

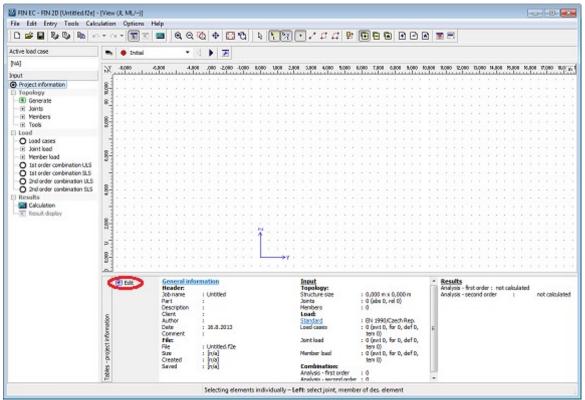
Timber truss

Task

In this example, the task is to design a symmetric roof timber truss of 13 *m* length and 25 *degrees* of the roof pitch. The truss consists of timber members of class *C24* and *40 mm* thickness; the spacing of the trusses is 1 *m* centres. The truss is subject to dead load 0.2 *kN/m* from both roofing and ceilings and to snow load 1.0 *kN/m* according to the Snow area 2 of the Czech snow map.

Setting up project

After running the program 2D, the main screen appears, consisting of the model space on the right-hand side, the control tree on the left-hand side and the input table in the bottom part. The table displays the project information which can be later used in headers and footers of the output documents. To enter or edit the project information, we can run the relevant dialog box by clicking the "**Edit**" button.



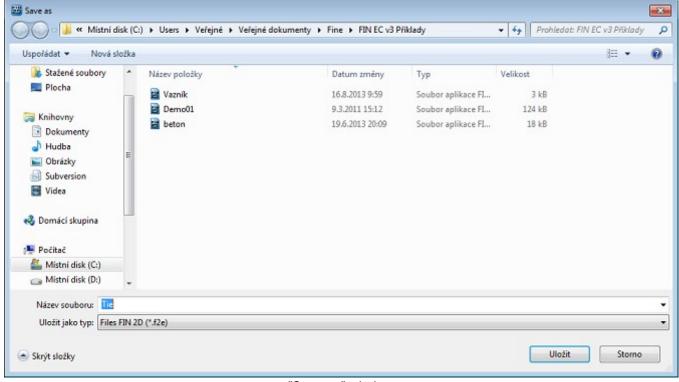
Button for running the "Project information" dialog box

In the dialog box we can enter e.g. the job title or the project author. After entering all necessary data we exit the dialog box by clicking the "**OK**" button.

Project informa	ation			X
Project Stan	dard			
Job:	Tie	Part:		
Description:		Client:		
Author:	Peter Smith Ph.D.	▼ Date:	16. 8.2013 -	
Remark:				*
				*
				¥.
				OK Cancel

"Project information" dialog box.

Before proceeding with work we should save the job using e.g. the "**Ctrl+S**" shortcut. The file name is entered and the destination folder selected in the standard 'Save as' window.



"Save as" window.

Structure generation

We can define the truss geometry either by entering individual nodes and members or we can simply make use of the Generator of 2D structures. In our example we will choose the latter approach. The generator is run by clicking the **"Generate"** button in the **"Topology"** section of the control tree.



Active load case			In	itial					•	4	•		F
[NA]	X	-8,0	000		6.00	0		-	4.00	0	,00,	0.	2,000
Input	N		1						i.				<u>. .</u>
Project information Topology	10,000		· ·	о. С.			•		•	•			
Generate	8_		 		е. 6.			•	•	•	•	•	·.
Hembers Tools	8.000 		 			13	2		•	•			о. К.

Running "Generator of 2d structures"

To generate the structure in the generator, we first click the "Wizard" button in the top left corner of the dialog box.

25	Generat	or of 2[) structures
Q	Q	2 ♦	🗆 🖸 🔀 🔇
S	tructure	_	
	Wizard		
	Outline	2	

Running wizard in Generator of 2D structures

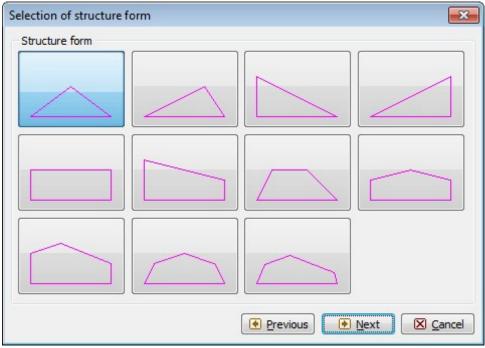
A dialog box appears enabling us to select one of the basic types of the structure. We select Basic truss types and proceed to the next step by clicking the "**Next**" button.

Select structure	e type 🛛 🔀
Structure type	2
	Basic truss types
	Collar-beam roofs
	Arch structures
	Frames, special structures
	Next Cancel

Selecting structure type

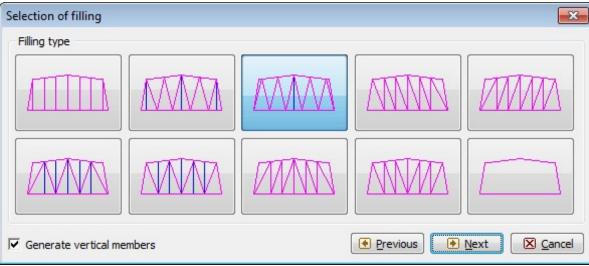
In the following dialog box, we select the desired form of the truss and click the "Next" button.





Selecting structure form

The following dialog box offers a selection of basic types of filling members' layouts. We select the desired filling type and in the bottom left corner we switch off automatic entering of verticals by unticking the "**Generate vertical members**" box.



Selecting filling type

The next dialog box "**Structure dimensions**" enables defining the main dimensions of the truss. If we enter the pitch and the length, the program will automatically calculate the height of the truss. In the field "**No. of bays on B.C.**" we define into how many segments the bottom chord will be divided by nodes. Additional nodes may be inserted between the main nodes of the upper and the bottom chords. In these nodes, the exact values of internal forces and deformations will be calculated and additional nodal forces can be defined. In the field "**No. of intermediate joints**" we can define how many nodes will be added to each segment.

	Right support height	0,000	[m]
\sim 1	Truss span (r)	13,000	[m]
	Truss height (v)	3,031	[m]
	Truss pitch	25,00	[°]
5 5	No. of bays on B.C.	5	
r /····································	No. of intermediate joints	1	[-]

"Structure dimensions" dialog box.

