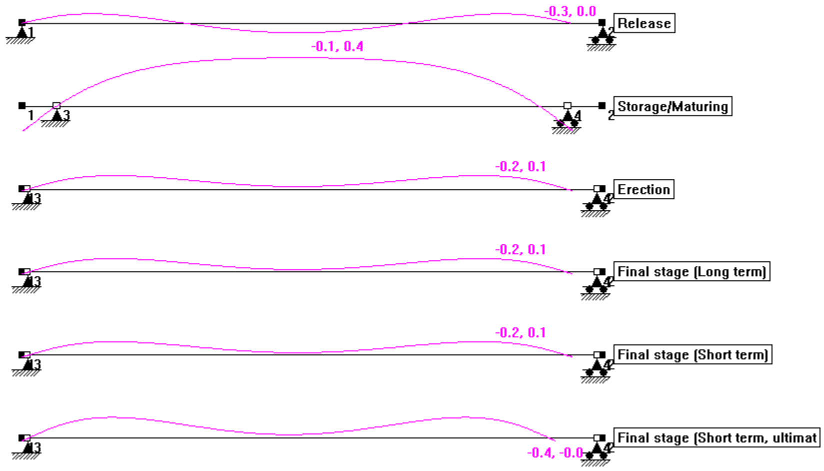
PRE-Stress

STRUSOFT

EXAMPLES PRE-STRESS 6.4



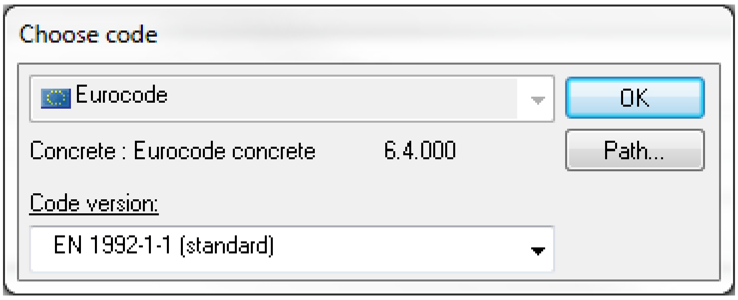
STEP BY STEP EXAMPLES

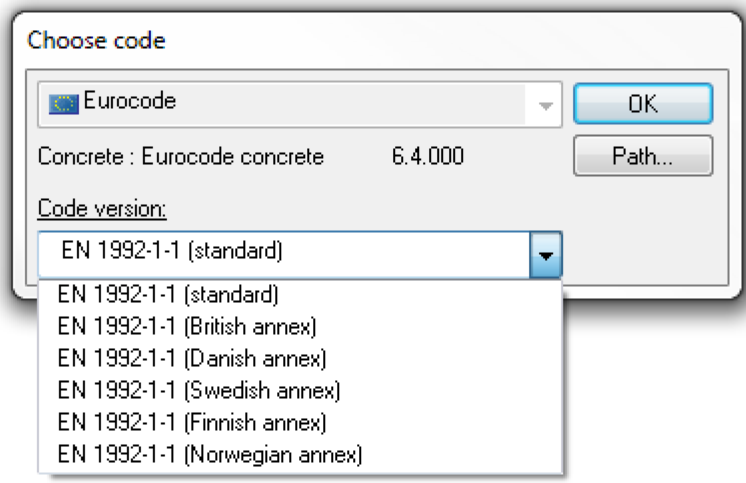
6.04.005 - 2014-07-018

CONTENTS

1 BASIC CONCEPT

1.1 CODES





When starting a new file, a question about selecting the code will appear. The possible codes to select depending on your license are:

Eurocode – EN 1992-1-1:2004 (Standard)

Eurocode – EN 1992-1-1:2004 (British NA)

Eurocode – EN 1992-1-1:2004 (Danish NA)

Eurocode – EN 1992-1-1:2004 (Swedish NA)

Eurocode – EN 1992-1-1:2004 (Finnish NA)

Eurocode – EN 1992-1-1:2004 (Norwegian NA)

Old national codes are only available up to version 6.3.

In the program there is an option to calculate shear and fire with EN1168 (Precast concrete products – Hollow core slabs).

1.2 LAYOUT OF THE PROGRAM

PRE-Stress is based on the StruSoft Frame Analysis-engine to make calculations. The Frame Analysis core is a 2D-FEM equation solver. The two programs have many similarities but there are many differences as well due to the prestressing forces/effects.

1.3 LIMITATIONS IN THE CURRENT VERSION

It is not possible to make sloped TT-slabs (STT), this feature will be added in a future update.

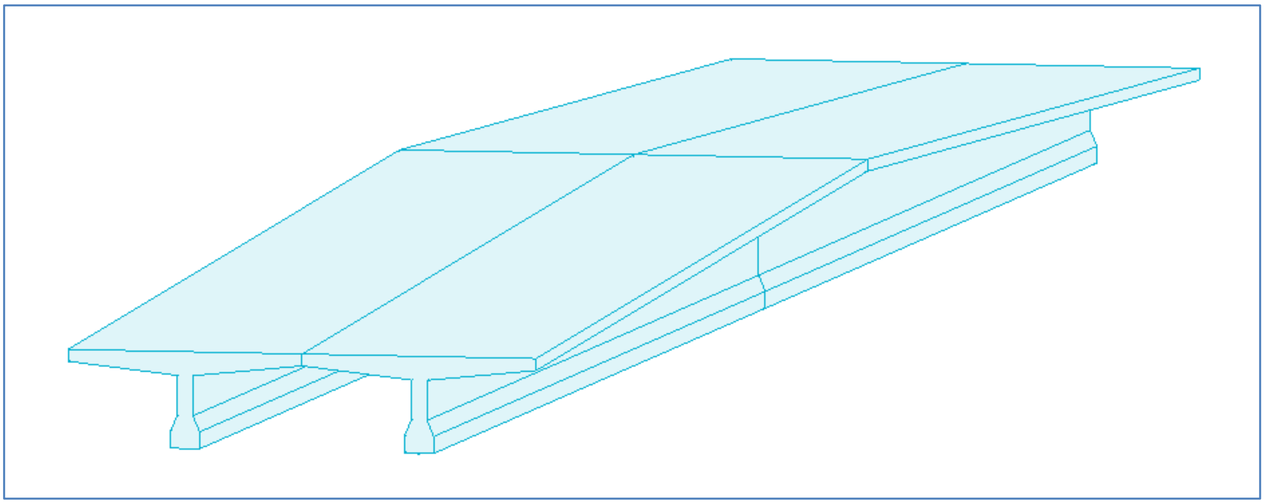


Figure 1-1 Sloped TT-slab (STT-slab), picture from StruSoft FEM-Design

One limitation with hollow core slabs is that they can only have cores of equal size and geometry.

2 EXAMPLES

2.1 MODELLING OF A TT/F 240/54 WITH NON-STRUCTURAL TOPPING

The first example is a 12m TT-slab. It will be calculated with Eurocode with Swedish annex.

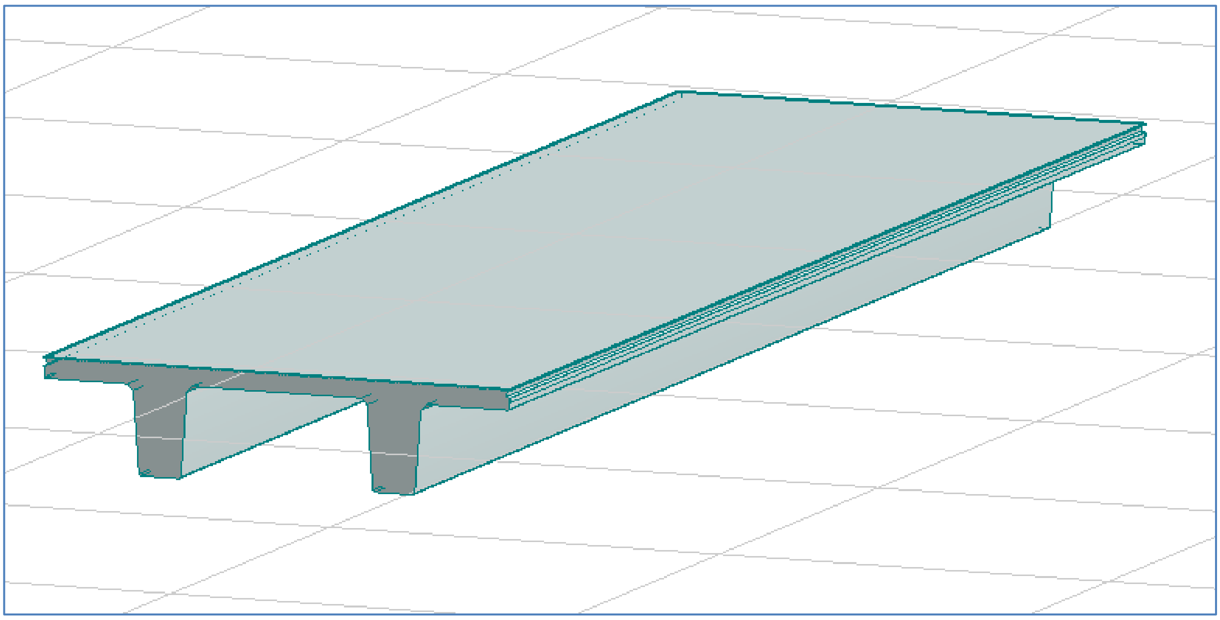


Figure 2-1 3D-view of a TT/F 240/54 (FEM-Design)

2.1.1 INPUT GEOMETRY

The width of the slab is 2.4m, but version 6.4 currently only supports single T-beams, so the calculations will focus on one of the legs. After selecting the code the project manager-window pops up. In the Input-tab to the right the name, length and cross-section can be entered, along with structural topping, if present.

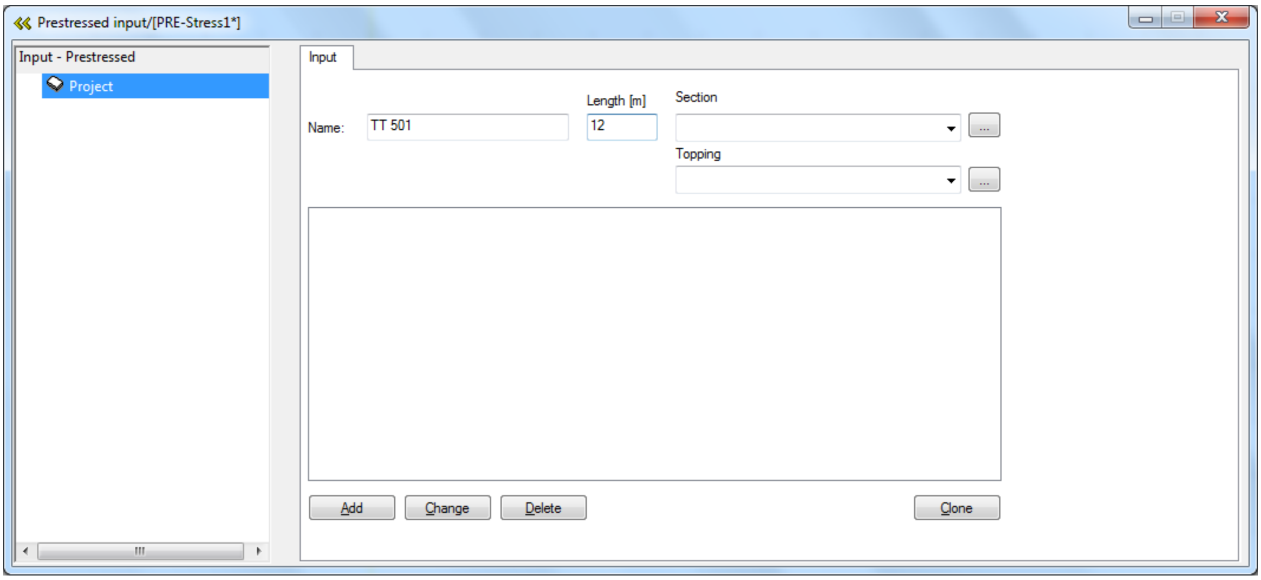


Figure 2-2 Input of elements

In the Name-box an identifier for the element should be entered, in this example we will use *TT 501*. The length of the slab is 12m. When entering the section of the slab it is possible to select already used sections in the dropdown-box, if none are present then click on the [...]-box to the right of the dropdown-box.

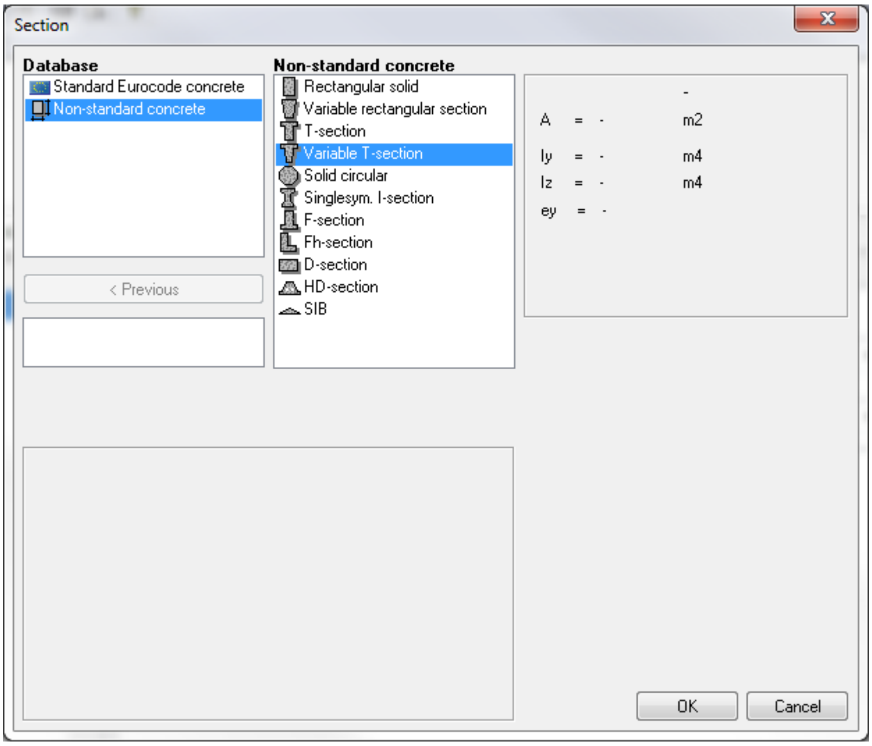


Figure 2-3 Crossection database, overview

Select Non-standard concrete to the left and Variable T-section to the right. (In order to use standard profiles from a few companies that want their cross-sections in the database, then we could use the Standard Eurocode concrete-database and directly select the section.)

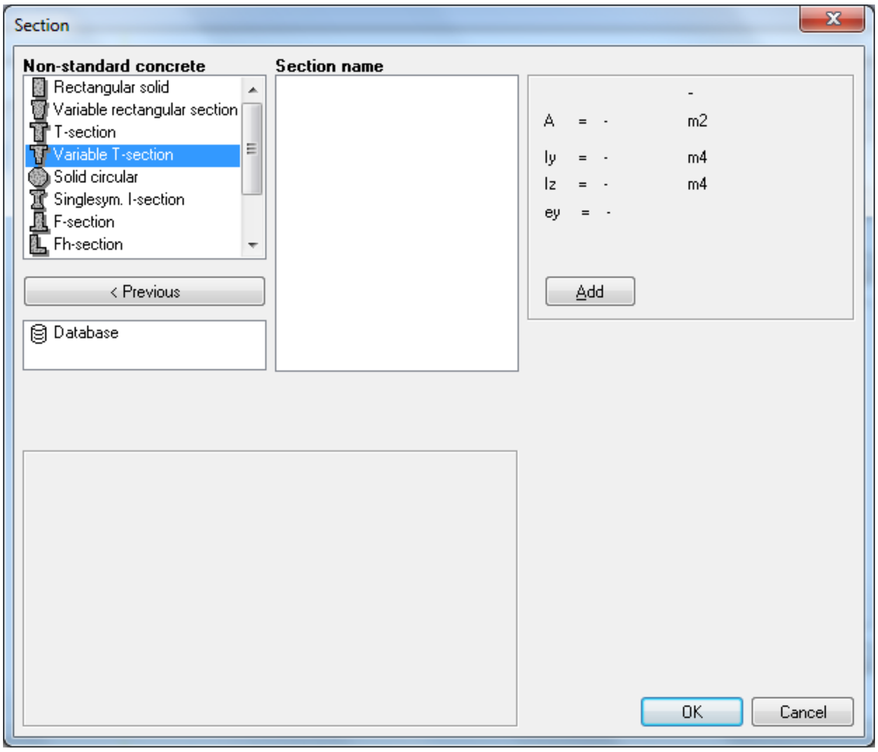


Figure 2-4 Crossection database

If this is the first time entering variable T-sections, then this sub-library is most likely empty. The more section you use the longer the selection of possible sections will be. To add a new section press [Add].

