



SECTION UTILITY in
Diamonds

www.buildsoft.eu



© BuildSoft, version 14.2

Any representation or reproduction, of all or parts of the materials is strictly prohibited without a prior written permission from BuildSoft.

By purchasing the **Diamonds** software, the purchaser acquires a license of use. Under no circumstances can the user assign the license in part or in total to a third party without a prior written consent of the publisher.

The publisher holds no responsibility for possible faults that the program and/or the here mentioned manual may include and declines any responsibility for damages arising from misuse of the **Diamonds** software and/or the here mentioned manual.

Table of content

1	INTRO	1
2	THE SECTION UTILITY WORKENVIRONMENT	2
2.1	STARTING UP	2
2.2	THE MENU AND THE TOOLBAR	3
2.2.1	The menu 'File'	3
2.2.2	The menu 'Edit'	3
2.2.3	The menu 'Screen'	4
2.2.4	The menu 'Help'	5
2.3	THE ICON PALETTE	5
2.4	THE STATUS BAR	6
3	EXAMPLES	7
3.1	EXAMPLE 1: SECTIONS FROM THE DEFAULT LIBRARY	7
3.2	EXAMPLE 2: COMPOSING SECTIONS	10
3.3	EXAMPLE 3: SECTIONS WITH A TYPE FORM	14
3.4	EXAMPLE 4: SECTIONS WITH AN ARBITRARY POLYGONAL SHAPE	16
3.5	EXAMPLE 5: COMPOSE DIFFERENT TYPES	21
3.6	EXAMPLE 6: SECTIONS COMPOSED OF DIFFERENT MATERIALS	24
3.7	EXAMPLE 7: HOLES IN SECTIONS	27
3.8	EXAMPLE 8: STEEL IN CONCRETE	30
3.9	EXAMPLE 9: THE USE OF ARCS AND ROUNDINGS	32
3.10	EXAMPLE 10: ALIGN CROSS SECTIONS	38
3.11	EXAMPLE 11: DUPLICATE	41
3.12	EXAMPLE 12: DIVIDE CROSS INTO TWO PARTS	43
4	OTHER FUNCTIONS	47
4.1	EDIT RESULTS	47
4.2	THE BUTTON 'IMPOSED CLASSES'	48
4.3	THE BUTTON 'INTERACTION DIAGRAM'	50
4.4	IMPORT AND EXPORT	51
4.5	SETTINGS	51
4.5.1	Tap page 'General'	52
4.5.2	Tap page 'View'	53
4.5.3	Tap page 'Polygon'	56
4.5.4	Tap page 'Print'	57
4.6	PRINTING	59

1 Intro

With the use of **Section Utility** it's possible to calculate the characteristics of composed cross sections.

The benefits of this program are:

- Multiple cross sections can be assembled
- Different materials can be combined
- The possibility to insert cross sections with an arbitrary polygonal shape

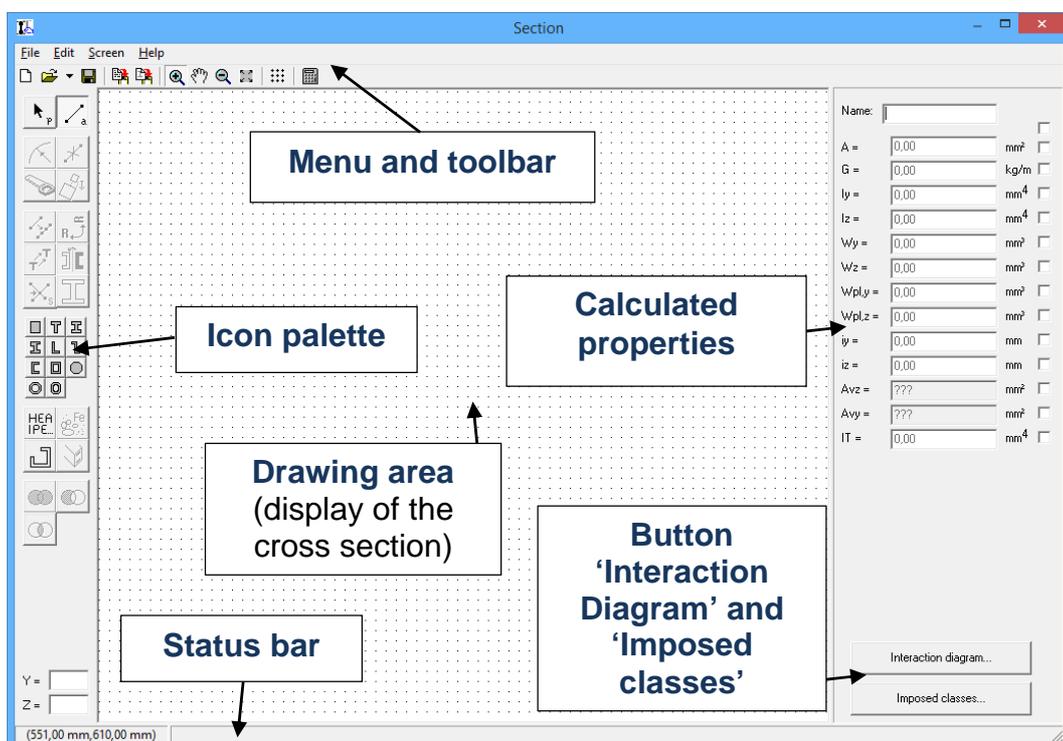
The cross sections made in this program can be used in other BuildSoft software like **PowerFrame** and **Diamonds**.

2 The Section Utility workenvironment

2.1 Starting up

The program is started by double clicking on the icon  or by calling it from PowerFrame / Diamonds.

After start up, we get the following window:



In this window there are a number of distinct parts:

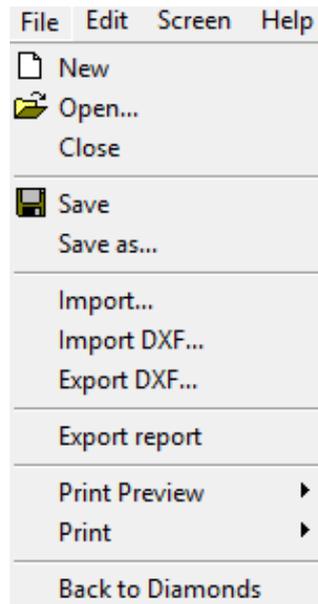
- At the top we see the **menu** and below the **toolbar**.
- In the middle there is the **drawing area** where we will draw the section and / or assemblies.
- On the left is the **icon palette** with at the bottom two editable fields.
- On the right we see the **calculated characteristics** and two buttons 'Interaction Diagram' and 'Imposed classes'.
- Below is the **status bar**.

In what follows, these portions are described.

2.2 The menu and the toolbar

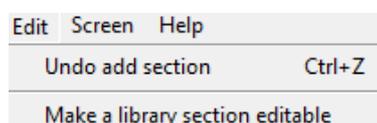
2.2.1 The menu 'File'

In the menu 'File' we find the following items:



-  : start new project
-  : open and existing file
- Close: close the current file
-  : save the current file
- Save as: save the current file under a certain name
- Import: import a cross section that was exported using Section Utility
- Import DXF: import a DXF containing the geometry of the cross section. **Make sure the DXF contains only lines! No polylines, blocks, hatches, annotations, planes, ...**
- Export DXF: export the geometry of a cross section to a DXF
- Export report: exports the report to a RTF. The RTF can be opened and edited in MS Word.
- Print preview and print: print a drawing or a calculation note (see §4.6)
- Stop: end the program (only visible in a stand alone version)
- 'Back to Diamonds' or 'Back to PowerFrame': take the cross section to Diamonds or PowerFrame to use in a global analysis.

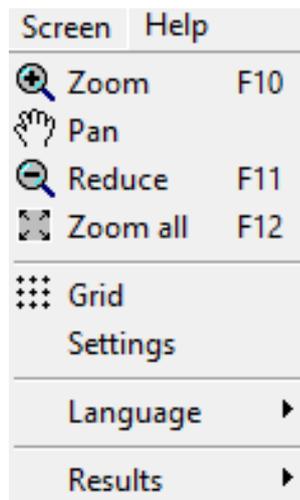
2.2.2 The menu 'Edit'



- Undo: undo the previous operation
- Make a library section editable: to make a cross section from the library editable (see §3.9)
- Align: to align a section with another section (see §3.10)

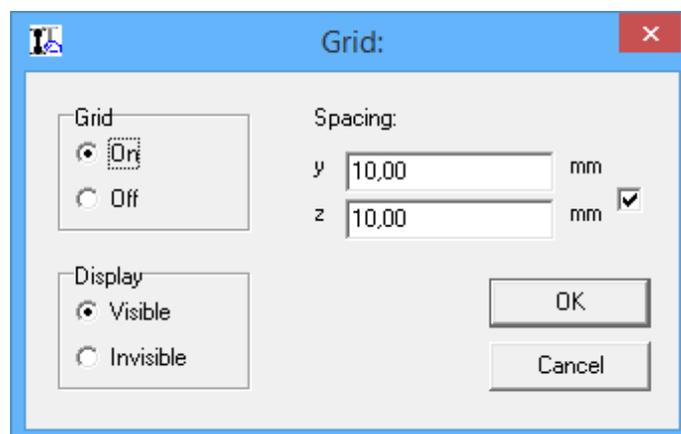
2.2.3 The menu 'Screen'

Then we have the menu 'Screen':



- : zoom in, drag a rectangle around the area on which you want to zoom in
- : pan
- : zoom out
- : show all as large as possible in the drawing area
- : grid

Select the menu item **Grid** or click on the icon to set the grid parameters. Then click 'OK' to confirm the adjustments.



The selection box on the right indicates that the step for y and z should stay the same. If this is not selected, different distances can be entered

for both directions. The grid can be turned on and off (off means that the mouse won't snap to the grid) and the grid can be set visible or not.

- Setting: all settings of the program can be changed here (see §4.5).
- Results: the results in the right column can be set invisible (to make more room for the drawing), editable (so that the user can change the values) or uneditable (see §4.1).

2.2.4 The menu 'Help'

Finally there's the menu 'Help' in which you can find the 'Help' function and the program information:



Next to the icons from the menu, the toolbar contains the following icons:

-  : calculate the characteristics of the drawn cross section
-  : copies the results to the clip board
-  : copies the drawing to the clip board

2.3 The icon palette

-  The selection arrow to indicate elements
-  Draw a new cross section (see §3.4).
-  Change a line into an arc or round a corner of a profile (see §3.9).
-  Delete elements.



Divide a cross section into two parts (see §3.12).



Align a cross section to another cross section (see §3.10).



Divide a line in multiple parts (see §3.12).



Rotate or mirror a cross section



Move or copy a cross section or a part of it



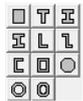
Duplicate a cross section



Determine the intersection point between two lines



Envelop profile



Insert a cross section with a particular shape



Insert a cross section from the library



Assign a material to the cross section



Create a cross section with thin walls



Union of the selected cross sections (see §3.5).



Difference between the selected cross sections (see §3.5).



Intersection between the selected cross sections (see §3.5).

Below the icon pallet there're two input fields that will be discussed further in this sect (see §3.2).

2.4 The status bar

In the status bar you'll find the coordinates of the mouse cursor. In addition, entries like the point number and the name of the cross section will also appear here.

It's the easiest to understand the program by looking at some examples.

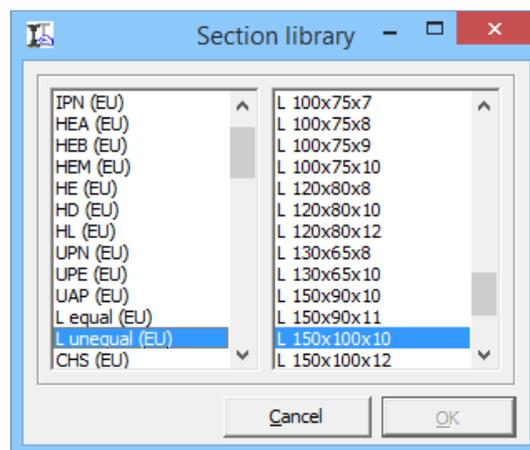
3 Examples

3.1 Example 1: Sections from the default library

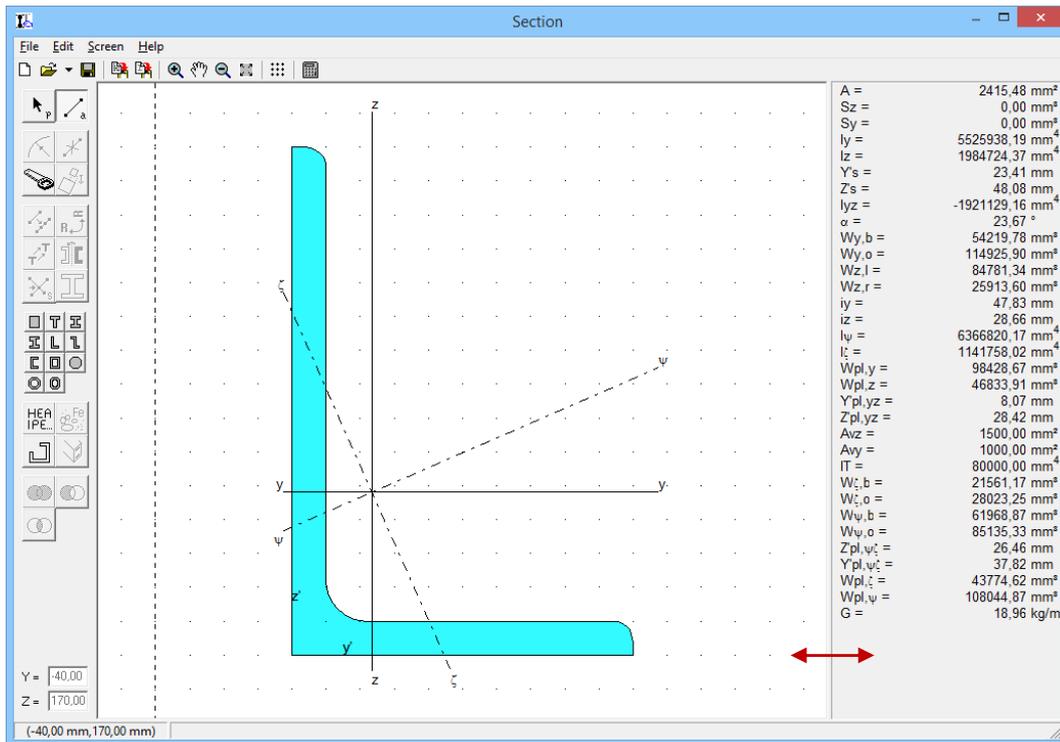
Start the program.

We turn off the option for editing the results: in the menu go to 'Screen' – 'Results' and choose 'Ineditable'.

Next we add a cross section from the library. Click on the button . The following dialog appears:



In the left column we select 'L unequal (EU)', these are European L-sections with unequal flanges. In the column on the right we select for example 'L150x100x10'. Click 'OK'. The screen will now look as follows:



When the values in the column on the right are not shown clearly, you can resize the column by dragging the column border to the left.

The y- and z-axis are drawn at the center of gravity, and also the the principal axes of inertia ξ and ψ .

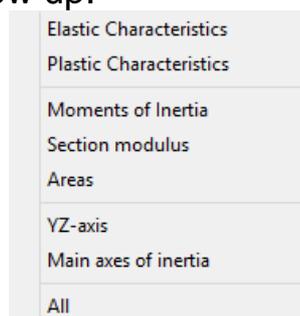
In the column on the right we find calculated characteristics:

- A : the gross area
- $S_{y'}$, $S_{z'}$: static moment of the cross-section according to axes y' and z'
- I_y , I_z : moment of inertia of the cross-section according to axes y' and z'
- Y'_s , Z'_s : the coordinates of the centre of gravity in relation to the coordinate system whose origin is situated in the lower left point of the cross section.
- I_{yz} : product of inertia of the cross-section according to axes y' and z'
- α : angle between the axes of principal inertia and the local axes y' and z' of the cross-section
- $W_{y,b}$: elastic moment of resistance according to bending around the axis y' , at the level of the top fibre of the cross-section
- $W_{y,o}$: elastic moment of resistance according to bending around the axis y' , at the level of the bottom fibre of the cross-section
- $W_{z,l}$: elastic moment of resistance according to bending around the axis z' , at the level of the extreme left fibre;
- $W_{z,r}$: elastic moment of resistance according to bending around the axis z' , at the level of the extreme right fibre;

- i_y, i_z : radius of inertia or radius of gyration of the cross-section in the directions y' and z'
- I_ψ, I_ζ : moment of inertia of the cross-section according to the principal inertia
- $W_{pl,y}$: plastic moment of resistance for bending around the axis y'
- $W_{pl,z}$: plastic moment of resistance for bending around the axis z'
- $Y'_{pl,yz}, Z'_{pl,yz}$: the position of the plastic middle point of the cross section in the local $y'z'$ -coordinate system: this is the point around which W_{pl} was calculated
- A_{vy}, A_{vz} : shear surface in the directions z' and y'
- I_t : torsion constant
- $W_{\zeta,b}, W_{\zeta,o}, W_{\psi,b}, W_{\psi,o}$: moment of resistance for bending around the principal axes based on the two extreme fibers
- $Y'_{pl,\psi\zeta}, Z'_{pl,\psi\zeta}$: the position of the plastic middle point of the cross section in relation to the local y,z – coordinate system;
- $W_{pl,\zeta}, W_{pl,\psi}$: plastic moment of resistance for bending around principal axes of inertia
- G : the weight of the section per running meter

In case the width of the column is insufficient to represent all the results, the results will be divided over two columns. It may then be useful to increase the width of the column. Another possibility is to not view all the values. To do so, there are two possibilities:

- Via the menu 'Screen' – 'Settings' (see §4.5.2)
- Or by clicking in the right column with the right mouse button. The following menu will show up:



By clicking on one of these items, we can determine which values should be displayed. The division is done as follows:

- Elastic properties: $A, S_{Z'I}, S_{Y'I}, I_y, I_z, Y'_s, Z'_s, I_{yz}, \alpha, W_{y,b}, W_{y,o}, W_{z,l}, W_{z,r}, i_y, i_z, I_\psi, I_\zeta, A_{vy}, A_{vz}, I_t, W_{\zeta,b}, W_{\zeta,o}, W_{\psi,b}, W_{\psi,o}, G$.
- Plastic properties: $W_{pl,y}, W_{pl,z}, Y'_{pl,yz}, Z'_{pl,yz}, Y'_{pl,\psi\zeta}, Z'_{pl,\psi\zeta}, W_{pl,\zeta}, W_{pl,\psi}$.
- Inertia properties: $S_{Z'I}, S_{Y'I}, I_y, I_z, I_{yz}, i_y, i_z, I_\psi, I_\zeta, I_t$.

