Version
November 2017

Program

SHAPE-THIN

Cross-section properties and design of
thin-walled sections

Introductory Example

© Dlubal Software, Inc. 2017
The Graham Building
30 South 15th Street
15th Floor
Philadelphia, PA 19102
USA

Phone: +1 267 702-2815
E-mail: info-us@dlubal.com
Web: www.dlubal.com/us

Contents

1 Introduction

This introductory example presents the fundamental functions of the SHAPE-THIN program, such as creating user-defined elements or setting a section from the library. Like in any other software, there are many ways how to reach a goal. Depending on the situation and your personal preferences, it may be useful to proceed in one way or another.

The example deals with a steel angle to which an unequal angle section is welded. We will model this section and apply two load cases. After that, we will calculate the cross-section properties and the stresses.

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

This basic section should also encourage you to discover the possibilities of the program on your own.

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

You can find the file *Section.du8* featuring similar metric proportions in the *Examples* project which was created during the installation.

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

2 Section and Internal Forces

2.1 Cross-Section

The example represents a welded steel angle consisting of steel plates with thicknesses of 1/2 in. and 3/8 in. to which a L 6x4x1/2 angle section is welded.

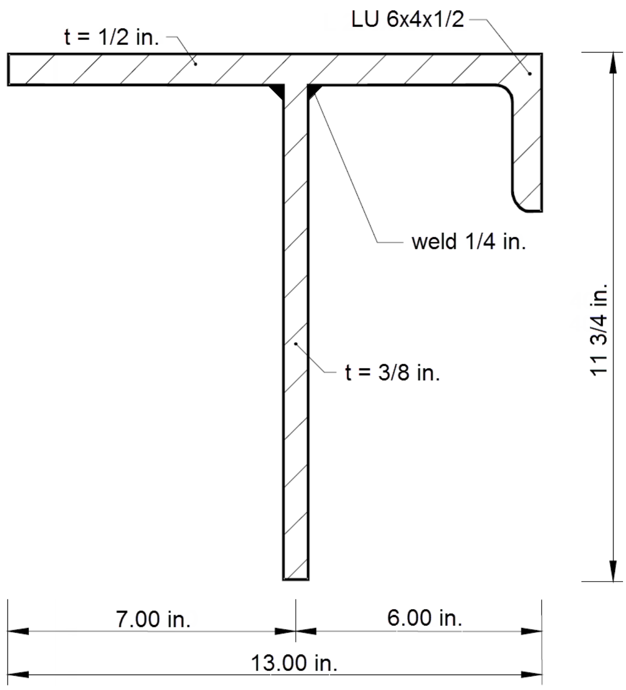
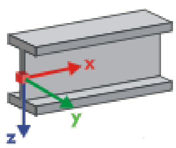


Figure 2.1: Sketch of section

The section is made of steel A36.

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.

$N = 8$ kip

$V_y = 4$ kip

$V_z = -6$ kip

$M_y = 40$ kipft

$M_z = 12$ kipft

Load case 2: Compression and bending

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

$N = -20$ kip

$V_y = -3$ kip

$V_z = -5$ kip

$M_y = 30$ kipft

$M_z = -3$ kipft

3 Creating Section

3.1 Starting SHAPE-THIN

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

The program can also be started on the taskbar: **All Apps** → **Dlubal** → **Dlubal SHAPE-THIN 8.xx**.

3.2 Defining General Data

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, we close it (menu **File** → **Close**). Then we open the *General Data* dialog box with the menu **File** → **New**.

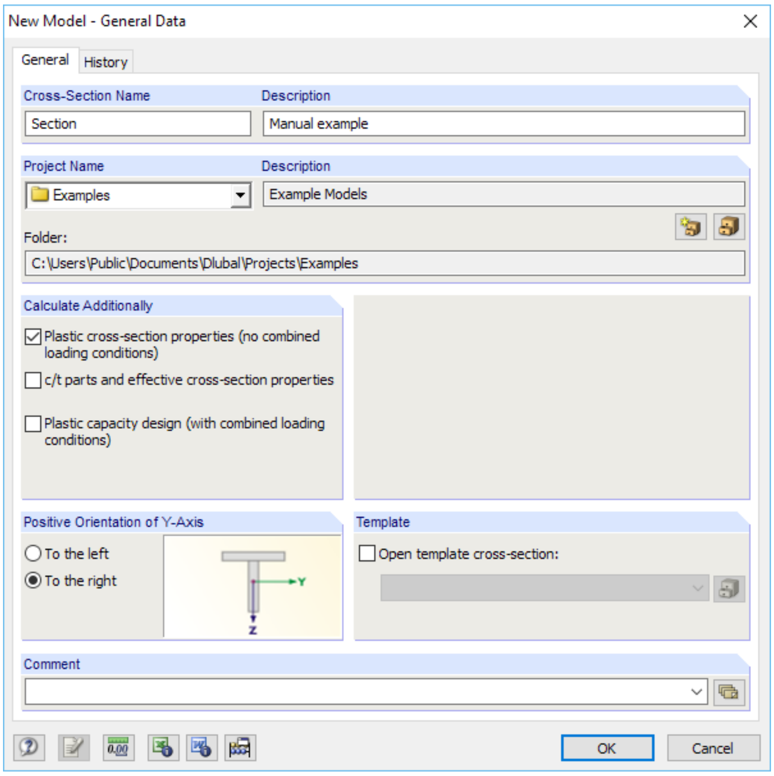


Figure 3.1: Dialog box *New Model - General Data*

In the *Cross-Section Name* text box, we enter **Section**; in the *Description* box, we enter **Manual example**. The name of the section must always be defined because it represents the name of the SHAPE-THIN file. The description is optional, however.

Then we select the **Examples** project from the *Project Name* list (unless it is preset anyway). The *Description* of the project and the *Folder* are shown automatically.

In the *Calculate Additionally* dialog section, we check whether the calculation of the **Plastic cross-section properties** is selected. We neither want to determine the effective cross-section properties (they refer to European standards) nor carry out a plastic capacity design.

We do not change the *Positive Orientation of Y-Axis*, i.e. we keep the presetting **To the right**.

When we have defined the general data of the section, we close the dialog box with [OK].

After this, the empty work window of SHAPE-THIN is displayed.

4 Geometry

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 [Move, Zoom, Rotate] button in the toolbar. The mouse pointer turns into a hand. While holding down the left mouse button, we can place the work space in any position. For the input of our section, we move the axes of coordinates a bit to the left, i.e. towards the navigator.

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

To close the *Move, Zoom, Rotate* function, we have several possibilities:

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

Units

For our example, we will use imperial 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 *Units and Decimal Places* dialog box by clicking on the menu

Edit → Units and Decimal Places.

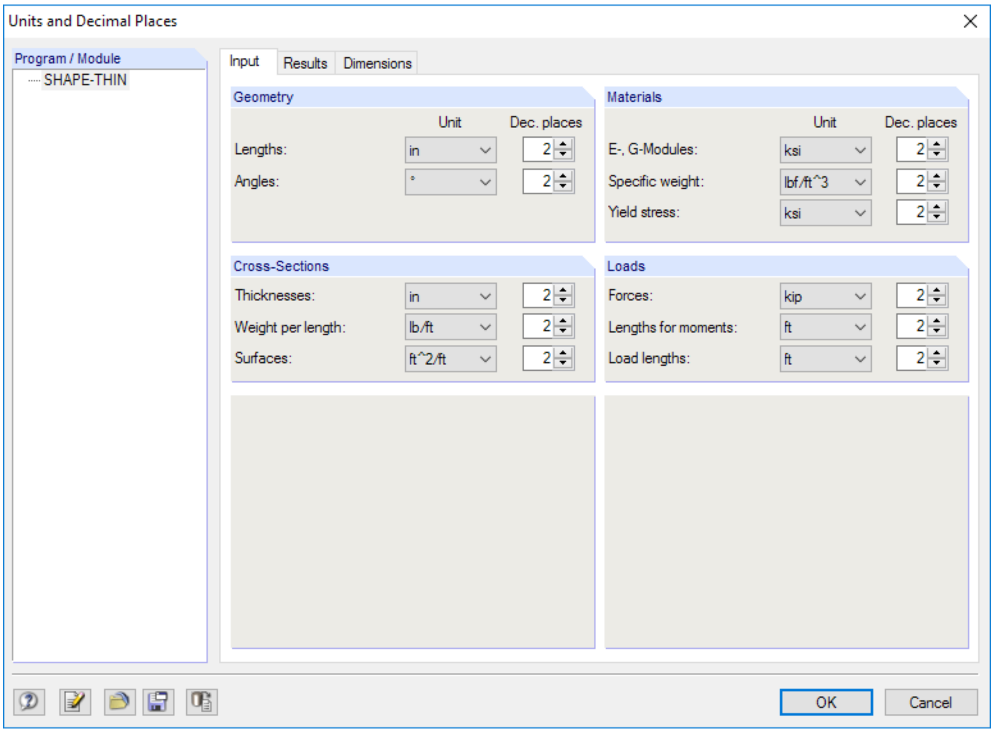


Figure 4.1: Dialog box *Units and Decimal Places*

We make sure that the settings correspond to those as seen in Figure 4.1 and Figure 4.2.

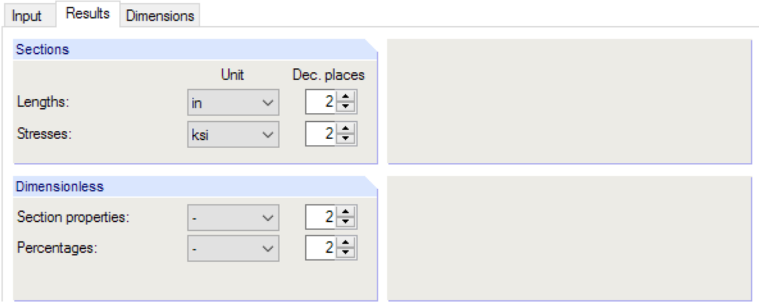


Figure 4.2: Dialog box *Units and Decimal Places*, tab *Results*

Grid

The background of the work space is marked by a grid. The spacing of grid points can be adjusted in the *Settings of Work Plane, Grid/Snap, Object Snap and Guide Lines* dialog box which we open with the button as seen to the left.

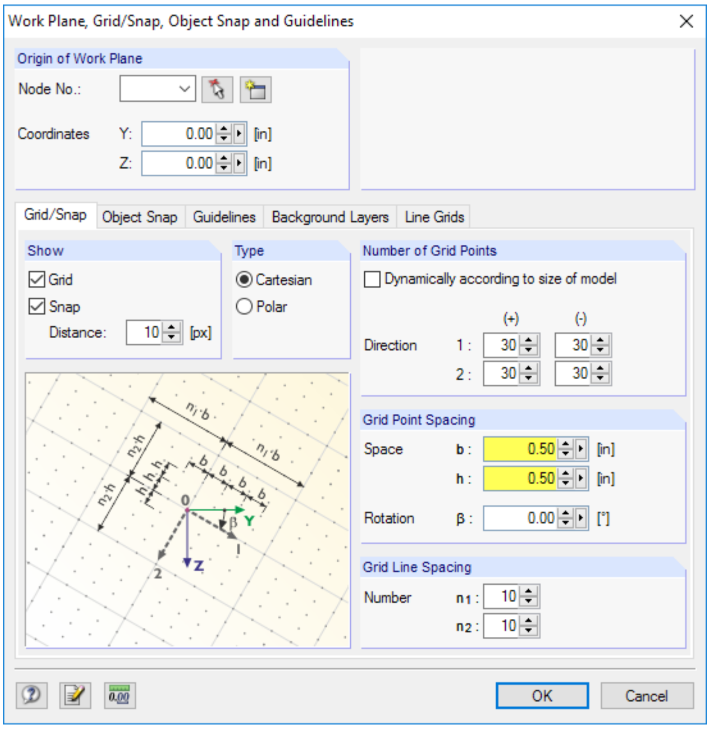


Figure 4.3: Dialog box *Work Plane, Grid/Snap, Object Snap and Guidelines*

We make sure that the grid point spacing is set to **0.5 in**. Then we close the dialog box with [OK].

SNAP GRID

For the graphical input, it is important that the *SNAP* and *GRID* buttons in the status bar are active. Then the grid is visible in the work space and the points are snapped on the grid.

Mouse functions

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

When you click an object with the **right** mouse button, its shortcut menu appears. It contains some useful commands and functions related to the object.

By scrolling the **wheel button**, we can maximize or minimize the view of the section. The position of the mouse pointer represents the center of the zoom area.

By holding down the wheel button, we can move the model directly without having activated the button. The pointer symbol shows the respective function.

4.2 Changing Material

Steel A992 is preset as material. As our section consists of steel A36, 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.

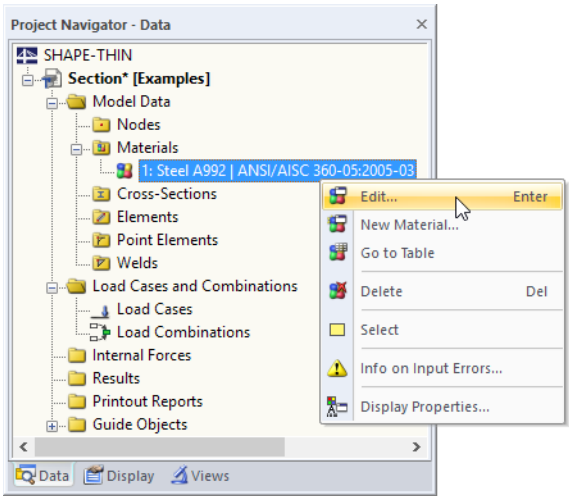


Figure 4.4: *Edit* materials in shortcut menu of navigator

The *Edit Material* dialog box appears.

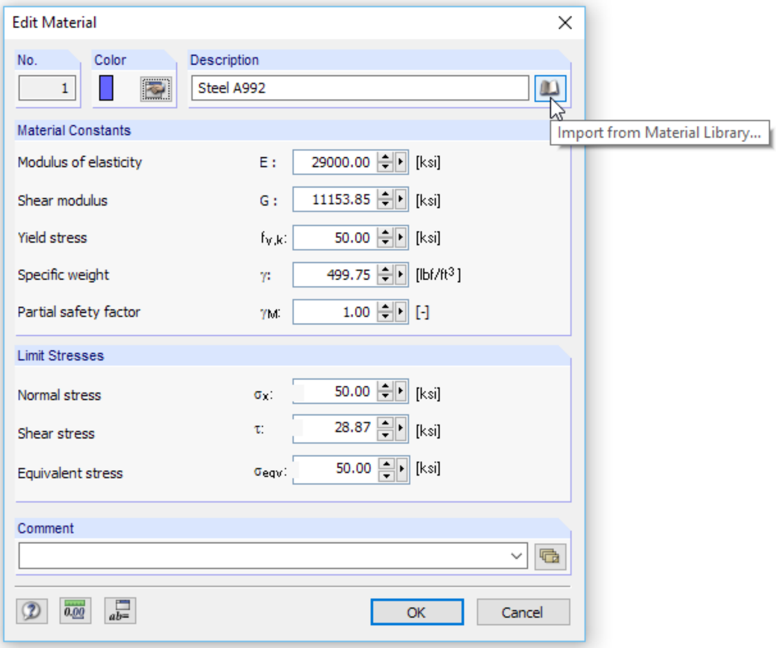


Figure 4.5: Dialog box *Edit Material* with access to library

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

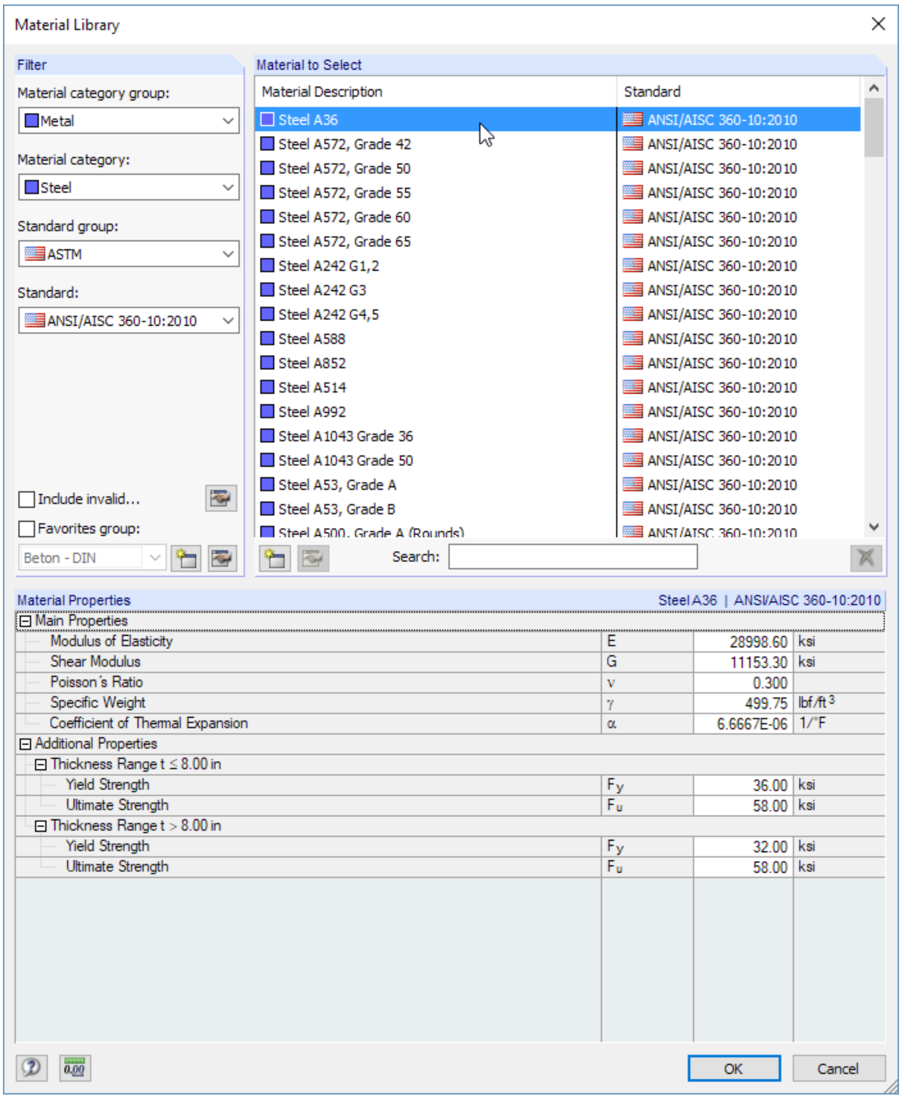


Figure 4.6: Selecting Steel A36 in library

The categories in the *Filter* dialog section enable us to select the materials according to specific criteria. When the filters are set as seen in Figure 4.6, the list *Material to Select* becomes clearer.

