NEMETSCHEK Scia

Canopy of Oman Cultural Complex
Setec Batiment © Architecture Studio

Scia Engineer

Concrete for Starters

Scia Engineer

Table of contents

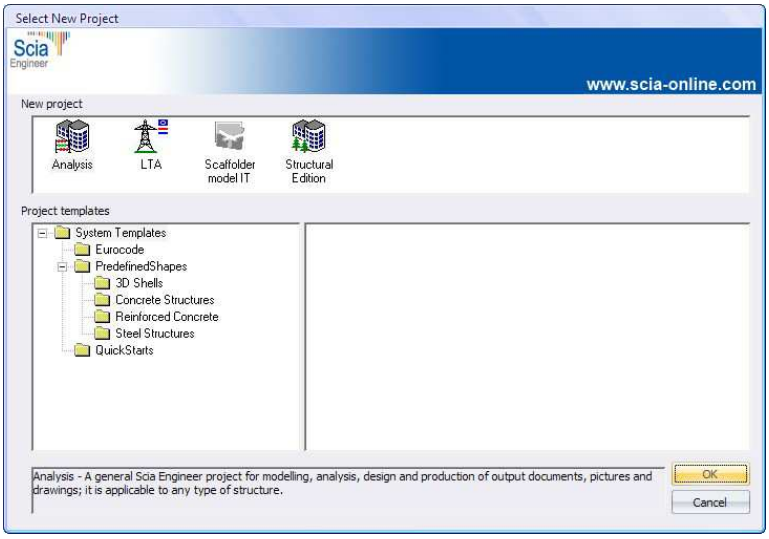
Note: this manual is created with version type *2012.0.116* of Scia Engineer.

1D Beam

Starting a project

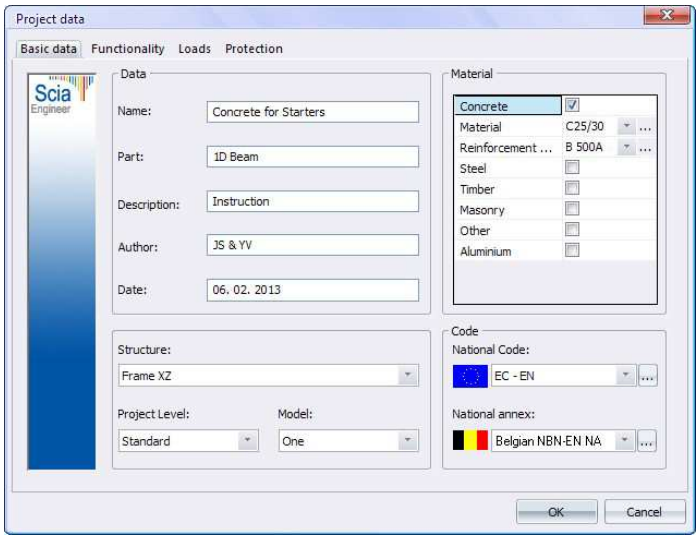
Starting a new project

To start a new project, one has to click on **New**  , which can be found in the toolbar. A new project will open in a new screen, namely **Select New Project**. Because calculations will have to be made, choose for the option **Analysis**.



Next the dialog box **Project data** will appear.

Entering the project data



In this dialog box, the general data of the project has to be entered (e.g. name, author, etc.). As construction type, choose for the option **Frame XZ**, this leads to a

structure of which all nodes are fixed. Consequently, the user will be able to introduce forces on the elements themselves (**Truss XZ** implies structures with hinged nodes, where forces can only be applied in these nodes).

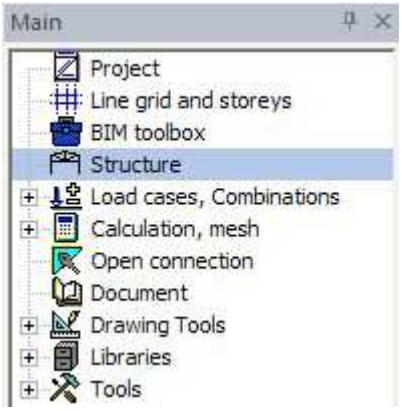
Additionally, the user can further specify the used materials. In this example is opted for **Concrete C25/30** and **Reinforcement steel, type B 500A**.

Finally, the calculations will be executed according to **NBN-EN NA**. Click on the **OK button** to confirm the project data.

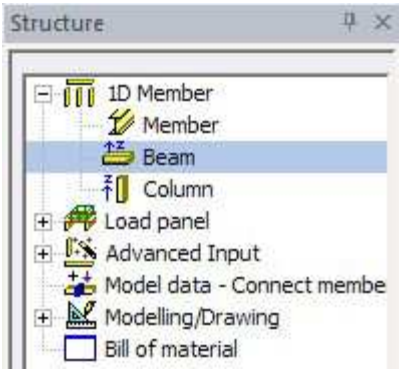
Defining the construction

The 1D-beam

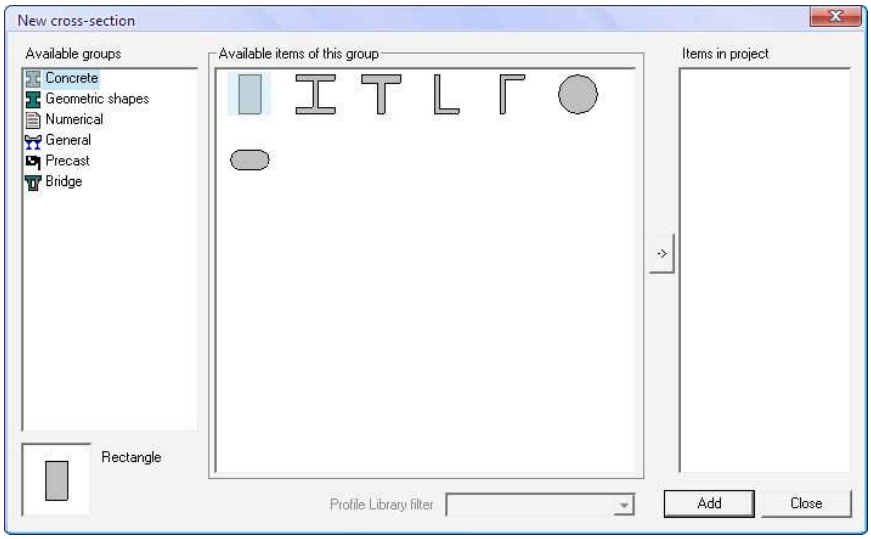
To determine the geometry of the structure, double-click on **Structure** in the **Main menu** option.



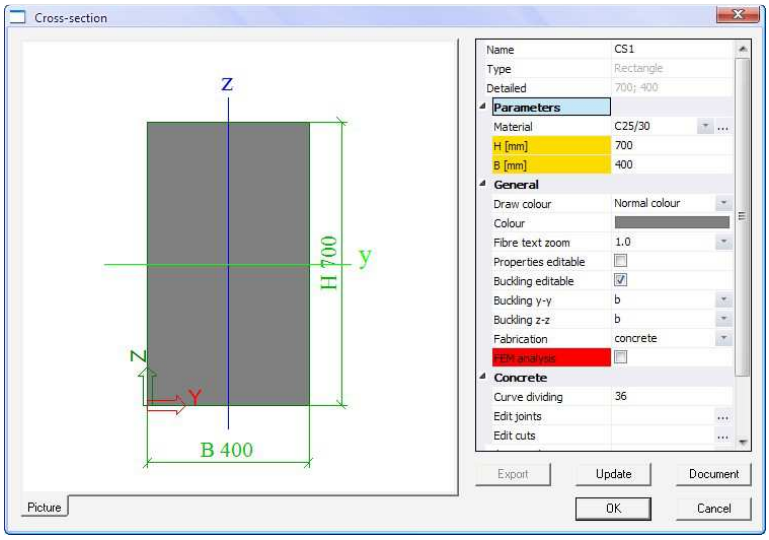
The selected menu will open. In this project, there is chosen for a 1D beam. This can be done through the submenu **1D member > Beam**.



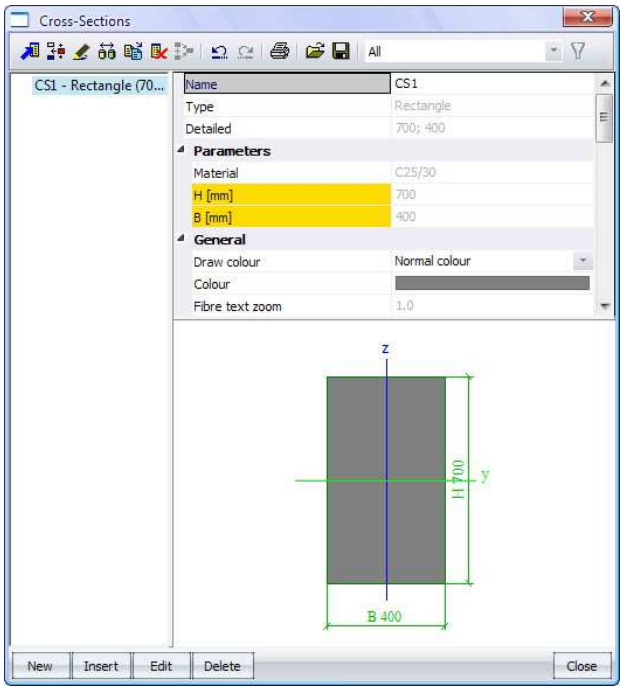
Because there is no section that has been defined yet, the window **New cross-section** will open up automatically. The beam in this project has a rectangular cross-section. To create this cross-section, one has to select **Rectangle** and click the **Add button**.



Now the dimensions of this cross-section have to be entered. Choose the following input parameters:

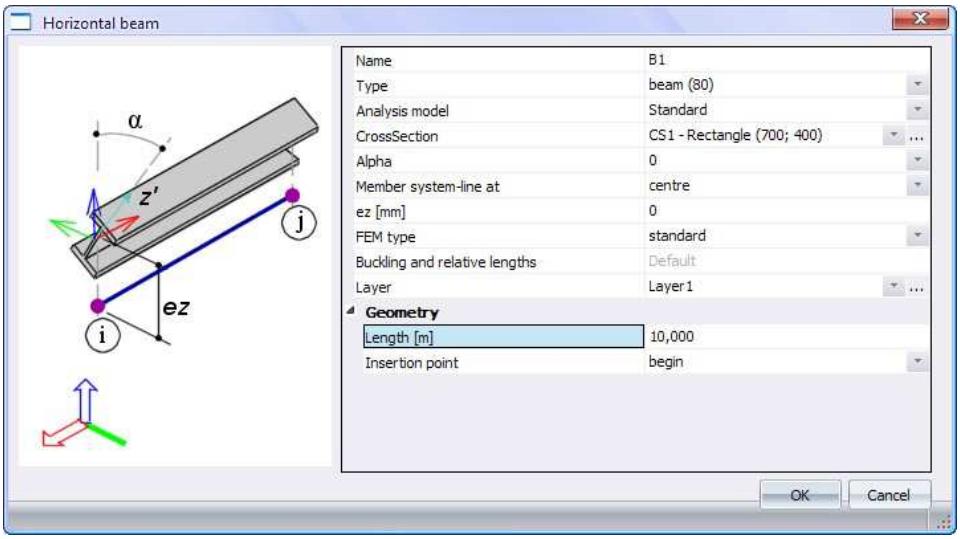


After entering the data, click on the **OK button**. As a result, this cross-section will become available for the user in the **Cross-section window**.