After definition of the truss dimensions, we need to specify cross-sections and materials of individual truss members. This will be done in the following dialog box "**Profiles in groups**" in which we can specify profiles and materials separately for the upper and the bottom chords and the filling studs by clicking the "**Profile**" button in the relevant tabs. However, we will use a quicker approach specifying first the material and profile for all members by clicking the "**Global profile**" and then only changing the profiles of the upper and the bottom chords.

				×
Upper chord	Bottom chore	d Filling		
Member pr		ross-section	Edit material]
		1055-500001		
			\bigwedge	
			\bigwedge	
	\langle	$\langle \rangle$		
	\checkmark	$\overline{\langle}$		
/	\checkmark	$\overline{\langle}$		
Global profile			Previous	

"Profiles in groups" dialog box

After clicking the "**Profile**" button, the "**Cross-section editor**" dialog box appears on screen. In this dialog box we select "**Timber**" and "**Solid squared**" type and define the dimensions of the rectangular profile as h = 80 mm and b = 40 mm.



Cross-section ed	litor - Timber, solid square	d				x
	ΤΙΊ					
	Cross-section desc	ription				
name	rectangle 40x80					
comment						
	Cross-section dime	ension				
cross-section heig	ght	h =	80,0	mm		
cross-section wid	th	b =	40,0	mm		
Information					Cance	el

Defining cross-section dimensions

After confirming the dimensions by clicking "**OK**", the "**Catalogue of materials – timber**" dialog box appears, offering a selection of the standard timber strength classes. We select **C24** and confirm by clicking the "**OK**" button.

Catalog of materials - Timber		×
Select from catalog of materials		
Timber EC 5 Timber EC 5, Czech Republic Timber EC 5, Slovakia	C 14 - coniferous C 16 - coniferous C 18 - coniferous C 20 - coniferous C 22 - coniferous C 24 - coniferous C 30 - coniferous C 30 - coniferous C 35 - coniferous C 40 - coniferous C 40 - coniferous C 45 - coniferous D 18 - hardwood D 24 - hardwood	
Information	СК	Cancel

Selection of strength class

After confirming, the selected material and cross-section data appear in all three tabs of the "**Profiles in groups**" dialog box.



Profiles in groups
Upper chord Bottom chord Filling
Member profile: Profile Edit cross-section Edit material
$ \begin{array}{c} \textbf{Section: rectangle 40x80} \\ A = 3,20E + 03 \ \text{mm}^2 & I_{\gamma} = 1,71E + 06 \ \text{mm}^4 \\ \textbf{Material: C24 - coniferous} \\ E_{0,mean} = 11,00E + 03 \ \text{MPa G}_{mean} = 690,0E + 00 \ \text{MPa} \\ \alpha_t = 5,000E - 06 \ 1/\text{K} & \gamma = 4,20 \ \text{kN/m}^3 \end{array} $
Global profile

Editing profiles of upper and bottom chords

Now we change the profiles of the upper and the bottom chords. First, in the **"Upper chord**" tab using the **"Profile**" button, we set a Pi-shaped profile and enter the dimensions.

Cross-section editor -	Timber, solid squared				— ×
	ΊΙ				
	Cross-section description				b
name	Pi-cross-section 200x240			1 1	
comment					
	Cross-section dimension			- I L	<u>۳</u>
ross-section height	h	= 240,0	mm		
cross-section width	b	= 200,0	mm		
tem thickness	t _w	= 40,0	mm		
lange thickness	tf	= 40,0	mm	-	
distance between inter	mal edges of cross-section v c	= 40,0	mm		
				4	, tw
Information					OK Cancel



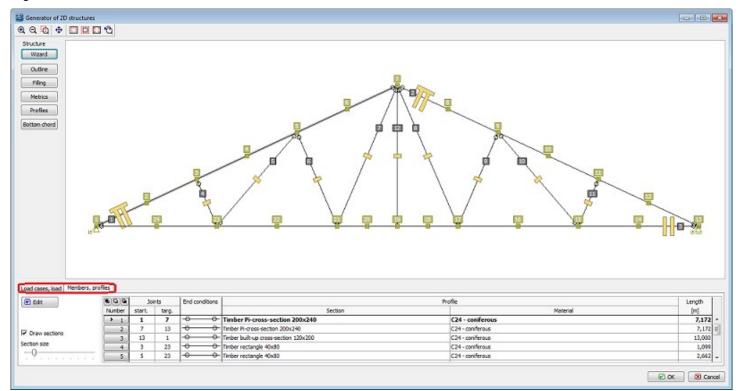
Upper chord's cross-section dimensions

Analogically, we define a compound cross-section of the bottom chord.

Cross-section ed	litor - Timber, composite							×
	Cross-section descript	ion				1.	b "	
name	built-up cross-section 120	0x200				1	1	
comment					1			
	Cross-section dimension	on						
cross-section heig	ght	h1 =	200,0	mm				
element height of	f built-up cross-section	b1 =	40,0	mm				
gap between eler	ments of built-up cross-section	b _m =	40,0	mm				
number of elemer	nts of bult-up cross-section	n =	2	pc	도			
							bm j	
Information							ОК	Cancel

Dimensions of bottom chord's compound cross-section

After changing the chords' profiles we exit the "**Cross-section editor**" by clicking "**OK**". The generated truss is now displayed on screen. We can go back to any of the previous steps using relevant buttons in the "**Structure**" frame to the left from the model space. In the frame in the bottom of the screen, the tables for managing load cases and members are organized into tabs.





Tabs for editing load cases

We select the "Load cases, load' tab and begin to define the load cases. Firstly, selecting "Self weight" we define a load case, which will contain automatically generated loads from the truss's self weight. In the dialog box we can edit the name of the load case or load factors.

Name:	G1 Self weight			
Code:	self weight Type	e: permanent		
.oad factor -	unfavourable effect of load :		γf,Sup =	1,35 [-]
oad factor -	favourable effect of load :		γf,Inf =	0,90 [-]
Category:	[default input]			
actor of per	manent load reduction in alternative com	bination :	ξ =	0,85 [-]
actor of com	bination value :		ψ0 =	[-]
actor of free	uent value :		ψ1 =	[-]
actor of qua	si-permanent value :		Ψ2 =	[-]

Parameters of "self weight" load case

Adding a new load case is confirmed by clicking "**OK**" and the loads generated from the self weight into this load case are instantly displayed in the model space. We continue with adding loads from the roofing, using the "**Roofing**" button. The "**Roofing load**" dialog box contains two tabs; the first is for specifying the load case parameters (similarly to self weight), in the second the load magnitude is defined. We switch to the second tab to enter the value *0.2 kN/m* and add the load case by clicking "**OK**". Then we exit the dialog box by clicking the "**Cancel**" button.

