Stade des Alpes - Grenoble
Etudes et Techniques Internationales (ETI)

Advanced Training

Parameters, Stages and Prestress

Scia Engineer

Contents

Introduction to construction stages, prestressing, TDA

Modern civil engineering structures can achieve considerable economy in construction by combination of hybrid systems of steel and concrete or precast and cast-in-place concrete. The design of such systems takes advantage of individual material properties. The economy and speed of construction are increased also by the application of hybrid methods of construction. The main load-bearing members, formed by suspension or stay cables, hangers, beams or arches, are very often constructed in advance and are used as auxiliary systems for other parts of the structure to reduce overall construction time and costs. The design of the structures combines both precast and cast-in-place techniques to obtain economy in construction, and maintains a high standard of quality while reducing the time needed to complete the construction.

During the construction these structures pass through different static systems; boundary conditions change, new structural members are assembled or cast, post-tensioning is applied and temporary support elements are removed. In many structures concrete structural elements of various ages are combined and the concrete is gradually loaded. Therefore, during both construction and throughout the service life of concrete structures, account must be taken of the creep and shrinkage of concrete. Rheological properties of concrete can influence the serviceability of the structure in decisive ways. The bearing capacity of the structure can also be influenced by the redistribution of internal forces caused by creep. Therefore sophisticated methods are needed for the structural analysis.

Construction stages, Prestressing, and TDA are modules of Scia Engineer designed for the analysis of prestressed concrete and composite structures with respect to step-by-step construction, change of boundary conditions, and rheological effects of concrete. The modules allow for the structural analysis of both prestressed concrete and composite structures, successive assembling or casting of structural elements, progressive construction of cross-sections, gradual application of loads and prestressing, and removal of temporary structural elements. Special construction technologies can be modeled, such as cantilever segmental construction with both precast and cast-in-place segments, launching, cable stayed structures, making simple beams continuous including successive casting of composite slab, or gradual construction of multi-storey buildings.

The implementation of these modules is the first step towards the change of design and analysis of concrete structures in Scia Engineer. But the possibility to run the calculation in smooth sequence with respect to the step-by step construction or the introduction of time as new variable in the analysis are not the only two aspects of the issue. Also new material parameters - rheological properties of concrete – are taken into account in the calculation and a new feature of a real value is that the program responds to modern concepts of the analysis of prestressing in theory of structures. The post-tensioned tendon is considered only as an external load at the moment of prestressing. This load is calculated as the load, which is equivalent to the effects of the tendon stressed just after short-term losses. The tendon becomes an integral part of the structure after anchoring. Its stiffness is added into the stiffness matrix of the structure. Since that, all loads carried by the structure will automatically cause the change of prestressing of that tendon. Both tendons and composite parts of cross-section are modeled by eccentric finite elements. Full strain compatibility between eccentric elements connecting two nodes is ensured along the whole length of elements. The TDA module in Scia Engineer allows for a new structural model of so far unattainable quality.

Each of the three mentioned modules can be used separately (e.g. module Prestressing in linear analysis, Construction stages for the analysis of 3D steel frame structure, etc.). However the user loses some of the features in such situation. Therefore, also the descriptions of these three modules will contain frequent links to other modules from this "little-family".

In precast structures there are often only some parameters that change in a project. Therefore the use of parameters in Scia Engineer is a very useful feature in precast calculations. The first part of this text will give a brief introduction into the use of parameters in Scia Engineer. In the other parts of this text user can parameterize the examples.

Part I: Use of parameters in Scia Engineer

Introduction to parametric input

The parametric input enables the user to define some of structure properties as parameters. Thus, for example, the geometry may be defined by means of parameters, loads can be defined as parameters, etc.

The parameter are fully editable and when changed they may lead to a very straightforward modification of the calculated model.

What's more, a model defined by means of parameters can be saved as a template. When opened, the user is first asked to fill in the table with all the parameters present in the model. This may be effectively used for creation of simple "programs" for e.g. calculation of continuous beam, simple frame, etc.

The user has to create the structure only once. Then he/she has to define the parameters and save the structure as template. In the future, he/she just fills in the table with a few parameters and can immediately proceed to calculation and evaluation of results.

There are numerous parameters types available in Scia Engineer. Each type may be used for other model entity, some are intended for geometry, others for loads, others for cross-sections, etc.

Available parameter types:

- ✓ Nothing: The parameter is not used.
- ✓ Integer: The parameter is used as an integer.
- ✓ Coefficient: The parameter is used as coefficient.
- ✓ Length: The parameter is used for definition of length in the model.
- ✓ Force: The parameter is used for definition of size of force load.
- ✓ Moment: The parameter is used for definition of size of moment load.
- ✓ Line load: The parameter is used for definition of size of line load.
- ✓ Surface load: The parameter is used for definition of size of surface load.
- ✓ Mass: The parameter is used for definition of size of masses.
- ✓ Line mass: The parameter is used for definition of size of line masses.
- ✓ Surface mass: The parameter is used for definition of size of surface masses.
- ✓ Cross-section length: The parameter is used for definition of length at cross-sections.
- ✓ Angle: The parameter is used for definition of angles.
- ✓ Relative: The parameter is used for definition of relative values.
- ✓ Boolean: This parameter can have two values only: True (ON) or False (OFF).
- ✓ Cross-section rolled: The parameter is used for definition of rolled cross-sections.
- ✓ Library: This parameter type can be used with any "library" item, i.e. any item that is selected from one of Scia Engineer internal databases, such as materials, cross-sections, subsoil, reinforcement pattern, etc.
- ✓ Combination factor: Combination factors for load cases inserted into a combination.
- ✓ Relative humidity: Applicable in the calculation of long term losses in prestress.
- ✓ Time (history): Time of individual construction stages on time-line.
- ✓ Stress: (i) Stress in concrete that can be defined in measured values when the Time Dependant Analysis is performed or (ii) the initial stress of the strands for a strand pattern.
- ✓ Temperature: Used for temperature.
- ✓ Length for stiffness: Used to define length unit in stiffness.
- ✓ Point stiffness: Used to define point stiffness.
- ✓ Line stiffness: Used to define line stiffness.
- ✓ Reinforcement diameter: Used to define the diameter of reinforcement bars.

It is also possible to work with formulas in combination with the parameters in Scia Engineer. The formula may consist of several values, parameters, functions and operands:

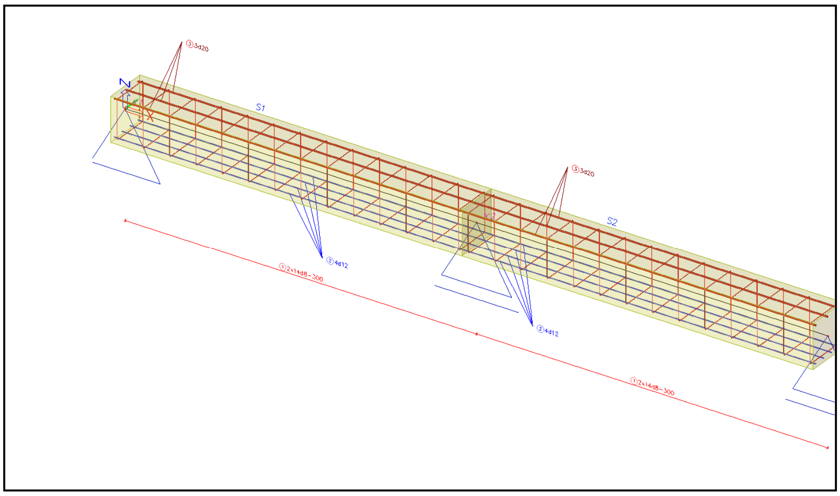
- ✓ +, -, *, /, sqrt(x), ^ (Raises the given number to a given power)
- ✓ sin(x), cos(x), tg(x)
- ✓ ln(x), log(x), exp(x)
- ✓ sign(x) (Returns the sign of parameter x)

More information about those operands is stated in the help of Scia Engineer.

Work method:

- Input of total structure
- Define the parameters
- Link the parameters
- Group the Parameters (Make a suitable layout)
- Save as template

Example: parameterized beam with practical reinforcement



- Project data - Construction type: Frame XZ
  - Project level Advanced
- Input:
  - Support
    - in node
    - point on beam
- Load cases:
  - LC 1: Self weight
  - LC 2: Life load: q [kN/m]
- Parameters:
  - Length of span
  - Loads
  - Cross section parameters
  - Reinforcement parameters
- Parameters input Tools Parameters
- Group parameters Tools Parameters template settings

  Pictures out of gallery
  - .wmf
  - 550 x 550
- Control Sjabloondialoog
- Save file in C:\Documents and Settings\#USER#\ESA80\templates: -
- Remark: Use Range to avoid wrong input of parameters
- Userblocks (C:\Documents and Settings\#USER#\ESA80\ userblocks) or ODA

Part II a: Introduction in staged calculations

Part II a: Introduction to staged calculations

Introduction to construction stages

The module Construction stages allows general modeling of the construction process. In combination with the TDA module the analysis also takes account of time as a new input variable. For the purpose of the time dependent calculation a global, local and detailed time axis is introduced and individual time nodes are generated. The over-time development and changes in the structure, cross-sections or loading, though, are modelled via individual "stages of construction", where each of these stages is given a number, name and global time. The responses (results) of the loading increments are saved into separate loading cases separately for the effect of the permanent loading increment, prestressing, and the sum of the rheology effects during the previous time interval.

Preparatory operations

Input of geometry and other data

Prior to input of data related directly to the Time Dependent Analysis or Analysis of Construction Stages, certain preparatory operations have to be done. All structural members, prestressed members, boundary conditions and loads, which will ever appear in the structure, must be defined in advance. After their definition, all elements, tendons, supports, etc. will be gradually "inserted" into the structure in the Construction stages module. The input of members, nodes, supports and loads is performed in the standard Scia Engineer environment.

Adjustment of parameters

It is necessary to input some specific data for a TDA analysis or ACS (Analysis of Construction Stages) analysis. These data can be input in one setup dialogue. This dialogue contains item for both TDA and Construction stages analysis. In addition, a few other parameters for mesh generation, calculation, materials, etc. must be set in a specific way. Individual parameters are described in separate chapters dealing with:

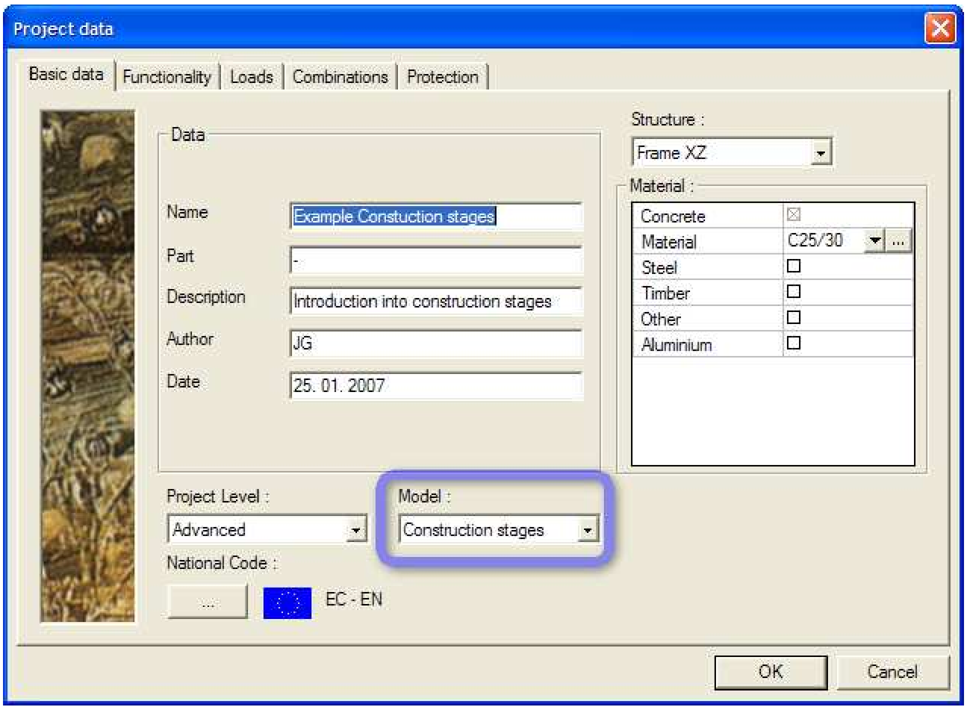
- ✓ Construction stages setup,
- ✓ TDA setup,
- ✓ Mesh and calculation setup,
- ✓ Material setup.

Note:

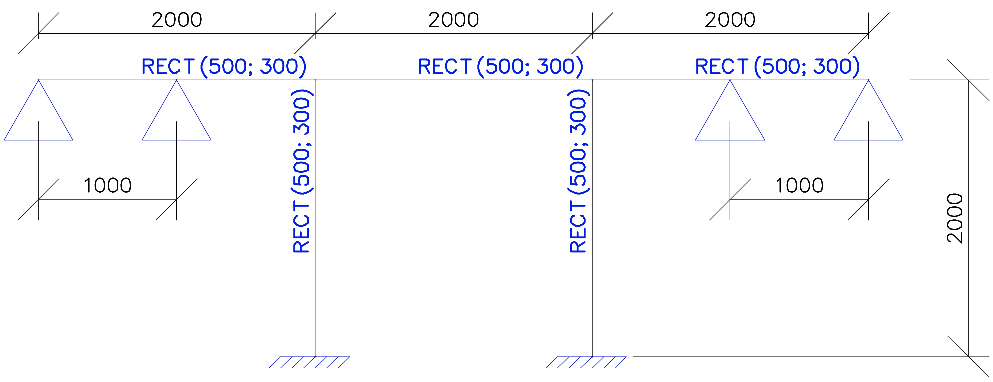
The TDA part of the setup dialogue is available ONLY if TDA module is available, i.e. if e.g. the project is of Frame XZ type.

Creating a Construction stages project

If you want to perform an analysis of construction stages, you must make the appropriate settings in the Project Setup dialogue on the Basic data tab:
Select Construction stages in the Model combo box:

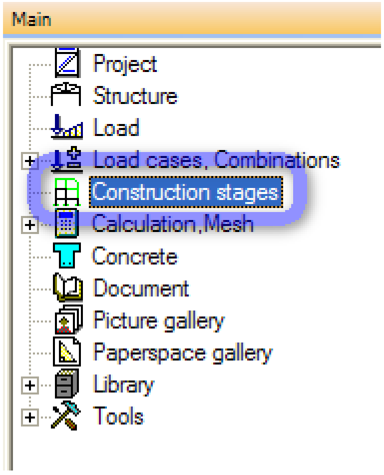


The following example will be used to explain the Construction stages module in Scia Engineer.

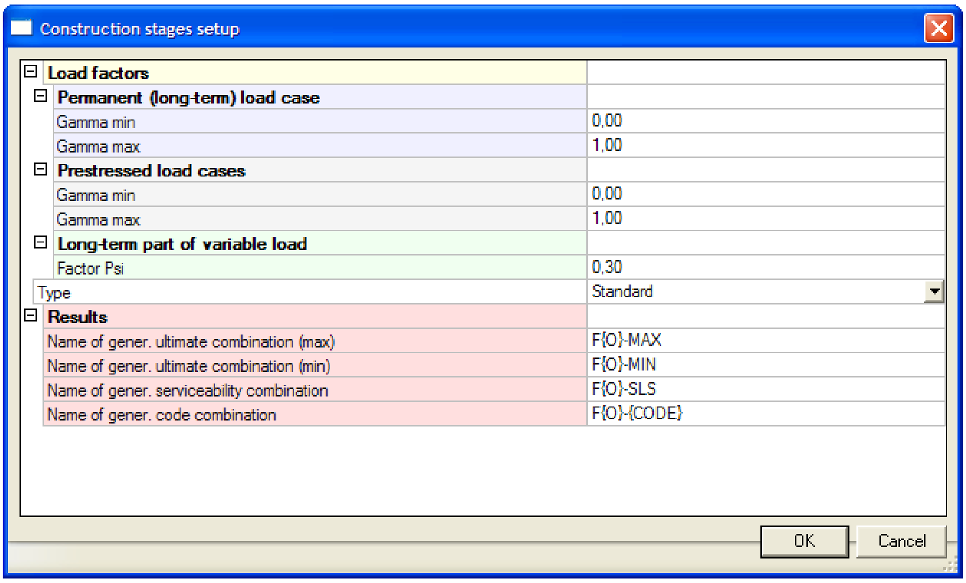


Construction stages setup

After a double click at the '**Construction stages**' in the main menu:



The Setup dialogue for Construction stages will appear it enables you to input the default values of the parameters that appear in the input dialogues for individual construction stages:



Load factors

'**Permanent (long-term) load case**': Gamma min and Gamma max are attached to permanent load cases of both types (Permanent load (γ_G) and prestress (γ_P)). The load factors γ_{Gmin}(<=1), γ_{Gmax}(>=1), γ_{Pmin}(<=1), γ_{Pmax}(>=1) are specified (for each load case) in each construction (or service) stage.

'**Prestressed load case**': See above.

'**Long-term part of variable load**': Factor ψ specifies the long-term part of the load.

If the dead, prestressing or variable load case is applied in a construction stage, it can **never** be applied again (exclusivity), because the configuration of the structure could be changed in the next construction steps and the results would be different.

Results

‘**Name of generated ultimate combination (max)**’: Specifies the mask for the automatic generation of names of maximal load case combinations.

‘**Name of generated ultimate combination (min)**’: Specifies the mask for the automatic generation of names of minimal load case combinations.

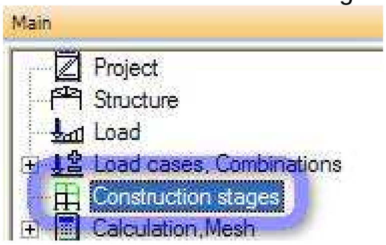
‘**Name of generated serviceability combination**’: Specifies the mask for the automatic generation of names of serviceability load case combinations.

‘**Name of generated code combination**’: Specifies the mask for the automatic generation of names of EC combinations.

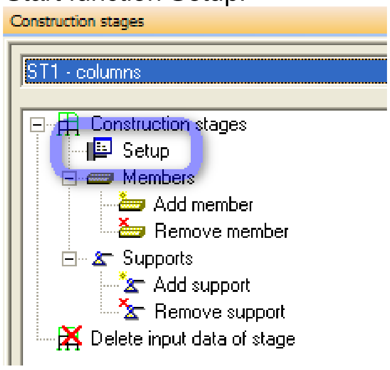
Note: Concerning the generated names of load case combinations, the sequence {O} is substituted with the appropriate number during the generation of the combinations. For example, the combination name mask F{O}-MAX gives combinations named F1-MAX, F2-MAX, F3-MAX, etc.
Note: This very same dialogue may also contain parameters for TDA. These TDA parameters are available only in a project that supports time dependent analysis. See also TDA setup.

Procedure to adjust Construction stages parameters

1. Open the service Construction stages:



2. Start function Setup:



3. Input the required parameters.
4. Confirm with [OK].
5. Close the Setup dialogue.

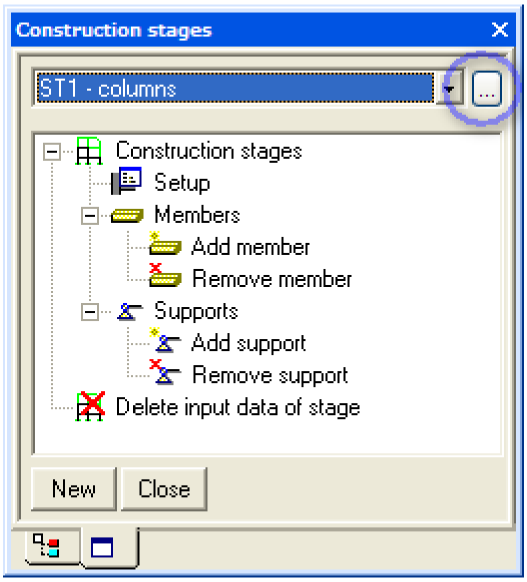
Construction stages manager

The Construction stages manager enables you to input, review, copy, print and delete individual construction stages. It is a standard Scia Engineer database manager. We will use this manager later to create the following stages in our example:

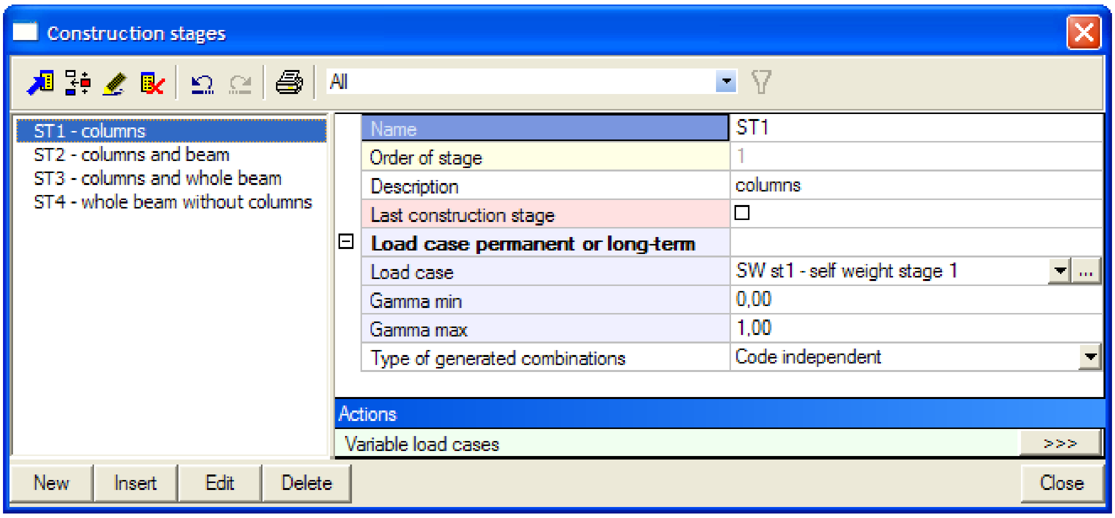
- ✓ Stage 1 : Columns
- ✓ Stage 2 : Columns and beam
- ✓ Stage 3 : Columns and whole beam
- ✓ Stage 4 : Whole beam without columns
- ✓ Stage 5: Whole beam on two supports

Procedure to open the Construction stages manager

1. The project must be of Construction stages type.
2. In the main tree menu open service Construction stages.
3. At the top part of the service dialogue click the "dot"-button:



4. The manager is opened on the screen:



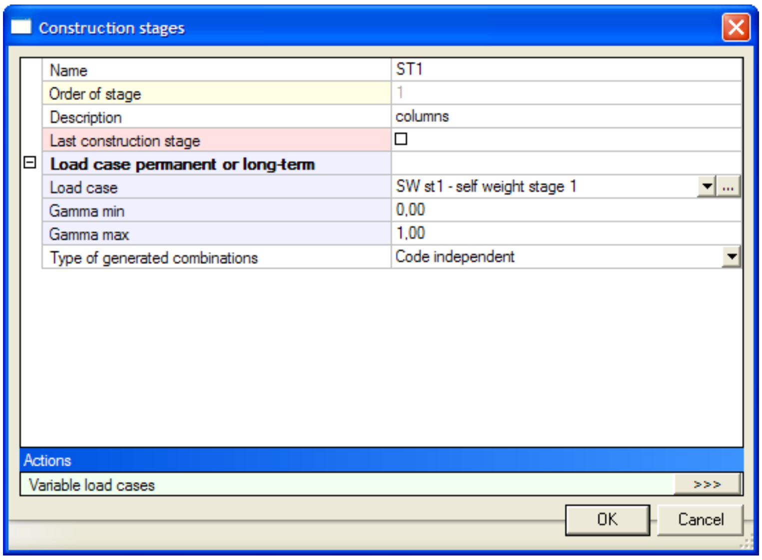
Note:

The Construction stages manager opens automatically when you open the service for the first time.
Note: When a new construction stage is created, the parameters are set to values defined in the Construction stages setup.

Creating a new construction stage

Procedure to create a new construction stage

- ✓ Open the Construction stages manager (see above).
- ✓ Click button [New]. If no suitable load case is available, you are asked to create one.
  A new construction stage is added to the list.
  Note: It is best if user creates on forehand the load cases that he will need.
- ✓ Click button [Edit] to open the editing dialogue:



- ✓ Fill in the parameters (see below).
- ✓ Confirm with [OK].
- ✓ Close the Construction stages manager.

Parameters of a construction stage

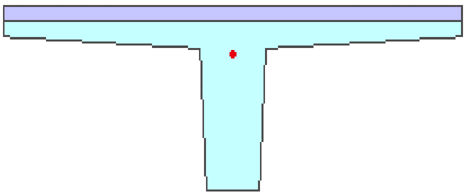
'**Name**': Defines the name of the stage.
'**Order of stage**' (informative): Gives the sequence number of the stage.
'**Description**': Contains a short description of the construction stage. It is useful to say in a few words what happens in the current construction stage. The comment helps the user to keep a clear image of the construction process. The name is also used in the generated names of result classes and generated load case combinations. E.g. for combinations, this description is the only unambiguous identifier of the generated load case combinations.
Note: It is highly recommended to use this field.
'**Last construction stage**': Defines, whether the current stage is the last construction stage. If ON, then the next construction stage is the first service stage. The user cannot change the structure from that time, but he can add dead load and variable load (no prestressing!). Therefore no changes in configuration of the structure (changes of cross-section, prestressing, etc.) are possible in service stages. If a variable load is assigned to a construction stage before (including) the last construction stage, it is "consumed" and cannot be used again in another construction stage. If a variable load is assigned to a service stage (i.e. into a stage following after the last construction stage), it can be reused freely in another construction stage.
'**Load case**': Selects the load case that is assigned to the construction stage.
'**Gamma min**', '**Gamma max**': Load factors (see also next pages).
'**Psi Factor**' for variable load.
'**Name of generated code combination**': Specifies the mask for the automatic generation of names of EC combinations (see later).
Action button '**Variable load cases**': Allows input of a variable load case into the construction stage.

Load case permanent or long-term

A load case of type permanent or self-weight must be defined and assigned to a construction stage. One load case of this type must be exclusively assigned to each stage. The load case may be empty. As mentioned above, the load case applied in a construction (or service) stage can be of two types: permanent or self-weight. The input of a permanent load is done by standard way, but concerning the self-weight, there are two possibilities of the input:
(1) First possibility is via permanent loads. The self-weight of the beam is calculated by the user in advance and is specified as a uniform or trapezoidal load distributed along the length of the beam. This method can be used in combination with other permanent loads, e.g. the self-weight of diaphragms, surfacing, etc. The self weight of the beam can be applied in any construction stage (at any time) independently from the time of installation of this beam. Therefore the input of the permanent load is not tied up with the beams or composite parts of beams added at the current (active) construction stage.
(2) Second possibility of the input of own weight can be applied only on beams added at the current (active) construction stage on added composite parts of beams. The appropriate load case must be of "self weight " type. No input of other loads is possible in such a load case. Therefore, if the "self-weight" load case is specified in current construction stage, only an increment of structural own weight is applied. The increment is defined as the self-weight of that part of the structure (structural elements or their composite parts) that is installed in the current construction stage. The two approaches will be demonstrated on a simple example.

Example: Introduction of self-weight into the model

Let's assume a beam of a T-cross-section that is made in two phases: (i) core cross-section, (ii) composite slab:



The cross-section consists of two stages: 1 = the "core" beam, 2 = the composite slab. We'll describe three model situations, two of which employ the first approach (user-calculated permanent load) and one of which uses the second one. We do not say which situation is better and which one is worse, we just describe them here to explain the consequences of different approaches. It is up to the user to decide which procedure of modeling best reflects the unique conditions of a particular project.

Situation A (user-calculated self-weight)

Stage	Action	Load case assigned to construction stage
1	casting the beam (phase 1 of the cross-section is introduced into the model)	empty permanent-standard load case
2	casting the composite slab (phase 2 of the cross-section is introduced into the model)	empty permanent-standard load case
3	introduction of the manually calculated self weight	permanent-standard load case with defined load that represents the self-weight of the beam member

In this situation, the user is fully responsible for the introduction of the self-weight into the model. On the other hand, the process is fully under his/her control. First, the "core" beam is produced. Then, the composite slab is cast. And only at the very end, the self-weight is introduced in its full size. To sum up, until the composite beam is completed, it is not subject to any load.

Situation B (user-calculated self-weight)

Stage	Action	Load case assigned to construction stage
1	casting the beam (phase 1 of the cross-section is introduced into the model)	empty permanent-standard load case
2	introduction of the manually calculated self weight	permanent-standard load case with defined load that represents the self-weight of the beam member
3	casting the composite slab (phase 2 of the cross-section is introduced into the model)	empty permanent-standard load case

Once again, in this situation, the user is fully responsible for the introduction of the self-weight into the model. First, the "core" beam is made and is subjected to no load. Then the self-weight is introduced in its full size. Finally, the composite slab is cast. To sum up, the "core" beam is subjected to the self-weight of the whole cross-section before the composite slab is made.

Situation C (automatically calculated self-weight)

Stage	Action	Load case assigned to construction stage
1	casting the beam (phase 1 of the cross-section is introduced into the model)	permanent-self-weight load case
2	casting the composite slab (phase 2 of the cross-section is introduced into the model)	permanent-self-weight load case

In this situation, the self-weight is introduced automatically and in parts. First, the core beam is cast and is automatically subjected to the self-weight of the completed part of the cross-section, i.e. of the "core" beam. When the composite slab is made, its self-weight is calculated and added to the existing self-weight of the "core" beam.

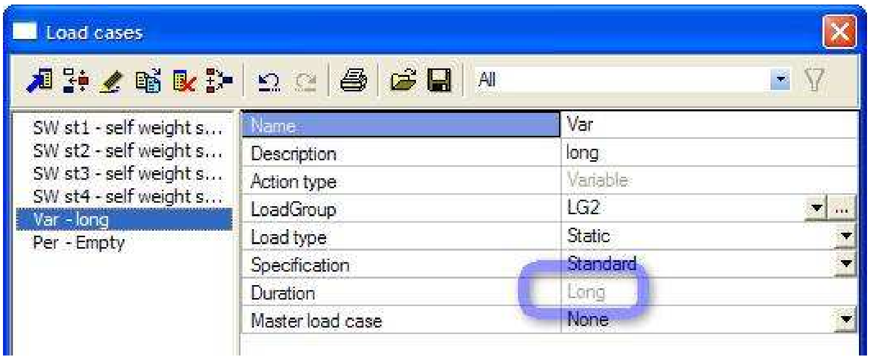
Conclusion:

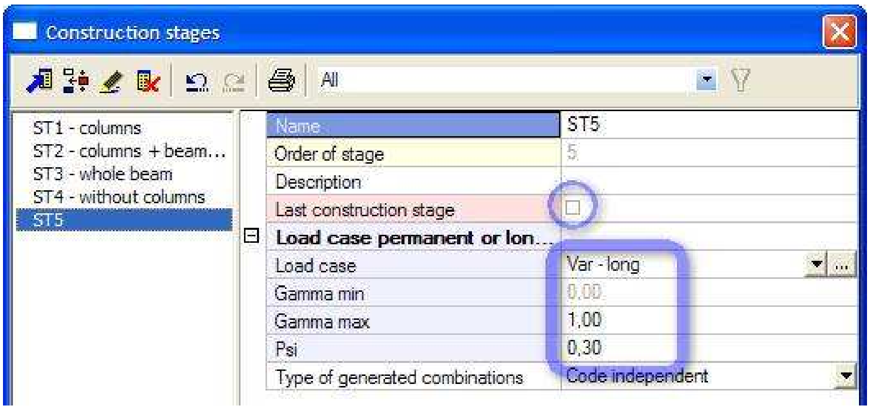
It is clearly seen from this very simple example that the calculation stages analysis allows for almost innumerable possibilities. The user must therefore think in advance and must be aware of (i) what he/she wants to model and (ii) what he/she in fact created.

Note: If the Construction stages module is combined with TDA module, other possibilities open. E.g. it is possible to model casting on a formwork (so that even the situation C does not result in the instant introduction of the self-weight), removal of the formwork in a specified time (and possible simultaneous introduction of the self-weight), etc. This will be discussed in the TDA chapter of this text.

'**Gamma min Gamma max**': Gamma min and Gamma max are attached to permanent load cases of both types (Permanent load (γ_G) and prestress (γ_P)). The load factors γ_{Gmin}(<=1), γ_{Gmax}(>=1), γ_{Pmin}(<=1), γ_{Pmax}(>=1) are specified (for each load case) in each construction (or service) stage.

If a long-term variable load is selected in the combo box '**Load case permanent or long-term**', only maximum factor γ_{Qmax} is asked, because γ_{Qmin} is automatically taken as zero (when all variable load is removed):

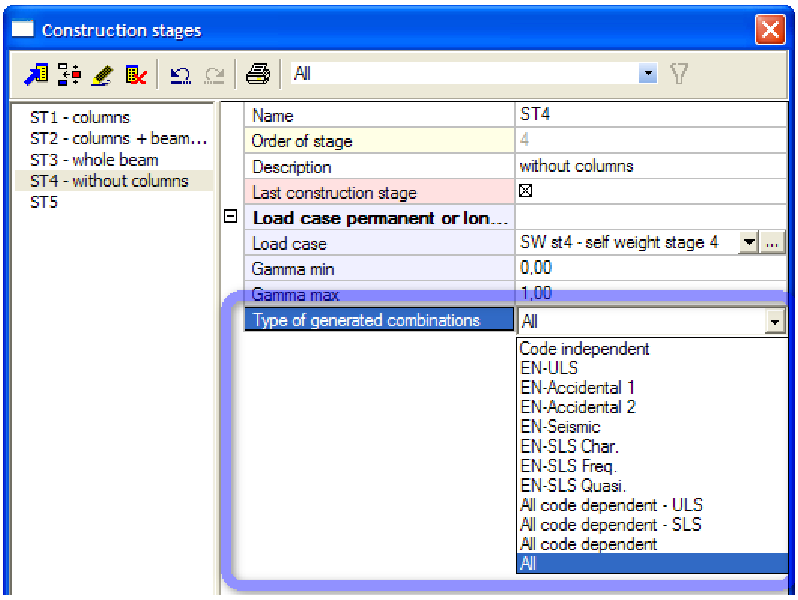




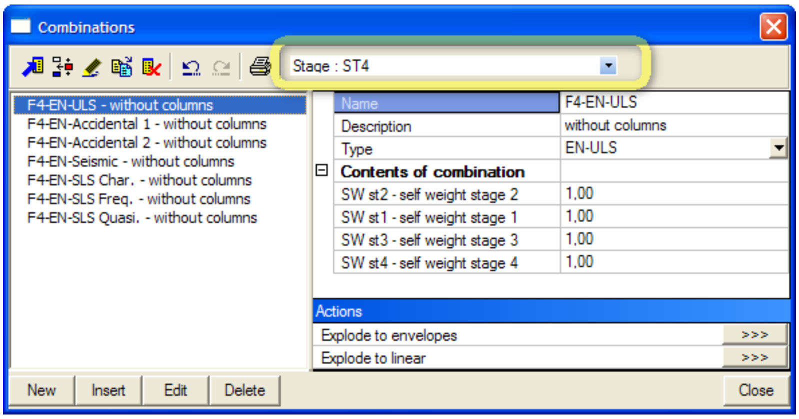
At the same time additional factor ψ<1 appears. Factor ψ specifies the long-term part of the load.

If a permanent, prestressing or variable LC is applied in a construction stage, it can never be applied again (exclusivity), because the configuration of the structure could be changed in next construction steps and the results would be different. In fact, no load factors are applied in TDA calculation (creep analysis) itself. Therefore, the results of creep load cases that are generated by TDA will also have no load factors included in themselves (better said load factor equals to 1.0).

After the calculation has been performed, both SLS and ULS combinations are generated automatically. For ULS combinations all factors for dead load γ_G, prestressing γ_P, quasi-permanent γ_Q load, and creep γ_C are applied using both their maximum (>=1) and minimum (<=1) values.
All combinations required by the codes (for EC2 persistent and transient, accidental, seismic, rare, frequent, quasi-permanent) can be defined manually as "envelope combinations". Or the user can use Scia Engineer to generate all combinations:



In the '**Construction stages manager**' user can choose a EC-EN combination type. The chosen combination type will automatically by generated after the linear staged calculation. The combinations can be reviewed for each stage (use filter on top) in the '**Combination manager**':



As you can see in this window, Scia Engineer has generated combinations of type 'EN'. So the combinations can be used the same way as standard combinations in Scia Engineer.

Two types of variable load can be applied in *service stages* (after construction stage):

- ✓ long-term load case (quasi-permanent)
- ✓ short-term load case

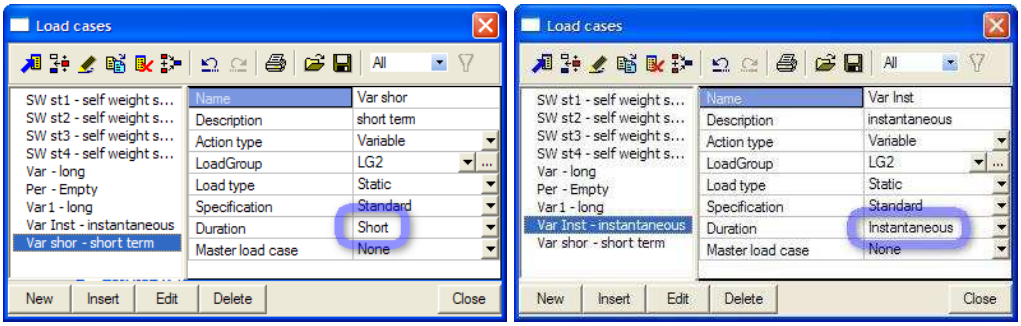
This classification has no connection to types of load cases specified elsewhere in Scia Engineer. Therefore, the long-term variable load case is identified only by specifying the long-term part of the load (using coefficient $0 < \psi < 1$). The long-term part of the load is then used for creep analysis in TDA. Quasi-permanent load is applied together with other loads at specified service stage in TDA.

Once the long-term load case is applied, it can never be applied again, because the quasi-permanent part (ψ) of the load will be applied in TDA and it will cause an additional creep of concrete. The quasi-permanent part (ψ) of the load case is assumed to stay on the structure until the end of its *service life*. The results of long-term load cases are also calculated by standard ESA and they are applied (by zero or full value) in all combinations generated for this and following service stages. It means, it is assumed that the quasi-permanent part (ψ) of variable load can be removed from the structure (or variable load can be applied by its full value) for short time (with no influence on creep).

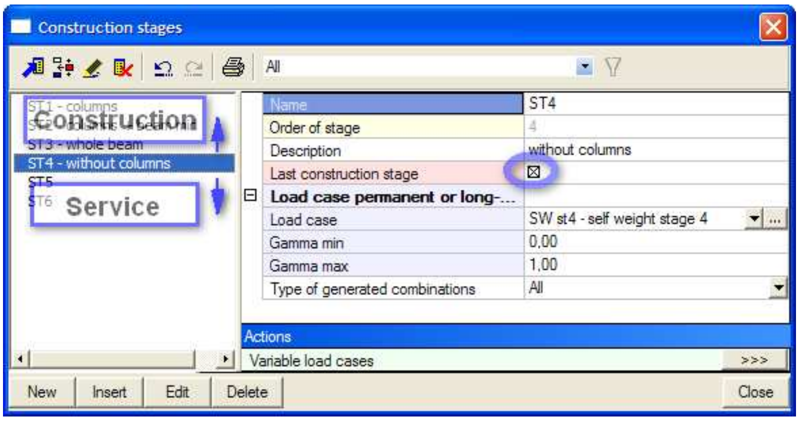
Note:

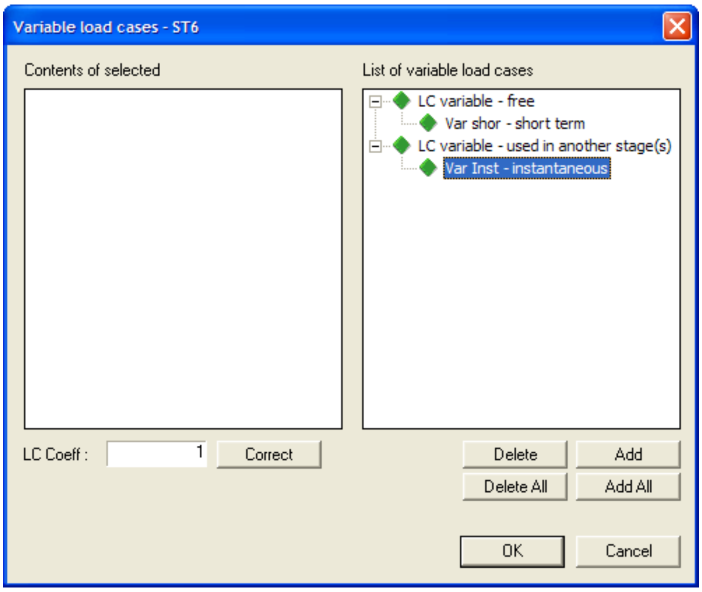
The long-term load cases cannot be applied in construction stages (only in service stages).

'**Variable load cases**': Variable loads (instantaneous and short-term) can be added into current stage:



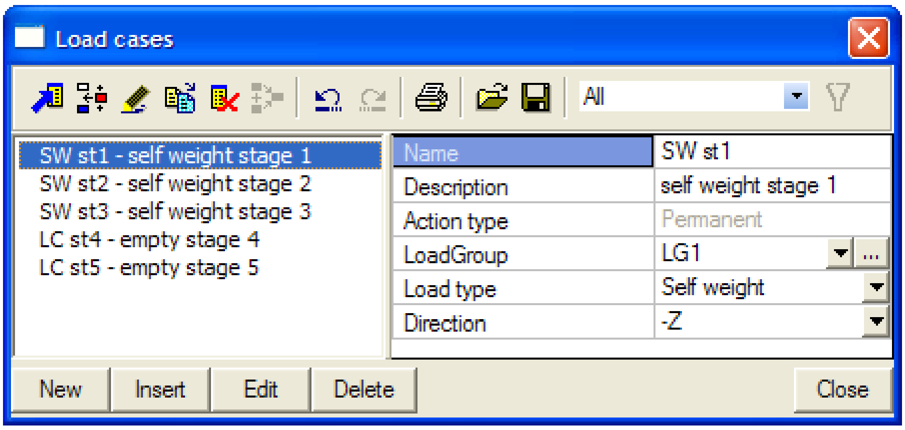
It is possible to add an arbitrary amount of load cases defined in advance. The load defined in this dialog is assumed to be temporary one and is not taken to TDA analysis. Once the variable load case is applied in *construction stage*, it must be copied into new load case before it can be used in some other construction stage. We have to realize, that the results of the same load can be different for different construction stages, because the configuration of the structure changes.
Starting from first *service stage* the short-term load case can be applied repeatedly, because the structure does not change during service, and the response (results) are identical for all service stages:



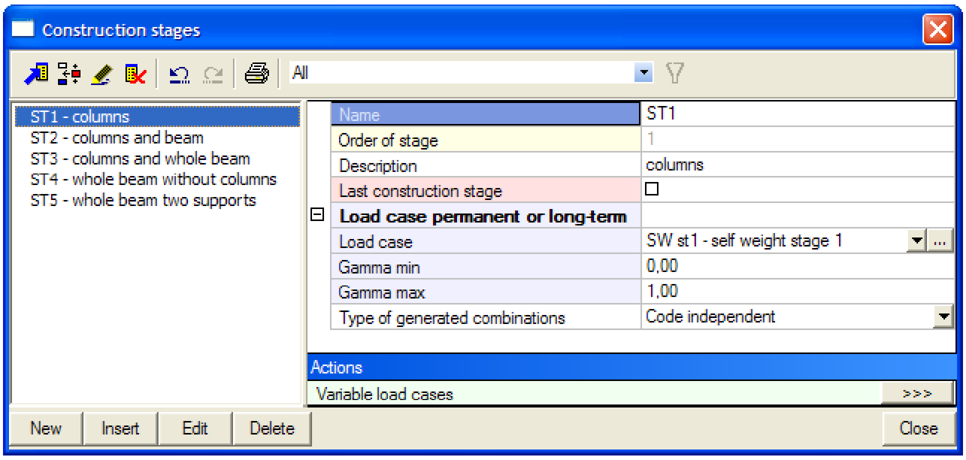


The short-term load cases are calculated by standard Scia Engineer with no influence of age of concrete, and with all materials at age of 28 days.

Let's define some load cases in our example. We will begin to define three permanent load cases of the type 'Self weight' (for stages 1,2 and 3. Because in those first stages each time a new member will be added). For stage 4 and 5 will need an empty permanent load case:



Afterwards we can also define the five stages that will be used and couple the predefined load cases to them:



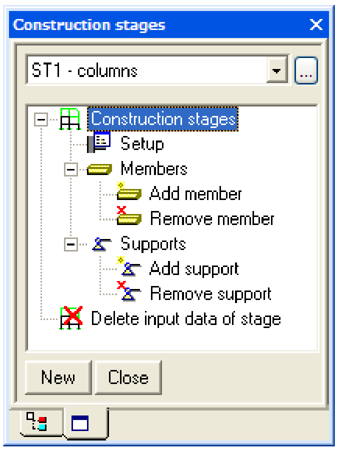
Defining the changes to the structural scheme

Before construction stages may be input, it is necessary to define all load-bearing elements, tendons, boundary conditions and load cases that are relevant for the structure. Following the real progress of construction, all elements, tendons, supports etc. are then gradually included into the structure. If any element is removed or if any boundary condition is changed, internal forces and corresponding reactions are automatically added to the load that the structure is subject to.

