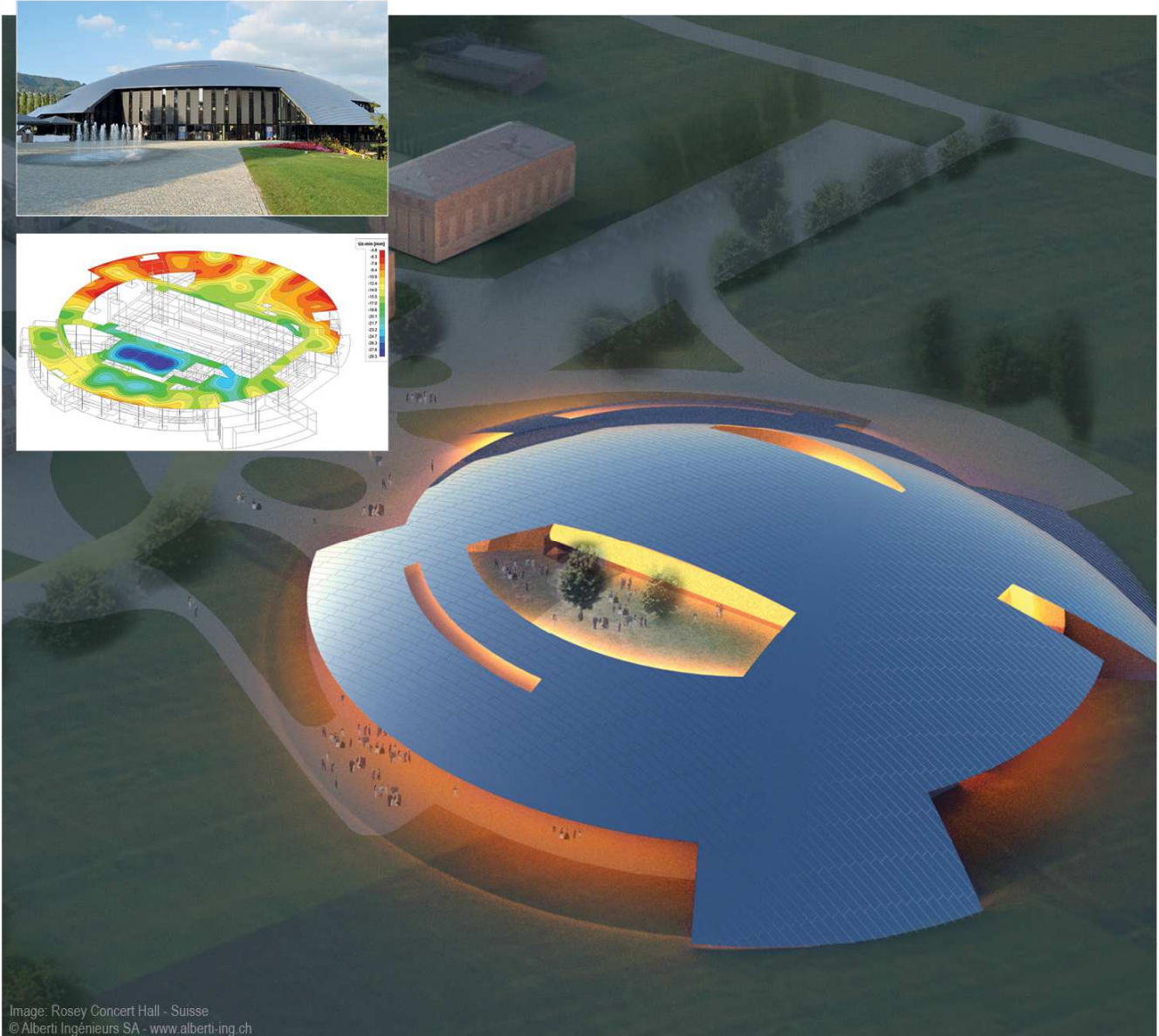


# SCIENGINEER



## Advanced Expert Training Construction stages

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2015 SCIA nv. All rights reserved.

---

## Table of contents

Introduction.....	5
Linear construction stages – Example: simple frame.....	6
1_Input geometry .....	6
2_Definition of the construction stages.....	7
3_Construction stage analysis .....	10
Linear construction stages – Example: phased cross-section.....	14
1_Input geometry .....	14
2_Definition of the construction stages.....	15
3_Construction stage analysis .....	17
Other types of construction stages.....	19



## Introduction

Construction stages are available in the **Expert Edition** and the **Precast Edition** of SCIA Engineer. But construction stage analysis in the Precast edition is restricted to 2D environment. Construction stage analysis in the Expert edition can also be done in general xyz environment.

Construction stages can be applied for all types of structures. It is supported for both 1D and 2D elements, concrete and steel or any other material. A linear construction stage analysis can be done in a general xyz environment (if license esas.01 is available (note: not in precast edition)).

In this training document only the linear construction stages are treated.

Also non linear construction stages are possible to analyze in SCIA Engineer.

There is also a time dependant analysis of construction stages, which can take into account the effect of creep and shrinkage (and long term relaxation of prestress reinforcement) between the stages. Examples of such a TDA calculation can be found in training document for prestress, but it is also possible to use TDA for reinforced concrete structures without prestress. TDA analysis however is limited to frame xz environment.

In this training document, the first chapter will treat the example of a simple frame structure which is usefull to explain how construction stages are created.

The second chapter will treat the example of a two-phased cross-section.