We select the material **Steel A36**. 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 modified material.

Now we can start to enter the geometry of the section.

4.3 Defining Elements

We could define the nodes graphically or in tables at first and then connect them by elements. For our example, the direct graphical input of elements is more convenient, however, where the nodes are created automatically.

4.3.1 Placing Elements

We open the *New Element* dialog box via the menu

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

or – faster – use the corresponding button in the toolbar.

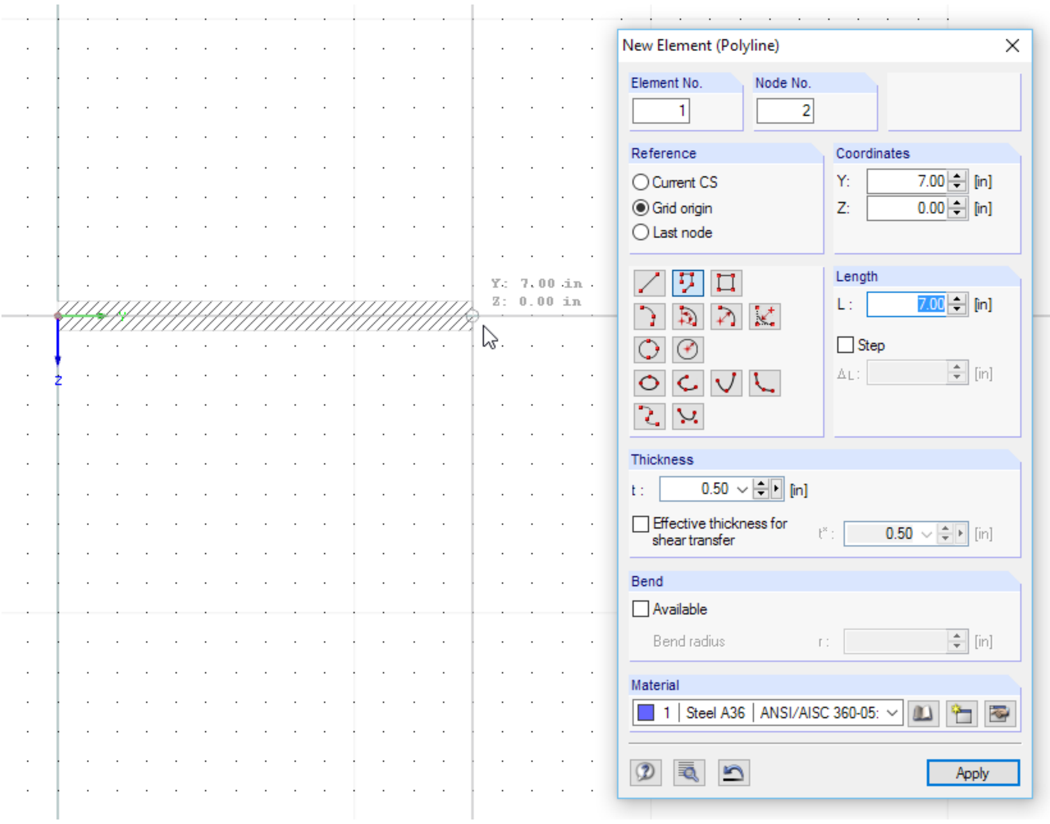


Figure 4.7: Dialog box *New Element (Polyline)*

Element No. **1** is preset with a *Thickness* of **0.50 in.** in the dialog box. The *Material* is **Steel A36**. Those settings are appropriate for our flange element.

When we move the mouse across the work space, the coordinates of the pointer are indicated. The reticle snaps at the grid points of 0.50 in.

With a click of the left mouse button, we place node **1** as the start node in the zero point (Y/Z-coordinates **0.00/0.00**) .

With another mouse click, we define node **2** in the grid point **7.00/0.00**. It is the end node of the first element.

As we have chosen the *Continuous* option, node 2 represents the start node of the next element no. **2**. Thus, we can go on with placing node **3** in the grid point **7.00/11.50** (see Figure 4.8). We maintain the thickness of 0.50 in. because we are going to adjust the thickness of the web later.

After that, we quit 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 the corresponding function on the menu

View → Show All

or use the button in the toolbar. Pressing the function key [F8] gives the same result.

Displaying numbers

Before we continue, we switch on the numbering of the nodes and elements. The fastest way to do this by right-clicking an empty area of the work window. A shortcut menu appears. It contains some useful functions.

We click the **Show Numbering** item.

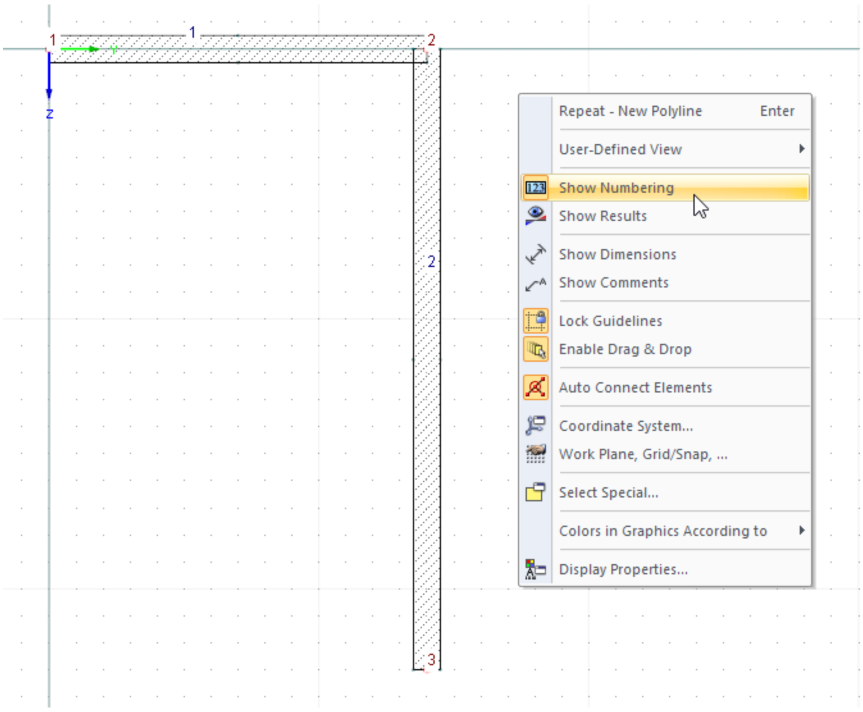


Figure 4.8: *Show Numbering* option in shortcut menu

4.3.2 Modifying Element

We still have to assign the correct thickness of 3/8 in. to the web element. For this, we double-click element 2.

In the *Edit Element* dialog box, we set the *Thickness t* to **0.375 in.** (see Figure 4.9).

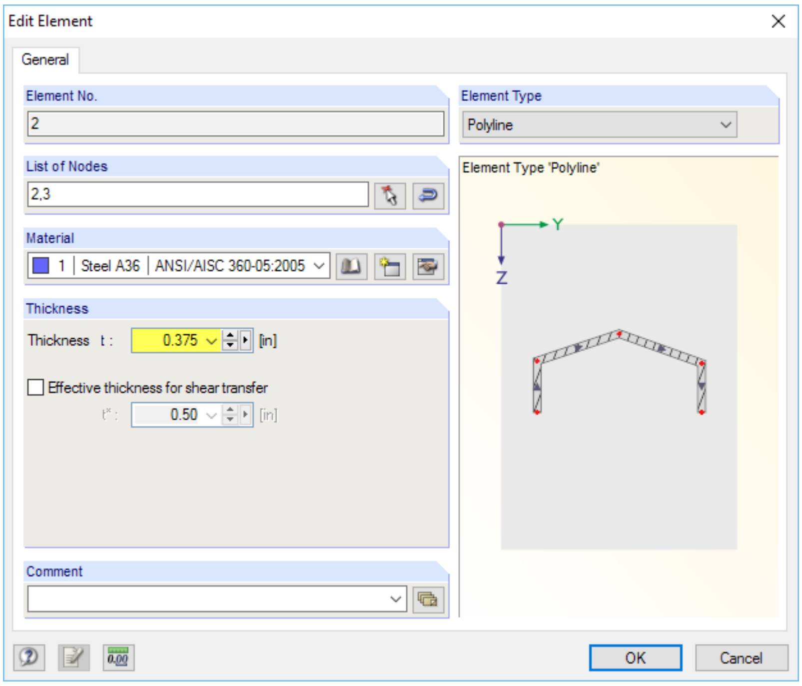


Figure 4.9: Dialog box *Edit Element*

After [OK], the thickness of the web is updated in the work window.

4.4 Defining Angle

Next we connect an unequal angle 6x4x1/2 to the horizontal element.

4.4.1 Placing Section

We open the cross-section library with the [Insert Section] button.

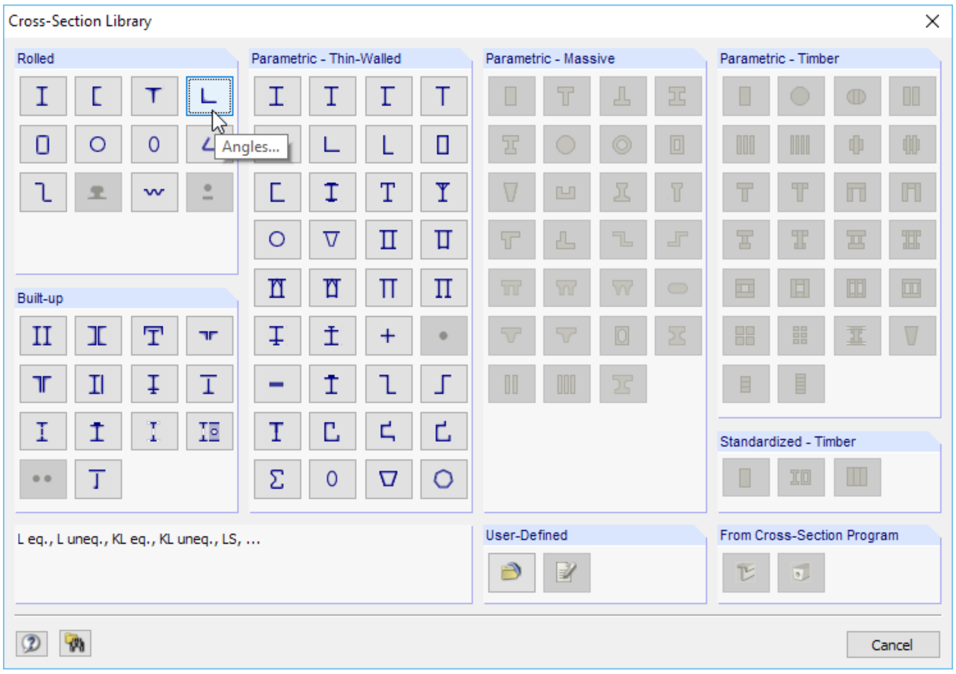


Figure 4.10: Cross-Section Library

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

In the *Rolled Cross-Sections - Angles* dialog box, we select the cross-section **L 6x4x1/2** in the **LU** *Table* (the selection is easier when *AISC* has been selected in the *Filter* section).

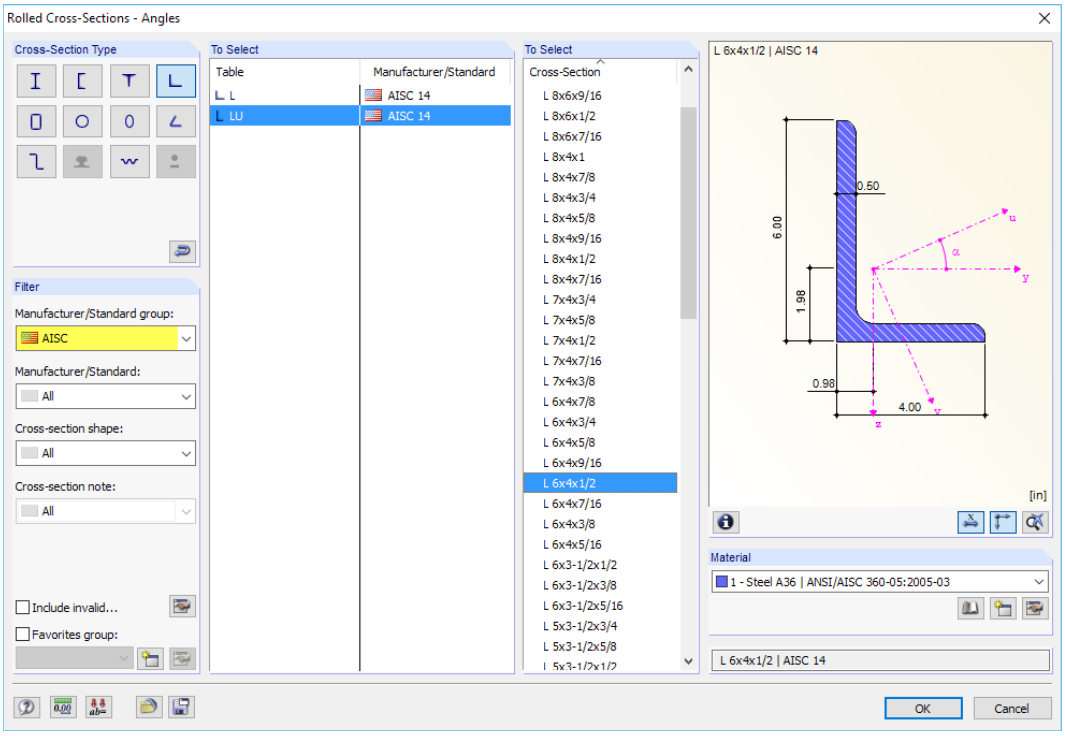


Figure 4.11: Selecting angle section in library

It is possible to check the properties of the angle with the [Info] button.

Steel A36 is preset as *Material*. Thus, we can confirm the dialog box with [OK].

The *Set Section* dialog box appears.

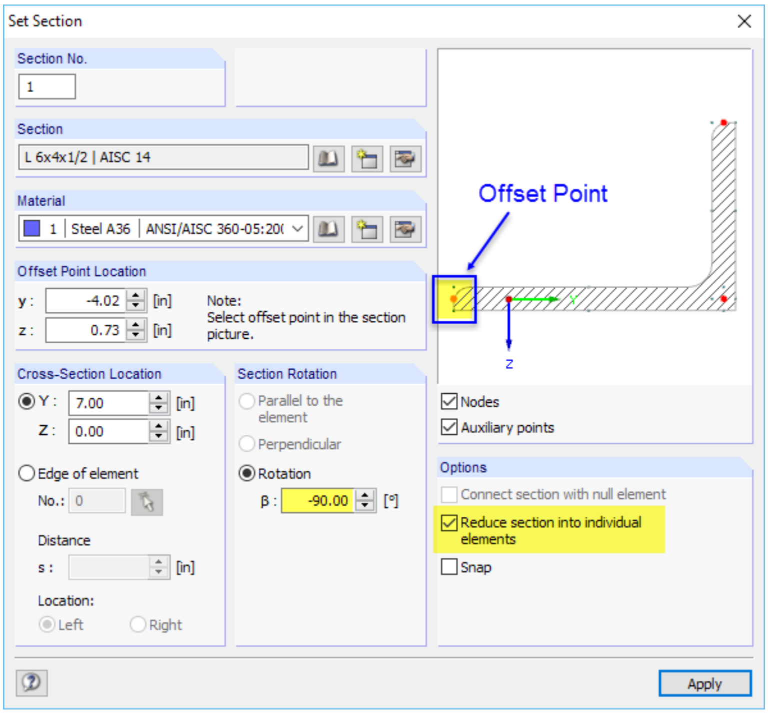


Figure 4.12: Dialog box *Set Section*

We specify a *Section Rotation* of **-90°** for a better arrangement of the section. Furthermore, we select the **Reduce section into individual elements** option.

When we move the angle with the mouse across the screen, we see that the current "grab point" (offset point) lies in the centroid of the section. Because we want to connect the angle with its long leg, we change the offset point: In the picture of section in the dialog box, we click the **red node** at the left section end so that the *Offset Point Location* corresponds to Figure 4.12.

