Version
November 2017

Program

SHAPE-THIN

Cross-section properties and designs of thin-walled cross-sections

Introductory Example

Am Zellweg 2
93464 Tiefenbach
Germany

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

1 Introduction

With this introductory example we want to show you the most important functions of the program SHAPE-THIN. Like in any other software there are many ways to reach a goal in SHAPE-THIN. Depending on the situation and your personal preferences, it may be useful to proceed in one way or another.

The example describes a steel angle to which an unequal angle section is welded. We will determine the cross-section values, the stresses and the effective widths of the angle. Furthermore, this simple cross-section should encourage you to discover the possibilities of SHAPE-THIN on your own.

Once the 90 day testing phase has expired, you can still enter and calculate the angle as the restrictions for the demo version of a maximum of four elements are met.

The described buttons are given in square brackets; for example, [Apply]. In addition, they are pictured on the left. Expressions appearing in dialog boxes, tables, and menus are set in *italics*. This way you can better follow the explanations. Required input is written in **bold**.

To find a description of the program functions, read the manual of SHAPE-THIN that you can download on our website.

The file **Section.du8** including cross-section properties is also stored in the *Examples* project that was created during the installation. However, for the first steps with SHAPE-THIN we recommend entering the section manually.

2 Cross-Section and Loads

2.1 Cross-section

The example is a welded steel angle consisting of steel plates with a thickness of 12 mm and 10 mm to which an angle section L 200x150x12 is welded.

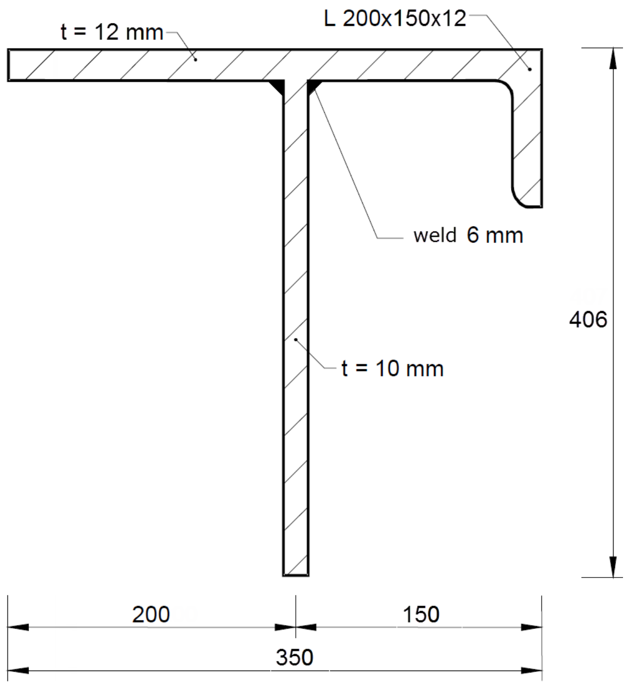
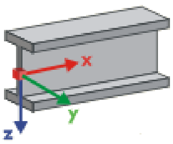


Figure 2.1: Sketch of cross-section

The section is made of **steel S 355**.

2.2 Internal Forces

Load case 1: Tension and bending



In the first load case, a tensile force is applied together with biaxial bending. The internal forces are acting in the direction of the member axes x, y and z.

$N = 35$ kN

$V_y = 15$ kN

$V_z = -25$ kN

$M_y = 60$ kNm

$M_z = 15$ kNm

Load case 2: Compression and bending

In the second load case, we will analyze a compressive force together with bending moments.

$N = -80$ kN

$V_y = -10$ kN

$V_z = -20$ kN

$M_y = 40$ kNm

$M_z = -5$ kNm

3 Creating the Cross-Section

3.1 Starting SHAPE-THIN

We start SHAPE-THIN with the icon **Dlubal SHAPE-THIN 8.xx** on the desktop.

You can also start the program on the taskbar
Start → **All Programs** → **Dlubal** → **Dlubal SHAPE-THIN 8.xx**.

3.2 Creating the Cross-section

The SHAPE-THIN work window opens and we can see a dialog box. We are asked to define the general data of a new cross-section.

If a section is already displayed in the window, we close it by using the menu **File** → **Close**. Then, we open the *General Data* dialog box with the menu **File** → **New**.

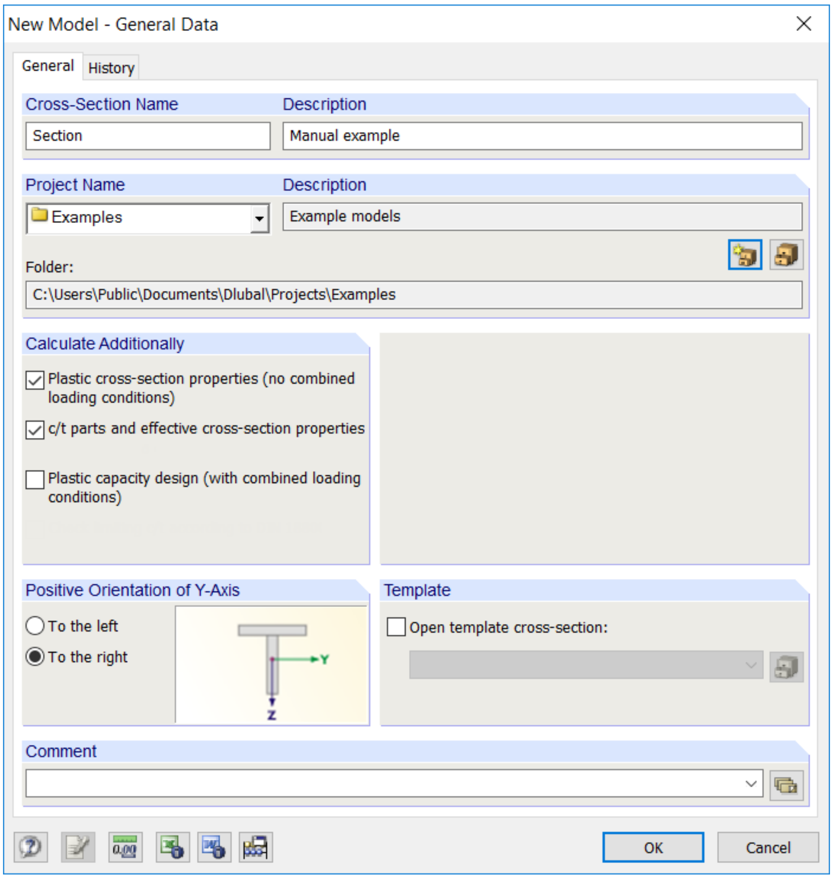


Figure 3.1: Dialog box *New Model - General Data*

In the *Cross-Section Name* text box we enter **Section**, and in the *Description* box we enter **Manual example**. A model name must always be defined because it determines the name of the SHAPE-THIN file. A description, however, does not necessarily need to be used.

Then, we select the **Examples** project from the list in the field *Project Name* if it is not preset. The *Description* of the project and the *Folder* are shown automatically.

In the dialog section *Calculate Additionally*, we select the checkbox for **c/t parts and effective cross-section properties** because we want to perform a classification.

We don't change the *Positive Orientation of Y-Axis*, and we keep the default **To the right**.

Finally, we have defined the general data of the model. We close the dialog box with [OK].

The empty work window of SHAPE-THIN is displayed.

4 Cross-Section Data

4.1 Checking Default Settings

Work window

Firstly, we maximize the work window with the corresponding button in the title bar. In the work space, we see the axes of coordinates with the global directions Y and Z.

To change the position of the axes, we click the button [Move, Zoom, Rotate] in the toolbar above. The mouse pointer turns into a hand. While holding down the left mouse button we can now place the work space in any position. For entering data we recommend moving the axes of coordinates to the left in the direction of the navigator.

The hand also allows zooming in and out: Hold down the shift key and the left mouse button and move the pointer up and down.

To close the function, you have different possibilities:

- Click the toolbar button again
- Press the [Esc] key
- Right-click into the work space

Units

For our example, we set the metric units. The units and decimal places can be changed anytime during the input and evaluation. The values will be converted and adjusted automatically.

We open the dialog box *Units and Decimal Places* by clicking on the menu

Edit → **Units and Decimal Places**.

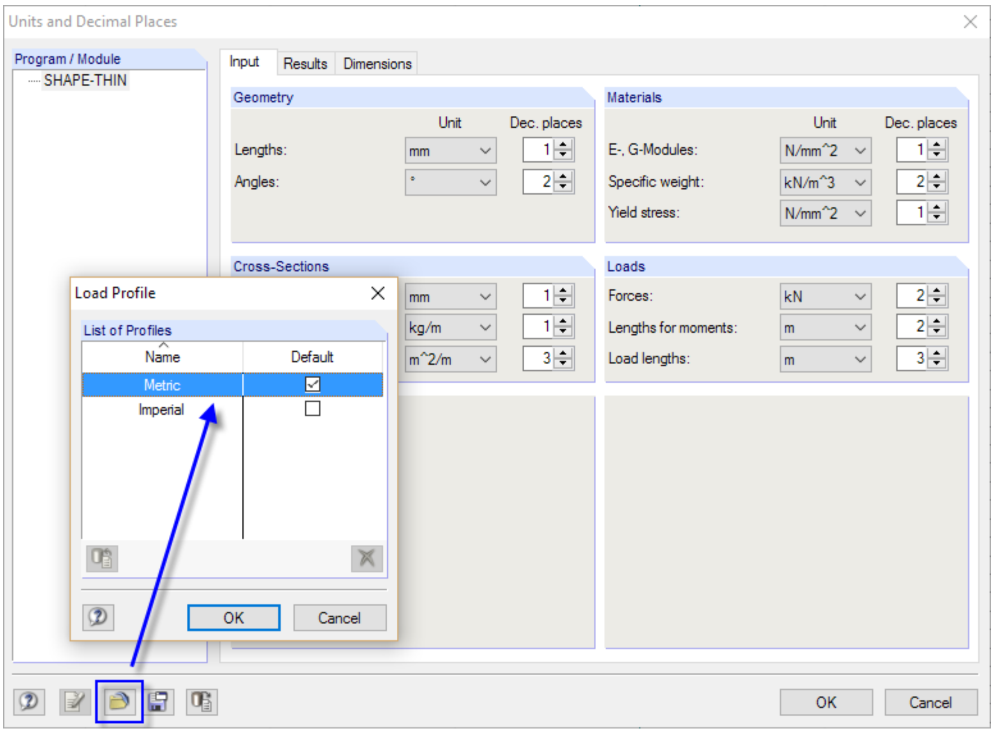


Figure 4.1: Dialog box *Units and Decimal Places*

We click the [Load Saved Profile] button. In the *Load Profile* dialog box, we select the **Metric** profile.

When clicking [OK], the modified units are set for the *Input*, *Results* and *Dimensions*.

We save our changes by [OK], thus closing the *Units and Decimal Places* dialog box.

Grid

The grid forms the background of the work space. The spacing of grid points can be adjusted with the button *Settings of Work Plane, Grid/Snap, Object Snap, Guide Lines* shown on the left.

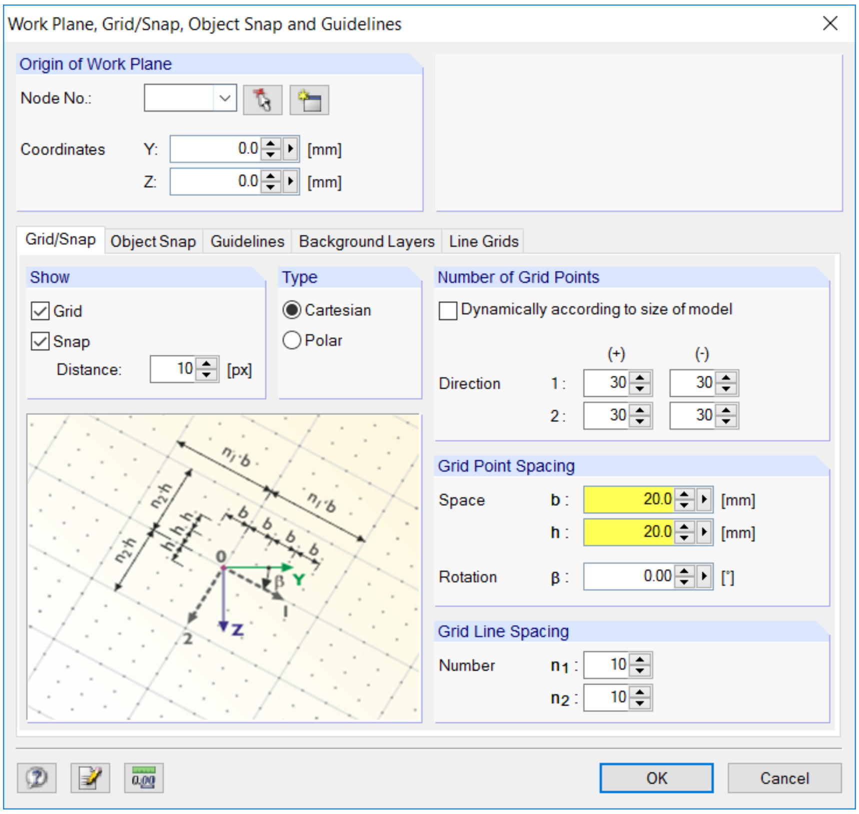


Figure 4.2: Dialog box *Work Plane, Grid/Snap, Object Snap and Guidelines*

For our example, we adjust the grid point spacing to **20 mm**. We close the dialog box with [OK].

SNAP GRID

For entering data later it is important that the control fields *SNAP* und *GRID* in the status bar are set active. This way the grid becomes visible in the work space and the points will be snapped on the grid when clicking.

Mouse functions

The mouse functions follow the general standards for Windows applications. To select an object for further editing, we click it once with the **left** mouse button. To open its Edit dialog box, we double-click the object.

When we click an object with the **right** mouse button, its shortcut menu appears showing us object-related commands and functions.

By scrolling the **wheel button** we can maximize or minimize the current model representation. The position of the mouse pointer is always taken as the center of the zoom area.

By holding down the wheel button we can move the model directly without previously activating the button [Move, Zoom, Rotate]. The pointer symbols show the selected function.

4.2 Changing Material

Steel A992 is preset as material. As our section consists of steel S355, we have to change the material.

In the navigator, we click on the ⊞ symbol to open the **materials**. Then, we click on the entry *1: Steel A992* with the right mouse button.

In the shortcut menu, we select the **Edit** option.

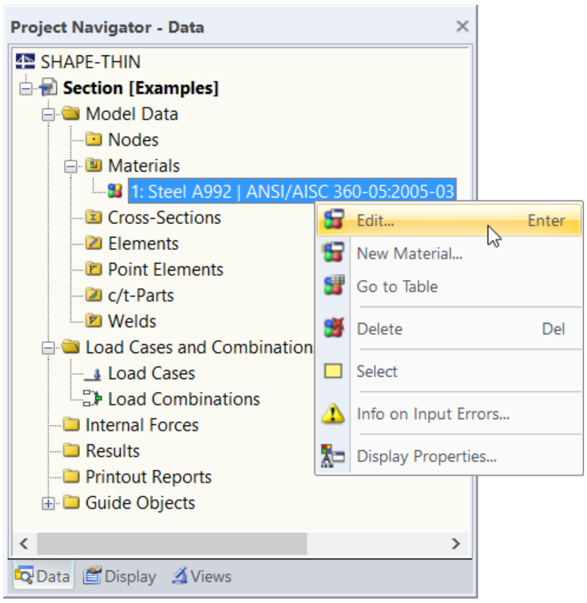


Figure 4.3: *Edit* materials in shortcut menu of navigator

The dialog box *Edit Material* appears.

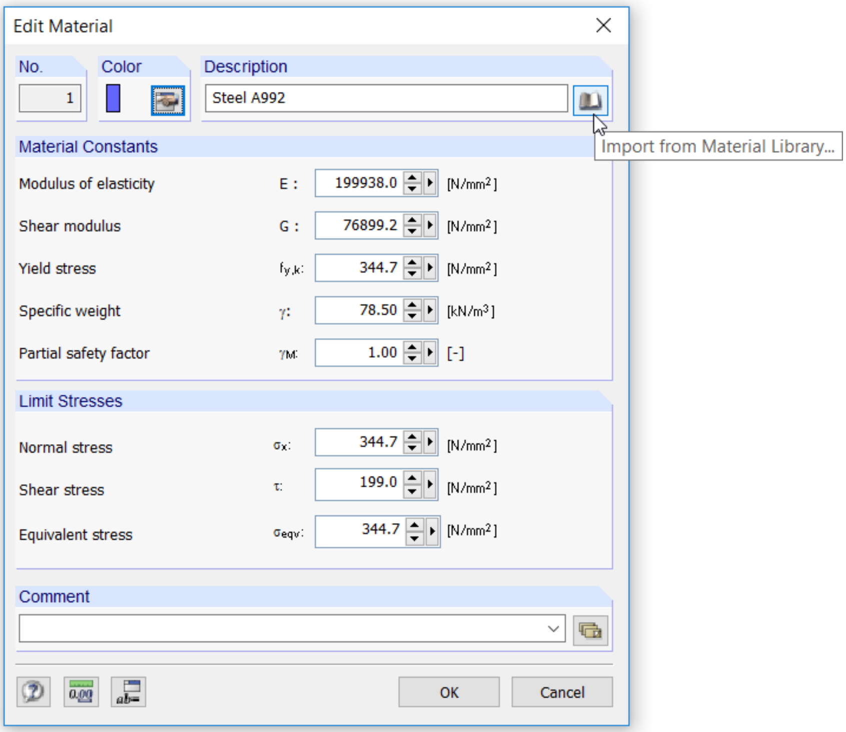


Figure 4.4: Dialog box *Edit Material* with access to library

We click the [Library] button to access the material library.