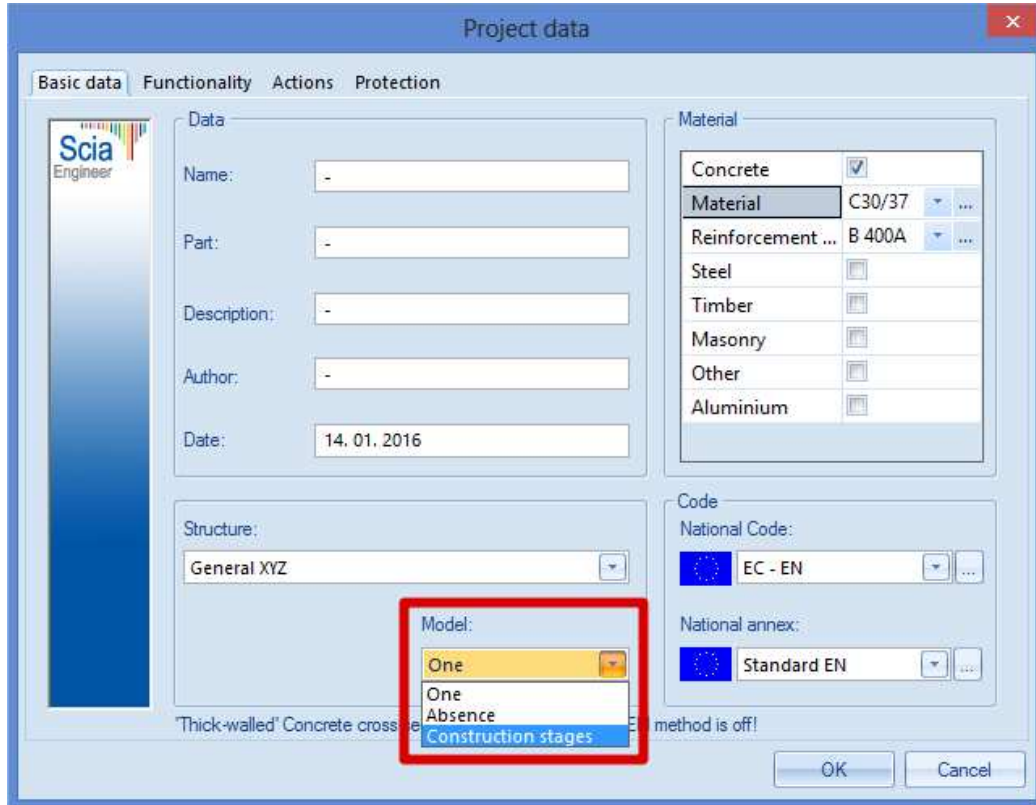
The third chapter will briefly explain the other types of construction stages.

## Linear construction stages – Example: simple frame

### 1\_Input geometry

- Project data:

In project data the model should be set to Construction stages:

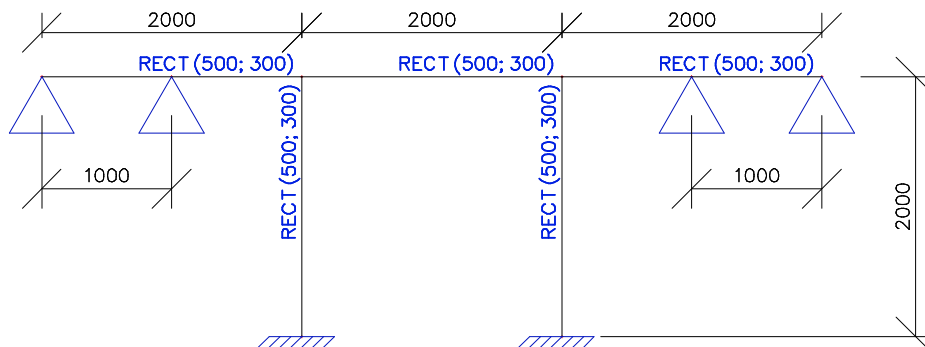


The model will be built up of construction stages, instead of one model.

This means more construction stages will be in one .esa file. In fact a separate analysis will be run for each stage, but the advantage of the use of construction stages is that the results can be summarized afterwards.

- The geometry should be modeled completely before starting to define the construction stages

The drawing below is showing the geometry of the example. Note that the beam elements should be 3 separate elements, because the middle beam and side beams will be added in different stages.



## 2\_Definition of the construction stages

- Before starting to define the different stages, it is important first to think through which stages should be created. Every stage should be given a clear name, which makes it easier afterwards for interpretation of the results. Especially when externals should be able to read the output.

In this project the following stages will be defined:

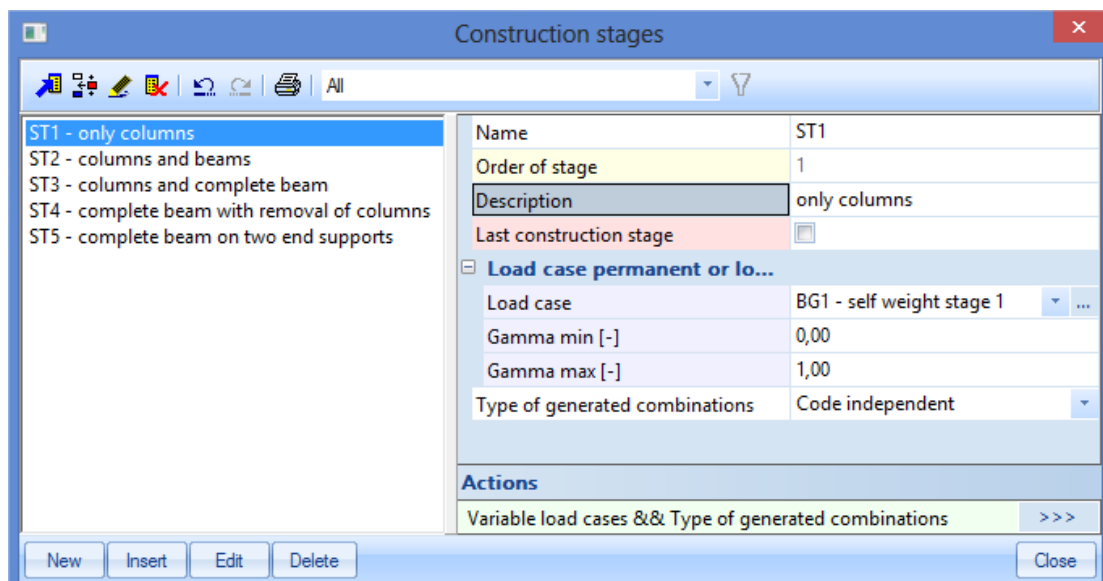
Stage 1: only columns  
 Stage 2: columns and middle beam  
 Stage 3: columns and full beam  
 Stage 4: complete beam without columns  
 Stage 5: complete beam on two supports

- The next step will consist of the creation of the load cases. It is necessary to link one permanent load case to every construction stage. Even if no loading is added in that stage. The reason is that the effect of the changes in a construction stage, is stored in that load case. Consider for instance a stage in which a temporary support or member is removed. This will cause effects on the rest of the structure and they are stored in this load case. This also means that every load case, can only be added once to a construction stage. Apart from permanent load cases (or long term variable load cases), the user can add multiple (short term) variable load cases to a certain stage, but the (short term) variable load cases will only be taken into account in that stage. For the envelope results on the next stage, these (short term) variable load cases will no longer be taken into account.

