Continuous composite beam
IDEA Beam
IDEA Tendon
Tutorial

IDEA RS s.r.o. | Jihomoravské inovační centrum, U Vodárny 2a, 616 00 BRNO
tel.: +420 - 511 205 263, fax: +420 - 541 143 011, www.idea-rs.cz, www.idea-rs.com

Content

1 Introduction 3
2 Input of geometry 3
2.1 Creating a new project 3
2.1.1 Modification of cross-sections in second beam span 18
2.1.2 Modification of cross-sections of diaphragms 19
2.1.3 Modification of geometry 21
3 Construction stages 23
3.1 General settings 23
3.2 History of members 23
3.3 Construction stages 28
4 Loads 29
4.1 Load cases 29
4.2 Dead loads on members 30
4.3 Assigning new variable load case to combinations 32
5 Design of tendons in IDEA Tendon 35
5.1 Launching IDEA Tendon 35
5.2 Input of construction stages 36
5.3 Input of tendons 37
5.3.1 Input of pre-tensioned tendons group to member 3 37
5.3.2 Input of next tendon groups to member 3 38
5.3.3 Input of pre-tensioned tendons group to member 6 40
5.3.4 Zadání dalších skupin kabelů na prvek 6 42
5.1 Closing IDEA Tendon 43
6 Analysis 44

1 Introduction

This tutorial describes input of composite continuous beam composed of prefabricated members and made continuous by casting diaphragms and composite deck in **IDEA Beam** application.

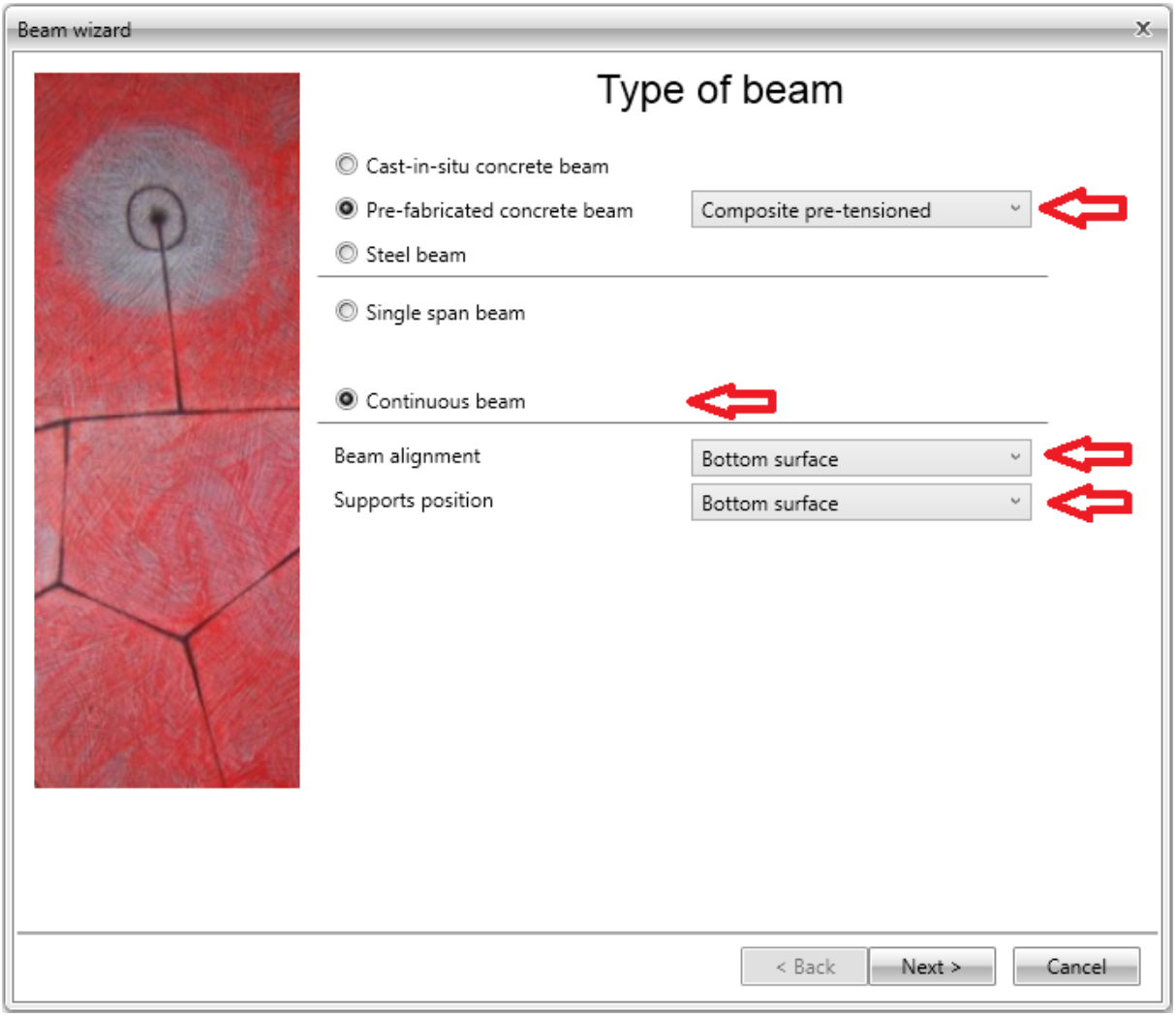
2 Input of geometry

2.1 Creating a new project

Run the IDEA Beam application.

To create a new project click **New** in ribbon group **Project**.

The **Beam wizard** starts.

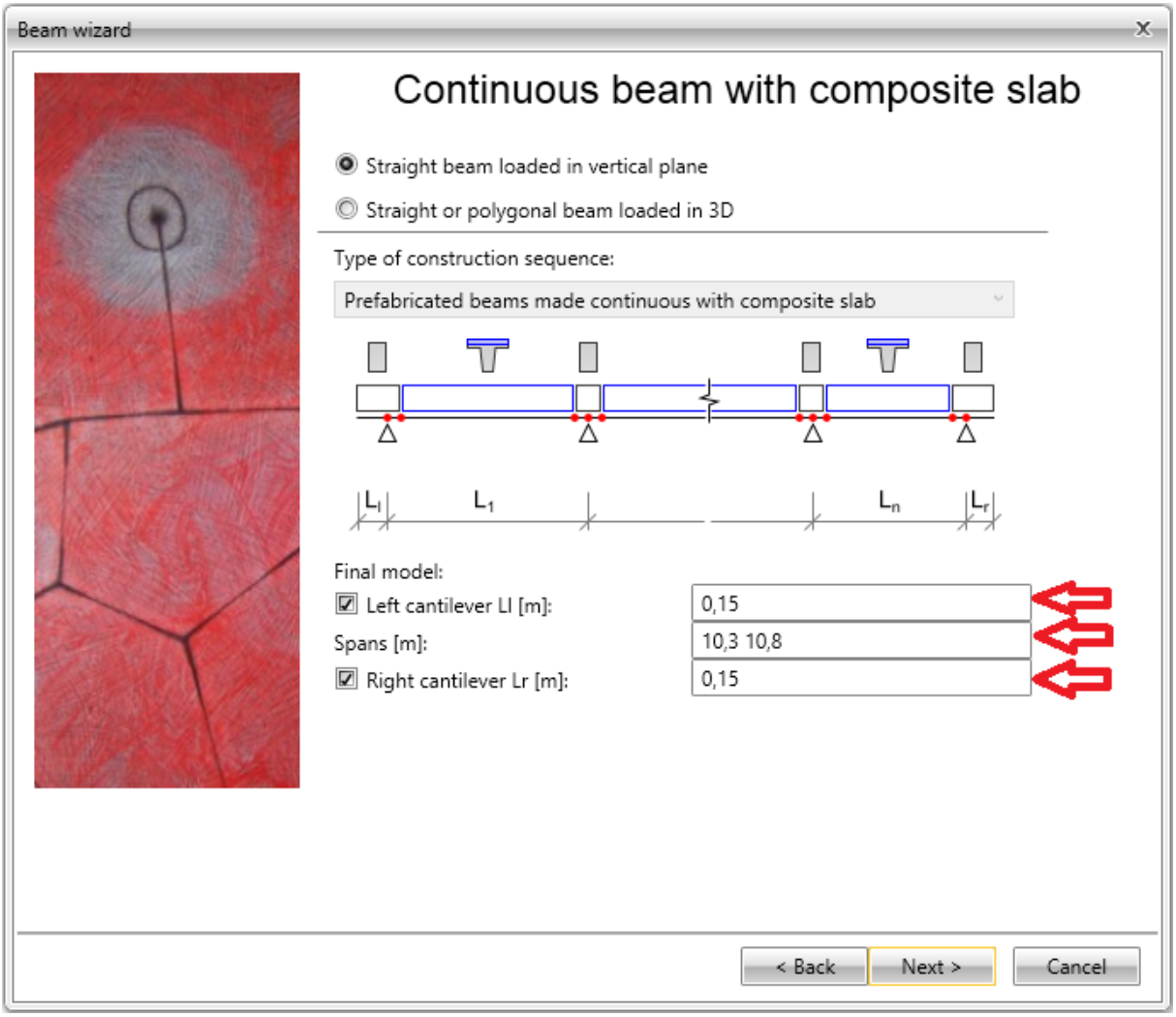


Set first dialog of **Beam wizard** as follows:

- Select the radio button **Pre-fabricated concrete beam**.

- Select the type of beam **Composite pre-tensioned**.
- Select the radio button **Continuous beam**.
- Select **Bottom surface** in combo box **Beam alignment**.
- Select **Bottom surface** in combo box **Supports position**.

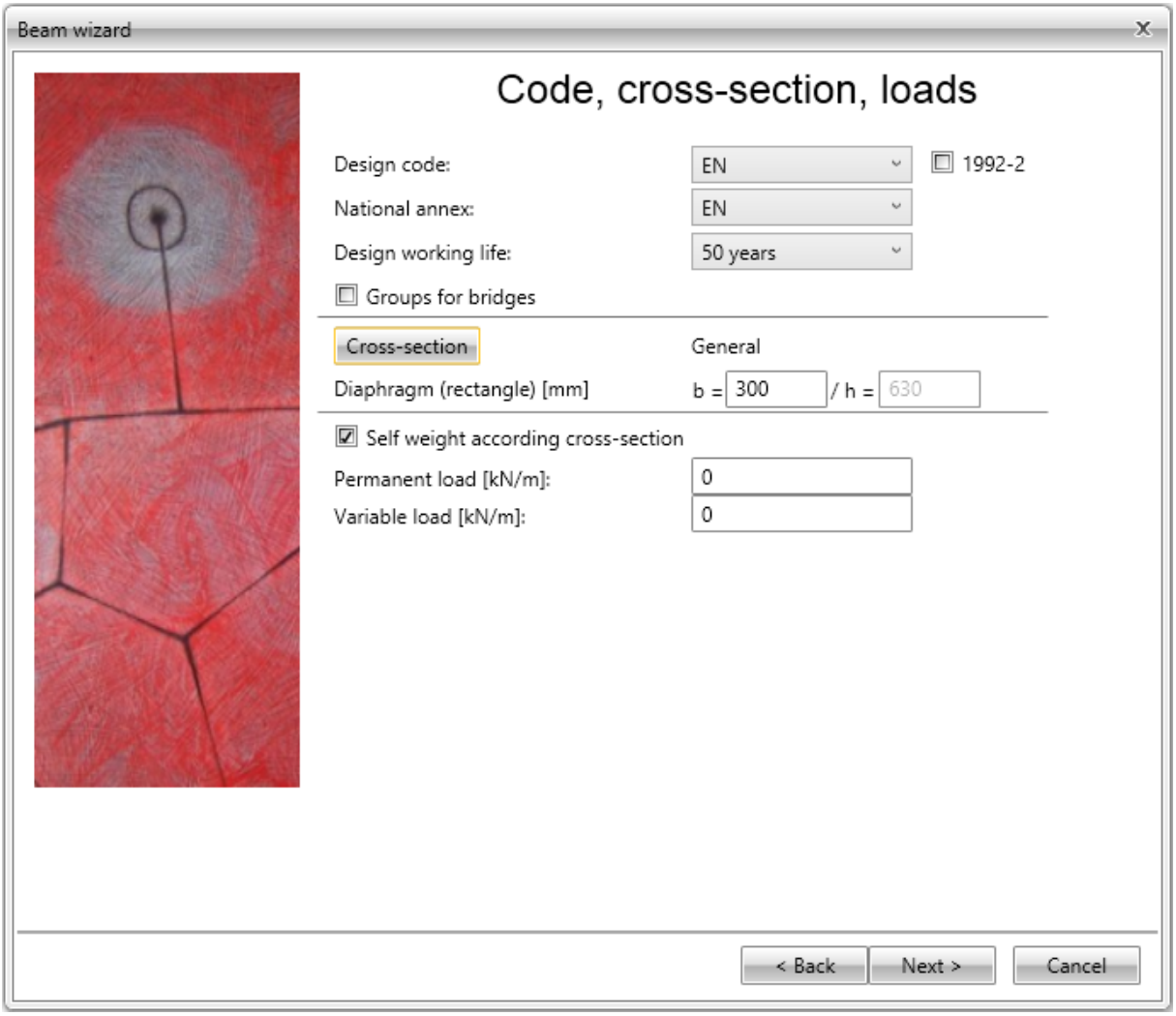
Click **Next** to continue to the following wizard step.



Set beam properties in the second wizard dialog **Continuous beam with composite slab** as follows:

- Select radio button **Straight bema loaded in vertical plane**.
- Type of construction sequence is limited to **Prefabricated beams made continuous with composite slab** and cannot be changed.
- Select checkbox **Left cantilever Ll [m]**and input the length of cantilever at the beam beginning **0,15**.
- Enter the string to define the spans lengths of the final beam to the **Spans [m]**edit box: **10,3 10,8**.
- Select checkbox **Right cantilever Ll [m]**and input the length of cantilever at the beam end **0,15**.

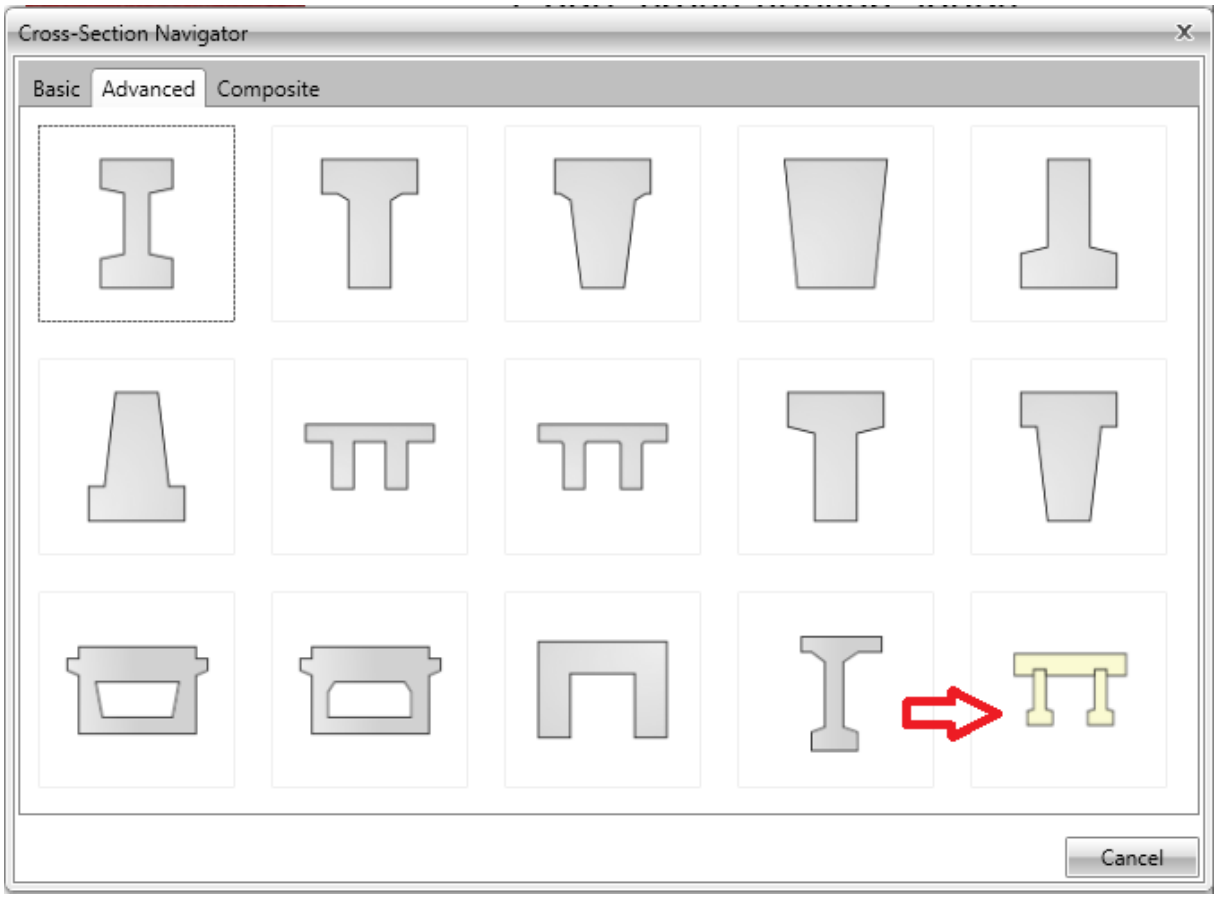
Click **Next** to continue to the following wizard step.



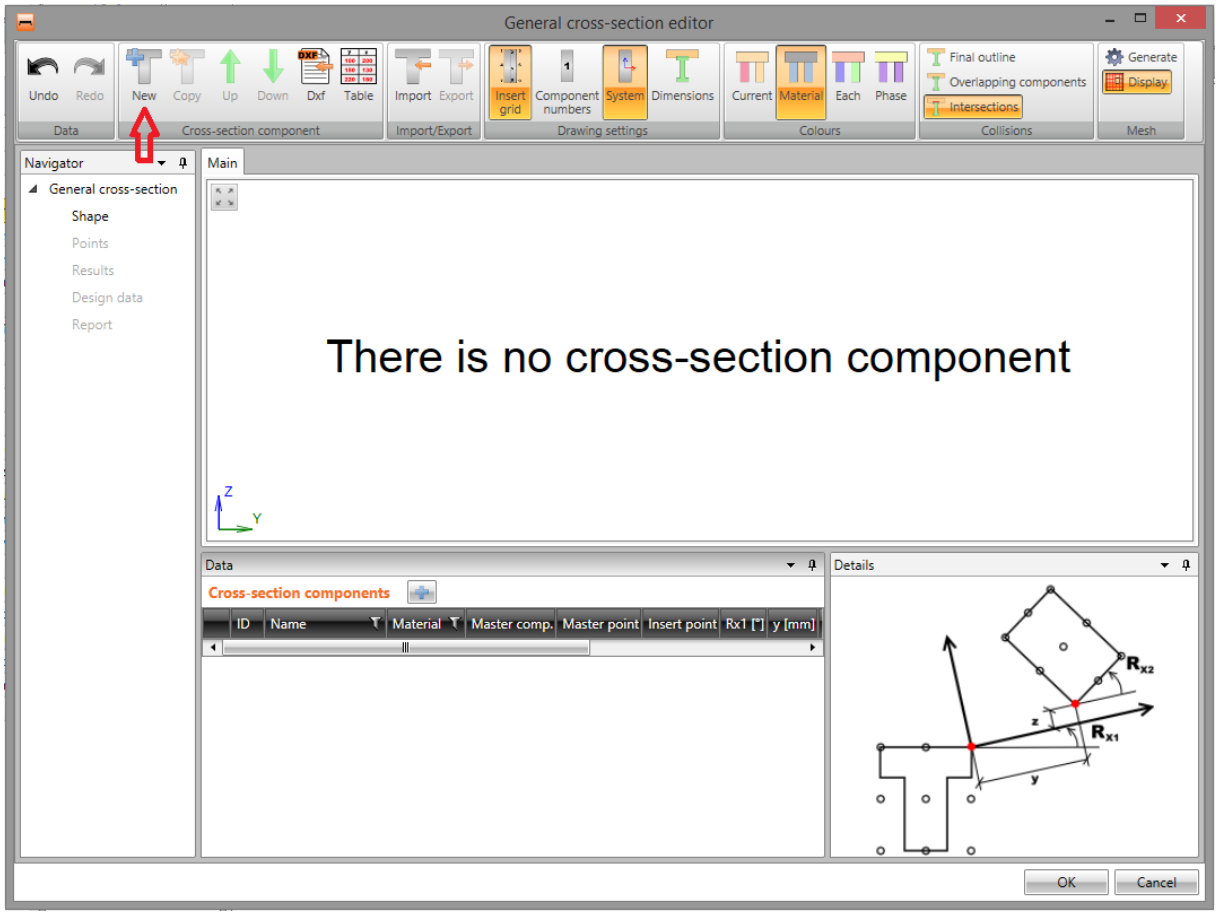
Set properties in dialog **Code, cross-section, loads** as follows:

- Set **Design code** and **National annex** to **EN**.
- Set **Design working life** to **50 year**s.
- The beam wizard enables to input rectangular cross-section of diaphragm. Cross-section of diaphragm can be changed after finishing the wizard. Height of default rectangular cross-section is calculated automatically according to height of composite cross-section. Set the value of **Diaphragm [mm] b** to **300** mm.
- Select the option **Self weight according to cross-section**.
- Set the value of **Permanent load [kN/m]** to **0** kN/m.
- Set the value of **Variable load [kN/m]** to **0** kN/m.