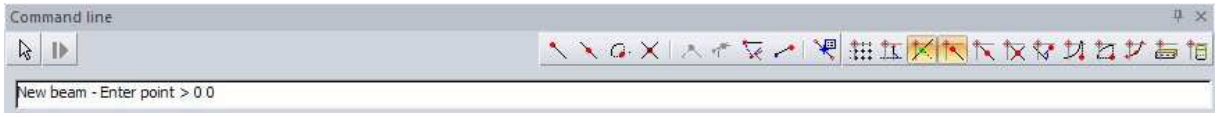


Then click **Close**.

Now it is possible to adjust the specifications of the beam. In this case, one has to choose for a length of **10 meters**.



Confirm the input by clicking the **OK button**. In the command line appears **New beam – Enter point**. The insertion point here, should be the origin of the GCS (Global Coordinate System), this can be done by entering the x,z-coordinates as follows:



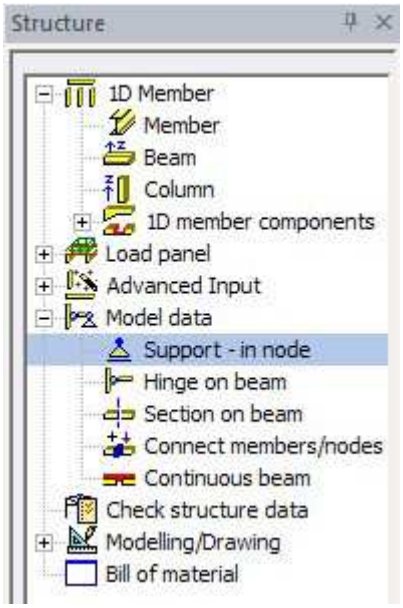
The coordinates can also be entered, separated by a semicolon (;). To complete the input, click the **Esc key**. Finally, to deselect the beam, click the **Esc key** again.

The beam will be displayed as follows:

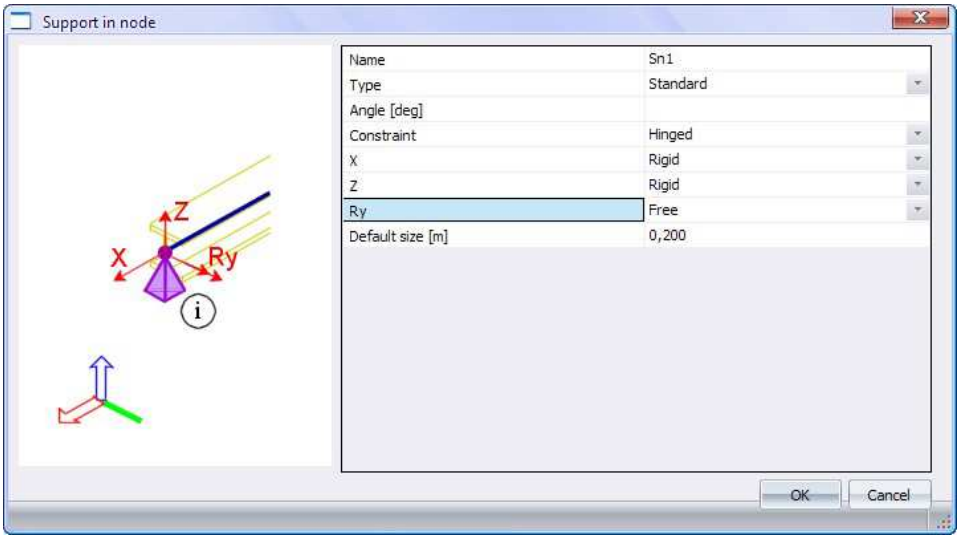
Remark: through the button, other cross sections can be defined.

Adding the supports

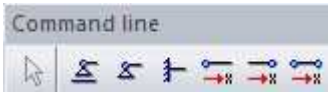
Now the supports of the structure have to be added. This is done by using the option **Model data** > **Support – in node** in the **Structure menu** that is already active.



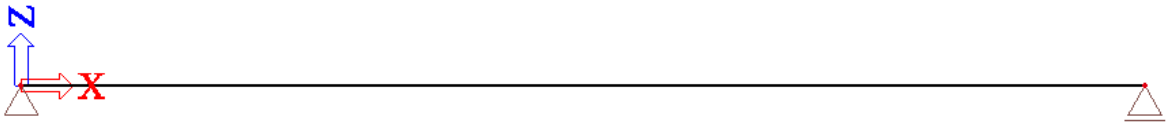
In the dialog box **Support in node**, one can adjust the details of the supports. At the leftmost point of the beam, the beam is supported by a hinge, so the following entry must be confirmed with the **OK button**.



Select the appropriate node to add the previously defined support. Then deselect by double-clicking the Esc key twice. To add a sliding support at the right most point of the beam, use the same method (use other parameters) or click on the icon **Sliding support**.



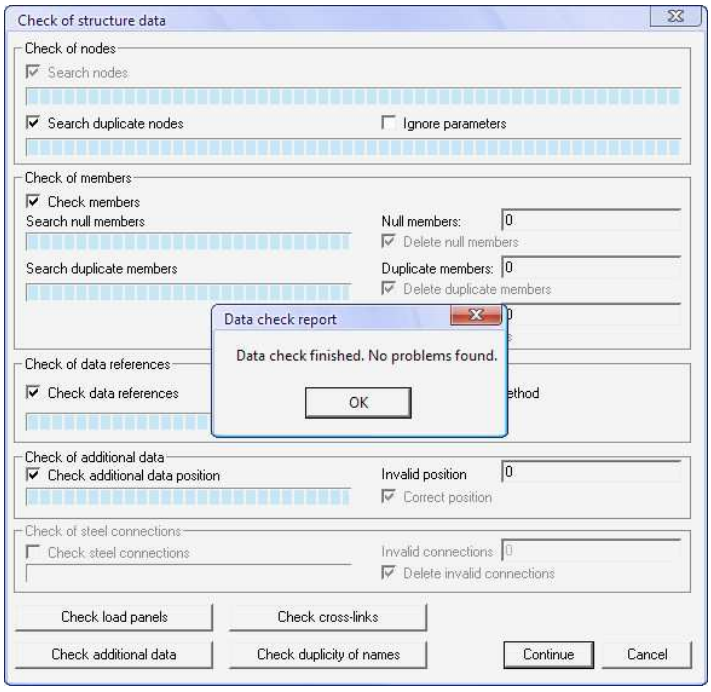
Finally, the following output will we obtained:



Remark: With the buttons, located above the **Command line**, the representation of the structure can be adjusted.

Checking the construction

In order to check the construction, one can use the **Check structure data button**  . By clicking the **Check button** in the dialog box **Check of structure data**, the construction check will be executed. When there are no problems to be found, the following output will appear:



The structure check can be closed, through the **OK button**. Next click the **Close button** in the submenu Construction to return to the Main menu. It is also possible to use the following icon  .

Applying the loads

Creating the load cases

The loads can always be divided into permanent and imposed loads. In this project there are 3 permanent loads and 1 imposed load, these are:

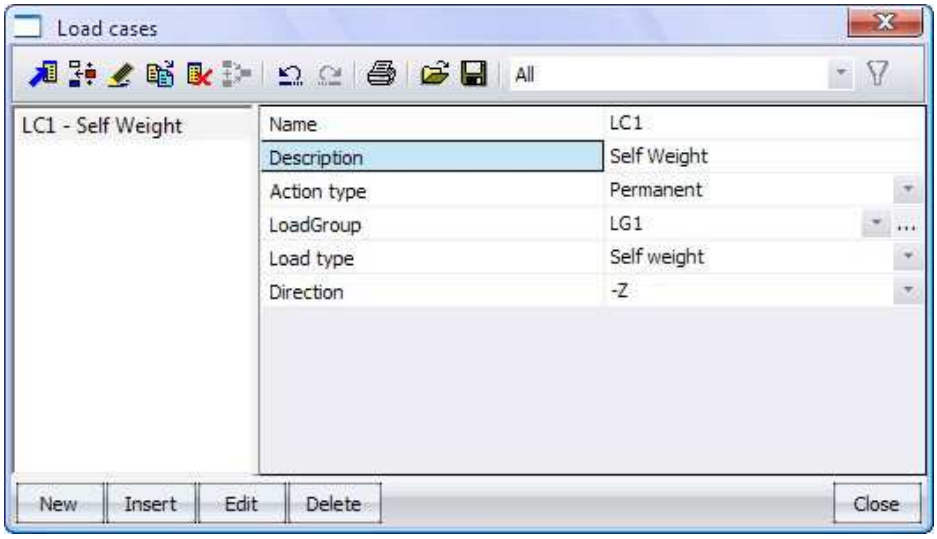
Permanent loads

- Self weight of the beam
- Weight of the slab, located on the beam
- Weight of the finishing layer

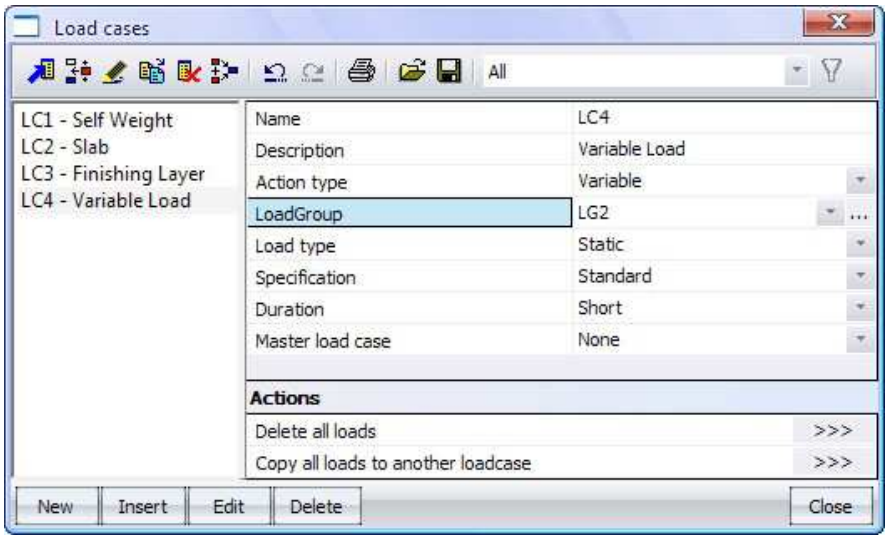
Imposed load

- Category B (offices)

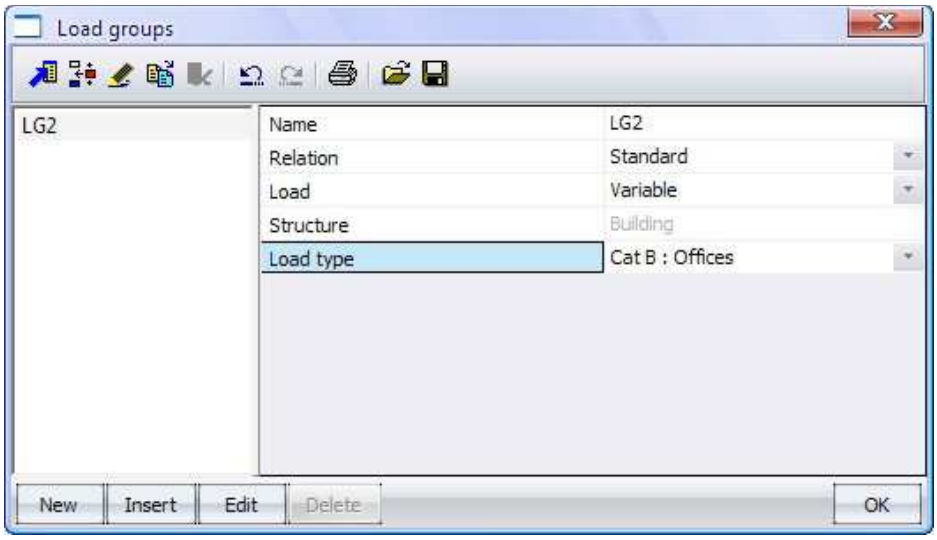
To apply the loads, double-click the option **Load cases, Combinations** in the Main menu. Because there are no loads yet defined, the dialog box **Load cases** will open automatically. Here, the user can define the characteristics of the loads. First enter the self weight of the beam as follows:



By clicking the **New button**, the remaining permanent loads can be entered as well. Be sure to select **Standard** as **Load type**, when creating the permanent loads, other than the self weight. Finally, to add the variable load, insert following parameters.