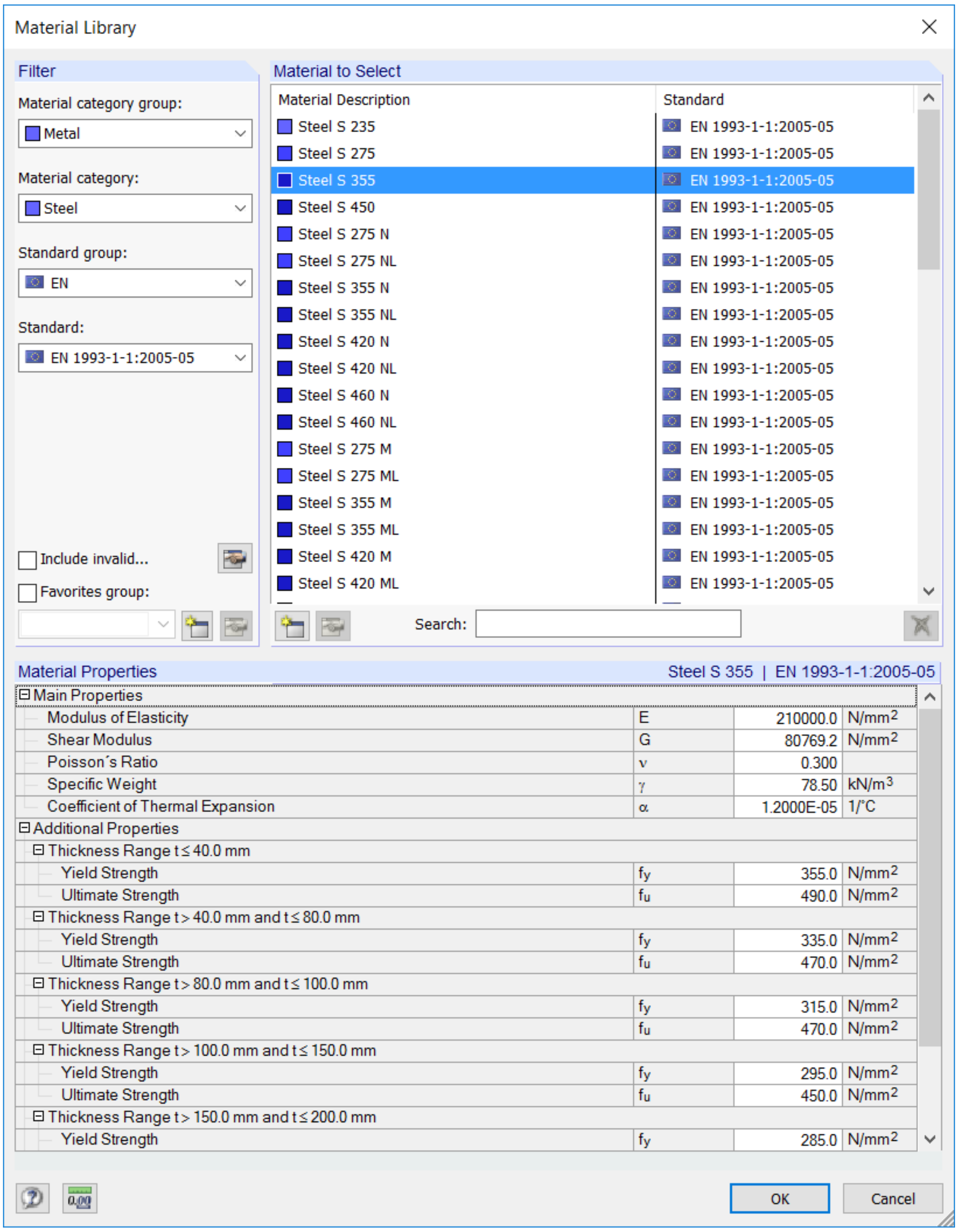


Figure 4.5: Selecting Steel S 355 in the library

The dialog section *Filter* offers several categories so that we can restrict the materials according to certain criteria. We set the filters as shown in the figure above. We can see that the list *Material to Select* becomes clearer.

We click on the material **Steel S 355**. In the dialog section below, we can check the *Material Properties* including the different ranges of element thicknesses.

We use the [OK] button in both dialog boxes to import the changed material properties.

Now we can start to enter the cross-section geometry.

4.3 Defining Elements

It is possible to first define the nodes graphically or in tables, and to later connect them with elements. For our example, however, it is better to use the direct graphical input of elements where nodes are created automatically.

4.3.1 Placing Elements

We open the dialog box for placing an element graphically on the menu

Insert → **Model Data** → **1.4 Elements** → **Polyline** → **Graphically** → **Continuous**

or use the corresponding button in the toolbar, which is faster.

The dialog box *New Element (Polyline)* appears.

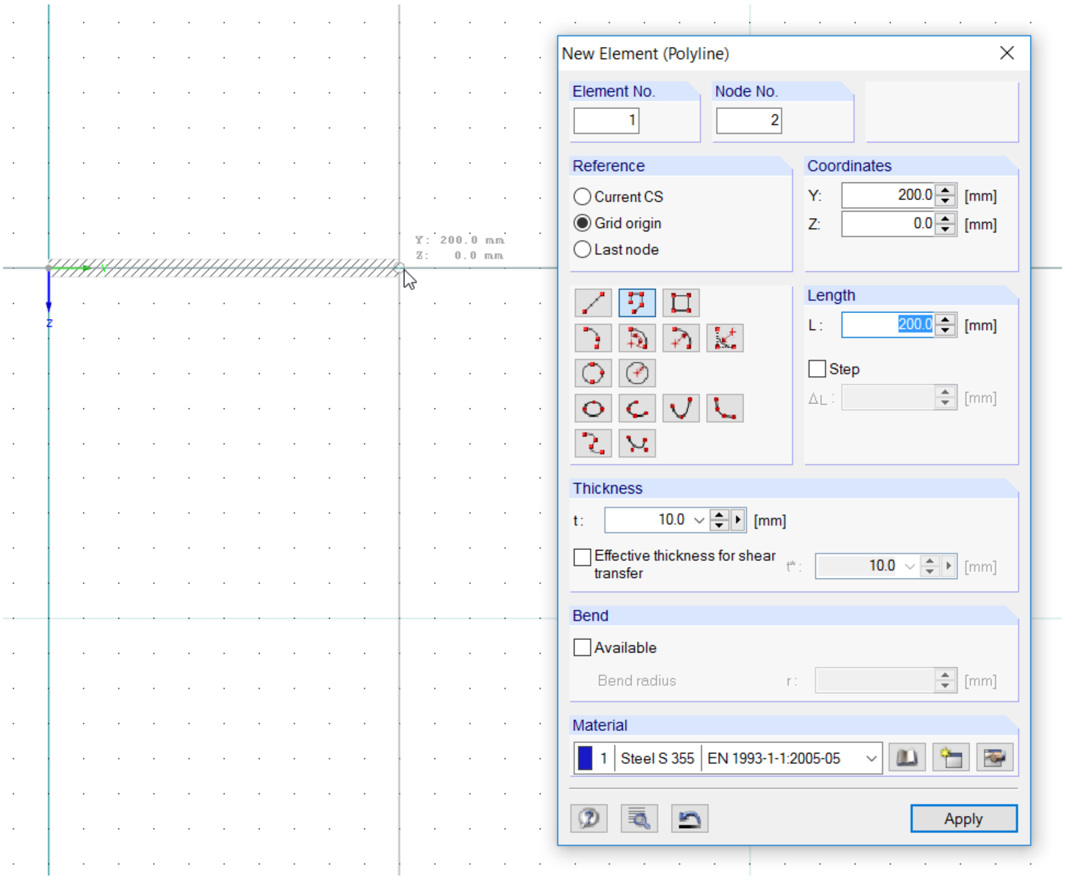


Figure 4.6: Dialog box *New Element (Polyline)*

The *Element No.* **1** and the *Thickness* of **10 mm** are preset. The *Material* is **Steel S 355**. We accept these settings because we will adjust the flange thickness subsequently.

When we move the mouse across the work space, we see the coordinates of the pointer displayed in the window. The reticle snaps at the points of the 20 mm grid.

With a click on the left mouse button we place node **1** as the initial point in the zero point (Y/Z-coordinates **0.0/0.0**) .

With another mouse click we define node **2** in the grid point **200.0/0.000** as the element's end point.

As we have chosen the *Continuous* option, node 2 represents the initial node of our next element no. **2**. Thus, we can go on with placing node **3** in the grid point **200.0/400.0**.

We close the function with a right mouse click into the empty work window or with [Esc].

Changing the view

For the full screen view we select on the menu

View → **Show All**.

We can also use the corresponding toolbar button or the function key [F8].

Displaying numbers

Before we enter more data, it is recommended to activate the numbering of the nodes and elements. The quickest way is to right-click into an empty area of the work window.

A shortcut menu with useful functions appears. We click the entry **Show Numbering**.

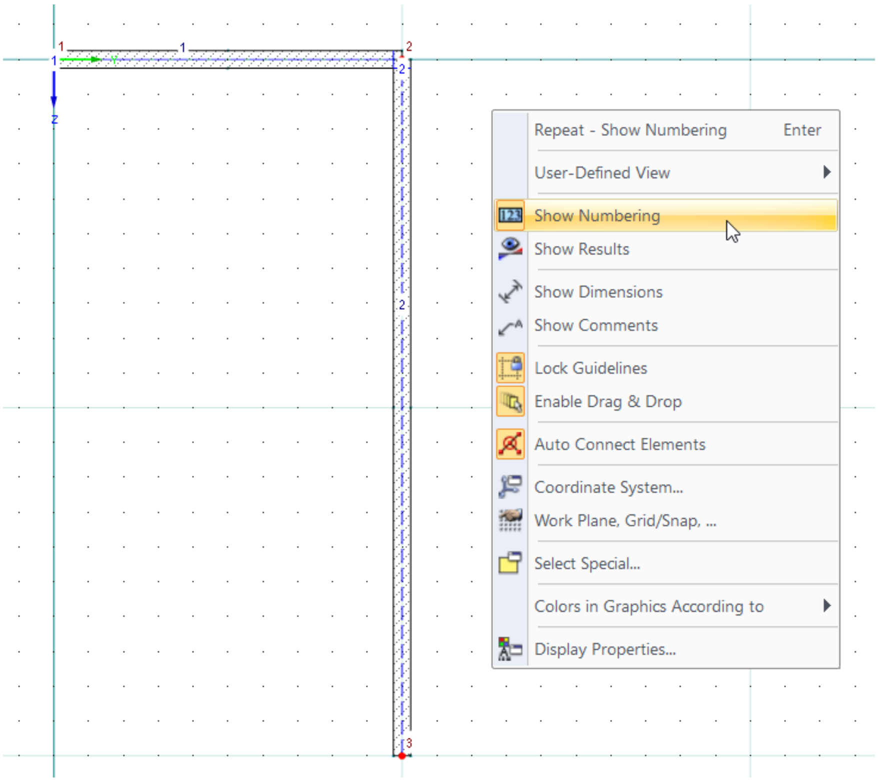


Figure 4.7: Activating the numbering in the shortcut menu

4.3.2 Editing Elements

When we go to element 1 and hover the pointer over it briefly, the quick info shows us, among other information, the element thickness *t* of 10 mm.

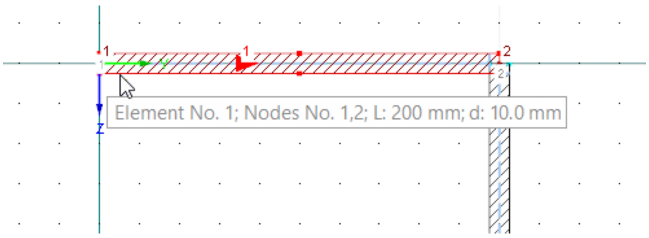


Figure 4.8: Quick info of element

As our flange has a thickness of 12 mm, we need to correct it. We double-click element 1 to open the *Edit Element* dialog box.

Please make sure to double-click the element on its edge to avoid "catching" the blue c-t zone!

We correct the *Thickness t* to **12 mm**.

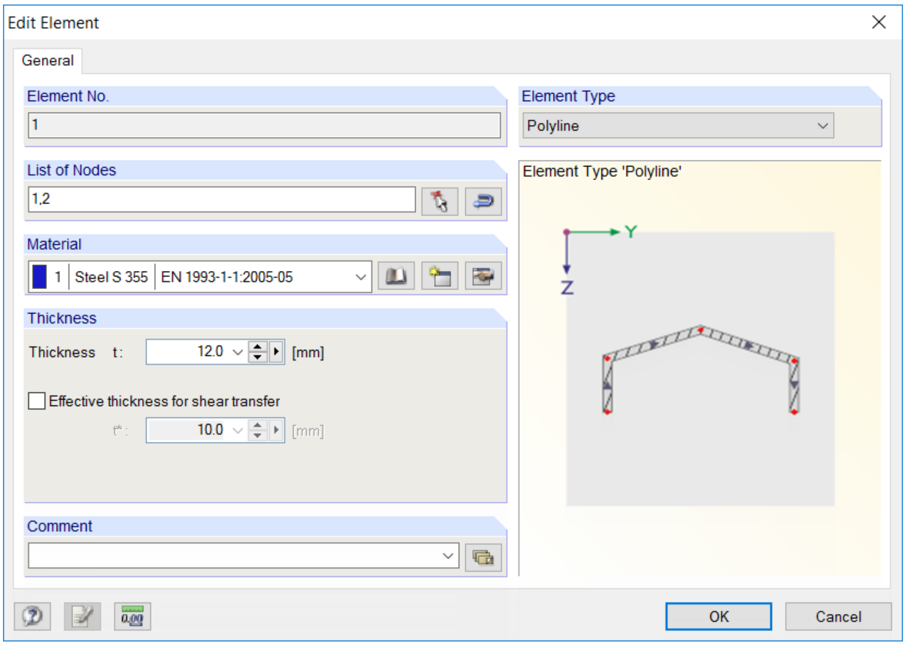


Figure 4.9: Dialog box *Edit Element*

After [OK] the model represented in the work window is refreshed.

4.4 Defining Angle

Now, we connect an unequal angle 200x150x12 to the horizontal element.

4.4.1 Placing Section

We open the cross-section library with the button shown on the left.

Figure 4.10: Cross-Section Library

In the dialog section *Rolled*, we click the button [Angles].

In the dialog box *Rolled Cross-Sections - Angles*, we select the cross-section **L 200x150x12** in the *Table* **LU**.

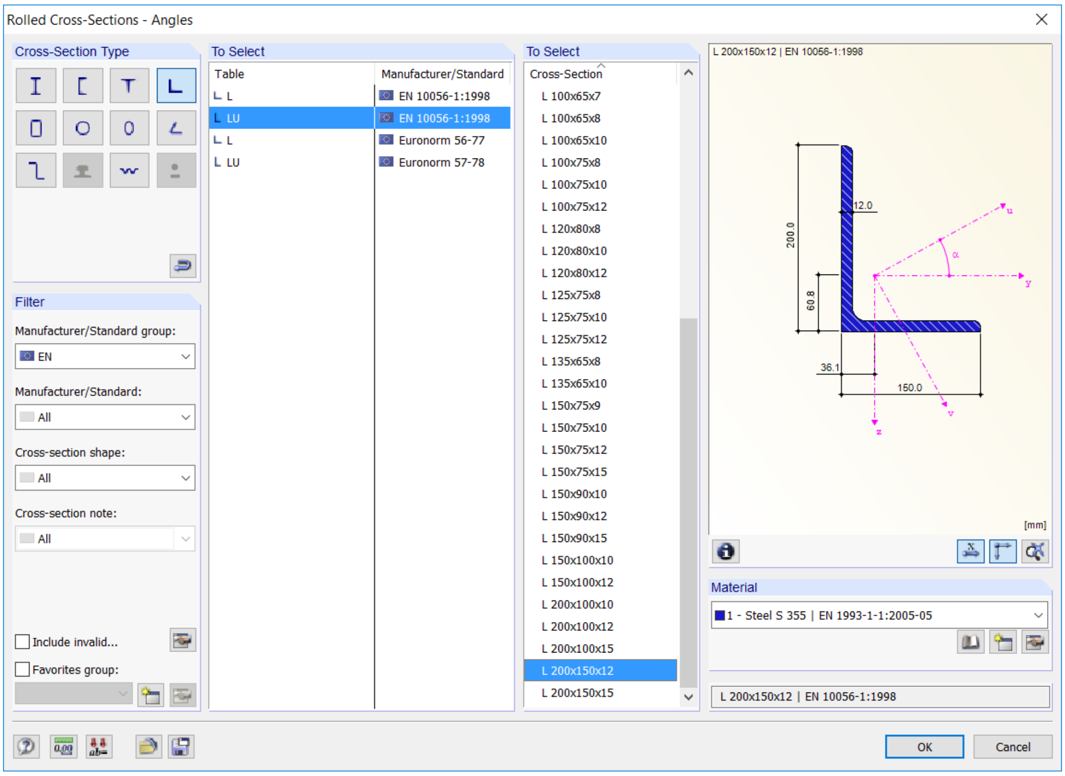


Figure 4.11: Selecting angles in the library

We can check the properties of the angle with the [Info] button.

Furthermore, **Steel S 355** is preset as *Material*. We confirm the dialog box with [OK].

The dialog box *Set Section* appears.

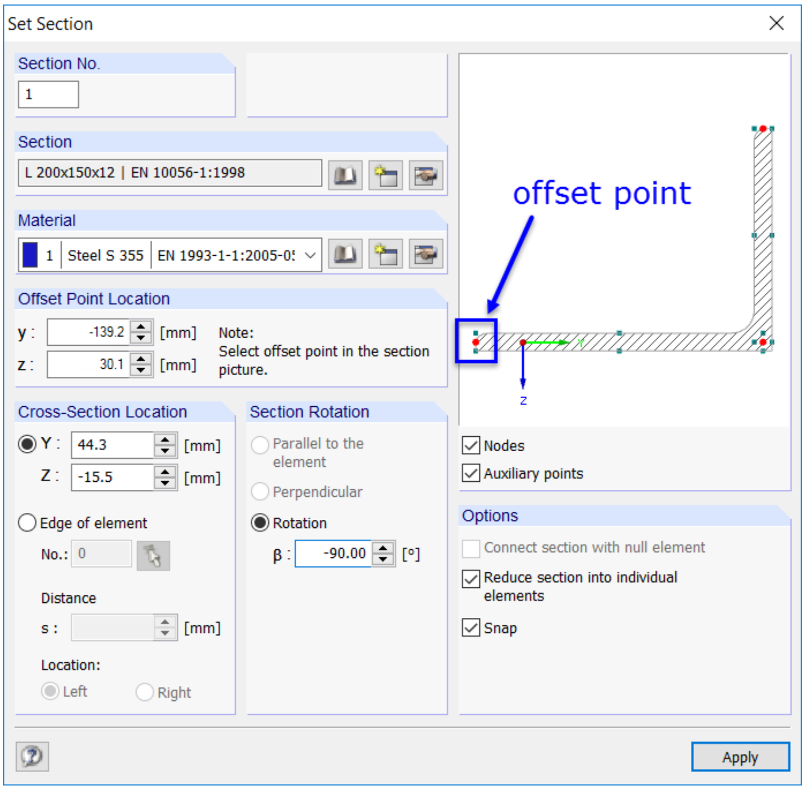


Figure 4.12: Dialog box *Set Section*

We specify a *Section Rotation* β of **-90°** to put the section in a more favorable position.