Roofing load		×
Load case Roofing load		
⊂Type of structure load	Per member	
Load value Roofing load	0,20 [kN/m]	
Children the	THE REAL PROPERTY OF THE REAL	
	Can OK	ncel

Tabs in the 'Roofing load' dialog box



We repeat the same procedure to define the ceiling loads.

Ceiling load
Load case Ceiling load
Type of structure load Per joint Per member
Load value
Ceiling load 0,20 [kN/m]
Add Cancel

Defining ceiling loads.

For snow loads, due to the variable nature of the loading, the dialog box for the load case properties contains different data than that for the permanent loads. Short or medium term loading type can be selected, as well as the "**Category**" which sets the combination factors in accordance with EN 1990.

oad case	Snow	load								
Name:		54	Snow load 1							
Code:	f	orce		-	Type:	:	hort-term v	ariable snow load	1	
Load facto	r - unfa	vourab	le effect of load	d:				Yf,Sup =	1,50	[-]
.oad facto	r - favo	urable	effect of load :					Yf, Inf =		[-]
Category:	5	now loa	ad - other mem	bers of (CEN, for si	ites	located at a	t. H<=1000m a.	s.l.	
Factor of p	erman	ent load	reduction in al	ternative	e combinat	tion	:8	ξ =		[-]
Factor of c	ombina	tion val	ue :					vo =	0,50	[-]
Factor of f	requen	t value	:					ψ1 =	0,20	[-]
Factor of o	quasi-pe	rmaner	nt value :					¥2 =	0,00	[-]

Snow load properties



In the second tab, we can define the loads separately for the left and the right side of the truss; it is possible to define non-uniformly distributed load caused by snowdrifts. The magnitude of the loading can be entered as the basic value obtained from the snow map; automatic redistribution on the inclined plane is run by ticking the "**Recalculate**" box. First we define the load case with uniformly distributed loads *1.0 kN/m* applied to both halves of the truss; the load is applied to the structure by clicking the "**Add**" button. Then we can change the value *s1* to *0.5 kN/m*; thus we obtain a non-uniformly distributed load case which we again apply to the truss by clicking the "**Add**" button. Finally, we switch the values *s1* and *s2* to obtain a load case symmetric to the previous. We apply it by clicking the "**Add**" button and the "**Cancel**" button to exit the dialog box.

ow load				×
oad case	Snow load	ł		
Type o	f structure lo	ad		
O Per	joint	0	Per member	
Load v	alues			
s1	1,00	[kN/m]	s3	[kN/m]
s2	1,00	[kN/m]	s4	[kN/m]
-			s5	[kN/m]
Rec	alculate			
•••		s1		s2
			-	
_	1			

Defining snow loads

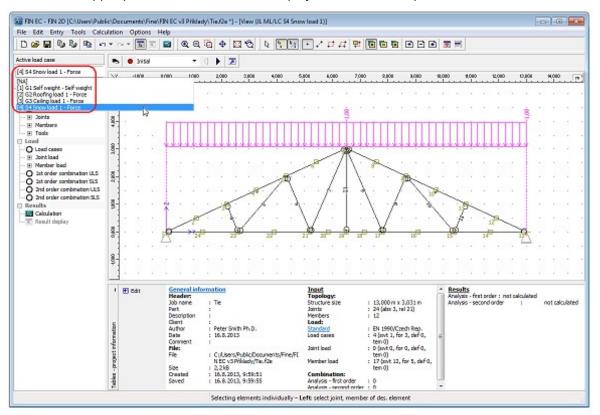
We can check and amend the defined load cases using the table in the bottom part of the Generator of 2D structures. If the load cases are correctly defined, we can insert the generated structure into the 2D program by clicking "**OK**". We can define structure placing and rotation in the table located in the bottom part of the main screen.



Inserting method	Struct	ture placing	
insert as part of existing structure	Y:	0,000	[m]
Substitute current structure by inserted	Z:	0,000	[m]
Named selections	- α:	0,00	[°]
allow multiple inserting	<u> </u>		<u>C</u> ancel

"Insert structure" table.

After inserting, the truss is displayed in the program's model space. Geometry can be further edited in the **"Topology"** part of the control tree, load cases and loads can be edited in the **"Loads"** part. Only the active load case selected from the drop down list in the upper part of the control tree is displayed in the model space.

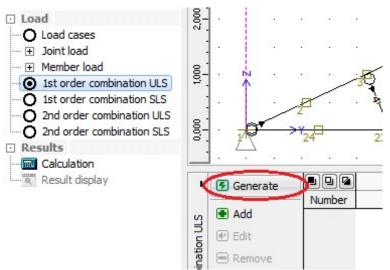


Displaying particular load cases

Definition of combinations

We can proceed to defining the load combinations which are defined separately for the ultimate and the serviceability limit states. First we define the combination for the ultimate limit state. We switch to "**1st order combination ULS**" in the control tree and run automatic definition of combination by clicking the "**Generate**" button in the table of combinations.





Button for automatic definition of combinations.

The automatic generation of combination is run in the "**Generator of combinations**" dialog box. The dialog box contains three tables. In the first, the load cases which act simultaneously are combined. In the second, we can set mutual exclusion of some load cases in one combination. The last table contains a list of variable loads which shall be considered main. In our example, we need to exclude simultaneous action of the defined snow load cases; hence we create a new exclusion group by clicking the "**Add**" button in the table "**Excluded interaction of load cases**".

Generator of combinatio	ns - combina	tions	s 1st order				×
Conditions of generator Mutually interacting load Create Resc Create Resc D D Interacting Count: 6 from these G 2 G2 3 G3 4 S4	cases alve load cases : 3; Q: 3	E (i 1st order Excluded interaction of load cases. Add Modify Remove Excluded mutual interaction Count: 0	_	load.	I groups acting as the main varia Ily create main variable loads Modify Remove Main variable loads 54 55 55 56	_
5 55 6 56		-		Ŧ			*
Characteristics of genera	-		_	_			
Existing combinations:	remove all co	mbina	(iii)			s act only unfavourably	
	Basic		Alternative Accidental	1	All permanent lo	ads always in combination	
Accidental load:	L		· · · · · · · · · · · · · · · · · · ·				
Factor for main variable I	oad:			Exp	ected numbe	r of combinations : 13	
						🕑 Generate 🛛 🔀 Ca	ncel

"Generator of combinations" dialog box.