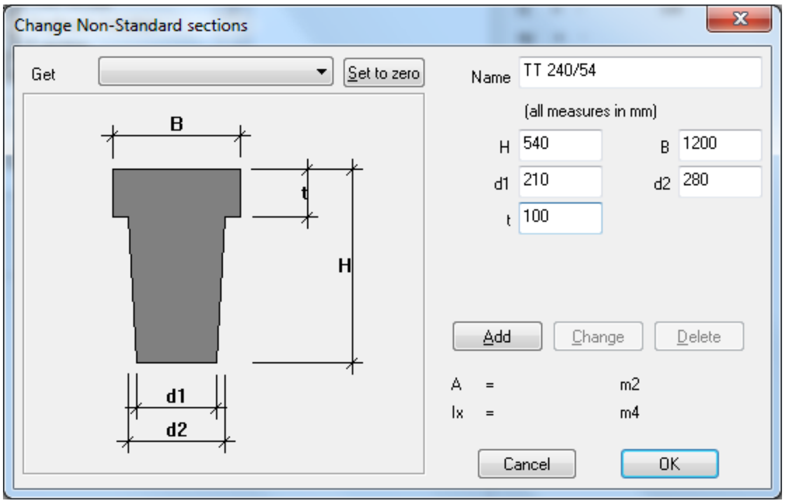


Figure 2-5 Entering a new variable T-section.

Enter the values according to Figure 2-5 and press [Add] and then [OK].

The cross section name should now be visible in the list of available cross sections. Select *TT 240/54* in the list and press [OK]

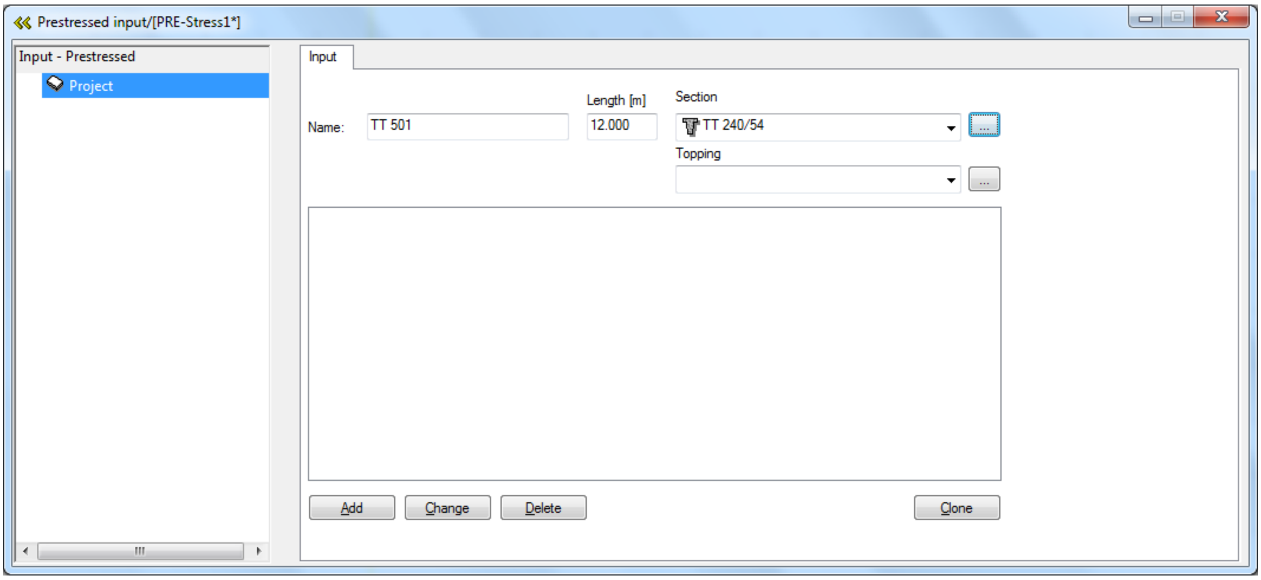


Figure 2-6 Project overview

In this example we will not use any structural topping so leave this box empty. To finalize the input of the geometry and add the element to the project-list press [Add] as shown in Figure 2-6.

Now a window will appear (Figure 2-7) where different kinds of support conditions and automatic calculation of dead loads can be set.

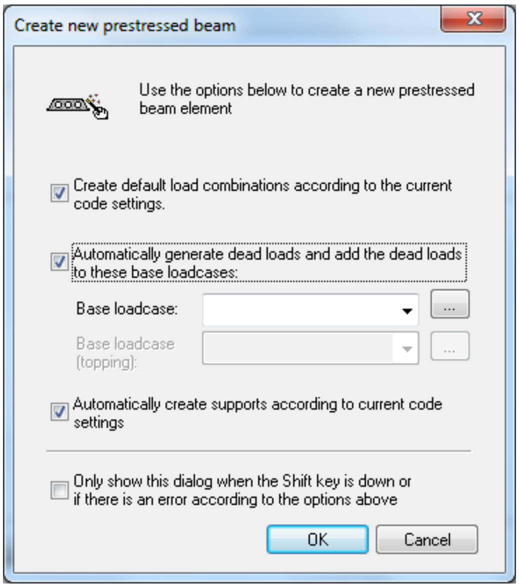


Figure 2-7 Create a new prestressed beam

We want to generate the dead loads automatically, select the [...]-button to the right of the Base load case-dropdown box.

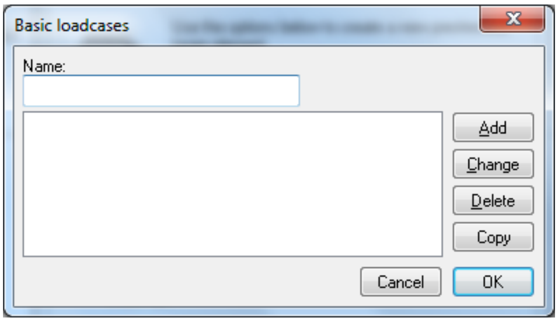


Figure 2-8 Basic load cases without entered loads

Now a dialogue box will appear (Figure 2-8) to define the basic load cases. After entering the dead load case it is now also possible to enter all the other ordinary loads acting on the element. Enter the names and add them. Enter all the loads according to Figure 2-9. When ready, select the load case where the dead load will be automatically generated in: select *Dead load*. If you should miss to select the right load case then you have the possibility to change it afterwards (Figure 2-10).

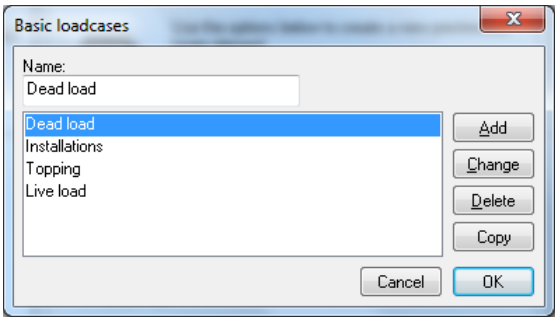


Figure 2-9 Entering basic loadcases

Since we haven't defined any structural topping the *Base load case (topping)* is inactive, Figure 2-10. If you want other support options then it is better to go in and change existing supports, rather than defining new. Press [OK] to create the beam, and the supports for the beam.

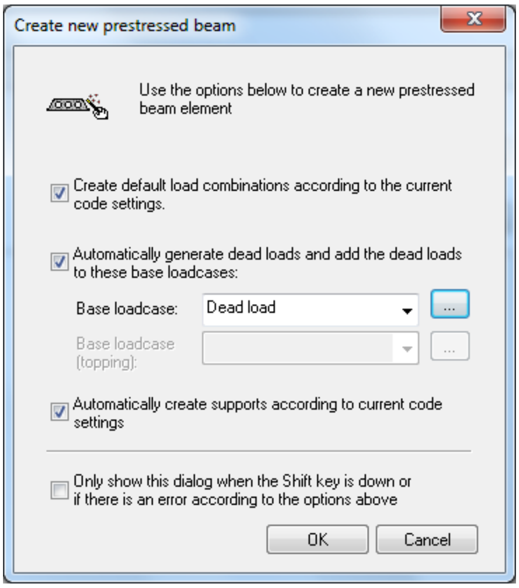


Figure 2-10 Basic load case for dead load entered

Now the beam is created with support, nine load combinations and some assorted information about the element. It is now possible to continue entering additional elements with various spans, geometries and load cases. With one element in the project manager the view should be similar to Figure 2-11.

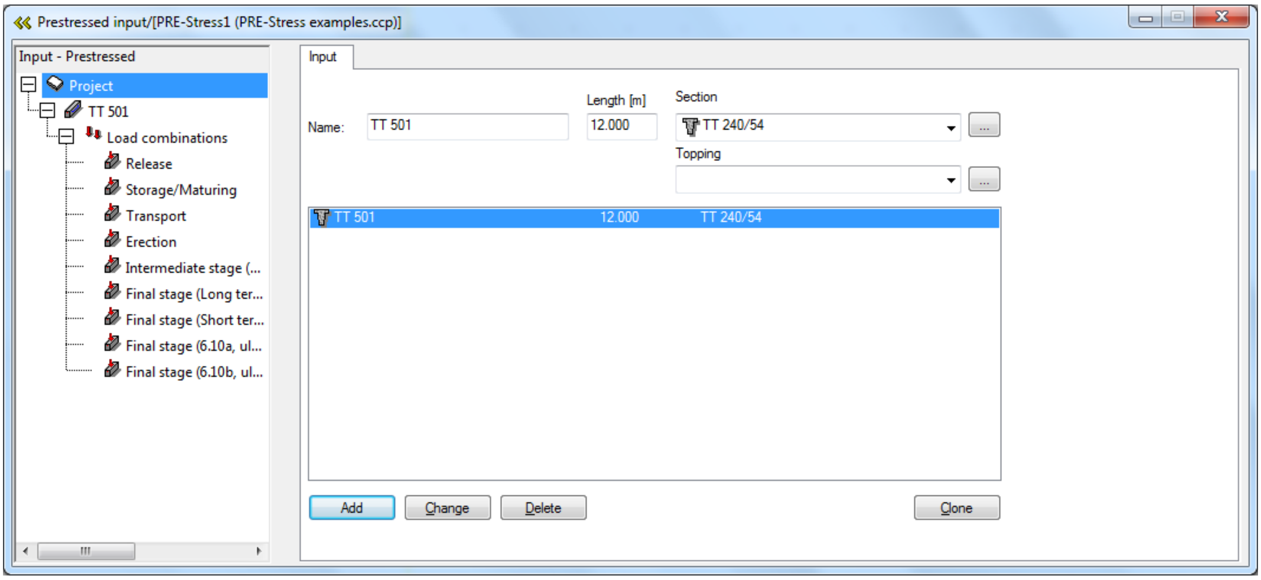


Figure 2-11 Project overview with one element

To get an overview of the element and its load combinations, select the name (*TT 501*) and then it is possible to graphically show the beam with all the combinations. In some cases the project overview window may cover. To make the project window smaller, click on << in the upper left corner. To restore the window again, click on >> instead. To open and close the project window, press the beam as shown in Figure 2-12.

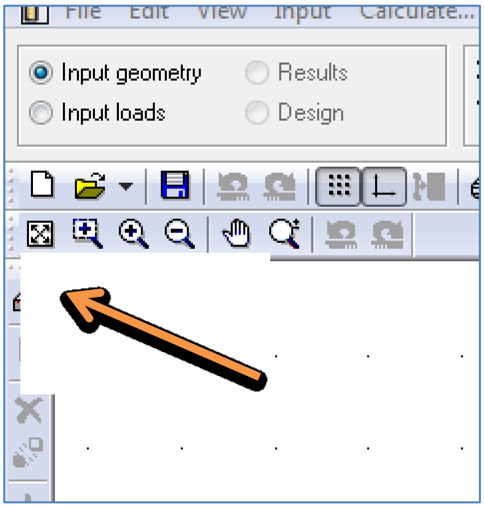


Figure 2-12 Open/close the project window

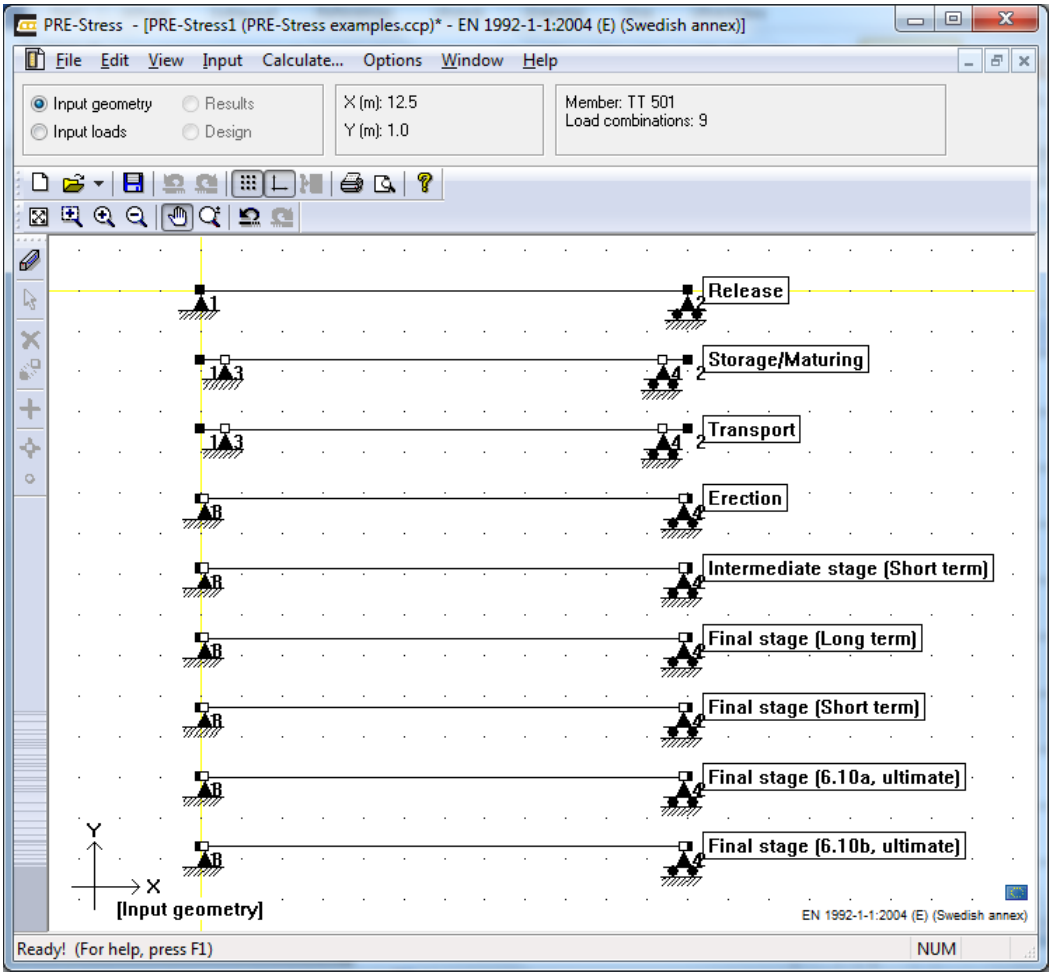


Figure 2-13 Overview of the load combinitions

To add more load combinations look at 2.3 Hollowcore slab (coming soon) or 2.4 How to add a fire loadcombination.

Open up the project overview and select *TT 501* to the left. On the right side the input form for the reinforcement will now be visible (Figure 2-15).