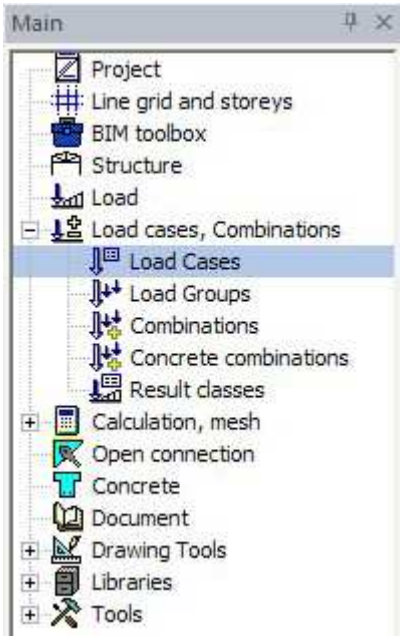
This example concerns an office (category B), therefore it is necessary to change the load type by clicking ... . Choose for the option **Cat B: Offices**.



Click the **OK button** to return to the dialog box **Load cases**. If the necessary loads are created, one closes the entry by using the **Close button**.

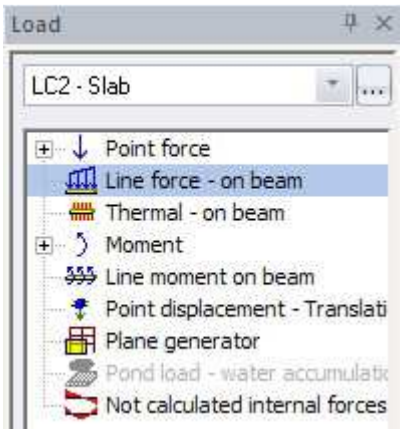
Remark: **Load groups** determine the way the individual loads will be combined with each other, when making load combinations.

Remark: Load cases can be adjusted through the option **Load cases, Combinations < Load Cases.**

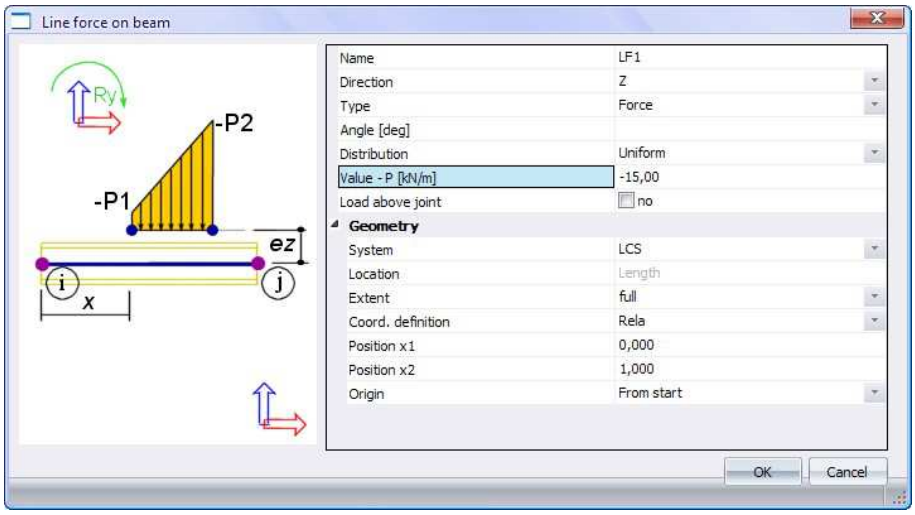


Applying the loads

The submenu **Load** will open automatically. Within this menu, all the different loads can be applied. By clicking , the user can choose the proper load case. Scia Engineer will automatically calculate the self weight of the structure, therefore no further attention needs to be spend on this load case. The weight of the slab **(LC2 – Slab)** on the beam can be represented as a **Line force** of 15 kN/m. This can be done by double-clicking **Line force – on beam**.

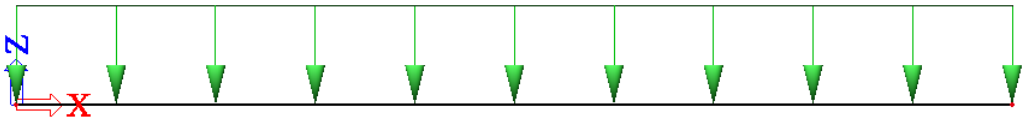


In the dialog box **Line force on beam**, the following parameters have to be entered:



Note that the value of the load is negative, this is because the load disposes of a negative sense in the Z-direction. The **System** is set to **LCS** (Local Coordinate System), this means that all parameters are related to the coordinate system of the structure. Choose **Rela** as **Coord. definition**, hereby the values of **Position x1** and **Position x2** will be applied, relative to the beam. The value 0,000 for **Position x1** will therefore propose the starting point of the beam, the value 1,000 for **Position x2** the endpoint.

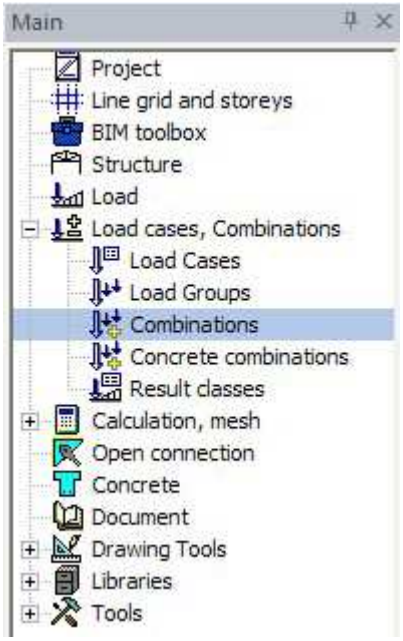
After clicking the **OK button**, one obtains a graphical representation of the applied load.



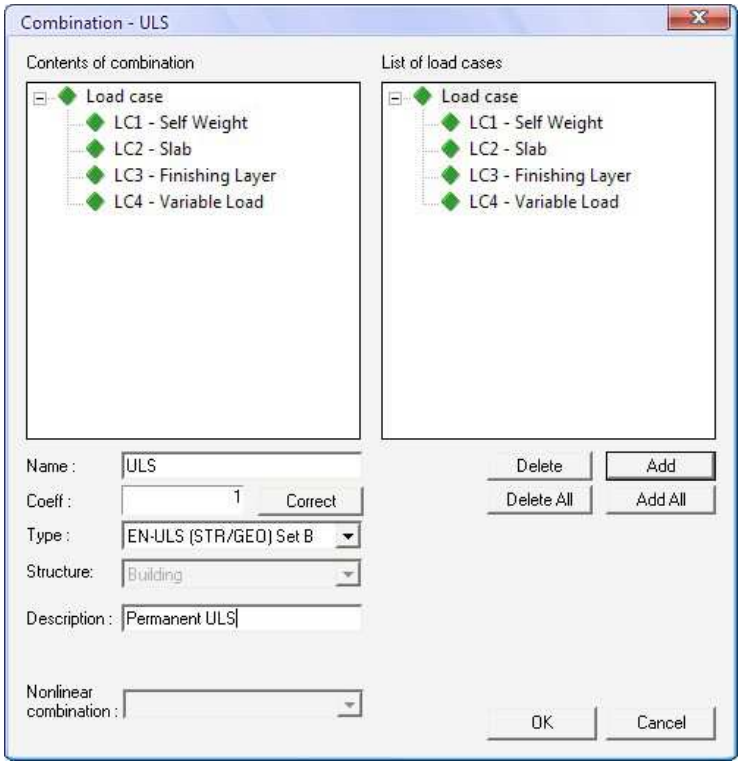
In this example, only **Line forces** occur. This means all loads can be applied as described above. **LC3 – Finishing layer** has a value of **-6,9 kN/m** and **LC4 – Variable load** a value of **-9 kN/m.**

Creating the load combinations

The load combinations can be created with the option **Load cases, Combinations > Combinations**, these will be generated according to the national annex NBN-EN NA.

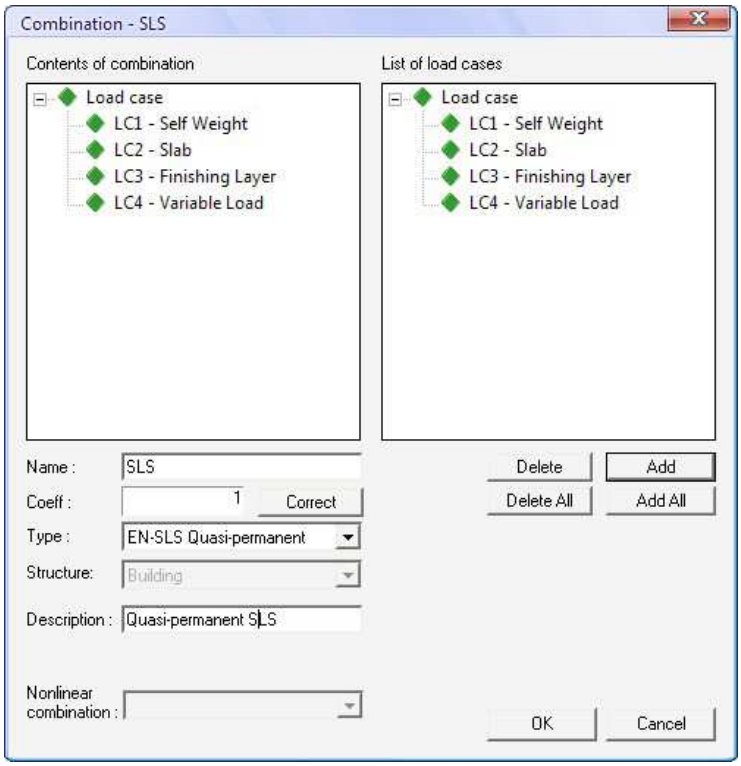


First, one has to create the combinations in the Ultimate Limit State (ULS) as follows:

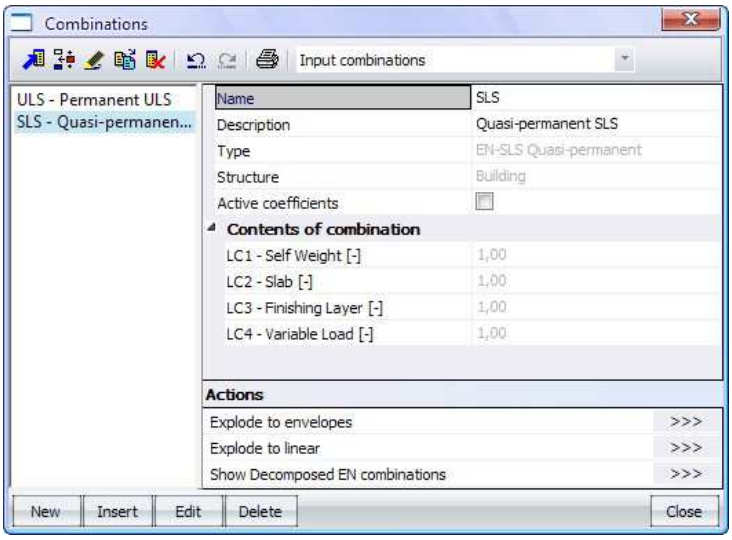


This ULS-combination contains all created loads, one can add this to **Contents of combination** through the **Add All button**. The user also has the possibility to add an additional **Description**. Choose the **type EN-ULS (STR/GEO) Set B,** as this is the permanent situation without any geotechnical effects. Confirm with the **OK button**.