Now we move the angle towards node **2**. When the node and its coordinates are shown in the status bar, we press the left mouse button.

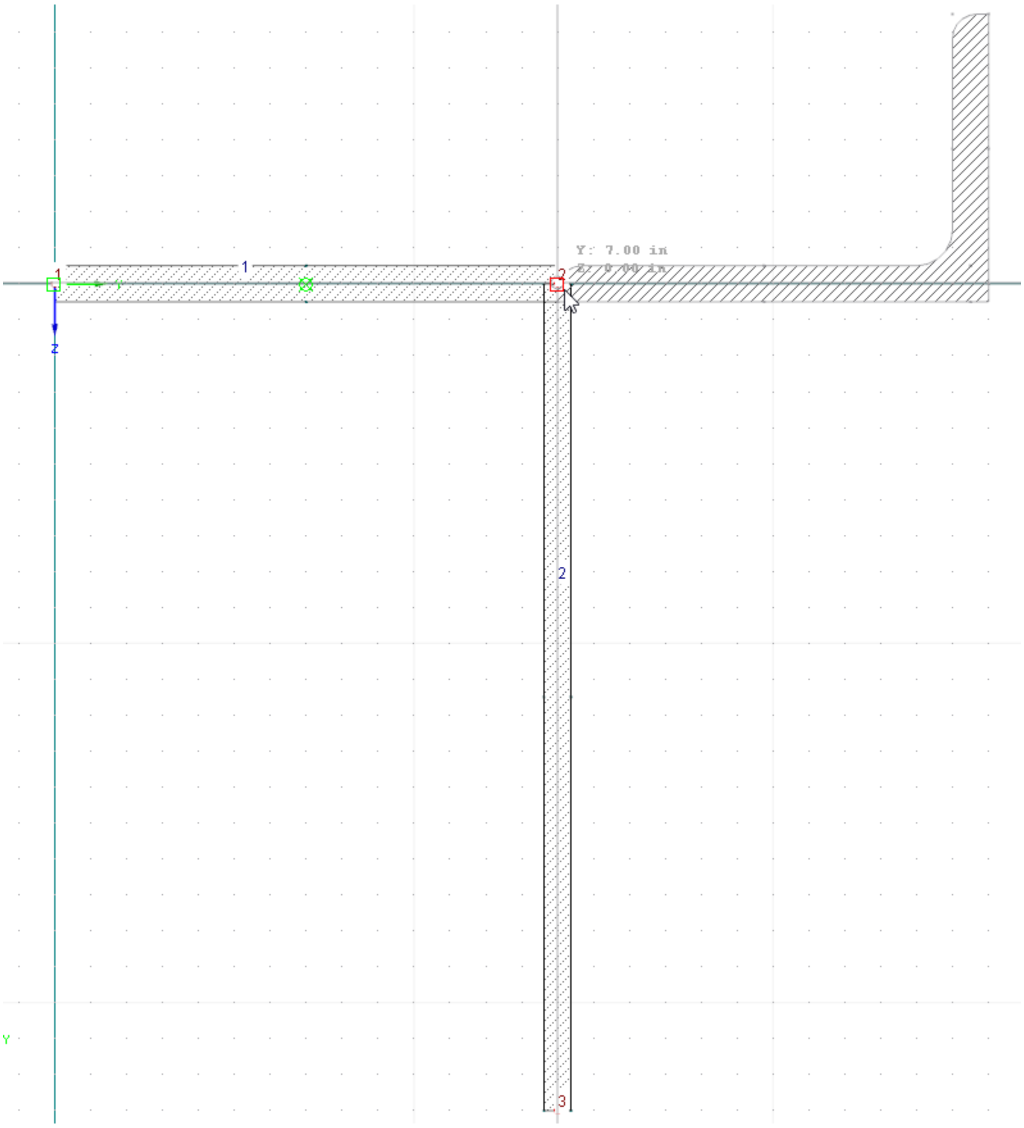


Figure 4.13: Placing angle at node 2

A message appears with the numbers of objects that were created when reducing the angle section into elements and point elements.

Strictly speaking, the angle ought to be set to the edge of the element. In this case, however, one extra element would be created to connect the angle, which exceeds the demo restrictions concerning the maximum number of elements.

We quit 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 using the *Mirror* function.

Selecting objects

Before the editing functions (copy, rotate, or mirror) can be applied, we have to determine the relevant objects, i.e. "select" them.

We draw a window across the section that we have just set – from the left to the right. Make sure that the window completely includes elements 3 and 4.

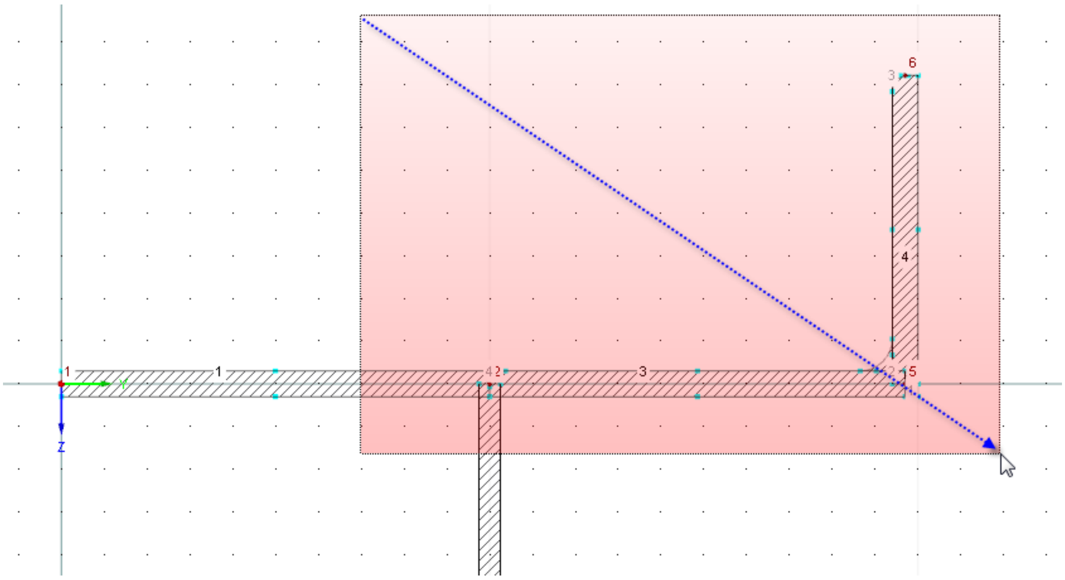


Figure 4.14: Selecting elements with window

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 also contains all 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 is canceled. Only the new object is selected.
- To add an 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.

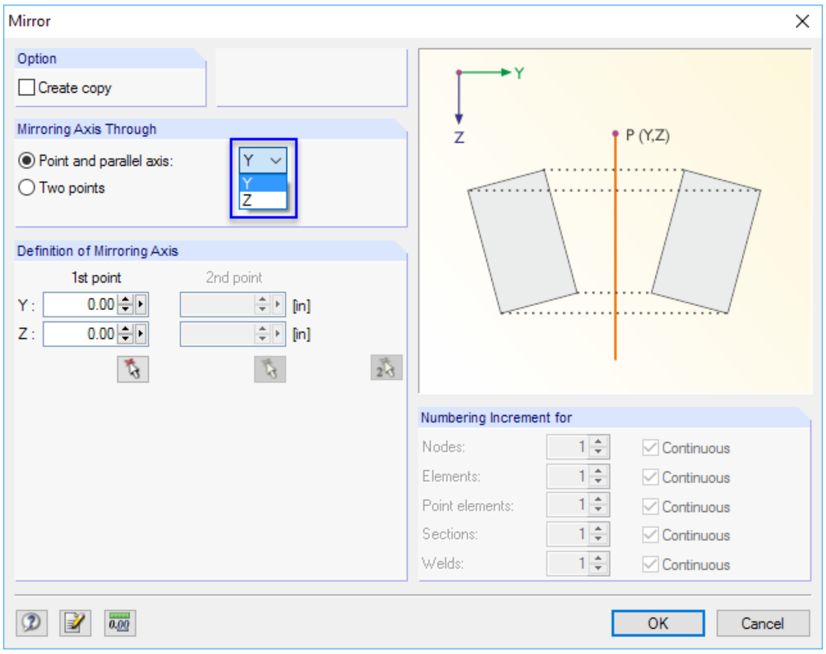


Figure 4.15: Dialog box *Mirror*

We make sure that the *Create copy* check box is <u>not</u> selected.

Now we mirror the angle about the *parallel axis* to **Y** with reference to the origin (0.00/0.00).

We click [OK] to put the angle into the correct position.

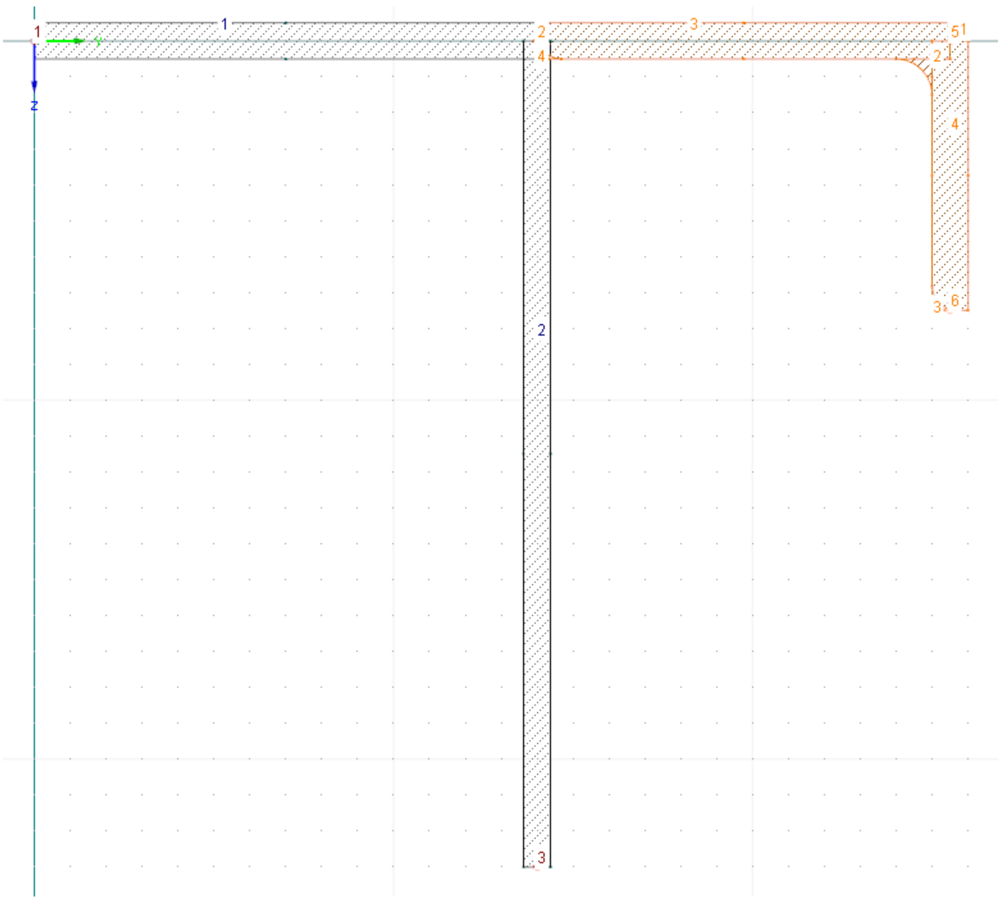


Figure 4.16: Mirrored angle

4.5 Defining Welds

Finally, we adjust the connection zones between the elements by applying welds.

4.5.1 Deleting Point Elements

We zoom in the area of the connection between the elements and the angle. For zooming in, we can use the wheel button of the mouse or apply the function [Zoom with Window] using the corresponding button.

There is an hole in the connection zone. It was created when we mirrored the angle. We eliminate it by right-clicking point element **4** and selecting the **Delete Point Element** option on the shortcut menu.

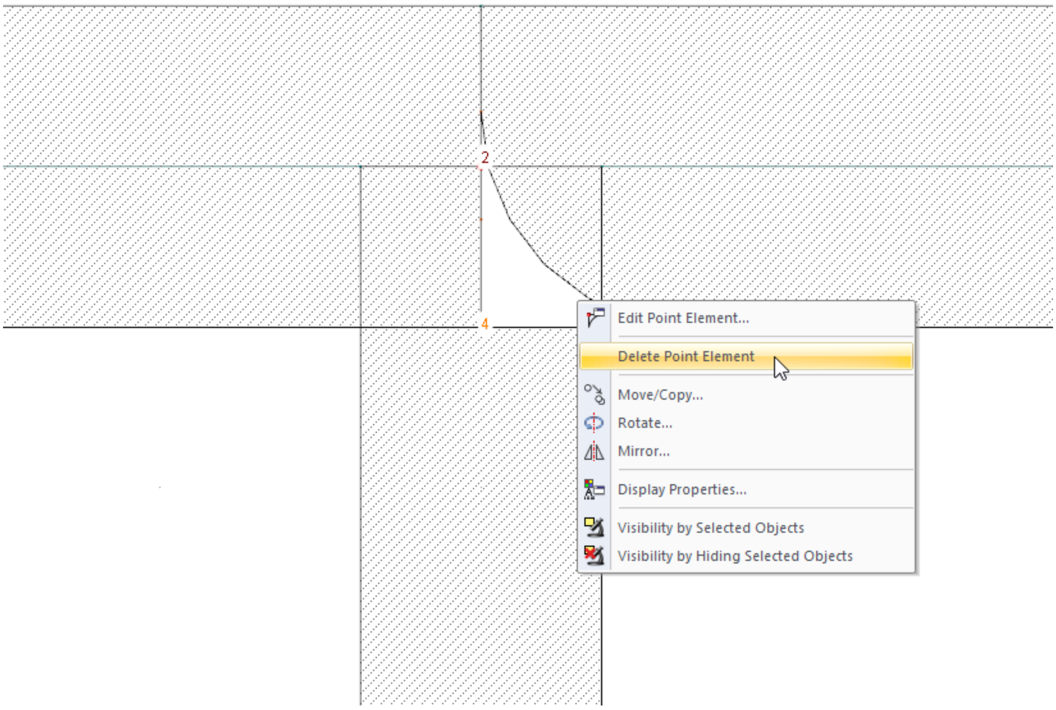


Figure 4.17: Deleting Point Element

4.5.2 Placing Welds

We define two fillet welds of 1/4 in. each between the web and the flange.

To open the *Set Weld* dialog box, can use the button as seen to the left or the menu

Insert → **Model Data** → **Welds** → **Graphically**.

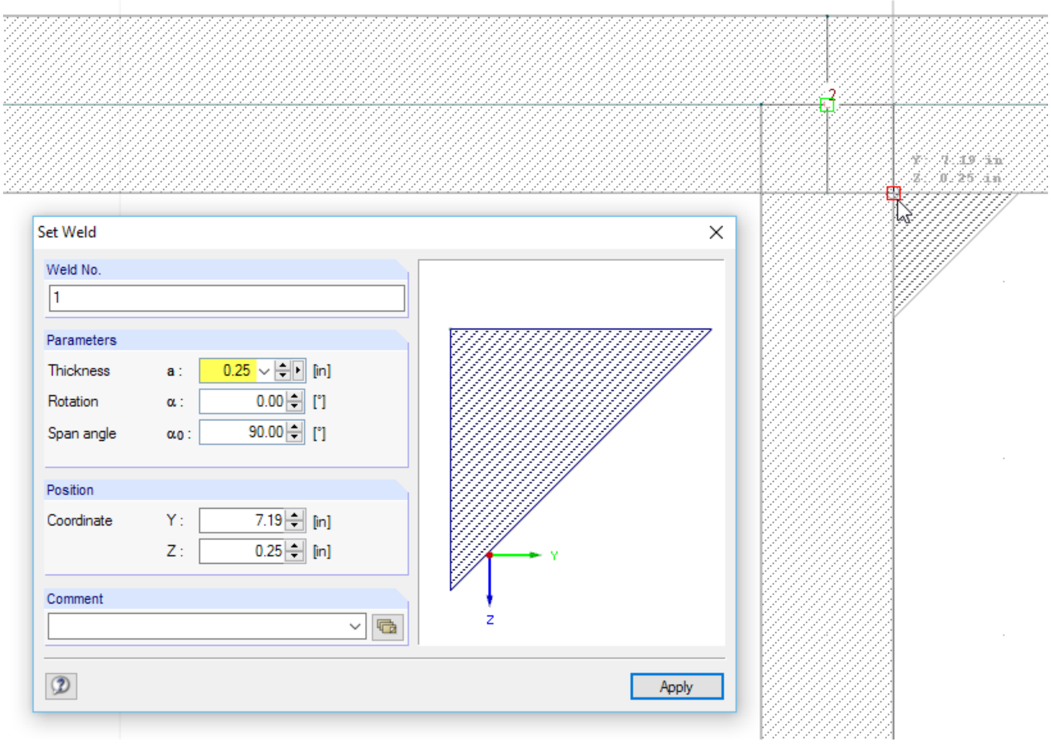


Figure 4.18: Dialog box *Set Weld*

We change the *Thickness* to **0.25 in.** To set the weld, we go to the point with the coordinates (**7.19/0.25**) 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.