Before defining reinforcement we need to set the different exposure classes in the later stages and ultimate limit stages, this to get the correct cover from the code. Go through the load combinations release to the final stages to verify the correct exposure class, life class and if possible, the concrete class as well, according to Figure 2-14. For this example we use either X0 or XC1.

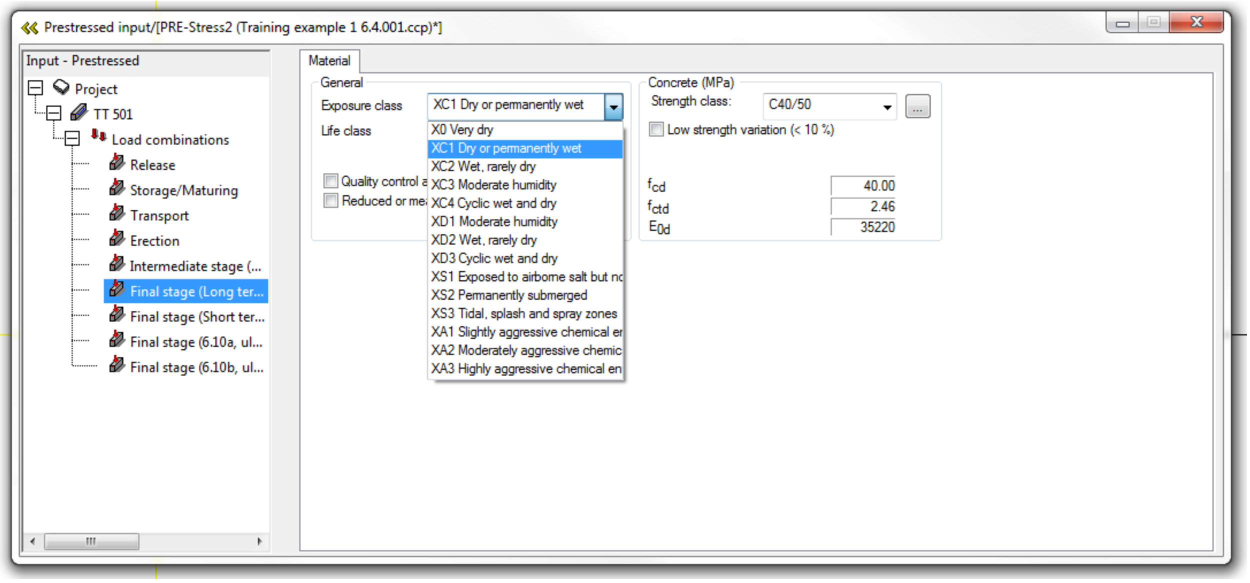
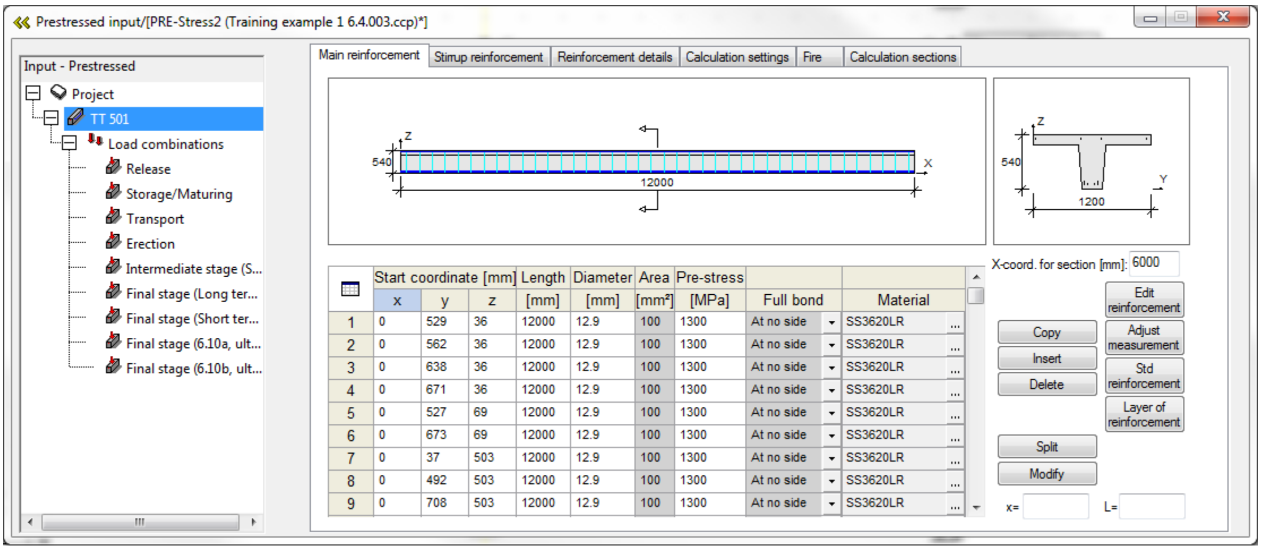


Figure 2-14 Exposure classes



	Start coordinate [mm] x	y	z	Length [mm]	Diameter [mm]	Area [mm²]	Pre-stress [MPa]	Full bond	Material
1	0	529	36	12000	12.9	100	1300	At no side	SS3620LR
2	0	562	36	12000	12.9	100	1300	At no side	SS3620LR
3	0	638	36	12000	12.9	100	1300	At no side	SS3620LR
4	0	671	36	12000	12.9	100	1300	At no side	SS3620LR
5	0	527	69	12000	12.9	100	1300	At no side	SS3620LR
6	0	673	69	12000	12.9	100	1300	At no side	SS3620LR
7	0	37	503	12000	12.9	100	1300	At no side	SS3620LR
8	0	492	503	12000	12.9	100	1300	At no side	SS3620LR
9	0	708	503	12000	12.9	100	1300	At no side	SS3620LR

Figure 2-15 Reinforcement view

Click [Std reinforcement] and select *Reinforcement Proposal 5*, finish with [OK].

Now the reinforcement list will be filled with the predefined reinforcement position, these can be modified in the table, and more reinforcement wires/bars can be added below. The start coordinate is the left end of the wire/bar and with length it is possible to define the position along the element.

To control the prestressing stress, enter the value in the Pre-stress column and Material governs the material and possible wire/bar diameters. Full bond is a function to solve reinforcement with a variable location in the cross section (sloped I-beams and the function will also be used for sloped TT-elements in the future). It is not possible to change it for variable T-sections, hence the default value is set to *At no side*.

Next tab, Stirrup reinforcement, contains information about the stirrups in the element. Enter ø8 s300 along the whole element, according to Figure 2-16. Be sure to have the group(s) fill the whole element, do not start 40mm in from the end, the program will see that small part as unreinforced and might fail the shear calculation.

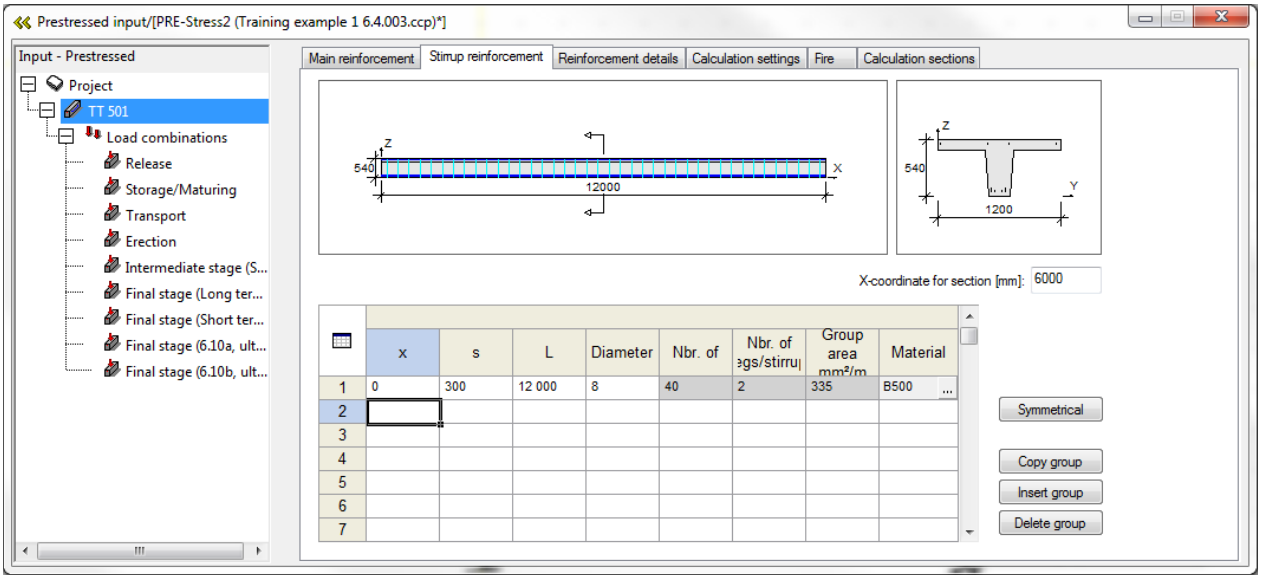


Figure 2-16 Stirrup reinforcement

The next two tabs, Reinforcement details (Figure 2-17) and Calculation settings (Figure 2-18), contains settings for code checks for the reinforcement and the calculations.

According to Swedish NA, 6.2.3 (2), can cot(θ) = 3,0 be used for prestressed members (Figure 2-18), check this setting if it is possible to use other values than the normal (1 ≤ cot(θ) ≤ 2.5) for your current code.

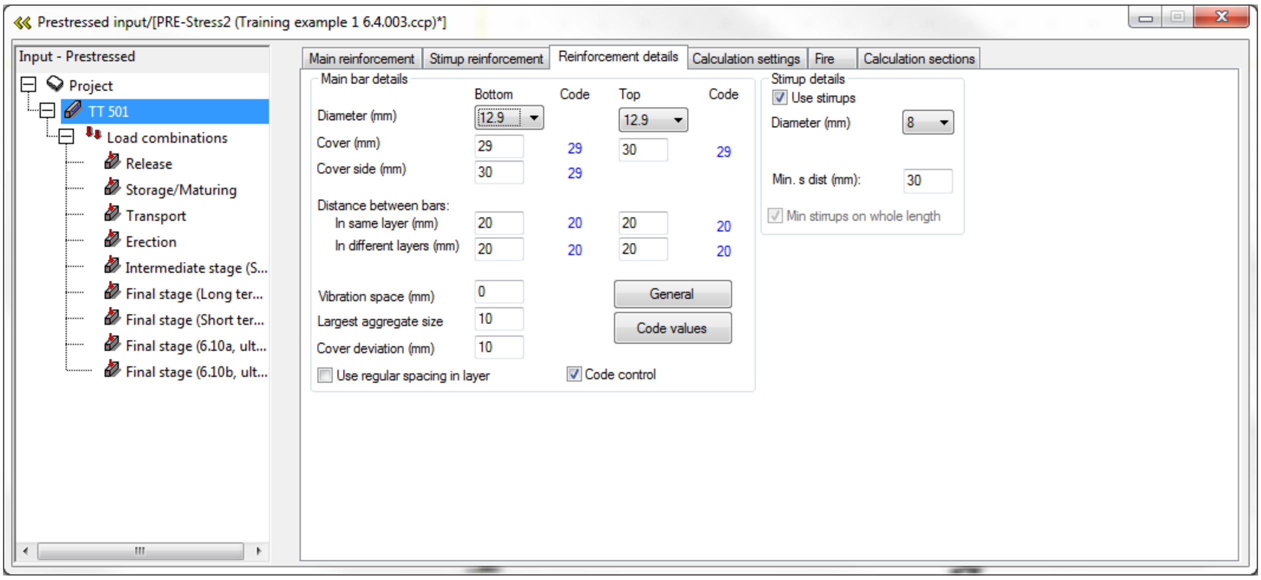


Figure 2-17 Code check concerning the reinforcement against the current code.

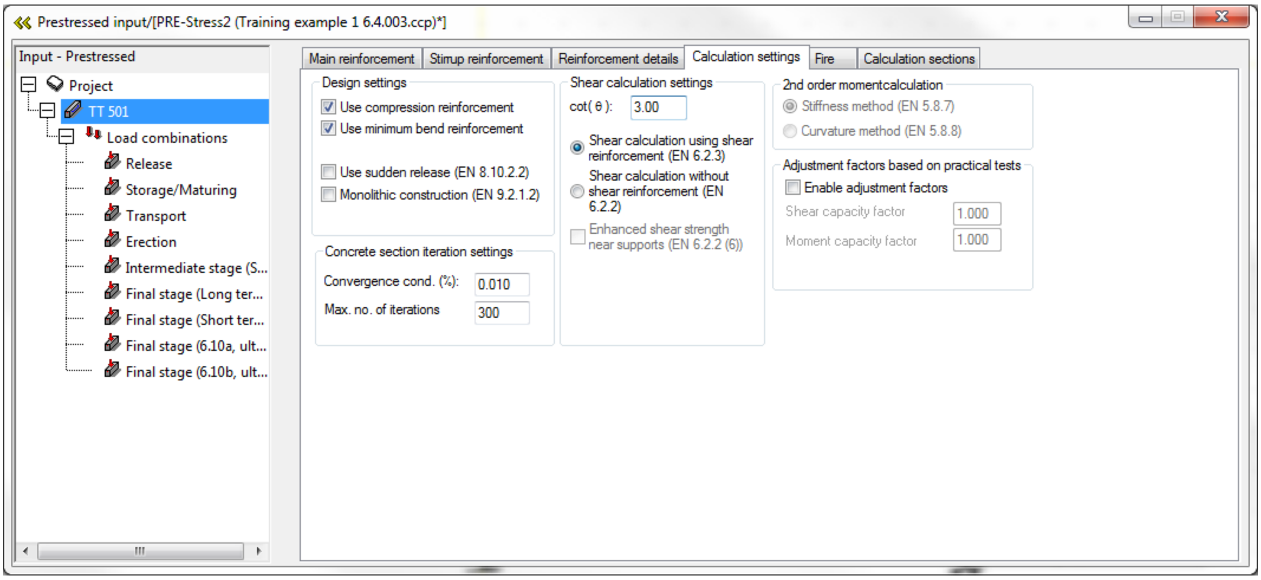


Figure 2-18 Calculation settings

In the last tab, Calculation sections, a good spacing between the sections should be used. As a rule of thumb, use a value slightly less than the cross section height. In the current example 0.5[m] is used then press [s]-button. The result should look like Figure 2-19. This setting is auto generated by the program, but it is always a good idea to verify that there are calculation sections spread evenly over the whole element.

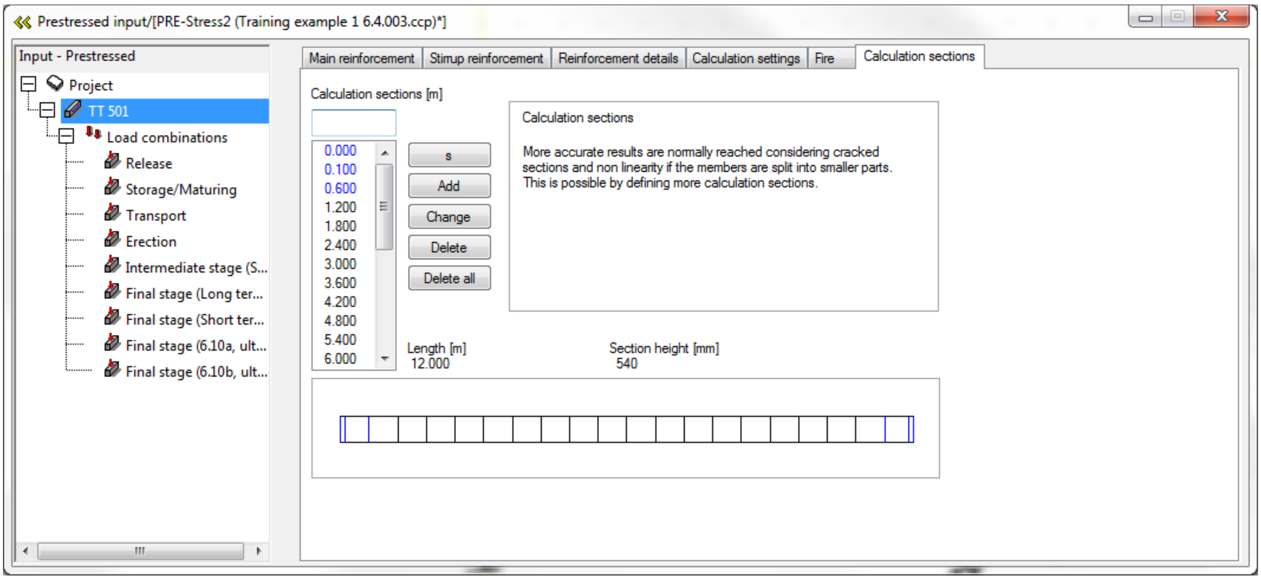


Figure 2-19 Calculation sections

2.1.2 INPUT LOADS

Now with the geometry and calculation settings done, the next step will be to combine the loads. Select *Load combinations* to the left. To the right two tabs will appear *Load combinations* and *Long term parameters*. In the *Load combinations*-tab combine the loads to the very right according to Figure 2-20.