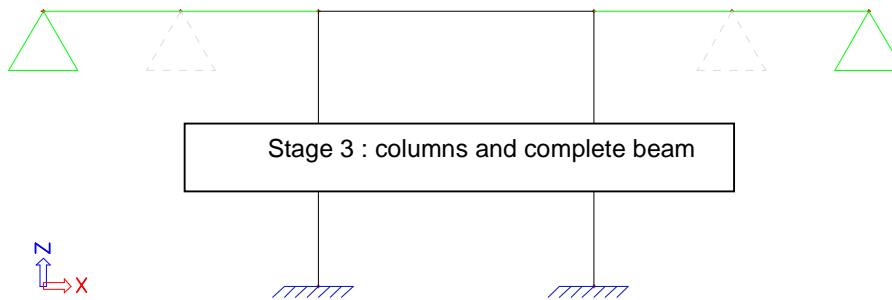
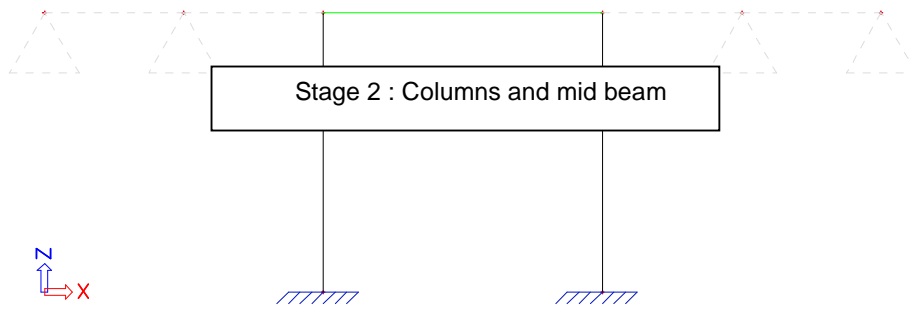
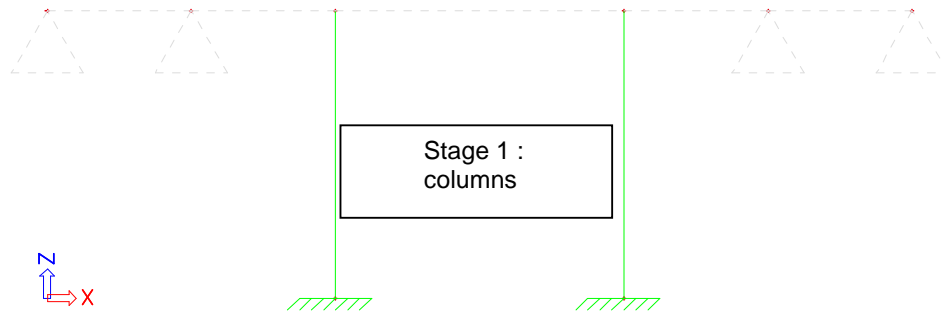
In this example, there will be no external loading. Only the self weight will be taken into account. 5 permanent load cases should be created. The first three should be of type self weight, because they will contain the self weight of the members that are added in that stage. A load case of type self weight will contain the self weight of the members added in that stage. The fourth and the fifth load case will be left empty (note that it will do no harm if they are defined as self weight, since in these stages, no new members will be added, so the self weight would be zero). As with the definition of the stages, it is advised to clearly fill in the description of every load case.

- In the definition of a construction stage, the name and description should be filled in. The load case(s) that appear in that stage should be assigned. For EN also the type of generated combinations could be chosen. In this example the generated combinations will be code independent (unfactorred) envelope combinations.

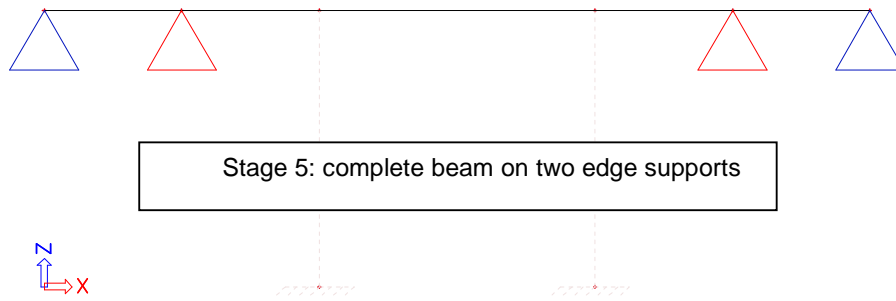
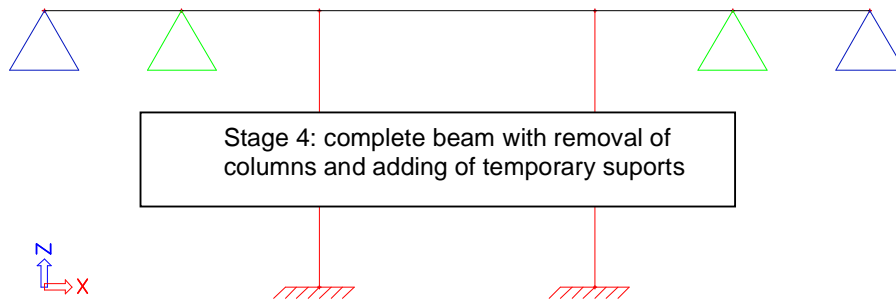
Below a print screen of the definition of the first stage is shown:



After definition of the stages, the user should indicate which members or supports shall be added or removed in every stage.







