

3D structure

Introduction

This tutorial shows the input and verification of the structure shown in the picture below. Columns are made of RHS profiles, beams are *I*-profiles. Material class *Fe 360* is used. Beams are loaded by linear load *18 kN/m*. Two columns are loaded by trapezoidal load (*12kN/m* at the top and *19kN/m* at the bottom). Supports are rigid. The programs "**Fin 3D**" and "**Steel**" will be used.



The main window appears after running the program "Fin 3D".

EN EC - FIN 3D (Unitided f3)	e - [View]									-	•
File Edit Entry Tools Opt	tions Help										
1 🖷 🖷 👘 👘 Krai	× • • •										
Project information Generate	· · ivia	-	Z Active is	ad case MA	. and	-	o ano [4]		- <u>* %</u> / E		E.
+ Add joints	Y YI W Y	E A A George			a onsec		0,000 (#)	81			
Add members	Q (2)										
2 Information	0 9										
O Tools	0										
- G+ Rotate - ++ Mirror	+										
-X Add scissor joint -% Convert joint to abs.	1	Z									
O Load cases O Load - I' Add sont load	50 L										
Add member load) 1本			Y.							
O Combinations O Dynamics	ы ця	×									
Calculation	12 12:										
	4ª cát	General inform Header: Job name	untited		Input Topology: Structure size		: 0,000 m	x 0,000	Analysis Ist order analysis 2nd order analysis	: not ca : do no	ekulated t calculate
		Part Description Client Author			Joints Members		m x 0,00 i 0 (abs 0, hidden i 0, 0 hidd	nel (1), 0	Dynamics - eigenn	: do na iades : do na	t calculate t calculate
ves		Date Project ID Archive ID Comment	29. 4. 2016		Load: Standard Service dam 5 Load cases	5	: EN 1990 : 2 : 0 (swt 0,	Default for 0,			
Add picture unb 0		File Size Created Saved	Untitled.f3e [n/w] [n/w]		Joint load Member load		<pre>card, te i 0 (swt 0, clef 0, te i 0 (swt 0, clef 0, te</pre>	m (1) for (1, m (1) for (1, m (1)			
and a start and			C.L. C.L						*		

Main application window

Basic part of structure

The structure will be created with the help of manual input of joints and members. This input style shows most of particular procedures which appear during the work with the software. The input will be done using following steps:

- •Input of a first frame (two columns and a beam)
- Input of load
- •Copy of the existing frame



• Input of transverse beams and their load

This procedure is not the easiest way, however, it shows variety of functions and tools included in the software. First, we save the project. The name (for example "**3D project**") appears in the heading of the window and is also copied into designing modules.

We select the mode "Add members".



This mode launches the window "**Member prototype**". This window contains properties (cross-section, material, end conditions) that will be assigned to new members. We will continue with the help of the button "**Profile**".

Member prototy	pe		×
- Member profile	Edit section	Edit material	
Material: no inpu	it		
- Member			
End conditions:	Fixed -	Fixed 💌	Load from structure
- Beginning (local	coordinate sys.):	— — End (local coor	dinate sys.):
Displ.:	Rot.:	Displ.:	Rot.:
1: 🔽 Fixed	1: 🔽 Fixed	1: 🔽 Fixed	1: 🔽 Fixed
2: 🔽 Fixed	2: 🔽 Fixed	2: 🔽 Fixed	2: 🔽 Fixed
3: 🔽 Fixed	3: 🔽 Fixed	3: 🔽 Fixed	3: 🔽 Fixed
Hinge 2,3	Fixed	Hinge 2,3	Fixed
Special			OK Cancel

The button for input of cross-section and material

The window "**Profile prototype**" which contains options for input of cross-section and material. We will use the button "**Steel**".



rofile prototype		×
Section	- Section type:	
	Steel	Timber
	Concrete	Masonry
	User defined	Editor
	Edit i	nput
	- Section rotation: $\alpha = $,00 [°]
Material		
Catalogue		
User defined		
Load from structure	C OK	X Cancel

The button for input of steel cross-section

The window "**Steel section**" that appears after the clicking on the button contains an option to select type of crosssection. We select database of rolled cross-sections (the option "**Solid hot-rolled**") and open the window "**Profiles catalogue**" by pressing the button "**OK**".

Steel section	×
Section types:	
Solid hot-rolled Solid welded Solid Built-up hot-rolled Built-up rolled Built-up welded General section	R
• ОК	X Cancel

Choice of cross-section type

We select the section type "Seamless tube circular cross-section" (RHS) in the first column of the database and the item "TK 168x16" in the second column. The choice of cross-section has to be confirmed by the button "OK".





Database of cross-sections

After the confirmation a window with material grades appears. We select the strength class "EN 10025: Fe 360".

Materials catalogue - Structural steel		×
Structural steel EN Stainless steel EN	EN 10025 : Fe 360 EN 10025 : Fe 430 EN 10025 : Fe 510 prEN 10113 : Fe E 275 prEN 10113 : Fe E 355 EN 10210-1 : S 235 EN 10210-1 : S 275 EN 10210-1 : S 355	
Information	Cance	al

Choice of strength grade

The properties entered in the window "Profile prototype" are displayed in the following figure.