To define the second weld, we change the *Rotation* α to **90.0°**. Then we set the weld at the point with the coordinates (**6.81/0.25**).

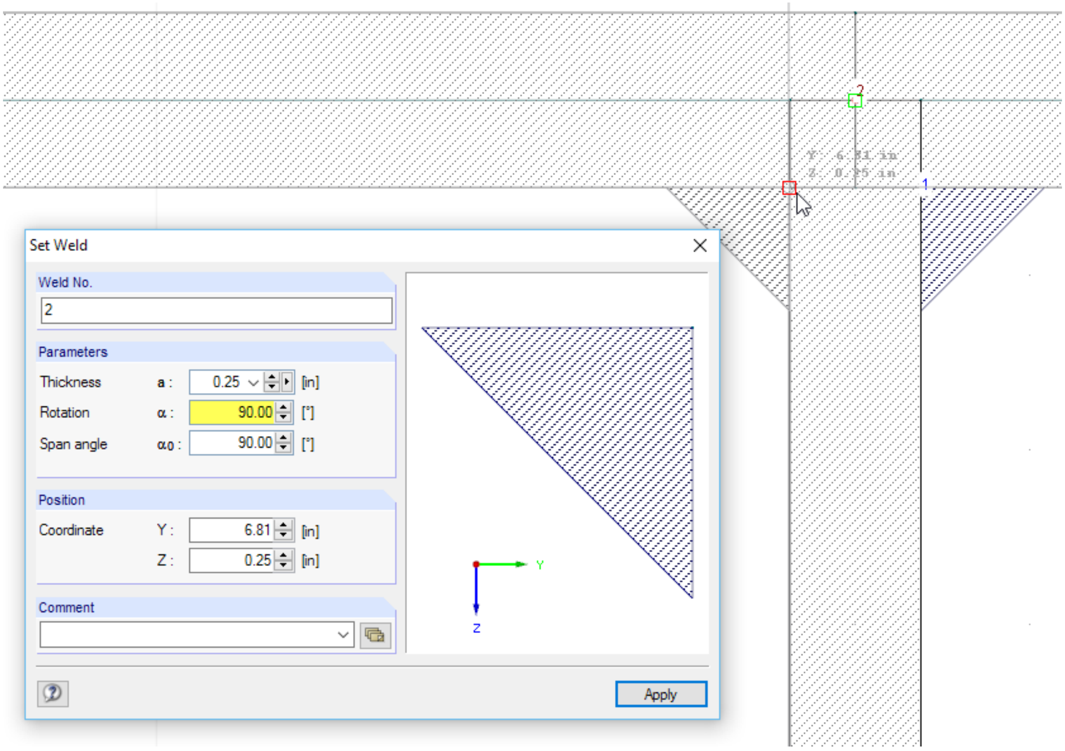


Figure 4.19: Dialog box *Set Weld*

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

4.6 Checking Data

Checking navigator and tables

All defined objects can be found in the directory tree of the *Data* navigator and in the tabs of the table. We can access the navigator items by clicking the ⊞ sign. To switch between the tables, we click can click the different tabs.

On the menu, we can apply **View** → **Navigator** or **View** → **Table** to hide or show the navigator and tables. The corresponding toolbar buttons can be used alternatively.

In the tables, the objects are organized in different tabs. The *Cross-Section* LU 6x4x1/2 is missing, though, as we selected the option *Reduce section into individual elements* when setting the angle.

Graphics, navigator and tables are interactive: To find an element in the table, for example, we select Table *1.4 Elements*. When we click an element in the work window, its corresponding row is highlighted in the table.

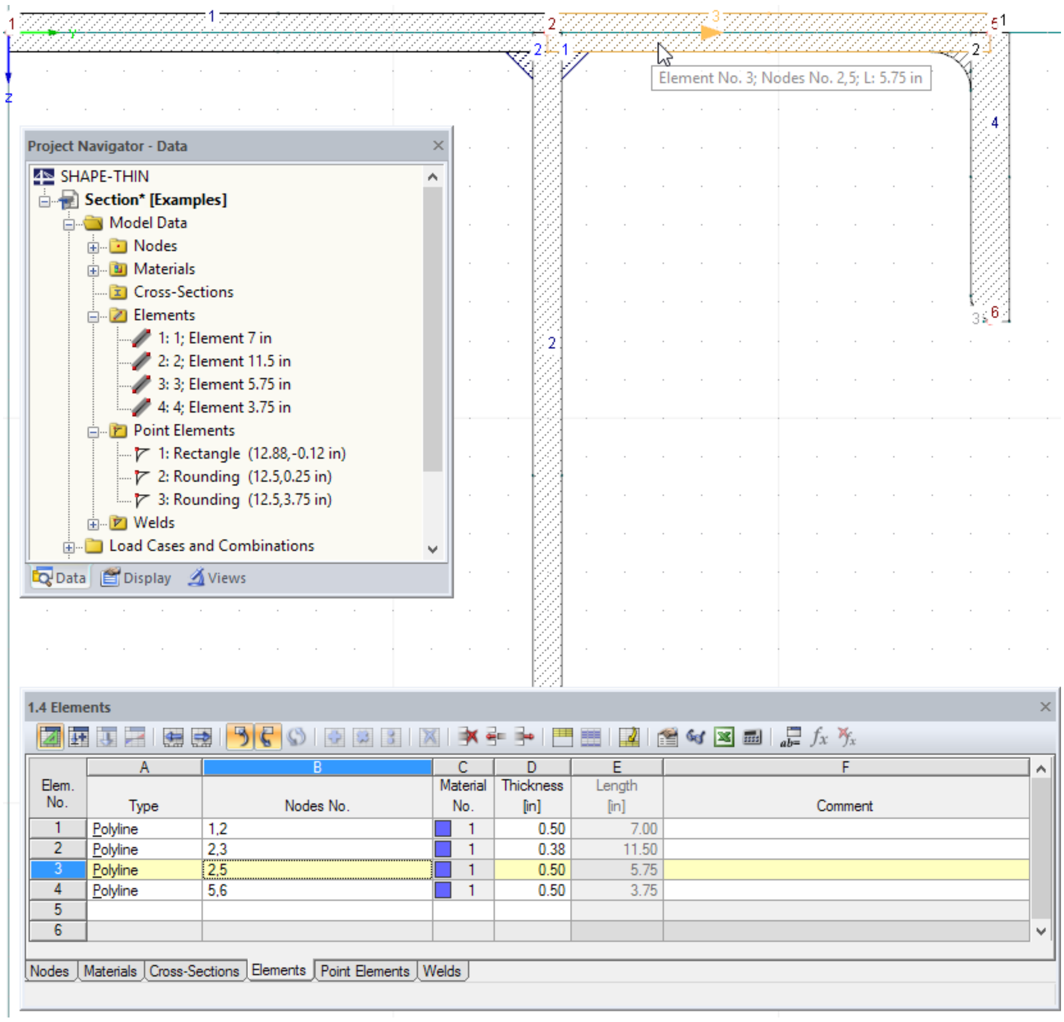


Figure 4.20: Elements in work window, navigator, and table

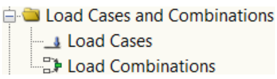
Saving data

Finally, we save our cross-section by selecting the menu

File → **Save**

or by clicking the corresponding button in the toolbar.

5 Loads



The *Data* navigator contains two items in the folder *Load Cases and Combinations*:

- Load cases
- Load combinations

In *load cases*, the specific internal forces due to dead weight, snow or wind loads, etc. can be defined. In *load combinations*, the internal forces of several load cases can be handled which are superimposed with safety factors to account for certain combination rules.

As specified in Chapter 2.2, we will define the internal forces in two separate load cases.

5.1 Load Case 1: Tension and Bending

Creating load case

We use the button to create a new load case.

Figure 5.1: Button [New Load Case]

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

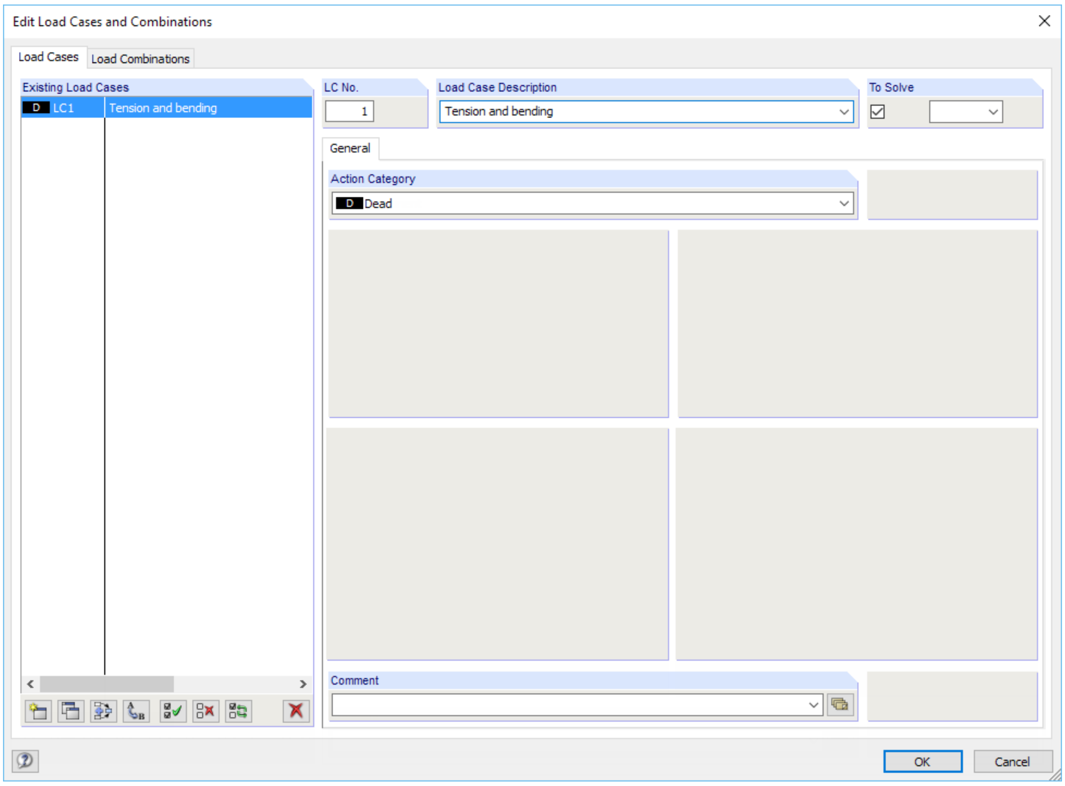


Figure 5.2: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

Load case number *1* is preset with the *Dead* action category. 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 .

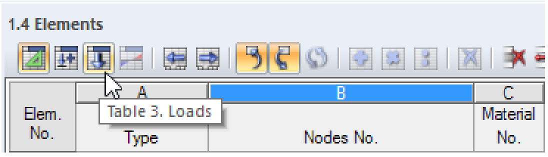
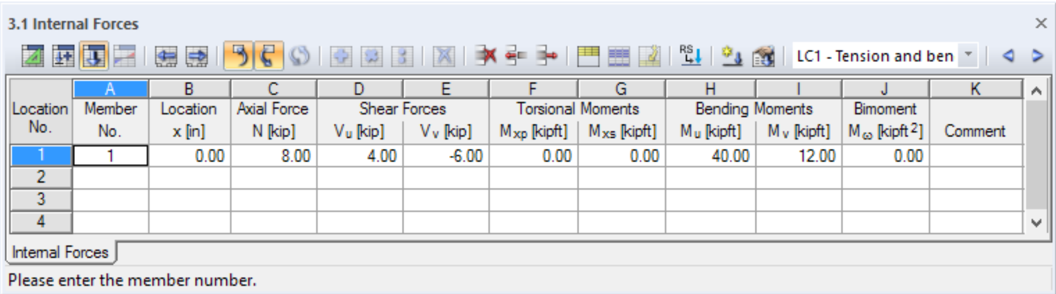


Figure 5.3: Table button [Table 3. Loads]

The table *3.1 Internal Forces* is displayed.

We enter the forces and moments as follows.



Location No.	A Member No.	B Location x [in]	C Axial Force N [kip]	D Shear Forces V_u [kip]	E V_v [kip]	F Torsional Moments M_{xp} [kipft]	G M_{xs} [kipft]	H Bending Moments M_u [kipft]	I M_v [kipft]	J Bimoment M_ω [kipft^2]	K Comment
1	1	0.00	8.00	4.00	-6.00	0.00	0.00	40.00	12.00	0.00	
2											
3											
4											

Figure 5.4: Table *3.1 Internal Forces*

The analysis requires a *Member No.* and a *Location x*. For our example, those references are of minor importance. The allocation of members and design locations is relevant when importing internal forces from RFEM or RSTAB.

SHAPE-THIN handles the algebraic signs of forces and moments as follows:

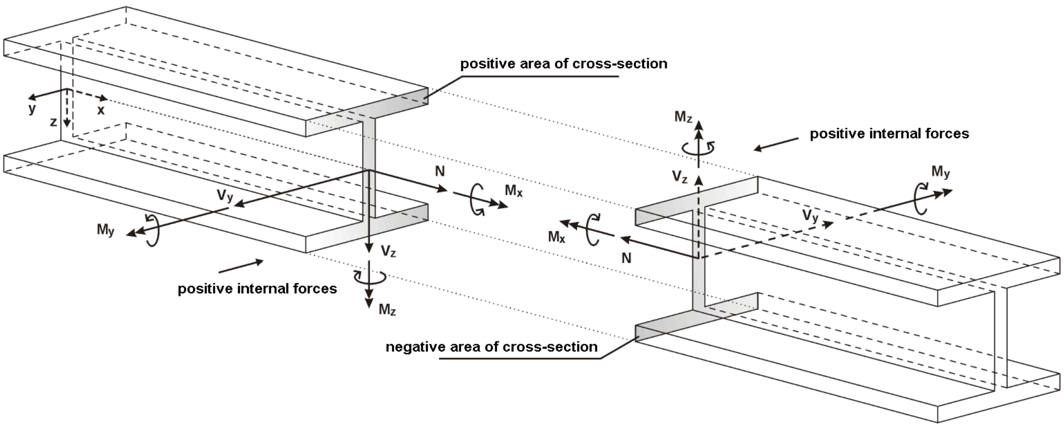


Figure 5.5: Definition of internal forces

Bending moments M_y are 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: Those internal forces are positive if they act on the positive section in a positive direction.

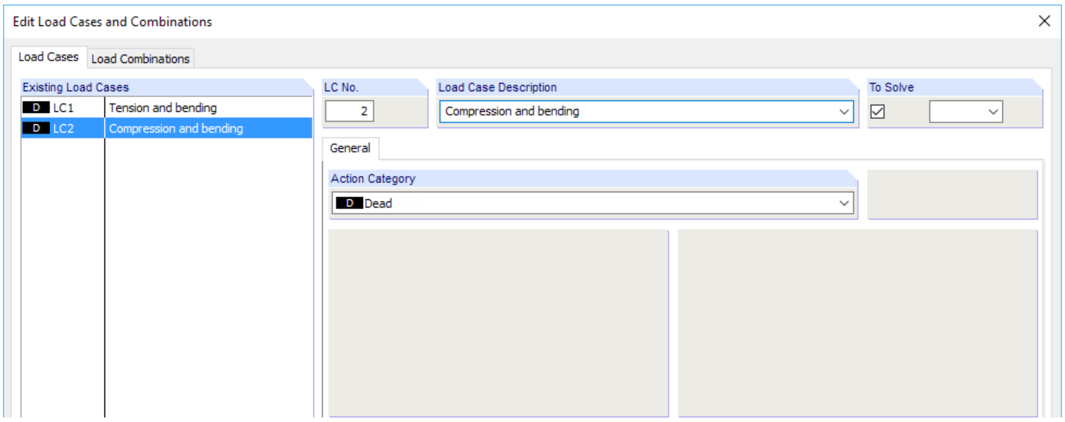
As we can see in the column titles of the shear forces and the moments in Table 3.1, the internal forces are not related to the global axes *y* and *z* as specified in Chapter 2.2 but to the principal axes **u** and **v**. We will adjust that later (see Chapter 6.1, page 23). The defined values will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

5.2 Load Case 2: Compression and Bending

We create a new load case for the second line-up 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).



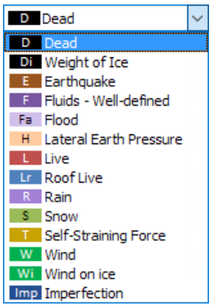


Figure 5.6: Dialog box *Edit Load Cases and Combinations*

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

Then we set the *Action Category* to **Dead** again by selecting it in the list.

After [OK], we make sure that the new load case is set in the toolbar. If it is not, we select *LC2* in the list or set it by clicking the button ▷.

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.