Click **Cross-section** to input the cross-section, which will be assigned to all prefabricated beams. The cross-section of prefabricated beam in second span will be modified later. Dialog **Cross-section navigator** appears. Click the picture of general cross-section on the tab **Advanced** to start input of general cross-section of prefabricated beams.

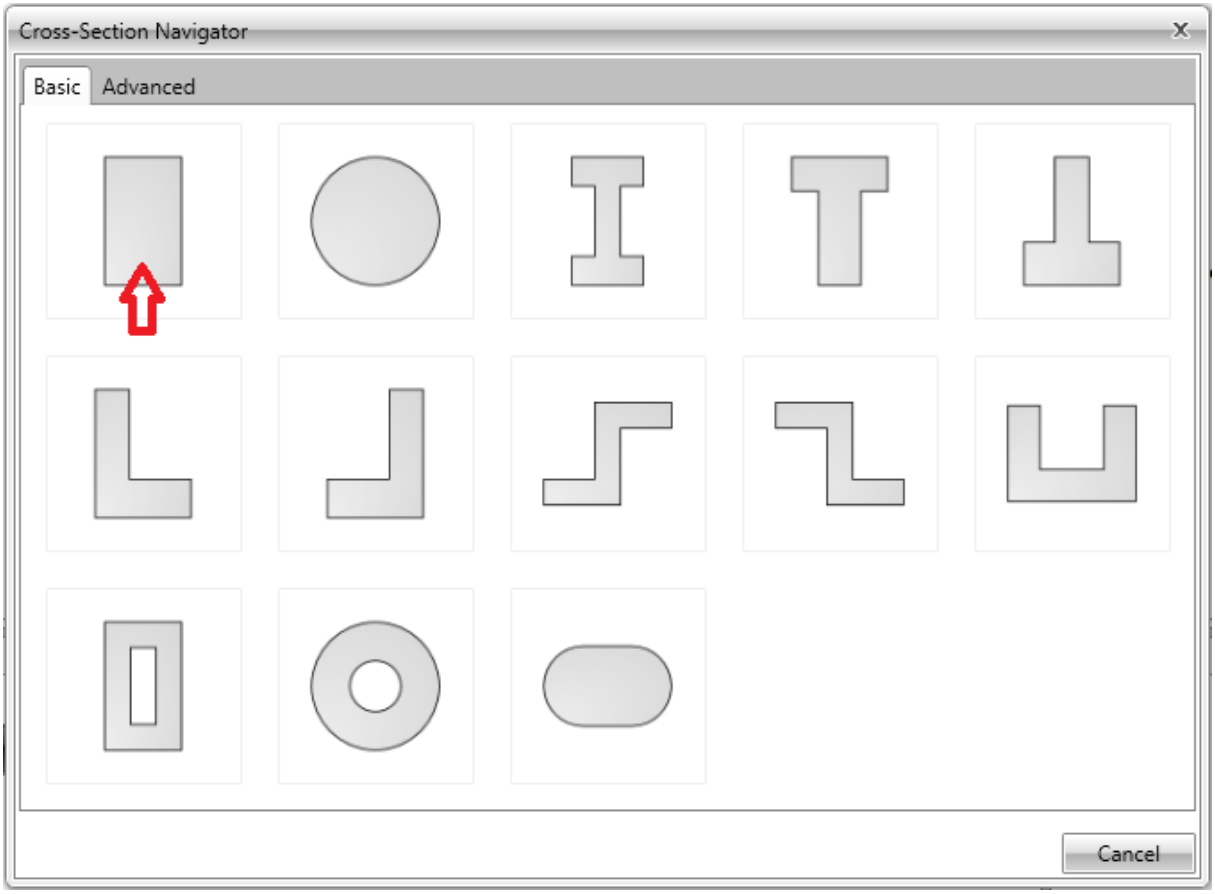


Dialog **General cross-section editor** appears.

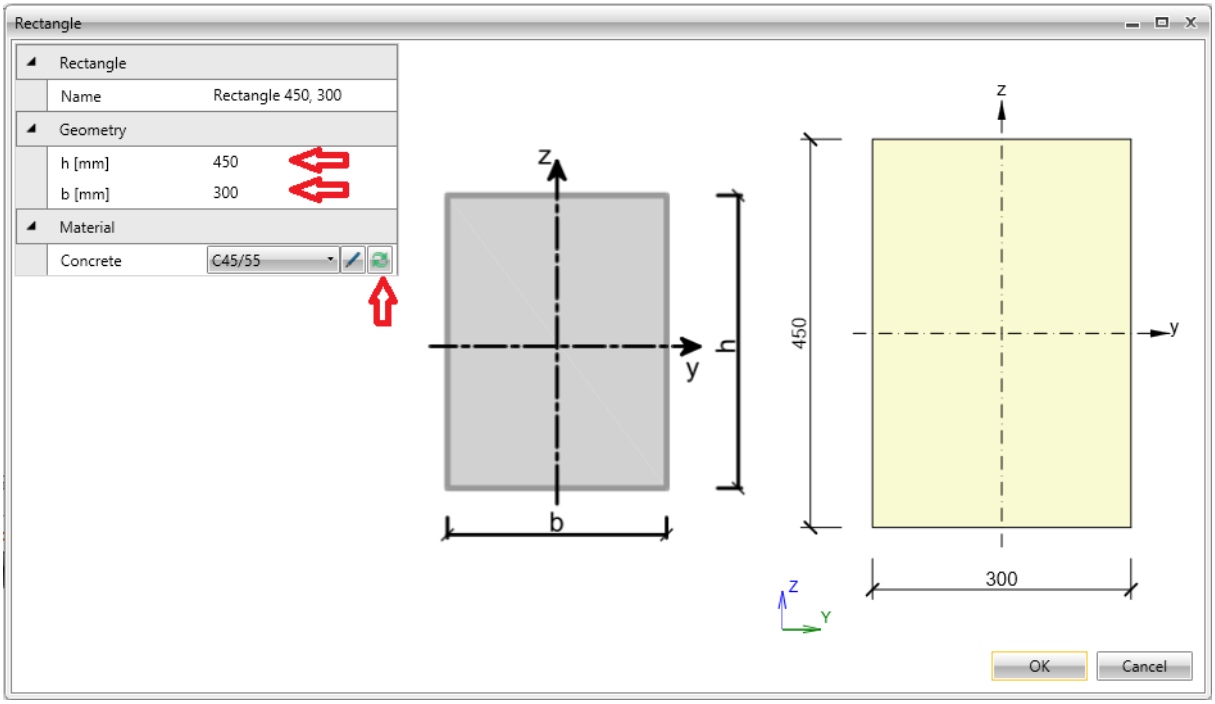


Click **New** in ribbon group **Cross-section component** to input the firs component of general cross-section.

Dialog **Cross-section** navigator appears. Click picture of rectangular cross-section on tab **Basic** to input the prefabricated component of composite cross-section.

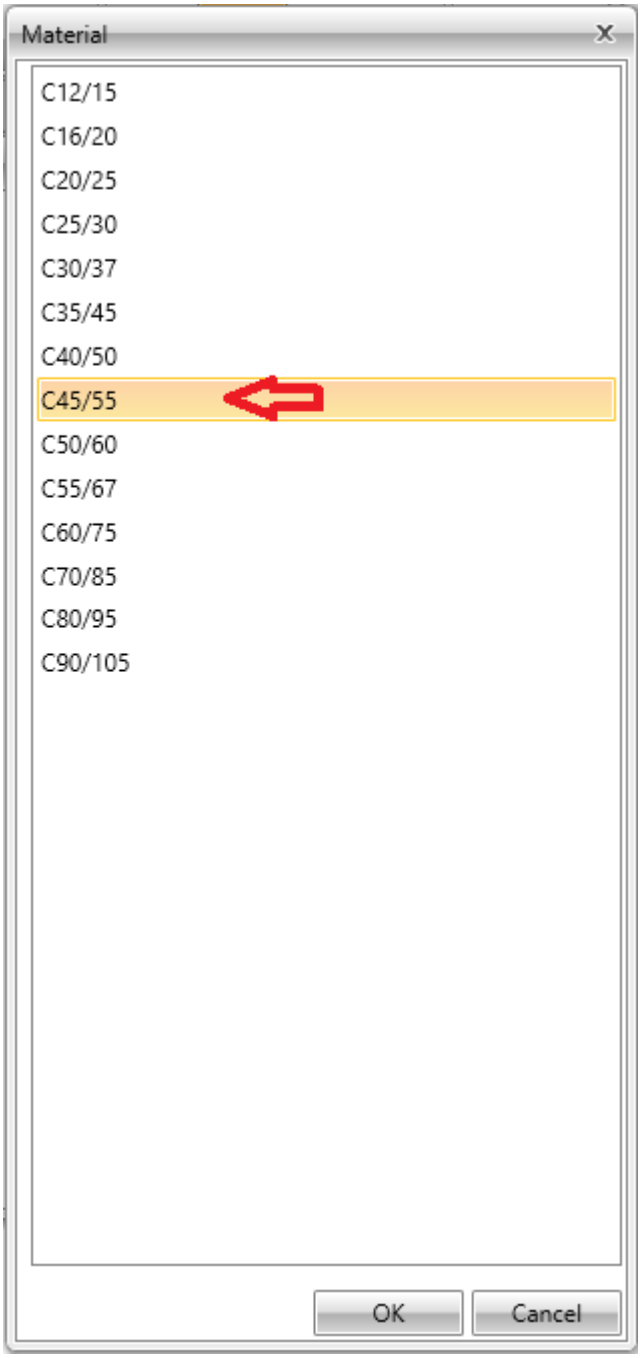


Dialog for definition of rectangular cross-section appears.



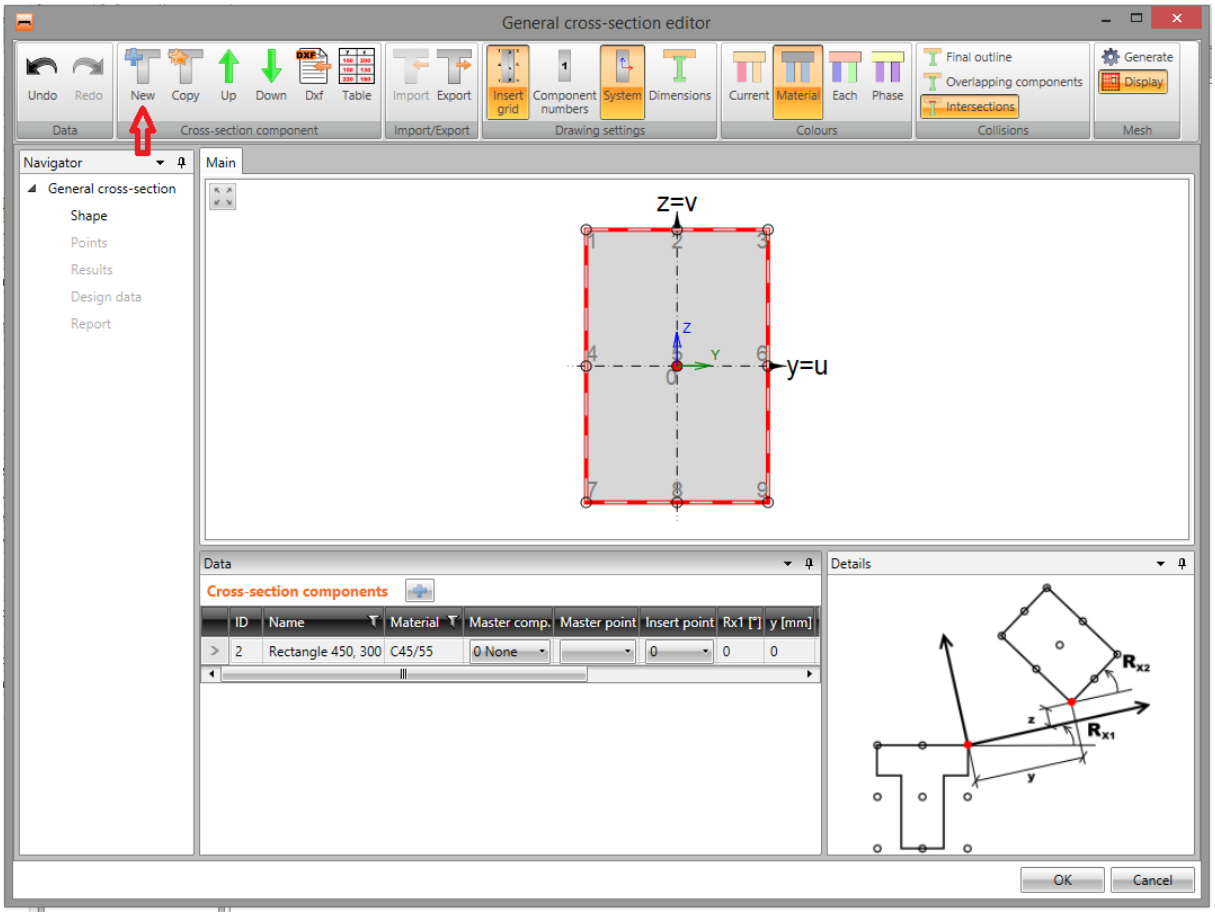
Set properties of rectangular cross-section in **Rectangle** dialog as follows:

- set height of cross-section **h [mm]** to **450 mm**;
- set width of cross-section **b [mm]** to **300 mm**.
- click  to add new material and assign it to the cross-section. Select material **C45/55**.

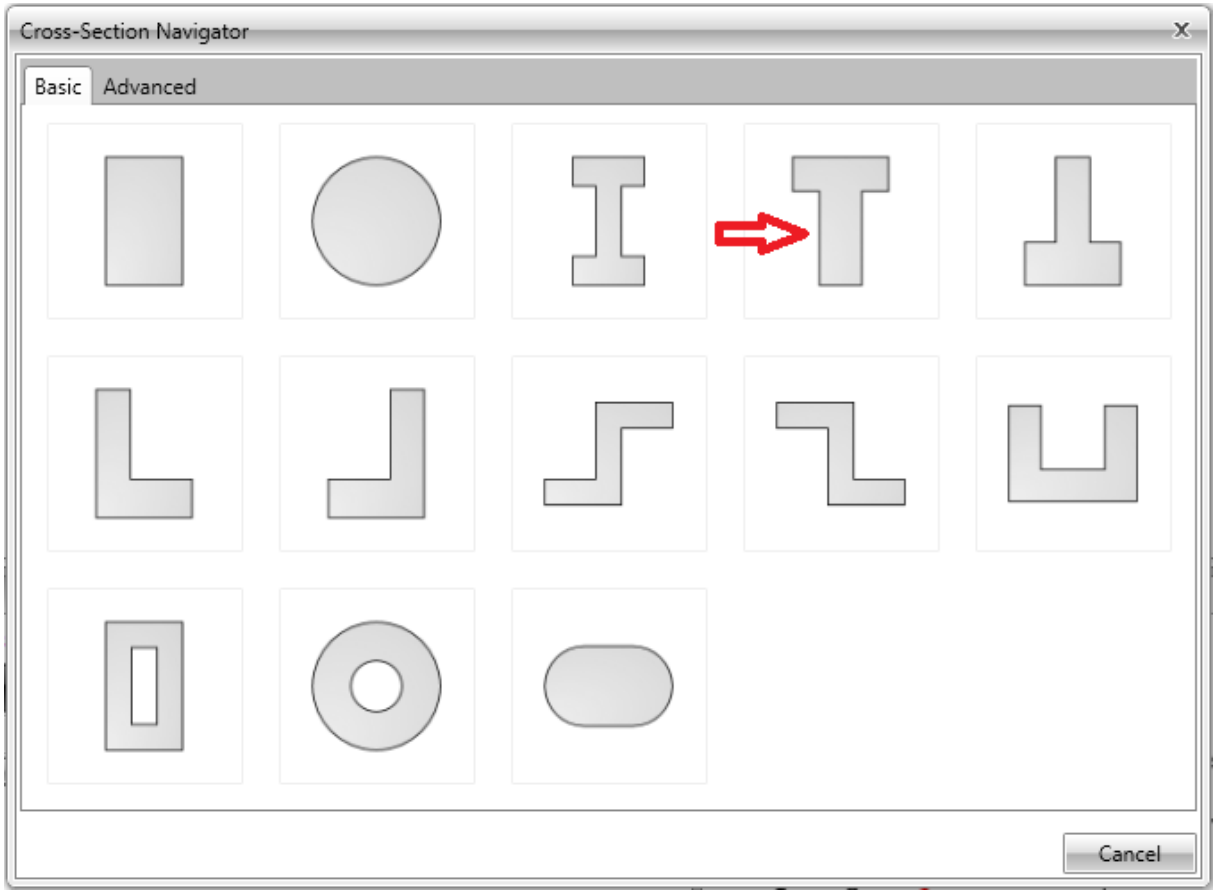


Click **OK** in **Rectangle** dialog to finish input of rectangular component of general cross-section.

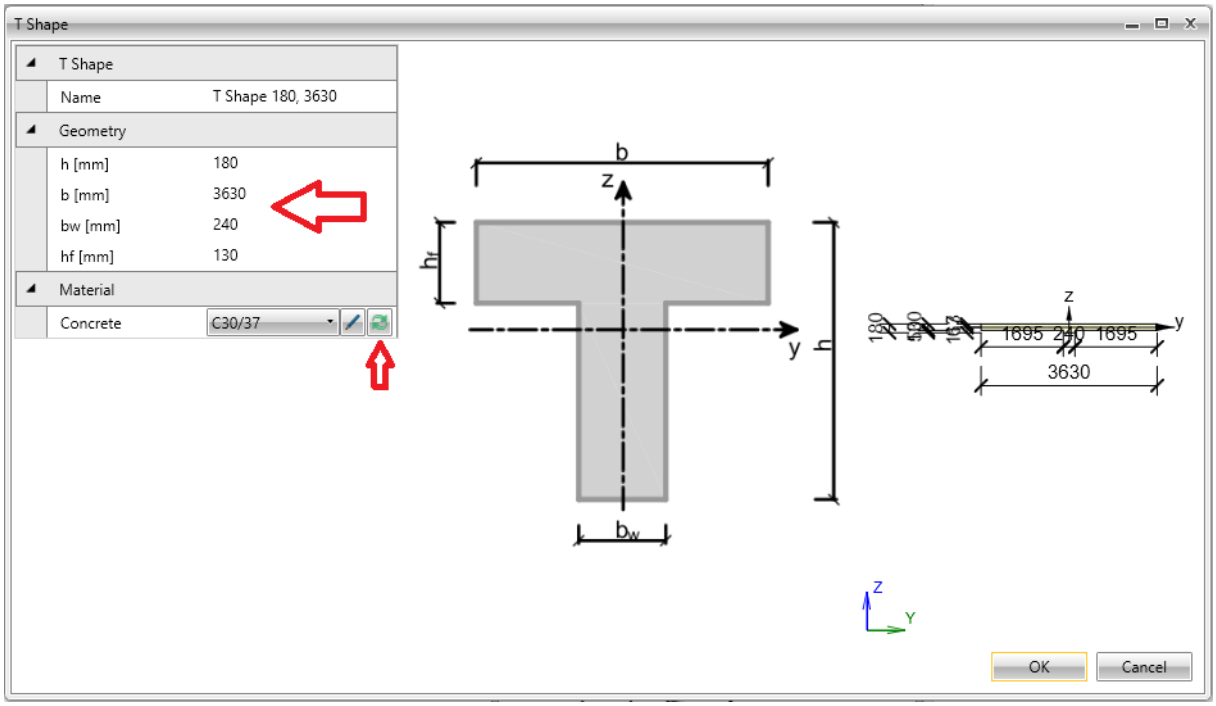
The composite slab will be defined as the second component of general cross-section. Click **New** in ribbon group **Cross-section component** to input the cross-section of composite slab.



Dialog **Cross-section** navigator appears. Click picture of T-shaped section on tab **Basic** to start input of composite slab component.