- Resistance properties: $W_{y,b}, W_{y,o}, W_{z,l}, W_{z,r}, W_{pl,y}, W_{pl,z}, W_{\zeta,b}, W_{\zeta,o}, W_{\psi,b}, W_{\psi,o}, W_{pl,\zeta}, W_{pl,\psi}$
- Area: $A, Y'_s, Z'_s, \alpha, Y'_{pl,yz}, Z'_{pl,yz}, A_{vy}, A_{vz}, Y'_{pl,\psi\zeta}, Z'_{pl,\psi\zeta}, G$.
- YZ - as: $A, S_{Zl}, S_{Yl}, I_y, I_z, Y'_s, Z'_s, I_{yz}, W_{y,b}, W_{y,o}, W_{z,l}, W_{z,r}, i_y, i_z, W_{pl,y}, W_{pl,z}, Y'_{pl,yz}, Z'_{pl,yz}, A_{vy}, A_{vz}, I_t, G$.
- Principal axes: $\alpha, l_\psi, l_\zeta, W_{\zeta,b}, W_{\zeta,o}, W_{\psi,b}, W_{\psi,o}, Y'_{pl,\psi\zeta}, Z'_{pl,\psi\zeta}, W_{pl,\zeta}, W_{pl,\psi}$.

3.2 Example 2: Composing sections

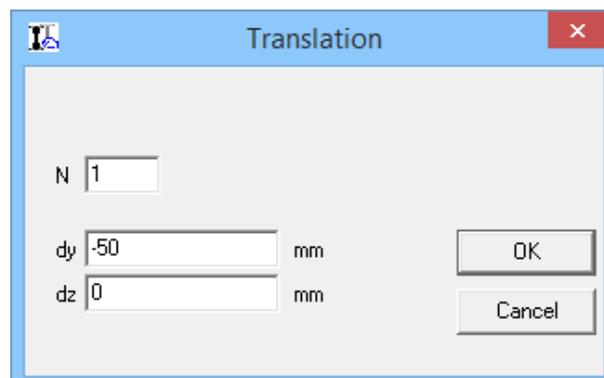
With Section Utility multiple sections can be combined. Suppose we'd like to add an additional section to the profile from §3.1. We could take the profile from the library  again. But we could also copy the profile from example 1.

Assume we want to copy this profile:

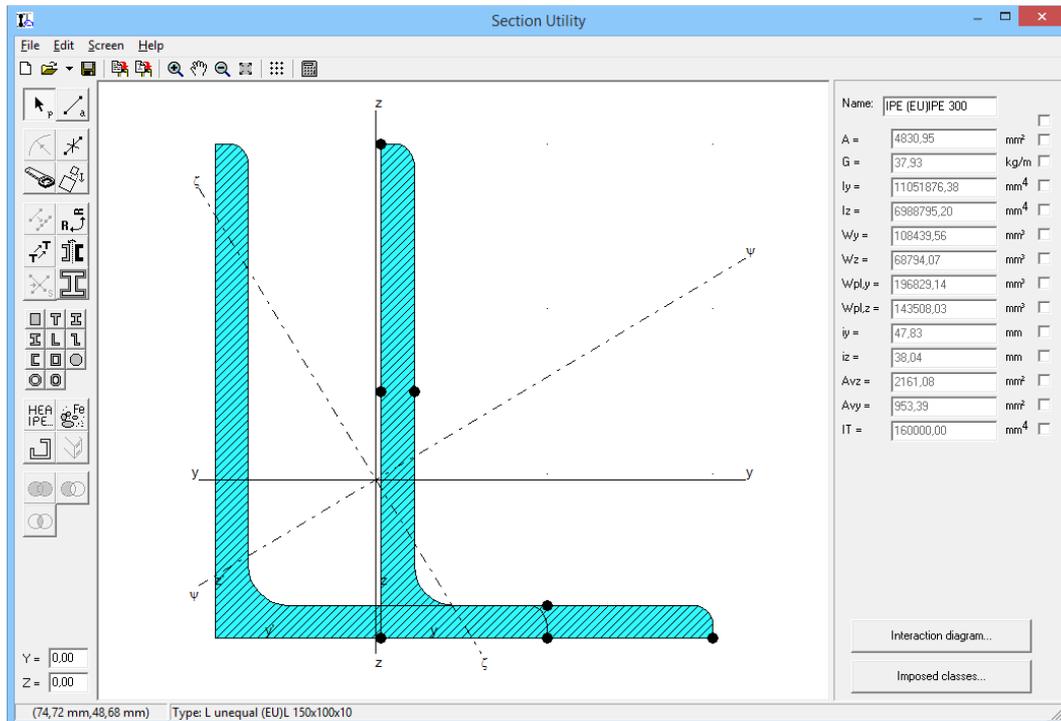
- Click on the selection arrow .
- Next select the L-section. An element can be selected by:
 - o clicking with the mouse on the element;
 - o or by drawing a selection window around it
When you draw the selection window from left to right only the elements completely within the window will be selected. When you draw the selection window from right to left, all the elements partially or completely within the window will be selected.
 - o Using the SHIFT-button elements can be added to the selection (if not yet selected) or be removed (if yet selected) from the selection.

When we select the profile a few privileged points will illuminate. The purpose of these points is discussed later in this topic. When a profile is selected, its name will be written in the status bar.

- After selecting the profile, press the translation button . The following dialog appears:

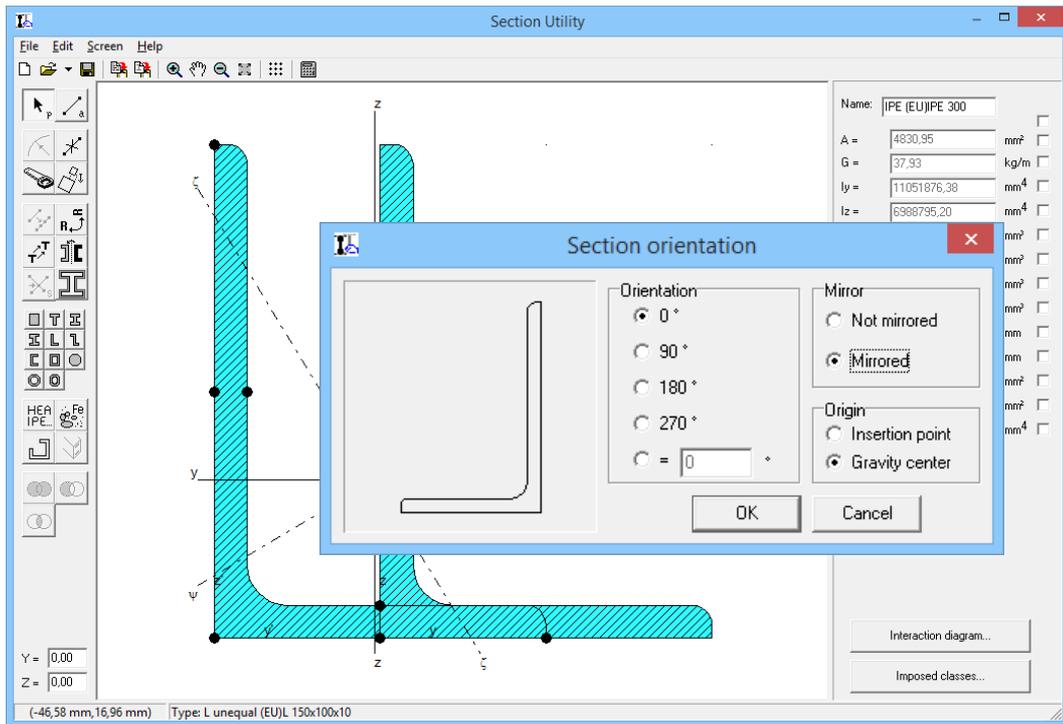


- We want to copy the profile once, so $N=1$. Now we can enter dy and dz to determine how far the copy should be translated from the original. We enter $dy = '-50'$ and $dz = '0'$.
- This will give the following result:

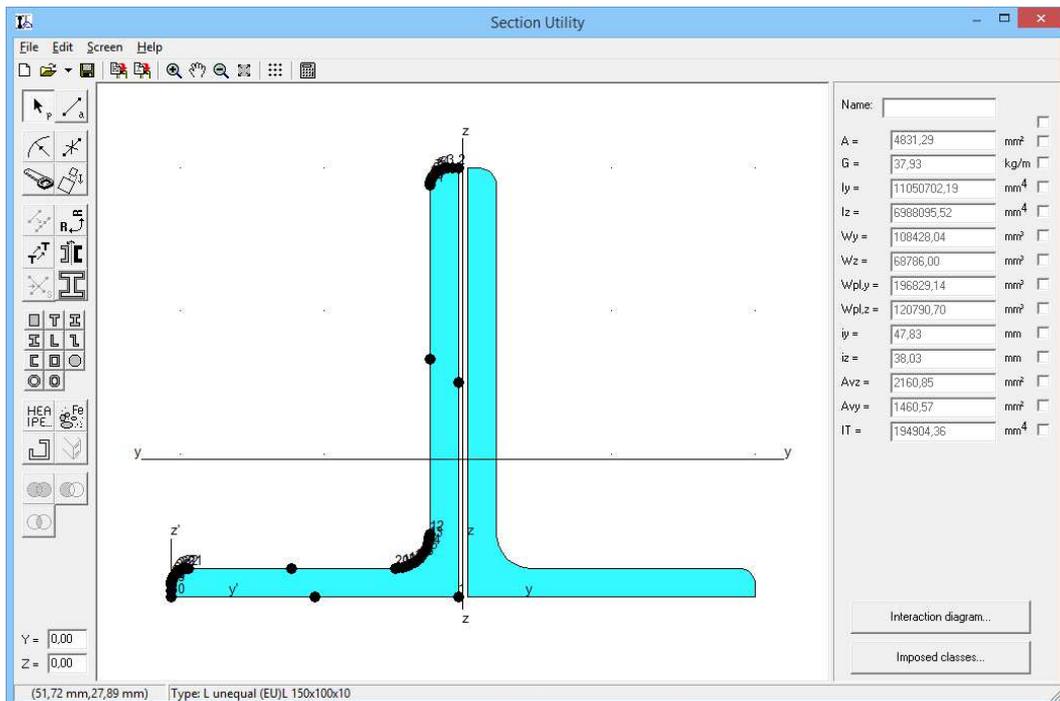


The hatched profiles indicate that one or more profiles overlap or intersect each other. The section can not be constructed this way.

- To mirror the profile on the left:
 - o Select the profile on the left
 - o Click on the icon . The dialog below appears.

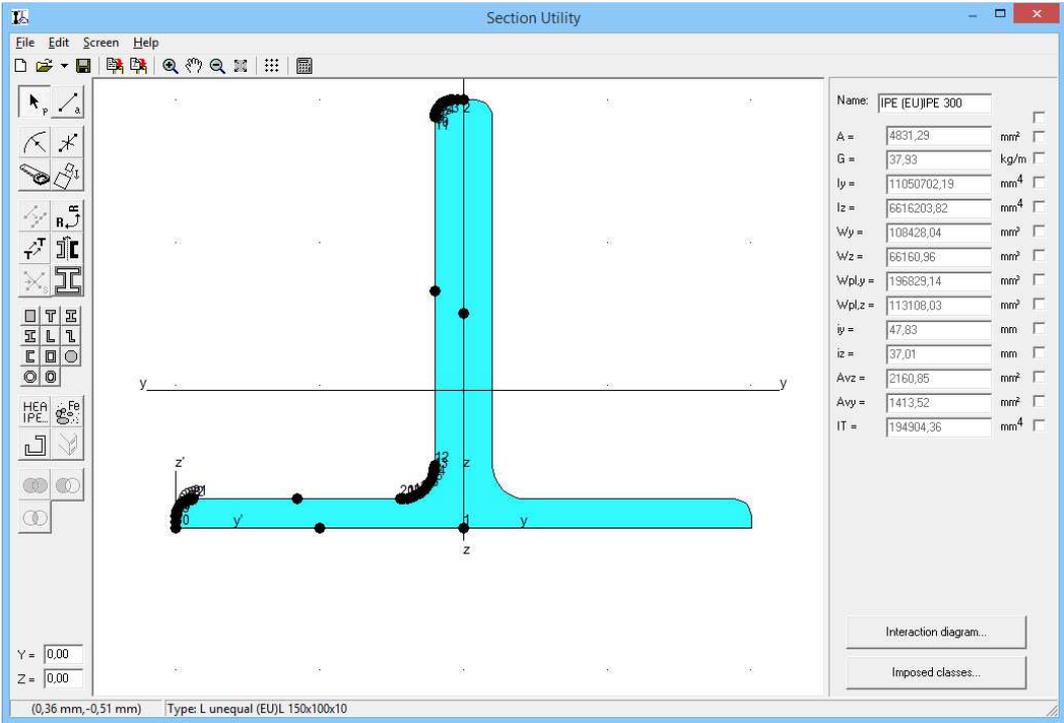


- In this dialog we can change the angle of the profile because we can also mirror the cross section. Also the rotation center can be established: either the insertion point of the section. This is the origin of the coordinate system before you start drawing, or either the center of gravity (not for composed sections). We just want to mirror the L-section and select 'Mirrored', for the angle we choose '0°'.



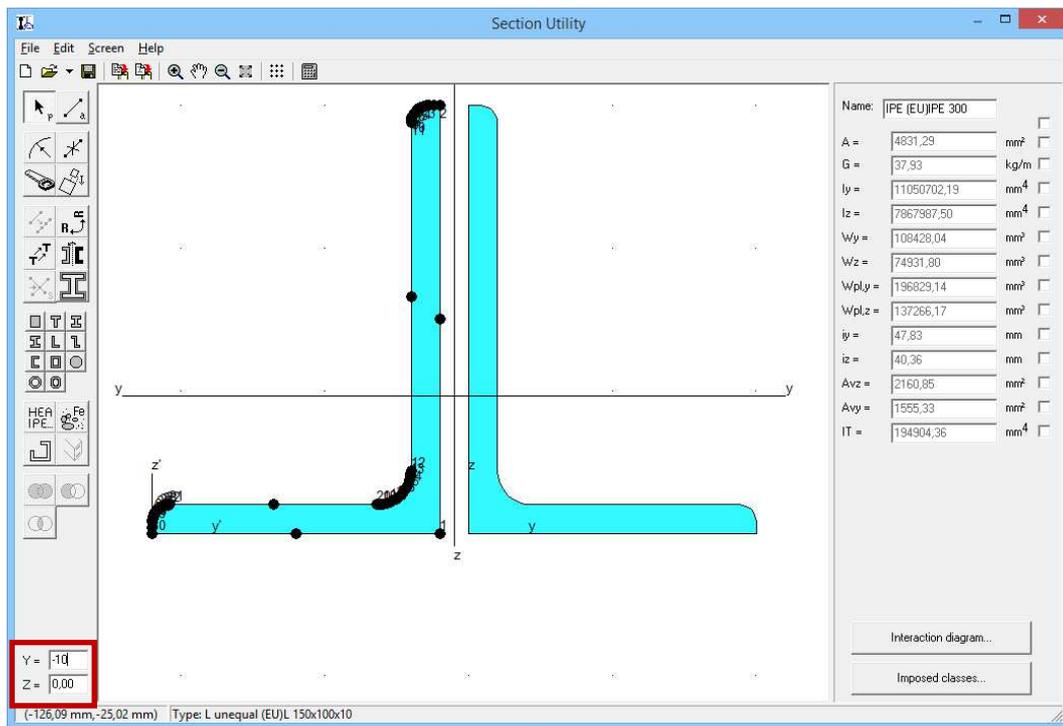
These profiles are now an arbitrary distance from each other. To precisely position them, we use the privileged points. When we select a profile, dots (or squares) will become visible on certain places. These privileged points can be used to attach the profiles to each other. When we click on one of these points and keep the left mouse button pressed in we can drag the profile. You will notice that the section properties change immediately.

We select the left profile. Now select the privileged point in the lower right corner. When we drag this profile near the profile on the right, the privileged points of this profile will light up. Move the mouse to one of these privileged points and only the selected privileged point will remain visible. We now know that the section will snap to this point. Then the sections will be perfectly aligned, whether we use a grid or not.



It may be possible that these sections will not be constructed against each other but with a certain offset. This can be done by selecting the profile that should have an offset with respect to the other profile, and enter the offset in the two edit boxes in the lower left corner of the screen.

Select for example the left profile and enter an offset 'y = -10';'z=0':



3.3 Example 3: Sections with a type form

In this example we will be using profiles with a default shape. First we erase the screen by starting a new  project. The program will ask if the section should be saved or not.