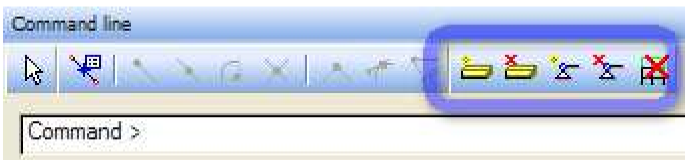
In each construction stage you may:

- ✓ add a new member to the structural scheme,
- ✓ remove the existing member from the structural scheme,
- ✓ add a new support into the structural scheme,
- ✓ remove the existing support from the structural scheme.

All these actions can be done in service '**Construction stages**':



These actions are also available in the command line when user is working in the '**Construction stages**' service:



Procedure to add member

1. Open service Construction stages.
2. Select (or define) the required construction stage (at the bottom of the service tree dialogue.
3. Call function **Members > Add member**.
4. Select the defined members that should be added to the structural scheme in the current stage.
5. End the function and if required, close the service.

Procedure to remove member

1. Open service Construction stages.
2. Select (or define) the required construction stage (at the bottom of the service tree dialogue.
3. Call function **Members > Remove member**.
4. Select the defined members that should be removed from the structural scheme in the current stage.
5. End the function and if required, close the service.

Procedure to add support

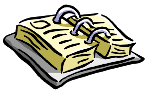
1. Open service Construction stages.
2. Select (or define) the required construction stage (at the bottom of the service tree dialogue.
3. Call function **Supports > Add support**.
4. Select the defined supports that should be added to the structural scheme in the current stage.
5. End the function and if required, close the service.

Procedure to remove support

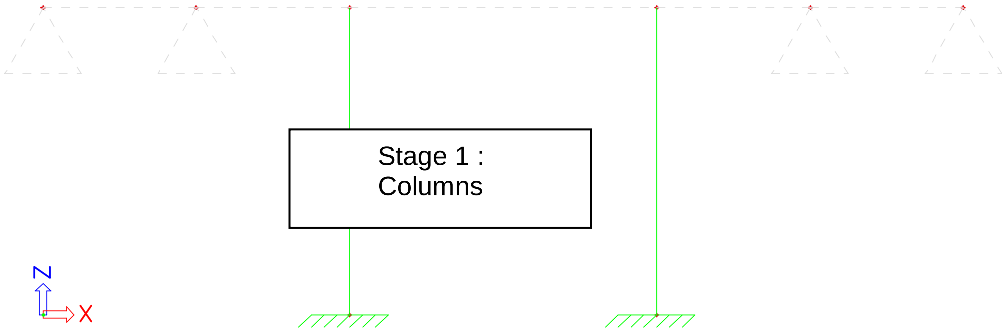
1. Open service Construction stages.
2. Select (or define) the required construction stage (at the bottom of the service tree dialogue.
3. Call function **Supports > Remove support**.
4. Select the defined supports that should be removed from the structural scheme in the current stage.
5. End the function and if required, close the service.

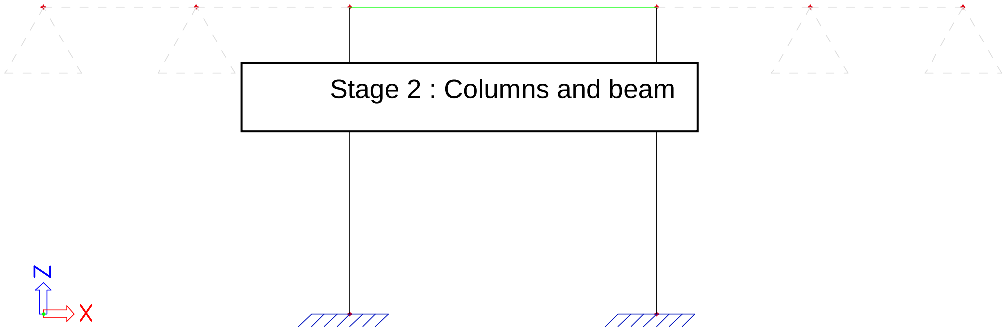
Clearing the whole stage

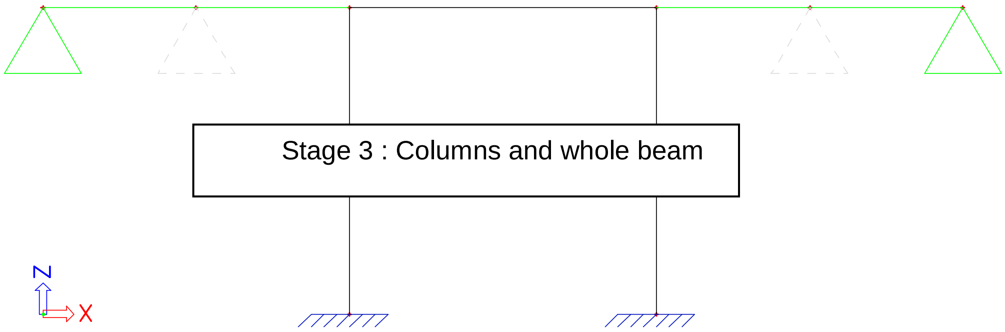
If required, it is possible to delete the complete definition of the current construction stage. Use function Delete input data of stage from the service Construction stages.

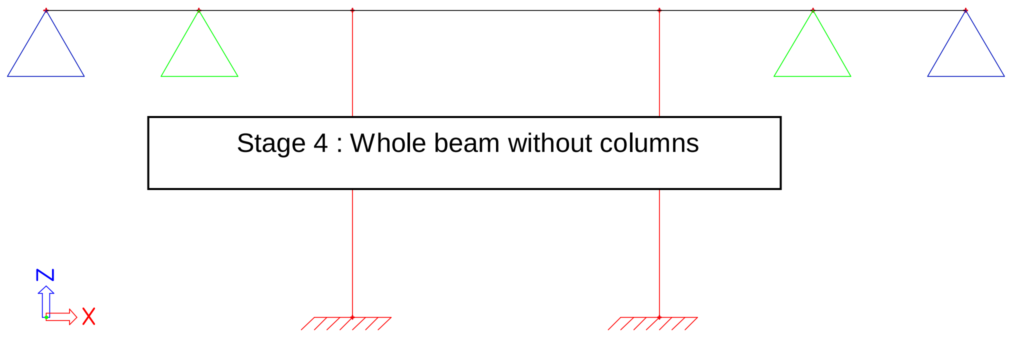


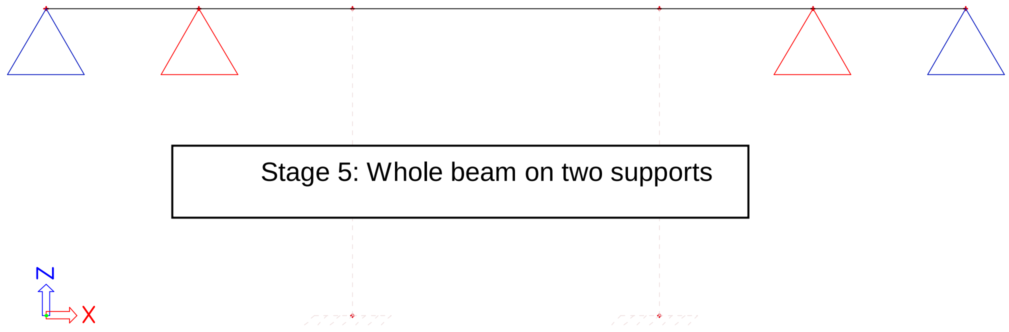
Let’s use these procedures in our example:









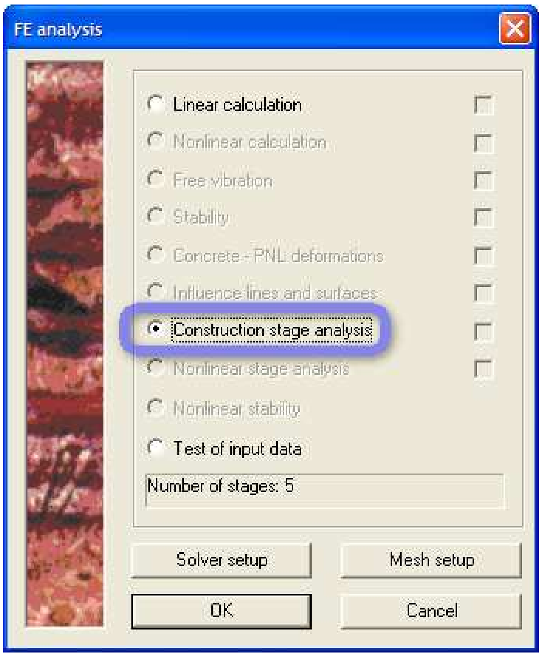


Running the calculation

Both Analysis of Construction Stages and Time Dependent Analysis are run the same way.

Procedure to run ACS

1. Call menu function Calculation, Mesh > Calculation.
2. Select Construction stages analysis.
3. Click [OK] to start the calculation.



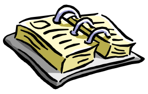
Results of construction stages analysis

When the Construction Stages Analysis (sometimes abbreviated CSA in Scia printed materials) has been performed, the results can be reviewed.

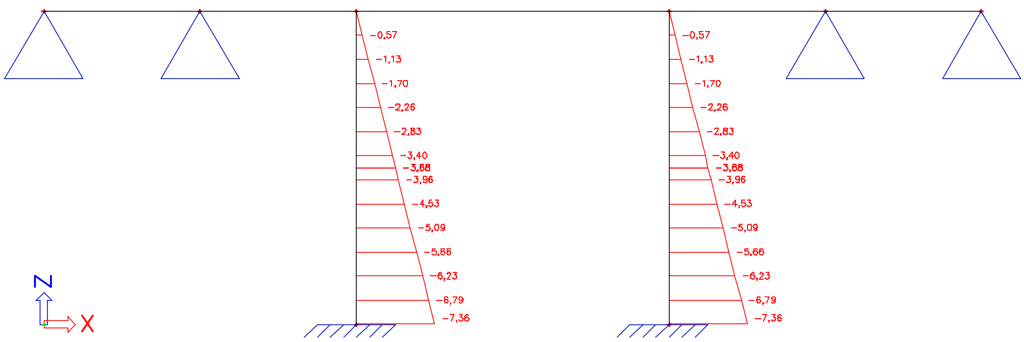
In general, you will be interested in two types or groups of results:

Results for load cases: As each construction stage is assigned to one load case (and this load case is exclusive for that construction stage, i.e. it is not used with any other construction stage), the results for load classes show the contribution of the particular construction stage to the overall distribution of a given quantity.

Results for load classes: The program automatically generates result classes during the Construction Stages Analysis. Two result classes are generated for each stage: ULS class and SLS class. (ULS class takes into account load factors gamma, SLS takes them equal to one (1)). The classes are numbered from 1 to the number of the last analysed stage. The results in each class show the current overall state (condition) of the structure after the particular construction stage.

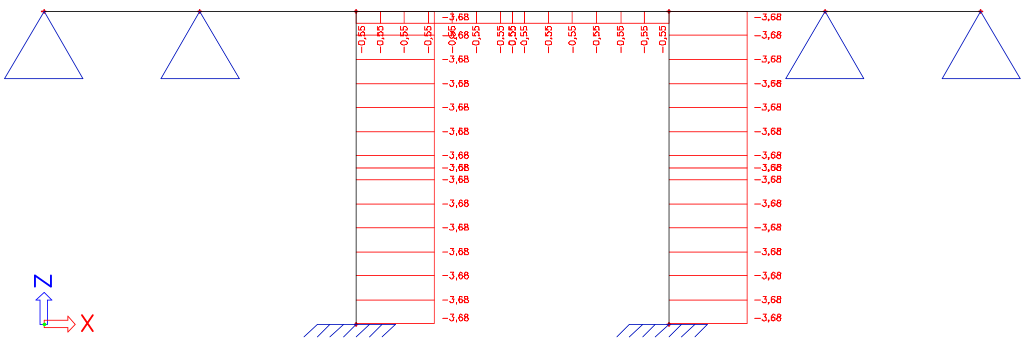


Let’s look into our example. If we ask the normal force for LC1:



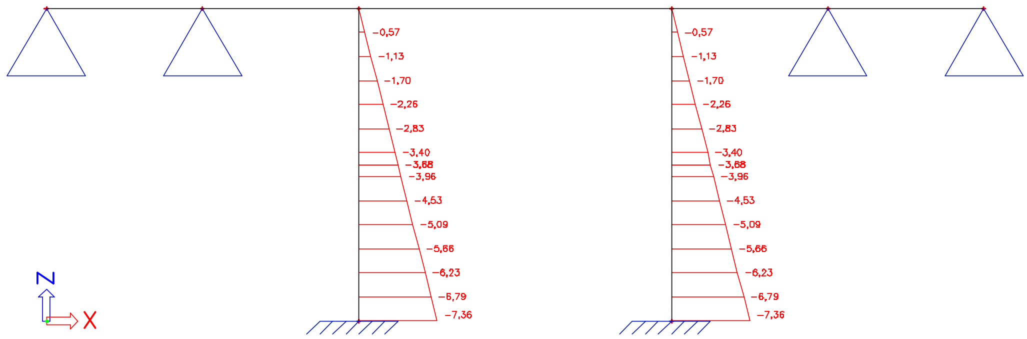
This is the normal force caused by the two columns.

For LC2 the normal force looks like:

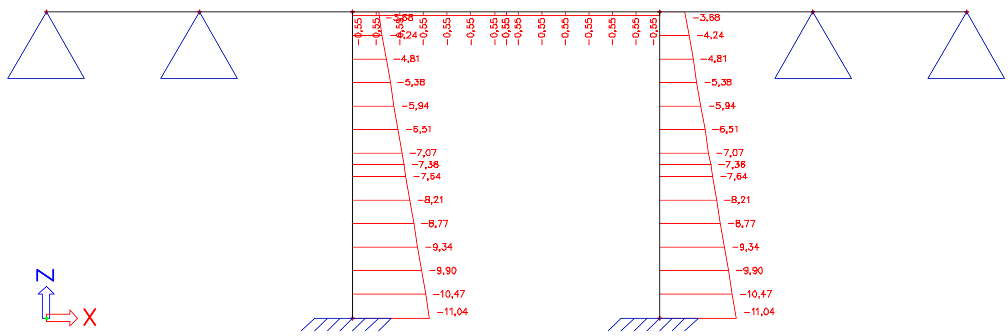


This is the normal force caused by the introduction of the beam in the middle only. It’s clear in these examples here that only the effect of the specific load case is taken into account.

If we look now for a certain class we will see this effect: e.g. the normal force for Class ST1 (ULS):

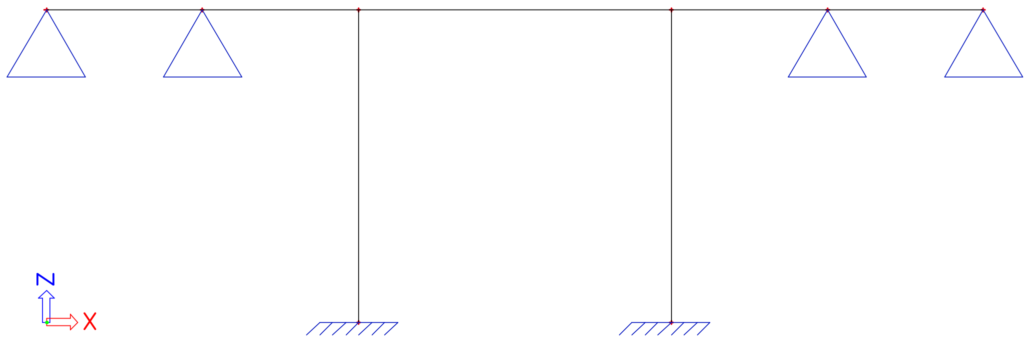


Or better the normal force for Class ST2 (ULS):

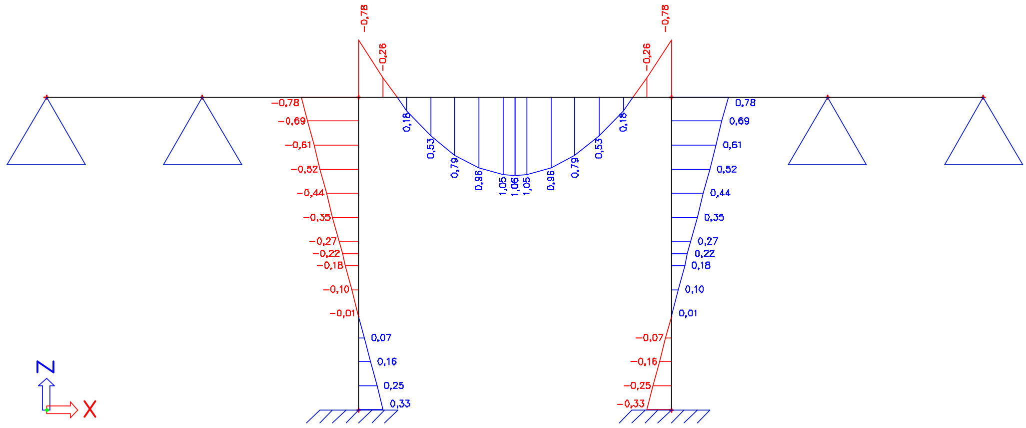


We can see clearly here that this is the sum of the normal force in LC1 and in LC2. This is the principle of linear construction stages.

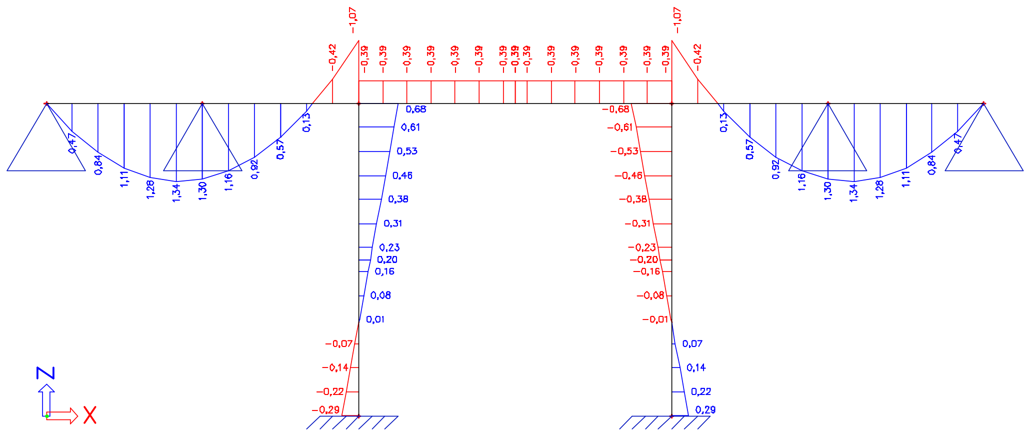
Let's take another example. Let's look at the moments. First for the loadcases:
LC1:



LC2:

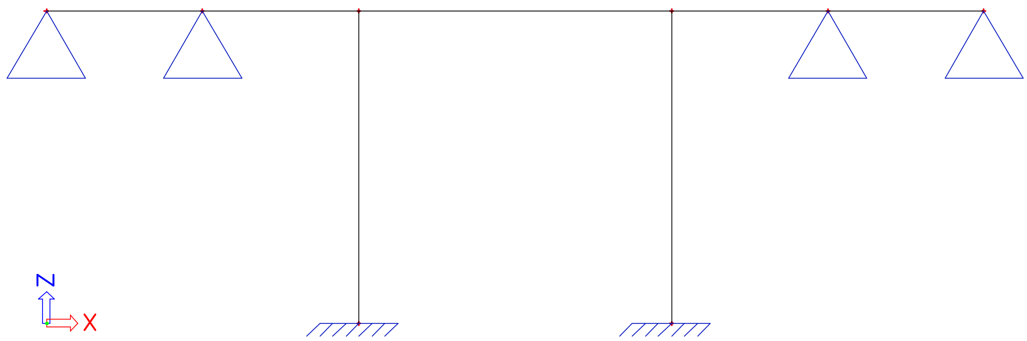


Here you can see that the introduction of a beam causes a moment in the columns.
LC3:

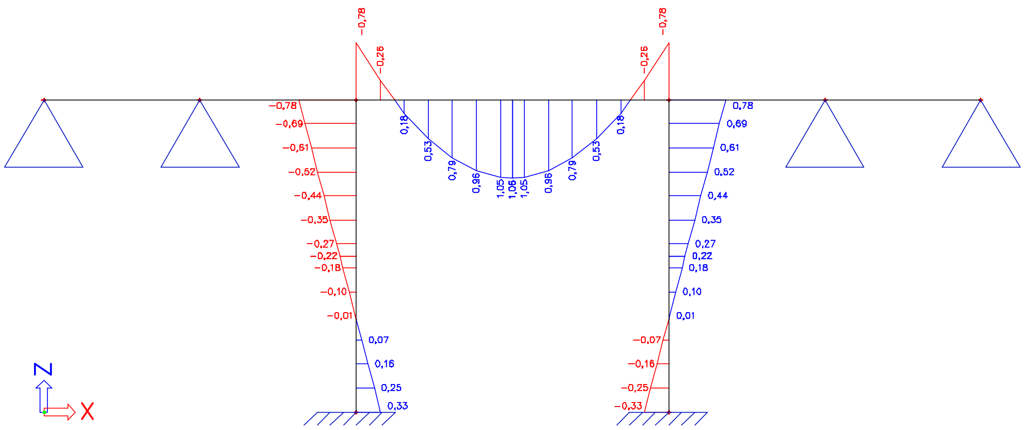


In this loadcase the left and the right beam are introduced. You can see that this action will cause extra moments (total 1,07 kNm) in the middle beam (-0,39 kNm) and in the columns (-0,68 kNm).

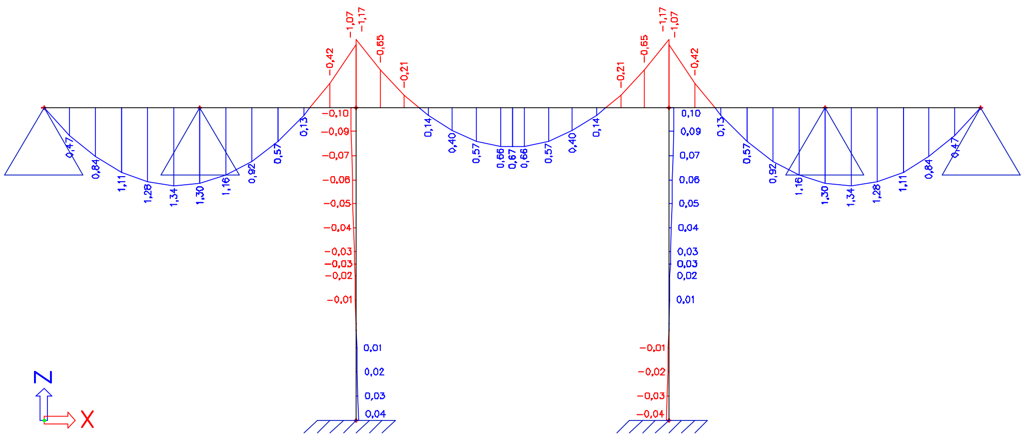
In the different stages the effects will be summed again:
Class ST1



Class ST2

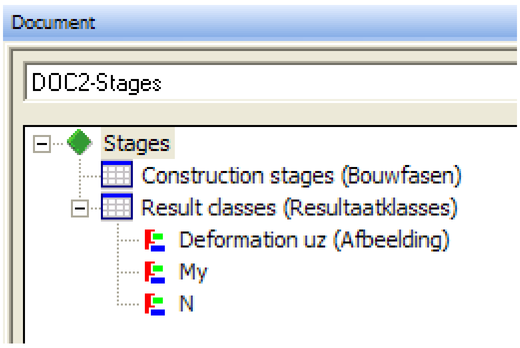


Class ST3



Note

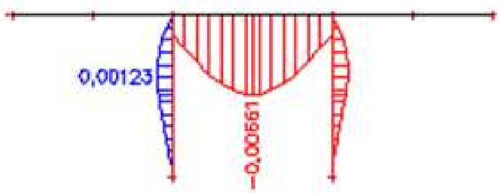
It can be very usefully when using the option nested tables in the document of Scia Engineer. This option will automatically generate all the results for each load case and/or stage:



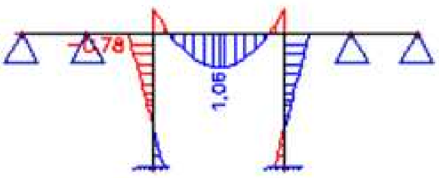
2.2. Result classes - ST2 (ULS)

Name	List
ST2(ULS)	F2-MAX

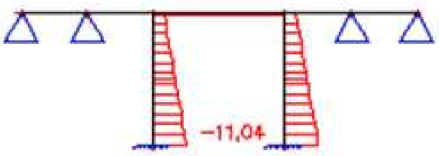
2.2.1. Deformation uz



2.2.2. My



2.2.3. N

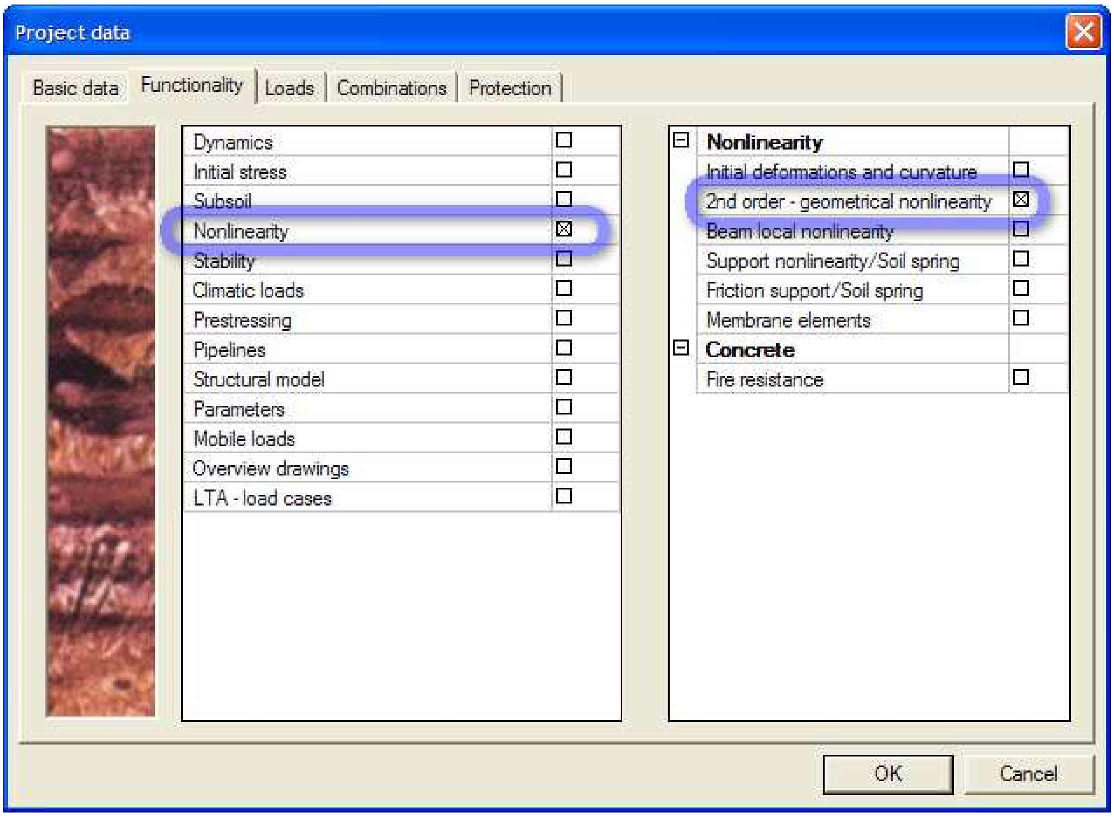


Nonlinear construction stages

The Analysis of Construction Stages (ACS) can be performed also as a non-linear analysis. Everything that was explained for linear Analysis of Construction Stages is valid also for this advanced type of calculation. There are a few differences.

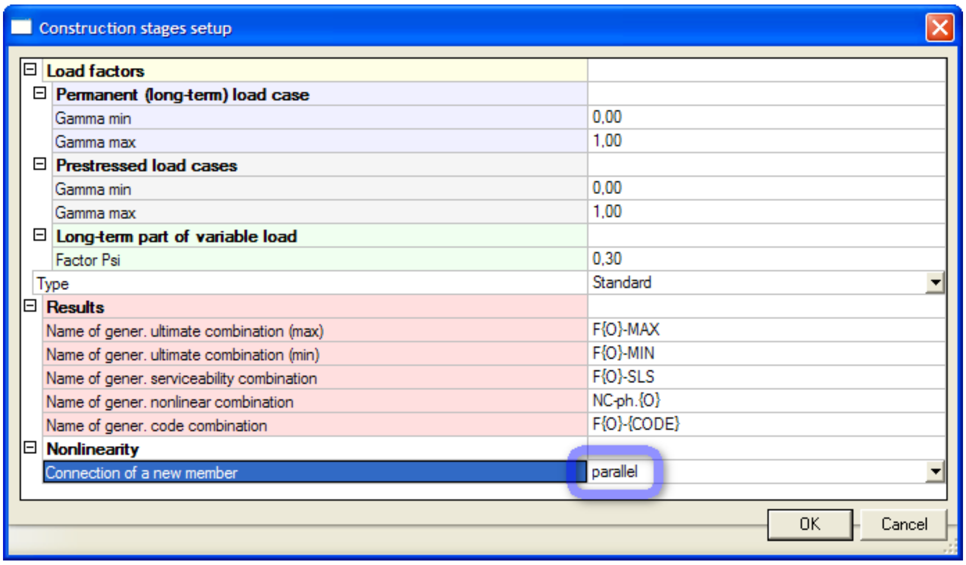
Project parameters

In the **Project setup > Functionality** item **Nonlinearity** and sub item **2nd order – geometrical nonlinearity** must be selected:



Tangent versus parallel connection of a new member

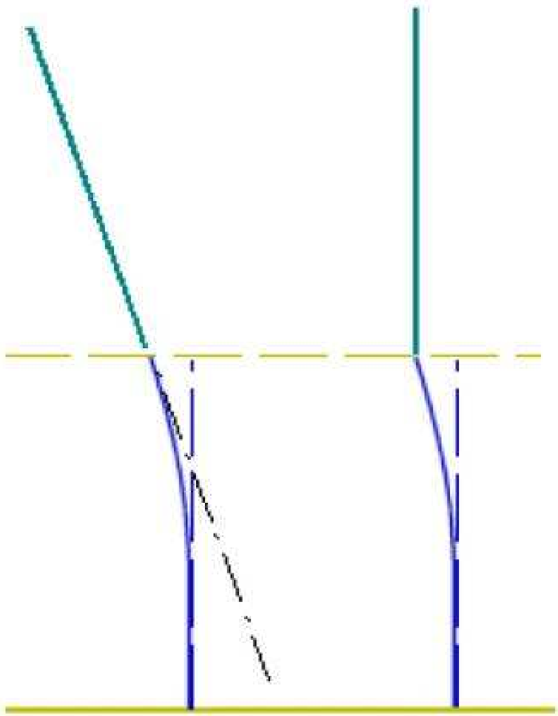
The setup dialogue for construction stages offers one more parameter:



'Connection of a new member': This parameter defines the geometrical configuration used for the connection of a new member in a new construction stage.

- ✓ **tangent**: the new member is attached to the "old" member in the direction of the tangent to the deformation line of the "old" member
- ✓ **parallel**: the new member is attached to the end of the deformed "old" member in the direction parallel to the direction of the new member on an undeformed structure.

The picture demonstrates the two options. The left hand side of the picture shows the tangent option. The right hand side, on the other hand, contains the second option.



TDA – Time Dependent Analysis

It is not possible to perform Time Dependent Analysis in combination with nonlinear analysis of construction stages. Therefore we will only describe this and we will not make an example in this workshop. Examples for nonlinear analysis of construction stages can be found in a specialized PIPFAS training.

Running the calculation

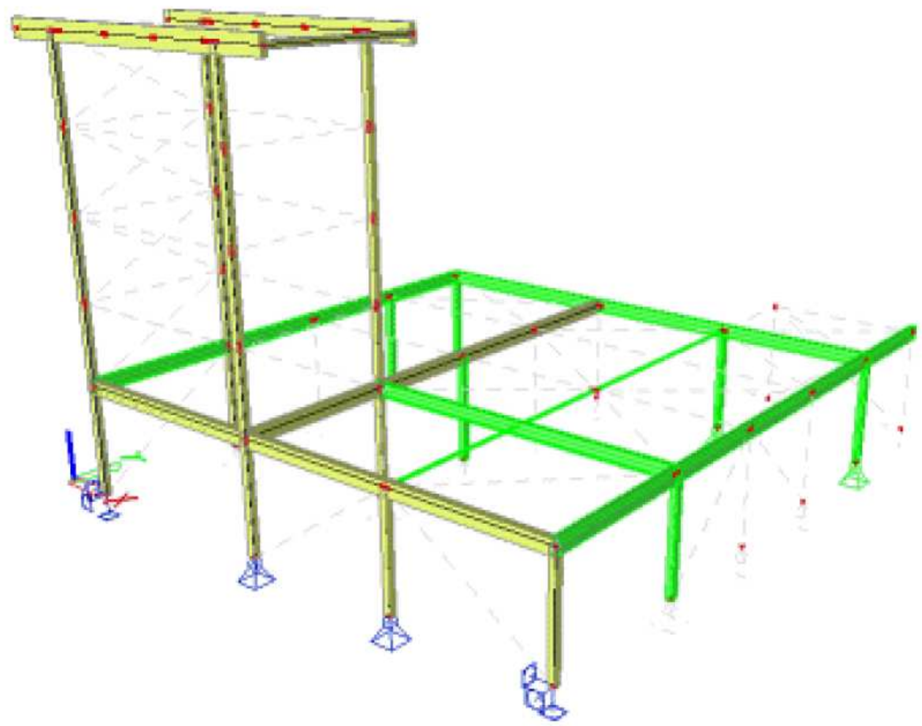
Procedure to run nonlinear ACS
1. Call menu function Calculation, Mesh > Calculation.
2. Select Nonlinear stage analysis.
3. Click [OK] to start the calculation.

Linear versus Non-linear construction stages

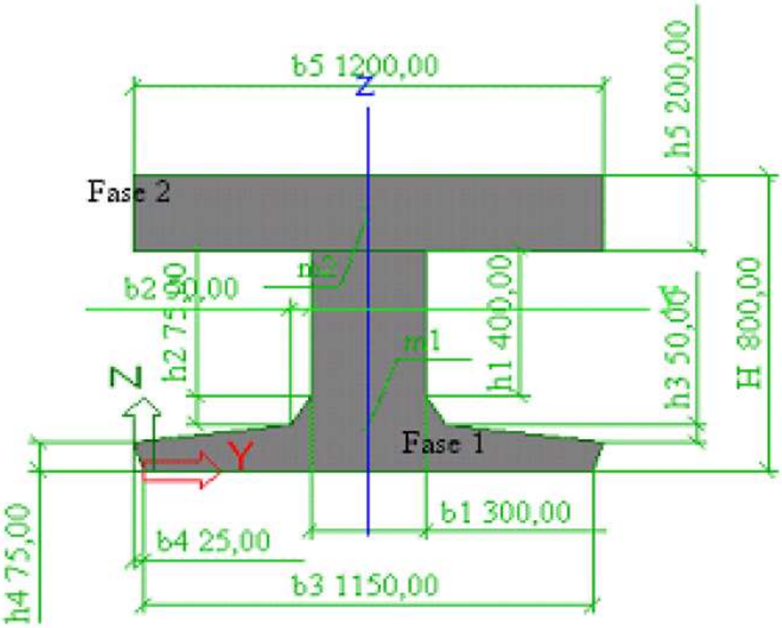
Let's look into the difference between Linear and Non-linear Construction Stages

Linear construction stages

The linear construction stages were developed mainly for the calculation of prestressed structures. It allows the user to model the construction process and life cycle of the structure. Despite the original purpose to apply this module to concrete structures, it can be in general used with any material. The user can add or remove supports, members and tendons:



For each construction stage, the safety factors can be set for the permanent and variable load cases including the prestress load cases, thus resulting in a bandwidth of min/max stresses/forces/deformations/reactions. Additionally, the user is able to model the segmentally constructed cross-sections by adding the newly cast (concrete) or installed (steel/timber/other) materials during the construction stage (see also next chapter):



The module of Linear construction stages is based on the superposition (linearization) of load cases. Therefore, the user can easily verify results by adding and removing individual load cases.

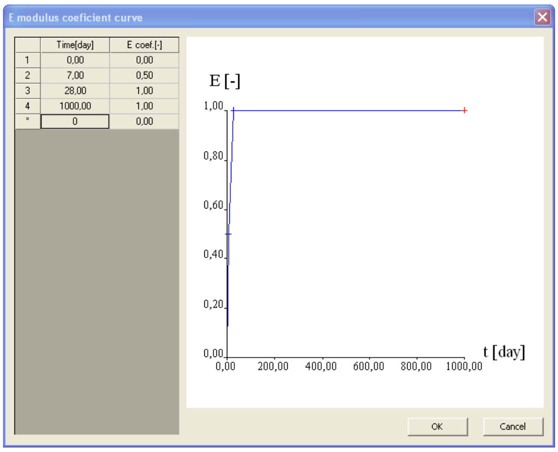
Note

Module Time dependant analysis (TDA) cannot run without this module.
The drawback of this module is that 2D members can only be added to the structure and not removed. Also, Young's modules cannot change and hinges can be neither added nor removed.

Construction Stages –E modulus changes

Up to now, Scia Engineer enabled the users to apply two approaches when analysing construction stages:

- Use the standard solver and calculate the construction stages without any effect of time. Only a sequence of individual models was analyzed and the internal forces changed according to changing boundary conditions.
- Use TDA (Time Dependent Analysis, more information will follow in one of the next chapters) calculation in which the full process of aging is taken into account, including relaxation of the reinforcement, creep and shrinkage of concrete.

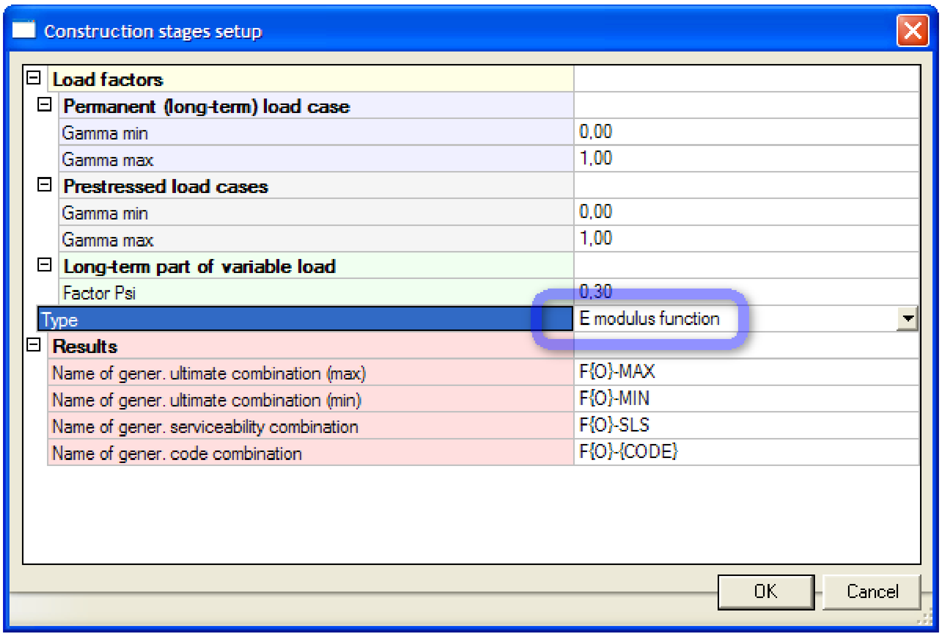


Scia Engineer version 2007 now newly introduces a third possibility:

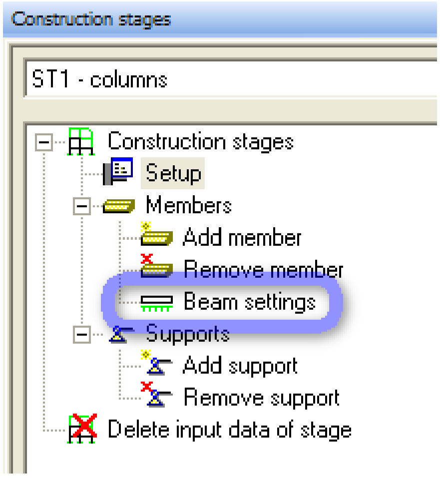
- Use the standard solver, but consider the change of modulus of elasticity (E) over time, i.e. aging of material, through a diagram that defines the changes of the modulus of elasticity over time (E-modulus diagram).

This new approach can be equally applied to both frame and plate-wall structures.
In one project, the user may define several E-modulus diagrams. It is even possible that each material used in the project has its own E-modulus diagram. The E-modulus diagrams can be assigned to all or just some materials used in the project.