Next, a combination for the Serviceability Limit State (SLS) has to be created. This can be done by clicking the **New button** en choosing the **type EN-SLS Quasi-permanent**.



The created combinations can always be consulted in the dialog box **Combinations**.



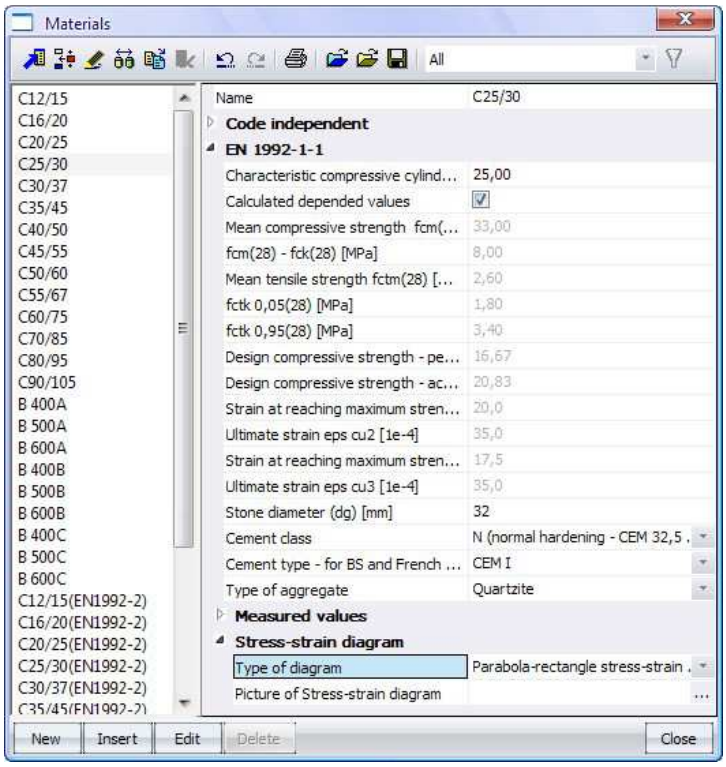
Click the **Close button** to close this window.

Remark: through the >>> behind **Explode to linear** in the dialog box **Combinations**, all possible linear combinations will be generated from the selected combination, in order to perform any checkups.

Materials library

The characteristics of the used materials can be consulted and modified in the materials library. This option is located in the menu bar.

The dialog box **Materials** will open. All materials are listed on the left and the specific characteristic on the right.

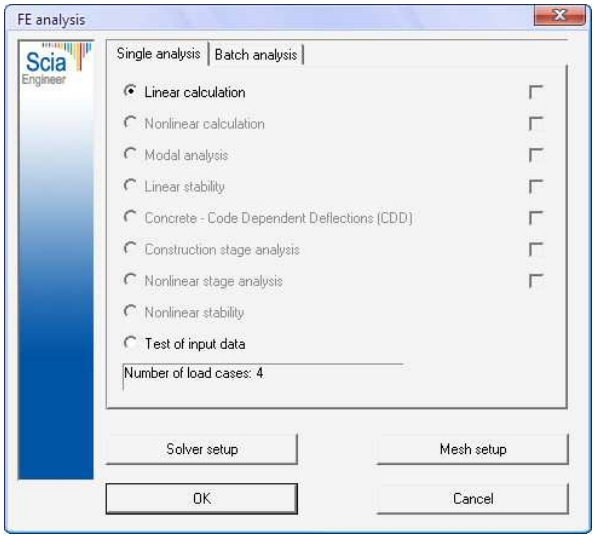


Any further calculations will be performed according to a **parabola-rectangle stress-strain diagram** as **Type of diagram** for the material **C25/30.**

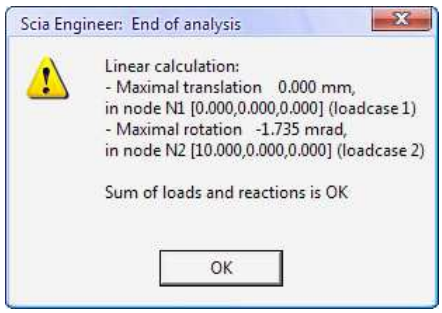
Choose also for a **Bi-linear without an inclined top branch** for steel **B 500A**. To obtain a graphical representation, click ... after **Picture of Stress-strain diagram**.

Linear calculation

After creating and applying the loads, the construction needs to be calculated. This is done through . A new dialog box, **FE analysis**, will appear. Perform a linear calculation by clicking the **OK button**.



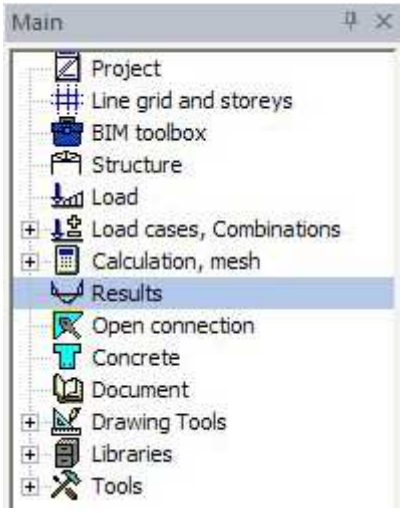
After performing this linear calculation, Scia Engineer reports the end of the calculation:



Click the **OK button** to close this window.

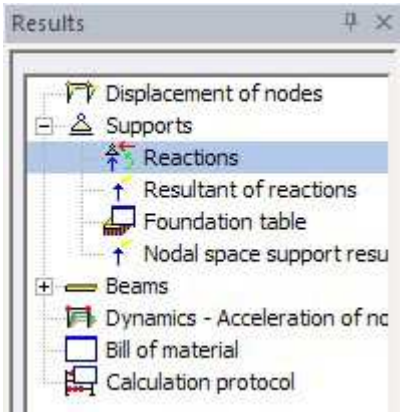
Results

After completing the calculation, the user gains access to the **Results menu.**

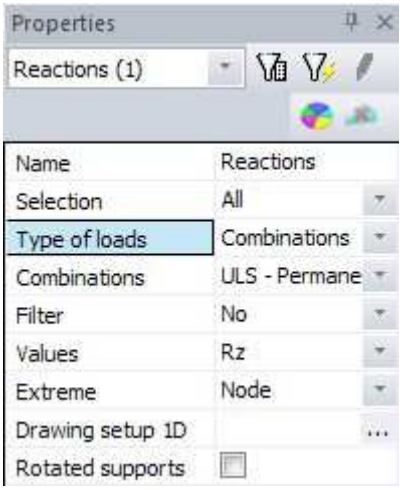


Reaction forces

Through **Results > Supports > Reactions**, the reaction forces at the supports can be determined.

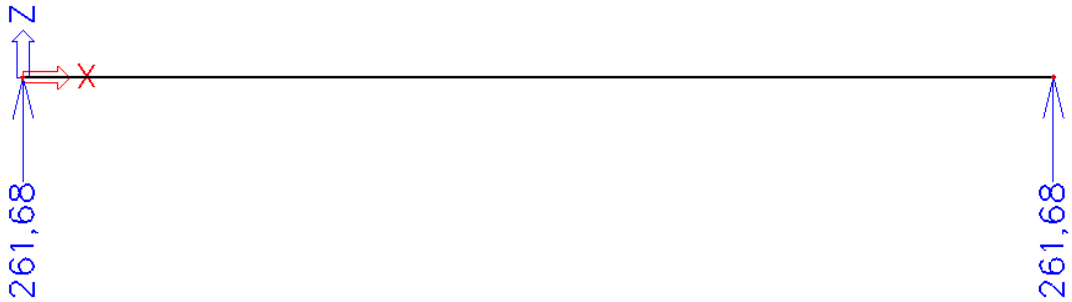


In the **Properties window**, the user can adjust any settings. To display the reaction forces in the Z-direction, choose as **Values** for **Rz**. Be sure **Combinations** are set to **ULS-Permanent**.



Next, to visualize the reaction forces, according to the chosen properties, click >>> , located behind **Refresh**.

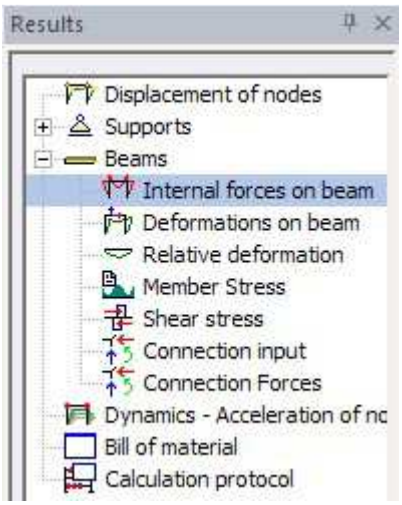
The result will be shown as follows:



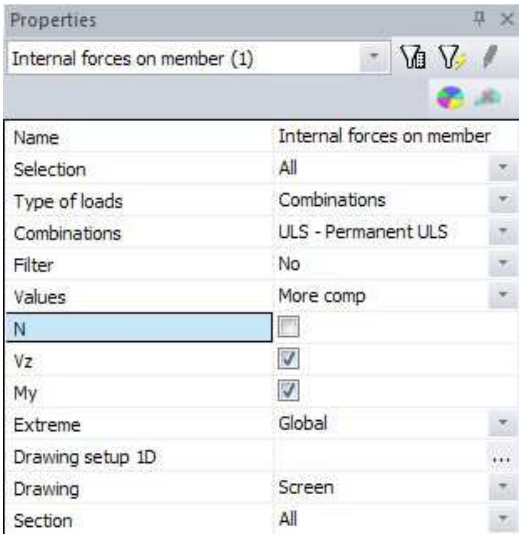
The results in **SLS**, following result will be obtained:

Internal forces

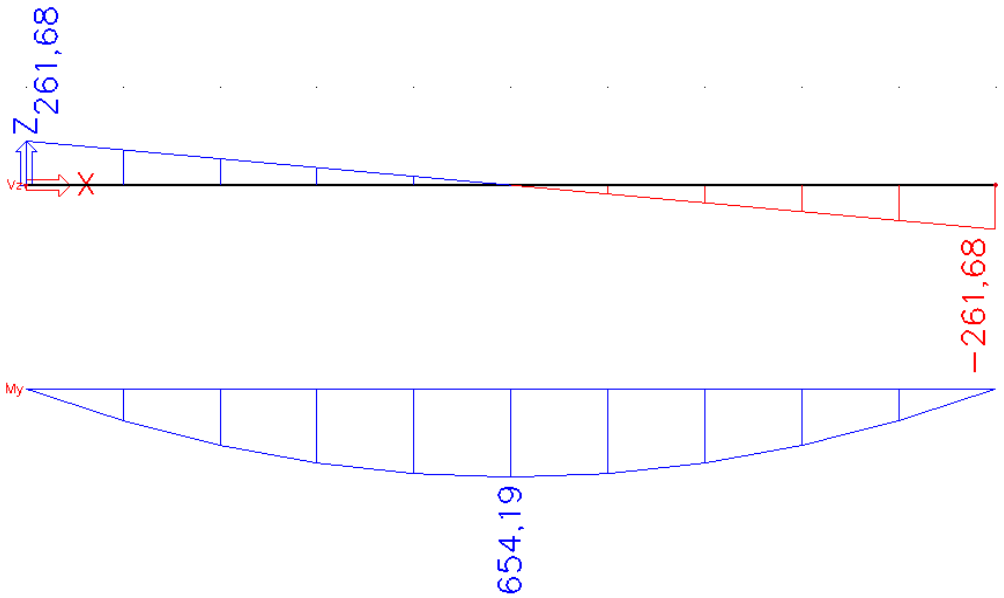
Through **Results > Beams > Internal forces on beam**, the internal forces can be requested.