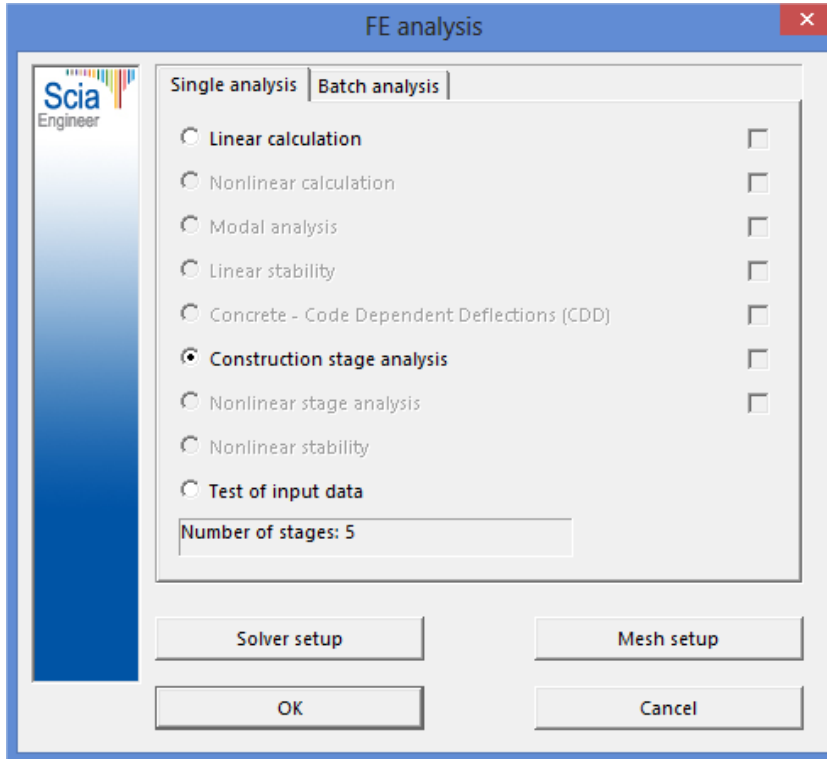
### 3\_Construction stage analysis

The results are available after running the construction stage analysis.

It is very important to run the construction stage analysis and not just the linear calculation.

If a linear calculation would be run, the structure is calculated as 'one model', so everything defined in the construction stages is ignored.

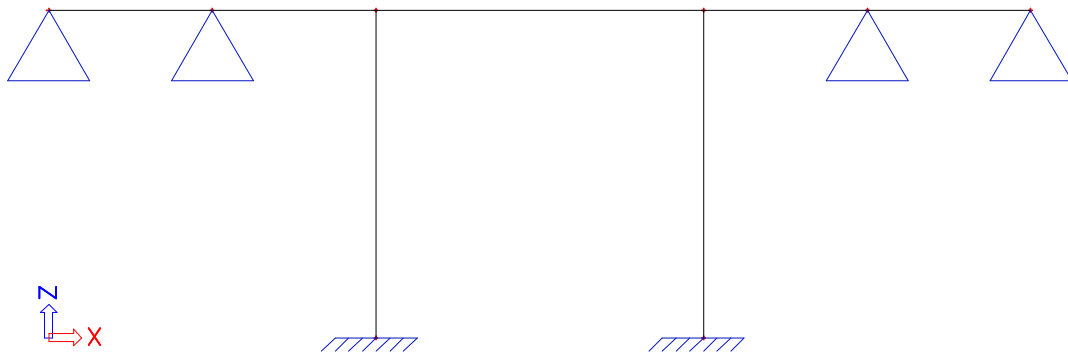
So pay attention to run the construction stage analysis:



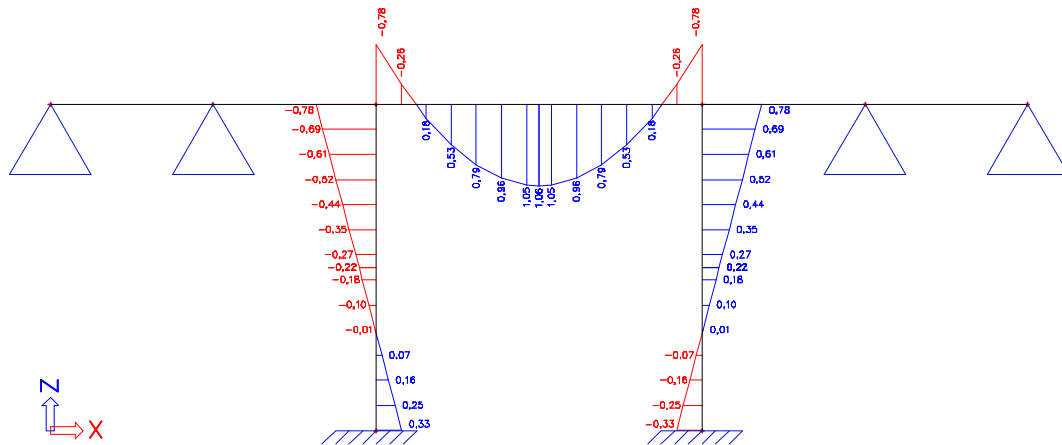
Next to the results for every load case individually, also the results for the summation of the different linear load cases can be viewed. For every stage, automatically an envelope combination and a class is created which contains the summation of the results in that stage including the history of all previous stages.

If we consider for instance the moment lines  $M_y$  for the first 3 load cases, we will have the following results.

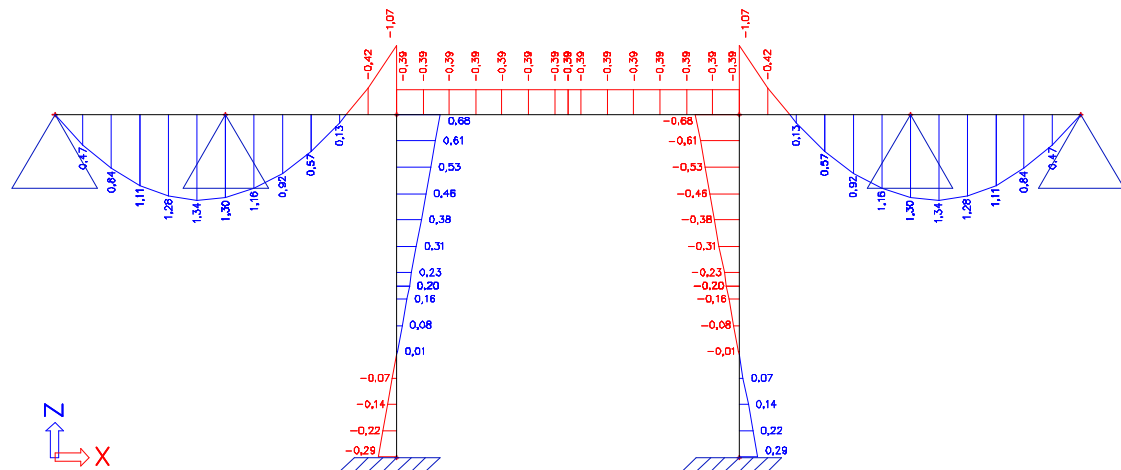
LC1: No moment, since only columns in this stage.



LC 2: the introduction of the beam element causes a moment in the beam, but also in the columns, because of the fixed connection.

