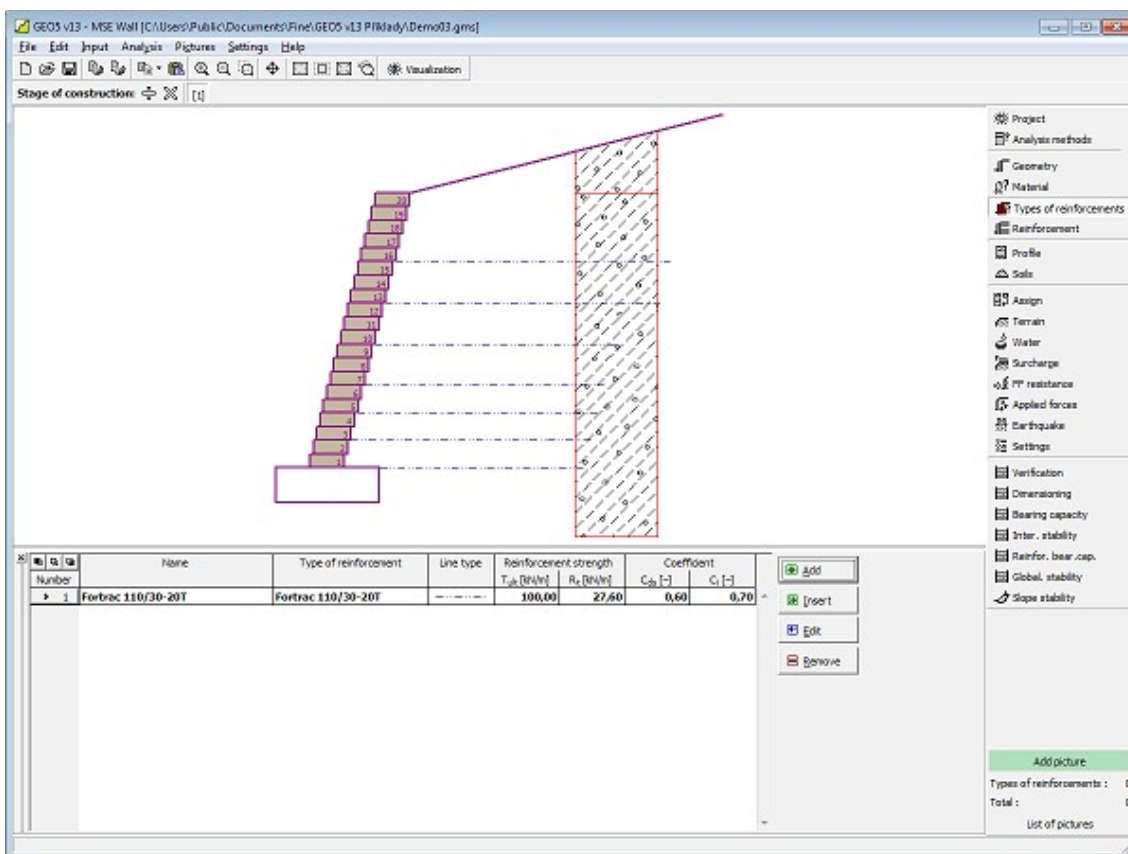
If the soil between the reinforcements is different than soil assigned to geological profile, then program allows to specify this soil by checking the option "Input different reinforced soil". Subsequently, in combo list user selects the type of soil (the combo list contains soils introduced in the frame "Soils").



Frame "Material"

## Types of Reinforcements

The "Types of reinforcements" frame contains a table with the list of input georeinforcements and their parameters (long-term bearing capacity of reinforcements and coefficients of interaction). Adding (editing) reinforcement is carried in the "New type of reinforcement (Edit type of reinforcement)" dialog window.



Frame "Types of reinforcements"

## Adding and Editing Type of Reinforcement

The "New type reinforcement (Edit type of reinforcement, Inserted type of reinforcement)" dialog window contains the following items:

- Reinforcement group and Type of reinforcement**
  - a combo list contains reinforcement group and individual types of reinforcements from a database, or its allows for inputting the "user-defined" type of reinforcement
- Short-term characteristic strength**
  - the strength value can be changed only for reinforcements stored with the help of "User's catalog"
- Analysis of long-term strength**
  - a combo list enables to choose the way of analysis of long-term strength: "input reduction factors" (direct input of factors), "calculate reduction factors" (factors are determined based on the selected lifetime of a reinforcement, soil Ph and grain size) or "input strength" (already reduced long-term strength is input)
- Reduction factors**
  - the values of factors reducing a short-term tensile strength - can be input directly or calculated based on the selected options in combo lists (lifetime, chemistry, grain size)
- Overall coefficient of model uncertainty**
  - the value of factor to reduce a short-tem strength is input
- Long-term design**
  - calculated value of s long-term tensile strength

**strength**

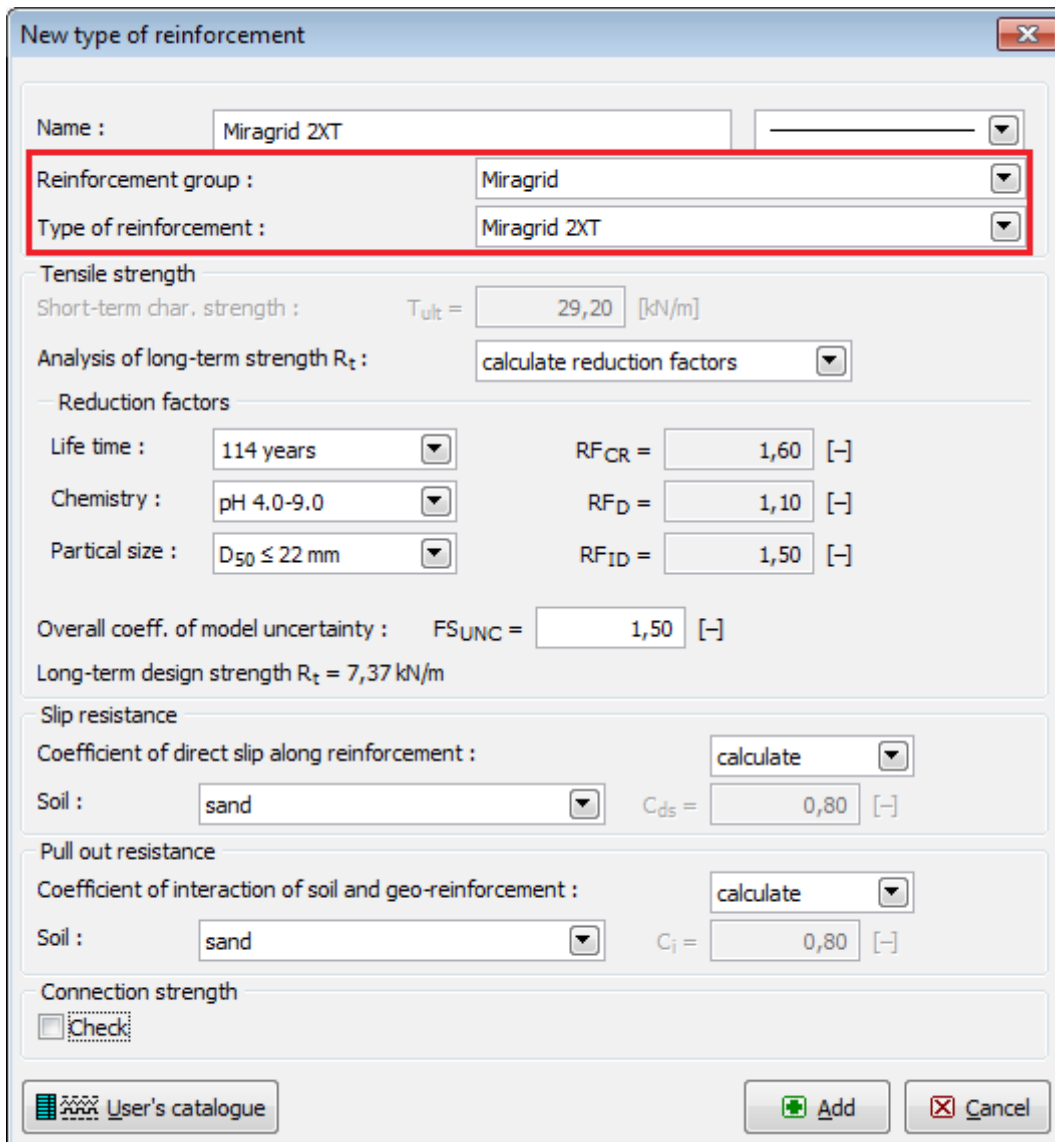
**Slip resistance**

- the "**Coefficient of direct slip along reinforcement**" can be input directly or calculated based on the type of soil

**Pull out resistance**

- the "**Coefficient of interaction of soil and geo-reinforcement**" can be input directly or calculated based on the type of soil

The "**User's catalog**" button in the bottom part of the window opens the "**User's catalog**" dialog window.



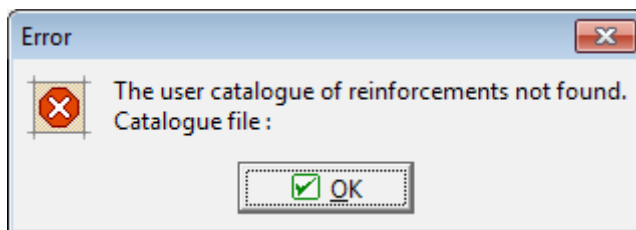
*Dialog window "New type of reinforcement"*

**User's Catalog**

The user's catalog allows to define and store own types of reinforcements and their material characteristics. At first use of the catalog (has not been yet created) the program prompts a warning message that no catalog was found. Then, pressing the button "**OK**" opens the "**Save**

as" dialog window that allows for entering the catalog name and saving it into a specified location by pressing the "Save" button (by default a folder used for saving the project data is assumed).

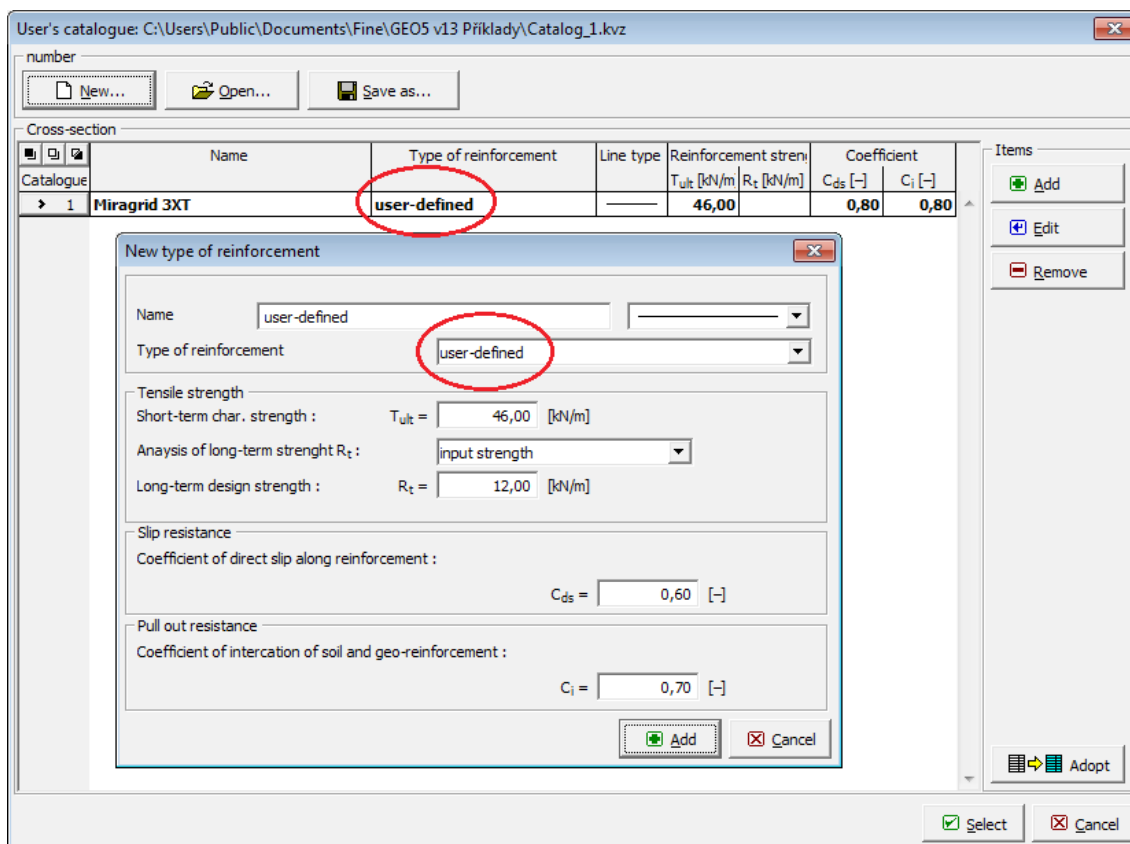
The program allows the user to create more than one catalog. The next catalog is created by pressing the "New" button - the program asks, whether the current catalog should be replaced (**the currently loaded catalog is not DELETED!**) and saves the new catalog under a new name. The "Open" button allows for load an arbitrary user catalog and by pressing the "Save as" button for saving it under a different name.



Dialog window at first use - user catalog of types of reinforcements

The "User catalog" dialog window contains a table listing the user defined reinforcements. The "Add" button opens the "New type of reinforcement" dialog window that allows for specifying and subsequent saving of characteristics of a new reinforcement into the catalog. Buttons "Edit" and "Remove" serve to edit individual items in the table.

The "Adopt" button accepts the current reinforcement characteristics specified in the "New type of reinforcement" dialog window and opens the "New type of reinforcement" dialog window that allows for their modification and saving.

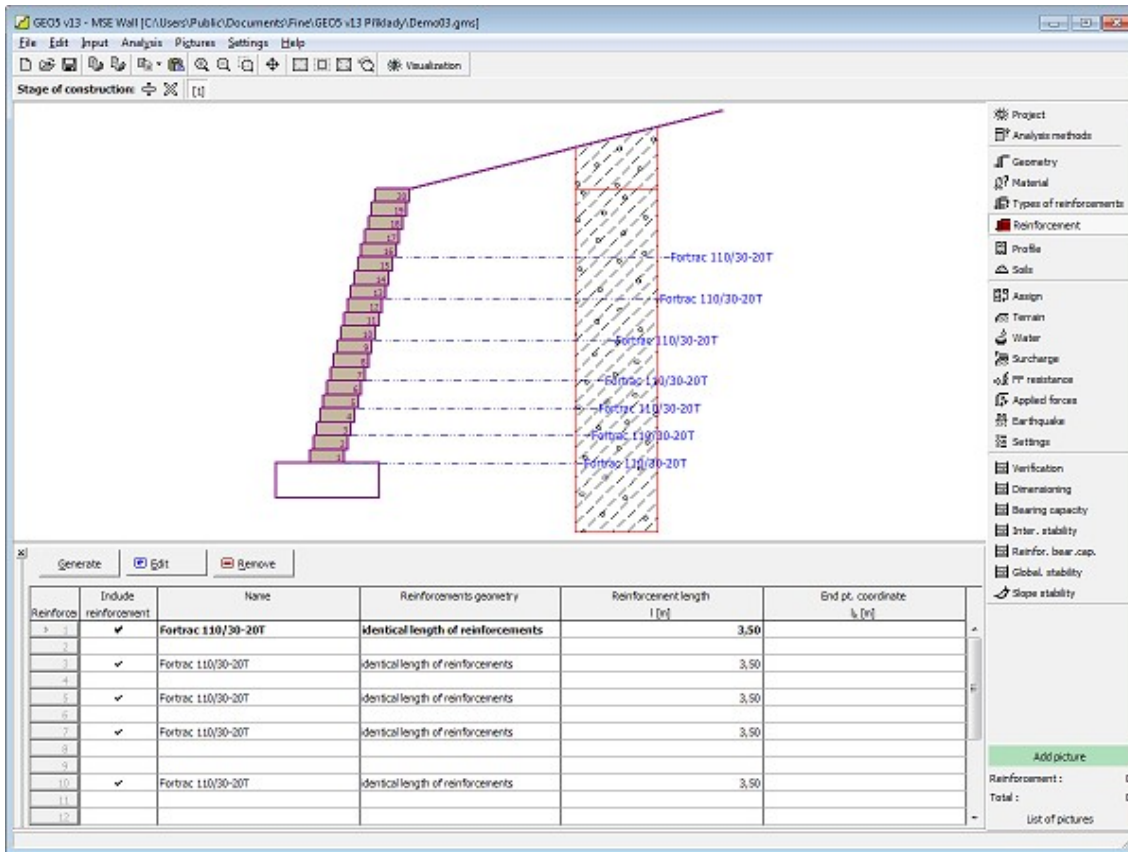


Dialog window "User's catalog"



# Reinforcement

The "**Reinforcement**" frame contains a table with the list of input geo-reinforcements and their geometries.

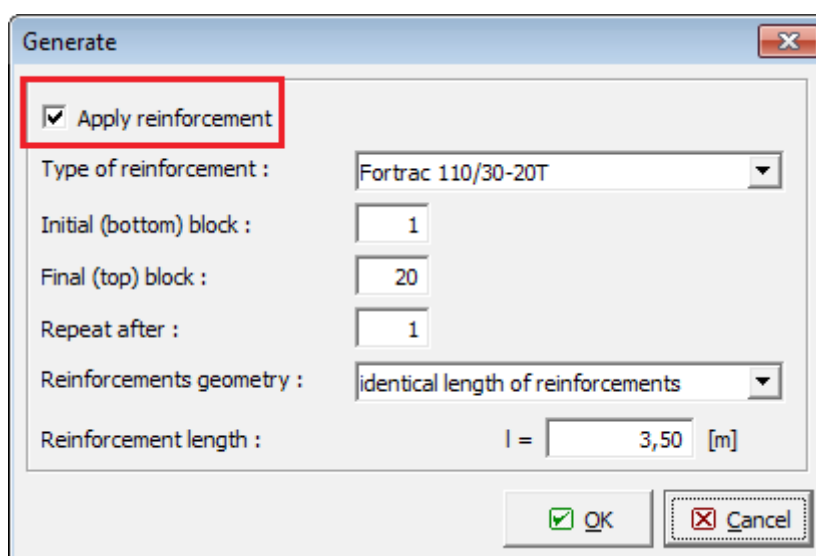


Frame "Reinforcement"

The "**Generate**" button opens the "**Generate**" dialog window that enables to set automatic parameters of generating group of reinforcements. Geo-reinforcements can be positioned only in joints between the blocks (checking the option "**Apply reinforcement**"). Next step is to define the type of reinforcement, the initial and the last block, the number of blocks to reenter the reinforcement, reinforcement geometry (the same length of reinforcements or the same type of reinforcement finishing). The input reinforcements can also be edited on the desktop with the help **active dimensions** or **active objects**, respectively.

The "**Edit**" button opens the "**Edit block**" dialog window that enables to change the type of reinforcement, its geometry or to specify whether the reinforcement between the blocks is to be considered. The "**Remove**" button removes **all** geo-reinforcements.

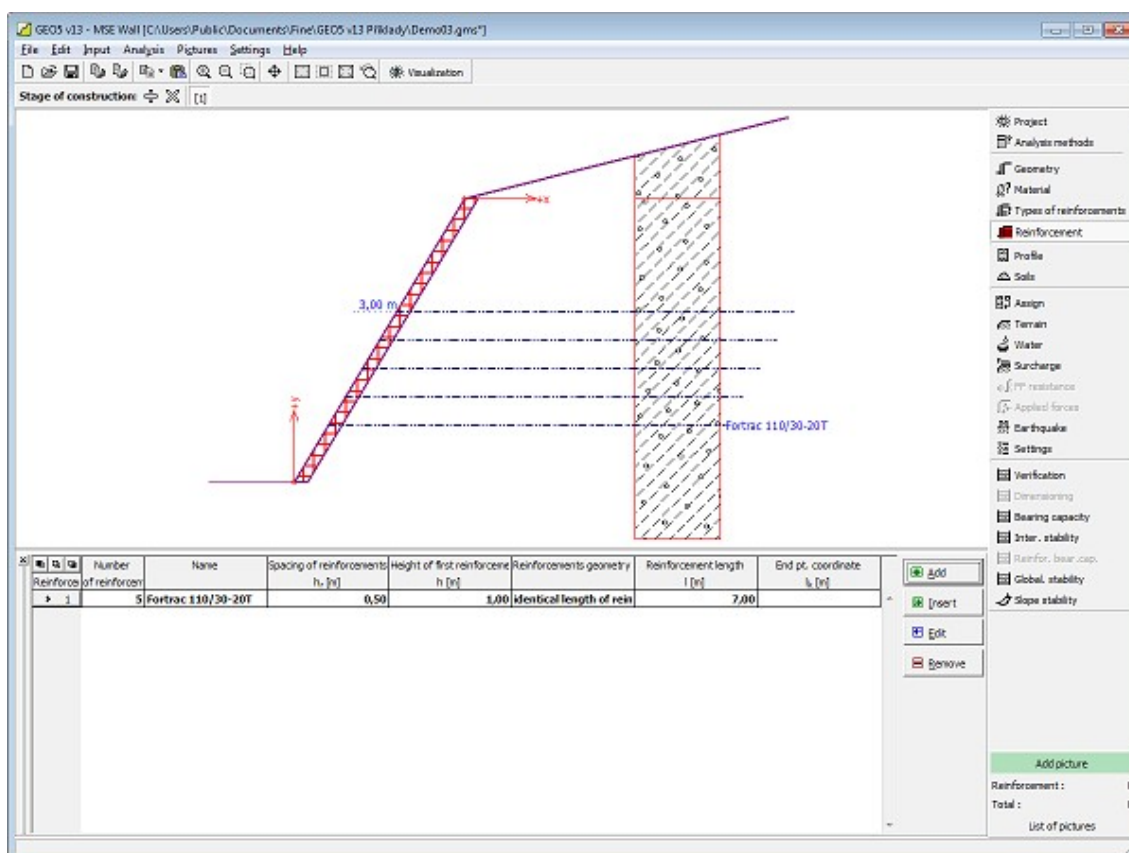




Dialog window "Generate"

## Reinforcement

The "**Reinforcement**" frame contains a table with the list of input groups of reinforcements and their geometries.

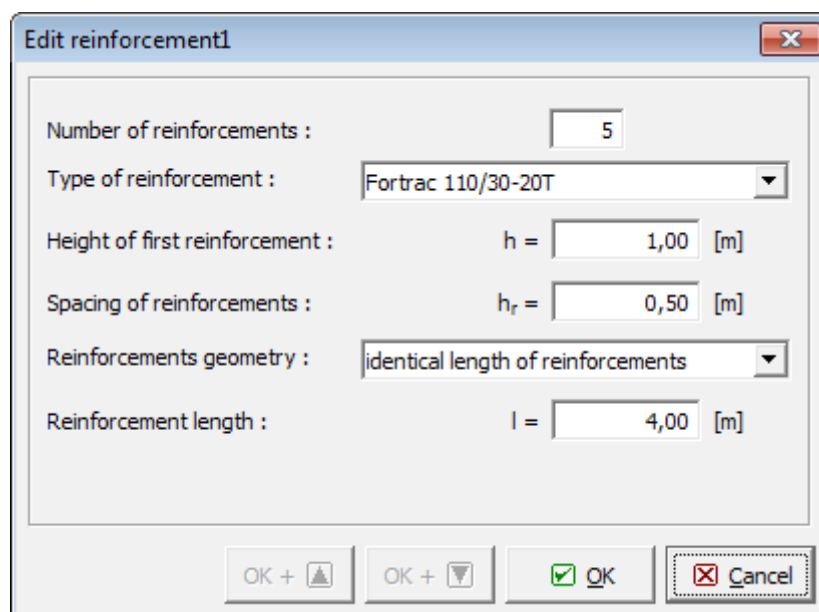


Frame "Reinforcement"

Adding (editing) groups of geo-reinforcements is performed in the "**New (edit)**"

**reinforcement" dialog window.** The input reinforcements can also be edited on the desktop with the help **active dimensions** or **active objects**, respectively. Each input group of reinforcements requires inputting in the dialog window the number of reinforcements and type, height of the first reinforcement, reinforcement spacing and their geometry.

The program allows for inserting (insert) another group of reinforcements in between the already input groups. Inserting a new group is performed in the "**Inserted reinforcement**" dialog window that is identical with the "**New reinforcement**" dialog window. The newly introduced (inserted) block is put below the currently selected block of a structure. The "**Remove**" button deletes a group of reinforcements.



*Dialog window "Edit reinforcement"*

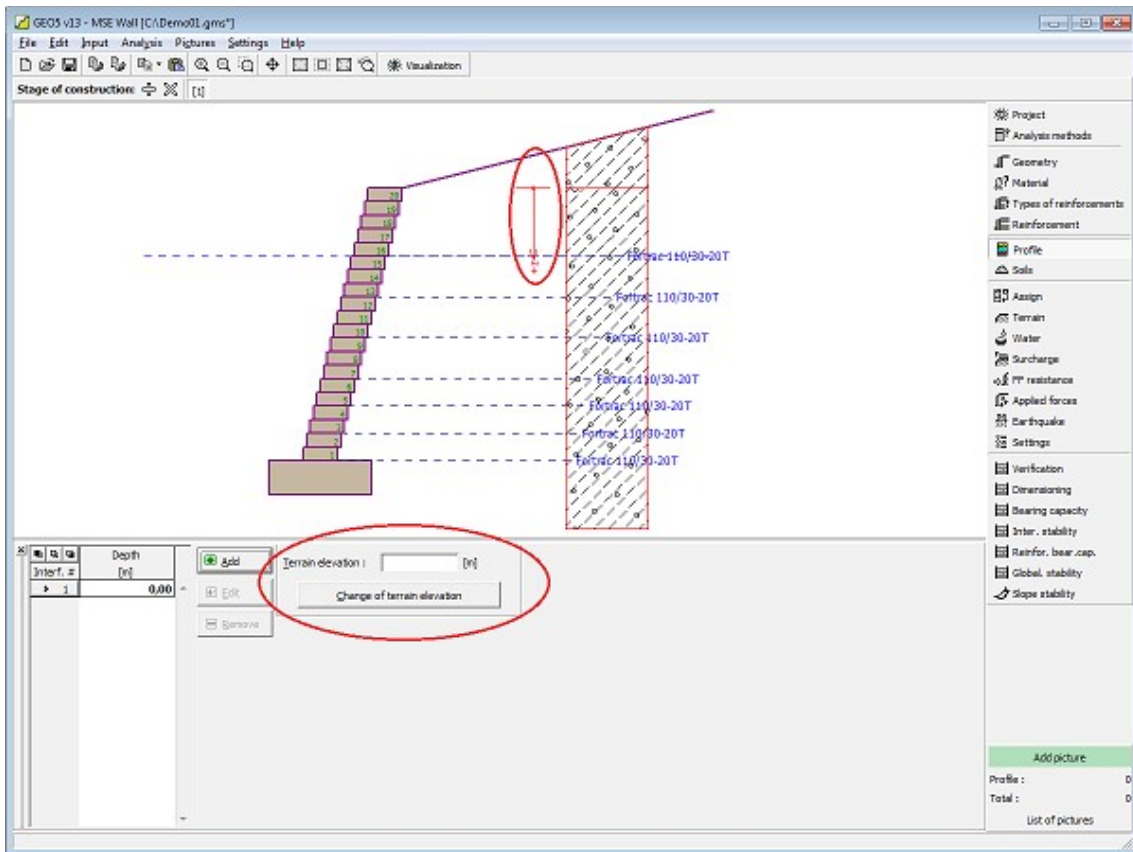
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of **active dimensions**.

**Adding** (editing) layer is performed in the "**New interface**" dialog window. The  $z$ -coordinate measured from the top point of a structure is specified ( $z$ -axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

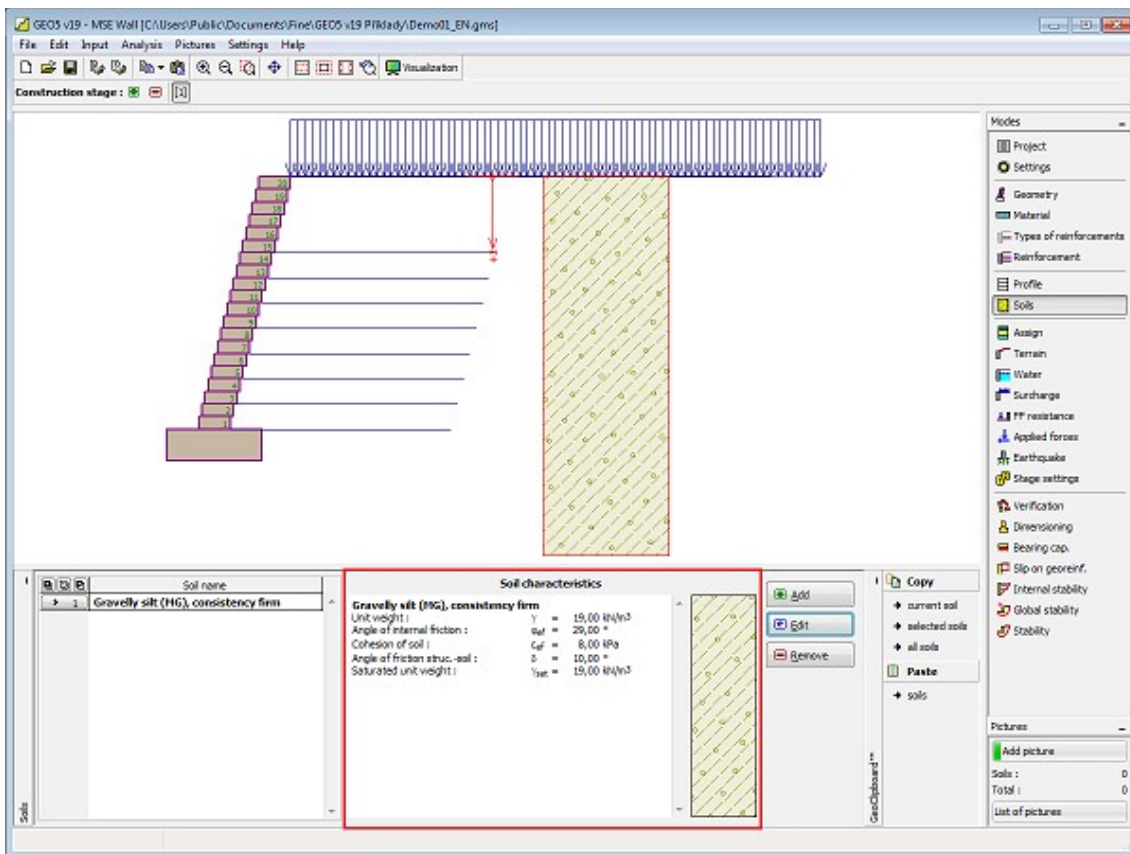
## Soils

The **"Soils"** frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the **"Add new soils"** dialog window.

The soil characteristics needed in the program are further specified in the following chapters: **"Basic data"** and **"Uplift pressure"**.

The program makes it possible to **import soils in the gINT format**. Data of input soils can be copied within all GEO5 programs using **"GeoClipboard"**.



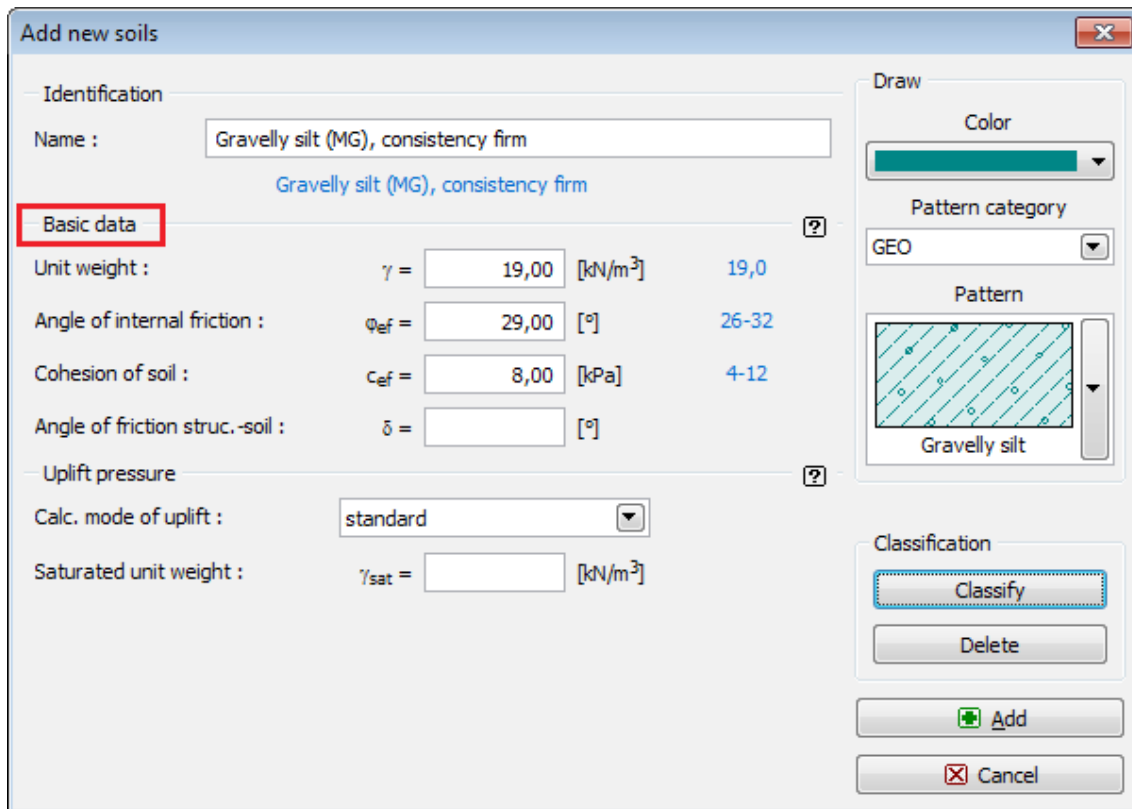
Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in [database of soils](#), which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

One further needs to specify the [angle of internal friction between the soil and structure](#), which depends on the structure material and the type of soil. Possible values of this parameter are listed in the [table of recommended values](#).

The associated theory is described in detail in chapter "[Earth pressures](#)".



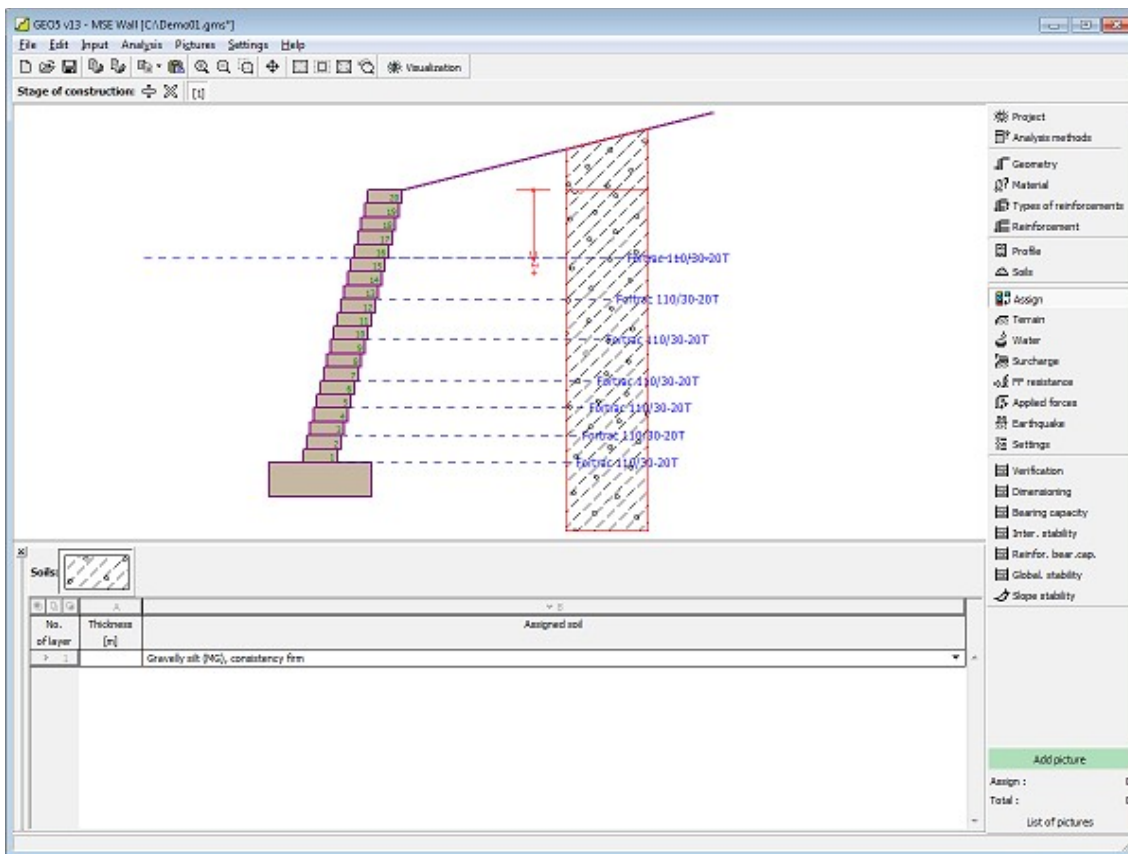
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



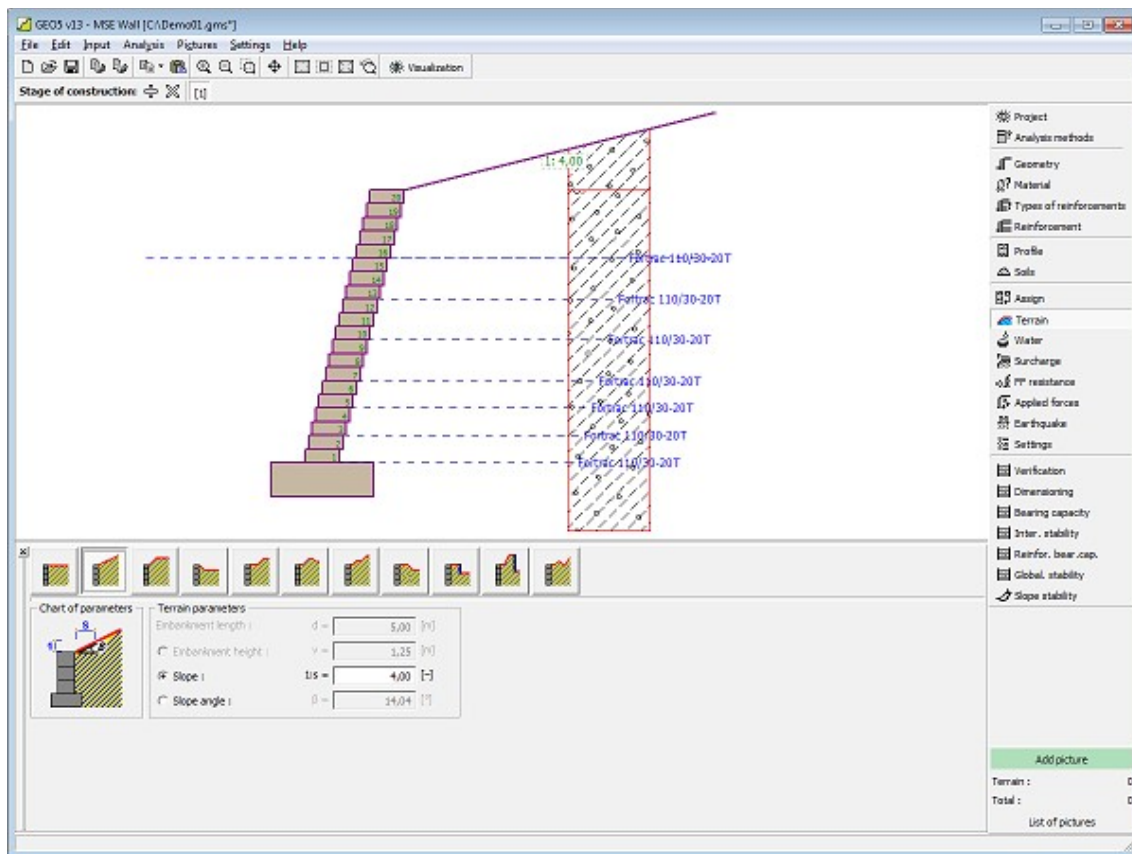
Frame "Assign"

## Terrain

The "Terrain" frame allows, by pressing the button, for specifying the terrain shape. The selected shape with graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The last option to choose from is a general shape of a terrain. In this case the frame contains a table with a list of terrain points. The first point with coordinates [0, 0] coincides with the top point of a structure.

Analysis of earth pressures in case of inclined terrain is described in the theoretical part of the help, chapter "**Distribution of earth pressures for broken terrain**".



Frame "Terrain"

## Water

The "**Water**" frame allows, by pressing the button, for selecting the type of water. The selected type together with a graphic hint ("**Chart of parameters**") of input values is displayed in the left part of the frame. Water parameters ( $h_1, h_2...$ ) can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The combo list serves to specify whether the influence of uplift pressure of water due to different tables at the foundation joint is considered. The uplift pressure can be assumed to be **linear, parabolic** or it may not be considered at all. When verifying the wall, the uplift pressure in base of footing joint due to different water tables is introduced in terms of a special force.

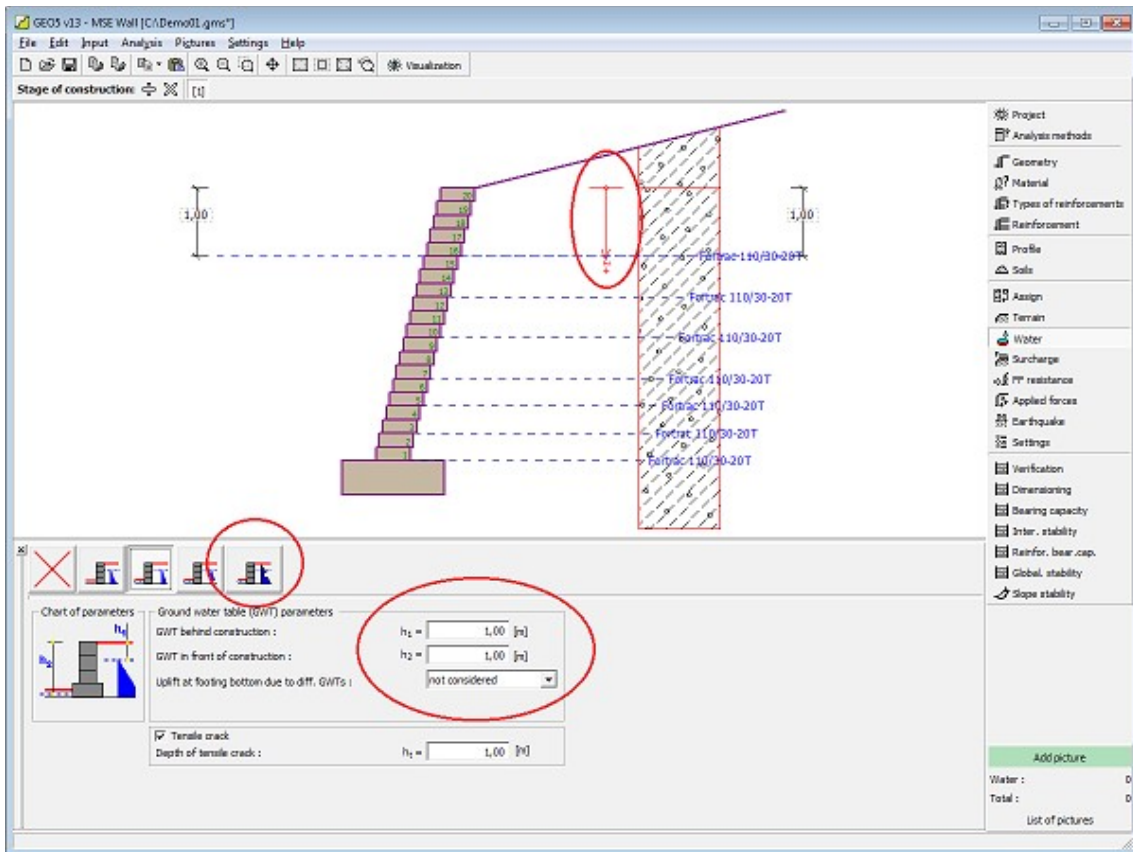
The last option is a manual input of pore pressure both in front and behind the structure. Two tabs "**In front of structure**" and "**Behind structure**" appear with **tables**. The table is filled with values of pore pressure in front, or behind the structure at a depth of "z" (z-axis).

The ground water table can also be specified **above the structure** or earth profile, respectively - in such a case the depth of water is input with a negative value.

Analysis of earth pressures with influence of water is described in the theoretical part of the help, chapter "**Influence of water**".

The program further allows for specifying a depth of **tensile cracks** filled with water.





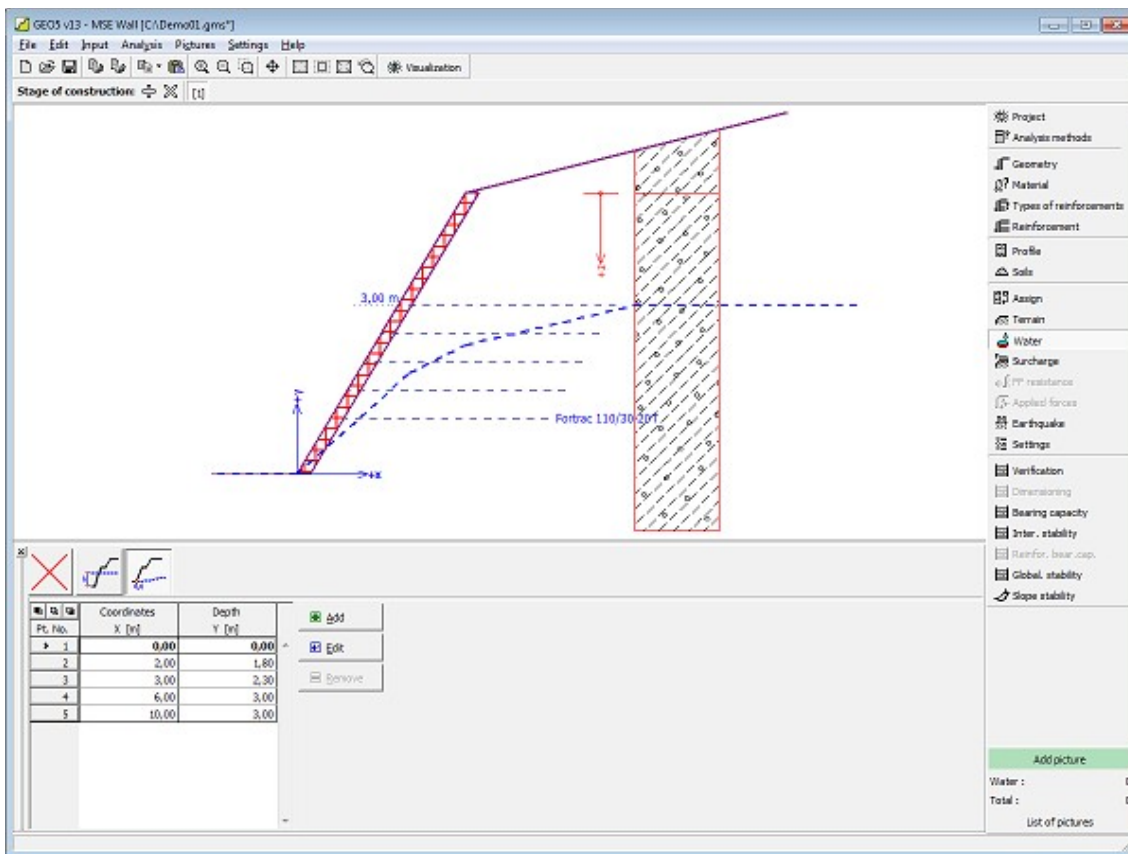
Frame "Water"

## Water

The **"Water"** frame allows for selecting the type of water. The ground water table can be specified in two ways. The first option allows for specifying the height of a flat ground water table. The second option enables to **define** an arbitrary shape of the ground water table with the help of coordinates.

The water parameters (water table height, coordinates of points) can be edited either in the frame by inserting values into the input fields or on the desktop using the **active dimensions** or active blocks.





Frame "Water"

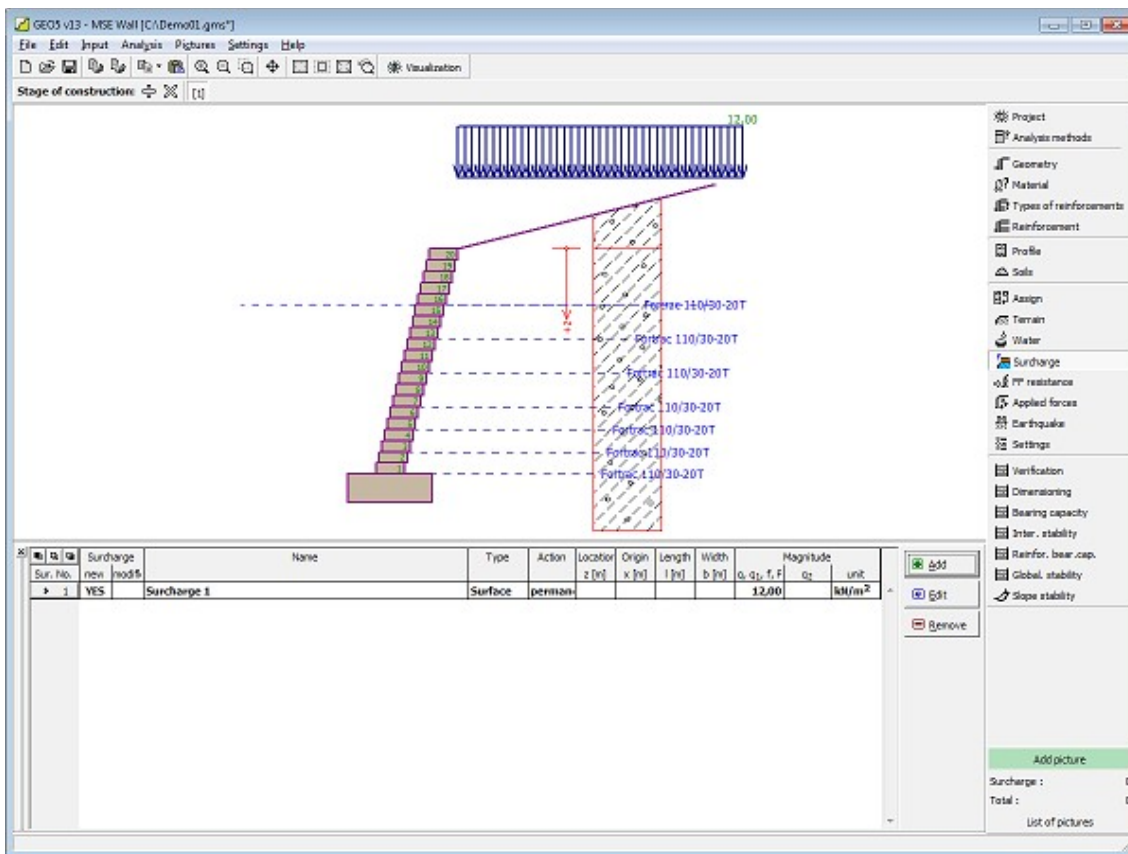
## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active dimensions or by active objects.

The z-coordinate measured from the top point of a structure is specified (positive direction downwards) when inputting the surcharge at a certain depth. In case when the surcharge is found out off the terrain the program prompts an error message before calculation.

Either **permanent**, **variable** or **accidental** surcharge can be specified. Selecting the particular type of surcharge also renders the corresponding design coefficient to multiply the resulting load action. Accidental surcharge with favorable effect is not considered in the analysis.

Analysis of earth pressures due to surcharges is described in the theoretical part of the help, chapter "**Influence of surcharge**".



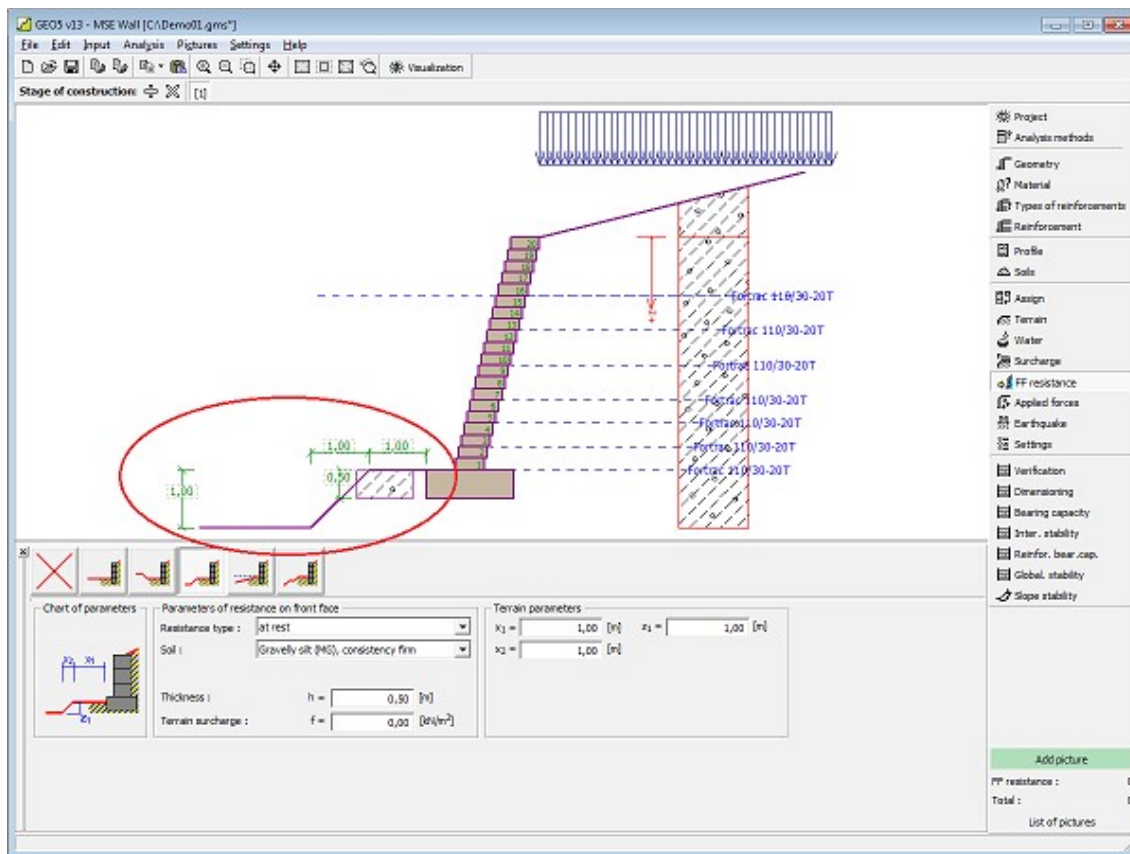
Frame "Surcharge"

## Front Face Resistance

The "Front face resistance" frame allows by pressing the button for specifying the terrain shape and parameters of front face resistance. The selected shape with a graphic hint ("Chart of parameters") of input values is displayed in the left part of the frame. The terrain shape can be edited either in the frame by inserting values into input fields, or on the desktop with the help of active dimensions.

Combo lists in the frame allows the user to select the type of resistance and a soil (the combo list contains soils introduced in the frame "Soils"). The magnitude of terrain surcharge in front of the wall or soil thickness above the wall lowest points can also be specified in the frame.

The resistance on a structure front face can be specified as a pressure at rest, passive pressure or reduced passive earth pressure. The resulting force due to reduced passive pressure is found as a resultant force caused by passive pressure multiplied by a corresponding coefficient, which follows from the input type of reduced passive pressure.



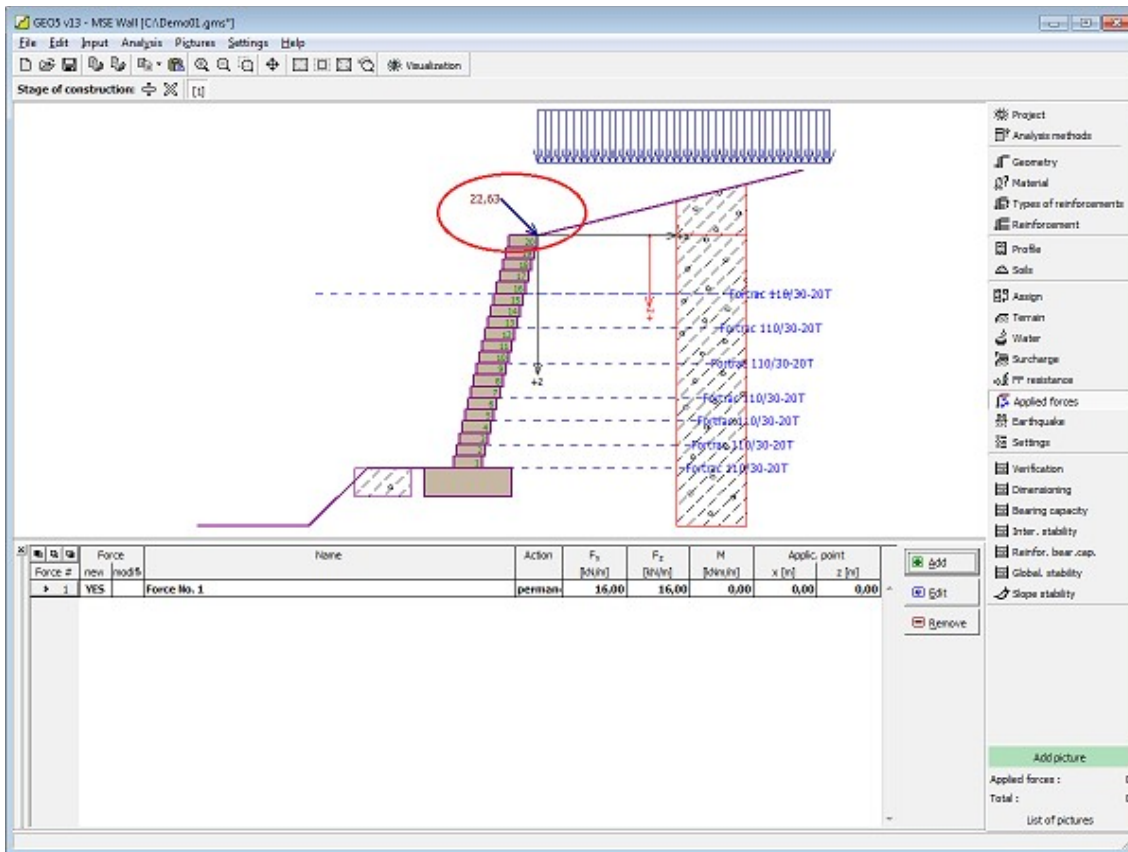
Frame "Front face resistance"

## Applied Forces

The "**Applied forces**" frame contains a table with a list of forces acting on a structure. Adding (editing) forces is performed in the "**New force (edit force)**" dialog window. The input forces can also be edited on the desktop with the help of active objects.

**Applied forces** represent an additional load on the structure of the wall, sheeting or MSE wall. We can model such as an anchoring crash barrier, crash vehicle, load from billboards and hoardings etc. Program doesn't adjust the applied forces in the calculation.

External load acting to the ground surface is necessary to define as surcharge.



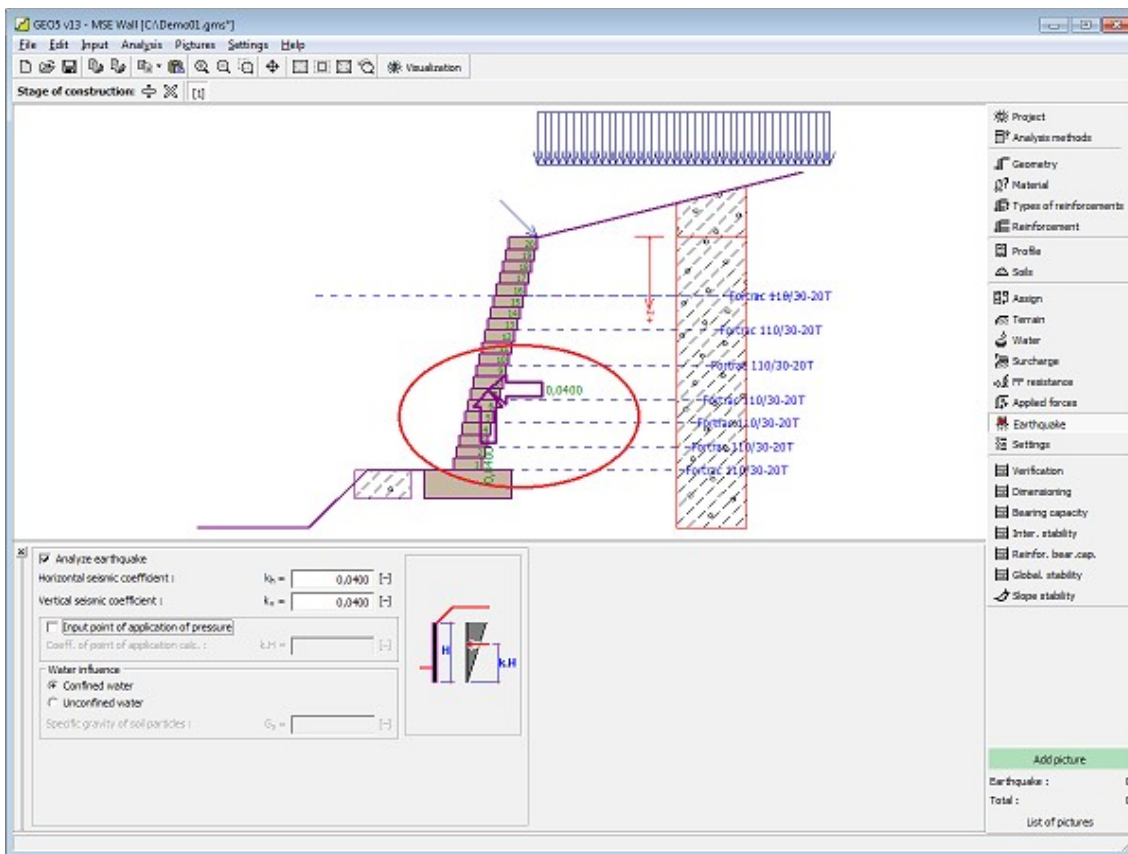
Frame "Applied forces"

## Earthquake

The "Earthquake" frame serves to input earthquake parameters. Directions of input earthquake effects are displayed on the desktop.

If not provided by measurements the coefficients  $k_h$  and  $k_v$  can be calculated following the approach adopted from EN 1998-5.

Analysis of earth pressures while accounting for earthquake is described in the theoretical part of the help, chapter "Influence earthquake".



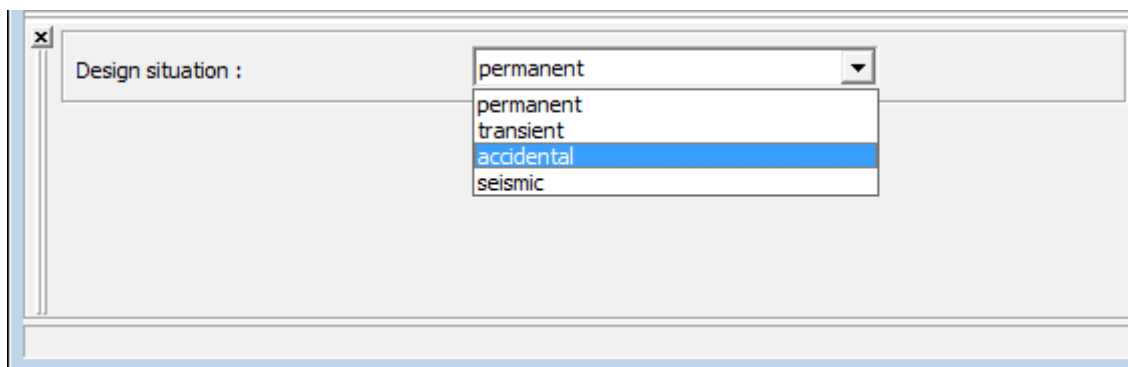
Frame "Earthquake"

## Stage Settings

The **frame "Stage settings"** serves to input settings valid for a given construction stage.

Selected **design situation** determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected **verification methodology**. LRFD 2012 introduces new types of **design situations (Strength I, Service I, Extreme I)**.



Frame "Stage settings"

## Verification

The frame "Verification" shows the analysis results.

To perform verification of external stability the program creates a **fictitious structure** (wall) which is then checked for **overturning and slip**. The fictitious wall consists of the structure front face and a curve bounding the end points of geo-reinforcements. The fictitious structure is loaded by an **active earth pressure**. The **actual verification procedure** is described in the theoretical part of the help.

The frame appearance is adjusted based on the selected **verification methodology**:

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces. These forces are displayed on the desktop and are updated for every change of data and setting in the frame.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations".
- **Analysis according to LRFD** - in the case the last column is not displayed.

**Several computations** can be carried out for a single task. The computed forces are displayed on the desktop and are automatically updated with every change of input data and setting. The right part of the frame shows the result of verification of a wall against **overturning and slip**. The "In detail" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Visualization settings".

No. of force	Force	Fx [kN]	Fz [kN]	Application point x [m]	z [m]	Minor load
1	Gravity force	0,00	340,42	3,40	-2,07	
2	Active pressure	-37,20	18,94	1,55	-0,80	
3	Weight - wall	0,00	46,00	1,22	-2,50	

Verification	Result	Percentage
OVERTURNING :	SATISFACTORY	(4,5%)
SLIP :	SATISFACTORY	(23,0%)



*Frame "Verification"*

## Dimensioning

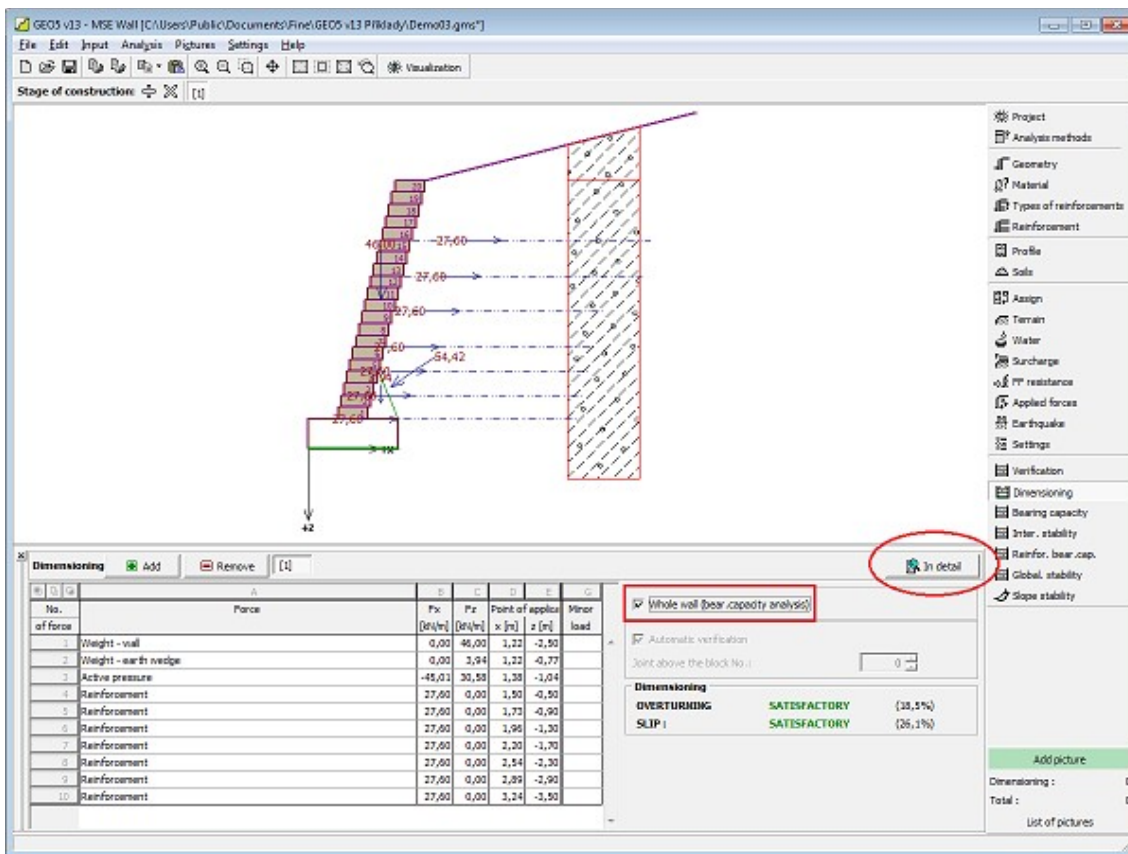
The "**Dimensioning**" frame enables to **verify** individual joints between blocks for **overturning** and **slip**. The option "**Entire wall**" allows for verifying the overall structure above the foundation joint as well as the foundation soil bearing capacity in the "**Bearing capacity**" frame. Checking the option "**Automatic verification**" provides verification of the most critical joint above the block. Or it is possible to input the "**Joint above block number**" to prompt the program to perform the analysis for a given joint only. The procedure for **wall dimensioning** is described in the theoretical part of the help.

The frame appearance is adjusted based on the selected **verification methodology**:

- Verification according to the **factor of safety**, or the theory of **limit states** - the last column in the table allows for inputting the **design coefficients**, which multiply the calculated forces.
- **Analysis according to EN 1997** - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "**Load combinations**".
- **Analysis according to LRFD** - in the case the last column is not displayed.

The frame enables to perform more analyses of individual joints of the wall blocks. Various **design coefficients** of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Dimensioning"

## Bearing Capacity

The "Bearing capacity" frame displays the results of the analysis of foundation soil bearing capacity. Stress in the foundation joint (assumed constant) is determined from all forces calculated in the "Verification" frame. In case of the input foundation, the bearing capacity is determined from all forces calculated in the "Dimensioning" frame (the option "Entire wall" must be selected). The program "Spread footing" adopts individual verifications as load cases.

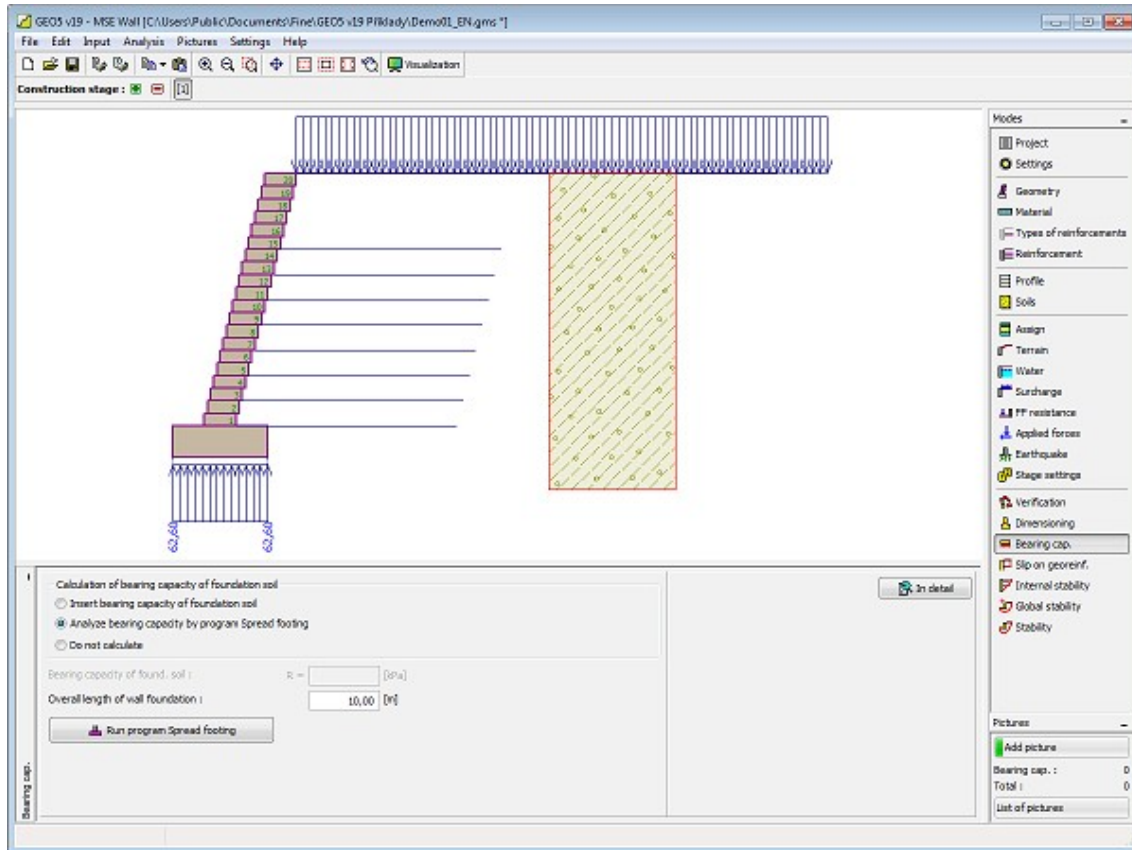
Three basic analysis options are available in the frame:

- Insert bearing capacity of foundation soil** The foundation soil bearing capacity is input. The **eccentricity** and **bearing capacity** analysis results are displayed in the right part of the frame. The "In detail" button opens a dialog window that displays detailed listing of the results.
- Analyse bearing capacity by program "Spread footing"** Pressing the "Run program Spread footing" button opens the program "Spread footing" which allows to calculate the soil bearing capacity or settlement and rotation of the footing. Pressing the "OK" button leaves the analysis mode - the results and all plots are transferred into the program "MSE Wall". The "Spread footing" program must be installed for the button to be active. Overall length of wall foundation is input.



- **Do not calculate (pile footing)** The foundation soil bearing capacity is not calculated.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Bearing capacity"

## Slip on Georeinforcement

The "Slip on georeinforcement" frame enables to verify a slip of the reinforced soil block along a geo-reinforcement checking the field "Reinforcement number". Selecting the option "Automatic verification" provides verification of the most critical reinforcement. The reinforced soil block is bounded by the wall front face, the checked geo-reinforcement, a vertical line passing through the geo-reinforcement end point and terrain. The reinforced soil block is loaded by an active earth pressure and by stabilizing forces due to geo-reinforcements exceeding the boundary of the reinforced block.

The solution procedure of slip on georeinforcement is described in the theoretical part of the help.

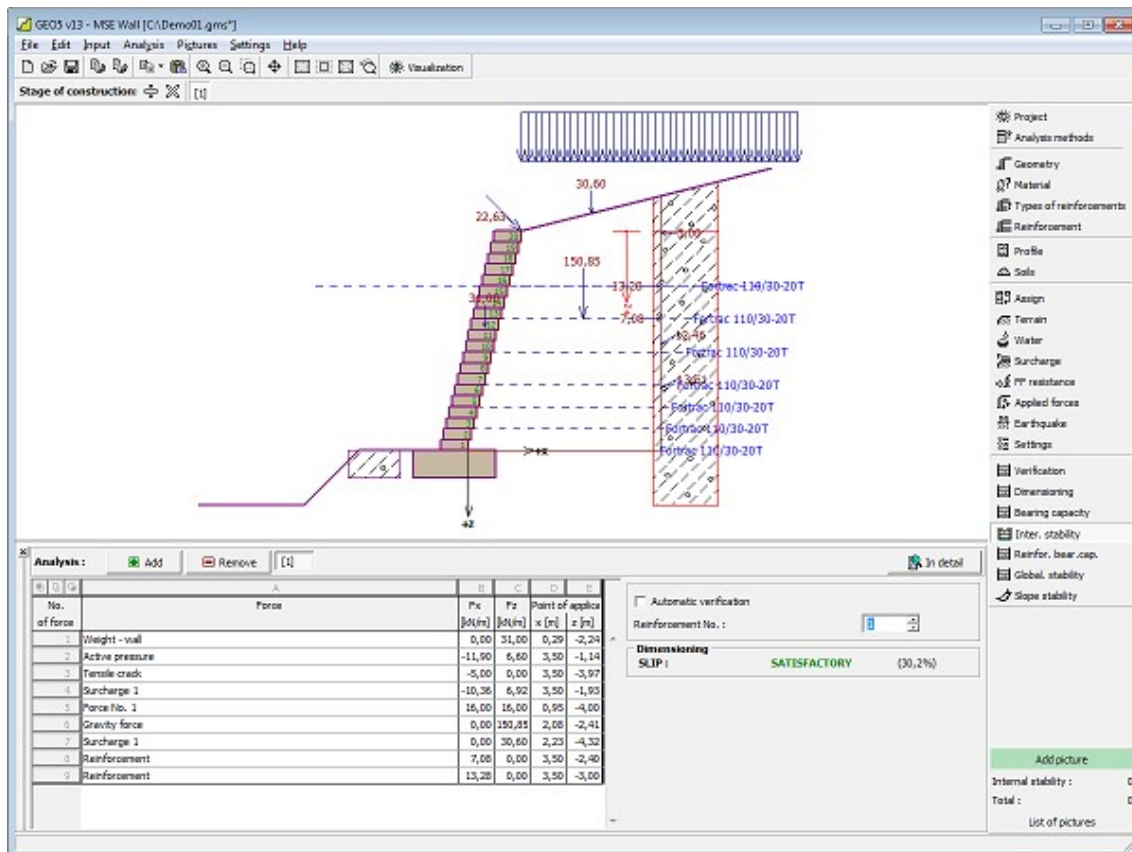
The frame appearance is adjusted based on the selected verification methodology:

- Verification according to the factor of safety, or the theory of limit states - the last column in the table allows for inputting the design coefficients, which multiply the calculated forces.
- Analysis according to EN 1997 - the last column in the table allows for specifying whether the load acting on a structure is considered as secondary one. This is explained in more detail in section "Load combinations".

- Analysis according to LRFD - in the case the last column is not displayed.

The frame enables to perform more verification analyses of individual geo-reinforcements. Various design coefficients of individual forces can also be specified. The resulting forces are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "In detail" button opens the dialog window, which contains a detailed listing of the results of internal stability.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Internal stability"

## Internal Stability

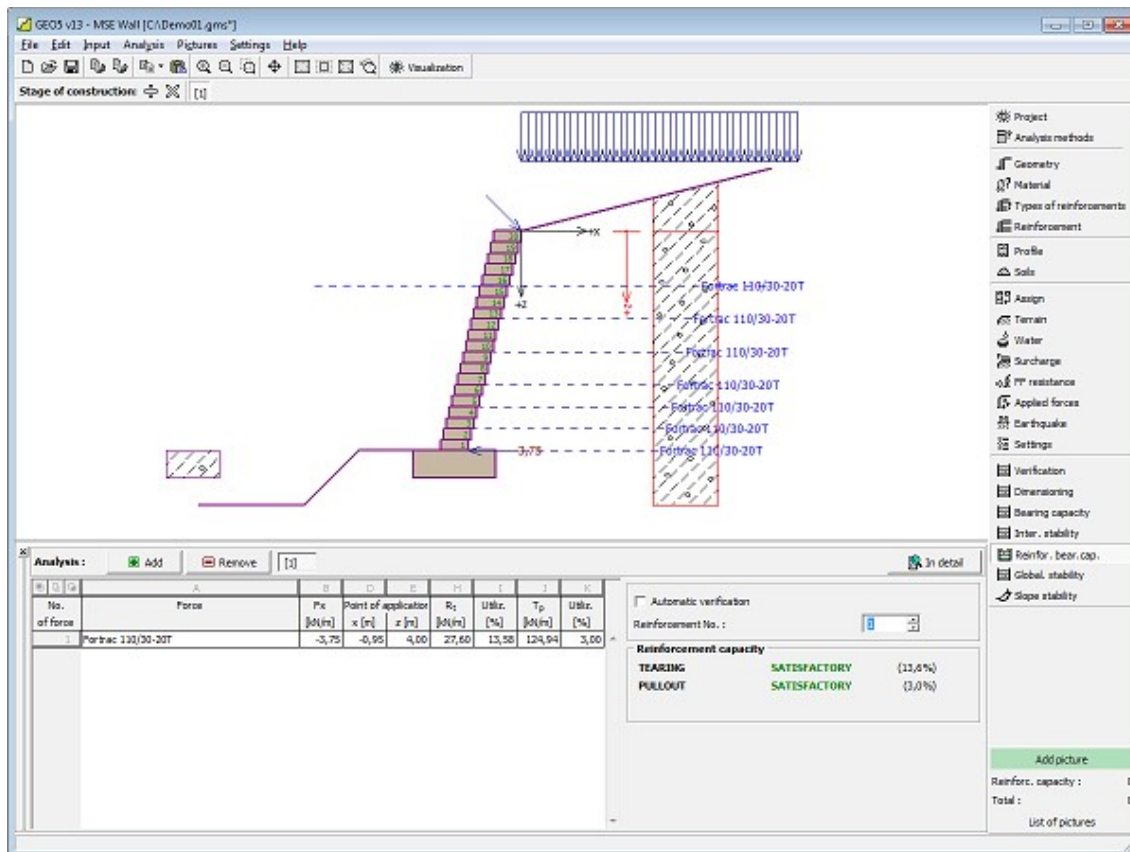
The "Internal stability" frame enables to check a strength of geo-reinforcement, bearing capacity for pull-out from the earth body and strength of connections. Checking the field "Reinforcement number" yields verification for individual reinforcements only. Selecting the option "Automatic verification" provides verification of all reinforcements. The result for the most critical reinforcement is displayed on the right part of the desktop. The solution procedure of internal stability is described in the theoretical part of the help.

The B column shows forces caused by an active earth pressure acting on the front face of the wall in individual geo-reinforcements. The D, E columns store points of application of these forces. The H, J columns represent bearing capacity of geo-reinforcements against tearing and the resulting utilization for tearing. The J, K columns represent bearing capacity of geo-reinforcements against pull-out from the earth body and the resulting utilization of geo-reinforcement for pull-out.

The frame enables to perform more analyses of individual geo-reinforcements. The calculated

forces are displayed on the desktop are automatically updated with every change of input data. The **"In detail"** button opens the dialog window, which contains a detailed listing of the results of reinforcement bearing capacity.

Visualization of results can be adjusted in the frame **"Visualization settings"**.



Frame "Reinforcement bearing capacity"

## Global Stability

The **"Global stability"** frame enables to perform the **slope stability** analysis along a circular slip surface. It is required to input parameters of the slip surface (center and radius or 3 points input) and the analysis method (**Spencer**, **Bishop**).

By pressing the **"Substitute"** button, it is possible to input points of the slip surface using mouse on the desktop.

If the **"Optimize"** option is checked, the stability analysis is performed on the most critical slip surface. The program allows to "keep" the end points of the slip surface (by checking the **"Keep the left end point of the slip surface"** or **"Keep the right end point of the slip surface"** option).

The **"Initial slip surface"** enables to input the circular slip surface automatically. The analysis is then performed after the **"Analyze"** button is pressed.

The **actual slope stability verification analysis** is carried out depending on the setting in the **"Stability analysis"** tab:

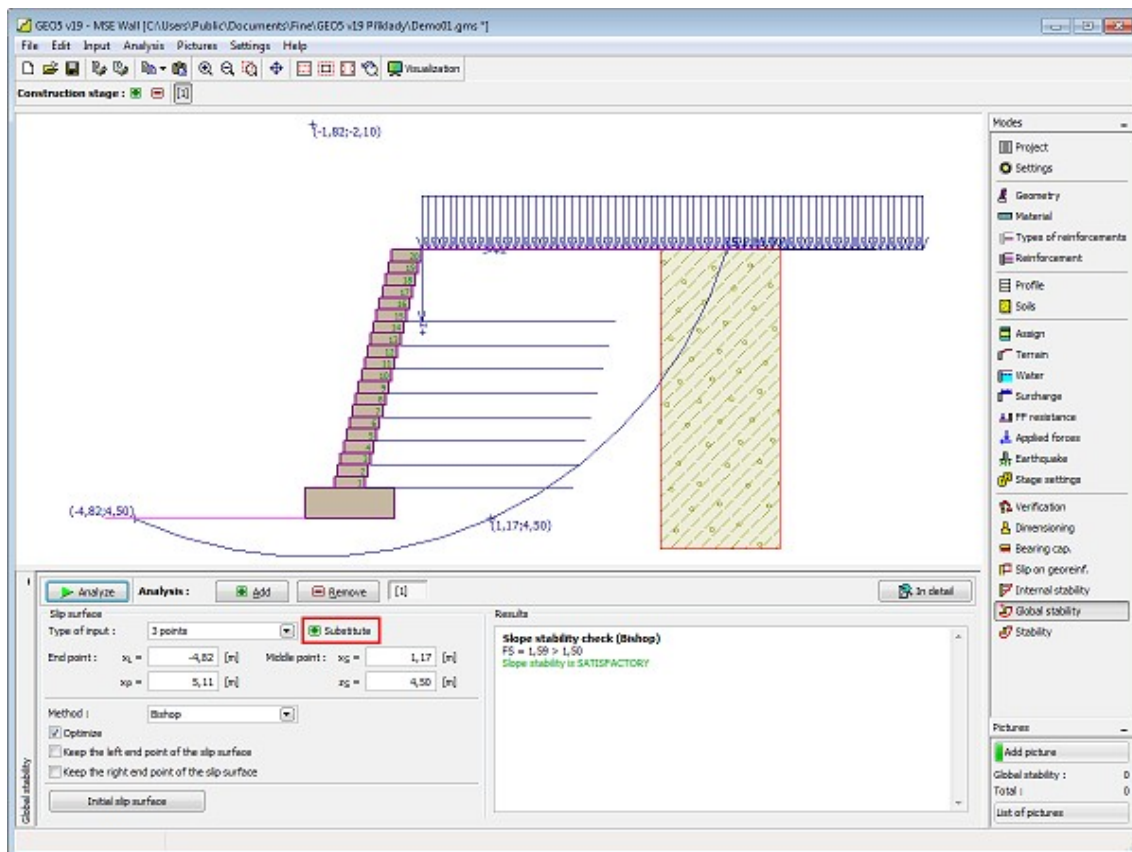
- According to EN 1997, where **load is reduced** by the partial factors of analysis and the verification is performed based on the theory of limit states.

- According to **LRFD**, the analysis is carried out similarly to the theory of limit states
- According to the **safety factor** / the theory of **limit states** depending on the setting in "Wall analysis" tab.

More analyses can be performed for a single task. The **"In detail"** button opens the dialog window, which contains a detailed listing of the results of stability analysis, i.e. parameters of the resulting slip surface and the factor of safety, alternatively utilization (for limit states).

The results are displayed in the right part of the frame, the optimized slip surface on the desktop.

Visualization of results can be adjusted in the frame **"Visualization settings"**.

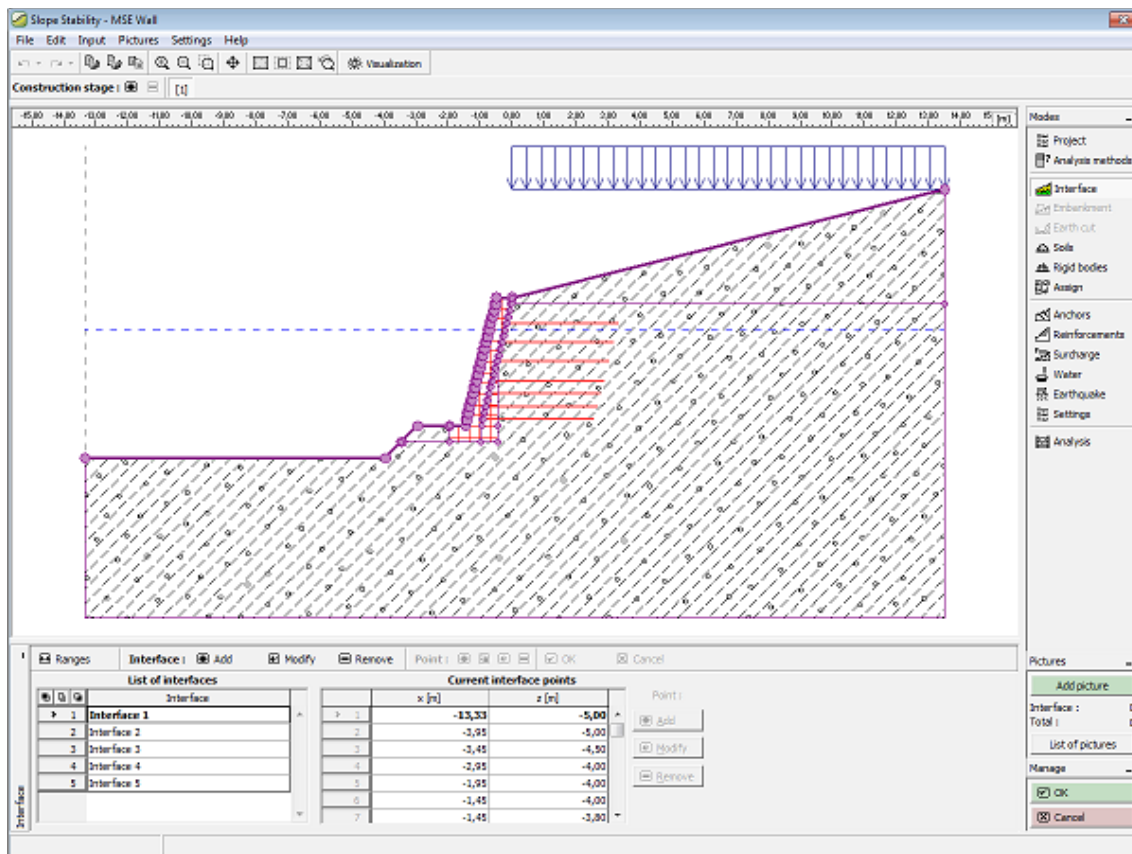


Frame "Global stability"

## Stability

Pressing the **"Stability"** button launches the **"Slope stability"** program. This program then allows us to check the overall stability of the analyzed structure. The button is available only if the program **"Slope stability"** is installed.

After completing all analyses press the **"OK"** button to leave the program - all data are then carried over to the analysis protocol of the **"MSE Wall"** program.



Frame "Stability"

## Program Spread Footing

This program is used for the design of spread footings subject to general load. It computes vertical and horizontal bearing capacity, settlement and rotation of a footing, and determines required longitudinal and shear reinforcement (punching).

The help in the program "Spread Footing" includes the following topics:

- Input of data into individual frames:

Project	Settings	Profile	Dilatometric Soils Tests (DMT)	Assign	Foundation
Load	Geometry	Footing Bottom	Sand-Gravel Cushion	Surcharge	Water, Incompressible Subsoil
Stage Settings	Bearing Capacity	Settlement and Rotation	Dimensionin		

- Standards and analysis methods



- Theory for analysis in the program "**Spread Footing**":
 

Stress in Soil Body	Parametry hornin	Analysis of Foundation Bearing Capacity	Settlement Analysis	Dimensioning of Concrete Structures
------------------------	------------------	---	------------------------	---
- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The "**Select**" button allows to choose an already created setting from the "**Settings list**".

The "**Settings Administrator**" button opens the "**Administrator**" dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

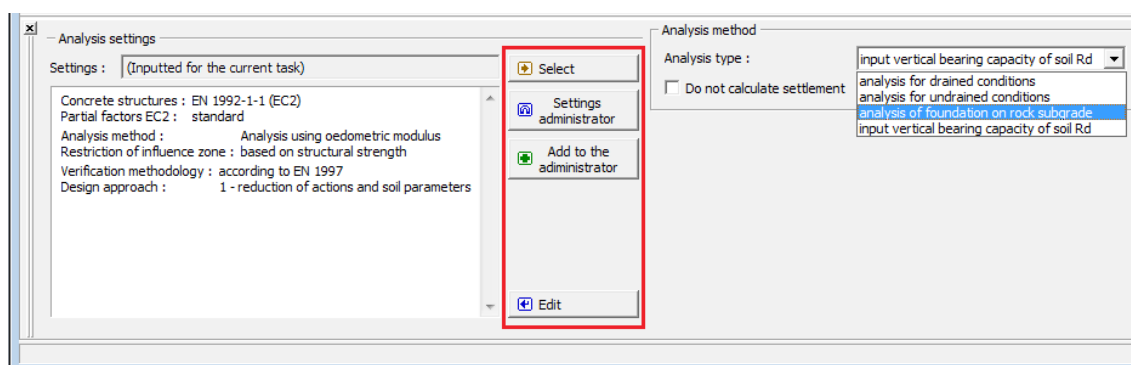
The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Materials and standards**", "**Settlement**" and "**Spread Footing**" tabs.

Four options are available to calculate the vertical bearing capacity of a spread footing:

- **analysis for drained conditions**
- **analysis for undrained conditions**
- **analysis of foundation on rock subgrade**
- **input vertical bearing capacity of soil  $R_d$**



Frame "Settings"

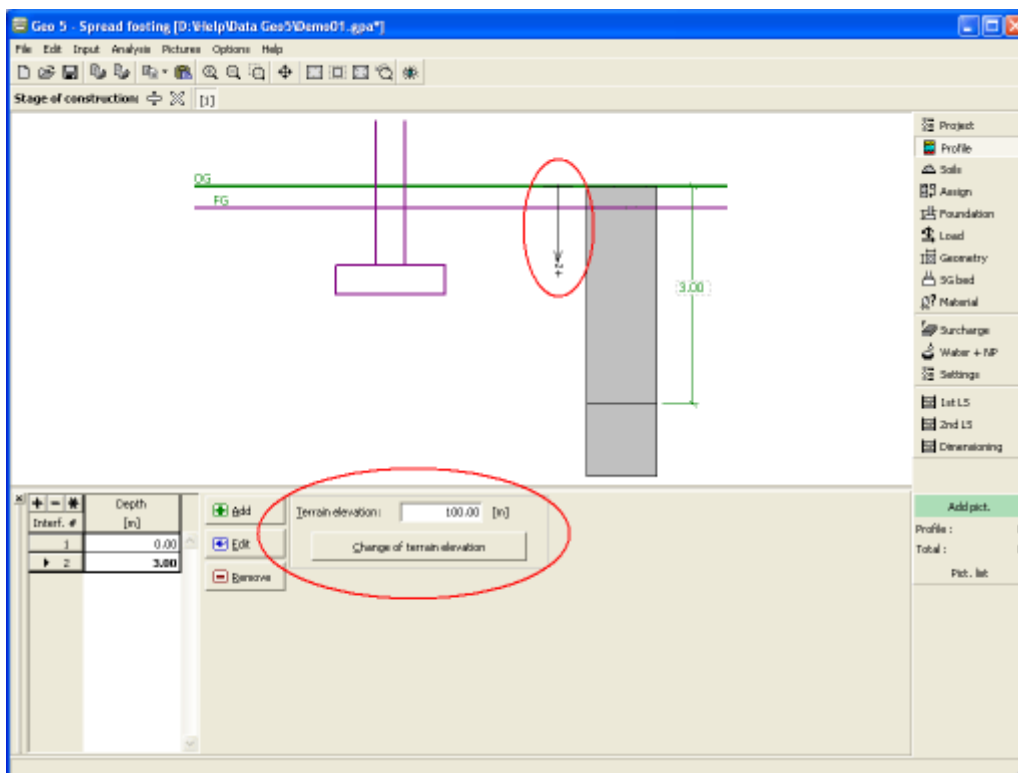
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of **active dimensions**.

**Adding** (editing) layer is performed in the "**New interface**" dialog window. The  $z$ -coordinate measured from the top point of a structure is specified ( $z$ -axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

## Dilatometric Tests (DMT)

The frame "DMT" serves to input the way of introducing of **constrained soil modulus** into the program - either as a parameter of **soil** (by checking the option "**Input  $M_{DMT}$  as a soil parameter**"), or by importing of a **dilatometric test (DMT)**.

The results of **dilatometric test (DMT)** are imported into the program by inserting the file in format **UNI (\*.uni)**. It's a **standardized and universal format** for import of the measured data obtained from **dilatometric tests**, which is used in the world.

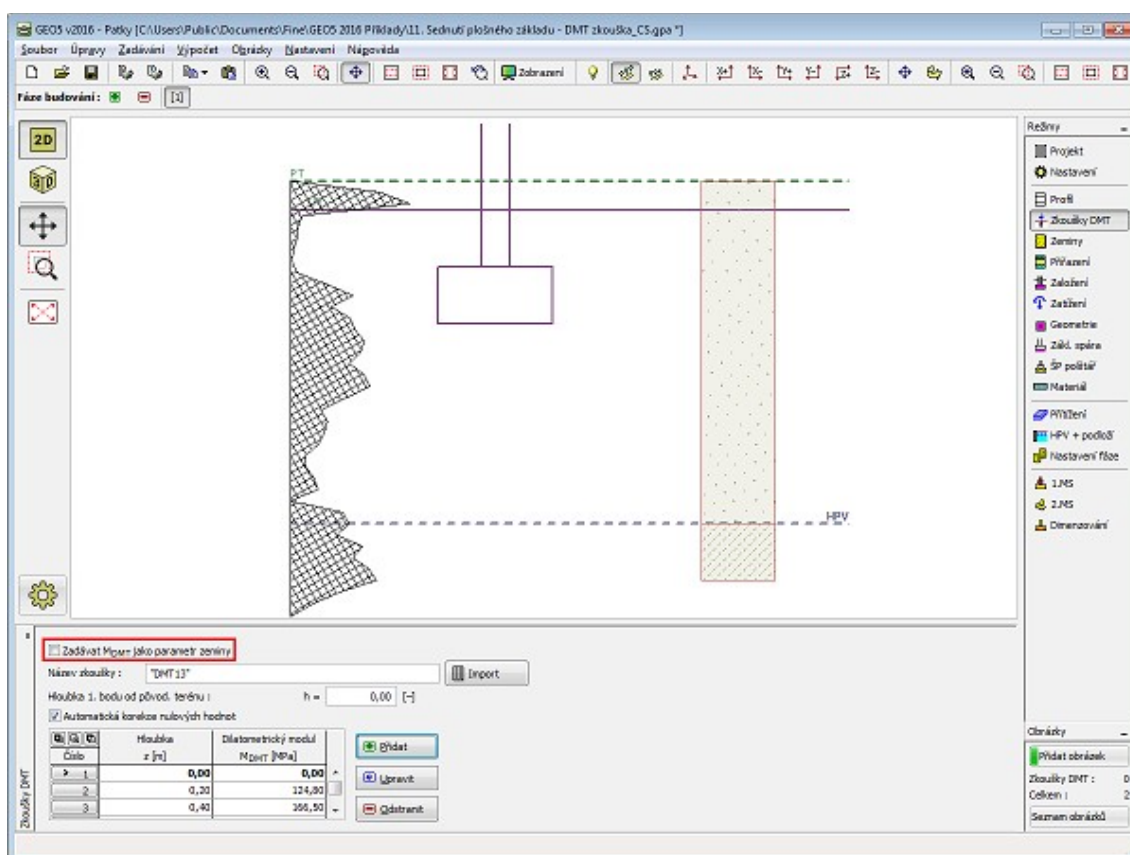
Then in the frame it's possible to edit the name of **dilatometric test** and the depth of the first point of DMT  $h$  [m] from the finished grade.

This frame contains a **table** with list of the input values of dilatometric test (DMT). Adding (editing) of values is performed in the "**New value of test, or Edit the value of test**" dialog **window**, where is specified the depth  $z$  [m] and the measured values of **constrained soil modulus  $M_{DMT}$  [MPa]**.

If during the evaluation of **dilatometric test** is measured the zero value of **constrained soil modulus  $M_{DMT}$** , then program allows the automatic correction of measurement errors - instead of zero value is considered the arithmetic average of the next upper and lower non-zero value of  $M_{DMT}$  in the calculation.

**Note:** The frame is accessible only in the case, when in the "Settings" frame is selected method of calculation with option "**Analysis using constrained modulus**" ("Settlement" tab).





Frame "DMT"

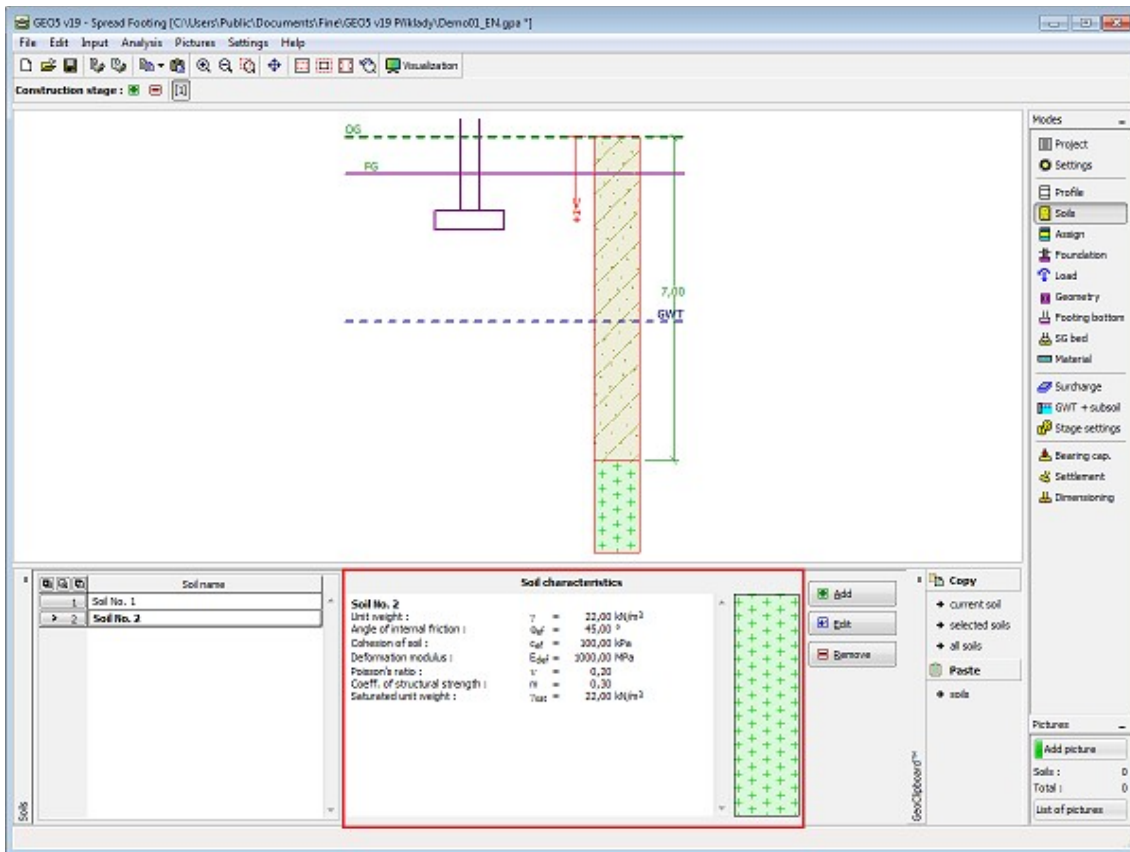
## Soils

The **"Soils"** frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Uplift pressure", "Foundation bearing capacity" and "Settlement".

The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

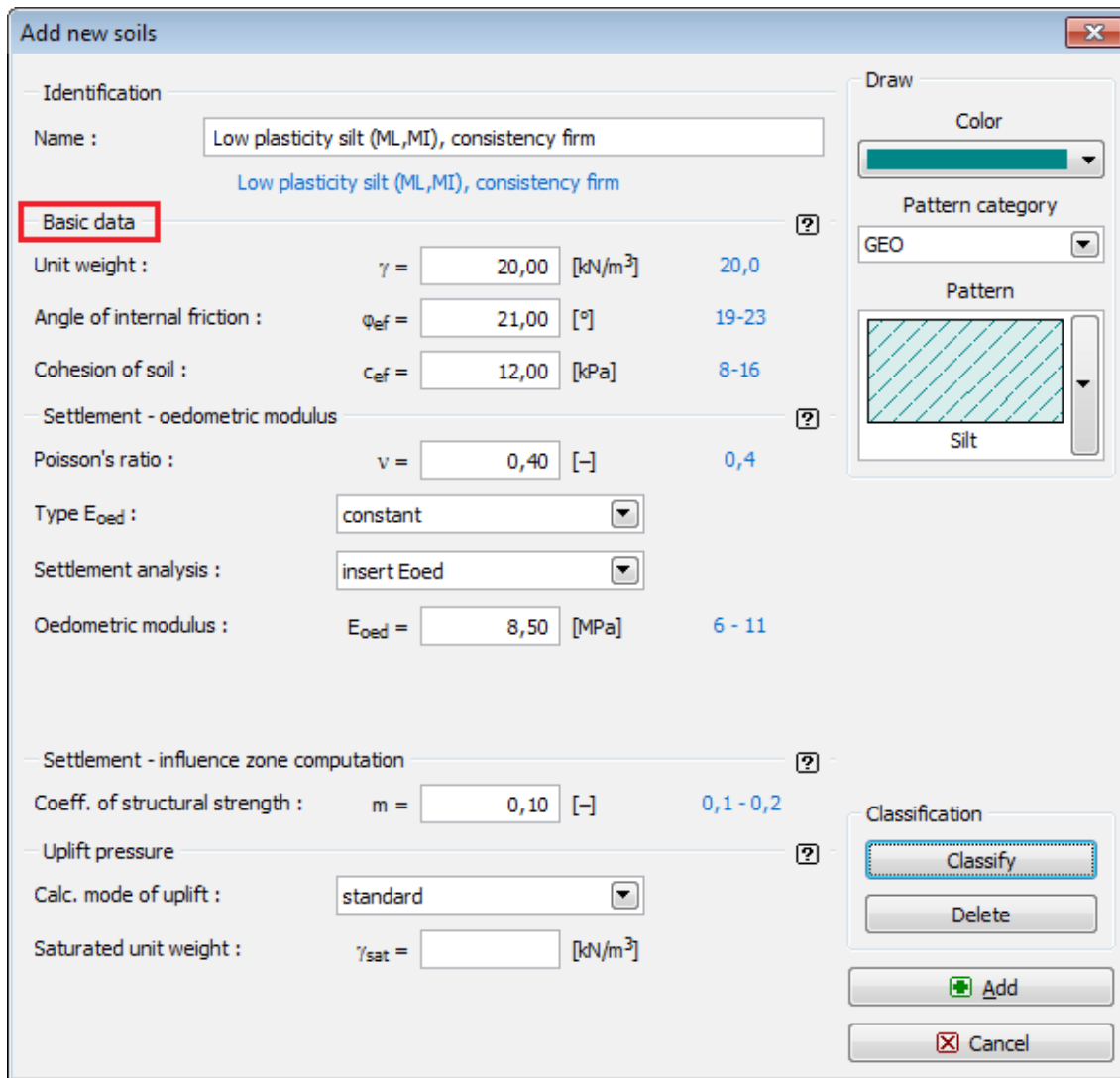
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Soil parameters differ according to the analysis type and analysis method ("**Settings**" frame, "**Spread Footing**" and "**Settlement**" tabs).

The analysis type differs according:

- **analysis for drained conditions:** **effective** parameters of shear strength of soil  $c_{ef}, \phi_{ef}$  are used commonly.
- **analysis for undrained conditions:** vertical bearing capacity of foundation depends on **undrained** shear strength of soil  $c_u$ . **Effective** angle of internal friction  $\phi_{ef}$  is defined only for calculation of **earth pressure** to solve **horizontal bearing capacity of foundation**.
- **analysis of foundation on rock subgrade:** for this analysis method program defines **angle of internal friction of rock  $\phi$ , compressive strength  $\sigma_c$ , coefficient of damage of rock  $D$ , coefficient of structural strength  $m_i$**  and Geological Strength Index.

The associated theory is described in detail in chapter "**Analysis of bearing capacity of foundation**".



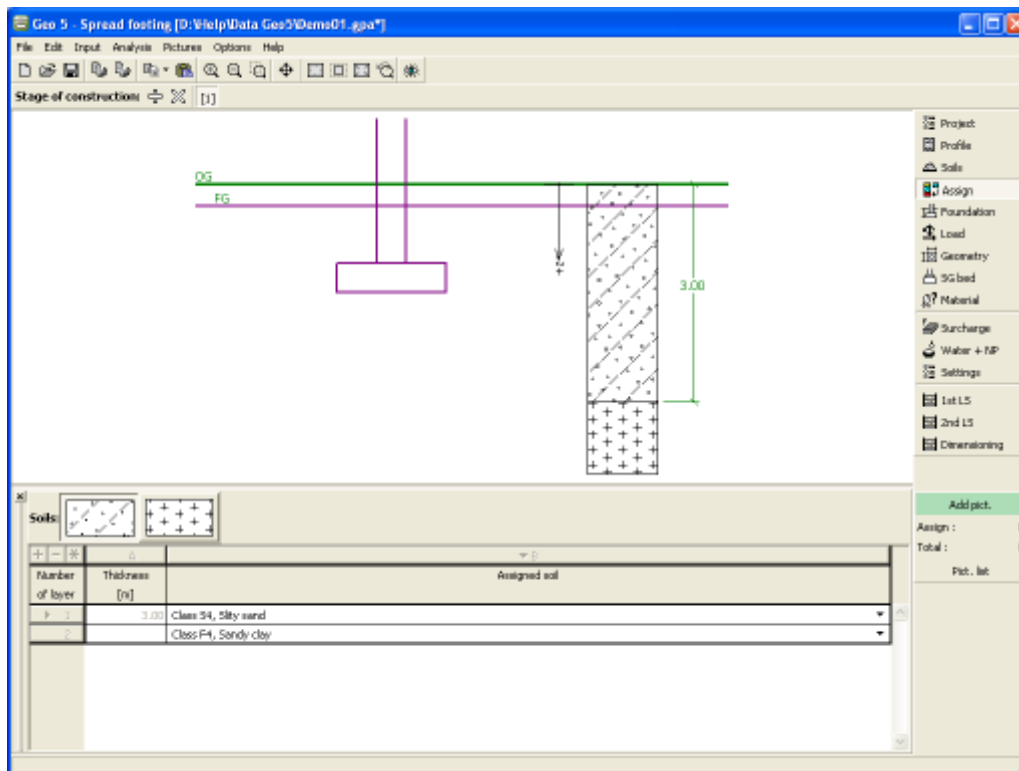
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

## Foundation

The "**Foundation**" frame allows to select a type of foundation. The selected type with graphic hint ("**Geometry scheme**") of input values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**. The frame also serves to specify the unit weight of overburden.

The following types of foundations can be selected:

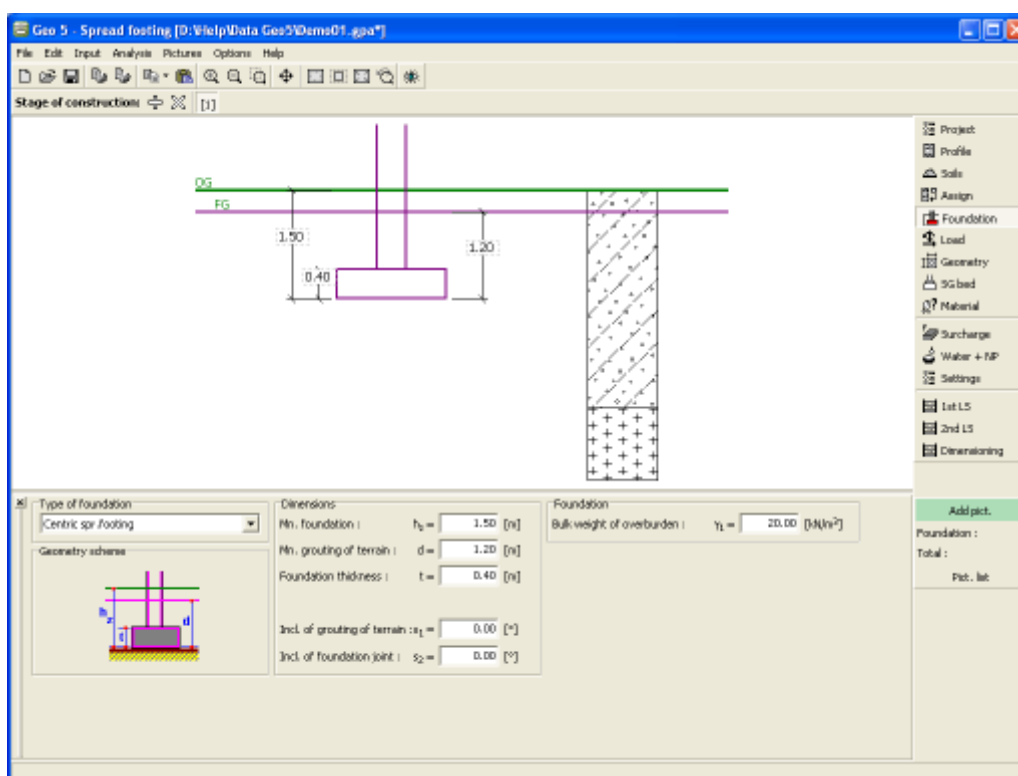
- **Centric spread footing**
- **Eccentric spread footing**
- **Strip footing**
- **Stepped centric spread footing**
- **Stepped eccentric spread footing**
- **Circular spread footing**
- **Circular stepped spread**
- **Centric spread footing with batter**
- **Eccentric spread footing with batter**

The soil profile is specified from the **original ground**. The foundation bearing capacity depends mainly on the depth **of foundation measured from the finished grade**. When the finished grade is found above the original ground then it is required to assign the same depth to both, the finished grade and original ground, and introduce into subsoil a layer with a new made-up-ground. This frame also allows for inputting the **foundation thickness**.

When completed the foundation is usually filled up with a soil - its unit weight must be specified (**Overburden unit weight  $\gamma_1$** ). Providing the analysis follows the theory of **limit states** its weight is multiplied by the coefficient  $\gamma_{my}$  input in the "**Spread Footing**" tab.

For foundations with **drained subsoil** (type of analysis is selected in the frame "**Settings**") it is

possible to introduce an **inclination of the finished grade and footing bottom**. In all other cases both the ground and footing bottom are horizontal.



Frame "Foundation"

## Load

The "**Load**" frame contains a table with a list of input loads. Adding (editing) loads is performed in the "**New (edit) load**" dialog window. Input of individual forces follows the sign convention displayed in the right part of the dialog window.

The following types of load can be specified:

- **design load** serves to verify the foundation bearing capacity
- **service load** serves to compute the foundation settlement and rotation

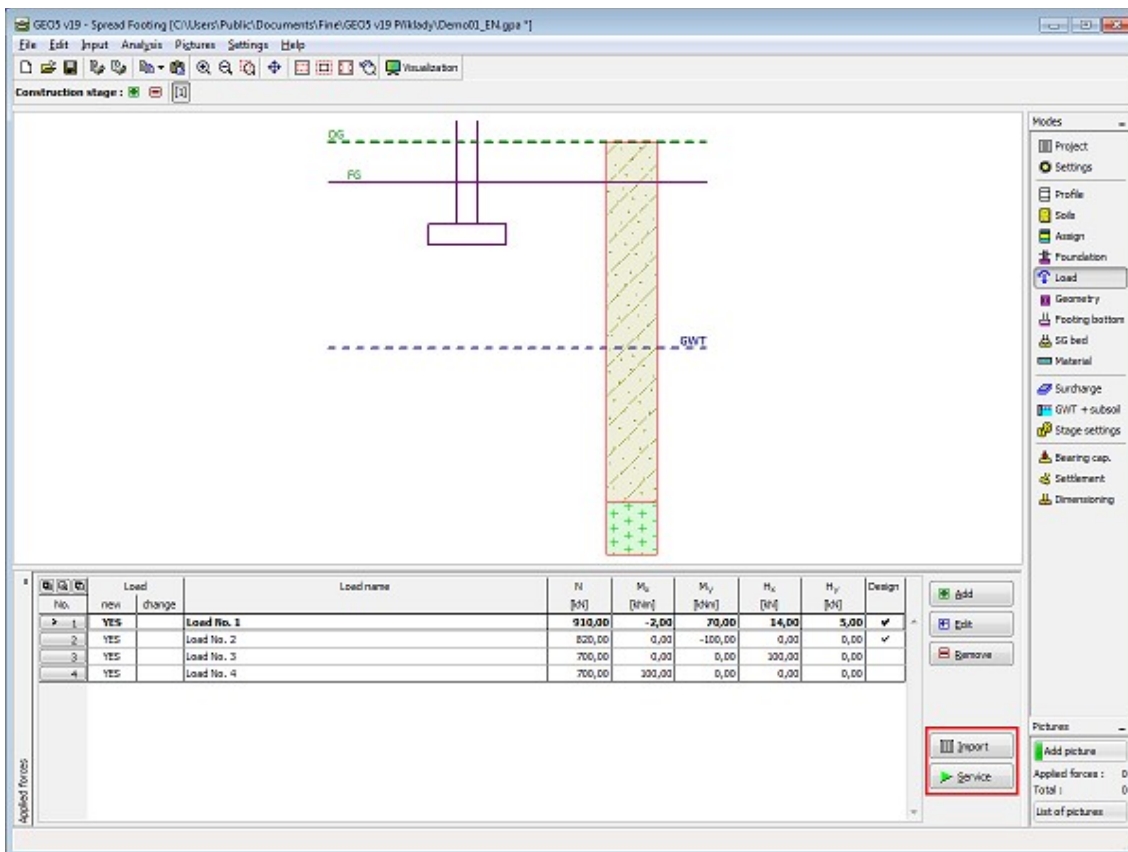
Dimensioning of reinforcements assumed for the foundation is carried out for both types of load.

When performing the analysis according to **EN 1997 or LRFD** (selected in the "**Spread Footing**" tab) it is assumed that the design load is determined in accordance with the corresponding standards and individual components of load are **already pre-multiplied** by corresponding partial factors - the program does not modify the input load **any further**.

The foundation is loaded always at the contact point between column and foundation. The program automatically computes the **foundation self weight** and the **weight of overburden**.

The "**Service**" button allows for creating the service loads from the already input design loads (analysis according to the factor of safety or the theory of limit states).

The program also allows for import of load using the "**Import**" button.



Frame "Load"

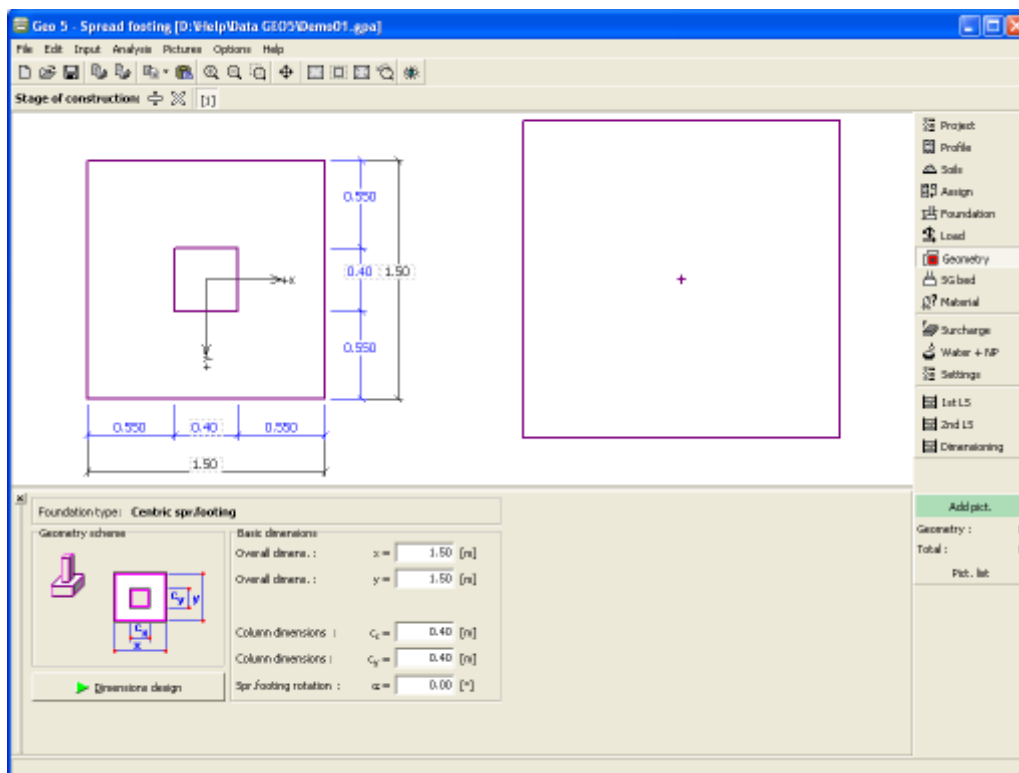
## Geometry

The **"Geometry"** frame allows to specify the foundation shape. The selected shape with graphic hint (**"Geometry scheme"**) of input values is displayed in the left part of the frame. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

**Foundation type** and its **thickness** are specified in the **"Foundation"** frame.

The program automatically computes the **self weight of both foundation and overburden above the foundation**. The foundation self weight is specified in the **"Material"** frame. Providing the analysis is carried out employing the theory of limit states the footing self weight is multiplied by coefficients specified in the **"Spread Footing"** tab.

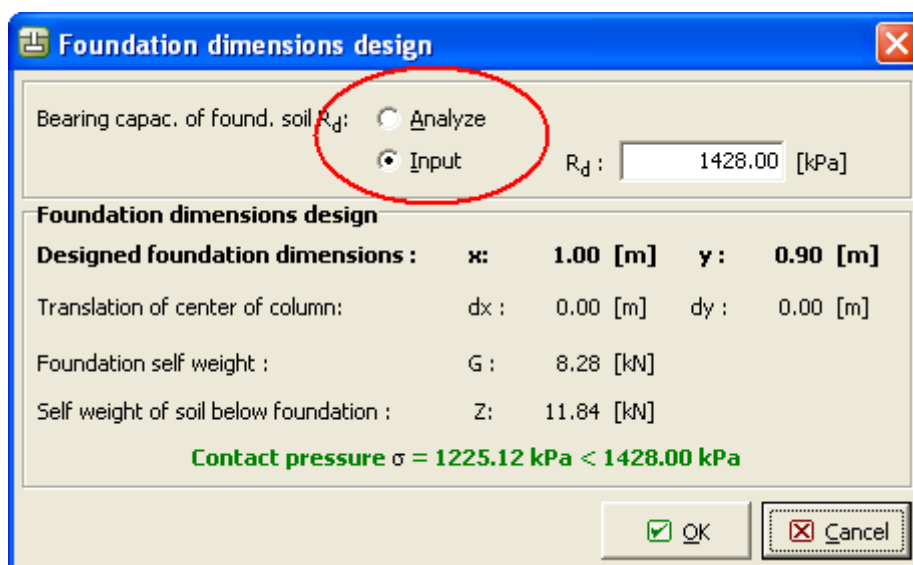
The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Geometry"

The "**Dimensions design**" button opens the "**Foundation dimensions design**" that serves with the help of the program to compute dimensions of a foundation. The dialog window allows for inputting the bearing capacity of foundation soil  $R_d$  or to select the option "**Analyze**". In such a case the program determines all dimensions of a foundation based on **input parameters** (soils, profile, water impact, sand-gravel-cushion, setting, etc.).

While leaving the dialog window by pressing the "**OK**" button the specified dimensions are loaded into the "**Geometry**" frame.



Dialog window "Foundation dimensions design"

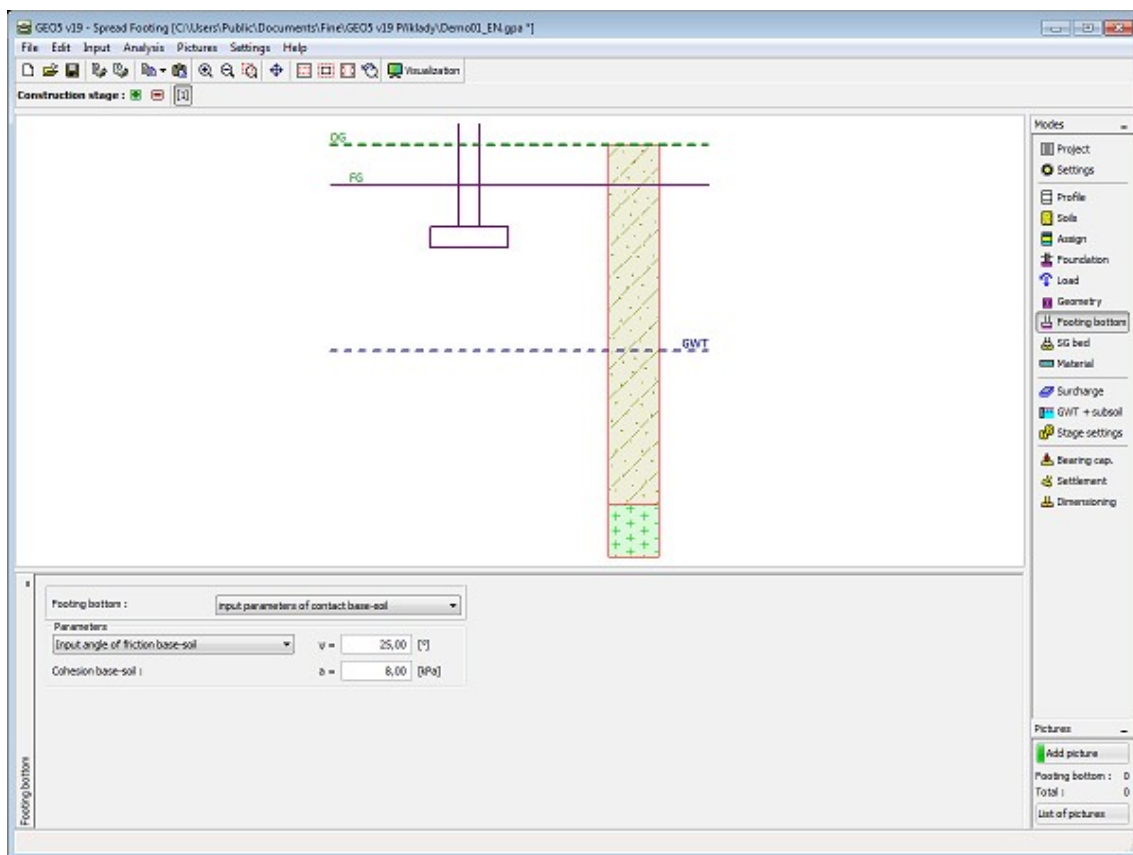


## Footing Bottom

The "**Footing bottom**" frame serves to input characteristics of the action of footing bottom:

- **soil from geological profile** - the spread footing is founded on the soil assigned from the geological profile specified in the "**Profile**" frame.
- **input parameters of contact base-soil** - parameters of the contact between footing bottom and soil are specified. Option "**input angle of friction base-soil**" requires inputting the friction angle  $\psi$  [°] between foundation and soil. Option "**input friction coefficient**" requires specifying the friction coefficient  $\mu$  [-]. Both options require inputting the cohesion  $a$  [kPa] between foundation (base) and soil.

The input data introduced in this frame influence the **spread footing bearing capacity**.



Frame "Footing bottom"

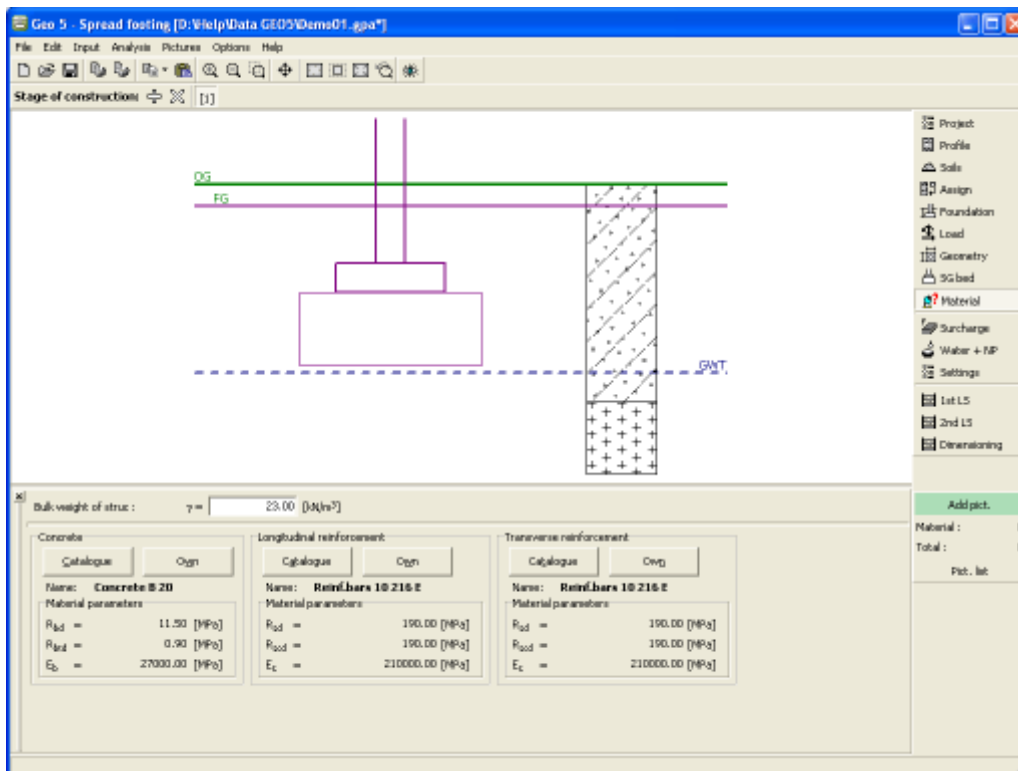
## Sand-Gravel Cushion

The "**Sand-gravel cushion**" frame allows to input parameters of the sand-gravel cushion below foundation. The cushion thickness and overhang over foundation edge are required. The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The cushion filling can be selected from a combo list that contains soils specified in the frame "**Soils**".

The input sand-gravel cushion influences the analysis of both the **foundation load bearing capacity** and **settlement**.





Frame "Sand - gravel cushion"

## Material

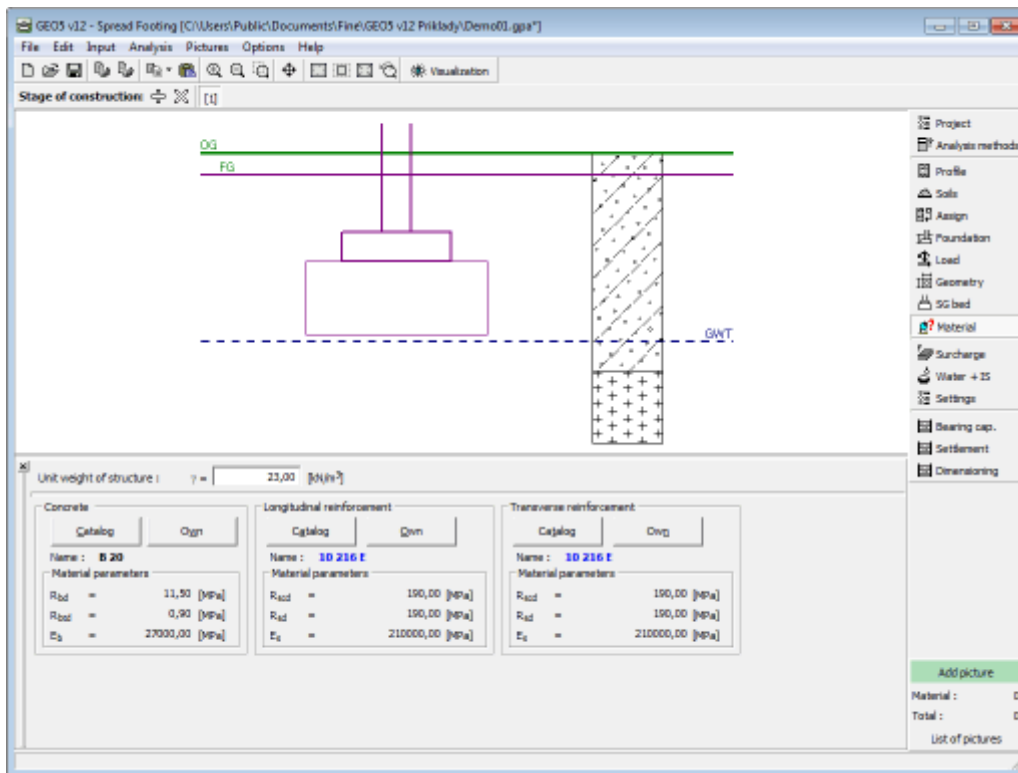
The "Material" frame allows to select material parameters for concrete and longitudinal and transverse steel reinforcements.

Two options are available when selecting the material type:

- the "Catalog" button opens the "Material catalog" dialog window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "Own" button opens the "Concrete" dialog window (for concrete) or the "Reinforcing steel bars" dialog window (for longitudinal and transverse steel reinforcements), which allows for manual specification of material parameters

The catalogs content depends on the selection of standard for the design of concrete structures set in the "Materials and standards" tab.

The input field in the upper part of the frame serves to specify the **wall unit weight**.



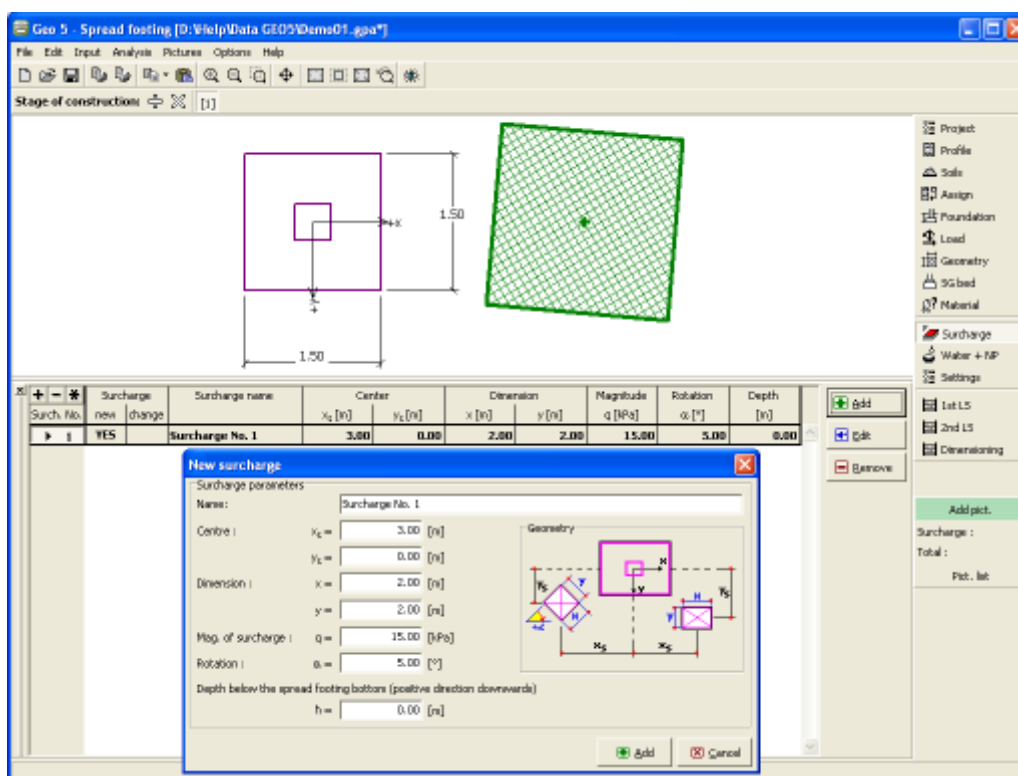
Frame "Material"

## Surcharge

The "Surcharge" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "New surcharge" dialog window. The values are specified according to "Geometry" chart displayed in the right part of the dialog window. The input surcharges can also be edited on the desktop with the help of active objects.

The z-coordinate measured from the foundation joint of a structure is specified (positive direction downwards) when inputting the surcharge at a depth different from the depth of foundation joint.

The surcharge is considered only when **computing settlement** and rotation of a foundation, in which case it **increases the stress** in soil below foundation. When **computing the foundation bearing capacity**, the surcharge is not considered - its presence would increase the bearing capacity.



Frame "Surcharge"

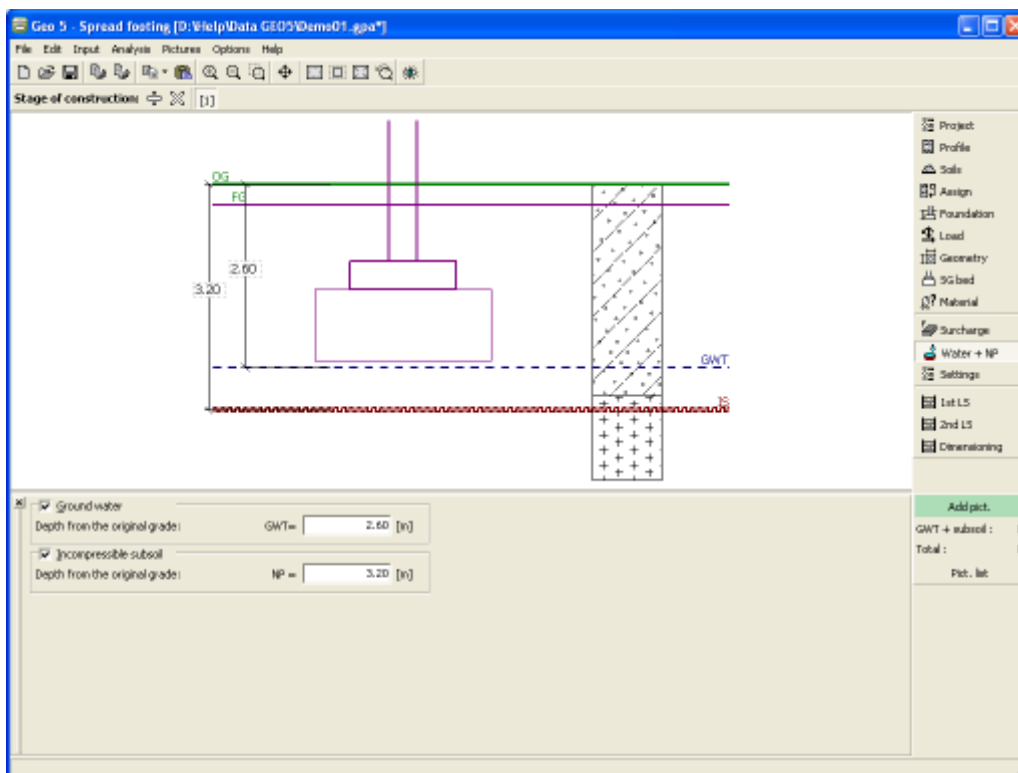
## Water, Incompressible Subsoil

The "Water + IS" frame serves to specify the **depth of ground water table** and level of **incompressible subsoil**.

The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The **GWT** changes the **geostatic stress** in the soil profile.

The **incompressible subsoil** cuts off the **influence zone** below foundation and also influences reduction in settlement.



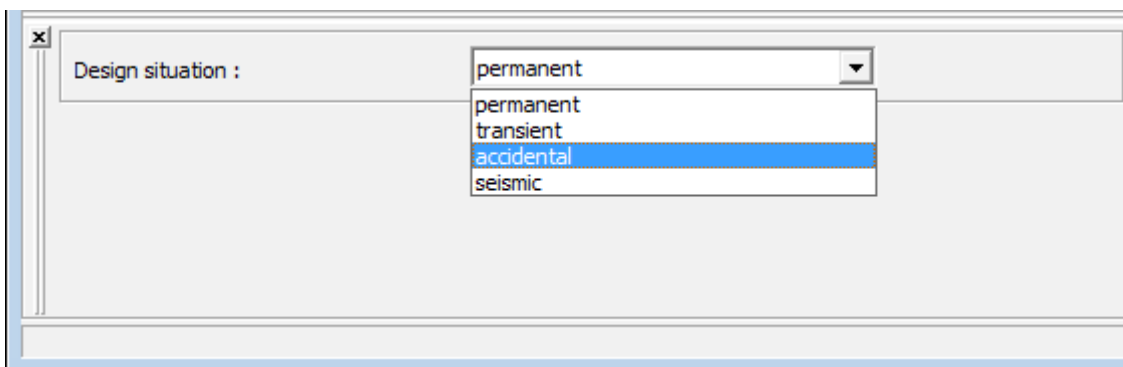
Frame "Water, incompressible subsoil"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.

The frame view depends on the selected verification methodology. LRFD 2012 introduces new types of design situations (**Strength I, Service I, Extreme I**).



Frame "Stage settings"

## Bearing Capacity

The "Bearing capacity" frame serves to verify the vertical and horizontal bearing capacity of a footing. More computations can be performed in the frame. The verification can

be performed either for individual loads or the program finds the **most critical one** (can be selected from a combo list).

The analysis follows the theory approach selected in the "Spread Footing" tab. This tab serves to choose the **verification methodology** (according to EN 1997, LRFD, factor of safety, limit states).

The **vertical bearing capacity** analysis requires selection of the **type of contact pressure** (general shape, rectangle). The shape of contact pressure is plotted in the left part of the desktop.

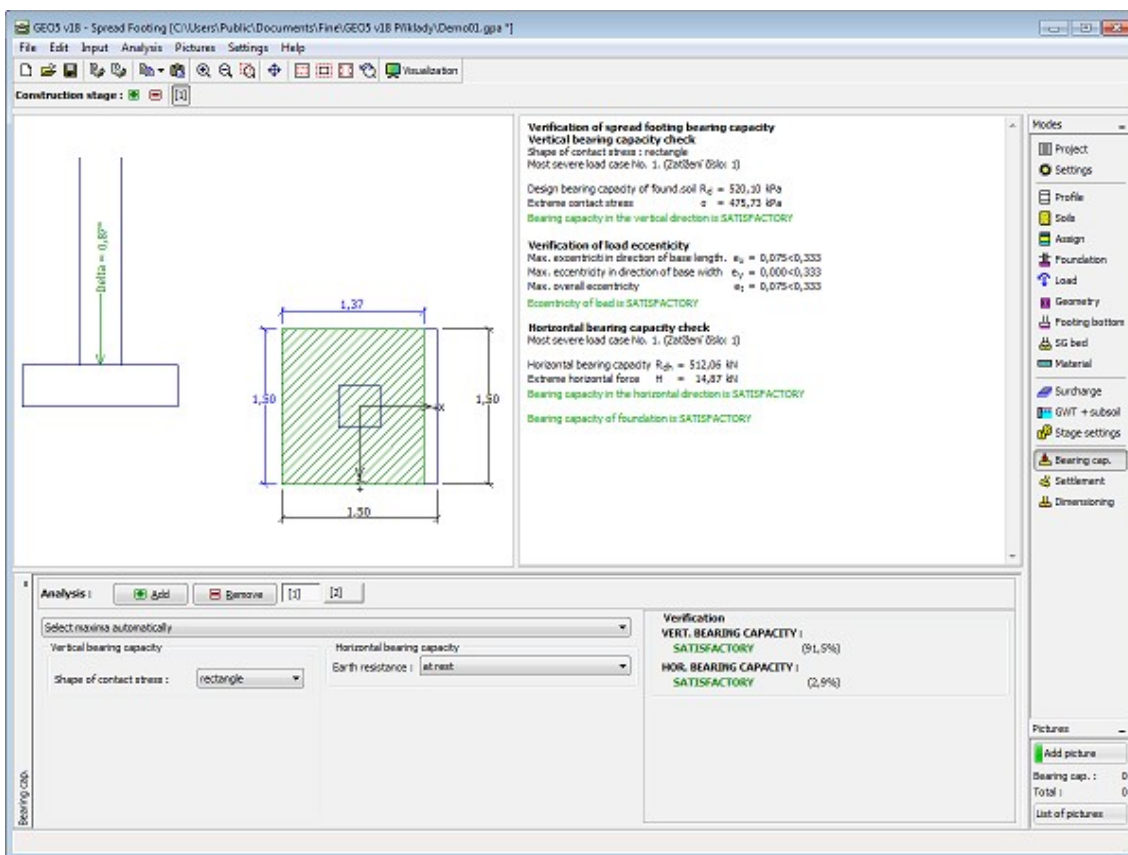
For both limit states (strength, usability) program assesses the **eccentricity of the foundation**. The value of the maximum allowable eccentricity of foundation  $e_{alw}$  is assumed in the frame "Settings" in the "Spread Footing" tab.

The **horizontal bearing capacity** analysis requires selection of the type of earth resistance that can be assumed as the **pressure at rest, passive pressure** or the **reduced passive pressure**.

The **soil parameters** (friction angle structure-soil, cohesion structure-soil) can be further **reduced** when computing the horizontal bearing capacity.

When **evaluating the uplift resistance** the view of the "Verification on uplift" dialog window is adjusted according to the analysis method selected in the "Settings" frame.

Detailed listing of the results is displayed in the right part of the desktop. Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Bearing capacity"

## Settlement and Rotation

The "**Settlement**" frame serves to compute the foundation settlement and rotation. The frame allows for **more analyses**. The verification can be performed either for individual loads or the program finds the **most critical one** (can be selected from combo list).

The **analysis of foundation settlement and rotation** is carried out according to the theory specified in the frame "**Spread Footing**" tab.

For both limit states (strength, usability) program assesses the **eccentricity of the foundation**. The value of the maximum allowable eccentricity of foundation  $e_{alw}$  is assumed in the frame "**Settings**" in the "**Spread Footing**" tab.

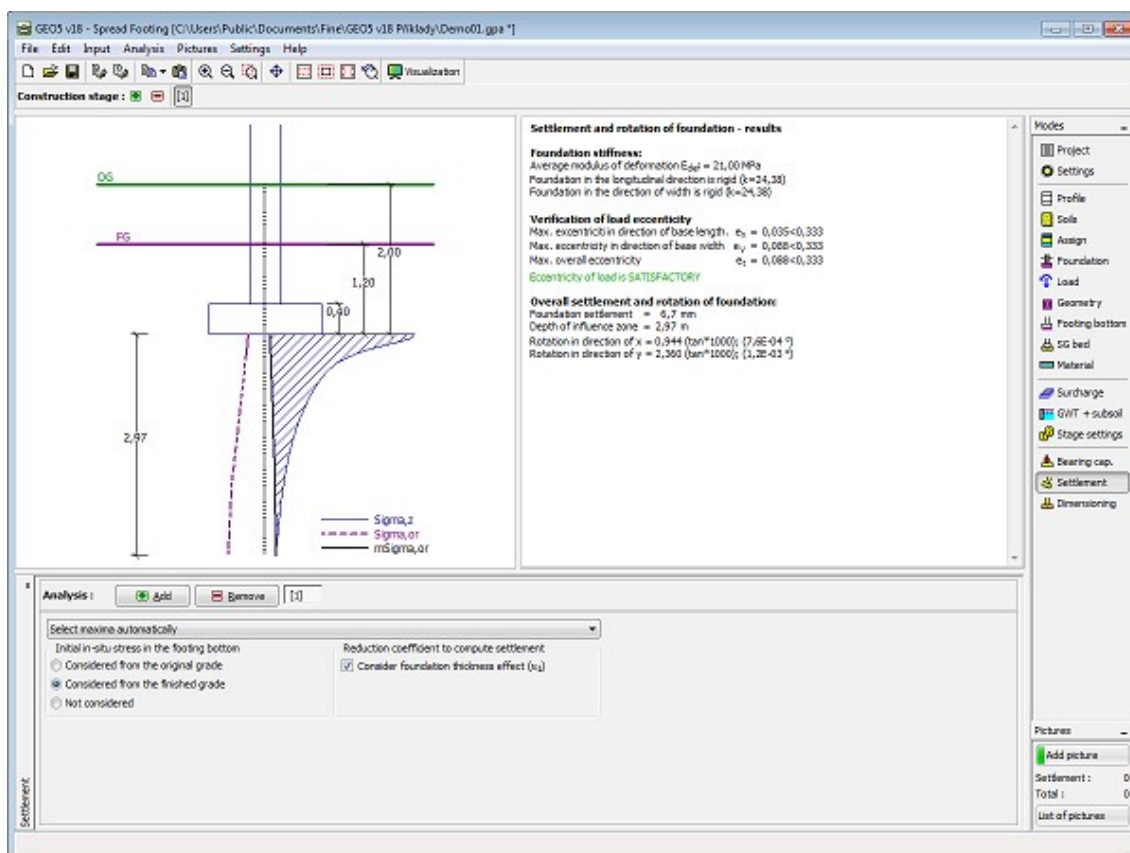
The **stress in the footing** button can be subtracted from the geostatic stress given by:

- **original ground**
- **finished grade**
- **not specified**

Distributions of the **geostatic stress** and the **stress increment** below foundation are displayed in the left part of the desktop. The label below footing represents the **depth of deformation zone**. The stress is drawn below footing at the point with a **characteristic deformation**.

The frame also allows for specifying the **coefficient of reduction of computation of settlement**.

The detailed listing of the verification analysis results is displayed in the right part of the desktop. Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Settlement"

## Dimensioning

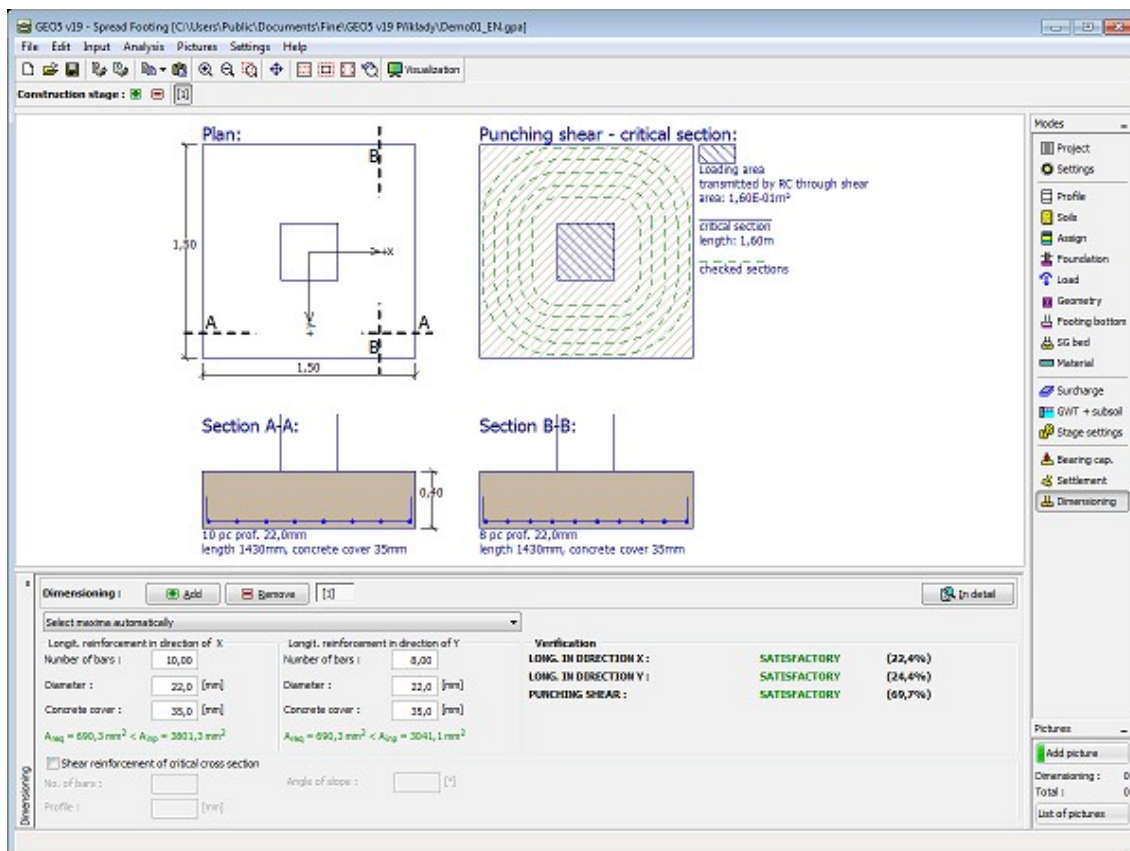
The "**Dimensioning**" frame allows for designing and verifying the longitudinal reinforcement of a foundation and also for verifying the foundation against being pushed through. The verification can be performed either for individual loads or the program finds the **most critical one** (can be selected from a combo list).

The program derives the stress in the construction joint and **determines the internal forces** in individual cross-sections.

**Dimensioning of the reinforced concrete** is performed according to the standard set in the "Materials and standards" tab.

The resulting information are displayed on the desktop and are updated with an arbitrary change in data or setting specified in the frame. The "**In detail**" button opens the dialog window that contains detailed listing of the dimensioning results.

Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Dimensioning"

## Program Pile

This program is used for analysis of vertical bearing capacity of a single pile loaded both in tension or compression, pile settlement as well as horizontal bearing capacity of a single pile.

**The help in the program "Pile" includes the following topics:**

- Input of data into individual frames:

Project	Settings	Profile	Modulus of Subsoil Reaction	Soils	Assign	Load
Geometry	Material	Water, Incompressible Subsoil	Negative Skin Friction	Stage Settings	Vertical Bearing Capacity (Analytical Solution)	Vertical Bearing Capacity (Spring Method)
Settlement (Poulos)	Settlement (Masopust)	Horizontal Bearing Capacity	Horizontal Bearing Capacity (Brom's Method)			

- Standards and analysis methods

- Theory for analysis in the program "**Pile**":

Stress in Soil Body

Pile Analysis

Dimensioning of Concrete Structures

- Outputs
- General information about the work in the [User Environment](#) of GEO5 programs
- [Common input](#) for all programs

## Project

The [frame "Project"](#) is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in [text and graphical outputs](#).

The frame also allows user to switch analysis units ([metric / imperial](#)). Project data can be copied within all GEO5 programs using "[GeoClipboard](#)".



Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

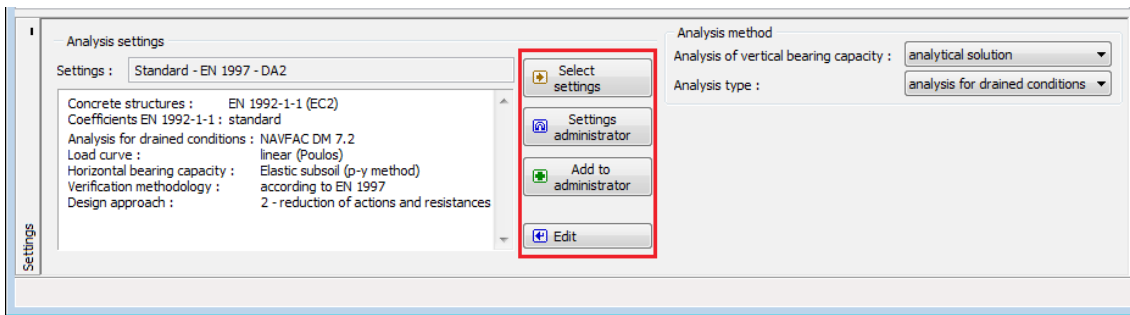
The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Materials and standards"** and **"Piles"** tabs.

The pile vertical bearing capacity can be found either by using the **analytical solution** or the **spring method**. Analytical solution is defined for:

- **analysis for drained conditions** (CSN 73 1002, Effective stress method, NAVFAC DM 7.2)
- **analysis for undrained conditions** (Tomlinson, NAVFAC DM 7.2)



Frame "Settings"

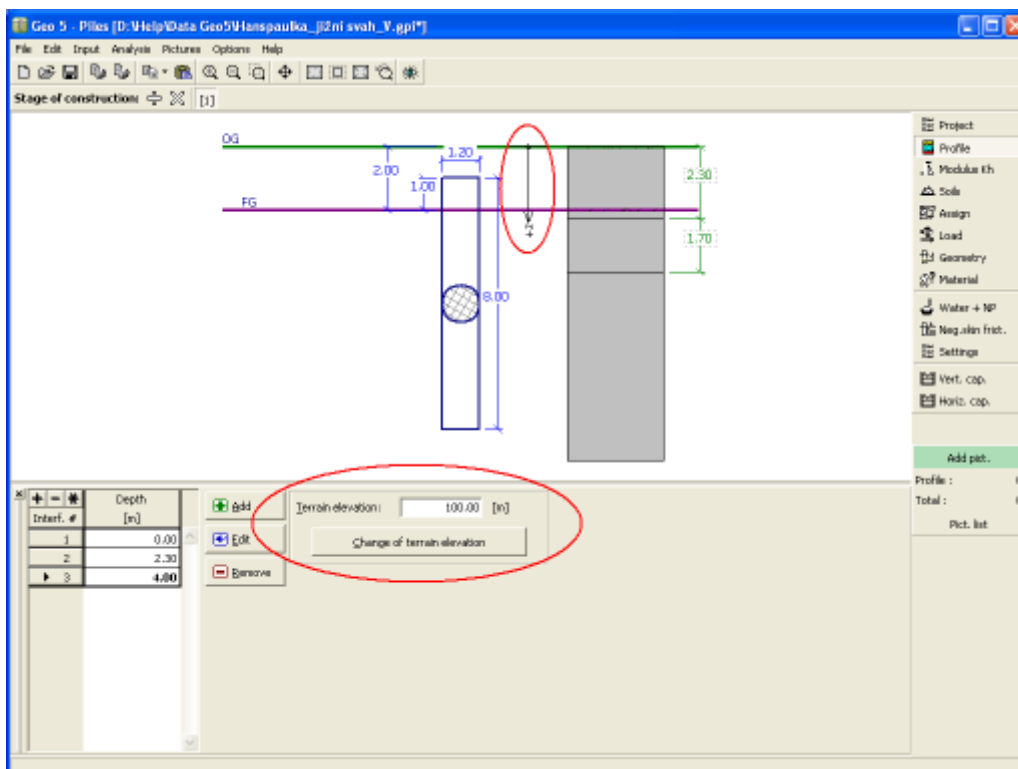
## Profile

The "Profile" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of active dimensions.

Adding (editing) layer is performed in the "New interface" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "Change of terrain elevation" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "Terrain".

The program makes it possible to import a profile in the gINT format.

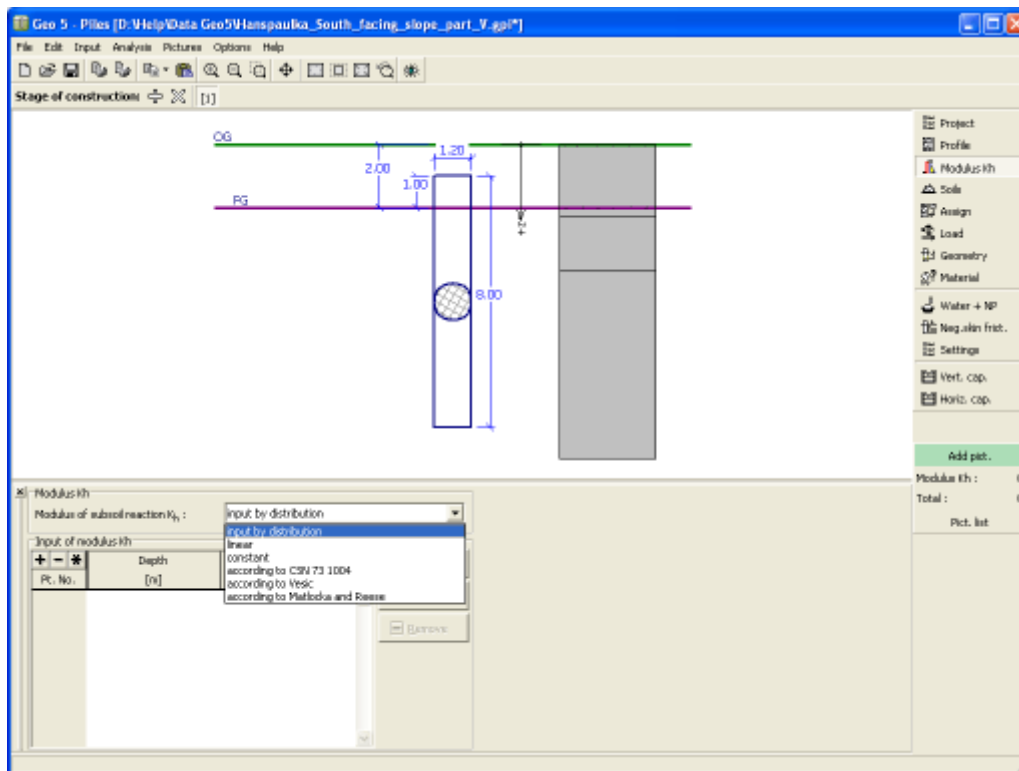


Frame "Profile"

## Modulus of Subsoil Reaction

The combo list serves to select one of the **methods for the evaluation of modulus of subsoil reaction** - the required material parameters of soils are input in the frame "Soils" based on the selected method.

Selecting the option "**Input by distribution**" opens a **table** that allows for **specifying** the values of the modulus of subsoil reaction along the pile.



Frame "Modulus of subsoil reaction"

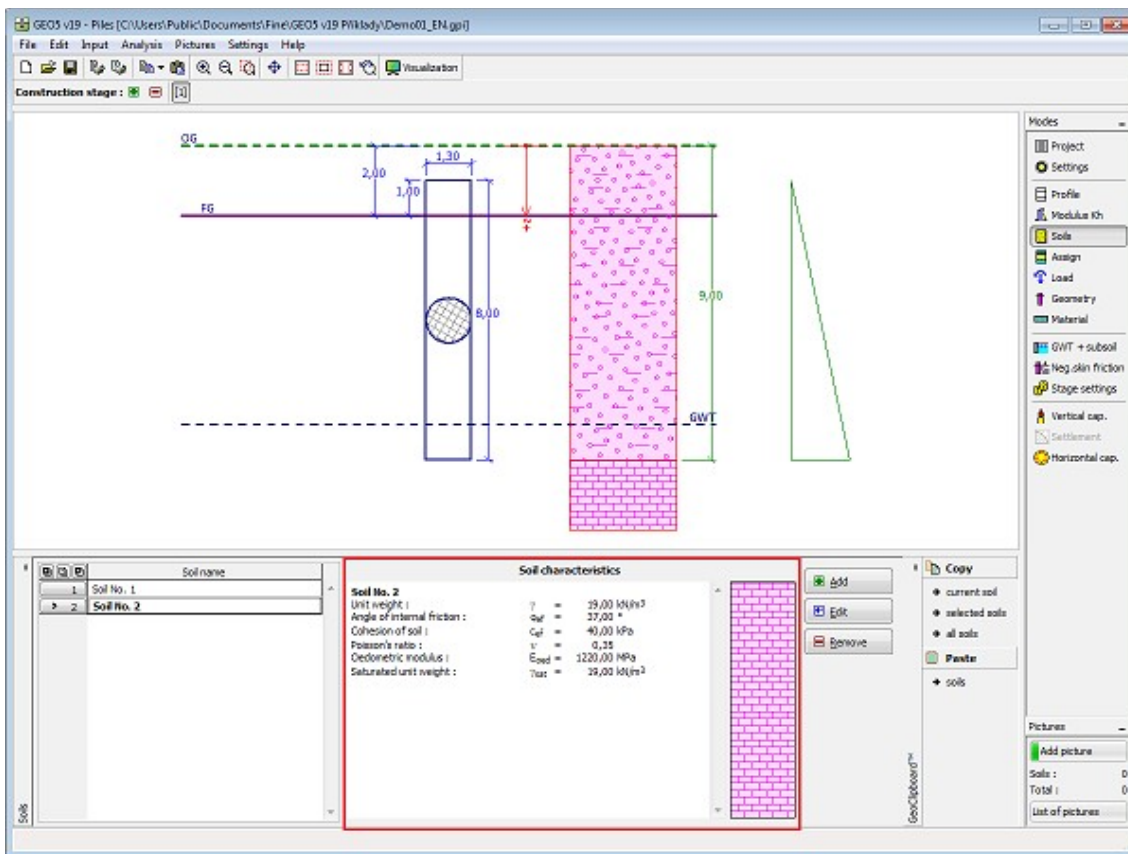
## Soils

The "**Soils**" frame contains a **table** with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame.

**Adding** (editing) a soil is performed in the "**Add new soils**" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "**Basic data**", "**Uplift pressure**", "**Oedometric modulus**", "**Modulus of subsoil reaction**", "**Empirical coefficient of adhesion**" and "**Consistency coefficient**". The specified soil parameters depend on the set up of **modulus of subsoil reaction** and selected theory of analysis specified in the "**Piles**" tab.

The program makes it possible to **import soils in the gINT format**. Data of input soils can be copied within all GEO5 programs using "**GeoClipboard**".



Frame "Soils"

## Basic Data

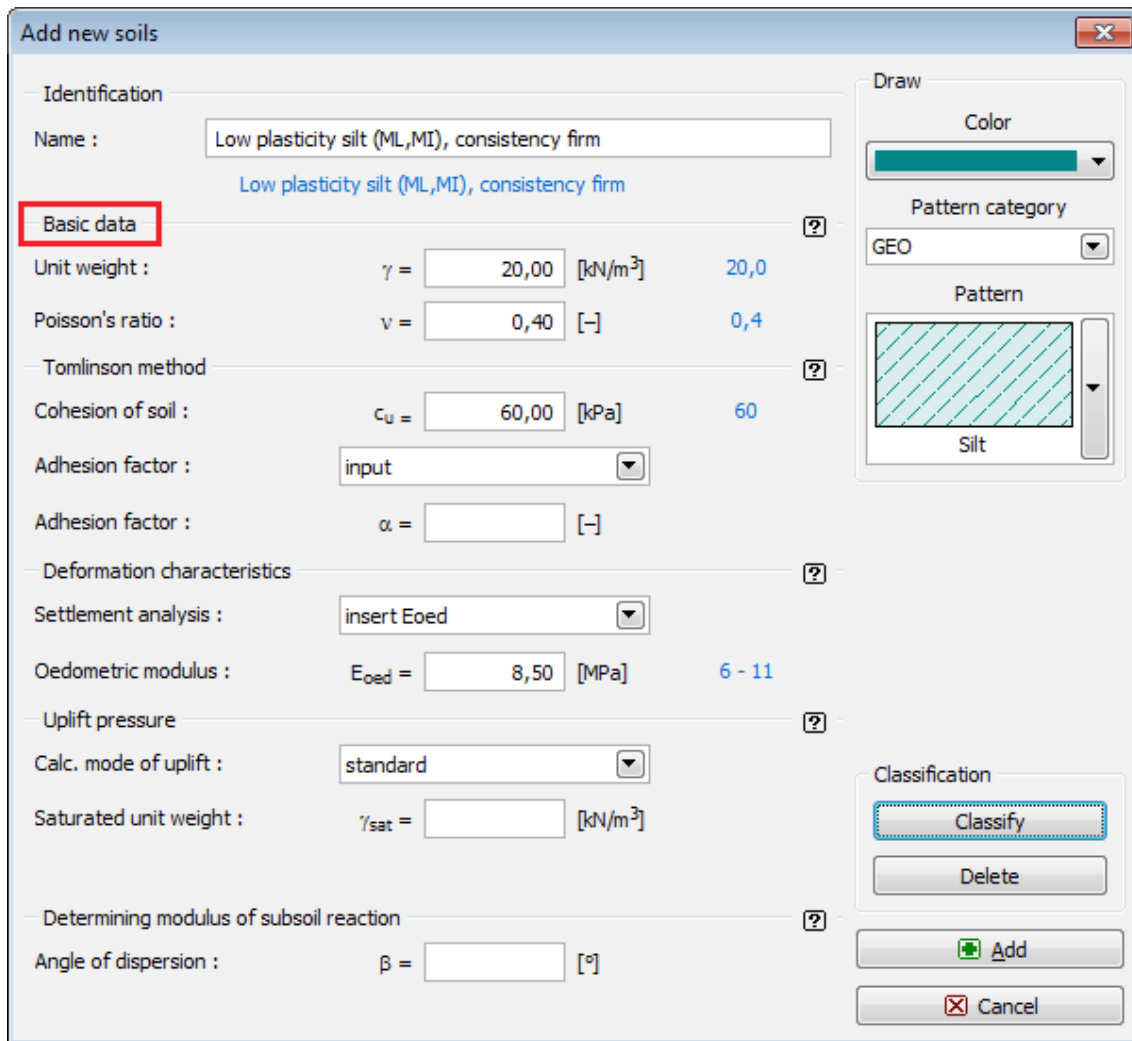
This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

Soil parameters differ according to the analysis type and analysis method ("**Settings**" frame, "**Piles**" tab).

The analysis type differs according:

- **Analysis for drained conditions:** **effective** parameters of shear strength of soil  $c_{ef}$ ,  $\varphi_{ef}$  are used commonly ("**CSN 73 1002**", "**Effective stress**").
- **Analysis for undrained conditions:** in program is defined **total** shear strength of soil  $c_u$  ("**Tomlinson**").
- **Method NAVFAC DM 7.2:** this method combines two types of calculation. For each layer of soil is defined, whether the soil is calculated as drained (cohesionless) or undrained (cohesive).

The associated theory is described in detail in chapter "**Pile analysis**".



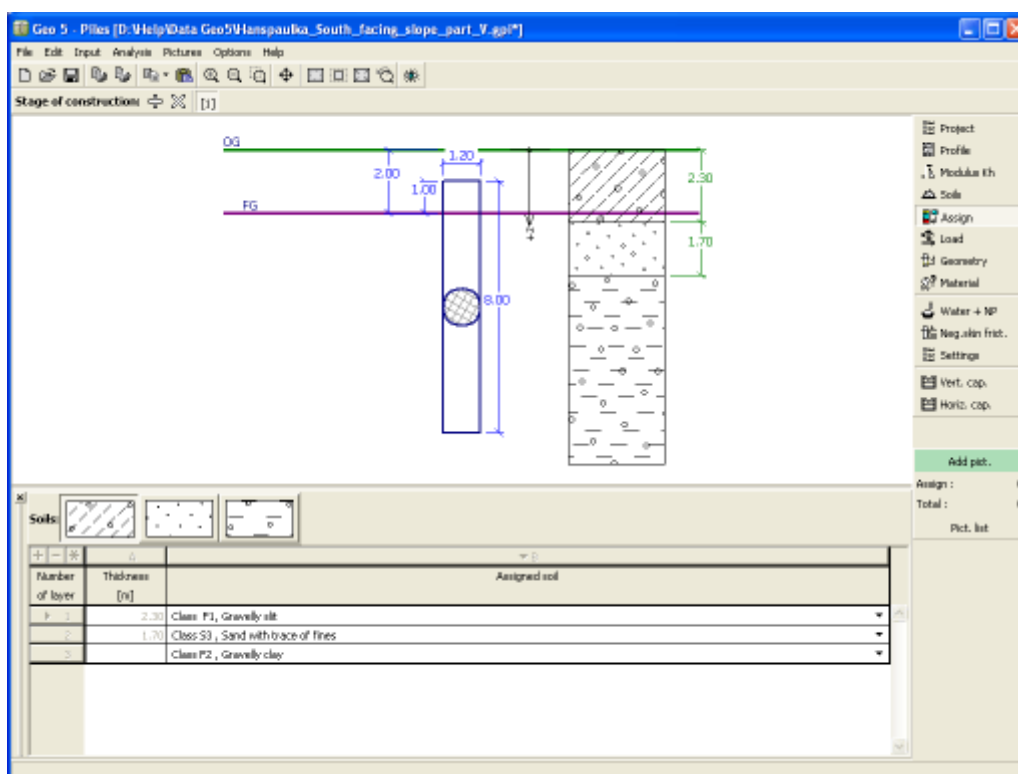
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

## Load

The "**Load**" frame contains a table with a list input loads. Adding (editing) load is performed in the "**New (edit) load**" dialog window. The forces are input following the sign convention displayed in the upper part of the dialog window.

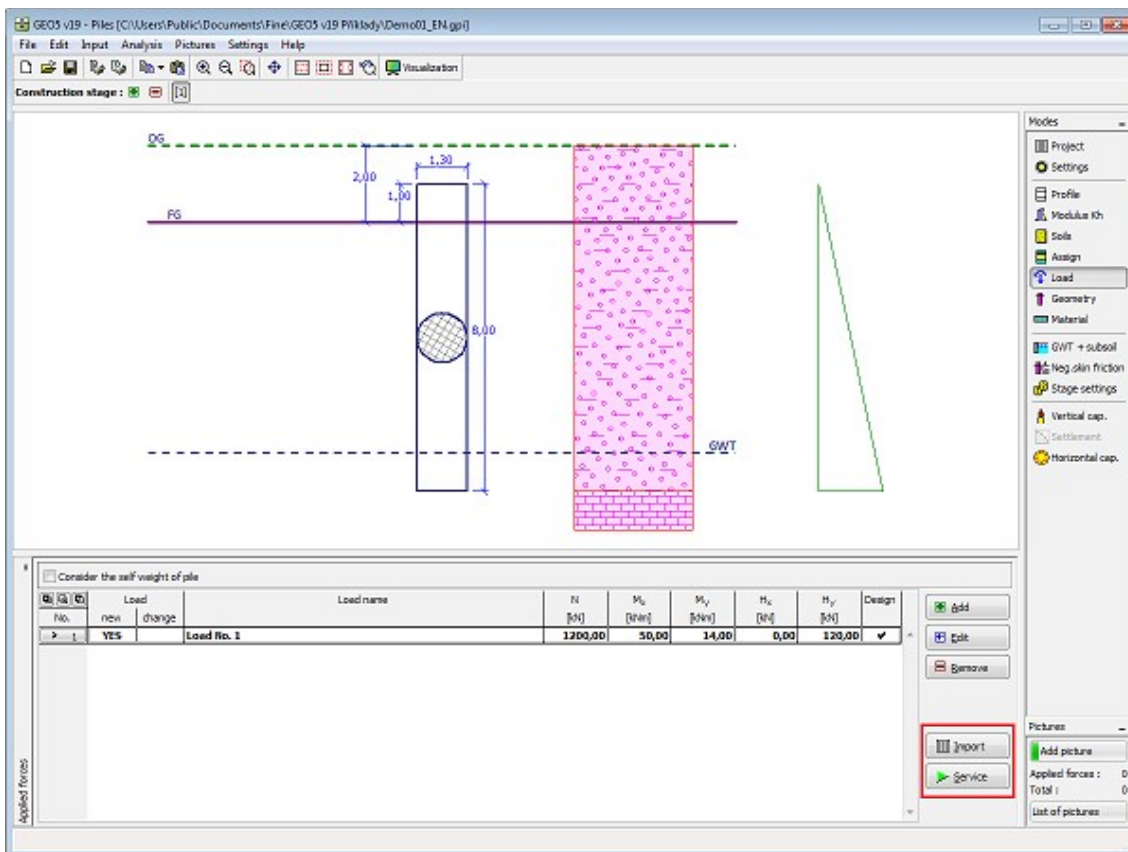
The following types of load can be specified:

- design load serves to verify the vertical and horizontal bearing capacity
- service load serves to compute the settlement of pile (Poulos, Masopust)

When performing the analysis according to **EN 1997 or LRFD** (selected in the "Piles" tab) it is assumed that the design load is determined in accordance with the corresponding standards and individual components of load are **already pre-multiplied** by corresponding partial factors - the program does not modify the input load **any further**.

The "**Service**" button allows for creating the service loads from the already input design loads (analysis according to the factor of safety or the theory of limit states).

The program also allows for import of load using the "**Import**" button.



Frame "Load"

## Geometry

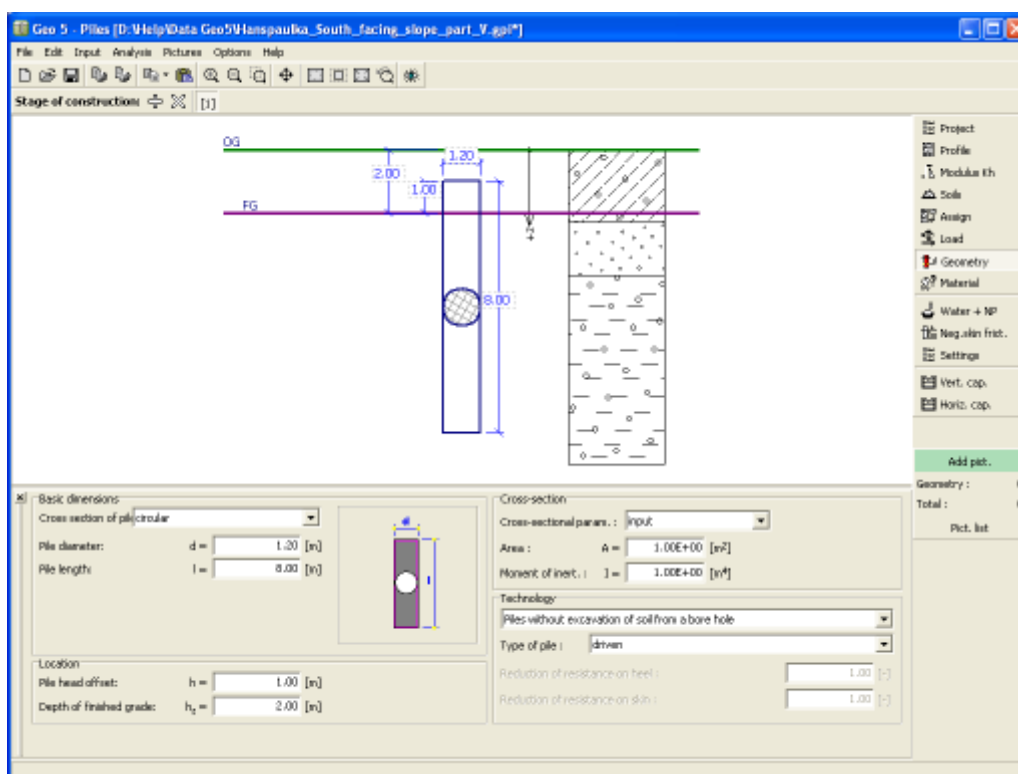
The **"Geometry"** frame allows for specifying the **pile cross-section** (circular, circular variable, rectangle, I-type cross-section) based on the **theory of analysis** (specified in the "Piles" tab). The selected shape with graphic hint is displayed in the central section of the frame. Input fields serve to specify dimensions of the selected cross-section.

**Cross-sectional characteristics** (area and moment of inertia) are computed by default, but they can also be specified (tubes, hollow cross-sections, steel **I**-profiles).

The bottom part of the frame serves to specify the pile location (pile lift out and the depth of finished grade). The pile lift out can also be negative - in such a case the pile is placed **"in-ground"**.

For piles analyzed using the **spring method** method and **analytical solution**, it is possible to account for the **influence of pile technology** by selecting the specific type of pile or directly by inputting coefficients.

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Geometry"

## Material

The "**Material**" frame allows for specifying the material parameters. The **unit weight of a structure** and material of a pile (**concrete, timber, steel**) are introduced in the input field in the right part of the frame.

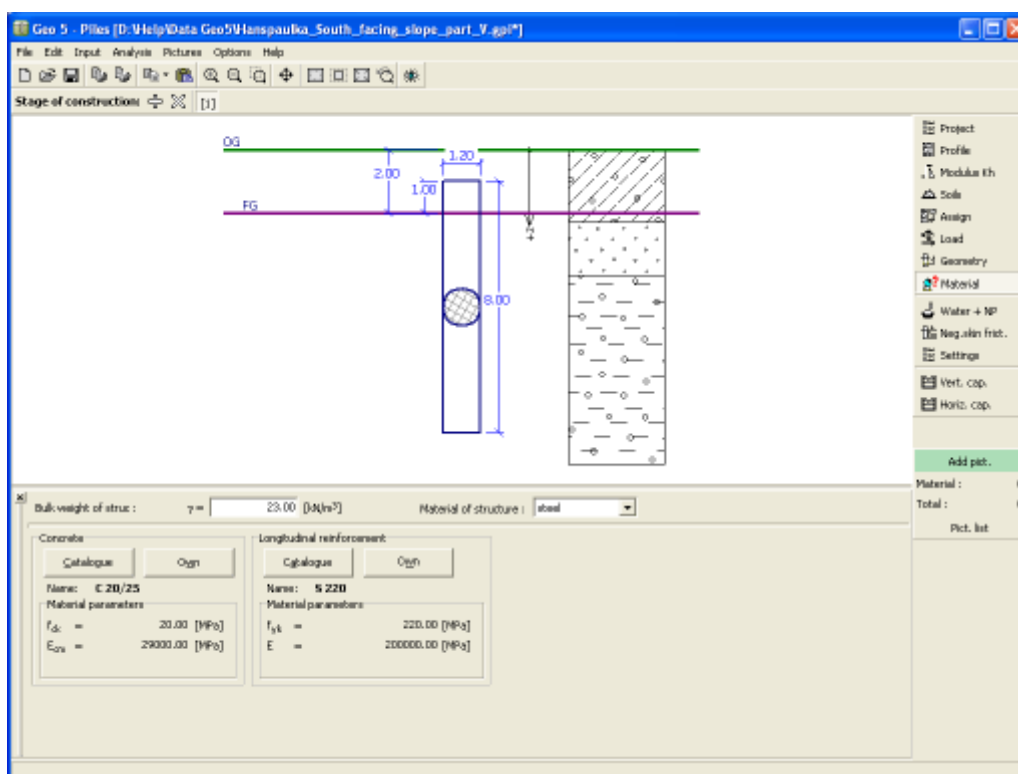
The elastic and shear moduli need to be specified when assuming **timber** or **steel** piles.

In case of a **concrete pile** the concrete material and parameters of transverse and longitudinal steel reinforcements are required. Two options are available when selecting the type of material:

- the "**Catalog**" button opens the "**Catalog of materials**" dialog window (for concrete or steel reinforcements), the list of materials then serves to select the desired material
- the "**Own**" button opens the "**Editor of material - Concrete**" dialog window (for concrete) or the "**Editor of material - Reinforcing steel bars**" dialog window (for longitudinal steel reinforcements), which allows for manual specification of material parameters

The catalogs content depends on the selection of standard for the design of concrete structures set in the "**Materials and standards**" tab.





Frame "Material"

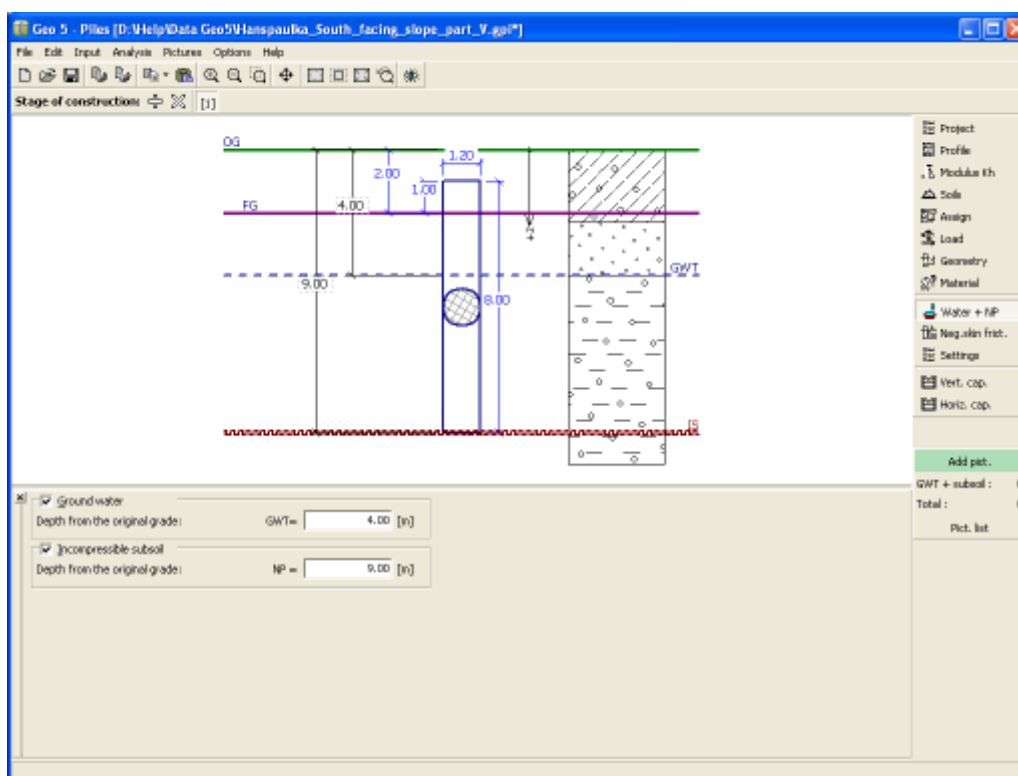
## Water, Incompressible Subsoil

The "**Water + IS**" frame serves to specify the **depth of ground water table** and level of **incompressible subsoil**.

The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**.

The **GWT** changes the **geostatic stress** in the soil profile.

The **incompressible subsoil** cuts off the **influence zone** below foundation and also influences reduction in settlement.

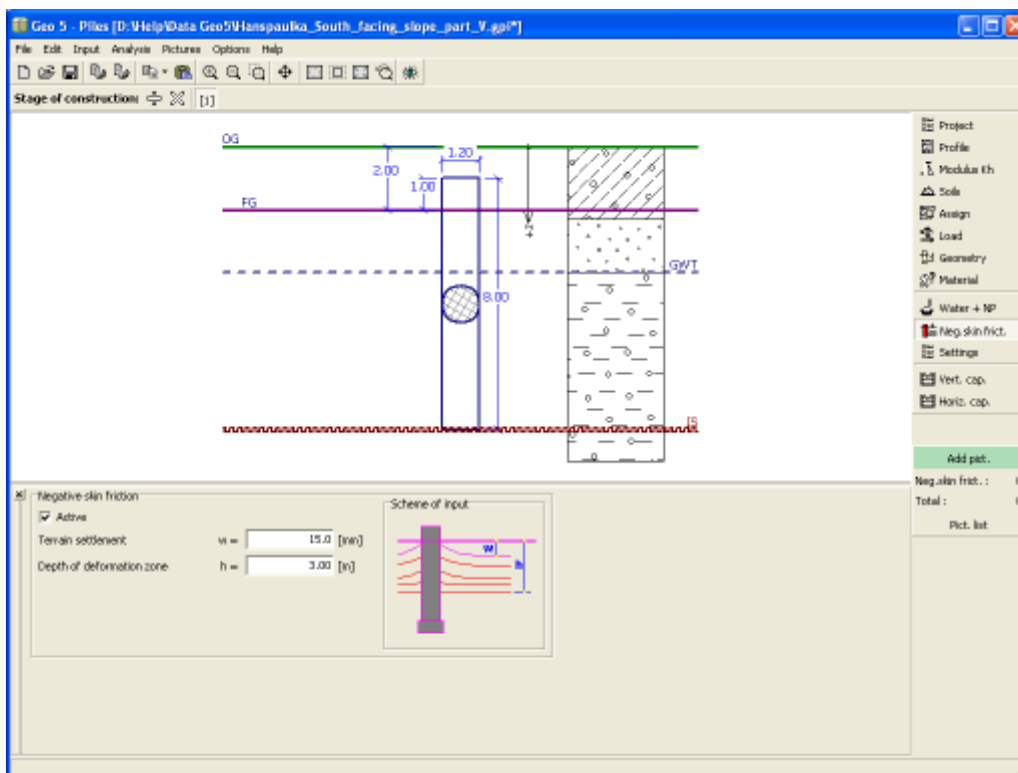


Frame "Water, incompressible subsoil"

## Negative Skin Friction

The "Negative skin friction" frame serves to specify the settlement of surrounding terrain and the depth of influence zone. For more information on the influence of negative skin friction the user is referred to theoretical section.

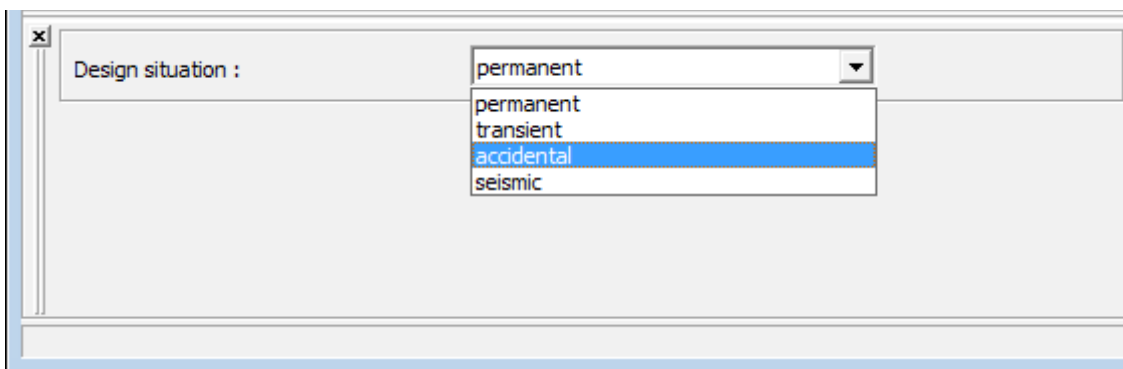
The setting option in the frame is active only when the **spring method** is selected for the analysis in the frame "Settings".



Frame "Negative skin friction"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage. Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.



Frame "Stage settings"

## Vertical Bearing Capacity - Analytical Solution

The "Vertical bearing capacity" frame serves to verify the vertical bearing capacity of a pile. Several analyses can be carried out in the frame. The verification can be performed for individual loads, or the program locates the most critical one (can be selected from a combo list). The frame appearance changes depending on the analysis type setting in the frame

"Settings".

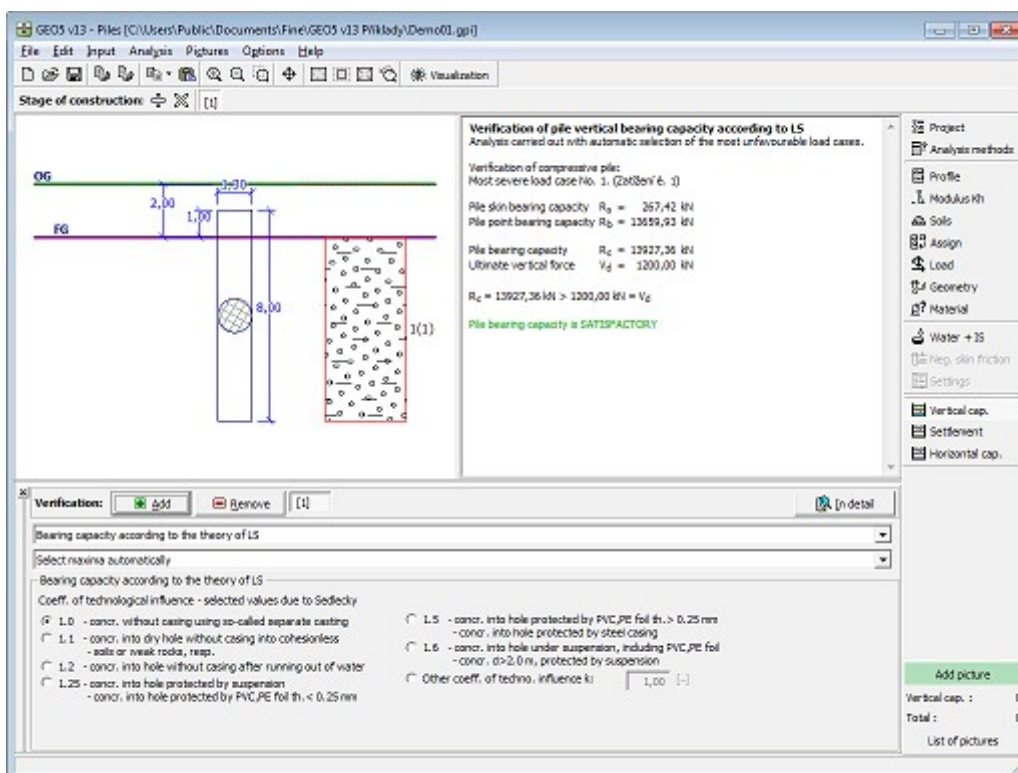
The analysis is performed based on the theory defined in the "Piles" tab. This tab serves to choose the **verification methodology** (EN 1997-1, factor of safety, limit states).

Calculation of the pile vertical bearing capacity by using the **analytical solution** is performed for:

- **drained conditions** (CSN 73 1002, Effective stress method, NAVFAC DM 7.2)
- **undrained conditions** (Tomlinson, NAVFAC DM 7.2)

The "In detail" button opens the dialog window containing detailed listing of the verification results.

The analysis results are displayed in the right part of the desktop. Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Vertical bearing capacity" - analysis based on classic theory

## Vertical Bearing Capacity - Spring Method

The "Vertical bearing capacity" frame allows for verifying the pile **vertical bearing capacity**. The analysis is performed automatically when switching to this frame. **More computations** can be performed in the frame. The verification can be performed either for individual loads or the program finds the **most critical one** (can be selected from a combo list).

The analysis is performed with the help of **spring method**. The results are automatically updated whenever one of the analysis parameters "**Maximal deformation**", "**Coefficient increasing limit skin friction due to technology**" or "**Procedure determining influence zone**" is changed.

Two options are available to determine influence zone:

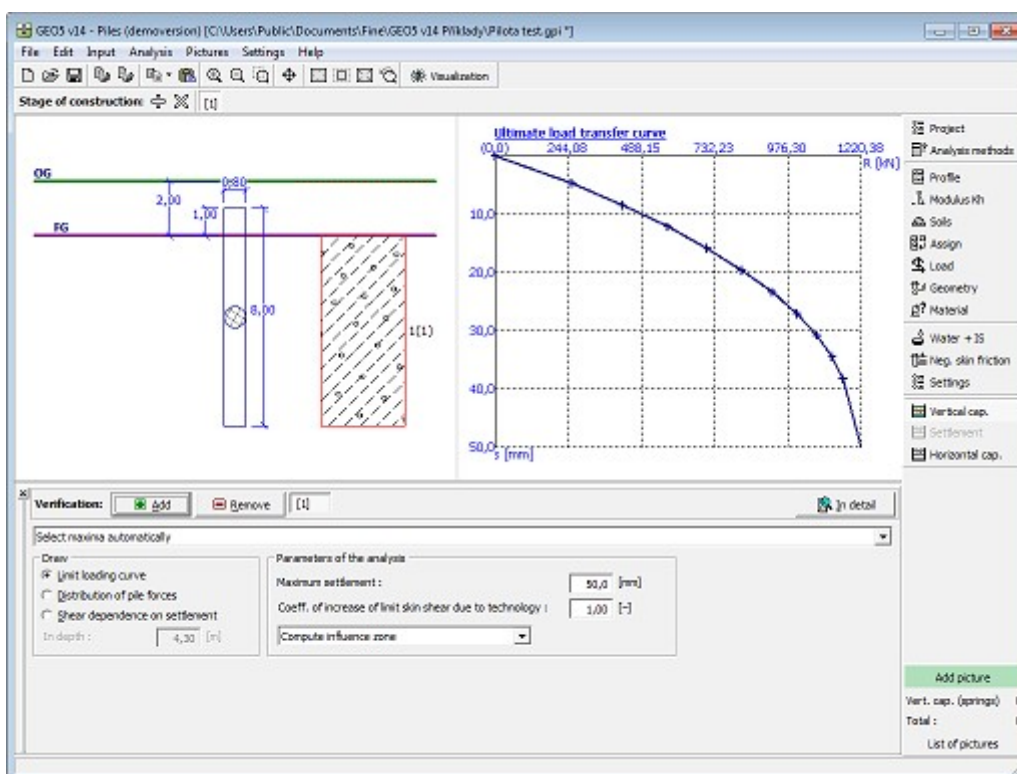
- By default the evaluation of the depth of influence zone below the pile base follows the procedure described in the theoretical part of the help in section "Depth of influence zone". The depth of influence zone on the pile skin is determined as a  $k$ -multiple of the pile diameter. The value of  $k$  increases from 1 for zero load to the value of 2.5 when exceeding the limit skin friction.
- The second option assumes the depth of influence zone below the foot and on the skin to be set conservatively a  $k^{th}$  multiple of the pile diameter, where the value of  $k$  can be selected. During a gradual increase of pile surcharge the value of  $k$  for the depth of influence zone on the pile skin is continuously changed from 1 at the onset of load to the specified value when exceeding the limit skin friction. The value of  $k$  for the influence zone below the pile base remains constant during the analysis.

The second method, originally used in the old version GEO4, with the value of  $k = 2.5$  offers less accurate results and usually underestimates the pile bearing capacity. Therefore a new option that allows for specifying the depth of influence zone through analysis is offered and is also recommended by the default setting.

Switching between results is available in the left part of the frame (load-settlement curve, distributions of internal forces, dependence of shear on displacement). The shear-displacement relationship is derived for a given depth measured from the pile head. The results are updated whenever the depth is changed.

The "In detail" button opens the dialog window, which contains detailed listing of the results of verification analysis.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Vertical bearing capacity" - analysis using spring method

## Settlement - Linear Load-Settlement Curve (Poulos)

The "Settlement" frame serves to display the **pile load-settlement curve**. More computations can be performed in the frame.

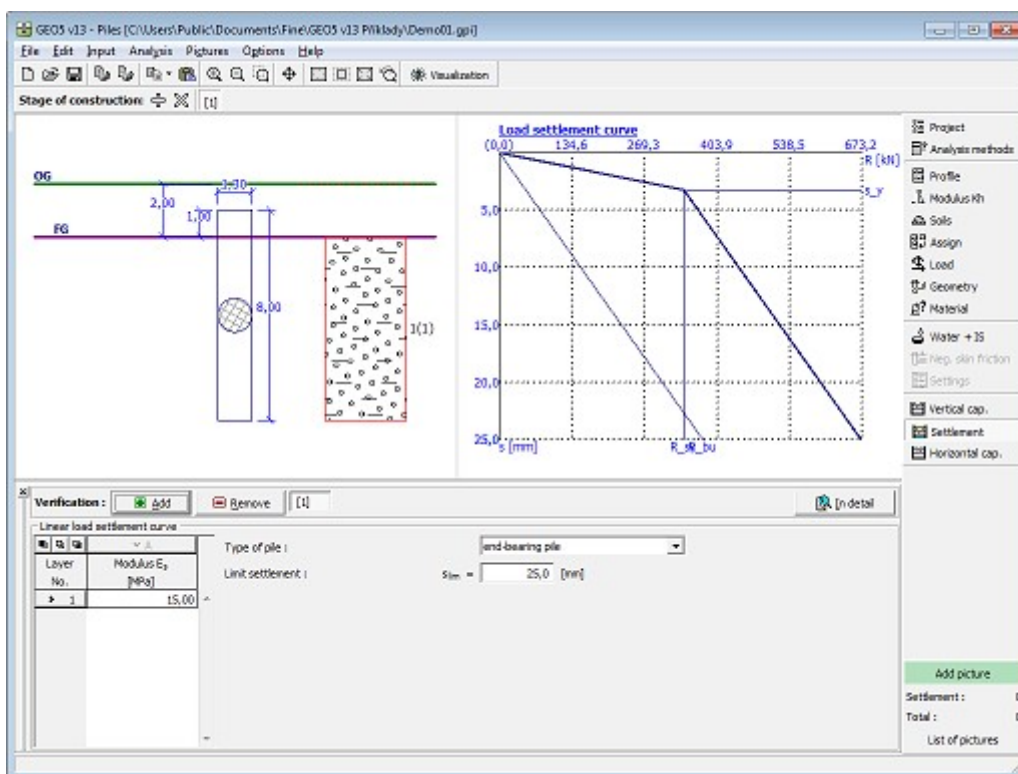
Next, it is necessary to enter the value of limit settlement. The program **constructs the load-settlement curve** of pile always such that this **limit settlement should be reached**.

The analysis is carried out according to the selected theory of settlement analysis (**linear**). The analysis theory is selected in the "Piles" tab. The table in the left bottom part of the frame directly allows to specify values of the **secant modulus of soil** for the relevant layers of soil.

The **analysis results** are displayed in the right part of the desktop. The "In detail" button opens the dialog window containing detailed listing of the verification results.

The analysis results (load-settlement curve of pile) are displayed in the right part of the desktop.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Settlement" - linear load-settlement curve of pile (Poulos)

## Settlement - Non-Linear Load-Settlement Curve (Masopust)

The "Settlement" frame serves to display the **pile load-settlement curve**. More analyses can be performed in the frame.

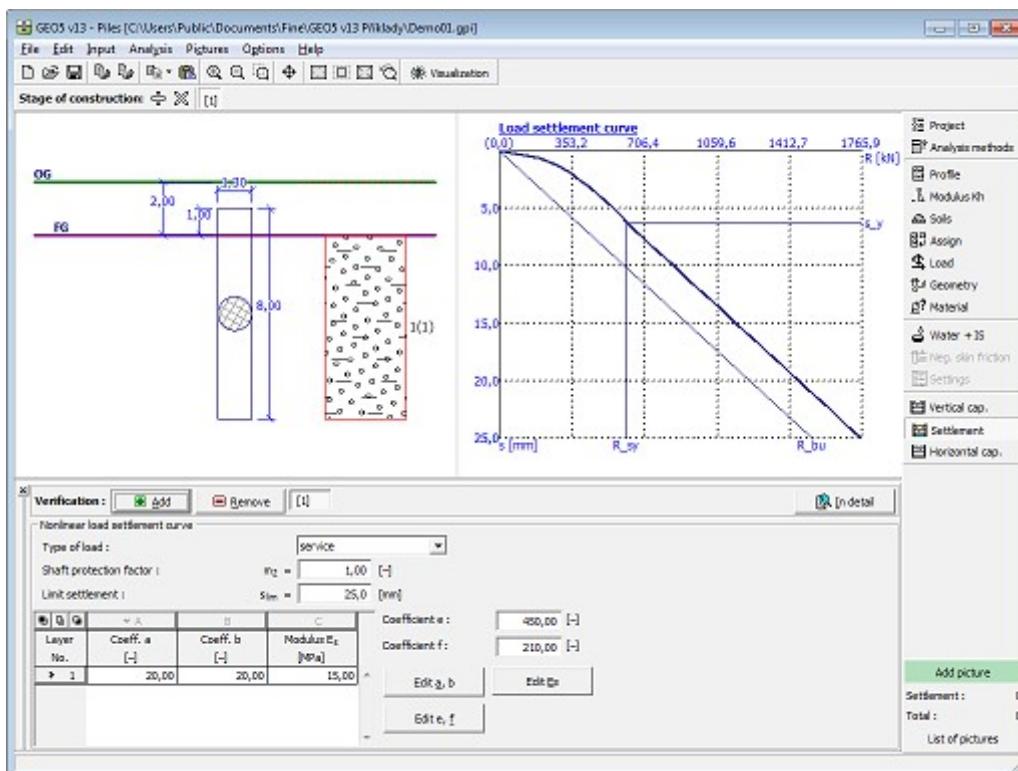
The combo list serves to choose the type of load (**design, service**). Next, it is possible to enter the **coefficient of influence of pile shaft**. The analysis of load-settlement of pile is always performed up to the **limit settlement** of 25 mm.

The analysis is performed according to the selected theory of settlement analysis (**nonlinear**). The analysis theory is selected in the "Piles" tab. The table in the bottom part of the frame directly allows with the help of the mouse for editing the defined parameters. Buttons "Edit  $a$ ,  $b$ ", "Edit  $e, f$ " and "Edit  $E_s$ " open dialog windows with a help section for entered parameters of **regression coefficients** and **secant modulus of soil**. Pressing the "OK" button in a particular window **stores the input parameters** for a given layer into the table.

The **analysis results** are displayed in the right part of the desktop. The "In detail" button opens the dialog window containing detailed listing of the verification results.

The analysis results (load-settlement curve of pile) are displayed in the right part of the desktop.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Settlement" - nonlinear load-settlement curve of pile (Masopust)

## Horizontal Bearing Capacity - Elastic Subsoil (p-y Method)

The **horizontal bearing capacity of a pile** is verified in the "Horizontal bearing capacity" frame. Several analyses can be carried out. The verification analysis can be carried out for individual loads, prescribed displacements, or the program finds the **most critical load** (can be selected from a combo list). Assuming the prescribed displacement type of load requires introduction of **boundary conditions in pile head** (translation and rotation).

The **fixed end** type of boundary condition prescribed in the pile heel can be assumed for all types of load.

For **steel reinforced concrete** pile the programs allow for **verifying the reinforcement** based on the relevant standard selected in the frame "Materials and standards" tab. In this case, here

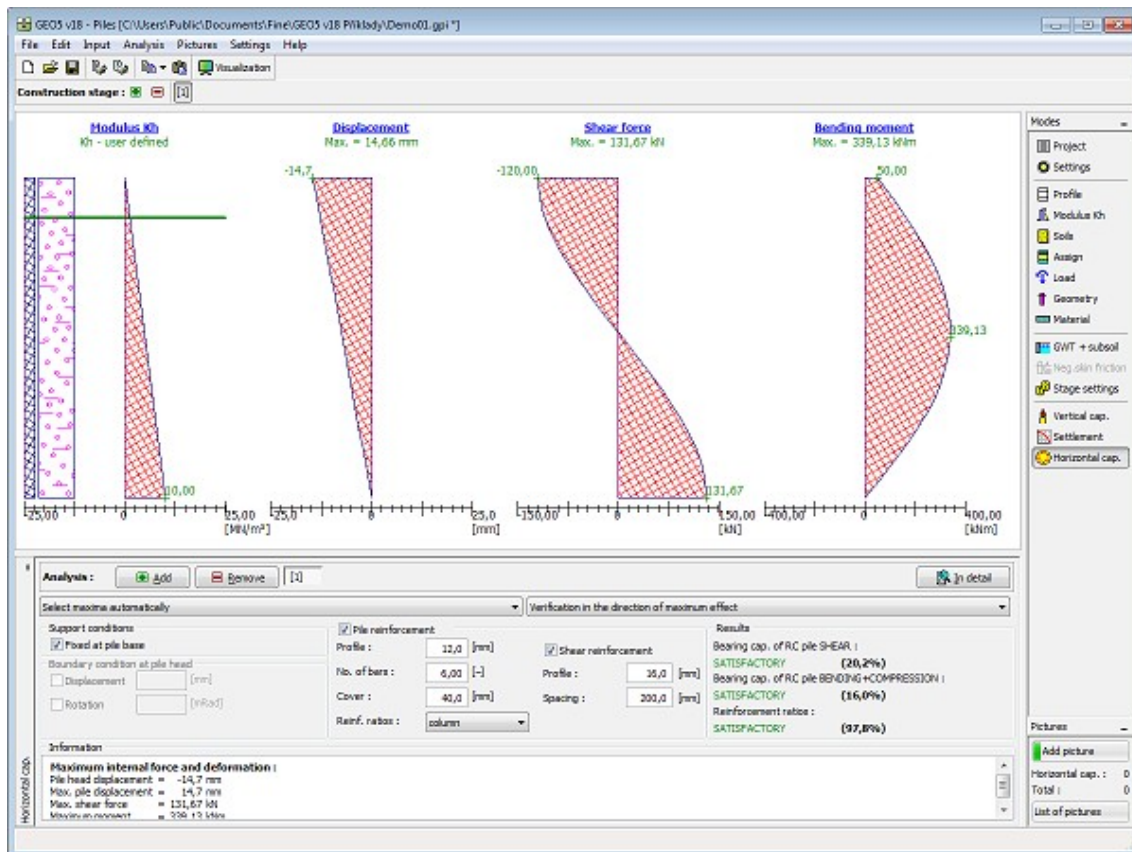


is assumed **profile of reinforcement, number of profiles and concrete cover of reinforcement**.

The combo list serves to specify the direction of pile verification ( $x, y$ ); for a circular pile the program allows for displaying the results in the **most stressed direction**.

The "In detail" button opens the dialog window that contains detailed listing of the verification results.

The analysis results are displayed on the desktop. Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Horizontal bearing capacity" - Elastic subsoil (p-y method)

## Horizontal Bearing Capacity - Brom's Method

The **horizontal bearing capacity of a pile** is verified in the "Horizontal bearing capacity" frame. Several analyses can be carried out. The verification analysis can be carried out for individual loads, prescribed displacements, or the program finds the **most critical load** (can be selected from a combo list). Assuming the prescribed displacement type of load requires introduction of **boundary conditions in pile head** (translation and rotation).

The **reinforced concrete** pile requires inputting the **reinforcement profile, number of profiles and reinforcement cover**, which influence the pile bending stiffness.

The input parameters for the analysis of **pile horizontal bearing capacity** are the **pile material characteristics** (modulus of elasticity and strength of a given material), **pile geometry** (pile length  $l$  and its diameter  $d$ ) and also the **pile load** due to shear force and bending moment.

When adopting the analysis according to the **Broms method** the program ignores up to now



input soil layers and checks the **single pile horizontal bearing capacity** only for the soil defined in the "**Horizontal capacity**" frame. Soil parameters are specified based on the **type of soil**:

- **cohesive** - requires inputting the **undrained cohesion of soil**  $c_u$ , **modulus of subsoil reaction**  $k_h$ , coefficient of cross-section bearing capacity  $\gamma_k$  and the coefficient of bearing capacity reduction  $\gamma_{Qu}$ .
- **cohesionless** - requires inputting the **effective angle of internal friction**  $\varphi$ , **unit weight of soil**  $\gamma$ , furthermore the **coefficient of subsoil reaction**  $n_h$ , coefficient of cross-section bearing capacity  $\gamma_k$  and the coefficient of pile bearing capacity reduction  $\gamma_{Qu}$ .

The frame further allows for inputting the **criteria of type of pile**:

- **standard** - in this case the **pile stiffness coefficient**  $\beta^*l$ , respectively  $\eta^*l$  is calculated automatically by the program.
- **user-defined** - this option allows the user to set the **pile stiffness coefficient**  $\beta^*l$ , respectively  $\eta^*l$  to check short as well as medium piles.

**Type of pile** can be considered in two ways:

- **free head** - rotation at pile head is not constrained
- **restrained** - pile is constrained against rotation at its head. In such cases we typically deal with piles that are part of a planar pile grid or a **pile group**.

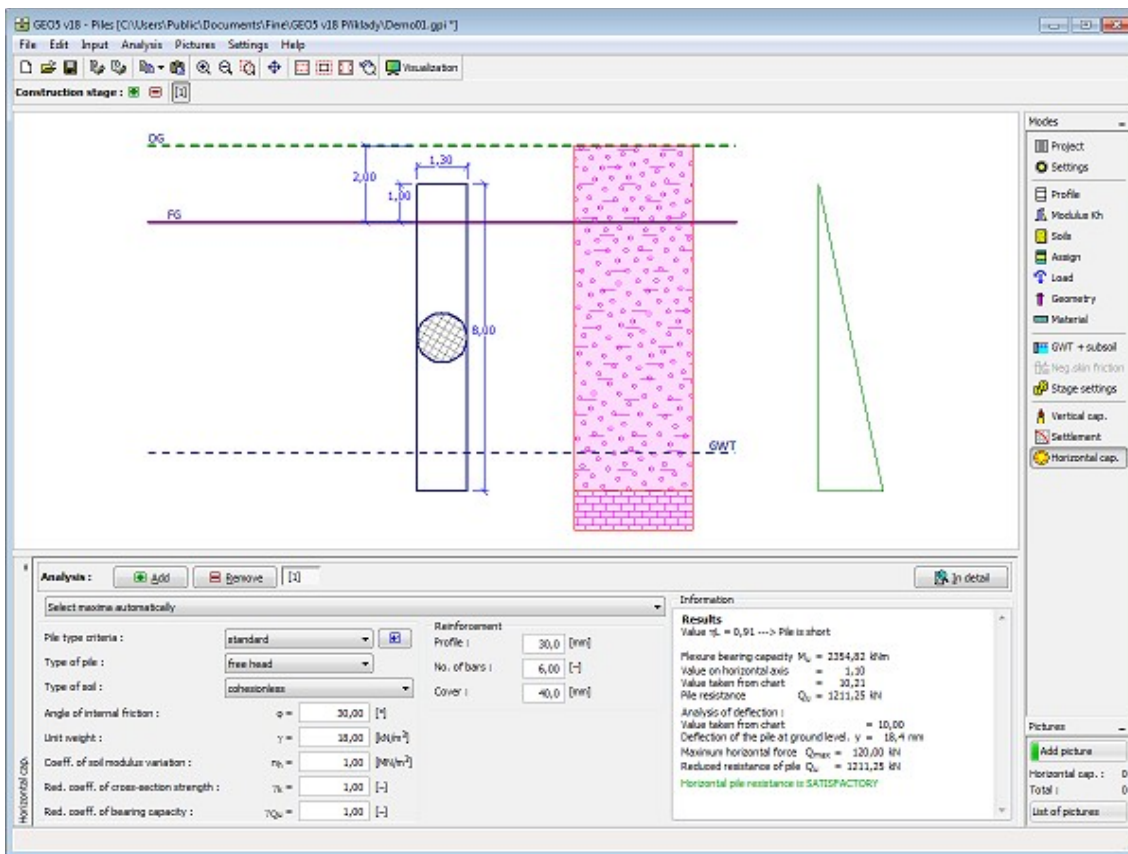
The **reduction coefficient of cross-section strength**  $\gamma_k$  serves to reduce the **flexure bearing capacity**  $M_u$ .

The **reduction coefficient of bearing capacity**  $\gamma_{Qu}$  serves to reduce the overall value **horizontal bearing capacity of a single pile**  $Q_u$ .

The result of an analysis is **horizontal bearing capacity of a single pile**  $Q_u$  and displacement of a pile at the terrain surface  $u$ .

The "**In detail**" button opens the dialog window that contains detailed listing of the verification results.

The analysis results are displayed on the desktop. Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Horizontal bearing capacity" - Broms method

## Program Pile CPT

This program verifies the vertical bearing capacity and settlement of a single pile or a group of piles, based on the results provided by (static) cone penetration tests (CPT).

The help in the program "Pile CPT" includes the following topics:

- Input of data into individual frames:

Project	Settings	CPT	GWT + NSF	Soil Classification	Profile	Soils
Assign	Construction	Geometry	Bearing Capacity	Settlement		

- Standards and analysis methods

- Theory for analysis in the program "Pile CPT":

Stress in Soil Body	Pile CPT
---------------------	----------

- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.

*Frame "Project"*

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

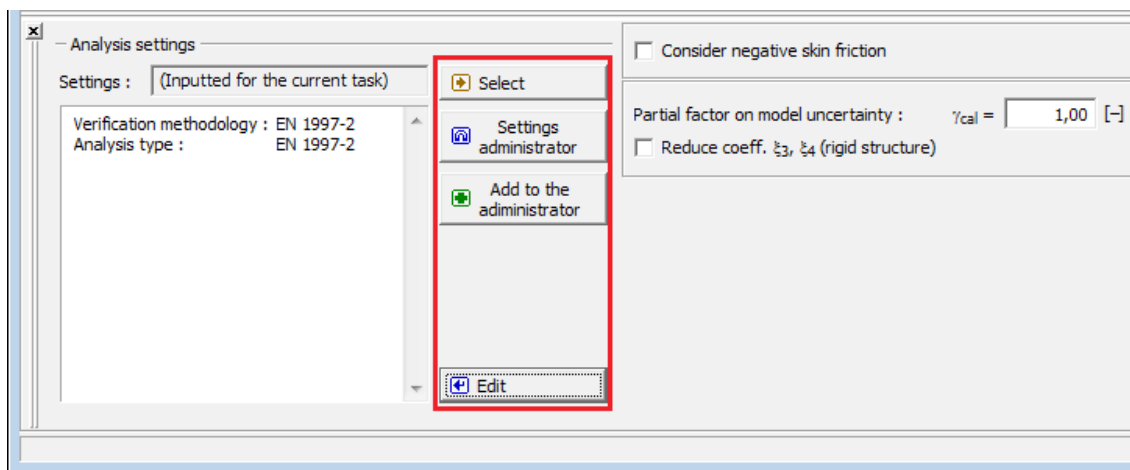
The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Piles CPT**" tab.

This frame also allows to introduce **negative skin friction**. Parameters of the negative skin friction are defined in the frame "**GWT+NSF**".

This frame also allows to introduce soil behavior type (SBT) - **classification of soils**, which is performed in the frame "**Soil Classification**".

Partial factor of model uncertainty  $\gamma_{cal}$  reduces **vertical bearing capacity** of single pile or **pile group**.



Frame "Settings"

## CPT

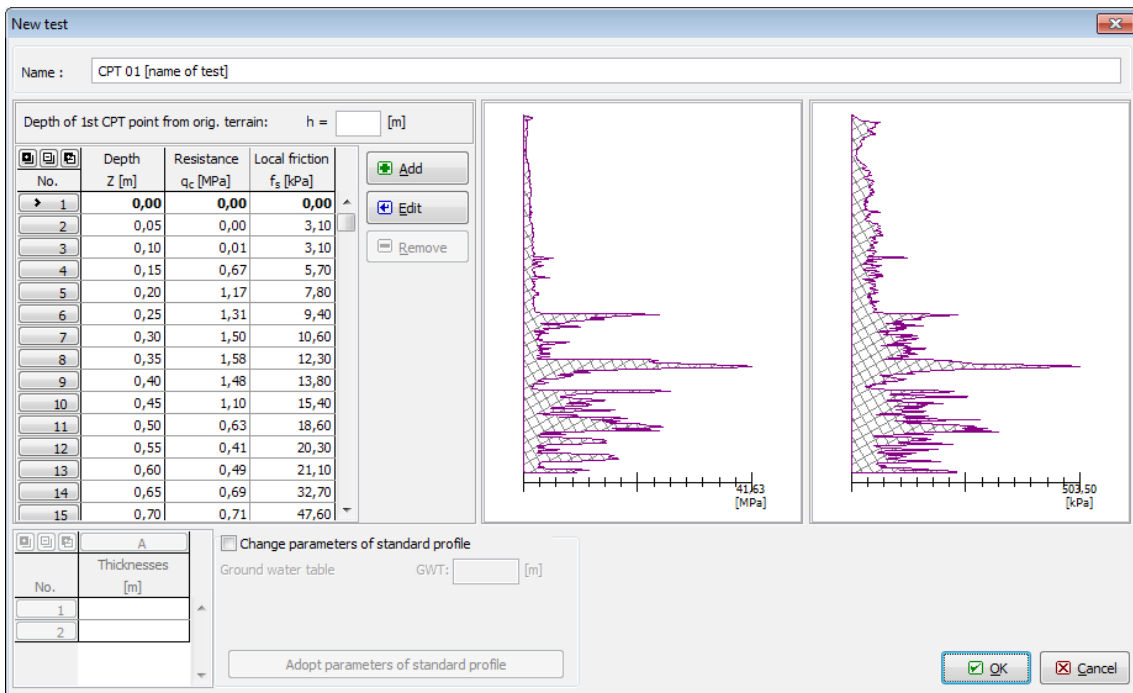
The **frame "CPT"** contains a **table** with list of **input tests of cone penetration (CPT)**. The results of CPT are **imported** by pressing the "**Import**" button. The procedure of an import of table data is more described **herein**.

Individual parameters of the test are defined in the "**New CPT**" ("**Edit CPT**") **dialog window**. This window serves to input a name of the test, a depth of the first point of the CPT  $h$  [m] from the finished grade, depth  $Z$  [m] (in the  $x, y$  coordinate system)  $Z$  [m] and measured values of **penetration resistance**  $q_c$  [MPa] with respect to a depth  $z$  measured from the original ground (when calculating according to **Schmertmann local friction**  $f_s$  [kPa] must be specified). When performing **classification of soils** in this **frame** entered values of pore pressure  $u_2$  [kPa] additionally.

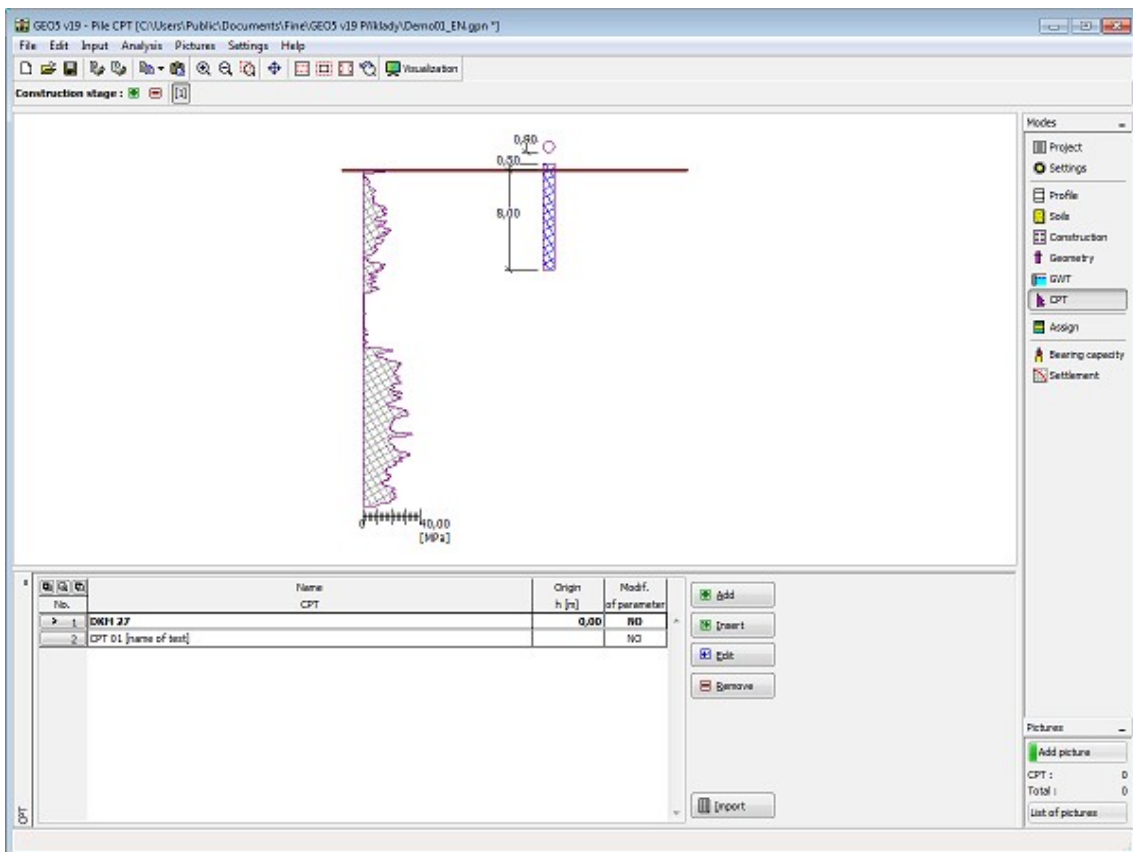
The "**Add**" button in the dialog window opens the "**New value of test**" ("**Edit value of test**") dialog window, which serves to input the measured values of penetration resistance.

Each input penetration test is automatically linked to the standard profile specified using the "**Profile**" frame.

User can change entered values in dialog window "**Edit CPT**".



Dialog window "New test"



Frame "CPT"

## Import CPT

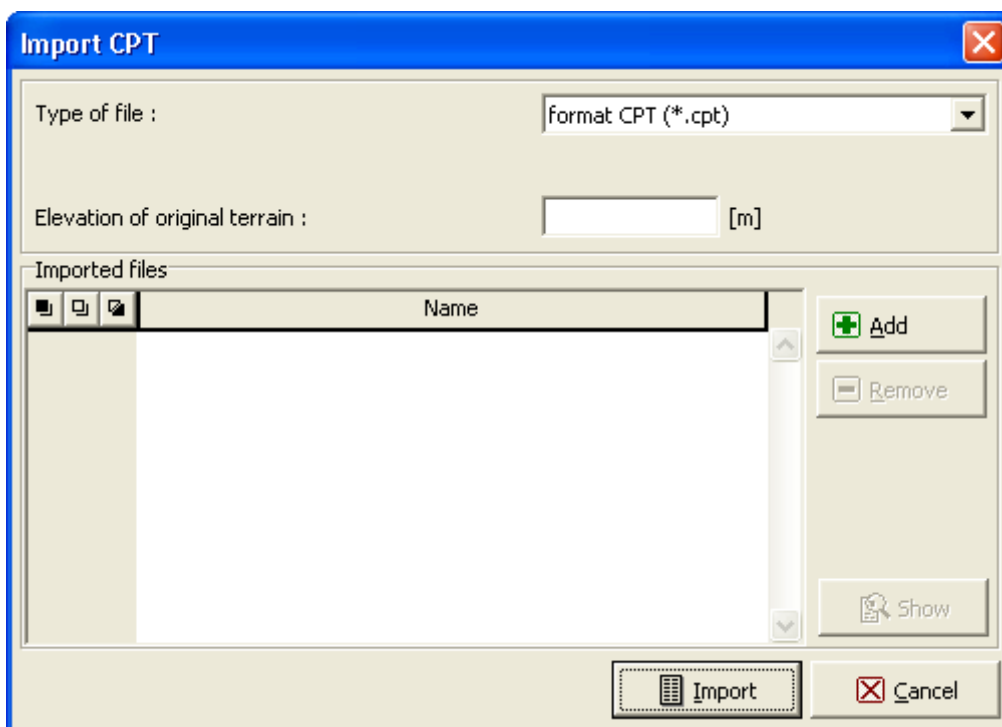
The program "**Pile CPT**" allows for importing the CPT test results in the **CPT** format (\*.cpt), **gINT** (\*.gi3) a **TXT** (\*.txt). The "**Import CPT**" dialog window contains a table with the list imported tests. The combo lists serve to select the type of file and the desired system of units.

- \*.**CPT** - a text file standard particularly for Netherlands (used e.g., in programs Geodelft M-Serie), which serves to input elevations of individual points and values of penetration resistance (may contain more CPTs)
- \*.**GI3** - a text format, which transports tests data from program gINT Software - this file contains a set of units, individual test points are specified by their depth from the origin
- \*.**TXT** - a general text format with the following definition:
  - 1st row - [test name] - string
  - 2nd row - [elevation of the test origin] - number (an empty row can be entered)
  - 3rd row - [point depth][penetration resistance] - points separated by space

A **TXT format** allows for selecting a particular **system of units** to store data of the test. When importing the program automatically converts the adopted system of units to the one used in the program.

For a correct calculation the test must be introduced into the soil body - therefore the window requires us to input an **elevation of the original ground**. The particular test is then inserted into the soil body according to its specified elevation. If no elevation is given the origin of the test is automatically placed on the **original ground**.

Providing you use a certain standard of a CPT text file not supported by the program, feel free to contact us at **hotline@fine.cz** - it will be introduced into the forthcoming version.



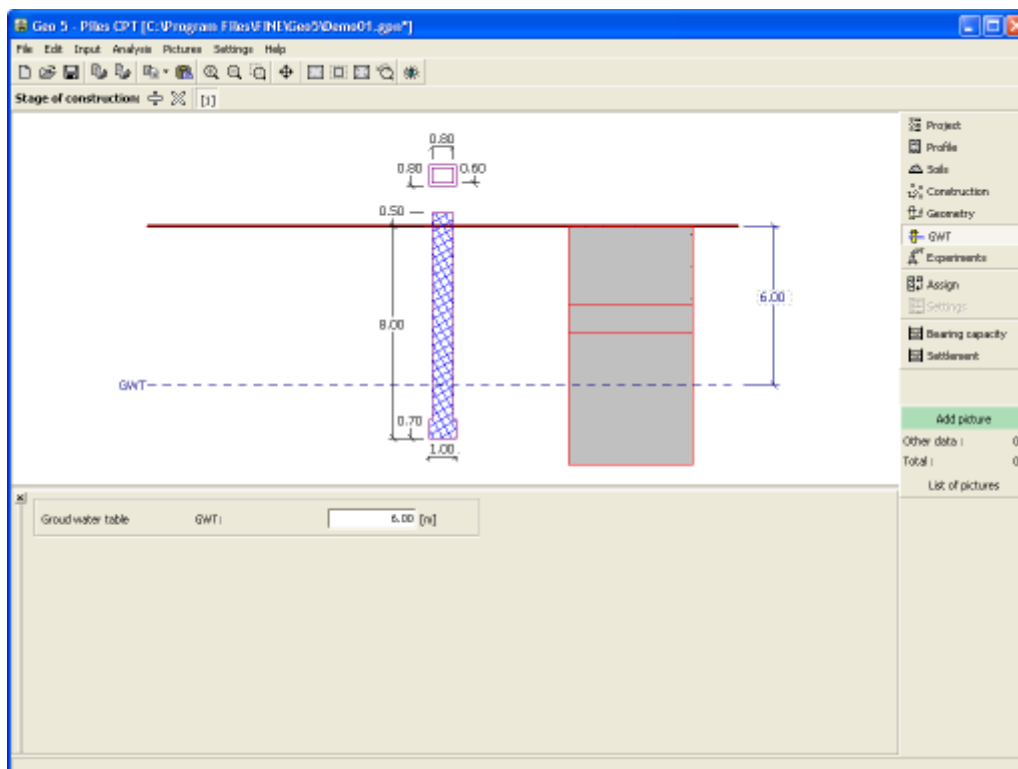
Dialog window "Import CTP"

## GWT + NSF

The frame **"GWT"** ("**GWT + NSF**") serves to specify the **depth of ground water table** and the level of **incompressible subsoil**.

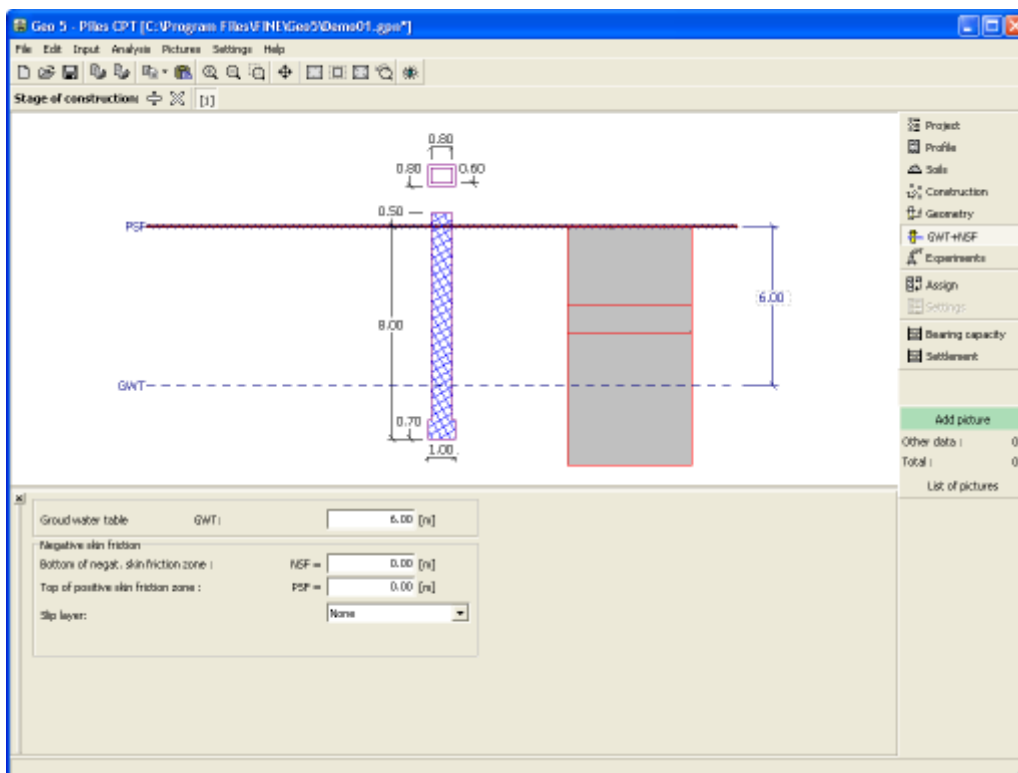
The values can be edited either in the frame by inserting values into input fields, or on the desktop with the help of **active dimensions**. The **GWT** changes the **geostatic stress** in the soil profile.

The **incompressible subsoil** cuts off the **influence zone** below foundation. It also influences a reduction of the settlement.



Frame "GWT" - without influence of NSF

If the option **"Consider influence of negative skin friction"** is set in the frame **"Settings"**, then it becomes possible to enter parameters of the negative skin friction using the **"GWT + NSF"** frame - boundaries of the region, where the influence of **negative skin friction** is considered, or sliding region and its material and cohesion.



Frame "GWT" - with influence of NSF

## Soil Classification

Classification of soils according to Robertson (1986 or 2010) allows to specify soil behavior type (SBT) and other parameters directly from the results of CPT - then input parameters of soils it's not necessary to input. We recommend to check generated soil parameters before the calculation.

In the frame "Soil classification" is selected **test for classification** (defined in the "CPT" frame). Classification of soils is performed according to Robertson (1986 or 2010).

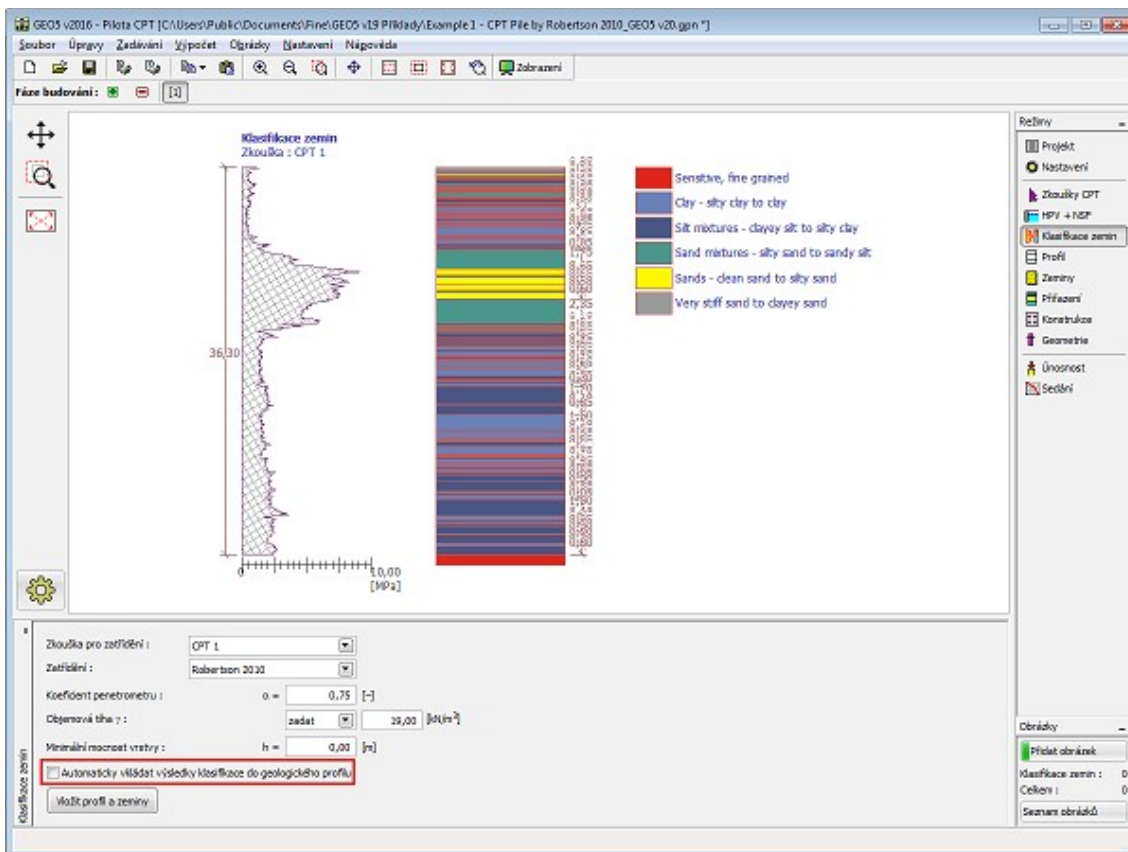
This frame serves to input **penetrometer net area ratio**  $\alpha$  [-].

The unit weight of soil  $\gamma$  [ $kN/m^3$ ] is possible to input with the same value for all layers of soils or it's **calculated** automatically from the results of CPT (for each layer separately). The frame serves to input a minimum thickness of the layer of soil  $h$  [ $m$ ]. It affects the distribution and number of individual layers of soil in the geological **profile** of solved task.

By checking the option "**Insert the results of classification into a geological profile automatically**" assigns generated geological **profile** to the current task automatically (when user change any data in the frame).

During deactivation of previous option the manual **assign** of soil is performed by pressing the "**Insert profile and soils**" button.





Frame "Soil classification"

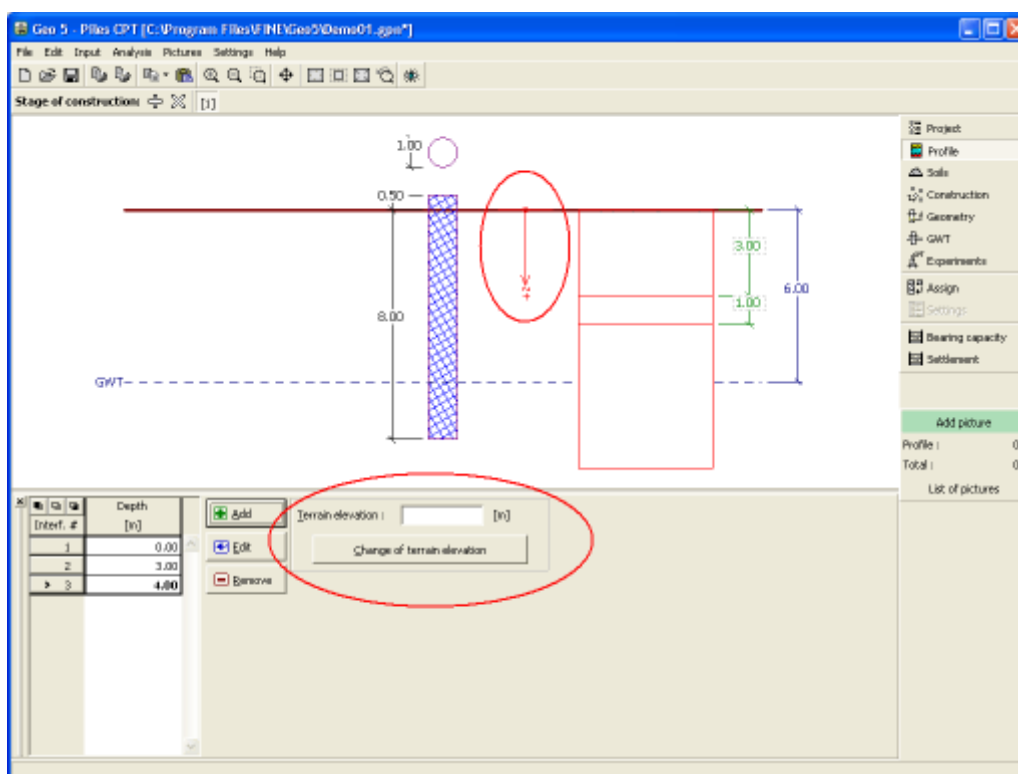
## Profile

The **frame "Profile"** contains a **table** with the list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of **active dimensions**.

**Adding** (editing) layer is performed in the **"New interface" dialog window**. The *z*-coordinate measured from the top point of a structure is specified (*z*-axis).

The program allows for raising or lowering the top point of a structure in the **"Change of terrain elevation"** dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program **"Terrain"**.

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

## Soils

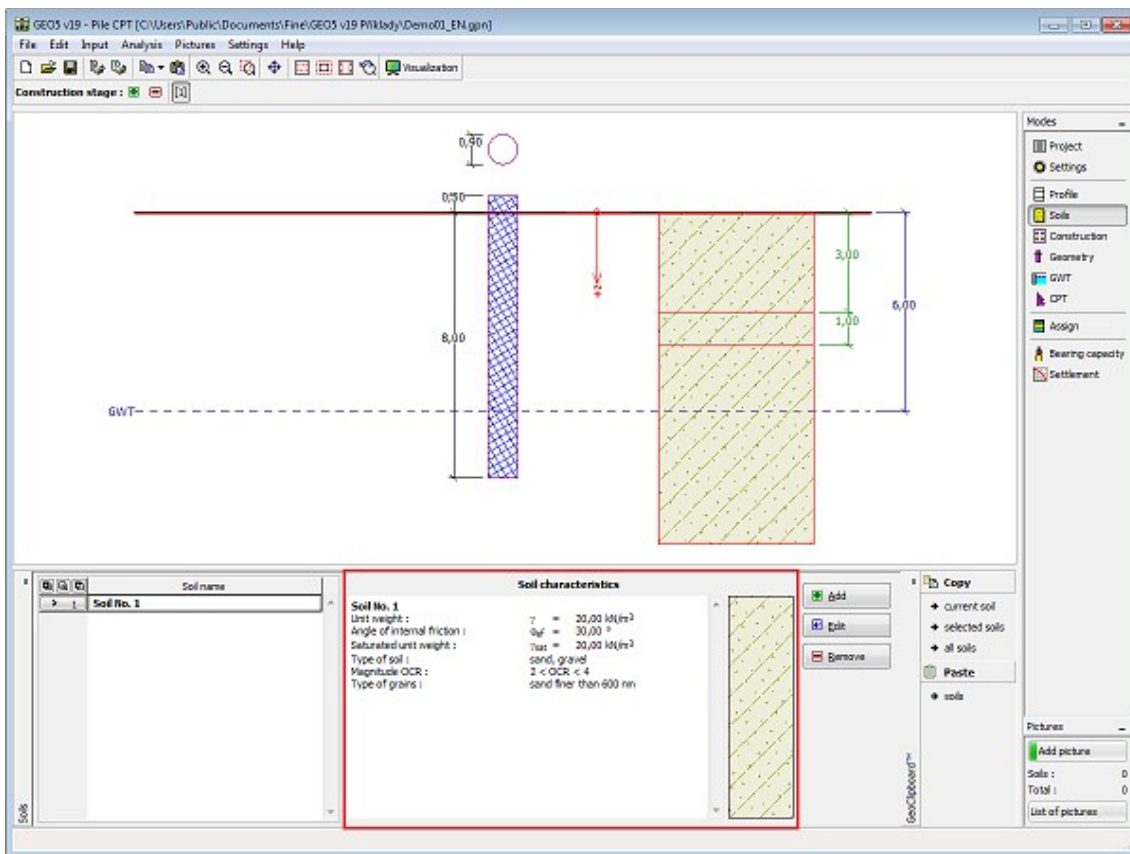
The frame **"Soils"** contains a table with the list of input soils. The table also provides information about the currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data" and "Uplift pressure".

These parameters depend on the theory of analysis specified in the "Piles CPT" tab.

The program makes it possible import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



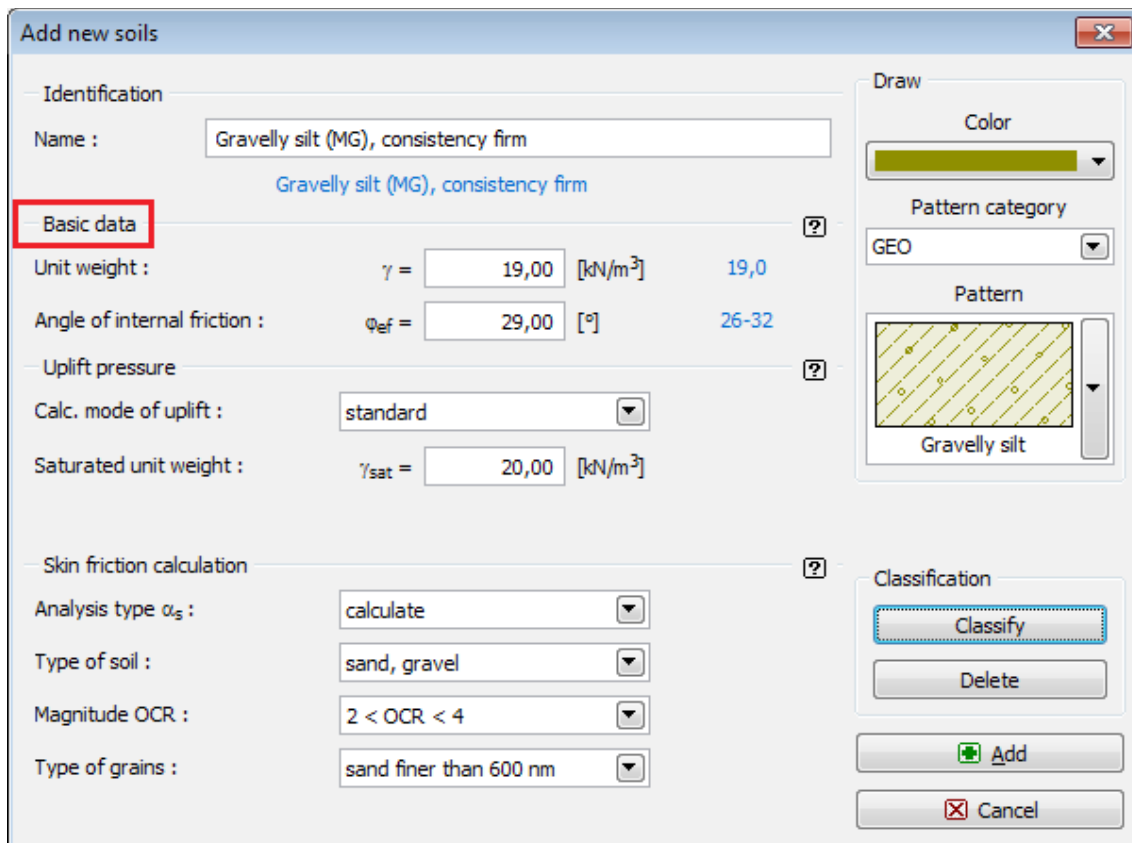
Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

The associated theory is described in detail in chapter "[Analyses in program Pile CPT](#)".

For calculation of **shaft resistance** according to the [EN 1997-2](#), [NEN 6743](#) and [LCPC \(Bustamante\)](#) further needs to specify **coefficient reducing the shaft friction**  $\alpha_s$ . For coarse-grained soils - **sands** and **gravels** further needs to specify the magnitude of **overconsolidation (OCR)** and type of grains.



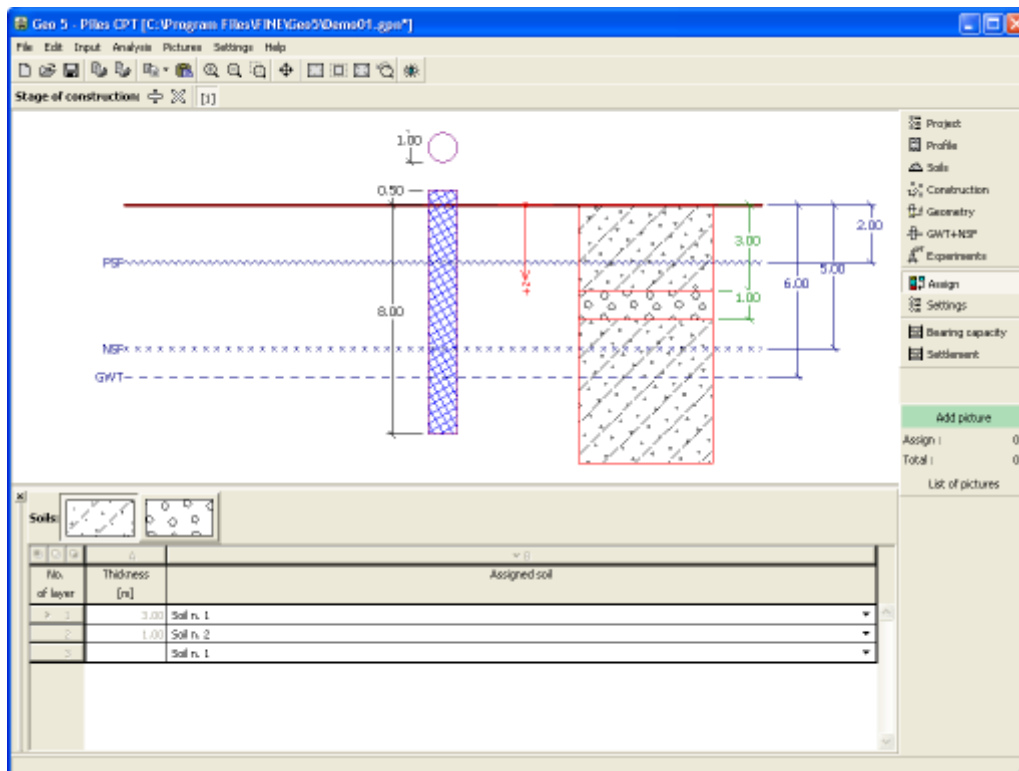
*Dialog window "Add new soils" - "Basic data"*

## Assign

The **frame "Assign"** contains the list of layers of the profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or it is accessible from a combo list associated with each layer of the profile.

The procedure to assign a soil to a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).

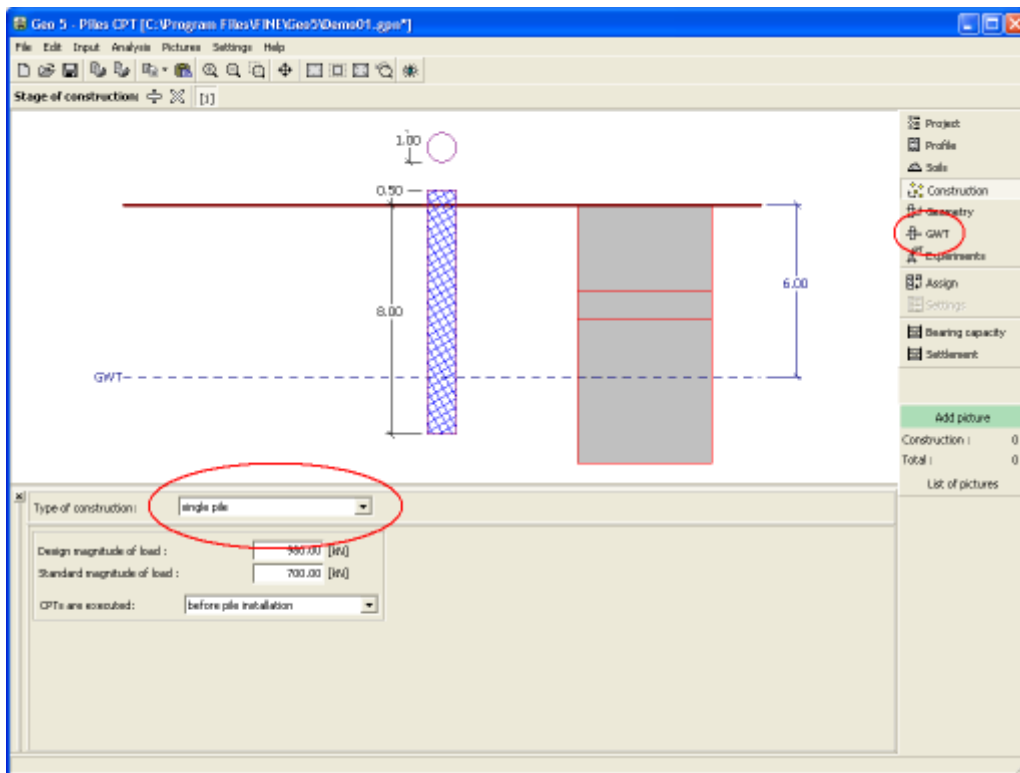


Frame "Assign"

## Construction

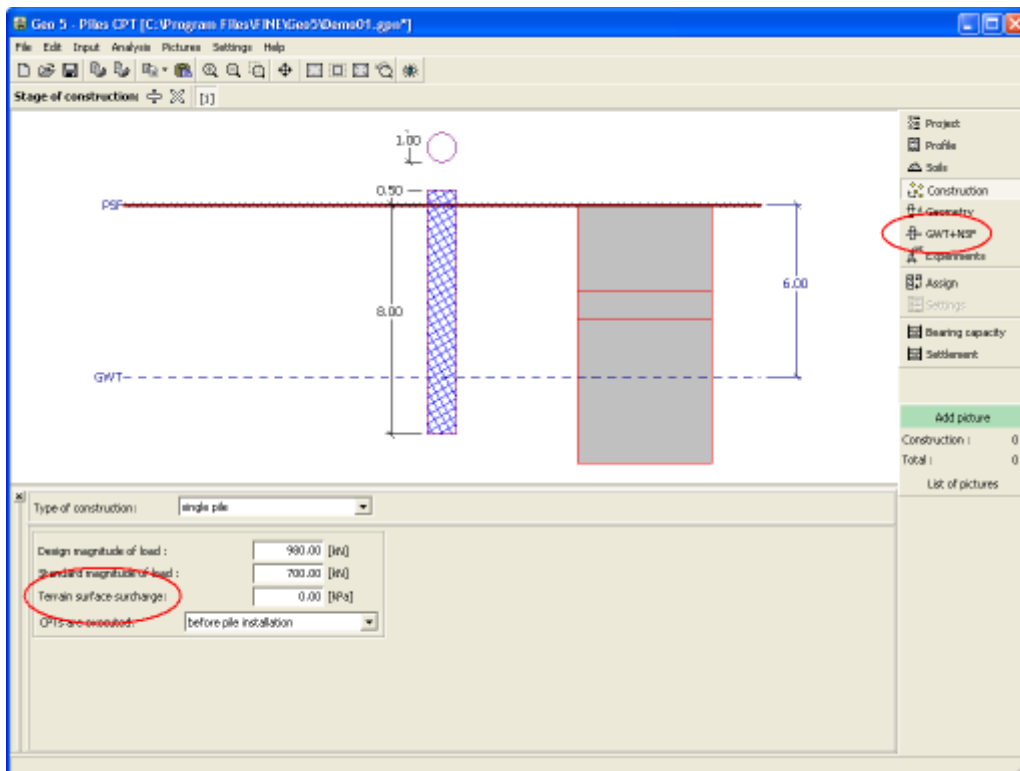
The **frame "Construction"** serves to select the type of structure - a single pile or a **group of piles**. This frame also serves to input the values of surcharge - design and standard value. The design value is used to calculate the **pile bearing capacity**, while the standard value is used to calculate the **pile settlement**, for both types of load when the NEM standard is employed (state 1B and 2).

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Structure" - single pile

If the option "**Consider influence of negative skin friction**" is set in the frame "Settings", then it is also possible to enter the **surface surcharge** using the "**GWT + NSF**" frame.

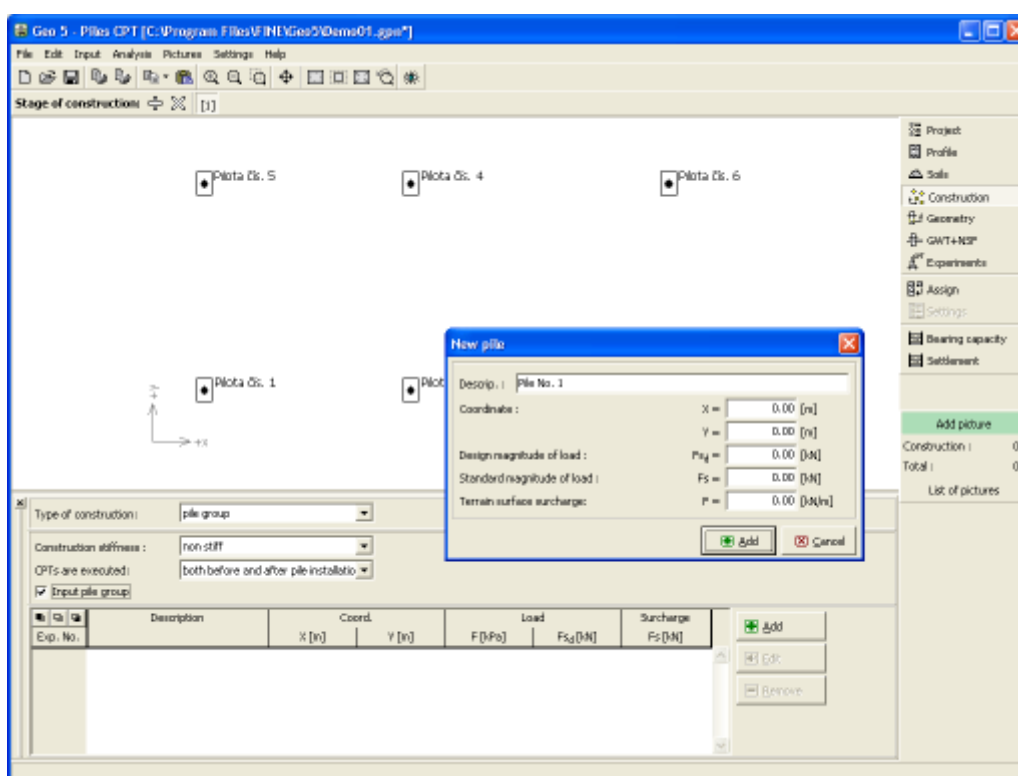


Frame Structure - single pile (influence NSF)

## Group of Piles

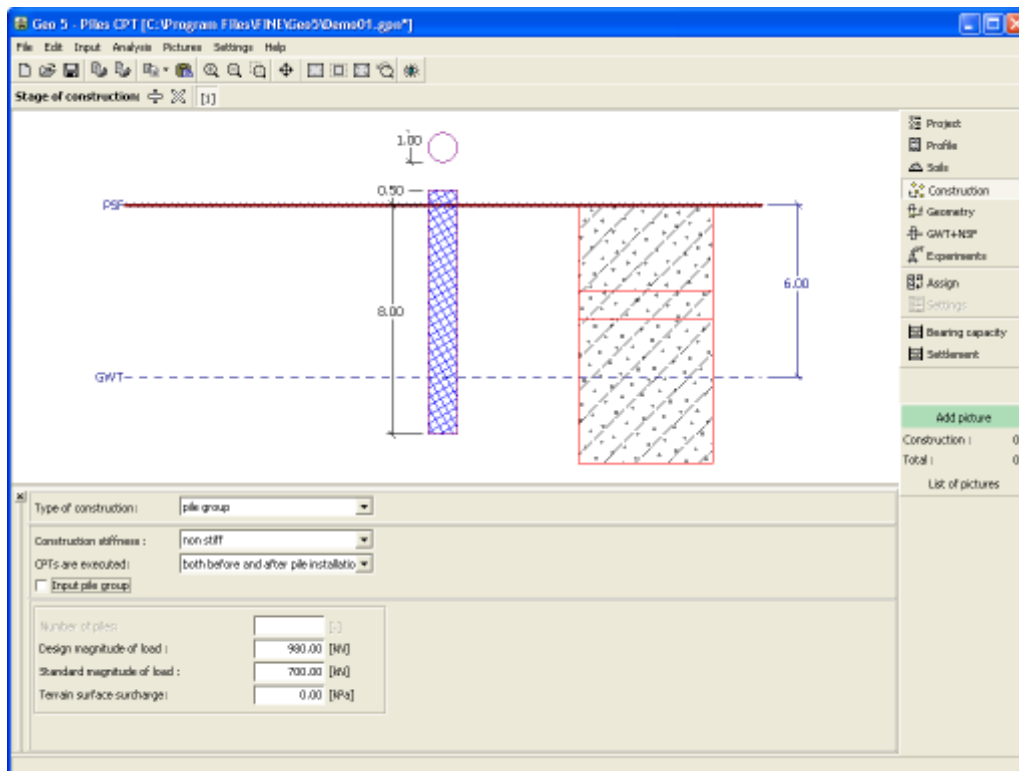
When defining a **group of piles** it is necessary to input the structure stiffness, which then drives the analysis and verification of the structures. The basic assumption is that for a stiff structure all piles experience the same settlement, while for a compliant structure each pile deforms independently. When running the analysis according to NEM6473 this frame also serves to select the way the CPT is carried out.

For both stiff and compliant structures the program allows for defining locations of individual piles using their coordinates. In such a case the coordinates of each pile are required (in the  $x, y$  coordinate system) and the load acts on each input pile. If the option **"Consider influence of negative skin friction"** is set in the frame **"Settings"**, then it is also possible to enter the **surface surcharge** using the **"GWT + NSF"** frame. Adding (editing) a new pile is performed in the **"New pile"** dialog window.



Frame "Group of piles" - entering locations of piles using their coordinates

If the user does not enter the coordinates of piles locations, then their parameters are defined directly in the frame **"Construction"**. Selecting the stiff structure allows for specifying the number of piles below the structure (the piles are then spread uniformly).



Frame "Group of piles"

## Geometry

The **frame "Geometry"** serves to input the **pile cross-section** (circular, rectangular, circular with enlargement, rectangular with enlargement) and a type of the pile (cast in place screw piles, prefabricated screw pile, continuous Flight Auger - CFA). Using input fields the cross-section dimensions are then specified for the selected cross-section.

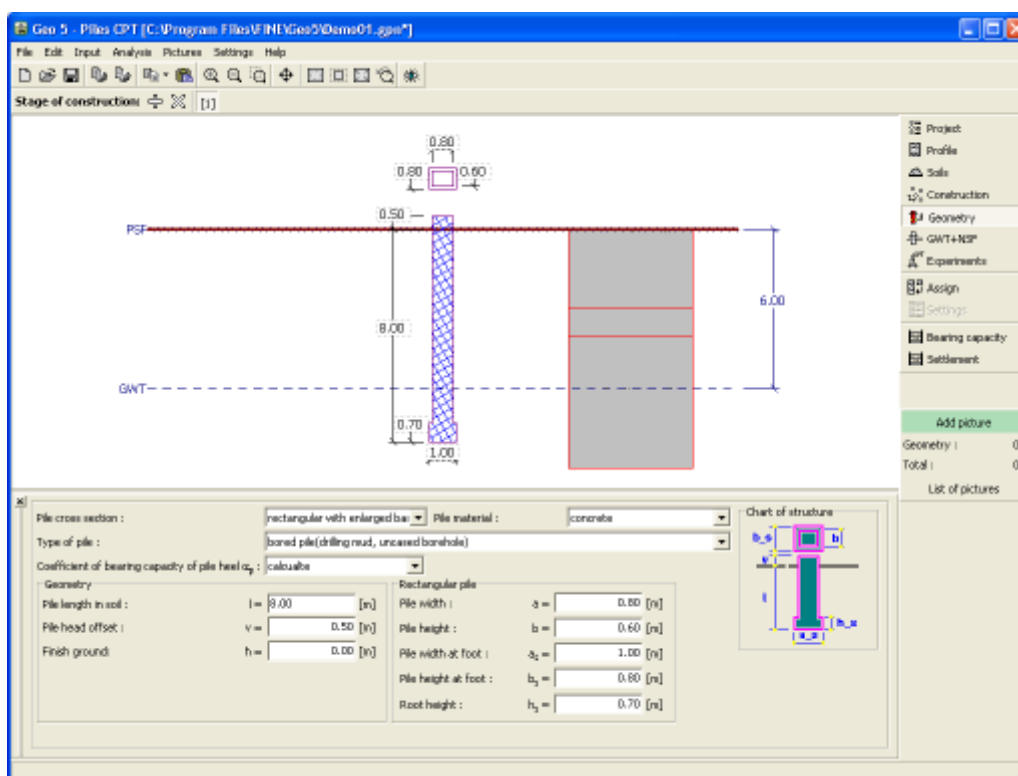
This frame also serves to input a **material of the pile** (timber, concrete, steel) a **geometry of position of the pile** (a pile length in the soil, a pile head offset and a depth of finished grade). The selected shape with a graphical hint of input values is displayed in the right part of the frame.

The **toe bearing capacity coefficient**  $\alpha_p$  is specified in the center part of the frame. This coefficient is by default automatically calculated based on the selected procedure while taking into account the type of pile and the surrounding soil.

When analyzing rectangular piles the **pile shape coefficient**  $s$  is introduced to reduce the toe bearing capacity. When analyzing piles with enlargement the expanded **pile toe coefficient**  $\beta$  is introduced to adjust the expanded toe bearing capacity.

The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



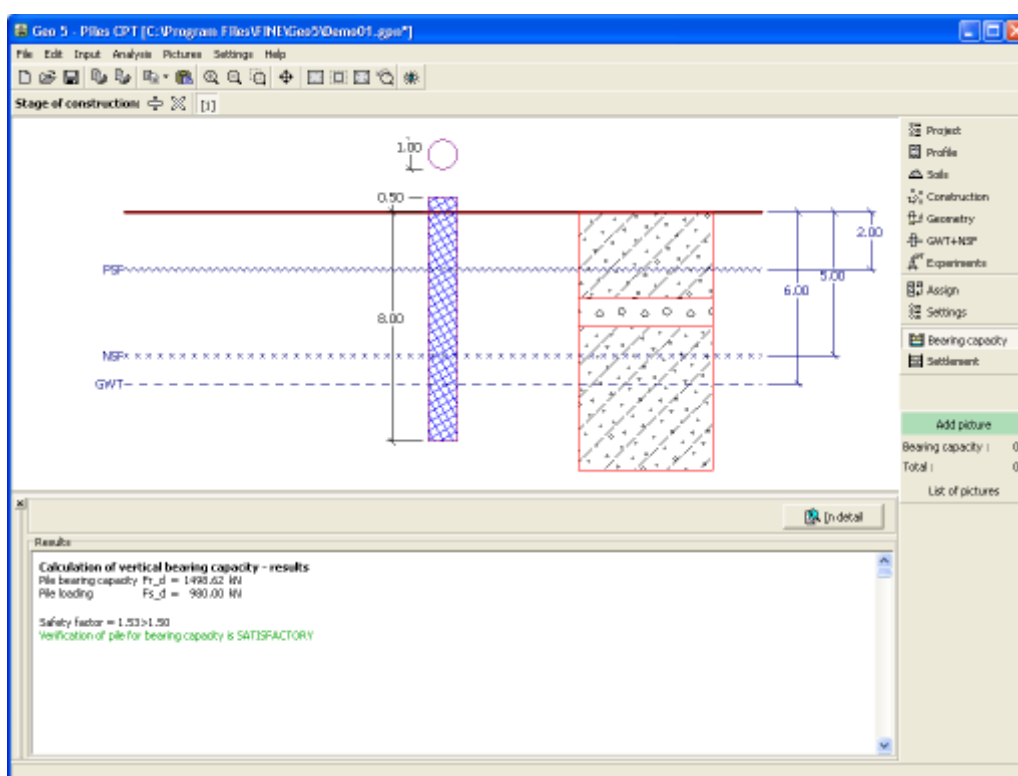


Frame "Geometry"

## Bearing Capacity

The **frame "Bearing capacity"** serves to verify the pile **vertical bearing capacity**. The **analysis results** are plotted in the right bottom part of the frame. The **"In details"** button opens the dialog window, which contains a detailed printout of results from the verification analysis.

Visualization of results can be adjusted in the frame **"Visualization settings"**.



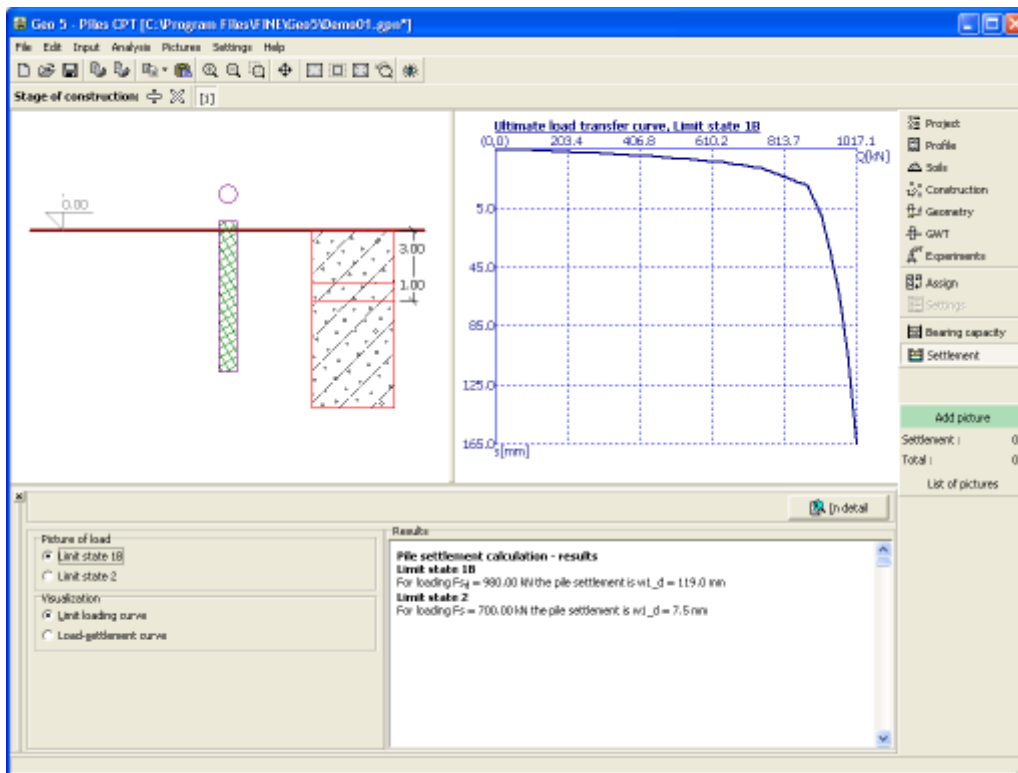
Frame "Bearing capacity"

## Settlement

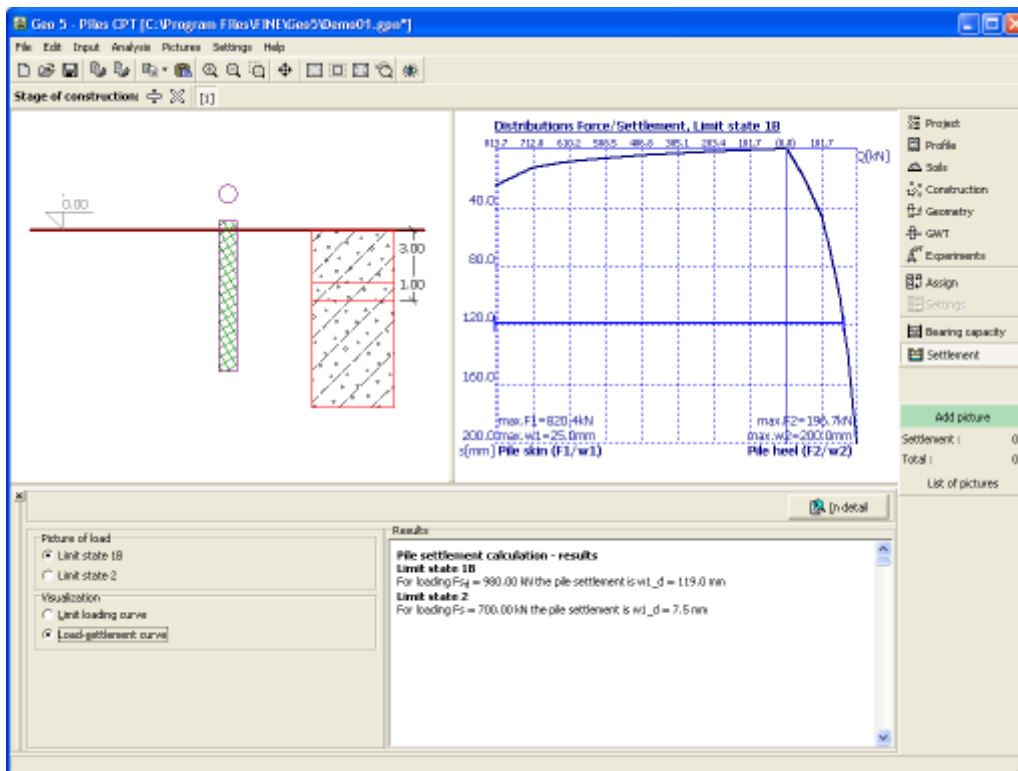
The frame "**Settlement**" serves to verify the pile settlement. The **analysis results** are plotted in the right bottom part of the frame. The "**In details**" button opens the dialog window, which contains a detailed printout of results from the verification analysis.

When calculating settlement according to the NEN 6743 standard the program plots, apart from the **load-settlement curve**, also the load diagram (**force/displacement curve**).

The analysis results are displayed in the top part of the frame. Visualization of results can be adjusted in the frame "**Visualization settings**".



Frame "Settlement" - Load-settlement curve



Frame "Settlement" - Distributions force/settlement

## Program Pile Group

This program is used to analyze a pile group (pile raft foundation with a rigid pile cap) using both spring method (FEM), or analytical solutions. Both floating piles and piles fixed into subsoil can be considered.

The help in the program "Pile Group" includes the following topics:

- Input of data into individual frames:

Project	Settings	Structure	Geometry	Material	Load	Profile
Soils	Assign	Water	Negative Skin Friction	Vertical Springs	Horizontal Modulus	Stage Settings
Vertical Bearing Capacity	Settlement (Cohesive Soil)	Settlement (Cohesionless Soil)	Analysis (Spring Method)	Dimensioning		

- Standards and analysis methods

- Theory for analysis in the program "Pile Group":

Stress in Soil Body	Pile Group	Dimensioning of Concrete Structures
---------------------	------------	-------------------------------------

- Outputs

- General information about the work in the User Environment of GEO5 programs

- Common input for all programs

## Project

The frame "Project" is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in text and graphical outputs.

The frame also allows user to switch analysis units (metric / imperial). Project data can be copied within all GEO5 programs using "GeoClipboard".

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Materials and standards"** and **"Pile Group"** tabs.

The right part of the frame allows to select the type of analysis - **analytical solution** or **spring method**.

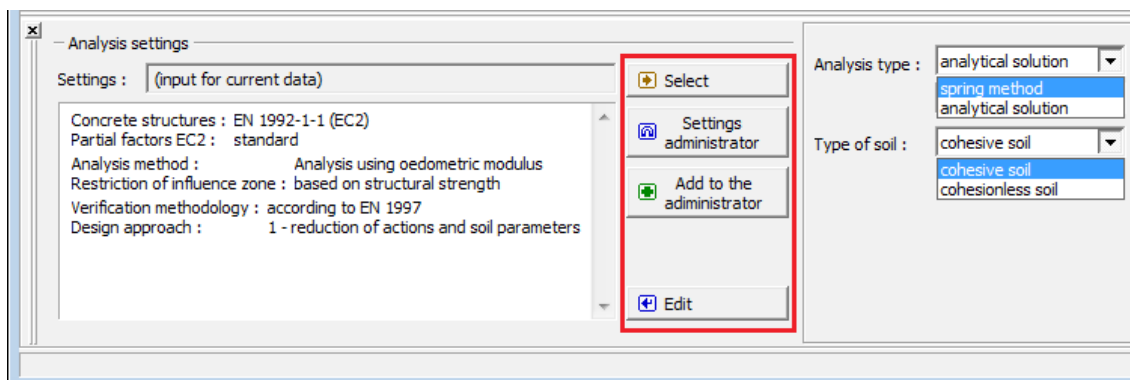
**The analytical solution requires defining the type of subsoil:**

- **cohesionless soil** (analysis for drained conditions)
- **cohesive soil** (analysis for undrained conditions)

**The spring method requires input of:**

- **type of pile** (pile acts vertically)

- connection piles / pile cap
- modulus of subsoil reaction (pile acts horizontally)



Frame "Settings"

## Structure

The frame "Structure" allows to input the **dimensions of the pile cap** according to the defined scheme, **number of piles**, their **diameter** and **spacing**.

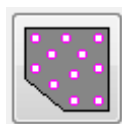


Defined pile group shapes

Individual piles in the group share the same diameter. A correct and reliable design of a pile group requires meeting the construction rules regarding:

- the number of piles in a group (3 - 20)
- the diameter of piles (from 0,3 m to 4,0 m)
- the spacing of piles ( $s = 1,5d$  to  $6d$ ), where  $d$  is the diameter of individual piles in the group
- the overhang of a pile cap from the obverse of outer piles ( $o = 0$  to  $2d$ )

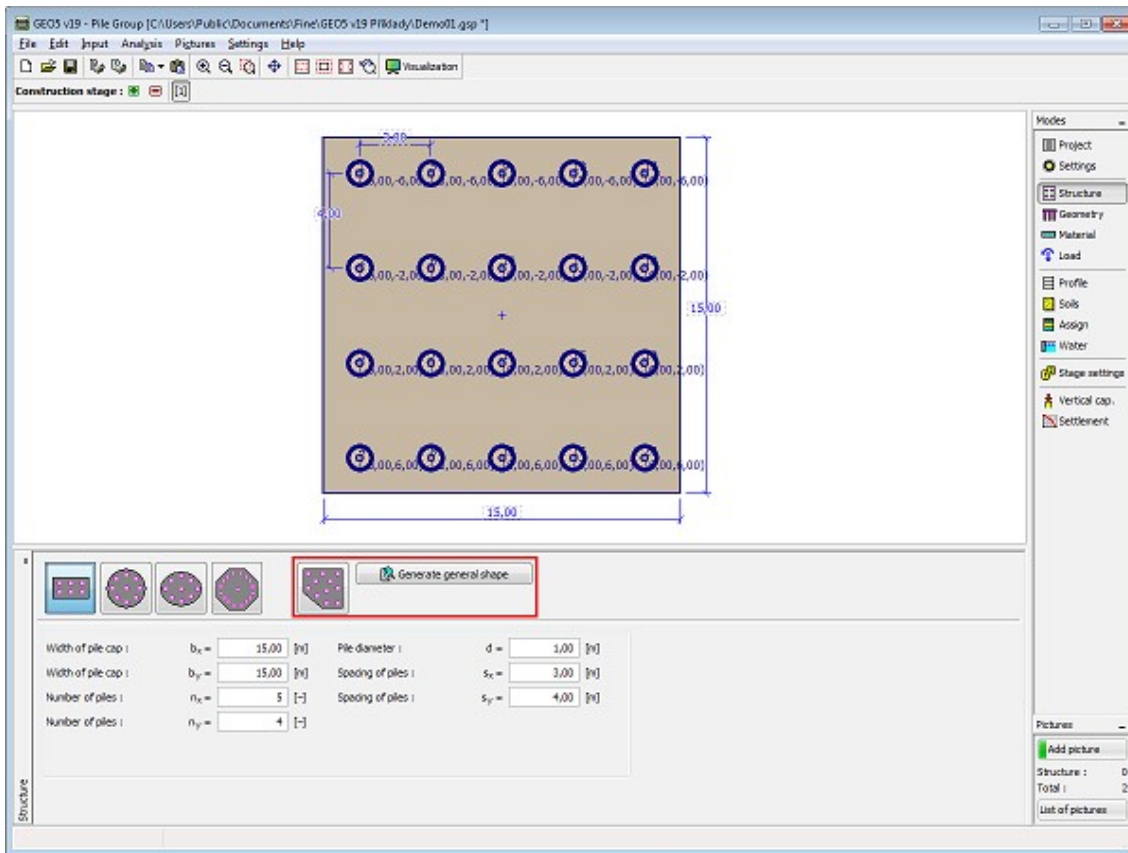
If defined pile cap shapes are not satisfactory for the pile group geometry input, program allows to enter **general shape of pile group**. General shape of pile cap is entered by coordinates of points, but it is also possible (by pressing the button "**Generate general shape**") to generate coordinates of structure from already input predefined pile cap.



Input of general pile cap shape

If the **general shape** of a pile cap is selected, it is possible to input inclination of piles  $\alpha$ . This option ( $\alpha \neq 0^\circ$ ) is available only for **spring method**, not for analytical solution. For **analytical solution** all piles are considered as vertical ( $\alpha = 0^\circ$ ). So, for calculation of **vertical bearing capacity** or **settlement of pile group** an inclination of piles is not considered.

The program allows to **export** the geometry of the structure in \*.DXF format.



Frame "Structure"

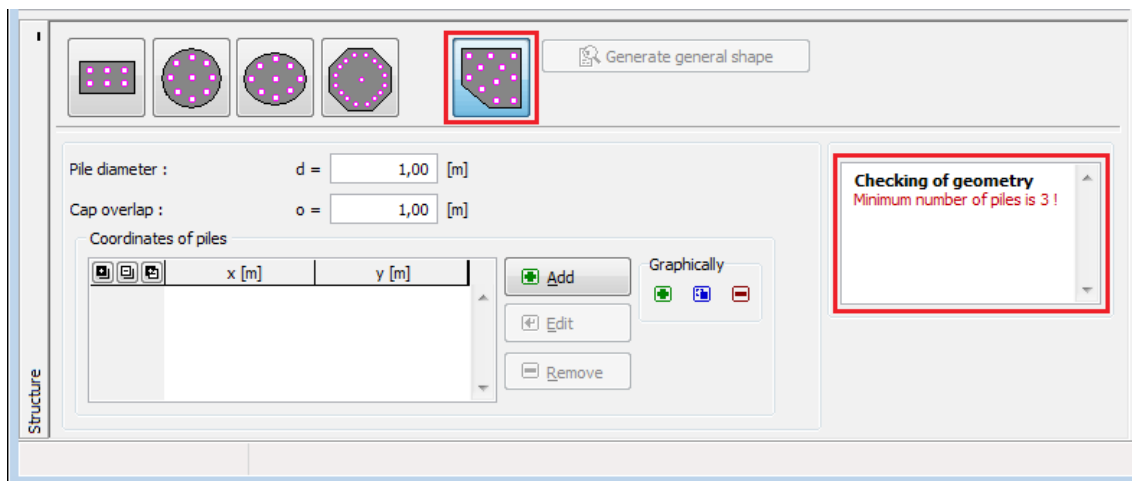
## General Pile Group Shape

### Input of general pile group shape in a new task

The program allows to input general pile group shape in two ways:

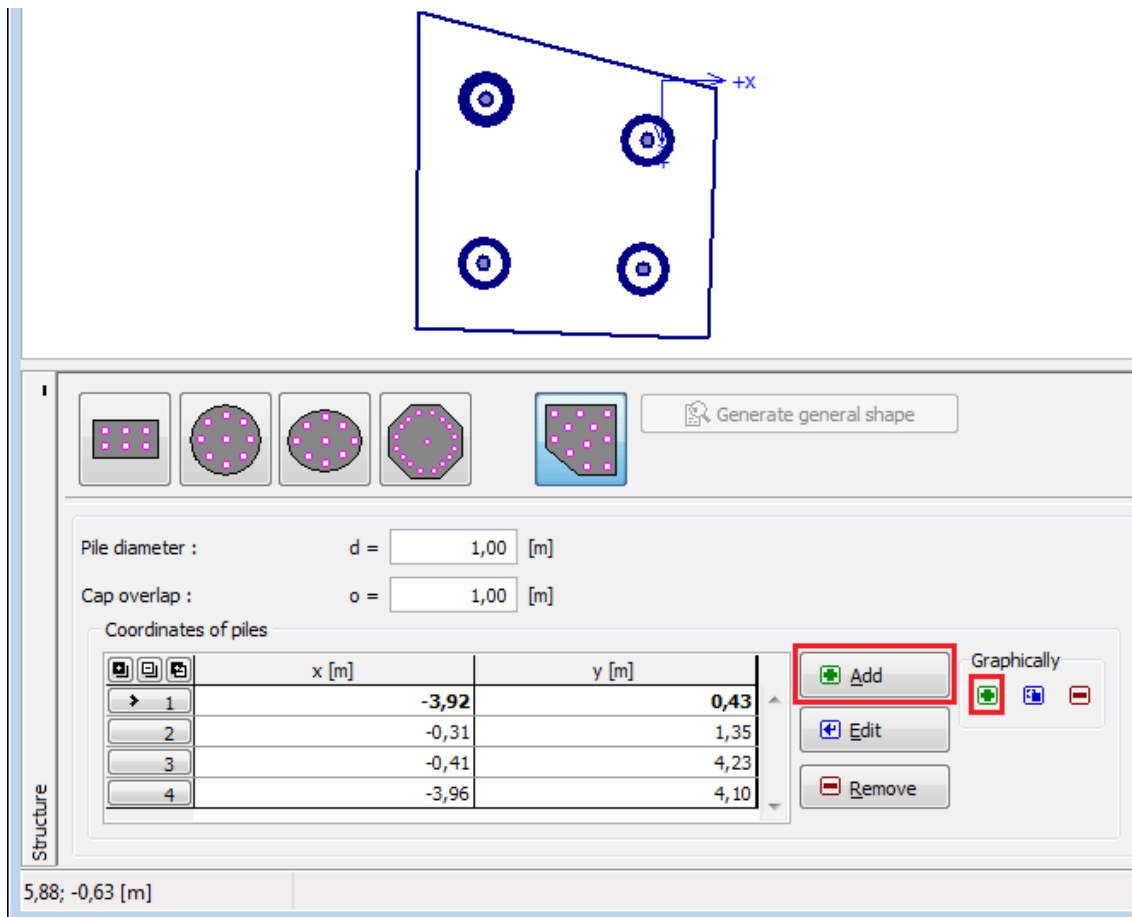
#### 1. Input of general pile group shape using points

By pressing the icon for creating a general pile group shape on the tool bar, the program will delete the desktop. Minimum input number of piles is 3 (in case less number is input, the program will show an error message).




Frame "Structure" - new task

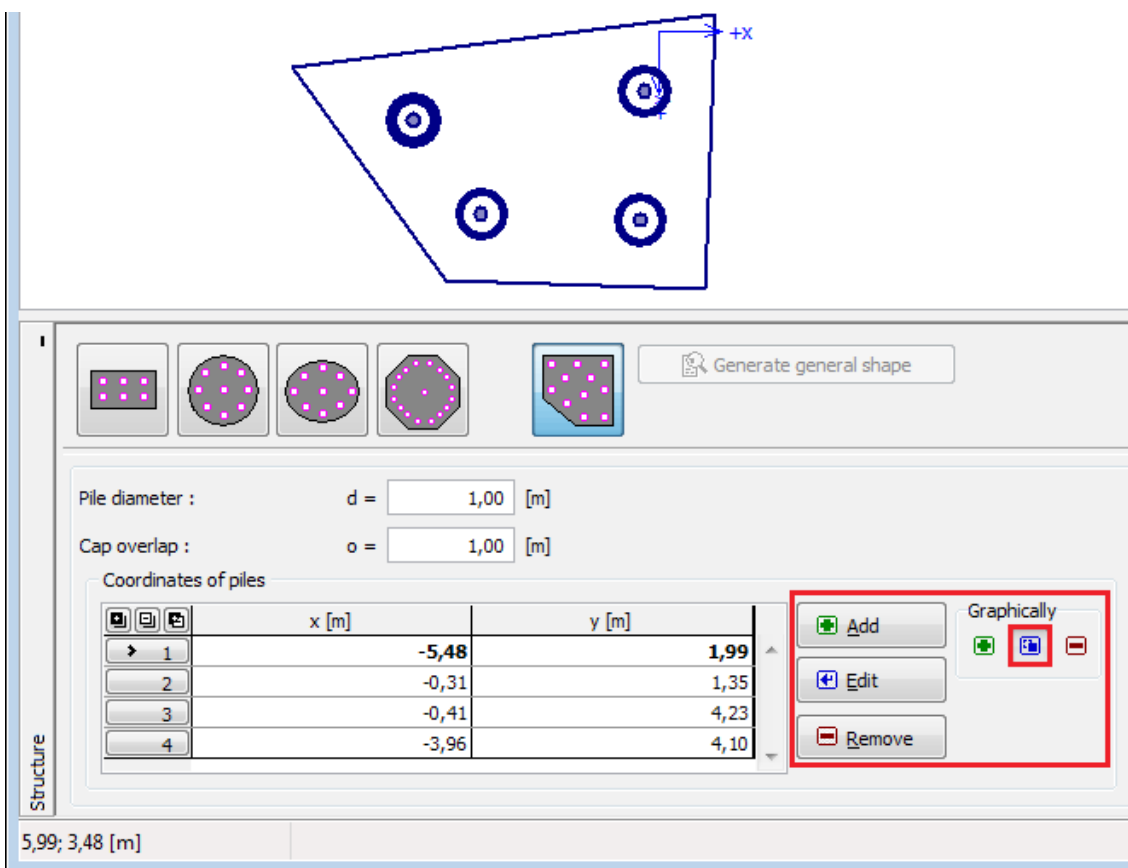
Using the **"Add"** button, which opens a dialog window **"New point"**, pile center points are input (it is possible to input points by clicking on the desktop).



Frame "Structure" - input of general pile group shape using points

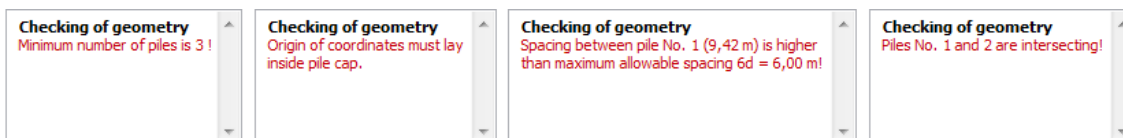
Input points are being added to the table, and it is then possible to edit them, and delete them using the desired buttons **"Edit"** and **"Remove"** or by clicking the points on the desktop in the corresponding mode. Points can be moved right on the desktop by mouse after clicking on the special icon .





Frame "Structure" - edit points

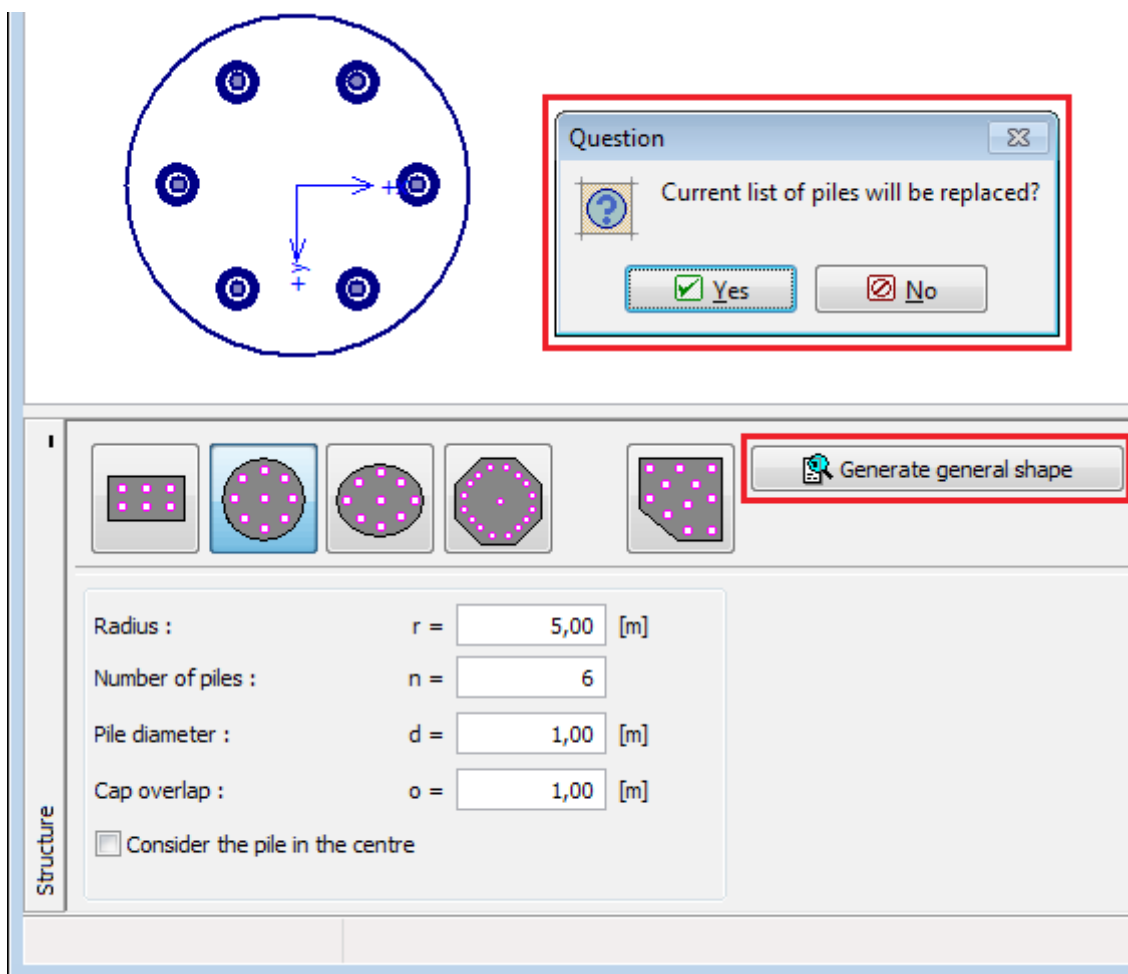
The minimum number of piles in a group is 3. In case of incorrect input (contravention of maximum allowable spacing of piles, intersecting piles) the program checks the geometry and warns the user of an error. In that case, it is necessary to change the location of the piles.



Frame "Structure" - error messages of general pile group shape input

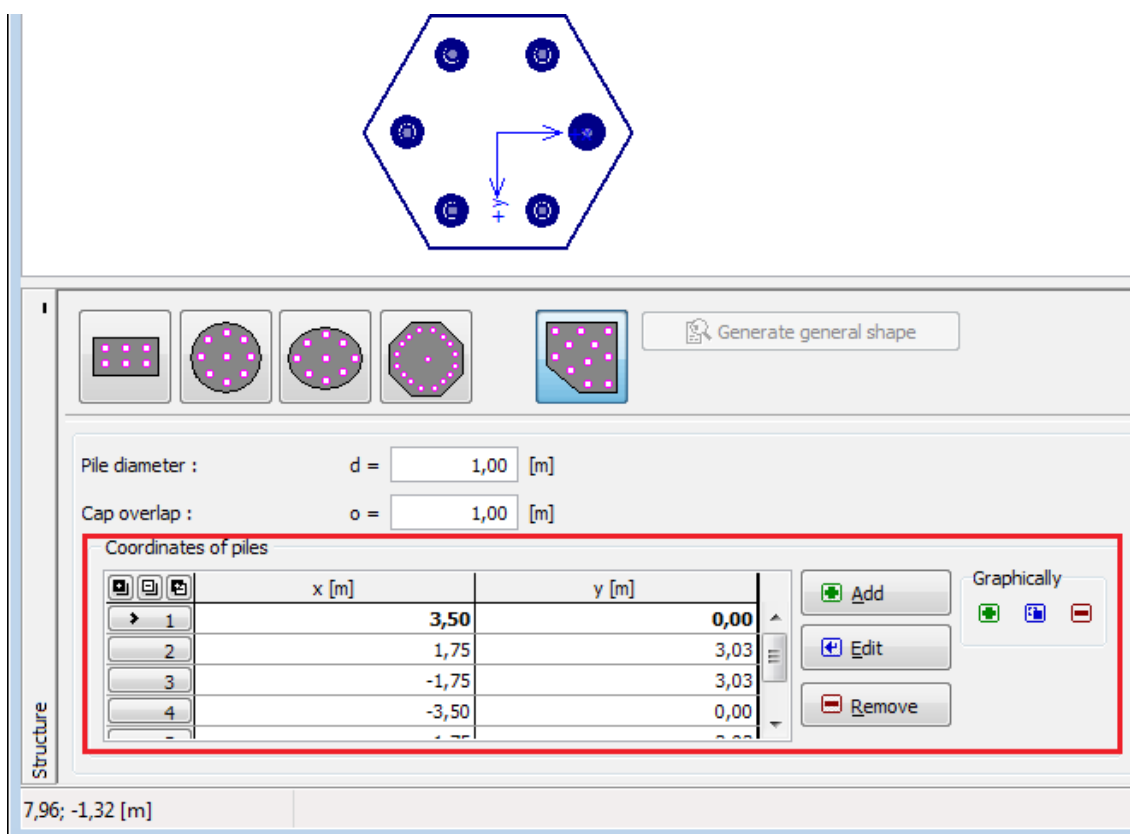
## 2. Input of general pile group shape using general shape generator

The structure defined by the scheme of construction and its dimensions can be taken to the general pile group shape input by pressing the "**Generate general shape**" button. It is possible to work with the newly generated points and edit the generated pile cap shape.



Frame "Structure" - input of pile group shape using general shape generator

Frame appearance is then changed as in the first case of general pile group shape input. It is possible to work with the picture of the structure as already described.