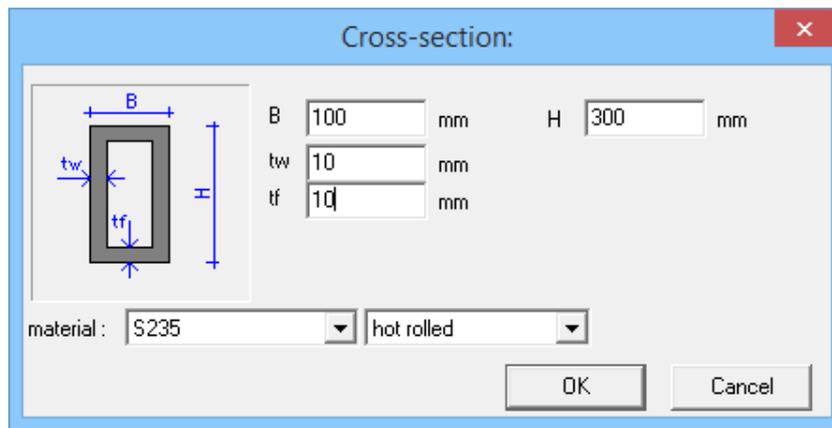
On the icon pallet on the left we see 11 default shapes:

By pressing on one of these icons, a dialogbox will appear allowing you to enter the dimensions and the material of the cross section.

The possible shapes are: a rectangle, T-section, I-section, unequal I, L-section, Z-section, rectangular hollow section, a pipe and a round section.

As example we take a rectangular hollow section with dimensions 100x300mm and thickness 10mm.

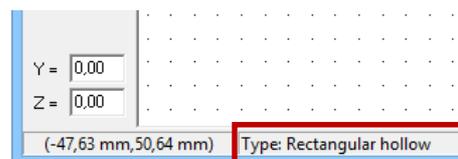
- We click on the icon  and complete the screen that appears:



- Then we press 'OK'.

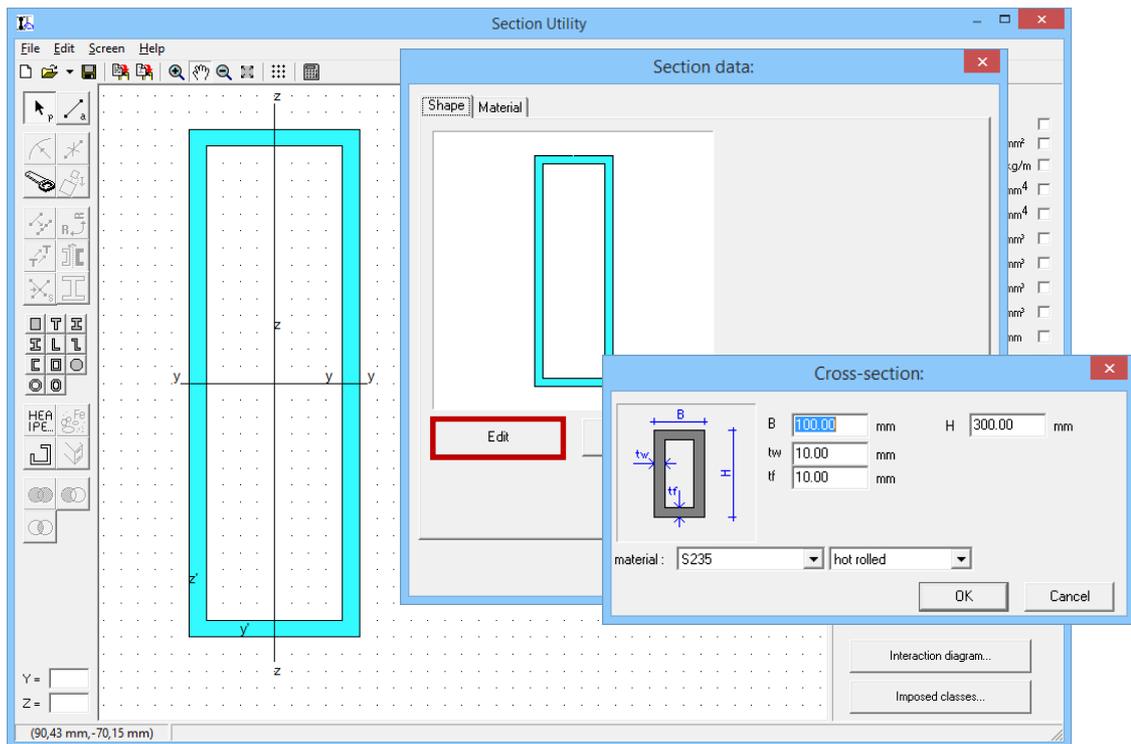
These section can also be copied, mirrored, translated and composed, ...

How can we know whether we are dealing with a profile from the library or with a profile according to one on the default shapes? When we select the profile we see the bottom of the status bar: the name of the profile (if the profile is from the library) or the default shape of the profile (if the profile was entered as a default shape).



The dimensions of the profile can be easily changed by double-clicking the profile. When a profile from the library is double clicked, the library will open again and we can change the profile.

When we double-click a profile made from a default shape, the corresponding dialog box opens, the dimensions and the material can be changed.



3.4 Example 4: Sections with an arbitrary polygonal shape

We will now enter a polygonal section, which is completely free of shape.

We start with a blank sheet and now click on the icon .

If we now move the cursor over the drawing screen, we see that it consists of two intersecting lines. We can now start drawing the section as follows.

We click on the place where the starting point of a line should be and then hold down the mouse button to draw a line. At the end point of the line we let go of the mouse button. While drawing, we can make use of the grid. The SHIFT-key can be used to draw lines with a slope of 0, 15, 30, 45, 60, ... degrees.

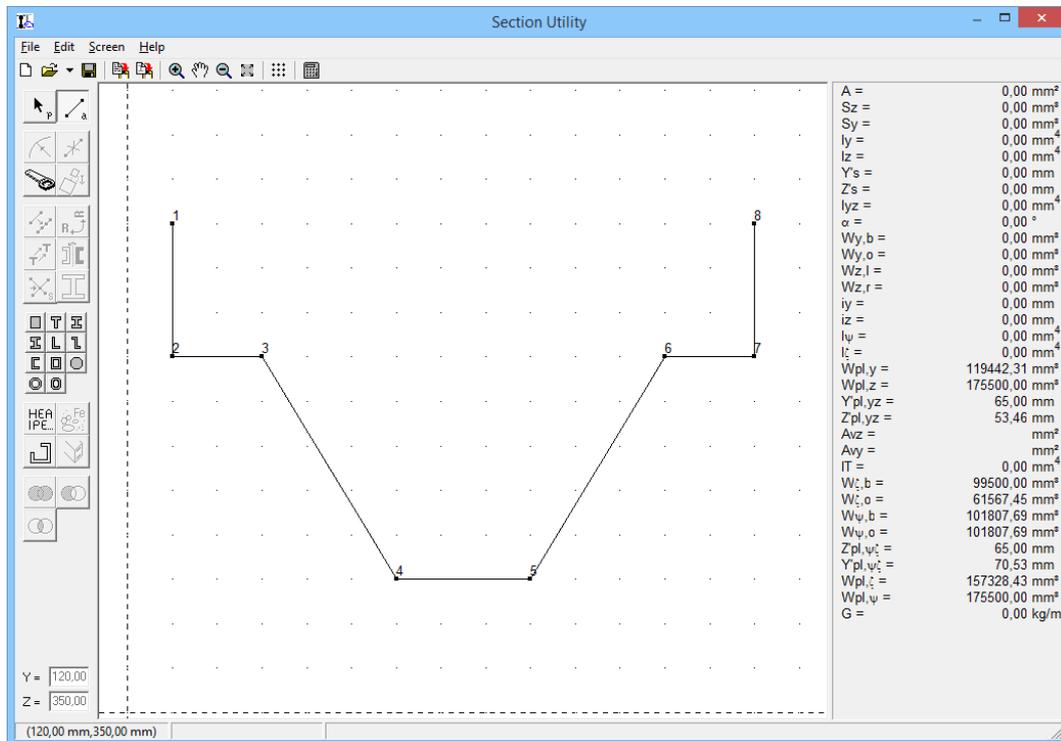
When drawing a line the following information will be shown in the status bar:

- The coordinates of the end point;
- The horizontal and vertical distance between the start and end points (dy and dz);
- The angle of the line to the horizontal and the length of the line.

The entire section must be drawn at once without using a different function. Only the zoom and pan functions may be used. Whenever you do something else (eg. selecting an element with the selection arrow) and then press back on the icon , you start a new section.

Also, the lines have to be entered in the correct order. When you add a line, the program will automatically place its origin at the last input node.

We now draw following section:

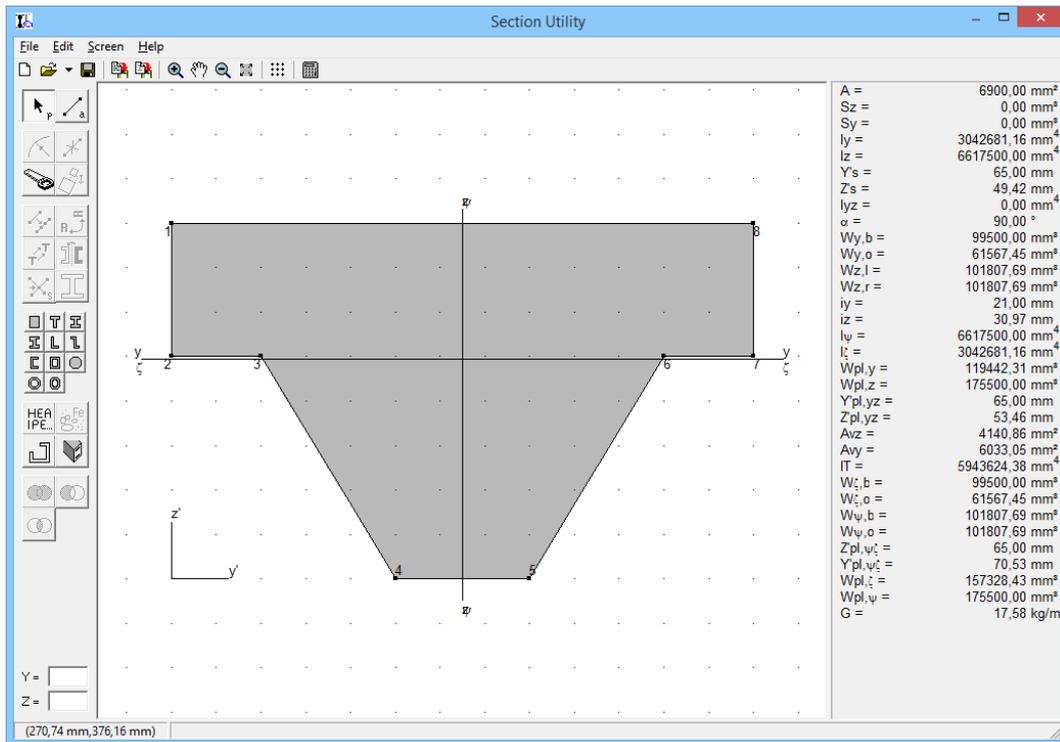


Since it's only possible to use closed polygons, the last line should in principle not be drawn. The program will automatically connect the last point with the first.

Click  to calculate the characteristics.

If we had drawn the last line anyway, than the characteristics would be calculated automatically.

The window looks as follows:



The nodes are numbered. As a result, we can easily see the difference between a self-drawn section and a section from the library or with a predetermined shape.

This drawing can now be modified in various ways.

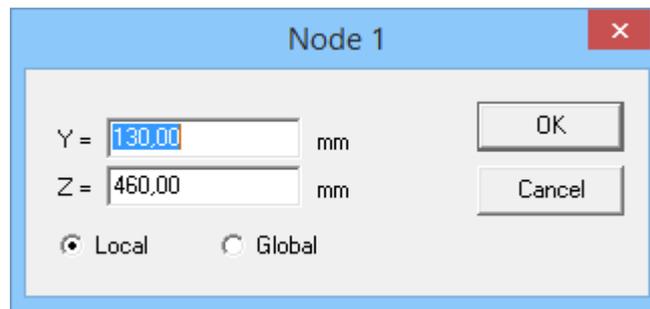
When we select a point by clicking on it with the selection arrow, we see in the point number in the status bar. In the two edit boxes at the left bottom of the window, we see the exact coordinates of the point. We can change the coordinates by entering them in the edit boxes.

We can also drag the point by clicking on it and keeping the mouse pressed as we move the mouse. However, it is not possible to bring the point to any place on the screen. Section Utility prevents the edges of the section to intersect or cross each other because the proper shape of the section is then unclear.

For instance, when we want to move point 4 up then that can only be done up to line 1-8. When we move the mouse over this line item 4 refuse to cross this line.

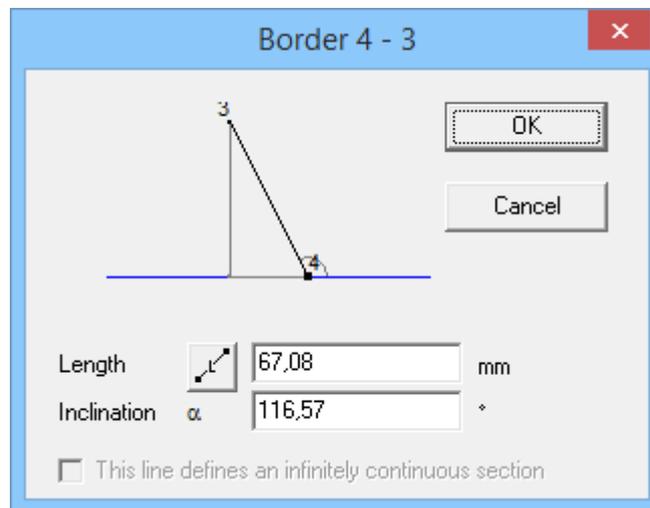
Also lines and selections can be selected and dragged. Each time you move elements, the program will make sure that no intersections occur within one section.

The exact coordinates of a point can also be changed by double-clicking it. The following dialog box:



Sections can be moved. The entered coordinates (Local) will no longer correspond to the actual coordinates (Global). However, in the dialog box, we can choose what we want to view or modify coordinates.

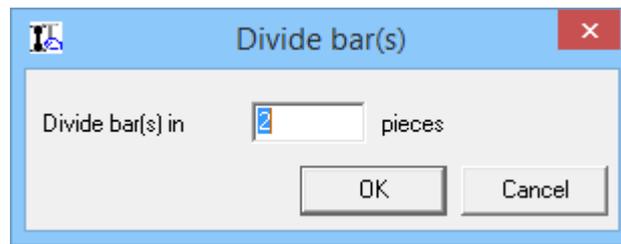
The lines can be changed by double-clicking them. The following dialog box shows up:



Here we have clicked the edge 4-3. When the point where we click the line is closer to point 4 than to point 3, then point 4 is fixed and we can only change the coordinates of the point 3. In the dialog box, this is referred to as follows: Point 4 is marked with a square and the angle is also shown in this point. By the icon  the length can be entered horizontally or diagonally.

Selected elements can be deleted using the icon  or by pressing the DEL-key.

It is also possible to add nodes to the section. We do this by dividing one or more edges in a number of pieces of equal length. So first select an edge or a number of edges and then click on the icon :

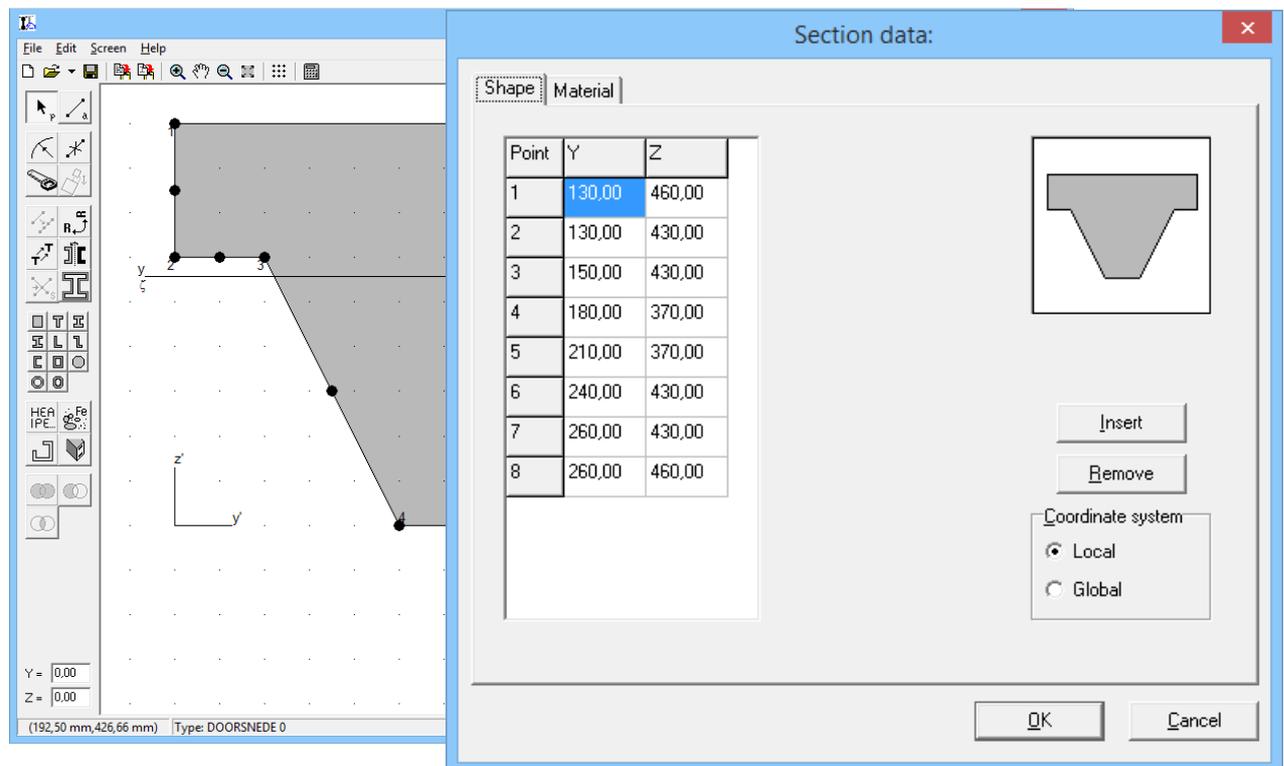


Indicate how many divisions you'd like.