Profile prototype			×
Section			
		Section type:	· · · · · · · · · · · · · · · · · · ·
	Å.	Steel	Timber
		Concrete	Masonry
		User defined	Editor
	Y	Edit in	nput
Structural st A = 7,64E+03	eel TK 168 x 16 mm ² P = 955,0 m	m	
$I_{\gamma} = 22,3E+06$	mm^4 $I_z = 22,3E+0$	06 mm ⁴	
Material			
Catalogue	EN 10025 : Fe 360 E = 210.0E+03 MPa	G = 81.00E+0	3 MPa
User defined	$\alpha_t = 12,00E-06 \ 1/K$	γ = 78,50 kN/n	1 ³
Load from strue	cture	C OK	Cancel

Profile prototype with specified cross-section

After the confirmation of the window **"Prototype profile**" by the button **"OK**", the prototype properties will be docked in the bottom frame. The input of members in the workspace is enabled now. First, we will insert a column. We specify the start of global coordinate system as a member beginning. The cursor snaps to this points automatically in close surrounding. The snapping is also indicated by the change of cursor appearance.



The cursor appearance when snapping to beginning of coordinate system

We show the orientation of the column by the cursor in the next step (the orientation should be in the direction of the axis "**Z**"). The alignment into this direction is automatically offered by the program (the program automatically snaps into 45° directions as a default).





Member orientation given by the position of the cursor

Paralelly, we specify the length of member (column height) 2.35 using keyboard. The lengths have to be specified in metres. The length entered on the keyboard is automatically inserted into the input field " Δr " in the bottom part of the tree menu. The input has to be confirmed by the button "**Enter**".



Input of member length

The input of beam follows. The beginning is identical to the end of the column, the end will be specified with the help of cursor (direction parallel to the axis "**Y**") and the length *4m*. We will finish the input of the length by the button "**Enter**".



We will continue with the input of the second column. The input will be done from the end of the beam against the direction of the axis "**Z**".





Frame made of two columns and a beam

We terminate the input of members by a right mouse button click. After that, we change the display settings to improve the visibility of input. The window "**Drawing settings**" can be opened with the help of the context menu, which can be launched by a right mouse button click everywhere in the workspace.



Drawing settings in the context menu

The window "**Drawing settings**" contains parameters, which affect the displayed objects in the workspace. We check settings "**Beginnings**" and "**Sections**". First setting highlights beginnings of members by black arrows, second shows member masses. The window has to be closed by the button "**OK**".



			×
Structure topology Struct	ture load		
- Common			
Global coordinate sys	stems	Draw grid	
- Joints	n	Members	
Numbering	Г	Numbering	
Symbols	Γ	Local axes	
Supports	F	End conditions	
Coordinate systems	; F	 Beginnings 	
Common Load, internal forces a Show units for load, in	and deformation Iternal force and	only on selected me deformation values	mbers
Common Load, internal forces a Show units for load, in Correction of display size	and deformation Internal force and	only on selected me d deformation values	mbers
Common Load, internal forces a Show units for load, in Correction of display size for load, int. forces and schematic deformations:	and deformation Internal force and smallest	only on selected me d deformation values default	mbers Iargest
Common Load, internal forces a Show units for load, in Correction of display size for load, int. forces and schematic deformations: Text size:	and deformation Internal force and smallest	only on selected me deformation values default	mbers largest
Common Load, internal forces a Show units for load, in Correction of display size for load, int. forces and schematic deformations: Text size:	and deformation iternal force and smallest smallest	only on selected me deformation values default default	mbers largest largest
Common Load, internal forces a Show units for load, in Correction of display size for load, int. forces and schematic deformations: Text size: Size of support symbols:	and deformation iternal force and smallest smallest	only on selected me deformation values default default	mbers largest largest
Common Load, internal forces a Show units for load, in Correction of display size for load, int. forces and schematic deformations: Text size: Size of support symbols:	and deformation iternal force and smallest smallest smallest	only on selected me deformation values default default default default	mbers largest largest largest

Changes in the window "Drawing settings"

The workspace shows now, that both columns and beam have identical cross-section (RHS). Therefore, next step is to change the cross-section of the beam. We open the context menu for the beam by right button mouse click on this member. We select the option "**Edit member**" in this menu.



The context menu for the beam

This action opens the window "**Properties of member**". We change the cross-section using method already described above. The button "**Profile**" launches the window "**Edit profile**". We select the option "**Steel**" in this window.



Properties of memi	dit profile		×
Geometry	Section		
Beginning:		Section type:	
Member profile	×	Steel T	imber
Profile		Concrete Ma	asonry
Section: TK 168 x A = 7,64E+03 mm ²		User defined E	ditor
Material: EN 100		Edit input	
- Member Ind conditions:		α = 0,00	[°]
Displ.: R	Structural steel TK 168 x 16		
1: ▼ Fixed 1 2: ▼ Fixed 2	$A = 7,042+03 \text{ mm}^2$ $P = 955,0 \text{ m}^2$ $I_y = 22,3E+06 \text{ mm}^4$ $I_z = 22,3E+06 \text{ mm}^4$	-06 mm ⁴	
3: 🔽 Fixed 3	Material		
Hinge 2,3	Catalogue EN 10025 : Fe 360 E = 210,0E+03 MPa	G = 81,00E+03 MPa	
Special Nu	User defined $\alpha_t = 12,00E-06 \ 1/K$	$\gamma = 78,50 \text{ kN/m}^3$	
> 2 2	Load from structure	🕑 ОК	🔀 Cancel
3 3			

The change of beam cross-section

The cross-section type "**Solid hot-rolled**" should be selected for opening the database of rolled cross-sections. The beam should have the cross-section "I(IPN) 200".