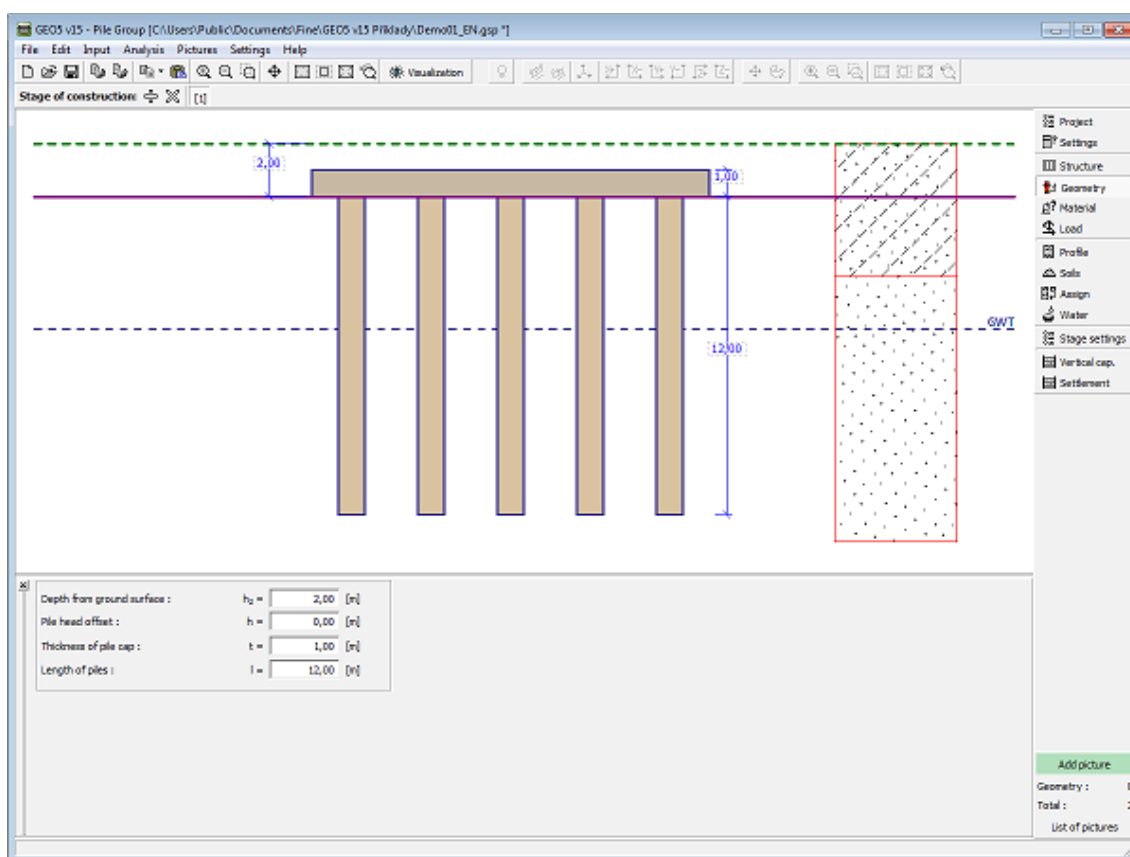


Frame "Structure" - frame appearance after point input

## Geometry

The frame "Geometry" allows to input the **depth from ground surface**, **pile head offset**, **thickness of pile cap** and **length of piles**.

Individual piles in the group are of the same length.



Frame "Geometry"

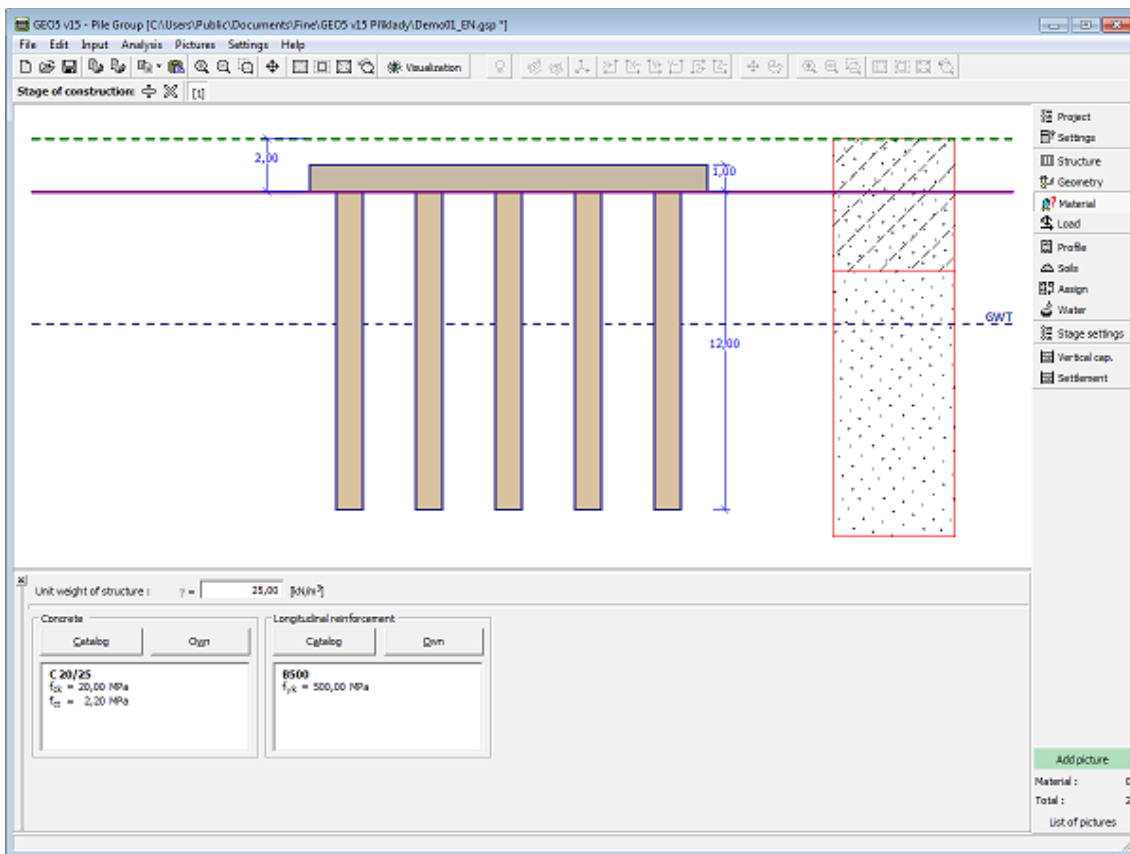
## Material

The **frame "Material"** allows to select material parameters for concrete and longitudinal steel reinforcements.

Two options are available when selecting the material type:

- The **"Catalog"** button opens the **"Catalog of materials"** dialog window (for concrete and steel reinforcements), the list of materials then allows to select the required material.
- The **"Own"** button opens the **"Editor of material - Concrete"** dialog window (for concrete) or the **"Editor of material - Reinforcing steel bars"** dialog window (for longitudinal steel reinforcements), which allows to input the specification of material parameters manually by user.

The content of catalogs depends on the selection of relevant standard for the **dimensioning of concrete structures** set in the **"Materials and standards"** tab. The input field in the upper part of the frame allows to specify the unit weight of structure.



Frame "Material"

## Load

The frame "Load" contains a table with the list of input loads. Adding (editing) a load is performed in the "New (edit) load" dialog window. The forces are input according to the sign convention displayed in the right part of the dialog window.

The program also allows to import a load using the "Import" button.

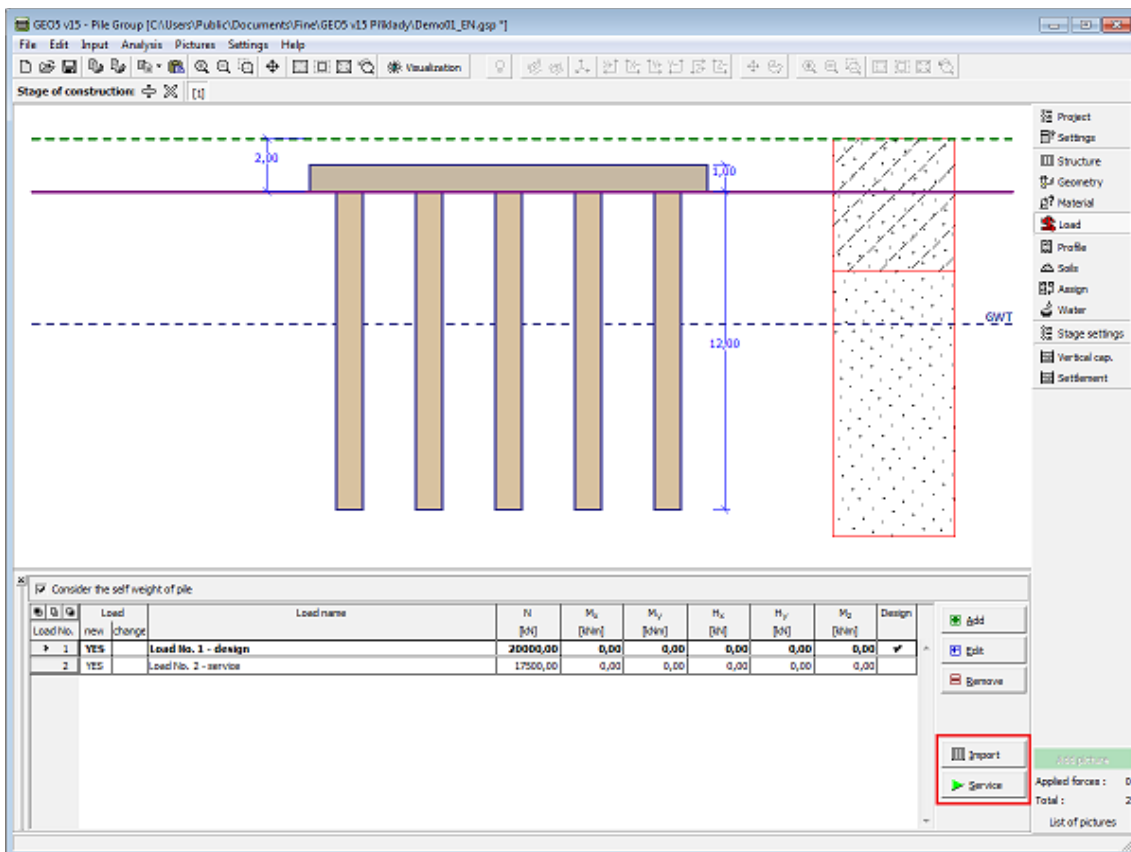
The load applied to a pile group acts at the level of the pile cap upper base at point [0,0]. This point cannot be located outside the pile cap. These values can be easily obtained from the analysis by an arbitrary program that performs static analysis.

The program automatically back-calculates the pile cap self weight to be added to the already existing load. This program further enables to consider the self weight of pile (using the button in the left part of the desktop)

The pile cap self weight  $G_{cap}$  is provided by:

$$G_{cap} = A_{cap} t \gamma$$

- where:
- $A_{cap}$  – base area of pile cap [ $m^2$ ]
  - $t$  – thickness of pile cap [ $m$ ]
  - $\gamma$  – unit weight of structure [ $kN/m^3$ ]



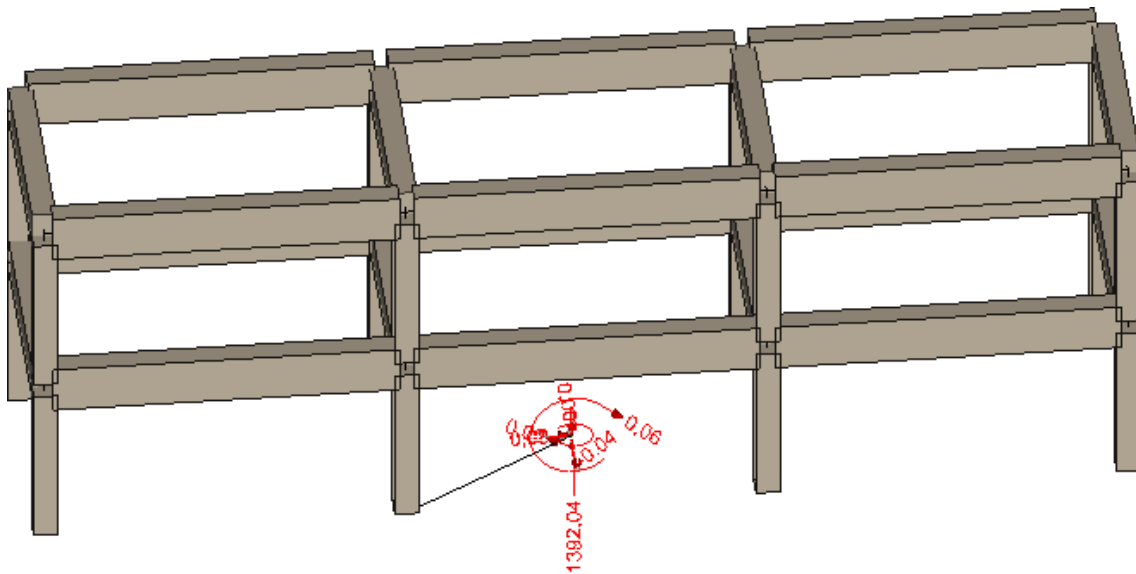
Frame "Load"

## Load Acting on a Pile Group

The pile group can be used to found both bridge abutments and arbitrary civil engineering structures. To determine the load acting on a pile group can be rather complicated. The load can be applied at several locations as concentrated load (column), distributed along the line (wall) or over the area. The procedure shows a simple way of determining the load at a given point by adopting an arbitrary static program.

1. We start from the model of a structure in the static program
2. Providing no joint is defined in the **center of the pile cap**, we introduce it.
3. A **fixed support in all 6 directions** is assigned to the joint (fixed, fixed, fixed, fixed, fixed, fixed)
4. If the joint is not found on the pile cap (beam model), we connect it with the actual structure (The stiffness should correspond to other elements)
5. Apart from the new joint we remove **all boundary conditions** on the analyzed model
6. Perform analysis - the **joint reactions correspond to the load**, which is input into the **"Pile group"** program - the function for importing the load can also be utilized

**Note:** **internal hinges** found in the structure must be changed to fixed-end supports for the static program to find a solution.



Structure with a fixed support

Load - NotePad

G1+G2	0,00	0,00	1879,25	-0,05	0,08	0,00	
W4 :G1+G2	0,00	-162,00	1879,25	728,95	0,08	0,00	
Q3 :G1+G2	0,00	0,00	3499,25	1079,95	0,08	0,00	
Q3 :G1+G2+W4	0,00	-97,20	3499,25	1517,35	0,08	0,00	
W4 :G1+G2+Q3	0,00	-162,00	3013,25	1484,95	0,08	0,00	
G1+G2	0,00	0,00	1392,04	-0,04	0,06	0,00	
W4 :G1+G2	0,00	-108,00	1392,04	485,96	0,06	0,00	
Q3 :G1+G2	0,00	0,00	2472,04	719,96	0,06	0,00	
Q3 :G1+G2+W4	0,00	-64,80	2472,04	1011,56	0,06	0,00	
W4 :G1+G2+Q3	0,00	-108,00	2148,04	989,96	0,06	0,00	

Support reactions ready for import into the "Pile group" program

Consider the self weight of pile

Load No.	new	change	Load name	N [kN]	M <sub>x</sub> [kNm]	M <sub>y</sub> [kNm]	H <sub>x</sub> [kN]	H <sub>y</sub> [kN]	M <sub>z</sub> [kNm]	Design
1	YES		2_W4:G1+G2 (2)	0,00	-162,00	1879,25	728,95	0,08	0,00	✓
2	YES		3_Q3:G1+G2 (3)	0,00	0,00	3499,25	1079,95	0,08	0,00	✓
3	YES		4_Q3:G1+G2+W4 (4)	0,00	-97,20	3499,25	1517,35	0,08	0,00	✓
4	YES		5_W4:G1+G2+Q3 (5)	0,00	-162,00	3013,25	1484,95	0,08	0,00	✓
5	YES		1_G1+G2 (6)	0,00	0,00	1392,04	-0,04	0,06	0,00	✓
6	YES		2_W4:G1+G2 (7)	0,00	-108,00	1392,04	485,96	0,06	0,00	✓
7	YES		3_Q3:G1+G2 (8)	0,00	0,00	2472,04	719,96	0,06	0,00	✓
8	YES		4_Q3:G1+G2+W4 (9)	0,00	-64,80	2472,04	1011,56	0,06	0,00	✓
9	YES		5_W4:G1+G2+Q3 (10)	0,00	-108,00	2148,04	989,96	0,06	0,00	✓

Buttons: Add, Edit, Remove, Import, Service

Load imported into "Pile group" in the frame "Load"

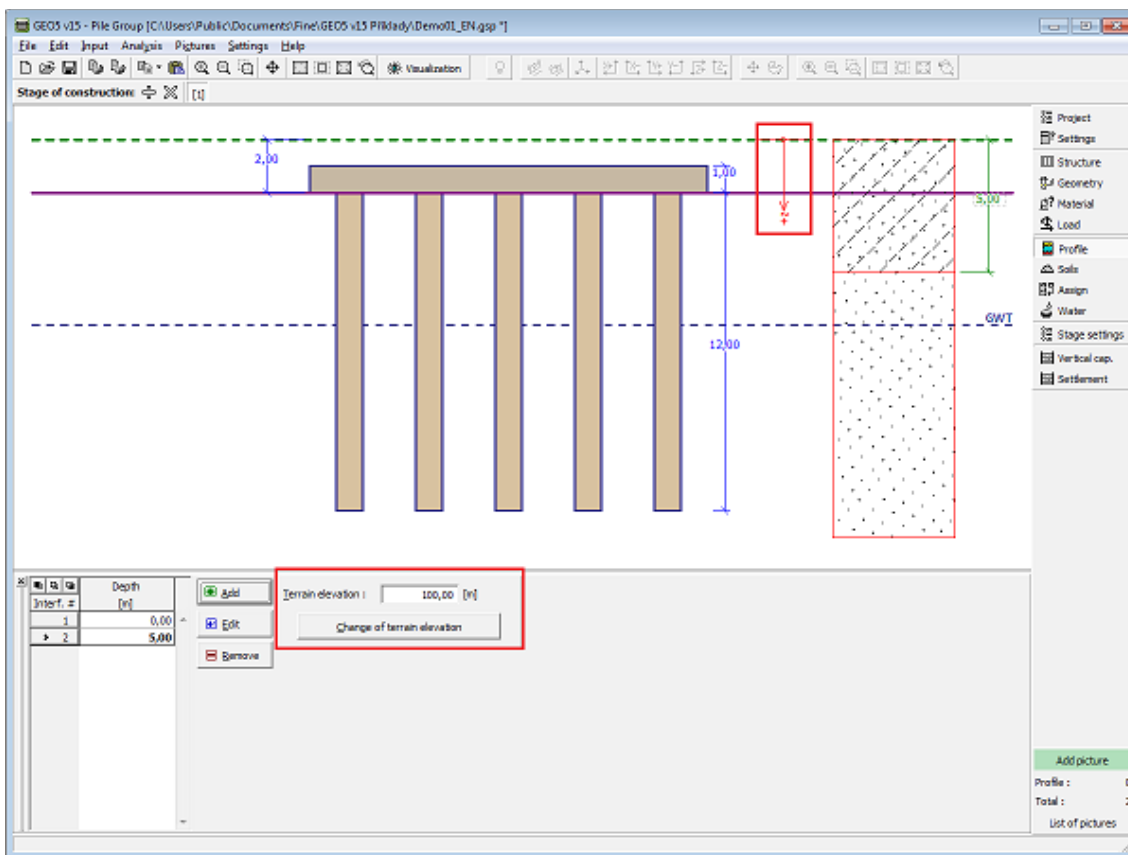
## Profile

The "**Profile**" frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of **active dimensions**.

**Adding** (editing) layer is performed in the "**New interface**" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows for raising or lowering the top point of a structure in the "**Change of terrain elevation**" dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program "**Terrain**".

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

## Soils

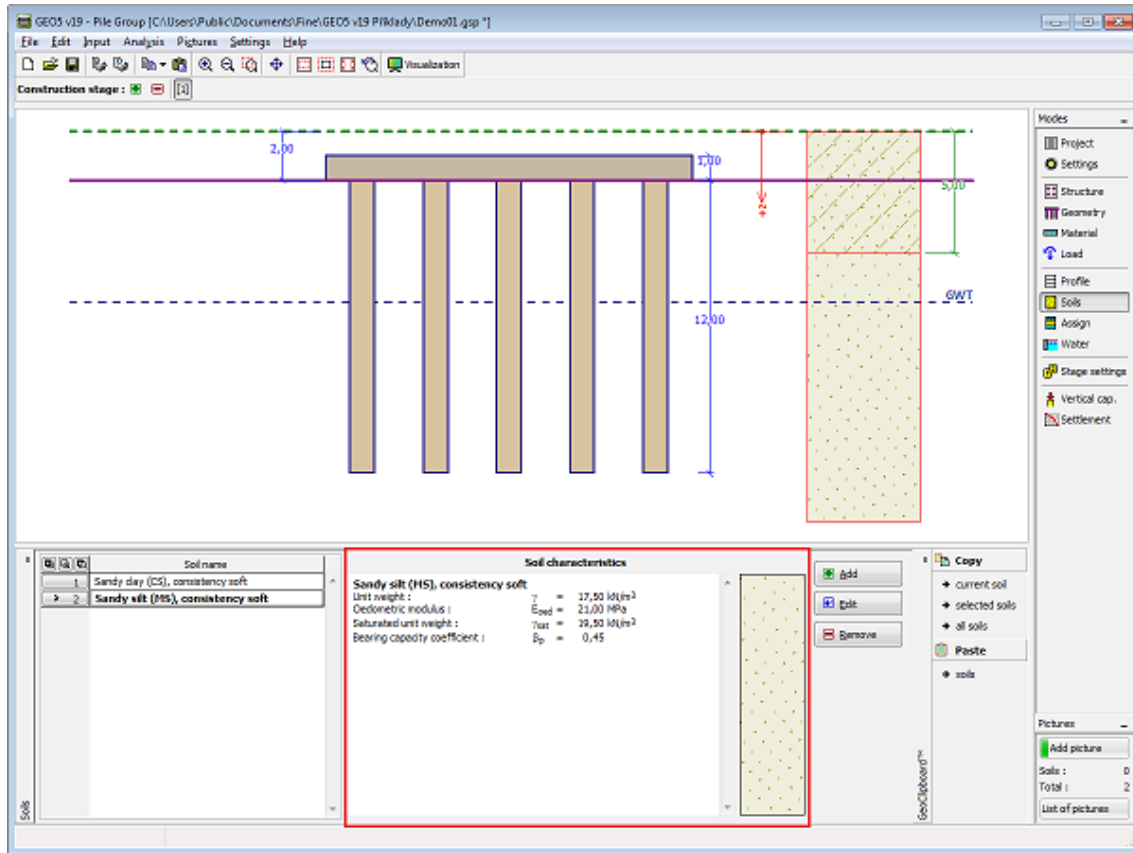
The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

**Adding** (editing) a soil is performed in the "**Add new soils**" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "**Basic data**", "**Uplift pressure**", "**Settlement**", "**Modulus of subsoil reaction**". These parameters depend on the type of soil specified in the frame "**Settings**" and the theory of analysis specified in the "**Pile Group**" tab.



The program makes it possible to **import soils in the gINT format**. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit the built-in **database of soils**, which contains values of the selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the help [herein](#).

The calculation of "Pile Group" differs according to the type of subsoil:

- **cohesionless soil: effective** parameters of shear strength of soil  $c_{ef}$ ,  $\varphi_{ef}$  are used commonly.
- **cohesive soil:** in program is defined only the value of **total** shear strength of soil  $c_u$ , which determines vertical bearing capacity of pile group (or earth block).

Additional parameters are input depending on the settings in the frame "Settings" and in the "Pile Group" tab.

The associated theory is described in detail in chapter "Pile Group".

The screenshot shows the 'Add new soils' dialog window with the following details:

- Title:** Add new soils
- Identification:** Name: Sandy silt (MS), consistency soft | Třída S3, středně ulehlá
- Basic data (highlighted in red):**
  - Unit weight:  $\gamma = 17,50$  [kN/m<sup>3</sup>] 17,5
  - Effective stress method: ?
  - Bearing capacity coefficient:  $\beta_p = 0,45$  [-]
  - Settlement - oedometric modulus: ?
  - Poisson's ratio:  $\nu = 0,30$  [-] 0,30
  - Settlement analysis: insert Eoed
  - Oedometric modulus:  $E_{oed} = 21,00$  [MPa] 16 - 26
  - Uplift pressure: ?
  - Calc. mode of uplift: standard
  - Saturated unit weight:  $\gamma_{sat} = 19,50$  [kN/m<sup>3</sup>]
- Draw:**
  - Color: [Yellow-Green]
  - Pattern category: GEO
  - Pattern: Sand (visualized as a yellow field with small green dots)
- Classification:**
  - Buttons: Classify, Delete, Add, Cancel

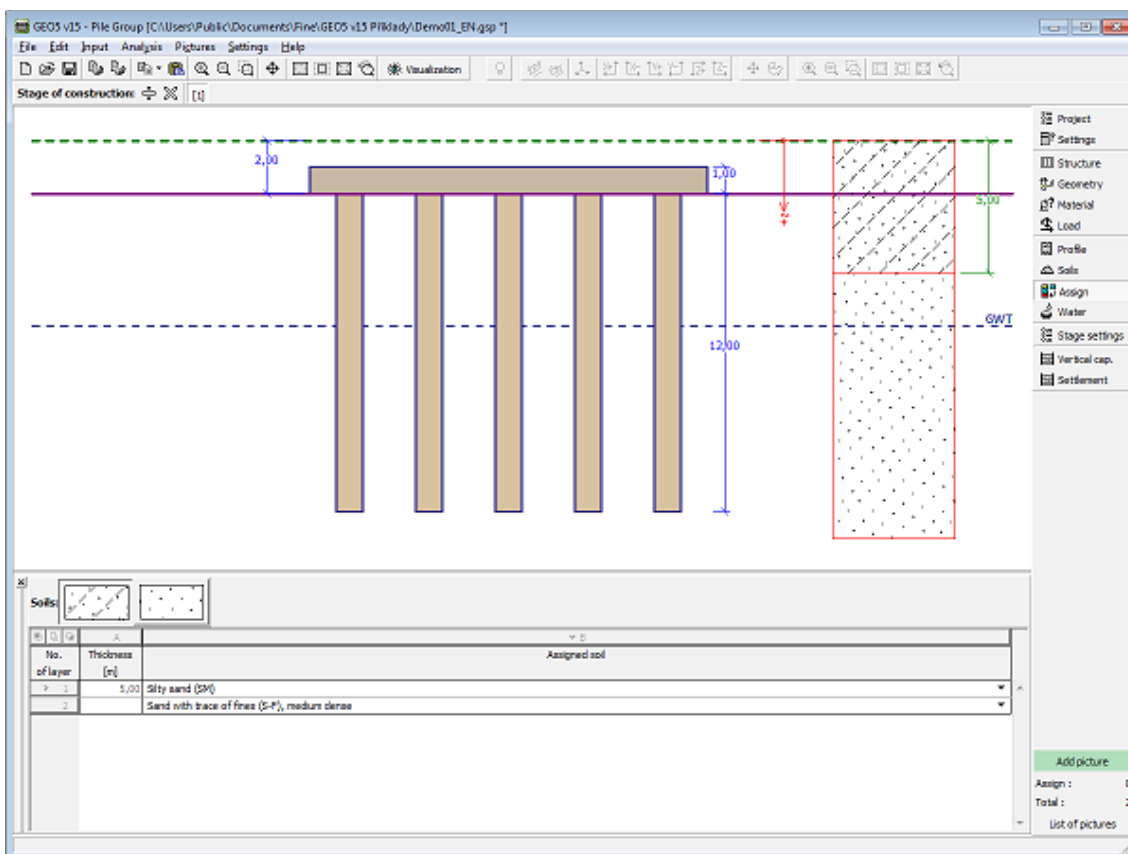
Dialog window "Add new soils" - "Basic data"

## Assign

The "**Assign**" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#).



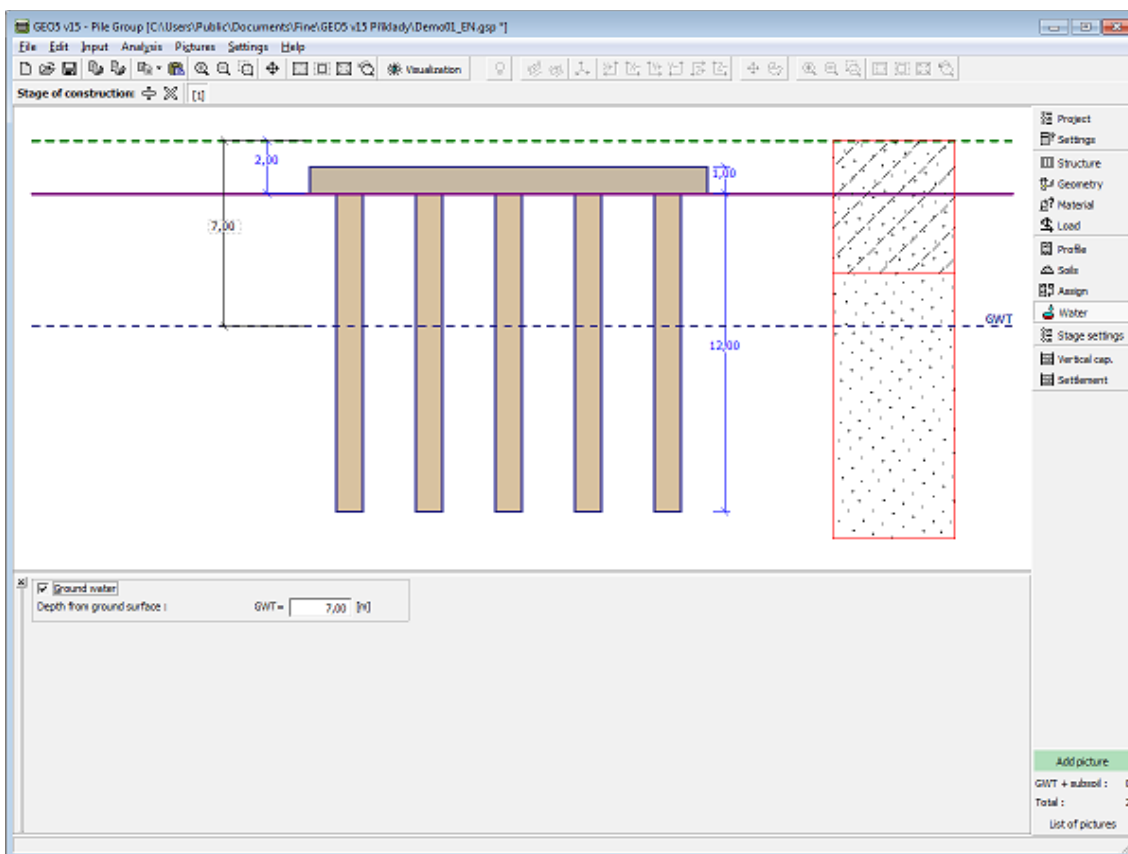
Frame "Assign"

## Water

The "Water" frame serves to specify the **depth of ground water table**.

The value can be edited either in the frame by inserting the value into the input field, or on the desktop with the help of **active dimensions**.

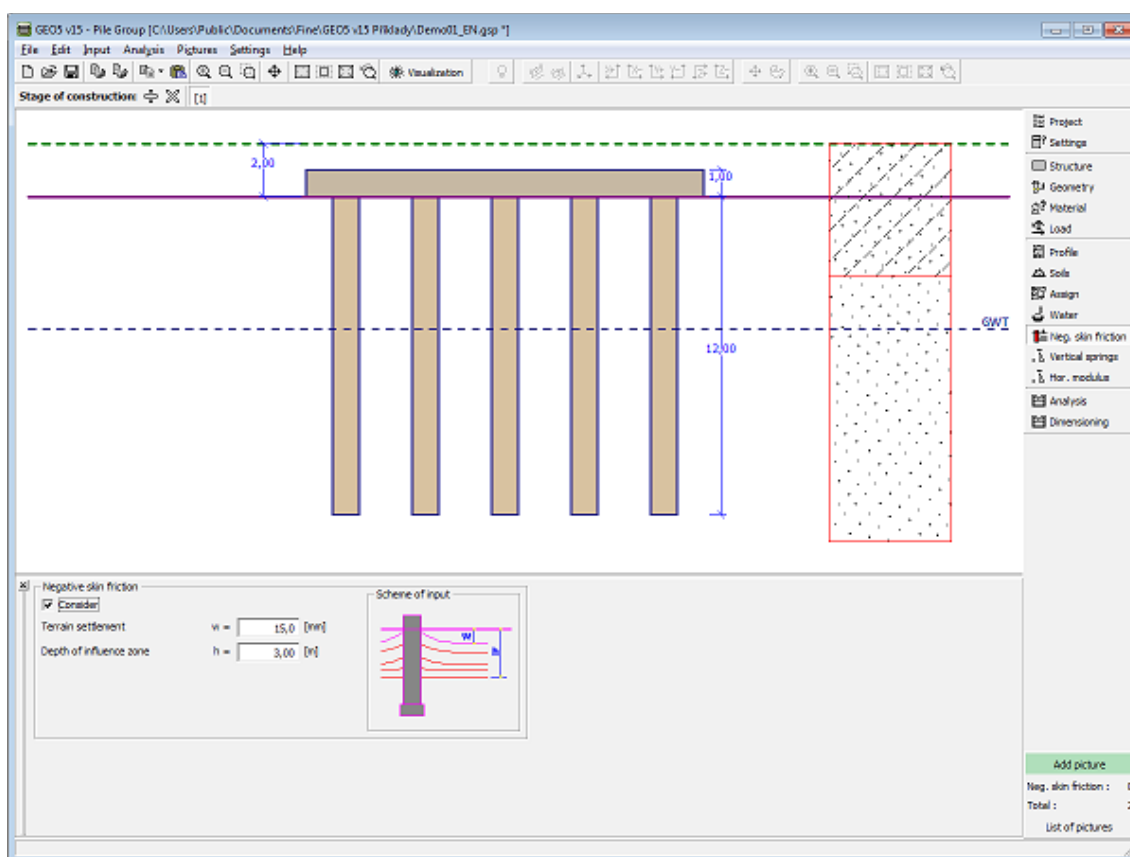
The **GWT** changes the **geostatic stress** in the soil profile.



Frame "Water"

## Negative Skin Friction

The "Negative skin friction" frame serves to specify the settlement of surrounding terrain and the depth of influence zone. For more information on the influence of negative skin friction the user is referred to theoretical section.



Frame "Negative skin friction"

## Vertical Springs

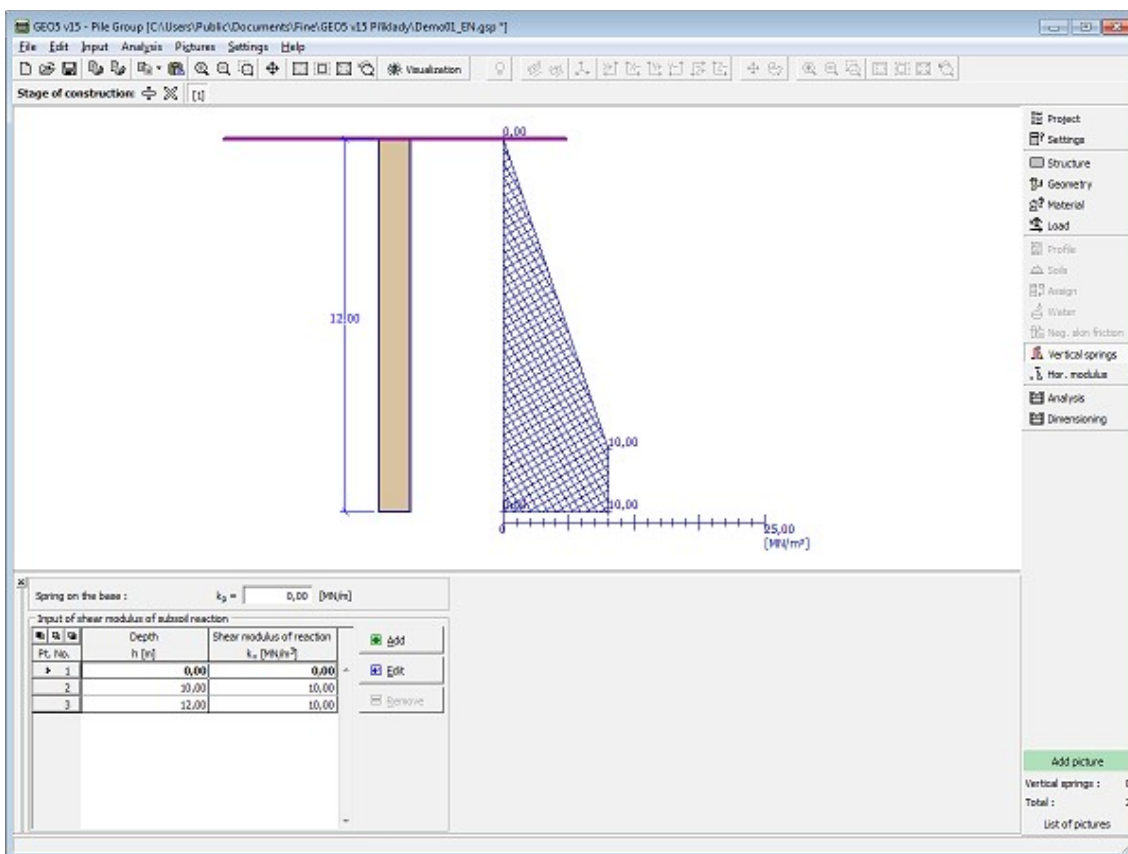
The frame "Vertical springs" is active only when analyzing a **floating pile**. The input springs are displayed in the [table](#).

The option "**input the stiffness of springs**" requires inputting:

- spring at the pile base [ $MN/m$ ]
- shear modulus of subsoil reaction along the pile [ $MN/m^3$ ].

The input values are the same for all piles. In the analysis the vertical stiffnesses of inner and outer piles in the group are **reduced** by particular coefficients.

The option "**Compute the stiffness of springs from soil parameters**" requires inputting the **typical load** to obtain the spring stiffnesses from calculation.

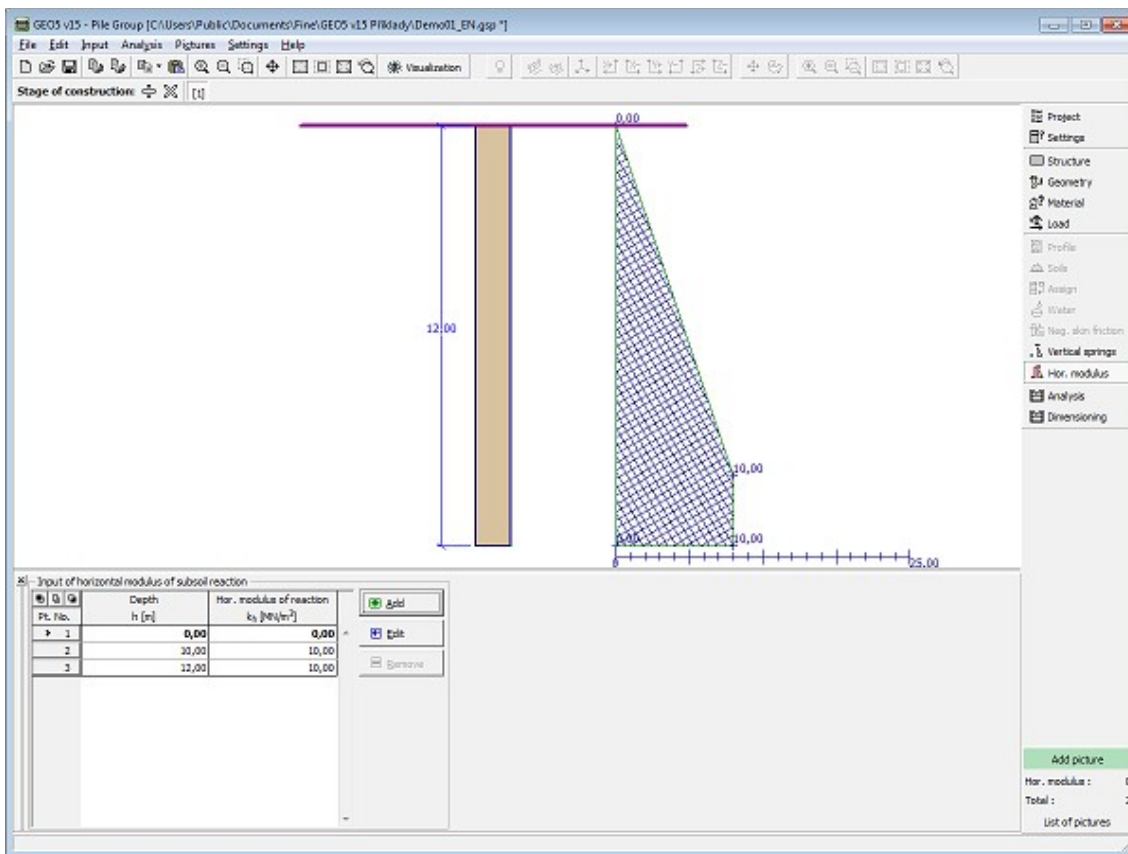


Frame "Vertical springs"

## Horizontal Modulus

The **frame "Horizontal modulus"** serves to input the horizontal modulus of subsoil reaction characterizing the pile response in the horizontal direction.

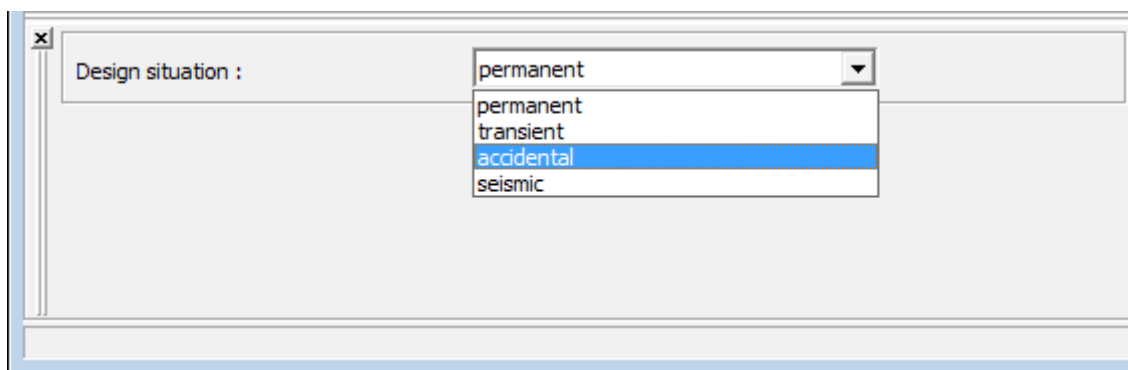
The **input** values of the modulus of subsoil reaction at a given depth of the profile are displayed in the **table**.



Frame "Horizontal modulus"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage. Selected design situation determines the safety coefficients to be used in the analysis of a given construction stage.



Frame "Stage settings"

## Vertical Bearing Capacity - Analytical Solution

The frame "Vertical bearing capacity" serves to verify the pile group vertical bearing capacity. Several analyses can be carried out in the frame.

The verification analysis can be carried out for individual loads or the program identifies **the most critical one** (can be selected from the combo list).

The analysis is performed according to the theory set in the frame "Settings" (analytical solution):

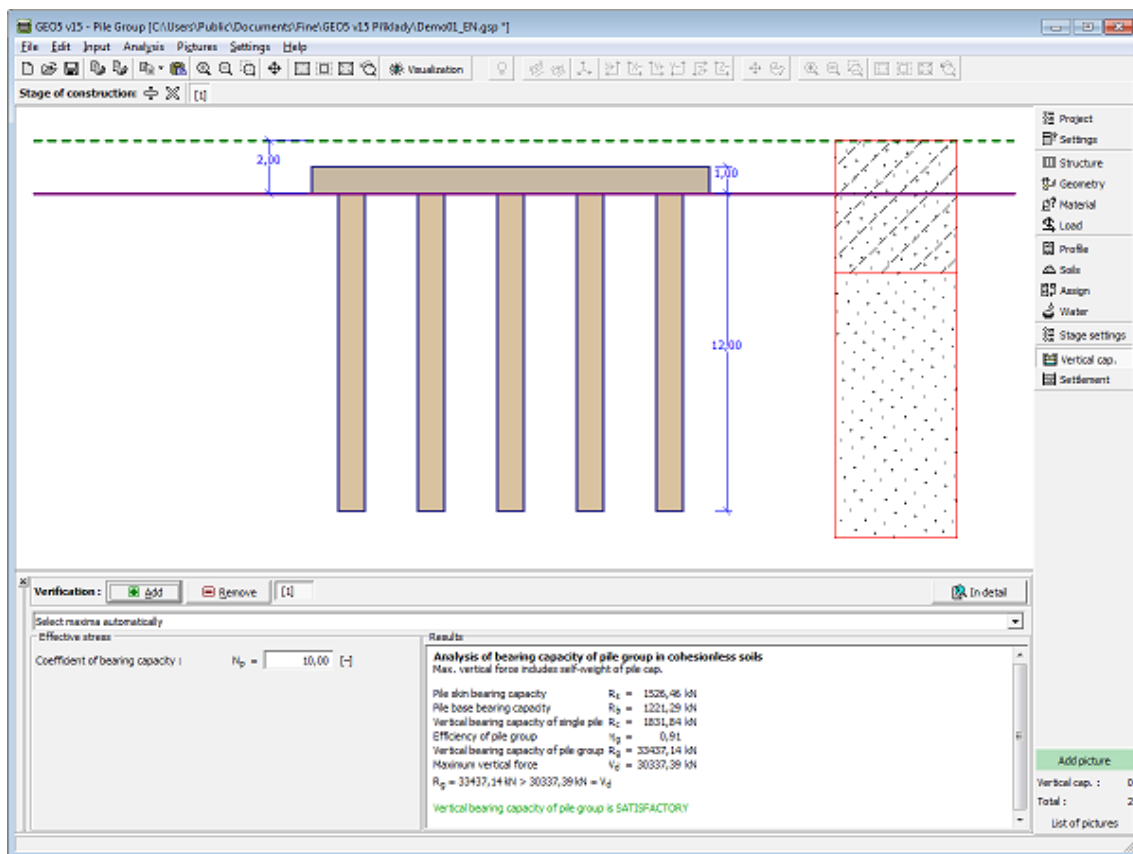
- for **cohesive soil** (undrained conditions) - analysis of bearing capacity of an earth block according to FHWA
- for **cohesionless soil** (drained conditions) - NAVFAC DM 7.2, Effective stress, CSN 73 1002

The parameters needed for the pile group analysis are introduced for individual methods in left part of the frame.

The verification analysis is carried out according to the **verification methodology** selected in the "Pile Group" tab (factors of safety, theory of limit states, EN 1997-1).

The "In detail" button opens the dialog window containing detailed listing of the verification results.

The analysis results are displayed in the right part of the desktop.



Frame "Vertical bearing capacity" - analytical solution

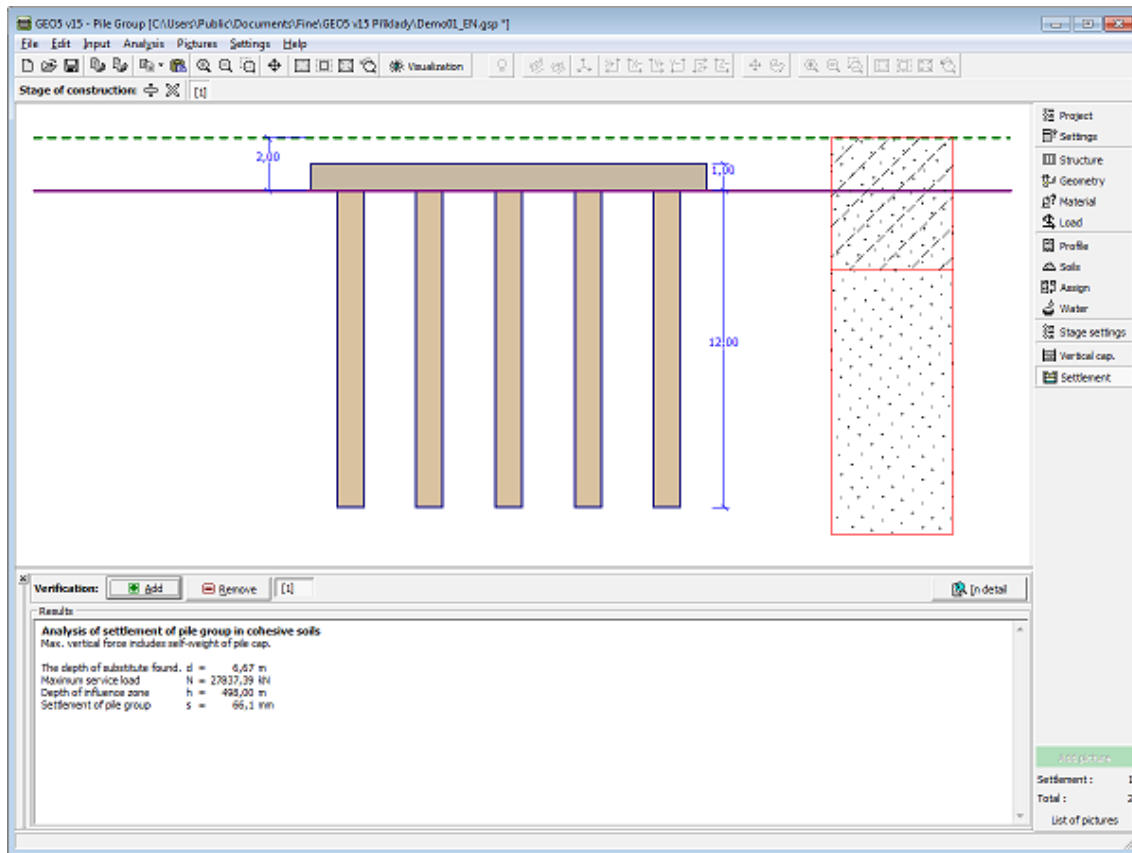
## Settlement - Cohesive Soil

This **frame** serves to calculate the pile group **settlement** for cohesive soils. The analysis of **settlement** is performed according to the selected theory set in the "Pile Group" tab. The analysis results are displayed in the right part of the desktop.

The "In detail" button opens the dialog window that contains a detailed description of the



results of the verification analysis.



Frame "Settlement" - cohesive soil

## Settlement - Cohesionless Soil (Load-Settlement Curve)

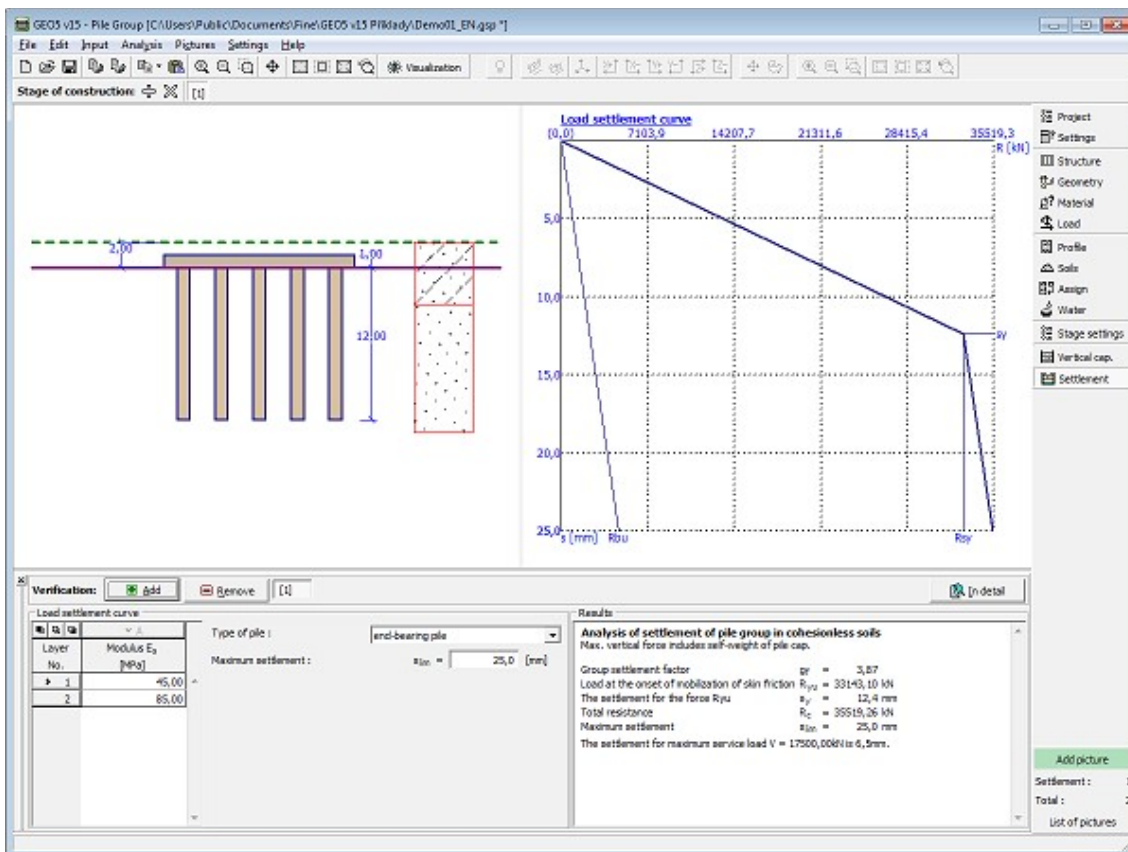
The frame "Settlement" displays linear load-settlement curve for the settlement of a pile group in a cohesionless soil. Several analyses can be carried out in the frame.

The load-settlement curve of pile group is calculated always for the input **limit settlement**.

The table in the left bottom part of the frame directly allows to specify values of the **secant modulus of soil** for the relevant layers of soil.

The **analysis results** are displayed in the right part of the frame. The "In detail" button opens the dialog window that contains a detailed description of the results of the verification analysis.

The analysis results (load-settlement curve of pile group) are displayed in the right part of the desktop.



Frame "Settlement" - cohesionless soil (load-settlement curve of pile group)

## Analysis - Spring Method

This **frame** serves to analyze a group of piles using the **spring method**. The analysis is run by pressing the **"Analysis"** button.

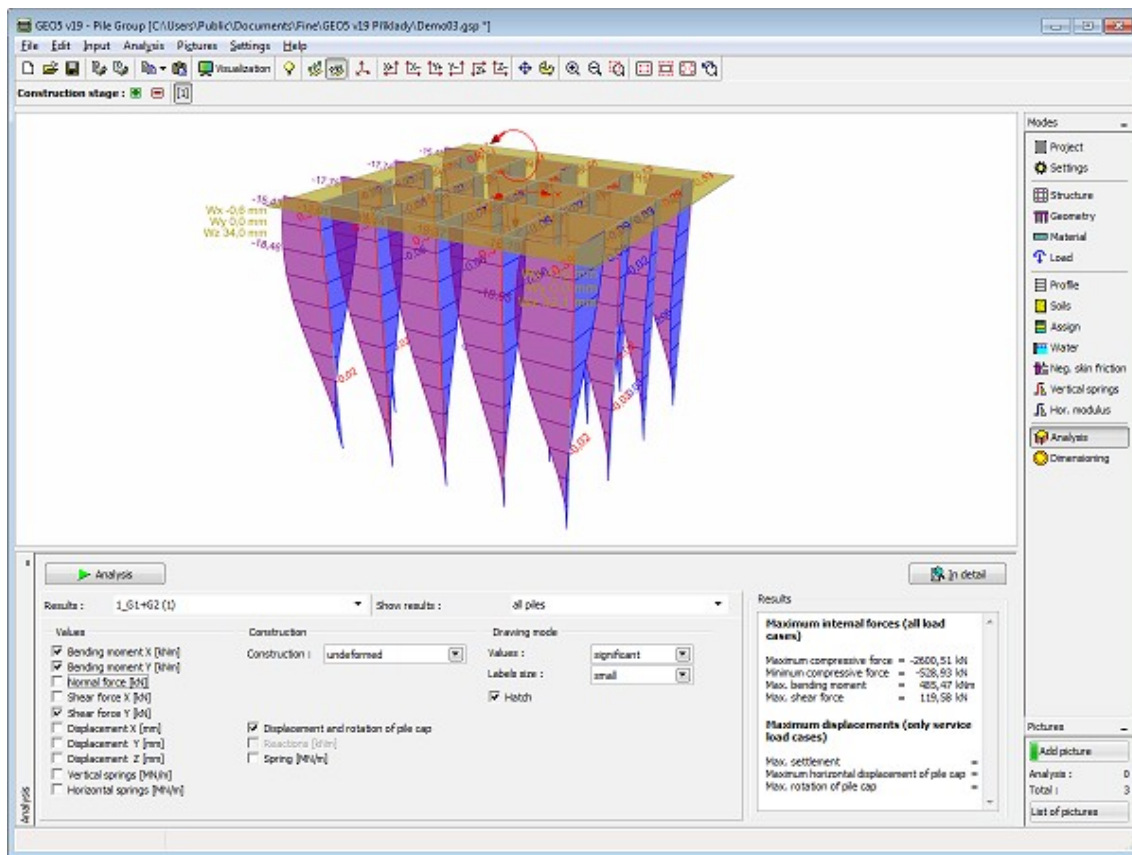
Upon performing the analysis, the results appear in the right part of the frame (**"Results"**) providing information about the **maximum internal forces, displacements and rotations of a structure**. Displacements of the structure shown in the window are determined for service loads. The **"In detail"** button opens the dialog window that contains a detailed description of the results of the verification analysis.

The left part of the frame allows for defining the way of plotting the results on the screen:

- **Results** - the results can be displayed for individual load cases or for their envelope
- **Show results** - the results can be displayed for all piles or for individual piles only
- **Values** - visualization of values of individual variable (moments, normal and shear forces, displacements, springs)
- **Structure** - allows for plotting a deformed structure (only undeformed structure can be displayed for envelopes of load cases), next it is possible to show the magnitudes of pile cap deflections, reactions and magnitudes of springs at the pile base.
- **Drawing mode** - defines the style of describing the results

The displayed results can be added to the **"List of pictures"** at any time and used in the **analysis protocol**.

Rotation, zooming and illumination of a structure can be adjusted with the help of "Visualization" tool bar. Visualization of results can be adjusted in the frame "Visualization settings".



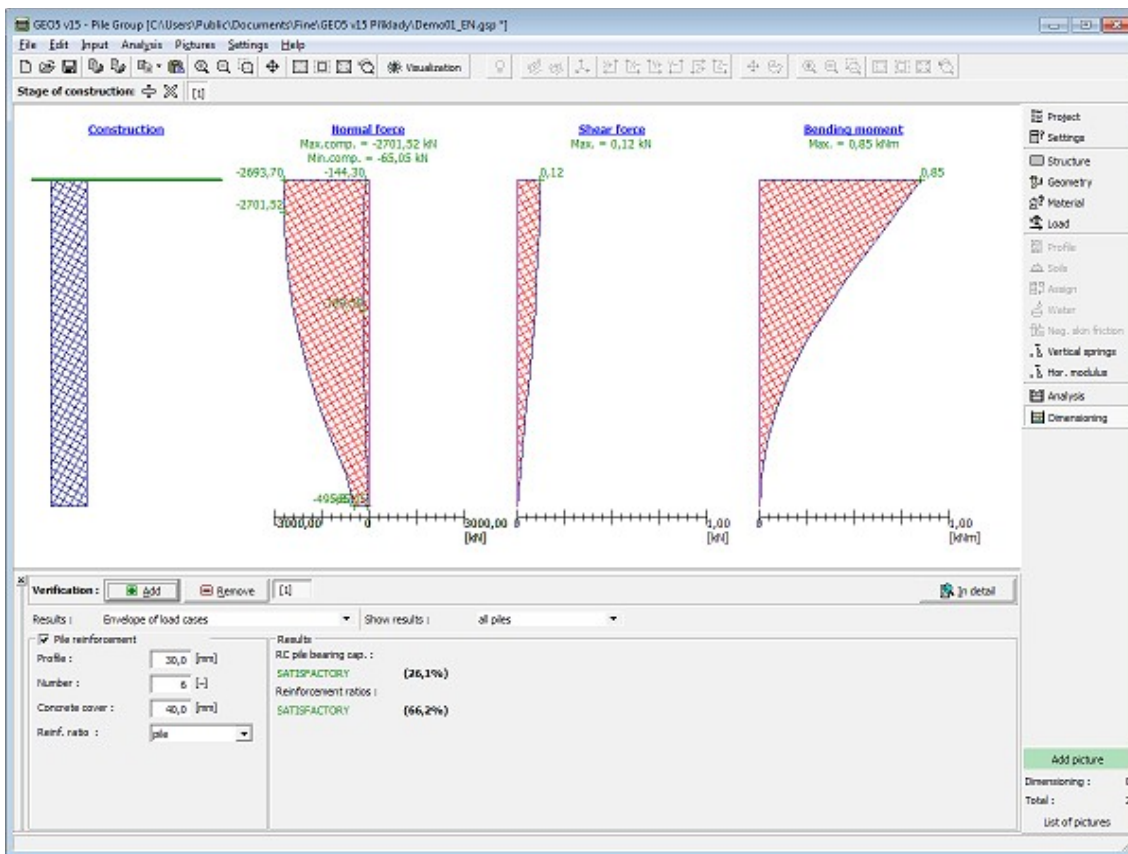
Frame "Analysis" - spring method

## Dimensioning

The frame "Dimensioning" allows the programs to adopt the results obtained from the calculations performed in the frame "Analysis". Both the envelope of loads and individual load cases can be selected. The reinforcement can be design for the selected pile or the same reinforcement for all piles in the group can be assumed.

The verification analysis of a steel-reinforced concrete pile is carried out according to the standard selected in the "Materials and standards" tab.

Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Dimensioning" - spring method

## Program Micropile

This program is used for verification of steel tube micropiles. When calculating the micropile bearing capacity, the program verifies both the root and shaft.

**The help in the program "Micropile" includes the following topics:**

- Input of data into individual frames:

Project	Settings	Profile	Soils	Geometry	Material	Assign
Load	Water	Verification of Cross-Section	Root Verification			

- Standards and analysis methods

- Theory for analysis in the program "**Micropile**":

Micropile

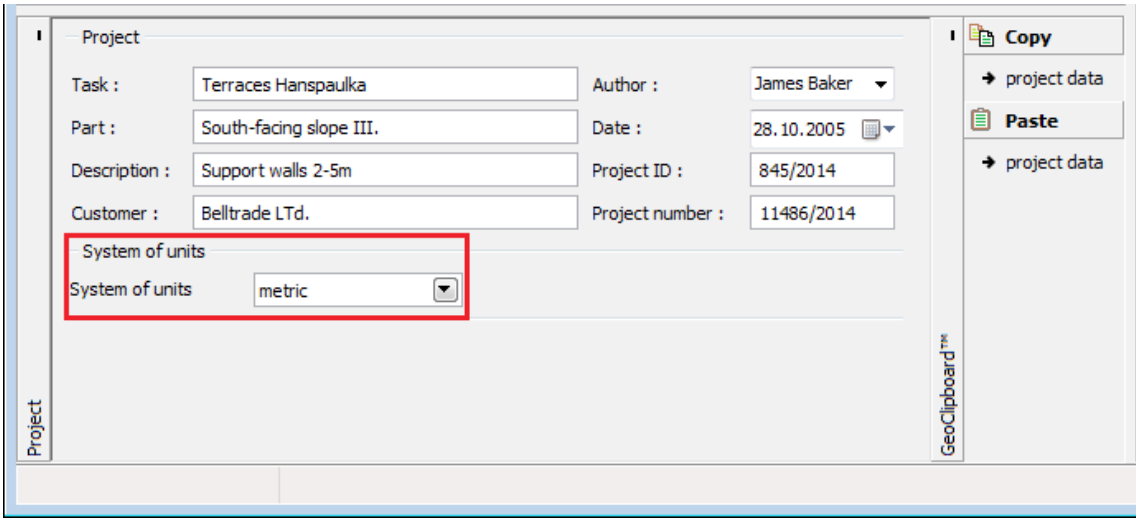
Field Testing

- **Outputs**
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.



Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

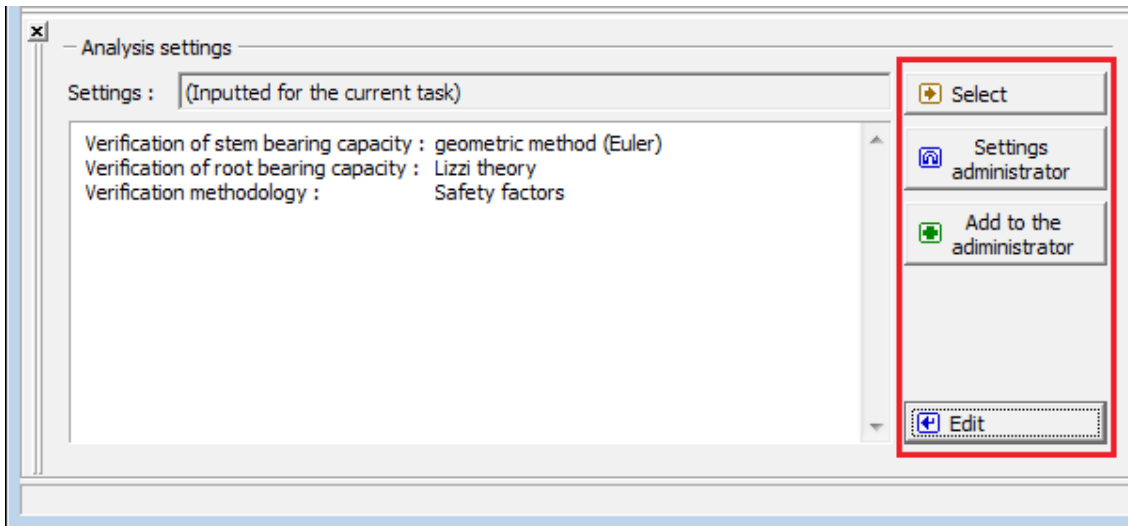
The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The **"Modify"** button enables a quick visualization and editing of the current Setting in the

opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the **"Micropiles"** tab.



Frame "Settings"

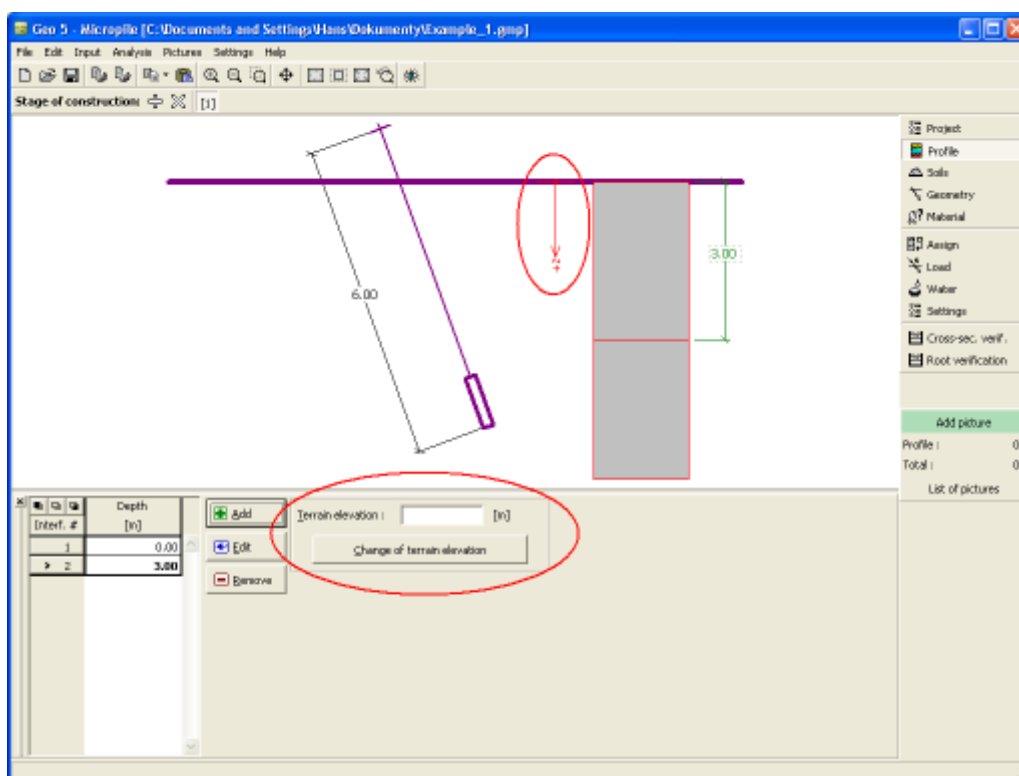
## Profile

The **"Profile"** frame contains a table with a list of input interfaces. After specifying interfaces it is possible to edit thicknesses of individual layers with the help of **active dimensions**.

**Adding** (editing) layer is performed in the **"New interface"** dialog window. The *z*-coordinate measured from the top point of a structure is specified (*z*-axis).

The program allows for raising or lowering the top point of a structure in the **"Change of terrain elevation"** dialog window so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from program **"Terrain"**.

The program makes it possible to **import a profile in the gINT format**.



Frame "Profile"

## Soils

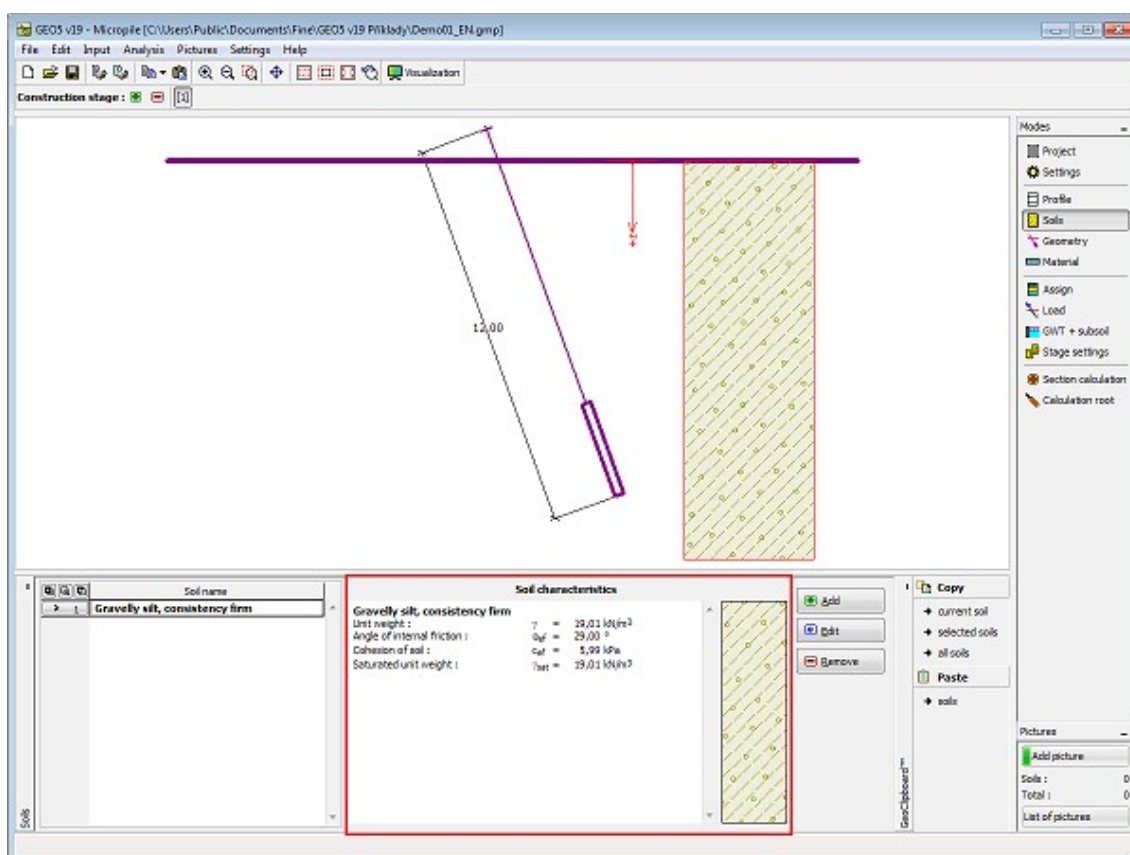
The **"Soils"** frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data" "Uplift pressure". These parameters depend on the theory of analysis specified in the "Micropiles" tab.

The program makes it possible import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".





Frame "Soils"

## Basic Data

This part of the window serves to introduce basic parameters of soils - **unit weight, angle of internal friction and cohesion**. The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in [database of soils](#), which contains values of selected characteristics of soils. The characteristics of rocks are not listed in the database built, these parameters must be defined manually. Approximate parameters of rocks are presented in the theoretical part of the Help [herein](#).

When calculating the tube bearing capacity according to [Salase](#), moreover, enters elastic modulus  $E$ .

The associated theory is described in detail in chapter "[Micropile](#)".



Dialog window "Add new soils" - "Basic data"

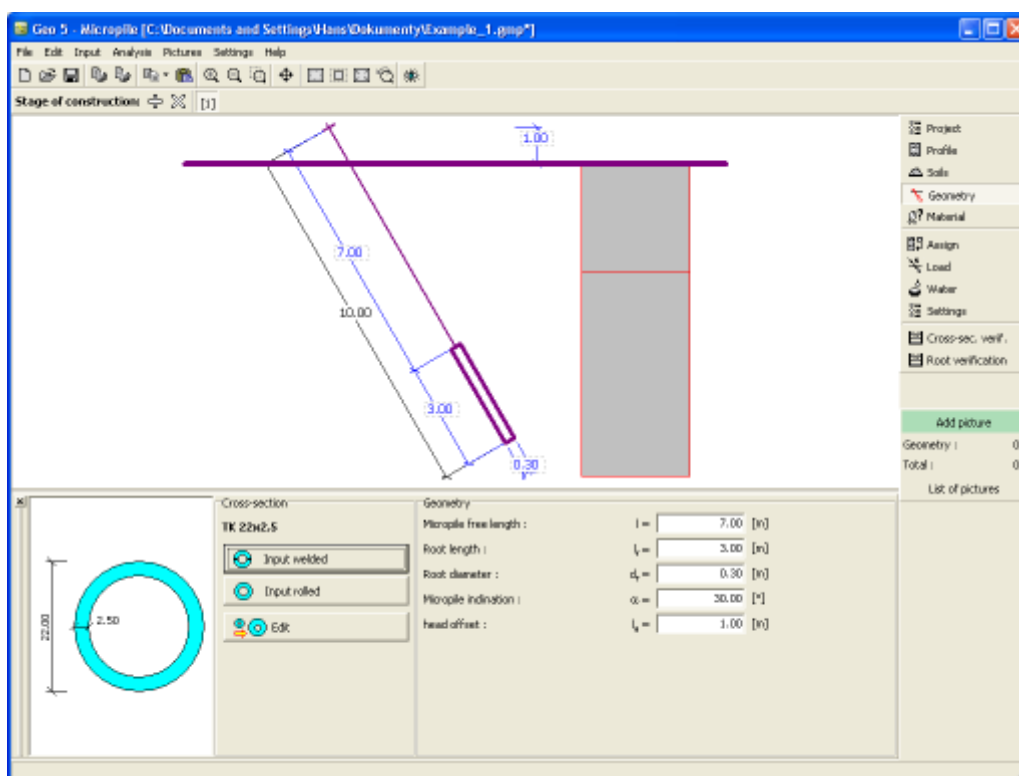
## Geometry

The **frame "Geometry"** serves to input a **micropile cross-section** (welded, rolled). The selected shape with a graphical hint of input values is displayed in the left part of the frame. The micropile cross-section is selected in dialog windows opened by pressing the "Enter welded" "Enter rolled" buttons (the selection for rolled cross-sections is performed from a catalog in the dialog window). An info window, displaying a detailed description of data of the selected cross-section, can be activated in the window. The selected data can be edited after choosing the type of micropile cross-section.

The basic geometrical data are specified in the right top part of the frame:

- free length of micropile (distance between the micropile head and the origin of micropile base is considered)
- root length
- root diameter
- micropile inclination (range from  $-60^\circ$  to  $60^\circ$  measured from vertical, a positive value of an inclination angle is measured counterclockwise)
- head offset (end of micropile above terrain (range from 0 to 10 m).

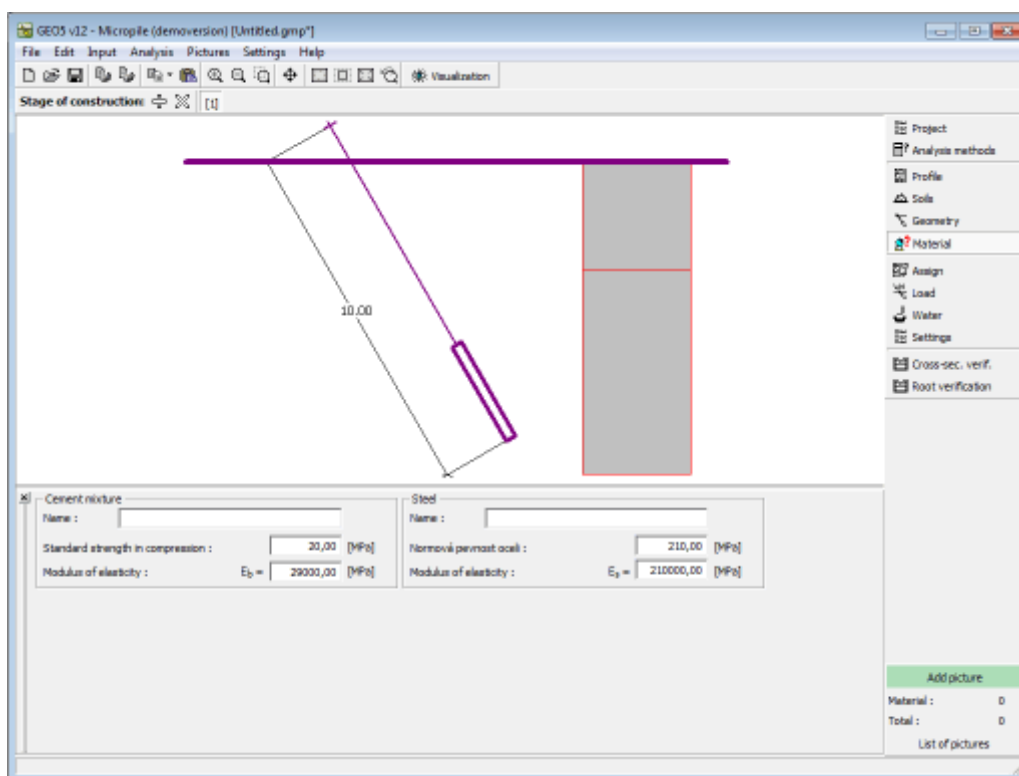
The program makes it possible to **export** the geometry of a structure in the \*.DXF format.



Frame "Geometry"

## Material

The frame "Material" serves to specify material parameters of cement mixture and steel. **Standard strength** of cement mixture in compression, standard strength of steel and **modulus of elasticity** of the selected steel and concrete mixture are entered. These values are required for verification of the micropile tube - **coupled section bearing capacity**.



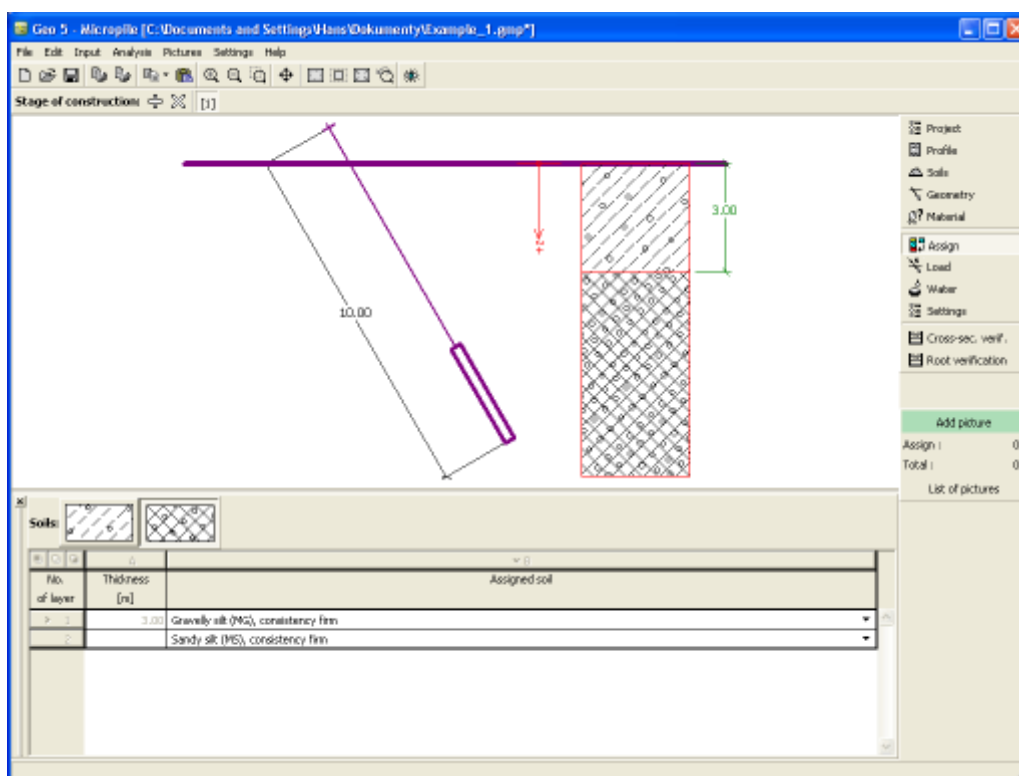
Frame "Material"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

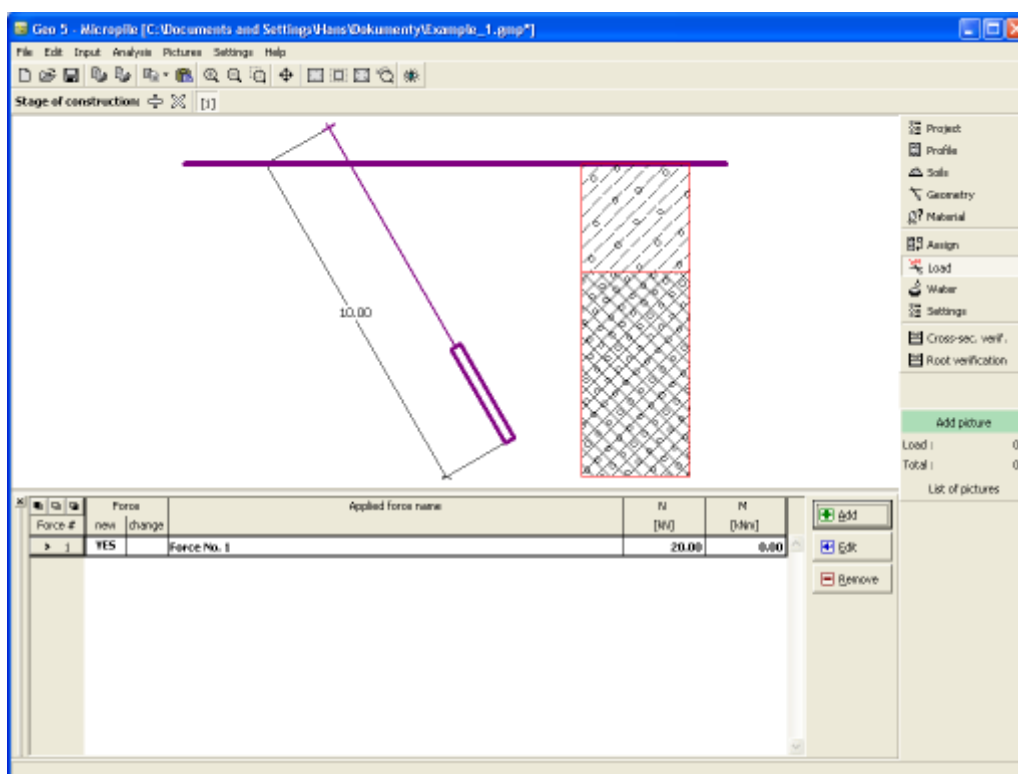
The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

## Load

The "Load" frame contains a table with a list input loads. Adding (editing) load is performed in the "New (edit) load" dialog window. Forces and moments are entered according to the sign convention displayed in the right part of the dialog window.

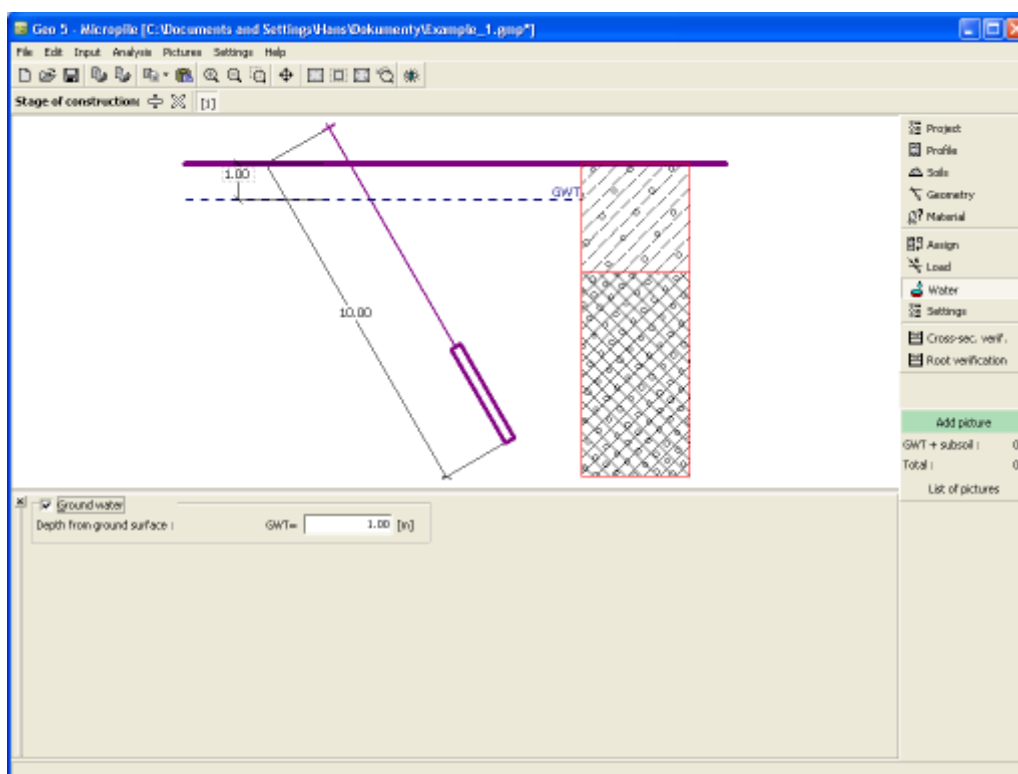


Frame "Load"

## Water

The frame "**Water**" serves to enter a **depth of ground water table**.

The values can be edited either in the frame by entering values into particular fields, or on the desktop with the help of **active dimensions**.



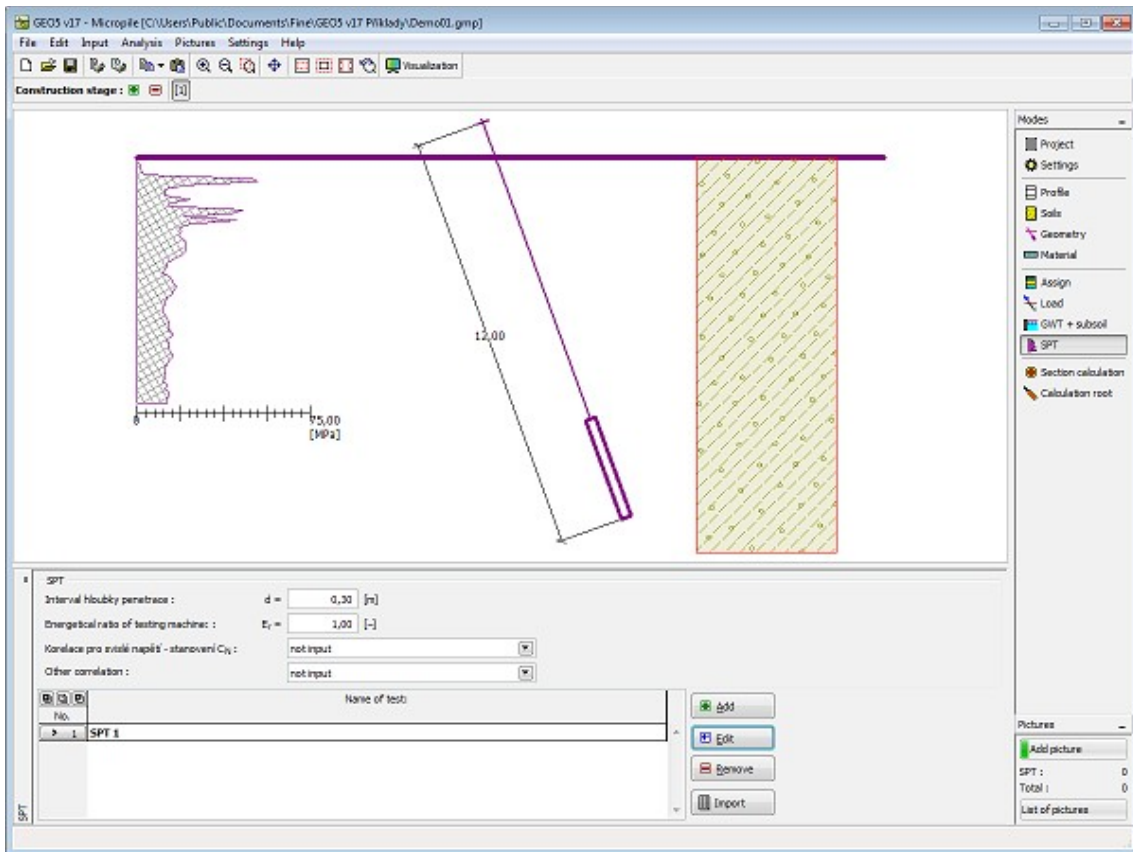
Frame "Water"

## Standard Penetration Tests (SPT)

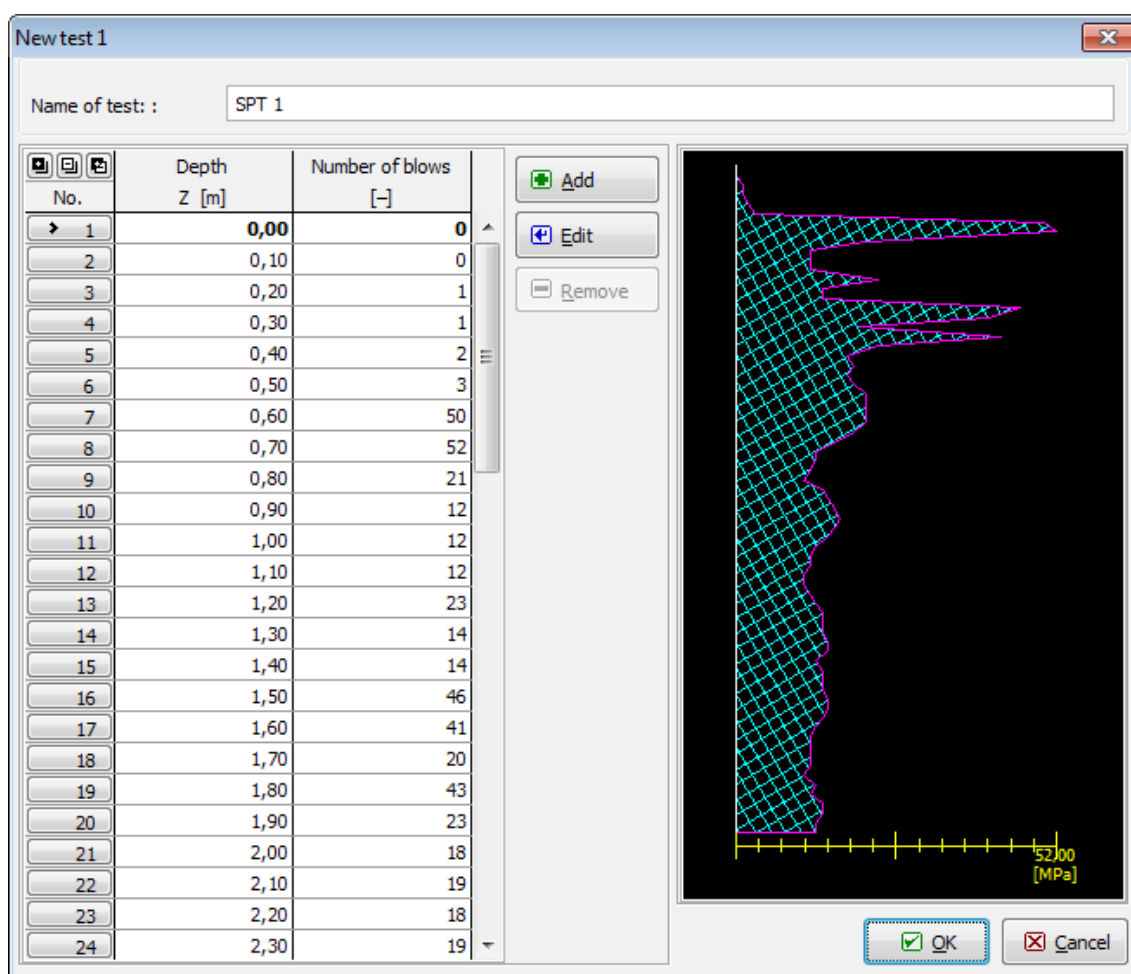
The "SPT" frame contains a table with the list of input standard penetration tests (SPT). The frame serves to define the test parameters (interval of the penetration depth  $d$ , energetical ratio of testing machine  $E_r$ ) and correlation of measured data. These are explained in more details in the theoretical part.

The name of test is specified in the "New test" ("Edit test") dialog window. The "Add" button in this dialog window opens another "New value of test" ("Edit value of test") dialog window, which allows for specifying the depth  $z$  measured from the terrain level and a given number of blows.

The results of standard penetration tests (SPT) can also be imported in the \*.TXT format.



Frame "SPT"



Dialog window "New test"

## Pressiometric Tests

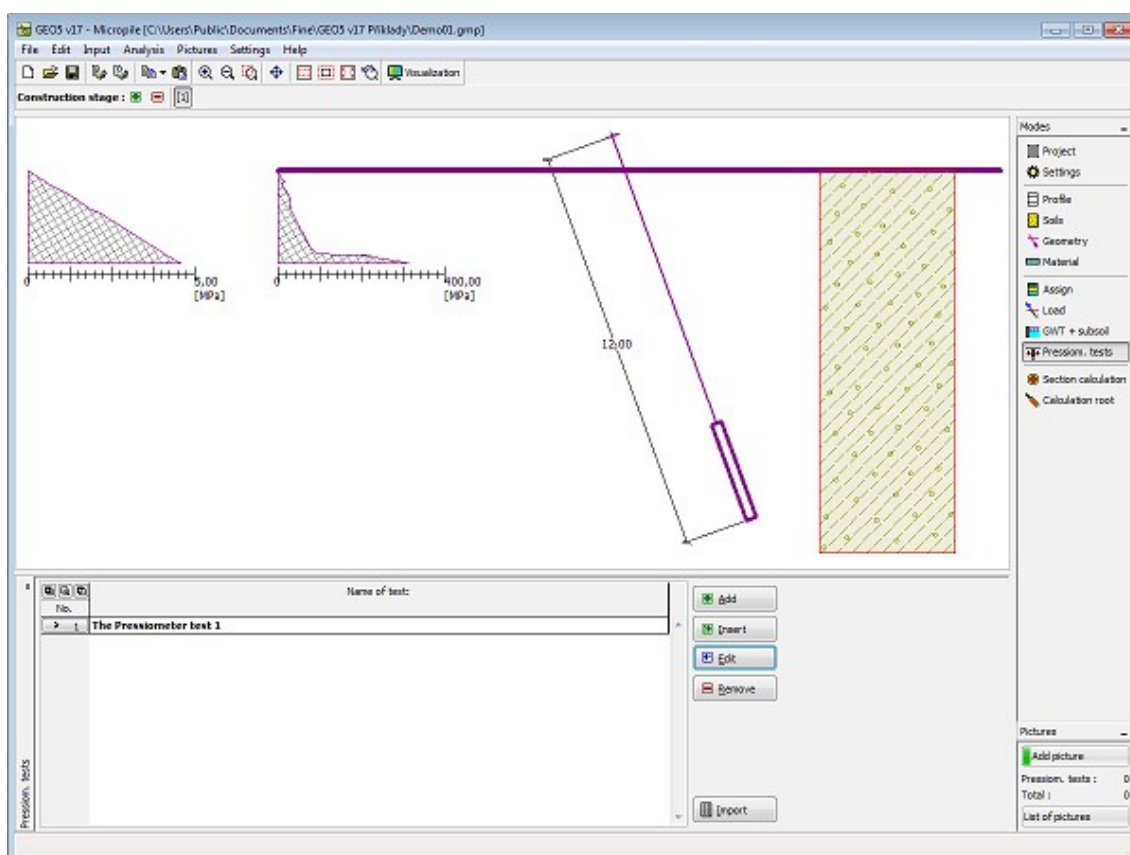
The frame "Pressiometric tests" contains a table with list of input points of pressiometric test (PMT).

The "Add" button in this dialog window opens another "New value of test" ("Edit value of test") dialog window, which allows for specifying the depth  $z$  measured from the terrain level, the limit pressure  $p_{LM}$  and the Menard modulus  $E_m$ . These parameters are explained in more detail in the theoretical part.

User can change entered values in dialog window "Edit test". Inserting values between already entered values is realized in dialog window "Inserted test".

The results of pressiometric tests can also be imported in the \*.TXT format.





Frame "Pressiometric tests"

## Verification of Cross-Section

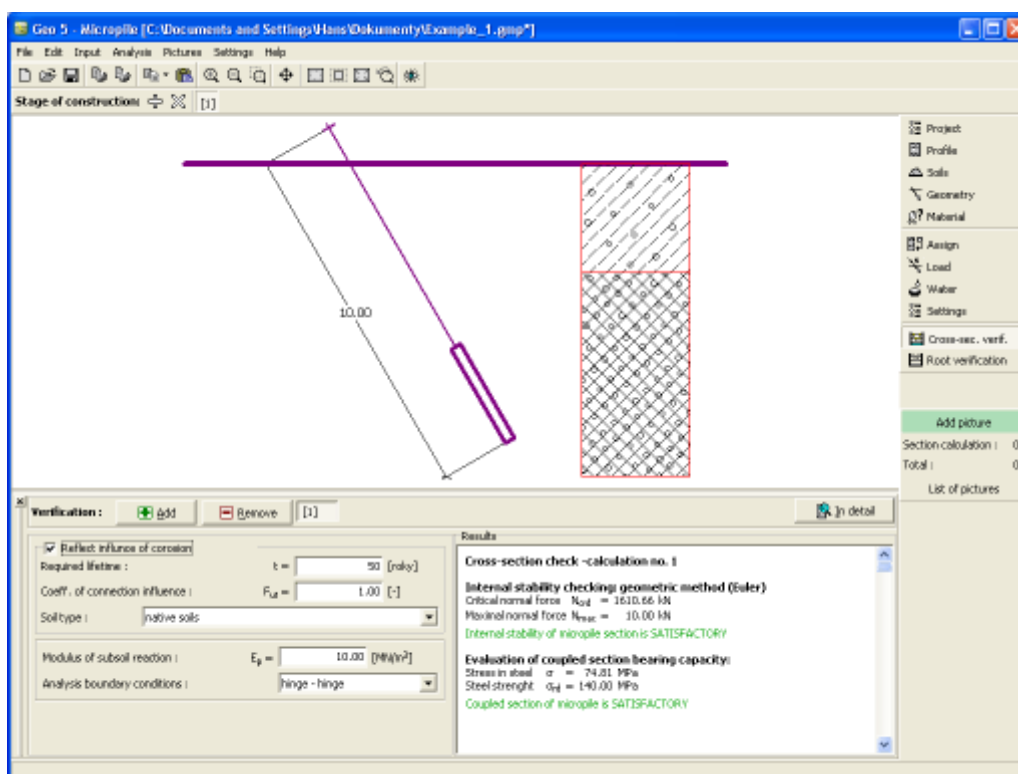
The results of the analysis of the micropile tube bearing capacity loaded either in tension or compression are displayed in the frame "**Verification of cross-section**". **More computations** can be carried out for a single task. The left part of the frame allows for inputting the modulus of subsoil reaction and to account for the **influence of corrosion** on the analysis.

When calculating the **tube bearing capacity** (micropile cross-section) the program differentiates between a micropile loaded in tension or in compression.

In case of tension the program determines **coupled section bearing capacity** (strength of cement mixture is not considered).

In case of compression the program examines both, **coupled section bearing capacity** and **internal stability of section**, depending on the method set in the "Micropiles" tab.

The results of the verification analysis are displayed in the right part of the window. The "**In detail**" button opens a dialog window listing in detailed the results of the analysis. Visualization of results can be adjusted in the frame "**Visualization settings**".



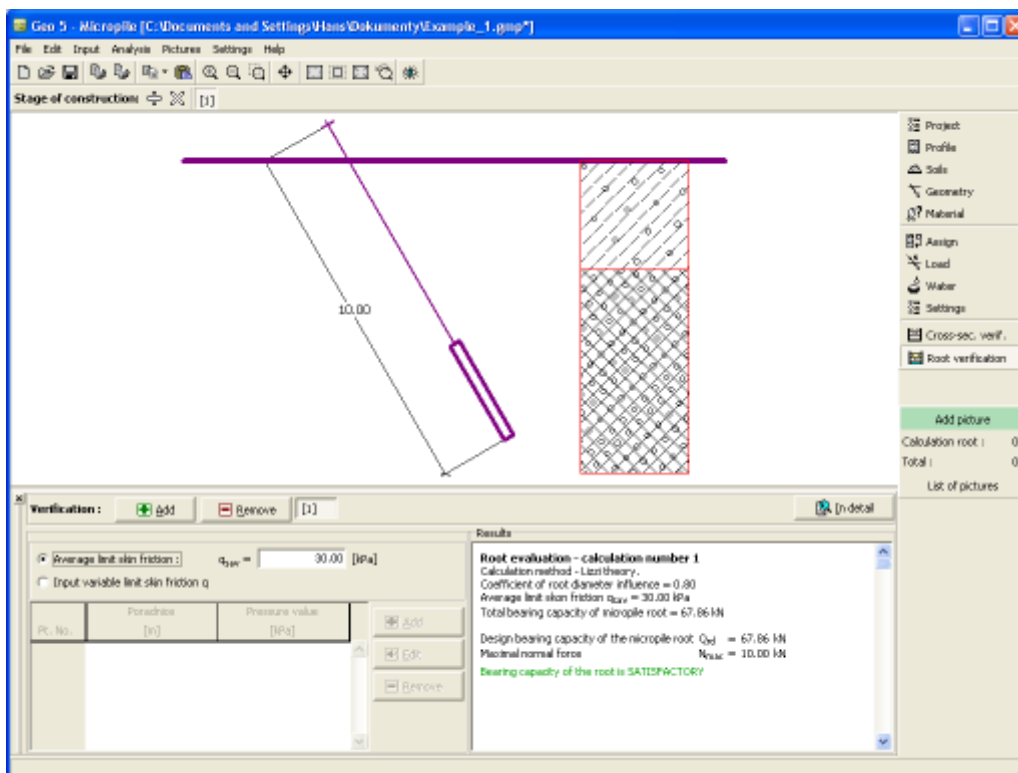
Frame "Verification of cross-section"

## Root Verification

The analysis results are displayed in the frame "Root verification". Several calculations can be carried out for a single task. The limit skin friction can be specified in the left part of the frame.

The procedure to examine the micropile root is described in detail [herein](#).

The results of the verification analysis are displayed in the right part of the window. The "In detail" button opens a dialog window listing in detailed the results of the analysis. Visualization of results can be adjusted in the frame "Visualization settings".



Frame "Root verification"

## Program Slab

The program Slab (*formerly: Plate*) is used for analysis of foundation mats and slabs of any shape on elastic subsoil, using the Finite Element Method.

The help in the program "Slab" includes the following topics:

- Input of data into individual frames:

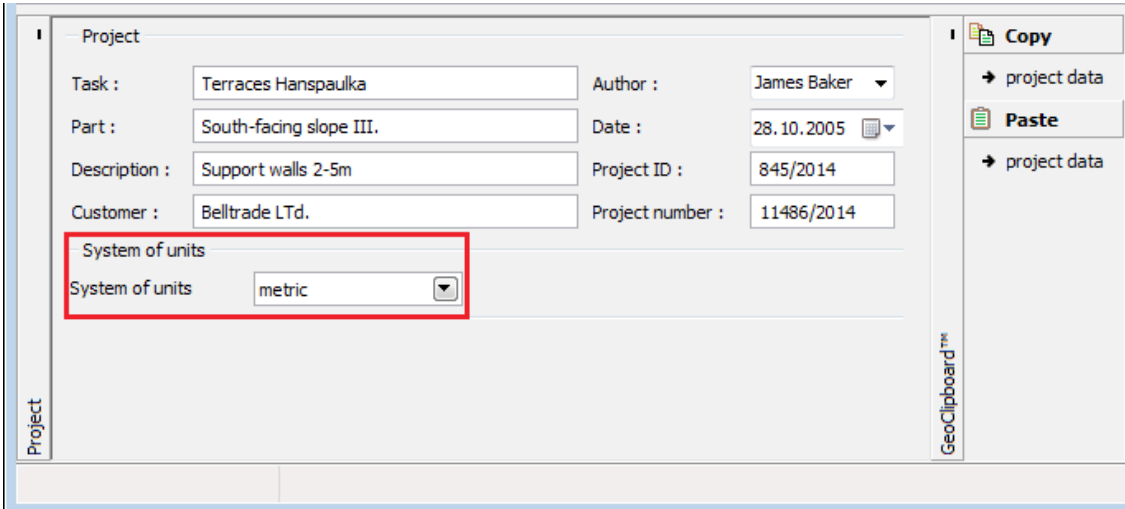
Project	Settings	Joints	Lines	Macroelement	Openings	Joint Refinements
Line Refinements	Macroelement Refinements	Mesh Generation	Joint Supports	Line Supports	Beams	Internal Hinges
Macroelement Subsoils	Load Cases	Joint Loads	Line Loads	Macroelement Loads	Free Point Loads	Free Line Loads
Free Area Loads	Combination ULS	Combination SLS	Dimensioning Parameters	Macroelement Dimensioning	Analysis	Values
Distributions						

- [Standards and analysis methods](#)
- [Theory](#) for dimensioning of concrete structures
- [Outputs](#)
- General information about the work in the [User Environment](#) of GEO5 programs
- [Common input](#) for all p

## Project

The [frame "Project"](#) is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in [text and graphical outputs](#).

The frame also allows user to switch analysis units ([metric / imperial](#)). Project data can be copied within all GEO5 programs using ["GeoClipboard"](#).



*Frame "Project"*

## Settings

The [frame "Settings"](#) allows to introduce the basic [settings](#) of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the ["Settings list"](#).

The **"Settings Administrator"** button opens the ["Administrator"](#) dialog window, which allows

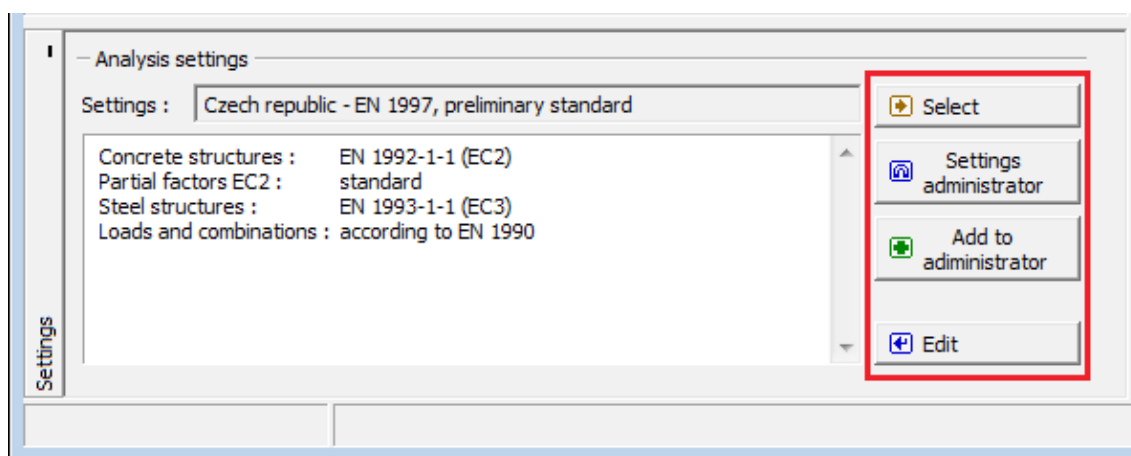
for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The "**Add to the administrator**" button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The "**Modify**" button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to "**Input for the current task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Materials and standards**" tab.



Frame "Settings"

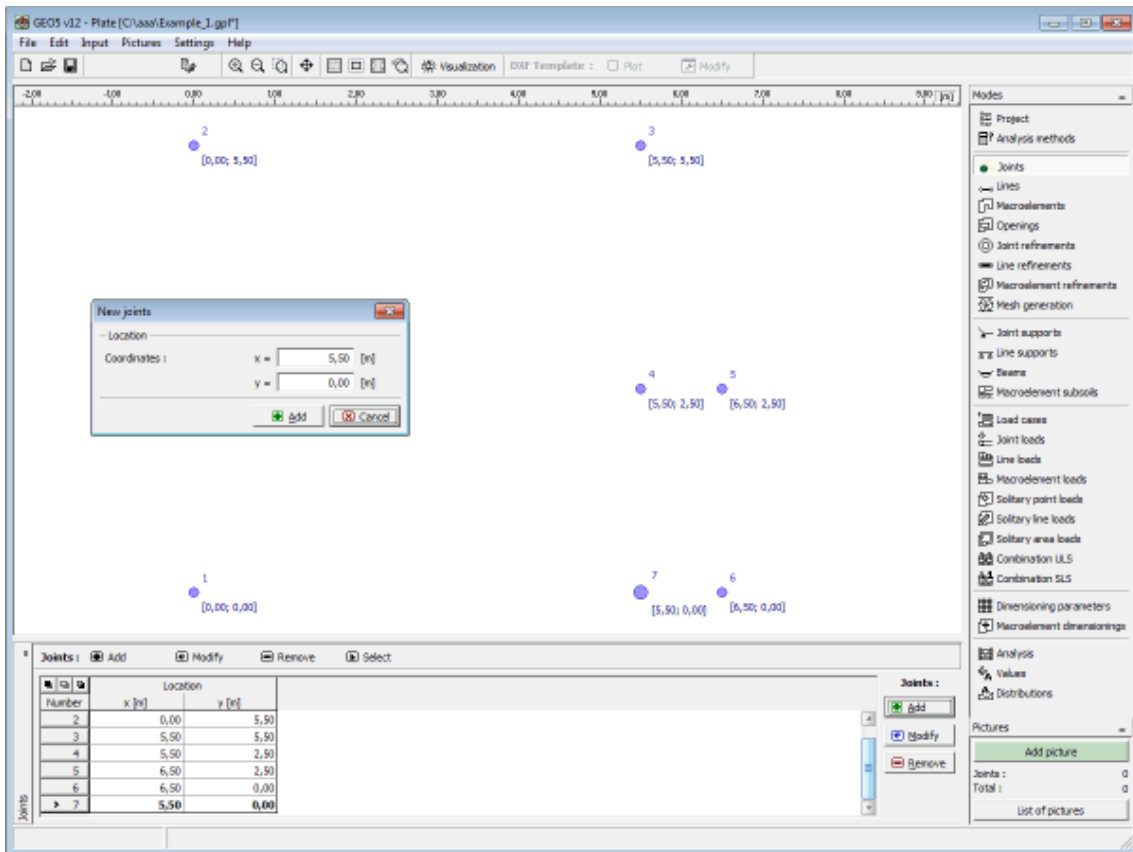
## Joints

The **frame "Joints"** contains a **table** with the list of input joints. Adding (editing) joints is performed in the "**New joints**" **dialog window**.

Joints can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Joints**". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the joint location.
- **Modify** Clicking the left mouse button on already existing joint opens the "**Modify properties of joint**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the joint opens the **remove joint** dialog window - accepting this action removes the selected joint.
- **Select** Clicking the left mouse button on the joint highlights the selected joint. The joint is simultaneously marked in the **table** list. The option allows for editing several joints at once (e.g. deleting).

The input joints can also be edited on the desktop with the help of **active objects**.



Frame "Joints"

## Lines

The frame "Lines" contains a table with the list of input lines. Adding (editing) lines is performed in the "New lines" dialog window.

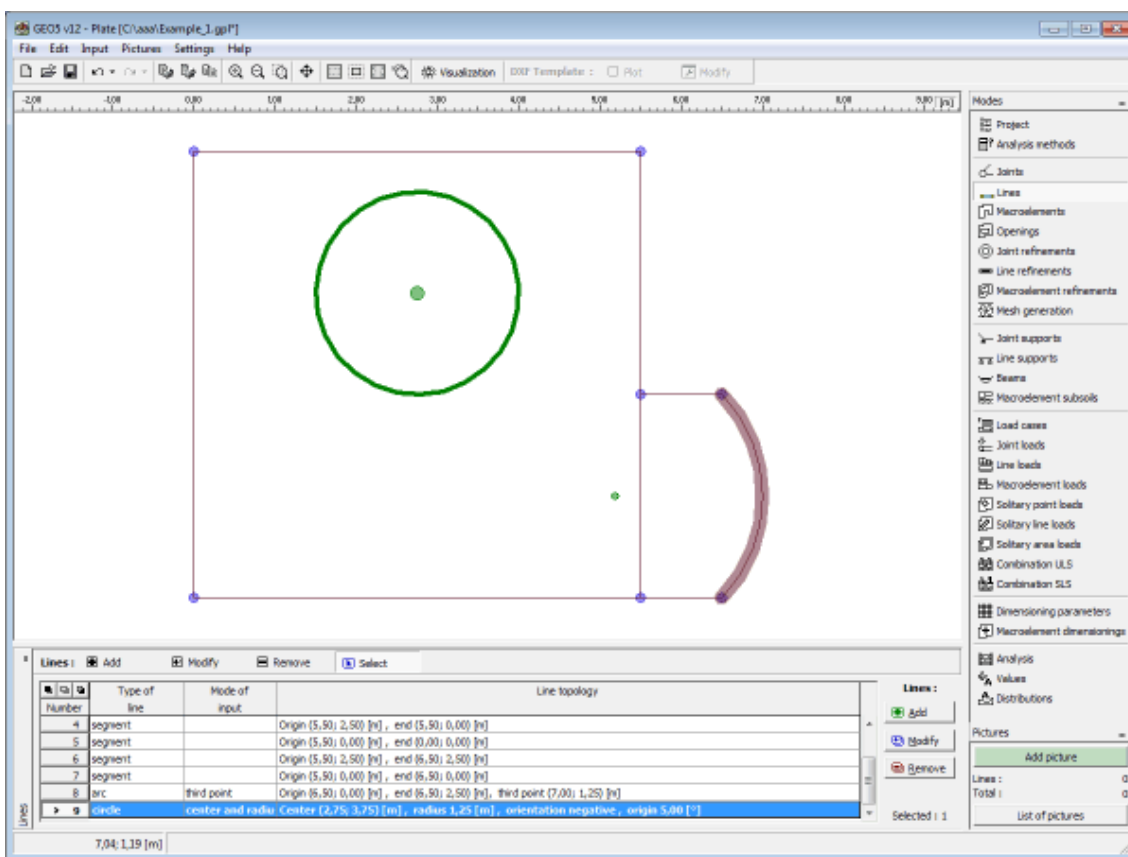
The lines are defined **between individual points** (segments, arcs, circles) or around individual points (circles). The lines may arbitrarily cross or touch each other - intersection of input lines are identified by the program automatically when correcting the input geometry.

Lines can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Lines". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the line location.
- **Line type** A combo list is used to select the desired type of line (segment, arc, circle).
  - **segment** Clicking the left mouse button on the joint introduces the line location.
  - **arc** Use the combo list to choose a particular mode of defining an arc segment (third point, center, radius, height). Clicking the left mouse bottom on the desktop then selects points to define the arc. When selecting one of the following options - center, radius or included angle, you are further requested to select the orientation (positive, negative) from the combo list.

- **circle** Use the combo list to choose a particular mode of defining a circle (center and radius, three points). Clicking the left mouse button on the desktop then selects points to define the circle. The combo list is also used to select the orientation (positive, negative).
- **Modify** Clicking the left mouse button on already existing points opens the "**Modify properties of line**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing free points opens the **remove line** dialog window - accepting this action removes the selected line.
- **Select** Clicking the left mouse button on the line highlights the selected line. The line is simultaneously marked in the **table** list. The option allows for editing several lines at once (e.g. deleting).

The lines can also be edited on the desktop with the help of **active objects**.



Frame "Lines"

## Macroelements

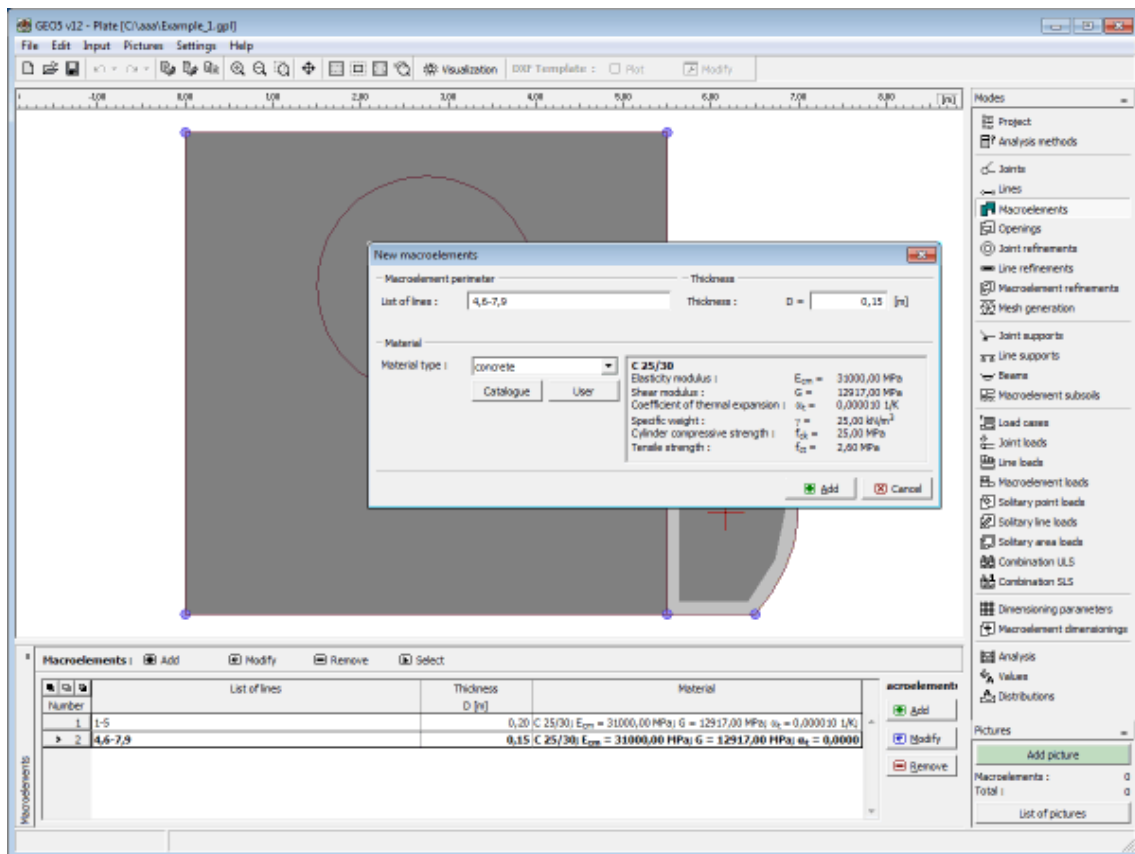
The **frame "Macroelements"** contains a **table** with the list of input macroelements. Adding (editing) macroelements is performed in the "**New macroelements**" dialog window. The dialog window servers to input a list of lines defining the macroelement outline, its thickness and material. The macroelement material can be either selected from the catalog of materials or its material parameters can be input manually using the "**Edit material**" dialog window.

Macroelements can also be introduced using mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Macroelements**". The following modes are

available:

- **Add** Clicking the left mouse button on the lines introduces the borders macroelement. Adding macroelement is performed in the "**New macroelement**" dialog window.
- **Modify** Clicking the left mouse button on already existing macroelement opens the "**Modify properties of macroelement**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the macroelement opens the **remove macroelement** dialog window - accepting this action removes the selected macroelement.
- **Select** Clicking the left mouse button on the macroelement highlights the selected macroelement. The macroelement is simultaneously marked in the **table** list. The option allows for editing several macroelements at once (e.g. deleting).

The input macroelements can also be edited on the desktop with the help of **active objects**.



Frame "Macroelements"

## Openings

The **frame "Openings"** contains a **table** with the list of input openings. Adding (editing) openings is performed in the "**New openings**" **dialog window**.

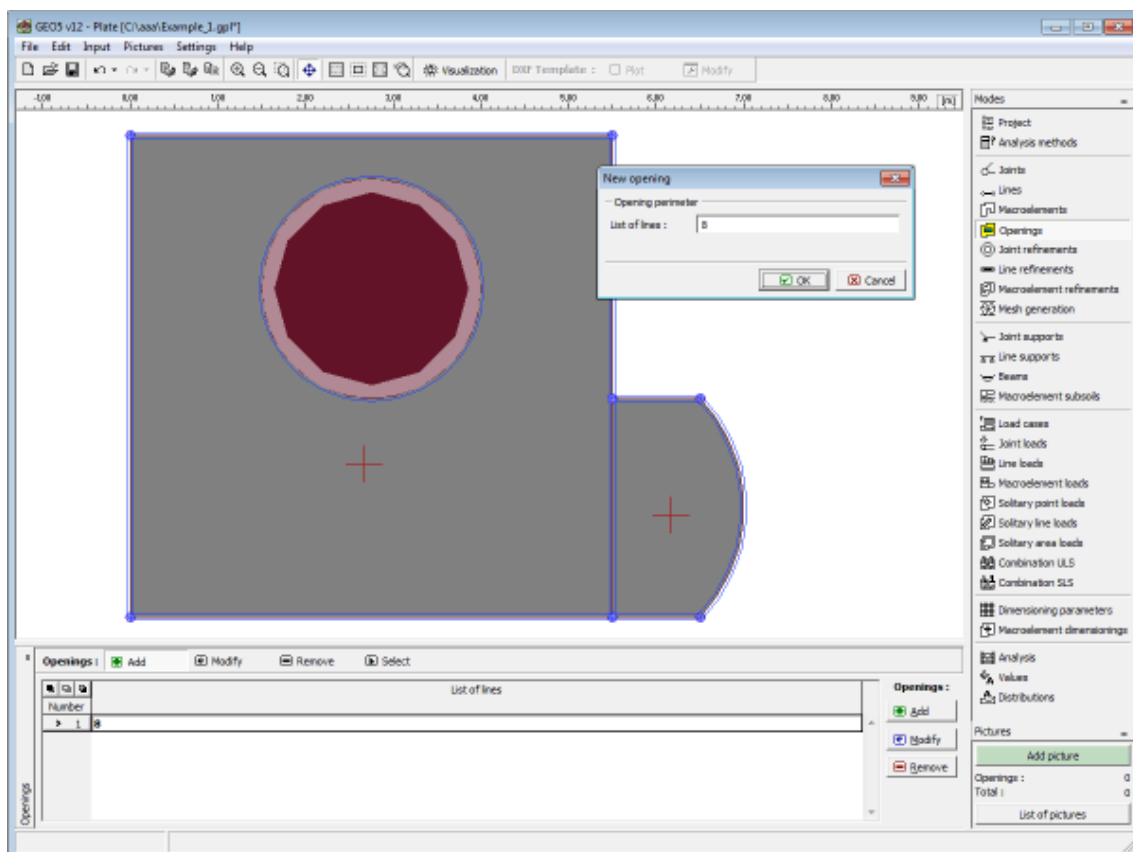
Openings can also be introduced using the mouse. This inputting mode is activated by clicking



an appropriate button on the horizontal tool bar "**Openings**". The following modes are available:

- **Add** Clicking the left mouse button on the lines introduces the borders openings. Adding opening is performed in the "**New opening**" dialog window.
- **Modify** Clicking the left mouse button on already existing macroelement opens the "**Modify properties of opening**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the opening opens the **remove opening** dialog window - accepting this action removes the selected opening.
- **Select** Clicking the left mouse button on the opening highlights the selected opening. The opening is simultaneously marked in the **table** list. The option allows for editing several openings at once (e.g. deleting).

The input openings can also be edited on the desktop with the help of **active objects**.



Frame "Openings"

## Joint Refinements

The **frame "Joint refinements"** contains a **table** with the list of input joint refinements. Adding (editing) joint refinements is performed in the "**New joint refinements**" **dialog window**.

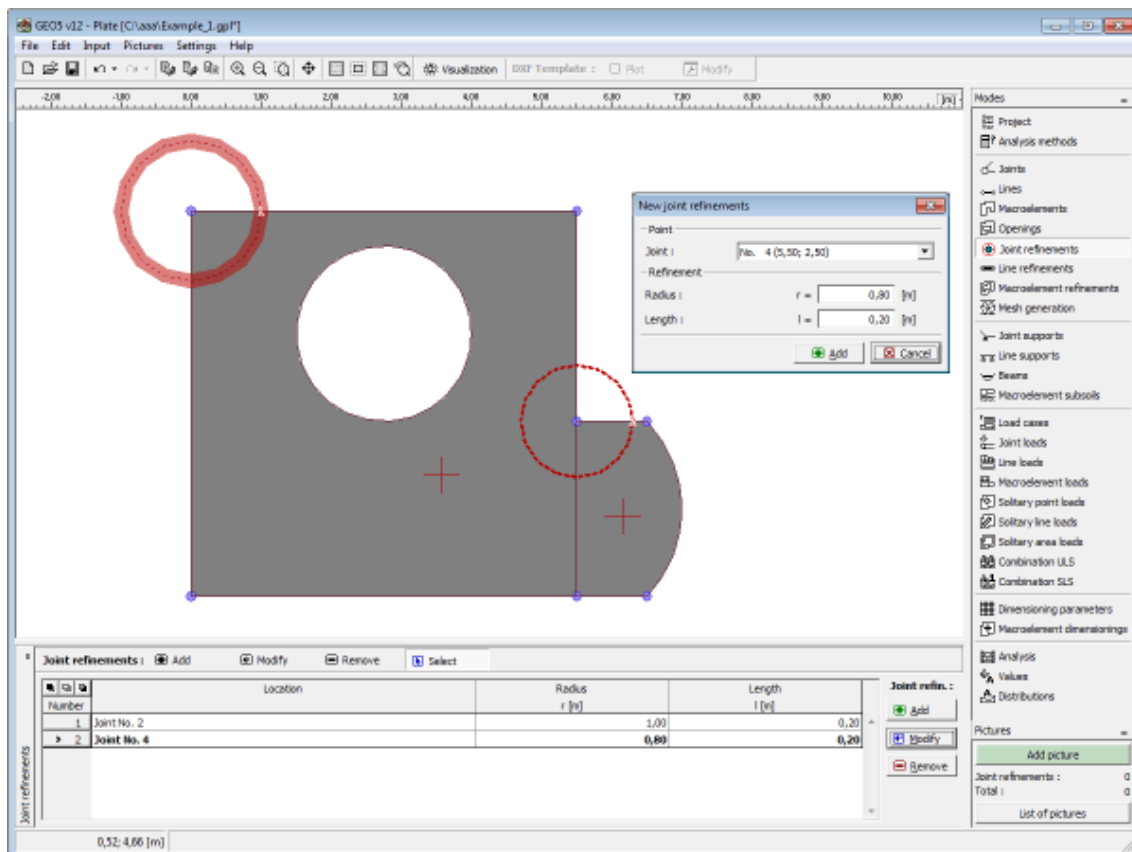
**Refining the finite element mesh around joints** is an important feature, which allows us to

create an appropriate **finite element mesh**.

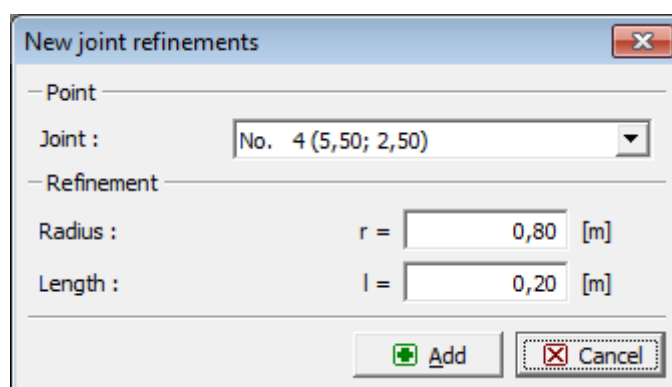
Joint refinements can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Joint refinements**". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the joint. Adding joint refinement is performed in the "**New joint refinement**" dialog window.
- **Modify** Clicking the left mouse button on already existing joint refinement opens the "**Modify properties of joint refinement**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the opening opens the **remove joint refinement** dialog window - accepting this action removes the selected joint refinement.
- **Select** Clicking the left mouse button on the joint refinement highlights the selected joint refinement. The joint refinement is simultaneously marked in the **table** list. The option allows for editing several joint refinements at once (e.g. deleting).

The input joint refinements can also be edited on the desktop with the help of **active objects**.



Frame "Joint refinements"



Dialog window "New joint refinements"

## Line Refinements

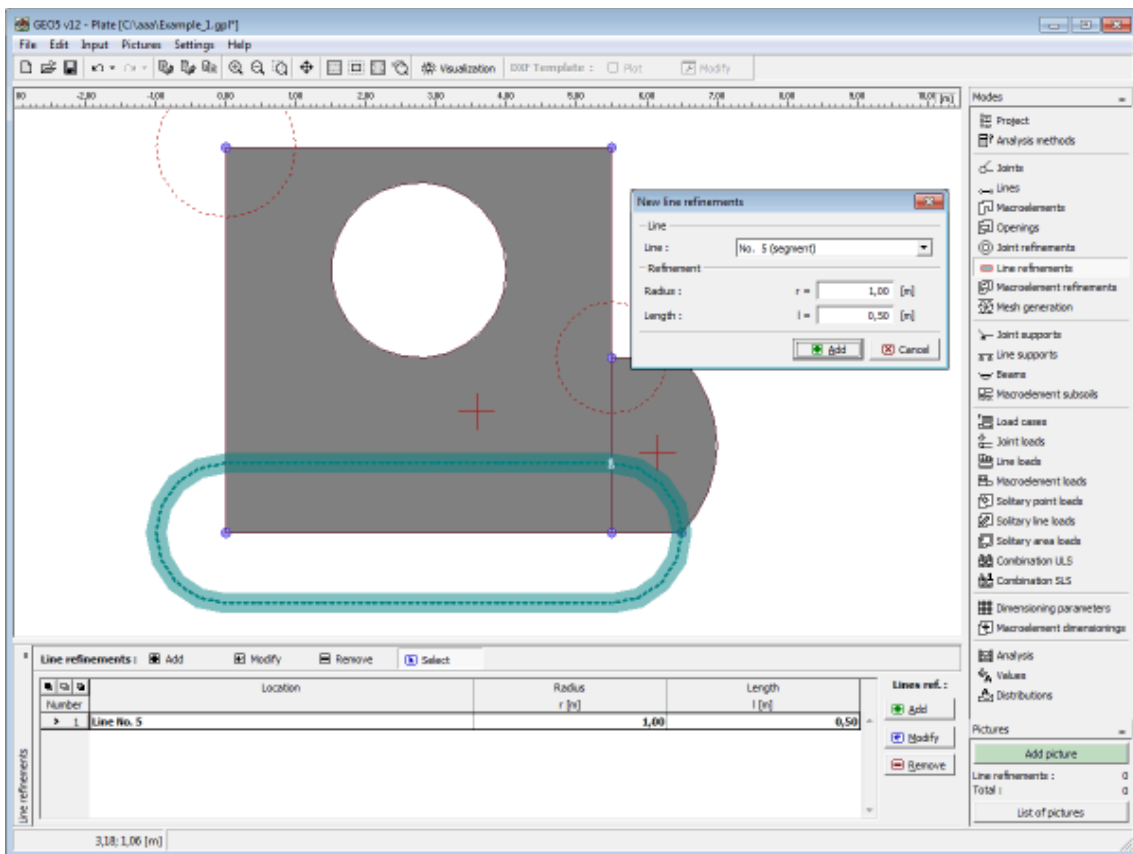
The frame "**Line refinements**" contains a table with the list of input line refinements. Adding (editing) line refinements is performed in the "**New line refinements**" dialog window.

**Refining the finite element mesh around lines** is an important feature, which allows us to create an appropriate finite element mesh.

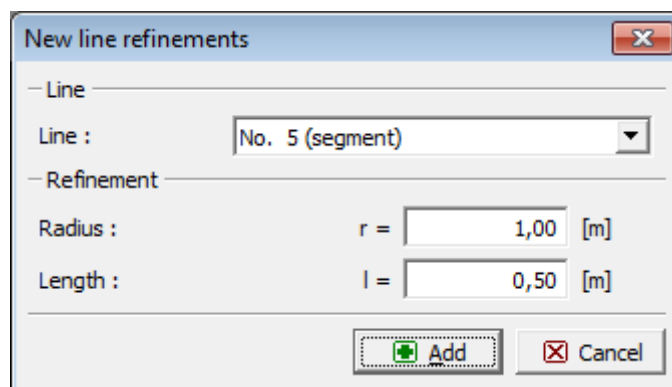
Line refinements can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Line refinements**". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the line. Adding line refinement is performed in the "**New line refinement**" dialog window.
- **Modify** Clicking the left mouse button on already existing line refinement opens the "**Modify properties of line refinement**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the line refinement opens the **remove line refinement** dialog window - accepting this action removes the selected line refinement.
- **Select** Clicking the left mouse button on the line refinement highlights the selected line refinement. The line refinement is simultaneously marked in the table list. The option allows for editing several line refinements at once (e.g. deleting).

The input line refinements can also be edited on the desktop with the help of active objects.



Frame "Line refinements"



Dialog window "New line refinements"

## Macroelement Refinements

The frame "Macroelement refinements" contains a table with the list of input macroelement refinements. Adding (editing) macroelement refinements is performed in the "New macroelement refinements" dialog window.

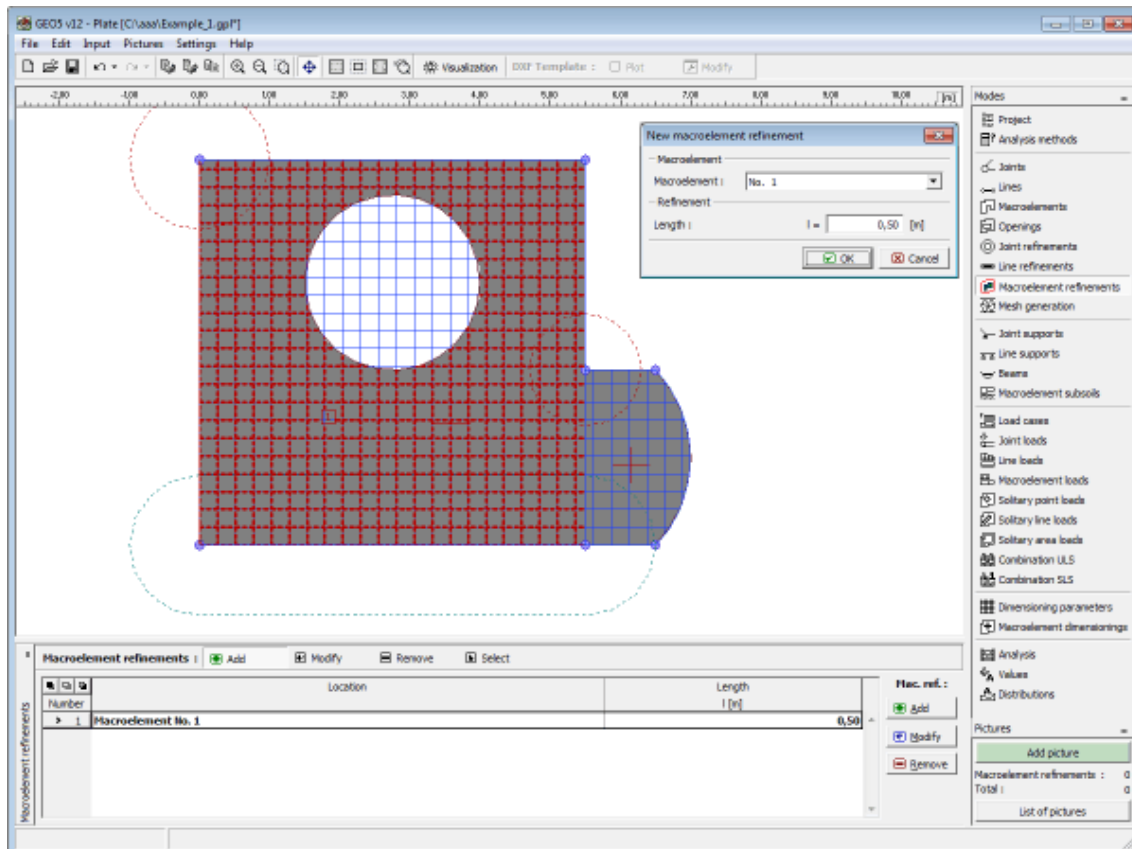
**Refining the finite element mesh of macroelements** is an important feature, which allows us to create an appropriate finite element mesh.

Macroelement refinements can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Macroelement

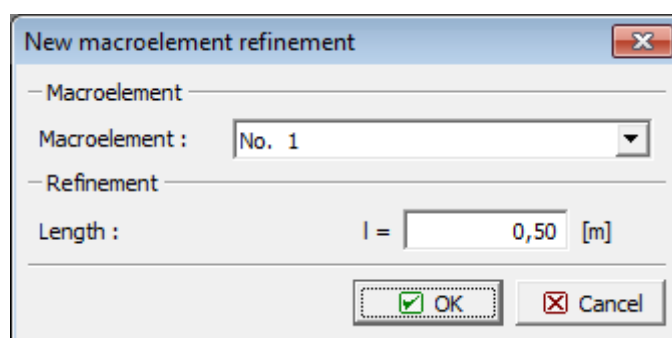
**refinements".** The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the macroelement. Adding line refinement is performed in the "**New macroelement refinement**" dialog window.
- **Modify** Clicking the left mouse button on already existing macroelement refinement opens the "**Modify properties of macroelement refinement**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the macroelement refinement opens the **remove macroelement refinement** dialog window - accepting this action removes the selected macroelement refinement.
- **Select** Clicking the left mouse button on the macroelement refinement highlights the selected macroelement refinement. The macroelement refinement is simultaneously marked in the **table** list. The option allows for editing several macroelement refinements at once (e.g. deleting).

The input macroelement refinements can also be edited on the desktop with the help of **active objects**.



Frame "Macroelement Line refinements"



Dialog window "New macroelement refinements"

## Mesh Generation

The frame "**Mesh generator**" serves to define the basic setting to generate mesh (element edge length, mesh type, mesh smoothing) and to view information about the generated mesh (right part). The "**Error analysis**" button allows for visualization of error listing in the right part of the frame (list of problems the structure has).

Information about the resulting **mesh including warnings** for possible weak points in the mesh is displayed in the right bottom window.

An arbitrary part of the slab specified by **lines** (segments, arches and circles) can be meshed. The slab can be formed by one or more **macroelements** all having a constant thickness and identical material properties and may contain an arbitrary number of **openings**. In addition, it is possible to introduce internal points and lines which are then considered as mesh nodes and edges. The **joints** along **lines** and inside **macroelements** allow for mesh refinement, which is characterized by the required length of element edges in the center of the refinement and by the refinement radius. The user may choose either a purely triangular mesh or a hybrid mesh consisting of both triangular and quadrilateral elements. The meshing algorithm is based on Delaunay triangulation enhanced by several methods to modify and optimize the finite element mesh. The mesh nodes are automatically renumbered to minimize the computational effort.

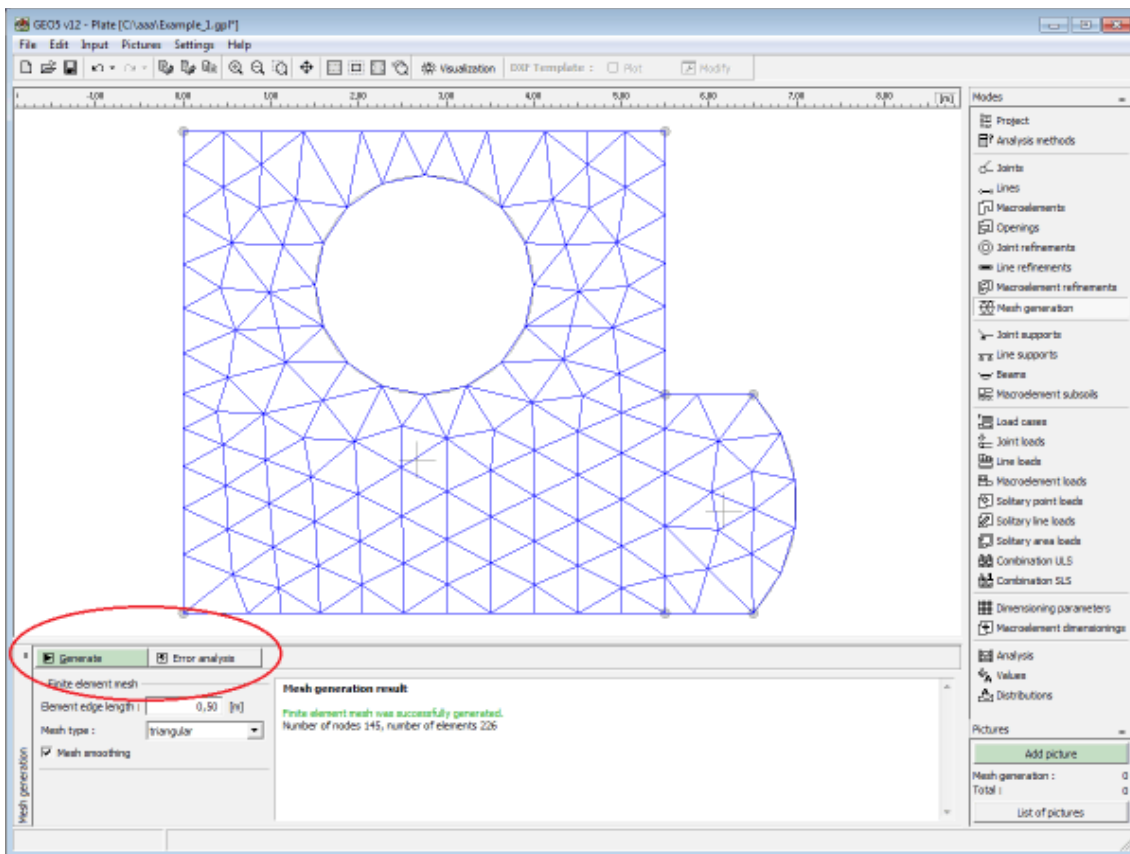
Properly generated finite element mesh is the stepping stone for obtaining accurate results. Optimal are equilateral triangular and square quadrilateral elements. The program contains a built-in automatic mesh generator considerably simplifying this task. The basic mesh density is specified in the "**Mesh generator**" window. Refining the mesh increases accuracy of the results. However, high mesh density considerably slows down both the solution and subsequent visualization of the results. The goal is thus to create an optimally refined mesh - this strongly depends on user's experience.

Thanks to efficiency of the mesh generator there is no problem to adjust input parameters until obtaining an optimal mesh. The mesh quality is further maintained with the help of built-in smoothing algorithm, which can be turned off. The actual analysis step is extremely fast even for relatively dense meshes.

### **The following procedure to generate the finite element mesh is recommended:**

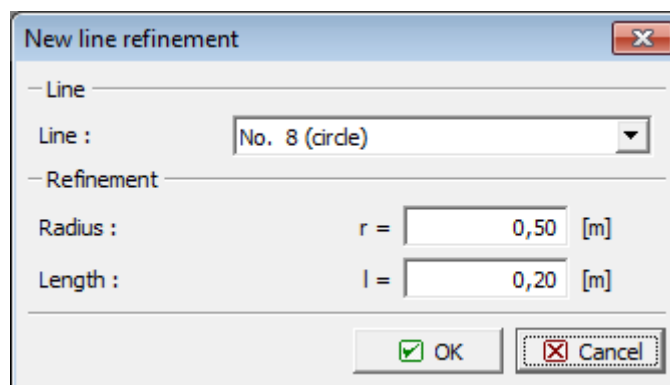
Correctly generated finite element mesh is the major step in achieving accurate and reliable results. The program FEM has an automatic mesh generator, which may substantially simplify this task. Nevertheless, **certain rules should be followed** when creating a finite element mesh:

- First, a uniform mesh linked to the slab thickness (1-5 multiple of its thickness) is generated throughout the slab.

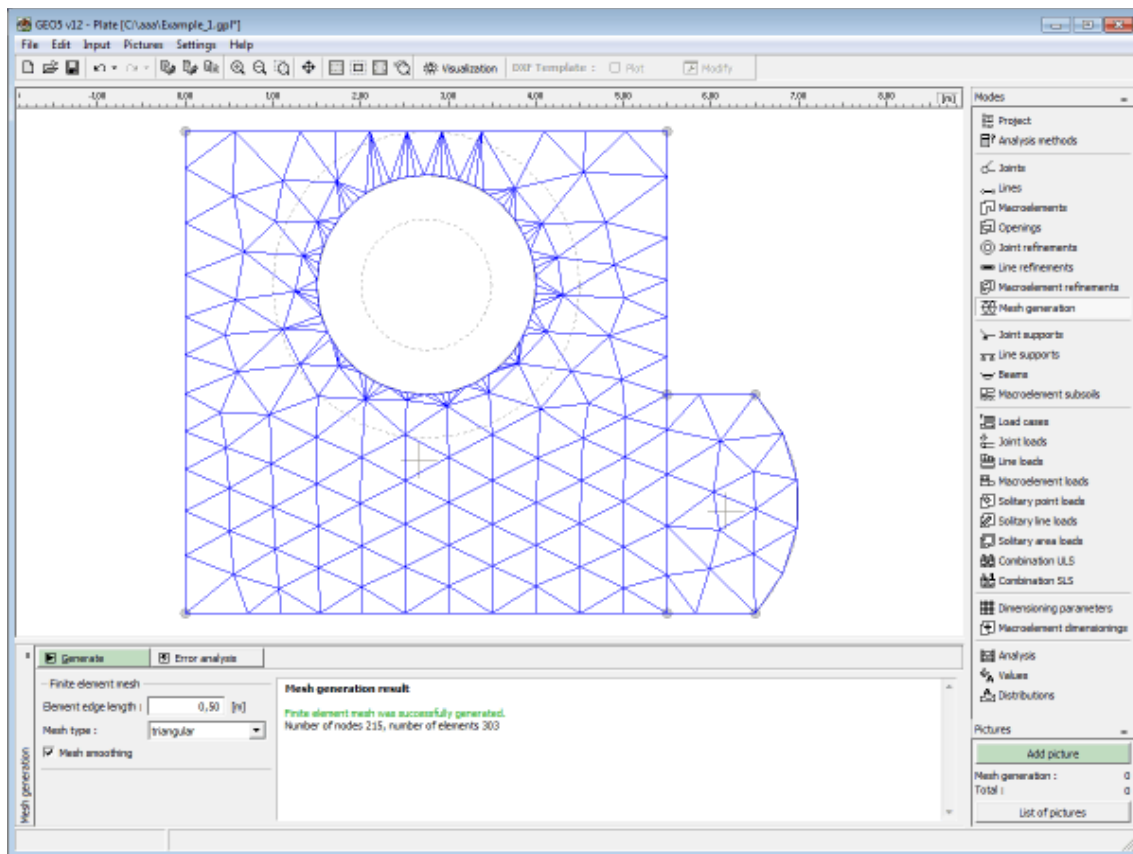


Frame "Mesh generation" - a mesh with no local refinement

- The finite element mesh should be sufficiently fine in the locations where large stress gradients are expected (point supports, corners, openings, etc.). The mesh refinement can be specified around individual **joints**, **lines** and on the **macroelements**. Its radius should be at least 2-3 multiple of the density assumed in the center of the refinement and both values (density, radius) should be reasonable with respect to the refinement prescribed for the neighboring regions. This assures a smooth transition between regions with different mesh densities. Singular lines should be tackled in the same way.



Defining mesh refinement around a circular line



*New mesh after refining the original mesh around a circular line*

## Mesh Generator Warning

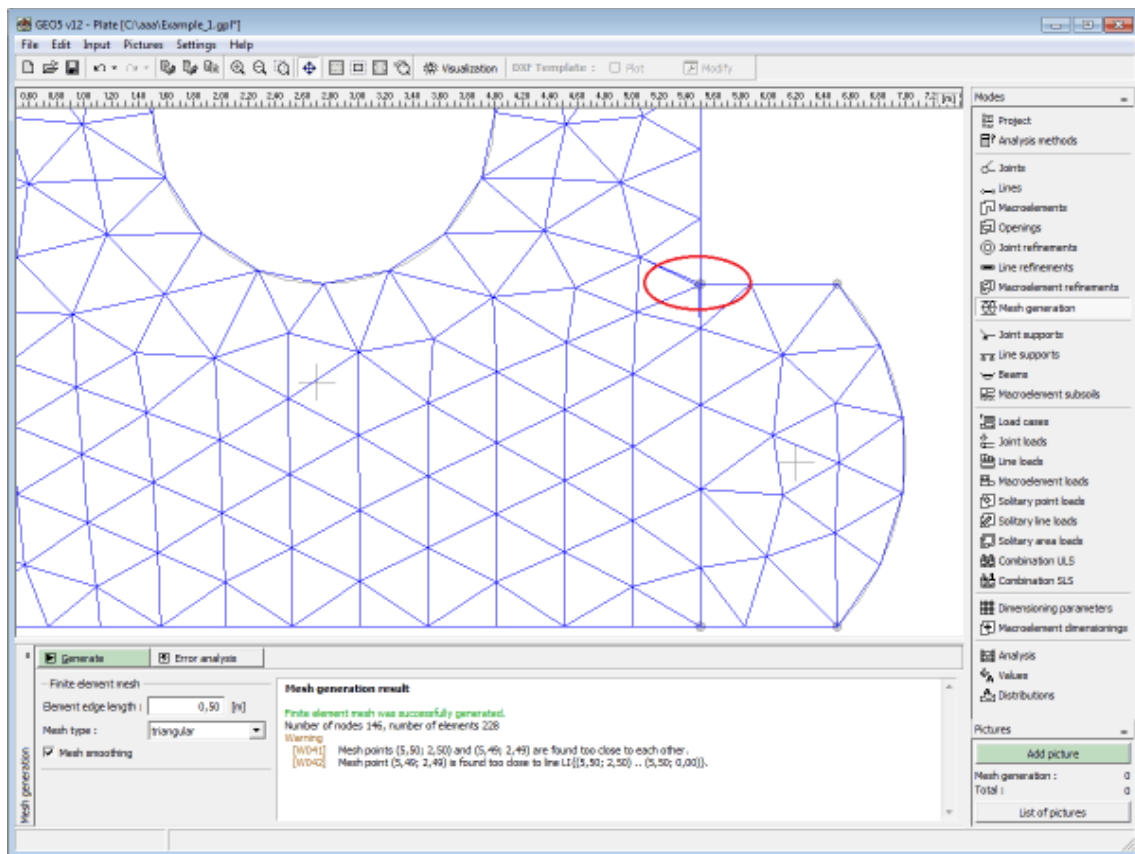
In the "**Mesh generation result**" dialog window the user is prompted for possible locations on the structure that may cause problems during automatic mesh generation. When positioning the cursor on individual warnings the corresponding critical region on a structure is highlighted with a red color. The following items are checked:

- whether the distance between two points is greater than one tenth of the required element edge length
- whether the distance between a point and a line is greater than one tenth of the element edge length
- whether the area of a region is greater than twice the element edge length
- whether points and/or lines are found inside the structure (in the soil)

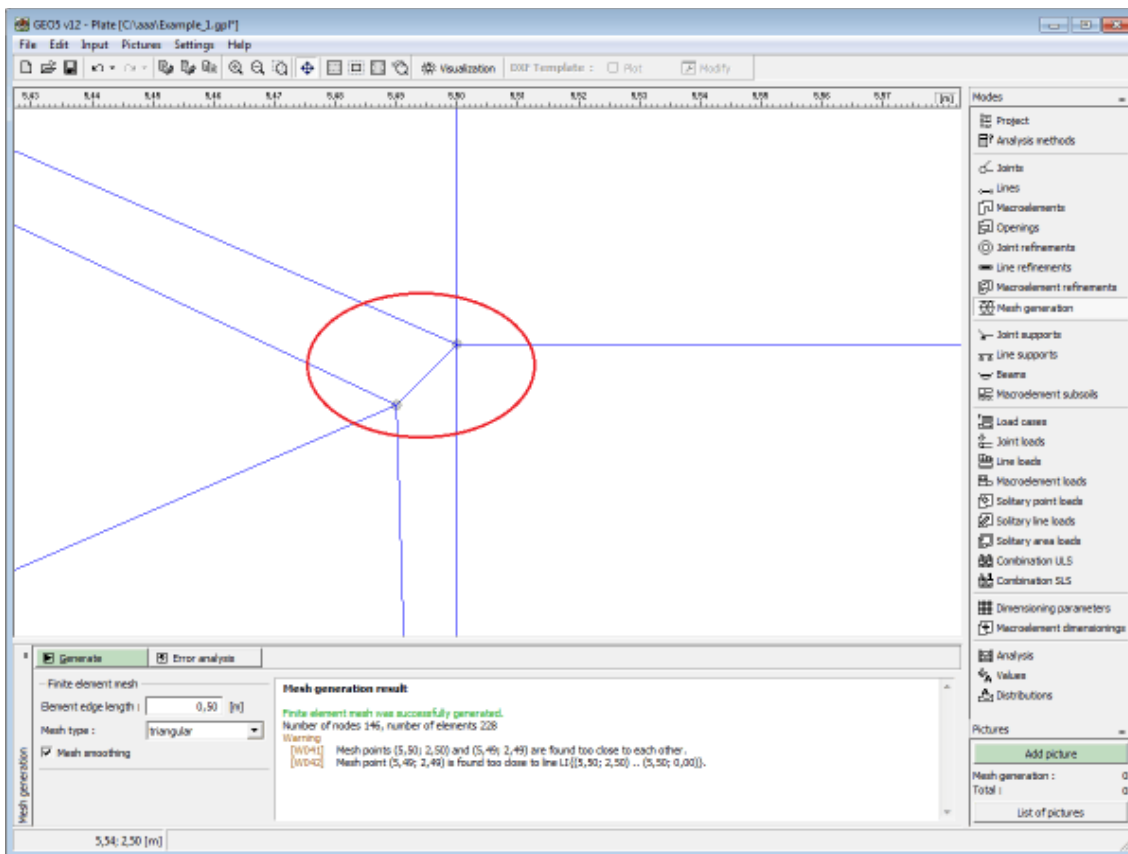
These warnings suggest locations, in which the mesh generator experience problems. The following possibilities may occur:

- the mesh is not generated => this calls for a new input of geometrical data
- the mesh is generated => in this case it is up to the user to decide whether the mesh is reasonable - in any case, the warning can be further ignored and the analysis can be carried out





*Warning after identifying critical sections in FE mesh*



*Critical section after zooming in - two points are too close to each other*

## Joint Supports

The frame **"Joint supports"** contains a table with the list of input joint supports. Adding (editing) joint supports is performed in the **"New joint supports"** dialog window.

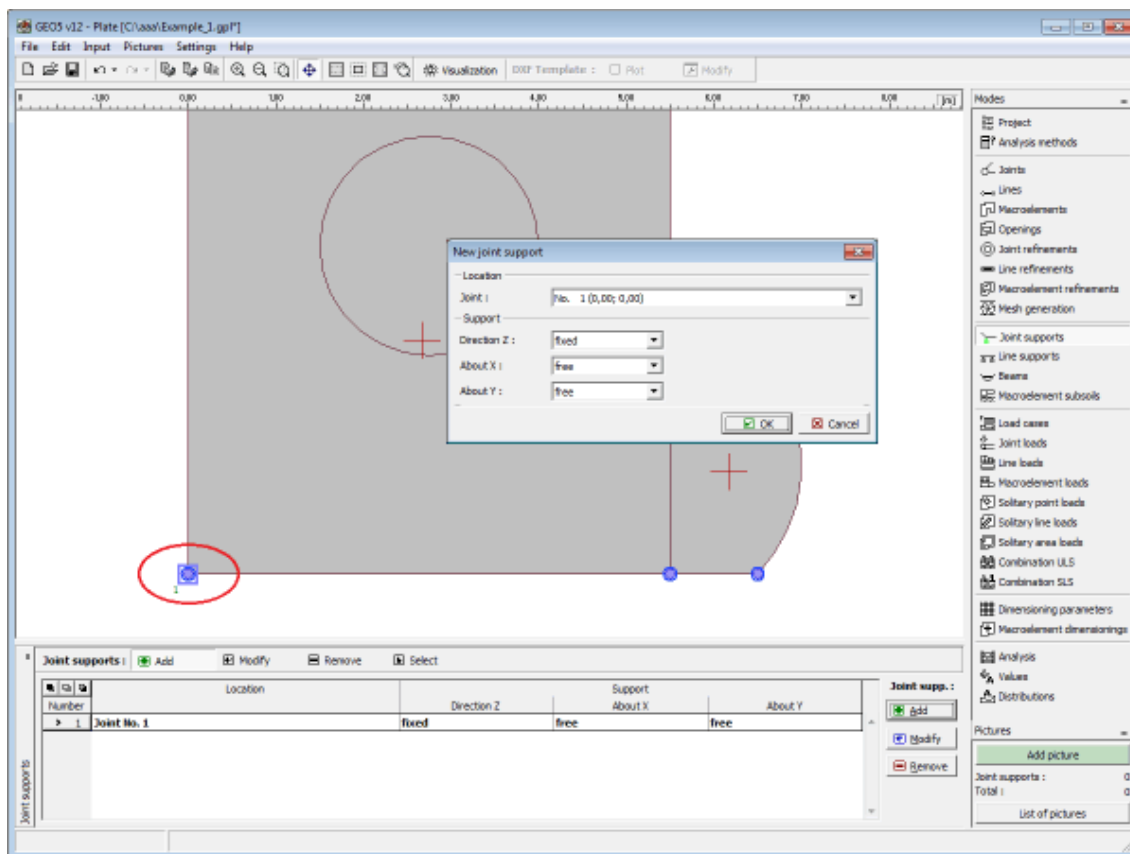
Joint supports can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Joint supports"**. The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the joint location. Adding (editing) joint support is performed in the **"New joint supports"** dialog window.
- **Modify** Clicking the left mouse button on already existing joint support opens the **"Modify properties of joint support"** dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the joint support opens the **remove joint support** dialog window - accepting this action removes the selected joint support.
- **Select** Clicking the left mouse button on the joint support highlights the selected joint support. The joint support is simultaneously marked in the table list. The option allows for editing several joint supports at once (e.g. deleting).

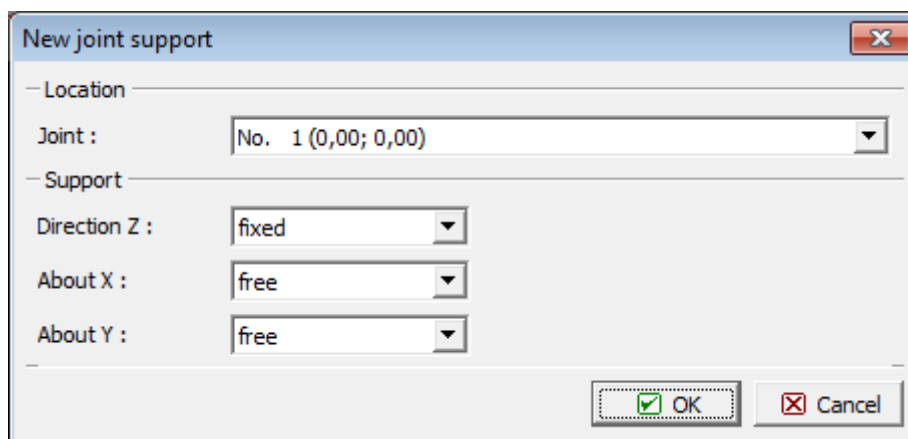
The input joint supports can also be edited on the desktop with the help of **active objects**.

The following types of joint supports are considered:

- free
- fixed
- spring



Frame "Joint supports"



Dialog window "New joint supports"

## Line Supports

The frame "**Line supports**" contains a table with the list of input line supports. Adding (editing) line supports is performed in the "**New supports of lines**" dialog window.

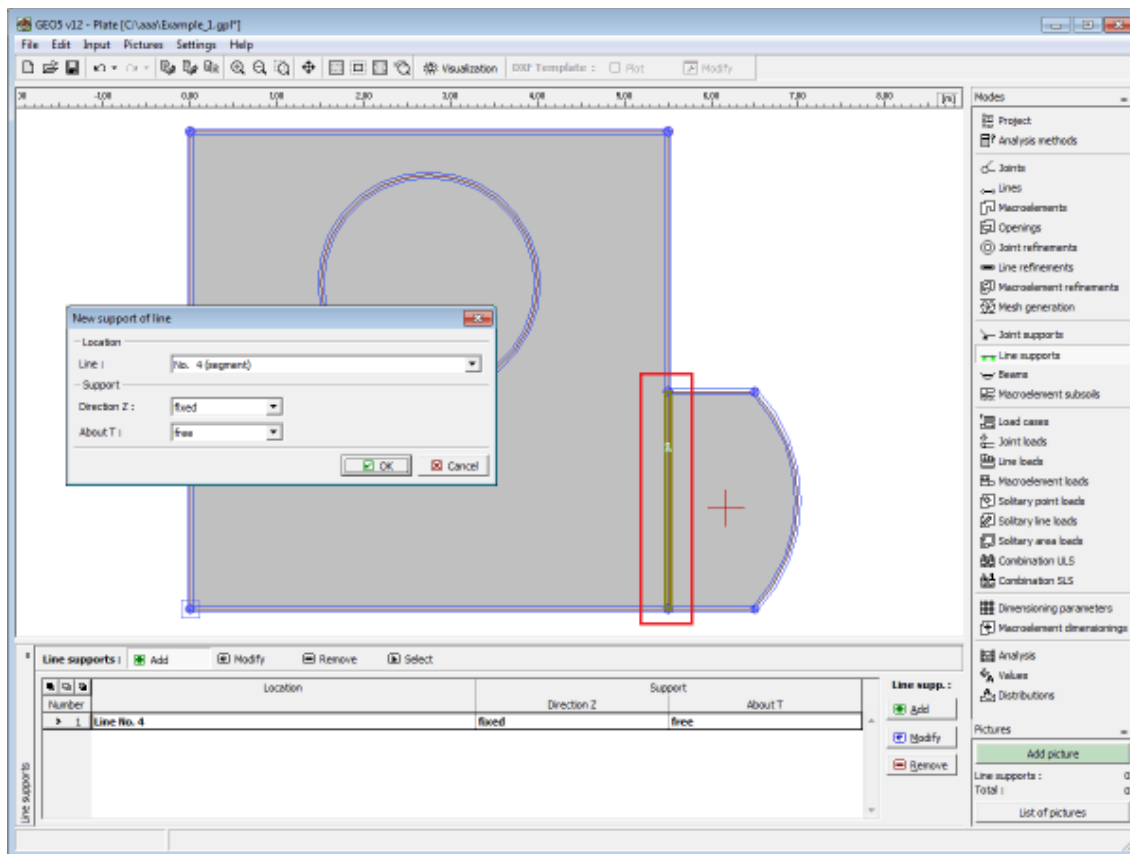
Line supports can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Line supports**". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the line support location. Adding line support is performed in the "**New supports of lines**" dialog window.
- **Modify** Clicking the left mouse button on already existing line support opens the "**Modify properties of support of line**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the line support opens the **remove line support** dialog window - accepting this action removes the selected line support.
- **Select** Clicking the left mouse button on the line support highlights the selected line support. The line support is simultaneously marked in the table list. The option allows for editing several line supports at once (e.g. deleting).

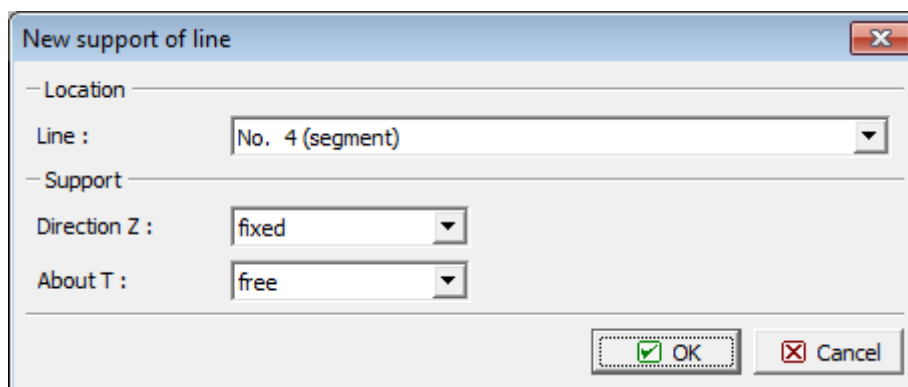
The input line supports can also be edited on the desktop with the help of **active objects**.

**The following types of line supports** are considered:

- free
- fixed
- spring



Frame "Line supports"



Dialog window "New supports of lines"

## Beams

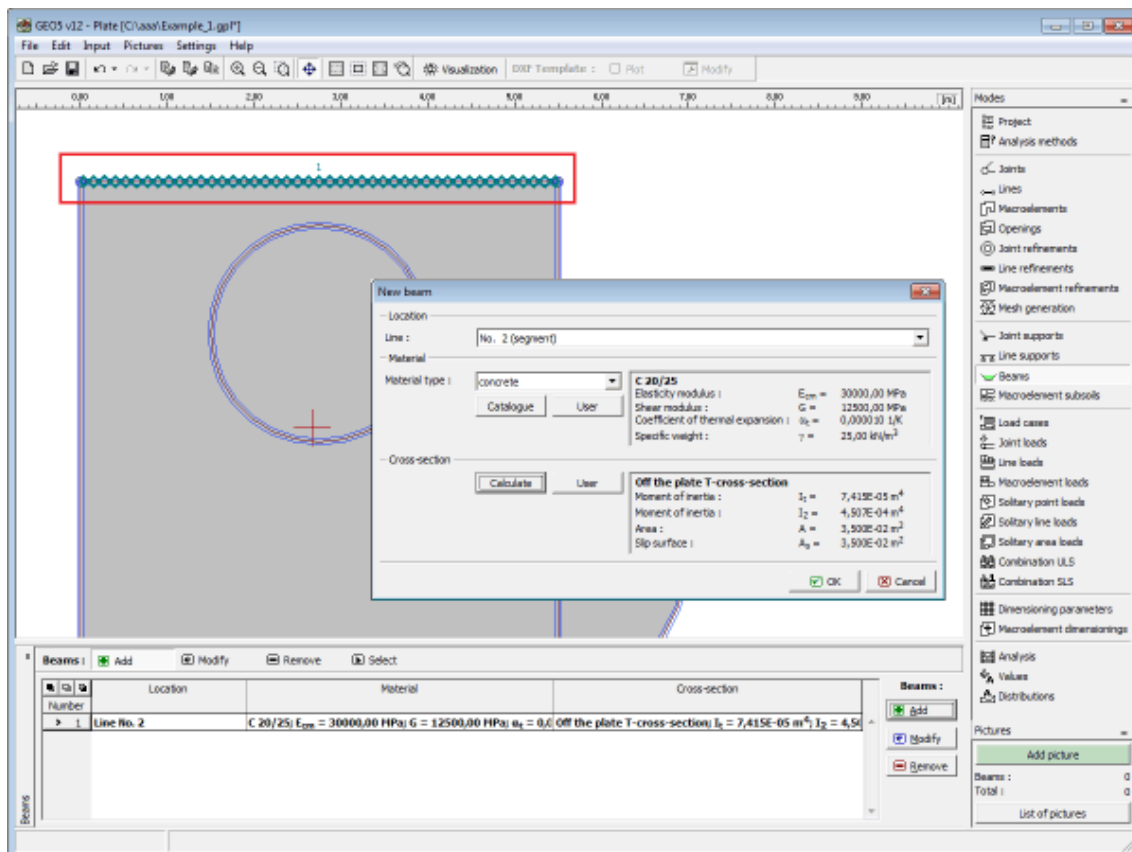
The frame **"Beams"** contains a table with the list of beams. Adding (editing) beams is performed in the **"New beams"** dialog window.

The dialog window serves to define the line number of the beam location and material and cross-section of the beam. Choosing the type of material (concrete, steel, other) then allows for assigning the material parameters either from the **catalog of materials** or manually using **editor of materials**. The cross-section parameters (based on **type of cross-section**) can be either calculated in the window **"Calculation of cross-sectional parameters"** or manually input in the **"Input of cross-sectional parameters"** window.

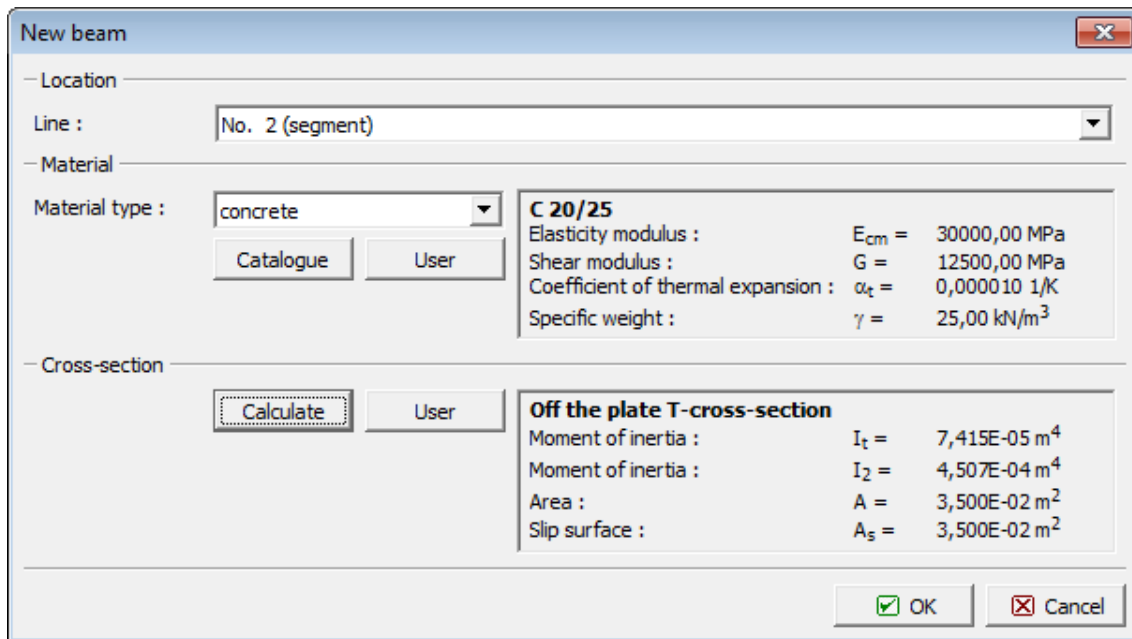
Beams can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Beams**". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the beam location. Adding beam is performed in the "**New beams**" dialog window.
- **Modify** Clicking the left mouse button on already existing beam opens the "**Modify properties of beam**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the beam opens the **remove beam** dialog window - accepting this action removes the selected beam.
- **Select** Clicking the left mouse button on the beam highlights the selected beam. The beam is simultaneously marked in the **table** list. The option allows for editing several beams at once (e.g. deleting).

The input beams can also be edited on the desktop with the help of **active objects**.



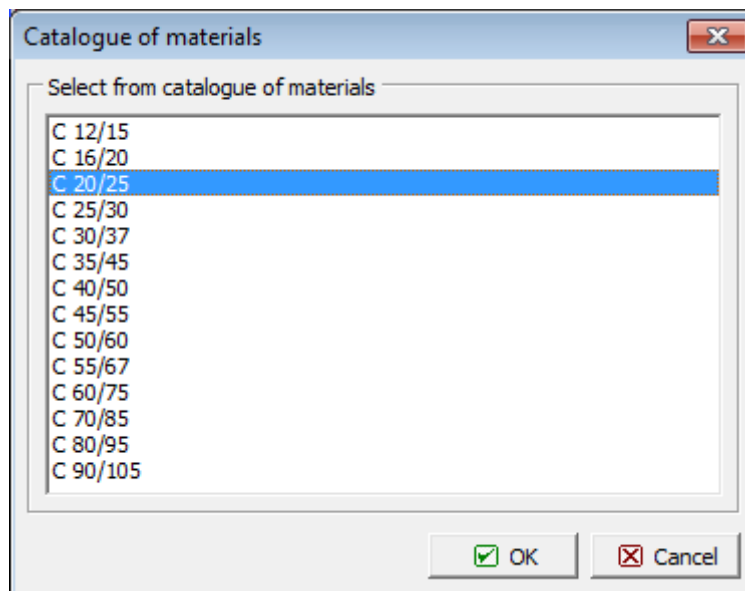
Frame "Beams"



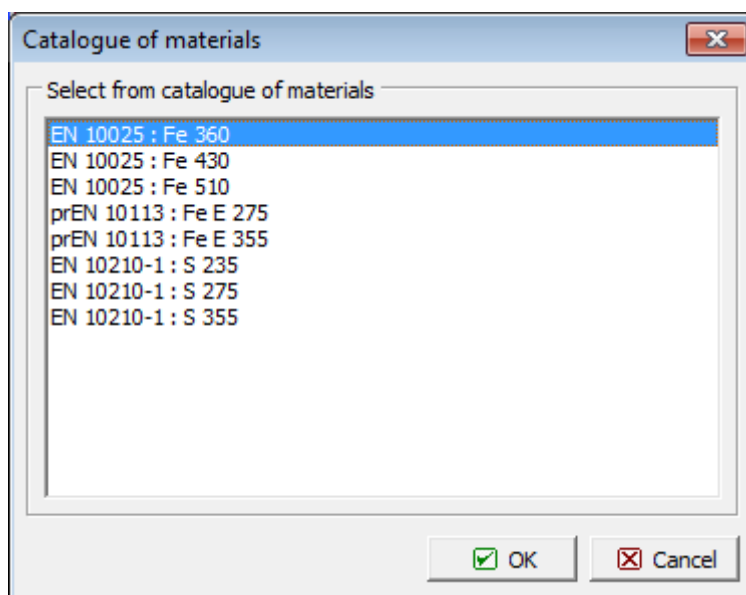
Dialog window "New beams"

## Catalog of Materials

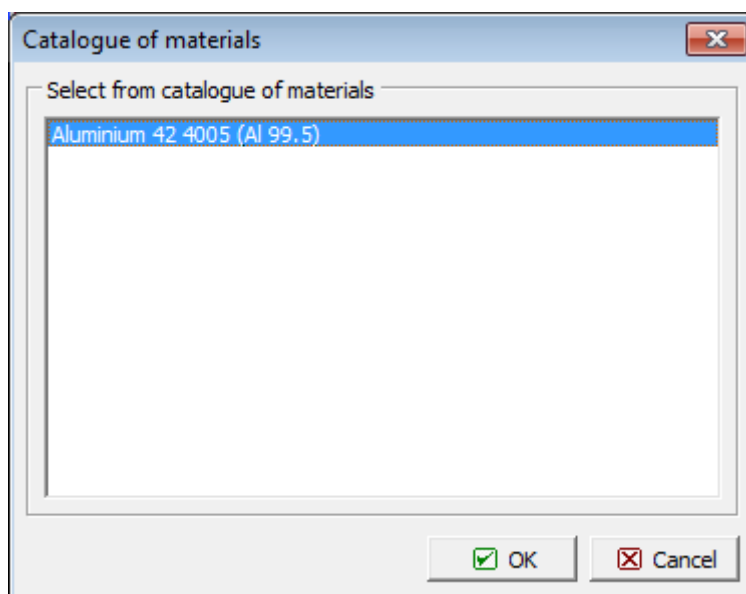
The program contains a built-in catalog of materials for concrete, steel and other materials. Only the type of material has to be specified in the dialog window. The **type of cross-section** is selected from the "**Calculation of cross-sectional parameters**" dialog window or "**Input of cross-sectional parameters**" dialog window.



Dialog window "Catalog of materials" - concrete



*Dialog window "Catalog of materials" - steel*

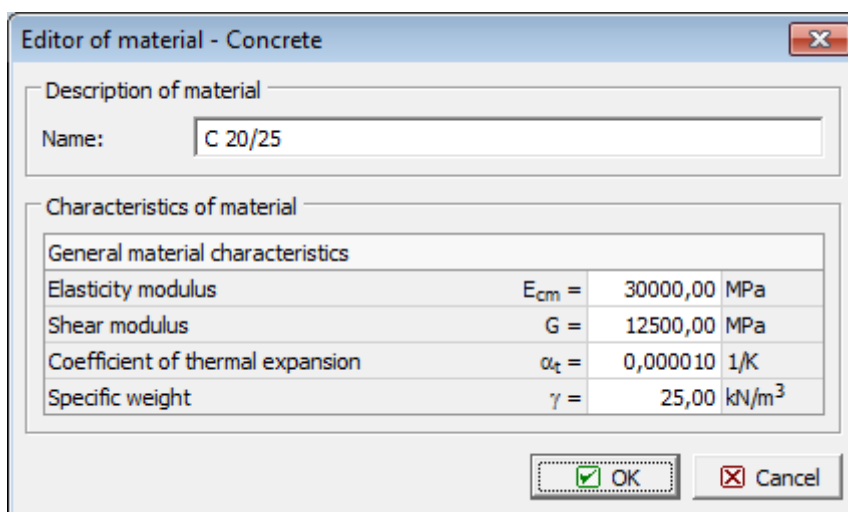


*Dialog window "Catalog of materials" - other*

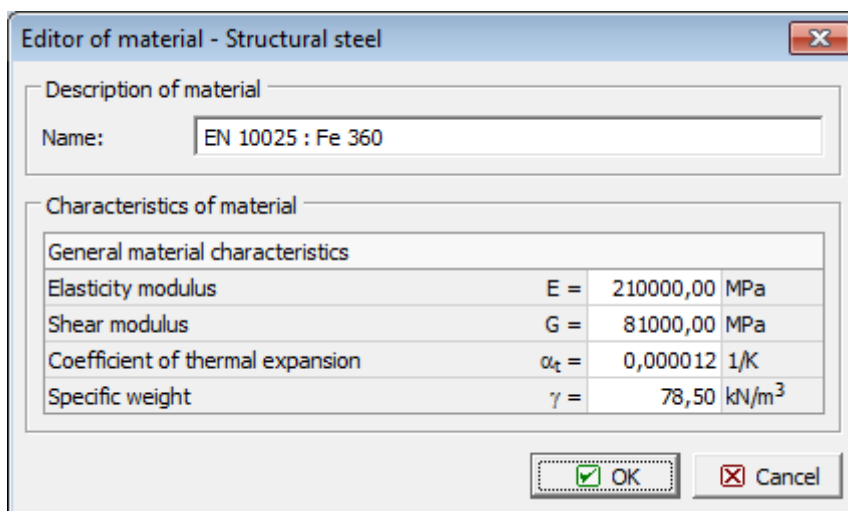
## Editor of Materials

Apart from using the "Catalog of materials" the program allows the user to enter the material parameters for steel, concrete and other materials (dialog window "**Editor of material - General**") digitally. Only the type of material (material parameters) has to be specified in the dialog window. The **type of cross-section** is selected from the "**Calculation of cross-sectional parameters**" dialog window or "**Input of cross-sectional parameters**" dialog window.

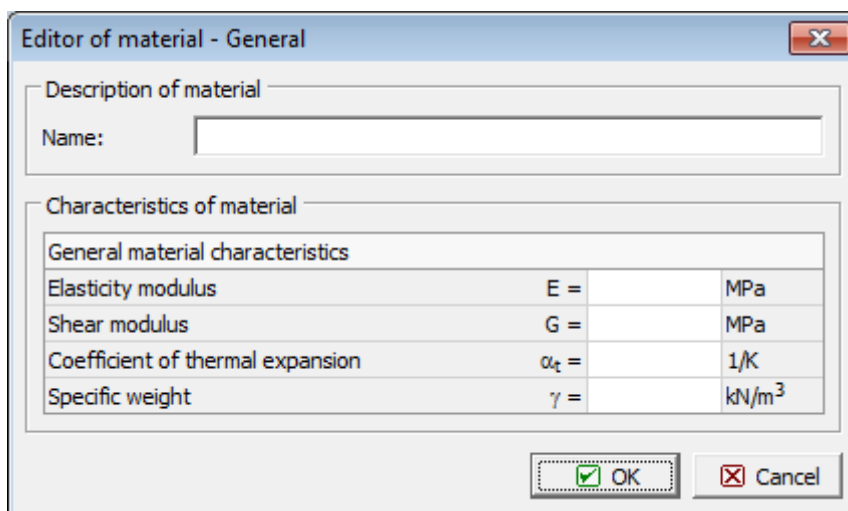




*Dialog window "Editor of material - Concrete"*



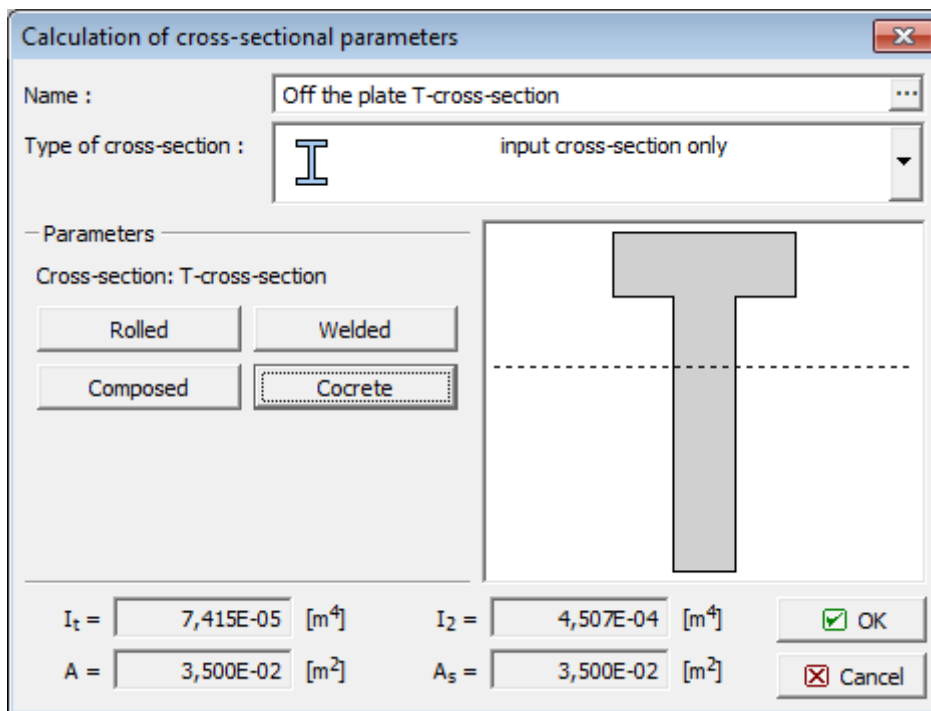
*Dialog window "Editor of material - Structural steel"*



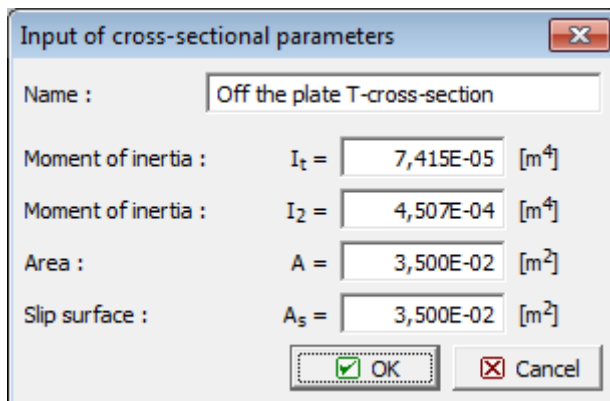
*Dialog window "Editor of material - General"*

## Types of Cross-Section

The program allows the user to either input the **cross-section parameters** in "Calculation of cross-sectional parameters" and "Input of cross-sectional parameters" dialog windows. The cross-sectional characteristics are selected from the [catalog of profiles](#), [cross-section editor](#) dialog windows.



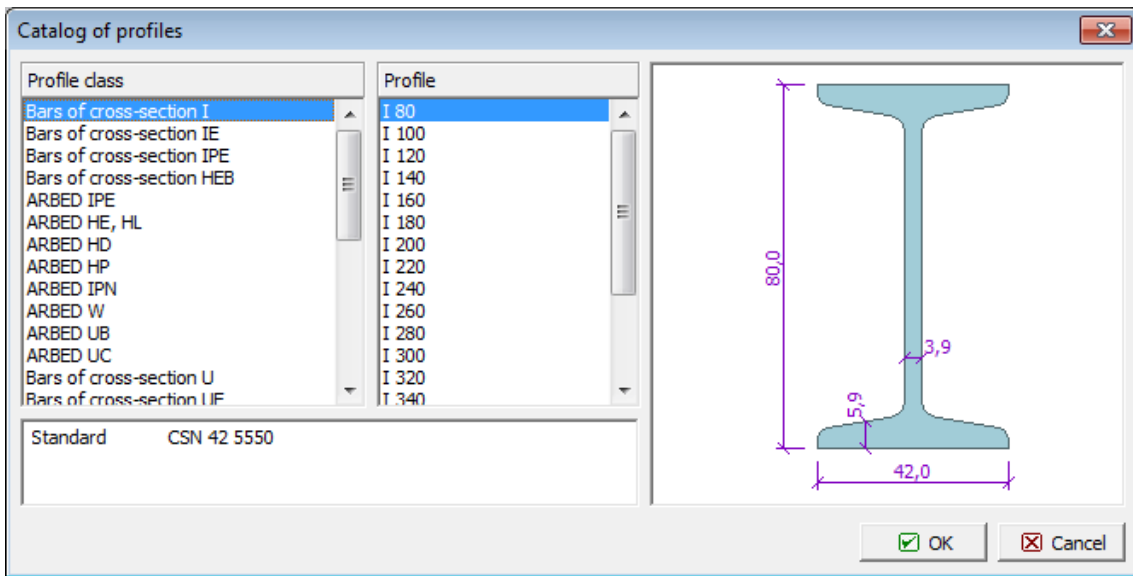
Dialog window "Calculation of cross-sectional parameters"



Dialog window "Input of cross-sectional parameters"

## Catalog of Profiles

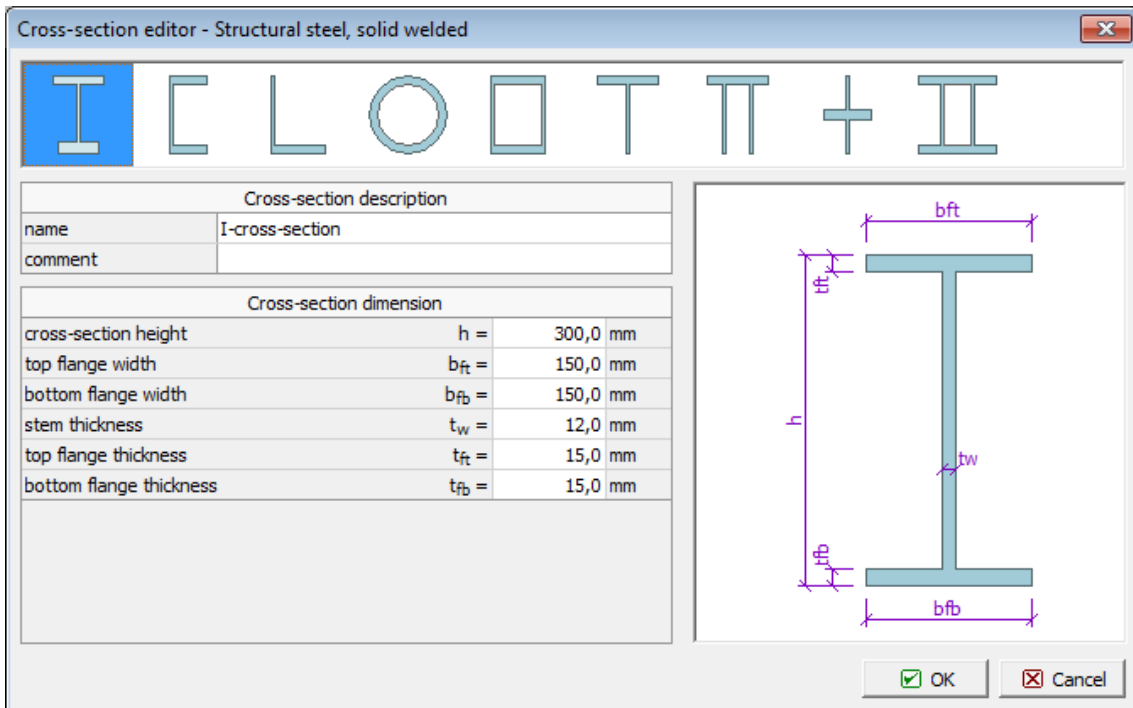
In the case of steel cross-sections the program allows for choosing a particular cross-section from the catalog of profiles. Only the type of cross-section has to be specified in the dialog window. The type of material of the cross-section is selected from the "Catalog of materials", or defined in the "Editor of materials".



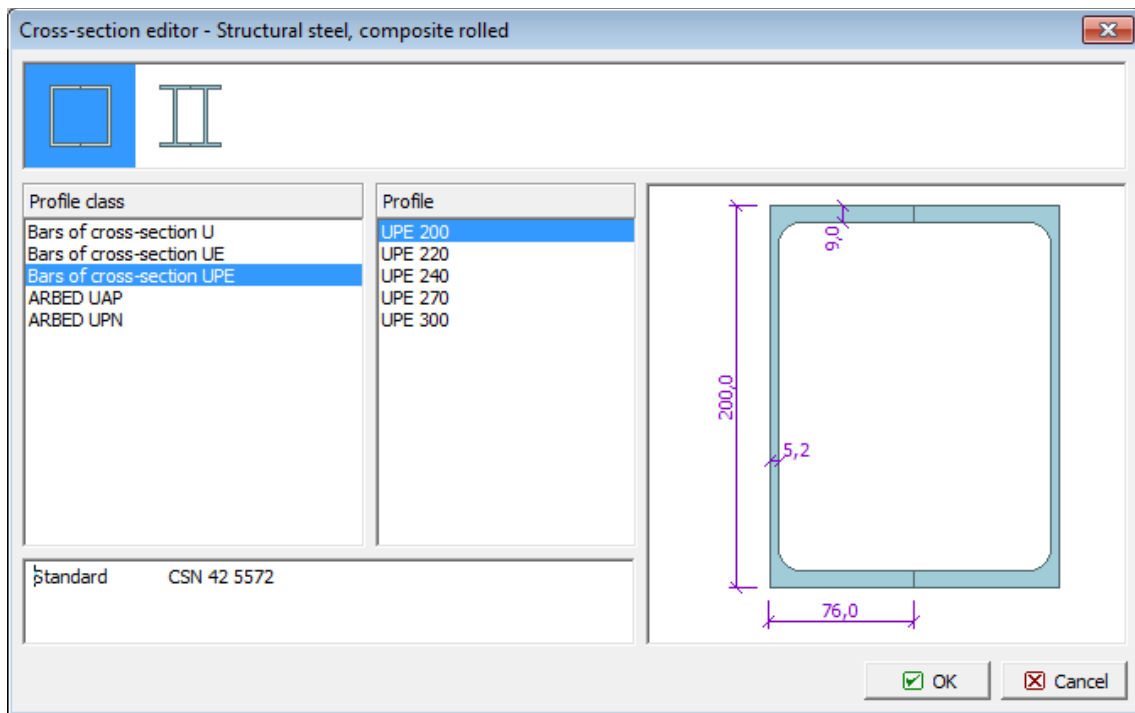
Dialog window "Catalog of profiles"

## Cross-Section Editor

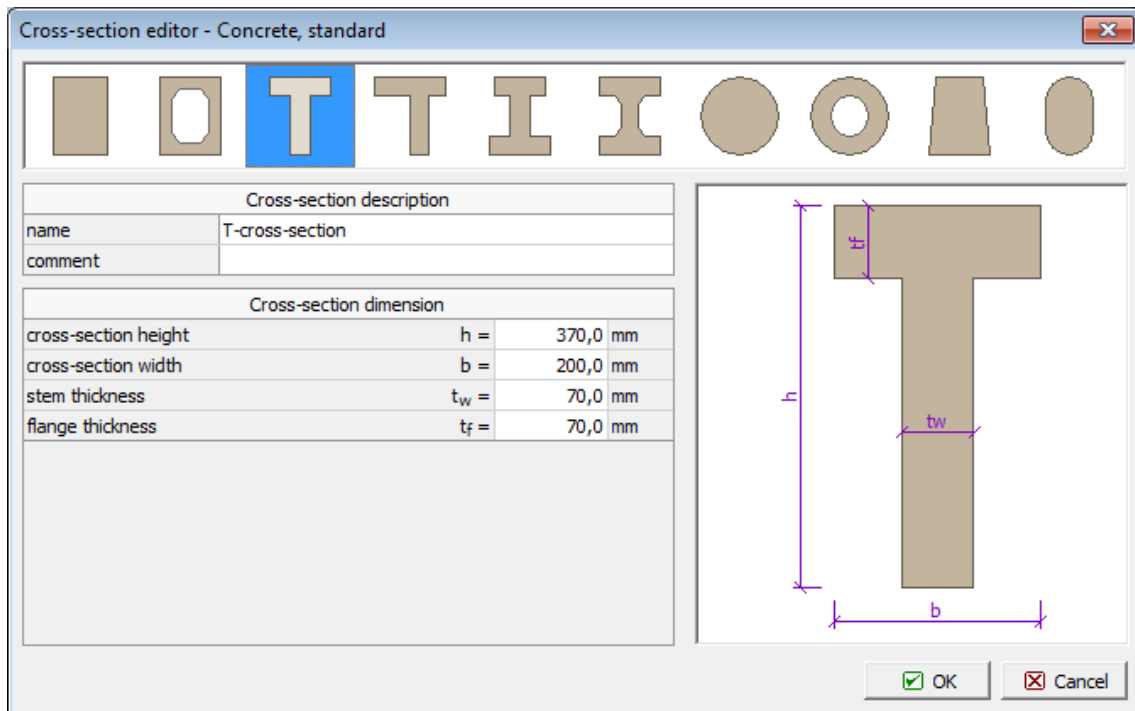
In the case of steel and concrete cross-section the program allows for introducing the user defined cross-section. Only the shape of cross-section has to be specified in the dialog window. The cross-sectional characteristics are selected from the [catalog of materials](#), [editor of materials](#) dialog windows.



Dialog window "Cross-section editor - Structural steel, solid welded"



Dialog window "Cross-section editor - Structural steel, composite rolled"



Dialog window "Cross-section editor - Concrete, standard"

## Internal Hinges

The frame "**Internal hinges**" contains a table with the list of internal hinges. Adding (editing) internal hinges is performed in the "**New internal hinges**" dialog window. The dialog window serves to define the line number of the internal hinges location and type of internal hinge

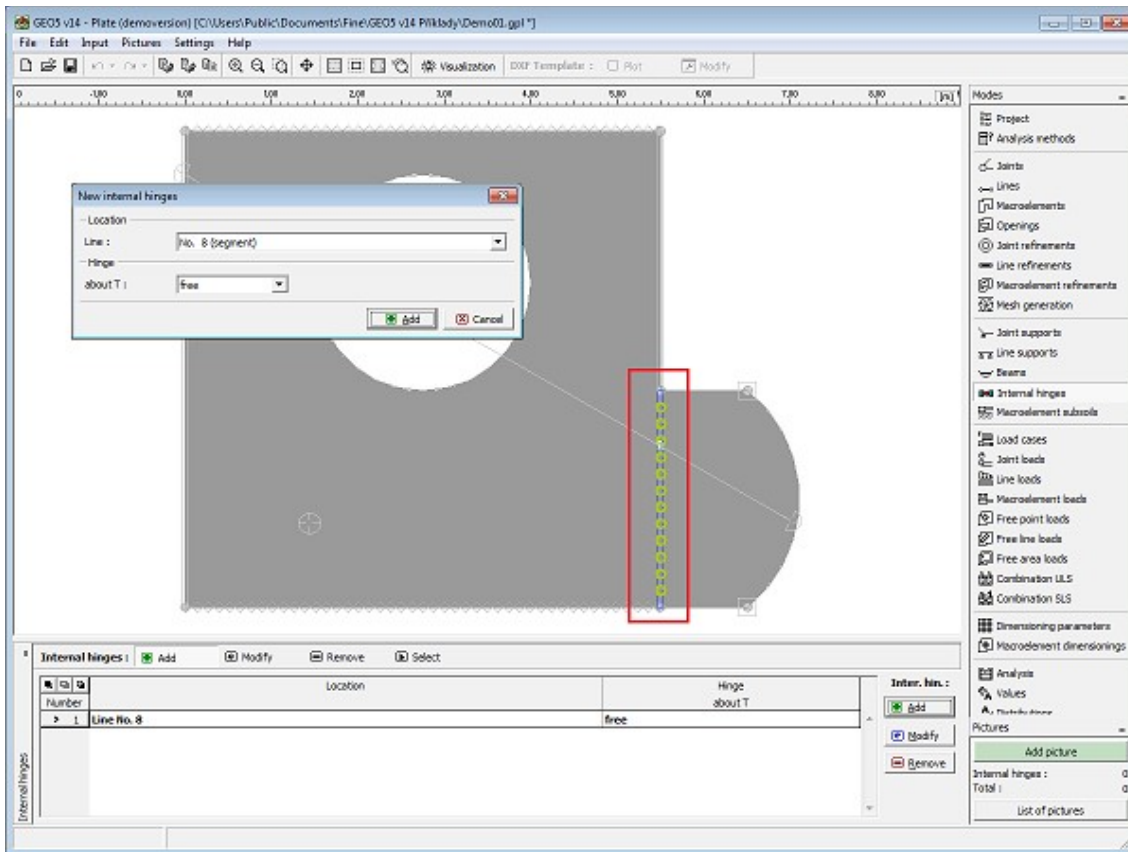
(free, spring).

The internal hinge is a boundary condition that allows for the introduction of independent rotation about the  $x$  and  $y$  axis between the two parts of the slab along a specified line while keeping the vertical deflection along this line the same. The internal hinge can be prescribed along an arbitrary line creating a boundary between two macroelements. The rotation can be either free or controlled by the spring torsional stiffness  $K_{\phi,T}$ .

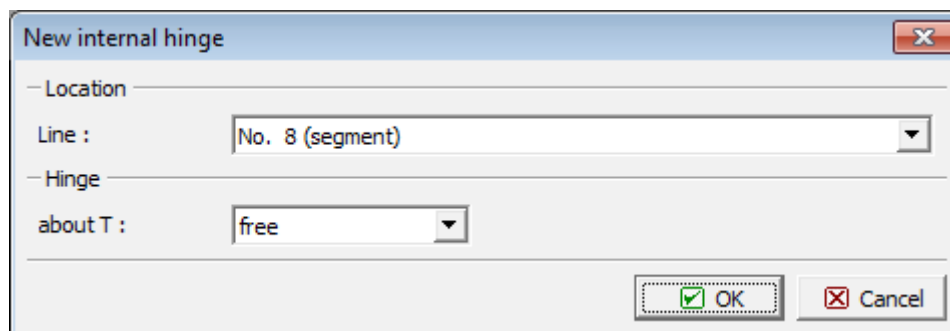
Internal hinges can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Internal hinges**". The following modes are available:

- Add** Clicking the left mouse button on the desktop introduces the internal hinge location. Adding internal hinge is performed in the "**New internal hinge**" dialog window.
- Modify** Clicking the left mouse button on already existing internal hinges opens the "**Modify properties of internal hinge**" dialog window, which allows for modifying its parameters.
- Remove** Clicking the left mouse button on the internal hinge opens the **remove internal hinges** dialog window - accepting this action removes the selected internal hinge.
- Select** Clicking the left mouse button on the internal hinge highlights the selected internal hinge. The internal hinge is simultaneously marked in the **table** list. The option allows for editing several internal hinges at once (e.g. deleting).

The input internal hinges can also be edited on the desktop with the help of **active objects**.



Frame "Internal hinges"



Dialog window "New internal hinge"

## Macroelement Subsoils

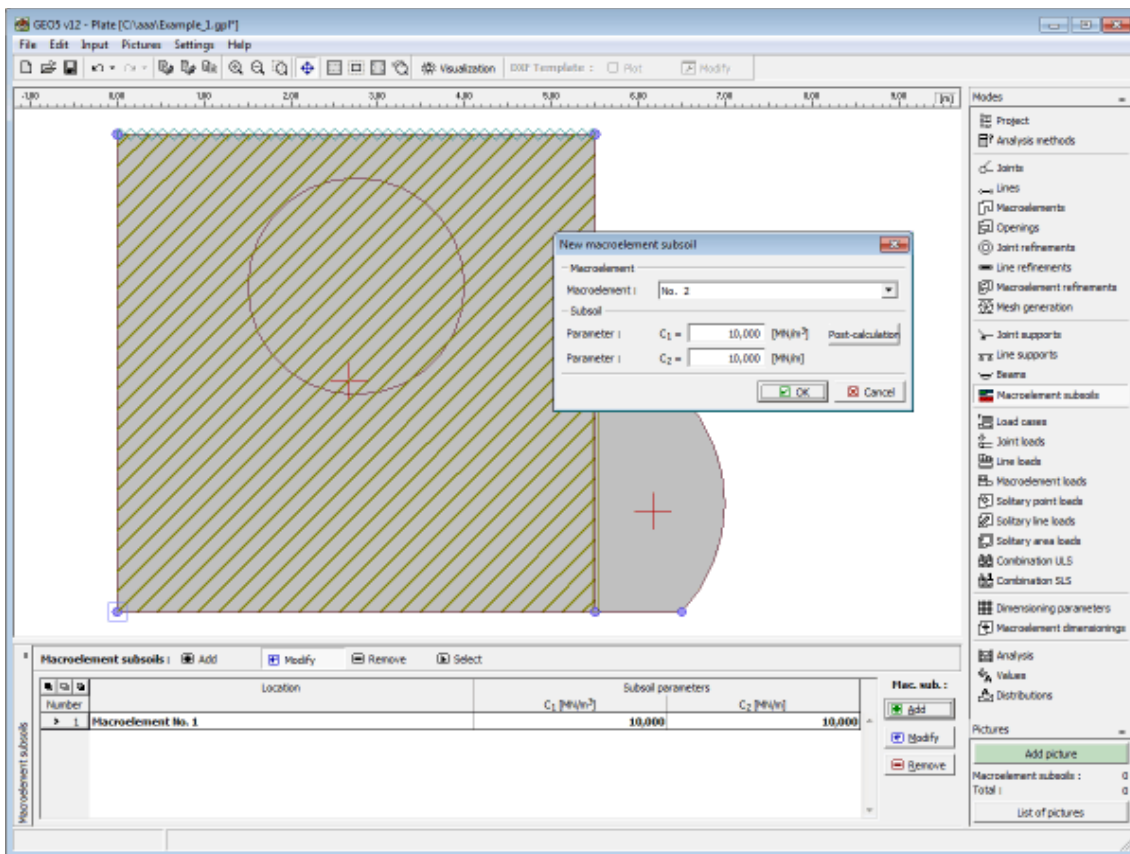
The frame "**Macroelement subsoils**" contains a **table** with the list of input macroelement subsoils. Adding (editing) line supports is performed in the "**New macroelement subsoils**" dialog window.

The dialog window serves to define the macroelement number and parameters  $C_1$  and  $C_2$ . The **Winkler-Pasternak constants**  $C_1$  and  $C_2$  can be specified either directly or **calculated** by the program. The latter option further requires inputting deformation parameters of soils (deformation modulus, Poisson's ratio and depth of influence zone) in the "**Compute  $C_1$  and  $C_2$** " dialog window. These parameters can be determined using the program "**Spread footing**" (2. limit state) and introduced into the program.

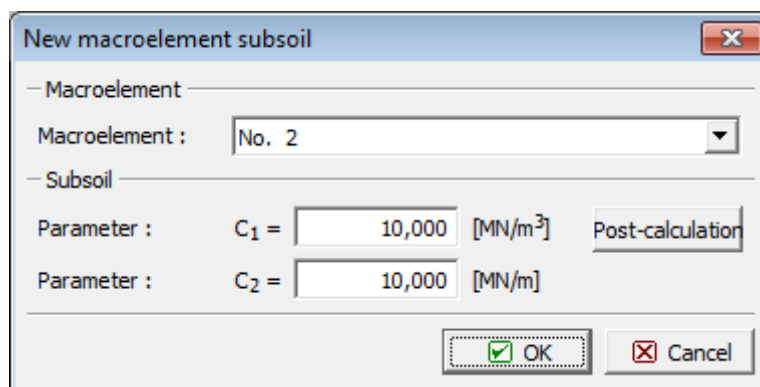
Macroelement subsoils supports can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Macroelement subsoils**". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the macroelement subsoil location. Adding macroelement subsoil support is performed in the "**New macroelement subsoils**" dialog window.
- **Modify** Clicking the left mouse button on already existing macroelement subsoil opens the "**Modify properties of macroelement subsoil**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the macroelement subsoil opens the **remove macroelement subsoil** dialog window - accepting this action removes the selected macroelement subsoil.
- **Select** Clicking the left mouse button on the macroelement subsoil highlights the selected macroelement subsoil. The macroelement subsoil is simultaneously marked in the **table** list. The option allows for editing several macroelement subsoil at once (e.g. deleting).

The input macroelement subsoils can also be edited on the desktop with the help of **active objects**.



Frame "Macroelement subsoils"



Dialog window "New macroelement subsoils"

## Winkler-Pasternak Parameters C1 a C2

The Winkler - Pasternak model for the solution of an elastic layer introduces the balance equation in the vertical direction as:

$$c_1 \cdot w - c_2 \cdot \Delta w = f_z$$

where:  $c_1, c_2$  constants characterizing the Winkler - Pasternak model  
 $w$  displacement in the vertical direction

$f_z$  vertical load acting on a layer

The elastic subsoil is introduced into the program using local stiffness matrices which are added to the stiffness matrices of individual elements resting on the subsoil. The contact stress  $\sigma$  is provided as an output.

## Calculation of Winkler-Pasternak Constants from Deformation Parameters of Soils

The Winkler - Pasternak constants  $C_1$  and  $C_2$  are calculated in the program from the condition of equal compliance matrices of infinitely stiff infinite strip footing resting on the Winkler - Pasternak and elastic subsoil. This condition is represented by the following equalities:

$$[C] = \begin{bmatrix} \frac{1}{2 \left[ \sqrt{C_{1WP} C_{2WP}} + b C_{1WP} \right]} & 0 \\ 0 & \frac{1}{2 \left[ b^2 \sqrt{C_{1WP} C_{2WP}} + b C_{2WP} + \frac{b^3}{3} C_{1WP} \right]} \end{bmatrix} = \begin{bmatrix} \sum_{n=0}^{\infty} \frac{1}{2\sqrt{H} \left[ (2n+1) \sqrt{C_{1w} C_{2w}} + (2n+1)^2 b C_{1w} \right]} & 0 \\ 0 & \sum_{n=0}^{\infty} \frac{1}{2\sqrt{H} \left[ (2n+1) b^2 \sqrt{C_{1w} C_{2w}} + b C_{2w} + (2n+1)^2 \frac{b^3}{3} C_{1w} \right]} \end{bmatrix}$$

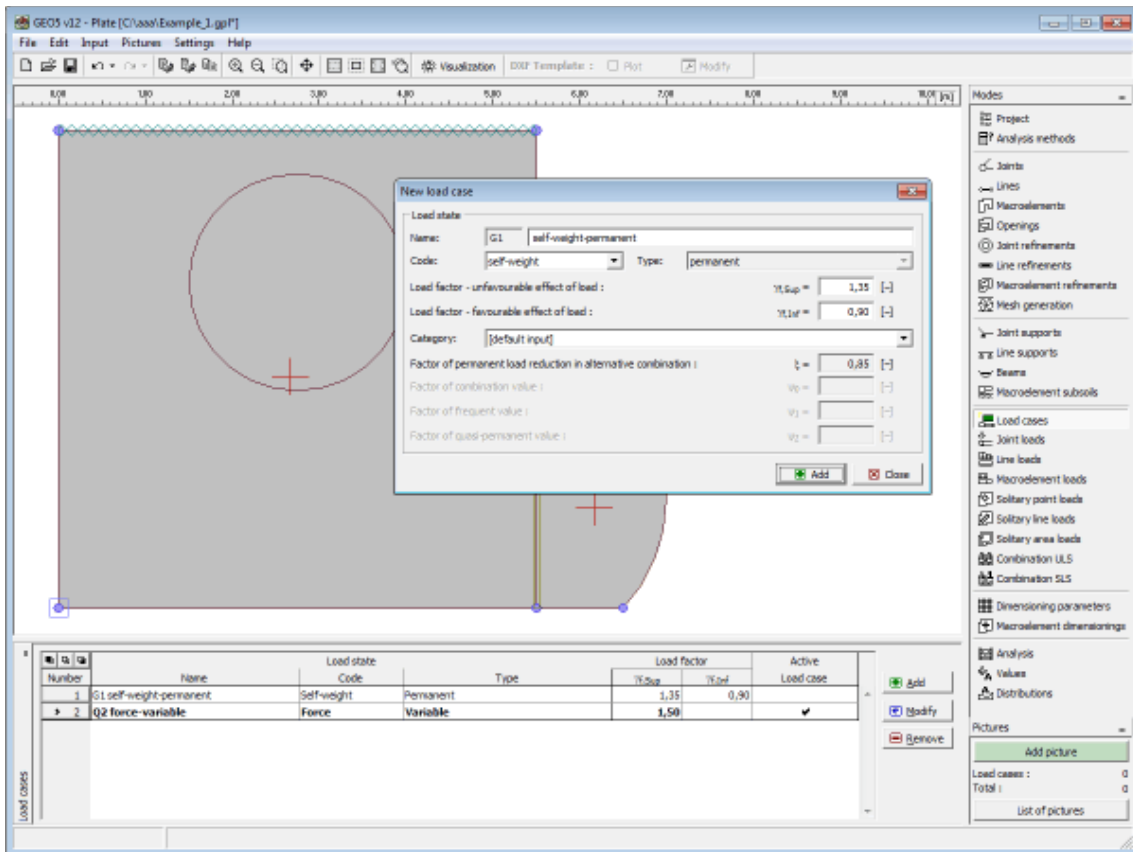
where:

$[C]$	matrix of constants $C_1$ a $C_2$
$b$	half width of foundation
$c_{1w}, c_{2w}$	Winkler's constants
$H$	depth of deformation zone

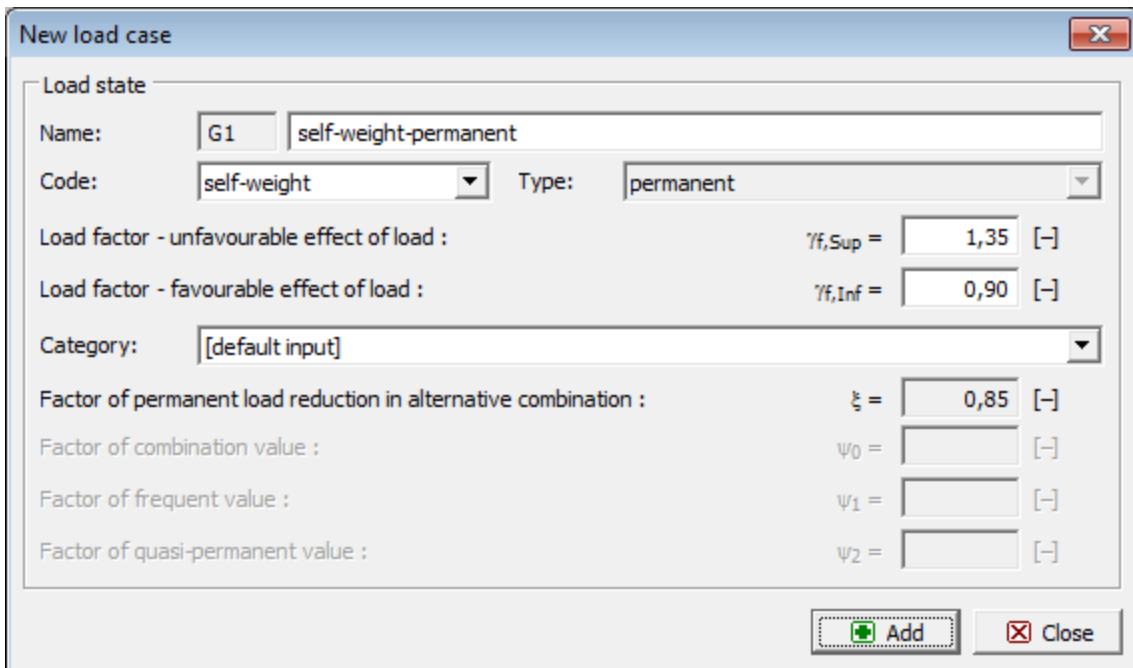
## Load Cases

The frame "Load cases" contains a table with the list of input load cases. Adding (editing) load cases and their parameters is performed in the "New load case" dialog window. Editing can be carried out with the help of "Modify" button or by clicking the row with the required load case in the list using the left mouse button.





Frame "Load cases"



Dialog window "New load case"

## Load Case Parameters

The following parameters are defined in the "**New load case**" dialog window:

### Load case identifier

The load case identifier, which is composed of the load case number and a uniliteral prefix, is displayed in front of the field for entering the name of the load case. The prefix is determined by the type of load case:

<b>G</b>	-	permanent load
<b>Q</b>	-	variable load
<b>A</b>	-	accidental load

The load case identifier is mainly used in printouts of combinations.

### Load case code

The load case code determines, what load can be specified for this load. The following options are available.

<b>Self-weight</b>	In this load case the load represents the structure self-weight and it is generated automatically by the program. Only one load case with this code can be considered in each task.
<b>Force</b>	An arbitrary type of force load (forces, moments) can be introduced into the load case with this code. The number of LCs is not limited.

### Load type

It determines the character of load cases based on their variability in time. Selecting a particular type of load corresponds to classification according to EN 1990 standard, art. 4.1.1.

### Load coefficients

It allows for specifying the load partial factor  $\gamma_f$ . This coefficient accounts for unfavorable deviations of values of loads from the representative ones. For permanent load it is necessary to introduce different values for favorable ( $\gamma_f, inf$ ) and for unfavorable ( $\gamma_f, sup$ ) load action in a combination. If the load input follows EN 1990 the default values of coefficients are taken from table A1.2(B).

### Category

Classification of load cases into categories corresponds to the classification of load according to table A1.1 of EN 1990 standard. Based on this the variable load cases are assigned combination coefficients  $\psi_0$ ,  $\psi_1$  a  $\psi_2$ . The category of "**User-defined input**" allows for defining the user self-values of these coefficients. Choosing a category is possible only for load cases input according to EN 1990 (the "**Material and standards**" tab serves to select the particular standard).

### Combination coefficients

Basic values of coefficients to create combinations arise from EN 1990 standard and depend on the load case category. When user input is assumed, it is possible to define the user self-values of these coefficients. The following coefficients are used to create a combination:

$\xi$	-	<b>Coefficient of reduction of permanent loads in alternative combination</b> - this coefficient is assigned to all permanent loads and is used when compiling alternative combinations for the bearing capacity limit state (combination to relation 6.10.b, EN 1990).
-------	---	---

- $\psi_0$  - **Coefficient of combination value** - coefficient for variable loads, it is used when compiling combinations for both the bearing capacity and service limit states
- $\psi_1$  - **Coefficient of frequent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state
- $\psi_2$  **Coefficient of quasi-permanent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state

The combination coefficients are available only for load cases input according to EN 1990 (the "Material and standards" tab serves to select the particular standard).

Dialog window "New load case"

## Joint Loads

The frame "Joint loads" contains a table with the list of input joint loads. Each joint load is assigned to a certain load case and input joint. Selection of the load is performed in the "Active load case" combo list. Adding (editing) joint loads is performed in the "New load of joints" dialog window.

The program allows for specifying either mechanical (e.g. forces) or deformational (e.g. prescribed displacements of supports) actions.

Joint loads can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Joint loads". The following modes are available:

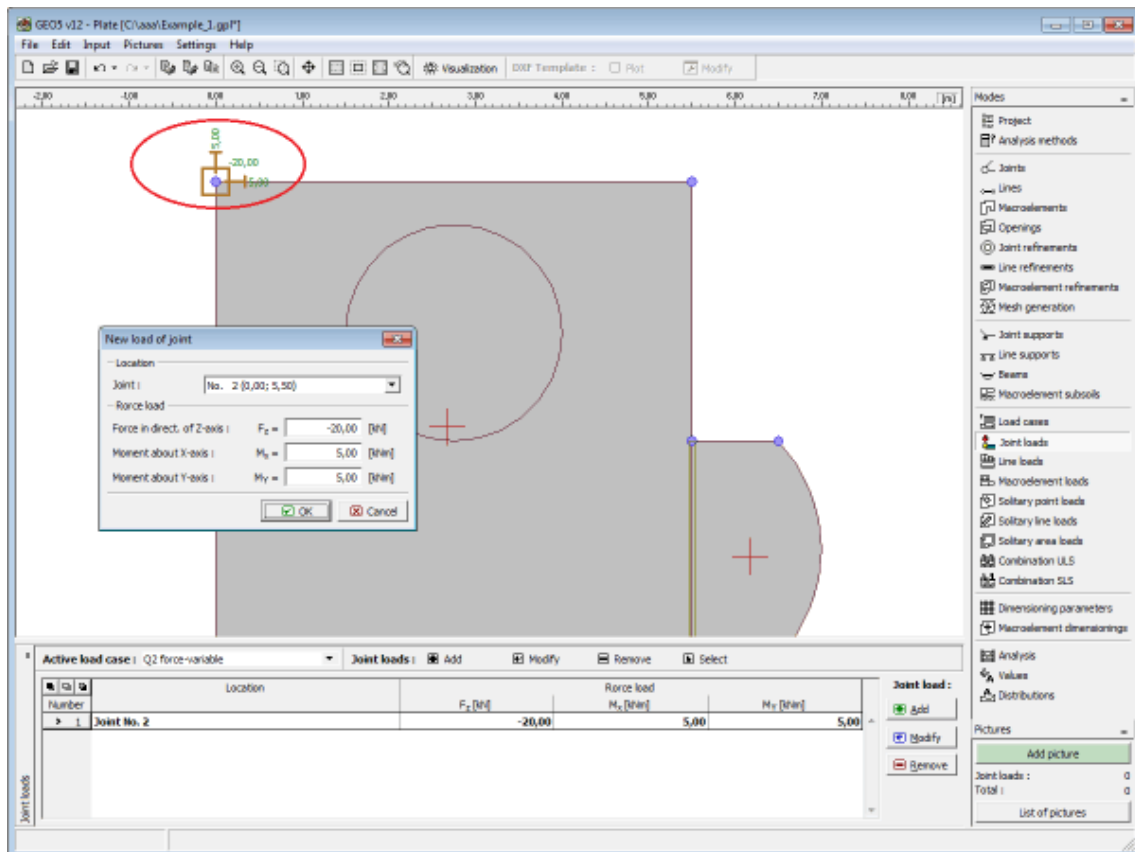
- **Add** Clicking the left mouse button on the desktop introduces the joint location. Adding joint load is performed in the "New load of joints" dialog window.
- **Modify** Clicking the left mouse button on already existing joint load opens the "Modify properties of joint load" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the joint load opens the **remove joint**

**load** dialog window - accepting this action removes the selected joint load.

- Select** Clicking the left mouse button on the joint load highlights the selected joint load. The joint load is simultaneously marked in the **table** list. The option allows for editing several joint loads at once (e.g. deleting).

The input joint loads can also be edited on the desktop with the help of **active objects**.

The program employs the following **coordinate systems** (sign convention).



Frame "Joint loads"

## Line Loads

The **frame "Line loads"** contains a **table** with the list of input line loads. Each line load is assigned to a certain load case and input **lines**. Selection of the load is performed in the **"Active load case"** combo list. Adding (editing) line loads is performed in the **"New loads of lines"** dialog window.

The program allows for specifying either mechanical (e.g. forces), deformational (e.g. prescribed displacements of supports), or **temperature** actions.

Joint loads can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Line loads"**. The following modes are available:

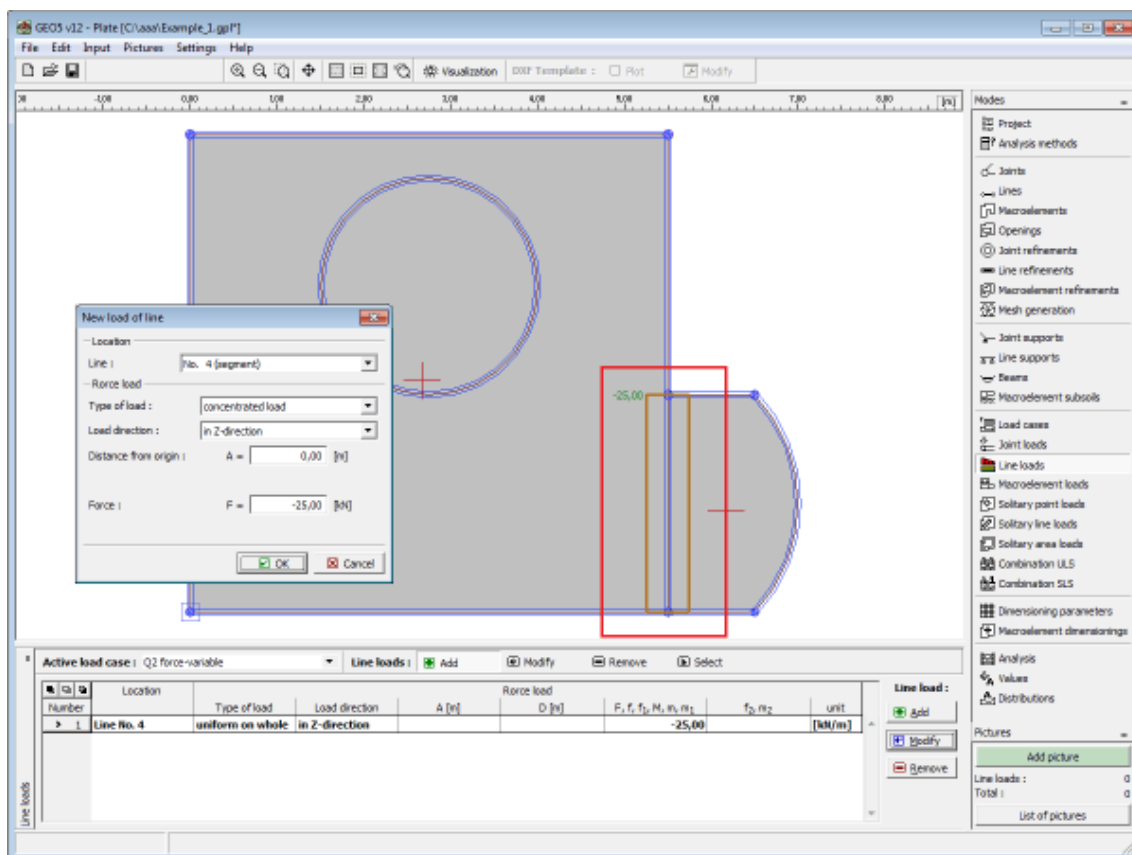
- Add** Clicking the left mouse button on the desktop introduces the line location. Adding line load is performed in the **"New loads of lines"** dialog window.
- Modify** Clicking the left mouse button on already existing line load opens the

"**Modify properties of load of line**" dialog window, which allows for modifying its parameters.

- **Remove** Clicking the left mouse button on the line load opens the **remove line load** dialog window - accepting this action removes the selected line load.
- **Select** Clicking the left mouse button on the line load highlights the selected line load. The line load is simultaneously marked in the **table** list. The option allows for editing several line loads at once (e.g. deleting).

The input line loads can also be edited on the desktop with the help of **active objects**.

The program employs the following **coordinate systems** (sign convention).



Frame "Line loads"

Dialog window "New loads of lines"

## Temperature Load

Temperature load assumes a linear distribution of temperature throughout the slab thickness. Such thermal gradient causes moments in the slab given by:

$$m_t = \frac{E \cdot h^2 \cdot \alpha \cdot \Delta t}{12 \cdot (1 - \nu)}$$

where:

- $E$  - elastic modulus
- $\nu$  - Poisson's ratio
- $h$  - slab thickness
- $\alpha$  - coefficient of thermal expansion
- $\Delta t$  - temperature difference

## Macroelement Loads

The **frame** "Macroelement loads" contains a **table** with the list of input macroelement loads. Each macroelement load is assigned to a certain load case and input **macroelement**. Selection of the load is performed in the "Active load case" combo list. Adding (editing) macroelement loads is performed in the "New macroelement loads" dialog window.

The program allows for specifying either mechanical (e.g. forces) or **temperature** actions.

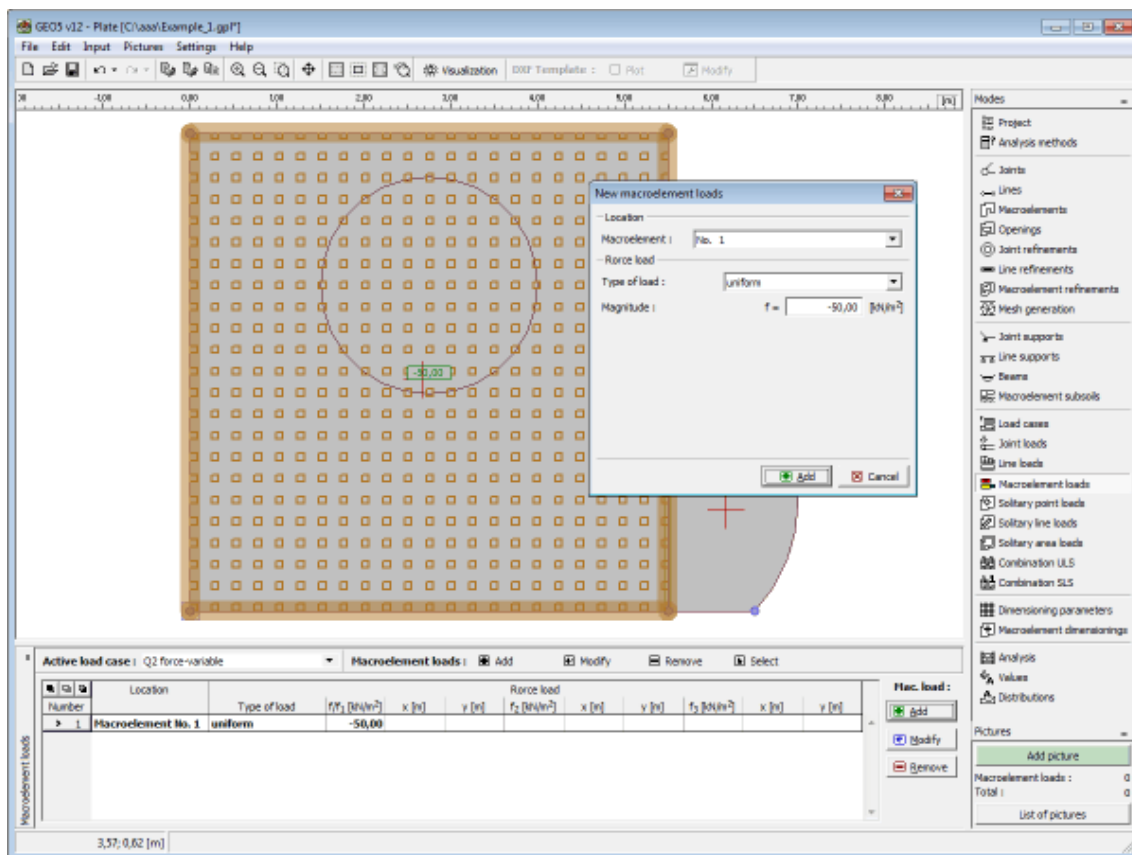
Joint loads can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Macroelement loads". The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the macroelement location. Adding macroelement load is performed in the "New macroelement loads" **dialog window**.

- **Modify** Clicking the left mouse button on already existing macroelement load opens the "**Modify properties of macroelement load**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the macroelement load opens the **remove macroelement load** dialog window - accepting this action removes the selected macroelement load.
- **Select** Clicking the left mouse button on the macroelement load highlights the selected macroelement load. The macroelement load is simultaneously marked in the **table** list. The option allows for editing several macroelement loads at once (e.g. deleting).

The input macroelement loads can also be edited on the desktop with the help of **active objects**.

The program employs the following **coordinate systems** (sign convention).



Frame "Macroelement loads"

## Free Point Loads

The **frame "Free point loads"** contains a **table** with the list of input free point loads. Each free point load is assigned to a certain load case and can receive an arbitrary location on the slab surface. Selection of the load is performed in the "**Active load case**" combo list. Adding (editing) free point loads is performed in the "**New free point loads**" **dialog window**.

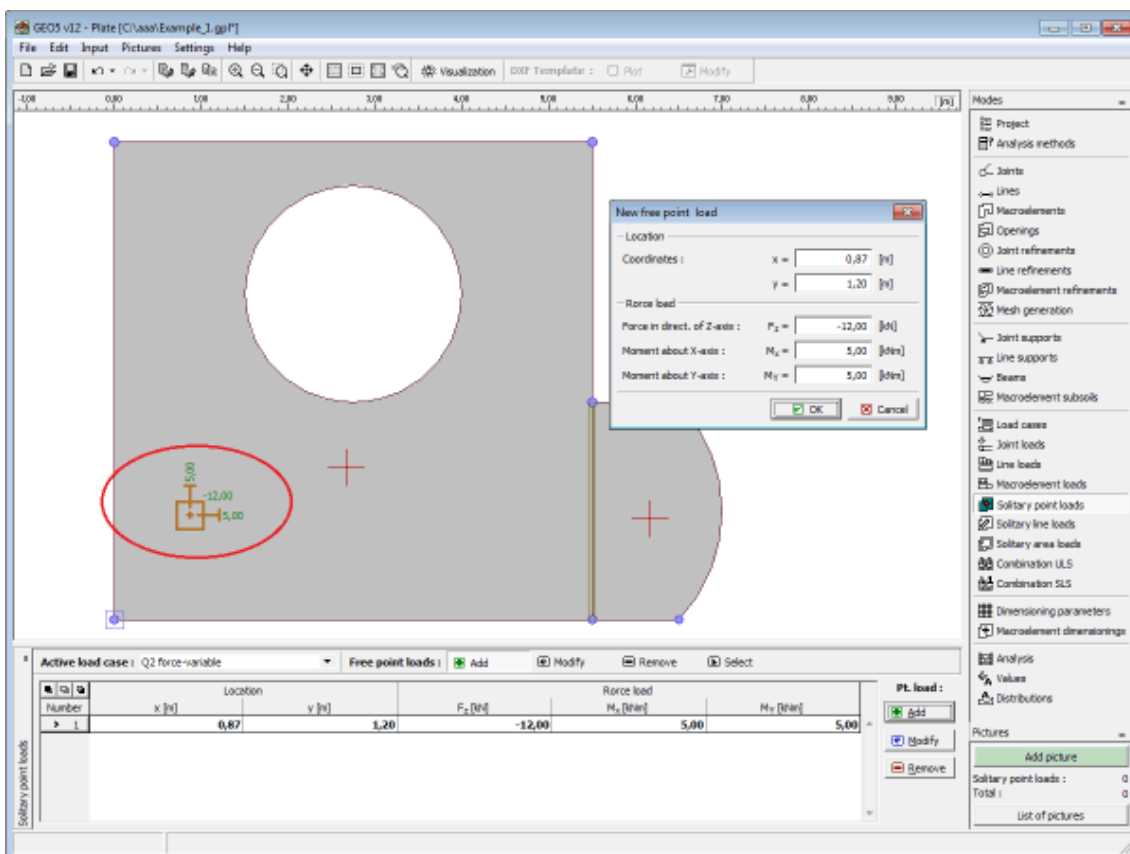
Free point loads can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Free point loads**". The following

modes are available:

- **Add** Clicking the left mouse button on the desktop defines point specifying the loaded. The load parameters are defined in the "**New free point loads**" dialog window.
- **Modify** Clicking the left mouse button on already existing free point load opens the "**Modify properties of free point load**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the free point load opens the **remove point load** dialog window - accepting this action removes the selected free point load.
- **Select** Clicking the left mouse button on the free point load highlights the selected free point load. The free point load is simultaneously marked in the **table** list. The option allows for editing several free point loads at once (e.g. deleting).

The input free point load can also be edited on the desktop with the help of **active objects**.

The program employs the following **coordinate systems** (sign convention).



Frame "Free point loads"

## Free Line Loads

The **frame "Free line loads"** contains a **table** with the list of input free line loads. Each free line load is assigned to a certain load case and with using points it can receive an arbitrary



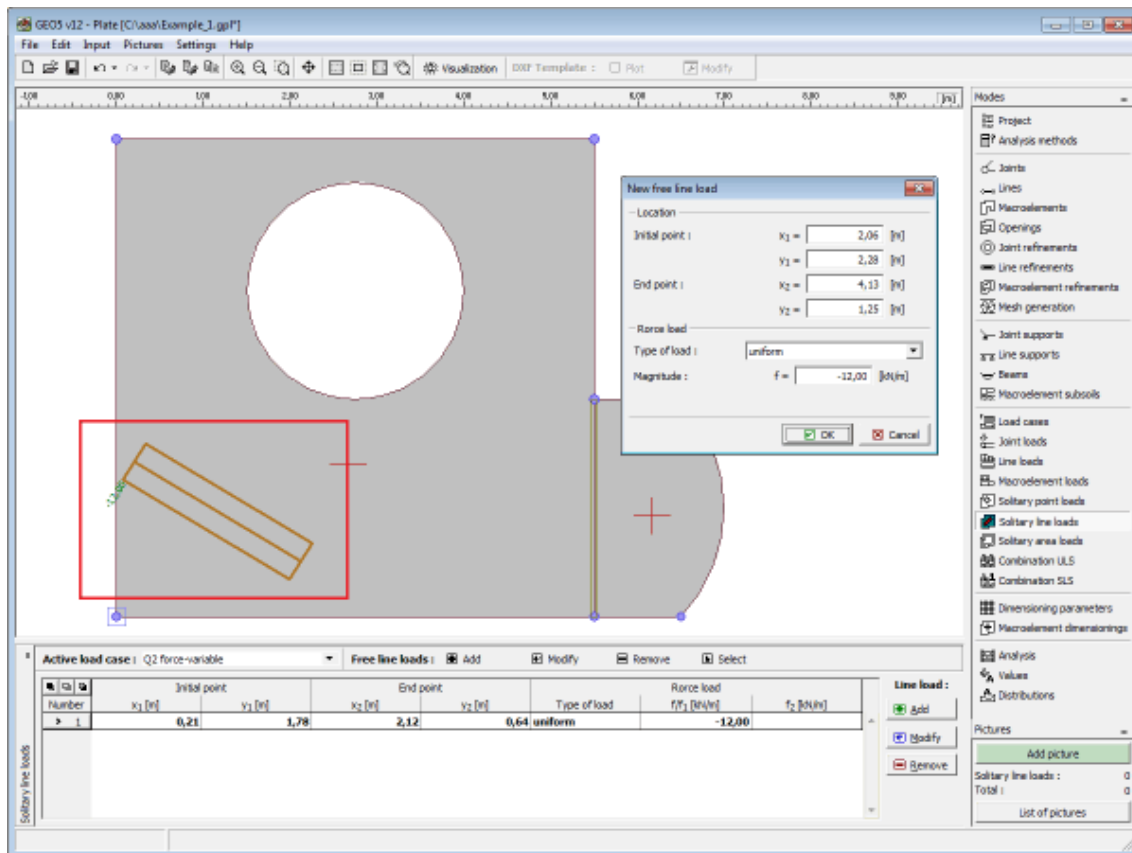
direction and location on the slab surface. Selection of the load is performed in the **"Active load case"** combo list. Adding (editing) free line loads is performed in the **"New free line loads"** dialog window.

Free line loads can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Free line loads"**. The following modes are available:

- **Add** Clicking the left mouse button on the desktop defines points specifying the loaded line. The load parameters are defined in the **"New free line loads"** dialog window.
- **Modify** Clicking the left mouse button on already existing free line load opens the **"Modify properties of free line load"** dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the free line load opens the **remove line load** dialog window - accepting this action removes the selected free line load.
- **Select** Clicking the left mouse button on the free line load highlights the selected free line load. The free line load is simultaneously marked in the **table** list. The option allows for editing several free line loads at once (e.g. deleting).

The input free line load can also be edited on the desktop with the help of **active objects**.

The program employs the following **coordinate systems** (sign convention).



Frame "Free line loads"

## Free Area Loads

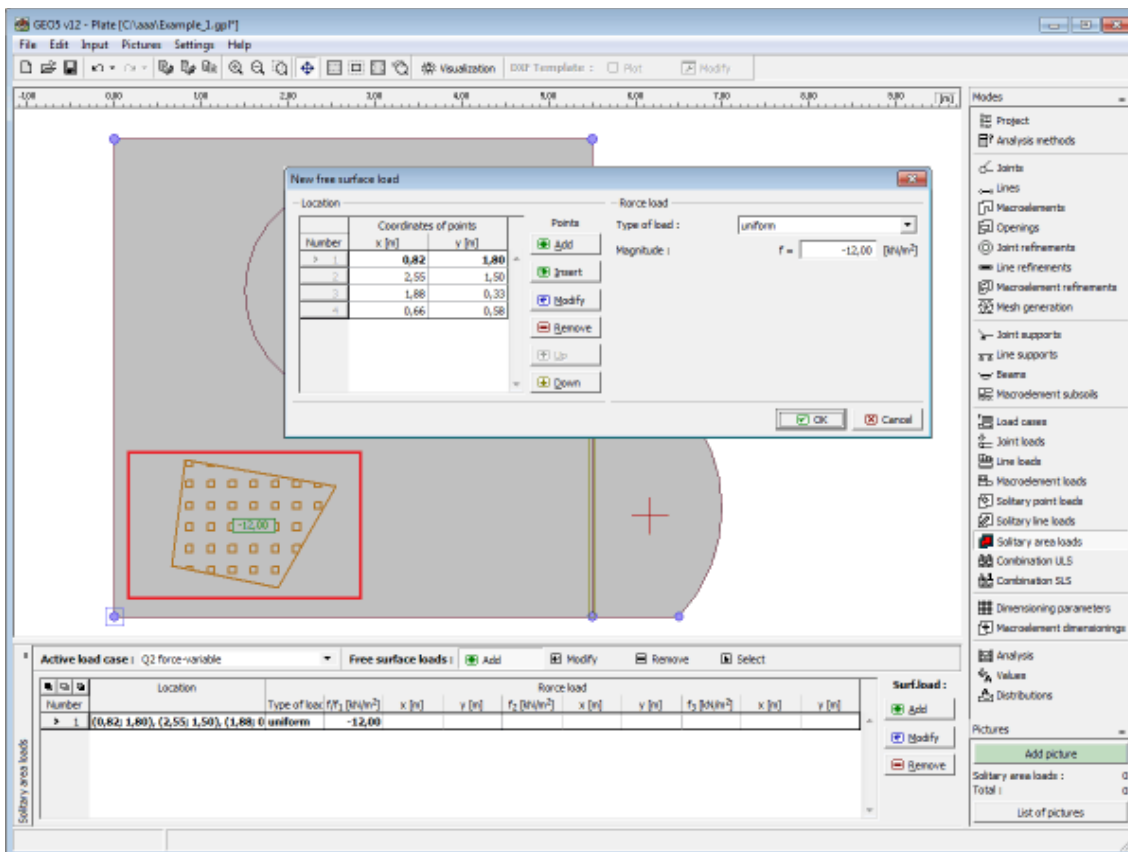
The frame "**Free area loads**" contains a **table** with the list of input free area loads. Each free area load is assigned to a certain load case and with using points it can receive an arbitrary shape and location on the slab surface. Selection of the load is performed in the "**Active load case**" combo list. Adding (editing) free area loads is performed in the "**New free surface loads**" dialog window.

Free area load can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Free surface loads**". The following modes are available:

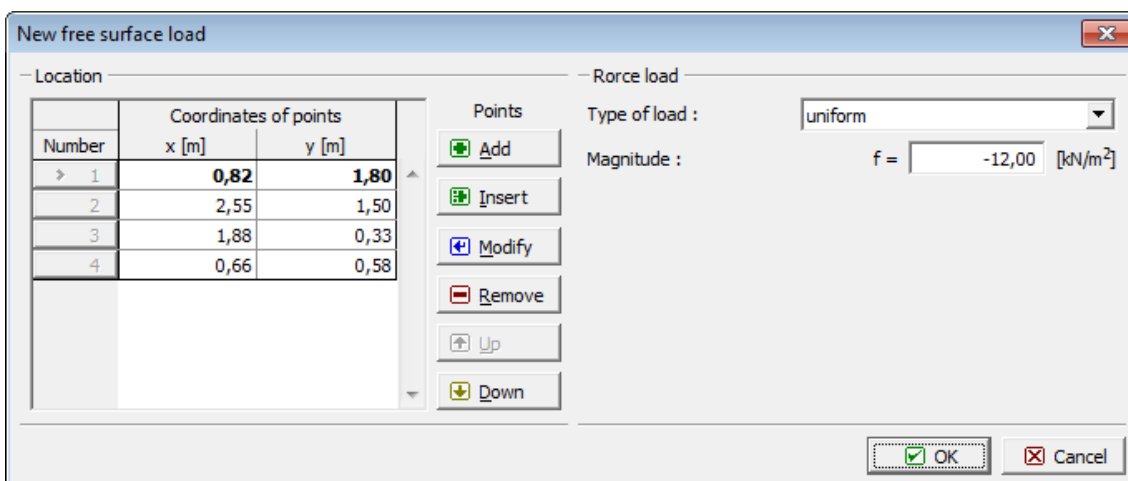
- **Add** Clicking the left mouse button on the desktop defines points specifying the loaded region. The load parameters are defined in the "**New free surface loads**" dialog window.
- **Modify** Clicking the left mouse button on already existing free area load opens the "**Modify properties of free surface load**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the free area load opens the **remove free area load** dialog window - accepting this action removes the selected free area load.
- **Select** Clicking the left mouse button on the free area load highlights the selected free area load. The free area load is simultaneously marked in the **table** list. The option allows for editing several free area loads at once (e.g. deleting).

The input free area load can also be edited on the desktop with the help of **active objects**.

The program employs the following **coordinate systems** (sign convention).



Frame "Free area loads"

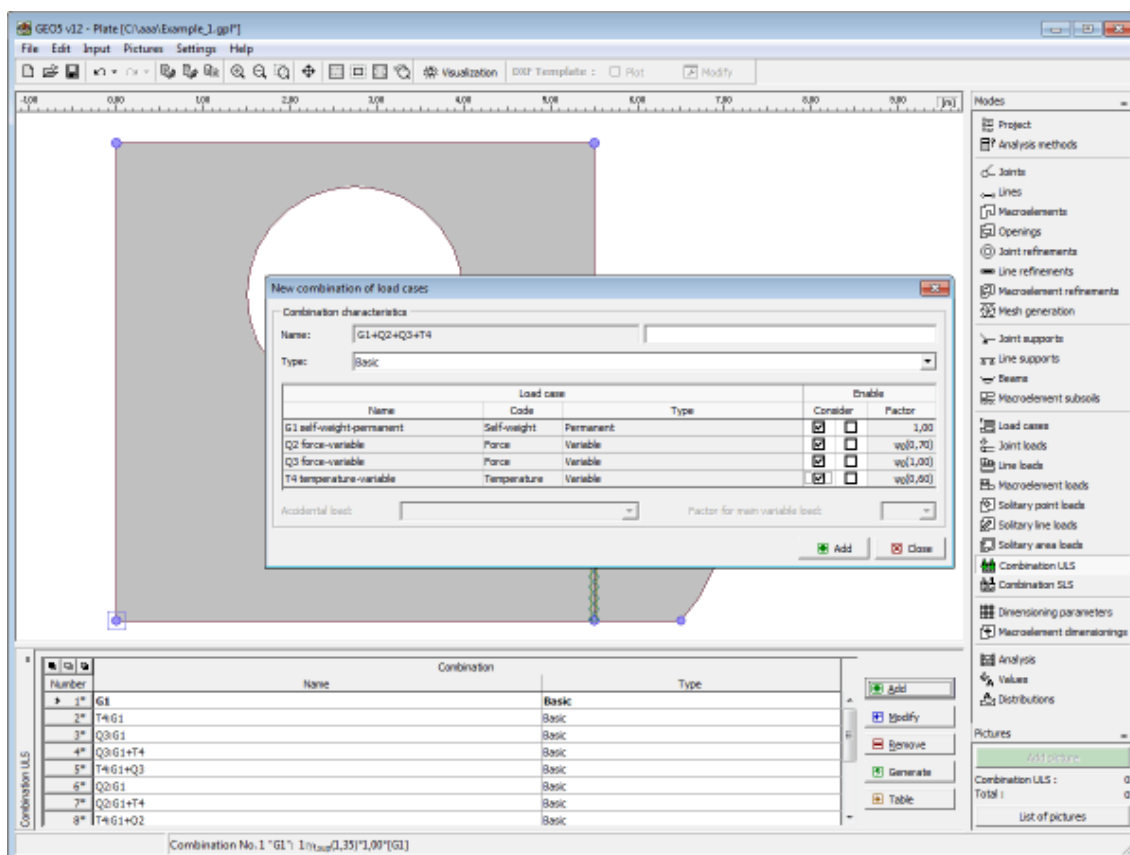


Dialog window "New free surface loads"

## Combination ULS

The frame "Combinations ULS" contains a table with the list of input combinations of the bearing capacity limit state. Adding (editing) combinations and their parameters is performed in the "New combination of load cases" dialog window. Editing can be carried out with the help of "Modify" button or by clicking the row with the required combination in the list using the left mouse button.

The built-in generator of combinations of load cases can be used to create individual combinations.



Frame "Combination ULS"

## Parameters of ULS Combinations

The following parameters are specified in the "New combination of load cases" dialog window:

### New combination

A brief description of combination is displayed in front of the field where the combination is defined. All considered load cases are tagged using their identifiers. The major variable loads are moved at the beginning of the list and separated from the remaining LCs by colon.

### Type of combination (for combinations based on EN 1990 only)

The following combinations can be created for the bearing capacity limit state:

- Basic** Basic combination based on expression 6.10 of EN 1990 standard
- Alternative** Combinations based on expressions 6.10a and 1.10b of EN 1990 standard. In this case, two variants of combination are used in the analysis, one with reduced permanent LCs and the other with reduced major variable LC.
- Accidental** Accidental combination based on 6.11 of EN 1990 standard.

### Selection of load cases

The table listing individual load cases allows for their selection to create a combination. The

load case can be introduced into a combination by checking the field in the column "**Consider**" for a particular LC. Further setting in the table depends on the selection of way of inputting loads in the "**Material and standards**" tab.

### Load according to EN 1990

A second field is available for each load case in the column "**Consider**". This field allows for assigning a favorable effect of action to permanent LCs (adopting coefficient  $\gamma_f, inf$ ) or for specifying a variable load as the major one, respectively. The number of major variable loads in the combination is not limited. An accidental load can be introduced into combinations tagged as "**Accidental**" (only LCs tagged as "**Accidental**" are available for the selection). For accidental combinations it is also necessary to choose, whether a major variable load should be reduced by the coefficient  $\psi_1$  or  $\psi_2$ .

### General load

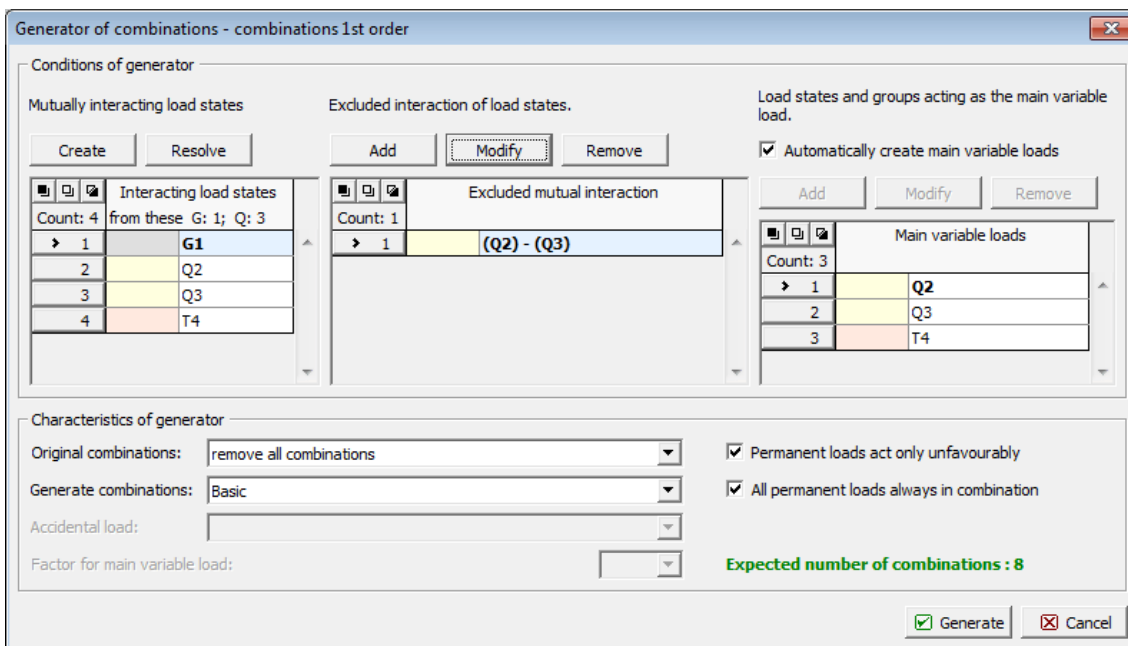
A coefficient of usability can be specified for each load case to adjust the degree of usability of the load case in the combination.

Load case			Enable	
Name	Code	Type	Consider	Factor
G1 self-weight-permanent	Self-weight	Permanent	<input checked="" type="checkbox"/>	1,00
Q2 force-variable	Force	Variable	<input checked="" type="checkbox"/>	$\psi_0(0,70)$
Q3 force-variable	Force	Variable	<input checked="" type="checkbox"/>	$\psi_0(1,00)$
T4 temperature-variable	Temperature	Variable	<input checked="" type="checkbox"/>	$\psi_0(0,60)$

Dialog window "New combination of load cases"

## Generator of ULS Combinations

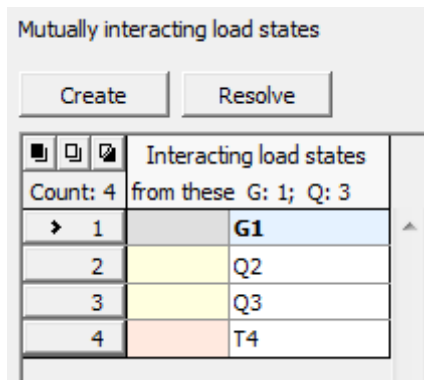
The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases based on the introduced combination rules. Referring to standard EN 1990 the number of generated combinations can be relatively large and in extreme cases could considerably slow down calculations. Owing to this, information about expected number of combinations to be generated is displayed in the right bottom corner. Therefore, before launching generation the user may check, how many combinations will be generated and possibly adjust generator conditions. The top part of the window serves to define conditions for generating combinations; the bottom part contains various generator settings.



Dialog window "Generator of combinations - 1st order"

**Mutually interacting load states and groups**

This part makes it possible to merge those load states that should appear in combinations always together. Permanent and variable loads cannot be merged into one group. If the field **All permanent loads always in combination** is checked in the Generator parameters, the creation of groups of permanent loads has no effect on their appearance in combinations as each generated combination will always contain all permanent LCs. In such case, merging permanent LCs will only influence consideration of favorable/unfavorable effects of LCs providing the field **Permanent loads act only unfavorably** is not checked.



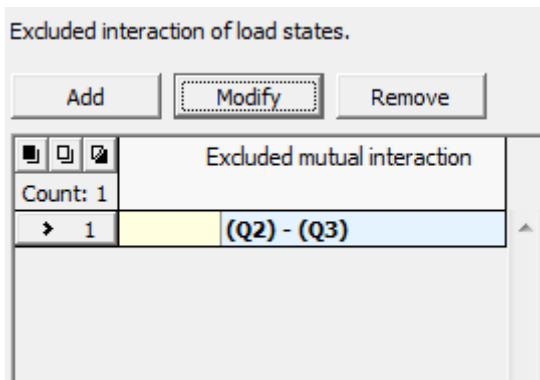
Dialog window "Generator of combinations" - Mutually interacting load states

**Excluded interaction of load states**

This part makes it possible to define, which LCs should not appear in a combination together. Arbitrary load cases or merged groups can be mutually excluded in dependent of the type of load case. Two options are available to define groups to be excluded:

**Mutual exclusion** An arbitrary number of load cases can be introduced into one group. In such a case, the program will not generate any combination that contains at least two load cases from this group.

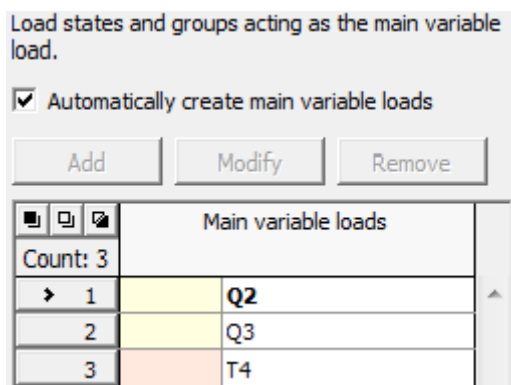
**Exclusion by pairs** Providing it is necessary to create a larger number of excluding groups of two sorts, where one LC is the same (e.g. exclusion of assembly variants of permanent loads with all service load cases), it is possible to adopt this option. A load case to be excluded is first selected in the first column. The second column is then used to select an arbitrary number of LCs, which are needed to create excluding groups.



Dialog window "Generator of combinations" - Excluded interaction of load states

**Load cases and groups acting as the main variable load**

This part is available only when inputting loads according to EN 1990 is considered (the standard is selected in the "Material and standards" tab). When automatic regime is assumed then each variable load is taken as major in created combinations. If this regime is turned off, it is possible to manually adjust the list of major variable loads. For example, it is possible to remove an arbitrary load case from the list so that it will not be considered as major variable in combinations. If a new item with more load cases is add to the list then all load cases will be considered as major variable in those combinations, where they appear together.



Dialog window "Generator of combinations" - Load cases and groups acting as the main variable load

**Generator parameters (parameters that can be set in the bottom part of the dialog window)**

**Combo list "Original combinations"**

**Retain original combinations**

- By pressing the "Generate" button the program will add new combinations, created according to the specified rules, to the original ones

**Remove all combinations**

- By pressing the "Generate" button the program will delete all original combinations and will replace them by the new ones

- Remove generated combinations** - By pressing the "**Generate**" button the program will delete older combinations and will add new ones created according to the specified rules
- Remove all combinations of the current type** - By pressing the "**Generate**" button the program will delete all combinations of a given type and will replace them by the new ones
- Remove generated combinations of the current type** - By pressing the "**Generate**" button the program will delete older combinations of a given type and will add new ones created according to the specified rules

### Combo list "Generate combinations"

The following types of generated combinations can be chosen for loads based on EN 1990:

- Basic** - Generates basic combinations for the bearing capacity limit state based on expression 6.10 of EN 1990 standard
- Alternative** - Generates combinations for the bearing capacity limit state based on expressions 6.10a and 1.10b of EN 1990 standard. This variant generates two times more combinations but it provides better results.
- Accidental** - Generates accidental combinations for the bearing capacity limit state based on 6.11 of EN 1990 standard. An accidental load case to appear in accidental combinations can be specified. It is also necessary to choose, whether a major variable load will be reduced by the coefficient  $\psi_1$  or  $\psi_2$ .

### Permanent loads act only unfavorably

If this setting is not checked, the program creates all possible combinations, where introduction of all variants of favorable and unfavorable actions of permanent loads will be considered.

### All permanent loads always in combination

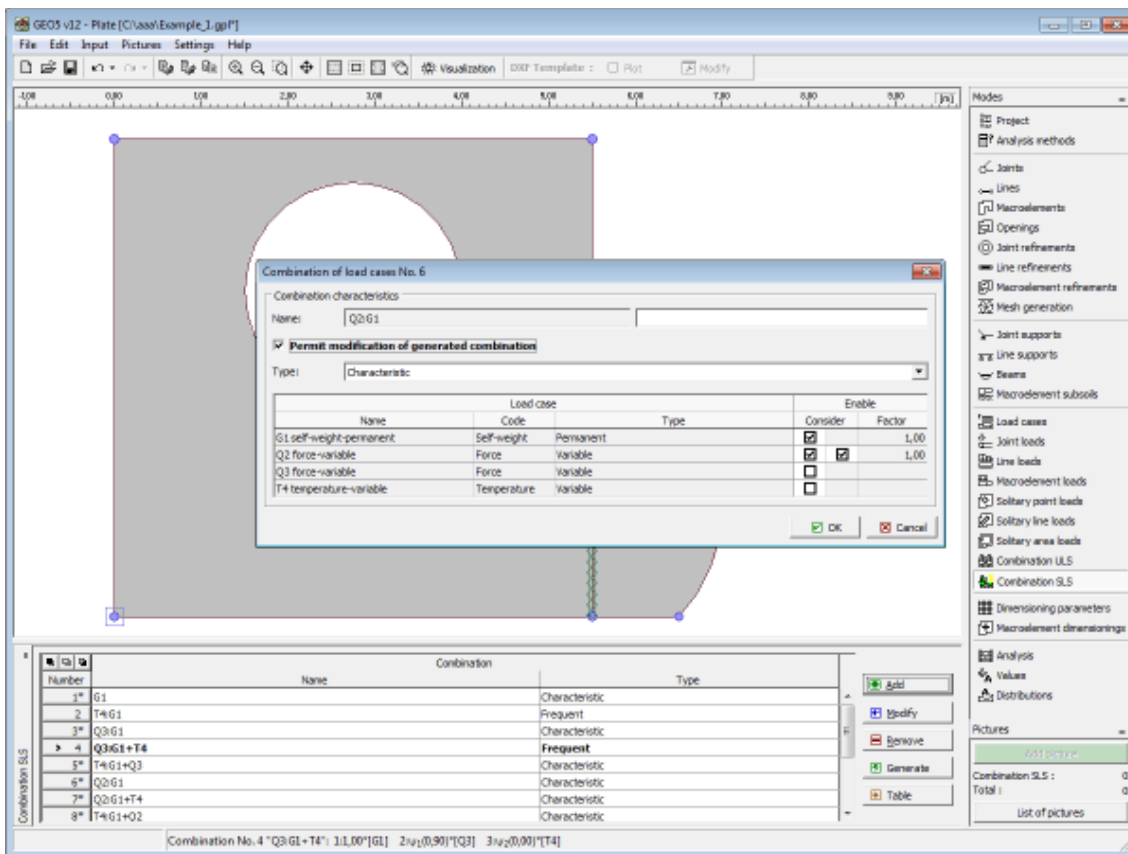
If this setting is not checked, the program creates combinations in such a way that a successive introduction of all LCs into a combination will be considered.

## Combination SLS

The **frame "Combinations SLS"** contains a **table** with the list of input combinations of the service limit state. Adding (editing) combinations and their **parameters** is performed in the "**New combination of load cases**" dialog window. Editing can be carried out with the help of "**Modify**" button or by clicking the row with the required combination in the list using the left mouse button.

The built-in **generator of combinations** of load cases can be used to compile individual combinations.





Frame "Combination SLS"

## Parameters of SLS Combinations

Combinations SLS serve to evaluate states that refer to the structure appearance, comfort of people or to functioning of a structure while in ordinary use. Typically, only deformations, vibrations, etc. are checked. The **"New combination of load cases"** dialog window (similarly to combinations for ULS) serves to define the following parameters:

### Type of combination according to EN 1990

The following combinations can be created for the service limit state:

- Characteristic** - combination based on expression 6.14 of EN 1990 standard
- Frequent** - combination based on expression 6.15 of EN 1990 standard
- Quasi-permanent** - combination based on expression 6.16 of EN 1990 standard

### Selection of load cases

The table listing individual load cases allows for their selection to create a combination. The load case can be introduced into a combination by checking the field in the column **"Consider"** for a particular LC. A coefficient of usability can be specified for generally input combinations (select in the **"Material and standards"** tab) to adjust the degree of usability of the load case in the combination.

Name	Load case		Enable		Factor
	Code	Type	Consider	Factor	
G1 self-weight-permanent	Self-weight	Permanent	<input checked="" type="checkbox"/>		1,00
Q2 force-variable	Force	Variable	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1,00
Q3 force-variable	Force	Variable	<input type="checkbox"/>		
T4 temperature-variable	Temperature	Variable	<input type="checkbox"/>		

Dialog window "Combination of load cases"

## Generator of SLS Combinations

The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases for the service limit state. Functions of generator of combinations are explained in section devoted to the **generator of combinations for the bearing capacity limit state**.

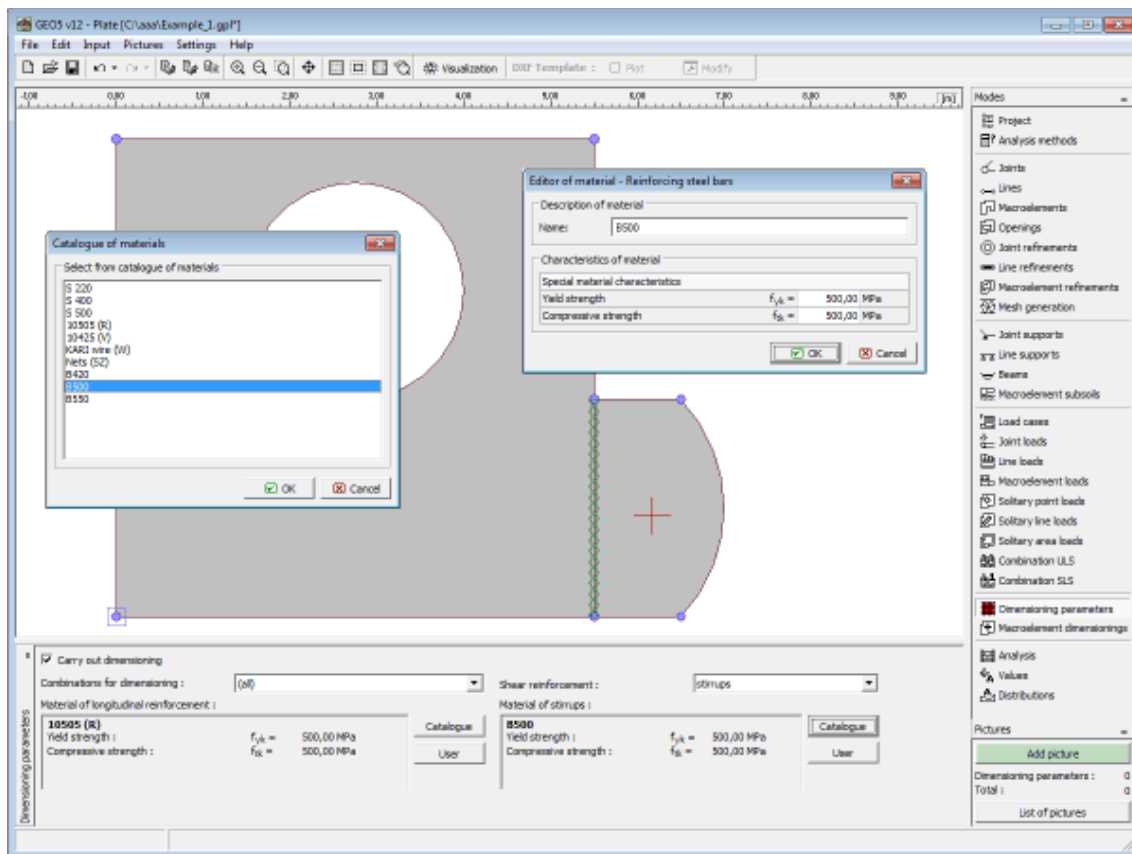
If inputting loads according to EN 1990 is set in the "**Material and standards**" tab, it is possible to generate the following combinations for the service limit state:

- Characteristic** - combination based on expression 6.14 of EN 1990 standard
- Frequent** - combination based on expression 6.15 of EN 1990 standard
- Quasi-permanent** - combination based on expression 6.16 of EN 1990 standard

## Dimensioning Parameters

The frame "**Dimensioning parameters**" serves to define data for dimensioning **longitudinal** and **shear reinforcement**. The **combination number** (or all combinations) of a combination to be analyzed must be specified. The material of longitudinal reinforcements is selected either from "**Catalog of materials**", or can be introduced manually in the "**Editor of materials**". Shear reinforcement is specified in terms of **crooks**, or **stirrups** (crooks require to define their angle).

When running the **dimensioning** analysis the program generates **values** of the following quantities. The analysis is carried out according to the standard set in the "**Material and standards**" tab.



Frame "Dimensioning parameters"

## Macroelement Dimensioning

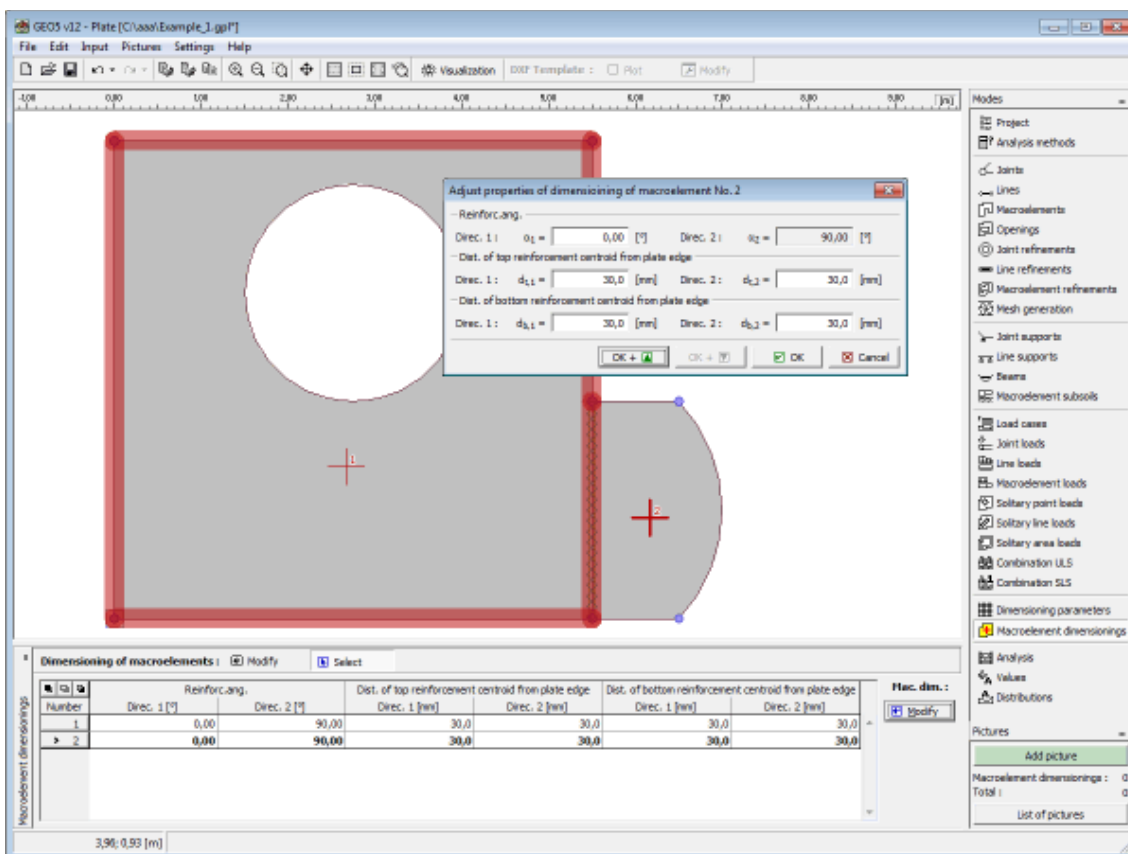
The frame "Macroelement dimensioning" contains a table with the input macroelements. Editing reinforcement properties (**reinforcement direction** and **distance of centroid of top and bottom reinforcement from the slab edge** can be modified) is performed in the "Modify properties of macroelement dimensioning" dialog window.

When running the dimensioning analysis the program generates values of the following quantities. The analysis is carried out according to the standard set in the "Material and standards" tab.

Macroelement dimensioning can also be modify using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Dimensioning of macroelements". The following modes are available:

- **Modify** Clicking the left mouse button on already existing solitary point load opens the "Modify properties of dimensioning of macroelement" dialog window, which allows for modifying its parameters.
- **Select** Clicking the left mouse button on the macroelement dimensioning highlights the selected macroelement dimensioning. The macroelement dimensioning is simultaneously marked in the table list. The option allows for editing several macroelement dimensionings at once (e.g. deleting).

The macroelement dimensioning can also be edited on the desktop with the help of active objects.



Frame "Macroelement dimensioning"

## Analysis

The analysis results are displayed in the frame "Analysis". The "Analysis" is carried out using the finite element method. The dimensioning analysis is performed according to the standard set in the "Material and standards" tab.

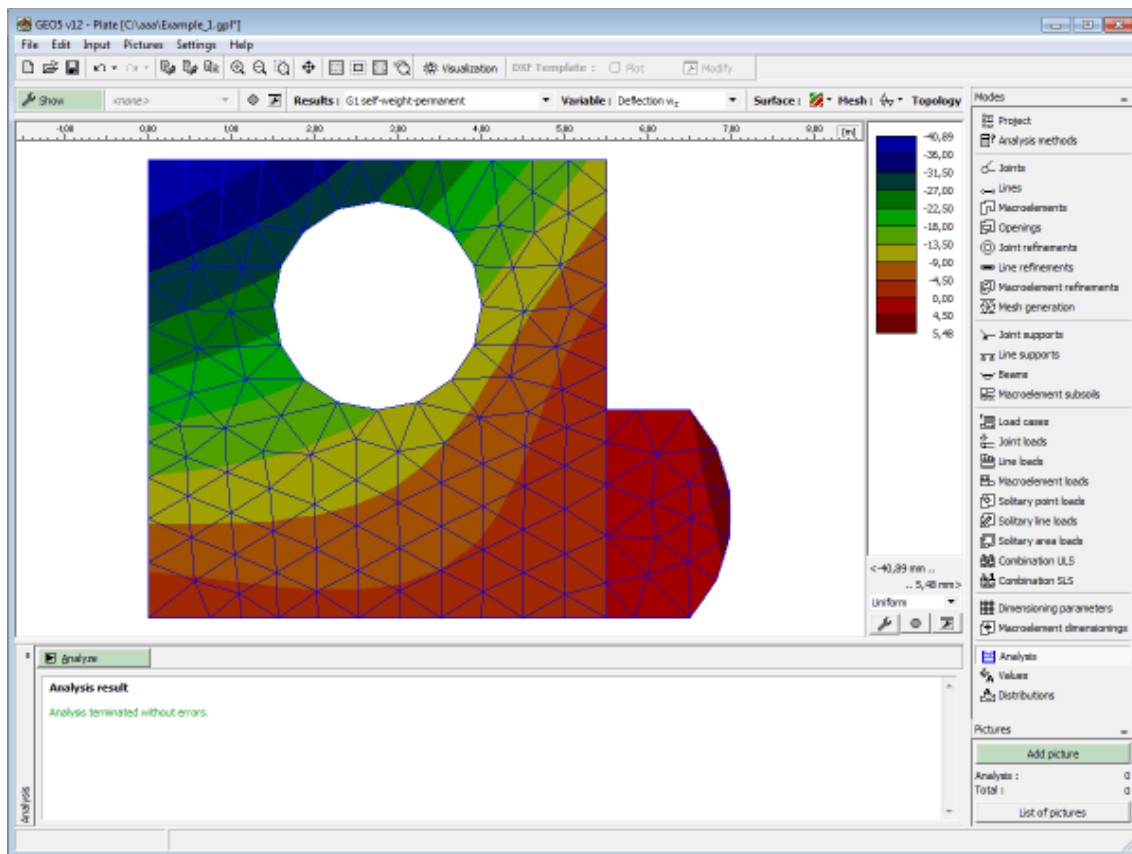
The analysis can be stopped any time by pressing the "Interrupt" button.

Upon completing the analysis the program immediately displays the results and information about the solution process. This information (with possible listing of errors) is shown in window in the bottom part of the frame. The principal output tool is the visualization of results on the screen. The tool bar in the top part of the screen serves to manage the graphical representation of output quantities.

The color range and the bottom for its setting are found in the top part of the desktop.

The program employs the following coordinate systems (sign convention).

The way the results appear on the screen can be set in the "Visualization style settings" dialog window.



Frame "Analysis" - screen after completing analysis

## Analysis Procedure

The solution procedure is split into several steps including localization of the global stiffness matrix while taking into account the support conditions (fixed or spring supports at joint or along lines, elastic subsoil), setting up the load vector and analysis of the system of equations using the Gaussian method with the Cholesky decompositions of the global stiffness matrix, which in this case is symmetric and band. The values of primary variables  $w_z$ ,  $\varphi_x$  a  $\varphi_y$  calculated at mesh nodes are then used to determine the internal forces  $m_x$ ,  $m_y$ ,  $m_{xy}$ ,  $v_x$  and  $v_y$  together with the derived quantities  $m_1$ ,  $m_2$  and the values of reactions developed in supports.

### 2D-elements

The quality of results of the slab problem derived using the finite element method is strongly influenced by the type of slab element. The present formulation exploits a deformation variant of the finite element method to derive triangular and quadrilateral elements denoted as DKMT and DKMQ (Discrete Kirchhoff-Mindlin Triangle a Quadrilateral).

Formulation of the slab element implemented in the program is based on the discrete Kirchhoff theory of bending of thin slabs, which can be considered as a special case of the Mindlin plate theory developed upon the following assumptions:

- compression of slab in the  $z$ -direction is negligible compare to the vertical displacement  $W_z$
- normals to the mid-plane of the slab remain straight after deformation but not necessarily normal to the deformed mid-plane of the slab
- normal stress  $\sigma_z$  is negligible compare to stresses  $\sigma_x$ ,  $\sigma_y$

DKMT and DKMQ elements have 9 and 12 degrees of freedom, respectively - three independent displacements at each node:

$W_z$  - deflection in the direction of  $z$ -axis

$z$

$\varphi_x$  - rotation about  $x$ -axis

$x$

$\varphi_y$  - rotation about  $y$ -axis

$y$

The elements satisfy the following criteria:

- the stiffness matrix has correct rank (no zero energy states are generated)
- fulfill the patch test
- are suitable for the analysis of both thin and thick slabs
- they show good convergence properties
- not computationally expensive

In case of well generated mesh the quadrilateral elements are preferable as show better behavior compare to triangular elements.

### 1D-elements

The slab can be reinforced by beams formulated on the basis of one dimensional beam element with embedded torsion and is compatible with slab elements (details can be found in literature). The primary variables are  $W_z$ ,  $\varphi_x$  and  $\varphi_y$  and corresponding internal forces are  $M_1$ ,  $M_2$  and  $V_3$  (twisting and bending moments and shear force). The beam is characterized by the moment of inertia  $I_t$  and  $I_2$  (torsion, bending), area  $A$  and shear area  $A_s$ . These parameters can be calculated by the program based on the type of cross-section. The analysis constructs  $6 \times 6$  local stiffness matrices subsequently localized in to the global stiffness matrix of the structure.

*Literature:*

*I. Katili, A new discrete Kirchhoff-Mindlin element based on Mindlin-Reissner plate theory and assumed shear strain fields - part I: An extended DKT element for thick-plate bending analysis, Int. J. Numer. Meth. Engng., Vol. 36, 1859-1883 (1993).*

*I. Katili, A new discrete Kirchhoff-Mindlin element based on Mindlin-Reissner plate theory and assumed shear strain fields - part II: An extended DKQ element for thick-plate bending analysis, Int. J. Numer. Meth. Engng., Vol. 36, 1885-1908 (1993).*

*Z. Bittnar, J. Sejnoha, Numericke metody mechaniky, CVUT, Praha, 1992.*

## Results

Visualization and interpretation of results is one of the most important parts of the program.

Based on the **tool bar** setting the program displays **variables** (deflection, moments, rotation) for an arbitrary load case or LC combination, or if needed **variables for dimensioning** (values of necessary reinforcement areas calculated according to the standard selected in the "**Material and standards**" tab).

Calculation of values in user-defined points, or on lines, can be set in frames "**Values**" or "**Distributions**", respectively.

The program provides several basic types of graphical output defined in the "**Slab - results**

visualization settings" dialog window.

- plotting structure
- surface plot of quantities
- plotting finite element mesh
- plotting grid-plotting distributions (diagrams)
- plotting values on surface
- plotting directions of moments and reactions

The tool bar "Results" in the upper part of the screen serves to selected variables to be displayed and the way they should appear on the screen. The color range is shown in the right part of the desktop. Its particular setting can be adjusted using the "Color range" tool bar.

Because properly setting outputs might be often time consuming, the program disposes of a comfortable system of selecting and storing view settings.

All outputs and selected results can be further printed out from the analysis protocol.

## Tool Bar - Results

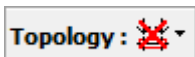
The tool bar contains the following operating elements:



Tool bar "Setting visualization of graphical outputs"

Individual elements operate as follows:

	<b>Plotting style setting</b>	<ul style="list-style-type: none"> <li>• opens the "Slab - results visualization settings" dialog window which allows the user to be more specific in defining the plotting style</li> </ul>
	<b>List of plots</b>	<ul style="list-style-type: none"> <li>• a combo list containing names of plots saved by the user</li> </ul>
	<b>Save plot</b>	<ul style="list-style-type: none"> <li>• saves the current plot displayed on the desktop, the dialog window serves to enter the name of the plot</li> </ul>
	<b>Manager of plots</b>	<ul style="list-style-type: none"> <li>• opens the "Manager of plots" dialog window which serves to manage (delete, change order, rename) already saved plots</li> </ul>
	<b>Results</b> (loads, load cases, combination, dimensioning)	<ul style="list-style-type: none"> <li>• displays the selected load cases, combination (ULS, SLS), envelopes (ULS, SLS) or dimensioning</li> </ul>
	<b>Type of variable</b>	<ul style="list-style-type: none"> <li>• displays the selected variable or variable of dimensioning</li> </ul>
	<b>Surface plot</b>	<ul style="list-style-type: none"> <li>• turns on/off plotting of isolines, isosurfaces</li> </ul>
	<b>Mesh</b>	<ul style="list-style-type: none"> <li>• turns on/off the style of plotting the FE mesh (only edges, or according to the setting in the "Slab - results</li> </ul>



### Plotting of topology

- visualization settings" dialog window
- plotting of topology of construction

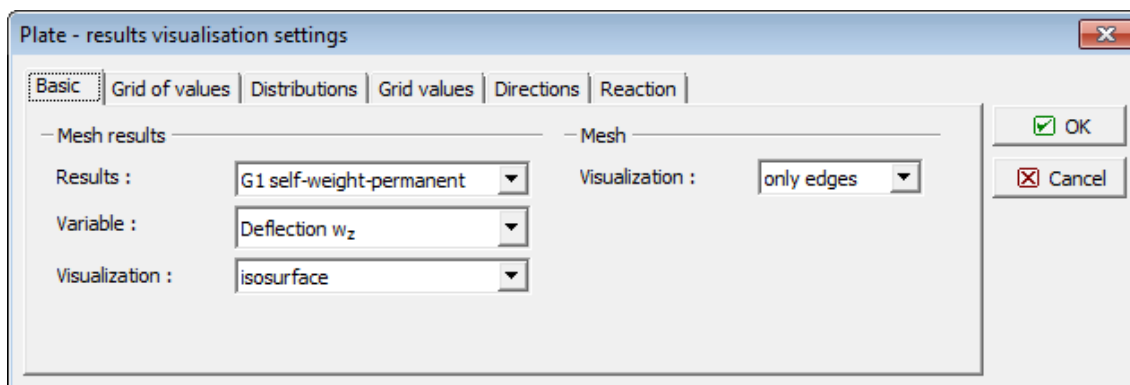
The tool bar contains **the most often used operating elements** needed to view the results on the desktop. Detailed setting of the style of plotting the results is available in the "Slab - results visualization settings" dialog window.

Similar to our other programs the results can be saved and printed. The plotting style can be adjusted in the "Visualization settings" dialog window.

## Results Visualization Settings

The "Slab - results visualization settings " dialog window serves to specify the values to be plotted and the way of their visualization. Individual settings can be later **saved** using the tool bar "Results".

The item "Basic" serves to set the basic parameters for plotting the results, quantities and mesh information - additional items can be then used to define visualization of other outputs.



Dialog window "Slab - results visualization settings"

## List of Variables

List of quantities displayed by the program for individual load cases, combinations of load cases (ULS, SLS) or envelopes (ULS, SLS)

Variable	Unit	Description
Deflection $W_z$	[mm]	Displacement in the Z-direction
Rotation $\phi_x$	[mrad]	Rotation about X-axis
Rotation $\phi_y$	[mrad]	Rotation about Y-axis
Moment $m_x$	[kNm/m]	Value of the moment about X-axis
Moment $m_y$	[kNm/m]	Value of the moment about Y-axis
Moment $m_{xy}$	[kNm/m]	Value of moment



Shear force $V_x$	[kN/m]	Value of the shear force in the $X$ -direction
Shear force $V_y$	[kN/m]	Value of the shear force in the $Y$ -direction
Moment $m_1$	[kNm/m]	Value of the principal (extreme) moment
Moment $m_2$	[kNm/m]	Value of the principal (extreme) moment
Shear force $V_{max}$	[kN/m]	Value of the shear force (extreme)
Contact stress $\sigma$	[kN/m <sup>2</sup> ]	Value of the contact stress

## List of Variables of Dimensioning

To perform **dimensioning** analysis it is first necessary to choose the option "**Carry out dimensioning**" in the frame "**Dimensioning parameters**". Visualization of values for dimensioning can be set in the tool bar "**Results**". Notation of variables (particularly indexes of variables) changes according to the standards used for dimensioning of concrete and steel structures set in the "**Material and standards**" tab.

### List of quantities displayed by the program for dimensioning

Variable	Unit	Description
Moment $M_{dim1, min}$	[kNm/m]	Minimal dimensioning moment in direction of reinforcement 1
Moment $M_{dim1, max}$	[kNm/m]	Maximal dimensioning moment in direction of reinforcement 1
Moment $M_{dim2, min}$	[kNm/m]	Minimal dimensioning moment in direction of reinforcement 2
Moment $M_{dim2, max}$	[kNm/m]	Maximal dimensioning moment in direction of reinforcement 2
Reinforcement area $A_{u1}$	[mm <sup>2</sup> /m]	Area of upper reinforcement in direction 1
Reinforcement area $A_{b1}$	[mm <sup>2</sup> /m]	Area of bottom reinforcement in direction 1
Reinforcement area $A_{u2}$	[mm <sup>2</sup> /m]	Area of upper reinforcement in direction 2
Reinforcement area $A_{b2}$	[mm <sup>2</sup> /m]	Area of bottom reinforcement in direction 2
Ratio of reinforcement $\mu_{h1}$	[%]	Reinforcement ratio of upper reinforcement in direction 1

Ratio of reinforcement $\mu_{d1}$	[%]	Reinforcement ratio of bottom reinforcement in direction 1
Ratio of reinforcement $\mu_{h2}$	[%]	Reinforcement ratio of upper reinforcement in direction 2
Ratio of reinforcement $\mu_{d2}$	[%]	Reinforcement ratio of bottom reinforcement in direction 2
Shear force $V_{Ed}$	[kN/m]	Dimensioning shear force
Reinforcement area $A_{b,nut}$	[mm <sup>2</sup> /m <sup>2</sup> ]	Requested area of shear reinforcement
Shear force $V_{Rd,c}$	[kN/m]	Shear strength of cross-section without shear reinforcement
Shear force $V_{Rd,max}$	[kN/m]	Maximal allowable shear force

## Values

The frame **"Values"** allows for defining points (or joints) arbitrarily placed on the slab surface. For these points (joints) it is possible to display **variables** (deflections, moments, rotations) for an arbitrary load case or LC combination, or if needed **variables for dimensioning** (values of necessary reinforcement areas calculated according to the standard selected in the **"Material and standards"** tab).

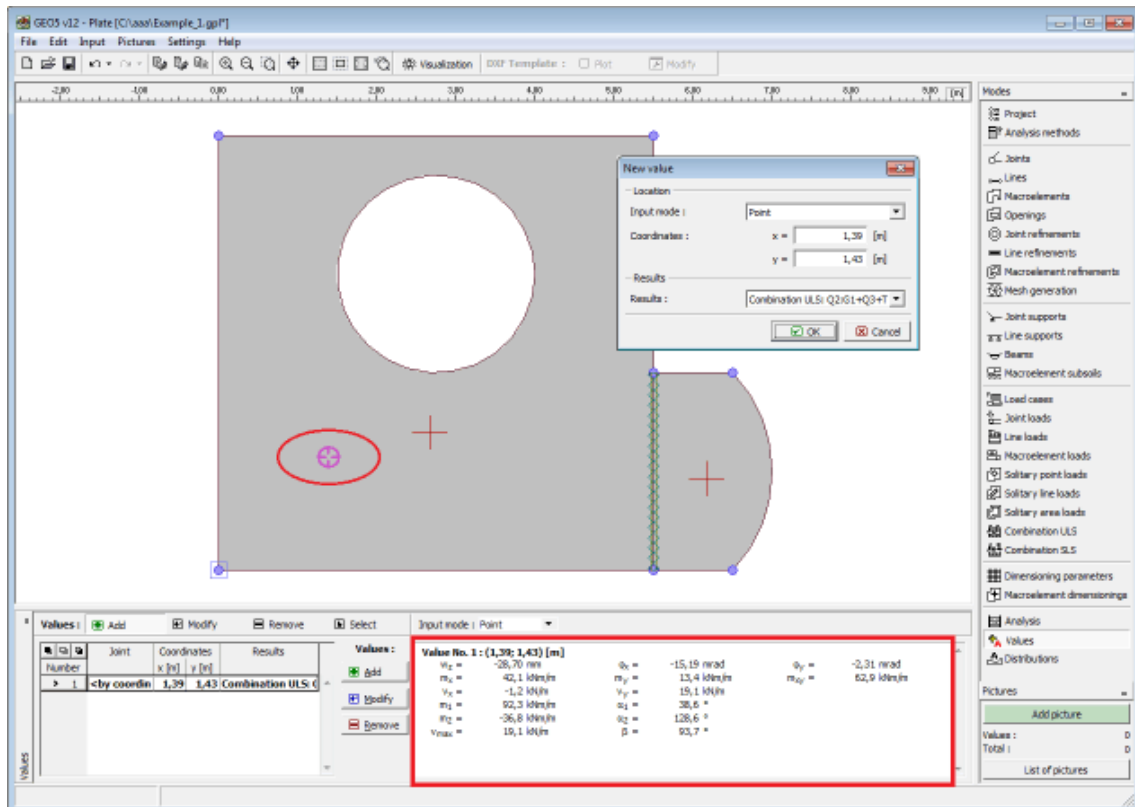
The frame contains a **table** with the list of input points (joints). Adding (editing) is performed in the **"New values" dialog window**. The window serves to specify the type of input (point, joint), coordinates and for what load case, combination, or dimensioning the resulting quantities should be displayed. The value in the supported joint corresponds to the reaction force at this support.

In the **dimensioning** analysis some quantities can be denoted by symbol **[\*]**. In such a case the necessary reinforcement area and minimal degree of reinforcement is required. If the point is found on the boundary of two macroelements, the program displays two sets of values for dimensioning.

Values (points) can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Values"**. The following modes are available:

- **Add** Clicking the left mouse button on the desktop introduces the point location (value).
- **Modify** Clicking the left mouse button on already existing point (value) opens the **"Modify properties of value"** dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the point (value) opens the **remove point** dialog window - accepting this action removes the selected point (value).
- **Select** Clicking the left mouse button on the point (value) highlights the selected point (value). The point (value) is simultaneously marked in the **table** list. The option allows for editing several points (values) at once (e.g. deleting).

The input points (values) can also be edited on the desktop with the help of **active objects**.



Frame "Values"

## Distributions

The frame "Distributions" serves to define general lines or lines located on the slab surface. For these segments (lines) it is possible to display **variables** (deflections, moments, rotations, etc.) for an arbitrary load case or LC combination, or if needed **variables for dimensioning** (values of necessary reinforcement areas calculated according to the standard selected in the "Material and standards" tab).

The frame contains a **table** with the list of input segments (lines). Adding (editing) is performed in the "New distributions" dialog window. The window serves to specify the type of input (segment, line), coordinates of the first and the last point, load case, combination, **dimensioning** and quantity.

The frame displays:

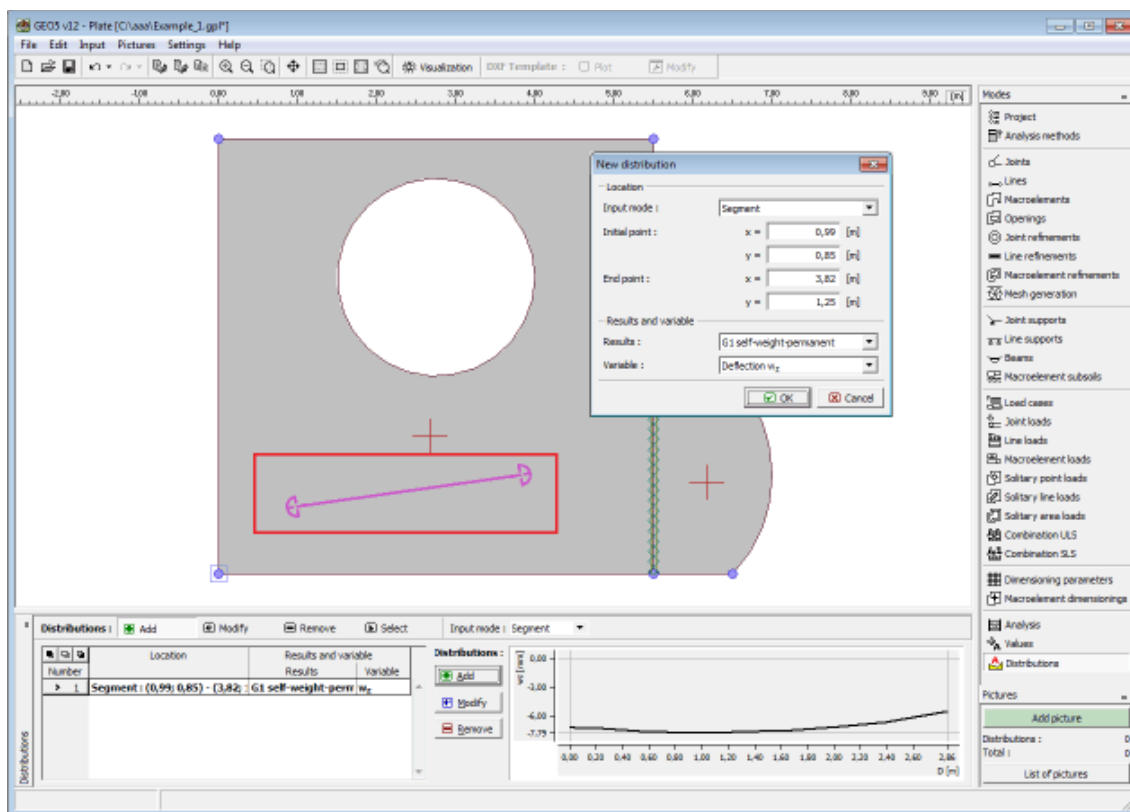
- **General distributions** general distributions (diagrams) on an arbitrary segment (line)
- **Distributions on beams** if a **beam** is assigned to the line it is possible to display distributions (diagrams) of other quantities (shear force  $V_3$ , bending moment  $M_2$ , twisting moment  $M_1$ )
- **Distributions on supported line** if the line is **supported** it is possible to display distributions (diagrams) of other quantities (vertical reaction  $r_z$ , moment reaction  $r_{m, t}$ ).

Distributions can also be introduced using the mouse. This inputting mode is activated by

clicking an appropriate button on the horizontal tool bar "**Distributions**". The following modes are available:

- **Add** Clicking the left mouse button on the joint introduces the distribution location.
- **Modify** Clicking the left mouse button on already existing distribution opens the "**Modify properties of distribution**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on the distribution opens the **remove distribution** dialog window - accepting this action removes the selected distribution.
- **Select** Clicking the left mouse button on the distribution highlights the selected distribution. The distribution is simultaneously marked in the **table** list. The option allows for editing several distributions at once (e.g. deleting).

The input distributions can also be edited on the desktop with the help of **active objects**.



Frame "Distributions"

## Coordinate System (Sign Convention)

### Internal forces

Internal forces are defined as:

$$m_x = \int_{-h/2}^{+h/2} \sigma_x z dz$$

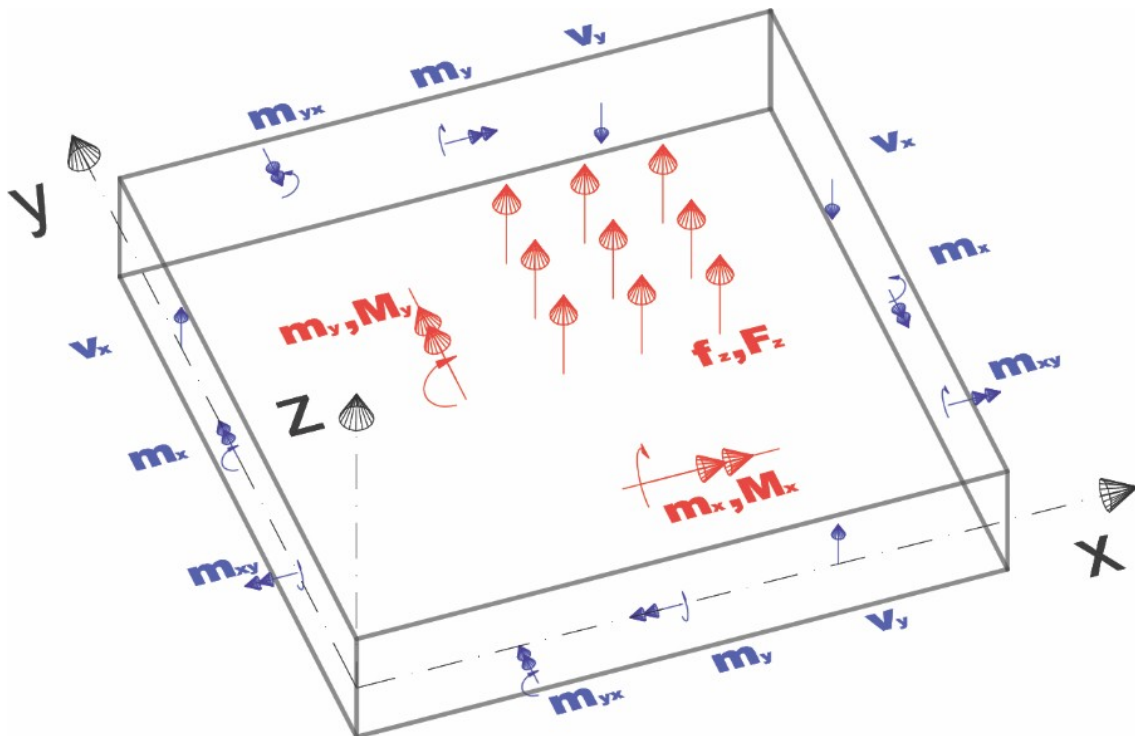
$$m_y = \int_{-h/2}^{+h/2} \sigma_y z dz$$

$$m_{xy} = \int_{-h/2}^{+h/2} \sigma_{xy} z dz$$

$$v_x = \int_{-h/2}^{+h/2} \sigma_{xz} dz$$

$$v_y = \int_{-h/2}^{+h/2} \sigma_{yz} dz$$

The positive direction of internal forces is evident from the following figure:



The principal moments and directions of principal axes are provided by:

$$m_{1,2} = \frac{1}{2} \left( m_x + m_y \pm \sqrt{(m_x - m_y)^2 + 4m_{xy}^2} \right)$$

$$tg 2\alpha_{1,2} = \frac{2m_{xy}}{m_x - m_y}$$

The meaning of individual variables is the following: internal forces can be transformed from

the  $(x, y)$  coordinate system to the  $(x', y')$  coordinate system by rotating the  $(x, y)$  plane through a certain angle about the  $z$ -axis. The angle  $\alpha$ , in particular, corresponds to a rotation angle for which the transformed  $m_{x'y'}$  moment attains a zero value whereas the  $m_{x'}$  and  $m_{y'}$  moments attain their maximum and minimum values  $m_1$  and  $m_2$ , respectively.