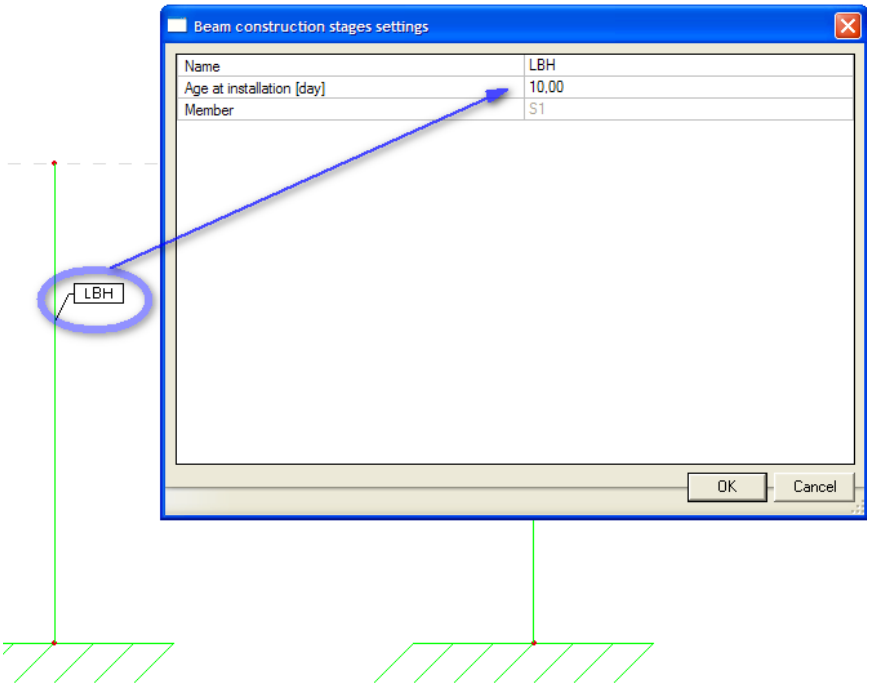
How does it work? In the '**Construction stages setup**' user can choose for E modulus function:



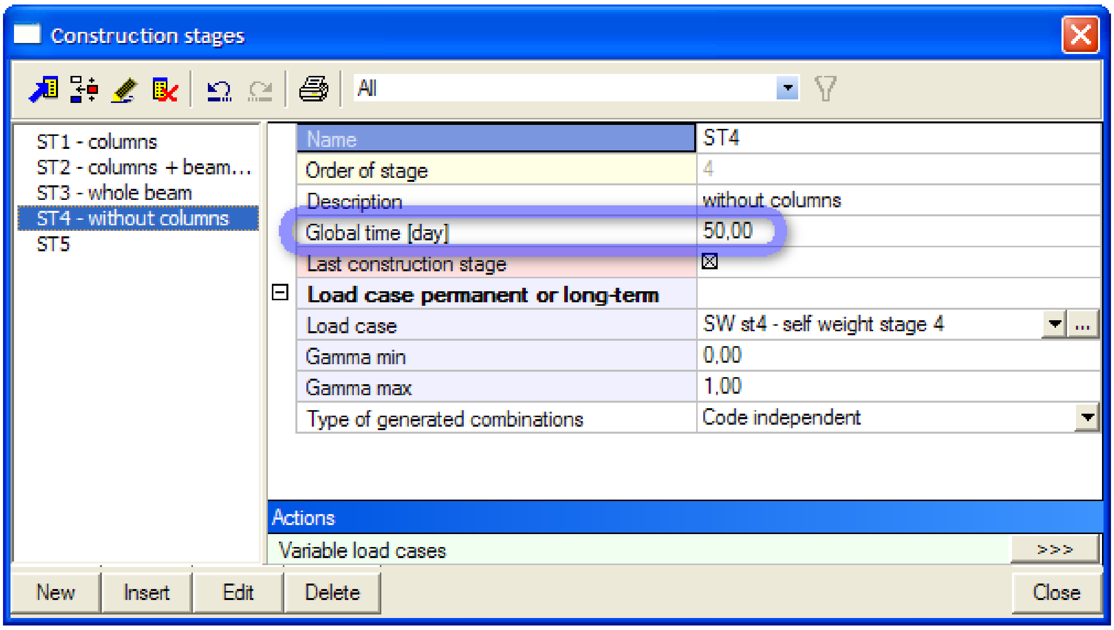
After this a new option will appear in the '**Construction stages**' service:



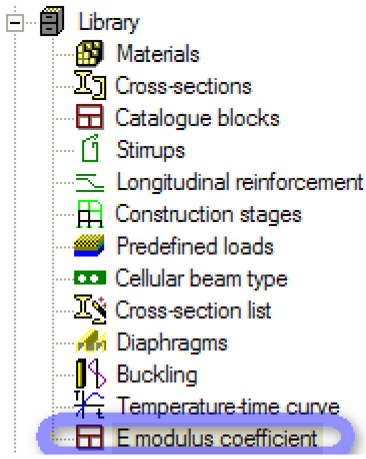
With '**Beam settings**' user can put a local beam history on a member in order to input the local age of the member (more information about this will be given in the chapter TDA):

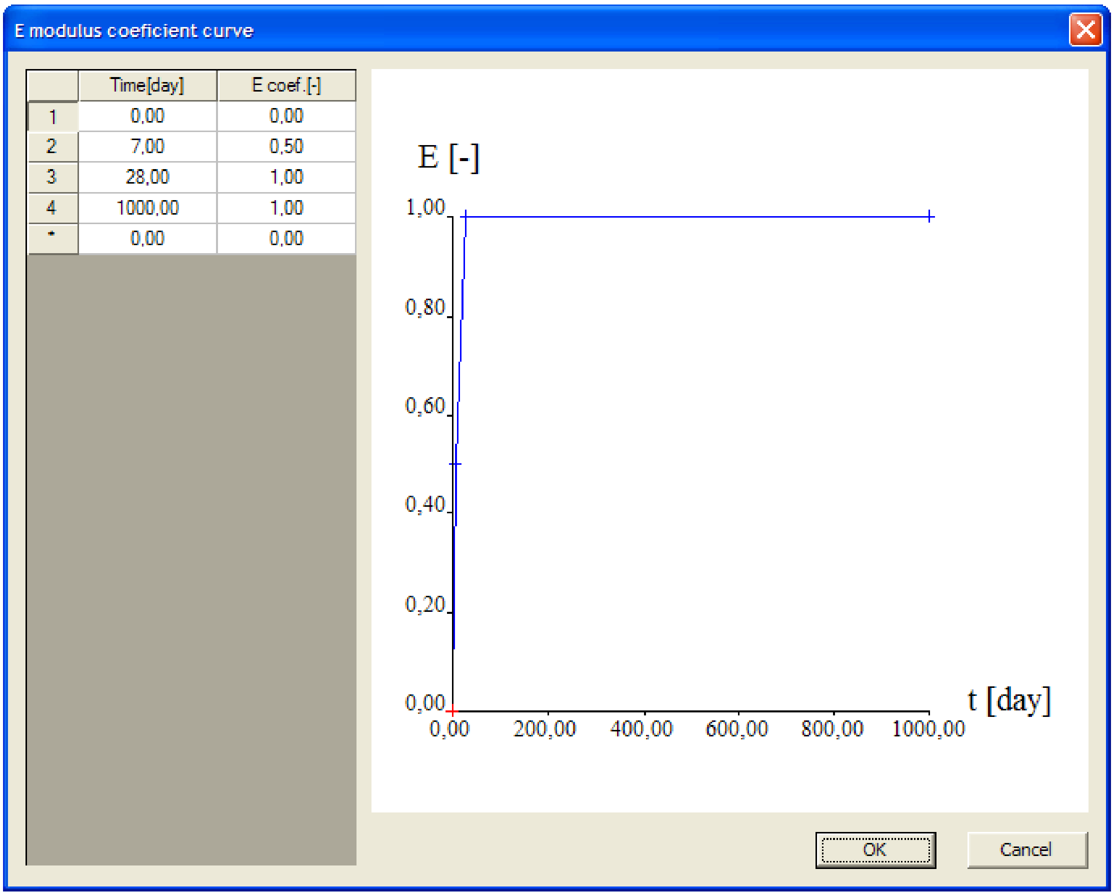


In the '**Construction stages**' manager it is now also possible to input a '**Global time**':

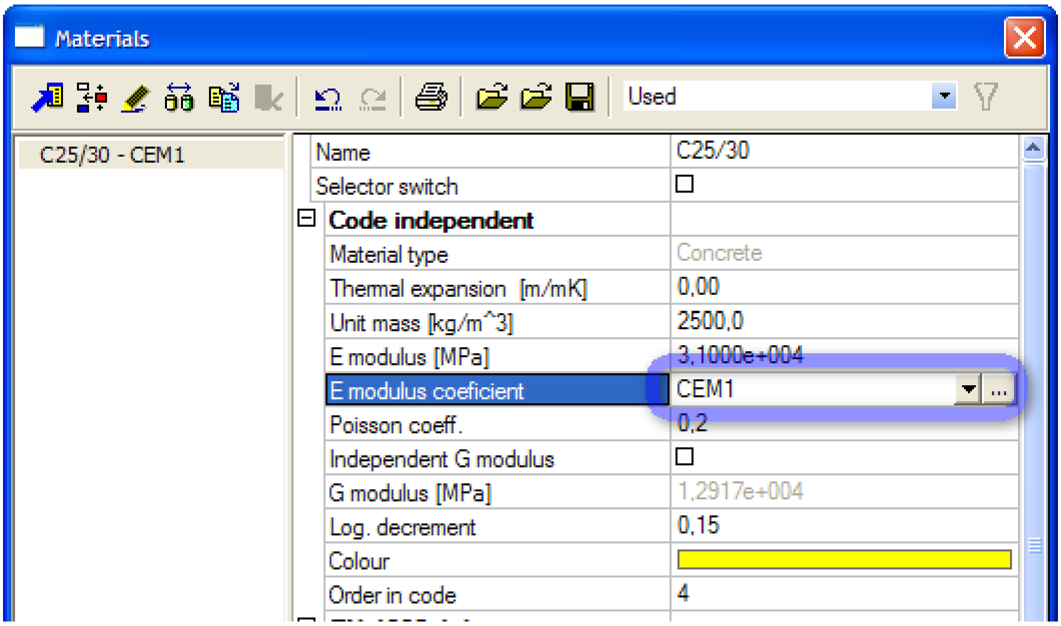


User can define in '**Library → E modulus coefficient**' his own E modulus function:





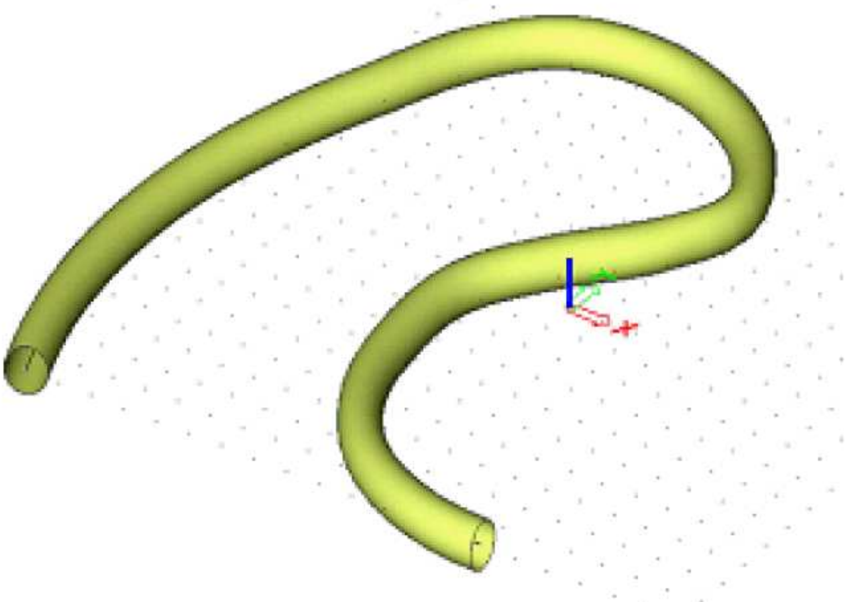
This function can now be taken into the linear staged calculation. User can use this function in relation to a certain material:



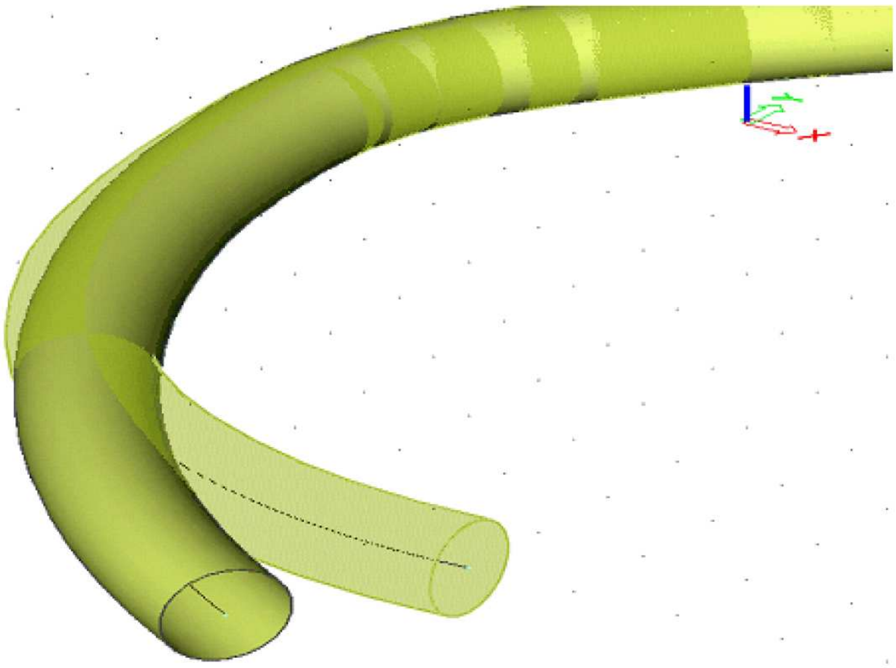
If user performs a linear staged calculation the function will be taken into account.

Non-linear construction stages

The non-linear construction stages were originally developed for the analysis of pipeline systems. In essence, it takes into account the deformed structure of the previous stage, whilst calculating a new construction stage.
Therefore, the term "non-linear" is used. This module can work in collaboration with the non-linear conditions and physical and geometrical non-linearity. The input of construction stages for this module follows the same principal and uses the same dialogues as for the linear construction stages.



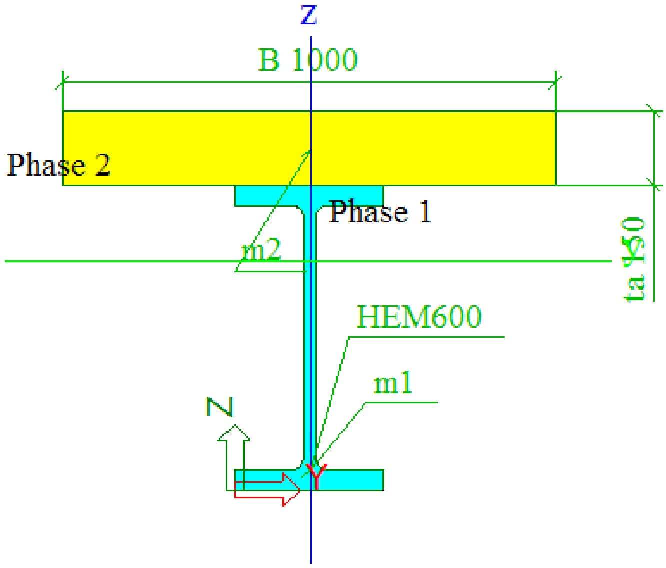
This module is based on the 2nd order-theory of Newton-Raphson method and requires a proper mesh and incrementing of the load. It generates non-linear combinations, each representing a construction stages.
Safety factors are all equal to 1.0, i.e. there is no bandwidth of results (min/max).



The drawback of this module is that it does not work for 2D members and TDA.

Phased cross-section

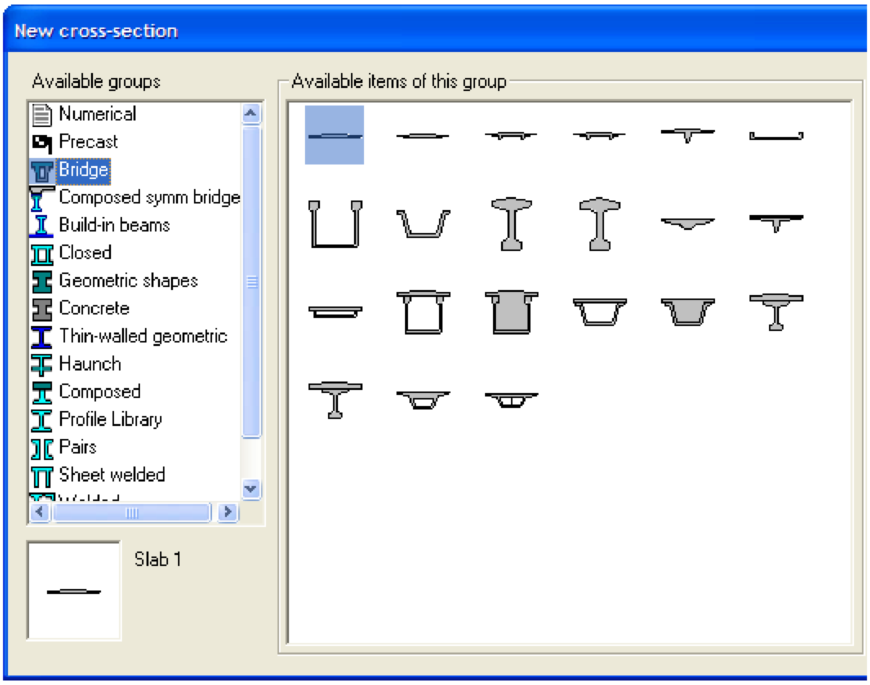
This chapter is related to the Analysis of construction stages and Time dependent analysis. Modules Construction stages and TDA can use all types of database cross-sections in Scia Engineer. A new feature called "phased cross-section" has been introduced for those modules. Phased cross-sections consist of two or more parts, each of which can be of different material.



Phased cross-sections allow for modelling of composite structures. The cross-section is built-up step-by-step starting by phase 1. Each phase of the crosssection is modelled by means of separate finite elements with eccentricity in the longitudinal direction. Therefore, stress redistribution between two different phases of cross-section will appear in TDA analysis due to creep and shrinkage of concrete. If any phase consists of more separate parts (of the same or different materials), only one finite element will be generated for that phase between two nodes of the FEM mesh.

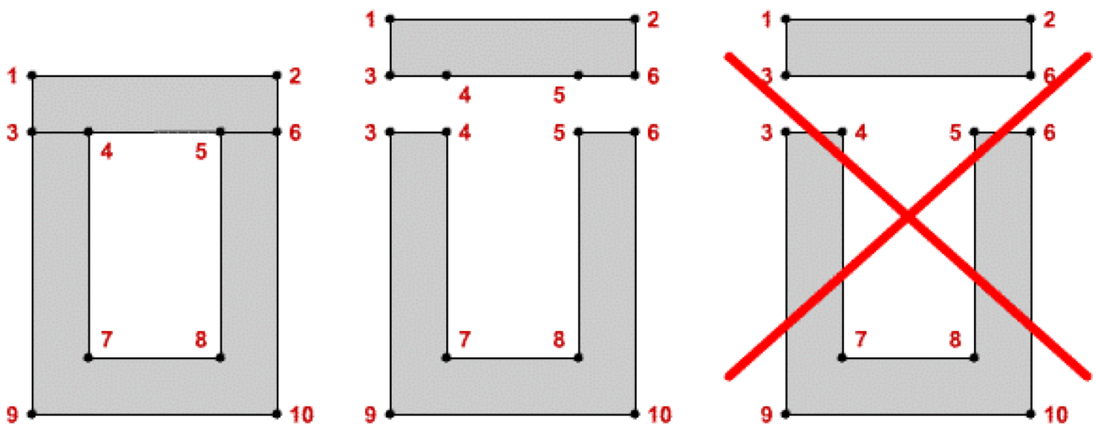
Sectional characteristic of individual parts will be transformed to one material. The generated finite element will have transformed cross-sectional characteristic. For that reason no stress redistribution can be expected in the analysis between individual parts of one phase.

Phased cross-sections can be created as a General cross-section. General cross-section can be defined by means of a polygon drawing or by conversion from other types of database cross-sections. Also some predefined bridge cross-sections can be defined as phased:



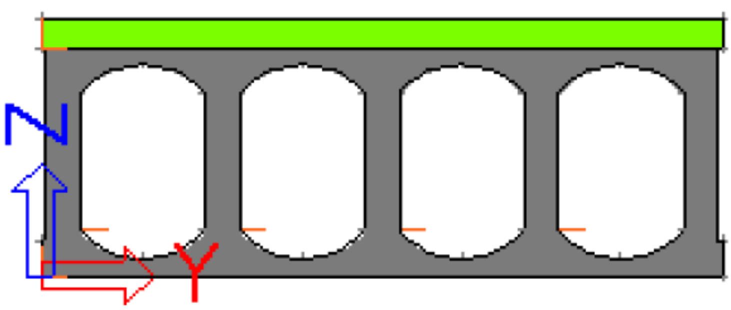
Currently, only two phases can be defined for one cross-section.

One important condition must be fulfilled when a phased general cross-section is created. The condition is clear from the following picture.



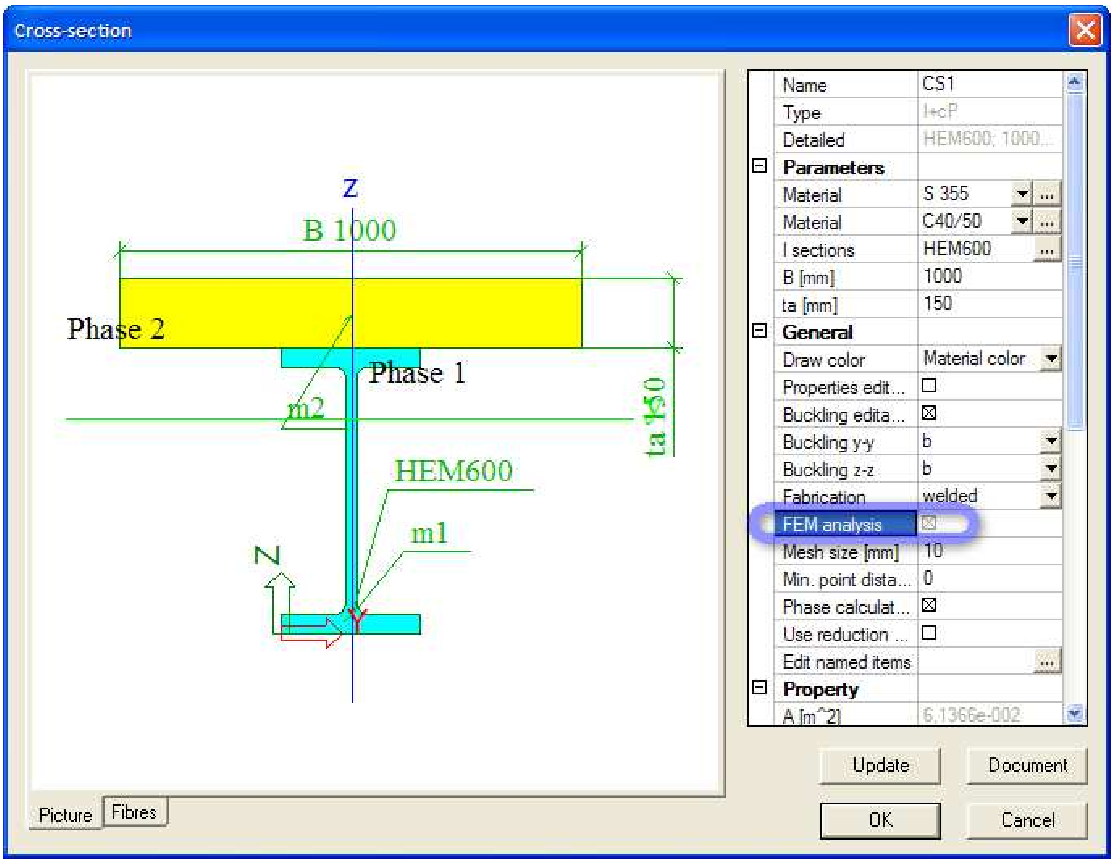
Example of a phased cross-section

The following picture shows a hollow core floor slab [phase 1] (400 mm high) with a 50 mm topping [phase 2]:

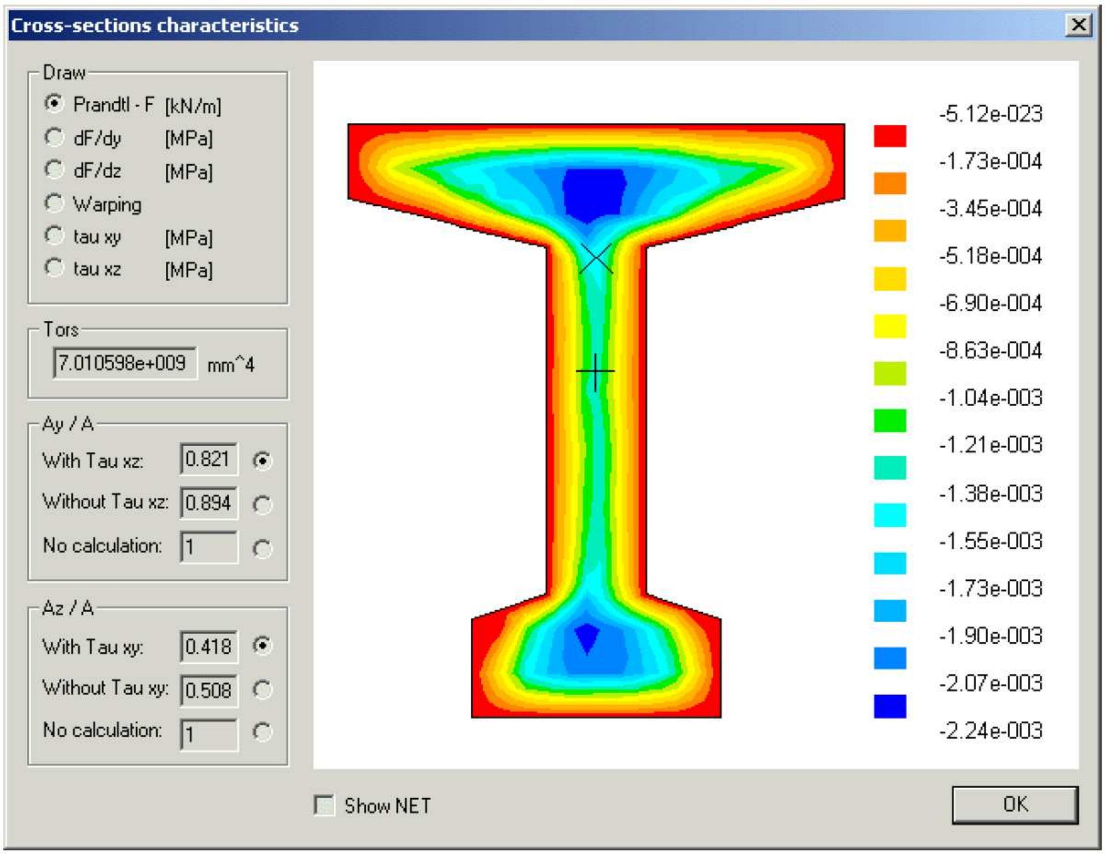


Sectional characteristics of phased cross-section

It is recommended to use FEM analysis for the calculation of sectional characteristics of a phased cross-section. This can be done in the editing dialogue of a cross-section by ticking (selecting) the option FEM analysis:



When this option is ON, the program starts a special engine to calculate the sectional characteristics. The result of the analysis is shown in a separate dialogue:



It is possible to view some results and also to select the way for the determination of shear-related parameters: Ay/A and Az/A (see the note below).
Note: It is up to the user to review the shear-related values and select the correct (or most correct) one manually.

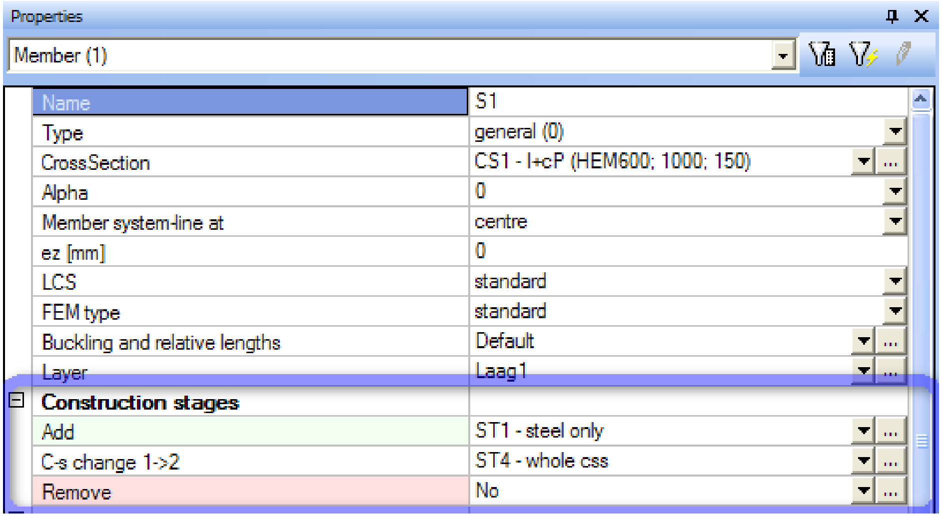
Defining the introduction of a new phase of the cross-section

Chapter Defining the changes to the structural scheme describes how to introduce a new member or a new support in a specific construction stage.

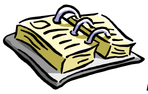
This chapter deals with the introduction of a new part of a phased cross-section, e.g. casting of composite slab, etc.

Procedure to install a new cross-section phase

1. Select the beam with the phased cross-section.
2. The property window displays the properties of the beam.

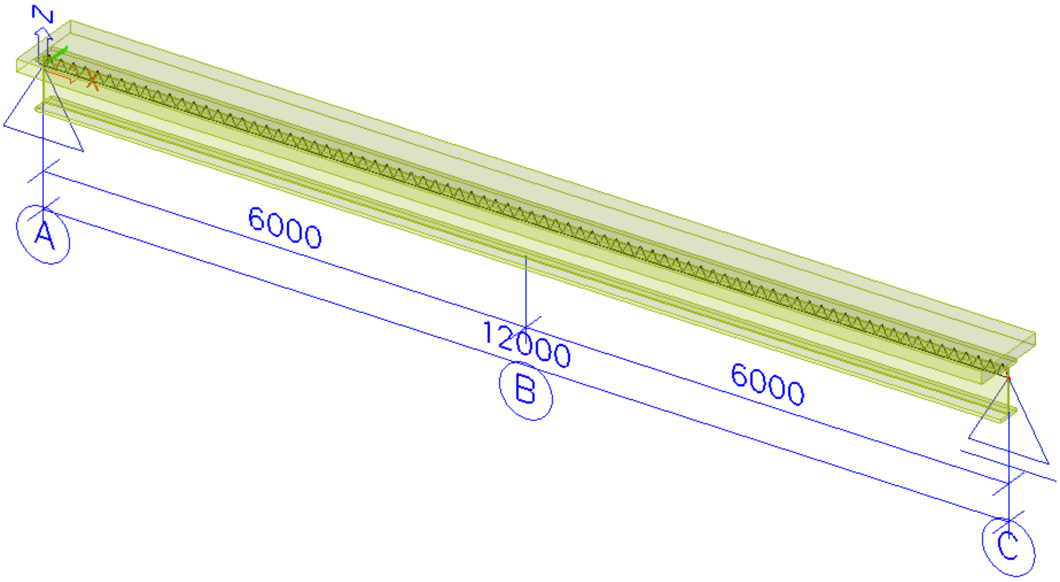


3. One of the property groups is named ‘**Construction stages**’.
4. Use item Add to define the stage in which the base part (phase one) of the cross-section is installed.
5. Use item ‘**Css change 1->2**’ to define the stage in which the second part (phase two) of the cross-section is installed.

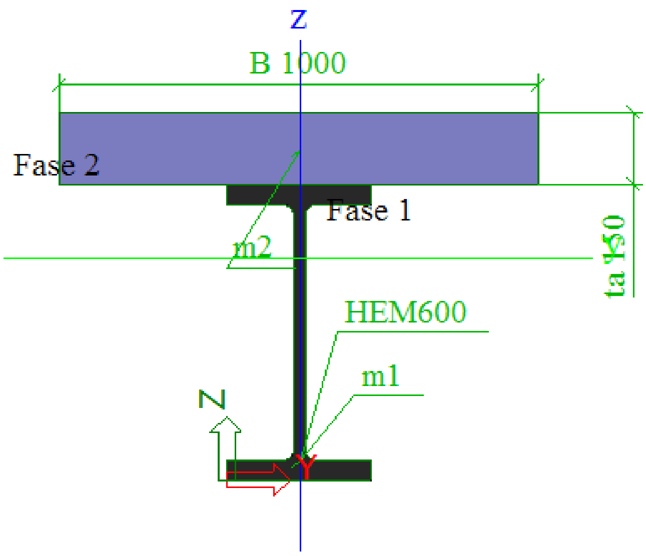
 *Let's make an example.*

Practical example of staged css: bridge

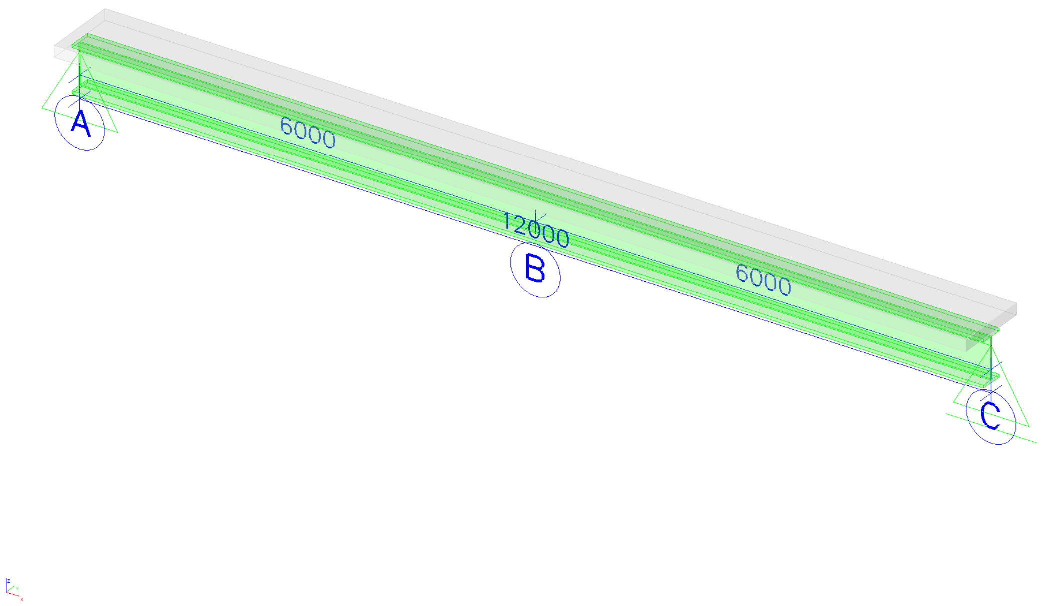
Beam on two support with a staged steel and concrete cross section:



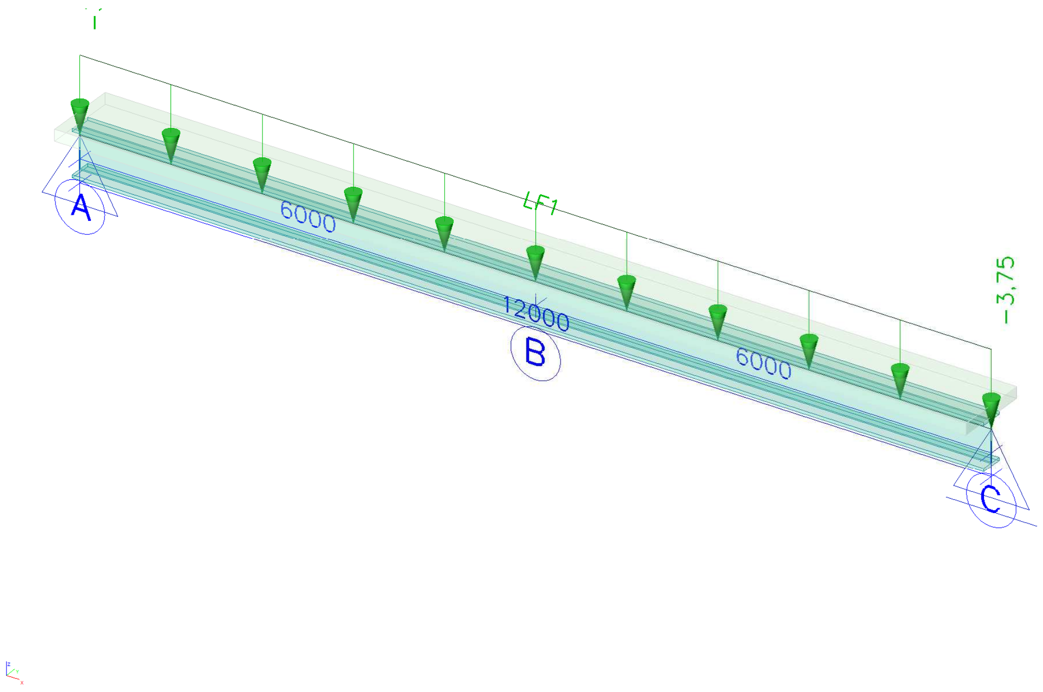
Steel: HEM600 S355
Concrete: 1000x0,15mm C40/50



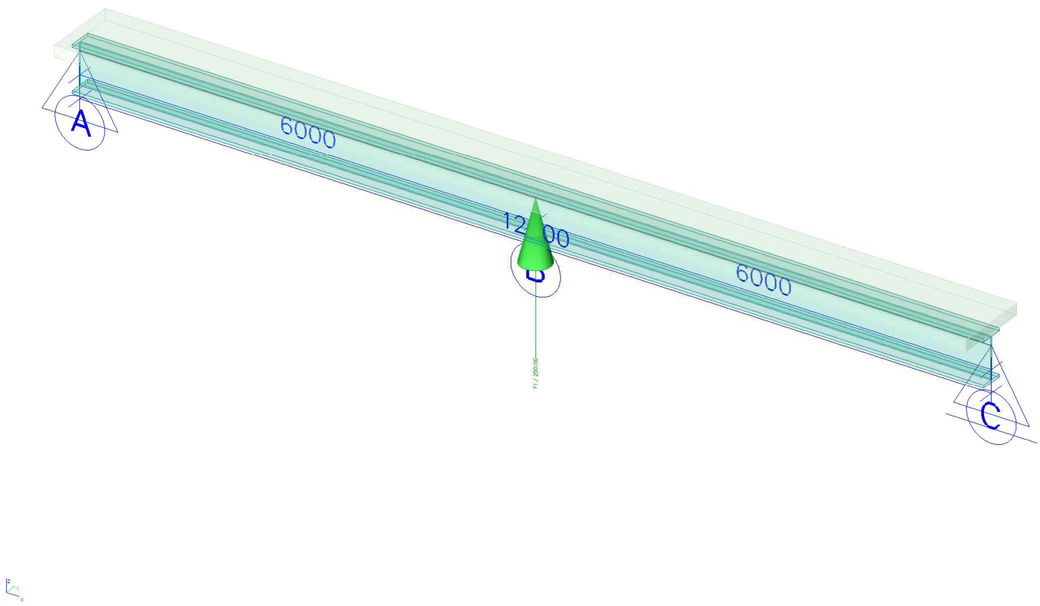
<u>Stage 1</u>
Steel cross-section only
Load case: Self weight steel (Permanent)



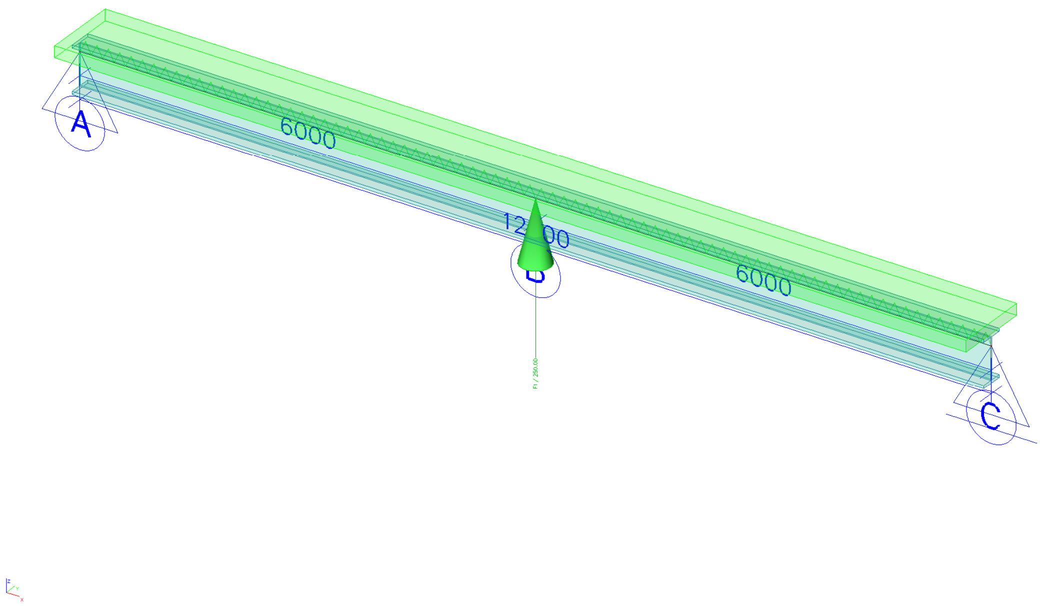
<u>Stage 2</u>
Cast of concrete
*Load case: distributed load Weight of wet conrete = 25*0,150*1 = 3,75kN/m (Permanent)*



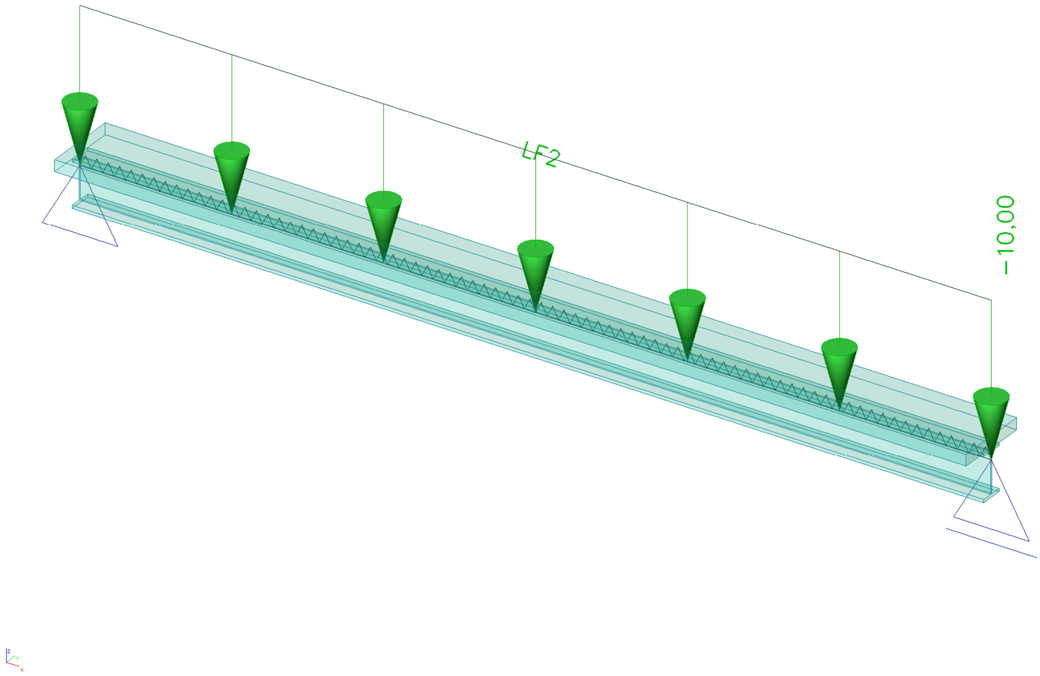
<u>Stage 3</u>
Input of intermediate support
Load case: Point force 250kN (Permenant)



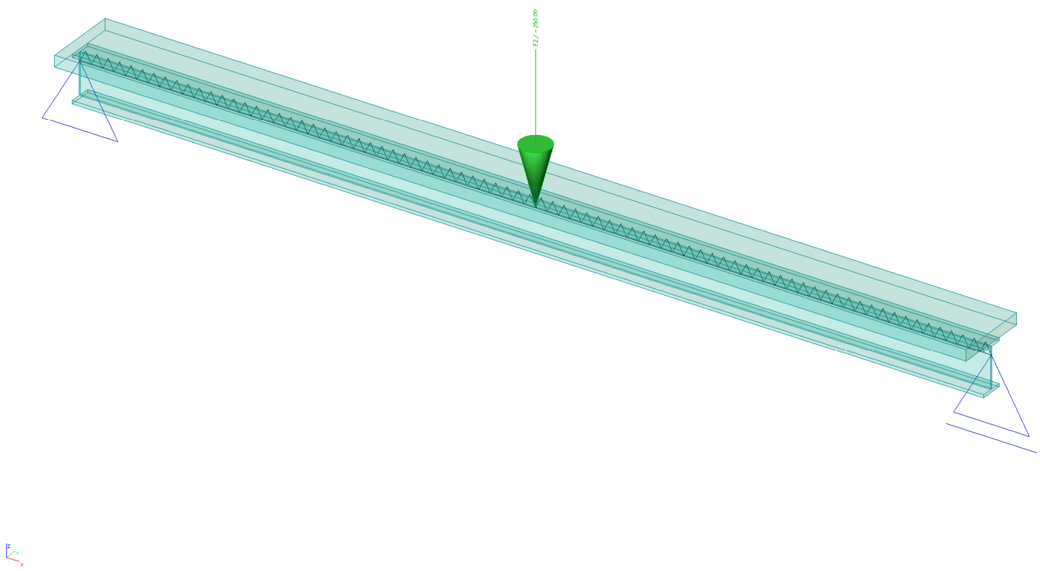
<u>Stage 4</u>
Collaboration of whole cross-section
Load case: Empty permanent loadcase



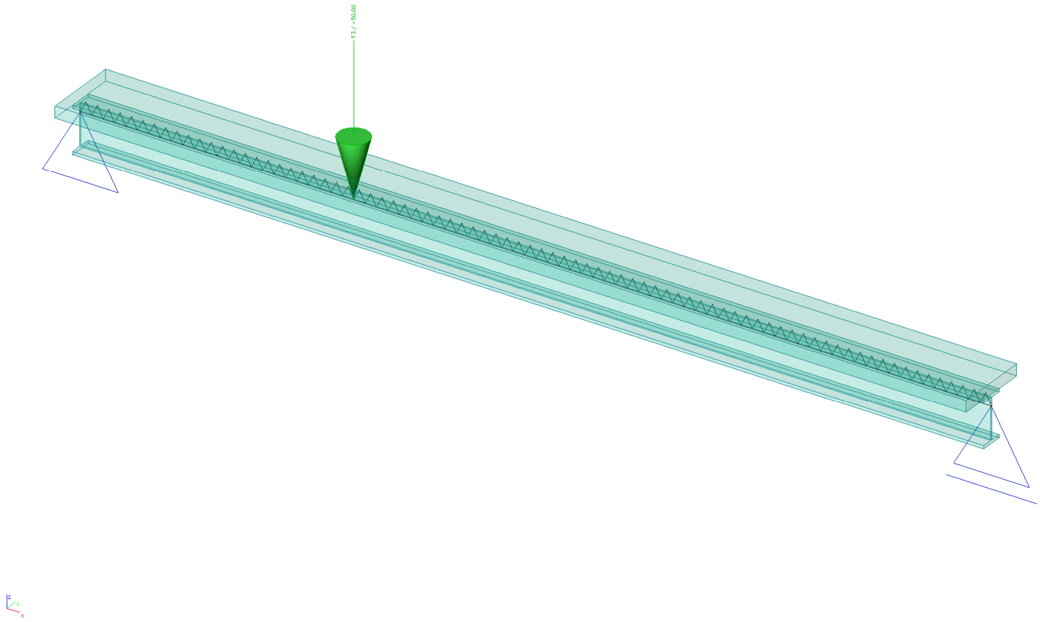
<u>Stage 5</u>
Casting the deck
Load case: Line force 10kN/m (permanent)