Profiles catalogue		×
Profiles catalogue Profile class Cross-section I(IPN) Cross-section IE Cross-section IPE Cross-section HE Cross-section HE Cross-section HE Cross-section HE Cross-section HE Cross-section HE Cross-section HD Cross-section UB Cross-section UC	Profile I(IPN) 80 I(IPN) 100 I(IPN) 120 I(IPN) 140 I(IPN) 160 I(IPN) 180 I(IPN) 200 I(IPN) 220 I(IPN) 220 I(IPN) 240 I(IPN) 260	
Cross-section J Cross-section UBP Cross-section S Cross-section HP American Standard Source ArcelorMittal, Ferona	I(IPN) 280 I(IPN) 300 I(IPN) 320 I(IPN) 340 25-1, CSN 42 5550	7,5 90,0 90,0 C OK X Cancel

Cross-section of beam

After finishing the changes of beam's cross-section, we will continue by modifying the second column. We open the window **"Properties of member 3"** for the right column and change the orientation of the member (the order of start and



end joints). The orientation is not significant for an analysis of internal forces, however, may be critical for a correct verification in the designing module when using unsymmetrical design properties (parameters of buckling etc.). The orientation can be changed by the button "<>" between numbers of beginning and end joints. The weindow has to be closed by the button "OK".

Properties of me	ember 3		×
— Geometry — Beginning: — Member profile	3 💌	End:	4
Profile	Edit section	Edit material	3 2
Section: TK 10 A = 7,64E+03 m Material: EN 1 E = 210,0E+03 $\alpha_t = 12,00E+06$ Member	58×16 nm ² $I_{\gamma} = 22$, $0025 : Fe 360$ MPa G = 81,0 $1/K$ $\gamma = 78,5$	3E+06 mm ⁴ 00E+03 MPa 50 kN/m ³	2Y
End conditions:	Fixed I	- Fixed	Load from structure
- Beginning (loca	l coordinate sys.): -		dinate sys.):
Displ.:	Rot.:	Displ.:	Rot.:
1: 🔽 Fixed	1: 🔽 Fixed	1: 🔽 Fixed	1: 🔽 Fixed
2: 🔽 Fixed	2: 🔽 Fixed	2: 🔽 Fixed	2: 🔽 Fixed
3: 🔽 Fixed	3: 🔽 Fixed	3: 🔽 Fixed	3: 🔽 Fixed
Hinge 2,3	Fixed	Hinge 2,3	Fixed
Special	Number 3		OK Zancel

Change of column orientation

After that, both columns have identical orientation parallel to the axis "Z".



The structure with identical orientation of columns

Load input

The input of loads follows. We switch the tree menu into the mode "**Load cases**" and add new load cases with the help of the button "**Add**" in the toolbar, that is located on the left side of the input frame.



Load Load cases Load Load Load Load Load Load	1	Y
Add member load Edit load Delete load Combinations	 Add ↓ Edit X Remove 	Number

The button for insertion of new load cases

The window "**New load case**" automatically starts with properties of the load case "**Self weight - permanent**". This load case represents a self weight of the structure and the loads included in this load case are generated automatically according to the member properties. We insert this load case by the button "**Add**".

Load case			
Name:	G1 self weight-permanent		
Code:	self weight Type:	permanent	
Load factor -	unfavourable effect of load :	γf,Sup = 1,	35 [-]
Load factor -	favourable effect of load :	γ _{f,Inf} = 0,	90 [-]
Category:	[default input]		
Factor of peri	manent load reduction in alternative combination	on : ξ = 0,	85 [-]
actor of com	bination value :	ψ ₀ =	[-]
actor of free	quent value :	ψ1 =	[-]
	si-permanent value :	W2 =	[-]

Load case "Self-weight"

Analogously, we add two other load cases with selected type "Long-term variable".

Name:	Q2 force -long	g-term variable			
Code:	force	▼ Type:	long-term variable		
.oad factor -	unfavourable effect of	load :	Yf, Sup	= 1,50	[-]
.oad factor -	favourable effect of loa	id :	7f, Inf	=] [-]
Category:	Category A: domest	iic, residential areas			
Factor of per	manent load reduction ir	n alternative combina	tion : Ę	=	[-]
actor of con	bination value :	Ψο	= 0,70	[-]	
actor of fre	quent value :	Ψ1	= 0,50	[-]	
actor of gua	asi-permanent value :	Ψ2	= 0,30	[-]	

Properties of long-term load case



The list of specified load cases is displayed in the bottom frame.

+ Add				Load case	
	Number	Name	Code	Туре	Category
- Eult	1	G1 self weight-permanent	Self weight	Permanent	[default input]
X Remove	2	Q2 force -long-term variable	Force	Long-term variable	Category A: domestic, resid
1 Up	> 3	Q3 force -long-term va	Force	Long-term variable	Category A: domestic, r

List of load cases

The next step is to insert loads into these load cases (except self weight). The active load case (we start with "Q2") has to be selected in the drop-down menu in the heading of the workspace.

•	<pre><none></none></pre>	 Active load case 	Q3 force -long-term variable - Force	
Q	¥.		[NA] G1 self weight-permanent - Self weight	
Q		×	Q2 force -long-term variable - Force	
Q			los force folig-term variable if orce v	

Choice of the active load case

We switch the tree menu into the mode "Add member load" and specify linear load with the value -18 kN/m in the window "Prototype of member load". The prototype properties has to be confirmed by the button "OK".