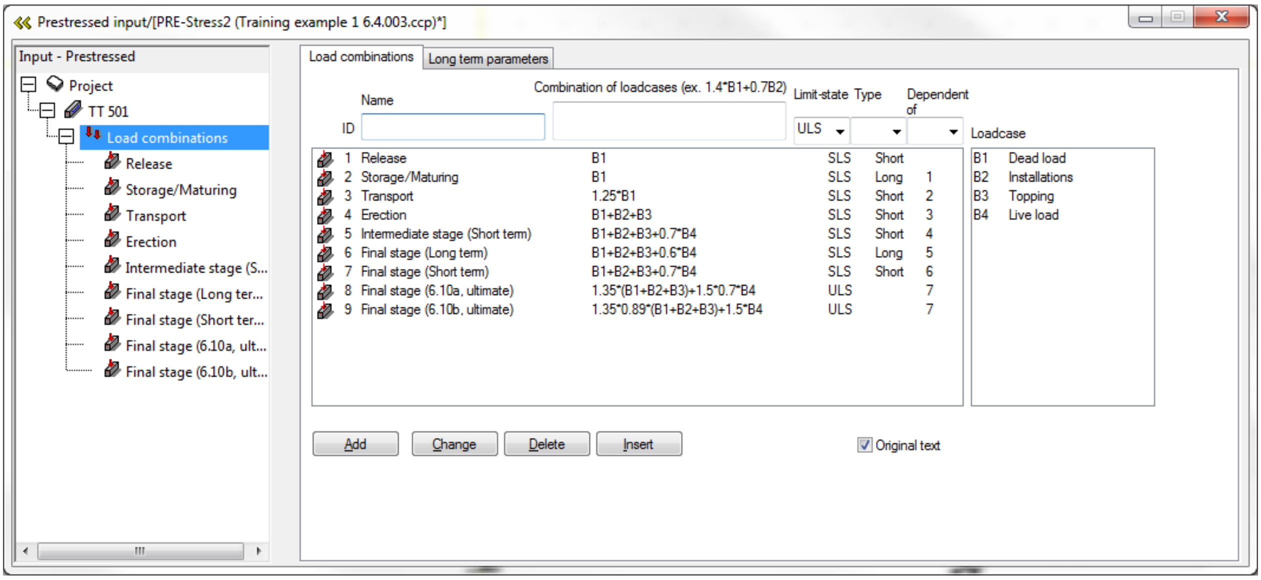


Figure 2-20 Load combinations

As for the long term parameters a creep value of 2 shall be divided in the different fields. If 25% of the creep is introduced at the Storage/maturing and 75% at the Final stage (Long term) then the creep should be entered as 2*0.25 = 0.5 and 2*0.75 = 1.5. The same goes for the shrinkage, but here we divide the 0.2‰ equal between the two long term cases.

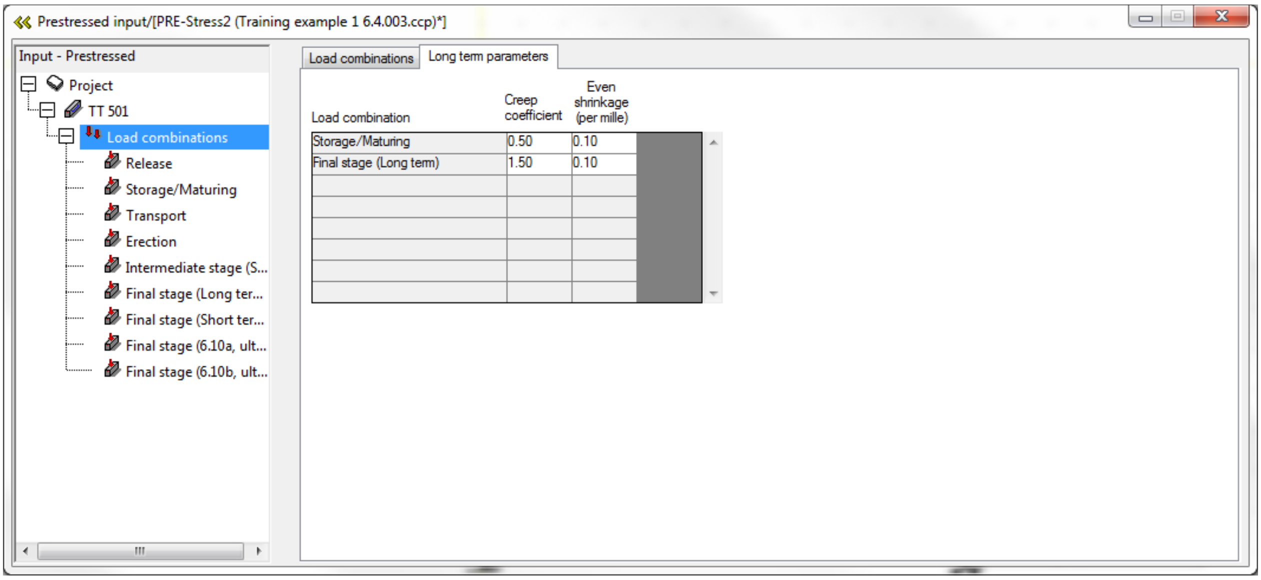


Figure 2-21 Long-term parameters

Next step is to check the supporting conditions and the material properties for each specific load combination. It is in this stage that it will be possible to change the supports etc. Select Release to get the option to change the material properties and supports at the release stage in the factory.

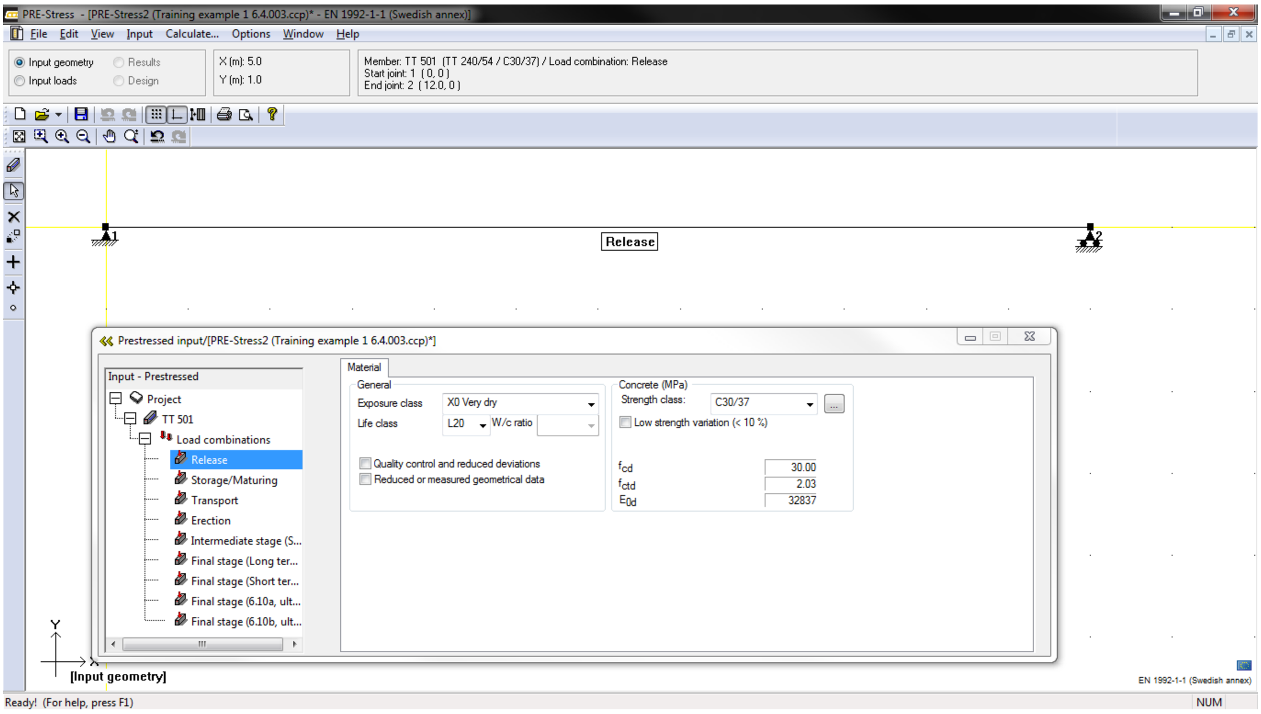


Figure 2-22 Release

In order to change the support conditions, double click on support 1 or 2, or node 3 or 4, depending on if the support isn't located at the end. It can be a little bit tricky to get the support, but with a little bit of experience and a lot of technique then it will be possible. When it comes to the material of the element then the default option is that there is a bit of increase in the concrete strength from release to the final stages.

Continue to *Input loads* (Figure 2-23). Here is a overview of each load case (**1**) and the input of loads are inactive in the overview (**2**). To be able to enter loads in the different load combinations a load combination in the project overview must be selected.

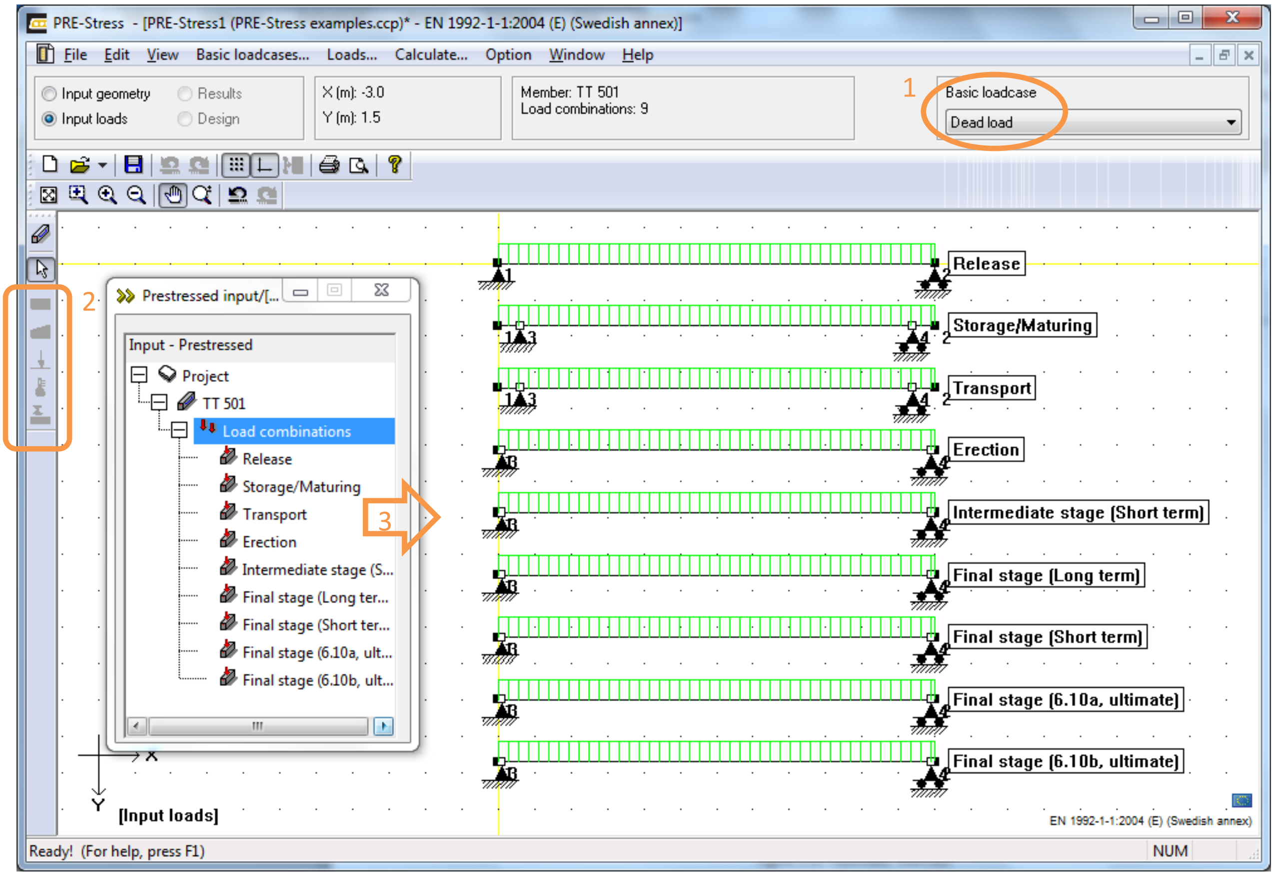


Figure 2-23 Inputloads, overview

The reason for the load combination *Intermediate stage* is that the cracking would be as fully developed before going into the long term stage. That gives a more correct picture of the deflection and cracking in the long term checks.

The dead load is defined automatically, so there is no need to define it again manually.

Enter an installation load of $0.5kN/m^2 * 1.2m =$ **0.6 kN/m** in the following load combinations:

- Erection
- Intermediate stage (Short term)
- Final stage (Long term)
- Final stage (Short term)
- Final stage (6.10a, ultimate)
- Final stage (6.10b, ultimate)

Enter a 50mm topping as a permanent load. $50mm * 25kN/m^3 * 1.2m =$ **1.5 kN/m** in the following load combinations:

- Erection
- Intermediate stage (Short term)
- Final stage (Long term)
- Final stage (Short term)
- Final stage (6.10a, ultimate)
- Final stage (6.10b, ultimate)

And finally the live load 8 kN/m^2 * 1.2m = **9.6 kN/m**. It should be applied as a characteristic load on the intermediate and final load combinations.

- Intermediate stage (Short term)
- Final stage (Long term)
- Final stage (Short term)
- Final stage (6.10a, ultimate)
- Final stage (6.10b, ultimate)

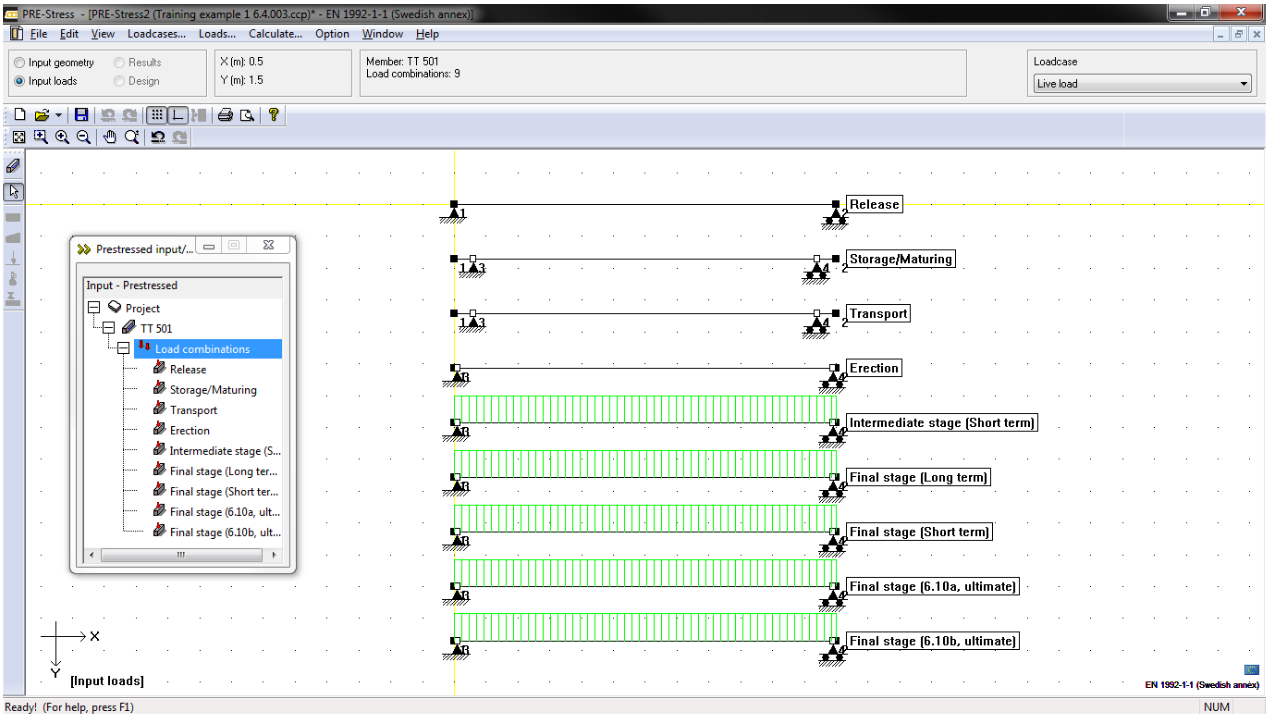


Figure 2-24 Loads defined

2.1.3 CALCULATION

To start the calculation, select *Calculate…* in the menu bar. Unselect *Second order analysis* in the calculation since it will not be relevant for calculation of single elements (see Figure 2-25).