We also note that Section Utility has an "undo" function so that the last instruction can be always undone (hot key: CTRL+Z).

When we refer to the full profile by clicking it in the center, we see that privileged points are placed on the corners and in the middle of each side.

When we double-click on the profile, the next dialog box will appear:



We get a list of all the points of the profile. These points are easy to modify by clicking on a coordinate. You can also inserted or deleted points there.

When the 'Delete' button is pressed, the point where the cursor is at the moment will be removed. When the 'Insert' button is pressed, a point is inserted just after the point where the cursor is.

Again, we can choose between local or global coordinates. At the top the name of the profile appears.

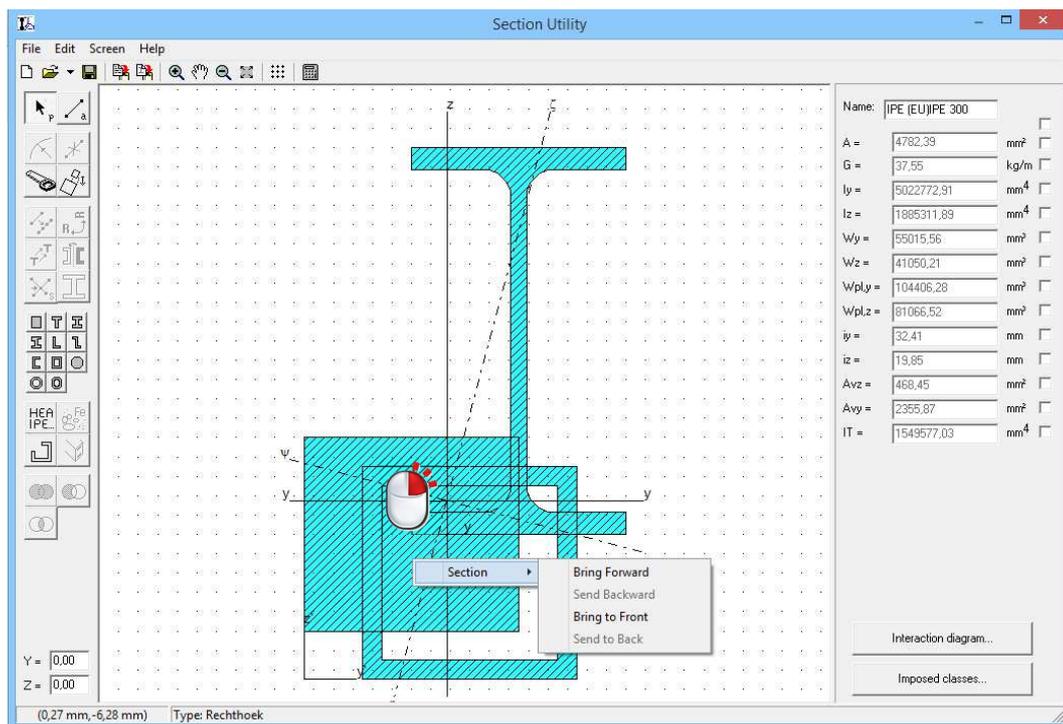
3.5 Example 5: Compose different types

The three types discussed in the previous exercises (section from library, section with default shape, self-drawn section) we can also be assembled.

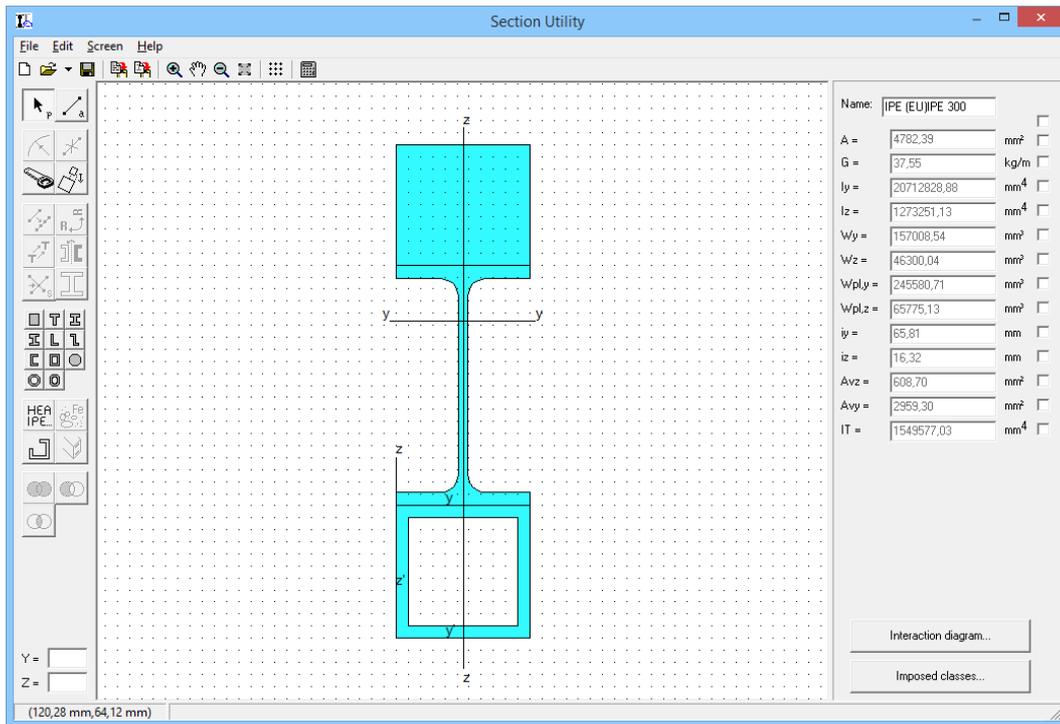
As an example, we take an I-profile IPE100 on which we attach a rectangle (55 x 50) and including a tube (55 x 55 x 5).

- We take the IPE-100 profile from the library .
- We will draw the rectangle ourselves .
- The tube will be defined as a default shape .

The sections are shaded when they overlap! We can move sections forward or back by clicking with the right mouse button on it so the following menu shows up:

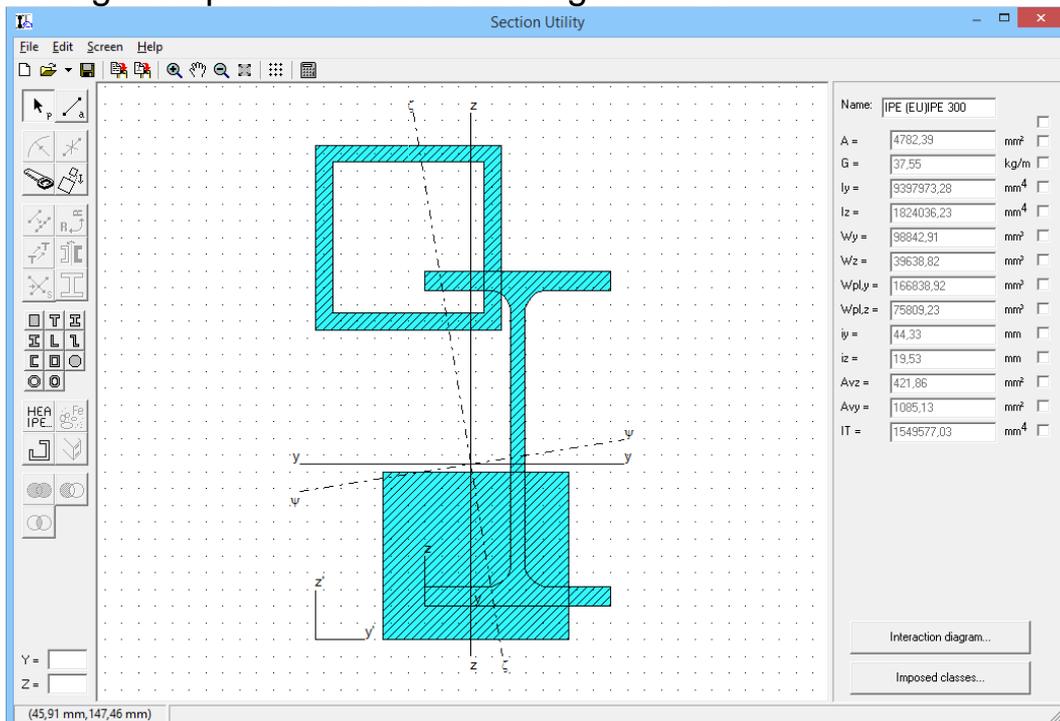


After arranging the sections, the drawing could look like this:

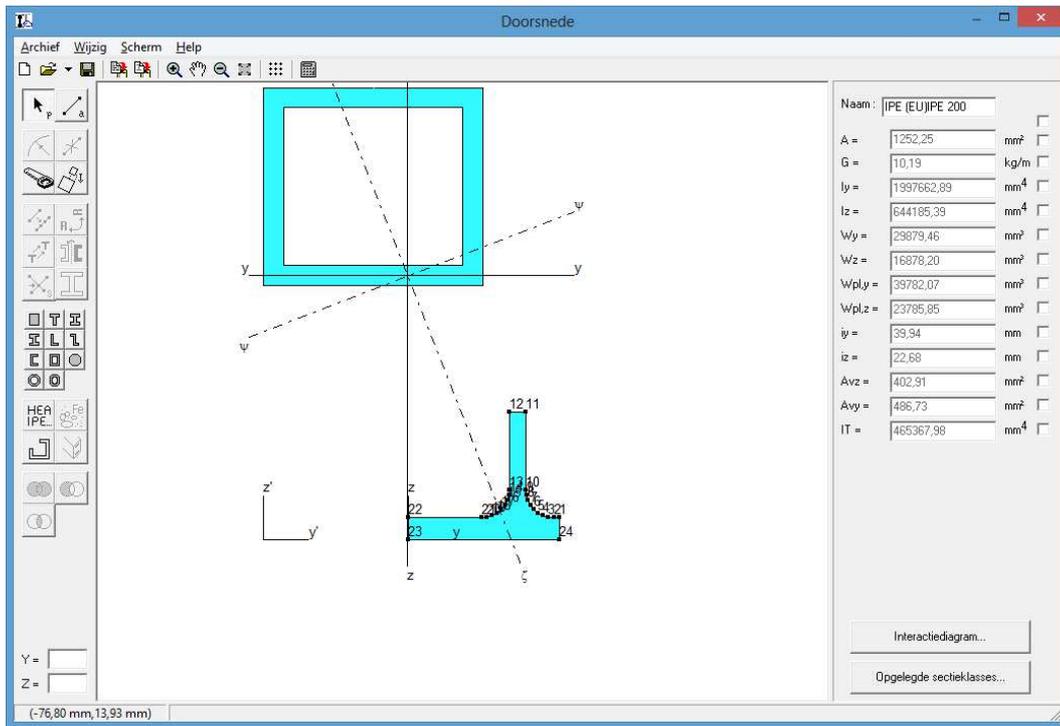


Furthermore we could also illustrate the functionalities of these buttons :

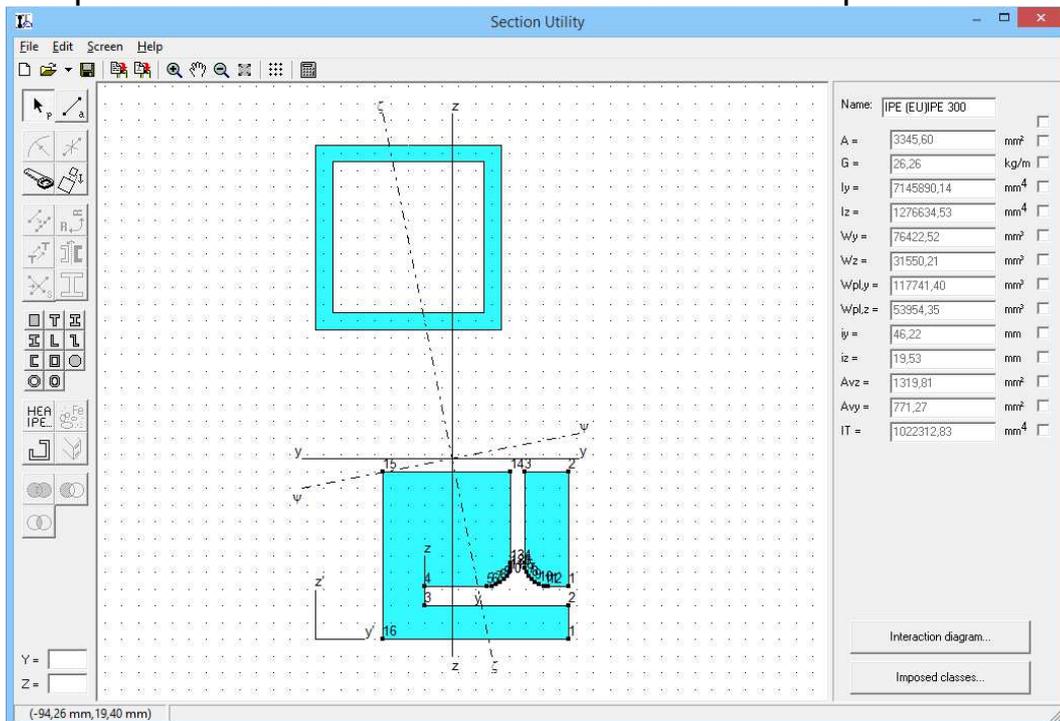
- Arrange the profiles like in the image below.



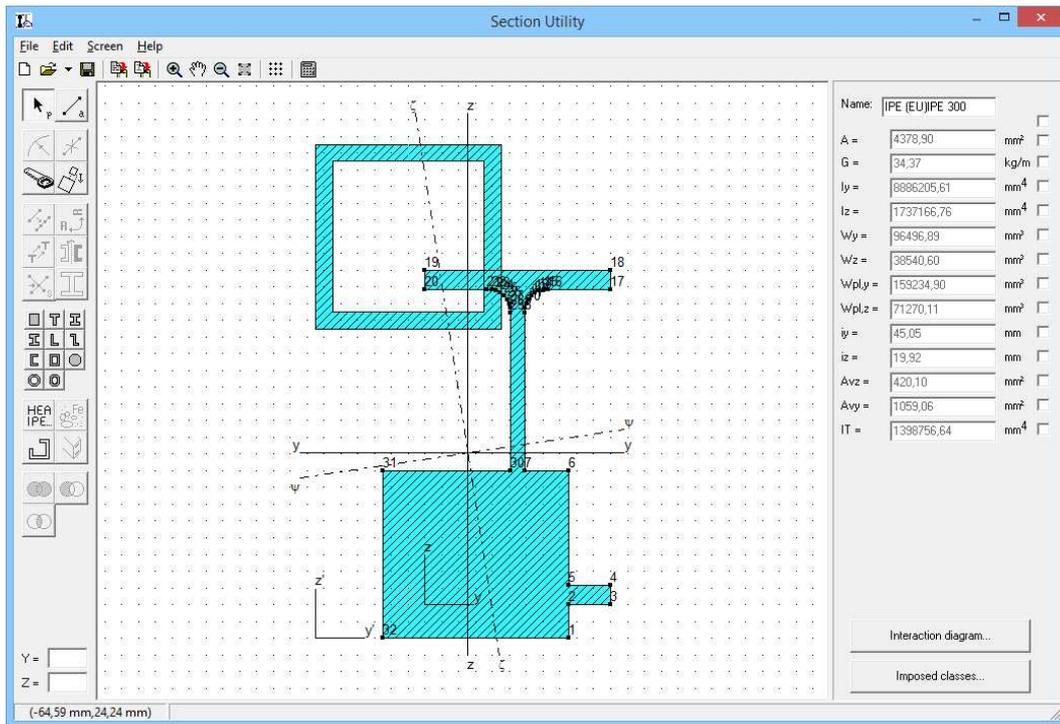
- Select the IPE 100 and the rectangle.
- Click on . Only
 - o the not selected cross section
 - o and the shared cross section **of the selected profiles** remains.



- In the menu click 'Edit' – 'Undo intersection'.
- Place the rectangle to the back using the right mouse button.
- Select the rectangle and the IPE 100.
- Now click .
- The profile in the front will be subtracted from the profile in the back.