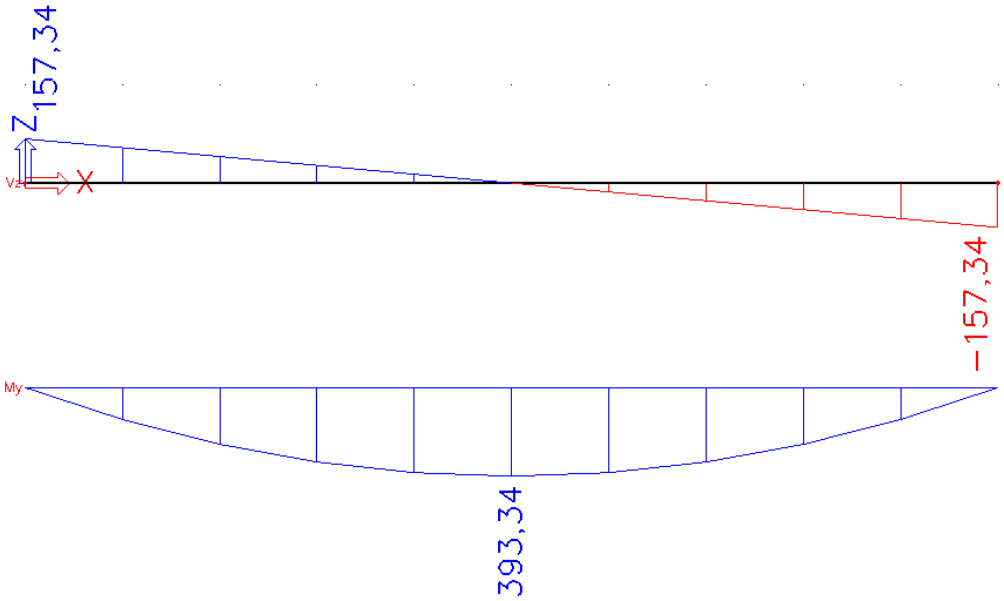
In the **Properties,** the user has to select the appropriate properties:



The results will be visible after clicking >>> , behind **Refresh**. Following internal forces will be given, in case of the **ULS**:

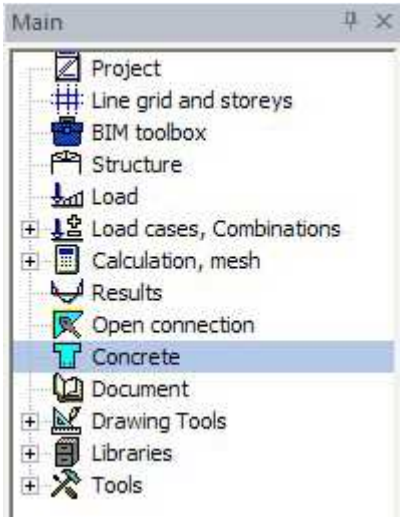


And for the **SLS**:

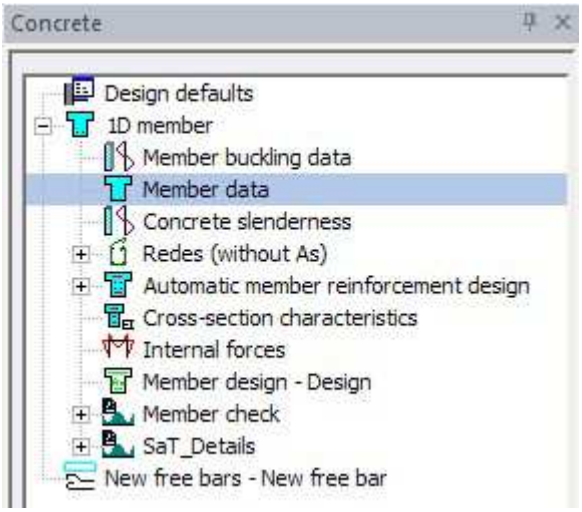


Concrete – general

Through the **Concrete menu** in Main menu, the user has the ability to perform specific concrete related checkups.

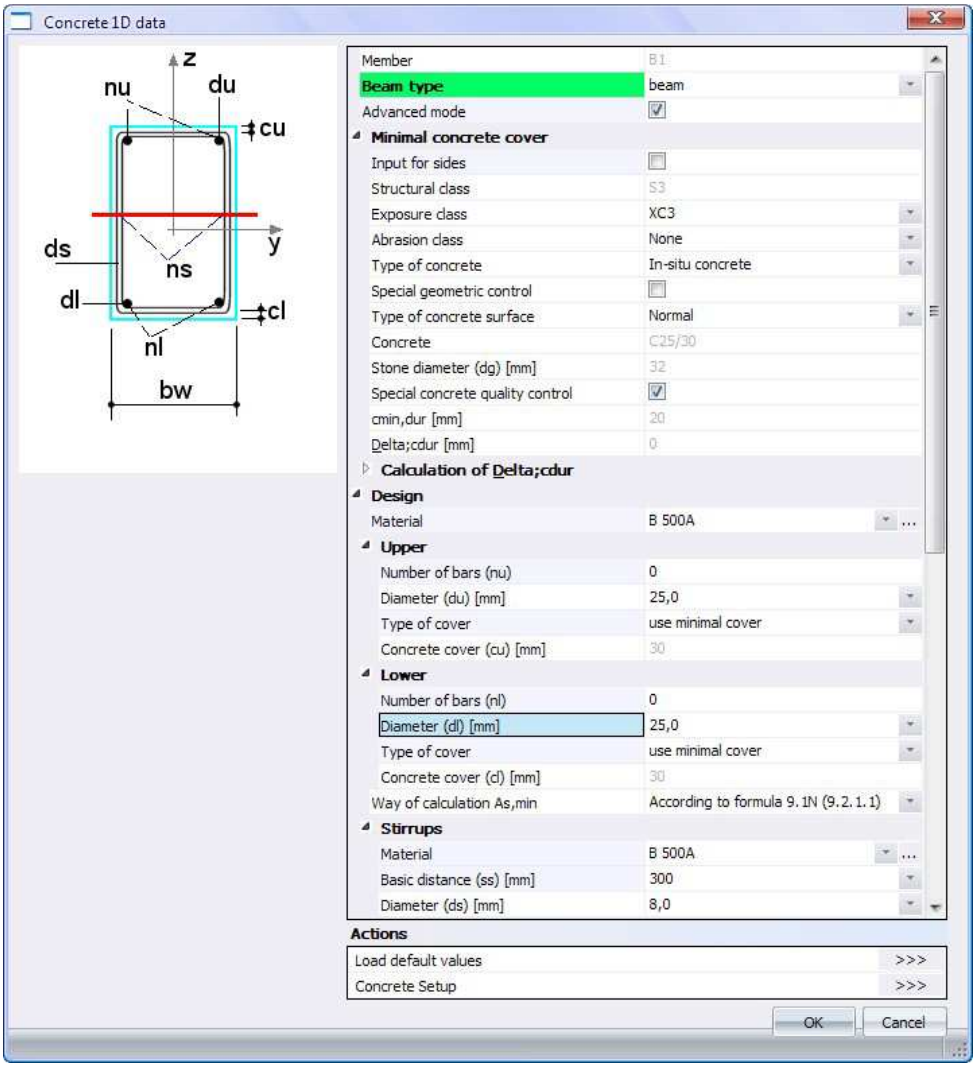


To adjust the properties of a concrete element, consult the option **1D member > Member data**.

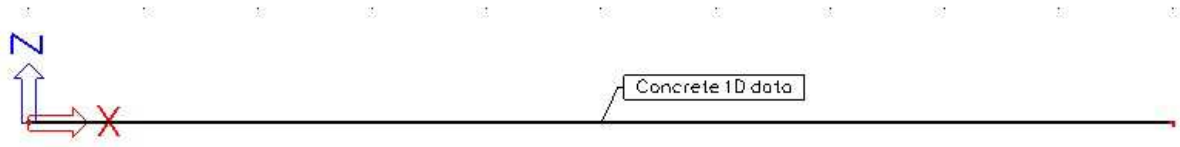


The concrete element, of which the properties need to be changed, has to be selected. This is indicated in the command line.

The dialog box **Concrete 1D data** will open. After ticking the **Advanced mode**, the user can perform several adjustments concerning the characteristics of the selected element. These adjustments need to be done as follows:



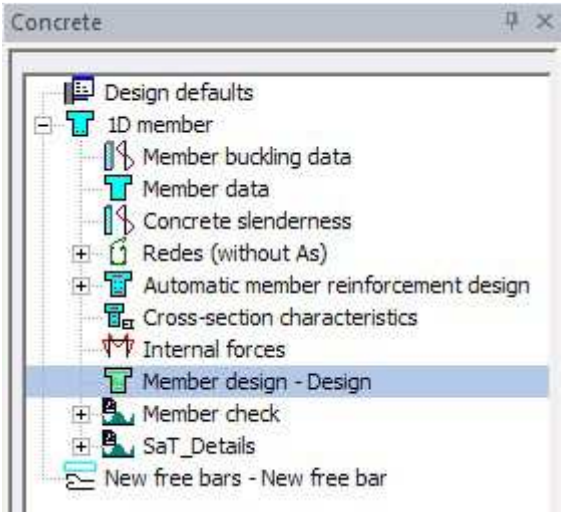
After clicking the **OK button**, a label will appear next to the concrete member. By selecting this label, the properties can be consulted or changed.



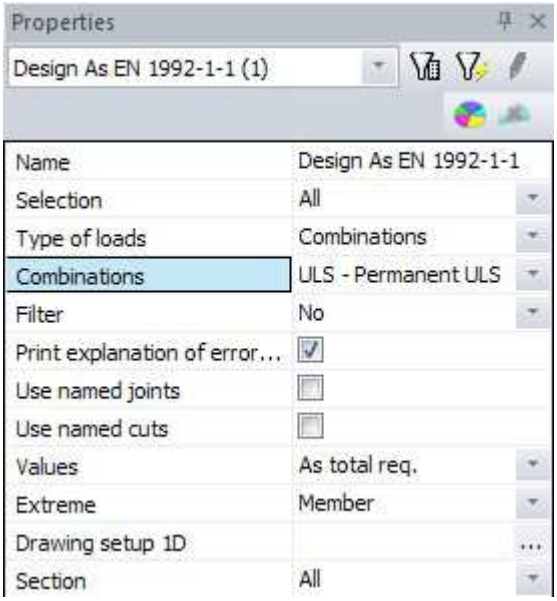
Remark: the user is also able to change the overall settings of all the different elements through the **Design defaults**. However these changes will not affect the elements that have been provided with member data.

Concrete – theoretical reinforcement

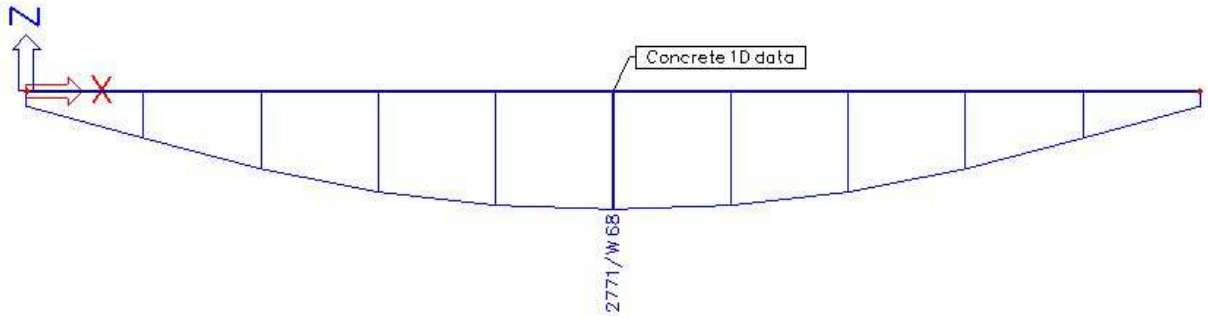
Through the option **1D member > Member design – Design**, several aspects of the theoretical reinforcements can be calculated.



