Finite Element Method (FEM) – Introduction

The objective of this engineering manual is to explain the basic terms of a particular field of problems and the practical application of the GEO 5 – FEM program in solving geotechnical problems.

GEO 5 – FEM program allows for the modelling of various types of problems and analyses. The text below explains some basic terms and general procedures in a more detailed way – individual analysis modules are described in other chapters.

By **problem type** the GEO 5 – FEM program distinguishes two basic cases:

− **a planar problem**: this analysis module is used in solving linear structures (a tunnel, embankment, open cut, dam, etc.), whose longitudinal dimension is by an order of magnitude larger than the lateral dimensions of the area being solved.

In such cases, it is proper to relate the analysis to $1\ rm$ of the structure and solve the problem *under the assumption of a plane-strain deformation*. Deformations in planes parallel to the longitudinal axis of the structure can be disregarded in this case. Thus, the development of deformations and stresses in the massif is taken into consideration only in the plane perpendicular to the longitudinal axis and, as a result of lateral contraction, also the normal stress in the longitudinal direction. Regarding beam elements, it is the case of solving a 1m wide plate strip (for more details visit Help – F1).

− **axial symmetry**: this analysis module is suitable for solving rotationally symmetrical problems. This assumption must be satisfied by both the geometrical arrangement of the structure and the loading. A suitable example is the analysis of a vertically loaded single pile, circular excavation or pumping groundwater from a circular borehole.

Similar to the case of the plane-strain deformation, this is the case of a generally three-dimensional problem, which, however, can be again converted into the planar problem solution. The solution is then related to $1\ rad$ of the $x\,(r)$-radius arch. The axis of symmetry always represents the $x\,(r)$ coordinate origin. Shear components of deformation in the direction of rotation can be disregarded. The evolution of a circumferential normal component of stress and deformation is also taken into consideration, in addition to the components of stress and deformation in the cross-sectional plane (for more details visit Help – F1).

From the **type of analysis** point of view, the program allows the following cases to be solved using individual modules:

− **Stress**: it serves to solve basic geotechnical problems in ground environment and a rock mass (e.g. for determining the vertical or horizontal geostatic stress, pore pressure, deformations, volumetric changes, and sub-grade deformations, as well as analyzing internal forces along a diaphragm wall structure length (height) etc.).

− **Steady flow**: it assumes a zero change in the saturation degree with time; individual construction stages are completely independent of each other (in contrast with the unsteady flow).

− **Unsteady flow**: this analysis module allows for the determination of the evolution of pore pressures (the total head) and the current degree of saturation with time possible. In this case, the analysis methodology is similar to that of the stress analysis.

− **Slope stability**: during the analysis, this program reduces the input values of the internal friction angle φ_{ef} or the soil cohesion c_{ef} and seeks the onset of failure associated with the evolution of the critical region of localized plastic deformation. The result is the factor of safety corresponding to classical slope stability analysis methods. The definition and input of a model in this regime is completely identical with that of the "Stress" module.

− **Tunnels**: this module allows for performing the analysis of an underground excavation (modelling of the 3D effect of excavation attributed to the New Austrian Tunnelling Method), accounting for the degradation of beams, temperature-induced loads acting on beams, swelling-induced loads acting within specified regions and monitoring the results.

− **Consolidation:** this module is used to perform time dependent settlement analysis under foundations, embankments and surface loads (surcharges). The coupled analysis provides the evolution of displacements, stresses, strains, plastic zones, pore pressure distribution, ground water table and water flow through the model for given external load and the mechanical and hydraulic boundary conditions.

"Settings" frame

⊿GEO5

The GEO 5 – FEM program incorporates the possibility of a so-called advanced specification, where it is possible to define complementary input parameters of soils for individual material models in more detail, to develop mixed meshes with multi-nodal elements and to visualize larger numbers of output variables.

Note: The standard setting assumes drained boundary conditions. In such cases, the analysis assumes steady-state conditions, where the skeleton deformation has no influence on the development of pore pressures. The pressures only have the character of external loads and do not vary during the course of the analysis. In the case of undrained conditions, where the entire boundary of the particular area behaves as being fully impermeable, we the opposite is solved, i.e. the fully coupled problem of the skeleton deformations and pore pressures under the assumption that all changes are immediate and the influence of time does not take effect.

