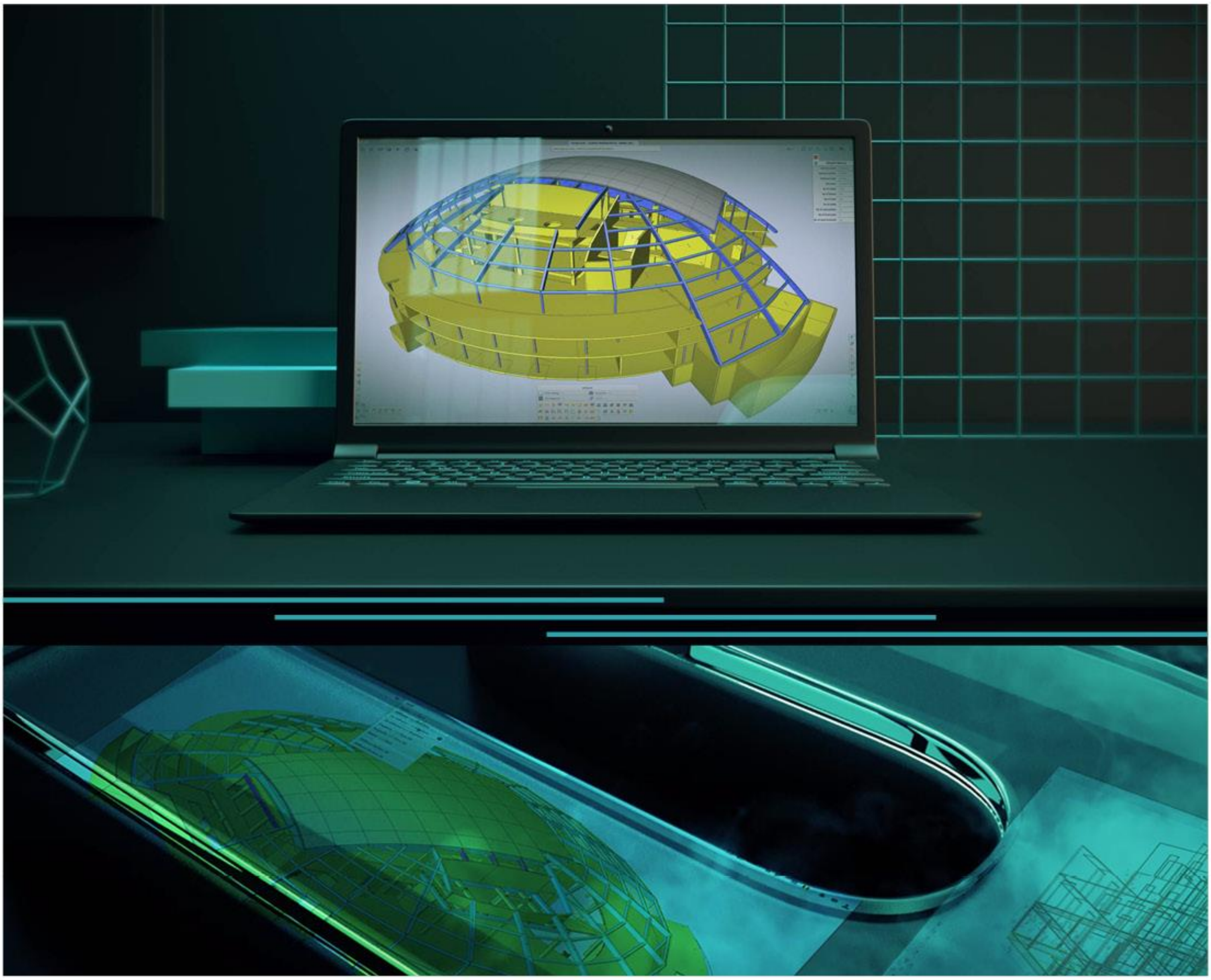
SCIA ENGINEER



TUTORIAL

INTEGRATION MEMBER

Table of Contents

Table of Contents **3**

Introduction **4**

Input and use the integration member **5**

- **Input integration member** **5**
- **Extra parameters** **6**
  - Integration relative to position of 6
  - Number of sections 6
  - Selection of members for integration 6
  - LCS 7
- **Results of the integration member** **7**