In the "**Excluded interaction**" dialog box we select the load cases *S4*, *S5* and *S6* and confirm selection by clicking the "**Add**" button.



Input mode:	mutua	exclusion	
		G1	A
		G2	
		G3	
		S4	
		S5	
		S6	

Defining mutual exclusion of load cases

After closing the dialog box a new group of mutually exclusive load cases appears in the relevant table. Thus it is guaranteed that only one of these load cases can appear in one combination.

Add	Modify Remove	2
Count: 1	Excluded mutual interact	ion
> 1	(54) - (55) - (56)	

Added group of mutually exclusive load cases

Once we have finished entering data, we can create the combinations by clicking the "**Generate**" button. A list of the generated combinations appears in the table in the bottom part of the screen; we can add, edit or erase the combinations as necessary. We can also display the list in a comprehensive table by clicking the "**Table**" button.

1	🖲 Generate			Combination	
		Number	Name	Туре	
S	Add	> 1*	G1+G2+G3	Basic	
combination ULS	🛃 Edit	2*	S6:G1+G2+G3	Basic	
latic	Remove	3*	S5:G1+G2+G3	Basic	
mbir	<u>8</u>	- 4*	S4:G1+G2+G3	Basic	
	🕀 Up				
orde	🛃 Down				
- 1st order	🕑 Table	-			
Tables -					
La la					

Button for running "Table of combinations"

In the "**Table of combinations**" we can check the generated combinations; for the active combination, a detail description including the used load factors is displayed in the bottom part of the table.



		Combinatio	ons	G1 Self weight	G2 Roofing loa	G3 Ceiling load	S4 Snow load	S5 Sn	ow load	S6 S	now load	
0/4	Name	Туре	Accidental	Permanent	Permanent		Short-term var	Sector Sector			A state of the	
umber			load	Enable	Enable	Enable	Enable	Er	able	1	inable	1
> 1*		Basic		1,00	1,00	1,00]
2*		Basic		1,00	1,00	1,00				1	1,00	
3*		Basic		1,00	1,00	1,00		×	1,00			
4*		Basic		1,00	1,00	1,00	✓ 1,00			1		1
ombinati	< 2n G1+G	2+G3: typeBas	ic: automatically	openerated							Þ	
hort des	on G1+G	2+G3 ; typeBas + γ _{7,50,2} (1,35)*		Sec. and							Þ	
hort desi i.sup.1(1, ong desc	on G1+G cription: 35)*[G1] ription:		"[G2] + 7F, sup.])(1,35)*[G3]	1] + 19,540,3(1,5	15) * [G3 Celing	1 load 1]				Þ	C

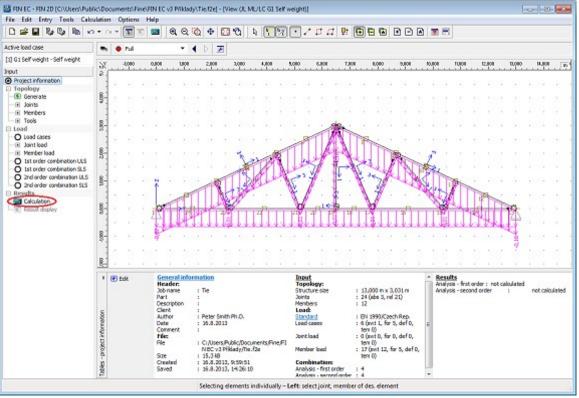
Table of combinations

Analogically we will generate the characteristic combinations switching to "1st order combinations SLS" in the control tree.

Calculation and results display

Now we can finally proceed to running the "**Calculation**" of internal forces by clicking the eponymous button in the control tree.





Running calculations

The "**Calculations properties**" dialog box appears; we can confirm the settings by clicking "**OK**" after which the calculation is executed and a window with information about the calculation process displayed. After clicking the "**Cancel**" button, program switches automatically to post-processor.

alculation p	properties		X
Calculation	Calculation setting		
- 1st order	analysis		
Calcul	ate	Load cases count:	6
		Combinations count:	4
2nd order	r analysis		
	ate	Combinations count:	4
Design			
Total com	binations count:		4
standar	d design can be launched		
Fire (accid	dental) combinations count:		0
fire che	ck cannot be launched		
Save da	ata prior to calculation		
		🛛 ок 🛛 🖾 (Cancel

"Calculation properties" dialog box.

After finishing calculation, deformation resulting from the combination *No. 1* is displayed in the model space. The program offers, apart from many other functions, a variety of settings for results displaying, e.g. enables saving views into the **"Named selections"** and printing all views subsequently. In our example we will show how to display the envelope of the bending moments. First we select **"ULS, envelope of 1st order combinations"** in the drop down list.



B 92	•	Deformation	•	•	• •	Diagrams:	ULS, envelope of 1st order o	Set
					1		[NA] ULS, load case SLS, load case ULS, 1st order combination ULS, load case envelope SLS, load case envelope SLS, load case envelope	

Selecting envelope display

To define an envelope of all combinations we click the "**All**" button in the dialog box; thus all combinations in the list on the left-hand side are automatically selected.

Envelope of 1st order comb.			×
List of 1st order combinations: ♥ [1] G1+G2+G3 ♥ [2] S6:G1+G2+G3 ♥ [3] S5:G1+G2+G3 ♥ [4] S4:G1+G2+G3	Al None Inverse Original From - to	Internal forces Per member Per section Envelope key All N V3 M2	Reaction Minimum Maximum Both extremes Envelope key All Fy F ₂ M _x
			OK Scancel

Selecting combinations for envelope

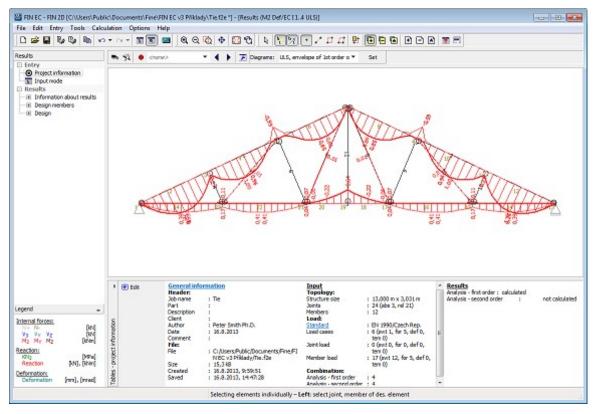
Then we run the "Cross-section display settings" dialog box and select "Bending moment".