- In the menu click 'Edit' – 'Undo intersection'.
- Select the rectangle and the IPE100.
- Click on .
- Both profiles will now be joint into one cross section

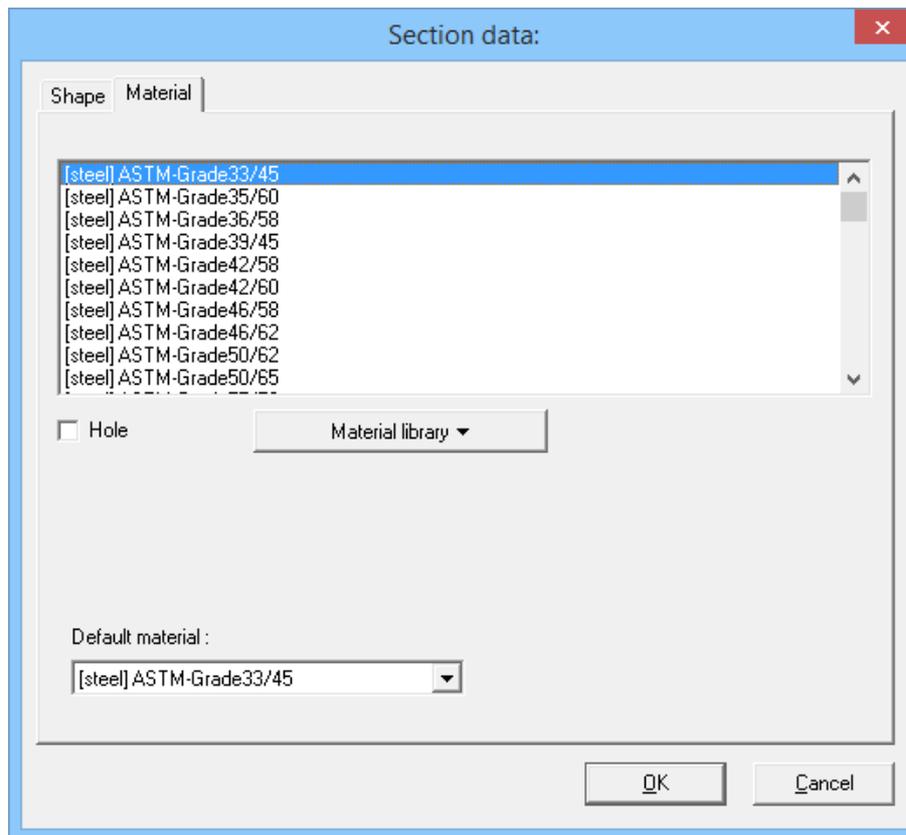


3.6 Example 6: Sections composed of different materials

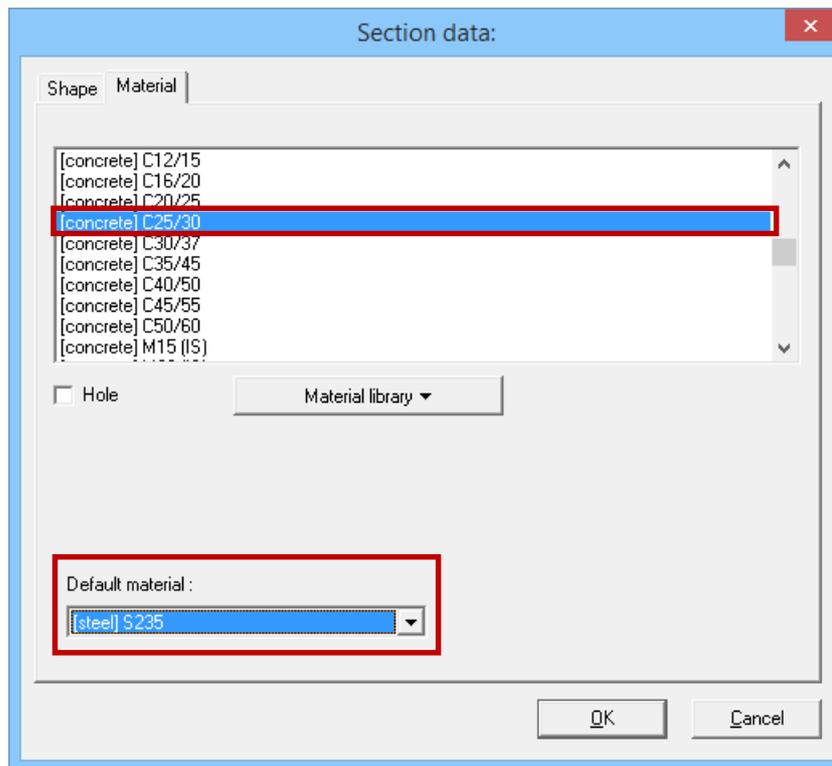
With Section Utility, it is possible to compose a cross section using different materials.

We take back the drawing of Example 5. We wish to remove the box section and adapt the materials. The IPE100 consists of steel, the rectangle of concrete.

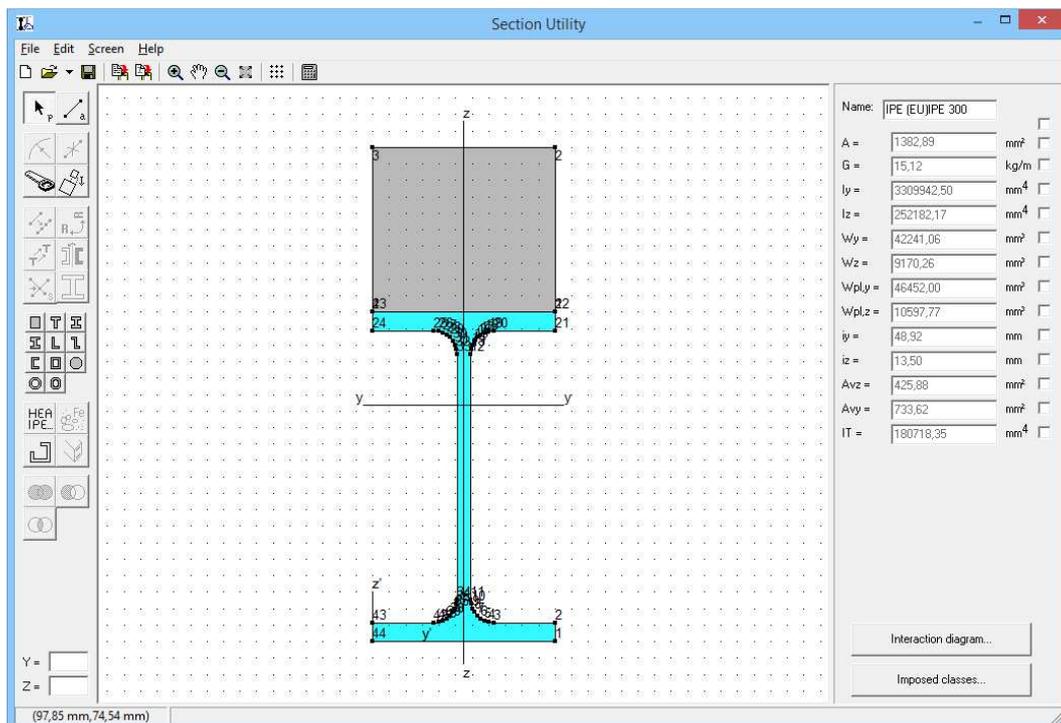
We select the rectangle and click . The following dialog appears:



- At the top we find a list of the already available materials.
- Materials can be added, edited and removed by clicking the button 'Material library' and then 'Edit'.
- Choose 'Concrete C25/30' from the list.
- **At the button the default material can be defined. The composed section will then be seen as a fictive section in this default material. Choose as default material 'Steel S235'.**



The drawing now looks as follows:



We chose 'Steel' as the default materials. So for all other materials we use a correction factor α .

For the area of the fictive section we obtain: $A = 1425 \text{ mm}^2$.
This is calculate like this:

$$\alpha = \frac{E_{concrete}}{E_{steel}} = \frac{30\,000\text{ N/mm}^2}{210\,000\text{ N/mm}^2} = 0,1428$$

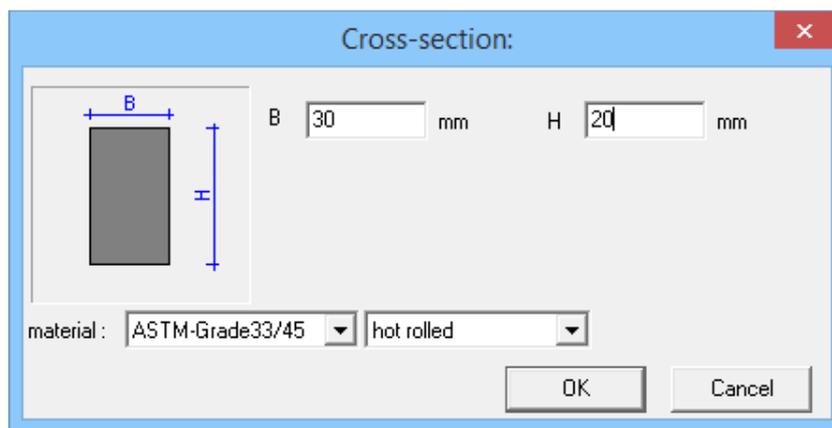
$$A_{fict} = A_{IPE100} + \alpha \cdot A_{concrete,55x50} = 1032\text{ mm}^2 + 0.1428 \times 2750\text{ mm}^2 = 1425\text{ mm}^2$$

3.7 Example 7: holes in sections

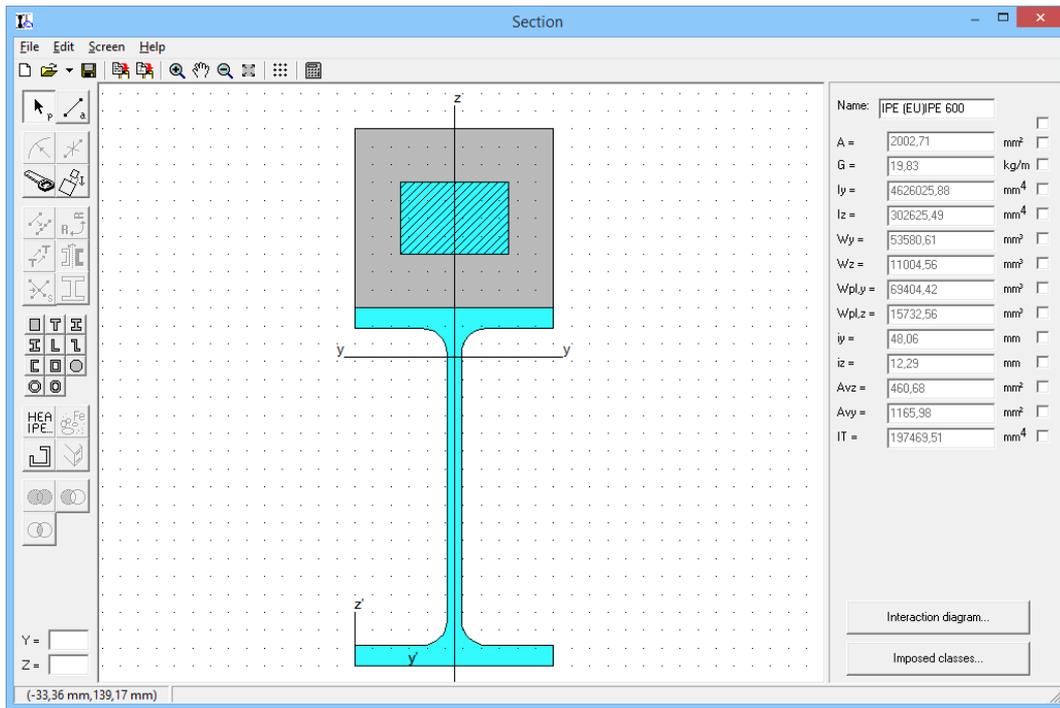
In the dialog box for the materials, there is a check box that we haven't discussed yet, namely 'holes'. With the help of this box we can mark a particular section as an opening.

We start from the profile in § 3.6. We now wish to make an opening in the concrete.

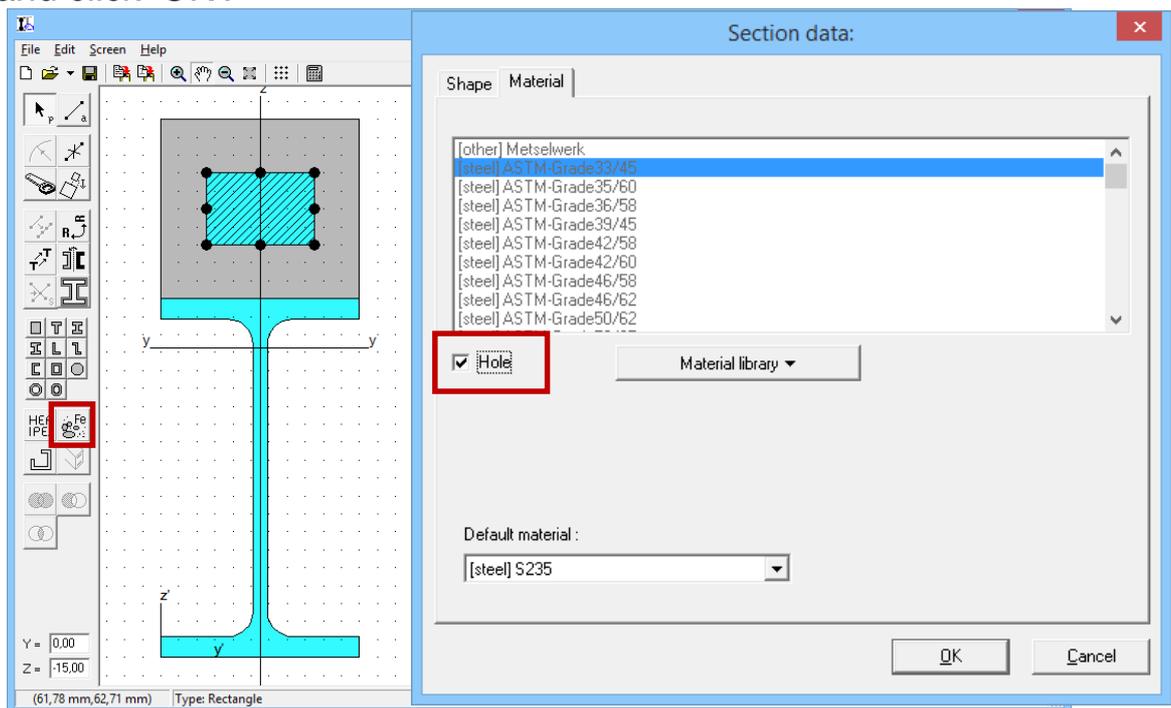
- Use the default shape  to define a rectangle 30x20mm.



- Drag the rectangle so that it lies on top of the concrete section:

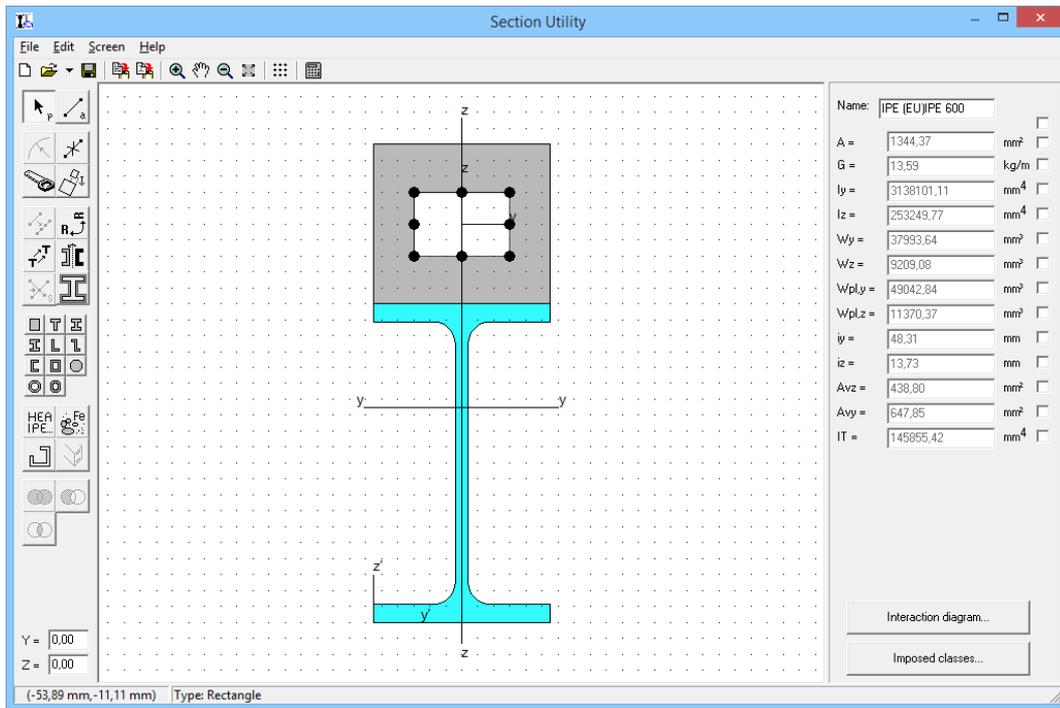


- Select the rectangle and click . In the material window select 'Cut out' and click 'OK'.



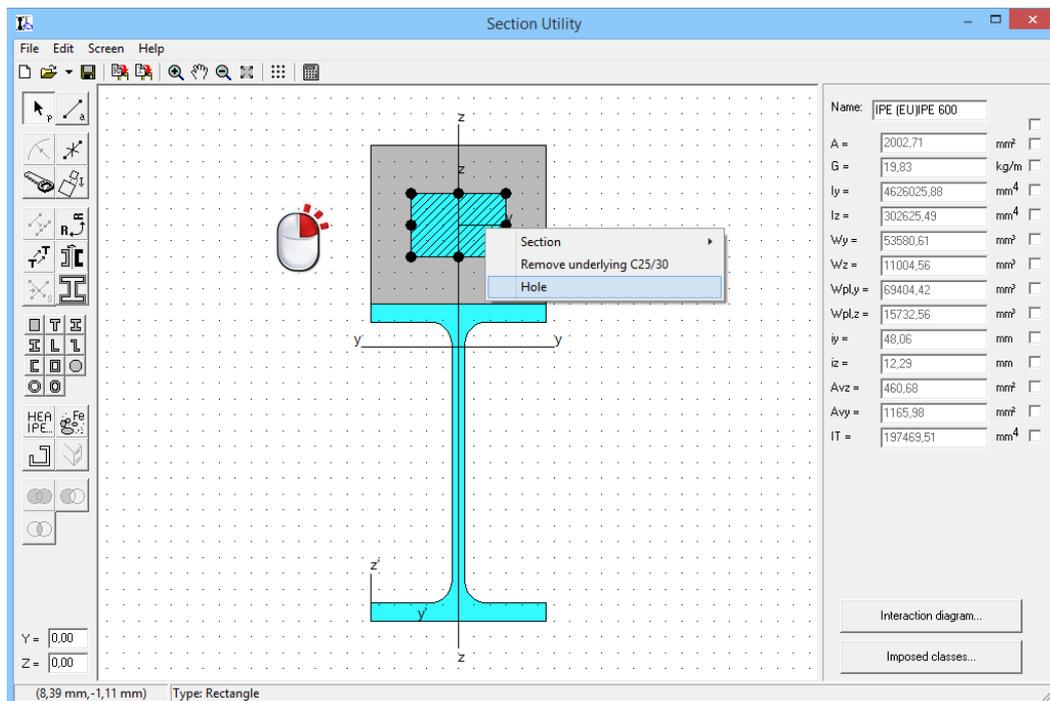
The program automatically searches for the underlying section and makes a hole in it. When Section Utility can not find the underlying section for some reason, it will ask the user to indicate the section.