Design and check of an integration member **8**

- **Settings of the integration member** **8**
- **Example** **9**

Introduction

The integration member is a tool that enables integration of results from one or more 2D and 1D members and their subsequent presentation in the form of the internal forces and deformations of 1D members. Meaning that it enables the presentation of a set of 7 components of 1D internal forces: N, V_y, V_z, M_x, M_y, M_z, V_r and a set of 6 components of 1D deformation: u_x, u_y, u_z φ_x, φ_y, φ_z, U_{total}
For concrete applications – for example: lintels, deep beams, beam slab, core wall, etc. – you can use this representation to design the reinforcement and check the element.

This tutorial will explain how to input the integration member, explain its properties and how to get the results. It will also discuss an example how you can design and check an integration member used in concrete applications.

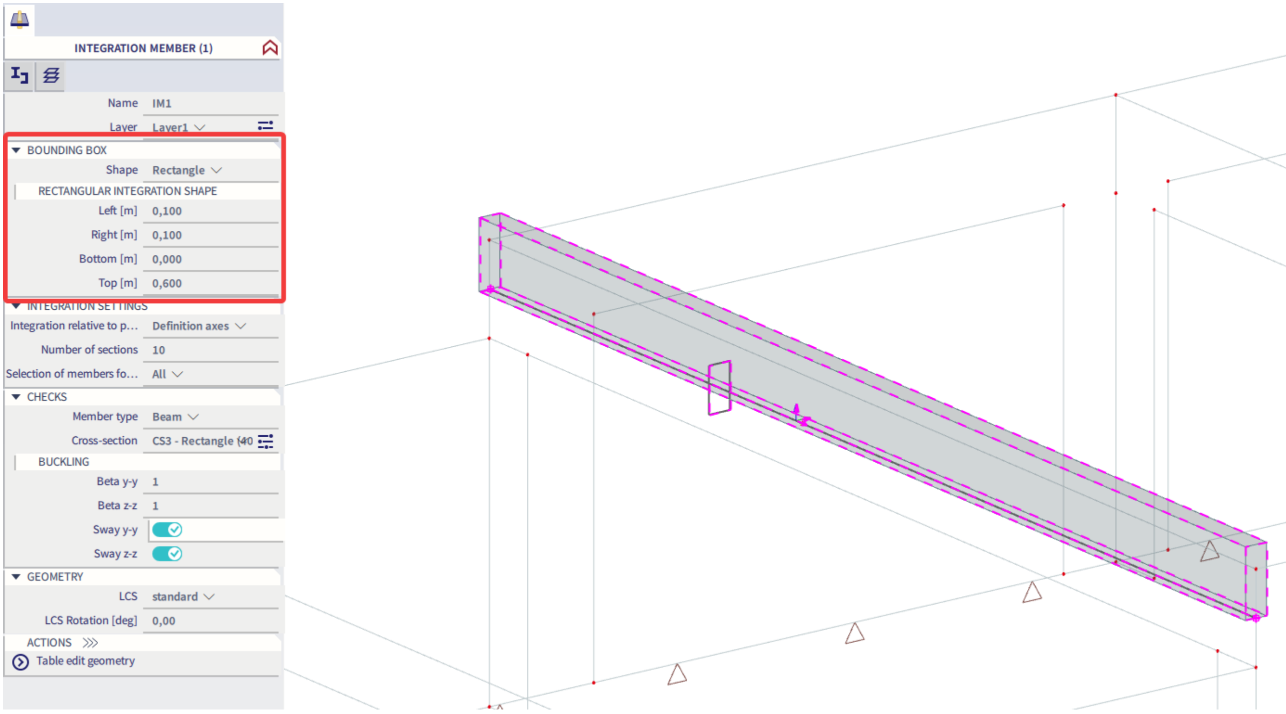
Input and use the integration member

Input integration member

To input an integration member, you can simply type this into the SCIA Spotlight:

Another way is to find the functionality in the results toolbar:

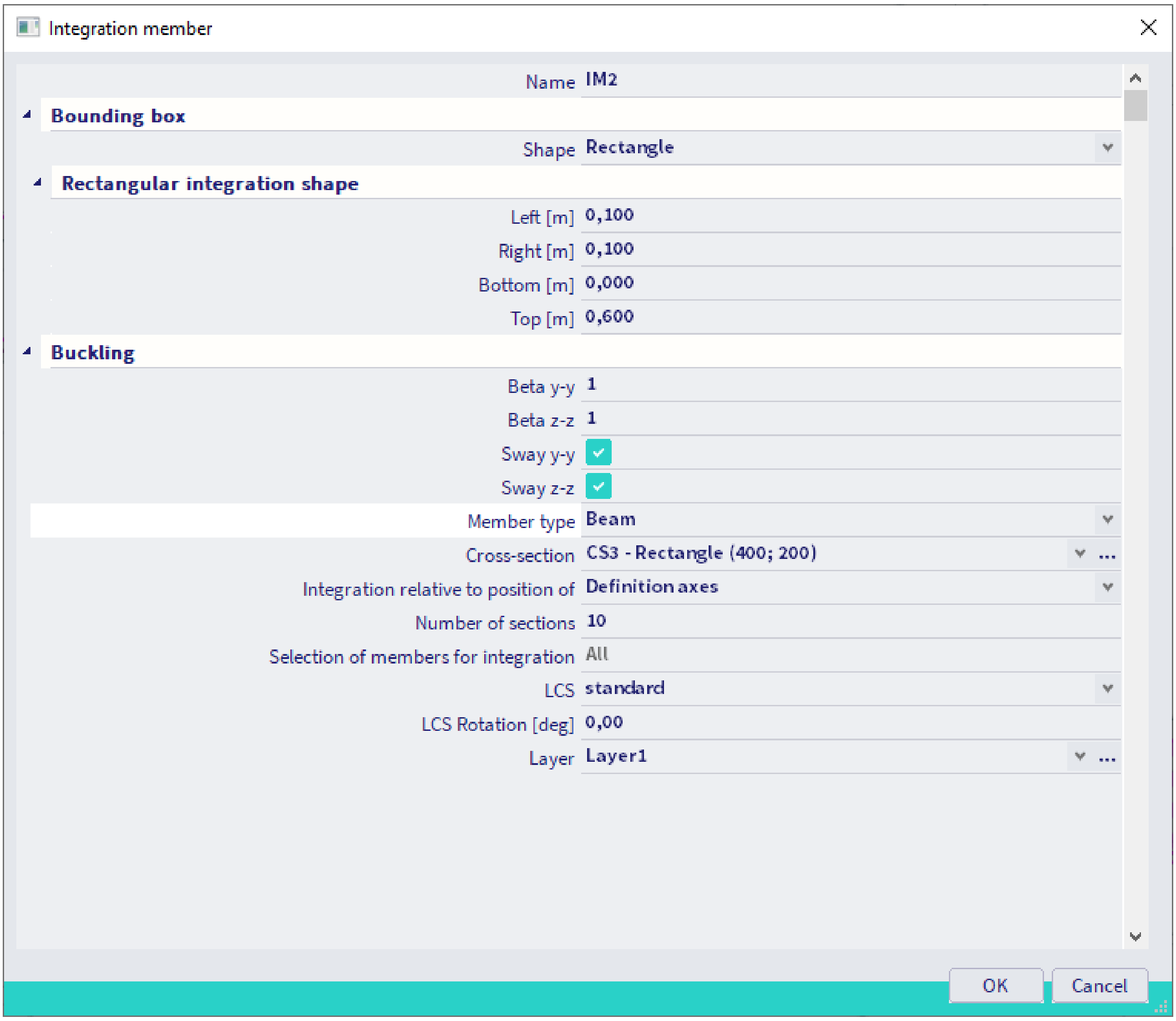
You can input the integration member simply by clicking on the start and end point of the integration member. Next you need to check the bounding box. This bounding box defines which entities are considered in the integration:



Note that you have several options for the shape of the bounding box. You can use this to your advantages and use the optimal shape for your projects.