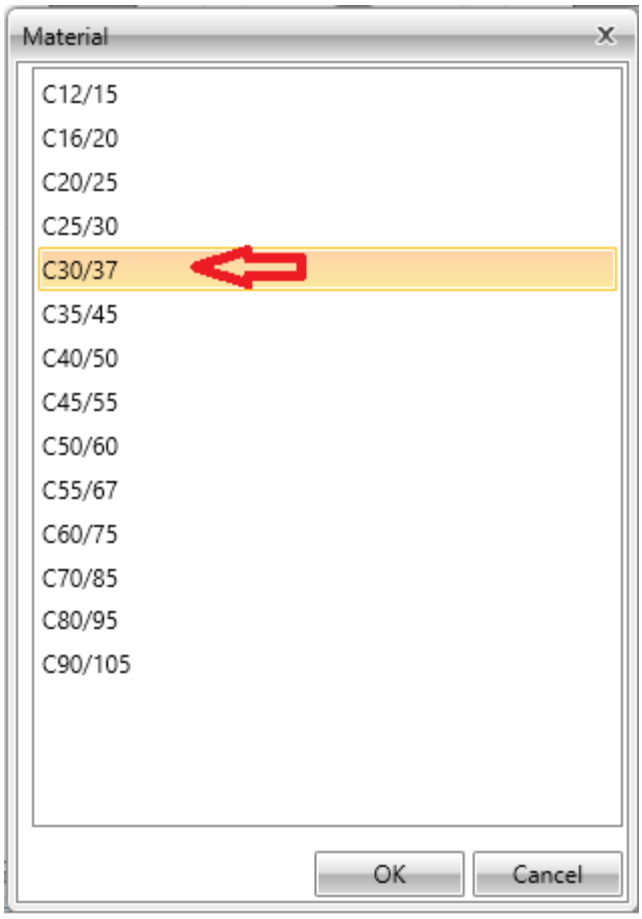
Dialog for definition of T-shaped cross-section appears.



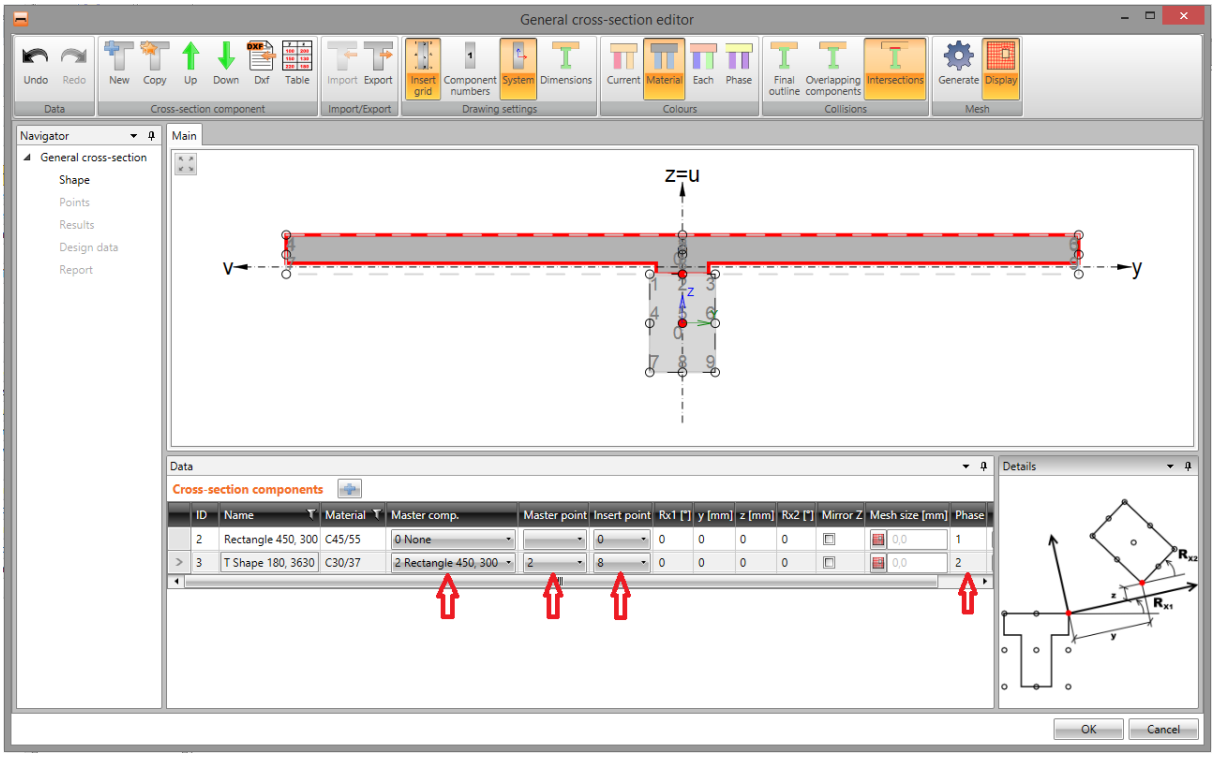
Set properties of T-shaped cross-section in **T-shape** dialog as follows:

- set cross-section height **h [mm]** to **180 mm;**
- set cross-section width **b [mm]** to **3630 mm;**
- set web width **bw [mm]** to **240 mm** ;

- set top flange thickness **hf [mm]** to **130 mm**.
- click  to add new material. Select concrete **C30/37**.



Click **OK** in **T-shape** dialog to finish input of composite slab cross-section.



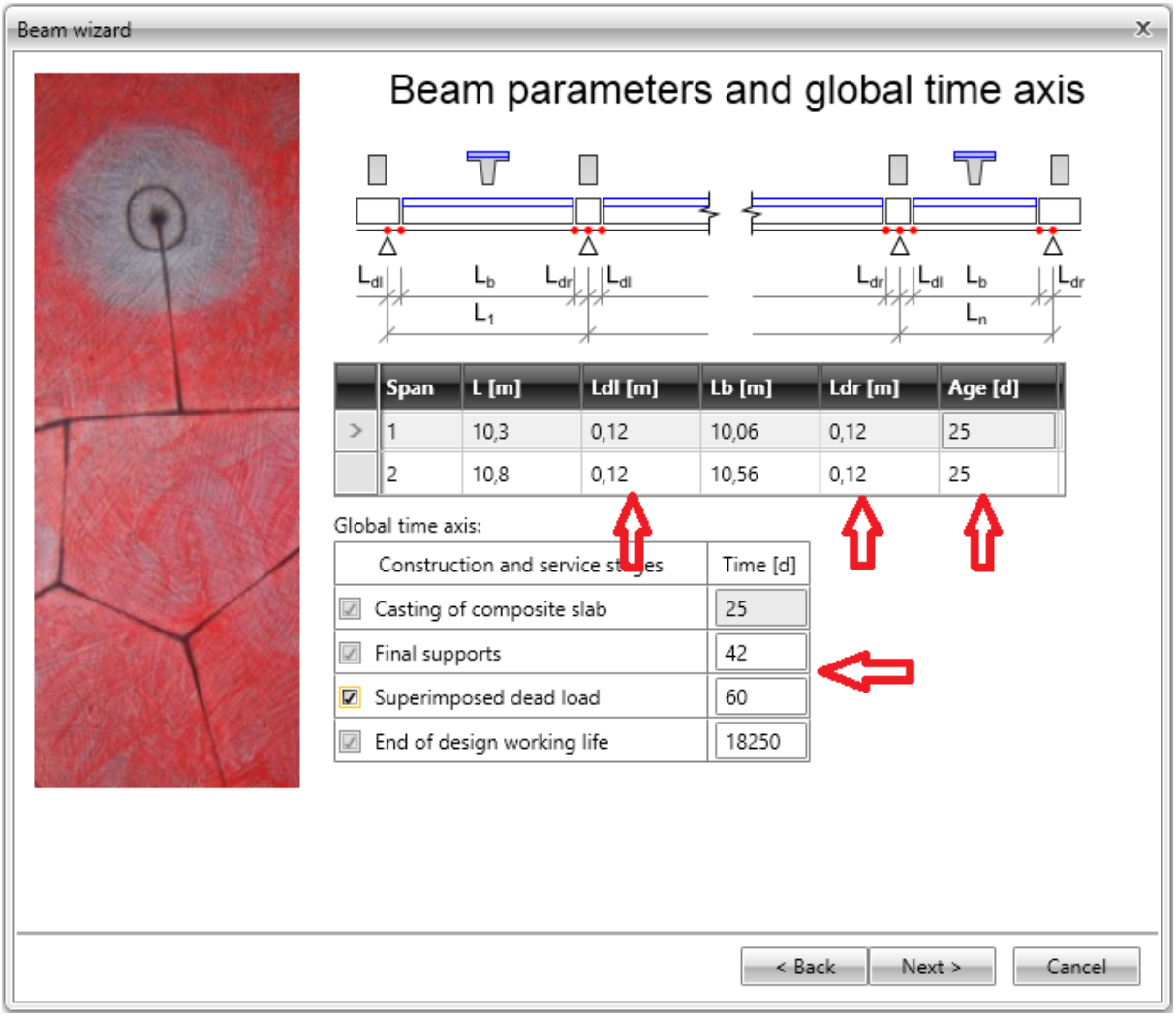
The next step is to change the position of composite slab component – to get the bottom edge of slab coincident with the top edge of the rectangular component of the first phase.

Select the second line (**T-shape**) in **Cross-section components** table and set component properties as follows:

- select **Rectangle 450,300** in column **Master comp.;**
- select number **2** in column Master point**;**
- select number **8** in column **Insert point**. The composite slab component moves so that point 8 of slab outline is coincident with point 2 of rectangle outline.
- verify the proper settings in **Phase** column - value **1** for **Rectangle** and value **2** for **T-shape**.

Click **OK** in **General cross-section editor** to finish input of composite cross-section.

Click **Next** in dialog **Code, cross-section, loads** to continue to the following wizard step.



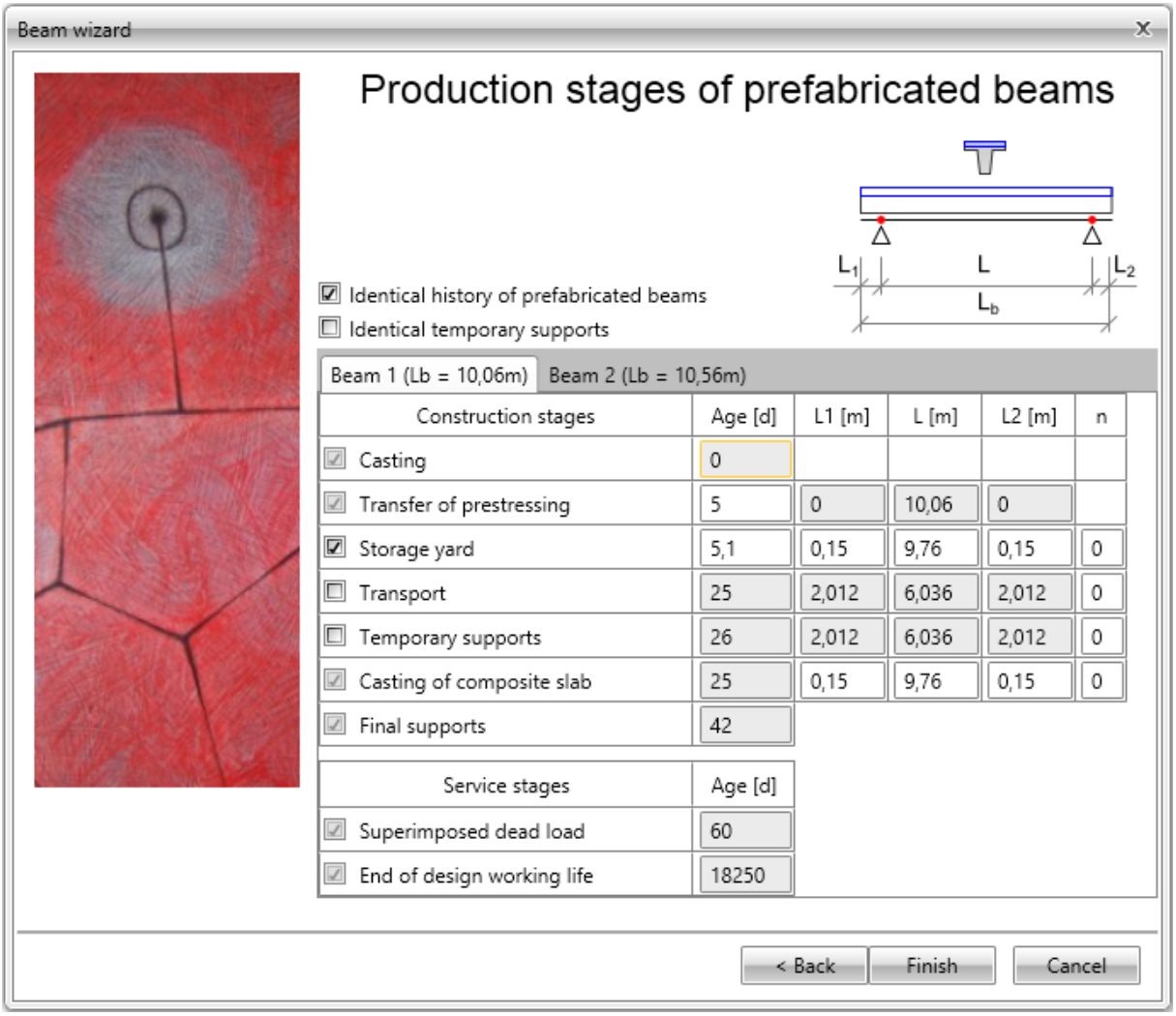
Set the table of spans properties in dialog **Beam parameters and global time axis** as follows:

- for both spans set the lengths of left adjacent diaphragms in column **Ldl [m]** to **0,12**;
- for both spans set the lengths of right adjacent diaphragms in column **Ldr [m]** to **0,12**. Lengths of prefabricated beams **Lb [m]** should recalculate automatically to **10,06** and **10,56**;
- set value in column **Age [d]** (equivalent age of prefabricated beams at the moment of casting of diaphragms and composite slab) to **25**.

Set times in the table **Global time axis** in column **Time [d]**:

- for **Final supports** stage set value **42;**
- for **Superimposed dead load** stage set value **60**.

Click **Next** to continue to the following wizard step.

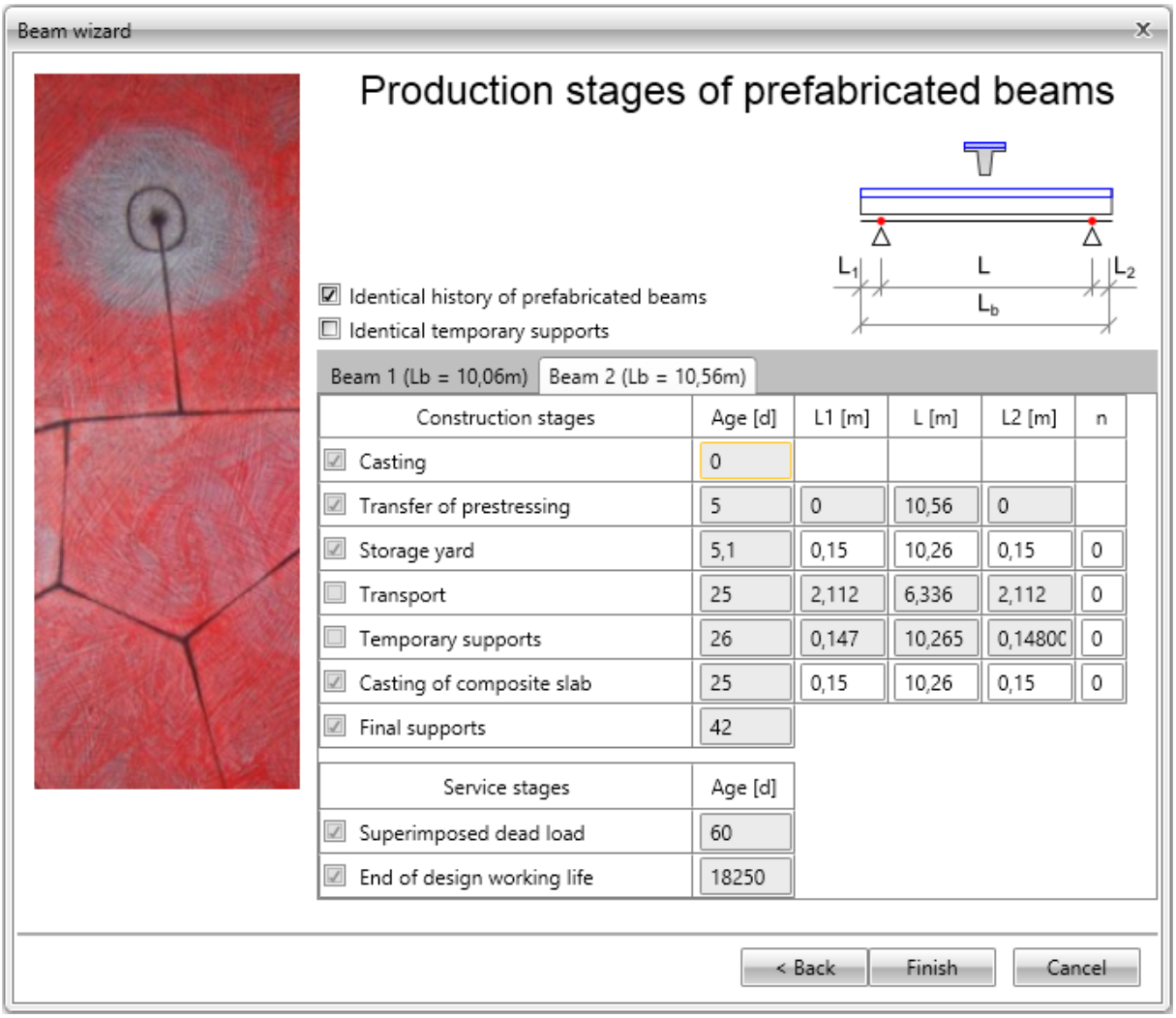


Set dialog **Production stages of prefabricated beams** as follows:

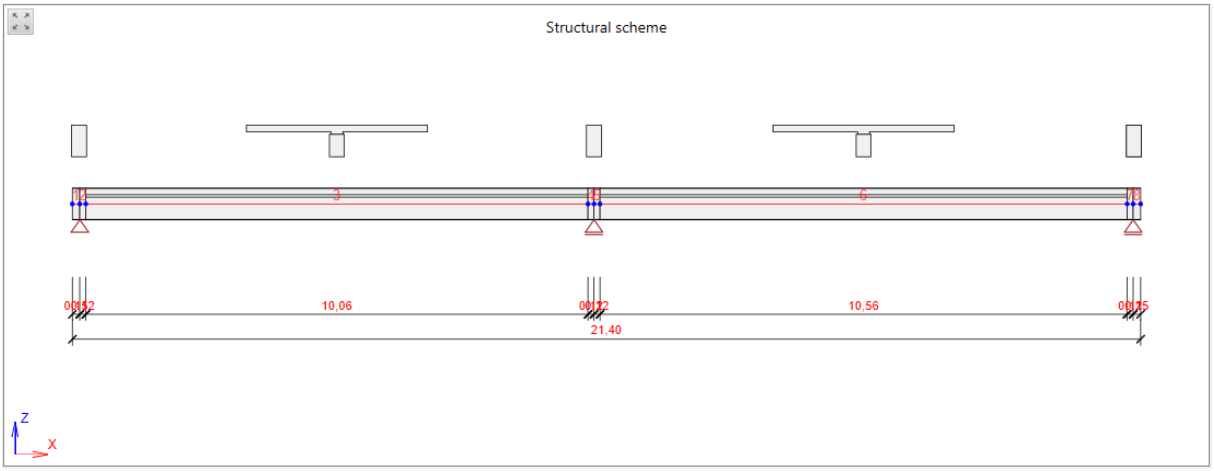
- select checkbox Identical history of prefabricated beams;
- select tab **Beam 1 (Lb = 10,06m)**;
- unselect stages **Transport** and **Temporary supports**.
- set value in column **Age[d]** for stage **Transfer of prestressing** to **5;**
- for stage **Storage yard** :
  - set value in column **Age[d]** to **5,1**;
  - set length of left diaphragm in column **L1 [m]** to **0,15**;
  - set length of right diaphragm in column **L2 [m]** to **0,15**;
  - set number of internal supports in column **n** to **0**;
- for stage **Casting of composite slab**:
  - set length of left diaphragm in column **L1 [m]** to **0,15**;
  - set length of right diaphragm in column **L2 [m]** to **0,15**;
  - set number of internal supports in column **n** to **0**;

- select tab **Beam 2 (Lb = 10,56m)**;
- for stage **Storage yard**:
  - set length of left diaphragm in column **L1 [m]** to **0,15**;
  - set length of right diaphragm in column **L2 [m]** to **0,15**;
  - set number of internal supports in column **n** to **0**;

- for stage **Casting of composite slab**:
  - set length of left diaphragm in column **L1 [m]** to **0,15**;
  - set length of right diaphragm in column **L2 [m]** to **0,15**;
  - set number of internal supports in column **n** to **0;**



Click **Finish** in dialog **Production stages of prefabricated beams** to finish the wizard. The beam data is generated.