The maximum shear force is obtained similarly:

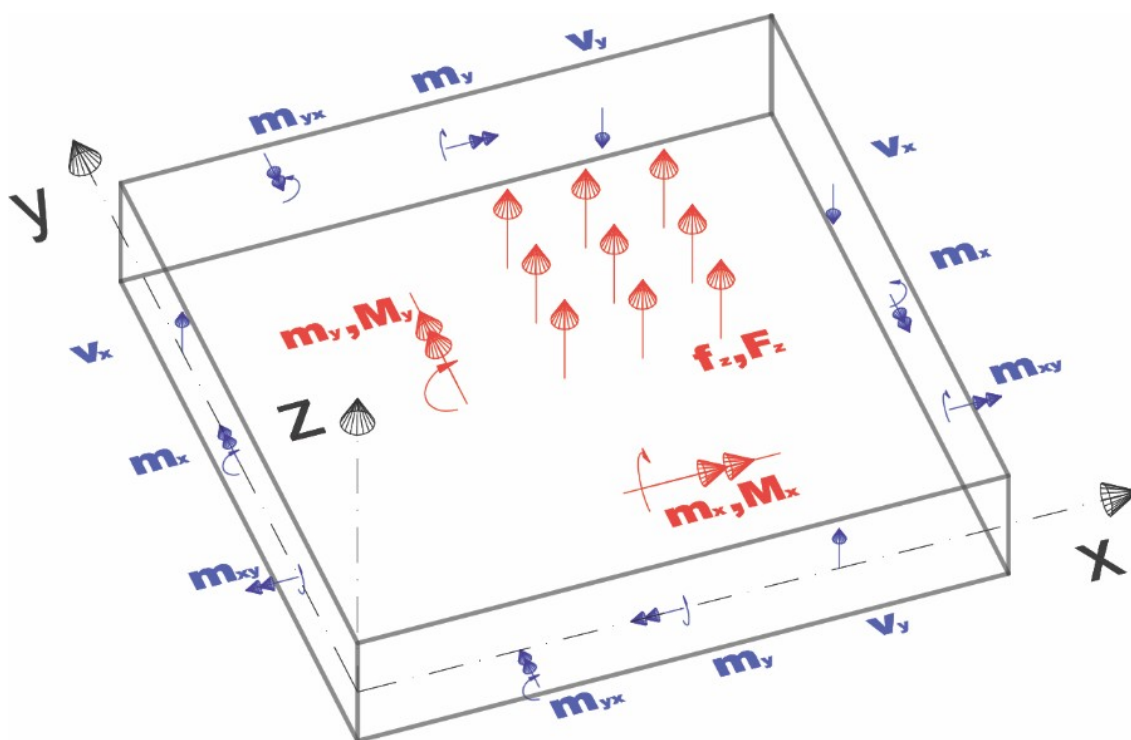
$$v_{max} = \sqrt{v_x^2 + v_y^2}$$

and the angle between  $v_{max}$  and the  $x$ -axis:

$$\beta = \text{arctg} \frac{v_y}{v_x}$$

**Load**

The sign convention of the applied force and moment load is evident from the following figure:



It is worth to point out a different sign convention applied to the load moment  $M$  (at a point or along a line) and to internal moment  $m$ . While the  $M_x$  moment rotates about the  $x$ -axis (as usual for beams), the internal moment rotating about the  $x$ -axis is denoted as  $m_y$ .

**Program Beam**

This program provides the analysis of foundation beams resting on elastic subsoil.

**The help in the program "Beam" includes the following topics:**

- Input of data into individual frames:

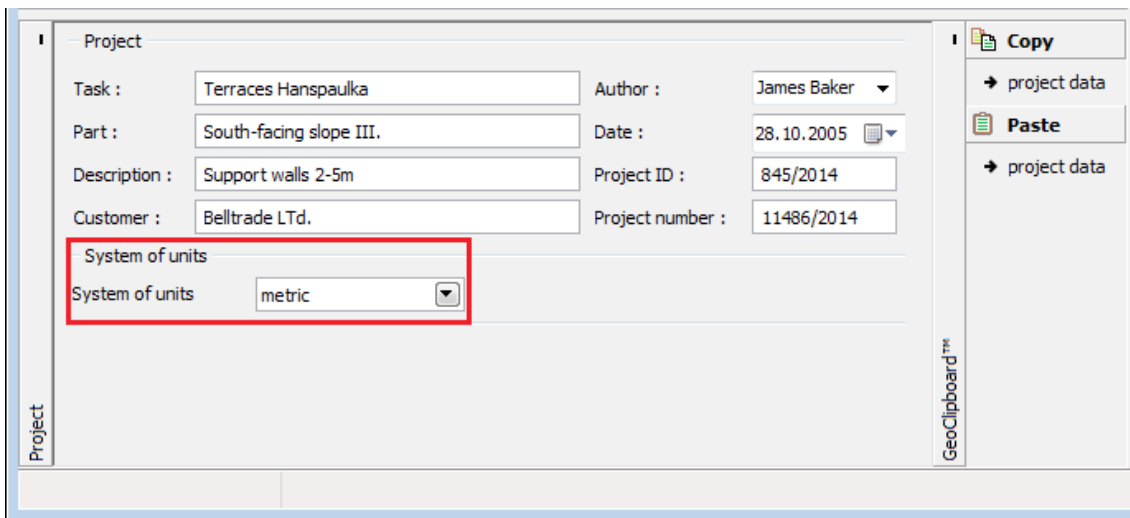
Project	Settings	Geometry	Subsoil	Interface	Location	Soils
Assign	Water	Supports	Load Cases	Load	Combination ULS	Combination SLS
Analysis						

- Standards and analysis methods
- Outputs
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.



Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently

used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current task"**. Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the **"Settings administrator"** by pressing the **"Add to the administrator"** button.

The **"Input for the current task"** setting is usually created when **importing older data**.

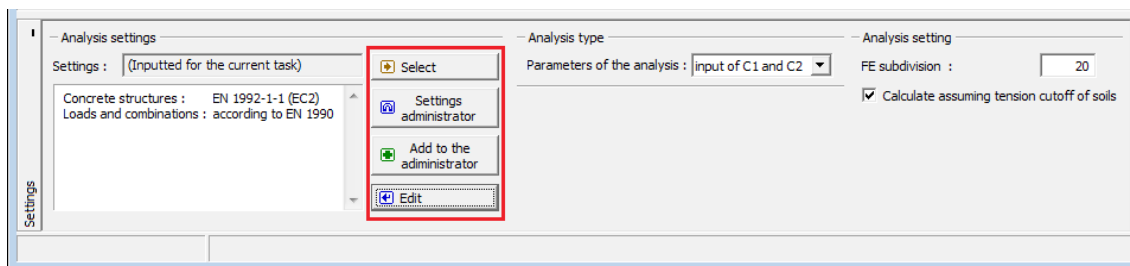
Settings of analysis parameters are performed in the **"Materials and standards"** tab.

This frame also allows to specify the **subdivision of beam into finite elements** (default setting is 20 elements). Next, it is possible to define whether the soil acts in tension - we always recommend **calculating assuming tension cutoff of soils**.

A combo list allows to select one of three ways of defining the **Winkler-Pasternak subsoil**:

- Calculation of  $C_1$  and  $C_2$  - the Winkler-Pasternak parameters of subsoil are **calculated** by the program from input parameters of the geological profile.
- Input of  $C_1$  and  $C_2$  - the Winkler-Pasternak parameters of subsoil are **directly specified**.
- Input of  $E_{def}$ ,  $n_y$ ,  $h_z$  - the Winkler-Pasternak parameters of subsoil are **calculated** from the deformation modulus  $E_{def}$ , Poisson's ratio  $\nu$  and depth of influence zone  $h_z$ .

In the first case, when parameters  $C_1$  and  $C_2$  are calculated, the frame **"Subsoil"** is not accessible. For the remaining approaches the frames **"Interface"**, **"Location"**, **"Soils"**, **"Assign"**, and **"Water"** are not accessible.



Frame "Settings"

## Winkler-Pasternak Parameters C1 a C2

**The Winkler - Pasternak model for the solution of an elastic layer** introduces the balance equation in the vertical direction as:

$$c_1 \cdot w - c_2 \cdot \Delta w = f_z$$

where:  $c_1, c_2$  parameters of the Winkler - Pasternak model  
 $w$  deflection in the vertical direction



$f_z$  vertical load acting on a layer

The program allows to calculate the parameters  $C_1, C_2$  from deformation parameters of soils, or from geological profile.

## Calculation of Winkler-Pasternak Parameters C1 and C2 from Geological Profile

A characteristic combination of load must be chosen when calculating the Winkler-Pasternak parameters ( $C_1, C_2$ ) from geological profile. This combination should be considered as service and should correspond to the most frequently appearing load. Using this combination, a surcharge at the footing bottom is calculated next followed by calculation of the influence zone.

Deformation parameters (Poisson's ratio, deformation modulus) are determined for the calculated influence zone as a weighted average of deformation parameters of soils. Given these parameters the Winkler-Pasternak constants ( $C_1, C_2$ ) are then calculated in the following way.

## Calculation of Winkler-Pasternak Constants from Deformation Parameters of Soils

The Winkler - Pasternak constants  $C_1$  and  $C_2$  are calculated from the condition of equal compliance matrices of infinitely stiff infinite strip footing resting on the Winkler - Pasternak and elastic subsoil. This condition is represented by the following equalities:

$$[C] = \begin{bmatrix} \frac{1}{2 \left[ \sqrt{C_{1WP} C_{2WP}} + b C_{1WP} \right]} & 0 \\ 0 & \frac{1}{2 \left[ b^2 \sqrt{C_{1WP} C_{2WP}} + b C_{2WP} + \frac{b^3}{3} C_{1WP} \right]} \end{bmatrix} =$$

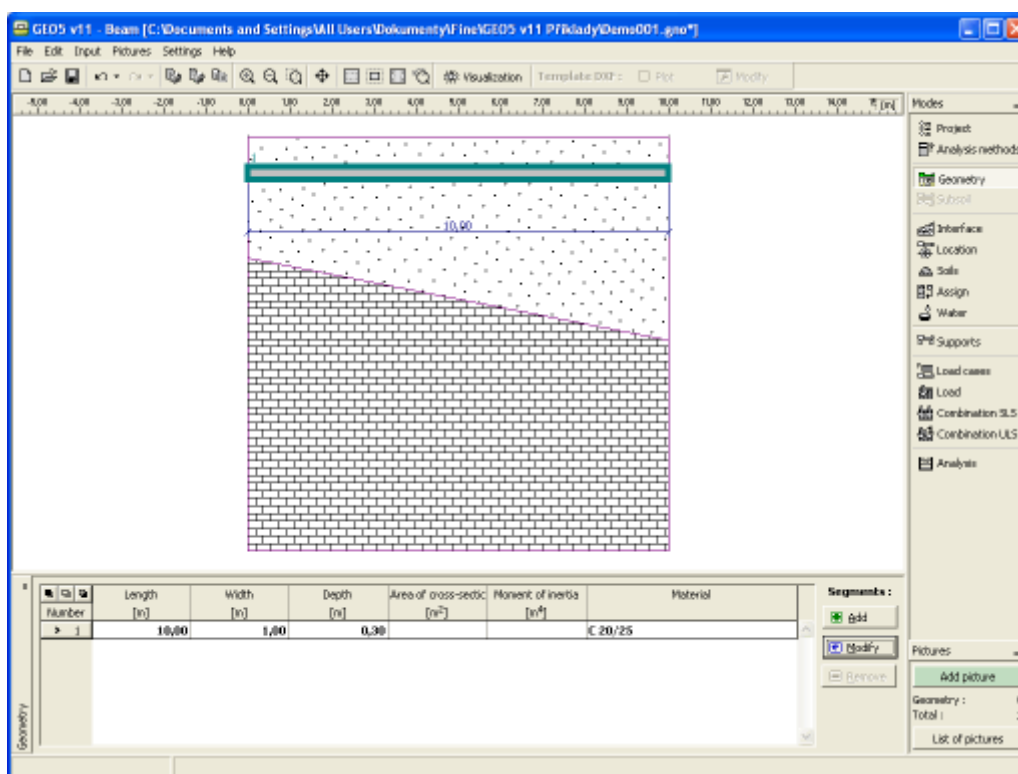
$$= \begin{bmatrix} \sum_{n=0}^{\infty} \frac{1}{2 \sqrt{H} \left[ (2n+1) \sqrt{C_{1w} C_{2w}} + (2n+1)^2 b C_{1w} \right]} & 0 \\ 0 & \sum_{n=0}^{\infty} \frac{1}{2 \sqrt{H} \left[ (2n+1) b^2 \sqrt{C_{1w} C_{2w}} + b C_{2w} + (2n+1)^2 \frac{b^3}{3} C_{1w} \right]} \end{bmatrix} \cdot d$$

where:  $[C]$  matrix of constants  $C_1$  a  $C_2$   
 $b$  half width of foundation  
 $c_{1w}, c_{2w}$  Winkler's constants  
 $H$  depth of deformation zone

## Geometry

The frame **"Geometry"** contains a table with a list of input beam sections. Adding (editing) points is performed in the **"New segments (Edit segment properties)"** dialog window. The window requires defining the section length, width and height (for **rectangular cross-section**). The program allows for defining a general cross-section of a beam (cross-sectional area and moment of inertia are specified).

**Material of the cross-section** is specified next, either using the program catalog, or by entering the material parameters (modulus of elasticity, shear modulus, self-weight).



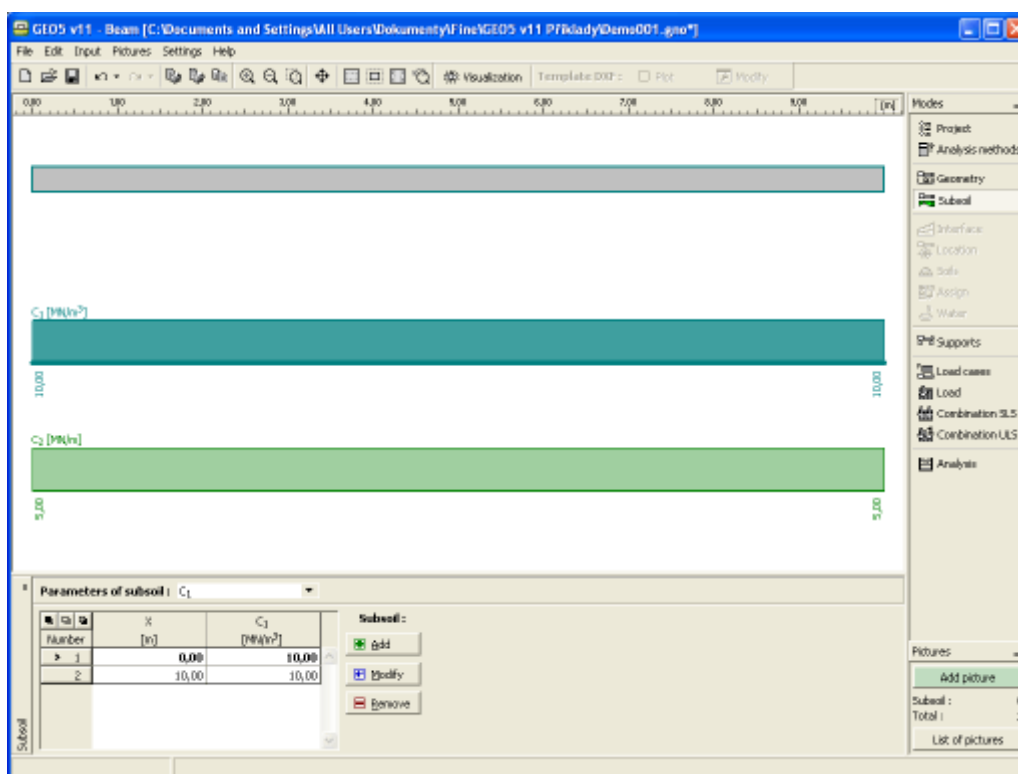
Frame "Geometry"

## Subsoil

The frame **"Subsoil"** contains a table with the list of values of the parameters of **Winkler-Pasternak subsoil**  $C_1$  and  $C_2$  or **deformation parameters of soils** ( $E_{def}$ ,  $n_y$ ,  $h_z$ ), respectively, depending on the setting in the frame **"Settings"**.

Inputting or editing parameters mode is selected from a combo list.

The table shows values of the parameter that is selected from the combo list above the table. Adding (editing) points is performed in the **"New subsoil parameters"** dialog window. The window serves to specify the  $X$ -coordinate and the value of the parameter.

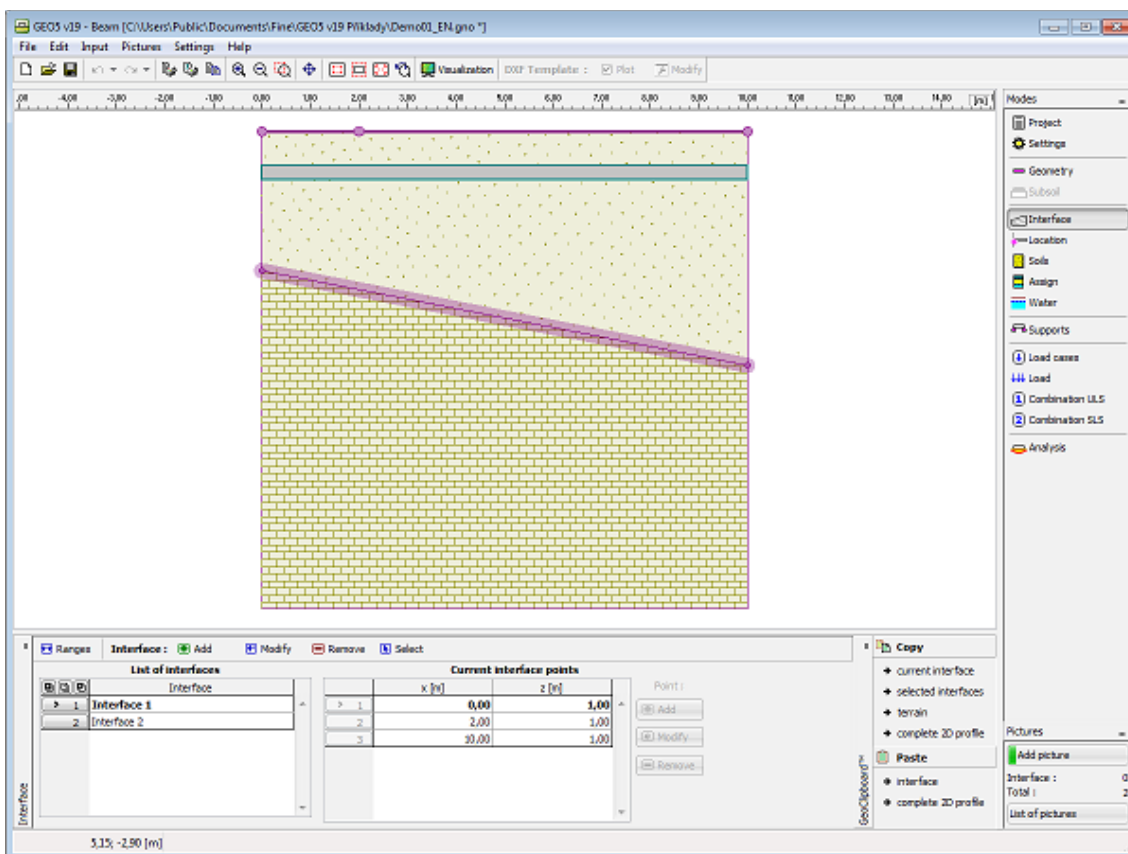


Frame "Subsoil"

## Interface

The frame "Interface" serves to introduce individual soil interfaces into the soil body. Detailed description on how to deal with interfaces is described [herein](#).

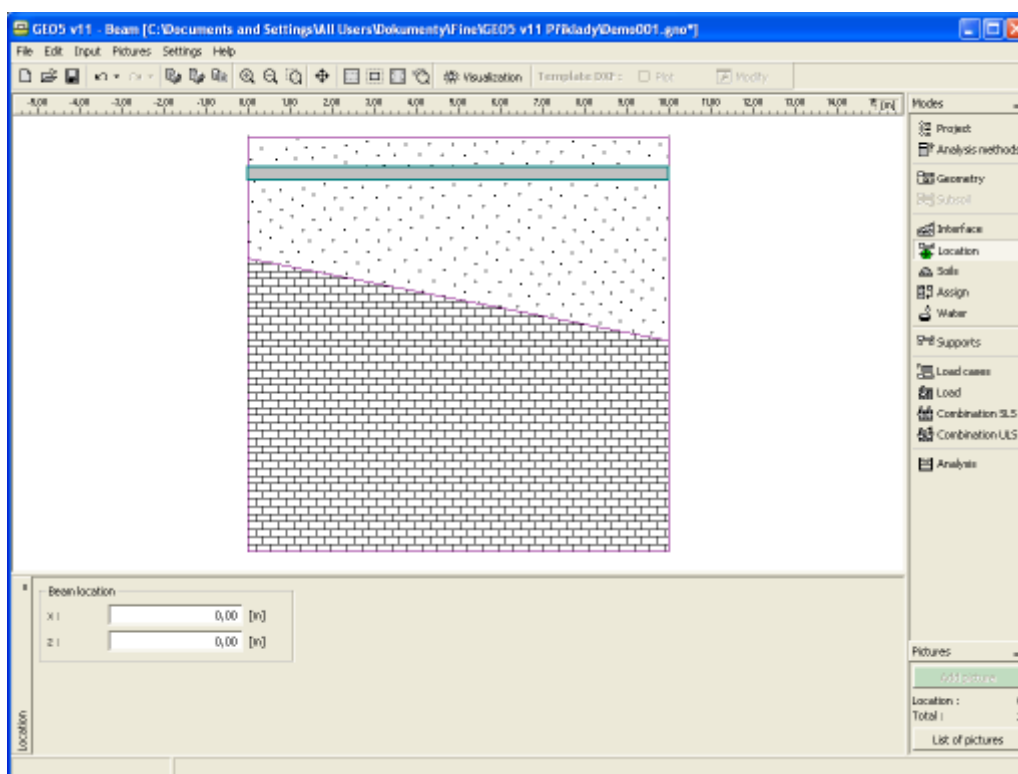
The program makes it possible to **import or export** interfaces in the \*.DXF format. They can also be imported in the **gINT format**. Input interfaces can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Interface"

## Location

The frame "**Location**" serves to specify the beam location. One needs to specify the beam origin - point having  $x$  and  $z$  coordinates.



Frame "Location"

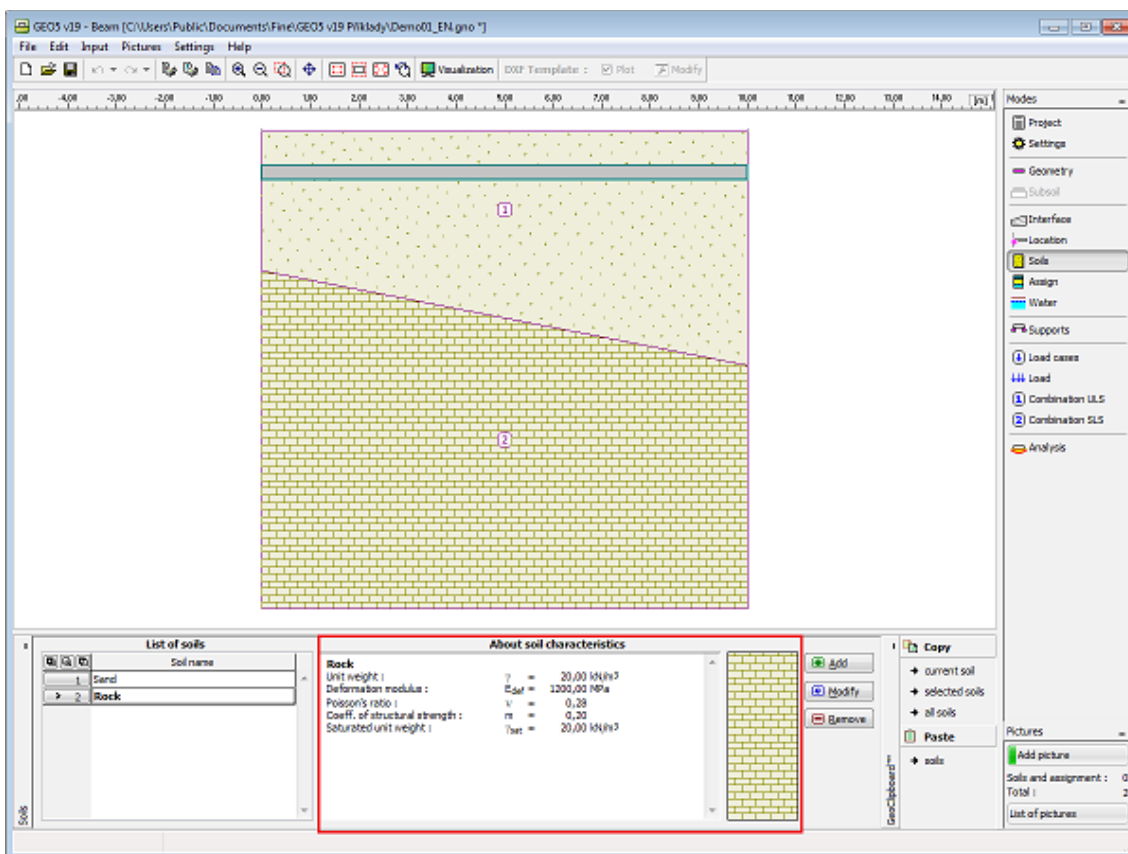
## Soils

The frame **"Soils"** contains a table with the list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Basic data", "Settlement - oedometric modulus", "Settlement - determination of the depth of influence zone" and "Uplift pressure".

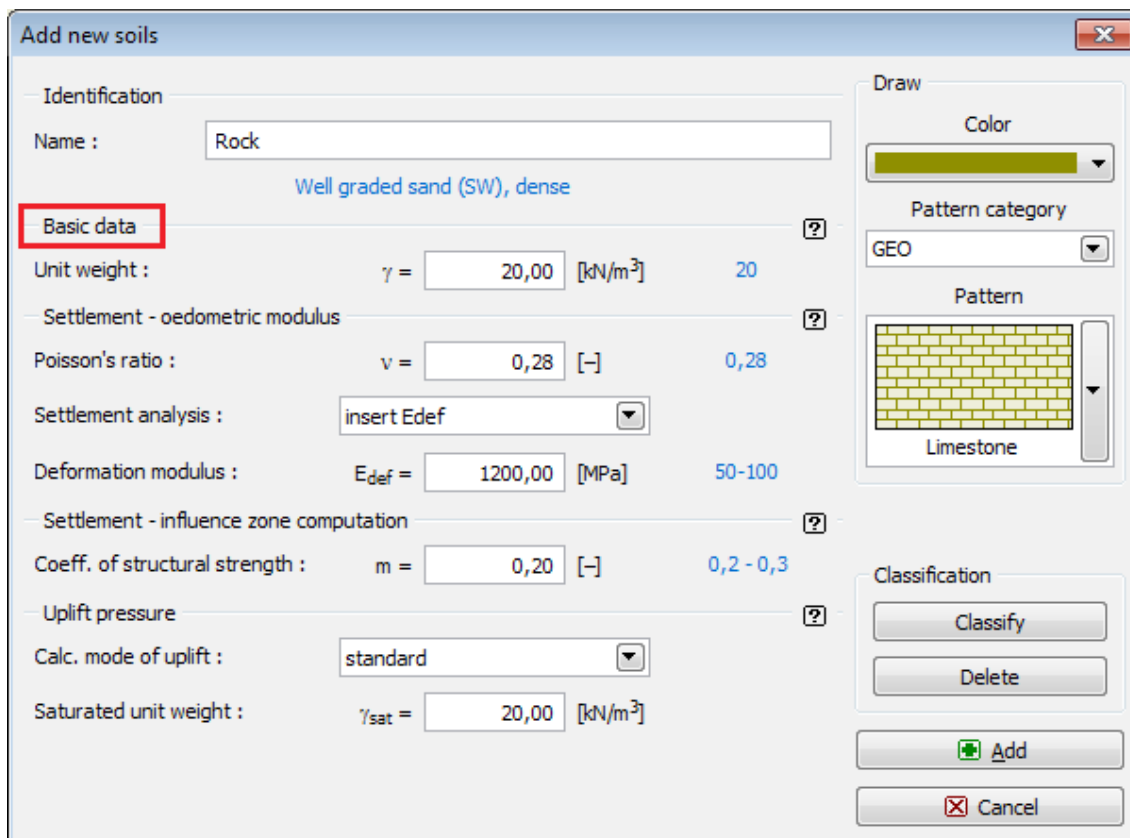
The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

This part of the dialog window serves to specify **the unit weight**.



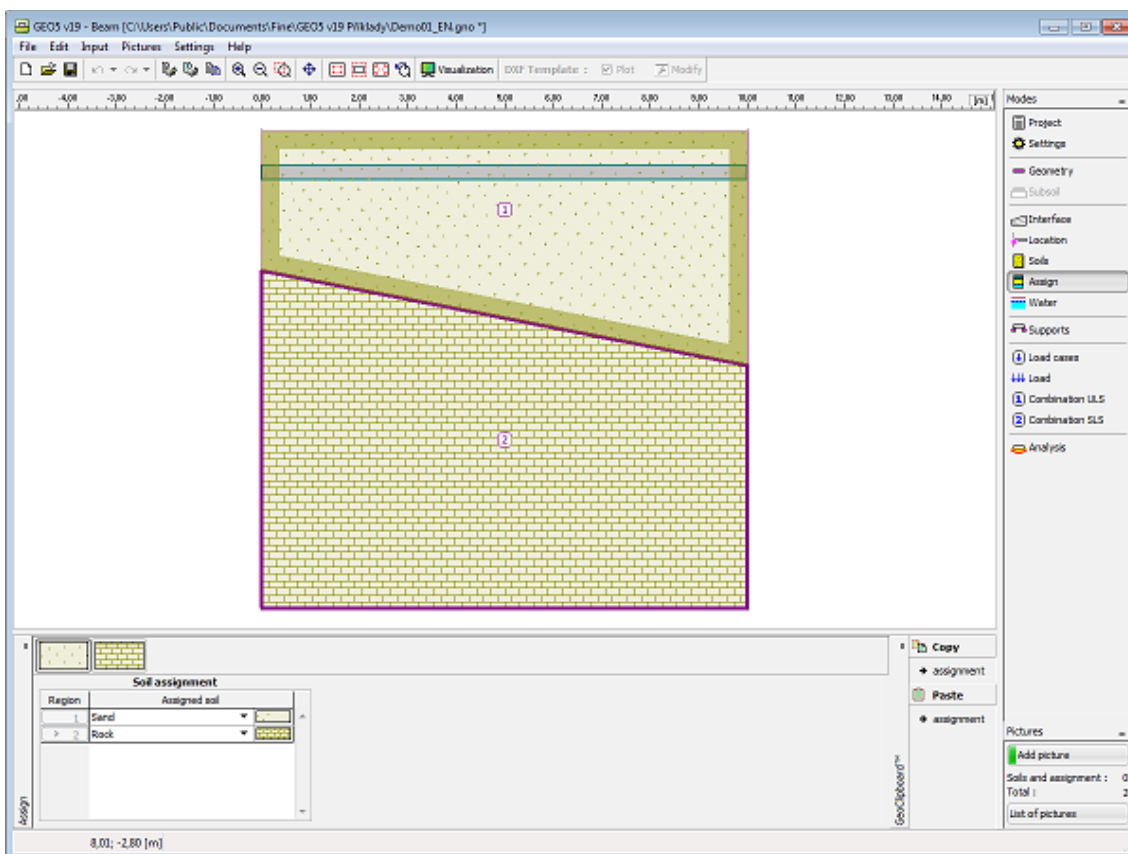
Dialog window "Add new soils" - "Basic data"

## Assign

The frame "Assign" contains a list of layers of the profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or it is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to import soil assignment in the [gINT format](#). Assign of soils can be copied within all 2D GEO5 programs using "GeoClipboard".



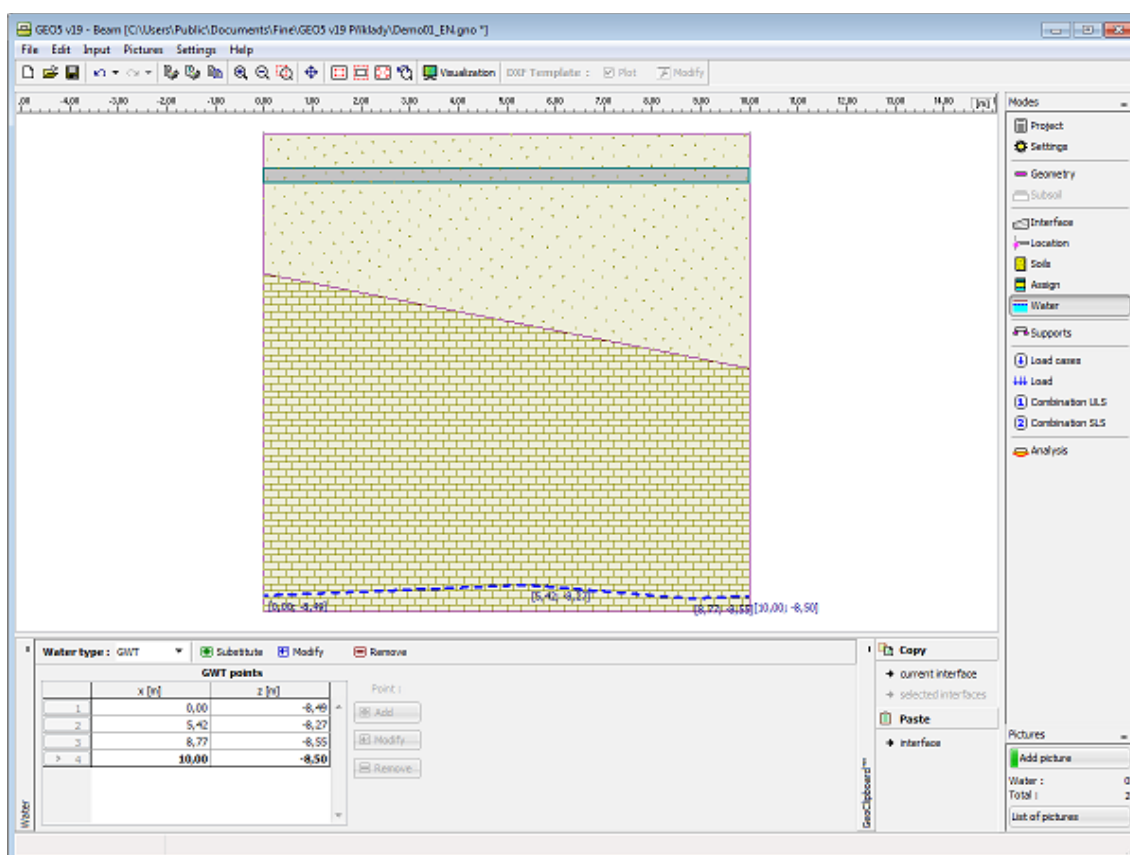
Frame "Assign"

## Water

The frame **"Water"** serves to input the ground water table. Distribution GWT is introduced the same way as **interfaces** of soils.

Input interfaces of water can be copied within all 2D GEO5 programs using **"GeoClipboard"**.





Frame "Water"

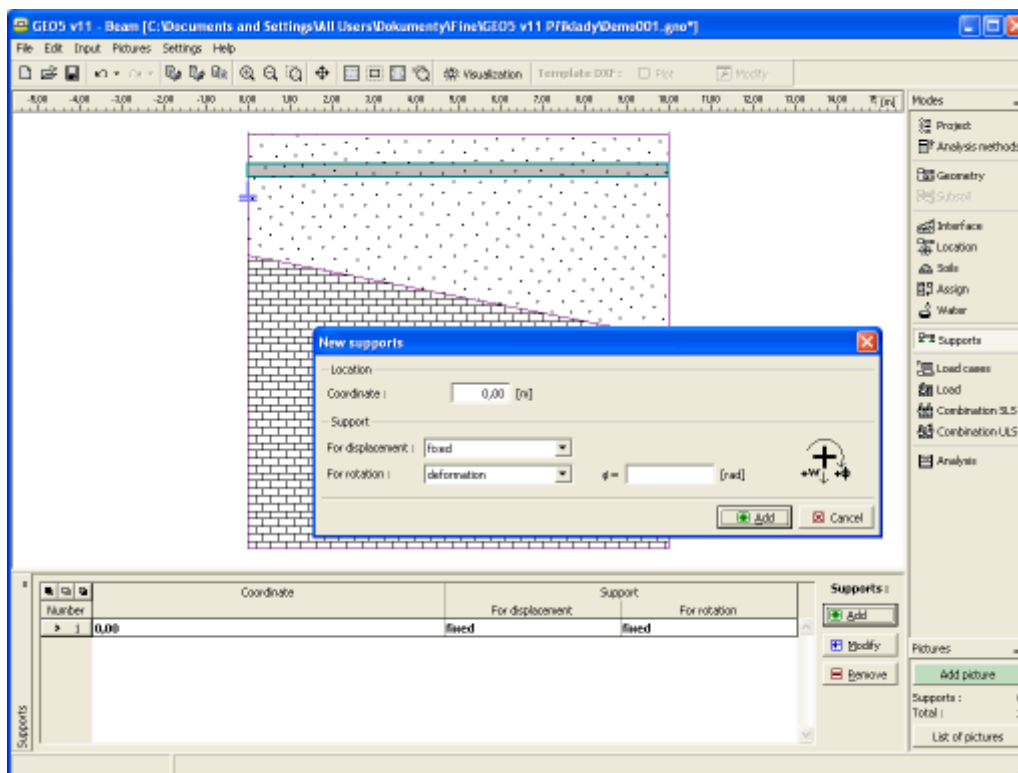
## Supports

The frame **"Supports"** contains a table with the list of input supports. Adding (editing) supports is performed in the **"New supports (Edit support properties)"** dialog window. Editing can be carried out with the help of **"Modify"** button or by clicking the row with the required support in the list using the left mouse button.

The type of support is determined according to a particular boundary condition specified at a given point (translation, rotation).

The following **boundary condition can be specified at a point:**

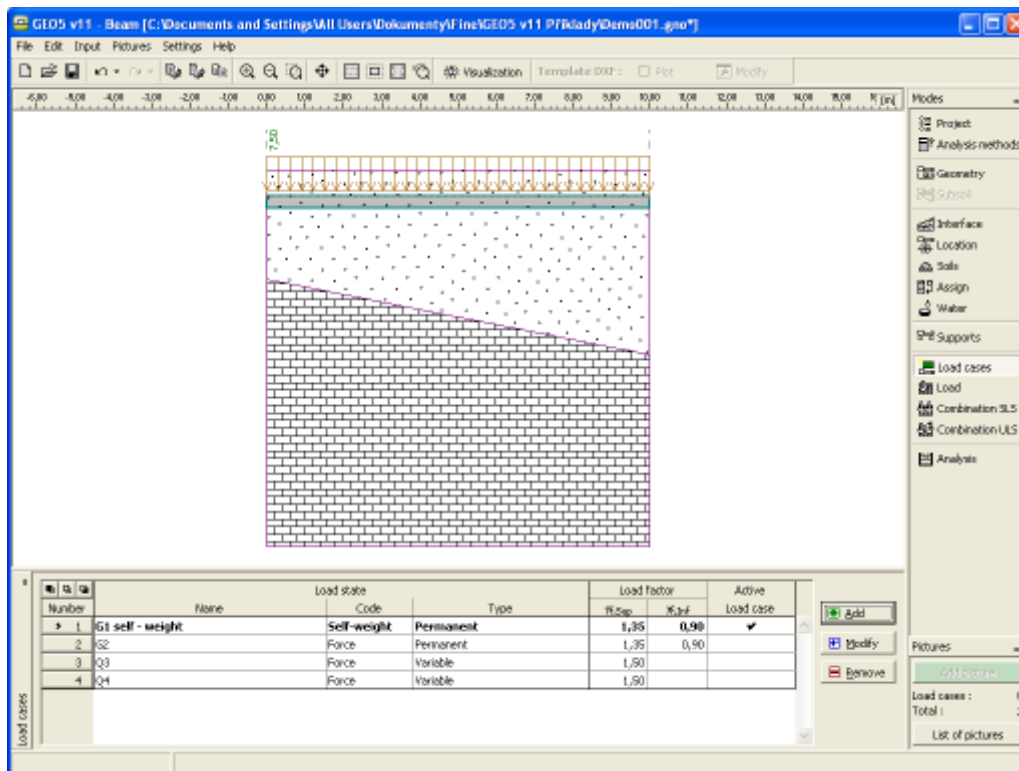
- free
- fixed
- deformation
- spring



Frame "Supports"

## Load Cases

The frame "Load cases" contains a table with the list of input load cases. Adding (editing) load cases and their parameters is performed in the "New load case" dialog window. Editing can be carried out with the help of "Modify" button or by clicking the row with the required load case in the list using the left mouse button.



Frame "Load cases"

## Load Case parameters

The following parameters are defined in the "New load case" dialog window:

### Load case identifier

The load case identifier, which is composed of the load case number and a uniliteral prefix, is displayed in front of the field for entering the name of the load case. The prefix is determined by the type of load case:

- G** - permanent load
- Q** - variable load
- A** - accidental load

The load case identifier is mainly used in printouts of combinations.

### Load case code

The load case code determines, what load can be specified for this load. The following options are available.

**Self-weight** In this load case the load represents the structure self-weight and it is generated automatically by the program. Only one load case with this code can be considered in each task.

**Force** An arbitrary type of force load (forces, moments) can be introduced into the load case with this code. The number of LCs is not limited.

### Load type

It determines the character of load cases based on their variability in time. Selecting a particular type of load corresponds to classification according to EN 1990 standard, art. 4.1.1.

### Load coefficients

It allows for specifying the load partial factor  $\gamma_f$ . This coefficient accounts for unfavorable deviations of values of loads from the representative ones. For permanent load it is necessary to introduce different values for favorable ( $\gamma_f, inf$ ) and for unfavorable ( $\gamma_f, sup$ ) load action in a combination. If the load input follows EN 1990 the default values of coefficients are taken from table A1.2(B).

### Category

Classification of load cases into categories corresponds to the classification of load according to table A1.1 of EN 1990 standard. Based on this the variable load cases are assigned combination coefficients  $\psi_0$ ,  $\psi_1$  and  $\psi_2$ . The category of "**User-defined input**" allows for defining the user self-values of these coefficients. Choosing a category is possible only for load cases input according to EN 1990 (the "**Materials and standards**" tab serves to select the particular standard).

### Combination coefficients

Basic values of coefficients to create combinations arise from EN 1990 standard and depend on the load case category. When user input is assumed, it is possible to define the user self-values of these coefficients. The following coefficients are used to create a combination:

- $\xi$  - **Coefficient of reduction of permanent loads in alternative combination** - this coefficient is assigned to all permanent loads and is used when compiling alternative combinations for the bearing capacity limit state (combination to relation 6.10.b, EN 1990).
- $\psi_0$  - **Coefficient of combination value** - coefficient for variable loads, it is used when compiling combinations for both the bearing capacity and service limit states
- $\psi_1$  - **Coefficient of frequent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state
- $\psi_2$  - **Coefficient of quasi-permanent value** - coefficient for variable loads, it is used when compiling accidental combinations and combinations for the service limit state

The combination coefficients are available only for load cases input according to EN 1990 (the frame "**Analysis methods**" serves to select the particular standard).

**New load case**

Load state

Name: Q5 force-variable

Code: force Type: variable

Load factor - unfavourable effect of load :  $\gamma_{f,Sup} =$  1,50 [-]

Load factor - favourable effect of load :  $\gamma_{f,Inf} =$  [-]

Category: [user-defined input]

Factor of permanent load reduction in alternative combination :  $\zeta =$  [-]

Factor of combination value :  $\psi_0 =$  [-]

Factor of frequent value :  $\psi_1 =$  [-]

Factor of quasi-permanent value :  $\psi_2 =$  [-]

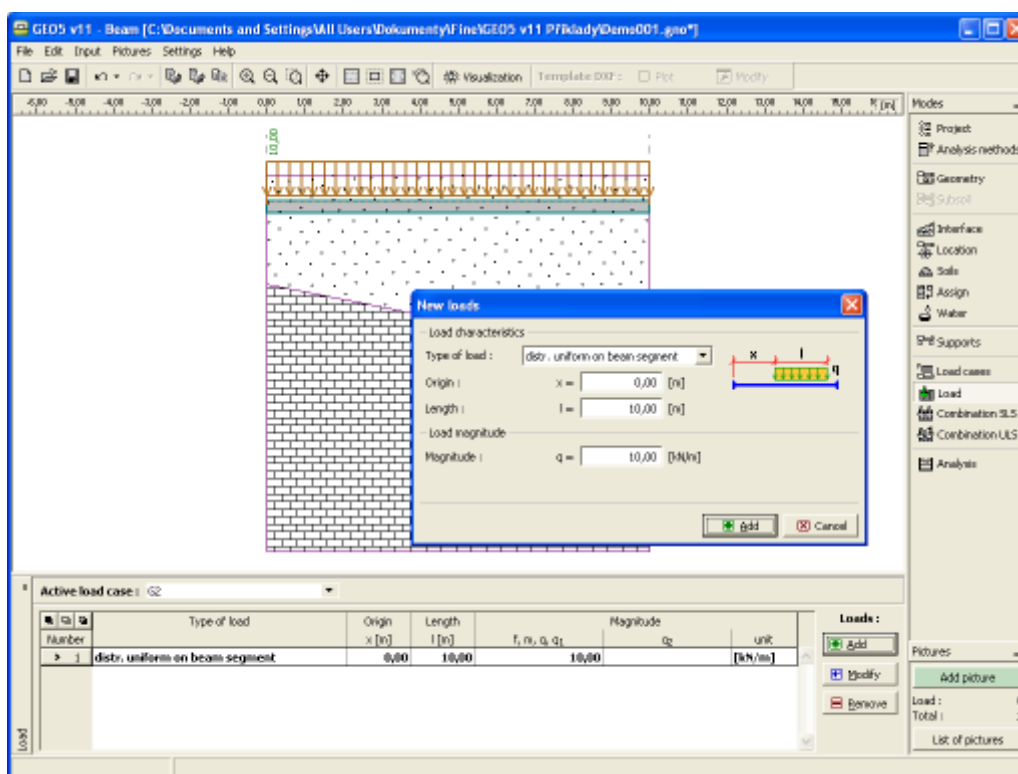
Add Close

*Dialog window "New load case"*

## Load

The frame "**Load**" contains a table with the list of input loads. Adding (editing) a load is performed in the "**New loads (Edit loads)**" dialog window. Editing can be carried out with the help of "**Modify**" button or by clicking the row with the required load in the list using the left mouse button.

Each load is assigned to a load case. The load case can be selected from the "**Active load case**" combo list above the table.

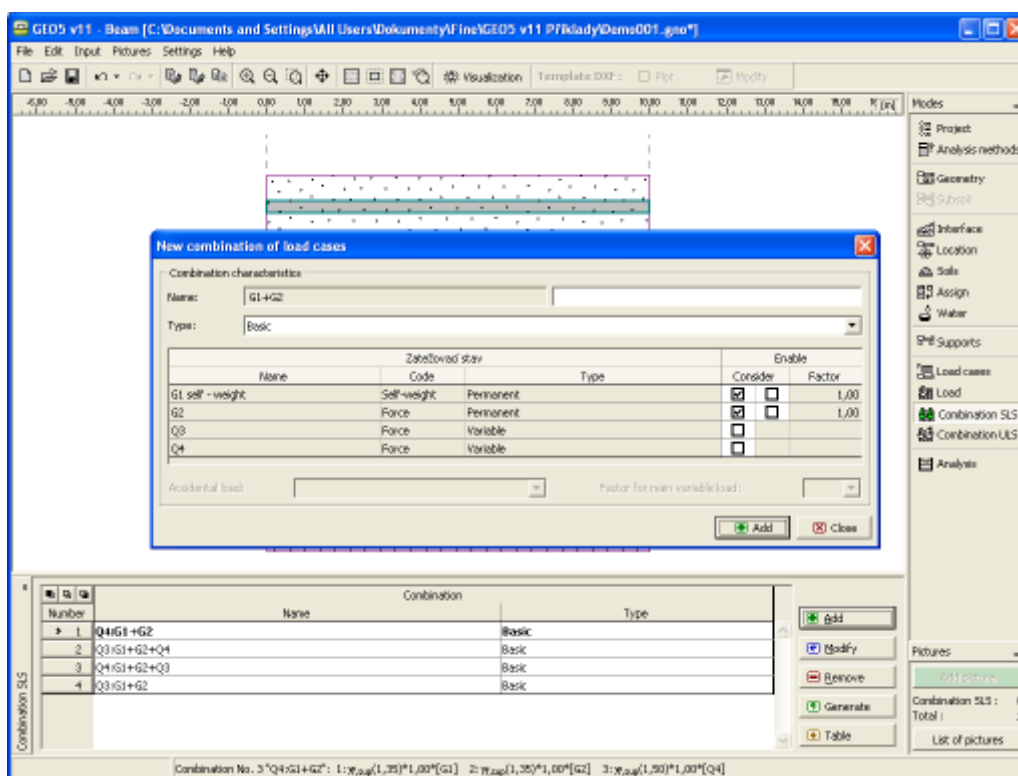


Frame "Load"

## Combination ULS

The **frame "Combinations ULS"** contains a table with the list of input combinations of the bearing capacity limit state. Adding (editing) combinations and their **parameters** is performed in the **"New combination of load cases"** dialog window. Editing can be carried out with the help of **"Modify"** button or by clicking the row with the required combination in the list using the left mouse button.

The built-in **generator of combinations** of load cases can be used to create individual combinations.



Frame "Combination of ULS"

## Parameters of ULS Combinations

The following parameters are specified in the "New combination of load cases" dialog window:

### New combination

A brief description of combination is displayed in front of the field where the combination is defined. All considered load cases are tagged using their identifiers. The major variable loads are moved at the beginning of the list and separated from the remaining LCs by colon.

### Type of combination (for combinations based on EN 1990 only)

The following combinations can be created for the bearing capacity limit state:

- Basic** Basic combination based on expression 6.10 of EN 1990 standard
- Alternative** Combinations based on expressions 6.10a and 1.10b of EN 1990 standard. In this case, two variants of combination are used in the analysis, one with reduced permanent LCs and the other with reduced major variable LC.
- Accidental** Accidental combination based on 6.11 of EN 1990 standard.

### Selection of load cases

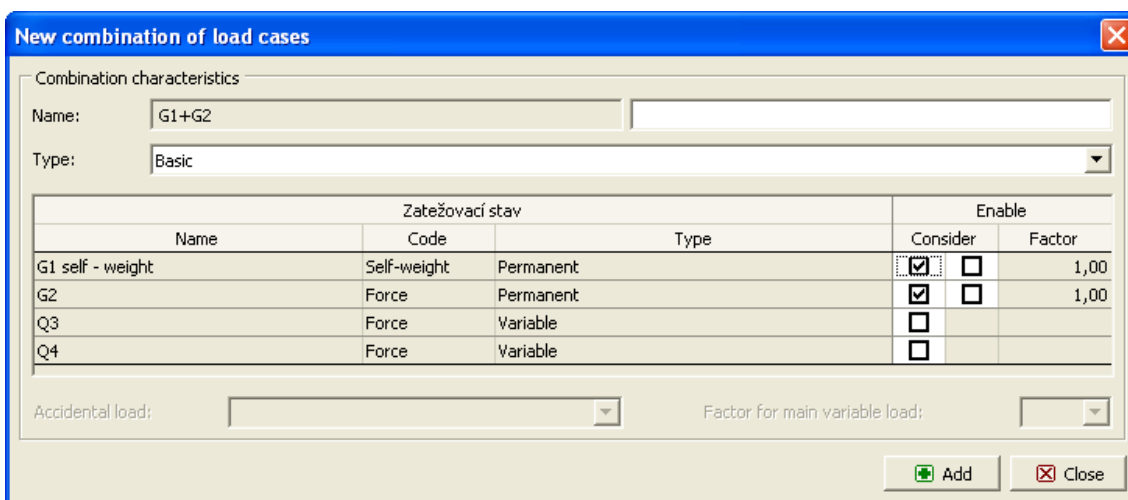
The table listing individual load cases allows for their selection to create a combination. The load case can be introduced into a combination by checking the field in the column "Consider" for a particular LC. Further setting in the table depends on the selection of way of inputting loads in the "Materials and standards" tab.

### Load according to EN 1990:

A second field is available for each load case in the column "**Consider**". This field allows for assigning a favorable effect of action to permanent LCs (adopting coefficient  $\gamma_f, inf$ ) or for specifying a variable load as the major one, respectively. The number of major variable loads in the combination is not limited. An accidental load can be introduced into combinations tagged as "**Accidental**" (only LCs tagged as "**Accidental**" are available for the selection). For accidental combinations it is also necessary to choose, whether a major variable load should be reduced by the coefficient  $\psi_1$  or  $\psi_2$ .

### General load

A coefficient of usability can be specified for each load case to adjust the degree of usability of the load case in the combination.

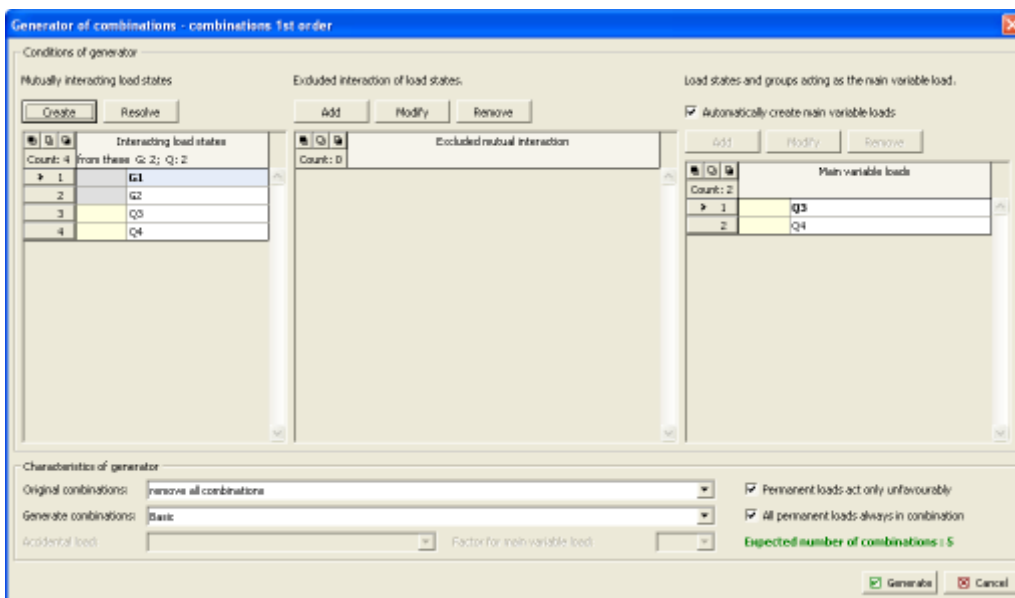


Dialog window "New combination of load cases"

## Generator of Combinations

The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases based on the introduced combination rules. Referring to standard EN 1990 the number of generated combinations can be relatively large and in extreme cases could considerably slow down calculations. Owing to this, information about expected number of combinations to be generated is displayed in the right bottom corner. Therefore, before launching generation the user may check, how many combinations will be generated and possibly adjust generator conditions. The top part of the window serves to define conditions for generating combinations; the bottom part contains various generator settings.

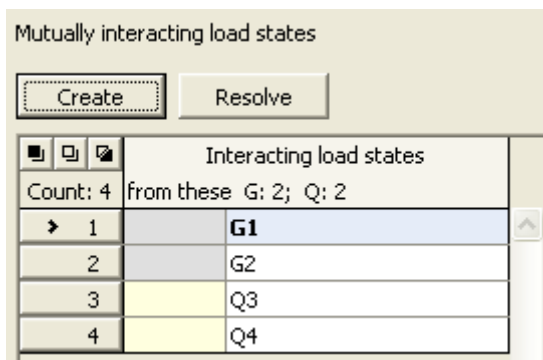




Dialog window "Generator of combinations - 1st order"

**Mutually interacting load states and groups**

This part makes it possible to merge those load states that should appear in combinations always together. Permanent and variable loads cannot be merged into one group. If the field **"All permanent loads always in combination"** is checked in the Generator parameters, the creation of groups of permanent loads has no effect on their appearance in combinations as each generated combination will always contain all permanent LCs. In such case, merging permanent LCs will only influence consideration of favorable/unfavorable effects of LCs providing the field **"Permanent loads act only unfavourably"** is not checked.



Dialog window "Generator of combinations" - Mutually interacting load states

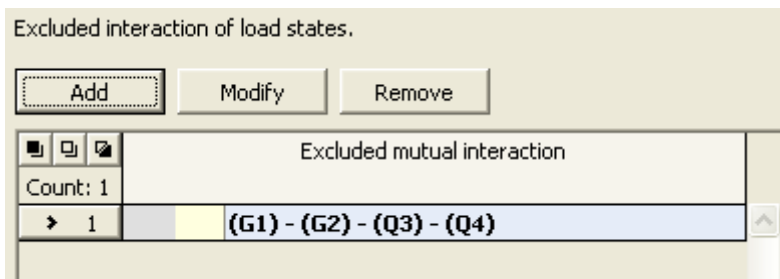
**Excluded interaction of load states**

This part makes possible to define, which LCs should not appear in a combination together. Arbitrary load cases or merged groups can be mutually excluded in dependent of the type of load case. Two options are available to define groups to be excluded:

**Mutual exclusion** An arbitrary number of load cases can be introduced into one group. In such a case, the program will not generate any combination that contains at least two load cases from this group.

**Exclusion by pairs** Providing it is necessary to create a larger number of excluding groups of two sorts, where one LC is the same (e.g. exclusion of assembly variants of permanent loads with all service load cases), it is possible to adopt this option.

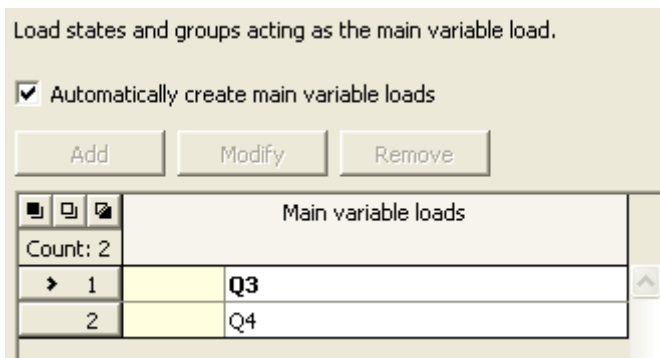
A load case to be excluded is first selected in the first column. The second column is then used to select an arbitrary number of LCs, which are needed to create excluding groups.



Dialog window "Generator of combinations" - Excluded interaction of load states

**Load cases and groups acting as the main variable load**

This part is available only when inputting loads according to EN 1990 is considered (the standard is selected in the "Materials and standards" tab). When automatic regime is assumed then each variable load is taken as major in created combinations. If this regime is turned off, it is possible to manually adjust the list of major variable loads. For example, it is possible to remove an arbitrary load case from the list so that it will not be considered as major variable in combinations. If a new item with more load cases is add to the list then all load cases will be considered as major variable in those combinations, where they appear together.



Dialog window "Generator of combinations" - Load cases and groups acting as the main variable load

**Generator parameters** (parameters that can be set in the bottom part of the dialog window).

**Combo list "Original combinations"**

**Retain original combinations**

- By pressing the "**Generate**" button the program will add new combinations, created according to the specified rules, to the original ones

**Remove all combinations**

- By pressing the "**Generate**" button the program will delete all original combinations and will replace them by the new ones

**Remove generated combinations**

- By pressing the "**Generate**" button the program will delete older combinations and will add new ones created according to the specified rules

**Remove all combinations of the current type**

- By pressing the "**Generate**" button the program will delete all combinations of a given type and will replace them by the new ones

**Remove generated combinations of the current type** - By pressing the "**Generate**" button the program will delete older combinations of a given type and will add new ones created according to the specified rules

### Combo list "Generate combinations"

The following types of generated combinations can be chosen for loads based on EN 1990:

- Basic** - Generates basic combinations for the bearing capacity limit state based on expression 6.10 of EN 1990 standard
- Alternative** - Generates combinations for the bearing capacity limit state based on expressions 6.10a and 1.10b of EN 1990 standard. This variant generates two times more combinations but it provides better results.
- Accidental** - Generates accidental combinations for the bearing capacity limit state based on 6.11 of EN 1990 standard. An accidental load case to appear in accidental combinations can be specified. It is also necessary to choose, whether a major variable load will be reduced by the coefficient  $\psi_1$  or  $\psi_2$ .

### Permanent loads act only unfavorably

If this setting is not checked, the program creates all possible combinations, where introduction of all variants of favorable and unfavorable actions of permanent loads will be considered.

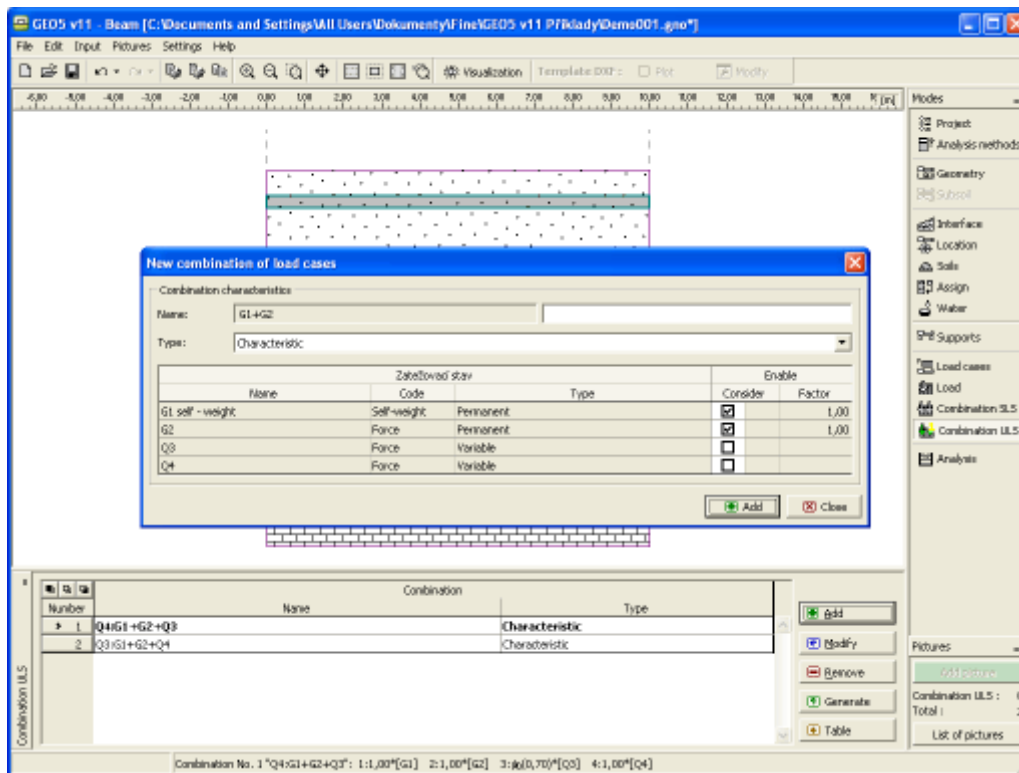
### All permanent loads always in combination

If this setting is not checked, the program creates combinations in such a way that a successive introduction of all LCs into a combination will be considered.

## Combination SLS

The **frame "Combinations SLS"** contains a table with the list of input combinations of the service limit state. Adding (editing) combinations and their **parameters** is performed in the "**New combination of load cases**" dialog window. Editing can be carried out with the help of "**Modify**" button or by clicking the row with the required combination in the list using the left mouse button.

The built-in **generator of combinations** of load cases can be used to compile individual combinations.



Frame "Combination SLS"

## Parameters of SLS Combinations

Combinations SLS serve to evaluate states that refer to the structure appearance, comfort of people or to functioning of a structure while in ordinary use. Typically, only deformations, vibrations, etc. are checked. The **"New combination of load cases"** dialog window (similarly to combinations for ULS) serves to define the following parameters:

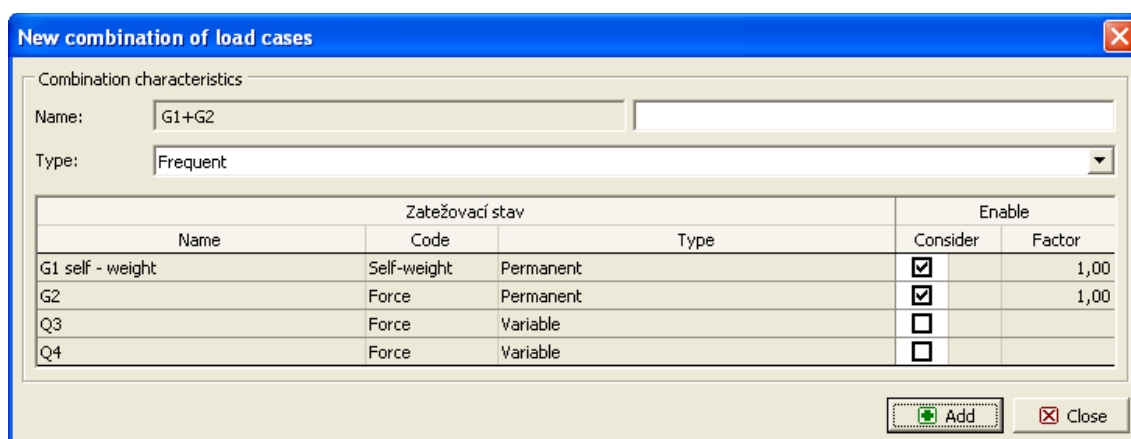
### Type of combination according to EN 1990

The following combinations can be created for the service limit state:

- Characteristic** - combination based on expression 6.14 of EN 1990 standard
- Frequent** - combination based on expression 6.15 of EN 1990 standard
- Quasi-permanent** - combination based on expression 6.16 of EN 1990 standard

### Selection of load cases

The table listing individual load cases allows for their selection to create a combination. The load case can be introduced into a combination by checking the field in the column **"Consider"** for a particular LC. A coefficient of usability can be specified for generally input combinations (select in the **"Materials and standards"** tab) to adjust the degree of usability of the load case in the combination.



Dialog window "New combination of load cases"

## Generator of Combinations

The "**Generator of combinations - 1st order**" dialog window allows for a collective compilation of combinations of load cases for the service limit state. Functions of generator of combinations are explained in section devoted to the [generator of combinations for the bearing capacity limit state](#).

If inputting loads according to EN 1990 is set in the "[Materials and standards](#)" tab, it is possible to generate the following combinations for the service limit state:

- Characteristic** - combination based on expression 6.14 of EN 1990 standard
- Frequent** - combination based on expression 6.15 of EN 1990 standard
- Quasi-permanent** - combination based on expression 6.16 of EN 1990 standard

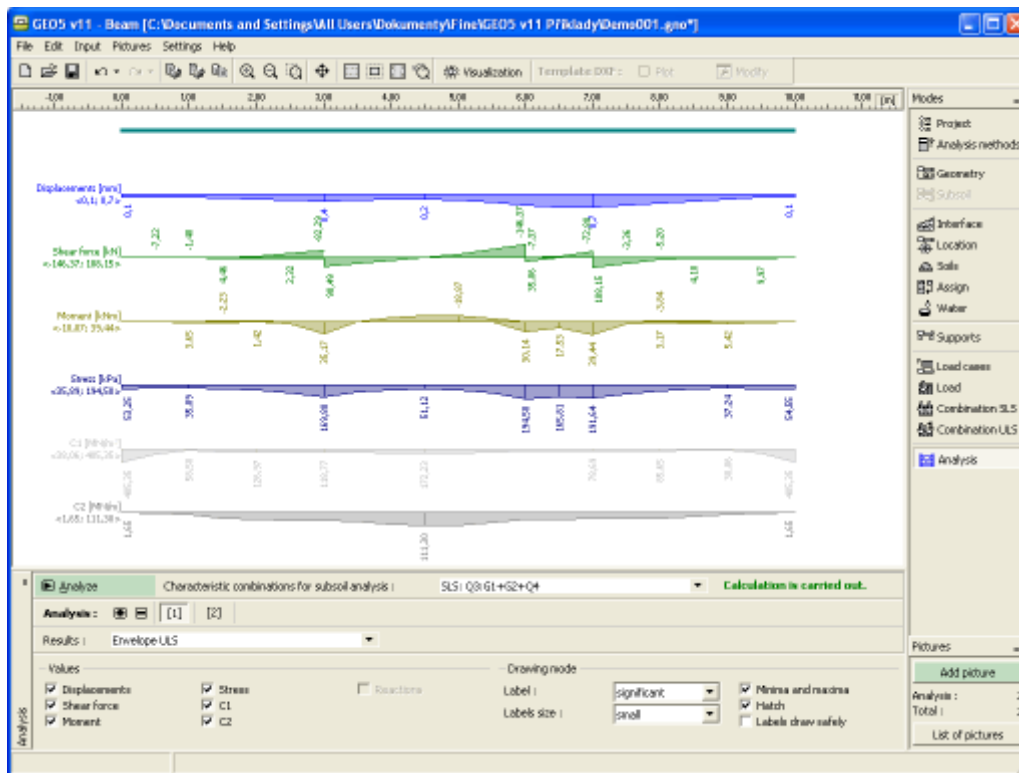
## Analysis

The frame "**Analysis**" also serves to display the analysis results. The analysis is carried using the **finite element method** incorporating the [Winkler-Pasternak](#) subsoil. [Several analyses](#), including presentation of results, can be carried out for one task. Information about the performed analysis is displayed in the top right corner of the frame. Should the analysis parameters change it is necessary to re-run the analysis by pressing the "**Analyze**" button.

If the subsoil parameters are [calculated from the geological profile](#), it is necessary to choose "**Characteristic combination for subsoil analysis**" in the combo list.

The "**Results**" combo list serves to set combinations of load for [ULS](#) or [SLS](#) (possibly envelopes of combinations of load cases) for which the results should be displayed on the desktop.

The bottom part of the window serves to define, which variables are visualized (Displacements, Shear force, Moment...) and the way of their appearance on the desktop.



Frame "Analysis"

## Program Settlement

This program is used to determine vertical settlement and time-dependent consolidation of soils under embankments, foundations, earth dams and surface loads (surcharges).

The help in the program "Settlement" includes the following topics:

- Input of data into individual frames:
 

Project	Settings	Interface	Embankment	Earth Cut	Incompressible
Assign	Surcharge	Water	Stage	Analysis	Soils
			Settings		Subsoil
- Standards and analysis methods
- Theory for analysis in the program "Settlement":
 

Stress in Soil Body	Settlement Analysis
---------------------	---------------------
- Outputs

- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.

Frame "Project"

## Settings

The **frame "Settings"** allows to introduce the basic **settings** of the program, such as standards and theories of analysis, the way of proving safety of a structure and individual coefficients of the analysis.

The programs not only contain the pre-defined **basic Settings** for individual countries, but also allow the user to create their own **user-defined Settings**, which can be subsequently used in all GEO5 programs.

The **"Select"** button allows to choose an already created setting from the **"Settings list"**.

The **"Settings Administrator"** button opens the **"Administrator"** dialog window, which allows for viewing and modifying individual Setting. It is also possible to identify the visible settings in the Settings list. Data in the Settings administrator can be also **exported and imported**.

The **"Add to the administrator"** button allows to **create user-defined Settings**, which are subsequently added to the Settings administrator.

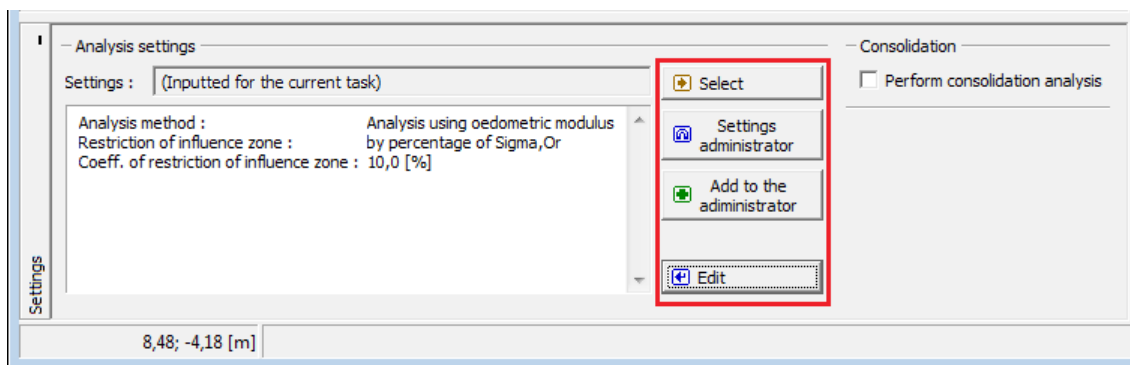
The **"Modify"** button enables a quick visualization and editing of the current Setting in the opened program. Modifying any of the parameters changes the title to **"Input for the current"**

**task**". Individual analyses are then performed with this **local setting**. Should we consider this setting as suitable also for other tasks, we add the setting into the "**Settings administrator**" by pressing the "**Add to the administrator**" button.

The "**Input for the current task**" setting is usually created when **importing older data**.

Settings of analysis parameters are performed in the "**Settlement**" tab.

The frame also allows to specify whether to perform the **consolidation analysis**.



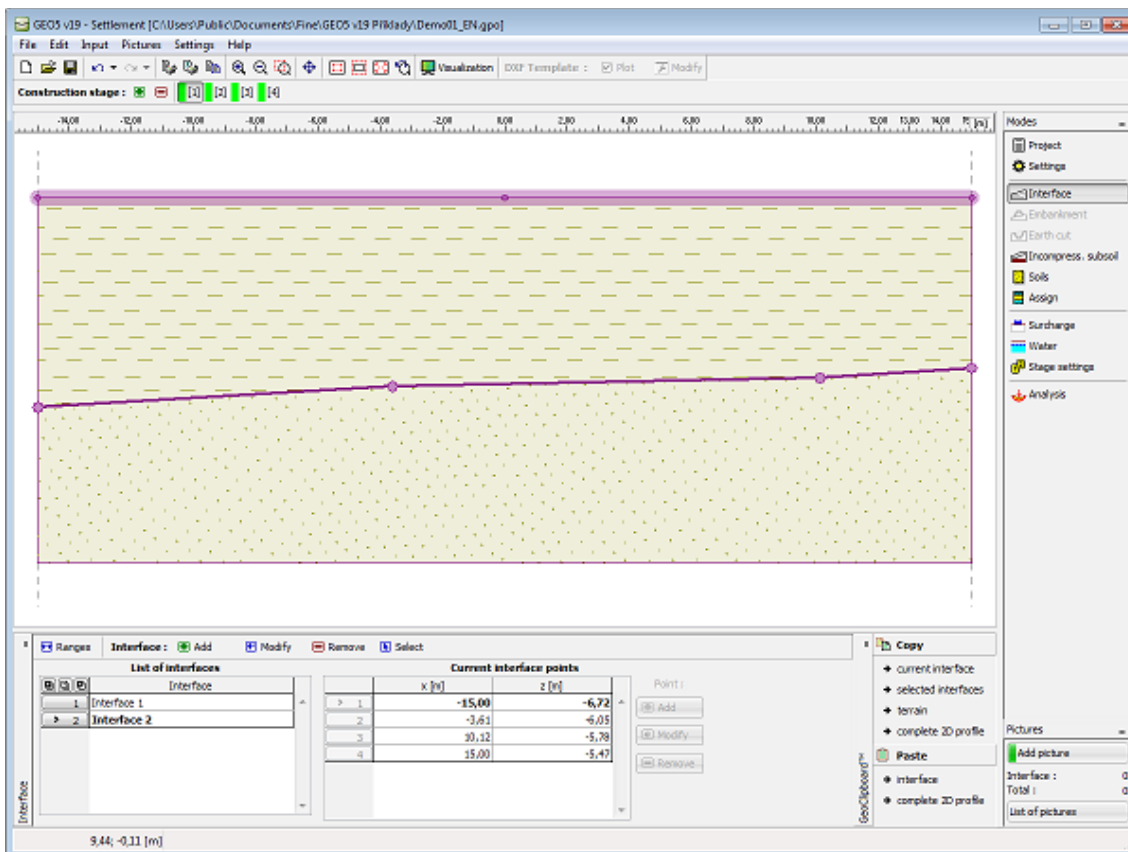
*Frame "Settings"*

## Interface

The "**Interface**" **frame** serves to introduce individual soil interfaces into the soil body. Detailed description how to deal with interfaces is described **herein**.

The program makes it possible to **import or export** interfaces in the \*.DXF format. They can also be **imported in the gINT format**. Input interfaces can be copied within all 2D GEO5 programs using "**GeoClipboard**".





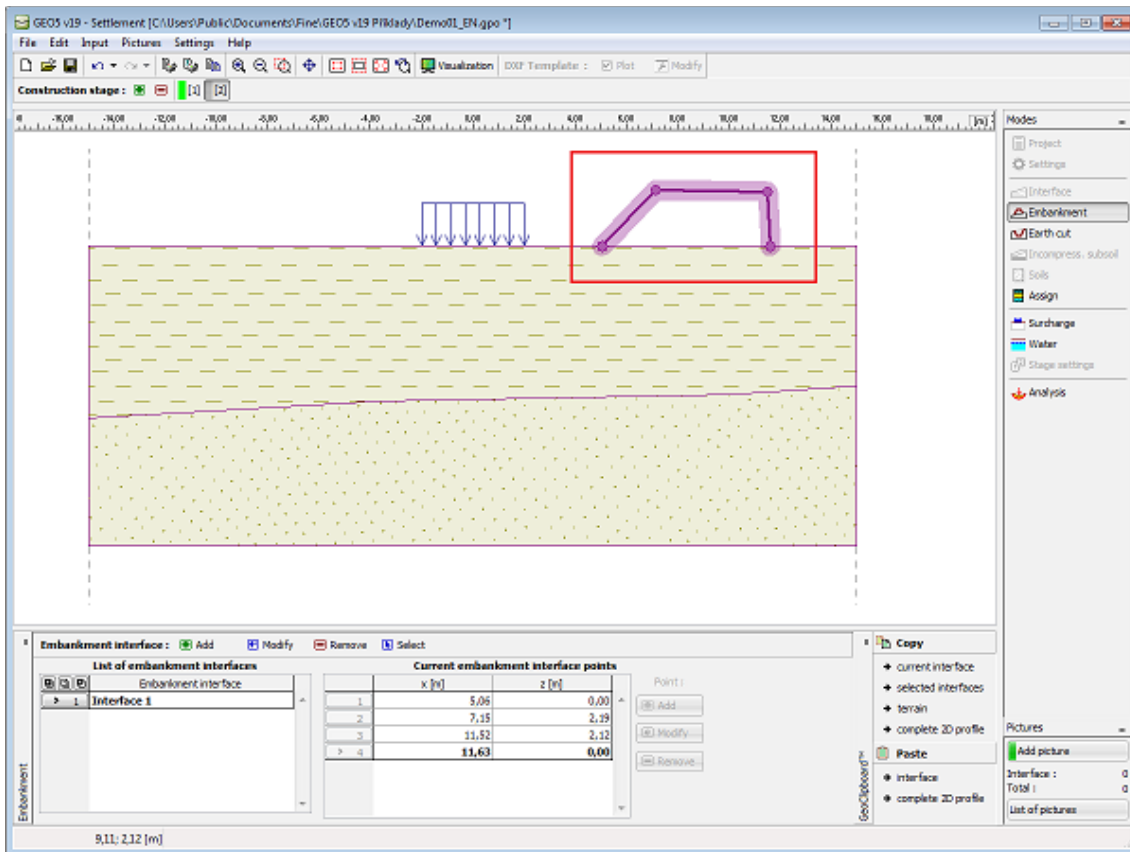
Frame "Interface"

## Embankment

The "**Embankment**" frame allows for **inputting interfaces** to create an embankment above the current terrain. The frame contains a **table** with a list of interfaces forming the embankment. A table listing the points of currently selected interface of the embankment is displayed in the mid section of the frame. Inputting an embankment interface follows the same steps as used for **standard interfaces**.

An embankment cannot be specified in the first **stage of construction**. An embankment cannot be built if there is an **earth cut** already specified in a given stage - in such a case either a new stage of construction must be introduced for embankment input or the existing open cut must be first removed.

Input interfaces of an embankment can be copied within all 2D GEO5 programs using "**GeoClipboard**".



Frame "Embankment"

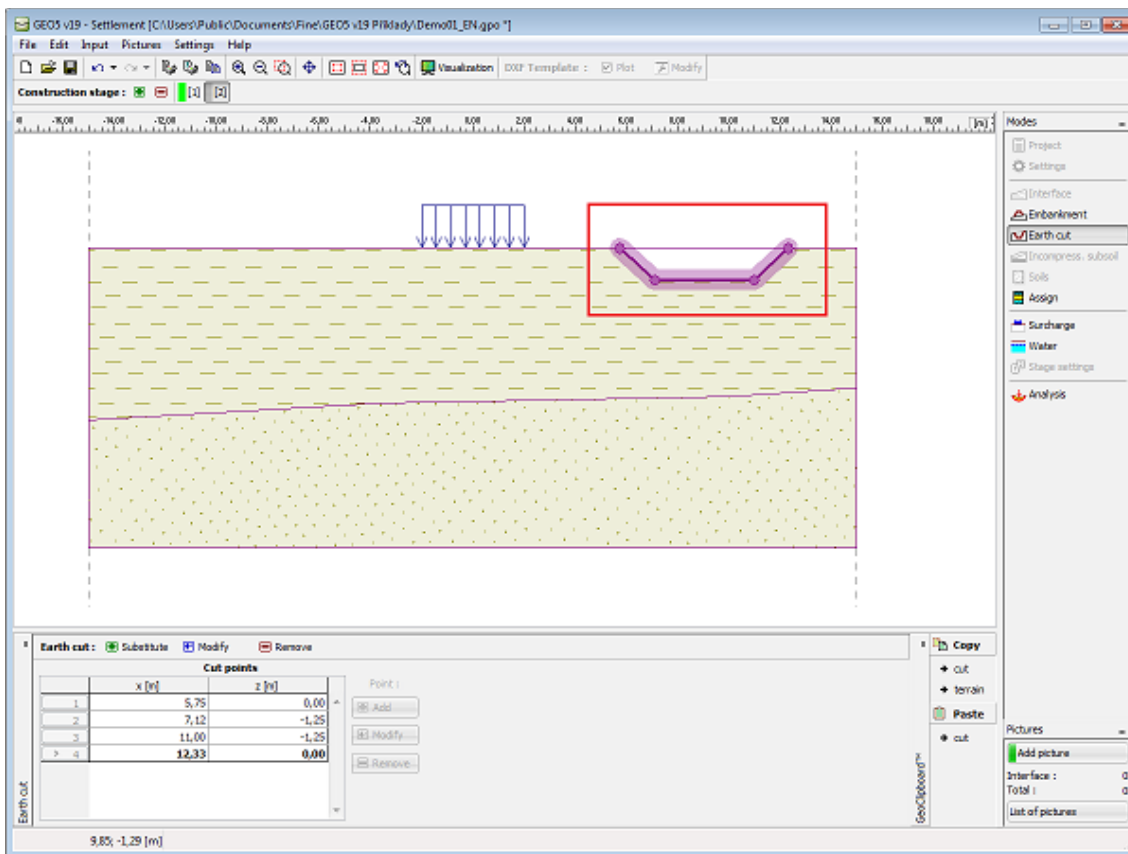
## Earth Cut

The "**Earth cut**" frame serves to specify the shape of an open cut. This function allows for modifying the terrain profile within a given **stage of construction**. Several **earth cuts** can be introduced at the same time. In such a case some of the lines in the cut appear partially above the terrain.

A **table** listing individual interface points is displayed in the left part of the frame. Inputting an earth cut interface follows the same steps as used for **standard interfaces**.

An open cut cannot be specified in the first **stage of construction**. An earth cut cannot be built if there is an **embankment** already specified in a given stage - in such a case either a new stage of construction must be introduced for earth cut input or the existing embankment must be first removed.

Input interfaces of an earth cut can be copied within all 2D GEO5 programs using "**GeoClipboard**".



Frame "Earth cut"

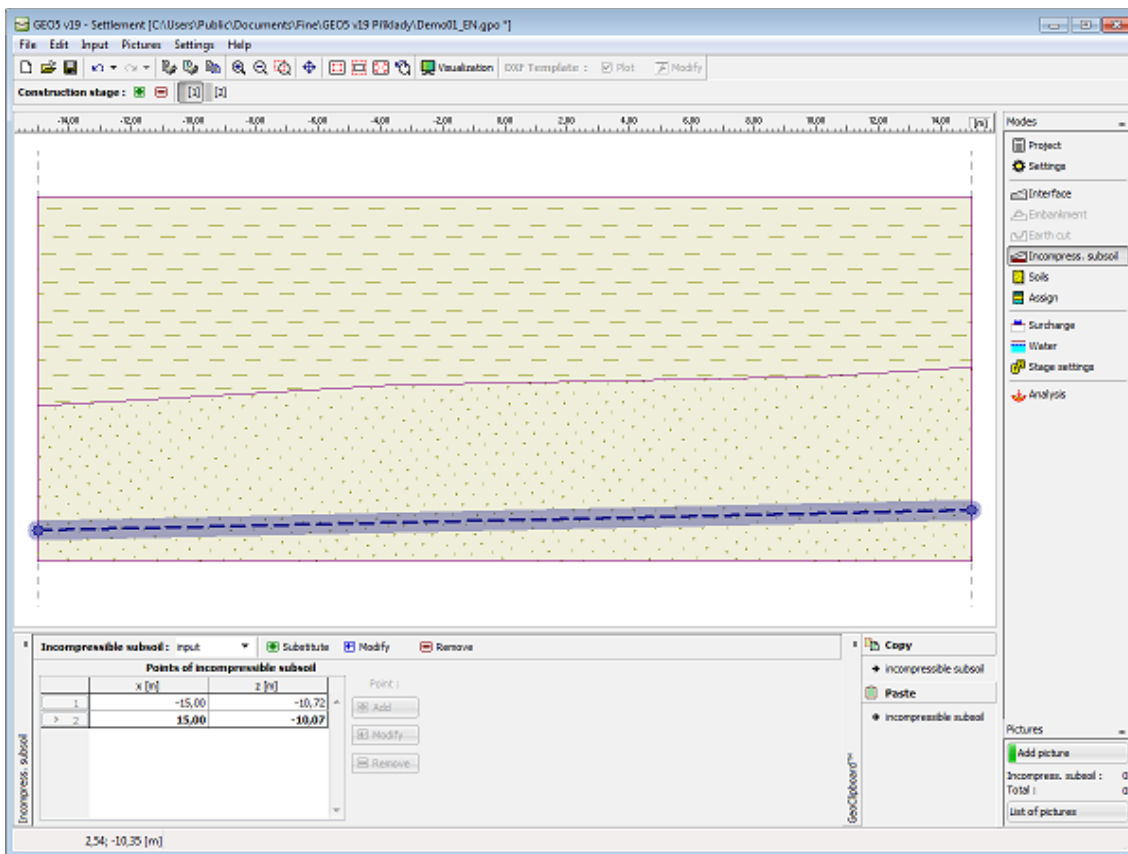
## Incompressible Subsoil

The frame "Incompressible subsoil" serves to input a depth of **incompressible subsoil**.

Inputting the depth of incompressible subsoil is the same as when inputting **standard interfaces**.

Inputting an incompressible subsoil is one the options how to restrict an influence zone - if input, then both ranges and tilted sections are drawn up to a depth of incompressible subsoil. No ground deformation appears below the incompressible subsoil.

Input interfaces of an Incompressible subsoil can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Incompressible subsoil"

## Soils

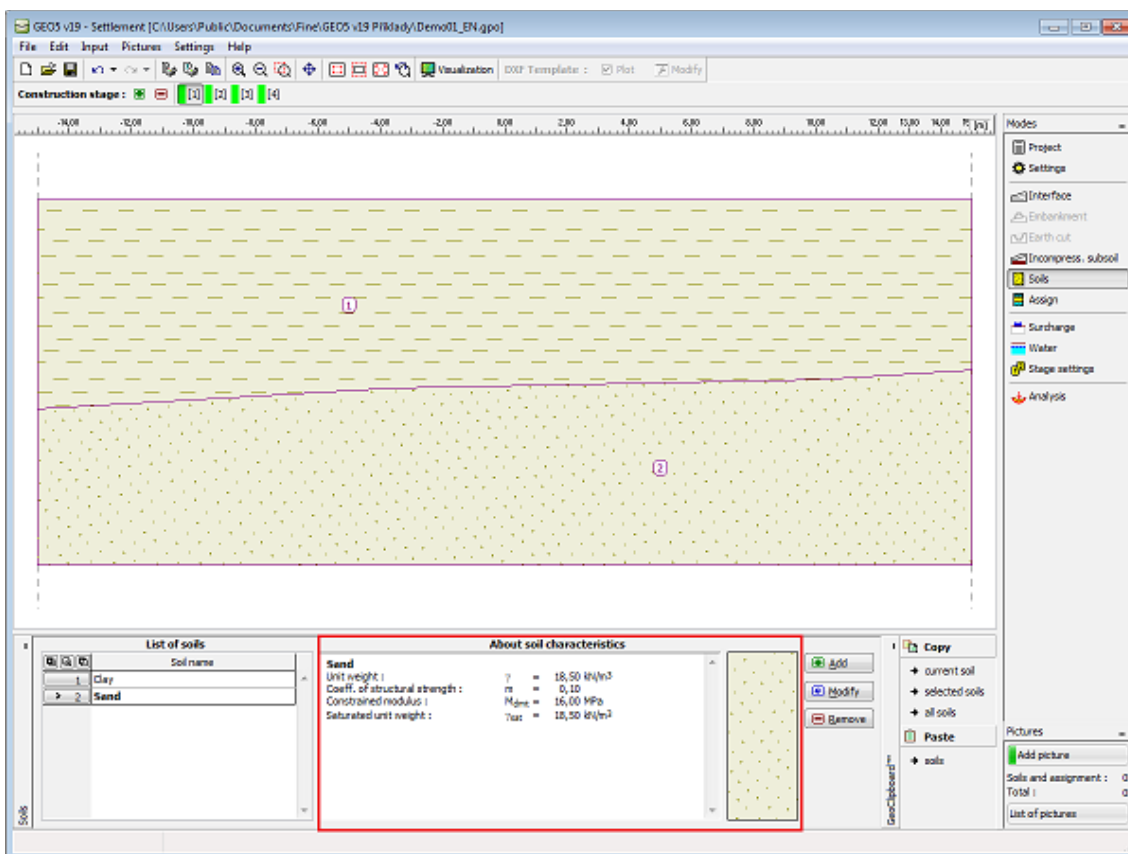
The **"Soils"** frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window.

The soil characteristics needed in the program are further specified in the following chapters: "Uplift pressure" and "Settlement analysis". In consolidation analysis the coefficient of permeability or consolidation coefficient must be entered. The input parameters of soils are determined based on the selected theory of analysis in the "Settlement" tab.

The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in database of soils, which contains values of selected characteristics of soils.

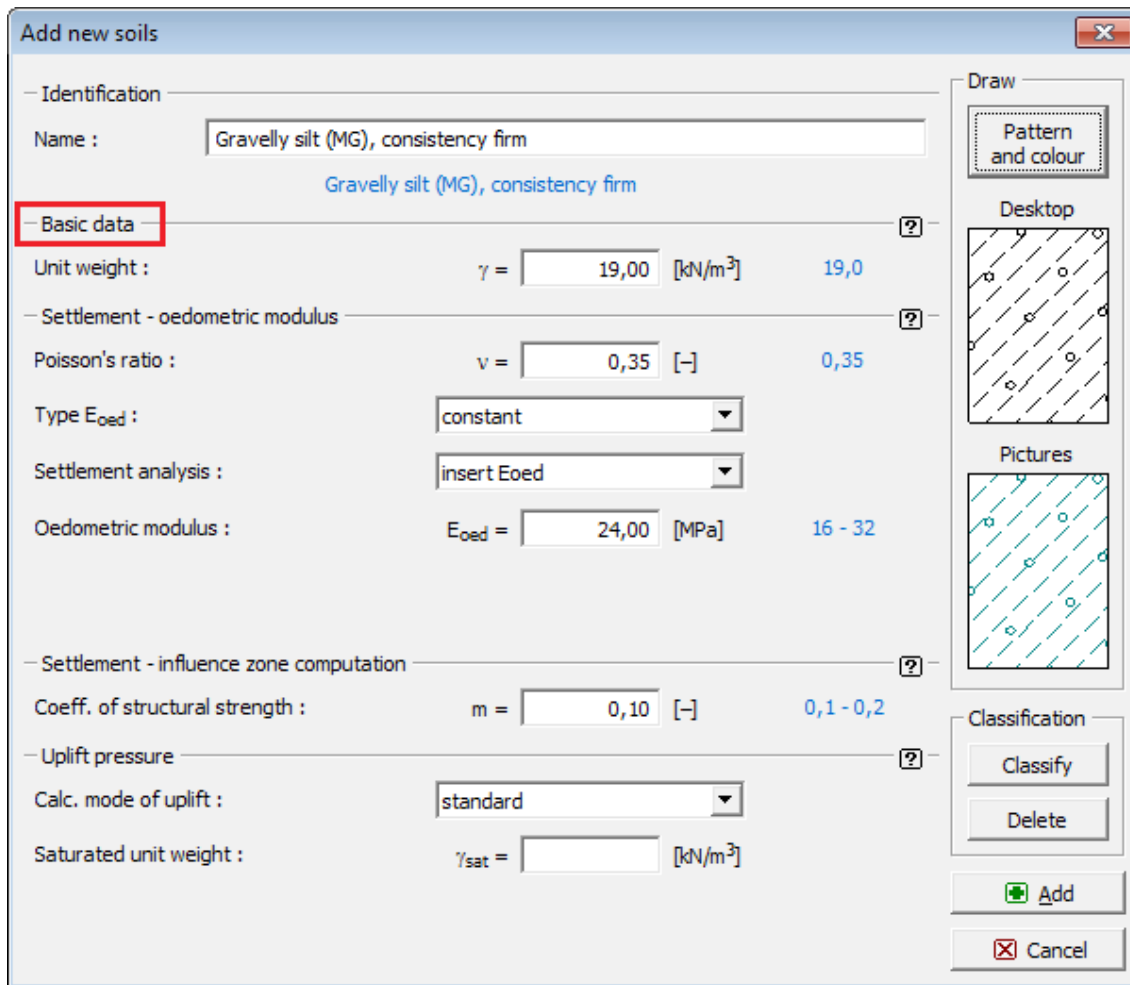
The program makes it possible to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

## Basic Data

This part of the dialog window serves to specify **the unit weight of soil**.



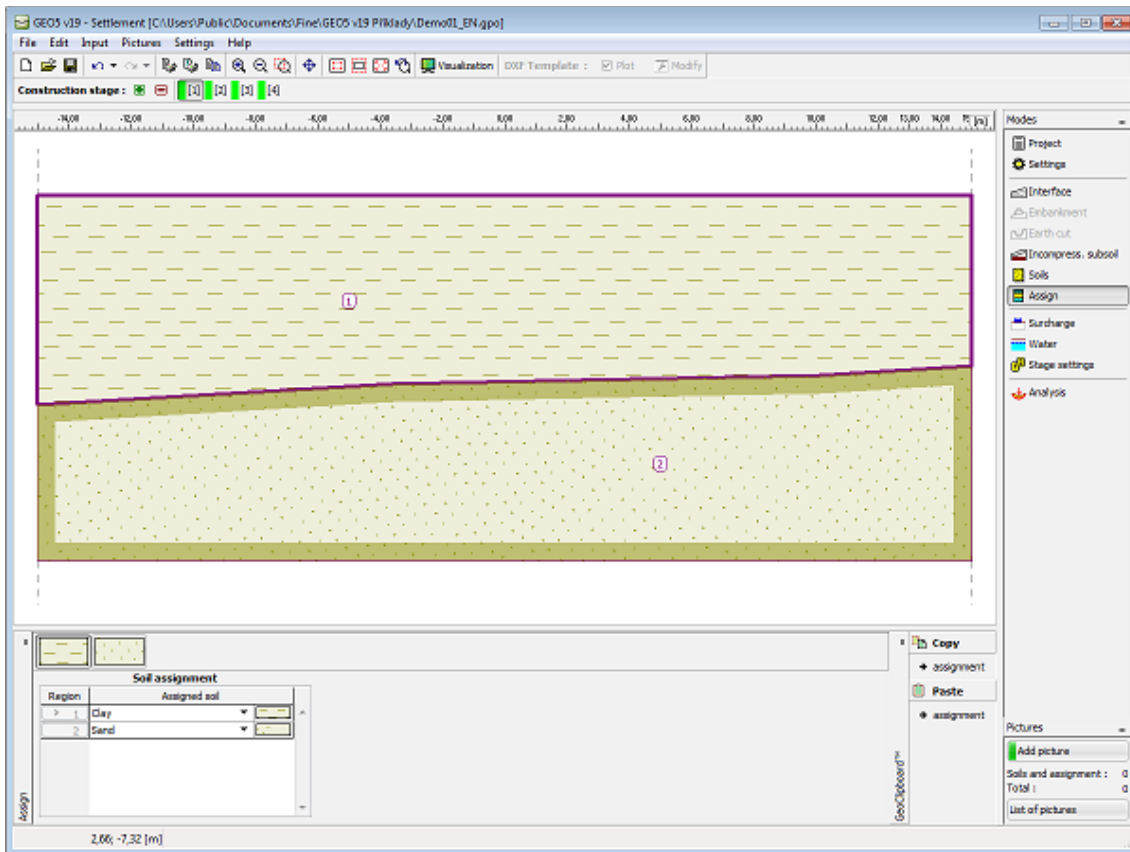
Dialog window "Add new soils" - "Basic data"

## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

The program makes it possible to [import soil assignment in the gINT format](#). Assign of soils can be copied within all 2D GEO5 programs using "GeoClipboard".



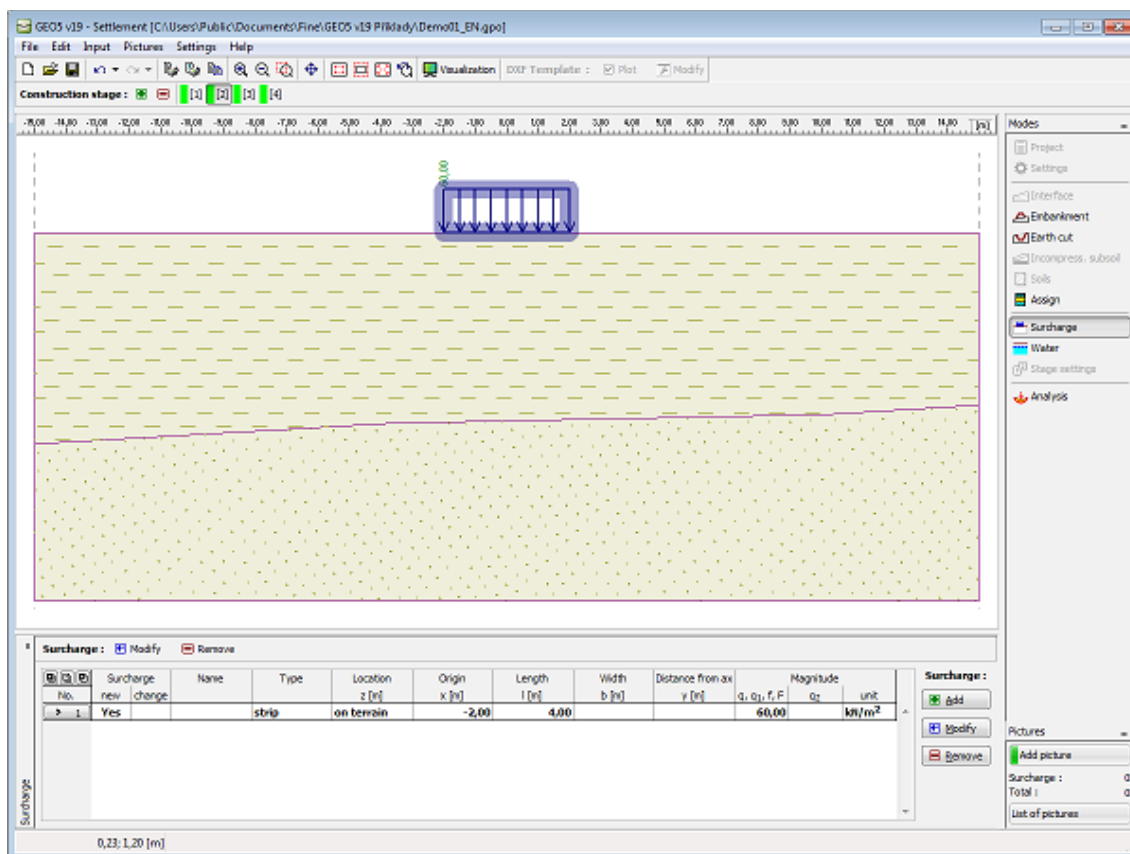
Frame "Assign"

## Surcharge

The "**Surcharge**" frame contains a table with a list of input surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can be edited on the desktop with the help of active objects.

All input parameters of a surcharge can be modified in the construction stage where the surcharge was specified. Only the surcharge magnitude can be modified in all subsequent construction stages (option "**Adjust surcharge**").

**Influence of surcharge** on the change of stress in the soil body is described in the theoretical part of the help section.



Frame "Surchage"

## Water

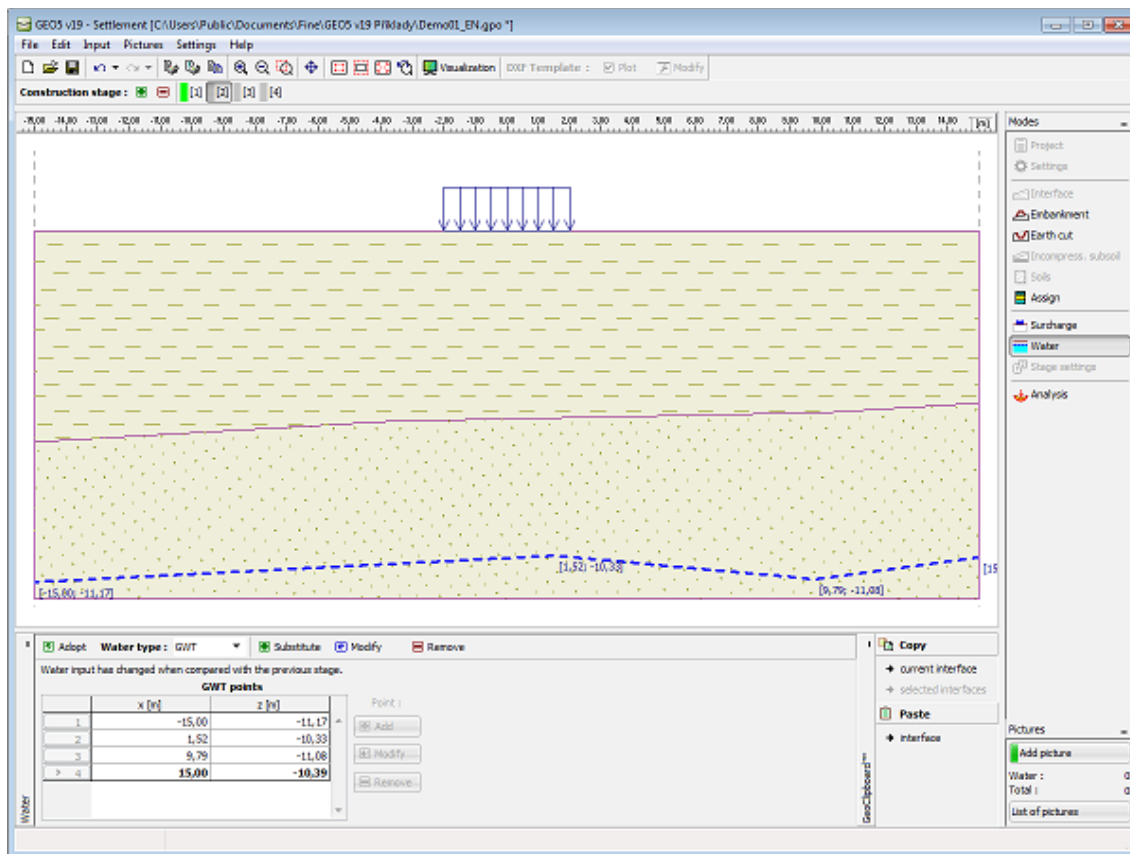
The "**Water**" frame serves to set the type of ground water table.

Inputting the ground water table or isolines, respectively, is identical with the standard input of interfaces.

If the input data in individual stages are different, the program then allows for accepting the data from the previous stage of construction by pressing the "**Accept**" button.

Input interfaces of water can be copied within all 2D GEO5 programs using "GeoClipboard".





Frame "Water"

## Stage Settings

The frame "Stage settings" serves to input settings valid for a given construction stage.

The frame allows for specifying the position of control holes and thicknesses and locations of layers where the stress values are calculated.

The program determines **stresses at individual control holes**. The terrain is always subdivided into twenty holes with even spacing. Additional holes are automatically generated in points specifying terrain, embankment, GWT, soil layer interfaces and end points of surcharge. The control (calculation) holes can be plotted in the frame "Analysis".

Individual holes are **divided into layers** according to the input values. The first layer always coincides with the original ground. In addition, **all points** specifying interfaces, GWT and incompressible subsoil are included. The default setting of thicknesses of layers **ensures reasonable speed and accuracy** of the analysis.

The layers are introduced up to depth of 250 m. In actual analyses, however, the depth of influence zone is restricted either by the **input incompressible subsoil** or by the **reduction of magnitude of stress change** or by the **structural strength**, respectively (depending on the setting in the "Settlement" tab).

The number and location of calculation holes can be adjusted when selecting the option "**User-defined**". In such a case it is possible to select both the location of holes and thicknesses and location of layers. The holes are then created according to the input - in addition, the program automatically includes all important points. When selecting the **option exact distribution**, the holes are included into all terrain points, soil layer interfaces, embankments, GWP and into

end points of surcharge. When selecting the **option minimal distribution**, the holes are not included into points of interfaces of soil and embankment layers.

For standard analyses we **recommend keeping** the default setting of the analysis.

No.	From depth [m]	Refinement [m]
1	0,00	0,10
2	2,00	0,30
3	5,00	0,50
4	10,00	2,00
5	30,00	10,00

Frame "Settings"

## Analysis

The "**Analysis**" frame displays the analysis results.

It is always required to model the structure using **construction stages**. **First construction stage** represents the **original state**, so the settlement is null. New surcharges or embankments are added in **further construction stages**, where the **terrain settlement** is calculated. Information about the analysis process, maximum settlement, depth of deformation zone are listed in the window in the bottom part of the frame.

In **consolidation analysis** (set in the frame "Settings") this section of the frame allows to enter **consolidation parameters**.

**The settlement is calculated** using the **analysis theory**, which is input in the tab "Settlement". The **depth of deformation zone** is defined either by input **incompressible subsoil**, **method of restriction of the primary stress magnitude**, or by **theory of structural strength**.

The results, as the main output, are displayed on the screen. To view the results, use the **horizontal bar** in the upper section of the screen, which allows for adjusting the way the resulting values are plotted. **The bar contains the following control items:**

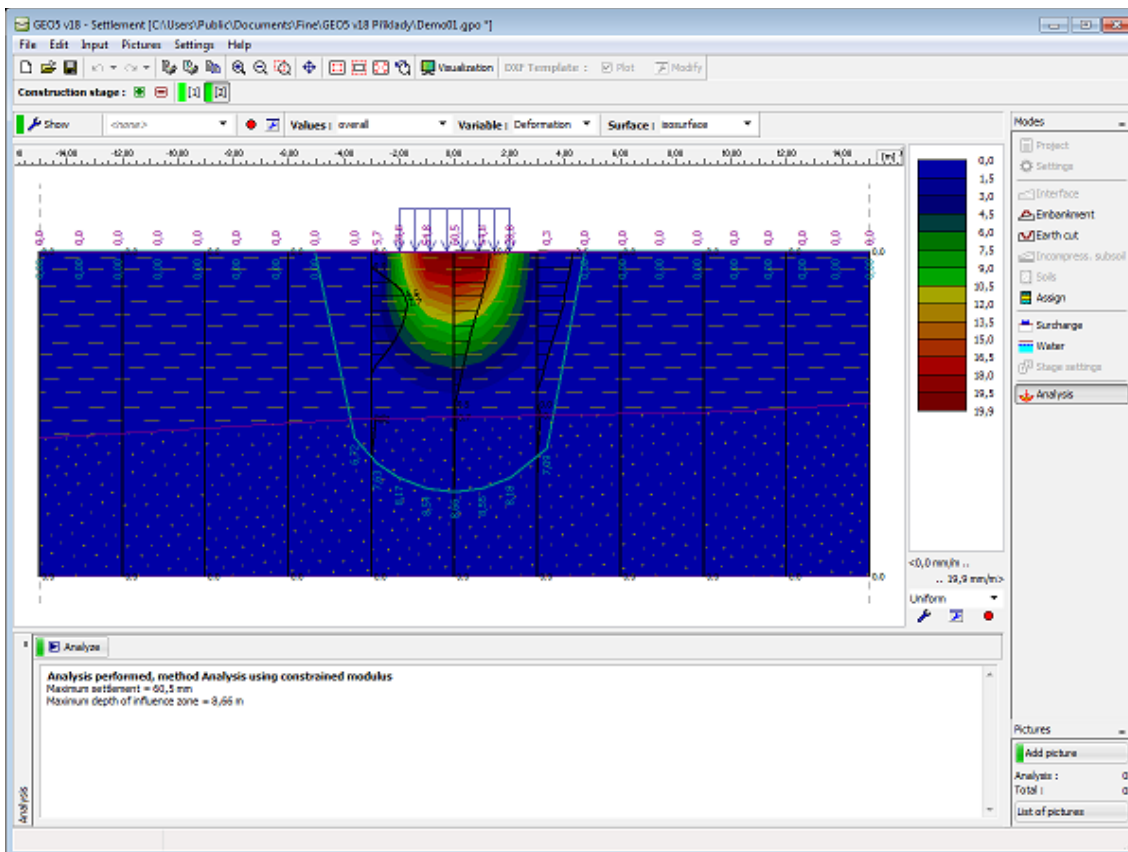
- **the button to display** the "Settlement - results visualization settings" dialog window. This dialog window allows for specifying all drawing parameters: parameters to display depression line and influence zone, to set color range, to draw tilted sections, isosurfaces and isolines, etc.
- **option to store individual views**
- **selecting values for visualization** - either **total** values, or their change during the **last calculation stage** or their change **in comparison with previous stages** can be plotted. The setting is available only in problems where it makes sense. It is therefore possible to display the change of stress, settlement or deformation in comparison with previous stages - however, always the current depth of influence zone is plotted.
- **selecting variables**
  - SigmaZ,tot - overall vertical total stress [kPa, ksf]
  - SigmaZ,eff - overall vertical effective stress [kPa, ksf]

- Pore pressure - stress due to water [*kPa, ksf*]
- Settlement - settlement of a point [*mm, feet*]
- Deformation - relative settlement of a layer [-]\*1000

- **plotting option** (do not plot, isosurfaces, isolines)

The **color range** is visible on the right part of the desktop. The buttons for **setting the color range** are located below.

Visualization of results can be adjusted in the frame "Visualization settings".



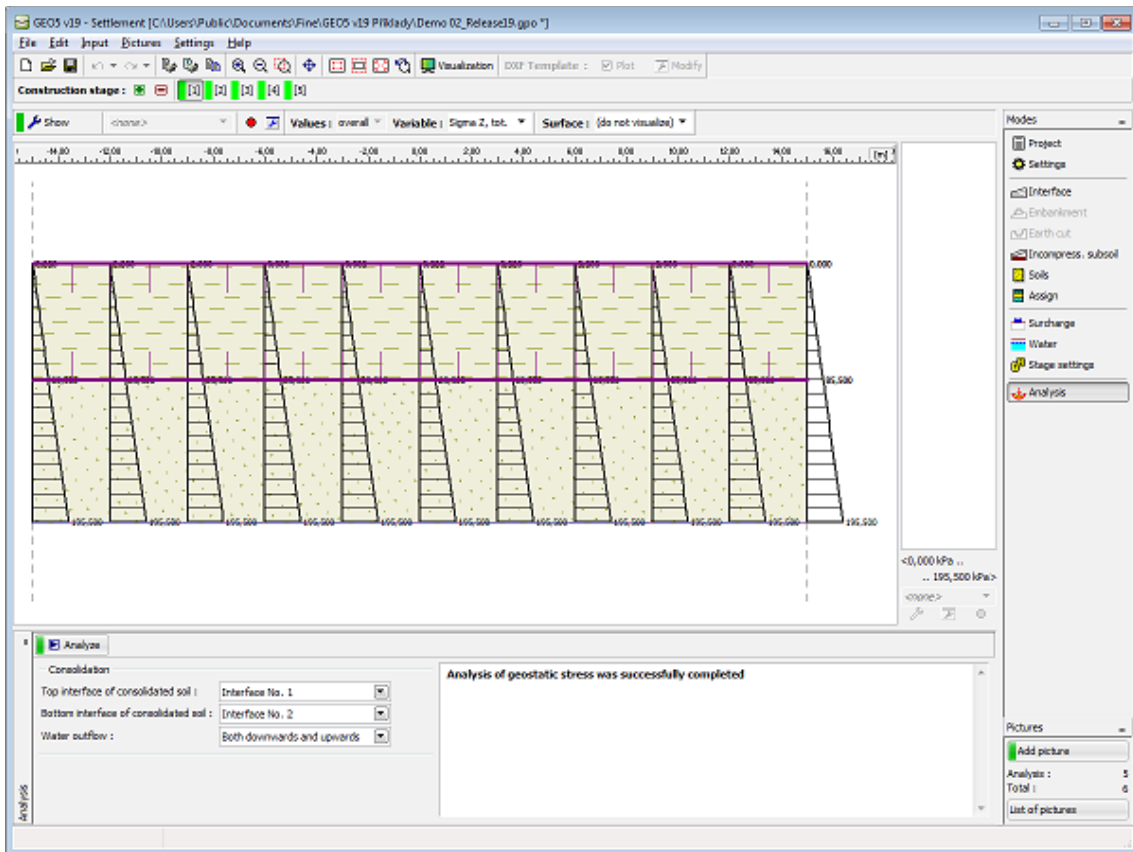
Frame "Analysis"

## Consolidation Parameters

In **consolidation analysis** (set in the frame "Settings") the bottom window in the frame "Analysis" allows to enter consolidation parameters.

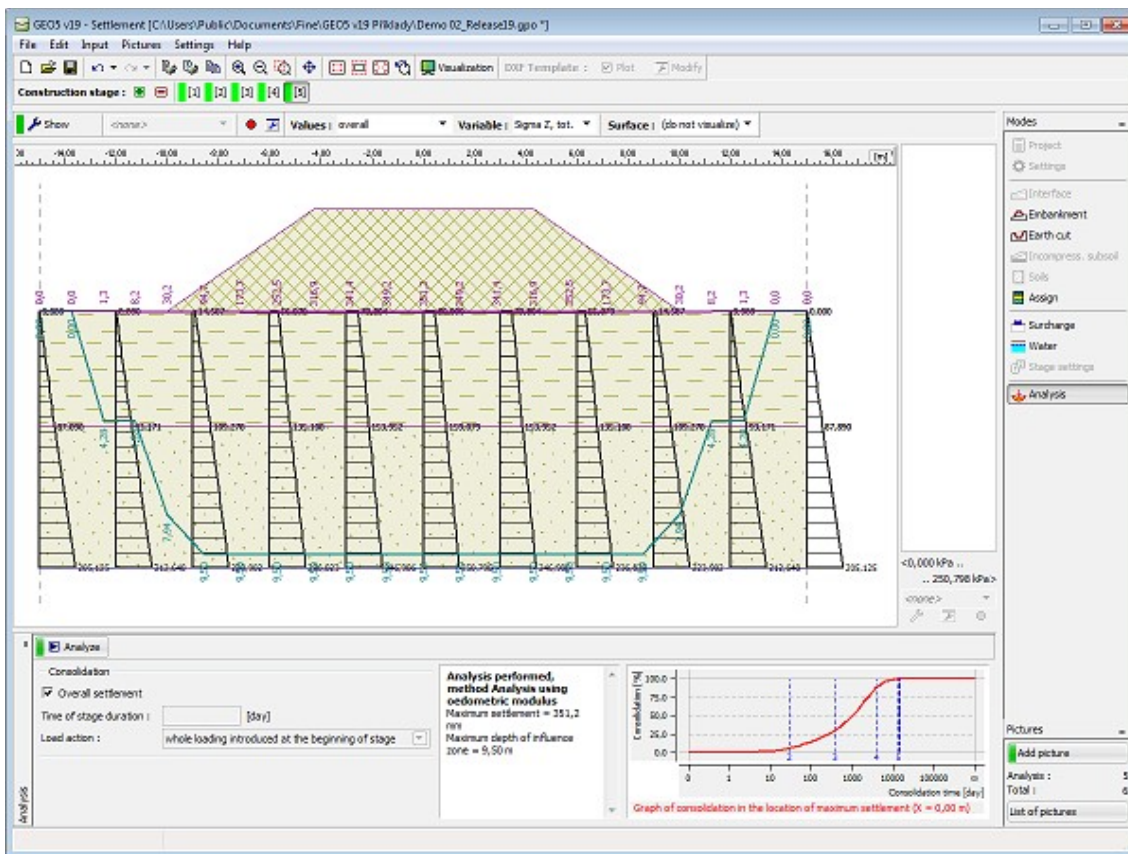
The first construction stage of calculation represents only geostatic stress at initial time of construction. Top and bottom interface of consolidated soil layer and direction of water outflow from this layer (upwards, downwards and both downwards and upwards) is entered in the first construction stage.

Program allows to draw a time course of settlement (graph on the right of the desktop) according to relevant **theory of settlement**. The vertical axis shows **degree of consolidation  $U$  [%]**, on the horizontal axis is shown time of settlement  $t$  [*days*].



Frame "Analysis" - primary geostatic stress (first construction stage)

In other construction stages **time of stage duration** and load action are entered. Program allows for choosing from two options of load acting: whole load introduced at the beginning of stage or load linearly increases during stage duration. The calculation is then launched from the first construction stage to the construction stage where "**Overall settlement**" is checked (may be checked in whatever stage except first one).



Frame "Analysis" - consolidation (other construction stage)

Example: determine settlement from surcharge after 5 days, 1 month, 1 year and 5 years?  
 Enter construction stages according to following scheme:

1. stage            Only geostatic stress
2. stage            Surcharge, time: 5 days
3. stage            No changes, time: 30 days
4. stage            No changes, time: 365 days
5. stage            No changes, time: 1825 days
6. stage            Check option "**Overall settlement**" and run the calculation

## Program Ground Loss

This program is used for analysis and determination of the shape of subsidence trough above excavations and to evaluate the damage to buildings situated in the affected area.

**The help in the program "Ground Loss" includes the following topics:**

- Input of data into individual frames:

Project      Settings      Buildings      Profile      Soils      Assign      Geometry

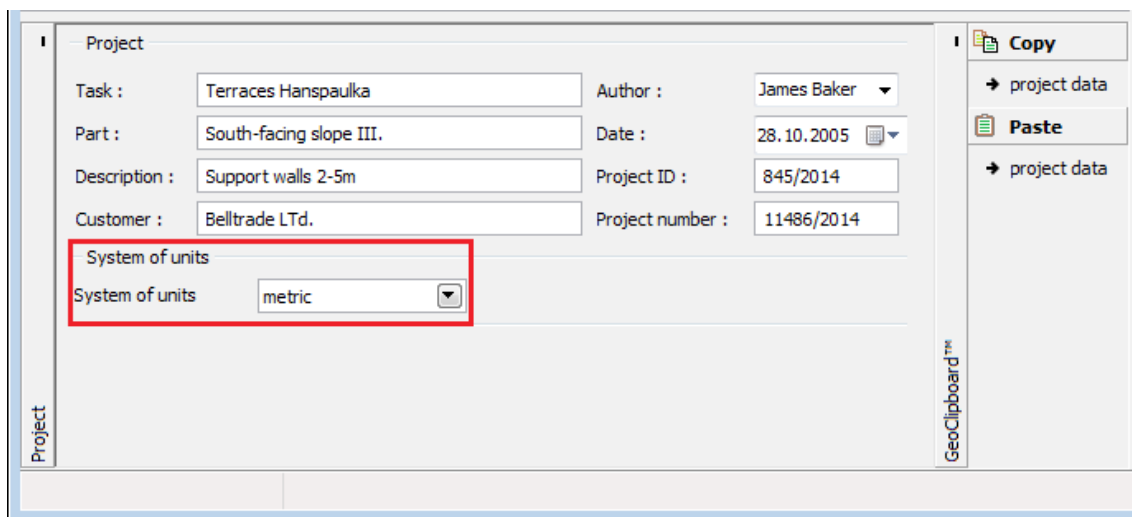
Measurements      Analysis      Damage  
 Settings

- Standards and analysis methods
- Theory for analysis in the program "**Ground Loss**":  
     Stress in Soil Body                      Ground Loss
- Outputs
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using "**GeoClipboard**".

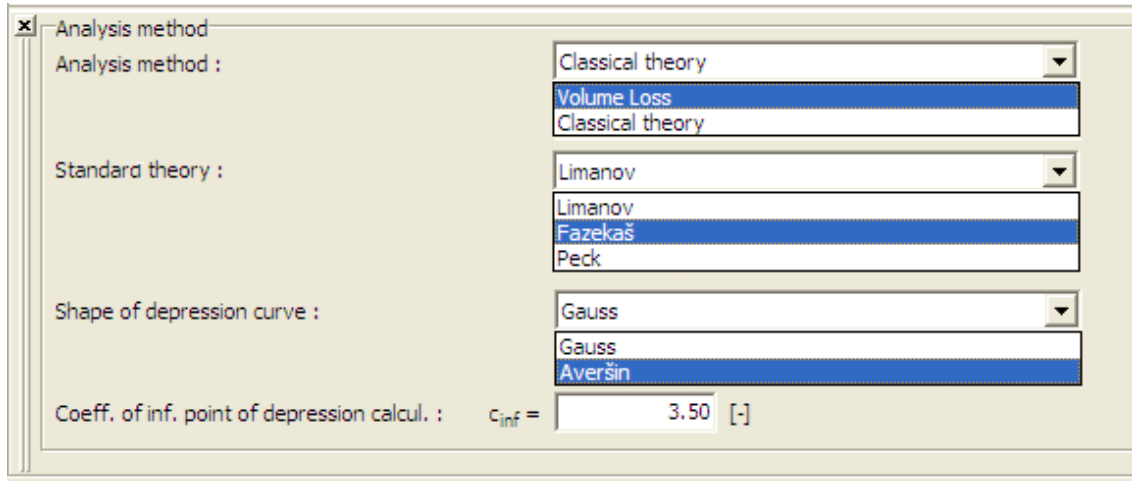


Frame "Project"

## Settings

The **frame "Settings"** allows to specify standards or methods that are used to perform the analysis.

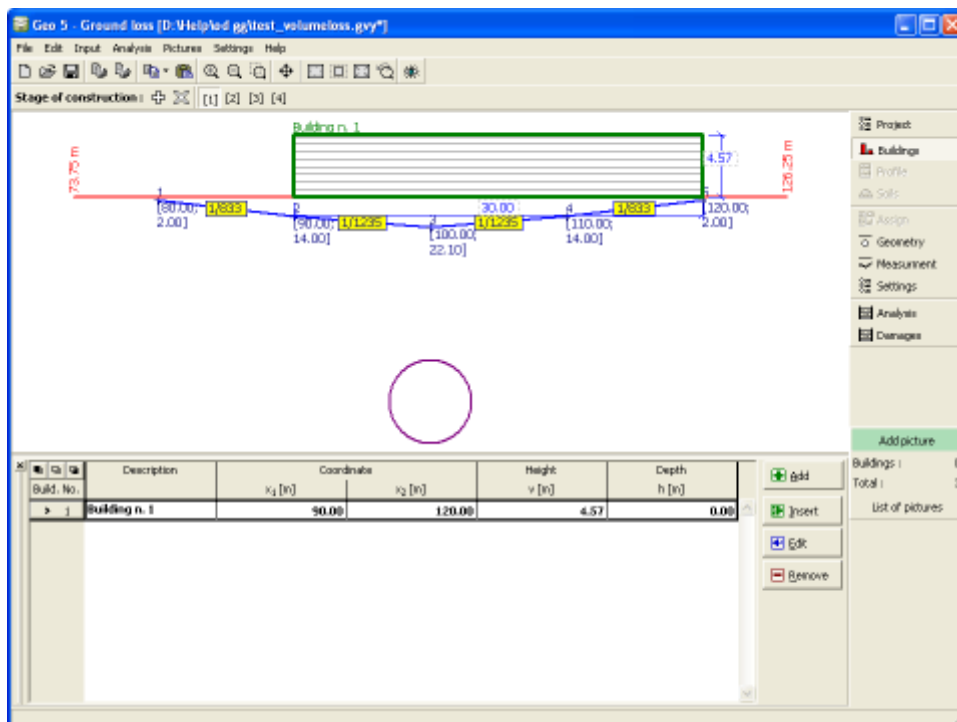
The frame "**Settings**" allows to select the method for determining subsidence trough (Volume loss, classic theories) and its shape (Gauss, Aversin). It also allows to input the coefficient of calculation of inflection point, (for classic theories only), which influences the shape of subsidence trough.



Frame "Analysis methods"

## Buildings

The frame "**Buildings**" allows to input objects above excavation. An arbitrary number of buildings can be specified both on a ground surface and at a given depth.



Frame "Buildings"



## Profile

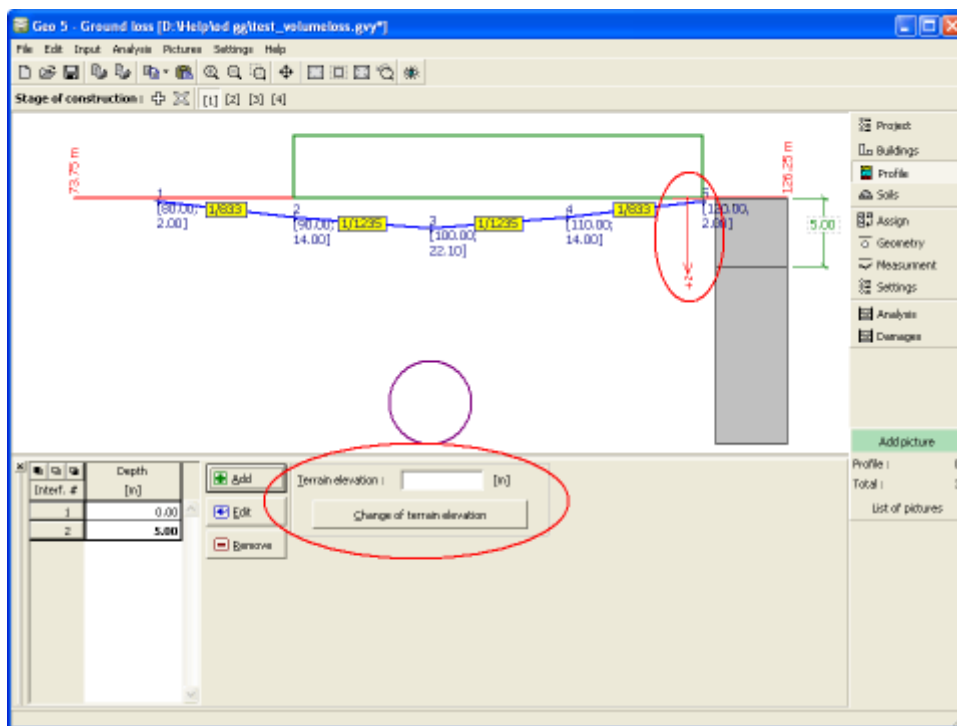
The "**Profile**" frame contains a table with a list of input interfaces. After interfaces specification, it is possible to edit thicknesses of individual layers using the active dimensions.

Adding (editing) layer is performed in the "**New interface**" dialog window. The z-coordinate measured from the top point of a structure is specified (z-axis).

The program allows to raise or lower the top point of a structure in the "**Change of terrain elevation**" dialog window, so that the whole interface can be translated while keeping the thicknesses of individual layers. This function is important when copying the profile from the "**Terrain**" program.

Data input in the frame is allowed if the **classic theory of analysis** is selected in the frame "**Settings**".

The program allows to import a profile in the gINT format.



Frame "Profile"

## Soils

The frame "**Soils**" contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "**Add new soils**" dialog window.

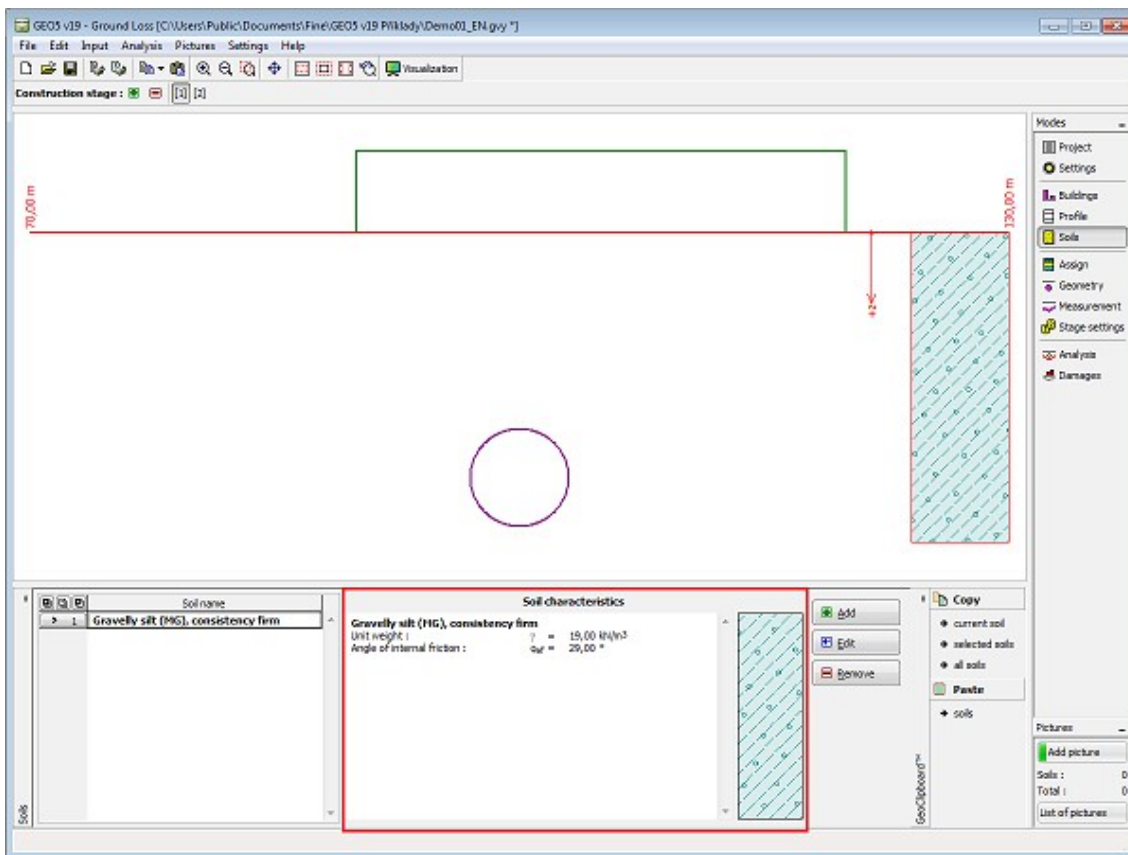
Inputting data into the frame is allowed if the **classic theory of analysis** is selected in the frame "**Settings**". The particular values are obtained from geotechnical survey or from laboratory experiments. If these data are not available, it is possible to exploit built-in **database of soils**, which contains values of selected characteristics of soils.

Possible values of the angle of internal friction and cohesion are available in chapter "**Rocks**



parameters".

The program allows to import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

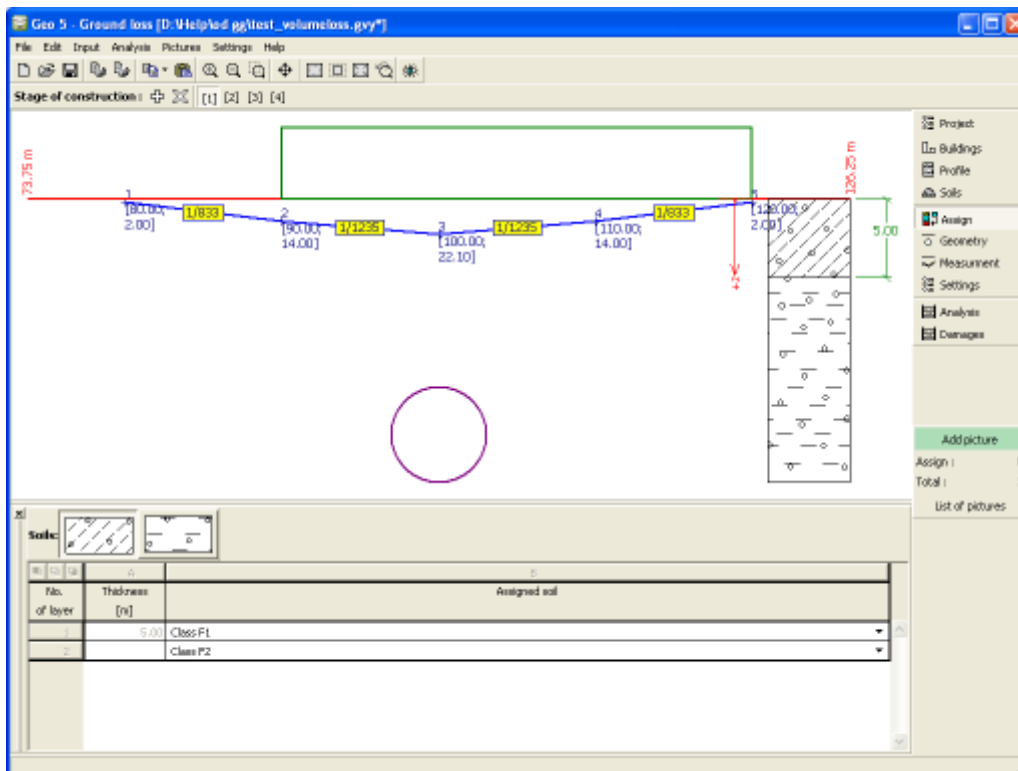
## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

Inputting data in the frame is allowed if the **classic theory of analysis** is selected in the frame "Settings".

The program allows to import soil assignment in the gINT format.



Frame "Assign"

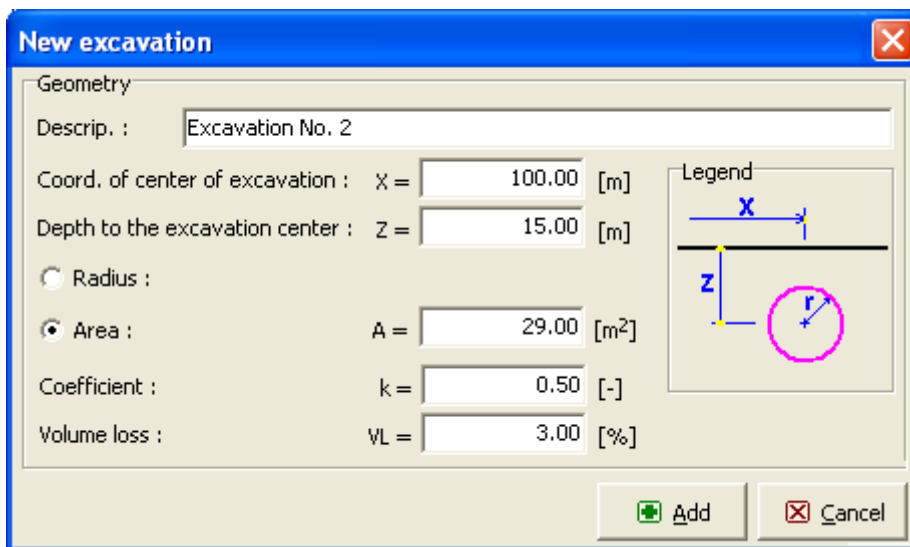
## Geometry

The frame "Geometry" contains a table with a list of input excavations. The "New excavation (Edit excavation)" dialog window allows to add (edit) excavations. The input excavations can also be modified on the desktop using the active objects.

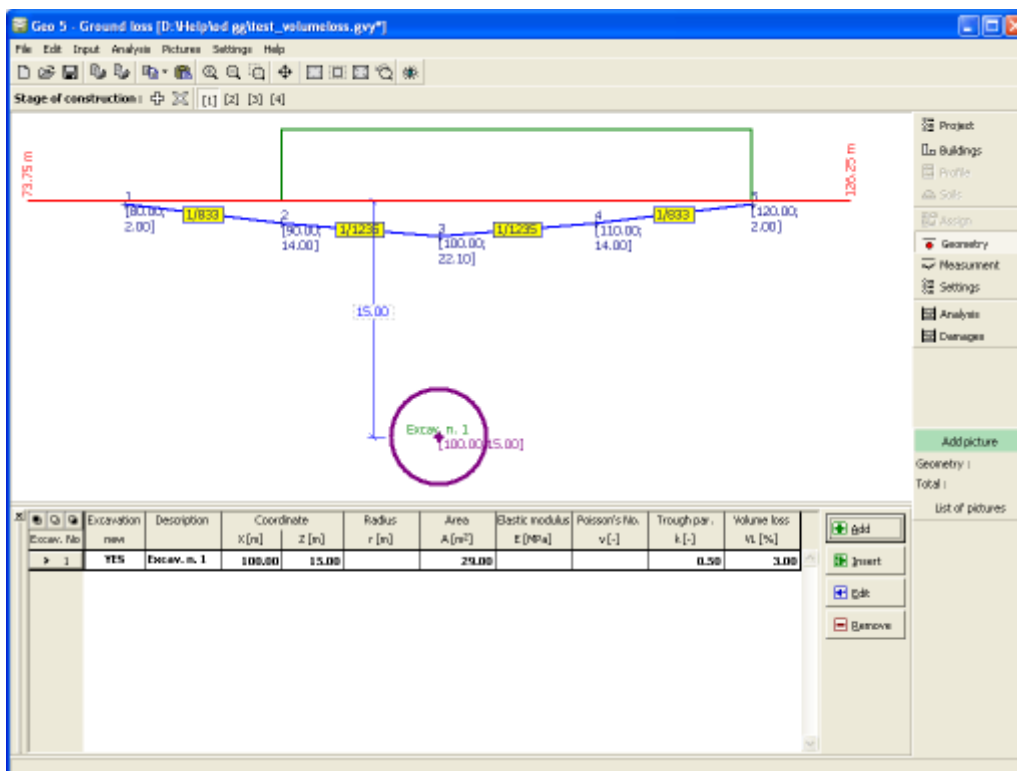
Parameters of excavation differ depending on the analysis method selected in the frame "Settings". Each excavation can be specified either by the radius or the area of excavation. Providing a sequential excavation is being input it is useful to specify the excavation area and place a fictitious center of excavation to a center of gravity of this area.

Additional input parameters are explained in more detail when describing individual analysis methods (Volume loss, classic theories).

The program allows to export the geometry of a structure in the \*.DXF format.



Dialog window "New excavation"



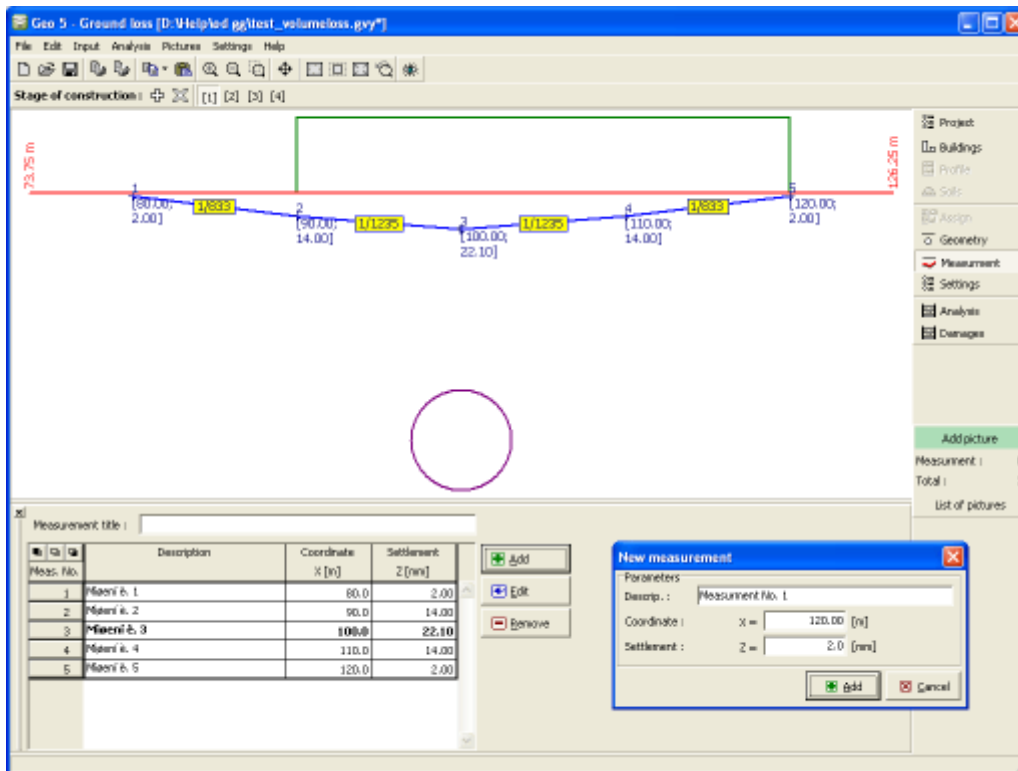
Frame "Geometry"

## Measurement

The frame "Measurement" contains a table with a list input measurements. The "New measurement (Edit measurement)" dialog window allows to add (edit) measurements. The input measurements can also be modified on the desktop using the active objects.

Input measurements **do not influence the actual analysis** - their introduction into the program has resulted purely from designers needs. After excavating the first part of a

sequential tunnel it is useful to input the values measured in the construction site into the program and subsequently to add the excavation input parameters such that the **calculated and measured values are the same**. Practical experience shows that the values of input parameters acquired from this procedure (e.g. coefficient of volume loss) are **valid also for subsequent stages**.



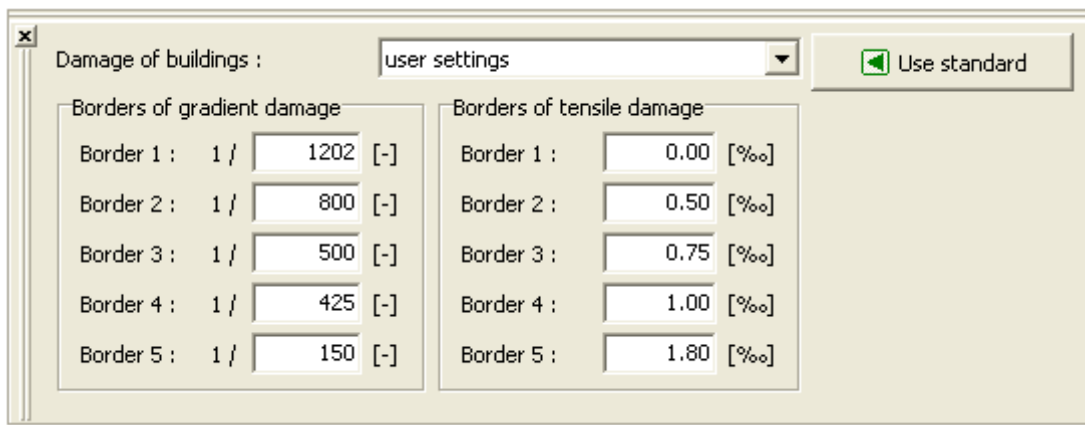
Frame "Measurement"

## Stage settings

The frame "Stage settings" allows to input settings valid for a given construction stage.

The frame allows to introduce bounds on the tensile and gradient damage. These values allow to verify the building damage in the frame "Damages". The program offers a default pre-setting (default setting for **masonry buildings**) and a user-defined setting - here, it is possible to define arbitrary criteria recommended by standards or gained from practical experience for arbitrary types of buildings.

The boundary values must be defined either in a descending or ascending order, respectively. Should you need to define fewer regions than specified in the program, it is possible to characterize certain boundaries by the same value.

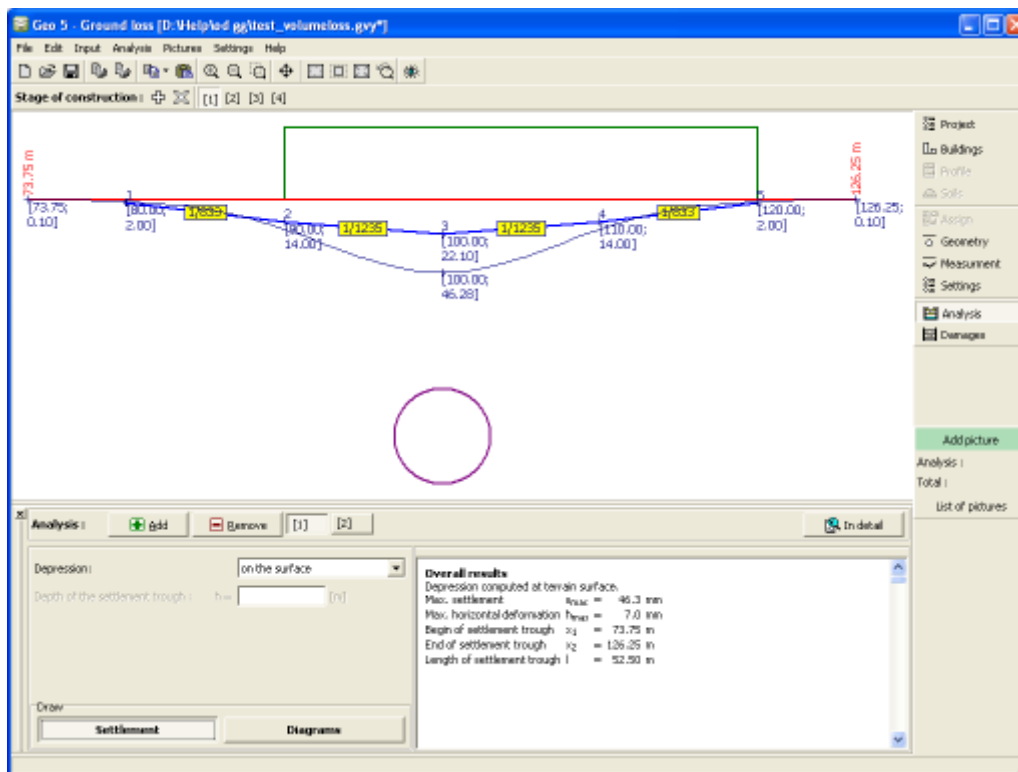


Frame "Settings"

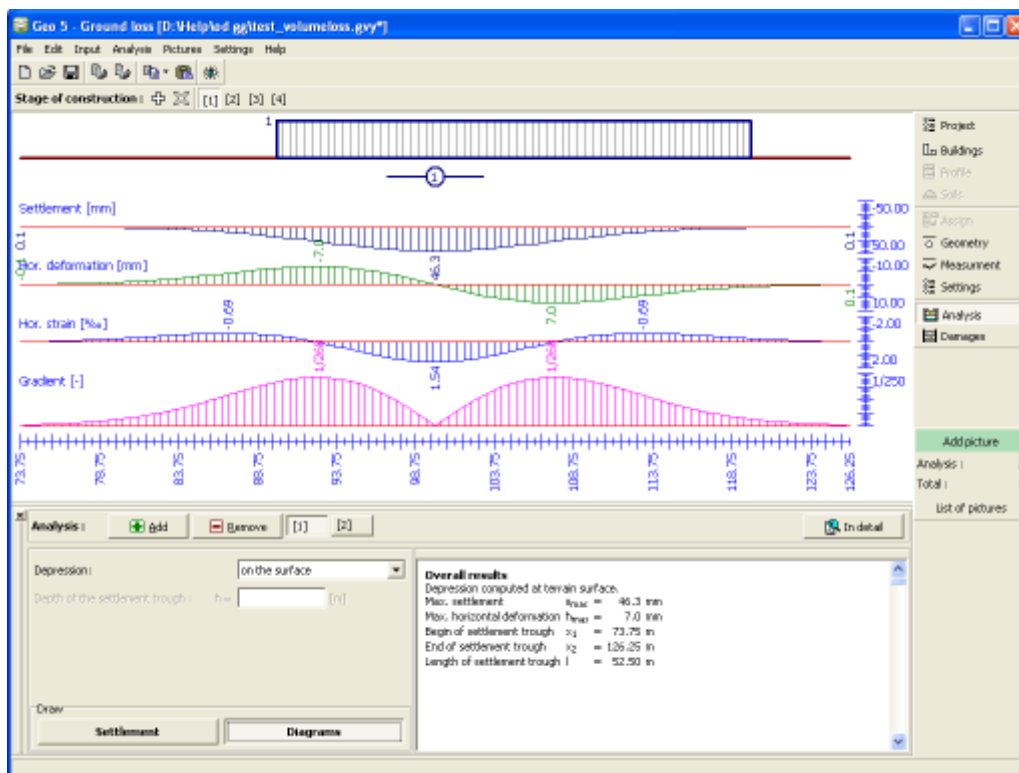
## Analysis

The frame "Analysis" provides the results from the analysis of subsidence trough. More than one analysis at different depths below the terrain surface can be performed for a single task. The computed values are displayed on a desktop and are continuously updated whenever a certain change in the input data or setting in the frame is introduced.

Visualization of results can be adjusted in the frame "Visualization settings". For a quick switch between different styles of graphical presentation of results (**subsidence trough, distribution of values**), the user may use the buttons in section "Visualization".



Frame "Analysis" - "Settlement"



Frame "Analysis" - "Diagrams"

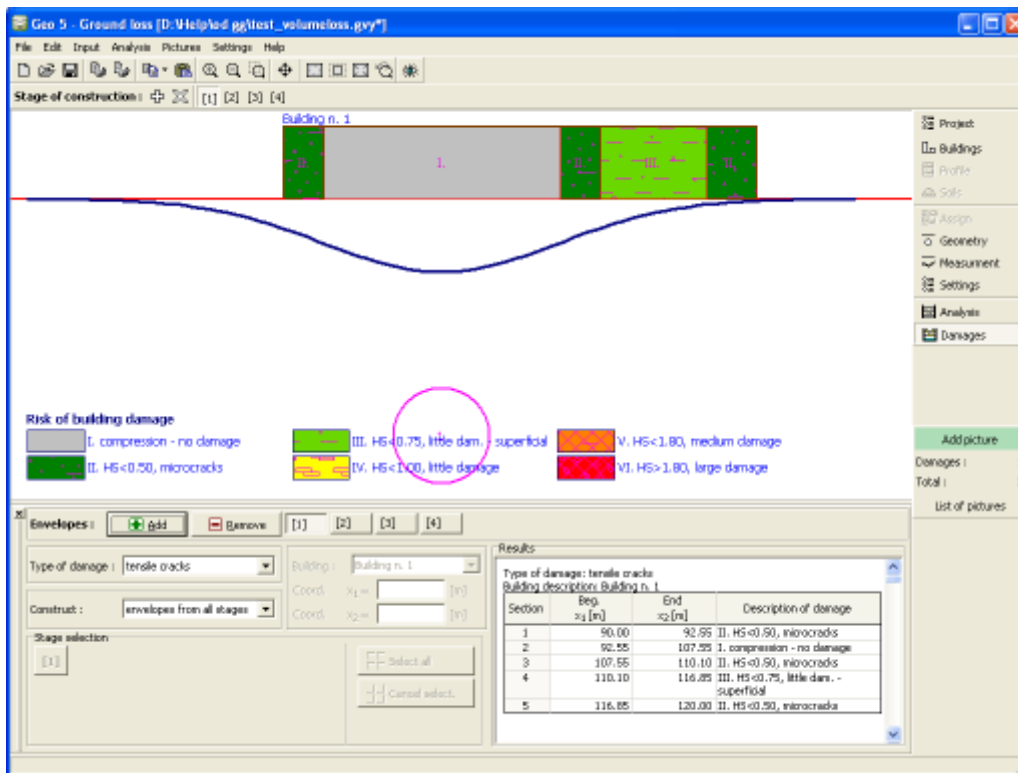
## Damage

The frame **"Damages"** provides the results of failure analysis of buildings. The program offers four types of verification:

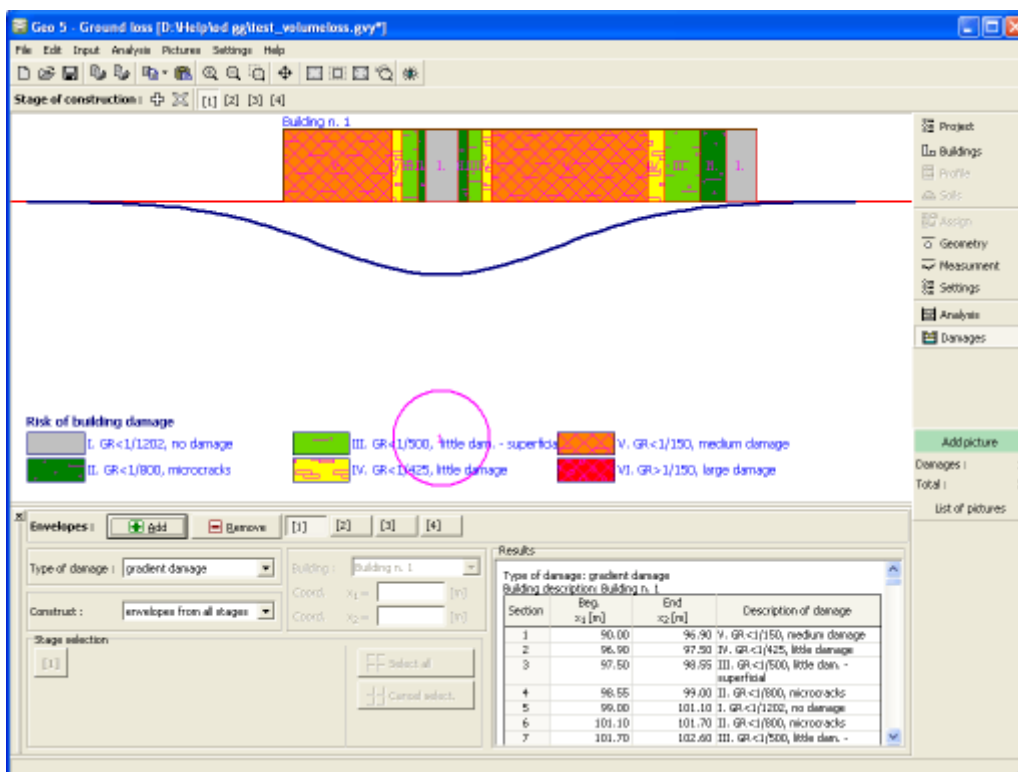
- Verification of **tensile cracks**
- Verification of **gradient damage**
- Verification of **relative deflection of buildings** (hogging, sagging)
- Verification of the **input section of a building**

The program allows to perform an analysis for the current and all previous stages (**envelope from all stages**) or it is possible to input individual stages and evaluate their influence. Such a procedure makes it possible to find, e.g. an optimal process of excavation of sequential tunnels.

**Several analyses** can be carried out for a single task. Visualization of results can be adjusted in the frame **"Visualization settings"**.



Frame "Damages" - visualization of tensile cracks



Frame "Damages" - visualization of gradient damage

## Program Terrain

This program is used to create digital terrain models (DEM, DTM) from inputted points and holes. It calculates volumes of excavation and also serves as task manager for other GEO5 programs.

**The help in the program "Terrain" includes the following topics:**

- Input of data into individual frames:

Project	Basic Data	Soils	Assign	Points	Edges	Water
Bore Holes	Earth Grading	Generate	Point Construction S	Line Construction S	Launching	

- Standards and analysis methods
- Outputs
- General information about the work in the **User Environment** of GEO5 programs
- **Common input** for all programs

## Project

The **frame "Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

In this input mode, the assumed setting can be modified only in the **first construction stage**. The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.



Frame "Project"

## Basic Data

The **frame "Basic data"** serves to input basic parameters of the task.

The frame contains a **table** with a list of specified layers. Layers can be added, inserted or removed using the buttons on the right from the table. The first layer can be neither removed, nor can another layer be inserted in front of it.

The frame section **"Basic setting"** serves to define world dimensions of the task. When increasing or decreasing these dimensions the program prompts possible consequences of this action.

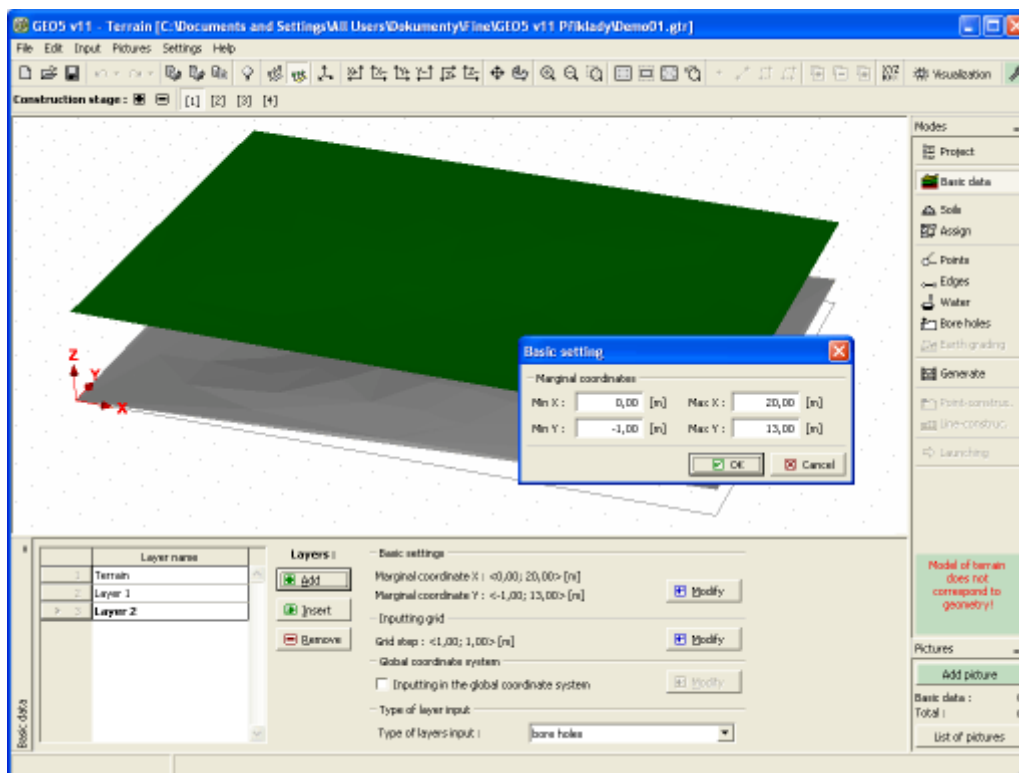
The section **"Inputting grid"** serves to define an origin and step of the grid in the *X* and *Y* directions. The dialog window, which allows for setting these parameters, is described in the help section **"User-defined environment"**, chapter **"Options - Input"**.

Crossing the item **"Input in the global coordinate system"** opens the way for introducing the data in the **global coordinate system** (JTSK, Gauss-Krüeger).

In the **"Type of layers input"** combo list it is possible to determine the way of inputting layers. Layers can be introduced with the help of layers **thicknesses** or their **points**.

In this input regime the assumed setting can be modified only in the **first construction stage**.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the **"Visualization settings"** dialog window and with the help of buttons on tool bar **"Visualization"**, "Scale and shift" and "Plot setting".



Frame "Basic data"

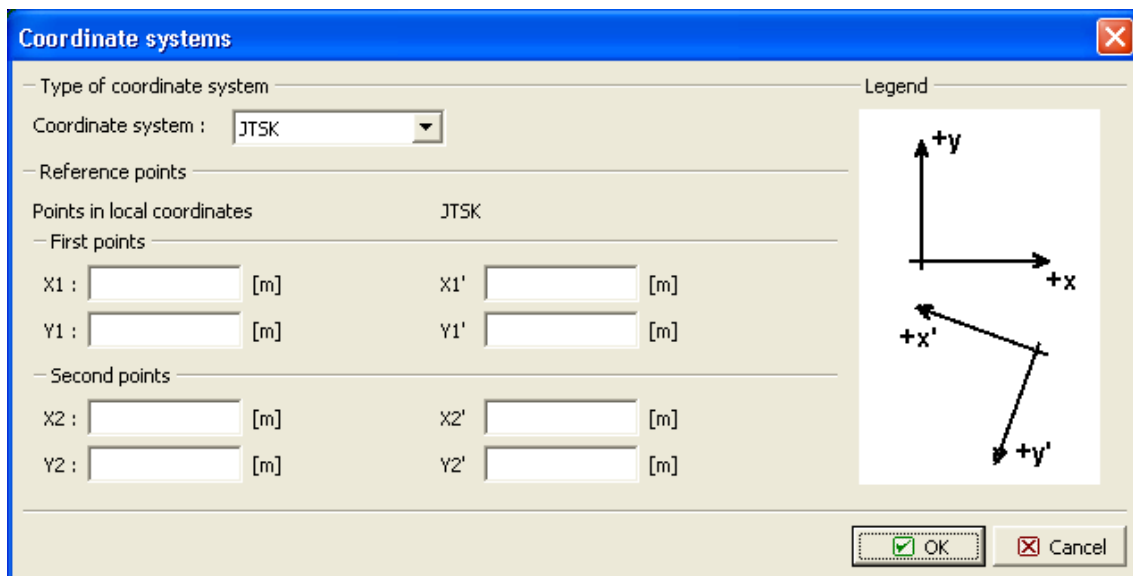
## Global Coordinate System

The "**Coordinate systems**" dialog window allows for defining the type of the global coordinate system.

Essential advantage is the possibility of specifying the coordinates of points and bore holes both in the local and global coordinate systems and switching between the two systems.

Orientation of the global coordinate system with respect to the local one is defined using two points, where one point is always introduced in the local coordinate system and its image in the global coordinate system.

Direction and sign convention is displayed for each type of the global coordinate system in the legend chart.



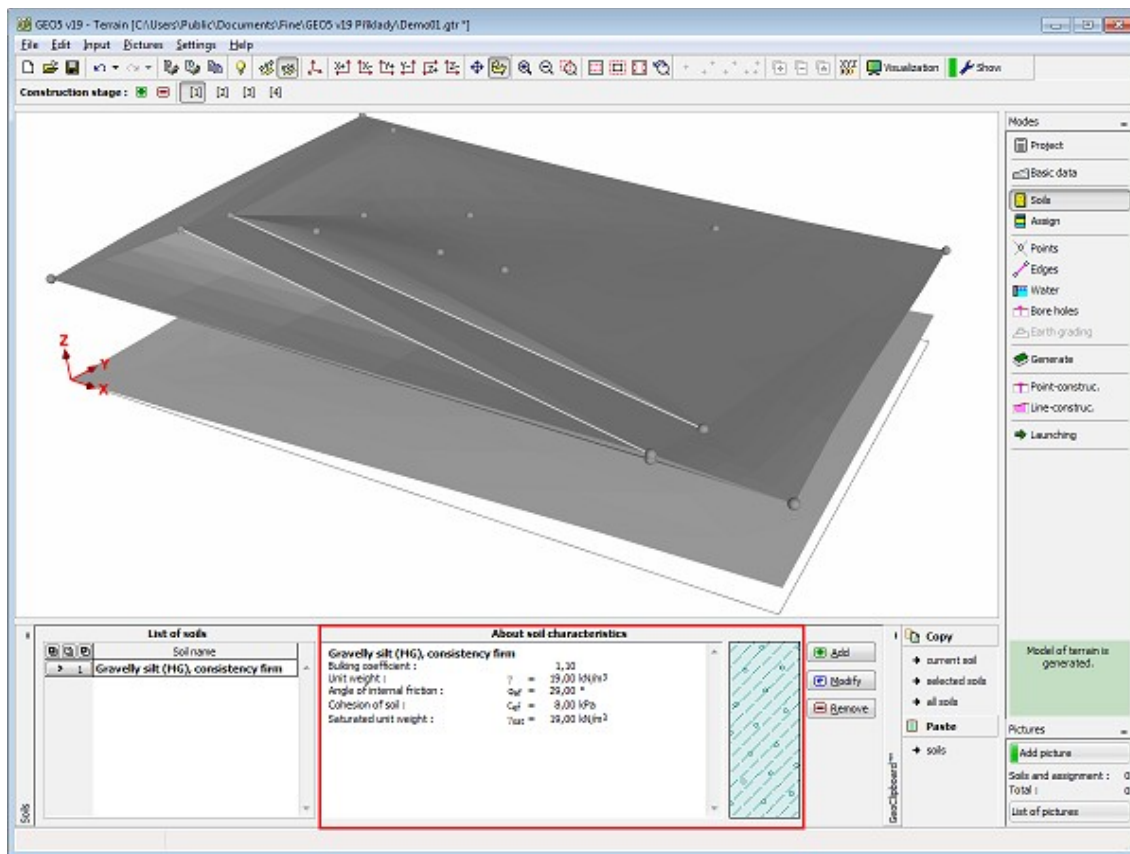
*Dialog window "Coordinate system"*

## Soils

The "**Soils**" frame contains a table with a list of input soils. The table also provides information about currently selected soil displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

Adding (editing) a soil is performed in the "Add new soils" dialog window. The program Terrain calls only for specification of the coefficient of bulking to compute yardage of excavation pits or embankments. The remaining data are used only for possible export into our other programs and have no effect on actual calculations performed in program "**Terrain**".

The program makes it possible import soils in the gINT format. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

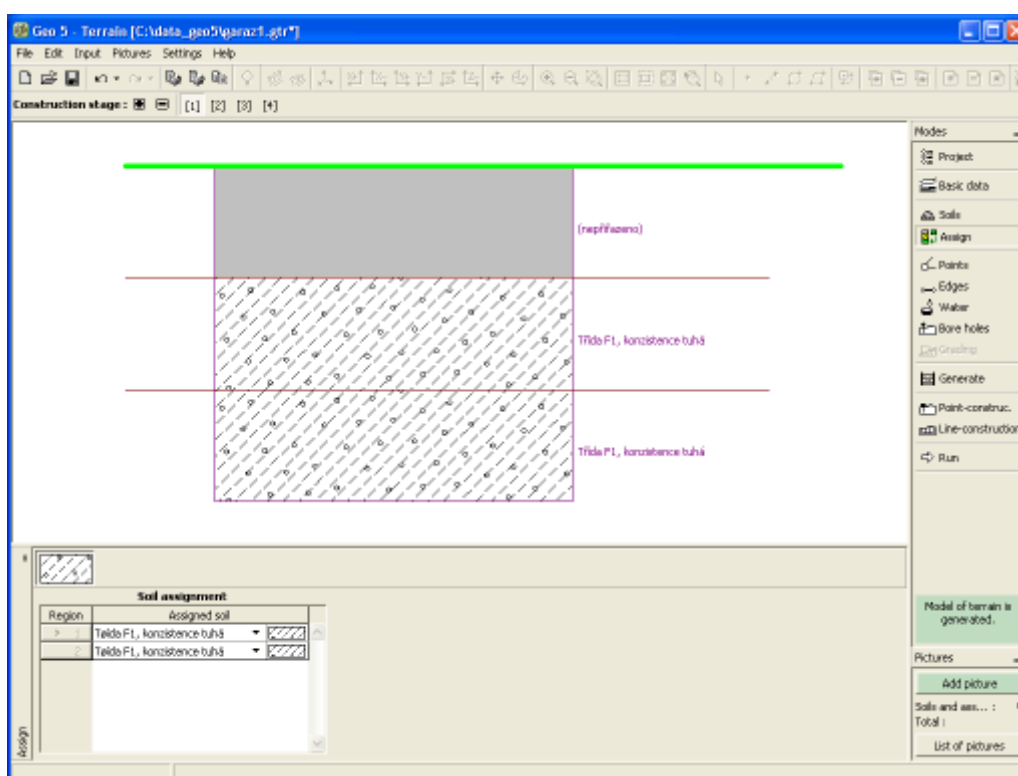
## Assign

The "Assign" frame contains a list of layers of profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign soil into a layer is described in detail [herein](#).

In subsequent stages of construction the program automatically adds a new layer, to which a soil adjacent to terrain is automatically assigned. In many cases (excavation pits) this layer may have no volume - its introduction is necessary providing the new terrain is found above the terrain of the previous stage. The soil is always assigned, since it is not possible to a prior estimate, whether some part of the terrain in the new stage will be located above the original one.

The program makes it possible to [import soil assignment in the gINT format](#).



Frame "Assign"

## Points

The **frame "Points"** serves to define the coordinates of terrain points. There are two options available to define coordinates of individual points:

**With the help of table:** points are defined in the **table**. Pressing the **"Add"** button opens the **"New point"** dialog window; coordinates of points are then specified and by pressing the **"Add"** button added to the table. An arbitrary number of points can be defined in this way. The **"Cancel"** button is used at last to close the window. These points can be further modified (in the dialog window) using the **"Edit"** button or removed with the help of **"Remove"** button (more points can be marked in the table to remove them all at once - before removing, the selected points are displayed on the desktop in red). Each change is immediately reflected on the desktop.

**With the help of the mouse:** this inputting mode is turned on by pressing a respective button on the horizontal bar. The following options are available:

- Add**
- to input a point, click the left mouse button on the desktop (the mouse cursor changes - see picture) - the program then opens the **"New point"** dialog window, which allows for modifying the point coordinates, or to input its Z-coordinate - after pressing the **"OK"** button the program adds this point into the table. Providing the point cannot be added (e.g. duplicity of coordinates) the program prompts a warning message
  - grid functions can also be used when specifying a point
- Edit**
- clicking an already existing point (see **active object**) using the left mouse button opens the **"Edit point"** dialog window, which allows for editing the point coordinates - in the dialog window the following buttons (**"OK+↑"** and **"OK+↓"**) can be used

- Remove** • clicking the point using the left mouse button opens a dialog window, which requests to confirm deletion of the selected point
- Select** • activates the regime of graphical selection of points (type of selection is set in the tool bar "Selection")

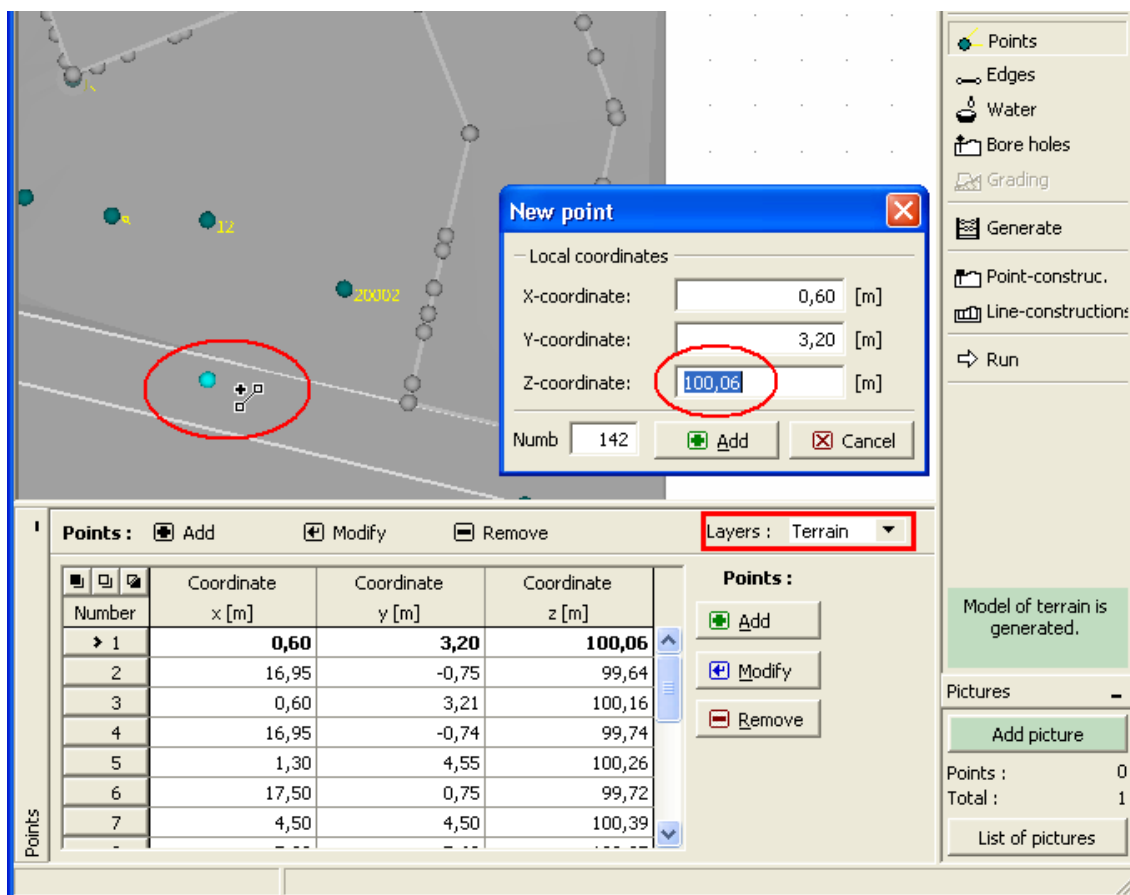
The selected points can also be imported from files in formats **TXT**, **Atlas DMT**, **DXF**, **LandXML** and **gINT**.

The program allows for importing further points and edges into the processed task (e.g. in subsequent stages).

When defining points the program in some cases automatically calculates their Z-coordinates. Only one point can be assigned to a single coordinate X, Y.

Providing the option of inputting layers with the help of "Points of layers" in the "Basic data" frame is selected, the "Points" bar contains the "Layers" combo list. This list serves to choose a layer into which the points will be input.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the "Visualization settings" dialog window and with the help of buttons on tool bar "Visualization", "Scale and shift" and "Plot setting".



Frame "Points"

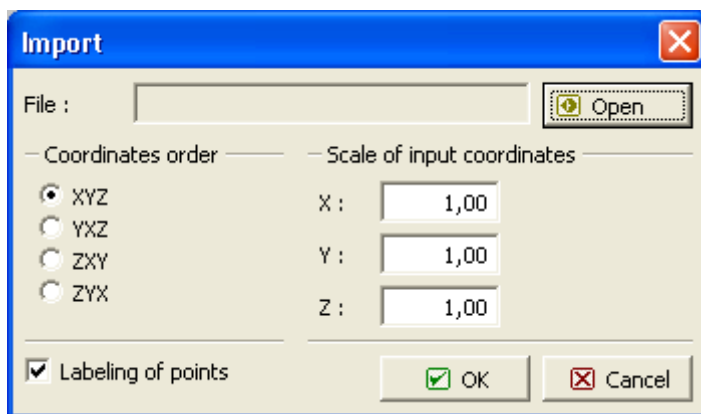
## Import of Points

The program allows for data import in formats **DXF**, **LandXML**, **ATLAS DMT** and **ASCII**. When

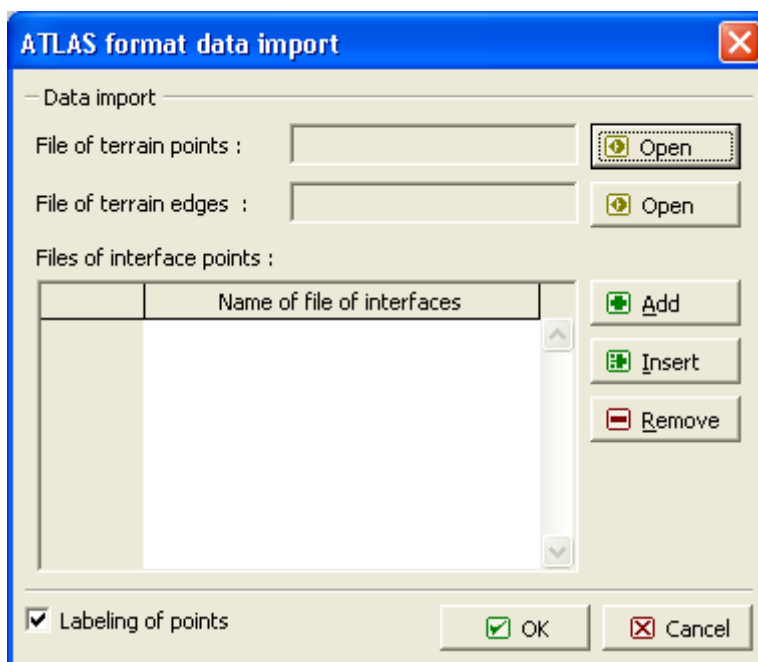
importing, **all old data are deleted** and replaced by the new ones. The world dimensions are automatically determined according to minimal and maximal values of coordinates  $x$  and  $y$  - it is therefore desirable to subsequently adjust the world dimensions in the frame "Basic data".

The program allows for importing **ASCII data from respective files**. Each point is written on a separate line of the file, coordinates are separated by space or tab character. If the file contains for each point first its name, it is necessary to check the item "Labeling of points". In the dialog window it is then necessary to specify the order of coordinates. If the data has an opposite sign convention, it possible to multiply the corresponding line by the value -1. The data import is performed after pressing the "OK" button.

The program also allows for importing terrain points in format gINT.



Dialog window "Import" - format "TXT"



Dialog window "Import" - format "Atlas"

## Automatic Calculation of Height

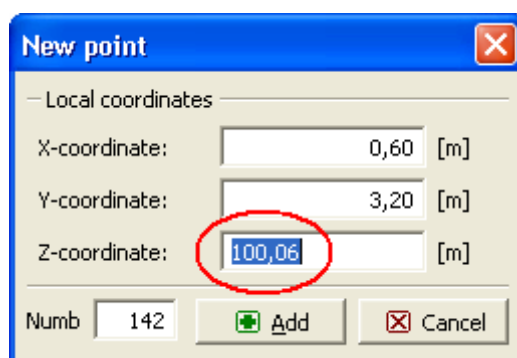
When defining points, bore holes and points of ground water table the program in some cases automatically calculates the point height ( $z$ -coordinate) and eventually the layer thickness. This

function is particularly valuable when editing terrain or layers.

The possibility of height calculation depends on the **status of generated terrain**:

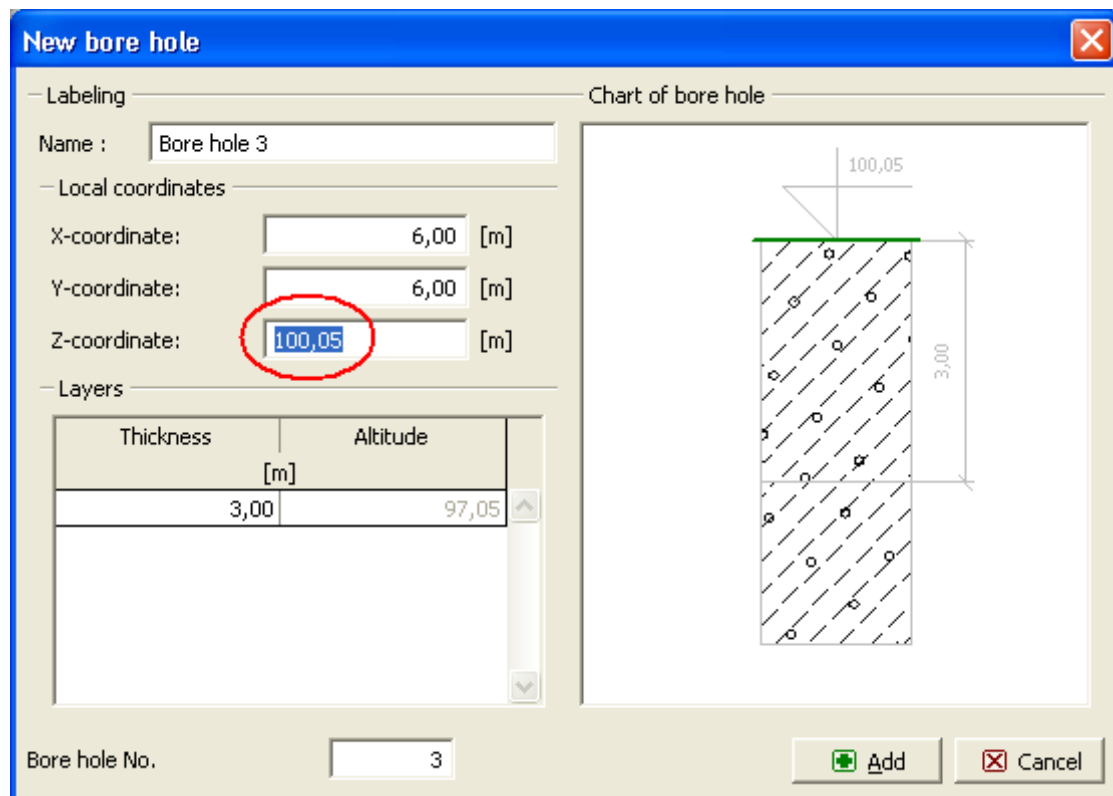
- If no terrain is generated, the height is not calculated and the respective field remains empty (blank).
- If terrain is generated for the current data (displayed on the desktop in a non-transparent mode and in regimes **Generation**, **Point constructions**, **Line constructions** and **Launching** also in a color mode), the required values are then automatically calculated from the model of terrain - for a point it is the  $z$ -coordinate, for a bore hole the program further determines the layer thickness and possibly also the depth of the ground water table - when a point or a bore hole is specified the status of generated terrain is changed and the drawing is switched to a transparent mode (the terrain is generated for the original input, not for the current input).
- When the terrain is generated, but it is not the current one, the  $z$ -coordinates and layers thicknesses are automatically calculated for the last generated terrain.

**Information regarding the terrain status** (not generated, generated, generated for the original data) are displayed on the vertical tool bar. The frame "**Generate**" allows for terrain generating or removing the generated model.



*Dialog window - add new point and calculate the Z-coordinate*





Dialog window - add new bore hole and calculate the Z-coordinate, thickness of GWT and layer thicknesses

## Edges

The frame "Edges" serves to input edges connecting the terrain points. Two options are available to define edges:

**With the help of table:** edges are defined in the table. Pressing the "Add" button opens the "New edge" dialog window; sequence numbers of the starting and end points are then specified and by pressing the "Add" button added to the table. An arbitrary number of edges can be defined in this way. The "Cancel" button is used at last to close the window. These edges can be further modified (in the dialog window) using the "Edit" button or removed with the help of "Remove" button (more edges can be marked in the table to remove them all at once - before removing, the selected edges are displayed on the desktop in red). Each change is immediately reflected on the desktop.

**With the help of the mouse:** this inputting mode is turned on by pressing a respective button on the horizontal bar. The following options are available:

### Add

- to input an edge, click the starting and end points using the left mouse button (the mouse cursor changes - see picture) - after clicking the end point the program adds the corresponding edge into the table and at the same time displays this edge on the desktop. Providing the edge cannot be added (duplicity reason, crossing, etc.) the program prompts a warning message

### Edit

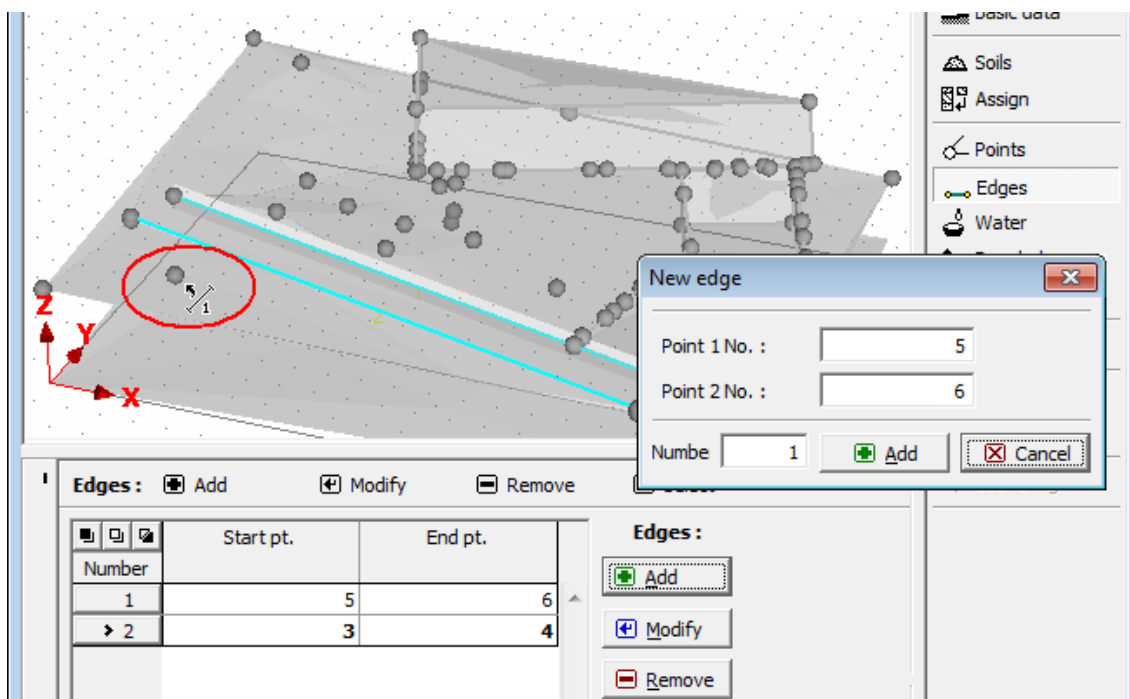
- clicking an already existing edge (see active objects) using the left mouse button opens the "Edit edge" dialog window, which allows for editing the sequence numbers of the starting and end points of the edge - in the dialog

window the following buttons ("OK+↑ " and "OK+↓") can be used

- Remove**
  - clicking the edge using the left mouse button opens a dialog window, which requests to confirm deletion of the selected edge
- Select**
  - activates the regime of graphical selection of edges (type of selection is set in the bar "Selections")

Edges can intersect neither other edges nor earth grading. Only one edge can be defined between two points.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the "Visualization settings" dialog window and with the help of buttons on tool bar "Visualization", "Scale and shift" and "Plot setting".



Frame "Edges"

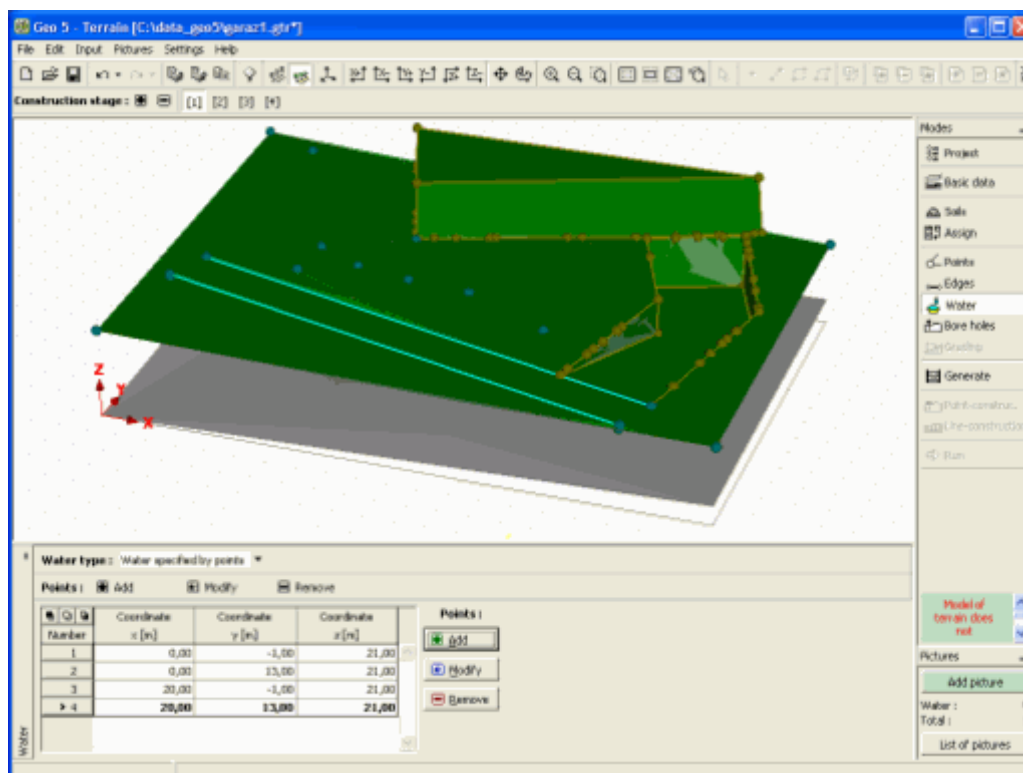
## Water

The frame "Water" serves to specify the ground water table (GWT). A combo list "Type of water" contains the following items:

- **No water** - no water specified
- **Water specified by points** - the GWT points are defined in the table in the same way as when defining terrain points. This approach is particularly suitable if having a horizontal water table - then it is sufficient to define only one point of a given coordinate Z and the program automatically generates a horizontal line representing the GWT.
- **Water specified in bore holes** - Ground water is defined within bore holes. A particular depth of GWT measured from terrain surface is specified. This approach is suitable when bore holes with measured depths of GWT are available.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the "Visualization settings" dialog window and with the help of buttons on

tool bar "Visualization", "Scale and shift" and "Plot setting".



Frame "Water"

## Bore Holes

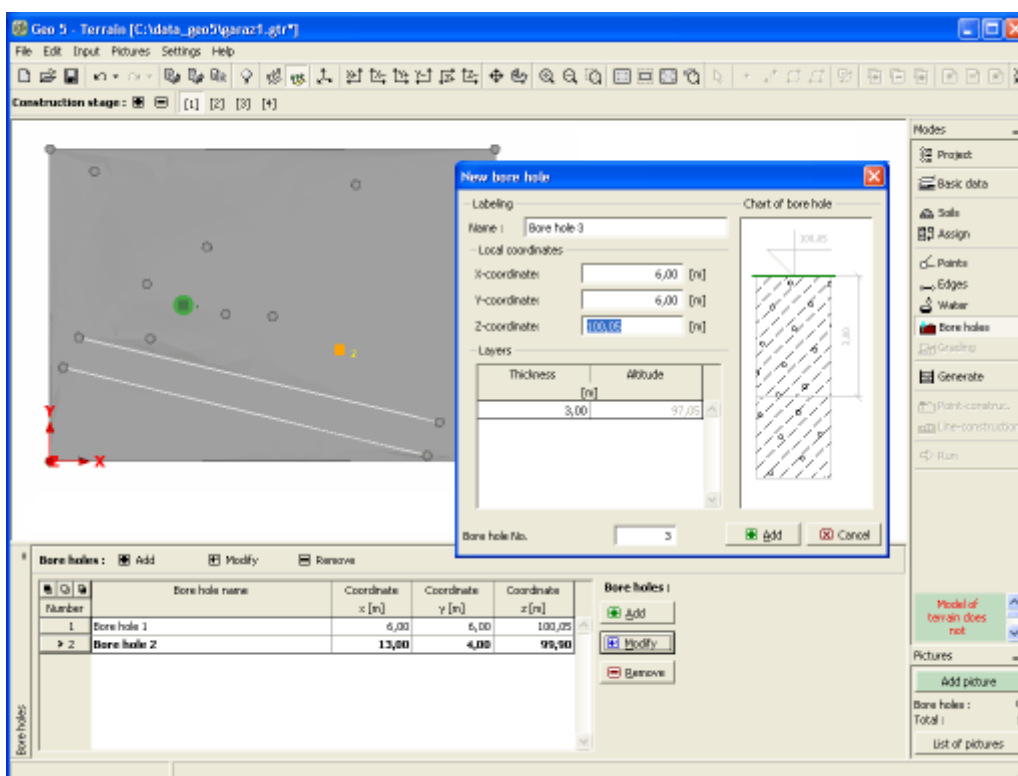
The frame "Bore holes" serves to define bore holes, which allow for the modeling individual **geological layers** (depending on the setting in the frame "Basic data") or **ground water tables** (depending on the setting in the frame "Water").

To input points that determine the location of individual bore holes proceed in the similar way as when defining **terrain points**. Apart from coordinates it is necessary to enter the bore-hole name and thicknesses of layers. The generated geological profile can be easily modified exploiting the option of **automatic calculation of height z** from the thicknesses of individual layers.

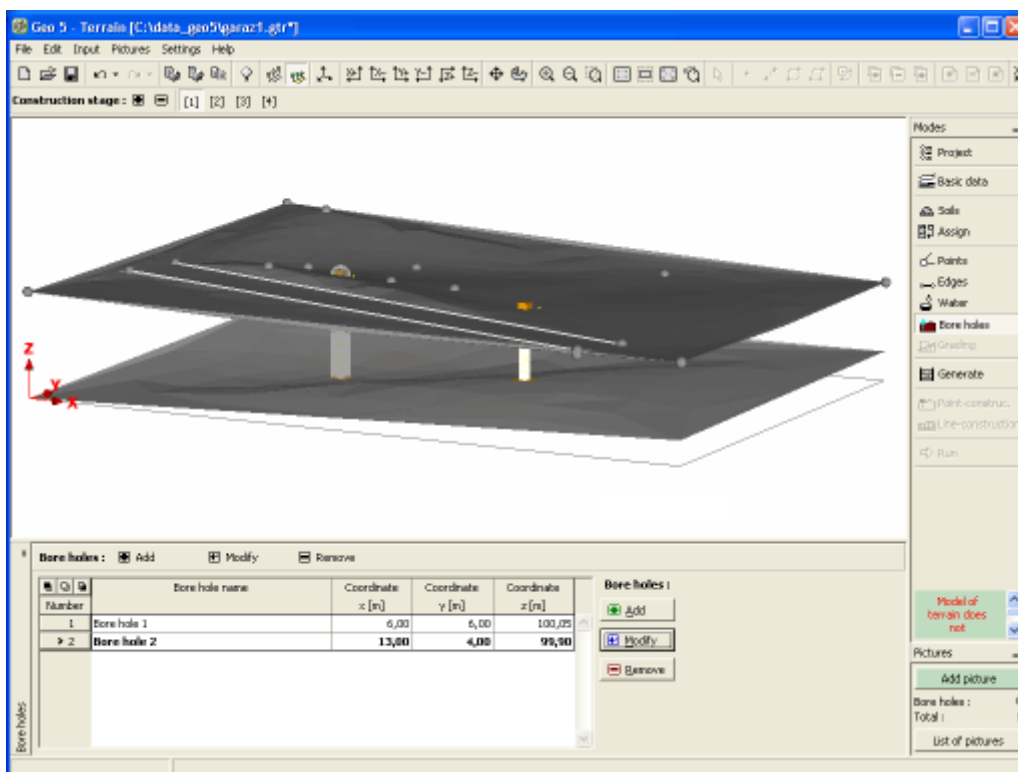
Bore holes can be defined **only in the first stage of construction**. The program automatically assures that a lower layer always lies below an upper layer - "**Crossing of layers**" is not acceptable - the **dominant layer** is always the **upper layer**.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the "Visualization settings" dialog window and with the help of buttons on tool bar "Visualization", "Scale and shift" and "Plot setting".

The program also allows for **importing bore holes in format gINT** at once.



Frame "Bore holes" - input, edit



Frame "Bore holes" - defined bore holes

## Earth Grading

The **frame "Earth grading"** serves to define terrain earth grading. The earth grading cannot be defined in the first stage of construction.

The earth grading should considerably simplify an input of excavation pits or embankments. The essential part of earth grading is **the shape of bottom**, from which the slopes of excavations or embankments are directed towards the original terrain. The original terrain points and edges, found in the region of earth grading, are automatically removed during generation.

More than one earth grading can be defined within a single stage of constructions. **They must not, however, cross each other**. If that happens, they need to be combined into a single earth grading. No part of earth grading can also exceed the world dimensions - in such a case one should realize that faces of earth grading may exceed the world dimensions even if the bottom is defined well inside.

The earth grading can be edited only in the stage, where it is defined. In the next **stage of construction**, the earth grading is transferred in terms of terrain new points and edges.

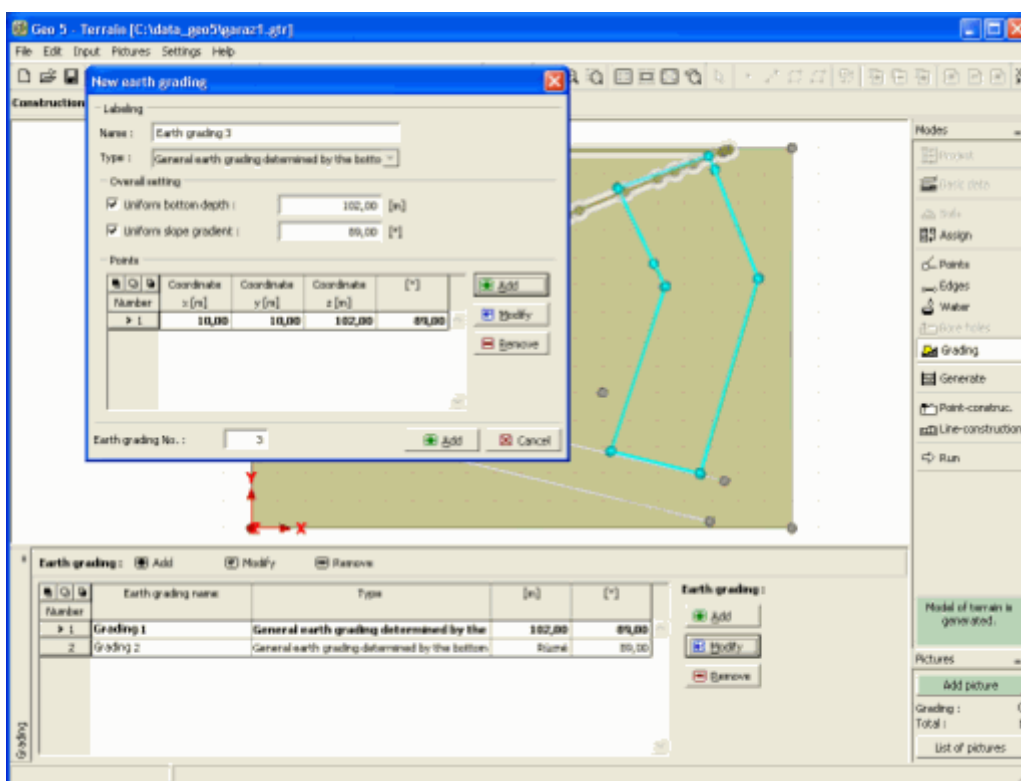
**With the help of table:** earth grading is defined in the **table**. The **"Add"** button opens the **"New earth grading"** dialog window, which allows for specifying the name of earth grading (by checking individual boxes it is possible to define a uniform depth of the bottom and a uniform gradient of the slope). This dialog window contains a table to introduce points, which define the ground plan (general polygon) of earth grading. To enter these points, proceed in the similar way as when defining **terrain points**. Pressing the **"Add"** button closes the dialog window and the new earth grading is inserted into the table.

The earth grading can be further modified (in the dialog window) using the **"Edit"** button or removed with the help of **"Remove"** button (more than one earth grading can be marked in the table to remove them all at once - before removing, the selected earth grading is displayed on the desktop in **red**). Each change is immediately reflected on the desktop.

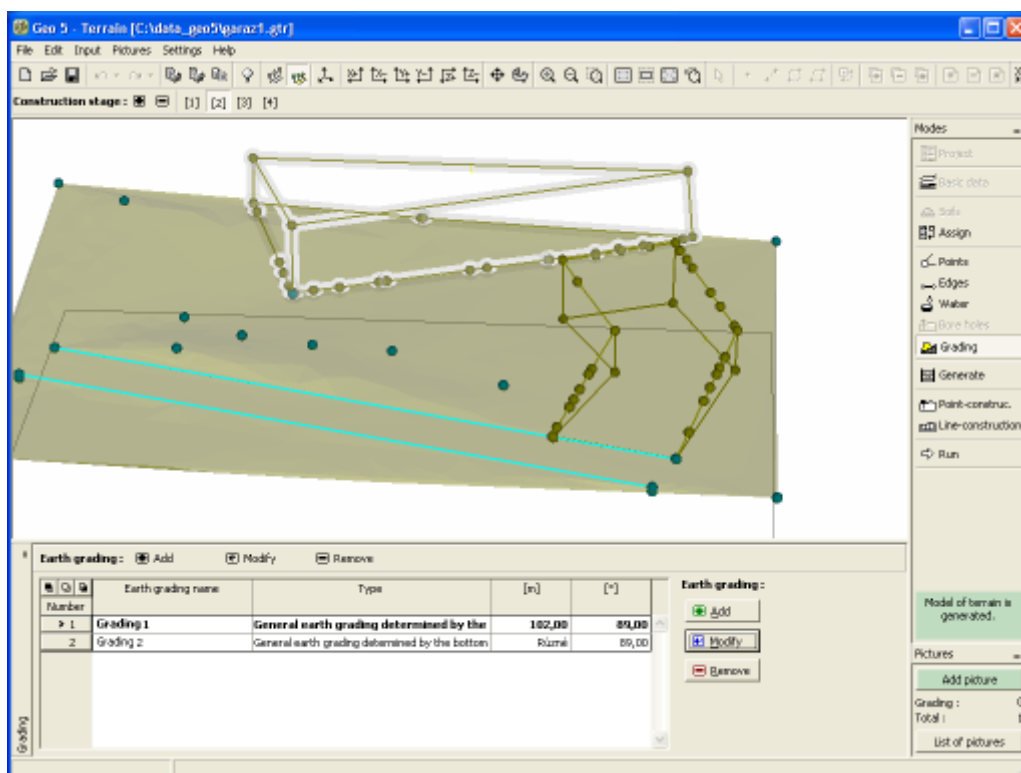
**With the help of the mouse:** this inputting mode is turned on by pressing a respective button on the horizontal bar. The following options are available:

- |                             |   |
|-----------------------------|---|
| <b>Add earth grading</b>    | <ul style="list-style-type: none"><li>• To input an earth grading, click the left mouse button on the desktop to successively define individual points of the polygon, which determines a ground plan of an earth grading - the polygon must be closed (the last clicked point serves as the first point of the polygon) - after closing the polygon the program opens the <b>"New earth grading"</b> dialog window; to continue follow the same steps as when an input using the table is assumed - providing the earth grading cannot be defined, or it overlays an already existing one, the program prompts a warning message</li></ul> |
| <b>Edit earth grading</b>   | <ul style="list-style-type: none"><li>• clicking an already existing earth grading using the left mouse button (see <b>active objects</b>) opens the <b>"Edit earth grading"</b> dialog window, which allows for editing the respective grading (is possible using buttons in the <b>dialog window "OK+↑ "</b> a <b>"OK+↓"</b>)</li></ul>   |
| <b>Remove earth grading</b> | <ul style="list-style-type: none"><li>• clicking an earth grading using the left mouse button opens a dialog window, which requests to confirm deletion of the selected earth grading</li></ul>   |

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the **"Visualization settings"** dialog window and with the help of buttons on tool bar **"Visualization"**, **"Scale and shift"** and **"Plot setting"**.



Frame "Earth grading" - input, edit



Frame "Earth grading" - defined earth grading

## Generate

The frame "Generate" serves to generate a model of the terrain.

**Parameters to generate the model**, which are valid for all subsequent stages, are specified in the first stage of construction.

These are:

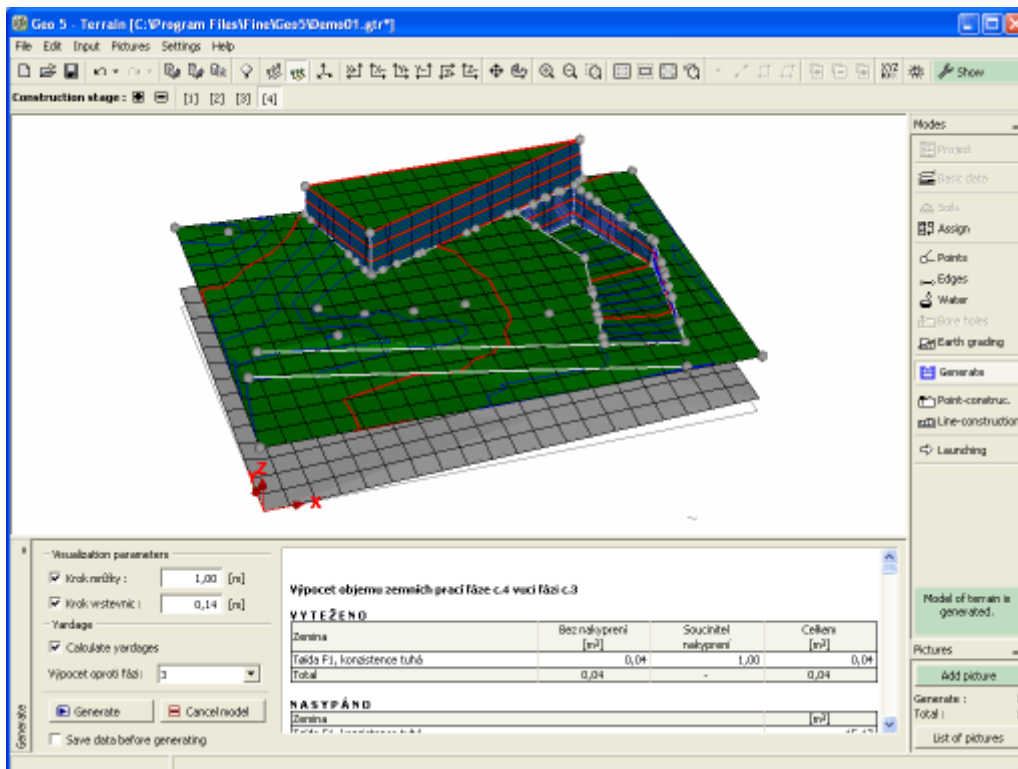
- **model smoothing** (none, medium, maximal)
- **active edge** - allows for modeling of terrain along edges

The frame further serves to define **drawing parameters** (grid step, contour line step).

The actual model is generated by pressing the "Generate" button. The generated model can be canceled by pressing the "Cancel model" button - this can be useful to enhance clarity of input.

Selecting the option "Compute yardage" allows for yardage calculation (in a combo list it is possible input the construction stage number for which the calculation should be carried out). This choice is not available in first stage of construction.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the "Visualization settings" dialog window and with the help of buttons on tool bar "Visualization", "Scale and shift" and "Plot setting".



Frame "Generate"

## Modeling Terrain on Edges

A special attention has to be given to **boundary condition** to correctly create a digital model of terrain - heights of points in corners and boundaries (edges) of the world (world



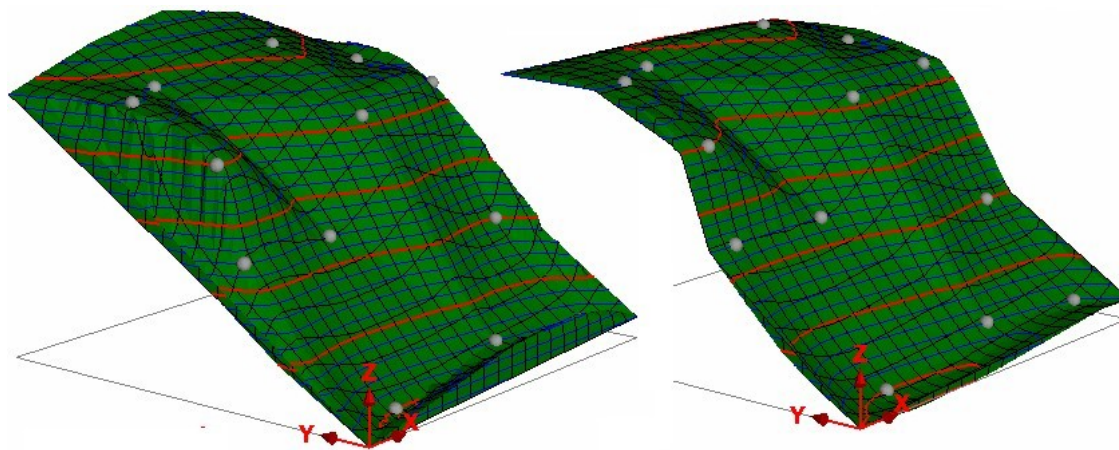
dimensions).

**The corner points** can be either entered or they are inserted automatically during the first stage of construction. When automatically generated, the corner point receives the same height as the closest point has or bore hole already defined.

When generating terrain **the corner points are connected by an edge**. In some cases (slopes) we wish the edges to model the overall **shape and inclination of terrain**. In such cases an active edge option can be used. An active edge is introduced as a **percentage fraction of the world dimensions**. All points **found on an active edge** are, during generation, automatically projected in the normal direction on to an edge - new points are then created at the same locations (on edge) having the same z-coordinate. The new points are stored in data associated with the next stage of construction.

Subsequent layers of the terrain model behave the same way. The thicknesses of these layers on edges are calculated according to thicknesses of layers of the closest bore holes.

The role of an active edge is evident from the following figure.



*Terrain generated without and with an active edge*

## Point Constructions

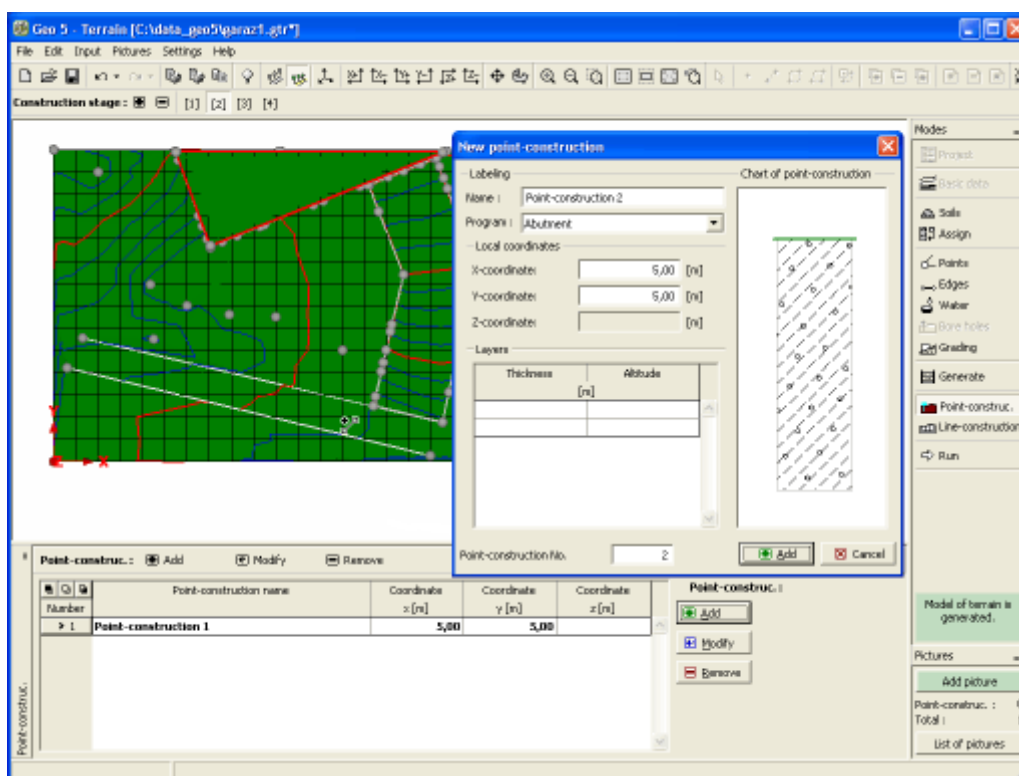
The frame "**Point constructions**" serves to introduce point constructions into the terrain.

To input points that determine the location of individual point constructions proceed in the similar way as when defining **terrain points** (using either table or mouse). The "**New point construction**" ("**Edit point construction**") dialog window allows also for specifying the name of the program to analyze the corresponding construction. The frame "**Launching**" is then used to run the calculation program and to transfer thicknesses of layers and assignment of soils into the program.

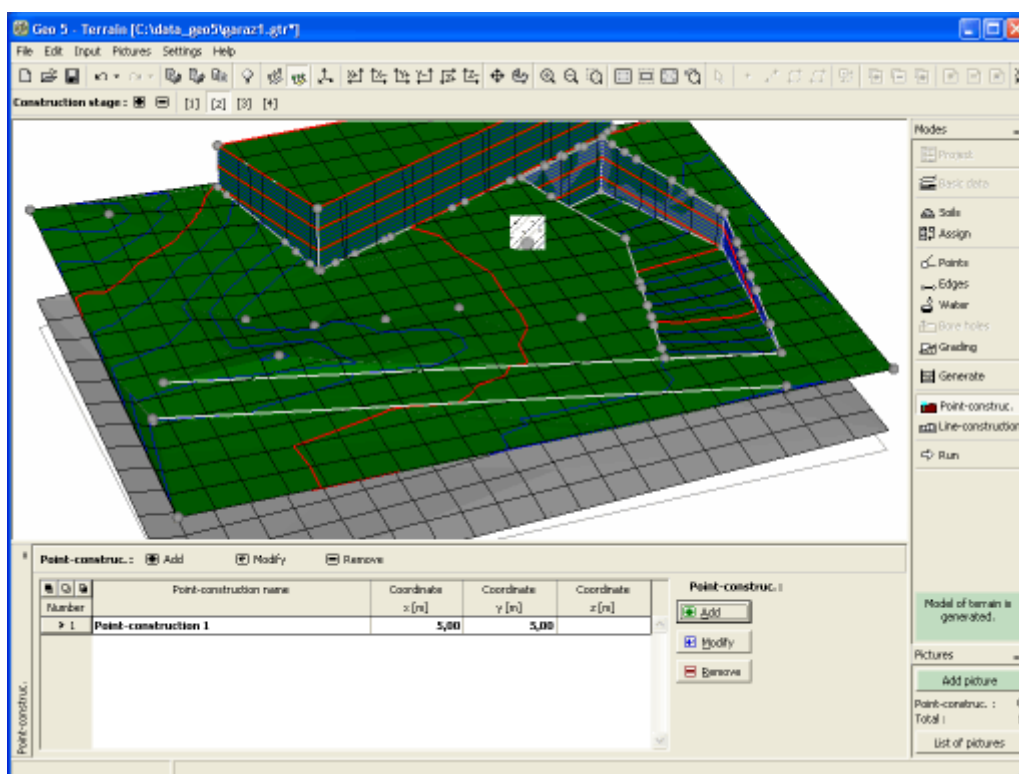
Point constructions can be defined only if a correct model of the terrain is **generated**.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the "**Visualization settings**" dialog window and with the help of buttons on tool bar "**Visualization**", "Scale and shift" and "Plot setting".





Frame "Point constructions" - input, edit



Frame "Point constructions" - defined constructions

## Line Constructions

The frame "Line constructions" serves to introduce line constructions into the terrain.

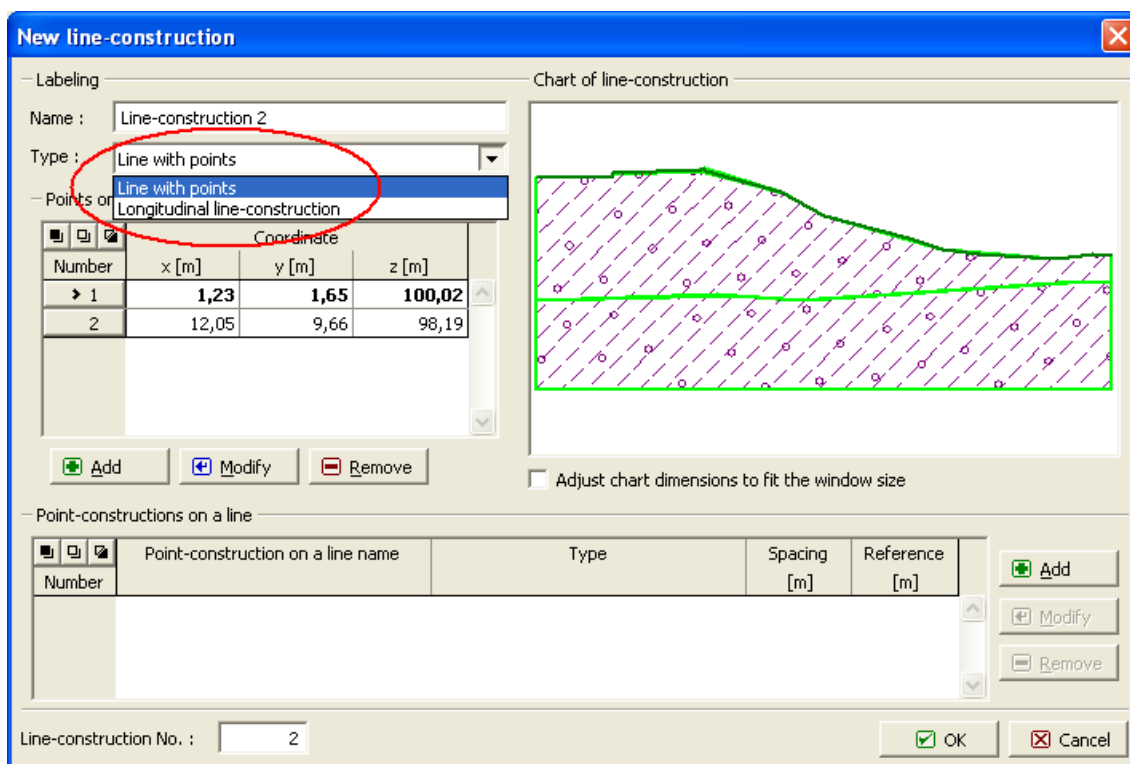
To input lines that determine the location of individual line constructions proceed in the similar way as when defining terrain edges (using either table or mouse). The "New line construction" ("Edit line construction") dialog window allows for specifying the name and type of a construction line:

"Longitudinal line construction" is defined by coordinates of the starting and end points (the table is a part of the dialog window). A combo list serves to select a particular calculation program (Settlement, Slope stability, FEM...). To run the program, use the frame "Launch". Terrain shape and interfaces are transferred in the same way as when assigning soils to layers.

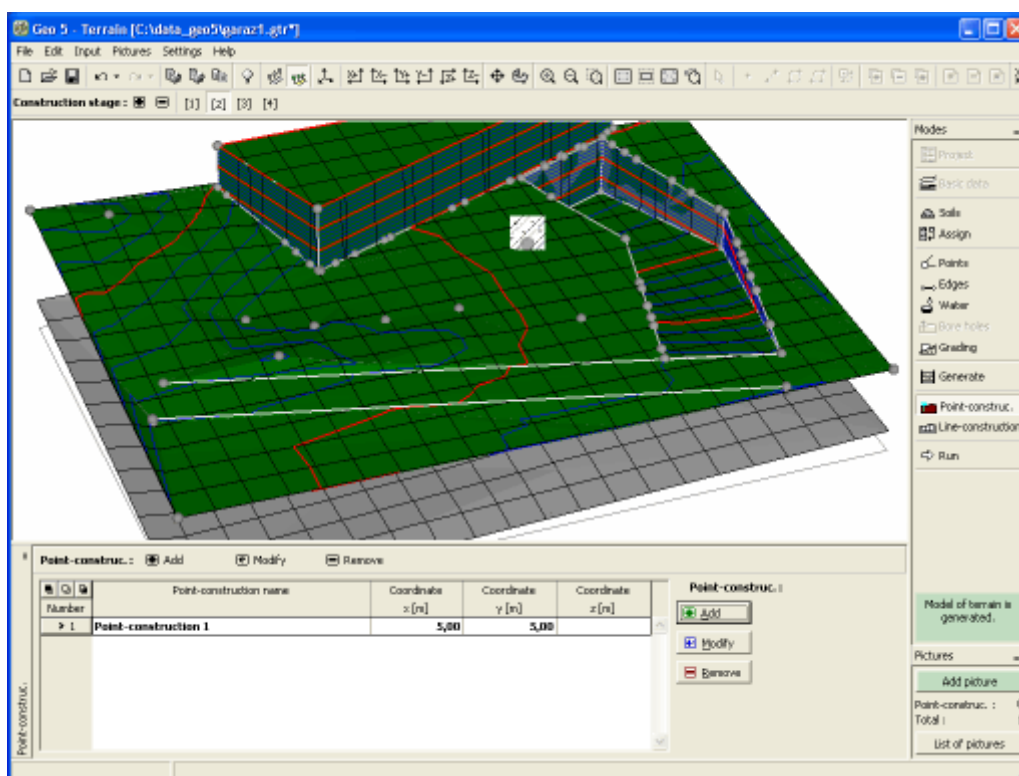
"Line with points" is defined by coordinates of a broken line and can be used to specify new point construction. Point constructions are defined in the table "Point constructions on line", which is a part of the "New line construction" dialog window. The frame "Launching" is then used to run the calculation program and to transfer thicknesses of layers and assignment of soils into the program.

Line constructions can be defined only if a correct model of the terrain is generated.

Visualization of drawing on the desktop can be modified in any input regime based on the setting adjusted in the "Visualization settings" dialog window and with the help of buttons on tool bar "Visualization", "Scale and shift" and "Plot setting".



Frame "Line constructions" - input, edit

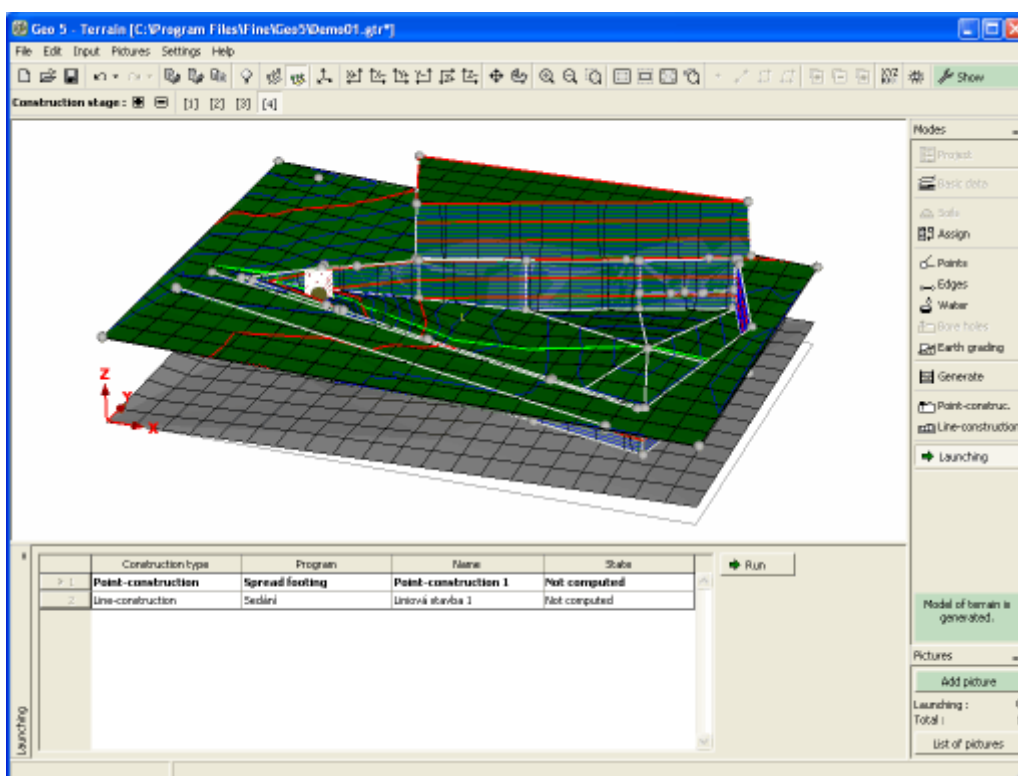


Frame "Line constructions" - defined constructions

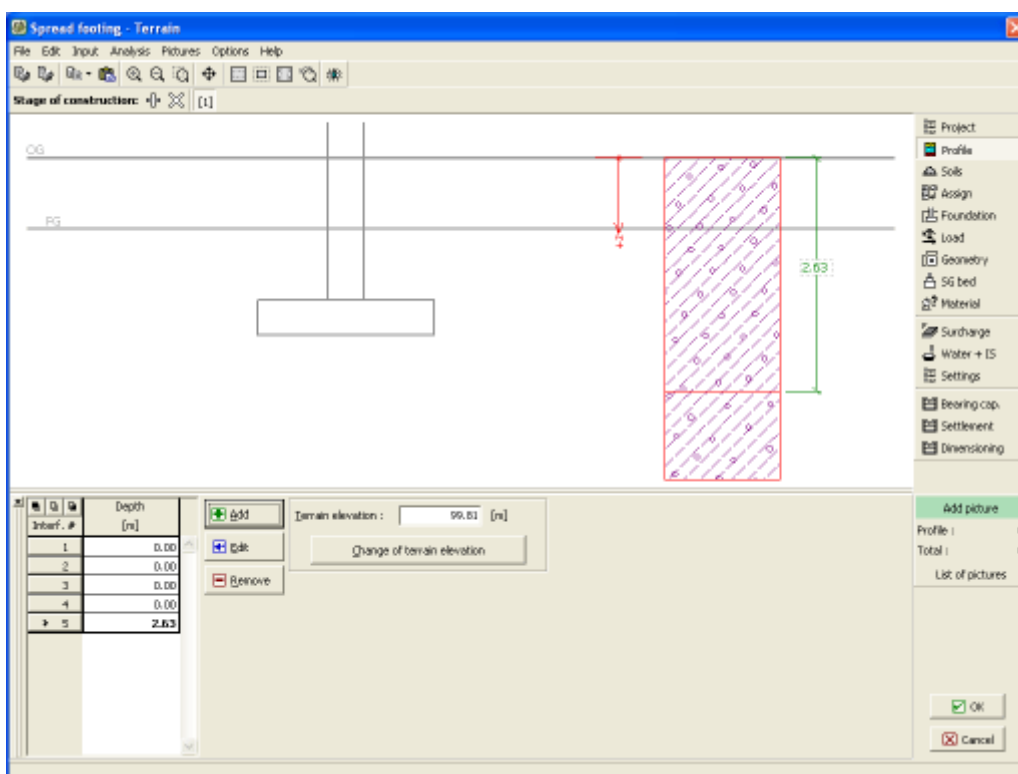
## Launching

The frame "Launching" contains a table with a list of defined point or line constructions. Based on the selection in the table and after pressing the "Launch" button the program associated with a particular task is launched (the corresponding calculation module must be purchased). The required data are transferred into the program. The program then allows for performing the specific calculations and verifications. If the program is not purchased, the launching button is not accessible.

When all calculations are completed the program is exited by pressing the "OK" button - the results and defined pictures are transferred back into a corresponding calculation protocol in program "Terrain".



Frame "Launching"



Launching program "Spread footing" from program "Terrain"

## Program FEM

Program FEM (and modules [Consolidation](#), [Water Flow](#), [Tunnel](#)) can model and analyze a wide range of geotechnical problems including:

- terrain settlement, consolidation
- sheeting structures
- anchored support structures
- slope stability
- excavation, tunnel analysis
- calculations of tunnels, ground losses
- calculation of water flow etc.

**The help in the program "FEM" includes the following topics:**

- Input of data into individual frames:

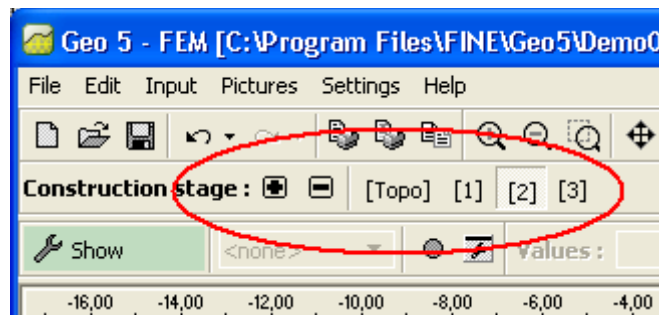
<a href="#">Topology:</a>	<a href="#">Project</a>	<a href="#">Settings</a>	<a href="#">Interface</a>	<a href="#">Soils</a>	<a href="#">Rigid Bodies</a>	<a href="#">Assign</a>
<a href="#">Contact Types</a>	<a href="#">Lining</a>	<a href="#">Free Points</a>	<a href="#">Free Lines</a>	<a href="#">Point Refinement</a>	<a href="#">Line Refinement</a>	<a href="#">Mesh Generation</a>
<a href="#">Construction Stages:</a>	<a href="#">Activation</a>	<a href="#">Assign</a>	<a href="#">Lining</a>	<a href="#">Beams</a>	<a href="#">Contacts</a>	<a href="#">Contacts and Beams</a>
<a href="#">Point Supports</a>	<a href="#">Point Flow</a>	<a href="#">Line Supports</a>	<a href="#">Line Flow</a>	<a href="#">Anchors</a>	<a href="#">Props</a>	<a href="#">Reinforcements</a>
<a href="#">Surcharge</a>	<a href="#">Beam Loads</a>	<a href="#">Water</a>	<a href="#">Analysis</a>	<a href="#">Monitors</a>	<a href="#">Graphs</a>	<a href="#">Stability</a>

- [Outputs](#)
- General information about the work in the [User Environment](#) of GEO5 programs
- [Common input](#) for all programs

## Topology

To input data in the FEM program slightly differs from our other programs in that it requires defining the topology of the structure prior to any calculation. This step includes introduction of interfaces between individual layers of soils, line constructions, parameters of soils and interfaces and at last generation of the finite element mesh. To avoid unexpected errors when creating a computational model the user should first become familiar with available [coordinate systems](#).

The topology input regime is selected by clicking the Topo button on the horizontal bar.



Bar "Stages of construction" - switching between "Topology" regime and calculation stages

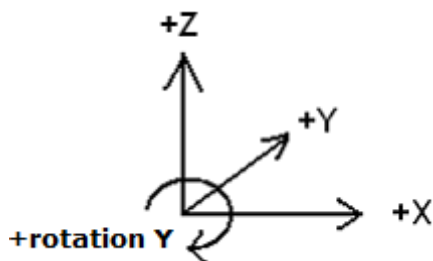
The actual analysis is performed in individual **stages of construction** (calculation stages), which allow the user to define activity of structures, to input beams, anchors and surcharge, to model the effect of water, etc.

Depending on the selected regime the **vertical tool bar** also adjusted.

## Coordinate Systems

### Global coordinate system

- is right-handed
- the positive  $X$  axis is directed from the left to the right
- the positive  $Z$  axis is directed from the bottom to the top
- the positive  $Y$  axis drills in to the  $XZ$  plane
- the rotation about the  $Y$  axis is positive when measured clockwise

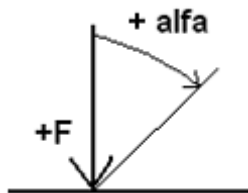


- The GCS is used for coordinates
- in general, the positive surcharge is assumed to act in the opposite direction to the positive axis and the positive rotation follows the positive sense of the global rotation
- particular definitions of the positive direction must be carefully examined for all cases

### Surcharge

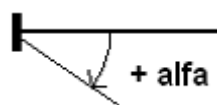
- is always assumed to act along the horizontal line (or at a point)
- the origin (point) and the length are the required input data
- the positive surcharge at zero angle is assumed to act in the opposite direction to the positive direction of the  $Z$  axis
- the zero angle corresponds to vertical surcharge
- the angle increases clockwise

- the angle ranges from  $-180^\circ$  to  $180^\circ$



### Anchors

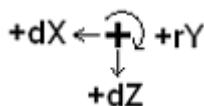
- an anchor can also be specified by the origin and an angle
- the zero angle corresponds to the direction of the  $X$  axis
- the angle increases clockwise



- the angle ranges from  $-180^\circ$  to  $180^\circ$

### Prescribed displacements and rotation of supports

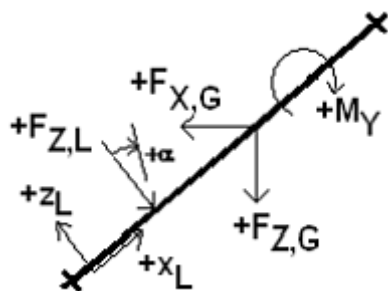
- prescribed displacements are positive in the directions of the  $X$ ,  $Z$  axes and about the  $Y$  axis
- displacements are positive when developed in the opposite direction to the positive directions of the coordinate axes



- the positive rotation is measured clockwise

### Load of beams

- the local coordinate system is right-handed
- the positive  $X_L$  axis of the beam is assumed in direction from the starting to the end point
- the positive  $Z_L$  axis is perpendicular and rotated counterclockwise by  $90^\circ$  from the beam axis
- load can be applied three directions:
  - global  $Z$
  - global  $X$
  - local normal ( $Z$ )
- the positive load in the global direction acts in the opposite direction to the positive direction of the corresponding axis
- the positive load in the normal direction acts in the opposite direction to the positive direction of the local  $Z_L$  axis
- the positive load angle  $\alpha$  is measured clockwise
- the moment is positive when acting clockwise



- definition of load along the  $X_L$  axis
  - coordinates, coordinates of the origin
  - load span
- types of load (always in above mentioned directions)
  - concentrated force
  - concentrated moment
  - distributed uniform over the whole beam
  - distributed trapezoidal over the whole beam
  - distributed uniform over a segment of the beam
  - distributed trapezoidal over a segment of the beam

### Stresses and strains

- positive normal stress  $\sigma$  corresponds to compression, negative to tension
- positive normal strain  $\epsilon$  corresponds to compression, negative to tension

### Internal forces along beams

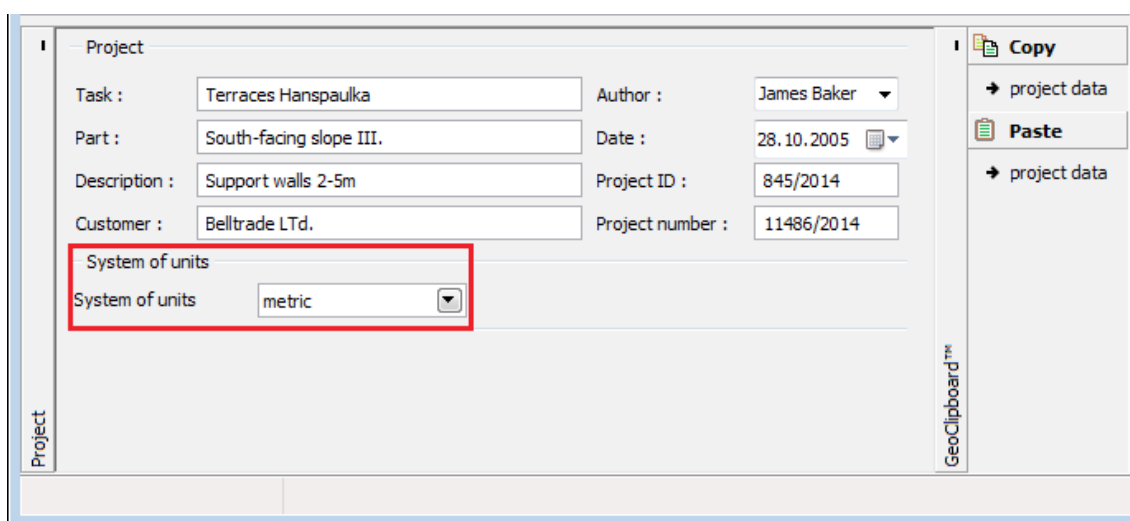
- positive normal force corresponds to tension, negative to compression
- positive normal strain  $\epsilon$  corresponds to compression, negative to tension

## Project

The frame **"Project"** is used to input basic project data and to specify overall settings of the analysis run. The frame contains an input form to introduce the basic data about the analyzed task, i.e. project information, project description, date, etc. This information is further used in **text and graphical outputs**.

The frame also allows user to switch analysis units (**metric / imperial**). Project data can be copied within all GEO5 programs using **"GeoClipboard"**.





Frame "Project"

## Settings

The **frame "Settings"** serves to specify standards or methods that are used to perform the analysis.

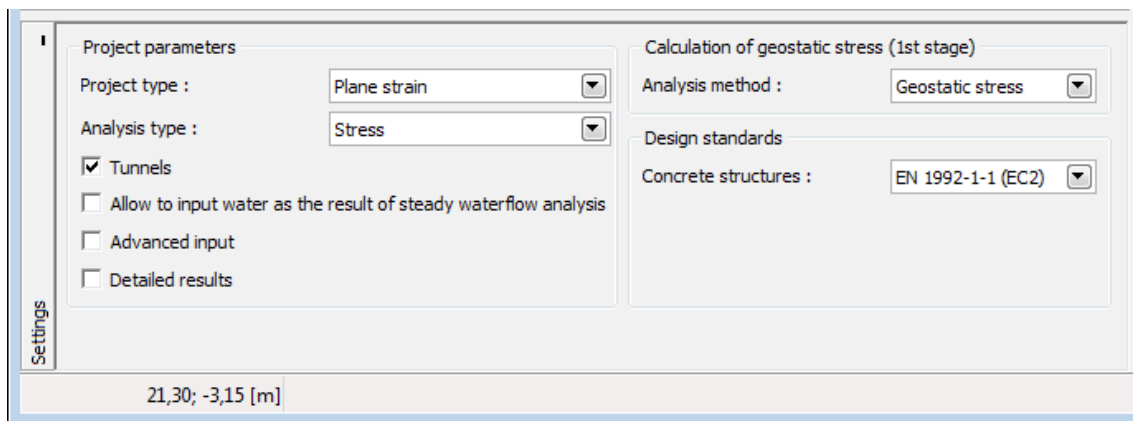
The **"Analyses"** tab allows the user to define the basic characteristics of the analysis to be carried out including the type of the problem and analysis, the method of calculating the initial stress (geostatic stress,  $K_0$  procedure) and available standards for concrete and steel structures.

**The available problem (plane strain analysis, axial symmetry) and analysis (stress, slope stability, water flow, tunnels, consolidation) types depend on the purchased configuration of the program.**

**Providing all modes are available we recommend to proceed with extreme caution when selecting the type analysis - more complex types require distinctively larger number of input data and may unnecessarily complicate the use of the program.**

**This tab also allows for choosing the option "Enhanced input", which affects both the input parameters of the program and possibilities of presenting the analysis results.**

This tab also serves to select the method for calculating the initial stress in the first stage of construction - either standard calculation of **geostatic stress** or the  **$K_0$  procedure**.



Frame "Settings"

## Stability Analysis

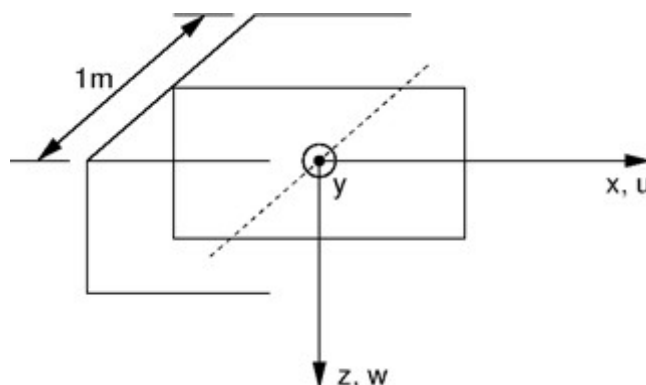
There are two options available in the FEM program to solve the slope stability problem:

1. To set the solution type to **"Slope stability"** in the frame "Settings".
2. To run the module in the **"Slope stability"** regime in an arbitrary **stage of construction** of a standard analysis by pressing the **"Stability"** button - in such a case, a new secondary task (which can be saved independently) is generated. The solution then proceeds as in the step 1.

Creating model and inputting data in the **"Slope stability"** regime is performed in the same way as in the **"Stress"** regime - just the **"Analysis"** button **launches the slope stability analysis** of a given structure. Individual slope stability analyses in construction stages are completely independent and have no relation to **the previous stages and calculations**.

## Plane Strain Analysis

This computational module is suitable for the analysis of longitudinal structures (**tunnel, embankment, dam**, etc.) characterized by a longitudinal dimensions being of the orders of magnitude larger than transverse dimensions of the analyzed domain.



Plane strain analysis

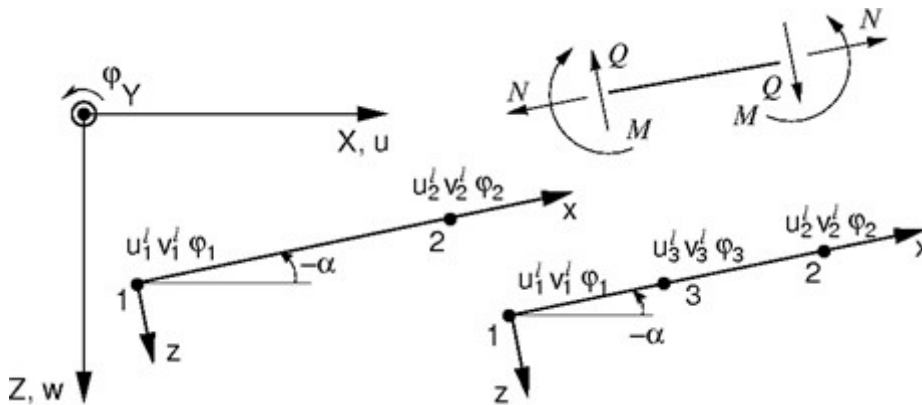
In such a case the analysis can be carried per 1 m run of the structure, see Figure. This complies with the plane strain assumption. Components of the strain vector developed on planes normal to the longitudinal axis can be then neglected. Therefore, we assume the soil body be loaded by the components of the strain and stress vector pertinent to the transverse

plane normal to the longitudinal axis and by the longitudinal normal stress that arises due to Poisson's effect. The corresponding non-zero components of the stress and strain vector are:

$$\sigma^T = \{ \sigma_{xx} \ \sigma_{zz} \ \tau_{xz} \ \sigma_{yy} \}$$

$$\varepsilon^T = \{ \varepsilon_{xx} \ \varepsilon_{zz} \ \gamma_{xz} \ \varepsilon_{yy} = 0 \}$$

Considering beam elements the analysis corresponds to the solution of a plate strip having the cross-section width equal to 1 m. Non-zero components of nodal generalized displacements are evident from the following Figure for a two-node beam element compatible with a three-node triangular plane element and a three-node beam element compatible with a six-node triangular plane element.



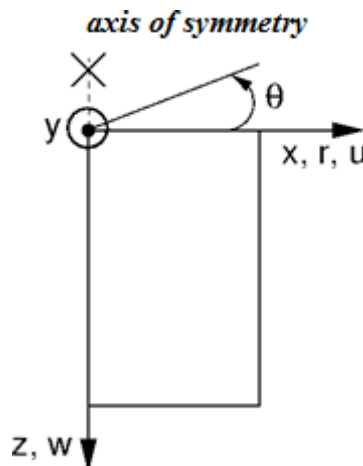
Two-node and three-node beam element

The corresponding components of internal forces, see Figure, assumed with respect to 1 m of cross-sectional width are given by:

$$\sigma^T = \{ N \equiv n_{x1}, M \equiv m_y, Q \equiv q_{z1} \}$$

### Axial Symmetry

This computational model is suitable for the analysis of structures of revolution. This assumption must be satisfied from both the construction and load point of view. A typical example is the solution of vertically loaded isolated pile, excavation of circular ditch, or pumping of water from a circular hole.



*Axisymmetric analysis*

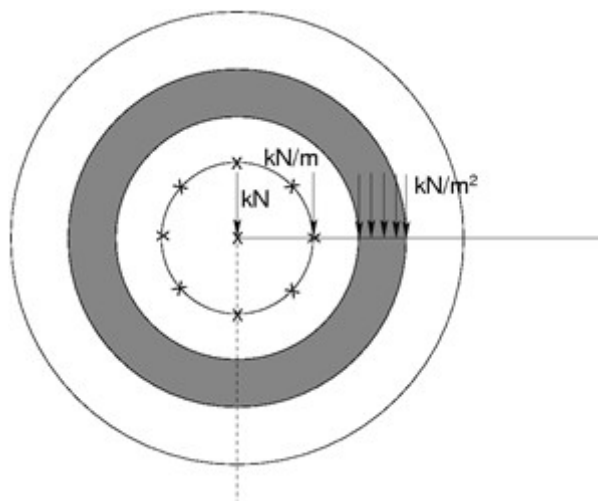
Similarly to the plane strain analysis the computational problem is three dimensional, which can, however, be transformed into a two-dimensional problem, see Figure. The analysis is then performed with respect to 1 m of arc length having diameter equal to  $x(r)$ . The axis of symmetry always corresponds to the origin of the  $x(r)$  coordinate. Shear strain components in the direction of rotation can be neglected. We are then left with the stress and strain components acting on the plane of symmetry cut and the normal strain and stress component in the hoop (circumferential direction) direction. The corresponding non-zero components of the stress and strain vector are:

$$\sigma^T = \{ \sigma_{xx} \sigma_{zz} \tau_{xz} \sigma_{\theta\theta} \}$$

$$\varepsilon^T = \left\{ \varepsilon_{xx} \varepsilon_{zz} \gamma_{xz} \varepsilon_{\theta\theta} = \frac{u}{r} \right\}$$

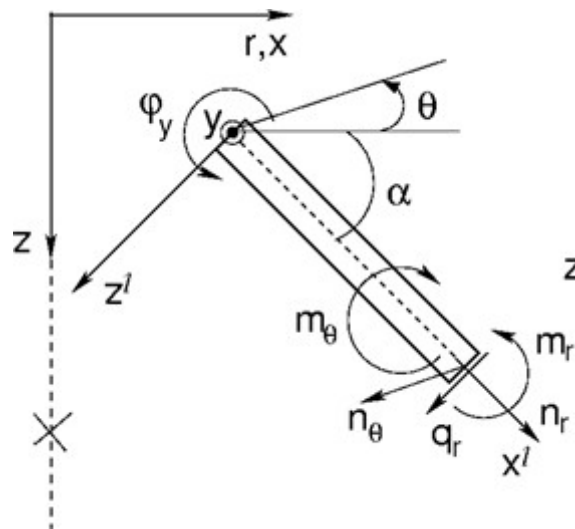
It is clear that the hoop strain, and therefore also the affected normal stresses, attains an infinite value at the axis of symmetry. Thus regarding the finite element approximation, arriving at reliable and sufficiently accurate estimates of these values requires a relatively fine mesh along the symmetry axis.

The application of line and surface load is also worth mentioning. Several examples of applying the load on terrain surface are displayed in the following Figure. Clearly, their effect increases with the distance from the axis of symmetry. Introducing such a type of load directly on the axis of symmetry has, therefore, no effect. In such a case it is necessary to choose the type of load of axis of symmetry. The program allows for the application of concentrated forces only.



*Examples of load applied on terrain surface*

Considering beam elements the analysis corresponds to the solution of a plane membrane of revolution including bending effects. Non-zero degrees of freedom are identical adopted for beam elements in plane strain analysis. Apart from axial (meridian) effects it is necessary to consider also membrane and bending effects in the hoop direction, see the following Figure.

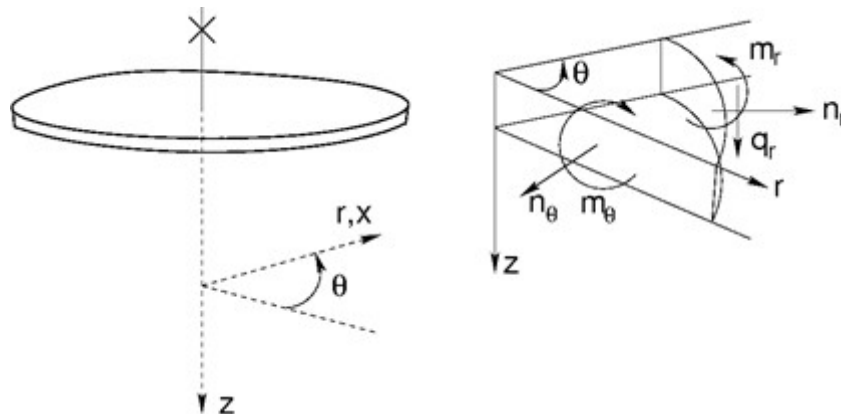


Beam element in axial symmetry

The corresponding components of internal forces, see Figure, assumed with respect to 1 m of cross-sectional width are given by:

$$\sigma^T = \{n_r, m_r, q_r, n_\theta, m_\theta\}$$

In a special case of a circular plate (angle  $\alpha = 0$ ) we may refer to radial and hoop components of internal forces, see the following Figure.



Internal forces acting on a circular plate

Internal forces are related to corresponding strains and curvatures as follows:

$$\begin{Bmatrix} n_r \\ m_r \\ q_r \\ n_\theta \\ m_\theta \end{Bmatrix} = \frac{1}{1-\nu^2} \begin{bmatrix} EA & 0 & 0 & \nu EA & 0 \\ 0 & EI_y & 0 & 0 & \nu EI_y \\ 0 & 0 & \frac{(1-\nu)kEA}{2} & 0 & 0 \\ \nu EI_y & 0 & 0 & EA & 0 \\ 0 & \nu EI_y & 0 & 0 & EI_y \end{bmatrix} \begin{Bmatrix} \frac{du^t}{dx} \\ \frac{d\varphi_y}{dx} \\ \varphi_y + \frac{dw^t}{dx} \\ \frac{u^t \sin \alpha - w^t \cos \alpha}{r} \\ \frac{\varphi_y \cos \alpha}{r} \end{Bmatrix}$$

After setting r equal to infinity we arrive at plane strain conditions. It is worth noting that in case of shear forces their magnitudes are, unlike the plane strain analysis, significantly

dependent on the refinement of the finite element mesh. This holds also for vertical reactions.

### Note to water flow

Recall that similarly to the reactions forces in the stress analysis the point fluxes at nodes with prescribed pore pressures are evaluated with respect to 1 *m* of arc length having diameter equal to *x(r)*. In case of plane strain analysis the corresponding values are taken per 1 *m* run. The corresponding overall fluxes (inflow/outflow) can be determined from point fluxes [*m*<sup>3</sup>/*day/m*] as follows:

### Plane strain analysis

$$\sum Q = \sum_{i=1}^N Q_i \text{ [m}^3/\text{day/m]}$$

### Axisymmetric analysis

$$\sum Q = \sum_{i=1}^N 2\pi x_i Q_i \text{ [m}^3/\text{day]}$$

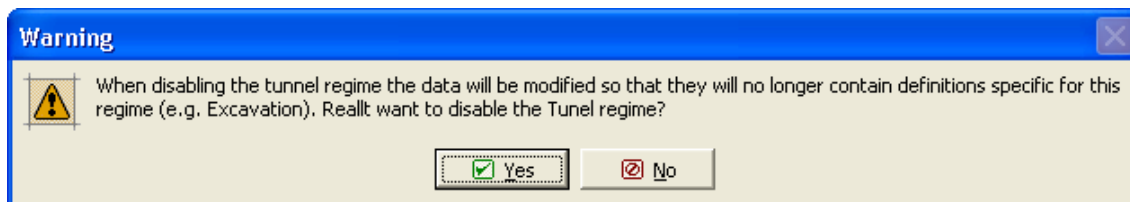
where *N* is the number of nodes along a given mesh line, in which the point fluxes *Q<sub>i</sub>* [*m*<sup>3</sup>/*day/m*] are calculated. In the case of axisymmetric analysis *x<sub>i</sub>* represents the *x*-coordinate of a given point. Therefore, the axisymmetric analysis provides total the inflow/outflow [*m*<sup>3</sup>/*day*] through for example a cylindrical surface (vertical line) or circular surface (horizontal line).

## Tunnels

The frame "Settings" in conjunction with the "Analyses" tab allows for selecting the option "Tunnels". (This module has to be **purchased** by the user - otherwise this option is not available). When selecting the "Tunnels" regime it is possible to define and calculate:

- **excavations** (Modeling a 3D effect at the tunnel face assuming the New Austrian method)
- **gradual degradation of beams**
- **subjecting beams to thermal load**
- **thermal load applied to selected regions** (Advanced input is required)
- **prescribing swelling stress to selected regions**
- **monitoring results**

The mode "Tunnels" can be switched on/off at any time. The previous results will be, however, deleted. While switching from a standard regime to the "Tunnels" regime is safe, proceeding in the opposite direction results into deleting all input data - a warning message, however, appears before this action is completed.



*Warning message about data modification when canceling the "Tunnels" regime*

## Consolidation

**Consolidation** analysis is the optional module of **FEM** program. We can switch the consolidation analysis on in the "**Settings**" frame by switching "**Analysis type**" to "**Consolidation**".

The input has the following limitations:

- the "**Tunnels**" regime cannot be used
- only some **materials models** can be used: **Elastic**, **Mohr-Coulomb**, **Modified Mohr-Coulomb** and **Drucker-Prager**
- active regions can be fully specified in the 1st construction stage only, in the following construction stages only **activating** the regions located above the terrain of the previous construction stage is allowed. No openings or cuts are possible.
- mesh is obligatorily generated using **multinode elements**
- **water** can be entered in the 1st construction stage only and "**No water**" or "**GWT**" can only be used
- **beams** are always in transverse direction impermeable, only the drain along the beam on each side of the beam can be specified
- no point flow is available, the **line flow** on the borders have only "**permeable**" or "**impermeable**" boundary conditions

The data input and calculation is as follows:

- **topology** is defined the same way as in stress analysis, the finite element mesh is **generated**
- in the first **construction stage** in the "**Water**" frame the pore pressure is specified by entering **ground water table**
- in the following construction stages are entered: the data for stress analysis, the **flow boundary conditions**, beam and **contact flow parameters**, analysis **parameters**, i.e. phase time and load action

The **results** of the analysis are presented in the same way as in stress **analysis**, in addition, the flow velocities are available.

The principle of the numerical solution is [here](#).

## Principle of Numerical Solution of Consolidation

### Consolidation

In standard stress analysis the program GEO5 FEM allows for two specific approaches to the modeling of pore pressure action on a soil body. In case of undrained conditions it is assumed that all boundaries surrounding the undrained soil are impermeable, the soil is considered as volumetrically incompressible, and the applied load results in the generation of excess pore pressure within this layer. Introducing suitable boundary conditions that allow for a gradual dissipation of excess pore pressure provides passage to drained conditions. In case of drained conditions we assume that the resulting pore pressure is no longer influenced by the deformation of the soil body. Transition from drained to undrained conditions describes the theory of consolidation.

The term consolidation stands for the soil deformation in time caused by external load, which can be either constant or time dependent. This is a reological process. For the present case we limit our attention to so called primary consolidation characterized by the reduction of volume of pores and thus the change of the internal soil structure due to load accompanied by the

escape of water from pores. The analysis assumes a fully saturated soil. Consolidation analysis in a partially saturated soil is not addressed by the program. The governing equation describing the flow of water (continuity equation,  $\dot{\mathbf{a}}$  represents a time derivative of a given quantity) in a fully saturated ( $S = 1, \dot{S} = 0$ ) deforming soil body is provided by (recall the Richards equation for the description of **transient (unsteady) water flow**).

$$\frac{1}{M} \dot{p} + \alpha \dot{\epsilon}_v + \nabla^T \left( -\frac{\mathbf{K}_{sat}}{\gamma_w} (\nabla p - \gamma_w i_g) \right) = 0 \quad (1)$$

- where:  $M$  - Biot's modulus, assumed in the range of  $M = (100-1000)K_{sk}$  ( $K_{sk}$  is unit modulus of skeleton). In general, it is a large number enforcing a volumetric incompressibility of a given fully saturated soil at very short times at the onset of consolidation. A default setting is  $M = 10^6 \text{ kPa}$ .
- $\alpha$  - Biot's parameter, typically assumed  $\alpha = 1$
- $p$  - pore pressure
- $\nabla p$  - pore pressure gradient
- $\mathbf{K}_{sat}$  - permeability matrix storing coefficients of permeability for a fully saturated soil, typical values for selected soils are stored in [Table](#)
- $i_g$  - hydraulic gradient

The rate of change of total pressure is given by:

$$\dot{\sigma} = \mathbf{D}^{ep} \dot{\epsilon} - \alpha 3 \mathbf{m} \dot{p}_{ex} \quad (2)$$

- where:  $\mathbf{D}^{ep}$  - current stiffness matrix
- $p_{ex}$  - excess pore pressure

$$\mathbf{m} = \left\{ \frac{1}{3}, \frac{1}{3}, 0, \frac{1}{3} \right\}^T \text{ for plane strain or axial symmetry}$$

Note that the total pore pressure  $p$  is a sum of steady state pore pressure  $p_{ss}$  and excess pore pressure  $p_{ex}$ . It holds:

$$\dot{p}_{ss} = 0 \quad (3)$$

The continuity equation (1) can be thus written as:

$$\frac{1}{M} \dot{p}_{ex} + \alpha \dot{\epsilon}_v + \nabla^T \left( -\frac{\mathbf{K}_{sat}}{\gamma_w} \nabla p_{ex} \right) = 0 \quad (4)$$

adopting the zero excess pore pressure boundary condition at the boundary with prescribed pressure as:

$$p_{ex}(t) = 0 \quad (5)$$

and the zero flow ( $q(t) = 0$ ) across the boundary with prescribed water flux density:

$$\mathbf{n}^T \left( \frac{\mathbf{K}_{sat}}{\gamma_w} \nabla p_{ex}(t) \right) = 0 \quad (6)$$

- where:  $\mathbf{n}$  - components of outward unit normal



See: **Setting hydraulic boundary conditions.**

The overall total stress is then provided by:

$$\sigma = \mathbf{D}^{el}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{pl}) - \alpha 3 \mathbf{m}(p_{ss} + p_{ex}) \quad (7)$$

where:  $\mathbf{D}^{el}$  - elastic stiffness matrix  
 $\boldsymbol{\varepsilon}$  - vector of overall strains  
 $\boldsymbol{\varepsilon}^{pl}$  - vector of overall plastic strains

The current values of strains and excess pore pressure in equation (7) follow from the application of static equations of equilibrium and continuity equation (4) within the solution of coupled stress and water transport problem using the principal of virtual displacements. As in the case of **transient water flow** analysis a fully implicit forward Euler method is adopted to perform time discretization of equation (4). Further details can be found in [1,2,3].

### Consolidation analysis

As in the case of transient water flow analysis the first calculation stage serves to set initial conditions, i.e. the distribution of geostatic stress and steady state pore pressure. The steady state pressure values also equal to the overall pressure values at the end of consolidation. The initial pore pressure values are set by specifying the ground water table (GWT) only. It is worth to note that even if GWT table is found inside the analyzed soil body the soil below and above GWT is assumed fully saturated. This applies also to soils being introduced into the analysis in subsequent calculation stages (activating new regions). Removing or excavating soils (deactivating existing regions) is not possible with the current version of the program. The actual consolidation analysis is performed from the second stage on and requires setting the hydraulic boundary conditions, setting the time duration of a given calculation stage, setting the expected number of time steps and setting the way of introducing the load in to the analysis.

### Setting hydraulic boundary conditions

The program allows for introducing two types of hydraulic boundary conditions, recall equations (5) and (6):

- Zero pore pressure condition ( $p = 0$ ), which allows for free water outflow from the soil body, i.e. condition representing a permeable boundary. More specifically, this condition corresponds to the zero value of the excess pore pressure  $p_{ex}$ . The overall value of pore pressure along this boundary is thus equal to  $p_{ss}$ . This is a default boundary condition and is it assumed along all external domain boundaries, therefore also along the external boundaries of new regions.
- Zero flux density condition (no inflow/outflow,  $q = 0$ ), i.e. condition representing an impermeable boundary. If needed, this condition has to be introduced manually.

The choice of a given boundary condition influences the rate of consolidation. Further details can be found in [1].

### Setting the time step size - expected number of time steps in a given stage

Unlike the **transient water flow** analyses the initial time step size (discrete value of time increment when solving equation (4)) is in case of consolidation not directly assigned. Instead, this step is set on the bases of the specified duration of calculation stage and the input number of time steps expected for a given stage. In case of linear consolidation (all soils are assumed linearly elastic) the input number of time steps is performed. A time step reduction can take place in case of nonlinear consolidation if the lack of convergence for the current time step is encountered. This increases the number of steps to complete the stage analysis. When

specifying the number of steps in relation to the stage duration one should keep in mind that at the onset of consolidation the time step should be relatively small (particularly when referring to a load stage in conjunction with nonlinear consolidation), while with gradual progress of consolidation the time step size may reach several tens of days. Further details can be found in [1].

### Introducing load into the analysis

As in the case of **transient flow analysis** the program offers two options only:

- The load is applied at one step at the beginning of the calculation stage. More specifically, a linear increase of load over the first time step is assumed. Thus if we are interested in the behavior at  $t \rightarrow 0$ , it is necessary to suitably choose the combination of number of time steps and duration of the first stage (e.g. 1 and 0.001). In case of a very short time step and impermeable boundaries ( $q = 0$ ) we simulate the response of volumetrically incompressible soil ( $K \rightarrow \infty$ ) with a finite value of the shear modulus. The results for  $t \rightarrow 0$  will then agree well with the results derived from standard stress analysis with undrained soils. Further details can be found in [1].
- The load linearly increases over the calculation stage. The load increment then depends on the current time step size. Especially in case of nonlinear consolidation one should properly specify the time span over which the load is introduced to avoid convergence difficulties.

Providing there is no load change within a given stage the above setting options are irrelevant.

### Application of beam elements in consolidation

The beam permeability depends on its location and the choice of hydraulic boundary conditions. A beam found inside the soil body is in its normal direction impermeable. On domain boundaries the beam permeability in normal direction, as in case of water flow analyses, is driven by the selected boundary condition. In case of permeable boundary ( $p = 0$ ) the beam on this boundary is fully permeable, while in case of impermeable boundary ( $q = 0$ ) the beam on this boundary is also impermeable.

### Application of contact elements in consolidation

The reason for introducing contact elements into the analysis is twofold. First, we wish to allow for a relative mutual shift between two soils, soil and rock or soil and beam element, e.g. in the analysis of interaction of soil and sheeting structure. Second, the aim is at modeling potential drain along the beam or in general along a line to which the contact element is assigned. In every case one should realize a coupled simulation of both phenomena, i.e. stress and water flow analysis being carried out simultaneously. If not specified otherwise the program assumes the flow within a contact element being dependent on permeabilities of surrounding soils both in the longitudinal and transverse direction. In case of contact attached to the beam element the normal permeability  $k_n$  is irrelevant as the beam is assumed in this direction either fully impermeable ( $k_n = 0$ ) or fully permeable ( $k_n \rightarrow \infty$ ), see Application of beam elements in consolidation.

### General comments

Time evolution of individual variables, e.g. settlement or excess pore pressure, will be in case of linear consolidation always bounded by the solution of stress analysis when considering either undrained soils (all active soils in the analyzed domain are specified as undrained) or drained soils (standard setting, all active soils in the analyzed domain are specified as drained). The latter case coincides with the steady state analysis with total dissipation of excess pore pressure. The results of linear stress analysis with drained soils and linear consolidation derived at  $t \rightarrow \infty$  must be identical. However, this does not hold for nonlinear analyses as in such cases we cannot rely on the principle of superposition. Further details can

be found in [1].

Unlike water flow analyses, the solution of consolidation requires application of higher order elements (e.g. 6-node triangular or 8-node quadrilateral elements). While displacements are calculated at all nodes of a given element (quadratic approximation of displacement field), pore pressure is calculated at the corner nodes only (linear approximation of pore pressure).

Unlike one-dimensional consolidation implemented in program Settlement the two-dimensional consolidation yields at  $t \rightarrow 0$  zero volumetric strain and thus also zero mean effective stress only. Individual components of the displacement field are generally nonzero.

*Literature:*

[1] M. Šejnoha, T. Janda, H. Pruška, M. Brouček, *Metoda konečných prvků v geomechanice: Teoretické základy a inženýrské aplikace, předpokládaný rok vydání (2015)*

[2] Z. Bittnar and J. Šejnoha, *Numerické Metody Mechaniky II. České vysoké učení technické v Praze, 1992*

[3] Z. Bittnar and J. Šejnoha, *Numerical methods in structural engineering, ASCE Press, 1996*

## Advanced Input

The frame "Settings" in conjunction with the "Analyses" tab allows to select the option "Advanced input". The advanced input offers in addition:

- defining additional **material parameters** of soils (e.g. the Biot parameter, unit modulus of water, coefficient of thermal expansion)
- **three node elements** are available
- **mixed mesh** is available (triangular and quadrilateral elements)
- **additional output parameters** are available

By default this option is turned off. On the other hand, it may become particularly useful when doing research or during teaching. This input mode can be turned on/off at any time, but this action will always result into deleting all previous results.

Adopting Enhanced input option in the frame "Settings" allows the user to further specify the selected material model by choosing the type of soil and setting the Biot parameter  $\alpha$  and the coefficient of thermal expansion  $\alpha_t$ .

Default setting assumes drained (long term) boundary conditions (steady state conditions). In such a case the deformation of soil does not change the values of the prescribed pore pressure. Pore pressures then appear as an additional source of external load and stay constant during the analysis (state at the end of consolidation after full dissipation of the excess pore pressure  $u_e$ ). This particular component of total active pore pressure  $u$  is denoted as  $u_s$ . On the contrary, undrained boundary conditions assume that entire boundary surrounding the particular soil is impermeable. This results in the solution of a fully coupled problem of evolution of pore pressures in dependence on the deformation of soil assuming that all changes are instantaneous (short term conditions) with no influence of time (state at the beginning of consolidation). These excess pore pressures are denoted in the program as  $u_e$ . The overall active pore pressure then follows from:

$$u = u_s + u_e$$

The analysis requires specification of the effective unit modulus of water  $K_e$ :

$$k_e = \frac{\alpha}{\frac{\alpha - n}{K_s} + \frac{n}{K_w}}$$

- where:
- $K_e$  - effective unit modulus
  - $K_s$  - unit modulus of grains
  - $K_w$  - unit modulus of water
  - $n$  - porosity (volume of pores / volume of skeleton)
  - $\alpha$  - Biot's parameter

The actual value of this parameter does not have a major influence on the resulting pore pressures providing it is sufficiently high. Typically it is chosen from the interval 1000 - 10000.

The Biot parameter  $\alpha$  depends on the unit modulus of skeleton  $K_{sk}$  and grains  $K_s$  of the basic material in the form:

$$\alpha = 1 - \frac{K_{sk}}{K_s} < 1$$

- where:
- $\alpha$  - Biot's parameter
  - $K_{sk}$  - unit modulus of skeleton
  - $K_s$  - unit modulus of grains

Default setting assumes incompressibility of grains ( $K_s \gg K_{sk}$ ) and therefore  $\alpha = 1$ . The total stress then provided by:

$$\sigma_{ij} = \sigma_{ij}^{eff} + u\delta_{ij} = D_{ijkl}\varepsilon_{kl} + \alpha u\delta_{ij}$$

- where:
- $\sigma_{ij}$  - total stress tensor
  - $\sigma_{ij}^{eff}$  - effective stress tensor
  - $\varepsilon_{kl}$  - overall strain tensor
  - $D_{ijkl}$  - elastic stiffness tensor
  - $\delta_{ij}$  - Kronecker's delta
  - $u$  - active pore pressure

## Ko Procedure

$K_o$  procedure is the method that allows for the calculation of **geostatic stress** ( $I^{st}$  stage) when

particular ratio between vertical and **horizontal stress components** is needed. For example, when dealing with **overconsolidated soils** the actual horizontal stress can attain much higher values than found in normally consolidated soils.

When adopting **standard analysis** the initial stress is determined through the application of the finite element method. Nonlinear material models can be used to account for evolution of possible failure surfaces already in the 1<sup>st</sup> calculation stage. In the case of elastic response the ratio between vertical  $\sigma_z$  and horizontal  $\sigma_x$  stress components is provided by:

$$\sigma_x = \frac{\nu}{(1-\nu)} \cdot \sigma_z$$

where:  $\sigma_z$  - vertical normal stress  
 $\sigma_x$  - horizontal normal stress  
 $\nu$  - Poisson's ratio

This analysis may lead to evolution of plastic strains.

The  $K_0$  **procedure** generates only elastic response. The horizontal stress in the 1<sup>st</sup> stage of construction follows from:

$$\sigma_x = K_0 \cdot \sigma_z$$

where:  $K_0$  - coefficient of horizontal stress at rest defined by the user  
 $\sigma_z$  - vertical normal stress  
 $\sigma_x$  - horizontal normal stress

The  $K_0$  coefficient is assumed to be a soil parameter. If the  $K_0$  parameter is not assigned, it is derived from the relation:

$$K_0 = \frac{\nu}{1-\nu}$$

The resulting stresses may, however, violate the plasticity condition in the 2<sup>nd</sup> stage of construction when nonlinear material models are used. Iteration of equilibrium is then carried out even if no changes occur in the 2<sup>nd</sup> stage.

## Water Flow

The program allows for performing either the **steady state or the transient flow analysis in a soil body**. The transient flow analysis allows for monitoring the evolution of pore pressure (pressure head) and the degree of saturation in time. Time after which the distribution of pores pressure no longer changes determines the time needed to reach the steady state conditions. This value depends both on the soil flow characteristics (coefficient of permeability, parameters of models describing the retention curve - dependence of the degree of saturation or water content on the negative pressure head or suction) and type of the analyzed problem (e.g. confined/unconfined flow). In case of steady state flow analysis, the individual stages of calculation are independent from each other. In case of transient flow analysis, the solution is

performed similarly to standard stress analysis. Individual calculation stages then depend on each other. The first stage of construction stays independent and serves to set initial conditions, i.e. to assign initial pore pressures/pressure heads and degree of saturation at the onset of time dependent analysis in both a fully saturated (positive pore pressures) and a partially saturated (negative pore pressures - suction) soil. The subsequently defined stages require inputting the time duration of a given stage together with the load history (time history of hydraulic boundary conditions). The current version of the program allows us to either introduce the entire load at once at the beginning of the calculation stage or to assume that it linearly increases with time during the course of stage calculation.

In both cases (steady state/transient flow) the program describes in general the flow in an unsaturated or partially saturated medium. The flow in a fully saturated medium appears only below the ground water table. Above the ground water table (flow in a partially saturated medium) the flow is driven by a suitable material model. To analyze problems of unconfined flow the program introduces three **material models**: the **Log-linear model**, the **Gardner model** and the **van Genuchten model**. When performing the transient flow analysis we recommend adopting the van **Genuchten model**, because this model is capable of credibly representing the retention data of a soil. Since the choice of the material model influences the setting of initial conditions (initial value of degree of saturation) the program does not allow for changing material models in subsequent calculation stages. In the same spirit changing geometry in comparison to the initial stage is also not possible.

When performing the transient soil analysis it is first necessary to set in the 1st stage the initial values of the pore pressure/pressure head at time  $t = 0$ , particularly above the ground water table in the unsaturated or partially saturated soil (suction region). The program offers three options to introduce suction, either by performing the state analysis, or assuming an equilibrium distribution given by  $p = -\gamma_w z$ , where  $z$  is measured from the current location of the ground water table, or the initial values of suction can be specified directly by the user. When solving practical region we recommend not to specify values of the negative pressure head  $h_p$  smaller than  $-10m$  ( $p > -100 kPa$ ), especially in case of coarse texture soils. For example, for sands the retention curve is almost flat for the values of  $h_p < -1m$  and for large changes in pressure head there is almost no change in the degree of saturation. This holds also for the coefficient of relative permeability  $K_r$ , which serves to reduce the fully saturated permeability in the unsaturated or partially saturated zones. A general recommendation for setting minimal values of negative pressure head is, however, rather complicated since for fine texture soils these values may reach several hundred and for clays even thousand meters.

The next step calls for defining the boundary conditions, either at a **point** or along a boundary **line** at the beginning of a new calculation stage.

**Beam** and **contact** elements can be introduced inside the soil body. The **analysis results** are presented in the form of pore pressure and total head distributions, suction, velocities and directions of flow and information about the total inflow/outflow into or out of the soil body. In case of transient flow it is also possible to plot the distribution of the degree of saturation inside the soil body.

## Flow Analysis

### Transient flow

Transient flow analysis in a partially saturated medium is driven by the solution of a general Richard's equation (equation of continuity):

$$n \dot{S} + \text{div}(-K_r K_{sat} \nabla h) = 0$$

where: $n$	- material porosity
$\dot{S}$	- rate of change of degree of saturation
$K_r$	- coefficient of relative permeability
$\mathbf{K}_{sat}$	- permeability matrix of fully saturated soil
$\nabla h$	- gradient of total head

Time discretization of Richard's equation is based on a fully explicit Picard's iteration scheme [1]. This corresponds to a hybrid formulation ensuring conservation of mass. Owing to the solution of a generally nonlinear problem, the analysis is performed incrementally. Standard Newton-Raphson iteration scheme is used to satisfy equilibrium conditions.

Note that speed and stability of the iteration process is influenced to a large extent by the choice of the material model (the way of calculating the coefficient of relative permeability  $K_r$ ,

$$C = \frac{dS}{dh_p}$$

degree of saturation  $S$  and the approximation of capacity term  $C$ ) in relation to the nonlinear properties of a given soil. A significantly nonlinear behavior is for example typical of sands where improperly prescribed initial conditions may cause numerical problems. Details can be found in [2,3].

### Steady state flow

The steady state analysis assumes no change of the degree of saturation in time. The governing equation then reduces to:

$$\text{div}(-K_r \mathbf{K}_{sat} \nabla h) = 0$$

Unlike transient flow, the analysis is therefore time independent and requires introduction of the flow boundary conditions only. However, it is still a generally nonlinear problem (e.g. unconfined flow analysis) calling for the application of the Newton-Raphson iteration method. Details can be found for example in [2,3].

*Literature:*

[1] M. A. Celia and E. T. Bouloutas, *A general mass-conservative numerical solution for the unsaturated flow equation*, *Water Resources Research* 26 (1990), no. 7, 1483-1496.

[2] M. Šejnoha, *Finite element analysis in geotechnical design, to appear (2015)*

[3] M. Šejnoha, T. Janda, H. Pruška, M. Brouček, *Metoda konečných prvků v geomechanice: Teoretické základy a inženýrské aplikace, předpokládaný rok vydání (2015)*

## Interface

The frame "**Interface**" serves to input interfaces between individual soils. Detailed description how to deal with interfaces is described [herein](#).

The width of the geometrical model can be usually estimated without much of a trouble (care must be taken in the stability analysis to provide for sufficiently large space surrounding the critical region). The depth of a mesh however is quite important. The lowest point of a mesh can be imagined as incompressible subsoil. If there is no such layer of the soil or rock material in the geological profile it is possible to assume that at a certain depth from the ground the internal forces will vanish so that there will be no deformation. This will be the lowest point of the geometrical model.

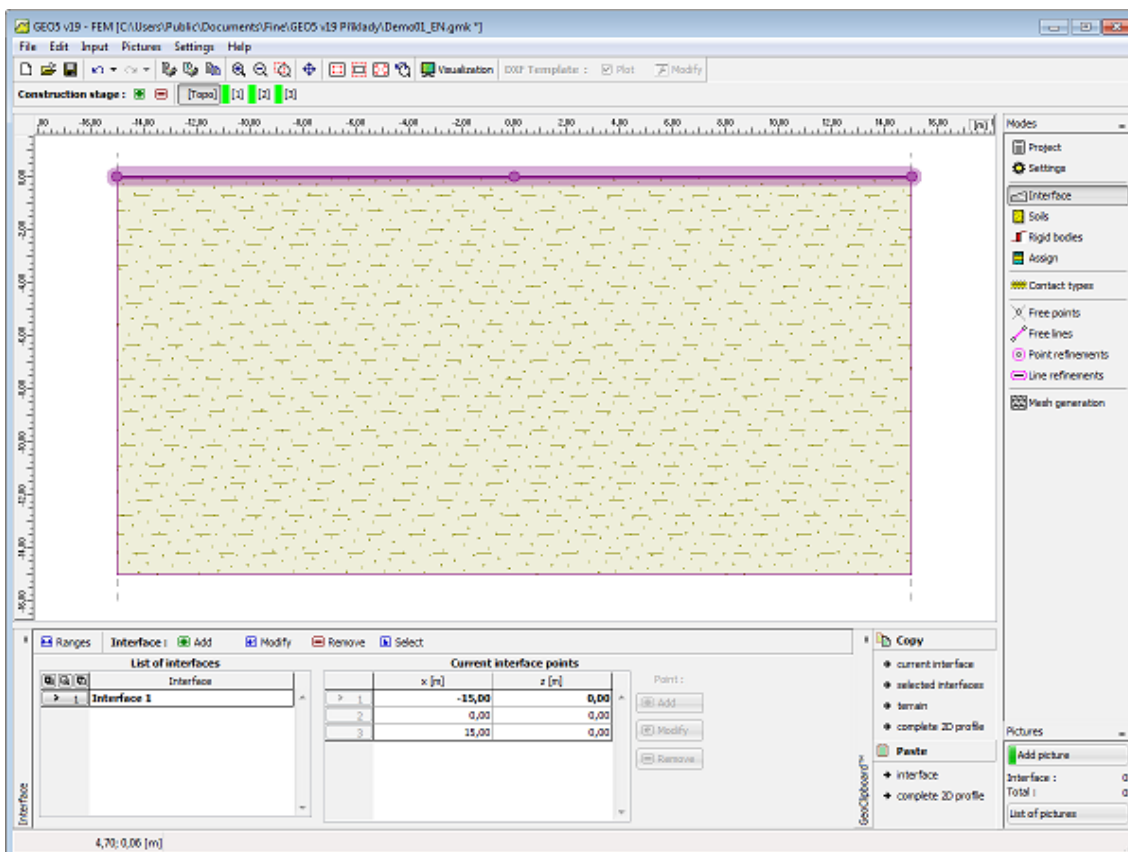
If you are not certain about the [margins of the geometrical model](#) it is useful to proceed as follows:



- First enter larger margins, use coarser mesh and compute changes in the stress distributions within a soil body.
- In the next step modify the initial margins (regions with no apparent deformation or changes in stresses can be cut off), generate new and finer mesh and carry out a new and more accurate analysis.

Interfaces can be also imported from our other programs using clipboard.

The program makes it possible to **import or export** interfaces in the \*.DXF format. They can also be **imported in the gINT format**. Input interfaces can be copied within all 2D GEO5 programs using "GeoClipboard".



Frame "Interface"

## Soils

The **frame "Soils"** contains a **table** with the list of input soils. Basic information regarding the current soil is displayed in the right part of the frame. If there are more items (soils) selected in the table, the information about individual soils is ordered consecutively.

The soil input parameters depend on the selected **material model**, or **material model in flow analysis**.

The basic material parameters are the **Young modulus of elasticity E** and the **Poisson's ratio** (they are needed in all models).

Most **nonlinear models** require defining **the angle of internal friction** and **cohesion** of the soil. The program allows for modeling either drained boundary conditions (analysis is carried out under steady state conditions after full dissipation of excess pore pressure) or undrained



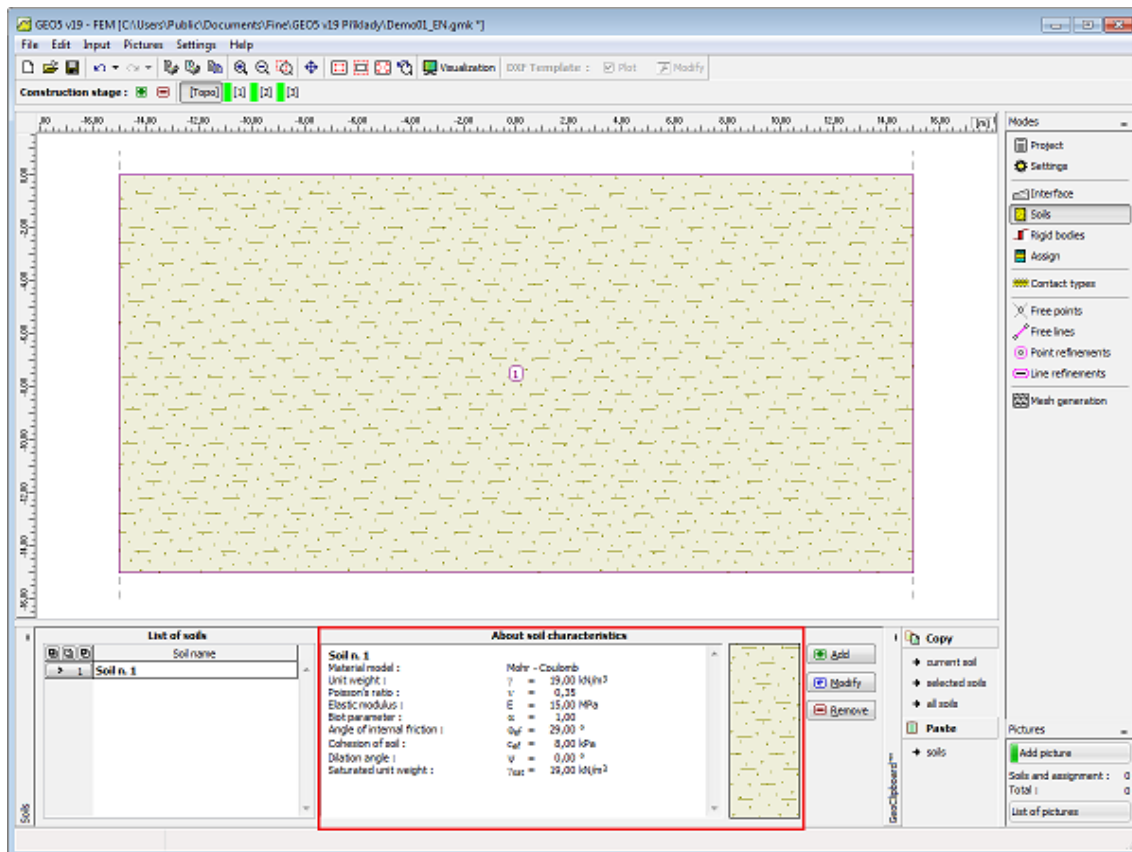
boundary conditions (the state at the onset of consolidation is represented so that the pore pressure distribution follows directly from the analysis assuming full saturation and no outflow of water). In both cases the analysis adopts **effective parameters** of the angle of internal friction  $\phi_{ef}$  and cohesion  $c_{ef}$ .

The required list of material parameters to input also depends on the selected input mode. Assuming **advanced input** (can be selected in the frame "Settings") allows us to define additional (advanced) material parameters (e.g. the Biot parameter, Effective unit modulus of water etc.). In most practical applications these material parameters are not particularly important and mostly serve to academic purposes.

Individual material models can be combined in the analysis - each soil can be assigned its own **material model**.

**Adding** (editing) a soil is performed in the "Add new soils" dialog window.

The program makes it possible to **import soils in the gINT format**. Data of input soils can be copied within all GEO5 programs using "GeoClipboard".



Frame "Soils"

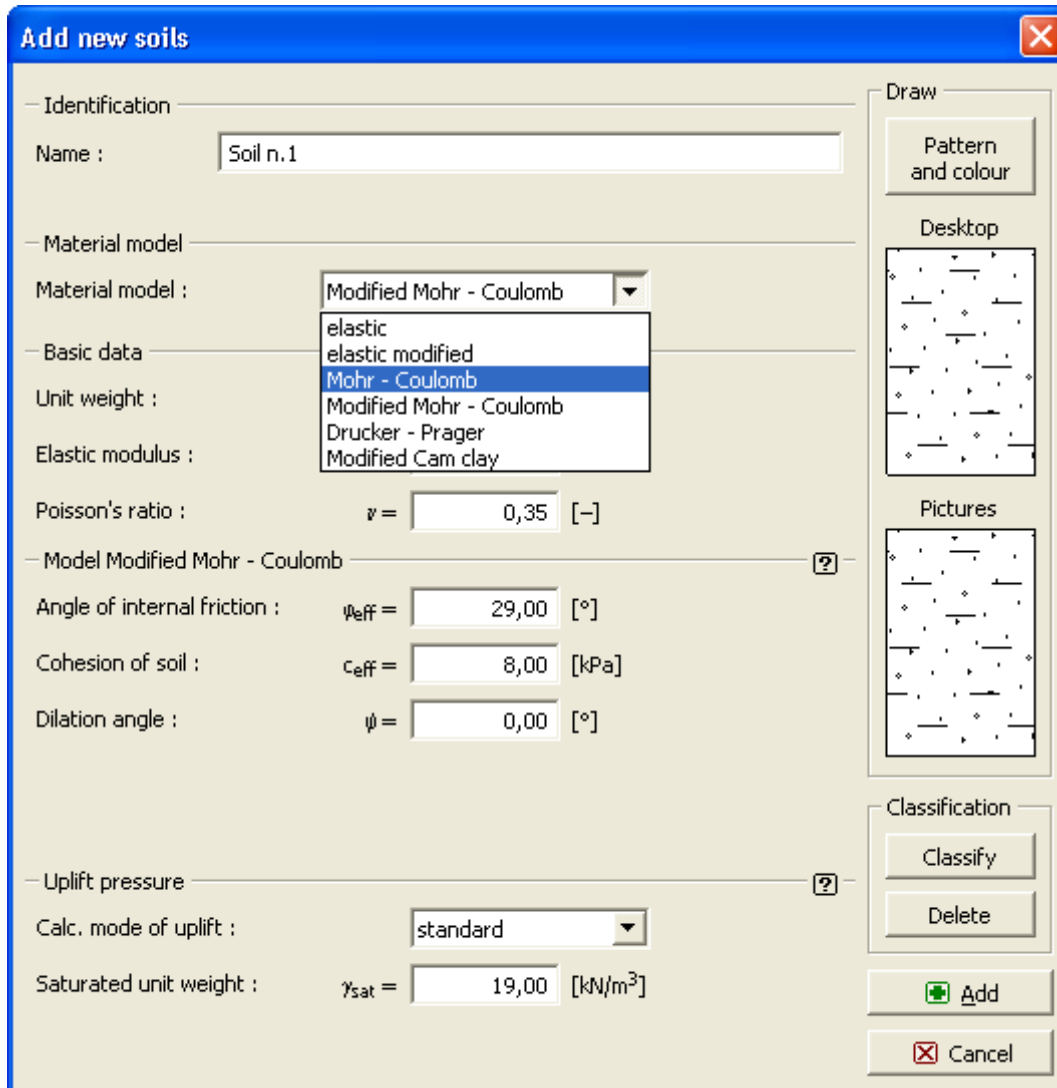
## Materials Models

Selecting the most suitable material model together with inputting the required material parameters is one of the most important but also one of the most difficult tasks when **modeling structures** using the finite element method.

The material model attempts to describe the soil (or rock) behavior as close to reality as possible. They can be divided into two basic groups - **linear** and **nonlinear** models. Selecting a

proper material model is **essential** for the prediction of a **real soil response**.

Most tasks require **nonlinear models** (e.g., modeling of sheeting structures with linear models yields totally wrong results). In some cases, however, using **linear models** may prove useful and adequate and may considerably simplify the analysis.



*Dialog window "Add new soils" - selection of a material model*

## Linear Models

Linear models give relatively fast, but not very accurate estimate of the true material response. These models can be used in cases, where only the stress or deformation states of a soil mass are of interest. They provide no information about locations and possible mechanisms of failure.

They can be used to model soil behavior in regions, where only the local failure with no effect on the evolution of global failure occurs, but which may cause premature loss of convergence. Providing the main interest is in a reliable description of the soil behavior it is necessary to employ **nonlinear models**.

The linear models include:

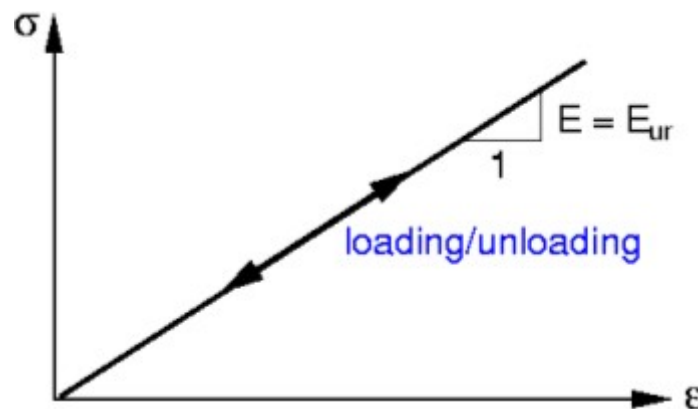
- Elastic model
- Modified elastic model

## Elastic Model

The linear model is the basic material model that assumes a linear relationship between the stress and strain given by the Hooke law. The following data are required:

- $\gamma$  - unit weight of soil  
 $\nu$  - Poisson's ratio  
 $E$  - modulus of elasticity

In a one dimensional problem the Hooke law describes the linear dependence of stress  $\sigma$  on strain  $\varepsilon$  via the Young modulus  $E$  (modulus of elasticity), see the Figure below. In this framework the linear model provides a linear variation of displacements as a function of applied loads.



Stress-strain relationship for LM

## Modified Elastic Model

It is clear that for soils the linear behavior is acceptable only for relatively low magnitudes of applied loads. This becomes evident upon unloading that usually shows a rather small amount of elastic deformation compare to the overall deformation. The modified linear model attempts at least to some extent to take this into account by considering different modulus for loading and unloading as plotted in the Figure.

A drop in the material stiffness along a given loading path attributed to the plastic yielding is reflected through a deformation modulus  $E_{def}$ , which can be imagined as a secant modulus associated with a certain stress level.

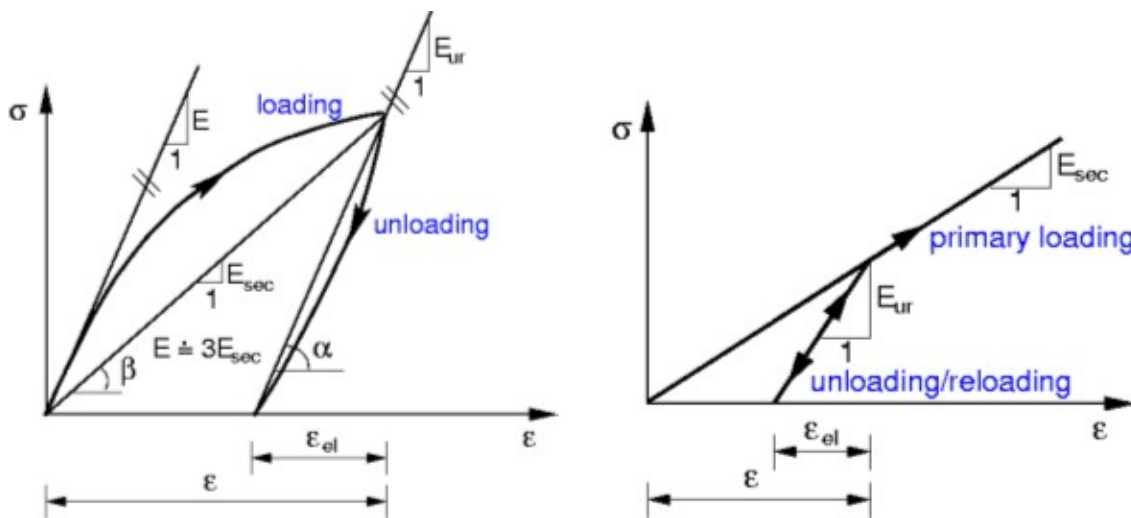
An elastic response is assumed upon unloading. To increase the clarity of model formulation the elastic modulus for the unloading branch is replaced by the unloading-reloading modulus  $E_{ur}$  that governs the response of a soil upon unloading and subsequent reloading up to the level of stress found in the material point prior to the unloading.

With reference to the following Figure these moduli are given by:

$$E_{def} = \tan \beta = \Delta \sigma / \Delta \varepsilon$$

$$E_{ur} = \tan \alpha = \Delta \sigma / \Delta \varepsilon^{el}$$

where:  $E_{def}$  -Deformation modulus [MPa]  
 $E_{ur}$  -Unloading/reloading modulus [MPa]



(a) Real stress-strain diagram of soil, (b) Simplified stress-strain diagram for MLM

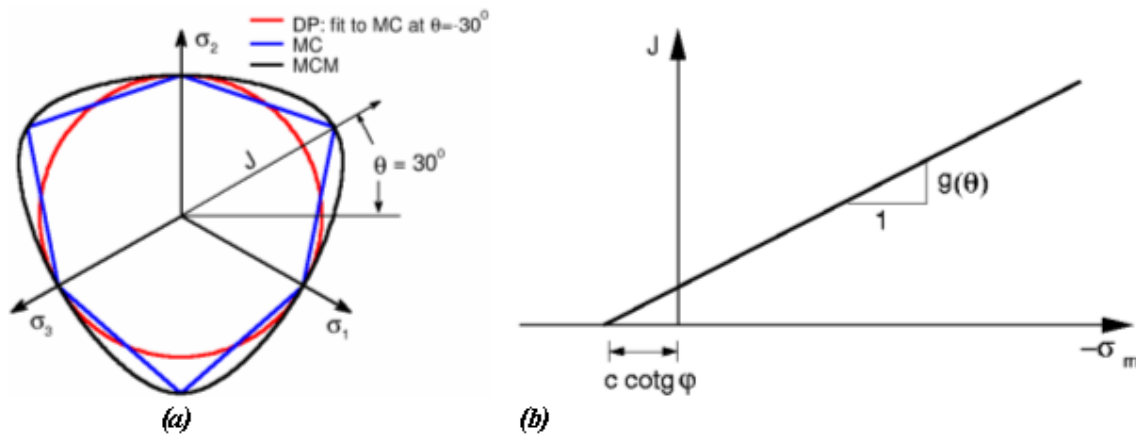
During the primary loading the response of a soil is therefore governed by the secant modulus while upon unloading it follows the path set by the *unloading-reloading modulus*  $E_{ur}$ . An approximate value of this modulus is **3\*secant modulus**  $E_{def}$ . In every case, both parameters should be obtained from reliable experimental measurements.

## Nonlinear Models

The basic nonlinear models can be again divided into two groups.

The first class of models originates from the classical Mohr-Coulomb failure criterion. In particular, the **Drucker-Prager**, **Mohr-Coulomb** and **Modified Mohr-Coulomb** models fall in this category. These models can also model the **hardening and softening**. A common feature to these models is the evolution of unbounded elastic strains when loaded along the geostatic axis. This is evident from the figure below that shows projections of the yield surfaces into deviatoric and meridian planes, respectively. An example of the effect of the selected model is given [here](#).

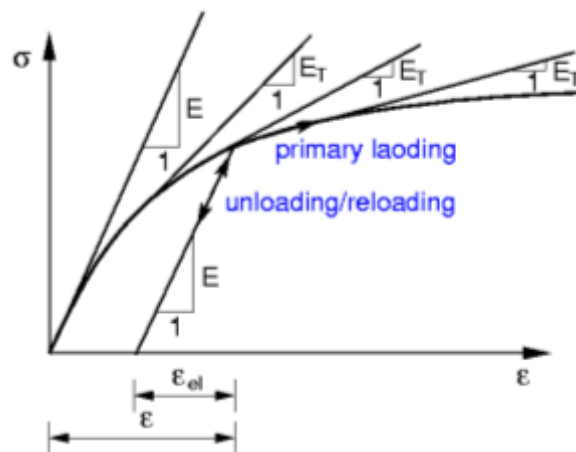
The second group of basic material models is represented by the **Modified Cam-clay**, **Generalized Cam clay** and **Hypoplastic clay** models employing the concept of the critical state of soils.