As described in Chapter 2.2, the forces and moments are as follows:

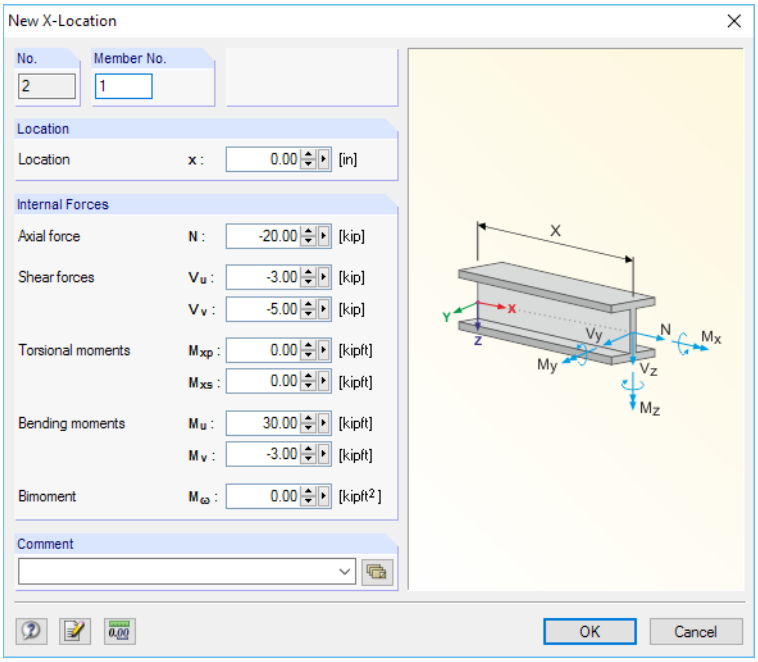


Figure 5.7: Dialog box *New X-Location*

When we have entered the values, we close the dialog box with [OK].

Then we check the data in Table *3.1 Internal Forces* (see Figure 5.8).

5.3 Checking Load Cases

Having defined the two load cases, we can use the buttons ◁ and ▷ in the table toolbar to switch between them.

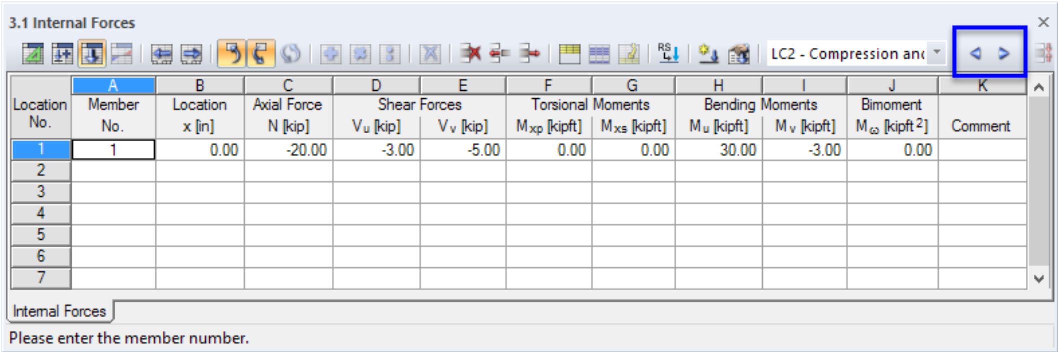


Figure 5.8: Switching between load cases

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

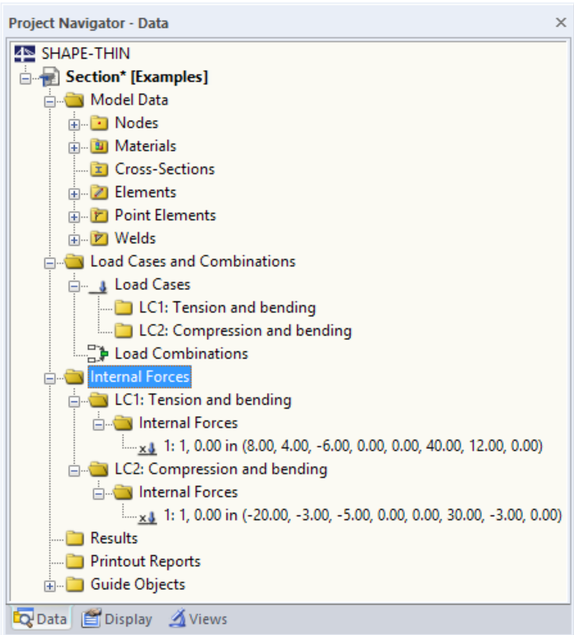


Figure 5.9: Load cases and internal forces in *Data* navigator

We do not define any internal forces for *Load Combinations*.

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

6 Calculation

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

6.1 Adjusting Calculation Parameters

We can access the calculation parameters via the menu

Calculate → Calculation Parameters

or the corresponding button in the toolbar.

The *Calculation Parameters* dialog box opens.

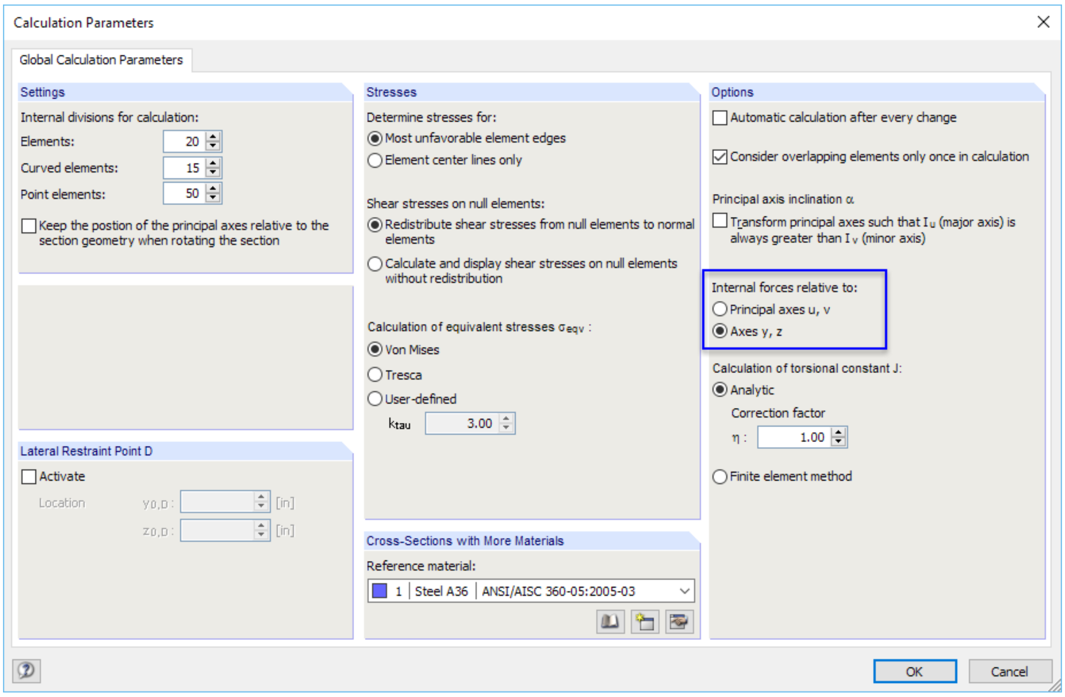
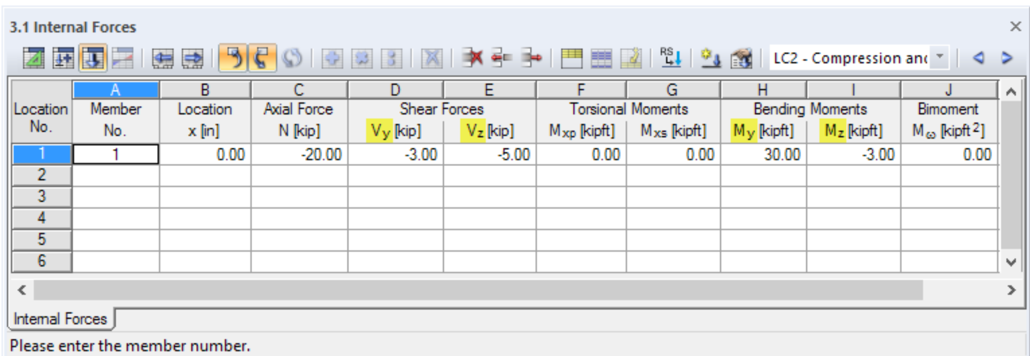


Figure 6.1: Dialog box *Calculation Parameters*, tab *Global Calculation Parameters*

We set the reference of the internal forces to the **Axes y, z**. This change will not affect the defined values of forces and moments.

We click [OK] to confirm the modification.

The table headers now show the correct indices of the shear forces and bending moments.



3.1 Internal Forces

	A	B	C	D	E	F	G	H	I	J
Location No.	Member No.	Location x [in]	Axial Force N [kip]	Shear Forces V_y [kip]	V_z [kip]	Torsional Moments M_{xp} [kipft]	M_{xs} [kipft]	Bending Moments M_y [kipft]	M_z [kipft]	Bimoment M_ω [kipft²]
1	1	0.00	-20.00	-3.00	-5.00	0.00	0.00	30.00	-3.00	0.00
2										
3										
4										
5										
6										

Figure 6.2: Table *3.1 Internal Forces*

6.2 Checking Input Data

SHAPE-THIN includes several possibilities of checking data.

6.2.1 Plausibility Check

We select on the menu

Tools → Plausibility Check

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

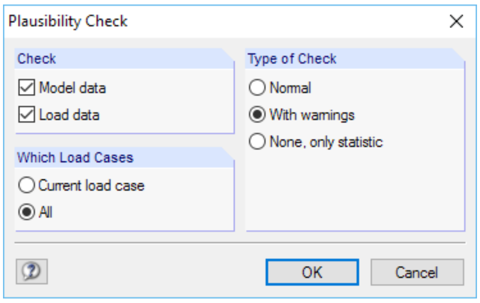


Figure 6.3: Dialog box *Plausibility Check*

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

6.2.2 Checking Interconnecting Elements

Next we make sure that we have modeled an interconnecting cross-section, i.e. the angle is properly connected. We access this check by selecting on the menu

Tools → Model Check → Check for Interconnecting Elements.

SHAPE-THIN displays the following result.

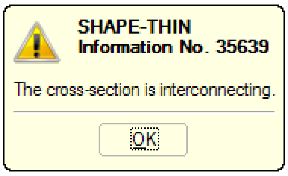


Figure 6.4: Result of model check

If this was not the case, SHAPE-THIN would calculate the section as a bracing shear wall system, without considering the parallel axis theorem (STEINER).

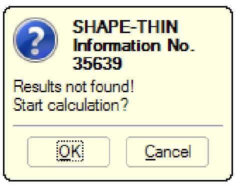
6.3 Calculating Cross-Section

We start the calculation on the menu

Calculate → Calculate All

or by clicking the corresponding button in the toolbar.

Immediately after the calculation, the results are displayed.



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 principal axis *u* of the section which is also displayed in the graphic.

Table *4.1 Section Properties* lists the cross-section parameters.

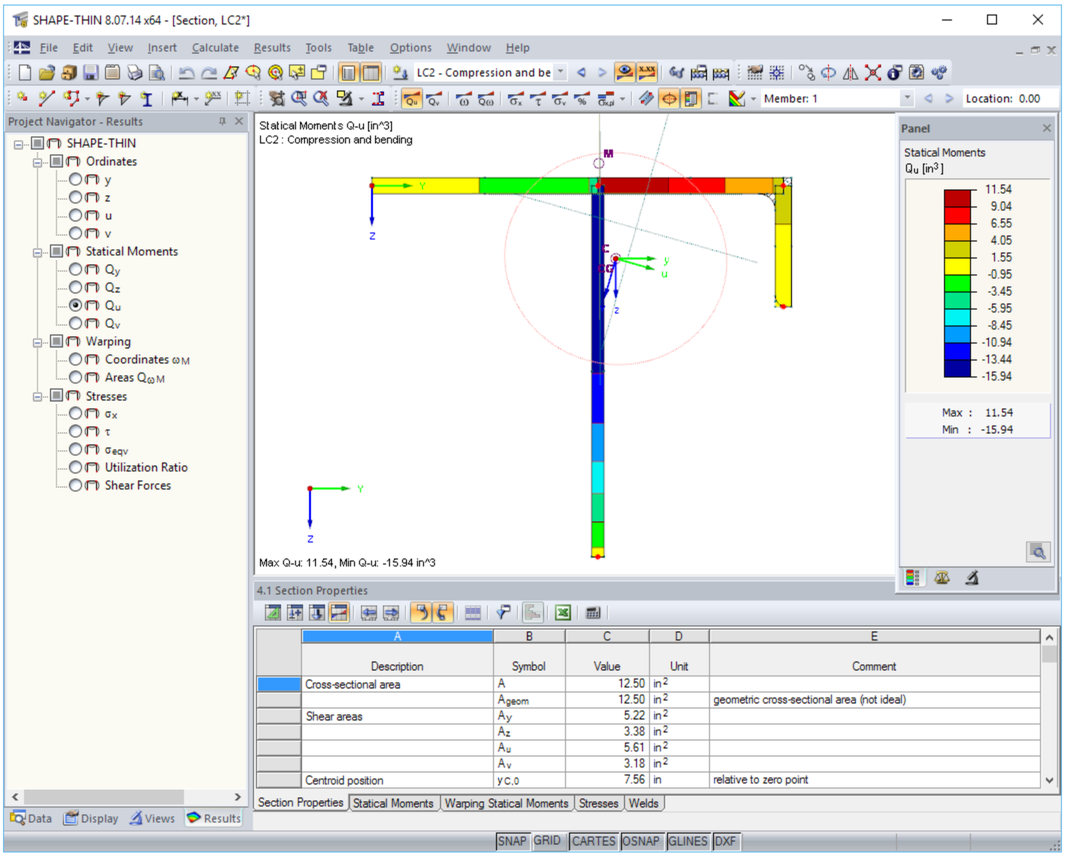
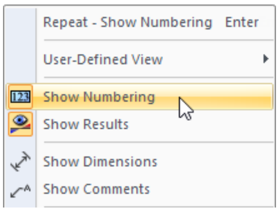
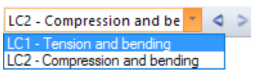


Figure 7.1: Cross-section with gradient of statical moments and principal axes



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 4.8, page 10). There we disable the *Show Numbering* option.

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. Alternatively, we use the list.

Selecting results in navigator

A new navigator, the fourth in its series, manages the result categories for the graphical display (see Figures 7.1 and 7.2).

The *Results* navigator is only shown when the display of results is active. The results can be switched on and off in the *Display* navigator or – faster – by clicking the [Show Results] button which is shown on the left.

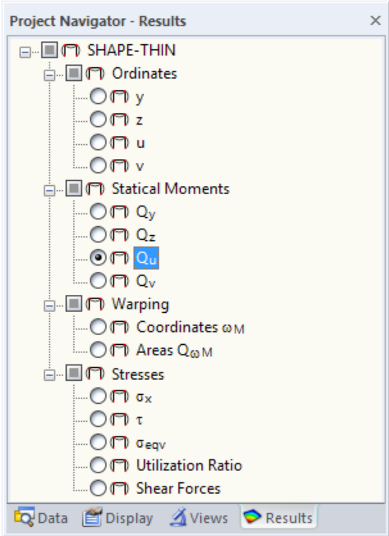
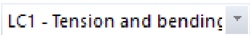


Figure 7.2: *Results* navigator

With the option buttons, we can set the different cross-section properties or the stresses of both load cases for the graphical display.

Normal stresses

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



We make sure that load case **LC1** is set in the toolbar.

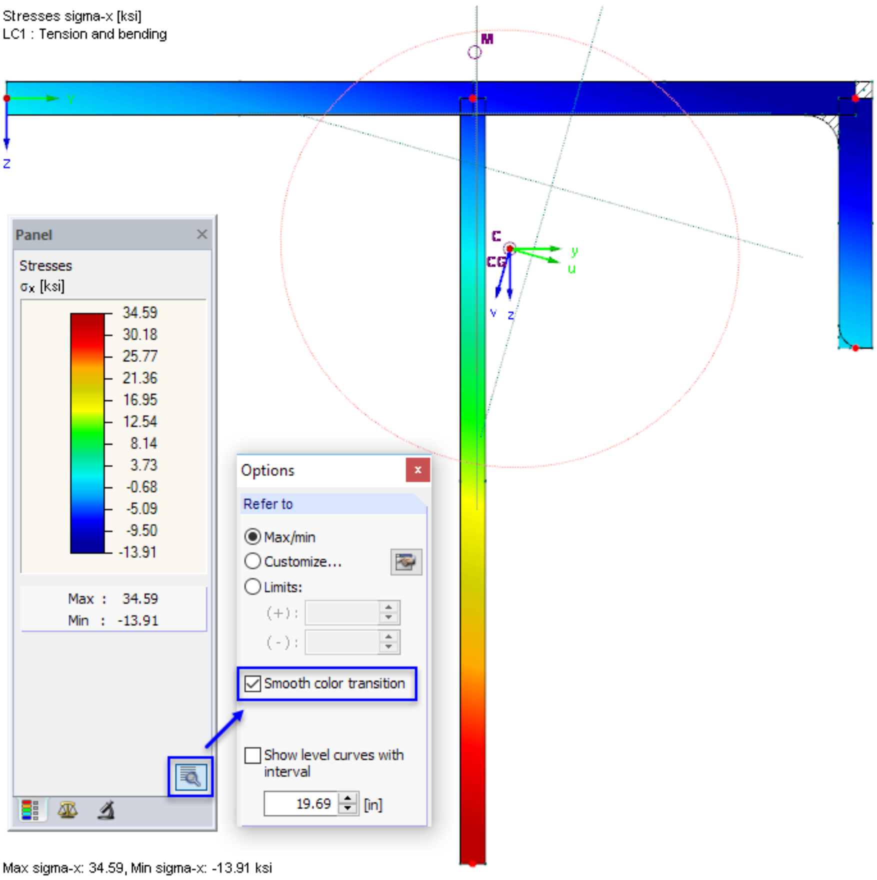
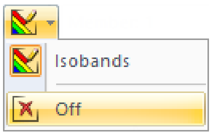


Figure 7.3: Normal stresses σ_x with smooth color transition

When we click the button  in the panel, another dialog box opens. There we can activate the **Smooth color transition** for the graphics of the stresses.