Add scissor joint	225	Prototype of m	ember load	×
 Load O Load cases O Load cases O Load ✓ Add joint load ✓ Add member load ✓ Edit load ✓ Delete load ✓ Combinations ✓ Dynamics Image Calculation 	● 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	- Load values - Type: Orientation: f: -18,0	distributed on entire member Orientation along global Z-axis 0 kN/m]	Preview

Prototype of member load

We assign this load to the beam by clicking on the member in the workspace. The load appears in the workspace immediately.



Structure with inserted load

We change the active load case to "Q3" in the heading of the workspace. As the load will be identical for both columns, we will use a tool for insertion of loads to selected members. First, we select columns in the workspace. Selected members are highlighted by a green colour. After that, we select the option "Add member load" in the context menu.



<u>↓</u> ×+1		 Delete selected elements (2) Modify selected elements (2)
		 Copy properties to selected members (2) Edit profiles of selected members (2) Edit end conditions of selected members (2)
Z+	2	🛓 Add member load
	,¢Y	Cancel selection Cancel settings X Cancel

The context menu for columns

We select the load type "**trapezoid on part of member**" and the orientation "**Along global X-axis**". We also specify values at the beginning and end and the length.



Load for columns

The load is applied to both columns after the confirmation by the button "**OK**". The selection of columns can be cancelled by the key "**Esc**".



The structure with column loads

The last part of the load input is a creation of combinations. The combinations are organized separately for ultimate and serviceability limit states in the program. First, we specify combinations for ultimate limit states. We will create two combinations. Both of them will contain all three load cases, however, the main variable load will differ. The tree menu has to be switched to the mode "**Combinations**". We add new combinations with the help of the button "**Add**" in the tab"**1st order combination ULS**" in the bottom frame.



Edit load	A X		
Combinations Dynamics	1st order combinatio	n ULS (0)	1st orde
Calculation	+ Add	Numb	er
	∉ Edit	-	

The button for input of load combinations

Combination properties are organized in the window "**New combination**". Load cases can be added into the combination with the help of the left check boxes in the column "**Consider**" of the table, which can be found in the mid part of the window. We also tick the second check box for the load case "**Q2**" in the column "**Consider**". This setting sets the load case "**Q2**" as main variable load. We insert the combination into the project with the help of the button "**Add**".

Combination	characteristics				
Name:	Q2:G1+Q3				
Г <mark>у</mark> ре:	Basic				•
		Load c	ase	Ena	ble
Name		Code	Туре	Consider	Factor
G1 self weig	ght-permanent	Self weight	Permanent		1,00
Q2 force -lor	ong-term variable	Force	Long-term variable		1,00
Q3 force -lor	ng-term variable	Force	Long-term variable		ψ ₀ (0,70)
Q2 force -lor Q3 force -lor Accidental loa	ong-term variable ong-term variable ad:	Force Force	Long-term variable Long-term variable Factor for m	ain variable load;	1 ψ ₀ (0,

The main variable load

We change the setting in the second column "**Consider**" and set the load case "**Q3**" as main variable load. Again, this new combination has to be added by the button "**Add**". We close the window by the button "**Cancel**". Created combinations can be displayed in the dedicated window "**Table of combinations**". This window can be opened with the help of the button "**Table**" in the toolbar on the left side of the bottom frame.



998		Combinatio	ons	G1 self weight	Q2 force -long	Q3 force -long	
0/2	Name	Type	Accidental	Permanent	Long-term var	Long-term var	
Number			load	Enable	Enable	Enable	
> 1		Basic		1,00	✓ 1,00	ψ ₀ (0,70)	1
2		Basic		1,00	ψ ₀ (0,70)	✓ 1,00	
							2
Combinatio Main varia	on Q2:G 1	L+Q3; typeBasic	; erm variable				
Short desu Yf,sup,1(1,	<i>cription:</i> 35)*[G1]	+ yf,sup,2(1,50)*	^ε [Q2] + γ _{f,sup,3} (1,50)* _{¥0,3} (0,70)	*[Q3]		
Long desc Yf,sup,1(1,	<i>ription:</i> 35) * [G1	self weight-perm	nanent] + _{Yf,sup}	,2(1,50) * [Q2 fo	rce -long-term	variable] +	

Table of combinations

The input of combinations for serviceability limit states follows. We will use an automatic tool "Generator of combinations" in this case. This tool can be opened by the button "Generate" in the tab "1st order combination SLS".

1st order combination	n ULS (2)	1st	order combination SLS (0)	2nd order combination ULS (0)
🗲 Generate 📐	9		2		
+ Add	N	umb	er		
∉ Edit					

The button for generating of combinations

The window "Generator of combinations" contains rules, which affect the count of created combinations. We add a new rule which ensures, that specified variable load cases will be included in combinations together. We will use the button "Create" for the table "Mutually interacting load cases".

utually inte	eracting load cases	Excluded int	eraction of load cas	es.	Load cases load.	and groups acting a	s the main variabl
Create	Resolve	Add	Modify	Remove	Automa	tically create main va	ariable loads
	Interacting load cases		Excluded mu	tual interaction	Add	Modify	Remove
> 1	rom these G: 1; Q: 2 G1	Count: 0				Main variab	e loads
2	Q2 Q3				Count: 2	Q2	
		~			~	Q3	
aracteristic	cs of generator						
sting comb	inations: remove all co	ombinations		•	Permanent	loads act only unfav	ourably
nerate:	🔽 Characteristic 🔲 Fi	requent 🗌 Qua	si-permanent 🗌	Final deformation	All permane	nt loads always in co	mbination
					Expected pur	where of combinati	ione i F

The button for input of interacting load cases

We tick load cases "Q1" and "Q2" in the window "Mutual interaction" and insert this rule by the button "Merge".

lutual interaction		>
	G1	1
	Q2	
	Q3	
	🔳 Merge	E X Cancel

Mutual interaction of load cases

Other parameters of generator may stay unchanged. The tool creates a set of combinations for serviceability limit states after closing the generator window by the button "**Generate**".