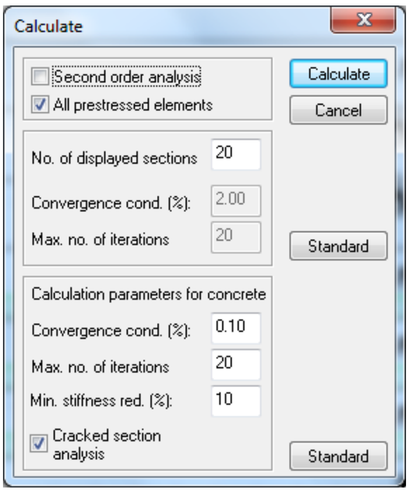


Figure 2-25 Calculate… [Calculate]

2.1.4 PRESENTING THE RESULTS

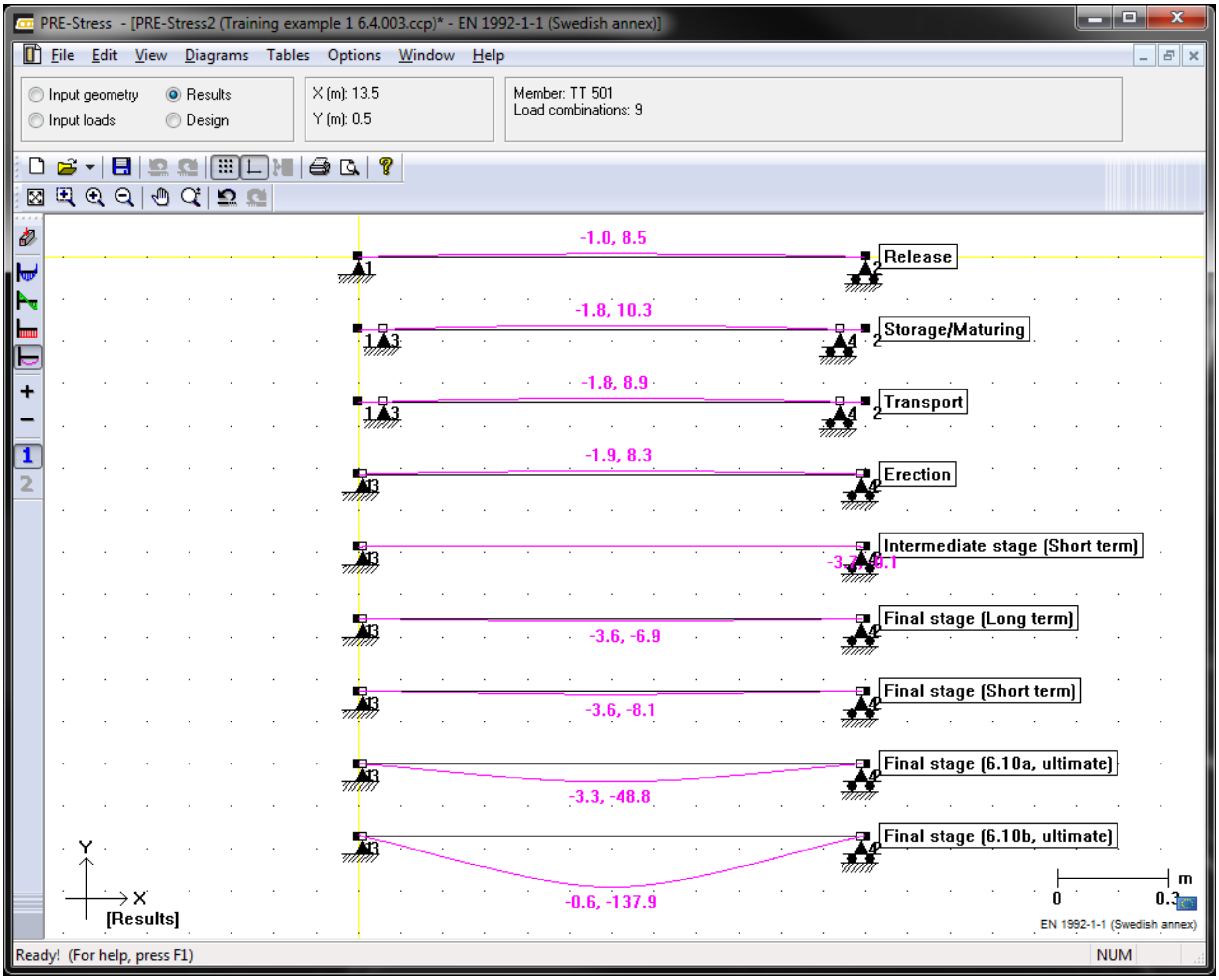


Figure 2-26 Deflections

The deflections are shown as two values, the first is the shortening of the element the second value is camber (pos.)/deflection (neg.) ie. deformation in (x,y). The program also presents values for deflection in ultimate limit state, but these can be ignored since deflections are not relevant and that the program uses material strengths with safety factors. After checking support reactions, moment curves and equilibrium, then continue to (select) the Design-part of the program. Select *Calculate* in the menu bar.

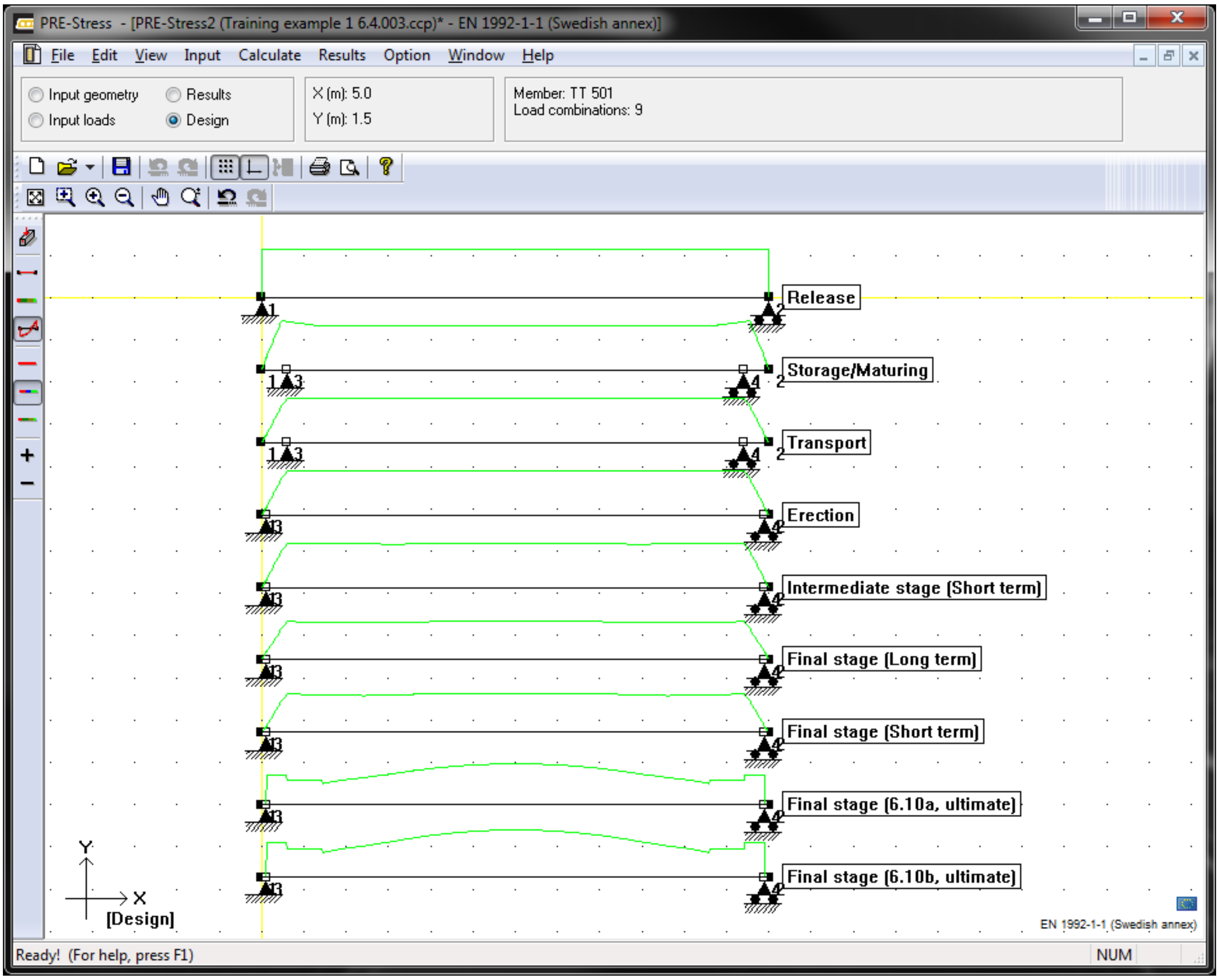


Figure 2-27 Design, utilization

With a quick overview of the utilizations (Figure 2-27) it is possible to get an immediate idea if the element is ok, or if the utilization is too high the curve turns red at those sections.

The results that will be presented more thorough can be found under the menu item *Results*,Figure 2-28.

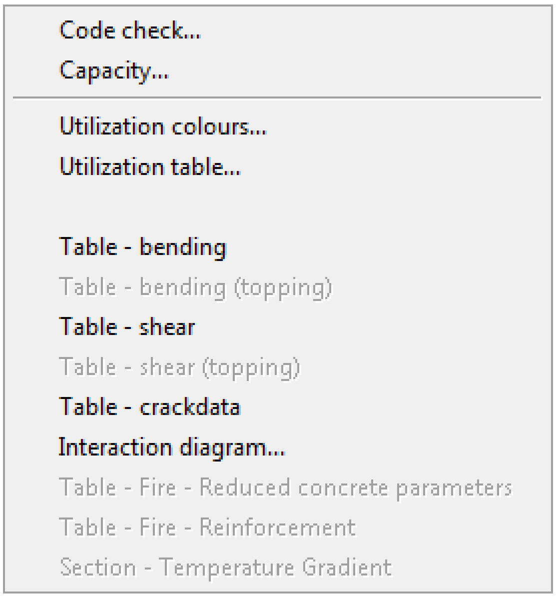


Figure 2-28 Design, Result menu

To check the different load combinations more in detail, select *Results* > Code check… (Figure 2-29)

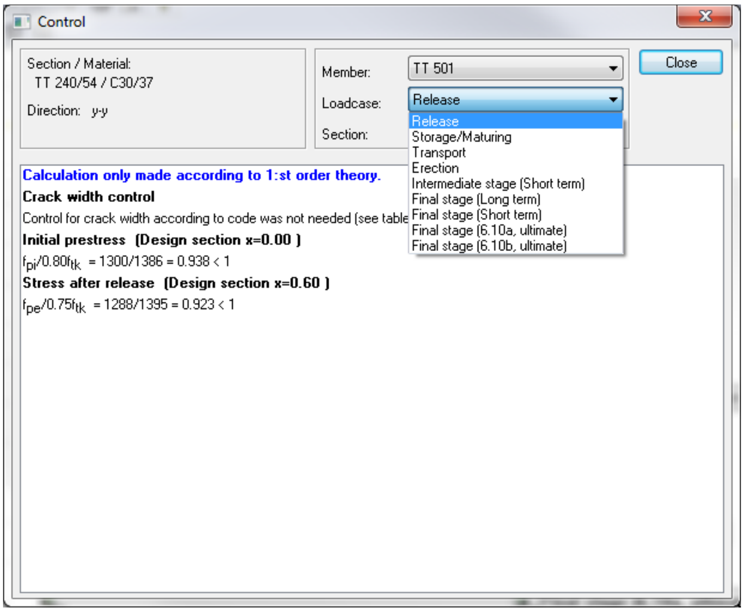


Figure 2-29 Results, Code check…

It is possible to flip through the different load combinations to get a numerical verification of the element. Depending on different types of load combinations (SLS or ULS), the program makes different checks at different stages.

- For serviceability limit states the program presents checks of stresses and cracks, there is also a possibility to check deflection criteria.
- For ultimate limit states the program present checks containing moment and shear capacities. It does check the most critical section, not always necessary mid-span or at the supports.

Next item in the list is *Capacity*. In this list the moment and shear capacities are shown along different sections of the element.

Figure 2-30 and Figure 2-31 shows values for bending moment capacity. These tables can be found from the *Table – Bending*-option from Figure 2-28.

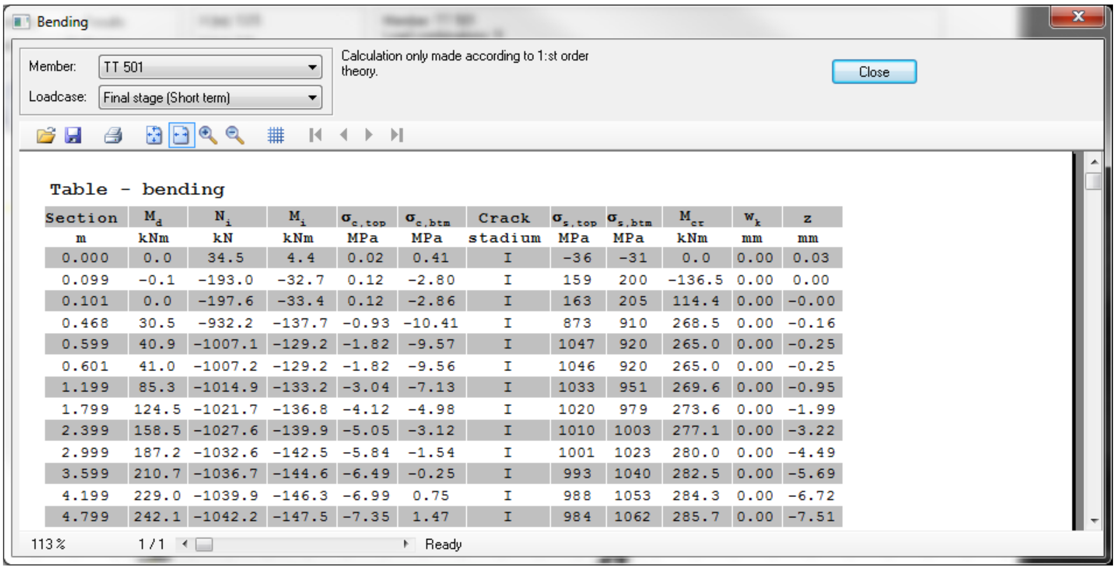


Table - bending

Section	M_d	N_i	M_i	$\sigma_{c,top}$	$\sigma_{c,btm}$	Crack	$\sigma_{s,top}$	$\sigma_{s,btm}$	M_{cr}	w_k	z
m	kNm	kN	kNm	MPa	MPa	stadium	MPa	MPa	kNm	mm	mm
0.000	0.0	34.5	4.4	0.02	0.41	I	-36	-31	0.0	0.00	0.03
0.099	-0.1	-193.0	-32.7	0.12	-2.80	I	159	200	-136.5	0.00	0.00
0.101	0.0	-197.6	-33.4	0.12	-2.86	I	163	205	114.4	0.00	-0.00
0.468	30.5	-932.2	-137.7	-0.93	-10.41	I	873	910	268.5	0.00	-0.16
0.599	40.9	-1007.1	-129.2	-1.82	-9.57	I	1047	920	265.0	0.00	-0.25
0.601	41.0	-1007.2	-129.2	-1.82	-9.56	I	1046	920	265.0	0.00	-0.25
1.199	85.3	-1014.9	-133.2	-3.04	-7.13	I	1033	951	269.6	0.00	-0.95
1.799	124.5	-1021.7	-136.8	-4.12	-4.98	I	1020	979	273.6	0.00	-1.99
2.399	158.5	-1027.6	-139.9	-5.05	-3.12	I	1010	1003	277.1	0.00	-3.22
2.999	187.2	-1032.6	-142.5	-5.84	-1.54	I	1001	1023	280.0	0.00	-4.49
3.599	210.7	-1036.7	-144.6	-6.49	-0.25	I	993	1040	282.5	0.00	-5.69
4.199	229.0	-1039.9	-146.3	-6.99	0.75	I	988	1053	284.3	0.00	-6.72
4.799	242.1	-1042.2	-147.5	-7.35	1.47	I	984	1062	285.7	0.00	-7.51