In addition, we tick the check box **Reduce section into individual elements**.

When we move the angle with the mouse across the screen, we see that the current "auxiliary point" (offset point) for placing the section lies in its centroid. In the angle's section sketch, the auxiliary point is shown in red.

Because we want to connect the section with its long leg, we change the offset point. In the sketch, we click the red node at the left section end so that it is shown in light red.

Now, to place the section, we go to node **2** in the work window. As soon as the node is displayed with its coordinates in the status bar, we press the left mouse button.

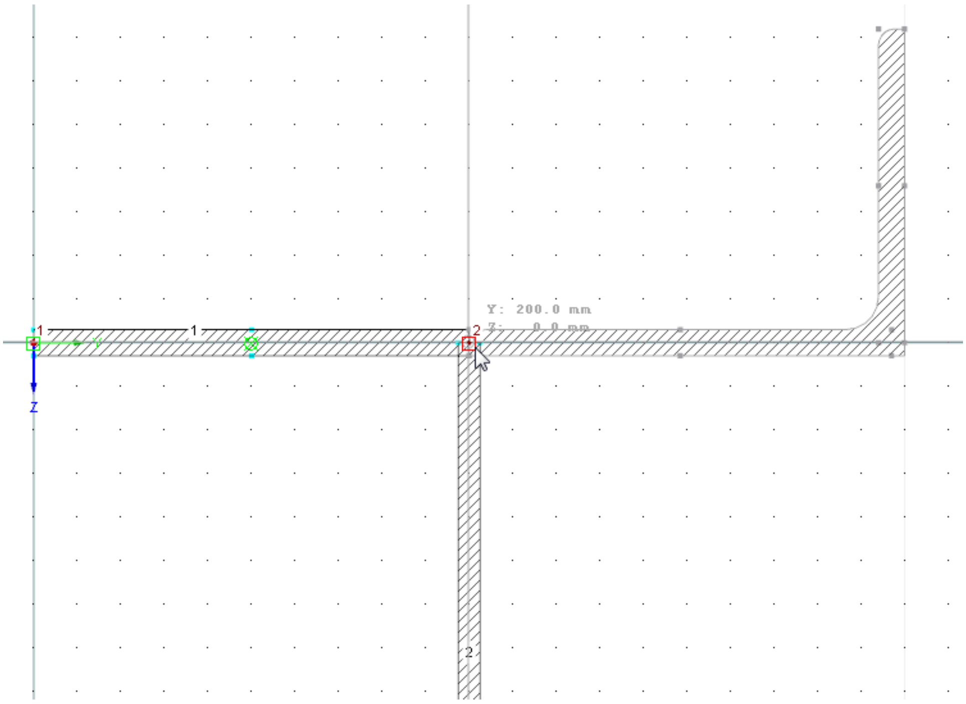


Figure 4.13: Placing angle at node 2

A message appears documenting the result of the section reduction to elements and point elements.

Strictly speaking, it would be correct to connect the angle to the element's edge. But then another element of the thickness zero would be created ensuring the shear stiff connection. The demo restrictions of a maximum of four elements would be exceeded for our example.

We close the function with a right mouse click into the empty work window or with [Esc].

4.4.2 Rotating Section

We adjust the position of the angle with the *Mirror* function.

Selecting objects

Before we can use the editing functions (copy, rotate, mirror), the relevant objects must be determined or "selected".

We draw a window across the section that has just been set – from the left to the right. It is necessary that the window completely includes the elements 3 and 4.

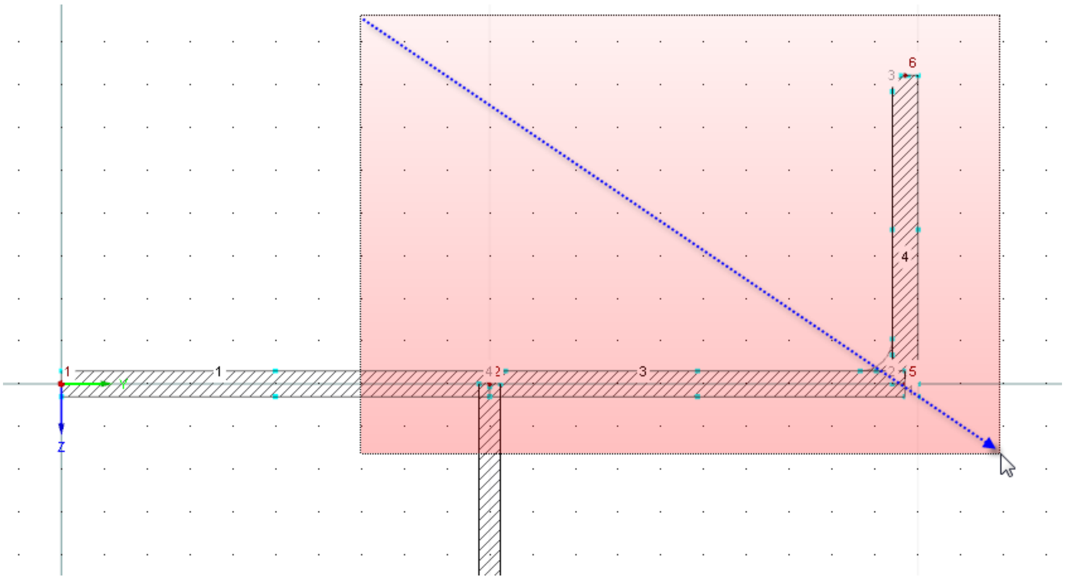


Figure 4.14: Selecting elements with window

How to select objects:

- If you pull up the window from the left to the right, the selection contains only objects that are completely within this window. If you pull up the window from the right to the left, the selection additionally contains those objects that are cut by the window.
- The selection is acting "alternatively": When you click an object (node, element, point element), the selection of an already selected object will be canceled. Only the new object is selected. To add the object to an existing selection, hold down the [Shift] key when clicking.

Mirroring objects

We open the **Mirror** function with the button shown on the left.

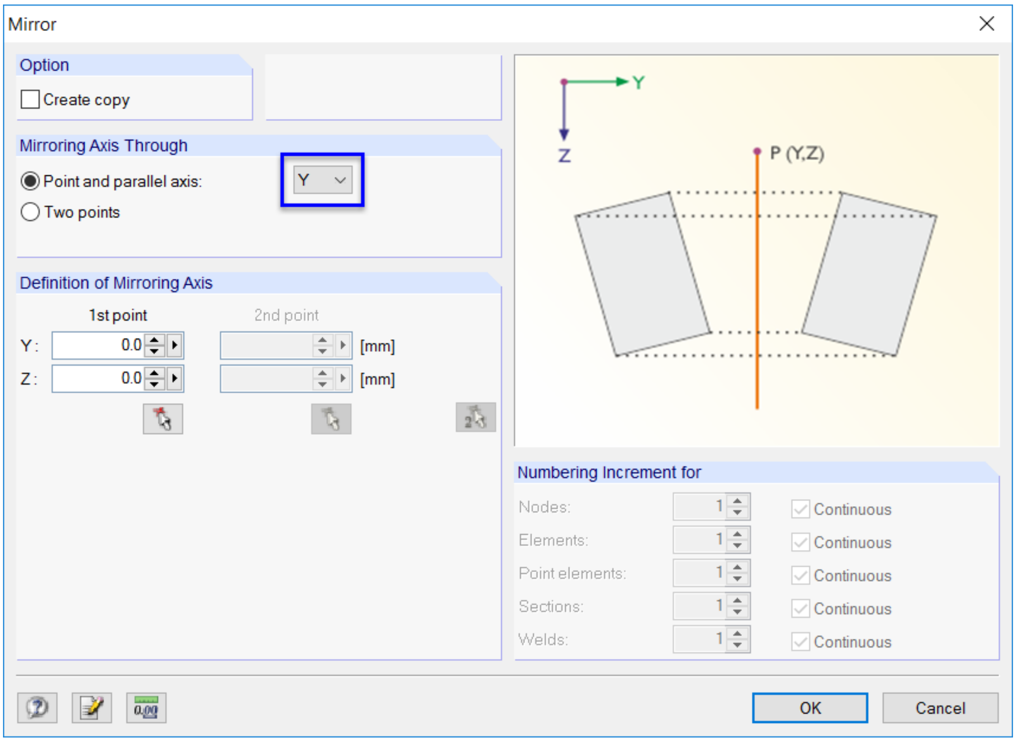


Figure 4.15: Dialog box *Mirror*

It is important that the check box for *Create copy* is not ticked in the *Mirror* dialog box.

Now, we mirror the angle around the origin (0.0/0.0) and the *parallel axis* **Y**.

After [OK] we see that SHAPE-THIN has put the angle into the correct position.

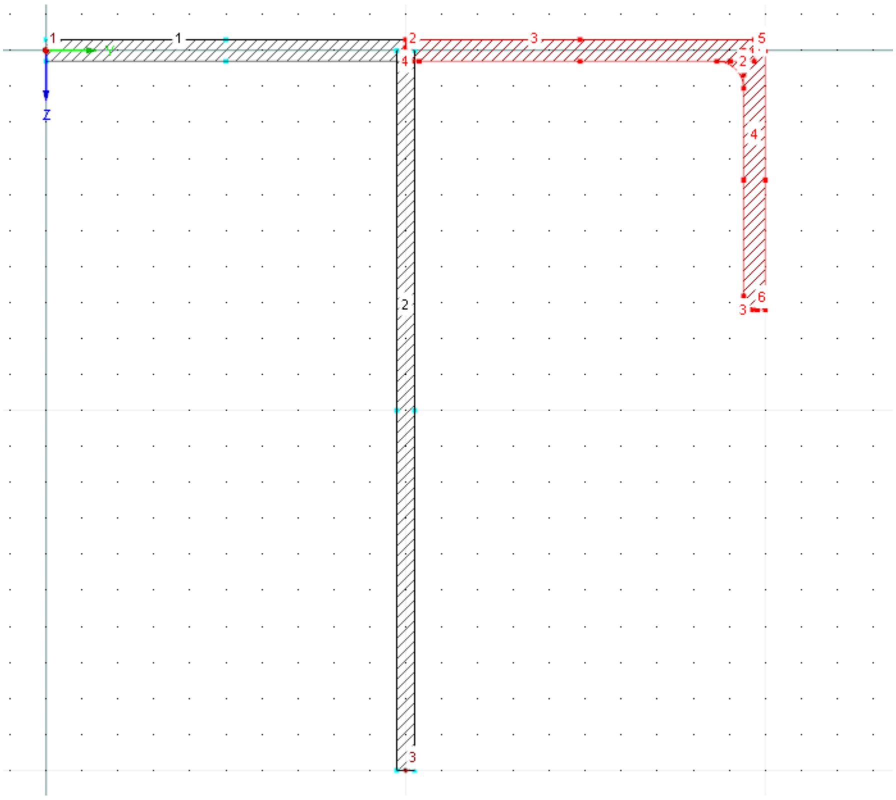


Figure 4.16: Mirrored angle

4.5 Defining Welds

Now, we adjust the connection zone between the elements. Then, we model the weld connections.

4.5.1 Deleting Point Elements

We zoom in the zone of the connection between the elements and the angle. For zooming in we can use the wheel button or the button [Zoom with Window].

We see an opening in the connection zone which has been caused by the mirrored angle. We eliminate it by right-clicking the point element **4** and selecting the option **Delete Point Element** in the shortcut menu.

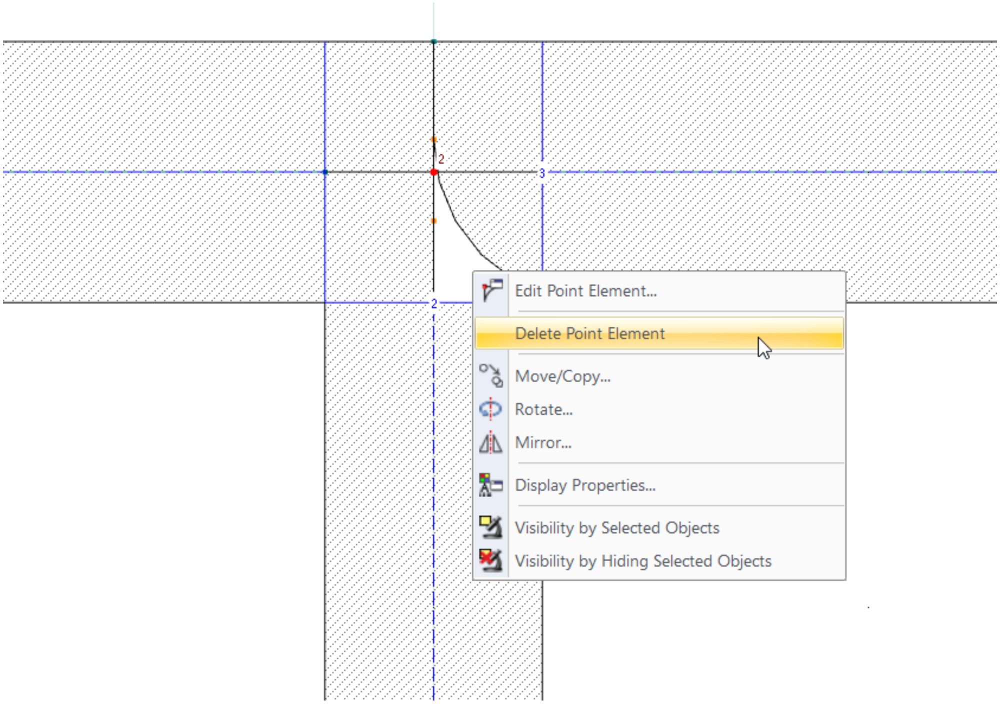


Figure 4.17: Deleting Point Elements

4.5.2 Placing Welds

We define a double fillet weld of 6 mm between the web and the flange.

With the button shown on the left or selecting on the menu

Insert → **Model Data** → **Welds** → **Graphically**,

we open the dialog box *Set Weld*.

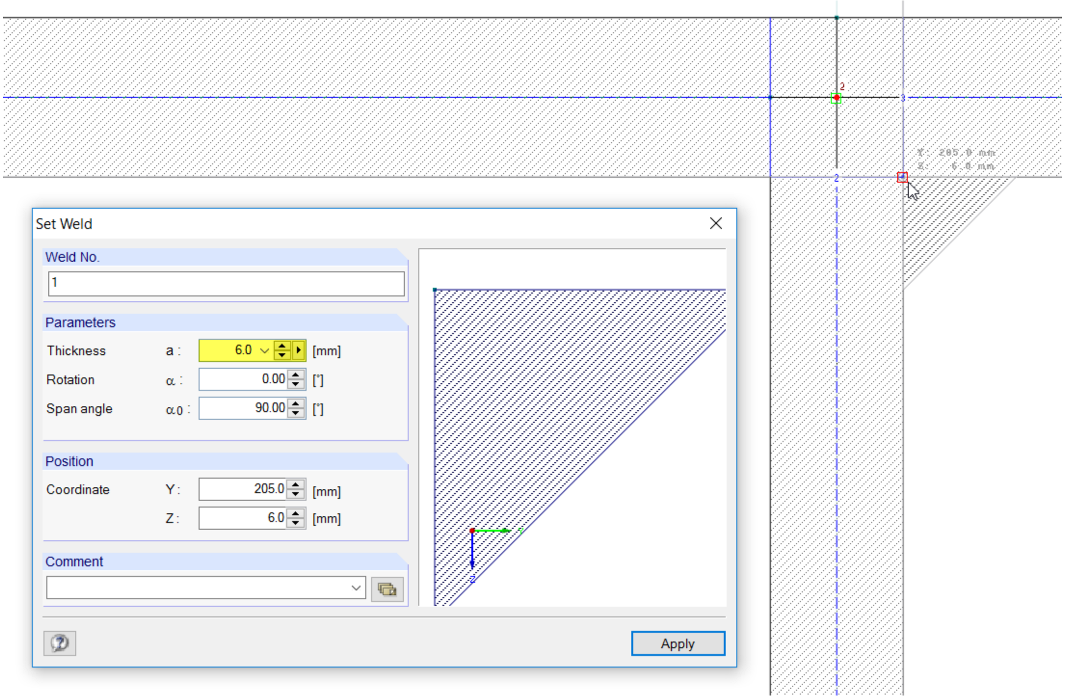


Figure 4.18: Dialog box *Set Weld*

We change the *Thickness* to **6 mm**. To set the weld, we go to the point with the coordinates (**205.0/6.0**) in the work window. As soon as the reticle is snapped in the corner (red square on the pointer), we press the left mouse button.

For setting the second weld we change the *Rotation* α to **90.0°**. Then, we place the weld at the point with the coordinates (**195.0/6.0**).

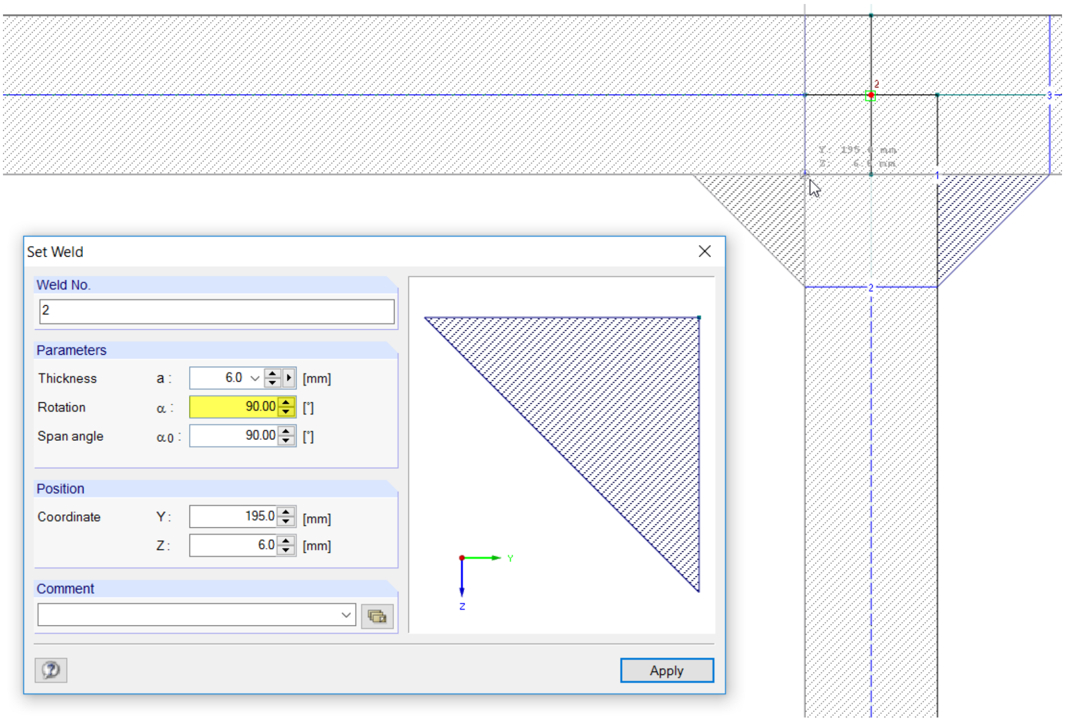


Figure 4.19: Dialog box *Set Weld*

We close the function with a right mouse click into the empty work window or with [Esc].