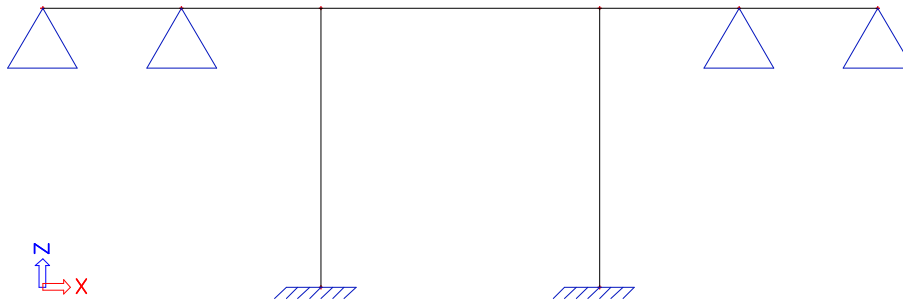


LC3: In this load case the side beams are added. Notice the extra moments. Since the end beams are connected rigidly with the existing frame, the moment in the end beams (1,07 kNm) will be equal to the sum of the moment in the middle beam (-0,39 kNm) and the column (-0,68 kNm)..

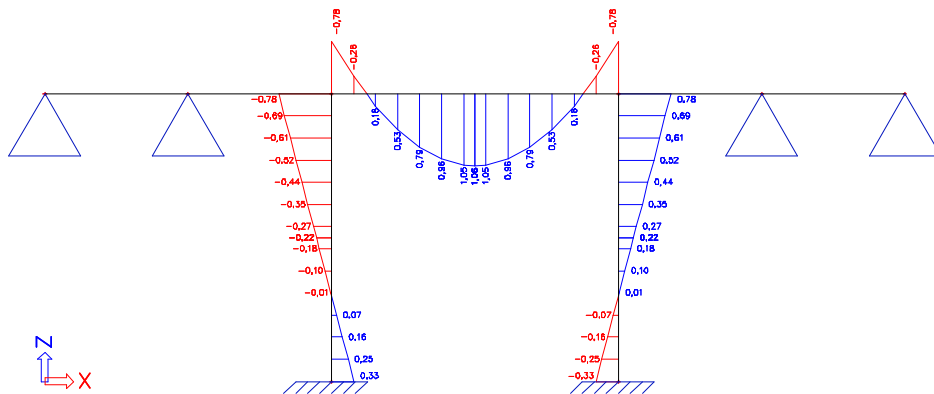


In the classes these results are summarized.

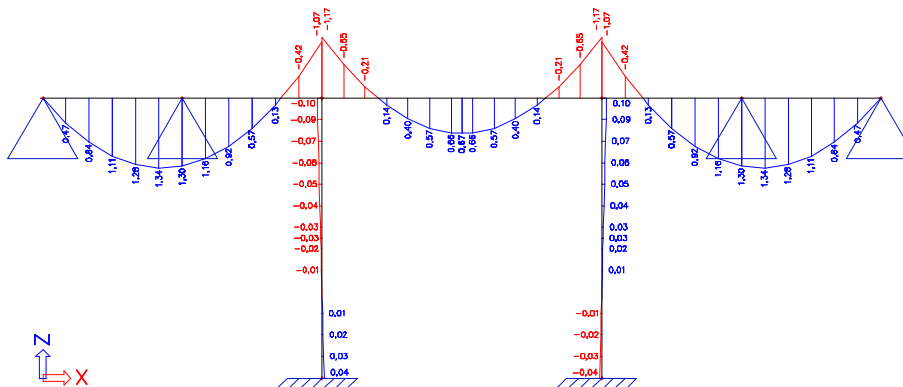
Class stage 1



Class stage 2



Class stage 3

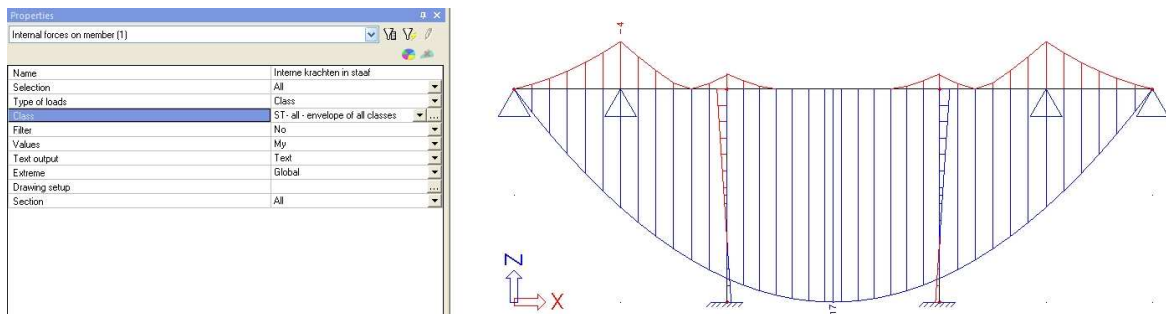


Note that the result that is given for class 3 is different from the result that would be obtained if the structure of stage 3 is modeled as 'one model'. Because of the construction stage analysis, the results from the previous stages are remembered and summarized. This leads to more exact results. For instance in stage 2 there are already internal forces and deformations in the structure. The effects of adding the end beams in stage 3 is just added to the existing forces of stage 2. This leads to more exact results as those that can be obtained in separate 'one model' .esa files.

This is of course a big advantage of the construction stage analysis. These effects become more important if the structure becomes larger. For instance in bridge construction where the different spans are constructed in different stages or in multi storey building design.

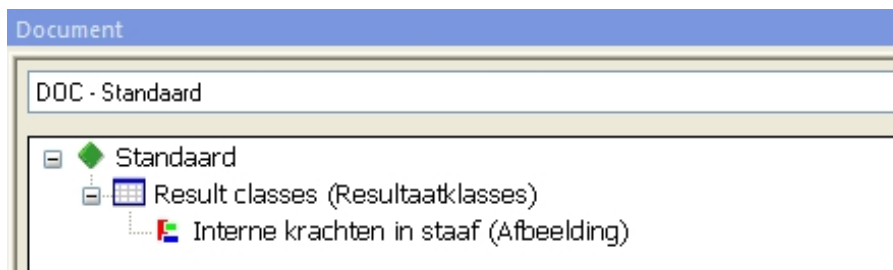
Another advantage is that all the results for the different stages are in the same .esa file. This means that an envelope class could be created from the envelopes of every stage. If the design is made based on this envelope, the user is sure to take into account all the internal forces that could occur in the previous stages. Note that the envelope class that is the envelope of all stages, is not created automatically and should be created by the user.