Projection of yield surfaces into (a) deviatoric, (b) meridian plane

Employing nonlinear models allows us to capture the typical nonlinear response of soils.

These models describe the evolution of permanent (plastic) deformation of a soil material. The onset of plastic deformation is controlled by so-called yield surface. The yield surface can be either constant (elastic-perfectly plastic material), or it can depend on the current state of stress (material with hardening/softening).



Stress-strain diagram for nonlinear models

Unlike the modified linear model the nonlinear models require specifying only the elastic modulus. A drop in the material stiffness is a result of evolution of plastic strains and corresponding redistribution of stresses. This consequently yields an instantaneous tangent material stiffness as a function of the current state of stress represented in the Figure below by an instantaneous tangent modulus  $E_T$ .

In addition to basic material parameters described in section "Elastic model" the nonlinear models call for the introduction of certain strength characteristics of the soil needed in the definition of a given yield surface. With reference to the first group of materials the following parameters must be specified.

- $\varphi$  - angle of internal friction [°]
- $c$  - cohesion of soil [kPa]
- $\psi$  - dilation angle [°]

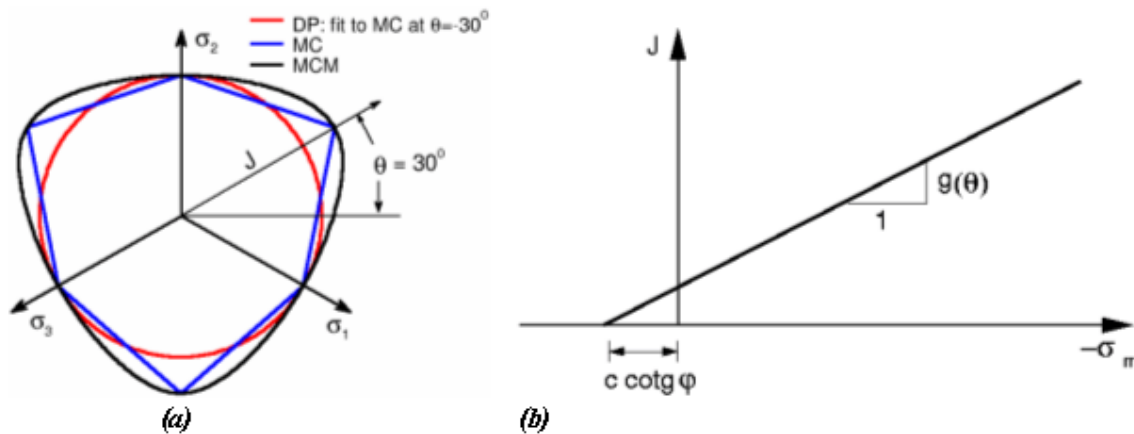
The angle of internal friction and cohesion determine the **onset of plastic deformation**. The

angle of dilation controls the evolution of plastic volumetric strain (dilation).

## Mohr-Coulomb (MC)

The model requires inputting the following parameters: modulus of elasticity  $E$ , the Poisson's ratio, angle of internal friction and cohesion. The latter two parameters serve to define the yield condition. The formulation of constitutive equations assumes effective parameters of angle of internal friction  $\varphi_{eff}$  and cohesion  $c_{eff}$ . The angle of dilation must also be specified.

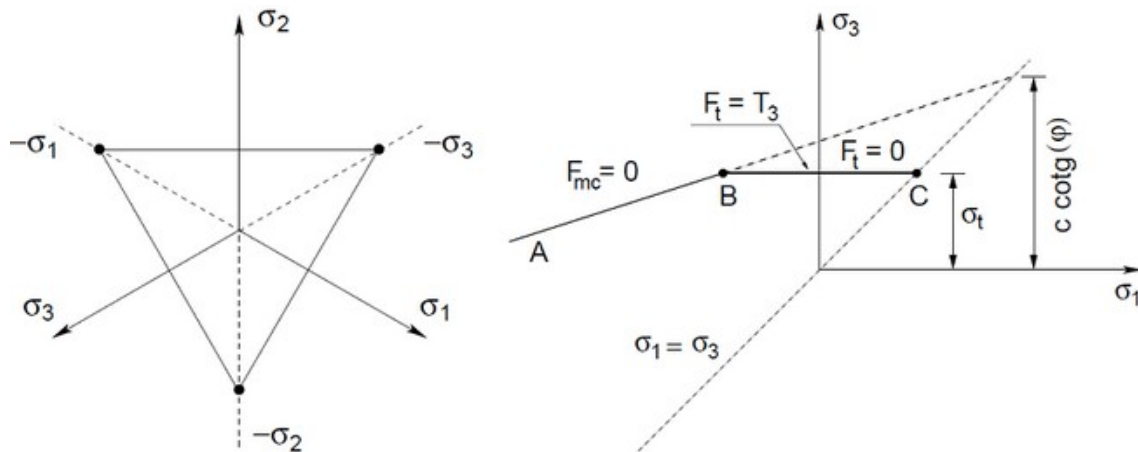
The Mohr-Coulomb yield surface can be defined in terms of three limit functions that plot as a non-uniform hexagonal cone in the principal stress space. Projections of this yield surface into deviatoric and meridian planes appear in the Figure. As evident from this Figure (part a) the MC yield function has corners, which may cause certain complications in the implementation of this model into the finite element method. The advantage on the other hand is the fact that the traditional soil mechanics and partially also the rock mechanics are based on this model.



Projection of yield surfaces into: (a) deviatoric, (b) meridian plane

## Mohr-Coulomb Model with Tension Cut Off

The original formulation of the Mohr-Coulomb material model is extended by introducing the Rankine type of plasticity condition, see Figure (a), allowing for the reduction of tensile strength of soil, which in the case of standard Mohr-Coulomb model is given by  $c \cdot \cotg \varphi$ , where  $c$  is the cohesion and  $\varphi$  the angle of internal friction. This value can be reduced by specifying the value of tensile strength  $\sigma_t$  as evident from Figure (b). If  $\sigma_t > c \cdot \cotg \varphi$  the program automatically sets  $\sigma_t = c \cdot \cotg \varphi$ . This model can be used if the advanced input option is on.

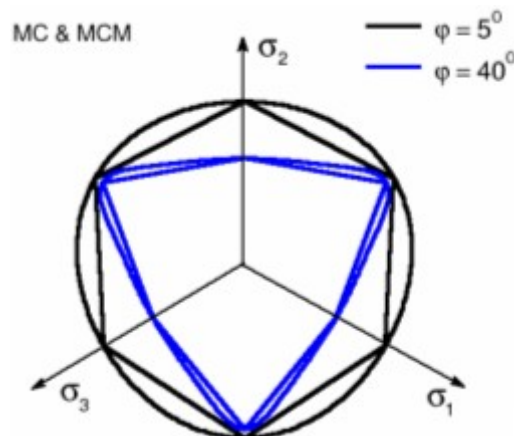


a) Projection of the Rankine yield condition in the deviatoric plane, b) Projection of the extended Mohr-Coulomb yield condition into the  $\sigma_1, \sigma_3$  plane

### Modified Mohr-Coulomb (MCM)

The model requires inputting the following parameters: **modulus of elasticity E**, the **Poisson's ratio**, **angle of internal friction** and **cohesion**. The latter two parameters serve to define the yield condition. The formulation of constitutive equations assumes **effective parameters** of angle of internal friction  $\phi_{eff}$  and cohesion  $c_{eff}$ . The **angle of dilation** must also be specified.

Similarly to the DP model the Modified Mohr-Coulomb model smoothes out the corners of the MC yield surface. As suggested in the Figure the projection of the MCM yield surface into the deviatoric plane passes through all corners of the Mohr-Coulomb hexagon and as the MC yield function the MCM yield function depends on the effective mean stress  $\sigma_m$  and the Lode angle  $\theta$ . With reference to its definition a slightly stiffer response of the material can be expected with the MCM plasticity model when compared to the MC and DP models.

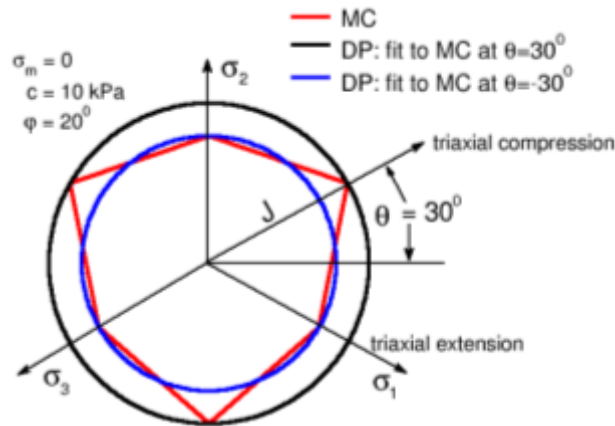


MCM and MC yield surfaces in the deviatoric plane

### Drucker-Prager

The model requires inputting the following parameters: **modulus of elasticity E**, the **Poisson's ratio**, **angle of internal friction** and **cohesion**. The latter two parameters serve to define the yield condition. The formulation of constitutive equations assumes **effective parameters** of angle of internal friction  $\phi_{eff}$  and cohesion  $c_{eff}$ . The **angle of dilation** must also be specified.

The Drucker-Prager model (sometimes also known as the extended von Mises model) modifies the Mohr-Coulomb yield function to avoid singularities associated with corners. Unlike the Mohr-Coulomb model the Drucker-Prager yield surface is smooth and plots as a cylindrical cone in the principal stress space. Similarly to the MC model the DP yield surface depends on the effective mean stress  $\sigma_m$ . The current version of the DP model implemented in FEM builds upon the assumption of triaxial extension. In other words, the yield surface projection into the deviatoric plane touches the inner corners of the Mohr-Coulomb hexagon ( $\theta = -30^\circ$ ), where  $\theta$  is the Lode angle.



DP and MC yield surfaces in the deviatoric plane

### Softening and Hardening

Standard formulation of the **Drucker-Prager** and **Modified Mohr-Coulomb** models assumes elastic rigid-plastic behavior of the soil, when the strength parameters of soil  $c$  and  $\varphi$  remain constant during the analysis. The enhanced version of both models (**advanced input** option on) allows for the evolution of these parameters as a function of the equivalent deviatoric plastic strain:

$$c = c(E_d^{pl})$$

$$\varphi = \varphi(E_d^{pl})$$

where:  $E_d^{pl}$  - equivalent deviatoric plastic strain is given by the following expressions:

$$E_d^{pl} = \sqrt{2e_{ij}^{pl}e_{ij}^{pl}}$$

$$e_{ij}^{pl} = \varepsilon_{ij}^{pl} - \frac{1}{3}\varepsilon_v^{pl}\delta_{ij}$$

$$\varepsilon_v^{pl} = \varepsilon_x^{pl} + \varepsilon_y^{pl} + \varepsilon_z^{pl}$$

where:  $E_d^{pl}$  - equivalent deviatoric plastic strain

$e_{ij}^{pl}$  - deviatoric plastic strain tensor

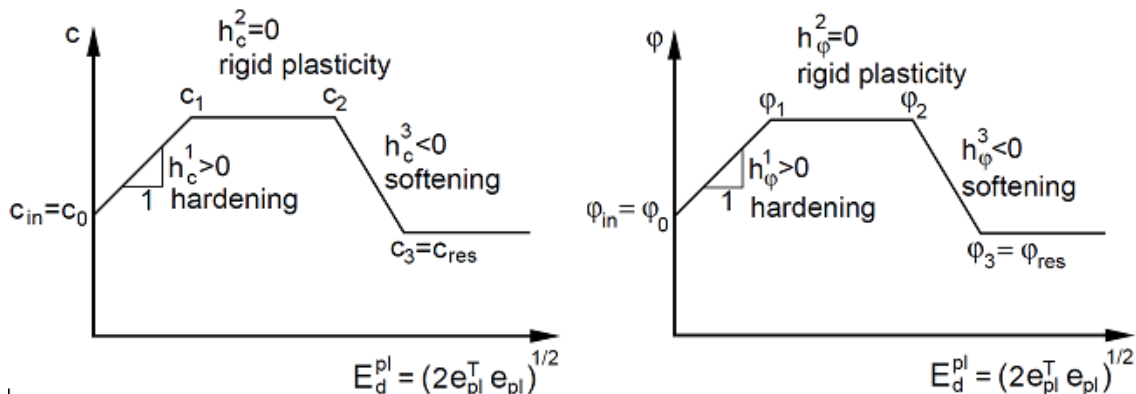
$\varepsilon_{ij}^{pl}$  - plastic strain tensor

$\varepsilon_v^{pl}$  - volumetric plastic strain

$\delta_{ij}$  - Kronecker's delta



The assumed, piecewise linear, variation of strength parameters is evident from figure.



Multi-linear hardening-softening law: dependence of a) cohesion and b) angle of internal friction on equivalent deviatoric plastic deformation  $E_d^{pl}$

Dilation angle  $\psi$  can be assumed either constant or it may evolve as a function of the angle of internal friction  $\phi$  following the Rowes dilation theory:

$$\sin \psi = \frac{\sin \phi - \sin \phi_{cv}}{1 - \sin \phi \sin \phi_{cv}}$$

where  $\phi_{cv}$  is the angle of internal friction at constant volume consistent with the critical state of soil (state at which the soil deforms at zero volumetric plastic strains). To prevent an infinite increase of the dilation angle (increase of tensile volumetric plastic strains) it must be bounded, e.g. in dependence on the maximum void ratio  $e_{max}$ , acceptable for a given material. The Rowes dilation theory requires introduction of the following parameters:

- $\phi_{cv}$  - angle of internal friction at constant volume [-]
- $e_0$  - initial void ratio
- $e_{max}$  - maximum void ratio [-]

The current void ratio  $e$  can be expressed in terms of the current volumetric strain  $\epsilon_v$  and the value of initial void ratio  $e_0$  as:

$$\lim_{\Delta V \rightarrow 0} \frac{\Delta V}{V} = \epsilon_v = \frac{e - e_0}{1 + e_0}$$

$$\epsilon_v = \epsilon_x + \epsilon_y + \epsilon_z$$

- where:
- $e$  - current void ratio
  - $e_0$  - initial void ratio
  - $\epsilon_v$  - overall volumetric strain

When the current void ratio  $e$  exceeds the maximum void ratio  $e_{max}$ , the dilation angle  $\psi$  is set to  $\theta$ .

### Angle of Dilation

The **angle of dilation** controls an amount of plastic volumetric strain developed during plastic shearing and is assumed constant during plastic yielding. The value of  $\psi = 0$  corresponds to the

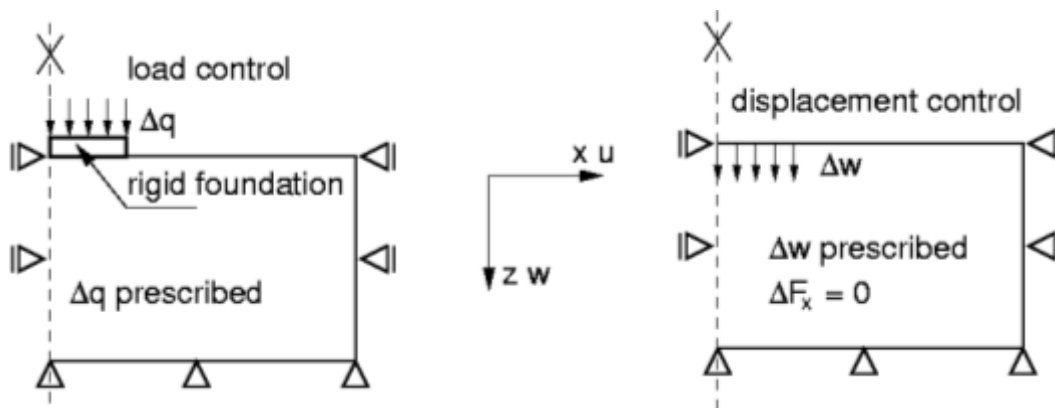
volume preserving deformation while in shear.

Clays (regardless of overconsolidated layers) are characterized by a very low amount of dilation ( $\psi \approx 0$ ). As for sands, the angle of dilation depends on the angle of internal friction. For non-cohesive soils (sand, gravel) with the angle of internal friction  $\varphi > 30^\circ$  the value of dilation angle can be estimated as  $\psi = \varphi - 30^\circ$ . A negative value of dilation angle is acceptable only for rather loose sands. In most cases, however, the assumption of  $\psi = 0$  can be adopted.

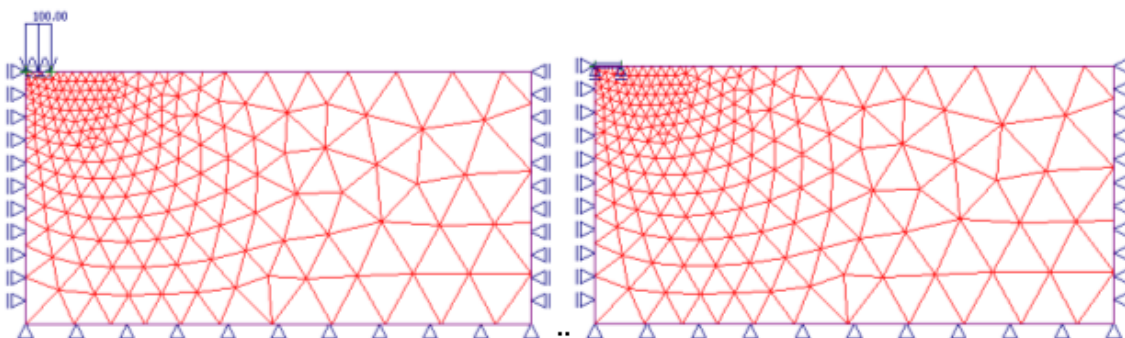
### Influence of Material Model

To illustrate the effect of a particular model used to predict a structural response we present an example of a shallow foundation loaded by the distributed load  $q$ . A certain simplification of this task is the assumption of an infinitely stiff foundation loaded by the prescribed displacements.

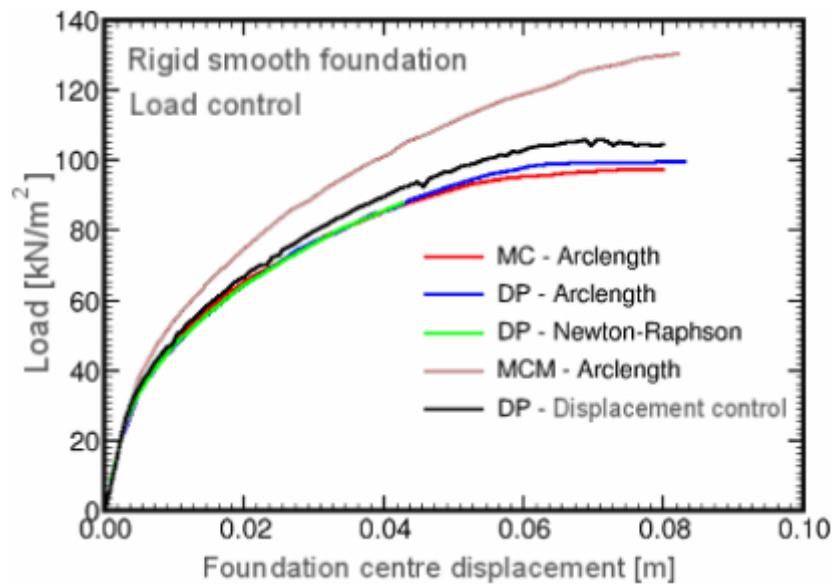
The geometrical model and finite element mesh for individual tasks appear in the Figure. The influence of soil and foundation self-weight on the resulting response is neglected. Owing to the symmetry of the model, only one half of the structure is analyzed.



Task assignment: strip foundation



Geometrical model and the finite element mesh



*Analysis results*

The results suggest a considerably stiffer response of the soil to the external load when using the MCM model in comparison to the DP and MC models, which in the present example show a similar behavior.

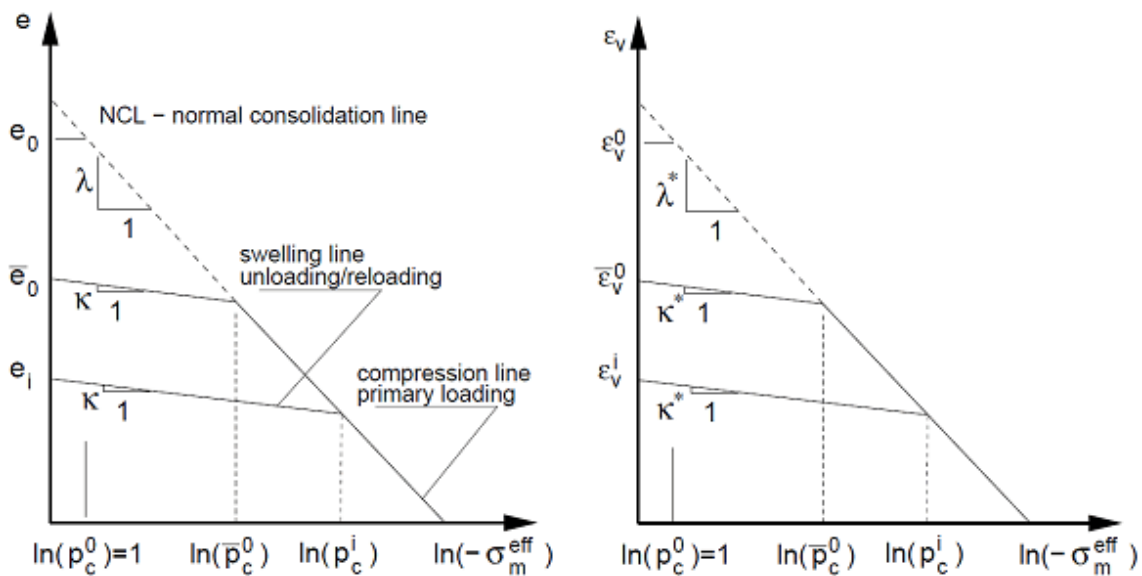
### Modified Cam-Clay Model (MCC)

The MCC model was originally developed for triaxial load conditions. Experimental measurements on soft clays provided the background for the development of the constitutive model expressing the variation of void ratio  $e$  (volumetric strain  $\varepsilon_v$ ) as a function of the logarithm of the effective mean stress  $\sigma_m^{eff}$ , as evident from the following Figure. Both graphs are related as follows:

$$\lambda^* = \frac{\lambda}{1+e}$$

$$\kappa^* = \frac{\kappa}{1+e}$$

- $\kappa$  - slope of swelling line [-]
- $\lambda$  - Slope of NCL (normal consolidation line) [-]
- $e$  - current void ratio [-]



*Material response during isotropic consolidation (constitutive model)*

The graph consists of a normal consolidation line (NCL) and a set of swelling lines. On the first load the virgin soil moves down the NCL. Next, suppose that the soil was consolidated to a certain level of stress, which is termed the preconsolidation pressure  $p_c$ , and subsequently unloaded up the current swelling line. Then, upon reloading the soil initially moves down the swelling line until reaching the stress state given by the parameter  $p_c$ , which existed prior to the unloading. At this point the soil begins to move again down the normal consolidation line (primary loading - compression line).

Parameters  $\kappa$  and  $\lambda$  can be estimated from the following expressions:

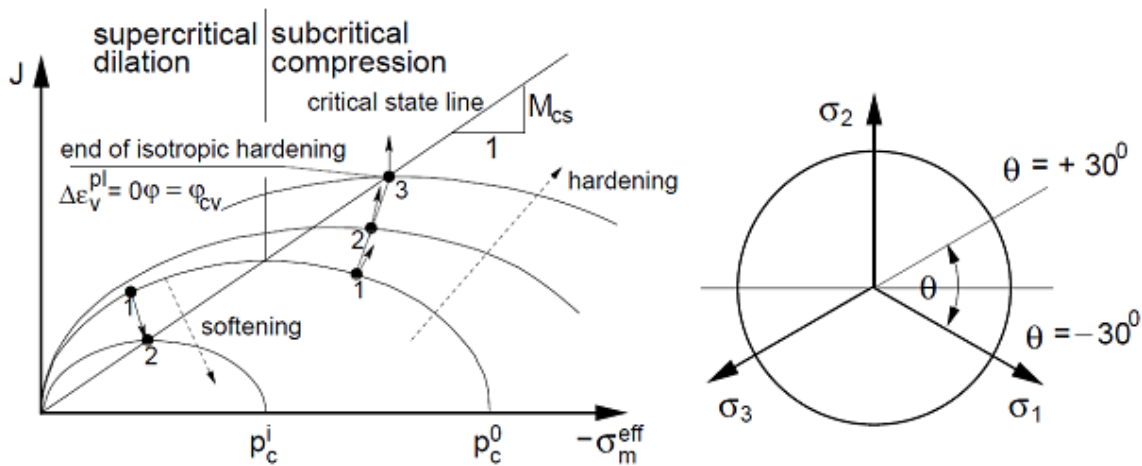
$$\lambda = \frac{C_c}{2,3}$$

$$\kappa = 1,3 \frac{1 - \nu_\epsilon}{1 + \nu} C_s$$

where:  $C_c$  one-dimensional compression index  
 $C_s$  one-dimensional swelling index

These parameters follow from a simple oedometric test.

The yield surface is smooth without the possibility of evolution of tensile stresses. The MCC model allows, unlike the first group of models, a direct modeling of strain hardening or softening for normally consolidated or overconsolidated soils, a nonlinear dependence of the volumetric strain on the effective mean stress and limit conditions of ideal plasticity. When using the MCC model the soil loaded in shear can be plastically deformed without collapse (points 1,2 for hardening, point 2 for softening) until reaching the critical state (points 3 and 2 for hardening and softening, respectively). The soil deforms further in shear under the assumption of ideal plasticity without the change of  $e$  and  $\sigma_m^{eff}$ . Upon unloading, a linear response of soil is assumed.



Projection of yield function into meridial and deviatoric planes

Evolution of the yield surface (hardening/softening) is driven by the current preconsolidation pressure  $p_c$ :

$$p_c^{i+1} = p_c^i \exp \left[ \frac{-\Delta \varepsilon_v^{pl}}{\lambda^* - \kappa^*} \right]$$

where:  $p_c^{i+1}$  - current preconsolidation pressure  
 $\Delta \varepsilon_v^{pl}$  - increment of volumetric plastic strain

Apart from parameters  $\kappa$  and  $\lambda$ , the self-weight and the Poisson's ratio, the MCC model requires specifying the following three parameters:

- $M_{cs}$  - slope of the critical state line [-]
- $OCR$  - overconsolidation ratio [-]
- $e_0$  - initial void ratio [-]

Reliable initialization of the model is described in section "Numerical implementation of MCC and GCC models".

The slope of the critical state line  $M_{cs}$  can be determined from the expression:

$$M_{cs}^{+30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3} \sin \varphi_{cv}}{3 - \sin \varphi_{cv}} \quad , \text{ for triaxial compression}$$

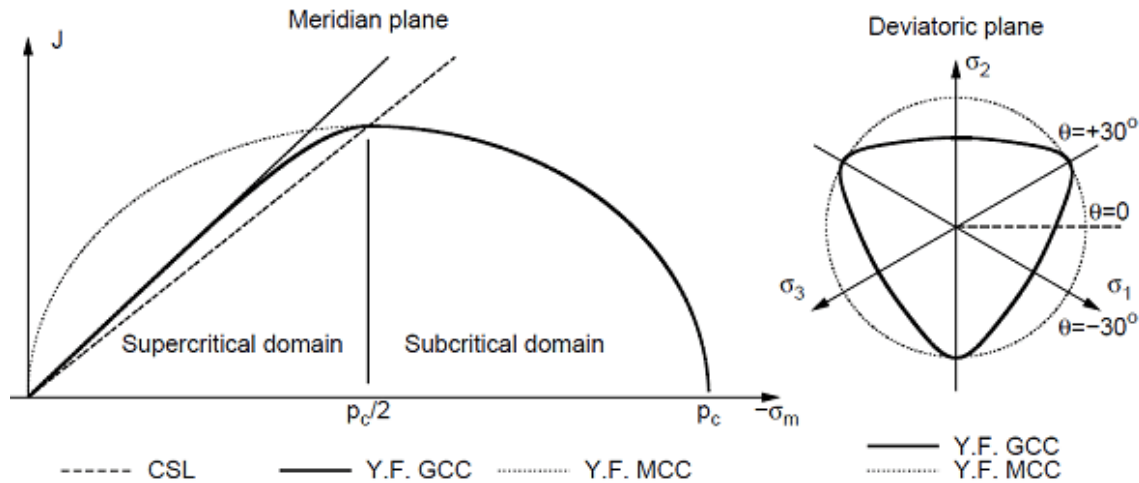
$$M_{cs}^{-30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3} \cdot \sin \varphi_{cv}}{3 + \sin \varphi_{cv}} \quad , \text{ for triaxial extension}$$

where  $\varphi_{cv}$  is the angle of internal friction for constant volume corresponding to the critical state.

### Generalized Cam-Clay Model (GCC)

This model represents a considerable improvement of the Modified Cam clay (MCC) model, particularly when modeling soils in the supercritical domain, see Figure, where the failure

surface follows the classical models of **Mohr-Coulomb**, **Drucker-Prager** and **Modified Mohr-Coulomb** models. Unlike the Modified Cam clay model (dashed line) the GCC model plots, similarly to the MMC model, as a rounded triangle in the deviatoric plane. The MCC model plots, similarly to the Drucker-Prager model, as a circle. In the subcritical domain both models behave identically. Upon unloading, a linear response of soil is assumed. GCC model is available only when **advanced input** is on. Reliable initialization of the model is described in section "**Numerical implementation of MCC and GCC models**".



*Projection of yield surface of MCC and GCC models into meridial and deviatoric planes*

The material parameters required for the Generalized Cam clay model are identical to the material data of MCC and MMC models:

- $\kappa$  - slope of swelling line
- $\lambda$  - slope of normal consolidation line (NCL)
- $e_0$  - initial void ratio
- $OCR$  - overconsolidation ratio
- $c$  - cohesion
- $\varphi$  angle of internal friction
- $\varphi_{cv}$  angle of internal friction at constant volume [-]
- $\nu$  Poisson's ratio

Parameters  $\kappa$  and  $\lambda$  can be estimated from the following expressions:

$$\lambda = \frac{C_c}{2,3}$$

$$\kappa = 1,3 \frac{1-\nu}{1+\nu} C_s$$

- where:  $C_c$  one-dimensional compression index
- $C_s$  one-dimensional swelling index

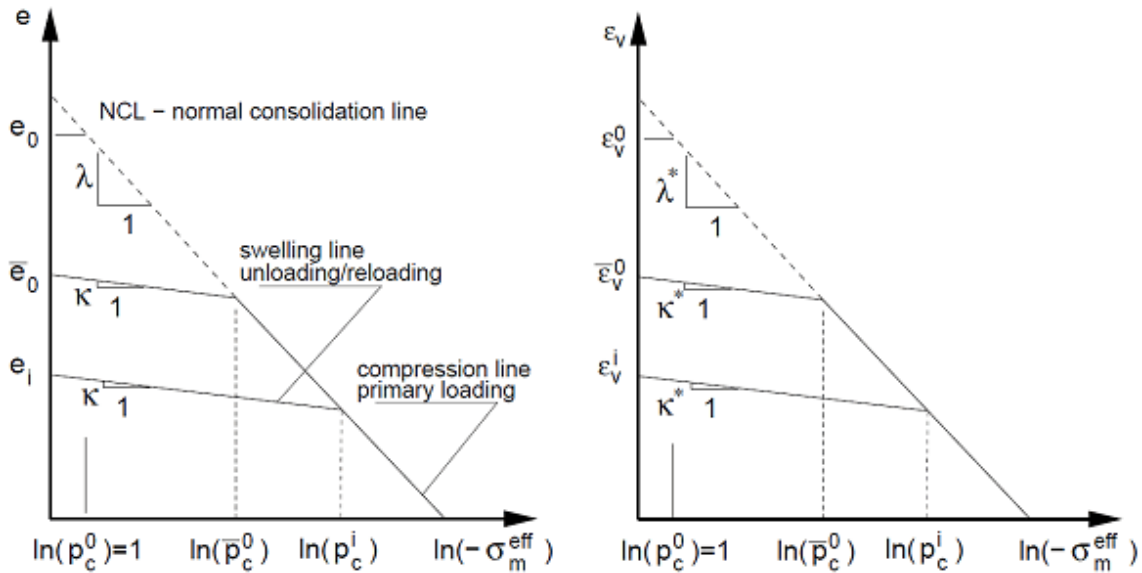
These parameters follow from a simple oedometric test.

Similarly to the MCC model the formulation of the GCC model is based on the relation between the void ratio (volumetric strain) and the mean effective stress as shown in the following

Figure. Both graphs are linked as follows:

$$\lambda^* = \frac{\lambda}{1+e}$$

$$\kappa^* = \frac{\kappa}{1+e}$$



Response of material during isotropic compression (constitutive law)

Evolution of the yield surface (hardening/softening) is driven by the current preconsolidation pressure  $p_c$

$$p_c^{i+1} = p_c^i \exp \left[ \frac{-\Delta \varepsilon_v^{pl}}{\lambda^* - \kappa^*} \right]$$

where:  $p_c^{i+1}$  - current preconsolidation pressure  
 $\Delta \varepsilon_v^{pl}$  - increment of volumetric plastic strain

### Numerical Implementation of MCC and GCC Models

An important step, ensuring a reliable application of MCC and GCC models, is the determination of the initial preconsolidation pressure  $p_c^{in}$  and the corresponding unit modulus  $K^{in}$ . These two parameters, however, are not directly specified by the user. Instead, they are derived by the program based on the assumed distribution of initial geostatic stress. Recall three basic options to derive the initial geostatic stress:

#### 1. Using $K_0$ procedure

The use of  $K_0$  procedure yields the following value of the initial mean stress:

$$\sigma_m = \frac{1}{3} \gamma h(1 + 2K_0)$$

where:  $K_0$  - coefficient of earth pressure at rest

- $\gamma$  - unit weight of soil  
 $h$  - current depth below terrain

Assuming normal consolidation, the value of  $p_c^{in}$  is determined such that the stress derived using the  $K_0$  procedure fulfils the yield condition:

$$p_c^{in} = -\frac{J^2}{M_{cs}^2 \sigma_m} - \sigma_m$$

- where:  $M_{cs}$  - slope of critical state line  
 $J$  - equivalent deviatoric stress  
 $\sigma_m$  - mean stress

Values of  $J$  and  $\sigma_m$  are defined by the following expressions:

$$S_{ij} = \sigma_{ij} - \sigma_m \delta_{ij}$$

$$\sigma_{ij} = D_{ijkl} \varepsilon_{ij}$$

$$\sigma_m = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$$

$$e_{ij} = \varepsilon_{ij} - \frac{1}{3}\varepsilon_v \delta_{ij}$$

$$S_{ij} = 2G e_{ij}$$

$$J = \sqrt{\frac{1}{2} S_{ij} S_{ij}}$$

$$J = G E_d$$

$$\sigma_m = K \varepsilon_v$$

$$e_v = \varepsilon_x + \varepsilon_y + \varepsilon_z$$

$$E_d = \sqrt{e_{ij} e_{ij}}$$

$$K = \frac{E}{3(1-2\nu)} = \frac{GE}{3(3G-E)} = \frac{2(1+\nu)}{3(1-2\nu)} G$$

- where:  $E_d$  - equivalent deviatoric stress  
 $e_{ij}$  - deviatoric strain tensor  
 $\varepsilon_{ij}$  - overall strain tensor  
 $\varepsilon_v$  - volumetric strain  
 $\sigma_{ij}$  - stress tensor  
 $s_{ij}$  - deviatoric stress tensor  
 $\delta_{ij}$  - Kronecker's delta  
 $D_{ijkl}$  - elastic stiffness tensor  
 $G$  - elastic shear modulus  
 $K$  - elastic unit modulus  
 $E$  - Young's modulus  
 $\nu$  - Poisson's ratio

In case of triaxial compression or extension it is possible to determine the slope of critical state line  $M_{cs}$  from the following expressions:



$$M_{\sigma}^{+30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3} \sin \varphi_{cv}}{3 - \sin \varphi_{cv}}$$

$$M_{\sigma}^{-30^\circ}(\varphi_{cv}) = \frac{2\sqrt{3} \sin \varphi_{cv}}{3 + \sin \varphi_{cv}}$$

In case of overconsolidated soils the initial value of  $p_c^{in}$  is modified as:

$$p_c^{in} = p_c^{in} OCR$$

The initial value of the unit modulus follows from:

$$K^{in} = -\frac{1+e}{\kappa} \sigma_m$$

where the current void ratio  $e$  is written as:

$$e = e_0 - \lambda \ln(p_c^{in}) + \kappa \ln\left(-\frac{p_c^{in}}{\sigma_m}\right)$$

For small stresses  $|\sigma_m^{in}| < 1$  we get:

$$p_c^{in} = 1$$

$$K_{in} = -\frac{1+e^0}{\kappa}$$

## 2. Standard (elastic) analysis

Recall that the program allows for replacing the material model between stages of construction. Providing the  $K_o$  procedure cannot be used it is possible to carry out the analysis assuming elastic response of the clayey soil. The resulting stresses are used to derive the initial values of  $p_c^{in}$  and  $K^{in}$  employing the previously defined expressions. In the next stages of construction the original elastic material model is replaced by the required MCC or GCC models.

## 3. Standard (plastic) analysis

This option allows the soil to be consolidated under the assumption of nonlinear behavior when generating the geostatic stress. This results in the evolution of plastic strains already in the first stage of construction. As in the  $K_o$  procedure we consider a normally consolidated soil which, during the course of deformation, moves down the normal consolidation line with the initial values of  $p_c^{in}$  and  $K^{in}$  given by:

$$p_c^{in} = 1$$

$$K_{in} = -\frac{1+e^0}{\kappa}$$

Before the next analysis step the resulting plastic strains are set equal to zero. In some cases such an analysis may fail to converge.

## Hypoplastic Clay

Hypoplastic clay is applicable for the modeling of soft fine grain soils. Similarly to all other models it belongs to the family of standard phenomenological models. As for the description of the soil response it falls into the group of critical state models (**Cam clay**, **Generalized cam clay**). This model, however, accounts for the nonlinear response of soils both in load and unloading. In comparison to other models based on the theory of plasticity, it allows for the calculation of total strains only. It thus makes no difference between elastic and plastic strains. Indication of type and location of a potential failure, in other models provided by the plot of equivalent deviatoric plastic strain, can be in the case of hypoplastic clay represented by the distribution of the mobilized angle of internal friction.

When describing the soil response, the model allows for reflecting a different stiffness in loading and unloading, softening or hardening in dependence on the soil compaction and the change of volume in shearing (dilation, compression). The current stiffness depends on only the load direction, but also on the current state of soil given by its porosity. Unlike **Cam clay** models, it strictly excludes tensile stresses in soil, see Figure 1a.

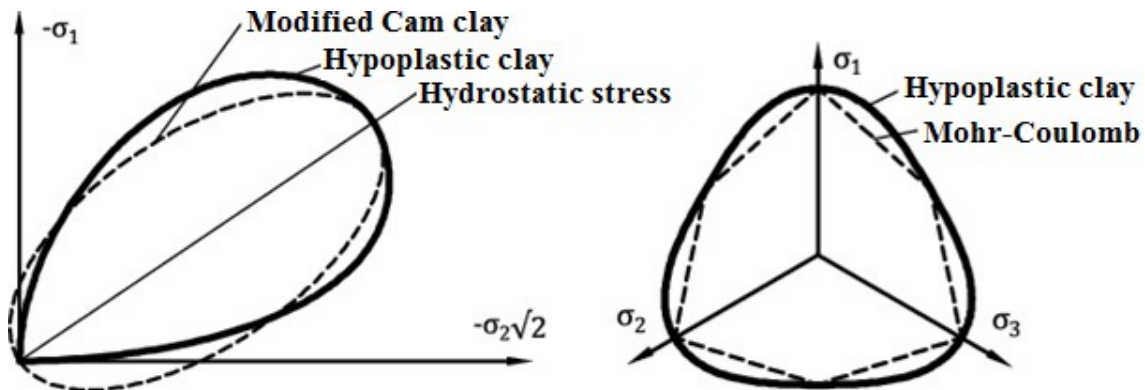


Figure 1: State boundary of hypoplastic model - (a) comparison with the yield surface of Cam clay model in the meridial plane, (b) comparison with the yield surface of Mohr-Coulomb model in the deviatoric plane

In case of hypoplastic model the standard yield surface is replaced by so called Boundary state surface. Its projection into the deviatoric plane is similar to the model, see Figure 1b. The flow rule is nonassociated resulting into a nonsymmetric stiffness matrix (compare e.g. with the **Mohr-Coulomb** model when having different values for the angle of internal friction  $\varphi$  and the dilation angle  $\psi$ ). Details regarding the model formulation can be found in [1].

### Model parameters

The basic variant of the model requires inputting five material parameters:

- Angle of internal friction for constant volume (critical angle of internal friction)  $\varphi_{cv}$
- Slope of swelling line  $\kappa^*$
- Slope of normal consolidation line (NCL - normal consolidation line)  $\lambda^*$
- Origin of the normal consolidation line  $N$
- Ratio of unit and shear modulus  $r$

Parameters  $\kappa^*$ ,  $\lambda^*$  and  $N$  determine a bilinear diagram of isotropic consolidation in a log-log scale, Figure 2a. Providing the parameters of the bilinear **Cam clay** model (in semi-logarithmic scale, Figure 2b) are available, it is possible to input these values and the parameters of the hypoplastic model are back calculated. Parameters of the bilinear **Cam clay** model are:

- Slope of swelling line  $\kappa$  (in semi-logarithmic scale)
- Slope of normal consolidation line  $\lambda$  (in semi-logarithmic scale)
- Void ratio  $e_{max}$  for normal isotropic consolidation by pressure of  $1kPa$

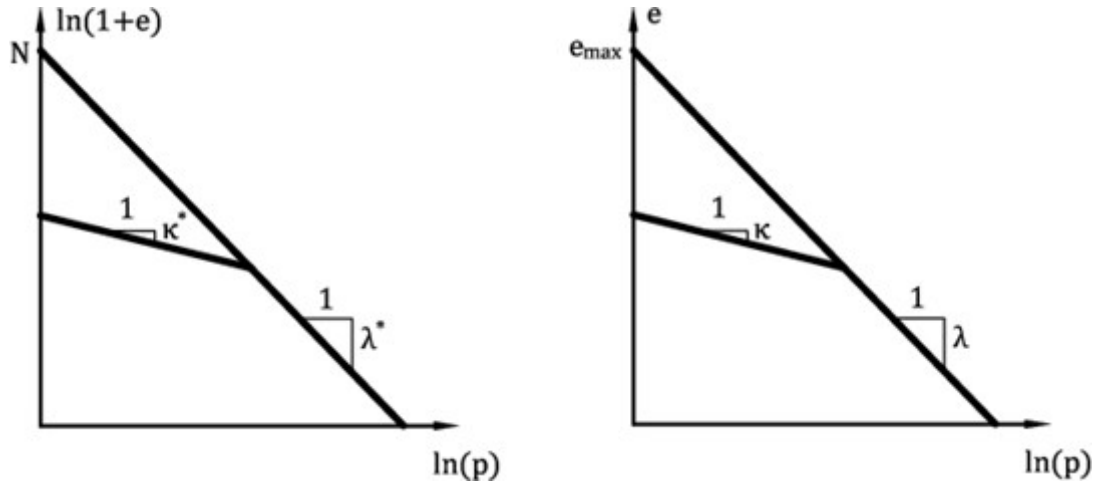


Figure 2: Bilinear diagram of isotropic consolidation - (a) Hypoplastic clay, (b) Cam clay model

#### Critical angle of internal friction $\phi_{cv}$

- Identical for both original (undisturbed) and reconstituted subsequently consolidated sample
- Can be determined from standard triaxial test applying different cell pressures on a reconstituted sample
- Both drained and undrained (faster) test can be performed
- Most common values are in range of  $18^\circ - 35^\circ$

#### Slope of normal consolidation line $\lambda^*$

- It is determined graphically from the loading branch of oedometric or isotropic consolidation test, see Figure 3
- For stiff clays it is preferable to run the test on a reconstituted sample
- Most common values are in range of 0.04 - 0.15

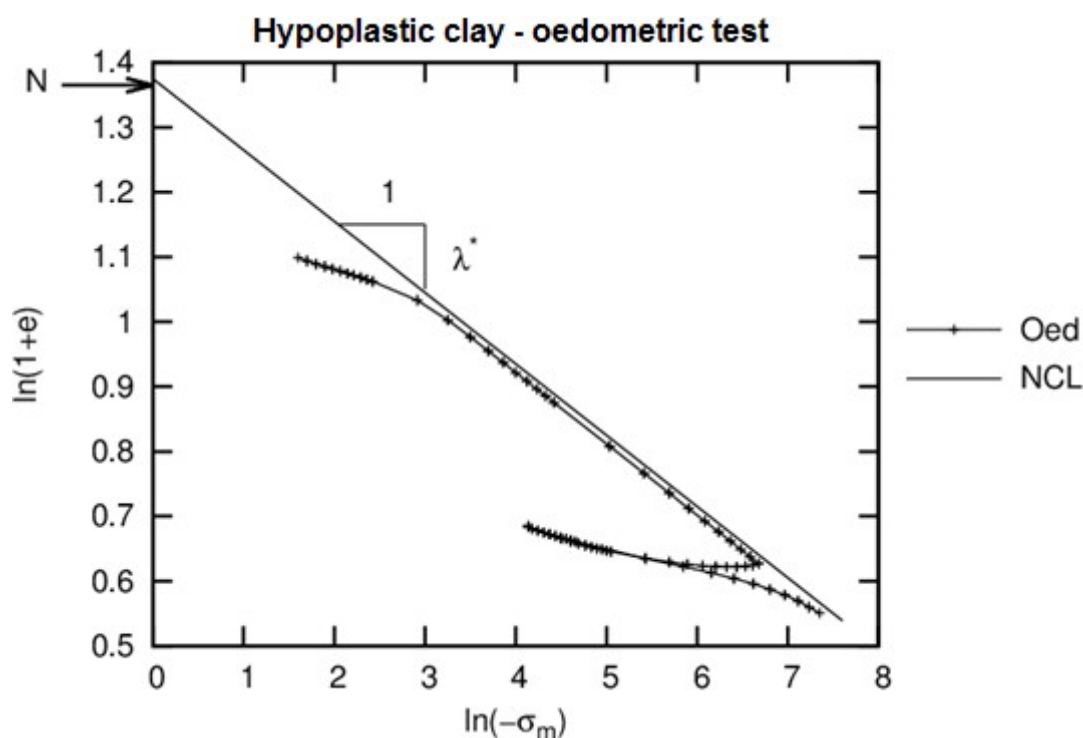


Figure. 3: Simulation of oedometric test with hypoplastic model

#### Slope of swelling line $\kappa^*$

- It can be determined similarly as parameter  $\lambda^*$  graphically or by performing a parametric study - comparing measurements and simulation along the unloading branche of oedometric or isotropic consolidation test, see Figure 3
- Most common values  $\kappa$  are in range of 0.01 - 0.02
- Ratio  $\lambda/\kappa$  should be large than 4.0

#### Origin of normal consolidation line $N$

- It is determined graphically from the loading branche of oedometric or isotropic consolidation test
- The test should be performed on an undisturbed sample - when surching for the intersection lambda line with the vertical axis it is possible to determine the slope lambda obtained from a reconstituted sample, see Figure 3
- Most common values are in range 0.8 - 1.6

#### Ratio of unit and shear modulus $r$

- The physical meaning of this parameter is given by the expressio  $r = K_i/G_i$
- $K_i$  corresponds to the tangent unit modulus from isotropic compression according to the normal consolidation line
- $G_i$  corresponds to the tangent shear modulus for undrained shear test assuming the same stress state
- Parameter  $r$  can be determined by a parametric study of shear triaxial test
- Most common values are in range 0.05 - 0.7

#### Setting initial state of soil

In hypoplastic clay the current state of soil is associated with the current compaction represented by the void ratio. Model implementation allows for inputting the initial or current void ratio either directly or it can be back calculated using the input preconsolidation pressure *OCR*. In the first case, the input value  $e_0$  corresponds to the void ratio measured on an unloaded sample extracted from a given depth. In the second case, the inputted value of  $e_{curr}$  corresponds to the void ratio of a stressed soil. In the last case, the value of *OCR* is specified. This parameter represents the ratio between the mean stress on NCL and the initial mean stress, see Figure 4b.

When initializing the task using the *K<sub>o</sub> procedure*, the initial stress state at the beginning of the second stage is assigned the current stress state. If adopting standard analysis in the first stage (the hypoplastic clay model is introduced already in the first calculation stage) where the soil is loaded by its self weight, the value of initial stress  $p_{in} = 1 \text{ kPa}$  is assumed and it holds  $e_{curr} = e_0$ . Providing a different material (e.g., elastic material is considered in the first calculation stage) is replaced by the hypoplastic clay model, the initial state of stress derived in the previous stage is adopted. Recall that when using elastic material in the first calculation stage the resulting stress state corresponds to the results provided by *K<sub>o</sub> procedure* for *K<sub>o</sub>* ( $\nu$  is the Poisson's ratio).

$$K_o = -\frac{\nu}{1-\nu}$$

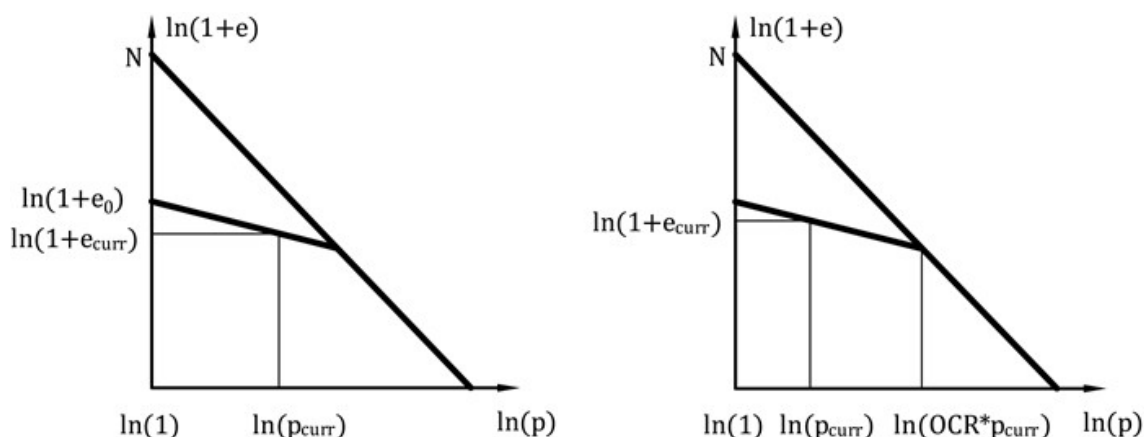


Figure 4: Initiation of void ratio - (a) with the help of initial void ratio, (b) initiation by OCR

It is clear from Figure 5 that for normally consolidated soils the state for which *OCR* = 1.0 corresponds to an isotropic consolidation only, thus for *K<sub>o</sub>* = 1.0. If the soil experiences a non-zero deviatoric stress state the corresponding *OCR* for a normally consolidated soil is greater than 1.0. An exact value of depends on both the soil parameters and stress path (the value of *K<sub>o</sub>*). Figure 5 shows the dependence of the minimum for various values of *K<sub>o</sub>* and different types of claye soils. Particular values are also stored in Table 1. The basic material parameters of this set of soils are listed in Table 2.

The choice of *OCR* = 1.0 for normally consolidated soils with *K<sub>o</sub>* not equal to 1.0 creates a non-acceptable stress state which may result in the loss of convergence.

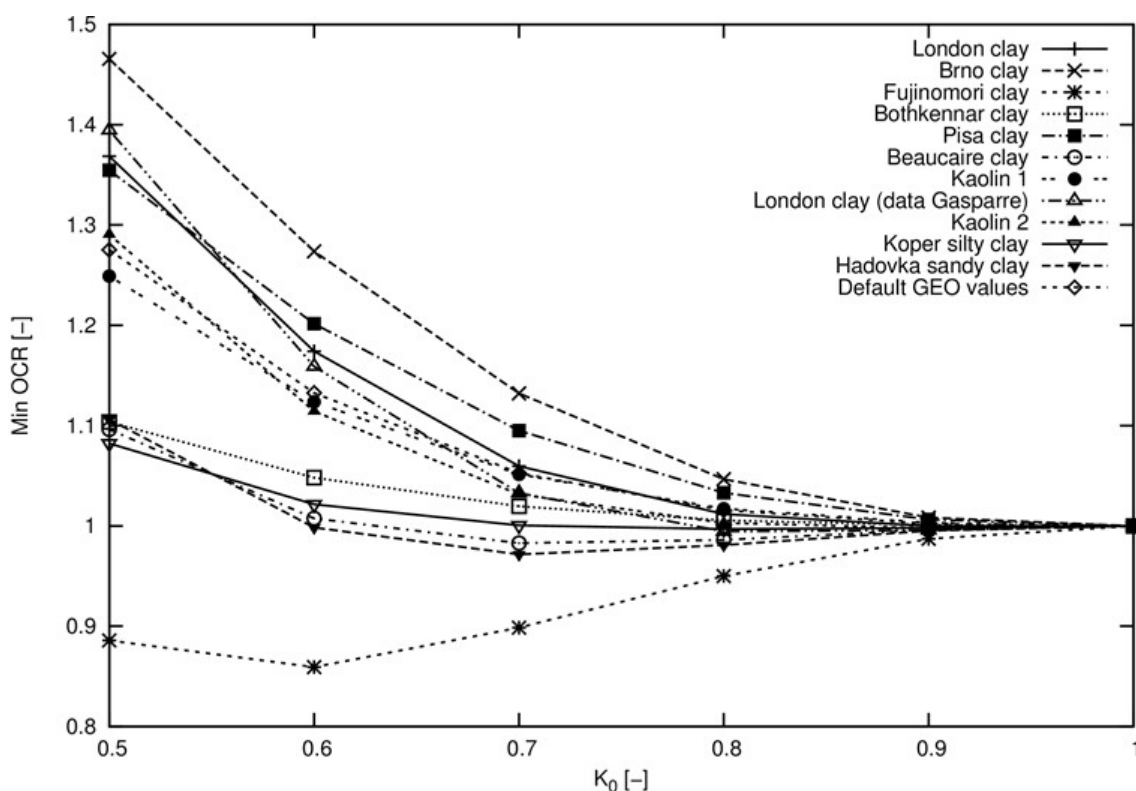


Figure 5: Dependence of OCR on the coefficient of earth pressure at rest  $K_0$

Soil/ $K_0$	0.5	0.6	0.7	0.8	0.9	1.0
	OCR					
London clay	1.369	1.174	1.059	1.011	1.000	1.0
London clay (data Gasparre)	1.394	1.159	1.033	0.994	0.995	1.0
Fujinomori clay	0.886	0.859	0.898	0.950	0.987	1.0
Bothkennar clay	1.104	1.048	1.019	1.001	1.001	1.0
Pisa clay	1.354	1.202	1.095	1.033	1.006	1.0
Beaucaire clay	1.096	1.008	0.983	0.986	0.996	1.0
Kaolin 1	1.249	1.123	1.051	1.017	1.003	1.0
Kaolin 2	1.291	1.114	1.031	1.001	0.998	1.0
Koper silty clay	1.081	1.021	1.001	0.997	0.998	1.0
Brno clay	1.466	1.274	1.132	1.047	1.008	1.0
Evropská (Hadovka) sandy clay	1.106	0.998	0.972	0.981	0.995	1.0
GEO FEM default values	1.275	1.132	1.052	1.016	1.002	1.0

Table 1: Oveconsolidation ratio OCR of the selected soils as function of  $K_0$  value

Soil	$\varphi_{cv}$	$\lambda$	$\kappa$	$N$	$r$
London clay	22.6	0.11	0.016	1.375	0.4
London clay (data Gasparre)	21.9	0.1	0.02	1.26	0.5
Fujinomori clay	34.0	0.045	0.011	0.887	1.3
Bothkennar clay	35.0	0.12	0.01	1.34	0.07
Pisa clay	21.9	0.14	0.01	1.56	0.3
Beaucaire clay	33.0	0.06	0.01	0.85	0.4
Kaolin 1	27.5	0.11	0.01	1.32	0.45
Kaolin 2	27.5	0.07	0.01	0.92	0.67
Koper silty clay	33.0	0.103	0.015	1.31	0.3
Trmice clay	18.7	0.09	0.01	1.09	0.18
Brno clay	19.9	0.13	0.01	1.51	0.45
Evropská (Hadovka) sandy clay	32.4	0.0411	0.0078	0.593	0.2
GEO FEM default values	27.0	0.1	0.01	1.2	0.4

Table 2: Material parameters of the selected soils

### Intergranular strain

The basic version of the model is suitable in analyses with a prevailing direction of the stress loading path. In cases with cyclic loading (loading-unloading-reloading) it is more suitable to use an advanced formulation with the concept of intergranular strain. This allows for constraining a unacceptable increase of permanent deformation arising during small repeating changes in load (ratcheting). Introducing intergranular strain allows for the modeling of large stiffness, which clays experience during small strains. This option is not part on any other models implemented in GEO FEM. The concept of intergranular strain assumes that the total soil deformation consists of a small deformation of an intergranular layer (intergranular strain) and deformation caused by mutual sliding of grains. Changing the load path changes first the intergranular strain. Upon reaching the limit value of the intergranular strain, the deformation associated with the motion of grains sets on.

Adopting the concept of intergranular strain requires five additional parameters:

- Range of elastic intergranular strain  $R$
- Parameters  $m_R$  and  $m_T$  control the small strain stiffness
- Parameters  $\beta_r$  and  $\chi$  control the degree of stiffness degradation with increasing shear strain

These parameters are calibrated after knowing already the material data of the basic hypoplastic model.

### Margin of elastic intergranular deformation $R$

- It determines the range of maximal intergranular strain
- It can be determined by a parametric study of the degradation curve  $G = G(\varepsilon_s)$ , Figure 5
- Alternatively it can be considered as material independent constant  $R = 10^{-4}$
- Most common values are in range  $2 \cdot 10^{-5} - 1 \cdot 10^{-4}$

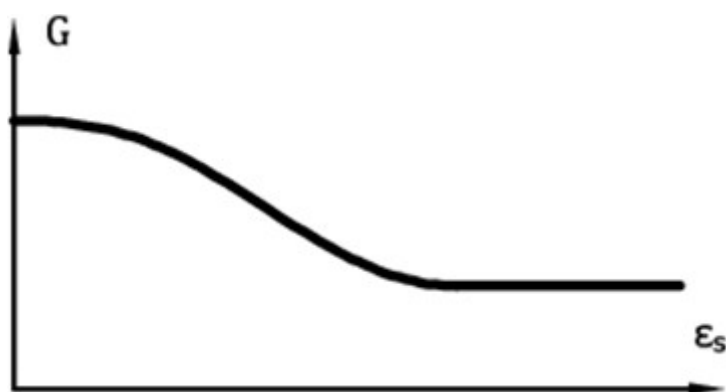


Figure 6: Curve describing the loss of stiffness of shear modulus

#### Parameter $m_R$

- It determines the magnitude of the shear modulus when changing the loading path in the meridial plane ( $\sigma_m - J$ ) o  $180^\circ$
- Linear ratio between parameter  $m_R$  and the intial shear modulus  $G_0$  is provided by  $G_0 = p^*(m_r/(r^*\lambda^*))$
- The initial shear modulus can be determined from the measurement of shear wave propagation [2]
- Most common values are in range 4.0 - 20.0

#### Parameter $m_T$

- It determines the magnitude of the shear modulus when changing the loading path in the meridial plane ( $\sigma_m - J$ ) o  $90^\circ$
- It holds  $m_R/m_T = G_0/G_{90}$
- The ratio of initial moduli can be estimated from the ratio of these moduli for larger strains. The value of the  $m_R/m_T$  ratio is commonly in the range of 1.0 - 2.0
- Most common values of  $m_T$  are in range 2.0 - 20.0

#### Parameters $\beta_r$ and $\chi$

- Determine the rate of stiffness degradation with increasing shear strain
- It can be determined by a parametric study of the degradation curve  $G = G(\varepsilon_s)$
- Most common values of parametr  $\beta_r$  are in fange 0.05 - 0.5
- Most common values of parametr  $\chi$  are in range 0.5 - 6

#### Literature:

[1] D. Mašín, A hypoplastic constitutive model for clays, *International Journal for Numerical and Analytical Methods in Geomechanics.*, 29:311-336, 2005.

## Material Models in Flow Analysis

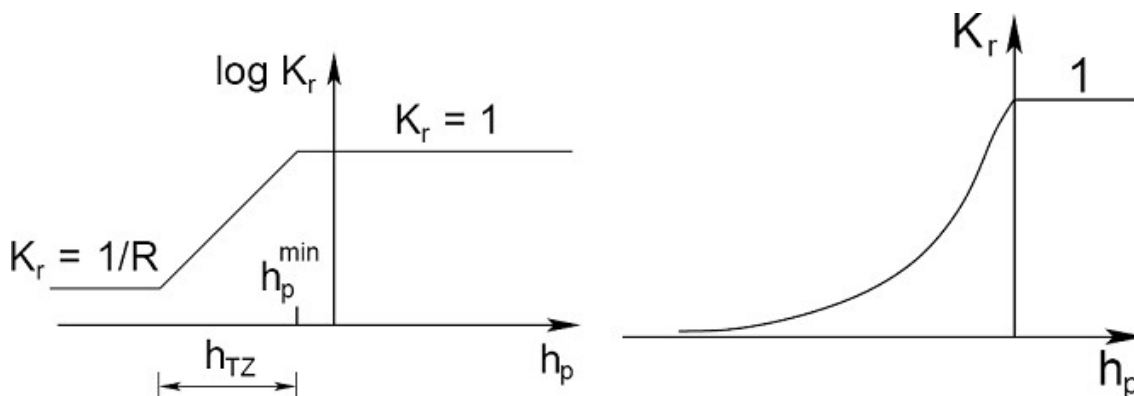
The steady state flow analysis is driven by Darcy's law specifying the relationship between the flow velocity and the gradient of hydraulic head. The current version of the program assumes constant values of **coefficients of permeability** independent of pore pressure.

The program also requires specifying the initial **void ratio**  $e_0$  for the determination of current



porosity  $n$  and subsequently the actual velocity of water flowing through pores only  $v_s = v/n$ , where  $v$  is the average flow velocity through the whole seepage area. Generally, the void ratio  $e_0 = 1$  corresponds to soil porosity of  $n = 50\%$ .

By introducing the relative coefficient of permeability  $K_r$ , the program allows for tracking the transition zone between fully saturated ( $S = 1, K_r = 1$ ) and unsaturated ( $K_r < 1$ ) region of the soil body. As an example we may consider the problem of unconfined flow. The process of tracking the transition zone is governed by one of the three models of transition zone determining the evolution of relative coefficient of permeability  $K_r$  as a function of pore pressure head, see Figures.



(a) Log-linear model [1], (b) Van Genuchten model [2]

**Log-linear model**

The Log-linear transition zone model described e.g. in [1] is defined by the following parameters:

- $h_p^{min}$  - minimum value of pressure head in fully saturated region [kPa]
- $h_{TZ}$  - transition zone width [m]
- $R$  - reduction parameter, a sufficiently large number  $R = 100$  až  $1000$  [-], default setting assumes  $R = 1000$  [-]

The relative coefficient of permeability  $K_r$  is given by:

$$K_r(h_p) = 10^{\frac{(h_p - h_p^{min}) \log R}{h_{TZ}}}$$

**Gardner model**

This is an equivalent model depending on a single parameter  $\alpha$  [1/m] only. The relative coefficient of permeability  $K_r$  is in this case given by [4]:

$$K_r(h_p) = e^{-\alpha h_p}$$

**Van Genuchten model**

In this case the value of relative coefficient of permeability  $K_r$  is given by:

$$K_r(h_p) = \frac{\left\{1 - (-\alpha h_p)^{n-1} [1 + (-\alpha h_p)^n]^{-m}\right\}^2}{[1 + (-\alpha h_p)^n]^{m/2}}$$

where  $\alpha$  [1/m],  $n$ ,  $m = 1 - 1/n$  are model parameters. Their values can be obtained from laboratory measurements of retention curves approximated by:

$$S = S_r + (S_{sat} - S_r)\Theta$$

$$\Theta = \left[ \frac{1}{1 + (\alpha h_p)^n} \right]^m$$

where:  $S_{sat}$  - degree of saturation of fully saturated soil, default setting  $S_{sat} = 1$   
 $S_r$  - residual degree of saturation  
 $\Theta$  - normalized water content

Parameter  $\Theta$  is in general provided by:

$$\Theta = \frac{\theta - \theta_r}{\theta_s - \theta_r}$$

$d$

where:  $\theta_r$  - residual water content [ $m^3/m^3$ ]  
 $\theta_s$  - water content of fully saturated soil [ $m^3/m^3$ ]

The current degree of saturation  $S$  can be expressed as a ratio of the water content  $\theta$  and porosity  $n$  as follows (it is necessary to distinguish between  $n$  representing porosity and  $n$ , which appears in the van Genuchten model, they are two different variables):

$$S_w = \frac{\theta}{n}$$

The Log-linear and Gardner models adopt a simplified version of the van Genuchten model according to [5]:

$$\Theta = K_r^b$$

where  $b > 0$  [-] is a fitting parameter allowing for a better approximation of the retention data of a given soil.

We recommend the following tables of parameters which are actually used in the program. These parameters are derived from those given in the original tables.

Optimal values of parameters of the van Genuchten model for various classifications based on USDA and FAO are presented in the following tables.

**Table with regression coefficients for grain size USDA according to Van Genuchten (1991)**

Soil (grain size)	$K_{sat}$	RETC	Rosetta
-------------------	-----------	------	---------

	[m/day]	$e$ [-]	$S_r$ [-]	$\alpha$ [1/m]	$n$ [-]	$e$ [-]	$S_r$ [-]	$\alpha$ [1/m]	$n$ [-]
Sand	7,13	0,75	0,11	14,5	2,68	0,60	0,14	3,5	3,18
Loamy sand	3,50	0,70	0,14	12,4	2,28	0,64	0,13	3,5	1,747
Sandy loam	1,06	0,70	0,16	7,5	1,89	0,63	0,10	2,7	1,448
Loam	0,25	0,75	0,18	3,6	1,56	0,66	0,15	1,1	1,474
Silt	0,06	0,85	0,07	1,6	1,37	0,96	0,10	0,7	1,677
Silt loam	0,11	0,82	0,15	2,0	1,41	0,78	0,15	0,5	1,663
Sandy clay loam	0,314	0,64	0,26	5,9	1,48	0,62	0,16	2,1	1,33
Clay loam	0,062	0,70	0,23	1,9	1,31	0,79	0,18	1,6	1,415
Silty clay loam	0,017	0,75	0,21	1,0	1,23	0,93	0,19	0,8	1,52
Sandy clay	0,029	0,61	0,26	2,7	1,23	0,63	0,30	3,3	1,207
Silty clay	0,0048	0,56	0,19	0,5	1,09	0,93	0,23	1,6	1,321
Clay	0,048	0,61	0,18	0,8	1,09	0,85	0,21	1,5	1,253

**Table with regression coefficients for grain size FAO according to Van Genuchten (1998)**

Soil (grain size)	$K_{sat}$ [m/day]	$e$ [-]	$S_r$ [-]	$\alpha$ [1/m]	$n$ [-]
<b>Top soil (up to depth 1 m)</b>					
Coarse (C)	0,600	0,68	0,062	3,83	1,3774
Medium (M)	0,121	0,78	0,023	3,14	1,1804
Medium fine (MF)	0,023	0,75	0,023	0,83	1,2539
Fine (F)	0,248	1,08	0,019	3,67	1,0120
Very fine (VF)	0,150	0,78	0,016	2,65	1,1033
<b>Soil at depth (&gt; 1 m)</b>					
Coarse (C)	0,700	0,58	0,068	4,30	1,5206
Medium (M)	0,108	0,65	0,026	2,49	1,1689
Medium fine (MF)	0,040	0,70	0,024	0,82	1,2179

Fine (F)	0,085	0,93	0,021	1,98	1,0861
Very fine (VF)	0,082	1,17	0,019	1,68	1,0730

**Table: FAO texture classification system**

Soil	Definition
Coarse (C)	clay < 18% and sand > 65%
Medium (M)	18% < clay < 35% a 15% < sand nebo: clay < 18% a 15% < sand < 65%
Medium fine (MF)	clay < 35% a sand < 15%
Fine (F)	35% < clay < 60%
Very fine (VF)	60% < clay

*Literature:*

Details can be found in [2].

[1] D.M. Potts, L. Zdravkovič, *Finite element analysis in geotechnical engineering - theory*, Thomas Telford, London, 1999.

[2] M. Th. Van Genuchten, *A closed formulation for predicting the hydraulic conductivity of unsaturated soils*, *Journal Soil Science Society of America* **44**, 239-259, 1988..

[3] M. Šejnoha, *Finite element analysis in geotechnical design*, to appear (2015)

[4] W. R. Gardner, *Some steady-state solutions of the unsaturated moisture flow equation to evaporation from a water table*, *Soil Science* **85(4)**, 228-232, 1958.

[5] M. Šejnoha, T. Janda, H. Pruška, M. Brouček, *Metoda konečných prvků v geomechanice: Teoretické základy a inženýrské aplikace, předpokládaný rok vydání (2015)*

[6] USDA 1951. *Soil Survey Manual. Soil Conservation Service. U.S. Department of Agriculture Handbook No. 18. US Government Printing Office. Washington DC.*

[7] Wösten, J.H.M., et. al. 1998. *Using existing soil data to derive hydraulic parameters for simulation models in environmental studies and in land use planning. Final Report on the European Union Funded project. DLO Winand Staring Centre. Report 156, Wageningen, NL. p. 106. ISSN 0927-04537.*

## Coefficient of Permeability

Ability of porous body (soils, rocks) to transport water of given properties (e.g. ground water) is denoted as seepage. The amount of water flowing through a certain area can be represented by the **coefficient of permeability**. The coefficient of permeability represents the slope of a linear dependence of water flow velocity on the gradient of total head (gradient of hydraulic head) in Darcy's law written as:

$$\mathbf{v} = n\mathbf{v}_s = -K_r \mathbf{K}_{sat} \nabla h$$

where:  $\mathbf{v}_s$  - velocity of water flowing through pores

$n$	porosity
$K_r$	relative coefficient of permeability
$\mathbf{K}_{sat}$	permeability matrix storing coefficients of permeability of fully saturated soil $k_x, k_y$ , which may be different along individual coordinate axes
$\nabla h$	gradient of total head

Total head at a given point of region of flow is defined as a sum of the pressure head and vertical coordinate and as such it determines the height of water in piezometer at a given point:

$$h = \frac{P}{\gamma_w} + z$$

where:  $\gamma_w$  - the weight of water

#### Example values of coefficients of permeability for various soils (Myslivec)

Type of soil	Coefficient of permeability $k$ [m/day]	Motion of water particle by $l$ cm for hydraulic gradient $i = 1$ per time
Soft sand	$10^2 - 10$	6 s - 10 min
Clayey sand	$10^{-1} - 10^{-2}$	100 min - 18 hrs
Loess loam	$10^{-2} - 10^{-4}$	18 hrs - 70 days
Loam	$10^{-4} - 10^{-5}$	70 days - 2 years
Clayey soil	$10^{-5} - 10^{-6}$	2 years - 20 years
Clay	$10^{-6} - 10^{-7}$	20 years - 200 years

There are several ways for determining the coefficient of permeability  $k$ . They grouped as follows:

#### a) Laboratory measurements

Several types are available for the range of  $k$   $10^4 - 10^{-6}$  m/day.

#### b) Field measurements

Dwell or sink tests, measurement of filtration velocity of flow, for the range of  $k$   $10^6 - 1$  m/day.

#### c) Using empirical expressions

$$k = 100 d_{10}^2 e^2$$

Suitable for non-cohesive soils,  $k$   $10^6 - 10$  m/day, they produce only guidance values - e.g. according Terzaghi:

where:  $k$  - coefficient of permeability [ $cm/s$ ]  
 $d_{10}$  - diameter of effective solid particle [ $cm$ ]  
 $e$  - void ratio [-]

#### d) By calculation from time dependent consolidation process

One must know the coefficient of consolidation  $c_v$  and consolidation curve (semi-logarithmic dependence of deformation on time). This is only an indirect determination from the expression:

$$k = \frac{c_v \rho_w g a_v}{1 + e_0}$$

where:  $e_0$  - initial void ratio  
 $c_v$  - coefficient of consolidation  
 $\rho_w$  - unit density of water  
 $g$  - gravitational acceleration  
 $a_v$  - coefficient of compressibility

## Basic Data

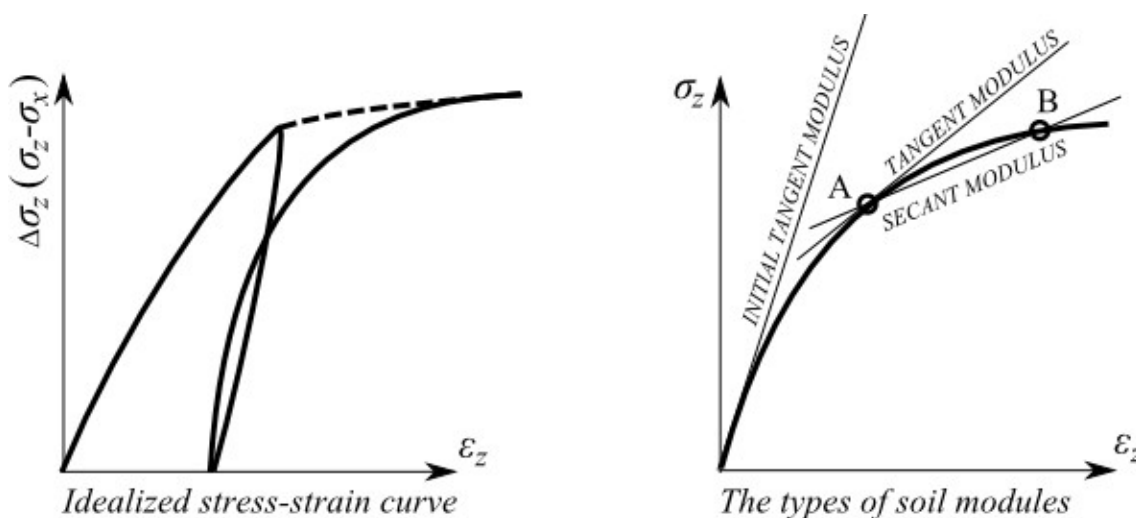
The following material parameters are required for all **material models**.

**Unit weight  $\gamma$**  - the unit weight of a dry soil (soil above the ground water table, GWT) is assumed. The unit weight of a soil below the GWT is calculated from other parameters introduced in section "Uplift".

### Modulus of elasticity $E$

The modulus of elasticity describes the material stiffness that is assumed constant over the entire load interval. In case of soils this assumption is, however, valid only for a very narrow interval of recoverable deformations. Modulus of elasticity  $E$  has no significant effect on soil behavior for **nonlinear models** after satisfying plasticity condition.

A straightforward answer to what definition and what value of this material parameter (initial, tangent, secant) one should use in a given material model is, unfortunately, not available. To select a given type of modulus one needs know the **soil behavior** in the analyzed geotechnical task and to assign a particular magnitude the results from a **triaxial test** for corresponding stress paths are necessary. Nevertheless, certain recommendations can be provided.



Distribution of idealized stress-strain curve and determination of individual types of soil modules

The following interpretation of Young’s modulus  $E$  of elasticity is available:

- **instantaneous modulus**  $E_0$  in case of small loads (assumption of linear dependence of strain and stress) or when instantaneous settlement is calculated
- **secant modulus**  $E_{50}$  is determined for a reference stress equal to 50% of stress at the onset of failure (used for example when analyzing spread foundations and settlement of pile foundations)
- **deformation modulus**  $E_{def}$  is determined from a stress-strain curve derived experimentally. This modulus is required when using the modified elastic model, which it assumes different behavior for load and unloading. Using this modulus when solving the problem of soil unloading (e.g., underground structures, heaving of bottom a foundation ditch) leads to larger deformations than when using the elastic modulus  $E_u$  determined from unloading branch  $r$  of the stress-strain curve. Determining of deformation modulus of soil is applicable by following approximate relation:

$$E_{ur} = 3E_{def}$$

- **oedometric modulus**  $E_{oed}$  which depends on the level of stress in the soil should be used depending on the expected range of stress in the soil may experience. The relation between  $E_{def}$  and  $E_{oed}$  is provided by:

$$E_{oed} = \frac{E_{def}}{\beta}$$

$$\beta = 1 - \frac{2\nu^2}{1-\nu}$$

where:  $\nu$  - Poisson's ratio  
 $E_{def}$  - deformation (secant) modulus

- **modulus of elasticity**  $E_{ur}$  determined from the unloading branch of stress-strain curve is used when solving the problem of soil unloading (excavations) - must be defined when using the **modified elastic model**

The values of modules of elasticity should be determined, if possible, from a triaxial shear test. If other methods (penetration tests, pressiometric tests etc.) are used then it becomes necessary to introduce some correlation coefficients are described in literature.

For actual modeling we recommend to perform an analysis according to the **elastic material model** at first and check the resulting strain field - such strains according to Hookes's law are linearly dependent on the applied load and the used elastic modulus. If the resulting strains (displacements) are already **too large** the user should **reassess the magnitude of the originally applied elastic modulus**.

**Poisson's ratio**  $\nu$  - coefficient of transverse contraction is in the case elastic homogeneous material loaded by normal stress in one direction given by:

$$\nu = \frac{\varepsilon_y}{\varepsilon_x}$$

where:  $\varepsilon_y$  - vertical strain  
 $\varepsilon_x$  - horizontal strain

The Poisson's ratio is relatively easy to determine. To select its value one may take advantage of the built-in soil database. If small loads are assumed and the instantaneous modulus  $E_0$  is used, then also the value of the Poisson's ratio  $\nu_0$  determined for the initial loading should be employed.

## Geostatic Stress, Uplift Pressure

Stress analysis is based on existence of soil layers specified by the user during input. The program further inserts fictitious layers at the locations where the stress and lateral pressure (GWT, points of construction, etc.) change. The normal stress in the  $i^{th}$  layer is computed according to:

$$\sigma_i = \sum h_i \gamma_i$$

where:  $h_i$  - thickness of the  $i^{th}$  layer  
 $\gamma_i$  - unit weight of soil

If the layer is found below the **ground water table**, the unit weight of soil below the water table is specified with the help of input parameters of the soil as follows:

- for option "**Standard**" from expression:

$$\gamma_{su} = \gamma_{sat} - \gamma_w$$

where:  $\gamma_{sat}$  - saturated unit weight of soil  
 $\gamma_w$  - unit weight of water

- for option "**Compute from porosity**" from expression:

$$\gamma_{su} = (1 - n)(\gamma_s - \gamma_w)$$

where:  $n$  - porosity  
 $\gamma_s$  - specific weight of soil  
 $\gamma_w$  - unit weight of water



$$\gamma_s = \frac{G_d}{V - V_p}$$

where:  $V$  - volume of soil  
 $V_p$  - volume of voids  
 $G_d$  - weight of dry soil

Unit weight of water is assumed in the program equal to  $10 \text{ kN/m}^3$  or  $0,00625 \text{ ksi}$ .

Assuming inclined ground behind the structure ( $\beta \neq 0$ ) and layered subsoil the angle  $\beta$ , when computing the coefficient of earth pressure  $K$ , is reduced in the  $i^{\text{th}}$  layer using the following expression:

$$\text{tg}\beta_i = \frac{\gamma}{\gamma_i} \text{tg}\beta$$

where:  $\gamma$  - unit weight of the soil in the first layer under ground  
 $\gamma_i$  - unit weight of the soil in the  $i^{\text{th}}$  layer under ground  
 $\beta$  - slope inclination behind the structure

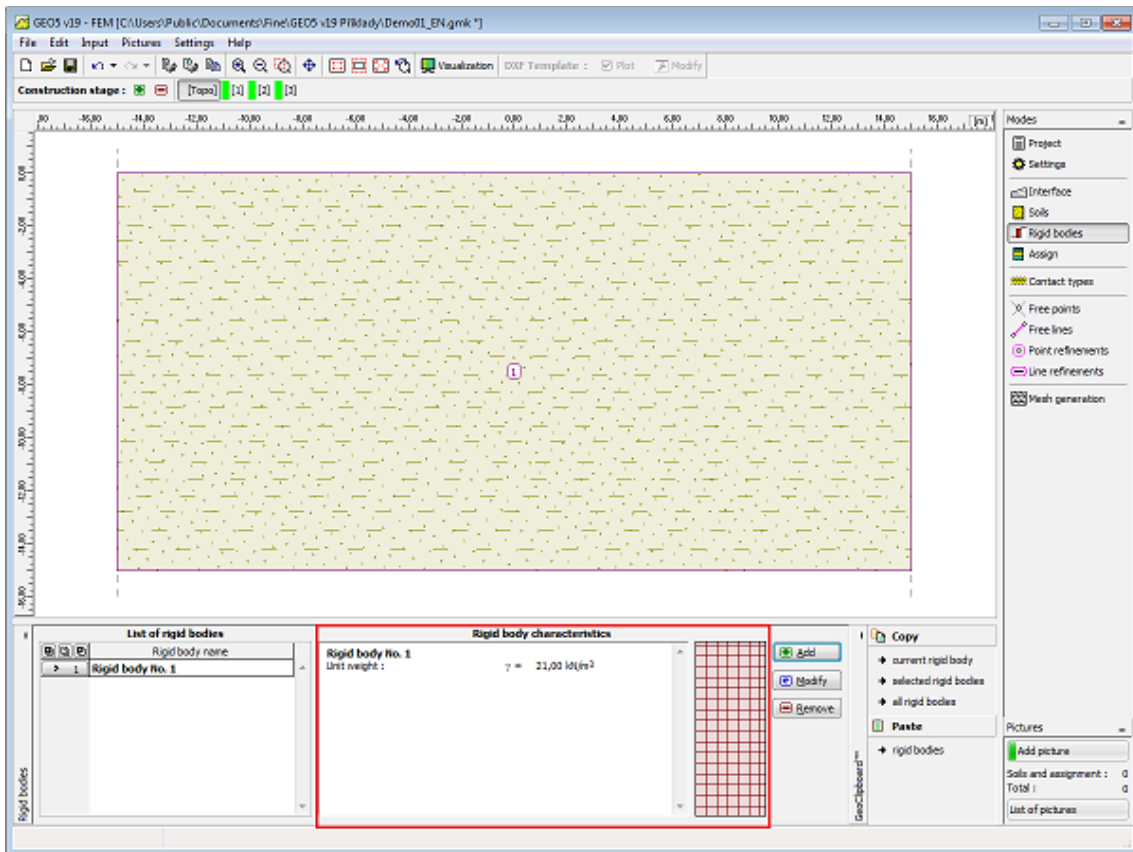
## Rigid Bodies

The **frame "Rigid bodies"** contains a **table** with the list of input rigid bodies.

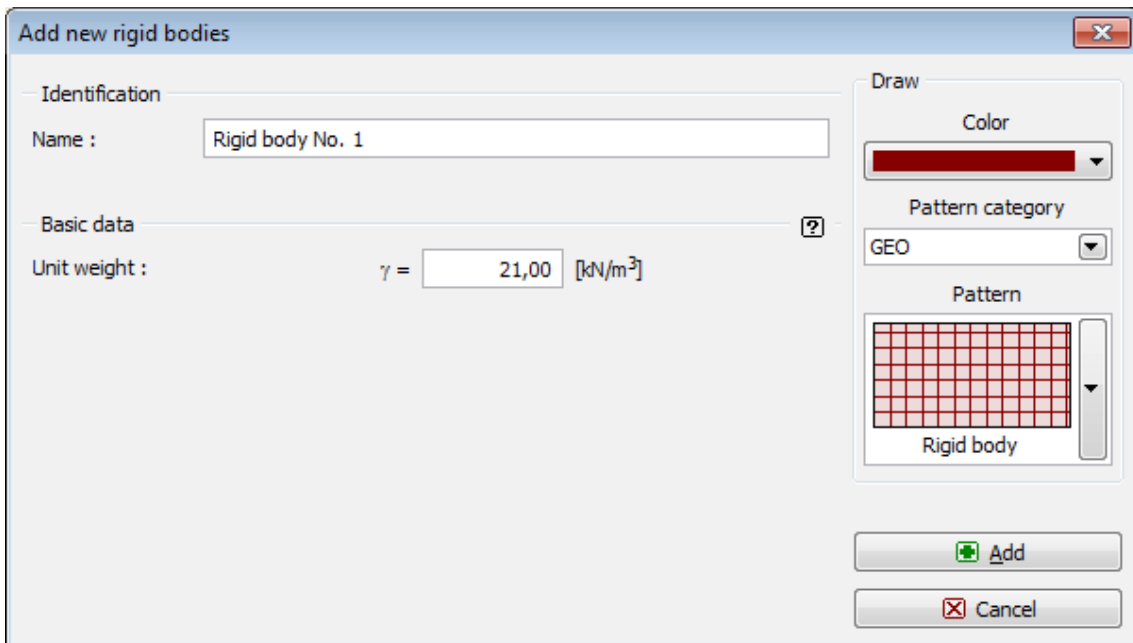
The program allows for adding the rigid bodies. Here the only required input parameter is the unit weight of the rigid body. The material of the rigid body is assumed an **infinitely stiff**. These bodies serve mainly to model massive concrete structures and walls in both standard and stability analyses.

Adding (editing) rigid bodies is performed in the **dialog window "Add new rigid body"**.

Input rigid bodies can be copied within all 2D GEO5 programs using **"GeoClipboard"**.



Frame "Rigid bodies"



Dialog window "Add new rigid body"

## Assign

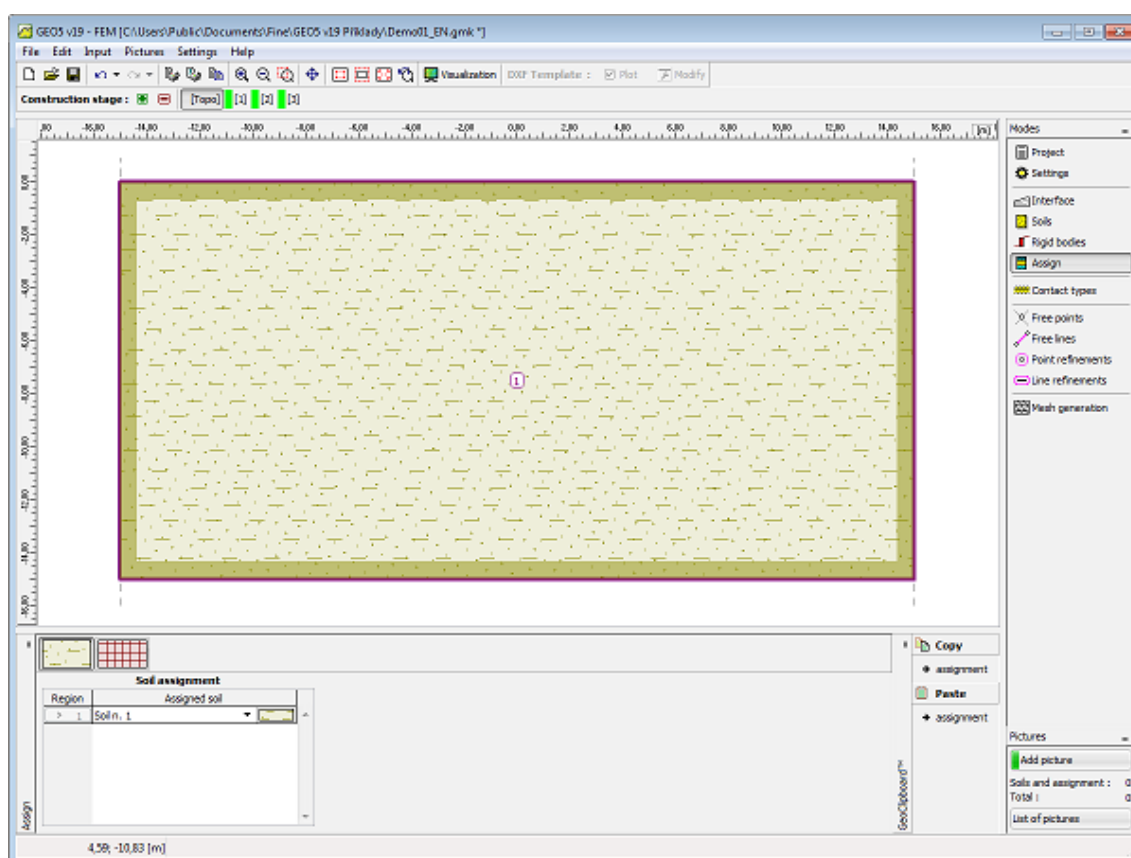
The **frame "Assign"** contains a list of layers of the profile and associated soils. The list of soils is graphically represented using buttons in the bar above the table, or is accessible from a combo list for each layer of the profile.

The procedure to assign a soil into a layer is described in detail [herein](#).

Unlike other programs the soils, which become active in calculations stages, are assigned to regions rather than to interfaces. The regions are created automatically when **creating the computational model**.

When a new soil is assigned in a topology regime, it is automatically assigned to all regions in a given geological layer.

The program makes it possible to **import soil assignment in the gINT format**. Assign of soils can be copied within all 2D GEO5 programs using **"GeoClipboard"**.

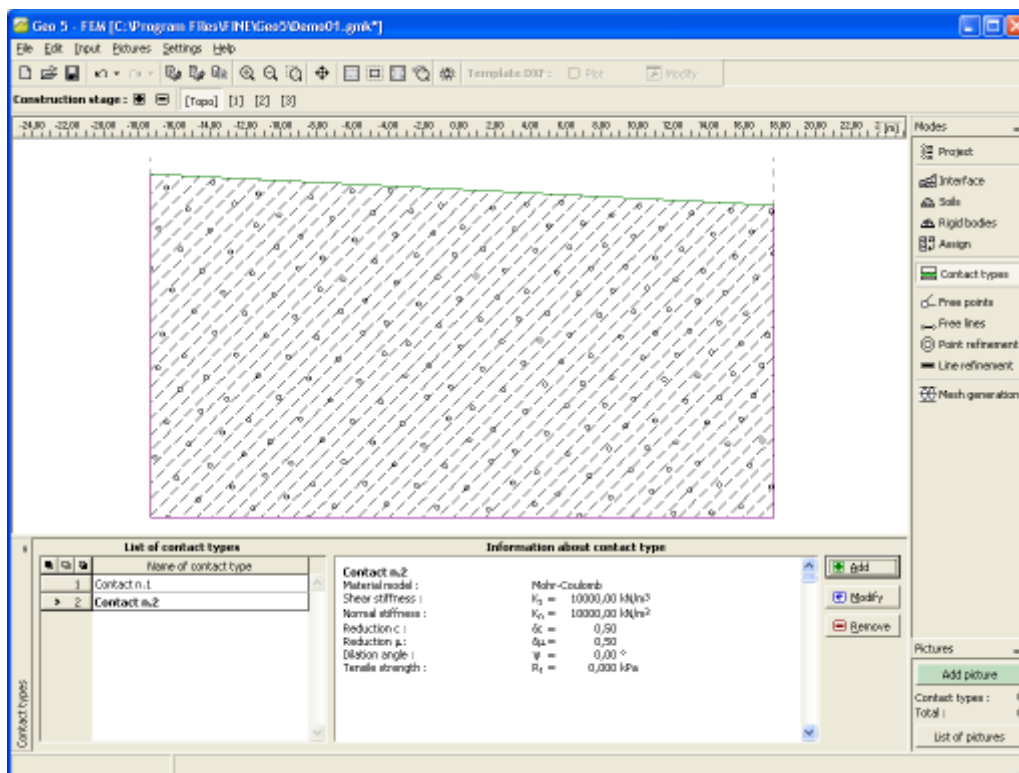


*Frame "Assign"*

## Contact Types

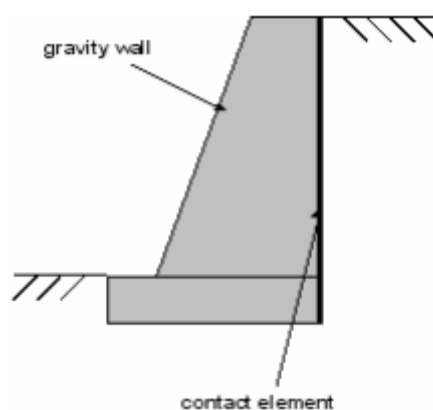
The **frame "Contact types"** contains a **table** with the list of types of contacts. Adding (editing) contacts is performed in the **"New types of contact" dialog window**.

This dialog window serves to define new **contact elements** which can be subsequently introduced into the program using the **"Beams"** and **"Contacts"** frames. The material model of a contact element can be either linear or nonlinear.



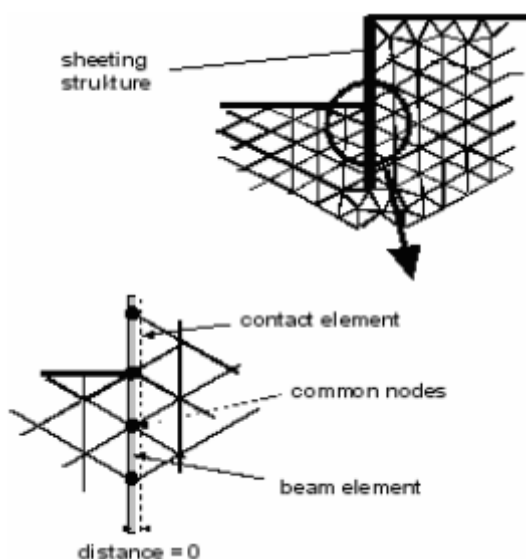
Frame "Contact types"

The **contact elements** are used in applications that require studying an interaction of a structure and a soil. They can be further used to model joints or interfaces of two distinct materials (soil - rock interface). A typical example of using contact elements is the **modeling of sheeting structures, retaining walls or tunnel lining**. In such applications the contact elements are used to model a relatively thin layer of a soil or rock loaded primarily in shear. Contacts can be defined also independently along **individual soil interfaces**.



Location of contact elements when modeling a gravity wall

The contact element is an element with a zero thickness allowing for calculating an interfacial stress as a function of a relative displacement developed along the interface.



Construction of a sheeting wall represented by beam and contact elements

## Contact Elements

Two options of the contact element material model are available. One may select either the **elastic model** with the possibility of plotting contact stresses while assuming the elastic behavior along the interface or the **plastic model**. The plastic model is based on the classical Mohr-Coulomb model extended by including the tension cut-off.

This model is therefore well suited when modeling tensile separation. In certain applications such as sheeting structures this model is vital for receiving meaningful predictions of the soil and structure response.

The basic model parameters are the cohesion  $c$ , coefficient of friction  $\mu$  and angle of dilation  $\psi$ . The parameters  $c$  and  $\mu$  can be specified also indirectly by reducing the soil strength parameters  $c$  and  $\tan(\varphi)$  of adjacent to the contact. If the contact is assumed between two soils (rocks) then the one having smaller values of  $c$  and  $\varphi$  is used in the reduction step.

The contact parameters are then defined as:

$$c = \sigma_z \cdot c_{zem}$$

$$\mu = \sigma_\mu \cdot \tan(\varphi_{zem})$$

If no better information regarding the reduction of parameters is available one may use the following values. For steel structures in sandy soils the reduction parameter equal to  $2/3$  is reasonable while for clays the value of  $1/3$  can be used. These parameters usually attain higher values when concrete structures are used. In general, the reduction parameters should be less than  $1$ . The dilation angle plays the same role as in the case of standard soil models. Just recall that by setting  $\psi = 0$  we prior assume elastic behavior in the tension/compression. The plastic deformation is thus limited to shear.

Additional parameters of the contact material model are the elastic stiffnesses in the **normal** and **tangential** directions  $k_n$  and  $k_s$ , respectively. They can be imagined as spring stiffnesses along a given interface. A reliable selection of the values of these parameters is not an easy task and is usually problem dependent. To shed a light on this subject one may relate these stiffnesses to the material parameters of the soil adjacent to the contact. The following

relations then apply:

$$K_n = \frac{E}{t}$$

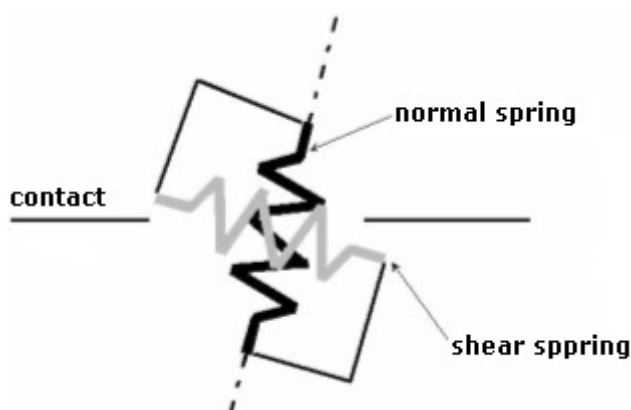
$$K_s = \frac{G}{t}$$

where:  $t$  -assumed (fictitious) thickness of contact (interface) layer  
 $G$  -shear modulus of elasticity  
 $E$  -Young's modulus of elasticity

In case of distinct materials ( $E_1, E_2, G_1, G_2$ ) we take the lower value of  $k_s$  and  $k_n$ .

Although in the case of a fully plastic behavior the selection of parameters  $k_s$  and  $k_n$  is not essential, the values assigned to these parameters are decisive for the success of the solution of a given nonlinear problem. Providing these values are too large (above  $100000 \text{ kN/m}^3$ ) the iteration process may oscillate. On the other hand, setting the values of  $k_s$  and  $k_n$  too low (below  $10000 \text{ kN/m}^3$ ) lead to nonrealistic deformations of structure.

The default setting in the program is  $10000 \text{ kN/m}^3$ .



Visualization of elastic stiffnesses

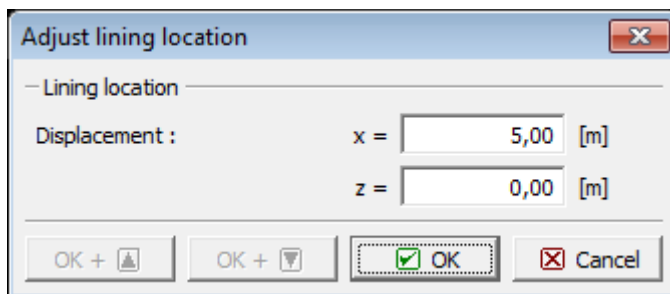
## Lining

The frame "**Lining**" contains a table with the list of input linings. This frame becomes accessible in the program once the "**Tunnel**" regime is activated in the frame "**Settings**". The "**Lining - FEM**" module simplifies modeling and positioning of individual tunnel linings.

The "**Lining - FEM**" module is an independent program used to design linings. Free points, free lines, line refinement, anchors, beams and beam loads created in this module are passed into the FEM program. Although behaving in a standard way, they cannot be edited in the FEM program. Editing is only possible in the "**Lining - FEM**" module.

Adding (editing) lining is performed in the "**Lining - FEM**" module. The following modes are available:

- **Add** Pressing the **"Add"** button launches the **"Lining - FEM"** module which allows for creating a new lining.
- **Position** Pressing the **"Position"** button opens the **"Adjust lining location"** dialog window, which allows for modifying coordinates of the lining location. To adjust lining in the FEM program is possible even without launching the **"Lining - FEM"** module.

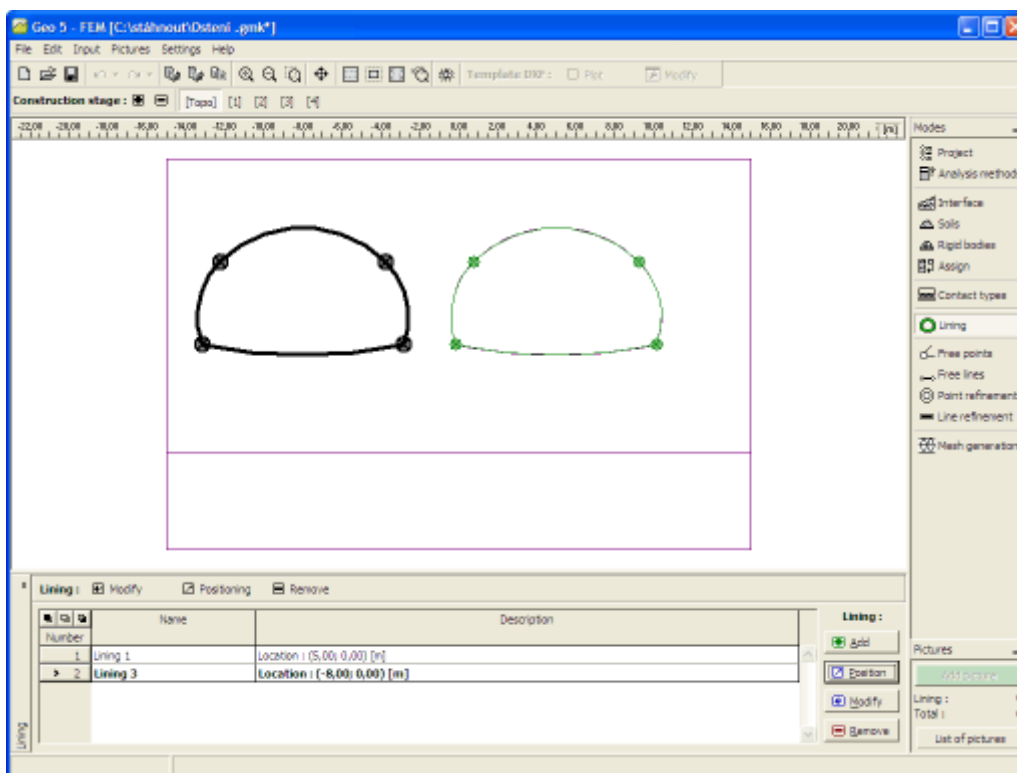


*Dialog window "Adjust lining location"*

- **Modify** Pressing the **"Modify"** button launches the **"Lining - FEM"** module, which allows for editing the selected lining
- **Remove** Pressing the **"Remove"** button opens the dialog window for confirming this action - upon accepting the selected lining is removed

The lining can also be modified, positioned and removed **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Lining"**. After choosing a particular mode the lining is selected on the desktop using the left mouse button. To continue follow the steps already described above.

Further details are available in chapter **"active objects"**.



Frame "Lining"

### Module Lining - FEM

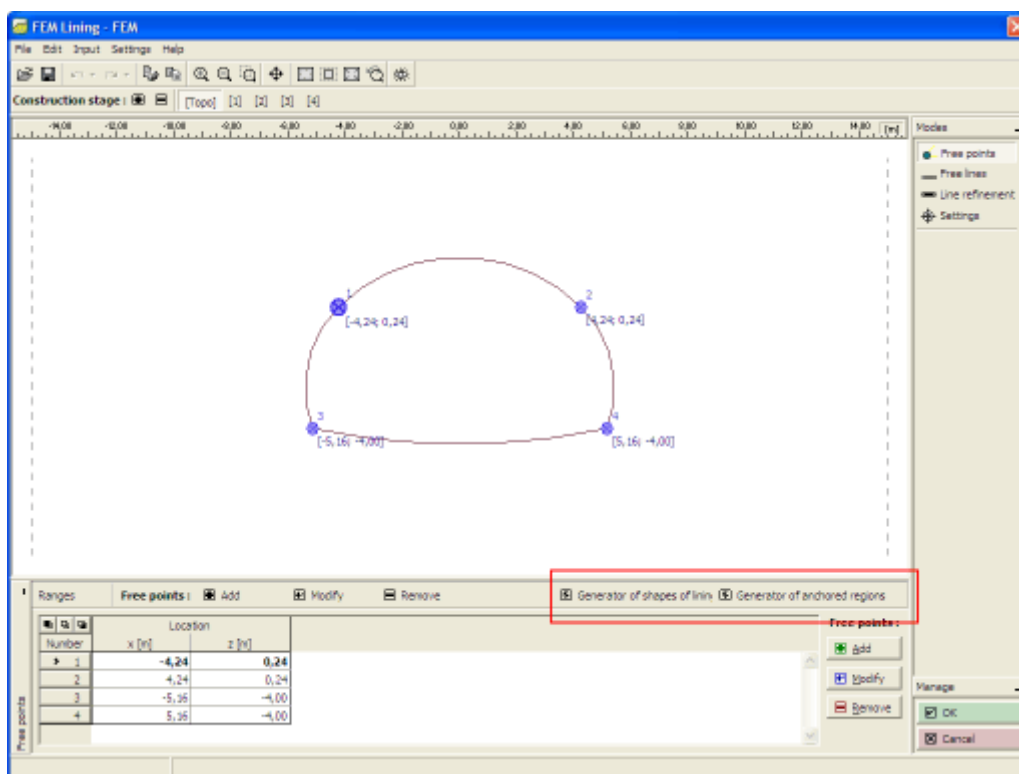
The "Lining - FEM" module simplifies modeling and positioning individual linings of tunnels. The module disposes of the features of the main FEM program including the "Topology" regime and stages of construction. In the "Topology" regime the module contains the "Free points", "Free lines", "Line refinement" and "Settings" frames. Frames accessible from stages of constructions are described within the stages of construction regime of the FEM program.

The "OK" button can be used to terminate the work in the module and to transmit data into the FEM program, whereas the "Cancel" button just terminates the work without data transmission.

The program makes it possible to import data in the \*.DXF format.

The data of the lining module can be independently saved or loaded while in this dialog using standard functions "Open" and "Save". This way allows for transmitted the lining between several analyzed tasks or within a single task.

Load a lining, having less number of stages than the current state, will add the remaining stages. In the case of lining having more stages, the corresponding stages are first added to the dialog and then to the main window. The data from the lining regime cannot be loaded directly into the main window.



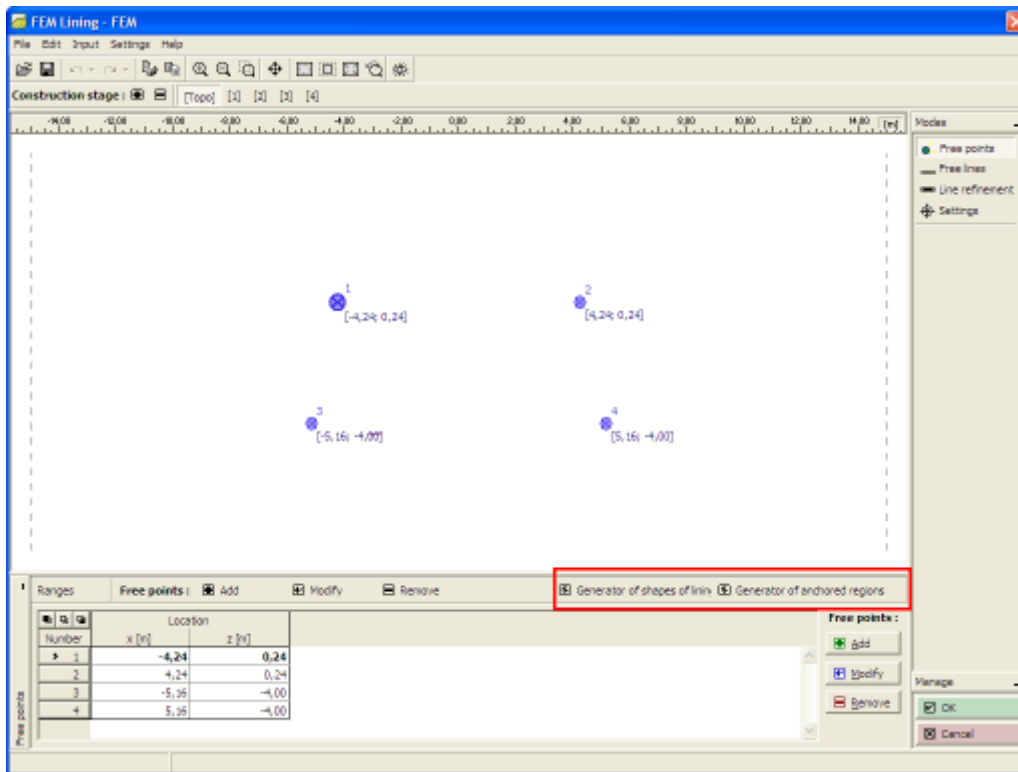
Module "Lining FEM"

### Free Points

The frame "Free points" contains a table with the list of input free points. Working with free points follows the same guidelines as in the FEM program - frame "Free points".



The frame differs by the functions on the horizontal tool bar, which contains the "Generator of shape of lining" and "Generator of anchored regions" buttons. The function of the "Range" button is identical to that in the FEM program - frame "Interface".

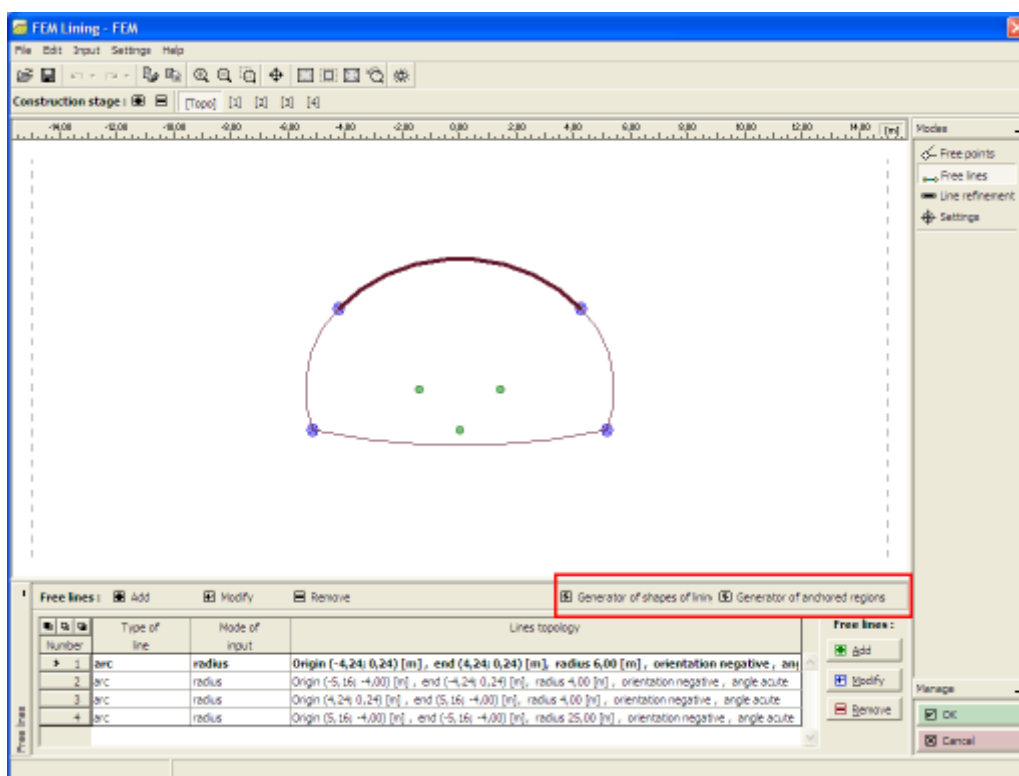


Frame "Free points"

## Free Lines

The frame "Free lines" contains a table with the list of input free points. Working with free lines follows the same guidelines as in the FEM program - frame "Free lines".

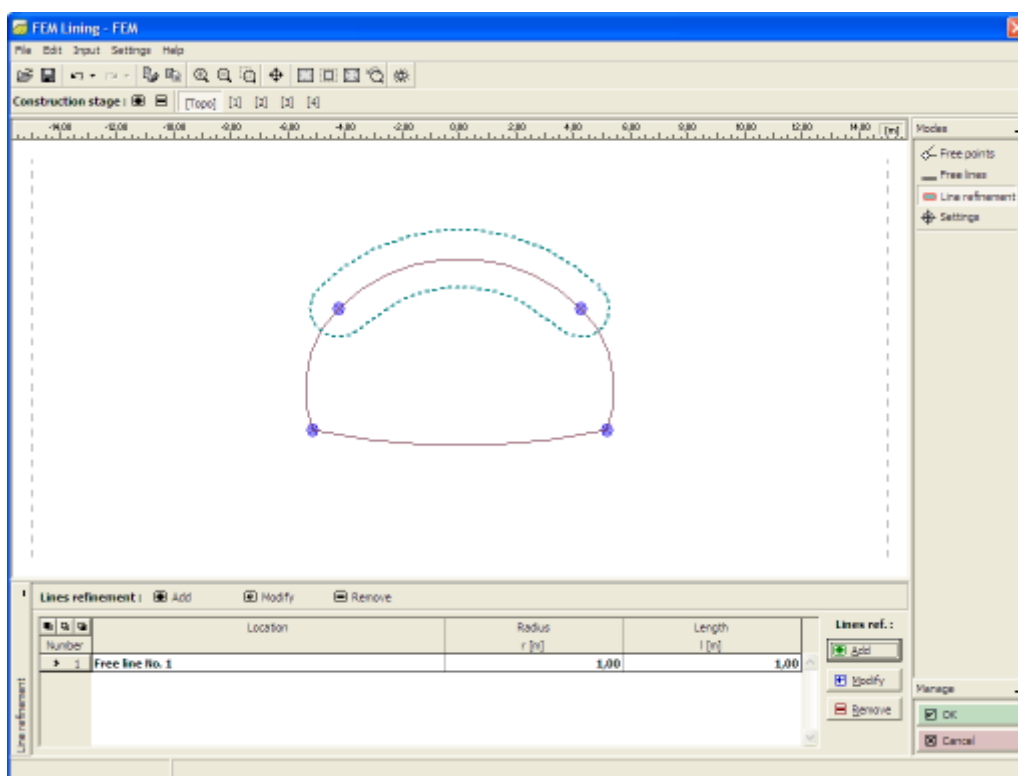
The frame differs by the functions on the horizontal tool bar, which contains the "Generator of shape of lining" and "Generator of anchored regions" buttons.



Frame "Free lines"

## Line Refinement

The frame "Line refinement" contains a table with the list of input point refinements. Working with free lines refinement follows the same guidelines as in the FEM program - frame "Line refinement".

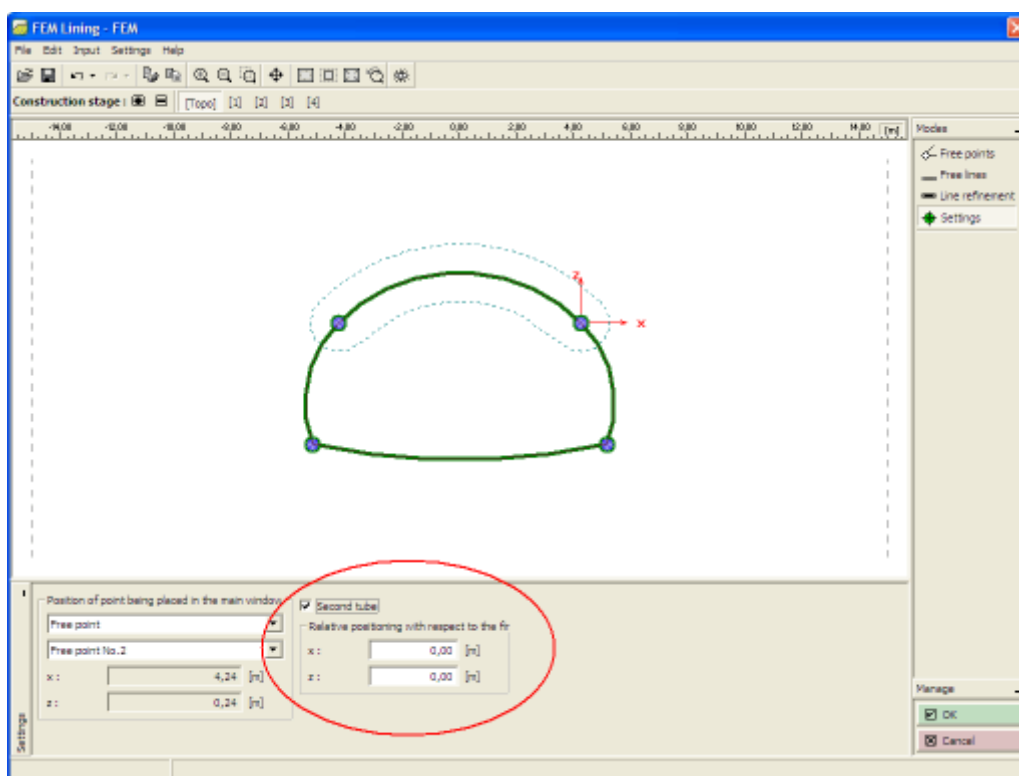


Frame "Line refinement"

## Settings

The frame **"Settings"** allows for redefining the location of a point to be subsequently positioned in the main window of the FEM program. The point location can be associated with the selected free point or determined by the coordinate system origin or by an arbitrary coordinate. This way allows for an exact positioning of a given point of the lining structure in the main window of the FEM program.

The use of a second tube can be activated in the right part of the frame. The second tube will appear in the frame **"Settings"** as a preview, and then after transmitting it into the FEM program. The second tube is a clone of the first one. It differs only in the positioning with respect to the originally defined structure.

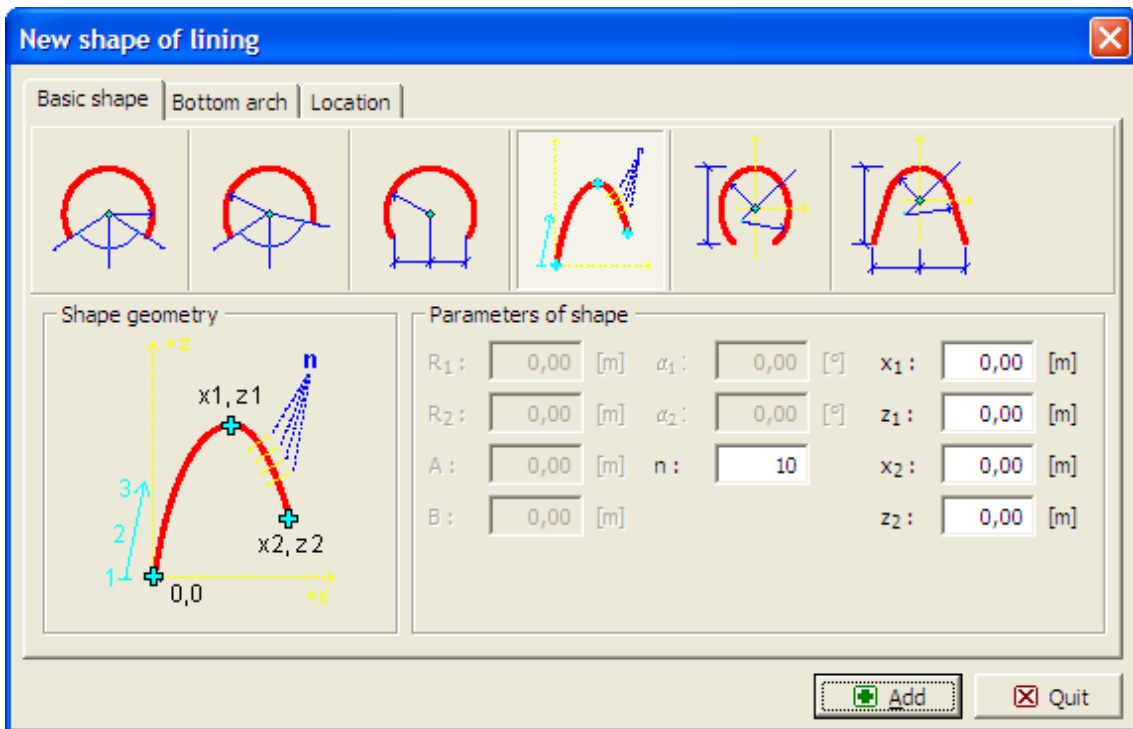


Frame "Settings"

## Generator of Lining Shape

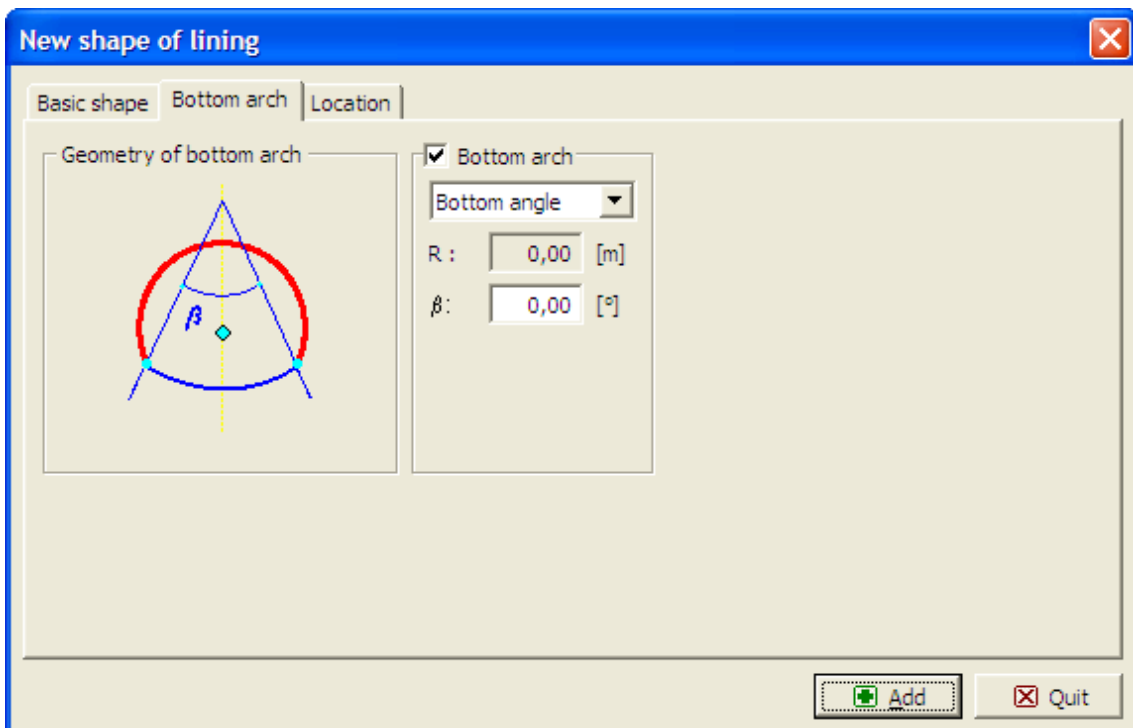
Depending on particular parameters the generator creates corresponding elements which are then operated on independently with no possibility for being parametrically modification. If the parameters of generation are acceptable, the program displays during their modification the current graphical representation of generated elements.

Six basic shapes of linings is available for generating free points and free lines in the "**New shape of lining**" dialog window. Each shape is defined by several parameters (radii, angles, height, spacing, subdivision number, control points).



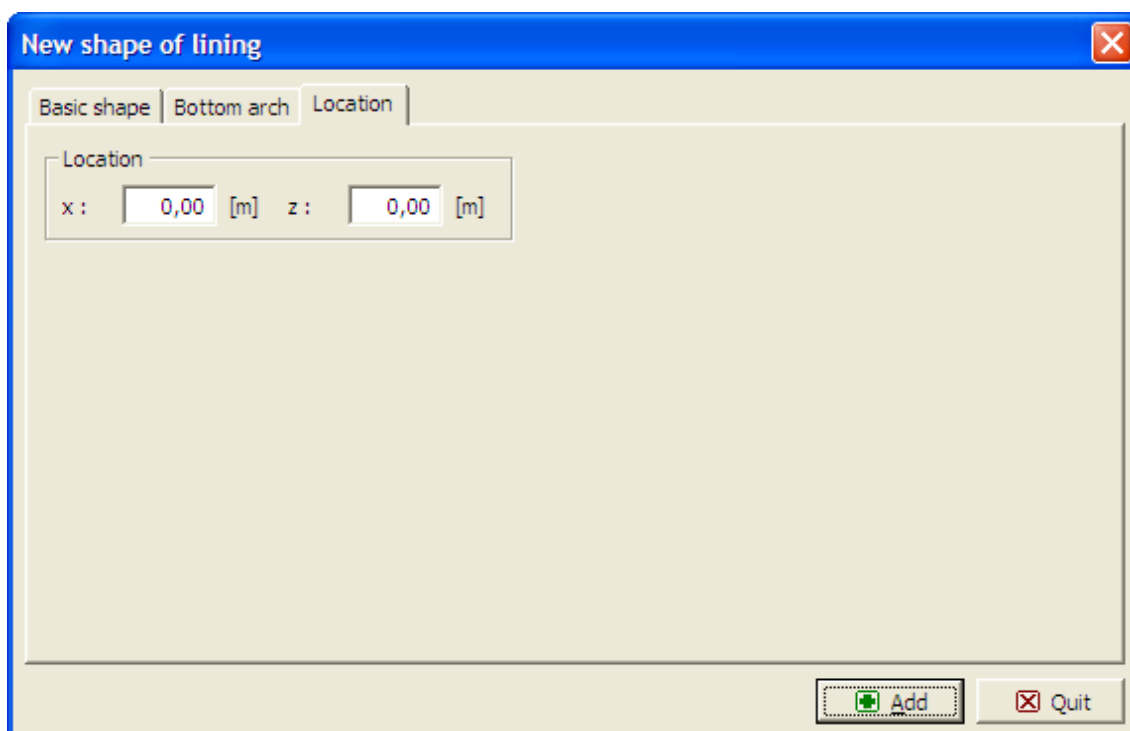
Dialog window "New shape of lining" - tab "Basic shape"

The "**Bottom arch**" tab sheep allows us to choose, whether the lining invert will be flat or arched, determined parametrically either by a radius or an angle.



Dialog window "New shape of lining" - tab "Bottom arch"

The "**Location**" tab allows, using coordinates, for changing the lining location.

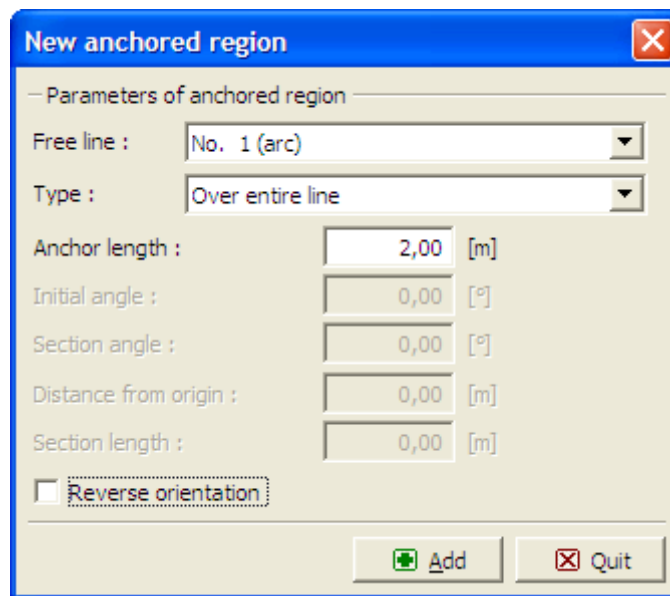


*Dialog window "New shape of lining" - tab "Location"*

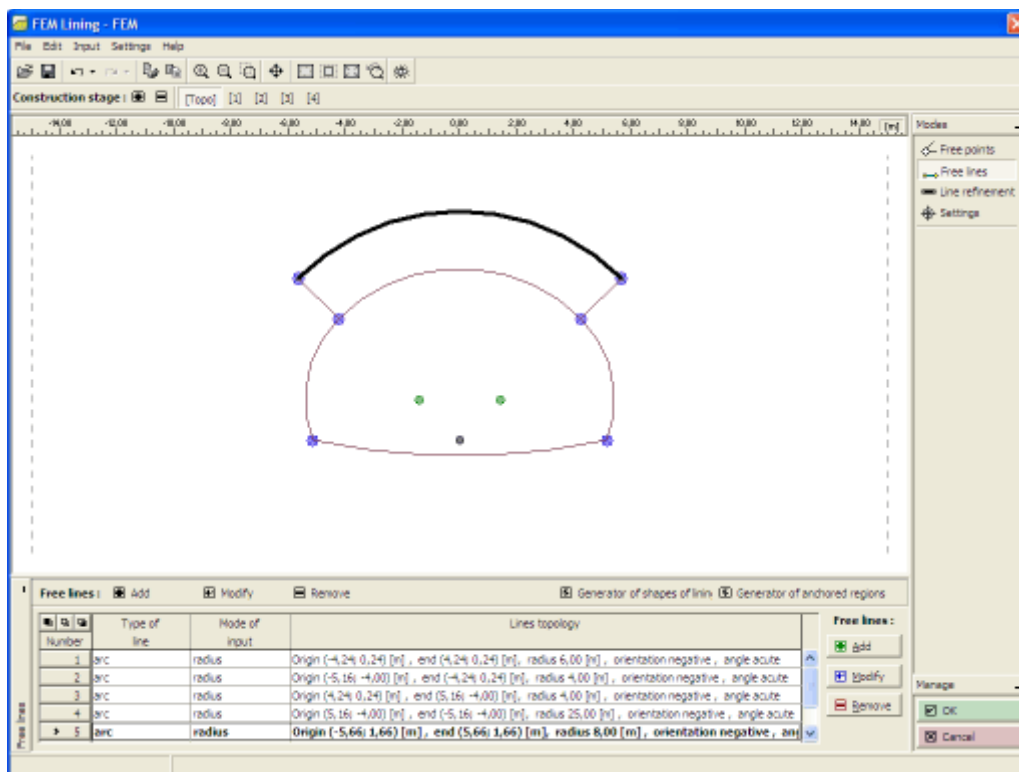
## Generator of Anchored Regions

Depending on particular parameters the generator creates corresponding elements which are then operated on independently with no possibility for being parametrically modification. If the parameters of generation are acceptable, the program displays during their modification the current graphical representation of generated elements.

The "**New anchored region**" dialog window serves to generate free points and free lines based, however, on already input lines. This generates a closed region, which is then assigned in the FEM program a special soil characterizing a densely anchored region. The dialog window requires specifying a line number and parameters based on the type anchoring system (over entire line, angle sector, origin and length).



Dialog window "New anchored region"



Defining anchored region

## Stages of Construction

Stages of construction in the "Lining - FEM" module and in the FEM program correspond to each other. They, however, may vary in several features.

Different behavior of stages in the "Lining - FEM" module:

- Possible to switch to stages of construction from the "**Topology**" regime without generating the FE mesh.
- Stages of construction added in the "**Lining - FEM**" module are, after confirming, transferred also into the FEM program.
- Stages of construction, preceding the stage from which the "**Lining - FEM**" module was launched, cannot be used.
- Stages of construction defined prior to launching the "**Lining - FEM**" module cannot be deleted.

## Free Points

The frame "**Free points**" contains a table with the list of input free points. Adding (editing) free points is performed in the "**New free point**" dialog window.

The free points can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Free points". The following modes are available:

- **Add**                    The point is introduced by clicking the left mouse button at a desired location on the desktop.
- **Adjust**                    Clicking the left mouse button on already existing free point opens the "**Adjust free point properties**" dialog window, which allows for modifying its parameters.
- **Remove**                    Clicking the left mouse button on already existing free point opens the **free point removal** dialog window - accepting this action removes the selected free point.

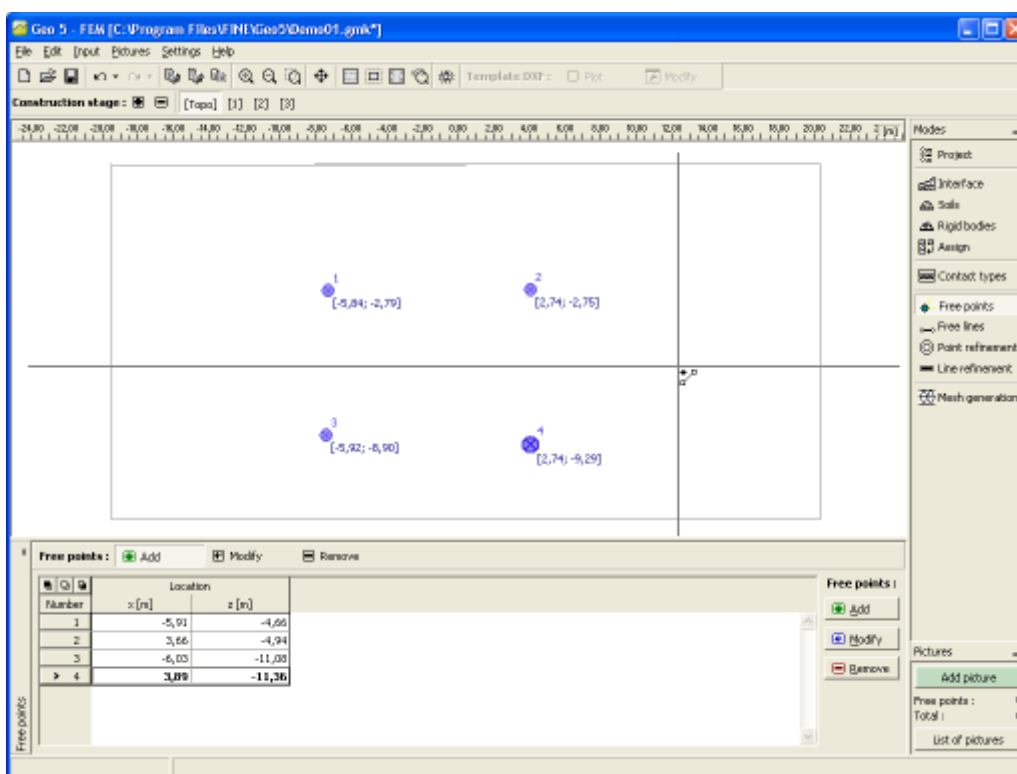
The free points can also be edited on the desktop with the help of **active objects**.

The program allows for inputting an arbitrary number of free nodes anywhere inside or outside the structure. Free nodes have several main functions:

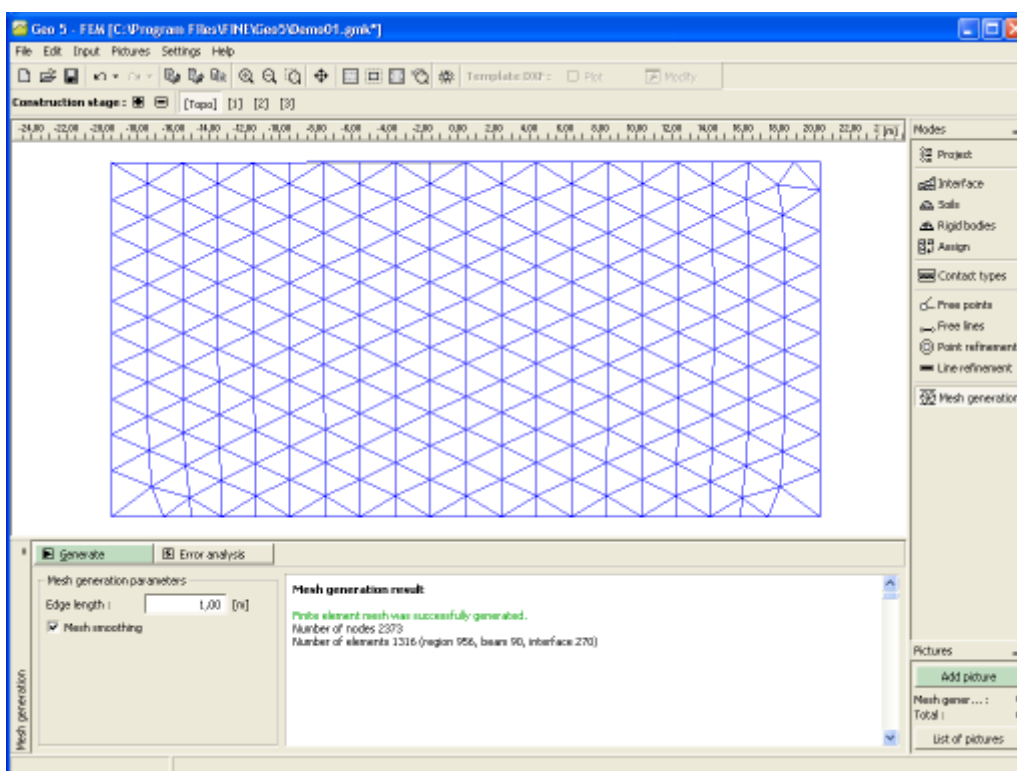
- **nodes to define structure** (tunnel opening, lining, sheeting, beams)
- **auxiliary points** for the **mesh refinement**
- points to **define a boundary condition**, to input forces, etc.

If a free node is found inside or on the boundary of a structure, it becomes **automatically a part of the finite element mesh**. This option allows an adjustment of the finite element mesh or makes it possible to create own mesh.

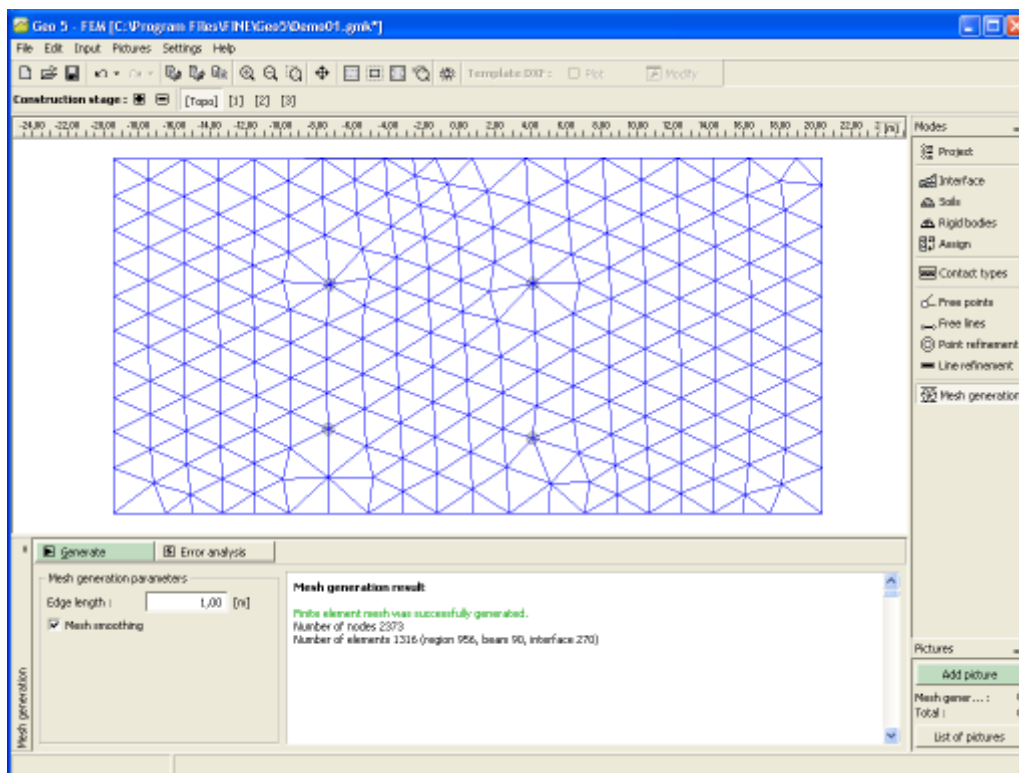




Frame "Free points"



Mesh generated without free points



*Mesh with free points*

## Free Lines

The frame "**Free lines**" contains a table with the list of input free points. Adding (editing) free points is performed in the "**New free line**" dialog window.

The lines are defined **between individual points** (segments, arcs, circles) or around individual points (circles). The lines can be introduced both between free points and between points located on interfaces including the terrain surface.

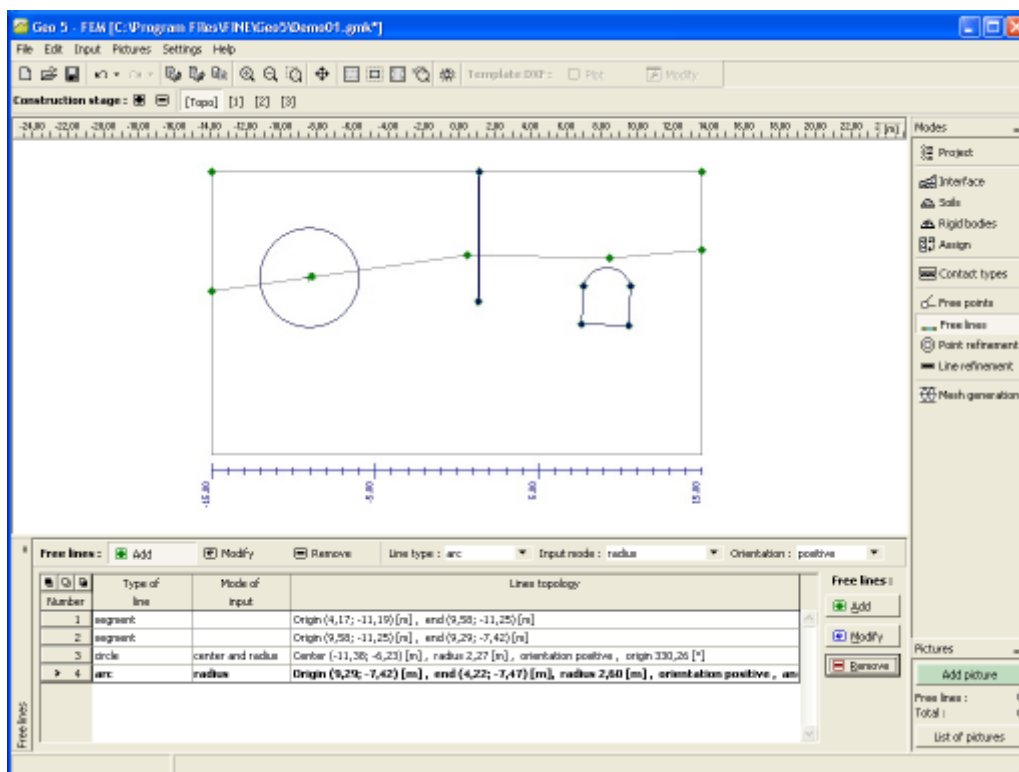
The lines may **intersect each other and may have an arbitrary number of contact points** - intersections of individual lines are determined by the program when **adjusting the geometrical model**. The free lines may be used to **introduce beam elements** into the model.

The free points can also be introduced using the mouse. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Free points". The following modes are available:

- **Add**            The line is introduced by clicking the left mouse button at a desired location on the desktop.
- **Line type** A combo list is used to select the desired line (segment, arc and circle).
  - **segment**    Clicking individual points on the desktop with the left mouse button creates a point to point line
  - **arc**            Use the combo list to choose a particular mode of defining an arc segment (third point, center, radius, height). Clicking the left mouse button on the desktop then selects points to define an arc. When selecting one of the following options - center, radius or, you are further requested to select from the combo list the orientation (positive, negative).

- **circle** Use the combo list to choose a particular mode of defining a circle (center and radius, three points). Clicking the left mouse button on the desktop then selects points to define a circle. The combo list is also used to select the orientation (positive, negative).
- **Adjust** Clicking the left mouse button on already existing free line opens the "**Adjust free line properties**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing free line opens the **free line removal** dialog window - accepting this action removes the selected free line.

The free lines can also be edited on the desktop with the help of **active objects**.



Frame "Free lines" - different types of free lines

## Point Refinement

The **frame "Point refinement"** contains a **table** with the list of input point refinements. Adding (editing) a point refinement is performed in the "**New point refinement**" dialog window.

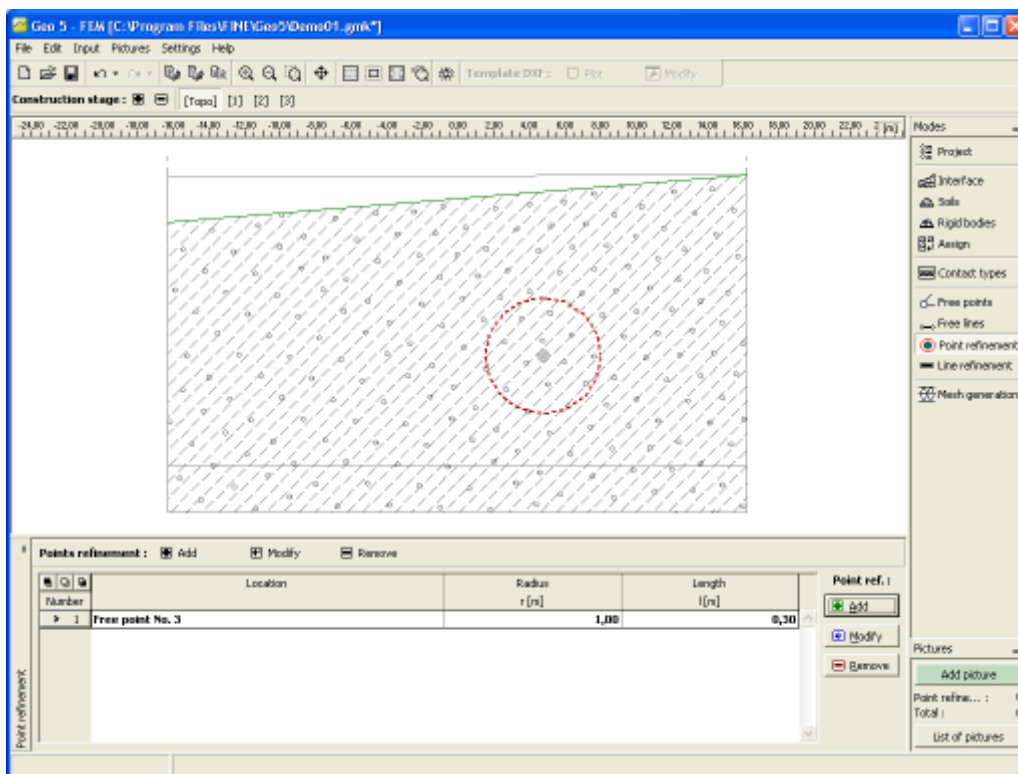
**Refining the finite element mesh around points** is an important feature, which allows us to create an **appropriate finite element mesh**. Both free points and points pertinent to individual interfaces including terrain can be used to refine the original finite element mesh.

Refining the finite element mesh around points can also be performed **using the mouse**. Several input modes are available depending on the selected button on the "**Point refinement**" horizontal bar:

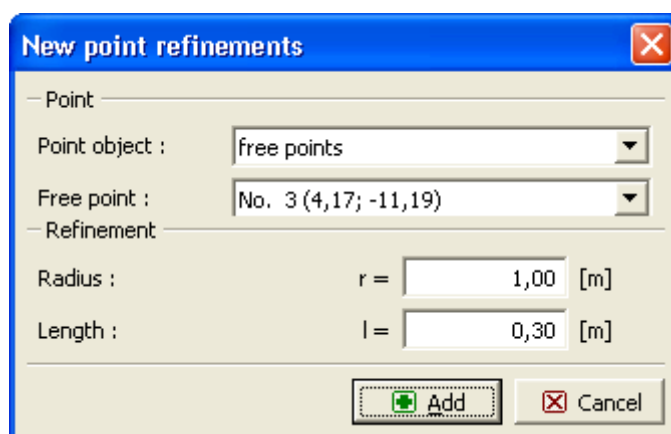
- **Add** Clicking the left mouse button on the desktop selects the point for refining the mesh. The "**New point refinement**" dialog window serves to input the required parameters.

- **Adjust** Clicking the left mouse button on already existing (refined) point opens the "**Adjust point refinement properties**" dialog window, which allows for modifying individual parameters of the refinement.
- **Remove** Clicking the left mouse button on already existing (refined) point opens the **point refinement removal** dialog window - accepting this action removes the selected point refinement.

The point refinement can also be edited on the desktop with the help of **active objects**.



Frame "Point refinement"



Dialog window "New point refinement"

## Line Refinement

The **frame "Line refinement"** contains a **table** with the list of input point refinements. Adding

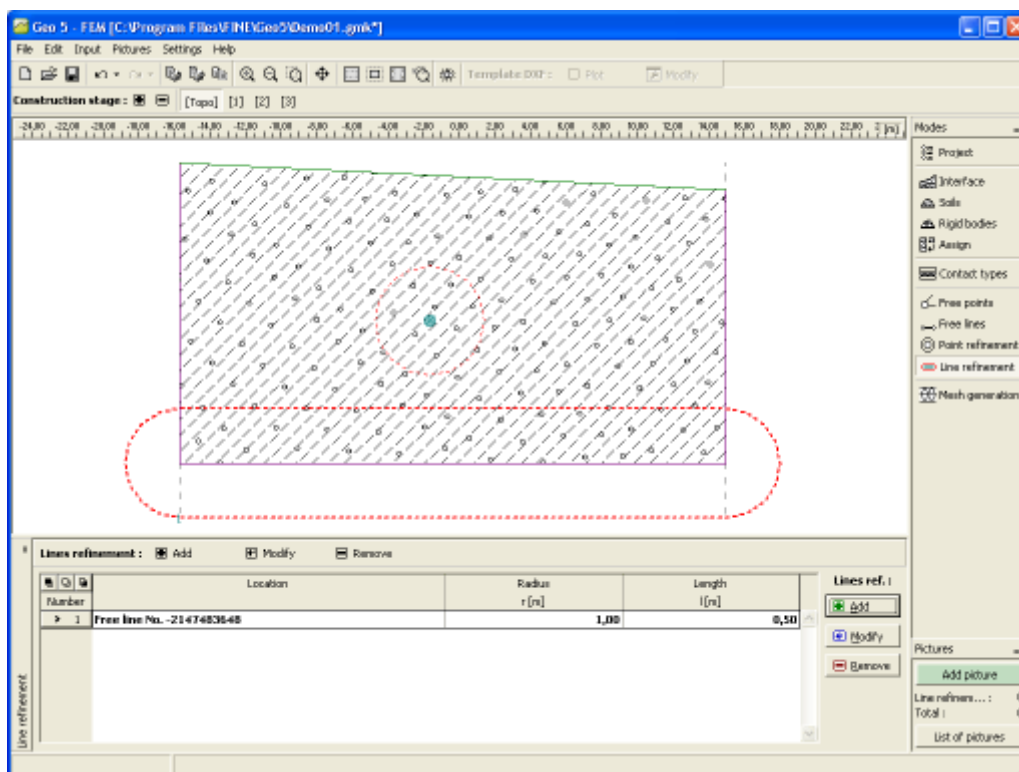
(editing) a line refinement is performed in the "**New line refinement**" dialog window.

**Refining the finite element mesh around lines** is an important feature, which allows us to create an **appropriate finite element mesh**. Both free lines and lines pertinent to individual interfaces including terrain can be used to refine the original finite element mesh.

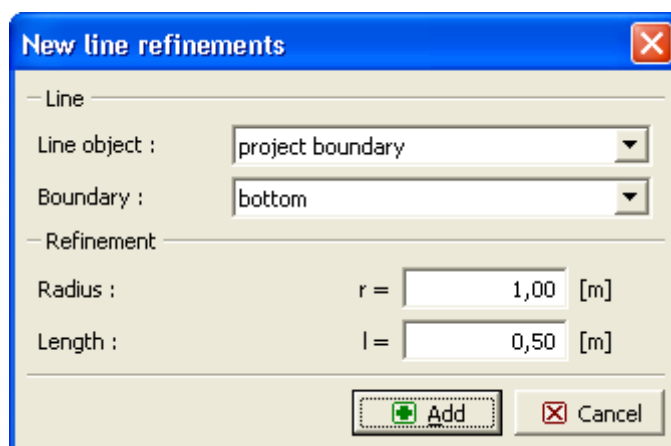
Refining the finite element mesh around points can also be performed **using the mouse**. Several input modes are available depending on the selected button on the "**Line refinement**" horizontal bar:

- **Add** Clicking the left mouse button on the desktop selects the line for refining the mesh. The "**New line refinement**" dialog window serves to input the required parameters.
- **Adjust** Clicking the left mouse button on already existing (refined) line opens the "**Adjust line refinement properties**" dialog window, which allows for modifying individual parameters of the refinement.
- **Remove** Clicking the left mouse button on already existing (refined) line opens the **line refinement removal** dialog window - accepting this action removes the selected line refinement.

The line refinement can also be edited on the desktop with the help of **active objects**.



Frame "Line refinement"



*Dialog window "New line refinement"*

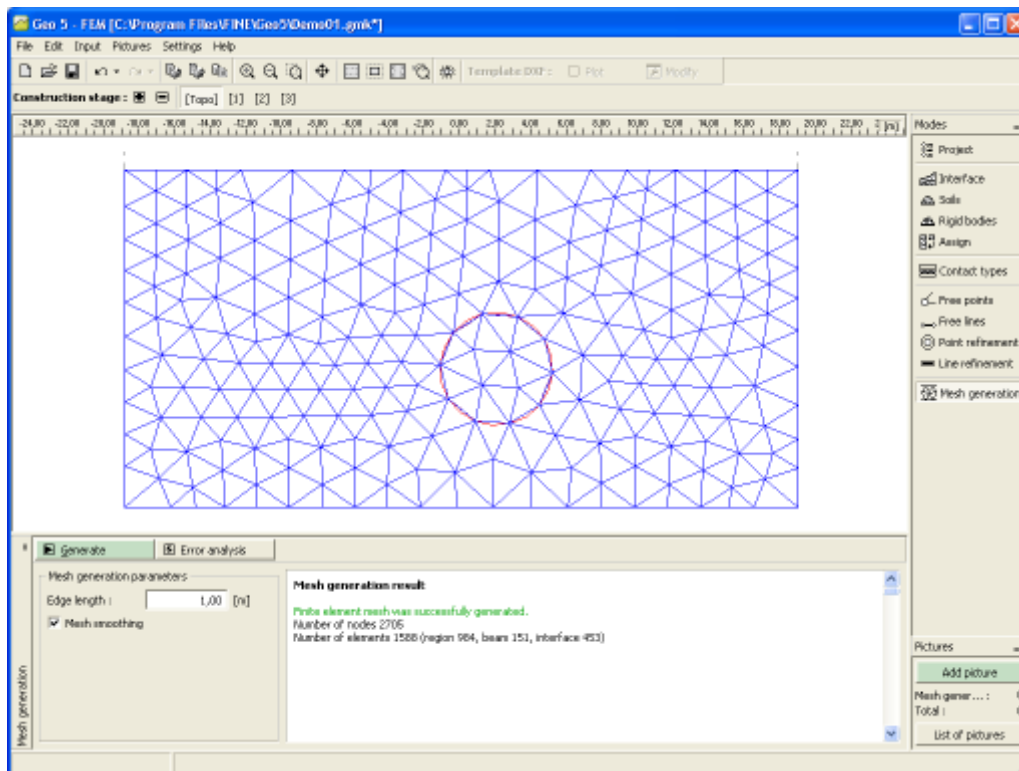
## Mesh Generation

The **frame "Mesh generator"** serves to define the basic setting to generate mesh (left part) and to view information about generated mesh (right part).

**A successfully generated mesh** completes the **topology input stage** - the analysis then proceeds with the **calculation stages**. When generating mesh the program **automatically** introduces **standard boundary conditions**. Information about the resulting mesh including **warnings** for possible weak points in the mesh is displayed in the right bottom window.

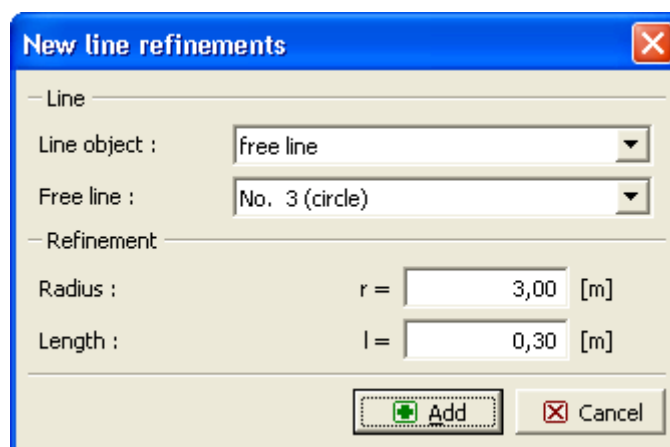
Correctly generated finite element mesh is the major step in achieving accurate and reliable results. The program FEM has an automatic mesh generator, which may substantially simplify this task. Nevertheless, **certain rules should be followed** when creating a finite element mesh:

- The basic mesh density can be specified in the **"Mesh generator"** dialog window. It is generally accepted that the finer the mesh the better the results - computation as well as post-processing, however, may slow down substantially. The goal thus becomes to find an optimum mesh density - this mainly depends on the user experiences. Meshes generated in example problems may serve as an initial hint.

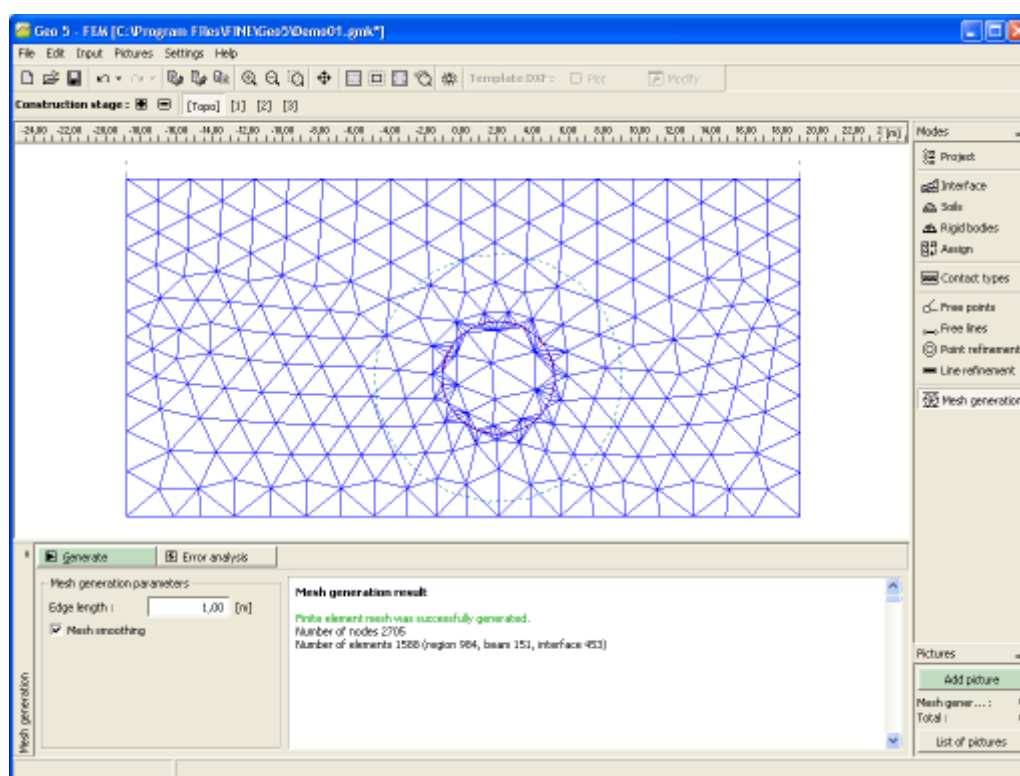


Frame "Mesh generation" - a mesh with no local refinement

- The finite element mesh should be sufficiently fine in locations in which large stress gradients are expected (point supports, corners, openings, etc.). To that end, it is possible to specify the mesh refinement in the neighborhood of these locations. The mesh refinement can be specified around individual **points** or **lines**. The spread of refinement should be at least 3-5 times the desired refinement in the center of the refinement. Also, both values (density and spread of refinement) should be reasonable in view of the prescribed mesh density that applies to the surrounding region. This assures a smooth transition between regions with different mesh densities. Singular lines should be tackled in the same way. For more complicated problems it is useful to first carry out the analysis with a rather coarse mesh and then after examining the results to refine the mesh accordingly.



Defining mesh refinement around a circular line



*New mesh after refining the original mesh around a circular line*

By default program assumes **6-node triangular elements** with mesh smoothing. The accuracy of the results more or less corresponds to twice as fine mesh composed of 3-node triangular elements. The 3-node elements are available only in the "Advanced input" mode (check box "**6-node elements**") and serve merely for research and testing purposes. The stability analysis, however, can be performed with 6-node triangular elements only. In case of nonlinear analysis, these elements should be used exclusively.

The "Advanced input" mode allows also for the generation of mixed mesh (triangular and quadrilateral elements).

## Mesh Generator Warning

In the "**Warning - structure critical locations**" dialog window the user is prompted for possible locations on the structure that may cause problems during automatic mesh generation. When positioning the cursor on individual warnings the corresponding critical region on a structure is highlighted with a red color. The following items are checked:

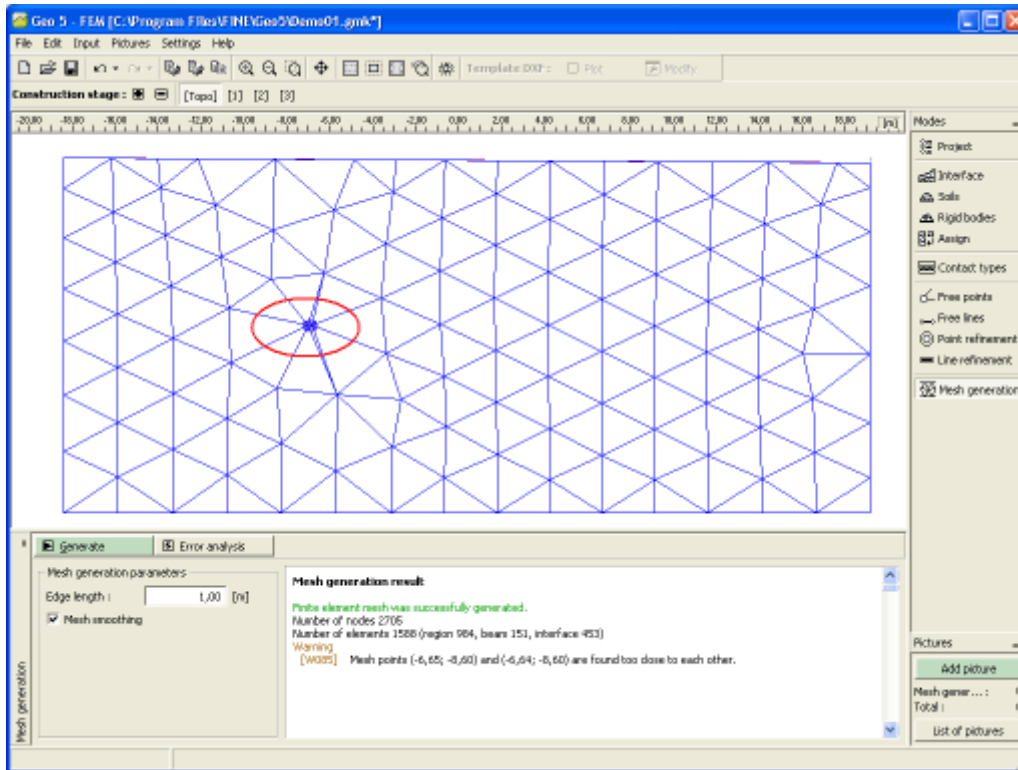
- whether the distance between two points is greater than one tenth of the required element edge length,
- whether the distance between a point and a line is greater than one tenth of the element edge length,
- whether the area of a region is greater than twice the element edge length,
- whether points and/or lines are found inside the structure (in the soil).

These warnings suggest locations, in which the mesh generator experience problems. The following possibilities may occur:

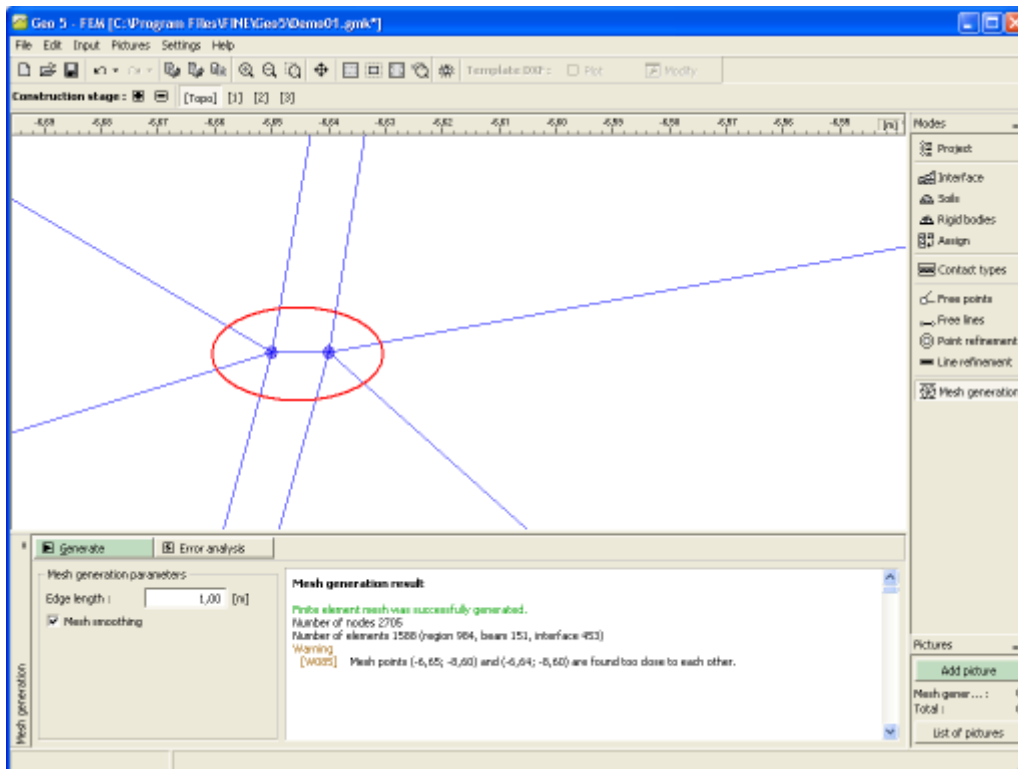
- the mesh is not generated => this calls for a new input of geometrical data,



- the mesh is generated => in this case it is up to the user to decide whether the mesh is reasonable - in any case, the warning can be further ignored and the analysis can be carried out.



*Warning after identifying critical sections in FE mesh*



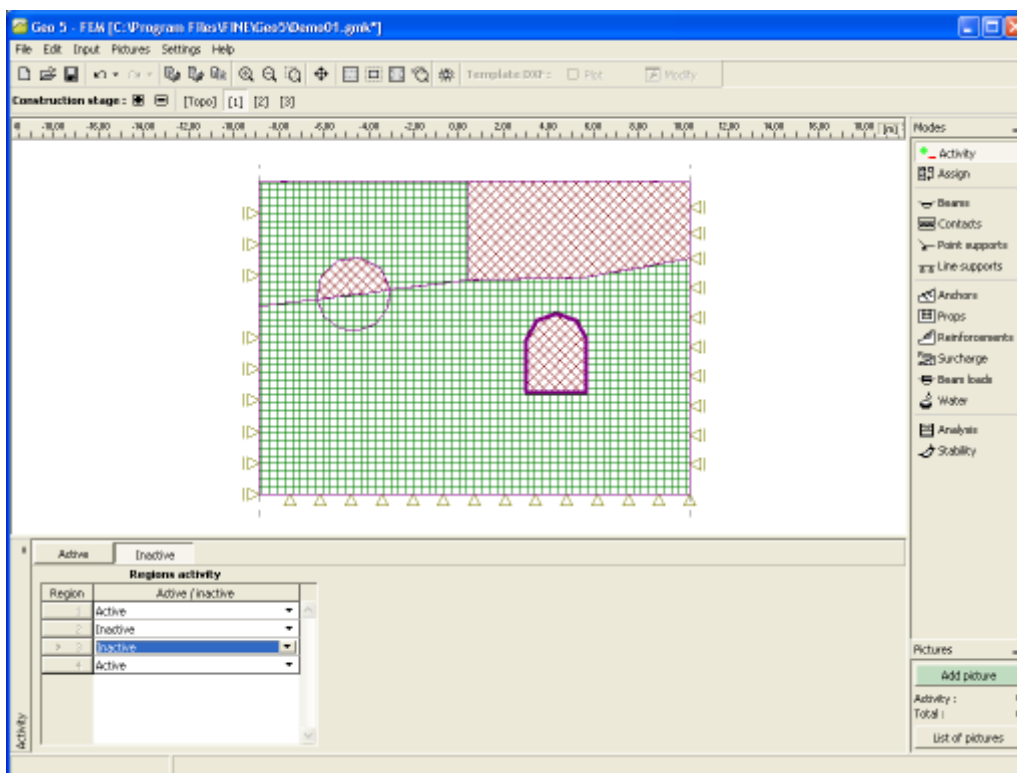
*Critical section after zooming in - two points are too close to each other*

## Adjusting Original Geometry

The program contains a built-in **automatic corrector of the specified geometry**. This means that prior to the mesh generation the program automatically locates all points of intersection of lines, locates all closed regions and creates a corresponding geometrical (calculation) model.

Such new regions can be then deactivated or they can be assigned a new soil. The main advantage of this system becomes evident when creating a geometrical model for tunnels (step by step excavation) or for sheeted structures. Creating even a very complicated model thus becomes rather simple and can be performed very efficiently.

Correcting the original geometrical model may cause some points in the model to be too close to each other or too small regions might be created. **Warning message** then appears in the right bottom window identifying such weak points in the model.



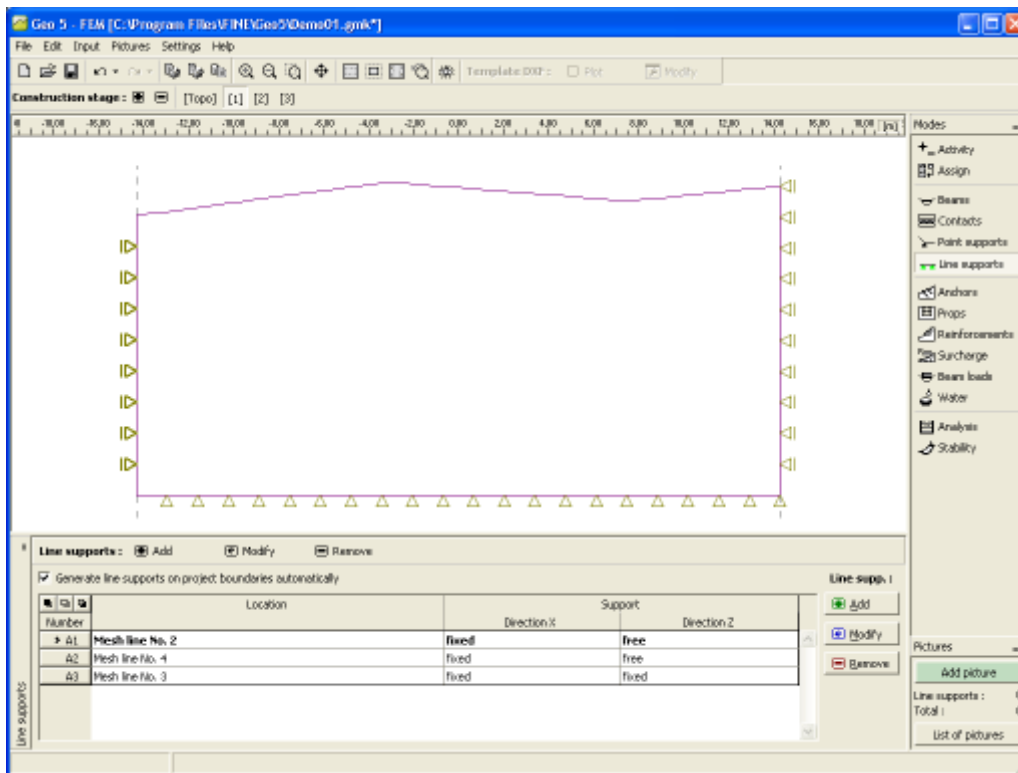
*Regions after performing an automatic adjustment of the geometrical model*

## Standard Boundary Conditions

The program automatically generates standard boundary conditions. Therefore, **in routine problems the user does not have to enter the step of specifying supports.**

The standard boundary conditions are:

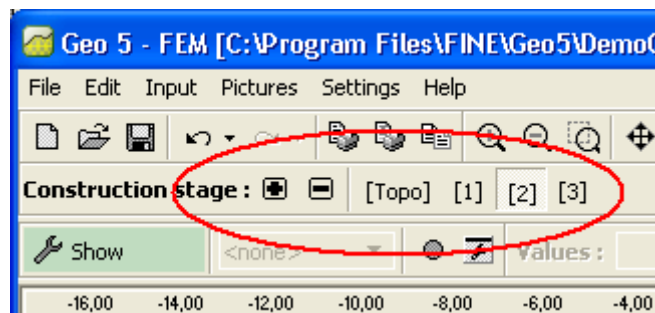
- smooth pin along the bottom edge of the geometrical model,
- sliding pin along vertical edges of the geometrical model.



Standard boundary conditions

## Construction Stages

The actual analysis is performed in individual stages of construction (calculation stages) after the geometrical model and **generating the finite element mesh (topology stage)**. One can move between calculation stages and the **"Topology"** regime using the buttons on the horizontal tool bar.

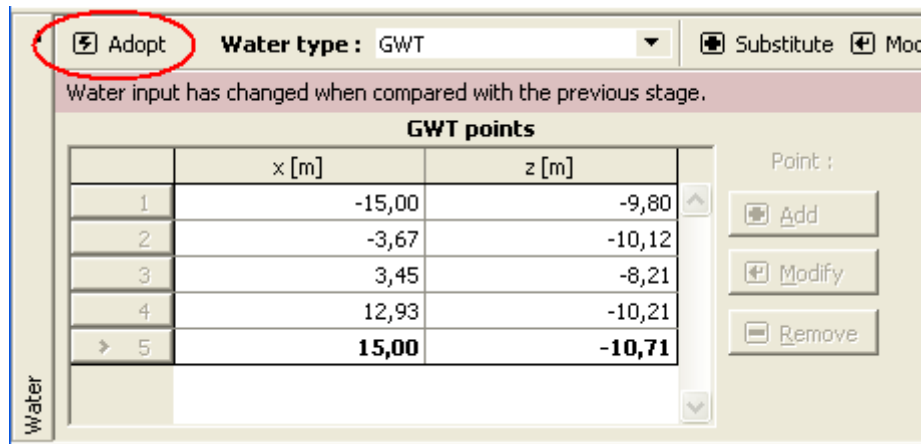


Tool bar "Construction stages" - switching between "Topology" regime and other stages of constructions

The calculation stages serve to model gradually build structures. Their correct definition and proper sequence is very important. The analysis of each stage builds (except for the stability analysis) upon the **results derived in the previous stage**. Information about individual objects and their properties are carried over from one stage to the other - when editing an existing stage or creating a new stage the program applies the **principle of heredity**.

Some frames ("Water", "Activity", "Assign") contain at the right part of the bar the **"Adopt"** button. The button becomes active once the data defined in the frame differ from those

defined in the previous stage. After pressing this button the corresponding data ("**Water**", "**Activity**") are adopted from the previous stage.



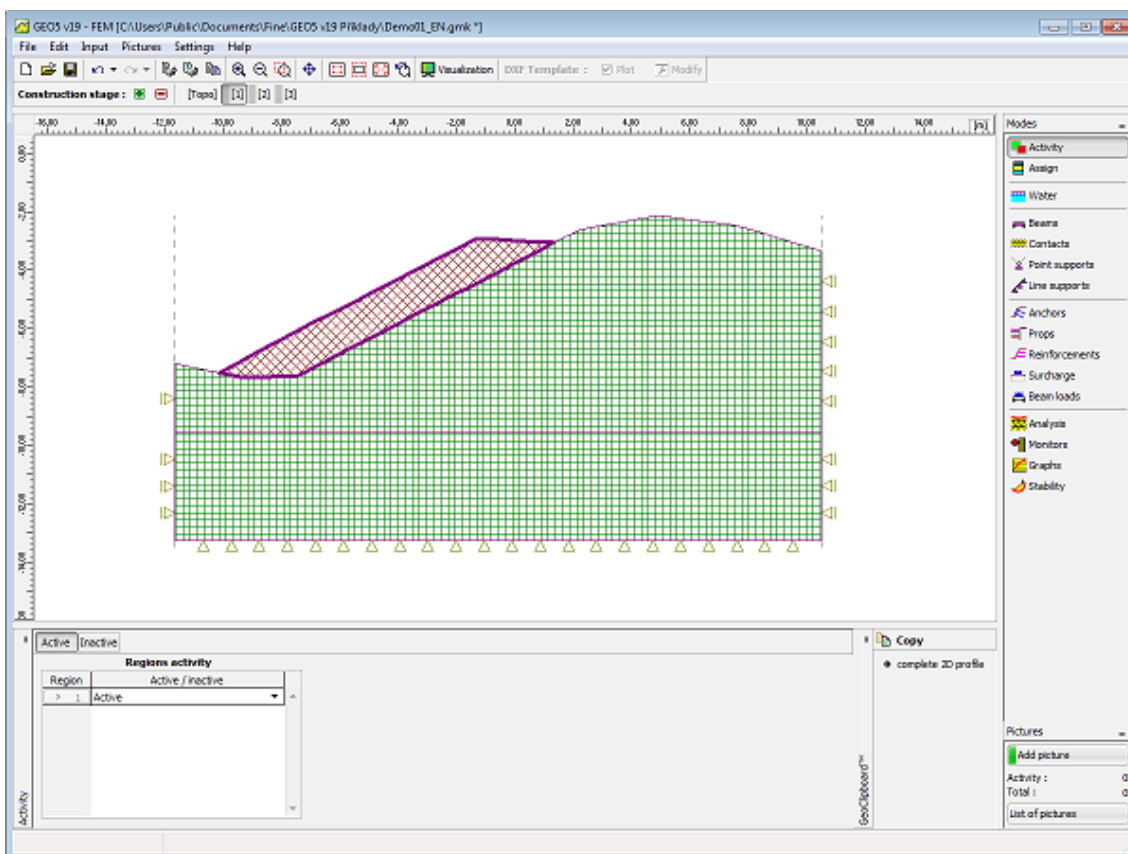
*Changing input data - accepting data from the previous stage of construction*

The first stage of construction (**calculation of geostatic stress**) represents the initial state of the soil (rock) body before the onset of construction - displacements associated with this stage are therefore set equal to zero.

Loss of convergence may occur for a certain stage of construction. In this case (the results are not available for non-converged structure) the subsequent stages cannot be analyzed. To avoid modeling errors we recommend the user to follow [the recommended way of the modeling](#) and analysis of a structure.

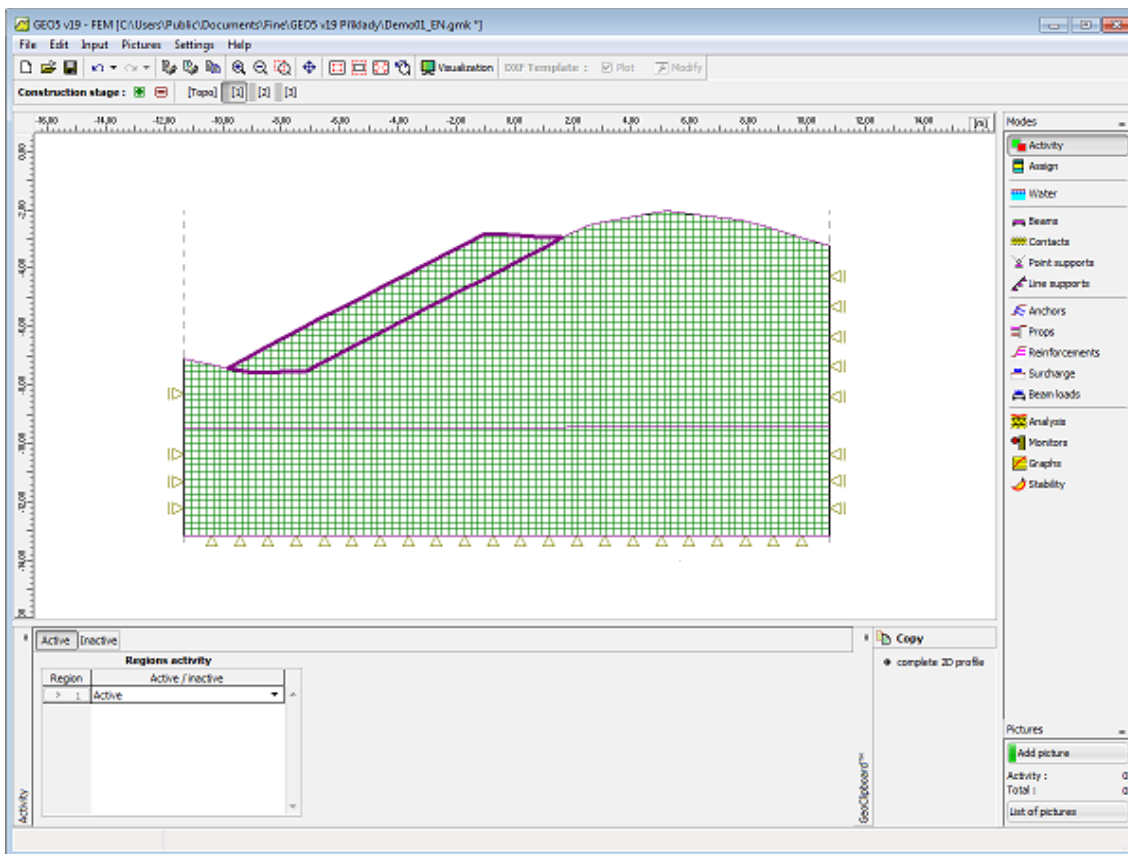
## Activation

The program allows for **removing (deactivating) soils** from individual regions. As an example we consider an embankment analysis. In such a case, it must be accounted for already in the topology regime when creating the overall geometrical model. In the  $1^{st}$  calculation stage, however, it can be deactivated. Similar approach applies also to underground or open excavations (tunnels, sheeting structures). When deactivating a region [below the ground water](#) it is necessary to correctly model the region boundary.



*Modeling embankment - 1st construction stage*

The embankment can be subsequently reactivated in the next construction stage.



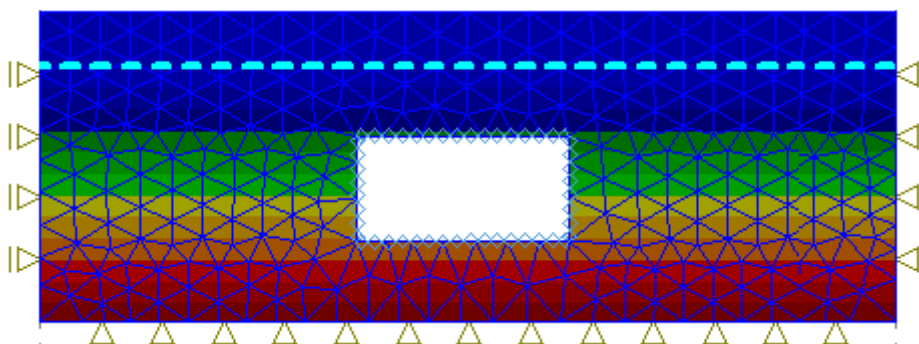
Modeling embankment - activity of embankment body

Using GeoClipboard there's a possibility to copy current profile as sorted interfaces and allow to copy profile to another program. Copied interfaces are corrected to follow specifications to 2D profile entered from top to bottom.

### Activity of Regions Below GWT

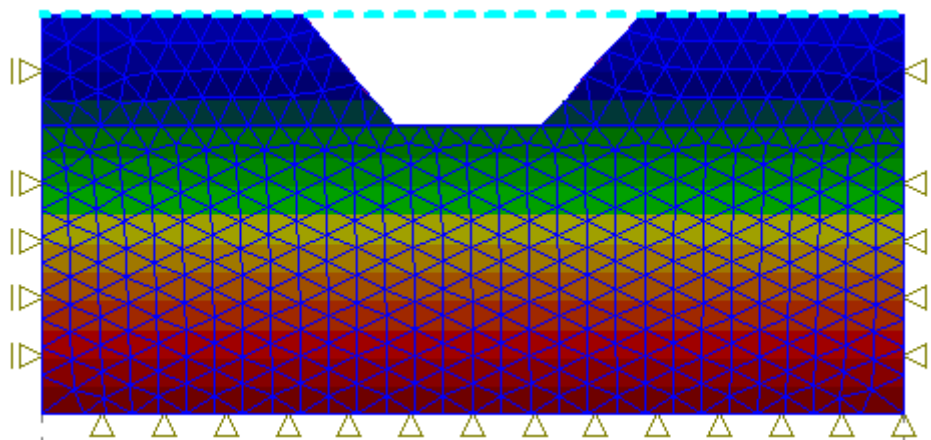
There are two cases to be considered when deactivating a region below the GWT.

**1)** The soil subjected to excavation is **completely enclosed by active beam elements**. The beam is then considered to be impermeable and both the soil and water are removed (removing total stresses - **inactive region is free of water**). Owing to impermeability of the beam elements, the pore pressure distribution remains unchanged, see the figure.



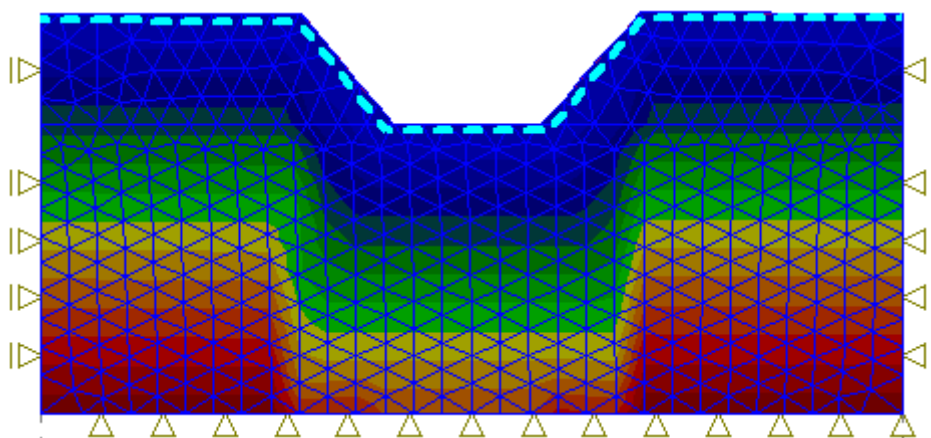
Pore pressure distribution after removing soil from region enclosed by active beams

2) The removed soil **is not enclosed by beam elements**. In such case we assume that water in the excavated region **is still active**. This state is evident from the pore pressure distribution in the figure.



*Pore pressure distribution after removing soil*

Its effect can be removed by **changing the ground water table**.



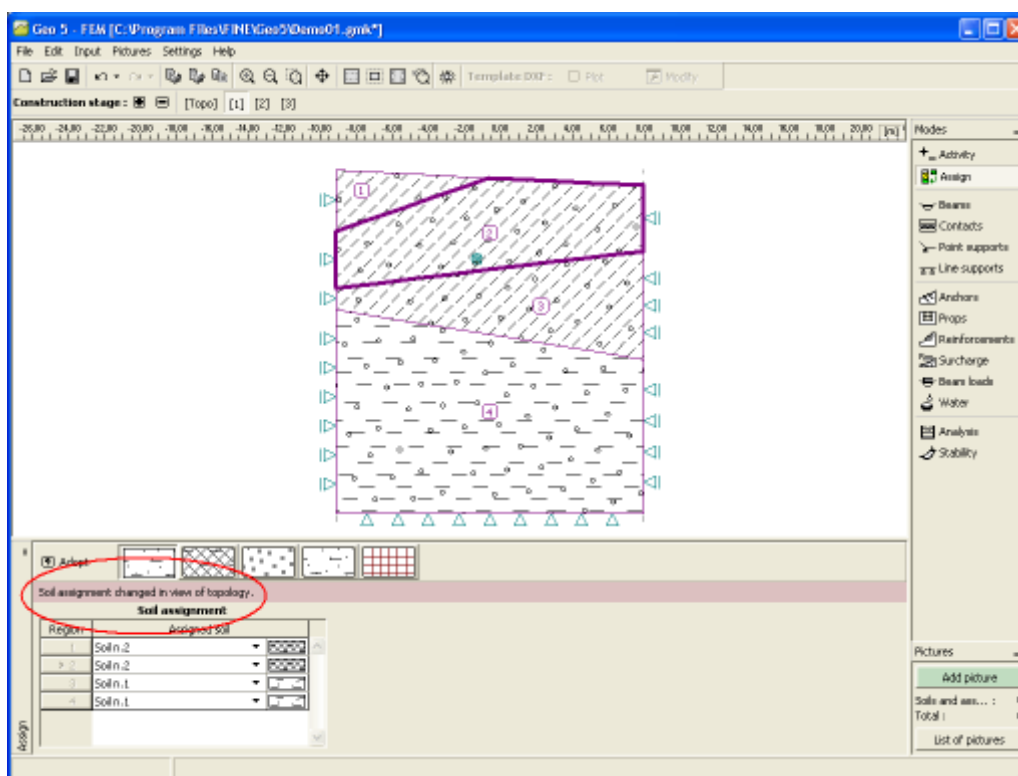
*Pore pressure distribution after modifying the ground water table*

## Assign

The frame "**Assign**" contains a list of layers of the profile and associated soils. Its functions are similar to the case of **assigning soils** in the **topology** regime.

In **calculations stages**, the active soils are assigned to regions rather than to interfaces. The regions are **created automatically** when creating the computational model.

Assign of soils can be copied within all 2D GEO5 programs using "**GeoClipboard**".



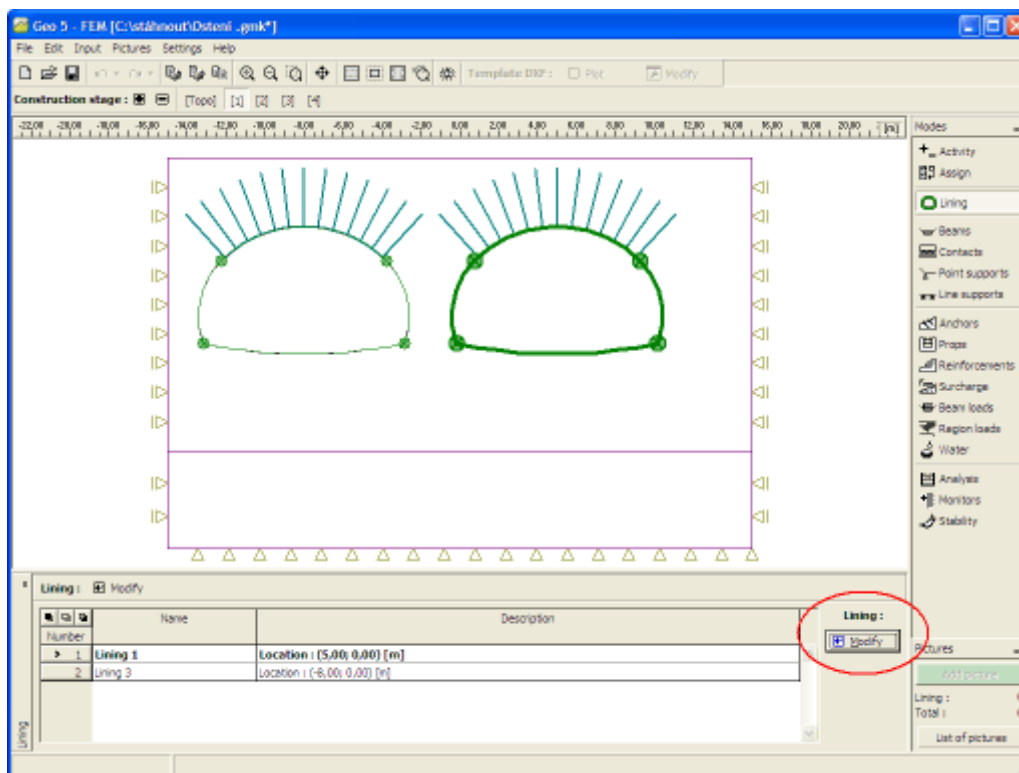
Frame "Assign"

## Lining

The frame "Lining" contains a table with the list of input linings. This frame becomes accessible in the program once the "Tunnel" regime is activated in the frame "Settings". Only editing is allowed in subsequent stages of construction.

To adjust the lining the program launches the module "Lining - FEM". Its function is described in detail in the "Topology" regime. In stages of construction the "Lining - FEM" module contains the "Beams", "Anchors" and "Beam loads" frames.



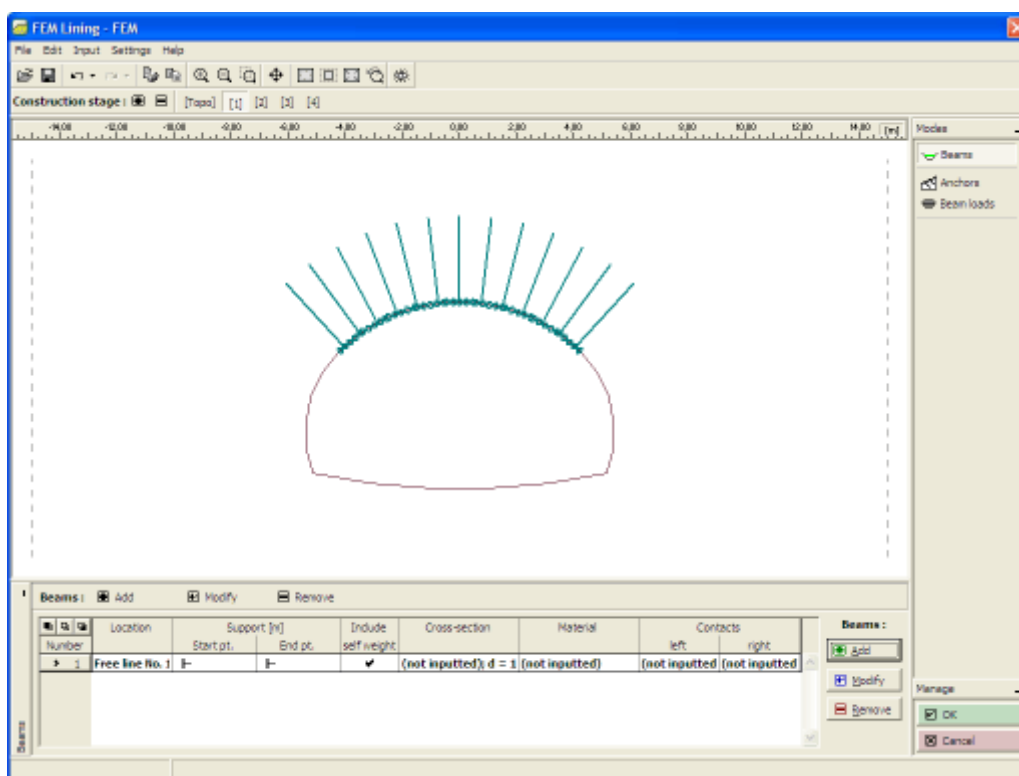


Frame "Lining"

## Beams

The frame "**Beams**" contains a table with the list of input beams. Actions applying to beams are identical to those used in stages of construction in the FEM program, frame "**Beams**".

Types of contacts to introduce contacts on beams are adopted from the FEM program.

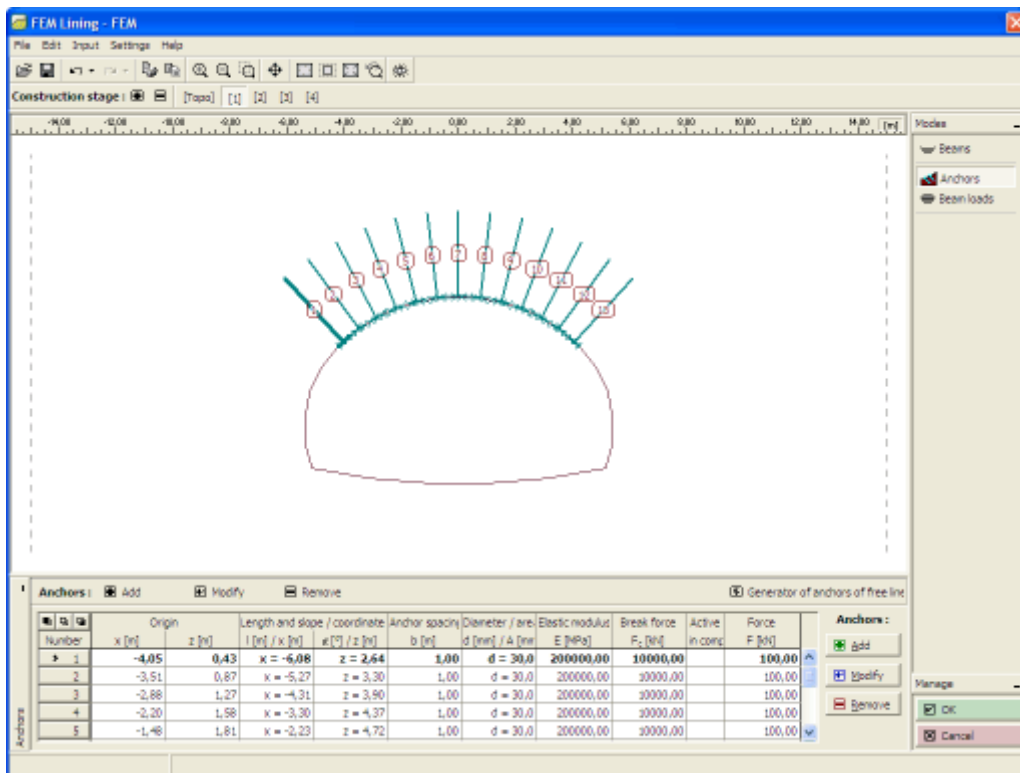


Frame "Beams"

## Anchors

The frame "Anchors" contains a table with the list of input anchors. Actions applying to beams are identical to those used in stages of construction in the FEM program, frame "Anchors".

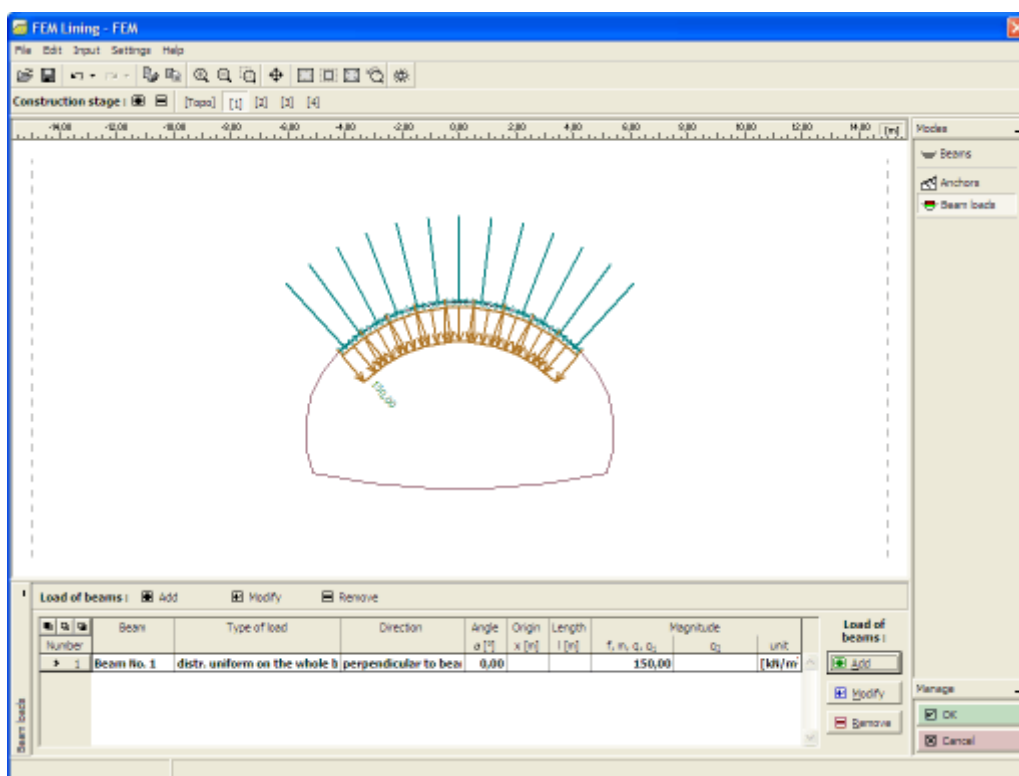
The frame differs by the function on the horizontal tool bar having the "Generator of anchors on free line" button.



Frame "Anchors"

## Beam Loads

The frame "Beam loads" contains a table with the list of input loads. Actions applying to beam loads are identical to those used in stages of construction in the FEM program, frame "Beam loads".



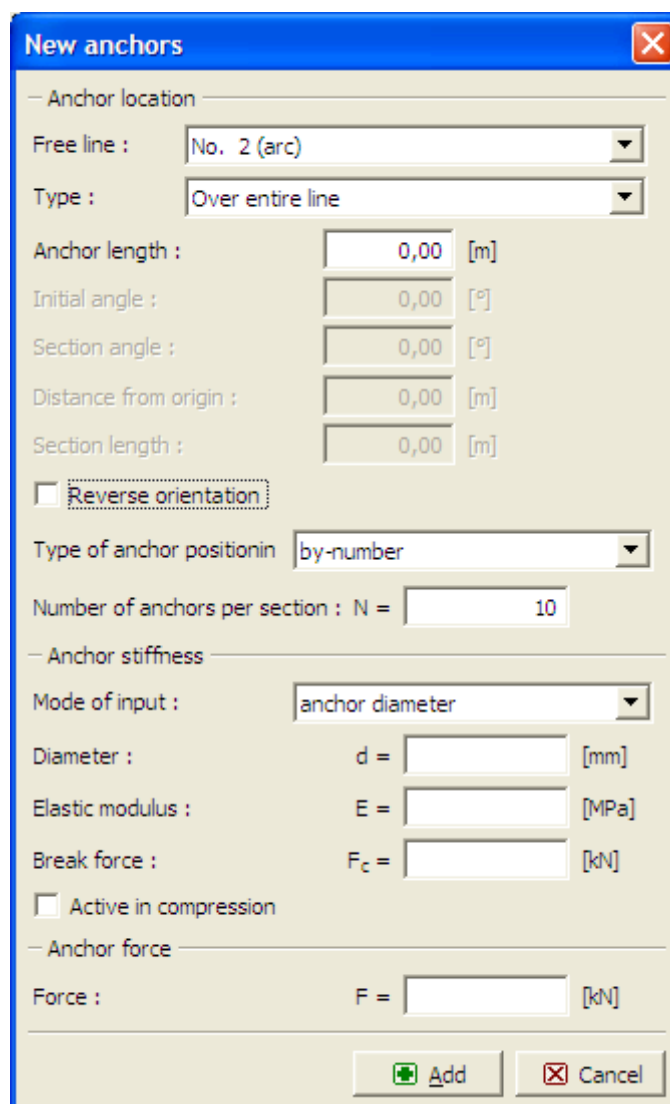
Frame "Beam loads"

## Generator of Anchors on Free Line

Depending on particular parameters the generator creates corresponding elements which are then operated on independently with no possibility for being parametrically modification. If the parameters of generation are acceptable, the program displays during their modification the current graphical representation of generated elements.

The "**New anchors**" dialog window is an extension of the standard dialog window allowing for a uniform distribution of several identical anchors along a line. Spacing of anchors is generated the same way as used in the [generator of anchored regions](#) (over the entire line, over a part defined by the angle or length). There are three options to generate the number of anchors: by the number over a length, by the angle or spacing between individual anchors.

The generated anchors are attached in the FEM program to the free line defined therein.



*Dialog window "New anchors"*

## Beams

The frame **"Beams"** contains a table with the list of beams. Adding (editing) beams is performed in the **"New Beams"** (**"Adjust beam properties"**) dialog window.

Beams can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Beams"**. The following modes are available:

- **Add**                The beam is introduced by clicking the left mouse button at a desired location on the desktop.
- **Adjust**             Clicking the left mouse button on already existing free point opens the **"Adjust beam properties"** dialog window, which allows for modifying its parameters.
- **Remove**            Clicking the left mouse button on already existing beam opens the **beam removal** dialog window - accepting this action removes the selected beam.

- **Location** The beam location is selected from the combo box (mesh line, terrain segment).

The input beams can also be edited on the desktop with the help of **active objects**. The program employs the following **coordinate systems**.

The **beam elements** serve to model **beams, linings, sheeting walls**, etc. **Distribution internal forces** such as moment, normal and shear forces developed along a beam axis can be derived from the beam element end forces.

Beams are assigned to already defined lines (**free lines, terrain segments**) - the corresponding line then represents the **beam axis**. The program offers several basic **types of cross-sections**. Nevertheless, the user is free to introduce the required cross-section independently.

An important step when modeling beams is the definition of **contact elements** characterizing the interface behavior between the beam and the soil. **Contact** (interface) elements can be assigned to **both sides of a beam**. A correct definition of contacts is essential especially when modeling sheeting walls.

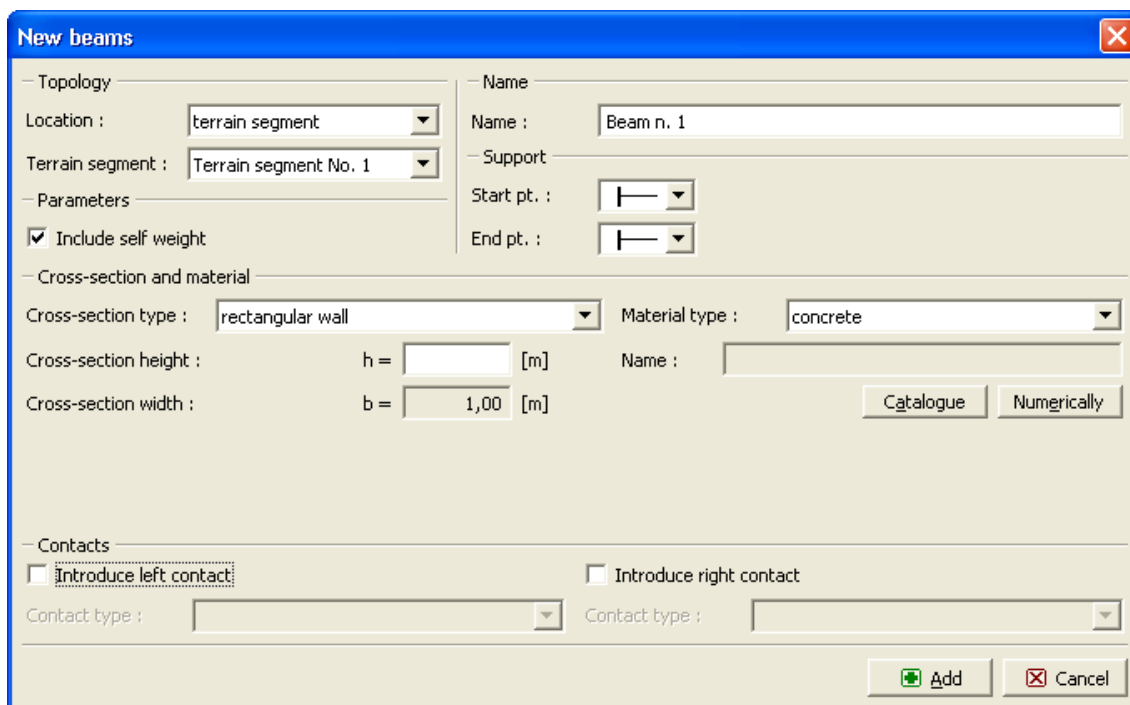
Types of **end points connections** can be specified for each beam.

In subsequent **stages** the beam can be either **strengthened or degraded**.

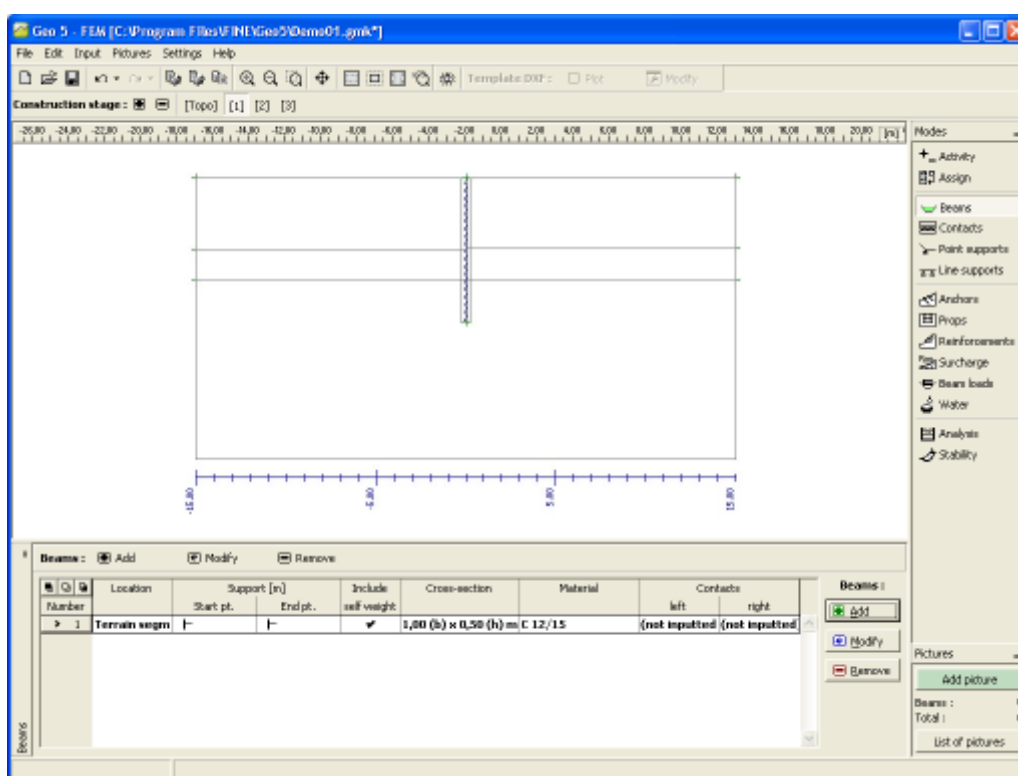
The program automatically includes the **beam self-weight** into the analysis. This feature, however, can be turned off when defining the beam.

Beams are modeled using the **beam elements** with three degrees of freedom at each node.

The beam elements are formulated on the basis of the Mindlin theory. The theory assumes that the plane cross-section normal to the beam axis before deformation remains plane after deformation but not necessarily normal to the deformed beam axis. At present, the internal forces are evaluated at the element nodes and from the beam end forces.



*Dialog window "New beams"*



Frame "Beams"

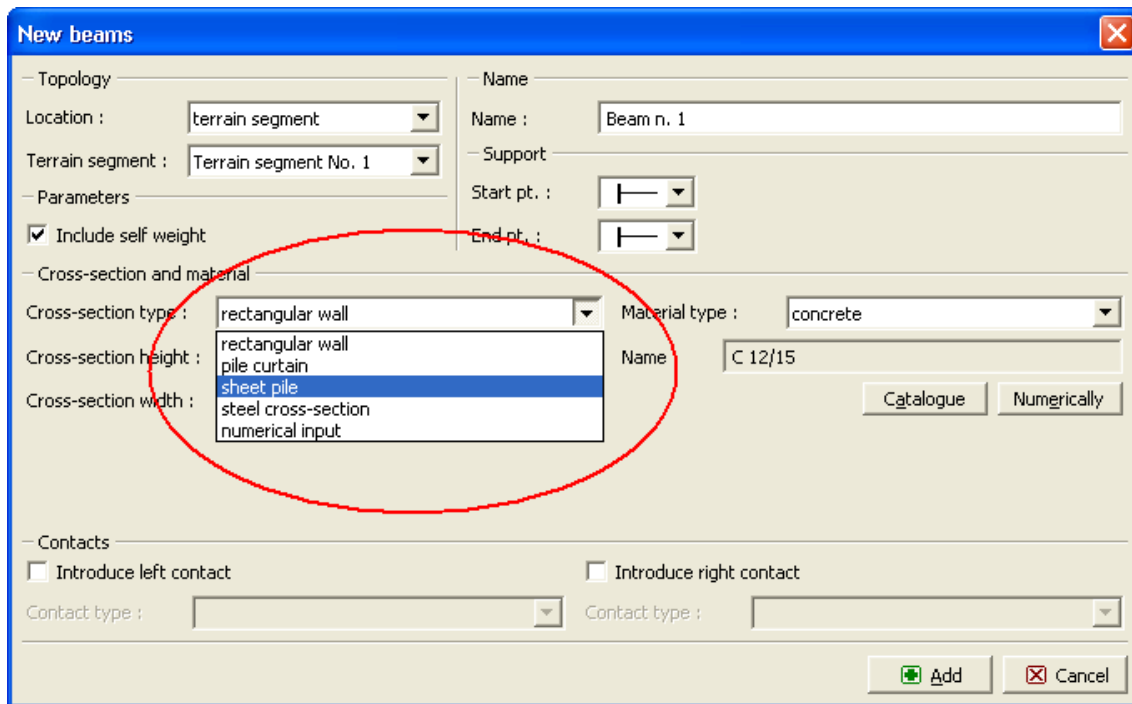
## Types of Cross-Section

The program allows the user to either input the **cross-section parameters digitally** or to choose from one of the predefined types of the cross-section. The type of material of the cross-section is selected from the **catalog of materials** or is introduced digitally using the **editor of materials**. The following types of the beam cross-section are implemented:

- **rectangular concrete wall** - a beam wall thickness must be specified
- **pile wall** - a pile diameter and their spacing must be specified
- **steel sheet pile** - selected from the built-in database
- **steel I cross-section** - a type of cross-section from the built-in database is selected, their spacing must be specified (the type of cross-section is selected from the "**Catalog of cross-sections**", or is defined in the "**Editor of cross-sections**", the type of material is selected from the "**Catalog of materials**" or is specified digitally in the "**Editor of materials**")

All input cross-sections **are automatically recalculated per 1 m (feet) run. The results of internal forces developed along the beams are also presented per 1 m (feet) run of a structure.** Thus if necessary, for piles or I cross-sections they must be adjusted depending on their spacing by the user.




Providing you have your own database of **sheet piles**, which is not yet built-in the program, we will be happy to implement it. You may reach us at **hotline@fine.cz**.



*Dialog window "New beams" - selection of the type of cross-section*

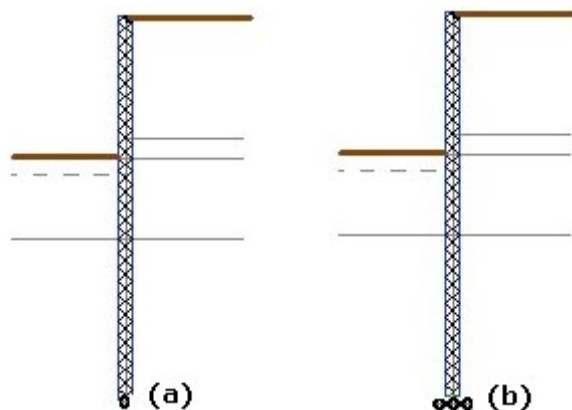
## Beam End-Points Connection

The program allows for three types of beam end-points connection:

-  **fixed**      standard type
-  **hinge**      (is used to introduce an internal hinge in between beams - locations with zero bending moment)
-  **foot**

**The foot is a special type** of a beam end-point support in the soil. It is applicable for **the beam end-point located in the soil body**. When the fixed type of connection is assumed the beam and the soil element are connected at one point (a singular connection) often causing evolution of plastic strains in the surrounding soil and loss of convergence. **The foot allows for more realistic redistribution of contact stresses** and prevents the beam from "penetrating" into the soil, consequently stabilizing the convergence process. **By default the foot length** is assumed to be equal to the beam width - it can be arbitrarily adjusted (for example to enlarge the pile heel).





Connection (a) without (b) with a foot

## Degradation and Strengthening of Beams

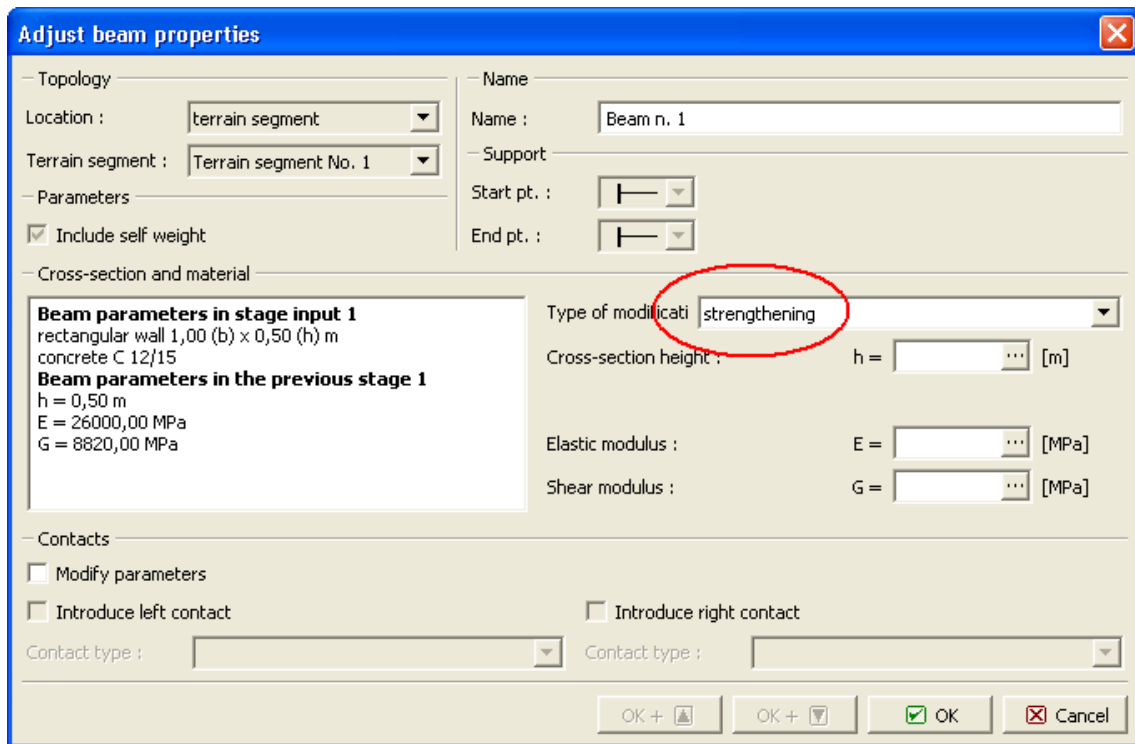
In subsequent stages the input beams cannot be edited in a standard way. Therefore, one of the following options must be selected to modify them:

- removing the selected beam from the analysis
- degrading the selected beam (applicable only in the "Tunnels" regime)
- strengthening the selected beam cross-section
- modifying the beam contact properties

The type of modification is selected from the "**Adjust beam properties**" dialog window.

A degree of **beam degradation** is specified in percentage, one hundred percent corresponds to beam removal.

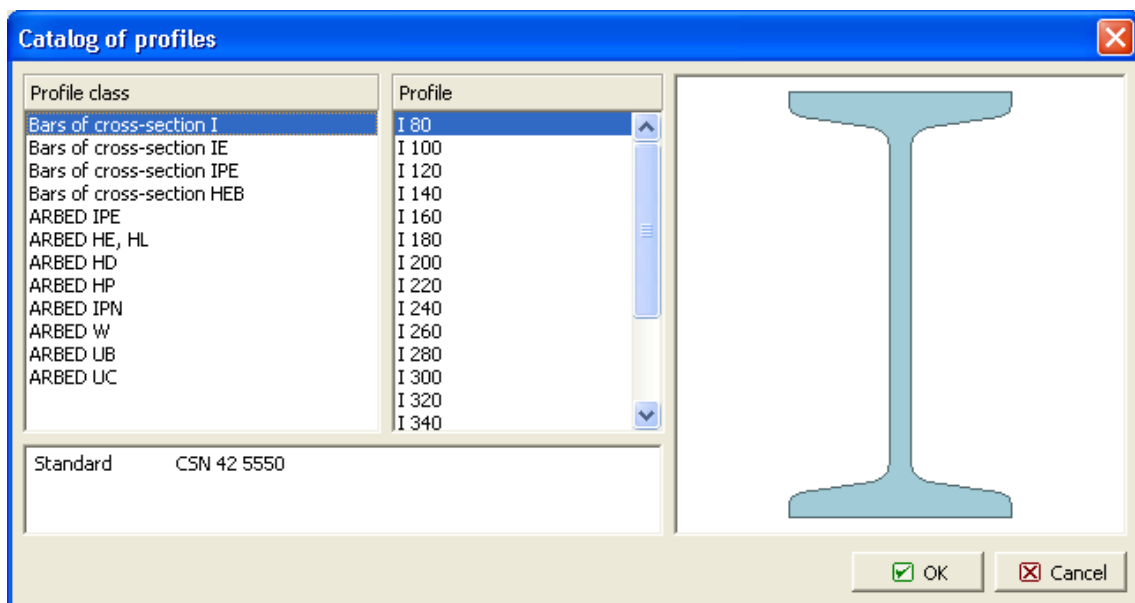
**Strengthening a beam element** with a rectangular cross-section can be achieved by enlarging its width (e.g., increasing the shotcrete thickness). Other cross-sections are modified by directly inputting new (larger) values of the cross-section parameters.



Dialog window "Adjust beam properties" - beam strengthening

## Catalog of Profiles

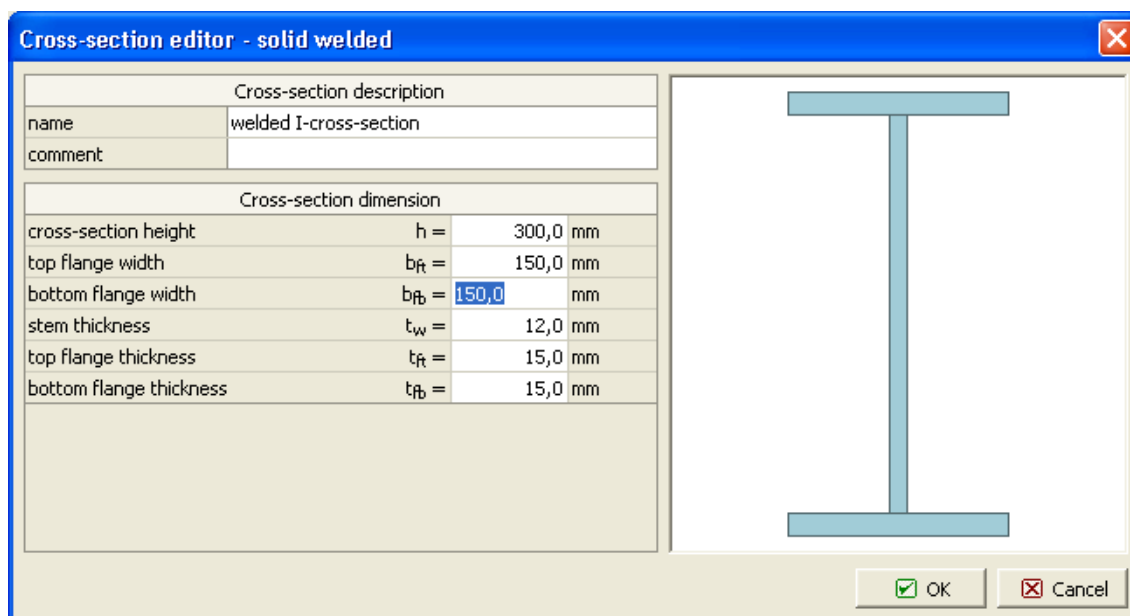
In the case of steel cross-sections the program allows for choosing a particular cross-section from the catalog of profiles. Only the type of cross-section has to be specified in the dialog window. The type of material of the cross-section is selected similarly to other cross-sections (rectangular wall, pile wall, sheet pile...) from the "Catalog of materials", or defined in the "Editor of materials". The type of cross-section (beam) is selected in the "New beams" dialog window.



*Dialog window "Catalog of profiles"*

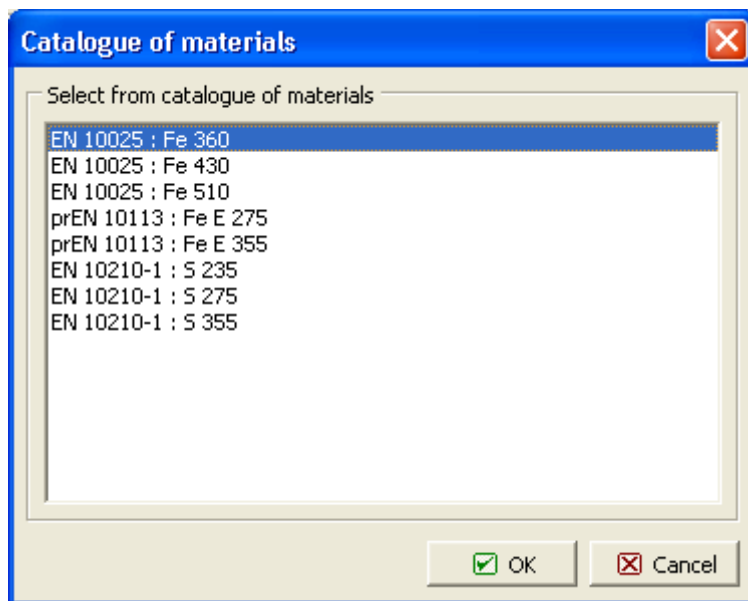
## Cross-Section Editor

In the case of steel cross-section the program allows for introducing the user defined cross-section. Only the shape of cross-section has to be specified in the dialog window. The type of material of the cross-section is selected similarly to other cross-sections (rectangular wall, pile wall, sheet pile...) from the "Catalog of materials", or defined in the "Editor of materials". The type of cross-section (beam) is selected in the "New beams" dialog window.

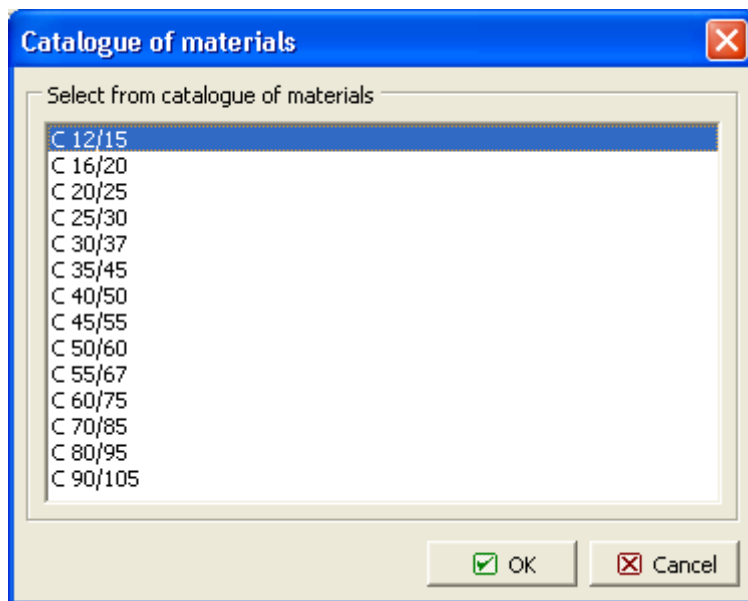
*Dialog window "Cross-section editor - solid welded"*

## Catalog of Materials

The program contains a built-in catalog of materials for concrete and steel. Only the type of material has to be specified in the dialog window. The shape of cross-section is selected from the "Catalog of profiles", or defined in the "Cross-section editor". For other types of cross-sections (rectangular wall, pile wall, sheet pile...) the type of cross-section is selected in the "New beams" dialog window.



*Dialog window "Catalog of materials" - steel*



*Dialog window "Catalog of materials" - concrete*

## Editor of Materials

Apart from using the "Catalog of materials" the program allows the user to enter the material parameters for steel and concrete digitally. Only the type of material (material parameters) has to be specified in the dialog window. The shape of cross-section is selected from the "Catalog of profiles", or defined in the "Cross-section editor". For other types of cross-sections (rectangular wall, pile wall, sheet pile...) the type of cross-section is selected in the "New beams" dialog window.

**Editor of material - Structural steel**

Description of material

Name:

Characteristics of material

General material characteristics		
Elasticity modulus	E =	MPa
Shear modulus	G =	MPa
Coefficient of thermal expansion	$\alpha$ =	1/K
Specific weight	$\gamma$ =	kN/m <sup>3</sup>

Special material characteristics		
Yield strength	$f_y$ =	MPa
Ultimate tensile strength	$f_u$ =	MPa

OK  Cancel

*Dialog window "Editor of material - Structural steel"*

**Editor of material - Concrete**

Description of material

Name:

Characteristics of material

General material characteristics		
Elasticity modulus	$E_{cm}$ =	26000,00 MPa
Shear modulus	G =	8820,00 MPa
Coefficient of thermal expansion	$\alpha$ =	0,000010 1/K
Specific weight	$\gamma$ =	25,00 kN/m <sup>3</sup>

Special material characteristics		
Cylinder compressive strength	$f_{ck}$ =	12,00 MPa
Tensile strength	$f_{ct}$ =	1,60 MPa

OK  Cancel

*Dialog window "Editor of material - Concrete"*

## Contacts

The **frame "Contacts"** contains a **table** with the list of contacts. Adding (editing) contacts is performed in the **"New contacts"** dialog window.

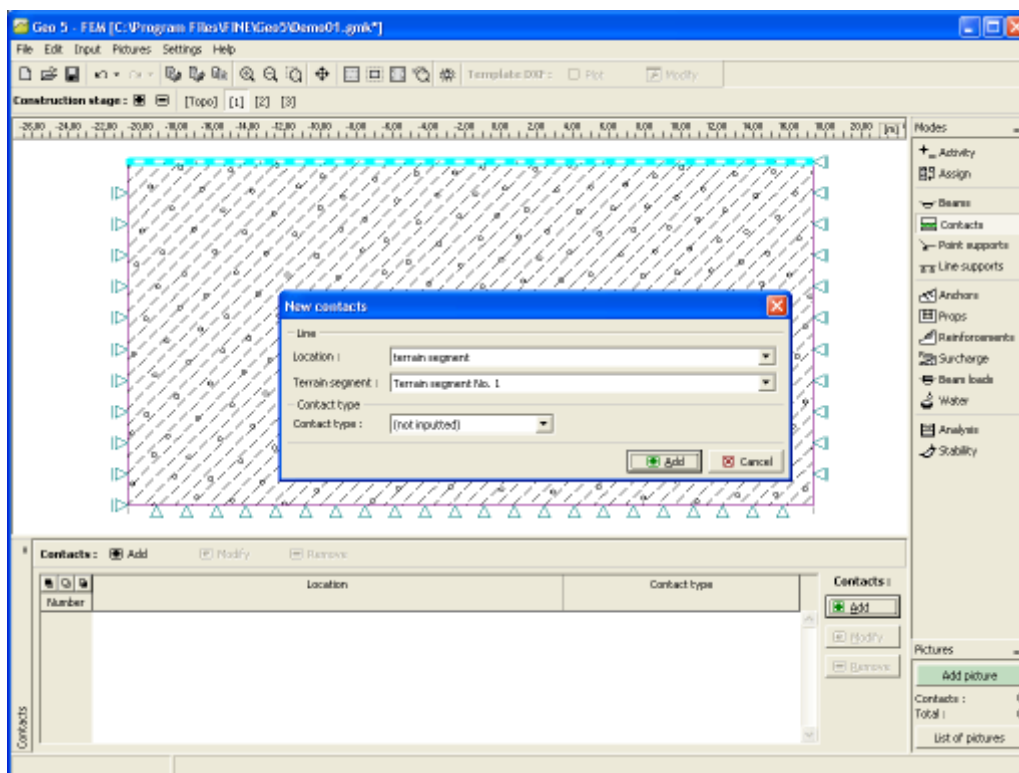
The **contact elements** are used in applications that require a proper representation of structure-soil interaction. They can be further used to model joints or interfaces of two distinct

materials (soil - rock interface). Contacts are assigned to already defined lines - free lines or mesh lines (interfaces). The contact is **defined by its type**.

Contacts can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Contacts**". The following modes are available:

- **Add** The contact is introduced by clicking the left mouse button at a desired location on the desktop.
- **Adjust** Clicking the left mouse button on already existing contact opens the "**Adjust contacts properties**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing contact opens the **contact removal** dialog window - accepting this action removes the selected contact.
- **Location** The contact location is selected from the combo box (mesh line, terrain segment).

The input contacts can also be edited on the desktop with the help of **active objects**.



Frame "Contacts"

## Contacts and Beams (Water Flow)

The **frame "Contacts"** ("**Beams**") contains (in mode "**Water flow**") a **table** with the list of contacts (beams). Adding (editing) contacts (beams) is performed in the "**New contacts**" ("**New beams**") **dialog window**.

The **contact elements** are used in applications that require a proper representation of structure-soil interaction. They can be further used to model joints or interfaces of two distinct

materials (soil - rock interface). Contacts are assigned to already defined lines - free lines or mesh lines (interfaces). The contact is defined by its type.

Contacts can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Contacts**". The following modes are available:

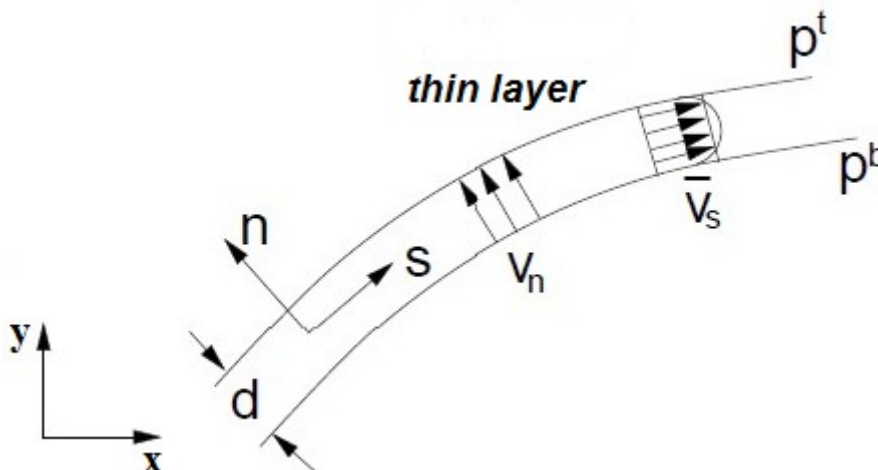
- **Add**                      The contact is introduced by clicking the left mouse button at a desired location on the desktop
- **Adjust**                    Clicking the left mouse button on already existing contact opens the "**Adjust contacts properties**" dialog window, which allows for modifying its parameters.
- **Remove**                  Clicking the left mouse button on already existing contact opens the **contact removal** dialog window - accepting this action removes the selected contact.

The input contacts can also be edited on the desktop with the help of **active objects**.

Beam or contact element can be defined as:

- **permeable**
- **impermeable**
- **partially permeable**

Contact elements allow us to model a certain barrier for flow in the soil body. Consider for example a sheeting wall represented in the stress analysis by beam elements. The sheeting wall anchored into the inside region can be considered either as fully permeable, or fully impermeable or partially permeable. Although the first two cases can also be treated using contact elements placed along the corresponding line, they are handled by the program automatically without needing these elements. The third case represents a problem of flow in a thin zone having a given thickness  $d$ , see figure:



*Partially permeable contact*

Corresponding fluxes in the tangent direction ( $s$ -direction)  $q_s$  and normal direction ( $n$ -direction)  $q_n$  are given by:

$$q_s = -k_s \frac{1}{2} \frac{\partial (h^t + h^b)}{\partial_s}$$

$$q_n = -k_n \frac{h^t + h^b}{d}$$

Defining contact elements therefore requires inputting the following parameters:

$k_s$  - permeability in tangent direction (permeability longitudinal), [m/day]

$k_n$  - permeability in normal direction (permeability transverse), [m/day]

## Point Supports

The frame "**Point supports**" contains a table with the list of point supports. Adding (editing) point supports is performed in the "**New point supports**" dialog window.

Point supports can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Point supports**". The following modes are available:

- **Add**                The point support is introduced by clicking the left mouse button at a desired location on the desktop. The required parameters are introduced in the "**New point supports**" dialog window.
- **Adjust**             Clicking the left mouse button on already existing point support opens the "**Adjust point supports properties**" dialog window, which allows for modifying its parameters.
- **Remove**            Clicking the left mouse button on already existing point support opens the **point support removal** dialog window - accepting this action removes the selected point support.

The input point supports can also be edited on the desktop with the help of **active objects**. The program employs the following **coordinate systems**.

The program contains a built-in automatic generator of **standard boundary conditions**.

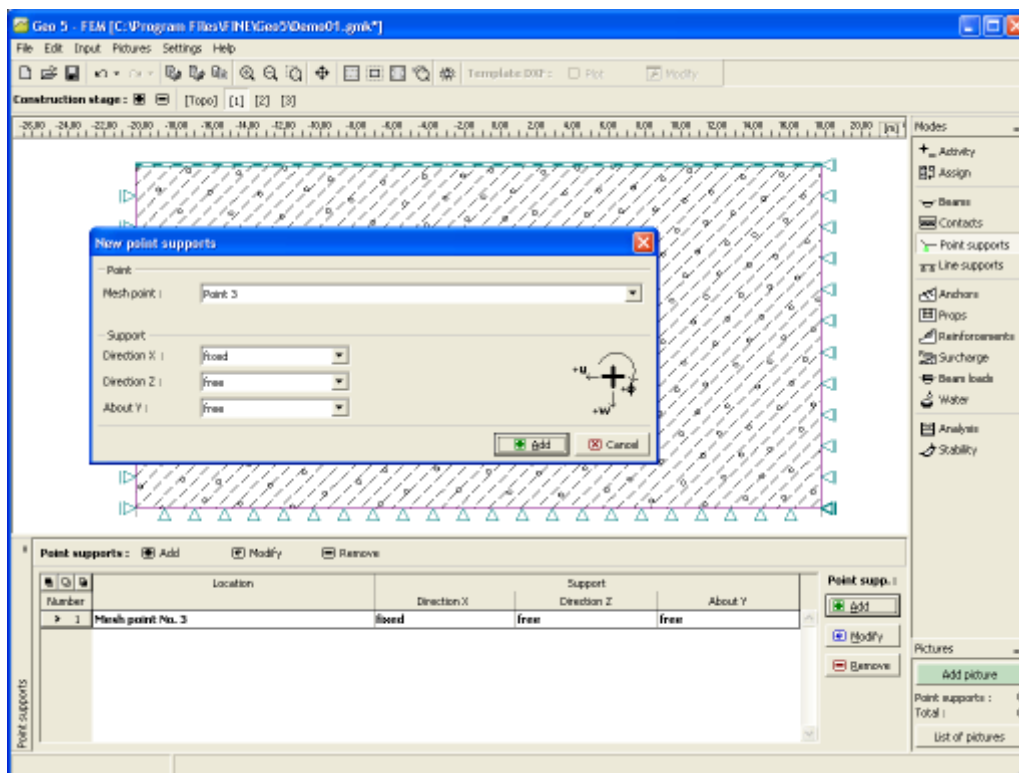
Therefore, in most problems the **boundary (support) conditions are not required to be specified**.

**The following types of point supports** are considered:

- **free**
- **fixed**
- **spring**
- **prescribed deformation**

Supports are defined in **the global coordinate system**.





Frame "Point supports"

## Point Flow

The **frame "Point flow"** contains a **table** with the list of point flows. Adding (editing) a point flow is performed in the **"New point flow" dialog window**.

A point flow can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Point flow"**. The following modes are available:

- **Add**                    The point flow is introduced by clicking the left mouse button at a desired location on the desktop. The required parameters are introduced in the **"New point flow"** dialog window.
- **Adjust**                 Clicking the left mouse button on already existing point flow opens the **"Adjust point flow properties"** dialog window, which allows for modifying its parameters.
- **Remove**                Clicking the left mouse button on already existing point support opens the **point flow removal** dialog window - accepting this action removes the selected point flow.

The input point flows can also be edited on the desktop with the help of **active objects**.

The following boundary conditions can be specified:

### a) Pore pressure at a point

- Numerically - the value of pore pressure at a given point is specified [*kPa, ksf*]
- By specifying the location of ground water table (total head) - coordinate of GWT is specified

## b) Point inflow/outflow

Pumping/injection rate is specified [ $m^3/day/m$ ,  $ft^3/day/ft$ ]

## Line Supports

The frame "Line supports" contains a table with the list of line supports. Adding (editing) line supports is performed in the "New line supports" dialog window.

Line supports can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Line supports". The following modes are available:

- **Add** The line support is introduced by clicking the left mouse button at a desired location on the desktop. The required parameters are introduced in the "New line supports" dialog window.
- **Adjust** Clicking the left mouse button on already existing line support opens the "Adjust line supports properties" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing line support opens the **line support removal** dialog window - accepting this action removes the selected line support.
- **Location** The line support location is selected from the combo box (free line, terrain segment, mesh line).

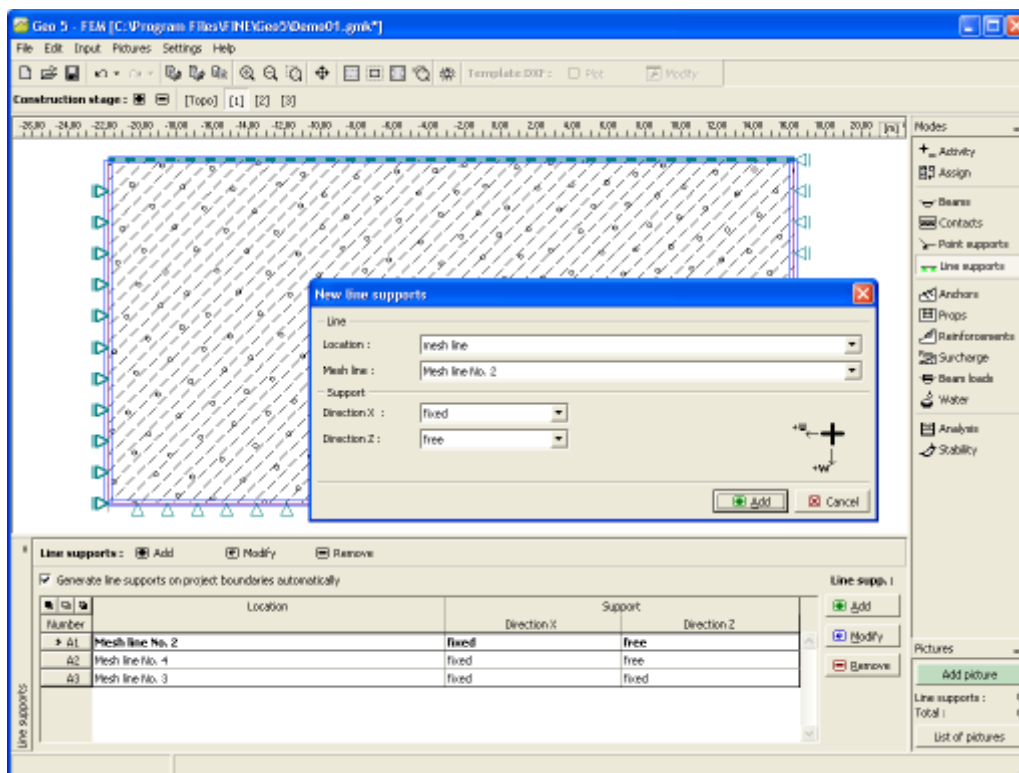
The input line supports can also be edited on the desktop with the help of **active objects**. The program employs the following **coordinate systems**.

The program contains a built-in automatic generator of **standard boundary conditions**. Therefore, in most problems the **boundary conditions are not required to be specified**.

When assigning supports to a line it is first necessary to select the type of line (**free line, interface, mesh line**).

**The following types of line supports** are considered:

- free
- fixed
- deformation



Frame "Line supports"

## Line Flow

The frame "Line flow" contains a table with the list of line flows. Adding (editing) a line flow is performed in the "New line flow" dialog window.

A line flow can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "Line flow". The following modes are available:

- **Adjust** Clicking the left mouse button on already existing line flow opens the "Adjust line flow properties" dialog window, which allows for modifying its parameters.

The input line flows can also be edited on the desktop with the help of active objects.

**Flow boundary** conditions **must be defined on all boundary lines**. The following boundary conditions can be specified:

**a) Impermeable**

**b) Permeable**

Pore pressure on a given line is equal to zero

**c) Pore pressure**

- distribution of pore pressure  $p$  can be specified numerically
- distribution of pore pressure can be specified by inputting the location of ground water table (by prescribing the total head  $h$ )

**d) Inflow/outflow on a line  $q$**  - it is specified in velocity units e.g. [m/day, ft/day] - the flow velocity into/out of the region is specified. The default setting corresponds to an impermeable

boundary for which  $q = 0$ .

**e) Seepage surface** - this boundary condition is introduced providing the boundary cannot be uniquely divided into the part with prescribed pore pressure and the part with prescribed inflow/outflow (the exit point is not known). In such a case the analysis is performed in two steps. In the first step the program locates the exit point. The actual flow analysis with known boundary conditions is then carried out in the second step. In some cases both steps must be repeated several times. When enhanced input is considered the program requires entering a fictitious permeability  $k_v$  in units [ $m/day$ ]. This is essentially a penalty term, a sufficiently large number in general, ensuring that along an impermeable boundary the value of total  $h$  will be equal to the  $y$ -coordinate of a given point ( $q = 0$ ). For a part of boundary with no flow condition we have  $k_v = 0$ . Variables  $q$  and  $h$  are then related by:

$$\bar{q}_n = k_v (h - y) \quad \text{if } h > 0 (S = 1) \text{ inside soil body}$$

$$\bar{q}_n = 0 \quad \text{if } h < 0 (S < 1) \text{ inside soil body}$$

**Note:** If in case of transient flow we directly define in the first calculation stage the location of the ground water table (phreatic surface) as an initial condition, we should define in the next calculation step along the boundaries below the water level in the region of fully saturated soil a boundary with the prescribed pore pressure having a corresponding value and not the seepage surface. In case of seepage surface the program would immediately label this boundary as a boundary with zero pore pressure and not the originally assumed boundary with the pore pressure distribution in accordance with the expected height of GWT. In such a case, the analysis will not converge, because water, instead of flowing out of the domain at  $p = 0$ , will have tendency to flow in, which not realistic.

## Anchors

The frame "**Anchors**" contains a table with the list of anchors. Adding (editing) anchors is performed in the "**New anchors**" dialog window ("**Adjust anchor properties**").

Anchors can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Anchors**". The following modes are available:

- **Add** By clicking the left mouse button on the desktop we define the starting and the end point of an anchor. Exploiting the function of **grid** may simplify this step. The starting point is hooked to the ground and its coordinates are rounded up to two significant digits - using the mouse or keyboard is therefore identical.
- **Adjust** Clicking the left mouse button on already existing anchor opens the "**Adjust anchor properties**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing anchor opens the **anchor removal** dialog window - accepting this action removes the selected anchor.

The input anchors can also be edited on the desktop with the help of **active objects**. The program employs the following **coordinate systems**.

The anchor head (starting point) can be **automatically hooked** to the ground, an arbitrary interface or opening (tunnel lining). The anchor head is then automatically positioned in to the

intersection of the anchor line determined by the input points and the selected line. The anchor can also be introduced directly by specifying coordinates of the two end points.

Anchors as stabilizing or reinforcing elements are represented by **elastic tensile-compressive bar element** with constant normal stiffness. The maximum allowable tensile force the element can sustain controls tensile failure of the anchor. The bar element is anchored into the soil only at its starting and end points. No mutual interaction between the soil and the anchor along the anchor length is considered.

Anchors are defined by their starting and end points and by their stiffness. The program automatically links the anchor element degrees of freedom to the actual degrees of freedom of the predefined finite element mesh. Therefore, the anchor can be introduced **anywhere in the structure**.

The **anchor stiffness** is specified in terms of the elastic modulus and its area. The program makes it also possible to enter the anchor diameter - the area is then determined automatically. In **stability analysis problems** the anchor stiffness is not considered. Its action is realized only through the prestress force introduced automatically as external compressive force acting at the anchor head.

Other important parameters are the **prestress force** and the **tensile strength** (the anchor breaks when the tensile strength is exceeded). For elements with no prestress the prestress force is set equal to zero. Sufficiently large value of the anchor tensile strength may be specified to avoid anchor failure.

By default the anchor **does not support a compressive force** - anchor elements loaded in compression during a certain stage of calculation are temporarily disabled. If tension occurs in subsequent analysis run (due to change in load, geometry or material parameters of soil), the program automatically introduces these elements back into the analysis. The program makes also possible to include compressive response of an anchor. However, for elements loaded primarily in compression we recommend to define these elements as **props**.

The anchor deforms during analysis. Such deformation together with the deformation of the surrounding soil may cause **reduction of the specified prestress force** in the anchor. Providing we wish to achieve a specific prestress force in the anchor, it is necessary to either post-stress the anchor to a given value in the next calculation stage or to use a sufficiently large magnitude of the prestress force right from the beginning to compensate for a possible drop (the resulting anchor force after completion of the calculation step is displayed at the anchor head below the prescribed prestress force).

In subsequent **stages** the program allows only for anchor post-stressing - change of the initial prestress force, or for removing the anchor from the analysis.

Introducing pre-stressed anchors into the soil may lead to plastic deformation of the soil in the vicinity of the anchor head or root. Some **modifications of the original input** are than required to avoid often encountered loss of converge.

**New anchors**

— Anchor location

Origin : attach to terrain of curr. stage ▾

x = 0,00 [m]

z = -2,90 [m]

End point : input length and slope of anchor ▾

Length : l = 12,00 [m]

Slope :  $\alpha$  = 15,00 [°]

Anchor spacing : b = 1,00 [m]

— Anchor stiffness

Mode of input : anchor diameter ▾

Diameter : d = 10,0 [mm]

Elastic modulus : E = 210000,00 [MPa]

Tensile strength :  $F_c$  = 185,00 [kN]

Active in compression

— Anchor force

Force : F = 145,00 [kN]

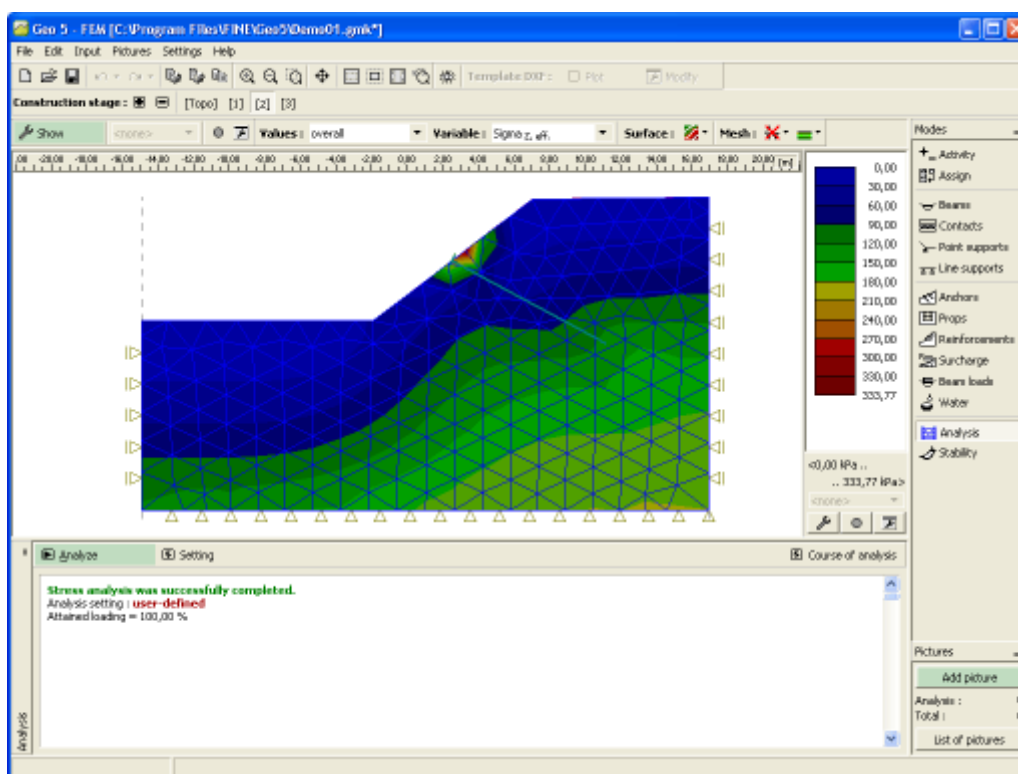
Anchor input

## Anchor End Points

Introducing pre-stressed anchors into the soil may lead to **plastic deformation** of the soil in the vicinity of the anchor head or root - the analysis then often fails to converge.

In such a case we recommend the following modifications of the original input:

- to place a **beam element** under the anchor head (this results into a better transition of load into the soil),
- to place the anchor root into a **sufficiently stiff soil** (use the elastic or modified elastic material model for the soil layer around the anchor).

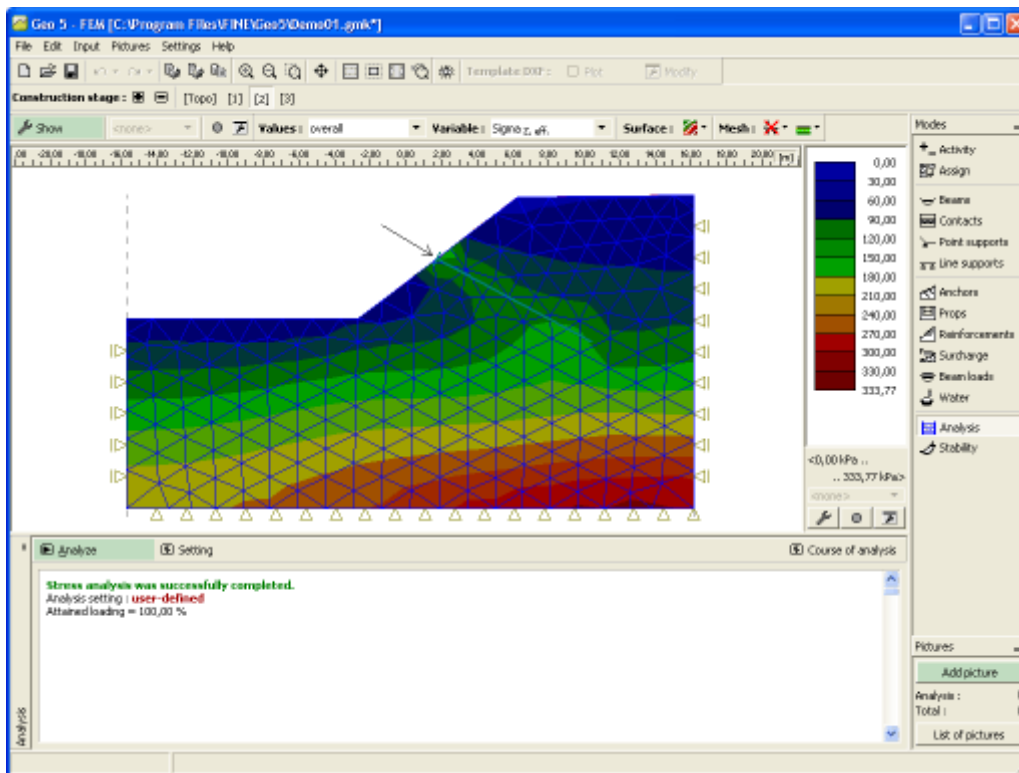


*Plastic regions in the vicinity of anchor head or root*

## Anchors in the Stability Analysis

When performing the **stability analysis** the actual pre-stressed anchor is automatically replaced by corresponding **compressive point forces** acting at the anchor head.

The soil at the point of the applied force may, however, undergo plastic deformation. One should therefore carefully assess the resulting distribution of plastic strains. Note that the localization of equivalent plastic strain identifies the location of the potential slip surface. Therefore, if the plastic strains at the anchor head become decisive, it is necessary to introduce some **modifications of the original input**.



*Modeling anchor in the slope stability analysis*

## Props

The frame "**Props**" contains a table with the list of anchors. Adding (editing) props is performed in the "**New props**" dialog window ("**Adjust prop properties**").

Props can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Props**". The following modes are available:

- **Add** By clicking the left mouse button on the desktop we define the starting and the end point of a prop. Exploiting the function of **grid** may simplify this step.
- **Adjust** Clicking the left mouse button on already existing prop opens the "**Adjust prop properties**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing prop opens the **prop removal** dialog window - accepting this action removes the selected prop.

The input props can also be edited on the desktop with the help of **active objects**. The program employs the following **coordinate systems**.

The prop end points can be **automatically hooked** to the ground, an arbitrary interface or opening (tunnel lining). These points are then automatically positioned in to the intersections of the prop line determined by the input points and the selected lines. The prop can also be introduced directly by specifying coordinates of the two end points.

Props are represented by **elastic compressive bar element** with constant normal stiffness. The props can sustain only compressive load. When found in tension they are removed from the analysis.



The prop is linked to the finite element mesh in its two end points. No interaction is considered between the soil and the prop along its length when placed into the soil.

Props are defined by their starting and end points and by their stiffness. The program **automatically** links the **prop element degrees** of freedom to the actual degrees of freedom of the predefined finite element mesh. Therefore, the prop can be introduced anywhere in the structure.

The **prop stiffness** is specified in terms of the elastic modulus and its area. The program makes also possible to enter the prop diameter - the area is then determined automatically.

In subsequent **stages** the prop cannot be edited - it can be either removed or input again.

**New prop**

— Prop location —

Point 1 : attach to interface No. 1

x = -2,69 [m]

z = 0,00 [m]

Point 2 : absolute position

x = 4,34 [m]

z = 5,72 [m]

Strut spacing : b = 1,00 [m]

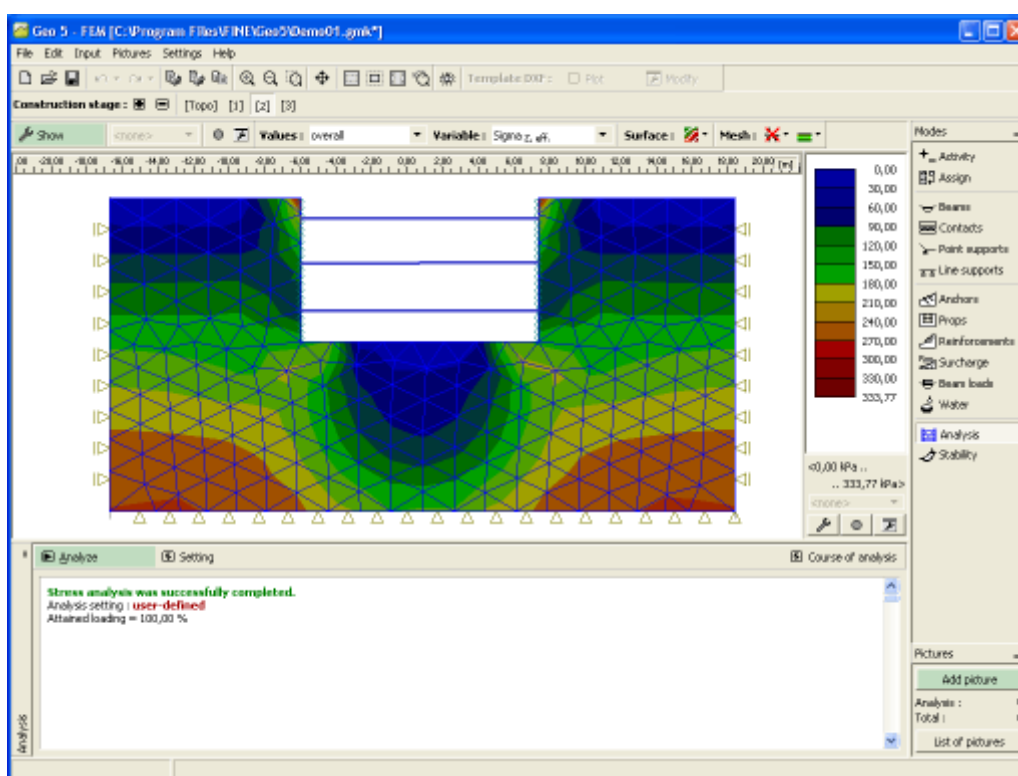
— Prop stiffness —

Area : A = 200,0 [mm<sup>2</sup>]

Elastic modulus : E = [MPa]

OK  Cancel

*Prop input*



*Props - Analysis*

## Reinforcements

The frame "**Reinforcements**" contains a table with the list of reinforcements. Adding (editing) reinforcements is performed in the "**New reinforcements**" dialog window ("**Modify reinforcement parameters**").

Reinforcements can also be introduced **using mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Reinforcements**". The following modes are available:

- **Add** By clicking the left mouse button on the desktop we define the starting and the end point of a reinforcement. Exploiting the function of **grid** may simplify this step.
- **Adjust** Clicking the left mouse button on already existing reinforcement opens the "**Modify reinforcement parameters**" dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing reinforcement opens the **reinforcement removal** dialog window - accepting this action removes the selected reinforcement.

The input reinforcements can also be edited on the desktop with the help of **active objects**. The program employs the following **coordinate systems**.

The reinforcement end points can be **automatically hooked** to the ground, an arbitrary interface or opening (tunnel lining). These points are then automatically positioned in to the intersections of the prop line determined by the input points and the selected lines. The reinforcement can also be introduced directly by specifying coordinates of the two end points.

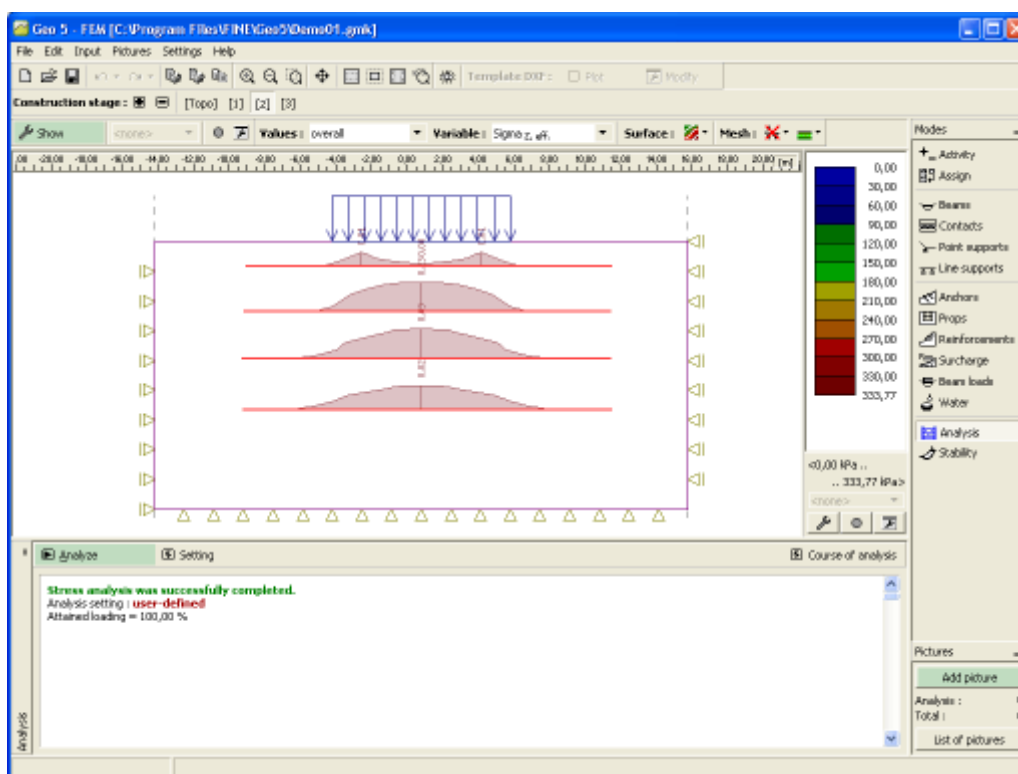
Reinforcements are **tensile reinforcing elements** (geotextiles, geodrids), which are defined by their starting and end points and their stiffness.

Unlike anchors or props, the reinforcement **is linked** to an underlying finite element mesh **along its entire length**. However, similar to anchors the program introduces the reinforcement end points into the finite element mesh automatically so the reinforcement can be specified anywhere within the mesh. Similar to anchors the reinforcement is modeled by a tensile/compressive bar element with the possibility of **transmitting only normal force**. Owing to its geometrical characteristics, the reinforcement calls for the input of the **cross-sectional stiffness taken per 1 m (foot) run of its width**. The user should contact the manufacturer for this information.

In subsequent **stages** the reinforcement cannot be edited - it can be only removed.

*Reinforcement input*

The program allows us to consider the reinforcement also in compression - by default however, the **part of reinforcement found in compression is disabled** for the analysis. This state is simulated in the figure showing the distribution of normal tensile forces over active parts of individual reinforcements. The compressive part of the reinforcement is **temporarily excluded** from the analysis. Similar to anchors, however, it can be automatically activated once loaded again in tension.

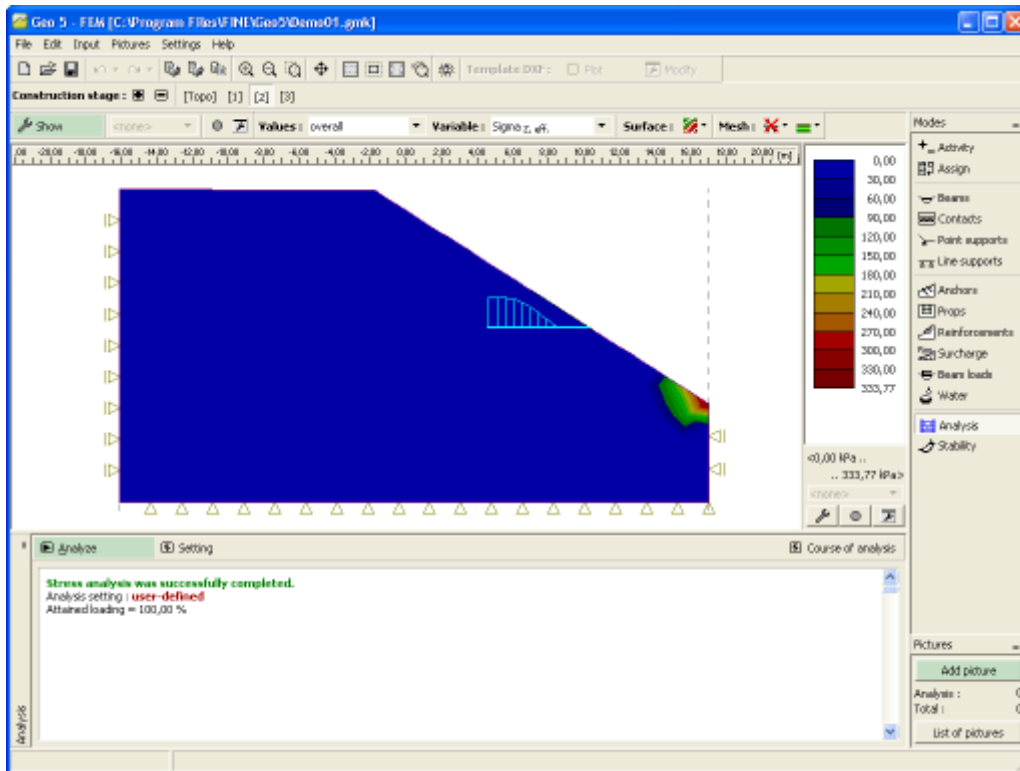


*Tensile stress in reinforcements*

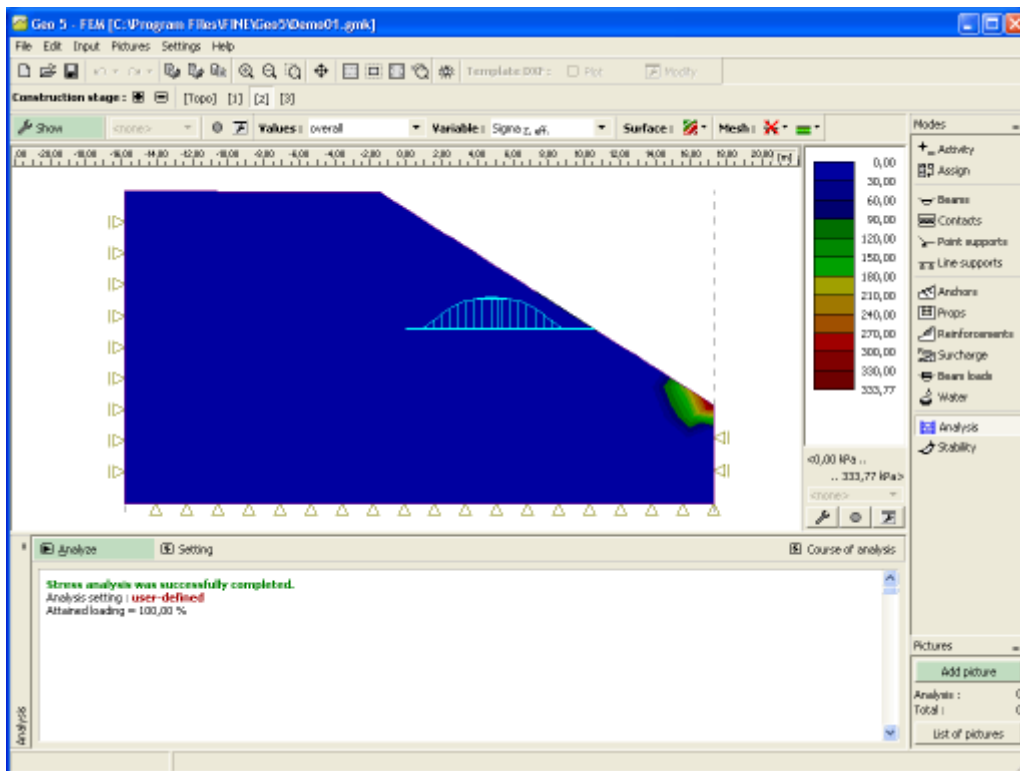
## Anchoring Geo-Reinforcements

When introducing the reinforcement into the soil body it is necessary to keep in mind a **sufficient anchorage of the reinforcement** since the program **does not check the reinforcement against the shear failure**. A sudden increase of the normal force as shown in the figure suggests singularity in contact stresses and probable shear failure of the reinforcement. From that point of view the displayed results are misleading and essentially unrealistic.

In such a case, the reinforcement should be either removed from the analysis or ensure its **sufficient anchorage** as plotted in the figure.



*Insufficiently anchored reinforcement*



*Correctly anchored reinforcement*

## Axial Stiffness of Geosynthetics

Geosynthetics are tensile reinforcing elements (geotextiles, geogrids) defined by their starting and end points and by the axial (normal) stiffness  $J_z$  [kN/m].

For **nonwoven fabrics** the axial stiffness is usually not considered since these elements typically serve as separating layers. **Woven geotextiles** experience for small deformations very low initial stiffness - in the small strain region (up to 5%) we encounter a considerable increase of deformations under constant load.

When designing geotextiles this property must be taken into account. We thus recognize both the **long-term tension strength** in dependence on partial reduction factors (reflecting damage of elements caused by installation, creep behavior of geosynthetics, biological and chemical effects) and **initial normal stiffness** in the small strain region in the interval of 0.5% to 2%.

To determine the **minimum axial stiffness** of georeinforcements it is possible to use the following expression where for the strength corresponding to the selected strain we accept maximally 10% deviation from a linear part of tension test:

$$T_{z-x} \geq \frac{0,9 \cdot \varepsilon \cdot T_{max}}{\varepsilon_{max}}$$

- where:
- $T_z$  - tensile strength at  $x\%$  strain [kN/m]
  - $\varepsilon$  -  $x\%$  strain (relative extension) according to EN ISO 10 31 [%]
  - $T_{max}$  - maximal tensile strength according to EN ISO 10 319 [kN/m]
  - $\varepsilon_{max}$  - maximal strain (relative extension) according to EN ISO 10 319 [%]

Suppliers and producers of geotextiles typically provide the value of tensile strength at 2% strain. The expression then becomes:

$$T_{z-2\%} \geq \frac{1,8 \cdot T_{max}}{\varepsilon_{max}}$$

The **minimal** (initial) **axial stiffness** of geotextiles from a short-term experiment (load rate according to EN ISO 10 319) for  $x\%$ -strain is given by:

$$J_{\varepsilon=x} \approx E \cdot A = \frac{T_{\varepsilon=x}}{\varepsilon}$$

- where:  $\varepsilon$  -  $x\%$ -strain (relative extension) according to EN ISO 10 319 [-]

The **maximum** (theoretically attainable) **axial stiffness** of geotextiles for a short-term axial strength is determined as follows:

$$J_{\varepsilon_{max}} \approx E \cdot A = \frac{T_{max}}{\varepsilon_{max}}$$

- where:  $\varepsilon_{max}$  - maximum strain (relative extension) according to EN ISO 10 319 [-]

Intervals of recommended values of axial (normal) stiffnesses of geosynthetics  $J_z$  [kN/m] are listed in the following table:

Variable description	Initial axial stiffness of geotextiles for	Theoretical (maximal) axial stiffness of

	$\varepsilon = 2\%$	<b>geotextile</b>
<b>Notation (unit)</b>	$J_{\varepsilon=x}$ [kN/m]	$J_{\varepsilon max}$ [kN/m]
<b>Georeinforcements category</b>	---	---
Non-woven geotextiles	-	-
Woven geotextiles	250 ÷ 500	1000
Unaxial geogrids	500 ÷ 1000	1500
Biaxial geogrids	100 ÷ 500 for $\varepsilon = 0.5\%$	2500
Triaxial geogrids	250 ÷ 500 for $\varepsilon = 0.5\%$	5000
Geomats	100 ÷ 500	1000
Drainage geocomposites	-	-
Composites	100 ÷ 500	1500
Geomeshes	-	-
Geocells	-	-

*Literature:*

GEOMAT Ltd. ([www.geomat.cz](http://www.geomat.cz)): *Types of geotextiles and their function in civil engineering structures*. Author: Martin Kašpar ([kaspar@geomat.cz](mailto:kaspar@geomat.cz)). In Czech.

HOLÝ, O., MIČA, L.: *Determination of axial stiffness of geosynthetics for numerical modeling - part 1*. TU Brno (paper in conference proceedings „Civil engineering structures in view of geomechanics“). In Czech.

EN ISO 10 319 (80 6125): *Geotextiles - Tensile test on a wide strip*. Czech standard institute, 2009. In Czech.

## Surcharge

The frame "**Surcharge**" contains a table with the list of surcharges. Adding (editing) surcharge is performed in the "**New surcharge**" dialog window. The input surcharges can also be edited on the desktop with the help of active objects. The program employs the following coordinate systems.

All input parameters of the surcharge can be modified in the stage of construction, in which the surcharge was introduced. In subsequent stages it is only possible to modify its magnitude (option "**Adjust magnitude**").

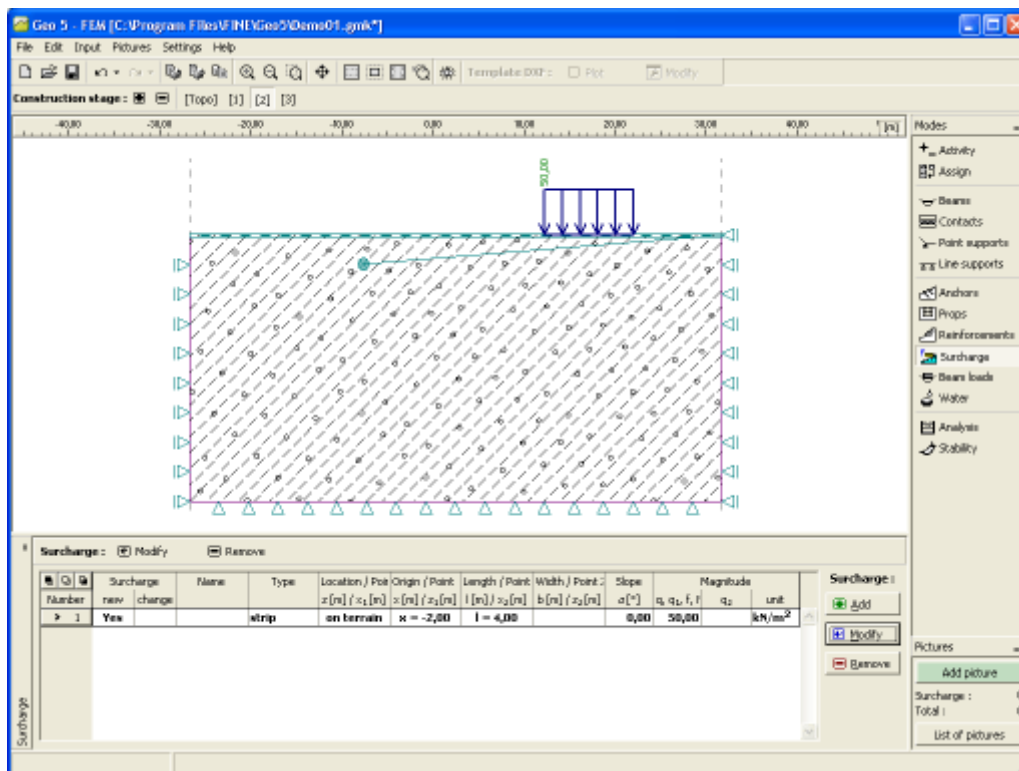
This frame serves to introduce **surcharges applied only to the soil body**. The surcharge

applied to a beam element is introduced in the frame - **beam load**.

An arbitrary number of surcharges can be specified in individual stages. The surcharge may act either on the **existing interface** (including ground surface) or can be applied **anywhere in the soil body**.

In subsequent **stages** we are free to either remove the input surcharge or to **modify its magnitude**.

Note that applying the surcharge directly on the ground surface may lead to **excessive plastic deformations** in the vicinity of the surcharge and the analysis may fail to converge. In such a case, one may either place a **beam element** under the applied surcharge, or to choose an **elastic** or modified elastic material model for the soil below the surcharge.



Frame "Surcharge"



**New surcharges**

– Surcharge name

Name : Surcharge n. 1

– Surcharge properties

Type : strip

Location : on terrain

Origin : x = -2,00 [m]

Length : l = 4,00 [m]

Slope :  $\alpha = 0,00$  [°]

– Surcharge magnitude

Magnitude : q = 50,00 [kN/m<sup>2</sup>]

Add Cancel

Dialog window "New surcharges"

## Beam Loads

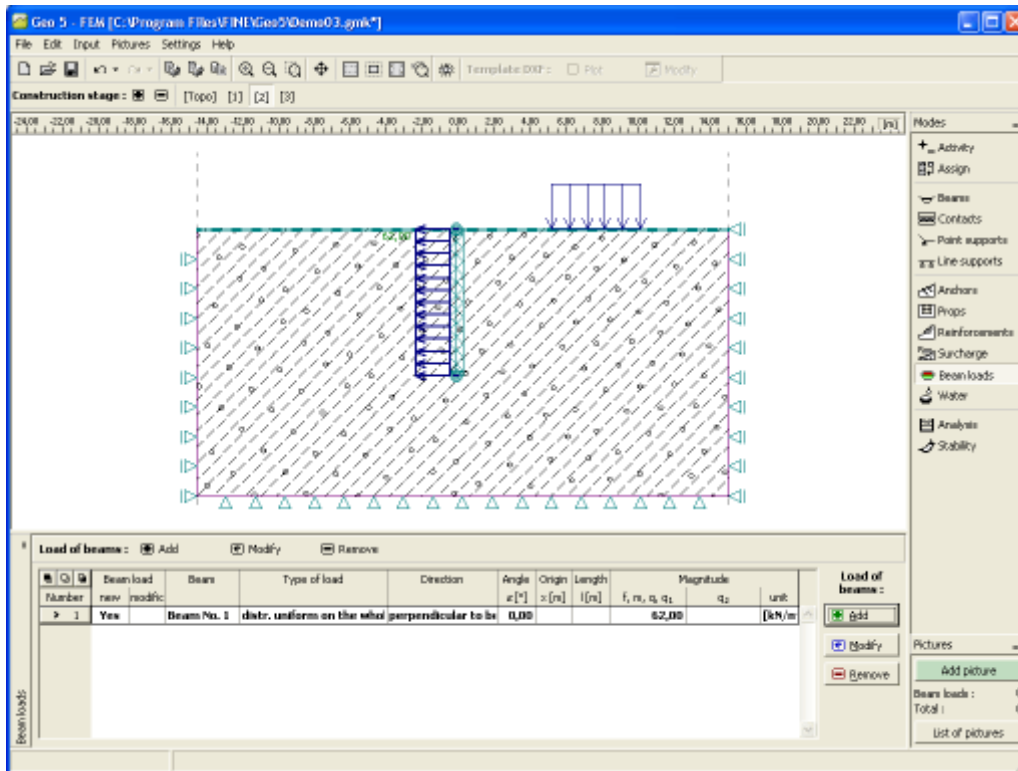
The frame **"Beam loads"** contains a table with the list of loads. Adding (editing) beam loads is performed in the **"New beam loads"** dialog window.

Beam loads can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Beam loads"**. The following modes are available:

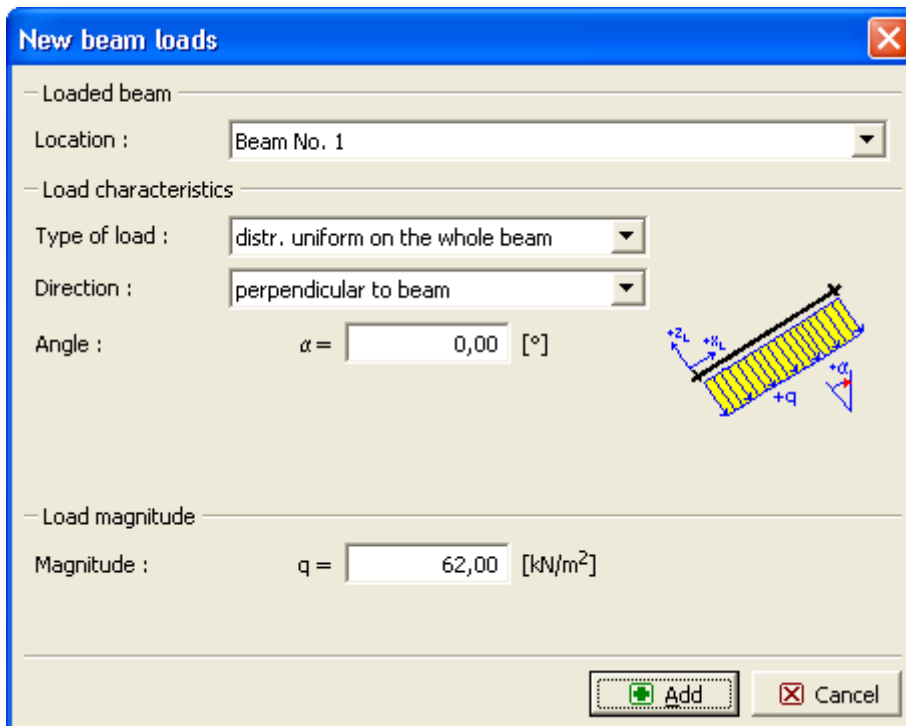
- **Add** By clicking the left mouse button on the selected beam. The load parameters are entered in the **"New beam loads"** dialog window.
- **Adjust** Clicking the left mouse button on already existing beam opens the **"Adjust beam loads"** dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing beam opens the **beam load removal** dialog window - accepting this action removes the selected beam load.

The input loads can also be edited on the desktop with the help of **active objects**. The program employs the following **coordinate systems**.

All input parameters of the load can be modified in the **stage of construction**, in which the load was introduced. In subsequent stages it is only possible to modify its magnitude (option **"Adjust magnitude"**).



Frame "Beam loads"



Dialog window "New beam loads"

# Water

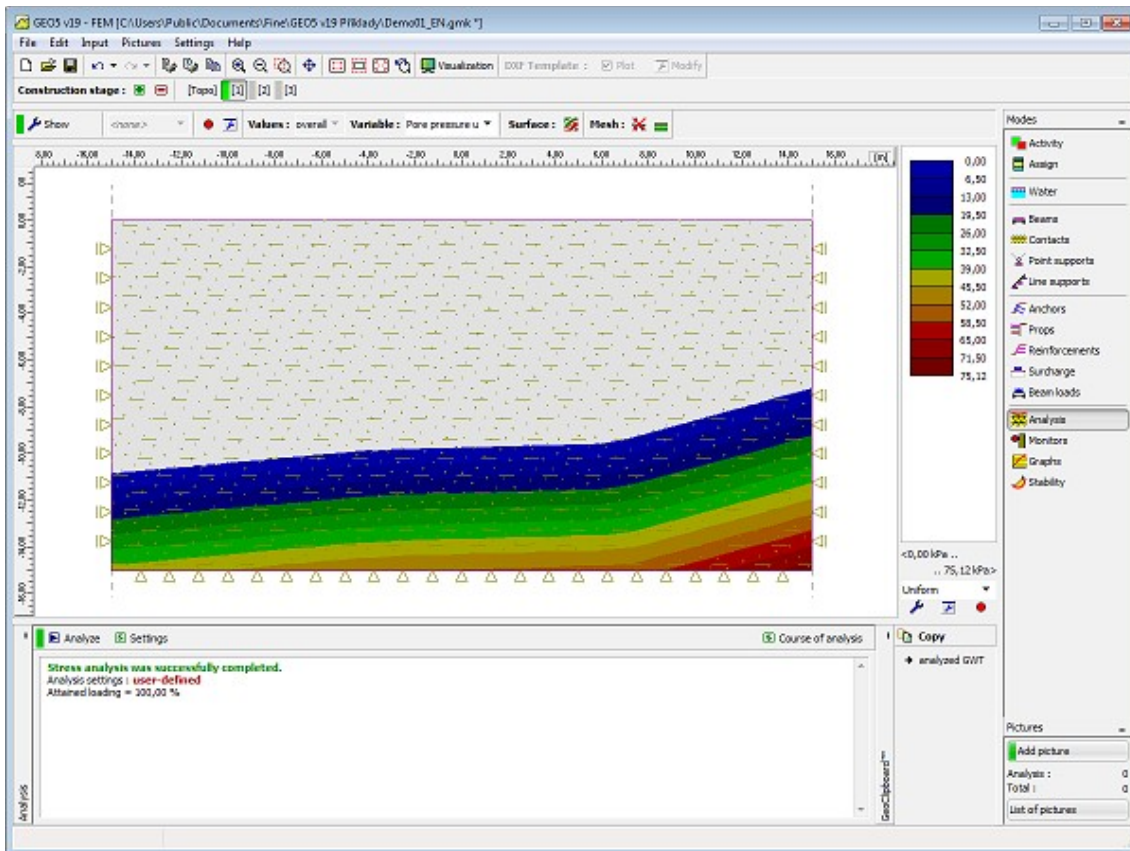
There are three options in the program to introduce ground water:

- The **ground water table** can be specified as a continuous interface below and above the ground surface. In such a case, the program automatically adjusts the soil self-weight **below the ground water table**.
- The **pore pressure** values are entered via isoline. Input is the same as **interface input**. The pore pressure values are inserted into the table "List of interfaces" in the left bottom part of the screen. The values between isolines follow from linear interpolation.
- **The pore pressure coefficient**  $r_u$  represents the ration between pore pressure and the geostatic stress in the soil. The values of the coefficient  $r_u$  are specified for individual isolines. The first isoline always coincides with the ground surface. The remaining isolines are introduced in the same way as interfaces between individual soil layers. The values are inserted into the table "List of interfaces" in the left bottom part of the screen. The values between isolines follow from linear interpolation.

When entering the values of pore pressure or the values coefficients  $r_u$  the **unit weight of soil is assumed in the whole body** to be equal to the unit weight  $\gamma$  regardless of the values of pore pressures or coefficients  $r_u$ .

The simplest way to check the input of water is to plot the distribution of **pore pressure** in the output window.

Input interfaces of water can be copied within all 2D GEO5 programs using "GeoClipboard".



Visualization of pore pressure

## Analysis

The analysis is performed for individual **calculation stages** in the frame **"Analysis"** after pressing the **"Analyze"** button.

**During analysis** the program attempts to arrive at such a solution that satisfies for given load and boundary conditions the **global equilibrium**. In most cases this step results into an iterative process. The **process of iteration** and convergence of the solution is displayed on the screen.

The analysis can be stopped any time by pressing the **"Interrupt"** button. The results are then available for the last converged load increment.

The correct results are obtained when **100% of the applied load is reached**. Due to **convergence failure** the program may stop **before reaching the desired load level** - only a fraction of the total applied load is reached. In such a case it is possible to adjust standard **parameters of the analysis** setting.

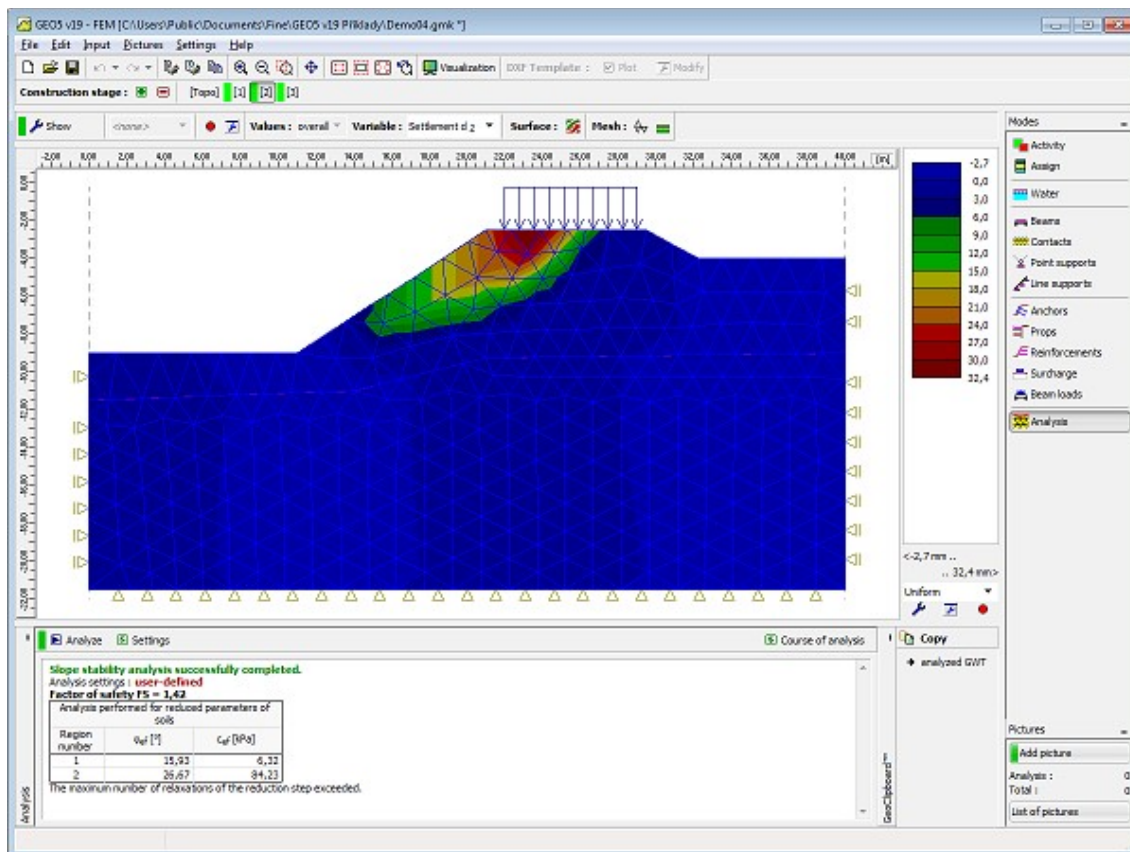
When modeling more complex engineering tasks we encourage the user to follow the **recommended modeling procedure**.

The **transient flow analysis** can be selected in the frame **"Settings"**.

The analysis **results** together with information about the course of analysis appear on the screen immediately after completing the analysis.

Detailed information about the actual modeling approach is presented in section **"Setting and analysis description"**. Visualization of results can be adjusted in the frame **"Visualization settings"**.

In case of considering water in analysis, in most cases there's a possibility to copy analyzed GWT to **GeoClipboard** and paste it into another program.



Screen after completing analysis

## Transient Flow Analysis

The actual analysis proceeds in two and more stages ("Water flow"), where the first stage serves to set the initial conditions, i.e. the distribution of initial pore pressure, initial pressure head, degree of saturation and relative permeability at the onset of transient flow analysis. Several options are available to set the initial pore pressure:

- With the help of ground water table
- Directly with the help of pore pressure interfaces
- Running the steady state flow analysis

The first option assumes a hydrostatic (linear) distribution of pore pressure over the height. Below GWT the program generates positive pore pressures, whereas above GWT the negative pore pressures (suction) are generated. The second option allows for considering a dry soil by prescribing e.g. negative pore pressures over the entire infiltrated region. The third option requires running the **steady state analysis**. Based on the assigned **material model** the program then determines the initial degree of saturation and relative permeability as a function of the initial pore pressure. Figure 1 shows the distribution of initial pore pressure provided by steady state analysis for the assumed hydraulic conditions. Clearly, only the pressures below GTW are presented. The initial state in an unsaturated or partially saturated region can be partially judged by plotting for example the distribution of initial degree of saturation as seen in Figure 2. When selecting the **"No water"** option the initial pore pressure values are set equal to zero.