Figure 2-30 Table Bending, serviceability limit state

The different columns in the serviceability limit state means:

- Section [m]: Position of calculation sections
- M_d [kNm]: Moment due to loads
- N_i [kN]: Prestressing force
- M_i [kNm]: The effect of the moment due to the prestressing force on the cross section
- $\sigma_{c,top}$ [MPa]: Concrete stress in the top of the cross section (shows dep. of cracking stadium)
- $\sigma_{c,btm}$ [MPa]: Concrete stress in the bottom of the cross section (shows dep. of cracking stadium)
- Crack stadium: Stadium I – uncracked, Stadium II - cracked
- $\sigma_{s,top}$ [MPa]: Steel stress in the top of the cross section
- $\sigma_{s,btm}$ [MPa]: Steel stress in the bottom of the cross section
- M_{cr} [kNm]: Moment when the cross section cracks
- w_k [mm]: Crack width
- z [mm]: Deformation – deflection (negative values) / camber (positive values)

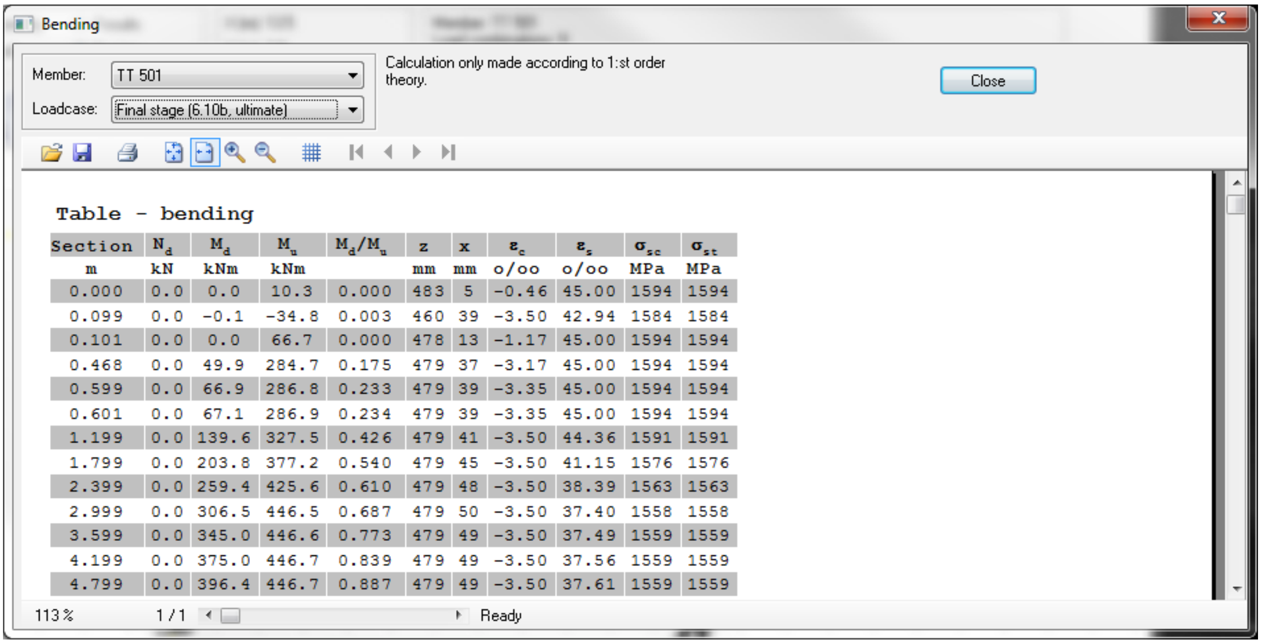


Table - bending

Section	N_d	M_d	M_u	M_d/M_u	z	x	ε_c	ε_s	σ_{sc}	σ_{st}
m	kN	kNm	kNm		mm	mm	o/oo	o/oo	MPa	MPa
0.000	0.0	0.0	10.3	0.000	483	5	-0.46	45.00	1594	1594
0.099	0.0	-0.1	-34.8	0.003	460	39	-3.50	42.94	1584	1584
0.101	0.0	0.0	66.7	0.000	478	13	-1.17	45.00	1594	1594
0.468	0.0	49.9	284.7	0.175	479	37	-3.17	45.00	1594	1594
0.599	0.0	66.9	286.8	0.233	479	39	-3.35	45.00	1594	1594
0.601	0.0	67.1	286.9	0.234	479	39	-3.35	45.00	1594	1594
1.199	0.0	139.6	327.5	0.426	479	41	-3.50	44.36	1591	1591
1.799	0.0	203.8	377.2	0.540	479	45	-3.50	41.15	1576	1576
2.399	0.0	259.4	425.6	0.610	479	48	-3.50	38.39	1563	1563
2.999	0.0	306.5	446.5	0.687	479	50	-3.50	37.40	1558	1558
3.599	0.0	345.0	446.6	0.773	479	49	-3.50	37.49	1559	1559
4.199	0.0	375.0	446.7	0.839	479	49	-3.50	37.56	1559	1559
4.799	0.0	396.4	446.7	0.887	479	49	-3.50	37.61	1559	1559

Figure 2-31 Table Bending, ultimate limit state

...and the equivalent for ultimate limit states:

- Section [m]: Position of calculation sections
- N_d [kN]: Normal force due to imposed loads, does not consider prestressing.
- M_d [kNm]: Moment due to loads
- M_u [kNm]: Design moment
- M_d/M_u: Utilization with regards to the moments
- z [mm]: Interior lever
- x [mm]: Height of compressed zone
- ε_c [‰]: Concrete strain
- ε_s [‰]: Steel strain
- σ_{sc} [MPa]: Equivalent steel stress for steel in compression
- σ_{st} [MPa]: Equivalent steel stress for steel in tension

There are a few results in the Results-menu that contains"… (topping)", these will be active only if you have a structural topping defined.

Next table is Shear, it can only be checked in ultimate limit states, Figure 2-32.

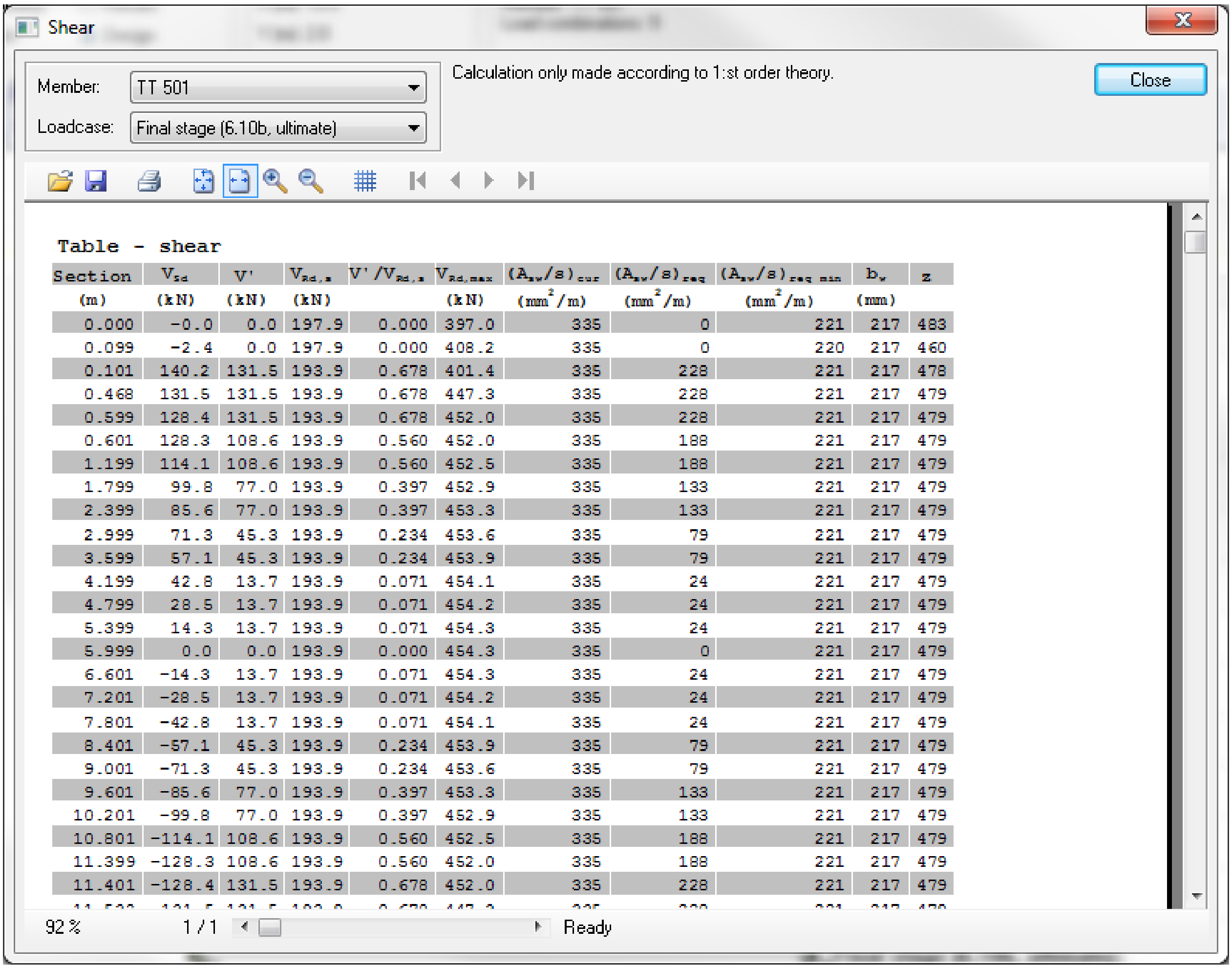


Table - shear

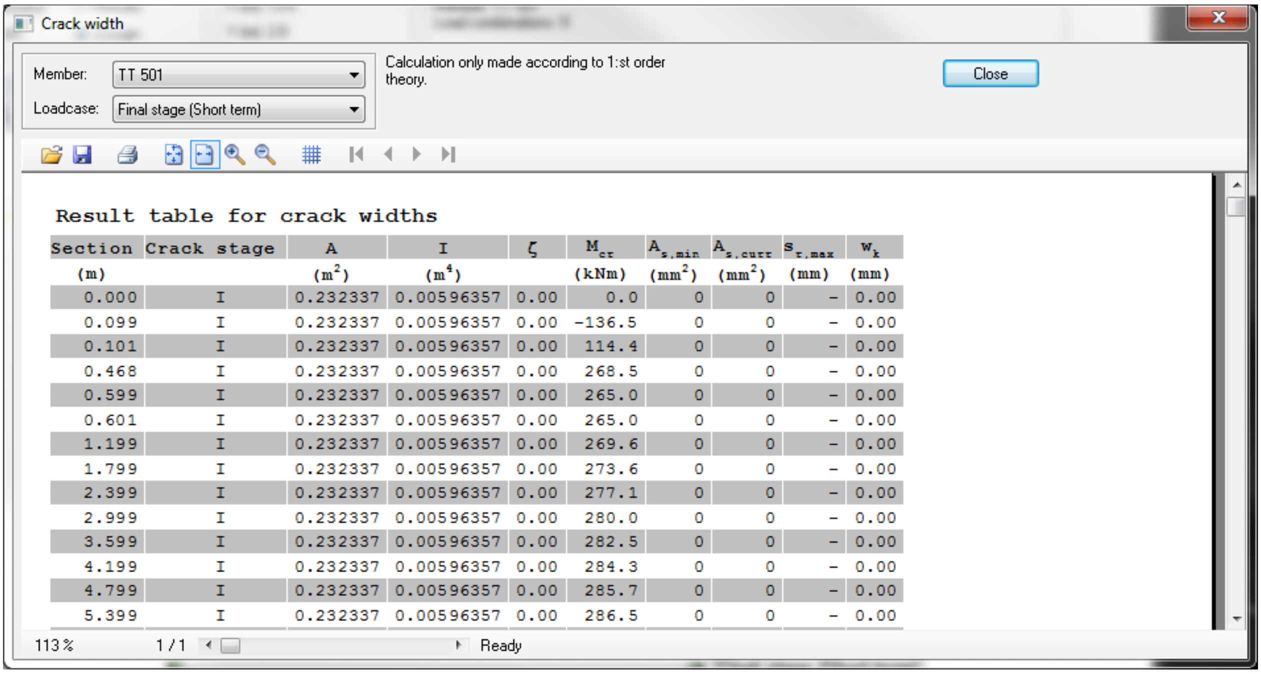
Section (m)	V_{Sd} (kN)	V' (kN)	$V_{Rd,s}$ (kN)	$V'/V_{Rd,s}$	$V_{Rd,max}$ (kN)	$(A_{sw}/s)_{cur}$ (mm²/m)	$(A_{sw}/s)_{req}$ (mm²/m)	$(A_{sw}/s)_{req\ min}$ (mm²/m)	b_w (mm)	z
0.000	-0.0	0.0	197.9	0.000	397.0	335	0	221	217	483
0.099	-2.4	0.0	197.9	0.000	408.2	335	0	220	217	460
0.101	140.2	131.5	193.9	0.678	401.4	335	228	221	217	478
0.468	131.5	131.5	193.9	0.678	447.3	335	228	221	217	479
0.599	128.4	131.5	193.9	0.678	452.0	335	228	221	217	479
0.601	128.3	108.6	193.9	0.560	452.0	335	188	221	217	479
1.199	114.1	108.6	193.9	0.560	452.5	335	188	221	217	479
1.799	99.8	77.0	193.9	0.397	452.9	335	133	221	217	479
2.399	85.6	77.0	193.9	0.397	453.3	335	133	221	217	479
2.999	71.3	45.3	193.9	0.234	453.6	335	79	221	217	479
3.599	57.1	45.3	193.9	0.234	453.9	335	79	221	217	479
4.199	42.8	13.7	193.9	0.071	454.1	335	24	221	217	479
4.799	28.5	13.7	193.9	0.071	454.2	335	24	221	217	479
5.399	14.3	13.7	193.9	0.071	454.3	335	24	221	217	479
5.999	0.0	0.0	193.9	0.000	454.3	335	0	221	217	479
6.601	-14.3	13.7	193.9	0.071	454.3	335	24	221	217	479
7.201	-28.5	13.7	193.9	0.071	454.2	335	24	221	217	479
7.801	-42.8	13.7	193.9	0.071	454.1	335	24	221	217	479
8.401	-57.1	45.3	193.9	0.234	453.9	335	79	221	217	479
9.001	-71.3	45.3	193.9	0.234	453.6	335	79	221	217	479
9.601	-85.6	77.0	193.9	0.397	453.3	335	133	221	217	479
10.201	-99.8	77.0	193.9	0.397	452.9	335	133	221	217	479
10.801	-114.1	108.6	193.9	0.560	452.5	335	188	221	217	479
11.399	-128.3	108.6	193.9	0.560	452.0	335	188	221	217	479
11.401	-128.4	131.5	193.9	0.678	452.0	335	228	221	217	479

Figure 2-32 Shear capacity

Here are the explanations for Figure 2-32:

- Section [m]: Position of calculation sections
- V_{Sd} [kN]: Shear force due to load
- V' [kN]: Shear force that you need to reinforce with stirrups for.
- $V_{Rd,s}$ [kN]: Capacity of the stirrups
- $V_{Rd,max}$ [kN]: Maximum shear capacity of the cross section (lower limit of compression failure)
- $(A_{sw}/s)_{cur}$ [mm2/m]: Amount of stirrups, current amount
- $(A_{sw}/s)_{req}$ [mm2/m]: Amount of stirrups, requirement due to loads
- $(A_{sw}/s)_{req\ min}$ [mm2/m]: Amount of stirrups, minimum requirement due to the code
- b_w [mm]: Calculated width
- z [mm]: Interior lever

The last table is Crack data, Figure 2-33.