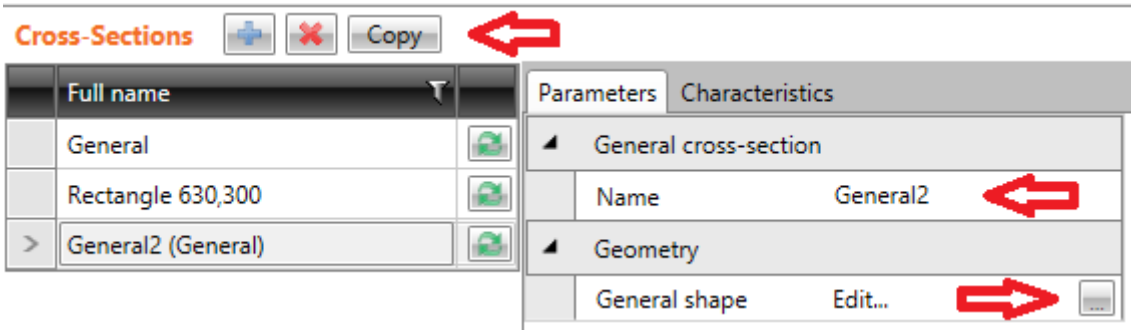
Data

Project data

Code and type settings	
Design code	EN
National annex	EN
EN 1992-2	☐
Type of bridge	No bridge
Type of beam	Prefabricated concrete beam
Prefabricated concrete beam	Composite pre-tensioned
Design working life	50 years
Geometry and loads settings	
Geometry and loads	Straight beam loaded in vertical plane
Flexible supports	☐
Ratio for unloaded spans	0
Beam alignment	Bottom surface
Supports position	Bottom surface

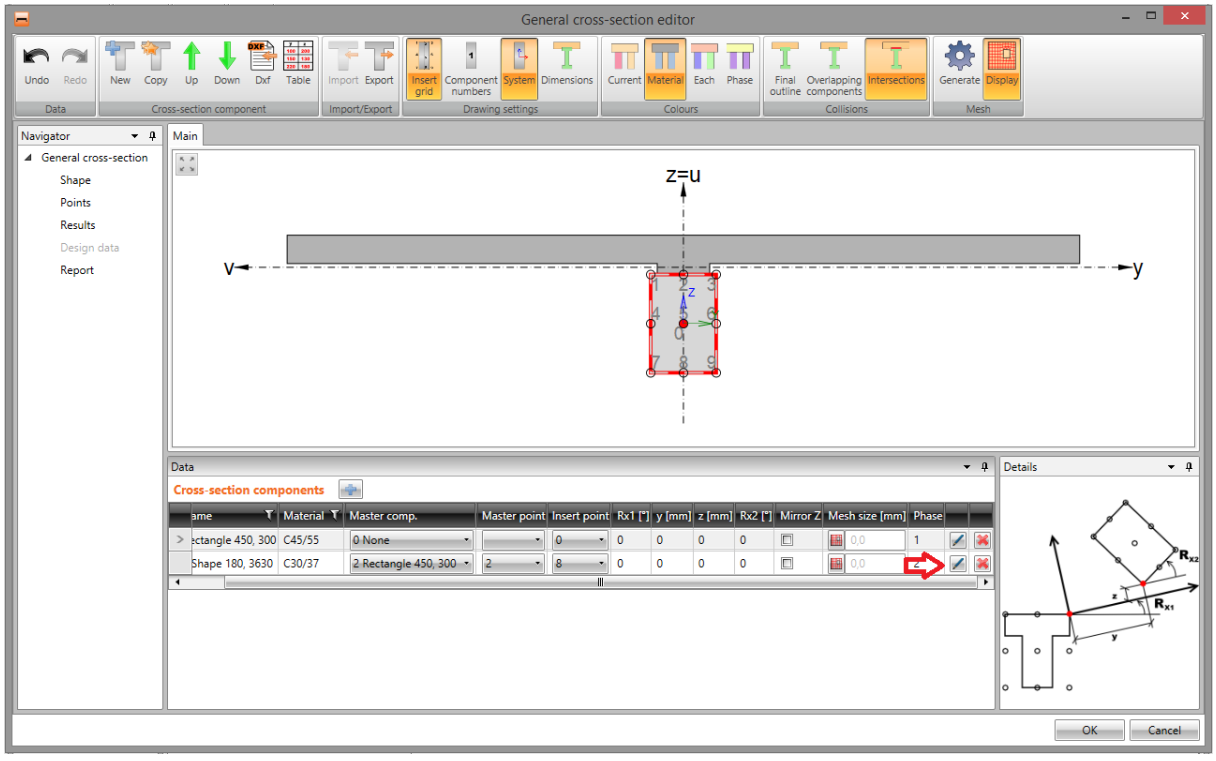
2.1.1 Modification of cross-sections in second beam span

Click navigator command **Project** > **Cross-sections** to start input of cross-sections. Select the first cross-section in the **Cross-sections** table (named **General**) and click **Copy** above the table to copy the selected cross-section.

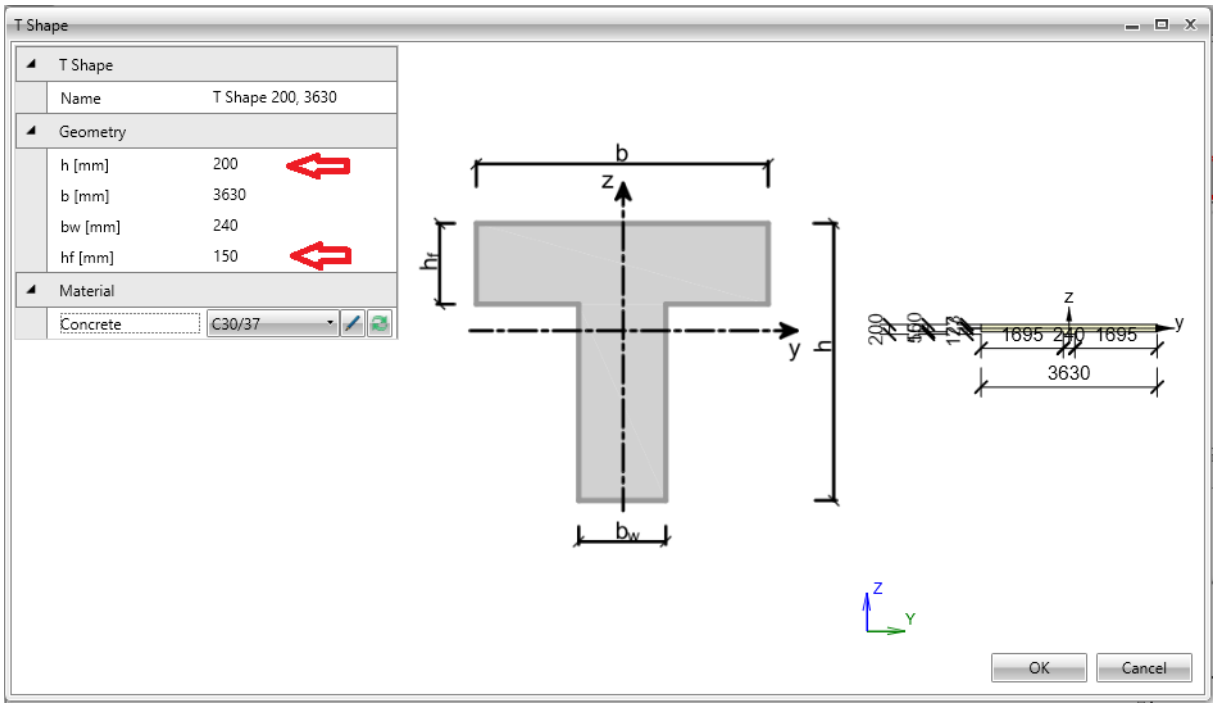


In the properties table on tab **Parameters**, rename the copied cross-section e.g. to **General2**. Click  to modify the cross-section.

Dialog **General cross-section editor** appears. Click  in the **T-shape** row of **Cross-section components** table to modify the properties of second component of cross-section – composite slab.



Dialog **T-shape** appears.



Modify T-shape cross-section properties as follows:

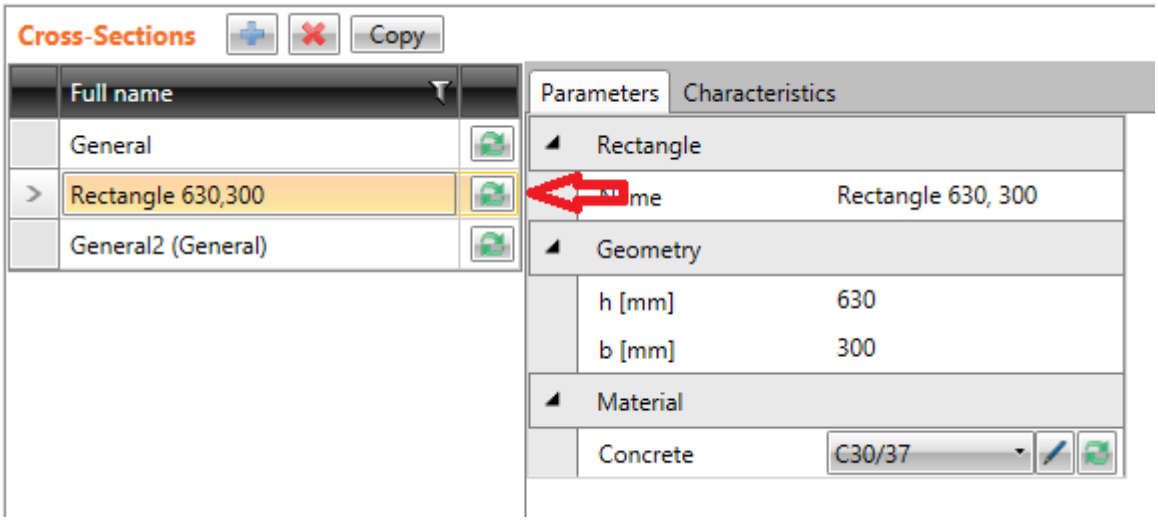
- set height **h [mm]** to **200;**
- set top flange thickness **hf [mm]** to **150 mm**.

Click **OK** in **T-shape** dialog to finish modification of composite slab component..

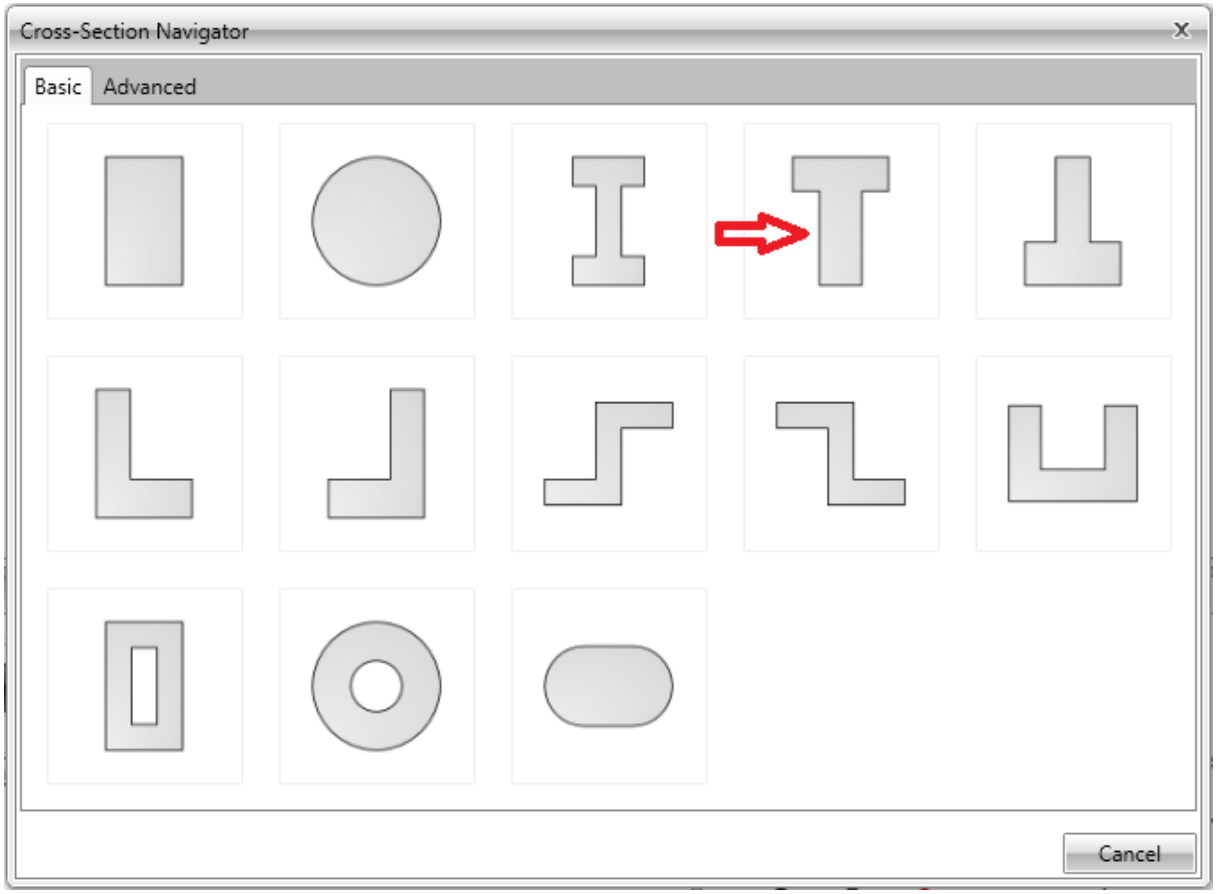
Click **OK** in **General cross-section editor** to finish modification of cross-section **General2**.

2.1.2 Modification of cross-sections of diaphragms

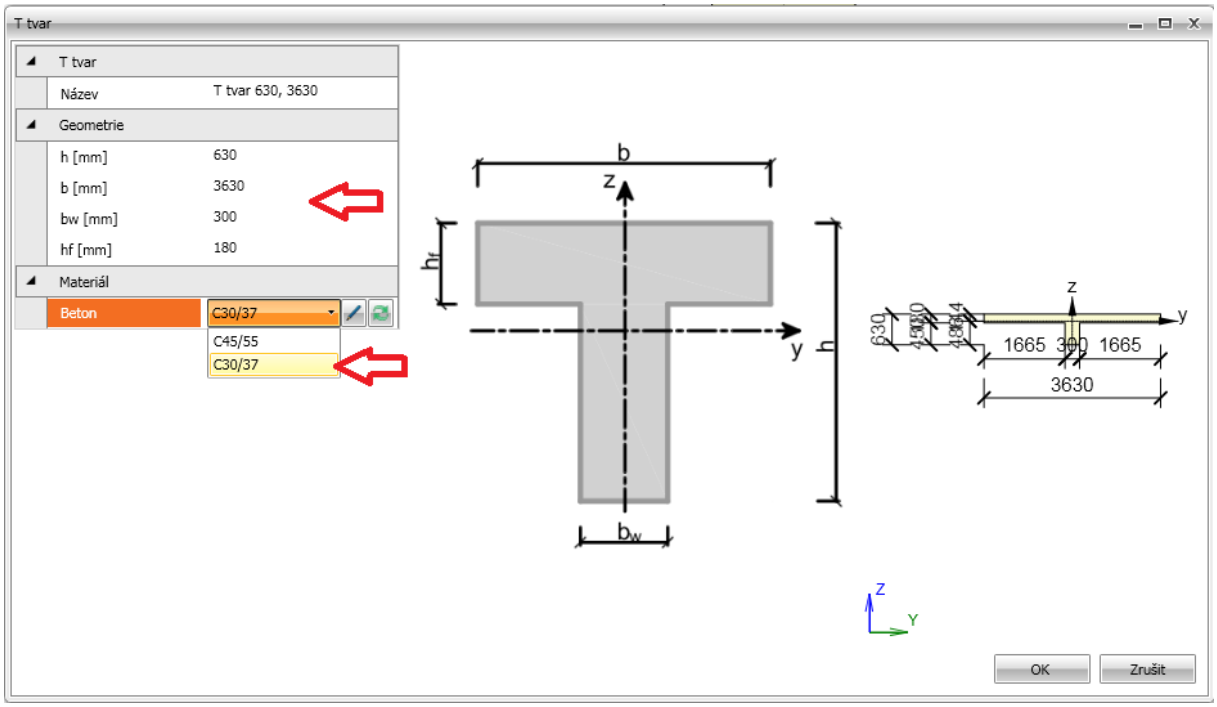
Click navigator command **Project > Cross-sections** to start input of cross-sections. Select the second cross-section in the **Cross-sections** table (named **Rectangle 630,300**) and click  to replace the cross-section by input of new cross-section.



Dialog **Cross-section navigator** appears. Click picture of T-shaped cross section to start input of diaphragm cross-section.



Dialog **T-shape** appears.



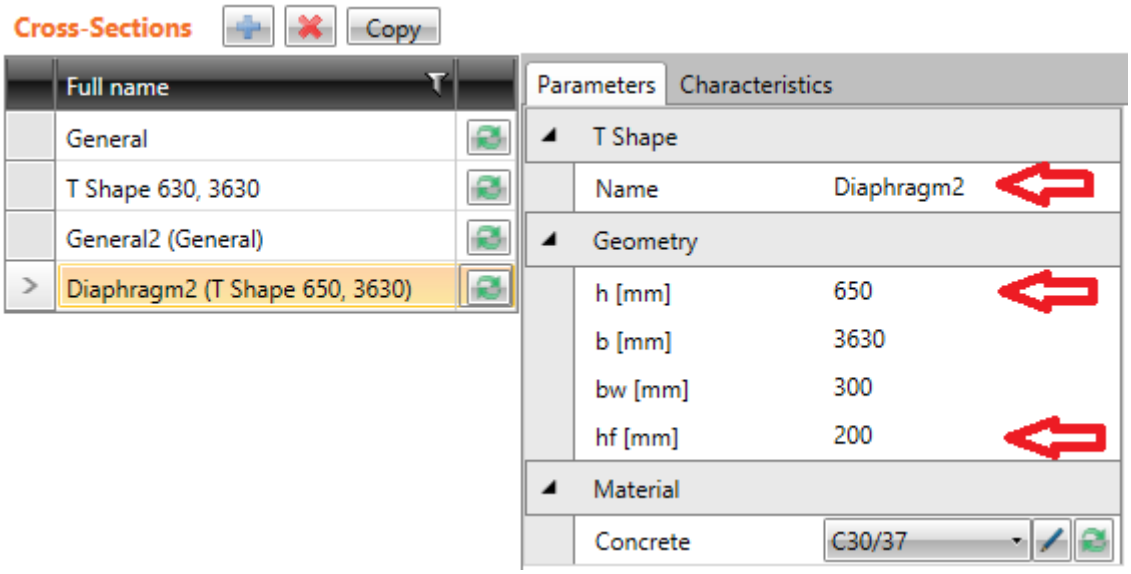
Set cross-section properties as follows:

- set height **h [mm]** to **630 mm**;
- set width **b [mm]** to **3630 mm**;
- set web thickness **bw [mm]**to **300 mm**;

- set top flange thickness **hf[mm]** to **180 mm**;
- select concrete **C30/37** in materials list.

Click **OK** in T-shape dialog to finish input of cross-section of diaphragm.

Select the second cross-section in the Cross-sections table (named **T-shape 630, 3630**) and click **Copy** above the table to copy the cross-section.