Finally, we have entered all geometry data.

4.6 Checking Cross-Section Parts

Checking data in navigator and tables

All entered objects can be found in the directory tree of the *Data* navigator and in the tabs of the table. The entries in the navigator can be opened with a click on the ⊞ sign. To switch between the tables, we click on the table tabs.

We can hide and display the navigator as well as the tables by selecting **View** → **Navigator** or **Table** on the menu. We can also use the corresponding toolbar buttons.

In the tables, the objects are organized in various tabs. The *Cross-Section* L 200x150x12 is missing as we have selected the option *Reduce section into individual elements* in the settings.

Checking c/t parts

SHAPE-THIN automatically creates cross-section parts that are required to determine the c/t ratios according to Eurocode 3. At the same time, the support conditions (outstand flanges or internal compression parts) and the reduced lengths due to point elements and welds are identified.

In the graphic, the *c/t-Parts* represented in the center lines of the elements are shown in light blue. In table 1.7, the data is listed numerically.

Graphics and tables are interactive: For example, to find a c/t part in the table, we go to table *1.7 Cross-Section Parts for Classification According to EN 1993-1*. When we click a c/t part in the work window, its corresponding table row is highlighted in color.

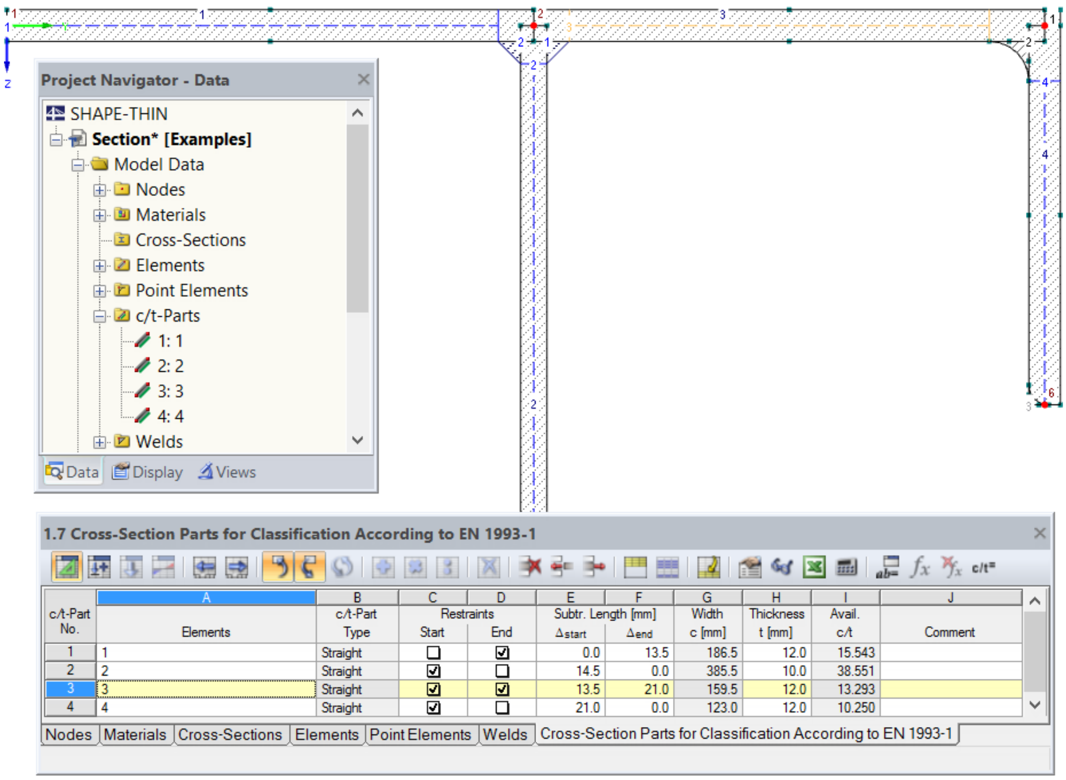


Figure 4.20: c/t parts in graphic, navigator and table

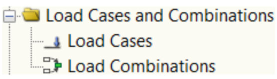
Saving data

Now, we have finally entered all model data. We save our cross-section by clicking on the menu

File → **Save**

or we use the corresponding button in the toolbar.

5 Loads



The *Data* navigator lists two entries in the folder *Load Cases and Combinations*:

- Load cases
- Load combinations

In load cases, it is possible to define the internal forces, for example, due to self-weight, snow or wind load. In load combinations, we can organize the internal forces of load cases that are superimposed with partial safety factors according to particular combination expressions.

Now, we define the internal forces in two separate load cases as specified in Chapter 2.2.

5.1 Load Case 1: Tension and Bending

Creating a load case

We use the button to create a new load case.

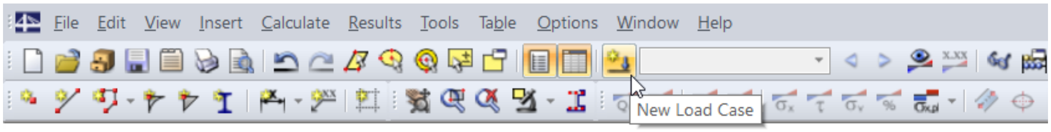


Figure 5.1: Button *New Load Case*

The dialog box *Edit Load Cases and Combinations* appears.

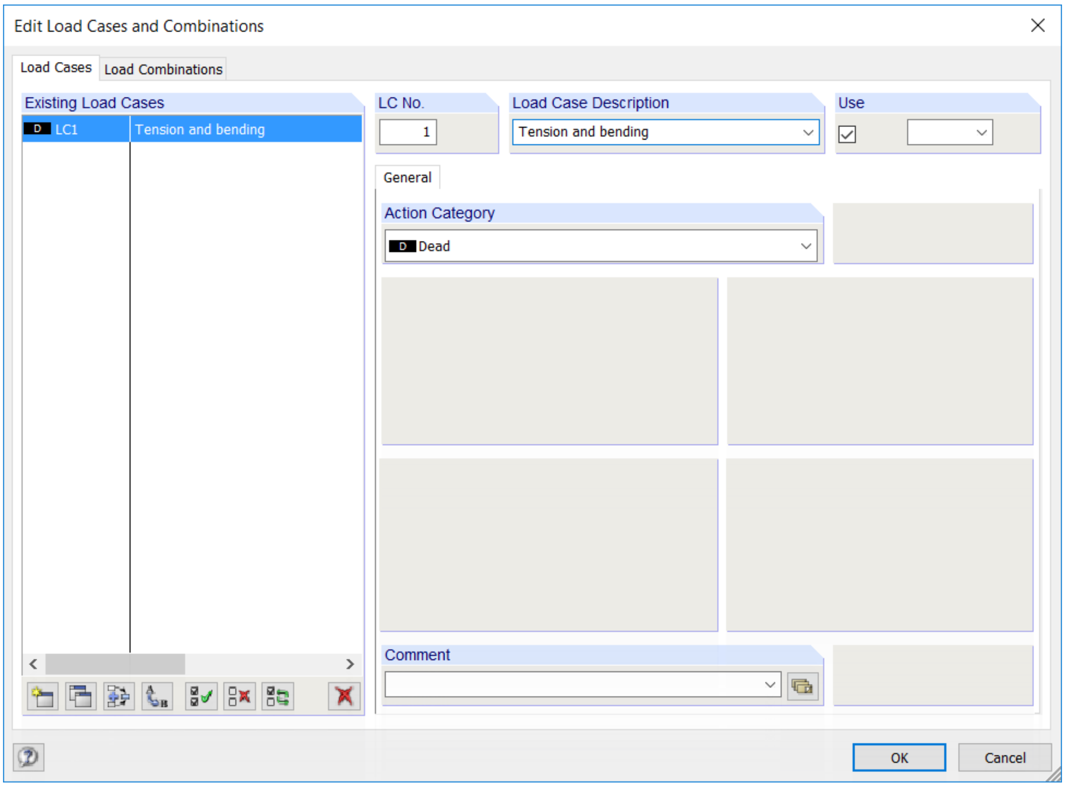


Figure 5.2: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

Load case number *1* is preset with the action category *Dead*. We enter the *Load Case Description* **Tension and bending**.

Then, we confirm the input with [OK] and close the dialog box.

Defining internal forces

In the tables toolbar, we click the button .

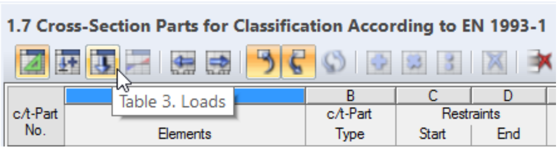
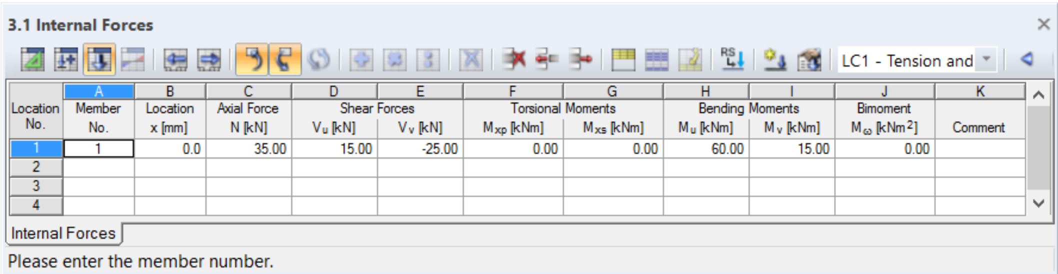


Figure 5.3: Button *Table 3. Loads*

The table *3.1 Internal Forces* is displayed.

We enter the forces and moments as follows.


3.1 Internal Forces

Location No.	A Member No.	B Location x [mm]	C Axial Force N [kN]	D Shear Forces V_u [kN]	E Shear Forces V_v [kN]	F Torsional Moments M_{xp} [kNm]	G Torsional Moments M_{xs} [kNm]	H Bending Moments M_u [kNm]	I Bending Moments M_v [kNm]	J Bimoment M_ω [kNm²]	K Comment
1	1	0.0	35.00	15.00	-25.00	0.00	0.00	60.00	15.00	0.00	
2											
3											
4											

Internal Forces

Please enter the member number.

Figure 5.4: Table *3.1 Internal Forces*

The analysis requires the definition of a *Member* number and a *Location x*. For our example, however, this data is not relevant. The allocation of members and member design locations is especially important for the import of internal forces from RSTAB/RFEM.

The algebraic signs of forces and moments are determined in SHAPE-THIN by the following rules:

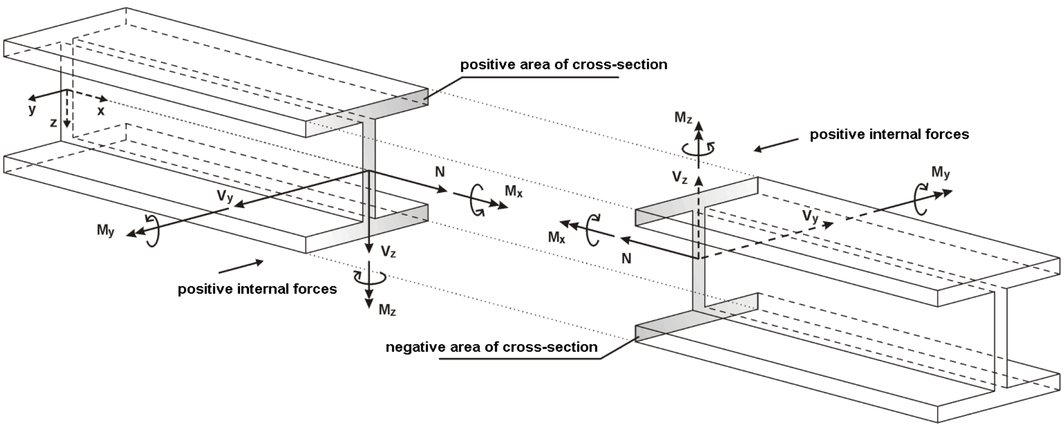


Figure 5.5: Definition of internal forces

The bending moment M_y is positive if tensile stresses occur on the positive member side (in the direction of the z-axis). M_z is positive if compressive stresses occur on the positive member side (in the direction of the y-axis). The sign definition for torsional moments, axial forces and shear forces conforms to the usual conventions: These internal forces are positive if they act on the positive section in a positive direction.

As we can see in the column titles of the shear forces and the moments in table 3.1, the internal forces are not related to the global axes *y* and *z* as specified in Chapter 2.2 but to the principal axes **u** and **v**. We will adjust it later (see Chapter 6.1, page 23). The entered values won't be converted; only the column titles will be changed.

5.2 Load Case 2: Compression and Bending

We create a new load case for the second constellation of internal forces. We can use the menu

Insert → Loads → New Load Case

or the button in the table toolbar (to the left of the load case list).

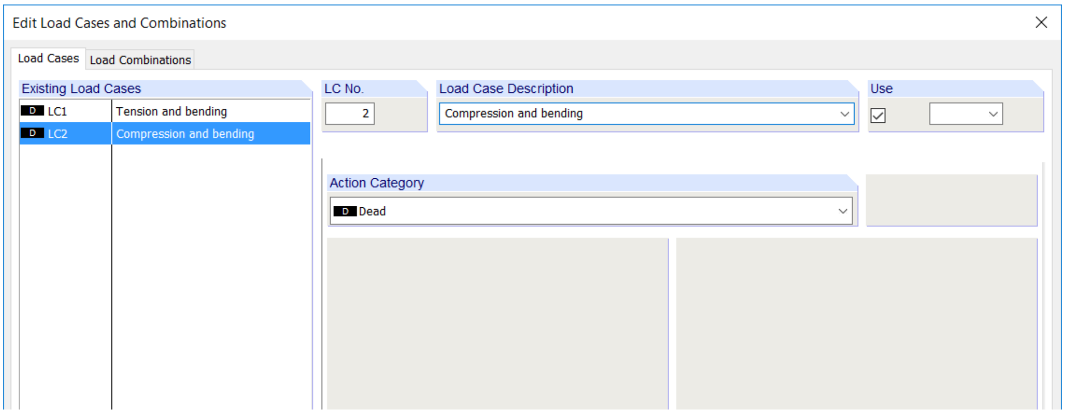
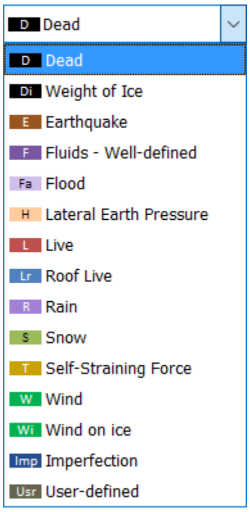


Figure 5.6: Dialog box *Edit Load Cases and Combinations*



We enter the *Load Case Description* **Compression and bending**.

Then, we change the *Action Category* and select **Dead** loads in the list.

This time, we enter the internal forces in a dialog box that we open on the menu

Insert → Loads → 3.1 Internal Forces → Dialog Box.

The forces and moments are as follows as described in Chapter 2.2:

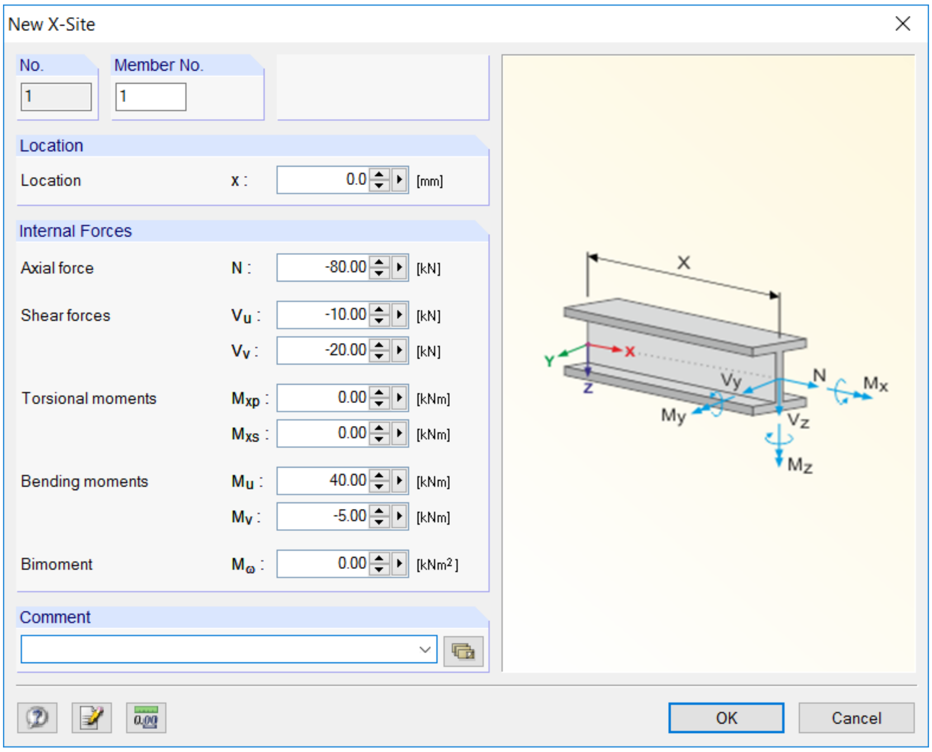


Figure 5.7: Dialog box *New X-Site*

We confirm the dialog box with [OK]. Then, we check the result in the table *3.1 Internal Forces*.

5.3 Checking Load Cases

Now, both load cases are defined. In the table, we can use the buttons ◁ and ▷ to switch between the load cases.