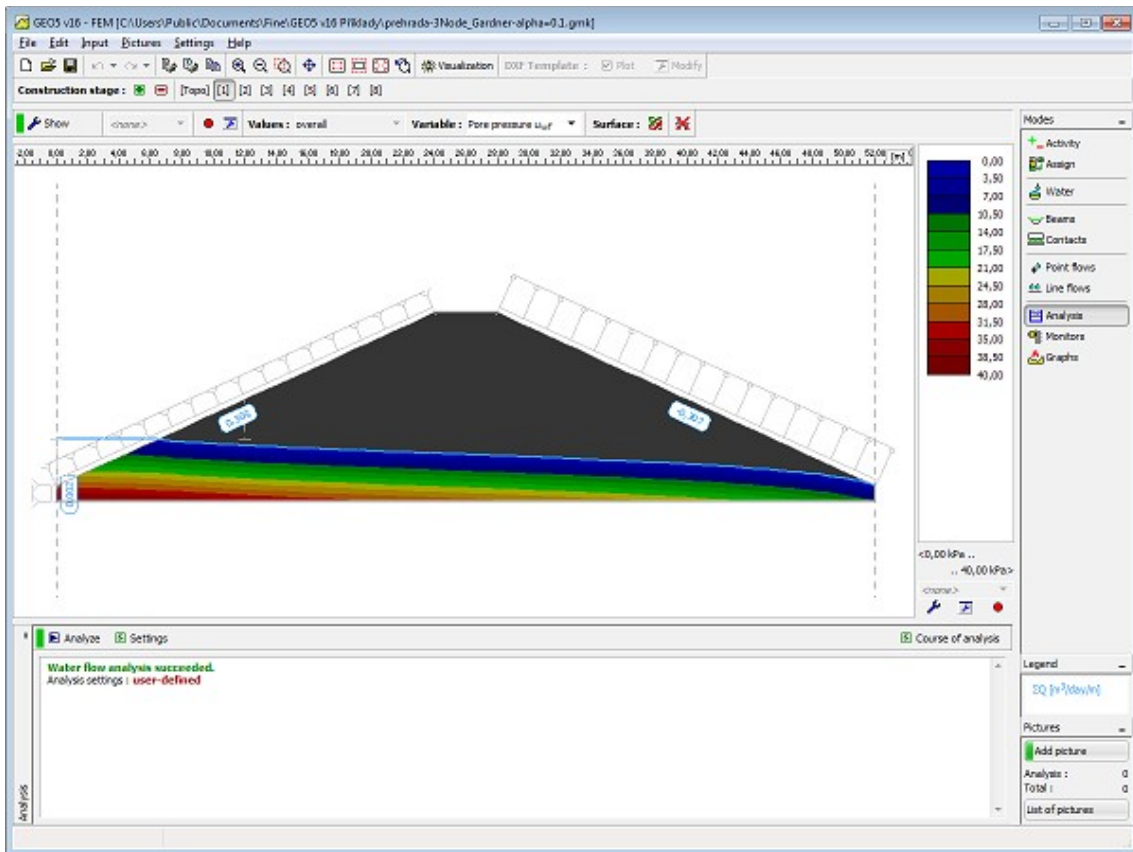


Figure 1 - First calculation stage: Distribution of initial pore pressure



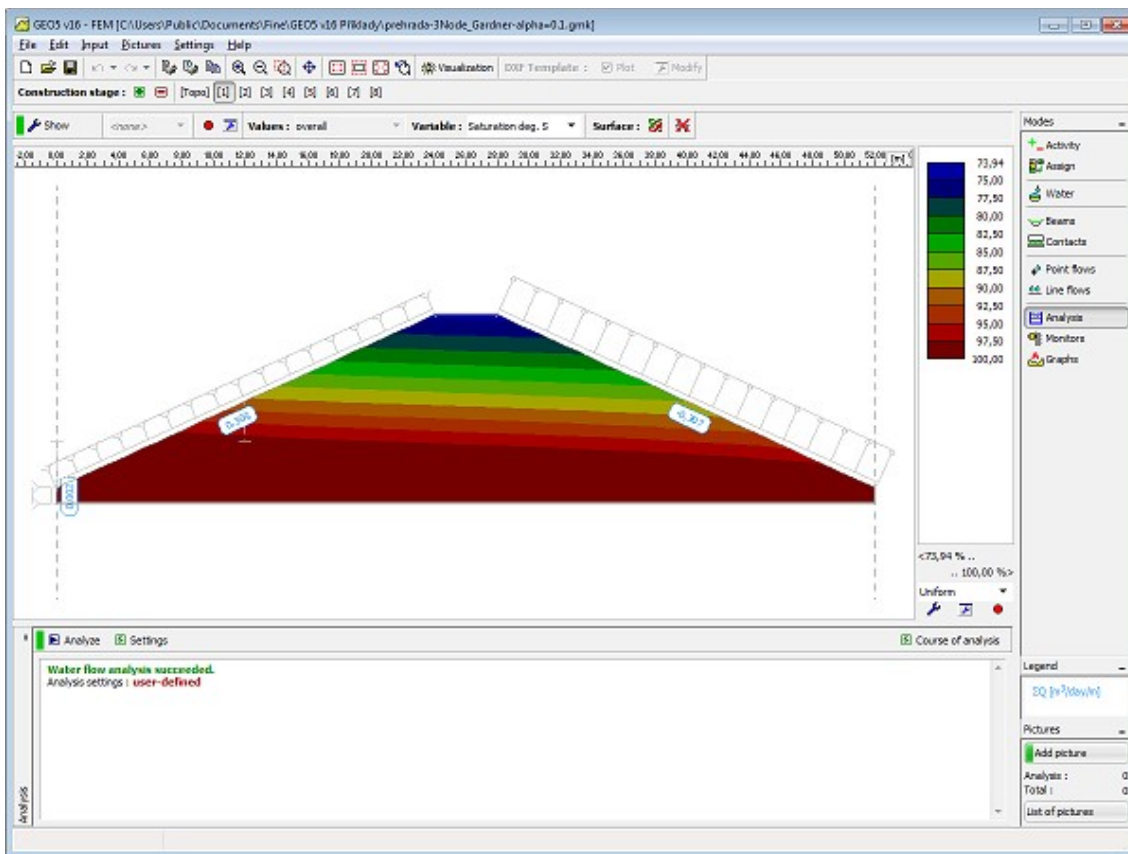


Figure 2 - First stage: Distribution of initial degree of saturation

The transient flow analysis is performed from the second stage on where the next stage follows the preceding one. Each stage requires setting the analysis time, time dependent variation of boundary (hydraulic) conditions and the time step length. The current version of the program allows us to either introduce the entire load at once at the beginning of the calculation stage or to assume that it linearly increases with time during the course of stage calculation ("Water flow"). In the first case the initial time step is set to 1/10 of the assigned time step. Next, the calculation continues with the assigned time step. It is reasonable to adjust the time step during the course of analysis. A shorter time step is recommended at the beginning of the analysis. With longer times, when the solution approaches the steady state conditions, the time step can be increased considerably (e.g. from 1/10 of day up to several days). Figures 3 and 4 display an intermediate state and a steady state solution, respectively, corresponding to a sudden increase of GWT in the second calculation stage. Figures 5 and 6 show similar states associated with a subsequent rapid drawdown simulated by resetting the original level of GWT at the seventh calculation stage.

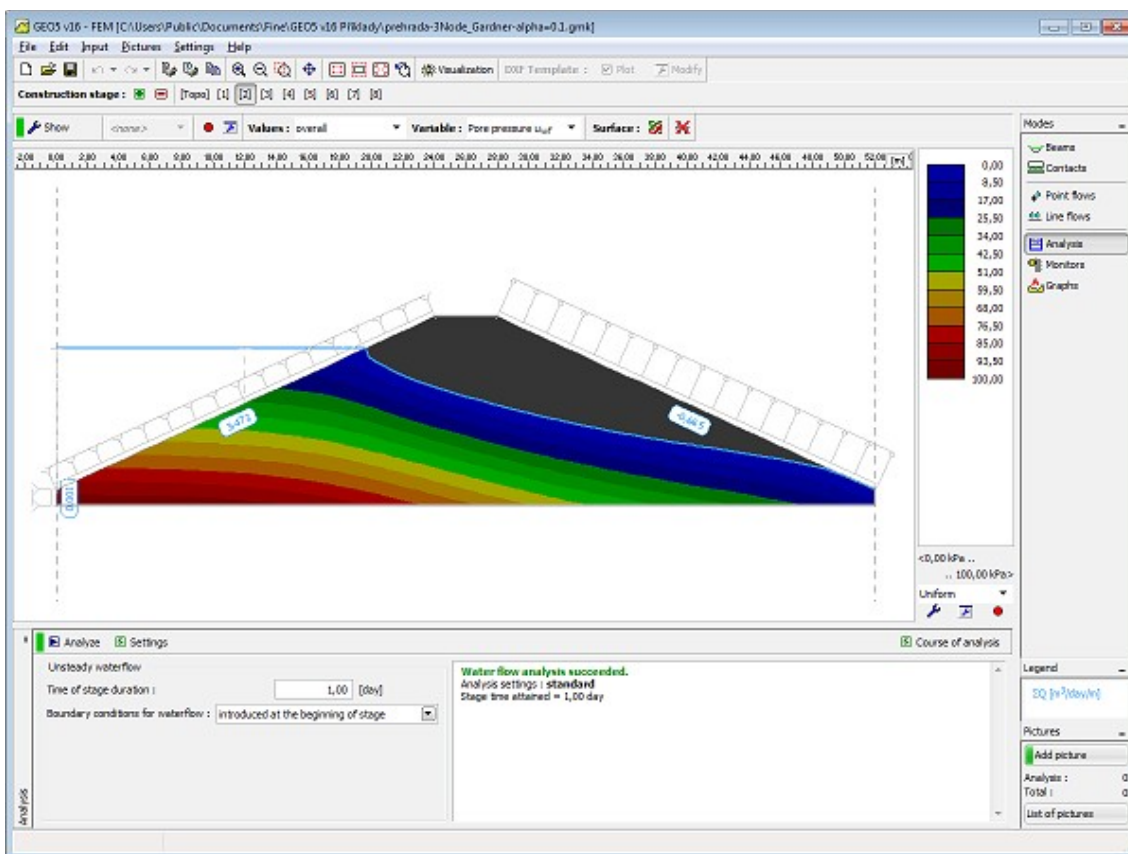


Figure 3 - Second calculation stage: Distribution of pore pressure at a given time of analysis



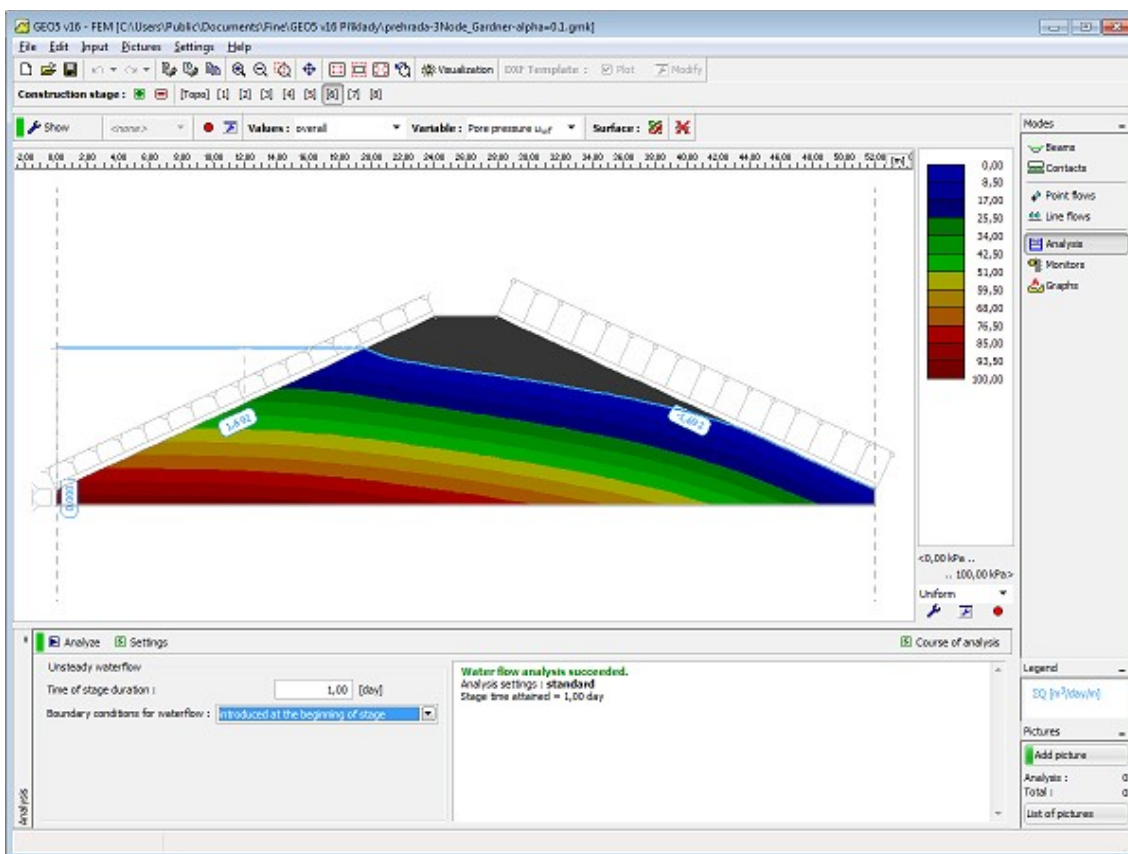


Figure 4 - Sixth calculation stage: Distribution of steady state pore pressure

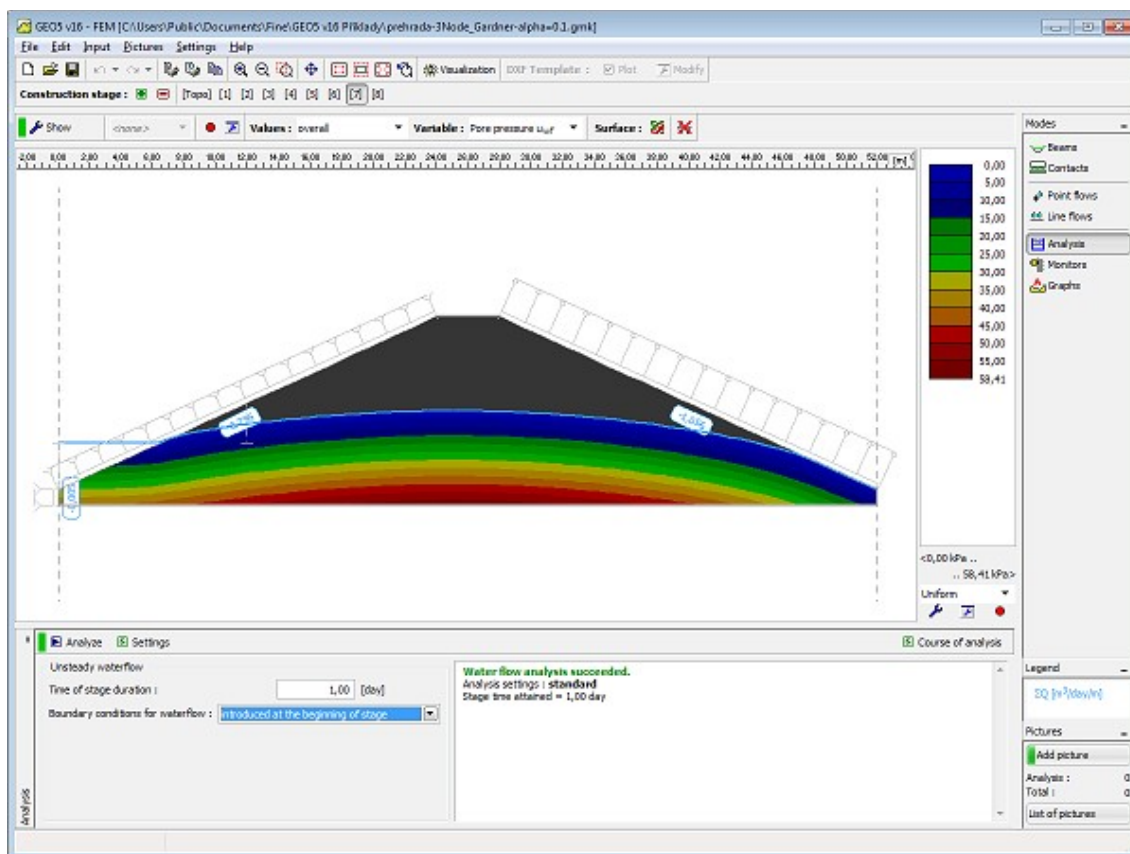


Figure 5 - Seventh calculation stage: Distribution of pore pressure at a given time of analysis

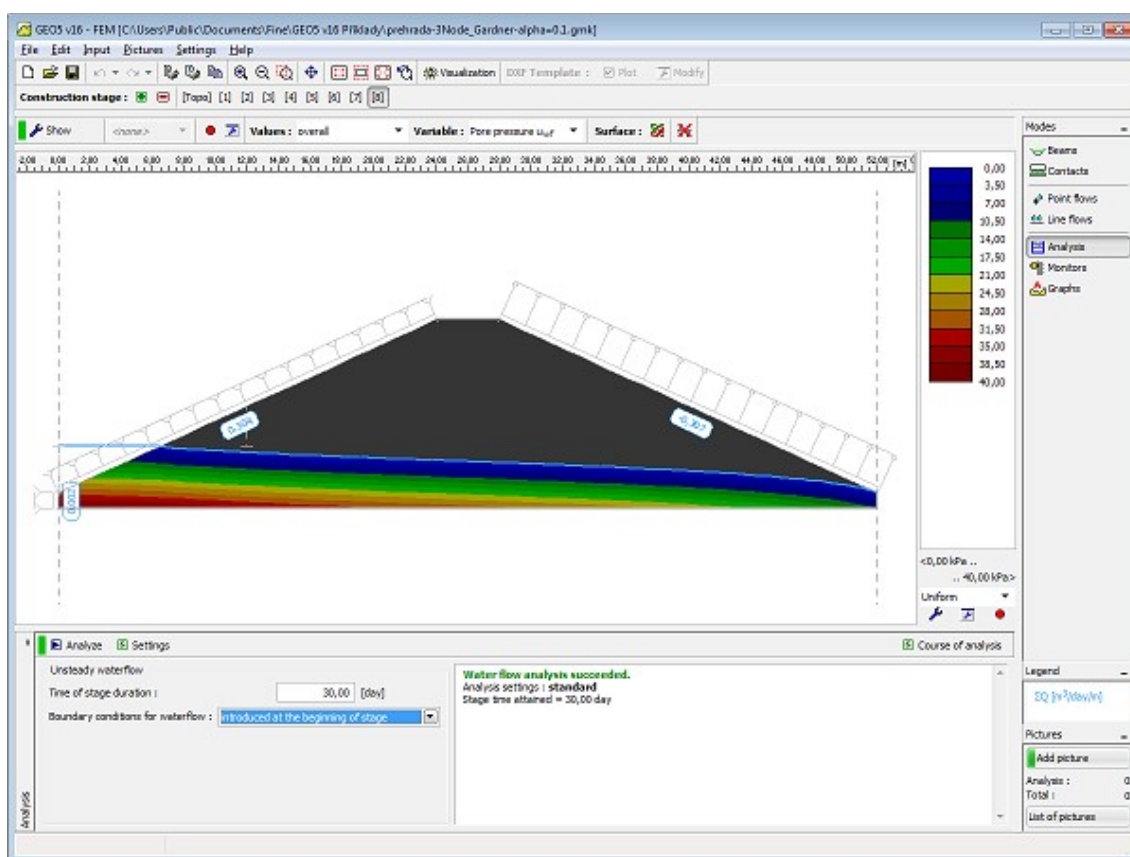


Figure 6 - Eighth calculation stage: Distribution of steady state pore pressure

## Recommended Modeling Procedure

Solving geotechnical problems using the finite element method is a relatively complex task. But yet, most users attempt to analyze the entire complex structure right from the beginning - to find the cause of possible loss of convergence may then become rather difficult. We therefore recommend the following approach:

- 1) Define the whole topology of the structure**
- 2) Assume elastic response of soils and contact elements (use [linear models](#))**
- 3) Generate coarse mesh**
- 4) Define all [calculation stages](#)**
- 5) Perform [analysis](#) of all calculation stages (it is sufficient to launch the analysis of the last stage of construction - analyses of all previous stages are carried out automatically).**
- 6) Asses the course of analysis**

If the analysis fails, the computational model is not correctly defined - e.g. beams have too many internal hinges resulting into a kinematically undetermined structure, props are not properly hooked to the structure, etc. The program contains a number of built-in checking procedures to warn the user for possible drawbacks in the model definition. Some of the errors, however, cannot be disclosed prior to running the program.

If all stages were successfully analyzed, we recommend the user to check the resulting displacements and this way also the objectivity of the used soil parameters and structure stiffness. Note that using nonlinear models always results into larger displacements in

comparison to the pure elastic response - should the elastic displacements be already excessively large, we must first adjust the computational model before adopting any of the available plasticity models.

If the analysis succeeded and the displacements are reasonable, we may proceed as follows:

**7) Replace linear models** with suitable **plastic models** (Mohr-Coulomb, Drucker-Prager)

**8) Perform analysis and evaluate the results according to step 6**

**9) Add nonlinear contact elements**

**10) Perform analysis and evaluate the results according to step 6**

**11) Refine and adjust the finite element mesh and perform the final analysis.**

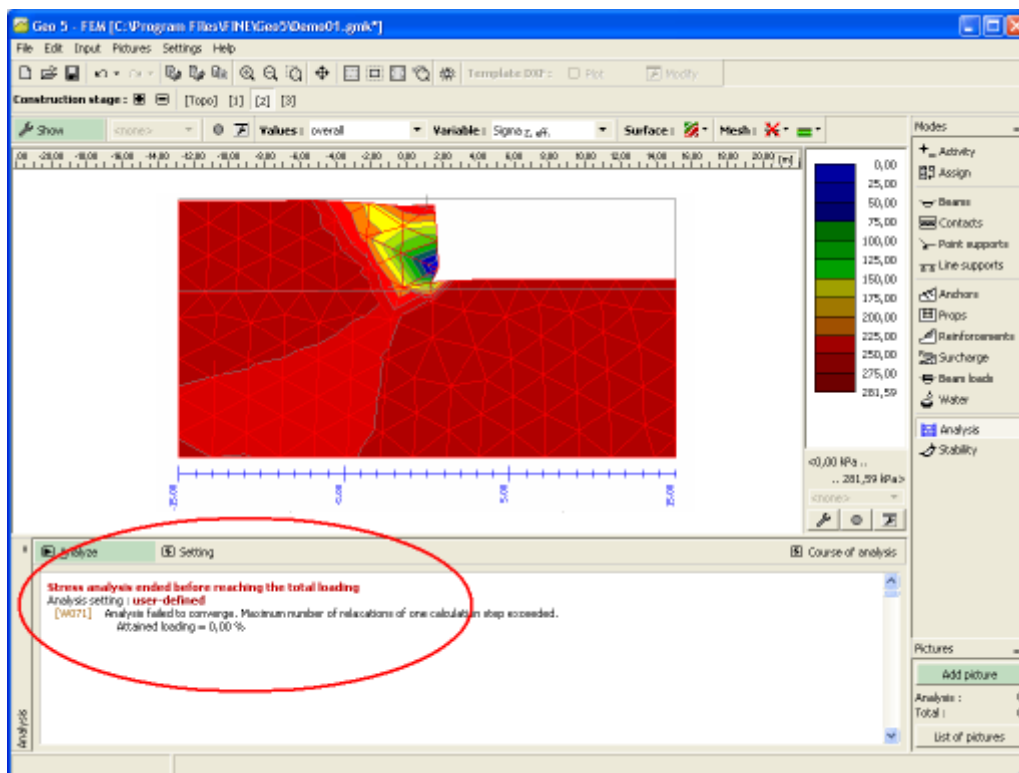
Although this approach may seem rather cumbersome and complicated, it may save a considerable amount of time when searching for the cause of failure (loss of convergence) of the analysis of complex problems.

## Loss of Convergence of Nonlinear Analysis

Loss of convergence of the solution of nonlinear analysis calls for certain **modifications of the underlying computational model** - the following steps can be adopted:

- Increase the stiffness of the structure
- Decrease the applied loads
- Split the soil excavation in more steps
- Improve material parameters of existing soils
- Change **material model** of soils in places of plasticity
- Add reinforcing members (beams, anchors)
- Add supports
- Change **parameters settings** affecting the iteration process (increase number of iterations).

**Distribution of plastic strains** may provide some explanation to why the analysis failed to converge. Note that the distribution of equivalent plastic strain locates the regions of probable evolution of critical failure surfaces.



*Analysis failed to converge - plot of equivalent plastic strain*

## Settings and Analysis Description

The **default settings of parameters that drive the solution** analysis are optimized to ensure sufficient accuracy and efficiency of the analysis. Nevertheless, an experienced user may require to change the default setting, or to examine the influence of parameters on the accuracy and course of the analysis. The parameters setting can be adjusted in the "**Analysis settings**" dialog window.

However, the change of standard setting deserves a **word of caution**. Prior to making any changes, the user should be well aware of possible consequences. In particular, improper setting may substantially slow down the computation process, may cause divergence and eventually lead to **incorrect results**.

- Solution method
- Change of stiffness matrix
- Initial solution step
- Maximum number of iterations
- Convergence criterion
- Newton-Raphson method setting
- Arc-length method setting
- Line search method
- Plasticity

The default setting can be always recovered by pressing the "**Standard**" button.

## Solution Method

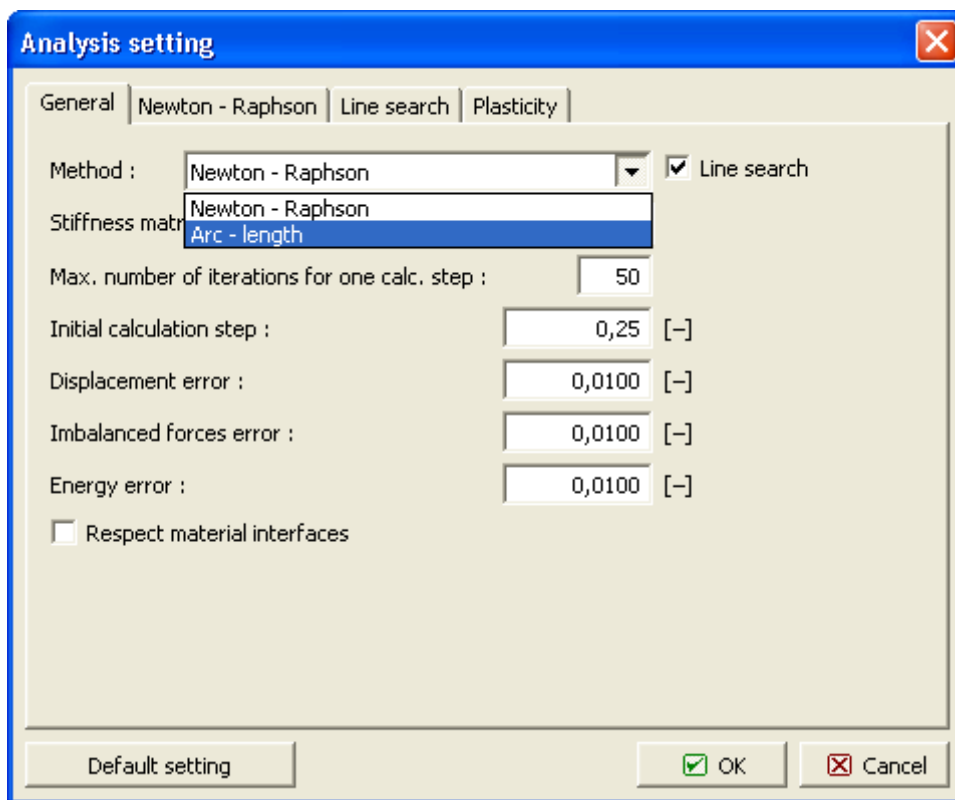
The program FEM serves to analyze geotechnical problems characterized by nonlinear response of the soil or rock body. A successful analysis of most of such problems calls for an iterative solution of a given boundary value problem. Applying the finite element method (FEM) then leads to an incremental form of the equilibrium conditions written as:

$$K_T \cdot \Delta u = \Delta f$$

where:  $K_T$  - instantaneous stiffness matrix  
 $\Delta u$  - vector of nodal displacement increments  
 $\Delta f$  - vector of out-of-balance force increments

This equation can be solved only approximately using a suitable numerical method. The goal of the method is to arrive, during the process of iteration, at such a state of stress and strain that satisfies the condition  $\Delta f = 0$ . To that end, **the program offers** two basic methods:

1. **Newton-Raphson method - NRM**
2. **Arc-length method - ALM**



*Analysis settings - setting the solution method*

## Change of Stiffness Matrix

The full **Newton-Raphson method** assumes that the instantaneous tangent stiffness matrix is formed at the beginning of each new iteration.

Forming a new tangent stiffness matrix only at the beginning of a new load increment leads to

so-called modified Newton-Raphson method.

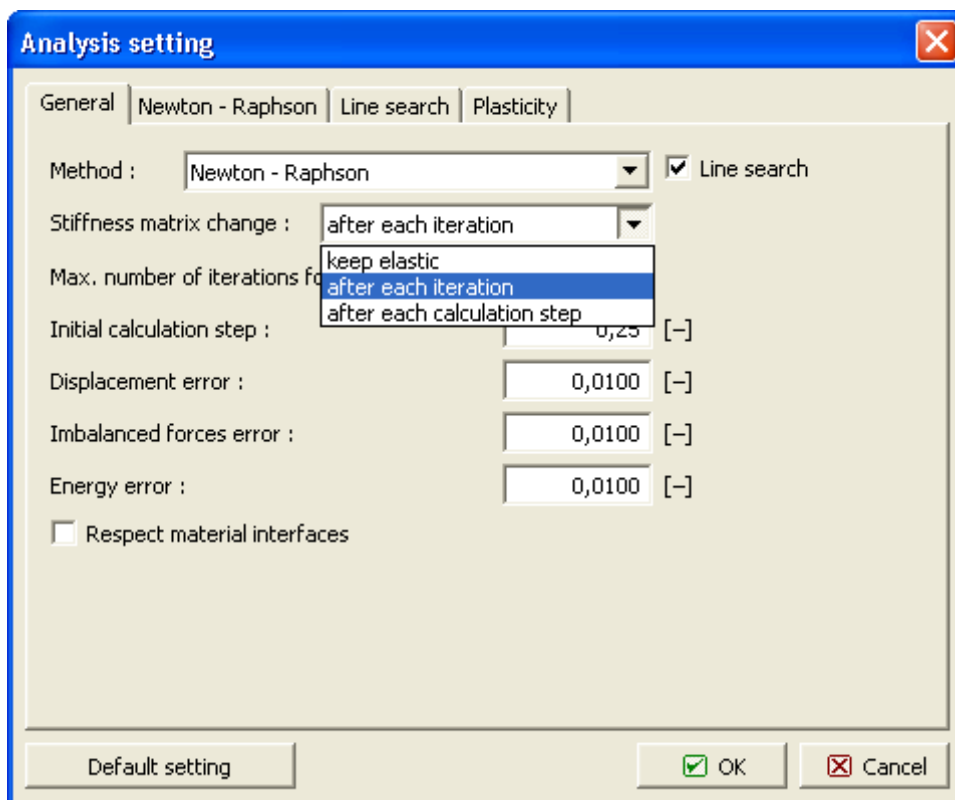
If the stiffness matrix is formed only once at the beginning of the solution analysis we obtain so called initial stress method.

Individual methods can be selected from the "**Analysis settings**" dialog window section "Stiffness update". The corresponding settings are:

1. **Keep elastic** - initial stress method,
2. **Each iteration** - full Newton-Raphson method,
3. **Each load step** - modified Newton-Raphson method.

The default setting assumes the full Newton-Raphson algorithm (*stiffness update after each iteration*). Note that the formulation of stiffness matrix is consistent with the stress update algorithm. Such a formulation then ensures quadratic convergence of the full Newton-Raphson (NRM) unlike the modified NRM or the initial stress method that, in comparison with the full NRM, require considerably more interactions to attain equilibrium.

On the other hand, it is fair to mention that the computational cost per iteration is mainly determined by the calculation and factorization of the tangent stiffness matrix. Assuming elastic response of a structure it is clearly meaningless to set up the structural stiffness matrix more then once (stiffness update - keep elastic). On the contrary, increasing the degree of nonlinearity suggests more frequent stiffness reformulations (stiffness update - Each iteration).



*Newton-Raphson method - stiffness matrix update options*

### Initial Solution Step

The actual analysis is carried out incrementally in several load steps until the overall prescribed load is reached.

The program requires setting the **initial load step** only.

This parameter represents the **ratio between the load applied in a given load step to the overall prescribed load**. Depending on the course of iteration this parameter is adaptively adjusted.

The default setting assumes 25% of the total prescribed load. Similarly to what we have already mentioned it holds that increasing the solution complexity from the nonlinear response point of view requires reduction of this parameter. However, in the case of **elastic response** this parameter can be set equal to 1, which corresponds to the solution of a given problem in one load step.

## Maximum Number of Iterations

This parameter represents the **maximum number of iterations allowed** for a single load step to reach the state of equilibrium.

Exceeding this value prompts the program to automatically **reduce the current value of the assumed load step** and restarts the solution from the last load level that complies with the state of equilibrium. Similar action is taken when oscillation or divergence of the program is imminent.

## Convergence Criterion

For the incremental solution strategy based on one of the iterative methods to be effective, it is necessary to select suitable criteria (preset tolerances for reaching equilibrium) for the **termination of the iteration process**.

Note that loose convergence criteria may result in inaccurate results while too tight convergence tolerances may lead to unjustified increase of computational cost spent to arrive at the results of superfluous accuracy.

In the program the convergence is checked against the change of nodal displacement increments, the change of out-of-balance forces and also the change of internal energy. The last criterion gives a certain idea about how both displacements and forces approach their equilibrium values. The corresponding settings are:

1. **Displacement error tolerance** - tolerance for the change of displacement increment norm.
2. **Out-of-balanced forces tolerance** - tolerance for the change of out-of-balance force norm.
3. **Energy error tolerance** - tolerance of the change of internal energy.

The default setting is 0.01 for **all convergence tolerances**.

## Setting Newton-Raphson Method

With the Newton-Raphson method the course of iteration can be driven by setting the following parameters:

**1) Relaxation factor** - it represents the value of reduction of the current load step for the restart providing the solution fails to converge. A new value of the assumed load step is found from the expression:

**new load step = old load step / relaxation factor.**

**2) Max. No. of relaxations** for a single load step - this parameter determines how many

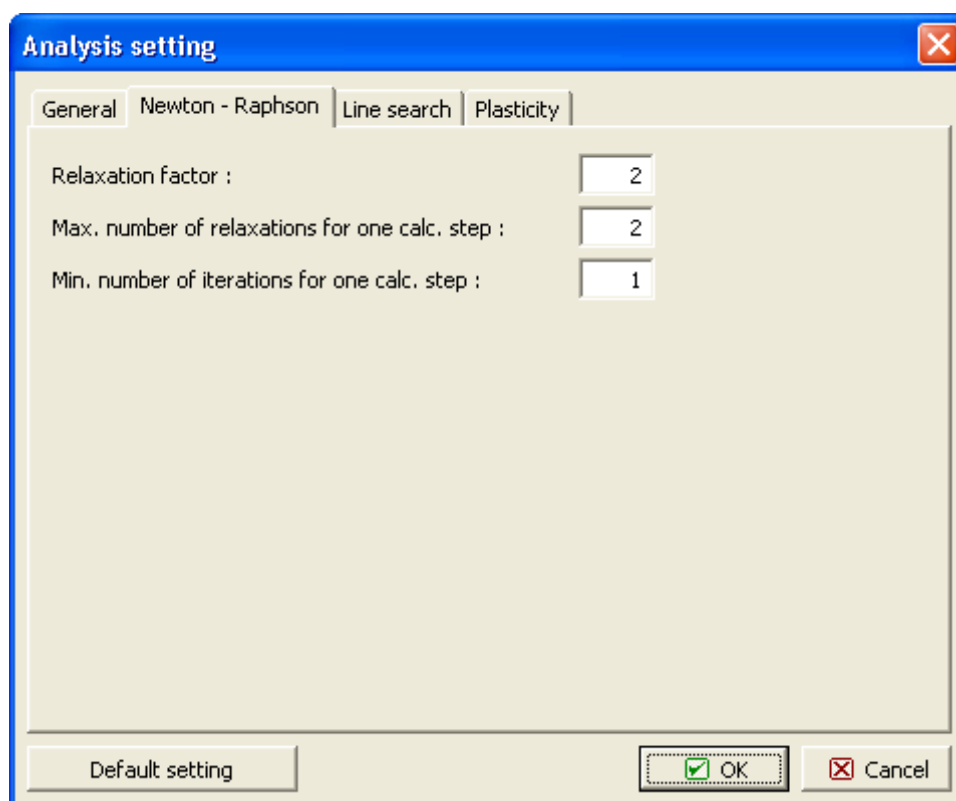


times it is possible to invoke the above action during the entire analysis. Exceeding this value prompts the program to terminate the analysis. The results are then available for the last successfully converged load level.

**3) Min. No. of iterations for a single load step** - this parameter allows for possible acceleration of the analysis. In particular, providing the number of iterations to converge in the last load step is less than the minimum one set, the load step for a new load increment is increased as follows:

**new load step = old load step \* relaxation factor.**

The default setting of the above parameters corresponds to values displayed in the figure:



*Parameters driving the iteration process*

## Setting Arc-Length Method

The Arc-length method (ALM) is relatively robust method particularly suitable for the solution of problems that require the search for the collapse load of a structure. Stability analysis of earth structures (slopes, embankments) is just one particular example of such a task. Unlike the NRM where the solution is driven purely by prescribing load increments, the ALM introduces an additional parameter representing a certain constraint on the value of load increment in a given load step. The value of the load step thus depends on the course of iteration and is directly related to the selected arc length.

The basic assumption of the method is that the prescribed load varies proportionally during the calculation. This means that a particular level of the applied load can be expressed as:

$$\bar{F} = \lambda.F$$

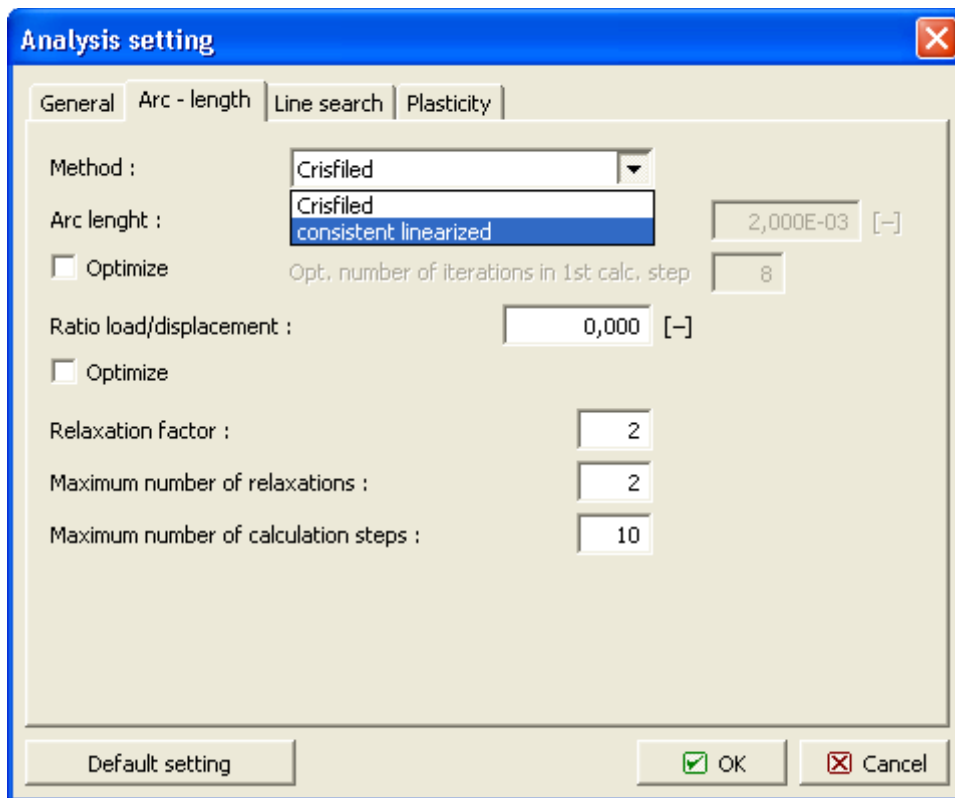
where:  $F$  - current fraction of the total applied load

- $\lambda$  - coefficient of proportionality
- $F$  - overall prescribed load

Note that with ALM the load vector **F** represents only a certain reference load that is kept constant during the whole response calculation. The actual value of the load at the end of calculation is equal to the  $\lambda$  multiple of **F**;  $\lambda < 1$  represents the state when the actual bearing capacity of a structure is less than the prescribed reference load; if  $\lambda$  at the end of response calculation exceeds 1, the program automatically adjusts the arc length in order for the solution to converge to value  $\lambda = 1$  within a selected tolerance equal to 0.01 (1% the maximum applied load). This value cannot be changed.

The literature offers a number of ALM formulations. The program supports the method suggested by Crisfield and the consistently linearized method proposed by Ramm. The latter one is considerably simple, at least from the formulation point of view, than the Crisfield method. On the other hand it is reportedly less robust. The default setting is the Crisfield method.

Other important parameters of the method are "Setting arc length" and "Automatic arc length control".



*Arc-length - setting the type of Arc-length method*

### Setting Arc Length

The arc length is the basic parameter affecting the response calculation. An indicator for the selection of arc length can be the course of iteration in the previous solution stage. Regardless of that the program enables the following setting:

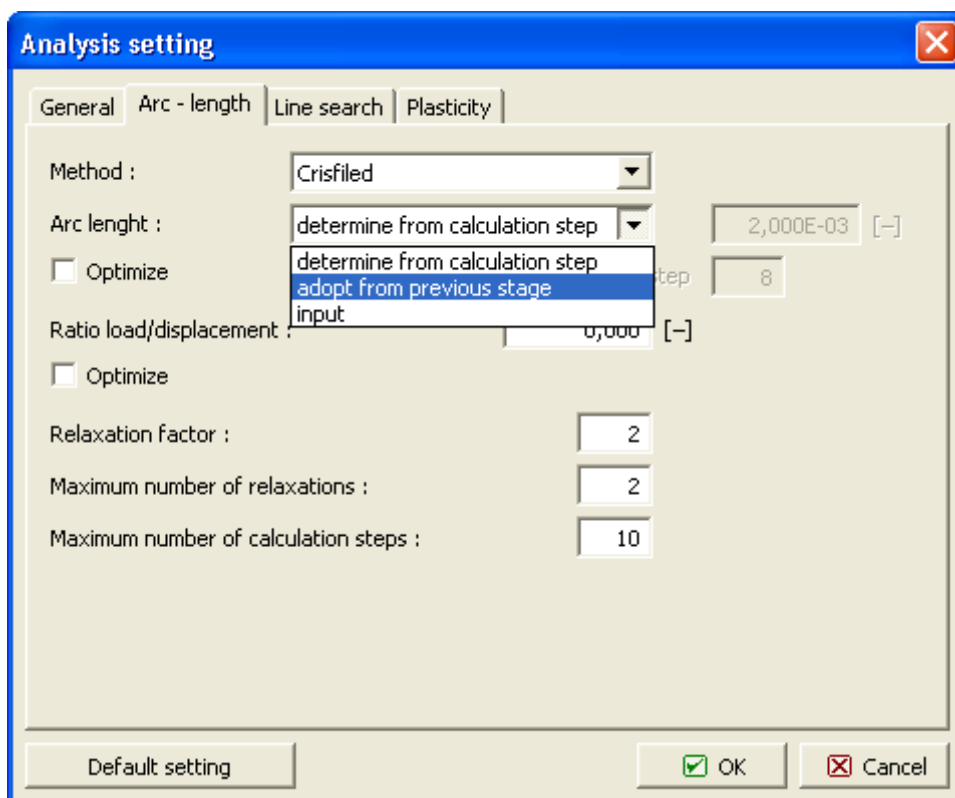
1. **Determine from load step** - the arc length is determined automatically from the initial load step.
2. **Adopt from the previous stage** - the value of arc length at the end of the previous calculation stage is used as a starting value for a new stage. This option becomes active in the second stage of construction.
3. **Input** - the value of arc length can be directly prescribed.

Providing the structure response cannot be determined prior we recommend using the first option. Depending on the course of calculation it is possible to adjust the value of arc length and repeat the calculation. At no event, however, is it possible to ensure convergence for an arbitrary value of arc length selected. Similarly to NRM, if the convergence problems occur the program allows for the reduction of the current arc length and restarts the calculation.

The next parameter driving the iteration process is the *Maximum No. of load steps*. The program always carries on the prescribed number of load steps providing:

- parameter  $\lambda$  exceeds 1,
- the maximum number of relaxations of arc length is exceeded.

Providing the analysis is terminated due to exceeding the maximum number of prescribed load steps and parameter  $\lambda$  is less than 1, it is necessary to increase the number of steps and restart the analysis.



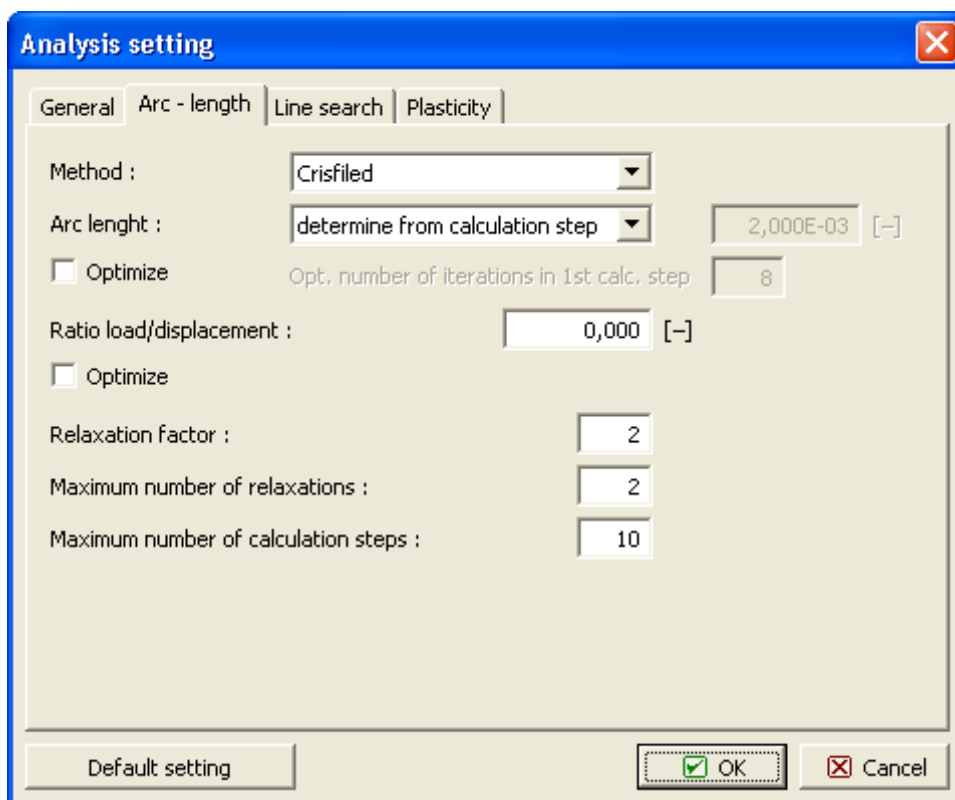
*Arc-length - arc length setting*

### Automatic Arc Length Control

Automatic arc length control strategy constitutes very important part of implementation of any numerical method. The program makes it possible to adaptively adjust the current arc length

for a new load step depending on the course of iteration in the previous step by activating option **Optimize**. The program will then attempt to select a value of arc length that keeps the desired number of iterations in each load step needed for convergence - option **Optim. No. of iter. in a single load step**.

The next parameter driving the process of iteration is the **Ratio load/displacement**. This parameter represents a scalar factor, which adjusts the scales of load given by parameter  $\lambda$  and displacement vector  $u$ . providing this parameter is sufficiently large the analysis is essentially driven by load increment. Setting this parameter equal to 0 (default setting) we obtain so-called cylindrical ALM and the analysis will be driven by displacement increment. This approach is more stable and recommended by the authors. Nevertheless, the program allows for optimization of this parameter by activating the option "**Optimize**". In such a case the current value of this parameter is set equal to the Bergan current stiffness parameter that provides a scalar measure of the degree of nonlinearity. With increasing the degree of nonlinearity this parameter is decreasing. In the vicinity of collapse load the value of this parameter approaches zero and the solution is driven by displacement increment. This strategy thus supports the use of cylindrical method having the **Ratio load/displacement** parameter equal to zero. As for the default setting this option is turned off.



*Arc-length - automatic arc length control*

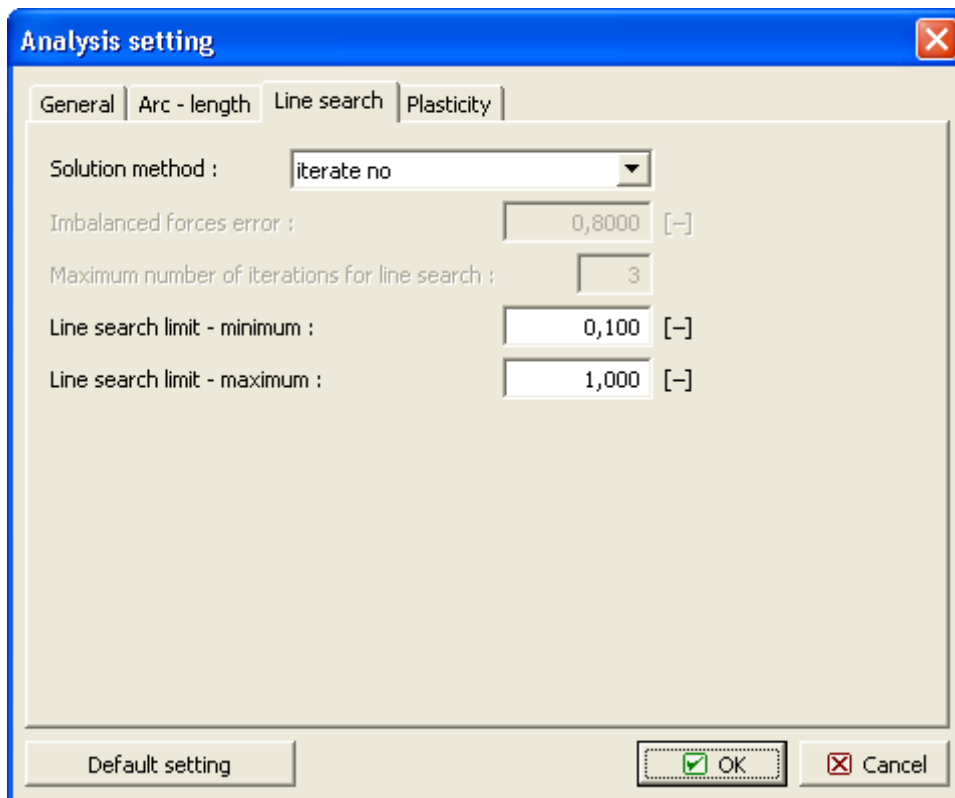
## Line Search Method

The basic goal of the Line search method is to determine a scalar multiplier  $\eta$  that is used to scale the current displacement increment so that the equilibrium is satisfied in a given direction. The actual displacement vector at the end of the  $i$ -th iteration thus becomes:

$$u_i = u_{i-1} + \eta \Delta u$$

Consequently, the calculation process is either accelerated,  $\eta > 1$ , or damped,  $\eta < 1$ . Obviously, with the Line search performed each iteration, the expense of the iteration increases. On the other hand, this drawback is compensated by less number of iterations needed for convergence and by the possibility of avoiding divergence or oscillation of the process of iteration. By default the use of the Line search is enabled.

An inexperienced user is recommended to employ the default setting evident from the figure.



*Line search method settings*

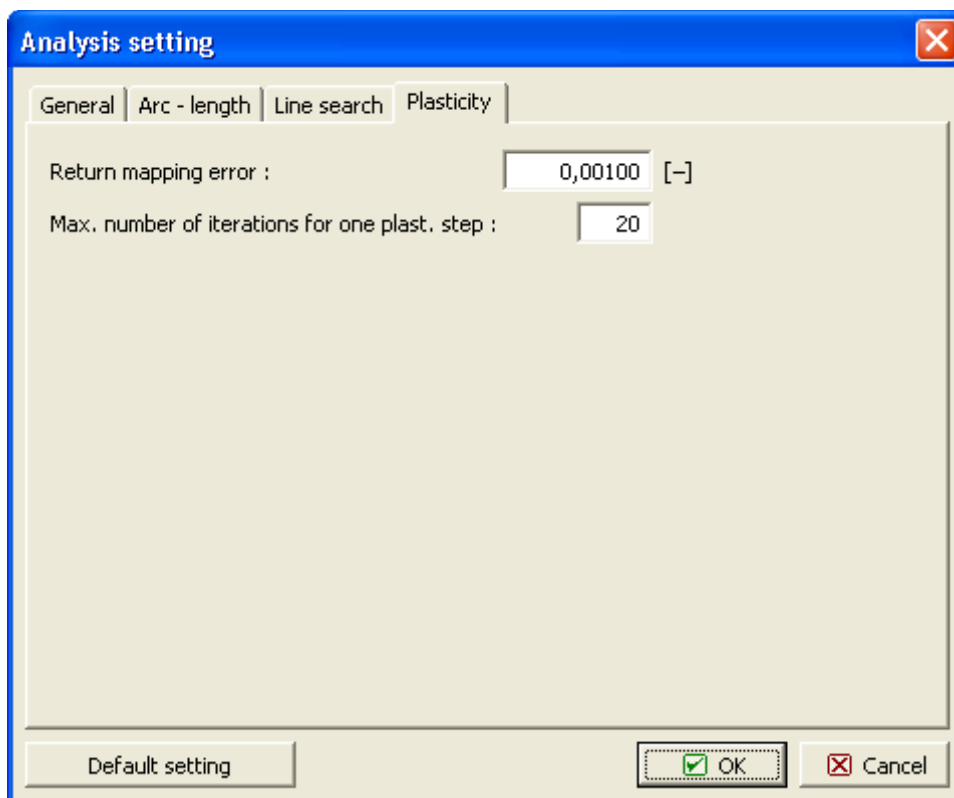
## Plasticity

The **Plasticity** dialog window serves to set parameters driving the stress update procedure.

The parameter **Return to yield surface tolerance** suggests the tolerance for satisfying the selected yield condition. Assuming nonlinear hardening/softening as in the case of modified cam clay model the stress return mapping requires an iteration process.

The maximum number of iterations allowed is then given by the **Max. No. of iterations for a single plastic step** parameter. When employing the rigid-plastic version of the Mohr-Coulomb, the Drucker-Prager or the modified Mohr-Coulomb model, these parameters will not apply.

The default setting, evident in the figure, is recommended.



*Parameters driving the stress return mapping*

## Course of Analysis

The course of analysis can be viewed in the bottom part of the screen.

An **elastic** analysis is completed in **one computational step**. A nonlinear analysis is performed in several steps - the external load is gradually increased in several **load (calculation) steps**. The analysis is completed successfully if there is no loss of overall convergence so that 100 percent of the required load is reached.

The **default setting of parameters** that drive the solution analysis is optimized to ensure sufficient accuracy and efficiency of the analysis. Nevertheless, an experienced user may require to change the default setting, or to examine the influence of parameters on the accuracy and course of the analysis. The parameters setting can be adjusted in the "**Analysis settings**" dialog window:

- The **Percent of the applied load** parameter gives percentage of the overall load (excepted value) at the end of the current load step assuming successful convergence for the current load step.
- The **Step size** parameter provides the current scaling factor for the determination of load increment in the current load step.
- The **Safety factor** parameter corresponds to the expected value of the safety factor assuming successful convergence for given parameters  $c, \varphi$ .

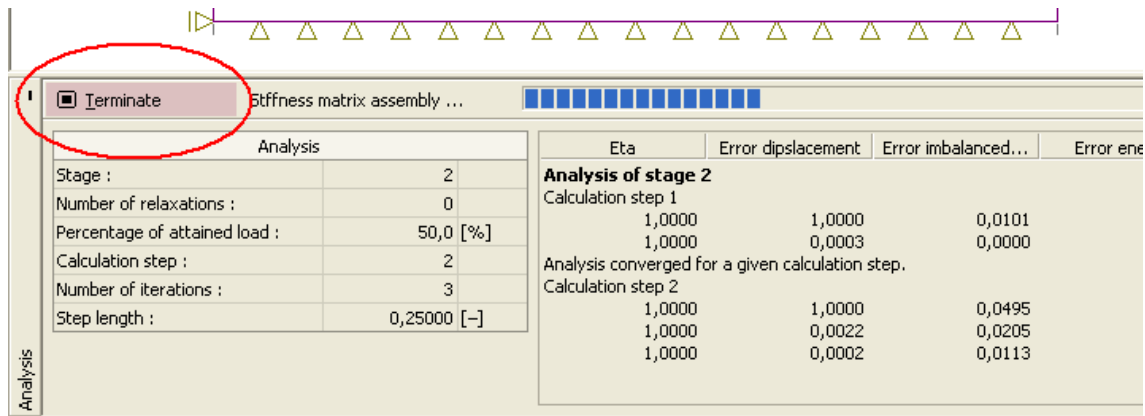
The course of iteration within a given load step is characterized by the change of convergence parameters:

- $\eta$  - Line search method parameter

- change of the displacement increment norm
- change of the out-of-balance force norm
- change of internal energy

If all three errors are smaller than the preset **error tolerance** (can be edited in "**Settings**" dialog window), the analysis is for the calculation step terminated.

The "**Interrupt**" button serves to terminate the calculation process. The results are then available for the last load level that complies with the state of equilibrium.



*Course of analysis*

## Results

Visualization (plotting) of results is one of the most important features of the program. The program allows us to select from several basic styles of graphical outputs, which are defined in the "**FEM - results visualization settings**" dialog window.

- draw **deformed mesh**
- surface plot of variables developed **inside the soil / rock body** (the total values or their increments with respect to other calculation stage can be displayed)
- **internal forces** distributed along beams, contacts
- **forces in anchors and reaction forces**
- **depression curve**
- **tilted sections** of variables
- **vectors and directions** of variables

To display results the program employs the following **coordinate systems**.

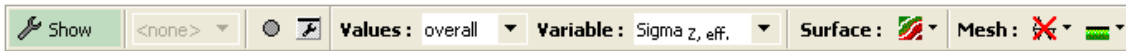
The tool bar "**Results**" in the upper part of the screen serves to selected variables to be displayed and the way they should appear on the screen. The color scheme is shown in the right part of the desktop. Its particular setting can be adjusted using the "**Color scheme**" tool bar.

Because properly setting outputs might be often time consuming, the program disposes of a comfortable system of **storing and managing various settings**.

All outputs and selected results can be further printed out from the **analysis protocol**.


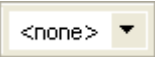


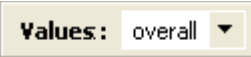
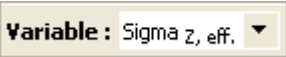
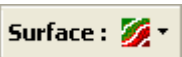
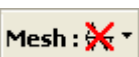

## Results Tool Bar

The tool bar contains the following operating elements:



*Tool bar "Setting visualization of graphical outputs"*

Individual elements operate as follows:

	<b>Plotting style setting</b>	<ul style="list-style-type: none"> <li>opens the "FEM - results visualization settings" dialog window which allows the user to be more specific in defining the plotting style</li> </ul>
	<b>List of plots</b>	<ul style="list-style-type: none"> <li>a combo list containing names of plots saved by the user</li> </ul>
	<b>Save plot</b>	<ul style="list-style-type: none"> <li>saves the <b>current plot</b> displayed on the desktop, the dialog window serves to enter the name of the plot</li> </ul>
	<b>Manager of plots</b>	<ul style="list-style-type: none"> <li>opens the "<b>Manager of plots</b>" dialog window which serves to manage (delete, change order, rename) already saved plots</li> </ul>
	<b>Values in stages of analysis</b>	<ul style="list-style-type: none"> <li>displays calculated values (either total or incremental values with respect to the selected <b>stage of construction</b> can be seen)</li> </ul>
	<b>Variable type</b>	<ul style="list-style-type: none"> <li>displays the selected <b>variable</b></li> </ul>
	<b>Surface plot</b>	<ul style="list-style-type: none"> <li>turns on/off plotting of isolines, isosurfaces</li> </ul>
	<b>Mesh</b>	<ul style="list-style-type: none"> <li>turns on/off the style of plotting the FE mesh (only edges, or according to the setting in the "FEM - results visualization settings" dialog window</li> </ul>
	<b>Displacements plot</b>	<ul style="list-style-type: none"> <li>selects the style of plotting deformed mesh - undeformed/deformed (deformed by the magnitude, deformed by the coefficient)</li> </ul>

The tool bar contains **the most often used operating elements** needed to view the results on the desktop. Detailed setting of the style of plotting the results is available in the "FEM - results visualization settings" dialog window.

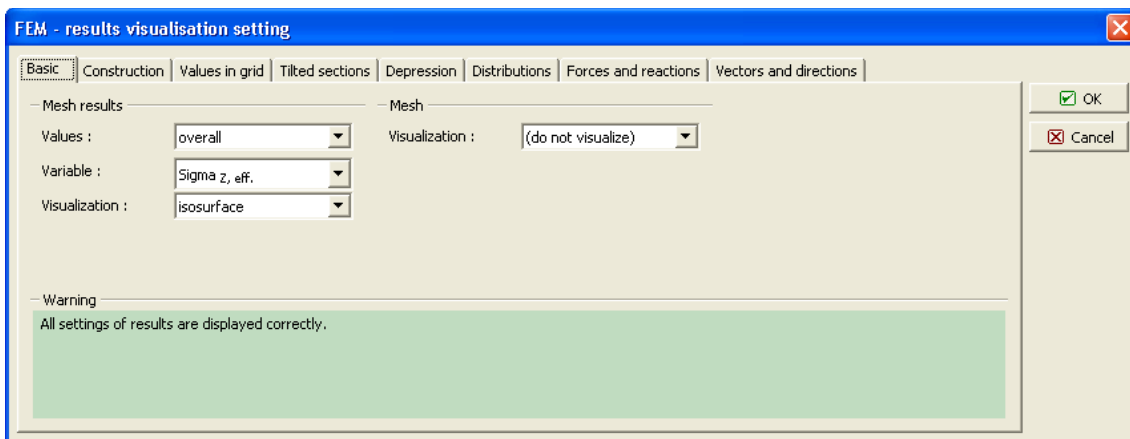
Similar to our other programs the results can be saved and printed. The plotting style can be adjusted in the "Visualization style settings" dialog window.

## Results Visualization Settings

The "FEM - results visualization settings" dialog window serves to select the type of variable to be displayed and the way it should appear on the screen. Individual settings can be later saved using the "Results" tool bar.

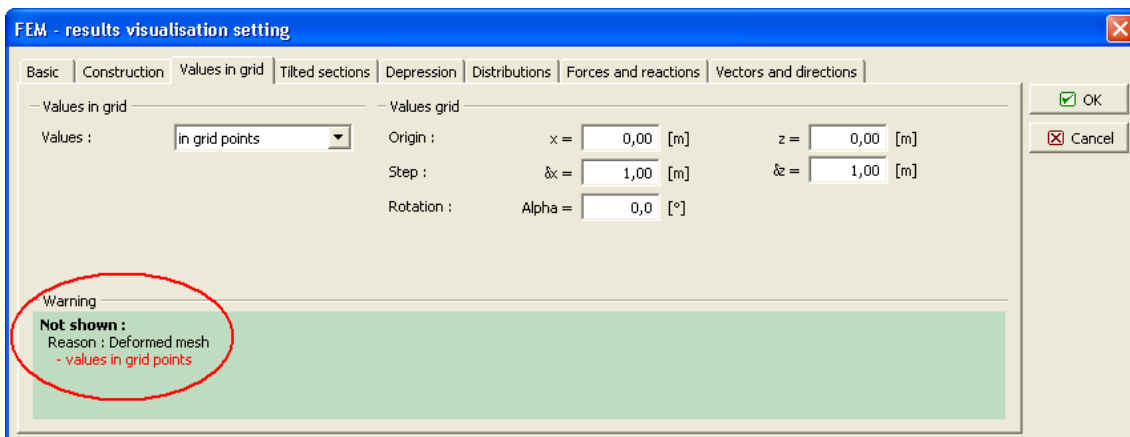
The tab "**Basic**" serves to set the basic parameters driving the visualization of surface variables and FE mesh - other tabs are used to define other types of outputs.





Dialog window "FEM - results visualization settings " - tab "Basic"

Owing to the clarity of graphical presentation it is not possible to plot some of the results **at the same time**. It is not possible to plot a deformed mesh together with distributions of internal forces along beams - only one option must be selected. If an unacceptable combination is selected, the displays a warning message in the bottom part of the dialog window. The present example shows an unacceptable combination of *deformed mesh/values in mesh grid* set in the tab "**Basic**".



Warning for conflict in plotting of results

## List of Variables

The following variables can be displayed (values in the soil/rock body):

### List of variables displayed by the program - basic variables

Notation	Description	Variable	Unit
Settlement $d_z$	Displacement in the $Z$ direction	$d_z$	[mm]
Settlement $d_x$	Displacement in the $X$ direction	$d_x$	[mm]
Sigma $Z, tot.$	Total normal stress in the $Z$ direction	$\sigma_{z,tot}$	[kPa]

Sigma $Z, eff.$	Effective normal stress in the $Z$ direction	$\sigma_{z,eff}$	[kPa]
Pore pressure $u$	Pore pressure	$u$	[kPa]
Sigma $X, tot.$	Total normal stress in the $X$ direction	$\sigma_{x,tot}$	[kPa]
Sigma $X, eff.$	Effective normal stress in the $X$ direction	$\sigma_{x,eff}$	[kPa]
Tau $X, Y.$	Shear stress	$\tau_{xz}$	[kPa]
Epsilon $eq.$	Equivalent strain	$\epsilon_{eq}$	[-]
Epsilon $eq., pl.$	Equivalent plastic strain	$\epsilon_{eq,pl}$	[-]

**List of variables displayed by the program - variables available in the regime "Extended input".**

Notation	Description	Variable	Unit
Epsilon $vol.$	Volumetric strain	$\epsilon_{vol.}$	[-]
Sigma $m, tot.$	Mean total normal stress	$\sigma_{m,tot}$	[kPa]
Sigma $m, eff.$	Mean total normal stress	$\sigma_{m,eff}$	[kPa]
Sigma $eq.$	Equivalent deviatoric stress	$J$	[kPa]
Epsilon $vol., pl.$	Volumetric plastic strain	$\epsilon_{vol,pl}$	[-]
Epsilon $X$	Normal strain in the $X$ direction	$\epsilon_x$	[-]
Epsilon $Z$	Normal strain in the $X$ direction	$\epsilon_z$	[-]
Gama $XZ$	Shear strain in the $XZ$ plane	$\gamma_{xz}$	
Epsilon $1, princ.$	Maximal principal strain	$\epsilon_{1,princ}$	[-]
Epsilon $2, princ.$	Intermediate principal strain	$\epsilon_{2,princ}$	[-]
Epsilon $3, princ.$	Minimal principal strain	$\epsilon_{3,princ}$	[-]

Sigma $1, princ.$	Maximal principal stress	$\sigma_{1,princ}$	[kPa]
Sigma $2, princ.$	Intermediate principal stress	$\sigma_{2,princ}$	[kPa]
Sigma $3, princ.$	Minimal principal stress	$\sigma_{3,princ}$	[kPa]
Epsilon $X, pl.$	Normal plastic strain in the $X$ direction	$\varepsilon_{x,pl}$	[-]
Epsilon $Z, pl.$	Normal plastic strain in the $Z$ direction	$\varepsilon_{z,pl}$	[-]
Gama $XZ, pl.$	Shear plastic strain in the $XZ$ plane	$\gamma_{xz,pl}$	

## Monitors

The frame "**Monitors**" contains a table with the list of input monitors. Adding (editing) monitors is performed in the "**New monitors**" dialog window.

Either point or line-monitors can be introduced. The dialog window then serves to specify coordinates of the monitor and monitor activity.

The monitors can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar "**Monitors**". The following modes are available:

- **Add**                    The monitor is introduced by clicking the left mouse button at a desired location on the desktop.
- **Modify**                Clicking the left mouse button on already existing monitor opens the "**Adjust monitor**" dialog window, which allows for modifying its parameters.
- **Remove**                Clicking the left mouse button on already existing monitor opens the **monitor removal** dialog window - accepting this action removes the selected free point.

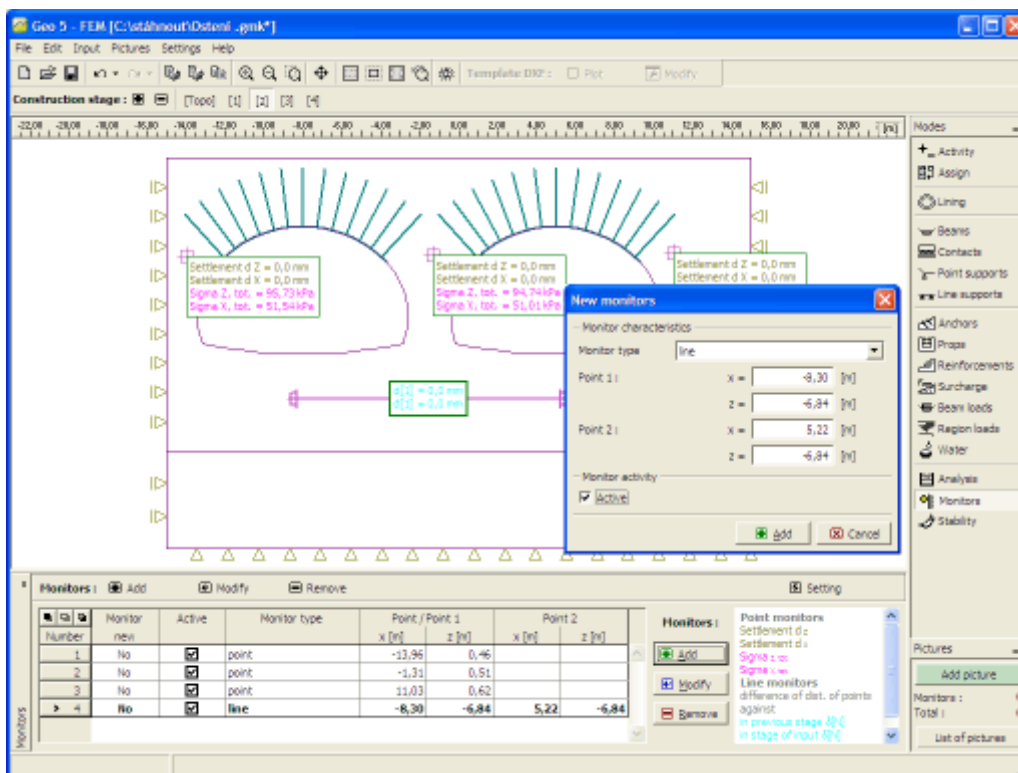
The monitors can also be edited on the desktop with the help of **active objects**.

The program allows for inputting an arbitrary number of point and line-monitors anywhere in the structure and also out of it. Monitors have several functions:

- displaying values of **variables** in a given point (point-monitor),
- displaying values of the difference of distance of two points in comparison with the **previous stage**  $d[N]$  or in comparison with the input stage, where  $N$  is the stage number (line-monitor).

The point monitors store also the values of variables recorded during the analysis in individual stages. These can be written into an output protocol or used to **create graphs**.

The list of variables plotted for individual monitors is set in the "**Monitor settings**" dialog window. To open the window use the "**Settings**" button in the horizontal tool bar "**Monitors**".



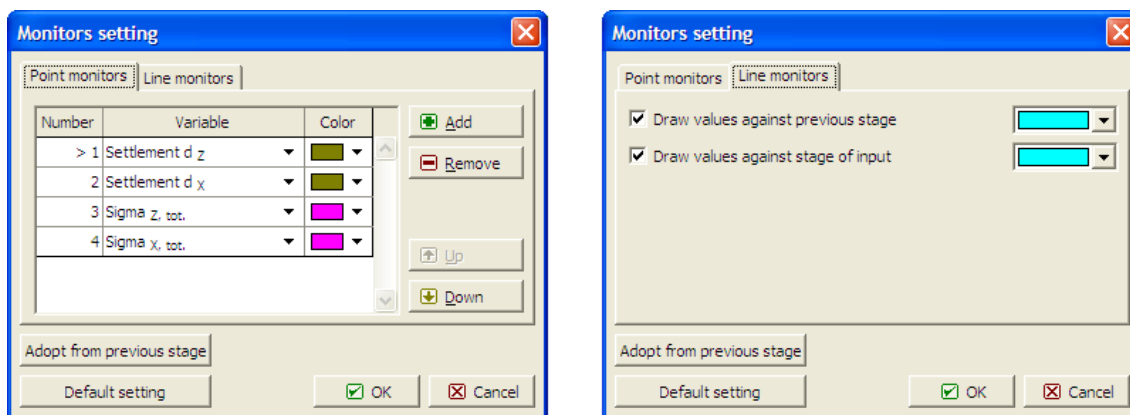
Frame "Monitors"

## Monitors Settings

The "**Monitor settings**" dialog window serves to set **variables** whose values will be plotted for a given monitor (point-monitors). Setting for a given list of variables can be adopted from the previous stage of construction using the "**Adopt from the previous stage**" button. Four variables are displayed by default. Additional variables can be added to the list using the "**Add**" button. The variable can be removed from the list using the "**Remove**" button.

For line-monitors the dialog window serves to activate the plot of values in comparison with the previous stage or the input stage, respectively.

For both point and line-monitors it is possible to specify the color range of plotted values.



Dialog window "Monitors settings"

## Graphs

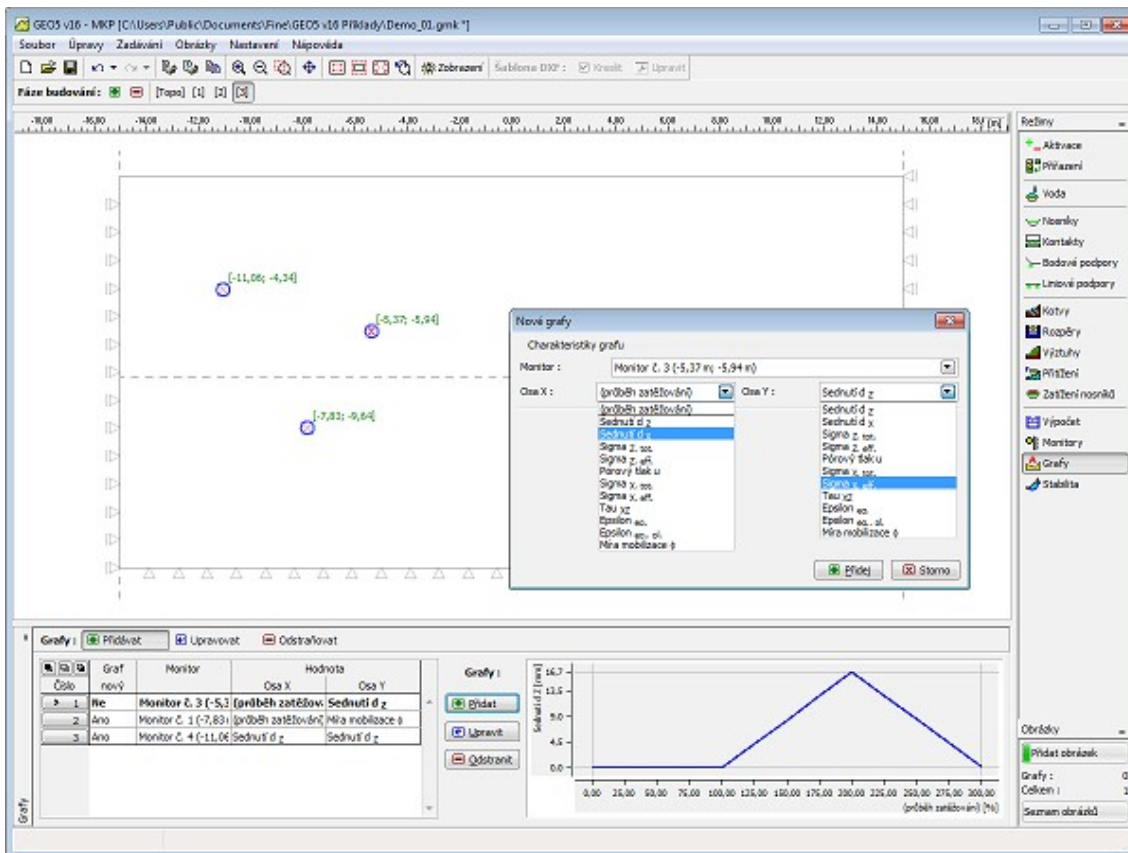
The **frame "Graphs"** contains a **table** with the list of input graphs. Adding (editing) monitors is performed in the **"New graphs"** dialog window. The dialog window serves to enter the **monitor** number for which the graph will be created and the variables adopted for the *X* and *Y*-axis respectively.

The graphs can also be introduced **using the mouse**. This inputting mode is activated by clicking an appropriate button on the horizontal tool bar **"Graphs"**. The following modes are available:

- **Add** The graph location is introduced by clicking the left mouse button at a desired monitor.
- **Modify** Clicking the left mouse button on already existing graph opens the **"Adjust graph"** dialog window, which allows for modifying its parameters.
- **Remove** Clicking the left mouse button on already existing graph opens the **graph removal** dialog window - accepting this action removes the selected graph.

The graphs can also be edited on the desktop with the help of **active objects**.

The program allows for inputting an arbitrary number of graphs at points of input **monitors**. Graphs allow for plotting a mutual dependence of individual variables stored in monitors during the course of analysis.



Frame "Graphs"

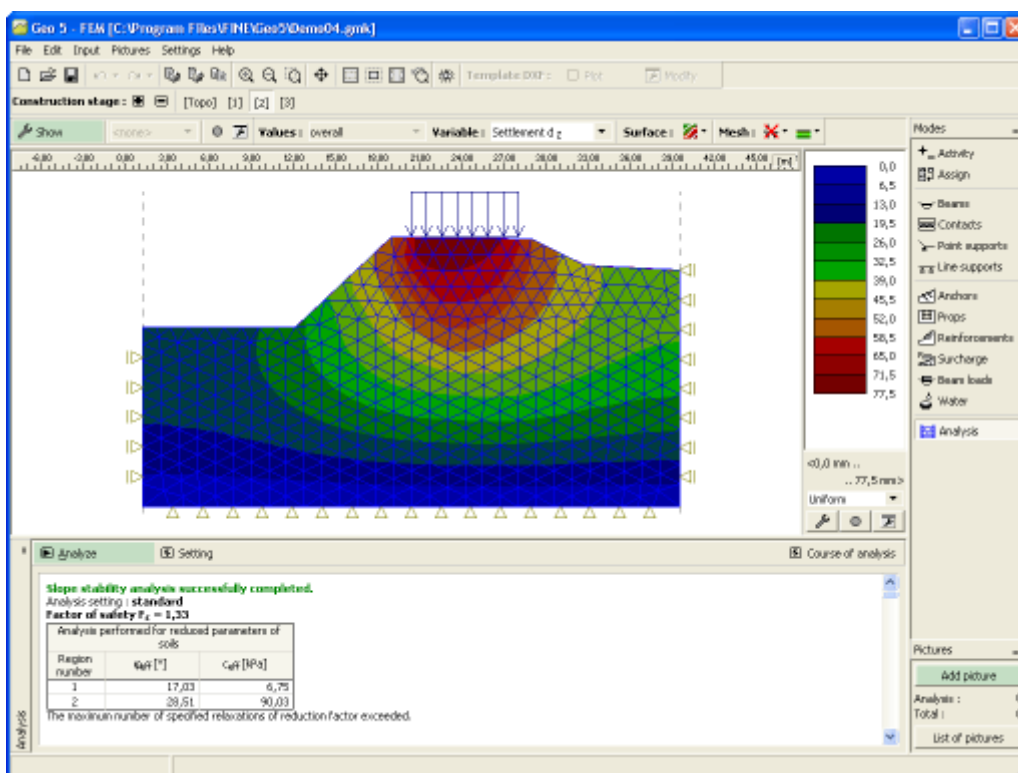
## Stability

In stability (safety factor) analysis the program **reduces the original strength parameters** - angle of internal friction and cohesion - until failure occurs. The analysis then results into a **factor of safety** that corresponds to the classical methods of limit equilibrium.

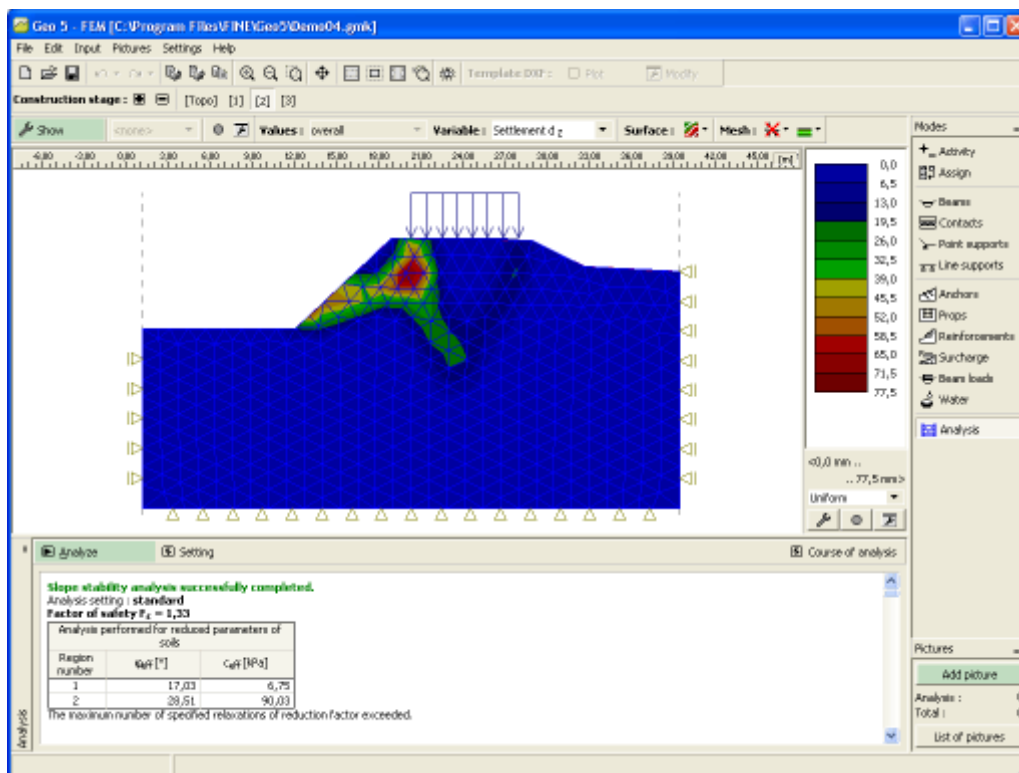
The safety factor **analysis** requires using **six-node elements**. Since plastic slip is the main failure mechanism we also require that the **Mohr-Coulomb**, the **modified Mohr-Coulomb** or the **Drucker-Prager** plasticity model be used for all soils. Default **setting** can be adjusted in the **"Analysis settings"** dialog window.

In the stability analysis mode the only variables available for graphical representation are **displacements** (in the Z and X-directions) and equivalent total and plastic **strains**. The deformation of a soil body corresponds to the state of failure attained for the reduced soil parameters - therefore, it does not correspond to real state of deformation of the soil body. Instead, it provides a good insight about the entire slope response of earth structure in general at the onset of failure.

A suitable way of presenting the stability analysis results are **vectors of displacements** plotted together with the **equivalent plastic strain**. The localized plastic deformation provides visible evidence about the possible location of the **critical slip surface**.



Frame "Stability"



Plot of equivalent plastic strain - slip surface

## Setting Basic Parameters of Slope Stability Analysis

The safety factor analysis is based on the assumption that the total load applied to the soil/rock body is introduced in a single load step. The actual factor of safety is evaluated using the **method of reduction of strength parameters**  $c, \varphi$ . Regarding this the factor of safety is defined as a scalar multiplier that reduces the original parameters  $c, \varphi$  to arrive at the state of failure.

Mathematically, the **factor of safety** is expressed as:

$$F = \frac{\tan \varphi^{orig}}{\tan \varphi^{failure}}$$

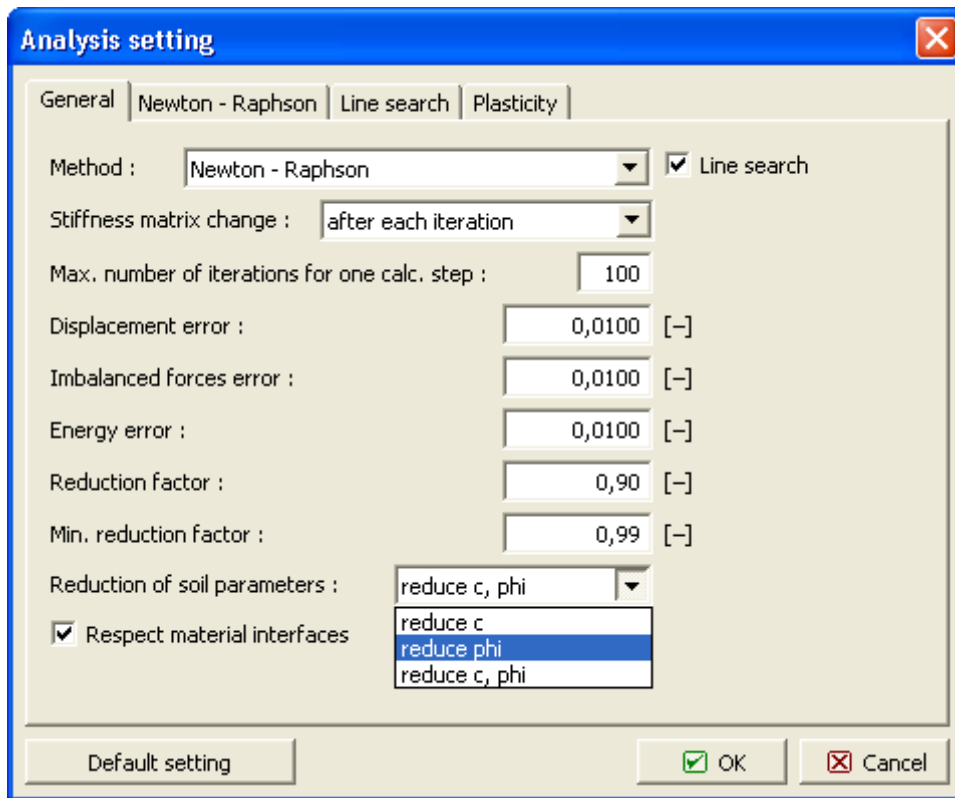
where:  $\varphi^{original}$  the original value of the angle of internal friction  
 $\varphi^{failure}$  the value of the angle of internal friction at failure

Searching for the critical value of the factor of safety requires a systematic modification (reduction) of strength parameters  $c, \varphi$  leading to failure. In the framework of the NRM the state of failure is determined as the state for which the solution fails to converge. The process of searching for critical  $c, \varphi$  is driven by the following parameters.

1. **Reduction** - reduction factor (scalar multiplier) to reduce parameters  $c, \varphi$ . During the course of analysis this parameter is **progressively updated**.
2. **Min. reduction factor** - the limit value, below which the value of reduction factor should not fall during the searching process. This parameter ensures that the

computation will not continue for needless low values of the reduction factor. It is one of the parameters to terminate the searching process.

3. **Reduction of soil parameters** - this parameter allows us to define which of the parameters  $c$ ,  $\phi$  should be reduced. The default setting assumes that both parameters are reduced at the same time.



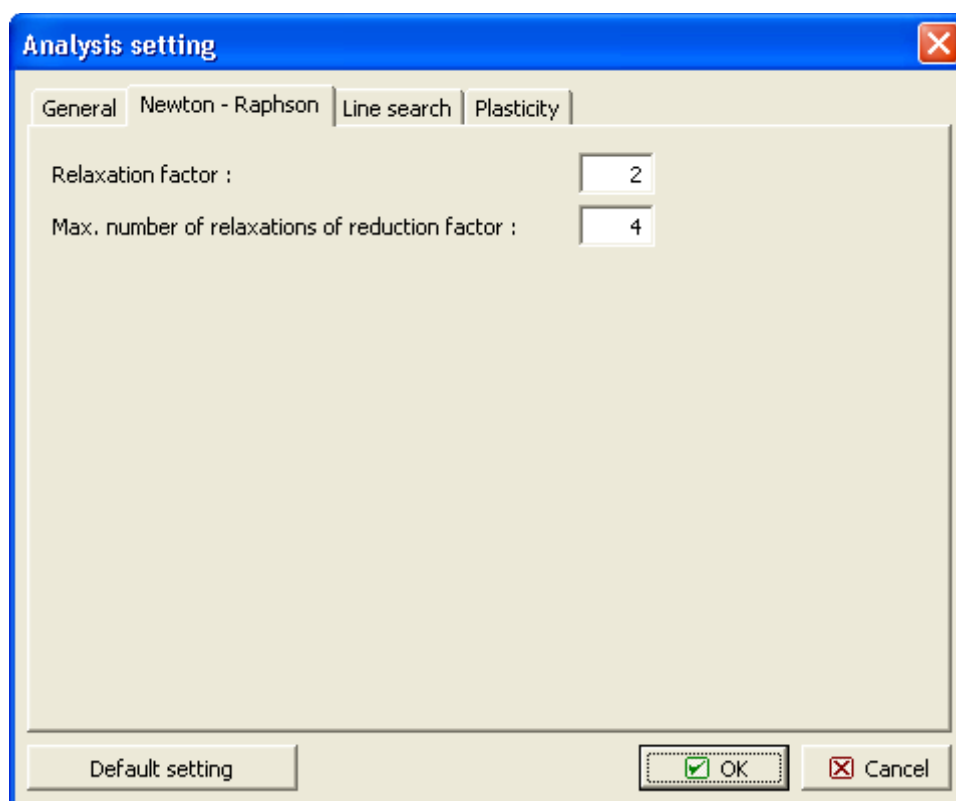
*Basic parameters of slope stability analysis*

## Setting Driving Parameters of Relaxation of Reduction Factor

Similar to standard analysis the program adaptively adjusts the value of reduction factor. Providing the solution fails to converge for a given set of parameters  $c$ ,  $\phi$ , the reduction factor is relaxed and the analysis is restarted. This approach is driven by the parameters set in the "Analysis settings" - tab **Newton Raphson**.

The "**Relaxation factor**" serves to reduce the current value of the Reduction factor of parameters  $c$ ,  $\phi$ . The analysis is terminated once the value of reduction factor drops below the minimum one or the maximum number of allowable reductions is exceeded. When selecting the NRM the program allows us to determine the parameters  $c$ ,  $\phi$  which bring a soil body to a stable state in cases, where the solution with the original parameters  $c$ ,  $\phi$  was not found. The program then precedes in the opposite way so that parameters  $c$ ,  $\phi$  are **systematically increased until the stable solution is found**.





*Parameters driving the process of reduction of strength parameters  $c$ ,  $\varphi$*

## Outputs

This chapter describes the work with outputs (export document, printing pictures) in GEO5 programs using the toolbar "**Outputs**".

GEO5 programs allows to create **output document** with saved pictures from any mode of input or analyses. The pictures can be printed or exported too.

- [Print and Export Document](#)
- [Print and Export Picture](#)

**Work with pictures** is described on pages:

- [Adding pictures](#)
- [List of pictures](#)

**Page properties** can be defined in each output document:

- [Settings Header and Footer](#)
- [Page Properties](#)
- [Page Numbering](#)

The information about the company can be input and print in output document:

- [About the Company](#)

## Adding Pictures

The program allows to store the current picture in all modes. Press the **"Add picture"** button on the [vertical tool bar](#). This opens the **"New picture"** dialog window and inserts the current view on the desktop into the window.

The picture is always linked to a certain input mode or analysis. (The current mode is displayed next to the picture name). When [printing a document](#) the picture is automatically added to a specific mode in the tree.

The program allows to define the picture either for a specific stage of construction (or for the current analysis) or adjusting the setting such that the picture is added to the document in all stages of construction (or all analyses). The latter option is assumed when selecting **"all"** in the **"Stages"** combo list (or **"Analysis"** list).

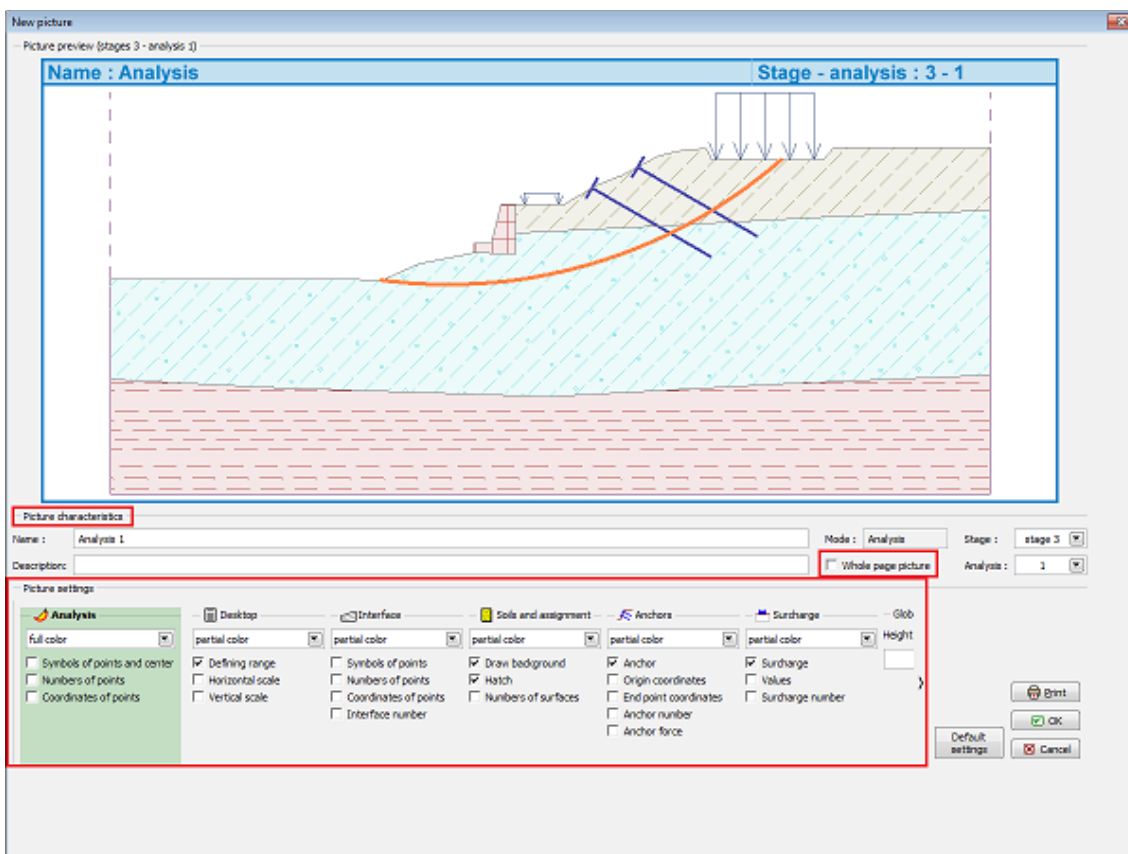
Checkbox **"Whole page picture"** allows to use whole page picture in the document.

**Warning: All input pictures are automatically regenerated whenever modifying data.**

The **"Picture settings"** frame in bottom part of the dialog window further allows to adjust colors and style of lines (object) drawing. Settings in this part of the frame is taken from the [visualization settings](#) for the desktop. The function of the frame is the same as the desktop visualization settings and is described in ["Visualization Settings"](#).

The **"OK"** button stores the picture into the ["Picture list"](#). It can then be opened and modified at any time.

The picture can also be printed out from this window by pressing the **"Print"** button, which opens a dialog window for [printing and exporting pictures](#). If the picture is active over all stages (or all analyses), then all possible combinations of pictures are printed all at once.



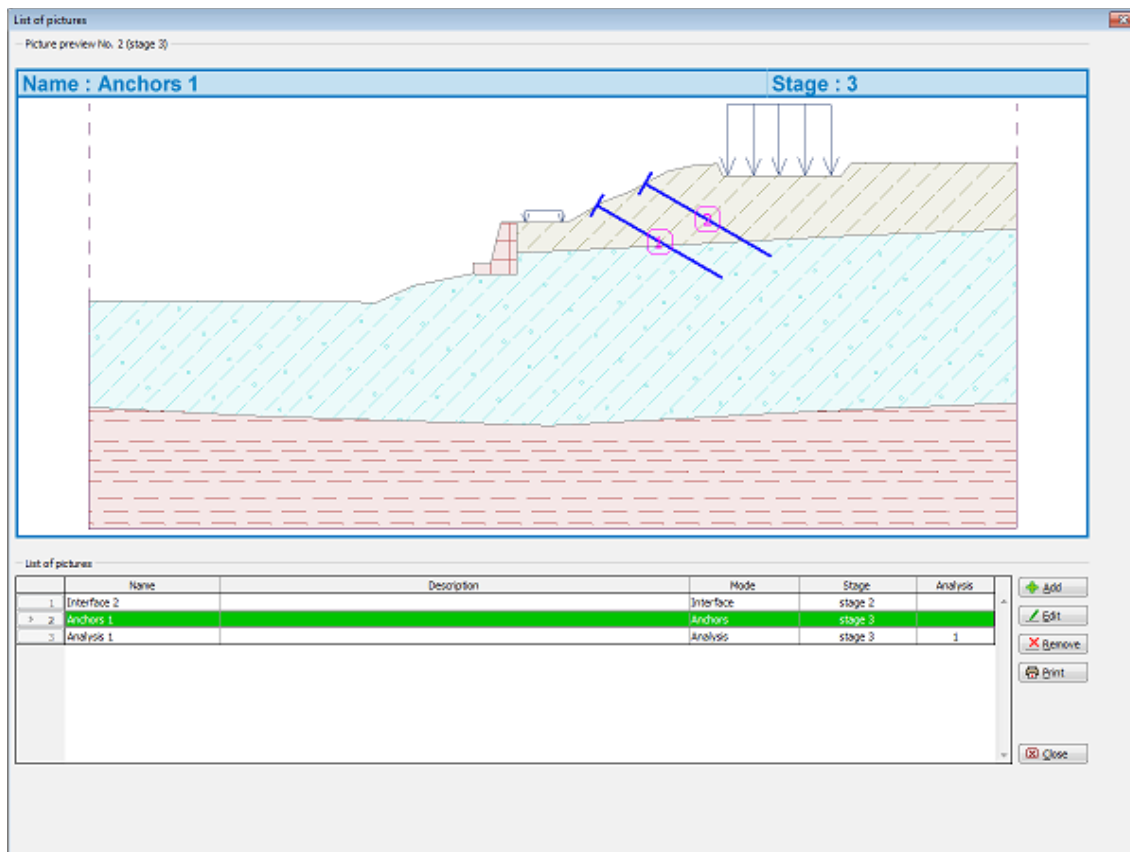
Dialog window "New picture"

## List of Pictures

Stored pictures by the "New picture" dialog window are ordered in the table in a "List of pictures". The "List of pictures" dialog window is opened using the button on the vertical tool bar. The table of the list of pictures contains the picture name and description, the mode in which it was created and stage of construction or the analysis number.

Individual pictures can be edited using the "Modify" button, which opens the "Edit picture" dialog window (this window corresponds to the "New picture" dialog window both in the way it looks and the way it functions).

These pictures can be printed out from the window by pressing the "Print" button that opens a dialog window for printing and exporting the picture. Providing the picture is active over all stages of construction (over all analyses, respectively) then the program prints all possible combinations of the picture. If more pictures are selected then all are printed out.



Dialog window "List of pictures"

## Print and Export Document

The "Print and export document" dialog window can be opened either from the **control menu** (items "Files", "Print document") or using the "Outputs" button on the tool bar. The page print preview with a generated text appears in the window.

This window generates the output document including pictures stored in the "Picture list". This window allows either to print the created protocol or export it for further use. The **document is always up to date** - the program creates the document again based on input data (even with regenerated pictures) whenever opening this window.

Only specific parts of the document including pictures can be generated by checking the corresponding "tree" item in the left part of the window. Selecting or deselecting an arbitrary item prompts the program to regenerate the document automatically.

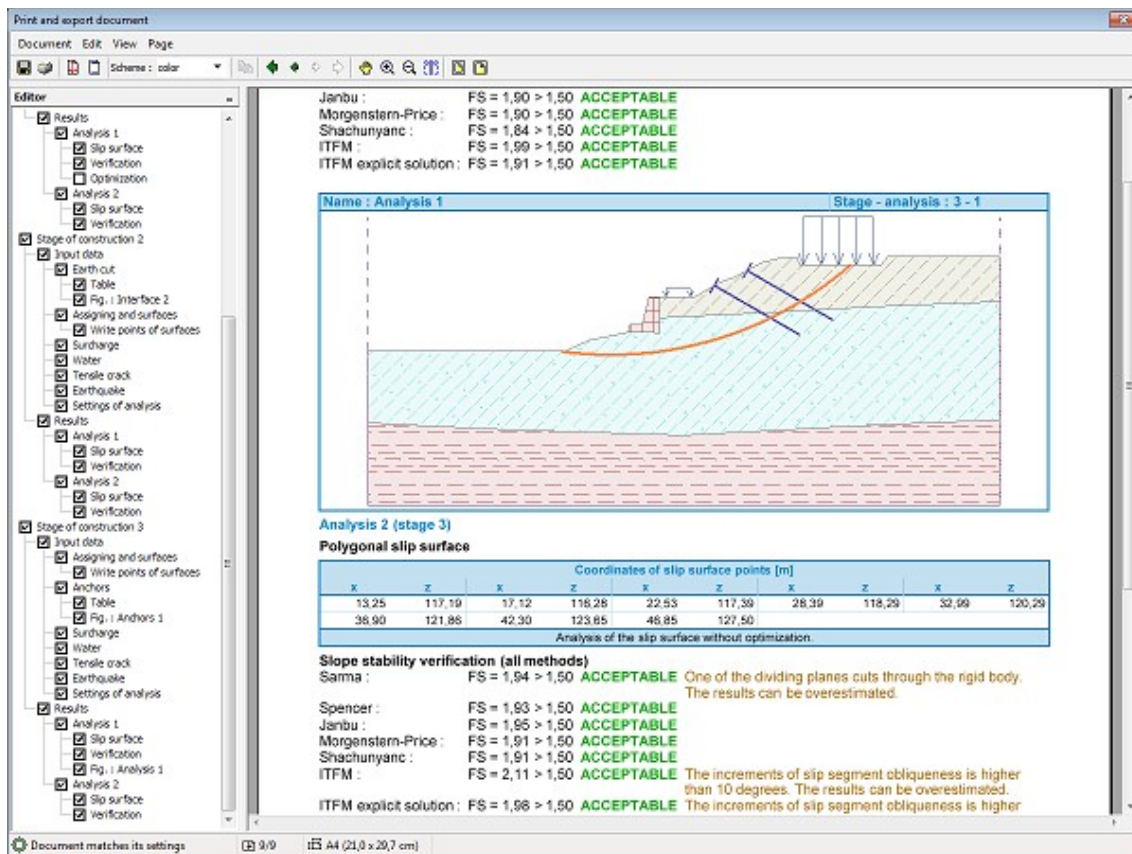
The document editor contains its own control elements:

- **Control menu - Print and Export**
- **Tool Bar - Print and Export**

These control elements are also used when customizing the appearance of the pages (**header and footer definition**, **page properties**, **page numbering**), print and export of the document.

The mouse scroll wheel or scroll bar on the right can also be used to view the document.

The button part of the dialog window displays current information (defined page size, current document page and the total number of pages).



Dialog window "Print and export document"

## Print and Export Picture

This dialog window allows to print and export one or more pictures. Three options are available to open this window:

- Using the **control menu** (items "**Files**", "**Print picture**") or the "**Files**" button on the tool bar to print data from the desktop.
- Using the "**New picture**" dialog window by pressing the "**Print**" button.
- Using the "**List of pictures**" dialog window by pressing the "**Print**" button.

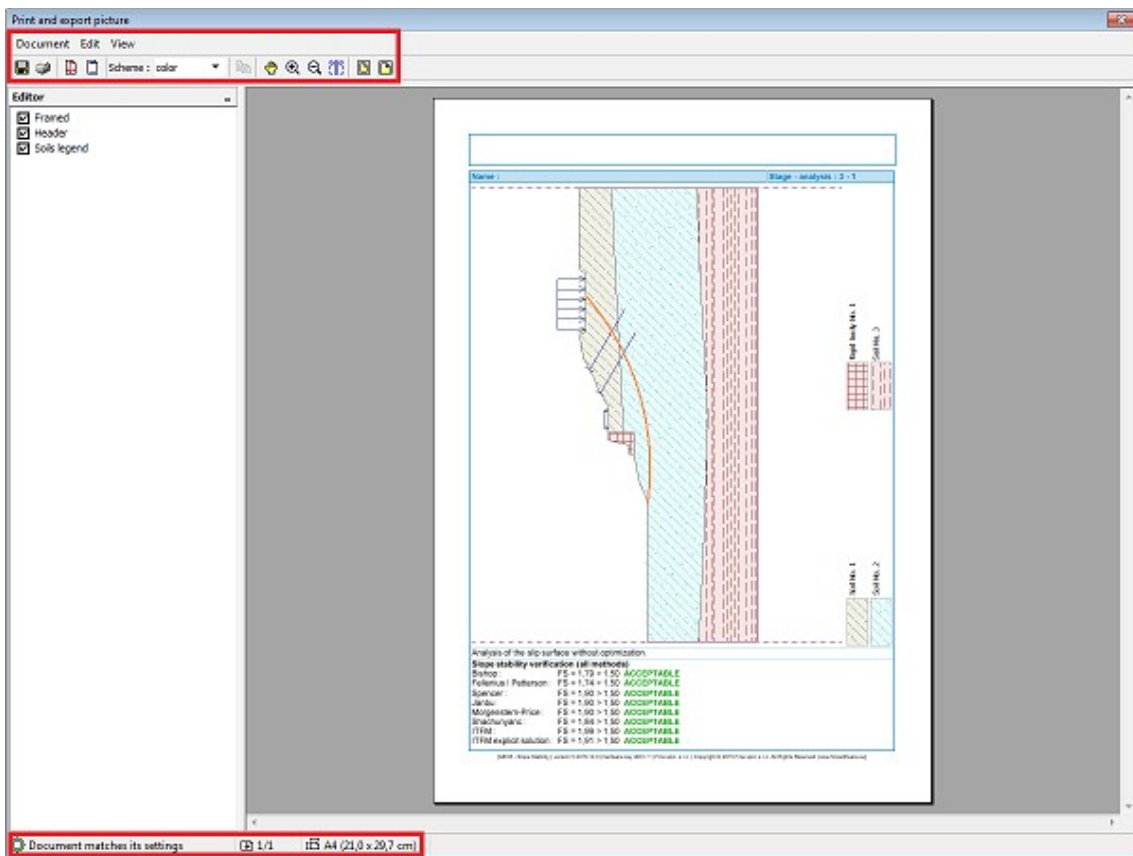
The window may contain more than one picture at the same time (when printing more construction stages or analyses) when printing more pictures from the list. Each picture is printed on a separate page. The picture preview can be adjusted by the buttons on the tool bar or by mouse scroll wheel.

The document editor contains its own control elements:

- **Control menu - Print and Export**
- **Tool Bar - Print and Export**

These control elements are also used when customizing the appearance of the pages (**header and footer definition**, **page properties**, **page numbering**), print and export of the document.

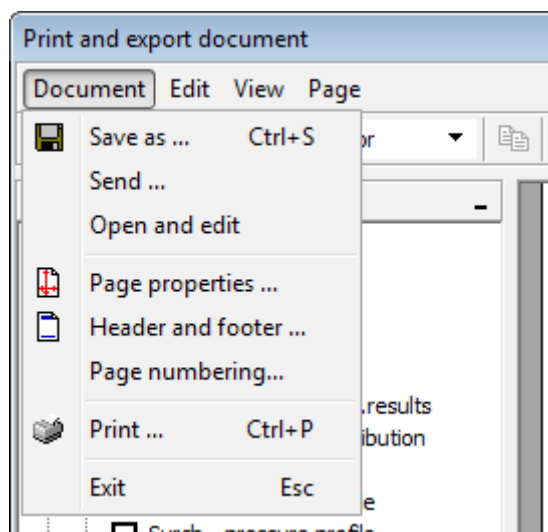
The button part of the dialog window displays current information (defined page size, current document page and the total number of pages).



Dialog window "Documents" - print and export current picture (view)

## Control Menu - Print and Export

The control menu of the "Print and export document" and "Print and export pictures" dialog windows contains the following items:



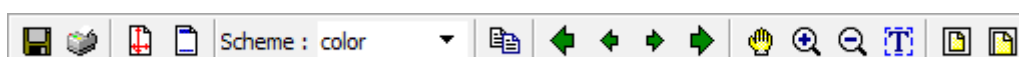
Control Menu

### Document

<b>Save as</b>	<ul style="list-style-type: none"> <li>• opens the "<b>Save as</b>" dialog window that allows to save the file in *.PDF, or *.RTF formats</li> </ul>
<b>Send</b>	<ul style="list-style-type: none"> <li>• opens a dialog window for mail client and adds the picture as an attachment in *.PDF format</li> </ul>
<b>Open and edit</b>	<ul style="list-style-type: none"> <li>• opens text editor (associated in the Windows system with *.RTF extension) that allows to edit the page manually</li> </ul>
<b>Page properties</b>	<ul style="list-style-type: none"> <li>• opens the "<b>Page properties</b>" dialog window that allows to specify the page style (size, edges, layout)</li> </ul>
<b>Header and footer</b>	<ul style="list-style-type: none"> <li>• opens the "<b>Header and footer</b>" dialog window that allows to input the document header and footer</li> </ul>
<b>Page numbering</b>	<ul style="list-style-type: none"> <li>• opens the "<b>Page numbering</b>" dialog window that allows to input the document page numbering</li> </ul>
<b>Print</b>	<ul style="list-style-type: none"> <li>• opens the dialog window for "<b>Print</b>"</li> </ul>
<b>Exit</b>	<ul style="list-style-type: none"> <li>• closes the dialog window</li> </ul>
<b>Edit</b>	
<b>Copy</b>	<ul style="list-style-type: none"> <li>• copies the selected picture (text) to clipboard - parameters are set in the "<b>Options</b>" dialog window - tab "<b>Copy to clipboard</b>"</li> </ul>
<b>Select all</b>	<ul style="list-style-type: none"> <li>• marks everything on the page (in the document) into block</li> </ul>
<b>Cancel selection</b>	<ul style="list-style-type: none"> <li>• cancels entire selection (picture, text)</li> </ul>
<b>View</b>	
<b>Whole page</b>	<ul style="list-style-type: none"> <li>• modifies the page size such that the entire page in the dialog window is visible</li> </ul>
<b>Page width</b>	<ul style="list-style-type: none"> <li>• fits the page to a maximum width of the document dialog window</li> </ul>
<b>Page</b> (this item appears in the menu only if the document has more than one page)	
<b>First page</b>	<ul style="list-style-type: none"> <li>• shows the document first page</li> </ul>
<b>Previous page</b>	<ul style="list-style-type: none"> <li>• shows the previous page</li> </ul>
<b>Next page</b>	<ul style="list-style-type: none"> <li>• shows the next page</li> </ul>
<b>Last page</b>	<ul style="list-style-type: none"> <li>• shows the document last page</li> </ul>
















## Tool bar - Print and Export

The tool bar of the "**Print and export document**" and "**Print and export picture**" dialog windows contains the following buttons:



*Tool bar "Print and export"*

Individual buttons function as follows:

	<b>Save as</b>	<ul style="list-style-type: none"> <li>• opens the "Save as" dialog window allowing to save the file in formats *.PDF, or *.RTF</li> </ul>
	<b>Print</b>	<ul style="list-style-type: none"> <li>• opens the dialog window for "<b>Print</b>"</li> </ul>
	<b>Page properties</b>	<ul style="list-style-type: none"> <li>• opens the "<b>Page properties</b>" dialog window that allows to specify the page style (size, edges, orientation)</li> </ul>
	<b>Header and footer</b>	<ul style="list-style-type: none"> <li>• opens the "<b>Header and footer</b>" dialog window that allows to input the document headers and footers</li> </ul>
Scheme : color ▾	<b>Color style</b>	<ul style="list-style-type: none"> <li>• determines the style of picture view (color, gray scale, black &amp; white)</li> </ul>
	<b>Copy</b>	<ul style="list-style-type: none"> <li>• copies the selected picture (text) to clipboard - parameters are set in the "<b>Options</b>" dialog window - tab "<b>Copy to clipboard</b>"</li> </ul>
	<b>First page</b>	<ul style="list-style-type: none"> <li>• shows the document first page</li> </ul>
	<b>Previous page</b>	<ul style="list-style-type: none"> <li>• shows the previous page</li> </ul>
	<b>Next page</b>	<ul style="list-style-type: none"> <li>• shows the next page</li> </ul>
	<b>Last page</b>	<ul style="list-style-type: none"> <li>• shows the document last page</li> </ul>
	<b>Move</b>	<ul style="list-style-type: none"> <li>• moves the current view in an arbitrary direction - to proceed move mouse in the desired location while keeping the left mouse button pressed</li> </ul>
	<b>Zoom in</b>	<ul style="list-style-type: none"> <li>• scales up the desktop view while keeping location of the point under the axis cross unchanged - this action is repeated using the left mouse button, the right button leaves the zooming mode</li> </ul>
	<b>Zoom out</b>	<ul style="list-style-type: none"> <li>• scales down the desktop view while keeping location of the point under the axis cross unchanged - this action is repeated using the left mouse button, the right mouse button leaves the zooming mode</li> </ul>
	<b>Text selection</b>	<ul style="list-style-type: none"> <li>• allows to select the text under the axis cross - to proceed move mouse over the desired text while keeping the left mouse button pressed</li> </ul>
	<b>Full page</b>	<ul style="list-style-type: none"> <li>• modifies the page size such that the entire page in the dialog window is visible</li> </ul>
	<b>Page width</b>	<ul style="list-style-type: none"> <li>• fits the page to maximum width of the document dialog window</li> </ul>

## Setting Header and Footer

The dialog window allows to define properties of the document header and footer. The "**print header (footer)**" check box determines whether to print the document header (footer).

Header and footer lines may contain an arbitrary text and inserted objects implicitly defined by the program. These objects receive program information such as:

- From the "**Company data**" dialog window (company name, logo, address)



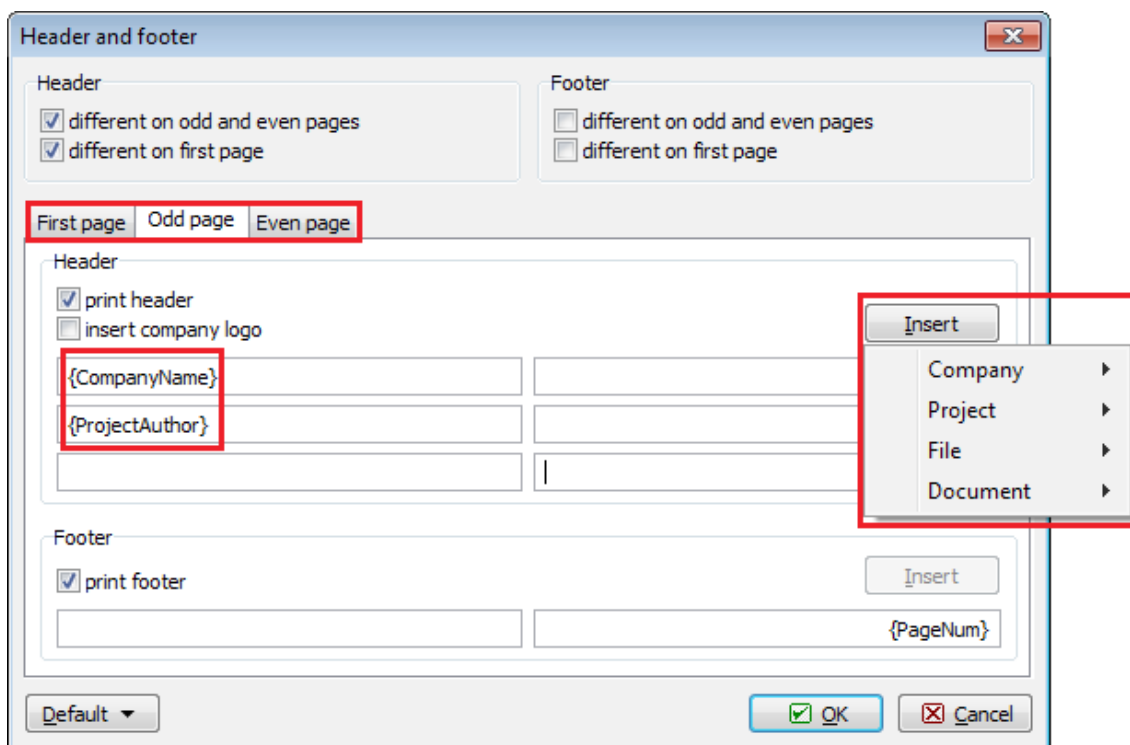
- From the "**Project**" frame (name and task description, author)
- From the document system data (date, time, page numbering)

Objects can be input using the "**Insert**" button (the button opens a list of objects). The button is active only if the cursor is found in one of the line that allows to insert text (object). Inserted objects are written in an internal format different from other text and placed in curly brackets.

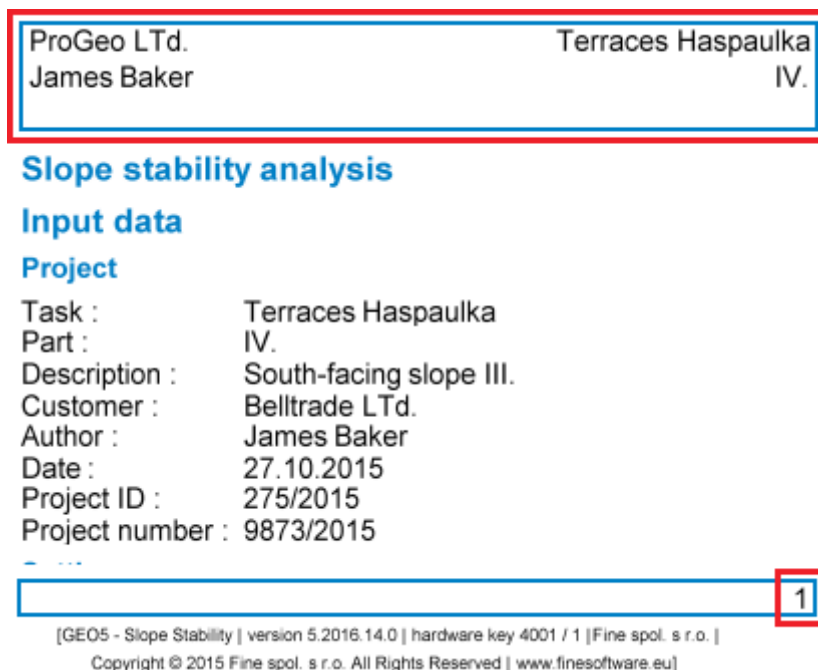
The program allows to defin various headers for the first page or odd and even pages, respectively. Individual headers are in such a case defined in separate tabs.

The "**Default**", selection "**Save settings as default**" option sets the input header and footer parameters as default for the newly created data. **The assumed default setting is common for all our programs.** Different computer users may use different settings. Selection "**Adopt default settings**" enables to adopt the default settings of the GEO5 programs to any opened task, which had the settings different.

Writing format and the resulting view are evident from the following pictures.



Dialog window "Header and footer"

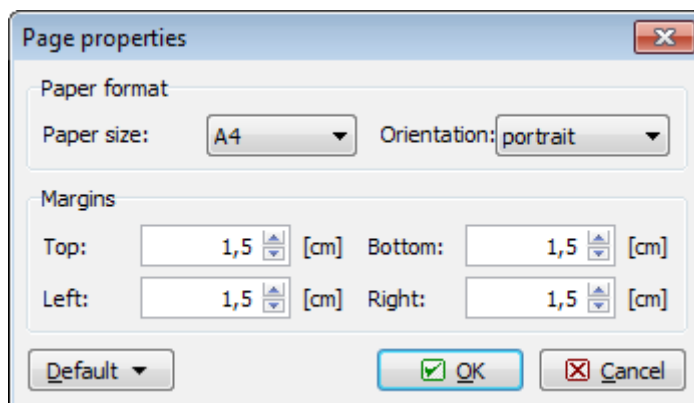


View of the document header and footer

## Page Properties

The dialog window allows to set the page layout (paper format, print orientation and edges).

The "Default", selection "Save settings as default" option sets the page properties as default for the newly created data. **The assumed default setting is common for all our programs.** Different computer users may use different settings. Selection "Adopt default settings" enables to adopt the default settings of the GEO5 programs to any opened task, which had the settings different.



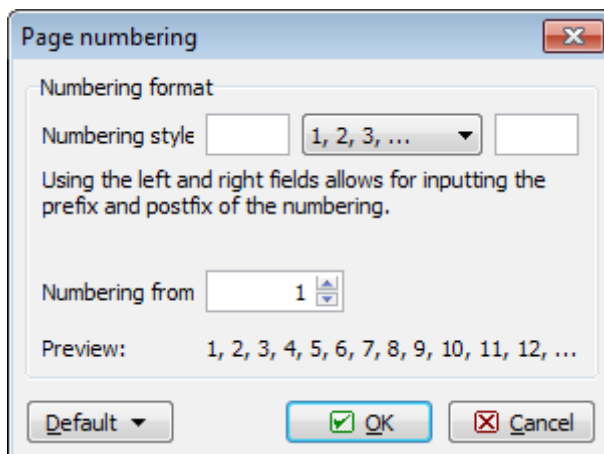
Dialog window "Page properties"

## Page Numbering

This dialog window allows to set page numbering. The combo list allows to define the numbering style (Arabic digits, Roman digits, using symbols). A constant text can be placed both in front and behind the page number. The "Numbering from" option allows to start the

page numbering from an arbitrary number.

The "**Default**", selection "**Save settings as default**" option sets the input page numbering properties as default for the newly created data. **The assumed default setting is common for all our programs.** Different computer users may use different settings. Selection "**Adopt default settings**" enables to adopt the default settings of the GEO5 programs to any opened task, which had the settings different.



*Dialog window "Page numbering"*

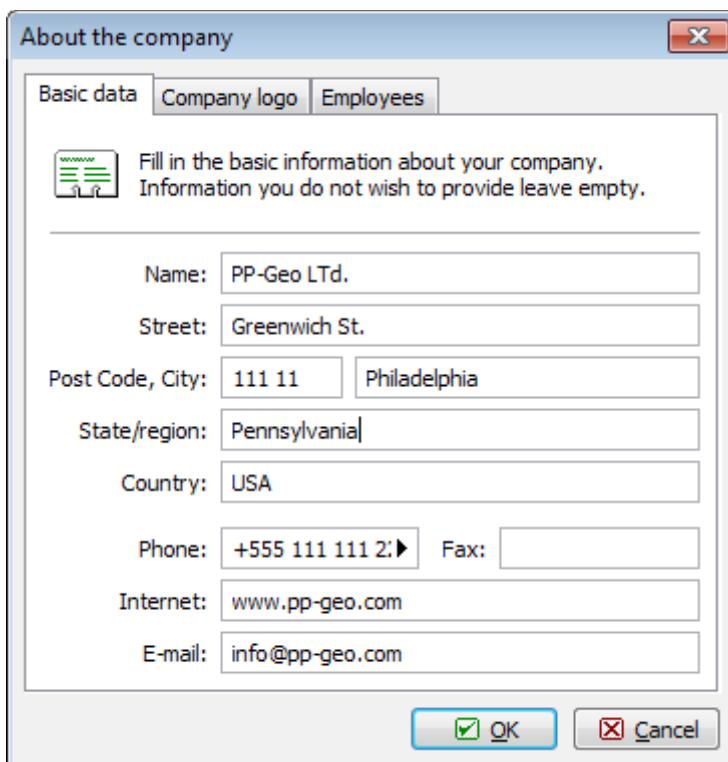
## About the Company

The dialog window is launched from the **managing menu** (items "**Settings**", "**Company**").

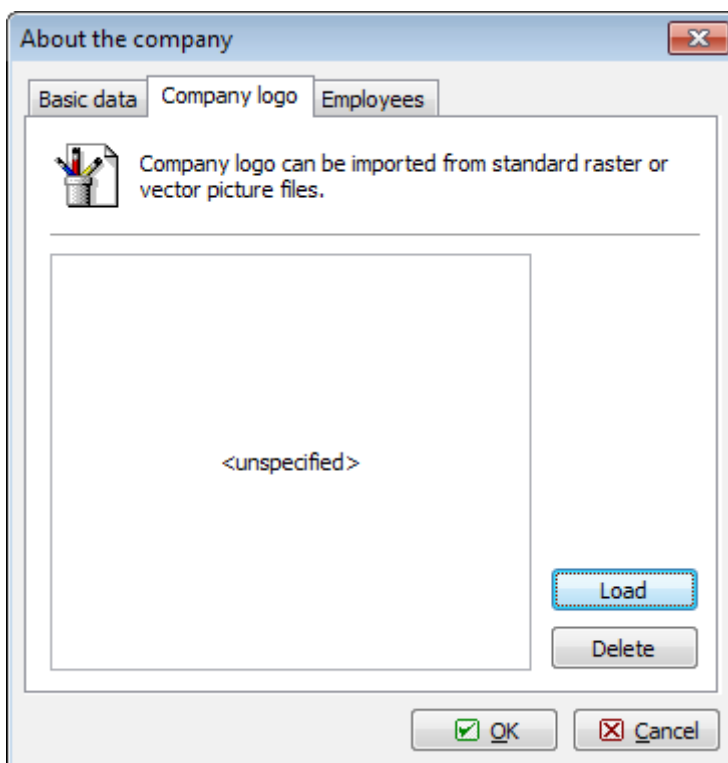
The "**Basic data**" tab allows to specify the basic information about the company. The input data is used by the program when printing and exporting documents (pictures), in the **document header and footer**.

The "**Company logo**" tab allows to load the company logo. The "**Load**" button opens a dialog window which allows to open the picture in various formats (\*.JPG, \*.JPEG, \*.JPE, \*.BMP, \*.ICO, \*.EMF, \*.WMF).

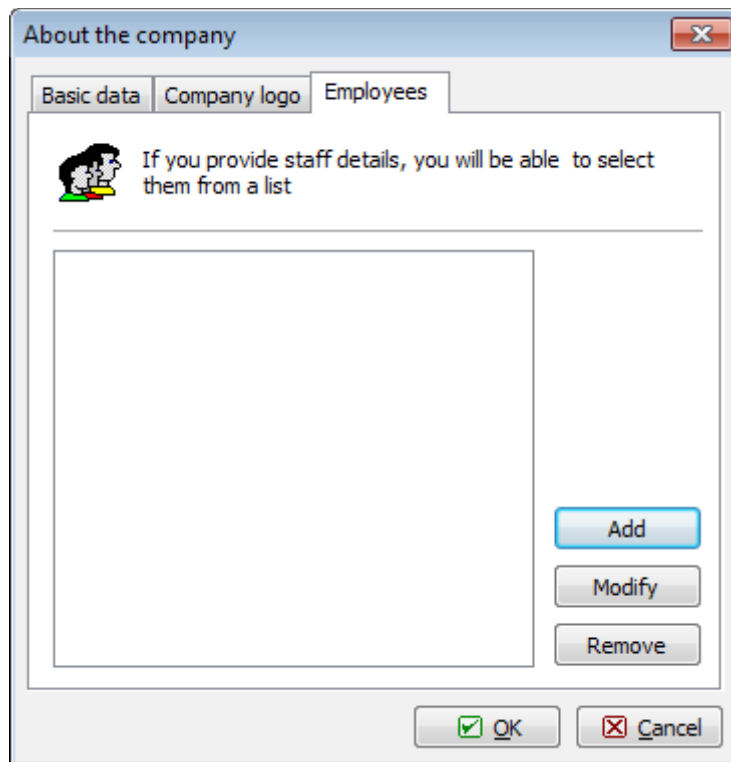
The "**Employees**" tab allows to input a list of program users (employees). When filling the name list it is no longer necessary fill the author's name in the frame "Project".



*Dialog window "About company" - tab "Basic data"*



*Dialog window "About company" - tab "Company logo"*



Dialog window "About company" - tab "Employees"

## Theory

The theoretical part of the help contains all theoretical basis employed in computations within GEO5 programs.

## Stress in Soil Body

Calculation of the stress in soil in our software is described in the following chapters:

- Geostatic stress in soil body, computation of uplift pressure
- Effective / total stress
- Stress increment due to surcharge
- Stress increment under footing

## Geostatic Stress, Uplift Pressure

Stress analysis is based on the existence of soil layers specified by the user during input. The program further inserts fictitious layers at the locations where the stress and lateral pressure (GWT, points of construction, etc.) change. The normal stress in the  $i^{th}$  layer is computed according to:

$$\sigma_i = \sum h_i \cdot \gamma_i$$

where:  $h_i$  - thickness of the  $i^{th}$  layer

$\gamma_i$  - unit weight of soil

If the layer is found below the **ground water table**, the unit weight of soil below the water table is specified with the help of input parameters of the soil as follows:

- for option "**Standard**" from expression:

$$\gamma_{su} = \gamma_{sat} - \gamma_w$$

where:  $\gamma_{sat}$  - saturated unit weight of soil

$\gamma_w$  - unit weight of water

- for option "**Compute from porosity**" from expression:

$$\gamma_{su} = (1 - n)(\gamma_s - \gamma_w)$$

where:  $n$  - porosity

$\gamma_s$  - specific weight of soil

$\gamma_w$  - unit weight of water

$$\gamma_s = \frac{G_d}{V - V_p}$$

where:  $V$  - volume of soil

$V_p$  - volume of voids

$G_d$  - weight of dry soil

Unit weight of water is assumed in the program equal to  $10 \text{ kN/m}^3$  or  $0.00625 \text{ ksi}$ .

Assuming inclined ground behind the structure ( $\beta \neq 0$ ) and layered subsoil the angle  $\beta$ , when computing the coefficient of earth pressure  $K$ , is reduced in the  $i^{\text{th}}$  layer using the following expression:

$$\text{tg}\beta_i = \frac{\gamma}{\gamma_i} \text{tg}\beta$$

where:  $\gamma$  - unit weight of the soil in the first layer under ground

$\gamma_i$  - unit weight of the soil in the  $i^{\text{th}}$  layer under ground

$\beta$  - slope inclination behind the structure

## Effective/Total Stress in a Soil

Vertical normal stress  $\sigma_z$  is defined as:

$$\sigma_z = \gamma_{ef} \cdot z + \gamma_w \cdot z$$

where:  $\sigma_z$  - vertical normal total stress

$\gamma_{ef}$  - submerged unit weight of soil

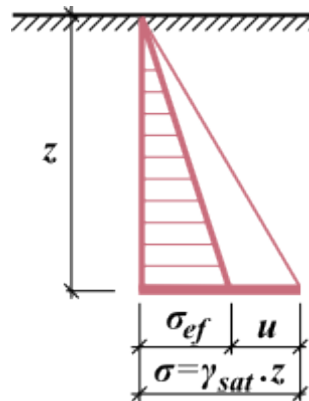
$z$  - depth below the ground surface

$\gamma_w$  - unit weight of water

This expression in its generalized form describes so called concept of effective stress:

$$\sigma_z = \sigma_{ef} + u$$

where:  $\sigma$  - total stress (overall)  
 $\sigma_{ef}$  - effective stress (active)  
 $u$  - neutral stress (pore water pressure)



*Total, effective and neutral stress in the soil*

Effective stress concept is valid only for normal stress  $\sigma$ , since the shear stress  $\tau$  is not transferred by the water so that it is effective. The total stress is determined using the basic tools of theoretical mechanics, the effective stress is then determined as a difference between the total stress and neutral (pore) pressure (i.e. always by calculation, it can never be measured). Pore pressures are determined using laboratory or in-situ testing or by calculation. To decide whether to use the total or effective stresses is no simple. The following table may provide some general recommendations valid for majority of cases. We should realize that the total stress depends on the way the soil is loaded by its self weight and external effects. As for the pore pressure we assume that for flowing pore water the pore equals to hydrodynamic pressure and to hydrostatic pressure otherwise. For partial saturated soils with higher degree of it is necessary to account for the fact that the pore pressure evolves both in water and air bubbles.

Assume conditions	Drained layer	Undrained layer
short - term	effective stress	total stress
long - term	effective stress	effective stress

In layered subsoil with different unit weight of soils in individual horizontal layers the vertical total stress is determined as a sum of weight of all layers above the investigated point and the pore pressure:

$$\sigma_z = \int_0^z \gamma dz + \gamma_w(z - d)$$

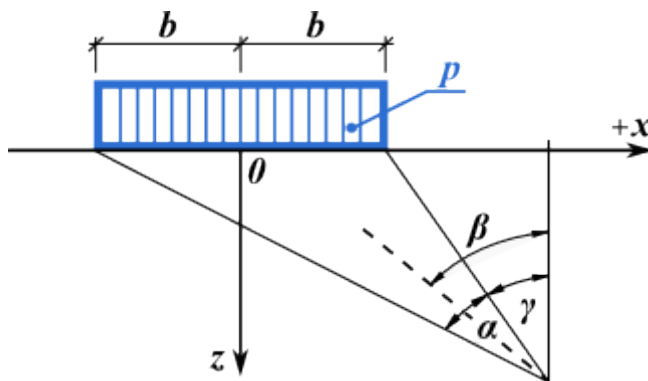
where:  $\sigma_z$  - vertical normal total stress

- $\gamma$  - unit weight of soil
  - unit weight of soil in natural state for soils above the GWT and dry layers
  - unit weight of soil below water in other cases
- $d$  - depth of the ground water table below the ground surface
- $z$  - depth below the ground surface
- $\gamma_w$  - unit weight of water

## Increment of Earth Pressure due to Surcharge

Earth pressure increment in a soil or rock body due to surcharge is computed using the theory of elastic subspace (Boussinesq).

Earth pressure increment in the point inside the soil or rock body due to an **infinite strip surcharge** is obtained from the following scheme:



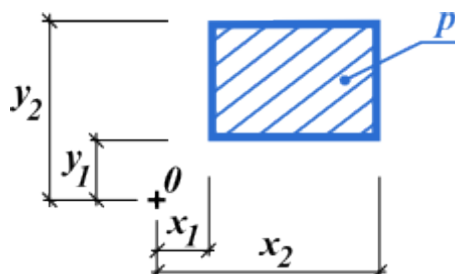
Computation of earth pressure due to infinite strip surcharge

$$\sigma_z = \frac{p}{\pi} (\alpha + \sin \alpha \cdot \cos 2\beta)$$

$$\beta = \gamma + \frac{\alpha}{2}$$

A **trapezoidal surcharge** is automatically subdivided in the program into ten segments. Individual segments are treated as strip surcharges. The resulting earth pressure is a sum of partial surcharges from individual segments.

Stress increment due to **concentrated surcharge** is computed as follows:



Surcharge related to point "O"



$$\Delta\sigma_z = \frac{p}{2\pi} \left( \frac{x_2 \cdot z \cdot S_2}{y_2 \cdot S_{2x}^2} + \frac{x_2 \cdot z^3}{y_2 \cdot S_{2y}^2 \cdot S_2} - \frac{x_2 \cdot z \cdot S_3}{y_1 \cdot S_{2x}^2} + \frac{x_2 \cdot z^3}{y_1 \cdot S_{1y}^2 \cdot S_3} - \frac{x_1 \cdot z \cdot S_4}{y_2 \cdot S_{1x}^2} + \frac{x_2 \cdot z^3}{y_2 \cdot S_{2y}^2 \cdot S_4} + \frac{x_1 \cdot z \cdot S_2}{y_1 \cdot S_{1x}^2} - \frac{x_1 \cdot z^3}{y_1 \cdot S_{2y}^2 \cdot S_1} + \operatorname{arctg} \frac{x_2 \cdot y_2}{z \cdot S_2} - \operatorname{arctg} \frac{x_2 \cdot y_1}{z \cdot S_3} - \operatorname{arctg} \frac{x_1 \cdot y_2}{z \cdot S_4} + \operatorname{arctg} \frac{x_1 \cdot y_1}{z \cdot S_1} \right)$$

where:

$$S_{2x} = \sqrt{x_2^2 + z^2}$$

$$S_1 = \sqrt{x_1^2 + y_1^2 + z^2}$$

$$S_3 = \sqrt{x_2^2 + y_1^2 + z^2}$$

$$S_{2y} = \sqrt{y_2^2 + z^2}$$

$$S_2 = \sqrt{x_2^2 + y_2^2 + z^2}$$

$$S_4 = \sqrt{x_1^2 + y_2^2 + z^2}$$

## Increment of Earth Pressure under Footing

In the program "Spread footing", the stress distribution below foundation is determined by combining the basic load diagrams:

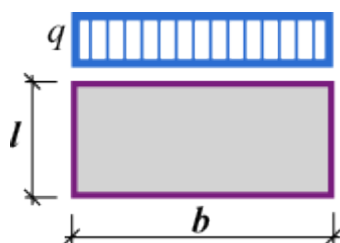


Chart of foundation load

$$\sigma_z = \frac{f}{2\pi} \left[ \operatorname{arctg} \frac{l \cdot b}{z \sqrt{l^2 + b^2 + z^2}} + \frac{l \cdot b \cdot z}{z \sqrt{l^2 + b^2 + z^2}} \left( \frac{1}{l^2 + z^2} + \frac{1}{b^2 + z^2} \right) \right]$$

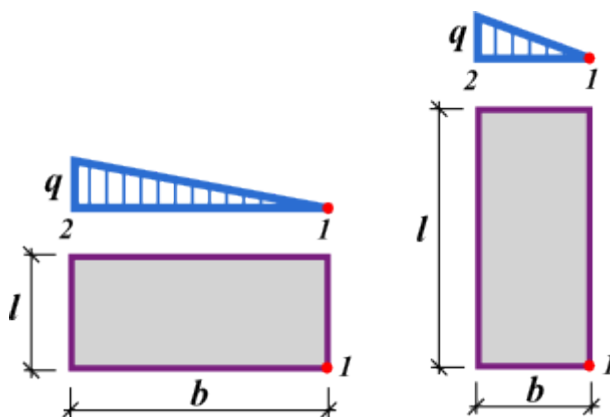


Chart of foundation load

$$\sigma_{z,1} = \frac{q}{2\pi} \left( \frac{l \cdot b \cdot z}{R(z^2 + b^2)} + \frac{l \cdot z \cdot R - \sqrt{l^2 + z^2}}{b \cdot R \sqrt{l^2 + z^2}} \right)$$

$$R = \sqrt{l^2 + b^2 + z^2}$$

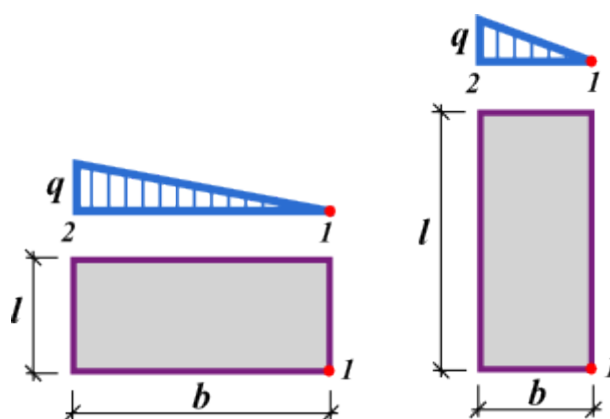


Chart of foundation load

$$\sigma_{z,z} = \frac{q}{2\pi} \left( \operatorname{arctg} \frac{l \cdot b}{z \cdot R} + \frac{l \cdot z}{l^2 + z^2} \frac{R - \sqrt{l^2 + z^2}}{b} \right)$$

## Earth Pressures

GEO5 software considers following earth pressure categories:

- active earth pressure
- passive earth pressure
- earth pressure at rest

When computing earth pressures, the program allows to distinguish between **effective and total stress** state and to establish several ways of **calculation of uplift pressure**. In addition, it is possible to account for the following effects having on the earth pressure magnitude:

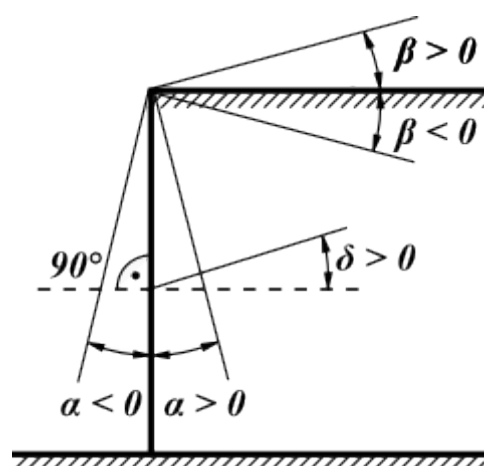
- influence of surcharge
- influence of water pressure
- influence of broken terrain
- friction between soil and back of structure
- adhesion of soil
- influence of earth wedge at cantilever jumps
- influence of earthquake

The following **sign convention** is used in the program, text and presented expressions.

When specifying rocks, it is also necessary to input both **cohesion of rock**  $c$  and the **angle of internal friction of rock**  $\varphi$ . These values can be obtained either from a geological survey or from the table of recommended values.

## Sign Convention

The following sign convention is used in the program, text and presented expressions.



Sign convention for calculation of earth pressures

- inclination of the ground surface  $\beta$  is positive, when the ground rises upwards from the wall
- inclination of the back of structure  $\alpha$  is positive, when the toe of the wall (at the back face) is placed in the direction of the soil body when measured from the vertical line constructed from the upper point of the structure
- friction between the soil and back of structure  $\delta$  is positive, if the resultant of earth pressure (thus also earth pressure) and normal to the back of structure form an angle measured in the clockwise direction

## Active Earth Pressure

Active earth pressure is the smallest limiting lateral pressure developed at the onset of shear failure by wall moving away from the soil in the direction of the acting earth pressure (minimal wall rotation necessary for the evolution of active earth pressure is about  $2 \text{ mrad}$ , i.e.  $2 \text{ mm/m}$  of the wall height).

The following theories and approaches are implemented for the computation of active earth pressure **assuming effective stress state**:

- The Mazindrani theory (Rankine)
- The Coulomb theory
- The Müller-Breslau theory
- The Caquot theory
- The Absi theory

For cohesive soils the tension cutoff condition is accepted, i.e. if due to cohesion the negative value of active earth pressure is developed or, according to more strict requirements, the value of "Minimum dimensioning pressure" is exceeded, the value of active earth pressure drops down to zero or set equal to the "Minimum dimensioning pressure".

The program also allows for running the analysis in **total stresses**.

## Active Earth Pressure - The Mazindrani Theory (Rankine)

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z \cdot K_a = \gamma \cdot z \cdot K'_a \cdot \cos \beta$$

where:

- $\sigma_z$  - vertical geostatic stress
- $K_a$  - coefficient of active earth pressure due to Rankine
- $\beta$  - slope inclination
- $\gamma$  - weight of soil
- $z$  - assumed depth
- $K'_a$  - coefficient of active earth pressure due to Mazindrani

$$K'_a = \frac{1}{\cos^2 \varphi} \left[ \frac{2 \cdot \cos^2 \beta + 2 \cdot \left(\frac{c}{\gamma \cdot z}\right) \cdot \cos \varphi \cdot \sin \varphi - \sqrt{4 \cdot \cos^2 \beta \cdot (\cos^2 \beta - \cos^2 \varphi) + 4 \cdot \left(\frac{c}{\gamma \cdot z}\right)^2 \cdot \cos^2 \varphi + 8 \cdot \left(\frac{c}{\gamma \cdot z}\right) \cdot \cos^2 \beta \cdot \sin \varphi \cdot \cos \varphi}}{2} \right] - 1$$

where:

- $\beta$  - slope inclination
- $\varphi$  - angle of internal friction of soil
- $c$  - cohesion of soil

Assuming cohesionless soils ( $c = 0$ ) and horizontal ground surface ( $\beta = 0$ ) yields the Rankine solution, for which the active earth pressure is provided by:

$$\sigma_a = \sigma_z \cdot K_a$$

and the coefficient of active earth pressure becomes:

$$K_a = \tan^2 \left( 45^\circ - \frac{\varphi}{2} \right)$$

where:  $\varphi$  - angle of internal friction of soil

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a \cdot \cos(\alpha + \delta)$$

$$\sigma_{az} = \sigma_a \cdot \sin(\alpha + \delta)$$

where:

- $\sigma_a$  - active earth pressure
- $\delta$  - angle of friction between structure and soil
- $\alpha$  - back face inclination of the structure

### Literature:

Mazindrani, Z.H., and Ganjali, M.H. 1997. Lateral earth pressure problem of cohesive backfill with inclined surface. *Journal of Geotechnical and Geoenvironmental Engineering, ASCE*, **123**(2): 110-112.

## Active Earth Pressure - The Coulomb Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z \cdot K_a - 2c_{ef} \cdot K_{ac}$$

where:

- $\sigma_z$  - vertical geostatic stress
- $c_{ef}$  - effective cohesion of soil
- $K_a$  - coefficient of active earth pressure
- $K_{ac}$  - coefficient of active earth pressure due to cohesion

The coefficient of active earth pressure  $K_a$  is given by:

$$K_a = \frac{\cos^2(\varphi - \alpha)}{\cos^2 \alpha \cdot \cos(\alpha + \delta) \left( 1 + \sqrt{\frac{\sin(\varphi + \delta) \cdot \sin(\varphi - \beta)}{\cos(\alpha + \delta) \cdot \cos(\alpha - \beta)}} \right)^2}$$

The coefficient of active earth pressure  $K_{ac}$  is given by:

for:  $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{\cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{\cos \varphi \cdot \cos \beta \cdot \cos(\delta - \alpha) \cdot (1 + \operatorname{tg}(-\alpha) \operatorname{tg} \beta)}{1 + \sin(\varphi + \delta - \alpha - \beta)}$$

for:  $\alpha \geq \pi/4$

$$K_{ac} = \sqrt{K_a}$$

where:

- $\varphi$  - angle of internal friction of soil
- $\delta$  - angle of friction between structure and soil
- $\beta$  - slope inclination
- $\alpha$  - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a \cdot \cos(\alpha + \delta)$$

$$\sigma_{az} = \sigma_a \cdot \sin(\alpha + \delta)$$

where:

- $\sigma_a$  - active earth pressure
- $\delta$  - angle of friction between structure and soil
- $\alpha$  - back face inclination of the structure

## Active Earth Pressure - The Müller-Breslau Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z \cdot K_a - 2 \cdot c_{ef} \cdot K_{ac}$$

where:

- $\sigma_z$  - vertical geostatic stress
- $c_{ef}$  - effective cohesion of soil
- $K_a$  - coefficient of active earth pressure
- $K_{ac}$  - coefficient of active earth pressure due to cohesion

The coefficient of active earth pressure  $K_a$  is given by:

$$K_a = \left[ \frac{\frac{\sin(\alpha + \pi/4 - \varphi)}{\sin(\alpha + \pi/4)}}{\sqrt{\sin(\alpha + \pi/4 + \delta) + \frac{\sqrt{\sin(\varphi + \delta) \cdot \sin(\varphi - \beta)}}{\sin(\alpha + \pi/4 - \beta)}}} \right]^2$$

where:

- $\varphi$  - angle of internal friction of soil
- $\delta$  - angle of friction between structure and soil
- $\beta$  - slope inclination
- $\alpha$  - back face inclination of the structure

The coefficient of active earth pressure  $K_{ac}$  is given by:

for:  $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{\cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{\cos \varphi \cdot \cos \beta \cdot \cos(\delta - \alpha) \cdot (1 + \operatorname{tg}(-\alpha) \cdot \operatorname{tg} \beta)}{1 + \sin(\varphi + \delta - \alpha - \beta)}$$

for:  $\alpha \geq \pi/4$

$$K_{ac} = \sqrt{K_a}$$

where:

- $\varphi$  - angle of internal friction of soil
- $\delta$  - angle of friction between structure and soil
- $\beta$  - slope inclination
- $\alpha$  - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a \cdot \cos(\alpha + \delta)$$

$$\sigma_{az} = \sigma_a \cdot \sin(\alpha + \delta)$$

where:  $\sigma_a$  - active earth pressure  
 $\delta$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure

*Literature:*

*Müller-Breslau's Erddruck auf Stützmauern, Stuttgart: Alfred Kroner-Verlag, 1906 (German).*

## Active Earth Pressure - The Caquot Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z \cdot K_a - 2 \cdot c_{ef} \cdot K_{ac}$$

where:  $\sigma_z$  - vertical geostatic stress  
 $c_{ef}$  - effective cohesion of soil  
 $K_a$  - coefficient of active earth pressure  
 $K_{ac}$  - coefficient of active earth pressure due to cohesion

The following analytical solution (Boussinesque, Caquot) is implemented to compute the coefficient of active earth pressure  $K_a$ :

$$K_a = \rho \cdot K_a^{Coulomb}$$

where:  $K_a$  - coefficient of active earth pressure due to Caquot  
 $K_a^{Coulomb}$  - coefficient of active earth pressure due to Coulomb  
 $\rho$  - conversion coefficient - see further

$$\rho = \left[ (1 - 0,9 \cdot \lambda^2 - 0,1 \cdot \lambda^4) (1 - 0,3 \cdot \lambda^3) \right]^{-n}$$

$$\lambda = \frac{\Delta + \beta - \Gamma}{4 \cdot \varphi - 2 \cdot \pi (\Delta + \beta - \Gamma)}$$

$$\Delta = 2 \cdot \tan^{-1} \left( \frac{|\cot \delta| - \sqrt{\cot^2 \delta - \cot^2 \varphi}}{1 + \operatorname{cosec} \varphi} \right)$$

$$\Gamma = \sin^{-1} \left( \frac{\sin \beta}{\sin \varphi} \right)$$

where:  $\beta$  - slope inclination behind the structure  
 $\varphi$  - angle of internal friction of soil  
 $\delta$  - angle of friction between structure and soil

The coefficient of active earth pressure  $K_{ac}$  is given by:

for:  $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{\cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{\cos \varphi \cdot \cos \beta \cdot \cos(\delta - \alpha) \cdot (1 + \operatorname{tg}(-\alpha) \cdot \operatorname{tg} \beta)}{1 + \sin(\varphi + \delta - \alpha - \beta)}$$

for:  $\alpha \geq \pi/4$

$$K_{ac} = \sqrt{K_a}$$

where:

- $\varphi$  - angle of internal friction of soil
- $\delta$  - angle of friction between structure and soil
- $\beta$  - slope inclination behind the structure
- $\alpha$  - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{ax} = \sigma_a \cdot \cos(\alpha + \delta)$$

$$\sigma_{az} = \sigma_a \cdot \sin(\alpha + \delta)$$

where:

- $\sigma_a$  - active earth pressure
- $\delta$  - angle of friction between structure and soil
- $\alpha$  - back face inclination of the structure

## Active Earth Pressure - The Absi Theory

Active earth pressure is given by the following formula:

$$\sigma_a = \sigma_z \cdot K_a - 2 \cdot c_{ef} \cdot K_{ac}$$

where:

- $\sigma_z$  - vertical geostatic stress
- $c_{ef}$  - effective cohesion of soil
- $K_a$  - coefficient of active earth pressure
- $K_{ac}$  - coefficient of active earth pressure due to cohesion

The program takes values of the coefficient of active earth pressure  $K_a$  from a database, built upon the values published in the book: Kérisel, Absi: Active and passive earth Pressure Tables, 3rd Ed. A.A. Balkema, 1990 ISBN 90 6191886 3.

The coefficient of active earth pressure  $K_{ac}$  is given by:

for:  $\alpha < \pi/4$

$$K_{ac} = \frac{K_{ahc}}{\cos(\delta + \alpha)}$$

$$K_{ahc} = \frac{\cos \varphi \cdot \cos \beta \cdot \cos(\delta - \alpha) \cdot (1 + \operatorname{tg}(-\alpha) \cdot \operatorname{tg} \beta)}{1 + \sin(\varphi + \delta - \alpha - \beta)}$$



for:  $\alpha \geq \pi/4$

$$K_{\alpha c} = \sqrt{K_a}$$

where:

- $\varphi$  - angle of internal friction of soil
- $\delta$  - angle of friction between structure and soil
- $\beta$  - slope inclination
- $\alpha$  - back face inclination of the structure

Horizontal and vertical components of the active earth pressure become:

$$\sigma_{\alpha c} = \sigma_a \cdot \cos(\alpha + \delta)$$

$$\sigma_{\alpha z} = \sigma_a \cdot \sin(\alpha + \delta)$$

where:

- $\sigma_a$  - active earth pressure
- $\delta$  - angle of friction between structure and soil
- $\alpha$  - back face inclination of the structure

*Literature:*

*Kérisel, Absi: Active and Passive Earth Pressure Tables, 3rd ed., Balkema, 1990 ISBN 90 6191886 3.*

## Active Earth Pressure - Total Stress

When determining the active earth pressure in cohesive fully saturated soils, in which case the consolidation is usually prevented (undrained conditions), the horizontal normal total stress  $\sigma_x$  receives the form:

$$\sigma_x = \sigma_z - K_{uc} \cdot c_u$$

where:

- $\sigma_x$  - horizontal total stress (normal)
- $\sigma_z$  - vertical normal total stress
- $K_{uc}$  - coefficient of earth pressure
- $c_u$  - total cohesion of soil

The coefficient of earth pressure  $K_{uc}$  is given by:

$$K_{uc} = 2 \cdot \sqrt{1 + \frac{a_u}{c_u}}$$

where:

- $K_{uc}$  - coefficient of earth pressure
- $c_u$  - total cohesion of soil
- $a_u$  - total adhesion of soil to the structure

## Passive Earth Pressure

Passive earth pressure is the highest limiting lateral pressure developed at the onset of shear

failure by wall moving (penetrating) in the direction opposite to the direction of acting earth pressure (minimal wall rotation necessary for the evolution of passive earth pressure is about  $10 \text{ mrad}$ , i.e.  $10 \text{ mm/m}$  of the wall height). In most expressions used to compute the passive earth pressure the sign convention is assumed such that the usual values of  $\delta$  corresponding to vertical direction of the friction resultant are negative. The program, however, assumes these values to be positive. A seldom variant with friction acting upwards is not considered in the program.

The following theories and approaches are implemented for the computation of passive earth pressure **assuming effective stress state**:

- The Rankine and Mazindrani theory
- The Coulomb theory
- The Caquot - Kérisel theory
- The Müller - Breslau theory
- The Absi theory
- The Sokolovski theory

The program also allows for running the analysis in **total stresses**.

## Passive Earth Pressure - The Rankine and Mazindrani Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z \cdot K_p = \gamma \cdot z \cdot K'_p \cdot \cos \beta$$

where:

- $\sigma_z$  - vertical geostatic stress
- $K_p$  - coefficient of passive earth pressure due to Rankine
- $\beta$  - slope inclination
- $\gamma$  - weight of soil
- $z$  - assumed depth
- $K'_p$  - coefficient of passive earth pressure due to Mazindrani

The coefficient of passive earth pressure  $K_p$  is given by:

$$K'_p = \frac{1}{\cos^2 \varphi} \cdot \left[ \frac{2 \cdot \cos^2 \beta + 2 \cdot \left( \frac{c}{\gamma \cdot z} \right) \cdot \cos \varphi \cdot \sin \varphi + \sqrt{4 \cdot \cos^2 \beta \cdot (\cos^2 \beta - \cos^2 \varphi) + 4 \cdot \left( \frac{c}{\gamma \cdot z} \right)^2 \cdot \cos^2 \varphi + 8 \cdot \left( \frac{c}{\gamma \cdot z} \right) \cdot \cos^2 \beta \cdot \sin \varphi \cdot \cos \varphi}}{2} \right] - 1$$

where:

- $\beta$  - slope inclination
- $\varphi$  - angle of internal friction of soil
- $c$  - cohesion of soil

If there is no friction ( $\delta = 0$ ) between the structure and cohesionless soils ( $c = 0$ ), the ground surface is horizontal ( $\beta = 0$ ) and the resulting slip surface is also plane with the slope:

$$\vartheta_p = 45^\circ - \frac{\varphi}{2}$$

The Mazindrani theory then reduces to the Rankine theory. The coefficient of passive earth pressure is then provided by:

$$K_p = \frac{1 + \sin \varphi}{1 - \sin \varphi} = \tan^2 \left( 45^\circ + \frac{\varphi}{2} \right)$$

where:  $\varphi$  - angle of internal friction of soil

Passive earth pressure  $\sigma_p$  by Rankine for cohesionless soils is given:

$$\sigma_p = \gamma \cdot z \cdot K_p$$

where:  $\gamma$  - unit weight of soil

$z$  - assumed depth

$K_p$  - coefficient of passive earth pressure due to Rankine

*Literature:*

Mazindrani, Z.H., and Ganjali, M.H. 1997. Lateral earth pressure problem of cohesive backfill with inclined surface. *Journal of Geotechnical and Geoenvironmental Engineering, ASCE*, **123**(2): 110-112.

## Passive Earth Pressure - The Coulomb Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z \cdot K_p + 2 \cdot c \cdot \sqrt{K_p}$$

where:  $\sigma_z$  - effective vertical geostatic stress

$K_p$  - coefficient of passive earth pressure due to Coulomb

$c$  - cohesion of soil

The coefficient of passive earth pressure  $K_p$  is given by:

$$K_p = \frac{\cos^2(\varphi + \alpha)}{\cos^2 \alpha \cdot \cos(\delta - \alpha) \cdot \left( 1 - \sqrt{\frac{\sin(\varphi + \delta) \cdot \sin(\varphi + \beta)}{\cos(\delta - \alpha) \cdot \cos(\beta - \alpha)}} \right)^2}$$

where:  $\varphi$  - angle of internal friction of soil

$\delta$  - angle of friction between structure and soil

$\beta$  - slope inclination

$\alpha$  - back face inclination of the structure

The vertical  $\sigma_{pv}$  and horizontal  $\sigma_{ph}$  components of passive earth pressure are given by:

$$\sigma_{px} = \sigma_p \cdot \cos(\alpha + \delta)$$

$$\sigma_{pz} = \sigma_p \cdot \sin(\alpha + \delta)$$

where:  $\delta$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure

## Passive Earth Pressure - The Caquot - Kérisel Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z \cdot K_p \cdot \psi + 2 \cdot c \cdot \sqrt{K_p \cdot \psi}$$

where  $K$  - coefficient of passive earth pressure for  $\delta = -\varphi$ , see the [table](#)

:  $p$

$\psi$  - reduction coefficient  $\psi$  for  $|\delta| < \varphi$ , see the [table](#)

$c$  - cohesion of soil

$\sigma$  - vertical geostatic stress

$z$

The vertical  $\sigma_{pv}$  and horizontal  $\sigma_{ph}$  components of passive earth pressure are given by:

$$\sigma_{px} = \sigma_p \cdot \cos(\alpha + \delta)$$

$$\sigma_{pz} = \sigma_p \cdot \sin(\alpha + \delta)$$

where:  $\delta$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure

## Coefficient of Passive Earth Pressure $K_p$

<b>Coefficient of passive earth pressure <math>K_p</math> for <math>\delta = -\varphi</math></b>											
$\alpha$ [°]	$\varphi$ [°]	$K_p$ when $\beta^\circ$									
		0	5	10	15	20	25	30	35	40	45
	10	1.17	1.41	1.53							
	15	1.30	1.70	1.92	2.08						
	20	1.71	2.08	2.42	2.71	2.92					
	25	2.14	2.81	2.98	3.88	4.22	4.43				
-30	30	2.78	3.42	4.18	5.01	5.98	8.94	7.40			

	35	3.7 5	4.7 3	5.8 7	7.2 1	8.78	10. 80	12. 50	13.80		
	40	5.3 1	8.8 7	8.7 7	11. 00	13.70	17. 20	24. 80	25.40	28.40	
	45	8.0 5	10. 70	14. 20	18. 40	23.80	90. 60	38. 90	49.10	60.70	69. 10
	10	1.3 6	1.5 8	1.7 0							
	15	1.6 8	1.9 7	2.2 0	2.3 8						
	20	2.1 3	2.5 2	2.9 2	3.2 2	3.51					
	25	2.7 8	3.3 4	3.9 9	4.8 0	5.29	5.5 7				
-20	30	3.7 8	4.8 1	8.5 8	8.8 1	7.84	9.1 2	9.7 7			
	35	5.3 8	8.8 9	8.2 8	10. 10	12.20	14. 80	17. 40	19.00		
	40	8.0 7	10. 40	12. 00	18. 50	20.00	25. 50	38. 50	37.80	42.20	
	45	13. 2	17. 50	22. 90	29. 80	38.30	48. 90	82. 30	78.80	97.30	11 1.0 4
	10	1.5 2	1.7 2	1.8 3							.
	15	1.9 5	2.2 3	2.5 7	2.8 8						
	20	2.5 7	2.9 8	3.4 2	3.7 5	4.09					
	25	3.5 0	4.1 4	4.9 0	5.8 2	8.45	8.8 1				
-10	30	4.9 8	8.0 1	7.1 9	8.5 1	10.10	11. 70	12. 80			
	35	7.4 7	9.2 4	11. 30	13. 80	18.70	20. 10	23. 70	26.00		
	40	12. 0	15. 40	19. 40	24. 10	29.80	37. 10	53. 20	55.10	61.80	

	45	21.2	27.90	38.50	47.20	80.80	77.30	908.20	124.00	153.00	178.00
	10	1.84	1.81	1.93							
	15	2.19	2.46	2.73	2.91						
	20	3.01	3.44	3.91	4.42	4.66					
	25	4.28	5.02	5.81	8.72	7.71	8.16				
0	30	8.42	7.69	9.19	10.80	12.70	14.80	15.90			
	35	10.2	12.60	15.30	18.80	22.30	28.90	31.70	34.90		
	40	17.5	22.30	28.00	34.80	42.90	53.30	78.40	79.10	88.70	
	45	33.5	44.10	57.40	74.10	94.70	120.00	153.00	174.00	240.00	275.00
	10	1.73	1.87	1.98							
	15	2.40	2.65	2.93	3.12						
	20	3.45	3.90	4.40	4.96	5.23					
10	25	5.17	5.99	6.90	7.95	9.11	9.67				
	30	8.17	9.69	11.40	13.50	15.90	18.50	19.90			
	35	13.8	16.90	20.50	24.80	29.80	35.80	42.30	46.60		
	40	25.5	32.20	40.40	49.90	61.70	76.40	110.00	113.00	127.00	
	45	52.9	69.40	90.90	116.00	148.00	188.00	239.00	303.00	375.00	431.00

	10	1.78	1.891	2.01						
	15	2.58	2.821	3.11	3.30					
	20	3.90	4.38	4.92	5.53	5.83				
20	25	6.18	7.12	8.17	9.39	10.70	11.40			
	30	10.4	12.30	14.40	16.90	20.00	23.20	25.00		
	35	18.7	22.80	27.60	33.30	40.00	48.00	56.80	62.50	
	40	37.2	46.90	58.60	72.50	89.30	111.00	158.00	164.00	185.00
	45	84.0	110.00	143.00	184.00	234.00	297.00	378.00	478.00	592.00 680.00

## Reduction Coefficient of Passive Earth Pressure

Reduction coefficient  $\psi$  for  $|\delta| < \varphi$

$\varphi$ [°]	$\psi$ for $ \delta  < \varphi$					
5	1.0	0.8	0.6	0.4	0.2	0.0
10	1.00	0.999	0.962	0.929	0.898	0.864
15	1.00	0.979	0.934	0.881	0.830	0.775
20	1.00	0.968	0.901	0.824	0.752	0.678
25	1.00	0.954	0.860	0.759	0.666	0.574
30	1.00	0.937	0.811	0.686	0.574	0.467
35	1.00	0.916	0.752	0.603	0.475	0.362
40	1.00	0.886	0.682	0.512	0.375	0.262
45	1.00	0.848	0.600	0.414	0.276	0.174

## Passive Earth Pressure - The Müller - Breslau Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z \cdot K_p + 2 \cdot c \cdot \sqrt{K_p}$$

where:  $K_p$  - coefficient of passive earth pressure  
 $p$   
 $c$  - cohesion of soil  
 $\sigma_z$  - vertical normal total stress

The coefficient of passive earth pressure  $K_p$  is given by:

$$K_p = \frac{\cos^2(\varphi - \alpha)}{\cos^2 \alpha \cdot \cos(\delta - \alpha) \left( 1 + \sqrt{\frac{\sin(\varphi - \delta) \cdot \sin(\varphi + \beta)}{\cos(\alpha - \delta) \cdot \cos(\alpha + \beta)}} \right)^2}$$

where:  $\varphi$  - angle of internal friction of soil  
 $\delta$  - angle of friction between structure and soil  
 $\beta$  - slope inclination  
 $\alpha$  - back face inclination of the structure

The vertical  $\sigma_{pv}$  and horizontal  $\sigma_{ph}$  components of passive earth pressure are given by:

$$\begin{aligned}\sigma_{px} &= \sigma_p \cdot \cos(\alpha + \delta) \\ \sigma_{pz} &= \sigma_p \cdot \sin(\alpha + \delta)\end{aligned}$$

where:  $\delta$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure

#### Literature:

Müller-Breslau's *Erddruck auf Stützmauern*, Stuttgart: Alfred Kroner-Verlag, 1906 (German).

## Passive Earth Pressure - The Absi Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z \cdot K_p + 2 \cdot c \cdot \sqrt{K_p}$$

where:  $K_p$  - coefficient of passive earth pressure  
 $c$  - cohesion of soil  
 $\sigma_z$  - vertical normal total stress

The program takes values of the coefficient  $K_p$  from a database, built upon the tabulated values published in the book: Kérisel, Absi: Active and passive earth Pressure Tables, 3rd Ed. A.A. Balkema, 1990 ISBN 90 6191886 3.

The vertical  $\sigma_{pv}$  and horizontal  $\sigma_{ph}$  components of passive earth pressure are given by:

$$\sigma_{px} = \sigma_p \cdot \cos(\alpha + \delta)$$



$$\sigma_{pz} = \sigma_p \cdot \sin(\alpha + \delta)$$

where:  $\delta$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure

Literature:

Kérisel, Absi: *Active and Passive Earth Pressure Tables, 3rd ed., Balkema, 1990 ISBN 90 6191886 3.*

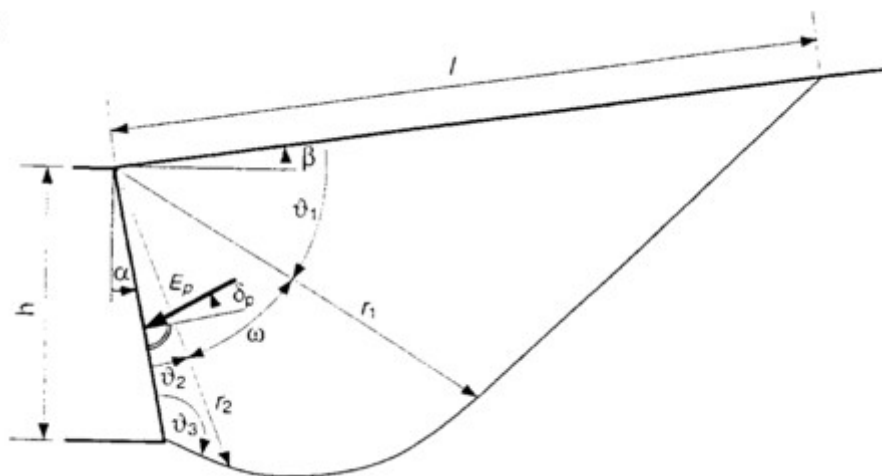
## Passive Earth Pressure - The Sokolovski Theory

Passive earth pressure follows from the following formula:

$$\sigma_p = \sigma_z \cdot K_{pg} + c \cdot K_{pc} + p_v \cdot K_{pp}$$

where:  $K_{pg}$  - passive earth pressure coefficient for cohesionless soils  
 $K_{pc}$  - passive earth pressure coefficient due to cohesion  
 $K_{pp}$  - passive earth pressure coefficient due to surcharge  
 $\sigma_z$  - vertical normal total stress

Individual expressions for determining the magnitude of passive earth pressure and slip surface are introduced in the sequel; the meaning of individual variables is evident from Fig.:



Passive earth pressure slip surface after Sokolovski

Angles describing the slip surface:

$$\vartheta_1 = \frac{\pi}{4} - \frac{\varphi}{2} - \frac{\varepsilon_1 - \beta}{2}$$

$$\vartheta_2 = \frac{\pi}{4} + \frac{\varphi}{2} - \frac{\varepsilon_2 - \delta_p}{2}$$

$$\vartheta_3 = \frac{\pi}{4} + \frac{\varphi}{2} + \frac{\varepsilon_2 - \delta_p}{2}$$

$$\omega = \frac{\pi}{2} - \alpha + \beta - \vartheta_1 - \vartheta_2$$

$$\varepsilon_1 = \frac{\sin \beta}{\sin \varphi}$$

$$\varepsilon_2 = -\frac{\sin \delta_p}{\sin \varphi}$$

where:  $\varphi$  - angle of internal friction of soil  
 $\delta_p$  - angle of friction between structure and soil  
 $\beta$  - slope inclination

Slip surface radius vector:

$$r_2 = \frac{h}{\cos \alpha} \cdot \frac{\sin \vartheta_3}{\sin(\vartheta_2 + \vartheta_3)}$$

$$r_1 = r_2 \cdot e^{\omega \cdot \tan \varphi}$$

$$I = r_1 \cdot \frac{\cos \varphi}{\cos(\varphi + \vartheta_1)}$$

Provided that  $\omega < 0$  the both straight edges of the zone  $r_1$  and  $r_2$  numerically overlap and resulting in the plane slip surface developed in the overlapping region. The coefficients of passive earth pressure  $K_{pg}$ ,  $K_{pp}$ ,  $K_{pc}$  then follow from:

$$K_{pg} = K_{pg,0} \cdot i_{pg} \cdot g_{pg} \cdot t_{pg}$$

$$K_{pp} = K_{pp,0} \cdot i_{pp} \cdot g_{pp} \cdot t_{pp}$$

$$K_{pc} = \cot \varphi \cdot \left( K_{pp,0} \cdot i_{pc} \cdot g_{pc} \cdot t_{pc} - \frac{1}{\cos \alpha \cdot \cos \delta} \right)$$

where:  $\varphi$  - angle of internal friction of soil  
 $\delta_p$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure

$$K_{pg,0} = K_{pp,0} = \frac{1 + \sin \varphi}{1 - \sin \varphi}$$

Auxiliary variables:  $i_{pg}$ ,  $i_{pp}$ ,  $i_{pc}$ ,  $g_{pg}$ ,  $g_{pp}$ ,  $g_{pc}$ ,  $t_{pg}$ ,  $t_{pp}$ ,  $t_{pc}$

$$i_{pg} = (1 - 0,53 \cdot \delta_p)^{0,26 + 5,96 \varphi}, \quad i_{pp} = (1 - 1,33 \cdot \delta_p)^{0,08 + 2,37 \varphi}, \quad i_{pc} = i_{pp}$$

f  
o  
r  
:

$$\beta \leq 0 \quad g_{pg} = (1 + 0,73 \cdot \beta)^{2,89}, \quad g_{pp} = (1 + 1,16 \cdot \beta_p)^{1,57},$$

$$g_{pc} = (1 + 0,001 \cdot \beta \cdot \tan \varphi)^{205,4 + 2232 \varphi}$$

$$\beta > 0 \quad g_{pg} = (1 + 0,35 \cdot \beta)^{0,42 + 8,15 \varphi}, \quad g_{pp} = (1 + 3,84 \cdot \beta_p)^{0,98 \varphi}, \quad g_{pc} = e^{2 \cdot \beta \cdot \tan \varphi}$$

$$\alpha \leq 0 \quad t_{pg} = (1 + 0,72 \cdot \alpha \cdot \tan \varphi)^{3,51 + 1,03 \varphi}$$

$$\alpha > 0 \quad t_{pg} = (1 - 0,0012 \cdot \alpha \cdot \tan \varphi)^{2910 - 1958 \varphi}$$

$$t_{pp} = \frac{e^{-2 \cdot \alpha \cdot \tan \varphi}}{\cos \alpha}$$

where:

$$t_{pc} = t_{pp}$$

For soils with zero value for the angle of internal friction the following expressions are employed to determine the coefficients of passive earth pressure:

$$K_{pp} = \cos \beta$$

$$K_{pc} = K_{pc,0} \cdot i_{pc} \cdot g_{pc} \cdot t_{pc}$$

where:

$$K_{pc,0} = 2$$

$$i_{pc} = 1$$

$$g_{pc} = 1 + \beta$$

$$t_{pc} = \frac{1 - \alpha}{\cos \alpha} \Rightarrow K_{pc} = \frac{2 \cdot (1 + \beta) \cdot (1 - \alpha)}{\cos \alpha}$$

Literature:

Sokolovski, V.V., 1960. *Statics of Soil Media*, Butterworth, London.

## Passive Earth Pressure - Total Stress

When determining the passive earth pressure in cohesive fully saturated soils, in which case the consolidation is usually prevented (undrained conditions), the horizontal normal total stress  $\sigma_x$  receives the form:

$$\sigma_x = \sigma_z - K_{uc} \cdot c_u$$

where:

$\sigma_x$	-	horizontal total stress (normal)
$\sigma_z$	-	vertical normal total stress
$K_{uc}$	-	coefficient of earth pressure
$c_u$	-	total cohesion of soil

The coefficient of earth pressure  $K_{uc}$  is given by:

$$K_{uc} = -2 \cdot \sqrt{1 + \frac{a_u}{c_u}}$$

where:  $K_{uc}$  - coefficient of earth pressure

- $c_u$  - total cohesion of soil  
 $a_u$  - total **adhesion of soil** to the structure

## Earth Pressure at Rest

Earth pressure at rest is the horizontal pressure acting on the rigid structure. It is usually assumed in cases, when it is necessary to minimize the lateral and horizontal deformation of the sheeted soil (e.g. when laterally supporting a structure in the excavation pit up to depth below the current foundation or in general when casing soil with structures sensitive to non-uniform settlement), or when structures loaded by earth pressures are due to some technological reasons extremely rigid and do not allow for deformation in the direction of load necessary to mobilize the active earth pressure.

Earth pressure at rest is given by:

$$\sigma_r = \sigma_z \cdot K_r$$

For **cohesive soils** the Terzaghi formula for computing  $K_r$  is implemented in the program:

$$K_r = \frac{\nu}{1 - \nu}$$

where:  $\nu$  - Poisson's ratio

For **cohesionless soils** the Jáky expression is used:

$$K_r = 1 - \sin \varphi$$

where:  $\varphi$  - angle of internal friction of soil

When computing the pressure at rest for cohesive soils  $\sigma_r$  using the Jáky formula for the determination of coefficient of earth pressure at rest  $K_r$ , it is recommended to use the **alternate angle of internal friction**  $\varphi_n$ .

The way of computing the earth pressure at rest can be therefore influenced by the selection of the type of soil (cohesive, cohesionless) when inputting its parameters. Even typically cohesionless soil (sand, gravel) must be introduced as cohesive if we wish to compute the pressure at rest with the help of the Poisson ratio and vice versa.

For **overconsolidated soils** the expression proposed by Schmertmann to compute the coefficient of earth pressure at rest  $K_r$  is used:

$$K_r = 0,5 \cdot (OCR)^{0,5}$$

where:  $K_r$  - coefficient of earth pressure at rest

$OCR$  - **overconsolidation ratio**

The value of the coefficient of earth pressure at rest can be **input also directly**.

The program calculates the influence of the **inclined ground surface or the back of structure** and increase of pressure at rest from the **surcharge**.

## Earth Pressure at Rest for an Inclined Ground Surface or Inclined Back of the Structure

For inclined ground surface behind the structure ( $0^\circ < \beta \leq \varphi$ ) the earth pressure at rest assumes the form:

$$\sigma_r = \frac{\sigma_z \cdot K_r \cdot \sin \varphi \cdot \cos \beta}{\sin \varphi - \sin^2 \beta}$$

where:

- $\varphi$  - angle of internal friction of soil
- $\beta$  - slope inclination
- $\sigma_z$  - vertical geostatic stress
- $K_r$  - coefficient of earth pressure at rest

For inclined back of wall the values of earth pressure at rest are derived from:

$$\sigma_r = \sigma_z \sqrt{\sin^2 \alpha + K_r^2 \cdot \cos^2 \alpha}$$

where:

- $\alpha$  - back face inclination of the structure
- $\sigma_z$  - vertical geostatic stress
- $K_r$  - coefficient of earth pressure at rest

Normal and tangential components are given by:

$$\begin{aligned} \sigma &= \sigma_z \cdot (\sin^2 \alpha + K_r \cdot \cos^2 \alpha) \\ \tau &= \sigma_z \cdot (1 - K_r) \cdot \sin \alpha \cdot \cos \alpha \end{aligned}$$

where:

- $\alpha$  - back face inclination of the structure
- $\sigma_z$  - vertical geostatic stress
- $K_r$  - coefficient of earth pressure at rest

The deviation angle from the normal line to the wall  $\delta$  reads:

$$\operatorname{tg} \delta = \frac{(1 - K_r) \cdot \operatorname{tg} \alpha}{K_r + \operatorname{tg}^2 \alpha}$$

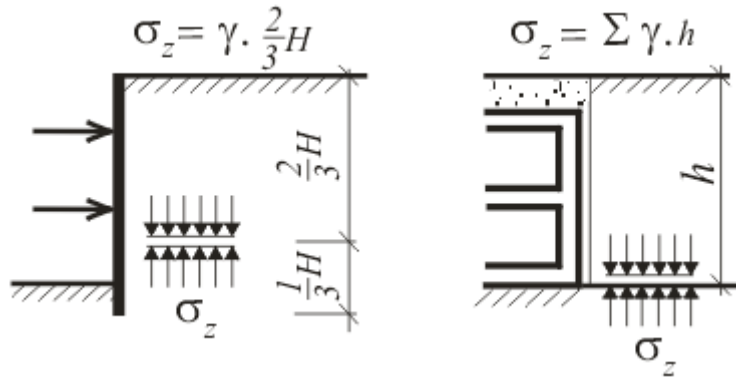
where:

- $\alpha$  - back face inclination of the structure
- $K_r$  - coefficient of earth pressure at rest

## Alternate Angle of Internal Friction of Soil

In some cases it appears more suitable for the analysis of earth pressures to introduce for cohesive soils an alternate angle of internal friction  $\varphi_n$  that also accounts for the influence of cohesive soil in conjunction with the normal stress developed in the soil. The magnitude of the normal stress for determining the value of alternate angle of internal friction depends on the type of geotechnical problem, foundation conditions, etc. For deep seated foundation pits or constructions in homogeneous or relatively simple environment the normal stress is introduced in the centroid of the load mass. In case of shallow pits or complex environment the normal

stress is assumed in the heel of the loading diagram - see figure:



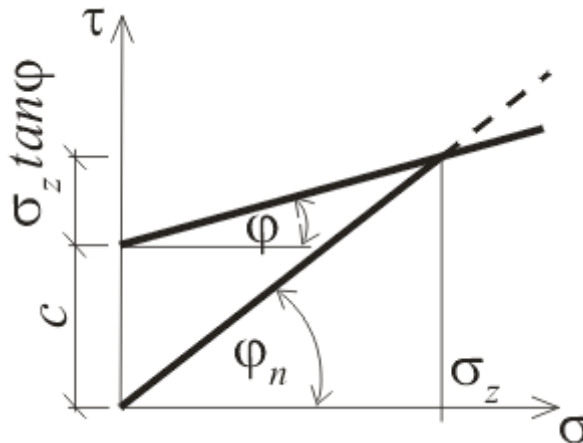
*Determination of normal stress for alternate angle of internal friction of soil  $\varphi_n$*

The alternate angle of internal friction of soil is given by:

$$\operatorname{tg} \varphi_n = \frac{c + \sigma_z \cdot \operatorname{tg} \varphi}{\sigma_z}$$

- where:
- $\sigma_z$  - vertical geostatic stress
  - $\varphi$  - angle of internal friction of soil
  - $c$  - cohesion of soil

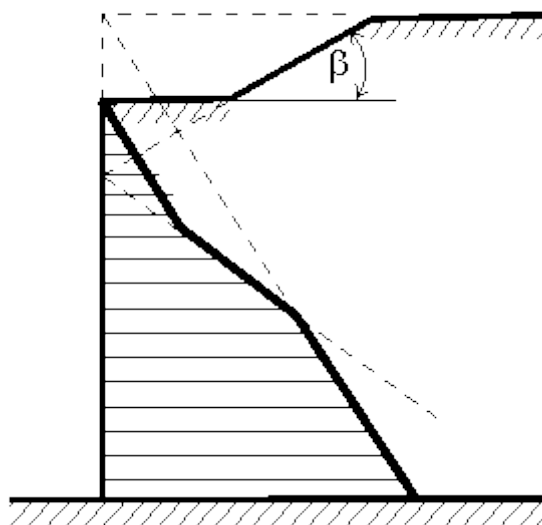
When computing the pressure at rest for cohesive soils  $\sigma_r$  using the Jáký formula for the determination of coefficient of earth pressure at rest  $K_0$ , it is recommended to use the alternate angle of internal friction  $\varphi_n$ .



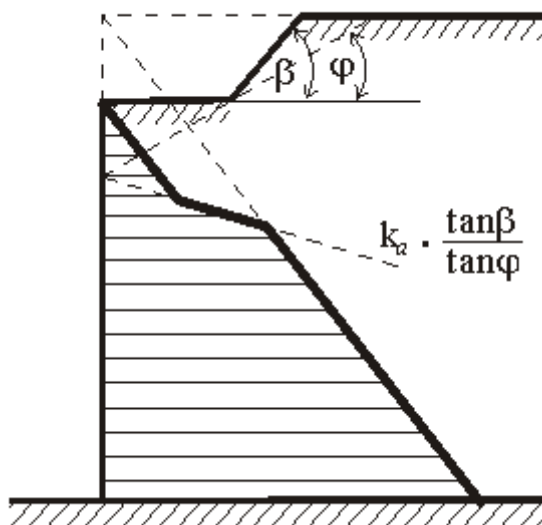
*Determination of alternate angle of internal friction of cohesive soil*

## Distribution of Earth Pressures in case of Broken Terrain

Figures show the procedure of earth pressure analysis in the case of sloping terrain. The resulting shape of earth pressure distribution acting on the construction is obtained from the sum of triangular distributions developed by individual effects acting on the construction.



Principle of the earth pressure computation in the case of broken terrain



Principle of the earth pressure computation in the case of broken terrain for  $\beta > \varphi$

## Influence of Water

The influence of ground water can be reflected using one of the following variants:

Without ground water, water is not considered



Hydrostatic pressure, ground water behind structure



Hydrostatic pressure, ground water behind and in front of structure



Hydrodynamic pressure



Special distribution of water pressure



## Without Ground Water, Water is not Considered



**Without ground water, water is not considered**

In this option the influence of ground water is not considered.

Complementary information:

If there are fine soils at and below the level of GWT, one should carefully assess an influence of full saturation in the region of capillary attraction. The capillary attraction is in the analysis reflected only by increased degree of saturation, and therefore the value of  $\gamma_{sat}$  is inserted into parameters of soils.

To distinguish regions with different degree of saturation, one may insert several layers of the same soil with different unit weights. Negative pore pressures are not considered. However, for layers with different degree of saturation it is possible to use different values of shear resistance influenced by suction (difference in pore pressure of water and gas ( $u_a - u_w$ )).

## Hydrostatic Pressure, Ground Water behind the Structure



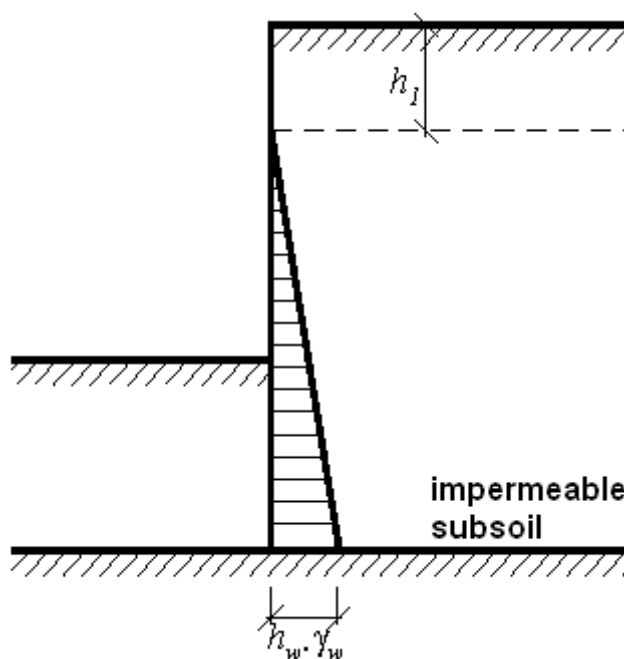
**Hydrostatic pressure, ground water behind structure**

The heel of a structure is sunk into impermeable subsoil so that the water flow below the structure is prevented. Water is found behind the back of structure only. There is no water acting on the front face. Such a case may occur when water in front of structure flow freely due to gravity or deep drainage is used. The back of structure is loaded by the hydrostatic pressure:

$$u = \gamma_w \cdot h_w$$

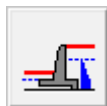
where:  $\gamma_w$  - unit weight of water  
 $h_w$  - water tables difference





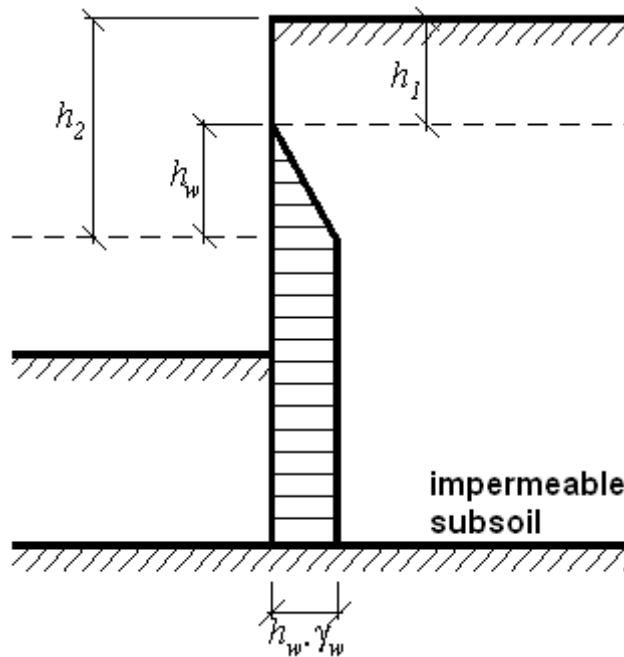
*Action of hydrostatic pressure*

## Hydrostatic Pressure, Ground Water behind and in front of the Structure



### Hydrostatic pressure, ground water behind and in front of structure

The heel of a structure is sunk into impermeable subsoil so that the water flow below the structure is prevented. The load due water is assumed both in front of and behind the structure. The water in front of structure is removed either with the help of gravity effects or is shallowly lowered by pumping. Both the face and back of structure is loaded by hydrostatic pressure due to difference in water tables ( $h_1$  and  $h_2$ ). The dimension  $h_w$  represents the difference in water tables at the back and in front of structure - see figure:



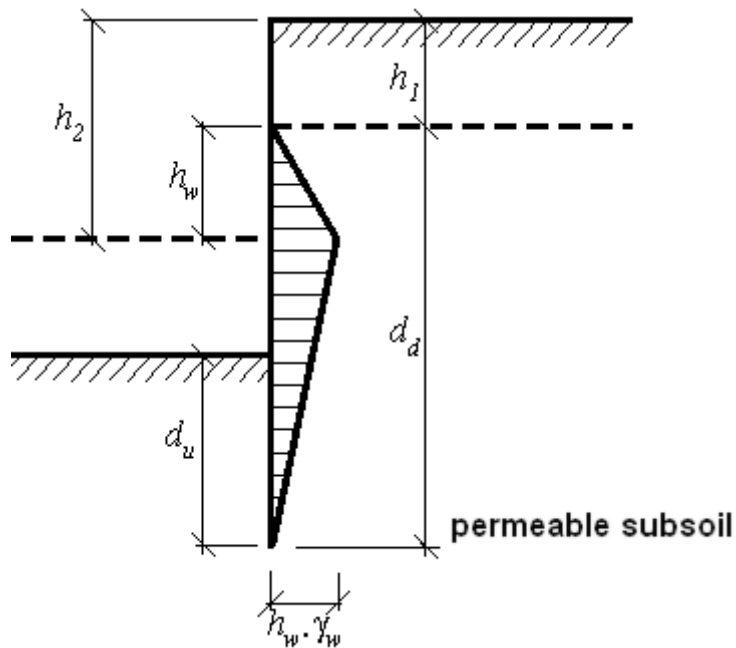
Action of hydrostatic pressure

## Hydrodynamic Pressure



### Hydrodynamic pressure

The heel of a structure is sunk into permeable subsoil, which allows free water flow below the structure - see figure. The unit weight of soil lifted by uplift pressure  $\gamma_{su}$  is modified to account for flow pressure. These modifications then depend on the direction of water flow.



### *Action of hydrodynamic pressure*

When computing the earth pressure in the area of descending flow the program introduces the following value of the unit weight of soil:

$$\gamma = \gamma_{su} + \Delta\gamma = \gamma_{su} + i \cdot \gamma_w$$

and in the area of ascending flow the following value:

$$\gamma = \gamma_{su} - \Delta\gamma = \gamma_{su} - i \cdot \gamma_w$$

where:

- $\gamma_{su}$  - unit weight of submerged soil
- $\Delta\gamma$  - alteration of unit weight of soil
- $i$  - an average seepage gradient
- $\gamma_w$  - unit weight of water

An average hydraulic slope is given:

$$i = \frac{h_w}{d_d + d_u}$$

where:

- $i$  - an average seepage gradient
- $h_w$  - water tables difference
- $d_d$  - seepage path downwards
- $d_u$  - seepage path upwards

If the change of unit weight of soil  $\Delta\gamma$  provided by:

$$\Delta\gamma = i \cdot \gamma_w$$

where:

- $i$  - an average seepage gradient
- $\gamma_w$  - unit weight of water

Is greater than the unit weight of saturated soil  $\gamma_{su}$ , then the leaching appears in front of structure - as a consequence of water flow the soil behaves as weightless and thus cannot transmit any load. The program then prompts a warning message and further assumes the value of  $\gamma = 0$ . The result therefore no longer corresponds to the original input - is safer.

## Special Distribution of Water Pressure

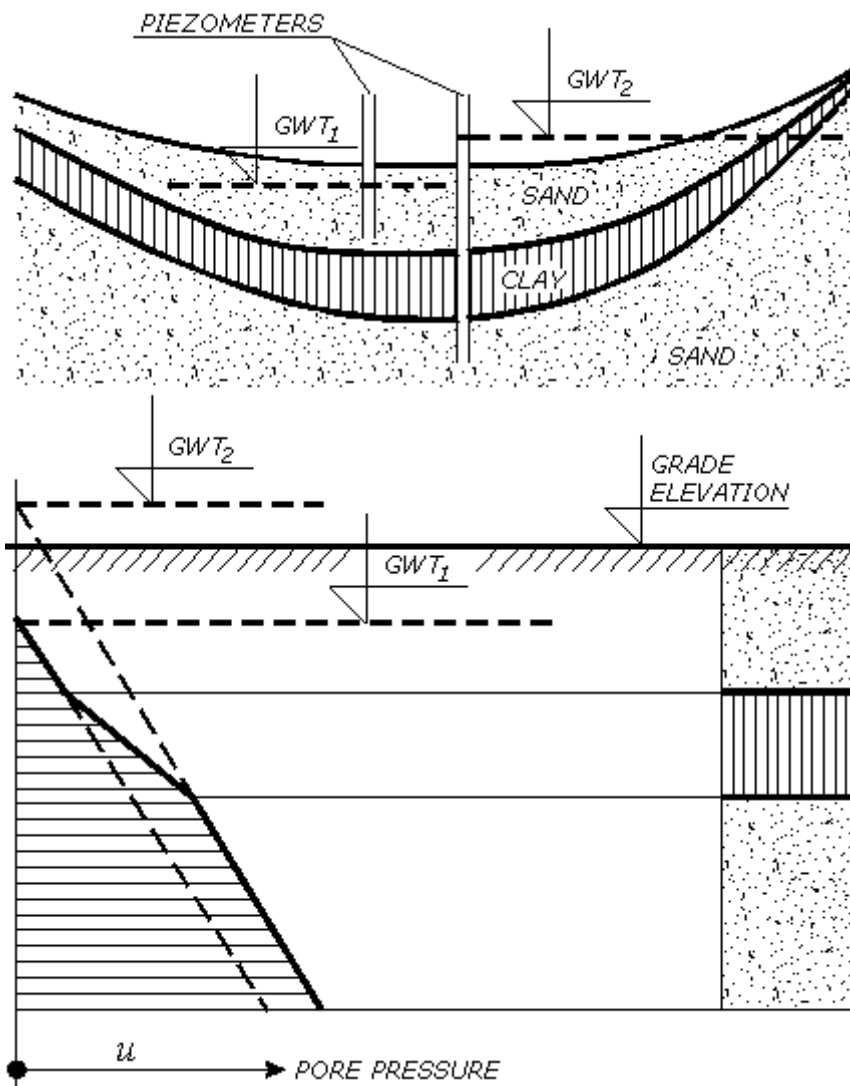


### Special distribution of water pressure

This option allows an independent (manual) input of distribution of load due to water at the back and in front of structure using ordinates of pore pressure at different depths. The variation of pressure between individual values is linear. At the same time it is necessary to input levels of tables of full saturation of a soil at the back  $h_1$  and in front  $h_2$  of structure including possible decrease of unit weight  $\delta\gamma$  in front of structure due to water flow.

**Example:** Two separated horizon lines of ground water.

There are two permeable layers (sand or gravel) with one impermeable layer of clay in between, which causes separation of two hydraulic horizon lines - see figure:



Example of pore pressure distribution

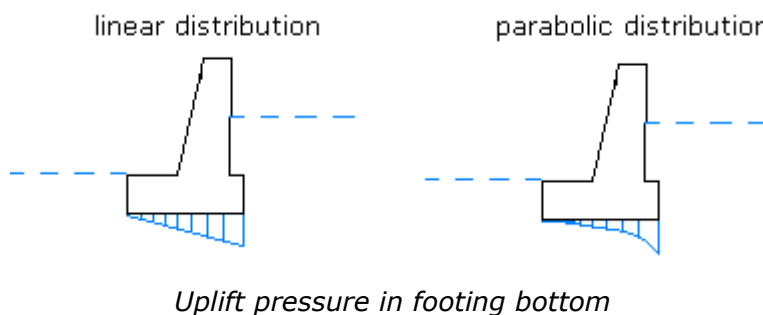
The variation of pore pressure above the clay layer is driven by free ground water table  $GWT_1$ . The distribution of pore pressure below the clay layer results from ratio in the lower separated ground water table  $GWT_2$ , where the ground water is stressed. The pore pressure distribution in clay is approximately linear.

The capillary attraction is in the analysis reflected only by increased degree of saturation, and therefore the value of  $\gamma_{sat}$  is inserted into parameters of soils.

To distinguish regions with different degree of saturation, one may insert several layers of the same soil with different unit weights. Negative pore pressures are not considered. However, for layers with different degree of saturation it is possible to introduce values of shear resistance influenced by suction.

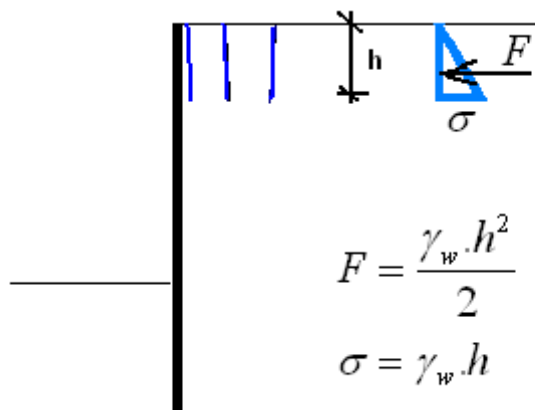
## Uplift Pressure in Footing Bottom

The variation of uplift pressure in the footing bottom due to difference in water tables is assumed according to expected effect linear, parabolic or is not taken into account.



## Influence of Tensile Cracks

The program makes it possible to account for the influence of tensile surface cracks filled with water. The analysis procedure is evident from the figure. The depth of tensile cracks is the only input parameter.

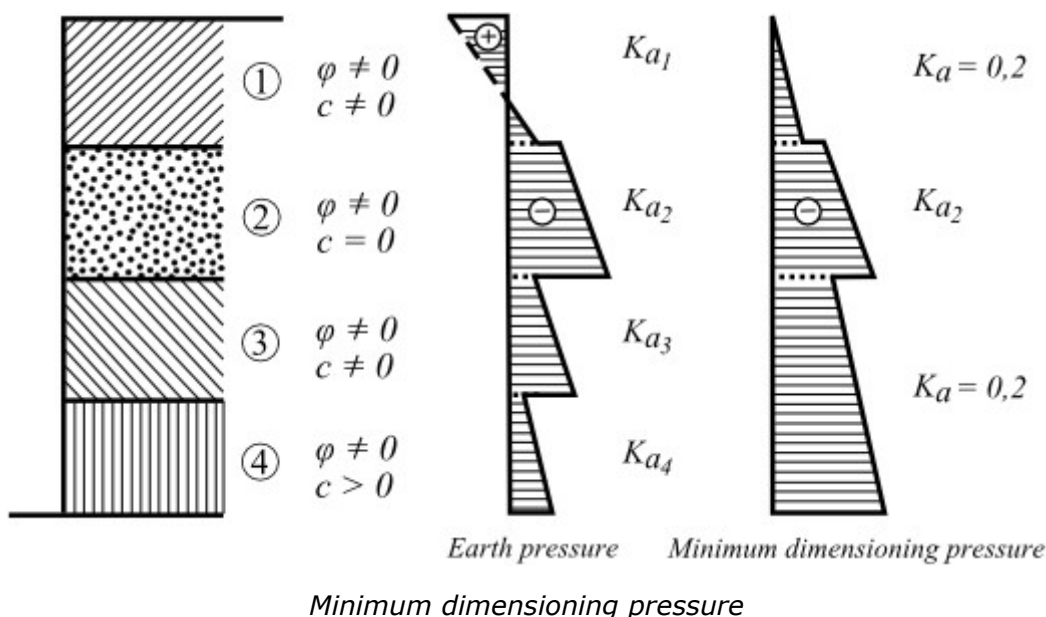


*Influence of tensile cracks*

## Minimum Dimensioning Pressure

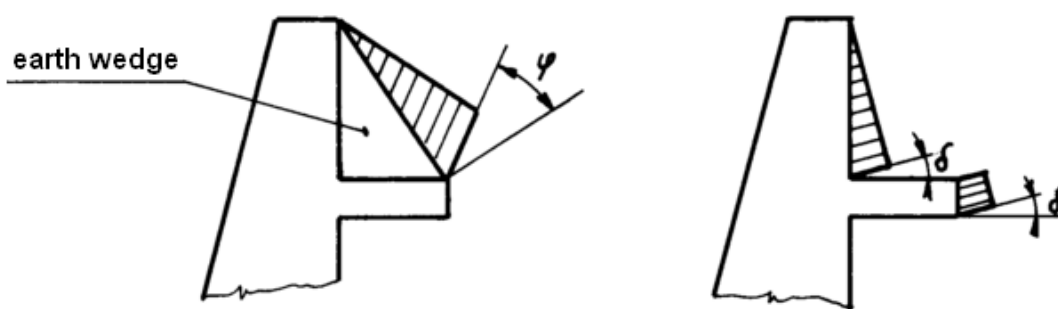
When determining the magnitude and distribution of earth pressures it is very difficult to qualify proportions of individual effects. This situation leads to uncertainty in the determination of earth pressure loading diagram. In reality we have to use in the design the most adverse distribution in favor of the safety of structure. For example, in case of braced structures in cohesive soils when using reasonable values of strength parameters of soil along the entire structure we may encounter tensile stresses in the upper part of the structure - see figure. Such tensile stresses, however, cannot be exerted on the sheeting structure (consequence of separation of soil due to technology of construction, isolation and drainage layer). In favor of the safe design of sheeting structure particularly in subsurface regions, where tensile stresses are developed during computation of the active earth pressure, the program offers the possibility to call the option "**Minimum dimensioning pressure**" in the analysis.

To determine the minimum dimensioning pressure the program employs for layers of cohesive soils as the minimal value of the coefficient of active earth pressure an alternate coefficient  $K_a = 0,2$ . Therefore it is ensured that the value of the computed active earth pressure will not drop below 20% of the vertical pressure ( $K_a \geq 0.2$ ) - see figure. Application of the minimum dimensioning pressure assumes for example the possibility of increasing the lateral pressure due to filling of joint behind the sheeting structure with rain water. If the option of minimum dimensioning pressure is not selected the program simply assumes tension cutoff ( $K_a \geq 0.0$ ).

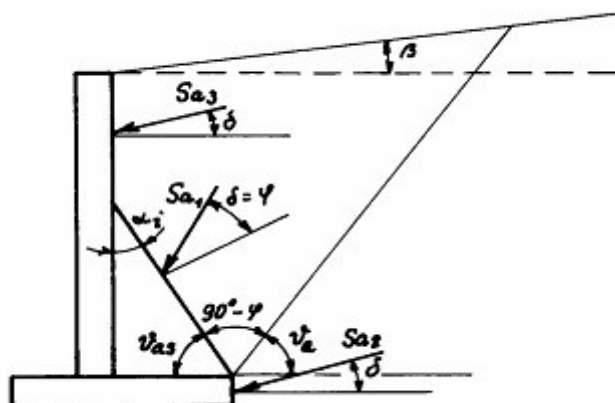


## Earth - Pressure Wedge

Providing a structure with a cantilever jump (foundation slab of cantilever wall, modification for reduction of earth pressures) is considered when computing earth pressures it is possible to compute earth pressures acting either on the real back of structure with the input angle of friction  $\delta \leq 2/3\phi$  or on an alternate back of structure. The alternate back of sheeting structure replaces the real broken one by a slip plane passing from the upper point of the back of wall towards the outer upper point of the jump and forms an earth wedge - see figure. A fully mobilized angle of friction  $\delta = \phi$  is assumed along this plane. The weight of earth wedge created under this alternate back further contributes to load applied to the structure. To introduce the alternate back of structure into the analysis it is necessary to select in the program Earth pressures the option "**Consider developing of earth-pressure wedge**". In other programs the earth wedge is introduced automatically.



Calculation with and without earth-pressure wedge



*Determination of earth-pressure wedge in case of active earth pressure*

The slip plane of the earth-pressure wedge is inclined from the horizontal line by angle  $\nu_a$  given by:

$$\nu_a = \varphi + \varepsilon$$

$$\tan \varepsilon = \frac{\cos(\varphi - \alpha) \cdot \sin(\varphi - \beta) \cdot \cos(\alpha + \delta) + B \cdot \cos(\varphi - \beta - \alpha - \delta)}{\sin(\varphi - \alpha) \cdot \sin(\varphi - \beta) \cdot \cos(\alpha + \delta) + B \cdot \sin(\varphi - \beta - \alpha - \delta) + M}$$

$$M = \sqrt{(\sin(\varphi - \beta) \cdot \cos(\beta - \alpha) + B) \cdot (\sin(\varphi + \delta) \cdot \cos(\alpha + \delta) + B)}$$

$$B = \frac{2 \cdot c \cdot \cos \alpha \cdot \cos(\beta - \alpha) \cdot \cos \varphi}{\gamma \cdot h \cdot \cos(\beta - \alpha) + \frac{2 \cdot \sigma_z \cdot \cos \alpha \cdot \cos \beta}{\gamma \cdot h}}$$

- where:
- $\varphi$  - angle of internal friction of soil
  - $\beta$  - slope inclination
  - $\delta$  - angle of friction between structure and soil
  - $\gamma$  - unit weight of soil
  - $\alpha$  - back face inclination of the structure
  - $h$  - height of earth wedge

The shape of the earth wedge in the layered subsoil is determined such that for individual layers of soil above the wall foundation the program computes the angle  $\nu_a$ , which then serves to determine the angle  $\nu_{as}$ . Next, the program determines an intersection of the line drawn under the angle  $\nu_{as}$  from the upper right point of the foundation block with the next layer. The procedure continues by drawing another line starting from the previously determined intersection and inclined by the angle  $\nu_{as}$ . The procedure is terminated when the line intersects the terrain or wall surface, respectively. The wedge shape is further assumed in the form of triangle (intersection with wall) or rectangle (intersection with terrain).

## Surcharge

The following types of surcharges are implemented in the program:

### Active earth pressure

- Surface surcharge
- Strip surcharge
- Trapezoidal surcharge
- Concentrated surcharge
- Line surcharge

### Earth pressure at rest

- Surface surcharge
- Strip surcharge
- Trapezoidal surcharge
- Concentrated surcharge

### Passive earth pressure

- Surface surcharge

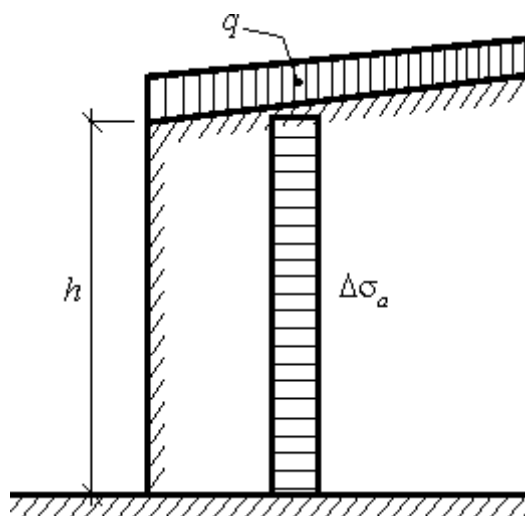
## Surface Surcharge - Active Earth Pressure

The increment of active earth pressure at rest due to surface surcharge is given by:

$$\Delta\sigma_a = p \cdot K_a$$

where:  $p$  - vertical uniform load  
 $K_a$  - coefficient of active earth pressure

The vertical uniform load  $p$  applied to the ground surface induces therefore over the entire height of the structure a constant increment of active earth pressure - see figure:



*Increment of active earth pressure due to vertical uniform ground surface surcharge*



## Strip Surcharge - Active Earth Pressure

For vertical strip load  $f_a$  acting parallel with structure on the ground surface along an infinitely long strip the trapezoidal increment of active earth pressure applied to the structure over a given segment  $h_f$  is assumed - see figure.

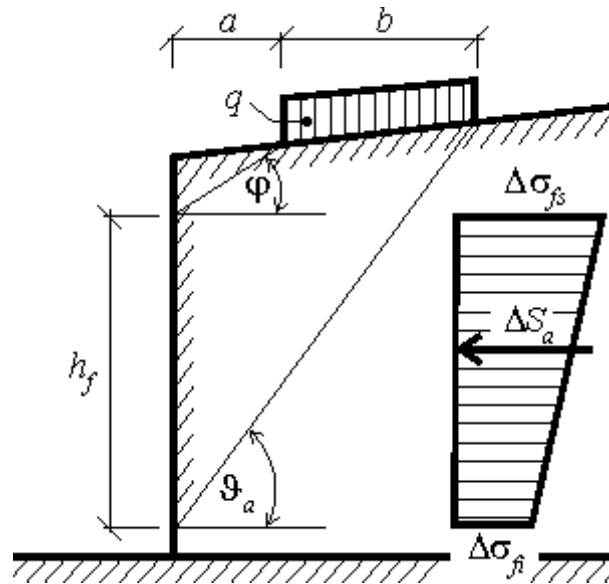


Diagram of increment of active earth pressure due to strip load  $f_a$

This segment is determined by intersection of the structure and lines drawn from the edge points of the strip load having slopes associated with angles  $\phi$  and  $\vartheta_a$ . The angle  $\vartheta_a$  corresponding to critical slip plane follows from:

$$\vartheta_a = \phi + \varepsilon$$

The formula is described in more detail in section "**Active earth pressure - line surcharge**".

Variation of pressure increment is trapezoidal; the larger intensity of  $\Delta\sigma_{fs}$  is applied at the upper end while the smaller intensity of  $\Delta\sigma_{fi}$  at the bottom end. The two increments are given by:

$$\Delta\sigma_{fs} = \frac{f_a \cdot b \cdot K_{\phi}}{h_f} \cdot \left(1 + \frac{a}{a+b}\right)$$

$$\Delta\sigma_{fi} = \frac{f_a \cdot b \cdot K_{\phi}}{h_f} \cdot \left(1 - \frac{a}{a+b}\right)$$

where:  $f_a$  - magnitude of strip surcharge  
 $b$  - width of the strip surcharge acting normal to the structure  
 $h_f$  - section loaded by active earth pressure increment

$$K_{\phi} = \frac{\sin(\vartheta_a - \phi)}{\cos(\vartheta_a - \phi - \delta)}$$

where:  $\vartheta_a$  - angle of critical slip plane

- $\varphi$  - angle of internal friction of soil  
 $\delta$  - angle of friction between structure and soil

The resultant of the increment of active earth pressure due to strip load  $f_a$  is provided by:

$$\Delta S_a = f_a \cdot b \cdot \frac{\sin(\vartheta_a - \varphi)}{\cos(\vartheta_a - \varphi - \delta)}$$

- where:
- $\vartheta_a$  - angle of critical slip plane
  - $\varphi$  - angle of internal friction of soil
  - $\delta$  - angle of friction between structure and soil
  - $f_a$  - magnitude of strip surcharge
  - $b$  - width of the strip surcharge

For non-homogeneous soils the program **proceeds as follows**.

## Trapezoidal Surcharge - Active Earth Pressure

The trapezoidal surcharge is subdivided in the program in ten segments. Individual segments are treated as **strip loads**. The resulting earth pressure is a sum of partial surcharges derived from individual segments.

## Concentrated Surcharge - Active Earth Pressure

The concentrated load (resultant  $F$  due to surface or concentrated load - see figure) is transformed into a **line load** with a limited length. If the width of surface load  $b$  is smaller than the distance  $a$  from the back of wall (see figure) the alternate line load  $f$  having length  $l+2(a+b)$  receives the form:

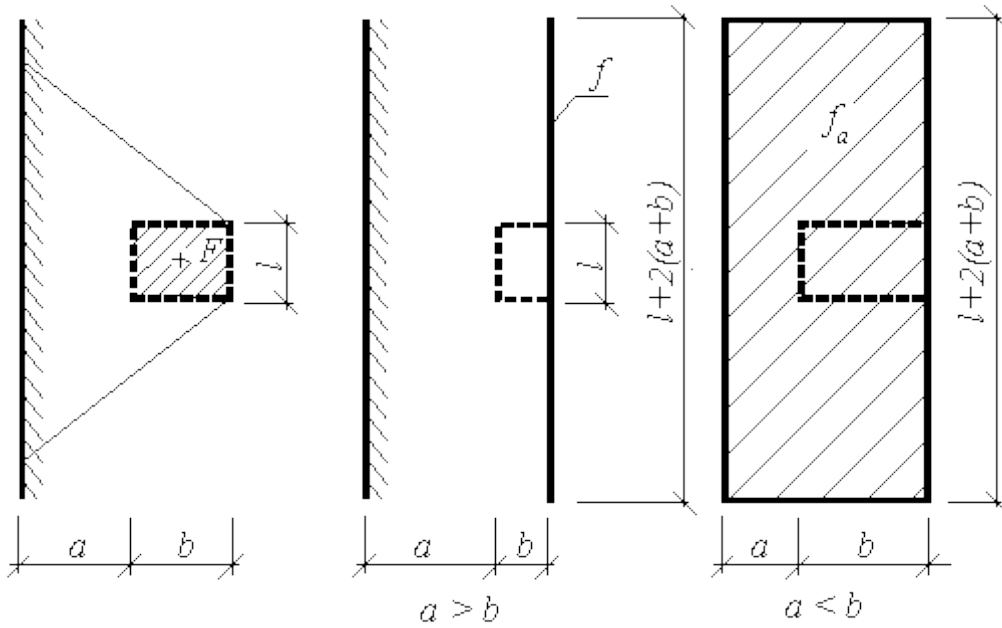
$$f = \frac{F}{l + 2 \cdot (a + b)}$$

- where:
- $F$  - resultant due to surface or concentrated load
  - $a$  - distance of load from the back of wall
  - $l$  - length of load
  - $b$  - width of surface load

If the width  $b$  of surface load is greater than the distance  $a$  from the back of wall (see figure) the alternate **strip load**  $f$  having length  $l+2(a+b)$  and width  $(a+b)$  reads:

$$f_a = \frac{F}{(l + 2 \cdot (a + b)) \cdot (a + b)}$$

- where:
- $F$  - resultant due to surface or concentrated load
  - $a$  - distance of load from the back of wall
  - $l$  - length of load
  - $b$  - width of surface load

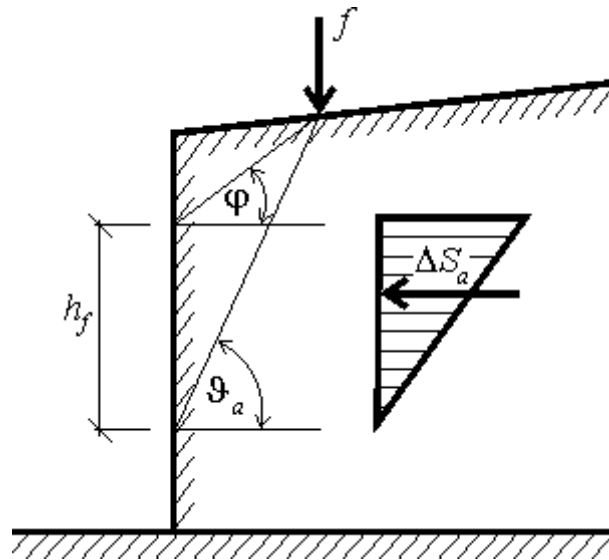


*Alternate load for calculation of increment of active earth pressure*

For non-homogeneous soils the program proceeds as follows.

### Line Surcharge - Active Earth Pressure

Vertical infinitely long line load  $f$  acting on the ground surface parallel with structure leads to a triangular increment of active earth pressure applied to the structure over a given segment  $h_f$  - see figure:



*Diagram of increment of active earth pressure due to vertical line load acting on ground surface*

Action of the line surcharge is determined such that two lines are drawn from the point of application following angles  $\varphi$  and  $\vartheta_a$  (corresponding to the critical slip surface), which is provided by:

$$\vartheta_a = \varphi + \varepsilon$$

where:  $\varphi$  - angle of internal friction of soil  
 $\varepsilon$  - angle derived from the following formulas

$$\tan \varepsilon = \frac{\cos(\varphi - \alpha) \cdot \sin(\varphi - \beta) \cdot \cos(\alpha + \delta) + B \cdot \cos(\varphi - \beta - \alpha - \delta)}{\sin(\varphi - \alpha) \cdot \sin(\varphi - \beta) \cdot \cos(\alpha + \delta) + B \cdot \sin(\varphi - \beta - \alpha - \delta) + M}$$

$$M = \sqrt{(\sin(\varphi - \beta) \cdot \cos(\beta - \alpha) + B) \cdot (\sin(\varphi + \delta) \cdot \cos(\alpha + \delta) + B)}$$

$$B = \frac{2 \cdot c \cdot \cos \alpha \cdot \cos(\beta - \alpha) \cdot \cos \varphi}{\gamma \cdot h \cdot \cos(\beta - \alpha) + \frac{2 \cdot \sigma_z \cdot \cos \alpha \cdot \cos \beta}{\gamma \cdot h}}$$

where:  $\beta$  - slope inclination  
 $\varphi$  - angle of internal friction of soil  
 $\delta$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure  
 $c$  - cohesion of soil  
 $\gamma$  - unit weight of soil  
 $h$  - assumed depth

For non-homogeneous soil and inclination of ground surface  $\beta$  smaller than the angle of internal friction of the soil  $\varphi$  the value of the angle  $\varepsilon$  is given by:

$$\cot g = \operatorname{tg}(\varphi - \alpha) + \frac{1}{\cos(\varphi - \alpha)} \sqrt{\frac{\sin(\varphi + \delta) \cdot \cos(\alpha - \beta)}{\sin(\varphi - \beta) \cdot \cos(\alpha + \delta)}}$$

where:  $\beta$  - slope inclination  
 $\varphi$  - angle of internal friction of soil  
 $\delta$  - angle of friction between structure and soil  
 $\alpha$  - back face inclination of the structure

The resultant of the increment of active earth pressure due to line load  $f$  is provided by:

$$\Delta S_a = f \cdot \frac{\sin(\vartheta_a - \varphi)}{\cos(\vartheta_a - \varphi - \delta)}$$

where:  $\vartheta_a$  - angle of critical slip plane  
 $\varphi$  - angle of internal friction of soil  
 $\delta$  - angle of friction between structure and soil  
 $f$  - magnitude of line surcharge

For non-homogeneous soils the program **proceeds as follows**.

## Surcharge in Non-Homogeneous Soil

For non-homogeneous soil we proceed as follows:

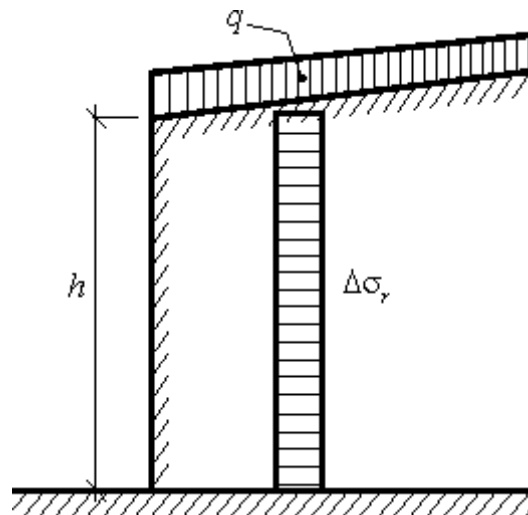
- Compute the angle  $\vartheta_a$  for a given soil layer.
- Determine the corresponding magnitude of force  $S_a$  and size of the corresponding pressure diagram.
- Determine the magnitude of earth pressure acting below the bottom edge of a given layer, and its ratio with respect to the overall pressure magnitude.
- The surcharge is reduced using the above ratio, then the location of this surcharge on the upper edge of the subsequent layer is determined.
- Compute again the angle  $\vartheta_a$  for the next layer and repeat the previous steps until the bottom of a structure is reached or the surcharge is completely exhausted.

## Surface Surcharge - Earth Pressure at Rest

An increment of uniformly distributed earth pressure at rest  $\Delta\sigma_r$  caused by the vertical surface load applied on the ground surface behind the structure is computed using the following formula:

$$\Delta\sigma_r = f \cdot K_r$$

where:  $f$  - magnitude of surface surcharge  
 $K_r$  - coefficient of earth pressure at rest



*Diagram of increment of earth pressure at rest due to vertical uniform load acting on ground surface*

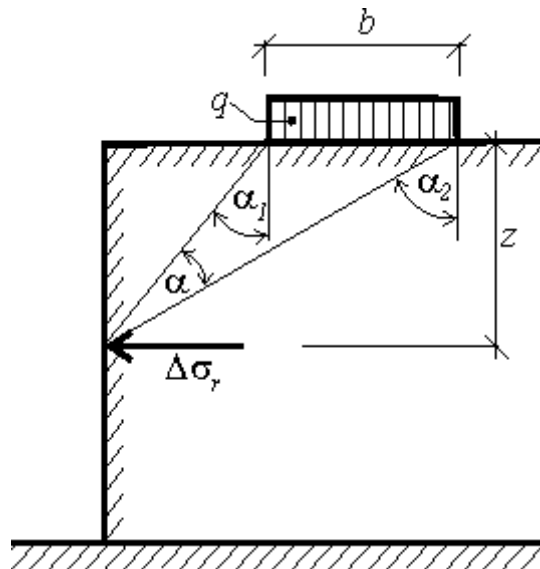
## Strip Surcharge - Earth Pressure at Rest

Uniform strip load  $f_a$  acting on the ground surface behind the structure parallel with vertical structure (see figure) creates an increment of earth pressure at rest  $\Delta\sigma_r$  having the magnitude given by:

$$\Delta\sigma_r = \frac{f_a}{\pi} (2.\alpha - \sin 2\alpha_2 + \sin 2\alpha_1)$$

where:  $f_a$  - vertical strip surcharge  
 $\alpha, \alpha_1, \alpha_2$  - evident from figure

The increase of the pressure can never be higher than pressure from surface surcharge of the same magnitude.



*Increment of earth pressure due to vertical strip surcharge*

### Trapezoidal Surcharge - Earth Pressure at Rest

The trapezoidal surcharge is subdivided in the program in five segments. Individual segments are treated as **strip loads**. The resulting earth pressure is a sum of partial surcharges derived from individual segments.

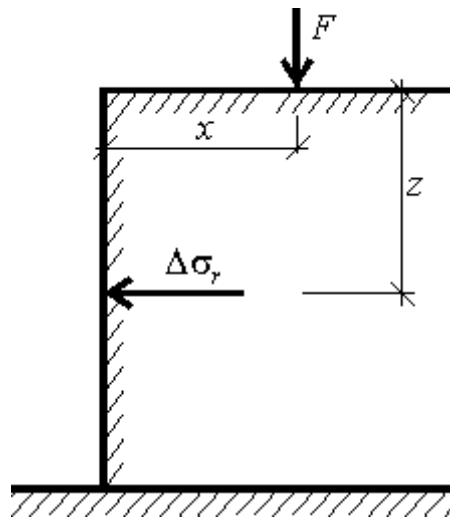
### Concentrated Surcharge - Earth Pressure at Rest

Application of concentrated force  $F$  yields an increment of earth pressure at rest  $\Delta\sigma_r$  acting on the vertical structure and having the magnitude of:

$$\Delta\sigma_r = \frac{3.F}{\pi} \left( \frac{x^2.z}{r^5} + \frac{1-2.\nu}{3} \left( \frac{1}{r.(r+z)} - \frac{(2.r+z).x^2}{(r+z)^2.x^3} - \frac{z}{r^3} \right) \right)$$

$$r = \sqrt{x^2 + z^2}$$

where:  $F$  - concentrated force acting on ground surface  
 $x, z$  - coordinates evident from figure



*Increment of earth pressure at rest due to vertical concentrated force*

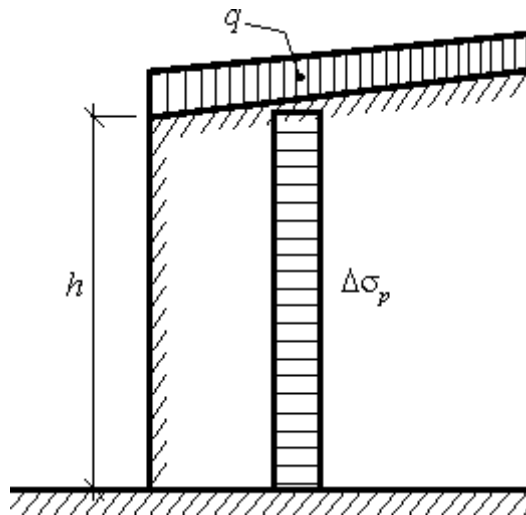
### Surface Surcharge - Passive Earth Pressure

For passive earth pressure only an increment due to vertical uniform load  $f_a$  is determined using the formula:

$$\Delta\sigma_p = f_a \cdot K_p$$

- where:
- $f_a$  - vertical surface surcharge
  - $K_p$  - coefficient of passive earth pressure

The vertical uniform load  $q$  acting on the ground surface therefore results in a constant increment of passive pressure applied over the whole length of wall - see figure.



*Increment of the passive earth pressure*

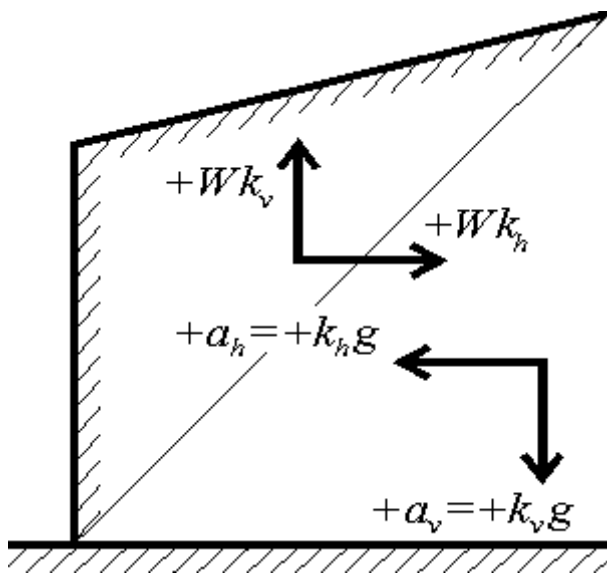
### Influence of Earthquake

Earthquake increases the effect of active pressure and reduces the effect of passive pressure.

The theories used in our programs (Mononobe-Okabe, Arrango, JTJ 004-89, JTS 146-2012, SL 203-97) are derived assuming cohesionless soils without influence of water. Therefore, all input soils are assumed cohesionless when employing these theories to address the earthquake effects. Earthquake effects due to surcharge are not considered in the program - the user may introduce these effects (depending on the type of surcharge) as **"Applied forces"**.

The coefficient  $k_h$  is assumed always positive and such that its effect is always unfavorable. The coefficient  $k_v$  may receive both positive and negative value. If the equivalent acceleration  $a_v$  acts downwards (from the ground surface) the inertia forces  $k_v W_s$  will be exerted on the soil wedge in the opposite direction (lifting the wedge up). The values of equivalent acceleration  $a_v$  (and thus also the coefficient  $k_v$ ) and inertia forces  $k_v W_s$  are assumed as positive. It is clearly evident that the inertia forces act in the direction opposite to acceleration (if the acceleration is assumed upwards -  $a_v = -k_v g$  then the inertia force presses the soil wedge downwards:  $-k_v W_s$ ). The direction with most unfavorable effects on a structure is assumed when examining the seismic effects.

For sheeting structures it is possible to neglect the effect of vertical equivalent acceleration  $k_v W_s$  and input  $k_v = 0$ .



Sign convention

The seismic angle of inertia is determined from the coefficients  $k_h$  and  $k_v$  (i.e. angle between the resultant of inertia forces and the vertical line) using the formula:

$$\psi = \tan^{-1} \left( \frac{k_h}{1 - k_v} \right)$$

where:  $k_v$  - seismic coefficient of vertical acceleration  
 $k_h$  - seismic coefficient of horizontal acceleration

**Pressure from seismic effects**

Increment of active earth pressure due to seismic effects (computed from the structure bottom) follows from:

$$\sigma_{ae,i} = \sigma_{0,i} (K_{ae,i} - K_{a,i})$$



$$\sigma_{0,i} = \sum_0^H \gamma_i \cdot h_i (1 - k_v)$$

- where:
- $\gamma_i$  - unit weight of soil in the  $i^{th}$  layer
  - $K_{ae,i}$  - coefficient of active earth pressure (static and seismic) in the  $i^{th}$  layer
  - $K_{a,i}$  - magnitude of earth pressure in the  $i^{th}$  layer due to **Coulomb**
  - $h_i$  - thickness of the  $i^{th}$  layer
  - $k_v$  - seismic coefficient of vertical acceleration

Reduction of passive pressure due to seismic load (computed from the structure bottom) is provided by:

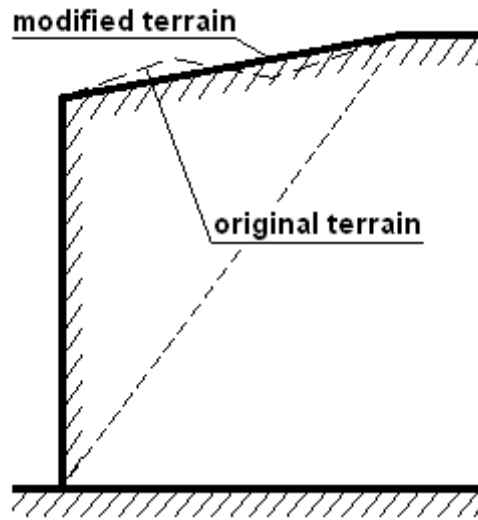
$$\sigma_{pe,i} = \sigma_{0,i} (K_{p,i} - K_{pe,i})$$

$$\sigma_{0,i} = \sum_0^H \gamma_i \cdot h_i (1 - k_v)$$

- where:
- $\gamma_i$  - unit weight of soil in the  $i^{th}$  layer
  - $K_{pe,i}$  - coefficient of passive earth pressure (static and seismic) in the  $i^{th}$  layer
  - $K_{p,i}$  - magnitude of earth pressure in the  $i^{th}$  layer due to **Coulomb**
  - $h_i$  - thickness of the  $i^{th}$  layer
  - $k_v$  - seismic coefficient of vertical acceleration

Active earth pressure coefficient  $K_{ae,i}$  and passive earth pressure coefficient  $K_{pe,i}$  are computed using the **Mononobe-Okabe theory** or the **Arrango theory**. If there is **ground water** in the soil body the program takes that into account.

The basic assumption in the program when computing earthquake is a flat ground surface behind structure with inclination  $\beta$ . If that is not the case the program approximates the shape of terrain by a flat surface as evident from figure:



Terrain shape approximation

### Point of application of resultant force

The resultant force is automatically positioned by the program into the center of the stress diagram. Various theories recommend, however, different locations of the resultant force - owing to that it is possible to select the point of application of the resultant force in the range of  $0.33 - 0.7H$  ( $H$  is the structure height). Recommended (implicit) value is  $0.66H$ . Having the resultant force the program determines the trapezoidal shape of stress keeping both the input point of application of the resultant force and its magnitude.

## Mononobe-Okabe Theory

The coefficient  $K_{ae}$  for active earth pressure is given by:

$$K_{ae} = \frac{\cos^2(\varphi - \psi - \alpha)}{\cos\psi \cos^2\alpha \cos(\psi + \alpha + \delta) \left( 1 + \sqrt{\frac{\sin(\varphi + \delta) \sin(\varphi - \psi - \beta)}{\cos(\delta + \psi + \alpha) \cos(-\beta + \alpha)}} \right)^2}$$

The coefficient  $K_{pe}$  for passive earth pressure is given by:

$$K_{pe} = \frac{\cos^2(\varphi - \psi + \alpha)}{\cos\psi \cos^2\alpha \cos(\psi - \alpha + \delta) \left( 1 - \sqrt{\frac{\sin(\varphi + \delta) \sin(\varphi - \psi + \beta)}{\cos(\delta + \psi - \alpha) \cos(\beta - \alpha)}} \right)^2}$$

- where:
- $\gamma$  - unit weight of soil
  - $H$  - height of the structure
  - $\varphi$  - angle of internal friction of soil
  - $\delta$  - angle of friction between structure and soil
  - $\alpha$  - back face inclination of the structure
  - $\beta$  - slope inclination
  - $k_v$  - seismic coefficient of vertical acceleration

- $k_h$  - seismic coefficient of horizontal acceleration  
 $\psi$  - seismic inertia angle

Deviation of seismic forces  $\psi$  must be for active earth pressure always less or equal to the difference of the angle of internal friction and the ground surface inclination (i.e.  $\varphi - \beta$ ). If the values  $\psi$  of is greater the program assumes the value  $\psi = \varphi - \beta$ . In case of passive earth pressure the value of deviation of seismic forces  $\psi$  must be always less or equal to the sum of the angle of internal friction and the ground surface inclination (i.e.  $\varphi + \beta$ ). The values of computed and modified angle  $\psi$  can be visualized in the output - in latter case the word **MODIFIED** is also displayed.

#### Analysis of earthquake effects (active earth pressure) - partial results

Layer No.	thick. [m]	$\phi_d$ [°]	$\psi$ [°]	$K_a$	$K_{ae}$	$K_{ae}-K_a$	Comment.
1	0.55	29.00	75.96	0.742	3.880	3.138	
2	0.48	29.00	75.96(17.69)	0.742	3.880	3.138	MODIFIED
3	1.30	29.00	75.96(17.69)	0.742	3.880	3.138	MODIFIED

Example of the program output

Literature:

Mononobe N, Matsuo H 1929, On the determination of earth pressure during earthquakes. In Proc. Of the World Engineering Conf., Vol. 9, str. 176.

Okabe S., 1926 General theory of earth pressure. Journal of the Japanese Society of Civil Engineers, Tokyo, Japan 12 (1).

## Arrango Theory

The program follows the Coulomb theory to compute the values of  $K_a$  and  $K_p$  while taking into account the dynamic values ( $\alpha^*$ ,  $\beta^*$ ).

**For active earth pressure:**

$$\beta^* = \beta + \psi$$

$$\alpha^* = \alpha + \psi$$

**For passive earth pressure:**

$$\beta^* = \beta - \psi$$

$$\alpha^* = \alpha - \psi$$

- where:
- $\beta$  - slope inclination
  - $\alpha$  - back face inclination of the structure
  - $\psi$  - seismic forces inclination

The coefficients of earth pressures  $K_{ae}$  and  $K_{pe}$  are found by multiplying the coefficients  $F_{ae}$  and  $F_{pe}$  by the values of  $K_a$  and  $K_p$ , respectively.

$$F_{ae} = \frac{\cos^2(\alpha + \psi)}{\cos \psi \cdot \cos^2 \alpha}$$

$$F_{pe} = \frac{\cos^2(\alpha - \psi)}{\cos \psi \cdot \cos^2 \alpha}$$

where:  $\alpha$  - back face inclination of the structure  
 $\psi$  - seismic forces inclination

If the value of the angle  $\beta^*$  becomes larger than  $\phi$  the program assumes the value ( $\beta^* = \phi$ ). The values of computed and modified angle  $\beta^*$  can be visualized in the output - in latter case the word **MODIFIED** is also displayed. It is the user's responsibility to check in such case whether the obtained results are realistic.

**Analysis of earthquake effects (active earth pressure) - partial results**

Layer No.	thick. [m]	$\phi_d$ [°]	$\beta$ [°]	$\psi$ [°]	$K_a$	$K_{ae}$	$K_{ae}-K_a$	Comment.
1	0.30	26.50	71.43(26.50)	57.99	0.427	3.973	3.546	MODIFIED
2	0.80	26.50	71.43(26.50)	57.99	0.427	3.973	3.546	MODIFIED
3	0.40	26.50	71.43(26.50)	57.99	0.427	3.973	3.546	MODIFIED

*Example of the program output*

*Literature:*

*Design of sheet pile walls, Pile Buck Inc., Vero Beach, Florida, [www.pilebuck.com](http://www.pilebuck.com).*

## Influence of Water

When examining the influence of ground water on the magnitudes of earth pressure the program differentiates between confined and unconfined water. Hydrodynamic pressure acting on the front face of the wall is calculated, if the wall is flooded at the front face side.

### Confined water

This type is used in soils with lower permeability - app. below the value of  $k = 1 \times 10^{-3} \text{ cm/s}$ . In such soils the water flow is influenced, e.g. by actual grains (by their shape and roughness) or by resistance of fraction of adhesive water. General formulas proposed by Mononobe-Okabe or Arrango are used to analyze seismic effects. The only difference appears in substituting the value of the seismic angle  $\psi$  by  $\psi^*$ :

$$\psi^* = \tan^{-1} \left( \frac{\gamma_{sat} \cdot k_h}{\gamma_{su} \cdot (1 - k_v)} \right)$$

where:  $\gamma_{sat}$  - unit weight of fully saturated soil  
 $\gamma_{su}$  - unit weight of submerged soil  
 $k_h$  - seismic coefficient of horizontal acceleration  
 $k_v$  - seismic coefficient of vertical acceleration

### Unconfined water

This type is used in soils with higher permeability - app. above the value of  $k > 1 \times 10^{-1} \text{ cm/s}$ . In such soils it is assumed that water flow in pores is more or less independent of soil grains (e.g.

turbulent flow in coarse grain soils). General formulas proposed by Mononobe-Okabe or Arrango are used to analyze seismic effects. The only difference appears in substituting the value of the seismic angle  $\psi_e$  by  $\psi_e^+$ :

$$\psi^+ = \tan^{-1} \left( \frac{k_{he}^+}{1 - k_v} \right)$$

$$k_{he}^+ = \frac{\gamma_d}{\gamma_{su}} \cdot k_h = \frac{G_s}{G_s - 1} \cdot k_h$$

where:  $\gamma_d$  - unit weight of dry soil  
 $\gamma_{su}$  - unit weight of submerged soil  
 $k_h$  - seismic coefficient of horizontal acceleration  
 $k_v$  - seismic coefficient of vertical acceleration  
 $G_s$  - specific gravity of soil particles

$$G_s = \frac{\rho_s}{\rho_w}$$

where:  $\rho_s$  - density of the soil solids  
 $\rho_w$  - density of water

Apart from dynamic pressure the structure is also loaded by hydrodynamic pressure caused by free water manifested by dynamic pressure applied to the structures. The actual parabolic distribution is in the program approximated by the trapezoidal distribution.

The resultant of hydrodynamic pressure **behind the structure**  $P_{wd}$  is distant by  $y_{wd}$  from the heel of structure:

$$y_{wd} = 0,4.H$$

where:  $H$  - height of the structure

and its magnitude follows from:

$$P_{wd} = \frac{7}{12} \cdot k_h \cdot \gamma_w \cdot H^2$$

where:  $\gamma_w$  - unit weight of water  
 $k_h$  - seismic coefficient of horizontal acceleration  
 $H$  - height of the structure

### Hydrodynamic pressure acting on the front face of the wall

The resultant of hydrodynamic pressure on the **front face of the wall**  $P_{wd}$  is distant by  $y_{wd}$  from the heel of structure:

$$y_{wd} = 0,4.H$$

where:  $H$  - height of the structure

and its magnitude follows from:

$$P_{wd} = \frac{7}{12} \cdot k_h \cdot \gamma_w \cdot H^2$$

where:  $\gamma_w$  - unit weight of water  
 $k_h$  - seismic coefficient of horizontal acceleration  
 $H$  - height of the structure

## EN 1998-5 Seismic Effects

If the coefficients  $k_h$  and  $k_v$  are not obtained from measurements it is necessary, providing the analysis is carried out according to 1998-5 Eurocode 8: Design of structures for earthquake resistance - Part 5: Foundations, retaining structures and geotechnical aspects, to input these coefficients as follows:

$$k_h = \alpha \cdot \frac{S}{R}$$

where:  $\alpha$  - ratio of the design ground acceleration on type A ground ( $a_g/g$ )  
 $S$  - soil factor defined in EN 1998-1:2004, chapter 3.2.2.2  
 $R$  - factor for the calculation of the horizontal seismic coefficient - see tab.

for:

$$\frac{a_{vg}}{a_g} > 0,6 \quad k_v = \pm 0,5 \cdot k_h$$

for other cases:

$$k_v = \pm 0,33 \cdot k_h$$

Type of sheeting structure	$R$
Free gravity walls that can accept a displacement up to $d_r = 300\alpha S$ (mm)	2
Free gravity walls that can accept a displacement up to $d_r = 200\alpha S$ (mm)	1.5
Flexural reinforced concrete walls, anchored or braced walls, reinforced concrete walls 1 founded on vertical piles, restrained basement walls and bridge abutments	1

**More detailed description can be found in EN 1998-5 chapter 7.3.2.2 Seismic effects**

## Forces from Earth Pressure at Rest Acting on the Rigid Structure

Earth pressure at rest acts on the rigid structures (e.g. stem of the cantilever wall) in earthquake analysis. Following resultant force of this pressure is taken into account:

$$F = k_h \cdot \gamma \cdot H^2$$

where:  $H$  - height of the structure  
 $k_h$  - seismic coefficient of horizontal acceleration  
 $\gamma$  - unit weight of soil

Resultant force acts in the half of the structure height.

## Influence of Earthquake according to Chinese Standards

Three different Chinese standards are implemented for calculation of seismic effect for wall design which are **JTJ 004-89 (Specifications of Earthquake Resistance Design for Highway Engineering)**, **SL 203-97 (Specification for seismic design of hydraulic structures)**, **JTS 146-2012 (Code for seismic Design of Water Transport Engineering)**. They are all based on **Mononobe-Okabe theory**. The main difference between Chinese standards and Mononobe-Okabe theory is that comprehensive influence factor  $C_z$  is introduced in Chinese standards which reduces seismic force by about 70 %.

The advantage of choosing Chinese standards as the option for earthquake analysis is that users only need to choose the intensity of the earthquake according to which the program automatically assign values of other parameters appropriate with standards.

### Influence of Earthquake according to JTJ 004-89

Only horizontal seismic force is considered according to JTJ 004-89.

#### Seismic force on structure

Seismic force acting on structure is provided by (Art. 3.1.5):

$$E_{ihw} = C_i C_z K_h \psi_{iw} G_{iw}$$

where:  $E_{ihw}$  - seismic force acting at the center of gravity of the wall above the  $i^{th}$  cross section [kN/m]

$K_h$  - coefficient of horizontal seismic acceleration

$G_{iw}$  - weight of the structure above the  $i$ th cross section [kN/m]

$C_z$  - comprehensive influence factor, usually it's 0.25

$C_i$  - importance coefficient for seismic design

$\psi_{iw}$  - distribution coefficient of horizontal earthquake along the wall

Recommended value of distribution coefficient  $\psi_{iw}$  (Tbl. 3.1.5):

Wall Height [m]	Security level		Calculation diagram for $\psi_{iw}$
	Highway, A class and B class motorway	C class and D class motorway	
$H \leq 12$	$\psi_{iw} = 1$	$\psi_{iw} = 1$	
$H > 12$	$\psi_{iw} = 1 + \frac{H_{iw}}{H}$	$\psi_{iw} = 1$	

$\psi_{iw}$  isn't considered when  $H \leq 12$  m which means parameter  $a$  and  $b$  don't work when  $H \leq 12$  m.  $a$  is the top value of the distribution map and  $b$  is the bottom value of the distribution map.

### Seismic earth pressure

When computing seismic earth pressure, Coulomb theory is used and unit weight of soil  $\gamma$ , internal friction angle of soil  $\varphi$  and angle of friction structure-soil  $\delta$  is replaced by  $\gamma / \cos \theta$ ,  $\varphi - \theta$ ,  $\delta + \theta$ , where  $\theta$  is seismic angle (Art. 3.1.6).

Seismic angle  $\theta$  is determined by different option of [seismic fortification intensity](#).

### Water influence

[Water influence according to Chinese standard](#) is a little different from the [water influence according to Mononobe-Okabe or Arrango theory](#) by reducing the water influence using comprehensive influence factor  $C_z$ .

### Seismic bearing capacity of subsoil

Seismic bearing capacity of subsoil is provided by (Art. 2.2.1):

$$f_{aE} = \zeta_a f_a$$

- where:
- $f_{aE}$  - seismic bearing capacity of subsoil
  - $\zeta_a$  - adjusting coefficient for seismic bearing capacity
  - $f_a$  - characteristic value of bearing capacity which has been modified by the geometry of foundation

The above formula is as same as Art. 4.2.3 in GB 50011-2010 (Code for seismic design of buildings). Suggested values of  $\zeta_a$  by different standards can be found [here](#).

## Influence of Earthquake according to JTS 146-2012

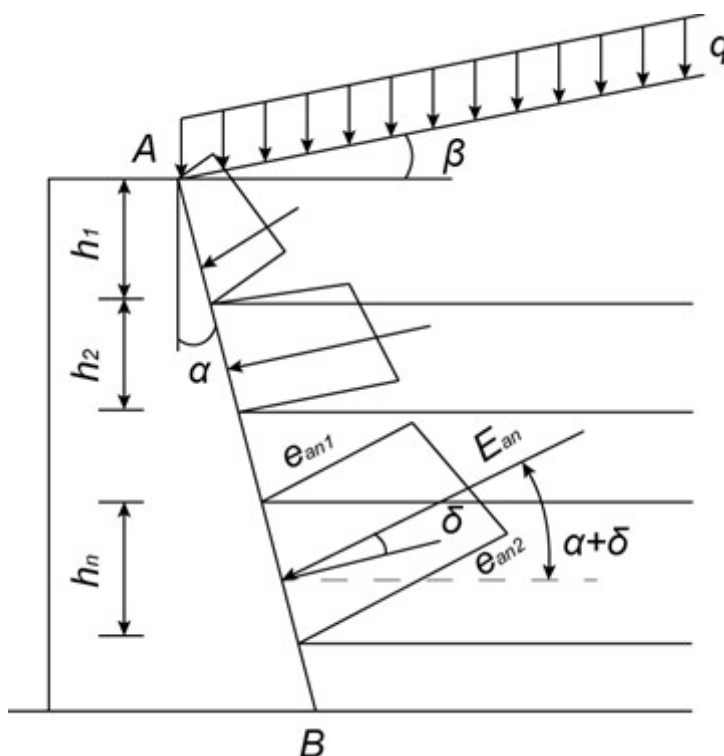
Only horizontal seismic force is considered according to JTS 146-2012.

### Seismic force on structure

Calculation of seismic force acting on structure is as same as [JTJ 004-89](#). Values of  $C_z$  can be set from 0.2 to 0.5 suggested by Art. 5.2 in JTS 146-2012.

### Seismic earth pressure





Distribution of seismic active pressure (Fig. 5.3.1)

The only difference between JTS 146-2012 and JTJ 004-89 is that seismic earth pressure according to JTS 146-2012 considers the influence of cohesion.

Active seismic earth pressure is provided by (Art. 5.3.1):

$$E_{an} = \frac{1}{2} (e_{an1} + e_{an2}) \frac{h_n}{\cos \alpha}$$

where:

$$e_{an1} = \left( K_q q + \sum_{i=0}^{n-1} \gamma_i h_i \right) K_{an} \cos \alpha - 2c_n K_{acn} \cos \alpha$$

$$e_{an2} = \left( K_q q + \sum_{i=0}^n \gamma_i h_i \right) K_{an} \cos \alpha - 2c_n K_{acn} \cos \alpha$$

$$K_q = \frac{\cos \alpha}{\cos(\alpha - \beta)}$$

$$K_{an} = \frac{\cos^2(\varphi_n - \alpha - \theta)}{\cos \theta \cos^2 \alpha \cos(\delta_n + \theta + \alpha) \left( 1 + \sqrt{\frac{\sin(\varphi_n + \delta_n) \sin(\varphi_n - \beta - \theta)}{\cos(\delta_n + \theta + \alpha) \cos(\alpha - \beta)}} \right)^2}$$

$$K_{acn} = \frac{\cos(\alpha - \beta) \cos \varphi_n}{\cos \theta \cos \alpha [1 + \sin(\varphi_n + \delta_n - \beta + \alpha)]}$$

$E_{an}$  overall active pressure acting on  $n^{th}$  layer [kN/m]

$e$   
 $r$   
 $e$   
 $:$

$e_a^n$  active pressure acting on the top of the  $n^{th}$  layer [ $kPa$ ]

$n$   
 $l$

$e_a^n$  active pressure acting on the bottom of the  $n^{th}$  layer [ $kPa$ ]

$n$   
 $2$

$h_n$  thickness of  $n^{th}$  layer [ $m$ ]

$\alpha$  back face inclination of the structure [ $^\circ$ ]

$K$  coefficient

$q$

$q$  uniform load acting on the terrain [ $kPa$ ]

$\gamma_i$  unit weight of  $i^{th}$  layer [ $kN/m^3$ ], below water - buoyant unit weight is accepted

$h_i$  thickness of the  $i^{th}$  layer [ $m$ ]

$K_a^n$  coefficient of active pressure of  $n^{th}$  layer

$a$   
 $n$

$c_n$  standard value of cohesion of  $n^{th}$  layer [ $kPa$ ]

$K_{sc}^n$  coefficient of seismic active pressure of  $n^{th}$  layer

$a$   
 $c$   
 $n$

$\beta$  slope inclination [ $^\circ$ ], and  $|\beta| < \varphi$

$\varphi_n$  internal friction angle of  $n^{th}$  layer [ $^\circ$ ]

$\theta$  seismic angle [ $^\circ$ ]

$\delta_n$  angle of friction between structure and soil of  $n^{th}$  layer [ $^\circ$ ]

Passive seismic earth pressure is provided by (Art. 5.3.2):

$$E_{pn} = \frac{1}{2} (e_{pn1} + e_{pn2}) \frac{h_n}{\cos \alpha}$$

where:

$$e_{pn1} = \left( K_q q + \sum_{i=0}^{n-1} \gamma_i h_i \right) K_{pn} \cos \alpha + 2c_n K_{pcn} \cos \alpha$$

$$e_{pn2} = \left( K_q q + \sum_{i=0}^n \gamma_i h_i \right) K_{pn} \cos \alpha + 2c_n K_{pcn} \cos \alpha$$

$$K_q = \frac{\cos \alpha}{\cos(\alpha - \beta)}$$

$$K_{pn} = \frac{\cos^2(\varphi_n + \alpha - \theta)}{\cos \theta \cos^2 \alpha \cos(\delta_n + \theta - \alpha) \left( 1 - \sqrt{\frac{\sin(\varphi_n + \delta_n) \sin(\varphi_n + \beta - \theta)}{\cos(\delta_n + \theta - \alpha) \cos(\alpha - \beta)}} \right)^2}$$

$$K_{pcn} = \frac{\cos(\alpha - \beta) \cos \varphi_n}{\cos \theta \cos \alpha [1 - \sin(\varphi_n + \delta_n + \beta - \alpha)]}$$

Seismic angle  $\theta$  is determined by different options of [seismic fortification intensity](#).

### Water influence

[Water influence according to Chinese standard](#) is a little different from the [water influence according to Mononobe-Okabe or Arrango theory](#) by reducing the water influence using comprehensive influence factor  $C_z$ .

### Seismic bearing capacity of subsoil

Calculation of seismic bearing capacity of subsoil is as same as [JTJ 004-89](#).

## Influence of Earthquake according to SL 203-97

Both horizontal and vertical seismic force can be considered according to SL 203-97. In SL 203-97, seismic angle  $\theta$  is derived automatically from  $K_h$ , so seismic angle  $\theta$  and seismic angle blow water  $\theta'$  are not visible in the frame.

### Seismic force on structure

Calculation of horizontal seismic force acting on structure is as same as [JTJ 004-89](#).

Vertical seismic force acting on structure is provided by (Art. 4.1.8):

$$E_{ivw} = C_0 \frac{E_{ihw}}{K_h} K_v$$

- where:  $E_{ivw}$  - vertical seismic force acting at the center of gravity of the wall above the  $i^{th}$  cross section [kN/m]
- $E_{ihw}$  - horizontal seismic force acting at the center of gravity of the wall above the  $i^{th}$  cross section [kN/m]
- $k_h$  - coefficient of horizontal seismic acceleration
- $k_v$  - coefficient of vertical seismic acceleration, usually, it's  $\pm 2/3 K_h$  (Art. 4.3.2)
- $G_{iw}$  - weight of the structure above the  $i^{th}$  cross section [kN/m]
- $C_0$  - meeting coefficient related to the influence of horizontal seismic effect,

usually, it's 0.5.

### Seismic earth pressure

Calculation of seismic earth pressure is as same as [JTJ 004-89](#). The only difference between SL 203-97 and JTJ 004-89 is that SL 203-97 has no "user defined - input  $K_h, \theta$ " as an option for [seismic fortification intensity](#).

### Water influence

[Water influence according to Chinese standard](#) is a little different from the [water influence according to Mononobe-Okabe or Arrango theory](#) by reducing the water influence using comprehensive influence factor  $C_z$ .

### Seismic bearing capacity of subsoil

Calculation of seismic bearing capacity of subsoil is as same as [JTJ 004-89](#).

## Seismic Fortification Intensity according to Chinese Standards

There are three main kinds of option for seismic fortification intensity according Chinese standards. Horizontal seismic acceleration coefficient  $K_h$  and seismic angle  $\theta$  are determined according to what option is selected for seismic fortification intensity.

- 7 degree (0.1g), 7 degree (0.15g), 8 degree (0.2g), 8 degree (0.3g), 9 degree (0.4g):  $K_h$  and seismic angle  $\theta$  are determined according to corresponding seismic fortification intensity based on standards.
- User defined - input  $K_h$ :  $K_h$  is input by user and  $\theta$  is determined by (Art. 4.9.1 from SL 203-97)

$$\theta = \arctan\left(\frac{C_z C_i K_h}{1 - C_z C_i K_v}\right)$$

- User defined - input  $K_h, \theta$ :  $K_h$  and  $\theta$  are both input by user.

**Note:** Third option is only valid for JTJ 004-89 and JTS 146-2012.

Values of  $K_h$  and  $\theta$  according to corresponding seismic fortification intensity are given by the following tables:

#### For [JTJ 004-89](#) (Tbl. 1.0.7 and Tbl. 3.1.6 from [JTJ 004-89](#))

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration $K_h$	0.10	0.15	0.20	0.30	0.40
Seismic angle $\theta$	1.5°	2.3°	3.0°	4.5°	6.0°

**Note:** 7 degree (0.15g) and 8 degree (0.30g) are not from JTJ 004-89, because there are no value for these two situations in JTJ 004-89. They are from JTS 146-2012.

#### For [JTS 146-2012](#) (Tbl. 5.3.1 from [JTS 146-2012](#))

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration $K_h$	0.10	0.15	0.20	0.30	0.40
Seismic angle $\theta$	1.5°	2.3°	3.0°	4.5°	6.0°

**For SL 203-97 (Tbl. 4.3.1 from SL 203-97)**

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration $K_h$	0.10	0.15	0.20	0.30	0.40

## Water Influence according to Chinese Standards

Seismic water influence can be divided into two parts - influence on seismic earth pressure and dynamic water pressure. Similar to [water influence according to Mononobe-Okabe](#) or [Arrango theory](#), water influence according to Chinese standards also has two types water influence - confined water and unconfined water. The main difference between Mononobe-Okabe theory and Chinese standards is that Chinese standards reduce the water influence using comprehensive influence factor  $C_z$ .

### Confined water

This type is used in soils with lower permeability - app. below the value of  $k = 1 \cdot 10^{-3} \text{ cm/s}$ . When confined water is chosen, dynamic water pressure is not considered. The only difference from soils without water is that seismic angle used in calculation of seismic earth pressure is replaced by seismic angle below water  $\theta'$ .

Value of seismic angle below water  $\theta'$  is determined by the following two options:

**1. By seismic fortification intensity - value of  $\theta'$  is provided by the following tables:**

**For JTJ 004-89 (Tbl. 3.1.6 from JTJ 004-89)**

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration $K_h$	0.10g	0.15g	0.20g	0.30g	0.40g
Seismic angle $\theta$	2.5°	4.5°	5.0°	9.0°	10.0°

**For JTS 146-2012 (Tbl. 5.3.1 from JTS 146-2012)**

Seismic fortification intensity	7 degree		8 degree		9 degree
Coefficient of horizontal seismic acceleration $K_h$	0.10g	0.15g	0.20g	0.30g	0.40g
Seismic angle $\theta'$	3.0°	4.5°	6.0°	9.0°	12.0°

**2. Input seismic angle - value of  $\theta'$  is input by users. When this option is chosen, the default value of  $\theta'$  is provided by:**

$$\theta' = \arctan \left( \frac{\gamma_{sat} C_z C_i K_h}{\gamma_{sat} (1 - C_o C_z C_i K_v)} \right)$$

If you have no idea about how to calculate the value of  $\theta'$ , you can use the default.

**Note:** For [SL 203-97](#), there are no additional options for confined water.  $\theta'$  is calculated automatically according to the above formula.

### Unconfined water

This type is used in soils with higher permeability - app. above the value of  $k > 1 \cdot 10^{-1} \text{ cm/s}$ . When unconfined water is chosen, both influence on earth pressure and dynamic water pressure is considered.

Value of seismic angle below water  $\theta'$  is determined by the following three options:

1. By seismic fortification intensity - same to confined water.
2. Input seismic angle - same to confined water.
3. Input specific gravity of soil particles - value of  $\theta'$  is provided by:

$$\theta' = \arctan\left(\frac{K_{hs}^+}{1 - C_o C_z C_i K_v}\right)$$

where:

$$K_{hs}^+ = \frac{\gamma_d}{\gamma_{su}} C_z C_i K_h = \frac{G_s}{G_s - 1} C_z C_i K_h$$

**Note:** For [SL 203-97](#), there are no additional options for unconfined water.  $\theta'$  is calculated automatically according to the above formula.

**Dynamic water pressure** is calculated according to the standard chosen.

**For [JTJ 004-89](#) (Art. 4.2.11 from [JTJ 004-89](#)):**

$$E_w = 0.24 C_i K_h \gamma_w d^2$$

where:  $E_w$  - over all dynamic water pressure acting on the structure [kN]

$C_i$  - importance coefficient for seismic design

$\gamma_w$  - unit weight of water [kN/m<sup>3</sup>]

$d$  - depth of water above the heel of the structure [m]

Distribution of dynamic water pressure is constant along the structure.

**For [JTS 146-2012](#) (Art. 5.4.1 from [JTS 146-2012](#)):**

$$p_z = \frac{7}{8} \eta C_i C_z K_h \gamma_w d^{\frac{1}{2}} Z^{\frac{1}{2}}$$

where:

$$\eta = th \frac{\pi b}{4d}$$

where:  $p_z$  - dynamic water pressure at depth  $Z$  [kPa]

$\eta$  - reduction factor, for walls, it equals to 1.0

$Z$  - distance between calculation point and water table [m]

- $d$  - depth of water above the heel of the structure [m]  
 $b$  - width of water table [m]

**For SL 203-97 (Art. 6.1.9 from SL 203-97):**

$$p_z = C_z C_i K_h \gamma_w d \psi(Z)$$

- where:  $p_z$  - dynamic water pressure at depth  $Z$  [kPa]  
 $d$  - depth of water above the heel of the structure [m]  
 $\psi(Z)$  - distribution coefficient of dynamic water pressure at depth  $Z$

**Value of  $\psi(Z)$  is provide by the following table (Tbl. 6.1.9 from SL 203-97):**

$Z/d$	$\psi(Z)$	$Z/d$	$\psi(Z)$
0.0	0.00	0.6	0.76
0.1	0.43	0.7	0.75
0.2	0.58	0.8	0.71
0.3	0.68	0.9	0.68
0.4	0.74	1.0	0.67
0.5	0.76	-	-

## Importance Coefficient for Seismic Design $C_i$

**Values of importance coefficient for seismic design  $C_i$  (Tbl. 1.0.4 from JTJ 004-89):**

The importance of the motorway	Important coefficient for seismic design $C_i$
Important Highway and A class motorway	1.7
Highway and A class motorway or important B class motorway	1.3
B class motorway or important C class motorway	1.0
C class motorway or important D class motorway	0.6

**Values of importance coefficient for seismic design  $C_i$  (Tbl. 1.0.4 from JTG TB02-01-2008):**

Importance of the bridge	E1 seismic effect	E2 seismic effect
A class	1.0	1.7
B class	0.43 (0.5)	1.3 (1.7)
C class	0.34	1.0
D class	0.23	-

## Adjusting Coefficient for Seismic Bearing Capacity $\xi_a$

Values of adjusting coefficient for seismic bearing capacity  $\xi_a$  (Tbl. 4.2.3 from GB 50011-2010):

Name and property of the subsoil	$\xi_a$
Rock, dense gravelly soil, dense gravel and coarse and medium coarse sand, clay and silt whose $f_{ak} \geq 300 \text{ kPa}$	1.5
Medium and moderate dense gravelly soil, medium dense and moderate dense gravel and coarse and medium coarse sand, dense and medium dense fine and silty sand, clay, silt and firm loess whose $300 \text{ kPa} \leq f_{ak} < 300 \text{ kPa}$	1.3
Moderate dense fine and silty sand, clay, silt and plastic loess whose $100 \text{ kPa} \leq f_{ak} < 150 \text{ kPa}$	1.1
Mud, muddy soil, loose sand, miscellaneous fill, newly accumulated loess and soft loess	1.0

**Note:** In JTJ 004-89, it's table 2.2.1 which is similar to the above table.

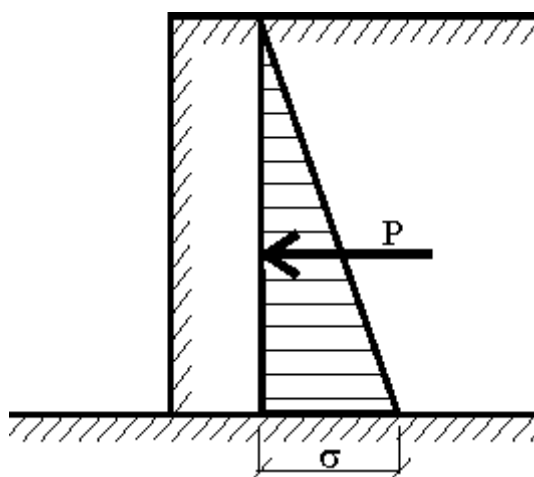
Values of adjusting coefficient for seismic bearing capacity  $\xi_a$  (Tbl. 5.5.1 from JTS 146-2012):

Subsoil	$\xi_a$
Loose sand, not in liquidation status	1.0
Normal sand soil, not in liquidation status	1.3
Dense gravelly soil and bedrock	1.5

## Influence of Friction between Soil and back of the Structure

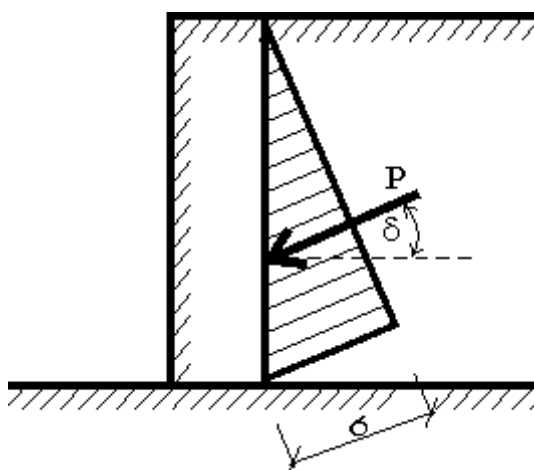
The magnitude of active or passive earth pressure, respectively, depends not only on the selected solution theory but also on friction between the soil and the back of wall and by the adhesion of soil to the structure face represented by the angle  $\delta$ . If  $\delta = 0$  then the pressure  $\sigma$  acts in the direction normal to the back of wall and the resultant of earth pressure  $P$  is also directed in normal to the back of wall - see figure:





*Distribution of earth pressure along structure for  $\delta = 0$*

Providing the friction between the soil and the back of wall is considered in the analysis of earth pressures, the earth pressure  $\sigma$  and also its resultant  $P$  are inclined from the back of wall by the angle  $\delta$ . Orientation of friction angles  $\delta$  from normal to the back of wall must be introduced in accord with the mutual movement of structure and soil. With increasing value of  $\delta$  the value of active earth pressure decreases, i.e. the resultant force of active earth pressure deviates from the normal direction - see figure:



*Distribution of earth pressure along structure for  $\delta \neq 0$*

The magnitude  $\delta$  can be usually found in the range of  $\delta \leq 1/3\varphi$  to  $\delta = 2/3\varphi$ . The values of orientation of the friction angle  $\delta$  between the soil and the structure are stored in [table of values](#) of  $\delta$  for various interfaces and in table of recommended values for  $|\delta|/\varphi$ . The value of  $\delta \leq 1/3\varphi$  can be used if assuming smooth treatment of the back of sheeting structure (foil and coating against ground water). For untreated face it is not reasonable to exceed the value of  $\delta = 2/3\varphi$ . When selecting the value of  $\delta$  it is necessary to reflect also other conditions, particularly the force equation of equilibrium in the vertical direction. One should decide whether the structure is capable of transmitting the vertical surcharge due to friction on its back without excessive vertical deformation. Otherwise it is necessary to reduce the value of  $\delta$ , since only partial mobilization of friction on the back of wall may occur. In case of uncertainty it is always safer to assume smaller value of  $\delta$ .

## Table of Ultimate Friction Factors for Dissimilar Materials

Values of the angle  $\delta$  for different interfaces (according to the NAVFAC standards)

Interface material	Friction factor $tg(\delta)$	Friction angle $\delta$ [°]
<b>Mass concrete on the following foundation materials:</b>		
Clean sound rock	0.70	35
Clean gravel, gravel-sand mixtures, coarse sand	0.55 - 0.6	29 - 31
Clean fine to medium sand, silty medium to coarse sand, silty or clayey gravel	0.45 - 0.55	
Clean fine sand, silty or clayey fine to medium sand	0.35 - 0,45	19 - 24
Fine sandy silt, nonplastic silt	0.30 - 0.30	17 - 19
Very stiff and hard residual or preconsolidated clay	0.40 - 0.50	22 - 26
Medium stiff and stiff clay and silty clay	0.30 - 0.35	17 - 19
<b>Steel sheet piles against the following soils:</b>		
Clean gravel, gravel-sand mixtures, well-graded rock fill with spalls	0.40	22
Clean sand, silty sand-gravel mixture, single size hard rock fill	0.30	17
Silty sand, gravel or sand mixed with silt or clay	0.25	14
Fine sandy silt, nonplastic silt	0.20	11
<b>Formed concrete or concrete sheet piling against the following soils:</b>		
Clean gravel, gravel-sand mixture, well-graded rock fill with spalls	0.40 - 0.50	22 - 26
Clean sand, silty sand-gravel mixture, single size hard rock fill	0.30 - 0.40	17 - 22
Silty sand, gravel or sand mixed with silt or clay	0.30	17
Fine sandy silt, nonplastic silt	0.25	14
<b>Various structural materials:</b>		
Dressed soft rock on dressed soft rock	0.70	35

Dressed hard rock on dressed soft rock	0.65	33
Dressed hard rock on dressed hard rock	0.55	29
Masonry on wood (Gross grain)	0.50	26
Steel on steel at sheet pile interlocks	0.30	17

## Adhesion of Soil

When performing the analysis in the **total stress** state for **active** or **passive earth pressure** it is necessary to consider the total (undrained) shear strength of soil  $c_u$  and the adhesion  $a$  of soil to the structure face. The value of adhesion  $a$  is usually considered as a fraction of the soil cohesion  $c$ . The typical values of  $a$  for a given range of the cohesion  $c$  are listed in the following table.

### Common values of the adhesion of soil $a$

Soil	Cohesion $c$ [kPa]	Adhesion $a$ [kPa]
Soft and very soft cohesive soil	0 - 12	0 - 12
Cohesive soil with medium consistency	12 - 24	12 - 24
Stiff cohesive soil	24 - 48	24 - 36
Hard cohesive soil	48 - 96	36 - 46

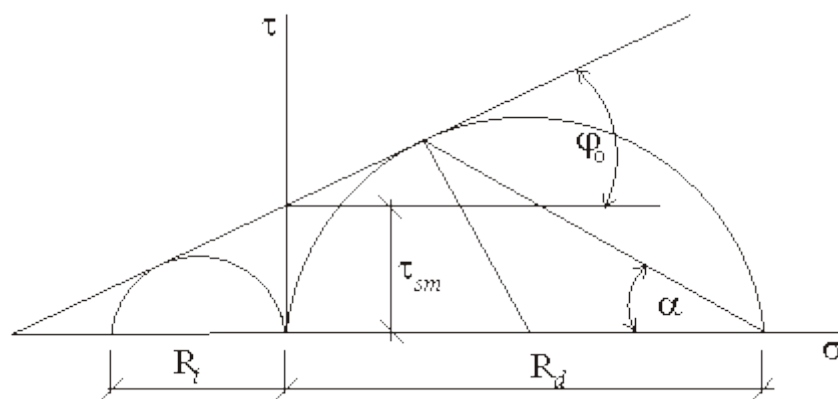
## Parameters of Rocks

Rock parameters of orientation with respect to strength of rock in pure compression

Compressive strength of rock	Strength parameter of rock after Hoek	GSI [-]	Cohesion of rock	Angle of internal friction of rock
$\sigma_{ci}$ [MPa]	$m_i$ [-]	[-]	$c$ [kPa]	$\varphi$ [°]
150	25	75	7000 - 13000	46 - 68
80	12	50	3000 - 4000	30 - 65
50	16	75	2000 - 4000	40 - 60
30	15	65	1000 - 2000	40 - 60
20	8	30	400 - 600	20 - 44
15	10	24	300 - 500	24 - 38
5	10	20	90 - 100	23 - 28

Unlike soils (both cohesive and cohesionless) the magnitude of the angle of internal friction (sometimes refer to as the angle of shear strength) varies and depends on the current state of stress in the rock body. Graphically it is represented by the angle of the tangent to the envelope of Mohr circles constructed for the ultimate stress state. The value of  $\varphi$  gradually decreases with the increasing value of stress  $\sigma$ . If the elastic regime is exceeded (onset of plastic deformation) we set  $\varphi = 0$ . As a representative value of the angle of internal friction  $\varphi$  we denote the value  $\varphi_0$  associated with the stress  $\sigma = 0$ . In practical applications the part of the Mohr envelope between tensile  $R_t$  and compressive  $R_d$  circles is usually replaced by the tangent to both circles (see Fig.) The magnitude of the angle of internal friction then follows from:

$$\sin \varphi_0 = \frac{R_d - R_t}{R_d + R_t}$$



Determination  $\varphi_0$  from Mohr circle

The angle of internal friction can be estimated by measuring angles of slip planes on remaining parts of tested specimens together with the following formula:

$$\alpha = 45^\circ - \frac{\varphi_0}{2}$$

**Some values of orientation:**

weathered sand conglomerate, lowly cracked	35 - 44°
unweathered clay slate, medium cracked	30 - 40°
unweathered tuff, medium cracked	33 - 42°
unweathered diabase	39 - 50°
unweathered phantinite, lowly cracked	45 - 52°

## Analysis of Walls

Verification analysis of walls can be performed using:

- the **theory of limit state** (when performing the analysis according to **EN 1997** or **LRFD** the structure is verified in this particular way)
- the **safety factor**

In addition, the **bearing capacity of foundation soil** is examined for both cases.

Following forces are used in the verification:

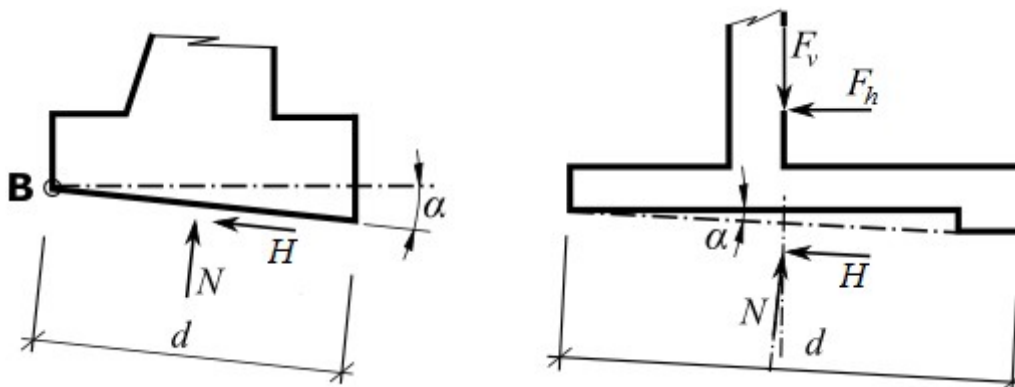
- **weight of wall** - depends on the shape and unit weight of wall (for input use the "**Material**" dialog window) - uplift pressure is introduced for walls found below the groundwater table
- **resistance on front face** - when inputting the resistance on front face the corresponding force acts as the **pressure at rest**, or **passive pressure** or **reduced passive pressure**
- gravity forces of **earth wedges** - an arbitrary number of these forces may occur depending on the shape of structure
- **active earth pressure** or **pressure at rest** acting on the structure - the basic load of structure due earth pressures - depending on the selected option in the frame "**Settings**" the pressure is computed either with or without reduction of input soil parameters
- forces due to **water effects** or pore pressure, respectively
- forces due to **surcharge** - a single force corresponds to each input surcharge. If the magnitude of force due to surcharge is equal to zero (the surcharge has no effect on a structure) then it does not appear in the picture but only in the table listing
- **input forces** - forces entering the analysis are displayed
- **forces due to earthquake** - several forces enter the analysis due to earthquake - increase of earth pressure acting on a structure, reduction of passive pressure on the front face of a structure, or force due to free water behind structure
- **mesh step joints and reinforcements** are displayed and included providing they appear in the analysis
- **base anchorage** of walls

## Evaluation of Forces in the Footing Bottom

After computing **forces acting on the structure** the program determines the overall vertical  $F_v$  and horizontal  $F_h$  forces, computes the forces acting in the footing bottom (normal force  $N$  and shear force  $H$ ):

$$N = F_v \cos \alpha + F_h \sin \alpha$$

$$H = F_v \sin \alpha + F_h \cos \alpha$$



Forces acting in the footing bottom

## Verification - Limit States

Program evaluates normal and shear force in the **footing bottom** and then verifies the wall against overturning and sliding. For walls with a flat footing bottom and specified jump it is possible to **account for the wall jump** either in the form of pressure acting on the front face or by considering a wall with an inclined footing bottom.

### Check for overturning stability:

$$\frac{M_{res}}{\gamma_o} > M_{ovr}$$

- where:
- $M_{ovr}$  - overturning moment
  - $\gamma_o$  - reduction coefficient of overturning
  - $M_{res}$  - resisting moment

### Check for slip:

$$\frac{[(N \tan \varphi_d + c_d(d - 2e) / \mu) + F_{res}]}{\gamma_s} > H$$

- kde:
- $N$  - normal force acting in the footing bottom
  - $\varphi_d$  - design angle of friction structure-soil
  - $c_d$  - design cohesion structure-soil
  - $d$  - width of wall heel
  - $e$  - eccentricity
  - $\gamma_s$  - reduction coefficient of sliding resistance
  - $H$  - shear force acting in the footing bottom
  - $F_{res}$  - resisting force (from georeinforcement and mesh overlap)
  - $\mu$  - reduction coefficient of contact base - soil

where eccentricity  $e$ :

$$e = \frac{M_{ovr} - M_{res} + \frac{Nd}{2}}{N}$$

where:  $M_{ovr}$  - overturning moment  
 $M_{res}$  - resisting moment  
 $N$  - normal force acting in the footing bottom  
 $d$  - width of wall heel

Horizontal components of forces are included in the shear force and overturning moment, vertical components of forces are included in the normal force and resisting moment. The resisting forces and moments also include horizontal forces from georeinforcements and overlapping meshes.

## Verification - Safety Factor

Program evaluates normal and shear force in the footing bottom and then verifies the wall against overturning and sliding. For walls with a flat footing bottom and specified jump it is possible to account for the wall jump either in the form of pressure acting on the front face or by considering a wall with an inclined footing bottom.

**Check for overturning stability:**

$$\frac{M_{res}}{M_{ovr}} > SF_o$$

where:  $M_{ovr}$  - overturning moment  
 $M_{res}$  - resisting moment  
 $SF_o$  - safety factor for overturning

**Check for slip:**

$$\frac{[(N \tan \varphi_d + c_d(d - 2e)/\mu) + F_{res}]}{H} > SF_s$$

kde:  $N$  - normal force acting in the footing bottom  
 $\varphi_d$  - design angle of friction structure-soil  
 $c$  - cohesion structure-soil  
 $d$  - width of wall heel  
 $e$  - eccentricity  
 $H$  - shear force acting in the footing bottom  
 $F_{res}$  - resisting force (from georeinforcement and mesh overlap)  
 $SF_s$  - safety factor for sliding resistance  
 $\mu$  - reduction coefficient of contact base - soil

where eccentricity  $e$ :

$$e = \frac{M_{ovr} - M_{res} + \frac{Nd}{2}}{N}$$

where:

$M_{ovr}$	-	overturning moment
$M_{res}$	-	resisting moment
$N$	-	normal force acting in the footing bottom
$d$	-	width of wall heel

Horizontal components of forces are included in the shear force and overturning moment, vertical components of forces are included in the normal force and resisting moment. The resisting forces and moments also include horizontal forces from georeinforcements and overlapping meshes.

## Internal Sliding

This limit state evaluates the possibility of structure to slide along the reinforcement. For the selected reinforcement the program searches for a critical slip surface in the range of  $45 - 90^\circ$  from the end of given reinforcement.

For each slip surfaces the program calculates the shear and resisting forces and performs verification.

### The shear forces include:

- active pressure on a fictitious wall
- forces due to surcharge behind the wall

### The resisting forces include:

- resistance of the wall structure against slip (it is calculated as for the wall dimensioning)
- friction between reinforcement and the sliding block
- forces due to other reinforcements

The resisting force due to friction between reinforcement and the sliding block is given by:

$$F = N \operatorname{tg} \varphi C_{ds}$$

where:

$N$	-	normal force acting on reinforcement (due to self weight of soil and surcharge behind the fictitious wall)
$\varphi$	-	angle of internal friction of soil surrounding the reinforcement
$C_{ds}$	-	coefficient of reduction of friction on reinforcement

The actual verification is then performed based on the input specified in the "Wall analysis" tab, according to the theory of limit states and factor of safety. It has to hold:

$$H_{res} > H_{act} \quad \text{resp.} \quad \frac{H_{res}}{H_{act}} > SF_{sr}$$

where:

$H_{res}$	-	resisting force
$H_{act}$	-	active force

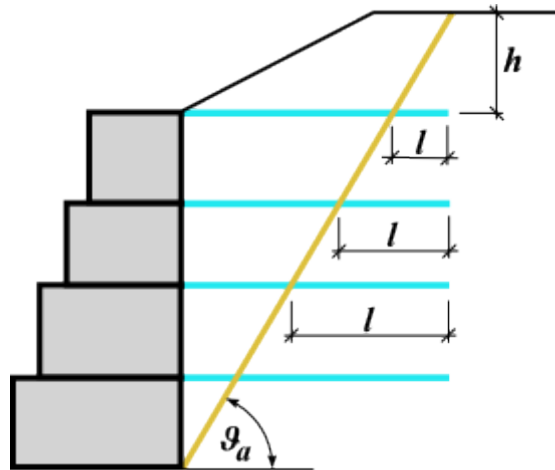


$SF_{sr}$  - safety factor for sliding along geo-reinforcement

## Reinforcements

Reinforcements or overhangs of mesh behind the wall, respectively, may considerably increase the wall stability. The basic parameter of reinforcement is the **tensile strength**  $R_t$ . A design value of this parameter is used in all programs (except for the Redi-Rock wall program), i.e. the tensile strength of reinforcement reduced by coefficients taking into account the effect of durability, creep, environment chemistry and installation damage. The force transmitted by reinforcement **can never exceed the assigned tensile strength**  $R_t$  (a default value of 40 kN/m is used for gabions).

The second characteristic is the **pull-out strength**  $T_p$ . This parameter determines the anchoring length, i.e. the required length of reinforcement in the soil, for which the reinforcement is fully stressed attaining the value  $R_t$ . Since the realistic values of the pull-out strength are difficult to determine, the program offers three options for their calculation, respectively for the calculation of the force  $F$  transmitted by the reinforcement.



*Length of mesh step joint or reinforcement behind blocks, respectively*

### Calculate pull-out force

The pull-out force  $F$  is given by:

$$F = 2 \cdot \sigma \cdot \tan \varphi \cdot C \cdot l$$

where:

- $\sigma$  - normal stress due to self weight at the intersection of mesh and slip surface
- $\varphi$  - angle of internal friction of soil
- $C$  - coefficient of interaction (0,8 by default)
- $l$  - length of mesh step joint behind the slip surface into the soil body

Computation of the angle  $\alpha_a$  is described in chapter [earth wedge](#).

### Input reinforcement anchor length $l_k$

An anchoring length  $l_k$  is specified. This parameter is determined by the shear strength developed between the mesh and the soil gradually increasing from zero to its limit value (measured from the end of reinforcement fixed in soil).

$$F = \frac{l}{l_k} \cdot R_t$$

where:  $l$  - length of mesh step joint behind the slip surface into the soil body  
 $l_k$  - anchoring length of reinforcement  
 $R_t$  - tensile strength

### Input mesh pull-out resistance $T_p$

The pull-out force  $F$  is given by:

$$F = T_p \cdot l$$

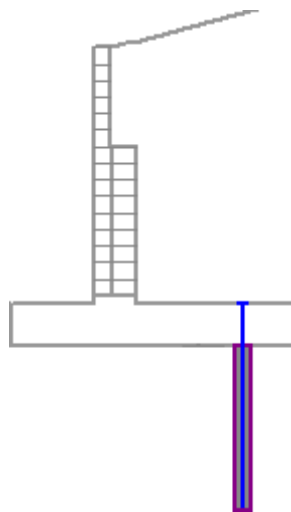
where:  $l$  - length of mesh step joint behind the slip surface into the soil body  
 $T_p$  - pull-out resistance of mesh

## Base Anchorage

An anchorage of wall footing can be specified in program "**Cantilever Wall**". It is necessary to specify an anchor location, dimensions of a drill hole, and spacing of anchors.

Two limit states of bearing capacity are defined for an anchor:

- **bearing capacity against pulling-out**  $R_e$  [kN/m]
- **strength of anchor**  $R_t$  [kN]



*Base anchorage*

Bearing capacities can be either input or computed from the input values using the following expressions:

$$T_p = \frac{\pi d a}{SF_s}$$

where:  $T_p$  - pull-out resistance  
 $d$  - drill hole diameter

- $a$  - ultimate bond  
 $SF_e$  - safety factor against pulling-out

$$R_t = \frac{\pi d_s^2 f_y}{4 SF_t}$$

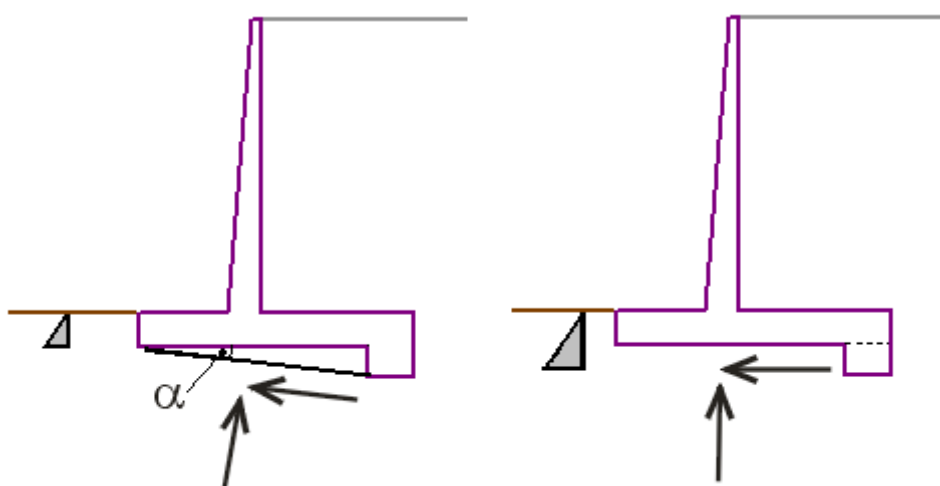
- where:  $R_t$  - strength of anchor  
 $d_s$  - anchor diameter  
 $f_y$  - yield strength of anchor  
 $SF_t$  - safety factor against pulling-apart

### Approximate values of bearing capacity against pulling-out

Material	Ultimate bond [N/mm <sup>2</sup> ]	Ultimate strength for specified hole diameter [kN/m]				
		65 mm	75 mm	90 mm	100 mm	150 mm
Soft shale	0.21 - 0,83	42 - 169	49 - 195	59 - 234	65 - 260	98 - 391
Sandstone	0.83 - 1,73	169 - 350	195 - 407	234 - 486	260 - 543	391 - 562
Slate, Hard Shale	0.86 - 1,38	175 - 281	202 - 325	243 - 390	270 - 433	405 - 562
Soft Limestone	1.00 - 1.52	204 - 310	235 - 358	282 - 429	314 - 477	471 - 562
Granite, Basalt	1.72 - 3.10	351 - 562	405 - 562	486 - 562	540 - 562	562 - 562
Concrete	1.38 - 2.76	281 - 562	325 - 562	390 - 562	433 - 562	562 - 562

## Accounting for Wall Jump

Two options are available to account for the foundation wall jump in the analysis as shown in the figure (programs "**Cantilever Wall**" and "**Masonry Wall**").

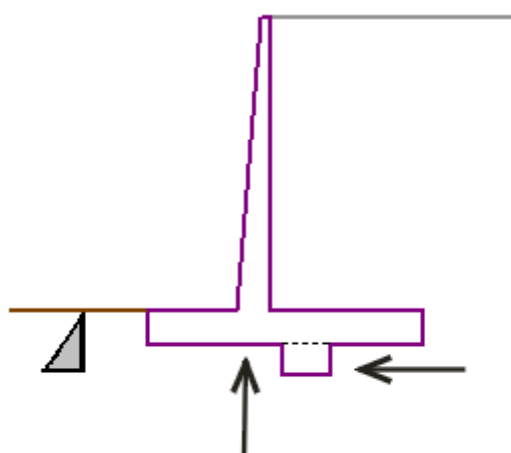


*Options to account for wall jump*

If the jump is assumed as an **inclined footing bottom**, then a new shape of the footing bottom is considered and the structure front face resistance is included only up to a depth of the wall front face.

If the jump influence is considered as a **front face resistance** the analysis assumes a flat footing bottom (as if there was no jump), but the structure front face resistance included up to a depth of jump. In such a case computation of the structure front face resistance must also be input - otherwise the jump influence is neglected.

The jump introduced below the wall foundation is always considered as a structure front face resistance.



*Assuming wall jump in the middle*

## Dimensioning of Masonry Wall According to AS 3700

Reinforced masonry is verified for load due to bending moment, shear force and combination of compressive normal force and bending moment. When load due normal force is considered, it is necessary to specify also the slenderness ratio  $S_r$ .

### Design for members in compression and bending

$$F_d \leq 0,85 \cdot \phi \cdot k_s (f'_m \cdot A_b + f_{sy} \cdot A_s)$$

$$f'_m = 0,35 \cdot f'_{mb}$$

$$f'_{mb} = 1,3 \cdot \sqrt{f'_{uc}}$$

- where:
- $F_d$  - the design compression force acting on the cross-section
  - $\phi$  - the capacity reduction factor - 0.75
  - $k_s$  - a reduction factor taken as  $1.18 - 0.03 \cdot S_r$  but not greater than 1.0
  - $f'_{uc}$  - the characteristic unconfined compressive strength of masonry
  - $f'_m$  - the characteristic compressive strength of masonry
  - $A_b$  - the bedded area of the masonry cross-section
  - $f_{sy}$  - the design yield strength of reinforcement
  - $A_s$  - the total cross-sectional area of main reinforcement

### Design for members in bending

$$M_d \leq \phi \cdot f_{sy} \cdot A_{sd} \cdot d \left( 1 - \frac{0,6 \cdot f_{sy} \cdot A_{sd}}{1,3 \cdot f'_m \cdot d} \right)$$

$$f'_m = 0,35 \cdot f'_{mb}$$

$$f'_{mb} = 1,3 \cdot \sqrt{f'_{uc}}$$

- where:
- $M_d$  - the design bending moment acting on the cross-section of member
  - $\phi$  - the capacity reduction factor - 0.75
  - $f_{sy}$  - the design yield strength of reinforcement
  - $A_{sd}$  - the portion of the cross-sectional area of the main tensile reinforcement used for design purposes in a reinforced masonry member

$$\frac{0,29 \cdot 1,3 \cdot f'_m \cdot d}{f_{sy}}$$

the lesser of  $f_{sy}$  and  $A_{st}$

- $f'_m$  - the characteristic compressive strength of masonry
- $d$  - the effective depth of the reinforced masonry member
- $f'_{uc}$  - the characteristic unconfined compressive strength of masonry

### Out-of-plane shear in wall

A reinforced wall subject to out-of-plane shear shall be such that:

$$V_d \leq \phi \cdot (f'_{vm} \cdot d + f_{sv} \cdot A_{st})$$

but not more than:

$$4 \cdot \phi \cdot f'_{vm} \cdot d$$

- where:
- $V_d$  - the design shear force acting on the cross-section of the masonry wall
  - $\phi$  - the capacity reduction factor - 0.75

- $f_{vm}$  - the characteristic shear strength of reinforced masonry - 0.35 Mpa  
 $d$  - the effective depth of the reinforced masonry wall  
 $f_{vs}$  - the design shear strength of main reinforcement - 17.5 Mpa  
 $f_{sy}$  - the design yield strength of reinforcement  
 $A_{st}$  - the cross-sectional area of fully anchored longitudinal reinforcement in the tension zone of the cross-section

## Dimensioning of Masonry Wall According to EN 1996-1-1

The reinforced masonry is verified for the load caused by the combination of the compressive normal force and the bending moment and for the load due to the shear force.

### Verification for pressure and bending

Analysis assumptions (Chapter 6.6):

- plane cross-sections remain plane
- the strain of steel is the same as the strain of the attached masonry
- the tensile strength of masonry is assumed equal to zero
- the limit strain of masonry in compression is 0.0035
- the limit strain of steel in tension is 0.01
- variation of stress as a function of strain of masonry is assumed parabolic-rectangular
- variation of stress as a function of strain of steel is assumed bounded by a horizontal upper branch
- the properties of filling concrete are considered the same as the properties of masonry (it is necessary to use the worse of the two materials)
- design strength of masonry (concrete) is provided by:

$$f_d = \frac{f_k}{\gamma_M}$$

where:  $f_k$  - characteristic strength of masonry (concrete)  
 $\gamma_M$  - 1.8

- If the slenderness ratio given by the ratio of the height and the width of the wall is greater than 12, the effect of the II-nd order theory is considered by including an additional design bending moment given by:

$$M_{ad} = \frac{N_{Ed} h_{ef}^2}{2000t}$$

where:  $N_{Ed}$  - design value of the normal force  
 $h_{ef}$  - buckling height of wall  
 $t$  - wall thickness

- If the slenderness ratio is greater than 27, then it is not possible to perform the analysis

and it is necessary by changing geometry to obtain more favorable slenderness ratio.

### Verification for shear

Chapter 6.7, Appendix J

$$V_{Ed} \leq f_{vd} t l$$

$$f_{vd} = \frac{\text{Min}(f_{vk} + 17.5\rho, 0.7)}{\gamma_M}$$

$$\rho = \frac{A_s}{bd}$$

where:	$V_{Ed}$	- design value of the shear force
	$f_{vd}$	- design value of the shear strength of masonry (concrete)
	$\rho$	- longitudinal reinforcement ratio
	$t$	- wall thickness
	$l$	- wall length - 1 running meter

## Dimensioning of Gravity Wall - Masonry According to EN 1996-1-1

The masonry is verified for the load caused by the combination of the compressive normal force and the bending moment and for the load due to the shear force.

### Verification of compression bearing capacity

Chapter 6.1.2.1

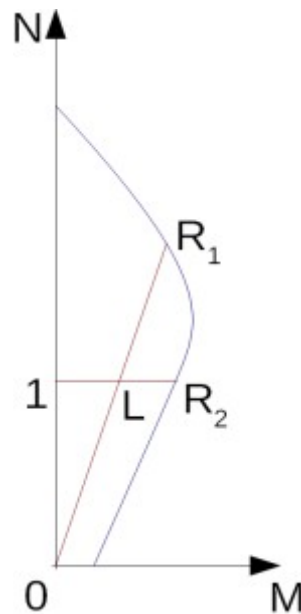
$$N_{Ed} \leq N_{Rd}$$

$$N_{Rd} = A_c f_k / \gamma_M$$

$$A_c = bh \left(1 - 2 \frac{e}{h}\right)$$

$$e = \frac{M_{Ed}}{N_{Ed}}$$

where:	$N_{Ed}$	- design value of normal force
	$N_{Rd}$	- compression bearing capacity
	$A_c$	- compressed area of cross section
	$f_k$	- characteristic value of compressive strength of masonry
	$\gamma_M$	- partial factor of masonry
	$b$	- width of cross section
	$h$	- depth of cross section
	$e$	- excentricity of normal force
	$M_{Ed}$	- design value of bending moment



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|OL|/|OR_1|$  or  $|IL|/|IR_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed excentricity and  $R_2$  is strength with prescribed normal force.

### Verification of shear bearing capacity

Chapter 6.2

$$V_{Ed} \leq V_{Rd}$$

$$V_{Rd} = A_c f_{vk} / \gamma_M$$

$$f_{vk} = \text{Min} \left( f_{vko} + 0.4 \frac{N_{Ed}}{A_c}; 0.065 f_b \right)$$

- where:
- $V_{Ed}$  - design value of shear force
  - $V_{Rd}$  - shear bearing capacity
  - $f_{vk}$  - characteristic value of shear strength of masonry
  - $f_{vko}$  - charakteristic value of original shear strength of masonry
  - $f_b$  - compressive strength of masonry unit

## Dimensioning of Gravity Wall - Masonry According to GB 50003-2011

The masonry is verified for the load caused by the combination of the compressive normal force and the bending moment and for the load due to the shear force.

### Verification of compression bearing capacity

Non-seismic design situation (Art 5.1.1):

$$\gamma_0 N \leq \varphi f A$$



Seismic design situation (Art 10.1):

$$N \leq \varphi f A / \gamma_{RE}$$

- where:
- $\gamma_0$  - coefficient of importance of structure
  - $N$  - design value of normal force
  - $f$  - design value of compressive strength of masonry
  - $A$  - area of cross section
  - $\varphi$  - influence factor due to eccentricity of normal force and depth-thickness ratio of structure
  - $\gamma_{RE}$  - seismic adjusting coefficient for compressive strength of masonry

$\varphi$  is provided by:

When  $\beta \leq 3$  (Art D.0.1-1)

$$\varphi = \frac{1}{1 + 12 \left(\frac{e}{B}\right)^2}$$

When  $\beta > 3$  (Art D.0.1-2, D.0.1-3)

$$\varphi = \frac{1}{1 + 12 \left[ \frac{e}{B} + \sqrt{\frac{1}{12} \left( \frac{1}{\varphi_0} - 1 \right)} \right]^2}$$

$$\varphi_0 = \frac{1}{1 + \alpha \beta^2}$$

- where:
- $e$  - eccentricity of normal force acting on the cross section
  - $B$  - depth of the cross section
  - $\varphi_0$  - stability coefficient of structure loaded with axial pressure
  - $\alpha$  - coefficient due to strength grade of mortar
  - $\beta$  - depth-thickness ratio of structure

$\beta$  is provided by:

$$\beta = \gamma_{\beta} \frac{2H}{B}$$

$$\varphi_0 = \frac{1}{1 + \alpha \beta^2}$$

- where:
- $\gamma_{\beta}$  - adjusting coefficient of depth-thickness ratio based on the type of masonry material
  - $H$  - height of the structure above cross section

### Verification of shear bearing capacity

Non-seismic design situation (Art. 5.5.1-1):

$$\gamma_0 V \leq (f_v + \alpha \mu \sigma_0) A$$

Seismic design situation (Art. 10.1):

$$V \leq (f_v + \alpha \mu \sigma_0) A / \gamma_{RE}$$

When  $\gamma_G \leq 1.2$  (Art 5.5.1-2):

$$\mu = 0.26 - 0.082 \frac{\text{Min}(0.8f; \sigma_0)}{f}$$

When  $\gamma_G \geq 1.35$  (Art 5.5.1-3):

$$\mu = 0.23 - 0.065 \frac{\text{Min}(0.8f; \sigma_0)}{f}$$

Intermediate values are interpolated.

where:	$\gamma_0$	- coefficient of importance of structure
	$V$	- design value of shear force
	$f_v$	- design value of shear strength of masonry
	$A$	- area of cross section
	$\sigma_0$	- average value of normal stress on cross section
	$f$	- design value of compressive strength of masonry
	$\gamma_G$	- partial factor for permanent actions
	$\alpha$	- correction factor; when $\gamma_G \leq 1.2$ : $\alpha = 0.64$ ; $\gamma_G \geq 1.35$ : $\alpha = 0.66$ Intermediate values are interpolated
	$\mu$	- influence factor for shear-compression load
	$\gamma_{RE}$	- seismic adjusting coefficient for shear strength of masonry

## Bearing Capacity of Foundation Soil

Verification analysis of the bearing capacity of foundation soil takes into account forces obtained from all already performed verifications of the overall stability of structure (the theory of **limit states**, **safety factor**). To that end, the following relationships are used:

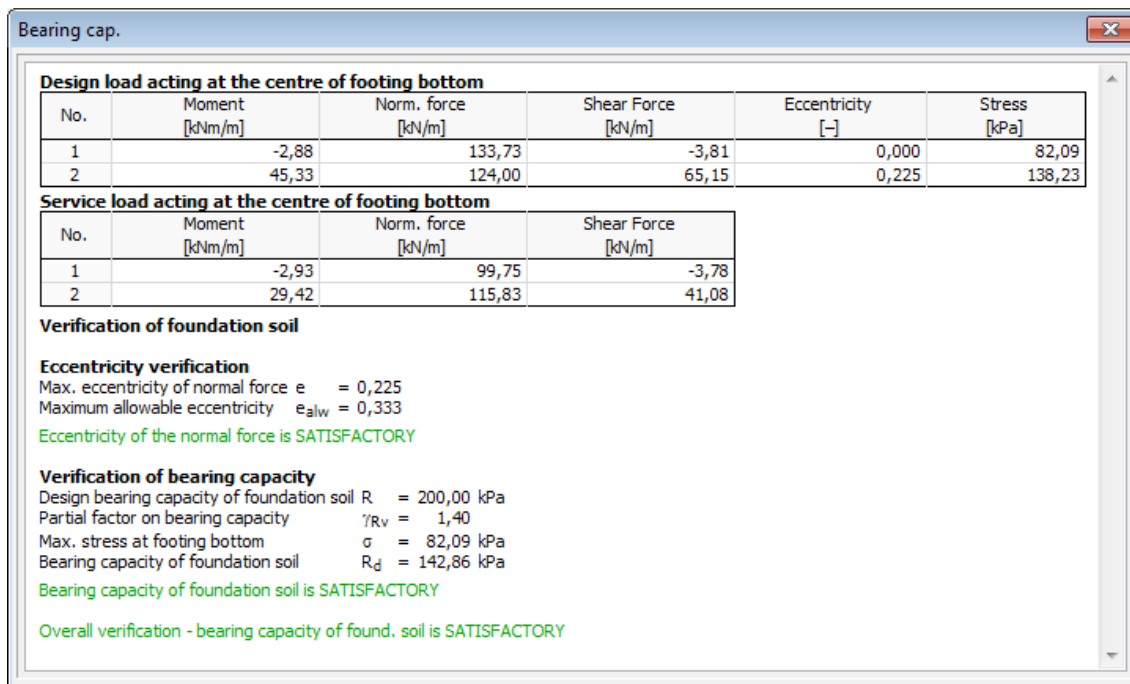
$$\sigma = \frac{N}{d - 2e} < R_d$$

$$e \leq e_{alw}$$

kde:	$N$	- normal force acting in the footing bottom
	$d$	- width of wall heel
	$e$	max. <b>eccentricity</b> of normal force
	$R_d$	- bearing capacity of foundation soil
	$e_{alw}$	- allowable eccentricity (this value is defined in the frame " <b>Settings</b> " in tab " <b>Wall analysis</b> ")

For calculation of bearing capacity of foundation soil (in the case of assuming **shallow**

**foundation** under the wall) program allows to calculate **design or service load**, that acts at the center of the footing bottom. When transferring data and results in the program "**Spread Footing**" it is possible to calculate **settlement** and **rotation of foundation** correctly. For assuming a **pile foundation** in the frame "**Foundation**" it is possible to view internal forces in the heads of piles (for one series of piles), respectively at the center of the footing bottom (for planar pile grid).