Extra parameters

After inputting the integration member, you can edit some other parameters to ensure the integration is correct:



Integration relative to position of

This parameter has two options:

- Center of gravity: This is the default option. With this option, the theoretical center of gravity is detected in each section. Integration is then referred to the center of gravity.
- Definition axes: This option refers to the definition axes. In cases where the position of definition axes differs from the position of the theoretical center of gravity, values of bending moments M_y and M_z are influenced by normal force acting on eccentricity.

Number of sections

This parameter defines on how many sections evenly spaced along the length of the integration member results will be integrated and presented. It is recommended to check the accuracy of the results. This means you might need to adjust the number of sections.

Selection of members for integration

This parameter has two options:

- All: With this option, internal forces are integrated from all members that are fully or at least partly inside the bounding box.

- User defined: This option allows you to manually define a set of members that will be excluded from the integration. In case this option is selected, additional action buttons "Exclude members from integration" appears in the properties. That may be used for the definition of a set of excluded members.

Remark: in case 1D or 2D members are only partly inside of the bounding box, the results are integrated only from the part of such members that are inside the bounding box.

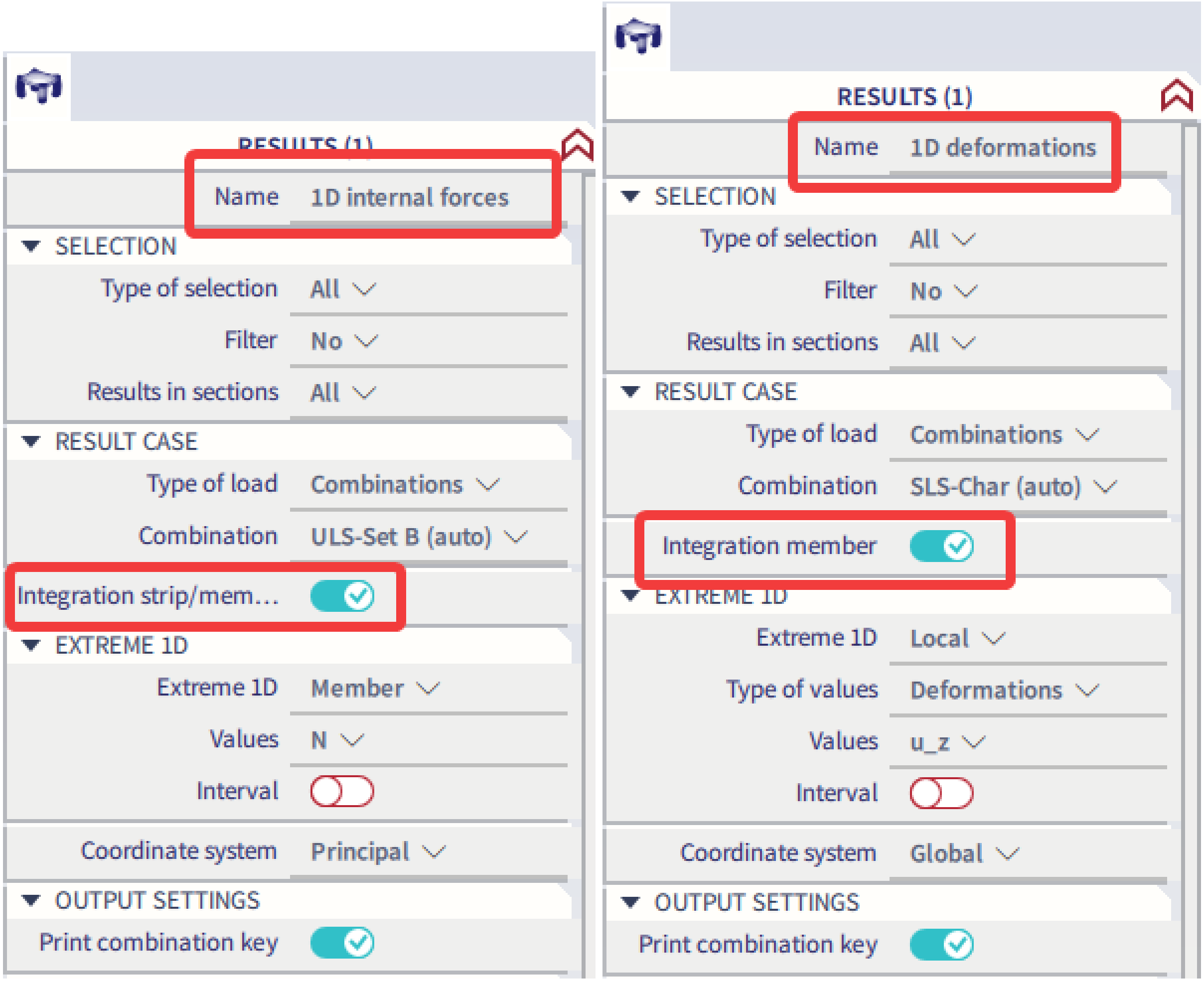
LCS

This property specifies the way the local axes of the integration member are determined.
You can use LCS Rotation to define a rotation of the local axes of the integration member. The rotation is measured around the integration members longitudinal axis, i.e.X-axis.

Note that there are still some other parameters. These have a function when designing the reinforcement and will be discussed further on.