Result table for crack widths

Section (m)	Crack stage	A (m^2)	I (m^4)	ζ	M_{cr} (kNm)	$A_{s,min}$ (mm^2)	$A_{s,curr}$ (mm^2)	$s_{r,max}$ (mm)	w_k (mm)
0.000	I	0.232337	0.00596357	0.00	0.0	0	0	-	0.00
0.099	I	0.232337	0.00596357	0.00	-136.5	0	0	-	0.00
0.101	I	0.232337	0.00596357	0.00	114.4	0	0	-	0.00
0.468	I	0.232337	0.00596357	0.00	268.5	0	0	-	0.00
0.599	I	0.232337	0.00596357	0.00	265.0	0	0	-	0.00
0.601	I	0.232337	0.00596357	0.00	265.0	0	0	-	0.00
1.199	I	0.232337	0.00596357	0.00	269.6	0	0	-	0.00
1.799	I	0.232337	0.00596357	0.00	273.6	0	0	-	0.00
2.399	I	0.232337	0.00596357	0.00	277.1	0	0	-	0.00
2.999	I	0.232337	0.00596357	0.00	280.0	0	0	-	0.00
3.599	I	0.232337	0.00596357	0.00	282.5	0	0	-	0.00
4.199	I	0.232337	0.00596357	0.00	284.3	0	0	-	0.00
4.799	I	0.232337	0.00596357	0.00	285.7	0	0	-	0.00
5.399	I	0.232337	0.00596357	0.00	286.5	0	0	-	0.00

Figure 2-33 Crack data

… and the explanations for Figure 2-33:

- Section [m]: Position of calculation sections
- Crack stage: Check if the crossection is uncracked (I) or cracked (II).
- A [m^2]: Crossection area, if section is cracked this value will be reduced accordingly.
- I [m^4]: Moment of inertia, if section is cracked this value will be reduced accordingly.
- ζ : Coefficient due to tension stiffening
- M_{cr} [kNm]: Moment when the crossection cracks
- $A_{s,min}$ [mm^2]: Minimum amount of crack reinforcement
- $A_{s,curr}$ [mm^2]: Current amount of crack reinforcement
- $s_{r.max}$ [mm]: Distance between cracks
- w_k [mm]: Crack width

2.2 SLOPED I-BEAM (COMING SOON)

2.3 Hollowcore slab (coming soon)

2.4 HOW TO ADD A FIRE LOADCOMBINATION

This chapter will show two different ways to add a fire load combination to the calculation and to show the unique results that will be available.

2.4.1 CREATING A LOAD COMBINATION THROUGH THE WIZARD

The first place where you will be able to add a fire load combination is with the Create new prestressed beam-wizard. Just check the second checkbox and a fire combination called “Fire (6.11)” will appear last in the combination-list.

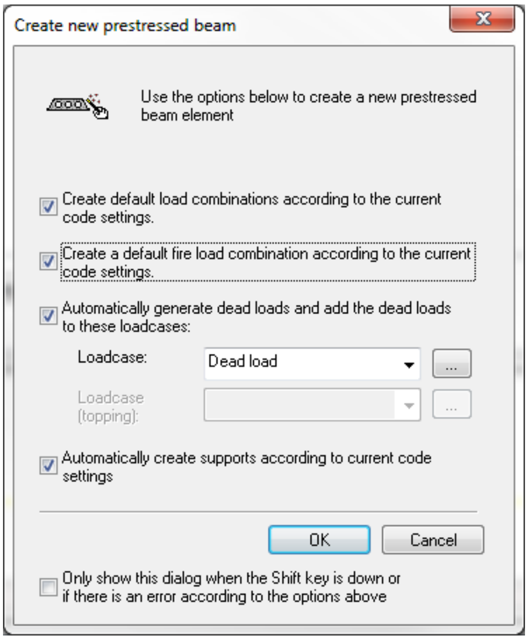


Figure 2-34 Adding a fire load combination

If this had been activated for example in chapter 2.1 the result would have looked like this:

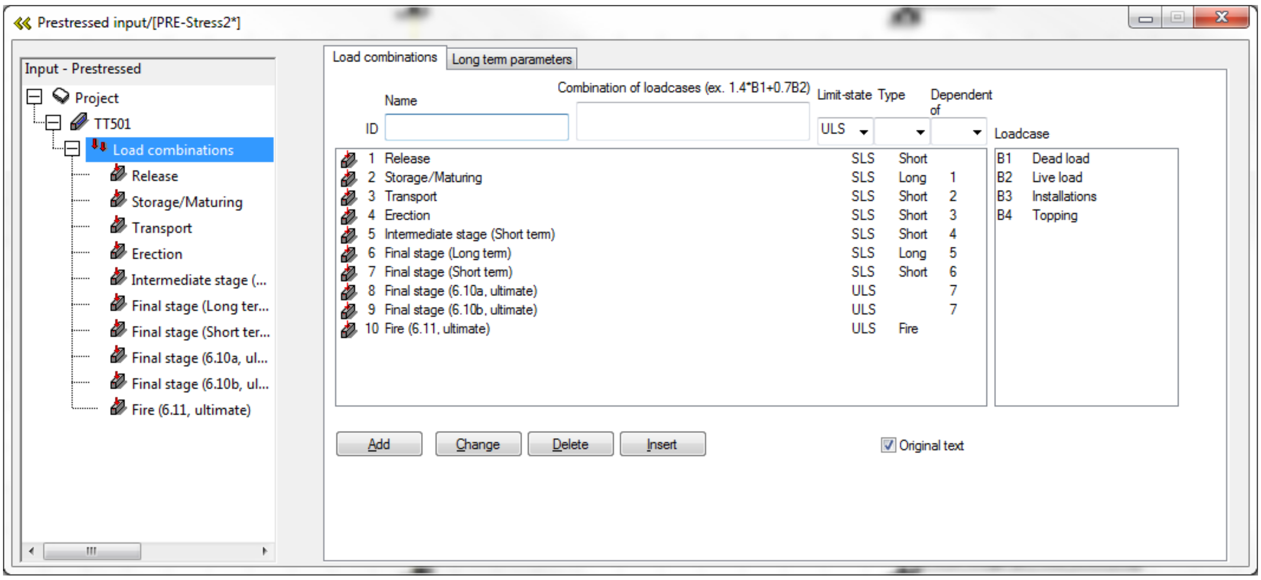


Figure 2-35 Fire load combination added

2.4.2 MAKING A LOAD COMBINATION FROM THE PROJECT MANAGER

Another way to add a fire load combination to an older file or if you forgot to check the checkbox in 2.4.1 is to do it manually.

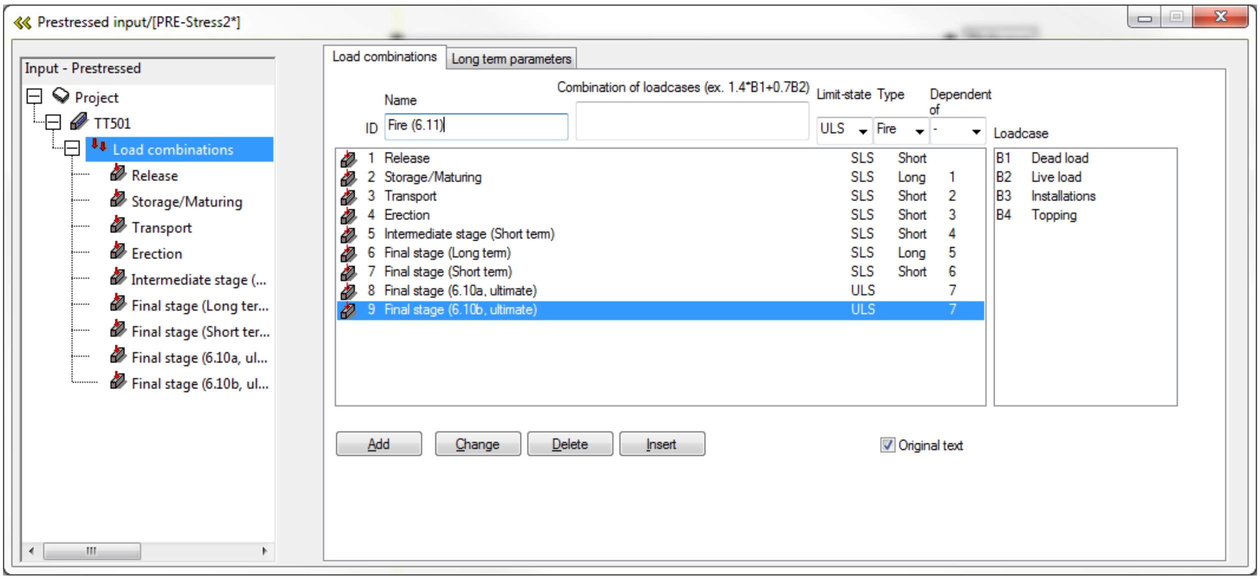


Figure 2-36 Entering a fire load combination

Go to Load combinations, click on an existing load combination that will be used as a template with regards to supports and material. Enter a unique name for the load combination (eg. *Fire (6.11)*), set limit state to *ULS*, Type as *Fire* and Dependent of to "-". Click *Add* to add the combination last in the list.

What will happen now is that you will get an option if you want to copy the selected load combination with regards to material and placement of the supports. See Figure 2-37. It is recommended to do this copy, by pressing Yes[1]. If you don't do this copy you will have to define the supports for this load combination manually.

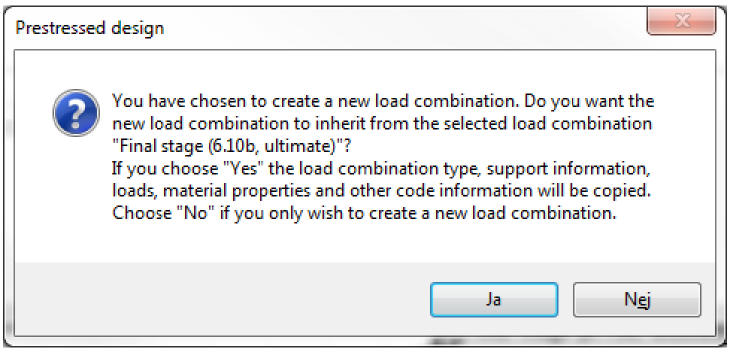


Figure 2-37 Create a load combination by copying an old combination

Verify that the correct type and dependency is selected. Dependency should be set to "-" because in the fire combination the cross section will be reduced depending on fire time. This will cause a different stress distribution. If you would try to set a dependency of a fire calculation you will get an error message, see Figure 2-38.

[1] Since these buttons are connected to the current language in Windows it will be "Yes" in your current language settings, (Swe: "Ja" = Eng. "Yes")

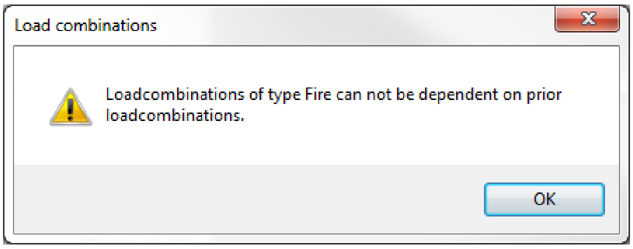


Figure 2-38 Setting an illegal dependency

If you successfully set the load combination you will se the result similar to Figure 2-35.

2.4.3 COMBINATION OF LOADS

According to EN 1990 6.4.3.3. (3) fire is an exceptional design situation, but only to some extent. EN 1992-1-1 2.4.2.4 mention that the fire is exceptional regarding combination of loads, but material properties should be picked from EN 1992-1-2 2.3 (1).

So to combine the loads, use formula 6.11 from EN 1990, see Figure 2-39.

6.4.3.3 Combinations of actions for accidental design situations

(1) The general format of effects of actions should be :

$$E_d = E\{G_{k,j}\,;P\,;A_d\,;(\psi_{1,1} \text{ or } \psi_{2,1})Q_{k,1}\,;\psi_{2,i}Q_{k,i}\} \quad j \geq 1; i > 1 \qquad (6.11a)$$

(2) The combination of actions in brackets { } can be expressed as :

$$\sum_{j\geq 1} G_{k,j}\text{"+"}P\text{"+"}A_d\text{"+"}(\psi_{1,1} \text{ or } \psi_{2,1})Q_{k,1}\text{"+"}\sum_{i>1}\psi_{2,i}Q_{k,i} \qquad (6.11b)$$

(3) The choice between $\psi_{1,1}Q_{k,1}$ or $\psi_{2,1}Q_{k,1}$ should be related to the relevant accidental design situation (impact, fire or survival after an accidental event or situation).

NOTE Guidance is given in the relevant Parts of EN 1991 to EN 1999.

Figure 2-39 Excerpt from EN 1990 6.4.3.3

For example: ”1.0 * Dead load + 1.0 * Permanent loads + ψ_1 * Main load + ψ_2 * Other free loads”

Note that the prestressing load P will be handled automatically in the program. The term A_d is unique in this accidental load combination, it will symbolize the indirect effet of the fire, but not the effect of the temperature on the material – coefficient of thermal expansion etc. For example 1 (2.1 Modelling of a TT/F 240/54 with non-structural topping) it would be looking something like Figure 2-40.