The method of determining the initial stress (1st construction stage) is assumed in the program as:

- **geostatic stress**: The standard method for the analysis of the vertical geostatic stress is based on the following relationship:

$$\sigma_z = \sum_{i=1}^{n} \gamma_i \cdot h_i \; [kPa]$$

where: γ_i – bulk density of soil in the ith layer,

h_i – thickness of ith layer.

- K_0 **procedure**: It is used in cases, where the user needs to define another *initial lateral stress*. For example, the actual lateral stress in over-consolidated soils can be significantly higher than that in normally consolidated soils (for more details visit Help – F1). The lateral pressure coefficient K_0 is set as a soil parameter. If this parameter is not specified, it is provided by:

$$K_0 = \frac{\nu}{1-\nu}$$

where: ν – Poisson's ratio.

The input parameters of soils also depend on the selected material model for stress or flow analysis. The most important input parameters for stress analysis comprise the modulus of elasticity E and Poisson's ratio ν (to be specified for all models), and for non-linear models, the angle of internal friction φ_{ef} and the soil cohesion c_{ef}. The steady-state condition after the redistribution of pore pressures is assumed in the program and the *effective parameters* of shear strength of soils or rocks are therefore used during the analysis (for more details visit Help – F1).

The selection of the material model and subsequent setting of soil parameters belong among the most important and at the same time most problematic tasks when FEM modelling of a structure is being carried out. Material models try to plausibly describe the behaviour of soil or rock. They can be divided into two basic groups, namely **linear** and **non-linear** models.

Note: The correct selection of the material model is absolutely necessary for giving a true picture of the real behaviour of the structure. Non-linear models are necessary for the majority of structures (e.g. the analysis of a sheeting wall structure using a linear model of soil will give completely unrealistic results), but the application of linear models can in many cases be appropriate, simplifying the entire analysis. A simplified procedure recommended for the modelling of problems using the FEM is presented in Help – F1).

Linear models provide a relatively fast but not very accurate assessment of the real material behaviour. They can be used in cases, where we are only interested in the stress or deformation of the groundmass and not in the area and mode of the potential failure. They can also be used in cases, where only a local failure develops, having no fundamental influence on the development of global failure, but which may result in premature termination of the analysis in the program (for more details visit Help – F1).

The group of linear models comprises:

- **elastic model**: It uses conversion relationships between the stress and deformation given by Hooke's law (within the linear elasticity scope).

- **modified elastic model**: It makes incorporating the influence of surcharge loading or unloading into the analysis possible, using the secant modulus E_{def} and the unloading/reloading modulus E_{ur}.

Although, if we try to obtain a plausible description of the groundmass behaviour or if we are interested in the distribution of areas of potential failures, it is necessary to adopt non-linear models. Basic non-linear models can be again divided into two groups. The first group of models is based on the classical Coulomb failure condition. It comprises the **Drucker-Prager**, **Mohr-Coulomb** and **Modified Mohr-Coulomb models**. Using these models, it is also possible to model the hardening or softening of soils. A common feature of these models lies in the unlimited elastic deformation under the assumption of geostatic stress (for more details visit Help – F1).

The second group of material models, which are based on the notion of the critical state of the soil, is represented by the **Modified Cam-clay, Generalized Cam-clay** and **Hypoplastic clay models**. These models provide a significantly better picture of the non-linear response of soil to external loading. Individual material models differ not only in their parameters but also in the assumptions made.

The boundary between the linear (elastic) and non-linear (plastic) response of the material is formed by the surface of plasticity. The mathematical expression of the surface of plasticity represents a certain failure condition (plasticity function). Exceeding this condition leads to the development of permanent (irreversible) plastic strains.

*Note: Apart from the basic material parameters adopted for linear models, the non-linear models require the introduction of certain strength-related characteristics of soils, which are necessary for the formulation of yield conditions. **The onset of the evolution of a plastic strain** depends on the value of the angle of internal friction φ and cohesion c. The dilatancy angle ψ controls the magnitude of the plastic volumetric strain (for more details visit Help – F1).*

The selection of a material model suitable for the analysis of geotechnical structures adheres first of all to the character of the soil/rock environment. In the Finite Element Method-based process of comprehensive modelling of more complex problems, the selection of the numerical model represents an absolutely essential influence on specifying the input data and assessing the analysis results.