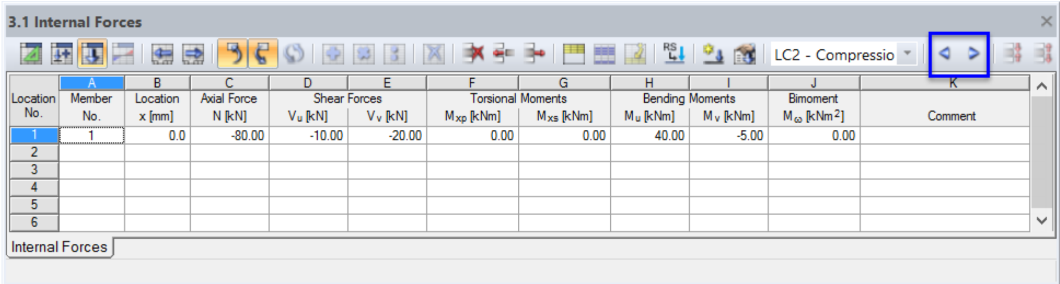


Figure 5.8: Switching between load cases

Again, the data entered for the internal forces is reflected in the tree of the *Data* navigator.

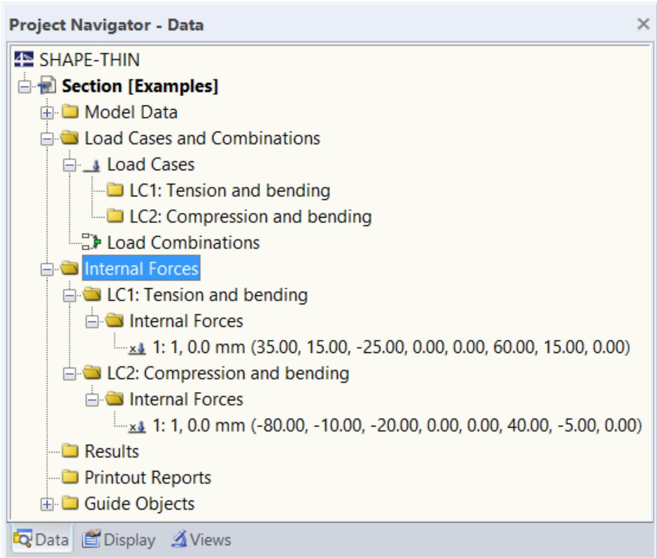


Figure 5.9: Load cases and internal forces in *Data* navigator

We don't define any internal forces for *Load Combinations*.

It is recommended to [Save] the entered data again.

6 Calculation

Before we start the calculation, we check the calculation parameters and the input data.

6.1 Adjusting Calculation Parameters

We access the calculation parameters by selecting on the menu

Calculate → Calculation Parameters

or by using the corresponding button in the toolbar.

The dialog box *Calculation Parameters* opens.

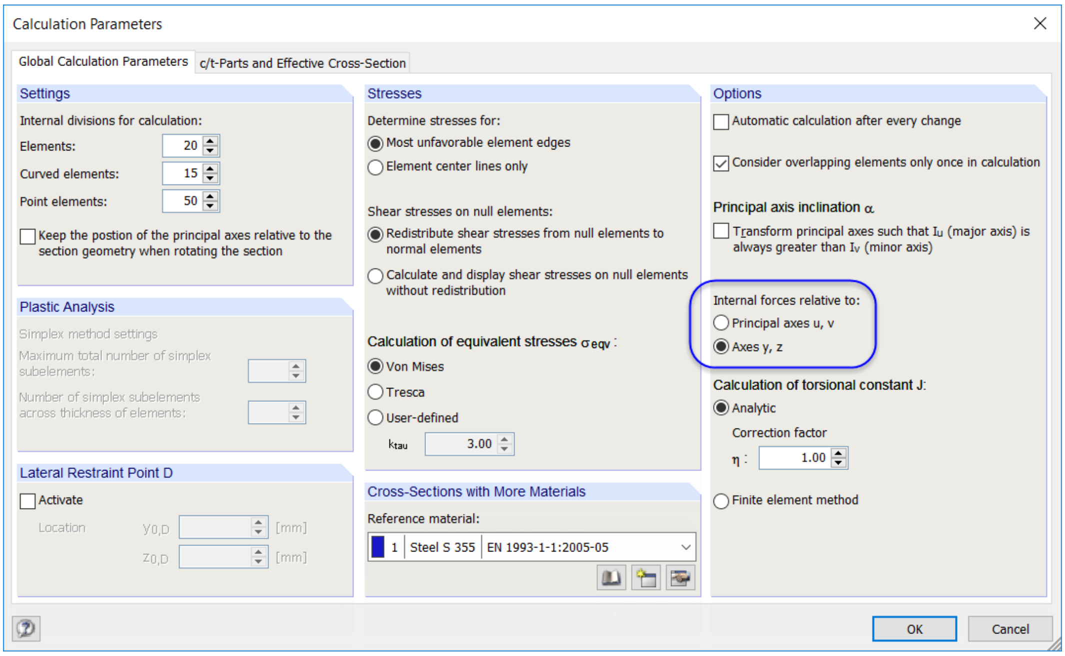


Figure 6.1: Dialog box *Calculation Parameters*, tab *Global Calculation Parameters*

In the first tab, we change the relation of the internal forces to the **Axes y, z**. The entered forces and moments won't be converted.

In the second tab, we check if the *Code* **EN 1993-1-1 and EN 1993-1-5** is set.

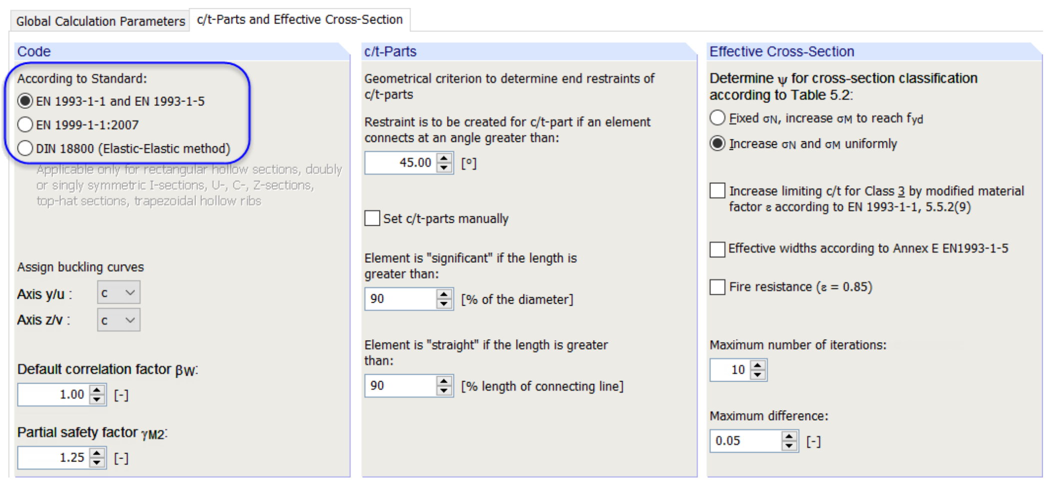


Figure 6.2: Dialog box *Calculation Parameters*, tab *c/t-Parts and Effective Cross-Section*

We confirm the changes with [OK].

6.2 Checking Input Data

SHAPE-THIN offers several possibilities to check data.

6.2.1 Plausibility Check

We select on the menu

Tools → Plausibility Check

and define the following settings in the dialog box *Plausibility Check*.

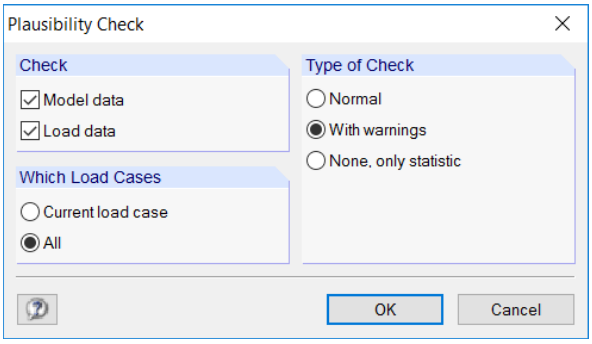


Figure 6.3: Dialog box *Plausibility Check*

If the program does not detect any inconsistencies after clicking [OK], a message appears showing a summary of the cross-section and load case data.

6.2.2 Checking Interconnecting Elements

Now, we check if there is an interconnecting cross-section. We can access this checking option by selecting on the menu

Tools → Model Check → Check for Interconnecting Elements.

SHAPE-THIN displays the following result.

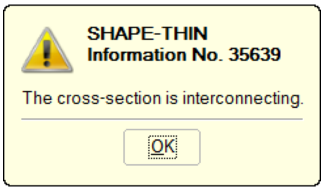


Figure 6.4: Result of model check

If this was not the case, SHAPE-THIN will calculate the cross-section according to the theory of stiffening shear wall systems without considering the parallel axis theorem (Steiner).

6.3 Calculating the Cross-Section

We start the calculation on the menu

Calculate → Calculate All

or by using the corresponding button in the toolbar.

The results are displayed immediately after the calculation.

7 Results

7.1 Graphical Results

The statical moments Q_u are represented as isobands on the cross-section. These gradients are related to the cross-section's principal axis *u* that is also represented in the graphic. Table *4.1 Gross Section Properties* lists the cross-section parameters.

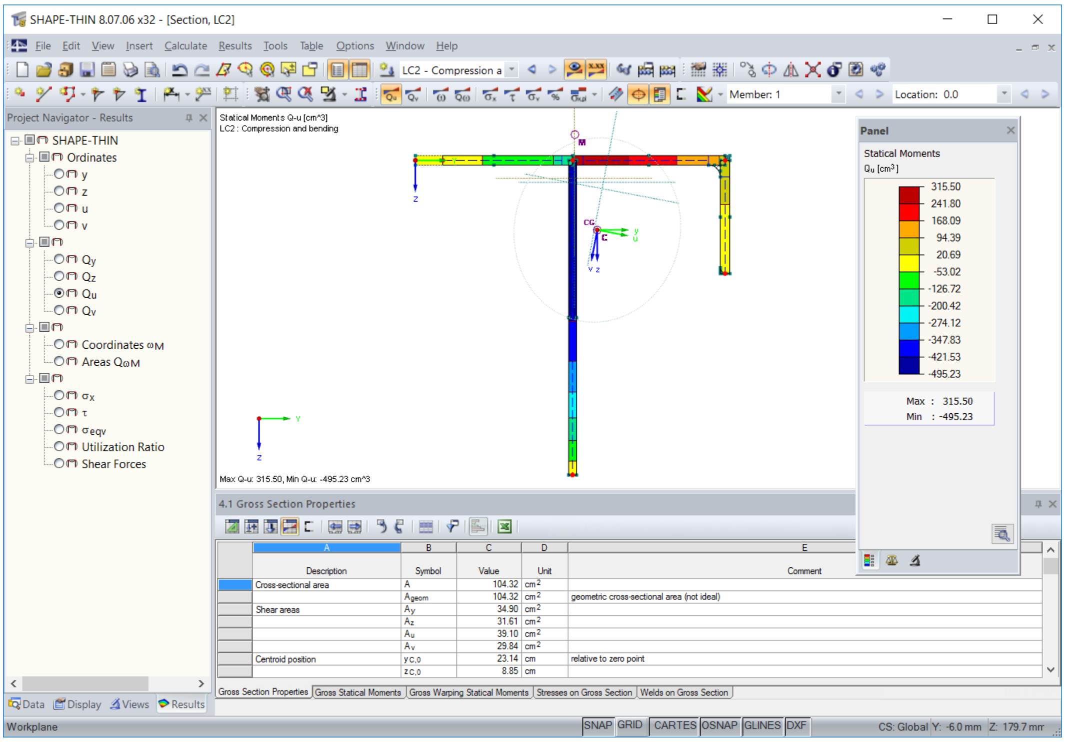
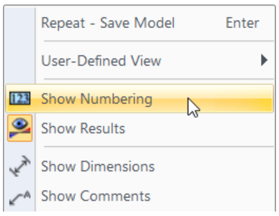


Figure 7.1: Cross-section with gradient of statical moments and principal axes (gross section)



Now, we hide the numbering of nodes and elements again: With a right-click in an empty space of the work window we open the general shortcut menu (see Figure 4.7, page 10) where we deactivate the entry *Show Numbering*.

Displaying effective cross-section

With the button [Effective Parts] we can switch between the result values of the gross cross-section (button "off") and of the effective section (button "on"). To evaluate the effective cross-section, we switch this function **on**.

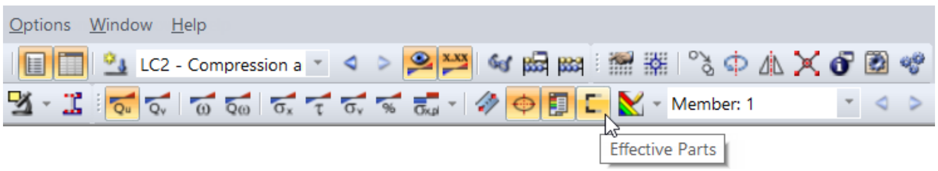


Figure 7.2: Button [Effective Parts]

Load case LC2 *Compression and bending* was last set. SHAPE-THIN displays the cross-section values taking into account the failing parts available in this constellation of internal forces. In the left flange, we can see the reduced width of the cross-section part subjected to compression (see Figure 7.3). The statical moments change accordingly.

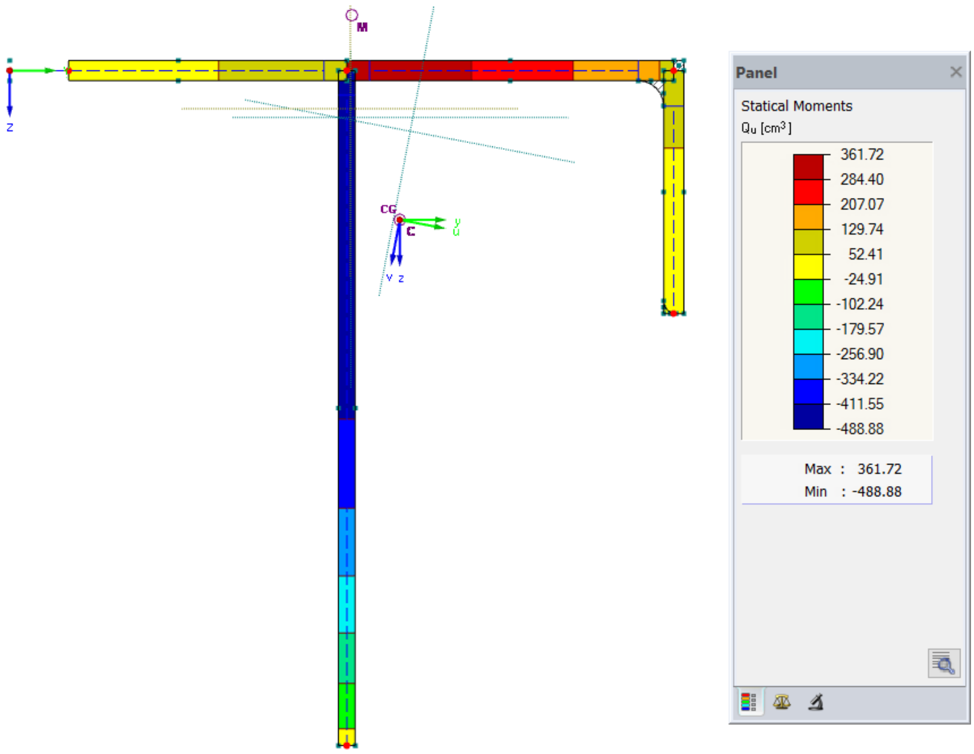
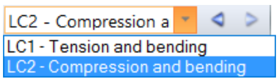


Figure 7.3: Statical moments Q_u for LC2 on effective cross-section

Selecting load cases



We can switch between the results of the load cases with the buttons ◄ and ► in the toolbar (to the right of the load case list) as we already know from the input. It is also possible to use the list.

Selecting results in navigator

A new navigator, the fourth in its series, manages the result categories for the graphical display. We have access to the *Results* navigator only when the results display is active. The results can be displayed and hidden in the *Display* navigator. We can also use the [Show Results] button.

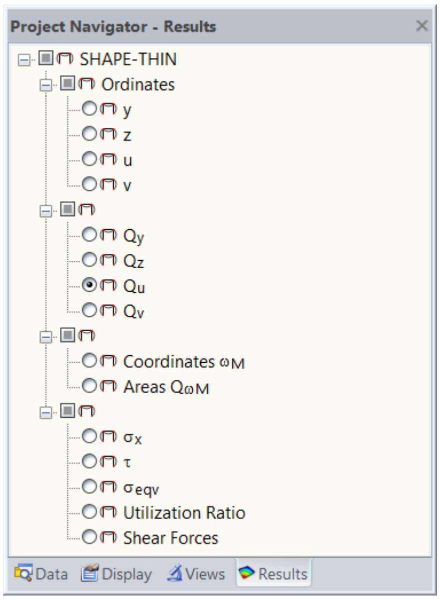


Figure 7.4: *Results* navigator

With the control fields we can set the cross-section properties and stresses available in both load cases for the graphical display.

Normal stresses

In the *Results* navigator, we select the results category **Stresses** σ_x. We can also use the button shown on the left.

In the toolbar, we set load case **LC1**.

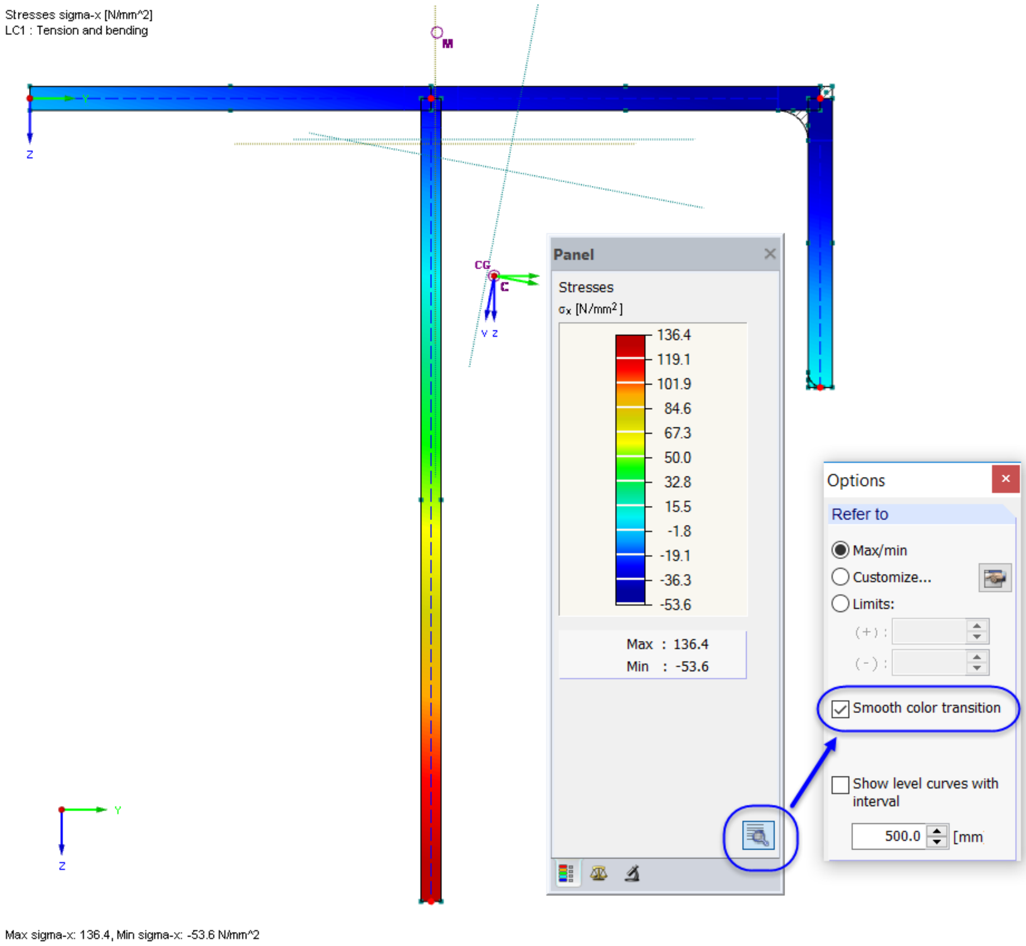


Figure 7.5: Normal stresses σ_x with smooth color transition

When we click the button  in the panel, another dialog box opens where we can activate **Smooth color transition** for the stress graphic.