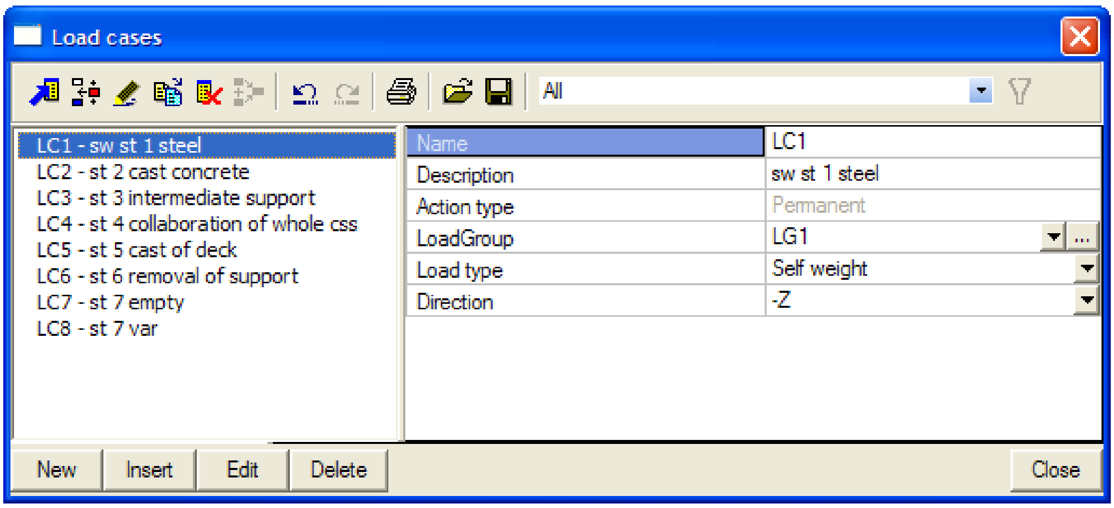
<u>Stage 6</u>
Removal of intermediate support
Load case: point force of 250kN oppositie of stage 3 (permanent)



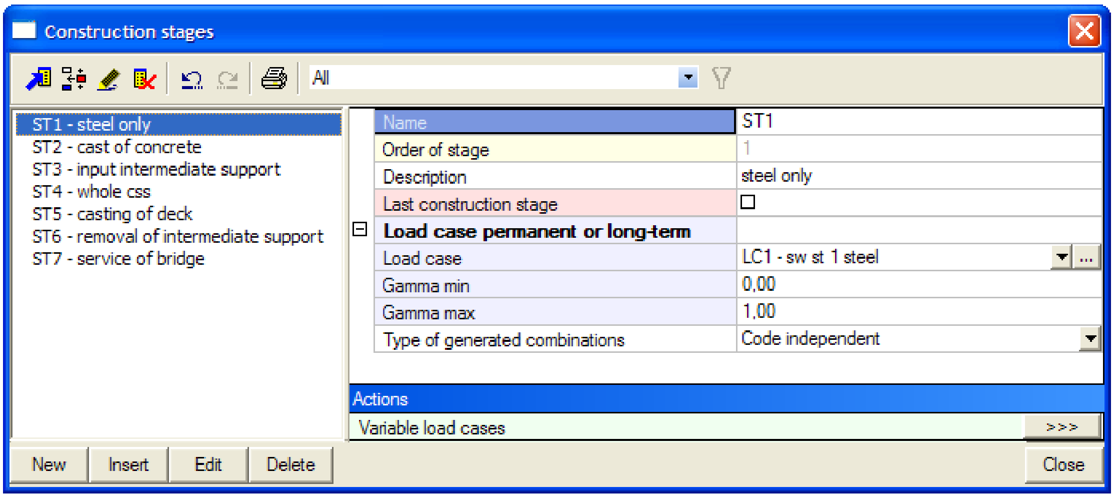
<u>Stage 7</u>
Bridge in service
Load case: point force on structure (variable)
Empty permanent loadcase



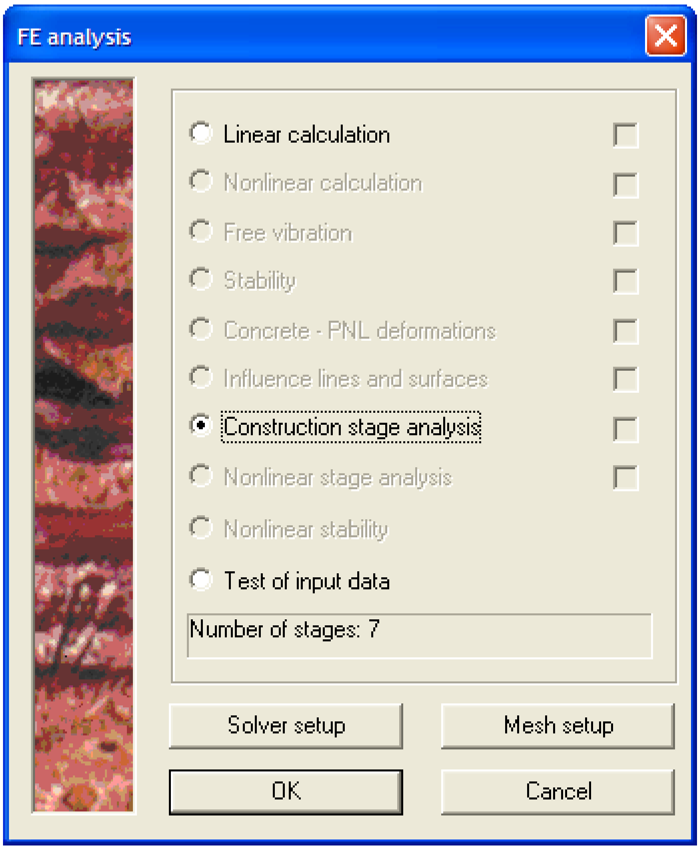
The load cases that are made in this project:



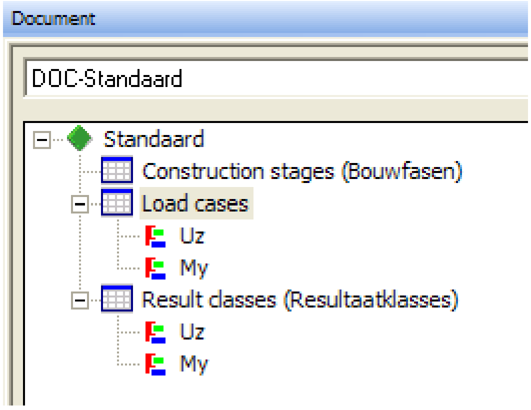
The stages should look like this:



Now we can launch the linear staged calculation:



Now user can use the document again to review the results:


Document
DOC-Standaard
Standaard
Construction stages (Bouwfasen)
Load cases
Uz
My
Result classes (Resultaatklasses)
Uz
My

Part III : Prestress

Brief introduction to prestressing

The module Prestressing allows for the analysis of pre-tensioned beams. Furthermore, it is possible to calculate and show the short-term losses. The data defining the location and shape of strands or wires are then used during the structural analysis for the automatic generation of the finite elements of the structural model and the calculation of its equivalent load including the short-term losses.
To sum up, the module Prestressing enables you to analyse the effects of prestressing for 2D. It can be used for the linear analysis of the final stage of the structure. When combined with the module Construction stages it is also possible to model the stepwise prestressing during the assembly of the structure. When also combined with the module TDA one can respect the impact of the rheology of concrete. However, the module TDA can solve 2D frame structures (project type Frame XZ) only (at this moment).

Modelling of prestressing

The prestressing force is not constant along the length of the tendon and over time. It has to be considered at various sections and at various construction stages respecting the prestressing losses. Some of the losses are calculated in advance by the "pre-processor". These are short-term losses and are marked "A" (in "a"dvance), see below. Since the tendons (or groups of tendons) are modelled as individual eccentric elements, the calculation of the other losses will be included in the "m"ethod (marked "M") for the structural analysis itself. The TDA (see chapter TDA) solver calculates these types of losses automatically and they are displayed in Results > Tendon stresses.

Losses during tensioning (before or during transfer of prestressing):

- ✓ Anchorage set loss, A Losses due to sequential prestressing (caused by the elastic deformation of concrete), M
- ✓ Losses due to deformation of stressing bed, A
- ✓ Losses due to elastic deformation of the joints of segmental structures sequentially prestressed, M (if the joints are included in the structural model
- ✓ Losses due to steel relaxation, A
- ✓ Losses caused by the temperature differences between prestressing steel and the stressing bed, A

Losses after transfer of prestressing (long-term losses):

- ✓ Losses due to steel relaxation, M
- ✓ Losses due to shrinkage of concrete, M
- ✓ Losses due to creep of concrete, M

Losses at service:

- ✓ Losses (changes of prestressing) caused by life load, M (calculated in standard Scia Engineer solver)

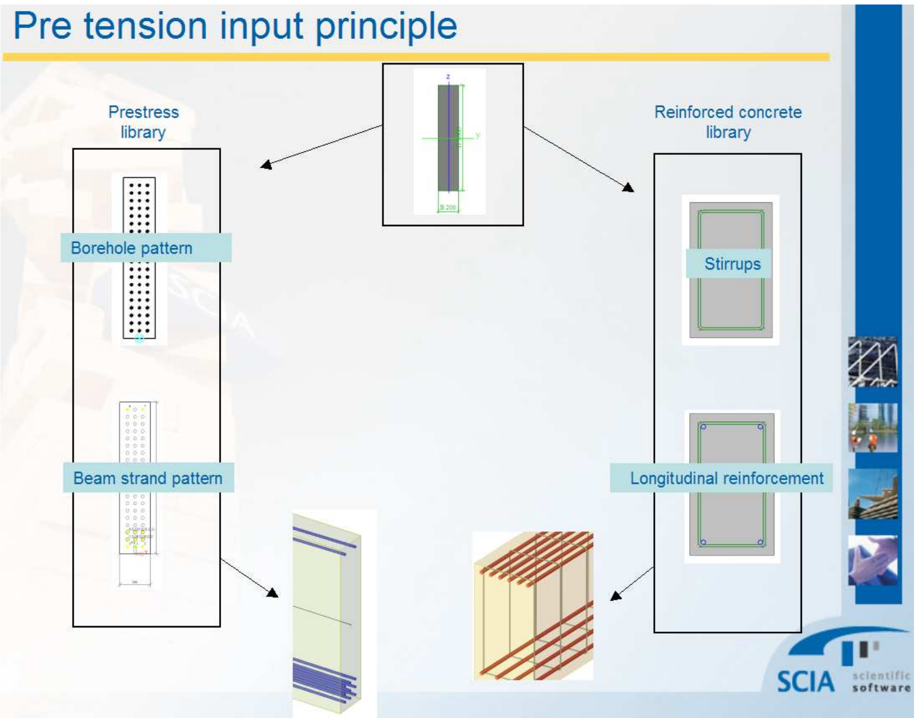
The local time axis for prestressed elements includes two nodes. The time of stressing is identical with the time of the stage. An additional time node is generated as the time of anchoring (installing of prestressed element into stiffness matrix). Time increment to anchoring is a virtual time. The purpose of introducing this time is to distinguish the moment of equivalent load application and the moment of installing the prestressed elements into stiffness matrix. These time nodes are identical for pre-tensioned tendons and they have no links to the time information required for calculation of losses. Strain equivalent to the relaxation to be passed in long-term is applied in several time nodes following the time of anchoring. The increments of internal forces and deformations caused by the relaxation are added to the results of dead load cases of construction or service stages, or they are added to "empty" load-cases generated automatically for creep and shrinkage effects. The effects of creep, shrinkage, and relaxation are mixed together (they cannot be separated, because they interact in the reality).

Pre-tensioned prestressed concrete

Introduction to prestressing

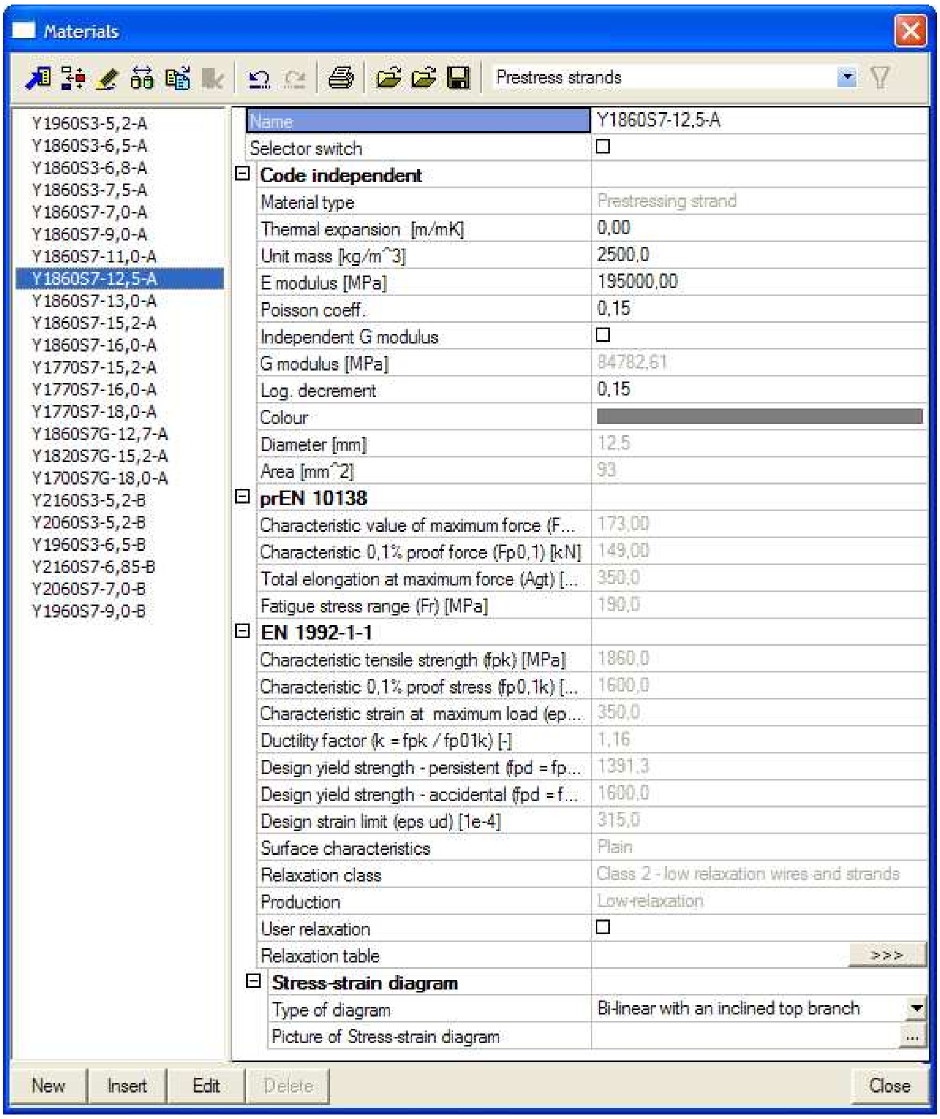
Module Prestressing makes it possible to define geometry, material and other properties of prestressed tendon. The tendon can be inserted into beams. It is possible to define pre-tensioned internal tendons. The tendons are defined through strand patterns that are supposed to be symmetrical in a beam, thus only one (symmetrical) half of the strand along the beam must be defined. The input is made in three steps:
1. Bore hole pattern is defined, i.e. the location of holes in the "face-plate" is defined.
2. Sectional strand pattern is specified, i.e. which holes of the "face-plate" are "filled" with a strand/wire/bar.
3. Beam strand pattern is input, which means that the shape of the strands/wires/bars along the beam is defined.

This method is written in the same philosophy as the input of practical reinforcement :

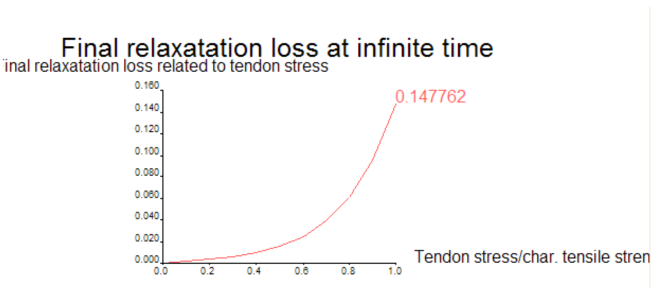


Materials of Prestressing Tendons

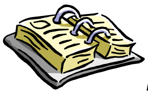
The system database contains all materials for prestressing tendons listed in EC2 Code, CSN 73 12 01 and CSN 73 62 07 (Czech standards). Considering the fact, that the material properties of prestressing tendons are dependent on diameter of prestressing unit (i.e. strand/wire/bar), the materials are listed in the system database not only according to the type, but also according to the diameter of the unit.



The dialog and parameters are code and type dependent. The relaxation table is defined in the system database for each prestressed material. Button Relaxation table can bring the relaxation table on view. Also diagrams of the relaxation values (after pressing button Graph) can be displayed, if required.



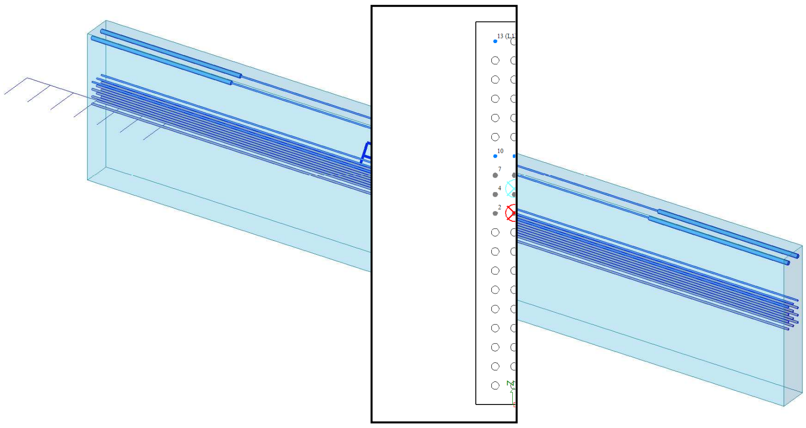
The user may also edit the values in the relaxation table. To do so, it is necessary to check option User relaxation first. Only then you can open the Relaxation table dialogue and edit the values in it.



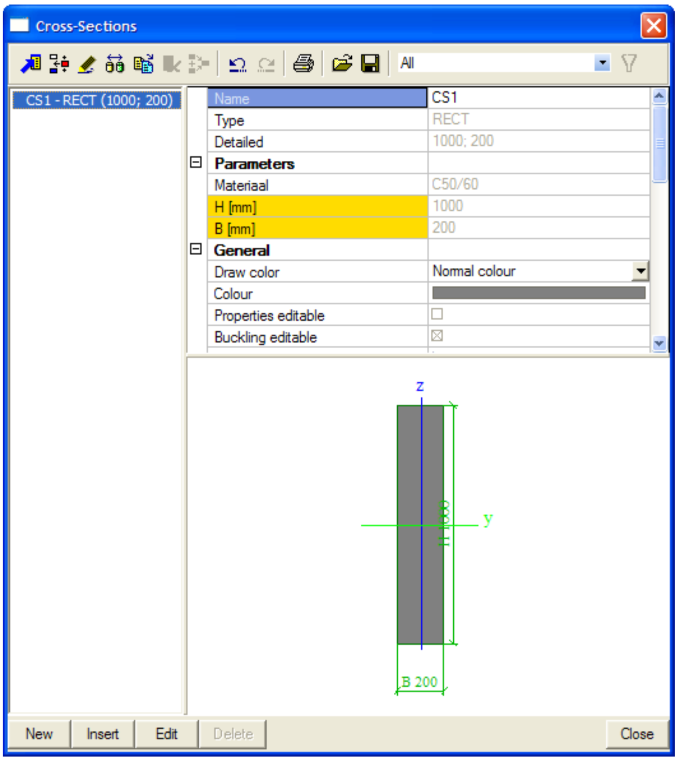
Let's prepare an example in order to better understand all the options:

Example: prestress with pre-tensioning

We will input a cantilever beam with some prestressed strands in it:



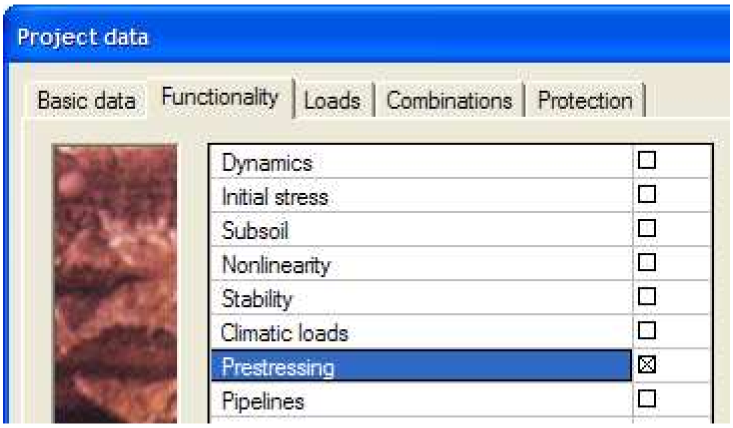
The beam has the following cross section:



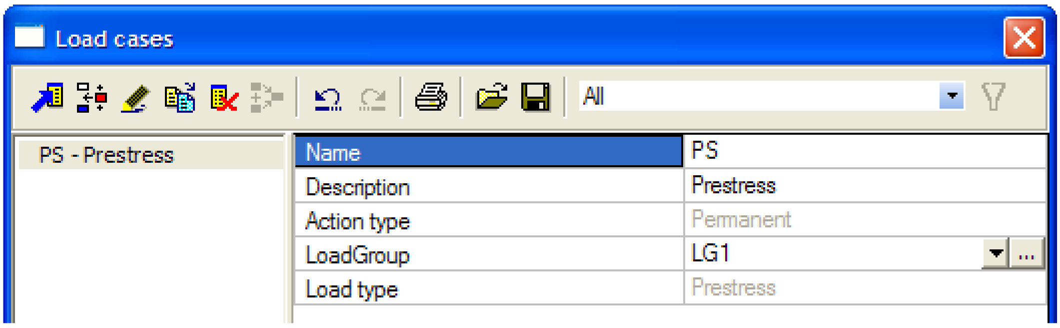
We will input the following prestress in the strands:
Prestress steel: Fep1860 Y1860S7-16
Prestress : 1250 N/mm²

Work method:

- Activate prestress functionality
-



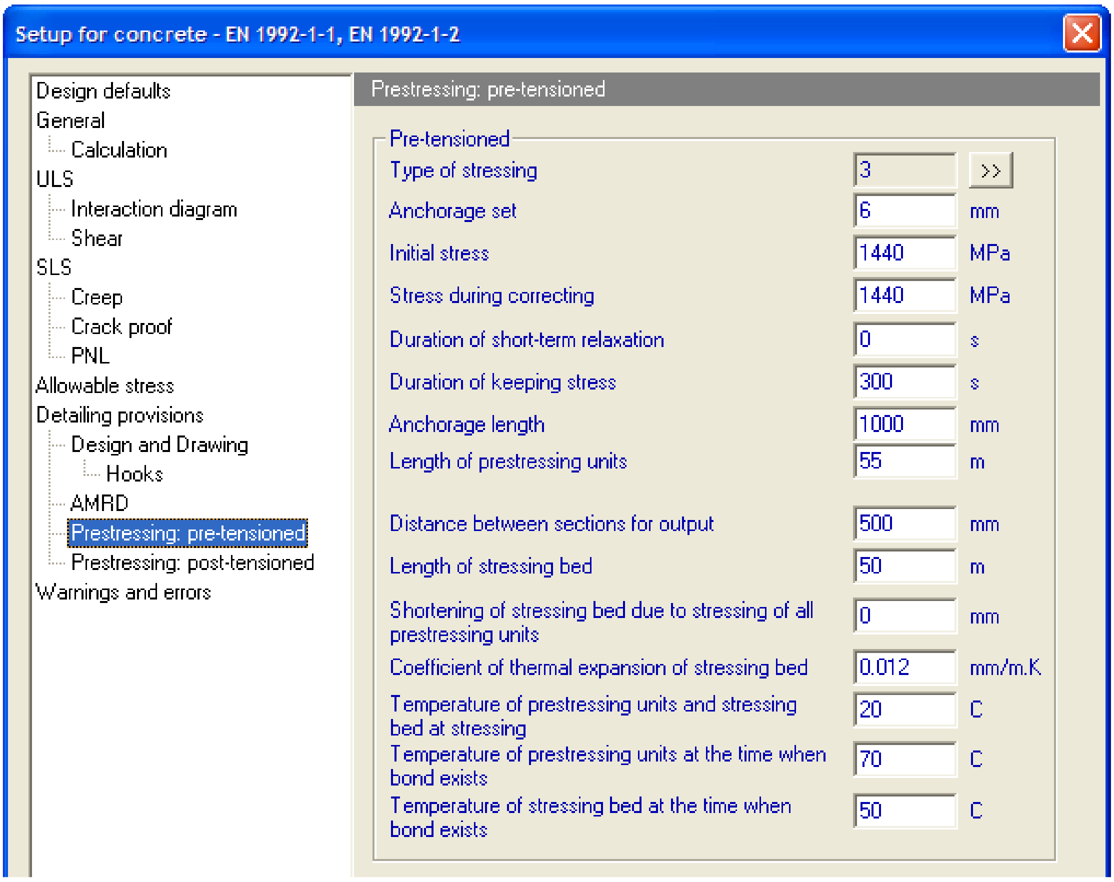
- Make a loadcase ‘Prestress’
-



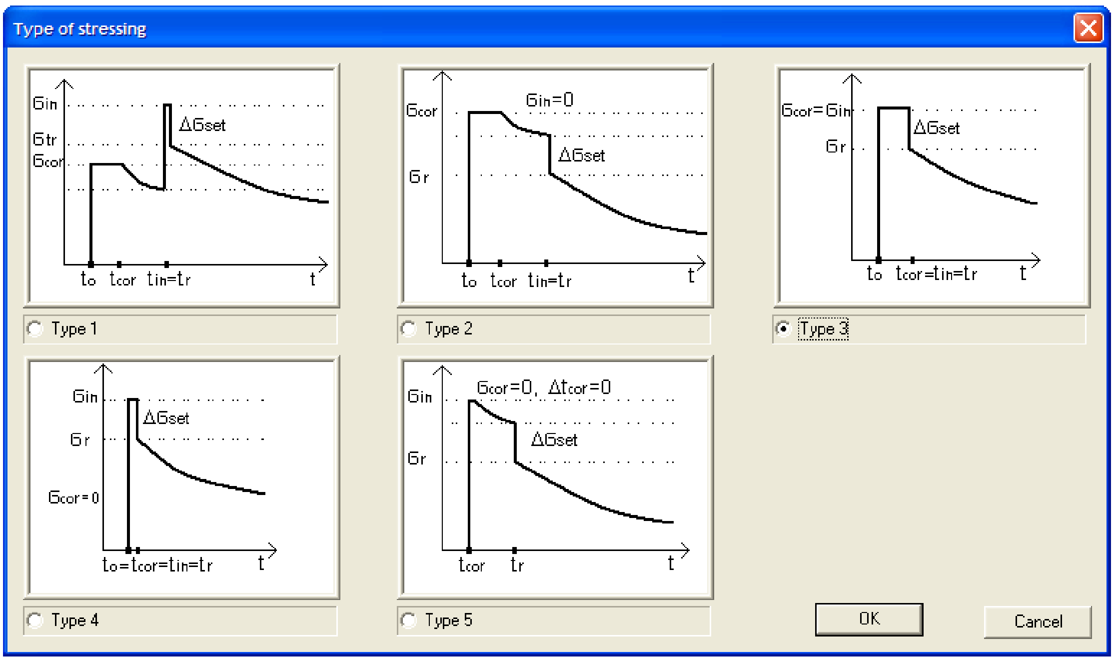
- Rekenen
- Bekijken resultaten: ux, ux, Rz, My, N, Kabelspanningen …
- Kabelspanningen: SAT, LED, LCS, Lmin, Lmax
- Geavanceerde voorspanning: wigzetting
- Controle voorspanrespons/voorspancapaciteit in beton menu

Properties of pre-tensioned tendons

The dialog Concrete > Setup can be used to define general properties and type of stressing of pre-tensioned tendons.



Type of stressing: See the figure:

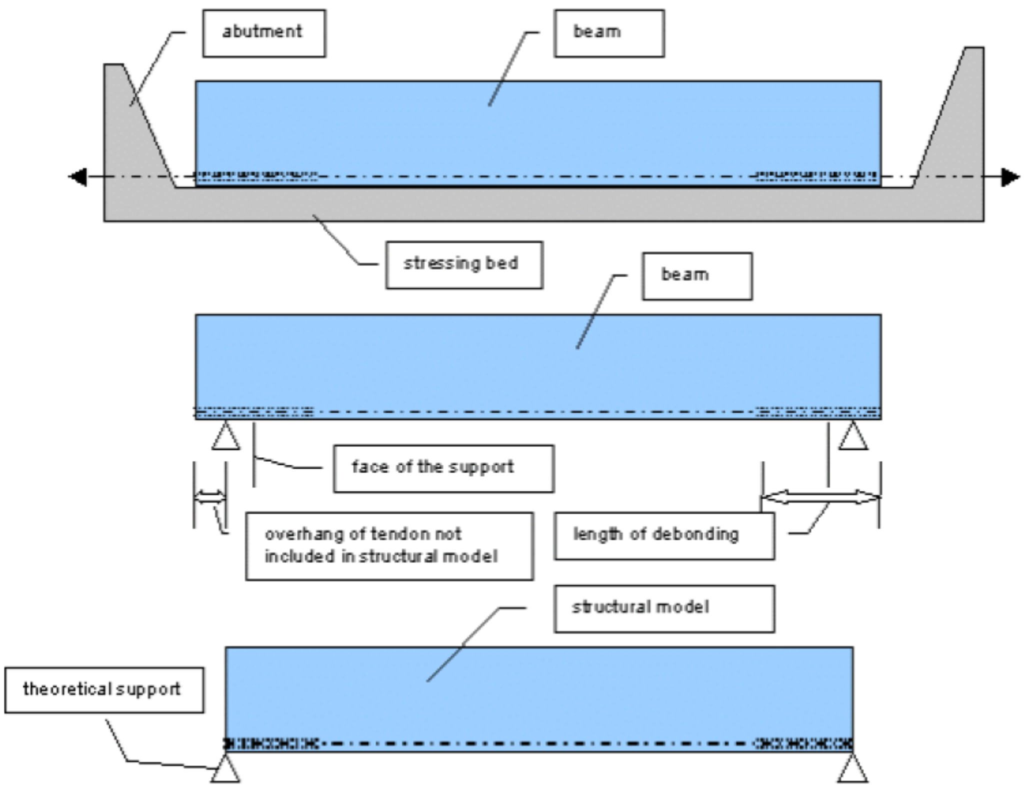


Anchorage set: Anchorage set at stressed end of tendon.
Initial stress: Specifies the initial stress at stressed end of tendon (before seating).
Stress during correcting: Defines the stress at stressed end of tendon. The amount of relaxation can be decreased by keeping the stress constant (so called correction of relaxation).
Duration of short-term relaxation: Specifies the time period between the end of correction of relaxation (if any) and time of anchoring.
Duration of keeping stress: The duration of keeping constant stress during correction of relaxation.
Anchorage length: The length of development of bond between the concrete and pre-tensioned tendon.
Length of prestressing units: Total length of wires or strands (between wedges); for pre-tensioned concrete it is equal to the length of stressing bed plus the length of abutments, see Fig. Pretensioned beam.
Distance between sections for output: Defines sections where results are given.
Length of stressing bed self-explanatory

Shortening of stressing bed due to stressing of all prestressing units: *self-explanatory*

Coefficient of thermal expansion of stressing bed: self-explanatory
Temperature of prestressing units and stressing bed at stressing: self-explanatory
Temperature of prestressing units at the time when bond exists: self-explanatory
Temperature of stressing bed at the time when bond exists: self-explanatory

Pre-tensioned beam



Types of Prestressing Units

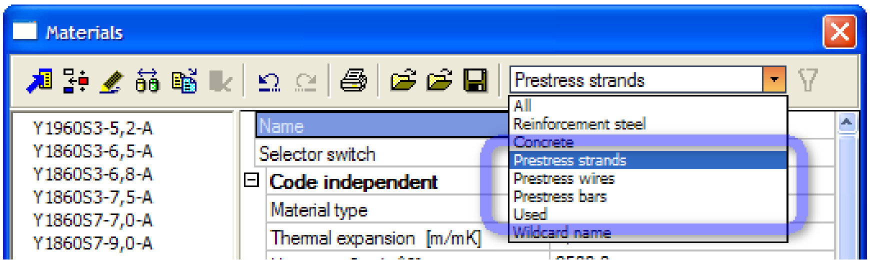
- ✓ Czech code CSN 73 12 01:
- ✓ Cold-drawn wires PD, PP, PH, PV, PN, PNV
- ✓ Strands LA, LB, LC, LD, LSA, LSB

Czech code CSN 73 62 07:

- ✓ Cold-drawn wires P
- ✓ Strands Lp, Ls

EC2:

- ✓ Cold-drawn wires w
- ✓ Indented wires w
- ✓ Strands s
- ✓ Plain round bars b
- ✓ Ribbed bars



Short-term losses

Short-term losses can be calculated in advance, before the solver is run:

- ✓ Anchorage set loss
- ✓ Losses due to deformation of stressing bed
- ✓ Losses due to steel relaxation
- ✓ Losses caused by the temperature differences in between prestressing steel and the stressing bed.

Besides of those losses, calculation of some other losses is included in the method for the structural analysis itself. Four codes are supported for the calculation of losses in Scia Engineer.

- ✓ EC2,
- ✓ NEN,
- ✓ CSN 73 6207,
- ✓ CSN 73 1201.

The losses are calculated according to assumptions given in these codes.

Anchorage set loss

There are two simplifications used in calculation of anchorage set loss:

1. EC2: we introduce effective cumulative angle $\gamma=\alpha+kx$ over a distance x, where x is horizontal coordinate, α is intended angle over a distance x, kx is unintended angle over a distance x.
2. CSN 73 1201 and CSN 73 6207: the exponential functions for friction calculations are approximated by first two members of power function.

Relaxation

The losses of prestressing caused by steel relaxation are introduced at three levels.

At first level - the correction of relaxation is calculated, namely the relaxation which appears during keeping the stress constant before anchoring. In fact, this is not a loss of prestressing. On the contrary the total relaxation considered in the structural analysis is decreased by this value. The relaxation at this level is applied for types 1, 2 and 3 of stressing sequence only (see Properties of pre-tensioned tendons > Type of stressing).

At second level – the short-term relaxation loss is calculated. The calculation is performed according to procedure of stressing and anchoring for types 1, 2 and 5 (see Properties of pre-tensioned tendons > Type of stressing).

At third level – the long-term relaxation loss is calculated for all five types of stressing sequence. This relaxation loss will occur after anchoring. Therefore it has an impact on the long-term behaviour of building structure and it should be applied as one of the loads in time-dependent analysis. In Scia Engineer implementation the strain equivalent to relaxation to be passed in long-term is applied in several time nodes following t_{tr} (see Properties of pre-tensioned tendons > Type of stressing).

The calculation of all of the relaxation losses mentioned above is based on the following principles. The differences between the procedures applied for different national codes are only in different definition of material characteristics. Firstly the final relaxation loss is calculated, which depends on the level of stress at given section related to the characteristic tensile strength. In the second step - the relative decrease of stress related to the final relaxation loss is calculated according to the duration of time interval when the stress is applied. The only simplification is that there is no effect of the changes of steel stress during the interval on the magnitude of relaxation in this time interval. For example - at third level – the losses of prestressing due to creep and shrinkage of concrete do not influence the amount of steel relaxation (insignificant).

The final relaxation loss is not defined in EC2 (only the relaxation up to 1000 hours). Therefore the estimate of the relaxation up to 30 years is used according to CEB FIP 1990 Model Code [2].

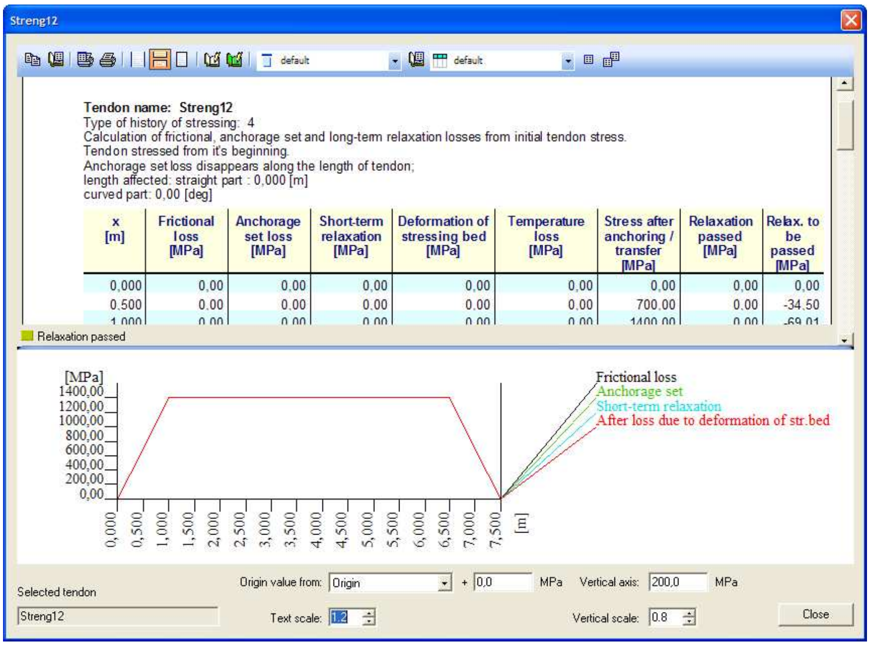
Running the losses

Having set all the input data described above, the losses can be calculated.

Procedure to calculate losses

1. Select the beam strand pattern for which the losses should be calculated.
2. The properties of the strand pattern are shown in the Property window.
3. Click button [Edit strand patterns].
4. The editing dialogue for the selected strand pattern is opened on the screen.
5. Select one strand which you are interested in.
6. Its properties are displayed in the bottom right corner of the dialogue.
7. Click action button Losses.
8. A preview window is displayed. The preview window is split into two parts. In first part some details of tendon parameters are displayed together with the table of results.

Using the toolbar at the top of the window, all the information can be exported to a file (HTML, TXT, PDF, RTF) or directly to printer. In the second part a diagram, the distribution of various losses along the length of the tendon is shown. It is possible to change the scale of the diagram or the text. And the pop-up menu (pressing right mouse button), offers some basic functions for the picture: zoom, print, copy to clipboard or save to an external file.



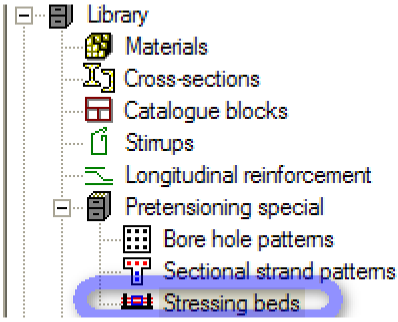
Stressing bed

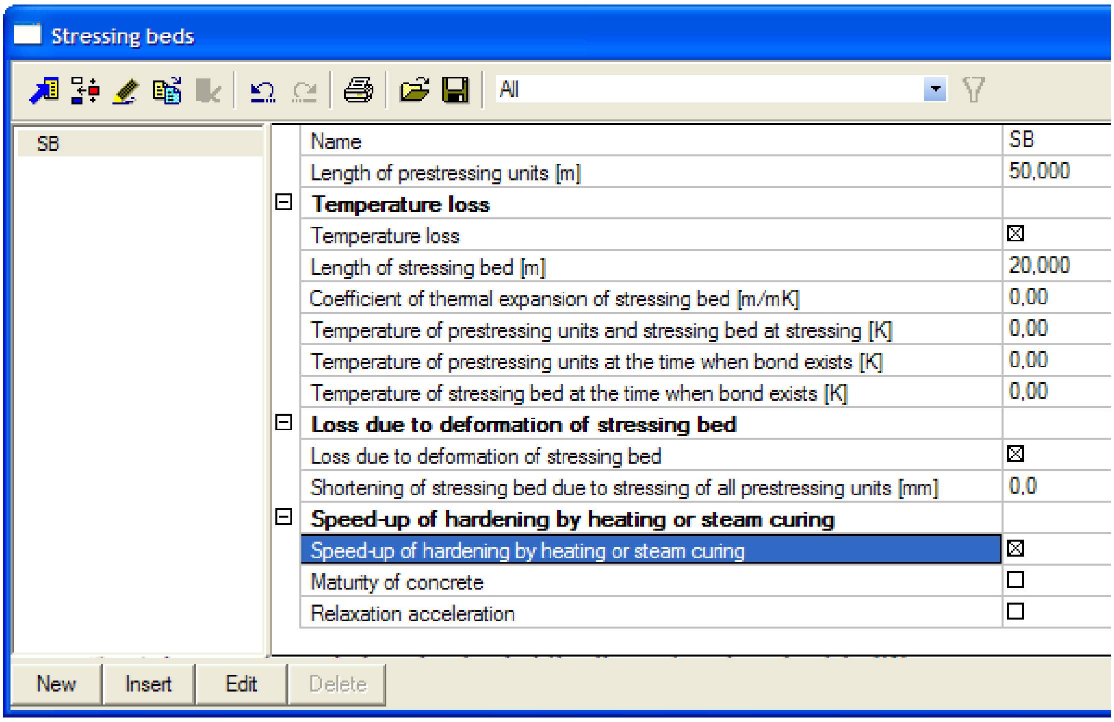
Stressing bed manager

The Stressing bed manager is one of the standard Scia Engineer database managers. It enables you to review, input, edit, delete, print, export or import individual stressing beds.

Procedure to open Stressing bed manager

1. Open service Library.
2. Open branch Pretensioning special.
3. Start function Stressing beds.
4. The Stressing beds manager opens on the screen.





Defining a new stressing bed

Procedure to define a new stressing bed
1. Open the Stressing bed manager.
2. Click button [New].

3. A new stressing bed is added to the manager and can be edited directly in the manager dialogue.
4. Define the parameters.
5. Close the manager.

Editing dialogue for stressing bed

Name self-explanatory
Length of prestressing units self-explanatory
Temperature loss It indicates whether the loss caused by the temperature differences between the prestressing steel and the stressing bed will be calculated.
Length of stressing bed self-explanatory
Coefficient of thermal expansion of stressing bed self-explanatory
Temperature of prestressing units and stressing bed at stressing self-explanatory
Temperature of prestressing units at the time when bond exists self-explanatory
Temperature of stressing bed at the time when bond exists self-explanatory
Loss due to deformation of stressing bed It indicates whether the loss caused by the shortening of stressing bed due to stressing of all prestressing units will be calculated. The stressing bed is an auxiliary structure between abutments where the formwork is placed. It might be of a limited stiffness relating to high forces applied during stressing.
Shortening of stressing bed due to stressing of all prestressing units self-explanatory

Editing the existing stressing bed

Procedure to edit the existing stressing bed
1. Open the Stressing bed manager.
2. Select the required stressing bed.
3. Click button [Edit].
4. The editing dialogue for a stressing bed opens on the screen.
5. Modify the required parameters.
6. Confirm with [OK].
7. Close the manager.

Bore hole pattern

Before casting a pre-tensioned member, a steel plate is installed at the end of the stressing bed. This plate contains holes that will define the position of strands in the end-section of the member. Not all the holes must be necessarily used (filled with a strand) at every member. Some holes may remain empty. That is why Scia Engineer distinguishes between bore hole pattern and sectional strand pattern. First, a bore hole pattern is defined and then it is used to define a sectional strand pattern. Both these "entities" define the location of strands at the end-section of the prestressed member.

Note: More information about the technology used for the production of pretensioned beams can be found in [3].

Bore hole patterns manager

The Bore hole pattern manager is one of the standard Scia Engineer database managers. It enables you to review, input, edit, delete, print, export or import individual bore hole patterns.