Working with interface, model dimensions

The description of the work with the specification of individual interfaces is presented in the program help in a more detailed way (see F1). The input data essential for numerical analyses using the FEM are the so-called *world coordinates* (specifying the magnitude of the area being solved), where it is necessary, especially for stability analyses, to secure a sufficient neighborhood (the interface width) to obtain meaningful results.

Note: The depth of the FE mesh is also very important. It is possible to visualise the end of the mesh as an incompressible sub-grade. If there is no incompressible sub-grade specified for the particular geological profile, it is possible to assume that the internal forces will disappear at a certain distance from the loading spot or from the contact of the structure with the sub-grade; no deformation will therefore develop. The world boundaries of the problem being solved are subsequently defined to be at this distance (for more details visit Help – F1).

The interface can be imported from other programs of the GEO 5 system using the clipboard. The program further allows the import and export of interfaces in *DXF format and the import of interfaces in gINT format. To simplify the specification of interface points (geometry) it is possible to use the so-called interface corrector (for more details visit Help – F1).

Mesh generation

The successful generation of a mesh is the last step in the process of setting the *topology of the structure* (interfaces between soil layers, lines of structures, parameters of soils and rocks, contacts, etc.). Individual construction stages are modelled and analyzed subsequently. When the mesh is being generated, the program also automatically generates the *standard boundary conditions*. The standard setting of the boundary conditions contains:

- smooth pin at the mesh nodes found along the horizontal bottom edge,

- sliding pin at the mesh nodes along the left-hand and right-hand vertical edges.

The GEO 5 – FEM program has a built-in *automatic corrector of the specified geometry*. This means that the program itself finds intersections of lines and all closed areas and develops an adequate model before the generation of the finite element mesh (for more details visit Help – F1).

The newly developed areas can be subsequently removed from the analysis or new soils can be assigned to them. The user will become aware of the main advantages of this system particularly when analyzing tunnels and braced structures. Specifying even a very complicated structure becomes a very simple and quick process (for more details visit Help – F1).

The correctly generated finite element mesh is the basic condition for obtaining results regarding the real behaviour of the structure. GEO 5 – FEM program has a built-in automatic mesh generator, which significantly simplifies this task. Despite this fact it is necessary to adhere to certain rules:

– The denser the mesh the more accurate the results. On the other hand, problem analysis becomes significantly slower. The objective, therefore, is to find the optimal density of the mesh. It depends both on the experience of the user and the finite element used.

– The mesh of elements should be sufficiently dense, especially in an area where large stress gradients can be expected (spot support, sharp corners, underground excavations, etc.). It is necessary to perform mesh refinement around individual points or lines. The density-refining radius has to be at least 3 to 5 times larger than the density in the density refining center and both values at the points (the density and radius) have to be in a reasonable proportion to the mesh density prescribed for the surrounding area. This way a smooth transition between the areas with different density will be secured.

Note: Singular lines must be dealt with similarly. In the cases of more complicated problems it is advisable to carry out a preliminary analysis of a course finite element mesh and on the basis of the analysis of the generated results, to carry out a local refinement of the mesh density (for more details visit Help – F1).

The program uses six-node triangular elements with automatic mesh smoothing by default (for more details visit Help – F1).

Construction stage

When the structure topology specification and the finite element mesh generation are finished, the analyses are then carried out for individual construction stages.

The construction stages correspond to the gradual building of the construction, and the correct specification and sequence are very important. The analysis of each stage (with the exception of stability analyses) is based on the **results of the preceding stage.** Information about individual construction objects and their properties is maintained between the construction stages. The rule of property inheritance is applied between the construction stages when a stage is being edited or specified (for more details visit Help – F1).

The first construction stage (**the analysis of the primary geostatic stress**) represents the initial state of the massif before the construction begins. For that reason, the analysis result refers to the stress in soil or rock mass, not to the deformations.