Results of the integration member

The results of the integration member are displayed as 1D member results. It is necessary to tick the checkbox "Integration strip/member" to see results on the integration member:



<u>1D internal forces</u>: Internal forces are integrated from all 1D and 2D members within the Bounding box. In case any object is only partly inside the Bounding box, only internal forces from that part which is inside the Bounding box are integrated.
<u>1D deflection</u>: For each section of the integration member, the average value of deflection of all members within the Bounding box is determined.

Design and check of an integration member

When modelling concrete structures, the integration member can be a useful tool for applications such as lintels, deep beams, beam slabs, core walls, etc. The modelling can be done as a 2D member, but by using the integration member you can get the results as a 1D member. On top of that you can design and check the reinforcement for these types of members.

Settings of the integration member

It is necessary to define some extra properties that will be used for the design and check of the integration member:

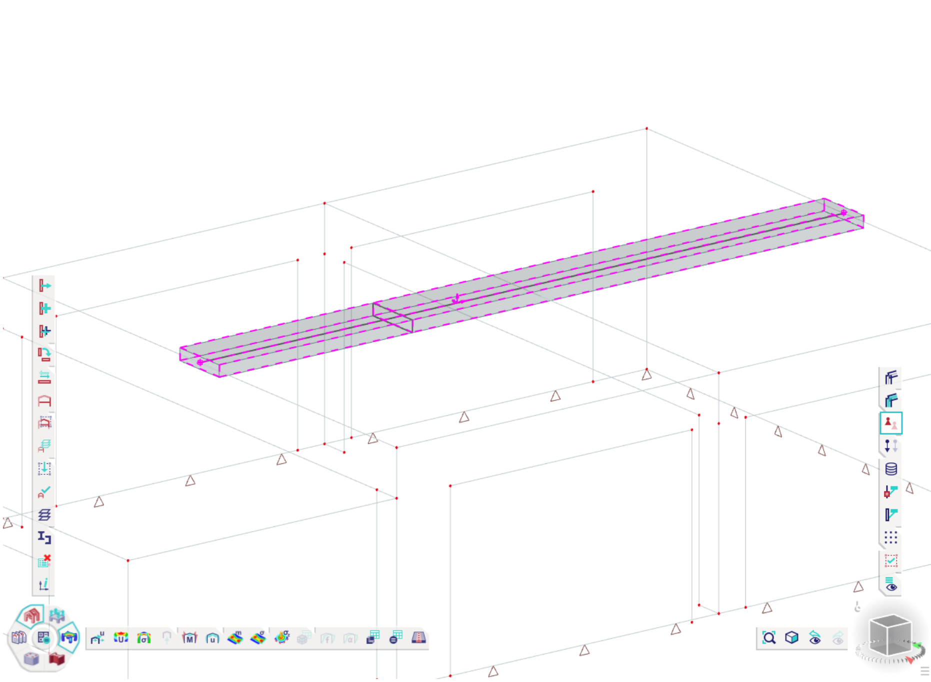
- Group Buckling: This group contains properties which are necessary for the buckling calculation:
    - Beta y-y: definition buckling coefficient around the y-axis of the integration member. Default value is 1.0.
    - Beta z-z: definition buckling coefficient around the z-axis of the integration member. Default value is 1.0.
    - Sway y-y: definition if integration member is prone to sway (unbraced) around y-axis of the integration member. Default vaulue is loading from Setup according to type of material of integration member.
    - Sway z-z: definition if integration member is prone to sway (unbraced) around z-axis of the integration member. Default value is loading from Setup according to type of material of integration member.