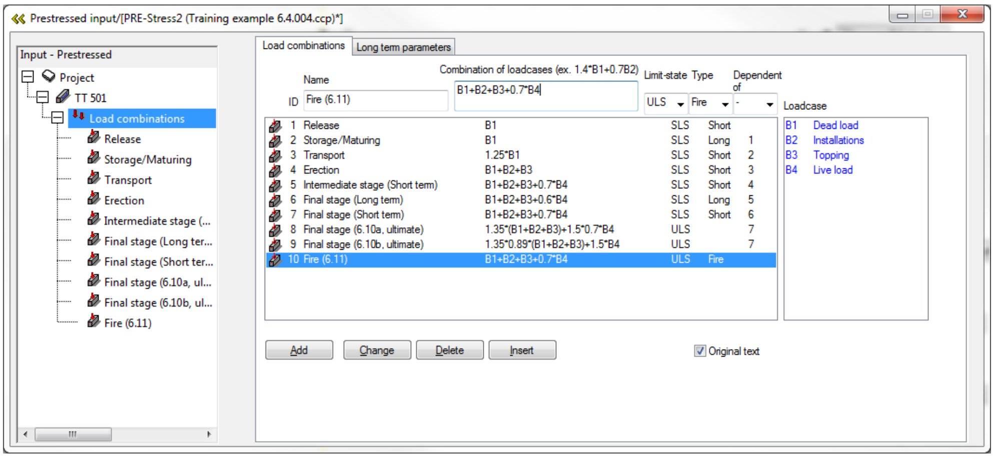
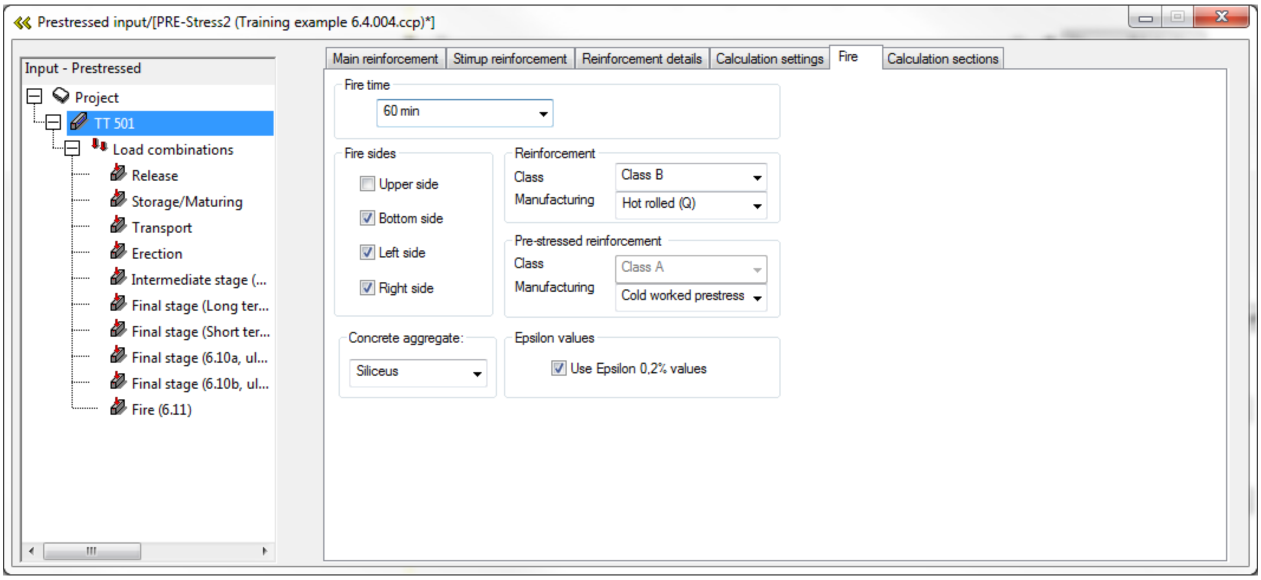


Figure 2-40 Load combination *Fire (6.11)* with loads combined

2.4.4 SETTINGS FOR THE FIRE



Under the Fire-tab there are a few settings (the manual will explain everything a bit furter in detail with references to EN 1992-2):

- Fire time: Fire resistance, time in minutes. R30, R60, R90, R120, R180 och R240
- Fire sides: What sides that will be subjected to fire, slabelement (hollowcores and slabs) can only be subjected by fire from the under and/or upper side. Beams and TT-elements can be subjected from the sides aswell.
- Concrete aggregate: Type of concrete aggregate. Siliconbased or limestonebased aggregate. This will affect the concrete strength at fire according to EN 1992 3.2.2.1.
- Reinforcement: Manufacturing class, see EN 1992-1-2 3.2.3. The class is shown in EN 1992-1-2, Annex C.
- Pre-stressed reinf: Manufacturing class for prestressing steel according to EN 1992-1-2 3.2.4
- Epsilon values: Strain limit used for fire calculation, according to EN 1992-1-2 3.2.3
  - ε=0.2%, default value.
  - If not 0.2% is used then ε=2% will be used. In order to use the 2%-limit the code demands justified assumptions for the calculation.

2.4.5 CALCULATION AND RESULTS

The calculation is being performed in the usual way, no need for activating anything, except be sure to have set the load combination to a frire-combination. So first perform an analysis then norm control (or 'design').

2.4.5.1 CODE CHECK

Here a verification of the actual load combination is made against the capacity for fire of the highest utilized sections. The three checks will be for highest positive moment, highest negative moment and highest shearforce.

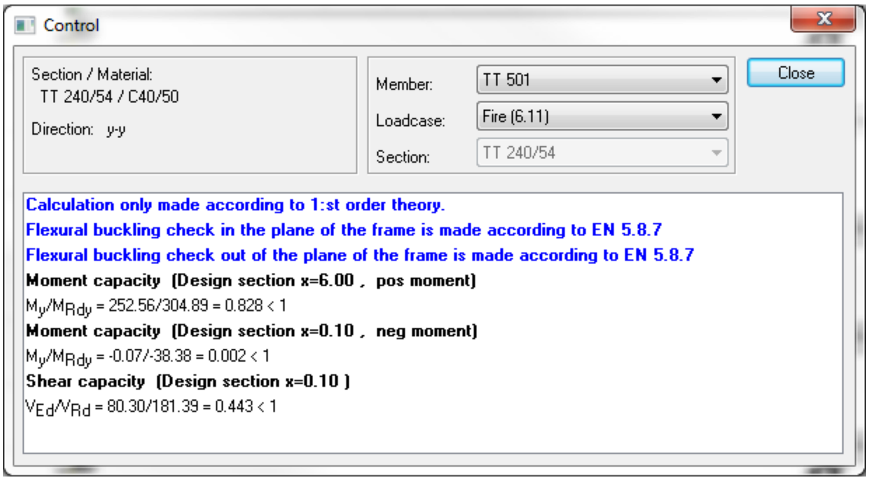


Figure 2-41 Control / code check

2.4.5.2 TABLE – BENDING

The result is not different from the ordinary bending-table. Values are considered using the reduced crossection, parameters and material depending on the fire.

2.4.5.3 TABLE – SHEAR

Same as Table – Bendning, no unique results due to fire. Calculation of shear is made from the fire load combination.

2.4.5.4 TABLE – FIRE – REDUCED CONCRETE PARAMETERS

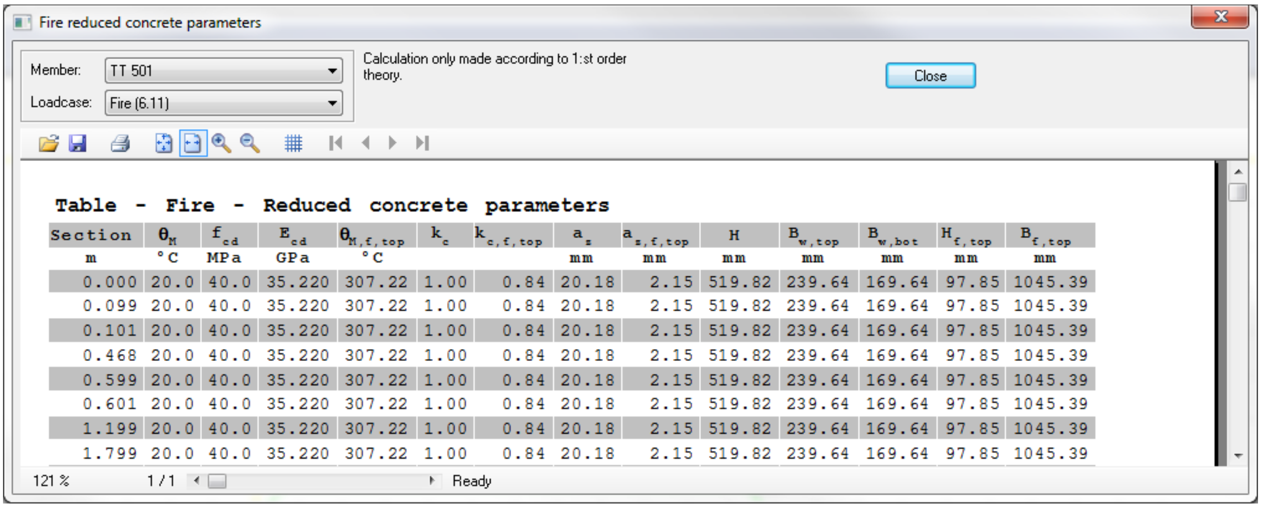


Table - Fire - Reduced concrete parameters

Section	θ_M	f_{cd}	E_{cd}	$\theta_{M,f,top}$	k_c	$k_{c,f,top}$	a_z	$a_{z,f,top}$	H	$B_{w,top}$	$B_{w,bot}$	$H_{f,top}$	$B_{f,top}$
m	°C	MPa	GPa	°C			mm	mm	mm	mm	mm	mm	mm
0.000	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39
0.099	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39
0.101	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39
0.468	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39
0.599	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39
0.601	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39
1.199	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39
1.799	20.0	40.0	35.220	307.22	1.00	0.84	20.18	2.15	519.82	239.64	169.64	97.85	1045.39

Figure 2-42 Table – Fire – Reduced concrete parameters

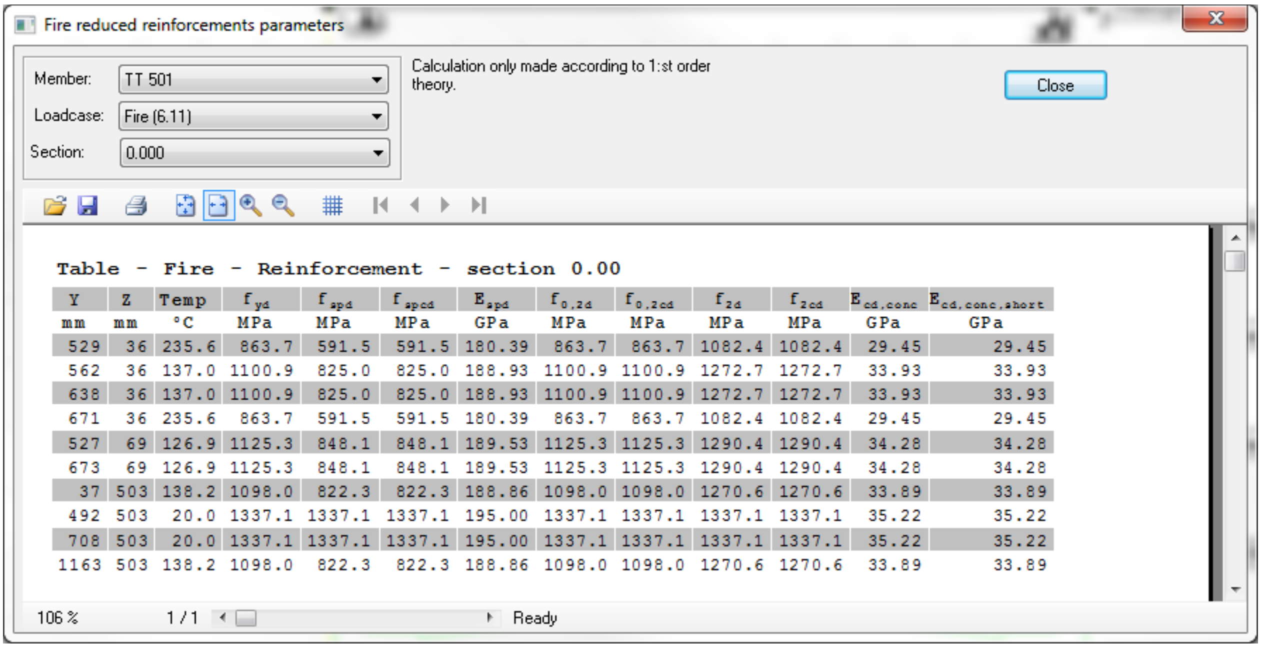
The different columns in the fire load combination means:

- Section [m]: Section along the element
- Θ_M [°C]: Temperature in the middle of the crossection
- f_{cd} [MPa]: Concrete strength
- E_{cd} [GPa]: Module of elasticity for the concrete (reduced due to fire)
- $\Theta_{M,f,top}$ [°C]: Den temperature in the top flange, see more info about the indexes below.
- k_c [-]: Factor used for reduction of the concrete material.
- $k_{c,f,top}$ [-]: Value of the reduction being used for different places of the section (indexes: see below)
- a_z [mm]: Height of the damaged zone of the section
- $a_{z,f,top}$ [mm]: Height of the damaged zone (indexes: see below)
- H [mm]: Remaining height of the section after reductions

Depending on the section there are different measurements and parameters being presented:

Parameter B [mm]: width. H [mm]: height
index: w: web, f: flange
index: top (upper, flange for instance) and bot (bottom)

2.4.5.5 TABLE – FIRE – REINFORCEMENT



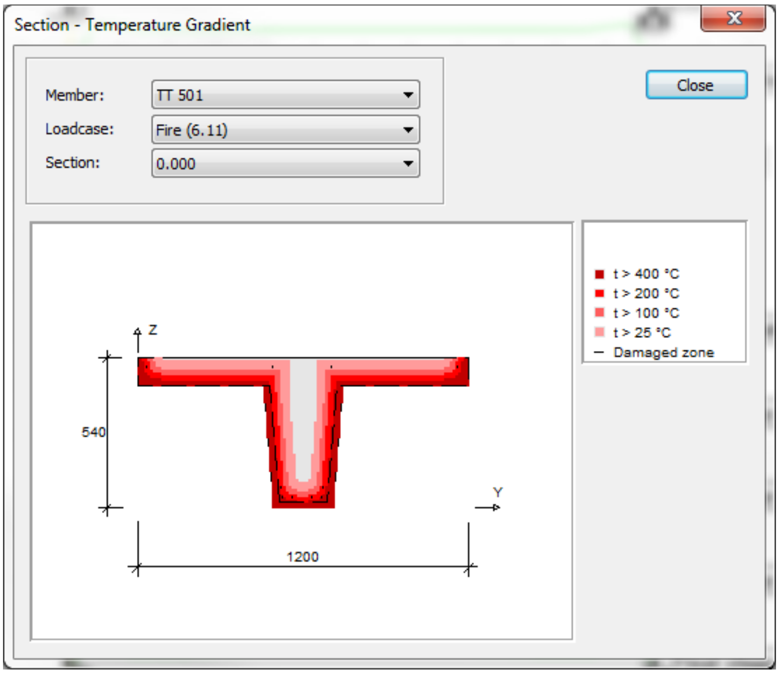
Y	Z	Temp	f_{yd}	f_{spd}	f_{spcd}	E_{spd}	$f_{0,2d}$	$f_{0,2cd}$	f_{2d}	f_{2cd}	$E_{cd,conc}$	$E_{cd,conc,short}$
mm	mm	°C	MPa	MPa	MPa	GPa	MPa	MPa	MPa	MPa	GPa	GPa
529	36	235.6	863.7	591.5	591.5	180.39	863.7	863.7	1082.4	1082.4	29.45	29.45
562	36	137.0	1100.9	825.0	825.0	188.93	1100.9	1100.9	1272.7	1272.7	33.93	33.93
638	36	137.0	1100.9	825.0	825.0	188.93	1100.9	1100.9	1272.7	1272.7	33.93	33.93
671	36	235.6	863.7	591.5	591.5	180.39	863.7	863.7	1082.4	1082.4	29.45	29.45
527	69	126.9	1125.3	848.1	848.1	189.53	1125.3	1125.3	1290.4	1290.4	34.28	34.28
673	69	126.9	1125.3	848.1	848.1	189.53	1125.3	1125.3	1290.4	1290.4	34.28	34.28
37	503	138.2	1098.0	822.3	822.3	188.86	1098.0	1098.0	1270.6	1270.6	33.89	33.89
492	503	20.0	1337.1	1337.1	1337.1	195.00	1337.1	1337.1	1337.1	1337.1	35.22	35.22
708	503	20.0	1337.1	1337.1	1337.1	195.00	1337.1	1337.1	1337.1	1337.1	35.22	35.22
1163	503	138.2	1098.0	822.3	822.3	188.86	1098.0	1098.0	1270.6	1270.6	33.89	33.89

Figure 2-43 Table - Fire – Reinforcement

The different columns in the fire load combination means:

- Y [mm]: Position of reinforcement in the local coordinatesystem of the element
- Z [mm]: Position of reinforcement in the local coordinatesystem of the element
- Temp [°C]: Temperature at the position of the reinforcement
- f_{yd} [MPa]: Design strength of the reinforcement at current temperature
- f_{spd} [MPa]: Tensile strength of prestressing wire
- f_{spcd} [MPa]: Compressive strength of the prestressing wire
- E_{spd} [GPa]: Module of elasticity of the prestressing wire
- $f_{0,2d}$ [MPa]: 0.2% design strength for tensile strength
- $f_{0,2cd}$ [MPa]: 0.2% design strength of compressive strength
- f_{2d} [MPa]: 2% design strength for tensile strength (not recommended to use)
- f_{2cd} [MPa]: 2% design strength of compressive strength (not recommended to use)
- $E_{cd,conc}$ [GPa]: Module of elasticity of the concrete (reduced due to fire)
- $E_{cd,conc,short}$ [GPa]: Short term module of elasticity of the concrete (reduced due to fire)

2.4.5.6 TEMPERATURE GRADIENT



Dashed line shows damaged zone. Concrete in the damaged zone is not being considered in calculations.