The results should look like this:



This opening can still be edited. But dots and lines can not be brought out of the section, ie, the opening must be entirely within the section. The program will also prevent that a point of the opening exceeds the outer edge of the section.

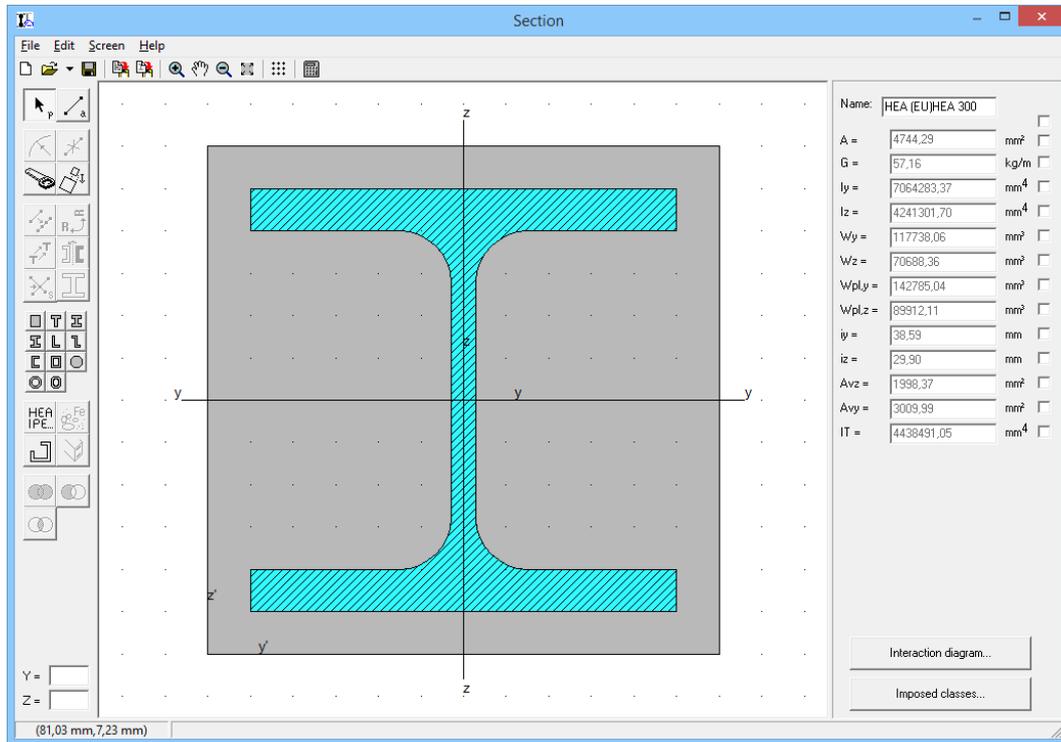
Note: a faster method to mark a section opening by clicking on the section with the right mouse button and then choose 'Hole'.



3.8 Example 8: Steel in concrete

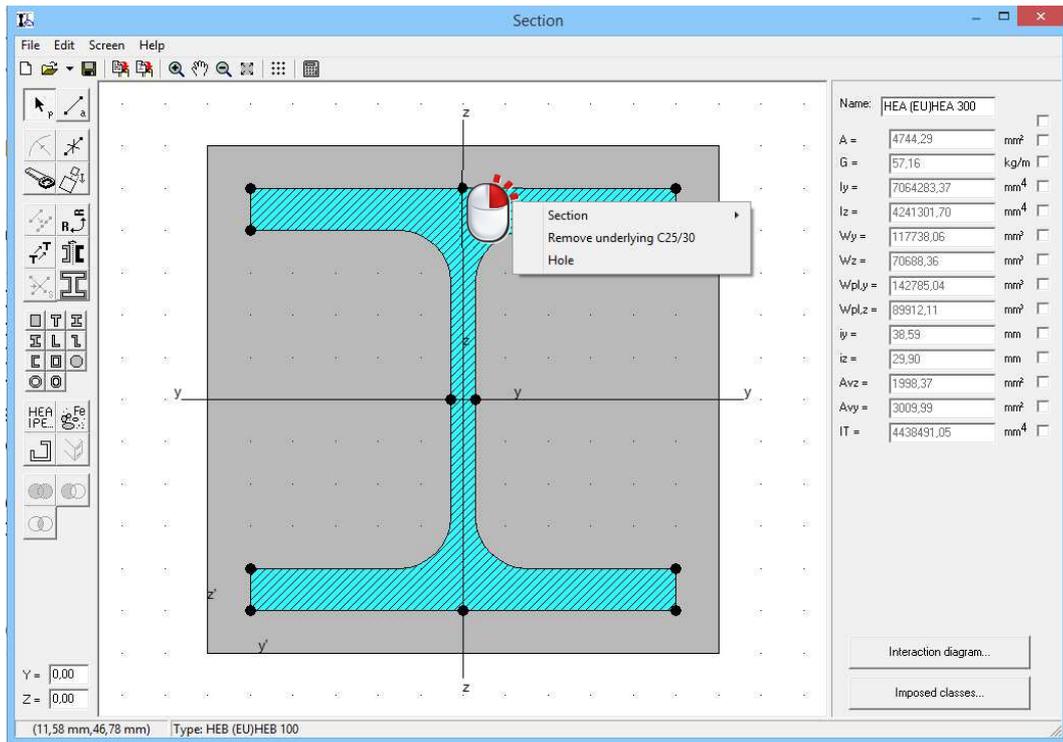
We want to encase an H-section into a concrete section:

- Draw a rectangle 120x120mm  in concrete C25/30 and take a HEB100 steel S235 from the library .
- We place the HEB into the rectangle. The drawing looks like this:

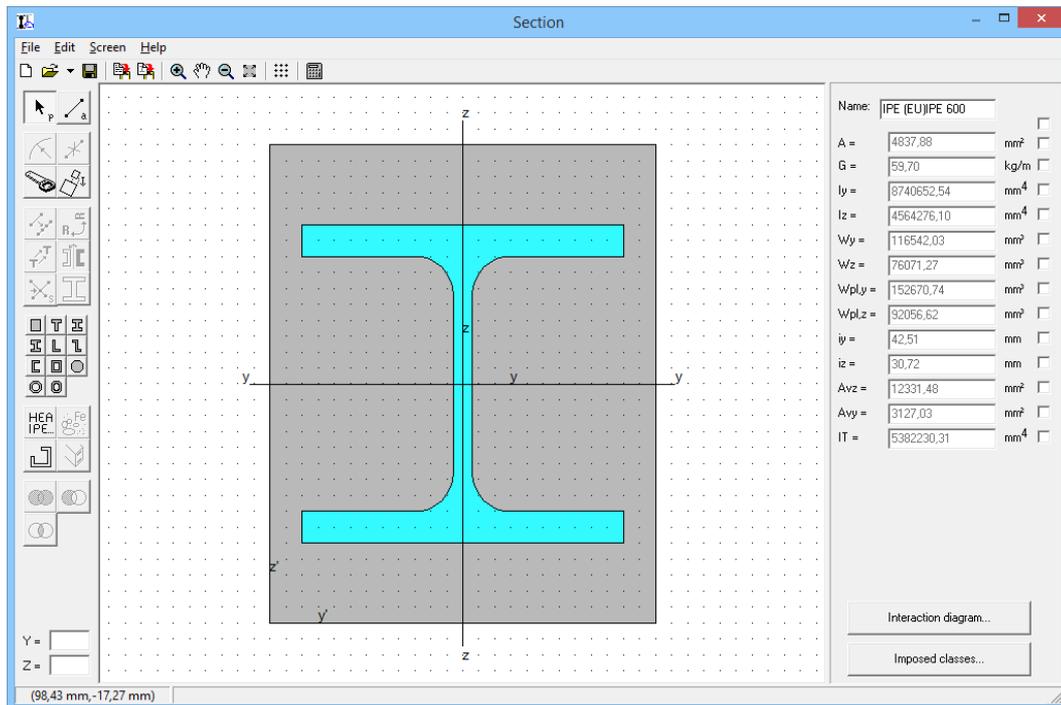


From the shading we see something is wrong: behind the H-section there's still concrete. To take this away, we should first make a hole in the concrete and then place the steel profile in it. However, Section Utility offers an easier method.

When you click with the right mouse button on the H-section, you'll see the following menu.



Select the option 'Remove underlying C25/30' to remove the concrete behind the H-section. The result:



The shading is now gone and the section properties will be calculated correctly.

When the default material was set to 'steel', a correction factor α will be used for all other materials in the cross section.

For example for the area of the fictive cross section we obtain this value:
 $A_{fict} = 4315 \text{ mm}^2$. This was calculated using this formula:

$$\alpha = \frac{E_{concrete}}{E_{steel}} = \frac{30\,472 \text{ N/mm}^2}{210\,000 \text{ N/mm}^2} = 0,1451$$
$$A_{fict} = A_{HEB100} + \alpha \cdot (A_{concrete,120x120} - A_{HEB100})$$
$$= 2604 \text{ mm}^2 + 0,1451 \cdot (14400 - 2604) \text{ mm}^2 = 4315 \text{ mm}^2$$

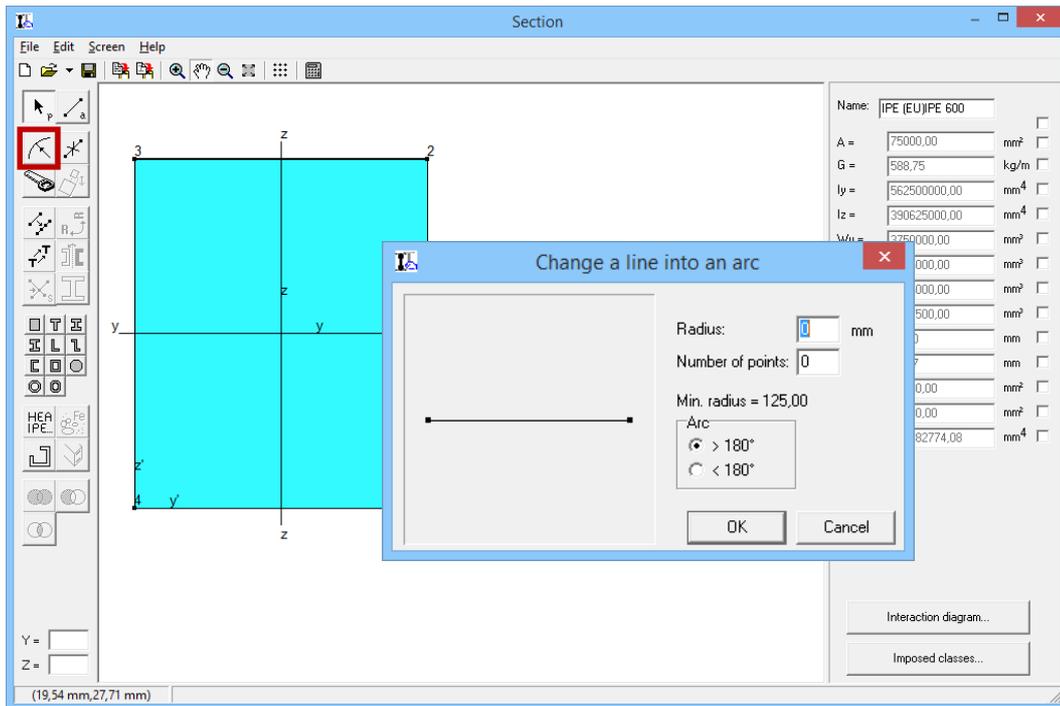
Notes:

- Steel-concrete sections can be used in an elastic analysis in Diamonds. In other words you can determine the moments, axial forces, reactions. **The stresses are always elastic! But a check according to Eurocode 4, is not implemented.**
- An interaction diagram can be requested in Section Utility (see §4.3).
- When you set the default material to 'Steel', you will be able to design the steel section according to Eurocode 3.

3.9 Example 9: The use of arcs and roundings

Earlier in this manual we mentioned you can enter arbitrary polygonal sections (see §3.4). This means that they are made up of lines. The introduction of an arc would therefore be impossible. However you can approach an arc by a number of lines. There are some built-in functions to do this automatically.

- Draw a rectangle 250x300mm. Use the drawing function  of a default shape . The material is not important. Choose something.
- Select the profile and click in the menu 'Edit' on 'Make library section editable'. All nodes will have numbers now.
- We now wish to replace the upper line with an arc. First select the line and then press the icon . The following dialog box appears:



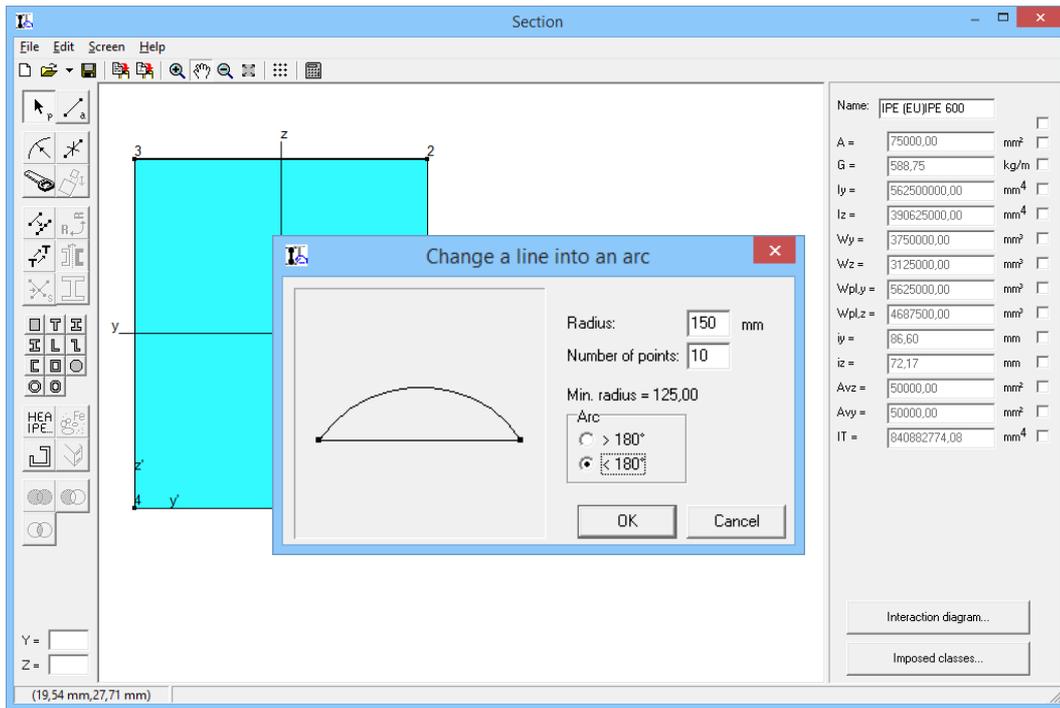
In this dialog box, the desired scale is entered. Note that there is a minimum value for the radius. When the entered radius is smaller than this minimum, the arc can not be constructed.

In fact, an arc is constructed with the help of two points and the radius. As a result, there are always two arcs possible. We can choose from these two arcs by indicating the value of the arc angle. In other words it must be greater than 180° or not.

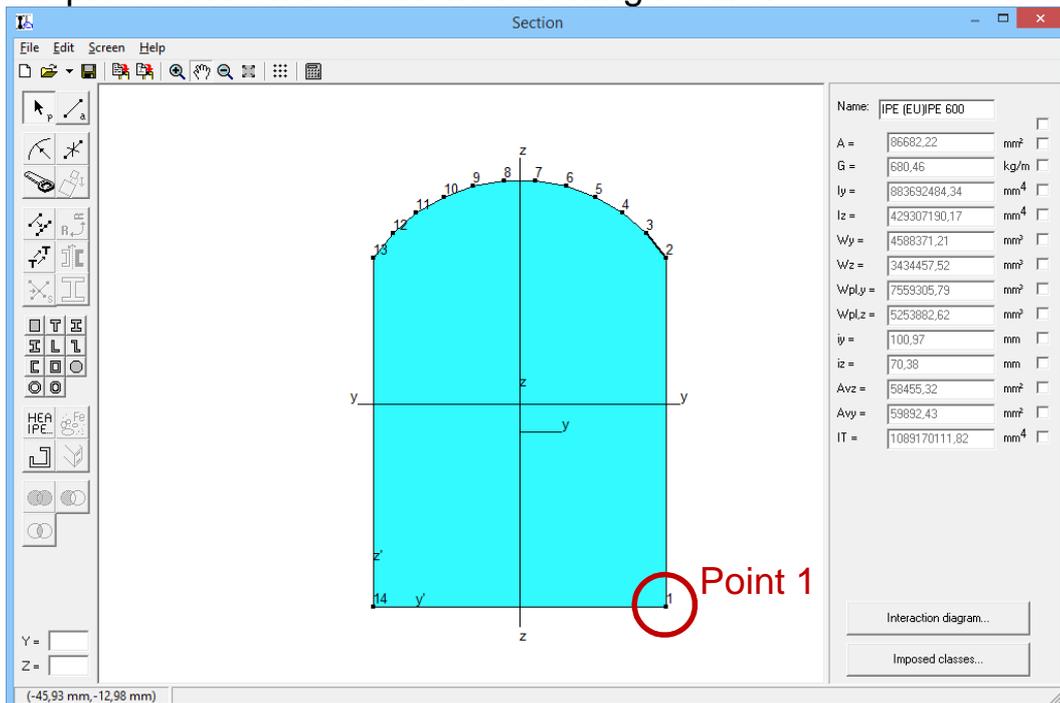
The arc is always constructed to the outside of the profile. If we wish to construct the arc to the inside, than we should enter a negative value for the radius.