Shear stresses

Next we select the **Stresses** τ in the *Results* navigator, or we use the button shown on the left.



We click the list button [Results As Isobands] to switch the surfaces representation **Off**. The shear stresses are then displayed with hatching.

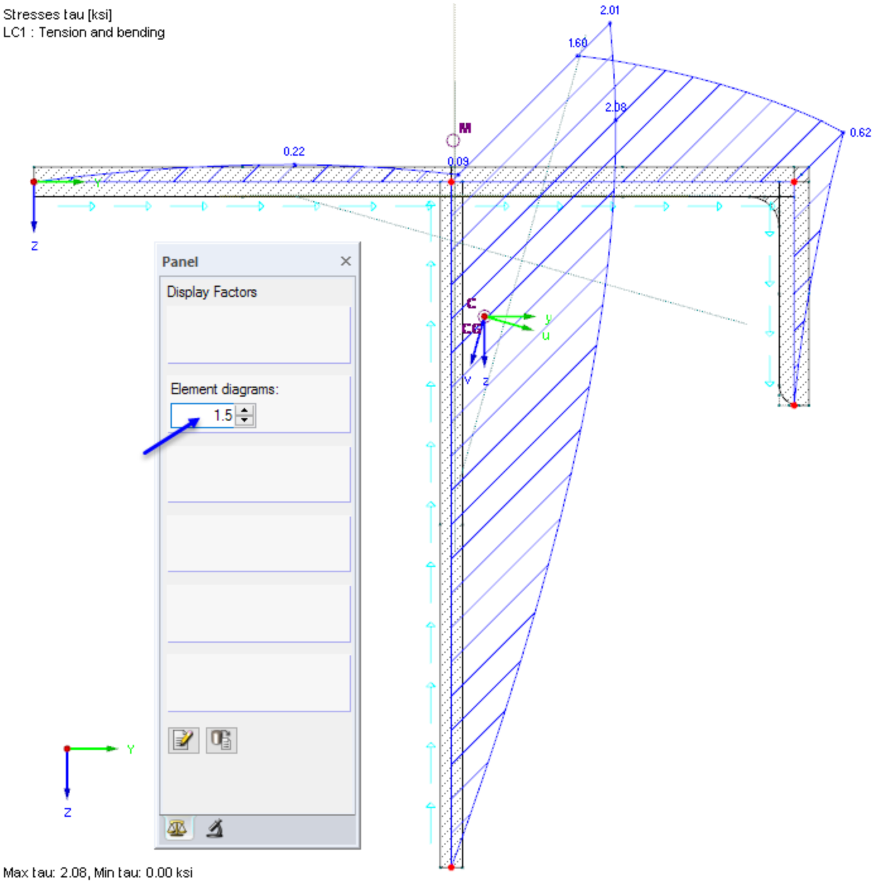


Figure 7.4: Shear stresses τ with scaled up diagrams

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

Toolbar

The *Results* toolbar offers more functions for us to evaluate the results.

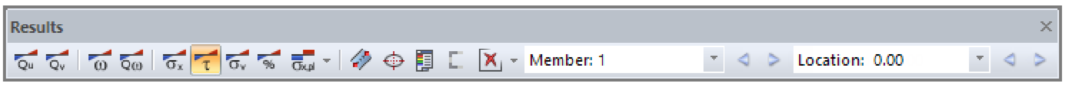


Figure 7.5: Toolbar *Results*

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 (European standards)
	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 listed numerically in tables.

After the calculation, the results tables are set automatically. Table *4.1 Section Properties* contains a multitude of cross-section parameters.

4.1 Section Properties

Description	Symbol	Value	Unit	Comment
Cross-sectional area	A	12.50	in^2	
	A_{geom}	12.50	in^2	geometric cross-sectional area (not ideal)
Shear areas	A_y	5.22	in^2	
	A_z	3.38	in^2	
	A_u	5.61	in^2	
	A_v	3.18	in^2	
Centroid position	$y_{C,0}$	7.56	in	relative to zero point
	$z_{C,0}$	2.26	in	
Moments of inertia	I_y	135.01	in^4	about centroidal axes y, z
	I_z	148.04	in^4	
	I_{yz}	4.01	in^4	
Inclination of principal axes	α	15.80	°	clockwise
Principal moments of inertia	I_u	133.88	in^4	about principal axes u, v in C
	I_v	149.17	in^4	
Polar moments of inertia	I_p	283.05	in^4	
	$I_{p,M}$	394.90	in^4	about shear center M
Radii of gyration	i_y	3.29	in	relative to centroid C
	i_z	3.44	in	
	i_{yz}	0.57	in	
Principal radii of gyration	i_u	3.27	in	about principal axes u, v in C
	i_v	3.45	in	
Polar radii of gyration	i_p	4.76	in	
	$r_{p,M}$	5.62	in	about shear center M
Warping radius of gyration	$i_{\omega,M}$	0.68	in	
Cross-section weight	G	42.53	lb/ft	
Cross-section perimeter	U	55.73	in	incl. inner side of cells
Torsional constant	I_t	0.87	in^4	calculated analytically
Secondary torsional constant	$I_{t,s}$	21.20	in^4	
Location of the shear center	$y_{M,0}$	7.03	in	relative to zero point
	$z_{M,0}$	-0.69	in	
	y_M	-0.53	in	relative to centroid C
	z_M	-2.95	in	
Warping constants	$I_{\omega,C}$	1502.45	in^6	relative to centroid C
	$I_{\omega,M}$	182.82	in^6	about shear center M
Auxiliary value for warp rotation	$r_{\omega,M}$	-0.960		
Section moduli	$W_{u,max}$	14.72	in^3	
	$W_{u,min}$	-34.38	in^3	in distance -3.89 in
	$W_{v,max}$	26.43	in^3	in distance 5.64 in
	$W_{v,min}$	-18.75	in^3	in distance -7.95 in
	$W_{y,max}$	14.61	in^3	in distance 9.24 in
	$W_{y,min}$	-53.84	in^3	in distance -2.51 in
	$W_{z,max}$	27.20	in^3	in distance 5.44 in
	$W_{z,min}$	-19.59	in^3	in distance -7.56 in
Warping section moduli	$W_{\omega,M,max}$	35.47	in^4	in node 4
	$W_{\omega,M,min}$	-11.22	in^4	in node 5
Torsional section modulus	W_t	1.75	in^3	
Stability parameters	r_u	2.98	in	
	r_v	-0.19	in	
	$r_{M,u}$	2.39	in	
	$r_{M,v}$	8.37	in	
Reduction factor	λ_M	0.04	1/in	
Max. plastic bending moments	$M_{pl,y,d}$	85.001	kipft	
	$M_{pl,z,d}$	95.503	kipft	
	$M_{pl,u,d}$	86.356	kipft	
	$M_{pl,v,d}$	101.177	kipft	
Max. plastic section moduli	$W_{pl,y}$	28.33	in^3	$\alpha_{pl,y}$: 1.94
	$W_{pl,z}$	31.83	in^3	$\alpha_{pl,z}$: 1.63
	$W_{pl,u}$	28.79	in^3	$\alpha_{pl,u}$: 1.96
	$W_{pl,v}$	33.73	in^3	$\alpha_{pl,v}$: 1.80

Section Properties | Statical Moments | Warping Statical Moments | Stresses | Welds

Inclination of principal axis with respect to centroidal y-axis, positive counter-clockwise

Figure 7.6: Table *4.1 Section Properties*

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

The statical moments, warping statical moments and stresses are each listed on the start and end nodes, as well as in the element centers.

Filtering stresses

We set Table *4.5 Stresses*. With the [Result Filter] button in the table toolbar, we can define which results are to be displayed in this table.

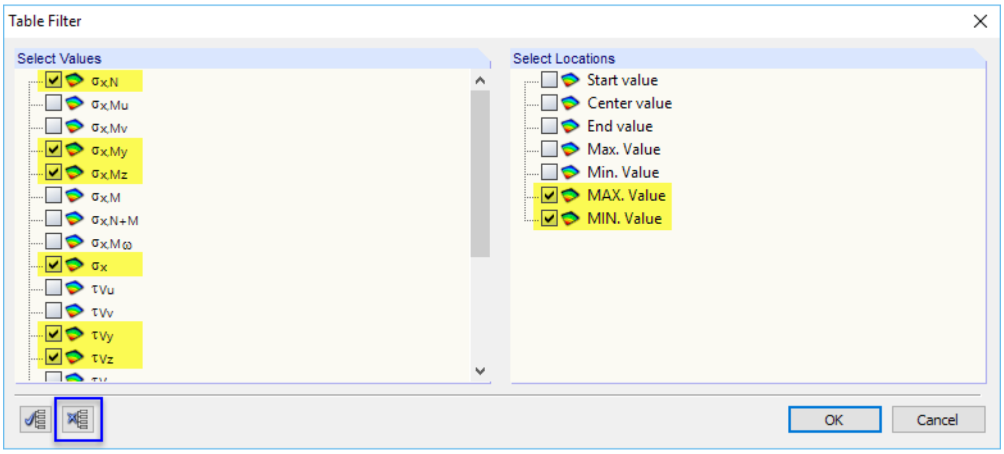
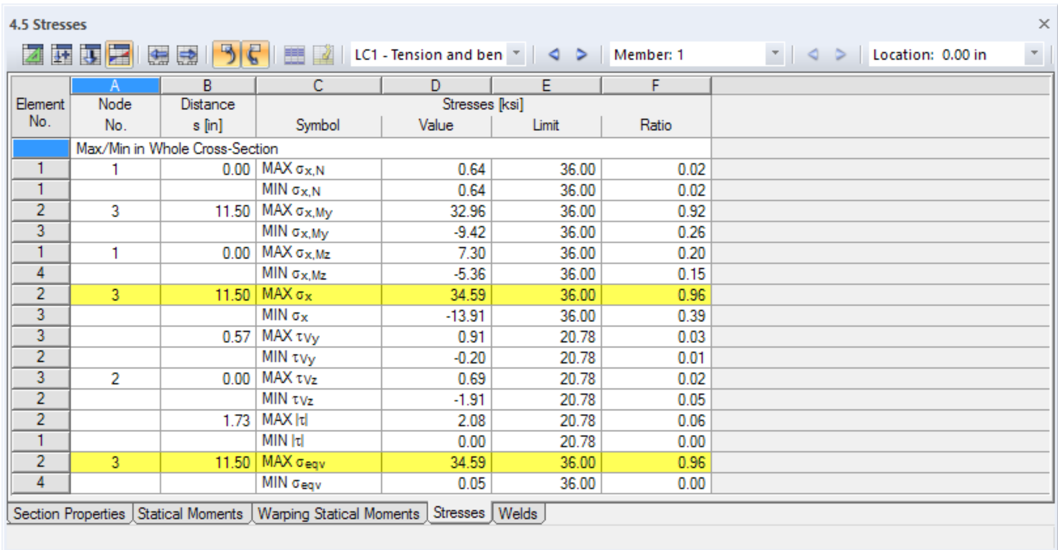


Figure 7.7: Dialog box *Table Filter*

We remove all check marks by clicking the button. Then we select some relevant stress types (see Figure 7.7). Furthermore, we want to display the **MAX** and **MIN** values of the entire section exclusively.

Having clicked [OK], we see only the extreme values of the selected stresses in Table *4.5 Stresses*.



4.5 Stresses — LC1 - Tension and ben — Member: 1 — Location: 0.00 in

Element No.	A Node No.	B Distance s [in]	C Symbol	D Stresses [ksi] Value	E Limit	F Ratio
	Max/Min in Whole Cross-Section					
1	1	0.00	MAX $\sigma_{x,N}$	0.64	36.00	0.02
1			MIN $\sigma_{x,N}$	0.64	36.00	0.02
2	3	11.50	MAX $\sigma_{x,My}$	32.96	36.00	0.92
3			MIN $\sigma_{x,My}$	-9.42	36.00	0.26
1	1	0.00	MAX $\sigma_{x,Mz}$	7.30	36.00	0.20
4			MIN $\sigma_{x,Mz}$	-5.36	36.00	0.15
2	3	11.50	MAX σ_x	34.59	36.00	0.96
3			MIN σ_x	-13.91	36.00	0.39
3		0.57	MAX τ_{Vy}	0.91	20.78	0.03
2			MIN τ_{Vy}	-0.20	20.78	0.01
3	2	0.00	MAX τ_{Vz}	0.69	20.78	0.02
2			MIN τ_{Vz}	-1.91	20.78	0.05
2		1.73	MAX $\lvert\tau\rvert$	2.08	20.78	0.06
1			MIN $\lvert\tau\rvert$	0.00	20.78	0.00
2	3	11.50	MAX σ_{eqv}	34.59	36.00	0.96
4			MIN σ_{eqv}	0.05	36.00	0.00

Section Properties | Statical Moments | Warping Statical Moments | Stresses | Welds

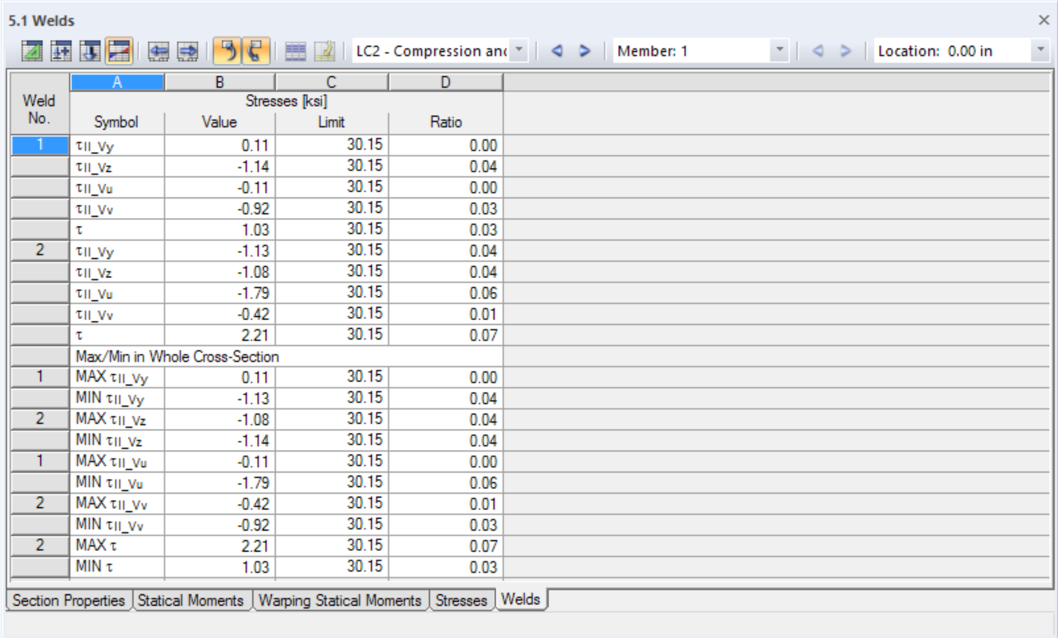
Figure 7.8: Table *4.5 Stresses* with filtered results

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

The maximum stresses are determined for the internal forces of LC1, with a *Ratio* of 0.96 in node 3.

Checking weld stresses

The stresses of the two fillet welds are listed in Table *5.1 Welds*.



5.1 Welds

Weld No.	Symbol	Stresses [ksi] Value	Limit	Ratio
1	τII_Vy	0.11	30.15	0.00
	τII_Vz	-1.14	30.15	0.04
	τII_Vu	-0.11	30.15	0.00
	τII_Vv	-0.92	30.15	0.03
	τ	1.03	30.15	0.03
2	τII_Vy	-1.13	30.15	0.04
	τII_Vz	-1.08	30.15	0.04
	τII_Vu	-1.79	30.15	0.06
	τII_Vv	-0.42	30.15	0.01
	τ	2.21	30.15	0.07
	Max/Min in Whole Cross-Section			
1	MAX τII_Vy	0.11	30.15	0.00
	MIN τII_Vy	-1.13	30.15	0.04
2	MAX τII_Vz	-1.08	30.15	0.04
	MIN τII_Vz	-1.14	30.15	0.04
1	MAX τII_Vu	-0.11	30.15	0.00
	MIN τII_Vu	-1.79	30.15	0.06
2	MAX τII_Vv	-0.42	30.15	0.01
	MIN τII_Vv	-0.92	30.15	0.03
2	MAX τ	2.21	30.15	0.07
	MIN τ	1.03	30.15	0.03

Figure 7.9: Table *5.1 Welds* for LC2

Similarly to the table of stresses, the contents can be controlled in the *Table Filter* dialog box.

7.3 Multiple Windows View

The cross-section diagrams and stresses can be displayed side by side in different windows. This function can be accessed on the menu

Results → **Arrange Result Windows**.