- Member type: Definition of the type of integration member. Three options are supported:
    - Beam
    - Column
    - Beam slab

- Cross-section: This property specifies which cross-section will be used for the design and check of integration members. The cross-section is presented graphically in the 3D window, if Draw cross-section is ON in View parameter setting.

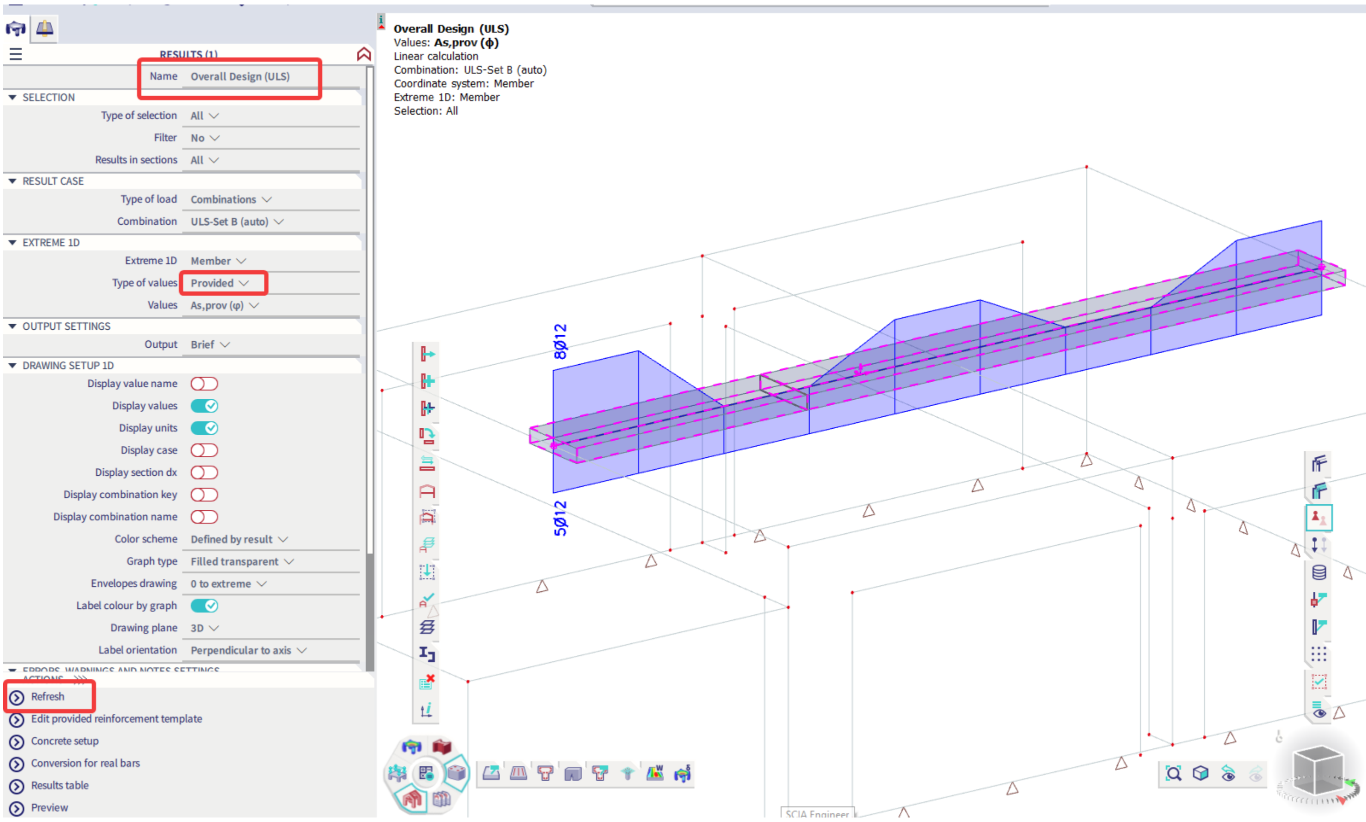
Example

In this example we would like to design and check a beam slab. We start by inputting the integration member and setting the correct bounding box. We use a total width of 1m (0.5m left and 0.5m right) and a thickness of 0.2m (0.1m top and 0.1m bottom):