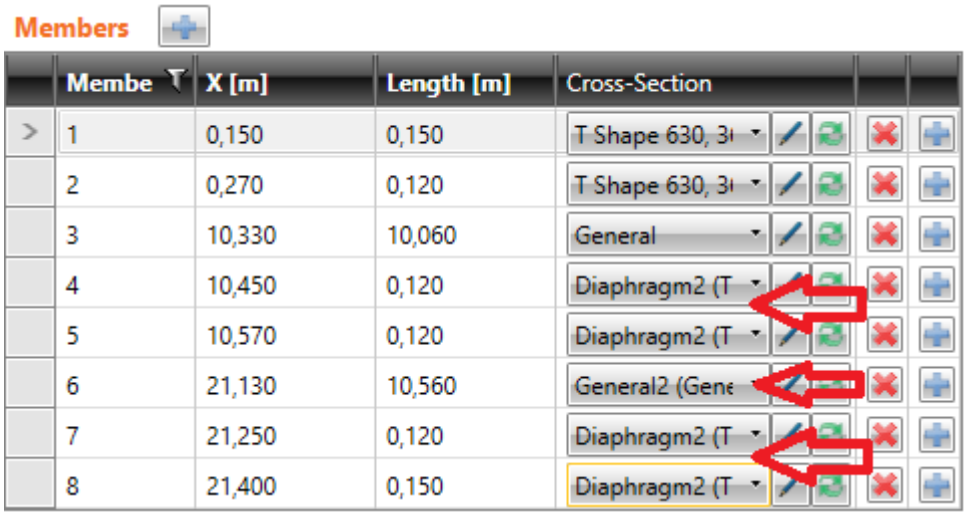
Modify following properties on the **Parameters** tab:

- change name to **Diaphraghm2;**
- set height **h [mm]** to **650 mm**;
- set thickness of top flange **hf[mm]** to **200 mm**.

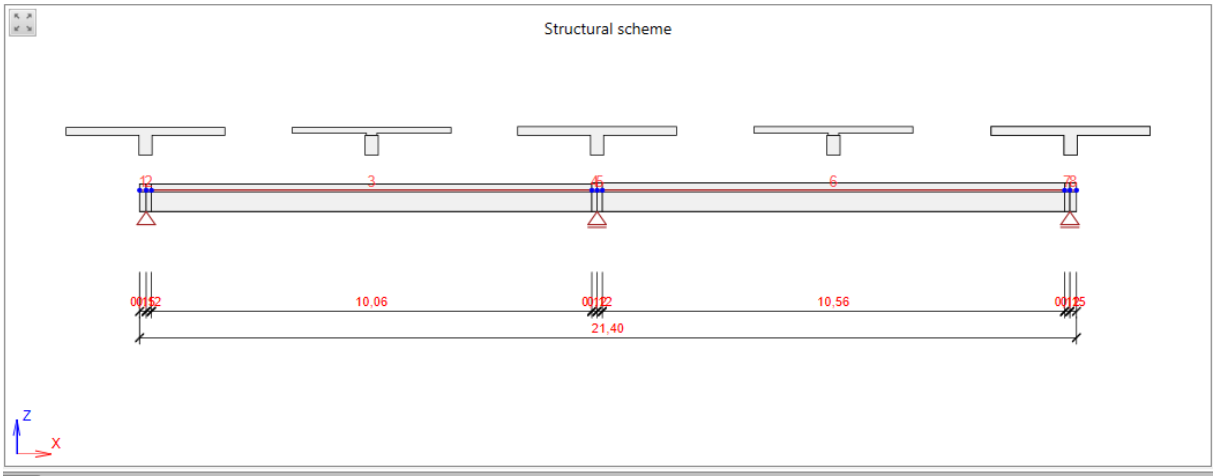
2.1.3 Modification of geometry

Click navigator command **Geometry** > **Members** to start modification of geometry. Change cross-sections assigned to members in the **Cross-section** column of **Members** table as follows:

- select **Diaphragm2** (fourth cross-section in the list) for members **4** and **5**;
- select **General2** (third cross-section in the list) for member **6**;
- select **Diaphragm2** for members **7** and **8**.



There is the beam after modification of cross-sections in the following picture:



Data

Members

	Membe	X [m]	Length [m]	Cross-Section
>	1	0,150	0,150	T Shape 630, 3
	2	0,270	0,120	T Shape 630, 3
	3	10,330	10,060	General
	4	10,450	0,120	Diaphragm2 (T
	5	10,570	0,120	Diaphragm2 (T
	6	21,130	10,560	General2 (Gene
	7	21,250	0,120	Diaphragm2 (T
	8	21,400	0,150	Diaphragm2 (T

Nodes

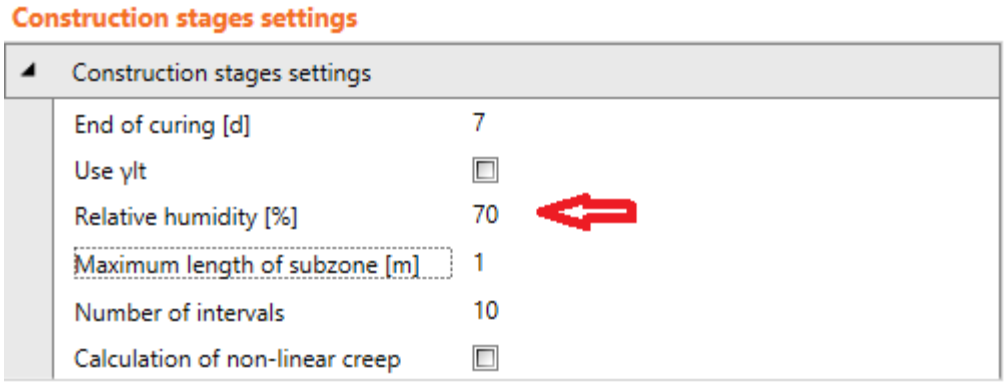
	Node	Nodal Support X	Z	Ry
>	1	☐	☐	☐
	2	☑	☑	☐
	3	☐	☐	☐
	4	☐	☐	☐
	5	☐	☑	☐
	6	☐	☐	☐
	7	☐	☐	☐
	8	☐	☑	☐
	9	☐	☐	☐

3 Construction stages

3.1 General settings

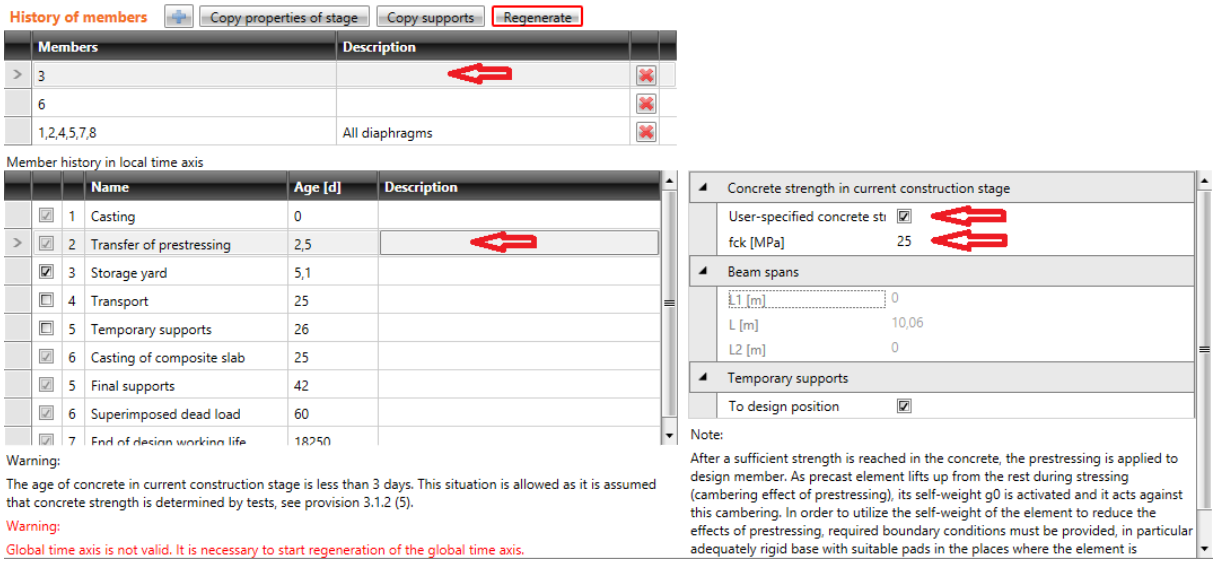
Click navigator command **Construction stages > Settings** to modify general settings of construction stages.

Set value of **Relative humidity** to **70%** in the properties table **Construction stages settings**.



3.2 History of members

Click navigator command **Construction stages > Members history** to modify the local time axes of individual structural members.



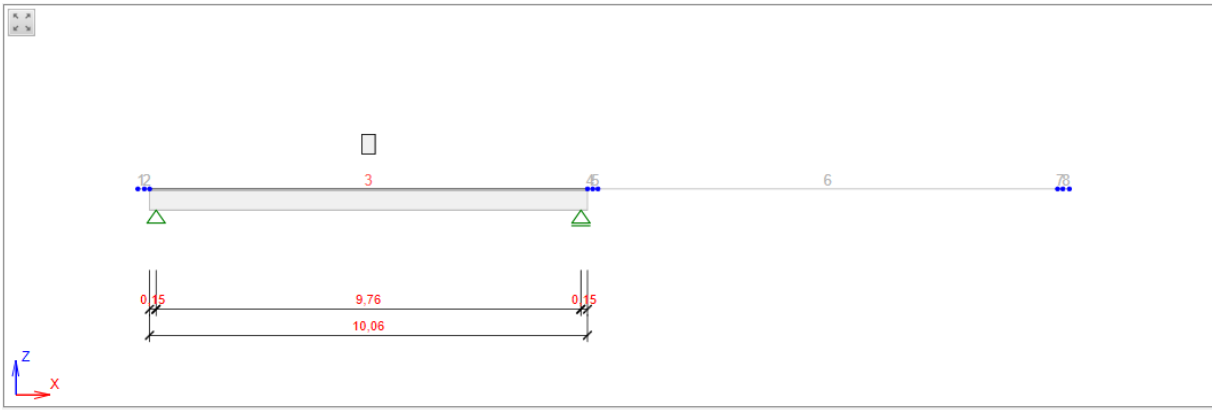
Select the first row in table **History of members**. This row represents the local history of prefabricated member number 3.

The history of selected member number 3 in its local time axis is described in the table **Member history in local time axis**. Select construction stage **2-Transfer of prestressing**.

In the properties table on the right select checkbox **User specified concrete strength**. Set the value of **fck [MPa]** to **25**. The age of stage **2-Transfer of prestressing** changes to **2,5** days.

Because the age of construction stage has changed, the global time axis gets invalid. Two warnings appear below the table. The first informs about concrete age less than 3 days. The second informs about invalid global time axis. The refresh of global time axis will be performed later.

Select construction stage **3-Storage yard** and change the value in column **Age [d]** to **2,7** days. The positions of temporary supports of prefabricated beam in this stage can be verified in properties table on right.



Data

History of members | Copy properties of stage | Copy supports | Regenerate

Members	Description
3	
6	
1,2,4,5,7,8	All diaphragms

Member history in local time axis

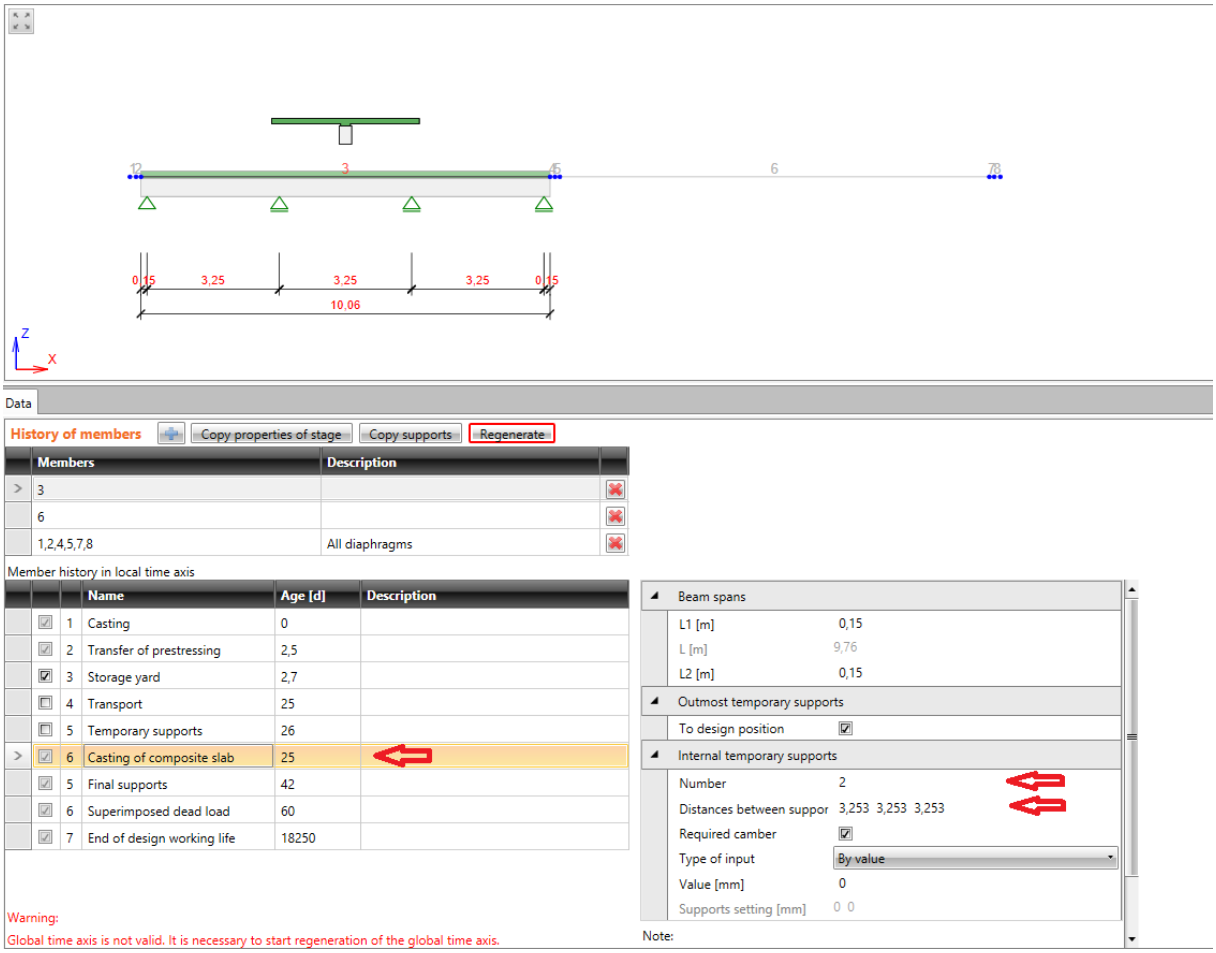
		Name	Age [d]	Description
☑	1	Casting	0	
☑	2	Transfer of prestressing	2,5	
☑	3	Storage yard	2,7	
☐	4	Transport	25	
☐	5	Temporary supports	26	
☑	6	Casting of composite slab	25	
☑	5	Final supports	42	
☑	6	Superimposed dead load	60	
☑	7	End of design working life	18250	

Beam spans	
L1 [m]	0,15
L [m]	9,76
L2 [m]	0,15
Outmost temporary supports	
To design position	☑
Internal temporary supports	
Number	0

Note:

Once prestressing is applied, the prefabricated elements are transferred to a storage yard where they are placed on a level, compacted and drained ground. They are stored in the same position as during production, piled one on top of another with a wooden or rubber stacking strips between individual layers.

Select construction stage **6 Casting of composite slab**.

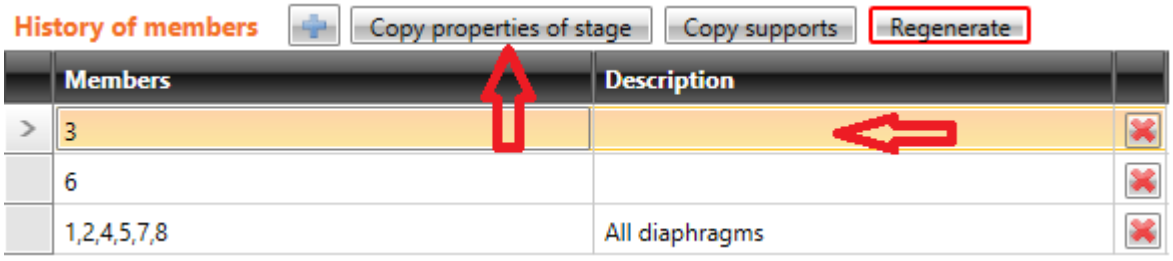


In the stage properties table on the right set the group **Internal temporary supports** as follows:

- Set value of **Numbe**r to **2**;
- Enter into **Distances between supports** the space delimited sequence of distances **3,88 2 3,88**.

Modification of history of member number **3** is finished. The history of member number **6** (prefabricated beam in the right span of beam) must be modified similarly.

Keep selected the first row in **History of members** table (member number **3**). Click Copy properties of stage above the table to copy times of stages from selected member number 3 to member number 6.



Select the second row in **History of members** table (member number **6**) and verify the properties of stages in **History of member in local time axis table**. The age of concrete in local time axis of member number **6** must be the same as the age of member number **3**.

History of members | Copy properties of stage | Copy supports | Regenerate

	Members	Description	
	3		
>	6		
	1,2,4,5,7,8	All diaphragms	

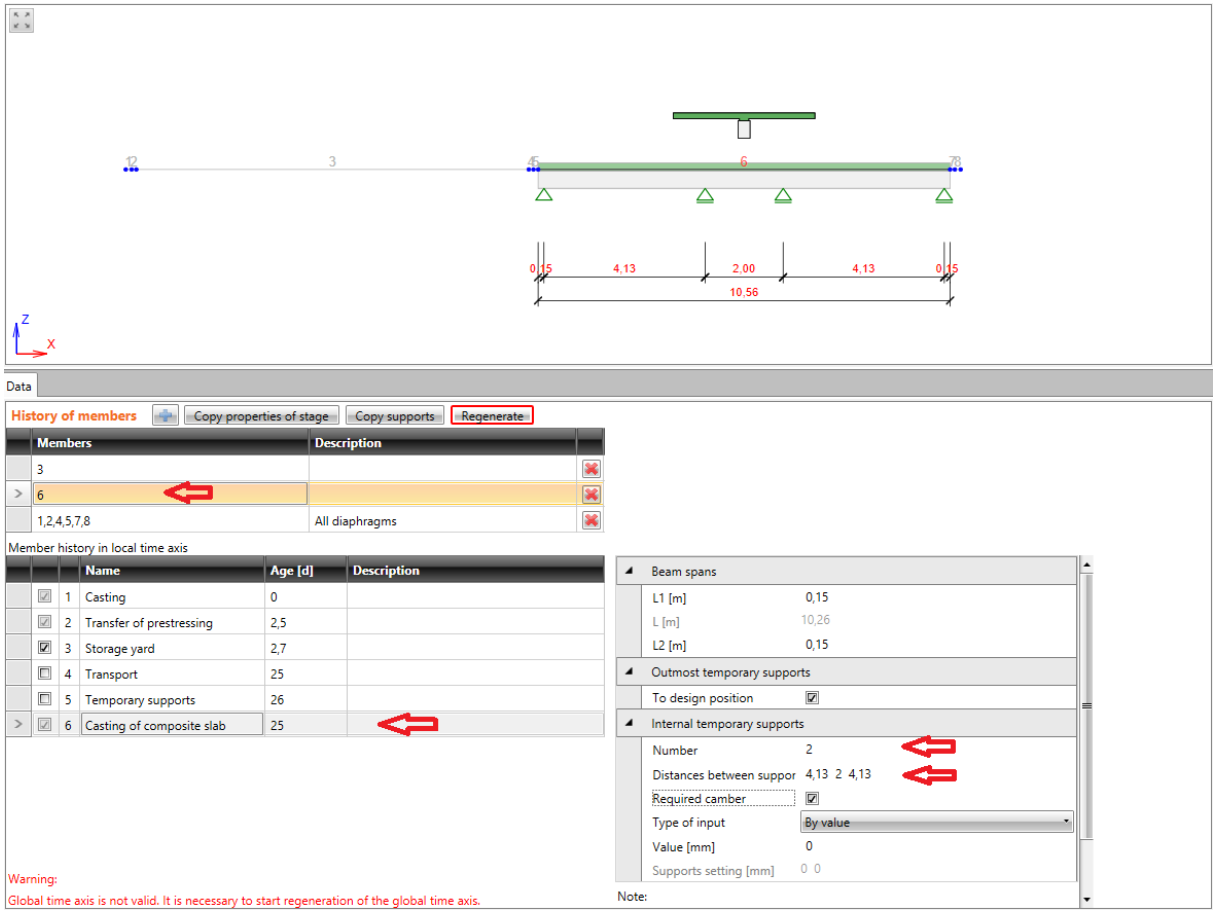
Member history in local time axis

			Name	Age [d]	Description
	☑	1	Casting	0	
>	☑	2	Transfer of prestressing	2,5	
	☑	3	Storage yard	2,7	
	☐	4	Transport	25	
	☐	5	Temporary supports	26	
	☑	6	Casting of composite slab	25	

Select stage **6 Casting of composite slab**.