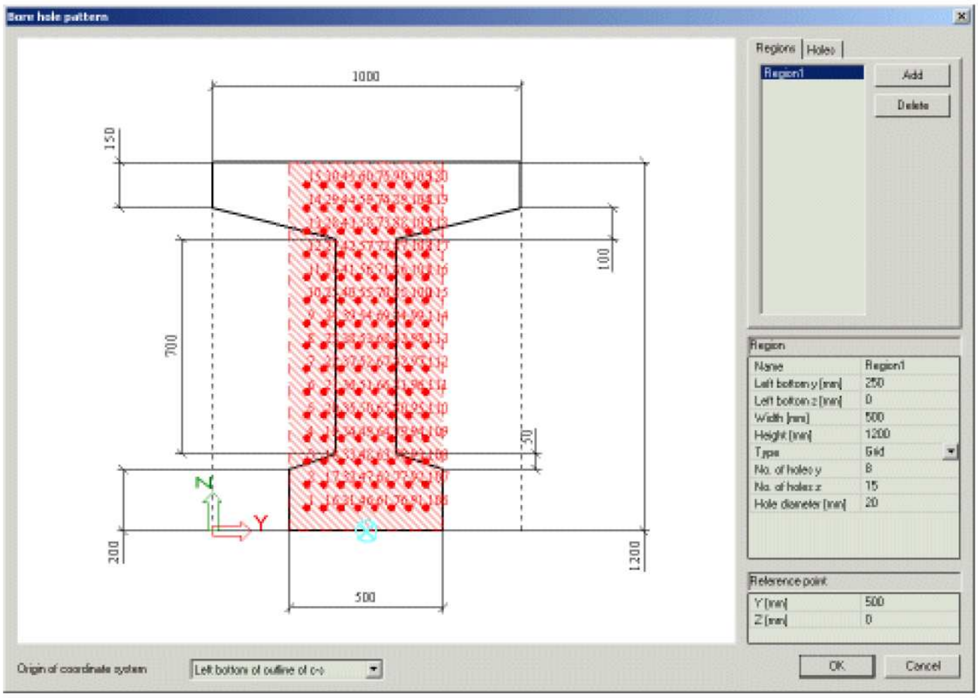
Procedure to open Bore hole pattern manager

1. Open service Library.
2. Open branch Pretensioning special.
3. Start function Bore hole patterns.
4. The Bore hole pattern manager opens on the screen. Defining a new bore hole pattern

Procedure to define a new bore hole pattern

1. Open the Bore hole pattern manager.
2. Click button [New].
3. The Cross-section database manager is opened on the screen.
4. Select the cross-section for which the new bore hole pattern should be defined.
5. Close the Cross-section database manager.
6. The editing dialogue for a bore hole pattern opens on the screen.
7. Define the bore hole pattern.
8. Confirm with [OK].

Editing dialogue for a bore hole pattern



The dialogue for the definition or editing of a bore hole pattern consists of the following parts:

- ✓ graphical window,
- ✓ definition of coordinate system,
- ✓ section of the input of regions and holes,
- ✓ definition of reference point.

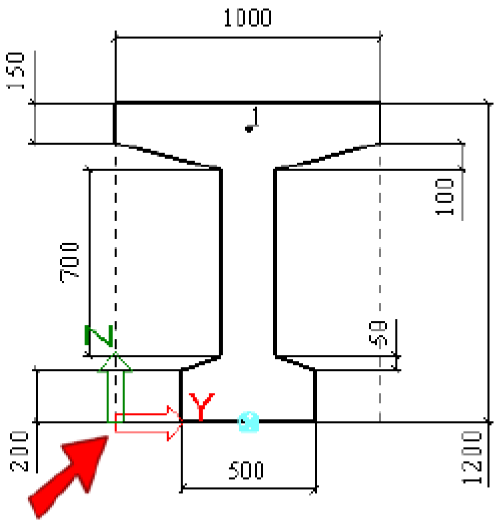
Graphical window

The graphical window displays the selected cross-section and the defined pattern of holes. It supports standard features of Scia Engineer graphical windows:

- ✓ pop-up menu with a set of zoom, print and export functions,
- ✓ [Ctrl] + [Shift] + right-click and drag to zoom in and out the drawing,
- ✓ [Shift] + right-click and drag to move the drawing.

Definition of coordinate system

You may select the origin of the input-coordinate-system. The selected system is marked in the graphical window. For example:



Input of regions and holes

The holes in the plate may be defined individually one-by-one or en-block in specified regions (even the region may contain just one hole).

Regions

A region is always rectangular and is defined by its position in the cross-section and its size. The holes are always regularly distributed across the region. You may specify either (i) the number of holes in the horizontal and vertical direction or (ii) the position of the first hole in each direction and the distance between the holes in that direction.

Name Specifies the name of the region.
Left bottom y Left bottom z Defines the coordinates of the bottom left the corner of region.
Width Specifies the width of the region.
Height Specifies the height of the region.
Type You may select the type of definition of the holes in the region:
Grid – you define the number of the holes in each direction
Increment - you define the position of the first hole in each direction and the distance between the holes in that direction
No. of holes y No. of holes z Inputs the holes for type Grid.
First hole y First hole z Inputs the holes for type Increment. Defines the position of the first hole.

***Dy Dz** Inputs the holes for type Increment. Specifies the distance between individual holes.*

Hole diameter Defines the hole diameter.

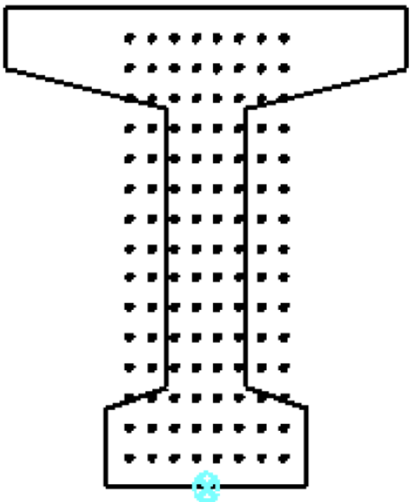
Holes

Holes may be input directly by their position in the cross-section.
ID (informative) Shows the number of the hole. The number are assigned automatically by the program.
Name Specifies the name of the hole.
Y Z The coordinates defining the position of the hole.
Hole diameter Defines the hole diameter.

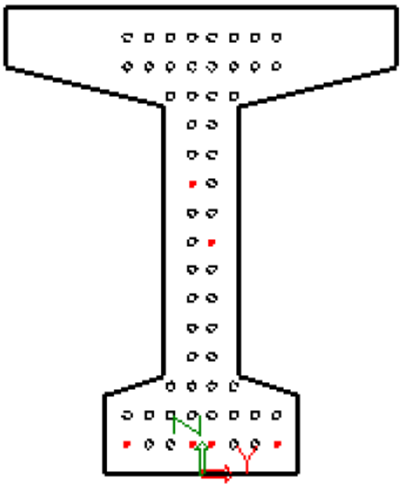
Holes can be copied. In that case you specify if you make just a single copy or a multiple copy, input the distance between the copies and, if required, also the number of copies. In order to copy a hole, simply use button [Copy] next to the list of holes and fill in the copy-dialogue.

Note: When you define the holes in regions, it may happen that some holes "fall" out of the cross-section (especially if the cross-section is not rectrangular). These outside holes do not have to be specially treated as they are automatically filtered when you define the sectional strand pattern. That means that strands can be put only into real and proper holes.

Defined holes in the bore hole pattern:



Available holes in the sectional strand pattern:

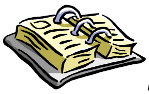


Reference point

The reference point can be used to position the bore hole pattern in the cross-section of the beam when the sectional strand pattern is created (you are asked to position the bore hole pattern on the cross-section). Normally, you may accept the default setting. Only if you decide on a special configuration of the bore hole pattern, it may be convenient to "play" with the reference point and use one bore hole pattern for different final location of strands in the beam.

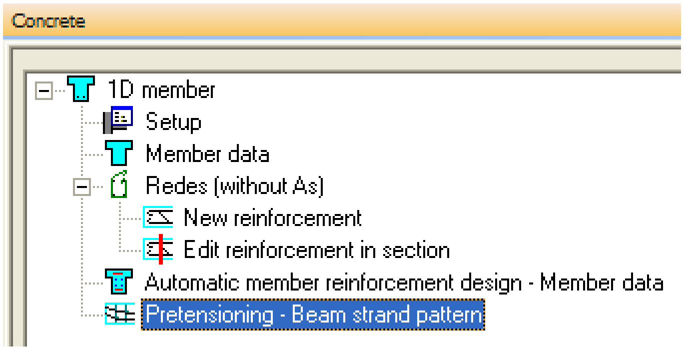
Editing the existing bore hole pattern

Procedure to edit the existing bore hole pattern
1. Open the Bore hole pattern manager.
2. Select the required bore hole pattern.
3. Click button [Edit].
4. The editing dialogue for a bore hole pattern opens on the screen.
5. Modify what necessary.
6. Confirm with [OK].

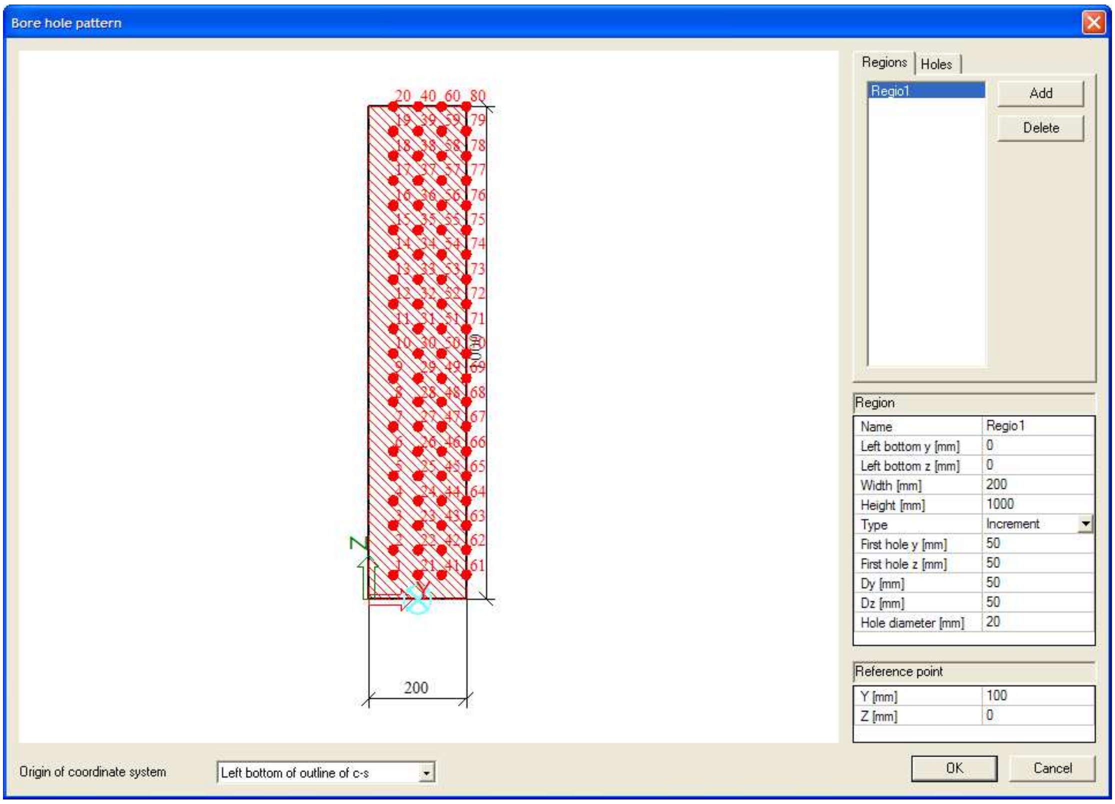


Let's create a borehole pattern in our example

In the concrete branch you can choose for:



Now you can define following borehole pattern:



Sectional strand pattern

Sectional strand pattern defines the position of strands at the end-section of a prestressed member. First, a bore hole pattern must be created and later the sectional strand pattern may be defined for it.

Note: More information about the technology used for the production of pre-tensioned beams can be found in [3].

Sectional strand pattern manager

The Sectional strand pattern manager is one of the standard Scia Engineer database managers. It enables you to review, input, edit, delete, print, export or import individual sectional strand patterns.

Procedure to open Sectional strand pattern manager

1. Open service Library.
2. Open branch Pretensioning special.
3. Start function Sectional strand pattern.
4. The Sectional strand pattern manager opens on the screen.

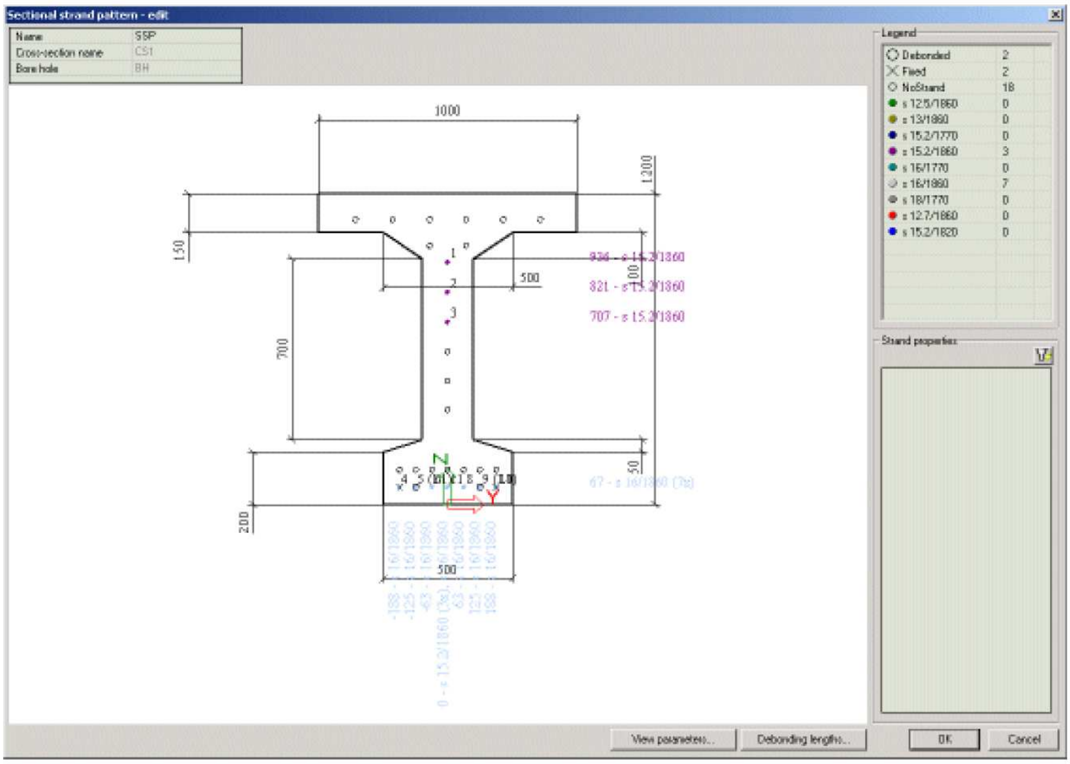
Defining a new sectional strand pattern

Procedure to define a new sectional strand pattern

1. Open the Sectional strand pattern manager.
2. Click button [New].
3. The Cross-section database manager is opened on the screen.
4. Select the cross-section for which a new strand pattern should be defined.
5. Close the Cross-section database manager.
6. Select the bore hole pattern for which a new sectional strand pattern should be defined.
7. Close the Bore hole pattern manager.
8. A small positioning dialogue is opened on the screen.
9. Position the bore hole pattern in the cross-section (see the note below).
10. The editing dialogue for a sectional strand pattern opens on the screen.
11. Define the new sectional strand pattern.
12. Confirm with [OK].

Note: This positioning of the bore hole pattern on the cross-section may become important later when you decide to change the height of the cross-section. The position of strands is related to the reference point and is not affected by the change of the dimension. You may choose the reference point which most suits your needs.

Editing dialogue for sectional strand pattern



The dialogue for the definition or editing of a sectional strand pattern consists of the following parts:

- ✓ graphical window,
- ✓ info-table,
- ✓ legend,
- ✓ strand properties,
- ✓ view parameters button,
- ✓ debonding length button,
- ✓ control buttons.

Graphical window

The graphical window displays the selected cross-section and the defined pattern of holes. It supports standard features of Scia Engineer graphical windows:

- ✓ pop-up menu with a set of zoom, print and export functions,
- ✓ [Ctrl] + [Shift] + right-click and drag to zoom in and out the drawing,
- ✓ [Shift] + right-click and drag to move the drawing.

Info-table

This table is located in the top left corner of the dialogue.
Name Specifies the name of the pattern.
Cross-section name (informative) Informs about the cross-section used in the sectional pattern.
Bore hole (informative) Informs about the bore hole used in the sectional pattern.

Legend

The legend has two roles:

1. (informative) it explains the symbols used in the graphical window (each type of strand uses a special graphical mark or colour),
2. (active) it is used for the input of individual strands into the bore holes.

Procedure to input a new strand

1. In the Legend select the required strand material and diameter.
2. In the graphical window click the holes where you want to have a strand.
3. If you need to combine more materials/diameters, simply repeat steps 1 and 2.

Procedure to delete the existing strand

1. In the Legend select item No Strand.
2. In the graphical window click the strand you want to remove.

Procedure to define debonded or fixed strand

1. In the Legend select the item Debonded or Fixed.
2. In the graphical window click the appropriate strand(s).

Alternatively, you may define these two properties in the property window of the required strand (see below).

Stand properties

Whenever a defined strand is selected in the graphical window, its properties are shown in the Properties window.
Name Specifies the name of the strand.
ID Specifies the ID of the strand.
Group (informative) The number of strand group.
Material Selects the material and diameter.
Fixed Specifies if the strand is fixed. The fixed strand has fixed position in the section along the whole length of the beam. It is straight.
Debonding length Defines whether the strand is debonded at its end and if so, over which distance. Also the debonded strand has fixed position in the section along the whole length of the beam. It is straight.
Stressing sequence Defines the sequence in which the strands are stressed.
Type of stressing See chapter Properties of pre-tensioned tendons.
Stress during correcting Stress at stressed end of tendon; the amount of relaxation can be decreased by keeping the stress constant (so called correction of relaxation)
Duration of keeping stress Duration of keeping constant stress during correction of relaxation.
Initial stress Initial stress at stressed end of tendon (before seating)
Anchorage set Defines the slip at the stressed end of the strand.
Anchorage length The length of development of bond between the concrete and pre-tensioned tendon
Distance between sections for output Specifies the distance for output.
Position Y, Z (informative) Shows the position of the strand.

Note:

In the property window you may use selection-by-property. Select one strand, in the property window select the property you are interested in, click the "funnel" icon in the top right corner of the property window and the program selects all the strands that have the same property. The selected strands are highlighted in the graphical window. This feature can be used for both active and informative properties.

Note:

It is possible to make a multiple selection of strands in the graphical window. Press and hold key [Ctrl] and click the left mouse button. When the multi-selection has been made, you can change a required property to all the selected items at once.

View parameters

This button opens a dialogue with view parameters. Their meaning is more or less self-explanatory.

Debonding lengths

A set of various debonding lengths can be defined here. One debonding length can be then assigned to a strand in the property window.

Editing the existing sectional strand pattern

1. Open the Sectional strand pattern manager.
2. Select the required strand pattern.
3. Click button [Edit].
4. The editing dialogue for a sectional strand pattern opens on the screen.
5. Change the required parameters of the selected sectional strand pattern.
6. Confirm with [OK].
7. Close the manager.

Beam strand pattern

When a sectional strand pattern has been defined (i.e. also a bore hole pattern must have been created), it is possible to define the shape of prestressing tendons along the member. In Scia Engineer this shape is defined by means of a beam strand pattern. It is in fact a set of strand patterns defined for individual sections of the beam. Both the beam and the prestressing reinforcement are assumed symmetrical, so just a half of the beam must be defined. It depends on the shape of the reinforcement how many sections must be created for each beam strand pattern. If the strands do not change their position in the section, one section (the sectional strand pattern) is sufficient.

Defining a new beam strand pattern

Procedure to define a new beams strand pattern from bore hole pattern

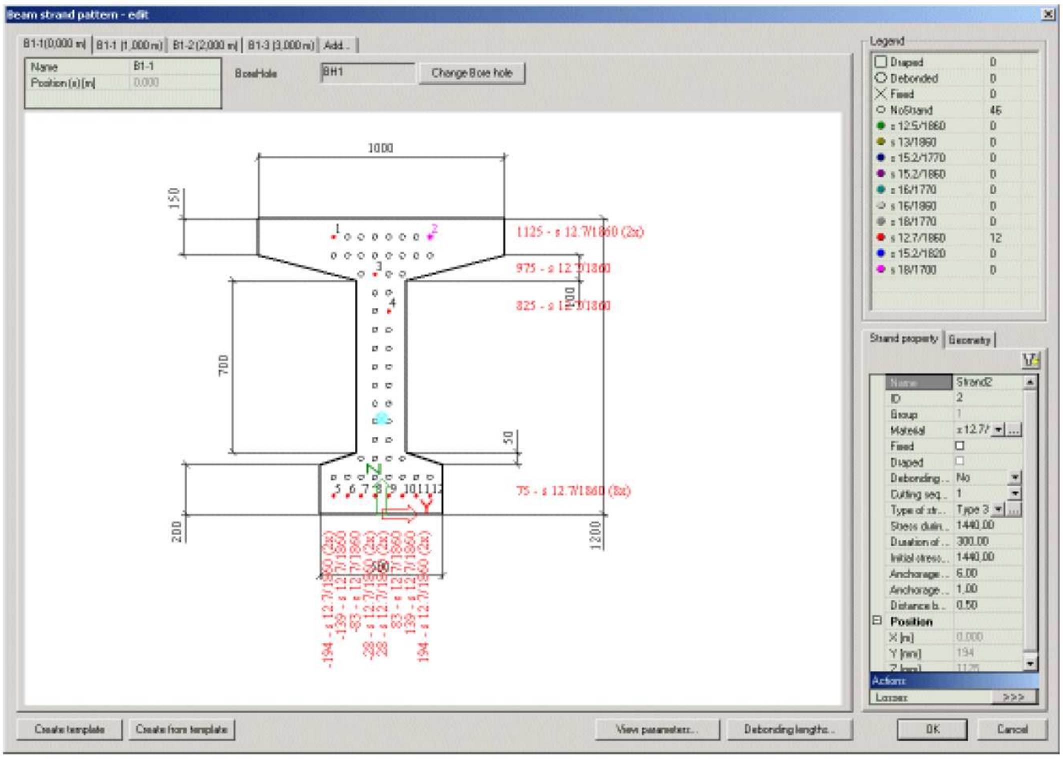
1. Open service Concrete.
2. Start (double-click) function Pretensioning – beam strand pattern.
3. Select the beam where the beam strand pattern is to be inserted.
4. The Select template dialogue opens on the screen.
5. Select item Bore hole.
6. In case of arbitrary beam the Cross-section manager opens on the screen and you must select the required cross-section.
7. The Bore hole pattern manager opens on the screen.
8. Select the required bore hole pattern.
9. Position the bore hole pattern in the cross-section.
10. The editing dialogue for the beam strand pattern opens on the screen.

11. Define the required sections of the beam strand pattern. Also the selection (or input) of a prestressing load case may be required during this. The load case is required for storage of results.
12. Confirm with [OK].

Procedure to define a new beam strand pattern from sectional strand pattern

1. Open service Concrete.
2. Start (double-click) function Pretensioning – beam strand pattern.
3. Select the beam where the beam strand pattern is to be inserted.
4. The Select template dialogue opens on the screen.
5. Select item Sectional strand pattern.
6. In case of arbitrary beam the Cross-section manager opens on the screen and you must select the required cross-section.
7. The Sectional strand pattern manager opens on the screen.
8. Select the required bore hole pattern.
9. Position the bore hole pattern in the cross-section.
10. The selection (or input) of a pre-stressing load case may be required during this. The load case is
required for storage of results.
11. The editing dialogue for the beam strand pattern opens on the screen.
12. Confirm with [OK].

Editing dialogue for the beam strand pattern



The dialogue for the definition or editing of a beam strand pattern consists of the following parts:

- ✓ tabs for individual sections with graphical window,
- ✓ tab for input of a new section,
- ✓ info-table,
- ✓ bore-hole information and button for its change,
- ✓ legend,
- ✓ strand properties,
- ✓ action button for calculation of losses in selected strand,

- ✓ strand geometry,
- ✓ view parameters button,
- ✓ debonding length button,
- ✓ template buttons,
- ✓ control buttons.

Graphical window

The graphical window displays the selected cross-section and the defined pattern of holes. It supports standard features of Scia Engineer graphical windows:

- ✓ pop-up menu with a set of zoom, print and export functions,
- ✓ [Ctrl] + [Shift] + right-click and drag to zoom in and out the drawing,
- ✓ [Shift] + right-click and drag to move the drawing.

Info-table

This table is located in the top left corner of the dialogue.
Name Specifies the name of the pattern.
Position (informative) Informs about the position of the section on the beam.

Bore hole information

This part of the dialogue shows the bore hole selected for the current strand pattern. The button [Change bore hole] can be used to change the pattern.

Note:

If the bore hole pattern is changed, all already defined strands are deleted.

Legend

The legend has two roles:

- ✓ (informative) it explains the symbols used in the graphical window (each type of strand uses a special graphical mark or colour),
- ✓ (active) it is used for the input of individual strands into the bore holes.

The procedures for input and removal of strands and for adjustment of special properties are described in chapter Defining a new sectional strand pattern.

Stand properties

Whenever a defined strand is selected in the graphical window, its properties are shown in the Properties window. The meaning of the properties is described in chapter Defining a new sectional strand pattern.

Calculation of losses in selected strand

Using the action button in the bottom part of the property dialogue you may calculate losses for the selected strand. The button opens a dialogue with a table and diagram of calculated short-term losses.
Note: See also chapter Short-term losses.

Strand geometry

Yp; Zp coordinate of the centre of gravity of entire strand pattern
Yp, deb; Zp, deb coordinate of the centre of gravity of all debonded strands
Yp, drap; Zp, drap coordinate of the centre of gravity of all draped strands
n total amount of strands in strand pattern
Ap total area of entire strand pattern
Ac total cross-sectional concrete area without the area of strands
Ap, deb total area of all debonded strands
Ap, drap total area of all draped strands
Y; Z coordinate of the centre of gravity of the combined crosssection - concrete plus pre-stressing steel
Iz; Iy moment of inertia for z/y-axis of combined cross-section. (zaxis being strong axis)
Iz,c; Iy, c moment of inertia for z/y-axis of combined cross-section without the strands (z-axis being strong axis)
Wy, top; Wy, bottom section modulus for z/y-axis of combined cross-section at top of section

View parameters

This button opens a dialogue with view parameters. Their meaning is more or less self-explanatory.

Debonding lengths

A set of various debonding lengths can be defined here. One debonding length can be then assigned to a strand in the property window.

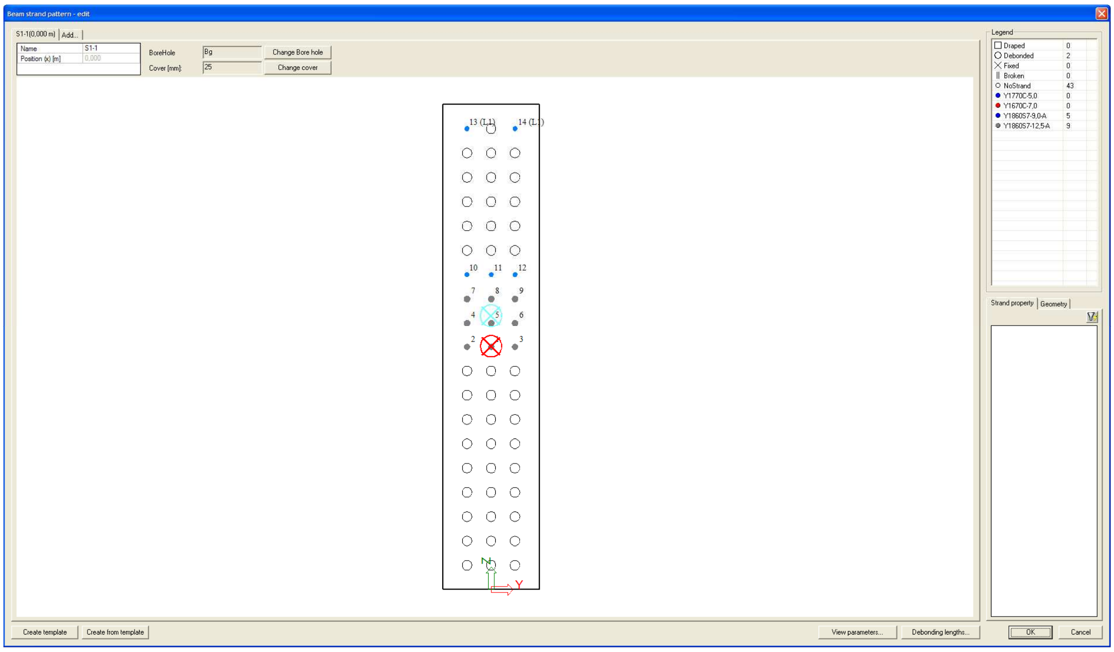
Template buttons

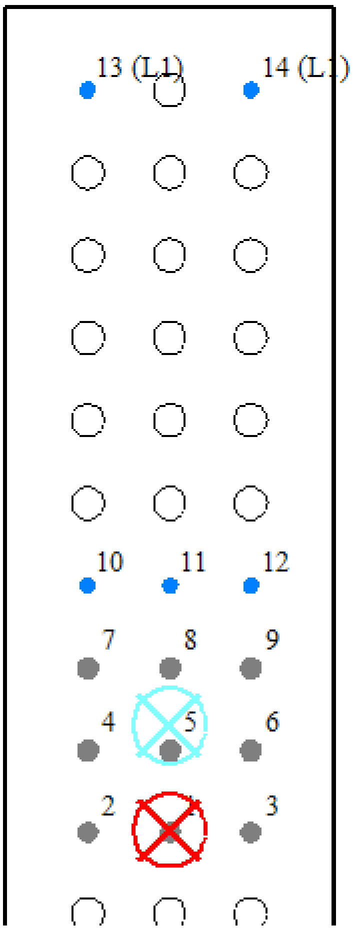
[Create template] A new template is saved to the project.
[Create from template] The strand pattern is loaded from an existing template.

Editing the existing beam strand pattern

Procedure to edit the existing beams strand pattern from bore hole pattern
1. Select the beam strand pattern for which the losses should be calculated.
2. The properties of the strand pattern are shown in the Property window.
3. Click button [Edit strand patterns].
4. The editing dialogue for the selected strand pattern is opened on the screen.
5. Make necessary changes.
6. Confirm with [OK].

Let’s create a beam strand pattern in our example:





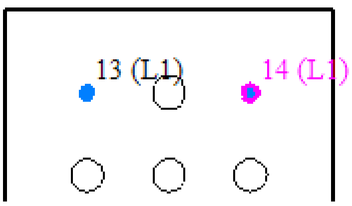
Draped	0
Debonded	2
Fixed	0
Broken	0
NoStrand	43
Y1770C-5,0	0
Y1670C-7,0	0
Y1860S7-9,0-A	5
Y1860S7-12,5-A	9

All strands have the same initial stress:

Name	Streng6
ID	6
Group	1
Material	Y1860S7-
Fixed	☐
Draped	☐
Debonding le...	No
Stressing seq...	1
Type of stres...	Type 4
Initial stress [...	1400,00
Anchorage s...	0,00
Anchorage le...	1,00
Distance bet...	0,500

The two upper strands have a debonding length of 1m:

Debonding lengths...



Name	Streng14
ID	14
Group	3
Material	Y1860S7-
Fixed	☐
Draped	☐
Debonding le...	L1 (1,000 m)
Stressing seq...	1
Type of stres...	Type 4
Initial stress [...	1400,00
Anchorage s...	0,00
Anchorage le...	1,00
Distance bet...	0,500
Position	
X [m]	0,000
Y [mm]	50
Z [mm]	950

Results of prestressing

Standard results

When the Prestress Analysis has been performed, you may review all the standard results as in case of a normal static linear calculation: deformations, internal forces, stresses, reactions.

Tendon stresses

Result diagrams in graphical window

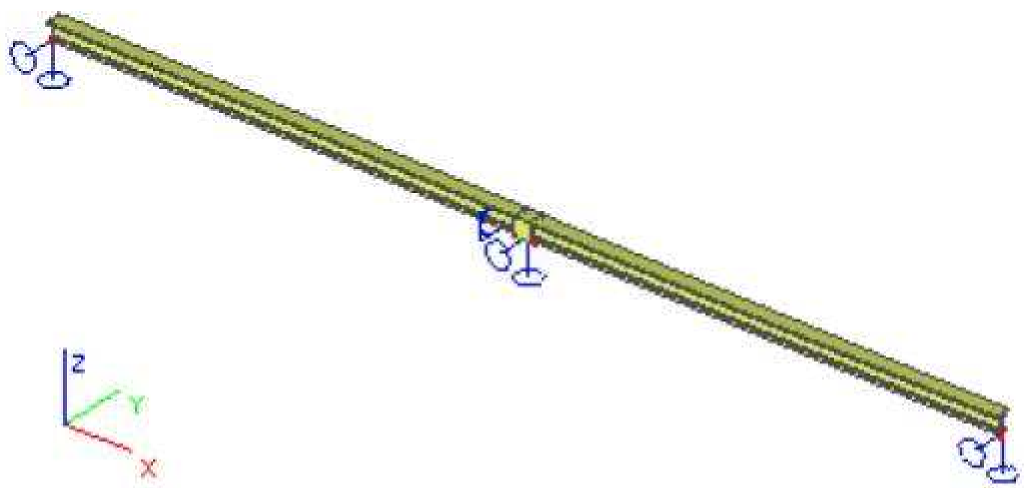
Procedure to display tendon stresses

1. Open service Results.
2. Start function Tendon stresses.
3. Select the load for the display.
4. Adjust the style of result diagrams.
5. Select the beam strand patterns for which the results should be drawn.
6. Use filter to specify the tendons to be displayed (see below).
7. Press button [Refresh] to regenerate the drawing.
8. When ready, close service Results.

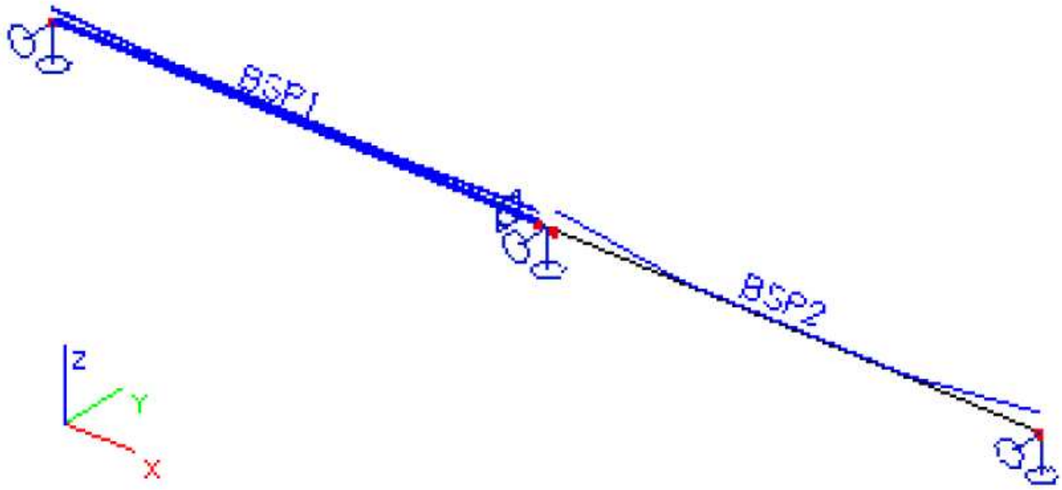
Filtering the results

Capabilities of filtering will be explained on a simple example.

Imagine a two span continuous beam built in two construction stages: left span in the first stage (assigned load case 1), second span in the second stage (assigned load case 2).



Both spans are prestressed and have a beam strand pattern defined. The left span contains 5 strands, the second one only one.

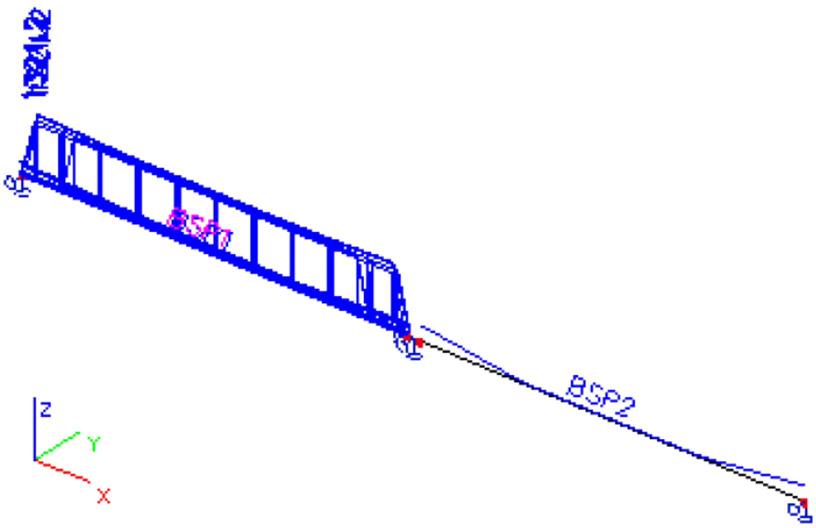


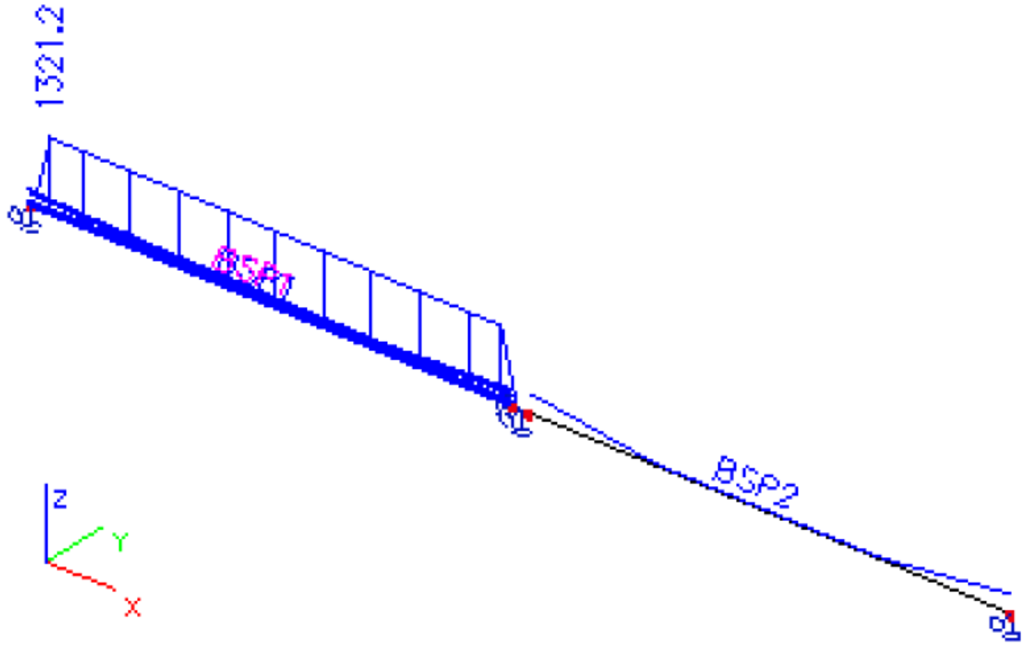
Open service **Results** and start function **Tendon stresses**.
Set **Selection** to **Standard** and select no beam strand pattern.

Open combo box **Tendons**, it contains just one option: All by selection.
Now select the strand pattern in the left span and open the same combo box again. It offers **All by selection** plus the five strands from the first beam.
Clear the selection, select the right hand span and look into the combo box. It offers All by selection plus the strand from the second beam.

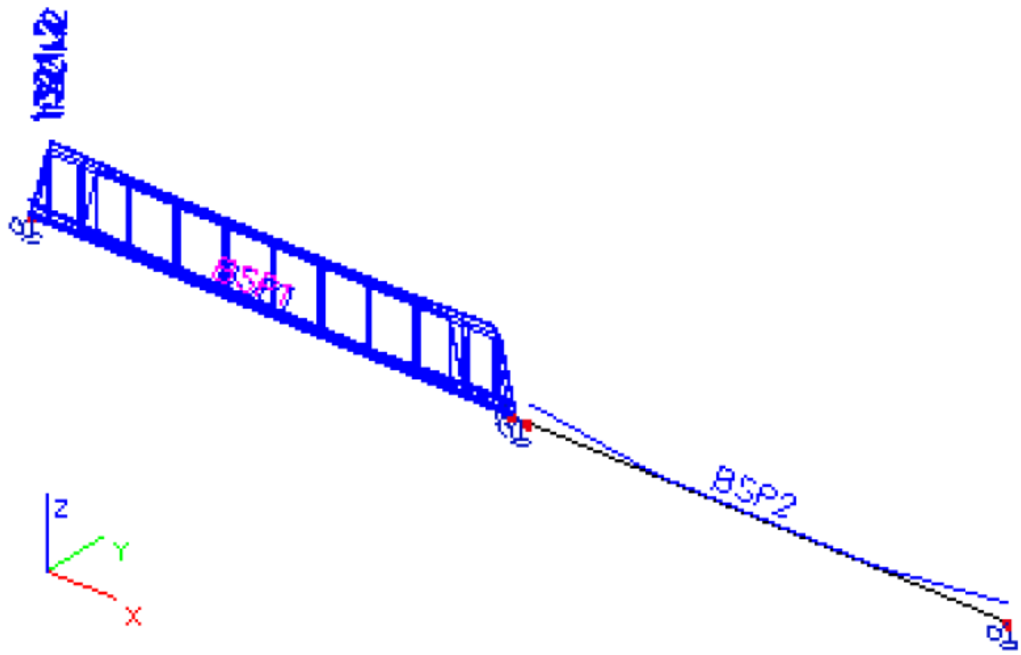
Select both strand patters and open the combo box once again. It offers All by selection, plus the five strands from the first beam, plus the strand from the second beam.

This way you may select just one tendon and display the results on it. This option is convenient especially if there are multiple strands in one beam. Compare the pictures below. The first one displays results on all strands in the beam, the second one on just one strand.

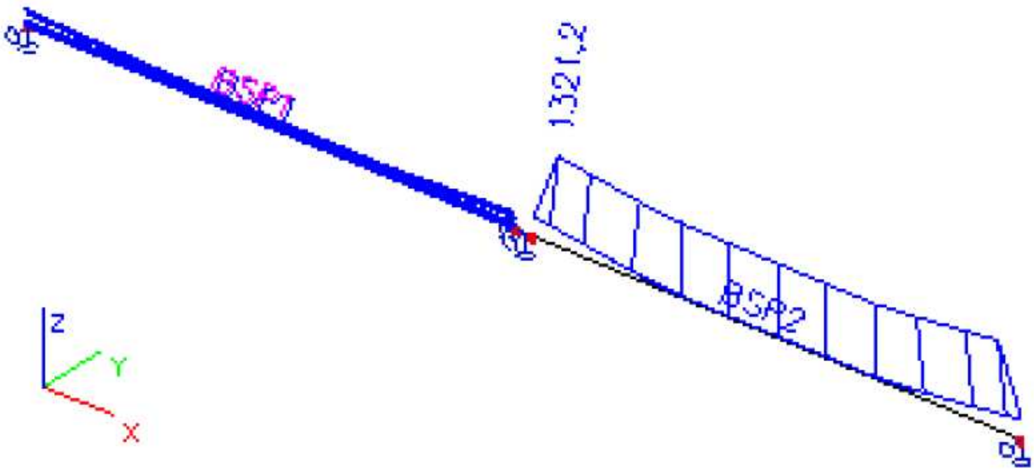




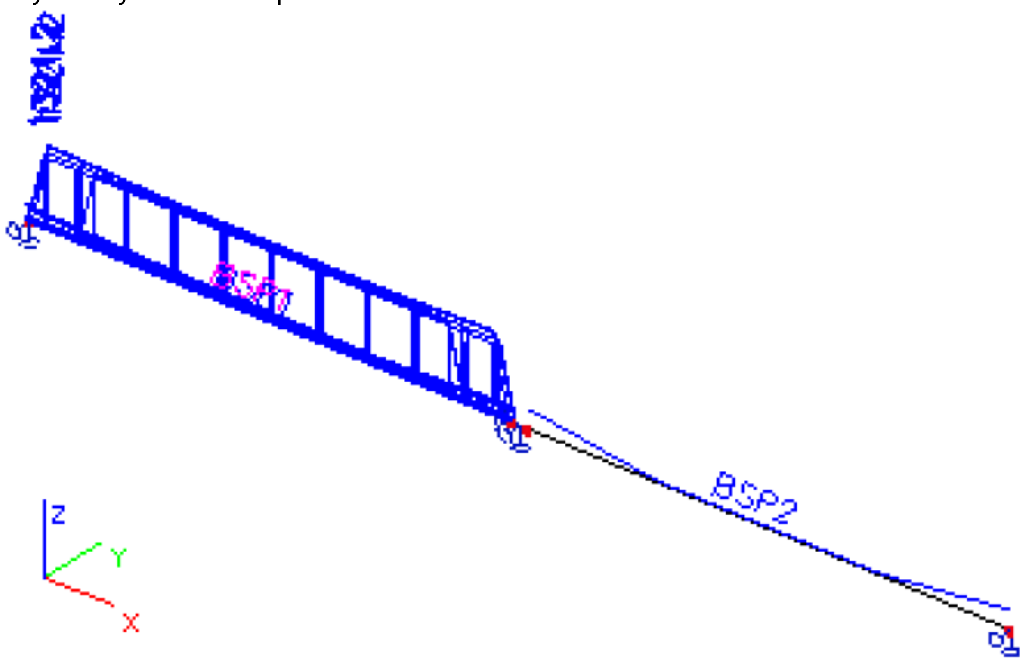
Now, set **Selection** to **All**. Select **All by selection** in combo box **Tendons**.
Set **Load** to **Load cases** and select **LC1**. This load case is assigned to the first construction stage when only the left span exists. When you press **[Refresh]**, tendon stresses are displayed only on the left span.