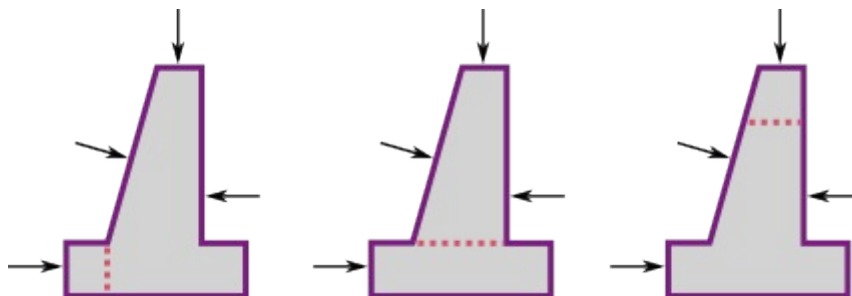


Dialog window "Bearing capacity"

## Wall Dimensioning

After **computing forces acting on the structure** the program determines all internal forces in the verified cross-section (normal force  $N$ , shear force  $Q$  and moment  $M$ ) and then **verifies the cross-section bearing capacity** employing one of the setting selected in the "**Wall analysis**" tab.

Only the forces found above the verified joint (see figure) are assumed for dimensioning. These forces are not multiplied by any design coefficients.



Forces entering the analysis

**The front jump of wall** as well as the back jump of wall is verified against the load caused by the bending moment and shear force. Stress in the footing bottom can be assumed either **constant** (CSN) or **linear** (EC).

Assuming **linear variation of stress** in the footing bottom the distribution of stress is provided by:

$$\sigma_1 = \frac{N}{d^2} \cdot \left( 4 \cdot d - 6 \cdot \left( \frac{d}{2} - e \right) \right)$$

$$\sigma_2 = \frac{N}{d^2} \cdot \left( -2 \cdot d + 6 \cdot \left( \frac{d}{2} - e \right) \right)$$

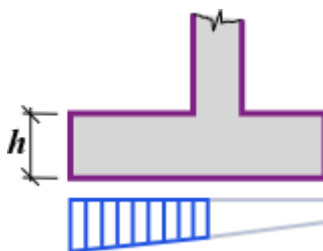
or when excluding tension:

$$\sigma = \frac{2 \cdot N}{3 \cdot \left( \frac{d}{2} - e \right)}$$

where:

- $e$  - eccentricity of normal force  $N$
- $d$  - width of wall foundation
- $N$  - normal force acting in the footing bottom (see verification according to **limit states** or **factor of safety**, respectively)

Bending moment and shear force are determined as reaction developed on the cantilever beam as shown in figure:



*Internal forces acting on wall jump*

**Verification of the back jump** of wall (top tensile reinforcement in the wall jump, respectively) is performed only in some countries and usually is not required. The programs "**Cantilever wall**" and "**Reinforced wall**" allow in version 5.5 for designing the reinforcement in the back jump of wall. The cross-section is then assumed to be loaded by the self weight of structure, earth wedge, surcharge, anchorage force and the force associated with contact pressure in the soil. Forces due to pressure are accounted for only if having a negative impact. Forces introduced by the user are not reflected at all.

The cross-section is checked against the load caused by the bending moment and shear force.

## Internal Stability of a Gabion

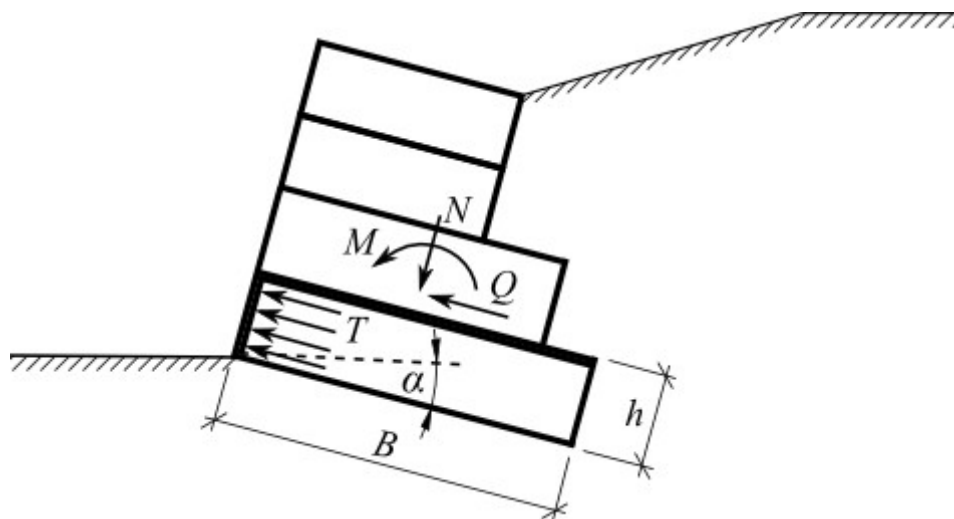
The internal stability of gabion wall can be examined with the help of:

- **the theory of limit states**
- **factor of safety**

Verification of joints between individual blocks is performed in the "**Dimensioning**" frame. The structure above the block is loaded by **active pressure** and corresponding forces are determined in the same way as for the **verification of the entire wall**. A loose filling is used in the analysis - not hand-placed rockfill - but its effect can be simulated using a very high angle

of internal friction. It can be assumed that after some time due to action of filling aggregate the stress in meshes will drop down. Individual sections of the gabion wall are checked for the maximum normal and shear stress. With the help of these variables it is possible to modify the slope of structure face by creating terraces or by increasing the slope of face of wall  $\alpha$ .

**Assuming load applied to the bottom block** is schematically represented as:



*Load on the bottom block*

Normal stress in the center of the bottom block is given by:

$$\sigma = \frac{2.N}{B-2.e} + \frac{\gamma.h.\cos \alpha}{2}$$

$$e = \frac{M}{N}$$

where:

- $N$  - normal force acting on the bottom block
- $B$  - width of upper block
- $e$  - eccentricity
- $M$  - moment acting on the bottom block
- $h$  - height of bottom block
- $\gamma$  - unit weight of the bottom block material
- $\alpha$  - gabion slope

**Pressure acting on the wall of the bottom block** is determined as an increased active pressure:

$$T = 0,5.T_r + 0,5.T_a$$

$$T_r = \sigma.T_r$$

$$T_a = \sigma.K_a - 2.c_d.\sqrt{K_a}$$

$$K_r = 1 - \sin \varphi_d$$

$$K_a = \operatorname{tg}^2 \left( 45 - \frac{\varphi_d}{2} \right)$$

where:

- $\varphi_d$  - design angle of internal friction of the bottom block material
- $c_d$  - design cohesion of the bottom block material
- $\gamma$  - unit weight of material of the bottom block
- $h$  - height of bottom block
- $B$  - width of upper block
- $\alpha$  - gabion slope
- $T$  - average value of pressure acting on face of the bottom block
- $\sigma$  - maximal normal stress acting on the bottom block

**Widths of meshes of the bottom block** per one running meter of the gabion wall are:

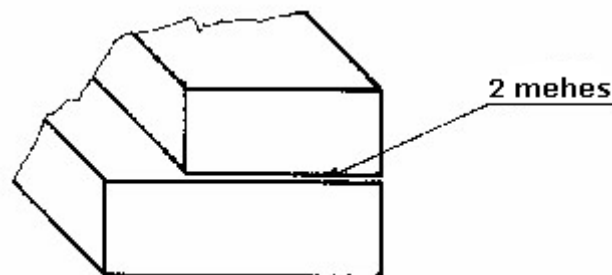
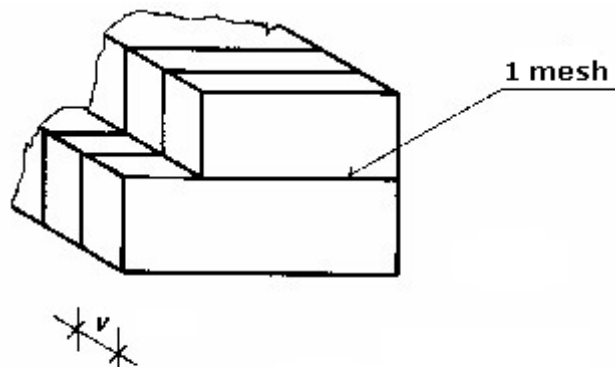
$$D_{upp} = 1$$

$$D_{total} = \frac{h}{v} + 1$$

where:

- $D_{upp}$  - width of upper mesh between blocks loaded in tension
- $D_{total}$  - overall width of meshes loaded in compression  $T$
- $v$  - spacing of vertical meshes
- $h$  - height of bottom block

The program allows for analysis of gabions with both simple and double mesh placed between blocks. For double meshes the input tensile strength of mesh (frame "Material" - the "**Edit material**" dialog window) should be twice as large as the value assumed for simple meshes.



*Geometry of gabions*

## Internal Stability of a Gabion Wall - Safety Factor

The following cases are assumed when examining the internal stability of the gabion wall using the concept of factor of safety:

### 1) Check for overturning stability:

$$\frac{M_{res}}{M_{ovr}} > SF_o$$

where:

$M_{ovr}$	-	overturning moment
$M_{res}$	-	resisting moment
$SF_o$	-	safety factor for overturning

### 2) Check for slip:

$$\frac{N \tan \varphi_d + c_d B}{Q} > SF_s$$

where:

$N$	-	normal force acting on the upper joint of the bottom block
$\varphi_d$	-	design angle of internal friction of the bottom block material
$B$	-	width of upper block
$c$	-	cohesion of the bottom block material
$Q$	-	shear force
$SF_s$	-	safety factor for sliding resistance

### 3) Check for bearing capacity with respect to the lateral pressure:

$$\frac{S_u}{S} > SF_n$$

$$S = \frac{Tbh}{D_{total}}$$

where:

$T$	-	average value of pressure acting on face of the bottom block
$S$	-	force per one running meter of the joint
$S_u$	-	joint bearing capacity (for input use the frame "Material")
$SF_n$	-	safety factor mesh strength (for input use the "Wall analysis" tab - default value is 1.5)
$b$	-	width = 1 m of structure width
$h$	-	height of the block
$D_{total}$	-	overall width of meshes loaded in compression $T$

### 4) Check for bearing capacity of joint between blocks:

$$\frac{N_u}{N} > SF_n$$

$$N_d = \frac{T b h}{D_{total}} + \frac{\max(0, Q - Q_{tr})}{D_{upp}}$$

$$Q_{tr} = \frac{N \tan \varphi_d + c_d B}{\gamma_f}$$

- where:
- $N_d$  - tensile force per one running meter of the upper joint
  - $N_u$  - strength of mesh (for input use the frame "Material")
  - $SF_n$  - safety factor mesh strength (for input use the "Wall analysis" tab - default value is 1.5)
  - $Q_{tr}$  - shear force transmitted by friction and cohesion between blocks
  - $\gamma_t$  - reduction coefficient of friction between blocks (for input use the "Wall analysis" tab)
  - $h$  - height of the block
  - $D_{total}$  - overall width of meshes loaded in compression  $T$
  - $D_{upp}$  - width of upper mesh between blocks loaded in tension

## Internal stability of a Gabion Wall - Limit States

Reduced parameters of the gabion material, which depend on the coefficients set in the "Wall analysis" tab, are used in the verification analysis.

### 1) Check for overturning stability:

$$M_{ovr} < M_{res}$$

- where:
- $M_{ovr}$  - overturning moment
  - $M_{res}$  - resisting moment

### 2) Check for slip:

$$N \tan \varphi_d + c_d B > Q$$

- where:
- $N$  - normal force acting on the upper joint of the bottom block
  - $\varphi_d$  - design angle of internal friction of the bottom block material
  - $B$  - width of upper block
  - $c_d$  - design cohesion of the bottom block material
  - $Q$  - shear force

### 3) Check for bearing capacity with respect to the lateral pressure:

$$S < S_u$$

$$S = \frac{T b h}{D_{total}}$$

- where:
- $T$  - average value of pressure acting on the face of bottom block
  - $S$  - force per one running meter of the joint



- $S_u$  - joint bearing capacity (for input use the "Material" frame)  
 $b$  - width =  $1m$  of structure width  
 $h$  - height of the block  
 $D_{total}$  - overall width of meshes loaded in compression  $T$   
 $l$

#### 4) Check for bearing capacity of joint between blocks:

$$N_d < N_u$$

$$Q_{tr} = \frac{N \tan \varphi_d + c_d B}{\gamma_f}$$

$$N_d = \frac{T b h}{D_{total}} + \frac{\max(0, Q - Q_{tr})}{D_{upp}}$$

- where:
- $N_d$  - tensile force per one running meter of the upper joint of the bottom block
  - $N_u$  - strength of mesh (for input use the frame "Material")
  - $Q_{tr}$  - shear force transmitted by friction and cohesion between blocks
  - $\gamma_f$  - reduction coefficient of friction between blocks (for input use the "Wall analysis")
  - $h$  - height of the block
  - $D_{total}$  - overall width of meshes loaded in compression  $T$   
 $l$
  - $D_{upp}$  - width of upper mesh between blocks loaded in tension

## Calculating Abutment Forces

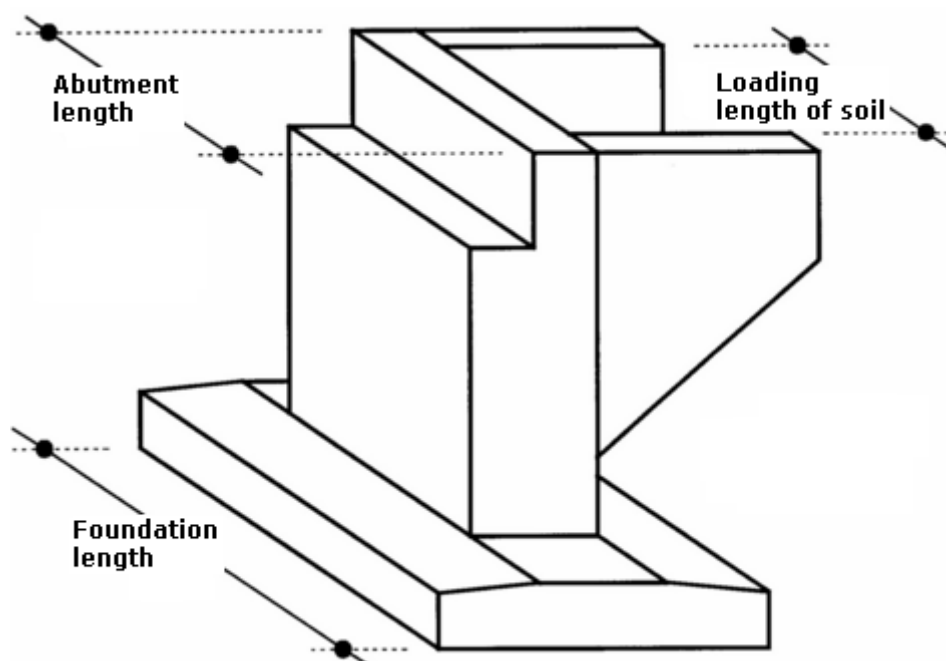
An abutment is analyzed per 1 m ( $lft$ ). All forces entering the analysis are therefore adjusted in the program as follows:

- the **abutment self weight**, assumed per 1 m ( $lft$ ), is calculated from the input transverse cross-section
- **reactions inserted by the bridge and the approach slab** are input in  $kN$  ( $kpi$ ) using the values for the whole abutment, these values are in the analysis divided by the **abutment length**
- **soil pressure** is determined per 1 m ( $lft$ ) and then multiplied by the ratio **length of load due to soil / abutment length**,
- **weight of soil wedges** is determined per 1 m ( $lft$ ) and then multiplied by the ratio **length of load due to soil / abutment length**,
- **surcharge** is determined per 1 m ( $lft$ ) and then multiplied by the ratio **length of load due to soil / abutment length**,
- **input forces** and **front face resistance** are assumed per 1 m without reduction
- **wing walls** - the wing walls self-weight is computed from their geometry; before introduced in the stem design and foundation verification it is divided by the **abutment**

**length** (it is the user responsibility to either include or exclude the effect of wing walls in from the analysis).

Computation of individual abutment forces is described in more detail in chapter "[Wall analyses](#)".

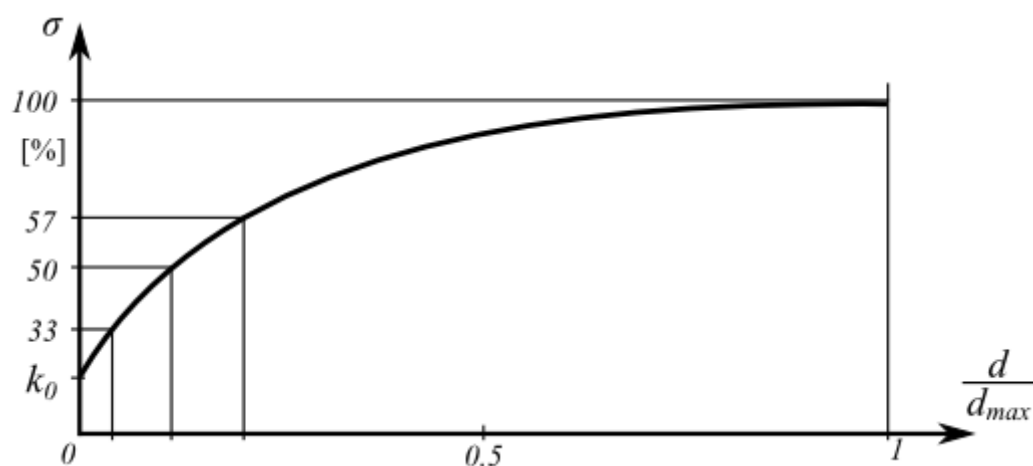
All forces acting in the foundation joint that are introduced in the verification analysis (except for the front face resistance) are multiplied by the ratio **abutment length / foundation length**.



*Geometry of bridge abutment*

## Reduced Passive Earth Pressure

Evolution of passive earth pressure  $\sigma_p$  corresponds to the maximal displacement of a structure pushed into the soil. Such a displacement might not, however, occur (e.g. in the case of fixed sheeting structures) and the structure is loaded by the reduced passive earth pressure  $\sigma_{ps}$ . The value of reduced passive earth pressure  $\sigma_{ps}$  can range from the value of earth pressure at rest  $\sigma_r$  (in the case of zero deformation) up to the value of passive earth pressure  $\sigma_p$ . Figure shows the dependence of values of earth pressure of a cohesionless soil (soil resistance) on the actual displacement  $d$  to maximal displacement  $d_{max}$  ratio (when activating passive earth pressure  $\sigma_p$ ).



Dependence of earth pressure values on the ratio of actual structure deformation

## Nailed Slope

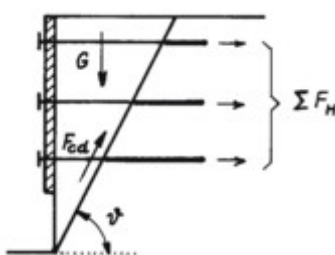
The program "**Nailed Slope**" allows the following verifications:

- Verification of structure **internal stability** (plane or broken slip surface, **bearing capacity of nails**)
- Verification of fictitious wall - the same as **gravity wall** verification
- Verification of structure **concrete cover** (dimensioning)
- Verification of overall stability using the program "**Slope Stability**"

## Analysis of Internal Stability

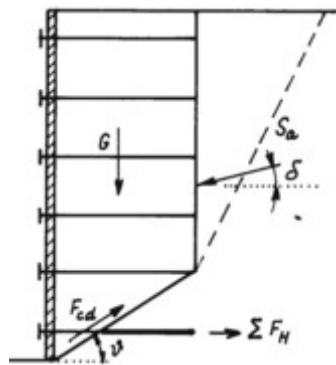
An internal stability of a structure is checked assuming two types of a slip surface:

- **Plane slip surface**



Plane slip surface

- **Broken slip surface**



Broken slip surface

In both cases a specific slip surface is examined for a variation of angle  $\vartheta$ .

When running an **optimization** analysis the calculation is carried out for all benches with a variation of the angle of slip surface  $\nu$  changing from 1 up to 89 degrees with a one degree step.

A verification analysis of internal stability can be performed using either the **factor of safety** or the **theory of limit states** depending on the setting in the "Wall analysis" tab.

The analysis checks whether a ratio of resisting and shear (driving) forces acting on a slip surface is greater than the input factor of safety. The following forces are employed:

#### Shear forces:

- component of gravity force parallel to slip surface
- in case of broken slip surface - component of active earth pressure acting on vertical part of structure and parallel to slip surface (pressure is determined without reduction of input parameters)
- horizontal forces due to earthquake

#### Resisting forces:

- soil friction and cohesion along slip surface
- sum of forces **transmitted by nails**

## Analysis of Bearing Capacity of the Nails

For each nail the following bearing capacities are either computed or **input**:

- $R_f$  - nail head strength
- $R_t$  - nail strength against breaking
- $T_p$  - pull-out nail bearing capacity

**Strength characteristics of a nail** represent the basic parameters to compute the **total bearing capacity of a nail**.

**The nail strength against breaking** follows from:

$$R_t = \frac{\pi d_s^2}{4} \frac{f_y}{SF_t}$$

where:  $R_t$  - strength against breaking

- $d_s$  - nail diameter  
 $f_y$  - strength of nail material  
 $SF_t$  - factor of safety against breaking

**The pull-out nail bearing capacity** is calculated by one of the following ways:

**1. calculate from ultimate bond strength:**

$$T_p = \frac{\pi d g_s}{SF_e}$$

- where:
- $T_p$  - pull-out nail bearing capacity [kN/m]
  - $d$  - hole diameter
  - $g_s$  - ultimate bond strength
  - $SF_e$  - factor of safety against pull-out

**2. calculate from effective stress**

$$T_p = \frac{\pi d (K_a \sigma_z \tan \varphi + c)}{SF_e}$$

where:

$$K_a = \frac{1 + K_0}{2} = \frac{1 + (1 - \sin \varphi)}{2}$$

- where:
- $T_p$  - pull-out nail bearing capacity [kN/m]
  - $d$  - hole diameter
  - $\sigma_z$  - vertical geostatic stress
  - $\varphi$  - effective angle of internal friction of soil
  - $c$  - effective cohesion of soil
  - $SF_e$  - factor of safety against pull-out

**3. calculate according to HA 68/94**

$$T_p = \frac{\pi d (\sigma_n \tan \varphi + c)}{SF_e}$$

- where:
- $T_p$  - pull-out nail bearing capacity [kN/m]
  - $d$  - hole diameter
  - $\sigma_n$  - average radial effective stress
  - $\varphi$  - effective angle of internal friction of soil
  - $c$  - effective cohesion of soil
  - $SF_e$  - factor of safety against pull-out

Average radial effective stress  $\sigma_n$  is calculated by following formula:

$$\sigma_n = \frac{(1 + K_L)\sigma_z}{2}$$

where:  $\sigma_z$  - vertical geostatic stress

$$K_L = \frac{1 + K_a}{2}$$

where:

$$K_a = \frac{1 - \sin \varphi}{1 + \sin \varphi}$$

**Nail head strength** is evaluated by formula:

$$R_f = \frac{\min(R_t; T_p l)(0,6 + 0,2(S_{max} - 1))}{SF_f}$$

where:  $l$  - nail length

$S_{max}$  - maximum spacing of nails in a structure

$R_t$  - nail strength against breaking

$T_p$  - pull-out nail bearing capacity

$SF_f$  - factor of safety of nail head strength

If the nail is not anchored to the structure cover, it is possible to set the nail head strength to zero.

## Estimated Bond Strength

**Estimated bond strength of soil nails in soil and rock** (source: Elias a Juran, 1991)

Material	Construction method	Soil / rock type	Ultimate bond strength $q_s$ [kPa]
Rock	Rotary drilled	Marl / limestone	300 - 400
		Phyllite	100 - 300
		Chalk	500 - 600
		Soft dolomite	400 - 600
		Fissured dolomite	600 - 1000
		Weathered sandstone	200 - 300
		Weathered shale	100 - 150

		Weathered schist	100 - 175	
		Basalt	500 - 600	
		Slate / hard shale	300 - 400	
<b>Cohesionless soils</b>	<b>Rotary drilled</b>	Sand / gravel	100 - 180	
		Silty sand	100 - 150	
		Silt	40 - 120	
		Piedmont residual	40 - 120	
		Fine colluvium	75 - 150	
	<b>Driven casing</b>	Sand / gravel -low overburden	190 - 240	
		-high overburden	280 - 430	
		Dense moraine	380 - 480	
			Colluvium	100 - 180
	<b>Augered</b>	Silty sand fill	20 - 40	
Silty fine sand		55 - 90		
Silty clayey sand		60 - 140		
<b>Jet grouted</b>	Sand	380		
	Sand gravel	700		
<b>Fine - grained soils</b>	<b>Rotary drilled</b>	Silty clay	35 - 50	
		<b>Driven casing</b>	Clayey silt	90 - 140
	<b>Augered</b>	Loess	25 - 75	
		Soft clay	20 - 30	
		Stiff clay	40 - 60	
		Stiff clayey silt	40 - 100	
		Calcareous sandy clay	90 - 140	

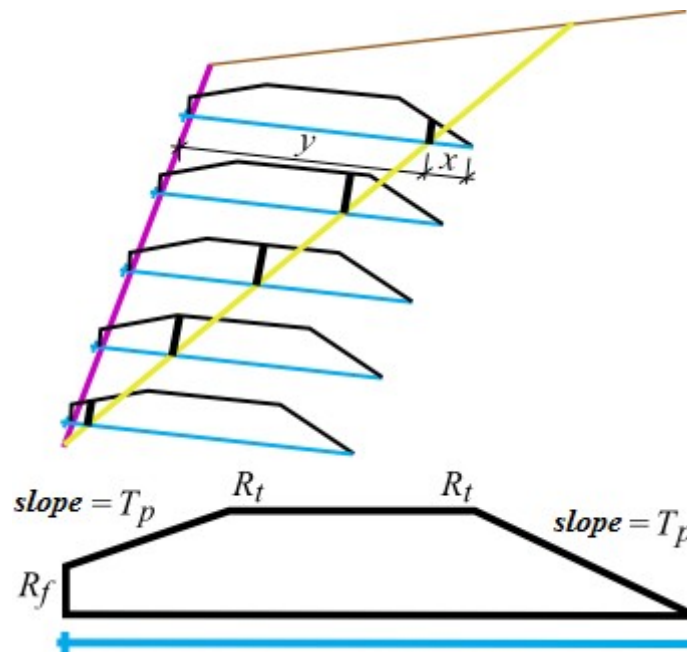
**Note:** Convert values in *kPa* to *psf* by multiplying by 20.9. Convert values in *kPa* to *psi* by multiplying by 0.145.

## Total Bearing Capacity of a Nail

Bearing capacity of nail is determined based on the location of its intersection with a slip surface. If a nail is found completely in front of the slip surface, then it does not enter the calculation. If a nail crosses the slip surface, then its bearing capacity is determined as:

$$F = \min (T_p x ; R_t ; R_f + T_p y)$$

- where:
- $x$  - nail length behind slip surface in direction of soil body
  - $y$  - nail length in front of slip surface
  - $R_f$  - nail cap bearing capacity
  - $R_t$  - nail strength against breaking
  - $T_p$  - pull-out nail bearing capacity



Distribution of tensile force along nail

## Verification - Factor of Safety

The analysis checks whether a ratio of **resisting** and **shear** (driving) forces acting on a slip surface is greater than the input factor of safety.

A factor of safety on the input slips forces is thus provided by:

$$SF = \frac{F_h \cos(\nu + \alpha) + F_{cd}}{(G + S_{a,vert}) \sin \nu + S_{a,hor} \cos \nu}$$

$$F_h = \sum F_{h,n}$$

$$F_{cd} = \sum \frac{d_i}{d} (G \cos \nu + F_h \sin(\nu + \alpha)) \tan \varphi_i + \sum d_i c_i$$



where:

$G$	- gravity force
$S_{a,vert}$	- vertical component of active pressure
$S_{a,hor}$	- horizontal component of active pressure
$d_i$	- length of $i^{th}$ section slip surface
$d$	- length of slip surface
$F_{h,n}$	- bearing capacity of $n^{th}$ nail behind slip surface per 1 running meter
$c_i$	- cohesion of $i^{th}$ soil layer
$\varphi_i$	- angle of internal friction of $i^{th}$ layer
$\nu$	- inclination of slip surface
$\alpha$	- inclination of nails from horizontal direction

## Verification - Theory of Limit States

The analysis checks whether the **passive** (resisting) forces  $F_p$  acting on a slip surface are greater than the **active** (shear) forces  $F_a$ :

$$F_p > F_a$$

$$F_p = F_h \cos(\nu + \alpha) + F_{cd}$$

$$F_a = (G + S_{a,vert}) \sin \nu + S_{a,hor} - \cos \nu$$

$$F_h = \sum F_{h,n}$$

$$F_{cd} = \sum \frac{d_i}{d} (G \cos \nu + F_h \sin(\nu + \alpha)) \tan \varphi_i + \sum d_i c_i$$

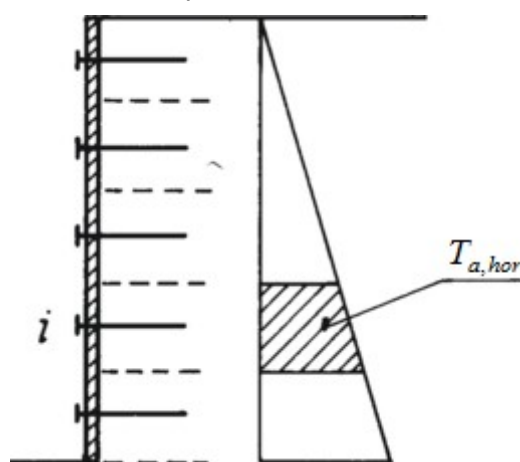
where:

$G$	- gravity force
$S_{a,vert}$	- vertical component of active pressure
$S_{a,hor}$	- horizontal component of active pressure
$d_i$	- length of $i^{th}$ section slip surface
$d$	- length of slip surface
$F_{h,n}$	- bearing capacity of $n^{th}$ nail behind slip surface per 1 running meter
$c_i$	- cohesion of $i^{th}$ soil layer
$\varphi_i$	- angle of internal friction of $i^{th}$ layer
$\nu$	- inclination of slip surface
$\alpha$	- inclination of nails from horizontal direction

## Nail Force

The magnitude of **active earth pressure** is reduced using a coefficient  $k_n$ . The recommended (experimentally determined) value is  $k_n = 0,85$ .

Forces transmitted by individual nails are determined such that a particular portion of the calculated earth pressure is assigned **to a given bench**. Each nail is then loaded by the corresponding portion of the active earth pressure.



*Forces transmitted by individual nails*

The **nail force** is provided by:

$$F_i = \frac{b k_n \sum T_{a,hor}}{\cos \alpha}$$

- where:
- $b$  - nail spacing
  - $\alpha$  - nail inclination
  - $k_n$  - reduction coefficient
  - $T_{a,hor}$  - active earth pressure acting on a given bench

## Dimensioning of Concrete Cover

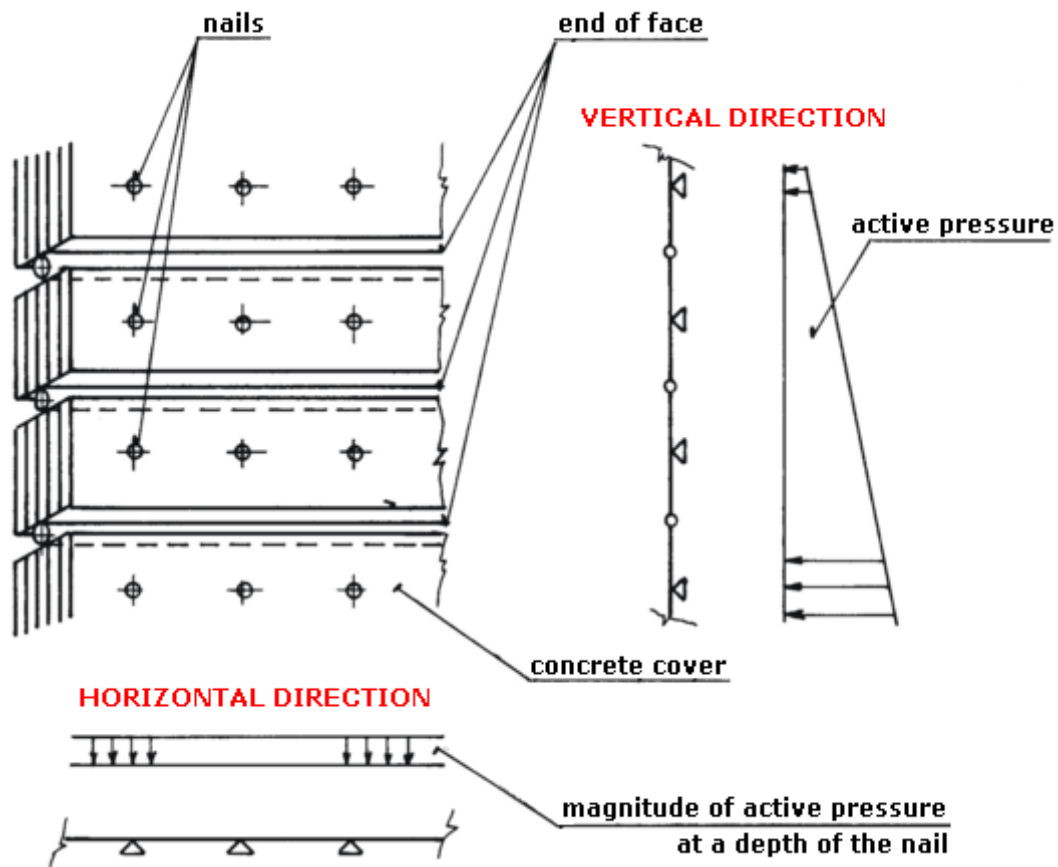
**The concrete cover of a nailed slope** is designed to sustain an active earth pressure. So, the structure is assumed to be subdivided into individual intermediate design strips.

In the **vertical direction** the nail cap is modeled as a support and joint between benches as an internal hinge.

In the **horizontal direction** the program generates (by default) a structure with five supports uniformly loaded by the magnitude of active pressure up to a depth of the nail cap.

The program further allows for the verification of **concrete cover reinforcement** of a structure loaded by the bending moment.

Constructing scheme of the **model design** including load is evident from the figure:



*Dimensioning of concrete cover*

## Sheeting Design

Analyses in the program "**Sheeting design**" can be divided into three groups:

- Analysis of anchor free walls (e.g. sheet pile wall)
- Analysis of anchored walls fixed at heel
- Analysis of anchored walls simply supported at heel

It is also possible to analyze braced sheeting using this program.

## Analysis of Pile Sheetting Wall

A pile sheeting wall is analyzed using a standard approach that account for the effect of earth pressures. In general, the active earth pressure develops behind the structure while the passive earth pressure appears in front of the structure.

Based on the **theory of limit states** the program searches in an iterative way a point on the wall to satisfy the moment equation of equilibrium in the form:

$$M_{\text{overturning}} = M_{\text{resisting}}$$

Once this is accomplished, the program continues by determining the wall heel location for which the equilibrium of shear forces is fulfilled (computation of depth of fixed end). The

overall length of the analyzed structure is found this way.

When applying approach based on the **factor of safety** the program searches, in an iterative way, a point to get:

$$\frac{M_{resisting}}{M_{overturning}} = FS$$

It is obvious that the distribution of internal forces resulting from this approach is not very realistic. In some countries, however, this approach is required.

The computation can be driven either by choosing a **minimal dimensioning pressure** or by reduction of **passive pressure**. Assuming the actual magnitude of the passive earth pressure provides deformations of the analyzed structure, which cannot usually occur. The actual passive pressure can attain for walls free of deformation the value of pressure at rest as well as all intermediate values up to the value of passive pressure for fully deformed wall (rotation app. 10 *mRad* - i.e. deformation 10 *mm* per 1 *m* of structure height). Therefore it is reasonable to consider reduced values of the passive earth pressure by setting the value of the "**Coefficient of reduction of passive pressure**" to less than or equal to one. The following values are recommended:

- 0.67 reduces deformations app. by one half
- 0.50 approximately corresponds to deformation of structure loaded by increased active earth pressure
- 0.33 approximately corresponds to deformation of structure loaded by the pressure at rest, structure reaches app. 20 percent of its original deformations

## Analysis of Anchored Wall Fixed in Heel

Anchored wall fixed in heel is analyzed as a continuous beam using the deformation variant of the finite element method in order to comply with the assumption of heel fixed in the soil. The actual analysis is preceded by the determination of load due to **earth pressure** applied to the structure. The pressure acting on the back of a structure is assumed to be the **active pressure**, while the front face is loaded by the **passive pressure**.

The passive pressure can be reduced with the help of the **coefficient of reduction of passive pressure**. Assuming the actual magnitude of the passive earth pressure provides deformations of the analyzed structure, which cannot usually occur. The actual passive pressure can attain for walls free of deformation the value of pressure at rest as well as all intermediate values up to the value of passive pressure for fully deformed wall (rotation app. 10 *mRad* - i.e. deformation 10 *mm* per 1 *m* of structure height). Therefore it is reasonable to consider reduced values of the passive earth pressure setting the value of the "**Coefficient of reduction of passive pressure**" to less than or equal to one. The following values are recommended:

- 0.67 reduces deformations app. by one half
- 0.33 deformations attain approximately twenty percent of their original values

The program offers two options to **determine active pressure**:

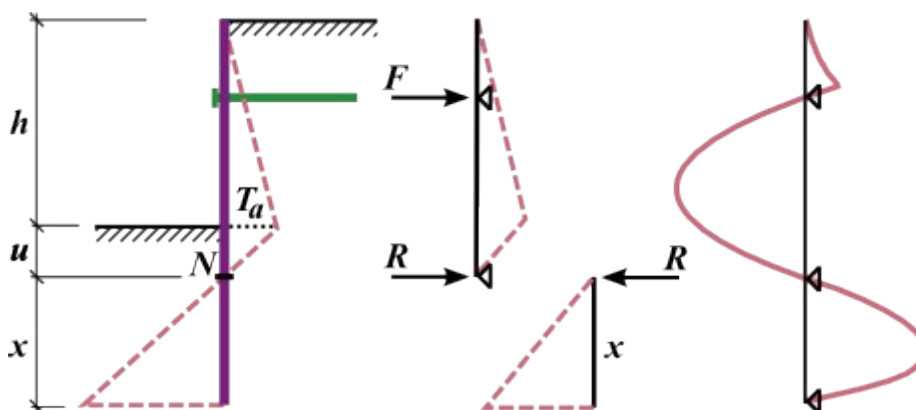
- calculation from input soil parameters, water, surcharge, terrain including introduction of the **minimum dimensioning pressure**
- inputting an arbitrary distribution of earth pressure up to the depth of zero point (this way it is possible to introduce an arbitrary redistribution of earth pressure)

Zero-value point, i.e. the point at which the overall pressure equals zero is determined by the following expression:

$$u = \frac{\sigma_a}{\gamma \cdot K}$$

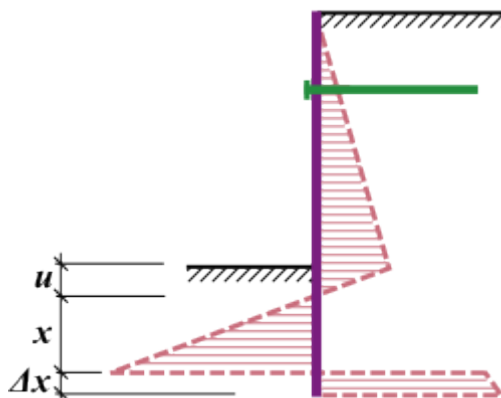
- where:
- $u$  - depth of zero-value point
  - $\sigma_a$  - magnitude of active pressure behind structure at the ditch bottom
  - $K$  - coefficient of overall pressure
  - $\gamma$  - unit weight of soil below the ditch bottom

The analysis of structure fixed at heel assumes that the point of zero load  $N$  (at depth  $u$ ) is identical with the point of zero moment. For the actual analysis the structure is divided into two parts - an upper part (upper beam) up to zero-value point and a lower beam:



*Analysis of anchored wall fixed in heel*

The upper beam is analyzed first together with evaluation of anchor forces  $F$  and the reaction force  $R$  at the zero-value point. Then, the lower beam length  $x$  is determined such that the moment equilibrium condition with respect to the heel is satisfied (the beam is loaded by the reaction  $R$  and by the difference of earth pressures). To satisfy the shear force equilibrium the computed length of fixed end is extended by the value  $\Delta x$  as shown in figure:



*Determination of the extension of the length of wall by  $\Delta x$*

## Analysis of Anchored Wall Simply Supported at Heel

Anchored wall simply supported at heel is analyzed as a continuous beam using the deformation variant of the finite element method in order to comply with the assumption of simply supported structure at heel. The actual analysis is preceded by the determination of

load due to **earth pressure** applied to the structure. The pressure acting on the back of a structure is assumed as **active pressure**, while the front face is loaded by **passive pressure**.

The passive pressure can be reduced with the help of the **coefficient of reduction of passive pressure**. Assuming the actual magnitude of the passive earth pressure provides deformations of the analyzed structure, which cannot usually occur. The actual passive pressure can attain for walls free of deformation the value of pressure at rest as well as all intermediate values up to the value of passive pressure for fully deformed wall (rotation app. 10 *mRad* - i.e. deformation 10 *mm* per 1 *m* of structure height). Therefore it is reasonable to consider reduced values of the passive earth pressure setting the value of the "**Coefficient of reduction of passive pressure**" to less than or equal to one. The following values are recommended:

- 0.67 reduces deformations app. by one half
- 0.33 deformations attain approximately twenty percent of their original values

The program offers two options to **determine active pressure**:

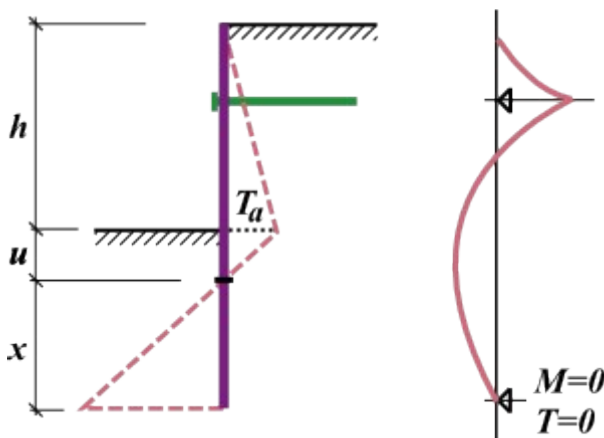
- calculation from input soil parameters, water, surcharge, terrain including introduction of the **minimum dimensioning pressure**
- inputting an arbitrary distribution of earth pressure up to the depth of zero point (this way it is possible to introduce an arbitrary redistribution of earth pressure).

Zero-value point, i.e. the point at which the overall pressure equals zero is determined by the following expression:

$$u = \frac{\sigma_a}{\gamma \cdot K}$$

- where:
- $u$  - depth of zero-value point
  - $\sigma_a$  - magnitude of active pressure behind structure at the ditch bottom
  - $K$  - coefficient of overall pressure
  - $\gamma$  - unit weight of soil below the ditch bottom

For simply supported structures it is assumed that the moment and shear force are zero at the heel. The program first places the end of a structure into the zero-value point, and then it looks for the end beam location  $x$ , where the above condition is fulfilled (see Fig.). Solution procedure for multiplied anchored walls is identical.



*Analysis of anchored wall simply supported at heel*

## Sheeting Check

The program verifies the input structure using the **method of dependent pressures** or using the **spring method according to JGJ 120-2012**. The load applied to the structure is derived from its deformation, which allows to realistically model its behavior and provides cost effective designs. The analysis correctly accounts for the **construction process** such as individual stages of progressive construction of the wall (**stages of constructions**) including gradual evolution of deformations and post-stressing of anchors, and makes it possible to model **braced sheeting**.

The use of the method of dependent pressures requires determination of the **modulus of subsoil reaction**, which is assumed either linear or **nonlinear**.

The program also allows the user to **check internal stability** of the anchorage system.

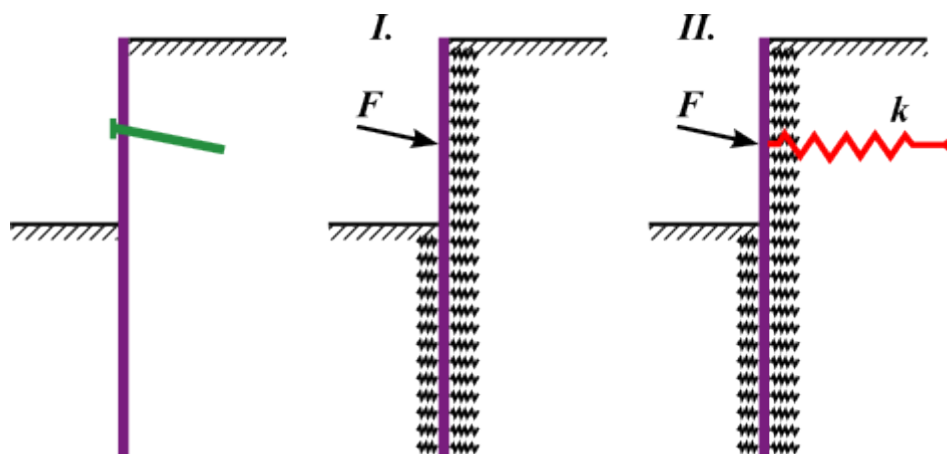
The **actual analysis** is carried out using the deformation variant of the finite element method. Displacements, internal forces and the modulus of subsoil reaction are evaluated at individual nodes.

The following procedure for dividing the structure into finite elements is assumed:

- First, the nodes are inserted into all topological points of a structure (starting and end points, points of location of anchors, points of soil removal, points of changes of cross-sectional parameters).
- Based on selected subdivision the program computes the remaining nodes such that all elements attain approximately the same size.

A value of the modulus of subsoil reaction is assigned to each element - it is considered as the Winkler spring of the elastic subsoil. **Supports** are placed onto already **deformed structure** - each support then represents a forced displacement applied to the structure.

**Anchors**, in the load case at which they were introduced or post-stressed, are considered as forces (variant I in Fig). In other load cases, the anchors are modeled as forces and springs of stiffness  $k$  (variant II. in Fig):



*Braced sheeting*

The change of anchor force due to deformation is provided by:

$$\Delta F = \frac{k \cdot v \cdot \Delta w}{\cos \alpha}$$

$$k = \frac{E \cdot A}{l}$$

where:

- $v$  - horizontal distance between anchors
- $\Delta w$  - deformation increment the point of anchor application
- $E$  - anchor Young's modulus
- $A$  - anchor cross-sectional area
- $l$  - anchor length
- $k$  - anchor stiffness
- $\alpha$  - anchor inclination

*Literature:*

Hurych, P.: *Metoda zavislych tlaku. Sbornik konference "Automatizacia projektovania", Vysoke Tatry, 1978.*

## Method of Dependent Pressures

The basic assumption of the method is that the soil or rock in the vicinity of wall behaves as ideally elastic-plastic Winkler material. This material is determined by the **modulus of subsoil reaction**  $k_h$ , which characterizes the deformation in the elastic region and by additional limiting deformations. When exceeding these deformations the material behaves as ideally plastic.

The following assumptions are used:

- The **pressure acting** on a wall may attain an arbitrary value between **active** and **passive** pressure - but it cannot fall outside of these boundaries.
- The **pressure at rest** acts on an undeformed structure ( $w = 0$ ).

The pressure acting on a deformed structure is given by:

$$\sigma = \sigma_r - k_h \cdot w$$

$$\sigma = \sigma_a \text{ for: } \sigma < \sigma_a$$

$$\sigma = \sigma_p \text{ for: } \sigma > \sigma_p$$

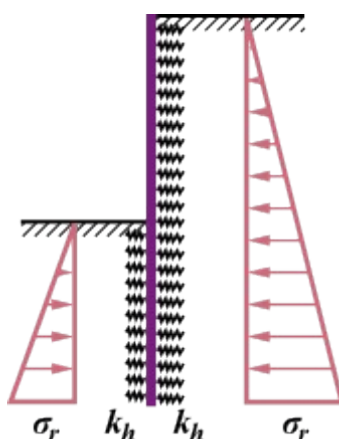
where:

- $\sigma_r$  - pressure at rest
- $k_h$  - modulus of subsoil reaction
- $w$  - deformation of structure
- $\sigma_a$  - **active earth pressure**
- $\sigma_p$  - **passive earth pressure**

The computational procedure is as follows:

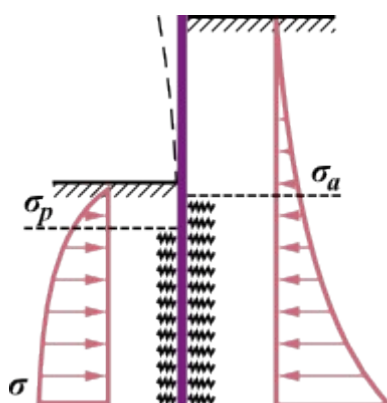
- The modulus of subsoil reaction  $k_h$  is assigned to all elements and the structure is loaded by the pressure at rest - see figure:





*Scheme of structure before first iteration*

- The analysis is carried out and the condition for allowable magnitudes of pressures acting on the wall is checked. In locations at which these conditions are violated the program assigns the value of  $k_h=0$  and the wall is loaded by active or passive pressure, respectively - see figure:



*Scheme of structure during the iteration process*

The above iteration procedure continues until all required conditions are satisfied.

In analyses of subsequent stages of construction the program accounts for plastic deformation of the wall. This is also the reason for specifying individual **stages of construction** that comply with the actual construction process.

*Literature:*

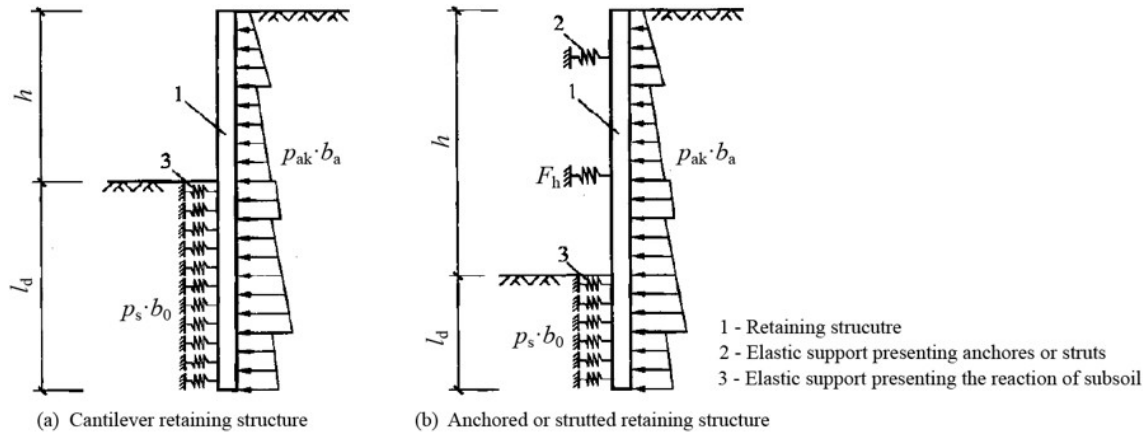
*Bartak, J.: Progresivni postupy navrhovani pazenych stavebnich jam. VUT Brno, 1991.*

*Hurych, P.: Metoda zavislych tlaku. Sbornik konference "Automatizacia projektovania", Vysoke Tatry, 1978.*

## Spring Method According to JGJ 120-2012

This method is used for analysis of **sheeting structures** and it is based on the **Chinese standard JGJ 120-2012** (Technical specification for retaining and protection of building foundation excavations). In principle, this theory is similar to calculation according to the **method of dependent pressures**, the difference is in consideration of **earth pressures**. The following figure shows that **behind the wall** (outside of the foundation pit) acts **active earth pressure**  $p_a$  or **earth pressure at rest**  $p_0$  (it's defined in the "Settings" frame).

**In front of the wall** there are considered springs (defined by using the **modulus of subsoil reaction**), which models reaction of the soil in a horizontal direction. In case of the attainment of ultimate pressures a limiting of the size of springs is the same as for **method of dependent pressures**.



Principle of spring method according to JGJ 120-2012 for solution of sheeting structures - (a) non-anchored structure, (b) anchored or strutted structure

Literature:

JGJ 120-2012 (Technical specification for retaining and protection of building foundation excavations).

## Modulus of Subsoil Reaction

The following options are available in the program to introduce the modulus of subsoil reaction:

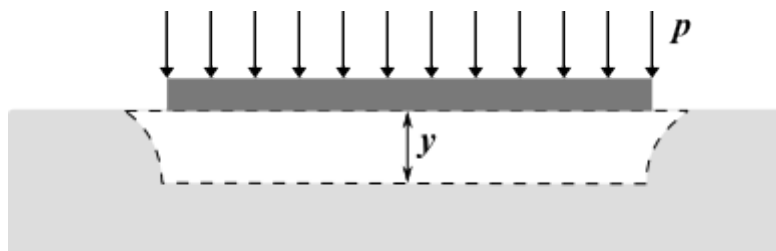
- **in the form of distribution** (input distribution of the modulus of subsoil reaction  $k_h$  in front and behind the structure is input)
- **as a soil parameter with a respective value** (either linear or **nonlinear - curve**)
- according to Schmitt
- according to Chadeisson
- according to CUR 166
- iterate by using deformation characteristics of soils
- input results of **pressiometric test** (according to NF P 94-282, according to Menard)
- input results of **dilatometric test (DMT)**
- according to **Chinese standards** ("c", "k" or "m" method)

The modulus of horizontal reaction of a soil body generally corresponds to spring stiffness in the Winkler model describing the relation between load applied to a rigid slab and the resulting soil deformation in the form:

$$p = k \cdot y$$

where:  $p$  - load acting along slab-soil interface  
 $k$  - stiffness of Winkler spring

$y$  - translation of slab into subsoil



Definition of the modulus of subsoil reaction

## Modulus of Subsoil Reaction According to Schmitt

This analysis of the **modulus of subsoil reaction** builds on the relation between **oedometric modulus** and bending stiffness of the structure introduced by Schmitt in *Revue Francaise de Géotechnique* no. 71 and 74:

$$k_h = 2,1 \left( \frac{E_{oed}^{4/3}}{(EI)^{1/3}} \right)$$

where:

- $EI$  - bending stiffness of the structure [ $MNm^2/m$ ]
- $E_{oed}$  - oedometric modulus [ $MPa$ ]

Literature:

Schmitt, P. (1995): "Estimating the coefficient of subgrade reaction for diaphragm wall and sheet pile wall design", in *French. Revue Française de Géotechnique*, N. 71, 2° trimestre 1995, 3-10.

## Modulus of Subsoil Reaction According to Chadeisson

Based on the measurements on sheeting structures in different soils and computation of a displacement of the structure needed to mobilize the limit value of passive pressure  $R$ . Chadeisson (1961) and A. Monnet (1994) derived expression for the determination of the **modulus of subsoil reaction** in the form:

$$k_h = \left[ 20 EI \left( \frac{K_p \gamma \left( 1 - \frac{K_0}{K_p} \right)}{0,015} \right)^4 \right]^{1/5} + A_p c' \frac{\operatorname{tgh} \left( \frac{c'}{30} \right)}{0,015}$$

where:

- $EI$  - bending stiffness of the structure [ $kNm^2/m$ ]
- $\gamma$  - unit weight of soil [ $kN/m^3$ ]
- $K_p$  - coefficient of passive earth pressure [-]

- $K_0$  - coefficient of earth pressure at rest [-]  
 $c'$  - effective cohesion of soil [ $kPa$ ]  
 $A_p$  - coefficient of influence of cohesion (1 - 15) [-]

*Literature:*

Chadeisson, R. (1961): *Parois continues moulées dans le sols. Proceedings of the 5<sup>th</sup> European Conference on Soil Mechanics and Foundation Engineering, Vol. 2. Dunod, Paris, 563-568".*

K. J. Bakker, A. Bezuijen, W. Broere, E. A. Kwast: *Geotechnical Aspects of Underground Construction in Soft Ground: Proceedings of the 5th International Symposium TC28. Amsterdam, the Netherlands, 15-17 June 2005. CRC Press, 2013, pp. 616, ISBN: 0415889138, 9780415889131.*

Monnet, A.: *Module de réaction, coefficient de décompression, au sujet des paramètres utilisés dans la méthode de calcul élastoplastique, Revue française de Géotechnique, 65, 1994, pp. 67 - 72.*

Mitew, M.: *Numerical analysis of displacements of a diaphragm wall. Warsaw University of Technology, Poland.*

N. M. ILIEȘ, T. A. HULPUȘ, A. POPA: *Design of Anchored Walls: The Influence of Design Approaches and Design Methods. Technical University of Cluj Napoca, Faculty of Civil Engineering, Romania, 2010.*

## **Modulus of Subsoil Reaction According to CUR 166**

The following table stores the values of the **modulus of subsoil reaction** derived from experimental measurements carried out in Netherlands (described in CUR 166). The table offers secant modules, which are in the program directly transformed into secant modules of subsoil reaction - see **nonlinear modulus of subsoil reaction**.

	$k_{h,1}$ [ $kN/m^3$ ] $p_0 < p_h < 0,5 p_{pas}$	$k_{h,2}$ [ $kN/m^3$ ] $0,5 p_{pas} \leq p_h \leq 0,8 p_{pas}$	$k_{h,3}$ [ $kN/m^3$ ] $0,8 p_{pas} \leq p_h \leq 1,0 p_{pas}$
<b>Sand</b>			
loose	12000 - 27000	6000 - 13500	3000 - 6750
medium dense	20000 - 45000	10000 - 22500	5000 - 11250
dense	40000 - 90000	20000 - 45000	10000 - 22500
<b>Clay</b>			
soft	2000 - 4500	800 - 1800	500 - 1125
stiff	4000 - 9000	2000 - 4500	800 - 1800
very stiff	6000 - 13500	4000 - 9000	2000 - 4500
<b>Peat</b>			
soft	1000 - 2250	500 - 1125	250 - 560
stiff	2000 - 4500	800 - 1800	500 - 1125

where:  $p_0$  - value of pressure at rest [ $kN/m^2$ ]  
 $p_{pas}$  - passive pressure [ $kN/m^2$ ]  
 $p_h$  - horizontal pressure corresponding to a given displacement of a structure [ $kN/m^2$ ]

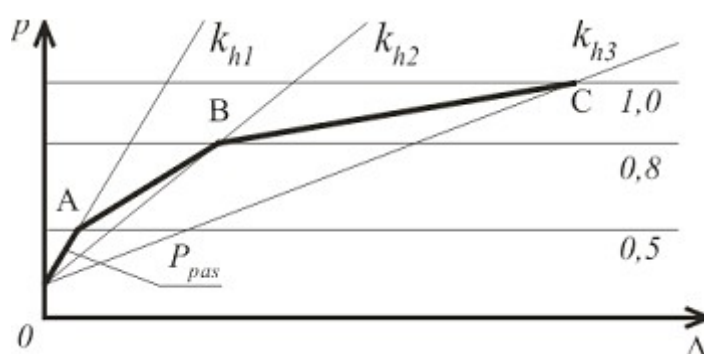


Diagram of determination of the modulus of subsoil reaction

Literature:

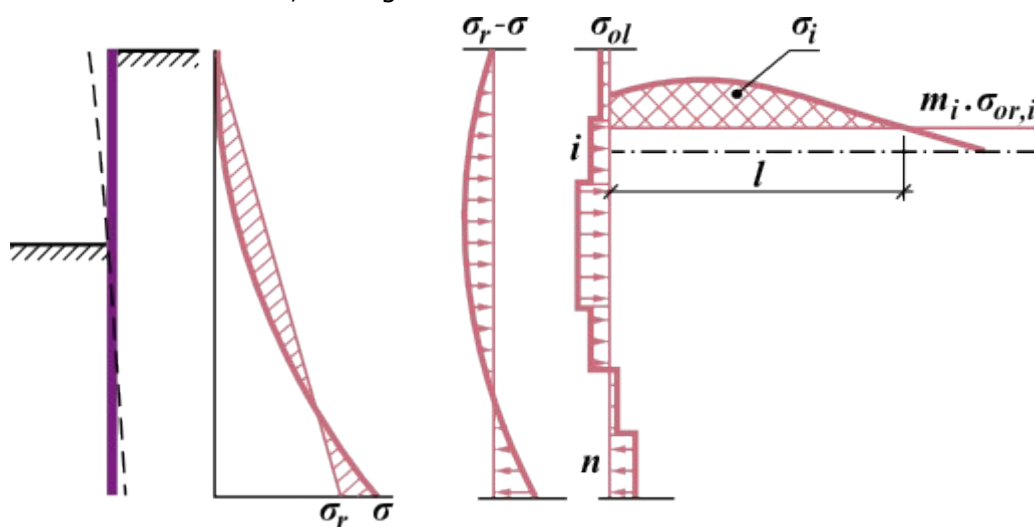
CUR 166 Damwandconstructies, available at Civieltechnisch Centrum Uitvoering Research en Regelgeving: P.O.Box 420, 2800 AK Gouda (NL).

## Modulus of Subsoil Reaction Determined from Iteration

The program allows automatic calculation of the modulus of subsoil reaction from

deformational characteristics of soil from iteration process. The procedure builds on the assumption that deformation of the elastic subspace characterized by the **deformation modulus**  $E_{def}$  [MPa] when changing the stress state associated with the change of earth pressures is the same as deformation of the underground wall.

The goal therefore is to find such values of  $k_h$  [MN/m<sup>3</sup>] so that the continuity of deformations of wall and adjacent soil is maintained. **Plastic deformation of structure is not considered** when performing analysis with  $k_h$  **manual iteration**. While the analysis of modulus  $k_h$  with **automatic iteration plastic deformation of structure is considered**. Principle of manual iteration is schematically cleared by computing the modulus of subsoil reaction of the  $i^{th}$  segment of wall free of anchor, see figure:



Determination of modulus of subsoil reaction of  $i^{th}$  segment

For change of stress  $\sigma_r - \sigma$  the program determines uniform load  $\sigma_{ol}$  [MPa] of individual segments of a structure. Next, the overall change of stress passing the  $i^{th}$  segment ( $\bar{\sigma}_{il}$  [MPa\*m]) is computed. This change is caused by additional load of the soil body due to segments  $1$  to  $n$  ( $\sigma_{ol,1} - \sigma_{ol,n}$ ). The overall change of stress  $\Delta\sigma_i$  is reduced by structural strength  $m_i \cdot \sigma_{or,i}$  [MPa]. The new value of the spring stiffness then follows from:

$$k_{n,i} = \frac{E_{def,i} \cdot \sigma_{ol,i}}{\bar{\sigma}_{il}}$$

- where:  $E_{def}$  - deformation modulus of elastic subspace [MPa]
- $\sigma_{ol}$  - uniform load applied to segments of structure [MPa]
- $\bar{\sigma}_{il}$  - overall change of stress behind  $i^{th}$  segment of structure [MPa\*m]

The change of stress inside the soil body is determined according to **Boussinesque**. Inserting the new value of  $k$  directly into the next calculation would cause instable iteration - therefore the value of  $k$  that is introduced into the next analysis of the wall is determined from the original value of  $k_p$  and the new value of  $k_n$  of the modulus of subsoil reaction.

$$k = k_p + 0,25 \cdot (k_n - k_p)$$

- where:  $k_p$  - original value of modulus of subsoil reaction [MN/m<sup>3</sup>]

$k_n$  - new value of modulus of subsoil reaction [ $MN/m^3$ ]

Maximum of modulus of subsoil reaction of the  $i^{th}$  layer is limited by the value:

$$k_{max,ip} = 10 \cdot E_{def,i}$$

where:  $E_{def,i}$  - deformation modulus of  $i^{th}$  layer [ $MPa$ ]

The **manual iterative procedure** used when computing the modulus of subsoil reaction is as follows:

1. Determine the matrix of influence values for deriving change of stress in a depth of the soil body passing the  $i^{th}$  segment of a structure due to surcharge caused by the change of stress in other segments.
2. The first approximation of the modulus  $k_h$  in front of the wall is introduced - a triangular distribution of values at the wall heel  $k_h = 10 \text{ MN/m}^3$  is assumed.
3. Perform analysis of the wall (sheeting structure).
4. Compute new values of  $k_h$  and determine new values for the next analysis.
5. The dialog window to check the iteration appears and the program waits till the next command. If the next  $n$  iterations are selected, the steps 3 and 4 are repeated  $n$ -times to arrive again at the step No. 5. The analysis is terminated in this dialog window by pressing the "**Stop**" button.

This **manual iterative process** is controlled by the user - he or she has to decide whether the results make sense. **Automatic iterative procedure** is performed without entering of further iterations for calculation of modulus  $k_h$ .

*Literature:*

*Bartak J.: Progresivni postupy navrhovani pazenych stavebnich jam, VUT Brno, 1991.*

## Modulus of Subsoil Reaction According to Menard

Based on the results from experimental measurements (**pressiometer**) of soil response loaded by rigid slab Menard derived the following expression for **modulus of subsoil reaction**:

$$k_h = \frac{E_M}{\frac{\alpha \cdot a}{2} + 0,133 \cdot (9 \cdot a)^\alpha}$$

where:  $E_M$  - **pressiometric (Menard) modulus**, if necessary it can be substituted by oedometric modulus of soil [ $MPa$ ]

$a$  - characteristic length depending on a depth of fixed-end structure, according to Menard assumed at a depth of 2/3 of length of fixed-end structure below final depth of sheeted ditch [ $m$ ]

$\alpha$  - rheological coefficient of soil [-]

**Approximate values of rheological coefficient of soil  $\alpha$**

	<b>Clay</b>	<b>Silt</b>	<b>Sand</b>	<b>Gravel</b>
Overconsolidated	1	2/3	1/2	1/3
Normally consolidated	2/3	1/2	1/3	1/4
Non-consolidated	1/2	1/2	1/3	1/4

*Literature:*

Menard, L. (1975): "The Menard Pressuremeter: Interpretation and Application of the Pressuremeter Test Results to Foundations Design", Sols-Soils, No. 26, Paris, France.

## Modulus of Subsoil Reaction According to NF P 94-282

The modulus of subsoil reaction  $k_h$  according to **NF P94-282:2009-03** depends on bending stiffness of sheeting structure  $E_{str} I_{str}$  and pressiometric (Menard) modulus  $E_M$ . The modulus of subsoil reaction is given by the following formula:

$$k_h = 2 \left( \frac{\left( \frac{E_M}{\alpha} \right)^{4/3}}{\left( \frac{E_{str} I_{str}}{B_0} \right)^{1/3}} \right) = 2 \left( \frac{\left( \frac{E_M}{\alpha} \right)^{4/3}}{(EI)^{1/3}} \right)$$

$$EI = \frac{E_{str} I_{str}}{B_0} = [MNm^2 / m]$$

- where:  $k_h$  - modulus of subsoil reaction [ $MN/m^3$ ]  
 $E_M$  - pressiometric modulus according to Menard [ $MPa$ ]  
 $\alpha$  - empirical coefficient depending on type of soil, or type of rock [-]  
 $E_{str} I_{str}$  - bending stiffness of sheeting structure [ $MNm^2$ ]  
 $B_0$  - characteristic (unit) length of sheeting structure [ $1 \text{ m}$ ]  
 $E$  - modulus of elasticity of material of sheeting structure [ $MPa$ ]  
 $I$  - moment of inertia [ $m^4/m$ ]

**Approximate values of empirical coefficient  $\alpha$  [-] for various types of soils**



Type of soil	Rašelina	Clay		Silt		Sand		Gravel	
	$\alpha$	$E_M/pLM$	$\alpha$	$E_M/pLM$	$\alpha$	$E_M/pLM$	$\alpha$	$E_M/pLM$	$\alpha$
Overconsolidated	-	> 16	1	> 14	2/3	> 12	1/2	> 10	1/3
Normally consolidated	1	9 - 16	2/3	8 - 14	1/2	7 - 12	1/3	6 - 10	1/4
Non-consolidated	-	7 - 9	1/2	5 - 8	1-2	5 - 7	1-3	-	-

**Approximate values of empirical coefficient  $\alpha$  [-] for various types of rocks (according to degree of violation)**

Type of rock	$\alpha$ [-]
Intact, strong	2/3
Slightly impaired, unweathered	1/2
Very poor, weathered	1/3
Metamorphic	2/3

Literature:

NF P94-282: March 2009, pp. 142 – 146.

## Modulus of Subsoil Reaction Specified by Dilatometric Test (DMT)

The program "**Sheeting check**" in the frame "**Dilatometric tests (DMT)**" defines characteristic length of sheeting structure (**coefficient of reduction**)  $B$ , which is used for correct the modulus of subsoil reaction  $k_h$  [ $MN/m^3$ ] and it's determined by the following formula:

$$k_h = \frac{M_{DMT}}{B}$$

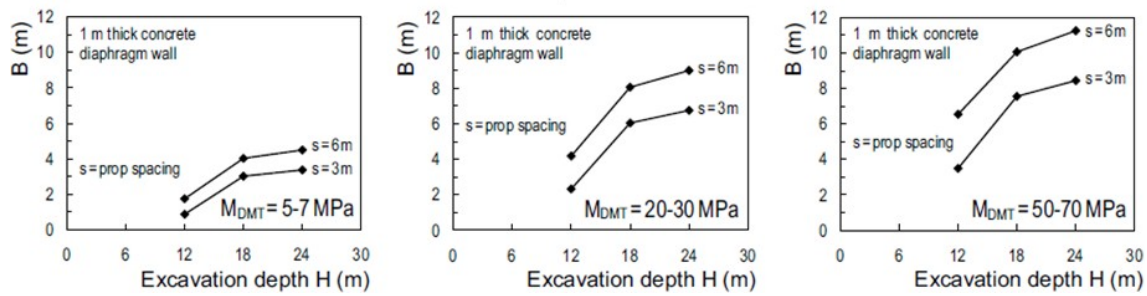
where:  $k_h$  - modulus of subsoil reaction [ $MN/m^3$ ]

$M_{DMT}$  - constrained soil modulus obtained from DMT [ $MPa$ ]

$B$  - characteristic length of sheeting structure (coefficient of reduction) [ $1\text{ }rm$ ]

The results obtained from performed **dilatometric tests (DMT)** pointed to the fact that in some cases the modulus of subsoil reaction  $k_h$  decreases due to a deeper excavation and subsequent displacement of the **sheeting structure**.

Values of characteristic length of sheeting structure (coefficient of reduction)  $B$  depending on the depth of ditch  $h$  [ $m$ ] and value of constrained soil modulus  $M_{DMT}$  [ $MPa$ ] are shown in the following figure.



Characteristic length  $B$  ( $M_{DMT}/k_h$ ) vs. excavation depth  $H$  for different values of  $M_{DMT}$  (average over total wall length) and prop spacing  $s$  (source: [2], Figure 7, pp. 999)

Literature:

Monaco, P. and Marchetti, S.: Evaluation of the coefficient of subgrade reaction for design of multi-propped diaphragm walls from DMT moduli. Millpress, Rotterdam, 2004, pp. 993 – 1002, ISBN 90 5966 009 9.

## Modulus of Subsoil Reaction According to Chinese standards

The calculation of modulus of subsoil reaction according to Chinese standards is based on the JGJ 120-2012 standard (Technical specification for retaining and protection of building foundation excavations) for "m" method.

For "m" method, the modulus of subsoil reaction  $k_h$  is given by the following formula:

$$k_h = m(z - h)$$

- where:  $m$  - proportional coefficient of modulus of subsoil reaction [ $kN/m^4$ ]  
 $z$  - depth of the calculation point from the original ground [ $m$ ]  
 $h$  - depth of the calculation point from the ditch bottom at current stage of construction [ $m$ ]

From previous formula it's obvious that calculation of modulus  $k_h$  is linear with depth of point during the analysis.

Proportional coefficient  $m$  should be determined from pile test with horizontal load. If there are no test data, Chinese standard JGJ 120-2012 suggest an empirical formula to estimate this coefficient:

$$m = \frac{0.2\varphi^2 - \varphi + c}{v_b}$$

- where:  $c$  - cohesion (shear strength) of soil [ $kPa$ ]  
 $\varphi$  - angle of internal friction of soil [ $^\circ$ ]  
 $v_b$  - horizontal displacement of sheeting structure at the ditch bottom [ $mm$ ]; when the displacement is smaller than 10 mm, then  $v_b = 10\text{ mm}$

Other methods ("c" method and "k" method) are not published in JGJ 120-2012 standard, but they are based on practical experience and they are used in China profusely. Then the modulus of subsoil reaction  $k_h$  is given by the following formula:

$$k_h = a (z - h)^n$$

If exponent  $n = 0,5$ , it's "**c**" **method** and then  $a = c (kN/m^{3.5})$ .

If exponent  $n = 0$ , it's "**k**" **method** and then,  $a = K (kN/m^3)$ .

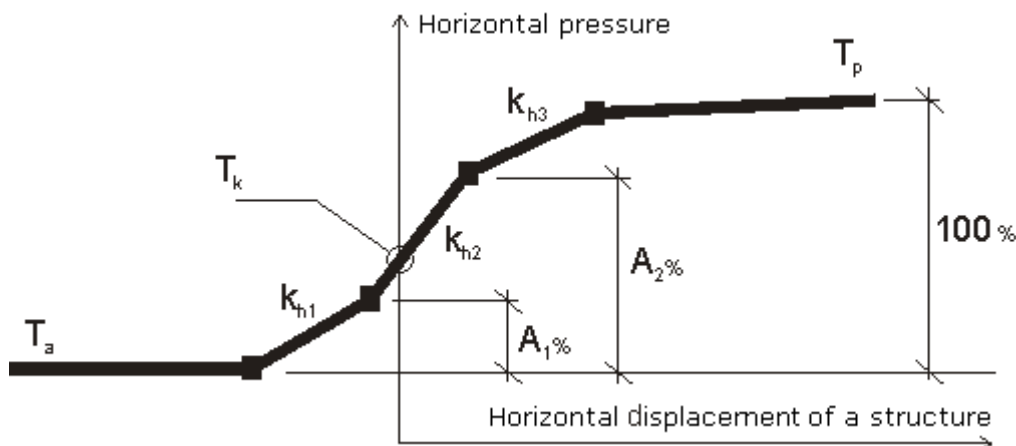
It's obvious that for "**m**" **method**,  $n = 1$ .

Literature:

JGJ 120-2012 (Technical specification for retaining and protection of building foundation excavations).

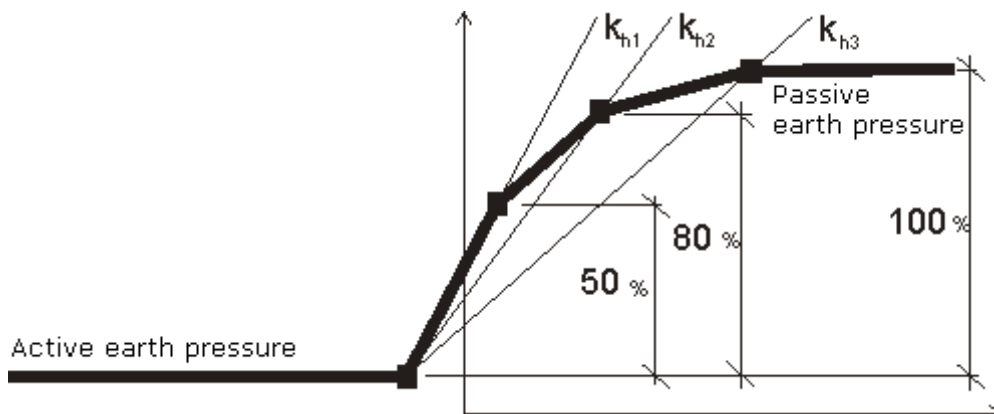
## Nonlinear Modulus of Subsoil Reaction

Nonlinear model describes dependence of the modulus of subsoil reaction  $k_h$  - i.e. change of  $k_h$  in between the threshold values corresponding to failure due to passive earth pressure  $T_p$  and active earth pressure  $T_a$  - see figure (the modulus of subsoil reaction is given by the slope of the curve; for pore pressure at rest acting on a structure it is possible to consider the value of  $k_{h1}$ ). This model also accounts for spring supports and forced deflections of the structure, various boundary conditions, application of struts and anchors, etc.



Interaction model to determine  $k_h$

The values of the modulus of subsoil reaction can be derived subsequently from the values of secant modulus of subsoil reaction (CUR 166) - see figure:



Interaction model to determine  $k_h$  - CUR 166

## Braced Sheet piling

When **analyzing** braced sheet piling (**pile curtain, steel I-section** or **user input of  $A, I, E, G$** ) the following approach is adopted to determine the **earth pressures**:

Up to the depth of ditch the pressures are determined with respect to 1 *rm* of the structure width. Below the ditch bottom the earth pressures are multiplied by the coefficient of reduction  $k$  (the "**Coefficient of pressure reduction below ditch bottom**"). This coefficient can be input (in the frame "**Geometry**" as a parameter of the section of a structure) or **automatically calculated**.

If "**Landfill of soil**" above the ditch (frame "**Excavation**") is input, then the pressures within this section are computed with respect to the whole width of wall ( $k = 1$ ).

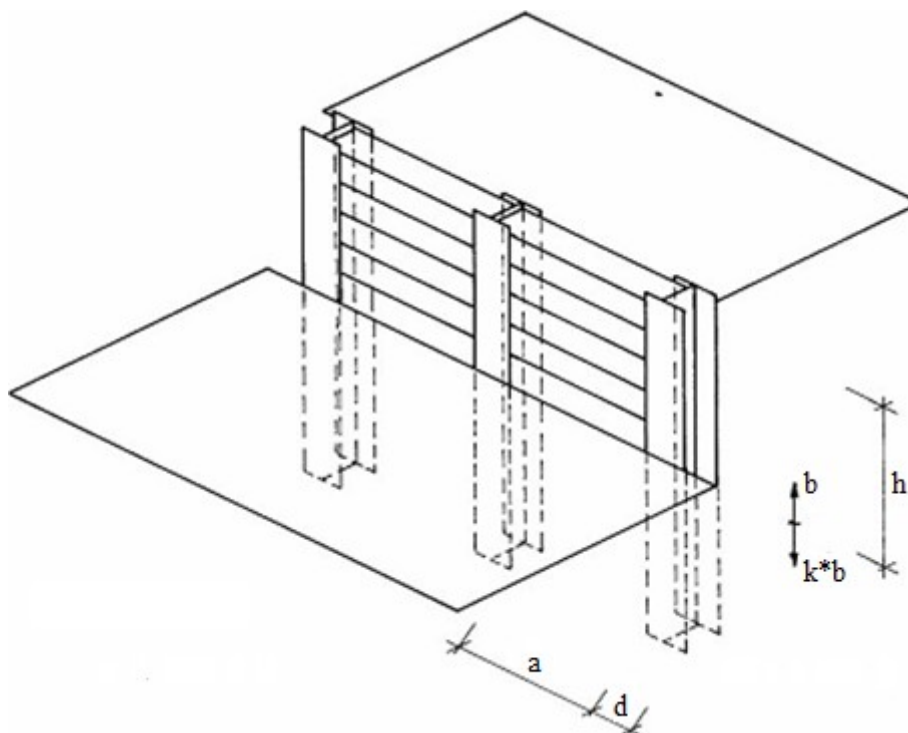
**The coefficient of pressure reduction below ditch bottom  $k$**  can be approximately determined (for very conservative design) according to:

$$k = \frac{d}{d + a}$$

where:

- $a$  - longitudinal spacing of soldier beams, or spacing between piles
- $d$  - width of soldier beam, or diameter of pile

**Real value of the coefficient  $k$**  also depends on the soil type and space effect of **earth pressure**. Real values of this coefficient (based on experiments) are **two to three-times higher** than values calculated with the help of the previous formula.



*Braced sheet piling - schematic view of retaining structure*

## Automatic Calculation of the Coefficient of Pressure Reduction Below Ditch Bottom

For **automatic calculation**, the coefficient of pressure reduction below ditch bottom  $k$  [-] is determined as follows:

- **circular pile curtain (a)**

$$k = 0.9 (1.5d + 0.5)/a \quad (d \leq 1 \text{ m})$$

$$k = 0.9 (d + 1)/a \quad (d > 1 \text{ m})$$

- **rectangular pile curtain or steel I-section (b)**

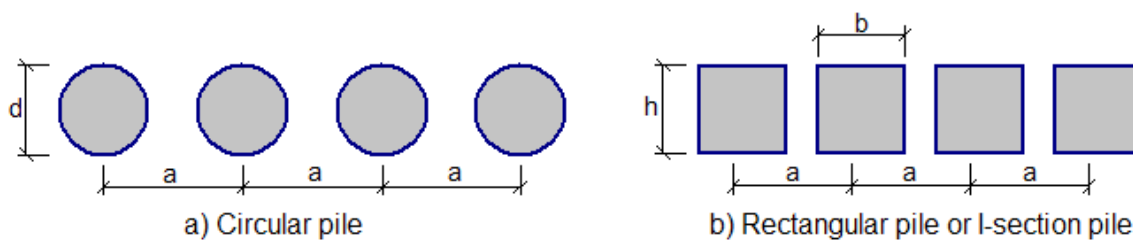
$$k = (1.5b + 0.5)/a \quad (b \leq 1 \text{ m})$$

$$k = (b + 1)/a \quad (b > 1 \text{ m})$$

**Note:** If coefficient  $k > 1$ , then  $k = 1$ .

where:

- $d$  - pile diameter
- $b$  - footprint of rectangular pile, or flange width of steel I-section
- $a$  - spacing of soldier beams, or spacing between piles

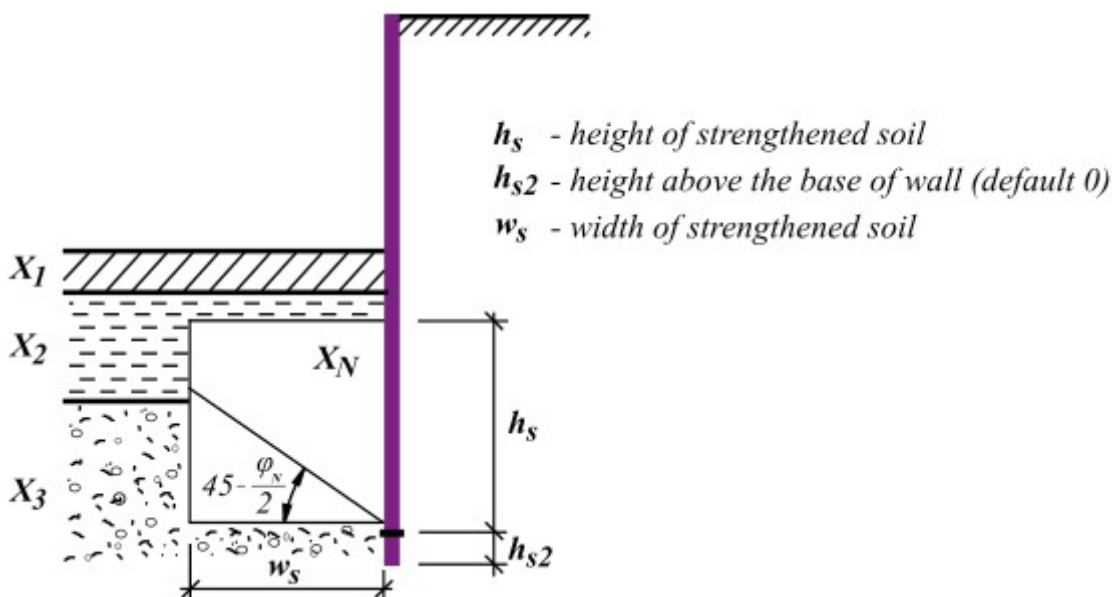


*The coefficient of pressure reduction below ditch bottom  $k$*

## Strengthening of the Soil

The program can model strengthening of the soil at the heel of **sheeting structure**. The strengthening of the soil is performed after installation of piles, or walls which are grouted at the base of the wall. The decisive parameters are **height of strengthened soil**  $h_s$ , or **width of strengthened soil**  $w_s$  and **parameters of strengthened soil** ( $\varphi, c$ ).

The principle of solution is shown in the following figure.



Strengthening of the soil at the base of sheeting structure - graphic presentation of the principle of solution

The principle of calculation for strengthening of the soil at the base of sheeting structure is described herein:

$$\mu = \text{Min} \left( \frac{w_s \tan \left( 45 - \frac{\phi_N}{2} \right)}{h}; 1 \right)$$

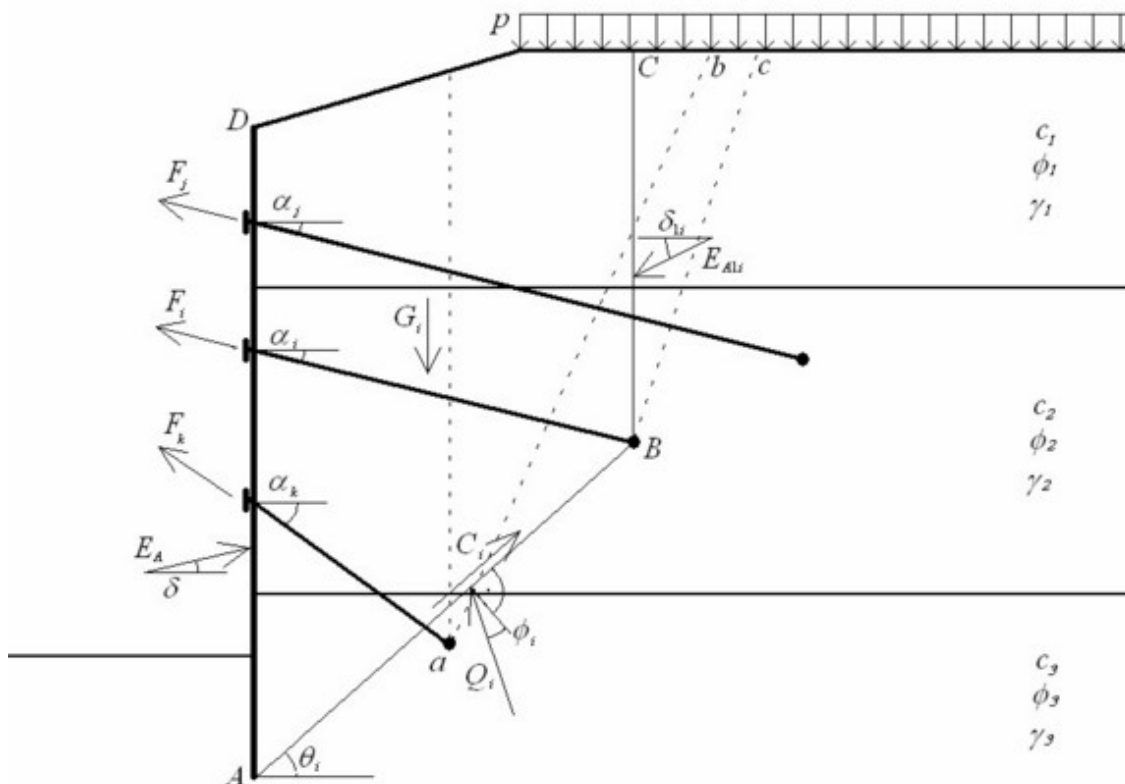
$$X_i = X_{oi}(1 - \mu) + \mu X_N$$

- where:
- $N$  - new layer of strengthened soil
  - $\mu$  - ratio (auxiliary parameter)
  - $X_i$  - arbitrary parameter at  $i^{th}$  layer of soil
  - $X_{oi}$  - original parameter at  $i^{th}$  layer of soil
  - $X_N$  - new parameter of strengthened soil
  - $w_s$  - width of strengthened soil [m]
  - $h_s$  - height of strengthened soil [m]

## Internal Stability of Anchors

The internal stability of an anchorage system of sheeting is determined for each layer independently. The verification analysis determines an anchor force, which equilibrates the system of forces acting on a block of soil. The block is outlined by sheeting, terrain, line connecting the heel of sheeting with anchor root and by a vertical line passing through the center of anchor root and terrain. The theoretical footing of sheeting construction is the point where the sum of horizontal forces under the bottom of the construction pit is equal to zero. If

this point lies under the footing of the sheeting wall, the theoretical point is the footing of this wall. The analysis is performed per one running meter of a sheeting structure. Anchor forces are therefore computed with respect to their spacing in individual layers.



Analysis of internal stability

Scheme for verification of the  $i^{th}$  layer of anchors is shown in the figure. The force equilibrium for the block  $ABCD$  is being determined. The following forces enter the analysis:

- $E_A$  - resultant of active earth pressure acting on sheeting (on line  $AD$ )
- $E_{Ai}$  - resultant of active earth pressure above the root of verified anchor (on line  $BC$ )
- $G_i$  - weight of the  $i^{th}$  the soil block  $ABCD$ ; in addition, this value incorporates the surcharge  $p$  applied on the ground surface providing the slope  $\theta_i$  of slip surface  $AB$  is greater than an average value of the angle of internal friction on this surface; in case of a smaller slope of slip surface  $AB$  the ground surcharge is not considered
- $C_i$  - resultant of soil cohesion on slip surface  $AB$
- $F_j, F_k$  - forces developed in other anchors, but some of them are not taken into account;
- ... only "**shorter**" anchors (comparing with the  $i^{th}$  anchor) will contribute in the equilibrium analysis of the  $i^{th}$  block; following principle is used to decide whether the given anchor (the  $m^{th}$ ) is included or excluded from equilibrium of the  $i^{th}$  block:

at first the lower anchor is selected (the  $m^{th}$  or the  $i^{th}$ ); then a plane slip surface is placed from the root center of the selected lower anchor; this plane is inclined  $45^\circ - \varphi_n/2$  from vertical line (line  $ab$  or  $Bc$  in the figure);  $\varphi_n$  is an average value of the angle of internal friction above the root of the lower anchor; if the  $i^{th}$  root is found

above the  $m^{th}$  one and the higher located root (the  $i^{th}$ ) is outside the area cut by the plane slip surface, then the  $m^{th}$  anchor is included into analysis; the same example of including the  $m^{th}$  anchor is when the  $i^{th}$  root lies under the  $m^{th}$  one and the  $m^{th}$  root is located inside the area cut by the slip surface; two opposite cases determine excluded anchors from analysis; first is the  $i^{th}$  root above the  $m^{th}$  one and the  $i^{th}$  inside the area, second is when the  $i^{th}$  root lies under the  $m^{th}$  one and the  $m^{th}$  is outside the area; from above definition resulting that "shorter" anchor  $F_k$  is included into analysis and "longer" anchor  $F_j$  is excluded from analysis (see figure)

$Q_i$  - reaction on slip surface  $AB$

$F_i$  force in the analyzed anchor, the maximum allowable magnitude of this force is the result of the equilibrium analysis carried out for the  $i^{th}$  block

Solution of the equilibrium problem for a given block requires writing down vertical and horizontal force equations of equilibrium. These represent a system of two equations to be solved for the unknown subsoil reaction  $Q_i$  and the maximum allowable magnitude of the anchor force  $F_i$ .

As a result the program provides the maximum allowable anchor forces for each row of anchors. These are them compared with those actually prescribed in anchors.

## Failure by Heave

### Failure by heave

The stability of soil against heave due to flow of water in the subsoil (HYD) is checked according to the **limit states** by equation:

$$u_{dst} \leq \frac{\sigma_{stb}}{\gamma_h}$$

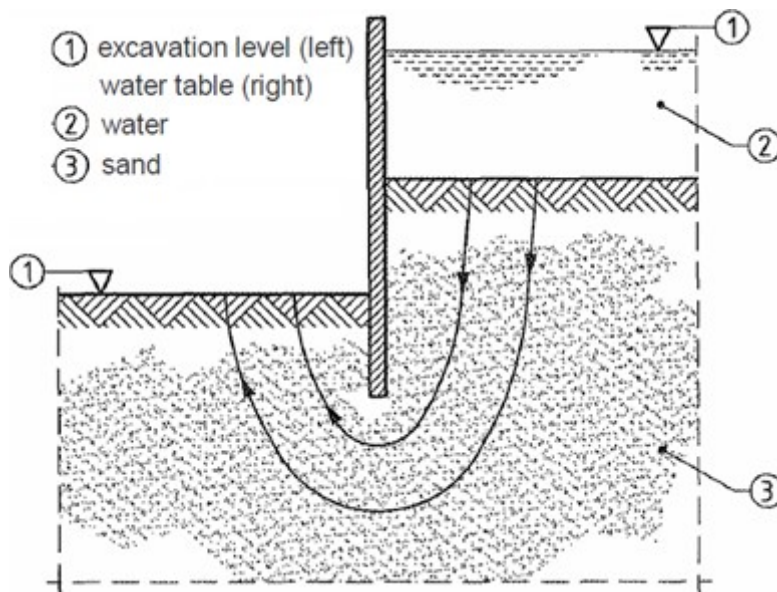
where:  $u_{dst}$  - unfavourable **water pressure**  
 $\sigma_{stb}$  - favourable **weight of soil**  
 $\gamma_h$  - reduction coefficient of failure by heave

The stability of soil against heave due to flow of water in the subsoil (HYD) is checked according to the **factor of safety** by equation:

$$\frac{\sigma_{stb}}{u_{dst}} \geq SF_h$$

where:  $u_{dst}$  - unfavourable **water pressure**  
 $\sigma_{stb}$  - favourable **weight of soil**  
 $SF_h$  - **safety factor** for failure by heave





Failure by heave - scheme of sheeting structure

### Failure by piping

Failure by piping is checked according to the **limit states** by equation:

$$i \leq \frac{i_c}{\gamma_p}$$

where:

- $i$  - hydraulic gradient
- $i_c$  - critical hydraulic gradient, where  $i_c = \gamma_{su}/\gamma_w$
- $\gamma_p$  - reduction coefficient of internal erosion of soil

Failure by piping is checked according to the **factor of safety** by equation:

$$\frac{i_c}{i} \geq SF_p$$

where:

- $i$  - hydraulic gradient
- $i_c$  - critical hydraulic gradient, where  $i_c = \gamma_{su}/\gamma_w$
- $SF_p$  - safety factor for internal erosion of soil

Literature:

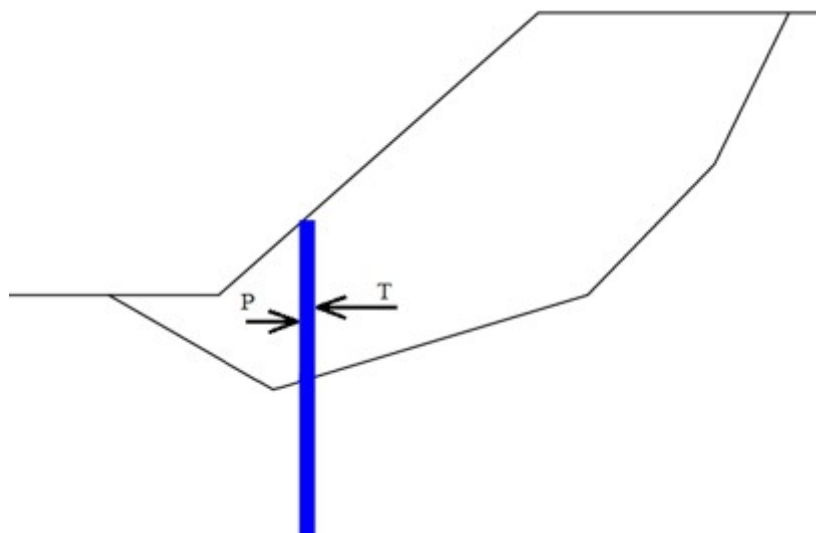
*Eurocode 7: Geotechnical design - Part 1: General rules.*

## Anti-Slide Pile

Program Anti-slide pile performs analysis of anti-slide piles (calculation of **internal forces + displacement, dimensioning of pile cross-section**). The analysis of structure is almost the same as in the program "**Sheeting check**", the different is **determination of pressures** above the slip surface and possibility to model pile **fixed into the rock**.

If there is for an input slope or structure found an unacceptable slip surface, then it is possible

to increase the slope stability on this surface by inserting an anti-slide pile ("Anti-Slide piles" frame serves to perform this step in the "Slope stability" program). The pile has to be placed such that it crosses the slip surface and its base is found sufficiently deep below the assumed slip surface. Above the slip surface the pile is loaded by an active force  $T$  having tendency to shift the pile and by a passive (resisting) force  $P$  stabilizing the pile (see figure below). The difference between the passive and active forces creates a load that must be transferred by the pile to increase stability on a input slip surface up to required value of  $SF_s$ .



*Scheme of active and passive forces acting on anti-slide pile*

If the slope stability  $SF$  without an anti-slide pile is not sufficient, the active and passive forces are in equilibrium - the pile is not loaded and there is no need to use it. For calculation of forces acting on a pile it is thus important that the required factor of safety  $SF_s$  is greater than the one calculated for a given slip surface  $SF$  without a pile, or the required safety is larger than the one associated with a input slip surface in the absence of an anti-slide pile.

## Determination of Forces Acting on an Anti-Slide Pile

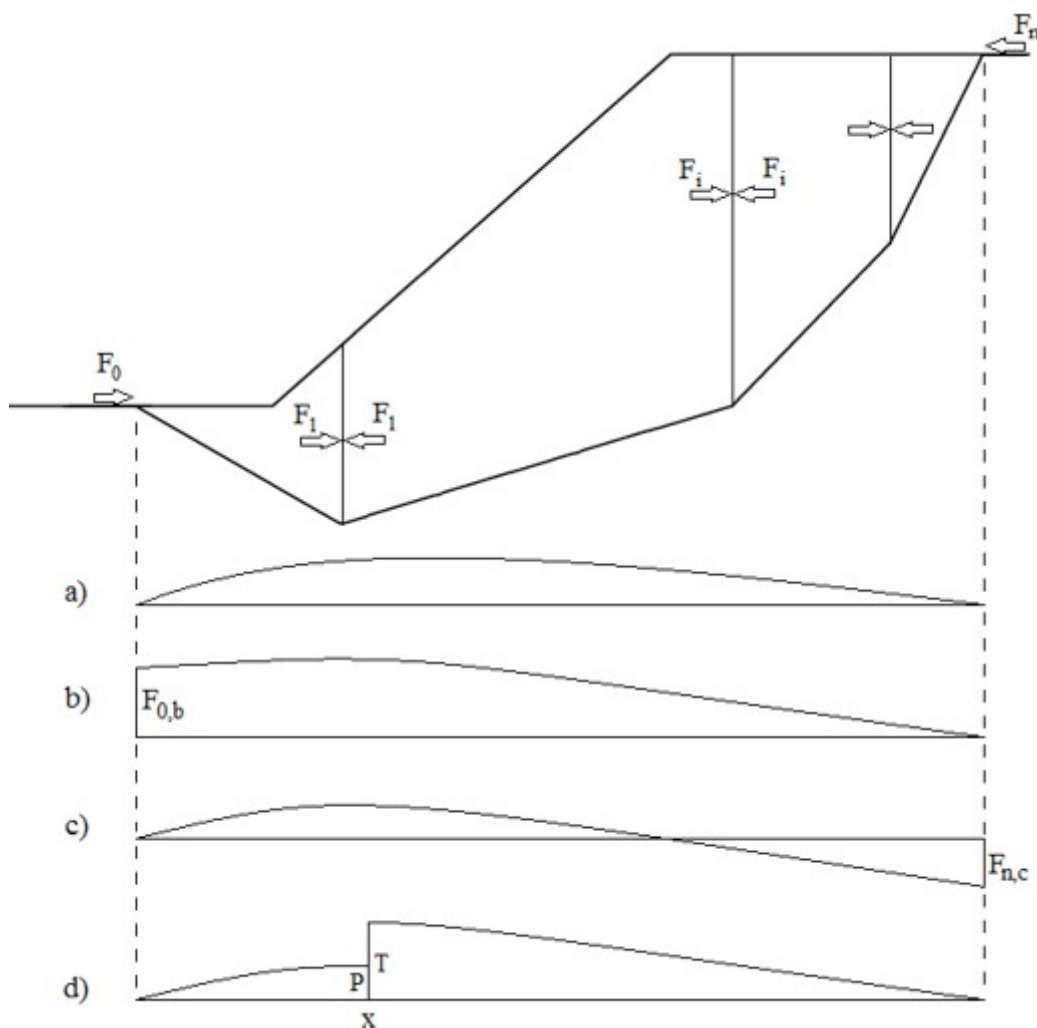
**Forces, acting on an anti-slide pile, are provided by the slope stability analysis.**

Calculation of slope stability  $SF$  is based on the equilibrium analysis of forces acting on blocks of soil above the slip surface. Vertical surfaces of individual blocks are subjected to action inter-block forces  $F_i$  and their determination is one of the steps of the slope stability analysis. If the soil blocks are exactly in the state of limit equilibrium, the inter-block forces at the beginning and at the end of the slip surface are equal to zero. The limit factor of safety  $SF_{lim}$ , for which the state of limit equilibrium is reached, is considered as a real factor of safety on an input slip surface. The distribution of inter-block forces along the sliding length is called the **pressure line**. The forces acting on an anti-slide pile are determined from the distribution of pressure lines calculated for the required factor of safety  $SF_s$ .

The following figure shows different distributions of inter-block forces  $F_i$  (pressure lines).

**Graph a)** displays the distribution of forces  $F_i$  in the state of limit equilibrium, where zero values of inter-block forces are found both at the beginning and at the end of the distribution. It means that this state has been found for the value of factor of safety  $SF_{lim}$ , which exactly expresses the degree of safety on a input slip surface. **Graph b)** displays the pressure line determined for a higher factor of safety than the one corresponding to  $SF_{lim}$ . An assumption of zero force  $F_n$  is adopted at the top of the slip surface and a non-zero force  $F_{0,b}$  at the bottom

of the slip surface is subsequently recorded. This means that in order to arrive at the factor of safety  $SF$ , the base of the slope would have to be loaded by the pressure force having the value of  $F_{0,b}$ . **Graph c)** displays the pressure line for a factor of safety  $SF$  greater than  $SF_{lim}$ . It results from the assumption of a zero force  $F_0$  on the bottom end of the slip surface at the top and then an unbalanced force  $F_{n,c}$  is encountered. Arriving at the state of equilibrium for the factor of safety  $SF$  would thus theoretically call for the action of a tensile force of this magnitude at the top end of the slip surface. **Graf d)** shows a distribution of inter-block forces for the case when an anti-slide pile is placed at point  $x$ . Part of the distribution corresponds to graph c) and serves to determine the magnitude of passive force  $P$  at point  $x$ . Above the pile the distribution according to graph b) is considered and serves to determine the active force  $T$  at point  $x$ . The difference between  $P$  and  $T$  is the force transmitted by the anti-slide pile.



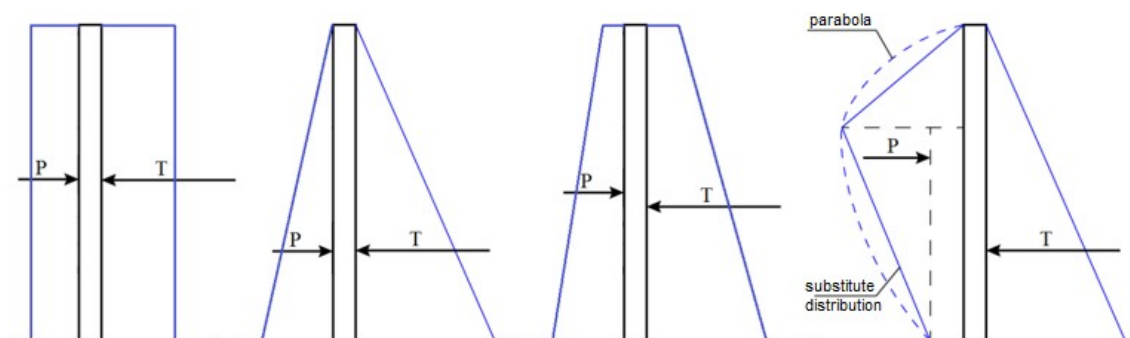
Distribution of inter-block forces  $F_i$

Figure caption (explanation):

- a)** distribution for factor of safety  $SF = SF_{lim}$  ( $F_0 = 0$  a  $F_n = 0$ )
- b)** distribution for factor of safety  $SF > SF_{lim}$  with zero value of force at the top end
- c)** distribution for factor of safety  $SF > SF_{lim}$  with zero force value at the bottom end
- d)** distribution for factor of safety  $SF$  with anti-slide pile at point  $x$

## Distribution of Pressures Above the Slip Surface

The distribution of load applied to an anti-slide pile above the slip surface is determined from the magnitudes of forces  $P$  and  $T$ . **Constant**, **triangular** or **trapezoidal** distributions are considered (for "**Anti-Slide Pile**" program the distribution of active and passive forces is introduced in the "**Determination of earth pressure**" frame). For the passive (resisting) force  $P$  can also be considered a **parabolic** distribution, which is approximated for the simplification by combining triangular and trapezoidal part.



*Types of distribution of load above the slip surface applied to an anti-slide pile above slip surface*

### Recommendation for active force distribution

- Triangle distribution - the layers above the slip surface are made from gravel or sand
- Rectangular distribution - the layers above the slip surface are made from fine grained soils (clay, silt)
- Trapezoidal distribution - the layers above the slip surface are made from different types of soils

## Shaft

In the frame "**Analysis**", program Shaft sets the **load** acting on the shaft.

The calculated load is then input for **analysis of internal forces** in the frame "**Dimensioning**".

## Calculation of Load Acting on a Shaft

The load from earth pressure and surcharge are computed in the frame "**Load analysis**". The stiffness of the shaft has a major influence on the earth pressure. Rigid structure does not allow deformation, so the earth pressure is much higher than on the flexible shaft.

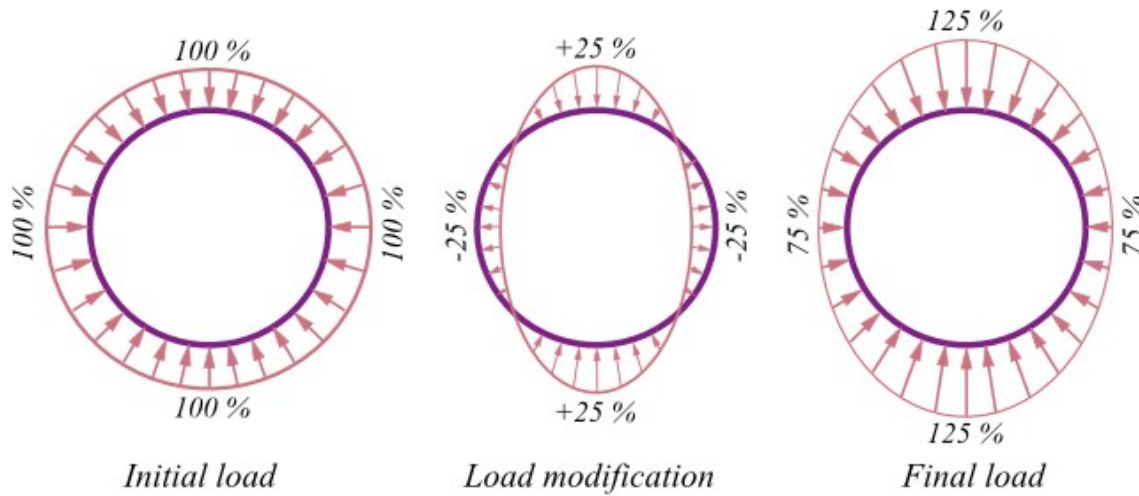
Three types of shafts can be modelled in the program:

- **Flexible** - spatial active pressure is considered (earth pressure, surface surcharge and **local surcharge**)
- **Semirigid**
- **Rigid** - spatial pressure at rest is considered (earth pressure, surface surcharge and **local surcharge**)

### Ways of load determination

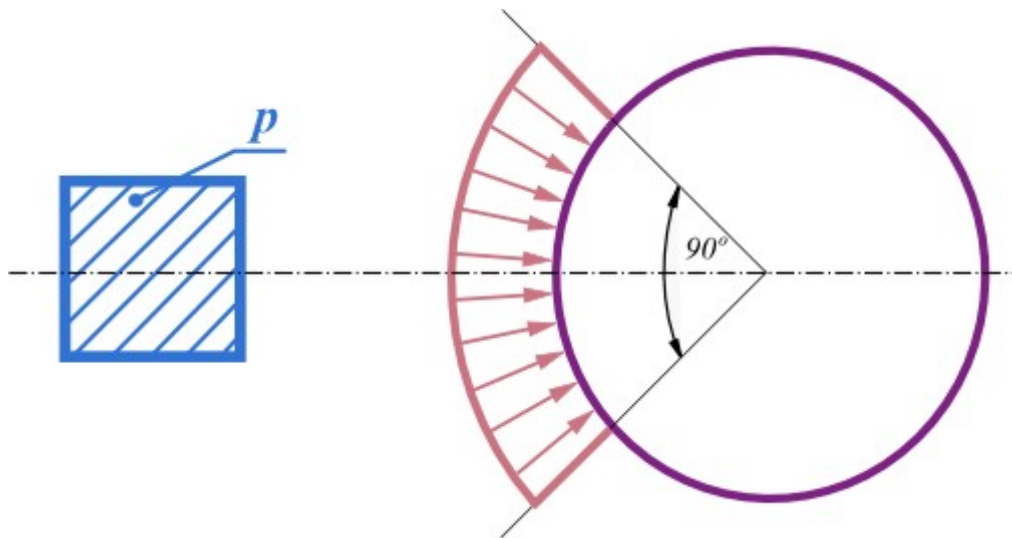
**Load from earth pressure** and surface surcharge acts as uniform load on the whole diameter. This load causes structure stress only by normal force - bending moment on the

shaft is theoretically equal to zero. For modelling of real behavior of the shaft, the program introduces the Reduction coefficient in compliance to the standards DIN V 4034-1 or СНиП II-94-80. Recommended value of reduction coefficient is 25 %.



*Modification of uniform load by reduction coefficient*

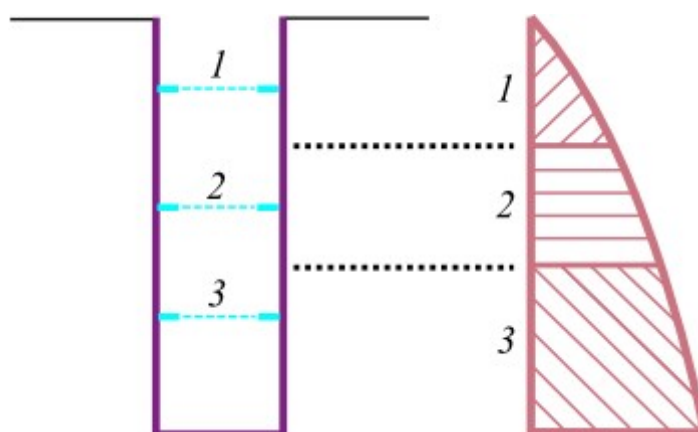
**Load from local surcharge** is considered as shown on the next picture. This load is not modified by reduction coefficient.



*Load from local surcharge*

**Recalculation of the load on walers**

If the walers are input the program computes the load on each of them. The load depends on the axial distance between walers as shown on the picture.



Calculation of load on the water

If no walers are input, the program computes load on unit depth (1 m, 1 ft).

If load is calculated, then program determines the distribution of **internal forces** on structure of the shaft.

*Literature:*

*Berezantzev, V. G.: Earth pressure on the cylindrical retaining walls, Brussels conference on Earth pressure problems, 1958.*

*ČSN 73 0037: Zemní tlak na stavební konstrukce, 1990.*

*DIN 4085: Berechnung des Erddrucks, 1987.*

*Exner, K.: Hloubení jam, VŠB v Ostravě, 1986.*

*Cheng, Y. M.; Hu, Y. Y.: Active earth pressure on circular shaft lining obtained by simplified slip line solution with general tangential stress coefficient. Chinese Journal of Geotechnical Engineering, 27 (1), 110-115, 2005.*

*Link, H.; Lutgendorf, H.; Stoss, K.: Richtlinien zur Berechnung von Schachtauskleidungen in nicht standfestem Gebirge, 1976.*

*Sedláček, M.: Zatížení kruhových šachet prostorovým zemním tlakem. Příspěvek ke konferenci Zakládání staveb, 2014.*

*Snášelová, K.: Hloubení a vyztužování jam v extrémních podmínkách, ODIS VTEI pro uhelný průmysl, 1987.*

*Tobar, T.; Meguid, M.: Distribution of active earth pressure on vertical shafts, Geo Halifax, 2009.*

*Valencia, T. T.: An experimental study of the earth pressure distribution on cylindrical shafts, McGill University, Montreal, 2009.*

*Walz, B.; Pulsfort, M.: Raumliche Erddruck auf Schachtbauwerke in Abhängigkeit von der Wandverformung, Bergische Universität Wuppertal, 1999.*

## Flexible Shaft Structure

Conventional excavation method is a typical example of using the shaft with **flexible** construction. Using this method, the rock is excavated in the first step and afterwards concrete is sprayed on to hardened surface to support the rock. There are specific technological downtime (ground excavation, shotcrete or arch timbering) with enabling stress rearrangement in the surrounding soil and the consequent value of the earth pressure acting

on the shaft will be equal to the active earth pressure. This fact is well described by V.G. Berezantsev's method (1958).

Load on flexible shaft is defined using this formula:

$$p_a = K_{a\gamma} \gamma h + K_{aq} q - K_{ac} c_{ef}$$

where:  $\gamma$  - unit weight of soil  
 $h$  - depth of cross-section  
 $q$  - magnitude of a surcharge  
 $c_{ef}$  - shear strength of soil

$$K_{a\gamma} = \frac{\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)}{\eta - 1} \left( \frac{r_0}{h} - \frac{r_0}{h R_b^{\eta-1}} \right)$$

$$K_{aq} = \frac{1}{R_b^\eta} \tan^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)$$

$$K_{ac} = \left[ \frac{1 - \lambda + \eta}{\eta} - \frac{\xi}{R_b^\eta} \tan^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right) \right] \cot \varphi_{ef}$$

where:  $r_0$  - radius of shaft  
 $\varphi_{ef}$  - angle of internal friction of soil  
 $\xi = 1$   
 $\lambda = 1$

$$\eta = \tan^2\left(\frac{\pi}{4} + \frac{\varphi_{ef}}{2}\right) - 1$$

$$R_b = 1 + \frac{h}{r_0} \tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)$$

*Literature:*

*Berezantsev, V. G.: Earth pressure on the cylindrical retaining walls, Brussels conference on Earth pressure problems, 1958.*

## Semirigid Shaft Structure

Sheet piles are a typical example of **semirigid** shaft support. The sheet piles are driven into the ground in the first construction stage and then the soil is excavated. The partial soil stress rearrangement in the shaft surrounding is permitted due to slowness of the construction. The resulting earth pressure value acting on the shaft construction moves between active and earth pressure.

Load on semirigid shaft is considered as an average value of load of flexible and rigid shafts.



## Rigid Shaft Structure

Shaft supported using secant pile walls is a typical example of the shaft with rigid construction. In the first construction stage the own shaft construction is built and therefore the soil is excavated. The load on the shaft due to earth pressure is equal to the at rest earth pressure because the rigid construction has minimal deformations. This construction behavior is well described by Y. M. and Y. Y. Hu theory (2005).

Load on rigid shaft is defined using this formula:

$$p_a = K_{ay} \gamma h + K_{aq} q - K_{ac} c_{ef}$$

where:  $\gamma$  - unit weight of soil  
 $h$  - depth of cross-section  
 $q$  - magnitude of a surcharge  
 $c_{ef}$  - shear strength of soil

$$K_{ay} = \frac{\tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)}{(1 - \sin \varphi_{ef}) Z} \left( \left[ 1 + Z \tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right) \right]^{1 - \sin \varphi_{ef}} - 1 \right)$$

$$K_{aq} = \frac{1}{\left[ 1 + Z \tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right) \right]^{\sin \varphi_{ef}}} \tan^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right)$$

$$K_{ac} = \left[ 2 - \frac{1}{\left[ 1 + Z \tan\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right) \right]^{\sin \varphi_{ef}}} \sec^2\left(\frac{\pi}{4} - \frac{\varphi_{ef}}{2}\right) \right] \cot \varphi_{ef}$$

wh  $r_0$  - radius of shaft  
ere

:

$\varphi_{ef}$  - angle of internal friction of soil

$$\eta = \sin \varphi_{ef}$$

$$Z = \frac{h}{r_0}$$

$r_0$  ratio of the depth of cut  $h$  to the radius of the shaft  $r_0$

$$\xi = \sec^2\left(\frac{\pi}{4} + \frac{\varphi_{ef}}{2}\right)$$



$$R_b = 1 + \frac{h}{r_0} \tan\left(\frac{\pi}{4} - \frac{\varphi_{sf}}{2}\right)$$

$$\lambda = 1 - \sin \varphi_{sf}$$

*Literature:*

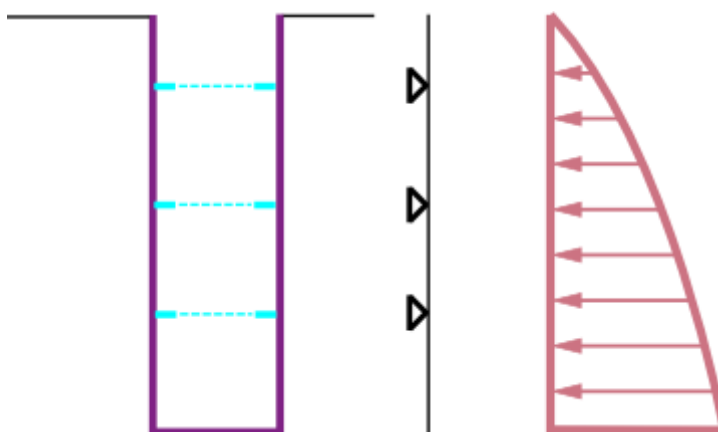
*Cheng, Y. M.; Hu, Y. Y.: Active earth pressure on circular shaft lining obtained by simplified slip line solution with general tangential stress coefficient. Chinese Journal of Geotechnical Engineering, 27 (1), 110-115, 2005.*

## Calculation of Internal Forces on a Shaft (Dimensioning)

The program allows analysis of internal forces acting on the structure in horizontal and vertical direction at an intended **load** acting on the shaft.

### Analysis of internal forces in vertical direction

Analytical model of the structure is shown on the picture. All walers are modelled as supports. The analysis is performed for a unit width (1 m, 1 ft) of the structure. Structures without walers or with one waler cannot be analysed in the vertical direction.

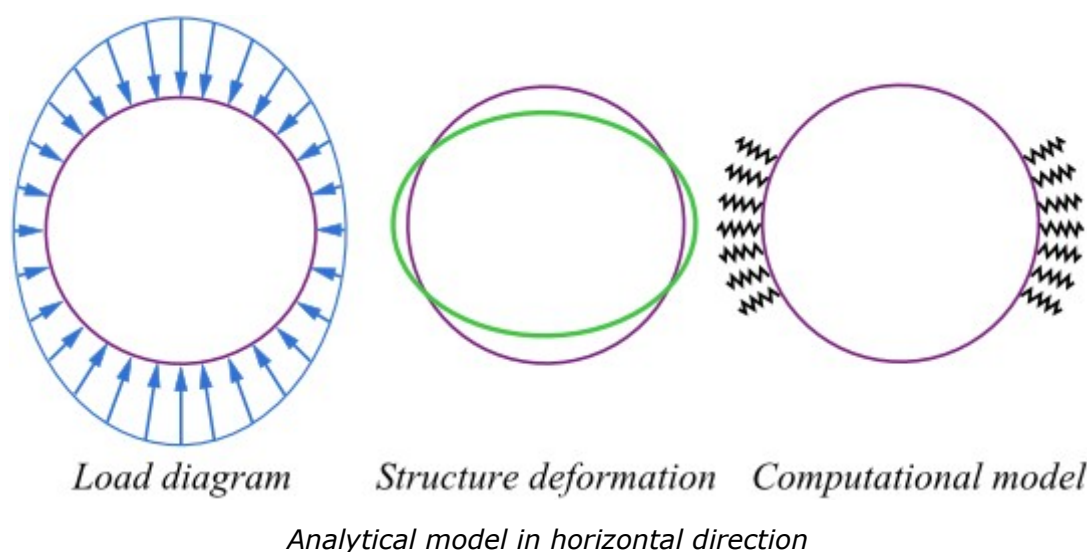


*Analytical model in vertical direction*

### Analysis of internal forces in horizontal direction (polygonal method)

Internal forces in the horizontal direction are computed by polygonal method, where circular structure is divided into 72 segments. Each segment is supported by non linear spring, acting only in compression. The stiffness of springs is equal to input modulus of subsoil reaction.

The way of analysis is shown on the picture - if the structure deforms in direction to the center, the springs are removed from the analysis.



## Slope Stability

**Slope stability** program compute stability of slopes and embankments with circular or polygonal slip surfaces.

The slope stability problem is solved in a two dimensional environment. The soil in a **slope body** can be found below the **ground water table**, water can also exceed the slope ground, which can be either partially or completely flooded. The slope can be loaded by a **surcharge** of a general shape either on the ground or inside the soil body. The analysis allows to include the **effect of anchors** expected to support the slope or for introduction of horizontal reinforcing elements - **reinforcements** or vertical elements - **anti-slide piles**. An **earthquake** can also be accounted for in the analysis.

**Two types of approaches** to the stability analysis are implemented in the program - classical analysis according to the factor of safety and the analysis following the theory of limit states.

The slip surface can be modeled in two different ways. Either as a **circular** one, then the user may choose either from the **Fellenius/Petterson, Bishop, Spencer, Janbu** or **Morgenstern-Price, Shahunyants, ITF** method, or as a **polygonal** one, in which case the program exploits the **Sarma, Spencer, Janbu** or **Morgenstern-Price, Shahunyants, ITF** method.

## Soil Body

The soil body is formed by a **layered profile**. An arbitrary number of **layers** can be used. Each layer is defined by its geometry and material. The material of a layer is usually represented by a **soil** with **specified properties**. The **geostatic stress** in soil body is determined during the analysis.

A layer can be specified also as a **rigid body**. Such layer then represents bedrock or a sheeting wall. The slip surface can never pass through the rigid body.

## Influence of Water

Ground water can be assigned to the slope plane section using one of the five options:

### 1) Ground water table

The ground water table is specified as a polygon. It can be arbitrarily curved, placed totally within the soil body or introduced partially **above the ground surface**.

Presence of water influences value of pore pressure acting within a soil and reducing its shear bearing capacity. The pore pressure is considered as the hydrostatic pressure, i.e. unit weight of water is multiplied by reduced height of the water table:

$$u = \gamma_w \cdot h_r$$

where:  $\gamma_w$  - unit weight of water  
 $h_r$  - reduced height of water table

$$h_r = h \cdot \cos^2 \alpha$$

where:  $h$  - vertical distance of point, where pore pressure is calculated and point on the water table  
 $\alpha$  - inclination of the water table

Resultant force of pore pressure at certain section of the block is used in the calculation:

$$U = u \cdot l$$

where:  $u$  - pore pressure in the point  
 $l$  - length of section

Below the ground water table the analysis proceeds using the unit weight of saturated soil  $\gamma_{sat}$  and uplift pressure; above the ground water table the analysis assumes the input unit weight of soil  $\gamma$ .

The shear forces along the slip surface are provided by:

$$T = (N - U) \operatorname{tg} \varphi + c \cdot d$$

where:  $T$  - shear force along slip surface segment  
 $N$  - normal force along slip surface segment  
 $U$  - pore pressure resultant along slip surface segment  
 $\varphi$  - angle of internal friction  
 $c$  - cohesion  
 $d$  - length of slip surface segment

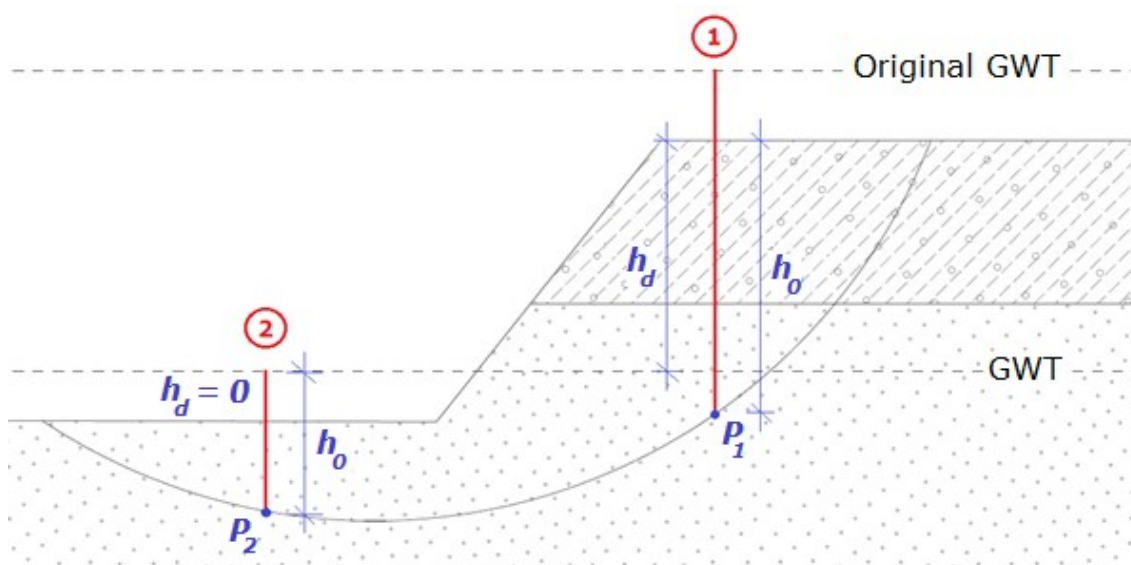
In case of total stress (entered in the "Soil" dialog window) total parameters are used and pore pressure is considered zero.

## 2) Ground water table including suction

**Suction table** can be introduced above the input ground water table. A negative value of pore pressure  $u$  is then assumed with the region separated by the two tables. Suction increases as negative hydrostatic pressure from the ground water table towards the suction table.

## 3) Rapid draw down

**Original table** can be introduced above the input ground water table. Original water table simulates state before rapid draw down.



Rapid draw down analysis

First of all, the initial pore pressure  $u_0$  is evaluated:

$$u_0 = \gamma_w \cdot h_0$$

where:  $h_0$  - height from original table to the point of evaluation P  
 $\gamma_w$  - unit weight of water

Height  $h_0$  is generally the distance from point of pore pressure evaluation (**P**) to original water table - this is valid for the case when original water table is under terrain surface. In case of original water table above terrain there is used the height  $h_0$  from point **P** to the level of terrain surface (**profile 1** in the figure). Another case is when original water table as well as ground water table are both above terrain - then height  $h_0$  is the distance from ground water table to point **P** (**profile 2** in the figure).

Second step is to calculate change of pore pressure in the area between original and ground water table:

$$\Delta u = \gamma_w \cdot h_d$$

where:  $h_d$  - height from original to ground water table  
 $\gamma_w$  - unit weight of water

As in previous calculation of pressure, there are three possibilities how to get height  $h_d$ . When both water tables are under terrain,  $h_d$  is the distance between original and ground water table. In case of original water table above terrain, then  $h_d$  is measured from ground water table to the level of terrain (**profile 1** in the figure). Last case is when both water tables are above terrain - then height  $h_d$  is zero (**profile 2** in the figure).

Third step is calculation of final value of pore pressure  $u$ . Change of pore pressure  $\Delta u$  is multiplied by coefficient of reduction of initial pore pressure  $X$ , which is required for all soils (dialog window "Soils").  $X$  coefficient of the soil in the area of point **P** is used (NOT soil in the area between original and ground water table). In case of permeable soil  $X = 1$ , in other case  $X = 0$ . Final pore pressure is evaluated as:

$$u = u_0 - X \cdot \Delta u$$

where:  $u_0$  - initial pore pressure  
 $X$  - coefficient of reduction of initial pore pressure  
 $\Delta u$  - change of pore pressure

#### 4) Coefficient of pore pressure $R_u$

The coefficient of pore pressure  $R_u$  represents the ratio between the pore pressure and geostatic pressure in a soil body. In the area, where  $R_u$  is positive, entered unit weight of saturated soil  $\gamma_{sat}$  is considered; in other case unit weight of soil  $\gamma$  is used.

The values of  $R_u$  are introduced with the help of isolines connecting points with the same value of  $R_u$ . Linear interpolation is assumed to obtain intermediate values. Pore pressure is established as geostatic stress reduced by coefficient  $R_u$ :

$$u = R_u \cdot \sum h_i \cdot \gamma_i$$

where:  $R_u$  - coefficient of pore pressure  
 $h_i$  - height of  $i^{th}$  soil layer  
 $\gamma_i$  - unit weight of  $i^{th}$  soil layer

#### 5) Pore pressure values

Ground water can be introduced directly through the pore pressure values  $u$  within the plane section of a soil body.

In the area, where  $u$  is positive, entered unit weight of saturated soil  $\gamma_{sat}$  is considered; in other case unit weight of soil  $\gamma$  is used.

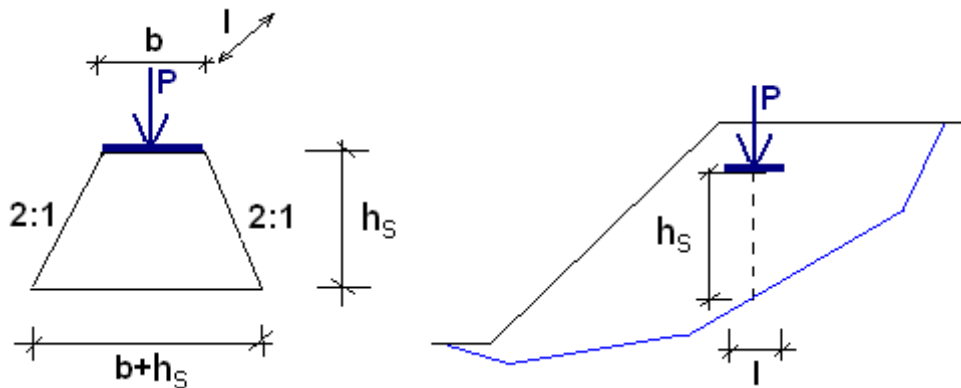
The pore pressure values are introduced with the help of isolines connecting points with the same value of pore pressure. Linear interpolation is assumed to obtain intermediate values. Pore pressure values are then derived from the values of pore pressure obtained in specific points within the slope plane section.

## Surcharge

The slope stability analysis takes into account even the surcharge caused by neighboring structures. The surcharge can be introduced either as a concentrated force or distributed load acting either on the ground surface or inside the soil body.

Since it is usually assumed that the surcharge is caused by the weight of objects found on the slope body, the vertical component of surcharge having the direction of weight (material component) is added to the weight of blocks. It means that if the **earthquake effects** are included this component is also multiplied by the factor of horizontal acceleration or vertical earthquake. Material surcharge component also influences the position of block centroid. The components that do not act in the direction of weight are assumed in equations of equilibrium written for a given block as weightless thus neither contribute to inertia effects of the earthquake nor position of block centroid.

The surcharge is always considered in the analysis with respect to one running meter. Providing the surcharge, essentially acting over the area  $b \cdot l$ , is introduced as a concentrated force it is transformed before running the analysis into a surface load spread up to a depth of slip surface along the slope 2:1 as displayed in figure.



*Scheme of spreading the concentrated load on the slip surface*

The analysis then proceeds with the resultant of surface load  $p$  having the value:

$$p = \frac{P}{(b + h_s)l}$$

## Anchors

Anchor is specified by two points and a force. The first point is always located on the ground surface; the force always acts in the direction of a soil body. The anchor force when computing equilibrium on a given block (slice) is added to the weightless surcharge of the slope.

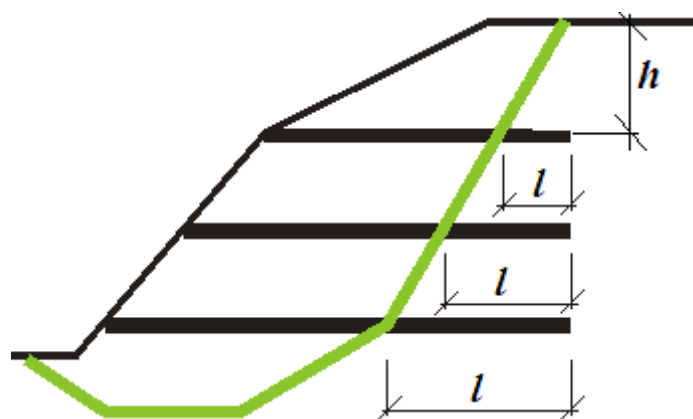
Two options are available to account for anchors:

1. **Compute anchor lengths** - analysis assumes infinite lengths of anchors (anchors are always included in the analysis) and computes the required lengths of links anchors (distance between the anchor head and intersection of anchor with the slip surface) subsequently. The anchor root is then placed behind the slip surface. This approach is used whenever we wish the anchor to be always active and thus contribute to increase the slope stability and we need to know its minimum distance.
2. **Analysis with specified lengths of anchors** - the analysis takes into account only those anchors that have their end points (center of roots) behind the slip surface. This approach is used always whenever we wish to evaluate the current state of slope with already existing anchors, since it may happen that some of the anchors may prove to be short to intersect the critical slip surface so that they do not contribute to increase the slope stability.

## Reinforcements

Reinforcements are horizontal reinforcing elements, which are placed into the soil to increase the slope stability utilizing their tensile strength. If the reinforcement intersects the slip surface, the force developed in the reinforcement enters the force equation of equilibrium of a given block. In the contrary case, the slope stability is not influenced.

The basic parameter of reinforcement is the **tensile strength**  $R_t$ . A design value of this parameter is used - i.e. the strength of reinforcement reduced by coefficients taking into account the effect of durability, creep and installation damage. The force transmitted by reinforcement **can never exceed the assigned tensile strength**  $R_t$ .



*Scheme of accounting for reinforcement*

The second characteristic is the **pull-out strength**  $T_p$ . This parameter determines the anchoring length, i.e. the required length of reinforcement in the soil, for which the reinforcement is fully stressed attaining the value  $R_t$ . Since the realistic values of the pull-out strength are difficult to determine, the program offers three options for their calculation, respectively for the calculation of the force  $F$  transmitted by the reinforcement.

### 1) Calculate reinforcement bearing capacity

The pull-out force  $F$  is given by:

$$F = 2 \cdot \sigma \cdot \tan \varphi \cdot C \cdot l$$

where:

- $\sigma$  - normal stress due to self weight at the intersection of reinforcement and slip surface - see Fig.
- $\varphi$  - angle of internal friction of soil
- $C$  - coefficient of interaction (0,8 by default)
- $l$  - length of reinforcement step joint behind the slip surface into the soil body

### 2) Input reinforcement anchor length $l_k$

An anchoring length  $l_k$  is specified. This parameter is determined by the shear strength developed between the reinforcement and the soil gradually increasing from zero to its limit value (measured from the end of reinforcement fixed in soil).

$$F = \frac{l}{l_k} \cdot R_t$$

where:

- $l$  - length of reinforcement behind the slip surface into the soil body
- $l_k$  - anchoring length of reinforcement
- $R_t$  - tensile strength

### 3) Input reinforcement pull-out resistance $T_p$

The pull-out force  $F$  is given by:

$$F = T_p \cdot l$$

where:

- $l$  - length of reinforcement behind the slip surface into the soil body
- $T_p$  - pull-out resistance of reinforcement

Forces in reinforcements determined on the basis of reinforcement strength may attain relatively large values. Introducing these forces in the analysis yields a higher factor of safety of a given slip surface. In case of rigorous methods (Spencer, Janbu, Morgenstern-Price) the introduction of such forces in the reinforcements may cause the loss of convergence. This appears mainly in cases when these forces are so high that it is not possible to achieve equilibrium of forces acting on blocks while maintaining the principal assumptions of individual methods, e.g. the assumption of zero moment at the end of slip surface. In such a case the forces in reinforcements are reduced as least as possible (to the highest acceptable values) so the method converges and attains acceptable results. The reduced values of forces are then written out as part of the stability analysis results. However, in case of no reduction these forces are not included in the final set of results.

## Reinforcement End

The reinforcement mounting is assumed in the program either as **fixed** or **free**.

Should the slope with **reinforcement** fail the one of the following **reinforcement** failure shown in the following figures may appear.

If the **reinforcement** at its starting point in front of the slip surface is fixed (for example fixed into the structure cladding) the 3rd type of failure is prevented - pullout of the reinforcement in front of the slip surface. The failure type 1 and 2 is always checked in the analysis, type of failure 3 is checked only for reinforcements having free end points that allow for such a type of failure.

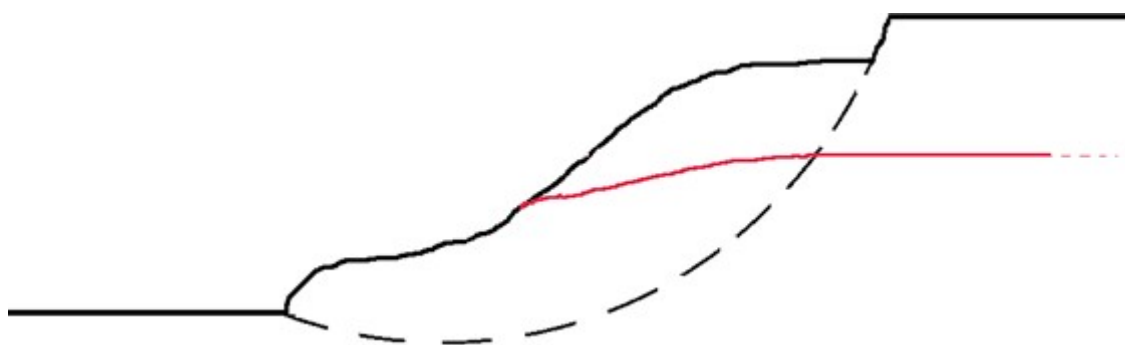
The image shows a software dialog box titled "New reinforcement". It contains several input fields and buttons. The "End of reinforcement" dropdown menu is highlighted with a red rectangular box and is currently set to "Fixed". Other visible fields include "Point to the left" (x=8,00, z=115,02), "Point to the right" (x=34,06, z=115,20), "Length" (L=26,06), "Tensile strength" (R<sub>t</sub>), "Analyses of bearing capa" (Calculate bearing capacity), and "Coefficient of interaction" (C=0,80). There are also buttons for "Extend to the left", "Extend to the right", "OK", and "Cancel".

"New reinforcement" dialog window - input of end of reinforcement





*Breaking of reinforcement - type of failure 1*



*Pullout of the reinforcement behind the slip surface - type of failure 2*

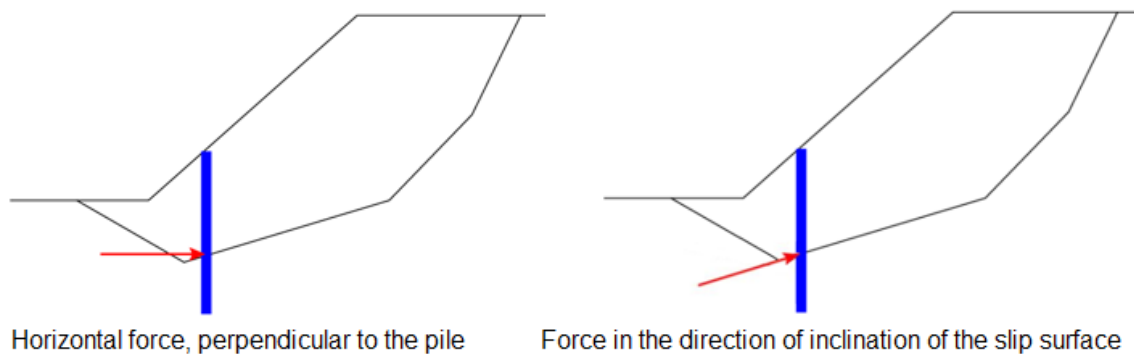


*Pullout of the reinforcement in front of the slip surface - type of failure 3*

## Anti-Slide Piles

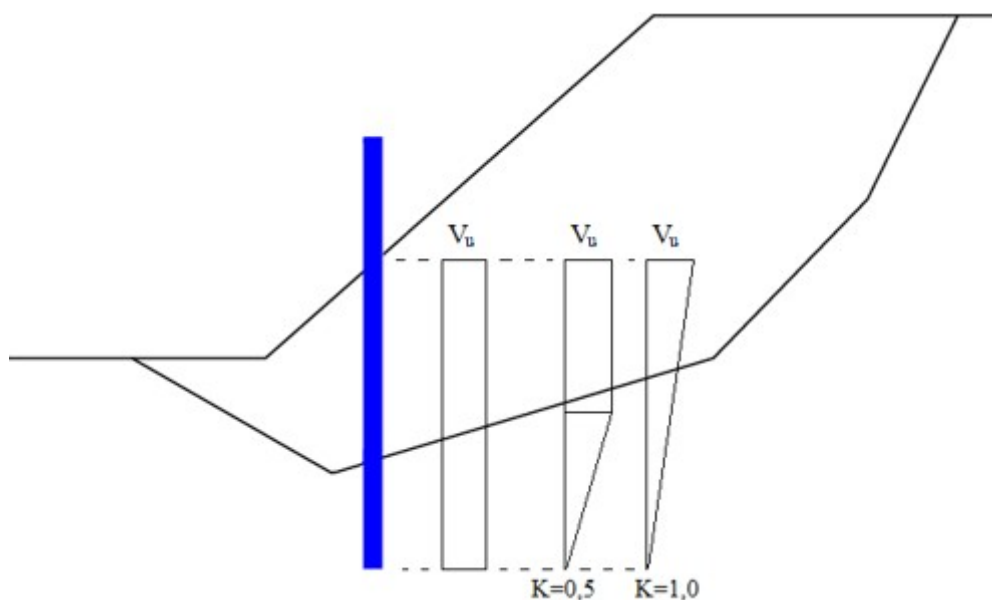
Anti-slide piles are vertical structural elements, which increase the slope stability. If the anti-slide pile intersects into assessed slip surface, then for the calculation of the factor of safety is introducing passive (resisting) force  $P$  which corresponds to bearing capacity of pile  $V_u$ . This step is accomplished by the higher value of safety factor  $SF$ .

It is assumed that the **pile is always vertical**. Passive (resisting) force  $P$  at the intersection point with the slip surface is considered either in the horizontal direction or in a direction which corresponds to the inclination of the slip surface at that location.



*Presentation of the direction of passive (resisting) force*

The value of passive (resisting) force  $P$  is always determined at 1 *rm* width of the slope with respect to spacing between piles. Bearing capacity of pile  $V_u$  can be specified as either with a **constant value** along the length of pile, or **increasing linearly** away from the pile base upwards.



*Constant and linear distribution of bearing capacity  $V_u$  along the pile length*

Linear increase of bearing capacity of pile is described by **gradient  $K$** , which is the ratio of the pile length, on which the ultimate bearing capacity  $V_u$  is achieved due to the length of pile below the ground surface. If the value of gradient  $K$  approaches zero, the linear distribution of bearing capacity  $V_u$  is close to constant distribution.

Program also determines **active and passive forces** acting on anti-slide piles above the slip surface and allows to send data to the program **Anti-slide pile**, where other analyses can be performed.

## Influence of an Earthquake

Program allows to calculate influence of earthquake according to the following standards:

- **Standard analysis**
- **Earthquake analysis according to the chinese standard JTJ 004-89**

- Earthquake analysis according to the chinese standard SL 203-97

The advantage of **chinese standards** is to establish the intensity of the earthquake, according to which the program automatically assigned values of the coefficient  $K_h$  appropriate standards.

## Earthquake Effect - Standard Analysis

The program allows to compute the earthquake effects with the help of two variables - factor of horizontal acceleration  $K_h$  or the coefficient of vertical earthquake  $K_v$ .

### Coefficient of vertical earthquake $K_v$

The coefficient of vertical earthquake either decreases ( $K_v > 0$ ) or increases ( $K_v < 0$ ) the unit weight of a soil, water in a soil and material **surcharge** by multiplying the respective values by  $1 - K_v$ . It is worth to note that the coefficient  $K_v$  may receive both positive and negative value and in case of sufficiently large coefficient of horizontal acceleration the slope relieve ( $K_v > 0$ ) is more unfavorable than the surcharge.

### Factor of horizontal acceleration $K_h$

In a general case the computation is carried out assuming a zero value of the factor  $K_h$ . This constant, however, can be exploited to simulate the effect of earthquake by setting a non-zero value. This value represents a ratio between horizontal and gravity accelerations. Increasing the factor  $K_h$  results in a corresponding decrease of the safety factor  $SF$ .

The coefficient of horizontal acceleration introduces into the analysis an additional horizontal force acting in the center of gravity of a respective block with the magnitude  $K_h * W_i$ , where  $W_i$  is the block overall weight including the material component of the slope surcharge.

The following table lists the values of the factor  $K_h$  that correspond to different degrees of earthquake based on M-C-S scale.

M-C-S degree	Horizontal acceleration		Factor of horizontal acceleration	
(MSK-64)	[mm/s <sup>2</sup> ]		$K_h$	
1	0,0	- 2,5	0,0	- 0.00025
2	2,5	- 5,0	0,00025	- 0.0005
3	5,0	- 10,0	0,0005	- 0.001
4	10,0	- 25,0	0,001	- 0.0025
5	25,0	- 50,0	0,0025	- 0.005
6	50,0	- 100,0	0,005	- 0.01
7	100,0	- 250,0	0,01	- 0.025
8	250,0	- 500,0	0,025	- 0.05
9	500,0	- 1000,0	0,05	- 0.1

10	1000,0	- 2500,0	0,1	- 0.25
11	2500,0	- 5000,0	0,25	- 0.5
12		>5000,0		> 0.5

## Earthquake Analysis According to JTJ 004-89

Earthquake effects are in stability analysis represented by horizontal and vertical forces acting at the centers of gravity of individual soil blocks. Magnitude of these forces is related to the weight of soil blocks and is calculated using horizontal and vertical earthquake coefficients. Horizontal earthquake force is always oriented out from the slope massif. Vertical force can be directed upwards or downwards, the orientation is defined by the sign of the force.

Horizontal earthquake force  $E_{hs}$  is given by formula:

$$E_{hs} = C_i C_z K_h G_s$$

and vertical earthquake force  $E_{vs}$  is determined by:

$$E_{vs} = C_0 C_i C_z K_v G_s$$

- where:
- $C_i$  - importance coefficient for seismic design
  - $C_z$  - comprehensive influence factor
  - $C_0$  - meeting coefficient related to the influence of horizontal seismic effect
  - $K_h$  - coefficient of horizontal seismic acceleration
  - $K_v$  - coefficient of vertical seismic acceleration
  - $G_s$  - weight of the soil block

## Earthquake Analysis According to SL 203-97

Earthquake effects are in stability analysis represented by horizontal and vertical forces acting at the centers of gravity of individual soil blocks. Magnitude of these forces is related to the weight of soil blocks and is calculated using horizontal and vertical earthquake coefficients. Earthquake coefficients are depended on position gravity center of each block. Therefore the coefficients have individual and different values for each one of soil blocks. Horizontal earthquake force is always oriented out from the slope massif. Vertical force can be directed upwards or downwards, the orientation is defined by the sign of the force.

Horizontal earthquake force  $E_{hs}$  is given by formula:

$$E_{hs} = C_i C_z \alpha_i K_h G_s$$

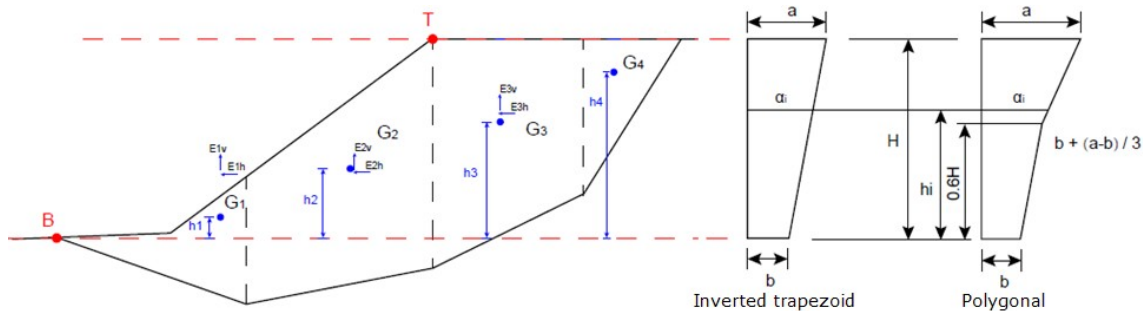
and vertical earthquake force  $E_{vs}$  is determined by:

$$E_{vs} = C_0 C_i C_z \alpha_i K_v G_s$$

- kde:
- $C_i$  - importance coefficient for seismic design
  - $C_z$  - comprehensive influence factor

- $C_0$  - meeting coefficient related to the influence of horizontal seismic effect
- $K_h$  - coefficient of horizontal seismic acceleration
- $K_v$  - coefficient of vertical seismic acceleration
- $G_S$  - weight of the soil block
- $\alpha_i$  - dynamic distribution coefficient of block  $i$

There are two types of dynamic distribution used for determining of  $\alpha_i$  value: inverted trapezoid and polygonal. Method for setting  $\alpha_i$  value could be seen at Figure.



Determination of dynamic distribution coefficient  $\alpha_i$

Height  $H$  of the range of  $\alpha_i$  is given by points **B** and **T**. The bottom point **B** is the lowest point of terrain above the slip surface and the top point **T** is the highest point of terrain above the slip surface.  $G_i$  denotes gravity center points of individual blocks and  $E_{ih}$ ,  $E_{iv}$  are horizontal and vertical earthquake forces.

## Verification According to EN 1997

When running the verification analysis according to EN 1997 the choice of a given "Design approach" and "Partial factors" is important. Forces and loads are reduced in all design approaches.

The value of capacity utilization  $V_u$  is calculated and then it is compared to 100%. The value of capacity utilization is given by:

$$V_u = \frac{M_a}{M_p} 100 < 100\%$$

- where:  $M_a$  - sliding moment  
 $M_p$  - resisting moment

In case of design approach 2 the resisting moment  $M_p$  is determined from non-reduced soil parameters but considering the reduction of resistance on the slip surface using the coefficient  $\gamma_{Rs}$ .

In case of design approach 1 and design approach 3 the program reduces for the determination of the overall resisting moment  $M_p$  the **strength parameters of soil** (angle of internal friction and cohesion).

## Analysis According to the Theory of Limit States / Safety Factor

The verification parameters are input in the "Stability analysis" tab. The structure can be verified according to the **factor of safety** or the theory of **limit states**.

Verification according to the **theory of limit states**:

Soil parameters (angle of internal friction, cohesion) are in this case **reduced using the design coefficients** introduced in the "Stability analysis" tab.

The value of utilization  $V_u$  is calculated and then compared with the value of 100%. The value of utilization is given by:

$$V_u = \frac{M_a}{M_p} 100 < 100\%$$

where:  $M_a$  - sliding moment  
 $M_p$  - resisting moment

The resisting moment  $M_p$  is determined considering the reduction with the help of overall stability of construction  $\gamma_s$ .

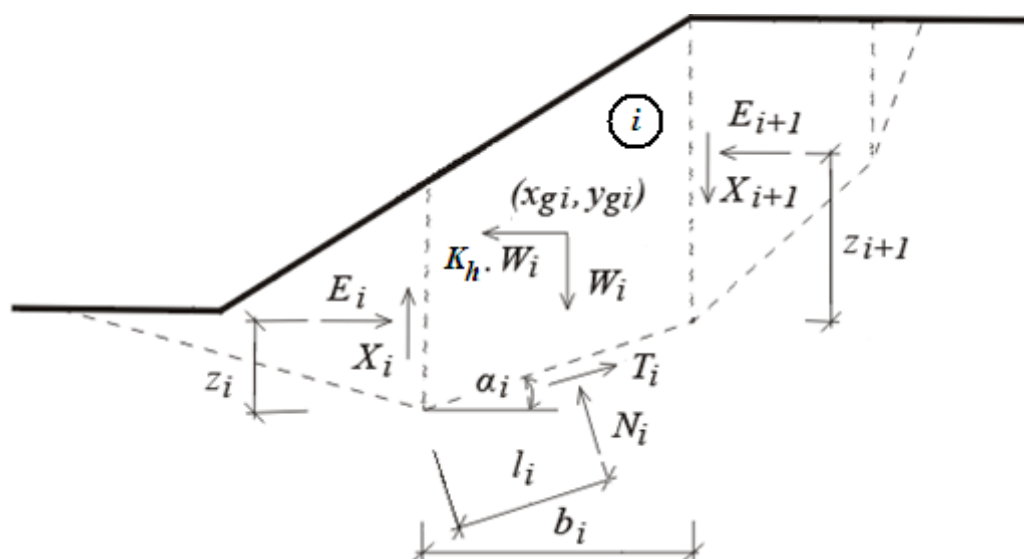
Verification according to the **factor of safety**:

$$\frac{M_p}{M_a} > SF_s$$

where:  $M_a$  - sliding moment  
 $M_p$  - resisting moment  
 $SF_s$  - factor of safety

## Polygonal Slip Surface

Solution of the slope stability problem adopting the polygonal slip surface is based on the determination of the limit state of forces acting on the soil body above the slip surface. To introduce these forces the slip surface above is subdivided into blocks by dividing planes. Typically, these planes are assumed vertical, but this is not a required condition, e.g. the Sarma method considers generally **inclined planes**.



Static scheme of block

The figure shows forces acting on individual blocks of soil. If the region above the slip surface is divided in blocks, then for the evaluation of unknowns we have:  $n$  normal forces  $N_i$  acting on individual segments and corresponding  $n$  shear forces  $T_i$ ;  $n-1$  normal forces between blocks  $E_i$  and corresponding  $n-1$  shear forces  $X_i$ ;  $n-1$  values of  $z_i$  representing the points of application of forces  $E_i$ ,  $n$  values of  $l_i$  representing the points of application of forces  $N_i$  and one value of the factor of safety  $SF$ . Forces  $X_i$  can be in some methods replaced by the values of inclination of forces  $E_i$ .

To following set of equations is available to solve the problem of equilibrium:  $n$  horizontal and  $n$  vertical equations of equilibrium written for individual blocks,  $n$  moment equations of equilibrium for individual blocks and  $n$  relations between  $N_i$  and  $T_i$  forces developed on blocks according to the Mohr-Coulomb theory. In total there are  $4n$  equations for  $6n-2$  unknowns. This suggests that  $2n-2$  unknowns must be chosen a priori. Individual methods differ from each other in the way these values are selected.

Most often points of application of individual forces acting between blocks or their inclinations are selected. Solving the problem of equilibrium it proceeds in an iterative manner, where the selected values must allow for satisfying both the equilibrium and kinematical admissibility of the obtained solution.

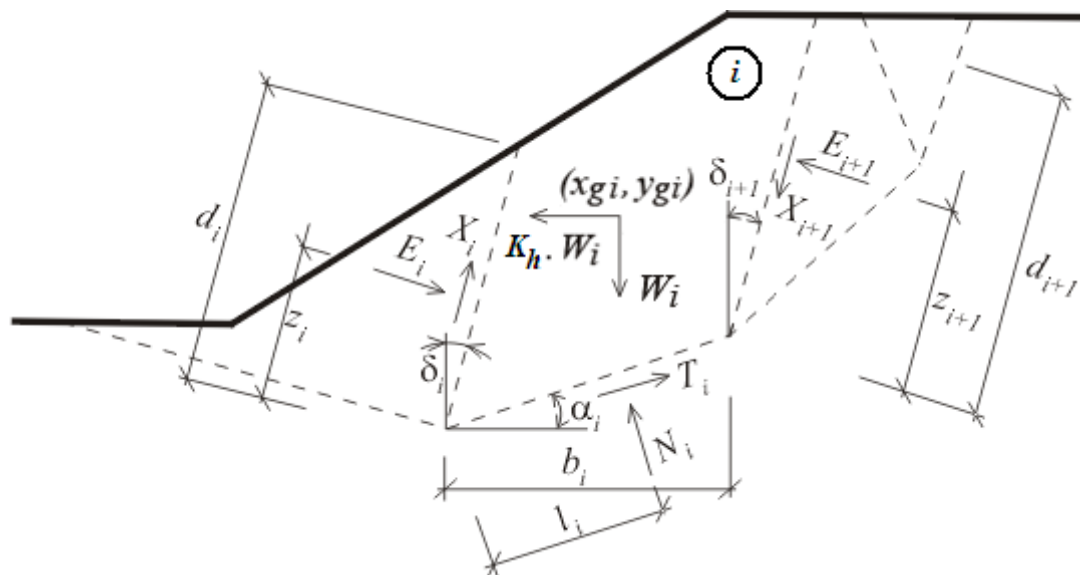
The program allows for adopting one of the following methods:

- Sarma
- Spencer
- Janbu
- Morgenstern-Price
- Shahunyants
- ITF Method

Optimization of polygonal slip surface searches the most critical surface (the lowest safety factor  $SF$ ).

## Sarma

The Sarma method falls within a category of general sliced methods of limit states. It is based on fulfilling the force and moment equilibrium conditions on individual blocks. The blocks are created by dividing the soil region above the potential slip surface by planes, which may have in general experience a **different inclination**. Forces acting on individual blocks are displayed in the following figure.



Static scheme - Sarma method

Here,  $E_i, X_i$  represent the normal and shear forces between blocks.  $N_i, T_i$  are normal and shear forces on segments of a slip surface.  $W_i$  is the block weight and  $K_h * W_i$  is the horizontal force that is used to achieve in the Sarma method the limit state. Generally inclined **surcharge** can be introduced in each block. This surcharge is included in the analysis together with the surcharge due to water having the free water table above the terrain, and with forces in **anchors**. All these forces are projected along the horizontal and vertical directions, which are then summed up into components  $F_{xi}$  and  $F_{yi}$ .

$K_h$  is a constant named the factor of **horizontal acceleration** and it is introduced into the analysis in order to satisfy the equilibrium on individual blocks. There is a relationship between  $K_h$  and the factor of slope stability  $SF$  allowing for the safety factor computation. In ordinary cases the analysis proceeds with the value of  $K_h$  equal to zero. A non-zero value of  $K_h$  is used to simulate the horizontal surcharge, e.g. due to earthquake (see below).

### Analysis process

#### Computation of limit equilibrium

The computation of limit equilibrium requires the solution of  $6n - 1$  unknowns, where  $n$  stands for the number of blocks dividing the soil region above the potential slip surface. These are:

- $E_i$  - forces developed between blocks
- $N_i$  - normal forces acting on slip surface
- $T_i$  - shear forces acting on a slip surface
- $X_i$  - shear forces developed between blocks



$z_i$  - locations of points of applications of forces

$l_i$  - locations of points of applications of forces

$K_h$  - factor of horizontal acceleration

$5n - 1$  equations are available for the required unknowns. In particular, we have:

**a) horizontal force equations of equilibrium on blocks:**

$$T_i \cdot \cos \alpha_i - N_i \cdot \sin \alpha_i = K_h \cdot W_i - Fx_i + X_{i+1} \cdot \sin \delta_i - X_i \cdot \sin \delta_i + E_{i+1} \cdot \cos \delta_i - E_i \cdot \cos \delta_i$$

**b) vertical force equations of equilibrium on blocks:**

$$N_i \cdot \cos \alpha_i - T_i \cdot \sin \alpha_i = W_i - Fy_i + X_{i+1} \cdot \cos \delta_{i+1} - X_i \cdot \cos \delta_i - E_{i+1} \cdot \sin \delta_{i+1} + E_i \cdot \cos \delta_i$$

**c) moment equations of equilibrium on blocks:**

$$N_i \cdot l_i - X_{i+1} \cdot b_i \sec \alpha_i \cdot \cos(\alpha_i + \delta_{i+1}) + E_{i+1} [z_{i+1} + b_i \sec \alpha_i \cdot \sin(\alpha_i + \delta_{i+1})] - E_i \cdot z_i - W_i \cdot (x_{gi} - x_i) + K_h \cdot W_i \cdot (y_{gi} - y_i) - Fx_i \cdot rx_i + Fy_i \cdot ry_i = 0$$

where  $rx_i$  and  $ry_i$  are arms of forces  $Fx_i$  and  $Fy_i$

**d) relationship between the normal and shear forces according to the Mohr-Coulomb theory:**

$$T_i = (N_i - U_i) \cdot \tan \varphi_i + c_i \cdot b_i \cdot \sec \alpha_i$$

$$X_i = (E_i - PW_i) \cdot \tan \bar{\varphi}_i + \bar{c}_i \cdot d_i$$

where:  $P \cdot W_i$  - resultant force of pore pressure on dividing planes

$i$

$\bar{\varphi}_i$  - average value of internal friction angle on dividing plane

$\bar{c}_i$  - average value of cohesion on dividing plane

It is evident that  $n - 1$  must be selected (estimated) a priori. Relatively small error is received when estimating the points of application of forces  $E_i$ . The problem then becomes statically determined. Solving the resulting system of equations finally provides the values of all remaining unknowns. The principal result of this analysis is the determination of the factor of horizontal acceleration  $K_h$ .

**Computation of factor of slope stability  $SF$**

The factor of slope stability  $SF$  is introduced in the analysis such as to reduce the soil strength parameters  $c$  and  $tg\varphi$ . Equilibrium analysis is then performed for the reduced parameters to arrive at the factor of horizontal acceleration  $K_h$  pertinent to a given factor of slope stability  $SF$ . This iteration is repeated until the factor  $K_h$  reaches either zero or a specified value.

**Influence of external load**

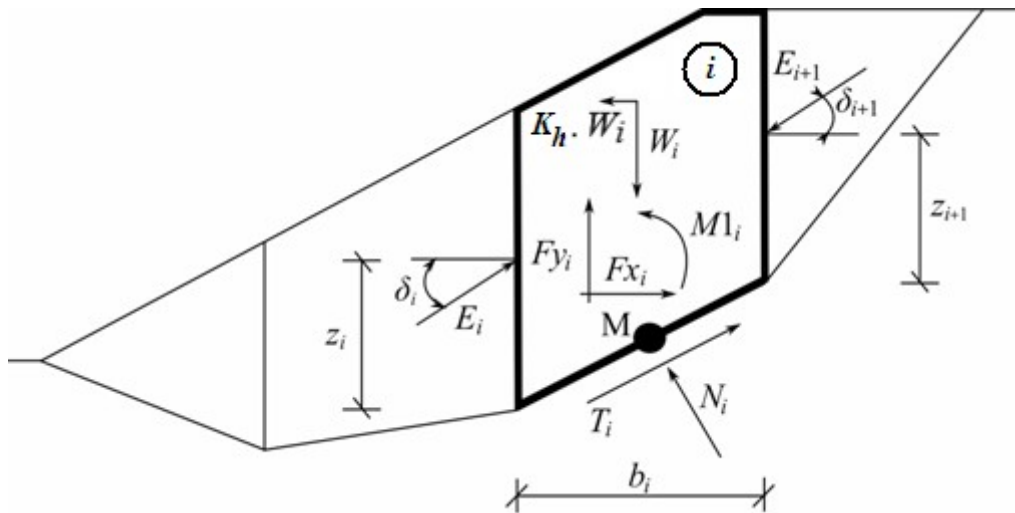
The analyzed slope can be loaded on its ground by inclined load having generally trapezoidal shape. This load enters the analysis such that its vertical material component (if having the direction of weight) is added to the weight of a corresponding block. This results in change of both the slice weight and its center of gravity. Providing the vertical component acts against the direction of gravity it is added to force  $Fy_i$ . The horizontal component is added to force  $Fx_i$ .

*Literature:*

Sarma, S. K.: *Stability analysis of embankments and slopes*, *Géotechnique* 23, 423-433, 1973.

## Spencer

The Spencer method is a general method of slices developed on the basis of limit equilibrium. It requires satisfying equilibrium of forces and moments acting on individual blocks. The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure.



Static scheme - Spencer method

Each block is assumed to contribute due to the following forces:

- $W_i$  - block weight, including material surcharge having the character of weight including the influence of the coefficient of vertical earthquake  $K_v$
- $K_h * W_i$  - horizontal inertia force representing the effect of earthquake,  $K_h$  is the factor of horizontal acceleration during earthquake
- $N_i$  - normal force on the slip surface
- $T_i$  - shear force on the slip surface
- $E_i, E_{i+1}$  - forces exerted by neighboring blocks, they are inclined from horizontal plane by angle  $\delta$
- $F_{x_i}, F_{y_i}$  - other horizontal and vertical forces acting on block
- $M_{l_i}$  - moment of forces  $F_{x_i}, F_{y_i}$  rotating about point  $M$ , which is the center of the  $i^{th}$  segment of slip surface
- $U_i$  - pore pressure resultant on the  $i^{th}$  segment of slip surface

The following assumptions are introduced in the Spencer method to calculate the limit equilibrium of forces and moment on individual blocks:

- dividing planes between blocks are always vertical
- the line of action of weight of block  $W_i$  passes through the center of the  $i^{th}$  segment of slip surface represented by point **M**

- the normal force  $N_i$  is acting in the center of the  $i^{th}$  segment of slip surface, at point **M**
- inclination of forces  $E_i$  acting between blocks is constant for all blocks and equals to  $\delta$ , only at slip surface end points is  $\delta = 0$

The solution adopts the following expressions:

$$N_i = N'_i + U_i \tag{1}$$

$$T_i = (N_i - U_i) \tan \varphi_i + c_i \frac{b_i}{\cos \alpha_i} = N'_i \tan \varphi_i + c_i \frac{b_i}{\cos \alpha_i} \tag{2}$$

$$N'_i + U_i - W_i \cos \alpha_i + k_h W_i \sin \alpha_i + Fy_i \cos \alpha_i - Fx_i \sin \alpha_i + E_{i+1} \sin(\alpha_i - \delta_{i+1}) - E_i \sin(\alpha_i - \delta_i) = 0 \tag{3}$$

$$N'_i \frac{\tan \varphi_i}{SF} + \frac{c_i}{SF \cos \alpha_i} \frac{b_i}{\cos \alpha_i} - W_i \sin \alpha_i - k_h W_i \cos \alpha_i + Fy_i \sin \alpha_i + Fx_i \cos \alpha_i - E_{i+1} \cos(\alpha_i - \delta_{i+1}) + E_i \cos(\alpha_i - \delta_i) = 0 \tag{4}$$

$$E_{i+1} \cos \delta_{i+1} \left( z_{i+1} - \frac{b_i}{2} \tan \alpha_i \right) - E_{i+1} \sin \delta_{i+1} \frac{b_i}{2} - E_i \cos \delta_i \left( z_i - \frac{b_i}{2} \tan \alpha_i \right) - E_i \sin \delta_i \frac{b_i}{2} + M1_i - k_h W_i (y_M - y_{g_i}) = 0 \tag{5}$$

Equation (1) represents the relationship between **effective and total** value of the normal force acting on the slip surface. Equation (2) corresponds to the Mohr-Coulomb condition representing the relation between the normal and shear forces on a given segment of the slip surface. Equation (3) represents the force equation of equilibrium in the direction normal to the  $i^{th}$  segment of the slip surface, whereas Equation (4) represents equilibrium along the  $i^{th}$  segment of the slip surface.  $SF$  is the factor of safety, which is used to reduce the soil parameters. Equation (5) corresponds to the moment equation of equilibrium about point **M**, where  $y_{g_i}$  is the vertical coordinate of the point of application of the weight of block and  $y_M$  is the vertical coordinate of point **M**. Modifying equations (3) and (4) provides the following recursive formula:

$$E_{i+1} = \frac{\left[ (W_i - Fy_i) \cos \alpha_i - (K_h W_i - Fx_i) \sin \alpha_i - U_i + E_i \sin(\alpha_i - \delta_i) \right] \frac{\tan \varphi_i}{SF} + \frac{c_i}{SF \cos \alpha_i} \frac{b_i}{\cos \alpha_i} - (W_i - Fy_i) \sin \alpha_i - (K_h W_i - Fx_i) \cos \alpha_i + E_i \cos(\alpha_i - \delta_i)}{\sin(\alpha_i - \delta_{i+1}) \frac{\tan \varphi_i}{SF} + \cos(\alpha_i - \delta_{i+1})}$$

This formula allows to calculate all forces  $E_i$  acting between blocks for given values of  $\delta_i$  and  $SF$ . This solution assumes that at the slip surface origin the value of  $E$  is known and equal to  $E_I = 0$ .

Additional recursive formula follows from the moment equation of equilibrium (5) as:

$$z_{i+1} = \frac{\frac{b_i}{2} \left[ E_{i+1} (\sin \delta_{i+1} - \cos \delta_{i+1} \tan \alpha_i) + E_i (\sin \delta_i - \cos \delta_i \tan \alpha_i) \right] + E_i z_i \cos \delta_i - M1_i + K_h W_i (y_M - y_{g_i})}{E_{i+1} \cos \delta_{i+1}}$$

This formula allows us calculating for a given value of  $\delta$  all arms  $z$  of forces acting between blocks, knowing the value on the left at the slip surface origin, where  $z_I = 0$ .

The factor of safety  $SF$  is determined by employing the following iteration process:

1. The initial value of  $\delta$  is set to zero  $\delta = 0$ .
2. The factor of safety  $SF$  for a given value of  $\delta$  follows from equation (6), while assuming

the value of  $E_{n+1} = 0$  at the end of the slip surface.

3. The value of  $\delta$  is provided by equation (7) using the values of  $E$  determined in the previous step with the requirement of having the moment on the last block equal to zero. Equation (7) does not provide the value of  $z_{n+1}$  as it is equal to zero. For this value the moment equation of equilibrium (5) must be satisfied.
4. Steps 2 and 3 are then repeated until the value of  $\delta$  does not change.

For the process of iteration to be stable it is necessary to avoid unstable solutions. Such instabilities occur at points where division by zero in expressions (6) and (7) takes place. In equation (7), division by zero is encountered for  $\delta = \pi/2$  or  $\delta = -\pi/2$ . Therefore, the value of angle  $\delta$  must be found in the interval  $(-\pi/2 ; \pi/2)$ .

Division by zero in expression (6) appears when:

$$SF = \tan \varphi_i \tan(\delta_{i+1} - \alpha_i)$$

Another check preventing numerical instability is verification of parameter  $m_\alpha$  - following condition must be satisfied:

$$m_\alpha = \cos \alpha_i + \frac{\sin \alpha_i \tan \varphi_i}{SF} > 0,2$$

Therefore before iteration run it is required to find the highest of critical values  $SF_{min}$  satisfying above mentioned conditions. Values below this critical value  $SF_{min}$  are in area of unstable solution, therefore iteration begins by setting  $SF$  to a value "just" above  $SF_{min}$  and all result values of  $SF$  from iteration runs are higher than  $SF_{min}$ .

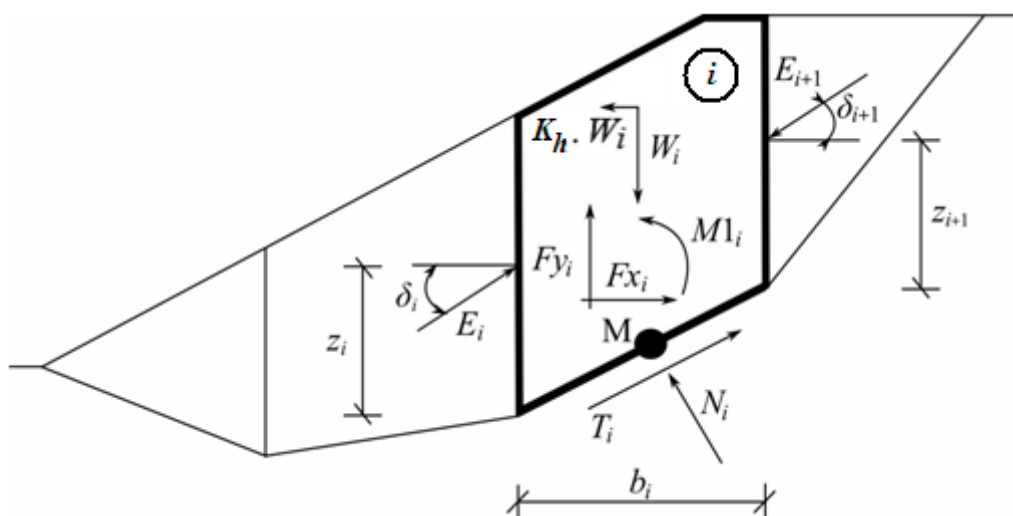
Generally rigorous methods converge worse than the simpler methods (Bishop, Fellenius). Examples with convergence problems include too steep sections of slip surface, complex geometry, a significant jump in surcharge etc. If no result is obtained, we recommend slight change of input data, e.g. less steep slip surface, input more points into the slip surface etc. or using of some of the simpler methods.

*Literature:*

*Spencer, E. 1967. A method of analysis of the stability of embankments assuming parallel interslice forces. Géotechnique, 17(1): 11-26.*

## Janbu

Janbu is a general method of slices developed on the basis of limit equilibrium. It requires satisfying equilibrium of forces and moments acting on individual blocks (only moment equilibrium at last uppermost block is not satisfied). The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure:



Static scheme - Janbu method

Each block is assumed to contribute due to the following forces:

- $W_i$  - block weight, including material surcharge having the character of weight including the influence of the coefficient of vertical earthquake  $K_v$
- $K_h * W_i$  - horizontal inertia force representing the effect of earthquake,  $K_h$  is the factor of horizontal acceleration during earthquake
- $N_i$  - normal force on the slip surface
- $T_i$  - shear force on the slip surface
- $E_i, E_{i+1}$  - forces exerted by neighboring blocks, they are inclined from horizontal plane by angle  $\delta_i$  resp.  $\delta_{i+1}$  and lie at the height  $z_i$  resp.  $z_{i+1}$  above slip surface
- $F_{x_i}, F_{y_i}$  - other horizontal and vertical forces acting on block
- $M I_i$  - moment from forces  $F_{x_i}, F_{y_i}$  rotating about point **M**, which is the center of the  $i^{th}$  segment of slip surface
- $U_i$  - pore pressure resultant on the  $i^{th}$  segment of slip surface

The following assumptions are introduced in the Janbu method to calculate the limit equilibrium of forces and moment on individual blocks:

- dividing planes between blocks are always vertical
- the line of action of weight of block  $W_i$  passes through the center of the  $i^{th}$  segment of slip surface represented by point **M**
- the normal force  $N_i$  is acting in the center of the  $i^{th}$  segment of slip surface, at point **M**
- position  $z_i$  of forces  $E_i$  acting between blocks is assumed, at slip surface end points is  $z = 0$

Choice of position  $z_i$  can have significant influence on convergency of method. If we make a bad assumption of position  $z_i$  for a given slope, it can become impossible to satisfy equilibrium conditions (algorithm does not converge). Heights  $z_i$  above slip surface are set approximately to one third of height of interface between the blocks. In case of unsatisfying equilibrium conditions algorithm changes heights to a different position, e.g. slightly higher within passive zone, near the toe, and lower within active zone, near the crest of slope.

The solution adopts the following expressions:

$$N_i = N'_i + U_i \quad (1)$$

$$T_i = (N_i - U_i) \tan \varphi_i + c_i \frac{b_i}{\cos \alpha_i} = N'_i \tan \varphi_i + c_i \frac{b_i}{\cos \alpha_i} \quad (2)$$

$$N'_i + U_i - W_i \cdot \cos \alpha_i + K_h \cdot W_i \cdot \sin \alpha_i + Fy_i \cdot \cos \alpha_i - Fx_i \cdot \sin \alpha_i + \quad (3)$$

$$E_{i+1} \cdot \sin(\alpha_i - \delta_{i+1}) - E_i \cdot \sin(\alpha_i - \delta_i) = 0$$

$$N'_i \cdot \frac{\tan \varphi_i}{FS} + \frac{c_i}{FS} \cdot \frac{b_i}{\cos \alpha_i} - W_i \cdot \sin \alpha_i - K_h \cdot W_i \cdot \cos \alpha_i + Fy_i \cdot \sin \alpha_i + \quad (4)$$

$$Fx_i \cos \alpha_i - E_{i+1} \cdot \cos(\alpha_i - \delta_{i+1}) + E_i \cdot \cos(\alpha_i - \delta_i) = 0$$

$$E_{i+1} \cdot \cos \delta_{i+1} \left( z_{i+1} - \frac{b_i}{2} \tan \alpha_i \right) - E_{i+1} \cdot \sin \delta_{i+1} \cdot \frac{b_i}{2} - \quad (5)$$

$$E_i \cdot \cos \delta_i \left( z_i - \frac{b_i}{2} \tan \alpha_i \right) - E_i \cdot \sin \delta_i \cdot \frac{b_i}{2} +$$

$$M1_i - K_h \cdot W_i (y_M - y_{gi}) = 0$$

Equation (1) represents the relationship between **effective and total** value of the normal force acting on the slip surface. Equation (2) corresponds to the Mohr-Coulomb condition representing the relation between the normal and shear forces on a given segment of the slip surface. Equation (3) represents the force equation of equilibrium in the direction normal to the  $i^{th}$  segment of the slip surface, whereas Equation (4) represents equilibrium along the  $i^{th}$  segment of the slip surface.  $FS$  is the factor of safety, which is used to reduce the soil parameters. Equation (5) corresponds to the moment equation of equilibrium about point **M**, where  $y_{gi}$  is the vertical coordinate of the point of application of the weight of block and  $y_M$  is the vertical coordinate of point **M**.

Modifying equations (3) and (4) provides the following recursive formula (6):

$$E_{i+1} = \frac{\left[ (W_i - Fy_i) \cdot \cos \alpha_i - (K_h \cdot W_i - Fx_i) \cdot \sin \alpha_i - U_i + E_i \cdot \sin(\alpha_i - \delta_i) \right] \cdot \frac{\tan \varphi_i}{FS} + \quad (6)$$

$$\frac{c_i}{FS} \cdot \frac{b_i}{\cos \alpha_i} - (W_i - Fy_i) \cdot \sin \alpha_i - (K_h \cdot W_i - Fx_i) \cdot \cos \alpha_i + E_i \cdot \cos(\alpha_i - \delta_i)}{\sin(\alpha_i - \delta_{i+1}) \cdot \frac{\tan \varphi_i}{FS} + \cos(\alpha_i - \delta_{i+1})}$$



This formula allows calculating all forces  $E_i$  acting between blocks for given values of  $\delta_i$  and  $SF$ . This solution assumes that at the slip surface origin the value of  $E$  is known and equal to  $E_1=0$ .

Formula for calculating angles  $\delta_i$  (7) follows from the moment equation of equilibrium (5) as:

$$\delta_{i+1} = \arctan\left(\frac{2 \cdot z_{i+1}}{b_i} + \tan \alpha_i\right) - \arcsin \frac{E_i \left( \cos \delta_i \left( z_i - \frac{b_i \cdot \tan \alpha_i}{2} \right) + \sin \delta_i \cdot \frac{b_i}{2} \right) - M1_i}{E_{i+1} \sqrt{\left( z_{i+1} + \frac{b_i \cdot \tan \alpha_i}{2} \right)^2 + \left( \frac{b_i}{2} \right)^2}} \quad (7)$$

This formula allows us calculating for a given value of  $\delta$  all arms  $z_i$  of forces acting between blocks, knowing the value on the left at the slip surface origin, where  $z_1 = 0$ .

The factor of safety  $FS$  is determined by employing the following iteration process:

1. The initial value of angles are set to zero  $\delta_i = 0$  and positions  $z_i$  to approximately one third of interface height.
2. The factor of safety  $FS$  for a given value of  $\delta_i$  follows from equation (6), while assuming the value of  $E_{n+1} = 0$  at the end of the slip surface.
3. The value of  $\delta_i$  is provided by equation (7) using the values of  $E_i$  determined in the previous step.
4. Steps 2 and 3 are then repeated until the value of  $FS$  does not change.

It is necessary to avoid unstable solutions for successful iteration process. Such instabilities occur at points where division by zero in expression (6) takes place, i.e.:

$$FS = \tan \varphi_i \cdot \tan (\delta_{i+1} - \alpha_i)$$

Another check preventing numerical instability is verification of parameter  $m_\alpha$  - following condition must be satisfied:

$$m_\alpha = \cos \alpha_i + \frac{\sin \alpha_i \cdot \tan \varphi_i}{FS} > 0,2$$

Therefore before iteration run it is required to find the highest of critical values  $SF_{min}$  satisfying above mentioned conditions. Values below this critical value  $SF_{min}$  are in area of unstable solution, therefore iteration begins by setting  $SF$  to a value "just" above  $SF_{min}$  and all result values of  $SF$  from iteration runs are higher than  $SF_{min}$ .

Generally rigorous methods converge worse than the simpler methods (Bishop, Fellenius). Examples with convergence problems include too steep sections of slip surface, complex geometry, a significant jump in surcharge etc. If no result is obtained, we recommend slight change of input data, e.g. less steep slip surface, input more points into the slip surface etc. or using of some of the simpler methods.

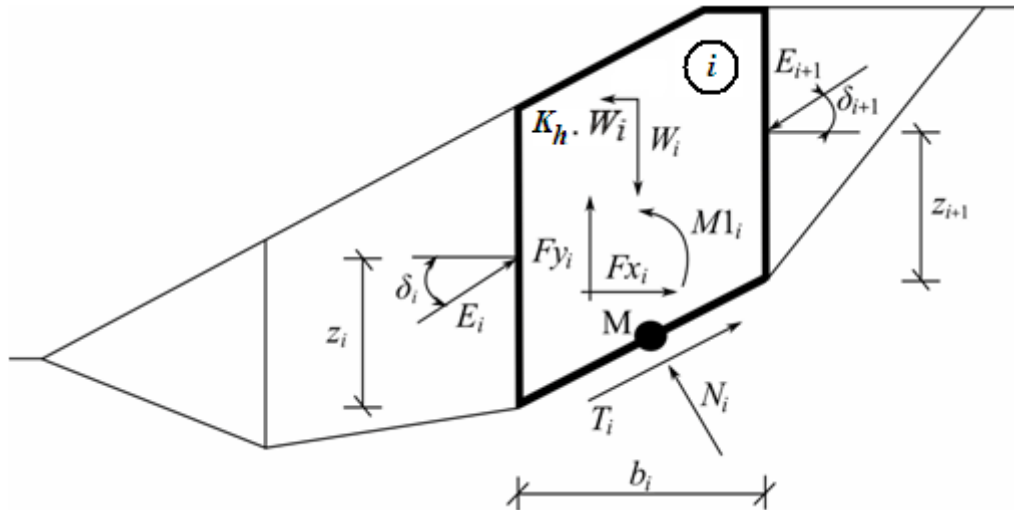
#### Literature:

Janbu, N. 1954. *Application of Composite Slip Surface for Stability Analysis*. European Conference on Stability Analysis, Stockholm, Sweden.

Janbu, N. 1973. *Slope Stability Computations*. Embankment Dam Engineering - Casagrande Volume, R.C. Hirschfeld and S.J. Poulos, eds., John Wiley and Sons, New York, pp 47-86.

## Morgenstern-Price

Morgenstern-Price is a general method of slices developed on the basis of limit equilibrium. It requires satisfying equilibrium of forces and moments acting on individual blocks. The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure:



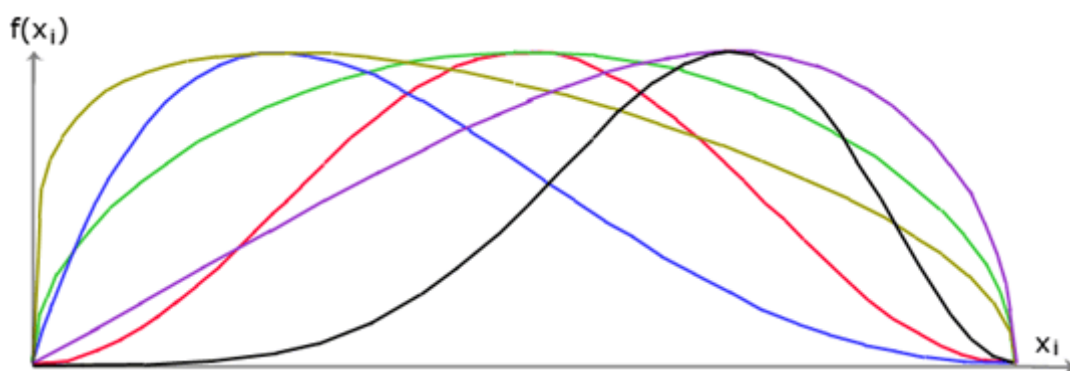
*Static scheme - Morgenstern-Price method*

Each block is assumed to contribute due to the same forces as in [Spencer](#) method. The following assumptions are introduced in the Morgenstern-Price method to calculate the limit equilibrium of forces and moment on individual blocks:

- dividing planes between blocks are always vertical
- the line of action of weight of block  $W_i$  passes through the center of the  $i^{th}$  segment of slip surface represented by point **M**
- the normal force  $N_i$  is acting in the center of the  $i^{th}$  segment of slip surface, at point **M**
- inclination of forces  $E_i$  acting between blocks is different on each block ( $\delta_i$ ) at slip surface end points is  $\delta = 0$

The only difference between [Spencer](#) and Morgenstern-Price method is shown in the above list of assumptions. Choice of inclination angles  $\delta_i$  of forces  $E_i$  acting between the blocks is realized with the help of Half-sine function - one of the functions in the following figure is automatically chosen. This choice of the shape of function has a minor influence on final results, but suitable choice can improve the convergency of method. Functional value of Half-sine function  $f(x_i)$  at boundary point  $x_i$  multiplied by parameter  $\lambda$  results the value of inclination angle  $\delta_i$ .





Half-sine function

The solution adopts the expressions (1) - (5), shown in **Spencer** method, i.e.:

$$N_i = N'_i + U_i \quad (1)$$

$$T_i = (N_i - U_i) \tan \varphi_i + c_i \frac{b_i}{\cos \alpha_i} = N'_i \tan \varphi_i + c_i \frac{b_i}{\cos \alpha_i} \quad (2)$$

$$N'_i + U_i - W_i \cdot \cos \alpha_i + K_h \cdot W_i \cdot \sin \alpha_i + Fy_i \cdot \cos \alpha_i - Fx_i \cdot \sin \alpha_i + \quad (3)$$

$$E_{i+1} \cdot \sin(\alpha_i - \delta_{i+1}) - E_i \cdot \sin(\alpha_i - \delta_i) = 0$$

$$N'_i \cdot \frac{\tan \varphi_i}{FS} + \frac{c_i}{FS} \cdot \frac{b_i}{\cos \alpha_i} - W_i \cdot \sin \alpha_i - K_h \cdot W_i \cdot \cos \alpha_i + Fy_i \cdot \sin \alpha_i + \quad (4)$$

$$Fx_i \cos \alpha_i - E_{i+1} \cdot \cos(\alpha_i - \delta_{i+1}) + E_i \cdot \cos(\alpha_i - \delta_i) = 0$$

$$E_{i+1} \cdot \cos \delta_{i+1} \left( z_{i+1} - \frac{b_i}{2} \tan \alpha_i \right) - E_{i+1} \cdot \sin \delta_{i+1} \cdot \frac{b_i}{2} - \quad (5)$$

$$E_i \cdot \cos \delta_i \left( z_i - \frac{b_i}{2} \tan \alpha_i \right) - E_i \cdot \sin \delta_i \cdot \frac{b_i}{2} +$$

$$M1_i - K_h \cdot W_i (y_M - y_{gi}) = 0$$

- (1) relationship between **effective and total** value of the normal force acting on the slip surface
- (2) Mohr-Coulomb condition representing the relation between the normal and shear forces on a given segment of the slip surface ( $N_i$  a  $T_i$ )
- (3) force equation of equilibrium in the direction normal to the  $i^{th}$  segment of the slip surface
- (4) force equation of equilibrium along the  $i^{th}$  segment of the slip surface

- (5) moment equation of equilibrium about point **M**

Modifying force equations (3) and (4) provides the following recursive formula (6):

$$E_{i+1} = \frac{\left[ (W_i - Fy_i) \cdot \cos \alpha_i - (K_h W_i - Fx_i) \cdot \sin \alpha_i - U_i + E_i \cdot \sin(\alpha_i - \delta_i) \right] \cdot \frac{\tan \varphi_i}{FS} + \frac{c_i}{FS} \cdot \frac{b_i}{\cos \alpha_i} - (W_i - Fy_i) \cdot \sin \alpha_i - (K_h W_i - Fx_i) \cdot \cos \alpha_i + E_i \cdot \cos(\alpha_i - \delta_i)}{\sin(\alpha_i - \delta_{i+1}) \cdot \frac{\tan \varphi_i}{FS} + \cos(\alpha_i - \delta_{i+1})} \quad (6)$$

This formula allows calculating all forces  $E_i$  acting between blocks for given values of  $\delta_i$  and  $SF$ . This solution assumes that at the slip surface origin the value of  $E$  is known and equal to  $E_I = 0$ .

Additional recursive formula (7) follows from the moment equation of equilibrium (5) as:

$$z_{i+1} = \frac{\frac{b_i}{2} \cdot [E_{i+1}(\sin \delta_{i+1} - \cos \delta_{i+1} \cdot \tan \alpha_i) + E_i(\sin \delta_i - \cos \delta_i \cdot \tan \alpha_i)] + E_i \cdot z_i \cdot \cos \delta_i - M_i + K_h W_i \cdot (y_M - y_{\bar{g}i})}{E_{i+1} \cdot \cos \delta_{i+1}} \quad (7)$$

This formula allows to calculate all arms  $z_i$  of forces acting between blocks for a given values of  $\delta_i$ , knowing the value on the left at the slip surface origin, where  $z_I = 0$ .

The factor of safety  $SF$  is determined by employing the following iteration process:

1. The initial value of angles  $\delta_i$  is set according to Half-sine function ( $\delta_i = \lambda * f(x_i)$ ).
2. The factor of safety  $SF$  for a given value of  $\delta_i$  follows from equation (6), while assuming the value of  $E_{n+1} = 0$  at the end of the slip surface.
3. The value of  $\delta_i$  is provided by equation (7) using the values of  $E_i$  determined in the previous step with the requirement of having the moment on the last block equal to zero. Functional values  $f(x_i)$  are same all the time during the iteration, only parameter  $\lambda$  is iterated. Equation (7) does not provide the value of  $z_{n+1}$  as it is equal to zero. For this value the moment equation of equilibrium (5) must be satisfied.
4. Steps 2 and 3 are then repeated until the value of  $\delta_i$  (resp. parameter  $\lambda$ ) does not change.

It is necessary to avoid unstable solutions for successful iteration process. Such instabilities occur at points where division by zero in expressions (6) and (7) takes place. In equation (7), division by zero is encountered for  $\delta_i = \pi/2$  or  $\delta_i = -\pi/2$ . Therefore, the value of angle  $\delta_i$  must be found in the interval  $(-\pi/2 ; \pi/2)$ .

Division by zero in expression (6) appears when:

$$FS = \tan \varphi_i \cdot \tan(\delta_{i+1} - \alpha_i)$$

Another check preventing numerical instability is verification of parameter  $m_\alpha$  - following condition must be satisfied:

$$m_{\alpha} = \cos \alpha_i + \frac{\sin \alpha_i \cdot \tan \varphi_i}{FS} > 0,2$$

Therefore before iteration run it is required to find the highest of critical values  $SF_{min}$  satisfying above mentioned conditions. Values below this critical value  $SF_{min}$  are in area of unstable solution, therefore iteration begins by setting  $SF$  to a value "just" above  $SF_{min}$  and all result values of  $SF$  from iteration runs are higher than  $SF_{min}$ .

Generally rigorous methods converge worse than the simpler methods (Bishop, Fellenius). Examples with convergence problems include too steep sections of slip surface, complex geometry, a significant jump in surcharge etc. If no result is obtained, we recommend slight change of input data, e.g. less steep slip surface, input more points into the slip surface etc. or using of some of the simpler methods.

*Literature:*

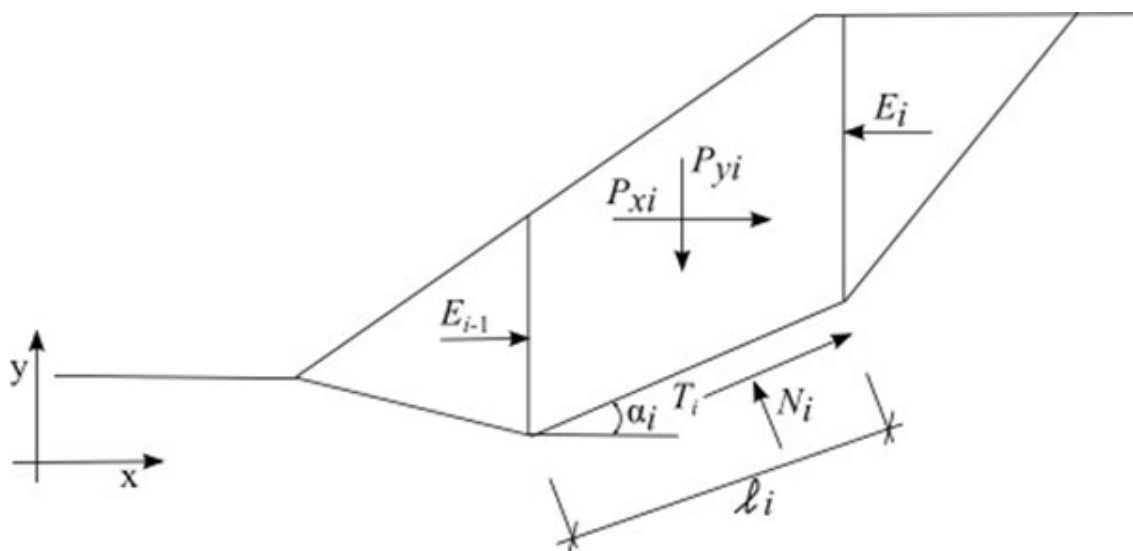
*Morgenstern, N.R., and Price, V.E. 1965. The analysis of the stability of general slip surfaces. Géotechnique, 15(1): 79-93.*

*Morgenstern, N.R., and Price, V.E. 1967. A numerical method for solving the equations of stability of general slip surfaces. Computer Journal, 9: 388-393.*

*Zhu, D.Y., Lee, C.F., Qian, Q.H., and Chen, G.R. 2005. A concise algorithm for computing the factor of safety using the Morgenstern-Price method. Canadian Geotechnical Journal, 42(1): 272-278.*

## Shahunyants

The Shahunyants method is a general method of slices developed on the basis of limit equilibrium. It requires satisfying equilibrium of forces and moments acting on individual blocks. The blocks are created by dividing the soil above the slip surface by dividing planes. Forces acting on individual blocks are displayed in the following figure:



*Static scheme - Shahunyants method*

Each block assumes action of the following forces:

where:  $P_{yi}$  - resultant of vertical forces acting on a given block (block weight, block surcharge, earthquake, anchor forces, ...)

- $P_{xi}$  - resultant of horizontal forces on a given block (block surcharge, earthquake, anchor forces, geo-reinforcements, ...)  
 $E_{i+1}, E_i$  - forces developed between blocks  
 $N_i$  - reaction below the block normal to the slip surface segment  
 $T_i$  - friction force on the slip surface segment  
 $\alpha_i$  - inclination of the slip surface segment  
 $l_i$  - length of the slip surface segment

The following assumptions are adopted in the Shahunyants method to calculate the limit state on a given block:

- dividing planes between blocks are always vertical
- slope of forces  $E_i$  acting between blocks is zero, forces act horizontally

#### **Solution procedure:**

Forces  $P_{yi}$  and  $P_{xi}$  are first transformed with the help of expressions (1) and (2) into directions of forces  $T_i$  and  $N_i$ . For a positive angle  $\alpha_i$  (the same way as in the schema) the force  $P_{Ni}$  acts in the direction opposite to  $N_i$ , the force  $P_{Qi}$  acts in the direction opposite to  $T_i$ .

$$P_{Ni} = P_{xi} \sin \alpha_i + P_{yi} \cos \alpha_i$$

(1)

$$P_{Qi} = P_{yi} \sin \alpha_i - P_{xi} \cos \alpha_i$$

(2)

The forces acting along the slip surface segment are related by:

$$T_i = (N_i - U_i) \tan \varphi_i + c_i l_i$$

(3)

where:  $U_i$  - pore pressure on the slip surface segment

#### **The force equations of equilibrium are fulfilled on the block:**

The equilibrium condition in the direction normal to the slip surface segment:

$$N_i = P_{Ni} + E_{i-1} \sin \alpha_i - E_i \sin \alpha_i$$

(4)

The equilibrium condition in the direction parallel to the slip surface segment:

$$T_i = P_{Qi} + E_i \cos \alpha_i - E_{i-1} \cos \alpha_i$$

(5)

Introducing Eq. (3) into Eq. (5) gives:

$$(N_i - U_i) \tan \varphi_i + c_i l_i = P_{Qi} + E_i \cos \alpha_i - E_{i-1} \cos \alpha_i$$

(6)

Next, substituting Eq. (4) into Eq. (6) gives:

$$(P_{Ni} + E_{i-1} \sin \alpha_i - E_i \sin \alpha_i - U_i) \tan \varphi_i + c_i l_i = P_{Qi} + E_i \cos \alpha_i - E_{i-1} \cos \alpha_i$$

(7)

After some algebra:

$$\begin{aligned} (P_{Ni} - U_i) \tan \varphi_i + (E_{i-1} - E_i) \sin \alpha_i \tan \varphi_i + c_i l_i &= P_{Qi} + (E_i - E_{i-1}) \cos \alpha_i \\ (P_{Ni} - U_i) \tan \varphi_i + c_i l_i - P_{Qi} &= (E_i - E_{i-1})(\cos \alpha_i + \sin \alpha_i \tan \varphi_i) \end{aligned}$$

(8)

Exploiting the following mathematical expression:

$$\cos \alpha + \sin \alpha \tan \beta = \frac{\cos \alpha \cos \beta + \sin \alpha \sin \beta}{\cos \beta} = \frac{\cos(\alpha - \beta)}{\cos \beta}$$

(9)

yields Eq. (8) in the form:

$$(P_{Ni} - U_i) \tan \varphi_i + c_i l_i - P_{Qi} = (E_i - E_{i-1}) \frac{\cos(\alpha_i - \varphi_i)}{\cos \varphi_i}$$

(10)

This can be modified as:

$$(P_{Ni} - U_i) \tan \varphi_i + c_i l_i - P_{Qi} + E_{i-1} \frac{\cos(\alpha_i - \varphi_i)}{\cos \varphi_i} = E_i \frac{\cos(\alpha_i - \varphi_i)}{\cos \varphi_i}$$

(11)

to provide the recurrent expression for  $E_i$  forces acting between blocks as:

$$E_i = \frac{[(P_{Ni} - U_i) \tan \varphi_i + c_i l_i - P_{Qi}] \cos \varphi_i}{\cos(\alpha_i - \varphi_i)} + E_{i-1}$$

(12)

At this stage the analysis enters the factor of safety  $K_u$ . The factor of safety is the value which bring the forces acting on individual blocks of soil into the state of limit states. This is achieved by multiplying active forces, i.e. forces contributing to sliding of the soil mass above the slip surface, by the factor of safety. Active forces are in Eq. (12) contained within the term  $P_{Qi}$ . This term contains on the one hand active forces that contribute to sliding and on the other hand forces that resist to sliding. The contributing forces will be denoted as  $P_{Qi,sd}$  whereas the resisting forces as  $P_{Qi,ud}$ . Eq. (12) then becomes:

$$E_i = \frac{[(P_{Ni} - U_i) \tan \varphi_i + c_i l_i - K_u P_{Qi,sd} - P_{Qi,ud}] \cos \varphi_i}{\cos(\alpha_i - \varphi_i)} + E_{i-1}$$

(13)

Providing the value of  $P_{Qi}$  is positive then it contributes to sliding and will be assumed as active force  $P_{Qi,sd}$ . Providing the value of  $P_{Qi}$  is negative then it resists to sliding and will be assumed as force  $P_{Qi,ud}$ . Therefore subtracting the value  $P_{Qi,ud}$ , which is negative, in Eq. (13) means essentially adding the positive value, so we can formally write:

$$E_i = \frac{[(P_{Ni} - U_i) \tan \varphi_i + c_i l_i - K_u P_{Qi,sd} + |P_{Qi,ud}|] \cos \varphi_i}{\cos(\alpha_i - \varphi_i)} + E_{i-1} \quad (14)$$

At the slip surface origin the value of  $E_0 = 0$ . The value of  $E_1$  is then given by:

$$E_1 = \frac{[(P_{N1} - U_1) \tan \varphi_1 + c_1 l_1 - K_u P_{Q1,sd} + |P_{Q1,ud}|] \cos \varphi_1}{\cos(\alpha_1 - \varphi_1)} \quad (15)$$

The value of  $E_2$  is then given by:

$$E_2 = \frac{[(P_{N2} - U_2) \tan \varphi_2 + c_2 l_2 - K_u P_{Q2,sd} + |P_{Q2,ud}|] \cos \varphi_2}{\cos(\alpha_2 - \varphi_2)} + \frac{[(P_{N1} - U_1) \tan \varphi_1 + c_1 l_1 - K_u P_{Q1,sd} + |P_{Q1,ud}|] \cos \varphi_1}{\cos(\alpha_1 - \varphi_1)} \quad (16)$$

Similarly we may determine the values of all forces acting between blocks. It further holds at the end point of the slip surface we have  $E_n = 0$ . Exploiting the previous expressions this can be written as:

$$E_n = \sum_{i=1}^n [(P_{Ni} - U_i) \tan \varphi_i + c_i l_i + |P_{Qi,ud}|] \frac{\cos \varphi_i}{\cos(\alpha_i - \varphi_i)} - K_u \sum_{i=1}^n P_{Qi,sd} \frac{\cos \varphi_i}{\cos(\alpha_i - \varphi_i)} = 0 \quad (17)$$

This equation directly provides the factor of safety  $K_u$  in the form:

$$K_u = \frac{\sum_{i=1}^n [(P_{Ni} - U_i) \tan \varphi_i + c_i l_i + |P_{Qi,ud}|] \frac{\cos \varphi_i}{\cos(\alpha_i - \varphi_i)}}{\sum_{i=1}^n P_{Qi,sd} \frac{\cos \varphi_i}{\cos(\alpha_i - \varphi_i)}} \quad (18)$$

## ITF Method (Imbalance Thrust Force Method)

The ITF method is a limit state method. It builds up on the equation of equilibrium of forces acting on individual blocks and does not consider the moment equation of equilibrium. The bases of the method and adopted assumptions are evident from following figure.

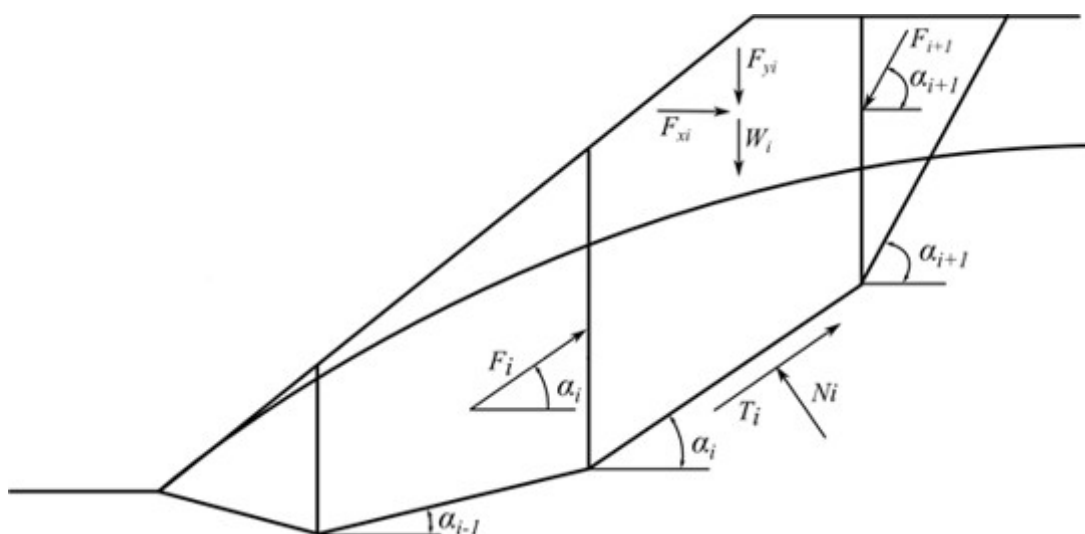


Fig. 1 Forces acting on a block - ITF Method

Consider the following assumptions to concerning the forces acting on the block:

- where:
- $W_i$  - weight of the  $i^{th}$  block, the weight of a part of the block below the ground water is determined from the saturated unit weight of soil  $\gamma_{sat}$
  - $F_{yi}$  - represents the remaining vertical load acting on the block
  - $F_{xi}$  - represents the remaining horizontal load acting on the block
  - $F_i, F_{i+1}$  - forces acting between blocks along directions given by angles  $\alpha_i$  and  $\alpha_{i+1}$

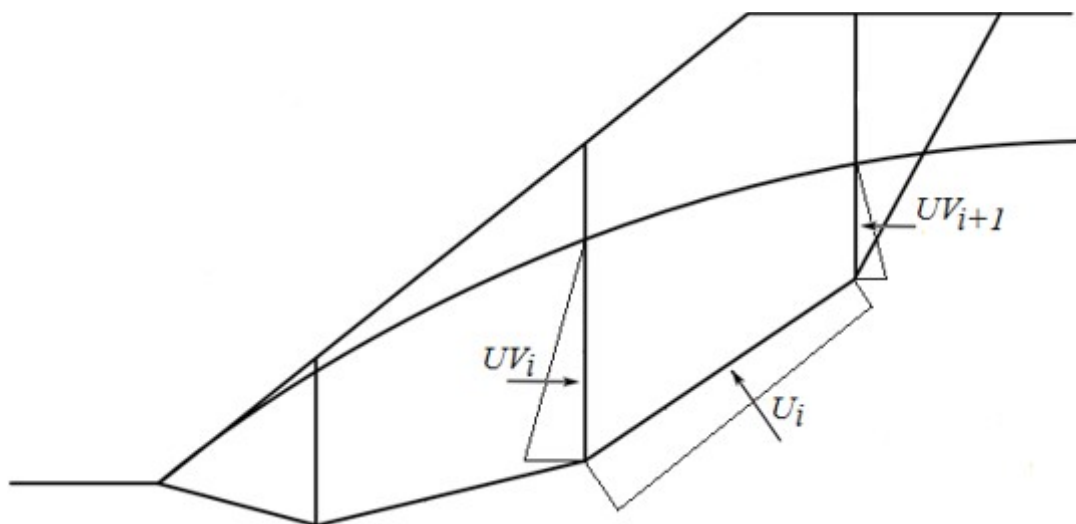


Fig. 2 Scheme of pore pressure action on block

- $U_i$  - pore pressure resultant on slip surface segment
- $UV_i, UV_{i+1}$  - pore presure resultants on dividing planes between blocks

The forces  $UV_i$  and  $UV_{i+1}$  are included in horizontal forces  $F_{xi}$ .



The force equation of equilibrium in the direction normal to the segment of the slip surface provides:

$$N_i = (W_i + Fy_i) \cos \alpha_i + Fx_i \sin \alpha + F_{i+1} \sin(\alpha_{i+1} - \alpha_i) - U_i \quad (1)$$

The forces on the segment of a slip surface are related by

$$T_i = N_i \tan \varphi_i + c_i l_i \quad (2)$$

where:  $\varphi_i$  - angle of internal friction of soil  
 $c_i$  - soil cohesion  
 $l_i$  - length of the slip surface segment associated with the  $i^{th}$  block

The force equation of equilibrium in the direction of the  $i^{th}$  segment of the slip surface (given by angle  $\alpha_i$ ) yields the force  $F_i$  acting between blocks in the form:

$$F_i = (W_i + Fy_i) \sin \alpha_i - Fx_i \cos \alpha_i - T_i + F_{i+1} \cos(\alpha_{i+1} - \alpha_i) \quad (3)$$

Introducing Eqs.(1) and (2) into Eq. (3) provides:

$$F_i = (W_i + Fy_i) \sin \alpha_i - Fx_i \cos \alpha_i - \\ - \{[(W_i + Fy_i) \cos \alpha_i + Fx_i \sin \alpha_i + F_{i+1} \sin(\alpha_{i+1} - \alpha_i) - U_i] \tan \varphi_i + c_i l_i\} + \\ + F_{i+1} \cos(\alpha_{i+1} - \alpha_i)$$

and after some formal algebra we arrive at the resulting form of the equation of equilibrium as:

$$F_i = (W_i + Fy_i) \sin \alpha_i - Fx_i \cos \alpha_i - \\ - \{[(W_i + Fy_i) \cos \alpha_i + Fx_i \sin \alpha_i - U_i] \tan \varphi_i + c_i l_i\} + \\ + F_{i+1} [\cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i) \tan \varphi_i] \quad (4)$$

The equilibrium condition will be fulfilled by introducing the factor of safety  $SFS$  into the analysis such that the strength parameters of a given soil  $c$  and  $\tan \varphi$  are divided by this value. Eq. (4) then becomes

$$F_i = (W_i + Fy_i) \sin \alpha_i - Fx_i \cos \alpha_i - \\ - \{[(W_i + Fy_i) \cos \alpha_i + Fx_i \sin \alpha_i - U_i] \tan \varphi_i + c_i l_i\} / SF + \\ + F_{i+1} [\cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i) \tan \varphi_i / SF] \quad (5)$$

Eq. (5) then gives the searched factor of safety  $SF$  through the process of iteration. This process proceeds such that the force  $F_n$  equal to  $0 \text{ kN}$  is applied at the highest (end) point of the slip surface. The forces  $F_i$  acting in between blocks are determined for a given value of the factor of safety  $SF$  from Eq. (5). This step is repeated for various values of  $SF$  until we find such  $SF$  for which the force  $F_0$  at the slope base becomes equal to  $0 \text{ kN}$ . No tension is assumed along the slip surface. If the equilibrium condition requires the value of normal force  $N_i$  being negative, which means that the soil is loaded in tension, then the value of this force is set equal to zero in the next iteration step and the shear force  $T_i$  acting on a given segment is determined based on the soil cohesion only.

The ITF method is quite sensitive with respect to the shape of the slip surface. In case the slip surface contains sharp segment discontinuities the resulting factor of safety is generally larger as compare to reality. It is recommended that the slope difference between adjacent segments of the slip surface be less than  $10^\circ$ . This is checked automatically by the program and if the



slope difference is found greater the programs prompts a warning that the results might be overestimated. This is usually not the problem of a circular slip surface but should be kept in mind in case of polygonal slip surfaces.

### ITF Method - explicit solution

The explicit solution of the ITF method assumes a different way of introducing the factor of safety into the analysis. The mathematical solution then does not require iterations and the resulting factor of safety is calculated directly in one step. With this approach the resulting factor of safety is typically higher which may the solution totally devalued, particularly in cases concerning polygonal slip surfaces with large slope differences of adjacent segments.

The solution exploits Eq. (4) to which the factor of safety  $SF$  is introduced such that it multiplies the active components of forces, i.e. the components acting in the sliding direction. The equilibrium condition then becomes :

$$F_i = [(W_i + Fy_i) \sin \alpha_i - Fx_i \cos \alpha_i] SF - \{ [(W_i + Fy_i) \cos \alpha_i + Fx_i \sin \alpha_i - U_i] \tan \varphi_i + c_i l_i \} + F_{i+1} [\cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i) \tan \varphi_i] \quad (6)$$

For lucidity we introduce the component of active forces as:

$$A_i = (W_i + Fy_i) \sin \alpha_i - Fx_i \cos \alpha_i$$

and next the component of passive forces as:

$$P_i = [(W_i + Fy_i) \cos \alpha_i + Fx_i \sin \alpha_i - U_i] \tan \varphi_i + c_i l_i$$

and an auxiliary function:

$$\psi_i = \cos(\alpha_{i+1} - \alpha_i) - \sin(\alpha_{i+1} - \alpha_i) \tan \varphi_i$$

Eq. (6) can be then written in more compact form as:

$$F_i = A_i SF - P_i + F_{i+1} \psi_{i+1} \quad (7)$$

The next step assumes the known force  $F_n=0$  to provide expressions of forces between blocks  $F$  in the form:

$$\begin{aligned} F_{n-1} &= A_{n-1} SF - P_{n-1} \\ F_{n-2} &= A_{n-2} SF - P_{n-2} + (A_{n-1} SF - P_{n-1}) \psi_{n-1} \\ F_{n-3} &= A_{n-3} SF - P_{n-3} + (A_{n-2} SF - P_{n-2}) \psi_{n-2} + (A_{n-1} SF - P_{n-1}) \psi_{n-1} \psi_{n-2} \end{aligned}$$

Etc....

$$F_0 = \left[ A_0 + \sum_{i=1}^{n-1} \left( A_i \prod_{j=1}^i \psi_j \right) \right] SF - P_0 - \sum_{i=1}^{n-1} \left( P_i \prod_{j=1}^i \psi_j \right) \quad (8)$$

And since the force on the bottom origin of the slip surface should be equal to  $0 \text{ kN}$ , we get the final form of the factor of safety  $SF$  as:

$$SF = \frac{P_0 + \sum_{i=1}^{n-1} \left( P_i \prod_{j=1}^i \psi_j \right)}{A_0 + \sum_{i=1}^{n-1} \left( A_i \prod_{j=1}^i \psi_j \right)} \quad (9)$$

## Optimization of Polygonal Slip Surface

The slip surface optimization proceeds such that the program changes subsequently locations of individual points of this surface and checks, which change of location of a given point results in the maximal reduction of the factor of slope stability  $SF$ . The end points of the optimized slip surface are moved on the ground surface, internal points are moved in the vertical and horizontal directions. The step size is initially selected as one tenth of the smallest distance of neighboring points along the slip surface. With every new run the step size is reduced by one half. Location of points of slip surface is optimized subsequently from the left to the right and it is completed when there was no point moved in the last run.

When optimizing the polygonal slip surface the iteration process may suffer from falling into the **local minimum** (with respect to gradual evolution of locations of nodal points) so not always the process is terminated by locating the critical slip surface. Especially in case of complex slope profile it is therefore advantageous to introduce several locations of the initial slip surface. Combination with the approach used for circular slip surfaces is also recommended. Therefore, the critical slip surface assuming circular shape is found first and the result is then used to define the initial polygonal slip surface.

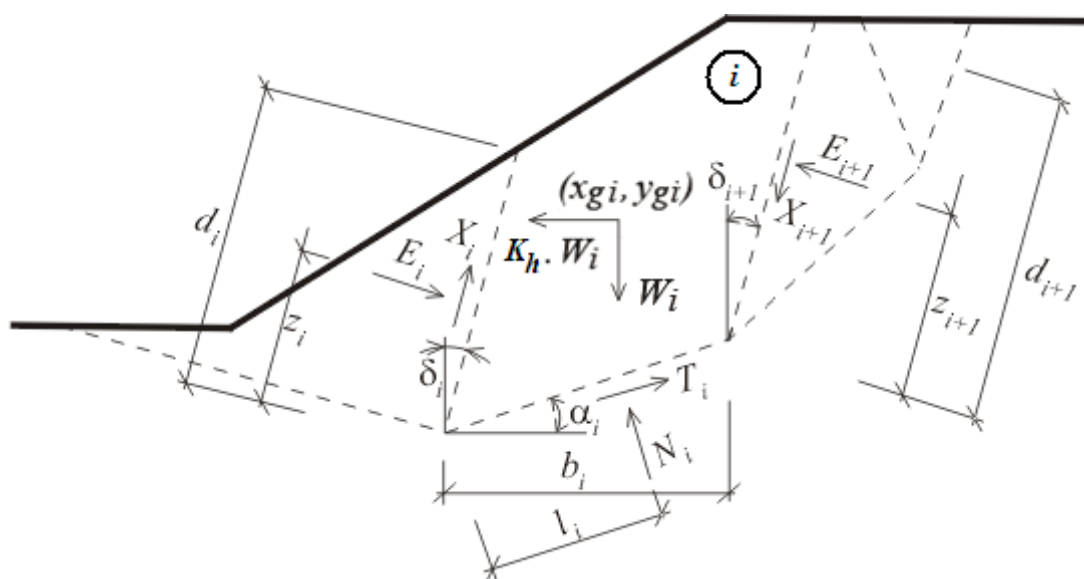
The optimization process can be **restricted by various constraints**. This becomes advantageous especially if we wish the searched slip to pass through a certain region or to bypass this region. The restriction on the optimization process can be performed in two different ways:

1. Optimization restrictions are specified as a set of segments in a soil body. The optimized slip surface is then forced to bypass these segments during optimization.
2. Another way of restricting the optimization process is to fix location of selected points along the optimized slip surface or allow for moving these points only in one of two directions, either vertically or horizontally.

## Changing the Inclination of Dividing Planes

Changing inclination of dividing planes is applied for Sarma's method only. It is evident from figure that the planes dividing individual blocks do not have to be vertical and not even mutually parallel. In the first stage of analysis when the **optimization** procedure moves points along the slip surface it assumes vertical alignment of dividing planes. To arrive at even smaller value of the slope stability it is possible to change the mutual alignment of dividing planes. This process is again performed in several runs with limited value of rotation step and this step is again reduced in the course of optimization. This stage of optimization is

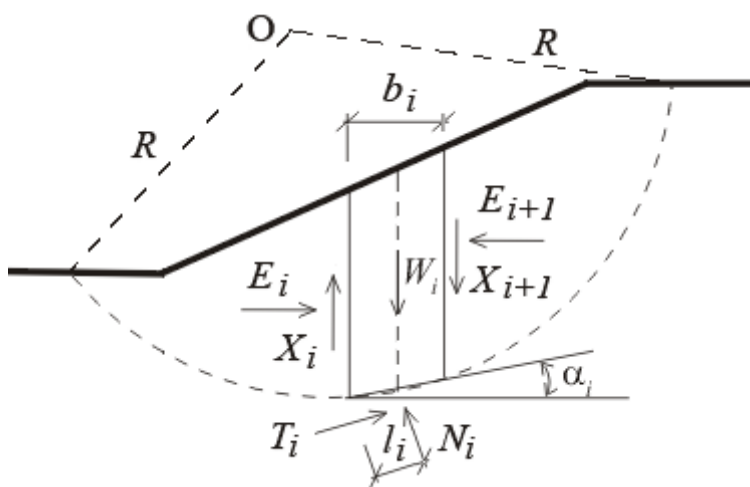
terminated once the rotation step drops below the value of  $1^\circ$  and no change of rotation occurred during the last optimization run.



Static scheme - Sarma method

## Circular Slip Surface

All methods of limit equilibrium assume that the soil body above the slip surface is subdivided into blocks (dividing planes between blocks are always vertical). Forces acting on individual blocks are displayed in figure.



Static scheme of slice

Here,  $X_i$  and  $E_i$  are the shear and normal forces acting between individual blocks,  $T_i$  and  $N_i$  are the shear and normal forces on individual segments of the slip surface,  $W_i$  are weights of individual blocks.

Individual methods of slices differ in their assumptions of satisfying the force equations of equilibrium and the moment equation of equilibrium with respect to the center **O**.

The program allows for adopting one of the following methods:

- Fellenius / Petterson
- Bishop

- Spencer
- Janbu
- Morgenstern-Price
- Shahunyants
- ITF Method

Ground water specified within the slope body (using one of the five options) influences the analysis in two different ways. First when computing the weight of a soil block and second when determining the shear forces. Note that the effective soil parameters are used to relate the normal and shear forces.

### Introducing anchor forces and water above the ground surface into the analysis

Anchor forces are considered as external load applied to the slope. They are taken with respect to one running meter [ $kN/m$ ] and introduced into the moment equation of equilibrium. These forces should contribute to additional stability, if that cannot be achieved in a different way. There is no limitation to the magnitudes of anchor forces and therefore it is necessary to work with realistic values.

Influence of water above the ground surface is considered as set forces acting perpendicular on the ground surface together with pore pressure along the slip surface, which is derived depending on the depth of slip surface measured from the ground water table. The forces acting on the ground surface enter the moment equation of equilibrium as forces acting on respective arms measured towards the center of the slip surface.

Optimization of circular slip surface searches the most critical surface (the lowest  $SF$ ).

## Fellenius / Petterson

The simplest method of slices assumes only the overall moment equation of equilibrium written with respect to the center of the slip surface. The shear and normal forces between blocks  $X_i$  and  $E_i$  are neglected. The factor of safety  $SF$  follows directly from the following expression:

$$SF = \frac{1}{\sum_i W_i \cdot \sin \alpha_i} \cdot \sum_i [c_i \cdot l_i + (N_i - u_i \cdot l_i) \cdot \tan \varphi_i]$$

- where:
- $u_i$  - pore pressure within block
  - $c_i, \varphi_i$  - effective values of soil parameters
  - $W_i$  - block weight
  - $N_i$  - normal force on the segment of the slip surface
  - $\alpha_i$  - inclination of the segment of the slip surface
  - $l_i$  - length of the segment of the slip surface

Literature:

Petterson KE (1955) *The early history of circular sliding surfaces. Geotechnique* 5:275-296.

## Bishop

The simplified Bishop method assumes zero  $X_i$  forces between blocks. The method is based on

satisfying the moment equation of equilibrium and the vertical force equation of equilibrium. The factor of safety  $SF$  is found through a successive iteration of the following expression:

$$FS = \frac{1}{\sum_i W_i \cdot \sin \alpha_i} \cdot \sum_i \frac{c_i \cdot b_i + (W_i - u_i \cdot b_i) \cdot \tan \varphi_i}{\cos \alpha_i + \frac{\tan \varphi_i \cdot \sin \alpha_i}{FS}}$$

where:

- $u_i$  - pore pressure within block
- $c_i, \varphi_i$  - effective values of soil parameters
- $W_i$  - block weight
- $\alpha_i$  - inclination of the segment of the slip surface
- $b_i$  - horizontal width of the block

*Literature:*

*Bishop, A.W. (1955) "The Use of the Slip Circle in the Stability Analysis of Slopes", Geotechnique, Great Britain, Vol. 5, No. 1, Mar., pp. 7-17.*

## Spencer

This method assumes non-zero forces between blocks. The resultants of shear and normal forces acting between blocks have constant inclinations. The Spencer method is a rigorous method in a sense that it satisfies all three equations of equilibrium - the force equations of equilibrium in the horizontal and vertical directions and the moment equation of equilibrium. The factor of safety  $SF$  is found through the iteration of inclination of forces acting between blocks and the factor of safety  $SF$ . Further details can be found in section describing the analysis of [polygonal slip surface](#).

## Janbu

Janbu method assumes non-zero forces between blocks. Method satisfies the force equations of equilibrium in the horizontal and vertical directions for all blocks and the moment equation of equilibrium for all but the last (uppermost) slice. Assumption of this method is choice of position of forces acting between the blocks. The factor of safety  $SF$  is found through the iteration of forces acting between blocks and then inclinations of these forces are calculated. Further details can be found in section describing the analysis of [polygonal slip surface](#).

## Morgenstern-Price

This method assumes non-zero forces between blocks. The resultants of shear and normal forces acting between blocks have different inclinations at each block (shape of Half-sine function). Morgenstern-Price is a rigorous method in a sense that it satisfies all three equations of equilibrium - the force equations of equilibrium in the horizontal and vertical directions and the moment equation of equilibrium. The factor of safety  $SF$  is found through the iteration of inclination of forces acting between blocks and the factor of safety  $SF$ . Further details can be found in section describing the analysis of [polygonal slip surface](#).

## Shahunyants

Further details can be found in section describing the analysis of [polygonal slip surface](#).

## ITF Method (Imbalance Thrust Force Method)

Further details can be found in section describing the analysis of [polygonal slip surface](#).

## Optimization of Circular Slip Surface

The goal of the optimization process is to locate a slip surface with the smallest factor of slope stability  $SF$ . The circular slip surface is specified in terms of 3 points: two points on the ground surface and one inside the soil body. Each point on the surface has one degree of freedom while the internal point has two degrees of freedom. The slip surface is defined in terms of four independent parameters. Searching for such a set of parameters that yields the most critical results requires sensitivity analysis resulting in a matrix of changes of parameters that allows fast and reliable optimization procedure. The slip surface that gives the smallest factor of slope stability is taken as the critical one. Parameters of individual slip surfaces and results from optimization runs can be displayed in output document.

This approach usually succeeds in finding the critical slip surface without encountering the problem of falling into a local minimum during iteration. It therefore appears as a suitable starting point when optimizing general slip surfaces such as the polygonal slip surface.

The optimization process can be [restricted by various constraints](#). This becomes advantageous especially if we wish the searched slip surface to pass through a certain region or to bypass this region. Optimization restrictions are specified as a set of segments in a soil body. The optimized slip surface is then forced to bypass these segments during optimization.

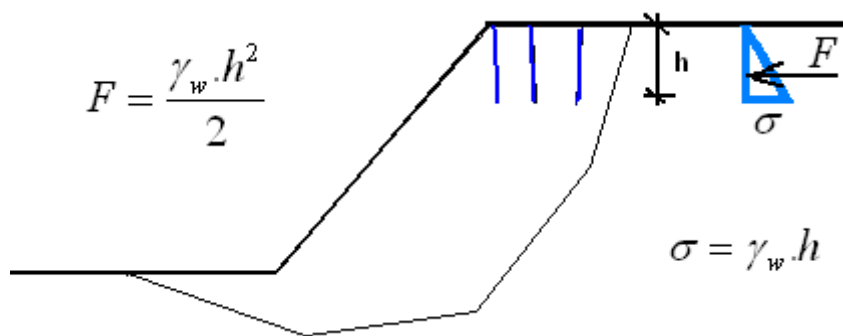
## Foliation

Soils can be introduced with foliation. It means that along an angle specified in terms of a certain interval, which in turn is introduced as one of the soil parameters **<Starting Slope; End Slope>** the soil experiences significantly different (usually worse) parameters ( $c$  and  $\varphi$ ).

If the slope of a slip surface segment or the slope of interface between blocks is assumed within the interval **<Starting Slope; End Slope>**, the analysis proceeds with the modified parameters of  $c$  and  $\varphi$ .

## Influence of Tensile Cracks

The program makes it possible to account for the influence of tensile cracks that appear on terrain surface and are filled with water  $h$ . The only input parameter is the depth of tensile cracks. The effect of cracks is incorporated when calculating normal and shear forces in sections of a slip surface containing cracks - in a section with tensile cracks the shear strength parameters are set to zero ( $c = 0, \varphi = 0$ ). Next, a horizontal force  $F$  due to presence water in a tensile crack is introduced in the analysis (see figure):

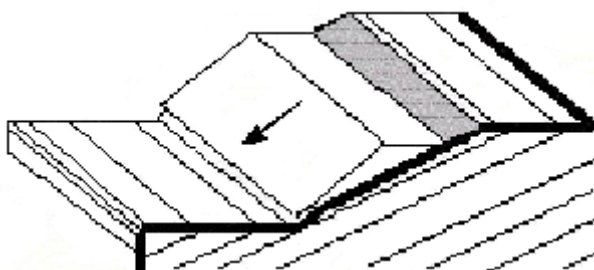


*Influence of tensile cracks*

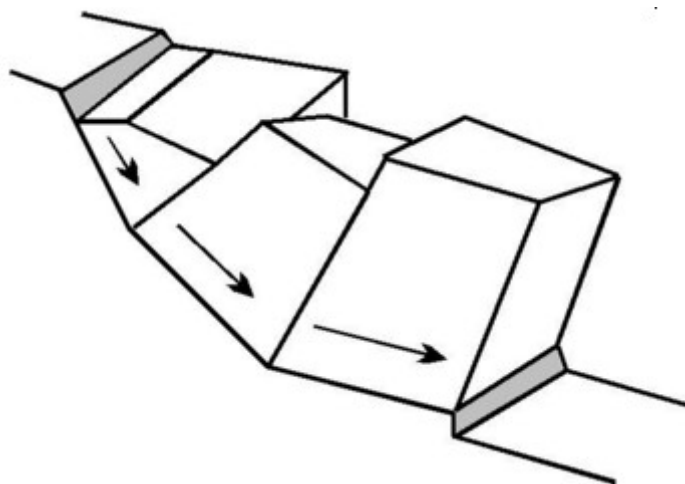
## Rock Stability

Program for stability analysis of rock slope treats the following types of failure of rock faces:

- Sliding on the plane slip surface
- Translation on the polygonal slip surface
- Fall of the rock wedge



*Failure of a rock face due to sliding on the plane slip surface*



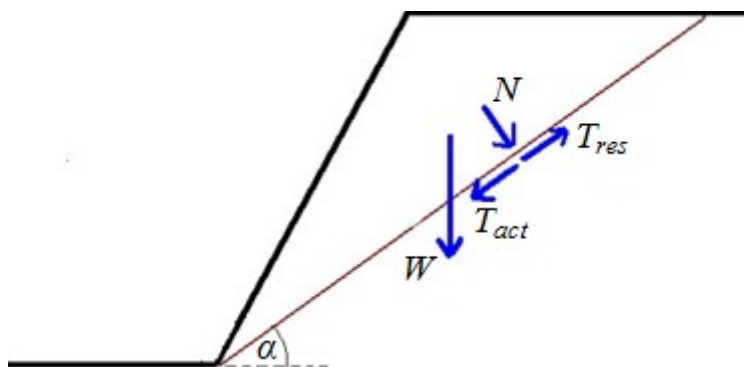
*Translation on the polygonal slip surface*



*Fall of the rock wedge*

## Plane Slip Surface

Failure on the plane slip surface is manifested by a rock block sliding down along this surface. The rock wedge can be specified with a tension crack. The solution procedure of stability requires determination of the **normal force**  $N$  acting on the slip surface, the **shear force**  $T_{act}$  (**active**) and the **resisting shear force**  $T_{res}$  (**passive**).



*Forces on the slip surface*

The **shear strength parameters** and the normal force  $N$  acting on the slip surface are the main input data for the determination of the resisting shear forces  $T_{res}$ . Calculation of the **active shear force**  $T_{act}$  and the **normal force**  $N$  is further influenced by the own weight of block (depends on the geometry and **unit weight of rock**), **anchorage**, **surcharge**, **influence of water** and **seismic effects**. The active force  $T_{act}$  and the normal force  $N$  are determined as a sum of all forces entering the analysis.

The program offers several types of plane slip surfaces:

- Smooth
- **Undulated**
- **Stepped**

The resulting verification can be carried out either according to the selected **verification methodology** based on the input in the "Settings" frame.

## Stepped Slip Surface

If the rock body contains a system of parallel discontinuous cracks inclined to the top face of a rock and the second system is indistinctive, then it is possible to consider a formation of a stepped (jagged) slip surface in the rock body. This surface can be introduced into the program using the Calla and Nicholas theory, which increases resistance on the stepped slip

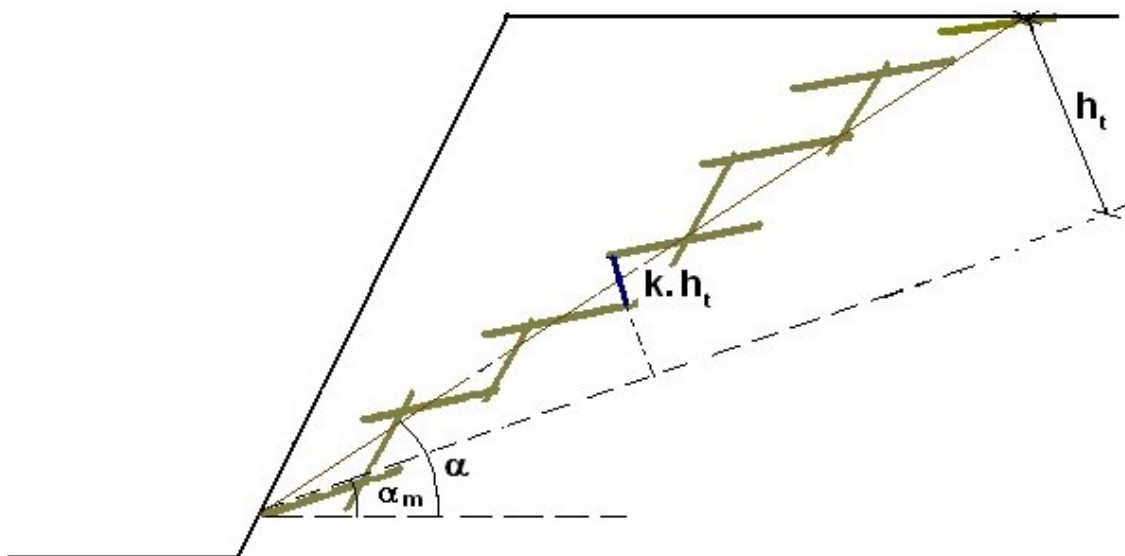


surface by  $\Delta\tau$ .

$$\Delta\tau = \sigma_n \operatorname{tg} \nu + T$$

$$T = \sum k \cdot h_t \cdot T_0$$

- where:
- $\sigma_n$  - normal stress acting in the direction normal to the slip surface
  - $\nu$  - waviness angle
  - $T$  - effective tensile strength of steps in the intact rock
  - $k$  - part of the height  $h_t$  associated with steps in the intact rock (not created by a secondary system of planes)  $\sum k \in \langle 0,1 \rangle$
  - $h_t$  - normal height of stepped wedge resting on an inclined plane of principal system of discontinuity planes
  - $T_0$  - **tensile strength of intact rock**



Stepped slip surface

Literature:

W.S. Dershowitz, H.H. Einstein - Characterizing rock joint geometry with joint system models  
*Journal Rock Mechanics and Rock Engineering, Springer Wien ISSN 0723-2632 , Issue Volume 21, Number 1 / January, 1988 Pages 21-51.*

## Tensile Strength of Rock

Tensile strength  $T_e$  is 20 to 30x smaller than the strength of rock in simple compression  $\sigma_c$ .

Strength in simple tension  $T_0$  for selected intact rocks [MPa]

Basalt	3 - 18
Gneiss	7 - 16
Granite	11 - 21
Limestone	3 - 5
Marble	7 - 12
Quarzite	4 - 23
Sandstone	5 - 11
Schist	5 - 12
Slate	2 - 17
Tuff	2 - 4

## Undulated Slip Surface

If undulated surface is considered (on scale 1 to 10 m) - it is possible to account for slip surface waviness by angle  $\nu$ :

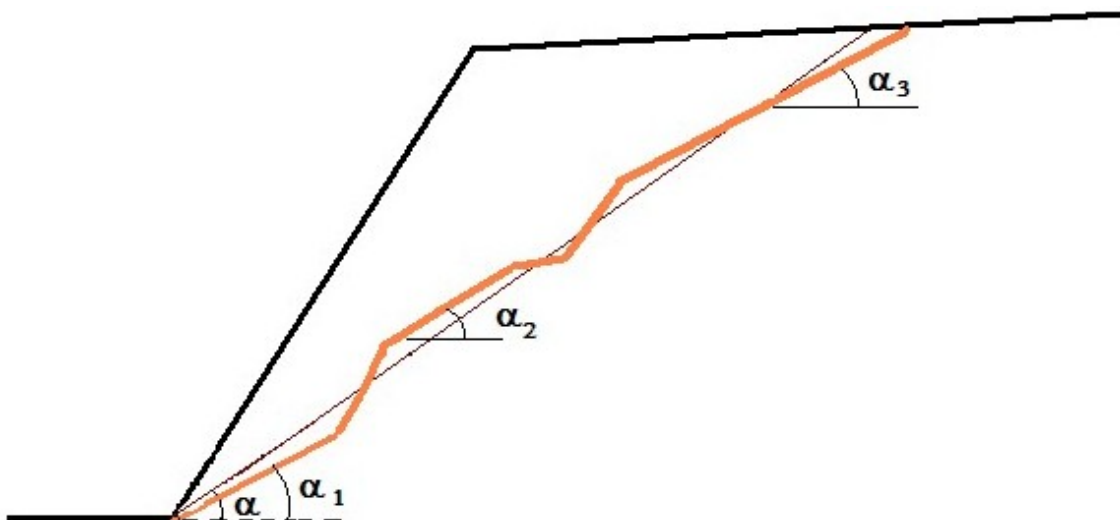
$$\nu = \alpha - \min(\alpha_i)$$

where:  $\alpha$  - slip surface gradient  
 $\alpha_i$  - gradient of the i-th fault of slip surface

The waviness increases the tensile strength  $\tau$  on slip surface by  $\Delta\tau$ :

$$\Delta\tau = \sigma_n \cdot \text{tg} \nu$$

where:  $\sigma_n$  - normal stress acting in the direction normal to the slip surface  
 $\nu$  - waviness angle



### Undulated slip surface

#### Literature:

Miller, S.M. (1988). *Modeling Shear Strength at Low Normal Stresses for Enhanced Rock Slope Engineering, Proc. Of 39th Highway Geology Symp, 346-356.*

## Anchorage of Rock Slope

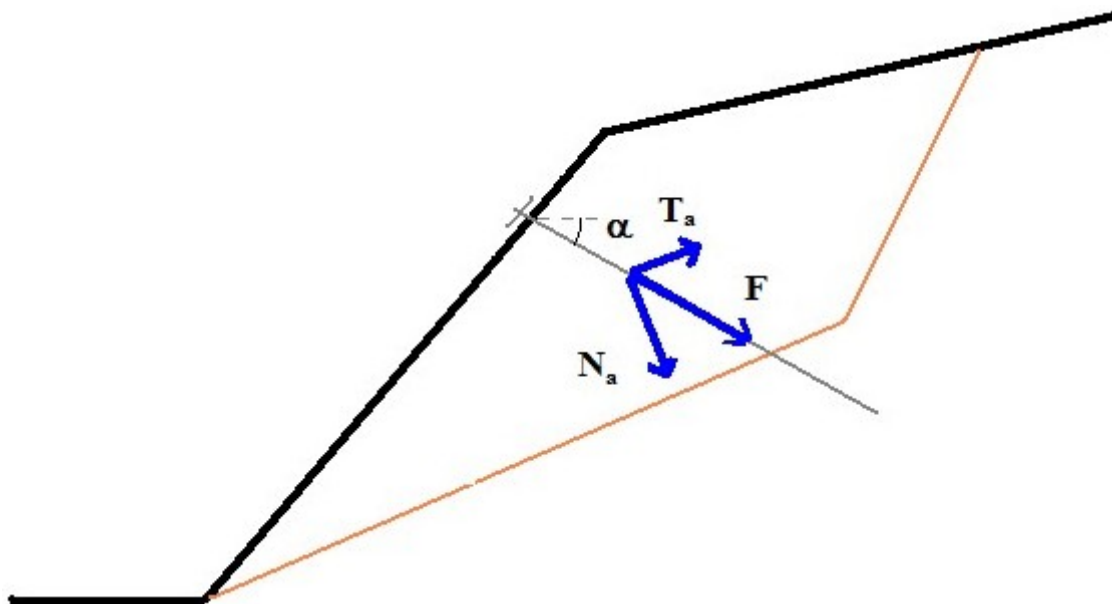
Two types of anchors can be defined when running the slope stability analysis on a plane slip surface:

### Active

An active anchor is represented by a pre-stressed anchor, for which the anchor forces are activated before the sliding of a rock block takes place. The normal force increases the normal stress on a slip surface and as such also the resisting forces; the tangent component of the normal force is either added to or subtracted from the shear (active) forces.

### Passive

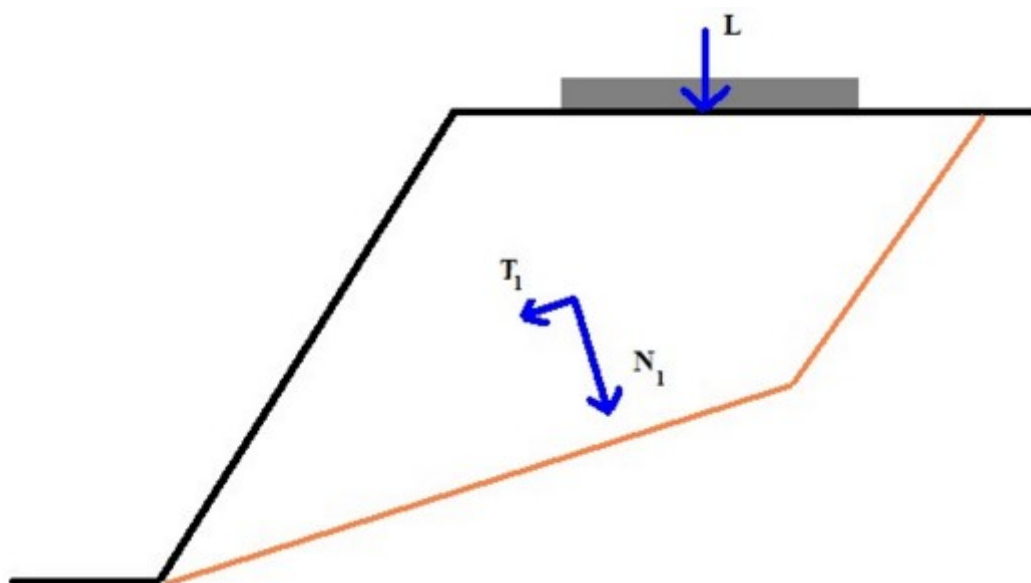
A passive anchor is activated by sliding of a rock block (i.e. not pre-stressed anchors). The normal force increases the normal stress on a slip surface and as such also the resisting forces; the tangent component of the normal force is added to the resisting forces.



*Resolution of anchor force*

## Surcharge of Rock Slope

The surcharge resultant is determined first. The normal component of the resultant force increases the normal stress on a slip surface and as such also the resisting forces  $T_{res}$ , the tangent component is either added to or subtracted from the shear (active) forces  $T_{act}$ .



*Resolution of surcharge*

## Influence of Water Acting on Slip Surface

The following options to account for water effects are available in the program:



Without ground water, water is not considered



Hydrostatic pressure, GWT above toe of slope



Hydrostatic pressure, GWT on tension crack



Hydrostatic pressure, GWT on tension crack, max



Hydrostatic pressure, water acting on tension crack only



Own water force acting on slip surface only



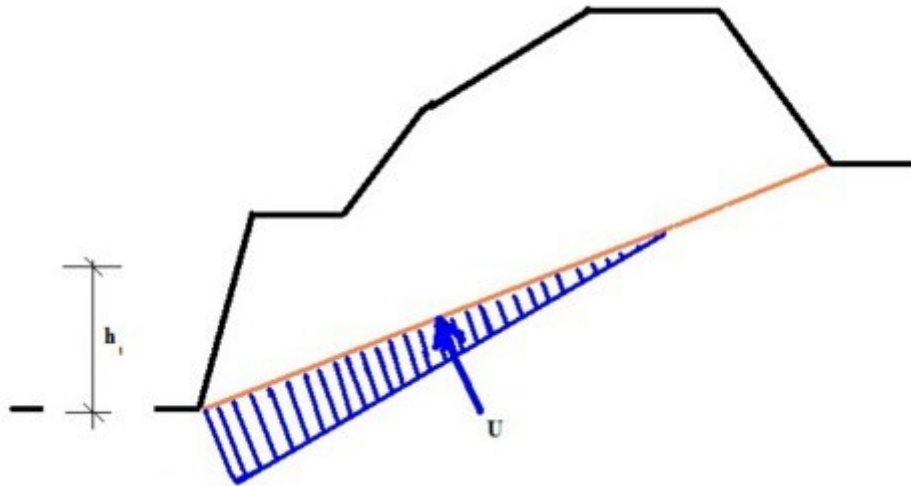
Own water force beha

## GWT Above Slope Toe



### Hydrostatic pressure, GWT above toe of slope

The slip surface is either entirely or partially below the ground water table (water can not outflow from slip surface), the maximal water pressure is at the toe of face.



*Hydrostatic pressure on slip surface*

The value of water pressure  $u$  at the heel of slope is given by:

$$u = \gamma_w \cdot h_t$$

where:  $\gamma_w$  - unit weight of water  
 $h_t$  - height of GWT above toe of slope

The resulting compressive water force acting in the direction normal to the slip surface is given by:

$$U = \frac{1}{2} \cdot \gamma_w \cdot h_t \cdot \left( \frac{h_t}{\sin \alpha} \right)$$

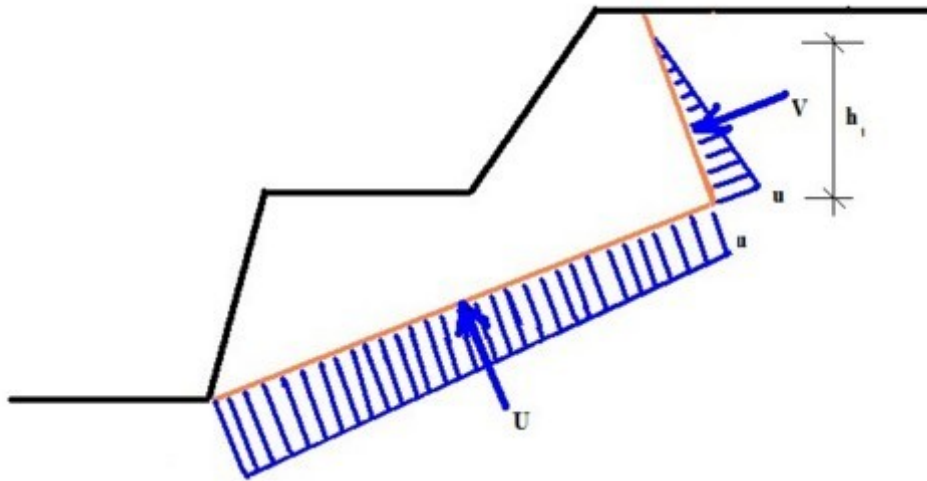
where:  $\gamma_w$  - unit weight of water  
 $h_t$  - height of GWT above toe of slope  
 $\alpha$  - deflection of slip surface from horizontal

## GWT on Tension Crack



### GWT on tension crack

The slip surface is entirely below the ground water table; the GWT either intersects the tension crack or is aligned with terrain, the maximal value of uplift pressure is at the toe of face.



*Hydrostatic pressure on slip surface and on tension crack, max. value at the toe of slope*

The value of uplift pressure  $u$  at the intersection of slip surface and tension crack is given by:

$$u = \gamma_w \cdot h_t$$

where:  $\gamma_w$  - unit weight of water

$h_t$  - height of GWT above the line of intersection of slip surface and tension crack

The resulting compressive water force  $V$  acting in the direction normal to the tension crack is given by:

$$V = \frac{1}{2} \cdot \gamma_w \cdot h_t \cdot \left( \frac{h_t}{\sin \varphi} \right)$$

where:  $\gamma_w$  - unit weight of water

$h_t$  - height of GWT above the line of intersection of slip surface and tension crack

$\varphi$  - deflection of tension crack from vertical

The value of hydrostatic pressure  $u_1$  at the toe of slope is given by:

$$u_1 = \gamma_w \cdot H_w$$

where:  $\gamma_w$  - unit weight of water

$H_w$  - height of GWT above toe of slope

The resulting compressive water force  $U$  acting in the direction normal to the slip surface is given:

$$U = \frac{1}{2} \cdot (u + u_1) \cdot \left( \frac{H_w - h_t}{\sin \alpha} \right)$$

where:  $u$  - water pressure acting on the line of intersection of slip surface and tension crack

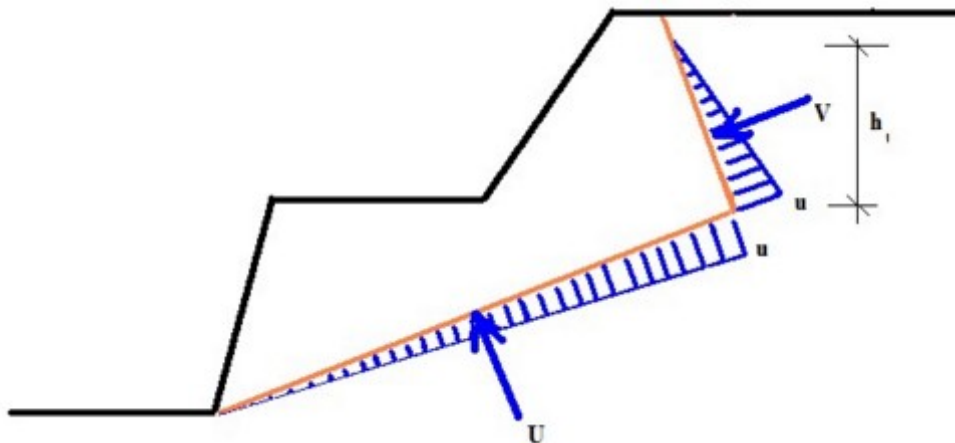
- $u_l$  - water pressure at toe of slope
- $h_t$  - height of GWT above the line of intersection of slip surface and tension crack
- $\alpha$  - deflection of slip surface from horizontal
- $H_w$  - height of GWT above toe of slope

## GWT on Tension Crack, Max. Tens. Crack



### GWT on tension crack

The slip surface is entirely below the ground water table, the GWT either intersects the tension crack or is aligned with terrain (water can outflow at the slope heel), the maximal value of uplift pressure is at the intersection of tension crack and slip surface.



*Hydrostatic pressure on slip surface and on tension crack*

The value of uplift pressure  $u$  at the intersection of slip surface and tension crack is given by:

$$u = \gamma_w \cdot h_t$$

where:  $\gamma_w$  - unit weight of water

$h_t$  - height of GWT above the line of intersection of slip surface and tension crack

The resulting compressive water force  $V$  acting in the direction normal to the tension crack is given by:

$$V = \frac{1}{2} \cdot \gamma_w \cdot h_t \cdot \left( \frac{h_t}{\sin \varphi} \right)$$

where:  $\gamma_w$  - unit weight of water

$h_t$  - height of GWT above the line of intersection of slip surface and tension crack

$\varphi$  - deflection of tension crack from vertical

The resulting value of pressure  $u_l$  at the toe of slope is equal to zero.

The resulting compressive water force  $U$  acting in the direction normal to the tension crack is given:

$$U = \frac{1}{2} u_l \left( \frac{H_w - h_t}{\sin \alpha} \right)$$

where:

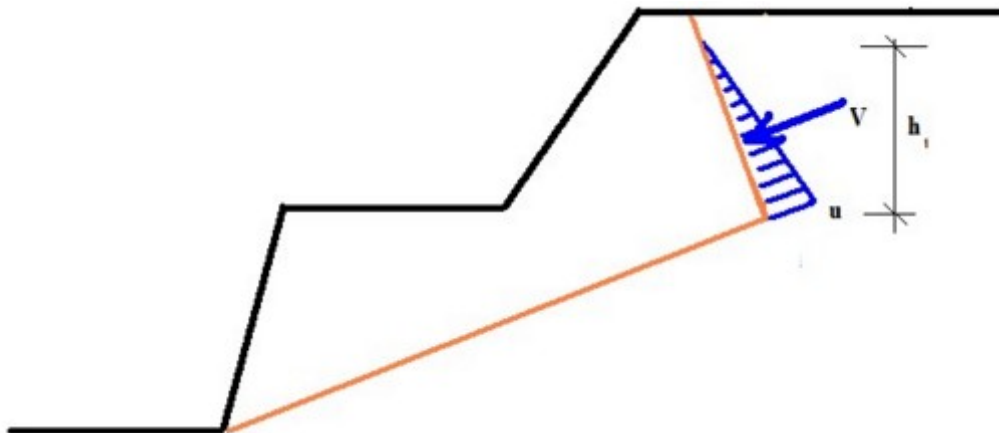
- $u$  - water pressure acting on the line of intersection of slip surface and tension crack
- $h_t$  - height of GWT above the line of intersection of slip surface and tension crack
- $\alpha$  - deflection of slip surface from horizontal
- $H_w$  - height of GWT above toe of slope

## Water Acting Only on Tension Crack



### Water acting on tension crack only

The slip surface is fully dry (seepage is not possible); the GWT either intersects the tension crack or is aligned with terrain, the maximal value of uplift pressure is at the intersection of slip surface and tension crack.



*Water acting on tension crack only*

The value of uplift pressure  $u$  at the intersection of slip surface and tension crack is given by:

$$u = \gamma_w \cdot h_t$$

where:

- $\gamma_w$  - unit weight of water
- $h_t$  - height of GWT above the line of intersection of slip surface and tension crack

The resulting compressive water force  $V$  acting in the direction normal to the tension crack is given by:



$$V = \frac{1}{2} \cdot \gamma_w \cdot h_t \cdot \left( \frac{h_t}{\sin \varphi} \right)$$

where:  $\gamma_w$  - unit weight of water  
 $h_t$  - height of GWT above the line of intersection of slip surface and tension crack  
 $\varphi$  - deflection of tension crack from vertical

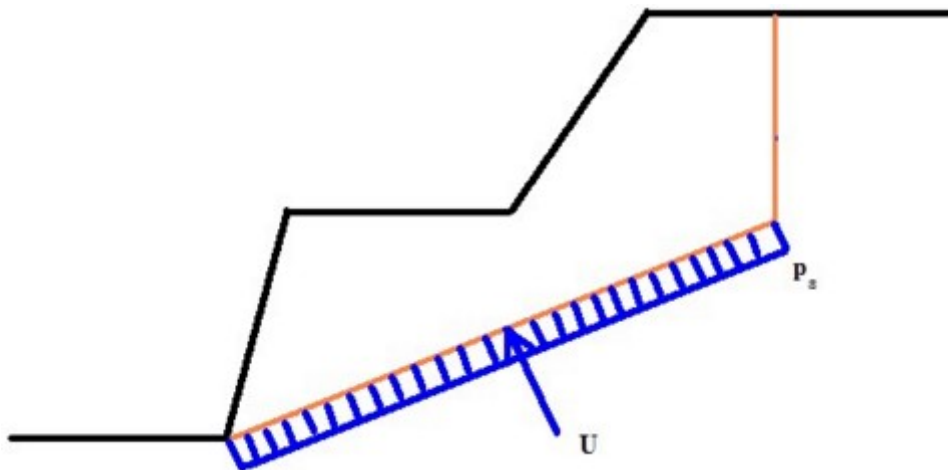
The value of water pressure acting on slip surface is equal to zero.

## Own Water Force Acting Only on Slip Surface



### Own water force acting on slip surface only

The program allows for a manual input of the constant value of water pressure  $p_s$  [kPa] acting on a slip surface.



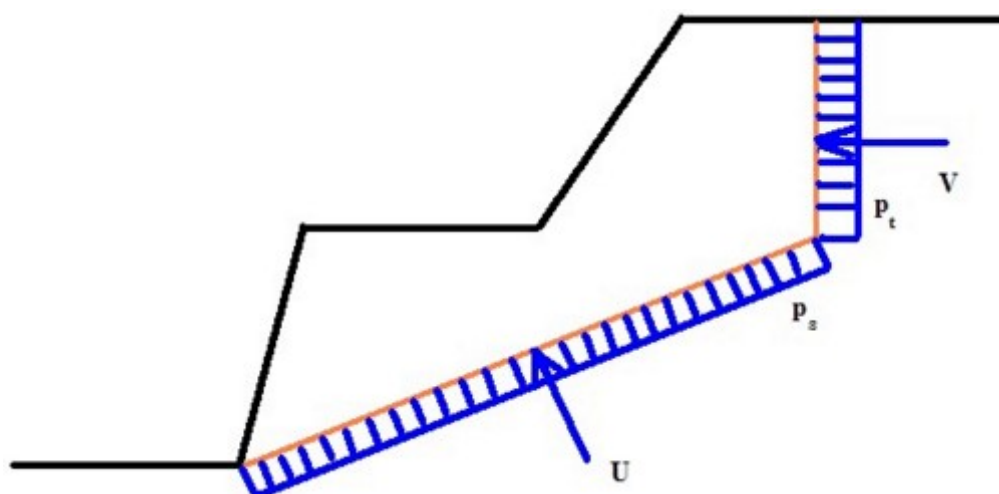
*Own value of water pressure acting on slip surface*

## Own Water Force Behavior



### Own water force behavior

The program allows for a manual input of the constant value of water pressure  $p_t$  [kPa] acting on the slip surface and  $p_t$  [kPa] acting on a tension crack.



*Own values of water pressure on slip surface and on tension crack*

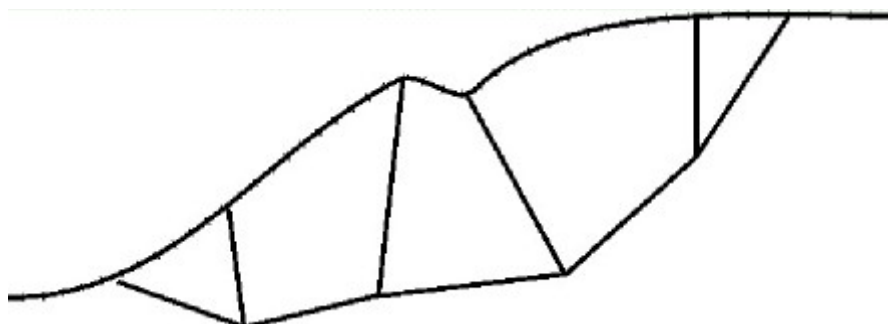
## Polygonal Slip Surface

The program performs stability analysis of rock blocks moving along the polygonal slip surface. Owing to the complexity of the general solution the program admits the following assumptions:

- Motion of rock blocks is only translational.
- Blocks translate along the polygonal slip surface formed either by planar planes or planes with moderate waviness.
- Rock blocks are divided by joints with known directions.
- Actual deformation of rock mass inside the blocks is negligible.
- Failure on the polygonal slip surface and along joints is driven by the **Mohr-Coulomb** failure criterion.
- The same factor of safety is assumed for whole polygonal slip surface.
- All rock blocks are in contact (opening of joints is not allowed).

The **Mohr-Coulomb** shear strength parameters on the slip surface and on joints separating individual block are the main input data for the determination of stability of rock blocks. The solution is further influenced by the weight of block (depends on the **geometry of block** and **unit weight of rock**), **anchorage**, **surcharge acting on the blocks**, **influence of water** and **seismic effects**.

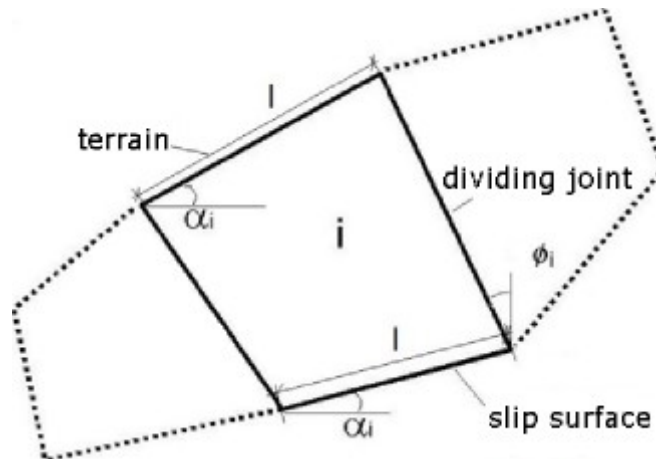
The basic theoretical grounds of the solution are described [here](#).



*Polygonal slip surface*

## Geometry of Rock Block

The block geometry is determined by the gradient  $\alpha$ , by the length of a given slip surface  $l$  and by the gradient of a dividing joints  $\phi$  separating the subsequent block as well as by the gradient  $\alpha$  and the length  $l$  of the top face of external surface of a rock slope (natural profile). Lengths of planes can be defined either by the total length or by the lengths of their horizontal and vertical projections. It is necessary to ensure the condition that all rock blocks are in contact (the opening between joints is not allowed).



Geometry of the  $i^{th}$  element

## Anchor Forces, Surcharge

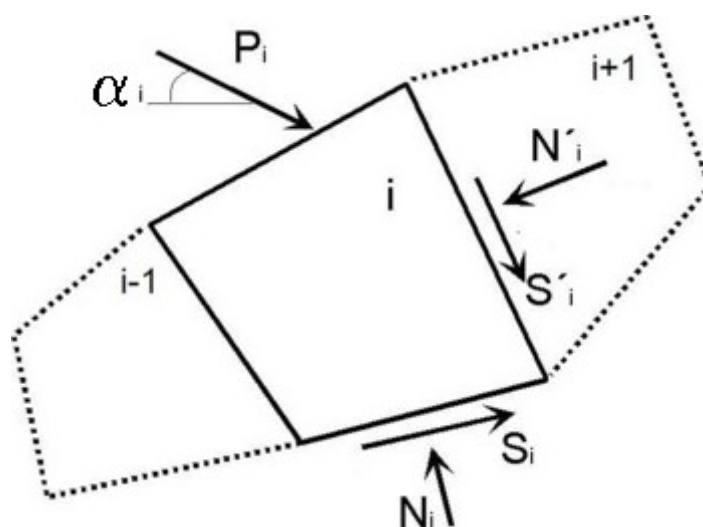
It is possible to introduce anchor forces and surcharges of rock blocks. The resultant of forces  $P_i$  [ $kN/m$ ] acting on the  $i^{th}$  block is then determined. All forces acting on the block including the water pressure on the slip surface and the block separating joints are taken into account.

### Surcharge acting on the block

It is possible to input surface, strip and trapezoidal surcharge of terrain. The program then determines their effect on individual rock blocks.

### Anchor forces

The applied anchor force is adjusted per 1  $m$  run based on the specified horizontal spacing of anchors.



External forces acting on the  $i^{th}$  element

### Influence of Water

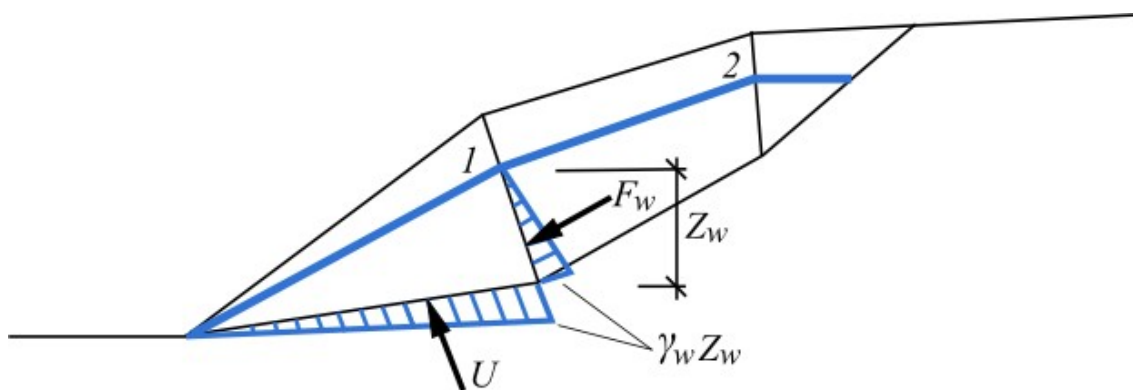
Influence of water can be considered using following options: **general shape of GWT**, **horizontal GWT** or directly by **water acting on the blocks**.

#### General shape of GWT

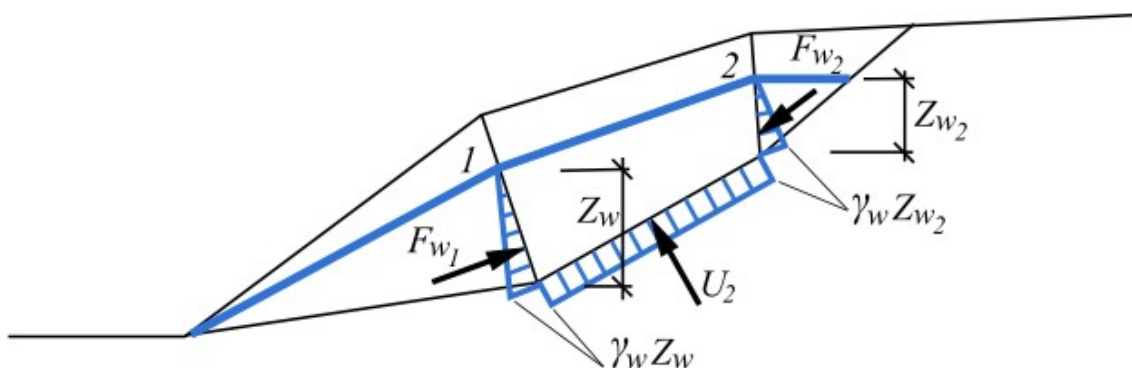
General shape of GWT is entered as a polygon. The pore pressure (stress) on the slip surface is considered linearly according to the equation:  $u = \gamma_w * z_w$ .

- where:
- $\gamma_w$  - unit weight of water
  - $z_w$  - height of GWT above slip surface in the joints

The resultant forces  $U$  (force due to water on slip surface) or  $F_v$  (force due to water on internal slip surface acting on the submerge part of joint per metre of width) are calculated from the pore pressure load diagrams.



Forces from the water acting on block - water can flow freely of the gap



*Forces from the water acting on block - water can't flow of the gap*

**Horizontal GWT**

The horizontal GWT is entered by constant height  $h_w$  over heel of the slope (from the origin of the coordinate system). Influence of water is considered from the water level (GWT) to a given point on the vertical.

**Water entered on blocks**

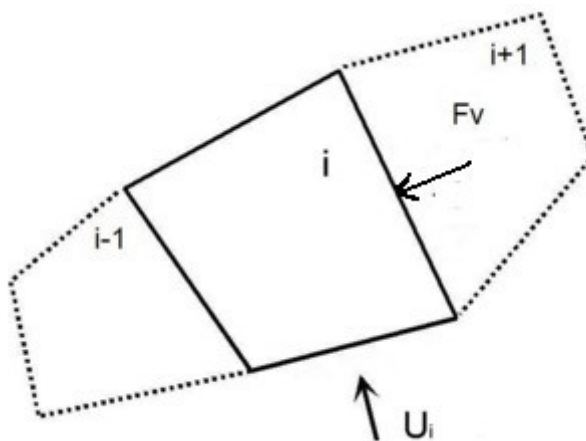
The water pressure along the joints and on the slip surface can be taken into account. It is introduced as external load:

**Force due to water on internal slip surface (water between blocks)  $F_v$**

It must be introduced into the analysis whenever the presence of water in the joints between blocks is expected. It is applied as a resultant force  $F_v$  in  $kN$  (the pressure acting on the immerse part of the joint per 1m run is considered).

**Force due to water on external slip surface (uplift pressure)  $U$**

It is defined as hydrostatic pressure on each slip surface of the polygon (external slip surface) separately and introduced as an external load (uplift pressure)  $U$  in  $kN$ , which can be reduced depending on the slip surface permeability (the pressure acting on the immerse part of the slip surface per 1m run is considered).

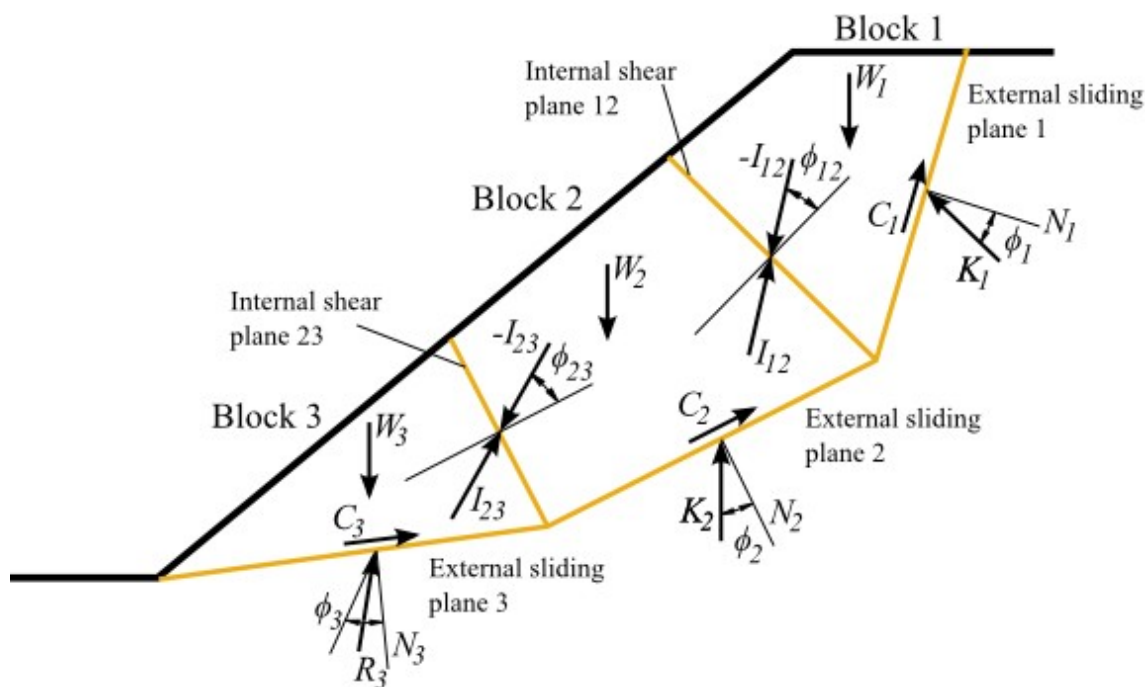


*Water forces acting on a rock block*

**Solution Procedure**

Principle of calculation of rock slope stability for polygonal slip surface is shown in following

figure.



*Forces acting on multiple-block sliding surface*

Suppose  $D_1$  is the vector representing the resultant of all the disturbing forces acting on block no. 1 given by:

$$D_1 = W_1 + E_1 + U_1 + V_1$$

- where:  $W_1$  - vector due to self weight of block which is acting vertically downwards
- $E_1$  - vector of external force due to earthquake
- $U_1$  - force vector due to uplift water force which acts normally to the sliding surface
- $V_1$  - vector of water force in tension cracks

If  $N_1$  denotes the unit vector for  $N_1$  and the angle of friction for plane no. 1 is  $\phi_1$ , block 1 would be an **active and unstable** block if the resultant falls outside the **friction cone** of plane no. 1 such that:

$$N_1 R_1 > \cos \phi_{m1}$$

- where:  $R_1$  - net resultant unit force of disturbing and resisting forces acting on block no. 1
- $N_1$  - unit vector representing the upward normal of plane no. 1
- $\phi_{m1}$  - mobilized angle of internal friction

For an active block, there will be a net transfer of interaction force from block 1 to the next lower block 2. The interaction force is denoted by vector  $I_{12}$  given by:

$$I_{12} = K_1 - R_1$$

where:  $K_1$  - reaction force for block 1 which acts at an angle  $\phi_1$  away from the normal of sliding plane 1 for the condition of equilibrium according to the Fig. 1

A similar method of analysis may be conducted for block 2 in addition to which an equal and opposite interaction force  $I_{12}$  has to be taken into account. The net resultant force vector  $R_2$  is given by:

$$R_2 = D_2 + C_{m2} + B_2 - I_{12}$$

where:  $D_2$  - resultant of all the disturbing forces acting on block no. 2

$C_{m2}$  - vector denoting the mobilised shear force

$B_2$  - vector denoting the resultant of external resisting forces acting on block 1 contributed by rock bolts or sprayed concrete

$I_{12}$  - interaction force vector from block 1 to the next lower block 2

The stability analysis is hence conducted in a sequential manner for all blocks ranging from the uppermost block 1 until the lowest block  $n$ . The entire system of blocks is deemed to be stable if the resultant force of the lowest block lies within the friction cone of the sliding surface. In Figure 1, for example, where the lowest block is numbered 3, the entire system of blocks would be stable, with passive blocks supporting the active blocks of the system, provided that:

$$N_3 R_3 \leq \cos \phi_{m3}$$

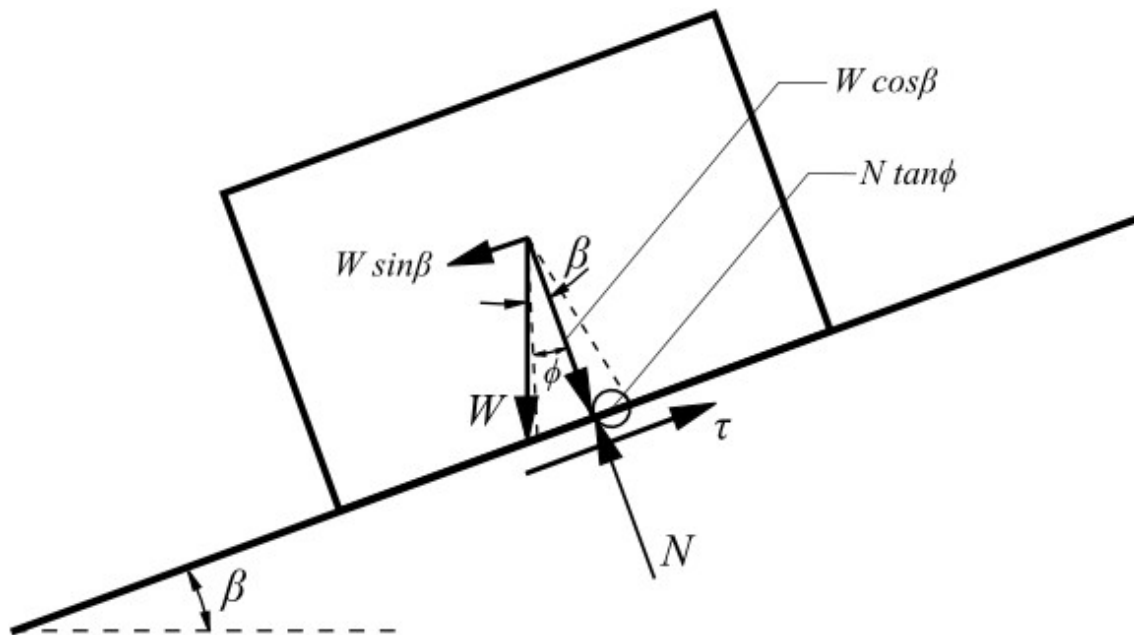
where:  $R_3$  - net resultant unit force of disturbing and resisting forces acting on block no. 3

$N_3$  - unit vector representing the upward normal of plane no. 3

$\phi_{m3}$  - mobilized angle of internal friction

## Cone Friction Concept

**Friction cone concept** is a combination of kinematic and kinetic calculation method, which is used to find potential **slip surface of failure**. The principle of solution is shown in the following figure.



The friction cone concept for a block resting on an inclined plane (polygonal slip surface)

The resistant forces are described using following condition:

$$N \tan \phi = c A + W \cos \beta \tan \phi$$

- where:
- $A$  - block area resting on the slip surface
  - $c$  - cohesion on the sliding surface
  - $W$  - vector due to self weight of block (self weight resultant of the rock block)
  - $N$  - normal to the slip plane
  - $\phi$  - angle of internal friction

Literature:

Goodman, R. E.: *Introduction to Rock Mechanics*: John Wiley & Sons, New York, 1989, 562 p.

## Rock Wedge

The program performs stability analysis of a rock wedge that is wedged in between two surfaces (planes) and slides in the direction of the line of intersection (tray) of these planes. The rock wedge can be specified with a tension crack. Gradient of this intersection must be considerably larger than the angle of internal friction dividing planes, whereas the falling line of both dividing planes must be directed towards the line of intersection. It is further assumed that the tray is located in a stable rock body.

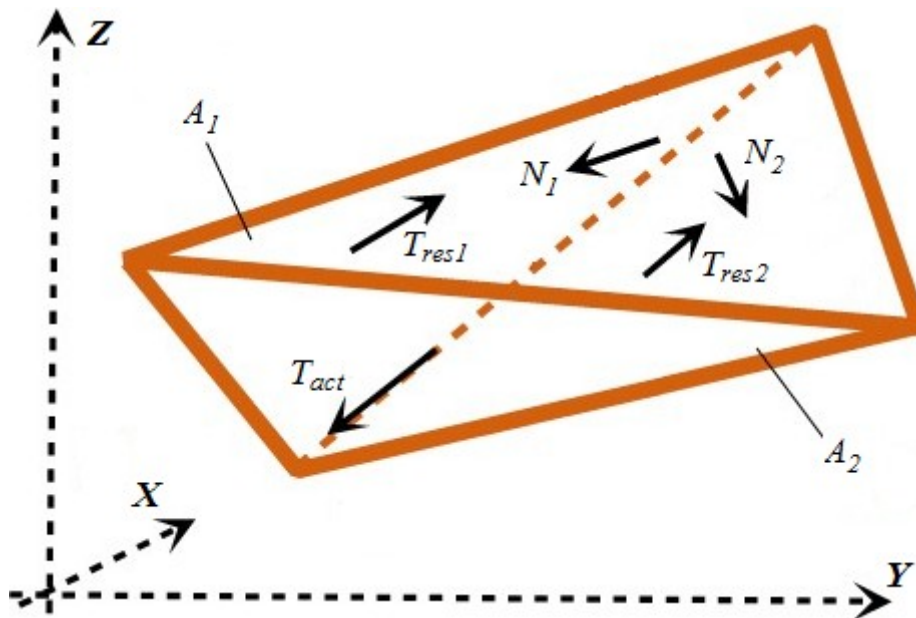
The solution requires determination of **the normal force**  $N$ , the **shear force**  $T_{act}$  (active) and the **resisting (passive) shear force**  $T_{res}$  acting on slip surfaces  $A_1$  and  $A_2$ . The active force  $T_{act}$  and the normal force  $N$  are obtained as a summation all forces entering the analysis after performing the space **resolution of these forces**.

The **Mohr-Coulomb** shear strength parameters and the **normal force**  $N$  acting on the slip surface are the main input data for the determination of the **resisting shear forces**  $T_{res}$ . Calculation of the **active shear force**  $T_{act}$  and the **normal force**  $N$  is further influenced by the



weight of wedge (depends on the **geometry of wedge** and **unit weight of rock**), anchorage of wedge, surcharge acting on the wedge, **influence of water** and **seismic effects**.

The resulting verification can be carried out according to the selected **verification methodology** based on the input in the "Settings" frame.

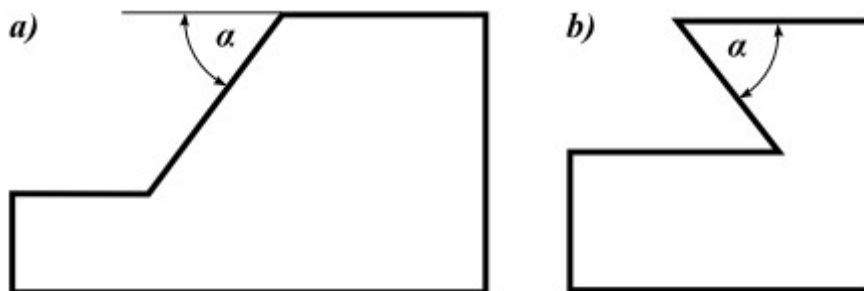


Components acting on a rock wedge

## Geometry of Rock Wedge

Entering geometry of a rock wedge using either gradient or falling line gradient direction requires definition of space orientation of the rock face, terrain (top face), slip surfaces  $N_1$  and  $N_2$  and/or tension crack, such that:

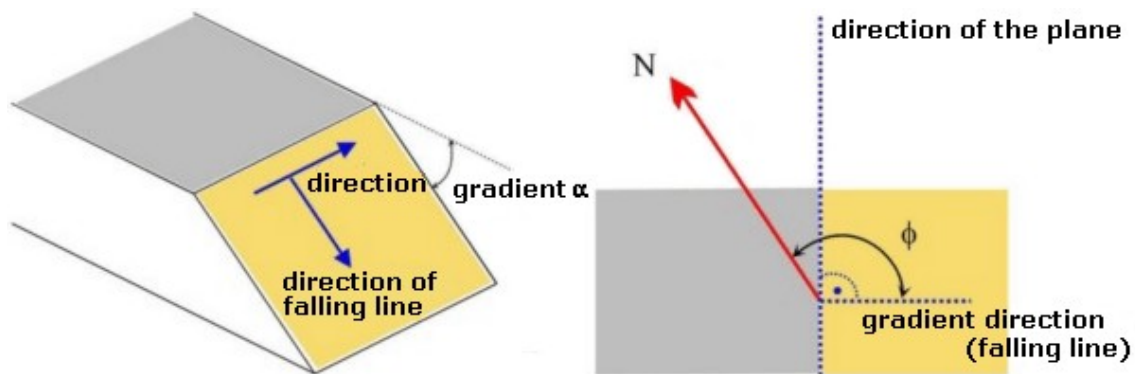
- Gradient of surface** (gradient angle) is an inclination angle  $\alpha$  representing inclination of surface from horizontal (it may receive values from  $0^\circ$  to  $90^\circ$ ). In the case of overhanging slope (the edge of slope is before the slope toe - wall tends from edge to the rock mass), then it must be checked button "**Overhang rock face**" and gradient of the face  $\varphi$  is considered in the half plane of the rock mass. Program checks the possibility of overturning failure of the rock mass with the overhang rock face. If the option of an earth wedge overturning is realistic, program notifies to the user in listing of the results. However, program does not make a real evaluation of the overturning or rotation of the rock wedge.



a) Pendulous rock face b) Overhang rock face

- **Gradient direction** (falling line) is an angle  $\beta$  between horizontal projection of the line normal to the strike direction measured as an azimuth angle from the north in the clockwise direction (the falling line corresponds to inclination of the plane), it may receive values from  $0^\circ$  to  $360^\circ$ .

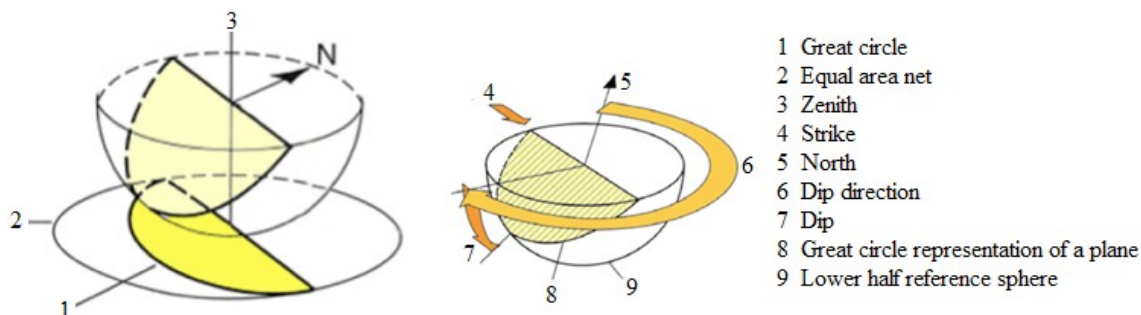
The program when defining space orientation of planes displays these planes using a **stereographic projection**.



*Description of orientation of surfaces - using gradient and direction*

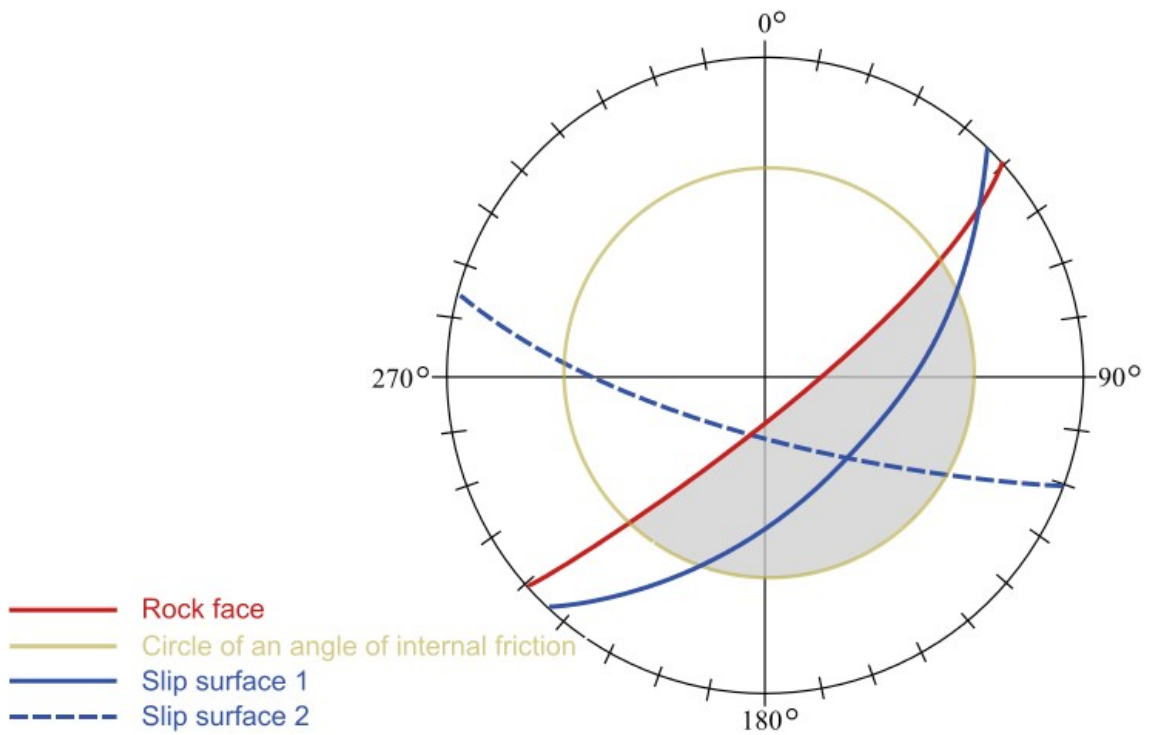
## Stereographic Projection

When defining geometry of the wedge and slip surfaces using space projection, the program displays individual surfaces with the help of great circles (in equal area net) of Lambert's hemispherical projection.



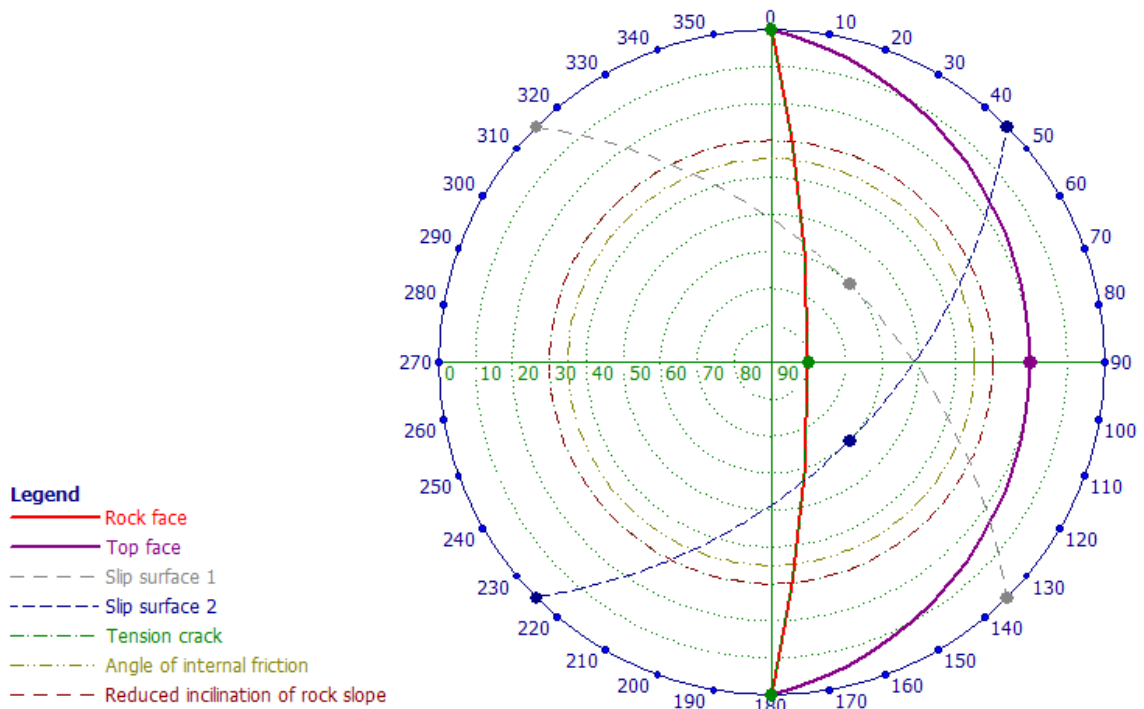
*Principle of Stereographic projection (Lambert's projection)*

Markland test utilizes a stereographic projection of the great circle for the kinematic possibility of the rock wedge failure. The geometry of the **rock wedge** is plotted on the stereographic projection using the great circle representing the slope face and discontinuities of slip surface 1 and 2 (see figure). The friction circle is plotted on the projection too, the friction angle on the discontinuities is measured from the north and the circle center is the center of the stereograph. The zone between the great circle representing the slope face and the friction circle is shaded area on the figure. When the plunge of the line of intersection of two discontinuities (1 and 2) is in the shaded area, then the failure is cinematically possible.



*Kinematic possibility of the rock wedge (Markland's test plot in a stereographic projection)*

View of the geometry of the rock wedge is supplement by elements of Markland's test plot. This makes it possible to assess the stereographic projection of the kinematics of the rock wedge.



*Markland's test plot*

## Influence of Ground Water

By default the program performs the stability analysis of a rock wedge without considering ground water. If interested on the influence of ground water on a rock wedge it is necessary to introduce the height of GWT from the line of intersection of slip surfaces and rock face (the GWT takes an arbitrary position over the entire height of a rock wedge). The program assumes that water can flow freely discontinuities located below the GWT (no restrictions, e.g. due to ice blocks, are considered).

The water pressure acts in the direction normal to the slip surfaces against normal components of the passive forces. If the height  $y_w$  above the point of maximal pressure  $P_{max}$  is equal or larger than  $Z/2$  and it is fully contained by the rock wedge, then its value is assumed to be equal to  $Z/2$  (case A). If the height  $y_w$  above the point of maximal pressure  $P_{max}$  is less than  $Z/2$  (case B), then its value reduced as:

$$y_w = \left( \frac{1}{2} \cdot L^* \cdot \sin \delta \right) \left( \frac{\text{tg} \alpha_1}{\text{tg} \delta} - 1 \right)$$

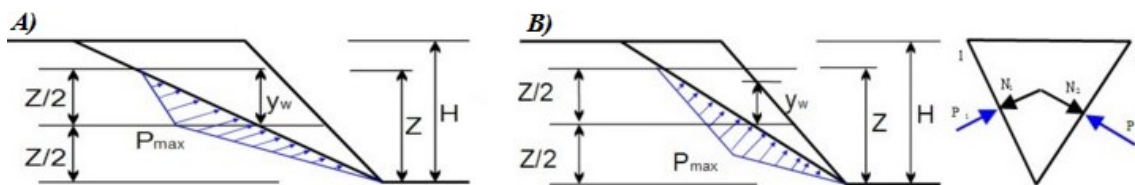
where:  $L^*$  - length of the line of intersection of slip surfaces  $A_1, A_2$   
 $\alpha_1$  - gradient of rock face  
 $\delta$  - gradient of the of line of intersection of slip surfaces

The resulting water pressure on slip surfaces 1 and 2 is given by:

$$P_1 = \frac{1}{3} \cdot P_{max} \cdot A_1^w = \frac{1}{3} \cdot \gamma_w \cdot y_w \cdot A_1^w$$

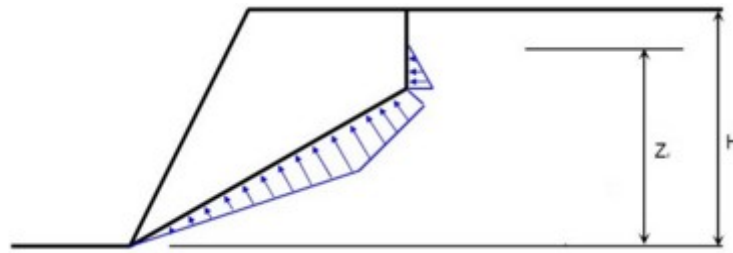
$$P_2 = \frac{1}{3} \cdot P_{max} \cdot A_2^w = \frac{1}{3} \cdot \gamma_w \cdot y_w \cdot A_2^w$$

where:  $Z$  - height of GWT above the line of intersection of slip surfaces and rock face  
 $P_{ma}$  - maximal water pressure on the line of intersection of slip surfaces  
 $x$   
 $\gamma_w$  - unit weight of water ( $\approx 10 \text{ kN/m}^3$ )  
 $A_1^w$  - are of the wetted part of the slip surface 1  
 $A_2^w$  - area of the wetted part of the slip surface 2



*Distribution of water pressure on the line of intersection of slip surfaces*

If a tension crack is found either entirely or partially below the GWT, then the influence of water pressure is reflected both on slip surfaces 1 and 2 through forces  $P_1$  and  $P_2$  acting on intersection of these surfaces and on tension crack through force  $P_3$  acting in the direction normal to the tension crack.



*Distribution of water pressure when considering GWT in tension crack*

## Resolution of Acting Forces

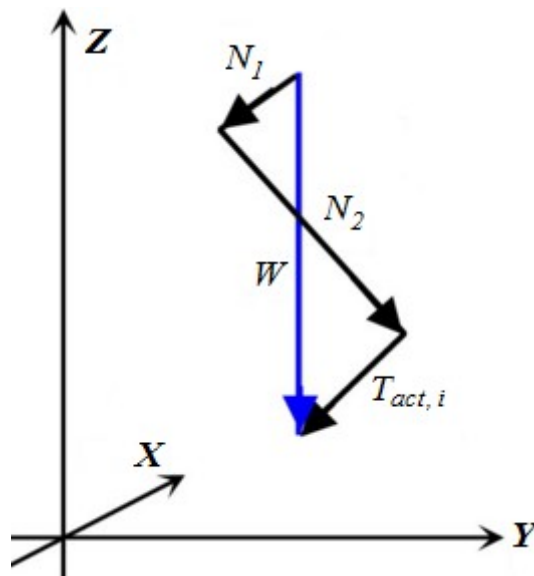
Forces acting on a rock wedge (weight of rock wedge, external load, anchor force) are resolved into directions normal to planes  $A_1$  and  $A_2$  (the block is wedged in between these surfaces) and into the direction of their intersection. The resolution of forces results into the normal forces  $N_1$ ,  $N_2$  acting on planes  $A_1$  and  $A_2$ , resisting (passive) forces  $T_{res1}$ ,  $T_{res2}$  acting along planes  $A_1$  and  $A_2$ .

This step further generates the **shear (active) force**  $T_{act}$  acting in the direction of the line of intersection of slip surfaces. The resulting **shear (active) force**  $T_{act}$  is obtained as a sum of individual shear forces  $T_{act,i}$ .

**The resisting (passive) force**  $T_{res}$  is found by summing up the components  $T_{res1}$ ,  $T_{res2}$  (e.g. due to external load) and friction forces on planes  $A_1$  and  $A_2$  due to normal forces:

$$T_{res} = \sum T_{res,1} + \sum T_{res,2} + \sum (N_1 \operatorname{tg} \varphi_1 + c_1 A_1) + \sum (N_2 \operatorname{tg} \varphi_2 + c_2 A_2)$$

where:	$c_1$	cohesion on slip surface $A_1$
	$c_2$	cohesion on slip surface $A_2$
	$\varphi_1$	angle of internal friction on slip surface $A_1$
	$\varphi_2$	angle of internal friction on slip surface $A_2$
	$T_{res1}$	resisting forces on slip surface $A_1$
	$T_{res2}$	resisting forces on slip surface $A_2$



Space resolution of self weight of earth wedge  $W$

## Verification

Verification can be carried out either according to the selected **verification methodology** based on the input in the "Settings" frame.