In the stage properties table on the right set the group **Internal temporary supports** as follows:

- Set value of **Number** to **2**;
- Enter into **Distances between supports** the space delimited sequence of distances **4,13 2 4,13**.

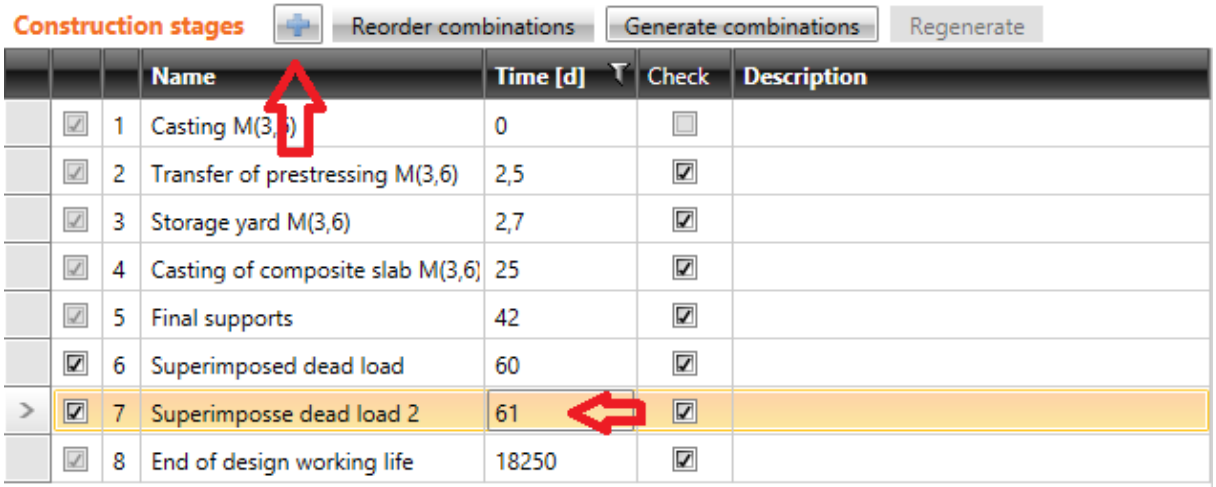


Modification of history of members in local time axes is finished. Click **Regenerate** above the **History of members** table to refresh the global time axis.

3.3 Construction stages

Click navigator command **Construction stages** > **Construction stages** to modify the structure history in global time axis.

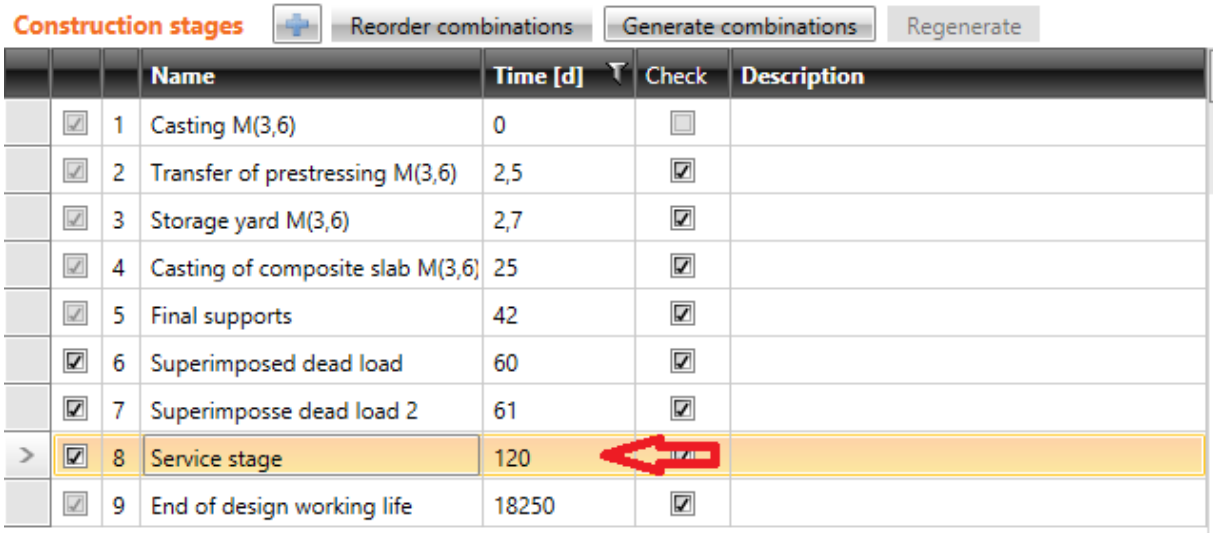
Click  above the table to add new service stage. The index of newly created stage is **7** and the predefined name is **Service stage**.



		Name	Time [d]	Check	Description
	1	Casting M(3,6)	0	☐	
	2	Transfer of prestressing M(3,6)	2,5	☑	
	3	Storage yard M(3,6)	2,7	☑	
	4	Casting of composite slab M(3,6)	25	☑	
	5	Final supports	42	☑	
	6	Superimposed dead load	60	☑	
>	7	Superimposse dead load 2	61	☑	
	8	End of design working life	18250	☑	

In the **Name** column of **Construction stages** table, change the name of newly created stage to **Superimposed dead load 2** and in the Time column set value **61** days. Appropriate load cases and combinations are assigned to the new stage automatically.

Click  above the table to add new service stage. The index of newly created stage is **8** and the predefined name is **Service stage 2**.



		Name	Time [d]	Check	Description
	1	Casting M(3,6)	0	☐	
	2	Transfer of prestressing M(3,6)	2,5	☑	
	3	Storage yard M(3,6)	2,7	☑	
	4	Casting of composite slab M(3,6)	25	☑	
	5	Final supports	42	☑	
	6	Superimposed dead load	60	☑	
	7	Superimposse dead load 2	61	☑	
>	8	Service stage	120	☑	
	9	End of design working life	18250	☑	

Modify name of stage to **Service stage** and set the time of stage to **120** days. Appropriate load cases and combinations are assigned to the new stage automatically.

4 Loads

4.1 Load cases

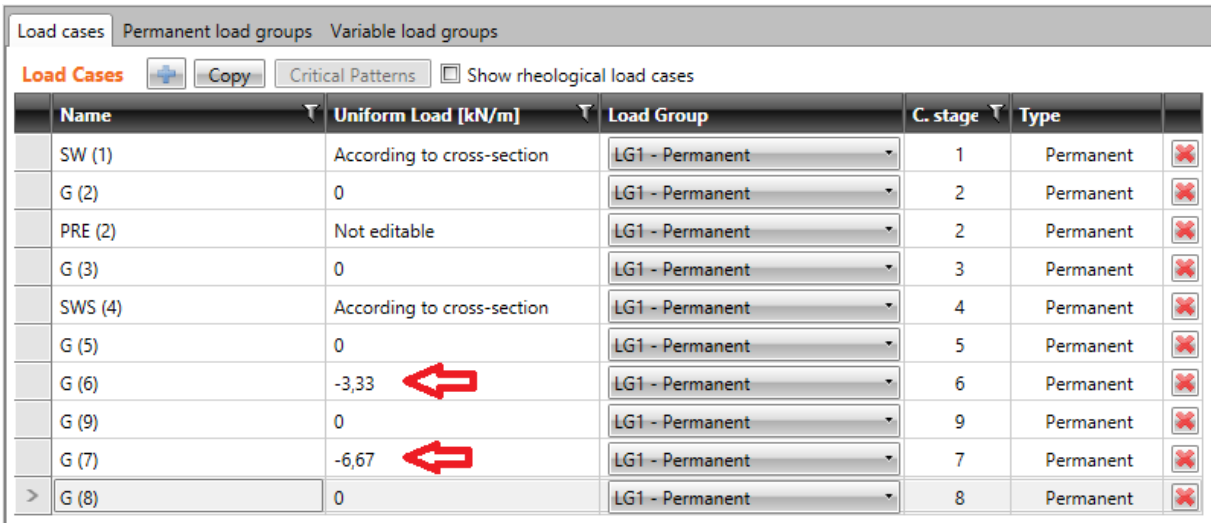
Click navigator command **Loads > Load cases** to edit load cases.

There are 10 load cases in the project, created by project wizard and by adding new service stages:

- **SW (1)** – load case for automatically generated self-weight of first phase of composite sections on members **3** and **6**. The load case is assigned to the construction stage **Casting**.
- **G (2), G (3), G (4), G (5), G (6), G (7), G (8), G (9)** – load cases for permanent loads. Load cases are assigned to appropriate construction stages except the stage **Casting**. No load cases have been assigned to the cases yet.
- **PRE (2)** – load case for transfer of effects of pre-stressing from pre-stressed tendons to the structural model. The load case is assigned to the stage **Transfer of prestressing**.

Into the **Uniform load [kN/m]** column of **Load cases** table, input the value **-3,33** kN/m of uniform continuous load, which acts on all members of the beam, into the load case **G(6)** – load case is assigned to stage **Superimposed dead load**.

Input the value **-6,67** kN/m of uniform continuous load, which acts on all members of the beam, into the load case **G(7)** – load case is assigned to stage **Superimposed dead load2**.



Name	Uniform Load [kN/m]	Load Group	C. stage	Type
SW (1)	According to cross-section	LG1 - Permanent	1	Permanent
G (2)	0	LG1 - Permanent	2	Permanent
PRE (2)	Not editable	LG1 - Permanent	2	Permanent
G (3)	0	LG1 - Permanent	3	Permanent
SWS (4)	According to cross-section	LG1 - Permanent	4	Permanent
G (5)	0	LG1 - Permanent	5	Permanent
G (6)	-3,33	LG1 - Permanent	6	Permanent
G (9)	0	LG1 - Permanent	9	Permanent
G (7)	-6,67	LG1 - Permanent	7	Permanent
G (8)	0	LG1 - Permanent	8	Permanent

Click  above the Load cases table to create a new load case. The newly created load case is appended to the end of the table.

Load Cases | Copy | Critical Patterns | Show rheological load cases

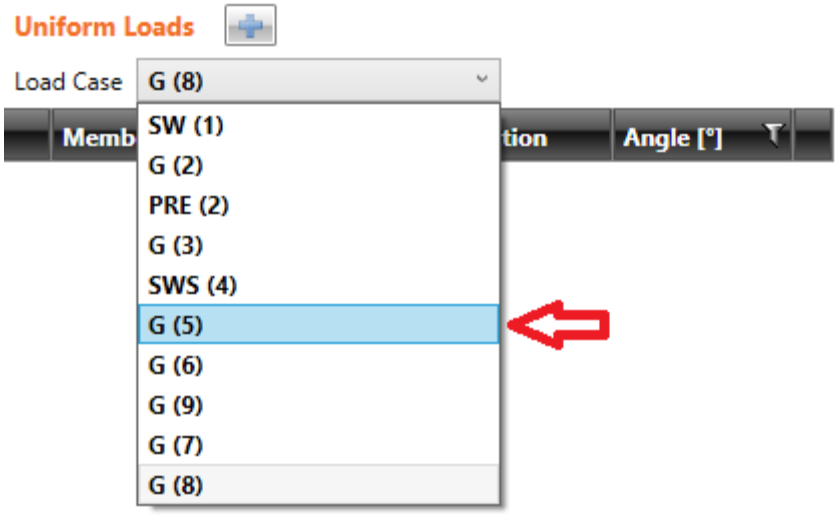
	Name	Uniform Load [kN/m]	Load Group	C. stage	Type	
	SW (1)	According to cross-section	LG1 - Permanent	1	Permanent	
	G (2)	0	LG1 - Permanent	2	Permanent	
	PRE (2)	Not editable	LG1 - Permanent	2	Permanent	
	G (3)	0	LG1 - Permanent	3	Permanent	
	SWS (4)	According to cross-section	LG1 - Permanent	4	Permanent	
	G (5)	0	LG1 - Permanent	5	Permanent	
	G (6)	-3,33	LG1 - Permanent	6	Permanent	
	G (9)	0	LG1 - Permanent	9	Permanent	
	G (7)	-6,67	LG1 - Permanent	7	Permanent	
	G (8)	0	LG1 - Permanent	8	Permanent	
>	Variable	-12,5	LG3 - Standard		Variable	

Modify properties of the new load case in **Load cases** table:

- change the load case name in **Name** column to **Variable**.
- set the value in **Uniform load [kN/m]** to **-12.5** kN/m.
- set the type of load in **Type** column to **Variable**.

4.2 Dead loads on members

Click navigator command Loads > Uniform loads to input uniform loads on individual members. Select load case **G(5)** as current load case – the load case is assigned to construction stage **Final supports**.



Click  above the table to add a new uniform load on member number **1**.

Set the value in column **Size [kN/m]** to **-13,83** kN/m.

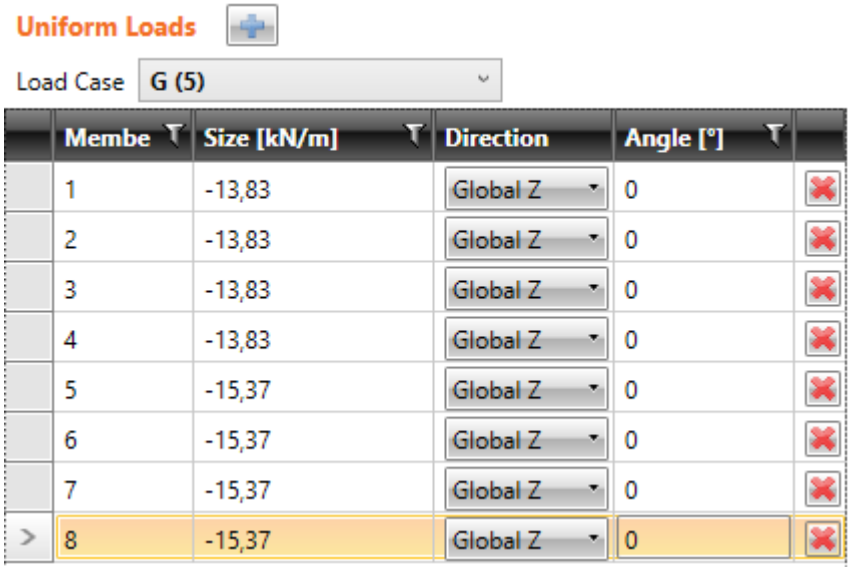
Click  above the table to add new uniform load on member number **2**. Load size is copied from the previous row.

Click  above the table to add new uniform load on member number **3**. Load size is copied from the previous row.

Click  above the table to add new uniform load on member number **4**. Set the size of load to **-15.37** kN/m.

Click  above the table to add new uniform load on member number **5**. Keep the load size **-15.37** kN/m.

Input the same load on members **6**, **7** and **8**.



Click navigator command **Loads > Point forces** to start input of point forces.

Select load case **G(6)** as the current load case– load case is assigned to construction stage **Superimposed dead load**. The uniform load **-3,33** kN/m acting on all members has been defined in this load case already.

Click  above the table to add a new point load.

Modify the new load as follows:

- set value in column **Member** to **6**;
- set value in column **Size** to **-16,67** kN;
- set value in column **Position** to **P 1/2**.

Select load case **G(7)** as the current load case– load case is assigned to construction stage **Superimposed dead load 2**. The uniform load **-6,67** kN/m acting on all members has been defined in this load case already.

Click  above the table to add a new point load.

Modify the new load as follows:

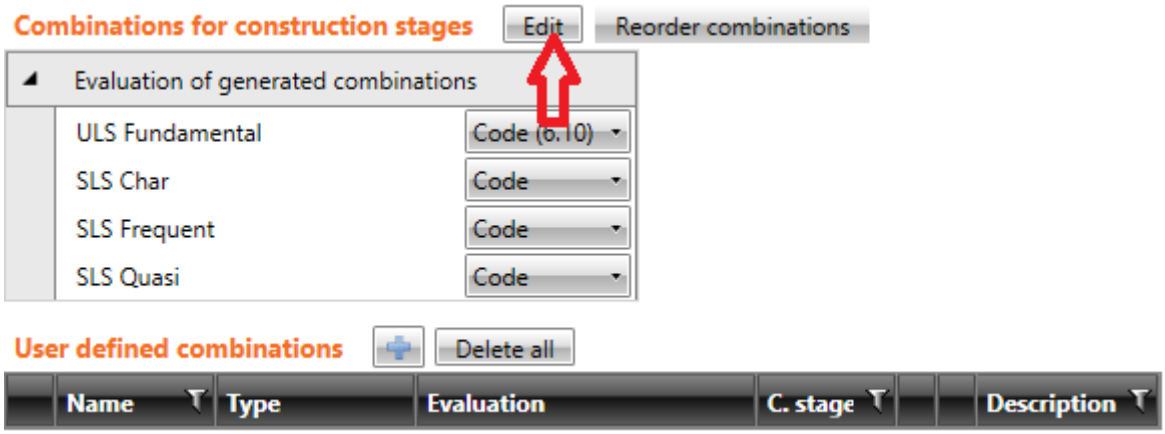
- set value in column **Member** to **6**;
- set value in column **Size** to **-33,33** kN;
- set value in column **Position** to **P 1/2**.

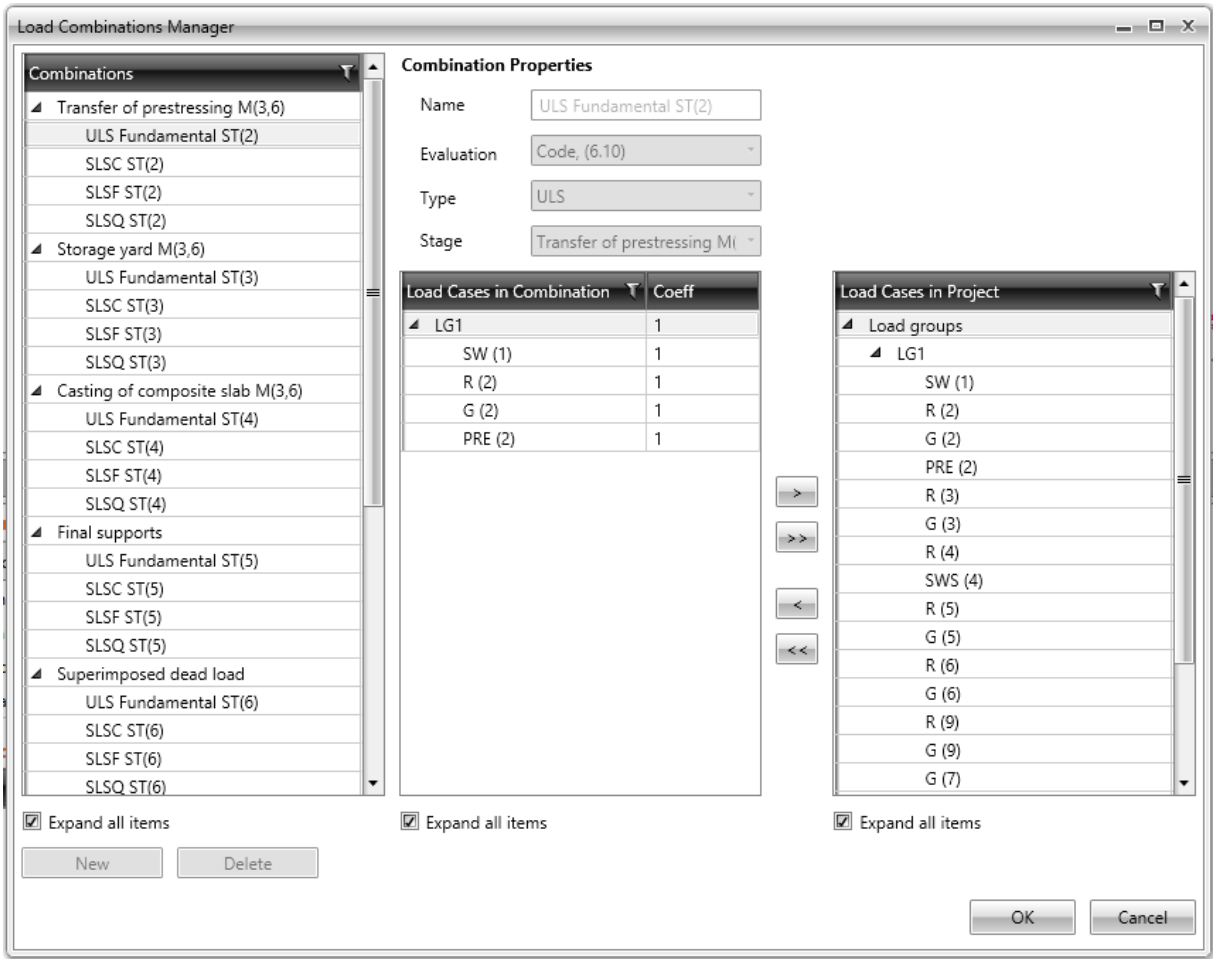
4.3 Assigning new variable load case to combinations

Combinations of load cases for all construction stages are generated automatically. New variable load case must be assigned to combinations for construction stages **Service stage** and **End of working life**.