FIN EC - FIN 2D [C:\Users\Public\Documen	ts\Fine\FIN EC v3 Příklady\	Tie.f2e] - [Results ([Def/EC I 14 ULS)]	
File Edit Entry Tools Calculation Op	tions Help			
	5 🔳 🔲 🔍 Q 🤅	\$ ⊕ ⊠ €		* 🗗 🕂 🛱 🔁 🖻
Results	Deformation	🔹 🕨 🗾 Diag	rams: ULS, envelope or	f 1st order o 🔻 Set
Entry				
Project information Input mode	Cross-section display set	ting	X	
Results Information about results	Result type:	Result drawing met	thod	
···· 🛨 Design members	Deformation	✓ Describe	Highlight maxima	
E. Design		Description type:	All values 👻	
	Reaction Fy	Describe	Highlight maxima	
	Reaction F _z Reaction M _x	 Describe Describe 	 Highlight maxima Highlight maxima 	
	Contact stress 3	✓ Describe	Highlight maxima	
	Draw internal forces along	beam span		
	Normal force - N	🔽 Describe	Mighlight maxima	× 13 &
	Shear force - V ₃	Describe	Highlight maxima	$I = \lambda f$
	Bending moment - M ₂	Describe	Highlight maxima	20 19 18 17
				20 19 16 17
	Draw everything	Compl. des.	Maxima everywhere	
	Draw nothing	No descr.	Maxima nowhere	
			<u>QK</u> <u>Cancel</u>	

"Cross-section display setting" dialog box.

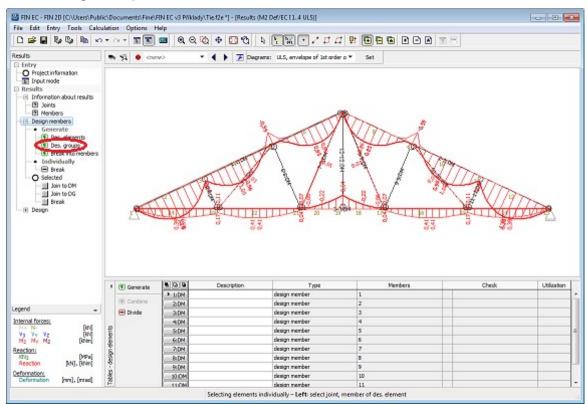
After confirming, the envelope of bending moments is displayed on the structure.





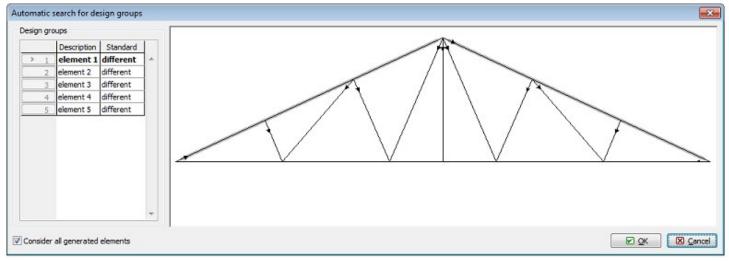
Envelope of bending moments on structure.

Now we proceed to design checks of the structure's cross-sections. First we switch to the "**Des. groups**" in the control tree. The structure consists of total of 11 elements, representing 11 design members. The program enables merging the members into design groups so that the assessment is as quick and straightforward as possible. Members merged into a design group are checked as one member; the loading is however considered on all members separately. This approach is beneficial for instance in case when we need to check a number of concrete columns in which we want to have unified reinforcement – it is sufficient to merge all of them into one design group. To create design groups automatically, we select "Generate – Design Groups" in the control tree.



Generating design groups

We can check in the dialog box which design groups were found by the program. If we want to create only some of the suggested groups, we can untick the "**Consider all generated elements**" box and then proceed only with selected design groups. In our example we will use all suggested groups, therefore we close the dialog box by clicking "**OK**".

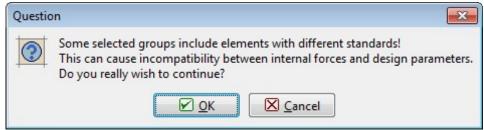


Suggested design groups

Because the orientation of particular elements vary we need to appreciate in case of which members this could cause difficulties – in our example it could be the upper chord. However; cross-sections and buckling parameters are constant



along the length of the upper chords, therefore varying axes orientation should not influence the results of the assessment.



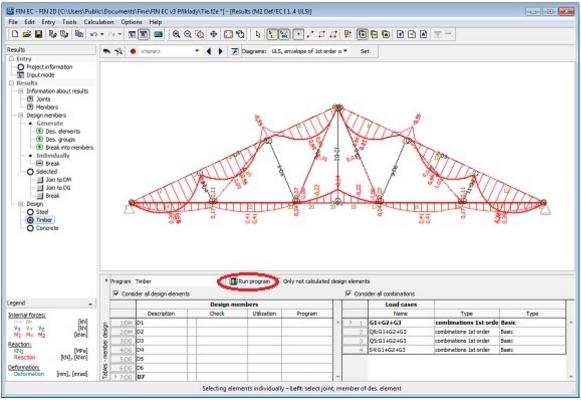
Notice on different element's orientation

Individual elements have been merged into 5 design groups and one design member. We can name the members and groups in the table in the bottom part of the screen.

Gene	1:DM D1	design member			
Com	bine and ba		3		
	2:DM D2	design member	12		
🔳 Divid	ie <u>3:DG</u> D3	design group	1, 2		
	4:DG D4	design group	4, 11		
	5:DG D5	design group	5, 10		
	6:DG D6	design group	6, 9		
	> 7:DG D7	design group	7, 8		

Table with entered names of design members and groups

Now we can proceed to the design itself. We select "**Design**" and "**Timber**" in the control tree and run the timber structures design program by clicking the "**Run program**" button.