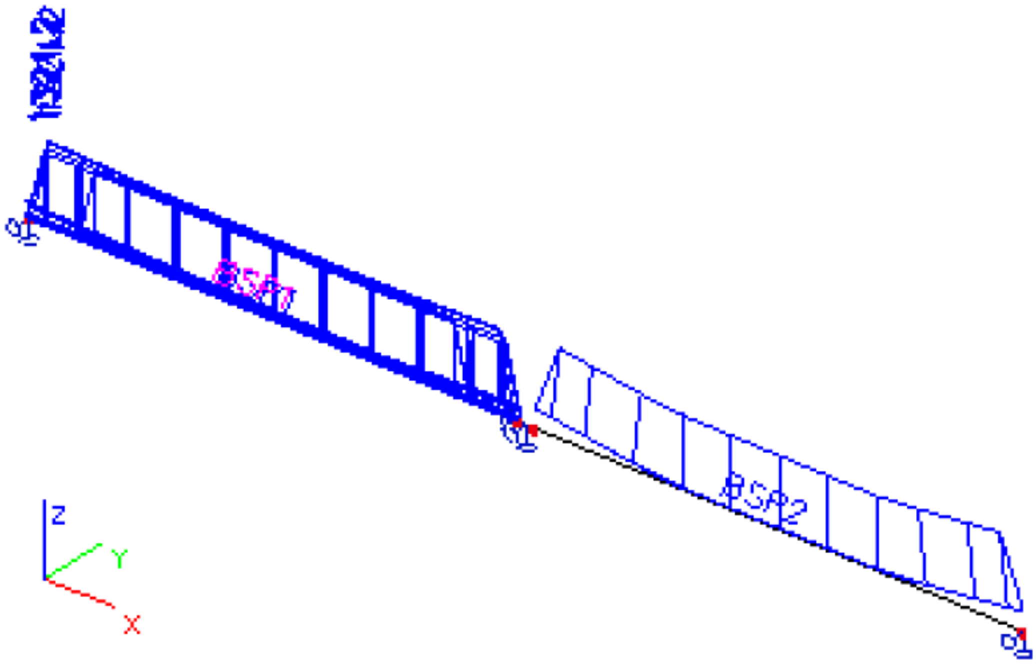
Similarly, if you select **LC2**, only tendon stresses in the right span are shown.



And finally,set **Selection** to **All** and keep **All by selection** in combo box Tendons.
Set **Load** to **Classes** and select **Class 1**. This class corresponds to the first construction stage when only the first span exists. Press **[Refresh]**, tendon stresses are displayed only on the left span.



Select **Class 2** that corresponds to the second construction stage, when both spans exist. Press **[Refresh]** and tendon stresses are displayed on both spans.



Detailed results
It is possible to display detailed results for a single beam strand pattern
Procedure to display detailed results
1. Open service Results.
2. Start function Tendon stresses.
3. Click action button [Detailed].
4. Select one beam strand pattern.
5. A window with detailed results opens on the screen.

Preview in the Preview window

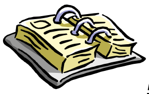
The results may be reviewed in tabular form in the preview window.

Procedure to view the preview

1. Open service Results.
2. Start function Tendon stresses.
3. Select the load for the display.
4. Adjust the style of result diagrams.
5. Select the beam strand patterns for which the results should be drawn.
6. Use filter to specify the tendons to be displayed (see below).
7. Press button [Preview] to view the table of results.

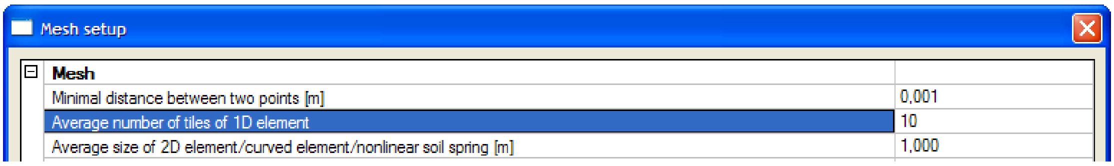
Explanation of abbreviations

SAT Stress after transfer.
LED Loss due to sequential prestressing + loss caused by the elastic deformation of concrete.
LCS Loss due to creep and shrinkage of concrete + loss due to long-term steel relaxation.
Lmin Loss (change of) prestressing caused by life load (min).
Lmax Loss (change of) prestressing caused by life load (max).
MinStress Minimal stress in phase.
MaxStress Maximal stress in phase.



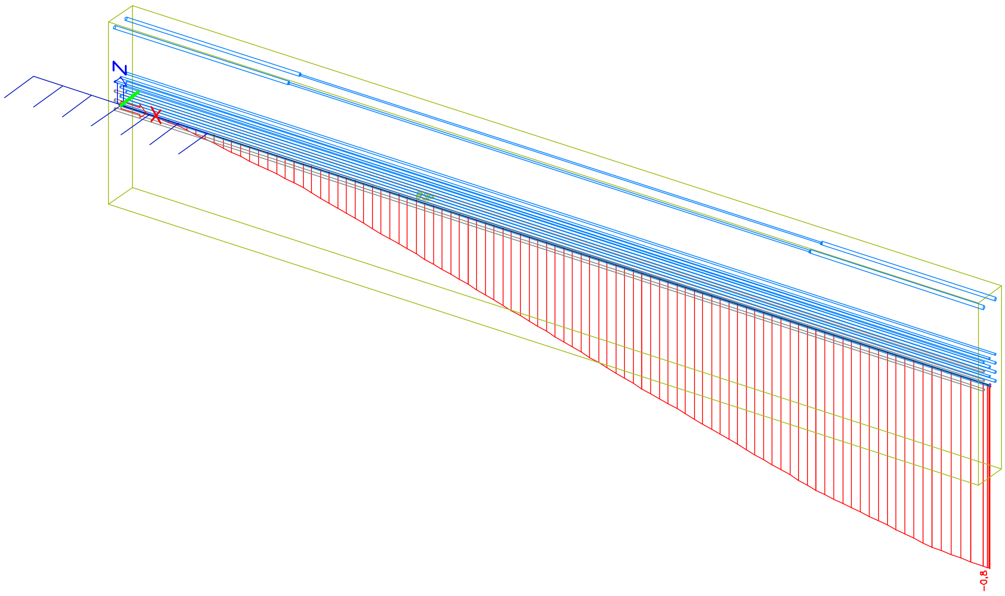
Let's calculate our example now:

Before launching the linear calculation it is best to always refine the 1D mesh:



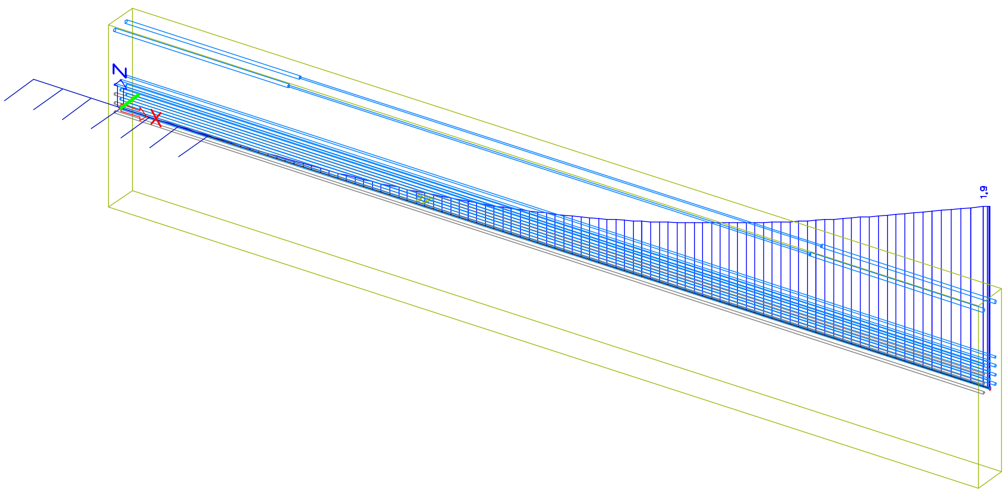
After the calculation we can review the results. Let's look first for the deformations:

ux:

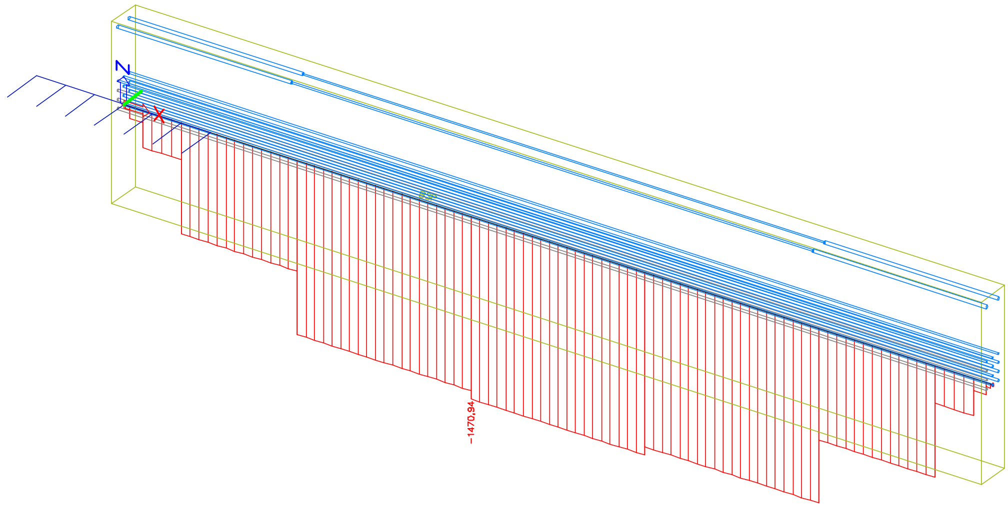


Because of the strands the beam will shorten.

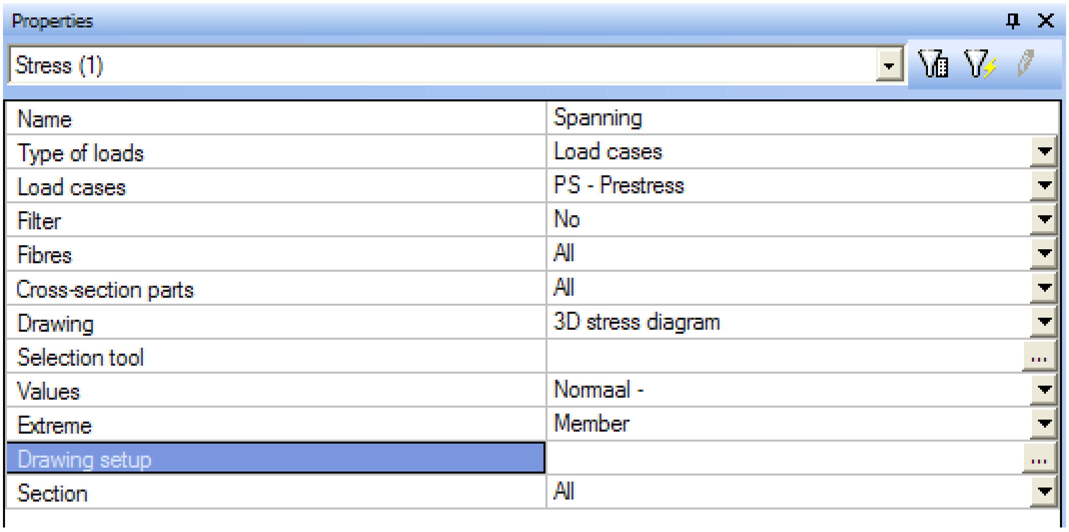
In the z direction we can see the expected upper deformation of the beam:

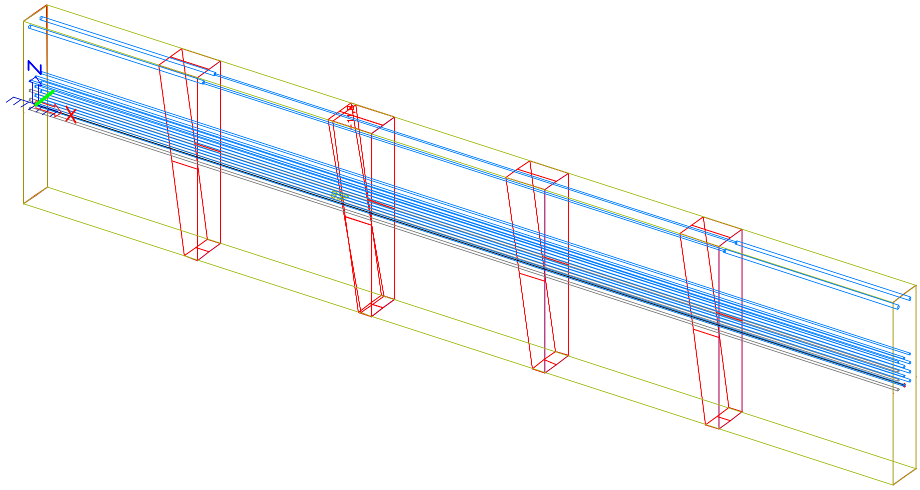


Let's now look into the internal forces.
The normal force N gives us:

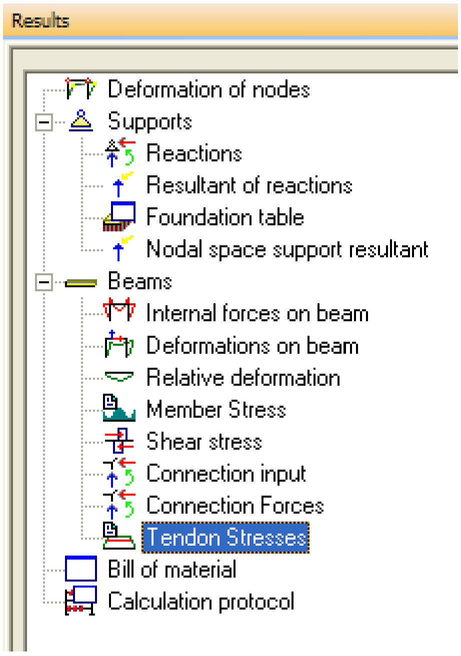


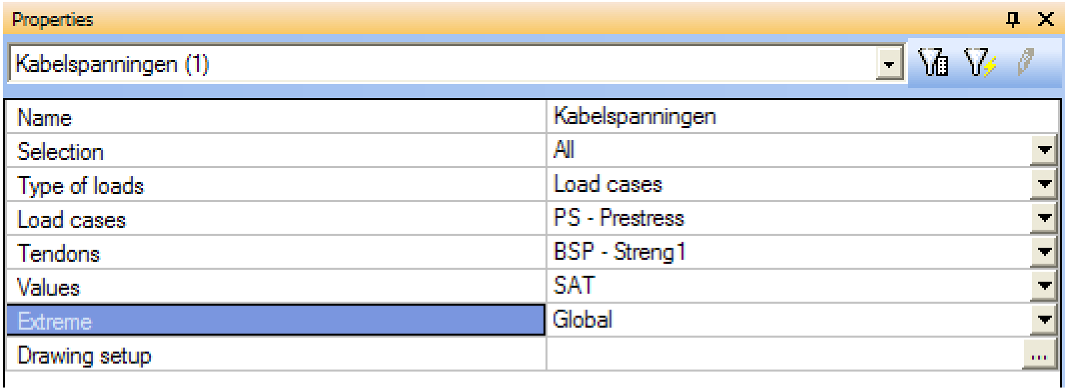
The 3D stresses are also interesting:

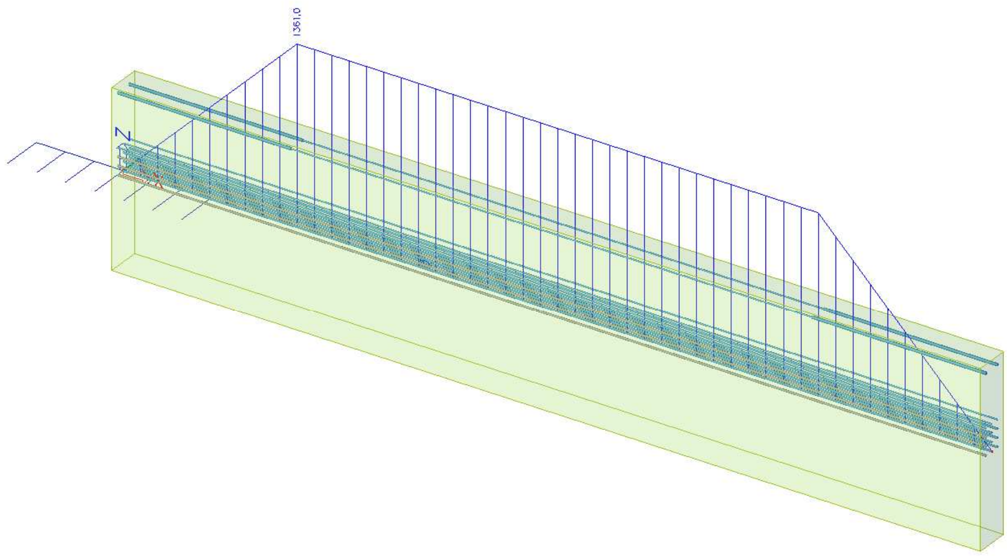




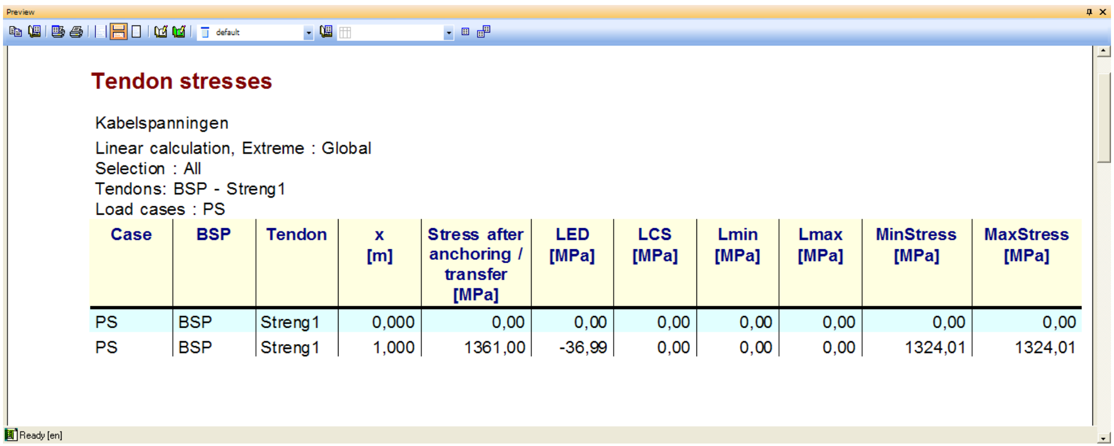
New in the results branch are the Tendon stresses:







With the preview action button we can see the results in detail:

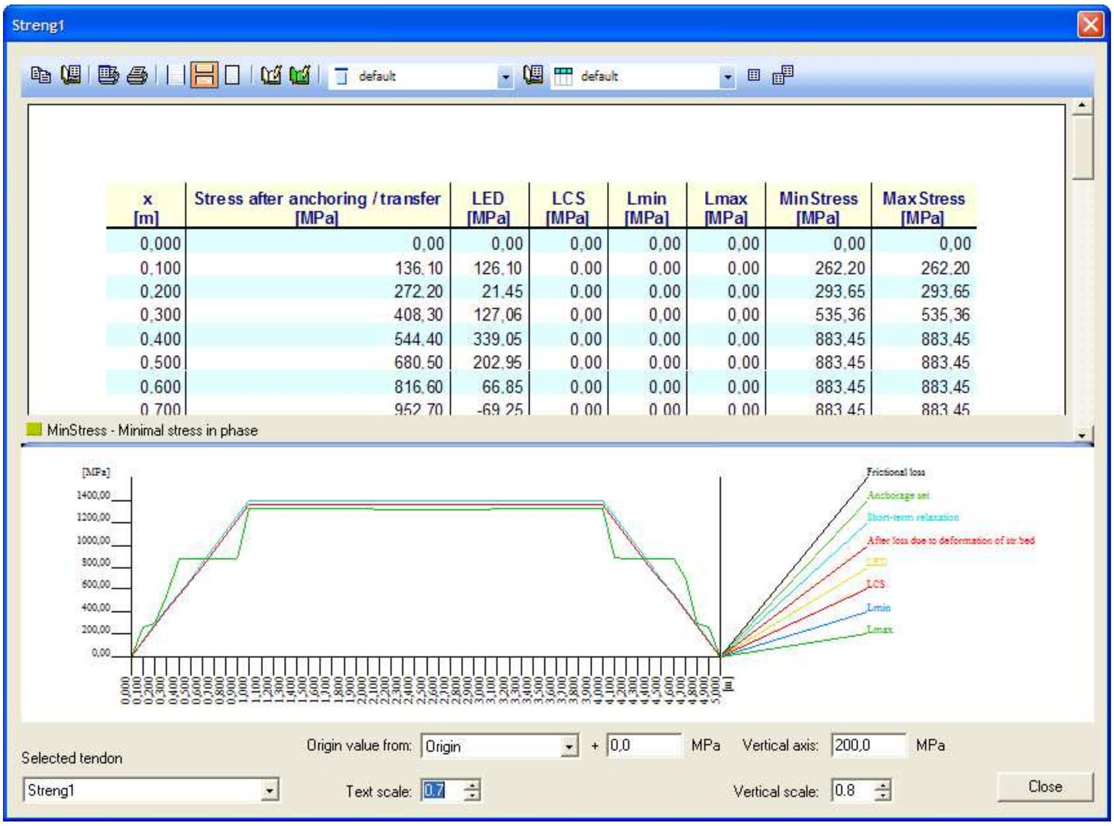


Tendon stresses

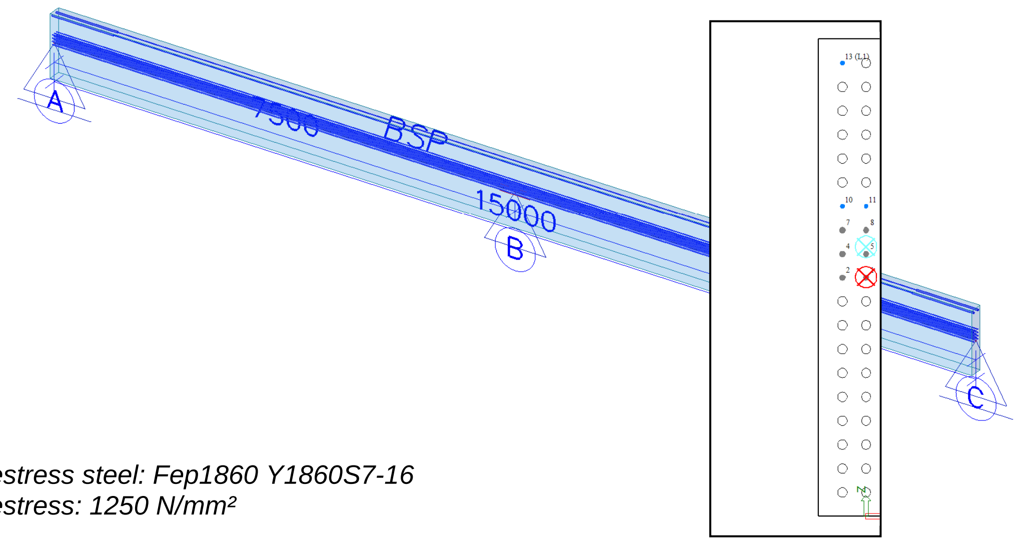
Kabelspanningen
Linear calculation, Extreme : Global
Selection : All
Tendons: BSP - Streng1
Load cases : PS

Case	BSP	Tendon	x [m]	Stress after anchoring / transfer [MPa]	LED [MPa]	LCS [MPa]	Lmin [MPa]	Lmax [MPa]	MinStress [MPa]	MaxStress [MPa]
PS	BSP	Streng1	0,000	0,00	0,00	0,00	0,00	0,00	0,00	0,00
PS	BSP	Streng1	1,000	1361,00	-36,99	0,00	0,00	0,00	1324,01	1324,01

With the action button detailed we can review each strand in detail:


Streng1
default
default
x [m]
Stress after anchoring / transfer [MPa]
LED [MPa]
LCS [MPa]
Lmin [MPa]
Lmax [MPa]
MinStress [MPa]
MaxStress [MPa]
0,000 0,00 0,00 0,00 0,00 0,00 0,00 0,00
0,100 136,10 126,10 0,00 0,00 0,00 262,20 262,20
0,200 272,20 21,45 0,00 0,00 0,00 293,65 293,65
0,300 408,30 127,06 0,00 0,00 0,00 535,36 535,36
0,400 544,40 339,05 0,00 0,00 0,00 883,45 883,45
0,500 680,50 202,95 0,00 0,00 0,00 883,45 883,45
0,600 816,60 66,85 0,00 0,00 0,00 883,45 883,45
0,700 952,70 -69,25 0,00 0,00 0,00 883,45 883,45
MinStress - Minimal stress in phase
[MPa]
1400,00
1200,00
1000,00
800,00
600,00
400,00
200,00
0,00
Frictional loss
Anchorage set
Short-term relaxation
After loss due to deformation of str bed
LED
LCS
Lmin
Lmax
Selected tendon
Streng1
Origin value from: Origin + 0,0 MPa
Vertical axis: 200,0 MPa
Text scale: 0.7
Vertical scale: 0.8
Close

Example 3: Pre-tensioned beam on three supports

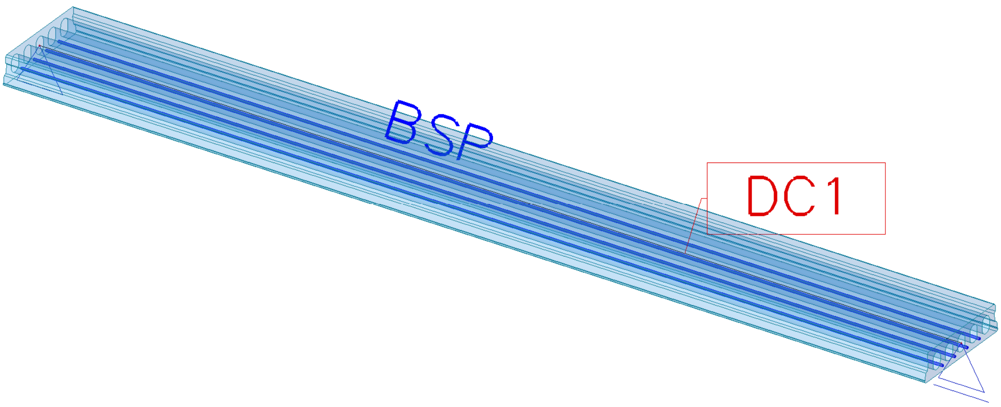


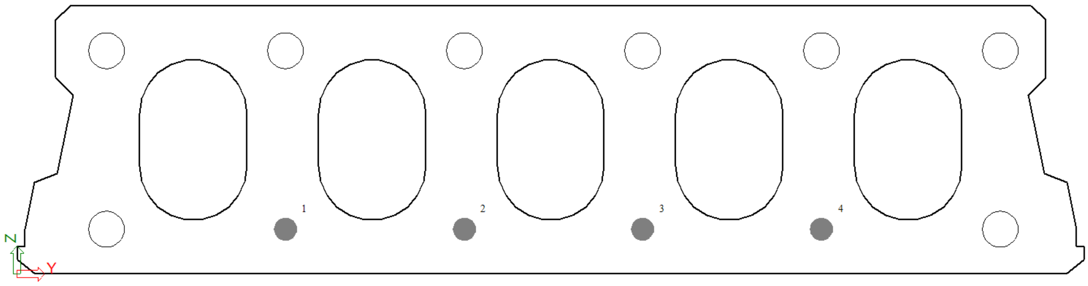
Prestress steel: Fep1860 Y1860S7-16
Prestress: 1250 N/mm²

Work flow:

- *Functionality Prestress*
- *Loadcase 'Prestress'*
- *Concrete branch*
    - *Bore hole patern*
    - *Strand pattern*
    - *Debond*
- *Calculate*
- *Results: ux, ux, Rz, My, N, stresses*
- *Cable stresses: SAT, LED, LCS, Lmin, Lmax*
- *Checks in concrete branch*

Example 4 (stages and prestress): hollow core slab (length 5m)





Concrete: C50/60
Strands: Y1860S7-12,5A
Prestress: 1250 N/mm²
Life load q: 1,5 kN/m

Stage 1
Prestress
Loadcases: s.w. and prestress

Stage 2
Life load
Load cases: empty and life load (variable)

Work flow:

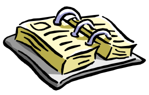
- *Prestress functionality*
- *Define loadcases*
- *Concrete brach*
    - *Bore hole*
    - *Strand patern*
- *Define stages*
- *Calculate*
- *Results: ux, ux, Rz, My, N, stresses*
- *Cable stresses: SAT, LED, LCS, Lmin, Lmax*
- *Control of respons and capacity in concrete branch*

Post-tensioned prestressed concrete

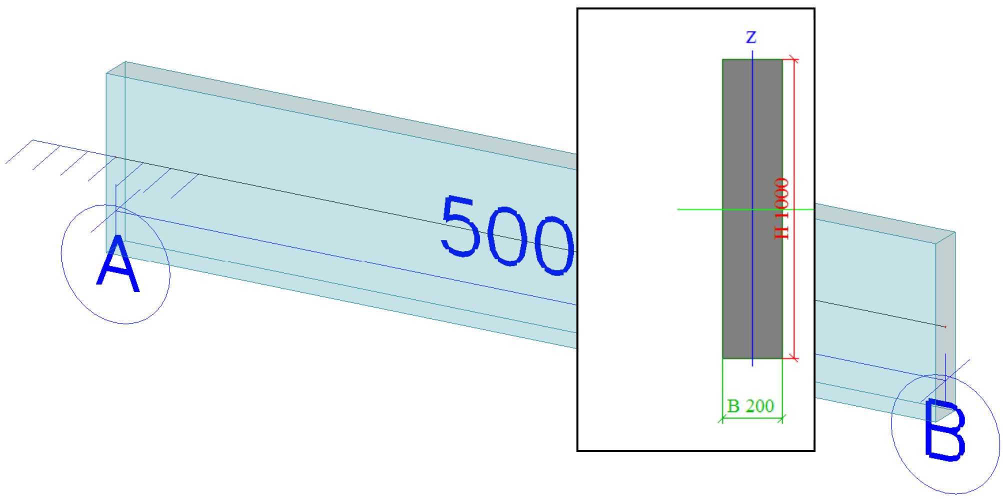
Source geometry

Tendon source geometry

When the shape (geometry) of a tendon is defined, it is possible to use what is termed Source geometry. The source geometry is in fact an independently prepared shape (geometry) of the tendon without any link to a particular structural entity that is to be reinforced. The advantage is clear. The user may prepare the shape of the tendon just once and later assign it to numerous beams, for example. Moreover, the source geometry is created as if intended for a straight beam. But, at the end it may be assigned even to a curved beam. The x axis (longitudinal axis) of the source geometry simply follows the x-axis of the beam regardless of the possible winding character of the beam axis. This feature significantly simplifies the input of tendons into curved beams.



Example: Prestress with post tensioned cables



Prestress steel: Fep1860 Y1860S7-16
Prestress: 1250 N/mm²

Workflow:

- *Prestress functionality*
- *Define loadcase prestress*
- *Structure branch*
    - *Create post tensioned cables*
- *Calculate*
- *Results: ux, ux, Rz, My, N, stresses*
- *Cable stresses: SAT, LED, LCS, Lmin, Lmax*
- *Control of respons and capacity in concrete branch*
- *Change cable e = -0,167mm*
- *Recalculate*
- *Review results and chacks*
- *Change cable geometry (bending of cable)*

Tendon source geometry manager

Source geometry for tendons is managed through a standard Scia Engineer database manager. It means that all the input source geometries are stored in a separate database and, for example, individual items can be exported from one project to another.
The Tendon source geometry manager slightly differs from the standard database manager in that the graphical window is split in two, in order to show both the side-view and plan-view of the source geometry.

The procedure to open the Tendon source geometry manager
Either:
Use tree menu function Library > Post-tensioning > Tendon source geometry.

Or:
When the property table of a tendon is displayed during its input or editing, click the three-dot button [...] in item Source geometry.

Defining a new source geometry for tendon

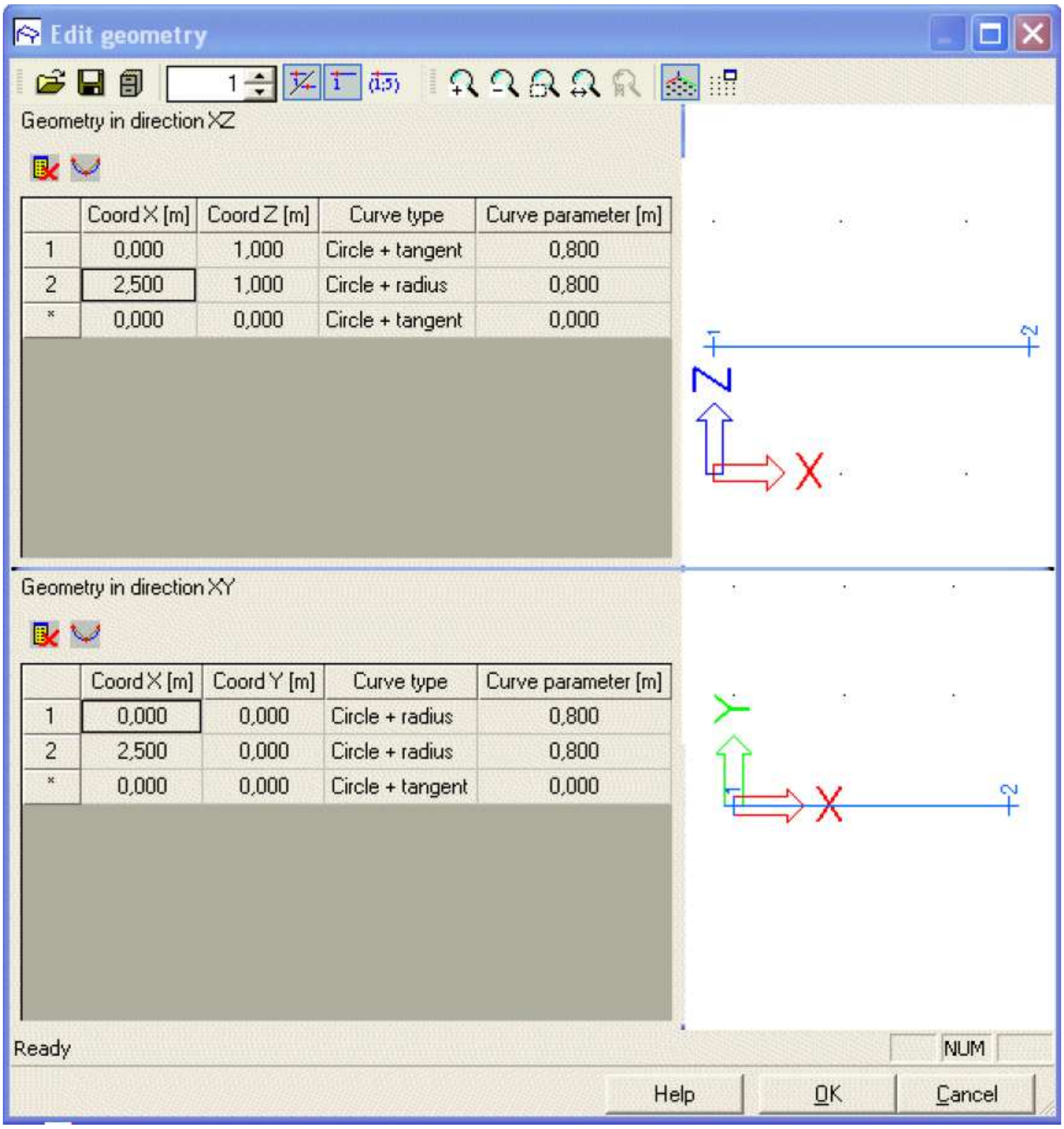
A new source geometry can be input from scratch or by importing another already defined source geometry.
Alternatively, the two ways can be combined. It means that the source geometry can consist of several parts
merged together in one "longer" source geometry.

Procedure to define a new source geometry

1. Open the Tendon source geometry manager.
2. Click button [New].
3. The Edit geometry dialogue is opened on the screen.
4. Input the source geometry.
5. Confirm with [OK].
6. Close the Tendon source geometry manager.

Edit geometry dialogue

The Edit dialogue provides for the numerical input of the shape of the tendon. The user must input individual vertices and types of curve in each vertex.



Note: The proportions of individual parts of the edit dialogue were deliberately distorted in order to fit the picture into one printed page.

Type of input

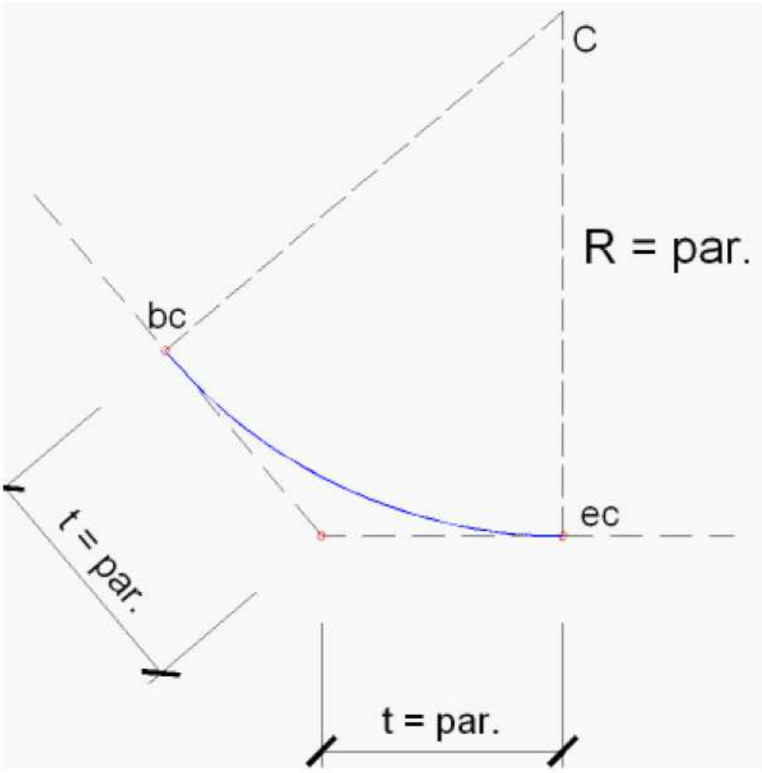
In general, there are two type of input:
(i) the tendon "passes" the individual vertices following the input type of curve, i.e. the tendon does not directly
goes through the vertex,
(ii) the tendon goes directly through the vertices – this is called "points fitting".
For the first input type, the following options are available.

Circle + tangent

Circle; parameter is the distance between vertex and tangent point, see Figure.

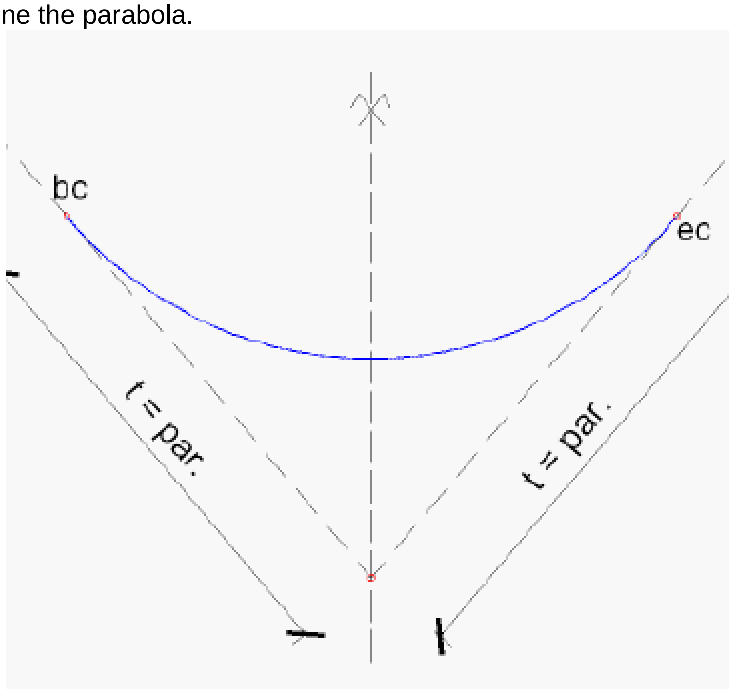
Circle + radius

Circle; parameter is the radius of circle, see Figure. Radius and two tangents determine the circle. Length of tangents is calculated automatically.



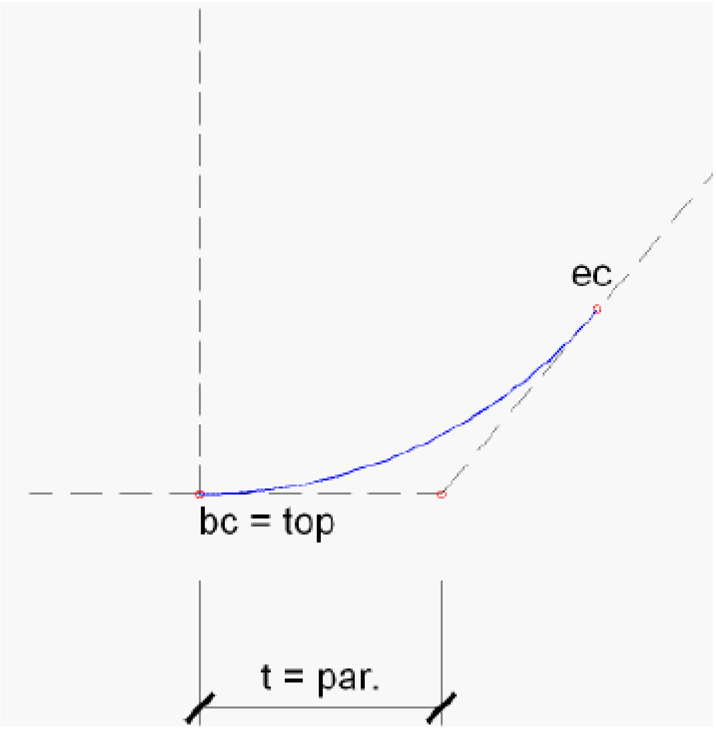
Symmetrical parabola + tangent

Parabola; parameter is the distance between vertex and tangent point (beginning or end of parabola), see Figure. The length of tangent & axis of symmetry of parabola determine the parabola.



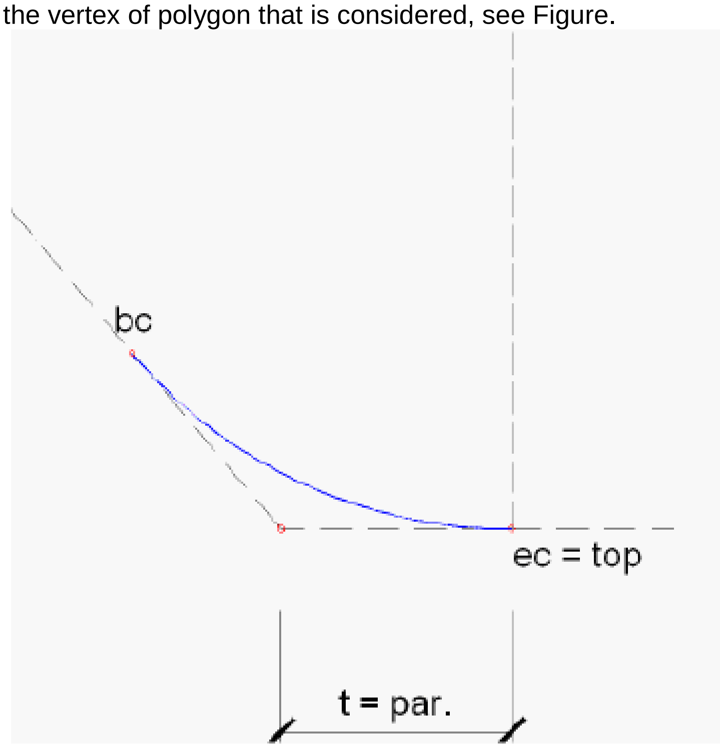
Parabola + tangent [begin]

Parabolic segment with the crown of parabola at the beginning of the curve; parameter is the distance between the beginning of the curve (tangent point at axis of symmetry of parabola) and the vertex of polygon that is considered, see Figure.