Finally, we must also enter the number of points. As already mentioned, the program creates the arc using a series of straight sections. The number of points is a measure for the accuracy of the arc and therefore also of the calculation.

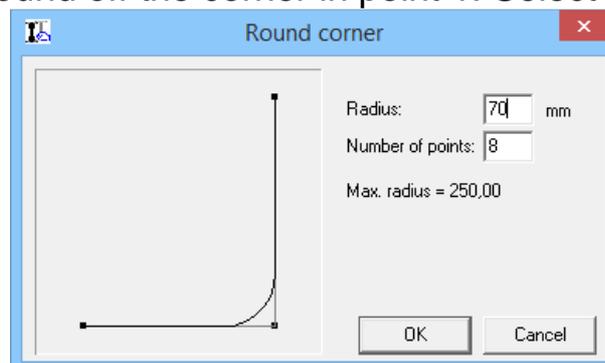
We set the radius to '150', '10' for the number of points and '<180' for the arc. Note that while entering the data the arc is immediately constructed on the left.

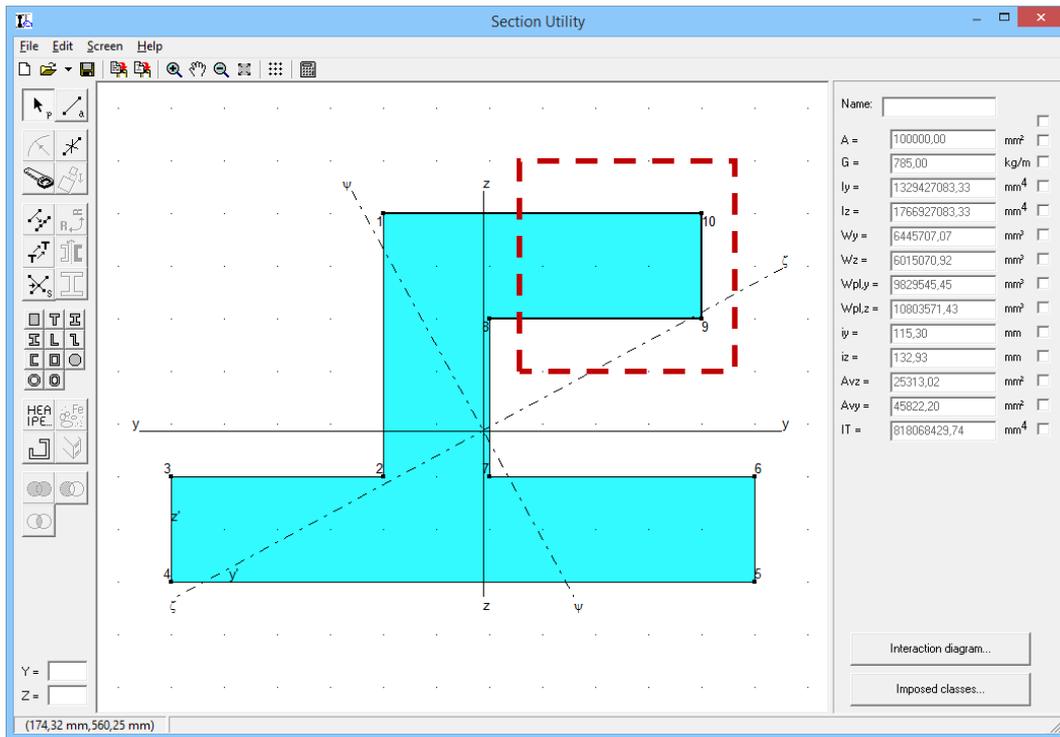


We press 'OK' and obtain the following results:

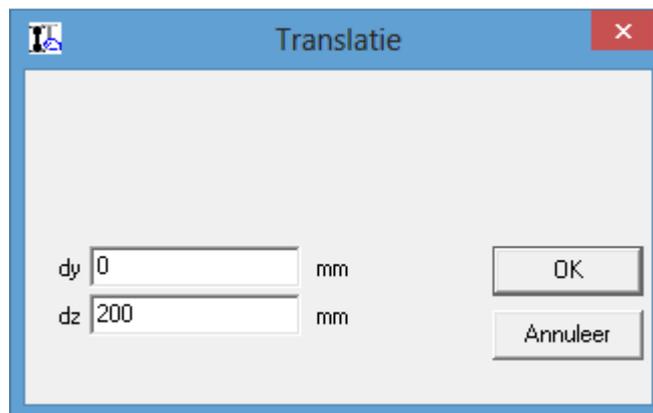


- Now we will round off the corner in point 1. Select point 1 and click :

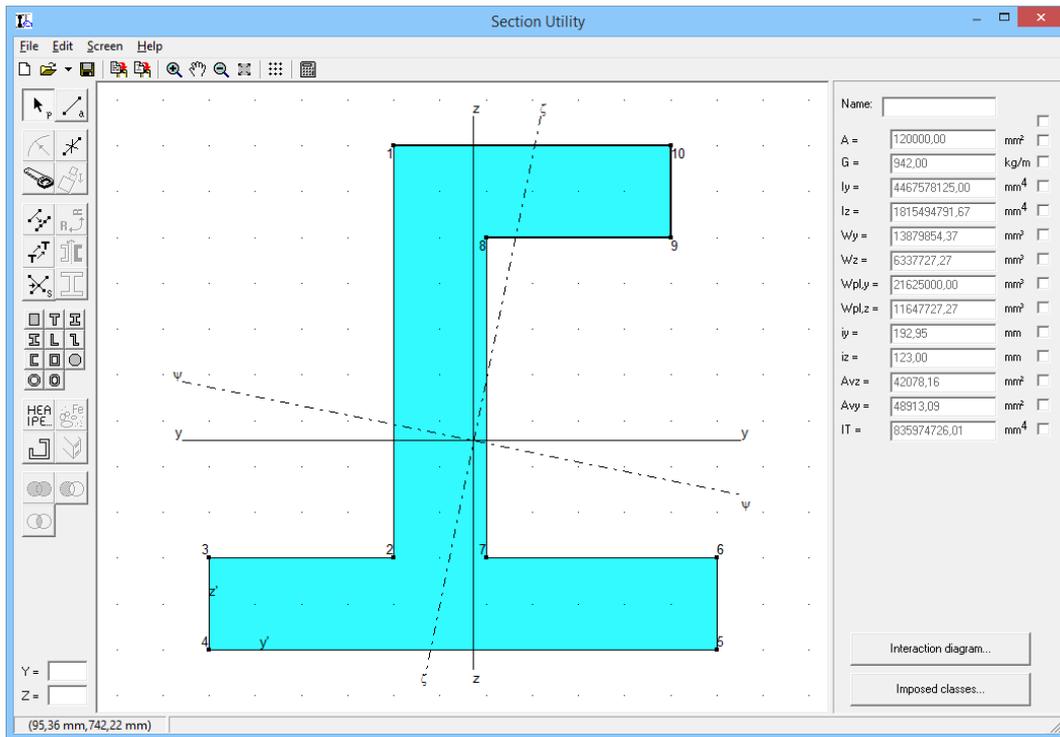




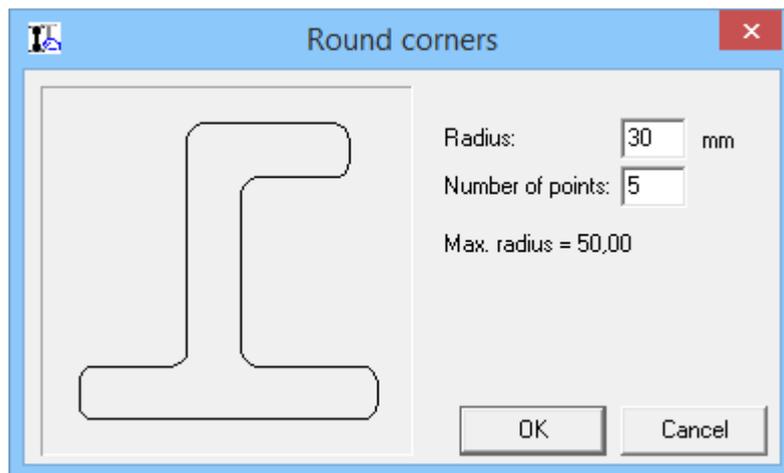
- Now click the icon . We enter '0' for dy and '200' for dz. Click 'OK'.



The results will look like this:

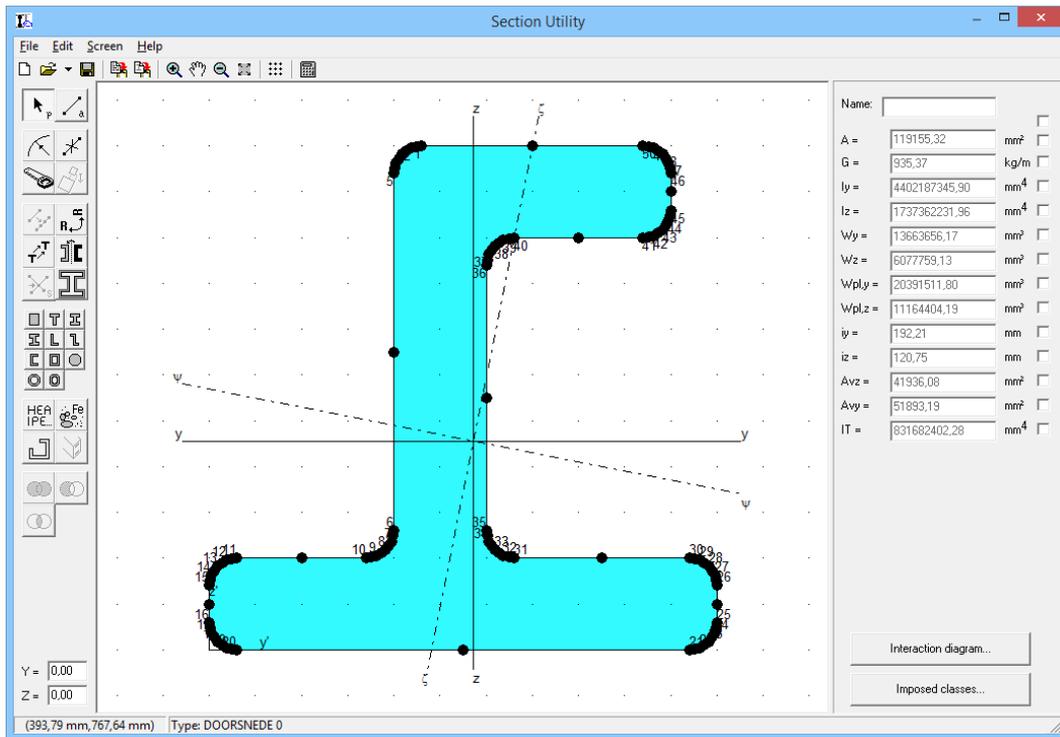


- In order to round off all corners, select the section and click the icon . The following dialogbox appears:



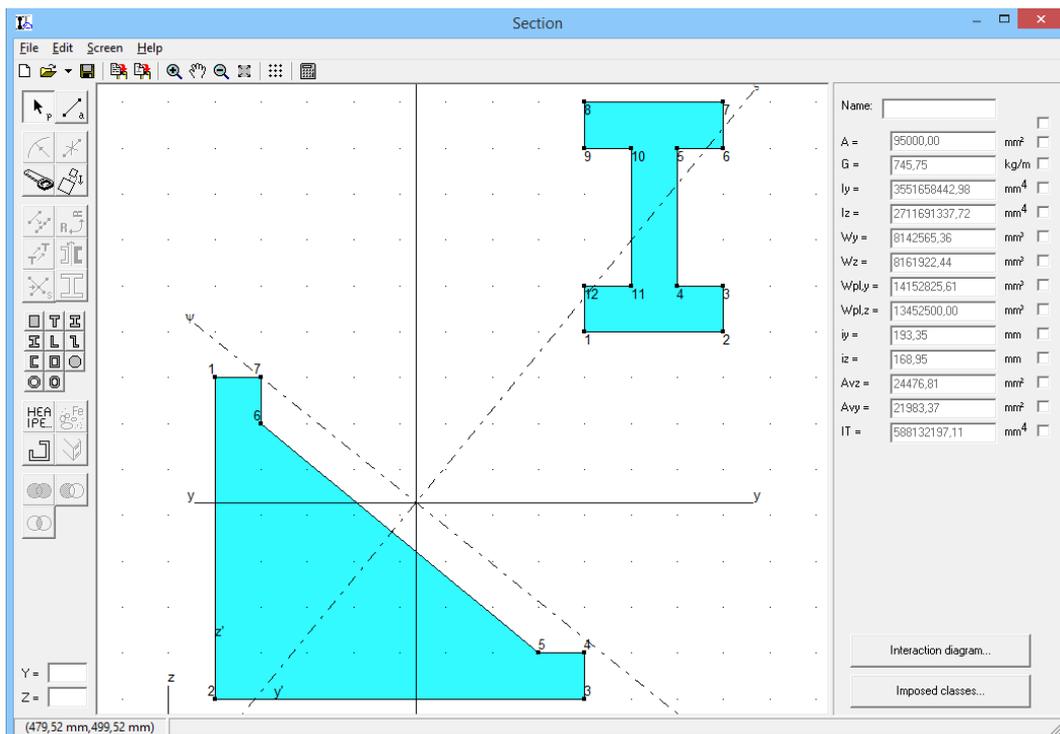
As with rounding off one point, we enter the radius and the number of points per rounding. Here too, there's a maximum associated with the radius. If the entered radius is greater than the maximum radius than the rounding can not be constructed.

We fill in '5' for number of points, and '30' for the radius. Hit 'OK'. Result:



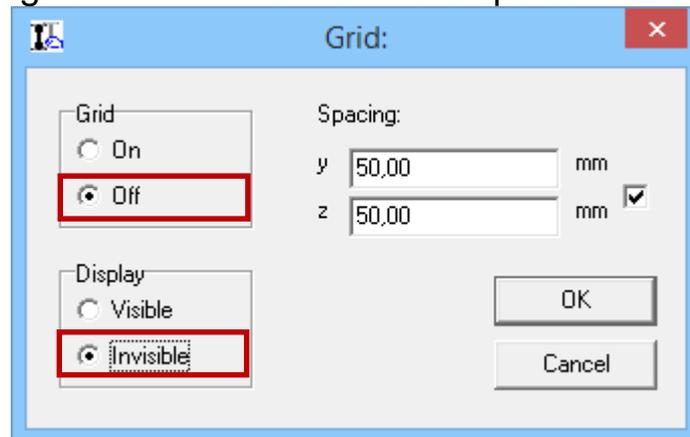
3.10 Example 10: Align cross sections

Suppose that in the example below, we'd like to place the H-profile on the sloping side. You could rotate the profile over a certain degrees and move it. However, it is simpler to make use of the alignment feature which we find in the 'Edit'-menu.

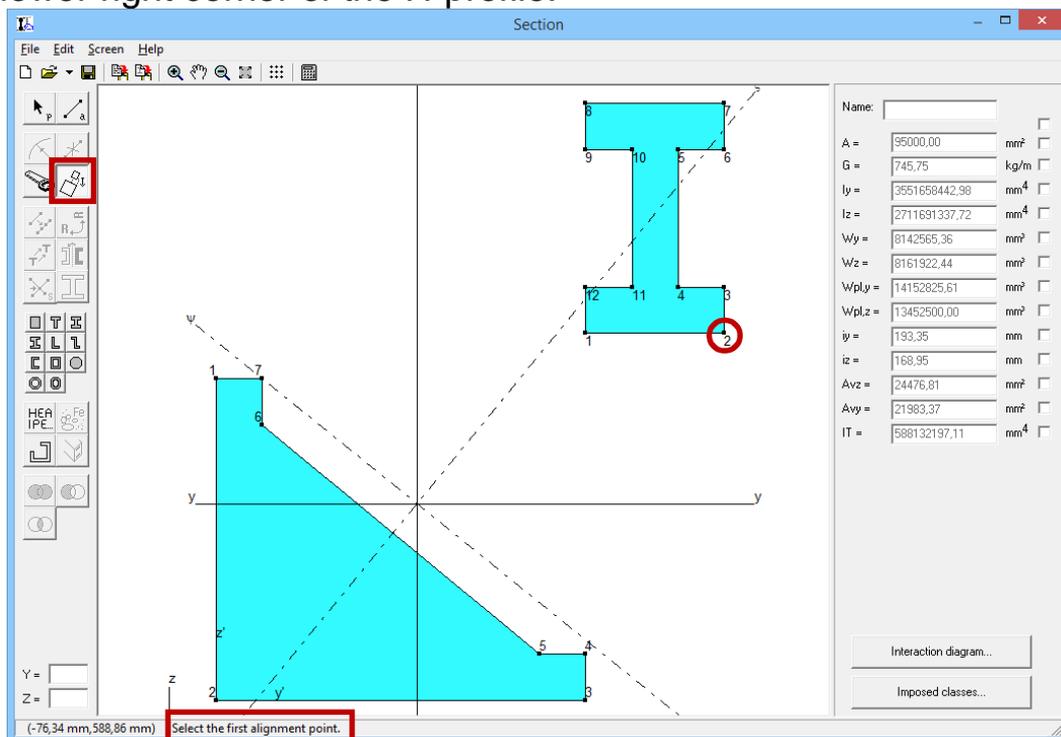


When we use this function we get the instruction in the status bar.