Running timber structures design program

Members design

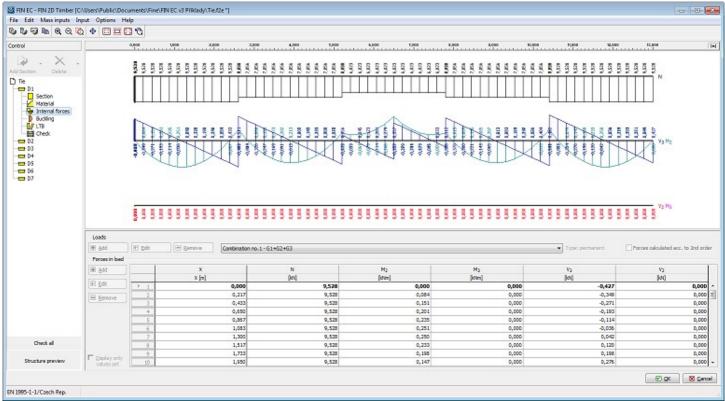
Program 2D Timber is run with all design members and groups automatically imported.



		icuments/Finel/FIN EC v3 PfRdady/Tie/2e "]	
File Edit Massinputs In	put Options	Help	
669 m			
Control	General proje	t data	
Q . X ·	⊞ Edit	Job name: Te	*
Add Section Delete		Date: 15.5.2013; Author: Peter Swift Ph.D.	
D Tre D 1 D 2 D 3 D 4 D 4 D 5 D 5 D 5 D 5			~
- D7	Standard		
	配 Edit	Calculation standard EN 1995-1-1	
		Collabor eccording to Caech national answer, (Caech Rep.) Pactor type frae according to accel trader = 1,300 Pactor type for basic contributions - glued triber = 1,250 Pactor type for accelental contributions = 1,000	
Check all Structure preview			
EN 1995-1-1/Caech Rep.			E OK Street

Design members in 2D Timber program

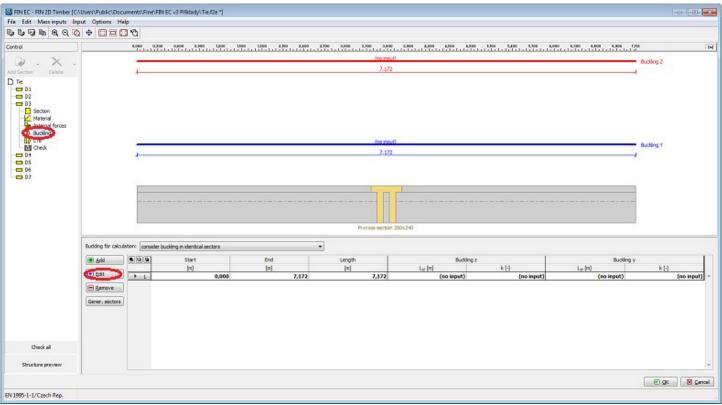
All data regarding geometry (members' lengths, cross-sections etc.) and loading (internal forces distribution for all combinations) have been imported into the program. The data can be checked in the relevant sections of the control tree. We can confirm position of a selected member in the structure by clicking the **"Structure preview"** button.



Distribution of internal forces in bottom chord

Proceeding to members design, we will demonstrate the procedure on the *upper chord* i.e. "**D1**" design group. The upper chord is subject to compression; therefore it is necessary to define the buckling parameters. In our example we assume that out-of-plane buckling is restrained by purlins at *0.6m* centres. We switch to "**Buckling**" in the upper chords section of the control tree and run the buckling parameters dialog box by clicking the "**Edit**" button.





Editing buckling parameters

In the "Edit buckling sector" dialog box we can define parameters for out-of-plane buckling ("Buckling Z") and in-plane buckling ("Buckling Y"). For out-of-plane buckling we define simple end conditions and sector length for buckling L_z = 0.6*m*; for in-plane buckling we define simple end conditions as well with sector length 2.4 m. The parameters are defined in the "Buckling Z" dialog box.

Buckling Z (Buckling in direction of axis Y)
Buckling check Image: Buckling Control of the sector length for buckling Image: Different sector length for buckling
Sector length for buckling Lz: 0,600 [m]
End conditions
Buckling length
L _{cr} = sector length * factor k
L _{crz} = 0,600 m

Defining in-plane buckling length

When the parameters are defined for both directions we can close the dialog box by clicking "OK".



Edit buckling sector				×
Sector Sector beginning :			0,000	[m]
Sector end :			7,172	[m]
Sector length :			7,172	[m]
Buckling parameters Buckling Z L _{crz} = Buckling Y L _{cry} =	0,600 m L _z = 2,400 m L _y =	k _z = 1,000 k _y = 1,000		<u>-</u> 2
				ancel

Defined buckling parameters

We switch to the "**Check**" section of the control tree and run the calculations by clicking the "**Calculate**" button. In the model space, the cross-section utilization curve is displayed along the length of the member; the critical section with the highest utilization is checked in the bottom right corner of the screen. If it is necessary to display detail checks in other sections, these can be added using the table in the bottom part of the screen or simply by double-clicking in the selected location of the utilization diagram in the model space.

🐸 FIN EC - FIN 2D Timber (0	Users\Public\Documents\Fine\FIN EC v3 Pfildady\Tie.f2e "]			
File Edit Massinputs I	aut Options Help			
	0 III 0			
Control Contro			2,2400 , 2,700 , 2,200 , 2,200 , 2,400 , 2,500 , 4,000	
D Bucking Uy UTS Uy UTS D4 D5 D5 D7			Prosestection 200x240	
	E Calculate alculation method: Calculate envelope of me	simum utilizations from all los	ada	•]
	Nember check: PASS Maximal utilization: 14,8%; Memi	er no.2 - Combination no	a.4 - \$461+62+63: X=5.977m.	
		Coordinates Utilization	Cross-section check X=5,977hr; 14,8%; Load: Member no. 2 - Combination no. 4 - 54:01402+03.	
	of cross-sector	[*] [%]	Decisive load: Member no. 2 - Combination no. 4 - 5%G14G24G3	
	E Edt > 1 Critical member cut "D3" - se		Internal forces: N = -29, S09 kH; M _V = 1,259 kHir; M _E = 0,000 kHir; V _E = 0,097 kH; V _V = 0,000 kH	
	E Banava		Compression and bending moment combination check: Remainment: Na ± 326 , 126 kit; Na, $k_{\rm R} \approx \pm 21$, 759 kits - 4, 090 + 4, 056 + 0, 000 + - 40, 140 checks	
	Re Printing		Shear forces checks Residence: Va = 24,025 M 0.004 < 1 Pages	
Check all				
Structure preview		-	Member slendersen: 32,5 Sector pl	*
			E Contraction of the second	O GK Sencel
EN 1995-1-1/Czech Rep.				