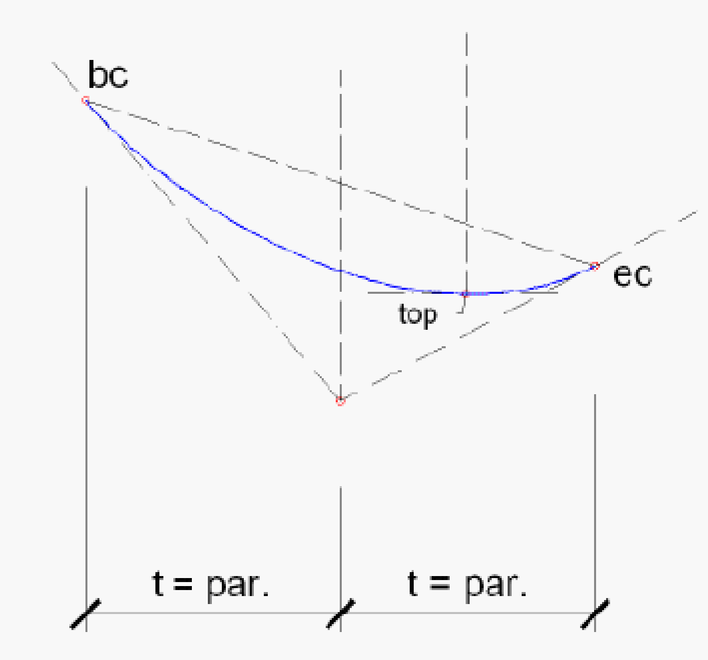
Parabola + tangent [end]

Parabolic segment with the crown of parabola at the end of the curve. Parameter is the distance between the end of the curve (tangent point at axis of symmetry of parabola) and the vertex of polygon that is considered, see Figure.



Parabola + vertical axis

Parabola with vertical axis with respect to macro co-ordinate system. Parameter is the length of projection of tangent into horizontal direction, see Figure.



For the second input type, there is just one option

Group of points fitting

The points defined in the tables of co-ordinates are the points, which the curve of the tendon must fit. The parameters are the tangents of the curve at those points.

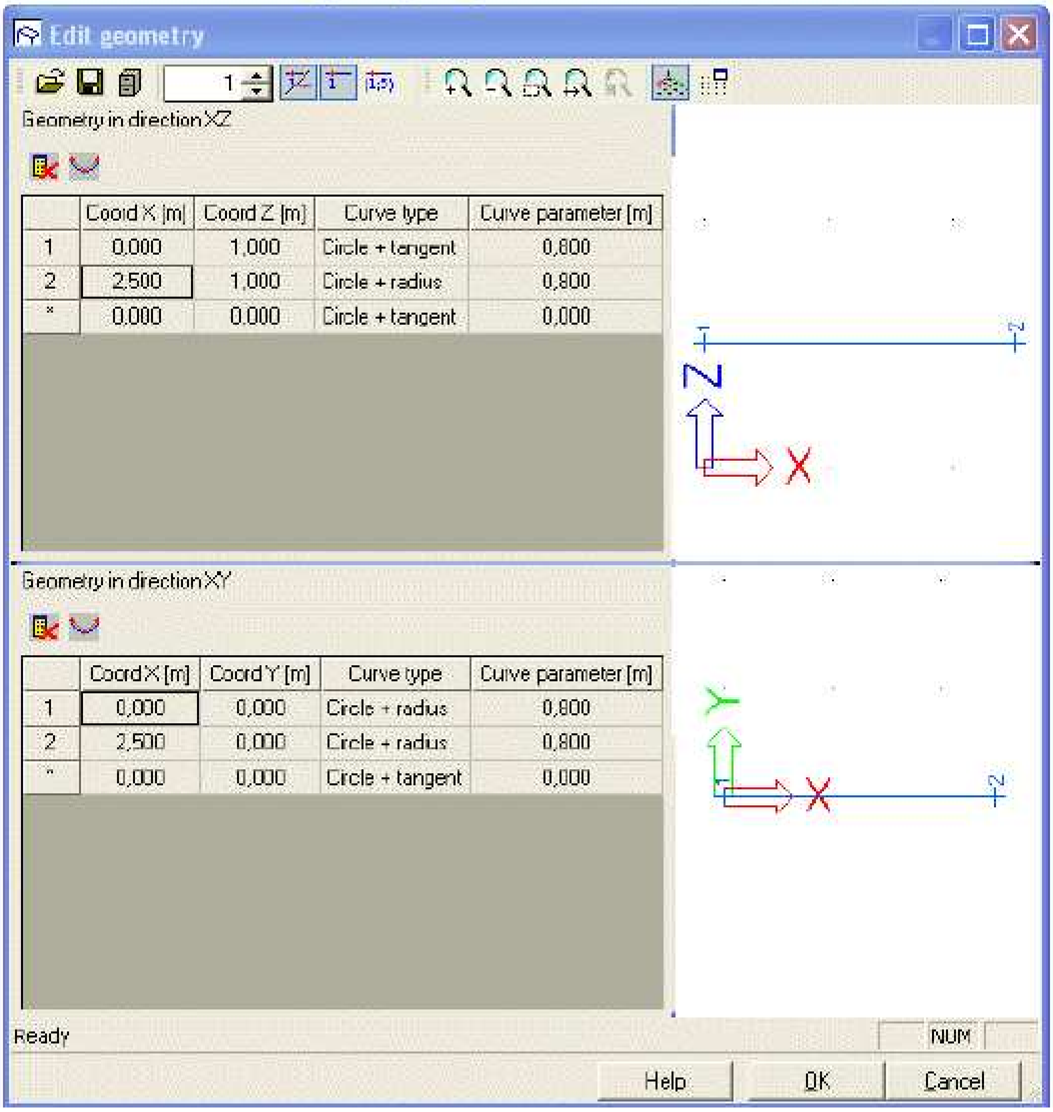
If you define the parameter equal to 100, then the tendon is oriented directly to the following point. If the parameter equals to –100, then the tendon is oriented back to the previous point. Therefore the pair of values
100 and –100 defines a straight part of the tendon between two points.

If the parameter equals to 1000, then the tangent is assumed unknown (arbitrary) and the program will calculate the tangent automatically.

As a result you can determine the limits -1.0 and +1.0 (angles 45°) for the parameters of reasonable tendon profile. For example the parameter equal 0.0 is the input for a horizontal tangent.

The principles of the algorithm is:
1. The user defined tangents are respected.
2. Three consecutive points with the identical y (z) co-ordinate will be fitted by a horizontal straight line.
3. The angles of tangent at anchors and at consecutive points are calculated (if they were not defined by the user). If possible, the whole first and last segment will be fitted by a straight line. If not, the straight line will be inserted into the half of the length of first and last segment.
4. If the distance of two consecutive points is significantly shorter than the distances of the other points, a straight line will be inserted into this segment.
5. The difference between radii of two consecutive curves is minimised.
6. The curved segment of opposite curvature is prohibited in connecting part between straight line and the curve.
7. If two tangents intersect in the half of the length of projection of the curve, a parabola with a vertical axis is applied.
8. In other cases two parabolas with vertical axis are applied.

Layout and controls of the Edit geometry dialogue



Toolbar

Import from file Imports the source geometry from an external TXT or XML file.
Export to file Exports the source geometry into the external TXT or XML file.
Import from library Opens the Tendon source geometry manager and allows you to import another tendon source geometry to the currently edited one.
Vertical scale Changes the vertical scale in the graphical preview windows.
Draw vertex label Switches ON/OFF the labels in the graphical preview window for plane XZ. No labels are drawn in the graphical preview window for plane XY.
Draw number of vertex in label If the labels are ON, the numbers of vertices are printed.
Draw vertex coordinates in labels If the labels are ON, the numbers of coordinates of the vertices are printed.
Zoom icons The set of five standard zoom functions.
Show/hide dot grid Shows/hides the dot grid.
Dot grid settings Enables you to adjust the dot grid so that it meets the needs of the current project.

Input table for XZ plane

Icon [Delete all nodes] This command deletes all the nodes in the table for plane XZ.
Icon [Group of points fittings] Switches the input mode – see paragraph Type of input above.

Input table

Coord X X-coordinate of the vertex of the source geometry.
Coord Z Z-coordinate of the vertex of the source geometry.
Curve type Type of curve "in" the vertex - see paragraph Type of input above.
Curve parameter The parameter of the curve selected in the item above – see paragraph Type of input above.

Graphical preview window

This graphical window shows the side-view the defined tendon shape. You can use the combination "Press-and-hold keys Ctrl+Shift" + "Press-and-hold the mouse right button" and zoom-in or zoom-out the drawing, or the combination "Press-and-hold key Shift" + "Press-and-hold the mouse right button" and move the drawing around the graphical window of the dialogue.

Input table for XY plane

Icon [Delete all nodes] This command deletes all the nodes in the table for plane XY.
Icon [Group of points fittings] Switches the input mode – see paragraph Type of input above.

Input table

Coord X X-coordinate of the vertex of the source geometry.
Coord Y Y-coordinate of the vertex of the source geometry.
Curve type Type of curve "in" the vertex - see paragraph Type of input above.
Curve parameter The parameter of the curve selected in the item above – see paragraph Type of input above.

Graphical preview window
This graphical window shows the plan-view the defined tendon shape. You can use the combination "Press-and-hold keys Ctrl+Shift" + "Press-and-hold the mouse right button" and zoom-in or zoom-out the drawing, or the combination "Press-and-hold key Shift" + "Press-and-hold the mouse right button" and move the drawing around the graphical window of the dialogue.

Main control buttons

The **[OK]** button confirms the input/changes made in the dialogue and closes it.
The **[Cancel]** button abandons the input/changes made in the dialogue and closes it.

Note: It is not allowed to input three successive points in one line. In other words, the direction ofthe tendon must change in each input vertex.

Editing the existing source geometry of tendon

Editing the tendon source geometry in the Tendon source geometry manager
The existing source geometry can be edited from the Tendon source geometry manager opened through the tree menu.

Procedure to edit the tendon source geometry through the tree menu

1. Open tree menu branch Library > Post-tensioning.
2. Start function Tendon source geometry.
3. The Tendon source geometry manager opens on the screen.
4. Select the source geometry that is to be edited.
5. Click button [Edit].
6. The Edit geometry dialogue is opened on the screen.
7. Modify the shape of the tendon.
8. Confirm the changes with [OK].
9. Close the Tendon source geometry manager.

Alternatively, the source geometry can be also edited through the property table of an existing tendon. Editing the tendon source geometry from the property table of the post-tensioned internal tendon

Procedure to edit the tendon source geometry through the property table

1. Select the tendon whose source geometry is to be edited (in fact you may select any tendon).
2. The properties of the selected tendon are displayed in the property table in the property window.
3. Click the three-dot button in item Source geometry.
4. The Tendon source geometry manager opens on the screen.
5. The source geometry of the selected tendon is highlighted in the list of all available source geometries.
6. Click button [Edit].
7. The Edit geometry dialogue is opened on the screen.
8. Modify the shape of the tendon.
9. Confirm the changes with [OK].
10. Close the Tendon source geometry manager.

Note:

It is important to keep in mind that the tendon that is input in a beam through the source geometry, remembers that it was created this way. Moreover, the tendon keeps the link to its source geometry. Consequently, once the source geometry is edited in the Edit geometry dialogue and the changes are confirmed, the shape of all (repeat: All) tendons based on this source geometry that have been already input into the model of the structure change their shape accordingly.

Internal tendons

Parameters of the post-tensioned internal tendon

General

Name Specifies the name of the tendon.
Description This item allows the user to add a short description, if required.
Number Defines the number of the tendon.
Type (informative value) Shows the type of the tendon (internal / external).
Layer Selects the layer of the tendon. Each layer can be assigned to a different layer, if necessary.

Geometry

Geometry input Selects the type of the geometry input.

Source geometry

For this type of geometry, the user must define the shape of the tendon geometry in advance. The predefined tendon is then allocated to the beam and, if necessary, modified in its shape to follow the shape of the beam. The tendon is not stretched to fit the length of the beam. But, it may be curved to follow the real shape of the beam. The latter may be used to simplify the input of tendons in curved beams. The tendon is defined by its "projection" into plane. Then, it is allocated to the curved beam. The shape of the tendon is modified, so that the local x-axis of the tendon follows precisely the local x-axis of the beam.

Direct input

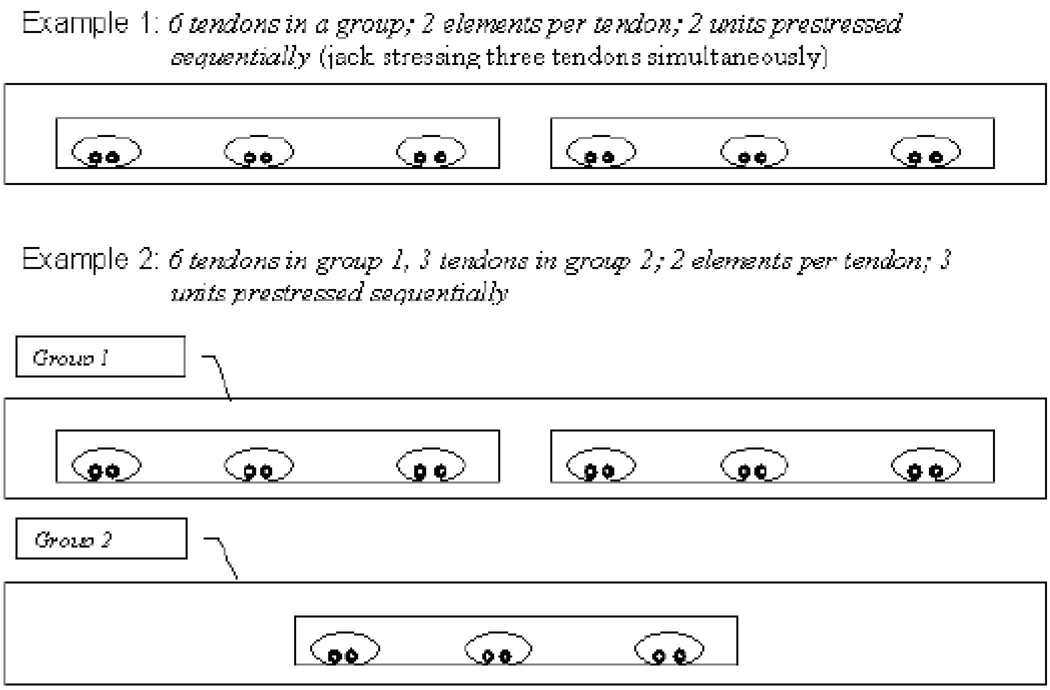
For this type of geometry input, the user defines directly the shape of the tendon in the graphical screen where the beam to be reinforced is displayed. In order to define exactly the shape that is required, an additional toolbar is added to the top of the command line. This added toolbar allows for the input of circular and parabolic intervals.
Note: The same toolbar is displayed when e.g. a new beam is input.
LCS Specifies the way in which the local coordinate system of the tendon (y- and z-axis) is defined.
Source geometry (This item is available only if the Geometry input is set to Source geometry) Here, the user must select the required source geometry for the tendon. It is also possible to invoke the Tendon source geometry manager and input a new source geometry.
Origin of source geometry (This item is available only if the Geometry input is set to Source geometry) It is necessary to define where the origin of the source geometry is to be put in the model. In other words, the user must position his/her tendon into the 3D space. The position is defined by (i) the offset from the origin of the coordinate the local coordinate system of the allocated beam or (ii) in the global coordinates.
Coord. X, Y, Z (This item is available only if the Geometry input is set to Source geometry) These three values define the position of the tendon source geometry origin. The exact meaning depends on the adjustment made in the item above.

Material

Material Specifies the material of the tendon.
Number of tendon elements in tendon Defines the number of wires or strands in tendon.
Number of tendons in group Specifies the number of identical tendons (e.g. in walls of single or multi box section, etc.) which create a group. For details see the picture below the table.
Area (informative value) Shows the sectional area of the tendon.
Diameter of duct Defines the diameter of the tendon duct. The parameter is used

for the test of tendon geometry only.
Allocation This item opens an extra dialogue where the user can select which beams are allocated to the tendon. In general, more than one beam can be allocated to one tendon, as e.g. in the case of several shorter beams running one after another that are reinforced by one long tendon.
Load case The user must select a load case from a list. The list contains only the load cases the Load type of which is set to Prestress. The effects of the prestressing of the tendon will be stored in this load case.

Picture: Tendon groups



Stressing

Type of stressing The type of stressing is analogous to pre-tensioned tendons.
Prestressing from The program offers four options. Simultaneous anchoring of both ends is neither economic nor practically feasible. The options offered in the list are self-explanatory.
Coefficient of friction in curved part of tendon Friction coefficient for curved part of tendon.
Coefficient of friction in straight part of tendon (only for CSN / STN standard) Friction coefficient for the straight part of tendon.
Unintentional angular displacement (only for EC2, NEN) The unintended angular displacement of the tendon.
Anchorage set Defines the anchorage set at the beginning of the tendon.
Stress during correcting Defines the anchorage set at the end of the tendon.
Duration of keeping stress Specifies the duration of keeping constant stress during the correction of relaxation.
Initial stress – **begin** Initial stress at the beginning of tendon (before seating).
Overhang of tendon not included in structural model – begin Defines the part of the tendon at its beginning which is taken into account when calculating the losses, but not when creating the structural model. This is useful in the case that the beam (and tendon) protrudes beyond the theoretical support and the user does not want to include this part of the structure into the model, but wants to have proper losses and tendon geometry for the export into CAD.
Overhang of tendon not included in structural model – end Analogous to the previous item.
Distance between sections for output Defines sections where results are presented.

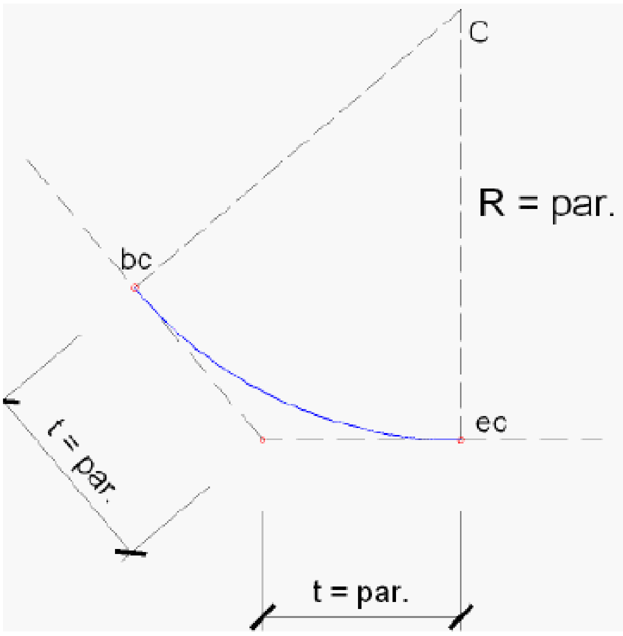
Arc

(These items are available only if the Geometry input is set to Direct input)

Curve type

Circle + tangent

Circle: the parameter is the distance between the vertex and the tangent point, see picture below.

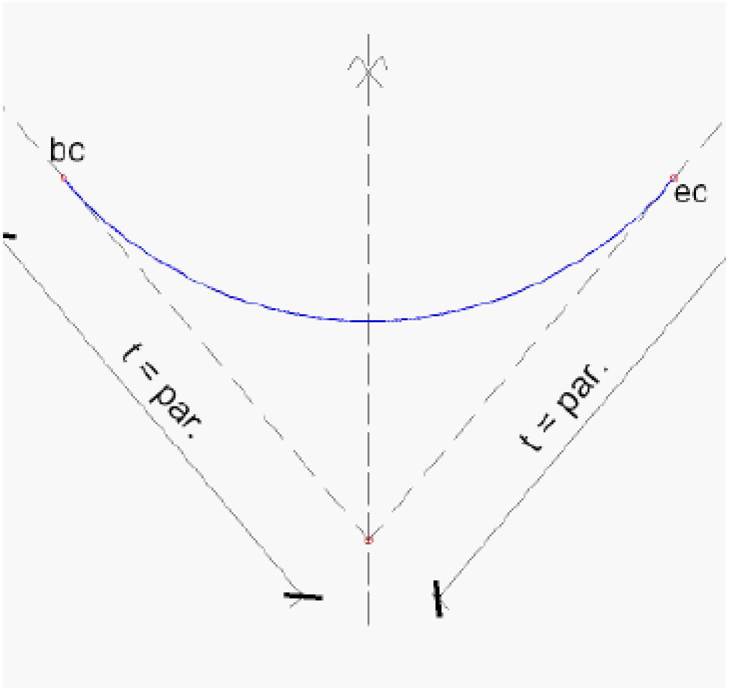


Circle + radius

Circle: the parameter is the radius of the circle, see picture below. The radius and two tangents determine the circle. The length of the tangents is calculated automatically.

Symmetrical parabola + tangent

Parabola: the parameter is the distance between the vertex and the tangent point (beginning or end of parabola), see picture below. The length of the tangent and the axis of symmetry of the parabola determine the parabola.



Curve parameter Here the corresponding curve parameter can be input.

Defining a new post-tensioned internal tendon

Procedure to input a post-tensioned internal tendon

1. Open service Structure.
2. Open branch Tendons.
3. Start function Post-tensioned internal tendon.
4. Fill in the required parameters.
5. Confirm with [OK].
6. Depending on the selected type of geometry input, (i) either define the allocation of the source geometry, or (ii) input directly the geometry of the tendon.
7. End the function.

Editing the existing internal tendon

When the internal tendon is already defined and there is a need to change any of its properties, the following procedure can be applied:

Procedure to edit a post-tensioned internal tendon

1. Select the tendon to be edited.
2. The property window displays its properties.
3. Change any parameter that needs to be modified.
4. If required, invoke any of the Action buttons at the bottom of the property window to carry out other possible alterations (available actions are listed below).
5. When ready, deselect the tendon.

Action buttons available during the editing of a post-tensioned internal tendon

Select allocation

The allocation of the tendon to particular beam or beams can be made through item Allocation in the property table of the tendon (which is displayed in the property window during the editing). This option (in the property table) allocates the beam in a simple table. On the other hand, action button Select allocation starts an interactive function that enables the user to select the allocated members directly in the graphical window.

Edit tendon geometry

This action button starts editing of the shape of the tendon directly in the graphical window.

Table edit geometry

This button opens a dialogue on the screen. The dialogue shows the table with all the vertices of the tendon. The coordinates and arc types including their parameters can be modified here.

Tendon losses

This action is not strictly an editing one, but it is useful during the design of the tendon as well. This action button starts the calculation of prestressing losses and shows the results in a separate dialogue. For more read chapter Prestressing losses in an internal tendon.

Calculation info

This button opens a report summarising the parameters of the tendon necessary for the calculation.

Default values

This action button sets all the tendon parameters to the default values (i.e. the values pre-adjusted by the manufacturer of the program).

Prestressing losses in an internal tendon

The procedure to calculate the losses
1. Select the tendon to be edited.
2. The property window displays its properties.
3. Click Action button [Tendon losses] at the bottom of the property window.
4. Review the results in a separate preview window – see below.
5. Close the dialogue.
6. Deselect the tendon.

Preview window with calculated tendon losses

The preview window is split into two parts. In first part some details of tendon parameters are displayed together with the table of results. Using the toolbar at the top of the window, all the information can be exported to a file (HTML, TXT, PDF, RTF) or directly to printer. In the second part a diagram is shown the distribution of various losses along the length of the tendon. It is possible to change the scale of the diagram or the text. And the pop-up menu (pressing right mouse button), offers some basic functions for the picture: zoom, print, copy to clipboard or save to an external file.

External tendons

Parameters of the post-tensioned external tendon

General

Name Specifies the name of the tendon.
Description This item allows the user to add a short description, if required.
Number Defines the number of the tendon.
Type (informative value) Shows the type of the tendon (internal / external).
Layer Selects the layer of the tendon. Each tendon can be assigned to a different layer, if necessary.
Material Material Specifies the material of the tendon.
Number of tendon elements in tendon Defines the number of wires or strands in tendon.
Number of tendons in group Specifies the number of identical tendons (e.g. in walls of single or multi box section, etc.) which create a group.
Area (informative value) Shows the sectional area of the tendon.
Load case The user must select a load case from a list. The list contains only the load cases the Load type of which is set to Prestress. The effects of the prestressing of the tendon will be stored in this load case.

Stressing

Stress after anchoring The stress in the tendon after anchoring.

Defining a new post-tensioned external tendon

Procedure to input a post-tensioned (free) internal tendon

1. Open service Structure.
2. Open branch Tendons.
3. Start function Post-tensioned free tendon.
4. Fill in the required parameters.
5. Confirm with [OK].
6. Input the geometry of the tendon.
7. End the function.

Editing the existing external tendon

When the external tendon has been already defined and there is a need to change any of its properties, the following procedure can be applied:

Procedure to edit a post-tensioned external tendon

1. Select the tendon to be edited.
2. The property window displays its properties.
3. Change any parameter that needs to be modified.
4. When ready, deselect the tendon.

Note: There are Action buttons for this type of tendon.

Changing the geometry of a post-tensioned external tendon

When you need to alter the shape (geometry) of an external tendon, follow the rules for the modification of a beam. In terms of geometry, the external tendon is in fact a beam. It has end nodes, possible intermediate nodes and the "body" connecting them. Therefore, you may simply edit the coordinates of the nodes, drag and drop the tendon or invoke any function for geometric manipulation.

Results for post-tensioned tendons

The results for post-tensioned tendons can be displayed the same way as for pre-tensioned tendons. Read chapter Pre-tensioned prestressed concrete > Results > Tendon stresses.

Part IV TDA

Brief introduction to TDA

The module TDA allows for the time dependent analysis of prestressed concrete, but also composite 2D frame structures, while taking into account the defined stages of construction, creep, shrinkage, and ageing of concrete.

The method used for the time-dependent analysis is based on a step-by-step procedure in which the time domain is subdivided by time nodes. The finite element analysis is performed in each time node.
Linear ageing viscoelastic theory is applied for the creep analysis. Due to symmetry of the long-term loads both the structure and the load can be adequately modeled in a vertical plane. On that account the plane frame structural model is used. The finite elements on eccentricity represent e.g. the concrete box girder (or separately concrete webs and layers of deck), prestressed tendons, diaphragms, piers, temporary anchoring ties, non-prestressed reinforcement, etc.
All operations in the construction are respected in the structural analysis according to the real production schedule. The elements are installed or removed according to the way of construction. Various operations used in the construction such as addition or removal of segments and prestressed tendons, changes of boundary conditions, loads and prescribed displacements may be modeled.

The prestressed tendons are assumed also as eccentric finite elements. When they are initially stressed, only load terms of the tendons are included in the global equilibrium equations. After anchoring also the stiffness of the tendon is considered. Both, the bonded and unbonded tendons may be modeled.
The long-term losses are automatically included in the analysis. If any element is removed or boundary condition is changed, the internal forces of the element and the appropriate reaction are automatically added to the load vector increment.

The total strain of concrete at the time t is subdivided into three parts: $\varepsilon_\sigma(t)$ is the stress-produced strain, $\varepsilon_s(t)$ the shrinkage and $\varepsilon_T(t)$ is the thermal expansion.
Neither shrinkage nor thermal strains are stress-dependent.
The shrinkage of structural members is predicted through the mean properties of a given cross-section taking into account the average relative humidity and member size.
The stress-produced strain consists of elastic instantaneous strain $\varepsilon_e(t)$ and creep strain $\varepsilon_c(t)$.
The development of modulus of elasticity over time due to ageing is respected. The creep prediction model is based on the assumption of linearity between stresses and strains to assure the applicability of linear superposition. The numerical solution is based on the replacement of Stieltjes hereditary integral by a finite sum.
The general creep problem is thus converted to a series of elasticity problems. The creep calculation is also based on the mean properties of a given cross-section. The creep, shrinkage and ageing effects may be taken into account according to design recommendations EUROCODE 2, CSN 73 1201 and CSN 73 6207 (Czech codes). The method respects stress history, does not require any iteration in single step and does not restrict the type of creep function.

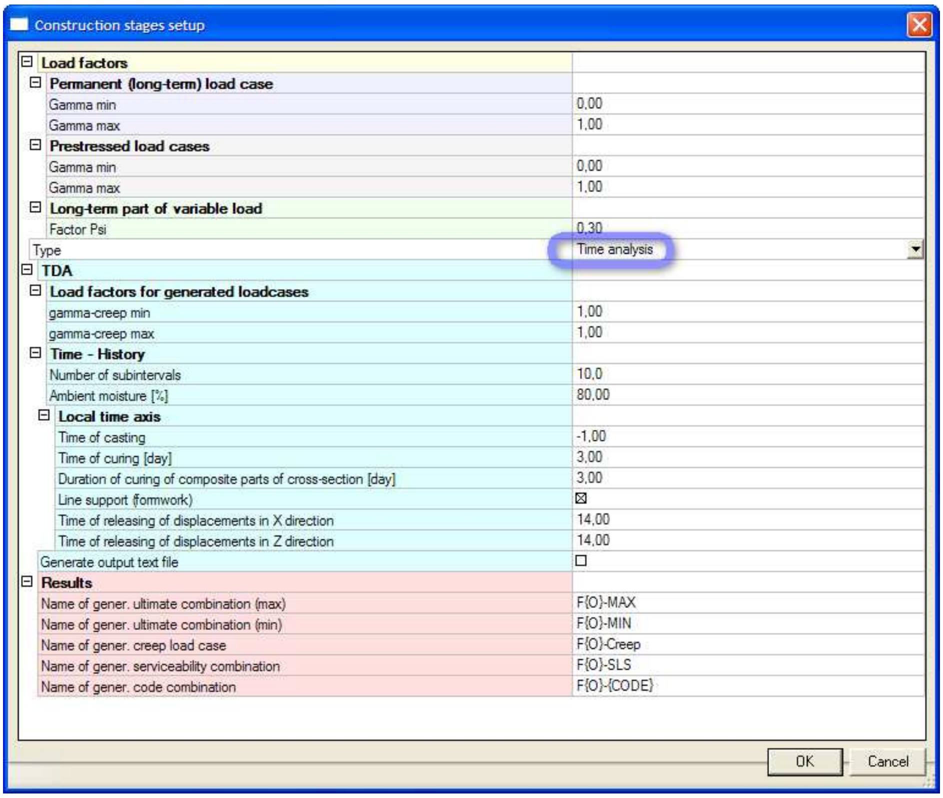
Implementation of construction stages and TDA

Time Dependent Analysis (TDA) is closely linked to the Analysis of Construction Stages (ACS) in Scia Engineer. The difference is that the rheological effects are not considered in ACS. On the other hand "the load-case" and "the combination of loadcases" are the basic "building units" for both TDA and ACS. ACS in fact runs independently on time. It is only a matter of form that each stage is linked to some time node. The increments of dead load in each building stage (construction or service) and the results (the increments of internal forces and deformations caused by this load) are stored in separate load-cases. This load is assumed to be present (applied on the structure) until infinite time. The unloading must be modelled as a new load with opposite sign. For example - the total internal forces in existing structural members caused by dead loads after third building stage are obtained as the results of the combination of three appropriate load-cases. A load-case representing the life load can be added to this combination.

If any prestressing is applied in the building stage, additional permanent load-case must be applied. Then two permanent load-cases are defined in one building stage – one for the dead load and one for prestressing. The user is not allowed to add loads to prestressing load-case. One additional (empty) load-case is generated automatically in each building stage in TDA analysis. These load-cases are used for storing of the increments of internal forces and deformations caused by creep and shrinkage calculated during passed time interval. They are marked as creep-loadcases in Scia Engineer.

TDA setup

These values must be set for the TDA:



Load factors

'**gamma-creep min**' (<=1); '**gamma-creep max**' (>=1): These parameters, that are common for all construction and service stages, can be specified for generated creep load cases. In fact, no load factors will be applied in TDA calculations. Therefore the results of creep load-cases that are generated by TDA will also have no load factors included in themselves (better said load factor = 1.0). After the calculation has been performed, both SLS and ULS combinations are generated. For ULS combinations all factors (gamma min and gamma max) for dead load, prestressing, quasi-permanent load, and creep will be applied using both their maximum and minimum values.

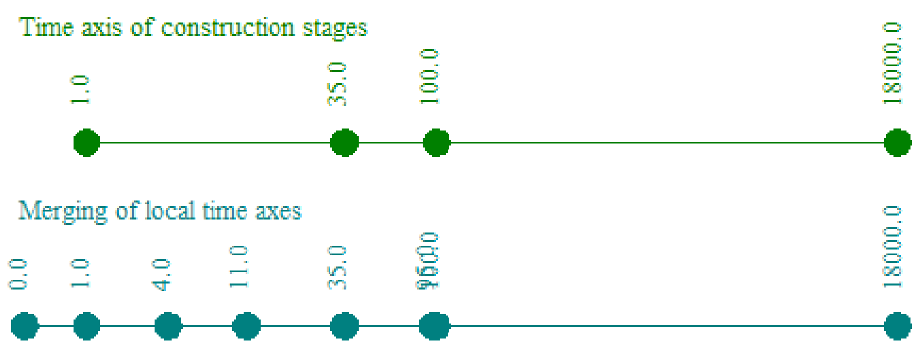
'**Generate output text file**': If this checkbox is ticked, a temporary output text-file is generated in the directory for analysis data and temporary files.

Time history

'**Number of subintervals**': The number of subintervals on the detail time axis. The subintervals following the first time subinterval are generated in log scale. This parameter has an impact on the accuracy of solution of concrete creep. See also Time axis.
'**Ambient moisture**' : Ambient moisture in percentages.

Beam history

Each member has it's own history of in local time axis containing e.g. time of casting, end of curing, etc. All data set in the setup dialog are related to local time axis of relevant 1D member. The origin of the local time axis (zero time) is set to the time, when the appropriate stiffness of macro is introduced (added) into global stiffness matrix of the whole structure ("birth of member"). The relevant member is highlighted in a special colour in the drawing in the main graphic window. The origin of local time axis is then located to global time of current construction stage.



Local time axis	
Time of casting [day]	-1,00
Time of end of curing [day]	3,00
Duration of curing of composite parts of cross-sect...	3,00
Line support (formwork)	☒
Time of instalation of formwork [day]	-1,00
Time of releasing of displacements in X direction [...	10,00
Time of releasing of displacements in Z direction [...	10,00
Time of releasing in rotation [day]	10,00
Member	S1

'**Time of casting**': time of casting of concrete in days. It is possible to input negative value. In such case the stiffness of the elements between the time of casting and the birth of macro (zero local time) is not included into global stiffness matrix. At the same time the age of concrete elements is correct.
'**Time of curing**': Time of curing of concrete in days. In case of "phased cross-section" it is time of curing of concrete of phase one in days.
'**Time of curing of composite parts of cross-section**': Significant only in case of "phased cross-section". It is the time of curing of concrete of the second phase (of composite parts) in days. This input value is in fact the duration of curing of concrete composite parts of the cross-section – it is related to the local time axis of the composite part.
'**Line support (formwork)**': The age of concrete is respected when calculating its modulus of elasticity. At early stages the fresh concrete should be properly supported by formwork (centering). It is therefore possible to define line support of 1D member.
'**Time of releasing of displacements in X direction**', '**Time of releasing of displacements in Z direction**': Self-explanatory. X, Z are axes of global co-ordinate system.

Procedure to adjust TDA parameters

1. Open the service Construction stages.
2. Start function Setup.
3. Input the required parameters.
4. Confirm with [OK].
5. Close the Setup dialogue.

Note:

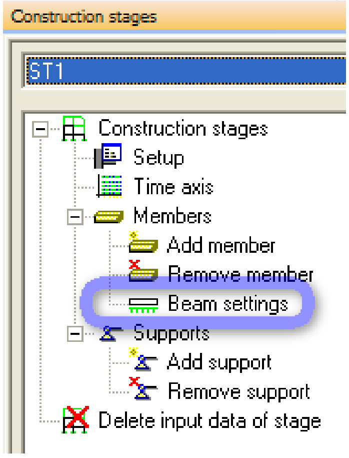
The TDA part of the setup dialogue is available ONLY if TDA module is available, i.e. if e.g. the project is of Frame XZ type.

Local beam history

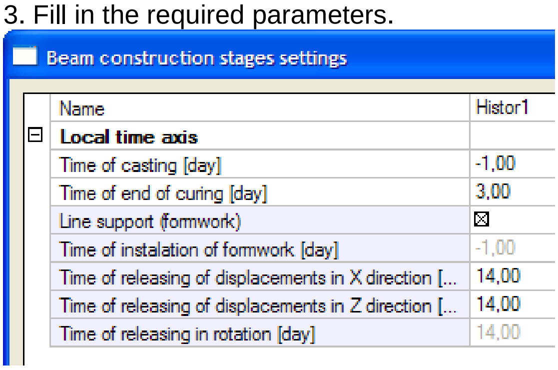
After adjusting the TDA parameters it is also possible to give local beam parameters to each member.

Procedure to input beam local history

1. Open the service Construction stages.
2. Start function Local history.



3. Fill in the required parameters.



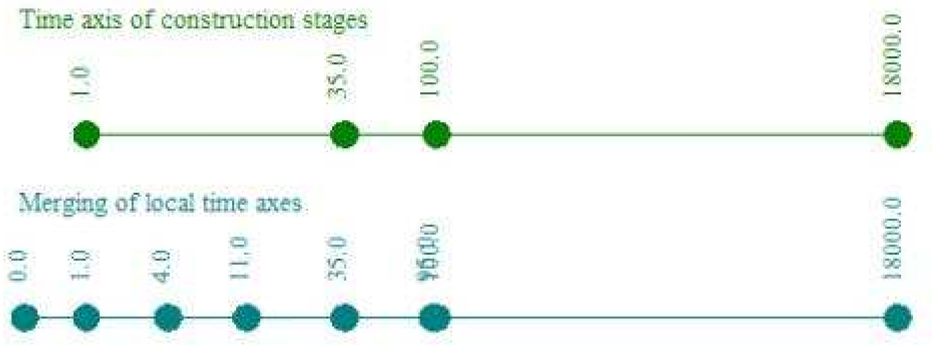
Note:

There are some extra informative parameters here:
*'**Time of installation of formwork**': (informative) The time is equal to the time of casting.*
*'**Time of releasing in rotation**': (informative) The time is equal to greater of previous two values.*

4. Confirm with [OK].
5. Select the required member where the defined history should be assigned.
6. End the function.

The beams with defined local beam history are marked with symbol Histor.

Note: When a beam is added into the model using the Construction stages service, it physically appears in the model and is added into the stiffness matrix. On the other hand, Local beam history may specify that such a beam is e.g. a few days old and the concrete already partially or fully hardened.



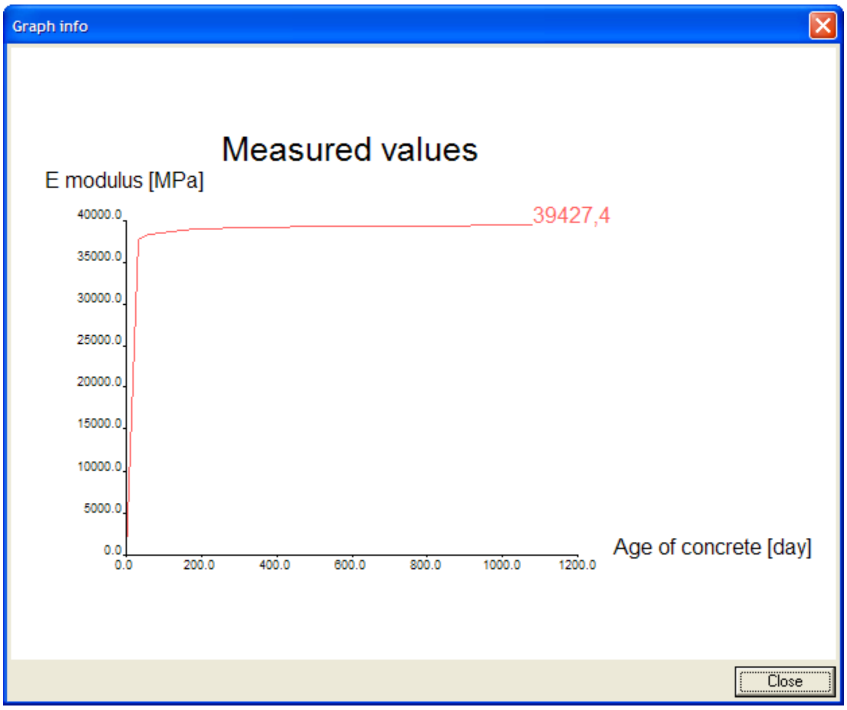
Material setup

These values must be set for the Analysis of construction stages and TDA.

Standard materials available in Scia Engineer material database can be used in TDA and ACS modules. The number of material characteristics for concrete is increased and the influence of ageing (code dependent) is introduced. Also the data of composition of concrete are added for TDA purposes to take into account creep and shrinkage of concrete. New possibility to define measured values of mean compressive strength of concrete is accessible for EC2 concrete.

Measured values	
Measured values of mean compressive strength (influence of ageing)	☒
Measured values I	
Age of concrete [day]	7,0
Mean value of compressive cylinder strength [MPa]	50,00
Emod, sec [MPa]	35654,4
Measured values II	
Age of concrete [day]	28,0
Mean value of compressive cylinder strength [MPa]	60,00
Emod, sec [MPa]	37658,9
Measured values III	
Age of concrete [day]	28,0
Mean value of compressive cylinder strength [MPa]	60,00
Emod, sec [MPa]	37658,94
Standard deviation [MPa]	5,0
Characteristic compressive cylinder strength (28) (Fck) [MPa]	51,8
Graph	

On ticking the checkbox Measured values of mean compressive strength (influence of ageing) (in the material editing dialogue opened from the Material database manager) new edit-boxes become accessible. The user can input measured values of mean compressive strength of concrete at age of concrete t1 and t2 (t1<t2). One of the input values can be equal 28 days. When clicking on the button after Graph, user will see a graphical interpretation of his inputted values:

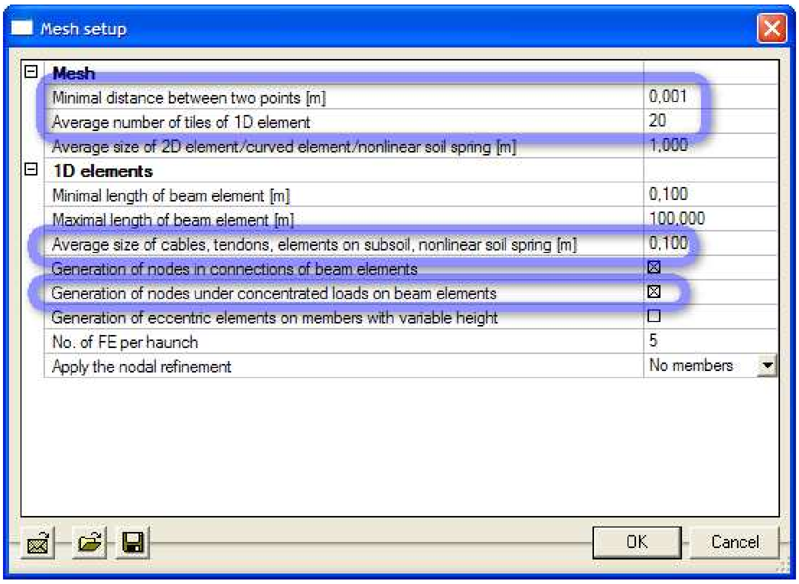


This feature of the program could be used especially for rapid hardening concrete or in case of any arrangement made to speed up the hardening of concrete (in prefab industry). Taking into account the input parameters, the modified CEB FIP 1990 [2] functions for the development of strength and modulus of elasticity (ageing) are used.

CSN: ‘**Water content**’: Content of water in concrete.
EC2: ‘**Type of cement**’: It can be slow hardening, normal hardening, rapid hardening.

Mesh setup

These values must be set for the TDA



- ✓ ‘**Minimal distance between two points**’ ≥ 0.001 m
- ✓ ‘**Average number of tiles of 1D element**’ must be ≥ 2.
- ✓ The geometry of finite elements representing the prestressed tendons is generated from the real tendon geometry inclusive of the curves at vertexes of basic (input) tendon polygon. The finite elements then make the polygon with the vertexes at the distances equal to ‘**Average size of cables, tendons, elements on subsoil**’. After the definition of the element geometry, the mesh is thickened according to the option ‘**Average number of tiles of 1D element**’, without backward influence on the geometry of tendon elements. Therefore the option ‘**Average size of cables, tendons, elements on subsoil**’ must be chosen according to needed accuracy of modelling of tendon geometry.
- ✓ Generation of nodes under concentrated loads on beam elements = on.