Click navigator command **Loads > Combinations** to define combinations of load cases.

Load cases are assigned to combinations using **Load combinations manager**. Click **Edit** above the table **Combinations for construction stages** to launch the combinations manager.





On the left, existing combinations categorized by construction stage are displayed in the tree control **Combinations**.

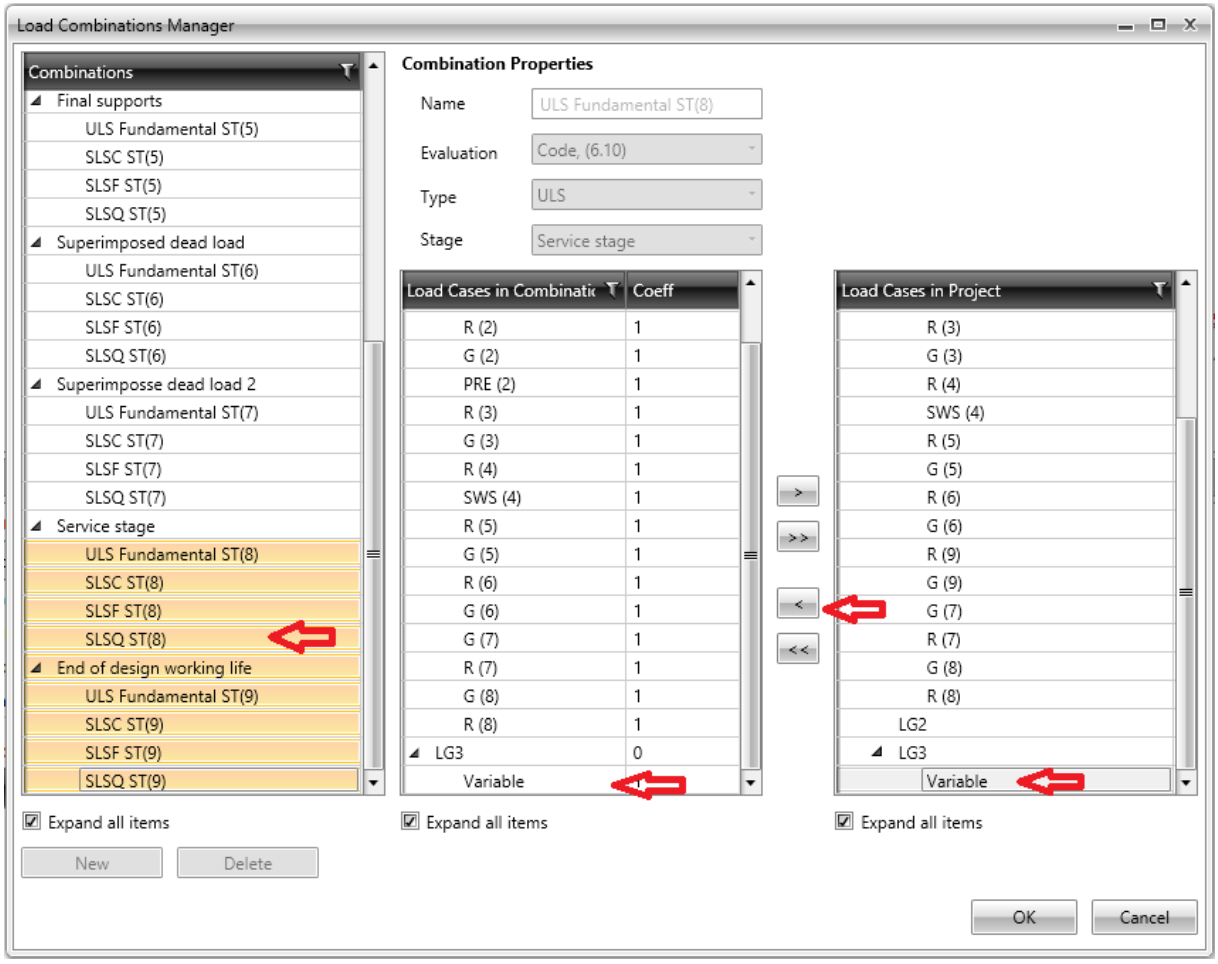
In the middle, properties for the selected combinations are displayed in the group **Combination properties**. Tree control **Load cases in combinations** displays list of load cases assigned to the combination, categorized by the load group and the coefficients of load cases in combination.

On the right, list of existing load cases categorized by load group is displayed in the tree control Load cases in project.

General process to modify the combination:

- Select required combination in the **Combinations** list**.**
- Add required load cases - select the load cases in the **Load cases in project list** and drag/drop it to the **Load cases in combination list** or click  .
- Remove not required load cases from combination – select load cases in **Load cases in combination** list and drag/drop it to **Load cases in project** list or click 
- Modify value of coefficients in **Load cases in combination** list.

Load cases can be added/removed to more selected combinations at once.



In the **Combinations** tree view, select all combinations assigned to construction stages **Service stage** and **End of working life**.

In the **Load cases in project** tree view select load case **Variable** and click  to add the load case to all selected combinations.

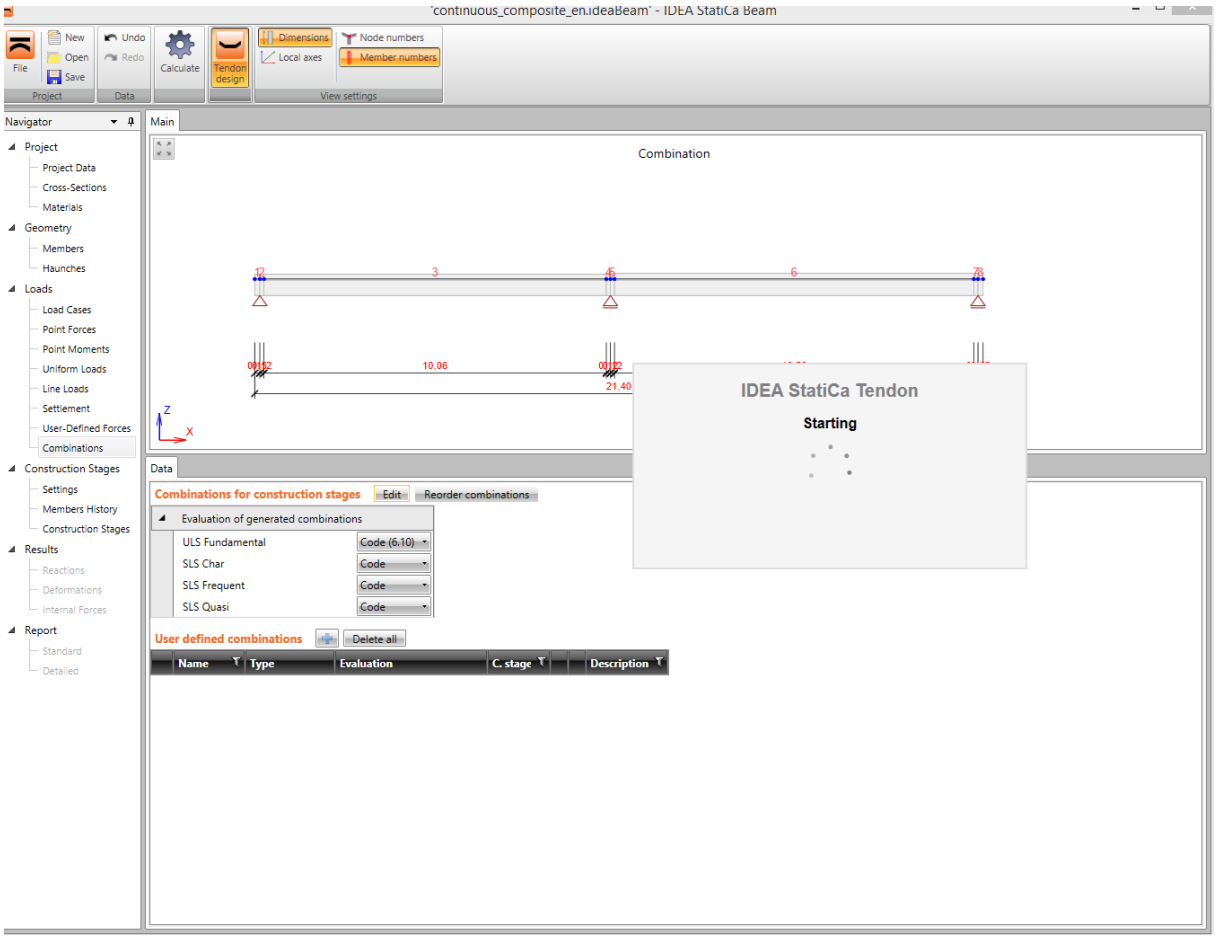
Click **OK** to close the combinations manager.

5 Design of tendons in IDEA Tendon

5.1 Launching IDEA Tendon

Click Tendon design in ribbon to launch IDEA Tendon.

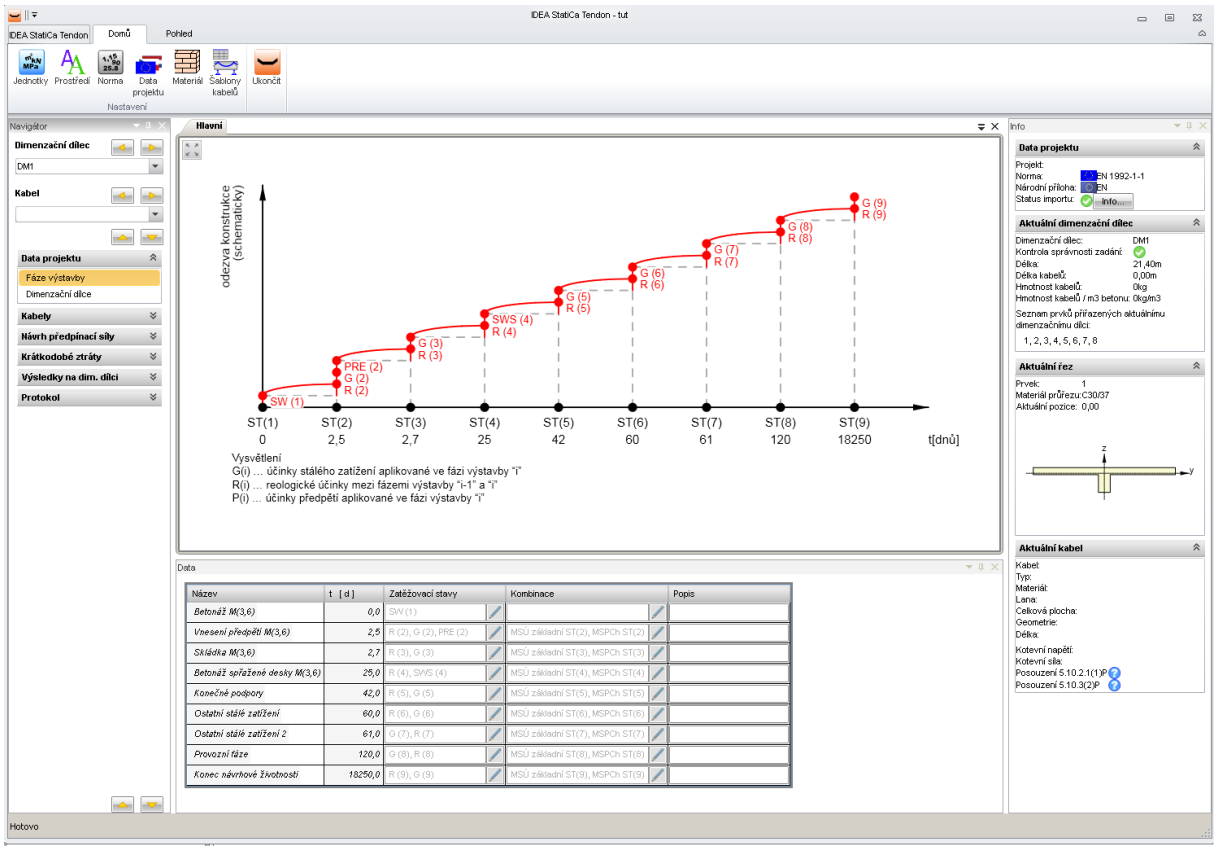
While launching IDEA Tendon the data from IDEA Beam analysis model are converted to IDEA Tendon data model.



Data validity is verified import into IDEA Tendon.

5.2 Input of construction stages

Navigator command **Project data** > **Construction stages** is opened automatically after the data is imported.



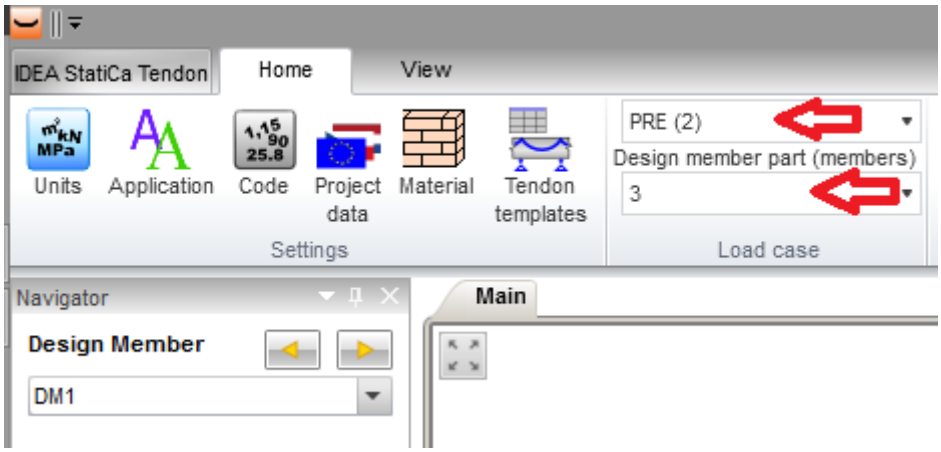
Definition of construction stages in IDEA Tendon is taken from definition of construction stages in including assignment of load cases and combinations of load cases to construction stages.

Name	t [d]	Load cases	Combinations	Description
Casting M(3,6)	*0,0*	SW (1)		
Transfer of prestressing M(3,6)	*2,5*	R (2), G (2), PRE (2)	ULS Fundamental ST(2), SLSC STi	
Storage yard M(3,6)	*2,7*	R (3), G (3)	ULS Fundamental ST(3), SLSC STi	
Casting of composite slab M(3,6)	*25,0*	R (4), SWS (4)	ULS Fundamental ST(4), SLSC STi	
Final supports	*42,0*	R (5), G (5)	ULS Fundamental ST(5), SLSC STi	
Superimposed dead load	*60,0*	R (6), G (6)	ULS Fundamental ST(6), SLSC STi	
Superimposse dead load 2	*61,0*	G (7), R (7)	ULS Fundamental ST(7), SLSC STi	
Service stage	*120,0*	G (8), R (8)	ULS Fundamental ST(8), SLSC STi	
End of design working life	*18250,0*	R (9), G (9)	ULS Fundamental ST(9), SLSC STi	

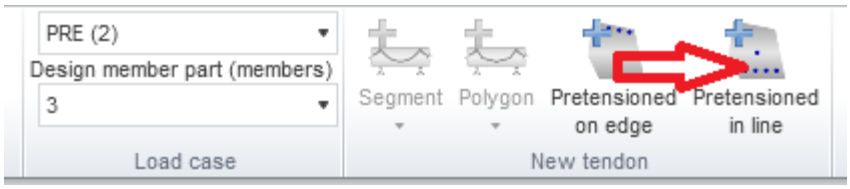
5.3 Input of tendons

5.3.1 Input of pre-tensioned tendons group to member 3

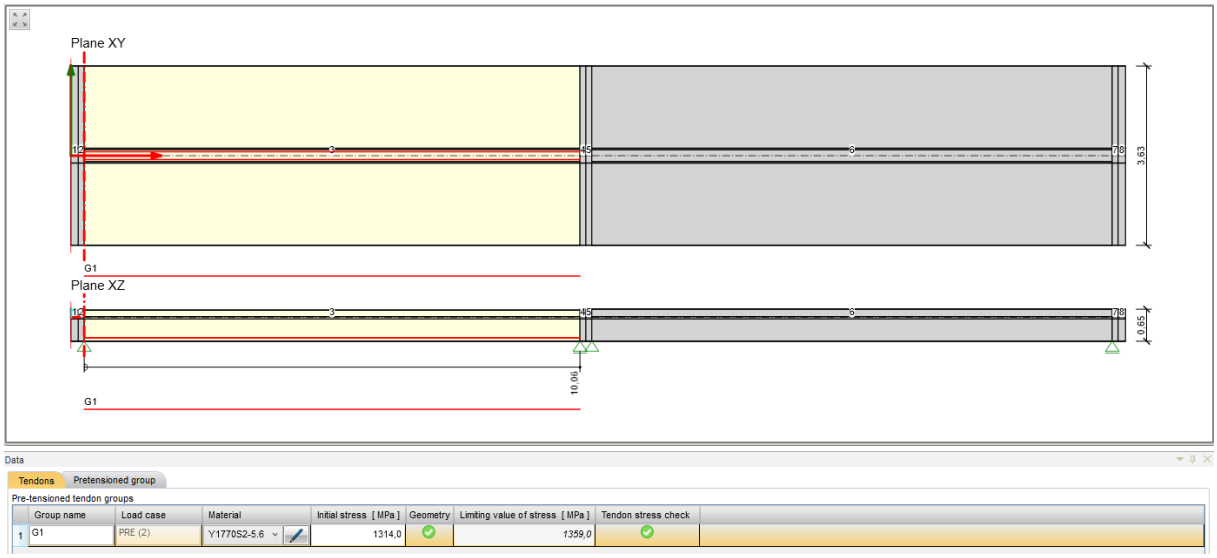
Click navigator command **Tendons > Tendons layout** to start input of tendons. In ribbon group Load case, select load case for transfer of prestressing **PRE (2)** and in the list **Part of design member** select member number **3** to input tendons to it.



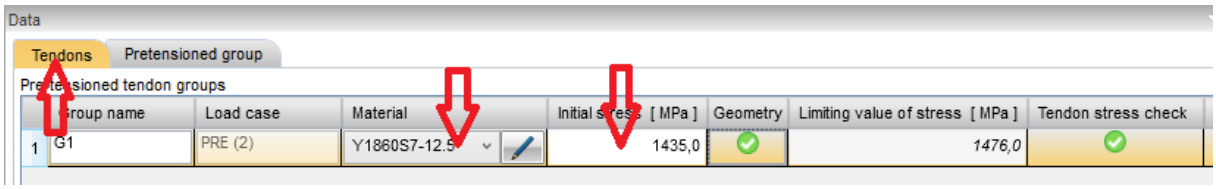
Click **Pretensioned in line** in ribbon group **New tendon** to add new group of pre-tensioned tendons to member **3**.



New tendon is drawn in uncoiled views XY and XZ.



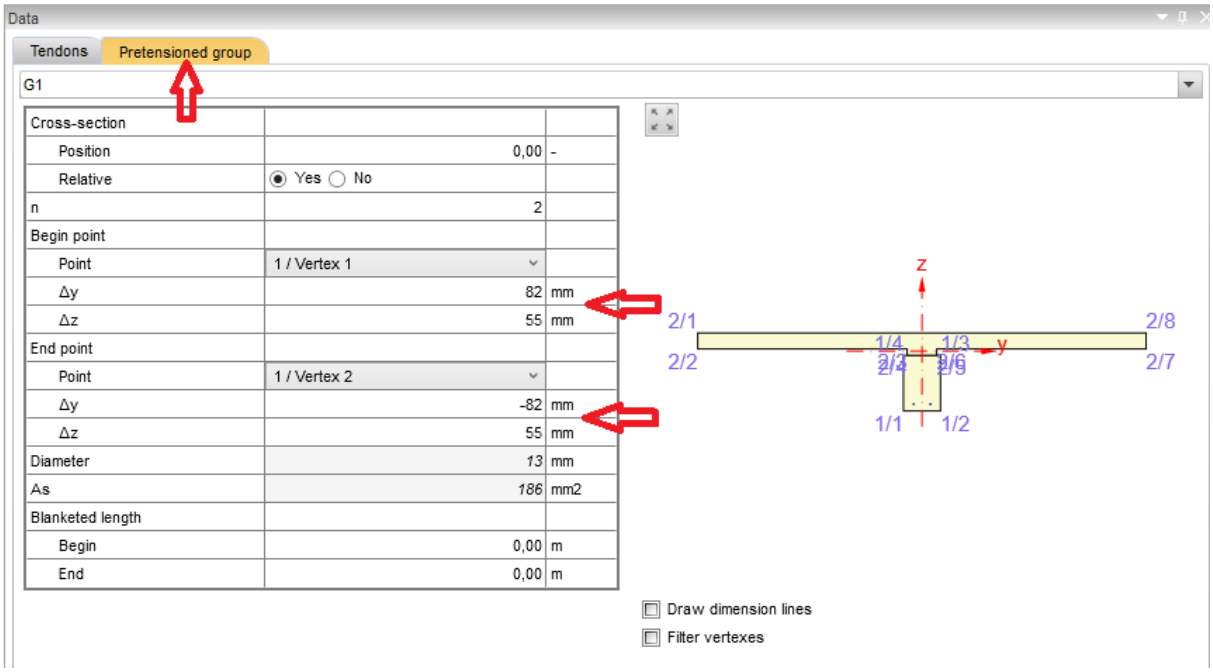
Properties of new tendon must be modified. Tendon properties table is displayed on tab **Tendons** in the **Data window**.