In the cross-section of LC1, which is subjected to tension, all cross-section parts are effective.

Shear stresses

We set the results category **Stresses** τ in the *Results* navigator. We can also use the button shown on the left.

Then, we click the list button [Results As Isobands] to switch **Off** the surface representation. Now, the shear stresses are displayed with hatching.

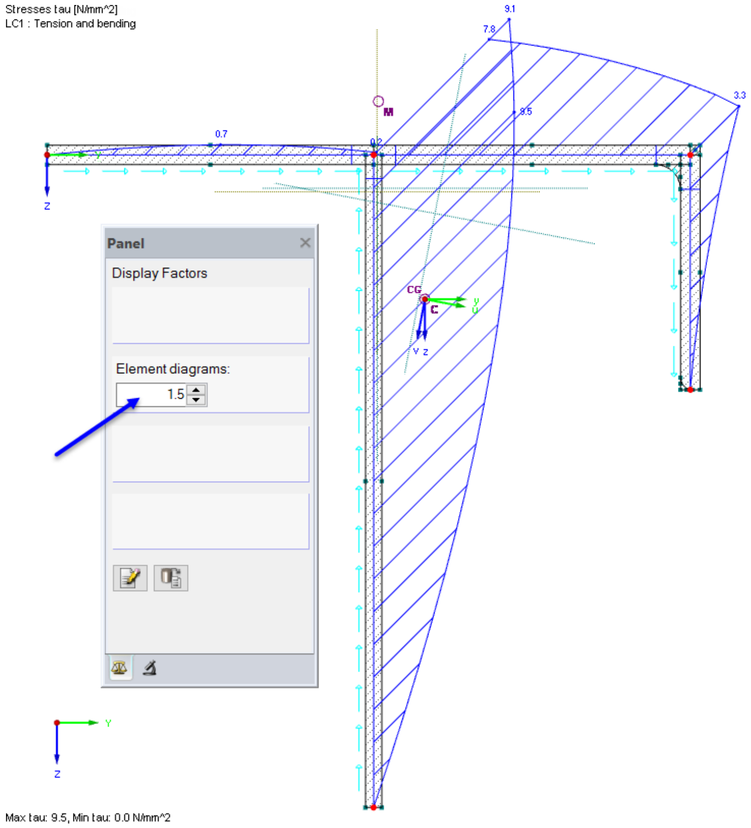


Figure 7.6: Shear stresses τ with scaled up diagrams

In the panel tab *Factors*, we can use the spin buttons to scale the *Element diagrams*.

Toolbar

The *Results* toolbar offers more functions for evaluating results.

Figure 7.7: Toolbar *Results*

The buttons next to the symbols of the section diagrams and stresses have the following meanings:

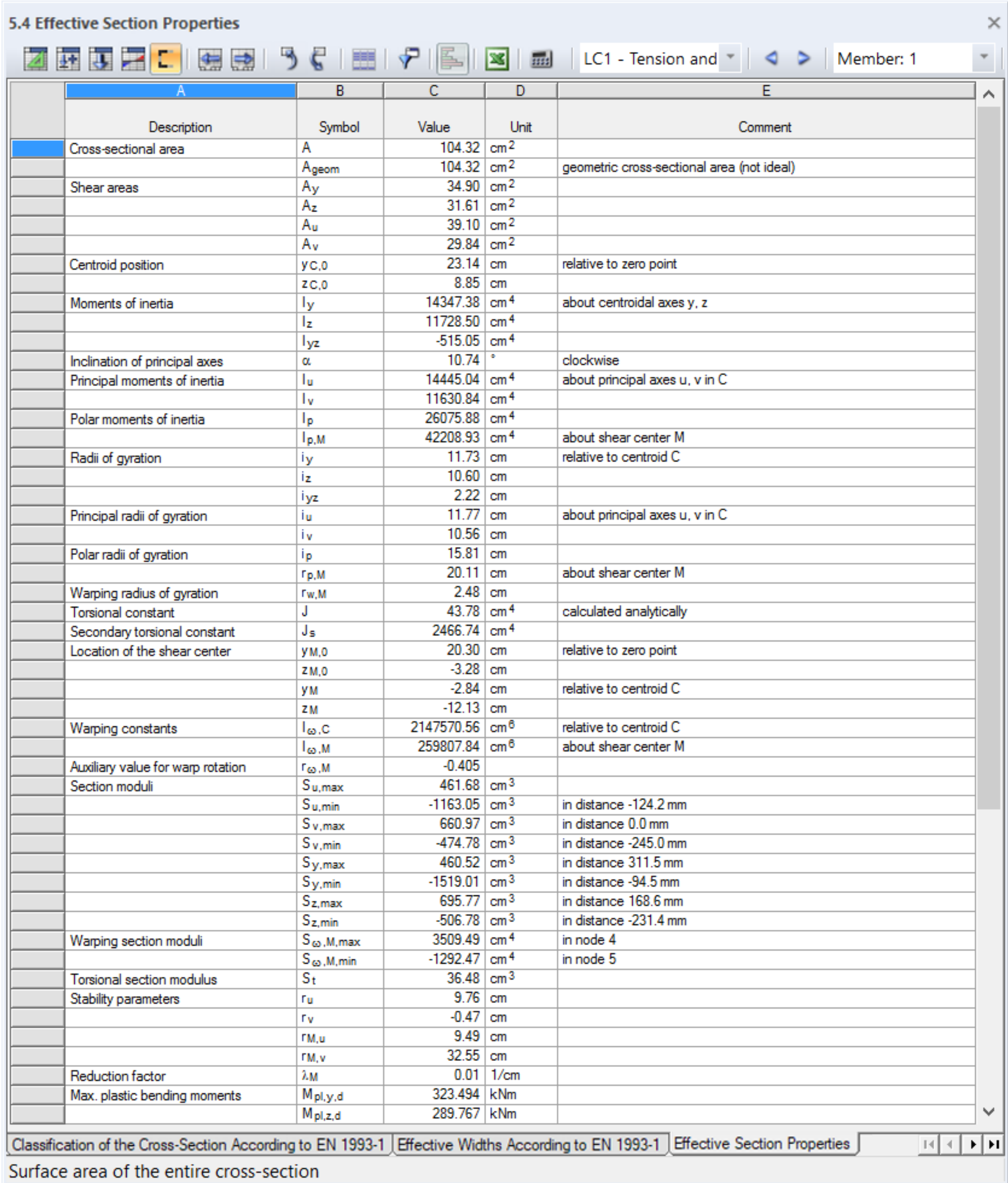
Button	Function
	Shows the element thicknesses or only the center lines
	Displays and hides the inertia ellipse
	Displays and hides the control panel
	Shows the cross-section with or without effective widths
	List button: shows the diagrams as isobands or two-colored lines

Table 7.1: Buttons in *Results* toolbar

7.2 Results Tables

The results are also numerically available and listed in tables.

We set Table *5.4.Effective Section Properties* to check on the section properties of the effective cross-section. They depend on the load case because the effective widths are taken into account.



5.4 Effective Section Properties

LC1 - Tension and | Member: 1

Description	Symbol	Value	Unit	Comment
Cross-sectional area	A	104.32	cm²	
	A_{geom}	104.32	cm²	geometric cross-sectional area (not ideal)
Shear areas	A_y	34.90	cm²	
	A_z	31.61	cm²	
	A_u	39.10	cm²	
	A_v	29.84	cm²	
Centroid position	$y_{C,0}$	23.14	cm	relative to zero point
	$z_{C,0}$	8.85	cm	
Moments of inertia	I_y	14347.38	cm⁴	about centroidal axes y, z
	I_z	11728.50	cm⁴	
	I_{yz}	-515.05	cm⁴	
Inclination of principal axes	α	10.74	°	clockwise
Principal moments of inertia	I_u	14445.04	cm⁴	about principal axes u, v in C
	I_v	11630.84	cm⁴	
Polar moments of inertia	I_p	26075.88	cm⁴	
	$I_{p,M}$	42208.93	cm⁴	about shear center M
Radii of gyration	i_y	11.73	cm	relative to centroid C
	i_z	10.60	cm	
	i_{yz}	2.22	cm	
Principal radii of gyration	i_u	11.77	cm	about principal axes u, v in C
	i_v	10.56	cm	
Polar radii of gyration	i_p	15.81	cm	
	$r_{p,M}$	20.11	cm	about shear center M
Warping radius of gyration	$r_{w,M}$	2.48	cm	
Torsional constant	J	43.78	cm⁴	calculated analytically
Secondary torsional constant	J_s	2466.74	cm⁴	
Location of the shear center	$y_{M,0}$	20.30	cm	relative to zero point
	$z_{M,0}$	-3.28	cm	
	y_M	-2.84	cm	relative to centroid C
	z_M	-12.13	cm	
Warping constants	$I_{\omega,C}$	2147570.56	cm⁶	relative to centroid C
	$I_{\omega,M}$	259807.84	cm⁶	about shear center M
Auxiliary value for warp rotation	$r_{\omega,M}$	-0.405		
Section moduli	$S_{u,max}$	461.68	cm³	
	$S_{u,min}$	-1163.05	cm³	in distance -124.2 mm
	$S_{v,max}$	660.97	cm³	in distance 0.0 mm
	$S_{v,min}$	-474.78	cm³	in distance -245.0 mm
	$S_{y,max}$	460.52	cm³	in distance 311.5 mm
	$S_{y,min}$	-1519.01	cm³	in distance -94.5 mm
	$S_{z,max}$	695.77	cm³	in distance 168.6 mm
	$S_{z,min}$	-506.78	cm³	in distance -231.4 mm
Warping section moduli	$S_{\omega,M,max}$	3509.49	cm⁴	in node 4
	$S_{\omega,M,min}$	-1292.47	cm⁴	in node 5
Torsional section modulus	S_t	36.48	cm³	
Stability parameters	r_u	9.76	cm	
	r_v	-0.47	cm	
	$r_{M,u}$	9.49	cm	
	$r_{M,v}$	32.55	cm	
Reduction factor	λ_M	0.01	1/cm	
Max. plastic bending moments	$M_{pl,y,d}$	323.494	kNm	
	$M_{pl,z,d}$	289.767	kNm	

Classification of the Cross-Section According to EN 1993-1 | Effective Widths According to EN 1993-1 | Effective Section Properties

Surface area of the entire cross-section

Figure 7.8: Table *4.1 Section Properties*

We can go to the other results tables by clicking the table tabs.

Statical moments, warping statical moments and stresses are displayed numerically each on the start and end nodes as well as in element centers.

Filtering stresses

We navigate to Table *5.7 Stresses on Effective Section*. With the table button [Result Filter] we can adjust the results rows in a separate dialog box.

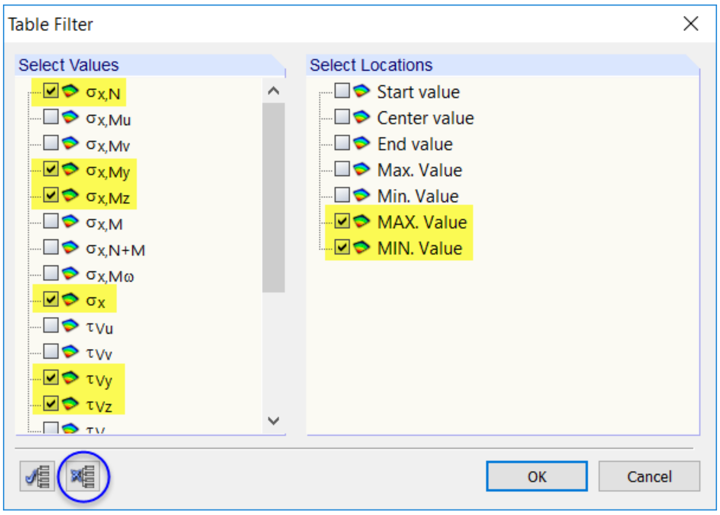
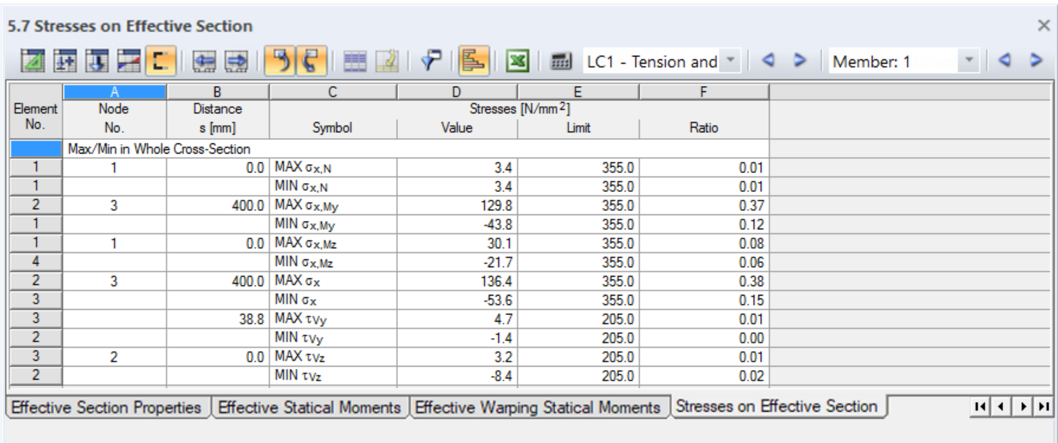


Figure 7.9: Dialog box *Table Filter*

We can remove all check marks by clicking the button. Then, we select some relevant stress types (see Figure 7.9). Furthermore, we want to display only the **MAX** and **MIN** values of the entire section.

After clicking [OK], Table *5.7 Stresses on Effective Section* shows only the extreme values of the selected stresses.



5.7 Stresses on Effective Section

Element No.	A Node No.	B Distance s [mm]	C Symbol	D Stresses [N/mm²] Value	E Limit	F Ratio
	Max/Min in Whole Cross-Section					
1	1	0.0	MAX $\sigma_{x,N}$	3.4	355.0	0.01
1			MIN $\sigma_{x,N}$	3.4	355.0	0.01
2	3	400.0	MAX $\sigma_{x,My}$	129.8	355.0	0.37
1			MIN $\sigma_{x,My}$	-43.8	355.0	0.12
1	1	0.0	MAX $\sigma_{x,Mz}$	30.1	355.0	0.08
4			MIN $\sigma_{x,Mz}$	-21.7	355.0	0.06
2	3	400.0	MAX σ_x	136.4	355.0	0.38
3			MIN σ_x	-53.6	355.0	0.15
3		38.8	MAX τ_{Vy}	4.7	205.0	0.01
2			MIN τ_{Vy}	-1.4	205.0	0.00
3	2	0.0	MAX τ_{Vz}	3.2	205.0	0.01
2			MIN τ_{Vz}	-8.4	205.0	0.02

Figure 7.10: Table *5.7 Stresses on Effective Section* with filtered results

As in the graphics, we can use the buttons and to switch between the two load cases.

Checking classification

The details of the classification are listed in Table *5.2 Classification of the Cross-Section According to EN 1993-1*.

5.2 Classification of the Cross-Section According to EN 1993-1

LC1 - Tension and | Member: 1

c/t-Part No.	A Elements	B Restraint Type	C Subtr. Length [mm] Δ_{start}	D Δ_{end}	E Width c [mm]	F Thickness t [mm]	G Description	H Symbol	I Value	J Unit
1	1	One side	0.0	13.5	186.5	12.0	Normal stress	$\sigma_{x,start}$	-10.4	N/mm²
							Normal stress	$\sigma_{x,end}$	-30.8	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_1	355.0	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_2	119.5	N/mm²
							Stress ratio	ψ	0.337	
							Material factor dependent on f_y	ε	0.814	
							Compression zone factor	α	0.591	
							c/t-ratio	c/t	15.543	
							Limiting proportions	λ_1	7.323	
								λ_2	8.136	-
								λ_3	11.391	-
							Class of the c/t-part		4	
2	2	One side	14.5	0.0	385.5	10.0	Normal stress	$\sigma_{x,start}$	-24.4	N/mm²
							Normal stress	$\sigma_{x,end}$	136.4	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_1	355.0	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_2	-1985.8	N/mm²
							Stress ratio	ψ	-5.594	
							Material factor dependent on f_y	ε	0.814	
							Compression zone factor	α	1.000	
							c/t-ratio	c/t	38.551	
							Limiting proportions	λ_1	7.323	
								λ_2	8.136	-
								λ_3	223.166	-
							Class of the c/t-part		3	

Classification of the Cross-Section According to EN 1993-1 | Effective Widths According to EN 1993-1 | Effective Section Properties

Figure 7.11: Table *5.2 Classification of the Cross-Section According to EN 1993-1* for LC1

In Table *5.3 Effective widths according to EN 1993-1*, we can check the details which are relevant to determine the effective widths.