### Verification According to the Factor of Safety

When performing verification according to the factor of safety the program directly determines the value of the factor of safety  $SF$ . Verification condition has the form:

$$SF = \frac{T_{res}}{T_{act}} > SF_s$$

- where:
- $T_{act}$  -shear forces along the slip surface
  - $T_{res}$  -passive forces along the slip surface
  - $SF$  -safety factor
  - $SF_s$  -required safety factor (input in the "Stability analysis" tab)

Typical values are e.g. - for walls of foundation pits  $SF_s = 1.1$  to  $1.25$ , - for rock cuts of highways  $SF_s = 1.2$  to  $1.5$ , etc.

### Verification According to the Theory of Limit States

When performing verification according to the theory of limit states the program reduces material parameters of rocks (angle of internal friction or tangent of the angle of internal friction, cohesion) using partial coefficients entered in the "Stability analysis" tab. Verification condition has the form:

$$T_{act} < \frac{T_{res}}{\gamma_s}$$

where:  $T_{act}$  - shear forces along the slip surface (active)  
 $T_{res}$  - passive forces along the slip surface  
 $\gamma_s$  - reduction coefficient of overall stability of construction (input in the "Stability analysis")

When analyzing the **polygonal slip surface** the program compares the calculated value with the value corresponding to the fully stressed design (state of equilibrium with zero reserve). Verification condition has the form:

$$SF > SF_s$$

where:  $SF$  - factor of safety calculated with the reduced material parameters  
 $SF_s$  - coefficient of the overall stability of the structure

## Rock - Shear Resistance Criteria

The shear strength is the basic criterion to determine resisting passive forces. The resisting force is given by the following expression:

$$T_{res} = \tau l$$

where:  $\tau$  - shear strength on the slip surface  
 $l$  - length of the slip surface

The shear strength for the planar slip surface can be written as:

- Mohr - Coulomb
- Hoek - Brown
- Barton - Bandis

## Mohr - Coulomb

The shear strength  $\tau$  [kPa] according to Mohr-Coulomb is given by equation:

$$\tau = c + \frac{N}{l} \operatorname{tg} \varphi$$

where:  $N$  - normal force acting on the slip surface  
 $l$  - length of the slip surface  
 $c$  - cohesion of rock on the slip surface  
 $\varphi$  - angle of internal friction of rock on the slip surface

Approximate ranges of parameters of the Mohr-Coulomb failure criterion for selected soils are given [here](#).

## Mohr - Coulomb Parameters

If possible the strength parameters should be determined in-situ measurements. The results of in-situ and laboratory experiments show that the angle of internal friction is found for majority of discontinuities in the rock mass in the range of 27° to 47°. Approximate values of the angle

of internal friction  $\varphi$  and cohesion  $c$  for rocks based on the RMR classification are stored in the following table:

Rock class	I	II	III	IV	V
RMR	100 - 81	80 - 61	60 - 41	40 - 21	< 20
Angle of internal friction $\varphi$ [°]	> 45	35 - 45	25 - 45	15 - 25	< 15
Cohesion $c$ [kPa]	> 400	300- 400	200 - 300	100 - 200	< 100

## Hoek - Brown

Hoek-Brown failure criterion describes the failure of a rock mass (based on the performed analyses of hundreds of underground structures and rock slopes) as:

$$\sigma_{1,ef} = \sigma_{3,ef} + \sigma_c \left( \frac{m_b \cdot \sigma_{3,ef}}{\sigma_c} + s \right)^a$$

- where:
- $\sigma_{1ef}$  - major principal stress during rock failure
  - $\sigma_{3ef}$  - minor principal stress during rock failure
  - $\sigma_c$  - strength of the intact rock in simple compression
  - $\sigma_{ci}$  - uniaxial compressive strength of intact pieces of rock
  - $m_{b,s}$  - nonlinear material constant depending on the rock quality
  - $a$  - coefficient depending on the rock breaking

Basic parameters of the modified Hoek-Brown model should be determined from in-situ and laboratory measurements. To become more acquainted with this model, a brief list of [ranges of individual parameters](#) is provided. If **rock mass classification using GSI** is known then it is possible to let the program to determine the H-B parameters [by itself](#).

For actual analysis the H-B parameters are transformed into the M-C parameters. The solution procedure then becomes identical to that of the [Mohr-Coulomb](#) criterion.

For conversion of Hoek-Brown parameters is used solution according to Hoek, Carranza-Torres and Corkum (2002) in case of the analysis of [rock slope stability](#):

Angle of internal friction  $\varphi$ :

$$\varphi' = \arcsin \left[ \frac{6 a m_b (s + m_b \sigma'_{3n})^{a-1}}{2(1+a)(2+a) + 6 a m_b (s + m_b \sigma'_{3n})^{a-1}} \right]$$

Cohesion (shear strength)  $c$ :

$$c' = \frac{\sigma_{ci} [(1+2a)s + (1-a)m_b \sigma'_{3n}] (s + m_b \sigma'_{3n})^{a-1}}{(1+a)(2+a) \sqrt{1 + (6 a m_b (s + m_b \sigma'_{3n})^{a-1}) / ((1+a)(2+a))}}$$

where:



$$\sigma'_{3n} = \frac{\sigma'_{3max}}{\sigma_{ci}}$$

The maximum value of a smaller principal stress  $\sigma'_{3max}$  is given by:

$$\frac{\sigma'_{3max}}{\sigma'_c} = 0,72 \left( \frac{\sigma'_c}{\gamma H} \right)^{-0,91}$$

where:

- $\gamma$  - unit weight of rock
- $H$  - height of rock slope
- $\sigma_c$  - strength of the intact rock in simple compression, or uniaxial compressive strength of intact rock mass

*Literature:*

*Stability analysis of rock slopes with a modified Hoek-Brown failure criterion, ANG Xiao-Li ; LIANG LI ; YIN Jian-Hua International journal for numerical and analytical methods in geomechanics. ISSN 0363-9061, 2004, vol. 28, no2, pp. 181-190.*

*Hoek E, Carranza-Torres CT, Corkum B.: Hoek-Brown failure criterion—2002 edition. Proceedings of the 5th North American Rock Mechanics Symposium, Toronto, Canada, vol. 1, 2002, pp. 267 - 273.*

## Parameters Hoek - Brown

### Parameter of rock breaking $a$

Parameter  $a$  is an exponent receiving values from 0.5 to 0.65 (for the original Hoek-Brown condition it is equal to 0.5) and depends on the degree of rock breaking.

**Nonlinear parameters**  $m_b = m_r$ ,  $s$  for  $a = 0.5$

(index  $r$  denotes residual values)

	<b>Carbonate rocks with well developed cleavage - dolomite, limestone, marble</b>	<b>Argillaceous rocks - mudstone, siltstone, shale, slate</b>	<b>Arenaceous rocks - sandstone, quartzite</b>	<b>Fine grained igneous crystalline rocks - andesite, dolerite, basalt, rhyolite</b>	<b>Coarse metamorphic and igneous rocks - gabbro, gneiss, granite</b>
Intact rock material, Laboratory specimens have no discontinuities, $RMR = 100$ , $Q = 500$	$m = 7.00$ $s = 1.00$ $m_r = 7.00$ $s_r = 1.00$	$m = 10.00$ $s = 1.00$ $m_r = 10.00$ $s_r = 1.00$	$m = 15.00$ $s = 1.00$ $m_r = 15.00$ $s = 1.00$	$m = 17.00$ $s = 1.00$ $m_r = 17.00$ $s = 1.00$	$m = 25.00$ $s = 1.00$ $m_r = 25.00$ $s = 1.00$
Very good quality rock mass, Rocks without isolated blocks with non-weathered discontinuities, $RMR = 85$ , $Q = 100$	$m = 2.40$ $s = 0.082$ $m_r = 4.10$ $s_r = 0.189$	$m = 3.43$ $s = 0.082$ $m_r = 5.85$ $s_r = 0.189$	$m = 5.14$ $s = 0.082$ $m_r = 8.78$ $s_r = 0.189$	$m = 5.82$ $s = 0.082$ $m_r = 9.95$ $s_r = 0.189$	$m = 8.56$ $s = 0.082$ $m_r = 14.63$ $s_r = 0.189$
Good quality rock mass, Slightly damaged rocks with non-weathered discontinuities spaced from 1 to 3 $m$ , $RMR = 65$ , $Q = 10$	$m = 0.575$ $s = 0.00293$ $m_r = 2.006$ $s_r = 0.0205$	$m = 0.821$ $s = 0.00293$ $m_r = 2.865$ $s_r = 0.0205$	$m = 1.231$ $s = 0.00293$ $m_r = 4.298$ $s_r = 0.0205$	$m = 1.395$ $s = 0.00293$ $m_r = 4.871$ $s_r = 0.0205$	$m = 2.052$ $s = 0.00293$ $m_r = 7.163$ $s_r = 0.0205$
Fair quality rock mass, Partially weathered discontinuities spaced from 0.3 to 1 $m$ ,	$m = 0.128$ $s = 0.00009$ $m_r = 0.947$	$m = 0.183$ $s = 0.00009$ $m_r = 1.353$	$m = 0.275$ $s = 0.00009$ $m_r = 2.030$	$m = 0.311$ $s = 0.00009$ $m_r = 2.301$	$m = 0.458$ $s = 0.00009$ $m_r = 3.383$

<i>RMR</i> = 44, <i>Q</i> = 1	$s_r = 0.00198$	$s_r = 0.00198$	$s_r = 0.00198$	$s_r = 0.00198$	$s_r = 0.00198$
Poor quality rock mass, Numerous weathered discontinuities spaced from 30 to 500 mm, <i>RMR</i> = 23, <i>Q</i> = 0.1	$m = 0.029$ $s = 0.000003$ $m_r = 0.447$ $s_r = 0.00019$	$m = 0.041$ $s = 0.000003$ $m_r = 0.639$ $s_r = 0.00019$	$m = 0.061$ $s = 0.000003$ $m_r = 0.959$ $s_r = 0.00019$	$m = 0.069$ $s = 0.000003$ $m_r = 1.087$ $s_r = 0.00019$	$m = 0.102$ $s = 0.000003$ $m_r = 1.598$ $s_r = 0.00019$
Very poor quality rock mass, Numerous extremely weathered discontinuities with filling spaced by less than 50 mm, fine grained waste rock, <i>RMR</i> = 3, <i>Q</i> = 0.01	$m = 0.007$ $s = 0.0000001$ $m_r = 0.219$ $s_r = 0.00002$	$m = 0.010$ $s = 0.0000001$ $m_r = 0.313$ $s_r = 0.00002$	$m = 0.015$ $s = 0.0000001$ $m_r = 0.469$ $s_r = 0.00002$	$m = 0.017$ $s = 0.0000001$ $m_r = 0.532$ $s_r = 0.00002$	$m = 0.025$ $s = 0.0000001$ $m_r = 0.782$ $s_r = 0.00002$

**Strength of rocks in simple compression  $\sigma_c$ , Poisson's ratio  $\nu$  and unit weight of rock  $\gamma$**

Rock strength	Types of rock (examples)	Strength $\sigma_c$ [MPa]	Poisson's ratio $\nu$	Bulk weight of rock $\gamma$ [kN/m <sup>3</sup> ]
Solid rock	most hard solid rock, intact, compact and dense quartz rock and basalt, other extraordinary hard rocks	>150	0.10	28.00 - 30.00
Highly hard rock	very hard granit rock, quartz porphyry, very hard granite, hard flinty shale, quartzite, very hard sand rock and very hard calcite	100 - 150	0.15	26.00 - 27.00
Hard rock	granite, very hard sandstone and calcite, quartzite lode, hard conglomerate, very hard ore, hard limestone, marble, dolomite, pyrite	80 - 100	0.20	25.00 - 26.00
Rock	sandstone, ore, medium sandy shale, flagstone	50 - 80	0.25	24.00
Medium rock	hard mudstone, softer sand rock and calcite, chalky clay	20 - 50	0.25 - 0.30	23.00 - 24.00
Soft rock	shale, soft limestone, calk, salt rock, frozen ground, anthracite, marl, remoulded sandstone, soft conglomerate, ground with fels	5 - 20	0.3 - 0.35	22.00 - 26.00
Weak soil	compact clay, soil eluvium, black coal	0.5 - 5	0.35 - 0.40	20.00 - 22.0  18.00 - 2

## Calculation of Hoek-Brown Parameters

If rock mass is described using GSI (Geological Structure Index) is known then it is possible to let the program to determine the H-B parameters as follows:

$$m_b = m_i \cdot e^{(GSI-100 / 28 - 14.D)}$$

$$s = e^{(GSI-100 / 9 - 3.D)}$$

$$a = \frac{1}{2} + \frac{1}{6} \left( e^{(-GSI / 15)} - e^{(-20 / 3)} \right)$$

where:  $GSI$  - Geological Structure Index

$D$  - damage coefficient of rock mass

$m_i$  - strength material constant of the intact rock for peak conditions

#### Values of damage coefficient $D$ for rock slope

Description of rock mass	Suggested value of coefficient $D$
Small scale blasting in engineering slopes results in modest rock mass damage, particularly if controlled blasting is used. However, stress relief results in some disturbance. (Good blasting).	0.7
Small scale blasting in engineering slopes results in modest rock mass damage, particularly if controlled blasting is used. However, stress relief results in some disturbance. (Poor blasting).	1
Very large open pit mine slopes significant disturbance due to heavy production blasting and due to stress relief from overburden removal. (Production blasting).	1
In some softer rocks excavation can be carried out by ripping and dozing and the degree of damage to the slope is less. (Mechanical excavation).	0.7

#### Approximate values of strength material constant of the intact rock $m_i$ (after Hoek)

Type of rock	Representative rocks	$m_i$ [-]
Limestone rocks with well developed crystalline cleavage	Dolomite, calcite, marble	$\approx 7$
Consolidated clayey rocks	Mudstone, siltstone, silty shale, slate	$\approx 10$
Sandy rocks with solid crystals and poorly developed crystalline cleavage	Sandstone, quartzite	$\approx 15$
Fine grained igneous crystalline rocks	Andesite, dolerite, diabase, rhyolite	$\approx 17$
Coarse grained and metamorphic rocks	Amphibolite, gabbro, gneiss, granite, diorite	$\approx 25$

## Barton - Bandis

The Barton-Bandis shear strength failure criterion for the rock mass takes the following form:

$$\tau = \sigma_n \cdot \text{tg} \left[ \varphi_b + JRC \cdot \log_{10} \left( \frac{JCS}{\sigma_n} \right) \right]$$

where:

- $JRC$  - joint roughness coefficient
- $\sigma_n$  - normal stress acting on the surface of the rock joint
- $JCS$  - joint compressive strength
- $\varphi_b$  - basic angle of internal friction of the slip surface

If possible the shear strength parameters should be determined from in-situ measurements. Approximate ranges of parameters of the Barton-Bandis failure criterion are given [here](#).

## Barton - Bandis Parameters

### Joint roughness coefficient JRC

If the value of JRC cannot be determined by direct measurements on the joint surface, it is possible to obtain this value from the Barton graph (see figure) showing the variation of the coefficient JRC as a function of length of profile and roughness depth.

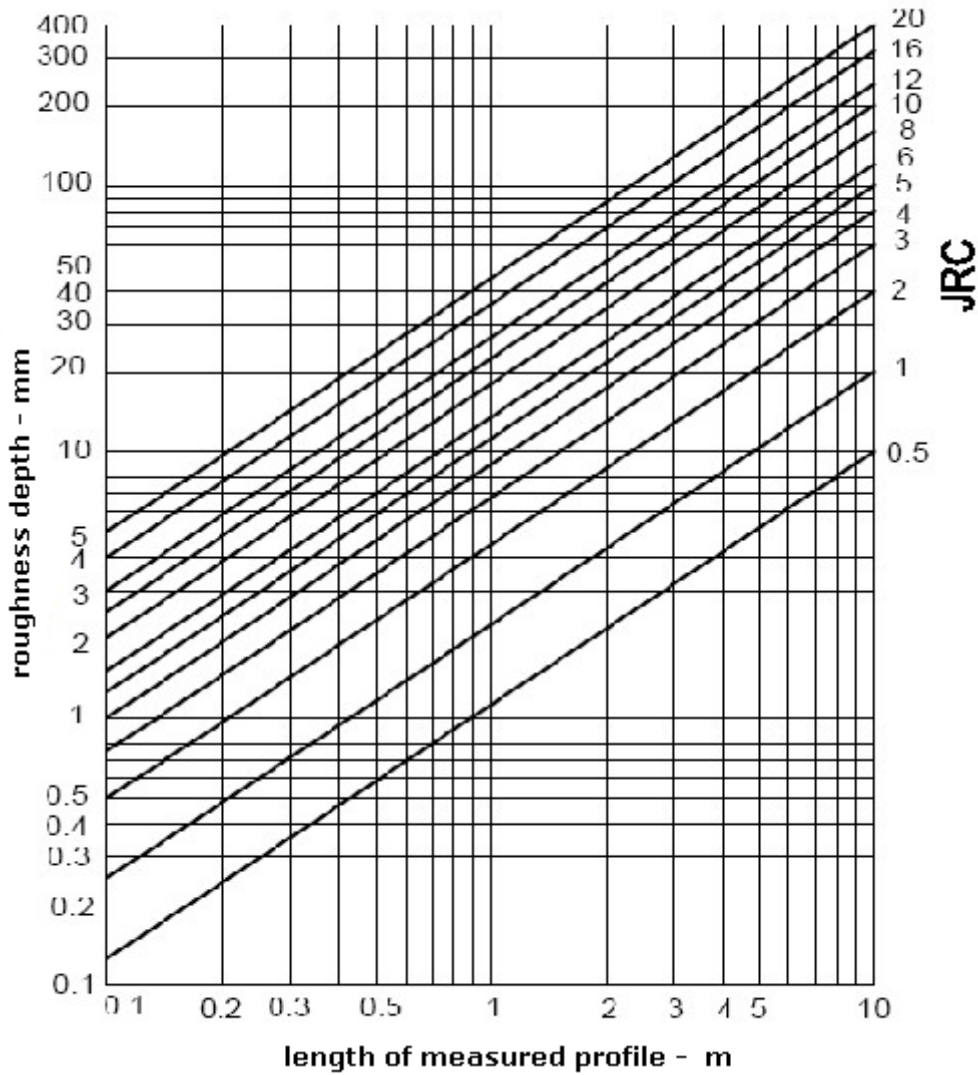
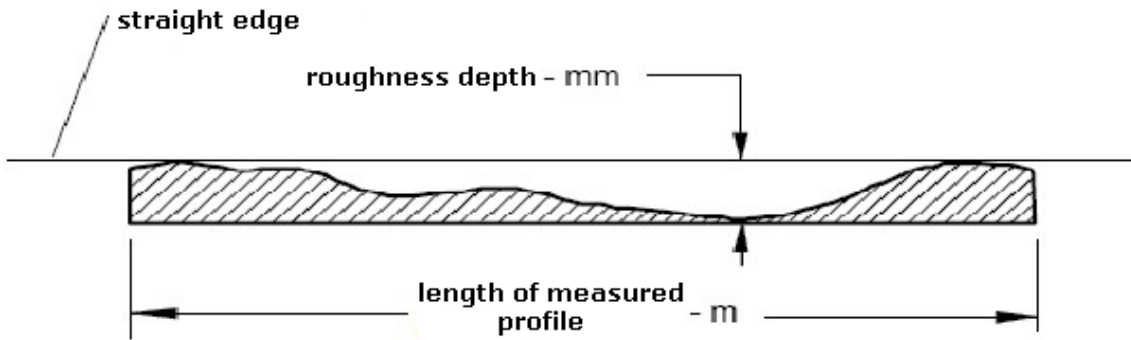
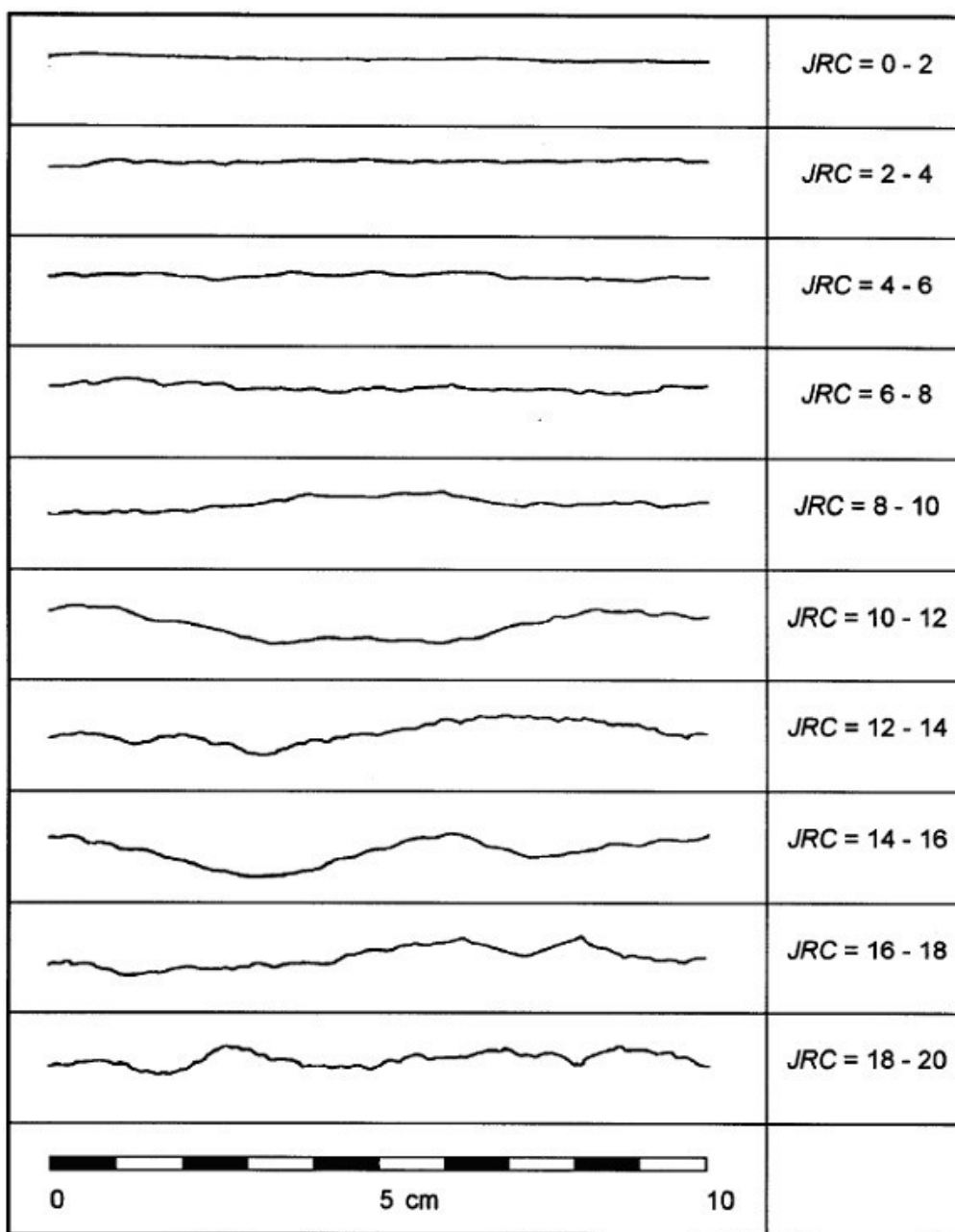


Diagram to determine JRC (after Barton)

Rock joint roughness profiles showing the typical range of JRC are plotted next.

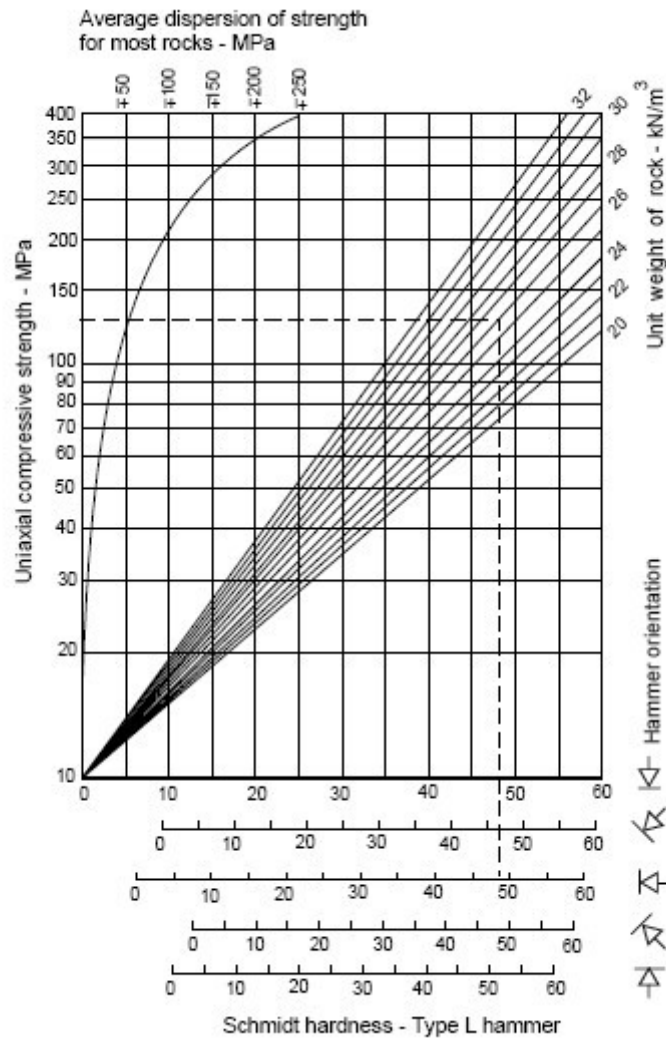


*Rock joint roughness profiles showing the typical range of JRC (Barton & Chubey 1977)*

### **Compressive strength of discontinuity JCS**

Methods allowing us to determine the compressive strength of discontinuity (slip surface) JCS are generally recommended by ISRM. The value of JCS can be obtained from the Deer-Miller graph showing its dependence on the rock strength found from the Schmidt hammer measurements, see figure below.





**Basic angle of internal friction on slip surface  $\phi_b$**

The basic value of the angle of internal friction on the surface is approximately equal to the residual value  $\phi_r$ . Nevertheless, it can be generally measured in laboratories using shear measurement devices (typical area of the specimen is 50\*50 mm). Typical ranges of the basic angle of internal friction for weathered rock surfaces are 25° to 35°.

**Unit Weight of Rocks**

**Unit weight or rock  $\gamma$**

<b>Rock strength</b>	<b>Rock category (examples)</b>	<b>Unit weight of rock <math>\gamma</math></b> [kN/m <sup>3</sup> ]
Solid rock	most hard solid rock, intact, compact and dense quartz rock and basalt, other extraordinary hard rocks	28.0 - 30.0
Highly hard rock	very hard granit rock, quartz porphyry, very hard granite, hard flinty shale, quartzite, very hard sand rock and very hard calcite	26.0 - 27.0
Hard rock	granite, very hard sandstone and calcite, quartzite lode, hard conglomerate, very hard ore, hard limestone, marble, dolomite, pyrite	25.0 - 26.0
Rock	sandstone, ore, medium sandy shale, flagstone	24.0
Medium rock	hard mudstone, softer sand rock and calcite, chalky clay	23.0 - 24.0
Soft rock	shale, soft limestone, calc, salt rock, frozen ground, anthracite, marl, remoulded sandstone, soft conglomerate, ground with fels	22.0 - 26.0
Weak soil	compact clay, soil eluvium, black coal	20.0 - 22.0  18.0 - 20.0

## Influence of Seismic Effects

The program takes into account the earthquake effects using two variables - factor of horizontal acceleration  $K_h$  and factor of vertical acceleration  $K_v$ .

The factor of acceleration is a dimensionless number, which represents the seismic acceleration as a fraction of the gravity acceleration. Earthquake effects are introduced through the seismic force  $S$ , which is determined by multiplying the weight of the rock subjected to earthquake (i.e. rock block) by the factor of acceleration. When assuming seismic effects only in the horizontal direction the seismic force is given by:

$$S = K_h \cdot W$$

where:  $K_h$  - factor of horizontal acceleration  
 $W$  - weight of the rock body

The seismic force always acts in the center of gravity of the rock body. Usually, only seismic effects in the horizontal direction are considered. Nevertheless, the program also allows for treating the vertical direction (with the help of vertical factor of acceleration  $K_v$ ). Effects in both directions are then combined.

<b>M_C_S grade</b>	<b>Horizontal acceleration</b>	<b>Factor of horizontal acceleration</b>
((MSK-64)	$[mm/s^2]$	$K_h$
1	0.0 - 2.5	0.0 - 0.00025
2	2.5 - 5.0	0.00025 - 0.0005
3	5.0 - 10.0	0.0005 - 0.001
4	10.0 - 25.0	0.001 - 0.0025
5	25.0 - 50.0	0.0025 - 0.005
6	50.0 - 100.0	0.005 - 0.01
7	100.0 - 250.0	0.01 - 0.025
8	250.0 - 500.0	0.025 - 0.05
9	500.0 - 1000.0	0.05 - 0.1
10	1000.0 - 2500.0	0.1 - 0.25
11	2500.0 - 5000.0	0.25 - 0.5
12	> 5000.0	> 0.5

**The values of factor  $K_h$  correspond to individual degrees of earthquake according to M-C-S scale**

## MSE Wall

The program allows the following verifications:

### Verification

The program verifies the external stability of so called **fictitious structure** consisting of the structure front face and a curve bounding the geo-reinforcements end points. The fictitious structure is loaded by **calculated forces acting on the structure** and checked for **overturning** and **slip** - similarly to the verification of a **gravity wall**.

### Dimensioning

From the **calculated forces acting on a structure**, the program determines the forces in the checker cross-section. Only the forces above the checked joint (see **figure**) are taken into account. Reinforcements introduce **stabilizing forces** which equal to the lower value of the two bearing capacities (against tearing and pull-out). The actual verification for **overturning** and **slip** follows afterwards. The program also allows for an automatic verification of the most critical cross-section.

### Bearing capacity

The **bearing capacity of foundation soil** below a **fictitious structure** is verified. The constant stress in the foundation joint is determined from all **forces acting on a structure** and calculated in the "**Verification**" frame. In case of the input foundation the bearing capacity is determined from all forces calculated in the "**Dimensioning**" frame (the option "**Entire wall**" must be selected).

### Slip on georeinforcement

The **slip** of a **reinforced soil block** along a geo-reinforcement is verified. The reinforced block is bounded the wall front face, the checked geo-reinforcement, a vertical line passing through the geo-reinforcement end point and terrain. The block is loaded by an **active earth pressure** and by stabilizing forces due to geo-reinforcements exceeding the boundary of the reinforced block and by **other forces**. The program also allows to automatically verificate **sliding** along individual reinforcements and find the most critical result.

### Internal stability

Individual geo-reinforcements are **analyzed** for **tearing** and **pull-out** from an earth body. The program also allows to automatically verify the most critical reinforcement.

### Global stability

The program allows to verify the overall **slope stability** along a **circular slip surface**. The slip surface can be automatically **optimized**, i.e. the program automatically selects the verification along the most critical surface. The actual **slope stability** analysis can be carried out with the help of two slice methods: **Spencer** (rigorous, more accurate method) and **Bishop** (more conservative, simpler, more easily finds the solution satisfying equilibrium conditions).

### Slope stability

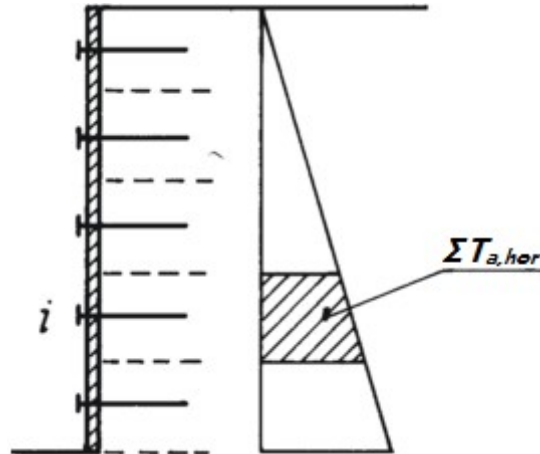
Verification of **overall (global) stability** by the "**Slope stability**" program.

## Internal Stability

### Geo-reinforcement force

Determination of forces in geo-reinforcements is performed by splitting and assigning the calculated earth pressure to individual layers. Each reinforcement accommodates part of the active pressure, which acts in the **corresponding layer**, i.e. force developed in the

reinforcement  $F_x = \Sigma T_{a,hor}$ . The calculation is always performed for flat terrain. Influence of broken terrain is considered as an additional **surcharge** (increase of **geostatic stress**  $\sigma_z$ ). **Earth pressure** is considered as an **active** for **extensible reinforcements** (Standard - straight slip surface, AASHTO - Extensible, FHWA NHI-10-024), or as a combination of pressures for **inextensible reinforcements** (AASHTO - Inextensible, JTGD30 - 2004 Highway China Code, TB 10025 Railway China Code, BS 8006 - Coherent Gravity Method). Slip surface has a **different shape** (straight, broken) according to the selected standard of calculation.



Forces transmitted by individual reinforcements

### Reinforcement strength check

Long-term design strength of geo-reinforcement  $R_t$  is calculated from the **input parameters** of the geo-reinforcement:

$$R_t = \frac{T_{ult}}{RF_{CR} \cdot RF_D \cdot RF_{ID} \cdot FS_{UNC}}$$

where:  $R_t$  - long-term design strength of reinforcement

$T_{ult}$  - short-term characteristic strength of geo-reinforcement

$RF_C$  - reduction coefficient of long-term deformation of reinforcement (determined based on geo-reinforcement lifetime)

$RF_D$  - reduction coefficient of durability of reinforcement (determined based on soil pH)

$RF_I$  - reduction coefficient of failure of reinforcement when inserting into the soil (determined based on the grain sizes of soil)

$FS_U$  - overall coefficient of model uncertainty

$NC$

### Reinforcement bearing capacity against pull-out

Strength of reinforcement against pull-out from the earth body is calculated from the **input parameters** of the geo-reinforcement and the normal force acting in the direction normal to its **area**:

$$T_p = 2 \cdot L \cdot C_i \cdot \sigma_z \cdot \tan \varphi$$

where:  $T_p$  - bearing capacity against tearing  
 $L$  - reinforcement length (from front face to its end)  
 $C_i$  - coefficient of interaction between soil and geo-reinforcement  
 $\sigma_z$  - vertical geostatic stress  
 $\varphi$  - angle of internal friction of soil

**Verification** of bearing capacity of reinforcement against pull-out can be carried out according to the **factor of safety** or the theory of **limit states**.

## Verification - Safety Factor

The main advantage of this verification is lucidity and uniqueness, since neither soil parameters nor acting forces are reduced.

### Check for tearing:

$$\frac{R_t}{F_x} > SF_{st}$$

where:  $F_x$  - **force** developed in reinforcement  
 $R_t$  - long-term design **strength** of reinforcement  
 $SF_{st}$  - safety factor for geo-reinforcement strength (input in the "Wall analysis" tab)

### Check for pull-out:

$$\frac{T_p}{F_x} > SF_{po}$$

where:  $F_x$  - **force** developed in reinforcement  
 $T_p$  - **bearing capacity** of reinforcement against pull-out  
 $SF_{po}$  - safety factor for pull out resistance of geo-reinforcement (input in the frame "Wall analysis" tab)

## Verification - Limit States

The soil parameters are **reduced** depending on the setting in the "Wall analysis" tab. The result is the reinforcement utilization compared with 100 %.

### Check for tearing:

$$\frac{F_x}{R_t} \cdot 100 < 100\%$$

where:  $F_x$  - **force** developed in reinforcement  
 $R_t$  - long-term design **strength** of reinforcement against tearing

### Check for pull-out:

$$\frac{F_x}{T_p} \cdot 100 < 100\%$$

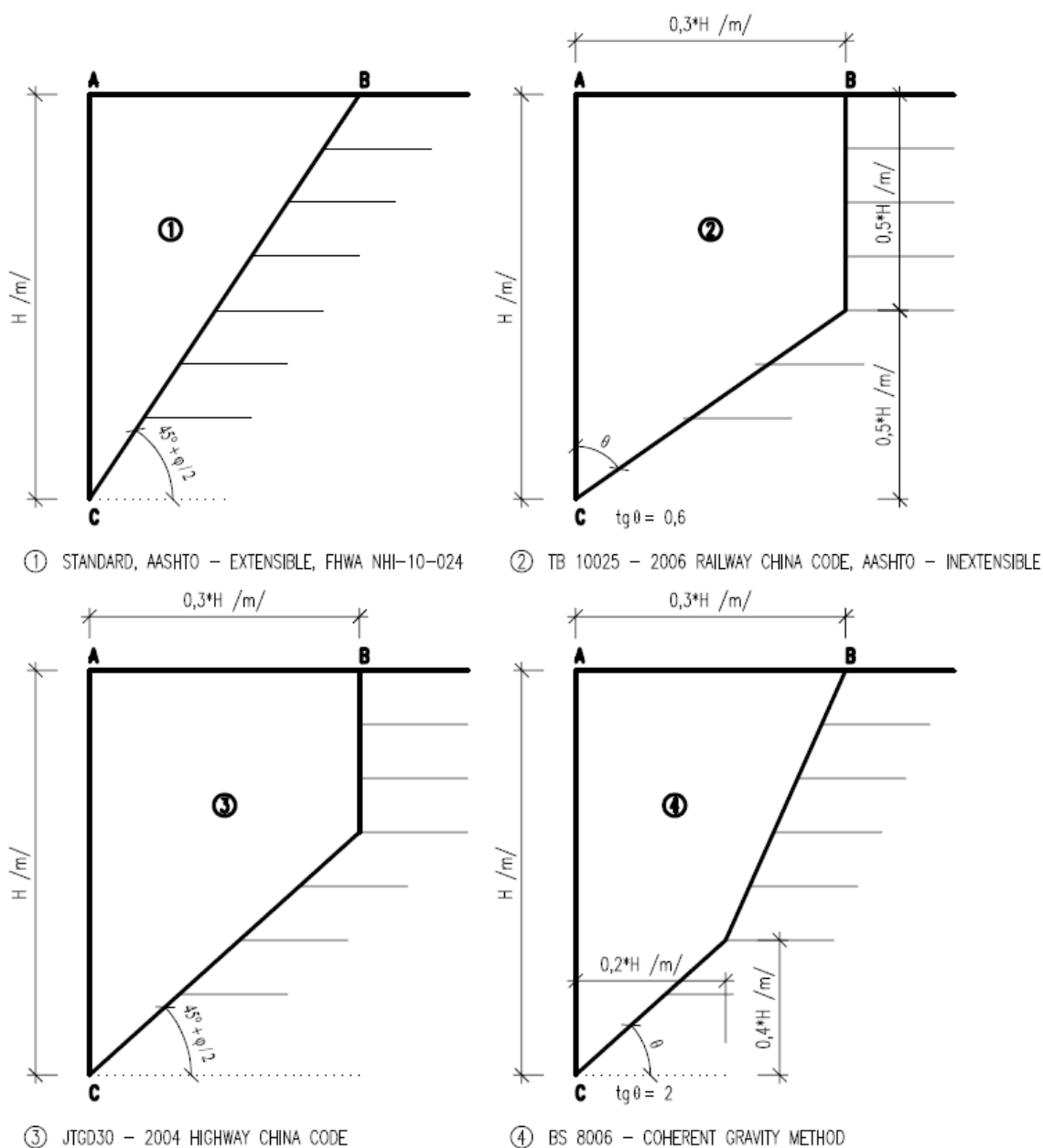
where:  $F_x$  - force developed in reinforcement

$T_p$  - bearing capacity of reinforcement against pull-out

## Shapes of Slip Surfaces

Slip surface has a different shape according to the selected standard of calculation:

- **Straight slip surface** - Standard, AASHTO - Extensible, FHWA NHI-10-024
- **Broken slip surface** - AASHTO - Inextensible, JTGD30 - 2004 Highway China Code, TB 10025 Railway China Code, BS 8006 - Coherent Gravity Method.

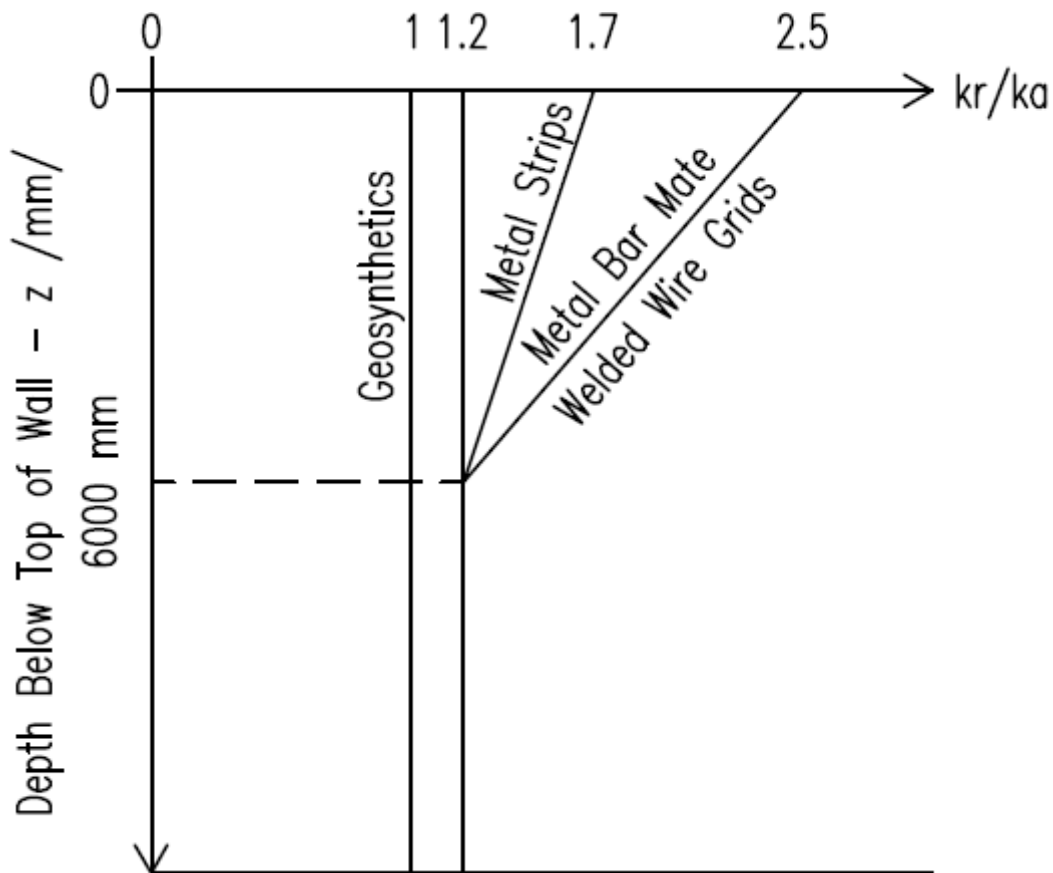


*Shapes of slip surfaces according to the individual standards of calculation*

## Extensible Reinforcements - Active Earth Pressure

For extensible reinforcements (Standard - **straight slip surface**, AASHTO - Extensible, FHWA NHI-10-024) is considered **active earth pressure** in the calculation of **internal stability**. The program allows to multiply the calculated **earth pressure** by coefficient  $k_r/k_a$  (according to the AASHTO standards). Recommended values are shown in the following picture.





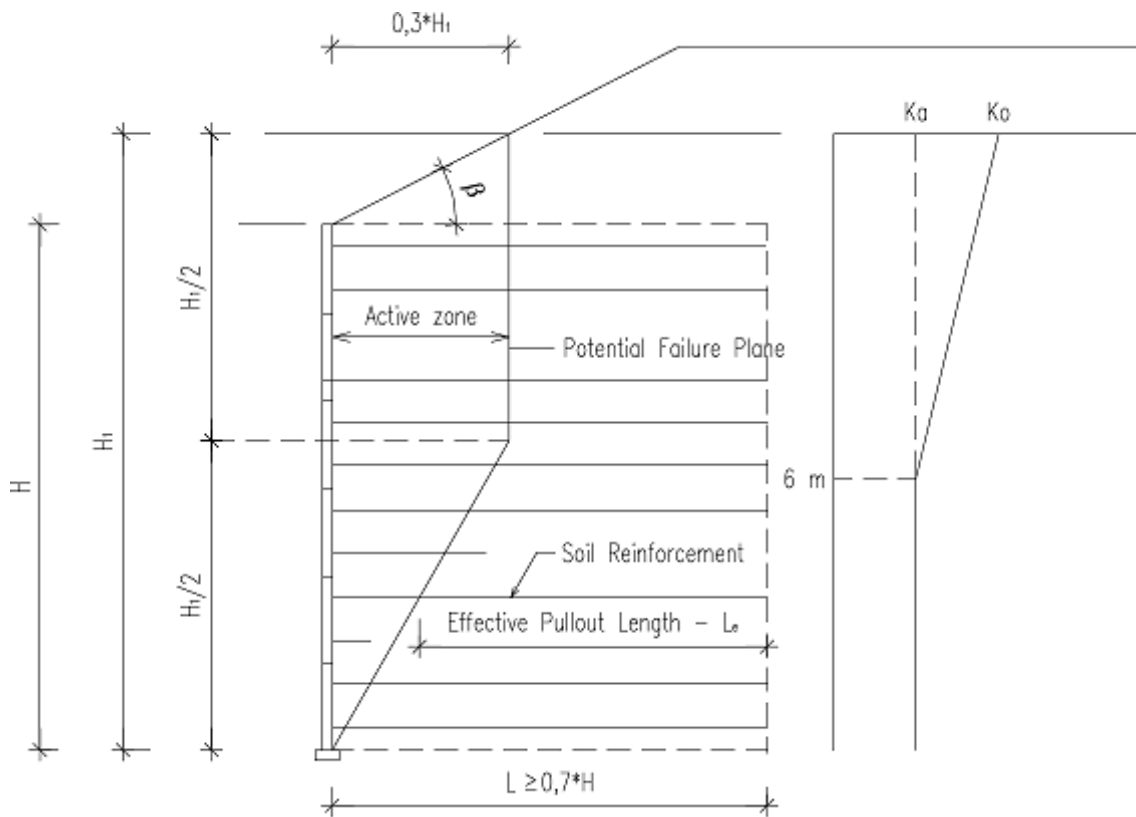
Variation of the coefficient of horizontal stress ratio  $k_r/k_a$  with depth for extensible reinforcements

Literature:

AASHTO LRFD Bridge Design Specifications 2004 (SI).

## Inextensible Reinforcements - Combination of Earth Pressures

For inextensible reinforcements (AASHTO - Inextensible, JTGD30 - 2004 Highway China Code, TB 10025 Railway China Code, BS 8006 - Coherent Gravity Method) is considered a combination of active earth pressure and earth pressure at rest in the calculation of internal stability.



Determination of failure plane location and variation of earth pressure coefficients  $K_a$ ,  $K_o$  with depth for inextensible reinforcements

Literature:

AASHTO Highway Bridges.

## Analysis of Foundation Bearing Capacity

The vertical bearing capacity of the foundation soil is verified according to the theory of limit states using the following inequality:

$$\sigma \leq \frac{R_d}{\gamma_{RV}}$$

or based on the factor of safety as:

$$\frac{R_d}{\sigma} \geq SF_v$$

- where:
- $\sigma$  - extreme design contact stress at the footing bottom
  - $R_d$  - design bearing capacity of foundation soil
  - $\gamma_{RV}$  - coefficient of vertical bearing capacity of foundation (for input use the "Spread Footing" tab)
  - $SF_v$  - safety factor for vertical bearing capacity

Extreme design contact stress at the footing bottom is assumed the form:

$$\sigma = \frac{V}{A_{ef}}$$

where:  $V$  - extreme design vertical force  
 $A_{ef}$  - **effective area** of foundation

The vertical bearing capacity of the foundation soil  $R_d$  is determined for three basic types of foundation conditions:

- **Drained subsoil**
- **Undrained subsoil**
- **Bedrock**

The above computations are applicable only for the homogeneous soil. If there is a **non-homogeneous** soil under the footing bottom (or there is ground water present), then the inserted profile is **transformed into a homogeneous one**.

## Bearing Capacity on Drained Subsoil

One of the following approaches is available to assess the bearing capacity of a foundation if drained conditions are assumed:

- **standard analysis**
- according to CSN 73 1001 "Základová půda pod plošnými základy" approved 8.6. 1987
- according to Polish standard PN-81 B - 03020 "Grunty budowlane, Posudowanie bezpořrednie budowli, Obliczenia statyczne i projektowanie" from year 1982
- according to Indian standard IS:6403-1981 "Code of Practice for Determination of Bearing Capacity of Shallow Foundations" from year 1981
- according to EC 7-1 (EN 1997-1:2003) "Design of geotechnical structures - Part 1: General rules"
- according to NCMA Segmental retaining walls manual, second edition
- according to Chinese standard GB 50007-2002
- according to Russian standard SNiP 2.02.01-83
- according to Danish standard DS/EN 1997-1 DK NA:2013

All approaches incorporate coefficients due to Brinch - Hansen (see **standard analysis**) to account for inclined ground surface and inclined footing bottom.

Assuming drained conditions during construction the soil below spread footing deforms including both shear and volumetric deformations. In such a case the strength of soil is assumed in terms of effective values of the angle of internal friction  $\varphi_{ef}$  and the effective cohesion  $c_{ef}$ . It is also assumed that there is an effective stress in the soil equal to the total stress (consolidated state). Effective parameters  $\varphi_{ef}$ ,  $c_{ef}$  represent the peak strength parameters.

Owing to the fact that the choice of drained conditions depends on a number of factors (rate of load, soil permeability, degree of saturations and degree of overconsolidation) it is the designer's responsibility to decide, depending on the actual problem being solved, if the effective parameters should be used.

## Standard Analysis

By default the solution proposed by J. Brinch - Hansen is used, where the bearing capacity of foundation soil follows from:

$$R_d = c \cdot N_e \cdot s_e \cdot d_e \cdot i_e \cdot b_e \cdot g_e + q_0 \cdot N_d \cdot s_d \cdot d_d \cdot i_d \cdot b_d \cdot g_d + \frac{b}{2} \cdot \gamma \cdot N_b \cdot s_b \cdot d_b \cdot i_b \cdot b_b \cdot g_b$$

where:

**coefficients of bearing capacity:**

$$q_0 = \gamma_1 \cdot d$$

$$N_e = (N_d - 1) \cdot \cot g \varphi \quad \text{for: } \varphi > 0$$

$$N_e = 2 + \pi \quad \text{for: } \varphi = 0$$

$$N_d = \text{tg}^2 \left( 45 + \frac{\varphi}{2} \right) \cdot e^{\pi \cdot \text{tg} \varphi}$$

$$N_b = 1,5(N_d - 1) \cdot \text{tg} \varphi$$

$$s_e = 1 + 0,2 \cdot \frac{b}{l}$$

$$s_d = 1 + \frac{b}{l} \cdot \sin \varphi$$

$$s_b = 1 - 0,3 \cdot \frac{b}{l}$$

**coefficients of influence of depth of foundation:**

$$d_e = 1 + 0,1 \cdot \sqrt{\frac{d}{b}}$$

$$d_d = 1 + 0,1 \cdot \sqrt{\frac{d}{b} \cdot \sin 2\varphi}$$

$$d_b = 1$$

$$i_e = i_d = i_b = (1 - \text{tg} \delta)^2$$

**coefficients of slope of footing bottom:**

$$b_e = b_d - \frac{(1 - b_d)}{N_e} \cdot \tan \varphi$$

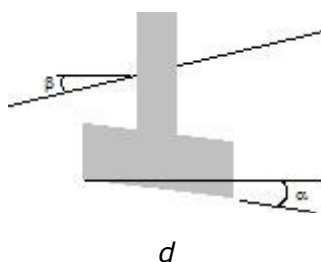
$$b_d = (1 - \alpha \cdot \tan \varphi)^2$$

$$b_b = b_d$$

**coefficients of influence of slope of terrain:**

$$g_e = 1 - \frac{2 \cdot \beta}{\pi + 2}$$

$$g_d = g_b = (1 - 0,5 \cdot \text{tg} \beta)^5$$



### Notation of angles and coefficients $b, g$

where:	$c$	-	cohesion of soil
	$q_0$	-	equivalent uniform load accounting for the influence of foundation depth
	$d$	-	depth of footing bottom
	$\gamma$	-	unit weight of soil above the footing bottom
	$b$	-	width of foundation
	$\gamma$	-	unit weight of soil
	$N_c, N_d, N_b$	-	coefficient of bearing capacity
	$s_c, s_d, s_b$	-	coefficients of shape of foundation
	$d_c, d_d, d_b$	-	coefficients of influence of foundation depth
	$i_c, i_d, i_b$	-	coefficients of influence of slope of load
	$g_c, g_d, g_b$	-	coefficients of influence of slope of terrain
	$\varphi$	-	angle of internal friction of soil
	$l$	-	length of foundation
	$\delta$	-	angle of deviation of the resultant force from the vertical direction
	$\beta$	-	slope of terrain
	$\alpha$	-	slope of footing bottom

#### Literature:

Brinch Hansen, J. (1970), *A revised and extended formula for bearing capacity*, Danish Geotechnical Institute, Bulletin 28, 5-11.

## Bearing Capacity on Undrained Subsoil

One of the following approaches is available to assess the bearing capacity of a foundation if undrained conditions are assumed:

- **standard analysis**
- according to CSN 73 1001 "Základová půda pod plošnými základy" approved 8.6. 1987
- according to Indian standard IS:6403-1981 "Code of Practice for Determination of Bearing Capacity of Shallow Foundations" from year 1981
- according to EC 7-1 (EN 1997-1:2003) "Design of geotechnical structures - Part 1: General rules"
- according to Danish standard DS/EN 1997-1 DK NA:2013

In addition the coefficients due to Brinch - Hansen are used to account for inclined footing bottom (see [standard analysis](#)).

In case of undrained conditions it is assumed that during construction the spread footing undergoes an instantaneous settlement accompanied by shear deformations of soil in absence of volumetric changes. When the structure is completed the soil experiences both primary and secondary consolidation accompanied by volumetric changes. The influence of neutral stress appears in the reduction of soil strength. The strength of soil is then presented in terms of total values of the angle of internal friction  $\varphi_u$  and the total cohesion  $c_u$  (these parameters can be considered as the minimal ones). Depending on the degree of consolidation the value of the total angle of internal friction  $\varphi_u$  ranges from 0 to  $\varphi_{ef}$ , the total cohesion  $c_u$  is greater than  $c_{ef}$ . Owing to the fact that the choice of undrained conditions depends on a number of factors (rate of load, soil permeability, degree of saturations and degree of overconsolidation) it is the designer's responsibility to decide, depending on the actual problem being solved, if the effective parameters should be used. Nevertheless, the total parameters are generally used for fine-grained soil.

## Standard Analysis

The following formula is used by default:

$$R_d = (\pi + 2) \cdot c_u \cdot s_c \cdot d_c \cdot i_c \cdot b_c + q$$

with dimensionless coefficients:

$$s_c = 1 + 0,2 \cdot \frac{b}{l}$$

$$d_c = 1 + 0,1 \cdot \sqrt{\frac{d}{b}}$$

$$i_c = (1 - \operatorname{tg} \delta)^2$$

$$b_c = 1 - \frac{2 \cdot \alpha}{\pi + 2}$$

where:

- $c_u$  - total cohesion of soil
- $b$  - width of foundation
- $l$  - length of foundation
- $d$  - depth of foundation
- $\delta$  - angle of deviation of the resultant force from the vertical direction
- $\alpha$  - slope of footing bottom from horizontal direction
- $q$  - overburden pressure at the level of foundation base

## Bearing Capacity of Foundation on Bedrock

The following methods can be used to compute the design bearing capacity  $R_d$  of the foundation with a horizontal footing bottom supported by the rock mass composed of rocks or weak rocks:

- [Standard approach](#)

- Solution according to CSN 73 1001
- Solution according to EC 7-1

## Standard Analysis

The bearing capacity of foundation soil composed of rocks or weak rocks is found from the expression proposed by Xiao-Li Yang and Jian-Hua Yin<sup>1</sup>:

$$R_d = s^{0,5} \sigma_c \cdot N_s + q_0 \cdot N_q + \frac{b}{2} \cdot \gamma_2 \cdot N_\gamma$$

where:

$$s = e^{\frac{GSI-100}{9-3 \cdot D}}$$

$$N_q = \frac{1}{2} \cdot \sec^2 \left( \frac{\pi}{4} + \frac{\varphi}{2} \right) \cdot e^{\left[ \left( \frac{2}{3} \cdot \pi - \varphi \right) \cdot \operatorname{tg} \varphi \right]}$$

$$N_\gamma = (N_q - 1) \cdot \frac{e^{\left[ \left( \frac{\pi}{2} - \varphi \right) \cdot \operatorname{tg} \varphi \right]}}{2 \cdot \cos \varphi}$$

- where:
- |                      |   |   |
|----------------------|---|---|
| $s$                  | - | nonlinear parameter depending on rock properties (according to Hoek and Brown)  |
| $GSI$                | - | Geological Strength Index   |
| $D$                  | - | coefficient reflecting damage of a rock mass                                    |
| $N_s, N_q, N_\gamma$ | - | coefficients of bearing capacity depending on the angle of internal friction    |
| $N_s$                | - | coefficient of strength of a rock depending on GSI and strength parameter $m_i$ |
| $\varphi$            | - | angle of internal friction of rock  |
| $\sigma_c$           | - | uniaxial compressive strength of rock > 0,5 MPa                                 |
| $q_0$                | - | equivalent uniform load accounting for the influence of foundation depth        |
| $\gamma_2$           | - | unit weight of soil above the footing bottom                                    |
| $b$                  | - | width of foundation   |

<sup>1</sup> Xiao-Li Yang, Jian-Hua Yin: Upper bound solution for ultimate bearing capacity with a modified Hoek-Brown failure criterion, International Journal of Rock Mechanics & Mining Sciences 42 (2005), str. 550-560.

## Solution According to CSN 73 1001

The bearing capacity of foundation soil composed of rocks or weak rocks follows from articles 97 - 99 of standard CSN 73 1001 "**Foundation soil below spread footing**" approved 8.6.

1987.

As input parameters the analysis requires the **unit weight of soil**  $\gamma$ , **uniaxial compression strength**  $\sigma_c$ , **Poisson's ratio**  $\nu$  and deformation modulus  $E_{def}$ .

## Analysis According to EC 7-1 (EN 1997-1:2003)

The bearing capacity of the foundation  $R_d$  with a horizontal footing bottom is determined according to a design method for the derivation of expected bearing capacity of spread footings resting on a bedrock outlined in a supplement G (informative) EC 7-1 (EN 1997-1:2003) "Design of geotechnical structures - Part 1: General rules". For low strength or damaged rocks with closed discontinuities including chalks with low porosity less than 35% the derivation of expected bearing capacity follows from classification of rocks into groups of rocks stored in the table below. The analysis further requires an input of discontinuity spacing  $S_d$ , **unit weight of rock**  $\gamma$ , **Poisson's ratio**  $\nu$  and **uniaxial compressive strength**  $\sigma_c$ . It is assumed that the structure is able to transmit a settlement equal to 0,5 % of the foundation width. The expected values of bearing capacity for other settlements can be estimated using direct proportion. For weak and broken rocks with opened or filled discontinuities it is recommended to use lower values than the expected ones.

### Rock groups

Group	Type of rock
1	Pure limestones and dolomites Carbonate sandstones of low porosity
2	Igneous Oolitic and marly limestones Well cemented sandstones Indurated carbonate mudstones Metamorphic rocks, including slates and schist (flat cleavage / foliation)
3	Very marly limestones Poorly cemented sandstones Slates and schist (steep cleavage / foliation)
4	Uncemented mudstones and shales

*Literature:*

*Eurocode 7:Geotechnical design - Part 1:General rules.*

## Parameters to Compute Foundation Bearing Capacity

### Parameters to compute vertical bearing capacity of a foundation resting on bedrock

The following parameters are used in the program to compute the foundation vertical bearing capacity:

- values of coefficient  $D$  reflecting a state of damage of a rock mass



- values of strength parameter  $m_i$
- strength of rocks in simple compression  $\sigma_c$
- Poisson's ratio of rocks  $\nu$
- unit weight of rocks  $\gamma$

#### Estimating disturbance coefficient $D$

Description of rock mass	Suggested value of $D$
Rock mass, intact strong rock, excavation by blasting or by open TBM	0
Rock mass, poor quality rock, mechanical excavation with minimal disturbance	0
Rock mass, poor rock, mechanical excavation, significant floor heave, temporary invert or horizontal geometry of excavation sequence	0.5
Rock mass, very poor rock often very altered, local damage of surrounding rock (app. 3 m )	0.8
Rock slope or rock outcrop, modification with controlled blasting	0.7
Rock slope or rock outcrop, modification with blasting results to the some disturbance	1.0
Open pit mines, excavation with blasting	1.0
Open pit mines, mechanical excavation	0.7

#### Values of strength parameter $m_i$

<b>Type of rock</b>	<b>Representative rocks</b>	$m_i$ [-]
Carbonate rocks with well developed cleavage	Dolomite, limestone and marble	$\approx 7$
Lithified argillaceous rocks	Mudstone, siltstone shale, slate	$\approx 10$
Arenaceous rock with strong crystal and poorly developed crystal cleavage	Sandstone and quartzite	$\approx 15$
Fine grained polyminerallic igneous crystalline rocks	Andesite, dolerite, diabase, rhyolite	$\approx 17$
Coarse grained polyminerallic igneous and metamorphic rocks	Amphibolite, gabbro, gneiss, granite and quartz diorite	$\approx 25$

**Uniaxial compressive strength  $\sigma_c$ , Poisson's ratio  $\nu$  and Unit weight of rock  $\gamma$**

<b>Strength of rocks</b>	<b>Types of rock (examples)</b>	<b>Uniaxial compr. strength</b> $\sigma_c$ [MPa]	<b>Poisson's ratio</b> $\nu$	<b>Unit weight of rock</b> $\gamma$ [kN/m <sup>3</sup> ]
Extremely hard rock	Very hard, intact rock strong and solid quartzite, basalt and other extremely hard rock	>150	0.10	28.00 - 30.00
Very hard rock	Very hard granite, quartz porphyry, quartz slate, very hard sandstones and limestones	100 - 150	0.15	26.00 - 27.00
Hard rock	Solid and compact granite, very hard sandstone and limestone, silicious iron veis, hard pudding stones, very hard iron ores hard calcite, not very hard granite, hard sandstone, marble, dolomite, pyrite	80 - 100	0.20	25.00 - 26.00
Fairly hard rock	Normal sandstone, medium hard iron ore, sandy shale, flagstone	50 - 80	0.25	24.00
Medium hard rock	Hard mudstones, not very hard sandstones and calcite, soft flagstone, not very hard shales, dense marl	20 - 50	0.25 - 0.30	23 - 24.00
Fairly weak rock	Soft schist, soft limestones, chalk, rock salt, frost soils, anthracite, normal marl, disturbed sandstones, soft flagstones and soils with	5 - 20	0.30 - 0.35	22.00 - 26.00

	aggregates			
Weak rock	Compact clay, hard soil (eluvium with soil texture)	0.5 - 5	0.35 - 0.40	22.00 - 18.00

## Horizontal Bearing Capacity of Foundation

The foundation horizontal bearing capacity is verified according to the theory of limit states using the following inequality:

$$H \leq \frac{R_{dh}}{\gamma_{RH}}$$

or based on the factor of safety as:

$$\frac{R_{dh}}{H} \leq SF$$

where:

$$R_{dh} = Q \cdot \tan \psi_d + a_d \cdot A_{ef} + S_{pd}$$

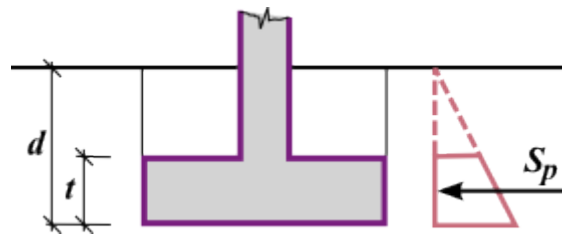
$$H = \sqrt{H_x^2 + H_y^2}$$

- where:
- $\psi_d$  - angle of internal friction between foundation and soil
  - $a_d$  - cohesion between foundation and soil
  - $A_{ef}$  - **effective area** of foundation
  - $S_{pd}$  - earth resistance
  - $H_x, H_y$  - components of horizontal force
  - $Q$  - extreme design vertical force
  - $\gamma_{RH}$  - coefficient of horizontal bearing capacity of foundation (for input use the "**Spread Footing**" tab)
  - $SF$  - safety factor

When adopting the analysis methodology according to EN 1997 the term with cohesion ( $a_d \cdot A_{ef}$ ) is excluded for drained conditions whereas the term with friction between foundation and soil ( $Q \cdot \tan \psi_d$ ) is excluded for undrained conditions.

The analysis depends on the design angle of internal friction below the footing bottom  $\varphi_d$ , the design value of cohesion below the footing bottom  $c_d$  and the design value of earth resistance  $S_{pd}$ . If the soil-footing frictional angle and the soil-footing cohesion are less than the values of soil below the footing bottom, then it is necessary to use those values.

The earth resistance is assumed as displayed in figure:



Earth resistance

The earth resistance  $S_{pd}$  is found with the help of the reduction of passive earth pressure or pressure at rest employing influence coefficients:

$$S_{pd} = \frac{S_p}{\gamma_{mR}}$$

where:  $S_p$  - passive earth pressure, pressure at rest or reduced passive pressure  
 $\gamma_{mR}$  - coefficient of reduction of earth resistance (for input used the "Spread Footing" tab) - for the analysis according to CSN it assumes the value  $\gamma_{mR} = 1.5$  for passive pressure,  $\gamma_{mR} = 1.3$  for pressure at rest

**Coefficients of earth pressures** are clear from the following formulas:

for passive pressure:

$$K_p = \tan(45 + 0,5 \cdot \varphi_d)$$

for pressure at rest in drained soils:

$$K_0 = 1 - \sin \varphi_d$$

for pressure at rest in other soils:

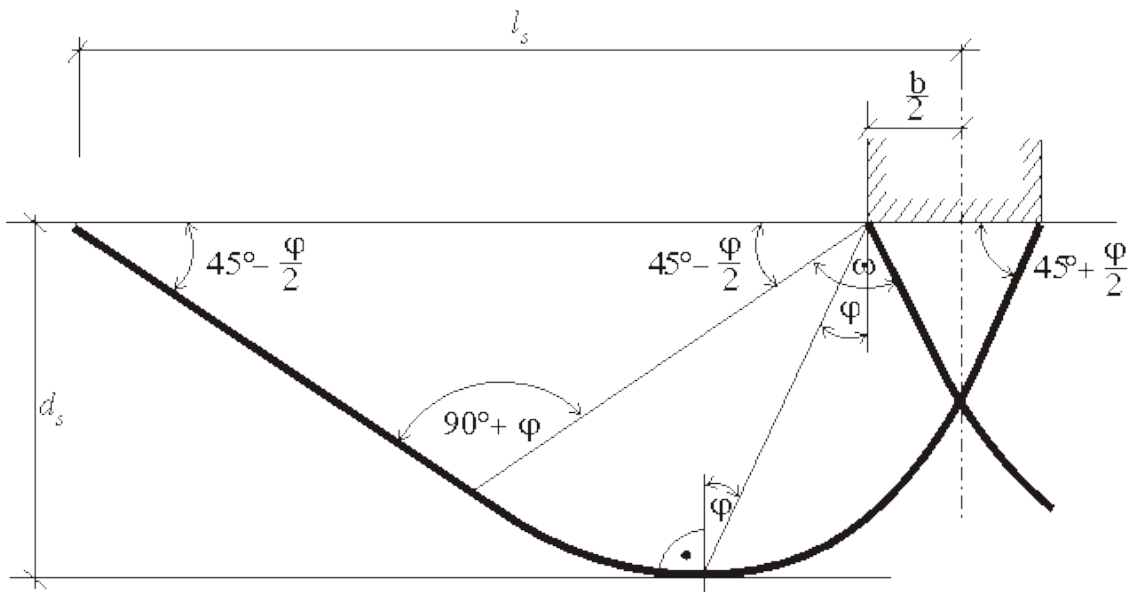
$$K_0 = \frac{\nu}{1 - \nu}$$

When determining the **reduced passive pressure**, the resultant force includes contributions due to the passive pressure and pressure at rest.

The passive pressure can be considered, if the deformation needed for its activation does not cause unallowable stresses or deformations in upper structure.

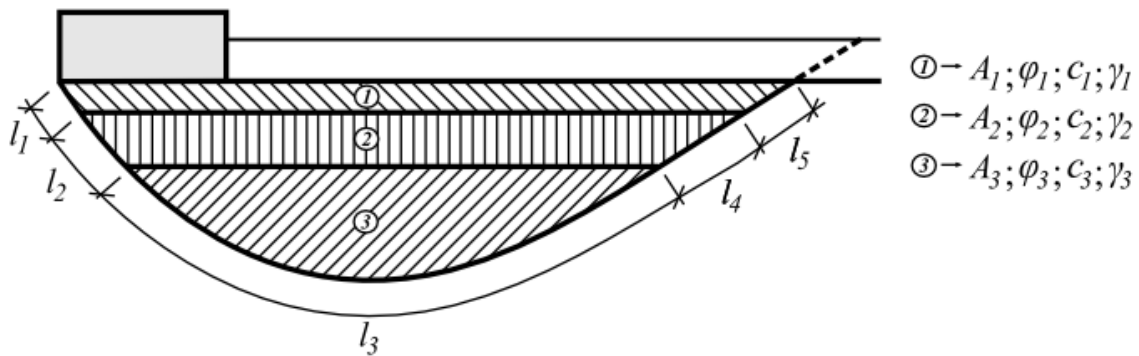
## Homogenization of Layered Subsoil

If the soil below the footing bottom is inhomogeneous (or if there is ground water present) then the input profile is transformed into a homogeneous soil based on the Prandtl slip surface (see Fig.), which represents the type and location of failure of the foundation.



The Prandtl slip surface

Determination of equivalent values of  $\phi$  (angle of internal friction),  $c$  (cohesion of soil)  $\gamma$  (unit weight of soil below footing bottom) is evident from the following formulas. The unit weight of soil above foundation is derived in the same way.



Procedure for computation of auxiliary values

$$\phi = \frac{\phi_1 \cdot (l_1 + l_5) + \phi_2 \cdot (l_2 + l_4) + \phi_3 \cdot l_3}{\sum_{i=1}^3 l_i}$$

$$c = \frac{c_1 \cdot (l_1 + l_5) + c_2 \cdot (l_2 + l_4) + c_3 \cdot l_3}{\sum_{i=1}^3 l_i}$$

$$\gamma = \frac{\gamma_1 \cdot A_1 + \gamma_2 \cdot A_2 + \gamma_3 \cdot A_3}{\sum_{i=1}^3 A_i}$$

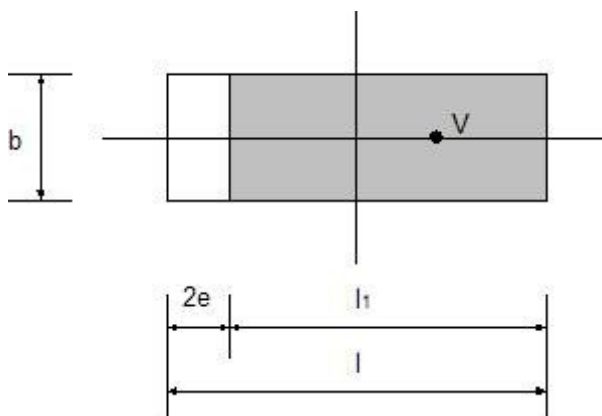
## Effective Area

When solving the problem of eccentrically loaded foundations the program offers two options to deal with an effective dimension of the foundation area:

- a rectangular shape of effective area is assumed
- general shape of effective area is assumed

### Rectangular shape

A simplified solution is used in such cases. In case of axial eccentricity (bending moment acts in one plane only) the analysis assumes a uniform distribution of contact stress  $\sigma$  applied only over a portion of the foundation  $l_1$ , which is less by twice the eccentricity  $e$  compared to the total length  $l$ .



*Determination of effective area in case of axial eccentricity*

An effective area ( $b \cdot l_1$ ) is assumed to compute the contact stress, so that we have:

$$\sigma = \frac{V}{b \cdot (l - 2 \cdot e)}$$

In case of a general eccentric load (foundation is loaded by the vertical force  $V$  and by bending moments  $M_1$  and  $M_2$  the load is replaced by a single force with given eccentricities:

$$e_1 = \frac{M_1}{V}$$

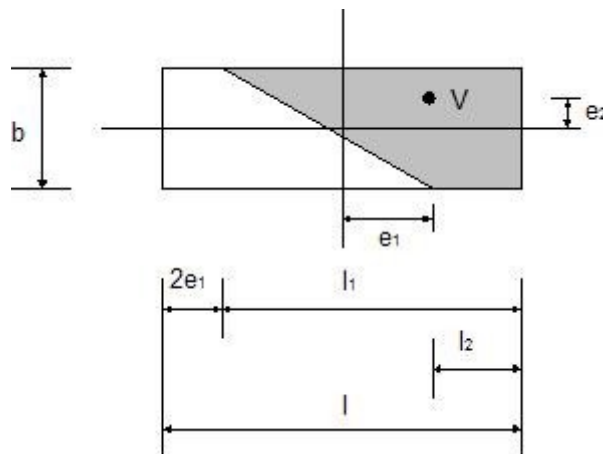
$$e_2 = \frac{M_2}{V}$$

The size of effective area follows from the condition that the force  $V$  must act eccentrically:

$$A_{ef} = b_{ef} \cdot l_{ef} = (b - 2 \cdot e_2) \cdot (l - 2 \cdot e_1)$$

### General shape of contact stress

In case of an eccentric load the effective area is determined from the assumption that the resultant force  $V$  must act in the center of gravity of the compressive area. The theoretically correct solution appears in Fig.



#### *Determination of contact stress for general eccentricity - general shape*

Owing to a considerable complexity in determining the exact location of the neutral axis, which in turn is decisive when computing the effective area, the program follows the solution proposed by Highter and Anders<sup>1)</sup>, where the effective areas are derived with help of graphs.

<sup>1)</sup> Highter, W.H. - Anders, J.C.: Dimensioning Footings Subjected to Eccentric Loads Journal of Geotechnical Engineering. ASCE, Vol. 111, No GT5, pp 659 - 665.

## Determination of Cross-Sectional Internal Forces

**Longitudinal reinforcement of a foundation** is checked for the load due to bending moment and shear force. The stress in the footing bottom is assumed as **linear**. Stresses in individual directions  $x, y$  are determined independently.

When the **linear distribution of stress** in the footing bottom is considered the distribution of stress over the cross-section is provided by:

$$\sigma_1 = \frac{N}{d^2} \cdot \left( 4 \cdot d - 6 \cdot \left( \frac{d}{2} - e \right) \right)$$

$$\sigma_2 = \frac{N}{d^2} \cdot \left( -2 \cdot d + 6 \cdot \left( \frac{d}{2} - e \right) \right)$$

or when excluding tension:

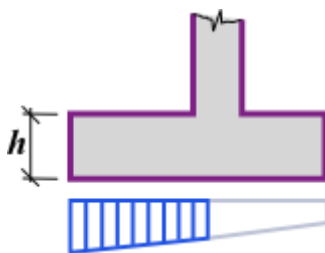
$$\sigma = \frac{2 \cdot N}{3 \cdot \left( \frac{d}{2} - e \right)}$$

where:

- $e$  - eccentricity of normal force  $N$
- $d$  - width of foundation
- $N$  - normal force acting at the footing bottom

Bending moment and shear force are determined as reaction developed on the cantilever beam as shown in figure:





*Internal forces acting on wall jump*

Internal forces in the cross-section corresponding to **constant distribution of stress** are provided by:

$$M = d_v^2 \cdot \sigma$$

$$Q = d_v \cdot \sigma$$

$$\sigma = \frac{N}{d - 2 \cdot e}$$

where:  $\sigma$  - maximal stress in the footing bottom  
 $d_v$  - length of jump  
 $e$  - eccentricity of normal force  $N$   
 $d$  - width of wall foundation  
 $N$  - normal force acting at the footing bottom

## Verification of Foundation Eccentricity

Verification of foundation eccentricity is carried out for the  $I^{st}$  LS (**foundation bearing capacity**) and the  $2^{nd}$  LS (**foundation settlement**) analysis.

During analysis the program performs verification for the following cases:

- maximum eccentricity in the direction of base length:  $e_x \leq e_{alw}$
- maximum eccentricity in the direction of base width:  $e_y \leq e_{alw}$
- maximum overall eccentricity:  $e_t \leq e_{alw}$

The value of maximum allowable foundation eccentricity  $e_{alw}$  is input in the "**Settings**" frame, tab "**Spread Footing**".

The value of overall eccentricity  $e_t$  is provided by:

$$e_t = \sqrt{e_x^2 + e_y^2}$$

where:  $e_x$  - maximum eccentricity in the direction of base length  
 $e_y$  - maximum eccentricity in the direction of base width

Procedure for calculating eccentricities needed for the determination an effective area of off-center (eccentrically) loaded foundation is described in more detail [here](#).

For a shallow foundation resting on a rock foundation or for a concrete slab type of foundation it is necessary in some cases to adopt different values of limit eccentricities.

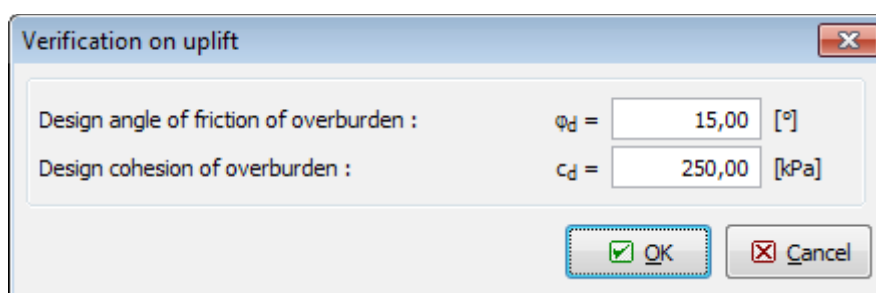
## Analysis of Uplift

The analysis of a spread footing in tension is performed when the load due to a negative normal force  $N$  (the force acts upwards) is assumed. The verification of such a footing is carried out according to the corresponding [verification methodology](#). During the analysis the program compares the maximum tensile force  $N_{t,max}$  with the uplift resistance  $R_t$ . The program considers the following three methods of the calculation of bearing capacity (uplift resistance  $R_t$ ) of footing:

- [standard approach](#)
- [cone method](#)
- [DL/T 5219-2005](#)

## Standard Approach

The uplift resistance  $R_t$  combines the self weight of the soil overburden and footing + friction along the footing walls + fictitious (substitute) block of soil above the footing. The "**Bearing cap.**" frame in the "**Verification on uplift**" dialog window allows to input the design angle of friction of overburden  $\varphi_d$  and the design cohesion of overburden  $c_d$ .

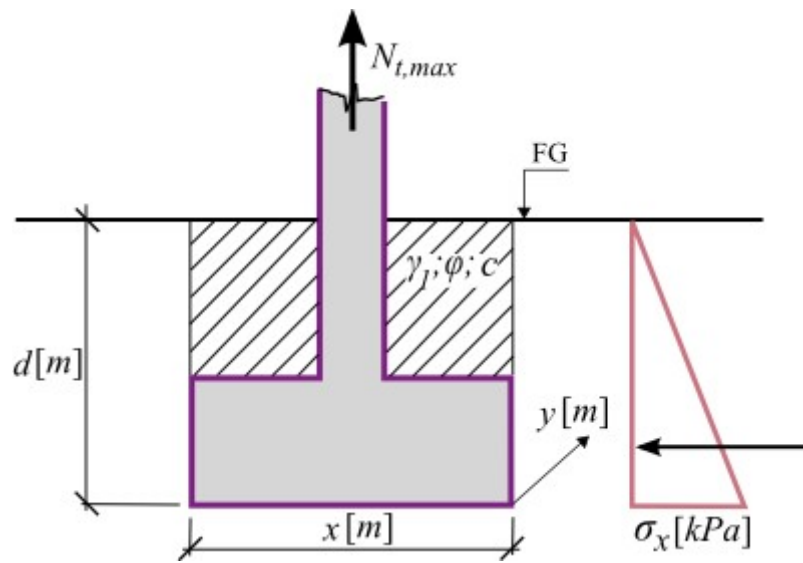


Dialog window "Verification on uplift" - standard approach

The vertical bearing capacity check - spread footing in tension (uplift resistance) follows from:

$$R_t = (\sigma_x \operatorname{tg} \varphi_d + c_d) d p + G_p$$

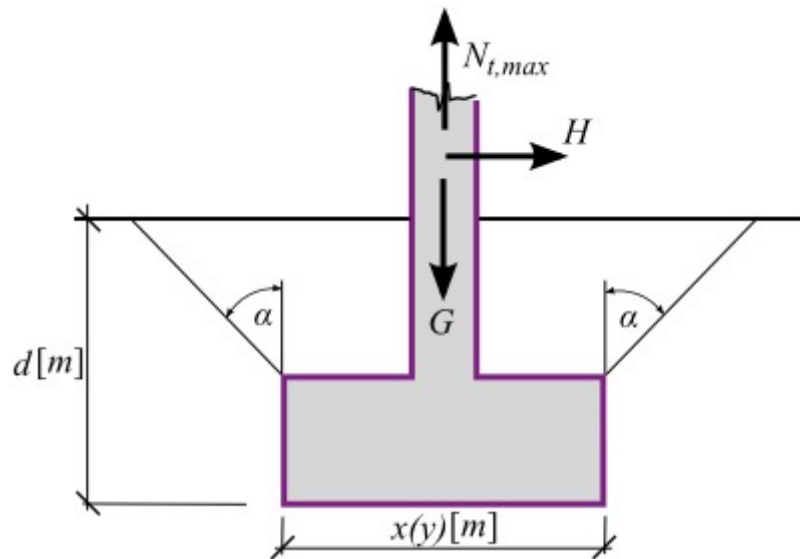
- where:
- $\sigma_x$  - [earth pressure at rest](#) due to overburden
  - $\varphi_d$  - design angle of internal friction of overburden
  - $c_d$  - design cohesion of overburden
  - $d$  - depth of footing bottom
  - $p$  - footing perimeter
  - $G_p$  - self weight of foundation



Verification on uplift - standard approach

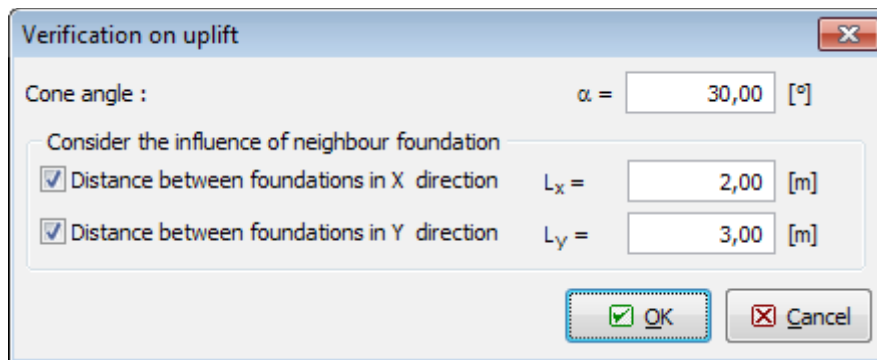
### Cone Method

The uplift resistance  $R_t$  combines the footing self weight and the self weight of the soil overburden in the shape of cone as evident from the following figure.

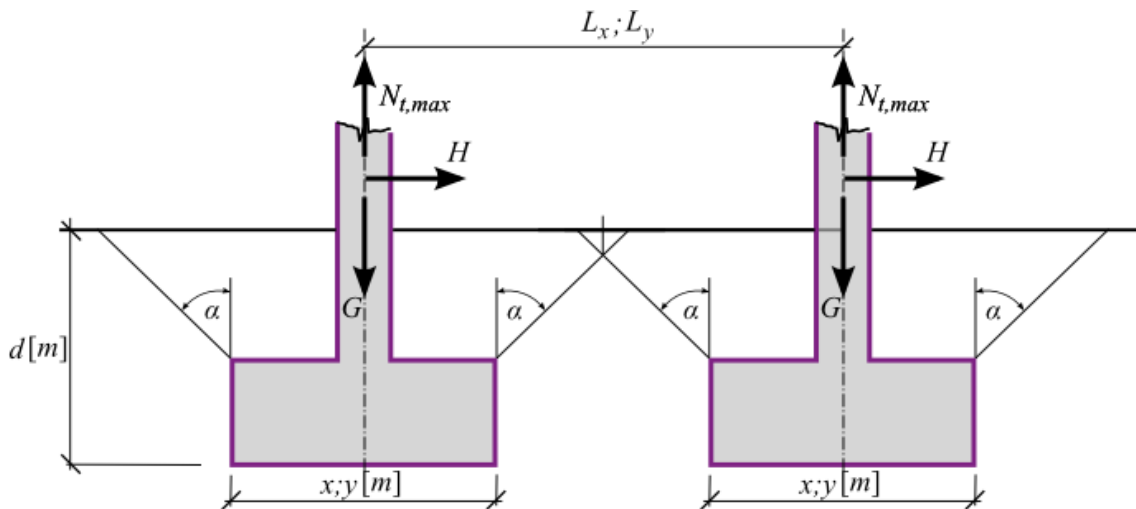


Basis of the cone method

The "**Bearing cap.**" frame in the "**Verification on uplift**" dialog window serves to input the cone angle  $\alpha$ . When calculating the uplift resistance  $R_t$  it is also possible to account for the influence of a neighbour foundation either in one or both directions that reduces the volume of the soil cone.



Dialog window "Verification on uplift" - cone method



Influence of a neighbour foundation

## DL/T 5219 - 2005

This type of verification originates from the Chinese standard **DL/T 5219 - 2005**. Unlike the [cone method](#) and the [standard approach](#) it introduces a critical depth  $h_c$ , which depends on the type of soil and the shape of foundation. The "**Verification on uplift**" dialog window allows either inputting the critical depth  $h_c$  directly or it can be determined by the program depending on the input type of soil and the shape of foundation according to table **6.3.1-1 - Critical depth  $h_c$** .

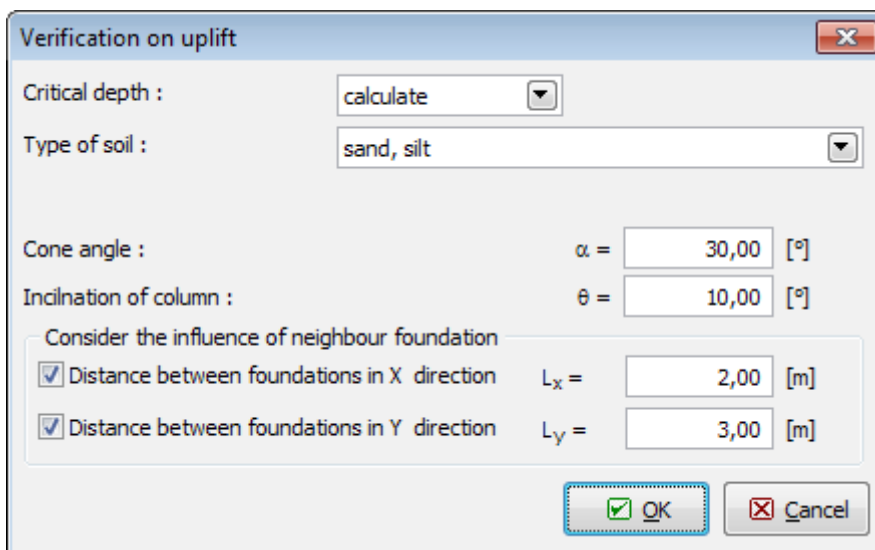
**Table 6.3.1-1 with the values of a critical depth based on the Chinese standard DL/T 5219-2005**

Type of soil	Natural state of soil	Critical depth $h_c$ for tensile foundation	
		Circular foundation	Square foundation
Sand or Silt	Dense ~ Slightly dense	$2.5D$	$3.0B$
Clay	Hard ~ Stiff	$2.0D$	$2.5B$
	Plastic	$1.5D$	$2.0B$
	Soft - plastic	$1.2D$	$1.5B$

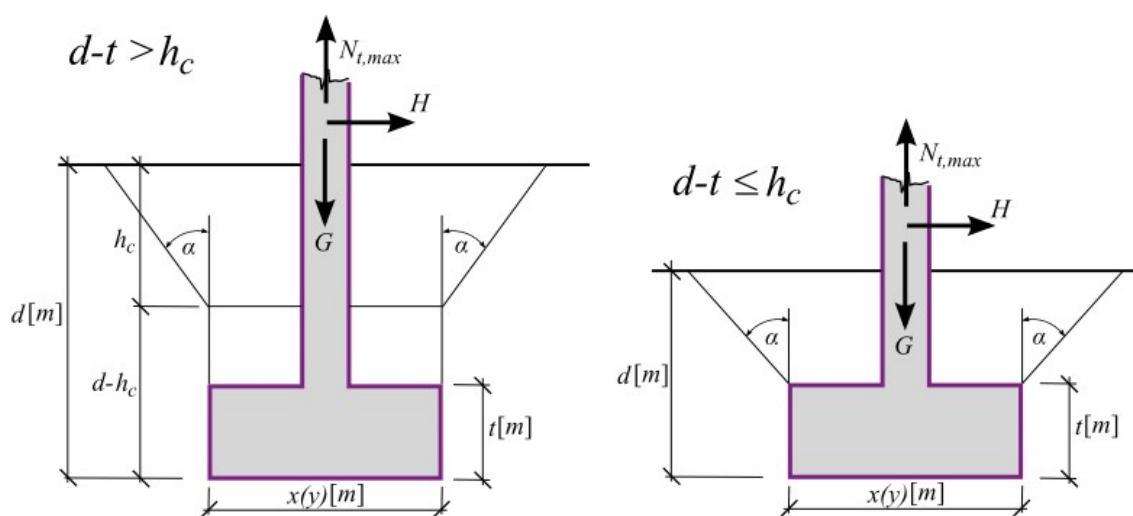
Note 1: For rectangular foundation, if the ratio between length  $L'$  and width  $B$  is smaller than 3, calculate  $h_c$  as circular foundation and  $D = 0,6*(B+L')$ .

Note 2: The soil must be in a natural state.

The "**Bearing cap.**" frame in the "**Verification on uplift**" dialog window serves to specify as another input parameter the cone angle  $\alpha$ . The influence of a neighbour foundation is described in the **cone method**. The inclination of column  $\theta$  has no influence on the calculation of bearing capacity of a footing in tension (the uplift resistance).



Dialog window "Verification on uplift" - DL/T 5219-2005



Verification of footing in tension according to DL/T 5219-2005

## Pile Analysis

Analyses available in the program "Pile" can be divided into three main groups:

- Analysis of vertical bearing capacity
- Pile settlement
- Analysis of horizontal bearing capacity

## Vertical Bearing Capacity

Analysis of pile vertical resistance can be carried out using:

- Analytical solution
- Spring method

## Analytical Solution

The analytical solution assumes that the pile total compressive resistance  $R_c$  is derived as a sum of the pile base resistance  $R_b$  and the pile shaft resistance  $R_s$  (developed due to friction of the surrounding soil along the shaft). The following generally accepted methods are implemented into the program:

- NAVFAC DM 7.2
- Tomlinson
- Effective stress method
- CSN 73 1002

For the above specified methods it is possible to choose one of the following verification methodologies:

- Classical way
- EN 1997-1

When running the **compression pile** analysis, the pile self-weight is introduced depending on the setting in the frame "Load". As for the **tensile pile**, the pile self-weight is always taken into account automatically. Based on the input load the program itself performs the verification analysis for **either compression or tensile pile**.

## NAVFAC DM 7.2

Calculation of vertical pile resistance is performed according to the publication: NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy 1984, where all approaches are described in more detail. The analysis provides the **pile base resistance**  $R_b$  and the **pile shaft resistance**  $R_s$ .

For non-cohesive, the program takes into account the **critical depth**.

### Pile Base Resistance

Pile base resistance for **non-cohesive soils** is given by:

$$R_b = \sigma_{efb} \cdot N_q \cdot A_b$$

where:  $\sigma_{efb}$  - effective stress on the pile base

$N_q$  - **bearing capacity factor**

$A_b$  - area of pile base

The **bearing capacity factor**  $N_q$  is back calculated by the program; however, its values can be manually modified.

For **cohesive soils** the following expression holds:

$$R_b = 9 \cdot c_u \cdot A_b$$

where:  $c_u$  - undrained shear strength at the base

$A_b$  - area of pile base

### Pile Shaft Resistance

Pile shaft resistance for **non-cohesive soils** is given by:

$$R_s = \sum_{j=1}^n K_j \cdot \sigma_{ef,j} \cdot \text{tg} \delta_j \cdot A_{s,j}$$

where:  $K_j$  - **coefficient of lateral earth pressure** in the  $j^{\text{th}}$  layer

$\sigma_{ef,j}$  - effective strength of soil in the  $j^{\text{th}}$  layer

$\delta_j$  - **pile skin friction angle** (between pile material and surrounding soil in the  $j^{\text{th}}$  layer)

$A_{s,j}$  - area of pile shaft in the  $j^{\text{th}}$  layer

The **lateral earth pressure coefficient**  $K$  is back calculated by the program; however, its values can be manually modified in the "Add new soils" dialog window.

For **cohesive soils** the following expression holds:

$$R_s = \sum_{j=1}^n \alpha_j \cdot c_{u,j} \cdot A_{s,j}$$

where:  $\alpha_j$  - skin friction coefficient in the  $j^{th}$  layer  
 $c_{u,j}$  - undrained cohesion in the  $j^{th}$  layer  
 $A_{sj}$  - area of pile shaft in the  $j^{th}$  layer

## Bearing Capacity Factor $N_q$

Reference values of the bearing capacity factor  $N_q$  are listed in the table. If jet grouting is used when constructing the pile, then the maximum angle of internal friction  $\varphi$  is equal to  $28^\circ$ .

**Bearing capacity factor  $N_q$**

Angle of internal friction $\varphi[^\circ]$	26	28	30	31	32	33	34	35	36	37	38	39	40
Bearing capacity factor $N_q$ for driven piles	10	15	21	24	29	35	42	50	62	77	86	120	145
Bearing capacity factor $N_q$ for bored piles	5	8	10	12	14	17	21	25	30	38	43	60	72

*Literature:*

NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy, 1984.

## Coefficient of Lateral Earth Pressure $K$

The soil around a driven pile is compressed during construction and the lateral earth pressure of this soil acting on the pile skin is greater than the earth pressure at rest (given by coefficient  $K_0$ ) and smaller than the maximum earth pressure (passive earth pressure given by coefficient  $K_p$ ):

$$K_0 < K < K_p$$

Reference values of the coefficient of lateral earth pressure  $K$  are listed later in the table. The coefficient of lateral earth pressure  $K$  is approximated as follows:

$$K = \frac{K_a + K_p + K_0}{3}$$

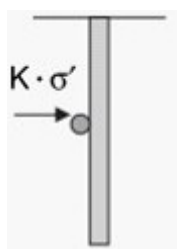
where:  $K_0$  - coefficient of earth pressure at rest  
 $K_0 = 1 - \sin\varphi$   
 $\varphi$  - angle of soil shear resistance  
 $K_p$  - coefficient of passive earth pressure



$$K_p = \operatorname{tg}^2\left(45^\circ + \frac{\varphi}{2}\right)$$

$K_a$  - coefficient of active earth pressure

$$K_a = \operatorname{tg}^2\left(45^\circ - \frac{\varphi}{2}\right)$$



*Pressure on the pile*

#### Reference values of the lateral earth pressure coefficient $K$

Type of pile	$K$ for compression piles	$K$ for tensile - uplifted piles
Driven H-piles	0.5 - 1.0	0.3 - 0.5
Driven displacement piles (round and square)	1.0 - 1.5	0.6 - 1.0
Driven displacement tapered piles	1.5 - 2.0	1.0 - 1.3
Driven jetted piles	0.4 - 0.9	0.3 - 0.6
Bored piles (less than 70cm)	0.7	0.4

*Literature:*

NAVFAC DM 7.2, Foundation and Earth Structures, U.S. Department of the Navy, 1984.

### Friction Angle on Pile Skin

Reference values of friction angle between the pile skin material and the surrounding non-cohesive soil are listed in the following table:

**Friction angle on pile  $\delta$  [°]**

<b>Pile material</b>	$\delta$ [°]
Steel piles	20
Timber piles	$0.75\varphi$
Steel reinforced concrete piles	$0.75\varphi$

where:  $\varphi$  - angle of internal friction of soil

*Literature:*

NAVFAC DM 7.2, *Foundation and Earth Structures*, U.S. Department of the Navy, 1984.

## Adhesion Coefficient

Reference values of the adhesion coefficient  $\alpha$  are listed in the following table:

**Empirical adhesion coefficient  $\alpha$**

<b>Pile material</b>	<b>Soil consistency</b>	<b>Cohesion range</b> $c_u$ [kPa]	<b>Adhesion coefficient <math>\alpha</math> [-]</b>
<b>Timber and concrete piles</b>	Very soft	0 - 12	0.00 - 1.00
	Soft	12 - 24	1.00 - 0.96
	Medium stiff	24 - 48	0.96 - 0.75
	Stiff	48 - 96	0.75 - 0.48
	Very stiff	96 - 192	0.48 - 0.33
<b>Steel piles</b>	Very soft	0 - 12	0.00 - 1.00
	Soft	12 - 24	1.00 - 0.92
	Medium stiff	24 - 48	0.92 - 0.70
	Stiff	48 - 96	0.70 - 0.36
	Very stiff	96 - 192	0.36 - 0.19

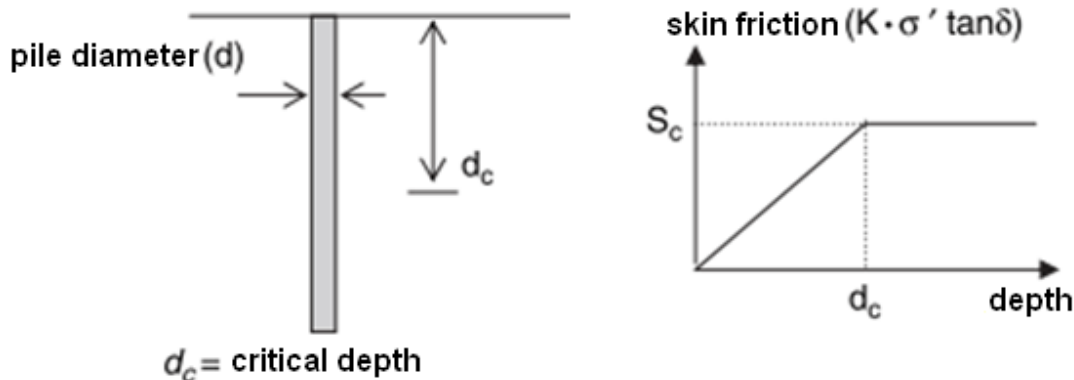
*Literature:*

NAVFAC DM 7.2, *Foundation and Earth Structures*, U.S. Department of the Navy, 1984.

## Critical Depth

For **non-cohesive** soils the skin friction does not increase infinitely with depth as e.g. **effective stress**, but from a certain so called **critical depth** it acquires a constant value - see the following figure, where  $d_c$  is the critical depth,  $S_c$  is the skin friction at critical depth and  $d$  is the pile diameter. Similar rule holds also for the pile base resistance in non-cohesive soils,

where the same values of the critical depth  $d_c$  are considered for simplicity.



*Critical depth*

The reference value of the critical depth for soft sand is  $10d$  ( $d$  is the pile diameter or its width), for medium compact sand and compact sand the values are  $15d$  and  $20d$ , respectively.

**The coefficient of critical depth**  $k_{dc}$  can be specified in the "Ver. capacity". The critical depth follows from:

$$d_c = k_{dc} \cdot d$$

where:  $k_{dc}$  - critical depth coefficient  
 $d$  - pile diameter

## Tomlinson

This widely used method adopts undrained shear strength parameters to calculate the pile bearing capacity. It further assumes that the pile shaft resistance depends on the pressure due to overburden surcharge.

The **pile shaft resistance** is given by:

$$R_s = \sum_{j=1}^n c_{a,j} \cdot A_{s,j} = \sum_{j=1}^n \alpha_j \cdot c_{u,j} \cdot A_{s,j}$$

where:  $c_{a,j}$  - **adhesion** in the  $j^{th}$  layer (shear stress between the pile skin and the surrounding soil)  
 $A_{s,j}$  - area of pile shaft in the  $j^{th}$  layer  
 $\alpha_j$  - **empirical adhesion coefficient** (depends on the type of soil, type of pile, etc.) in the  $j^{th}$  layer  
 $c_{u,j}$  - undrained cohesion in the  $j$ -th layer (undrained shear strength)

The **empirical adhesion coefficient**  $\alpha$  is back calculated by the program. Its values, however, can be manually adjusted in the "Add new soil" dialog window.

The **pile base resistance** is given by:

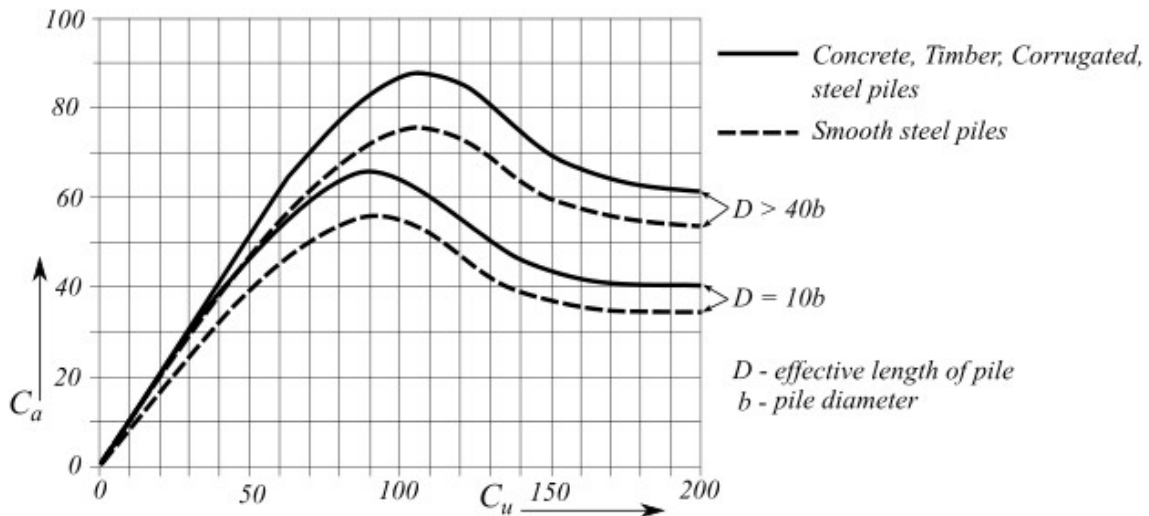
$$R_b = q_b \cdot A_b = 9 \cdot c_u \cdot A_b$$

where:  $q_b$  - unit pile base resistance

- $A_b$  - pile base area
- $c_u$  - undrained shear strength

### Adhesion Coefficient

The empirical adhesion coefficient  $\alpha$  takes into account the behavior of soil around the pile skin and depends on the pile material, quality of the pile skin surface and the type of surrounding soil. Values of this coefficient are introduced into the program employing the following graph taken from M.J. Tomlinson: Pile Design and Construction Practice.



Graph to determine adhesion coefficient

### Effective Length

The effective length  $D$  determines the pile length, which effectively transfers the load from the pile into the soil. If the whole pile is placed into the resistant soil, in which the load is transferred by skin friction, then the effective length corresponds to the pile length below terrain - see Fig. A. In case of layered medium, highly compressible layers (in which the load is not transferred into the soil by skin friction) and the layers above are not counted into the effective length  $D$  - see Fig. B. Introduction of effective length into the analysis and its magnitude are set in the frame "Vertical capacity".

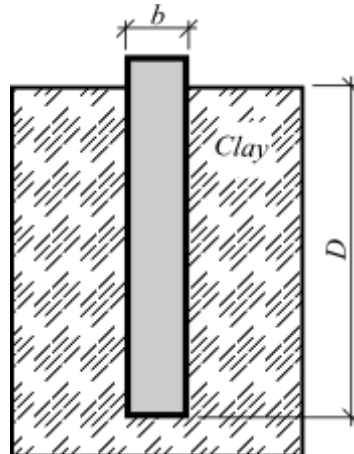


Fig. A Effective length in resistant soils

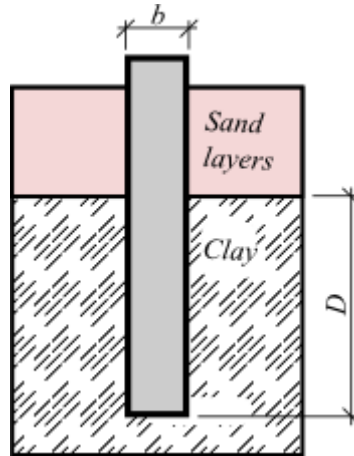


Fig. B Effective length in layered medium

## Effective Stress Method

The effective stress method allows to calculate the vertical bearing capacity of an isolated pile in both cohesive and non-cohesive soils. This method is suitable for drained conditions - i.e. conditions that prevail after sufficient time passed the construction.

The **pile shaft resistance** is given by:

$$R_s = \sum_{j=1}^n q_{s,j} \cdot A_{s,j} = \sum_{j=1}^n \beta_{p,j} \cdot \sigma_{0,j} \cdot A_{s,j}$$

- where:
- $q_{s,j}$  - shaft resistance in the  $j^{th}$  layer
  - $\beta_{p,j}$  - coefficients according to Bjerrum and Burland in the  $j^{th}$  layer
  - $\sigma_{0,j}$  - average effective stress due to overburden acting along the pile in the  $j^{th}$  layer
  - $A_{sj}$  - pile shaft area in the  $j^{th}$  layer

The **pile base resistance** is given by:

$$R_b = q_p \cdot A_b = N_p \cdot \sigma_p \cdot A_b$$

- where:
- $q_p$  - unit pile base resistance
  - $A_b$  - pile base area
  - $N_p$  - pile base resistance coefficient (according Fellenius)
  - $\sigma_p$  - effective stress due to overburden acting at pile base

## Coefficients of Pile Bearing Capacity

Recommended ranges of values of coefficients of pile base resistance  $N_p$  and coefficient  $\beta$  are listed in the following table. The coefficient  $\beta$  is usually found in the given range, it seldom exceeds the value 1,0.

**Range of coefficients  $N_p$  and  $\beta$  (Fellenius, 1991)**

Type of soil	$\varphi_{ef}$	$N_p$	$\beta$
Clay	25 - 30	3 - 30	0.23 - 0.40
Silt	28 - 34	20 - 40	0.27 - 0.50
Sand	32 - 40	30 - 150	0.30 - 0.60
Gravel	35 - 45	60 - 300	0.35 - 0.80

Literature:

Fellenius, B.H.: *Foundation Engineering Handbook*, Editor H.S. Fang, Van Nostrand Reinhold Publisher, New York, 1991, 511 - 536.

**CSN 73 1002**

There are two methods implemented in the program to compute the pile vertical bearing capacity following the Commentary to the **CSN 73 1002 standard "Pile foundation"**:

- **Analysis according to the theory of the 1st group of limit states**

The solution procedure is described in the Commentary to the CSN 73 1002 standard "Pile foundation" in Chapter 3 "Design" part B - general solution according to the theory of the 1st group of limit states (pp. 15). All computational approaches are based on formulas presented therein. The original geostatic stress  $\sigma_{or}$  is assumed from the finished grade. The coefficient of conditions of the behavior of foundation soil is considered for the depth  $z$  (measured from the finished grade).

$$z \leq 1 \Rightarrow \gamma_{r2} = 1,3$$

$$1 < z \leq 2 \Rightarrow \gamma_{r2} = 1,2$$

$$2 < z \leq 3 \Rightarrow \gamma_{r2} = 1,1$$

$$3 < z \Rightarrow \gamma_{r2} = 1,0$$

The effective pile length used for the computation of skin bearing capacity is reduced by a segment:

$$l_p = \frac{d \cdot N_d^{2/3}}{4}$$

where:  $d$  - pile diameter

- **Analysis of pile resting on incompressible subsoil**

Analysis of a pile resting on incompressible subsoil (rocks class R1, R2) is based on part G - Analysis of vertical bearing capacity  $R_c$  according to CSN 73 1004 - Commentary to CSN 73 1002 "Pilotové základy". The description begins in page 27 titled "Piles resting on incompressible subsoil". The solution procedures used in the program are identical. The influence coefficient of settlement  $I_{wp}$  is interpolated from Table 16, which is also built-in the program.

If checking the option "**analysis according to CSN 73 1002**" in the "Piles" tab the verification analysis is carried out exclusively according to CSN 73 1002 and other coefficients

ate not used. Providing this option is not checked the verification is performed based on the selected methodology adopting particular coefficients.

*Literature:*

Československá státní norma ČSN 73 1002 Pilotové základy, Normalizační institut, Praha, 1987.

Československá státní norma ČSN 73 1004 Velkopřůměrové piloty, Normalizační institut, Praha, 1981.

## Verification

Verification of pile **bearing capacity** depends on the verification methodology selected in the "Piles" tab:

- verification according to the **factor of safety**
- verification according to the theory of **limit states**
- verification according to **EN 1997**

Actual analyses (e.g. assessment of the pile base resistance) are the same for both options - they differ only by incorporation of design coefficients, combinations and in the way of demonstrating the structure safety. **Design coefficients** (verification parameters) are specified in the "Piles" tab.

If the verification analysis **according to CSN 73 1002** is selected, the verification is carried out exclusively according to the Commentary to CSN 73 10002.

## Verification According to the Theory of Limit States

When running the verification analysis according to the theory of **limit states**, it is possible to introduce the required values of **design coefficients** in the "Piles" tab.

The program performs verification of the **compression pile** as:

$$R_c = \frac{R_b}{\gamma_b} + \frac{R_s}{\gamma_s} \geq V_d + W_p$$

where:	$R_c$	-	pile compressive resistance
	$R_b$	-	pile base resistance
	$R_s$	-	pile shaft resistance
	$\gamma_b$	-	partial factor on pile base resistance
	$\gamma_s$	-	partial factor on pile shaft resistance
	$V_d$	-	extreme vertical load acting on a pile
	$W_p$	-	pile self-weight

For **tension pile** the following verification applies:

$$R_{sdt} = \frac{R_s}{\gamma_{st}} \geq V_d + W_p$$

where:	$R_{sdt}$	-	pile tensile resistance
--------	-----------	---	-------------------------

- $R_s$  - pile shaft resistance
- $\gamma_{st}$  - partial factor on tensile pile shaft resistance
- $V_d$  - extreme vertical load acting on a pile
- $W_p$  - pile self-weight

## Design Coefficients

The "Piles" tab allows to specify two groups of design (partial) coefficients:

### Partial factors on soil parameters

- $\gamma_{m\phi}$  - reduction coefficient of internal friction
- $\gamma_{mc}$  - reduction coefficient of cohesion
- $\gamma_{m\gamma}$  - coefficient of unit weight

It is also possible to choose reduction of  $tg\phi$ .

### Partial factors on pile resistance

- $\gamma_b$  - reduction coefficient of base resistance
- $\gamma_s$  - reduction coefficient of shaft resistance
- $\gamma_t$  - reduction coefficient of total resistance
- $\gamma_{st}$  - reduction coefficient of resistance in tension

The values of individual coefficients are listed in corresponding standards.

## Verification According to the Safety Factors

When running the verification analysis according to the **factor of safety**, it is possible to introduce the required value of factor safety  $SF$  for the vertical bearing capacity in the "Piles" tab.

The program performs verification of vertical bearing capacity of **compression pile** as:

$$\frac{R_c}{V_d + W_p} > SF_{cp}$$

- where:
- $V_d$  - extreme vertical load acting on a pile
  - $R_c$  - pile compressive resistance
  - $W_p$  - pile self-weight (introduction into the analysis based on the setting in the frame "Load")

and for **tension pile**:

$$\frac{R_{tdt}}{V_d + W_p} > SF_{tp}$$



where:  $V_d$  - extreme vertical load acting on a pile  
 $R_{St}$  - pile tensile resistance  
 $W_p$  - pile self-weight

## Vertical Bearing Capacity - Spring Method

The program module "**Pile - spring method**" is part of the "**Pile**" program. It allows to calculate the pile vertical bearing capacity in general layered subsoil. This analysis provides the **load-settlement curve** and distributions of forces and displacements developed along the pile.

The main advantage of this module is availability of the required input parameters of soils around the pile - the user is asked to specify the **angle of internal friction, cohesion, unit weight and deformation modulus** of a given soil.

The solution procedure in the module "**Pile - spring method**" is based on a semi-analytical approach. The response of soil surrounding the pile follows from the well known solution of layered subsoil as a generalization of the Winkler-Pasternak model. The elastic rigid plastic response in shear is assumed along the pile-soil interface in view of the Mohr-Coulomb failure criterion. The normal stress acting on the pile is determined from the **geostatic stress** and soil (concrete mixture) pressure at rest.

The **influence of water** in the vicinity of pile is not only introduced into the **shear bearing capacity** of the pile skin, but also affects the **depth of influence zone** below the pile heel.

Providing the pile reaches **incompressible subsoil** the spring method cannot be used.

The pile settlement can also be influenced by the settlement of the surrounding terrain. In particular, settlement of soil may reduce the pile bearing capacity. The pile settlement increases without increasing load. This phenomenon is modeled in the program as so called **negative skin friction**.

The analysis may also account for the **influence of technological process** of pile construction on the stiffness of pile foundation.

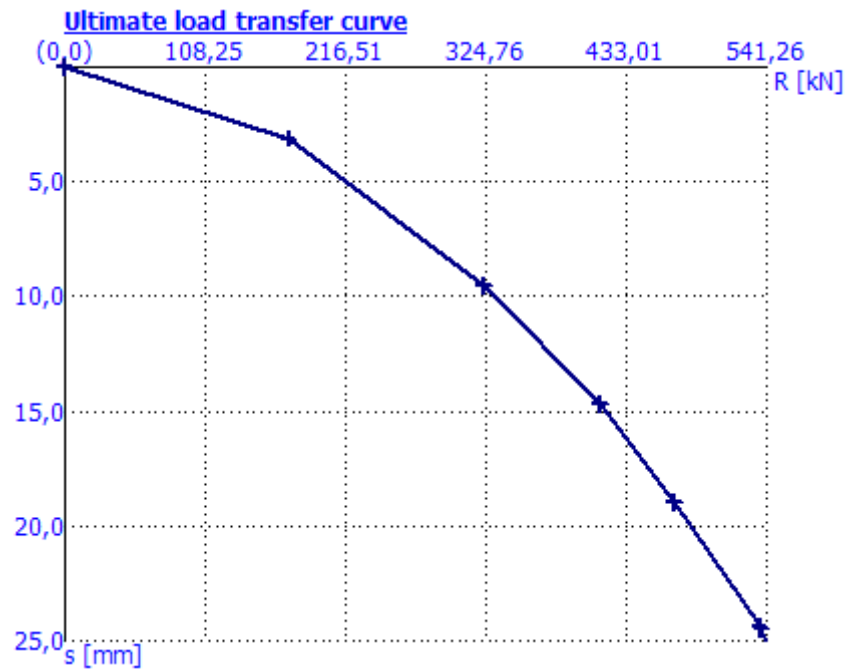
The solution procedure consists of several steps:

- 1) In analysis the pile subdivided into a number of segments. Subdivision into individual segments complies with the condition that the ratio between the pile segment and its diameter should be approximately equal to 2.5. The minimum number of segments is 10.
- 2) Each segment is in the analysis characterized by a spring. The spring stiffness serves to model both the **shear resistance of skin** and at the pile heel the **stiffness of soil below the pile heel**.
- 3) For each segment the **limit value of shear force**  $T_{lim}$  transmitted by the skin is determined.
- 4) The pile is loaded at its top end by increments of the vertical load. For each load increment the magnitude of spring force for each segment is determined. However, it cannot exceed the limit value of skin friction  $T_{lim}$ . It is clear that for a certain load level all springs will no longer be able to increase their force and with additional load increase the pile becomes supported by the base spring only. This spring has no restriction on the transmitted force.
- 5) As a result the analysis provides the **load-settlement curve, forces developed in the pile** and a graph showing variation of **shear as a function of deformation** at a given location.

## Load-Settlement Curve

The Load-settlement curve describes the variation of vertical load  $Q$  as a function of the pile settlement.

By default the program offers the construction of this curve for the maximal value of settlement equal to 25 mm. This magnitude, however, can be adjusted up to the value of 100 mm before running the calculation. An example showing a typical shape of the load-settlement curve appears in the figure.



*Load-settlement curve of single pile*

## Shear Strength of Skin

For each segment of the analyzed pile the program determines the limiting value of the force that can be transmitted by the pile skin at the location of a given segment. Its value depends on the geostatic stress  $\sigma_z$  found at a depth of a given segment.

$$\sigma_z = \sum \gamma \cdot h$$

where:  $\gamma$  - unit weight of soil  
 $h$  - depth below the ground surface

Summation sign denotes that  $\sigma_z$  is summed over individual layers of the soil.

The allowable shear stress is then given by:

$$\tau = \sigma_z \cdot k \cdot \text{tg} \varphi + c$$

where:  $c$  - cohesion of soil at the location of beam  
 $\varphi$  - angle of internal friction of soil at the location of beam  
 $k$  - coefficient of increase of allowable skin friction due to technology

If the beam is found below the ground water table, the allowable skin friction is then reduced

to receive the form:

$$\tau = (\sigma_z - u).k.tg\varphi + c$$

where:  $u$  - pore pressure below the ground water table

The allowable shear force then follows from:

$$T_{lim} = O.l.\tau$$

where:  $O$  - length of perimeter of pile skin

$l$  - length of pile beam

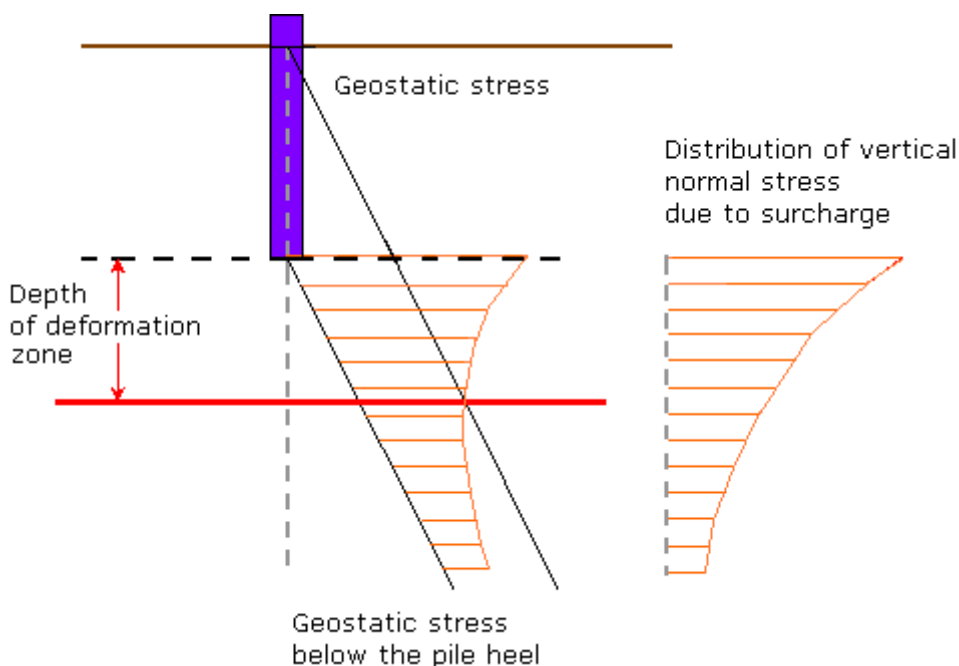
## Coefficient of Increase of Limit Skin Friction

A specific input parameter is the coefficient of increase of limit skin friction  $k$  due to applied technology of construction. By default the value of this coefficient is set equal to 1.0. There is no recommendation by standard for its specific value - its adjustment depends solely on the practical experiences of the designer. It has been found from the in situ measurements on real piles that the value of  $k$  is usually greater than 1.0 and may reach the value of 1.5. Theoretically, however, it may attain values even less than 1.0.

## Depth of Deformation Zone

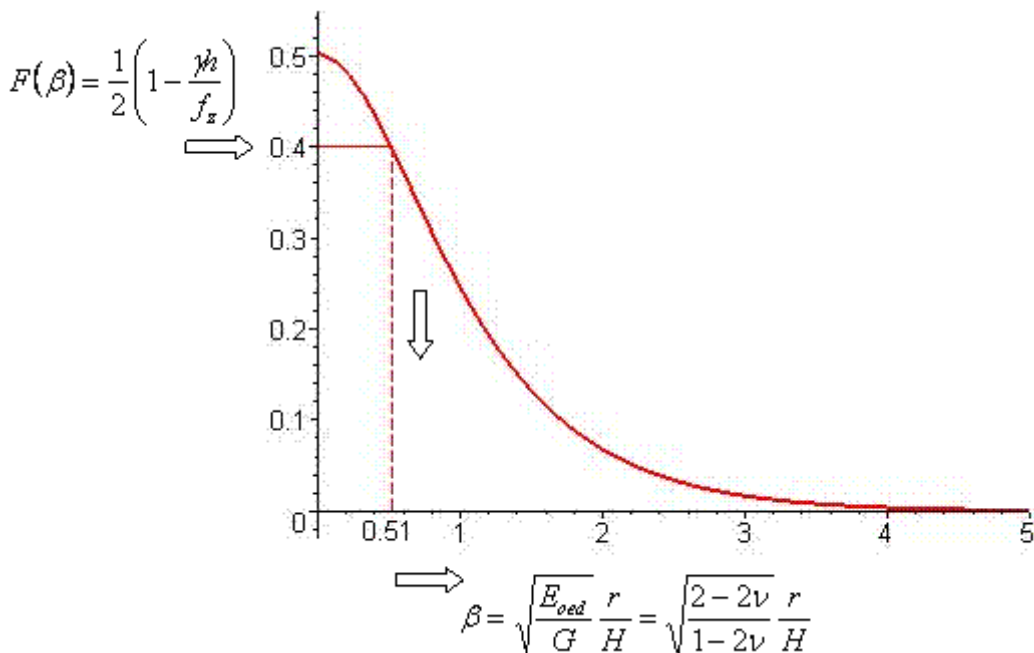
The assumed depth of influence is a variable, which considerably influences the stiffness of soil below the pile heel. It is one of the input parameters for the determination of parameters  $C_1$  and  $C_2$  of the Winkler-Pasternak model. The deeper the influence zone the smaller the stiffness of subsoil. When the depth of influence zone approaches in the limit zero the stiffness of subsoil tends to infinity.

The depth of influence zone depends both on subsoil parameters and magnitude of the applied surcharge, thus on stress below the pile heel. The program assumes that the depth of influence zone is found in the location, where the stress below the heel equals the geostatic stress. Such an idea is depicted in the following figure:



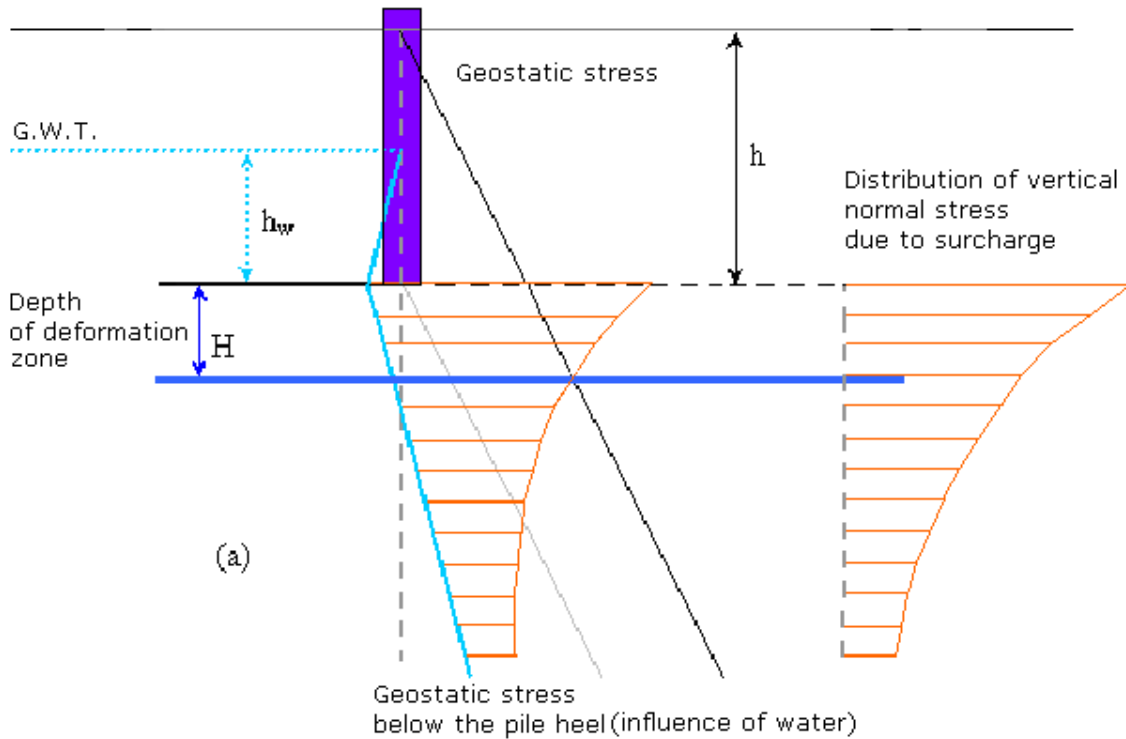
*Determination of the depth of influence zone below the pile heel*

For digital determination of the depth of influence zone  $H$  serves the function  $F(\beta)$ . Its distribution appears in figure. This function was derived using the above assumptions and in the program appears in the form of table. Its application is evident from the following steps. The values of  $F(\beta)$  are determined for the current value of stress  $f_z$  below the pile heel and for the original geostatic stress  $\gamma_h$ . For this value of  $F(\beta)$  we determine the parameter  $\beta$ . This value serves to determine for the actual value Poisson's ratio  $\nu$  and pile diameter  $r$  the corresponding depth of influence zone  $H$ .



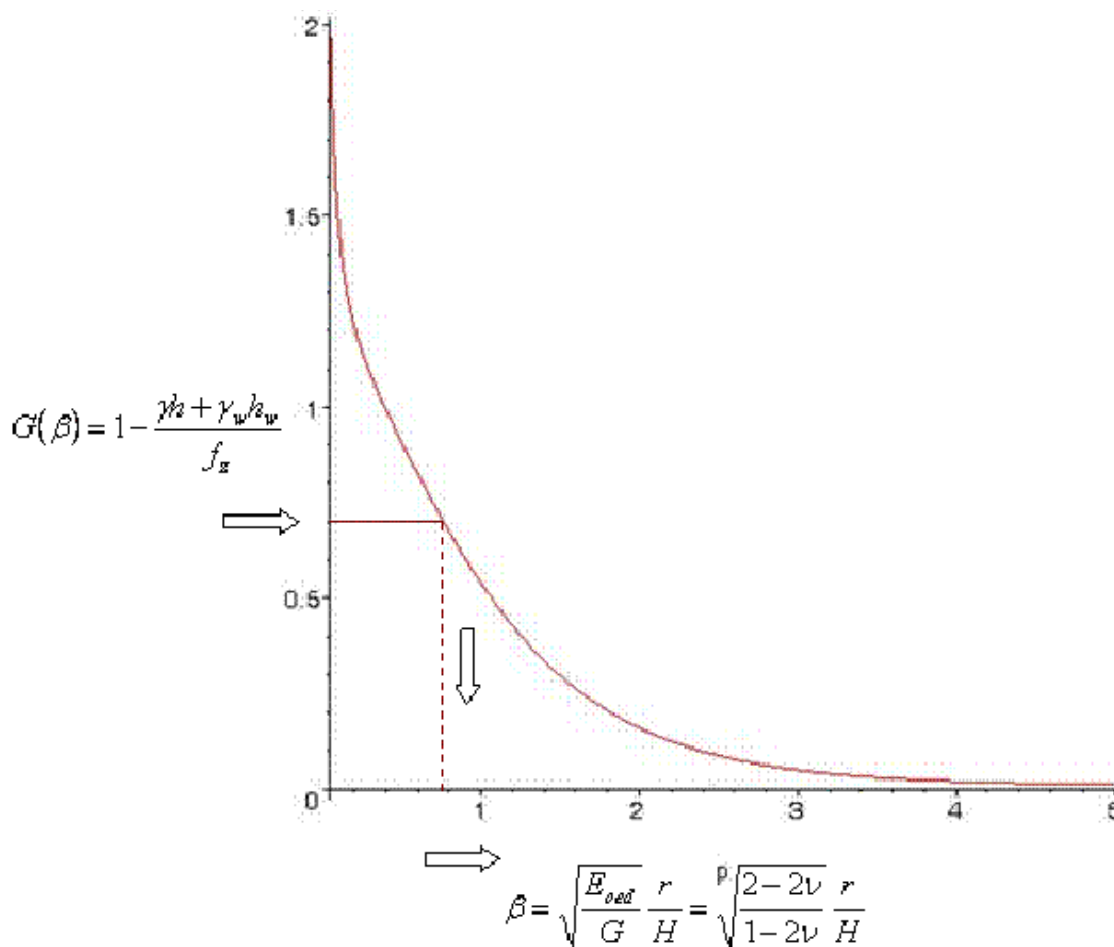
Variation of function  $F(\beta)$

The depth of influence zone can be affected by the presence of ground water. In such a case its determination is outlined in following figure:



*Determination of the depth of influence zone below the pile heel including water*

For digital determination of the depth of influence zone  $H$  is then used the function  $G(\beta)$ . Its distribution appears in figure. In the analysis this function is exploited in the similar way as function  $F(\beta)$ . The only difference when determining the values of  $G(\beta)$  appears in the use of hydrostatic pressure  $\gamma_w * h_w$ .



Variation of function  $G(\beta)$

### Incompressible Subsoil

At a certain depth below the ground surface it is possible to specify incompressible subsoil. If the pile exceeds this specified depth the spring method cannot be used, because the pile is assumed rigid and therefore no deformation can developed in its surrounding. If there is incompressible subsoil below the pile heel but not deeper than the depth of influence zone below the heel, the **depth of influence zone** for the stiffness computation is reduced such that the influence zone reaches the incompressible subsoil. This way also the incompressible subsoil below the base increases its stiffness and consequently also the bearing capacity of the pile base. If the incompressible subsoil is found below the depth of influence zone, it does not influence the analyzed pile.

### Negative Skin Friction

A negative skin friction is a phenomenon that arises from a settlement of soil in the vicinity of a pile. The soil deforming around the pile tends to pull the pile downwards thus reducing its bearing capacity for a given pile settlement.

The input parameters for assessing the influence of negative skin friction is the settlement of ground surface  $w$  and a depth of influence zone of this deformation  $h$ . For a uniformly distributed load around the pile the value of  $w$  should be measured in the distance equal to

three times the pile diameter from its outer face. The value then represents the depth influenced by the ground surface settlement and below which the soil is assumed incompressible with no deformation.

Computation of negative skin friction is carried out first while determining the limit shear forces transmitted by the pile skin  $T_{lim}$ . The solution procedure assumes that the soil settlement decreases linearly with depth from the value of  $w$  on the ground surface up to 0 at a depth of  $h$ . The specific value of the soil settlement is therefore assumed for each level below the ground surface till the depth of  $h$ . The forces developed in springs of pile segments due to their deformation are determined and then subtracted from  $T_{lim}$  to reduce the bearing capacity of the pile skin.

From the presented theory it is evident that for large settlement  $w$  or large depth  $h$  the values of  $T_{lim}$  may drop down to zero. In extreme cases the negative skin friction may completely eliminate the skin bearing capacity so that the pile is then supported only by the elastic subsoil below the pile heel.

## Influence of Technology

The pile bearing capacity is considerably influenced by technological processes applied during construction. The module "**Pile - Spring method**" allows to specify the technology of pile construction. The mobilized skin friction and the resistance at the pile heel are then reduced with the help of reduction coefficients depending on the selected technology. The values of these coefficients follow from the Dutch standard NEN 6743 Pile foundation.

Apart from technologies offered by the program and corresponding coefficients the users are free to assign to these coefficients their own values. This way the users may introduce their own practical experiences or information provided by other sources into the analysis.

## Shear Resistance on Skin

The shear resistance on pile skin is in the analysis represented by the stiffness of springs supporting individual pile segments. This stiffness is associated with material parameters of the Winkler-Pasternak model  $C_1$  and  $C_2$ . The values of  $C_1$  and  $C_2$  are determined from parameter  $E_{def}$ . They depend on the depth of influence zone, which varies with the pile deformation (settlement). The variability of influence zone is in the analysis determined such that for zero deformation it receives the value of  $1x$  the pile diameter and for deformation at the onset of skin failure equals  $kx$  the pile diameter, where  $k$  is the specified value, resp. 2,5.

The decisive parameter for the determination of magnitudes of  $C_1$  and  $C_2$  is the deformation modulus. Caution must be taken when estimating the value of  $E_{def}$  from deformational characteristics of soil using standards. In particular, in case of long piles we are essentially dealing with deep seated foundations and the soil at the pile heel will certainly experiences higher stiffness than that proposed by the standard for spread footings. This holds particularly for cohesive soils. The most reliable estimates are of course those obtained directly from experimental measurements.

Formulas given below serve to determine the stiffness of springs representing the shear resistance of pile skin as a function of computed parameters of the elastic subsoil. They depend on the shape of cross-section and for the implemented cross-sections they receive the following forms:

### Circle:

$$k = 2 \cdot \pi \cdot r \cdot \sqrt{C_1 C_2} \cdot \frac{K_1(\alpha \cdot r)}{K_2(\alpha \cdot r)}$$

where:  $r$  - radius of pile cross-section  
 $C_1, C_2$  - subsoil parameters  
 $K_1(\alpha r), K_2(\alpha r)$  - values of the modified Bessel functions

Parameter  $\alpha$  attains the value:

$$\alpha = \sqrt{\frac{C_1}{C_2}}$$

### Rectangle:

$$k = [2 \cdot (a + b) \cdot \sqrt{C_1 C_2} + 3 \cdot C_2] \cdot k_{red}$$

where  $a, b$  are lengths of rectangle edges and  $C_1, C_2$  are subsoil parameters and  $k_{red}$  is the reduction coefficient, which reduces the stiffness with respect to slenderness of the rectangle.

It receives the following values

$$k_{red} = 0,6 + 0,4 \cdot e^{0,5 \cdot \left(1 - \frac{b}{a}\right)} \quad H \geq 3 \cdot a$$

$$k_{red} = 1 - \frac{1 - 0,6 + 0,4 \cdot e^{0,5 \cdot \left(1 - \frac{b}{a}\right)}}{3 \cdot a} \cdot H \quad H < 3 \cdot a$$

where  $a$  is the length of a shorter edge of the rectangle and  $H$  is the depth of influence zone.

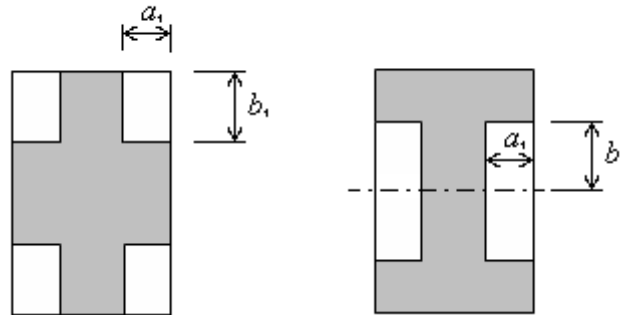
### Cross, "I-section":

For these cross-sections the stiffness is derived from the stiffness of rectangular cross-section reduced by subtracting the stiffness corresponding to four "removed" parts of the cross-section.

$$k = [2 \cdot (a + b) \cdot \sqrt{C_1 \cdot C_2} + 3 \cdot C_2] \cdot k_{red} - 4 \cdot \left(1 - e^{-e \cdot \sqrt{a_1^2 + b_1^2}}\right) \cdot \left[ C_1 \cdot \frac{a_1 \cdot b_1}{9} + C_2 \cdot \left( \frac{a_1}{3 \cdot b_1} + \frac{b_1}{3 \cdot a_1} \right) \right]$$

$a_1, b_1$  - evident from the following figure





### Stiffness of Subsoil Below the Pile Heel

The soil stiffness below the pile heel follows from the value of stiffness of the Winkler model  $C_I$ . The value of  $C_I$  is determined for soil parameters  $E_{def}$  and  $\nu$  at the location of pile heel. The value of  $C_I$  further depends on the **depth of influence zone beneath the heel**.

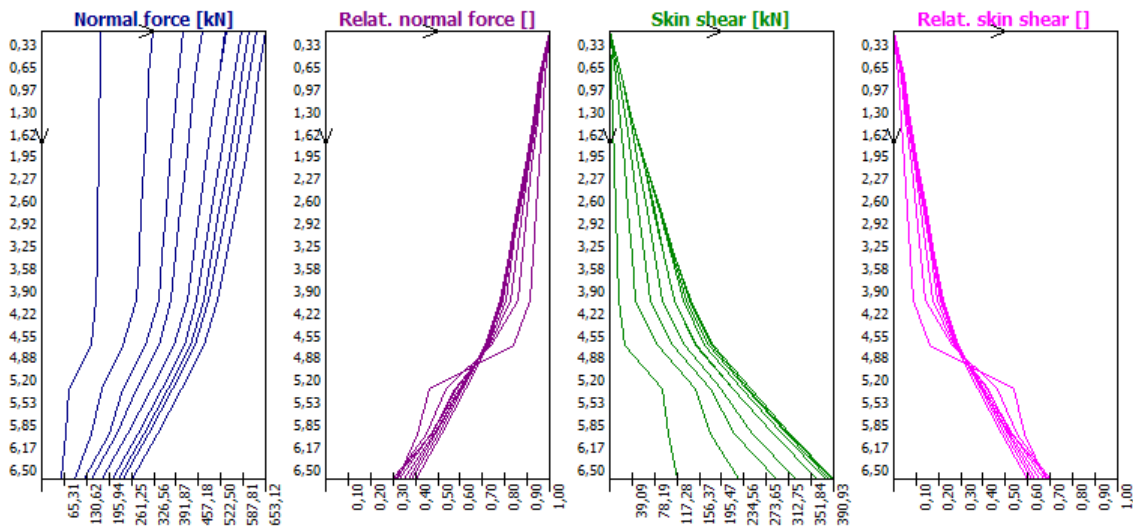
The spring stiffness introduced at the pile base is then provided by:

$$k_r = C_I \cdot A$$

where:  $A$  - cross-sectional area at the pile heel

### Distributions of Forces Acting on a Pile

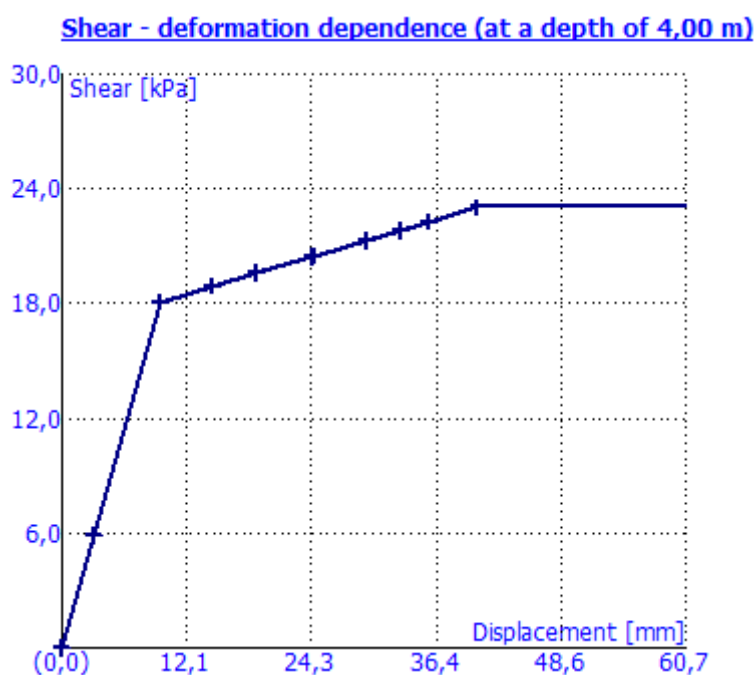
Apart from the load-settlement curve it is also possible to keep track of the distribution of normal force in the pile and the distribution of shear force developed along the pile skin. The normal forces decreases from the top to the bottom as the load is gradually taken by the shear force developed along the pile skin. Unlike the normal force the shear force thus increases from the top to the bottom. Both forces are evaluated in relative values related to the magnitude of vertical load.



Distribution of internal forces acting on pile

## Dependence of Shear on Deformation

At an arbitrary (selected) depth it is possible to view the distribution of skin friction as a function of displacement (settlement) of a given point of the pile. This graph shows the process of gradual reduction of shear stiffness of pile skin until zero with increasing deformation. This dependency is initially linear, particularly in stage, where the spring force does not exceed the value  $T_{lim}$ . When this value is exceeded the spring stiffness starts to gradually decrease manifested by the flattening of the curve.



*Dependence of shear on displacement (settlement) of pile*

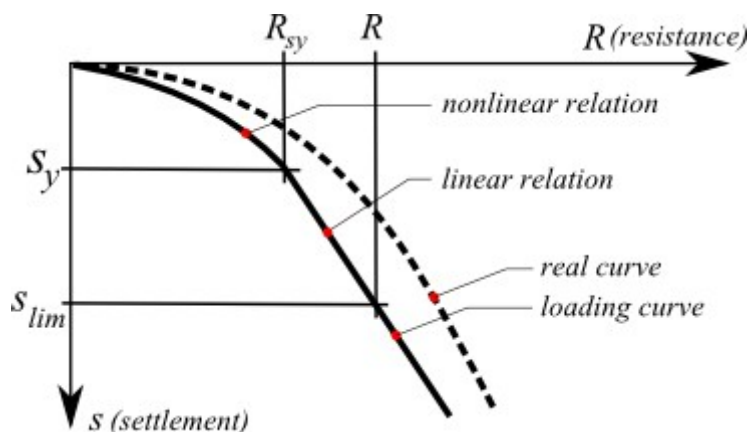
## Pile Settlement

Two options are available to perform the pile settlement analysis:

- Nonlinear theory (Masopust)
- Linear theory (Poulos)

### Nonlinear Theory (Masopust)

The nonlinear theory constructs load-settlement curve assuming that evolution of settlement as a function of resistance up to full mobilization of skin friction can be represented by parabola. After that the relationship is linear as displayed in figure. This method was derived from equations of regression curves constructed on the basis of statistical analysis of the results of static loading tests of piles and for the determination of vertical bearing capacity it employs [regression coefficients](#). Further details are provided [herein](#).



Load-settlement curve of pile

Literature:

Masopust, J.: Vrtane piloty. 1<sup>st</sup> edition, Prague, Cenek a Jezek, 1994, 263 p.

Masopust, J., Glisnikova, V.: Zakladani staveb Modul M01. 1<sup>st</sup> edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.

## Approach According to Masopust

The load-settlement curve of single pile is constructed in the following way:

1) The ultimate skin friction  $q_s$  is determined as follows:

$$q_s = a - \frac{b}{\frac{v_i}{d_i}}$$

- where:
- $a, b$  - regression coefficient of the specific skin friction
  - $v_i$  - depth from terrain up to the middle of the  $i^{th}$  layer [m]
  - $d_i$  - pile diameter in the  $i^{th}$  layer [m]

and pile skin bearing capacity is the provided by:

$$R_s = m_1 \cdot m_2 \cdot \pi \cdot \sum_{i=1}^n d_i \cdot h_i \cdot q_{si}$$

- where:
- $m_1$  - load type coefficient
  - $m_2$  - shaft protection coefficient
  - $d_i$  - pile diameter in the  $i^{th}$  layer [m]
  - $h_i$  - thickness of the  $i^{th}$  layer [m]
  - $q_{si}$  - ultimate skin friction in the  $i^{th}$  layer [MPa]

2) The pile base bearing capacity  $q_b$  follows from:

$$q_b = e - \frac{f}{D/d_b}$$

where:  $e, f$  - regression coefficients below pile base  
 $D$  - pile length inside soils [m]  
 $d_b$  - pile base diameter [m]

**3) The proportion of applied load transferred to pile base  $\beta$**  is written as:

$$\beta = \frac{q_b}{q_b + 4 \cdot \bar{q}_s \cdot D/d_b}$$

where:  $q_b$  - pile base bearing capacity [MPa]  
 $\bar{q}_s$  - weighted average of ultimate skin friction [MPa]  
 $D$  - pile length inside soils [m]  
 $d_b$  - pile base diameter [m]

The **load at mobilization of skin friction**  $R_{sy}$  is then given by:

$$R_{sy} = \frac{R_s}{1 - \beta}$$

where:  $R_s$  - pile skin bearing capacity [N]  
 $\beta$  - proportion of applied load transferred to pile base [-]

**4) The load at the shaft resistance activation (= mobilization of skin friction)  $R_{sy}$**  is given by:

$$s_y = I_s \cdot \frac{R_{sy}}{0,7 \cdot d \cdot E_s}$$

where:  $I_s$  - settlement-influence factor  
 $R_{sy}$  - load at the mobilization of skin friction [N]  
 $d$  - pile diameter [m]  
 $E_s$  - secant modulus of soil along the pile shaft [MPa]

**5) The load at the pile base for the prescribed settlement (for limiting settlement of 25 mm)** follows from:

$$R_{b,lim} = \beta \cdot R_{sy} \cdot \frac{s_{lim}}{s_y}$$

where:  $\beta$  - proportion of applied load transferred to pile base [-]  
 $R_{sy}$  - load at the mobilization of skin friction [N]  
 $s_{lim}$  - limit settlement (usually prescribed as 25 mm) [m]

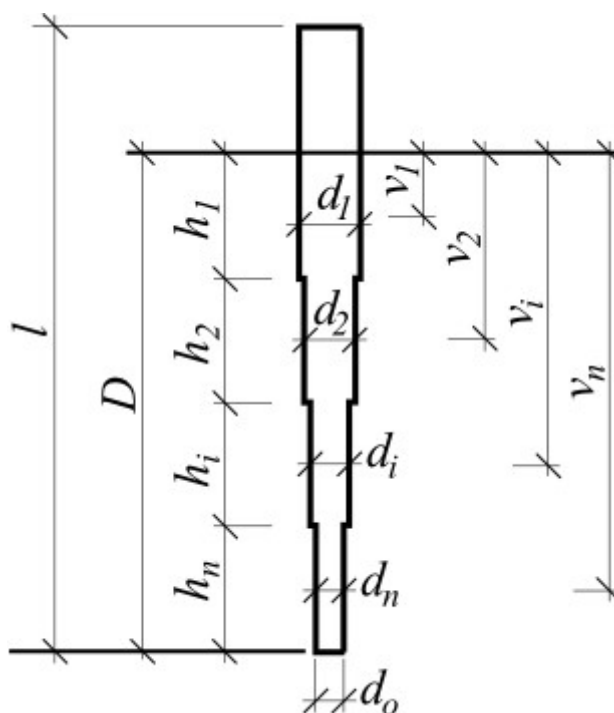
$s_y$  settlement at shaft resistance activation [m]

and the pile resistance attributed to a given **limit settlement**  $s_{lim}$  is then provided by:

$$R_c = R_{b,lim} + R_s$$

where:  $R_{b,lim}$  - load on pile base for prescribed settlement [N]

$R_s$  - pile shaft resistance [N]



Approach according to Masopust

*Literature:*

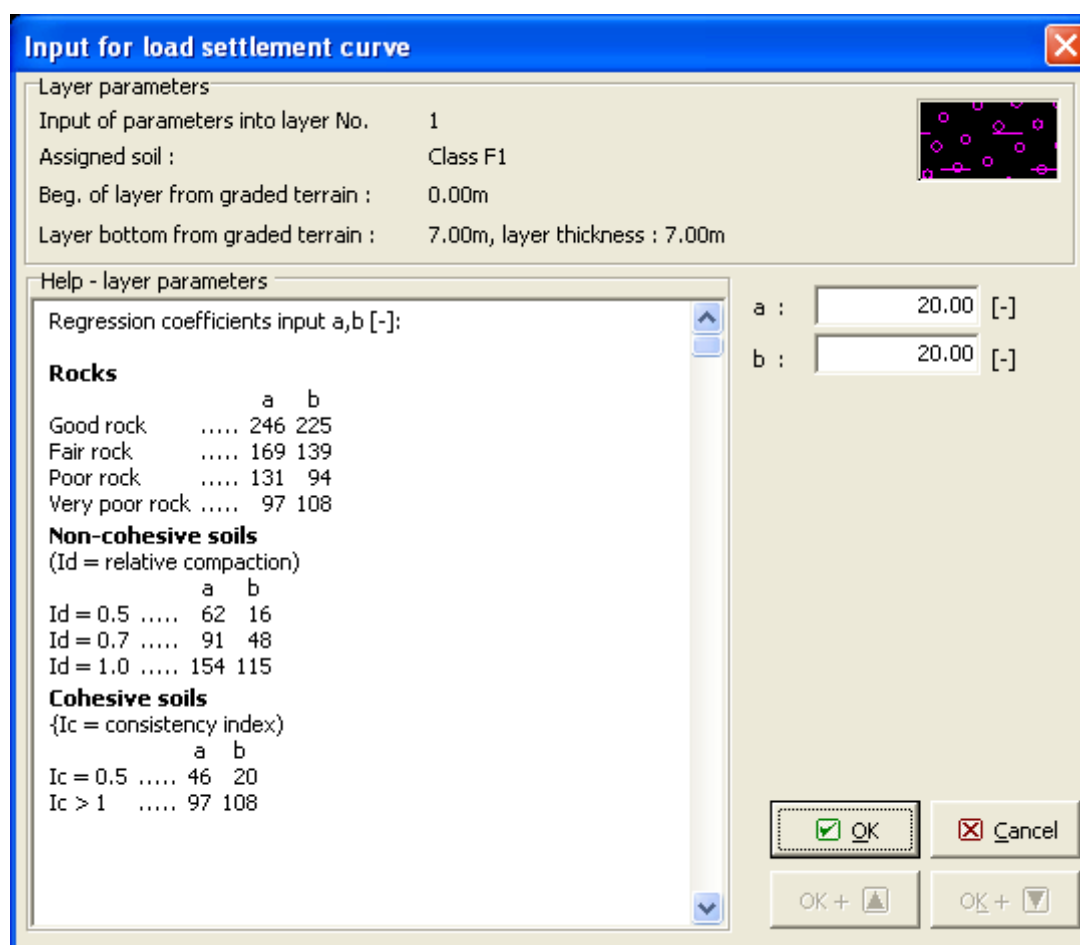
Masopust, J.: *Vrtane piloty*. 1<sup>st</sup> edition, Prague, Cenek a Jezek, 1994, 263 p.

Masopust, J., Glisnikova, V.: *Zakladani staveb Modul M01*. 1<sup>st</sup> edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.

## Regression Coefficients

The specific skin friction depends on regression coefficients  $a, b$ . The pile base resistance (at the full mobilization of skin friction) depends on the regression coefficients  $e, f$ . The values of these regression coefficients were derived from equations of regression curves constructed on the basis of statistical analysis of the results of approximately 350 static loading tests of piles.

The dialog window for entering regression coefficients can be displayed in the frame "Settlement" using the "Edit  $a, b$ ", "Edit  $e, f$ " buttons. When editing the dialog window displays the recommended values of regression coefficients for various types of soils and rocks.



Dialog window "Input for load settlement curve" - input of regression coefficients  $a$ ,  $b$  (e, f)

## Coefficients $m_1$ , $m_2$

### Load type coefficient $m_1$ :

- for service load 0.7
- for extreme load 1.0

### Shaft protection coefficient $m_2$ :

- for concreting in dry shaft or under water 1.0
- for concreting with bentonite slurry 0.9
- for PVC sheet pile protection (thickness over 0.7 mm) 0.7
- for sheet pile protection and B system mesh 0.5
- for steel casing tube protection 0.15

### Literature:

Masopust, J., Glisnikova, V.: *Zakladani staveb Modul M01. 1<sup>st</sup> edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.*

## Secant Modulus of Soil $E_s$

Value of secant modulus of soil  $E_s$  depends on pile diameter  $d$  and thickness of an individual layers of the soil  $h_i$ . The values of this modulus should be determined experimentally from pile-load tests.

For cohesionless soils its value also depends on the index of relative density  $I_d$ , for cohesive soils this value depends on the index of consistency  $I_c$ . Value of secant modulus of soil  $E_s$  increases with depth (thickness of soil layer).

Issue 60 in [2] states that the cumulative value of this module (applicable to all soil types along the shaft and the base of large-diameter pile) is given by following equation:

$$E_s = I_s \frac{Q}{s d}$$

where:  $I_s$  - settlement-influence factor [-]

$d$  - pile diameter [m]

$Q$  - relevant value of load (force) measured during the pile-loading test [N]

$s$  - relevant value of pile settlement measured during the pile-loading test [m]

Values of secant modulus  $E_{si}$  for various types of soils and different pile diameters and depths of piles are shown in the following tables [3]. Intermediate values of secant modulus of soil  $E_s$  can be interpolated linearly.

### Secant modulus of soil $E_s$ located in rocks and weak rocks

$h$ (m)	$d$ (m)								
	0.6			1.0			1.5		
	R3	R4	R5	R3	R4	R5	R3	R4	R5
1.5	50.3	28.2	20.2	72.3	35.0	24.7	85.5	33.5	22.3
3.0	64.5	43.1	30.8	105.5	57.3	41.0	138.3	58.8	41.2
5.0	-	58.2	41.3	-	75.3	54.8	-	87.9	63.7
10.0	-	87.5	61.6	-	114.5	83.2	-	133.0	97.0

### Secant modulus of soil $E_s$ located in cohesionless soils

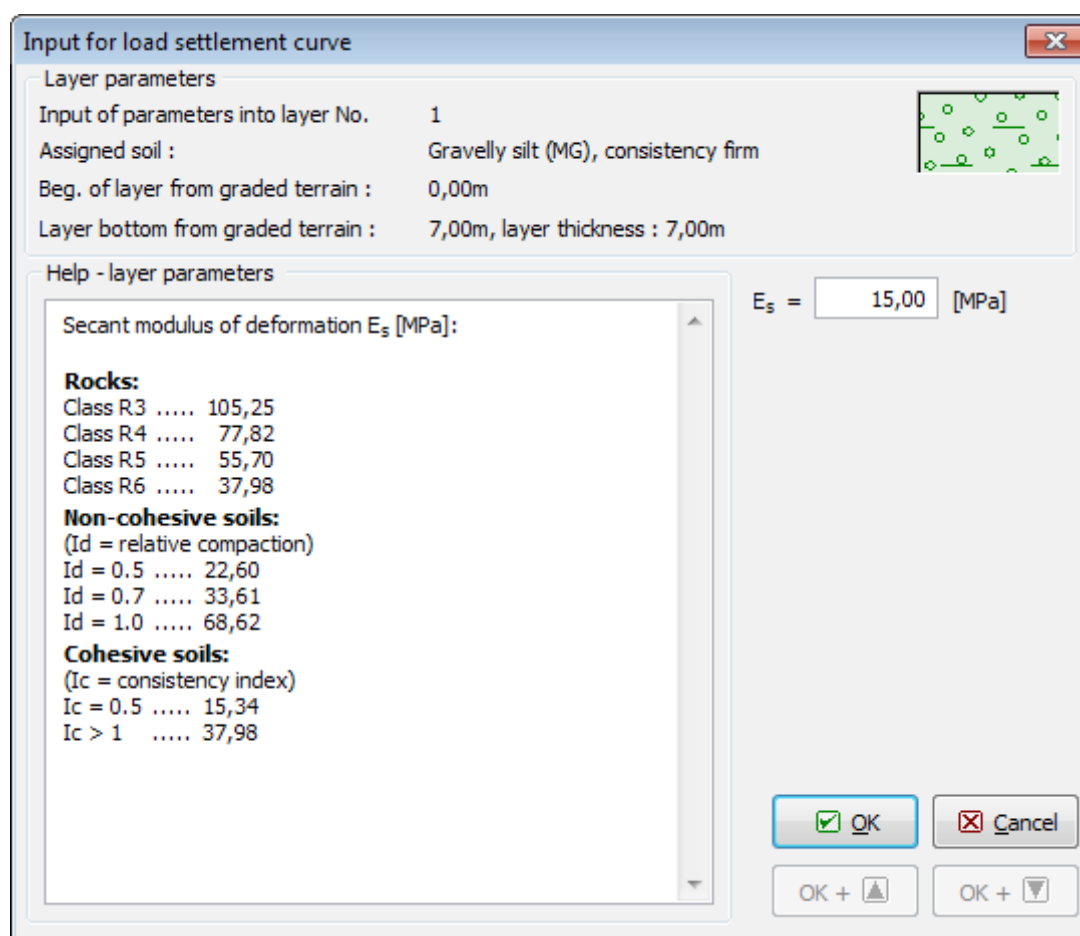
$h$ (m)	$d$ (m)								
	0.6			1.0			1.5		
	$I_d$								
	0.5	0.7	0.9	0.5	0.7	0.9	0.5	0.7	0.9
1.5	11.0	13.7	28.3	12.8	15.8	30.6	13.0	15.3	29.0
3.0	15.5	20.2	44.5	18.4	25.0	47.8	19.4	24.5	52.5
5.0	18.8	26.6	56.1	22.8	32.5	69.1	24.5	36.0	78.2
10.0	23.8	36.6	72.1	29.8	47.8	93.4	32.6	54.0	107.3

**Secant modulus of soil  $E_S$  located in cohesive soils**

$h$ (m)	$d$ (m)					
	0.6		1.0		1.5	
	$I_c$					
	0.5	$\geq 1.0$	0.5	$\geq 1.0$	0.5	$\geq 1.0$
1.5	6.9	13.2	7.9	13.4	8.6	12.3
3.0	10.0	22.0	12.5	23.9	13.7	23.0
5.0	12.5	31.2	15.9	35.4	18.4	36.7
10.0	15.5	44.3	21.3	51.3	24.6	57.4

The dialog window for entering the secant modulus  $E_S$  can be displayed in the frame "Settlement" using the "Edit  $E_S$ " button. When editing the window displays the recommended values of the secant modulus of soil  $E_S$ .





Dialog window "Input for load settlement curve" - Secant modulus of soil  $E_s$  [MPa]

#### Literature:

[1] CSN 73 1002: Pilotove zaklady. Praha, UNM, 1988, 28 p.

[2] CSN 73 1004: Velkopruerove piloty. Praha, UNM, 1981, 56 p.

[3] Masopust, J., Glisnikova, V.: Zakladani staveb Modul M01. 1<sup>st</sup> edition, Brno, AN CERM, 2007, 182 p., ISBN 978-80-7204-538-9.

[4] Pochman, R., Simek, J.: Pilotove zaklady - Komentar k CSN 73 1002. 1<sup>st</sup> edition, Prague, Vydavatelstvi norem, 1989, 80 p.

## Settlement-Influence Factor $I_s$

The settlement-influence factor depends on the depth of pile below the surface of a resistant layer  $D$  and the pile diameter  $d$ . The settlement-influence factor of pile  $I_s$  is given by:

$$I_s = I_0 \cdot R_k \cdot R_h$$

where:

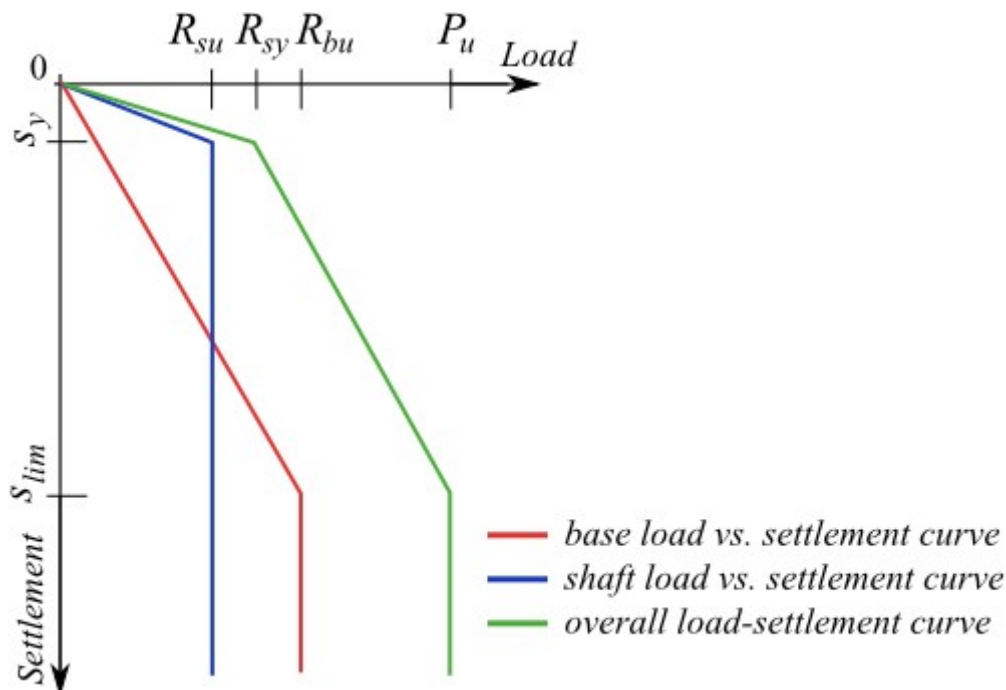
- $I_0$  - base settlement-influence factor
- $R_k$  - correction factor for pile compressibility
- $R_h$  - correction factor for finite depth of layer on a rigid base

## Linear Theory (Poulos)

Analysis of the load-settlement curve of **single pile** or **pile group** is based on the solution described in the book **Pile Foundations Analysis and Design** (H. G. Poulos et. E. H. Davis, 1980) and is based on the theory of elasticity and modifications attributed to in-situ measurements. Foundation soil is therefore characterized by the modulus of elasticity  $E$  and by the Poisson's ratio  $\nu$ . This method allows the construction of the load-settlement curve for **pile foundations** (single pile, pile group).

The basic input parameters of the analysis are pile base bearing capacity  $R_{bu}$  and pile skin bearing capacity  $R_{su}$ . Ultimate bearing capacity of pile foundation, respectively ultimate load is given by equation  $P_u = R_{su} + R_{bu}$ . These values are obtained by the program from the analysis of vertical bearing capacity of **single pile** or **pile group** and it depends on the selected method of analysis. All partial factors of the analysis are assumed equal to 1.0 so that the resulting resistance is greater than the one obtained from actual bearing capacity analysis.

During the analysis of settlement of **single pile** or **pile group** according to Poulos method (1980) program don't consider **influence of additional compression of the pile shaft** - that is why displacement of pile material is neglected.



Constructing of load-settlement curve of pile foundation according to Poulos method

Literature:

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 5, pp. 71 - 108.

## Settlement of Piles According to Poulos

The basic assumption of the analysis is the determination of the **load at the shaft resistance activation**  $R_{sy}$ . At this point the shaft resistance no longer increases, further load is taken by the pile base only. This force is given by equation:

$$R_{sy} = \frac{R_s}{1 - \beta}$$

where:  $R_s$  - pile shaft resistance [N]  
 $\beta$  - proportion of applied load transferred to pile base [-]

The **proportion of applied load transferred to pile base**  $\beta$  is provided by:

$$\beta = \beta_0 \cdot C_k \cdot C_b \cdot C_v$$

where:  $\beta_0$  - base-load proportion for incompressible pile  
 $C_k$  - correction factor for pile compressibility  
 $C_v$  - correction factor for Poisson's ratio of soil  
 $C_b$  - correction factor for stiffness of bearing stratum

The corresponding value of **settlement**  $s_y$  **at shaft resistance activation**  $R_{sy}$  is given by:

$$s_y = \frac{I \cdot R_{sy}}{d \cdot E_s}$$

where:  $I$  - settlement-influence factor [-]  
 $E_s$  - average value of **secant modulus of soil** along the pile shaft [MPa]  
 $d$  - pile diameter [m]  
 $R_{sy}$  - load at the shaft resistance activation [N]

The **settlement-influence factor**  $I$  is given by:

$$I = I_0 \cdot R_k \cdot R_b \cdot R_v$$

where:  $I_0$  - **basic settlement-influence factor**  
 $R_k$  - **correction factor for pile compressibility**  
 $R_b$  - **correction factor for stiffness of bearing stratum**  
 $R_v$  - **correction factor for Poisson's ratio of soil**

The **overall limit settlement**  $s_{lim}$  is provided by:

$$s_{lim} = \frac{I \cdot R_{bu}}{\beta \cdot d \cdot E_s}$$

where:  $I$  - settlement-influence factor [-]  
 $R_{bu}$  - ultimate pile base bearing capacity [N]  
 $\beta$  - proportion of applied load transferred to pile base [-]  
 $d$  - pile diameter [m]  
 $E_s$  - average value of **secant modulus of soil** along the pile shaft [MPa]

**Literature:**

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and

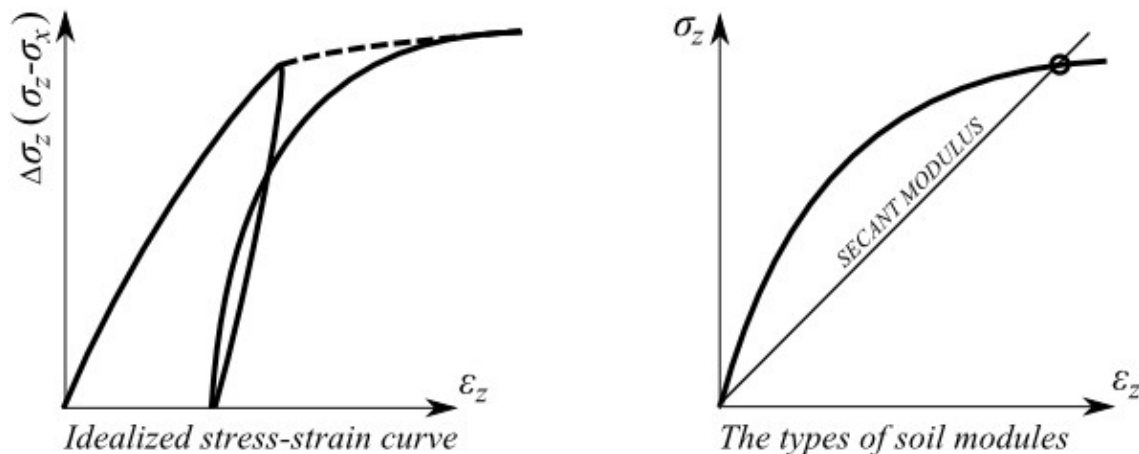
*Sons, 1980, chapters 5.3 and 5.4, pp. 84 - 100.*

## Secant Modulus of Soil $E_s$

In the literature, it appears double marking of soil modulus  $E_s$ . According to the Poulos et. Davis (1980) this parameter is referred as the **modulus elasticity of soil (Young's modulus)**, while Briaud (2001) and Gopal Ranjan (2000) named this parameter as the **secant modulus of soil**. Both titles of this modulus  $E_s$  has the same meaning. However, the soil behaves elastically only in the field of small strains (generally it is a heterogeneous material), and thus is more appropriate to speak rather about the **secant modulus of soil  $E_s$** .

**Modulus of elasticity of soil  $E$**  is obtained from the deviator stress-axial strain curve. The undrained modulus,  $E_u$  is obtained from the undrained triaxial test data while the drained modulus  $E_d$  is obtained from the drained test conditions.

At the initial stage of the stress-strain curve is nearly linear dependence, but elastic strain of soils is a very small due to overall value of the strain. There are defined several types of modules - **tangent modulus of soil**, **secant modulus of soil** and **initial tangent modulus**. The introduction of this simplifying assumption is possible to use the theory of elasticity for detecting of stress-strain state in soils.



*Distribution of idealized stress-strain curve and determination of individual types of soil modules*

**Secant modulus of soil  $E_s$**  is defined as the ratio of difference in deviator of normal stress to the corresponding axial strain of soil according to the following equation:

$$E_s = \frac{\Delta(\sigma_1 - \sigma_3)}{\Delta \varepsilon_E}$$

Lambe et. Whitman (1969) say that the elastic modulus for a soil is usually the secant modulus from zero deviator of normal stress to a deviator stress equal to one-half or one-third of the peak deviator stress.

The secant modulus  $E_s$  decreases as the strain level increases because the stress-strain curve has a downward curvature. There are three means of obtaining this parameter:

- laboratory triaxial tests (from calculation based on the tangent modulus of soil)
- pile-load test
- empirical correlations based on previous experience

Typical range of values for the static stress-strain (secant) modulus  $E_S$  for selected soils - field values depend on stress history, water content, density (Gopal Ranjan et. Rao, 2000):

Type of soil	Consistency or Density of soil	Modulus $E_S$ [MPa]
Silt	Very soft	0.2 - 2
Clay	Very soft	2 - 15
	Soft	5 - 25
	Firm, medium	15 - 50
	Hard	50 - 100
	Sandy	25 - 250
Loess sand	Silty	7 - 21
	Loose	10 - 24
	Dense	48 - 80
Sand and gravel	Loose	50 - 145
	Dense	100 - 190

*Literature:*

*Briaud, J.-L.: Introduction to Soil Moduli. Geotechnical News, June 2001, BiTech Publishers Ltd, Richmond, B.C., Canada.*

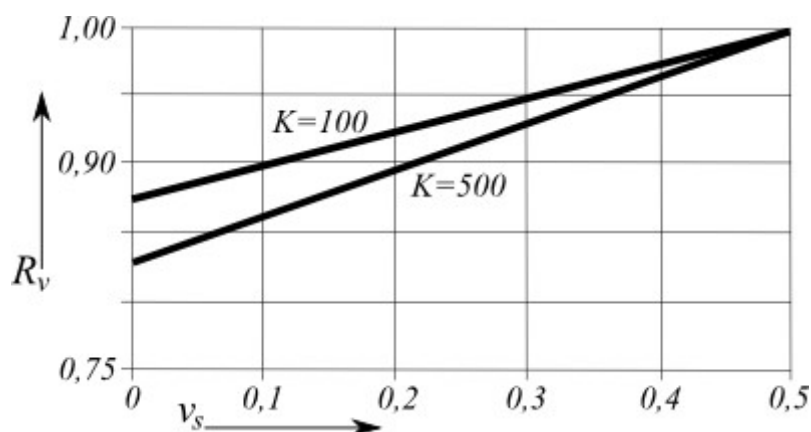
*Gopal Ranjan et. A. S. R. Rao: Basic and Applied Soil Mechanics. New Age International, 2000, chapter 10.11, pp. 328 - 330. ISBN: 8122412238, 9788122412239.*

*Lambe, T. W. et. Whitman, V. R.: Soil Mechanics. New York: John Wiley and Sons, 1969, 576 p. ISBN: 978-0-471-51192-2.*

*Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.5, pp. 101 - 104.*

## Correction Factor for Soil Poisson's Ratio $R_v$

The correction factor for the influence of Poisson's ratio  $R_v$  accounts for the influence of reduction of Poisson's ratio  $\nu$  of soils surrounding the pile on the values of pile settlement for constant modulus of elasticity of these soils. These values are generally presented as a function of Poisson's ratio of the surrounding soil  $\nu_s$  for various **pile-stiffness factor**  $K$ . These graphs are implemented in the program in a digital format.



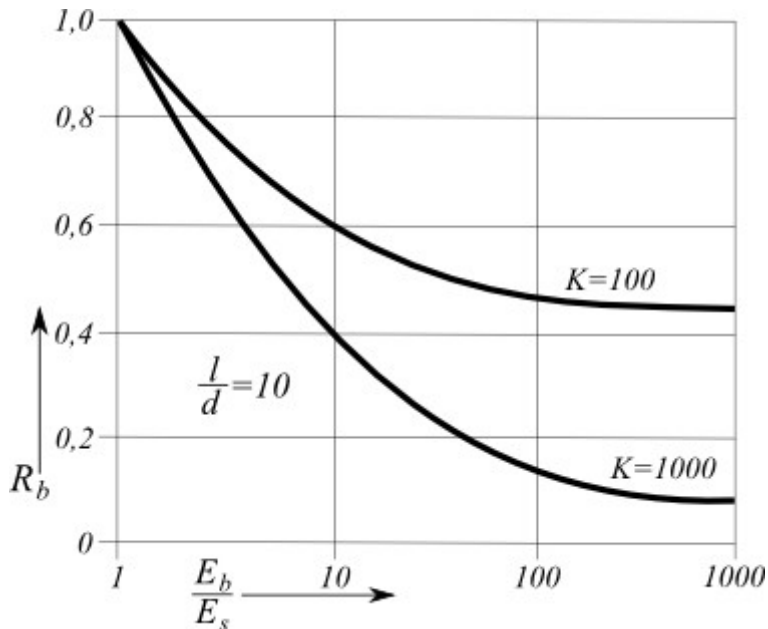
Poisson's ratio correction factor for settlement  $R_v$

Literature:

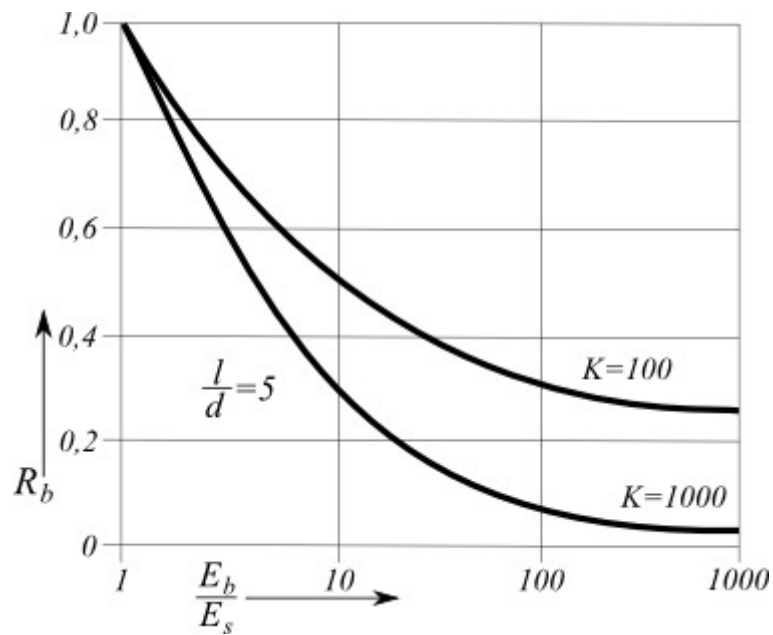
Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.21).

### Correction Factor for Stiffness of Bearing Stratum $R_b$

The values of the correction factor  $R_b$  are generally presented as a function of the ratio of modulus of elasticity of pile and secant modulus of soil at the pile base and the surrounding soil ( $E_b/E_s$ ) for various pile-stiffness factor  $K$  and various pile length to pile diameter ratios ( $l/d$ ). These graphs are implemented in the program in a digital format.



Base modulus correction factor for settlement  $R_b$  ( $L/d = 10$ )



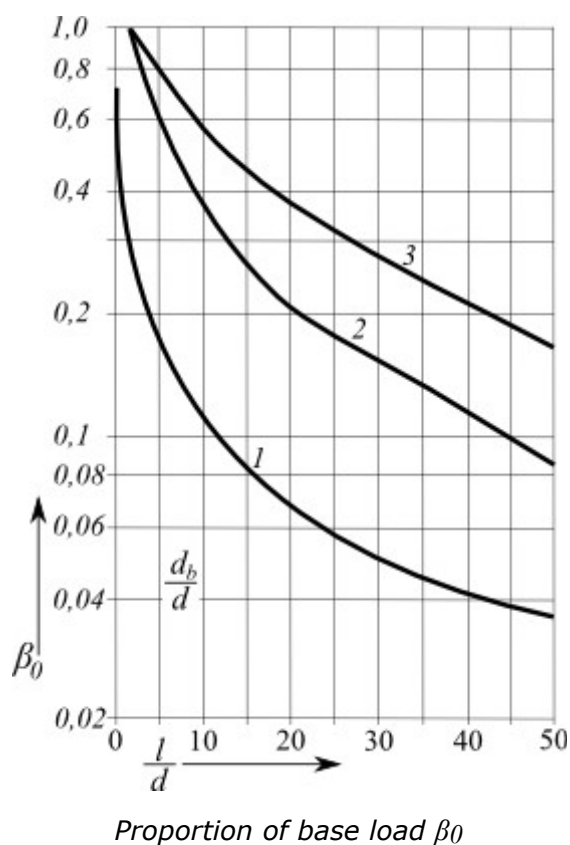
Base modulus correction factor for settlement  $R_b$  ( $L/d = 5$ )

Literature:

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 90 (figure 5.22).

## Base-Load Proportion for Incompressible Pile $\beta_0$

The base-load proportion for incompressible pile  $\beta_0$  represents the influence of compression of elastic half-space, which adopts the load transferred by the pile from incompressible soil. The values of this coefficient are generally presented as a function of the pile length to pile diameter ratio ( $l/d$ ) for various pile base diameter to pile diameter ratios ( $d_b/d$ ). These graphs are implemented in the program in a digital format.



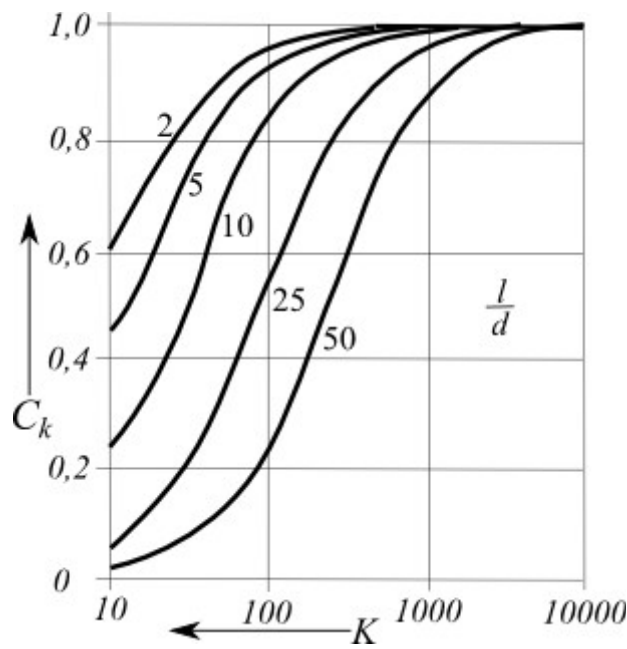
Literature:

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 86 (figure 5.11).

## Correction Factor for Pile Compressibility $C_k$

The values of the factor  $C_k$  are generally presented as a function of the **pile-stiffness factor  $K$**  for various pile length to pile diameter ratios ( $l/d$ ). These graphs are implemented in the program in a digital format.





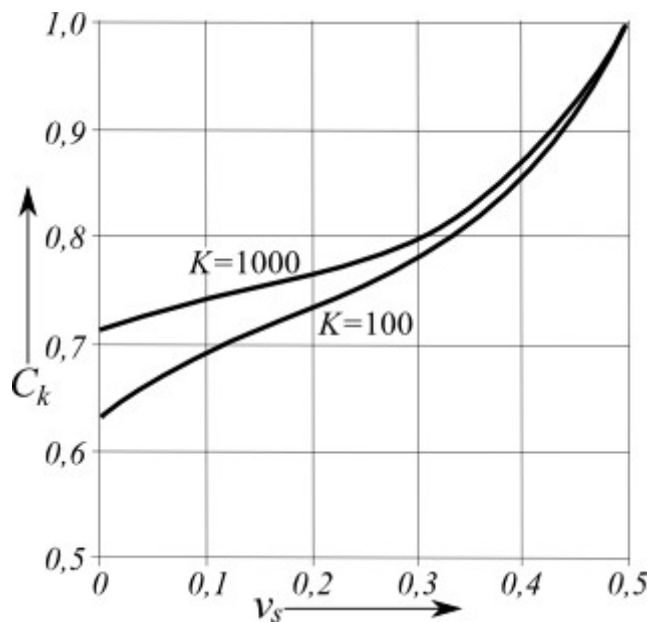
Compressibility correction factor for base load  $C_k$

Literature:

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 86 (figure 5.12).

### Correction Factor for Poisson's Ratio of Soil $C_v$

The values of the factor  $C_v$  are generally presented as a function of Poisson's ratio of the surrounding soil  $\nu_s$  for various pile-stiffness factor  $K$ . These graphs are implemented in the program in a digital format.



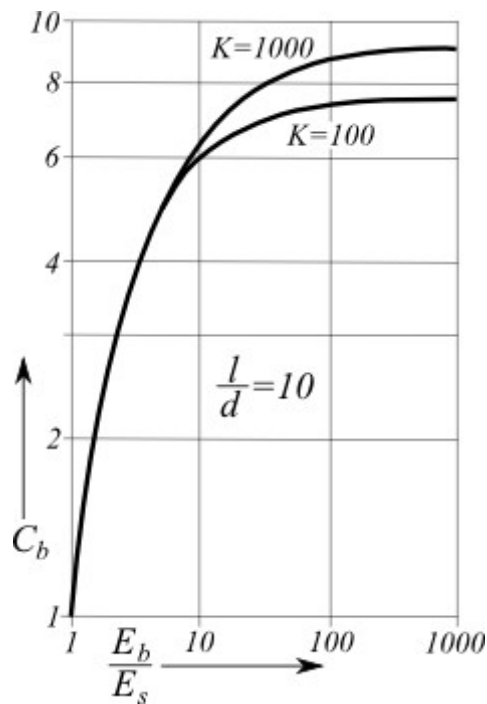
Poisson's ratio correction factor for base load  $C_v$

*Literature:*

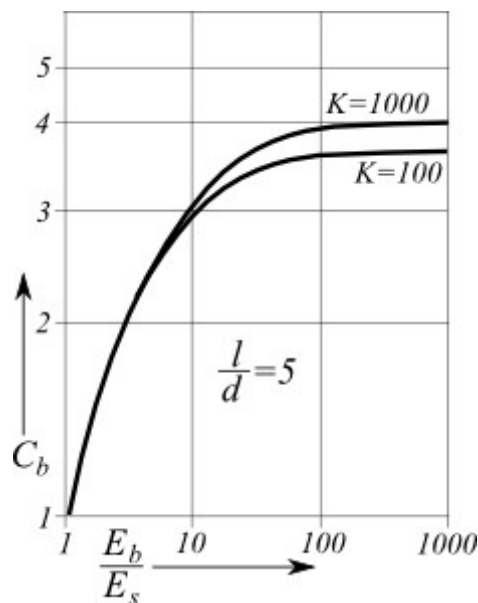
*Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 86 (figure 5.13).*

### Correction Factor for Stiffness of Bearing Stratum $C_b$

The values of the factor  $C_b$  are generally presented as a function of the ratio of modulus of elasticity of pile and **secant modulus of soil** at the pile base and the surrounding soil ( $E_b/E_s$ ) for various **pile-stiffness factor**  $K$  and various pile length to pile diameter ratios ( $l/d$ ). These graphs are implemented in the program in a digital format.



Base modulus correction factor for base load  $C_b$  ( $L/d = 10$ )



*Base modulus correction factor for base load  $C_b$  ( $L/d = 5$ )*

*Literature:*

*Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 87 - 88 (figure 5.14).*

## **Pile-Stiffness Factor K**

The pile-stiffness factor is defined as:

$$K = \frac{E_p \cdot R_a}{E_s}$$

where:  $E_p$  - elastic modulus of pile material [MPa]

$E_s$  - average value of **secant modulus of soil** along the pile shaft [MPa]

$R_a$  - ratio of area of pile section to area bounded by pile outer-circumference [-]

$$R_a = \frac{A_1}{A_2}$$

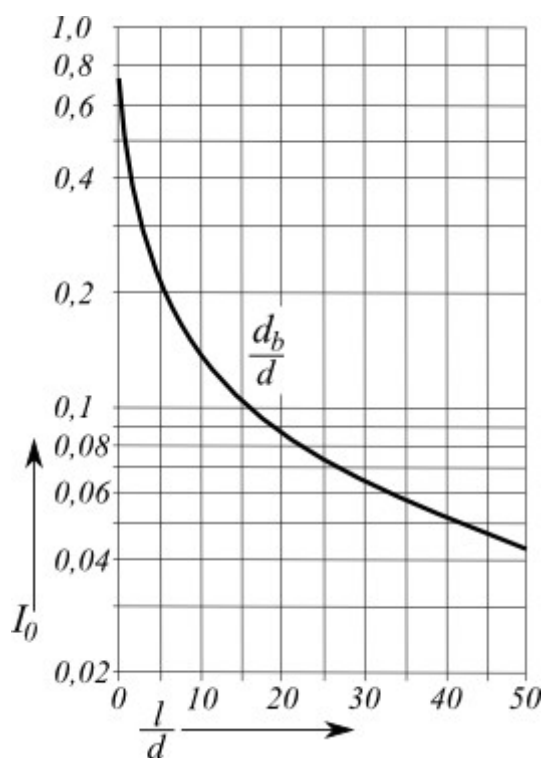
where:  $A_1$  - average area of cross-section of pile [ $m^2$ ]

$A_2$  - area of pile shaft [ $m^2$ ]

(for stiff piles  $R_a = 1$ )

## **Basic Settlement-Influence Factor $I_o$**

The basic settlement-influence factor  $I_o$  depends on the pile length  $l$  and diameter  $d$  and the values of this coefficient are generally provided by the following graph also showing their ranges:



Basic settlement-influence factor  $I_0$

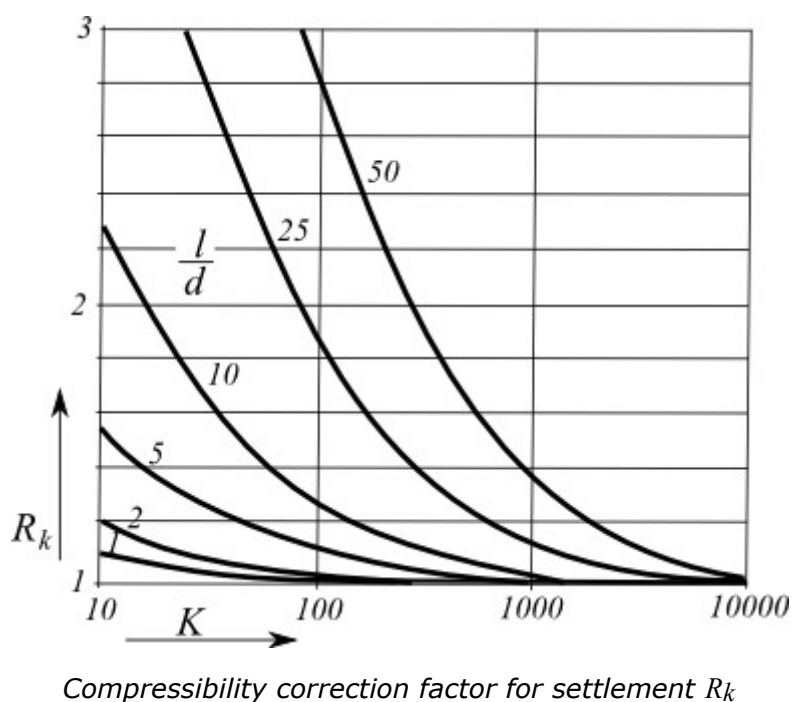
These graphs are implemented in the program in a digital format.

*Literature:*

*Poulos, H. G. et. Davis, E. H.: Pile Foundations Analysis and Design. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.18).*

## Correction Factor for Pile Compressibility $R_k$

The correction factor  $R_k$  represents the pile stiffness in dependence on the **pile-stiffness factor**  $K$  for various ratios of the pile length to pile diameter ( $l/d$ ). Its values are provided by the following graphs, that are implemented in the program in a digital format.

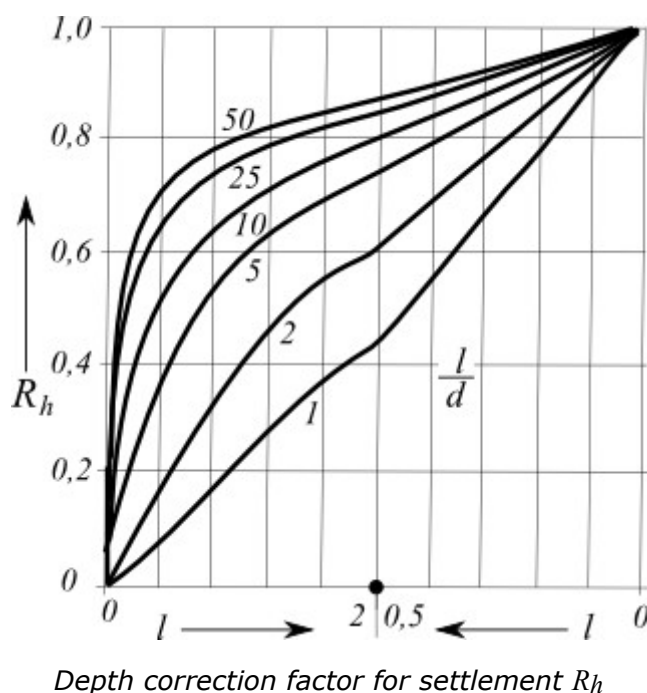


Literature:

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.19).

## Correction Factor for Finite Depth of Layer on a Rigid Base $R_h$

The correction factor  $R_h$  represents the influence of incompressible layer below the pile base. Its values are presented in the literature graphically for various pile length to pile diameter ratios ( $l/d$ ) and ratios of pile length to thickness of compressible layer above the incompressible layer ( $l/h$  or  $h/l$ ). These graphs are implemented in the program in a digital format.



Literature:

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 5.3.3, pp. 89 (figure 5.20).

## Horizontal Bearing Capacity - Elastic Subsoil (p-y Method)

### Horizontal bearing capacity of a pile, dimensioning

The horizontally loaded pile is analyzed using the finite element method as a beam on elastic Winkler foundation. The soil parameters along the pile are represented by the modulus of subsoil reaction. By default the pile is subdivided into 30 segments. For each segment the program determines the values of the modulus of subsoil reaction, internal forces and deformation (displacements). The program also allows for dimensioning of the **steel-reinforced concrete pile** based on the method specified in the frame "Settings" and on the parameters input in the "Piles" tab.

The program also enables to analyze a pile loaded by the **prescribed displacements** (translation or rotation of the pile head). In such a case the analysis is carried out only with the prescribed displacement. The input mechanical load is excluded.

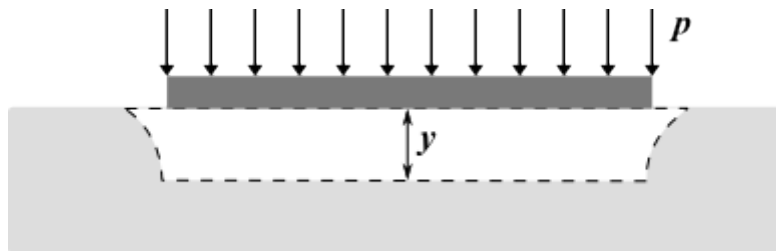
The following options for inputting the **modulus of subsoil reaction** are available in the program:

- **by distribution** (distribution of the modulus of subsoil reaction along the pile is specified)
- constant distribution
- linear distribution (Bowles)
- according to CSN 73 1004
- according to Matlock and Reese
- according to Vesic

In general, the modulus of subsoil reaction corresponds to the spring stiffness in the Winkler model. This model describes settlement of a rigid slab as a function of the applied load. The corresponding relationship is represented by the following formula:

$$p = ky$$

where:  $p$  - load acting along slab-soil interface  
 $k$  - stiffness of Winkler spring  
 $y$  - translation (displacement) of slab into subsoil



*Definition of the modulus of subsoil reaction*

## Constant Distribution of Modulus of Subsoil Reaction

The modulus of subsoil reaction of the  $i^{th}$  layer is provided by:

$$k_h = \frac{3E_{def}}{2r}$$

where:  $E_{def}$  - deformation modulus of soil [MPa]  
 $r$  - reduced width of pile [m], which is given by equation:

$$r = d + 2d \tan \beta$$

where:  $d$  - pile diameter [m]  
 $\beta$  - angle of dispersion - is input with respect to the angle of internal friction in the range of  $\varphi/4 \div \varphi$

*Literature:*

*Pochman, R., Simek, J.: Pilotove zaklady - Komentar k CSN 73 1002. 1<sup>st</sup> edition, Prague, Vydavatelstvi norem, 1989, 80 p.*

## Linear Modulus of Subsoil Reaction

The modulus of subsoil reaction at a depth  $z$  is given by equation:

$$k_h = k \left( 0,308 + 1,584 \frac{d}{l} \right) \frac{z}{rl}$$

where:  $d$  - pile diameter [m]  
 $l$  - length of pile [m]  
 $k$  - soil parameter (modulus) according to Bowles [MN/m<sup>3</sup>]

$r$  - reduced width of pile [m], which is given by equation:

$$r = d + 2d \tan \beta$$

where:  $d$  - pile diameter [m]

$\beta$  - angle of dispersion - is input with respect to the angle of internal friction in the range of  $\varphi/4 \div \varphi$

**Representative range of values of lateral modulus  $k$  [ $MN/m^3$ ] according to Bowles:**

dense sandy gravel	220 - 400
medium dense gravel	155 - 300
medium-graded sand	110 - 280
fine sand	80 - 200
stiff clay	60 - 220
saturated stiff clay	30 - 110
plastic clay	40 - 140
saturated plastic clay	10 - 80
soft clay	2 - 40

*Literature:*

Bowles, J. E.: *Foundations Analysis and Design*. 5<sup>th</sup> edition, New York: McGraw-Hill Book Company, 1997, ISBN 0-07-118844-4, chapter 16-15.2, s. 941 (table 16-4).

Pochman, R., Simek, J.: *Pilotové zaklady - Komentář k CSN 73 1002*. 1<sup>st</sup> edition, Prague, Vydavatelství norem, 1989, 80 p.

## Modulus of Subsoil Reaction According to CSN 73 1004

The modulus of subsoil reaction for **cohesive soil** assumes the form:

$$k_n = \frac{2 E_{def}}{3d}$$

where:  $E_{def}$  - deformation modulus of soil [MPa]

$d$  - pile diameter [m]

For **cohesionless soil** it is given by:

$$k_n = n_h \frac{z}{d}$$

where:  $n_h$  - modulus of horizontal compressibility [ $MN/m^3$ ]



- $d$  - pile diameter [m]  
 $z$  - depth of a given section from finished grade [m]

**Approximate values of modulus of horizontal compressibility  $n_h$  for cohesionless soils:**

Soil	$n_h$ [MN/m <sup>3</sup> ]		
	0.3	0.5	0.9
Relative density of soil $I_D$ [-]	0.3	0.5	0.9
Dry sand and gravel	2.5	7.0	18.0
Wet sand and gravel	1.5	4.5	11.0

*Literature:*

CSN 73 1004: Velkoprumerove piloty. Praha, UNM, 1981, 56 p.

Masopust, J.: Vrtane piloty. 1<sup>st</sup> edition, Prague, Cenek a Jezek, 1994, 263 p.

## Modulus of Subsoil Reaction According to Matlock and Reese

This method is applicable for **cohesionless soils**. The modulus of subsoil reaction is given by equation:

$$k_h = n_h \frac{z}{d}$$

- where:  $n_h$  - modulus of horizontal compressibility [MN/m<sup>3</sup>]  
 $d$  - pile diameter [m]  
 $z$  - depth of a given section from finished grade [m]

**Approximate values of modulus of horizontal compressibility  $n_h$  for cohesionless soils:**

Soil	$n_h$ [MN/m <sup>3</sup> ]
- density	
Dry sand and gravel	
- loose	1.8 - 2.2
- medium dense	5.5 - 7.0
- dense	15.0 - 18.0
Wet sand and gravel	
- loose	1.0 - 1.4
- medium dense	3.5 - 4.5
- dense	9.0 - 12.0

*Literature:*

Reese, L. C. et. Matlock, H.: *Non-Dimensional Solutions for Laterally Loaded Piles with Soil Modulus Assumed Proportional to Depth*. University of Texas, Austin, 1956.

Reese, L. C. et. Matlock, H.: *Generalized Solutions for Laterally Loaded Piles*. *Journal of the Soil Mechanics and Foundations Division, ASCE* 86, No. 5, 1960, pp. 63 - 91.

Reese, L. C. et. Matlock, H.: *Foundation analysis of offshore pile-supported structures*. *Proceedings of the 5<sup>th</sup> International Conference, ISSMFE, Paris, Vol. 2, 1961, pp. 91-7*

## Modulus of Subsoil Reaction According to Vesic

The modulus of subsoil reaction is provided by:

$$k_h = \frac{0,65}{d} \sqrt[12]{\frac{E_s d^4}{E_p I_p} \frac{E_s}{1-\nu^2}}$$

where:

- $E_p$  - modulus of elasticity of pile [MPa]
- $I_p$  - moment of inertia of pile [m<sup>4</sup>]
- $E_s$  - modulus of elasticity of soil [MPa]
- $d$  - pile diameter [m]
- $\nu$  - Poisson's ratio [-]

*Literature:*

Poulos, H. G. et. Davis, E. H.: *Pile Foundations Analysis and Design*. New York: John Wiley and Sons, 1980, chapter 8.2.3, pp. 174 (equation 8.43).

Vesic, A. S.: *Bending of Beams Resting on Isotropic Elastic Solid*. *JSMFD, ASCE, vol. 87, 1961, EM 2: pp. 35 - 53*.

Vesic, A.S.: *Design of Pile Foundations*. National Cooperative Highway Research Program Synthesis 42, Transportation Research Board, Washington D.C., 1977.

## Pile Horizontal Bearing Capacity - Brom's Method

Analysis of a single pile according to Broms is described in Broms, 1964. This method exclusively assumes a pile in the **homogeneous soil**. Thus the analysis method does not allow for layered subsoil. The type of analysis of the pile horizontal bearing capacity is specified in the "**Settings**" frame, tab "**Piles**".

When adopting the Broms method for the analysis of horizontal bearing capacity the program disregards up now input soil layers. The soil parameters are specified in the "**Horizontal bearing capacity**" frame based on the **type of soil** (cohesive, cohesionless).

The input parameters for the analysis of pile horizontal bearing capacity are the **pile material characteristics** (modulus of elasticity and strength of a given material), **pile geometry** (pile length  $l$  and its diameter  $d$ ) and also the **pile load** due to shear force and bending moment.

The coefficient of pile stiffness  $\beta$  for cohesive soils is given by:

$$\beta = \frac{k_h d}{4 E I}$$

where:  $E*I$  - bending stiffness of pile section [ $MNm^2$ ]

$k_h$  - modulus of subsoil reaction [ $MNm^3$ ]

$d$  - diameter of a single pile [ $m$ ] - in case of a pile with a **circular variable cross-section** the calculation of parameter  $\beta$  assumes a constant value of the pile diameter  $d_l$  input in the "**Geometry**" frame

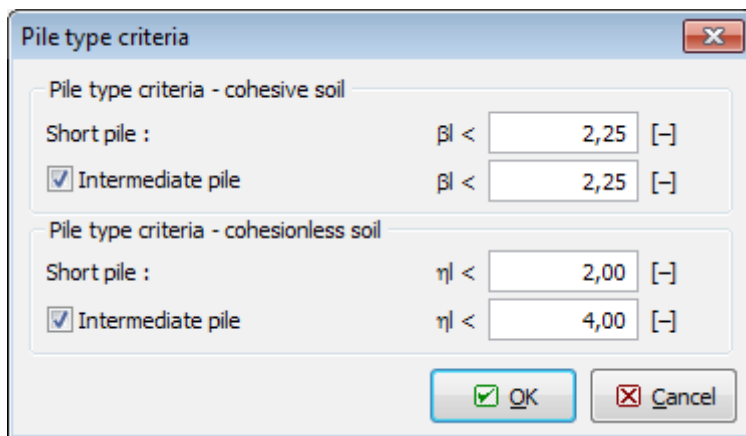
The coefficient of pile stiffness  $\eta$  for cohesionless soils follows from:

$$\eta = \left( \frac{n_h}{E I} \right)^{\frac{1}{5}}$$

where:  $E*I$  - bending stiffness of pile section [ $MNm^2$ ]

$n_h$  - coefficient of soil modulus variation [ $MNm^3$ ]

The program automatically determines whether to consider a long or a short pile based on the ratios  $\beta*l$  (for **cohesive soils**) and  $\eta*l$  (for **cohesionless soils**), respectively. Because literature offers different criteria for different types of piles, the program allows the user to define them. For an **intermediate** pile length the verification analysis considers both short and long piles and then program automatically chooses the result with the lowest value of the pile horizontal bearing capacity  $Q_u$ .



Dialog window "Pile type criteria"

**Type of pile criteria** (long, short, medium) are considered according to the following conditions:

- **free head**: for long piles it holds  $\beta * l > 2,5$  ; for short piles then  $\beta * l < 2,5$
- **restrained**: for long piles it holds  $\beta * l > 1,5$  ; for short piles then  $\beta * l < 1,5$

**Type of pile** (pile head support) can be considered in two ways:

- **free head** - rotation at pile head is not constrained
- **restrained** - pile is constrained against rotation at its head. In such cases we typically deal with piles that are part of a planar pile grid or a **pile group**.

Another important input parameter is the **flexure bearing capacity**. This quantity is automatically back calculated by the program using the following relation:

$$M_u = \gamma_k f W_y$$

- where:
- $W_y$  - section modulus of the pile section [ $m^3$ ]
  - $f$  - strength of the pile material [ $MPa$ ]
  - $\gamma_k$  - reduction coefficient of cross-section strength [-] - the cross-section bearing capacity is according to different standards pre-multiplied by different safety coefficients. This coefficient enables to adapt the program to these standards.

In case of a **steel-reinforced concrete pile** the flexure bearing capacity  $M_u$  depends on the amount of designed steel.

The reduction coefficient of bearing capacity  $\gamma_{Qu}$  reduces the overall magnitude of the **single pile horizontal bearing capacity** as:

$$Q_{u,red} = \frac{Q_u}{\gamma_{Qu}}$$

- where:
- $Q_u$  - horizontal bearing capacity of a single pile [ $kN$ ]
  - $\gamma_{Qu}$  - reduction coefficient of bearing capacity [-]

The result of an analysis is horizontal bearing capacity of a single pile  $Q_u$ , respectively  $Q_{u,red}$  and displacement of a pile at the terrain surface  $u$ .

### Literature:

[1] BROMS, BENGT. B.: *Lateral Resistance of Piles in Cohesive Soils. Proceedings of the American Society of Civil Engineers, Journal of the Soil Mechanics and Foundations Division, Vol. 90, SM2, 1964.*

[2] BROMS, BENGT. B.: *Lateral Resistance of Piles in Cohesionless Soils. Proceedings of the American Society of Civil Engineers, Journal of the Soil Mechanics and Foundations Division, vol. 90 SM3, 1964.*

## Pile CPT

The program Pile CPT allows to verify the bearing capacity and settlement of a single pile or a group of piles based on the results of **penetration tests**.

The main objective is to determine the **toe and shaft bearing capacities**. This analysis can be carried according to the following standards and approaches:

- EN 1997-2
- NEN 6743
- LCPC (Bustamante)
- Schmertmann

For all methods the essential input parameters are dimensionless coefficients adjusting the magnitude of bearing capacity and shaft friction, respectively. Different notation of these parameters can be encountered in various publications. The following notation is used in program Pile CPT:

$\alpha_p$       -pile toe coefficient

$\alpha_s$       -shaft friction coefficient

These coefficients are automatically calculated based on the type pile and the surrounding soil - these parameters can be, however, also specified manually ( $\alpha_p$  can be entered in the "Geometry" input mode,  $\alpha_s$  as a soil parameter).

When analyzing rectangular piles the **pile shape coefficient**  $s$  is introduced to reduce the toe bearing capacity. When analyzing piles with enlargement the expanded **pile toe coefficient**  $\beta$  is introduced to adjust the expanded toe bearing capacity. When calculating the toe bearing capacity the program automatically accounts for the influence of the change of terrain elevation.

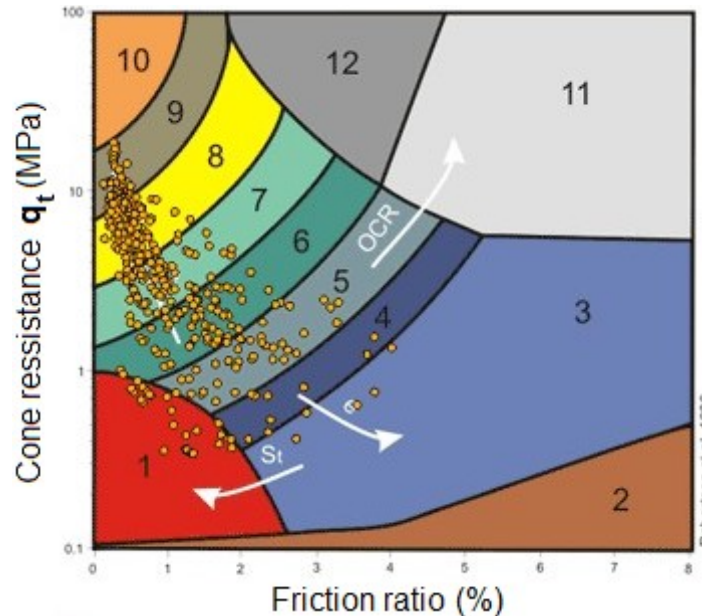
The program allows for the calculation of **load-settlement curve** and **pile settlement** for a given load. This analysis adopts the values of calculated toe and shaft bearing capacities and follows the NEN 6743 standard. A **negative skin friction** can also be taken into account when calculating pile settlement.

**Verification** of a pile bearing capacity depends on the verification methodology selected in the "Pile CPT" tab.

## Classification of Soils According to Robertson

During classification of soils according to Robertson (1986 or 2010) it's not necessary to **input** parameters of **soils**, the program performs this step automatically with their assignment to the geological **profile**. For this reason, the **assessment** of the performed **CPT** is very fast and especially clear.

Classification of soils according to Robertson (1986 or 2010) is based on the measured values of penetration resistance  $q_c$  [MPa], local skin friction  $f_s$  [kPa], pore pressure  $u_2$  [kPa] respectively. Based on the **corrected value of the cone resistance**  $q_t = q_c + u_2 * (1 - a)$ , or percentage ratio  $q_c/p_a$  and friction ratio  $R_f = f_s/q_t$  program automatically performs the assignment of soil behavior type (SBT) according to the following graphs.  $p_a$  - atmospheric pressure = 100 kPa (= 1 tsf).



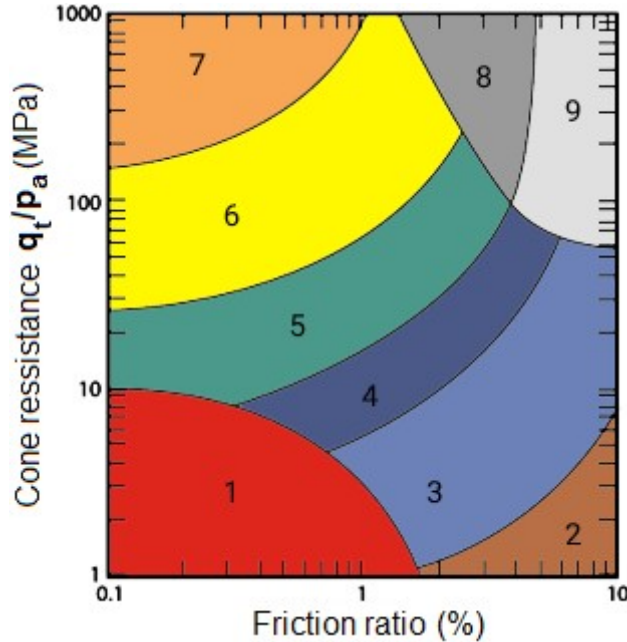
Non-normalized CPT Soil Behavior Type (SBT) chart according to Robertson, 1986 (source: Robertson et al., 1986)

**Soil classification according to Robertson, 1986 (source: Robertson et al., 1986)**

Zone	Soil Behavior Type (SBT)
1	Sensitive fine grained
2	Organic material
3	Clay
4	Silty Clay to clay
5	Clayey silt to silty clay
6	Sandy silt to clayey silt
7	Silty sand to sandy silt
8	Sand to silty sand
9	Sand
10	Gravelly sand to sand
11	Very stiff fine grained *

12	Sand to clayey sand *
----	-----------------------

\* Overconsolidated or cemented soil



*Non-normalized CPT Soil Behavior Type (SBT) chart according to Robertson, 2010 (source: [6], Figure 21, pp. 26)*

**Soil classification according to Robertson, 2010 (source: [6], Figure 21, pp. 26)**

Zone	Soil Behavior Type (SBT)
1	Sensitive, fine grained
2	Organic soils - clay
3	Clay - silty clay to clay
4	Silt mixtures - clayey silt to silty clay
5	Sand mixtures - silty sand to sandy silt
6	Sands - clean sand to silty sand
7	Gravelly sand to dense sand
8	Very stiff sand to clayey sand *
9	Very stiff fine grained *

\* Heavily overconsolidated or cemented

A newer classification of soils according to Robertson (2010) contains a smaller number of individual classes of soils than the original soil classification from 1986. However, the classification of soils according to Robertson (2010) is now more accurate and more used in the world.

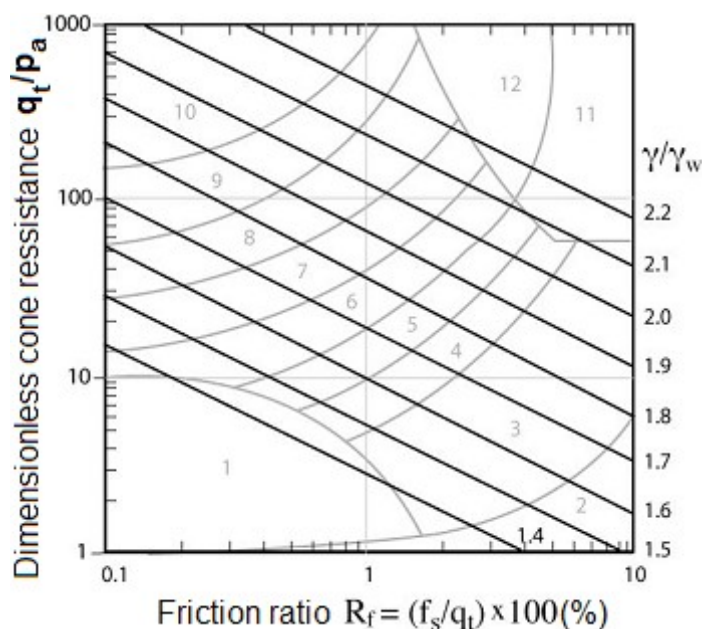
If it's chosen option **"calculate"** for unit weight of soil in the frame **"Soil Classification"**, then the **unit weight of soil**  $\gamma$  [ $kN/m^3$ ] is determined by the following formula:

$$\frac{\gamma}{\gamma_w} = 0,27 (\log R_f) + 0,36 \left( \log \frac{q_t}{p_a} \right) + 1,236$$

where:  $\gamma_w$  unit weight of water ( $\approx 10$ ) [ $kN/m^3$ ]  
 $p_a$  atmospheric pressure ( $\approx 100$ ) [ $kPa$ ]

$$R_f = \frac{f_s}{q_t} 100 \%$$

where:  $R_f$  friction ratio between skin friction and cone resistance



Dimensionless soil unit weight  $\gamma/\gamma_w$  based on CPT tests (source: [6], Figure 28, pp. 36)

Input of the thickness of soil layers influences what is the minimum thickness of layer of the  $i^{th}$  soil. In the case of the **zero layer of soil** there are assigned all layers of soils based on the soil classification according to Robertson (1986 or 2010) into geological **profile**.

When is input a **non-zero minimum thickness of layer**, then the number of layers of soil is reduced in the geological **profile**. The layout and number of soil layers affects the **vertical bearing capacity** and **settlement of piles** investigated by **CPT**.

*Literature:*

[1] EN ISO 22476-1: Geotechnical investigation and testing - Field testing. Part 1: Electrical cone and piezocone penetration test, 2013.  
 [2] EN ISO 22476-3: Geotechnical investigation and testing - Field testing. Part 3: Standard penetration test, 2005.  
 [3] EN ISO 22476-4: Geotechnical investigation and testing - Field testing. Part 4: Menard pressuremeter test, 2005.



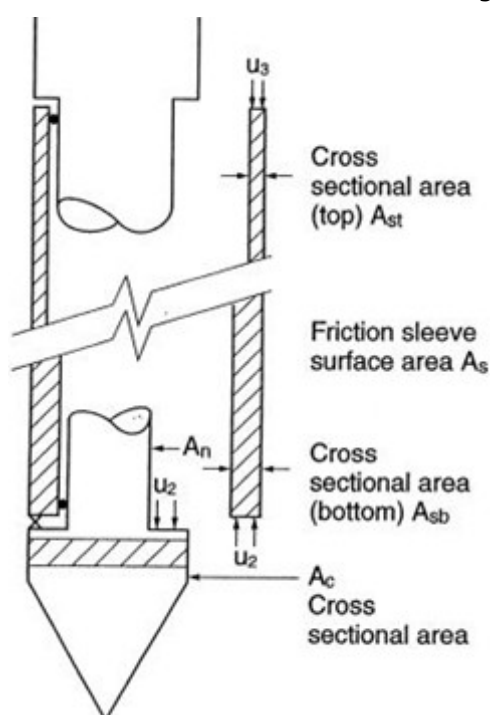
[4] EN ISO 22476-12: Geotechnical investigation and testing - Field testing. Part 12: Mechanical cone penetration test (CPTM), 2009.

[5] Robertson, P. K.: Interpretation of Cone Penetration Tests – a unified approach. Canadian Geotechnical Journal, 2009, No. 46, pp. 1337 – 1355.

[6] Robertson, P. K. and Cabal, K. L.: Guide to Cone Penetration Testing for Geotechnical Engineering. Gregg Drilling & Testing, Inc., USA, 6<sup>th</sup> edition, 2014, 133 p.

## Coefficient of Penetrometer (Net Area Ratio)

This coefficient  $\alpha$  [-] represents the **net area ratio** which is determined from calibration measurement at laboratory (the effort to eliminate the adverse effects of friction sleeve and unequal cone tip). Typical values of this coefficient are in the range from 0.7 to 0.85.



Unequal end area effects on cone tip and friction sleeve (source: [6], Figure 20, pp. 22)

Literature:

[1] EN ISO 22476-1: Geotechnical investigation and testing - Field testing. Part 1: Electrical cone and piezocone penetration test, 2013.

[2] EN ISO 22476-3: Geotechnical investigation and testing - Field testing. Part 3: Standard penetration test, 2005.

[3] EN ISO 22476-4: Geotechnical investigation and testing - Field testing. Part 4: Menard pressuremeter test, 2005.

[4] EN ISO 22476-12: Geotechnical investigation and testing - Field testing. Part 12: Mechanical cone penetration test (CPTM), 2009.

[5] Robertson, P. K.: Interpretation of Cone Penetration Tests – a unified approach. Canadian Geotechnical Journal, 2009, No. 46, pp. 1337 – 1355.

[6] Robertson, P. K. and Cabal, K. L.: Guide to Cone Penetration Testing for Geotechnical

Engineering. Gregg Drilling & Testing, Inc., USA, 6<sup>th</sup> edition, 2014, 133 p.

## Bearing Capacity

The maximum bearing capacity of a single pile based on the values of tip resistance  $q_c$  of the  $i$ -th cone penetration test is given by:

$$F_{max,i} = F_{max,toe,i} + F_{max,shaft,i}$$

where:

$F_{max,i}$	̄ maximum bearing capacity of the pile from $i^{th}$ CPT test
$F_{max,toe,i}$	̄ maximum toe resistance from $i^{th}$ CPT test
$F_{max,shaft,i}$	̄ maximum shaft resistance from $i^{th}$ CPT test

Providing there is  $n$  CPT tests then the bearing capacity of a single pile is obtained as an arithmetic average of  $n$  calculated bearing capacities:

$$F_{max} = \frac{\sum_{i=1}^n F_{max,i}}{n}$$

If performing the analysis according to the NEN 6743 standard then the approach for more CPT tests is different and follows directly the NEN 6743 standard (article 5.3.2.2).

**The maximum pile toe resistance**  $F_{max,toe}$  is provided by:

$$F_{max,toe} = A_{toe} \cdot p_{max,toe}$$

where:

$A_{toe}$	pile toe cross-sectional area
$p_{max, toe}$	maximum pressure at pile toe from CPT results

**The maximum shaft resistance**  $F_{max,shaft}$  is provided by:

$$F_{max,shaft} = O_p \cdot \int_0^{\Delta L} p_{max,shaft} dz$$

where:

$O_p$	pile periphery in bearing soil
$p_{max, shaft}$	maximum force on shaft (friction) from CPT results
$\Delta L$	pile length, either length of actived shaft friction or length of expanded toe
$z$	vertical dimension along pile axis

Actual calculation of the maximum pressure at pile toe  $p_{max,toe}$  and the maximum force developed along the shaft  $p_{max,shaft}$  (determined according to the selected type of analysis selected in the "Pile CPT" tab).

## EN 1997-2

The **EN 1997-2** standard determines the maximum pressure at pile toe (maximum resistance)  $p_{max,toe}$  from the corresponding  $i$ -th penetration test as follows:

$$P_{max,toe} = 0,5 \cdot \alpha_p \cdot \beta \cdot s \cdot \left( \frac{q_{c,I,m} + q_{c,II,m}}{2} + q_{c,III,m} \right)$$

where:	$q_{c,I,m}$	mean from values $q_{c,I}$ (see Addendum D.7 in EN 1997-2)
	$q_{c,II,m}$	mean of the minimum cone tip resistances $q_{c,II}$ (see Addendum B4 in EN 1997-3)
	$q_{c,III,m}$	mean of the cone tip resistance $q_{c,III}$ (see Addendum B4 in EN 1997-3)
	$\alpha_p$	pile toe coefficient (pile class factor)
	$s$	pile shape coefficient
	$\beta$	expanded pile toe coefficient

The **maximum value of penetration pressure**  $q_c$  is limited by the value of 15 MPa. In cohesionless soils the analysis takes into account the **influence of overconsolidation** (OCR).

The **maximum shaft friction** (shaft resistance)  $p_{max,shaft}$  is given by:

$$P_{max,shaft} = \alpha_s \cdot Q_{c,z,a}$$

where:	$\alpha_s$	shaft friction coefficient
	$Q_{c,z,a}$	tip resistance at depth $h$

*Literature:*

EN 1997-2 Geotechnical design. Ground investigation and testing.

## NEN 6743

The NEN 6743 "Piled Foundations" standard determines the **maximum pressure at pile toe**  $p_{max,toe}$  from the corresponding  $i^{th}$  penetration test as follows:

$$P_{max,toe} = 0,5 \cdot \alpha_p \cdot \beta \cdot s \cdot \left( \frac{q_{c,I,m} + q_{c,II,m}}{2} + q_{c,III,m} \right)$$

kde:	$q_{c,I,m}$	mean of the cone tip resistance $q_{c,I}$ (see article 5.3.3.3 in NEN 6743 standard)
	$q_{c,II,m}$	mean of the minimum cone tip resistance $q_{c,II}$ (see article 5.3.3.3 in NEN 6743 standard)
	$q_{c,III,m}$	mean of the cone tip resistance $q_{c,III}$ (see article 5.3.3.3 in NEN 6743 standard)
	$\alpha_p$	pile toe coefficient
	$s$	pile shape coefficient
	$\beta$	expanded pile toe coefficient

The **maximum value of penetration pressure**  $q_c$  is limited by the value of 15 MPa. In cohesionless soils the analysis takes into account the **influence of overconsolidation** (OCR).

The **maximum shaft friction**  $p_{max,shaft}$  is given by:

$$P_{max,shaft} = \alpha_s \cdot q_{c,z,a}$$

where:  $\alpha_s$  shaft friction coefficient  
 $q_{c,z,a}$  tip resistance at depth  $h$

*Literature:*

*NEN 6743:1991/A1:1997, Geotechniek - Berekeningsmethode voor funderingen op palen - Drukpalen.*

## LCPC (Bustamante)

The LCPC - Laboratoire Central des Ponts et Chausees method (also known as Bustamante method based on the works of Bustamante and Gianceselli) determines the **maximum pressure at pile toe**  $p_{max,toe}$  as follows:

$$P_{max,toe} = \alpha_p \cdot q_{c,eq}$$

where:  $\alpha_p$  pile toe coefficient  
 $q_{c,eq}$  equivalent average cone tip resistance

The **maximum shaft friction**  $p_{max,shaft}$  is given by:

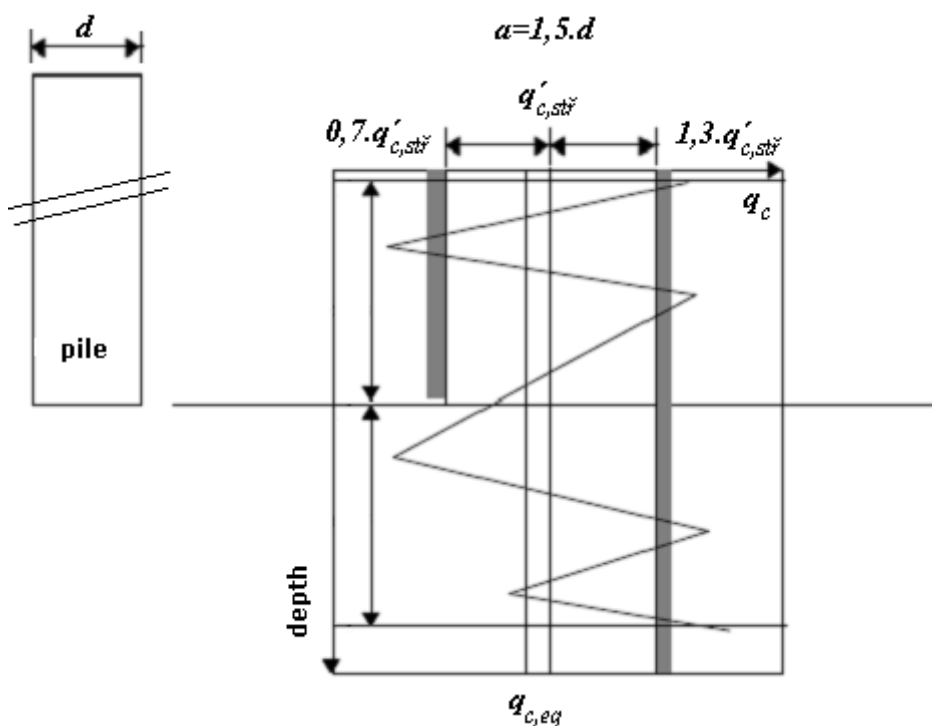
$$P_{max,shaft} = \alpha_s \cdot q_{c,z,a}$$

where:  $\alpha_s$  shaft friction coefficient  
 $q_{c,z,a}$  tip resistance

## Determination of Equivalent Average Cone Tip Resistance

An equivalent average cone tip resistance is obtained in the following way:

- 1) calculate the average tip resistance  $q_{c,m}$  at the tip of the pile by averaging  $q_c$  values over a zone ranging from  $1.5d$  below the pile tip to  $1.5d$  above the pile tip ( $d$  is the pile diameter)
- 2) eliminate  $q_c$  values in the zone which are higher than 1.3 multiple of the mean of the cone tip resistance  $q_{c,m}$  and those are lower than 0.7 multiple of the mean of the cone tip resistance  $q_{c,m}$  as shown in figure
- 3) calculate the equivalent average cone tip resistance  $q_{c,eq}$  by averaging the remaining cone tip resistance ( $q_c$ ) values over the same zone that were not eliminated (i.e. from values in the range 0.7 to 1.3 multiple of the cone tip resistance  $q_{c,m}$ )



Determination of equivalent average cone tip resistance  $q_{c,eq}$

Literature:

Tom Lunne, Peter K. Robertson, John J.M. Powell: *Cone Penetration Testing in Geotechnical Practice*, Spon Press, 1997, London.

## Schmertmann

The Schmertmann method determines the maximum pressure at pile toe  $p_{max,toe}$  as follows:

$$P_{max,toe} = \alpha_p \cdot q_{upr}$$

where:  $\alpha_p$  pile toe coefficient  
 $q_{upr}$  modified equivalent average cone tip resistance

$$q_{upr} = \frac{q_{c1} + q_{c2}}{2}$$

where:  $q_{c1}, q_{c2}$  minimum value of mean of the cone tip resistance

In cohesionless soils the analysis takes into account the influence of overconsolidation (OCR).

The maximum shaft friction  $p_{max,shaft}$  is given by following formulas:

- for cohesionless soils:

$$P_{max,shaft} = K \left[ 0,5 \cdot (\bar{f}_s \cdot A_s)_{0 \text{ to } 8d} + (\bar{f}_s \cdot A_s)_{8d \text{ to } D} \right]$$

where:  $K$  correlation coefficient of skin friction  
 $\bar{f}_s$  mean value of penetrometer sleeve local friction  $f_s$  in the interval given by bracket subscript

$A_s$	area of the pile shaft surface in given interval
$d$	diameter of pile
$D$	embedded pile length

- **for cohesive soils:**

$$P_{max,shaft} = \sum_i \alpha_{s,i} \cdot \bar{f}_{s,i} \cdot A_{s,i}$$

where:	$\alpha_{s,i}$	shaft friction coefficient according Tomlinson in the $i^{th}$ layer
	$\bar{f}_{s,i}$	mean value of penetrometer sleeve local friction $f_s$ in the $i^{th}$ layer
	$A_{s,i}$	area of the pile shaft surface in the $i^{th}$ layer

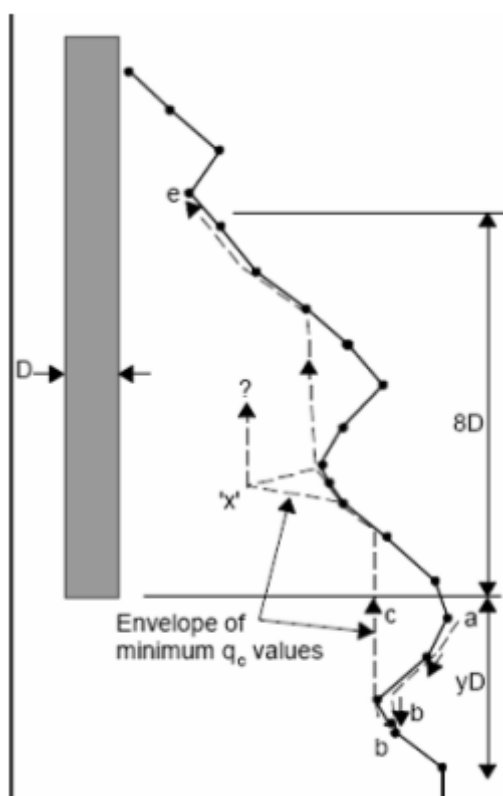
*Literature:*

Schmertmann J.H.: *Guidelines for Cone Penetration Test, Performance and design*, U.S. Departments of Transportation, report No. FHWA-TS-78-209, Washington, D.C., 1978.

## Determination of Average Cone Tip Resistance

The minimum mean value of the cone tip resistance  $q_c$  is determined by the minimum value of the mean of the cone tip resistance  $q_c$  over the influenced zone ranging from  $0.7d$  to  $4d$  below the pile toe ( $d$  is the pile diameter). The minimum mean value of the cone tip resistance  $q_{c2}$  is determined over the influence zone extending from  $8d$  above the pile toe ( $d$  is the pile diameter). The procedure for obtaining the mean value of the cone tip resistance  $q_{c1}$ ,  $q_{c2}$  is as follows (see figure):

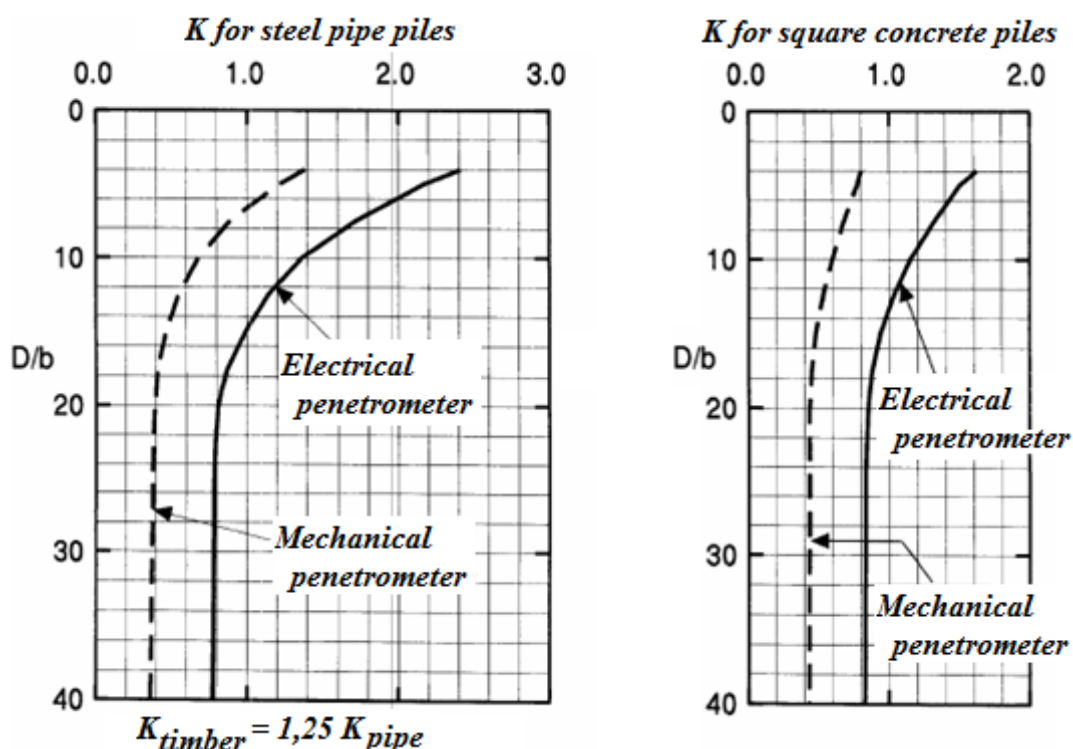
- 1) determine two averages of the cone stress within the zone below the pile toe, one for a zone depth of  $0.7d$  and one for  $0.4d$  along the path "a" through "b". The smaller of the two is retained. (The zone height  $0.7d$  applies to where the cone stress increases with depth below the pile toe).
- 2) determine the smallest cone stress within the zone used for the Step 1
- 3) determine the average of the two values per Steps 1 and 2. Step 4 is determining the average cone stress in the zone
- 4) determine the average cone stress in the zone  $8d$  above the pile toe that gives the value  $q_c$ . Finally, the average of the Step 3 and Step 4 values is determined.



Determination of average cone tip resistance  $q_{c1}$ ,  $q_{c2}$

## Correlation Coefficient $K$

Correlation coefficient of skin friction  $K$  is entered in the "Pile CPT" tab. Value of this coefficient is equal to ratio of unit pile shaft resistance and unit penetrometer sleeve local friction. Correlation coefficient  $K$  can be expressed for example by function of embedded pile length - see following graphs.



Function of embedded pile length ( $D$  - embedded pile length,  $b$  - pile width or diameter)

Literature:

FHWA HI 97-013: Design and Construction of Driven Pile Foundations, Workshop manual - Volume 1, National Highway institute.

## Negative Skin Friction

A negative skin friction is an effect that arises as a result of the settlement of soil around the pile. A soil deforming around the pile tends to pull the pile down thus reducing its bearing capacity. In extreme cases this effect may completely eliminate the influence of shaft friction. The pile is then supported only by elastic subsoil below the pile toe.

The **negative skin friction**  $F_{s,nk,rep}$  is given by:

$$F_{s,nk,rep} = O_p \cdot \sum_{i=1}^n h_i \cdot K_{0,i,rep} \cdot \tan(\delta_{i,rep}) \cdot \frac{\sigma_{v,i-1,rep} + \sigma_{v,i,rep}}{2} + p_{i,a,rep} - \Delta\sigma_{i,v,w,rep}$$

where:  $O_p$  pile periphery

$n$  number of layers in the negative friction zone

$h_i$  depth of  $i^{th}$  layer

$K_{0,i,rep}$  representative value of the coefficient of earth pressure at rest

$\delta_{i,rep}$  friction between soil and pile at  $i^{th}$  layer

$$\delta_{i,rep} = 0,75 \cdot \varphi_{i,rep}$$

$\varphi_{i,rep}$  representative value of the angle of internal friction at  $i^{th}$  layer



- $\sigma_{v,i-1,rep}$  horizontal stress in soil at  $i-1$  layer
- $\sigma_{v,i,rep}$  horizontal stress in soil at  $i^{th}$  layer
- $p_{i,a,rep}$  surcharge at  $i^{th}$  layer
- $\Delta\sigma_{i,v,w,rep}$  change of vertical stress  $\sigma_v$  at  $i^{th}$  layer

the following relation holds:  $K_{0,i,rep} \cdot \tan(\delta_{i,rep}) > 0,25$

If a slip surface is defined then the value of negative skin friction  $F_{s,nk,rep}$  is provide by:

$$F_{s,nk,rep} = O_p \cdot \sum_{i=1}^n h_i \cdot c_{i,rep}$$

- where:
- $O_p$  pile periphery
  - $h_i$  depth of  $i^{th}$  layer
  - $c_{i,rep}$  representative cohesion of slip surface
    - for bitumen  $10 \cdot 10^3 \text{ N/m}^2$
    - for bentonite  $20 \cdot 10^3 \text{ N/m}^2$
    - for synthetic material  $50 \cdot 10^3 \text{ N/m}^2$

The value of representative cohesion along a slip surface can also be introduced directly by the user.

## Shaft Friction Coefficient ALFAs

The coefficient reducing the shaft friction  $\alpha_s$  differs based on the applied method and the type of soil. The values of this coefficients are built into the program according to **EN 1997-2** and **NEN 6743** standards.

The values **for sands** and **sands with gravel** are listed in the following table:

Piles	NEN 6743	EN 1997-2
	$\alpha_s$ [-]	$\alpha_s$ [-]
prefabricated driven piles or steel piles	0.010	0.010
Franki piles	0.014	0.012
driven wooden piles	0.012	0.012
vibrating or vibropressed	0.012	0.012
cast in place screw piles	0.009	0.009
prefabricated screw piles	0.009	0.009

cast in place screw piles with additional grouting	0.006	0.006
prefabricated screw piles with additional grouting	0.006	0.006
steel tubular piles	0.0075	0.0075
Continuous Flight Auger piles (CFA)	0.006	0.006
bored piles or piles sheeted by bentonite suspense	0.006	0.006
bored piles with steel casing	0.005	0.005

For **very coarse-grained sands** and **gravels** the above values are reduced in both methods by a reduction coefficient (coarse-grained sand 0.75, gravel 0.5).

For **peat** the value of  $\alpha_s = 0$  is considered.

For **clay and silt** the values of  $\alpha_s$  according to the **EN 1997-2** are listed in the following table:

Type of soil	$q_c$ [MPa]	$\alpha_s$ [-]
clay	> 3	< 0.030
clay	< 3	< 0.020
silt		< 0.025

For **clay and silt** the values of  $\alpha_s$  according to the **NEN 6743** are listed in the following table:

$q_c$ [MPa]	$\alpha_s$ [-]
> 1	0.035
< 1	0.0 depth to quintuple of pile diameter
	0.025 depth from 5 to 20 multiple of pile diameter
	0,035 depth over 20 multiple of pile diameter

If the **LCPC (Bustamante)** method is used the shaft friction coefficient  $\alpha_s$  is used depending on the tip resistance  $q_c$  (orientation values are available in the following table).

**Orientation values of the shaft friction coefficient  $\alpha_s$  based on the cone tip resistance**

$q_c$

<b>LCPC (Bustamante) Soil type</b>	<b>Cone stress (tip resistance)</b> $q_c$ [MPa]	$\alpha_s$ <b>for piles of type "A"</b>	$\alpha_s$ <b>for piles of type "B"</b>	<b>Maximum shaft resistance</b> [kPa]
Clay	$< 1$	0.033	0.033	15
	$1 < q_c < 5$	0.025	0.011	35
	$5 < q_c$	0.017	0.008	35
Sand	$q_c < 5$	0.010	0.008	35
	$5 < q_c < 12$	0.010	0.005	80
	$12 < q_c$	0.007	0.005	120

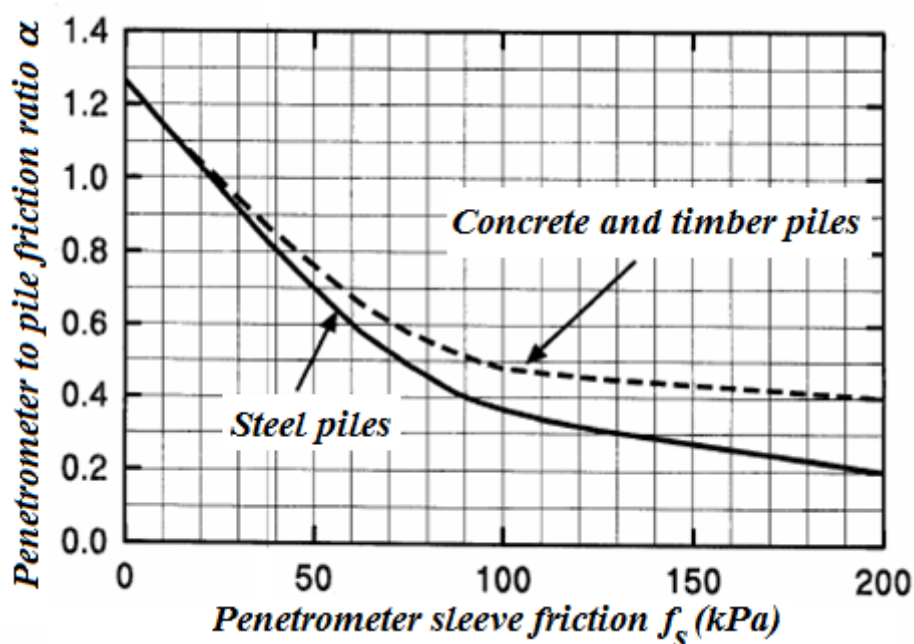
**Type "A"** includes these types of technology of installation of piles:

- **screw** (cast in place, prefabricated, cast in place with additional grouting, prefabricated with additional grouting, CFA piles, bored or piles sheeted by bentonite suspension)

**Type "B"** includes these types of technology of installation of piles:

- **driven** (prefabricated or steel, wooden)
- **Franki piles**
- **vibrating**
- **steel tubular**
- **bored with steel casing**

When using **Schmertmann** method, coefficient  $\alpha_s$  reducing shaft friction according to Tomlinson is considered. Values used in program are derived from following graph mentioned in publication M. J. Tomlinson: Pile Design and Construction Practice (1994).



Literature:

Tomlinson M. J.: *Pile Design and Construction Practice*, 4<sup>th</sup> edition, Taylor and Francis, 1994, ISBN 0 419 18450 3.

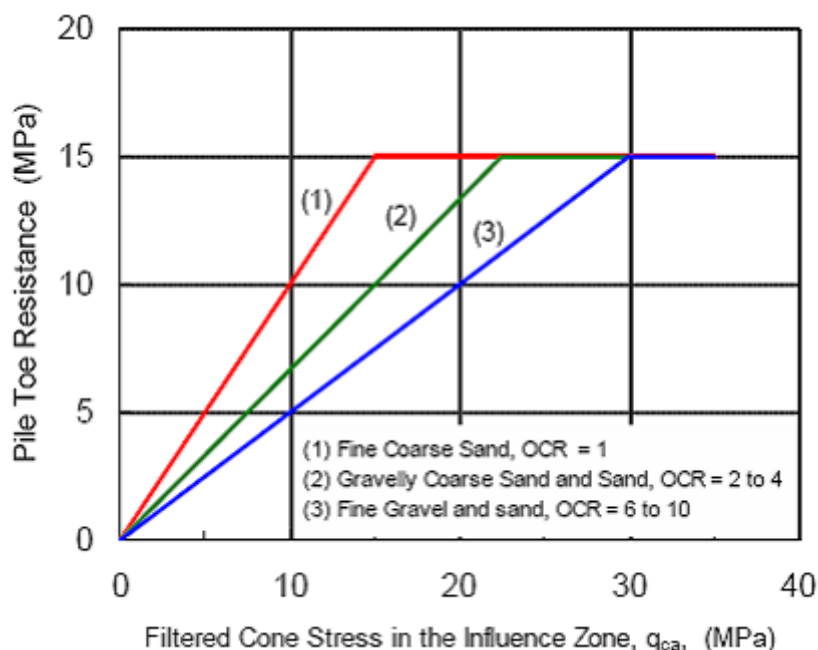
## Influence of Overconsolidation (OCR)

For sand and gravel the maximum pressure at pile toe  $p_{max,toe}$  (determined according to the selected type of analysis selected in the "Pile CPT" tab) is reduced depending on the value of overconsolidation OCR (defined as a soil parameter in the frame "Soils") as follows:

### Analysis according to EC 7-3, NEN 6743:

- for all cohesionless soils the maximum pressure at pile toe  $p_{max,toe}$  is 15 MPa
- for  $OCR \leq 2$  no reduction is performed
- for  $2 < OCR \leq 4$  the maximum pressure at pile toe  $p_{max,toe}$  is multiplied by 0.67
- for  $OCR > 4$  the maximum pressure at pile toe  $p_{max,toe}$  is multiplied by 0,50

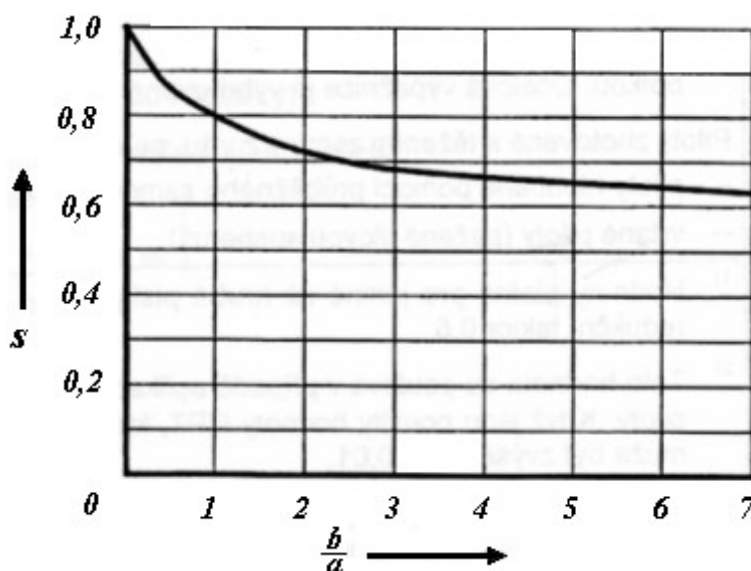
**When using the Schmertmann method** the reduction is performed according to the following graph:



*Reduction of equivalent mean cone tip resistance according to OCR (Schmertman)*

### Coefficient of Influence of Pile Shape *s*

This coefficient represents the influence of a **rectangular** pile cross-section, the *b/a* ratio in particular. Its values are evident from the following graph (function of *b/a*):

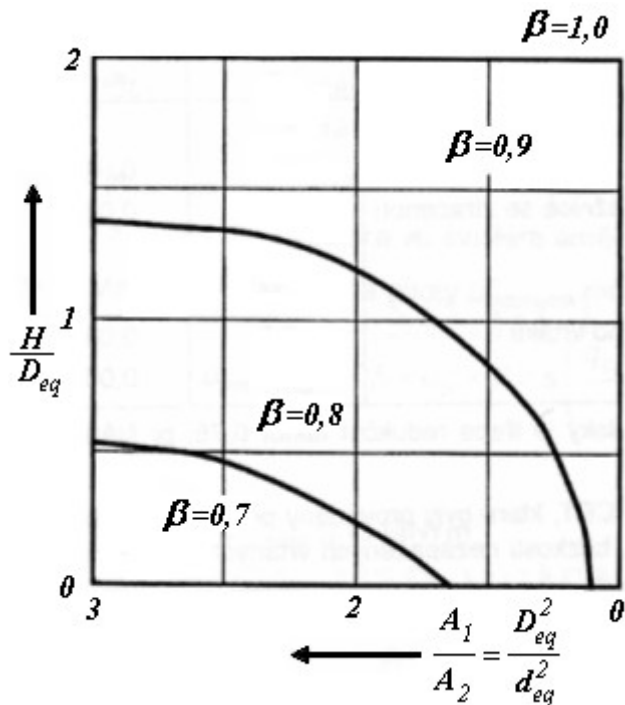


*Graph to determine pile shape coefficient *s* (*a* - length of the smallest side, *b* - the largest side)*

### Coefficient of Influence of Pile Widened Base **BETA**

This coefficient denoted as  $\beta$  represents the influence of an expanded pile base, its values are evident from the following figure:

(as a function  $\frac{D_{eq}^2}{d_{eq}^2}$  of  $\frac{H}{D_{eq}}$ ):



Graph to determine the coefficient  $\beta$

where:  $H$  pile length [m]  
 $D_{eq}$  equivalent pile diameter at pile base [m]  
 $d_{eq}$  equivalent pile shank diameter [m]

### Coefficient of Reduction of a Pile Base Bearing Capacity ALFA p

The coefficient of reduction of pile base bearing capacity  $\alpha_p$  identifies the type of pile. Its values are determined from one of the available calculation methods or they can be entered manually by the user.

For **NEN 6743** and **EN 1997-2** methods the following built-in values of the coefficient  $\alpha_p$  are available:

Piles	$\alpha_p$ [-]
prefabricated driven piles or steel piles	1.0
Franki pile	1.0
driving wooden pile	1.0

Vibrating	1.0
cast in place screw piles	0.9
prefabricated screw pile	0.8
cast in place screw piles with additional grouting	0.9
prefabricated screw pile with additional grouting	0.8
steel tubular piles	1.0
continuous Flight Auger (CFA)	0.8
bored piles or piles sheeted by bentonite suspense	0.5
bored piles with steel casing	0.5

For **LCPC** and **Schmertmann** the coefficient is back-calculated based on the value of cone resistance  $q_c$  (the values are presented in the following table):

**Orientation values of the coefficient  $\alpha_p$  based on the cone resistance  $q_c$**

<b>LCPC (Bustamante) Soil type</b>	<b>Cone resistance <math>q_c</math> [MPa]</b>	$\alpha_p$ <b>for bored piles</b>	$\alpha_p$ <b>for driven piles</b>
Clay	< 1	0.04	0.50
	$1 < q_c < 5$	0.35	0.45
	$5 < q_c$	0.45	0.55
Sand	$q_c < 12$	0.40	0.50
	$13 < q_c$	0.30	0.40

## Pile Group

Analysis of a group of piles depends on the **structure stiffness**. The basic assumption is that for a stiff structure all piles experience the same settlement, while for a compliant structure each pile deforms independently - no interaction is assumed.

The **maximum bearing capacity of a rigid pile foundation** is given by:

$$F_{r,found,max} = M \cdot F_{r,max,rep}$$

where:  $M$  number of piles in the pile foundation  
 $F_{r,max,rep}$  single pile bearing capacity in the pile foundation

If adopting the NEN6743 standard then a coefficient of capacity reduction  $\xi$  is introduced into the analysis depending on the number piles  $M$  and the number of CPT tests (article 5.3.2.1).

The maximum **bearing capacity of a compliant pile foundation** is determined according to the bearing capacity of the most stressed pile in the group as:

$$F_{r,found,max} = \max(F_{r,i})$$

where:  $F_{r,i}$  bearing capacity of fully stressed pile in the group

## Calculation of Pile Toe Settlement

The magnitude of pile head settlement  $w_{l,d}$  is determined as follows:

$$w_{l,d} = w_{toe,d} + w_{el,d}$$

where:  $w_{toe,d}$  pile toe settlement due to acting force

$$w_{toe,d} = w_{toe,d,1} + w_{toe,d,2}$$

$w_{toe,d,1}$  pile toe settlement due to force acting at toe

$w_{toe,d,2}$  pile toe settlement due to force acting on the shaft

$w_{el,d}$  pile settlement due to elastic compression

The magnitudes of settlements  $w_{toe,d,1}$  and  $w_{toe,d,2}$  are determined from **built-in graphs** according to the NEN6743 standard. The value  $w_{el,d}$  is given by:

$$w_{el,d} = \frac{LF_{mean,d}}{A_{plast}E_{p,mat,d}}$$

where:  $L$  pile length

$F_{mean,d}$  mean of force acting on the pile

$A_{plast}$  pile shank cross-sectional area

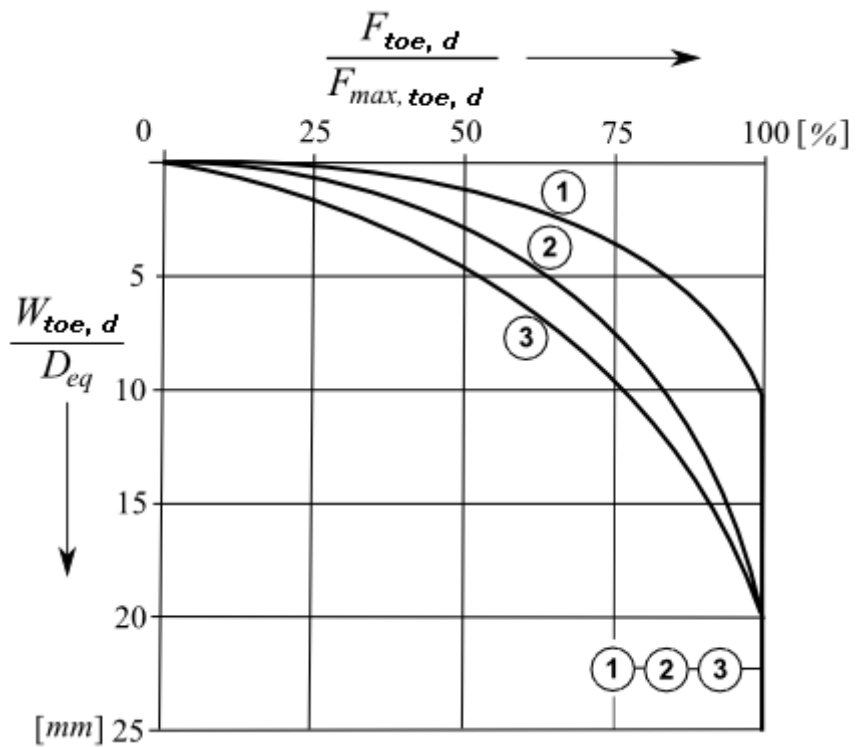
$E_{p,mat,d}$  modulus of elasticity of pile material

## Graphs to Calculate Settlement

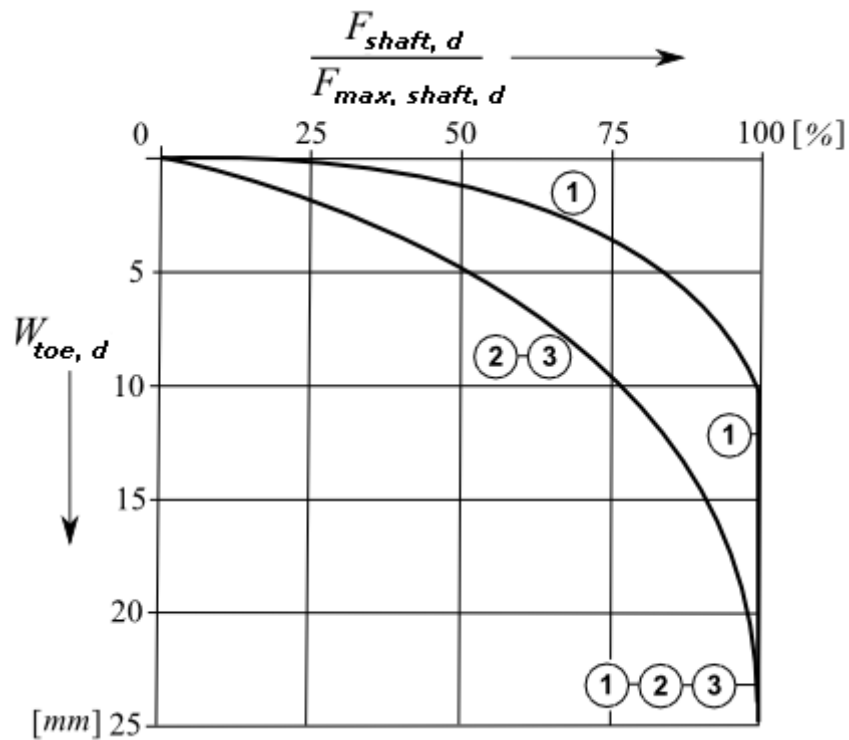
Graphs to calculate settlement are taken from the NEN6743 standard (article 6.2.1), which allow us to determine:

- Pile settlement due to toe vertical force (pile settlement in percentage of the equivalent pile diameter plotted as a function of the toe vertical force given in percentage of the maximum toe resistance  $F_{max,toe}$ ).
- Pile settlement due to shaft force (pile settlement in  $mm$  plotted as a function of the shaft force given in percentage of the maximum shaft resistance  $F_{max,shaft}$ ).





Graph to determine  $w_{toe,d,1}$  (1 - driven piles, 2 - continuous auger, 3 - bored piles)

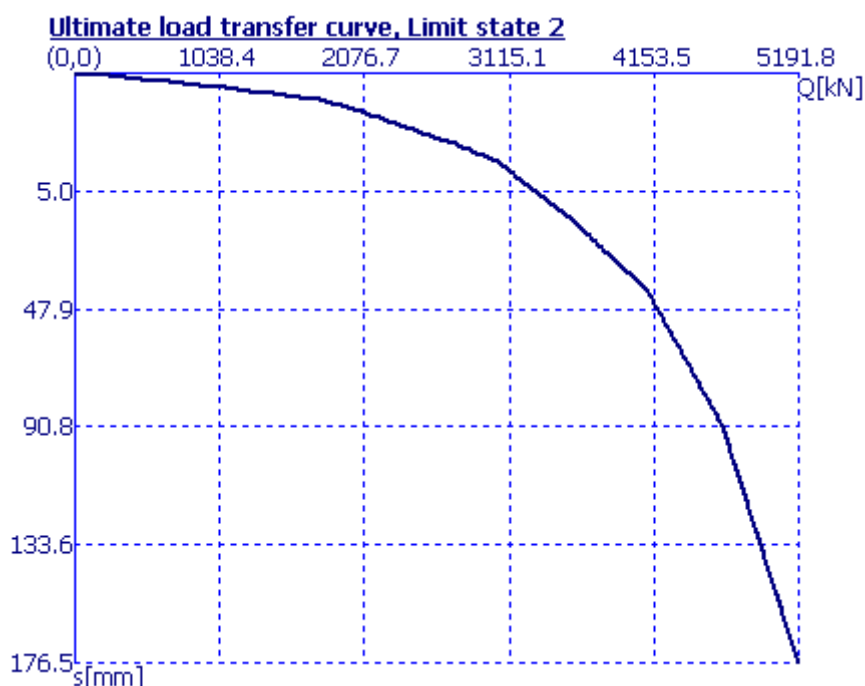


Graph to determine  $w_{toe,d,2}$  (1 - driven piles, 2 - continuous auger, 3 - bored piles)

## Calculation of Load-Settlement Curve

One of the program outputs is a load diagram of vertically loaded pile - **load-settlement curve**, which plots the pile vertical settlement as a function of the applied load.

The load-settlement curve is determined as a sum of the settlements due to forces at the pile toe and on the shaft derived from [graphs used to calculate](#) the pile settlement. A typical example of the load-settlement curve appears in the following figure.



*Load-settlement curve*

## Verification

Verification of pile (or group of piles) **bearing capacity** depends on the verification methodology selected in the "Pile CPT" tab:

- verification according to the [EN 1997-2](#)
- verification according to the [factor of safety](#) or the theory of [limit states](#)
- verification according to the [NEN 6743](#)

In **settlement** calculation it is possible to use either a [load-settlement curve](#) or a [load-displacement curve](#) when adopting the NEN 6743 standard.

## Verification According to EN 1997-2

When verification according to EN 1997-2 is set, required coefficients are specified in the "Pile CPT" tab.

Program determines the toe and shaft [bearing capacities](#). Result is  $N$  values of total bearing capacities for  $N$  CPT experiments:

$$R_{c,i} = R_{b,i} + R_{s,i}$$

- $R_{c,i}$  – total bearing capacity from  $i^{th}$  CPT experiment  
 $R_{b,i}$  – toe bearing capacity from  $i^{th}$  CPT experiment  
 $R_{s,i}$  – shaft bearing capacity from  $i^{th}$  CPT experiment

Each value of bearing capacity is reduced by model coefficient  $\gamma_{cal}$ :

$$R_{c,cal,i} = R_{b,cal,i} + R_{s,cal,i} = \frac{R_{b,i}}{\gamma_{cal}} + \frac{R_{s,i}}{\gamma_{cal}}$$

Model coefficient  $\gamma_{cal}$  is established according to design values and in-situ pile experiments (statistical evaluation). Coefficient can be set in the "Pile CPT" tab.

Program evaluates automatically standard values of pile resistance in compression:

- **from minimal value**

$$R_{c,k,min} = R_{b,k,min} + R_{s,k,min}$$

where:

$$R_{b,k,min} = \min \frac{R_{b,cal,i}}{\xi_4}$$

$$R_{s,k,min} = \min \frac{R_{s,cal,i}}{\xi_4}$$

- **from mean value**

$$R_{c,k,mean} = R_{b,k,mean} + R_{s,k,mean}$$

where:

$$R_{b,k,mean} = \frac{\sum_{i=1}^N R_{b,cal,i}}{\xi_3}$$

$$R_{s,k,mean} = \frac{\sum_{i=1}^N R_{s,cal,i}}{\xi_3}$$

Correlation coefficients  $\xi_3$  and  $\xi_4$  for evaluating standard values of bearing capacity are set automatically according to number of CPT experiments. Constructions with sufficient stiffness and resistance can be modelled by reducing of the correlation coefficients by value 1.1 (result cannot be less than 1.0 after dividing). Reduction of coefficients  $\xi_3$  and  $\xi_4$  is set in the frame "Settings".

Result standard value is minimal value from both values (minimal and mean bearing capacity):

$$R_c = \min(R_{c,k,min} : R_{c,k,mean})$$

Design values of bearing capacities are calculated from standard values:

- **from minimal value**

$$R_{c,d,min} = \frac{R_{b,k,min}}{\gamma_b} + \frac{R_{s,k,min}}{\gamma_s}$$

- **from mean value**

$$R_{c,d,mean} = \frac{R_{b,k,mean}}{\gamma_b} + \frac{R_{s,k,mean}}{\gamma_s}$$

Bearing capacity is reduced by coefficients  $\gamma_b$  and  $\gamma_s$  (toe and shaft). Default values are set to 1.0. Values of coefficients can differ depending on various methodologies and countries - user should specify them in the "Pile CPT" tab.

Result value of design bearing capacity is minimal value from both values (minimal and mean bearing capacity):

$$R_{c,d} = \min(R_{c,d,min} : R_{c,d,mean})$$

Verification of pile for bearing capacity is given by following formula:

$$F_{s,d} < R_{c,d}$$

where:  $F_{s,d}$  -design load  
 $R_{c,d}$  -design pile bearing capacity

## Correlation Coefficients for Evaluating Standard Values of Bearing Capacity

**Correlation coefficients  $\zeta$  for evaluating standard values of bearing capacity from results of soil experiments ( $n$  - number of CPT test profiles)**

$\zeta$ for $n =$	1	2	3	4	5	7	10
$\zeta_3$	1.40	1.35	1.33	1.31	1.29	1.27	1.25
$\zeta_4$	1.40	1.27	1.23	1.20	1.15	1.12	1.08

## Verification According to the Safety Factor

The verification analysis according to factor of safety is selected in the "Pile CPT" tab. This frame also allows to define the required factor of safety for bearing capacity. Pile verification then assumes the form:

$$\frac{F_{r,d}}{F_{s,d}} > SF_b$$

where:  $F_{s,d}$  pile load  
 $SF_b$  safety factor for bearing capacity  
 $F_{r,d}$  pile bearing capacity

## Verification According to Limit States

The verification according to limit states is selected in the "Pile CPT" tab, including setting of pile bearing capacity reduction coefficient. When using the NEN 6743 standard the program automatically performs the verification analysis as specified by this standard and therefore the frame "Settings" is not accessible. Pile verification for the **first limit state** assumes the formula:

$$F_{s,d} < \frac{F_{r,d}}{\gamma_t}$$

where:  $F_{s,d}$  design pile load  
 $\gamma_t$  reduction coefficient of bearing capacity  
 $F_{r,d}$  design pile bearing capacity

## Pile Group

Analyses performed in the "Pile Group" program can be divided into two groups:

- **Analytical solution** - calculation of the vertical bearing capacity of a pile group for **cohesive** and **cohesionless** soils and the determination of **settlement**
- Analysis of a pile group using the **spring method** together with the determination of **reinforcement** of piles

## Analytical Solution

Analysis of the vertical bearing capacity of a pile group can be performed for:

- **cohesionless soil** (analysis for drained conditions)
- **cohesive soil** (analysis for undrained conditions)

The actual verification analysis is carried out according to the **factors of safety** or the theory of **limit states**.

The verification is performed for the **vertical load** only. Load due to moments and shear forces is not considered. To account for horizontal actions of the pile group calls for choosing the **spring method** in the frame "Settings".

The analytical methods also allow for calculating the **pile group settlement**.

## Cohesionless Soil (Analysis for Drained Conditions)

The same methods as for the analysis of an isolated pile are used to calculate the vertical bearing capacity of a pile group:

- NAVFAC DM 7.2
- Effective stress
- CSN 73 1002

The pile group vertical bearing capacity is provided by:

$$R_g = \sum R_c = nR_c\eta_g$$

where:

- $n$  - number of piles in a group
- $R_c$  - vertical bearing capacity of an isolated pile
- $\eta_g$  - pile group efficiency

The actual verification analysis is carried out according to the factors of safety or the theory of limit states.

## Efficiency of a Pile Group

### UFC 3-220-01A

- $\eta_g \approx 0,7$  for axial spacing of piles in the group:  $3d$
- $\eta_g \approx 1,0$  for axial spacing of piles in the group:  $6d$

### La Barré (CSN 73 1002):

$$\eta_g = 1 - \psi \left[ \frac{(n_x - 1)n_y + (n_y - 1)n_x}{90n_x n_y} \right]$$

$$\psi = \arctg \frac{d}{s}$$

where:

- $n_x$  - number of piles in the  $x$  direction
- $n_y$  - number of piles in the  $y$  direction
- $\psi$  - angle having tangent  $tg\psi = d/s$ , expressed in degrees
- $s$  - axial spacing of piles
- $d$  - diameter of piles

### Seiler-Keeney formula:

$$\eta_g = \left[ 1 - 0,479 \left( \frac{s}{s^2 - 0,093} \right) \left( \frac{n_x + n_y - 2}{n_x + n_y - 1} \right) \right] + \frac{0,3}{n_x + n_y}$$

where:

- $n_x$  - number of piles in the  $x$  direction
- $n_y$  - number of piles in the  $y$  direction

$s$  - axial spacing of piles

### Input efficiency

User-defined input of the degree of efficiency in the range of 0.5 - 1.0.

*Literature:*

*Pochman, R.; Simek, J.: Pilotove zaklady - Komentar k CSN 73 1002. 1<sup>st</sup> edition, Prague, Vydavatelstvi norem, 1989, 80 p.*

*Unified Facilities Criteria (UFC 3-220-01A): Design of deep foundations - Technical instructions, Chapter 5-3, 1997.*

*Venkatramaiah, C.: Geotechnical Engineering. Second edition, New Delhi (India): New Age International Publishers, 1995.*

## Cohesive Soil (Analysis for Undrained Conditions)

The bearing capacity of an earth block is provided by:

$$R_{\varepsilon} = 2 \cdot l \cdot (b_x + b_y) \cdot c_{us} + N_{\varepsilon g} \cdot c_{ub} \cdot b_x \cdot b_y$$

where:  $l$  - length of piles

$b_x, b_y$  - plane dimensions of the base of an earth body in the form of a block

$c_{us}$  - average undrained shear strength along the piles ( $\varphi_u \approx 0$ )

$c_{ub}$  - undrained shear strength at the base of piles

$N_{\varepsilon g}$  - cohesion group bearing capacity factor

$$N_{\varepsilon g} = 5 \cdot \left[ \left( 1 + 0,2 \cdot \frac{b_x}{b_y} \right) \cdot \left( 1 + 0,2 \cdot \frac{l}{b_x} \right) \right] \text{ for condition: } \frac{l}{b_x} \leq 2,5$$

$$N_{\varepsilon g} = 7,5 \cdot \left( 1 + 0,2 \cdot \frac{b_x}{b_y} \right) \text{ for condition: } \frac{l}{b_x} > 2,5$$

where:  $b_x$  - minimum width of pile group (shorter layout size of the pile cap)

**Note:** The earth body is represented by a block with its base given by a plane containing feet of individual piles and having vertical walls found in the distance of one pile diameter from the axes of outer piles. This earth block subjected to overall load caused by the pile group resists by shear along the walls - **skin friction** and by bearing capacity at its **base**.

The actual verification analysis is carried out according to the **factors of safety** or the theory of **limit states**.

## Analysis According to the Safety Factor

When performing the analysis according to the **factor of safety** the program carries out the verification analysis for a **pile group in compression**:

$$\frac{R_g}{V_d + W_p} > SF_{cp}$$

- where:
- $R_g$  -vertical bearing capacity of a pile group
  - $V_d$  -maximum vertical force (including the pile cap self weight)
  - $W_p$  -self weight of piles (only when the option "**Consider the self weight of pile**" is checked)
  - $SF_{cp}$  -factor of safety for a pile group in compression

## Analysis According to the Theory of Limit States

When performing the analysis according to the theory of **limit states** the program carries out the verification analysis for a pile group in a **cohesionless soil**:

$$R_g = n \frac{R_c}{\gamma_t} \eta_g \geq V_d + W_p$$

- where:
- $R_g$  -vertical bearing capacity of a pile group
  - $n$  -number of piles in the group
  - $R_c$  -vertical bearing capacity of an isolated pile ( $R_b + R_s$ )
  - $\gamma_t$  -reduction coefficient of total resistance
  - $\eta_g$  -**pile group efficiency**
  - $V_d$  -maximum vertical force (including the pile cap self weight)
  - $W_p$  -self weight of piles (only when the option "**Consider the self weight of pile**" is checked)

When performing the analysis according to the theory of **limit states** the program carries out the verification analysis for a pile group in a **cohesive soil**:

$$\frac{R_g}{\gamma_t} \geq V_d + W_p$$

- where:
- $R_g$  -vertical bearing capacity of a pile group
  - $V_d$  -maximum vertical force (including the pile cap self weight)
  - $W_p$  -self weight of piles (only when the option "**Consider the self weight of pile**" is checked)
  - $\gamma_t$  -reduction coefficient of total resistance

When performing the verification analysis according to **EN 1997-1** the pile group vertical bearing capacity in a cohesive soil is reduced by the coefficient of base resistance ( $\gamma_t = \gamma_b$ ).

## Pile Group Settlement

### Cohesionless soil

The analysis of a pile group in a cohesionless soil is developed based on the **linear theory of**



settlement (Poulos). The load-settlement curve for a pile group and the value of the total settlement  $s_g$  is increased by so-called **group settlement factor**  $g_f$ .

An immediate settlement of the pile group increased by the group settlement factor is provided by:

$$s_g = g_f \cdot s_0$$

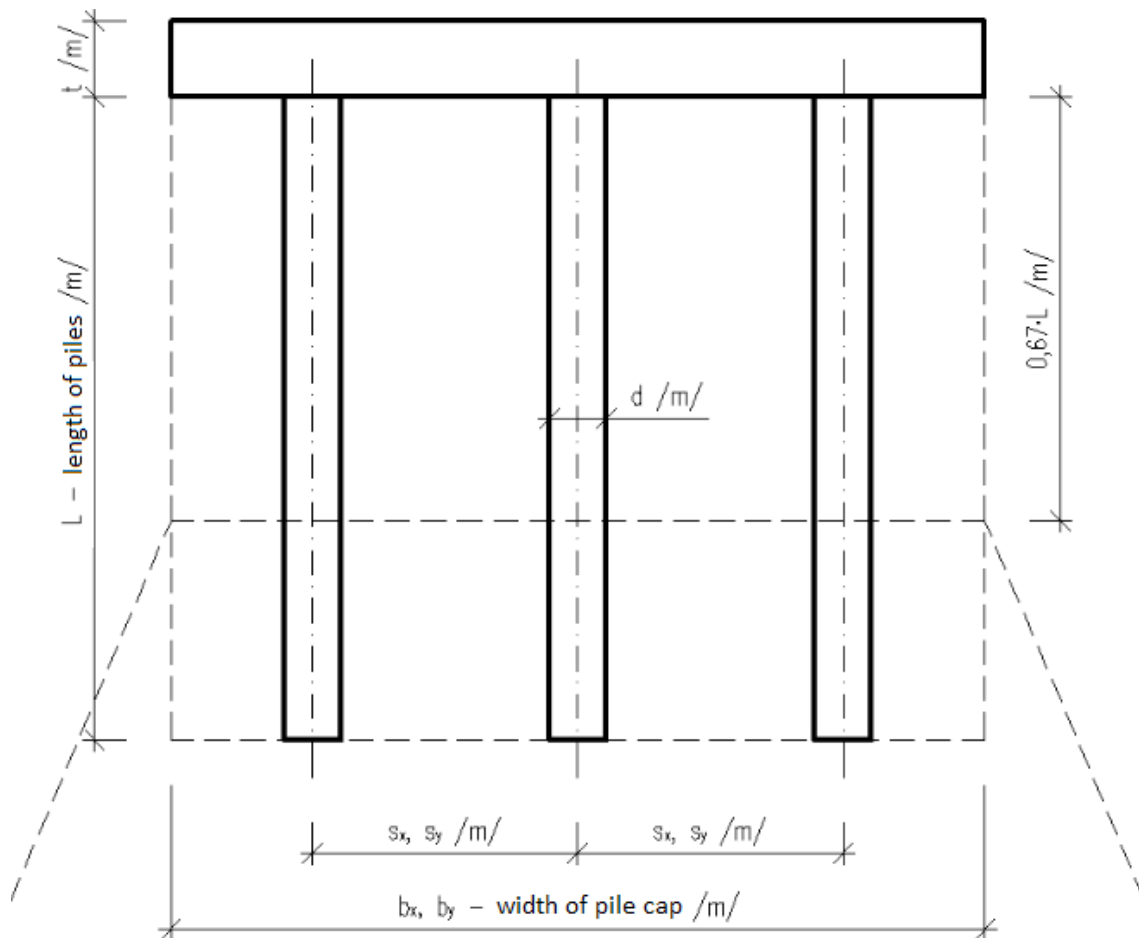
$$g_f = \sqrt{\frac{b_x}{d}}$$

- where:
- $s_g$  -pile group settlement
  - $g_f$  -group settlement factor for a cohesionless soil (according to Pile Buck Inc. 1992)
  - $s_0$  -settlement of a single pile (determined, e.g. from the load-settlement curve)
  - $d$  -pile diameter
  - $b_x$  -minimum width of pile group

**Cohesive soil**

The pile group settlement in a cohesive soil is determined as the settlement of a substitute foundation at a depth of  $0,67 \cdot L$ , having a width  $B$  and a length  $B'$ .

Analyses to calculate settlement are described in more detail in "Settlement analysis".

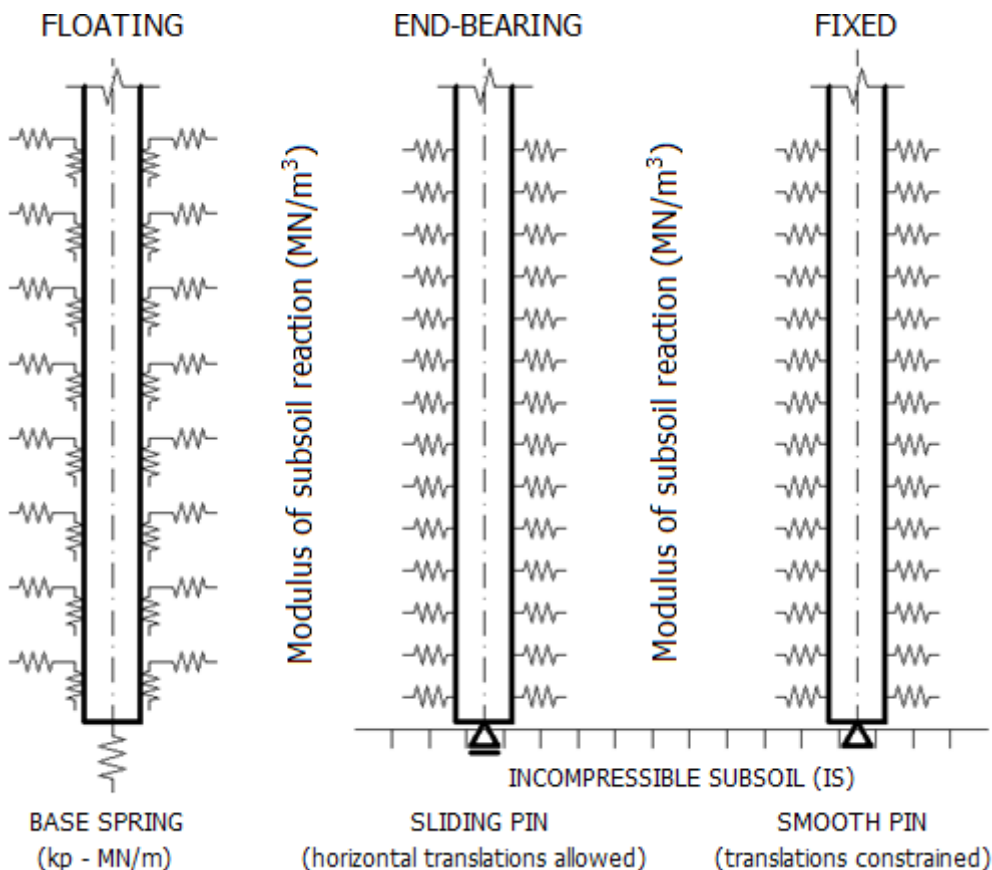


*Scheme of substitute foundation - settlement of pile group in cohesive soil*

## Spring Method

The pile group is analyzed using the Finite Element Method. The pile cap is considered as infinitely stiff. A general load is applied in the center of the cap and can be **imported** from an arbitrary program that performs static analysis.

The piles analyzed according to figure:



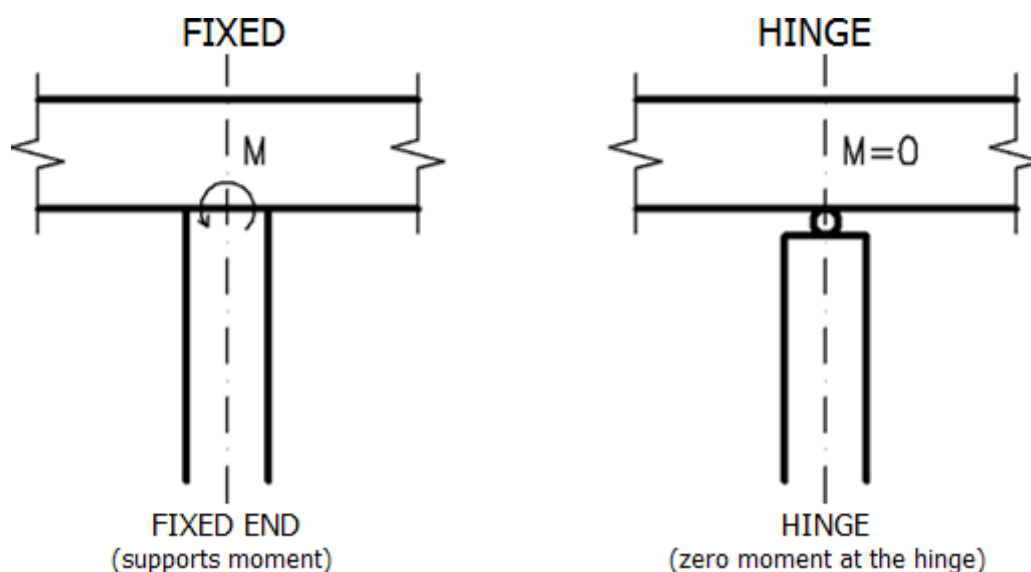
### *Modeling piles*

Four options to model piles are available in the frame "Settings":

1. Floating piles - compute the stiffness of springs from soil parameters
2. Floating piles - **input the stiffness of springs**
3. Piles resting on the rock subgrade
4. Piles fixed into the rock subgrade

All options require inputting the "**Horizontal modulus of subsoil reaction**" characterizing the pile behavior in the transverse direction. Floating piles further require determining the stiffness of vertical springs. The program allows for back **calculation of this stiffness** from the available soil parameters and the typical load. They can be also input directly in the frame "**Vertical springs**".

Either hinge or fixed-type connection of a pile to the pile cap can be considered.



*Connecting pile to the pile-cap (selected in the frame "Settings")*

The own analysis of structure is performed using the finite element method (FEM). Each pile is divided into ten elements. For each element the program defines the magnitude of horizontal and vertical springs. In comparison to a single pile, the stiffness of horizontal and vertical springs are further reduced for both the inner and outer piles - the horizontal stiffness is reduced by the coefficients equal to 0,5 and 0,25 for the outer and inner piles, respectively; the shear stiffness is reduced by the coefficients equal to 0,5 and 0,1 for the outer and inner piles, respectively. These reductions well represent the real behavior of a pile group. The springs at the pile-base are not reduced.

## Calculation of Stiffness of Vertical Springs

When back calculating the stiffnesses of vertical springs it is necessary to input a **typical load** in the frame "Vertical springs" that will serve to the determine the spring stiffnesses. This load should be selected such as to characterize the structure behavior as close as possible.

The stiffnesses are determined as follows:

1. Typical load is applied to individual piles
2. The stiffness of **shear vertical springs** distributed along the pile is calculated depending on the soil parameters.
3. The stiffness of vertical spring at the pile-base is calculated depending on the **stiffness of the subsoil below the pile-base** and the **depth of influence zone**. For tensile piles this stiffness is equal to zero.

These stiffnesses are further adjusted according to their location in the pile group - the shear stiffness is reduced by the coefficients equal to 0.5 and 0.1 for the outer and inner piles, respectively.

## Micropile

The program performs verification analysis of micropiles (reinforced by steel tube)

- based on **limit states**
- based on **factor of safety**

Both the **root section** and **micropile tube** (micropile cross-section) are examined for both cases. When examining the micropile tube the analysis may include expected **lifetime** of the micropile.

## Verification Based on Safety Factor

The program performs verification analysis of the micropile tube and root:

### Verification of the cross-section (tube)

Both, internal stability of section and coupled section bearing capacity, are verified.

#### 1. Internal stability of section

$$\frac{N_{cr}}{N_{max}} > SF_f$$

where:

- $N_{cr}$  -standard **critical normal force**, calculated in dependence on the method set in the "**Micropiles**" tab
- $N_{max}$  -maximal normal force, entered in the frame "**Load**"
- $SF_f$  -critical force safety factor, entered in the "**Micropiles**" tab

#### 2. Coupled section bearing capacity

$$\frac{R_s}{\sigma_s} > SF_s$$

where:

- $R_s$  -standard strength of steel, entered in the frame "**Material**"
- $\sigma_s$  -stress in steel, calculated according to the way of load (section loaded only by **normal force** or by **combination of bending moment and normal force**)
- $SF_s$  -safety factor of section resistance, entered in the "**Micropiles**" tab

### Verification of the root

$$\frac{Q}{N_{max}} > SF_r$$

where:

- $Q$  -standard root bearing capacity, calculated in dependence on used method (see "**Bearing capacity of the micropile root section**")
- $N_{max}$  -maximal normal force, entered in the frame "**Load**"
- $SF_r$  -root resistance safety factor, entered in the "**Micropiles**" tab

## Verification Based on Limit States

The program performs verification analysis of the micropile tube and root:

### Verification of the cross-section (tube)

Both, internal stability of section and coupled section bearing capacity, are verified.

### 1. Internal stability of section

$$N_{max} < N_{crd}$$

where:  $N_{max}$  -maximal normal force, entered in the frame "Load"  
 $N_{crd}$  -design critical normal force

$$N_{crd} = \frac{N_{cr}}{\gamma_{mf}}$$

where:  $N_{cr}$  -standard critical normal force, calculated in dependence on the method set in the "Micropiles" tab  
 $\gamma_{mf}$  -reduction coefficient of critical force, entered in the "Micropiles" tab (limit states)

### 2. Coupled section bearing capacity

$$\sigma_s < R_{sd}$$

where:  $\sigma_s$  -stress in steel, calculated according to the way of load (section loaded only by normal force or by combination of bending moment and normal force)  
 $R_{sd}$  -design strength of steel

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

where:  $R_s$  -standard strength of steel, entered in the frame "Material"  
 $\gamma_{ss}$  -reliability coefficient of steel, entered in the "Micropiles" tab (limit states)

### Verification of the root

$$N_{max} < Q_{rd}$$

where:  $N_{max}$  -maximal normal force, entered in the frame "Load"  
 $Q_{rd}$  -design root bearing capacity

$$Q_{rd} = \frac{Q}{\gamma_r}$$

where:  $Q$  -standard root bearing capacity, calculated in dependence on used method (see "Bearing capacity of the micropile root section")  
 $\gamma_r$  -reduction coefficient of root resistance, entered in the "Micropiles" tab (limit states)

## Verification of the Micropile Tube

When calculating the tube bearing capacity (micropile cross-section) the program differentiates between a micropile loaded in tension or in compression.

In case of tension the program determines **coupled section bearing capacity** (strength of cement mixture is not considered).

In case of compression the program examines both, **coupled section bearing capacity** and **internal stability of section**, depending on the method set in the "Micropiles" tab.

## Coupled Section Bearing Capacity

In the case of coupled section bearing capacity, the micropile tube is examined against the failure due to load caused by **normal force** or by **combination of bending moment and normal force**.

When determining the coupled section bearing capacity it is possible to involve influence of the expected **life time of the micropile**.

## Micropile Lifetime

The micropile life time is introduced by reducing the area of the reinforcing tube using the **reduction coefficient** of the influence of corrosion of steel tube  $r_e$  and **coefficient**  $F_{ut}$  taking into account connection of the micropile and the surrounding soil.

$$A_a = \frac{\pi}{4} \left[ (D - 2 \cdot r_e)^2 - (D - 2 \cdot t)^2 \right] F_{ut}$$

where:	$D$	-external diameter of reinforcing tube
	$t$	-wall thickness of reinforcing tube
	$F_{ut}$	-coefficient taking into account connection of micropile and surrounding soil
	$r_e$	-coefficient of influence of corrosion of steel tube

*Literature:*

*BS EN 14199:2005 Execution of special geotechnical works. Micropiles British-Adopted European Standard / 30-Mar-2005 / 52 pages ISBN: 0580457249.*

## Coefficient $F_{ut}$

**Coefficient  $F_{ut}$  taking into account connection of micropile and surrounding soil**

Type	$F_{ut}$ [-]
Using sleeve of external double spiral without reduction in cross-section	1.0
Spiral with increasing cross-section	1.0
Other types of connection	1.0
Other cases	1.0

## Coefficient of the Influence of Corrosion

Coefficient of influence of corrosion of steel tube  $r_e$  [mm] (based on EN 14199)

Type of soil	Required life time of micropile [years]				
	5	25	50	75	100
Soils in natural deposition	0.00	0.30	0.60	0.90	1.20
Soils in natural deposition contaminated	0.15	0.75	1.50	2.25	3.00
Organic soils	0.20	1.00	1.75	2.50	3.25
Loose soils	0.18	0.70	1.20	1.70	2.20
Special soils (containing soluble salts)	0.50	2.00	3.25	4.50	5.75

**Note:** Values of the coefficient of influence of corrosion of steel tube  $r_e$  are for intermediate values.

## Bearing Capacity of Cross-Section Loaded by Normal Force

### Tension normal force

In case of tension force, the stress in steel part of cross section is calculated using following formula:

$$\sigma_s = \frac{N}{A_s}$$

where:  $\sigma_s$  - stress in steel  
 $N$  - normal force acting in section  
 $A_s$  - area of the steel part of the micropile cross-section

### Compressive normal force

Bearing capacity of the cross-section in compression, reduced by buckling coefficient, is determined as:

$$N_{c,u} = \chi \cdot (A_s \cdot R_{sd} + A_c \cdot R_{cd})$$

where:  $\chi$  - buckling coefficient  
 $A_s$  - area of the steel part of the micropile cross-section  
 $A_c$  - area of the cement mixture part of the micropile cross-section  
 $R_s$  - design strength of steel  
 $d$

$R_c$  - design strength of cement mixture in compression  
 $d$

Design strengths are equal to standard values in the verification based on the **factor of safety**.

Design strengths of steel and cement mixture are calculated in the verification based on the theory of **limit states** as follows:

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

$$R_{cd} = \frac{R_c}{\gamma_{sc}}$$

where:  $R_s$  - standard strength of steel, entered in the frame "Material"  
 $\gamma_{ss}$  - reduction coefficient of steel strength, entered in the "Micropiles" tab  
 $R_c$  - standard strength of cement mixture in compression, entered in the frame "Material"  
 $\gamma_{sc}$  - reduction coefficient for cement mixture, entered in the "Micropiles" tab

The stress in the steel part of the cross-section is determined as:

$$\sigma_s = \frac{N}{N_{c,u}} \cdot R_{sd}$$

where:  $N$  - normal force acting in section  
 $N_{c,u}$  - bearing capacity of the cross-section in compression, reduced by **influence of buckling**  
 $R_{sd}$  - design strength of steel

## Bearing Capacity of Cross-Section Loaded by Combination of Bending Moment and Normal Force

A cross-section loaded by combination of bending moment and normal force requires the determination of neutral axis, dividing the cross-section into **tensile** and **compressed** part. When searching the position of neutral axis, influence of buckling is included, i.e. normal force is increased by dividing it by **coefficient of buckling**  $\chi$ . The neutral axis is searched following the procedure known from the dimensioning of concrete cross-sections, reinforced by steel, as a limit equilibrium method. Compression is transmitted by a part of a steel tube and cement mixture filling. Tension is taken by the remaining part of the steel tube, cement mixture in tension is not considered.

The bearing capacity in bending is determined by the following formula:

$$M_u = R_{sd} \cdot (A_{s,t} \cdot t_{s,t} + A_{s,c} \cdot t_{s,c}) + R_{cd} \cdot A_{c,c} \cdot t_{c,c}$$

where:  $R_s$  - design strength of steel  
 $d$



$A_{s,t}$  - area of the tensile part of the steel micropile cross-section

$A_{s,c}$  - area of the compressed part of the steel micropile cross-section

$A_{c,c}$  - area of the compressed part of the cement mixture cross-section

$t_{s,t}$  - location of the center of tensile steel part

$t_{s,c}$  - location of the center of compressed steel part

$t_{c,c}$  - location of the center of compressed cement mixture part

$R_c$  - design strength of cement mixture in compression

$d$

Design strengths are equal to standard values in the verification based on the **factor of safety**.

Design strengths of steel and cement mixture are calculated in the verification based on the theory of **limit states** as follows:

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

$$R_{cd} = \frac{R_c}{\gamma_{sc}}$$

where:  $R_s$  - standard strength of steel, entered in the frame "Material"

$\gamma_{ss}$  - reliability coefficient of steel, entered in the "Micropiles" tab

$R_c$  - standard strength of cement mixture in compression, entered in the frame "Material"

$\gamma_{sc}$  - reliability coefficient of cement mixture, entered in the "Micropiles" tab

The stress in the steel part of the cross-section is determined as:

$$\sigma_s = \frac{M}{M_u} \cdot R_{sd}$$

where:  $M$  - bending moment acting in section

$M_u$  - bearing capacity in bending

$R_{sd}$  - design strength of steel

## Influence of Buckling

The analysis is preceded by the determination of characteristics of an ideal cross-section, in which the effect of cement mixture cross-section is transformed into steel. Slenderness of element is determined as:

$$\lambda = \frac{l_{cr}}{i}$$

where:  $l_{cr}$  - element buckling length  
 $i$  - radius of gyration of the ideal cross-section

$$l_{cr} = \sqrt{\frac{E \cdot I \cdot \pi^2}{N_{cr}}}$$

where:  $E$  - modulus of elasticity of the ideal cross-section  
 $I$  - moment of inertia of the ideal cross-section  
 $N_{cr}$  - standard **critical normal force**, calculated in dependence on the method set in the "Micropiles" tab

Recounted slenderness  $\lambda_p$  is determined next:

$$\lambda_p = \lambda_e \sqrt{\frac{R_{sd}}{210}}$$

where:  $R_{sd}$  - design strength of steel (in calculation based on factor of safety design strength is equal to standard strength)

$$R_{sd} = \frac{R_s}{\gamma_{ss}}$$

where:  $R_s$  - standard strength of steel, entered in the frame "Material"  
 $\gamma_{ss}$  - reliability coefficient of steel, entered in the "Micropiles" tab (limit states)

Buckling coefficient  $\chi$  is determined according to slenderness  $\lambda_p$  with the help of following formulas:

$$\chi = \frac{1}{2} \left[ 1,26 + \left( \frac{93}{\lambda_p} \right)^2 \right] - \sqrt{\frac{1}{4} \left[ 1,26 + \left( \frac{93}{\lambda_p} \right)^2 \right]^2 - \left( \frac{93}{\lambda_p} \right)^2} \quad \text{for: } \lambda_p \leq 250$$

$$\chi = \left\{ \frac{1}{2} \left[ 1,26 + \left( \frac{93}{250} \right)^2 \right] - \sqrt{\frac{1}{4} \left[ 1,26 + \left( \frac{93}{250} \right)^2 \right]^2 - \left( \frac{93}{250} \right)^2} \right\} \left( \frac{250}{\lambda_p} \right)^{for: \lambda_p > 250}$$

## Internal Stability of Section

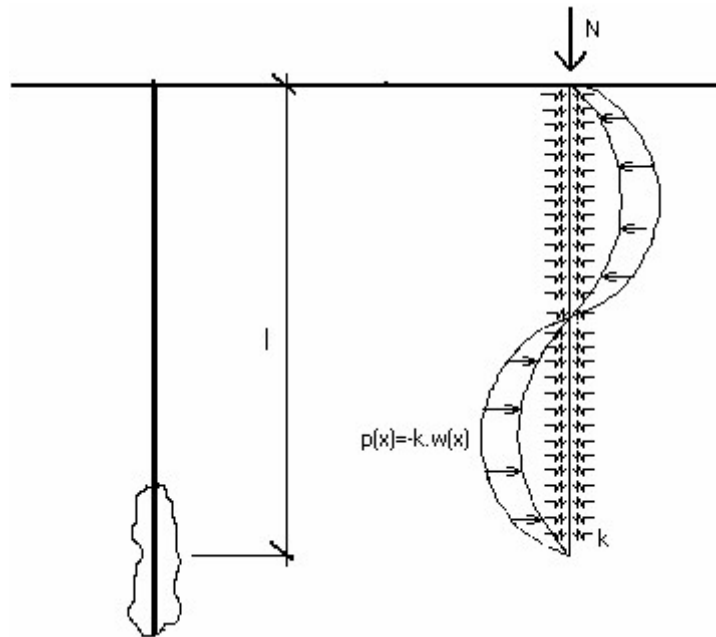
Internal stability of section examines the failure of a micropile due to buckling into the surrounding soil. The crucial step for the determination of internal stability of section is the calculation of the normal force  $N_{cr}$  that depends on the micropile length, the surrounding soil and other effects. User can choose one of the following solution methods in the "Micropiles" tab for calculating critical normal force  $N_{cr}$ :

- **Geometric method (Euler)**

- Salas theory
- Véas-Souche theory

## Geometric Method (Euler)

The soil surrounding the micropile is represented in the program by the modulus of subsoil reaction  $E_p$  (Winkler constant  $k$ ) defined by the user in the frame "Verification of cross-section". A model of a structure is displayed in the figure.



*Model of structure*

For a micropile in compression it is expected that a varying number of half waves occurs depending on the geometry and stiffness of the structure and surrounding soil, respectively. The solution of this case arises from the equation of bending of a straight beam.

$$w'''' = -\frac{M + N.w}{E.I}$$

After some manipulations the bending equation can be expressed as:

$$w_{(x)} = C_1 \cdot \cos(A.x) + C_2 \cdot \sin(A.x) + C_3 \cdot \cos(B.x) + C_4 \cdot \sin(B.x)$$

where:

$$A = \sqrt{\frac{\alpha^2}{2} + \sqrt{\frac{\alpha^4}{2} - 4.\beta^4}}$$

$$B = \sqrt{\frac{\alpha^2}{2} - \sqrt{\frac{\alpha^4}{2} - 4.\beta^4}}$$

$$\alpha = \sqrt{\frac{N}{E.I}}$$

$$\beta = \sqrt[4]{\frac{k}{4.E.I}}$$

Integration constants are found from four boundary conditions depending on the assumed end point supports.