Modify tendon properties in table row **G1** as follows:

- from the list in column **Material** select tendon material **Y1860S7-12.5**.
- set value in column in column **Initial stress** to **1435** MPa.

Position of pre-tensioned tendons in the cross-section can be modified in properties table on the tab **Pretensioned group** in the **Data window**.



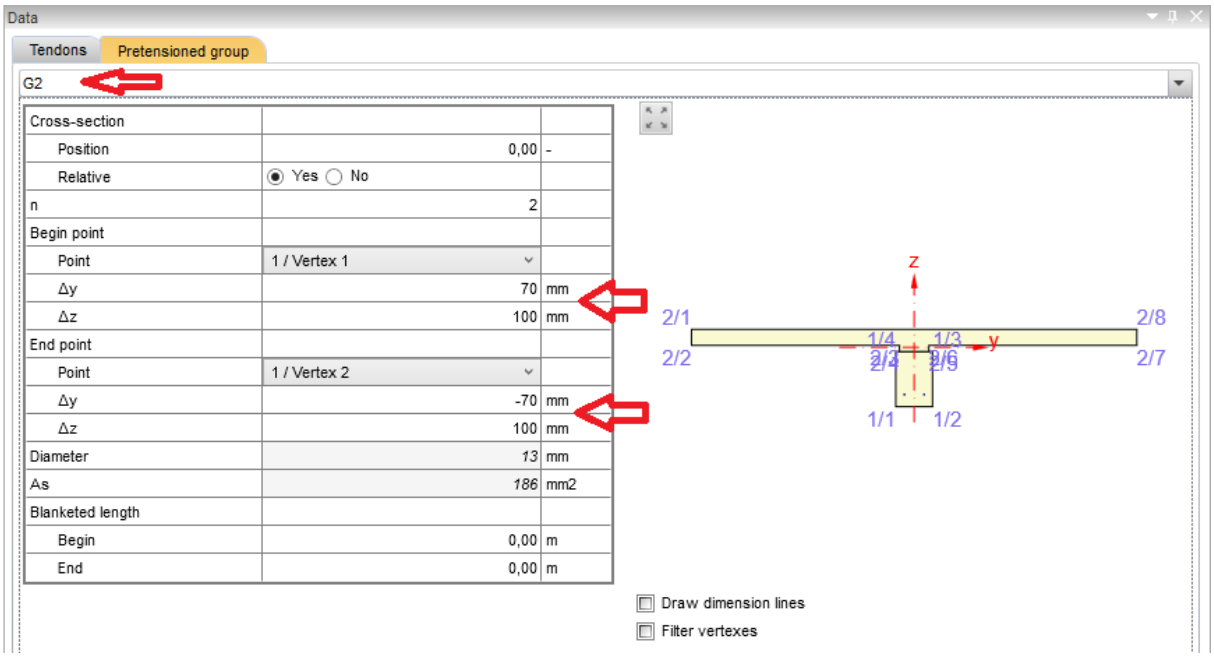
Modify properties of pretensioned group as follows:

- Group **Begin point**:
  - set the orgin point to **Vertex 1**;
  - set value of **Δy** to **82** mm;
  - set value of **Δz** to **55** mm;
- Group **End point**:
  - set the origin point to **Vertex 2**;
  - set value of **Δy** to **-82** mm;
  - set value of **Δz** to **55** mm.

Input of position of pretensioned tendons group **G1** is finished.

5.3.2 Input of next tendon groups to member 3

Click **Pretensioned in line** in ribbon group **New tendon** to add new group of tendons to member **3**. Newly created group **G2** is set as current group. Material properties are automatically copied from the first group **G1**. The positions of tendons in cross-section must be modified:



- Group **Begin point**:
  - set the orgin point to **Vertex 1**;
  - set value of **Δy** to **70** mm;
  - set value of **Δz** to **100** mm;
- Group **End point**:
  - set the origin point to **Vertex 2**;
  - set value of **Δy** to **-70** mm;
  - set value of **Δz** to **100** mm.

Modification of tendons position in **G2** group is finished.

Input tendon groups **G3** and **G4** in similar way.

Modify **G3** as follows:

- set beginning point to **Vertex 1**, set **Δy** to **70** mm and **Δz** to **145** mm.
- set end point to **Vertex 2**, set **Δy** to **-70** mm and **Δz** to **145** mm.

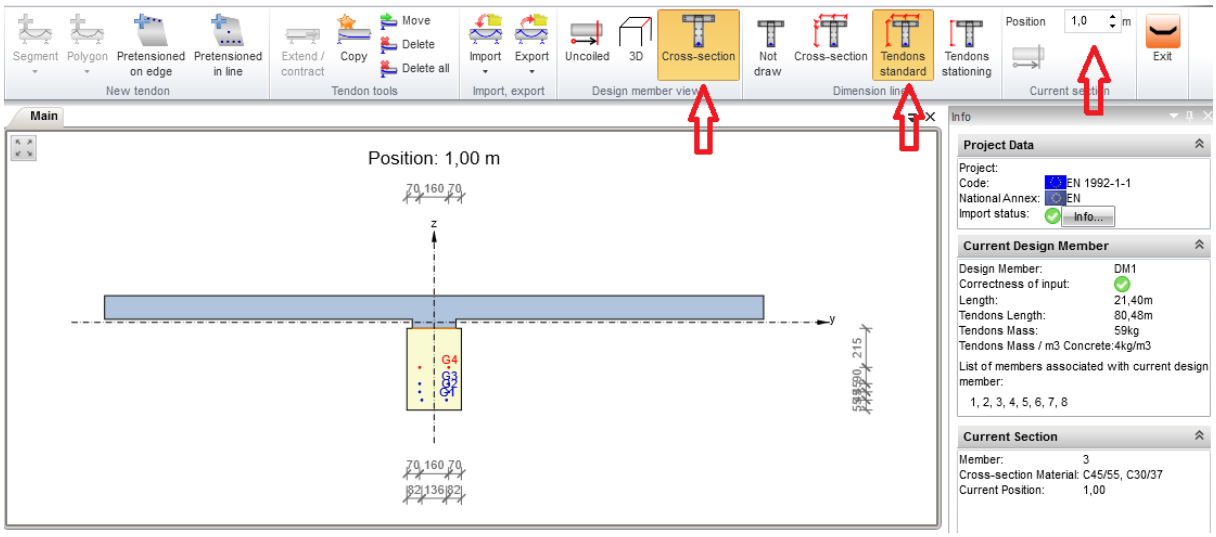
Modify **G4** as follows:

- set number of tendons in layer **n** to **3**.
- set beginning point to **Vertex 1**, set **Δy** to **70** mm and **Δz** to **235** mm.
- set end point to **Vertex 2**, set **Δy** to **-70** mm and **Δz** tp **235** mm.

Positions of tendons can be checked in the main window:

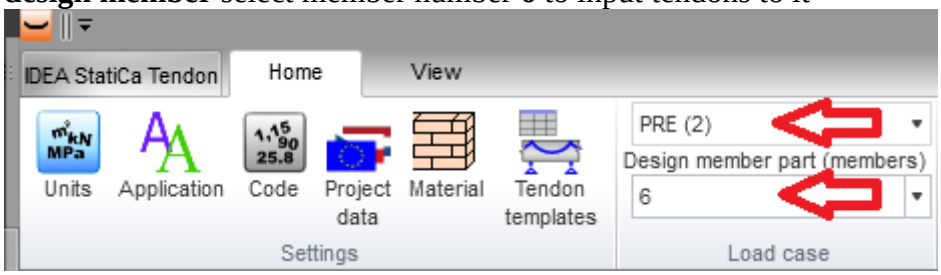
- switch on drawing mode **Cross-section** in ribbon group **Design member views**;
- switch on dimensioning mode **Standard-tendons** in ribbon group **Dimension lines**;
- in ribbon group **Current section** set value **Position** to **1** m.

Tendon groups in cross-sections defined on member **3** are drawn in the main window.



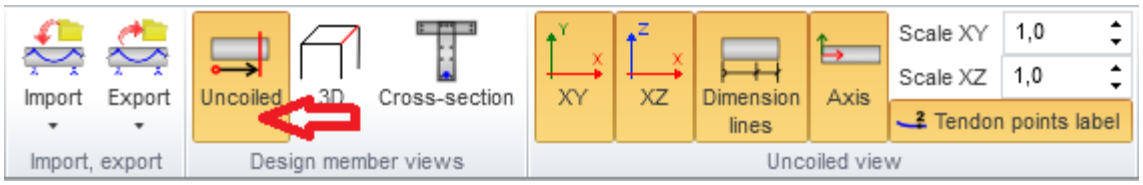
5.3.3 Input of pre-tensioned tendons group to member 6

Click navigator command **Tendons > Tendons layout** to start input of tendons. In ribbon group Load case, select load case for transfer of prestressing **PRE (2)** and in the list **Part of design member** select member number **6** to input tendons to it

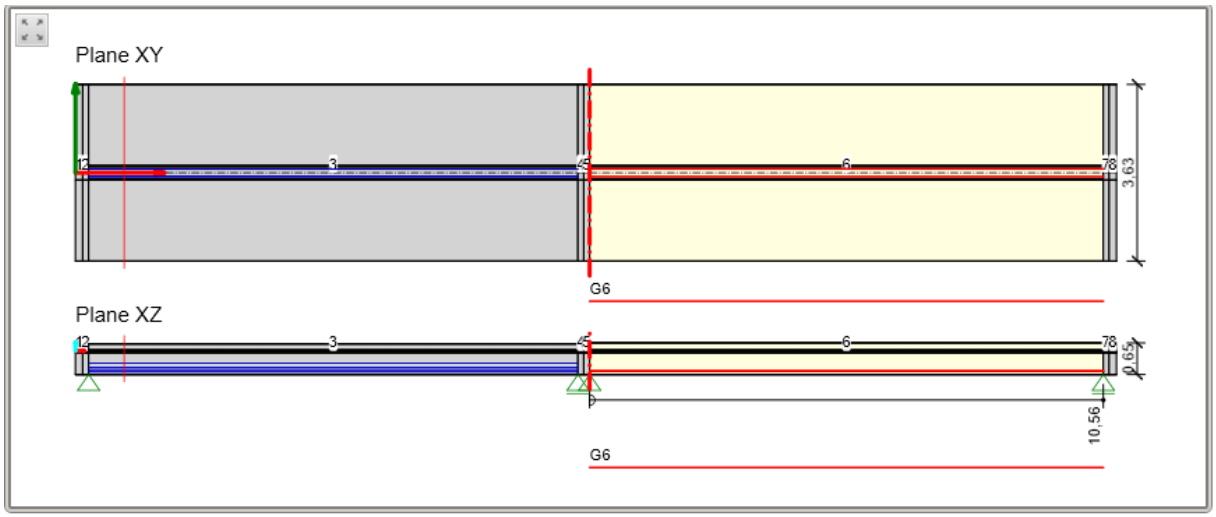


Click **Pretensioned in line** in ribbon group **New tendon** to add new group of pre-tensioned tendons to member **6**.

Click **Uncoiled** in ribbon group **Design member views** to switch back the drawing mode to uncoiled view.



Newly defined tendon is drawn in uncoiled views XY and XZ.



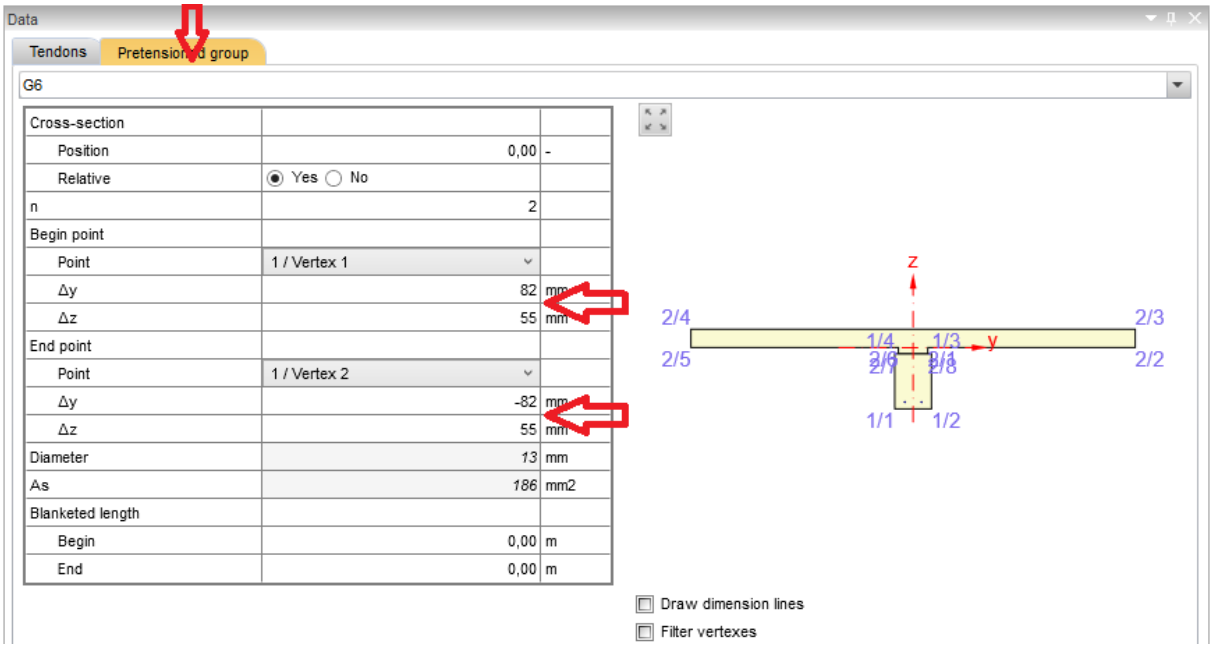
Data

Tendons | Pretensioned group

Pre-tensioned tendon groups

	Group name	Load case	Material	Initial stress [MPa]	Geometry	Limiting value of stress [MPa]	Tendon stress check
1	G1	PRE (2)	Y1860S7-12.5	1435,0	✓	1476,0	✓
2	G2	PRE (2)	Y1860S7-12.5	1435,0	✓	1476,0	✓
3	G3	PRE (2)	Y1860S7-12.5	1435,0	✓	1476,0	✓
4	G4	PRE (2)	Y1860S7-12.5	1435,0	✓	1476,0	✓
5	G6	PRE (2)	Y1860S7-12.5	1435,0	✓	1476,0	✓

Positions of pretensioned tendon groups in cross-section can be modified in tables on tab **Pretensioned groups** in the **Data window**.



Modify **G5** as follows:

- set beginning point to **Vertex 1**, set **Δy** na **82** mm and **Δz** to **55** mm;
- set end point to **Vertex 2**, set **Δy** to **-82** mm and **Δz** to **55** mm.

Modification of positon of pretensioned tendons group **G5** is finished.

5.3.4 Zadání dalších skupin kabelů na prvek 6

Input tendon groups **G6**, **G7**, **G8**, **G9** and **G10 in similar way**.

Modify **G6** as follows:

- set beginning point to **Vertex 1**, set **Δy** to **70** mm and **Δz** to **100** mm;
- set end point to **Vertex 2**, set **Δy** to **-70** mm and **Δz** to **100** mm.

Modify **G7** as follows:

- set beginning point to **Vertex 1**, set **Δy** to **70** mm and **Δz** to **145** mm;
- set end point to **Vertex 2**, set **Δy** to **-70** mm and **Δz** to **145** mm.

Modify **G8** as follows:

- set beginning point to **Vertex 1**, set **Δy** to **70** mm and **Δz** to **190** mm;
- set end point to **Vertex 2**, set **Δy** to **-70** mm and **Δz** to **190** mm.

Modify **G9** as follows:

- set beginning point to **Vertex 1**, set **Δy** to **70** mm and **Δz** to **235** mm;
- set end point to **Vertex 2**, set **Δy** to **-70** mm and **Δz** to **235** mm.

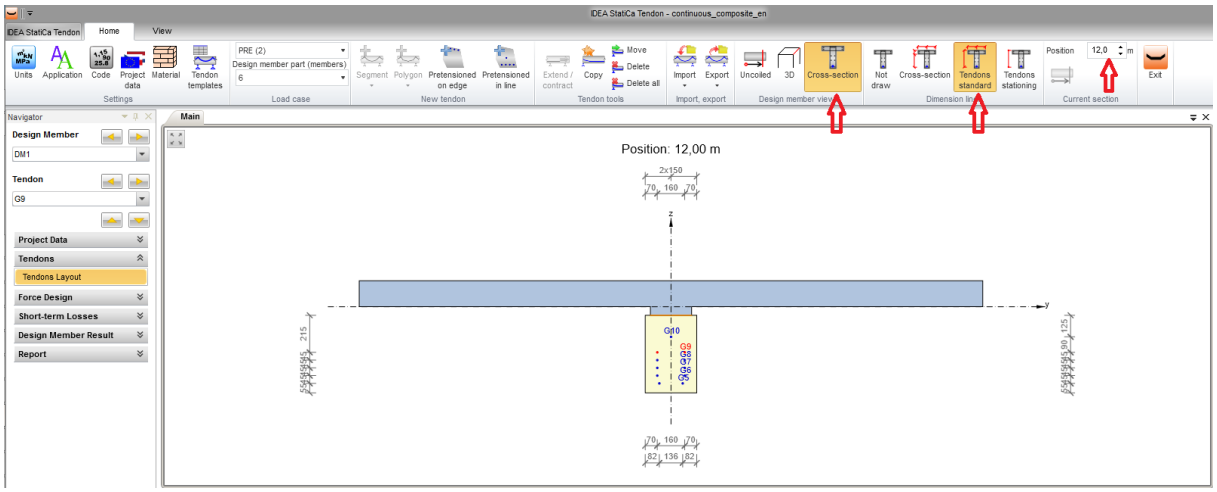
Modify **G10** as follows:

- set number of tendons in layer **n** to **1**;
- set beginning point to **Vertex 1**, set **Δy** to **150** mm and **Δz** to **325** mm.

Positions of tendons can be checked in the main window:

- switch on drawing mode **Cross-section** in ribbon group **Design member views**;
- switch on dimensioning mode **Standard-tendons** in ribbon group **Dimension lines**;
- in ribbon group **Current section** set value **Position** to **12** m.

Tendon groups in cross-sections defined on member **6** are drawn in the main window.



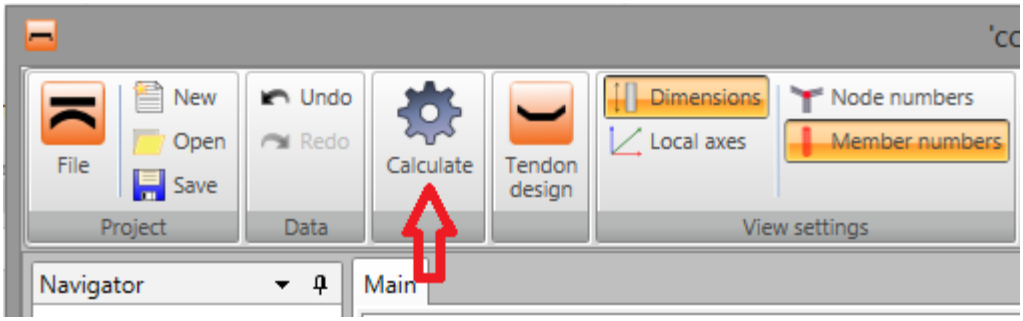
Input of pre-tensioned tendons is finished.

5.1 Closing IDEA Tendon

To close **IDEA Tendon** click **Exit** in ribbon group or click cross in the right top corner of the window. Also menu command **IDEA Tendon** > **Close** can be used.

6 Analysis

Click **Analysis** in ribbon group to run linear analysis of the beam.



After the linear analysis has finished, the results of linear analysis (reactions, internal forces and deformations) can be evaluated using commands group **Results** in navigator.