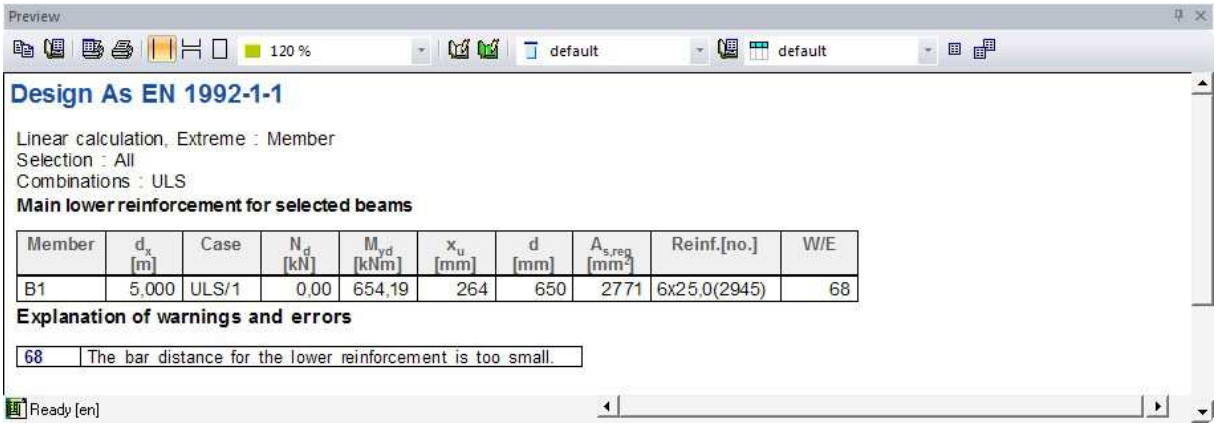
Choose the following options in the properties window:



By choosing **As total req.** and clicking >>> , Scia Engineer will calculate the reinforcement that is needed to resist the internal forces [mm²].



By clicking >>> behind **Preview**, more details will be shown in a table.



Remark: by ticking **Print explanation of errors**, any errors (E) or warning (W) will be explained.

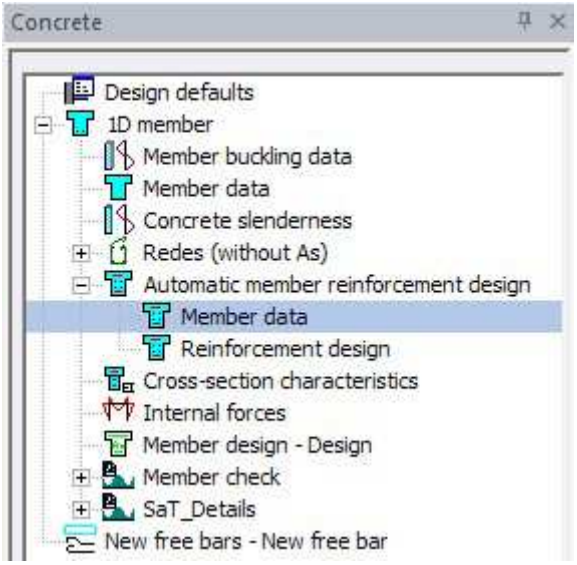
Remark: by selecting **As user defined** or **As add. req.**, respectively the reinforcement added by the user and the extra reinforcement that is needed will be given.

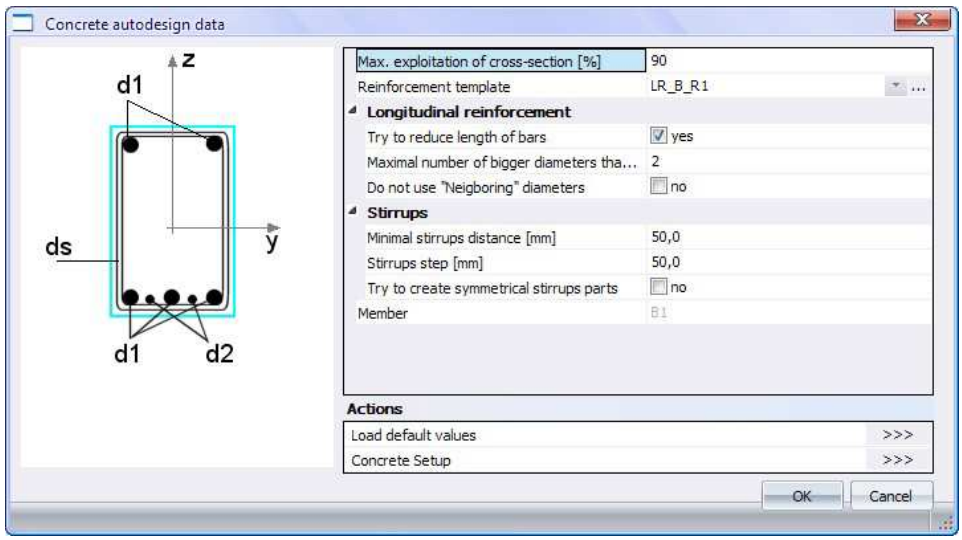
Concrete – AMRD

The user is also able to calculate and apply automatically the practical reinforcement.

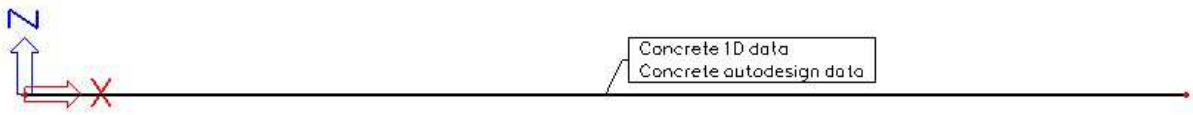
Member data

Within **1D member < Automatic member reinforcement design < Member data**, the user is able to determine the **Maximal numbers of bigger diameters than the default diameter** In this example, the value for this option is equal to **2**. Because there is already chosen for a diameter of 25,0 mm, Scia Engineer will try to implement reinforcement bars with diameters varying from 25,0, 28,0 and 32,0 mm.



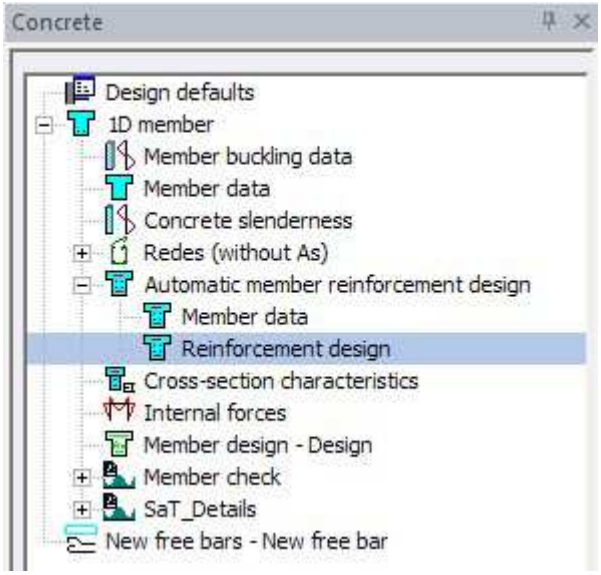


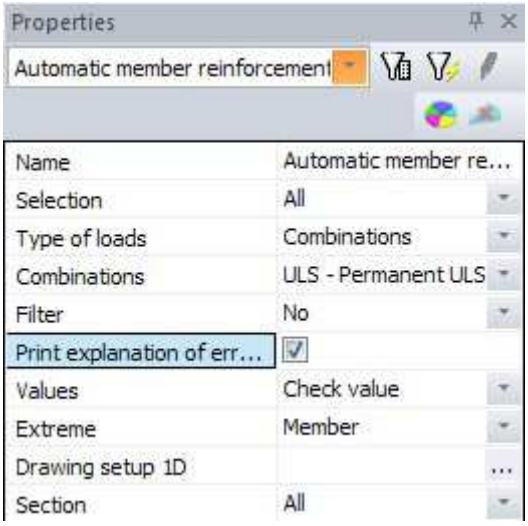
After confirming the properties, the selected element will obtain another label.



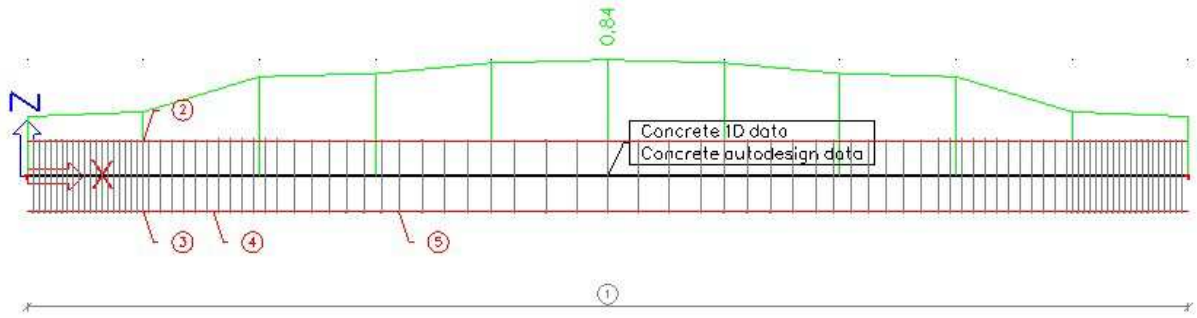
Reinforcement design

Through **1D member < Automatic member reinforcement design < Reinforcement design**, one can calculate the actual reinforcement, after entering the correct parameters and confirming these by clicking the **Refresh button.**

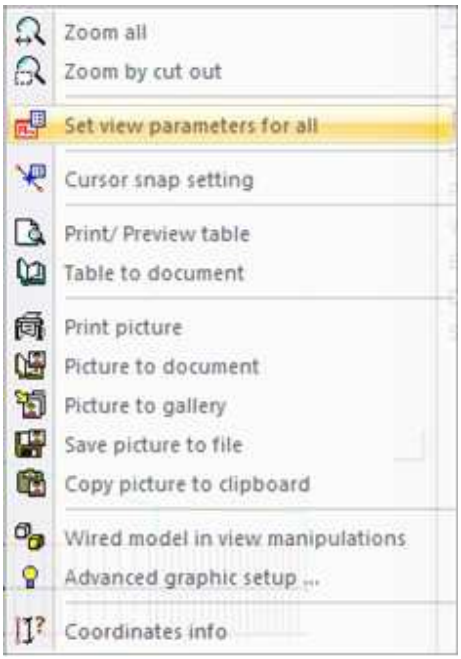




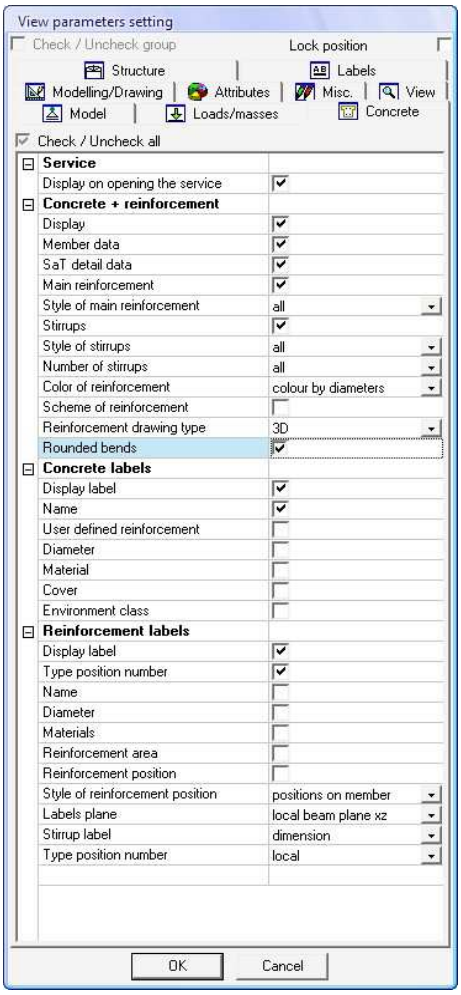
The following output will be displayed:



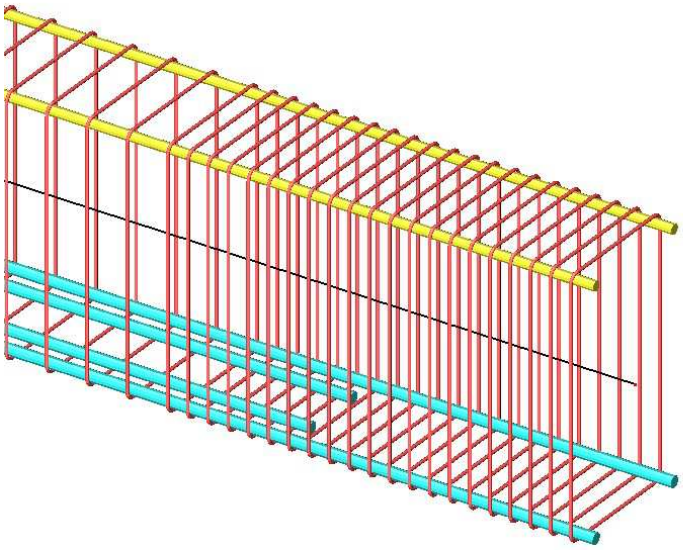
Remark: It is possible to display the reinforcement bars in a more realistic manner by changing the properties in the **Set view parameters for all** option.



More specifically by changing the settings in the **Concrete – tab** as follows:



Eventually, the reinforcement bars will be displayed like this:

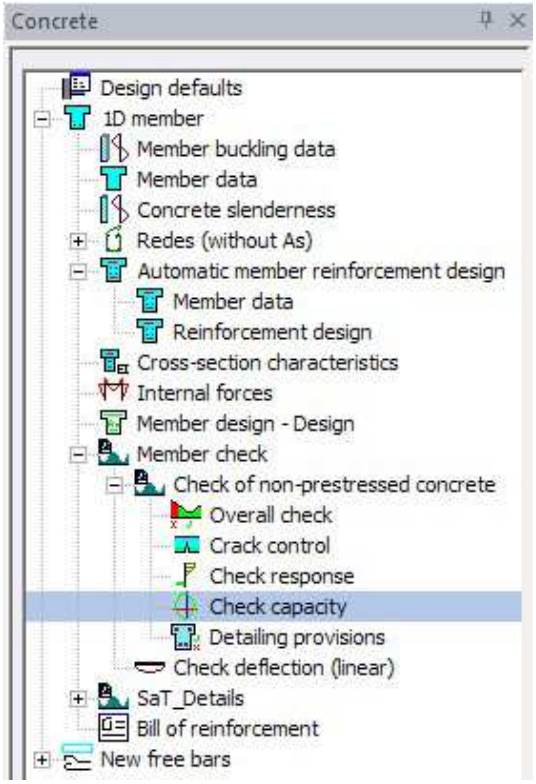


Remark: by using the following buttons  , the user is able to change the view or to zoom in and out.

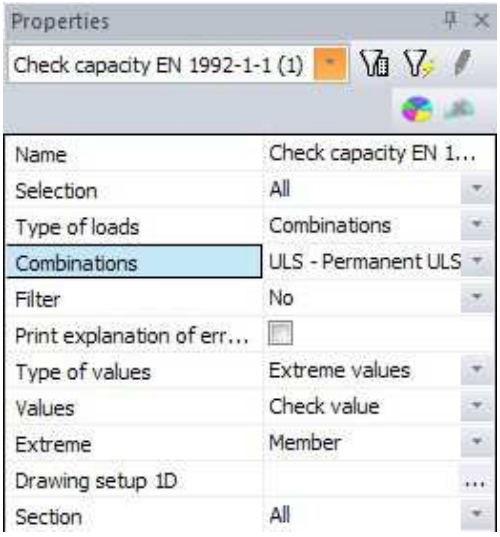
Concrete – checks

Capacity check (ULS)

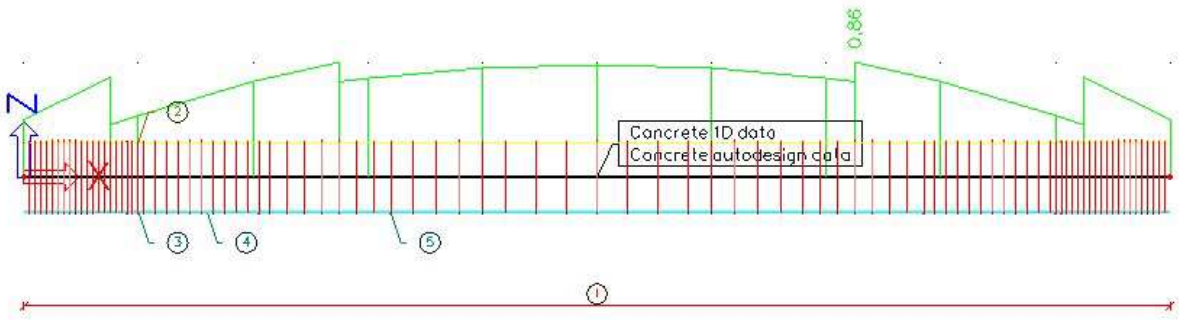
The capacity check is performed in order to check whether the interaction between the normal force N and the bending moments M_y and M_z is located within the fixed boundaries. This check determines the efficiency of the added reinforcement and is executed through the option **1D member < Member check < Check of non-prestressed concrete < Check capacity**.



Choose for the properties as shown below:

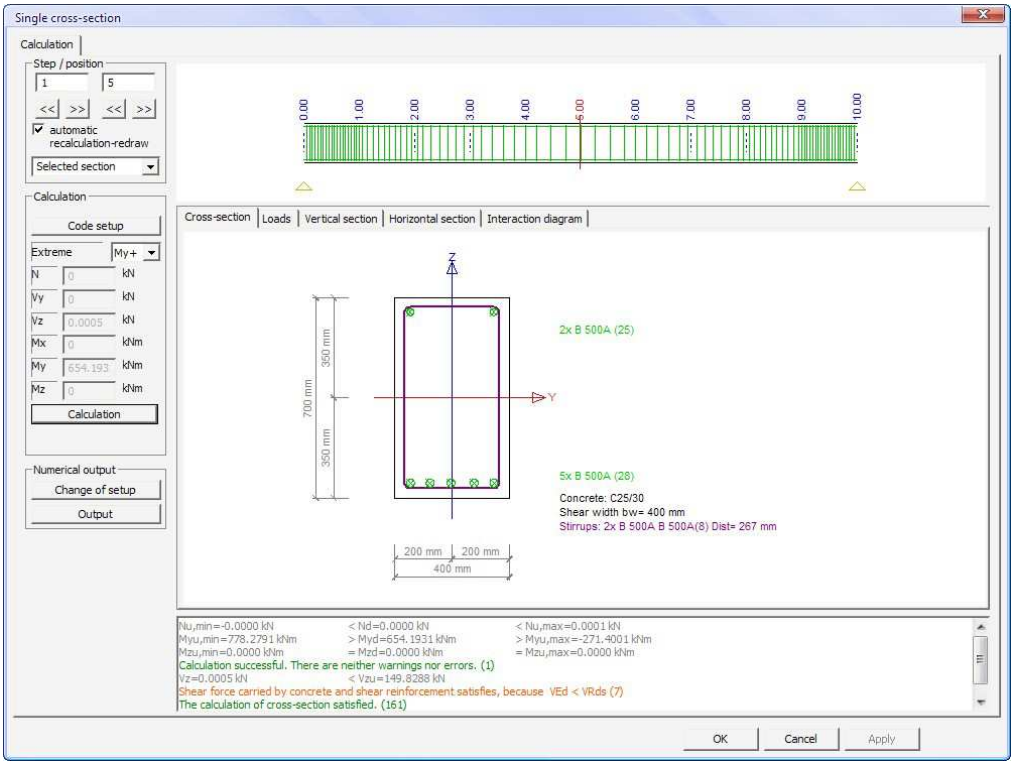


After refreshing, the output will be as follows:



Remark: the output will be the maximum value, which has to be smaller than 1 in order to be correct. A correct output will be displayed in green.

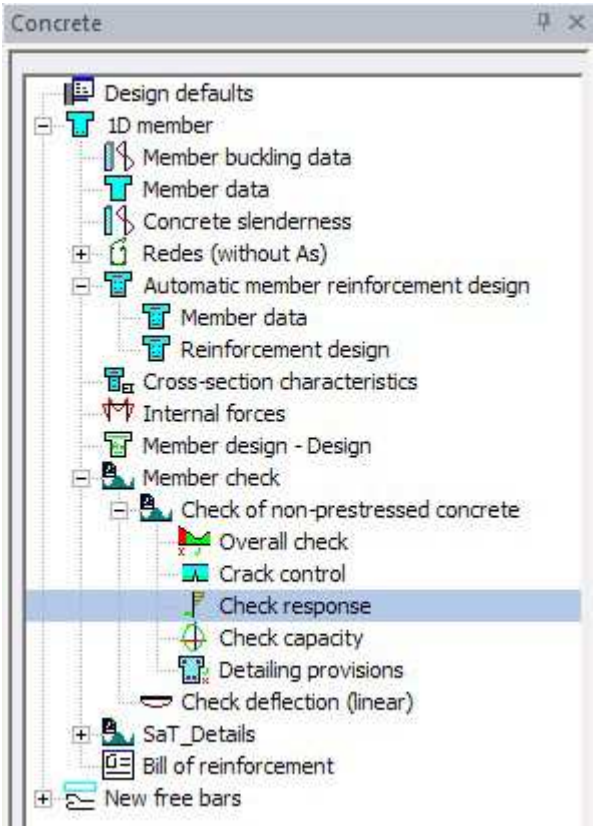
The results of each cross section can be observed by clicking >>>, behind **Single Check**. The command line will ask the user to select the member of which the cross section will be analyzed.



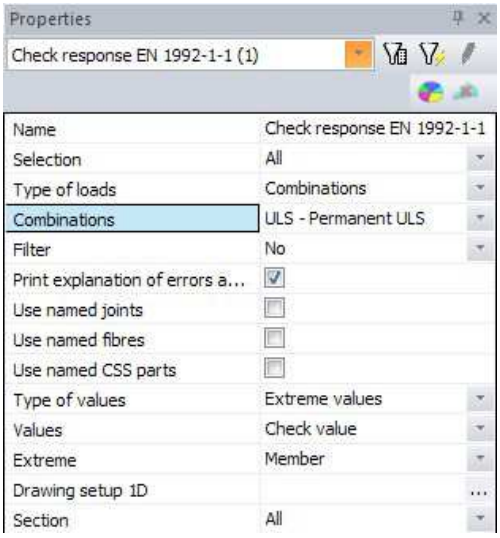
Remark: it is necessary to click **Calculation**, in order to obtain the output as shown above. The user is also able to choose the exact position of the cross section, as well as to scroll through the results by using the tabs.