Members check

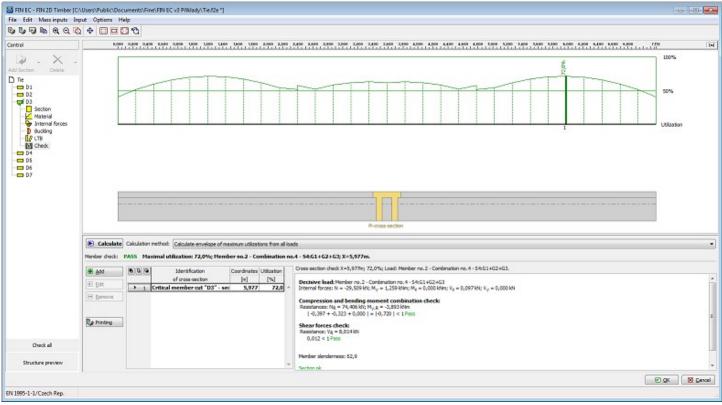
Because the maximum utilization of the member is very low, we can reduce the size of the cross-section. We switch to the "**Section**" part of the control tree and run the "**Cross-section editor**" dialog box where we can edit the cross-section geometry.



	Cross-section description				b	
name	Pi-cross-section					
comment						
	Cross-section dimension					
cross-section he	ight h =	125,0	mm			
cross-section wi	dth b =	120,0	mm			
stem thickness	t _w =	25,0	mm	1. 1. 1. 1. 1. 1.		
flange thickness	t _f =	25,0	mm	-		
distance betwee	n internal edges of cross-section v c =	40,0	mm			
					, tw	+
				│ ↓ └		

Edited dimensions of the upper chord

After editing the geometry as shown we return to the "**Check**" part of the control tree and re-calculate the structure, obtaining more acceptable check results.



Optimized member check

Now we proceed with the "**bottom chord**". Because the bottom chord is in tension it is not necessary to define buckling parameters. However, we need to define lateral torsional buckling parameters as lateral and torsional stability should be checked in members subject to combination of tension and bending. We switch to the "**LTB**" section of the control tree and analogically to the buckling of the upper chord we define the buckling parameters for bending moment M_y by clicking the "**Edit**" button.



Buckling for calculation: consider buc	kling	•	
LTB My LTB Mz			
	Start	End	Length
	[m]	[m]	[m]
	0,000	13,000	13,000
Remove			
Gener. sectors			

Editing buckling parameters

In the "**Buckling sector editing**" dialog box we define the LTB effective length and select appropriate beam and load type. We close the dialog box by clicking "**OK**".

Sector		
Sector beginning X :	0,000	[m]
Sector end :	13,000	[m]
Sector length :	13,000	[m]
LTB effect		
Neglect LTB (buckling pr	evented)	
Different sector length t	for LTB	
Sector length for LTB:	2,60	00 [m]
Beam and load type		
Beam type and load for My		
	beam	•
Load position with respect to	section height:	
at the bottom		•
Beams are fixed in supports		sverse
and torsional instability (LTB		

Buckling parameters

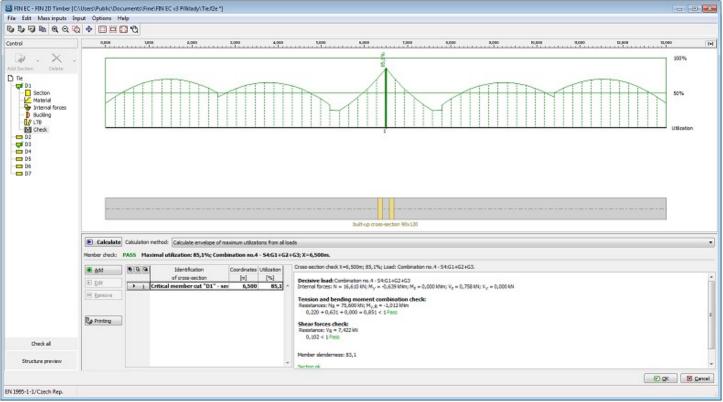
Then we switch to the **"Check"** section of the control tree again and carry out the members design check. Also for this member the utilization is too low therefore we edit the cross-section again.



	Cross-section descript	ion				b	
name	built-up cross-section 90>	120					1
comment							
	Cross-section dimension	on					
cross-section he	eight	h1 =	120,0	mm			
element height o	of built-up cross-section	b1 =	25,0	mm			
gap between ele	ements of built-up cross-section	b _m =	40,0	mm			
number of eleme	ents of bult-up cross-section	n =	2	pc	표		
						b1 . bm .	

Editing geometry of the bottom chord

We carry out the design check again confirming more economic design of the member.



Bottom chord's design check

Finally, the diagonals remain to be checked. Because the diagonals' properties are almost identical, we can define the calculation parameters for all of them together. This can be done using a function in the "**Mass input**" part of the main menu. First we define the buckling parameters.



		\Documents\Fine\FIN EC v3 Příklady\Tie.f2e *] ; Help
By By 5	Section- joint Material	1.000 2.000
Control	Buckling	
Add Section	Service class	

Mass input of buckling properties

Even though it would be sufficient to define the buckling parameters for the compressive members only, it is easier to assign them to all diagonals. Hence we select members D1 to D4 on the left-hand side. In the right part we tick the "**Take over sector geometry from analysis**" so that the program will use the actual lengths of the members as buckling lengths. Finally we define simple supports at both ends for both y and z directions. After clicking "**OK**" the entered parameters are assigned to all diagonals.

Buckling	
List of members for parameter input D1 Selection mode D2 all D3 no check D5 checked D6 sufficient not sufficient mot sufficient Selected : 5 Count : 7	Global Take over sector geometry from analysis Buckling for calculation: consider buckling in identical sectors Buckling parameters Buckling Z L_z =rom geometry $k_z = 1,000 \Delta$ Buckling Y L_Y =rom geometry $k_Y = 1,000 \Delta$

Mass input of buckling parameters

We continue with defining lateral torsional buckling parameters. Diagonals are generally only subject to axial forces therefore the LTB checks are not required; however, due to their self weight, small bending moments can occur. In such cases, the program demands lateral torsional buckling checks. In the "**Mass input**" section we select "**LTB**"; in the left part of the dialog box we again select all diagonals and from drop down list in the right part we select "**do not consider buckling**". Thus the influence of lateral torsional buckling will not be considered in the design checks of the selected members.