5.3 Effective Widths According to EN 1993-1

LC2 - Compressio | Member: 1

c/t-Part No.	A Elements	B Restraint Type	C Subtr. Length [mm] Δ_{start}	D Δ_{end}	E Width c [mm]	F Thickness t [mm]	G Description	H Symbol	I Value	J Unit
1	1	One side	0.0	13.5	186.5	12.0	Normal stress	$\sigma_{x,start}$	-57.9	N/mm²
							Normal stress	$\sigma_{x,end}$	-42.7	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_1	57.9	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_2	42.7	N/mm²
							Stress ratio	ψ	0.737	-
							Buckling factor	k_σ	0.453	-
							Euler relative stress	σ_e	785.7	N/mm²
							Max. compressive stress	σ	57.9	N/mm²
							Plate slenderness	$\lambda_{p,\sigma}$	0.999	-
							Reduction factor	ρ	0.813	-
							Effective width	b_{eff}	151.5	mm
2	2	One side	14.5	0.0	385.5	10.0	Normal stress	$\sigma_{x,start}$	-35.7	N/mm²
							Normal stress	$\sigma_{x,end}$	80.0	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_1	35.7	N/mm²
							Boundary stress relative to $f_{y,d}$	σ_2	-80.0	N/mm²
							Stress ratio	ψ	-2.240	-
							Buckling factor	k_σ	98.736	-
							Euler relative stress	σ_e	127.7	N/mm²
							Max. compressive stress	σ	35.7	N/mm²
							Plate slenderness	$\lambda_{p,\sigma}$	0.168	-
							Reduction factor	ρ	1.000	-

Classification of the Cross-Section According to EN 1993-1 | Effective Widths According to EN 1993-1 | Effective Section Properties

Figure 7.12: Table *5.3 Effective widths according to EN 1993-1* for LC2

7.3 Multiple Windows View

The cross-section diagrams and stresses can be displayed side by side in different windows. To access this function, we click on the menu

Results → **Arrange Result Windows**.

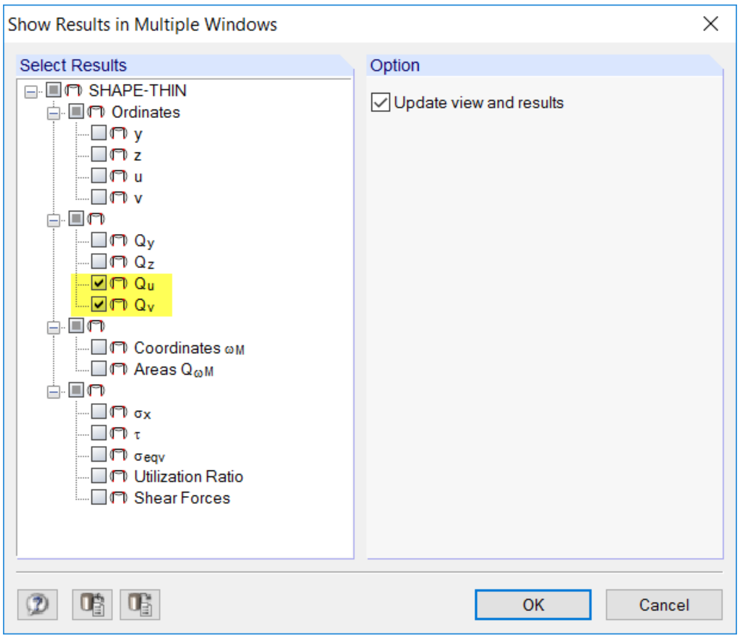


Figure 7.13: Dialog box *Show Results in Multiple Windows*

In the dialog box *Show Results in Multiple Windows*, we tick the check boxes selecting only the statical moments $\mathbf{Q_u}$ and $\mathbf{Q_v}$.

After [OK] we see both cross-section diagrams displayed next to each other in windows.

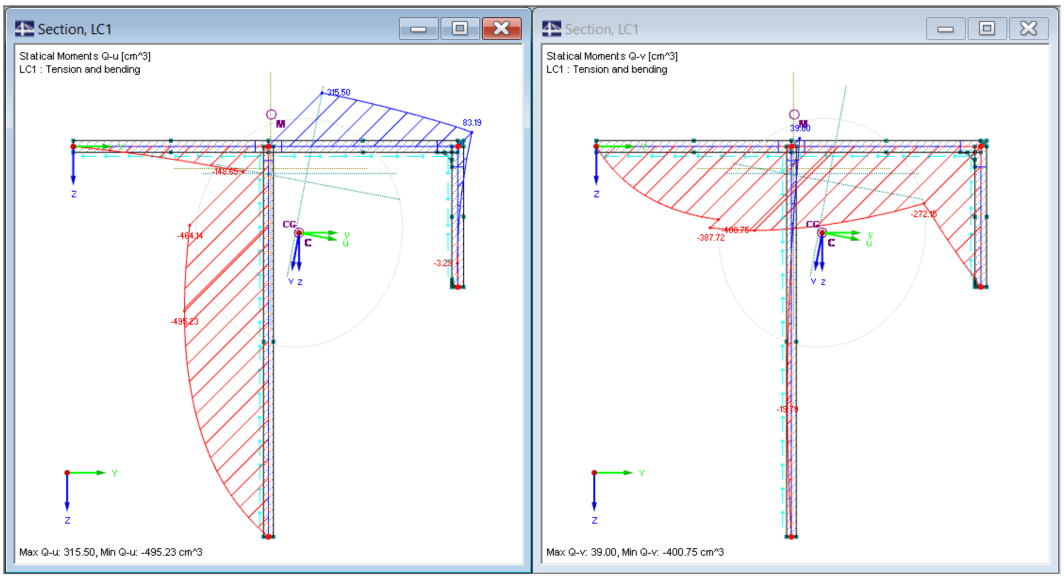


Figure 7.14: Statical moments Q_u and Q_v of gross cross-section

8 Documentation

8.1 Creating Printout Report

SHAPE-THIN offers a print preview for the documentation, which is called *printout report*. There, we can define the input data and results appearing in the report. Moreover, it is possible to add graphics, descriptions and scans.

We start the printout report with the button [Current Printout Report]. We find it in the toolbar to the right of the printer button.

A dialog box opens where we can choose a *Template* for the new printout report.

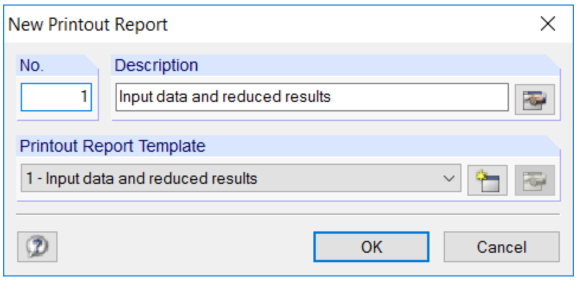


Figure 8.1: Dialog box *New Printout Report*

We accept the template *1 - Input data and reduced results* and create the print preview with [OK].

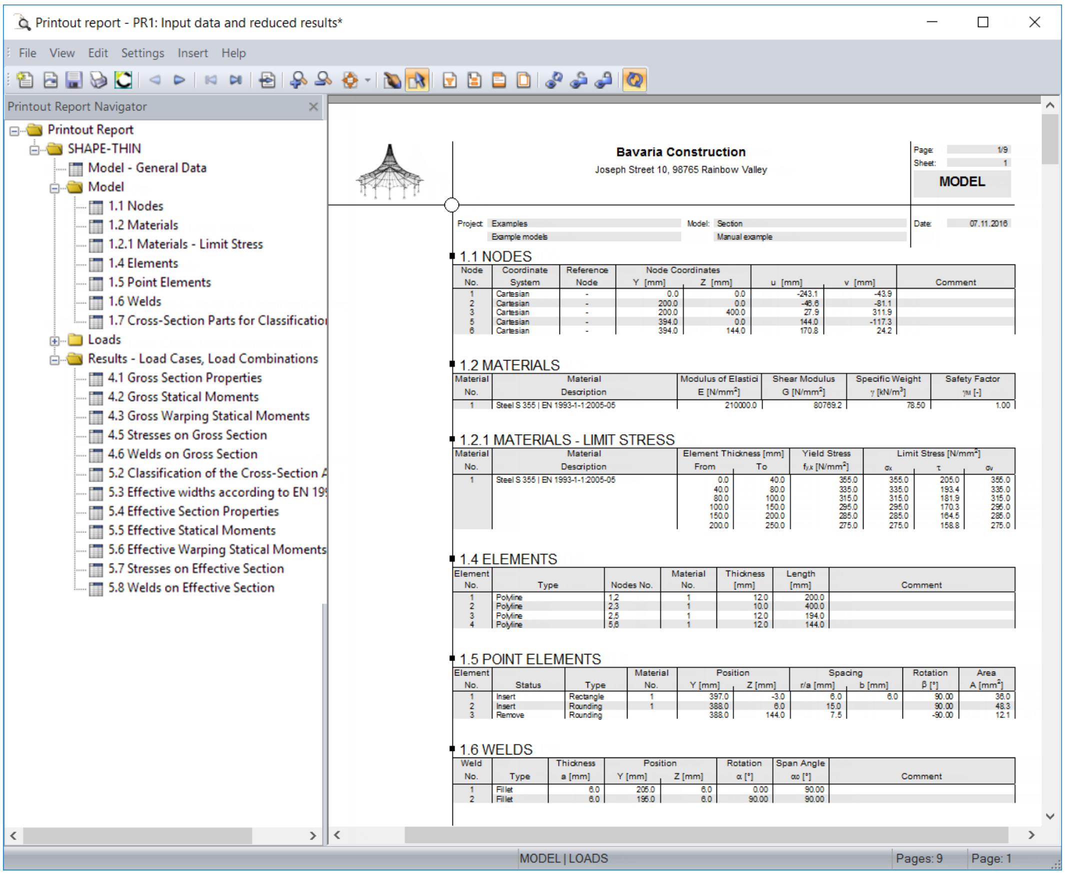


Figure 8.2: Print preview in printout report

8.2 Adjusting Printout Report

On the left, we see the navigator listing the selected chapters. When we click a navigator entry, the content of the corresponding chapter is displayed on the right.

The contents set by default can be adjusted individually. For our example we change the output of cross-section values: We right-click the chapter *Results - Load Cases, Load Combinations* and select *Selection* on the shortcut menu.

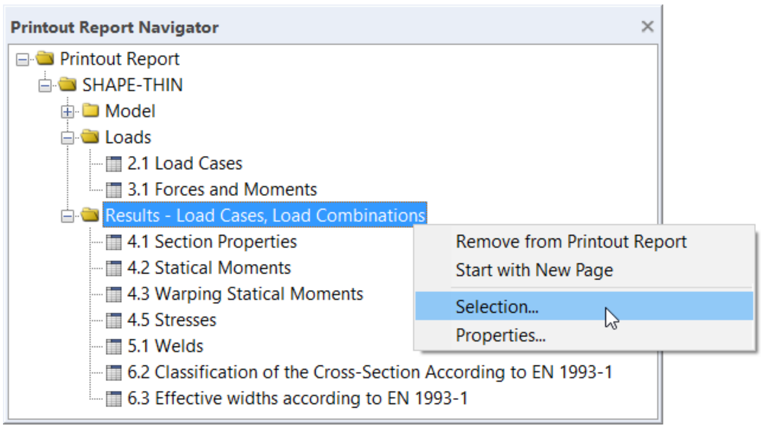


Figure 8.3: Shortcut menu *Results - Load Cases, Load Combinations*

In the dialog box *Printout Report Selection*, tab *LC/CO Results*, we remove the check marks of Tables **5.5 Effective Statical Moments** and **5.6 Effective Warping Statical Moments**.

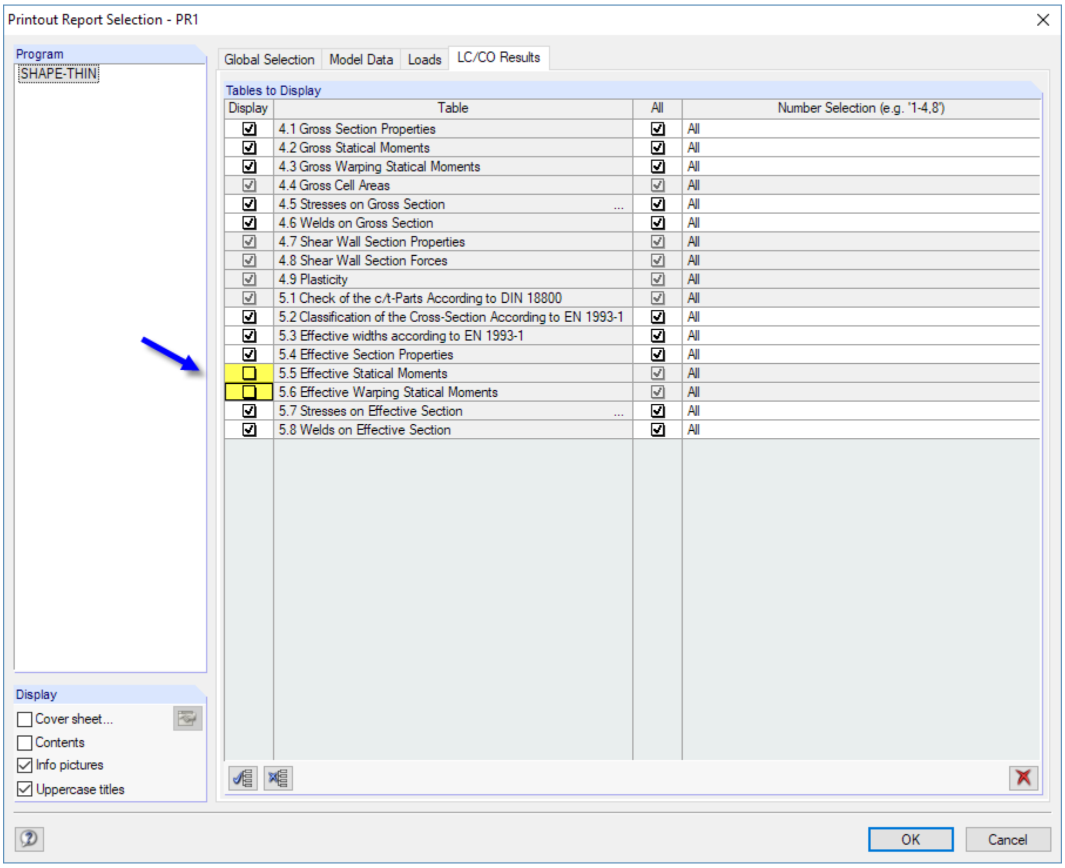


Figure 8.4: Deactivating statical and warping statical moments by using the *Printout Report Selection*

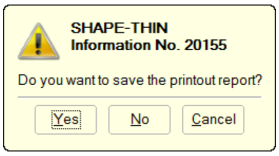
After we have confirmed the dialog box, the print preview is adjusted accordingly.

It is possible to move chapters in the navigator to another position by using the drag-and-drop mouse function. To delete a chapter, we can use the shortcut menu (see Figure 8.3) or the [Delete] key.

8.3 Printing Graphics in Printout Report

Often, graphics are integrated in the printout illustrating the documentation.

Printing statical moments



We close the printout report with [X].

Then, we answer the query for saving the changes with [Yes] and return to the SHAPE-THIN work window.

The result diagrams of the statical moments Q_u and Q_v were set last (see Figure 7.13, page 32). We print these two graphics in the printout report by selecting on the menu

File → Print Graphic

or by using the corresponding button in the toolbar.

In the dialog box *Graphic Printout*, we set the print specifications as shown in Figure 8.5. In addition, we enter **Statical moments** as the *Header of Graphic Picture*.

When printing several windows we can adjust the *Window Arrangement* with the button. We set option **3)** to print the two graphics one below the other.

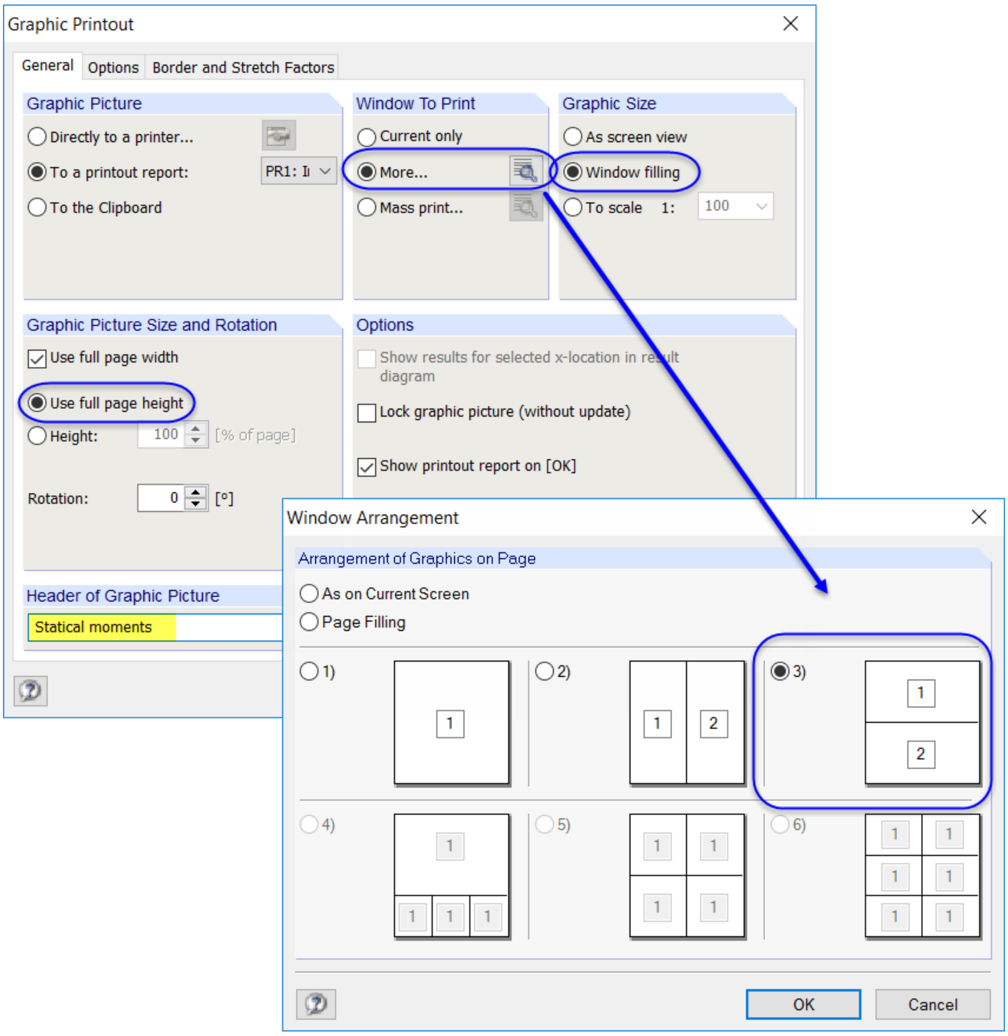


Figure 8.5: Dialog boxes *Graphic Printout* and *Window Arrangement*

We don't change the default settings of the other tabs.