The result for the manually created envelope class of all stages is shown below:

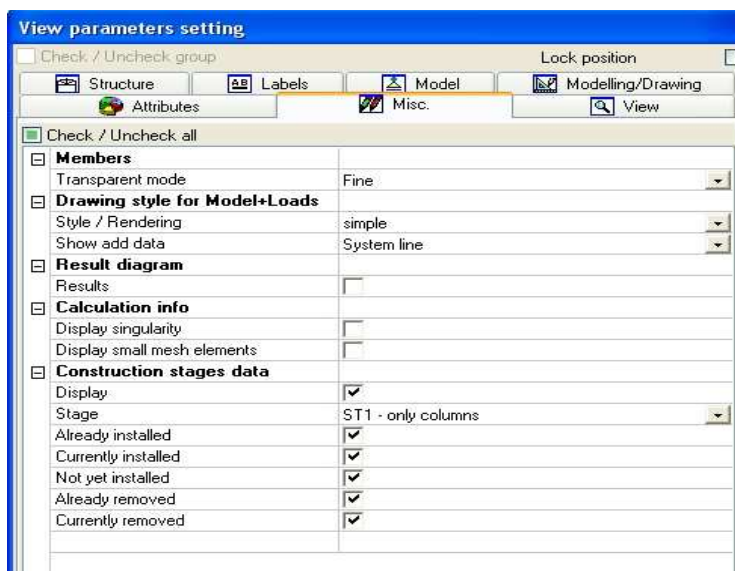


To conclude some extra tips and tricks concerning representation of results:

For clear and easy presentation of the output of this construction stage analysis, the user could use nested images in the document.



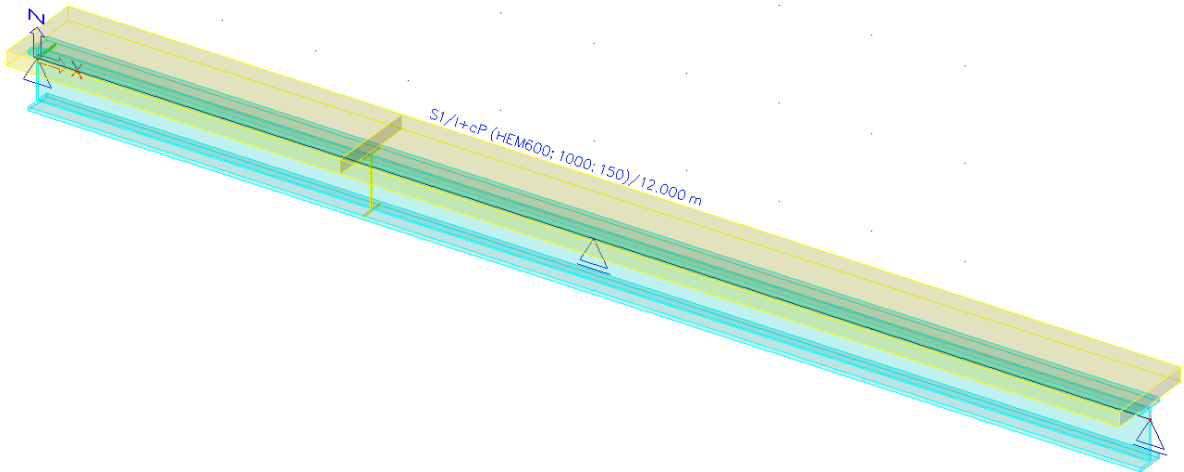
Moreover in the view parameters can be set to only show the current stage.



## Linear construction stages – Example: phased cross-section

### 1\_Input geometry

In this example a beam on two supports with a two-phased cross-section is modeled. In the middle of the beam, there will be a temporary support that will be used for temporary stutting of the beam while the cast concrete will dry and harden.

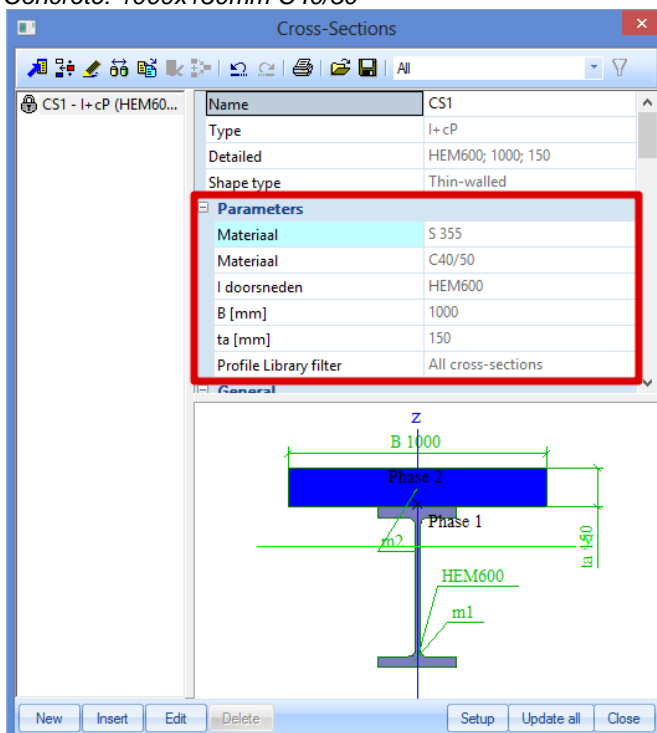


For the definition of the cross-section the user could create a general cross-section. Through the cross-section editor the user can create any shape of cross-section he would like.

In this example however a steel-concrete predefined shape is used. The properties are shown below