Mass input of LTB parameters	
List of members for parameter input D1 Selection mode D2 all D3 no check D5 checked D6 sufficient D7 not sufficient Selected : 5 Count : 7	Global Take over sector geometry from analysis Buckling for calculation: do not consider buckling Buckling parameters Buckling My l_{z1} = (no input) Buckling Mz l_{y1} = (no input)

Mass input of buckling parameters

Now we can carry out the design checks for all diagonals. The diagonal D6 does not pass the buckling check, however we can increase its capacity by reducing its buckling length.

	-	6/	80 8,100 8,300	0,308 8,480	0,100 0,100	0,700 0,800 0,800	5300 5300 520	00 1,300 1,40	a 1,500 1,400	Lion Then	1,880 2,808	2,00 2,0	10	
Delete Delete Section Astesid Unterval forces Duterval forces Duterval forces Duterval forces Duterval forces Duterval forces										1963			200%	
,														
	E Cakulate	Calculation met	hod: [Celculate envelop	e of maximum utiliza	itions from all load	8	rectangle 40x80							
7					ombination no.	4 - 54:61+62+63; X=1,	710m.							
y a no.		All Maxim		: Hember no.1 - C Coordinates [11]	Utilization no.	A - 54:G1+G2+G3; X=1, Cross section check X=1, 7 Decisive load: Member r Internal forces: N = 4,63 Compression and bea	710m. 10m; 158,2%; Load: Mer o. 1 - Combination no. 4 5164; M ₂ = 0,003 Mer; 1 Sing moment combin	- 54:61+62+63 M _e = 0,000 kHm; V _e		57630				
	Member check:	All Maxim	d utilization: 158,2% Identification of cross-section	: Hember no.1 - C Coordinates [11]	Utilization no.	A - 54:61 + 62 + 63; X = 1, Cross section check X = 1, 7 Decisive load: Member Internal forces: N = -6, 63 Compression and bea Resistances: N ₀ = -9, 202	710m. Idm; 158,2%; Load: Mer Is.1 - Combination no. 4 Stét: M _y = 0,000 Hén; 1 Sing moment combin dr; M _y ,g = -1,168 Hén 000 = -1,582 > 1 R	- 54:61+62+63 Mg = 0,000 kHn; Vg ation check		57630				
7 Check all	Member dheck:	All Maxim	d utilization: 158,2% Identification of cross-section	: Hember no.1 - C Coordinates [11]	Utilization no.	A - S4451+62+63; X=1, Cross-section check X=1, 7 Internal factors: N = 4,61 Compression and bea Resolutions: Ng = 4,202 (-1,579 + 4,003 + 0 Shear factors: Alg = 3,293 K Resolutions: Alg = 3,593 K	730m. 10m; 138,2%; Laed: Mer n. I - Combination no. 4 5 kH; M ₂ = 6,000 Mer; J Mig M ₂ = 4,000 Mer; J Mig M ₂ = -1,148 Mer 0000 = 1-1,582 > 1 Fri N	- 54:61+62+63 Mg = 0,000 kHn; Vg ation check		57630				

Diagonal D6 check

Therefore we design a longitudinal stiffener in the diagonal's centre which will reduce its buckling length to half. In the **"Buckling"** section of the control tree we adjust the buckling length in the **"Buckling Z"** dialog box.



Buckling Z (Buckling in direction of axis Y)
Buckling check Neglect buckling (buckling prevented) Different sector length for buckling
Sector length for buckling Lz: 1,100 [m]
End conditions
Buckling length
L _{cr} = sector length * factor k
L _{crz} = 1, 100 m

Adjusting buckling length

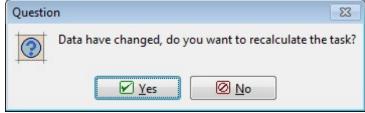
After re-calculating, the diagonal passes the design checks.

FIN EC - FIN 2D Timber [Cr			vel FIN EC v	3 Pilklady/ Tie.f2e	7												
File Edit Massinputs Ing	and the second se																
			0.000		0.495 0.599			0.955 1050	1900	1900 1900	1000 11		1959 1990	1995	1.000 1.000	1.110	
Control Addisecton Delate Te D1 Control Cont							0.00 6.00								2,000 2,000	2093 100% 50% Utilastion	TW
									ectangle 40x8							_	
					of maximum utilizatio			i2+63; X=1,221m.									•]
	BE Add			dentification	Coordinates			n check X=1,221m; 43	,2%; Load: M	ember no. 1 - Com	bination no. 4 - 5	PRG1+G2+G3.					
	E Edit		at	cross-sector	[*]	[%]	Decisive	oad: Henber no. 1 - C	ombination ne	4-5461+62+6	a		8				*
	B Bemave	- 1	Critical m	rmber cut "D6" ·	- sec 1,221	43,2 -	Compress	ces: N = -6,627 kH; N iion and bending m s: Ng = 15,400 kH; M 5 + -0,004 + 0,000 [-	oment com	ination check:		с «у = 0,000 к	a				
	Re Printing						Shear for	ves check: : Va = 3,958 kN	- Leivar L.c.	L P MAR							r
Check all							057633										
Structure preview						-	Henber sle Section ok	nderness: 95,3									
	1																gk 🛛 🖉 gencel
EN 1995-1-1/Caech Rep.																	

Checked design members

Now all design members and groups in the structure are checked, therefore we can exit the design module by clicking "**OK**". Program 2D has recognized that some members were changed. Because the stiffness of individual members changed and thus different internal forces can act on some of the them, it is necessary to re-calculate the structure. The program automatically offers this option. If we select "**No**", the program will erase results and return to the pre-processor. In our example, we select "**Yes**" after which the structure is re-calculated.





Question on re-calculating structure

After finishing the calculation, the program updates distribution of internal forces and deformations and erases the results in the design module. Therefore we run the design module again and make use of the command "**Check All**" located in the bottom part of the control tree. We do not have to re-define all parameters as they are saved after previous definition. The module will carry out design checks for all members and inform us about the results.

Informa	ation	23
	Item check co All items pass.	
	<u>Р</u> <u>о</u> к	

Members' design checks results

After returning to the program 2D, members which passed the checks are marked in green and the members which did not in red. In our case, all members are marked in green; therefore the structure's design is completed.