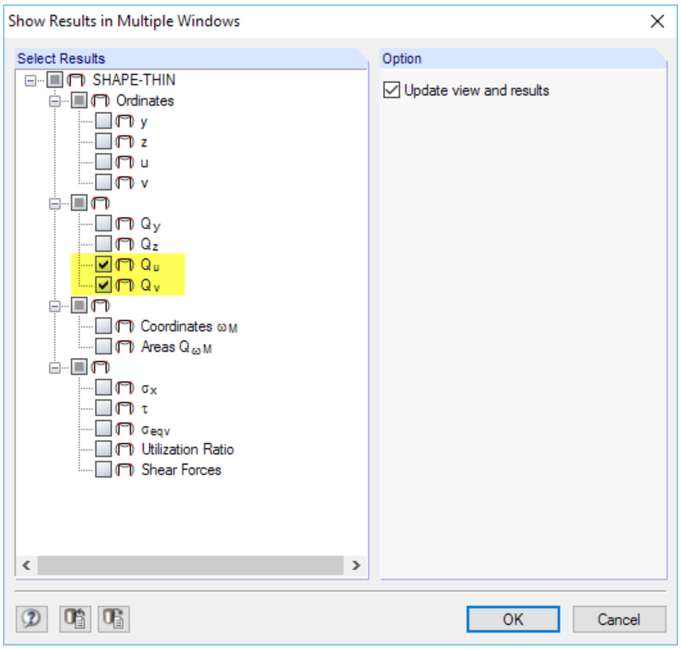


Figure 7.10: Dialog box *Show Results in Multiple Windows*

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

After [OK], both cross-section diagrams are displayed next to each other in two windows.

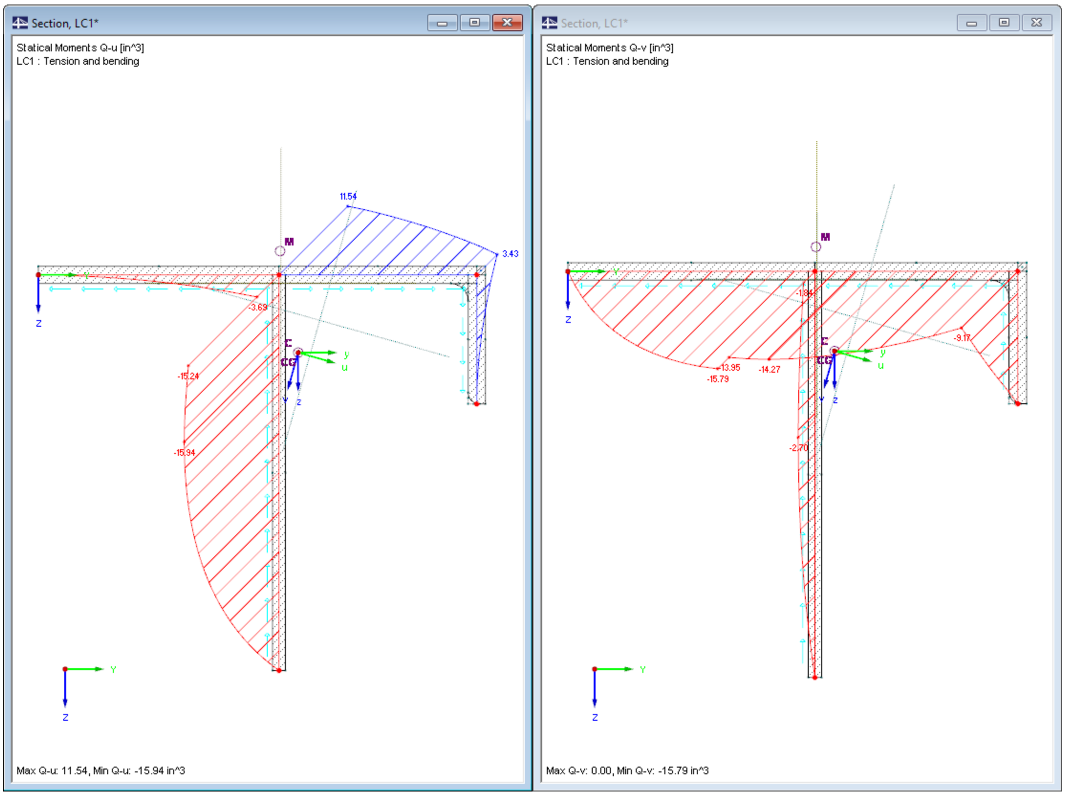


Figure 7.11: Statical moments Q_u and Q_v

8 Documentation

8.1 Creating Printout Report

For the documentation, SHAPE-THIN includes a print preview – the *Printout report*. This tool enables us to define which input data and results are to be printed. We can also add pictures or insert texts.

We initiate the printout report with the [Current Printout Report] button. It is located to the right of the printer button in the toolbar.

A dialog box opens where it is possible to define a *Description* and select a *Template* for the new printout report.

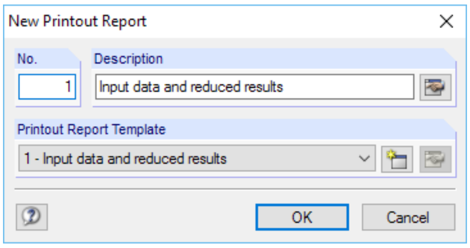


Figure 8.1: Dialog box *New Printout Report*

We accept the default settings as seen in Figure 8.1 and create the print preview by clicking [OK].

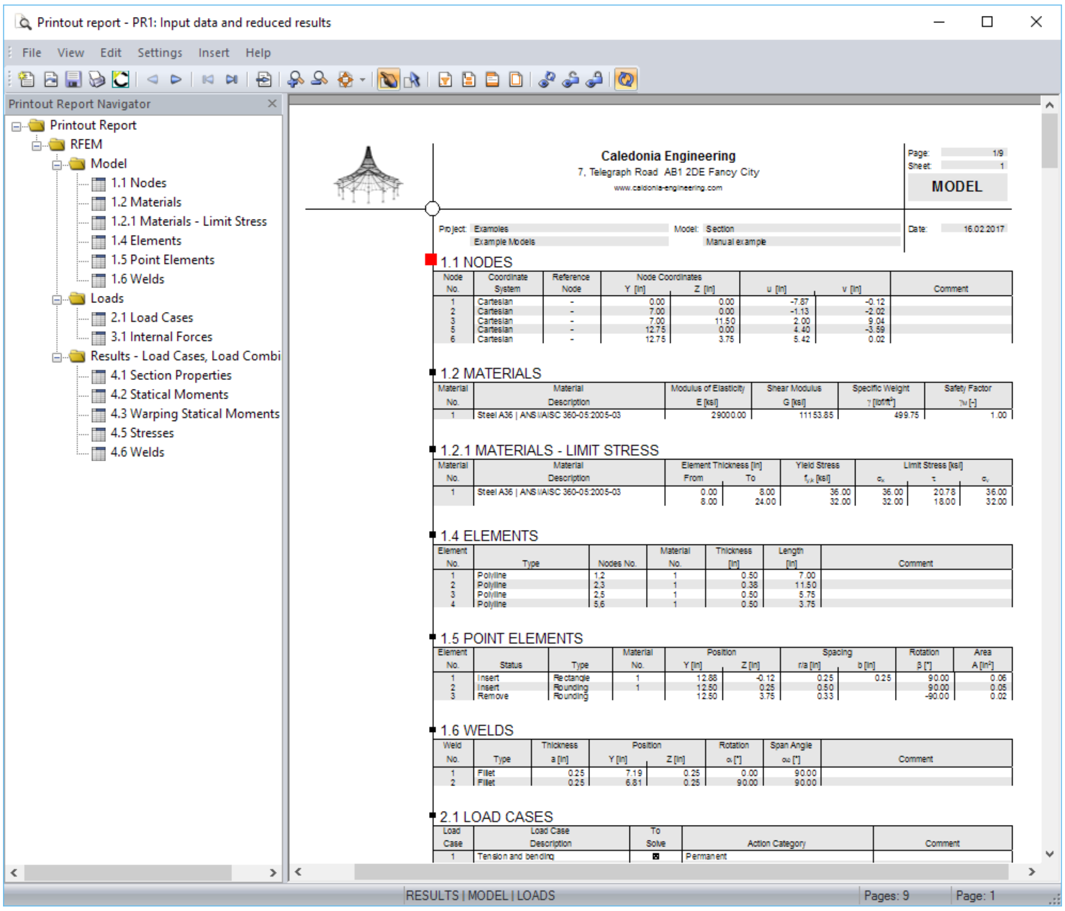


Figure 8.2: Print preview in printout report

8.2 Adjusting Printout Report

On the left, we see the navigator with the chapters selected for printing. When we click one of those items, the data of the corresponding chapter is shown in the preview.

The contents of the printout report can be adjusted individually. For our example, we will reduce the output of the results. We right-click the *Results - Load Cases, Load Combinations* item in the navigator. On the shortcut menu, we click *Selection*.

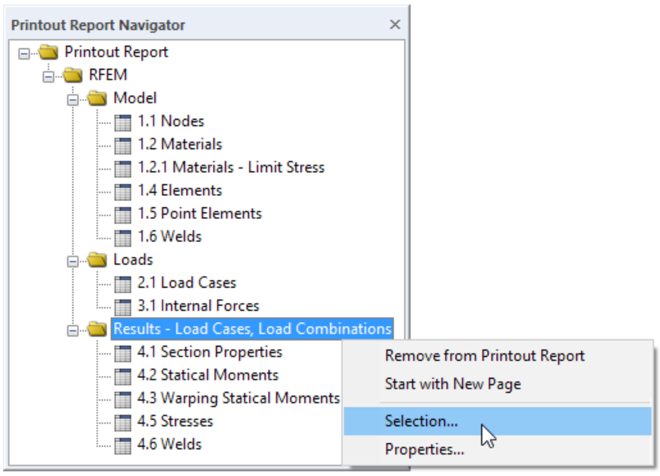


Figure 8.3: Shortcut menu for *Results - Load Cases, Load Combinations*

The *Printout Report Selection* dialog box opens. On the *LC/CO Results* tab, we remove the check marks of the **4.2 Statical Moments** and **4.3 Warping Statical Moments** tables.

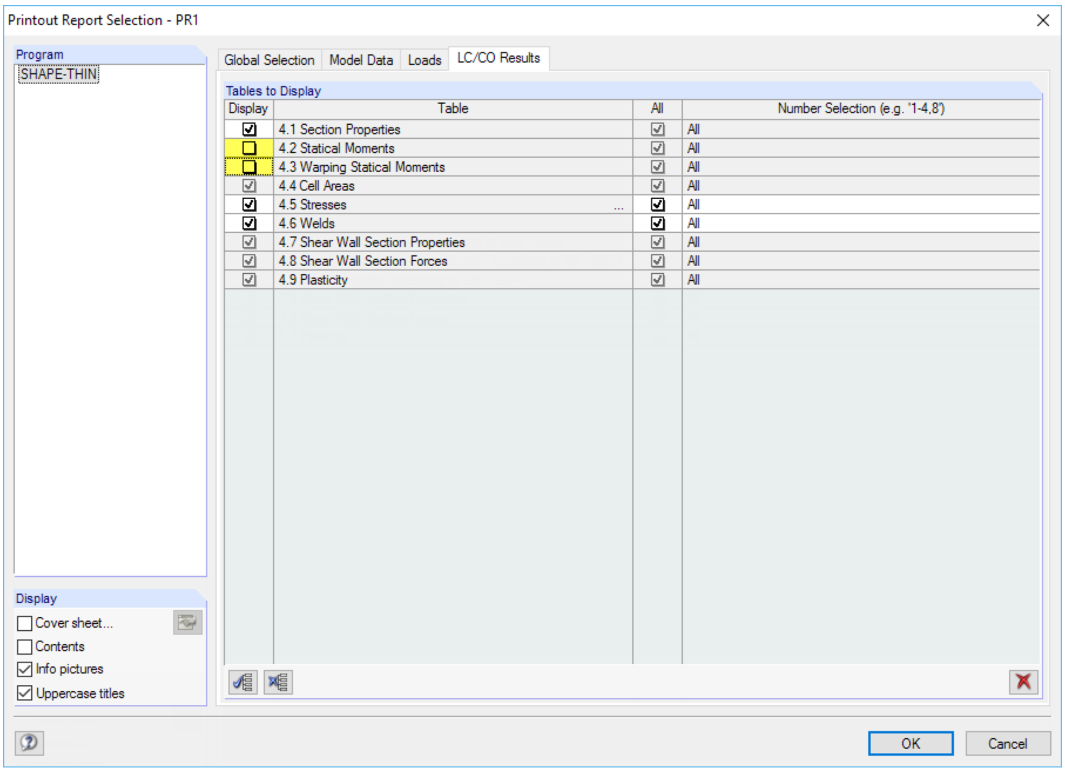


Figure 8.4: Deactivating statical and warping statical moments in *Printout Report Selection* dialog box

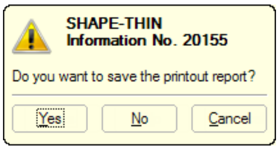
When we have closed the dialog box with [OK], the preview is adjusted accordingly.

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

8.3 Printing Graphics

The printout of the cross-section data can be illustrated by pictures.

Printing statical moments



We close the printout report with .

When we are asked whether to save our changes, we click [Yes]. We return to the work window.

As the result diagrams of the statical moments Q_u and Q_v were set last (see Figure 7.10, page 30), we print those two images to the printout report. The print option can be selected on the menu

File → Print Graphic

or by clicking the corresponding button in the toolbar.

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

As we print *More* windows, we adjust the *Window Arrangement* with the button . We set option number **3)** to print the two images one below the other.

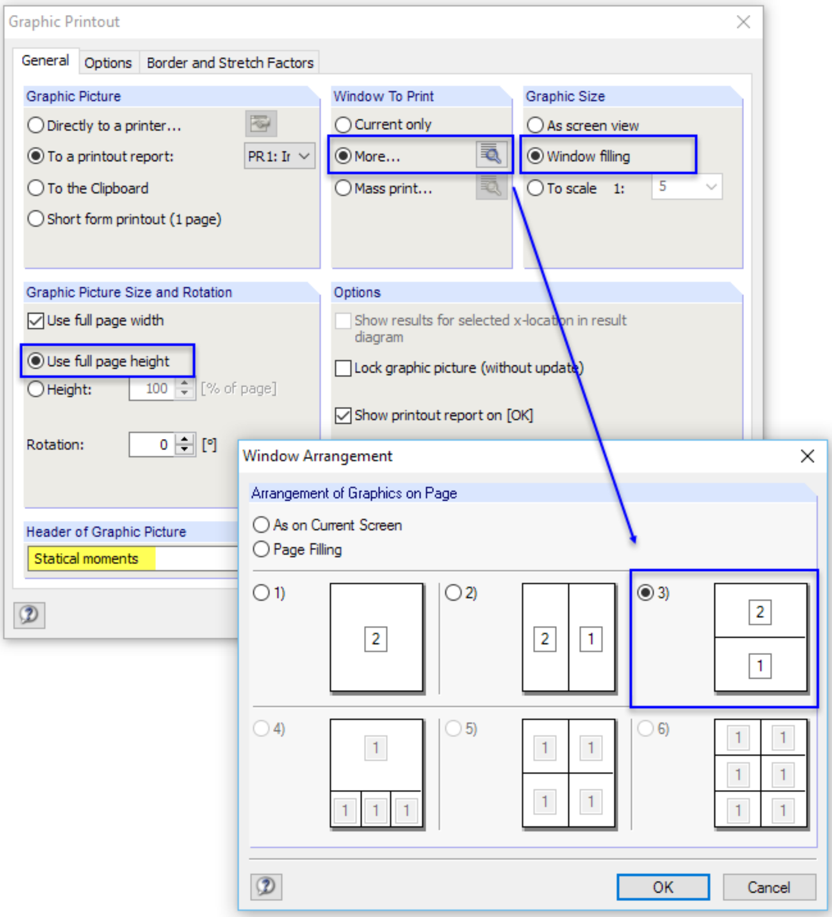


Figure 8.5: Dialog boxes *Graphic Printout* and *Window Arrangement*

We do not change the default settings of the other tabs.

By clicking [OK], we finally print the two diagrams of the statical moments to the printout report. The images are placed at the end of Chapter *Results - Load Cases, Load Combinations*.