Enlargement of structure

This chapter describes the enlargement of the structure with the help of tool for copying the elements. This tool will copy existing frame to a new position. First, we select upper joints, as transverse beams can be created automatically in the positions of selected joints.





Selection of joints in the workspace

We use the tool "Copy" in the tree menu and select the mode "Move in axis direction".

Tools - Copy	×
- Move methods	- Preview
move in axis direction	+
O move determined by two points	Z A X Y
	OK Cancel

The choice of vector input

The settings in the window "**Move/Copy parameters**" should be defined according to the figure below. We use also the setting "**Connect sel. joints by members**", which creates new beams connecting old and new frames in selected joints. The cross-section of these members has to be set with the help of the button "**Member - set**".



Move/Copy parameters	×
- Information	
Total joint count: 4	Total member count: 3
Count of selected joints: 2	Count of selected members: 0
- Manipulation method	Elements to be handled
Copy OMove	○ All
Copy parameters	
Filter identical joints and members	Copy supports
Copy concentrated masses	Copy loads
Create new saved selection	
Connect sel. joints by members	Member - set
Copy count: 1	'V
- Moving parameters	
Move in direction of the X-axis:	0,000 [m]
Move in direction of the Y-axis:	0,000 [m]
Move in direction of the Z-axis:	0,000 [m]

The button for input of connecting beams properties

The known window "**Profile prototype**" appears. It is possible to use the button "**Load from structure**" in the left bottom corner and copy cross-sectional properties from existing member.



Profile prototype			×
- Section		- Section type:	
r	V	Section type.	
		Steel	Timber
		Concrete	Masonry
		User defined	Editor
	→ Y	Edit in	nput
		- Section rotation: $\alpha = 0$,00 [9]
Structural st	eel I(IPN) 200		
A = 3,34E+03 I _y = 21,4E+06	mm^2 P = 707,1 m mm ⁴ I _z = 1,16E+	nm 06 mm ⁴	
Material			
Catalogue	EN 10025 : Fe 360 E = 210,0E+03 MPa	G = 81,00E+0	3 MPa
User defined	$\alpha_{t} = 12,00E-06 \ 1/K$	γ = 78,50 kN/n	1 ³
Load from strue	ture	🗹 ОК	Cancel

The button for copying properties

The last part of copy properties is the vector of transformation. We enter the value *-4m* into the input line "**Move in direction of the X-axis**". The parameters have to be confirmed by the button "**OK**".



Move/Copy parameters	×
- Information	
Total joint count: 4	Total member count: 3
Count of selected joints: 2	Count of selected members: 0
Manipulation method	Elements to be handled
● Copy ○ Move	All Oselected
Copy parameters	
✓ Filter identical joints and members	Copy supports
Copy concentrated masses	Copy loads
Create new saved selection	
Connect sel. joints by members	Member - set
Copy count: 1	
Moving parameters	
Move in direction of the X-axis:	-4,000 [m]
Move in direction of the Y-axis:	0,000 [m]
Move in direction of the Z-axis:	0,000 [m]

Input of vector

The tool copied the frame into a new position and also added new beams connecting old and new frame.



Created structure

Columns were copied including loading. As only two columns should be loaded, we have to delete loads on new columns. We select these columns and use the tool "**Delete load at selected members**" in the context menu.





Deletion of selected loads

We cancel the selection by the key "Esc". After changing the active load case to "Q2" in the heading of the workspace, the workspace shows, that there is not any variable load pplied to new transverse beams.



Load case "Q2"

We select these beams and insert loads with the help of the tool "Add member load" in the context menu.



Add load for selected members

The new load properties are identical to existing ones.

New load of sele	cted members		×
-Load values -			5
Type:	distributed on entire member		
Orientation:	Orientation along global Z-axis	▼ ²	
f: -18,00) [kN/m]	2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2	
		lok.	
			el

Load properties for new beams

The last topology editing is the input of supports. First, we select all base joints of columns.





Selection of base joints

We will use the tool "Edit supports of selected joints" in the context menu for selected joints.



Support properties for selected joints

The joinst should be supported in all directions. We can use pre-defined button for fixed support in the right part of the window. The window has to be closed by the button "**OK**".



The button for easy input of fixed support

The structure is complete and it is possible to start the analysis.





Completed structure

Analysis and results

The analysis can be run using the tool "**Calculation**" in the tree menu. The window "**Calculation properties**" with analysis parameters appears before the analysis. We start the calculation by the button "**OK**".

alculation p	properties		>	
Calculation	Calculation setting			
- 1st order	analysis			
Ist order analysis Ist order analysis Image: Calculate Load cases count: Combinations count: Design Total combinations count: standard design can be launched Fire (accidental) combinations count: fire check cannot be launched Image: Calculate Image: Calculate Combinations count: Save data prior to calculation	3			
		Combinations count:	5	
- Design -				
Total combinations count: standard design can be launched				
Fire (accid	dental) combinations count:		0	
fire che	ck cannot be launched			
Save da	ata prior to calculation			
			Cancel	

Confirmation before analysis

The window with analysis log is shown after the calculation is finished. We will go back to the main window with the help of the button "**Close**".