**Assuming hinges on both ends** the following equation can be derived:

$$N_{cr} = E_i \cdot I_i \cdot \frac{\pi^2}{l_p^2} \cdot n^2 + E_p \frac{l_p^2}{\pi^2} \cdot n^{-2}$$

providing the number of half waves in the form:

$$n^2 = \sqrt{\frac{E_p}{E_i \cdot I_i} \cdot \frac{l_p^2}{\pi^2}}$$

where:  $E_i$  - modulus of elasticity of ideal cross-section  
 $I_i$  - moment of inertia of ideal cross-section  
 $l_p$  - micropile length  
 $E_p$  - modulus of subsoil reaction  
 $n$  - number of half waves

**Assuming hinge on the one side and fixed end on the other side** the following equation holds:

$$N_{cr} = E_i \cdot I_i \cdot \frac{\pi^2}{2 \cdot l_p^2} \cdot n + E_p \frac{4l_p^2}{\pi^2} \cdot n^{-2}$$

providing the number of half waves in the form:

$$n^2 = \sqrt{\frac{E_p}{E_i \cdot I_i} \cdot \frac{4l_p^2}{\pi^2}}$$

where:  $E_i$  - modulus of elasticity of ideal cross-section  
 $I_i$  - moment of inertia of ideal cross-section  
 $l_p$  - micropile length  
 $E_p$  - modulus of subsoil reaction

$n$  - number of half waves

Force  $N_{cr}$  is determined from the following equation by iterations:

$$l_{cr} = \sqrt{\frac{E_i \cdot I_i \cdot \pi^2}{N_{cr}}}$$

where:  $E_i$  - modulus of elasticity of ideal cross-section  
 $I_i$  - moment of inertia of ideal cross-section  
 $N_{cr}$  - critical normal force  
 $l_{cr}$  - buckling length of micropile cross-section in compression

## Salas Theory

The critical force  $N_{cr}$  for basic support conditions in the micropile head (determining micropile deflection) follows from:

$$N_{cr} = \pi^2 \cdot \frac{E_a \cdot I_a}{(l + l_{ef})^2} \cdot A$$

where:  $E_a \cdot I_a$  -bending stiffness of micropile reinforcing tube  
 $l$  -free length of micropile length  
 $l_{ef}$  -length of fictitious fixed end  
 $A$  -constat reflecting the type of support in micropile head

$$l_{ef} = 1, 2 \cdot f \cdot l_e$$

where:  $f$  -coefficient depending on the ratio of modulus of elasticity of soil in micropile head and base  
 $l_e$  -elastic length of micropile given by:

$$l_e = \left( \frac{3 \cdot E_a \cdot I_a}{E_l} \right)^{\frac{1}{4}}$$

where:  $E_a \cdot I_a$  -bending stiffness of micropile reinforcing tube  
 $E_l$  -modulus of elasticicy of soil in the micropile base

*Literature:*

*Jiménez Salas J.A. a kol: Geotecnica y Cimientos III, Capitulo 3, Rueda, Madrid (Spanish).*

## Constant A Reflecting the Type of Support in the Micropile Head

Constant  $A$  reflecting the type of support in the micropile head

Type of support in the micropile head	$A$ [-]
Hinged	2.045
Free	0.25
Fixed	4.0
Horizontally movable	1.0

## Coefficient $f$

Coefficient  $f$

$E_o / E_l^{1)}$ [-]	$f$ [-]
0	1.70
0.5	1.25
1	1.00

<sup>1)</sup>  $E_o$  - the modulus of elasticity of soil below terrain surface (at the micropile head)

$E_l$  - the modulus of elasticity of soil at the micropile root

## Véas-Souche Theory

Calculation of the force  $N_c$  follows from graphs published by Véas and Souche (see literature). The graphs for the determination of the critical normal force  $N_{cr}$  are constructed for dimensionless quantities  $\omega, m$ :

$$\frac{N_{cr} \cdot l_p^2}{\pi^2 \cdot E_a \cdot I_a}, \omega, m = \frac{1}{\pi} \sqrt[4]{\frac{E_{rd}}{E_a \cdot I_a}}$$

where:

- $l_p$  - micropile length
- $E_a \cdot I_a$  - bending stiffness of micropile reinforcing tube
- $\omega$  - ratio of free length of micropile (from beginning of base) and its length in soil
- $E_{rd}$  - design value of modulus of horizontal reaction

$$E_{rd} = \frac{E_r}{F_w}$$

where:  $E_r$  -reaction of soil in horizontal direction  
 $F_w$  -coefficient reducing the value of  $E_r$  ( $F_w = 1.25$ )

*Literature:*

Véase, Souche: *Étude du flambement de pieux partiellement immergés dans offrant latéralement une réaction élastique pure, Annales de ITBTP, No. 423, Sene Soils et Foundations, 187, mars - avril 1984, str. 38 - 60 (French).*

## Modulus of Horizontal Reaction of Subsoil

The soil surrounding the micropile can be represented using horizontal springs along micropile characterized by the Winkler constant  $k_h$ . For buckling of micropile into the soil in the direction of the  $x$  axis it is possible to write:

$$p_h = k_h \cdot x = E_p \cdot x$$

where:  $p_h$  -reaction of soil caused by the displacement of micropile in direction of  $x$  axis (soil in compression)  
 $k_h$  -stiffness of Winkler spring (modulus of subsoil reaction  $E_p$ )  
 $x$  -shift of micropile in direction of  $x$  axis

Providing we consider the reaction of soil to pressing of micropile per one meter run of the micropile we arrive at:

$$p_h = E_r \cdot x$$

where:  $E_r$  -reaction of soil in horizontal direction  
 $p_h$  -reaction of soil caused by shift of micropile in direction of  $x$  axis per one meter run of micropile  
 $x$  shift of micropile in direction of  $x$  axis

The above equations identify the relation between the modulus of subsoil reaction  $E_p$  [ $kN/m^3$ ] and the reaction of soil in the horizontal direction  $E_r$  [ $kN/m^2$ ] (assuming constant  $E_r$  in the soil):

$$E_r = k_h \cdot D = E_p \cdot D$$

where:  $D$  -diameter of micropile  
 $k_h$  -stiffness of Winkler spring (modulus of subsoil reaction  $E_p$ )

Reaction of soil in the horizontal direction  $E_r$  can be post calculated based on the knowledge of the pressiometric modulus  $E_m$ .

## Calculation of the Modulus of Horizontal Reaction of Subsoil $E_r$

The modulus of horizontal reaction of subsoil can be determined when knowing the **pressiometric modulus**  $E_m$  and coefficient  $\alpha_p$  as:

$$E_r = E_m \cdot \frac{6}{\frac{4}{3}(2,65)^{\alpha_p} + \alpha_p}$$

kde:  $E_m$  - **pressiometric (Menard) modulus** [MPa]

$\alpha_p$  - rheological factor of soil (see the table below)

### Reference values $E_m$ and $P_{lim}$

Soils		$E_m$ [Mpa]	$P_{lim}$ [MPa]
cohesionless	loose	0 - 3.5	0 - 0.5
	medium dense	3.5 - 12	0.5 - 1.5
	dense	12 - 22.5	0.5 - 2.5
	very dense	> 22.5	> 2.5
cohesive	slush	0 - 2.5	0 - 0.2
	soft	2.5 - 5	0.2 - 0.4
	stiff	5 - 12	0.4 - 0.8
	solid	12 - 25	0.8 - 1.6
	hard	> 25	> 16

### Values of rheological factor $\alpha_p$ for various soil conditions

Type of soil	Peat		Clay, silt		Sediment		Sand		Sand and gravel	
	$\alpha_p$	$E_m / P_{lim}$	$\alpha_p$	$E_m / P_{lim}$	$\alpha_p$	$E_m / P_{lim}$	$\alpha_p$	$E_m / P_{lim}$	$\alpha_p$	
preconsolidated	1	> 16	1.0	> 14	0.67	> 12	0.5	> 10	0.33	
normally consolidated	1	9 - 16	0.67	8 - 14	0.5	7 - 12	0.33	6 - 10	0.25	
underconsolidated	-	7 - 9	0.5	5 - 8	0.5	5 - 7	0.5	-	0.25	

Literature:



Menard, L. F.: *Proceedings of the 6<sup>th</sup> International Conference on Soil Mechanics and Foundation Engineering, Montreal, Vol. 2, 1965, pp. 295 - 299 (table 2.29 a 2.30).*

## Values of the Modulus of Subsoil Reaction Ep

Values of the modulus of subsoil reaction  $E_p = k_h$  [ $MN/m^3$ ]

Soil	$E_p$ Min/Max [ $MN/m^3$ ]	Average value $k_h = E_p$ [ $MN/m^3$ ]
soft clay	2 - 5	3.5
stiff clay	3 - 8	5.5
solid clay	6 - 16	11
sand naturally wet loose	6 - 13	9.5
sand naturally wet medium dense	20 - 40	30
sand naturally wet dense	45 - 90	67.5
sand aquiferous loose	4 - 8	6
sand aquiferous medium dense	10 - 20	15
sand aquiferous dense	30 - 60	45
sandy clay soft	3 - 6	4.5
sand clay stiff	5 - 9	7
sandy clay solid	8 - 17	12.5
clayey sand wet loose	4 - 9	6.5
clayey sand wet medium dense	12 - 32	22
clayey sand wet dense	24 - 44	34
clayey sand aquiferous loose	3.5 - 6.5	5
clayey sand aquiferous medium dense	7 - 11	9
clayey sand aquiferous dense	11.5 - 13.5	12.5

## Bearing Capacity of the Micropile Root Section

The micropile bearing capacity can be determined computationally using one of the approaches available in the literature and standards. The program "**Micropile**" provides a set of methods representing the basic approaches to the solution of bearing capacity of the micropile root. The analysis is accrued out according to setting in the "**Micropiles**" tab employing one of the following procedures:

- Lizzi theory** - average limit friction on root skin is specified
- Littlejohn theory** - grouting pressure is specified
- Zweck theory** - method depends on geostatic stress and soil parameters of surrounding soil
- Bowles theory** - method depends on geostatic stress and soil parameters of surrounding soil
- Véas theory** - the way the micropile is built and soil parameters of surrounding soil are specified
- root in rock** - rock parameters of surrounding soil are specified
- Bustamante** - method depends on parameters of **SPT** or **pressiometric tests (PMT)**

### Lizzi Theory

The Lizzi method is currently the most popular method used. The root bearing capacity is provided by:

$$Q = \pi \cdot d \cdot l \cdot \tau_m \cdot J$$

- where:
- $d$  - root diameter
  - $l$  - root length
  - $\tau_m$  - average limit **skin friction**
  - $J$  - coefficient reflecting influence of bore hole

Coefficient  $J$  reflects the influence of the bore hole diameter - it ranges from 1.0 for hole up to 100 *mm* and 0.8 for hole from 200 *mm*.

**Average limit skin friction** of the micropile root can be found in the literature. The program contains three tables with reference values of limit skin friction. The first one is created by the program authors using various literature sources, the second one contains values of  $\tau_m$  according to DIN 4128, and the third one includes values published in by Klein and Mišov (Inženýrské stavby, 1984). Third table contains measured values of skin friction of anchor roots for various soils, root diameters, number if groutings, etc. - using this table yields rather realistic results.

*Literature:*

*Lizzi, F. (1982). "The pali radice (root piles)". Symposium on soil and rockimprovement techniques including geotextiles, reinforced earth and modern pilingmethods, Bangkok, D-3.*

## Skin Friction of the Micropile Root

**Reference values of limit skin friction (recommend by the authors)**

<b>Soil</b>	<b>Skin friction [kPa]</b>
soft clay	40 - 60
stiff clay	65 - 85
solid clay	130 - 170
sand naturally wet, loose	110 - 150
sand naturally wet, medium dense	140 - 180
sand naturally wet, dense	170 - 230
aquiferous sand, loose	80 - 130
aquiferous sand, medium dense	120 - 160
aquiferous sand, dense	160 - 200
sandy clay, soft	50 - 70
sandy clay, stiff	75 - 95
sandy clay, solid	125 - 165
clayey sand, wet, loose	90 - 135
clayey sand, wet, medium dense	135 - 165
clayey sand, wet, dense	150 - 170
clayey sand, aquiferous, loose	80 - 105
clayey sand, aquiferous, medium dense	90 - 130
clayey sand, aquiferous, dense	115 - 155

#### Values of limit skin friction according to DIN 4128

<b>Soil</b>	<b>Average limit skin friction</b>	
	<b>piles in compression [kPa]</b>	<b>piles in tension [kPa]</b>
medium to coarse-grain sand	200	100
sand and gravel sand	150	80
cohesive soils	100	50

**Recommended parameters of anchor roots** (Mišove, Klein, Inženýrské stavby 5/1986)

Type of support of micropile in head	Final grouting press. [MPa]	Number of groutings	Root diameter [mm]	Root length [m]	Skin friction [kPa]
bedrock	-	0	120	5 - 3	1000 - 1600
semirock	0.5 - 3.0	0 - 1	120 - 220	7 - 3	300 - 1000
gravel, injectable soils	1,0	1 - 2	250 - 400	7 - 5	250 - 320
gravel, non-injectable soils	2.0 - 4.0	1 - 2	280 - 350	7 - 5	230
medium and fine-grain sand	1.5 - 4.0	2 - 3	220 - 350	12 - 7	150 - 180
cohesive stiff and solid soils	1.5 - 3.0	1 - 3	200 - 280	17 - 8	130 - 190
cohesive solid to rigid plastic soils	1.0 - 2.5	2 - 3	150 - 400	20 - 9	100 - 130
cohesive soft plastic soils	0.5 - 2.0	3 - 4	300 - 450	27 - 13.5	50 - 70

## Littlejohn Theory

When using the Littlejohn method the root bearing capacity is provided by:

$$Q = \pi \cdot d \cdot l \cdot p_i$$

where:  $d$  -root diameter  
 $l$  -root length  
 $p_i$  -magnitude of grouting pressure

It follows from experimental measurements of micropiles that their bearing capacity also depends on the course of grouting and on the grouting pressure (grouting course often governs the micropile bearing capacity). The bearing capacity considerably increases with repeated grouting. Grouting pressures range from 0.1 to 3 Mpa, in some case they may reach up to 8 Mpa. The Littlejohn method gives the bearing capacity directly proportional to the grouting pressure.

*Literature:*

LITTLEJOHN, G. S. y BRUCE, D. A. (1975).: "Rock Anchors - State of the Art. Part 1. Design". *En Ground Engineering*, Vol. 8, N° 4.

## Zweck Theory

The Zweck and Bowles methods were developed for the analysis of anchor roots - they depend mainly on the geostatic stress in the location of the micropile root. These methods arise from the same principles - the pressure magnitude is however reduced using the coefficient of

pressure at rest  $K_o$ .

$$Q = \pi \cdot d \cdot l \cdot \frac{1 + K_o}{2} \cdot \sigma_z \cdot \tan \varphi$$

$$K_o = 1 - \sin \varphi$$

where:

- $d$  -root diameter
- $l$  -root length
- $K_o$  -magnitude of pressure at rest
- $\sigma_z$  -average geostatic stress at the micropile root
- $\varphi$  -average value of friction angle at the micropile root

## Bowles Theory

The Bowles solution allows incorporating the influence of the cohesion on the root bearing capacity - therefore it is more suitable for cohesive soils.

$$Q = \pi \cdot d \cdot l \cdot \sigma_z \cdot K_o \cdot \tan \varphi + \pi \cdot d \cdot l \cdot c$$

$$K_o = 1 - \sin \varphi$$

where:

- $d$  -root diameter
- $l$  -root length
- $K_o$  -coefficient of pressure at rest
- $\sigma_z$  -average geostatic stress at the micropile root
- $\varphi$  -average magnitude of angle of internal friction at the micropile root

*Literature:*

*J.E. Bowles - Foundation Analysis and Design, McGraw Hill book Company.*

## Véas Theory

This solution takes into account the effect of geostatic stress at the micropile root and course of grouting.

**Bearing capacity of the micropile root is provided by:**

$$Q = R_{bk} + R_{sk}$$

where:

- $R_{bk}$  -bearing capacity of the micropile root
- $R_{sk}$  -skin bearing capacity of the micropile root

**Micropile skin bearing capacity:**

$$R_{sk} = \sum_{i=1}^n A_{si} \cdot q_{si}$$

where:  $n$  – number of layers passed by the micropile root  
 $A_{si}$  – area of wall of the micropile base in the  $i^{th}$  layer  
 $q_{si}$  – skin friction in the  $i^{th}$  layer

**Bearing capacity of the micropile root is provided by:**

$$R_{bk} = 0,15 \cdot R_{sk}$$

Skin friction  $q_s$  at a depth of  $z$  below the terrain surface:

$$q_s(z) = \frac{c}{F_c} + \sigma_h(z) \frac{\text{tg} \delta}{F_\varphi}$$

where:  $z$  – depth  $z$  below the terrain surface, where the magnitude of skin friction is determined  
 $c$  – effective cohesion of soil at a depth of  $z$   
 $\delta$  – friction angle along the interface of the micropile root and the soil at a depth of  $z$ :

$$\delta \in \left\langle \frac{2}{3} \varphi'; \varphi' \right\rangle$$

$\varphi'$  – effective angle of internal friction of soil at a depth of  $z$   
 $\sigma_h(z)$  – horizontal component of geostatic stress at a depth of  $z$ :  
 )

for grouting course of type IR and IRS (with monitoring of grouting pressure) and depth  $z \geq 5$  m:

$$\sigma_h(z) = K_o \cdot \sigma_v(z) + \frac{P_i}{3}$$

other cases:

$$\sigma_h(z) = K_o \cdot \sigma_v(z)$$

$K_o$  – coefficient of earth pressure at rest  
 for normally consolidated soils:

$$K_o = 1 - \sin \varphi'$$

for overconsolidated soils:

$$K_o = (1 - \sin \varphi) \cdot \sqrt{OCR}$$

$\sigma_v(z)$  – vertical component of geostatic stress at a depth of  $z$

)

$p_i$  grouting pressure for grouting course of type IR and IRS and depth  
 $z \geq 5 \text{ m}$ , in other cases  $p_i = 0$

$F_c$ , coefficients of type of application of micropile

$F_\varphi$

Literature:

Véase, Souche: *Étude du flambement de pieux partiellement immergés dans offrant latéralement une réaction élastique pure, Annales de l'ITBTP, No. 423, Sene Soils et Foundations, 187, mars - avril 1984, str. 38 - 60 (French).*

## Coefficients of Type of Application of Micropile

### Coefficients of type of application of micropile

Type of application of micropile	$F_c$ [-]	$F_\varphi$ [-]
Newly constructed foundations	1.50	1.50
Existing foundations	1.20	1.20

## Bearing Capacity of the Root in Rock

This solution is suitable for the micropile root reaching into rocks with index  $RQD > 60$  or having the strength in simple compression  $\sigma_c > 20 \text{ MPa}$  (ISRM < III). The root bearing capacity is given by:

$$Q = A_s \cdot q_{sr} + A_b \cdot q_{br}$$

where:

- $A_s$  -area of wall of micropile root
- $q_{sr}$  -skin friction in rock
- $A_b$  -area of the micropile root
- $q_{br}$  -bearing capacity of the micropile root in rock

Literature:

*Guía para el proyecto y la ejecución de micropilotes en obras de carretera, Ministerio de fomento, 2005 (Spanish).*

## Skin Friction and Bearing Capacity of the Micropile Root in Rock

**Skin friction in rock  $q_{sr}$  and bearing capacity of the micropile root in rock  $q_{br}$**

Type of rock	$q_{sr}$ [MPa]	$q_{br}$ [MPa] <sup>1)</sup>
Sediments	0.15 - 0.40	$0.07\sigma_c$
Slates and fylits	0.20 - 0.30	$0.07\sigma_c$
Sandstones	0.30 - 0.45	$0.07\sigma_c$
Lime stones and dolomites	0.40 - 0.50	$0.10\sigma_c$
granites, basalts	0.40 - 0.60	$0.10\sigma_c$

1)  $\sigma_c$  - strength in simple tension MPa

## Bustamante (SPT, Pressiometer PMT)

The analysis of bearing capacity of the micropile root section is based on the results of standard penetration tests (SPT) or pressiometric tests (PMT).

The magnitude of skin friction of the micropile root  $q_s$  [MPa] is available from the graphs according to Bustamante, which depend on the type of soil and the injection technology.

Shaft resistance of the micropile root  $R_s$  follows from:

$$R_s = \sum \pi d_r l_r q_s$$

where:

- $d_r$  - diameter of the micropile root
- $l_r$  - length of the micropile root
- $q_s$  - skin friction of the micropile root (value determined from the graph)

Base resistance of the micropile root  $R_b$  may not be considered in the analysis or is taken as:

$$R_b = 0,15 R_s$$

where:  $R_s$  - Shaft resistance of the micropile root

Base resistance of the micropile root  $R_b$  is assumed in the program in the form:

$$R_b = A_p k_p p_{LM}$$

where:

- $A_p$  - cross-section area of the base of the micropile root
- $k_p$  - soil factor in the vicinity of the base of the micropile root
- $p_{LM}$  - limit pressure according to Menard

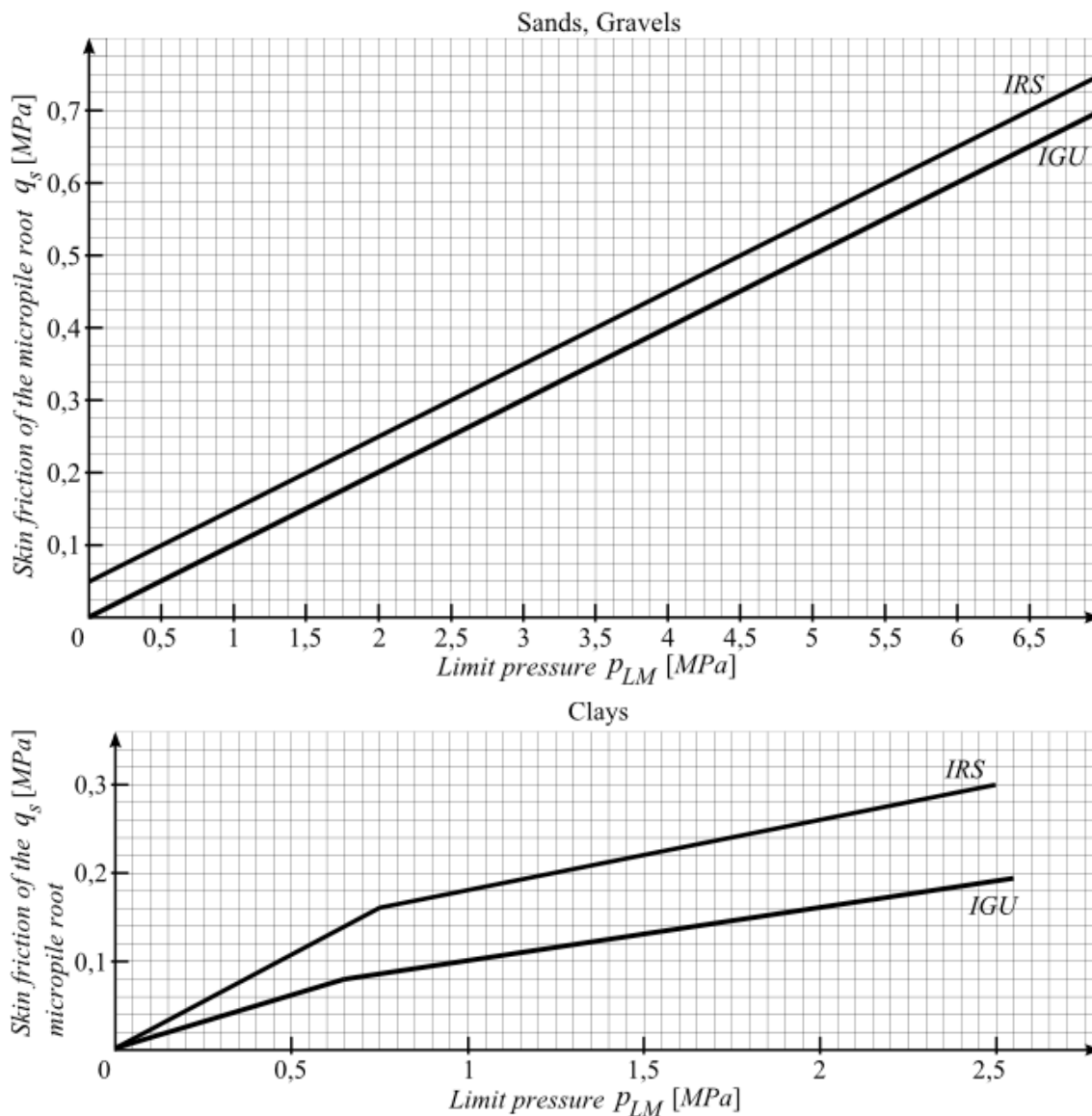
## Skin Friction of the Micropile Root - Graphs

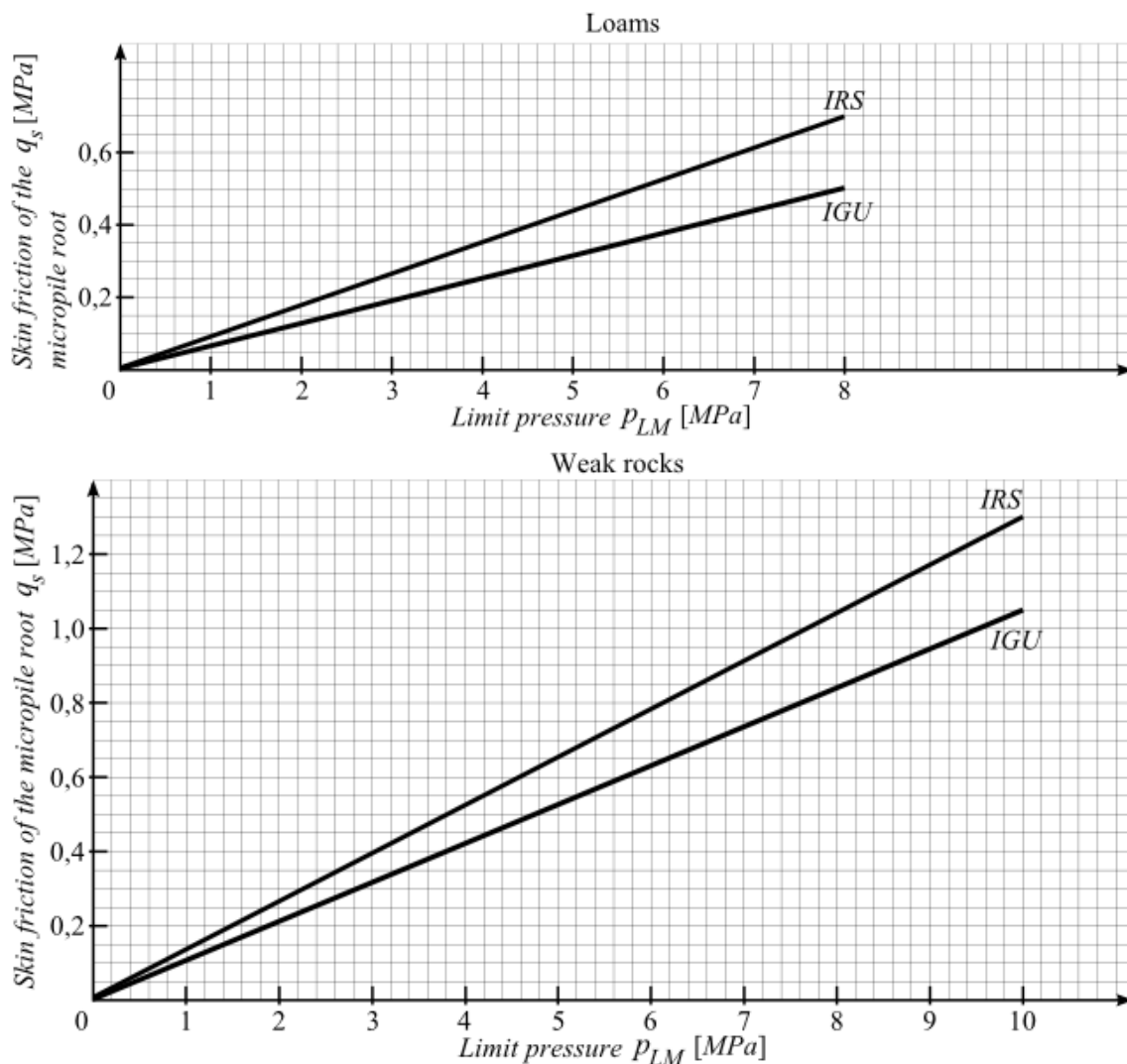
The analysis of shaft resistance of the micropile root  $R_s$  depends considerably on the type of injection of micropile root. The following options of injection are considered in the program:



- **IRS**: repeated selective injection of the micropile root over sleeves performed locally (**Tube-à-Manchette**),
- **IGU**: unified global pressure injection (**Looped Tube Systems**).

The following graphs for the analysis of **skin friction of the micropile root**  $q_s$  [MPa] are built in the program:





The displayed graphs consider on the horizontal axis the limit pressure  $p_{LM}$  determined from the **pressiometric tests (PMT)**. In case of **SPT tests** the same graphs are used, but the limit pressure  $p_{LM}$  [MPa] is then determined as the  **$n$ -multiple** of the number of blows  $N$  for the interval of penetration depth  $d = 0.3$  m, i.e.  $SPT [N/0.3 m]$ . For individual types of soils the values of limit pressure  $p_{LM}$  according to Menard are as follows:

- **sand, gravel, silt and weak rock:**  $p_{LM} = SPT / 20$ ,
- **clays:**  $p_{LM} = SPT / 15$ .

For example, for the sandy soil and the value of the multiple of number of blows  $SPT = 120$  the limit pressure is given by  $p_{LM} = SPT / 20 = 120 / 20 = 6.0$  MPa.

Next, for example, for the clayey soil and the value of the multiple of number of blows  $SPT = 30$  the limit pressure is provided by  $p_{LM} = SPT / 15 = 30 / 15 = 2.0$  MPa.

The vertical axis provides the value of skin friction of the micropile root  $q_s$  depending on the value of limit pressure  $p_{LM}$  and the applied type of injection (**IRS** or **IGU**, respectively).

## Field Testing

Some of the GEO5 programs exploit as input parameters for the analysis several types of field tests (in situ). The following tests are considered:

- **Cone penetration tests (CPT)** - "Pile CPT" program
- **Standard penetration tests (SPT)** - "Micropile" program (Bustamante method)
- **Pressiometric tests (PMT)** - "Sheeting Check" and "Anti-Slide Pile" program (modulus of subsoil reaction according to Menard or according to NF P 94-282), "Micropile" program (Bustamante method)
- **Dilatometric tests (DMT)** - "Spread Footing" program (settlement analysis using DMT), "Sheeting Check" and "Anti-Slide Pile" program (modulus of subsoil reaction specified by dilatometric test)

## Cone Penetration Tests (CPT)

The **cone penetration test (CPT)** is based on pushing a penetration cone using a system of penetration rods with constant velocity of (20 - 25 mm/s) into the soil. During the penetration test the values of the **cone resistance**  $q_c$  and the **local skin friction**  $f_s$ , respectively are recorded. The cone resistance thus represents in general the resistance against penetration of a cone spike into the soil (subsoil). The diameter of the tip of cone spike is typically in the range of 25 - 50 mm.

The **cone (penetration) resistance**  $q_c$  [MPa] represents the ratio of the measured force on the cone tip  $Q_c$  and the area of normal projection of the cone tip  $A_c$ .

The **local skin friction**  $f_s$  [kPa] represents the ratio of the measured force on the friction of sleeve  $F_s$  and the area of its skin  $A_s$ .

The result of cone penetration test is its distribution plotted as a graph. The evaluation of cone penetration tests (CPT) serve as an input parameter for the analyses in the "Pile CPT" program.

### Import of CPT

The results of cone penetration tests (CPT) can be imported into the program by inserting the file in different formats (eg. \*.TXT, \*.CPT, \*.GI3).

*Literature:*

*EN ISO 22476-1: Geotechnical investigation and testing - Field testing. Part 1: Electrical cone and piezocone penetration test, 2013.*

*EN ISO 22476-12: Geotechnical investigation and testing - Field testing. Part 12: Mechanical cone penetration test (CPTM), 2009.*

## Standard Penetration Tests (SPT)

The **standard penetration test (SPT)** primarily allows to determine the strength and deformation parameters of soils. The standard penetration test is based on a sampling device being punched by a ram weight  $m$  [kg] that is being dropped down from a height  $h$  [m] on to an anvil or punching head. The number of blows  $N$ , needed to penetrate a sampling device by a certain interval of penetration depth, is called the **penetration resistance**.

The results of standard penetration test are presented as a number of blows  $N$  over a certain distance, to which the device (ram with a head or anvil) punched into the soil or rock,

respectively. This distance is called the **interval of penetration depth**  $d$  [m]. The value of this parameter is commonly assumed being equal to 0.3 m. For some types of tasks this value can be changed.

**Energetic ratio of testes device**  $E_r$  [%] represents the ratio of real energy  $E_{meas}$  and the calculated energy  $E_{theor}$  of the ram.

Other important parameters to evaluate standard penetration test are so called **correlations** or correction factors (e.g. loss of energy due to length of the system of rods, influence of overburden in sands, etc.). The current design methods based on the principle of SPT tests have an empirical character and therefore it is necessary to use the corresponding parameters correctly modified. The program assumes the following two ways of adopting the correction factors:

- **correlation  $C_N$  for vertical stress  $\sigma'_V$**  - represents the influence of weight of overburden in sands. The values of the correction factor  $C_N$  greater than 1.5 must not be used (according to EN ISO 22476-3 recommendations).
- **user's correlation  $\lambda$  [-]** - represents the loss of energy due to the length of the system of rods for sandy soils. This correction factor can be specified in the programs in the range of (0.5 - 1.0).

**Table of built in types of correlations**

Type	Type of consolidation	Relative compactness $l_p$ [%]	Correlation factor $C_N$
Type 1 - EN ISO 22476-3 (Tab. A2)	Normally consolidated	40 - 60	$C_N = \frac{200}{100 + \sigma'_V}$
Type 2 - EN ISO 22476-3 (Tab. A2)		60 - 80	$C_N = \frac{300}{200 + \sigma'_V}$
Type 3 - EN ISO 22476-3 (Tab. A2)	Over-consolidated	-	$C_N = \frac{170}{70 + \sigma'_V}$
Type 4 - EN ISO 22476-3	Normally consolidated sands	-	$C_N = \sqrt{\frac{98}{\sigma'_V}}$
Type 5 - FHWA (1998), Peck (1974)	-	-	$C_N = \left(\frac{100}{\sigma'_V}\right)^{0,4}$

where:  $\sigma'_V$  - Effective vertical stress

The result of standard penetration test is its process plotted as a graph. The evaluation of standard penetration tests (SPT) are used as input parameters for the analyses in the "**Micropile**" program (Bustamante method).

### Import of SPT

The results of standard penetration tests (SPT) can be imported into the program by inserting

the file in different formats (eg. **\*.TXT**, **\*.CSV**, **\*.XLSX**, **\*.ODS**).

*Literature:*

*EN ISO 22476-3: Geotechnical investigation and testing - Field testing. Part 3: Standard penetration test, 2005.*

## Pressiometric Tests (PMT)

The **pressiometric test (PMT)** consists of pressiometric probe placed in the tested soil and gradually filled with water. The subsequent swelling of soil or rock around the hole is determined as a dependence of the measured volume of water on the pressure increment that is gradually increased in a priory defined time intervals.

The pressiometric test provides the following parameters as a function of depth  $z$  [m]:

- **pressiometric (Menard) modulus**  $E_m$  [MPa] - is obtained from the pressiometric test and depends on the type of **sheath of probe** (rubber sleeve, perforated casing).
- **limit pressure**  $p_{LM}$  [MPa] - represents an increment of water pressure in the testing probe depending on the volume change of soil or rock, respectively.

The result of pressiometric test is its process plotted as a graph. The evaluation of pressiometric tests (PMT) are used as input parameters for the analyses in the "**Micropile**" program (**Bustamante** method), "**Sheeting Check**" and "**Anti-Slide Pile**" program (modulus of subsoil reaction according to **Menard** or according to the **NF P 94-282**).

### Import of PMT

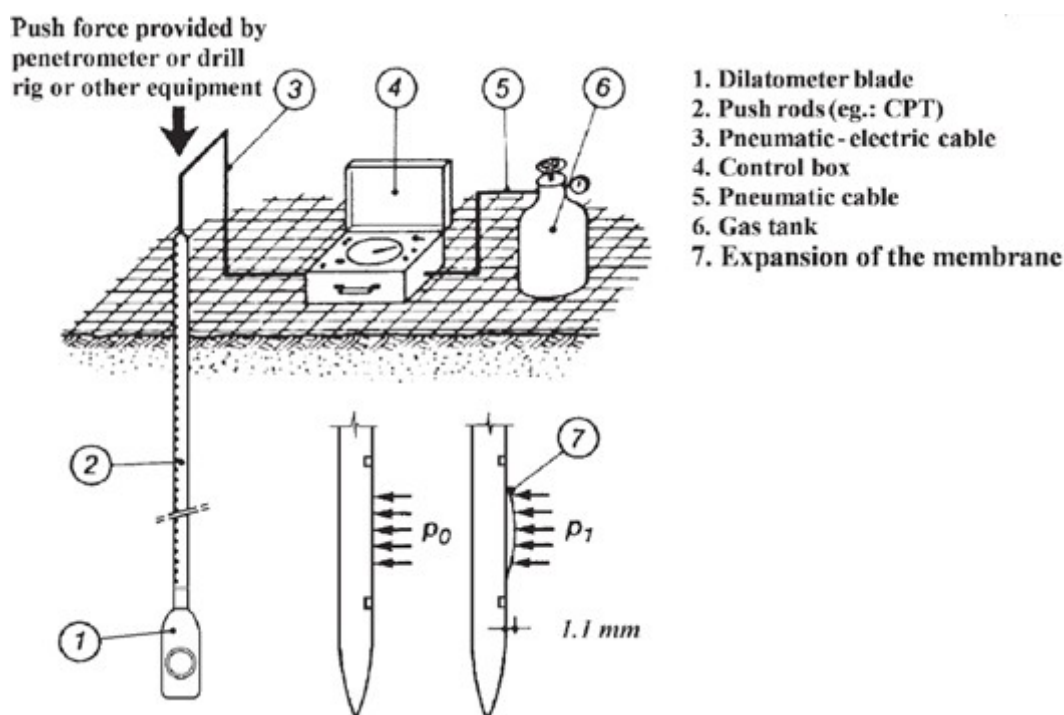
The results of pressiometric tests (PMT) can be **imported** into the program by inserting the file in different formats (eg. **\*.TXT**, **\*.CSV**, **\*.XLSX**, **\*.ODS**).

*Literature:*

*EN ISO 22476-4: Geotechnical investigation and testing - Field testing. Part 4: Menard pressuremeter test, 2005.*

## Dilatometric Test (DMT)

The **dilatometric test (DMT)** is performed by using a dilatometer, which operates on the principle of verification values by using the displacements of the inductive sensors (with a sensitivity of up to 0.001 mm). The advantage of these tests is a more accurate description of displacement and deformation of foundation soil.



General Layout of Dilatometer Test (source: [1], Figure 2, pp. 10)

The result of dilatometric test is its process plotted as a graph. The evaluation of dilatometric tests (DMT) are used as input parameters for the analyses in the "**Spread Footing**" program (settlement analysis using DMT), "**Sheeting Check**" and "**Anti-Slide Pile**" program (modulus of subsoil reaction specified by dilatometric test).

### Import of DMT

The results of dilatometric test (DMT) are imported into the program by inserting the file in format **UNI (\*.uni)**. It is a **standardized and universal format** for import of the measured data obtained from dilatometric tests, which is used in the world.

*Literature:*

Marchetti, S., Monaco, P., Totani, G. & Calabrese, M.: *The Flat Dilatometer Test (DMT) in soil investigations. A Report by the ISSMGE Committee TC16, University of L'Aquila, Italy, 2001, 48 p.*

## Settlement Analysis

One of the following methods is available to compute settlement:

- Using the oedometric modulus
- Using the compression constant
- Using the compression index
- According to NEN (Buismann, Ladde)
- Using the Soft soil model
- According to Janbu theory
- Using the DMT (constrained soil modulus)

The program offers two options to constrain the depth of influence zone:

- Exploiting the **theory of structural strength**
- Using the percentage of the **magnitude of geostatic stress**

The theory of elasticity (Boussinesq theory) is employed to determine **stress in a soil state** in all methods available for the settlement analysis.

General **theories of settlement** serve as bases in all the above methods.

When computing settlement below the footing bottom the programs first calculates the **stress in the footing bottom** and then determines the **overall settlement and rotation of foundation**.

The general approach in all theories draws on subdividing the subsoil into layers of a different thickness based on the depth below the footing bottom or ground surface. Vertical deformation of each layer is then computed - the overall settlement is then defined as a sum of partial settlements of individual layers within the influence zone (deformations below the influence zone are either zero or neglected):

$$s = \sum \Delta s_i$$

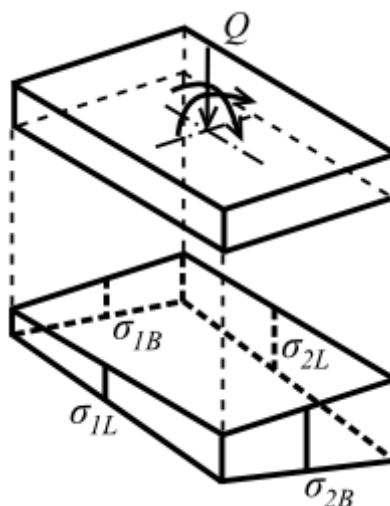
where:  $s$  - settlement  
 $s_i$  - settlement of the  $i^{th}$  layer

## Stress in the Footing Bottom

The stress in the footing bottom can be assumed as:

- **rectangular** (uniform in the footing bottom)
- **general** (trapezoidal) with different edge values

General distribution of stress follows from figure:



*Stress in the footing bottom*

where:

$$\sigma_{1B,z} = \frac{Q}{l \cdot b} \pm \frac{Q \cdot e_b}{W_b}$$

$$e_b = \frac{M_x + H_y \cdot t + N \cdot p_x}{Q}$$

$$W_b = \frac{1}{6} \cdot l \cdot b^2$$

$$\sigma_{1L,z} = \frac{Q}{l.b} \pm \frac{Q.e_t}{W_t} \quad e_t = \frac{-M_y + H_x.t + N.p_y}{Q} \quad W_t = \frac{1}{6}.l.b^2$$

where:

- $Q$  - vertical load of footing
- $l, b$  - footing width and length
- $e_b$  - load eccentricity
- $M$  - moment acting on the footing
- $H$  - horizontal force
- $N$  - normal force at eccentric footing
- $p$  - column axis offset from the footing center

If in some points the stress becomes negative, the program continues with adjusted dimensions  $b^*l$  while excluding tension from the analysis. Before computing the stress distribution due to surcharge the stress in the footing bottom is reduced by the geostatic stress in the following way:

$$\sigma_{ol} = \max(\sigma_{ol} - \sigma_{or,sp}; 0)$$

There are three options in the program to specify the geostatic stress in the footing bottom:

- **From the original ground** It is therefore considered, whether the footing bottom in the open pit measured from the original ground is free of stress for the time less than needed for soil bulking and subsequent loss of stress in the subsoil.
- **From the finished grade** The same assumptions as above apply.
- **Not considered at all.**

## Overall Settlement and Rotation of Foundation

The foundation settlement is substantially influenced by the overall stiffness of the system represented by foundation structure and foundation soil given by:

$$k = \frac{E_{basic}.t^3}{E_{def,av}.l^3}$$

where:

- $E_{basic}$  - modulus of elasticity of footing
- $t$  - foundation thickness
- $E_{def,av}$  - weighted average of the deformation modulus up to depth of influence zone
- $l$  - footing dimension in the direction of searched stiffness

For  $k > 1$  the foundation is assumed to be rigid and as a representative point for the determination of its settlement is assumed the **characteristic point** (distant by 0.37 times the foundation dimension from its axis).

For  $k < 1$  the foundation structure is assumed to be compliant and as a representative point for the determination of foundation settlement is assumed the **foundation center point**.

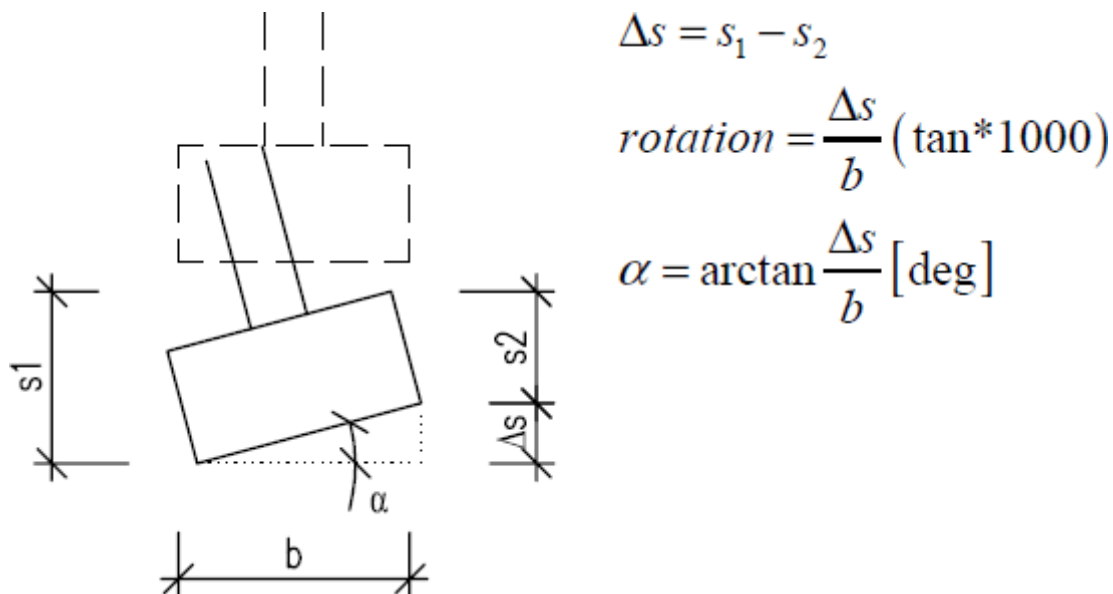
The **foundation rotation** is determined from the difference of settlements of center and of



individual edges.

For example, if the rotation in direction of  $x$  equals to  $x = 0.944(\tan^*1000)$ , then the value of rotation in degrees  $[\circ]$  equals to  $\alpha_x = (\arctan 0.944) / 1000 = 4.3 \cdot 10^{-2} \circ$ .

Next, for example, if the rotation in direction of  $y$  equals to  $y = 2.360(\tan^*1000)$ , then the value of rotation in degrees  $[\circ]$  equals to  $\alpha_y = (\arctan 2.360) / 1000 = 6.7 \cdot 10^{-2} \circ$ .



Rotation of spread footing - principle calculation

## Influence of Foundation Depth and Incompressible Subsoil

When computing settlement it is possible to account for the **influence of foundation depth** by introducing the reduction coefficient  $\kappa_1$ :

for strip footing:

$$\kappa_1 = 1 + 0,61 \cdot \arctg \frac{d}{z}$$

for spread footing:

$$\kappa_1 = 1 + 0,35 \cdot \arctg \left( 1,55 \cdot \frac{d}{z} \right)$$

where:  $d$  - depth of footing bottom  
 $z$  - depth under footing bottom

**Influence of incompressible layer** is introduced into the analysis by the reduction coefficient  $\kappa_2$ :

$$\kappa_2 = 1 - e^{\left( \frac{z_{ic}}{z} \cdot \ln 0,25 + \ln 0,8 \right)}$$

where:  $z_{ic}$  - depth of rigid base under footing bottom  
 $z$  - depth under footing bottom

Incorporating the above coefficients allows **transformation** of the vertical component of stress  $\sigma_z$  such that the actual depth is replaced by a **substitute value**  $z_r$  given by:

$$z_r = \kappa_1 \cdot \kappa_2 \cdot z$$

where:  $\kappa_1$  - coefficient of footing bottom depth  
 $\kappa_2$  - coefficient of rigid base  
 $z$  - depth under footing bottom

## Influence of Sand-Gravel Cushion

If the sand-gravel cushion is specified below the spread footing, the material parameters  $X$  in individual layers are computed in the following way:

For layer  $h_{a,i}$ :

$$X_i < X_c$$

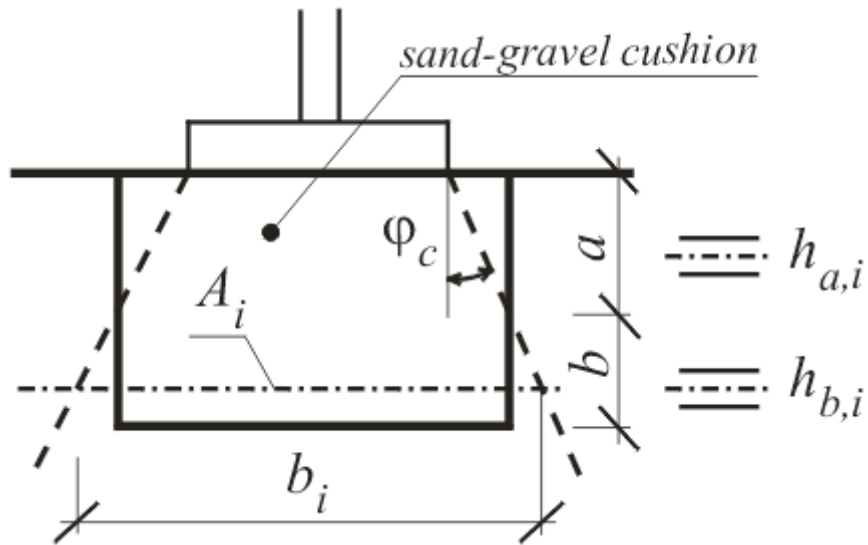
where:  $X_i$  - material parameters at  $i^{th}$  layer  
 $X_c$  - material parameters of sand-gravel cushion

For layer  $h_{b,i}$ :

$$X_i = \frac{(A_i - A_c) \cdot X_c + A_i \cdot X_c}{A_i}$$

$$A_i = b_i \cdot l_i$$

where:  $A_c$  - area of sand-gravel cushion  
 $X_c$  - material parameters of sand-gravel cushion  
 $X_{b,i}$  - material parameters of  $b,i$  layer  
 $b_i$  - cushion widths in the  $i^{th}$  layer  
 $l_i$  - cushion length in the  $i^{th}$  layer



Analysis  $X_i$  in the sand-gravel cushion

## Analysis Using the Oedometric Modulus

Equation to compute compression of an  $i^{th}$  soil layer below foundation having a thickness  $h$  arises from the definition of **oedometric modulus**  $E_{oed}$ :

$$s_i = \sum \frac{\sigma_{z,i} h_i}{E_{oed,i}}$$

- where:
- $\sigma_{z,i}$  - vertical component of incremental stress in the middle of  $i^{th}$  layer
  - $h_i$  - thickness of the  $i^{th}$  layer
  - $E_{oed,i}$  - oedometric modulus of the  $i^{th}$  layer

The oedometric modulus  $E_{oed}$  can be specified for each soil either as constant or with the help of an **oedometric curve** ( $\sigma_{eff}/\epsilon$  relation). When using the oedometric curve the program assumes for each layer the value of  $E_{oed}$  corresponding to a given range of original and final stress. If the value of oedometric modulus  $E_{oed}$  is not available, it is possible to input the deformation modulus  $E_{def}$  and the program carries out the respective transformation.

$$E_{oed} = \frac{E_{def}}{\beta}$$

where:

$$\beta = 1 - \frac{2\nu^2}{1 - \nu}$$

- where:
- $\nu$  - Poisson's ratio
  - $E_{def}$  - deformation modulus

## Analysis Using the Compression Constant

Equation to compute compression of an  $i^{th}$  soil layer below foundation having a thickness  $h$  arises from the definition of **compression constant**  $C$ :

$$s = \frac{h_i}{C_i} \cdot \ln \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

where:

- $\sigma_{or,i}$  - vertical component of original geostatic stress in the middle of  $i^{th}$  layer
- $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
- $h_i$  - thickness of the  $i^{th}$  layer
- $C_i$  - compression constant in the  $i^{th}$  layer

The program allows for inputting either the compression constant  $C_i$  or the **compression constant**  $C_{10}$  (the program itself carries out the transformation).

*Literature:*

Arnold Verruijt: *Soil mechanics, Delft University of Technology, 2001, 2006,*  
<http://geo.verruijt.net/>.

## Analysis Using the Compression Index

Equation for settlement when employing the **compression index**  $C_c$  of the  $i^{th}$  layer arises from the formula:

$$s_i = C_{c,i} \frac{h_i}{1 + e_0} \cdot \log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

where:

- $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer
- $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
- $e_0$  - initial void ratio
- $h_i$  - thickness of the  $i^{th}$  layer
- $C_{c,i}$  - compression index in the  $i^{th}$  layer

*Literature:*

Arnold Verruijt: *Soil mechanics, Delft University of Technology, 2001, 2006,*  
<http://geo.verruijt.net/>.

## Analysis According to NEN (Buismann, Ladd)

This method computes both the primary and secondary settlement. When computing the method accounts for overconsolidated soils and differentiates between two possible cases:

- Sum of the current vertical effective stress in the soil and stress due to external surcharge

is less than the preconsolidation pressure so that only additional surcharge is considered.

- Sum of the current vertical effective stress in the soil and stress due to external surcharge is greater than the preconsolidation pressure so that the primary consolidation is set on again. The primary settlement is then larger when compared to the first case.

### Primary settlement

Primary settlement of the  $i^{th}$  layer of overconsolidated soil ( $OCR > 1$ ) is provided by:

for:  $\sigma_{or} + \sigma_z \leq \sigma_p$  (sum of the current vertical stress and its increment is less than the preconsolidation pressure):

$$s_i = C_{r,i} \frac{h_i}{1 + e_0} \cdot \log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

for:  $\sigma_{or} + \sigma_z > \sigma_p$  (sum of the current vertical stress and its increment is greater than the preconsolidation pressure):

$$s_i = C_{r,i} \frac{h_i}{1 + e_0} \cdot \log \frac{\sigma_{p,i}}{\sigma_{or,i}} + C_{c,i} \frac{h_i}{1 + e_0} \cdot \log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{p,i}}$$

- where:
- $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer
  - $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
  - $\sigma_{p,i}$  - preconsolidation pressure in the  $i^{th}$  layer
  - $e_0$  - initial void ratio
  - $h_i$  - thickness of the  $i^{th}$  layer
  - $C_{c,i}$  - **compression index** in the  $i^{th}$  layer
  - $C_{r,i}$  - **recompression index** in the  $i^{th}$  layer

Primary settlement of the  $i^{th}$  layer of normally consolidated soil ( $OCR = 1$ ) reads:

$$s_i = C_{c,i} \frac{h_i}{1 + e_0} \cdot \log \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

- where:
- $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer
  - $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
  - $e_0$  - initial void ratio
  - $h_i$  - thickness of the  $i^{th}$  layer
  - $C_{c,i}$  - **compression index** the  $i^{th}$  layer

### Secondary settlement

Secondary settlement of the  $i^{th}$  layer assumes the form:

for:  $\sigma_{or} + \sigma_z \leq \sigma_p$  (sum of the current vertical stress and its increment is less than the

preconsolidation pressure):

$$s_{i,d} = C_{\alpha,i} \cdot h_i \cdot \left( \log \frac{t_s}{t_p} \right)$$

for:  $\sigma_{or} + \sigma_z > \sigma_p$  (sum of the current vertical stress and its increment is greater than the preconsolidation pressure):

$$s_{i,d} = C_{\alpha,i} \cdot h_i \cdot \left( \log \frac{t_s}{t_p} \right)$$

- where:
- $h_i$  - thickness of the  $i^{th}$  layer
  - $C_{\alpha,i}$  - secondary compression index below preconsolidation pressure in the  $i^{th}$  layer
  - $C_{\alpha}$  - index of secondary compression in the  $i^{th}$  layer
  - $t_p$  - time to terminate primary consolidation
  - $t_s$  - time required for secondary settlement

If we specify the value of preconsolidation index of secondary compression the same as for the index of secondary compression, the program does not take into account in the computation of secondary settlement the effect of preconsolidation pressure.

*Literature:*

*Netherlandish standard NEN6740, 1991, Geotechniek TGB1990 Basisen en belastingen, Nederlands normalisatie-Institut.*

## Analysis Using the Soft Soil Model

The analysis employs the modified compression index  $\lambda$  and is based on the Soft soil elastic-plastic model developed in university of Cambridge. The soil deformation assumes the volumetric strain to be linearly dependent on the change of effective mean stress  $\varepsilon$  plotted in natural logarithmic scale. The settlement of the  $i^{th}$  layer is then provided by:

$$s_i = \lambda_i \cdot h_i \cdot \ln \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

- where:
- $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer
  - $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
  - $h_i$  - thickness of the  $i^{th}$  layer
  - $\lambda$  - modified compression index in the  $i^{th}$  layer

The analysis requires inputting the modified compression index  $\lambda$  usually obtained from triaxial laboratory measurements.

If the modified compression index  $\lambda$  is not known, it is possible to specify the compression index  $C_C$  together with an average value of the void ratio  $e$  (if that is also not known it is

sufficient to provide the initial void ratio  $e_0$ ) and the program then performs an approximate computation of the modified compression index  $\lambda$  using the available information.

*Literature:*

*Burland J.B. The yielding and dilatation of clay (correspondence), Géotechnique, 15 (2), 1965, str. 211-214.*

## Analysis According to the Janbu Theory

It is based on principles of nonlinear elastic deformation, where the stress-strain relationship is described by a function of two dimensionless parameters unique for a given soil. The parameters are the **exponent  $j$**  and the **Janbu modulus  $m$** . Equations describing the settlement are obtained by specifying  $\varepsilon$  from the definition of deformation modulus  $E_t$  and by subsequent integration. The program allows the user to compute settlement for the following types of soil:

- Cohesionless soils
- Coarse - grained soil
- Sands and silts
- Overconsolidated sands and silts
- Cohesive soils
- Overconsolidated cohesive soils

*Literature:*

*Method of settlement computation for various types of soils, Soil Mechanics and foundation engineering, Springer, 7 (3), 1970, str, 201-206.*

## Analysis for Cohesionless Soils

For cohesionless soils the stress exponent is not equal to zero. For layered subsoil the resulting settlement equals to the sum of partial settlements of individual layers:

$$s_i = \frac{h_i}{m_i \cdot j_i} \left[ \left( \frac{\sigma_{or,i} + \sigma_{z,i}}{100} \right)^j - \left( \frac{\sigma_{or,i}}{100} \right)^j \right]$$

- where:
- $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer
  - $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
  - $j_i$  - **stress exponent** in the  $i^{th}$  layer
  - $m_i$  - **Janbu modulus** in the  $i^{th}$  layer
  - $h_i$  - thickness of the  $i^{th}$  layer

## Analysis for Coarse-Grained Soils

For dense coarse-grained soils (e.g. ice soil) the stress-deformation (settlement) relationship is usually assumed as "elastic", i.e. the **stress exponent  $j$**  is equal to one. Thus for  $j = 1$  and the

reference stress  $\sigma_r = 100 \text{ kPa}$  the resulting settlement equals to the sum of partial settlements of individual layers:

$$s_i = \frac{h_i}{100 \cdot m_i} \cdot (\sigma_{z,i})$$

where:  $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression - i.e. change of effective stress

$m_i$  - Janbu modulus in the  $i^{\text{th}}$  layer

$h_i$  - thickness of the  $i^{\text{th}}$  layer

## Analysis for Sands and Silts

For sands and silts the stress exponent  $j$  receives the value around 0,5, for the reference stress  $\sigma_r = 100 \text{ kPa}$  the resulting settlement equals to the sum of partial settlements of individual layers. It can be derived from the following formula:

$$s_i = \frac{h_i}{5 \cdot m_i} \cdot \left( \sqrt{\sigma_{or,i} + \sigma_{z,i}} - \sqrt{\sigma_{or,i}} \right)$$

where:  $\sigma_{or,i}$  vertical component of geostatic stress in the middle of  $i^{\text{th}}$  layer

$\sigma_{z,i}$  vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression

$m_i$  Janbu modulus in the  $i^{\text{th}}$  layer

$h_i$  thickness of the  $i^{\text{th}}$  layer

## Analysis for Overconsolidated Sands and Silts

Providing the final stress in soil exceeds the preconsolidation pressure ( $\sigma_{or} + \sigma_z > \sigma_p$ ), the settlement of layered subsoil is found from the following equation:

$$s_i = \frac{h_i}{5 \cdot m_{r,i}} \cdot \left( \sqrt{\sigma_{p,i}} - \sqrt{\sigma_{or,i}} \right) + \frac{h_i}{5 \cdot m_i} \cdot \left( \sqrt{\sigma_{or,i} + \sigma_{z,i}} - \sqrt{\sigma_{p,i}} \right)$$

where:  $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{\text{th}}$  layer

$\sigma_{p,i}$  - preconsolidation pressure in the  $i^{\text{th}}$  layer

$\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression

$m_i$  - Janbu modulus in the  $i^{\text{th}}$  layer

$m_{r,i}$  - Janbu modulus of recompression in the  $i^{\text{th}}$  layer

$h_i$  - thickness of the  $i^{\text{th}}$  layer

If the stress due to surcharge does not cause the final stress to exceed the preconsolidation pressure ( $\sigma_{or} + \sigma_z \leq \sigma_p$ ), it is possible to assume the following forms of equations for the



computation of settlement of layered sand or silt subsoil:

$$s_i = \frac{h_i}{5 \cdot m_{r,i}} \cdot \left( \sqrt{\sigma_{or,i} + \sigma_{z,i}} - \sqrt{\sigma_{or,i}} \right)$$

- where:
- $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer
  - $\sigma_{p,i}$  - preconsolidation pressure in the  $i^{th}$  layer
  - $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
  - $m_{r,i}$  - Janbu modulus of recompression in the  $i^{th}$  layer
  - $h_i$  - thickness of the  $i^{th}$  layer

## Analysis for Cohesive Soils

In case of cohesive soils the stress exponent is equal to zero. For normally consolidated soils we obtain from the definition of the tangent modulus of deformation (by modification and subsequent integration)  $E_t$  equation for the settlement of layered subsoil formed by cohesive soils in the form:

$$s_i = \frac{h_i}{m_i} \cdot \ln \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

- where:
- $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer
  - $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
  - $m_i$  - Janbu modulus in the  $i^{th}$  layer
  - $h_i$  - thickness of the  $i^{th}$  layer

## Analysis for Overconsolidated Cohesive Soils

Most cohesive soils in the original order except very young or organic clays are overconsolidated. If the final stress in the soil exceeds overconsolidation stress ( $\sigma_{or} + \sigma_z > \sigma_p$ ) than the settlement of the layered subsoil composites from cohesive soils is computed from following relation:

for:  $\sigma_{or} + \sigma_z > \sigma_p$

$$s_i = \frac{h_i}{m_{r,i}} \cdot \ln \frac{\sigma_{p,i}}{\sigma_{or,i}} + \frac{h_i}{m_i} \cdot \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{p,i}}$$

for:  $\sigma_{or} + \sigma_z \leq \sigma_p$

$$s_i = \frac{h_i}{m_{r,i}} \cdot \frac{\sigma_{or,i} + \sigma_{z,i}}{\sigma_{or,i}}$$

- where:  $\sigma_{or,i}$  - vertical component of geostatic stress in the middle of  $i^{th}$  layer

- $\sigma_{p,i}$  - preconsolidation pressure in the  $i^{th}$  layer
- $\sigma_{z,i}$  - vertical component of incremental stress (e.g. stress due to structure surcharge) inducing layer compression
- $m_i$  - Janbu modulus in the  $i^{th}$  layer
- $m_{r,i}$  - Janbu modulus of recompression in the  $i^{th}$  layer
- $h_i$  - thickness of the  $i^{th}$  layer

## Settlement Analysis Using DMT (Constrained Soil Modulus)

Constrained soil modulus  $M_{DMT}$  [MPa] is defined as the vertical drained confined tangent modulus at  $\sigma_{vo}$ . Modulus  $M_{DMT}$  is obtained from dilatometric tests (DMT).

If the value of the constrained soil modulus  $M_{DMT}$  is not available, it is possible to input the coefficient of volume compressibility  $m_V$  [ $m^2/MN$ ] (determined from the oedometric test) and the program carries out the respective transformation:

$$M_{DMT} = \frac{1}{m_V}$$

- where:
- $M_{DMT}$  - constrained soil modulus
  - $m_V$  - coefficient of volume compressibility

The analysis employs the constrained soil modulus  $M_{DMT}$  or coefficient of volume compressibility  $m_V$  and is based on Marchetti theory. This approach being based on linear elasticity, provides a settlement proportional to the load and is unable to provide non linear predictions.

The settlement of the  $i^{th}$  layer is then provided by:

$$s_i = \frac{\sigma_{z,i} \cdot h_i}{M_{DMT}}$$

- where:
- $\sigma_{z,i}$  - vertical component of incremental stress in the middle of  $i^{th}$  layer
  - $h_i$  - thickness of the  $i^{th}$  layer
  - $M_{DMT}$  - constrained soil modulus

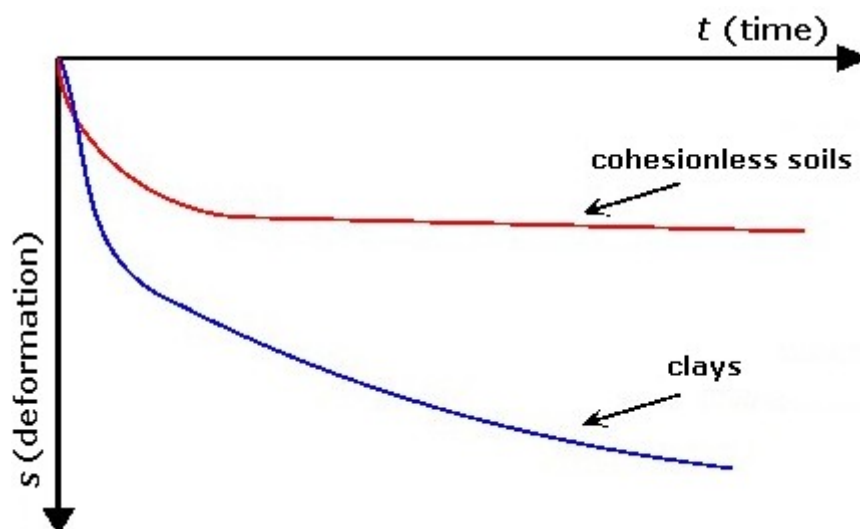
### Literature:

Marchetti, S., Monaco, P., Totani, G. & Calabrese, M.: *The Flat Dilatometer Test (DMT) in soil investigations. A Report by the ISSMGE Committee TC16, University of L'Aquila, Italy, 2001, 48 p.*

## Theory of Settlement

If the stress change in the soil or in the currently build earth structure, caused by ground

surface surcharge, is known, it is possible to determine the soil deformation. The soil deformation is generally inclined and its vertical component is termed the settlement. In general, the settlement is non-stationary dependent on time, which means that it does not occur immediately after introducing the surcharge, but it rather depends on consolidation characteristics of a soil. Permeable, less compressible soils (sand, gravel) deform fast, while saturated, low permeability clayey soils experience gradual deformation called consolidation.



*Time dependent settlement of soils*

Applied load yields settlement, which can be subdivided based on time dependent response into three separate components:

- Instantaneous settlement (initial)
- Primary settlement (consolidation)
- Secondary settlement (creep)

### **Instantaneous settlement**

During instantaneous settlement the soil experiences only shear deformation resulting into change in shape without volumetric deformation. The loss of pore pressure in the soil is zero.

### **Primary settlement**

This stage of soil deformation is characterized by skeleton deformation due to motion and compression of grains manifested by volume changes. If the pores are filled with water (particularly in case of low permeability soils), the water will be carried away from squeezed pores into locations with lower pressure (the soil will undergo consolidation). The consolidation primary settlement is therefore time dependent and is terminated by reaching zero pore pressure.

### **Secondary settlement**

When the primary consolidation is over the skeleton deformation will no longer cause the change in pore pressure (theoretically at infinite time). With increasing pressure the grains may become so closely packed that they will start to deform themselves and the volumetric changes will continue - this is referred to as creep deformation of skeleton or secondary consolidation (settlement). Unlike the primary consolidation the secondary consolidation proceeds under constant effective stress. Particularly in case of soft plastic or squash soils the secondary consolidation should not be neglected - in case of overconsolidated soils it may represent app. 10% of the overall settlement, for normally consolidated soils even app. 20%.

## Primary Settlement

The final primary settlement  $s$  is often substituted by the term settlement. Most of the computational approaches can be attached to one of the two groups:

- **Linear elastic deformation**
- **Nonlinear elastic deformation**

### Linear elastic deformation

The linear stress-strain relationship follows the Hook law:

$$\varepsilon = \frac{\Delta\sigma_{ef}}{E}$$

where:

- $\varepsilon$  - induced deformation of the soil layer
- $\Delta\sigma_{ef}$  - induced change of effective stress in the soil layer
- $E$  - Young's modulus in the soil layer
- $\nu$  - Poisson's ratio

The applicability of Young's modulus  $E$  of elasticity is substantiated only in cases, in which the stressed soil is allowed to stretch in the horizontal direction. This, however, is acceptable only for small spread foundations. When applying the load over a larger area, the stressed soil cannot, except for its edges, to deform sideways and experiences therefore only a vertical (one-dimensional) strain related to the oedometric modulus  $E_{oed}$ , that is larger than the elastic modulus  $E$ .

The settlement of a soil layer  $s$  is determined by multiplying the deformation of a soil layer  $\varepsilon$  by the layer thickness (height)  $H_o$ :

$$s = \varepsilon \cdot H_o$$

where:

- $\varepsilon$  - deformation of the soil layer
- $H_o$  - thickness of the soil layer

In case of layered subsoil we get the total settlement by summing up settlements of individual layers:

$$s = \sum s_i = \sum \varepsilon_i \cdot H_{oi}$$

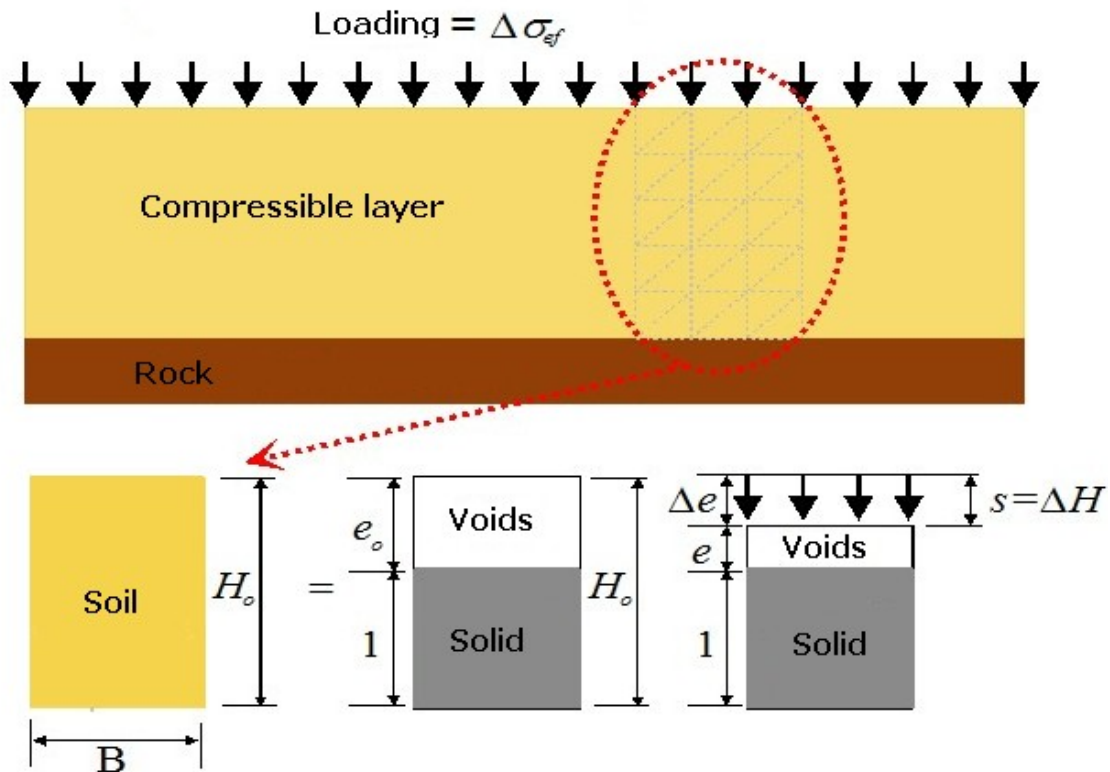
where:

- $s$  - settlement of the layered subsoil
- $\varepsilon_i$  - deformation of the  $i^{th}$  soil layer
- $H_{oi}$  - thickness of the  $i^{th}$  soil layer

### Nonlinear elastic deformation

For most soils the stress-strain relationship is nonlinear and often influenced by the load history. This nonlinearity cannot be neglected, particularly when computing the settlement of fine-grained soils (silts, clays). Clearly, the procedure based on application of Young's modulus of elasticity is not generally applicable. Even if employing the stress dependent **oedometric modulus of deformation**, it will not be possible to receive reasonable estimates of the behavior of certain overconsolidated soils. Nonlinear elastic deformation is generally modeled using the **void ratio** and deformation characteristics derived from one-dimensional deformation of a soil sample (e.g. **compression constant**, **compression index**, etc.).

The procedure for the computation of settlement of a compressible saturated soil layer using the void  $e$  is described on the following soil element having the height  $H_o$  and the width  $B = 1$  m:



*Analysis of settlement from phase diagram*

Owing to the fact that the soil is a three phase medium (it contains solid particles and pore filled with fluid and gas) it is possible to describe the solid particles (rock particles and mineral grains) by their volume  $V_s$  (and assumed to be equal to unity), while the porous phase can be described using the **void ratio**  $e$ .

The soil element is subjected on its upper surface to a uniform load  $q$  causing the change in stress inside the sample and also the vertical displacement  $\Delta H$ , which in turn leads to the reduction of pores  $V_p$  and therefore also to the reduction of void ratio (from its original value  $e_o$  to a new value  $e$ ). The vertical strain  $\epsilon$  of a soil sample is given by the ratio of  $\Delta H$  to the original sample height  $H_o$ , and can be expressed using the void ratio  $e$ :

$$\epsilon = \frac{\Delta H}{H_o} = \frac{s}{H_o} = \frac{\Delta e}{1+e}$$

- where:
- $\epsilon$  - vertical relative compression
  - $\Delta H$  - vertical deformation
  - $H_o$  - origin height of the element
  - $s$  - settlement
  - $e$  - void ratio
  - $\Delta e$  - change of void ratio

By modifying this equation we arrive at the formula describing the sample settlement with the help of void ratio:

$$s = \frac{\Delta e}{1+e} \cdot H_o = \varepsilon \cdot H_o$$

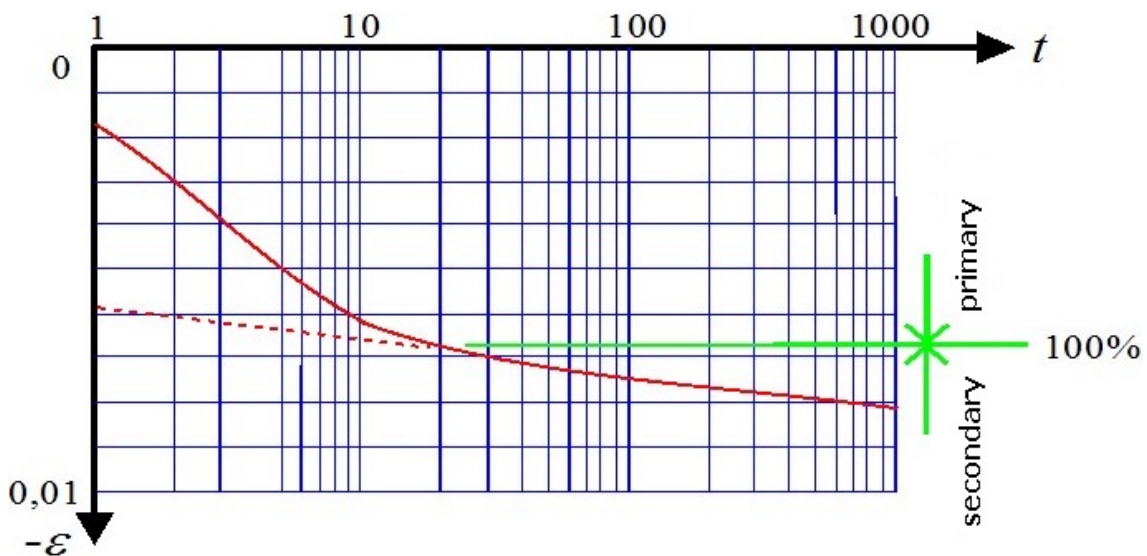
- where:
- $\varepsilon$  - vertical relative compression
  - $H_o$  - origin height of the element
  - $s$  - settlement
  - $e$  - void ratio
  - $\Delta e$  - change of void ratio

## Secondary Settlement

To describe a gradual creep of soil during secondary settlement the program employs the Buisman method (it incorporates the **index of secondary compression**  $C_\alpha$  derived by Ladd). From observations suggesting that the soil deformation follows a linear path when plotted in semi-logarithmic scale against time Buisman proposed the variation of  $\varepsilon$  due to long-term stress in the form:

$$\varepsilon = \varepsilon_p + \varepsilon_s \cdot \log\left(\frac{t}{t_0}\right)$$

- where:
- $\varepsilon$  - total deformation
  - $\varepsilon_p$  - deformation associated with primary consolidation
  - $\varepsilon_s$  - deformation associated with secondary consolidation
  - $t$  - time of consolidation
  - $t_0$  - reference time



Time dependent variation of strain (primary and secondary consolidation)

## Consolidation Analysis

Program allows analysis of consolidation when set in the frame "Settings". Consolidated layer, formed by impermeable, resp. lower permeable soil, subsequently settles with increasing time. Consolidation affects values of pore pressure. Soil parameters influencing consolidation analysis are entered in the frame "Soils", other consolidation parameters are set in the frame "Analysis" in individual construction stages.

Consolidation coefficient, depended on the soil properties, is calculated:

$$c_v = \frac{E_{oed} \cdot k}{\gamma_w}$$

where:  $E_{oed}$  - oedometric modulus of deformation  
 $k$  - coefficient of permeability  
 $\gamma_w$  - unit weight of water

When the consolidated layer is composed from non-homogeneous soil, coefficient  $c_v$  is evaluated as average of soil coefficients.

Consolidation analysis is also influenced by time factors, which are depended on the path of water outflow. This path is equal to the thickness of the consolidated layer in case of only one direction outflow (upwards or downwards) or half of the thickness in case of both directions outflow (upwards and downwards). Real time factor is evaluated according to the following formula:

$$T_v = \frac{c_v \cdot t}{H^2}$$

where:  $c_v$  - consolidation coefficient  
 $t$  - real time  
 $H$  - drainage path

Time factor of build duration is influenced by duration of load action. When the whole load is introduced at the beginning of stage, build time is equal to zero. When load linearly increases during stage duration, then build time is equal to the time of stage duration. Time factor of build duration is calculated by formula:

$$T_c = \frac{c_v \cdot t_c}{H^2}$$

where:  $c_v$  - consolidation coefficient  
 $t_c$  - build time  
 $H$  - drainage path

Degree of consolidation is evaluated by following formulas:

for:  $T_v \leq T_c$

$$U_{av} = \frac{T_v}{T_c} \left\{ 1 - \frac{2}{T_v} \sum_{m=0}^{\infty} \frac{1}{M^4} \left[ 1 - \exp(-M^2 \cdot T_v) \right] \right\}$$

for:  $T_v > T_c$

$$U_{av} = 1 - \frac{2}{T_c} \sum_{m=0}^{\infty} \frac{1}{M^4} [\exp(M^2 T_c) - 1] \exp(-M^2 T_v)$$

where:

$$M = (2m + 1) \frac{\pi}{2}$$

where:  $T_v$  - real time factor  
 $T_c$  - time factor of build duration

Original value of deformation in consolidated layer in certain construction stage is multiplied by corresponding degree of consolidation  $U_{av}$  to obtain result value of deformation:

$$\varepsilon_{fin} = U_{av} \cdot \varepsilon$$

where:  $\varepsilon_{fin}$  - result value of deformation  
 $\varepsilon$  - original value of deformation  
 $U_{av}$  - degree of consolidation

Consolidation analysis also influences pore pressure values in consolidated layer. In the time of introducing the load action, pore pressure values are the highest. When time increases to theoretical infinity, pore pressure decreases to zero.

Pore pressure:

for:  $T_v \leq T_c$

$$u = \sum_{m=0}^{\infty} \frac{2 \cdot u_0}{M^3 \cdot T_c} \cdot \sin \frac{M \cdot z}{H} [1 - \exp(-M^2 T_v)]$$

for:  $T_v > T_c$

$$u = \sum_{m=0}^{\infty} \frac{2 \cdot u_0}{M^3 \cdot T_c} \cdot [\exp(M^2 T_c) - 1] \sin \frac{M \cdot z}{H} \exp(-M^2 T_v)$$

where:

$$M = (2m + 1) \frac{\pi}{2}$$

where:  $T_v$  - real time factor  
 $T_c$  - time factor of build duration  
 $H$  - drainage path  
 $z$  - depth, where value of pore pressure is evaluated  
 $u_0$  - change of effective stress compared to previous stage (load)

*Literature:*

*Braja M. Das. Advanced Soil Mechanics; Taylor & Francis: London, 2008.pp278 - 316*  
*Verruijt A. Soil Mechanics, Delft University of Technology, 2010, pp97-123.*

<http://geo.verruijt.net/software/SoilMechBook.pdf>



## Determination of the Influence Zone Depth

From the theoretical point of view when applying a load on the ground surface we may expect the change of stress in subsoil into an infinite depth. The soil, however, deforms only up to a certain depth - within so called influenced zone.

The program offers two options to specify the influence zone:

- Using the **theory of structural strength**
- By specifying a certain percentage of the **primary geostatic stress**

## Theory of Structural Strength

The structural strength represents the resistance of soil against deformation for a load at the onset of failure of its internal structure. With decreasing coefficient  $m$  the soil responds tends to be linear.

If the structural strength is accounted for during settlement analysis, then:

**a)** the influence zone is characterized by the depth below the footing bottom at which the increment of vertical stress  $\sigma_z$  becomes equal to the structural strength of soil (determined by multiplying the original geostatic stress  $\sigma_{or}$  by the **coefficient  $m$** ):

$$\sigma_z = m \cdot \sigma_{or}$$

where:

$m$	-	coefficient of structural strength
$\sigma_{or}$	-	original geostatic stress

**b)** when computing the settlement of a layer, the increment of vertical stress  $\sigma_z$  due to surcharge and reduced by the structural strength of soil is provided by:

$$\sigma_z - m \cdot \sigma_{or}$$

where:

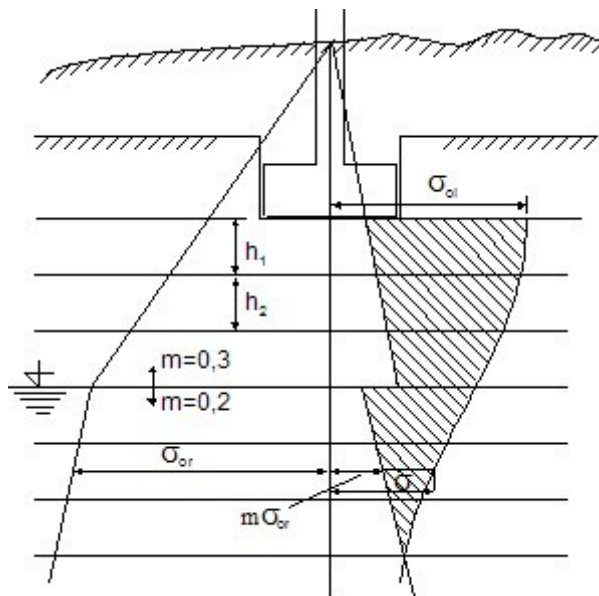
$m$	-	coefficient of structural strength
$\sigma_{or}$	-	original geostatic stress
$\sigma_z$	-	incremental stress in the middle layer

and the settlement  $s$  then follows from the stress denoted in figure by hatching and is given by:

$$s = f(\sigma_z, m, \sigma_{or})$$

where:

$m$	-	coefficient of structural strength
$\sigma_{or}$	-	original geostatic stress
$\sigma_z$	-	incremental stress in the middle layer



Depth of influence zone based on theory of structural strength (area of effective surcharge is hatched)

### Method of Restriction of the Primary Stress Magnitude

If we assume in the settlement analysis the constrains in terms of the percentage of primary geostatic stress, then:

**a)** the influence zone is represented by a depth below the footing bottom where the incremental stress  $\sigma_z$  reaches a certain percentage of the original geostatic stress:

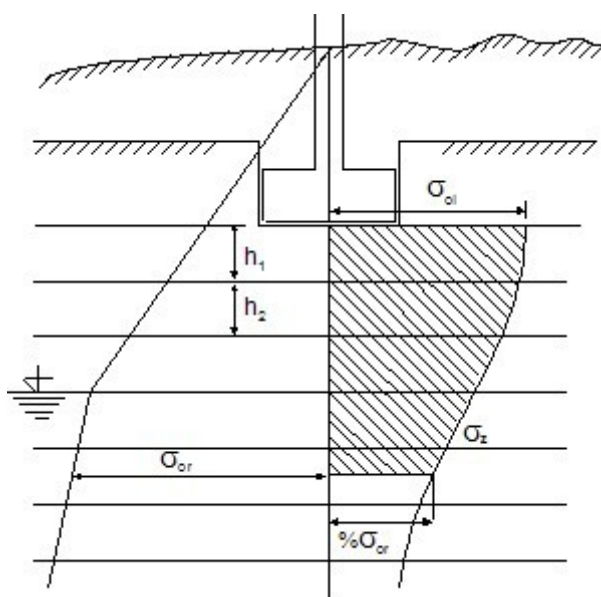
$$\sigma_z = x\% \cdot \sigma_{or}$$

where:  $x\%$  - considered magnitude of the geostatic stress  
 $\sigma_o$  - geostatic stress

**b)** the settlement  $s$  is derived from stress value denoted in figure by hatching and it receives the form:

$$s = f(\sigma_z, \sigma_{or})$$

where:  $\sigma_z$  - incremetal stress  
 $\sigma_{or}$  - geostatic stress



Depth of influence zone given by constraining the magnitude of primary stress

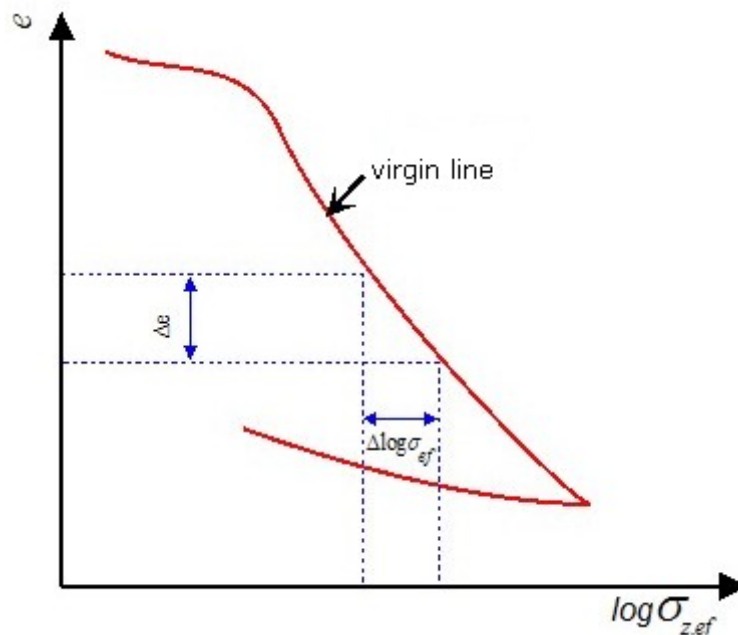
## Characteristics of Settlement Analyses

Depending on the selected solution method the program employs for the computation of settlement the following characteristics that may differ by the type of experiment needed for their determination or in the way of representation of measured variables:

- Compression index  $C_c$
- Oedometric modulus  $E_{oed}$
- Deformation modulus  $E_{def}$
- Compression constant  $C$
- Compression constant  $C_{10}$
- Void ratio  $e$
- Recompression index  $C_r$
- Janbu characteristics
- Correcting coefficient  $m$
- Modified compression index  $\lambda$
- Index of secondary compression  $C_\alpha$
- Overconsolidation index of secondary compression  $C_{ar}$

## Compression Index

It describes variation of the void ratio  $e$  as a function of the change of effective stress  $\sigma_{ef}$  plotted in the logarithmic scale:



Void ratio  $e$  versus effective stress  $\sigma_{ef}$

It therefore represents a deformation characteristic of overconsolidated soil:

$$C_c = \frac{\Delta e}{\Delta \log \sigma_{ef}}$$

where:  $\Delta e$  - variation of void ratio  
 $\Delta \log \sigma_{ef}$  - variation of effective stress

**Range of compression index  $C_c$**  (Naval Facilities Engineering Command Soil Mechanics DESIGN MANUAL 7.01)

A typical range of the compression index is from 0.1 to 10. Approximate values for homogeneous sand for the load range from 95 kPa to 3926 kPa attain the values from 0.05 to 0.06 for loose state and 0.02 to 0.03 for dense state. For silts this value is 0.20.

**For lightly overconsolidated clays and silts tested in USA Louisiana Kaufmann and Shermann (1964) present the following values:**

Soil	Effective consolidation stress $\sigma_{cef}$ [kPa]	Final effective stress in the soil $\sigma_{ef}$ [kPa]	Compression index $C_c$ [-]
CL soft clay	160	200	0.34
CL hard clay	170	250	0.44
ML silt of low plasticity	230	350	0.16
CH clay of high plasticity	280	350	0.84
CH soft clay with silt layers	340	290	0.52

**Prof. Juan M. Pestana-Nascimento (University of California, Berkeley) offers the following typical values of the compression index  $C_c$ :**

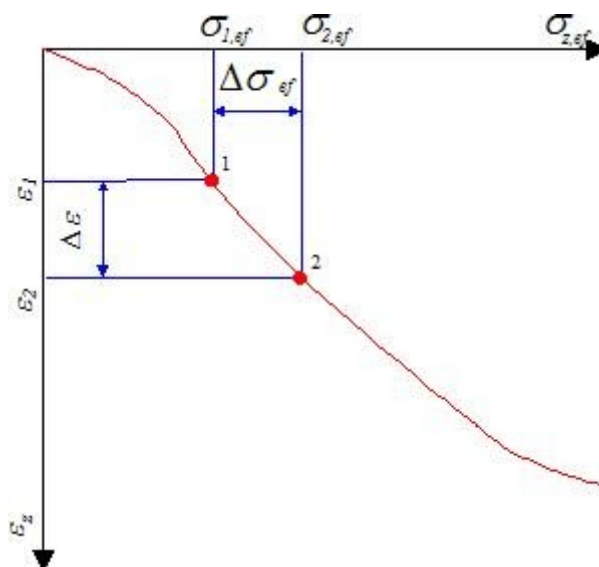
Soil	Compression index $C_c$ [-]
Normal consolidated clays	0.20 - 0.50
Chicago clay with silt (CL)	0.15 - 0.30
Boston blue clay (CL)	0.30 - 0.50
Vickburgs clay - dray falls into lumps (CH)	0.3 - 0.6
Swedish clay (CL - CH)	1 - 3
Canada clay from Leda (CL - CH)	1 - 4
Mexico City clay (MH)	7 - 10
Organic clays (OH)	4 and more
Peats (Pt)	10 - 15
Organic silts and clay silts (ML - MH)	1.5 - 4.0
San Francisco sediments (CL)	0.4 - 1.2
Clay in the old San Francisco Bay	0.7 - 0.9
Bangkok clay (CH)	0.4

**In addition, there are empirical expressions available to determine approximate values of  $C_c$  for silts, clays and organic soils; their applicability, however, is more or less local:**

Soil	Equations	Reference
Transformed clays	$C_c = 0,007(w_z - 7\%)$	Skempton 1944
Clays	$C_c = 1,15.(e_0 - 0,35)$	Nishida 1956
Brazilian clays Sao Paulo clays	$C_c = 0,256 + 0,43.(e_0 - 0,84)$ $C_c = 0,0046(w_z - 9\%)$	Cozzolino 1961
New York clays	$C_c = 0,009(w_z - 10\%)$	Terzaghi a Peck 1948
Clays of low plasticity	$C_c = 0,75.(w_0 - 0,50)$	Sowers 1970
Taipei clays and silts	$C_c = 0,54.(e_0 - 0,23)$ $C_c = 0,007.(w_z - 7\%)$	Moh a kol. 1989
Clays	$C_c = 2,203.\rho_c.e_0.\left(1 - \left(\frac{0,4}{e_0}\right)^2\right)$ $C_c = \frac{\alpha.w_z}{100}.\left(1 - \left(\frac{20}{w_z}\right)^2\right)$	Pestana 1994

### Oedometric Modulus

If the results from oedometric test are represented in terms of oedometric curve ( $\Delta\varepsilon = f(\Delta\sigma_{ef})$ ), it becomes evident that for each point on the curve we receive a different ratio  $\sigma_{ef} / \varepsilon$ .



Determination of oedometric modulus  $E_{oed}$

If the stress-strain curve is replaced for a certain interval of two neighboring stresses  $\sigma_{1ef} - \sigma_{2ef}$  by a secant line, it is acceptable to assume a linear behavior of soil within this interval and

represent the soil compressibility by as  $\Delta\sigma_{ef}/\Delta\varepsilon$  - called the oedometric modulus of deformation. The oedometric modulus of deformation is therefore a secant modulus linked to a certain stress interval  $\sigma_{1ef} - \sigma_{2ef}$  selected on the stress-strain diagram  $\Delta\varepsilon = (\Delta\sigma_{ef})$ :

$$E_{oed} = \frac{\Delta\sigma_{ef}}{\Delta\varepsilon} = \frac{\sigma_{2,ef} - \sigma_{1,ef}}{\varepsilon_2 - \varepsilon_1}$$

In general, the oedometric modulus of deformation  $E_{oed}$  tends to decrease its value with the increasing stress interval. Therefore we should consider for each layer a specific value of  $E_{oed}$  pertinent to a given stress interval (from original to final stress state). This is reflected in the program by the way of inputting  $E_{oed}$ , where it is possible to specify for each soil the respective oedometric curve ( $\sigma_{ef}/\varepsilon$  diagram).

Practical experience, however, suggests (e.g. for clays) a several orders of magnitude difference between the value of  $E_{oed}$  derived from the deformation modulus  $E_{def}$  and that provided by the in situ measured loading curve.

The relation between  $E_{def}$  and  $E_{oed}$  is provided by:

$$E_{oed} = \frac{E_{def}}{\beta}$$

$$\beta = 1 - \frac{2\nu^2}{1-\nu}$$

where:  $\nu$  - Poisson's ratio  
 $E_{def}$  - deformation modulus  
 $f$

**Approximate range of values of oedometric modulus of deformation  $E_{oed}$  for individual soils and typical stress range (prof. I. Vanicek: Soil mechanics):**

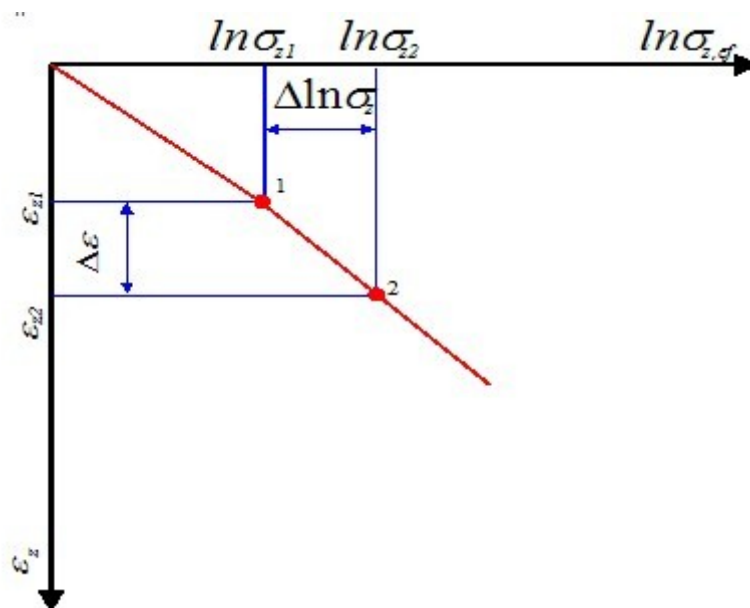
Soil	Oedometric modulus $E_{oed}$ [MPa]
gravels	60 - 600
medium dense sands to dense sands	7 - 130
cohesive	2 - 30

Literature:

Vanicek, I.: *Geomechanika 10: mechanika zemin. 3<sup>th</sup> edition, Prague, CTU, 2000, 229 s., ISBN 80-01-01437-1.*

## Compression Constant

When plotting the effective vertical stress against the vertical strain in the semi-logarithmic scale we often arrive at a linear dependency.



Determination of compression constant *C*

Slope of this curve is one of the soil parameters particularly in case of one-dimensional deformation and is referred to as the compression constant *C*:

$$C = \frac{1}{\Delta \epsilon} \cdot \ln \frac{\sigma_{z,ef}}{\sigma_{z1,ef}}$$

where:  $\sigma_{z1,ef}$  - initial effective stress of soil in oedometer  
 $\sigma_{z2,ef}$  - final effective stress of soil in oedometer

**Margins of compression constant *C* (J.Šimek: Mechanika zemin)**

Soil	Compression constant <i>C</i> [-]
Loess silt	15 - 45
Clay	30 - 120
Silts	60 - 150
Medium dense and dense sands	150 - 200
Sand with gravel	> 250

**Compression Constant 10**

In engineering practice the natural logarithm with base is sometimes replaced by logarithm with base 10 when plotting the stress  $\sigma_{ef}$ . In this case it is common to denote the compression constant with subscript 10:  $C_{10}$ . Since it holds:

$$\log(x) = \frac{\ln(x)}{2,3}$$



it is possible to derive a relationship between compression constant  $C$  and  $C_{10}$ :

$$C_{10} = \frac{C}{2,3}$$

**Arnold Verruijt (Soil Mechanics)** offers the following values of compression constant:

Soil	$C$	$C_{10}$
Sand	50 - 500	20 - 200
Silt	25 - 125	10 - 50
Clay	10 - 100	4 - 40
Peat	2 - 25	1 - 10

## Void Ratio

The void ratio  $e$  describes porosity of a soil and is provided by:

$$e = \frac{V_p}{V_s}$$

where:  $V_p$  - volume of voids  
 $V_s$  - volume of solid grains

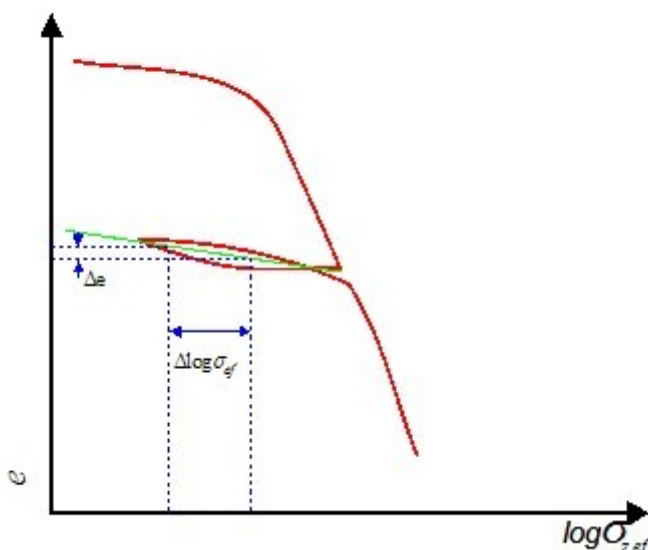
**Ranges of void ratio  $e$**  (*Braja M. DAS: Principles of Foundation Engineering*)

Soil	Void ratio $e$ [-]
Poorly graded sand with loose density	0.8
Well graded dense sand	0.45
Loose density sand with angular particles	0.65
Dense density sand with angular particles	0.4
Stiff clay	0.6
Soft clay	0.9 - 1.4
Loess	0.9
Soft organic clay	2.5 - 3.2
Glacial till	0.3

## Recompression Index

The recompression index  $C_r$  is determined from the graph representing the variation of **void ratio  $e$**  as a function of the effective stress  $\sigma_{ef}$  plotted in the logarithmic scale for unloading -

reloading sequence:



Determination of recompression index  $C_r$

$$C_r = \frac{\Delta e}{\Delta \log \sigma_{ef}}$$

- where:
- $\Delta e$  - change of void ratio for the unloading-reloading curve
  - $\Delta \log \sigma_{ef}$  - change of effective stress for the unloading-reloading curve

If no results from either laboratory or in situ measurements are available, the recompression index  $C_r$  can be approximately derived from:

$$C_r \cong \frac{1}{5} \sim \frac{1}{10} C_c$$

- where:
- $C_c$  - compression constant

## Janbu Characteristics

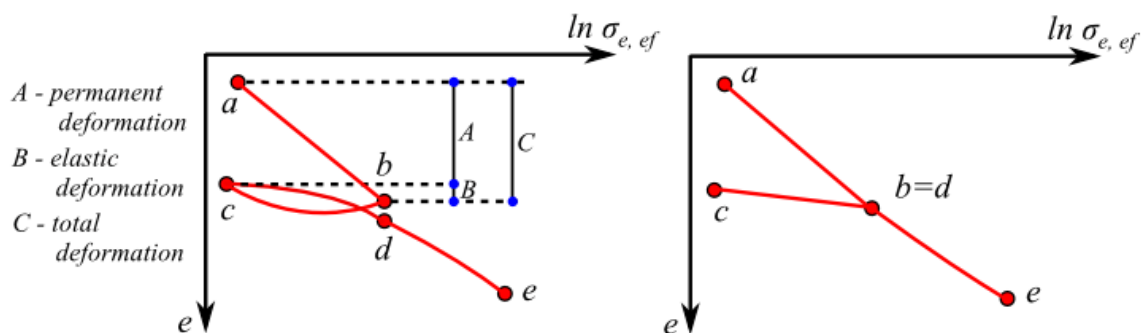
Values of the Janbu modulus  $m$  and of stress exponent  $j$  (according Canadian Foundation Engineering Manual 1992)

Soil	Janbu modulus $m$	Stress index $j$
Very dense to dense till, glacial till	1000 - 300	1
Gravel	400 - 40	0.5
Dense sand	400 - 250	0.5
Medium condition sand	250 - 150	0.5

Loose sand	150 - 100	0.5
Dense silt	200 - 80	0.5
Medium condition silt	80 - 60	0.5
Loose silt	60 - 40	0.5
Hard to very stiff clay	60 - 20	0
Medium to stiff clay	20 - 10	0
Soft claye silt	10 - 5	0
Soft marine clays	20 - 5	0
Organic clays	20 - 5	0
Peats	5 - 1	0

### Influence of Load History

The load history has a substantial influence on the distribution of deformation curve and therefore also on the values of deformation characteristics. The following figure displays the deformation curve ( $\Delta e = f(\Delta\sigma_{ef})$  diagram) derived from oedometric loading test corresponding, e.g. to natural dense sandy soil.



Load history a) Deformation curve for clayey soils from oedometric test b) Simplified interpretation of deformation curve

The soil sample was gradually loaded to reach the stress level  $\sigma_{bef}$ , the stress-strain relationship ( $\sigma_{bef} - \epsilon$ ) within the section  $a-b$  is linear and is denoted as primary or virgin (i.e., relative compression is encountered). Upon exceeding the stress level  $\sigma_{bef}$  the sample was elastically unloaded and the soil moved up the  $b-c$  section of the deformation curve. Upon reloading the soil moved down the  $b-c$  section till reaching the original stress  $\sigma_{bef}$  prior to unloading. When loading beyond  $\sigma_{bef}$  the deformation curve approaches asymptotically within the  $d-e$  section the primary line accompanied by inelastic deformation of a soil sample. Such a complex stress-strain curve is often simplified by the idealized deformation curve (fig. b). Such a curve characterizes so called overconsolidated soils, which were in the past subjected large stresses and subsequently unloaded. The overconsolidation ratio (**OCR**) then represents the ratio between the maximum preconsolidation stress the soil has ever experienced and the current vertical stress. Overconsolidated soils typically follow the deformation curve given by

points *c-d-e*. The change in slope along this line (given app. by point *d*) corresponds either to the vertical geostatic stress  $\sigma_o$  (normally consolidated soils) or to preconsolidation pressure  $\sigma_c$  (overconsolidated soils). This point influences the soil deformation, which is smaller within the *c-d* section when compared to the *d-e* section (where for the large degree of overconsolidation the soil deformation increases). Additional deformation characteristics such as deformation modulus upon unloading  $E_e$ , one-dimensional swelling index  $C_e$ , recompression index  $C_r$ , etc. were introduced to describe such a complex soil behavior. Currently the most often used parameter is the recompression index  $C_r$  suitable for the computation of settlement of overconsolidated soils.

## Coefficient $m$

Correction coefficient of surcharge due to structural strength  $m$  determines the structural strength of soil.

### Values of the correction coefficient of surcharge $m$

Type of fundamental soil	$m$
Very compressible fine soils class F1 -F8 - with deformation modulus $E_{def} < 4 \text{ MPa}$ - nonoverconsolidated - soft to hard consistency (all 3 attributes must be fulfilled), filling, made - ground secondary and tertiary sediments rocks class R1, R2	0.1
fine soils class F1-F8, not belonging to coefficient $m = 0.1$ nor $0.4$ nor $0.6$ sands and gravels class S1, S2, G1, G2 under GWT rock class R3, R4	0.2
Sands and gravels class S1, S2, G1, G2 above GWT sands and gravels with clay, silt or fine soil admixture soils class S3, S4, S5, G3, G4, G5 rocks class R5, R6	0.3
eluvium of igneous and metamorphic rocks	0.4

## Modified Compression Index

The analysis employing the Soft soil model builds on the elastic-plastic model developed in the university in Cambridge. Here, the vertical deformation of soil  $\varepsilon$  assumes linear dependence on the logarithmic variation of effective stress in a soil. Application of this model requires an

introduction of the modified compression index  $\lambda$  usually obtained from triaxial tests.

If the modified compression index  $\lambda$  is not available from laboratory measurements, it can be approximately found from the compression index  $C_C$ :

$$\lambda = \frac{C_c}{2,3 \cdot (1 + e)}$$

where:  $C_C$  - compression index

$e$  - average void ratio (if this value is not available, it can be approximately substituted by the initial void ratio  $e_0$ )

## Index of Secondary Compression

The index of secondary compression is proportional to the logarithm of time and the slope of primary consolidation (it is strongly dependent on the final effective stress in soil):

$$C_\alpha = \frac{\Delta \varepsilon}{\log t_2 - \log t_1}$$

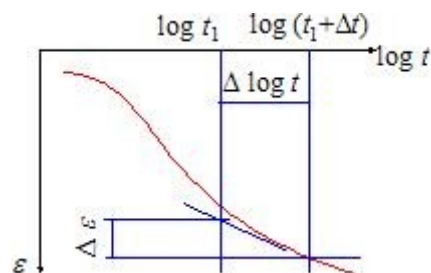
where:  $C_\alpha$  - index of secondary compression

$\varepsilon$  - deformation of a soil layer

$t_1$  - initial time of a period of monitoring (measured from the start of consolidation)

$t_2$  - final time of a period of monitoring

Determining the value of index of secondary compression  $C_\alpha$  requires either laboratory (e.g. one-dimensional consolidation in oedometer) or in-situ measurements:



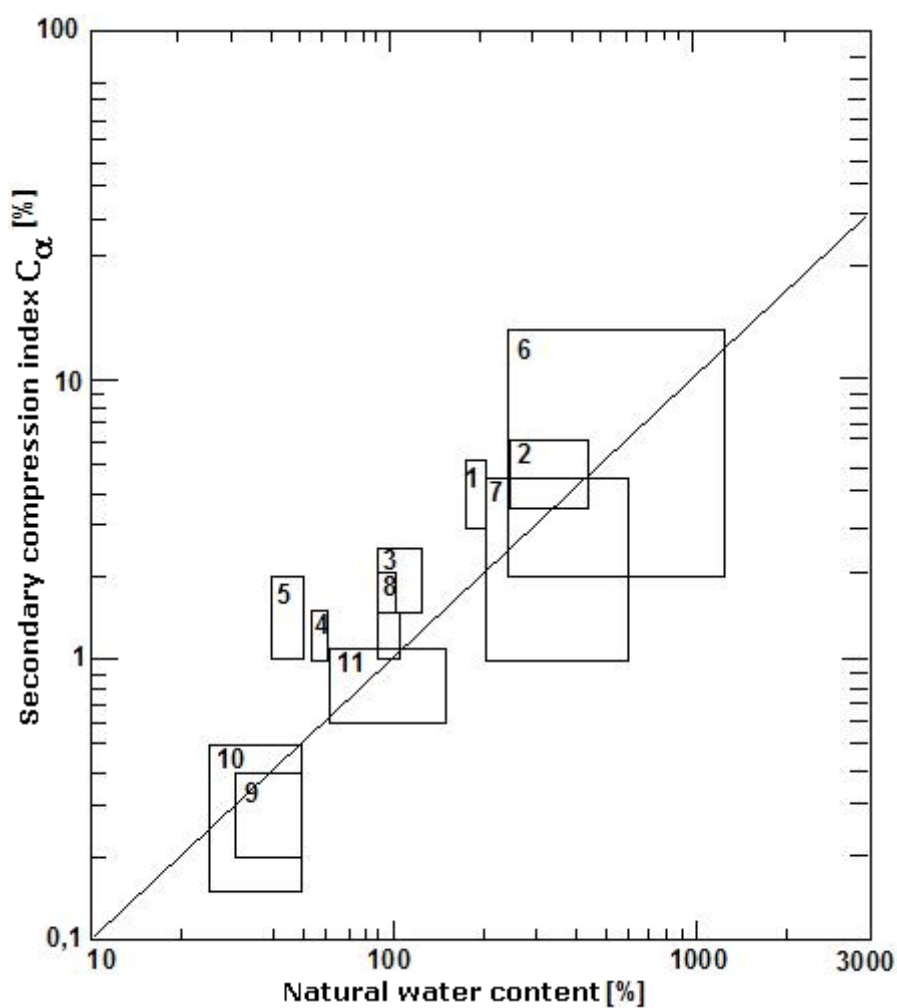
*Determination of index of secondary compression  $C_\alpha$*

### Ranges of values of index of secondary compression $C_\alpha$

sand	0.00003 - 0.00006
silty loess	0.0004
clay	0.01

The ratio between the index of secondary compression  $C_\alpha$  and the compression index  $C_C$  is approximately constant for most of the normally consolidated clays for load typical in engineering practice. Its average value is 0.05.

Variation of natural moisture of soil as a function of the index of secondary compression  $C_\alpha$  derived by Mesri appears in figure:



*Variation of natural moisture of soil as a function of the index of secondary compression  $C_{\alpha}$  after Mesri*

- 1 Whangamarino clay
- 2 Mexico City clay
- 3 Calcareous organic silt
- 4 Leda clay
- 5 Norwegian plastic clay
- 6 Amorphous and fibrous peat
- 7 Canadian muskeg
- 8 Organic marine deposits
- 9 Boston blue clay
- 10 Chicago blue clay
- 11 Organic silty clay

## Overconsolidation Index of Secondary Compression

The overconsolidation index of secondary compression depends on laboratory measurements (e.g. one-dimensional consolidation) and is proportional to the logarithm of time and slope of virgin consolidation line providing the preconsolidation pressure was not exceeded:

$$C_{\alpha} = \frac{\Delta\varepsilon}{\log t_2 - \log t_1}$$

where:

- $C_{\alpha}$  - overconsolidation index of secondary compression
- $\varepsilon$  - deformation of a soil layer
- $t_1$  - initial time of a period of monitoring (measured from the onset of consolidation)
- $t_2$  - final time of a period of monitoring

## Ground Loss

Analyses performed in the program "**Ground Loss**" can be divided into the following groups:

- Analysis of the shape of subsidence trough above excavations
- Analysis of failure of buildings

The failure analysis of building is based on the shape of subsidence trough.

## Analysis of Subsidence Trough

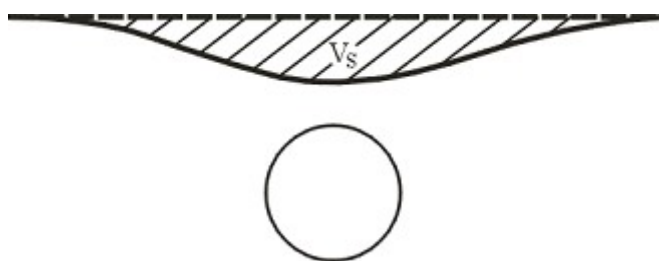
The analysis of subsidence trough consists of several sequential steps:

- Determination of the **maximum settlement** and **dimensions of subsidence trough** for individual excavations
- Analysis of the shape of subsidence trough
- Back calculation of the shape and dimensions of subsidence trough providing it is calculated at a given depth below the terrain surface
- Determination of the overall shape of subsidence trough for more excavations
- Post-processing of other variables (horizontal deformation, slope)

The analysis of maximum settlement and dimensions of subsidence trough can be carried out using either the theory of volume loss or the classic theories (Peck, Fazekas, Limanov).

## Volume Loss

The volume loss method is a semi-empirical method based partially on theoretical grounds. The method introduces, although indirectly, the basic parameters of excavation into the analysis (these include mechanical parameters of a medium, technological effects of excavation, excavation lining etc) using 2 comprehensive parameters (**coefficient  $k$  for determination of inflection point** and **a percentage of volume loss  $VL$** ). These parameters uniquely define the shape of subsidence trough and are determined empirically from years of experience.



*Settlement expressed in terms volumes*

The maximum settlement  $S_{max}$ , and location of inflection point  $L_{inf}$  are provided by the following expressions:

$$L_{inf} = k.Z$$

$$S_{max} = \frac{A.VL}{100} \cdot \frac{1}{\sqrt{2.\pi.L_{inf}}}$$

where:

- $A$  - excavation area
- $Z$  - depth of center point of excavation
- $k$  - coefficient to calculate inflection point (material constant)
- $VL$  - percentage of volume loss

The roof deformation  $u_a$  follows from:

$$u_a = \frac{2r - \sqrt{4r^2 - \frac{4r^2.VL}{100}}}{2}$$

where:

- $r$  - excavation radius
- $V$  - percentage of volume loss
- $L$

*Literature:*

<http://www.groundloss.com/>

## **Recommended Values of Parameters for Volume Loss Analysis**

Data needed for the determination of subsidence trough using the volume loss method:

**Coefficient to calculate inflection point  $k$**



Soil or rock	$k$
cohesionless soil	0.3
normaly consolidated clay	0.5
overconsolidated clay	0.6 - 0.7
clay slate	0.6 - 0.8
quartzite	0.8 - 0.9

#### Percentage of volume loss $VL$

Technology	$VL$
TBM	0.5 - 1
Sequential excavation method	0.8 - 1.5

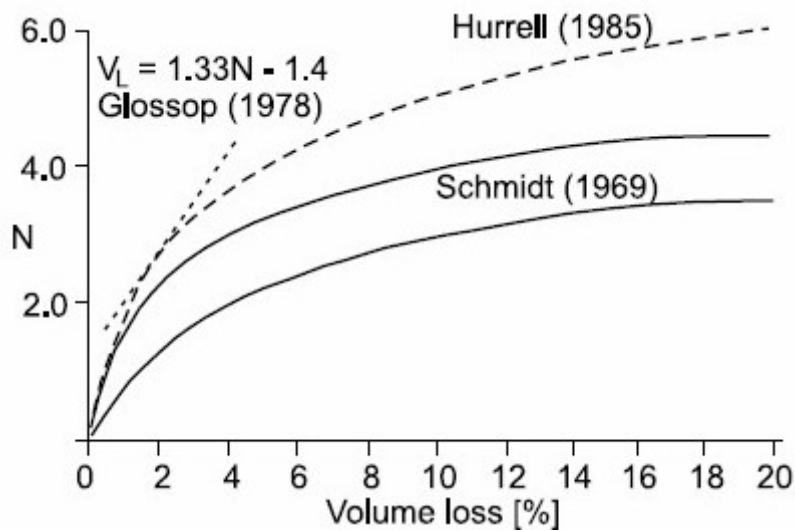
Several relationships were also derived to determine the value of lost volume  $VL$  based on stability ratio  $N$  defined by Broms and Bennermarkem:

$$N = \frac{\sigma_v \cdot \sigma_t}{S_u}$$

where:

- $\sigma$  overall stress along excavation axis
- $v$
- $\sigma$  excavation lining resistance (if lining is installed)
- $t$
- $S$  undrained stiffness of clay
- $u$

For  $N < 2$  the soil/rock in the vicinity of excavation is assumed elastic and stable. For  $N \in < 2, 4$  local plastic zones begin to develop in the vicinity of excavation, for  $N \in < 4, 6$  a large plastic zone develops around excavation and for  $N = 6$  the loss of stability of tunnel face occurs. Figure shows the dependence of stability ration and lost volume  $VL$ .



Literature:

Broms, B.B., Bennemark, H., 1967. Stability of clay at vertical openings. ASCE, Journal of Soil Mechanics and Foundation Engineering Division, SMI 93, 71-94.

## Classic Theory

Convergence analysis of an excavation and calculation of the maximum settlement in a **homogeneous body** are the same for all classic theories. The **subsidence trough analyses** then differ depending on the assumed theory (Peck, Fazekas, Limanov).

When calculating settlement the program first determines the radial load of a circular excavation as:

$$p = \sigma_z \cdot \frac{1 + K_r}{2}$$

where:  $\sigma$  geostatic stress in center of excavation  
 $z$   
 $K$  coefficient of pressure at rest of cohesive soil  
 $r$

The roof  $u_a$  and the bottom  $u_b$  deformations of excavation follow from:

$$u_a = (1 + \nu) \cdot \frac{p}{E} \cdot r \cdot \frac{Z + (1 - 2\nu) \cdot r}{Z + r}$$

$$u_b = -(1 + \nu) \cdot \frac{p}{E} \cdot r \cdot \frac{Z + (1 - 2\nu) \cdot r}{Z + r}$$

where:  $Z$  depth of center point of excavation  
 $r$  excavation radius  
 $E$  modulus of elasticity of rock/soil in vicinity of excavation  
 $\nu$  Poisson's ratio of rock/soil in vicinity of excavation

The maximum terrain settlement and the length of subsidence trough are determined as follows:

$$S_{max} = (1 - \nu^2) \frac{p}{E} r \cdot \frac{4r^2 \cdot Z}{Z^2 - r^2}$$

$$L = 2 \cdot \sqrt{Z^2 - r^2}$$

where:

- $Z$  depth of center point of excavation
- $r$  excavation radius
- $E$  modulus of elasticity of rock/soil in vicinity of excavation
- $\nu$  Poisson's number of rock/soil in vicinity of excavation

When the **tunnel roof displacement is prescribed** the maximum settlement is provided by the following expression:

$$S_{max} = 4 \cdot u_a \cdot \frac{Z \cdot (1 - \nu)}{(Z + r) \cdot (Z + r + 2 \cdot \nu r)}$$

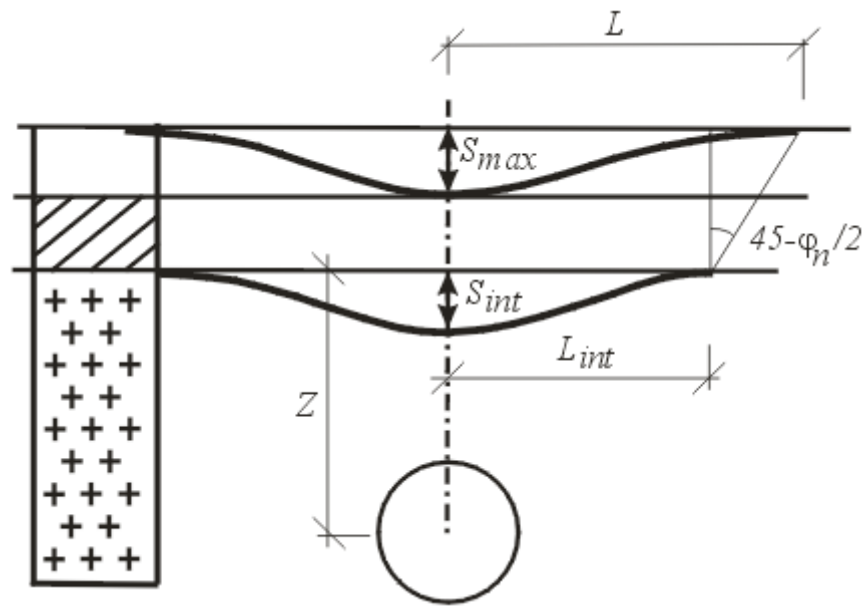
where:

- $Z$  depth of center point of excavation
- $r$  excavation radius
- $u_a$  tunnel roof displacement
- $a$
- $\nu$  Poisson's number of rock/soil in vicinity of excavation

## Analysis for Layered Subsoil

When determining a settlement of layered subsoil the program first calculates the settlement at the interface between the first layer above excavation and other layers of overburden  $S_{int}$  and determines the length of subsidence trough along layers interfaces. In this case the approach complies with the one used for a **homogeneous soil**.

Next (as shown in Figure) the program determines the length of subsidence trough  $L$  at the terrain surface.



*Analysis of settlement for layered subsoil*

The next computation differs depending on the selected analysis theory:

**Solution according to Limanov**

Limanov described the horizontal displacement above excavation with the help of lost area  $F$ :

$$S_{max} = \frac{L}{F}$$

where:  $L$  - length of subsidence trough  
 $F$  - volume loss of soil per 1m run determined from:

$$F = S_{int} \cdot \pi \cdot \frac{L_{int}}{2}$$

where:  $L_{int}$  - length of subsidence trough along interfaces above excavation  
 $S_{int}$  - settlement of respective interface

**Solution according to Fazekas**

Fazekas described the horizontal displacement above excavation using the following expression:

$$S_{max} = S_{int} \cdot \frac{L_{int}}{L}$$

where:  $L$  - length of subsidence trough  
 $L_{int}$  - length of subsidence trough along interfaces above excavation  
 $S_{int}$  - settlement of respective interface

**Solution according to Peck**

Peck described the horizontal displacement above excavation using the following expression:

$$S_{max} = S_{int} \cdot \frac{L_{int}}{L_{inf}}$$

where:  $L_{int}$  - length of subsidence trough along interfaces above excavation  
 $S_{int}$  - settlement of respective interface  
 $L_{inf}$  - distance of inflection point of subsidence trough from excavation axis at terrain surface

*Literature:*

*Széchy, Károly, The art of tunnelling, Budapest : Akadémiai Kiadó, 1966.*

## Shape of Subsidence Trough

The program offers two particular shapes of subsidence troughs - according to Gauss or Aversin.

### Curve based on Gauss

A number of studies carried out both in the USA and Great Britain proved that the transverse shape of subsidence trough can be well approximated using the Gauss function. This assumption then allows us to determine the horizontal displacement at a distance  $x$  from the vertical axis of symmetry as:

$$S_i = S_{max} \cdot e^{\left(\frac{-x_i^2}{2 \cdot L_{inf}^2}\right)}$$

where:  $S_i$  - settlement at point with coordinate  $x_i$   
 $S_{max}$  - maximum terrain settlement  
 $L_{inf}$  - distance of inflection point

### Curve based on Aversin

Aversin derived, based on visual inspection and measurements of underground structures in Russia, the following expression for the shape of subsidence trough:

$$S_i = S_{max} \cdot \left(1 - \frac{x_i}{L}\right) \cdot e^{\left(\frac{4 \cdot x_i}{L}\right)}$$

kde:  $S_i$  - settlement at point with coordinate  $x_i$   
 $S_{max}$  - maximum terrain settlement  
 $L$  - reach of subsidence trough

*Literature:*

*Széchy, Károly, The art of tunnelling, Budapest : Akadémiai Kiadó, 1966.*

## Coefficient of Calculation of Inflection Point

When the **classical methods** are used the input coefficient  $k_{inf}$  allows the determination of the

inflection point location based on  $L_{inf} = L/k_{inf}$ . In this case the coefficient  $k_{inf}$  represents a very important input parameter strongly influencing the shape and slope of subsidence trough. Its value depends on the average soil or rock, respectively, in overburden - literature offers the values of  $k_{inf}$  in the range 2.1 - 4.0.

Based on a series of FEM calculations the following values are recommended:

- gravel soil G1-G3  $k_{inf} = 3.5$
- sand and gravel soil S1-S5, G4, G5, rocks R5-R6  $k_{inf} = 3.0$
- fine-grained soil F1-F4  $k_{inf} = 2.5$
- fine-grained soil F5-F8  $k_{inf} = 2.1$

The coefficient for calculation of inflection point is input in the frame "Settings".

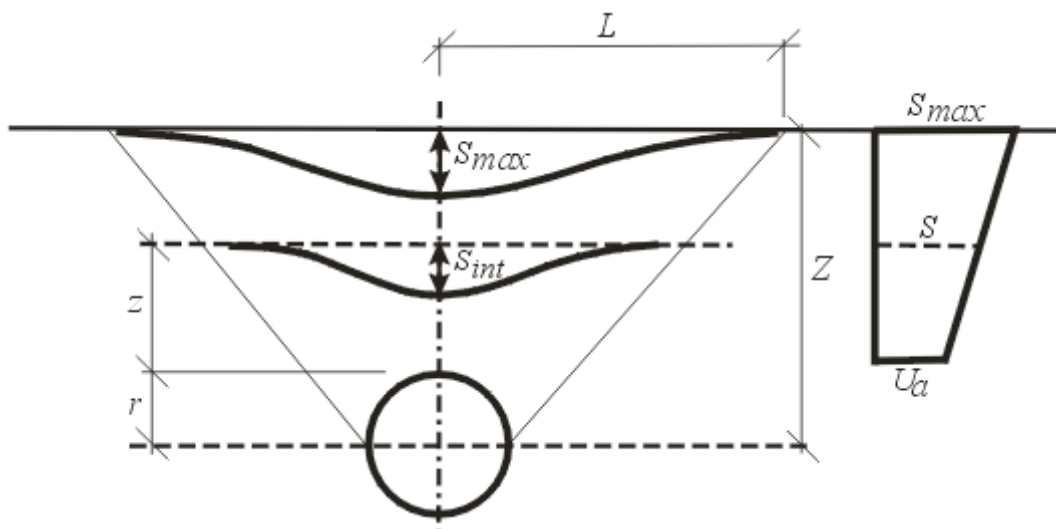
## Subsidence Trough with Several Excavations

The principal of superposition is used when calculating the settlement caused by structured or multiple excavations. Based on input parameters the program first determines subsidence troughs and horizontal displacements for individual excavations. The overall subsidence trough is determined subsequently.

**Other variables**, horizontal strain and gradient of subsidence trough, are post-processed from the overall subsidence trough.

## Analysis of Subsidence Trough in Depth

A linear interpolation between the maximal value of the settlement  $S_{max}$  at a terrain surface and the displacement of roof excavation  $u_a$  is used to calculate the maximum settlement  $S$  at a depth  $h$  below the terrain surface in a homogeneous body.



*Analysis of subsidence trough at a depth*

The width of subsidence trough at an overburden  $l$  is provided by:

$$l = \frac{(L - r)(z + r)}{Z} + r$$

where:  $L$  - length of subsidence trough at terrain surface  
 $r$  - excavation radius  
 $Z$  - depth of center point  
 $z$  - analysis depth

The values  $l$  and  $S$  are then used to determine the shape of subsidence trough in overburden above an excavation.

## Calculation of Other Variables

A vertical settlement is accompanied by the evolution of horizontal displacements which may cause damage to nearby buildings. The horizontal displacement can be derived from the vertical settlement providing the resulting displacement vectors are directed into the center of excavation. In such a case the horizontal displacement of the soil is provided by the following equation:

$$S_x = -\frac{s(x)}{Z-r}$$

where:  $x$  - distance of point  $x$  from axis of excavation  
 $s(x)$  - settlement at point  $x$   
 $Z$  - depth of center point of excavation  
 $r$  - excavation radius

The horizontal displacements are determined in a differential way along the  $x$  axis and in the transverse direction they can be expressed using the following equation:

$$\varepsilon_x = -\frac{s(x)}{Z-r} \left( \frac{x^2}{L_{inf}^2} - 1 \right)$$

where:  $x$  - distance of point  $x$  from axis of excavation  
 $s(x)$  - settlement at point  $x$   
 $Z$  - depth of center point of excavation  
 $L_{inf}$  - distance of inflection point  
 $r$  - excavation radius

## Analysis of Failure of Buildings

The program first determines the shape and dimensions of **subsidence trough** and then performs analysis of their influence on buildings.

The program offers four types of analysis:

- Determination of **tensile cracks**
- Determination of **gradient damage**
- Determination of a **relative deflection of buildings** (hogging, sagging)

- Analysis of the [input section of a building](#)

## Tensile Cracks

One of the causes responsible for the damage of buildings is the [horizontal tensile strain](#). The program highlights individual parts of a building with a color pattern that corresponds to a given class of damage. The maximum value of tensile strain is provided in the text output.

The program offers predefined zones of damage for masonry buildings. These values can be modified in the frame "[Stage settings](#)". Considerable experience with a number of tunnels excavated below build-up areas allowed for elaborating the relationship between the shape of subsidence trough and damage of buildings to such precision that based on this it is now possible to estimate an extent of compensations for possible damage caused by excavation with accuracy acceptable for both preparation of contractual documents and for contractors preparing proposals for excavation of tunnels.

Recommended values for masonry buildings from one to six floors are given in the following table.

Horizontal strains (per mille)

Proportional h.s. (per mille)	Damage	Description
0.2 - 0.5	Microcracks	Microcracks
0.5 - 0.75	Little damage - superficial	Cracks in plaster
0.75 - 1.0	Little damage	Small cracks in walls
1.0 - 1.8	Medium damage, functional	Cracks in walls, problems with windows and doors
1.8 -	Large damage	Wide open cracks in bearing walls and beams

## Gradient Damage

One of the causes leading to the damage of buildings is the slope subsidence trough. The program highlights individual parts of a building with a color pattern that corresponds to a given class of damage. The maximum value of tensile strain is provided in the text output.

The program offers predefined zones of damage for masonry buildings. These values can be modified in the frame "[Stage settings](#)". Considerable experience with a number of tunnels excavated below build-up areas allowed for elaborating the relationship between the shape of subsidence trough and damage of buildings to such precision that based on this it is now possible to estimate an extent of compensations for possible damage caused by excavation with accuracy acceptable for both preparation of contractual documents and for contractors preparing proposals for excavation of tunnels.

Recommended values for masonry buildings from one to six floors are given in the following table.

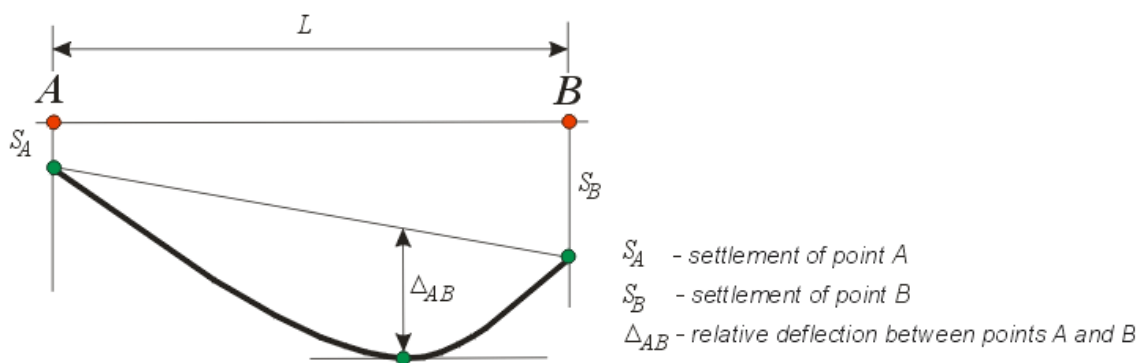
### Gradient



Gradient	Damage	Description
1:1200 - 800	Microcracks	Microcracks
1:800 - 500	Little damage - superficial	Cracks in plaster
1:500 - 300	Little damage	Small cracks in walls
1:300 - 150	Medium damage, functional	Cracks in walls, problems with windows and doors
1:150 - 0	Large damage	Wide open cracks in bearing walls and beams

## Relative Deflection

Definition of the term relative deflection is evident from the figure. The program searches regions on buildings with the maximum relative deflection both upwards and downwards. Clearly, from the damage of building point of view the most critical is the relative deflection upwards leading to "**tensile opening**" of building.



Verification of the maximum relative deflection is left to the user - the following tables list the ultimate values recommended by literature.

Type of structure	Type of damage	Ultimate relative deflection $\Delta/l$			
		Burland and Wroth	Meyerhof	Polshin a Tokar	ÈSN 73 1001
Unreinforced bearing walls	Cracks in walls	For $L/H = 1$ - 0.0004 For $L/H = 5$ - 0.0008	0.0004	0.0004	0.0015
	Cracks in bearing structures	For $L/H = 1$ - 0.0002 For $L/H = 5$ - 0.0004	-	-	-

## Failure of a Section of a Building

In a given section the program determines the following variables:

- maximum **tensile strain**
- maximum **gradient**
- maximum **relative deflection**
- relative gradient between input points of a building

Evaluation of the analyzed section is left to the user - the following tables list the recommended ultimate values of relative rotation and deflection.

Type of structure	Type of damage	Ultimate relative gradient				
		Skempton	Meyerhof	Polshin a Tokar	Bjerrum	ÈSN 73 1001
Frame structures and reinforced bearing walls	Structural	1/150	1/250	1/200	1/150	
	Cracks in walls	1/300	1/500	1/500	1/500	1/500

Type of structure	Type of damage	Ultimate relative deflection $\Delta/l$			
		Burland and Wroth	Meyerhof	Polshin a Tokar	ÈSN 73 1001
Unreinforced bearing walls	Cracks in walls	For $L/H = 1$ - 0.0004 For $L/H = 5$ - 0.0008	0.0004	0.0004	0.0015
	Cracks in bearing structures.	For $L/H = 1$ - 0.0002 For $L/H = 5$ - 0.0004	-	-	-

## Dimensioning of Concrete Structures

Concrete structures can be analyzed according to following standards:

- EN 1992-1-1 (EC 2) or EN 1992-2
- CSN 73 1201R
- CSN 73 6206 (only for Abutment)
- PN-B-03264:2002
- BS 8110:1997
- IS 456
- ACI 318-11
- AS 3600-2001
- SNiP 52-101-2003
- GB 50010-2002

- NZS 3101-2006

## EN 1992-1-1 (EC2) or EN 1992-2

This help contains the following computational methods:

- Materials, coefficients, notation
- Standard values of coefficients
- Verification of rectangular cross-section made from plain concrete
- Verification of rectangular RC cross-section under M, V
- Verification of rectangular RC cross-section under N, M, V
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

## Materials, Coefficients, Notation

The following notation for material parameters is used:

$f_{ck}$  - characteristic value of cylindrical strength of concrete in compression

$f_{cd}$  - design strength of concrete in compression

$f_{cm}$  - average value of tensile strength of concrete

$f_{ctk,0,05}$  - lower value of the characteristic tensile strength of concrete

$f_{ctd}$  - design strength of concrete in tension

$f_{yk}$  - characteristic strength of steel

$f_{yd}$  - design strength of steel in tension

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete - it serves to derive the remaining coefficients of reliability (Tbl. 3.1).

$$f_{cd} = \alpha_{cc} \cdot \frac{f_{ck}}{\gamma_c}$$

$$f_{cm} = f_{ck} + 8$$

$$f_{ctm} = 0,3 \cdot (f_{ck})^{\frac{2}{3}}$$

for:  $f_{ck} \leq 50 \text{ MPa}$

$$f_{ctm} = 2,12 \cdot \ln \left( 1 + \frac{f_{cm}}{10} \right)$$

for:  $f_{ck} > 50 \text{ MPa}$

$$f_{ctk,0,05} = 0,7 \cdot f_{ctm}$$

$$f_{ctd} = \alpha_{ct} \cdot \frac{f_{ctk,0,05}}{\gamma_c}$$

$$E_{cm} = 22 \cdot \left( \frac{f_{cm}}{10} \right)^{0,3}$$

$$f_{yd} = \frac{f_{yk}}{\gamma_s}$$

Standard values of coefficients  $\alpha_{cc}$ ,  $\gamma_c$ ,  $\alpha_{ct}$ ,  $\gamma_s$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

The most common notation for geometrical parameters:

- $b$  - cross-section width
- $h$  - cross-section depth
- $d$  - effective depth of cross-section
- $z$  - lever arm (arm of internal forces)

## Standard Values of Coefficients

The standard contains a number of coefficients, which can be adjusted in **National annexes**. The table provides description of individual coefficients, their values and corresponding article of the standard. In some cases the formula contains a variable, which has no symbol in the standard - in such a case the variable in the **expression is denoted by X**.

Coefficient	Value	Annotations	Article
$\gamma_c$	1,5		2.4.2.4
$\gamma_s$	1,15		2.4.2.4
$\alpha_{cc}$	1		3.1.6
$\alpha_{ct}$	1		3.1.6
$\alpha_{cc,pl}$	0,8		12.3.1
$\alpha_{ct,pl}$	0,8		12.3.1
$k$	1,5		12.6.3
$\rho_{min}$	0,0013		9.2.1.1
$X$	0,26	$\rho_{min} = X \cdot \frac{f_{ctm}}{f_{yk}}$	9.2.1.1
$\rho_{max}$	0,04		9.2.1.1
$\rho_{min}$	0,002		9.5.2

$X$	0,1	$\rho_{\min} = \frac{X N_{Ed}}{f_{yd} A_s}$	9.5.2
$\rho_{\max}$	0,04		9.5.2
$X$	0,18	$C_{Rd,c} = \frac{X}{\gamma_c}$	6.2.2
$v_{\min}$	-	$0,035 \cdot k^{\frac{3}{2}} \cdot f_{ck}^{\frac{1}{2}}$	6.2.2
$X$	0,5	$v_{\max} = X \cdot v \cdot f_{cd}$	6.2.2
$v$	-	$v = 0,6 \cdot \left(1 - \frac{f_{ck}}{250}\right)$	6.2.2
$\cotg \theta_{\min}$	1		6.2.3
$\cotg \theta_{\max}$	2,5		6.2.3

#### National Annex Czech Republic (CSN EN 1992-1-1 - 2010)

Coefficient	Value	Annotations	Article
$\alpha_{ct,pl}$	0,7		12.3.1

other values are standard

#### National Annex Slovakia (STN EN 1992-1-1 - 2008)

all values are standard

#### National Annex Poland (PN EN 1992-1-1 - 2008)

Coefficient	Value	Annotations	Article
$\gamma_c$	1,4		2.4.2.4
$\cotg \theta_{\max}$	2,0		6.2.3

other values are standard

#### EN 1992-2 - 2007

Coefficient	Value	Annotations	Article
$\alpha_{cc}$	0,85		3.1.6

other values are standard

## Verification of Rectangular Cross-Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M_{Ed}$ , normal force  $N_{Ed}$  (applied in the cross-section centroid) and by the shear force  $V_{Ed}$ . The shear strength is provided by (Art. 12.6.3):

$$V_{Rd} = \frac{f_{cvd} \cdot A_{cc}}{k}$$

where:  $A_{cc}$  - compressed area of concrete

$$f_{cvd} = \sqrt{f_{ctd}^2 + \sigma_{cp} \cdot f_{ctd} - \left( \frac{\text{Max}(0; \sigma_{cp} - \sigma_{c,lim})}{2} \right)^2}$$

$$\sigma_{cp} = \frac{N_{Ed}}{A_{cc}}$$

$$\sigma_{c,lim} = f_{cd} - 2 \cdot \sqrt{f_{ctd} \cdot (f_{cd} + f_{ctd})}$$

Standard value of the coefficient  $k$  is built-in the program (Art. 12.6.3) - this value can also be adjusted in the program based on the **selected National annex**.

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions (Art. 12.6.1) depending on the normal force eccentricity  $e$ :

As the greater of:

$$N_{Rd} = b \cdot x \cdot \eta \cdot f_{cd}$$

$$N_{Rd} = \text{Min} \left( \frac{b \cdot h \cdot f_{ctd}}{\frac{6 \cdot e}{h} - 1}, \frac{b \cdot h \cdot f_{cd}}{\frac{6 \cdot e}{h} + 1} \right)$$

Formula express the strength with linear stress-strain diagram of cross-section without the crack.

$$\eta = 1,0 - \frac{\text{Max}(f_{ck}; 50) - 50}{200}$$

$$x = h - 2 \cdot e$$

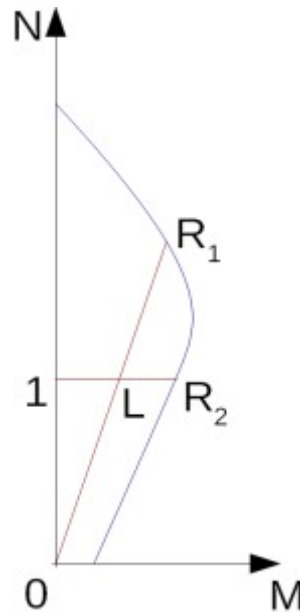
$$e = \text{Max} \left( \text{abs} \left( \frac{M_{Ed}}{N_{Ed}} \right); \frac{h}{30}; 20 \text{mm} \right)$$

Minimal values of excentricity are from article 6.1(3).

$$f_{cd} = \alpha_{cc,pl} \cdot \frac{f_{ck}}{\gamma_c}$$

$$f_{ctd} = \alpha_{ct,pl} \cdot \frac{f_{ctk,005}}{\gamma_c}$$

Standard values of coefficients  $\alpha_{cc,pl}$ ,  $\alpha_{ct,pl}$ ,  $\gamma_c$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|OL| / |OR_1|$  or  $|IL| / |IR_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed excentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M_{Ed}$ .

The permissible moment for a given area of reinforcements  $A_s$  reads (Art. 6.1, Art. 3.1.7(3)):

$$M_{Rd} = \lambda \cdot x \cdot b \cdot \eta \cdot f_{cd} \cdot \left( d - \frac{\lambda}{2} \cdot x \right)$$

$$x = \frac{A_s \cdot f_{yd}}{\lambda \cdot b \cdot \eta \cdot f_{cd}}$$

$$\lambda = 0,8 - \frac{\text{Max}(f_{ck}; 50) - 50}{400}$$

$$\eta = 1,0 - \frac{\text{Max}(f_{ck}; 50) - 50}{200}$$

The computed degree of reinforcement is checked using the following expressions (Art.

9.2.1.1):

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{min} = \text{Max} \left( 0,0013 ; 0,26 \cdot \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0,04$$

**Standard values of coefficients**  $\rho_{min}$ ,  $\rho_{max}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

### Shear

First, the program computes the ultimate shear strength of concrete  $V_{Rd,c}$  (Art. 6.2.2(1)).

$$V_{Rd,c} = \text{Min} \left[ v_{min} ; C_{Rd,c} k (100 \rho_l f_{ck})^{\frac{2}{3}} \right] d b$$

where:

$$k = 1 + \sqrt{\frac{200}{d}} \leq 2,0$$

$$\rho_l = \frac{A_{sl}}{b d} \leq 0,02$$

If the ultimate shear strength of concrete is exceeded, the maximum ultimate shear strength  $V_{Rd,max}$  is checked (Art. 6.2.3(3)).

$$V_{Rd,max} = 0,5 z v f_{cd} b$$

Next, the necessary reinforcement area is given by (Art. 6.2.3(3)):

$$A_{sw,l} = \frac{V_{Ed}}{f_{ywd} z} b$$

**Standard values of coefficients**  $v$ ,  $v_{max}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,002 to 0,0035. Compression reinforcement is not taken into account. Minimum eccentricity is applied: (Art. 6.1(3)):

$$e_0 = \text{Min} \left( \frac{h}{30} ; 20\text{mm} \right)$$



The computed degree of reinforcement is checked using the following expressions (Art. 9.2.1.1):

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{min} = \text{Max} \left( 0,0013 ; 0,26 \cdot \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0,04$$

**Standard values of coefficients**  $\rho_{min}$ ,  $\rho_{max}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

### Shear

First, the program computes the ultimate shear strength of concrete  $V_{Rd,c}$  (Art. 6.2.2(1)).

$$V_{Rd,c} = \text{Min} \left[ v_{min} ; C_{Rd,c} k (100 \rho_l f_{ck})^{\frac{2}{3}} \right] d b$$

where:

$$k = 1 + \sqrt{\frac{200}{d}} \leq 2,0$$

$$\rho_l = \frac{A_{sl}}{b d} \leq 0,02$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{Rd,max}$  is checked (Art. 6.2.3(3)).

$$V_{Rd,max} = 0,5 z v f_{cd} b$$

Next, the necessary reinforcement area is given by (Art. 6.2.3(3)):

$$A_{sw,l} = \frac{V_{Ed}}{f_{ywd} z} b$$

**Standard values of coefficients**  $v$ ,  $v_{max}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,002 - 0,0035. Concrete strength  $\eta \cdot f_{cd}$  is reduced by ten percent due to shape of cross-section (Art. 3.1.7).

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Pile** (Art. 9.8.5)

$$\rho = \frac{4A_s}{\pi d^2}$$

$$A_c < 0,5m^2$$

$$\rho_{min} = 0,005$$

$$A_c > 1m^2$$

$$\rho_{min} = 0,0025$$

where:  $A_c$  - cross-section area of pile

intermediate values are interpolated

$$\rho_{max} = 0.04$$

- **Column** - check for dominant compression (Art. 9.5.2)

$$\rho = \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \text{Max} \left( 0.002; \frac{0.1N_{Ed}}{f_{yd}A_s} \right)$$

$$\rho_{max} = 0.04$$

- **Beam** - check for dominant bending (Art. 9.2.1.1)

$$\rho = 0.5 \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \text{Max} \left( 0.0013; 0.26 \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0.04$$

where:  $d$  - pile diameter

$A_s$  - cross sectional area of reinforcement

**Standard values of coefficients**  $\rho_{min}$ ,  $\rho_{max}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

### Shear

First, the program computes the ultimate shear strength of concrete  $V_{Rd,c}$  (Art. 6.2.2(1)). Formulas are from Art. 6.2.2(1), where the section width ( $b_w$ ) is replaced by  $0.88 \times d$  and effective depth ( $d$ ) is replaced  $0.8 \times d$ .

$$V_{Rd,c} = \text{Min} \left[ \nu_{min}; C_{Rd,c} k (100 \rho_l f_{ck})^{\frac{2}{3}} \right] 0.704 d^2$$

where:

$$k = 1 + \sqrt{\frac{200}{0.8d}} \leq 2.0$$

$$\rho_l = 0.33 \frac{A_{sl}}{0.25 \pi d^2} \leq 0.02$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{Rd,max}$  and

strength of reinforced section  $V_{Rd,s}$  are checked (Art. 6.2.3(3)).

$$V_{Rd,max} = 0.5(0.72d)v f_{cd}0.88d$$

$$V_{Rd,s} = \frac{A_{sw}}{s}0.72df_{ywd}$$

Standard values of coefficients  $v$ ,  $v_{min}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

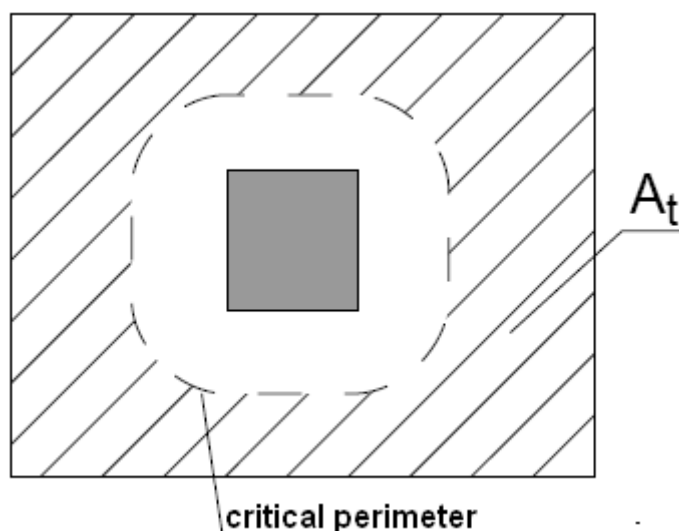
## Verification of Spread Footing for Punching Shear

It is loaded by the prescribed moments  $M_{Ex}$ ,  $M_{Ey}$  and by the shear force  $V_E$  provided by:

$$V_E = \frac{V \cdot A_t}{A}$$

where:

- $A$  - area of footing
- $V$  - assigned vertical force developed in column
- $A_t$  - hatched area in fig.



*Dimensioning of shear reinforcement area  $A_t$*

The program constructs **control sections** at **distances** from  $0,5d$  to  $2d$  in case of **footing without shear reinforcement**. In case of **reinforced footing**, the distances are from  $0,5d$  to  $4d$ , where  $d$  is the **effective depth of footing**. The shear reinforcement is considered in control sections, which are in the distance of less than  $2d$  from the column. The control sections are considered in intervals of  $0,25d$ .

The load stress  $V_{Ed}$  in each control section is found using 6.4.3 (3), the punching shear resistance of footing without shear reinforcement  $V_{Rd,c}$  follows from 6.4.4 (2) and if necessary the punching shear resistance of reinforced footing  $V_{Rd,cs}$  is given by 6.4.5 (1).

Furthermore, the **compression chord resistance** at the column perimeter  $V_{Rd,max}$  is calculated according to 6.4.5 (3).  $V_{Rd,max}$  depends on column dimensions and the footing thickness.

**The control section** with the worst ratio of load and resistance is considered as critical and

marked in the program.

Literature:

EN 1992-1-1 Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M_{Ed}$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as (Art. 3.1.7, Art. 6.1):

$$x = \frac{d - \sqrt{d^2 - \frac{M_{Ed}}{0,5 \cdot b \cdot \eta \cdot f_{cd}}}}{\lambda}$$

Providing the location of neutral axis is less than the allowable one ( $x < x_{max}$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{\lambda \cdot \eta \cdot b \cdot x \cdot f_{cd}}{f_{yd}}$$

Providing the location of neutral axis is greater than the allowable one ( $x > x_{max}$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max} (d - 0,5 \cdot \lambda \cdot x_{max})}{f_{yd} \cdot z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} \cdot f_{yd}}{f_{yd}}$$

$$F_{c,max} = \lambda \cdot \eta \cdot b \cdot x_{max} \cdot f_{cd}$$

The limit location of neutral axis is found from (Art. 5.6.3(2)):

$$x_{max} = 0,45 \cdot d \quad \text{for concrete C40/45 and lower}$$

$$x_{max} = 0,35 \cdot d \quad \text{for concrete C45/50 and higher}$$

The computed degree of reinforcement is checked using the following expressions (Art. 9.3.1.1):

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{min} = \text{Max} \left( 0,0013 ; 0,26 \cdot \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0,04$$

**Standard values of coefficients**  $\rho_{min}$ ,  $\rho_{max}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

If the maximum degree of total reinforcement  $\rho_{max}$  is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $V_{Rd,c}$  (Art. 6.2.2(1)) and the maximum allowable shear force  $V_{Rd,max}$  (Art. 6.2.3(3)).

$$V_{Rd,c} = \text{Min} \left[ v_{min} ; C_{Rd,c} \cdot k (100 \cdot \rho_l \cdot f_{ck})^{\frac{2}{3}} \right] \cdot d$$

where:

$$k = 1 + \sqrt{\frac{200}{d}} \leq 2,0$$

$$\rho_l = \frac{A_{sl}}{b \cdot d} \leq 0,02$$

$$V_{Rd,max} = 0,5 \cdot z \cdot v \cdot f_{cd}$$

As for stirrups the necessary reinforcement area is given by (Art. 6.2.3(3)):

$$A_{sw,l} = \frac{V_{Ed}}{f_{ywd} \cdot z}$$

As for hooks the necessary reinforcement area is given by (Art. 6.2.3(4)):

$$A_{sw,l} = \frac{V_{Ed}}{f_{ywd} \cdot z \cdot \sin \alpha \cdot (1 + \cot \alpha)}$$

**Standard values of coefficients**  $v$ ,  $v_{min}$  are built-in the program - these values can also be input by the user depending on the **selected National annex**.

## Verification of Crack Width

Crack width is evaluated according to chapter 7.3.4 of standard.

First, the maximal tension stress in concrete is calculated at ideal section. If the stress is less than concrete tension strength  $f_{ctm}$  than cracks don't develop.

If not fulfilled than the crack width is determined according to:

$$w_k = s_{r,max} (\varepsilon_{sm} - \varepsilon_{cm})$$

where

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma - k_t \frac{f_{ctm}}{\rho_{p,eff}} (1 + \alpha_e \rho_{p,eff})}{E_s} \geq 0.6 \frac{\sigma_s}{E_s}$$

where  $\sigma_s$  is stress in tensile reinforcement determined at ideal section with crack

$$\alpha_e = \frac{E_s}{E_{cm}}$$

$$\rho_{p,eff} = \frac{A_s}{A_{c,eff}}$$

$$A_{c,eff} = b \times \text{Min} (2.5(h - d), (h - x)/3, h/2)$$

$$k_t = 0.4$$

If the distance of reinforcement bar is less or equal than  $5(c+\phi/2)$ :

$$s_{r,max} = k_3 c + k_1 k_2 k_4 \phi / \rho_{p,eff}$$

where:

$k_1 = 0,8$

$k_2 = 0,5$

$k_3 = 3,4$

$k_4 = 0,425$

If the distance of reinforcement bar is greater than  $5(c+\phi/2)$ :

$$s_{r,max} = 1.3(h - x)$$

## CSN 73 1201 R

This help contains the following computational methods:

- [Materials, coefficients, notation](#)
- [Verification of rectangular cross-section made from plain concrete](#)
- [Verification of rectangular RC cross-section under M, V](#)
- [Verification of rectangular RC cross-section under N, M, V](#)
- [Verification of circular RC cross-section](#)
- [Verification of spread footing for punching shear](#)
- [Design of longitudinal reinforcement for slabs](#)
- [Design of shear reinforcement for slabs](#)

## Materials, Coefficients, Notation

The following notation for material parameters is used:

$R_{bd}$  - design strength of concrete in compression

- $R_{btd}$  - design strength of concrete in tension  
 $\gamma_u$  - coefficient of the shape of cross-section  
 $z$  - lever arm (arm of internal forces)

Coefficient  $\gamma_u$  is given by equation (Art. 5.2.2):

$$\gamma_u = \text{Max} \left( 1 - \frac{20}{1000 \cdot h + 50} ; 0,85 \right)$$

The most common notation for geometrical parameters:

- $b$  - cross-section width  
 $h$  - cross-section depth  
 $h_e$  - effective depth of cross-section  
 $z$  - lever arm (arm of internal forces)

## Verification of Rectangular Cross-Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M$ , normal force  $N$  (applied in the cross-section centroid) and by the shear force  $Q$ . The cross-section bearing capacity subjected to bending moment is given by (Art. 5.2.5):

$$M_u = \frac{b \cdot h^2}{6} \cdot R_{btd} \cdot \gamma_u$$

The shear strength is provided by (Art. 5.3.3, Appendix 9):

$$Q_u = \frac{1}{3} \cdot \kappa_h \cdot \kappa_n \cdot b \cdot h \cdot R_{btd}$$

$$\kappa_h = \text{Max} \left( 1 ; 1,4 - \frac{2}{3} h \right)$$

$$\kappa_n = \text{Min} \left( 2 ; 1 + 0,2 \cdot \frac{N}{b \cdot h \cdot R_{btd}} \right)$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity  $e$  (Art. 5.2.5):

for:

$$e < 0,9 \cdot a_{gc} \Rightarrow N_u = b \cdot x \cdot R_{bd} \cdot \gamma_u$$

$$e > 0,9 \cdot a_{gc} \Rightarrow N_u = \frac{b \cdot h \cdot R_{btd} \cdot \gamma_u}{\frac{6 \cdot e}{h} - 1}$$

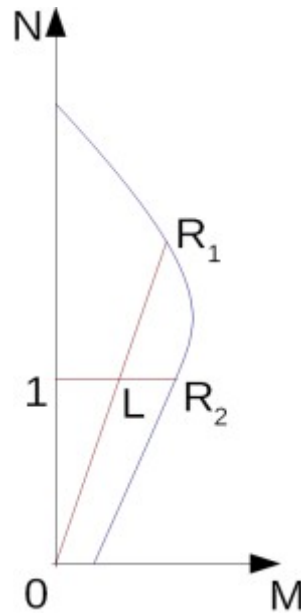
$$x_u = h - 2 \cdot e$$

$$e = \frac{\text{abs}(M)}{N}$$

$$a_{gc} = \frac{h}{2}$$

The ultimate bearing capacity is checked using the following formula (Art. 5.2.5.5):

$$N_{lim} = 0,8 \cdot b \cdot x_u \cdot R_{btd} \cdot \gamma_u$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|OL| / |OR_1|$  or  $|IL| / |IR_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed excentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M_d$ .

The ultimate moment is provided by (Art. 5.2.7):

$$M_u = b \cdot x_u \cdot R_{bd} \cdot \left( h_e - \frac{x_u}{2} \right) \cdot \gamma_u$$

$$x_u = \frac{A_s \cdot R_{sd}}{b \cdot R_{bd}}$$

The program further checks whether the location of neutral axis  $x$  is less than the limit location of neutral axis  $x_{lim}$  given by (Art. 5.2.7.1):



$$x_{lim} = \text{Min} \left( 0,533; \frac{1}{1,25 + \frac{R_{sd}}{420}} \right)$$

The computed degree of reinforcement is checked using the following expressions (Art. 3.1.4.3, Art. 3.1.4.6):

$$\mu_{st,min} = \frac{R_{btd}}{3 \cdot R_{sd}} < \mu_{st} < 0,03 = \mu_{st,max}$$

$$\mu_{st} = \frac{A_s}{b \cdot h}$$

### Shear

First, the program computes the ultimate shear strength of concrete  $Q_{bu}$  (Art. 5.3.3, Appendix 9).

$$Q_{bu} = \frac{1}{3} b h \kappa_q R_{btd}$$

where:

for:  $h \geq 0,3m$  is:  $\kappa_q = 1,25$

for:  $h > 0,15m$  je  $\kappa_q = 1,50$

for:  $h < 0,15m$  je  $\kappa_q = 1,60$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $Q_{max}$  is checked (Art. 5.3.2.1).

$$Q_{max} = \frac{1}{3} b h \text{Min} (R_{bd} ; 18)$$

Next, the necessary reinforcement area is given by (Art. 5.3.4):

$$A_b = \frac{Q - Q_{bu}}{R_{swd} c} b$$

where (Art. 5.3.5):

$$c = \text{Max} \left( \frac{1,2 \cdot b \cdot R_{btd} \cdot h_e^2}{Q - Q_{bu}} ; z \right)$$

The magnitude of  $c$  is bounded by the following expression:

$$c < 0,18 \frac{R_{bd} \cdot h}{\kappa_q \cdot R_{btd}}$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 5.2.8). The maximum allowable strain of concrete in compression is 0,0025. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions (Art. 3.1.4.3, Art. 3.1.4.6):

$$\mu_{st,min} = \frac{R_{btd}}{3 \cdot R_{sd}} < \mu_{st} < 0,03 = \mu_{st,max}$$

$$\mu_{st} = \frac{A_s}{b \cdot h}$$

### Shear

First, the program computes the ultimate shear strength of concrete  $Q_{bu}$  (Art. 5.3.3, Appendix 9).

$$Q_{bu} = \frac{1}{3} b h \kappa_q R_{btd}$$

where:

for: $h \geq 0,3m$	is: $\kappa_q = 1,25$
for: $h > 0,15m$	je $\kappa_q = 1,50$
for: $h < 0,15m$	je $\kappa_q = 1,60$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $Q_{max}$  is checked (Art. 5.3.2.1).

$$Q_{max} = \frac{1}{3} b h \text{Min} (R_{bd} ; 18)$$

Next, the necessary reinforcement area is given by (Art. 5.3.4):

$$A_b = \frac{Q - Q_{bu}}{R_{svd} \cdot c} b$$

where (Art. 5.3.5):

$$c = \text{Max} \left( \frac{1,2 \cdot b \cdot R_{btd} \cdot h_e^2}{Q - Q_{bu}} ; z \right)$$

The magnitude of  $c$  is bounded by the following expression:

$$c < 0,18 \frac{R_{bd} \cdot h}{\kappa_q \cdot R_{btd}}$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art. 5.2.8). The maximum allowable strain of concrete in compression is  $0,0025$ . The degree of reinforcement is checked using the formula:

- **Column** - check for dominant compression (Art. 3.1.4.3, Art. 3.1.4.6)

$$\mu_{st,min} = 0.0008 \leq \mu_{st} \leq 0.04 = \mu_{st,max}$$

$$\mu_{st} = \frac{4A_s}{\pi d^2}$$

- **Beam** - check for dominant bending

$$\mu_{st,min} = \frac{R_{btd}}{3R_{sd}} \leq \mu_{st} \leq 0.03 = \mu_{st,max}$$

$$\mu_{st} = 0.5 \frac{4A_s}{\pi d^2}$$

where:  $d$  - pile diameter  
 $A_s$  - reinforcement area

### Shear

First, the program computes the ultimate shear strength of concrete  $Q_{bu}$  (Art. 5.3.3, Appendix 9).

$$Q_{bu} = \frac{1}{3}(0.88d)(0.88d)\kappa_q R_{btd}$$

where:

for: $0.88d \geq 0.3m$	is: $\kappa_q = 1,25$
for: $0.88d > 0.15m$	je $\kappa_q = 1,50$
for: $0.88d < 0.15m$	je $\kappa_q = 1,60$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $Q_{max}$  and strength of reinforced section  $Q_u$  are checked (Art. 5.3.2.1).

$$Q_{max} = \frac{1}{3}(0.88d)(0.88d)\text{Min}(R_{bd}, 18)$$

$$Q_u = Q_{bu} + A_b R_{swd} c$$

where (Art. 5.3.5):

$$c = \text{Max} \left( \frac{1.2(0.88d)R_{btd}(0.8d)^2}{Q - Q_{bu}}; 0.9(0.8d) \right)$$

The magnitude of  $c$  is bounded by the following expression:

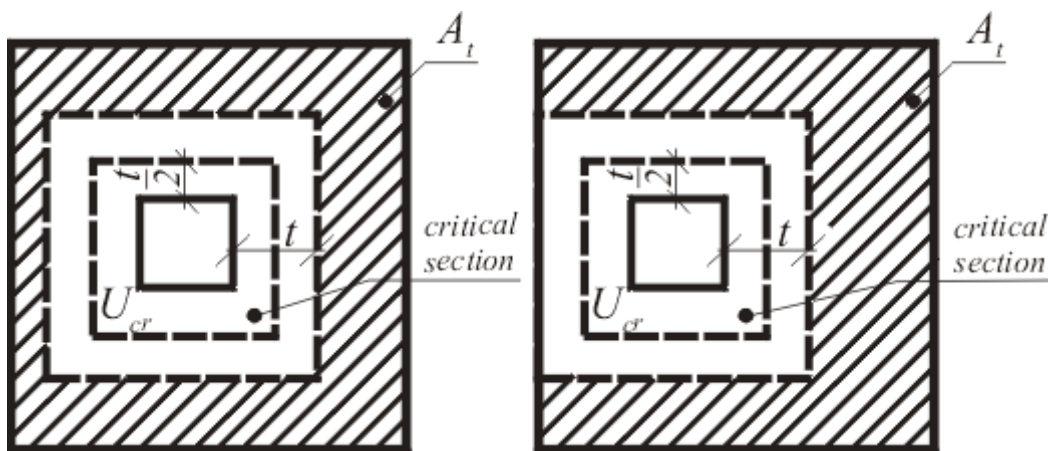
$$c \leq 0.18 \frac{R_{bd}(0.88d)}{\kappa_q R_{btd}}$$

## Verification of Spread Footing for Punching Shear

The program allows to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear  $U_{cr}$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x, M_y$  and by the shear force  $Q_r$  provided by:

$$Q_{dmax} = \frac{Q \cdot A_t}{A}$$

where:  $A$  - area of footing  
 $Q$  - assigned vertical force developed in column  
 $A_t$  - hatched area in fig.



Dimensioning of shear reinforcement area  $A_t$

The program computes the maximal shear force  $Q_{dmax}$  developed in the critical section, the shear force transmitted by concrete with no shear reinforcement  $Q_{bu}$ , and the maximal allowable force  $Q_{max}$ :

$$Q_{bu} = 0,42 \cdot \kappa_h \cdot \kappa_n \cdot \kappa_s \cdot t \cdot R_{bt,d}$$

$$Q_{max} = 2 \cdot Q_{bu}$$

where for:  $\mu_s > \mu_{min}$  is:  $\kappa_s = \text{Min}[1 + 50 \cdot (\mu_s - \mu_{min}); 1,5]$  or else:  $\kappa_s = 1$

$$\kappa_h = \text{Max}\left[1,4 - \frac{2}{3} \cdot h; 1\right]$$

$$\kappa_n = 1$$

For  $Q_{dmax} < Q_{bu}$  no shear reinforcement is needed.

For  $Q_{dmax} > Q_{bu}$  and  $Q_{dmax} < Q_{max}$  the shear reinforcement must be introduced. The ultimate shear force is given by:

$$Q_u = Q_{su} + Q_{bu}$$

$$Q_{su} = \frac{A_s \cdot R_{sd} \cdot \sin \alpha}{U_{cr}}$$

where:  $U_{cr}$  - critical cross-section span

- $\alpha$  - is angle of crooks  
 $A_s$  - overall area of crooks in footing

For  $Q_{dmax} > Q_{max}$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = \frac{h_e - \sqrt{h_e^2 - \frac{M}{0,5 \cdot b \cdot \gamma_u \cdot R_{bd}}}}{0,8}$$

Providing the location of neutral axis is less than the allowable one ( $x < x_{lim}$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{0,8 \cdot b \cdot x \cdot R_{bd}}{R_{sd}}$$

Providing the location of neutral axis is greater than the allowable one ( $x > x_{lim}$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{\frac{M}{\gamma_u} - N_{max} (h_e - 0,5 \cdot 0,8 \cdot x_{lim})}{R_{sd} \cdot z}$$

$$A_{st} = \frac{N_{max} + A_{sc} \cdot R_{scd}}{R_{sd}}$$

$$N_{max} = x_{lim} \cdot 0,8 \cdot b \cdot R_{bd}$$

The limit location of neutral axis is found from:

$$x_{max} = \text{Min} \left( 0,533 ; \frac{1}{1,25 + \frac{R_{sd}}{420}} \right) \cdot h_e$$

The computed degree of reinforcement is checked using the following expressions:

$$\mu_{st,min} = \frac{R_{btd}}{3 \cdot R_{sd}} < \mu_{st} < 0,03 = \mu_{st,max}$$

$$\mu_{st} = \frac{A_s}{b \cdot h}$$

If the maximum degree of tensile reinforcement ( $\mu_{st,max} = 0,03$ ) or total reinforcement ( $\mu_{max} = 0,04$ ), respectively, is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $Q_{bu}$  and the maximum allowable shear force  $Q_{max}$ .

$$Q_{bu} = \frac{1}{3} b \cdot h \cdot \kappa_q \cdot R_{btd}$$

$$Q_{max} = \frac{1}{3} b \cdot h \cdot \text{Min}(R_{bd} ; 18)$$

where:

for: $h \geq 0,3m$	is: $\kappa_q = 1,25$
for: $h > 0,15m$	je $\kappa_q = 1,50$
for: $h < 0,15m$	je $\kappa_q = 1,60$

As for stirrups the necessary reinforcement area is given by:

$$A_b = \frac{Q - Q_{bu}}{R_{swd} \cdot c}$$

As for hooks the necessary reinforcement area is given by:

$$A_b = \frac{Q - Q_{bu}}{R_{swd} (c \cdot \sin \alpha + 0,8 \cdot h_e \cdot \cos \alpha)}$$

where:

$$c = \text{Max} \left( \frac{1,2 \cdot b \cdot R_{btd} \cdot h_e^2}{Q - Q_{bu}} ; z \right)$$

The magnitude of  $c$  is bounded by the following expression:

$$c < 0,18 \frac{R_{bd} \cdot h}{\kappa_q \cdot R_{btd}}$$

## CSN 73 6206

When selecting "**CSN 73 6206**", frame "**Analysis methods**", the verification analysis of decisive joints is performed according to the standard CSN 73 6206 "Design of concrete and steel reinforced concrete bridge structures", including changes a-10/1989 a Z2/1994. The program allows to verify cross-sections from plain concrete or single-ended steel reinforced

concrete. All calculations related to concrete are carried out using the **theory of allowable stresses**.

The main difference when compared to other standards appears in the dimensioning of concrete joints where the earth pressure is computed **always without reduction of input parameters** independently of the input in the frame "Settings".

When performing the verification analysis of cross-sections made either from plain or steel reinforced concrete it is possible input the **coefficient of allowable stress** according to art. 47 **CSN 73 6206** to increase the material allowable stress.

The following joints can be verified by the program:

**Abutment stem - foundation, construction joint** - the cross-section can be made either from plain or steel reinforced concrete. The joint is verified for the load due to normal force and bending moment. The allowable stresses of concrete, steel and concrete in concentric pressure are checked. In case of reinforced concrete the program also checks the degree of reinforcement, cross-sections from plain concrete are then checked for overturning ( $h/2e < 1,35$ ) and translation ( $N*f < 1,5$ ); friction concrete-concrete is assumed as  $f = 0,5$ ).

**Closure wall - bearing block** - the cross-section is verified for the load due to normal force and bending moment. The steel reinforced concrete cross-section is always assumed. The allowable stresses of concrete and steel and the degree of reinforcement are checked.

**Wing wall - abutment** - the joint can be made either from concrete or steel reinforced concrete. The allowable stresses of concrete, steel and concrete in concentric pressure are checked. In case of reinforced concrete the program also checks the degree of reinforcement.

**Front jump of abutment foundation** - the front jump of abutment is verified according to its projection. In case of jump projection  $v < 0,5h_z$  ( $h_z$  is the height of foundation jump) the program checks the magnitude of stress in principal tension due to forces developed in the above-foundation joint. The stress is determined as:

$$\sigma = 0,15 \cdot \frac{N}{d - 2 \cdot \frac{M}{N}}$$

where:  $d$  - width of above-foundation joint

$M, N$  - moment and normal force in above-foundation joint

In case of jump projection  $v > 0,5 h_z$  the jump is analyzed as cantilever bended by the reaction (stress) of foundation soil. The joint can be made either from concrete or steel reinforced concrete. The allowable stresses of concrete, steel and concrete in concentric pressure are checked. In case of reinforced concrete the program also checks the degree of reinforcement.

## PN-B-03264:2002

This help contains the following computational methods:

- [Materials, coefficients, notation](#)
- [Verification of rectangular cross-section made from plain concrete](#)
- [Verification of rectangular RC cross-section under M, V](#)
- [Verification of rectangular RC cross-section under N, M, V](#)
- [Verification of circular RC cross-section](#)
- [Verification of spread footing for punching shear](#)

- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

## Materials, Coefficients, Notation

The following notation for material parameters is used:

- $f_{ck}$  - characteristic strength of concrete in compression  
 $f_{cd}$  - design strength of concrete in compression  
 $f_{ctk}$  - characteristic strength of concrete in tension  
 $f_{ctd}$  - design strength of concrete in tension  
 $f_{yk}$  - characteristic strength of steel  
 $f_{yd}$  - design strength of steel  
 $f_{ctm}$  - design strength of steel in tension

$$f_{cd} = \frac{f_{ck}}{\gamma_c} \cdot \alpha_{cc}$$

$$f_{ctd} = \frac{0,7 \cdot f_{ctm}}{\gamma_c} \cdot \alpha_{ct}$$

$$E_{cm} = 11000 \cdot (f_{ck} + 8)^{0,3}$$

$$f_{ctm} = 0,3 \cdot (f_{ck})^{\frac{2}{3}}$$

- where:  $\alpha_{cc} = 1$   
 $\alpha_{ct} = 1$   
 $\gamma_c = 1,5$  -for reinforced concrete structures  
 $\gamma_c = 1,8$  -for concrete structures

The most common notation for geometrical parameters:

- $b$  - cross-section width  
 $h$  - cross-section depth  
 $d$  - effective depth of cross-section  
 $z$  - lever arm (arm of internal forces)

## Verification of Rectangular Cross-Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M_{Sd}$ , normal force  $N_{Sd}$  (applied in the cross-section centroid) and by the shear force  $V_{Sd}$ . The cross-section bearing capacity subjected to bending moment is given by:



$$M_{Rd} = \frac{b \cdot h^2}{6} \cdot f_{ctd}$$

The shear strength is provided by:

$$V_{Rd,1} = 0,35 \cdot f_{ctd} \cdot k \cdot 1,2 \cdot b \cdot d$$

where:

$$k = \text{Max}(1,6 - d ; 1)$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity  $e$ :

As the greater of:

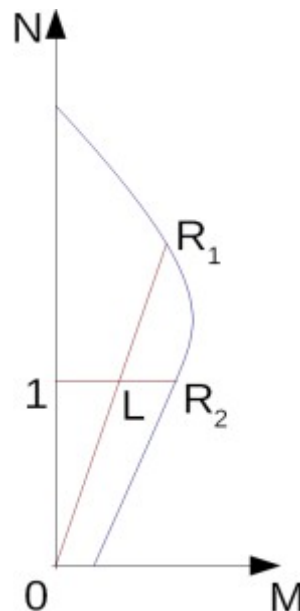
$$N_{Rd} = b \cdot x \cdot f_{cd}$$

$$N_{Rd} = \text{Min} \left( \frac{b \cdot h \cdot f_{ctd}}{\frac{6 \cdot e}{h} - 1}, \frac{b \cdot h \cdot f_{cd}}{\frac{6 \cdot e}{h} + 1} \right)$$

where:

$$x = h - 2 \cdot e$$

$$e = \text{abs} \left( \frac{M}{N} \right)$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|OL|/|OR_1|$  or  $|IL|/|IR_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed eccentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M_{Sd}$ .

The permissible moment for a given area of reinforcements  $A_s$  reads:

$$M_{rd} = 0,8 \cdot x \cdot b \cdot f_{cd} \cdot (d - 0,4 \cdot x)$$

$$x = \frac{A_s \cdot f_{yd}}{0,8 \cdot b \cdot f_{cd}}$$

The program further checks whether the location of neutral axis  $x$  is less than the limit location of neutral axis  $x_{lim}$  given by:

$$x_{max} = \frac{\varepsilon_{cu}}{\varepsilon_{cu} + \varepsilon_{yd}} \cdot d$$

where:

$$\varepsilon_{cv} = 0,0035$$

$$\varepsilon_{yd} = \frac{f_{yd}}{E_s}$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

where:

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{min} = \text{Max} \left( 0,0013 ; 0,26 \cdot \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0,04$$

### Shear

First, the program computes the ultimate shear strength of concrete  $V_{Rd1}$ .

$$V_{Rd1} = 0,35 k f_{ctd} (1,2 + 40 \rho_L) d b$$

where:

$$k = 1,6 - d$$

$$\rho_L = \frac{A_{sL}}{b d} \leq 0,01$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{Rd2}$  is checked.

$$V_{Rd2} = 0,5 v f_{cd} z b$$

where:

$$v = 0,6 \left( 1 - \frac{f_{ck}}{250} \right)$$

Next, the necessary reinforcement area is given by:

$$A_{sw1} = \frac{V_{Ed}}{f_{ywd1} z} b$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is  $0,002 - 0,0035$ . Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = \text{Max} \left( 0,0013 ; 0,26 \cdot \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0,04$$

### Shear

First, the program computes the ultimate shear strength of concrete  $V_{Rd1}$ .

$$V_{Rd1} = 0,35 k f_{ctd} (1,2 + 40 \rho_L) d b$$

where:

$$k = 1,6 - d$$

$$\rho_L = \frac{A_{sL}}{b d} \leq 0,01$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{Rd2}$  is checked.

$$V_{Rd2} = 0,5 v f_{cd} z b$$

where:

$$v = 0,6 \left( 1 - \frac{f_{ck}}{250} \right)$$

Next, the necessary reinforcement area is given by:

$$A_{sw1} = \frac{V_{Ed}}{f_{ywd1} z} b$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is  $0,002 - 0,0035$ . The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression

$$\rho = \frac{4 \cdot A_s}{\pi \cdot d^2}$$

$$\rho_{min} = \text{Max} \left( 0,003 ; \frac{0,15 N_{Ed}}{f_{yd} A_s} \right)$$

$$\rho_{max} = 0,04$$

- **Beam** - check for dominant bending

$$\rho = 0,5 \frac{4 A_s}{\pi d^2}$$

$$\rho_{min} = \text{Max} \left( 0,0013 ; 0,26 \cdot \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0,04$$

where:  $d$  - pile diameter  
 $A_s$  - reinforcement area

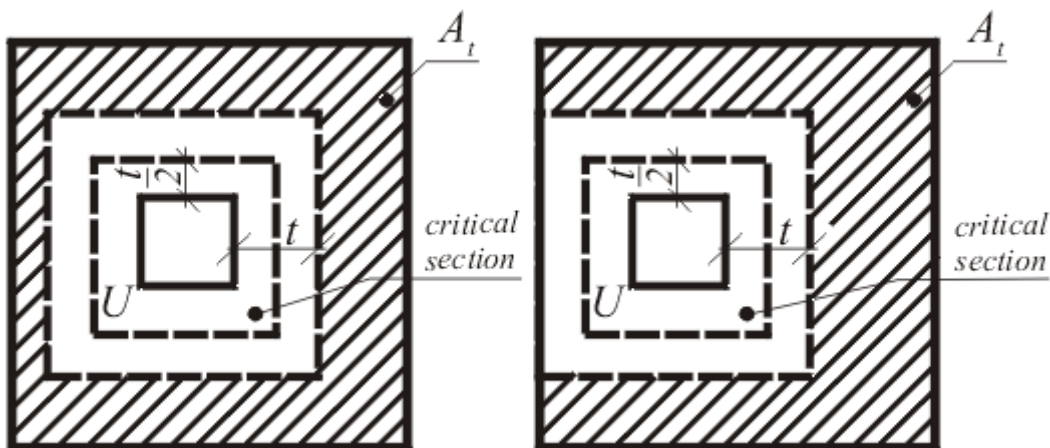
## Verification of Spread Footing for Punching Shear

The critical section loaded in shear  $u$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x$ ,  $M_y$  and by the shear force  $N_{Sd}$  provided by:

$$N_{Sd} = \frac{Q \cdot A_t}{A}$$

where:  $A$  - area of footing  
 $V$  - assigned vertical force developed in column

$A_t$  - hatched area in fig.



Dimensioning of shear reinforcement area  $A_t$

The program computes the maximal shear force  $N_{Sd}$  developed in the critical section, the shear force transmitted by concrete with no shear reinforcement  $N_{Rd1}$  and the maximal allowable force  $N_{Rd,max}$ :

$$N_{Rd} = f_{ctd} \cdot d$$

$$N_{Rd,max} = 1,4 \cdot N_{Rd}$$

For  $N_{Sd} < N_{Rd}$  no shear reinforcement is needed.

For  $N_{Sd} > N_{Rd}$  and  $N_{Sd} < N_{Rd,max}$  the shear reinforcement must be introduced. The ultimate shear force is given by:

$$N_{Rd} = \frac{\sum A_{sw} \cdot f_{yd} \cdot \sin \alpha}{u}$$

- where:
- $u$  - critical cross-section span
  - $\alpha$  - is angle of crooks
  - $A_{sw}$  - overall area of crooks in footing

For  $N_{Sd} > N_{Rd,max}$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M_{Sd}$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = \frac{d - \sqrt{d^2 - \frac{M_{sd}}{0,5 \cdot b \cdot f_{cd}}}}{0,8}$$

Providing the location of neutral axis is less than the allowable one ( $x < x_{max}$ ), the program

determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{0,8 \cdot b \cdot x \cdot f_{cd}}{f_{yd}}$$

Providing the location of neutral axis is greater than the allowable one ( $x > x_{max}$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max} (d - 0,4 \cdot x_{max})}{f_{yd} \cdot Z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} \cdot f_{yd}}{f_{yd}}$$

$$F_{c,max} = 0,8 \cdot x_{max} \cdot b \cdot f_{cd}$$

The limit location of neutral axis is found from:

$$x_{max} = \frac{\varepsilon_{cu}}{\varepsilon_{cu} + \varepsilon_{yd}} \cdot d$$

where:

$$\varepsilon_{cu} = 0,0035$$

$$\varepsilon_{yd} = \frac{f_{yd}}{E_s}$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

where:

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{min} = \text{Max} \left( 0,0013 ; 0,26 \cdot \frac{f_{ctm}}{f_{yk}} \right)$$

$$\rho_{max} = 0,04$$

If the maximum degree of total reinforcement  $\rho_{max}$  is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force

transmitted by concrete  $V_{Rd1}$  and the maximum allowable shear force  $V_{Rd2}$ .

$$V_{Rd1} = 0,35 \cdot k \cdot f_{ctd} \cdot (1,2 + 40 \cdot \rho_L) \cdot d$$

where:

$$k = 1,6 - d$$

$$\rho_L = \frac{A_{sL}}{b \cdot d} \leq 0,01$$

$$V_{Rd2} = 0,5 \cdot v \cdot f_{cd} \cdot z$$

where:

$$v = 0,6 \cdot \left( 1 - \frac{f_{ck}}{250} \right)$$

As for stirrups the necessary reinforcement area is given by:

$$A_{sw1} = \frac{V_{Ed}}{f_{ywd1} \cdot z}$$

As for hooks the necessary reinforcement area is given by:

$$A_{sw2} = \frac{V_{Ed}}{f_{ywd2} \cdot z \cdot \sin \alpha \cdot (1 + \cot \alpha)}$$

## BS 8110:1997

This help contains the following computational methods:

- Materials, coefficients, notation
- Verification of rectangular cross-sections made from plain concrete
- Verification of rectangular RC cross-section under M, V
- Verification of rectangular RC cross-section under N, M, V
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

## Materials, Coefficients, Notation

The following notation for material parameters is used:

$f_{cu}$  - characteristic cube compressive strength of concrete

$f_y$  - characteristic strength of reinforcement

$f_{yd}$  - design strength of steel in tension

$$f_{yd} = \frac{f_y}{1,05}$$

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete.

The most common notation for geometrical parameters:

- $b$  - cross-section width
- $h$  - cross-section depth
- $d$  - effective depth of cross-section
- $z$  - lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

## Verification of Rectangular Cross-Sections Made from Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M$ , normal force  $N$  (applied in the cross-section centroid) and by the shear force  $V$ .

Strength of concrete cross-section subject to the combination of bending moment and normal force with eccentricity  $e$  is derived from the following expressions:

$$N_u = x \cdot 0,45 \cdot f_{cu}$$

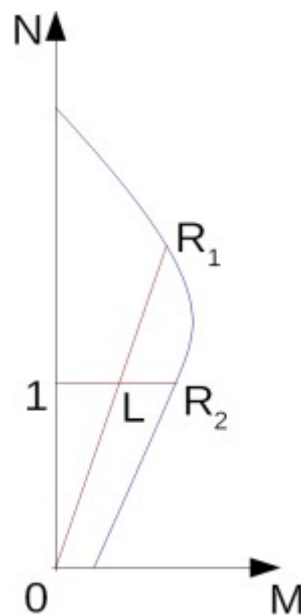
$$x = h - 2 \cdot e$$

$$e = \text{Max} \left( \frac{\text{abs}(M)}{N} ; 0,05 \cdot h ; 20 \text{mm} \right)$$

The shear strength is provided by:

$$V_u = v_c \cdot b \cdot h$$

where:  $v_c$  - is the design value of shear stress in concrete for degree of longitudinal reinforcement  $\rho = 0$  (see [Verification of spread footing for punching shear](#))



Interaction diagram N-M



Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|0L|/|0R_1|$  or  $|1L|/|1R_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed excentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M_u$ .

The permissible moment for a given area of reinforcements  $A_s$  reads:

$$\begin{aligned}M_u &= b \cdot F_c \cdot (d - 0,45 \cdot x) \\F_c &= 0,402 \cdot f_{cu} \cdot x \\x &= \frac{A_s \cdot f_{yd}}{b \cdot 0,402 \cdot f_{cu}}\end{aligned}$$

The program further checks whether the location of neutral axis  $x$  is less than the limit location of neutral axis  $x_{max}$  given by:

$$x_{max} = 0,5 \cdot d$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

where:

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{max} = 0,04$$

$$\rho_{min} = 0,0013 \quad - \text{ for } f_y = 460 \text{ N/mm}^2$$

$$\rho_{min} = 0,0024 \quad - \text{ for } f_y = 250 \text{ N/mm}^2$$

### Shear

First, the program computes the ultimate shear strength of concrete  $V_c$ .

$$V_c = v_c \cdot d \cdot b$$

where:

$$v_c = \frac{0,79 \left( \frac{100 A_s}{b h} \right)^{\frac{1}{3}} \left( \frac{400}{d} \right)^{\frac{1}{4}}}{1,25}$$

The  $v_c$  values are for  $f_{cu}$  above  $25 \text{ N/mm}^2$  multiplied by  $(f_{cu} / 25)^{1/3}$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is

checked.

$$V_{\max} = \text{Min}(5 ; 0,8\sqrt{f_{cu}})db$$

Next, the necessary reinforcement area is given by:

$$A_{sl} = \frac{V - V_c}{0,95 f_{yv} (d - d')} b$$

where:

$$f_{yv} \leq 460 \text{ MPa}$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,002 - 0,0035. Compression reinforcement is not taken into account. Minimum eccentricity is applied:

$$e_0 = \text{Min}(0,05.h ; 20\text{mm})$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{\min} \leq \rho \leq \rho_{\max}$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{\max} = 0,04$$

$$\rho_{\min} = 0,0013$$

- for  $f_y = 460 \text{ N/mm}^2$

$$\rho_{\min} = 0,0024$$

- for  $f_y = 250 \text{ N/mm}^2$

### Shear

First, the program computes the ultimate shear strength of concrete  $V_c$ .

$$V_c = v_c db$$

where:

$$v_c = \frac{0,79 \left( \frac{100 A_s}{bh} \right)^{\frac{1}{3}} \left( \frac{400}{d} \right)^{\frac{1}{4}}}{1,25}$$

The  $v_c$  values are for  $f_{cu}$  above  $25 \text{ N/mm}^2$  multiplied by  $(f_{cu} / 25)^{1/3}$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is checked.

$$V_{max} = \text{Min}(5 ; 0,8\sqrt{f_{cu}})db$$

Next, the necessary reinforcement area is given by:

$$A_{sl} = \frac{V - V_c}{0,95 f_{yv} (d - d')} b$$

where:

$$f_{yv} \leq 460 \text{ MPa}$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is  $0,002 - 0,0035$ .

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression

$$\rho = \frac{4 \cdot A_s}{\pi \cdot d^2}$$

$$\rho_{max} = 0,04$$

$$\rho_{min} = 0,0013$$

- for  $f_y = 460 \text{ N/mm}^2$

$$\rho_{min} = 0,0024$$

- for  $f_y = 250 \text{ N/mm}^2$

- **Beam** - check for dominant bending

$$\rho = 0,5 \frac{4 A_s}{\pi d^2}$$

$$\rho_{max} = 0,06$$

$$\rho_{min} = 0,004$$

where:  $d$  - pile diameter

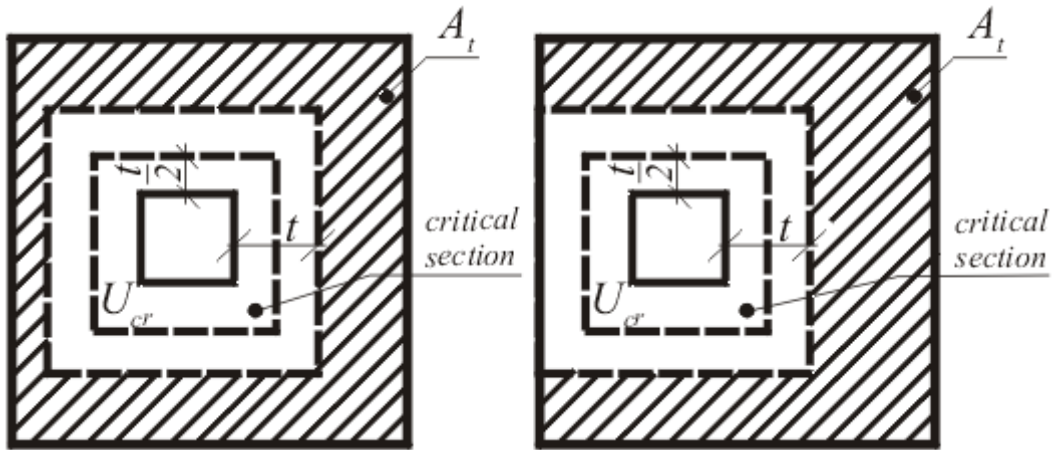
$A_s$  - reinforcement area

## Verification of Spread Footing for Punching Shear

The critical section loaded in shear  $U_{cr}$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x, M_y$  and by the shear force  $V$  provided by:

$$V = \frac{Q \cdot A_t}{A}$$

where:  $A$  - area of footing  
 $Q$  - assigned vertical force developed in column  
 $A_t$  - hatched area in fig.



Dimensioning of shear reinforcement area  $A_t$

The program computes the maximum shear force  $V$  developed in the critical section, the shear force transmitted by concrete with no shear reinforcement  $V_c$ , and the maximal allowable force  $V_u$ :

$$V_c = v_c \cdot d$$

$$V_u = v_u \cdot d$$

where:

$$v_c = \frac{0,79 \cdot \left( \frac{100 \cdot A_s}{b \cdot h} \right)^{\frac{1}{3}} \cdot \left( \frac{400}{d} \right)^{\frac{1}{4}}}{1,25}$$

The  $v_c$  values are for  $f_{cu}$  above  $25 \text{ N/mm}^2$  multiplied by  $(f_{cu} / 25)^{1/3}$

$$v_u = 0,8 \cdot \sqrt{f_{cu}} \text{ or } 5 \text{ N/m}^2$$

$v_u$  is ultimate shear stress

For  $V < V_c$  no shear reinforcement is needed.

For  $V > V_c$  and  $V_c < V_u$  it is necessary to design shear reinforcement. The permissible shear force is given by:

$$V_{rd} = V_c + V_{us}$$

$$V_{us} = \frac{\sum 0,95 A_{us} \cdot f_{yv} \cdot \sin \alpha}{u}$$

$$f_{yv} = \text{Min}(f_y ; 460)$$

where:  $u$  critical cross-section span

$\alpha$	angle of crooks
$A_{us}$	overall area of crooks in footing

For  $V > V_u$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section depth.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M_d$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = \frac{d - \sqrt{d^2 - \frac{2.M_d}{0,402.b.f_{cu}}}}{0,9}$$

Providing the location of neutral axis is less than the allowable one ( $x < x_{max}$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{0,402.b.f_{cu}.0,9.x}{f_{yd}}$$

Providing the location of neutral axis is greater than the allowable one ( $x > x_{max}$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max}(d - 0,45.x_{max})}{f_{yd}.z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc}.f_{yd}}{f_{yd}}$$

$$F_{c,max} = 0,9.x_{max}.0,67.\frac{f_{cu}}{1,5}$$

The limit location of neutral axis is found from:

$$x_{u,lim} = 0,5$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{max} = 0,04$$

$$\rho_{min} = 0,0013 \quad - \text{ for } f_y = 460 \text{ N/mm}^2$$

$$\rho_{min} = 0,0024 \quad - \text{ for } f_y = 250 \text{ N/mm}^2$$

If the maximum degree of reinforcement  $\rho_{max}$  is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $V_c$  and the maximum allowable shear force  $V_{max}$ .

$$V_c = v_c \cdot d$$

where:

$$v_c = \frac{0,79 \cdot \left( \frac{100 \cdot A_s}{b \cdot h} \right)^{\frac{1}{3}} \cdot \left( \frac{400}{d} \right)^{\frac{1}{4}}}{1,25}$$

The  $v_c$  values are for  $f_{cu}$  above  $25 \text{ N/mm}^2$  multiplied by  $(f_{cu}/25)^{1/3}$

$$V_{max} = \text{Min}(5 ; 0,8 \cdot \sqrt{f_{cu}}) \cdot d$$

As for stirrups the necessary reinforcement area is given by:

$$A_{sl} = \frac{V - V_c}{0,95 \cdot f_{yv} \cdot (d - d')}$$

As for hooks the necessary reinforcement area is given by:

$$A_{sb} = \frac{V - V_c}{0,95 \cdot f_{yv} \cdot (\sin \beta + \cos \beta) \cdot (d - d')}$$

where:

$$f_{yv} \leq 460 \text{ MPa}$$

## IS 456

This help contains the following computational methods:

- Materials, coefficients, notation
- Verification of rectangular cross-sections made from plain concrete
- Verification of rectangular RC cross-section under M, V
- Verification of rectangular RC cross-section under N, M, V
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

## Materials, Coefficients, Notation

The following notation for material parameters is used:

- $f_{ck}$  - characteristic cube compressive strength of concrete
- $f_{cd}$  - design strength of concrete in compression
- $f_{ctk}$  - characteristic strength of concrete in tension
- $f_{ctd}$  - design strength of concrete in tension
- $f_y$  - characteristic strength of steel
- $f_{yd}$  - design strength of steel in tension

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete - it serves to derive the remaining coefficients of reliability.

$$f_{cd} = 0,67 \cdot \frac{f_{ck}}{1,5}$$

$$f_{ctk} = 0,7 \cdot \sqrt{f_{ck}}$$

$$f_{ctd} = \frac{f_{ctk}}{1,5}$$

$$E_c = 5000 \cdot \sqrt{f_{ck}}$$

$$f_{yd} = \frac{f_y}{1,15}$$

The most common notation for geometrical parameters:

- $b$  - cross-section width
- $h$  - cross-section depth
- $d$  - effective depth of cross-section
- $z$  - lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

## Verification of Rectangular Cross-Sections Made from Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M$ , normal force  $N$  (applied in the cross-section centroid) and by the shear force  $V$ :

$$M_{rd} = \frac{b h^2}{6} f_{ctd}$$

The shear strength is provided by:

$$V_{rd} = \tau_c b h$$

where:  $\tau_c$  - is the design value of stress in concrete obtained from table 19 of the IS456 standard for degree of longitudinal reinforcement  $\rho = 0$ .

Strength of concrete cross-section subject to the combination of bending moment and normal force with eccentricity  $e$  is derived from the following expressions:

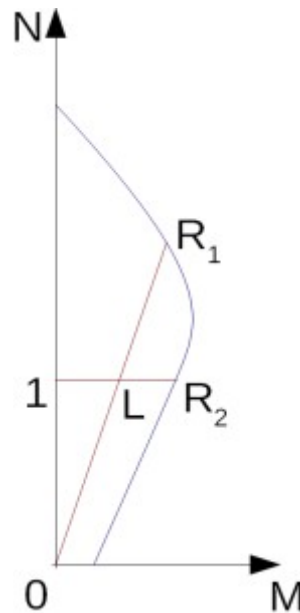
$$P_{rd} = b x f_{cd}$$

$$P_{rd} = \text{Min} \left( \frac{b h f_{ctd}}{\frac{6e}{h} - 1} ; \frac{b h f_{cd}}{\frac{6e}{h} + 1} \right)$$

where:

$$x = h - 2e$$

$$e = \frac{\text{abs}(M_u)}{P_u}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|OL|/|OR_1|$  or  $|IL|/|IR_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed eccentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M$ . The permissible moment for a given area of reinforcements  $A_s$  reads:

$$M_{rd} = b F_c (d - 0,42 x)$$

$$F_c = 0,36 f_{ck} x$$



$$x = \frac{A_s f_{yd}}{b 0,36 f_{ck}}$$

The program further checks whether the location of neutral axis  $x$  is less than the limit location of neutral axis  $x_{max}$  given by:

$$x_{max} = 0,53d \quad \text{- for steel Fe 250}$$

$$x_{max} = 0,48d \quad \text{- for steel Fe 400}$$

$$x_{max} = 0,46d \quad \text{- for steel Fe 500}$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

$$\rho_{min} = \frac{0,85}{f_y}$$

$$\rho_{max} = 0,04$$

### Shear

First, the program computes the ultimate shear strength of concrete  $V_{uc}$ .

$$V_{uc} = \tau_c db$$

where:  $\tau_c$  is determined according to table 19 standard IS 456 : 2000.

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{uc,max}$  is checked.

$$V_{uc,max} = \tau_{c,max} db$$

where:  $\tau_{c,max}$  is determined according to table 20 standard IS 456 : 2000.

Next, the necessary reinforcement area is given by:

$$A_{sv} = \frac{V_u - V_{uc}}{0,87 f_y d} b$$

where:

$$f_y \leq 415 \text{ MPa}$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is  $0,002 - 0,0035$ . Compression reinforcement is not taken into account. Minimum eccentricity is applied:

$$e_0 = \text{Max}\left(\frac{h}{30}; 20\text{mm}\right)$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

$$\rho_{min} = \frac{0,85}{f_y}$$

$$\rho_{max} = 0,04$$

### Shear

First, the program computes the ultimate shear strength of concrete  $V_{uc}$ .

$$V_{uc} = \tau_c db$$

where:  $\tau_c$  is determined according to table 19 standard IS 456 : 2000.

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{uc,max}$  is checked.

$$V_{uc,max} = \tau_{c,max} db$$

where:  $\tau_{c,max}$  is determined according to table 20 standard IS 456 : 2000.

Next, the necessary reinforcement area is given by:

$$A_{sv} = \frac{V_u - V_{uc}}{0,87 f_y d} b$$

where:

$$f_y \leq 415 \text{ MPa}$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is  $0,002 - 0,0035$ .

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression

$$\rho = \frac{4.A_s}{\pi.d^2}$$

$$\rho_{min} = 0,008$$

$$\rho_{max} = 0,04$$

- **Beam** - check for dominant bending

$$\rho = 0,5 \frac{4 A_s}{\pi d^2}$$

$$\rho_{min} = \frac{0,85}{f_y}$$

$$\rho_{max} = 0,04$$

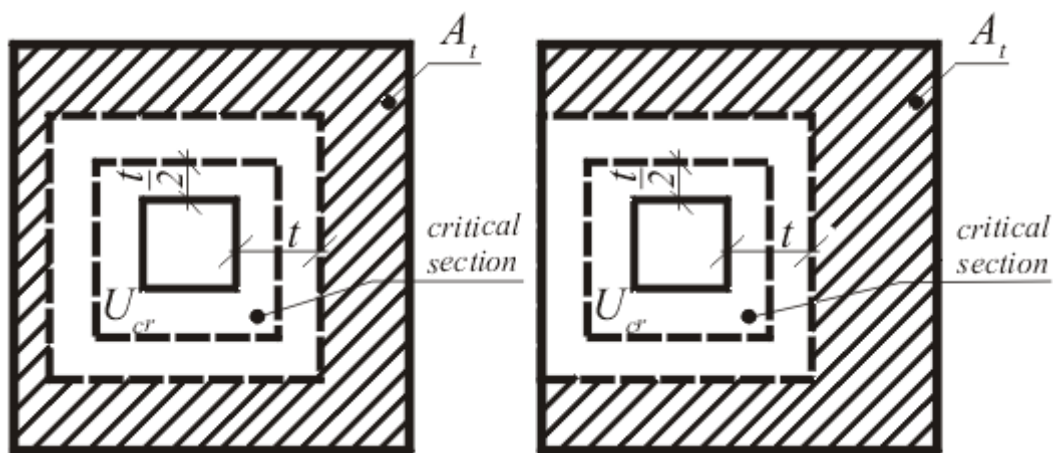
where:  $d$  - pile diameter  
 $A_s$  - reinforcement area

## Verification of Spread Footing for Punching Shear

The critical section loaded in shear  $U_{cr}$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x, M_y$  and by the shear force  $V_r$  provided by:

$$V_r = \frac{Q \cdot A_t}{A}$$

where:  $A$  - area of footing  
 $Q$  - assigned vertical force developed in column  
 $A_t$  - hatched area in fig.



Dimensioning of shear reinforcement area  $A_t$

The program computes the maximum shear force  $V$  developed in the critical section, the shear force transmitted by concrete with no shear reinforcement  $V_c$ , and the maximal allowable force  $V_{max}$ :

$$V_c = \tau_{rd} \cdot k_s \cdot h$$

$$V_{max} = 1,5 \cdot V_c$$

where:

$$\tau_c = 0,25 \cdot \sqrt{f_{ctk}}$$

$$k_s = \text{Min} \left( 0,5 + \frac{c_x}{c_y} ; 1 \right)$$

where:  $c_x, c_y$  - are dimensions of footing column

For  $V < V_c$  no shear reinforcement is needed.

For  $V > V_c$  and  $V < V_{max}$  it is necessary to design shear reinforcement. The permissible shear force is given by:

$$V_{rd,3} = \frac{1}{2} V_c + V_{us}$$

$$V_{us} = \frac{\sum 0,87 A_{sv} \cdot f_{yd} \cdot \sin \alpha}{u}$$

where:  $u$  critical cross-section span  
 $\alpha$  is angle of crooks  
 $A_{us}$  overall area of crooks in footing

For  $V > V_{max}$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section depth.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M_{rd}$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = \frac{d - \sqrt{d^2 - \frac{M_{rd}}{0,96 \cdot b \cdot f_{cd}}}}{0,84}$$

Providing the location of neutral axis is less than the allowable one ( $x < x_{max}$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = 0,36 \cdot b \cdot x \cdot f_{ck}$$

Providing the location of neutral axis is greater than the allowable one ( $x > x_{max}$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max} (d - 0,42 \cdot x_{max})}{f_{yd} \cdot z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} \cdot f_{yd}}{f_{yd}}$$

$$F_{c,max} = 0,36 \cdot x_{max} \cdot b \cdot f_{ck}$$

The limit location of neutral axis is found from:

$x_{max} = 0,53.d$	for steel Fe 250
$x_{max} = 0,48.d$	for steel Fe 400
$x_{max} = 0,46.d$	for steel Fe 500

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = \frac{0,85}{f_y}$$

$$\rho_{max} = 0,04$$

If the maximum degree of tensile reinforcement ( $\rho_{t,max} = 0,04$ ) or total reinforcement ( $\rho_{max} = 0,08$ ), respectively, is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $V_{cu}$  and the maximum allowable shear force  $V_{uc,max}$ .

$$V_{uc} = \tau_c \cdot d$$

where:  $\tau_c$  is determined according to table 19 standard IS 456 : 2000.

$$V_{uc,max} = \tau_{c,max} \cdot d$$

where:  $\tau_{c,max}$  is determined according to table 20 standard IS 456 : 2000.

As for stirrups the necessary reinforcement area is given by:

$$A_{sv} = \frac{V_u - V_{uc}}{0,87 \cdot f_y \cdot d}$$

As for hooks the necessary reinforcement area is given by:

$$A_{sv} = \frac{V_u - V_{uc}}{0,87 \cdot f_y \cdot (\sin \alpha + \cos \alpha) \cdot d}$$

where:

$$f_y \leq 415 \text{ MPa}$$

## IS Road Bridges

### ACI 318-11

This help contains the following computational methods:

- Materials, coefficients, notation
- Verification of rectangular cross-section made from plain concrete
- Verification of rectangular RC cross-section under M, V
- Verification of rectangular RC cross-section under N, M, V
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

### Materials, Coefficients, Notation

The following notation for material parameters is used:

- $f'_c$  - design strength of concrete in compression  
 $E_c$  - modulus of elasticity

The modulus of elasticity is provided by (Art. 8.5.1):

$$E_c = 57000 \cdot \sqrt{f'_c}$$

The most common notation for geometrical parameters:

- $b$  - cross-section width  
 $h$  - cross-section depth  
 $d$  - effective depth of cross-section

### Verification of Rectangular Cross-Section Made of Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M$ , normal force  $P$  (applied in the cross-section centroid) and by the shear force  $V_n$ .

The shear strength is provided by (Art. 22.5.4, Art. 9.3.5):

$$\phi \cdot V_n \geq V_u$$

$$V_n = \frac{4}{3} \cdot \sqrt{f'_c} \cdot b \cdot h$$

$$\phi = 0,6$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions (Art. 22.5.3, Art. 9.3.5):

for compression side:

$$\frac{P_u}{\phi \cdot P_n} + \frac{M_u}{\phi \cdot M_n} \leq 1$$

where:

$$P_n = 0,6 \cdot f'_c \cdot b \cdot h$$

$$M_n = 0,85 \cdot f'_c \cdot S$$

$$S = \frac{b \cdot h^2}{6}$$

$$\phi = 0,6$$

for tension side:

$$\frac{M_u}{S} - \frac{P_u}{b \cdot h} \leq 5 \cdot \phi \cdot \sqrt{f'_c}$$

where:

$$\phi = 0,6$$

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M_u$ .

The ultimate moment is provided by (Ch. 10, Art. 10.2.7.3, Art. 9.3.2.1):

$$M_u < \phi \cdot M_n$$

$$M_n = A_s \cdot f_y \cdot \left( d - \frac{a}{2} \right)$$

$$a = \frac{A_s \cdot f_y}{b \cdot 0,85 \cdot f'_c}$$

$$c = \frac{a}{\beta_1}$$

$$\beta_1 = 0,85 - 0,05 \cdot \frac{f'_c - 4000}{1000}$$

$$0,65 < \beta_1 < 0,85$$

$$\phi = 0,9$$

The computed degree of reinforcement is checked using the following expressions (Art. 10.5.1):

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{min} = \frac{Max(3\sqrt{f'_c}; 200)}{f_y}$$

The program further checks whether the location of neutral axis  $c$  is less than the limit location of neutral axis  $c_{max}$  given by (Art. 10.3.2):

$$c_{max} = \frac{0,003}{0,003 + 0,004} .d$$

### Shear

First, the program computes the ultimate shear strength of concrete  $V_c$  (Art. 11.2.1.1).

$$V_c = 2\sqrt{f'_c} db$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is checked (Art. 11.2.1.1 + Art. 11.4.7.9).

$$V_{max} = 10\sqrt{f'_c} db$$

Next, the necessary reinforcement area is given by (Art. 11.4.7):

$$A_v = \frac{V_u - \phi V_c}{\phi f_{yt} d} b$$

where (Art. 11.4.2, Art. 9.3.2.3):

$$f_{yt} \leq 60000 \text{ psi}$$

$$\phi = 0,75$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 10.3, Art. 10.4). The maximum allowable strain of concrete in compression is  $0,003$ . Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions (Art. 10.5.1):

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = \frac{Max(3\sqrt{f'_c}; 200)}{f_y}$$

### Shear



First, the program computes the ultimate shear strength of concrete  $V_c$  (Art. 11.2.1.1).

$$V_c = 2\sqrt{f'_c} db$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is checked (Art. 11.2.1.1 + Art. 11.4.7.9).

$$V_{max} = 10\sqrt{f'_c} db$$

Next, the necessary reinforcement area is given by (Art. 11.4.7):

$$A_v = \frac{V_u - \phi V_c}{\phi f_{yt} d} b$$

where (Art. 11.4.2, Art. 9.3.2.3):

$$f_{yt} \leq 60000 \text{ psi}$$

$$\phi = 0,75$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art. 10.3, Art. 10.4). The maximum allowable strain of concrete in compression is  $0,003$ . The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression (Art. 10.9.1)

$$\rho = \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = 0.01$$

$$\rho_{max} = 0.08$$

- **Beam** - check for dominant bending (Art. 10.5.1)

$$\rho = 0.5 \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \frac{\text{Max} \left( 3\sqrt{f'_c}; 200 \right)}{f_y}$$

where:  $d$  - pile diameter  
 $A_s$  - reinforcement area

### Shear

First, the program computes the ultimate shear strength of concrete  $V_c$  (Art. 11.2.1.1, Art. 11.2.3).

$$V_c = 2\sqrt{f'_c} 0.8d^2$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  and

strength of reinforced section  $V_s$  are checked (Art. 11.2.1.1 + Art. 11.4.7.9, Art. 11.2.3, Art. 11.4.7.2).

$$V_{max} = 10\sqrt{f'_c}0.8d^2$$

$$V_s = \frac{A_v f_{yt} 0.8d}{s}$$

where (Art. 11.4.2, Art. 9.3.2.3):

$$f_{yt} \leq 60000 \text{ psi}$$

$$\phi = 0.75$$

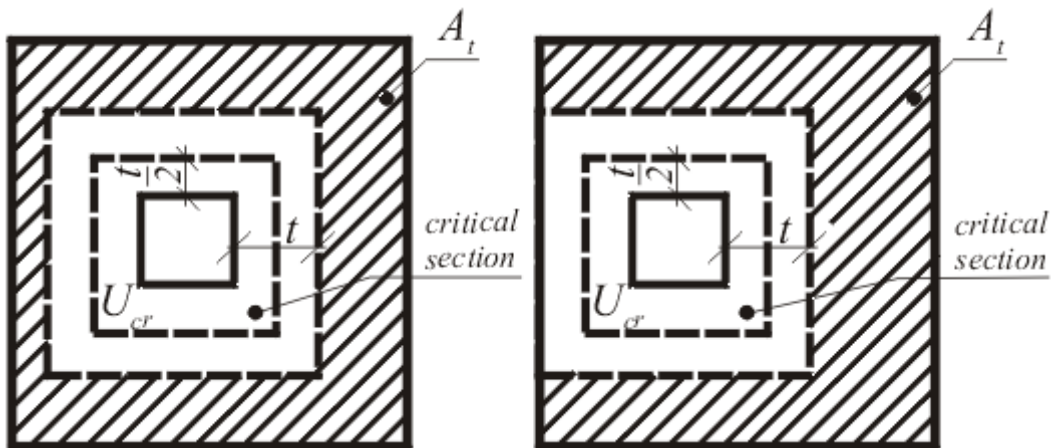
## Verification of Spread Footing for Punching Shear

The program allows to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear  $b_o$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x, M_y$  and by the shear force  $V_u$  provided by:

$$V_u = \frac{VA_t}{A}$$

where:

- $A$  - area of footing
- $V$  - assigned vertical force developed in column
- $A_t$  - hatched area in fig.



*Dimensioning of shear reinforcement area  $A_t$*

The program computes the maximal shear force  $V_u$  developed in the critical section, the shear force transmitted by concrete with no shear reinforcement  $V_c$  as minimum of values (Art. 11.11.2.1):

$$V_c = \left(2 + \frac{4}{\beta_c}\right) \sqrt{f'_c} b_o d$$

where  $\beta_c$  is ratio of long side to short side of column.

$$V_c = \left( 2 + \frac{\alpha_s d}{b_o} \right) \sqrt{f'_c b_o d}$$

where  $\alpha_s =$  40 - inner column  
 30 - edge column  
 20 - corner column

$$V_c = 4\sqrt{f'_c b_o d}$$

and the maximal allowable force  $V_{max}$  (Art. 11.11.3.2):

$$V_{max} = 6\sqrt{f'_c b_o d}$$

For  $V_u < \phi * V_c$  no shear reinforcement is needed.

For  $V_u > \phi * V_c$  and  $V_u < \phi * V_{max}$  the shear reinforcement must be introduced. The ultimate shear force is given by (Art. 11.11.3.1):

$$V_n = \phi \left( 2\sqrt{f'_c b_o d} + A_v f_{yt} \sin \alpha \right)$$

$$\phi = 0.75$$

where:  $b_o$  - critical cross-section span  
 $\alpha$  - is angle of crooks  
 $A_v$  - overall area of bends in footing

For  $V_c > \phi * V_{max}$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = d - \sqrt{d^2 - \frac{2.M_d}{0,85.\phi.b.f'_c}}$$

Providing the location of neutral axis is less than the allowable one ( $x < x_{lim}$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{0,85.\phi.f'_c.b.x.\beta_1}{f_y}$$

Providing the location of neutral axis is greater than the allowable one ( $x > x_{lim}$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{\frac{M}{\phi} - F_{c,max}(d - 0,45 \cdot x_{max})}{f_{yd} \cdot z}$$

$$A_{st} = \frac{M - A_{sc} \cdot f_y}{f_y}$$

$$F_{c,max} = 0,85 \cdot \phi \cdot b \cdot f'_c$$

where:

$$\phi = 0,9$$

The limit location of neutral axis is found from:

$$x_{u,lim} = \frac{0,003}{0,003 + 0,004} \cdot d$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

$$\rho = \frac{A_s}{b \cdot d}$$

$$\rho_{min} = \frac{\text{Max}(3 \cdot \sqrt{f'_c}; 200)}{f_y}$$

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $V_c$  (Art. 11.2.1.1) and the maximum allowable shear force  $V_{max}$  (Art. 11.2.1.1 + Art. 11.4.7.9).

$$V_c = 2 \cdot \sqrt{f'_c} \cdot d$$

$$V_{max} = 10 \cdot \sqrt{f'_c} \cdot d$$

As for stirrups the necessary reinforcement area is given by (Art. 11.4.7.2):

$$A_v = \frac{V_u - \phi V_c}{\phi \cdot f_{yt} \cdot d}$$

As for hooks the necessary reinforcement area is given by (Art. 11.4.7.4):

$$A_v = \frac{V_u - \phi V_c}{\phi \cdot f_{yt} \cdot (\sin \alpha + \cos \alpha) \cdot d}$$

where (Art. 11.4.2, Art. 9.3.2.3):

$$f_{yt} \leq 60000 \text{ psi}$$

$$\phi = 0,75$$

## AS 3600-2001

This help contains the following computational methods:

- Materials, coefficients, notation
- Verification of rectangular cross-sections made from plain concrete
- Verification of rectangular RC cross-section under  $M, V$
- Verification of rectangular RC cross-section under  $N, M, V$
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

### Materials, Coefficients, Notation

The following notation for material parameters is used:

$f'_c$	- characteristic compressive cylinder strength of concrete at 28 days
$E_c$	- mean value of the modulus of elasticity of concrete at 28 days
$f'_{cf}$	- characteristic flexural tensile strength of concrete
$f'_{ct}$	- characteristic principal tensile strength of concrete
$f_{sy}$	- yield strength of reinforcing steel

$$E_c = \rho^{1.5} \cdot 5,056 \cdot \sqrt{f'_c}$$

$$f'_{cf} = 0,6 \cdot \sqrt{f'_c}$$

$$f'_{ct} = 0,4 \cdot \sqrt{f'_c}$$

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete.

The most common notation for geometrical parameters:

$b$	- cross-section width
$D$	- cross-section depth
$d$	- effective depth of cross-section
$z$	- lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

### Verification of Rectangular Cross-Sections Made from Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M$ , normal force  $N$  (applied in the cross-section centroid) and by the shear force  $V$ .

The shear strength is provided by:

$$\phi \cdot V_u < V$$

$$V_u = 0,15 \cdot b \cdot D \cdot (f'_c)^{\frac{1}{3}}$$

$$\phi = 0,6$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions:

$$\frac{N}{\phi \cdot N_u} + \frac{M}{\phi \cdot M_u} \leq 1$$

where:

$$N_u = 0,45 \cdot f'_c \cdot A_g$$

$$M_u = \frac{b \cdot h^2}{6} \cdot f'_c$$

$$\phi = 0,6$$

where:  $A_g$  - loaded area

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment M. The permissible moment for a given area of reinforcements  $A_s$  reads:

$$M_x < \phi \cdot M_{uo}$$

$$\phi = 0,8$$

$$M_{uo} = A_s \cdot f_{sy} \cdot \left( d - \frac{c}{2} \right)$$

$$c = \frac{A_s \cdot f_{sy}}{b \cdot 0,85 \cdot f'_c}$$

$$\gamma = [0,85 - 0,007 \cdot (f'_c - 28)]$$

$$0,65 \leq \gamma \leq 0,85$$

The program further checks whether the location of neutral axis parameter  $k_u$  is less than the limit value:

$$k_u \leq 0,4$$

$$k_u = \frac{x}{d}$$

$$x = \frac{c}{\gamma}$$

where:  $x$  - depth of neutral axis

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = \left[ 0,22 \cdot \left( \frac{D}{d} \right)^2 \cdot \frac{f'_{cf}}{f_{sy}} \right] \leq \rho$$

The program further checks ultimate shear strength:

$$V < \phi \cdot V_{uc}$$

$$\phi = 0,7$$

where:

$$V_{uc} = \beta_1 \cdot b \cdot d \cdot \left( \frac{A_s \cdot f'_c}{b \cdot d} \right)^{\frac{1}{3}}$$

$$\beta_1 = \text{Max} \left[ 1,1 ; \left( 1,6 - \frac{d}{1000} \right) \cdot 1,1 \right]$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,002 - 0,0035. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = \left[ 0,22 \cdot \left( \frac{D}{d} \right)^2 \cdot \frac{f'_{cf}}{f_{sy}} \right] \leq \rho$$

The program further checks ultimate shear strength:

$$V < \phi \cdot V_{uc}$$

$$\phi = 0,7$$

where:

$$V_{uc} = \beta_1 \cdot b \cdot d \cdot \left( \frac{A_s \cdot f'_c}{b \cdot d} \right)^{\frac{1}{3}}$$

$$\beta_1 = \text{Max} \left[ 1,1 ; \left( 1,6 - \frac{d}{1000} \right) \cdot 1,1 \right]$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is  $0,002 - 0,0035$ . The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression

$$\rho = \frac{4 \cdot A_s}{\pi \cdot d^2}$$

$$\rho_{min} = 0,01$$

$$\rho_{max} = 0,04$$

- **Beam** - check for dominant bending

$$\rho = 0,5 \frac{4 A_s}{\pi d^2}$$

$$\rho_{min} = \left[ 0,22 \cdot \left( \frac{D}{d} \right)^2 \cdot \frac{f'_{cf}}{f_{sy}} \right]$$

$$\rho_{max} = 0,04$$

where:  $D$  - pile diameter  
 $A_s$  - reinforcement area

## Verification of Spread Footing for Punching Shear

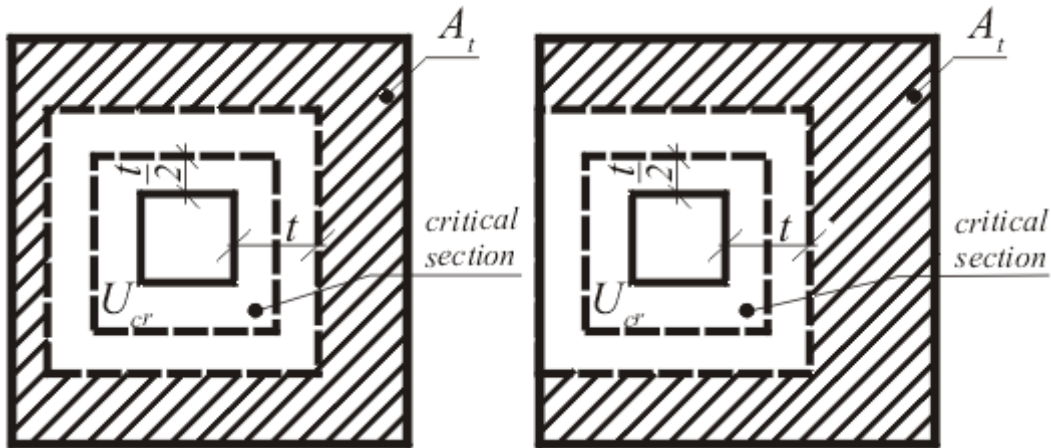
The program allows to verify spread footing for punching shear. The critical section loaded in shear  $U_{cr}$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x, M_y$  and by the shear force  $V^*$  provided by:

$$V^* = \frac{V \cdot A_t}{A}$$

where:  $A$  - area of footing



- $V$  - assigned vertical force developed in column
- $A_t$  - hatched area in fig.



*Dimensioning of shear reinforcement area  $A_t$*

The program checks, whether the cross-section bursting strength is sufficient according to the relation:

$$V^* = \phi \cdot V_u$$

where:

$$\phi = 0,7$$

$$V_u = \frac{V_{uo}}{1 + \frac{u \cdot M_v^*}{8 \cdot V^* \cdot a \cdot d}}$$

$$V_{uo} = u \cdot d \cdot f_{cv}$$

$$f_{cv} = \text{Min} \left[ 0,34 \cdot \sqrt{f'_c} ; 0,17 \cdot \left( 1 + \frac{2}{\beta_h} \right) \cdot \sqrt{f'_c} \right]$$

- where:  $\beta_h$  - the ratio of the longest overall dimension of the effective loaded area,  $Y$ , to the overall dimension,  $X$ , measured perpendicular to  $Y$
- $a$  - the dimension of the critical shear perimeter measured parallel to the direction of  $M_v^*$
- $M_v^*$  - the bending moment transferred from the slab to a support in the direction being considered

The analysis is carried out independently in directions  $x$  and  $y$ , as the decisive one the lower value of  $V_u$  is accepted.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It

takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = d - \sqrt{d^2 - \frac{2.M_d}{0,85.\phi.b.f'_c}}$$

Providing the location of neutral axis is less than the allowable one ( $x < k_u \cdot d$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{0,85.\phi.f'_c.b.x.\beta_1}{f_y}$$

Providing the location of neutral axis is greater than the allowable one ( $x > k_u \cdot d$ ), the program determines the areas of both compressive ( $A_{sc}$ ) and tensile ( $A_{st}$ ) reinforcement from the expressions:

$$A_{sc} = \frac{\frac{M}{\phi} - f_{c,max}(d - 0,45.k_u.d)}{f_y d \cdot Z}$$

$$A_{st} = \frac{M - A_{sc} \cdot f_y}{f_y}$$

$$F_{c,max} = 0,85.\phi.b.f'_c$$

where:

$$\phi = 0,8$$

$$k_u = 0,4$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} \leq \rho$$

where:

$$\rho = \frac{A_s}{b.d}$$

$$\rho_{min} = \left[ 0,22 \cdot \left( \frac{D}{d} \right)^2 \cdot \frac{f'_{cf}}{f_{sy}} \right] \leq \rho$$

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $V_{uc}$  and the maximum allowable shear force  $V_{u,max}$ .

$$V_{uc} = \beta_1.d \cdot \left( \frac{A_s \cdot f'_c}{d} \right)^{\frac{1}{3}}$$

where:

$$\beta_1 = \text{Max} \left[ 1,1 ; \left( 1,6 - \frac{d}{1000} \right) \cdot 1,1 \right]$$

$$V_{u,max} = 2 \cdot f'_c \cdot d$$

As for stirrups the necessary reinforcement area is given by:

$$A_{sv} = \frac{V^* - \phi \cdot V_{uc}}{\phi \cdot f_{sy,f} \cdot d}$$

As for hooks the necessary reinforcement area is given by:

$$A_{sv} = \frac{V^* - \phi \cdot V_{uc}}{\phi \cdot f_{sy,f} \cdot (\sin \alpha + \cos \alpha) \cdot d}$$

where:

$$\phi = 0,7$$

## SNiP 52-101-2003

This help contains the following computational methods:

- [Materials, coefficients, notation](#)
- [Verification of rectangular cross-sections made from plain concrete](#)
- [Verification of rectangular RC cross-section under M, V](#)
- [Verification of rectangular RC cross-section under N, M, V](#)
- [Verification of circular RC cross-section](#)
- [Verification of spread footing for punching shear](#)
- [Design of longitudinal reinforcement for slabs](#)
- [Design of shear reinforcement for slabs](#)

## Materials, Coefficients, Notation

The following notation for material parameters is used:

$R_{bd}$  - design strength of concrete in compression

$R_{bt}$  - design strength of concrete in tension

$R_{sc}$  - design strength of steel in compression

$R_s$  - design strength of steel in tension

The most common notation for geometrical parameters:

$b$  - cross-section width

$h$  - cross-section depth

$h_e$  - effective depth of cross-section

$z$  - lever arm (arm of internal forces)

## Verification of Rectangular Cross-Sections Made from Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M$ , normal force  $N$  (applied in the cross-section centroid) and by the shear force  $Q$ . The cross-section bearing capacity subjected to bending moment is given by:

$$M_{ult} = \frac{b \cdot h^2}{6} \cdot R_{bt}$$

The shear strength is provided by:

$$Q_{ult} = 1,5 \cdot b \cdot h \cdot R_b$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity  $e$ :

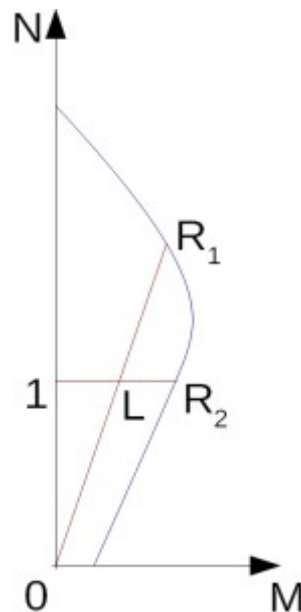
for:

$$N_{ult} = b \cdot x \cdot R_b$$

$$N_{ult} = \text{Min} \left( \frac{b \cdot h \cdot R_{bt}}{\frac{6 \cdot e}{h} - 1} ; \frac{b \cdot h \cdot R_b}{\frac{6 \cdot e}{h} + 1} \right)$$

$$x_u = h - 2 \cdot e$$

$$e = \frac{\text{abs}(M)}{N}$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|0L|/|0R_1|$  or  $|1L|/|1R_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed excentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under M, V

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M$ . The ultimate moment is provided by:

$$M_{ult} = b x R_b \left( h_0 - \frac{x}{2} \right)$$

$$x = \frac{A_s R_s}{b R_b}$$

The program further checks whether the location of neutral axis  $x$  is less than the limit location of neutral axis  $x_R$  given by:

$$x_R = \frac{0,8 h_0}{1 + \frac{R_s}{700}}$$

The computed degree of reinforcement is checked using the following expressions:

$$\mu_{st,min} = 0,001 < \mu_{st}$$

$$\mu_{st} = \frac{A_s}{b h_0}$$

### Shear

First, the program computes the ultimate shear strength of concrete  $Q_b$ .

$$Q_b = 2,5 R_{bt} h_0 b$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $Q_{max}$  is checked.

$$Q_{max} = 0,3 R_b h_0 b$$

Next, the necessary reinforcement area is given by:

$$A_{sw} = \frac{Q - 1,5 R_{bt} h_0 b}{0,75 R_{sw} h_0}$$

where:

$$R_{sw} = \text{Min}(0,8 R_s ; 300 \text{ MPa})$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,002 to 0,0035. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

$$\mu_{st,min} = 0,001 < \mu_{st}$$

$$\mu_{st} = \frac{A_s}{b h_0}$$

### Shear

First, the program computes the ultimate shear strength of concrete  $Q_b$ .

$$Q_b = 2,5 R_{bt} h_0 b$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $Q_{max}$  is checked.

$$Q_{max} = 0,3 R_b h_0 b$$

Next, the necessary reinforcement area is given by:

$$A_{sw} = \frac{Q - 1,5 R_{bt} h_0 b}{0,75 R_{sw} h_0}$$

where:

$$R_{sw} = \text{Min}(0,8 R_s ; 300 \text{ MPa})$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,0015 - 0,0035. The degree of reinforcement is checked using the formula:

- **Column** - check for dominant compression

$$\mu_{st,min} = 0,001 < \mu_{st}$$

$$\mu_{st} = \frac{4 \cdot A_s}{\pi \cdot d^2}$$

- **Beam** - check for dominant bending

$$\mu_{st,min} = 0,001 < \mu_{st}$$

$$\mu_{st} = 0,5 \frac{4 A_s}{\pi d^2}$$

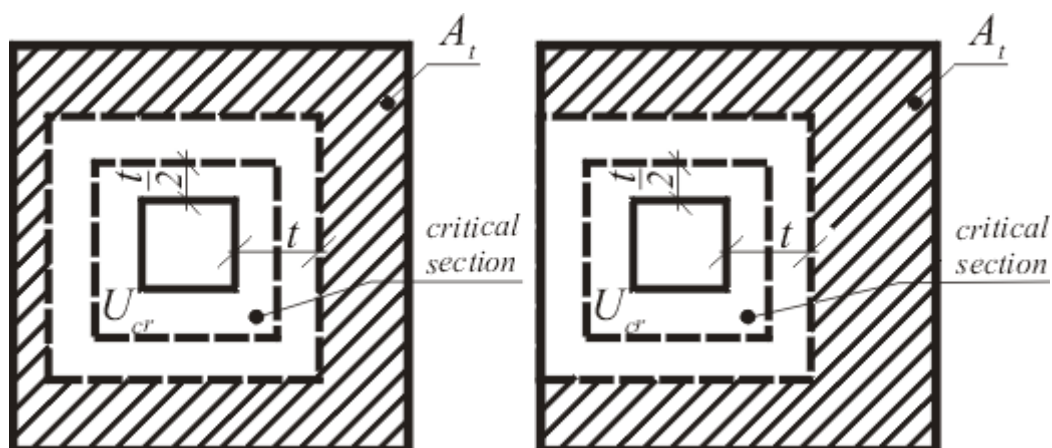
where:  $d$  - pile diameter  
 $A_s$  - reinforcement area

## Verification of Spread Footing for Punching Shear

The program allows to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear  $U_{cr}$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x, M_y$  and by the shear force  $F$  provided by:

$$F = \frac{Q \cdot A_t}{A}$$

where:  $A$  - area of footing  
 $Q$  - assigned vertical force developed in column  
 $A_t$  - hatched area in fig.



Dimensioning of shear reinforcement area  $A_t$

The program computes the maximal shear force  $F$  developed in the critical section, the shear force transmitted by concrete with no shear reinforcement  $F_{b,ult}$ , and the maximal allowable force  $F_{ult,max}$ :

$$F_{b,ult} = R_{bt} \cdot h_0$$

$$F_{ult,max} = 2 \cdot F_{b,ult}$$

For  $F < F_{b,ult}$  no shear reinforcement is needed.

For  $F > F_{b,ult}$  and  $F < F_{ult,max}$  the shear reinforcement must be introduced. The ultimate shear force is given by:

$$F_{ult} = F_{b,ult} + F_{sw,ult}$$

$$F_{sw,ult} = \frac{0,8 \cdot A_s \cdot R_{sw} \cdot \sin \alpha}{V_{cr}}$$

where:  $V_{cr}$  - critical cross-section span  
 $\alpha$  - is angle of crooks

$A_s$  - overall area of crooks in footing

For  $F > F_{ult,max}$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$x = h_0 - \sqrt{h_0^2 - \frac{M}{0,5 \cdot b \cdot R_b}}$$

Providing the location of neutral axis is less than the allowable one ( $x < x_{max}$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{b \cdot x \cdot R_{bd}}{R_{sd}}$$

Providing the location of neutral axis is greater than the allowable one ( $x > x_{max}$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{M - F_{c,max} (h_e - 0,5 \cdot x_{max})}{R_{sd} \cdot z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} \cdot R_{sd}}{R_{sd}}$$

$$F_{c,max} = x_{max} \cdot b \cdot R_{bd}$$

The limit location of neutral axis is found from:

$$x_{max} = 0,533 \cdot h_e$$

The computed degree of reinforcement is checked using the following expressions:

$$\mu_{st,min} = 0,001 < \mu_{st}$$

$$\mu_{st} = \frac{A_s}{b h_0}$$

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $Q_b$  and the maximum allowable shear force  $Q_{max}$ .



$$Q_b = 2,5 R_{bt} h_0$$

$$Q_{\max} = 0,3 R_b h_0$$

As for stirrups the necessary reinforcement area is given by:

$$A_{sw} = \frac{Q - 1,5 R_{bt} h_0}{0,75 R_{sw} h_0}$$

As for hooks the necessary reinforcement area is given by:

$$A_{sw} = \frac{Q - 1,5 R_{bt} h_0}{0,75 R_{sw} h_0 \sin \alpha}$$

where:

$$R_{sw} = \text{Min}(0,8 R_s ; 300 \text{ MPa})$$

## GB 50010-2010

This help contains the following computational methods:

- Materials, coefficients, notation
- Verification of rectangular cross-sections made from plain concrete
- Verification of rectangular RC cross-section under M, V
- Verification of rectangular RC cross-section under N, M, V
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

## Materials, Coefficients, Notation

The following notation for material parameters is used:

- $f_c$  - design strength of concrete in compression  
 $f_t$  - design strength of concrete in tension  
 $f'_y$  - design strength of steel in compression  
 $f_y$  - design strength of steel in tension

The most common notation for geometrical parameters:

- $b$  - cross-section width  
 $h$  - cross-section depth  
 $h_0$  - effective depth of cross-section

## Verification of Rectangular Cross-Sections Made from Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M$ , normal force  $N$  (applied in the cross-section centroid) and by the shear force  $V$ . The cross-section bearing capacity subjected to bending moment is given by (Art. D.3):

$$M_u = \frac{bh^2}{6} \gamma f_{ct}$$

where (Art. 7.2.4, Art. D.2.2):

$$\gamma = \left( 0.7 + \frac{120}{h} \right) 1.55$$

$$400\text{mm} \leq h \leq 1600\text{mm}$$

$$f_{ct} = 0.55 f_t$$

The shear strength is provided by (Art. 6.3.3):

$$V_u = 0.7 \beta_h f_t b h$$

where:

$$\beta_h = \left( \frac{800}{h} \right)^{\frac{1}{4}}$$

$$800\text{mm} \leq h \leq 2000\text{mm}$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity  $e_0$  (Art. D.2.1):

As the greater of:

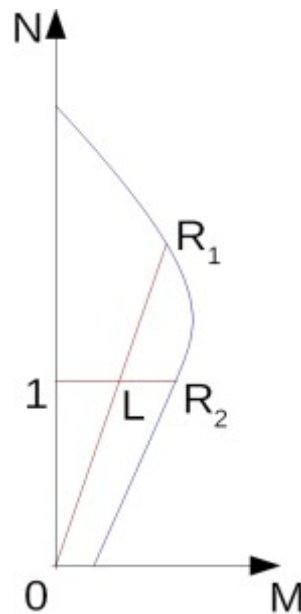
$$e_0 \leq 0.45h \Rightarrow N_u = f_{cc} b (h - 2e_0)$$

$$N_u = \text{Min} \left( \frac{bh\gamma f_{ct}}{6e_0 - h}; \frac{bh f_c}{6e_0 + h} \right)$$

where (Art. D.2.1):

$$f_{cc} = 0.85 f_c$$

$$e_0 = \left| \frac{M}{N} \right|$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|OL|/|OR_1|$  or  $|IL|/|IR_2|$ . Where  $L$  is load,  $R_1$  is strength with prescribed excentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under $M$ , $V$

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M$ . The ultimate moment is provided by (Art. 6.2.10):

$$M_u = \alpha_1 f_c b x \left( h_0 - \frac{x}{2} \right)$$

$$x = \frac{f_y A_s}{\alpha_1 f_c b}$$

$\alpha_1 = 1$  for:  $\leq$  **C50**

$\alpha_1 = 0.94$  for:  $\geq$  **C80**, intermediate values are obtained using linear interpolation method (Art. 6.2.6).

The program further checks whether the depth of compression zone  $x$  is less than the limit depth of compression zone  $\xi_b h_0$  given by (Art. 6.2.7):

$$\xi_b = \frac{\beta_1}{a + \frac{f_y}{E_s \varepsilon_{cu}}}$$

$\beta_1 = 0.8$  for:  $\leq$  **C50**

$\beta_1 = 0.74$  for:  $\geq$  **C80**, intermediate values are obtained using linear interpolation method (Art. 6.2.6).

The computed degree of reinforcement is checked using the following expressions (Art. 8.5.1):

$$\rho_{min} = \text{Max} \left( 0.002; 0.45 \frac{f_t}{f_y} \right) \leq \rho$$

$$\rho = \frac{A_s}{bd}$$

**Shear**

First, the program computes the ultimate shear strength of concrete  $V_c$  (Art. 6.6.3).

$$V_c = 0.7\beta_h f_t b h_0$$

where:

$$\beta_h = \left( \frac{800}{h_0} \right)^{\frac{1}{4}}$$

$$800\text{mm} \leq h_0 \leq 2000\text{mm}$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is checked (Art. 6.3.1).

for  $h_0/b \leq 4$

$$V_{max} = 0.25\beta_c f_c b h_0$$

for  $h_0/b \geq 6$

$$V_{max} = 0.2\beta_c f_c b h_0$$

intermediate values are obtained using linear interpolation method

$\beta_c = 1$  for:  $\leq$  **C50**

$\beta_c = 0.8$  for:  $\geq$  **C80**, intermediate values are obtained using linear interpolation method.

Next, the necessary reinforcement area is given by (Art. 6.3.4):

$$A_{sv} = \frac{V - 0.7 f_t b h_0}{f_{yv} h_0}$$

$$f_{yv} = \text{Min} (360\text{MPa}, f_y)$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation (Art. 6.2.1). The maximum allowable strain of concrete in compression is 0,002 to 0,0033. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions (Art. 8.5.1):

$$\rho_{min} = \text{Max} \left( 0.002; 0.45 \frac{f_t}{f_y} \right) \leq \rho$$

$$\rho = \frac{A_s}{bd}$$

## Shear

First, the program computes the ultimate shear strength of concrete  $V_c$  (Art. 6.6.3).

$$V_c = 0.7\beta_h f_t b h_0$$

where:

$$\beta_h = \left( \frac{800}{h_0} \right)^{\frac{1}{4}}$$

$$800\text{mm} \leq h_0 \leq 2000\text{mm}$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is checked (Art. 6.3.1).

for  $h_0/b \leq 4$

$$V_{max} = 0.25\beta_c f_c b h_0$$

for  $h_0/b \geq 6$

$$V_{max} = 0.2\beta_c f_c b h_0$$

intermediate values are obtained using linear interpolation method

$\beta_c = 1$  for:  $\leq \mathbf{C50}$

$\beta_c = 0.8$  for:  $\geq \mathbf{C80}$ , intermediate values are obtained using linear interpolation method.

Next, the necessary reinforcement area is given by (Art. 6.3.4):

$$A_{sv} = \frac{V - 0.7 f_t b h_0}{f_{yv} h_0}$$

$$f_{yv} = \text{Min} (360\text{MPa}, f_y)$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation (Art. 6.2.1). The maximum allowable strain of concrete in compression is  $0,002 - 0,0033$ . The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Column** - check for dominant compression (Art. 8.5.1, Art. 9.3.1)

$$\rho = \frac{4A_s}{\pi d^2}$$

for steel strength grade greater or equal to  $500\text{MPa}$

$$\rho_{min} = 0.005$$

for steel strength grade greater or equal to  $400\text{MPa}$

$$\rho_{min} = 0.0055$$

for steel strength grade less than  $335\text{MPa}$

$$\rho_{min} = 0.006$$

$\rho_{min}$  is increased by  $0.001$  for concrete strength grade greater than **C60**

$$\rho_{max} = 0.05$$

- **Beam** - check for dominant bending (Art. 8.5.1)

$$\rho = 0.5 \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \text{Max} \left( 0.002; 0.45 \frac{f_t}{f_y} \right)$$

where:  $d$  - pile diameter  
 $A_s$  - reinforcement area

### Shear

First, the program computes the ultimate shear strength of concrete  $V_c$  (Art. 6.3.3, Art. 6.3.15).

$$V_c = 0.7\beta_h f_t (0.88d)(0.8d)$$

where:

$$\beta_h = \left( \frac{800}{0.8d} \right)^{\frac{1}{4}}$$

$$800\text{mm} \leq 0.8d \leq 2000\text{mm}$$

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  (Art. 6.3.1, Art. 6.3.15) and strength of reinforced section  $V_s$  are checked (Art. 6.3.4, Art. 6.3.15).

for  $h_0/b \leq 4$

$$V_{max} = 0.25\beta_c f_c b h_0$$

for  $h_0/b \geq 6$

$$V_{max} = 0.2\beta_c f_c b h_0$$

$$V_{max} = 0.25\beta_c f_c (0.88d)(0.8d)$$

$\beta_c = 1$  for:  $\leq$  **C50**

$\beta_c = 0.8$  for:  $\geq$  **C80**, intermediate values are obtained using linear interpolation method.

$$V_{cs} = 0.7 f_t (0.88d)(0.8d) + f_{yv} \frac{A_{sv}}{s} (0.8d)$$

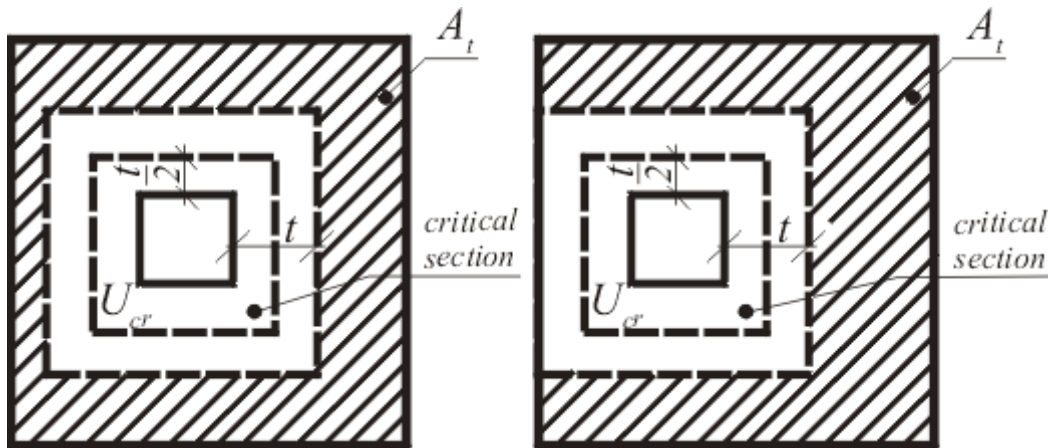
$$f_{yv} = \text{Min} (360\text{MPa}, f_y)$$

## Verification of Spread Footing for Punching Shear

The program allows to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear  $U_{cr}$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x$ ,  $M_y$  and by the shear force  $F_l$  provided by:

$$F_l = \frac{Q A_t}{A}$$

where:  $A$  - area of footing  
 $Q$  - assigned vertical force developed in column  
 $A_t$  - hatched area in fig.



Dimensioning of shear reinforcement area  $A_t$

The program computes the maximal shear force  $F_l$  developed in the critical section (the influence of unbalanced bending moment is added according to Appendix F of standard GB50010-2010), the shear force transmitted by concrete with no shear reinforcement  $F_c$  (Art. 6.5.1), and the maximal allowable force  $F_{max}$  (Art. 6.5.3). The shear forces are related to the unit length of critical section.

$$F_c = 0.7\beta_h f_t \eta h_0$$

$$\beta_h = 0.9 \text{ for } h \geq 2000\text{mm}$$

$\beta_h = 1$  for  $h \leq 800\text{mm}$ , intermediate values are obtained using linear interpolation method.

$$\eta = \text{Min}(\eta_1, \eta_2)$$

$$\eta_1 = 0.4 + \frac{1.2}{\beta_s}$$

where  $\beta_s$  is the size ratio of long side and short side of action area.

$$2 \leq \beta_s \leq 4$$

$$\eta_2 = 0.5 + \frac{\alpha_s h_0}{4u_m}$$

where  $\alpha_s$ : 40 - for interior column  
 30 - for edge column  
 20 - for corner column

$$F_{max} = 1.2 f_t \eta h_0$$

For  $F_l < F_c$  no shear reinforcement is needed.

For  $F_l > F_c$  and  $F_l < F_{max}$  the shear reinforcement must be introduced. The ultimate shear force is given by:

$$F_u = 0.5 f_t \eta h_0 + 0.8 f_{yv} A_{sbu} \sin \alpha$$

$$f_{yv} = \text{Min} (360\text{MPa}, f_y)$$

where:  $u_m$  - critical cross-section span  
 $\alpha$  - is angle of crooks  
 $A_{sb}$  - area of crooks in unit length of critical section  
 $u$

For  $F_l > F_{max}$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

Additional check according to article 8.2.9 of standard GB50007-2011 is done for narrow footing or strip footing.

$$V_c = 0.7\beta_h f_t b_0 h_0$$

where:

$$\beta_h = \left( \frac{800}{h_0} \right)^{\frac{1}{4}}$$

$$800\text{mm} \leq h_0 \leq 2000\text{mm}$$

$b_0$  is average width of footing.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the depth of compression zone as (Art. 6.2.10):

$$x = h_0 - \sqrt{h_0^2 - \frac{M}{0.5b\alpha_1 f_c}}$$

Providing the depth of compression zone is less than the allowable one ( $x < \xi_b h_0$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{\alpha_1 b x f_c}{f_y}$$

Providing the depth of compression zone is greater than the allowable one ( $x > \xi_b h_0$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{\frac{M}{h_0 - 0.5\xi_b h_0} - F_{c,max}}{f'_y}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} f'_y}{f_y}$$

$$F_{c,max} = \alpha_1 b \xi_b h_0 f_c$$



The limit depth of compression zone  $\xi_b h_0$  is found from (Art. 6.2.7):

$$\xi_b = \frac{\beta_1}{1 + \frac{f_y}{E_s \varepsilon_{cu}}}$$

The computed degree of reinforcement is checked using the following expressions (Art. 8.5.1):

$$\rho_{min} = \text{Max} \left( 0.0015; 0.45 \frac{f_t}{f_y} \right) \leq \rho$$

$$\rho = \frac{A_s}{bd}$$

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force transmitted by concrete  $V_c$  (Art. 6.3.3) and the maximum allowable shear force  $V_{max}$  (Art. 6.3.1).

$$V_c = 0.7 \beta_h f_t b h_0$$

where:

$$\beta_h = \left( \frac{800}{h_0} \right)^{\frac{1}{4}}$$

$$800\text{mm} \leq h_0 \leq 2000\text{mm}$$

for  $h_0/b \leq 4$

$$V_{max} = 0.25 \beta_c f_c b h_0$$

for  $h_0/b \geq 6$

$$V_{max} = 0.2 \beta_c f_c b h_0$$

intermediate values are obtained using linear interpolation method

$\beta_c = 1$  for:  $\leq \mathbf{C50}$

$\beta_c = 0.8$  for:  $\geq \mathbf{C80}$ , intermediate values are obtained using linear interpolation method.

As for stirrups the necessary reinforcement area is given by (Art. 6.3.4):

$$A_{sv} = \frac{V - 0.7 f_t b h_0}{f_{yv} h_0}$$

$$f_{yv} = \text{Min} (360\text{MPa}, f_y)$$

As for hooks the necessary reinforcement area is given by (Art. 6.3.5):

$$A_{sb} = \frac{V - 0.7 f_t b h_0}{0.8 f_{yv} h_0 \sin \alpha_s}$$

## NZS 3101-2006

This help contains the following computational methods:

- Materials, coefficients, notation
- Verification of rectangular cross-sections made from plain concrete
- Verification of rectangular RC cross-section under M, V
- Verification of rectangular RC cross-section under N, M, V
- Verification of circular RC cross-section
- Verification of spread footing for punching shear
- Design of longitudinal reinforcement for slabs
- Design of shear reinforcement for slabs

### Materials, Coefficients, Notation

The following notation for material parameters is used:

$f'_c$  - specified compressive strength of concrete

$E_c$  - modulus of elasticity of concrete at 28 days

$f'_y$  - lower characteristic yield strength of reinforcing steel

$$E_c = \left[ 3320 \sqrt{f'_c} + 6900 \right]$$

The characteristic compressive strength of concrete is the basic input parameter given by the class of concrete.

The most common notation for geometrical parameters:

$b$  - cross-section width

$h$  - cross-section depth

$d$  - effective depth of cross-section

$z$  - lever arm (arm of internal forces)

All computations are carried out according to the theory of limit states.

### Verification of Rectangular Cross-Sections Made from Plain Concrete

The cross-section is rectangular, loaded by the bending moment  $M^*$ , normal force  $N^*$  (applied in the cross-section centroid) and by the shear force  $V^*$ . The shear strength is provided by:

$$V^* \leq \phi V_n$$

where:

$$V_n = v_c \cdot b \cdot h$$

for cross-sections with height smaller than 200mm

$$v_c = 0.17\sqrt{f'_c}$$

for cross-sections with height greater than 400mm

$$v_c = 0.08\sqrt{f'_c}$$

intermediate values are obtained using linear interpolation method.

$f'_c$  is limited to value 50MPa.

$$\phi = 0.75$$

Strength of concrete cross-section subject to the combination of bending moment and normal force is derived from the following expressions depending on the normal force eccentricity  $e$ :

$$N^* \leq \phi N_n$$

Where  $N_n$  is determined as the greater of:

$$N_n = bx\alpha_1 f'_c$$

$$N_n = \text{Min} \left( \frac{bh0.36\sqrt{f'_c}}{\frac{6e}{h} - 1}; \frac{bhf'_c}{\frac{6e}{h} + 1} \right)$$

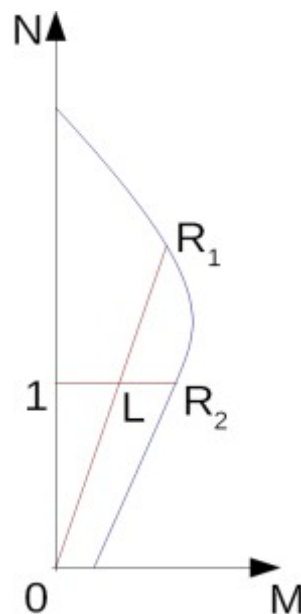
for  $f'_c < 55\text{MPa}$  is  $\alpha_1 = 0.85$

for concrete with greater strength is

$$\alpha_1 = \text{Max}(0.85 - 0.004(f'_c - 55); 0.75)$$

$$x = h - 2e$$

$$\phi = 0.6$$



Interaction diagram N-M

Usage ratio of concrete cross-section subject to the combination of bending moment and normal force is determined as  $|OL|/|OR_1|$  or  $|IL|/|IR_2|$ . Where  $L$  is load,  $R_1$  is strength with

prescribed excentricity and  $R_2$  is strength with prescribed normal force.

## Verification of Rectangular RC Cross-Section Under $M$ , $V$

The cross-section is rectangular, reinforced on one side and loaded by the bending moment  $M^*$ .

The ultimate moment is provided by:

$$M^* \leq \phi M_n$$

$$\phi = 0.85$$

$$M_n = \beta_1 c b \alpha_1 f'_c (d - 0.5 \beta_1 c)$$

$$c = \frac{A_s f_y}{\beta_1 b \alpha_1 f'_c}$$

for  $f'_c < 55 \text{MPa}$  is  $\alpha_1 = 0.85$

for concrete with greater strength is

$$\alpha_1 = \text{Max}(0.85 - 0.004(f'_c - 55); 0.75)$$

for  $f'_c < 30 \text{MPa}$  is  $\beta_1 = 0.85$

for concrete with greater strength is

$$\beta_1 = \text{Max}(0.65; 0.85 - 0.008(f'_c - 30))$$

The program further checks whether the location of neutral axis  $c$  is less than the limit location of neutral axis  $0.75c_b$  given by:

$$c_b = \frac{0.003}{0.003 + \frac{f_y}{E_s}}$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} = \frac{\text{Max}(0.25\sqrt{f'_c}; 1.4)}{f_y} < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

### Shear

$$V^* \leq \phi V_n$$

where:

$$\phi = 0.75$$

First, the program computes the ultimate shear strength of concrete  $V_c$ .

$$V_c = v_c b d$$

for cross-sections with height smaller than 200mm

$$v_c = 0.17\sqrt{f'_c}$$

$v_c$  is computed according to following formulas for cross-sections with height greater than 400mm, intermediate values are obtained using linear interpolation method.

$$v_c = k_d v_b$$

$$k_d = \text{Max}(0.9; \text{Min}(1; (400/d)^{0.25}))$$

$$v_b = \text{Min}(0.2; \text{Max}(0.08; 0.07 + 10\rho_w))\sqrt{f'_c}$$

where  $\rho_w$  is degree of reinforcement and  $f'_c$  is limited to value 50MPa.

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is checked.

$$V_{max} = \text{Min}(8\text{MPa}; 0.2f'_c)bd$$

Next, the necessary reinforcement area is given by:

$$A_v = \frac{V^* - \phi V_c b}{f_{yt} d}$$

## Verification of Rectangular RC Cross-Section Under N, M, V

The cross-section is rectangular, unilaterally reinforced and loaded by the bending moment and normal compression force. The program verifies a reinforced concrete section using the method of limit deformation. The maximum allowable strain of concrete in compression is 0,003. Compression reinforcement is not taken into account.

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} = \frac{\text{Max}(0.25\sqrt{f'_c}; 1.4)}{f_y} < \rho < 0.04 = \rho_{max}$$

$$\rho = \frac{A_s}{bd}$$

### Shear

$$V^* \leq \phi V_n$$

where:

$$\phi = 0.75$$

First, the program computes the ultimate shear strength of concrete  $V_c$ .

$$V_c = v_c b d$$

for cross-sections with height smaller than 200mm

$$v_c = 0.17\sqrt{f'_c}$$

$v_c$  is computed according to following formulas for cross-sections with height greater than 400mm, intermediate values are obtained using linear interpolation method.

$$v_c = k_n k_d v_b$$

$$k_n = 1 + \frac{N^*}{bh f'_c}$$

$$k_d = \text{Max}(0.9; \text{Min}(1; (400/d)^{0.25}))$$

$$v_b = \text{Min}(0.2; \text{Max}(0.08; 0.07 + 10\rho_w)) \sqrt{f'_c}$$

where  $\rho_w$  is degree of reinforcement and  $f'_c$  is limited to value  $50\text{MPa}$ .

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  is checked.

$$V_{max} = \text{Min}(8\text{MPa}; 0.2 f'_c) bd$$

Next, the necessary reinforcement area is given by:

$$A_v = \frac{V^* - \phi V_c b}{f_{yt} d}$$

## Verification of Circular RC Cross-Section

The program verifies a reinforced concrete pile using the method of limit deformation. The maximum allowable strain of concrete in compression is  $0,003$ .

The degree of reinforcement is checked using the formula:

$$\rho_{min} \leq \rho \leq \rho_{max}$$

- **Pile**

$$\rho = \frac{4A_s}{\pi d^2}$$

$$A_g < 0,5m^2$$

$$\rho_{min} = 2,4 / f_y$$

$$A_g > 2m^2$$

$$\rho_{min} = 1,2 / f_y$$

where:  $A_g$  - cross-section area of pile

intermediate values are calculated according to:

$$\rho_{min} = \frac{2.4}{f_y \sqrt{2A_g}}$$

$$\rho_{max} = 0.08$$

- **Column** - check for dominant compression

$$\rho = \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = 0.008$$

$$\rho_{max} = 0.08$$

- **Beam** - check for dominant bending

$$\rho = 0.5 \frac{4A_s}{\pi d^2}$$

$$\rho_{min} = \frac{\text{Max}(0.25\sqrt{f'_c}; 1.4)}{f_y}$$

$$\rho_{max} = 0.04$$

where:  $d$  - pile diameter  
 $A_s$  - cross sectional area of reinforcement

### Shear

$$V^* \leq \phi V_n$$

where:

$$\phi = 0.75$$

First, the program computes the ultimate shear strength of concrete  $V_c$ .

$$V_c = v_c b d$$

for cross-sections with height smaller than 200mm

$$v_c = 0.17\sqrt{f'_c}$$

$v_c$  is computed according to following formulas for cross-sections with height greater than 400mm, intermediate values are obtained using linear interpolation method.

$$v_c = k_n k_d v_b$$

$$k_n = 1 + \frac{N^*}{bh f'_c}$$

$$k_d = \text{Max}(0.9; \text{Min}(1; (400/d)^{0.25}))$$

$$v_b = \text{Min}(0.2; \text{Max}(0.08; 0.07 + 10\rho_w))\sqrt{f'_c}$$

where  $\rho_w$  is degree of reinforcement and  $f'_c$  is limited to value 50MPa.

If the ultimate shear strength of concrete is exceeded, the ultimate shear strength  $V_{max}$  and strength of reinforced section  $V_s$  are checked.

$$V_{max} = \text{Min}(8\text{MPa}; 0.2f'_c)bd$$

$$V_{cs} = \phi V_c + \phi A_v f_{yt}(0.8d)$$

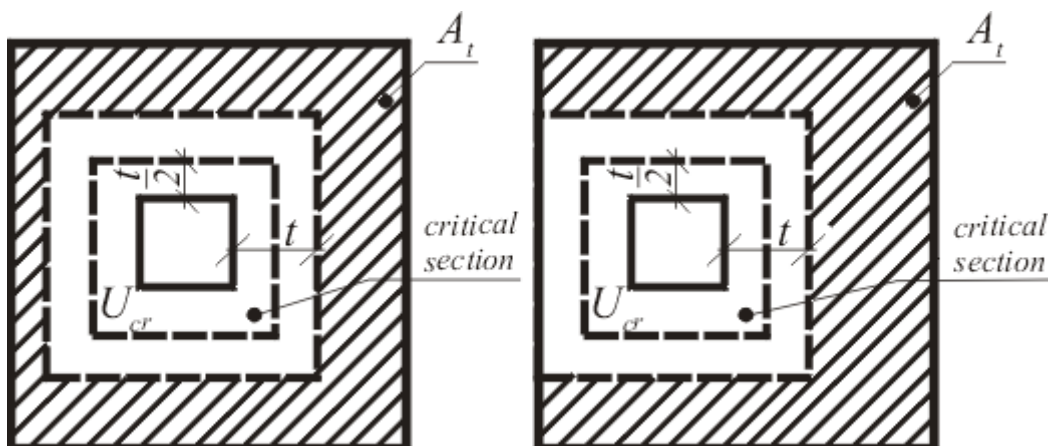
## Verification of Spread Footing for Punching Shear

The program allows to verify spread footing for punching shear or for the design of shear reinforcement. The critical section loaded in shear  $U_{cr}$  is distant from the column edge by one half of the footing thickness. It is loaded by the prescribed moments  $M_x^*$ ,  $M_y^*$  and by the shear force  $V^*$  provided by:

$$V^* = \frac{VA_t}{A}$$

where:  $A$  - area of footing  
 $V$  - assigned vertical force developed in column

$A_t$  - hatched area in fig.



Dimensioning of shear reinforcement area  $A_t$

The program computes the maximal shear force  $V^*$  developed in the critical section, the shear force transmitted by concrete with no shear reinforcement  $V_c$ , and the maximal allowable force  $V_{max}$ :

$$V_c = v_c b_o d$$

wher:

$$v_c = \frac{1}{6} k_{ds} \left( 1 + \text{Min} \left( 1; \frac{2}{\beta_c}; \frac{\alpha_s d}{b_o} \right) \right) \sqrt{f'_c}$$

- where  $\alpha_s$ :
- 20 - for interior column
  - 15 - for edge column
  - 10 - for corner column

$\beta_c$  is the ratio of the long side to the short side of the critical section

$$k_{ds} = \sqrt{\frac{0.2}{d}} \cdots \langle 0.5; 1 \rangle$$

$$V_{max} = 0.5 \sqrt{f'_c}$$

For  $V^* < \phi V_c$  no shear reinforcement is needed.

For  $V^* > \phi V_c$  and  $V^* < \phi V_{max}$  the shear reinforcement must be introduced. The ultimate shear force is given by:

$$V_n = \text{Min} \left( V_c; \frac{1}{6} \sqrt{f'_c} \right) + A_v f_{yv} \sin \alpha$$

- where:
- $b_o$  - critical cross-section span
  - $\alpha$  - is angle of crooks
  - $A_v$  - overall area of crooks in footing



For  $V^* > \phi V_{max}$  the shear reinforcement cannot be designed. It is therefore necessary to increase the cross-section height.

## Design of Longitudinal Reinforcement for Slabs

The design of reinforcement is performed for load caused by the bending moment  $M^*$ . The program provides the required area of tensile and compressive (if needed) reinforcement. It takes into account conditions for the minimum and maximum degree of reinforcement in a given cross-section. First, the program determines the location of neutral axis as:

$$c = \frac{d - \sqrt{d^2 - \frac{M^*}{0.5b\alpha_1 f'_c}}}{\beta_1}$$

Providing the location of neutral axis is less than the allowable one ( $c < 0.75c_b$ ), the program determines the area of tensile reinforcement  $A_{st}$  from the expression:

$$A_{st} = \frac{\beta_1 \alpha_1 b c f'_c}{f_y}$$

Providing the location of neutral axis is greater than the allowable one ( $c > 0.75c_b$ ), the program determines the areas of both compressive  $A_{sc}$  and tensile  $A_{st}$  reinforcement from the expressions:

$$A_{sc} = \frac{M^* - F_{c,max}(d - 0.5\beta_1 0.75c_b)}{f_y z}$$

$$A_{st} = \frac{F_{c,max} + A_{sc} f_y}{f_y}$$

$$F_{c,max} = \beta_1 \alpha_1 b 0.75c_b f'_c$$

The limit location of neutral axis is found from:

$$c_b = \frac{0.003}{0.003 + \frac{f_y}{E_s}}$$

The computed degree of reinforcement is checked using the following expressions:

$$\rho_{min} = \frac{\text{Max}(0.25\sqrt{f'_c}; 1.4)}{f_y} < \rho < 0.04 = \rho_{max}$$

If the maximum degree of total reinforcement  $\rho_{max}$  is exceeded, the program informs the user that the longitudinal reinforcement cannot be designed for a given cross-section.

## Design of Shear Reinforcement for Slabs

The program allows determination of the required amount of shear reinforcement form by stirrups and hooks, respectively.

First, the program computes the ultimate shear strength in a given section - the shear force

transmitted by concrete  $V_c$  and the maximum allowable shear force  $V_{max}$ .

$$V^* \leq \phi V_n$$

where:

$$\phi = 0.75$$

$$V_c = v_c b d$$

for cross-sections with height smaller than 200mm

$$v_c = 0.17 \sqrt{f'_c}$$

$v_c$  is computed according to following formulas for cross-sections with height greater than 400mm, intermediate values are obtained using linear interpolation method.

$$v_c = k_d v_b$$

$$k_d = \text{Max}(0.9; \text{Min}(1; (400/d)^{0.25}))$$

$$v_b = \text{Min}(0.2; \text{Max}(0.08; 0.07 + 10\rho_w)) \sqrt{f'_c}$$

where  $\rho_w$  is degree of reinforcement and  $f'_c$  is limited to value 50MPa.

$$V_{max} = \text{Min}(8\text{MPa}; 0.2 f'_c) b d$$

As for stirrups the necessary reinforcement area is given by:

$$A_v = \frac{V^* - \phi V_c}{f_{yt} d}$$

As for hooks the necessary reinforcement area is given by:

$$A_v = \frac{V^* - \phi V_c}{f_{yt} d \sin \alpha}$$

## Dimensioning of Steel Cross-Sections

Verification of steel cross-sections is carried out for two cases of load:

1. for the maximum value of bending moment and the corresponding shear force ( $M_{max} + Q$ )
2. for the maximum shear force and the corresponding bending moment ( $Q_{max} + M$ )

In both cases, the load enters to the assessment with an influence of normal force, which is defined separately. Its value is identical for both of load cases. Internal forces are, prior to analysis, pre-multiplied by the reduction coefficient of bearing capacity. This coefficient represents the degree of uncertainty of the determination of theoretical values of internal forces and as thus introduces into the analysis with such values certain reliability. The value of this coefficient is determined solely by the user.

The "**Sheeting Check**" program exploits for the dimensioning of steel cross-sections the following types of analyses:

- Verification according to EN 1993-1-1 (EC 3)
- Verification according to CSN 73 1401
- Verification according to safety factor

- Verification according to limit states
- Verification according to GB 50017-2003

Each cross-section is checked for free types of load:

### 1. Check for bending moment and normal force

The analysis checks the normal stress  $\sigma$  developed at the edge of cross-section given by:

$$\sigma = \frac{M}{W} + \frac{N}{A}$$

where:  $M$  - bending moment  
 $W$  - elastic modulus of cross-section  
 $N$  - normal force  
 $A$  - area of cross-section

### 2. Check for shear

The analysis checks the shear stress  $\tau$  at the cross-section center of gravity written as:

$$\tau = \frac{QS}{It}$$

where:  $Q$  - shear force  
 $S$  - 1st moment of area  
 $I$  - moment of inertia  
 $t$  - width (thickness) of cross-section at its center of gravity

### 3. Check for state of plane stress for the combination of stresses $\sigma_I$ and $\tau_I$ at the point of critical load

Equivalent stress for plane stress conditions is defined as:

$$\sigma_k = \sqrt{\sigma_1^2 + 3\tau_1^2}$$

All verifications are carried out assuming elastic response of the material, plasticity is not taken into consideration.

#### Verification of steel I-profiles of sheeting

Internal forces provided by the "Sheeting Check" program are considered per 1 m run of the structure width. Therefore, the units of shear force  $Q$  are  $kN/m$  and of bending moments  $M$  are  $kNm/m$ . For dimensioning of individual I-profiles these forces are, prior to verification analysis, automatically pre-multiplied by their spacing  $a$  [m] to get their values at the cross-section center of gravity, i.e. shear force  $Q$  in  $kN$  and bending moment  $M$  in  $kNm$ . The normal stress  $\sigma$  is checked at the outer face of flange. The shear stress  $\tau$  is checked at the center of gravity, thus at the center of web height. The equivalent stress  $\sigma_k$  is checked in the web at the flange-web connection (cut 1).

#### Verification of pile sheet wall

The verification analysis is carried out for a wall section of a unit length. All cross-sectional parameters are therefore determined not for individual sheet piles, but for a wall section of a unit length. The normal stress  $\sigma$  is checked at the outer face of back of sheet piles. The shear stress  $\tau$  is checked at the web center of gravity, thus for sheet piles of **U** shape at the location

of locks and for sheet piles of **Z** shape at the center of inclined sheet pile webs. The equivalent stress  $\sigma_k$  is checked in the sheet pile web at the location of connection of back of sheet piles (cut 1).

## Verification According to EN 1993-1-1 (EC3)

### Check for bending and stress caused by normal force

The bearing capacity in bending is given by:

$$M_{c,Rd} = \frac{W f_y}{\gamma_{M0}}$$

where:  $W$  - elastic modulus of cross-section  
 $f_y$  - steel yield stress  
 $\gamma_{M0}$  - coefficient of cross-section bearing capacity

The bearing capacity of normal force is given by:

$$N_{c,Rd} = \frac{A f_y}{\gamma_{M0}}$$

where:  $A$  - area of cross-section  
 $f_y$  - steel yield stress  
 $\gamma_{M0}$  - coefficient of cross-section bearing capacity

The bearing capacity is checked according to

$$\frac{M}{M_{c,Rd}} + \frac{N}{N_{c,Rd}} \leq 1,0$$

and the value of utilization is provided by:

$$\left( \frac{M}{M_{c,Rd}} + \frac{N}{N_{c,Rd}} \right) 100\%$$

### Check for shear

The shear bearing capacity is given by:

$$V_{c,Rd} = \frac{I t}{S} \frac{f_y}{\sqrt{3} \gamma_{M0}}$$

where:  $I$  - moment of inertia  
 $t$  - section thickness at the center of gravity  
 $S$  - 1st moment of area  
 $f_y$  - steel yield stress  
 $\gamma_{M0}$  - coefficient of cross-section bearing capacity

The bearing capacity is checked according to:

$$\frac{Q}{V_{c,Rd}} \leq 1,0$$

and the value of utilization is provided by:

$$\frac{Q}{V_{c,Rd}} 100\%$$

### State of plane stress verification:

The state of plane stress is checked exploiting the following conditions:

$$\left( \frac{\sigma_l}{f_y / \gamma_{M0}} \right)^2 + 3 \left( \frac{\tau_l}{f_y / \gamma_{M0}} \right)^2 \leq 1,0$$

where:  $\sigma_l$  - normal stress

$\tau_l$  - shear stress

The value of utilization is provided by:

$$\sqrt{\left( \frac{\sigma_l}{f_y / \gamma_{M0}} \right)^2 + 3 \left( \frac{\tau_l}{f_y / \gamma_{M0}} \right)^2} 100\%$$

*Literature:*

*Eurocode 3: Design of steel structures - Part 1-1: General rules and rules for buildings.*

## Verification According to CSN 731401

The CSN 73 1401 standard (from year 1998) adopts as the material parameter the steel design strength  $R_d$ . If this value is not determined for the used steel directly, it is back calculated from the steel yield stress as:

$$R_d = R_y / \gamma_m$$

where:  $\gamma_m$  - material coefficient taking value of 1.15 pro  $R_y \leq 300 \text{ MPa}$  and 1.25 for  $R_y > 300 \text{ MPa}$

### Check for bending

The normal stress  $\sigma$  is checked based on the following expression:

$$\sigma \leq R_d$$

and the value of utilization is provided by:

$$\frac{\sigma}{R_d} 100\%$$

### Check for shear

The shear stress  $\tau$  is checked based on the following expression:

$$\tau \leq 0,6R_d$$

and the value of utilization is provided by:

$$\frac{\tau}{0,6R_d} 100\%$$

### State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\sqrt{\sigma_1^2 + 3\tau_1^2} \leq 1,1R_d$$

where:  $\sigma_1$  - normal stress

$\tau_1$  - shear stress in the verified section

and the value of utilization is provided by:

$$\frac{\sqrt{\sigma_1^2 + 3\tau_1^2}}{1,1R_d} 100\%$$

*Literature:*

CSN 73 1401 (1998): *Design of steel structures.*

## Verification According to the Safety Factor

### Check for bending

The normal stress  $\sigma$  is checked based on the following expression:

$$\frac{f_y}{\sigma} \geq SF_s$$

where:  $f_y$  - steel yield stress

$SF_s$  - safety factor for bearing capacity of steel cross-section

and the value of utilization is provided by:

$$\frac{SF_s}{f_y/\sigma} 100\%$$

### Check for shear

The shear stress  $\tau$  is checked based on the following expression:

$$\frac{f_y}{\sqrt{3}\tau} \geq SF_s$$

and the value of utilization is provided by:

$$\frac{SF_s}{f_y/(\sqrt{3}\tau)} 100\%$$

### State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\frac{f_y}{\sqrt{\sigma_1^2 + 3\tau_1^2}} \geq SF_s$$

where:  $\sigma_l$  - normal stress  
 $\tau_l$  - shear stress in the verified section

and the value of utilization is provided by:

$$\frac{SF_s}{f_y / \sqrt{\sigma_1^2 + 3\tau_1^2}} 100\%$$

## Verification According to the Theory of Limit States

When performing the analysis according to the theory of limit states the steel yield stress  $f_y$  is reduced by the coefficient of material reliability  $\gamma_{ss}$ .

### Check for bending

The normal stress  $\sigma$  is checked based on the following expression:

$$\sigma \leq \frac{f_y}{\gamma_{ss}}$$

and the value of utilization is provided by:

$$\frac{\sigma}{f_y / \gamma_{ss}} 100\%$$

### Check for shear

The shear stress  $\tau$  is checked based on the following expression:

$$\tau \leq \frac{f_y}{\sqrt{3}\gamma_{ss}}$$

and the value of utilization is provided by:

$$\frac{\tau}{f_y / \sqrt{3}\gamma_{ss}} 100\%$$

### State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\sqrt{\sigma_1^2 + 3\tau_1^2} \leq \frac{f_y}{\gamma_{ss}}$$

where:  $\sigma_l$  - normal stress  
 $\tau_l$  - shear stress in the verified section

and the value of utilization is provided by:

$$\frac{\sqrt{\sigma_1^2 + 3\tau_1^2}}{f_y/\gamma_{ss}} 100\%$$

## Verification According to GB 50017-2003

The GB 50017-2003 standard adopts as the material parameter the steel design compressive, tension and bending strength  $f$  and shear strength  $f_v$ . If this value is not determined for the used steel directly, it is back calculated from the steel yield stress  $f_y$  as:

$$f = \frac{f_y}{\gamma_R}$$

$$f_v = \frac{f}{\sqrt{3}}$$

where:  $\gamma_R$  - resistance sub coefficient, which is 1,087 for  $f_y \leq 240 \text{ MPa}$  and 1,111 for  $f_y > 240 \text{ MPa}$

### Check for bending with influence of normal force

The bending stress with influence of normal force is checked according to this expression:

$$\frac{M}{\gamma_x W} + \frac{N}{A} \leq f$$

where:  $A$  - area of cross-section  
 $W$  - elastic modulus of cross-section  
 $\gamma_x$  - section plasticity develop factor  
 $f$  - design strength of steel

Section plasticity develop factor  $\gamma_x$  depends on shape of cross-section. For I-sections, sheet piles and casing is considered as  $\gamma_x = 1,05$ . The value of utilization is provided by:

$$\frac{\frac{M}{\gamma_x W} + \frac{N}{A}}{f} 100\%$$

### Check for shear

The shear stress  $\tau$  is checked based on the following expression:

$$\tau \leq f_v$$

and the value of utilization is provided by:

$$\frac{\tau}{f_v} 100\%$$

### State of plane stress verification

The state of plane stress is checked exploiting the following conditions:

$$\sqrt{\sigma^2 + 3\tau^2} \leq \beta_1 f$$

where:  $\sigma$  - normal stress



$\tau_l$  - shear stress in the verified section

$\beta_l$  - strength design value increase coefficient, which is 1,1

The value of utilization is provided by:

$$\frac{\sqrt{\sigma^2 + 3\tau^2}}{\beta_l f} 100\% /_o$$

## Index

3D Visualization.....	38
About the Company.....	851
ACI 318-11.....	1235
Activation.....	702
Active Dimensions and Objects.....	39
Active Earth Pressure.....	393
Add. 37, 45, 47, 49, 51-53, 69-72, 74, 75, 84, 85, 88, 106, 111, 114, 127, 151-153, 155, 158, 163, 165, 166, 168, 172, 174, 185, 188, 192, 193, 195, 199, 201, 209-212, 214, 218, 220, 230, 233, 234, 236, 240, 242, 251-255, 257, 261, 263, 271, 274, 276, 277, 279, 284, 286, 295, 297, 299, 300, 302, 305, 314, 315, 317, 319-321, 325, 326, 334, 336, 338, 339, 341, 344-346, 350-354, 366, 367, 369-371, 373, 376, 383-386, 394-397, 399, 401, 407, 408, 411, 413, 415-418, 429, 430, 432, 436, 437, 446, 447, 451, 453, 456, 458-460, 462, 466, 468, 480-482, 484, 486, 491, 498-500, 502, 503, 516, 517, 522, 523, 525, 528, 534, 537, 542, 545-547, 558-560, 562, 565, 567, 569, 574-582, 587, 589-591, 597-599, 601, 604, 605, 607-612, 617, 627-629, 633, 635, 638, 640, 642, 643, 646, 647, 652, 656, 657, 661, 663, 664, 666, 673, 675, 684, 686, 690, 694, 722, 723, 754-756, 758-760, 769, 771-774, 790, 798-805, 809, 811, 816, 818, 829, 844-846, 850, 851, 1004, 1011, 1088, 1092, 1140, 1221, 1305	
Administrator. 50, 113-116, 127, 128, 151, 163, 184, 209, 229, 251, 271, 295, 313, 334, 366, 394, 407, 429, 451, 480, 498, 516, 534, 558, 573, 633, 656	
Analysis. .62, 87-89, 91, 121-126, 131, 137-139, 141-143, 145-148, 150, 156-159, 161, 162, 170-172, 175, 178, 181, 183, 197-199, 202, 204, 206, 208, 216-218, 221, 223, 225, 228, 238-240, 243, 246, 248, 250, 259-261, 264, 266, 268, 270, 282-284, 287, 289, 292, 294, 303-306, 308, 309, 312, 323-325, 327-330, 333, 348-350, 355, 357-360, 365, 378, 381-383, 387, 389-393, 403, 404, 406, 421, 423-427, 440, 441, 448-450, 454, 463, 464, 466, 469, 471, 472, 474, 475, 479, 481, 483, 497, 501, 533, 555, 556, 572, 621, 622, 632, 645, 654, 655, 666-672, 678, 679, 702, 707, 712, 732, 811, 821, 830-832, 839, 847, 849, 851, 935, 956, 958, 959, 976, 981, 986, 991, 1001, 1008, 1022, 1087, 1095, 1115, 1116, 1118-1127, 1129, 1131, 1132, 1149, 1152, 1158, 1182, 1196, 1206, 1208, 1224, 1229, 1231, 1233, 1255	
Analytical Solution.....	497
Anchors.....	312, 319, 320, 333, 352, 353, 365, 385, 386, 406, 415, 416, 427, 437, 438, 447, 448, 702, 704, 785, 787, 788, 805, 806, 960, 1181
Anti-Slide Pile.....	118, 120, 149, 364, 365, 378, 406, 407, 417, 981, 1188, 1190, 1191
Application Window.....	38
Applied Forces.....	162, 183, 208, 228, 250, 269, 312, 333, 365, 449
AS 3600-2001.....	1235
Assign.....	68, 71, 87, 103, 107, 150, 155, 156, 162, 168, 169, 183, 195, 196, 208, 214, 215, 228, 236, 237, 250, 257, 258, 269, 280, 281, 293, 302, 303, 312, 317, 318, 332, 346, 347, 365, 379, 380, 393, 399, 400, 406, 413-415, 449, 462, 463, 478, 484, 485, 497, 502, 503, 515, 525, 526, 533, 547, 548, 557, 564, 565, 632, 633, 640, 641, 655, 663, 664, 670, 674, 675, 681, 685, 686, 702, 756, 780, 784, 785
Barton - Bandis.....	430, 1048
Base Anchorage.....	162, 228

---

Basic data...	153, 155, 166, 168, 193, 195, 212, 214, 234, 236, 255, 257, 277, 279, 300, 302, 315, 317, 339, 341, 371, 373, 397, 399, 411, 413, 460, 462, 482, 484, 500, 502, 523, 525, 545, 547, 560, 562, 638, 640, 663, 681-683, 687, 688, 692, 860, 861
Beam Loads.....	702
Beams.....	572, 590, 591, 702, 756, 785-787, 790-792, 799, 1131
Bearing Capacity.	162, 183, 184, 208, 228, 229, 250, 269, 270, 293, 449, 478, 479, 497, 515, 533, 1068, 1070
Bishop.....	406, 422, 476, 987, 1005, 1008, 1012, 1020-1022, 1061
Blocks.....	228, 433, 449, 1035
Bore Holes.....	681
Braced Sheeting.....	313
BS 8110:1997.....	1235
Buildings.....	670, 672
Combination SLS.....	572, 618, 632, 653
Combination ULS.....	572, 613, 632
Concentrated Surcharge - Earth Pressure at Rest.....	393
Consolidation.....	150, 702, 712-714, 1208, 1209
Construction....	45, 68, 112, 180, 206, 248, 291, 363, 392, 515, 526, 528, 681, 702, 758, 780, 951, 965, 1093, 1145, 1148, 1149, 1156
Contact Types.....	702
Contacts.....	702, 756, 757, 798-800
Control Menu.....	38, 855
Copy to Clipboard.....	39, 69
CPT.....	110, 125, 130, 131, 150, 515, 517-519, 521, 523, 524, 528, 1134-1139, 1144, 1149, 1152, 1155-1158, 1188
CSN 73 1002.	124, 126, 498, 501, 509, 553, 1087, 1095, 1096, 1114, 1128, 1129, 1159, 1160
CSN 73 6206.....	292, 1235, 1255, 1256
Damage.....	671, 677, 679, 680, 1233, 1234
Design Approaches.....	965
Dialog Windows.....	39
Dilatometric Tests (DMT).....	332, 365, 478
Dimensioning. . .	88, 140, 148, 162, 180-183, 206-208, 225, 226, 228, 248-250, 268-270, 272, 291-294, 310, 311, 333, 363-365, 392, 393, 403-406, 449, 472, 473, 478, 479, 486, 496, 497, 533, 556, 557, 572, 619, 620, 626, 627, 941, 956, 981, 1061, 1081, 1244, 1253, 1262, 1269, 1276, 1283, 1290, 1296, 1304, 1313
Distributions.....	357, 389, 495, 532, 572, 623, 628, 629
DL/T 5219 - 2005.....	1085
Drucker-Prager.....	712, 725, 729, 735, 829, 838, 847
DXF Import.....	69, 95-97, 100, 101

Earth Cut.....	406, 655
Earth Grading.....	681
Earth Pressure at Rest.....	393
Earth Pressures.....	43, 129, 131, 138, 150, 162, 183, 208, 228, 250, 270, 294, 313, 333, 365, 450
Earthquake...119-121, 148, 150, 159, 160, 162, 175, 176, 183, 202, 203, 208, 221, 222, 228, 243, 244, 250, 264, 265, 269, 287, 288, 293, 306, 307, 312, 327, 328, 333, 355, 356, 365, 387, 388, 406, 420, 421, 427, 438, 439, 449, 469, 470, 904, 905, 912, 995-997, 1059	
Edges.....	681, 690, 691
Edit 41, 43, 44, 53-55, 61, 68, 76-79, 114, 117, 165, 188, 191, 230, 232, 252, 253, 297, 319-321, 335, 336, 338, 341, 342, 344-346, 352-354, 368-370, 374, 376, 377, 386, 395, 429, 432-435, 453, 454, 456, 457, 459, 481, 505, 512, 517, 537, 541, 567, 569, 576, 593-595, 601, 612, 617, 619, 620, 635, 642, 643, 646, 647, 652, 675, 676, 686, 690, 694, 697, 699, 759, 792, 795, 796, 798, 852, 856, 943, 1095, 1110, 1113	
Embankment.....	62, 83, 406, 409, 410, 655, 658, 659, 1008
EN 1997-2.....	125, 524, 1134, 1139, 1140, 1146, 1147, 1151, 1155
Excavation.....	83, 129, 133, 151, 314, 329, 332, 334-336, 338, 345, 347, 348, 366, 368-370, 395, 973
External Stability.....	293, 333
Fellenius / Petterson.....	1020
Field Testing.....	557
File.....	41, 42, 94-97, 99, 108, 110, 111, 853, 854
Foliation.....	411
Footing. . .	82, 124, 130, 131, 133, 139, 150, 269, 274, 275, 478-480, 483, 485-487, 489, 494, 495, 940, 1067, 1077, 1078, 1081, 1082, 1188, 1191
Foundation. .	131, 162, 169, 170, 180, 183, 184, 196, 197, 205, 208, 215, 216, 224, 228, 237, 238, 247, 250, 258, 259, 267, 269, 270, 281, 282, 290, 478, 479, 482, 485-488, 911, 940, 965, 969, 1068, 1070, 1072, 1088-1091, 1095, 1115, 1116, 1118-1127, 1129, 1131, 1134, 1140, 1145, 1176, 1178, 1182, 1184, 1218, 1219, 1227
Frames.....	39, 47, 51, 84, 761
Free Area Loads.....	572
Free Line Loads.....	572
Free Lines.....	702
Free Point Loads.....	572
Free Points.....	702
Front Face .....	162, 183, 208, 228, 250, 269, 449
GB 50010-2010.....	148
Generate.....	93, 185, 189, 457, 458, 535, 538, 616, 617, 651, 652, 681, 689, 696, 828
Geometric Section.....	269
Geometry.....	47, 139, 150, 152, 162, 164, 165, 169, 176, 183, 185-190, 196, 208, 210, 211, 215, 225, 228, 231, 232, 237, 244, 250, 253, 254, 258, 269, 271-274, 281, 293, 295-298,

312, 318, 319, 332, 341, 342, 365, 373, 374, 393, 395, 396, 427, 441-443, 449, 451, 452, 478, 485, 487, 488, 491, 497, 504, 505, 515, 529, 530, 533, 540, 541, 557, 562, 563, 632, 635, 670, 675, 676, 943, 947, 973, 1036, 1132, 1134	
Geostatic Stress, Uplift Pressure.....	393
Global Stability.....	449
Graphs.....	702, 846, 1153
Ground Loss.....	150, 670, 671, 1224
GWT + NSF.....	515, 520, 527, 528
Heave Failure.....	333
Hoek - Brown.....	430, 1048
Horizontal Modulus.....	533
Import gINT.....	69, 111, 112
Import LandXML.....	69, 109
Incompressible Subsoil.....	478, 497, 655
Interface. .62, 68, 74, 75, 78-80, 406, 408, 409, 632, 633, 636, 637, 655, 657, 658, 702, 720, 721, 762, 923	
Internal Hinges.....	572
Internal Stability.....	293, 333, 449
IS 456.....	1235, 1274, 1275, 1278
Janbu 122, 406, 422, 987, 993, 1000, 1005, 1006, 1008, 1021, 1022, 1191, 1200-1203, 1212, 1219	
Joint Loads.....	572
Joint Refinements.....	572
Joint Supports.....	572
Joints.....	249, 572, 574, 575
Launching.....	681, 689, 697, 699-701
LCPC (Bustamante).....	125, 524, 1134, 1147, 1148
Line Flow.....	702
Line Loads.....	572
Line Refinement.....	572, 702
Line Supports.....	572, 702
Lines.....	572, 575, 576, 702
Lining.....	702, 759-761, 768, 769, 785, 786
Load. .95, 96, 110, 118, 124, 137-139, 141-143, 145-147, 161, 178, 181, 204, 206, 223, 225, 246, 248, 266, 268, 269, 271, 279, 280, 289, 292, 309, 393, 403, 404, 471, 472, 474, 478, 486, 487, 497, 503, 504, 532, 533, 542-544, 557, 565, 566, 572, 601-603, 614, 616, 631, 632, 643-648, 651, 702, 704, 761, 860, 942, 981, 982, 984, 985, 1081, 1088, 1097, 1099, 1108, 1111, 1131, 1155, 1158, 1165, 1166, 1220	
Location.....	39, 42, 632, 633, 637, 638, 757, 766, 767, 791, 799, 803, 1019

Macroelement Refinements.....	572
Macroelement Subsoils.....	572
Macroelements.....	572, 576, 577
Material 118, 148, 151, 162, 163, 165, 181, 183, 185, 190-192, 207, 210, 228, 230, 232, 233, 249, 250, 252, 253, 269, 271, 275, 276, 292, 293, 295, 298, 299, 311, 332, 334, 345, 365, 366, 377, 408, 430, 449, 451-453, 478, 480, 487, 490, 491, 496-498, 505, 506, 512, 533, 534, 541, 542, 556, 557, 563, 564, 574, 603, 604, 614, 616, 618-621, 623, 626-628, 633, 635, 645, 648, 651, 653, 654, 733, 744, 926, 932, 943-946, 951, 990, 1165, 1166, 1169-1171, 1236, 1247, 1256, 1264, 1271, 1279, 1286, 1292, 1298, 1307	
Measurement.....	671, 676, 677
Mesh Generation.....	572, 702
Micropile. .126, 130, 131, 150, 557, 559-561, 570, 1165-1167, 1169-1171, 1179, 1182, 1188-1190	
Modulus Kh.....	120, 332, 336, 365, 369
Modulus of Subsoil Reaction.....	497
Mohr - Coulomb.....	430, 1048
Monitors.....	702, 844, 845
Morgenstern-Price.....	406, 422, 987, 993, 1000, 1009, 1012, 1021, 1022
Mouse Context Menu.....	39
Mouse Functions.....	39
MSE Wall.....	119, 149, 449, 450, 473, 477
Nailed Slope.....	128, 131, 149, 293, 294, 310, 311, 948
NAVFAC DM 7.2.....	124, 126, 498, 501, 509, 553, 1087-1091, 1159
Negative Skin Friction.....	497, 533
NEN 6743.....	125, 524, 531, 1104, 1134, 1139-1141, 1146, 1147, 1149, 1151, 1155, 1158
NZS 3101-2006.....	1236
Openings.....	572, 577, 578
Options.....	37, 39, 40, 43, 44, 60, 65-68, 682, 856, 857, 933
Outputs.....	41, 48, 60, 150, 162, 184, 209, 229, 251, 270, 294, 313, 333, 365, 394, 407, 428, 450, 479, 497, 516, 533, 558, 573, 632, 655, 671, 681, 702, 850, 853
Page Numbering.....	850
Page Properties.....	850
Parameters of Rocks.....	407, 428
Passive Earth Pressure.....	874, 882
Pile 94, 110, 118, 120, 125, 127, 130, 131, 133, 139, 149, 150, 179, 180, 205, 224, 247, 267, 290, 364, 365, 378, 406, 407, 417, 496-498, 500, 501, 503, 504, 509, 511, 512, 515, 517, 519, 523, 524, 533, 534, 543-546, 553, 909, 981, 1087, 1088, 1091, 1093, 1095-1098, 1104, 1115, 1116, 1118-1127, 1131-1134, 1139, 1140, 1144-1146, 1148, 1149, 1151, 1153, 1155-1158, 1162, 1163, 1188, 1190, 1191, 1243, 1311	
Plane Slip Surface.....	427

Plasticity.....	830, 838
PN-B-03264:2002.....	1235
Point Flow.....	702
Point Refinement.....	702
Point Supports.....	702
Points.....	109, 189, 537, 681, 686, 687, 702
Polygonal Slip Surface.....	427
Pressiometric Tests.....	332, 365
Pressure Determination.....	312, 332, 365, 378
Print and Export Document.....	47, 850
Print and Export Picture.....	47, 850
Profile....	47, 150, 153, 162, 165, 166, 169, 183, 192, 193, 196, 208, 211, 212, 215, 228, 233, 234, 237, 250, 254, 255, 258, 269, 276, 277, 281, 293, 299, 300, 312, 314, 315, 332, 334, 335, 342, 365, 367, 368, 374, 393, 396, 397, 449, 459, 460, 478, 480, 481, 489, 497, 499, 515, 517, 522, 523, 533, 545, 557, 559, 560, 670, 673
Program FEM.....	702
Project....	47, 60, 150, 151, 162, 163, 183, 184, 208, 209, 228, 229, 250, 251, 269, 270, 293, 294, 312, 313, 332, 333, 365, 366, 393, 394, 406, 407, 427, 428, 449, 450, 478, 479, 497, 498, 515, 516, 533, 534, 557, 558, 572, 573, 632, 655, 656, 670, 671, 681, 682, 702, 705, 706, 726-728, 734, 735, 858, 860
Props.....	312, 320, 321, 333, 353, 354, 702, 809-811
Reinforcement.....	249, 406, 416, 417, 449, 454, 457, 458, 474-476, 626, 627, 702, 811, 812, 930, 991, 1061, 1062
Results.....	91, 555, 624-626, 654, 840, 841, 969
Rigid Bodies.....	702
Rigid Body.....	406
Rock....	119, 130, 131, 149, 365, 378, 379, 407, 427, 428, 430, 431, 438, 673, 921, 924, 930, 951, 1026, 1028, 1035, 1041, 1049-1051, 1053, 1056, 1057, 1059, 1072-1074, 1181
Root Verification.....	557
Sand-Gravel Cushion.....	478
Sarma.....	406, 422, 987, 999-1001, 1003, 1019, 1020
Schmertmann.....	125, 517, 885, 1134, 1142, 1143, 1148, 1149, 1152
Setting a Color Range.....	69
Settings.	38, 47, 49-51, 65, 72, 91, 92, 113-116, 127-129, 131, 136, 137, 144, 148, 150-152, 162-164, 183-185, 208-210, 228-230, 250-252, 269-271, 293-295, 312-314, 332-336, 338, 345, 365-370, 393-395, 406-408, 427-429, 440, 448-451, 478-481, 483, 485, 494, 495, 497-499, 501, 507, 509, 515-517, 520, 527, 528, 533-535, 545, 546, 553, 557-559, 572-574, 632, 633, 635, 655-657, 667, 668, 670-675, 678, 702, 706, 707, 711, 712, 716, 722, 759, 761, 764, 765, 785, 821, 840, 844, 850, 851, 860, 926, 939, 962, 1025, 1042, 1047, 1082, 1127, 1132, 1156, 1158, 1163, 1164, 1208, 1231, 1256
Settlement....	45, 49, 65, 68, 94, 112, 122, 150, 478-483, 495, 497, 511, 512, 515, 531-533, 545, 554, 555, 638, 655, 657, 661, 666-668, 678, 699, 716, 842, 1110, 1113, 1162, 1225

---

Shaft.....	149, 393, 981, 985, 1111, 1185
Shahunyants.....	422, 987, 1000, 1012, 1013, 1021
Sheeting Check.....	112, 118, 120, 129, 131, 133, 138, 149, 312, 332, 333, 365, 1188, 1191, 1315, 1316
Sheeting Design.....	129, 131, 138, 149, 312, 313, 332
Slip on Georeinforcement.....	208, 449
Slope Stability.....	130, 131, 149, 364, 406, 407, 412, 450, 948, 1008
SNiP 52-101-2003.....	1235
Soil Body.....	150, 162, 183, 208, 228, 250, 270, 294, 313, 333, 365, 407, 450, 479, 497, 515, 533, 655, 671
Soil Classification.....	515, 517, 1137
Soils.66-68,	128, 150, 153, 154, 162, 166, 167, 169, 173, 183, 193, 194, 196, 200, 208, 212, 213, 215, 219, 228, 234, 235, 237, 241, 250, 255, 256, 258, 262, 269, 277, 278, 281, 285, 293, 300, 301, 312, 315, 316, 332, 335, 336, 339, 340, 347, 365, 368, 371, 372, 393, 397, 398, 406, 411, 412, 449, 452, 460, 461, 467, 478, 482, 483, 489, 497, 500, 501, 515, 523, 524, 533, 545, 546, 557, 560, 561, 632, 633, 638, 639, 655, 661, 662, 670, 673, 674, 681, 684, 685, 702, 721, 722, 969, 989, 1023, 1134, 1149, 1168, 1176, 1177, 1184, 1208
Spencer.....	406, 422, 476, 987, 993, 1000, 1003, 1005, 1009, 1010, 1021, 1022, 1061
Spring Method.....	497, 533
Stability.....	122, 130, 131, 149, 162, 182, 183, 207, 208, 227, 228, 249, 250, 269, 293, 295, 308, 312, 330, 331, 333, 364, 406-408, 412, 422, 427-429, 449-451, 476-478, 702, 707, 834, 847, 948, 999, 1003, 1008, 1022, 1047, 1048, 1050, 1227
Stability Analysis.....	1008
Stage settings.....	131, 140, 144, 149, 160, 177, 178, 181, 203, 222, 245, 246, 248, 265, 288, 289, 292, 307, 328, 356, 388, 403, 421, 439, 470, 493, 508, 552, 666, 677, 1233
Structure	87, 162, 184, 229, 270, 294, 333, 365, 479, 497, 527, 533, 535-540, 544, 555, 986, 1053, 1088-1091
Subsoil.....	62, 478, 497, 572, 632, 633, 635, 636, 655, 921
Supports.....	312, 321, 322, 333, 354, 355, 365, 386, 387, 572, 632, 642, 643, 702, 801, 960
Surcharge.....	87, 140, 150, 158, 159, 162, 172, 173, 183, 199, 200, 208, 218, 219, 228, 240, 241, 250, 261, 262, 269, 272, 284, 285, 293, 305, 306, 312, 325, 326, 333, 350, 351, 365, 383, 384, 393, 401, 402, 406, 418, 419, 427, 436, 437, 446, 447, 449, 466, 467, 478, 491, 492, 655, 664, 665, 670, 702, 703, 816, 817, 865, 1036
Table Data Import.....	69
Tables.....	39, 873, 874, 881, 882
Template.....	41, 44, 98
Terrain..	108, 110-112, 150, 153, 156, 157, 162, 166, 170, 171, 183, 192, 197, 198, 208, 211, 216, 217, 228, 233, 238, 239, 250, 254, 259, 260, 269, 276, 282, 283, 293, 299, 303, 304, 312, 314, 323, 324, 332, 335, 348, 349, 365, 367, 381, 382, 396, 427, 429, 430, 449, 459, 463, 464, 480, 499, 522, 545, 559, 673, 681, 684, 697, 699-701, 907
Theory.	150, 162, 183, 208, 228, 250, 270, 294, 312, 333, 365, 393, 407, 428, 450, 479, 497, 515, 533, 557, 573, 655, 671
Tomlinson.....	124, 498, 501, 509, 1087, 1093, 1143, 1148, 1149

---



---

Topology.....	702, 703, 761, 769, 780, 785
Tunnels.....	84, 711, 712, 794
Types of Nails.....	293
Values...555, 572, 623, 627, 628, 737, 841, 913, 917, 920, 921, 923, 970, 1005, 1008, 1012, 1054, 1074, 1093, 1112, 1148, 1157, 1168, 1177, 1178, 1180, 1219, 1221	
Verification..68, 87, 88, 113, 119-121, 123-126, 129, 132, 139, 140, 161, 162, 178, 179, 181, 183, 203, 204, 206, 208, 222, 223, 225, 228, 246-248, 250, 265, 266, 268, 269, 279, 289-293, 308, 309, 403, 422, 449, 451, 452, 471-474, 494, 557, 570, 571, 679, 926, 935-939, 941, 948, 999, 1047, 1048, 1061, 1063, 1082-1087, 1096, 1134, 1155, 1157, 1165, 1166, 1172, 1234, 1236, 1247, 1256, 1264, 1265, 1271, 1279, 1286, 1292, 1298, 1307, 1315, 1316	
Vertical Springs.....	533
Visualization...38, 41, 46-51, 54, 72, 84, 90-92, 161, 178, 180, 181, 204, 205, 207, 210, 223-226, 246, 247, 249, 253, 266-268, 289, 290, 292, 309-311, 329, 357, 389, 392, 403, 405, 422, 426, 427, 440, 448, 471, 472, 474-477, 494-496, 509-514, 530, 531, 556, 570, 571, 621, 623, 625, 626, 668, 678, 679, 682, 687, 691, 692, 694, 696, 697, 699, 759, 820, 821, 840, 841, 851	
Water 49, 52, 62, 150, 157, 158, 162, 171, 172, 183, 198, 199, 208, 217, 218, 228, 239, 240, 250, 260, 261, 269, 283, 284, 293, 304, 305, 312, 324, 325, 332, 349, 350, 365, 382, 383, 393, 400, 401, 406, 419, 420, 427, 435, 436, 445, 446, 449, 464-466, 478, 492, 493, 497, 506, 507, 533, 548, 549, 557, 566, 567, 632, 633, 641, 642, 655, 665, 666, 681, 691, 692, 702, 712, 720, 780, 781, 799, 822, 824, 889, 912, 913, 916, 917, 1033, 1038	
Wings.....	269, 272, 273, 291

© Fine Ltd. 2016  
[www.finesoftware.eu](http://www.finesoftware.eu)