Steel: HEM600 S355

Concrete: 1000x150mm C40/50



## 2\_Definition of the construction stages

Stage 1: Steel beam on two supports

Stage 2: Casting of wet concrete and placing of extra support

Stage 3: Removal of extra support after hardening of concrete

LC 1: self weight of steel beam

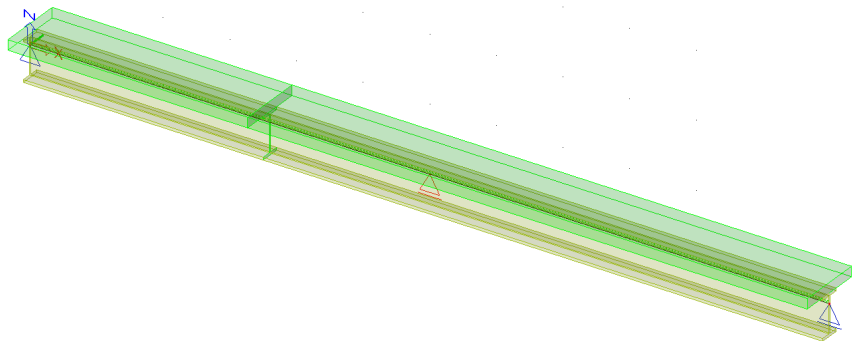
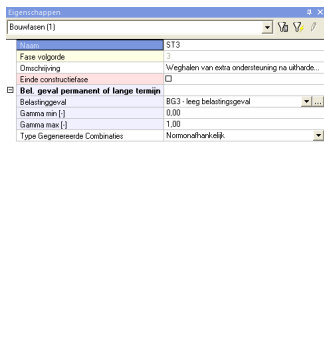
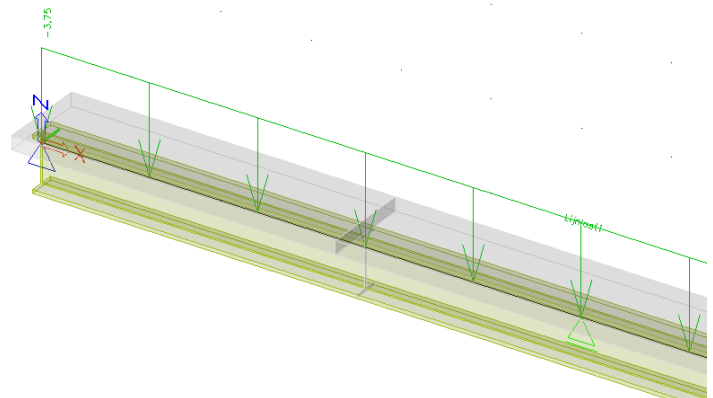
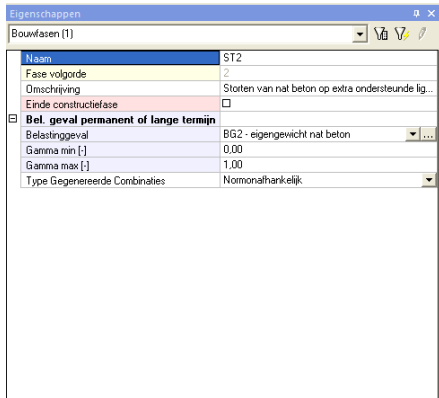
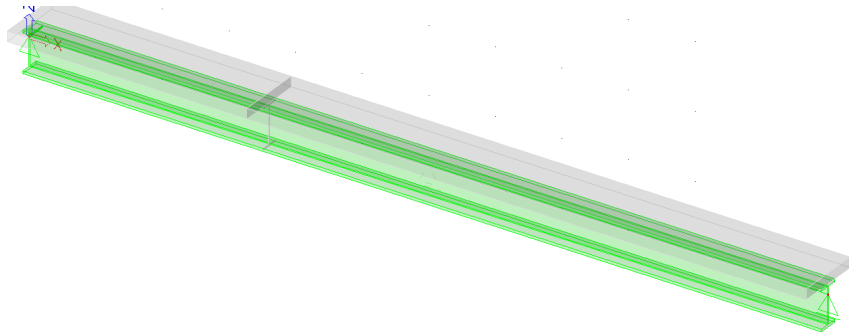
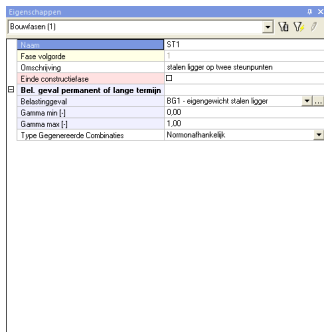
LC 2: weight of wet concrete (=3,75kN/m, to be entered manually!) (LC of type sw only works for 1<sup>st</sup> phase of css !)

LC 3: empty load case

Note that since this is a linear construction stage analysis, no time effects can be taken into account. That is why a member is always at full stiffness when added to the model.

Therefore the second phase of cross-section is only added in the 3<sup>rd</sup> stage, while its weight is already entered as a separate load case in stage 2.

The different stages are shown below:



The indication of the construction stage in which the phase change of the cross-section occurs is done in the properties of the member:

The screenshot shows the 'Properties' dialog box for a member. The 'Construction stages' section is expanded and highlighted with a red box. It contains the following data:

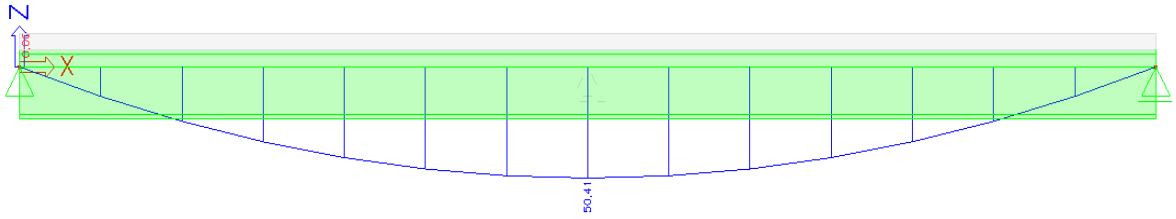
Construction stages	
Add	ST1 - steel beam on two support
C-s change 1->2	ST3 - Removal of extra support
Remove	No



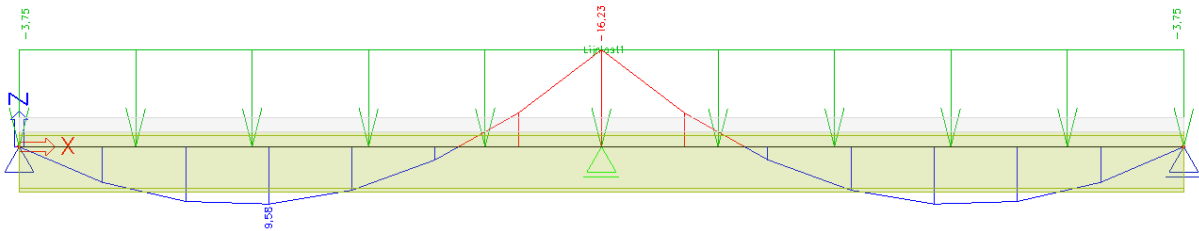
### 3\_Construction stage analysis

The moment lines  $M_y$  for the different stages are shown below:

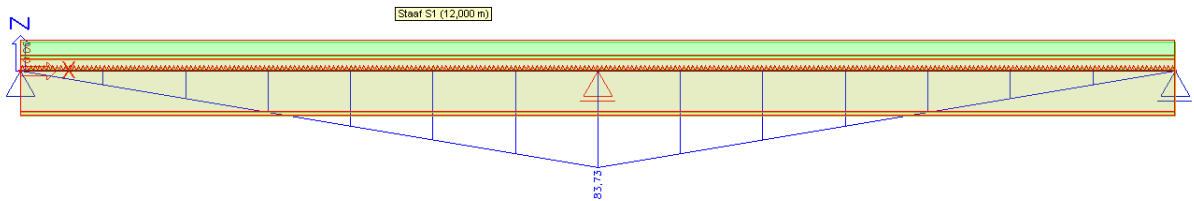
*LC1: self weight of steel beam*



*LC2: applying the weight of wet concrete on a beam which is extra supported in the middle*

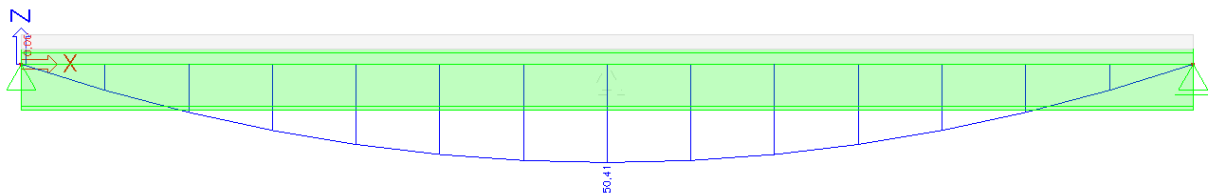


*BG3: effect of removal of the support (notice that this is the moment line corresponding to the application of the inverse of the reaction force of the removed support)*

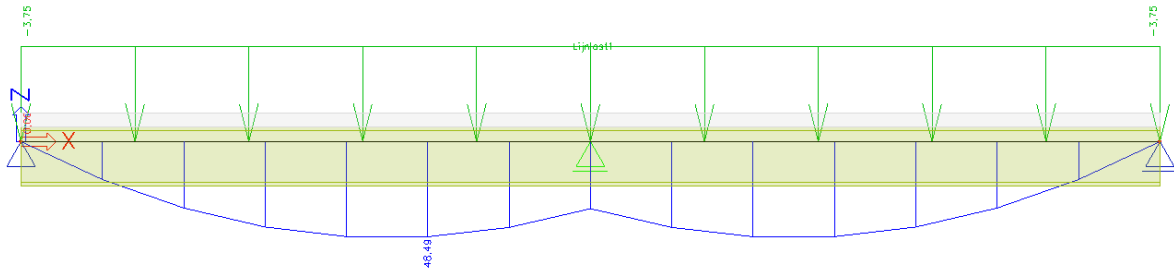


In the different stages, these results are summarized.

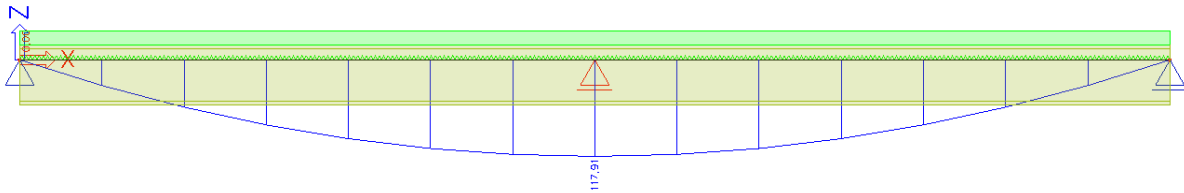
*Class stage 1: moment line of simple supported beam under its self weight*



**Class stage 2:** Clearly a superposition of both stages, and different result as calculated in a separate 'one model' esafire. The support is placed under the deformed stage of stage 1.



**Class stage 3:** By removing the support, the final moment line is that of a simply supported beam under self weight of both the steel (applied in stage 1) and the concrete (applied in stage 2).



Remark about deflection :

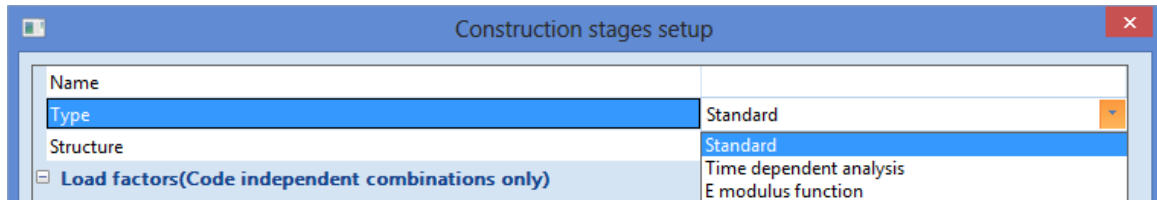
The deflection that would be given for class stage 3 is a summation of the deflection in the previous stages. The deflection of stage 1 is the deflection caused by the weight of the steel and calculated on the steel stiffness only. In stage 2 the deflection is limited because of the intermediate support. In stage 3 the support is removed and by this the deflection of the loading of stage 2, which is the weight of the wet concrete is calculated on the stiffness of the composite section.

So one could conclude that the final deflection is composed of the deflection by the weight of the steel on the steel stiffness only and the weight of the concrete on the composite stiffness.

It is clear that this effect can only be achieved by making use of construction stage analysis.

## Other types of construction stages

There are actually two other types of construction stages that can be chosen:



The standard type is the linear construction stage analysis which was treated in the two first examples in this course document.

The E-modulus function is in fact also a linear construction stage analysis. The only additional factor is that the E-modulus can be time dependant. For this also a global time should be assigned to each construction stage.

Consider example 2:

With the application of an E-modulus function the second phase of the cross-section could already be added in stage 2, but at the moment when it is added the E-modulus will still be very low. At the moment where the support is removed in stage 3, the E-modulus will have increased to maximum value.

The E-modulus function should be entered manually in the material properties.

It is still linear analysis, with an extra parameter to vary the E-modulus between stages. Since it is actually linear construction stage analysis, this type can also be used in general xyz environment.

The Time Dependant Analysis (TDA) is more complex and can only be run in frame xz environment.

The TDA solver uses a numerical procedure to calculate the losses that will occur between different stages. So it is not only a varying stiffness, but also real losses will be calculated between the stages. These losses can be due to a mix of shrinkage, creep and long term relaxation of the prestress steel. The TDA solver will create extra load cases of type F-creep to store the effect of the losses between the stages. At the end of TDA analysis also envelope combinations and classes are created that contain the user defined load cases together with these F-creep load cases that contain the losses that appear between different stages.

For an example of TDA analysis reference is made to the training document on prestress.