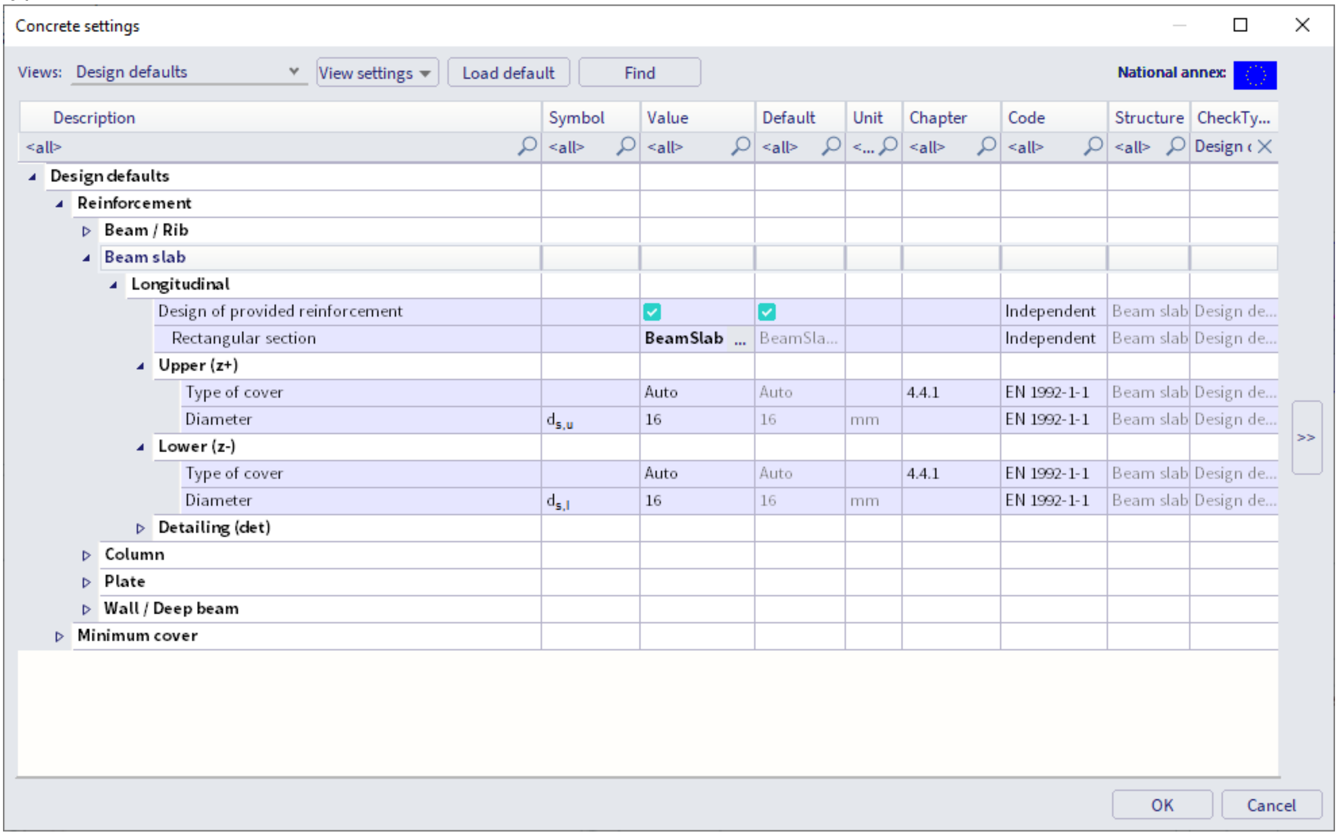


Next, we set the correct type of element, i.e. Beam slab, and choose the same cross-section. We also choose that the integration will be done according to the definition axes:

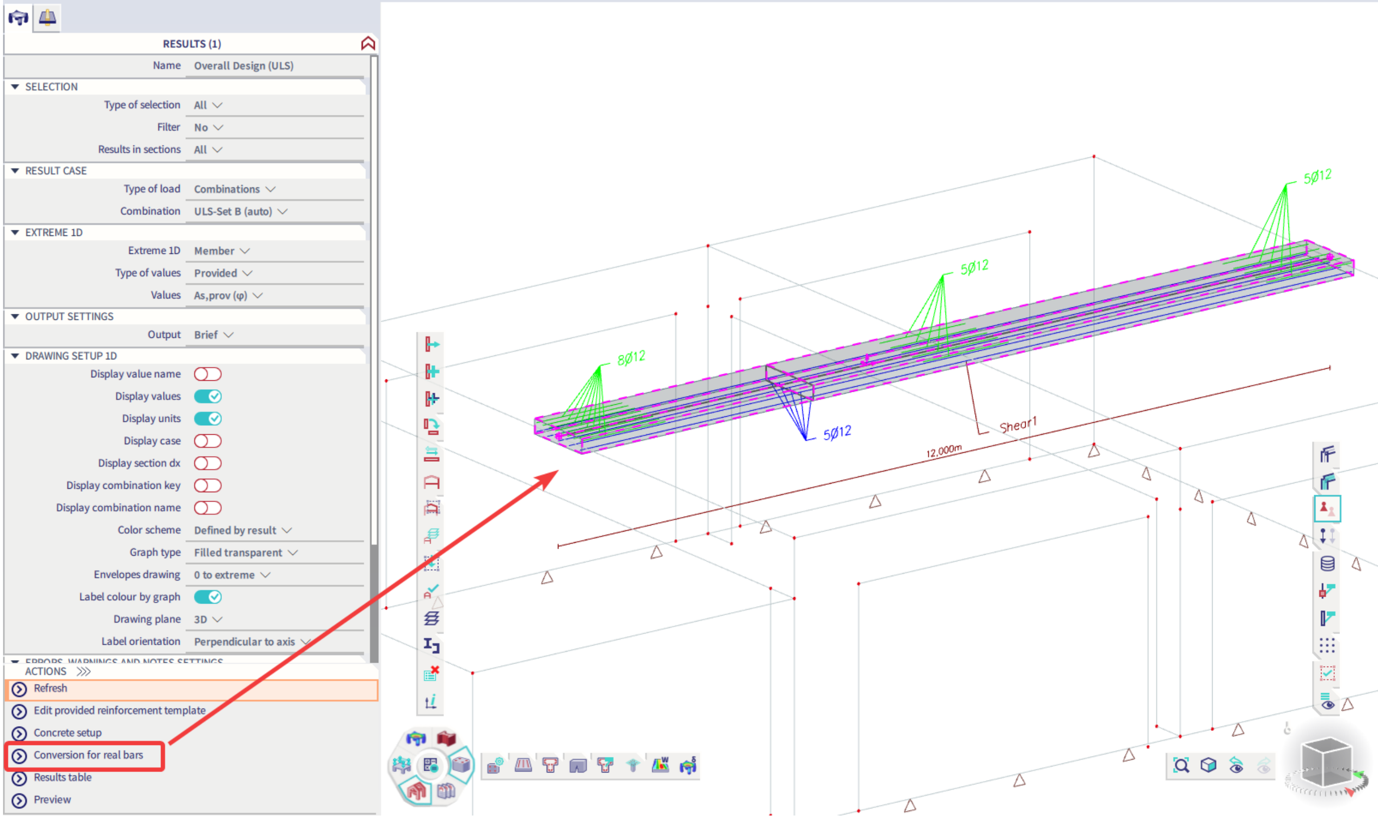
The next step is to check the 1D internal forces. If necessary, you can adjust the number of sections and the selected members for the integration.
Once the 1D internal forces are calculated, we can use these to design and check the reinforcement.
Simply go to the 1D Reinforcement design and check the provided reinforcement:



Note that you can change the template of the provided reinforcement in the concrete setup according to the type of member:



The following step is to convert the provided reinforcement:



Once the provided reinforcement is converted to practical reinforcement, you can use this reinforcement to perform the necessary concrete checks.