Calculation progress Calculation results Data recapitulation (i) joint count: 8 (i) members count: 8 (i) load cases count: 3 Preparing 1st order calculation (i) count of calculation joints: 8 (i) count of calculation members: 8 (i) combination count: 5 Static 1st order calculation (i) Calculation successful	alculati	ion	
Data recapitulation (i) joint count: 8 (i) members count: 8 (i) load cases count: 3 Preparing 1st order calculation (i) count of calculation joints: 8 (i) count of calculation members: 8 (i) combination count: 5 Static 1st order calculation (i) Calculation successful	Calcula	ation progress Calculation results	
 (i) joint count: 8 (i) members count: 8 (i) load cases count: 3 Preparing 1st order calculation (i) count of calculation joints: 8 (i) count of calculation members: 8 (i) combination count: 5 Static 1st order calculation (i) Calculation successful 	Dat	ta recapitulation	
 (i) members count: 8 (i) load cases count: 3 Preparing 1st order calculation (i) count of calculation joints: 8 (i) count of calculation members: 8 (i) combination count: 5 Static 1st order calculation (i) Calculation successful 	(i)	joint count: 8	
(i) load cases count is Preparing 1st order calculation (i) count of calculation joints: 8 (i) count of calculation members: 8 (i) combination count: 5 Static 1st order calculation (i) Calculation successful	(i)	members count: 8	
 (i) count of calculation joints: 8 (i) count of calculation members: 8 (i) combination count: 5 Static 1st order calculation (i) Calculation successful 	Pre	paring 1st order calculation	
 (i) count of calculation members: 8 (i) combination count: 5 Static 1st order calculation (i) Calculation successful 	(i)	count of calculation joints: 8	
(i) Calculation successful	(i)	count of calculation members: 8	
(i) Calculation successful	(I)	compination count: 5	
(I) Calculation successful	Sta		
	(1)	Calculation successful	
	(1)	Calculation successful	

Window "Calculation" with analysis report

The deformed structure appears in the workspace.



Deformations in the structure

The program contains variety of tools for displaying the results. The window **"Results view settings**", where the displayed quantities can be selected, can be launched by the button "\$" in the toolbar above the workspace.



We switch on the display of bending moments.

Result type: Result drawing method						
Describe	Highlight maxima					
Description type:	All values					
Describe	Highlight maxima					
Describe	Highlight maxima					
Describe	Highlight maxima					
🔽 Describe	Highlight maxima					
🔽 Describe	🔲 Highlight maxima					
🔽 Describe	🔲 Highlight maxima					
🔽 Describe	🔽 Highlight maxima					
🔽 Describe	🔲 Highlight maxima					
coordinate system	•					
🔽 Describe	🔲 Highlight maxima					
🔽 Describe	🔲 Highlight maxima					
🔽 Describe	Highlight maxima					
Describe	Highlight maxima					
Describe	Highlight maxima					
🔽 Describe	🔲 Highlight maxima					
sections:						
Describe	🔲 Highlight maxima					
Describe	🔽 Highlight maxima					
🗖 Describe	🔲 Highlight maxima					
Compl. description	Maxima everywhere					
No description Maxima no						
	Result drawing met Pescribe Describe Compl. description					

Bending moments

Results can be shown for given load case, combination or envelope. Envelopes show extreme values for more selected load cases or combinations. We will define the envelope of both combinations for ultimate limit states. We select the option **"Envelope of 1st order combination, ULS"** in the drop-down menu **"Diagrams"**.



The window "Envelope of 1st order combinations" appears. It is necessary to tick both combinations in the left part of the window.



Envelope of 1st order comb.			×
 List of 1st order combinations: [1] Q2:G1+Q3 [2] Q3:G1+Q2 	All None Inverse Original	Internal forces Minimum Maximum Both extremes Envelope key Per member Per section All N V ₂ V ₃ M ₂ M ₃ M ₁	Reaction Minimum Maximum Both extremes Envelope key All F _x F _y F _z M _x M _y M _y M _z
			OK Cancel

Choice of included combinations

New diagrams show extreme values of bending moments in all points of the structure.



Envelope of bending moments for ULS

Verification

The last part of the work is verification of members in designing modules. To reduce the necessary inputs, we merge eight members into two design groups (columns and beams). Design group is verified as one member, however, more results of analysis are considered during the verification. The design groups can be created automatically using the tool "**Design members**" "**Generate**" in the tree menu.



Calculation	tz-
Results Diagrams	8
	2
Ecoend	

The tool "Generate" in the tree menu

We will use the option "Generate design groups" in the window "Generate".

Generate	×
Generate design members	
Generate design groups	
Divide design elements to members	
🔀 <u>C</u> ancel	

Generation of design groups

The window "**Automatic search for design groups**" shows, that two design groups (columns and beams) were found. The right part of the window shows the structure view with highlighted active group.

Au	itor	nati	c search for de	sign groups		×
-	Des	ign g	proups			
			Description	Direction		
	*	1	element 1	identical	~	
		2	element 2	different		
						. /
						+ 1
						t
					~	
-	C	onsid	er all generated	d <mark>elem</mark> ents		☑ OK Zancel
	15		36			

Automatic search of design groups

The warning regarding the different orientation of beams appears after the confirmation of design group search. As beams have symmetrical verification parameters (buckling properties etc.), we can continue using the button **"OK**".



Warning regarding different directions



The created design groups can be described in the bottom frame for the mode "Design members" of the tree menu.