Note: A principal problem of the FEM analysis usually lies in the non-convergence of some of the construction stages. If the results of one phase are not available, the analysis of the subsequent stage is impossible. As for the correct modelling, the program authors recommend following a recommended procedure for the modelling of a construction process (for more details visit Help – F1).

Analysis settings and description

During the course of the analysis, the program tries to find (iterate) a solution at which the equilibrium conditions in the massif are met for the specified boundary conditions. The iteration process and the analysis convergence can be followed on-screen (for more details visit Help – F1).

The analysis can be interrupted at any time; in such a case the results are available for the last successful convergence of the loading increment. Correct results can be obtained when 100 percent is reached. Nevertheless, it may happen that the analysis only goes up to a certain percentage of the loading, which means that the program has not succeeded in finding the correct solution and the analysis does not converge (for more details visit Help – F1). In that case, it is possible to change the parameter settings of the standard analysis or to carry out some modifications of the model:

- increase the structure stiffness,

- reduce the applied loads,

- reduce the soil excavation area,

- improve soil characteristics,

- change the material model of soils in plasticity locations,

- add beam reinforcing elements or tensional elements,

- add boundary conditions,

- change the course of the iteration in the analysis settings (e.g. increasing the number of iterations).

Note: The visualization of plastic strains showing critical locations with the anticipated development of failure surfaces can advise you why the analysis does not converge (for more details visit Help – F1).

The program provides a default set of analysis parameters ensuring sufficient accuracy and speed. However, an experienced user may want to change certain parameters or test their influence on the analysis accuracy (for more details visit Help – F1).

Note: The authors of the program recommend that any changes in the setting of analysis parameters should be approached very carefully, after thoroughly studying the problems. Incorrectly selected settings may cause not only an incorrect iteration of the solution and a significantly slower analysis but also inaccurate results (for more details visit Help – F1).

It is possible to change the following parameters in the "Settings" window in the analysis stage:

- **the solving method (and its settings)**:   Newton-Rapshon method – NRM,

  Arc-length method – ALM.

- **the stiffness matrix**: the initial stress method; the full or modified NRM.

- **the initial computation step**: the ratio between the load in the particular loading step and the prescribed overall load (the standard-setting value is 25 percent of the overall load).

- **the maximum number of iterations**: for achieving equilibrium within the framework of the given loading increment.

- **the convergence criterion**: setting the error tolerance (changes in the vector standard) for displacement, unbalanced forces, and internal energy.

- **the line search method**: determination of the weight coefficient η for meeting the equilibrium condition, resulting in the acceleration or dampening of the computation process.

- **plasticity**: setting the tolerance for the error in return to the yield surface, expressing the accuracy required to meet the yield condition.

Note: Individual analysis settings, including basic equations for meeting the equilibrium conditions or yield conditions, are described in more detail in the program Help (for more details visit Help – F1).

Visualization and interpretation of results

The visualisation and interpretation of results is one of the most important parts of the program. GEO 5 – FEM program allows several basic types of graphical outputs:

– drawing of the deformed structure

– planar representation of quantities in the groundmass (it is possible to represent absolute values or values compared with another construction stage)

– internal forces in beams and on contacts

– forces in anchors and reactions

– settlement trough

– tilted cross-sections or vectors and directions of the quantities

Note: The program uses certain coordinate systems for the presentation of results. All outputs and selected results can be printed in the analysis protocol (for more details visit Help – F1).

Some results cannot be drawn concurrently because of the necessity for transparency and comprehensibility. It is, for example, impossible to draw a deformed structure and, at the same time, the distribution of internal forces along a beam. It is always necessary to choose only one option. The program gives a warning at the bottom of the dialogue window if inadmissible combinations of outputs are set.

The program makes the setting of an arbitrary quantity of point monitors and line monitors possible at any place of the structure or outside the structure. **Monitors** have several functions:

– visualizing the values of quantities at a particular point (the point monitor),

– visualizing the values of the difference at a distance between two points compared with the previous construction stage $d\left[N\right]$, or compared with the setting phase where N is the construction stage number (*the line monitor*).

List of chapters relating to the FEM

– *Chapter 20*:   Finite Element Method (FEM) – Introduction.

– *Chapter 21*:   Terrain settlement analysis.

– *Chapter 22*:   Settlement of a circular silo foundation.

GEO5