Response check (ULS)

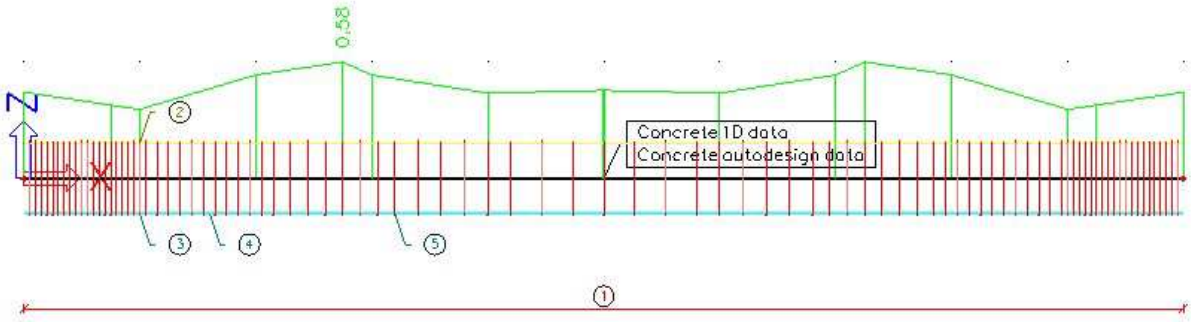
This check is executed in order to make sure the stresses, in both concrete and reinforcement steel, do not excess their limit values. The check can be performed through **1D member < Member check < Check of non-prestressed concrete < Check response**.



Select the following settings in the properties window:



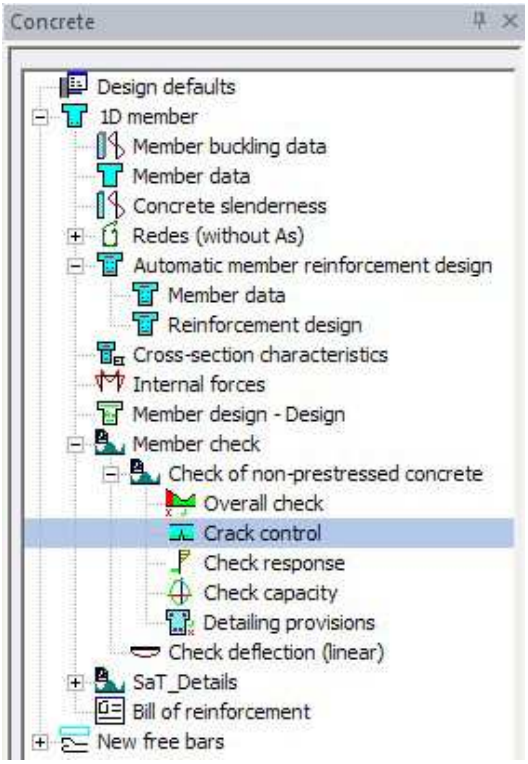
After clicking the **Refresh button**, the following output will be obtained:



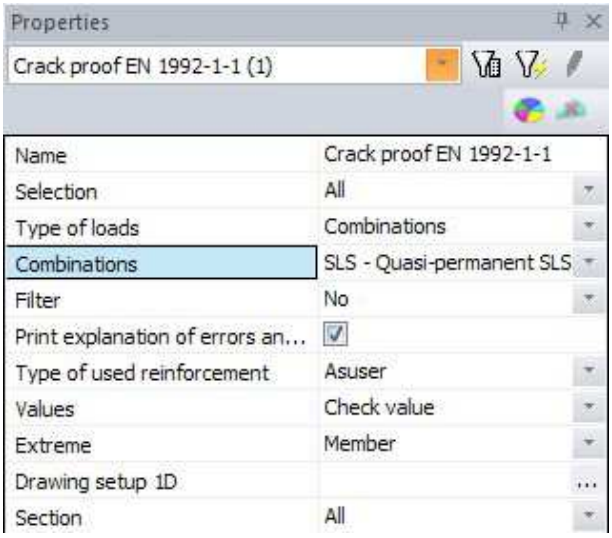
The user is also able to request a **Preview** and to perform a **Single Check**.

Crack control (SLS)

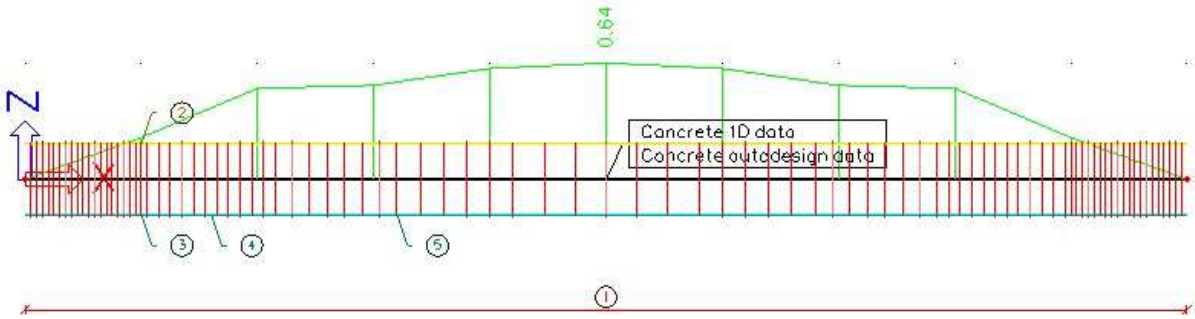
Scia Engineer also offers the opportunity to check the construction for possible cracks. This is done with the option **1D member < Member check < Check of non-prestressed concrete < Crack control**.



The properties should be changed as shown below:



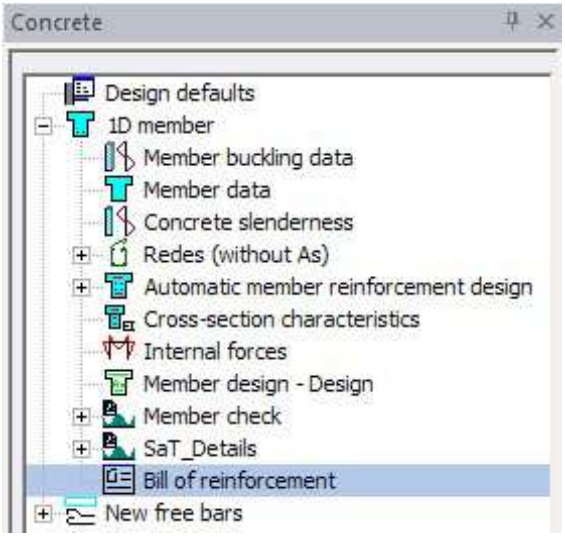
Make sure to choose the option **Asuser**, this value is equal to the previously added reinforcement during the AMRD. After clicking the Refresh button, the output will be as follows:



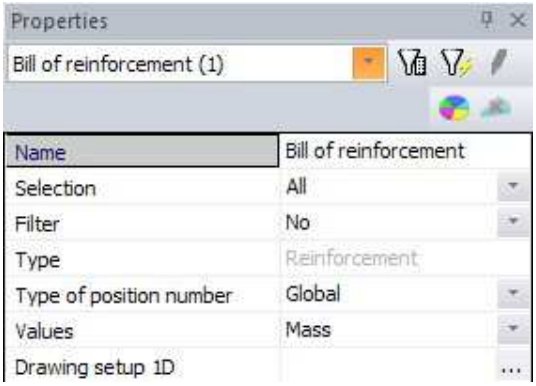
The user is also able to request a **Preview** and to perform a **Single Check**.

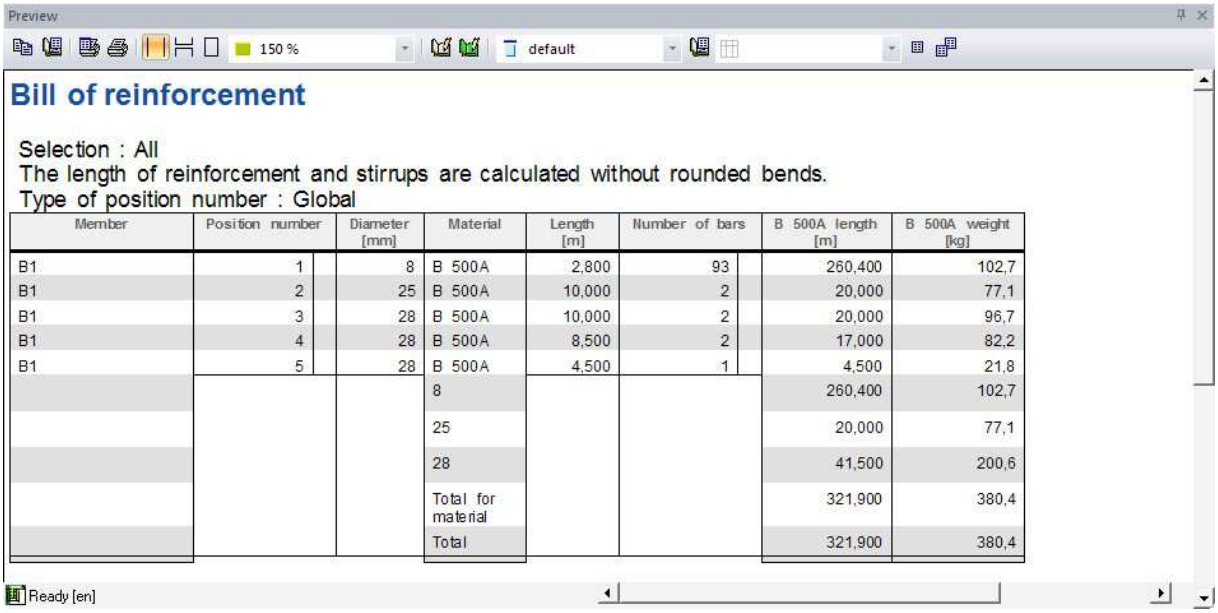
Concrete – Bill of reinforcement

The length and mass of the added reinforcement can be requested through the option **1D member < SaT_Details < Bill of reinforcement**.



After refreshing the following properties, the user will be able to view the output:

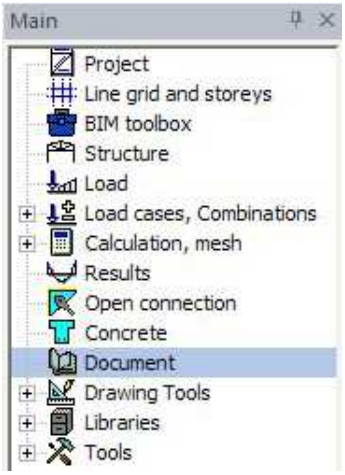




Member	Position number	Diameter [mm]	Material	Length [m]	Number of bars	B 500A length [m]	B 500A weight [kg]
B1	1	8	B 500A	2,800	93	260,400	102,7
B1	2	25	B 500A	10,000	2	20,000	77,1
B1	3	28	B 500A	10,000	2	20,000	96,7
B1	4	28	B 500A	8,500	2	17,000	82,2
B1	5	28	B 500A	4,500	1	4,500	21,8
			8			260,400	102,7
			25			20,000	77,1
			28			41,500	200,6
			Total for material			321,900	380,4
			Total			321,900	380,4

Document

All required results can be put together in one document. This function can be activated in the Main menu.



By clicking New, the required results can be selected. These will be included in the Document. In the dialog box **New document item**, all items are summed up. These can be added by the user by clicking <<< Add. By adding for example **Results < Internal forces on member**, the following output will be obtained:

1. Internal forces on member

Linear calculation, Extreme : Global, System : LCS
Selection : All
Load cases : LC1

Member	Case	dx [m]	N [kN]	Vz [kN]	My [kNm]
B1	LC1	0,000	**0,00**	**34,34**	**0,00**
B1	LC1	10,000	0,00	**-34,34**	0,00
B1	LC1	5,000	0,00	0,00	**85,84**

Remark: in order to gain more insight, it is useful to add the **Combination key**.