Calculation Calcul	8 .34,52	34,82 14.07 34,83 14.07 34,83 14.07 34,83 14.07						
Break	Generate		Description	Туре				
C Design	,	1:DG	Columns	design group	1,			
Legend	Combine	> 2:DG	Beams	design group	2,			
Internal forces N+ N- [kN]	🗙 Divide							

Names of design groups

We switch the tree menu into the mode "Design" and run the design module "Steel" by the button "Run program".

Calculation Calculation Results O Diagrams O Design members Generate Reak Break O Design	Steel 1	.54,63	14.04 54,63
Legend	Program	Steel	🔞 Run program 🔲 Only not
Internal forces	Cons	ider all design elements	45
N+N- [KN] Ma, Ma, Ma, Ma, [KNm]			Design mer
V3, Vz V2, Vy [kN]		Description	Analysis
M ₁ T _t T _o B [kNm, k	Nm ²] 1:DG	Columns	
Reaction [kN, kNr	m] > 2:DG	Beams	
Contact stress			

Launching a designing module

All necessary data (geometry, cross-section, material, internal forces) are transferred into the design module. The design members are organized into the tree menu in the left part of the window.

100 FIN EC - FIN 3D - Steel (D: VFin EC	data)/Fin 10,30 projec	-0e *]			
File Edit Massinput Input Op	tions Help				
B B 0 h					
Project	- General projec	t data			
Counna Beanis	lef Çát	Job name : 30 project Dete : 38. 4. 2016			~
	Standard				v
	4ª Gát	Standard EN 1993-1-1, EN 1993-1-4/Czech Rep. Patters for steel structures Section capacity : THE = 1,000 Section capacity : THE = 1,000 Perforated section capacity : THE = 1,250 Patters for stankes steel Section capacity : THE = 1,200 Section capacity : THE = 1,000 Perforated section capacity : THE = 1,200 Perforated section capacity : THE = 1,200			< y
Check al					
Structure preview					
EN 1993/Czech Rep.					
			Øge	Care	cal



Designing module for steel structures

We switch to the mode "**Columns**" "**Check**" and run the verification by the button "**Analyse**". The program shows warnings, that parameters of buckling and lateral torsional buckling were not set and it is not possible to run analysis.

Analyse Analysis me Member check: no check .	ethod: Maximum utilization	envelope				-
	Section name	Coordinates [m]	Utilization [%]	In Ca [x	o check ncomplete input. Add or adjust the data needed. alculation failed () Buckling length Z on sector no. 1 must be set () Length of buckling Y on sector no. 1 must be set	^ t

Button for running the analysis

We switch to the part "**Buckling**" of the tree menu. The parameters can be defined in the bottom frame. As the parameters are constant for the whole length of the column, we will use only default length sector with analysis properties. We will edit it by the button "**Edit**" in the toolbar on the left side of the bottom frame.

Buckling for calcul	ation: consi	der buckling	in identical	sectors			•		
+ Add		Start	End	Length	Buckling Z (Buckling in d	Buckling Y (Buckling in di	
		[m]	[m]	[m]	L _{cr} [m]	k [-]	L _{cr} [m]	k [-]	
	≯ 1	0,000	2,350	2,350	(no input)	(no input)	(no input)	(no input)	~
X Remove									

The button for editing buckling properties

The window "**Buckling sector edit**" contains buckling parameters for directions *z* and *y*. The buttons "**Buckling z**" and "**Buckling y**" has to be used for the input. "**Buckling z**" contains parameters for buckling in the direction perpendicular to the axis *z*, "**Buckling y**" parameters for buckling perpendicular to the axis *y*.

Buckling sector edit	×
- Sector Sector beginning :	0,000 [m]
Sector end :	2,350 [m]
Sector length :	2,350 [m]
Buckling parameters	
Buckling Z $L_{crz} =$ (no input) $L_z =$ (2,350) m $k_z =$ (no input)	<u>्र</u> इत्
Buckling Y $L_{cry} =$ (no input) $L_y =$ (2,350) m $k_y =$ (no input)	
	🗹 OK 🛛 Cancel

The window "Buckling sector edit"

The buckling length has to be specified in the window "**Buckling**". To be on a safety side, we will select general end conditions (the button with question marks at ends) and specify value 2.0 for buckling factors " $\mathbf{k_v}$ " and " $\mathbf{k_z}$ ".



Buckling Z (Buckling in direction of axis Y)
Buckling check Sector length for buckling Lz: 2,350 [m]
- End conditions
Factor k _z : 2,000 [-]
- Buckling length
L _{cr} = sector length * factor k
L _{crz} = 4,700 m
- Buckling curve
Edit curve
Cancel

Properties of buckling length

Also shape of bending moment area and end conditions have to be specified for the analysis. We switch to the part "**LT buckling**" of the tree menu. The parameters of lateral torsional buckling are organized into two tabs according to the bending moments M_y and M_z .

buckling My	.T buckling M	z					
Add		Start	End	Length	Buckling		Moment area shape
4 1 - 10		[m]	[m]	[m]	length Iz1		
Edit	> 1	0,000	2,350	2,350	(no inpl	~	
X Remove							End conditions for buckling:
-							k7

Tabs "LT buckling M_y " and "LT buckling M_z "

The properties can be edited in the same way as buckling parameters. We specify identical properties for both directions.