Finally, we print both statical moment diagrams in the printout with [OK]. The graphics are placed at the end of chapter *Results - Load Cases, Load Combinations*.

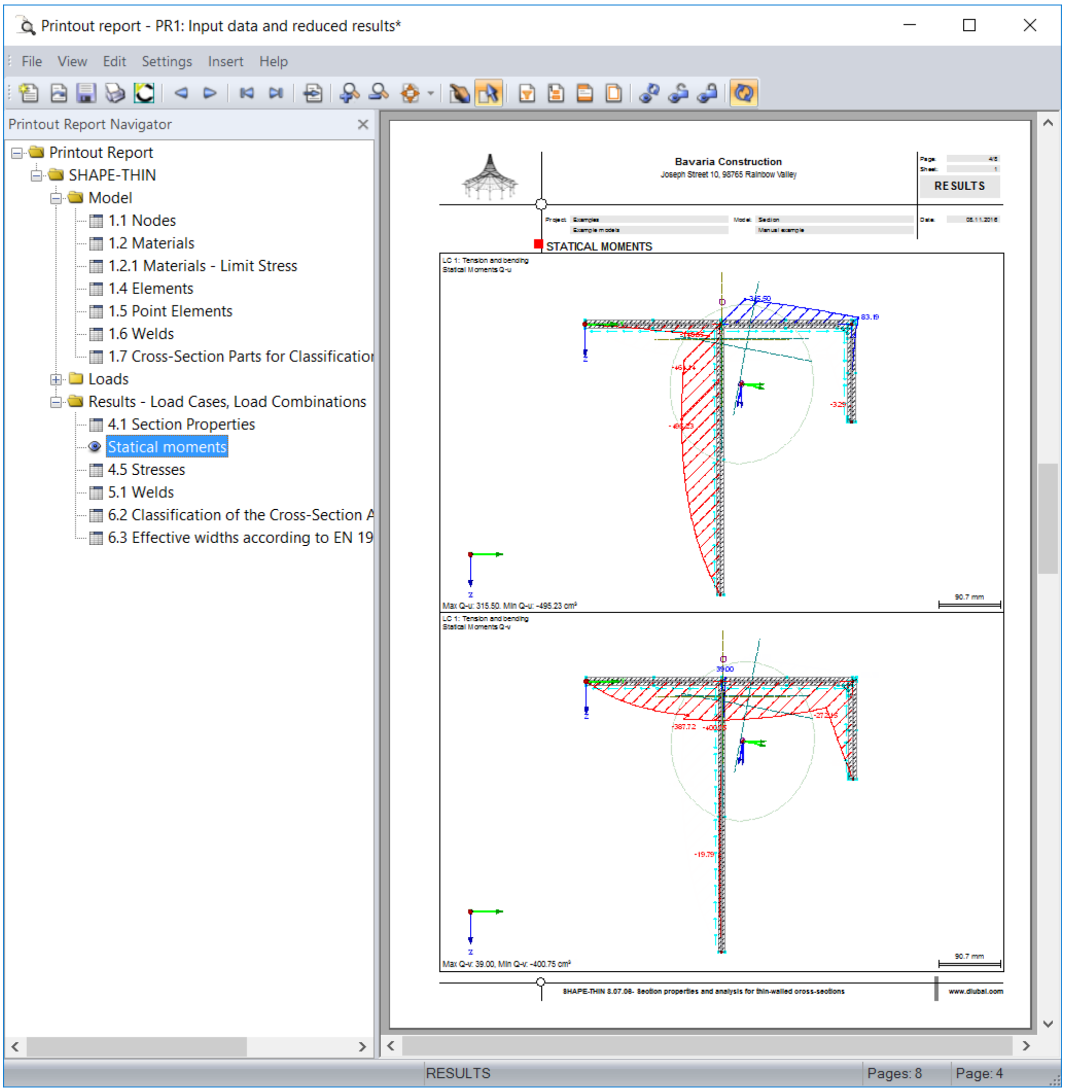
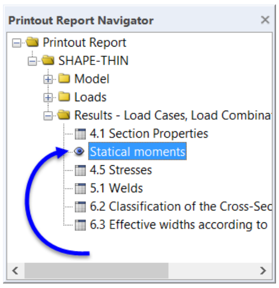


Figure 8.6: Statical moments in printout report



We move the graphic up to the entry *Section Properties* by using the drag-and-drop function (hold down left mouse button).

Printing equivalent stresses

We close the printout report and save the modifications.

Now, we close one of the windows in the SHAPE-THIN work window with . If a query for saving the data appears, we confirm it with [Yes].

We maximize the remaining window with .

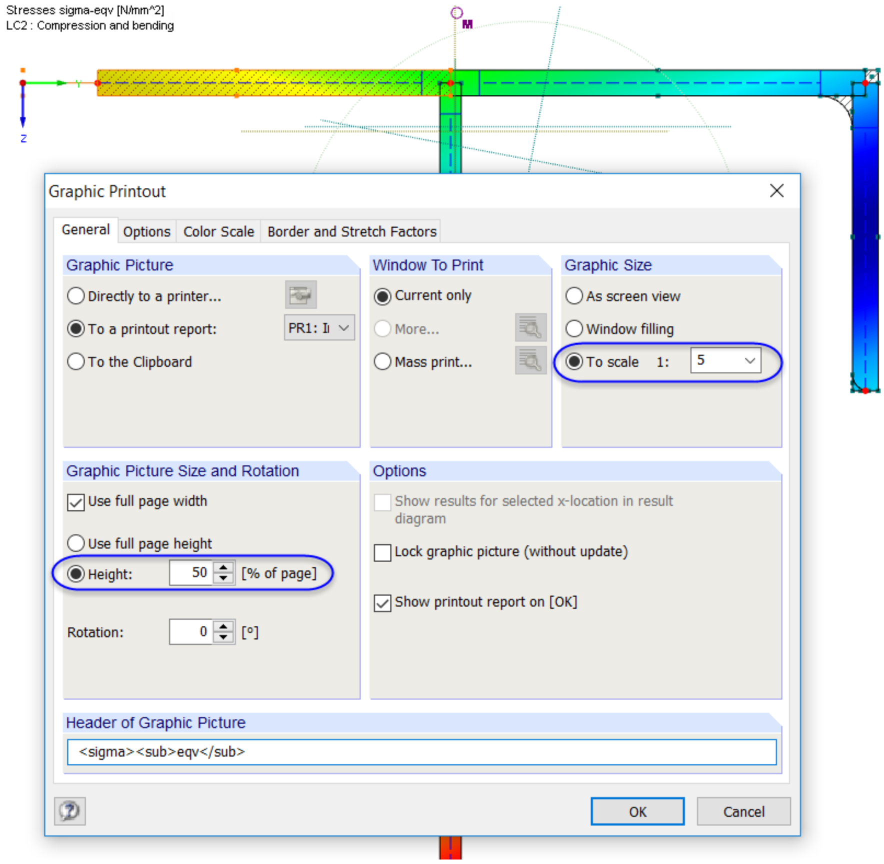
Then, we set the window filling view and show the whole model with the button or the function key [F8].

Now we display the **equivalent stresses** σ_{eqv} of load case **LC2**. Furthermore, we switch the display of the *Effective Parts* **on**.

Then, we select the [Isobands] again for a colored representation of the stresses.

We click the button to open the dialog box *Graphic Printout* again (see Figure 8.7).

Now, for the printing we set the *Height* of the graphic picture to **50 %**. Thus, the graphic will use exactly a half page in the printout report. We want *To Scale* the graphic with **1:5**.



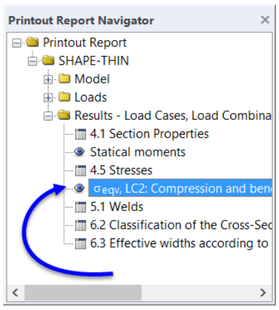


Figure 8.7: Printing the equivalent stresses

We click [OK]. The graphic is printed in the printout report where it is placed again at the end of the chapter *Results - Load Cases, Load Combinations*. We move it to the entry *Stresses*.

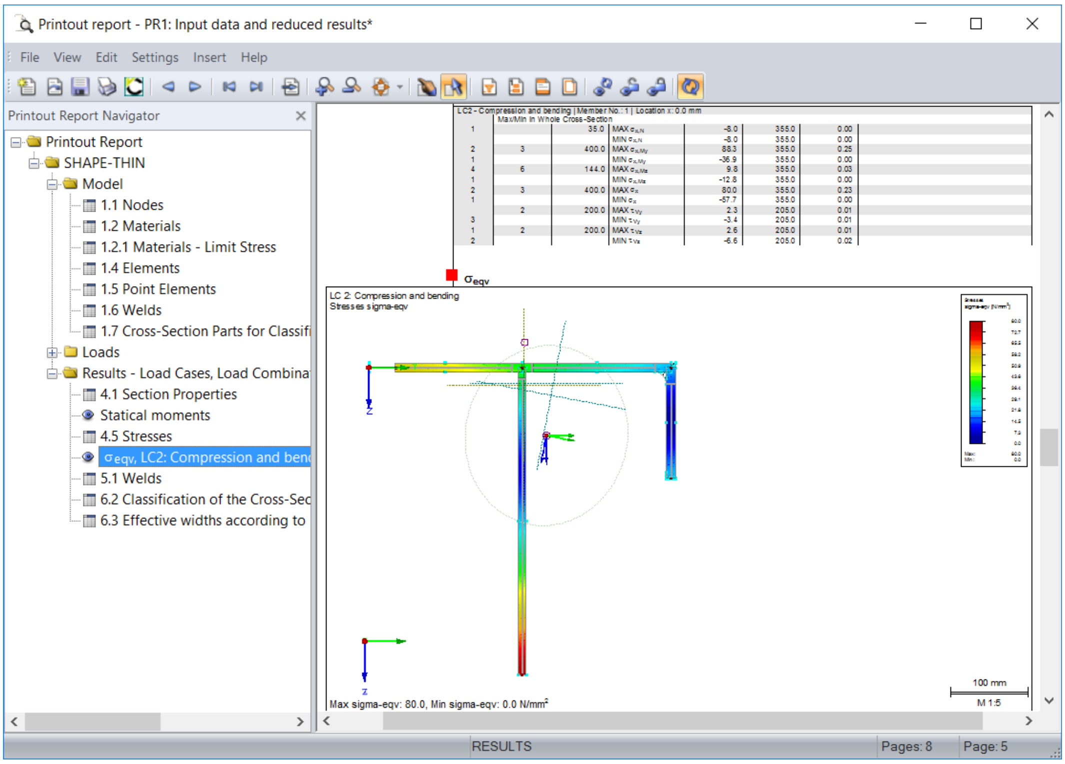


Figure 8.8: Graphic of equivalent stresses in printout report

Printing the printout report

Now, we can send the printout report to the printer by using the [Print] button.

The integrated PDF printer enables printing data of the printout report in a PDF file. We use this function and select on the printout report's menu

File → **Export to PDF**.

The Windows dialog box *Save As* opens where we specify the storage location and the file name.

Save

We click [Save], and then a PDF file is created with bookmarks making the navigation in the digital document easier.

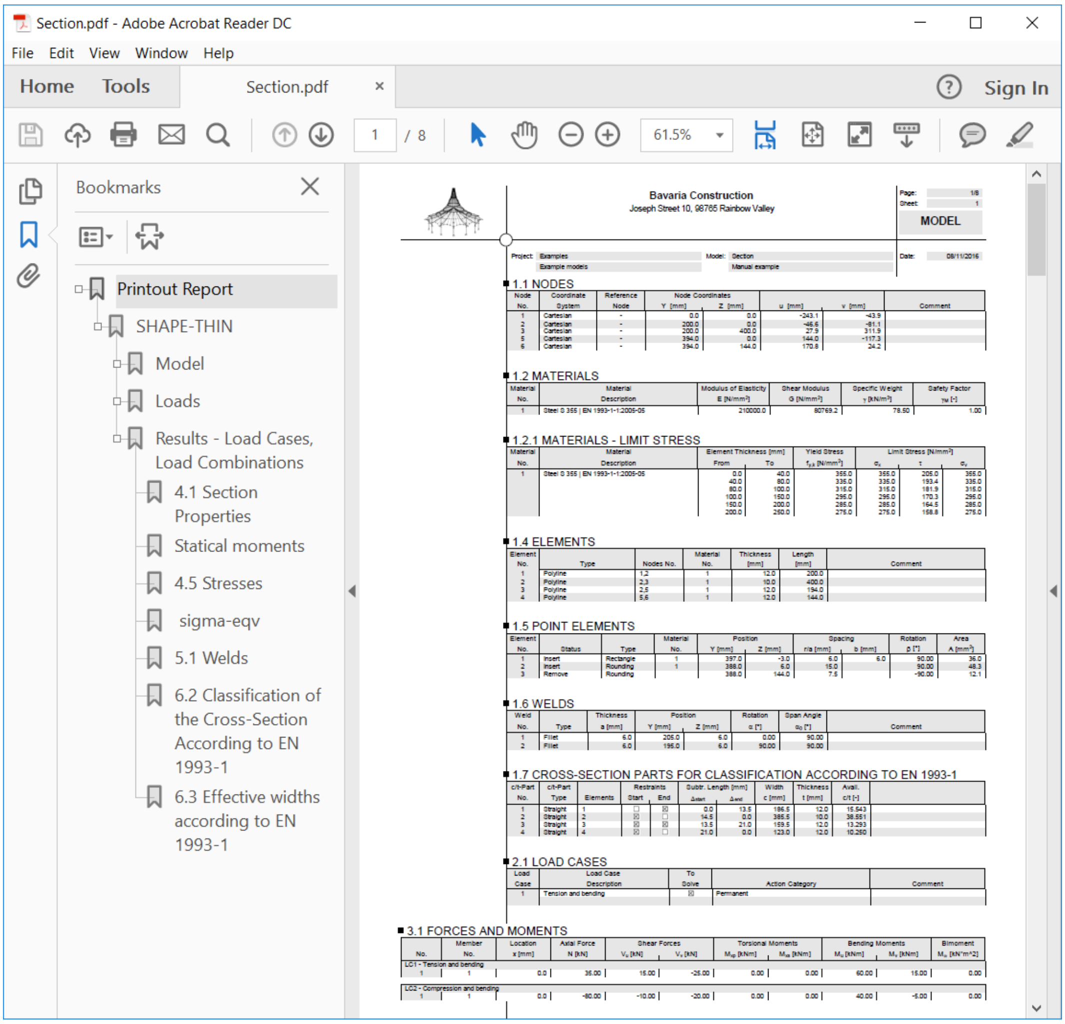


Figure 8.9: Printout report as PDF file with bookmarks

9 Outlook

Now, we are at the end of our example. We hope that this introduction helps you to easily find access to SHAPE-THIN and makes you curious about unknown functions. Find a detailed program description in the user manual of SHAPE-THIN that you can download on our website.

You can access the program's online help on the **Help** menu or with [F1] and search for certain expressions as seen in the manual. The online help is based on the SHAPE-THIN manual.

You can also contact our hotline team and ask any question by e-mail. Or have a look at the FAQ and the Knowledge Base pages on our website.

It is possible to import the cross-section of our example to an RSTAB or RFEM model. You can also use it for designs in the add-on modules RF-/STEEL or RF-/STEEL EC3.

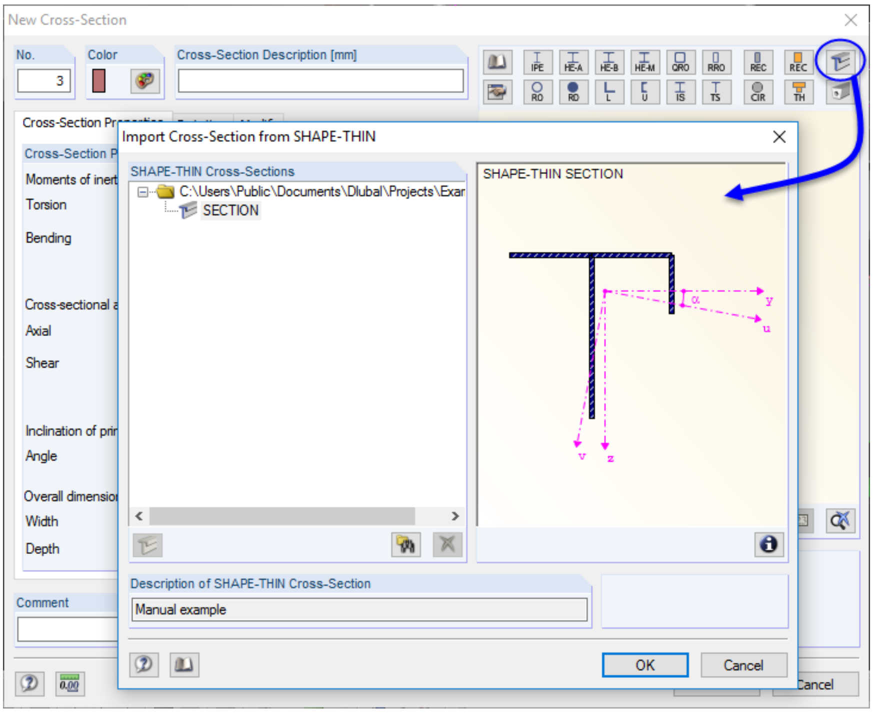


Figure 9.1: Importing SHAPE-THIN section to RSTAB/RFEM

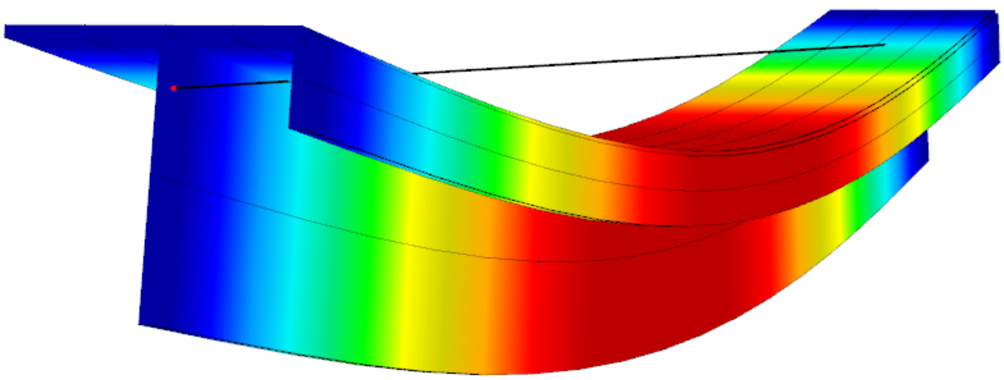


Figure 9.2: Deformations in RSTAB/RFEM model