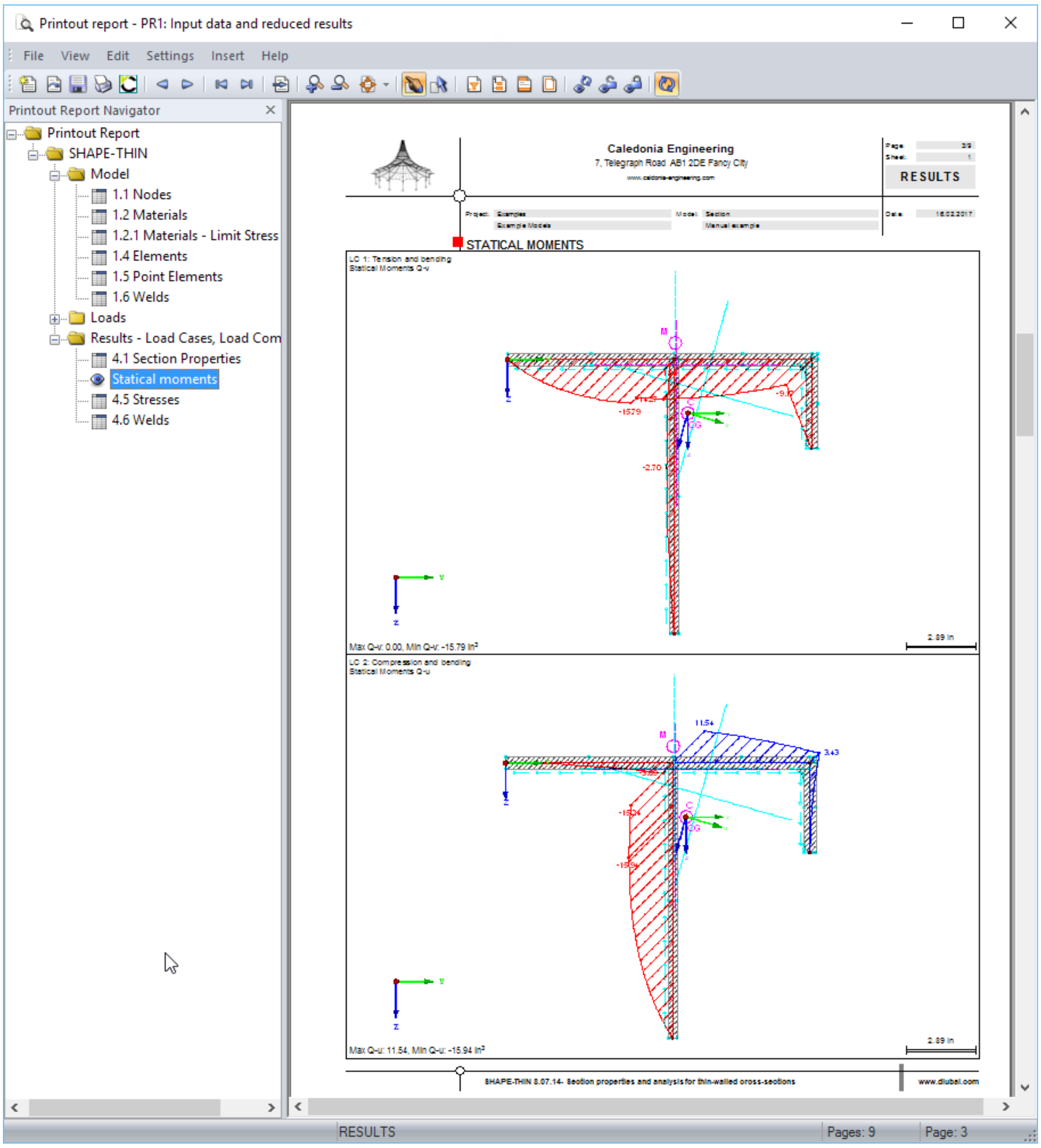
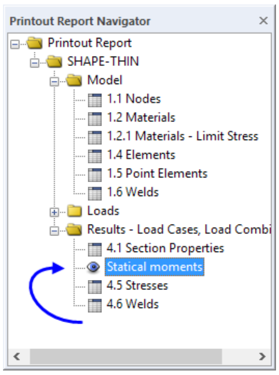


Figure 8.6: Statical moments in printout report



We move the image up to Chapter *4.1 Section Properties* by using the drag-and-drop function.

Printing equivalent stresses

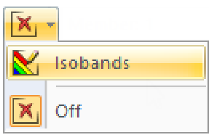
We close the printout report and save the modifications.

Now we close one of the windows in the SHAPE-THIN work window with . If a message appears asking us whether we want to save the data, we confirm it with [Yes].

Then we maximize the remaining window with .

With the button or the function key [F8], we set the window filling view of the section.

Next we display the **equivalent stresses** $\sigma_{\mathbf{eqv}}$ of load case **LC2**.



We select the [Isobands] again for a colored representation of the stresses.

We click the button to open the *Graphic Printout* dialog box again.

For the print, we set the *Height* of the image to **50 %**. Thus, the graphic will be half of a page in the printout report. Furthermore, we want *To Scale* the graphic with **1:5**.

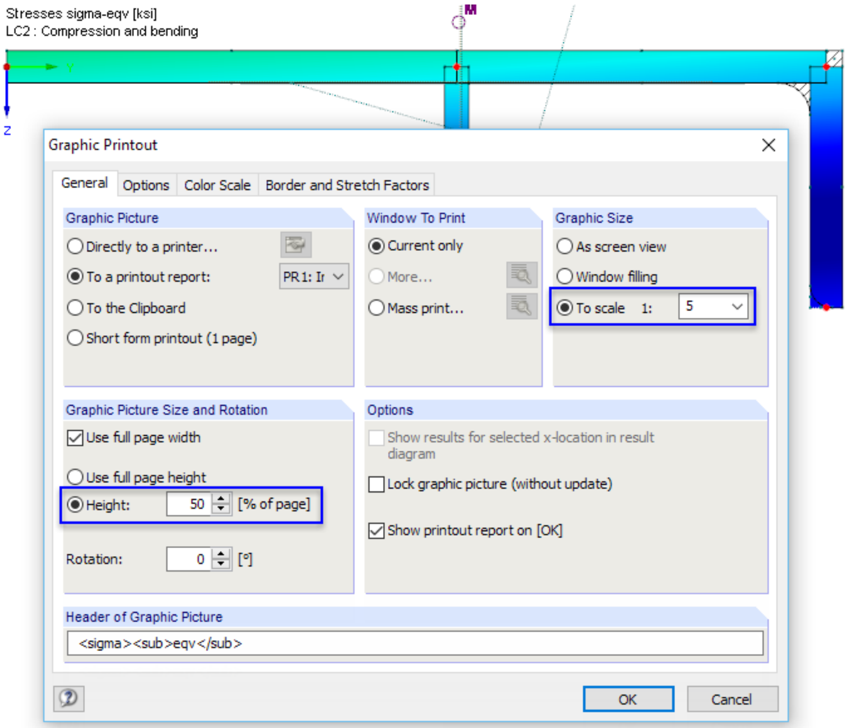
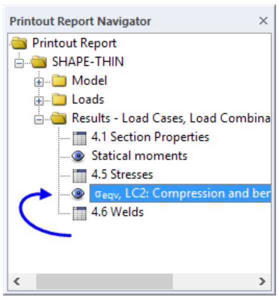


Figure 8.7: Printing equivalent stresses



We click [OK]. The image is printed to the printout report where it is placed at the very end again. We move it up to Chapter *Stresses*.

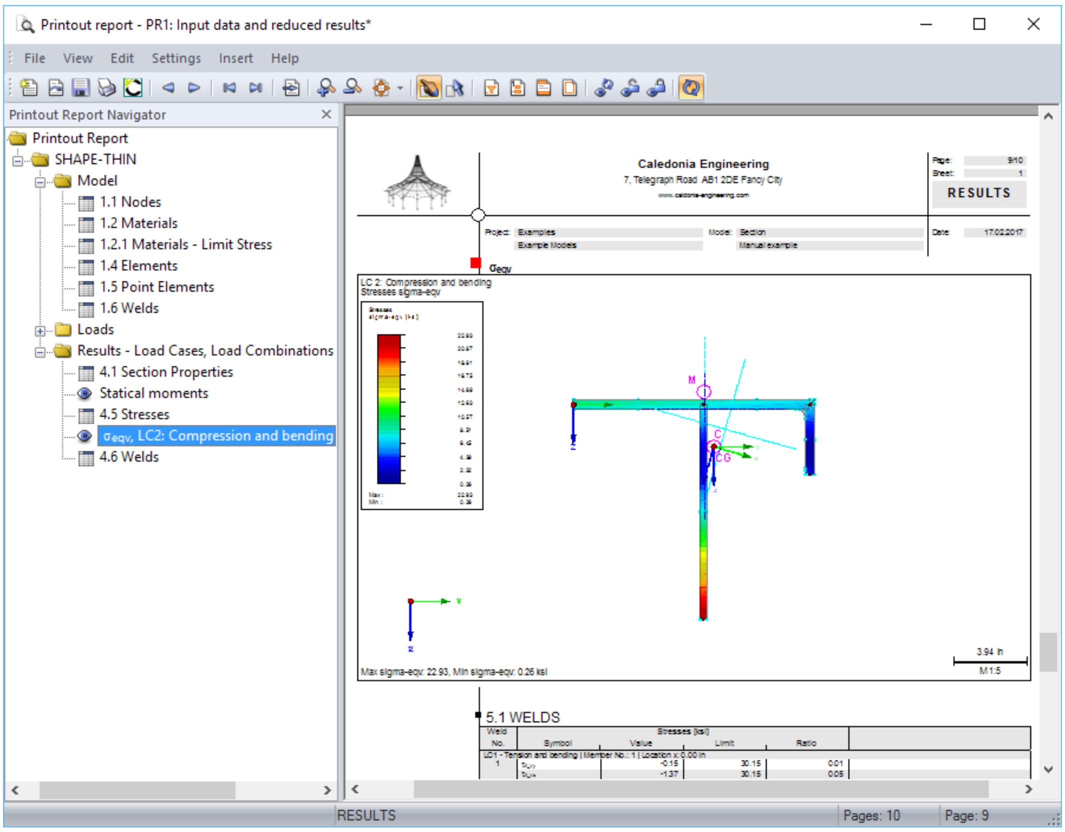


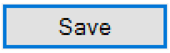
Figure 8.8: Graphic of equivalent stresses in printout report

Printing the printout report

To print the printout report now, we could send it to the printer by clicking the [Print] button.

We wil use the integrated PDF printer, however, and create a PDF file of the data. This option is available on the printout report menu

File → Export to PDF.



The Windows *Save As* dialog box opens where we can specify the folder and the file name. Then we click [Save].

The PDF file is created. The single chapters are listed as bookmarks which facilitate the navigation.

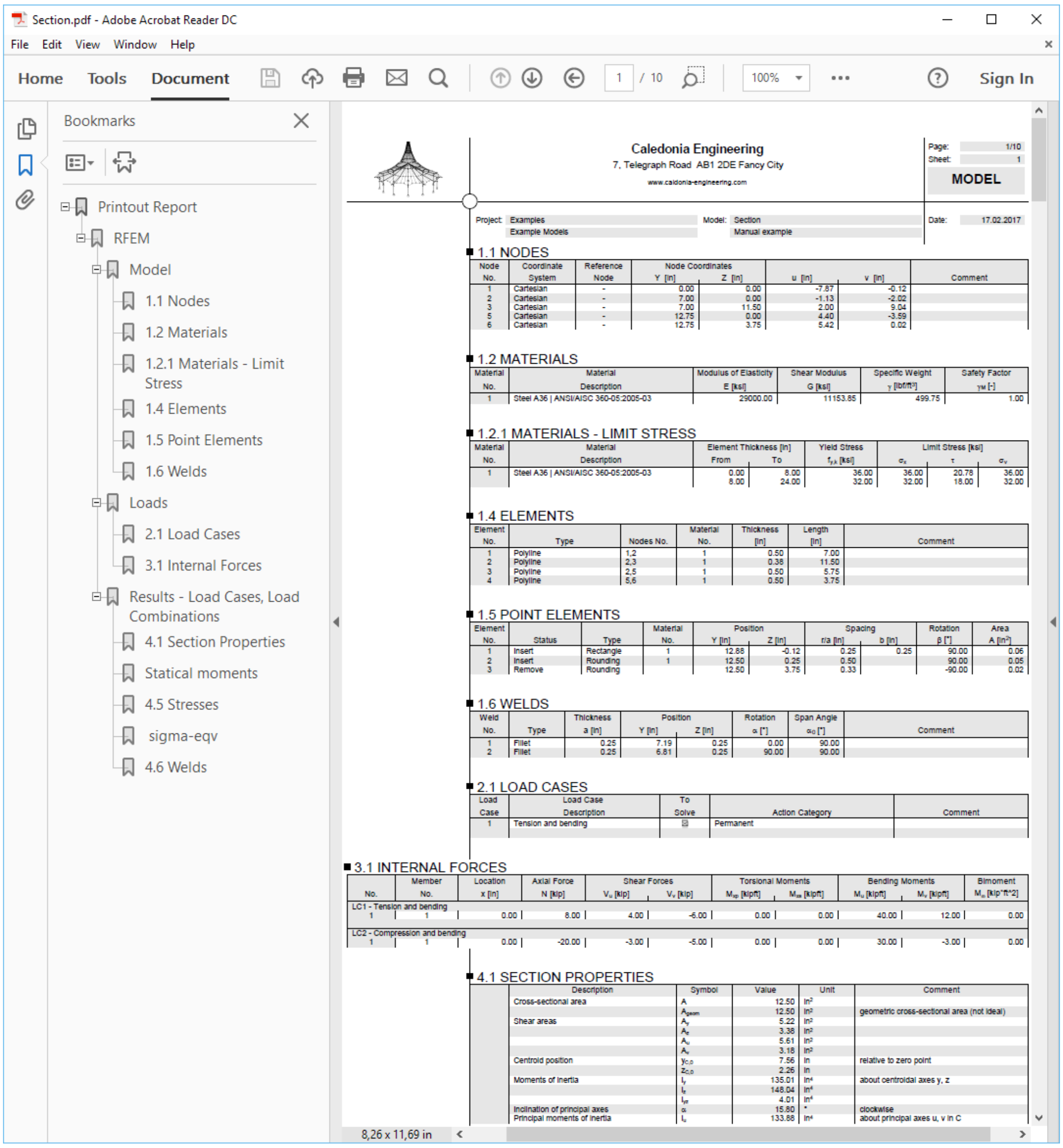


Figure 8.9: PDF file of printout report

9 Outlook

Having arrived at the end of our example, we hope that this introduction was useful for you to get familiar with SHAPE-THIN. It should also have made you curious about some more functions of the program. You can find the detailed description of SHAPE-THIN in the user manual that you can download on our website.

The online help of the program, which is based on the SHAPE-THIN manual, can be accessed on the *Help* menu or with [F1].

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.

The cross-section of our example can be imported to a model of RFEM or RSTAB. In those programs, you can access any SHAPE-THIN section in the library of cross-sections.

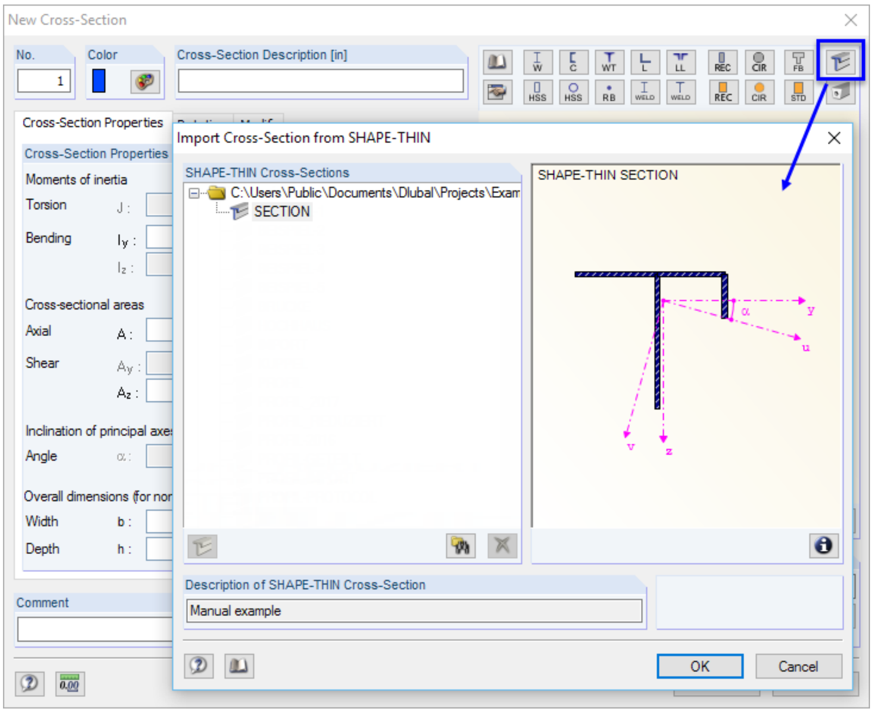


Figure 9.1: Importing SHAPE-THIN section to RFEM/RSTAB

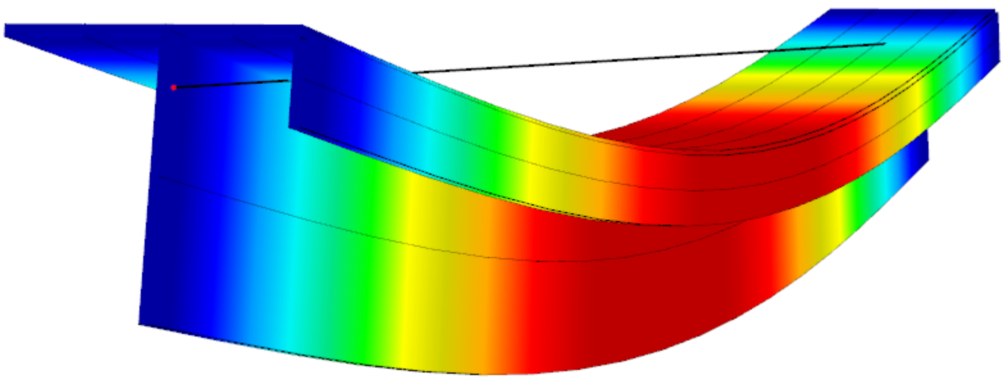


Figure 9.2: Deformations in RFEM/RSTAB model

Furthermore, you can analyze SHAPE-THIN sections in the RF-/STEEL AISC or RF-/ALUMINUM ADM design modules.