For reasons of numerical stability of TDA solver it is recommended to adjust:

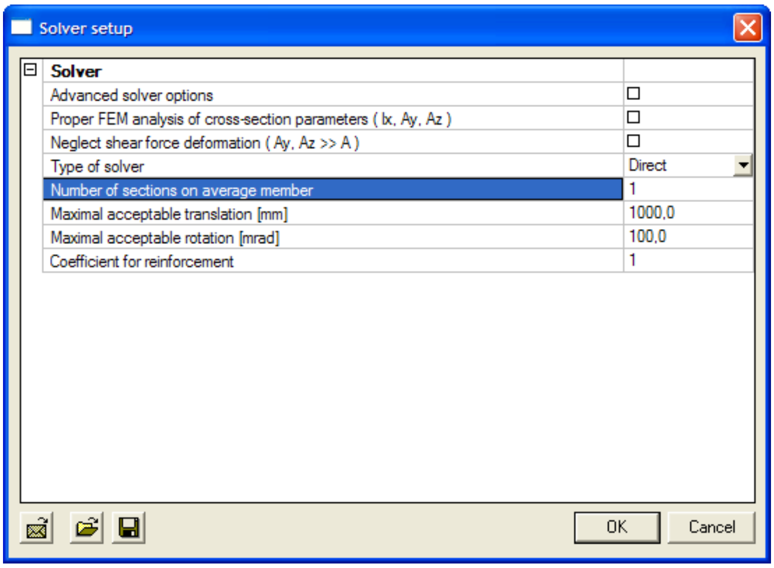
- ✓ Minimal length of beam element = 0.05 m.

Procedure to adjust mesh parameters

1. Start menu function Setup > Mesh.
2. Adjust the required parameters.
3. Confirm with [OK].

Calculation setup

These values must be set for the TDA.
Number of sections on average member (element) = 1 (detail results of internal forces at intermediate sections can be obtained by refining mesh).

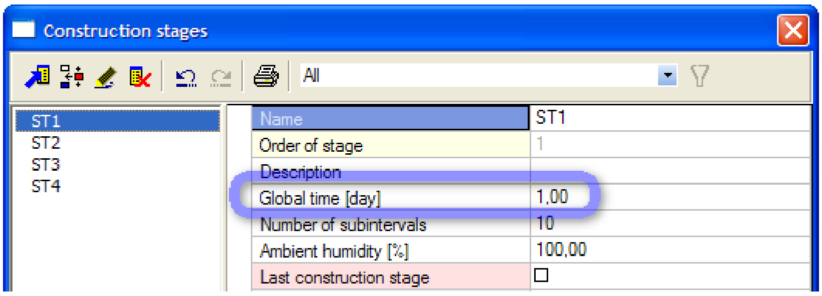


Procedure to adjust solver parameters

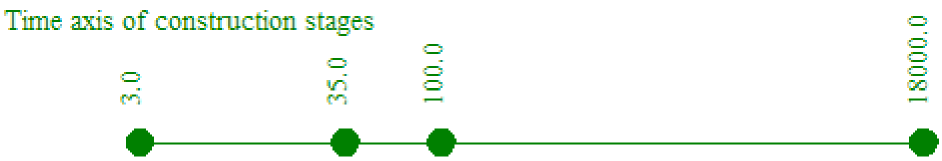
1. Start menu function Setup > Solver.
2. Adjust the required parameters.
3. Confirm with [OK].

Time axis

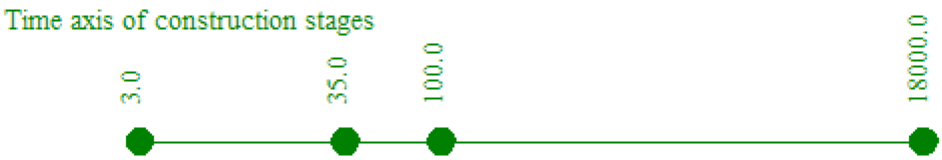
The time is a new quantity in TDA analysis. First of all, the user defines global time as one of the parameters of each construction phase. This time is be assigned to the current stage.

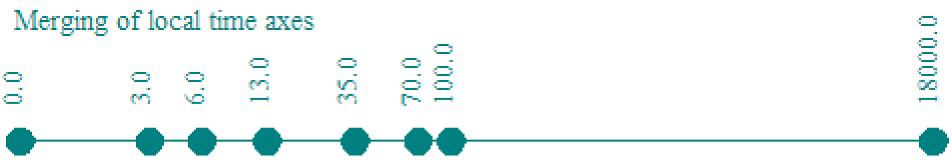


Additional time nodes are generated for TDA, see local history of a beam. Therefore, number of time nodes is greater than the number of stages. Additional time nodes required for proper accuracy of creep analysis can be generated in the dialog Time axis edit. Logarithmic scale is used for the time axis drawing in the dialog '**Time axis edit**'. First of all the user inputs the global time of the first construction stage, which is the first time node on the "Time axis of construction stages".

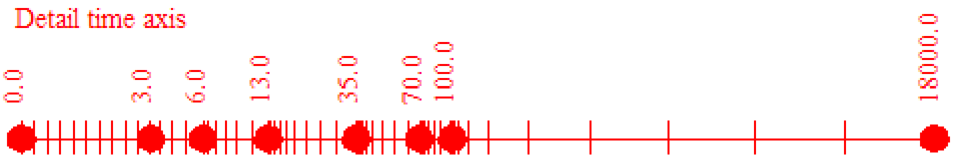


For example if we assume (local) time of casting of first macro –3 days,
then it is recommended to input time of the first stage +3 days (but it is not necessary). If we do so, the time of first construction stage in global time axis will be still +3 days also after merger of the local axis to the time axis of construction stages.





The origin of the time axis is always moved to the time of casting of the first member! Now we have "Merging of local time axes" (time axis of stages + local time axes): t0, t1, t2, …, ti, tj, … tn, and we need to generate detail time nodes ti+k, k=1,2, … until ti+k<tj (to ensure the required accuracy of creep analysis). It is done automatically.



Time axis edit dialogue

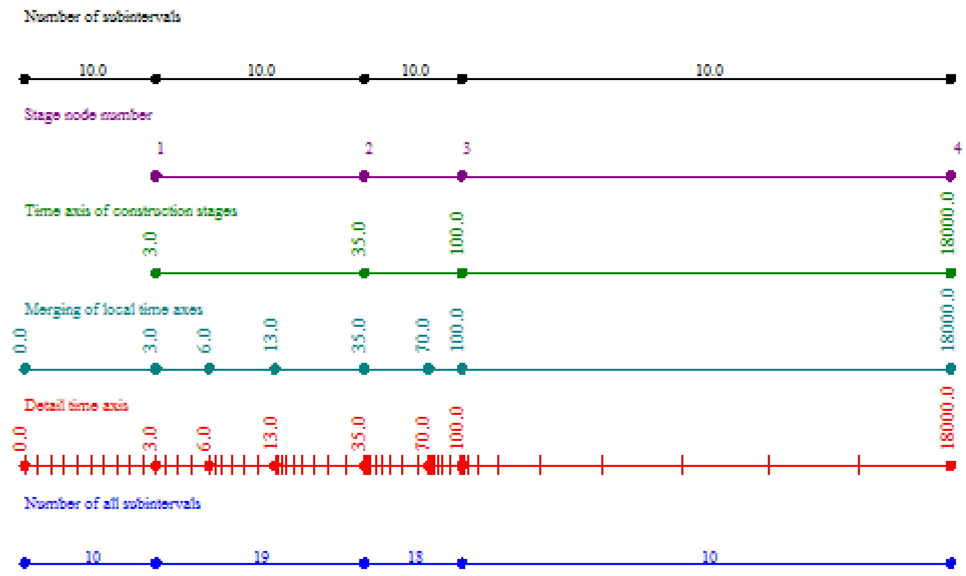
The Time axis edit dialogue consists of three parts:

- ✓ graphical window,
- ✓ property window,
- ✓ input window.

Graphical window

The graphical window shows all the information related to the time axis:

- ✓ number of subintervals,
- ✓ stage node number,
- ✓ time axis of construction stages,
- ✓ merge of local time axis,
- ✓ detail time axis,
- ✓ number of all subintervals.

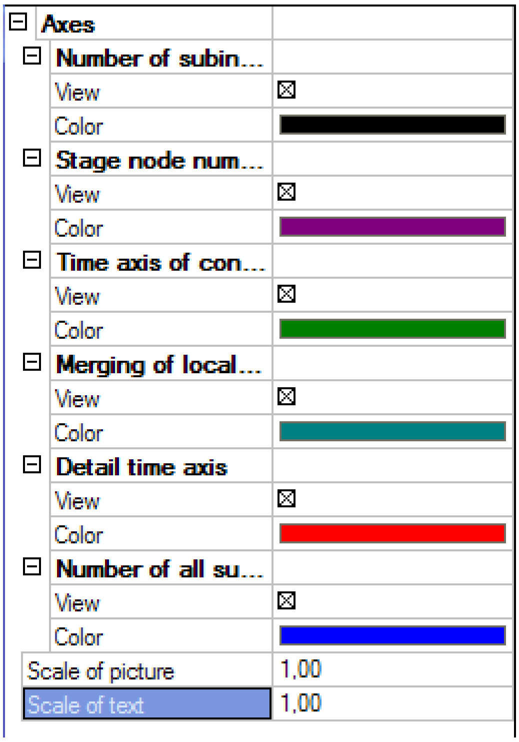


The graphical window supports standard features of SCIA ENGINEER graphical windows:

- ✓ pop-up menu with a set of zoom, print and export functions,
- ✓ [Ctrl] + [Shift] + right-click and drag to zoom in and out the drawing,
- ✓ [Shift] + right-click and drag to move the drawing.

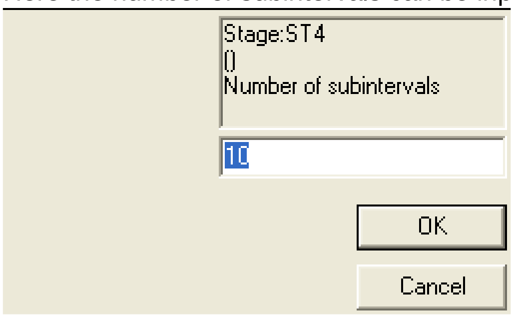
Property window

This window enables you to show or hide required information in the graphical window. It is also possible to adjust colours of individual diagrams and set the scale of the text and the picture.



Input window

Here the number of subintervals can be input for the selected interval.



Procedure to edit time axis (change the number of subintervals)

1. Open the service Construction stages.
2. Start function Time axis.
3. The Time axis edit dialogue is opened on the screen.
4. You may:

- ✓ review the input data: to display or hide the required axis, just click the appropriate ‘View’ checkbox in the property window of the dialogue. To change the colour of the required axis, click the colour box in the property window of the dialogue, a "three-dot-button" becomes enabled, press it and select the required colour.
- ✓ change the number on subintervals: On the axis ‘**Number of subintervals**’ select the interval you want to edit. The input box in the bottom right corner of the dialogue becomes accessible and shows the currently defined number of subintervals for the selected interval. You may change the number.

5. When ready, close the dialogue.

Note: A small number of subintervals is suitable for the first analysis and tuning of the model. The accuracy is not perfect, but the calculation is fast and necessary re-calculations do not take too much of your precious time. Once the model has been tuned, it is highly recommended to increase the number of subintervals in order to obtain satisfactory accuracy of the results.

Analysis

Finite Element formulation

The method used for the time-dependent analysis is based on a step-by-step procedure in which the time domain is subdivided by time nodes. The finite element analysis is performed in each time node. Linear ageing viscoelastic theory is applied for the creep analysis.

The cross-sections of the structural members usually consist of various materials, e.g. concrete girder or composite slab, prestressed tendons or reinforcement that are modelled by individual elements. Therefore the centroidal axis of the element is to be placed in an eccentricity, relating to the reference axis, which connects the nodes. Full compatibility at adjacent surfaces of two eccentric elements must be ensured. That's why the finite element with two external and one internal nodes is used. The internal node is situated at the centre of the element. To fulfill the requirement for compatibility of two eccentric elements fixed to common nodes, the axial and transverse displacements are approximated by the polynomial function of order 2 and 3, respectively. Stated quite simply, polynomial functions are functions with x as an input variable, made up of several terms, each term is made up of two factors, the first being a real number coefficient, and the second being x raised to some non-negative integer power.
All elements with different eccentricity, which connect identical nodes, create the substructure. The static condensation of internal node parameters is used, thus the full compatibility between eccentric elements is fulfilled.

The cross section of the element is constant along the length of element. It is assumed linear distribution of normal forces and bending moments and constant distribution of shear forces along the length of the element. Relatively detailed subdivision of the structural member on finite elements is therefore required.

Solution strategy

Creep and shrinkage of structural members are predicted through the mean properties of a given crosssection, taking into account the average relative humidity and member size. The creep, shrinkage and ageing effects may be taken into account according to the design recommendations of EC2, CSN 73 1201 and CSN73 6207 (the latter two being Czech standards). The method used for creep analysis does not require any iteration in one step and does not restrict the type of creep function. It is based upon the assumption of linearity between stresses and strains to assure the applicability of linear superposition. The development of modulus of elasticity over time due to ageing is taken into account. The method used for the time-dependent analysis is based on a step-by-step computer procedure in which the time domain is subdivided by discrete time nodes ti (i = 1,2, ... n) into time intervals. The solution in the time node i is as follows:

1. The increments of strains, curvatures and shear strains caused by creep during the interval <ti-1,ti> are calculated. Correspondingly the shrinkage strains are also calculated.
2. The load vector dFp is assembled as equivalent to the effects of generalised strains calculated in the step 1.
3. The stiffness matrices K of the elements are calculated for the time ti and the stiffness matrix of the whole structure Kg is assembled.
4. The system of equation $Kgd\Delta g = dFp$ is analysed. The vector of increments of nodal displacements $d\Delta g$ is added to the vector of total nodal displacements Δg.

5. The elements are analysed in the central co-ordinate system (the co-ordinate system, in which x– axis is created by centroids of cross-sections of the element). The increments of internal forces and increments of elastic strains are calculated from the increments of displacements of the element nodes.
6. The changes of the structural configuration carried on in the time node ti are introduced.
7. The increments of generalised strains of the elements that are prestressed (or loaded by changes of temperature) in the time node ti are calculated. The losses of the prestressing due to the deformation of the structure are automatically included in the analysis through the increments of internal element forces.
8. The load vector dFz is assembled as equivalent to the effects of generalised strains calculated in the step 7. The increments of other types of the long-term load applied in the time node ti are added to the load vector dFz.
9. The system of equation $Kgd\Delta g = dFz$ is analysed. The vector of increments of nodal displacements $d\Delta g$ is added to the vector of total nodal displacements Δg.
10. The increments of internal forces and increments of elastic strains are calculated from the increments of displacements of the element nodes.
11. The increments of internal forces calculated in the steps 5 and 10 are added to the total internal forces. The increments of elastic strains calculated in the steps 5 and 10 are added together and saved to the history of elastic strains as the increments in the time node ti.
12. Go to the first step of the time node i+1.

Running the calculation

Both Analysis of Construction Stages and Time Dependent Analysis are run the same way.

Procedure to run ACS / TDA

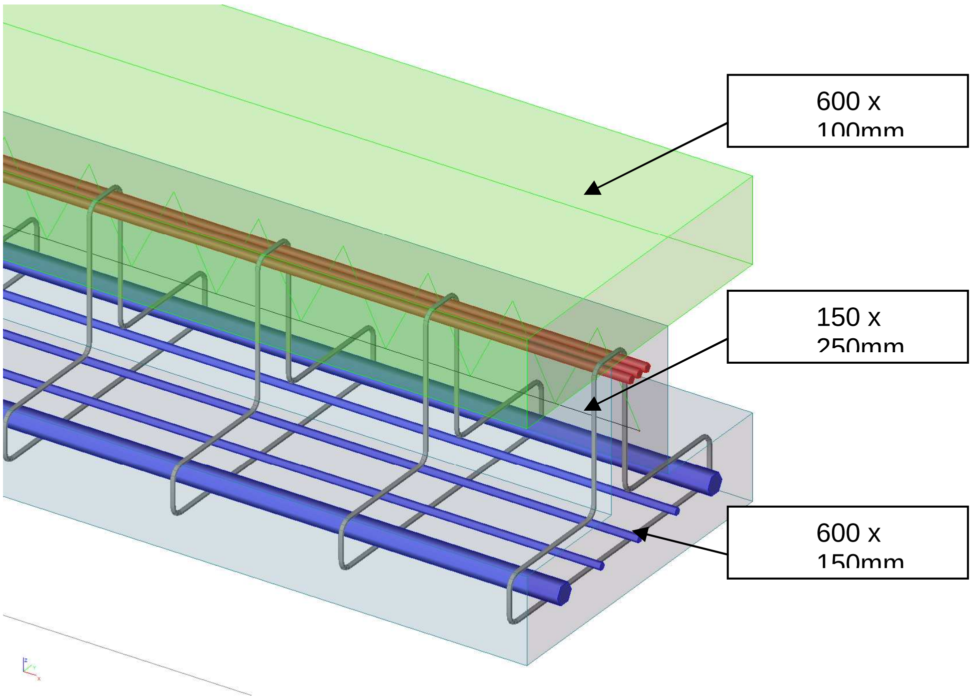
1. Call menu function Calculation, Mesh > Calculation.
2. Select Construction stages analysis.
3. Click [OK] to start the calculation.

Note:

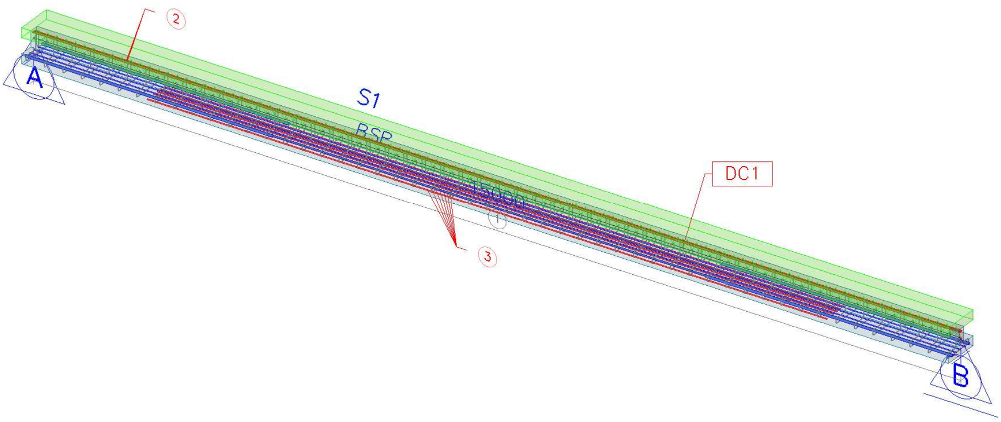
When the Time Dependent Analysis is started, the program may issue a warning that some solver and mesh parameters must be re-adjusted in order to meet the analysis requirements. You may either select the automatic re-adjustment and continue with the calculation, or cancel the calculation and make manual adjustment according to chapters Mesh setup and Calculation setup.

Example (stages/prestress/TDA): Bridge

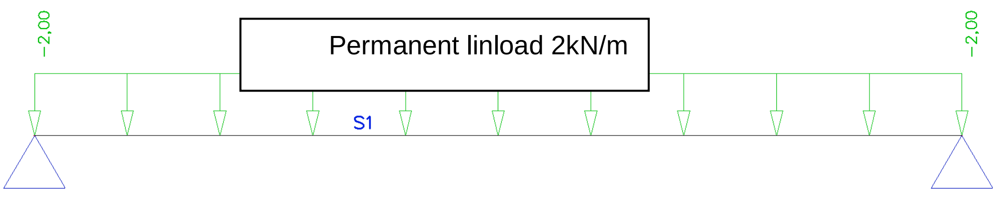
Geometry:

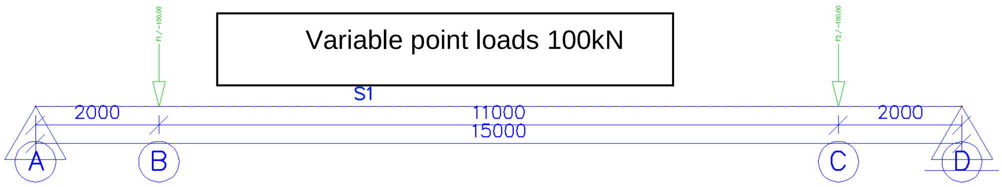


Beam on two supports L = 15m:



Loads:





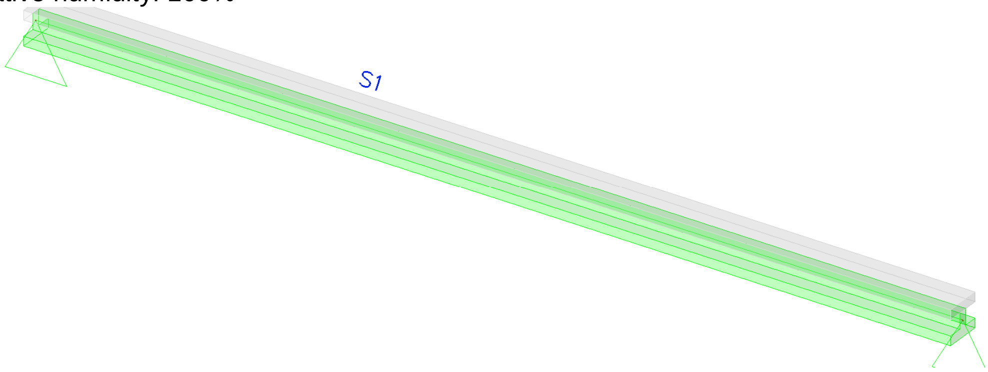
Stage 1

Prestress and selfweight beam

Loadcases: Prestress and self weight beam

Time: day 1

Relative humidity: 100%



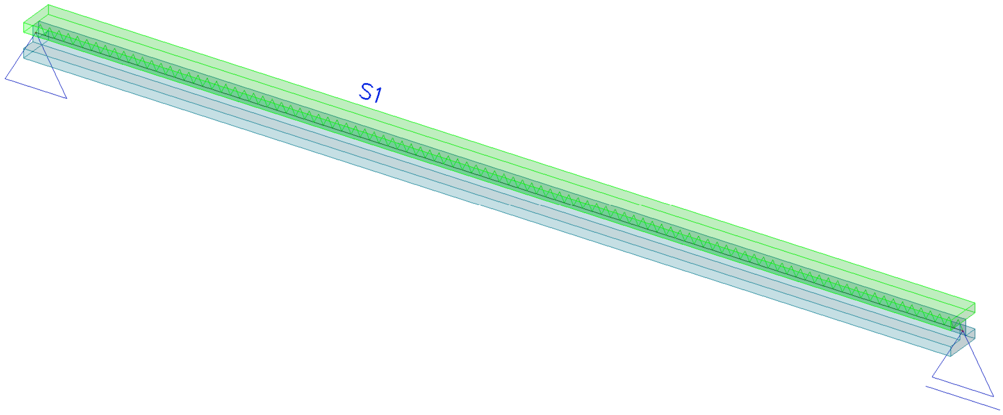
Stage 2

Cast of deck

Loadcases: self weight deck

Time: day 35

Relative humidity: 80%



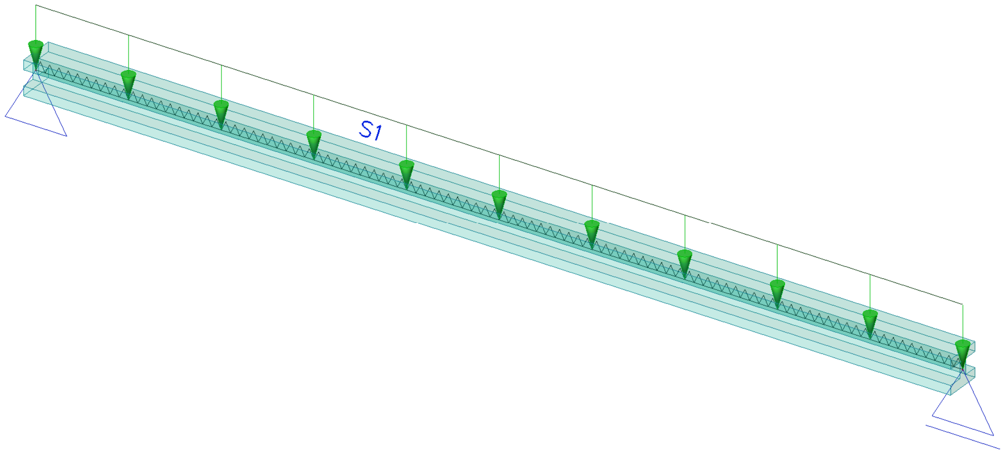
Stage 3

Permanent loading

Loadcase: Permanent

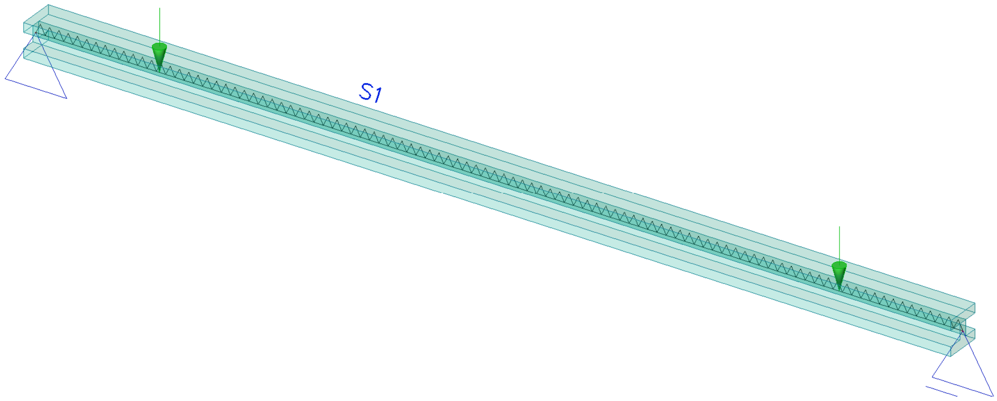
Time: day 100

Relative humidity: 80%

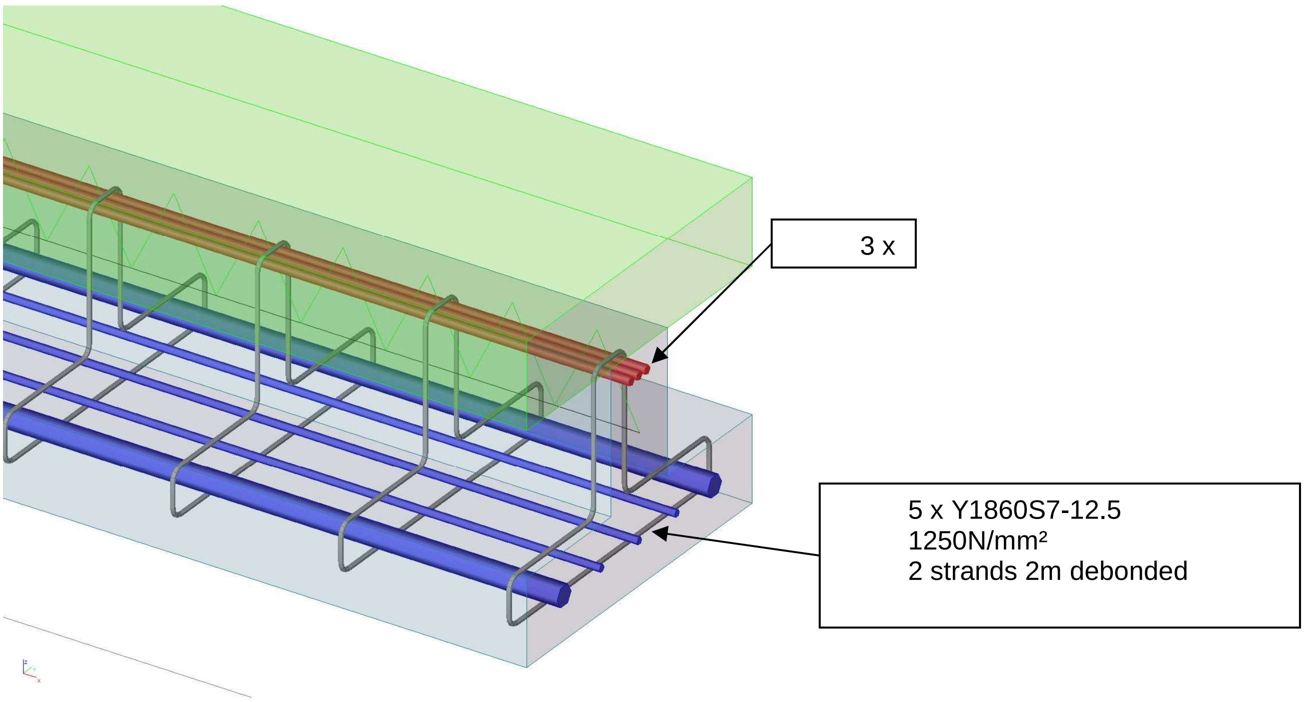


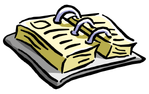
Stage 4

Service stage
Loadcase: Variable & empty
Time: day 18000
Relative humidity: 80%



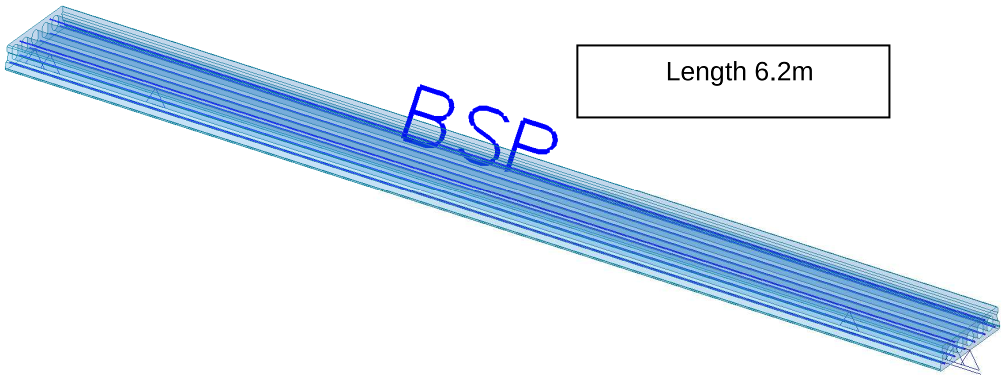
Reinforcement and prestress:

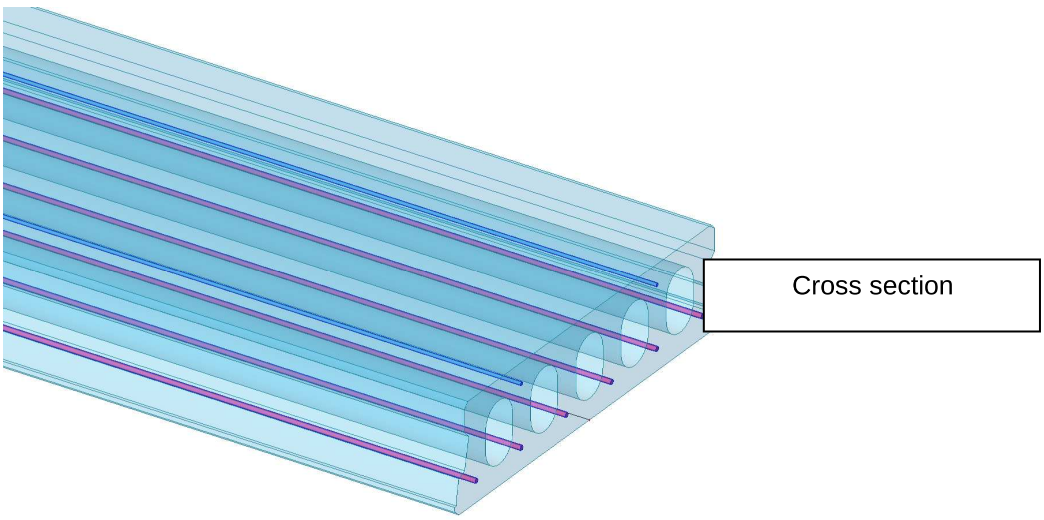


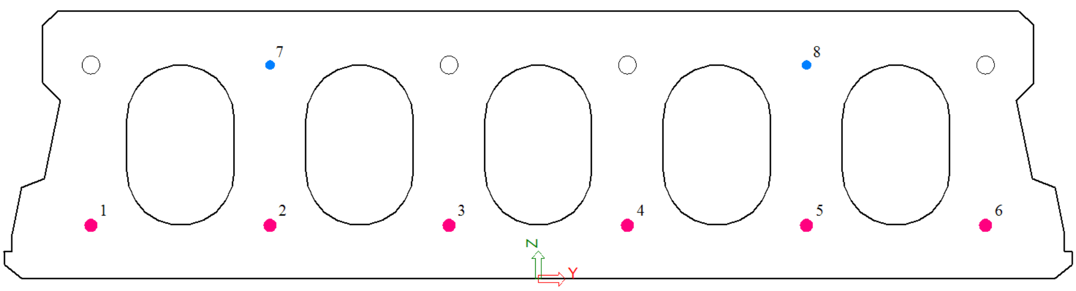


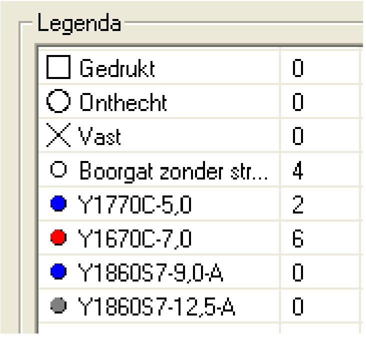
Example (Stages/prestress/TDA/Parameters): hollow core slab

Geometry:







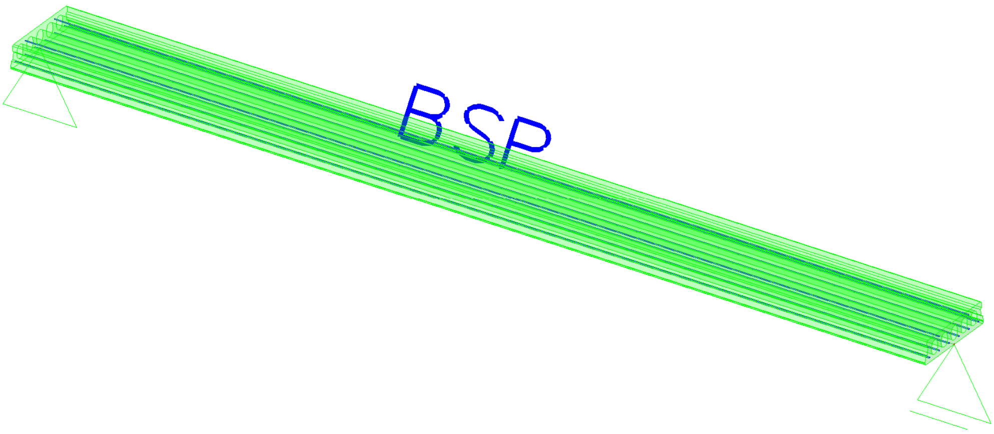


Legenda

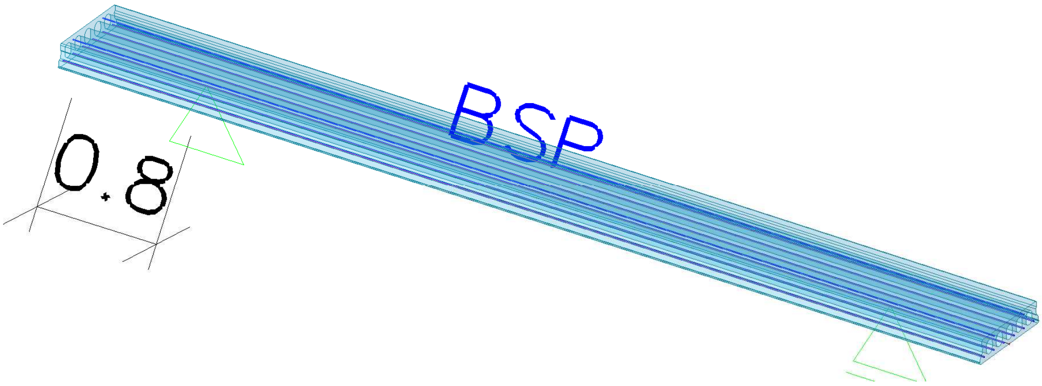
Gedrukt	0
Onthecht	0
Vast	0
Boorgat zonder str...	4
Y1770C-5,0	2
Y1670C-7,0	6
Y1860S7-9,0-A	0
Y1860S7-12,5-A	0

Initial stress 1250N/mm²

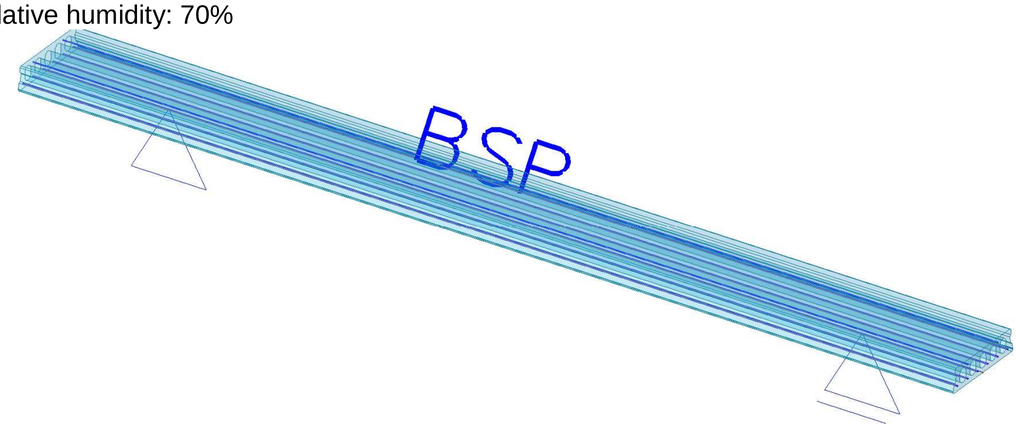
<u>Stage 1</u>
Prestress and selfweight beam
Loadcases: Prestress and self weight beam
Time: day 1
Relative humidity: 100%



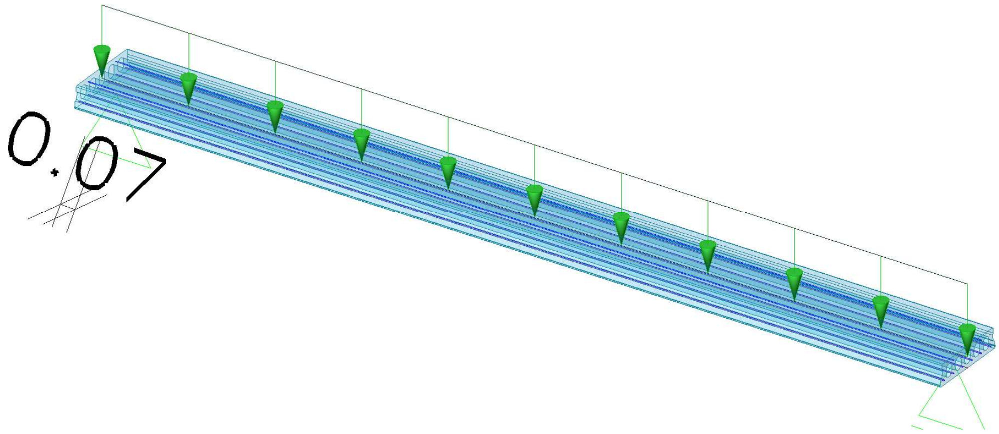
<u>Stage 2</u>
Transport
Loadcases: empty
Time: day 1.1
Relative humidity: 100%



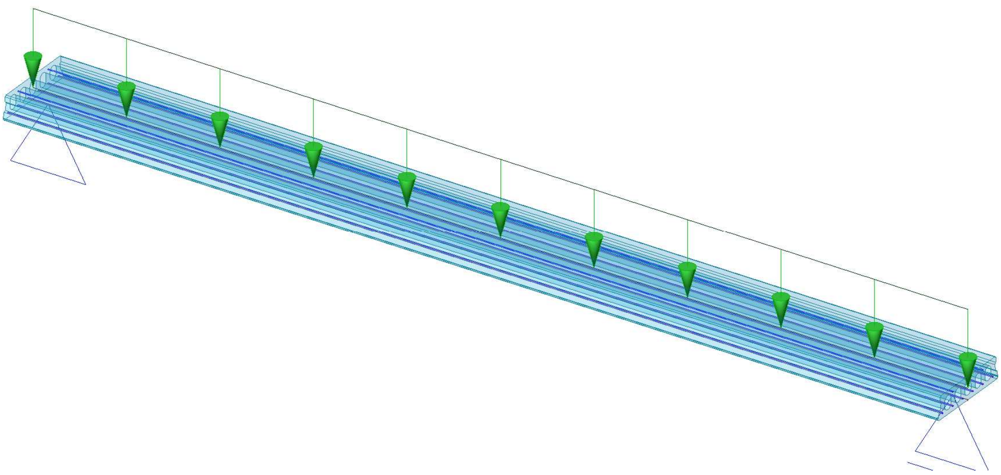
Stage 3
Stock
Load cases: empty2
Time: day 1.2
Relative humidity: 70%



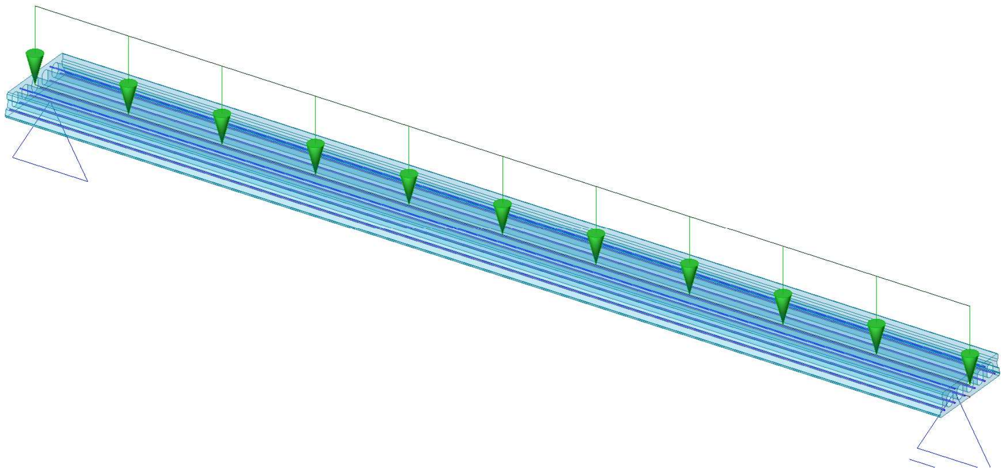
Stage 4
In situ, placing of slab
Load cases: q wall 0.6 kN/m
Time: day 28
Relative humidity: 70%



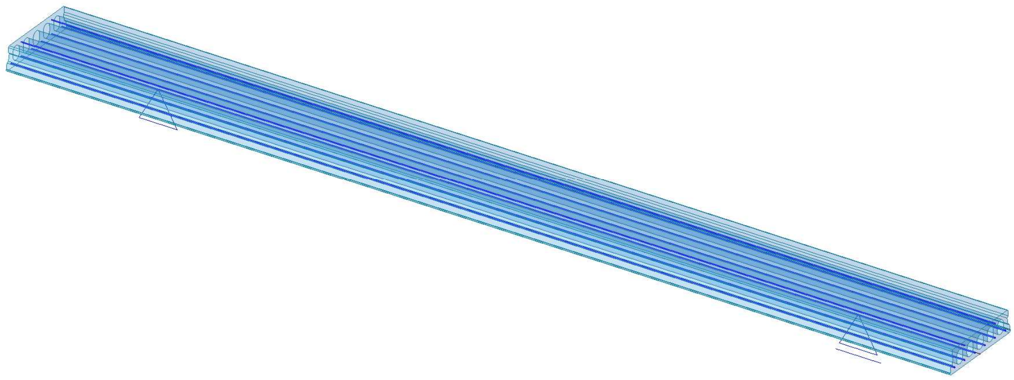
Fase 5
Casting of deck layer
Load cases: deck 1.2 kN/m
Time: day 50
Relative humidity: 70%



<u>Stage 6</u>
Service
Load cases: long term Var 1.2 kN/m
Time: day 100
Relative humidity: 70%



<u>Fase 7</u>
After 50 years
Load cases: empty3
Time: day 18000
Relative humidity: 70%



References

[1] Navrátil J.: Time-dependent Analysis of Concrete Frame Structures (in Czech), Stavebnicky casopis, 7 (40), 1992, pp. 429-451

[2] CEB-FIP Model Code 1990, Final Draft 1991, BULLETIN D'INFORMATION No 203, Comite Euro-International Du Beton, Lausane, 1990.

[3] Navrátil, J.: Prestressed concrete structures, Akademické nakladatelství CERM, s.r.o., Brno, 2004. (available at www.scia-online.com)

[4] Scia Engineer reference guide 2007.0.