Lateral torsional buckling	sector edit		
Sector			
Sector beginning :	0,000	[m]	
Sector end :	2,350	[m]	
Sector length :	2,350	[m]	
Buckling effect			
Do not consider bucklin	ng - beam is res	trained	<u> </u>
Different buckling sect	or length		
Buckling sector length:	2,350	[m]	Ā
Moment area			
Moment area shane M			
ionent area shape my			
	2	-	
]		
Load position zp:] [-]	
Ratio ψ (M _{start} /M _{end});	-1,000		
Parameters			Z
End conditions k _z :			0.0
ш "	fixed-fixed	-	
End conditions in torsion k_{W}	r.		
Δ Δ	hinged-hinge	d 🔻	

The window "LT buckling sector edit"

We switch back to the mode "Check" and run the analysis again. The results show that the member is OK.



Die golt Massinput Input Opt	Kions Help							
Ro Ko You No Project Column Column Datations D	Q Open Open Q Image: Comparison open Q Image: Comparison open Q Image: Comparison open Q Image: Comparison open V Image: Comparison open <th>A 6480 6480 6480</th> <th>TK L68</th> <th>00 1499</th> <th>10.2 - Q261+Q2; X=6 Section analysis Hember no.5 - Cc</th> <th>2,000 2,200 1.1.101 1.1.111</th> <th>2.71% (e) 100% 50% Utilization </th>	A 6480 6480 6480	TK L68	00 1499	10.2 - Q261+Q2; X=6 Section analysis Hember no.5 - Cc	2,000 2,200 1.1.101 1.1.111	2.71% (e) 100% 50% Utilization 	
	el Dát ,	Critical member out "Co	umi 0,000	70,4	Decisive load: Member no. 6 - Cambrination no. 2 - Q3/61+Q2 Cross section class 1 Check of shear clast to shear force V ₂ : 12,719/64 < 593,311/64 Page			
					CHECK OF MININF CU	IN MARKENET POPULATION AND AND AND AND AND AND AND AND AND AN		
Check all	C Printing				67,356 kN < 518,3 Internal forces: N = 54,636 kNm	11 kN Pass -86,240 kN; M _y = 9,9	12 kNim; Mg =	

Results for column

The verification of beams follows. The buckling parameters will be specified in the same way as we did for column. Difference is, that we will select pinned end conditions with buckling factors " $\mathbf{k_y}$ " and " $\mathbf{k_z}$ " equal to 1.0.

Buckling sector edit				×
- Sector			0,000	[m]
Sector end :			4,000	[m]
Sector length :			4,000	[m]
Buckling parameters		15N12-601-001-0		
Buckling Z L _{crz} =	4,000 m Lz =	(4,000) m k _z = 1,000 Δ		Δ
Buckling Y L _{cry} =	4,000 m Ly =	(4,000) m k _y = 1,000 ∆		Δ
			✓ OK	<u>C</u> ancel

Properties of buckling sector for beams

The parameters for lateral torsional buckling induced by the bending moment M_y have to be also specified. The lateral torsional buckling properties are shown in the figure below.



Lateral torsional buckling	sector edit					×
Sector						
Sector beginning :	0,000	[m]	(dz)	ņ		
Sector end :	4,000	[m]	1.0			
Sector length :	4,000	[m]	1			-
- Buckling effect						
Do not consider bucklin	ng - beam is restr	ained				
Different buckling sect	tor length					
Buckling sector length:	4,000	[m]				
- Moment area						
Moment area shape M						
Homent area shape My		_				-
	<u>∕</u> 6	-	V			~ 2
			1			
Load position zp:	1,000	[-]				6
Ratio w (Metart/Mend);		[-]				-
- Parameters						
Ford conditions k :						- 1
End conditions k _z :						
<u>м</u>	fixed-fixed	-				
End conditions in torsion ky	v					
μ μ	fixed-fixed	•	0.0	N		
					<u>o</u> k 🗵	Cancel

Properties of LT-buckling M_y

The analysis shows that beams have passed the verification.



FIN EC - FIN 3D - Steel (D:\Fin E	C\data\/Fin 10\30 p	roject.file *									×
Eile Edik Massinput (nput Q	ptions Help										
Columns Columns Columns Columns Columns Columns Columns Dearts-section Dearts-section	e e e e € e e \$		0,000 . 5.7			2,00 2,000	2,780	3,080 3,3		400 300% 50% Utilization 4 1 Cass	
		_			1 (PAR)	200					
	Member che	yse Analysi dc PASS	Nacimal utili	anum utilization enve zation: 87,7%; Her	nber na.8 - (Combination	no.1 - Q2:61	+Q3; X=4,0	100m.		12
	+ add			Section	Coordinates	Utilization	Section	analysis X=	4,000m; 87,7%; 1	oad:	,
	el pár	-	Critical or	nane ember cut "Beam	[11]	[%]	Hember	no.8 - Com	bination no.1 - Q	261+03	
	× Beau	e	CHRONN	chief of beam	4000	67,7	Decisive Q2/61+Q Check o 59,833	load: Nenb (3) Cross-si i shear due (4) < 211,69	er no.8 - Combinato ection classs 1 to shear force V ₂ 1kN Pass	n no. 1 -	ļ
and a	E Printing						Internal f = 0,000 k Critical e	promit N = -1 Nos combinatios	8,070 kH; Ny = -32	,646 khinç Ma	
							and ben Bucklin	ding mome y Y: Resistan	901 DBBC Ng = -717,398	N; My, R =	
Structure preview											
Structure preview											

Results for beams

Both design groups are verified. We will close the design module by the button **"OK**" in the right bottom corner and go back to the program **"Fin 3D**". The table in the mode **"Design"** shows the analysis results.



Verified structure