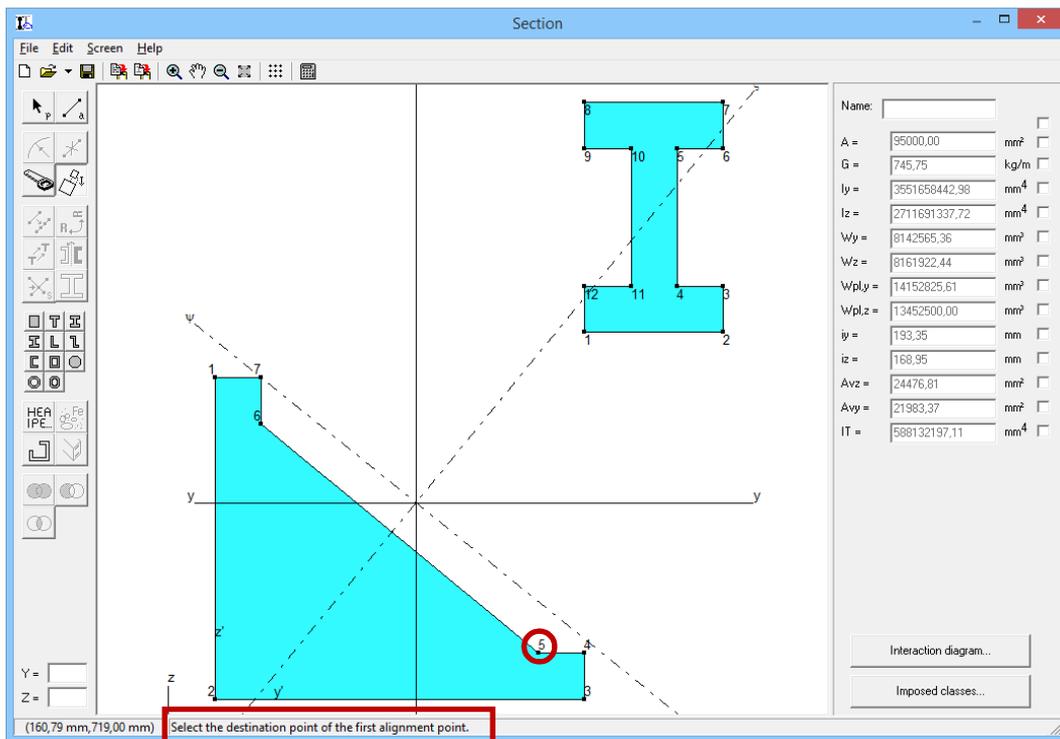
- Turn off the grid. It will make the next steps easier.



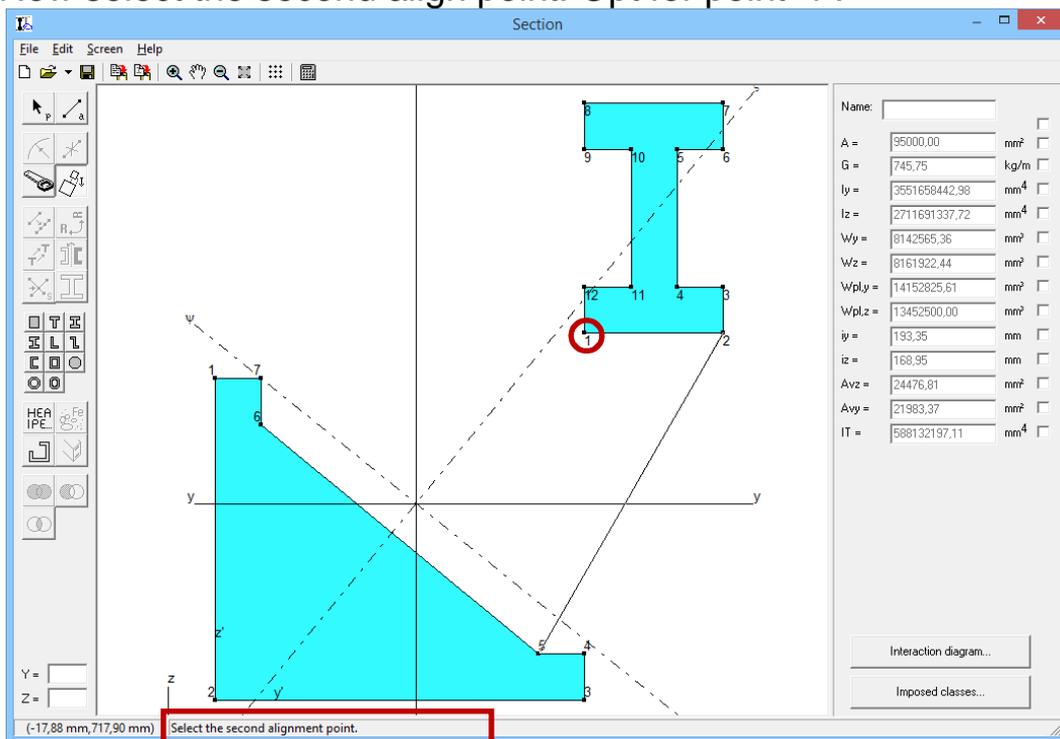
- Click on .
- The status bar will ask you to assign the first align point. Choose the lower right corner of the H-profile.



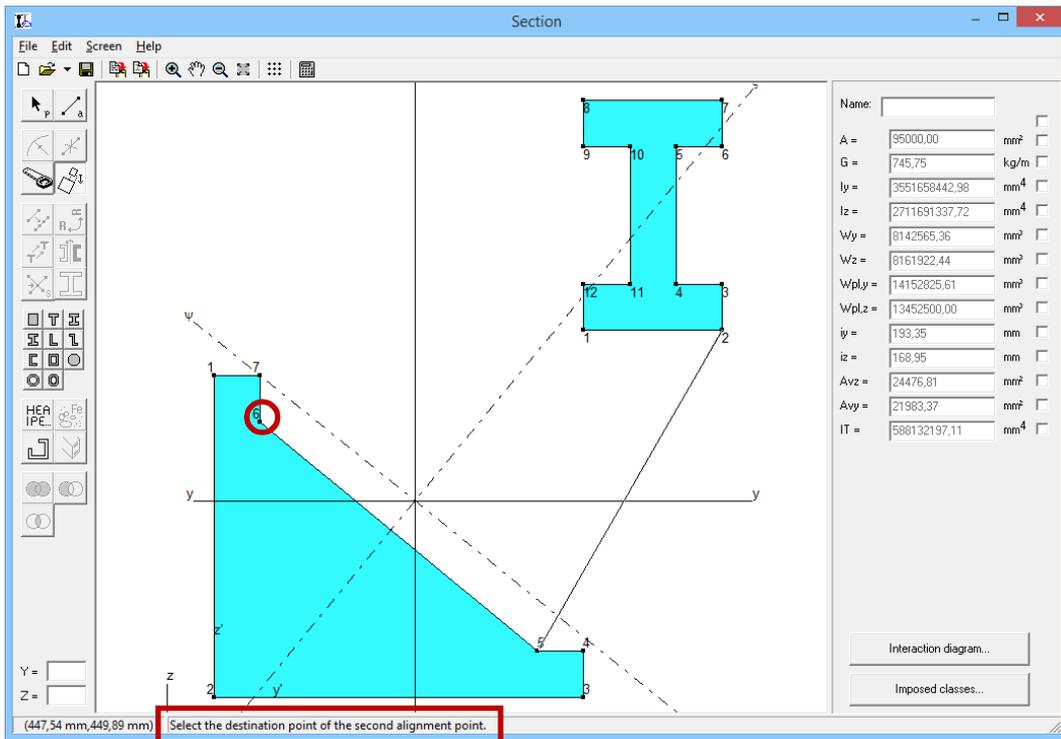
- Next Section utility will ask you the destination of this point. Opt for point '5'.



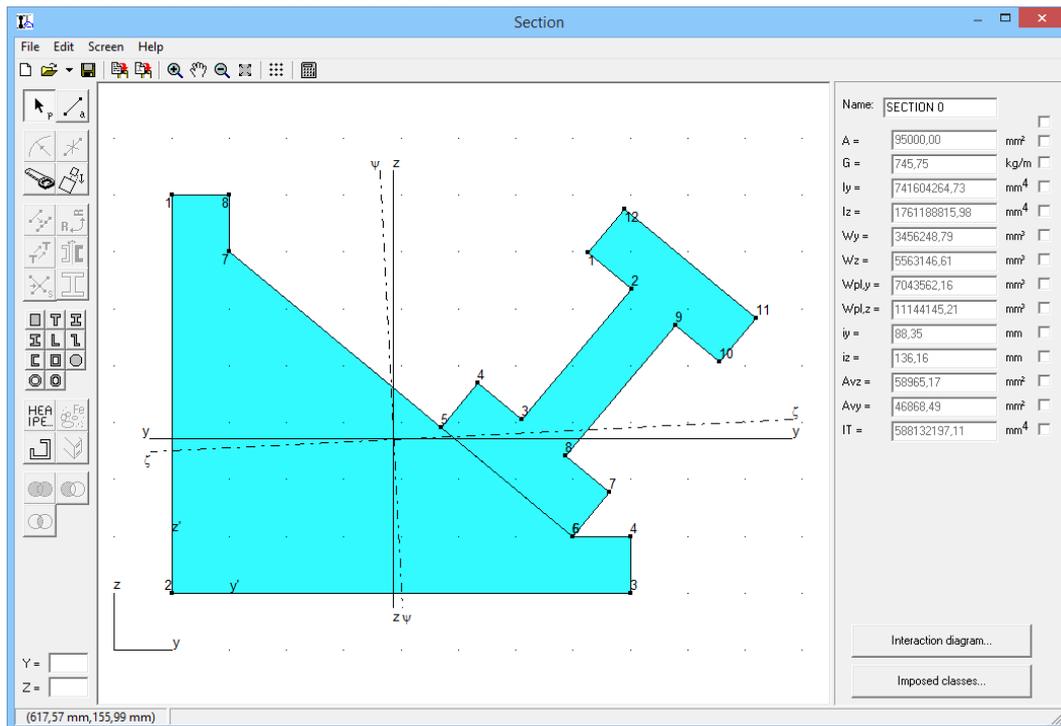
- Now select the second align point. Opt for point '1'.



- Finally indicate the destination of the second align point. Choose point '6'.



- The result should look like this:



3.11 Example 11: Duplicate

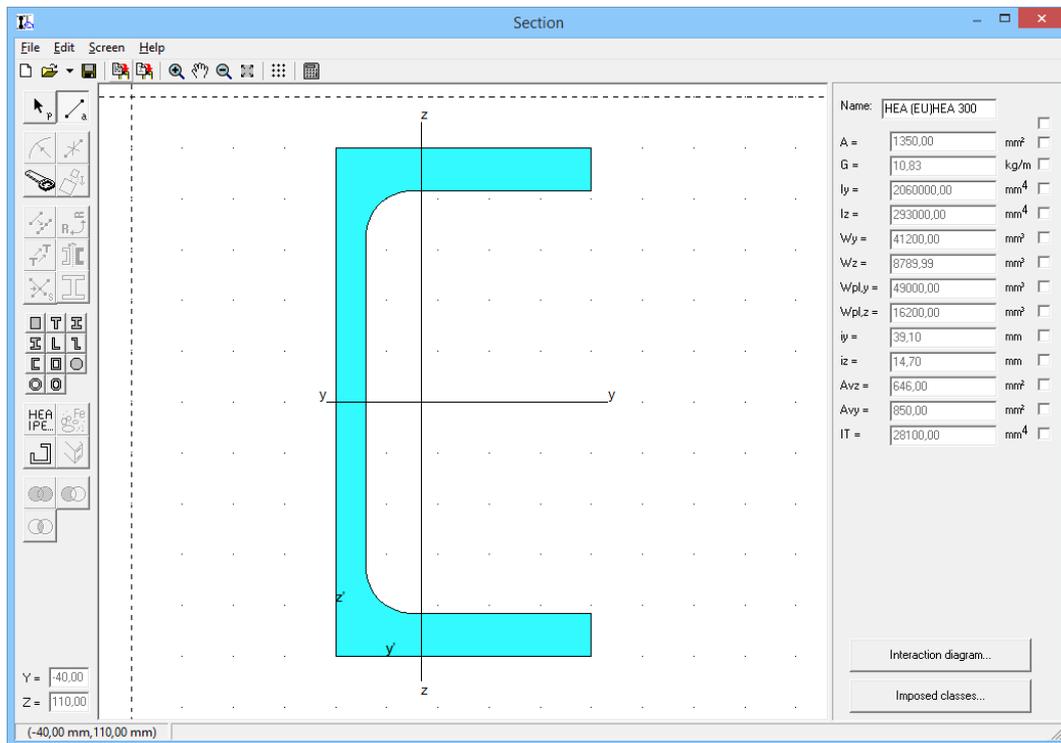
In §0 we placed 2 L-section next to each other. Such forms are common and therefore Section Utility is provided with a function to assemble easily such profiles, namely, the duplication of profiles.

Duplicating a profile is in fact a succession of three operations.

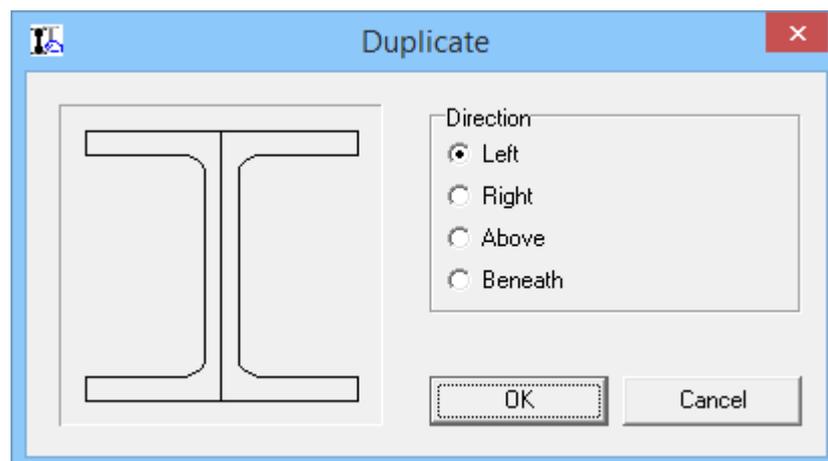
- First a copy is taken from the profile
- Next this profile is mirrored
- And finally, the two profiles are placed against each other.

As an example we're going to combine 2 U-sections

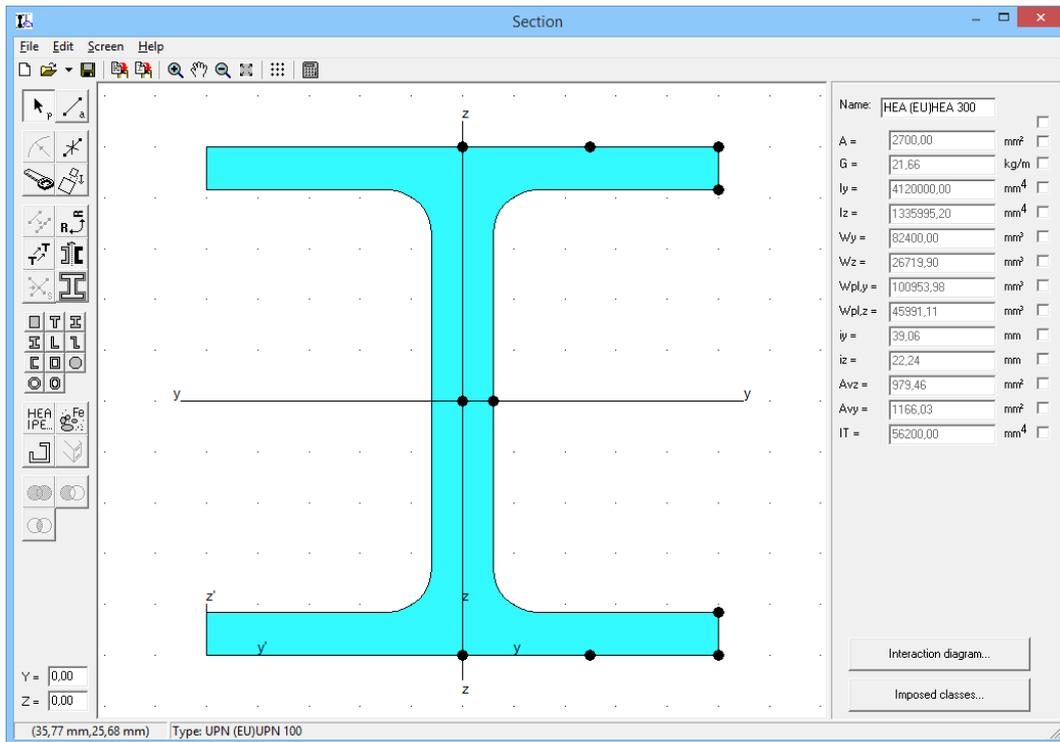
- Take an UPE 100 profile from the library:



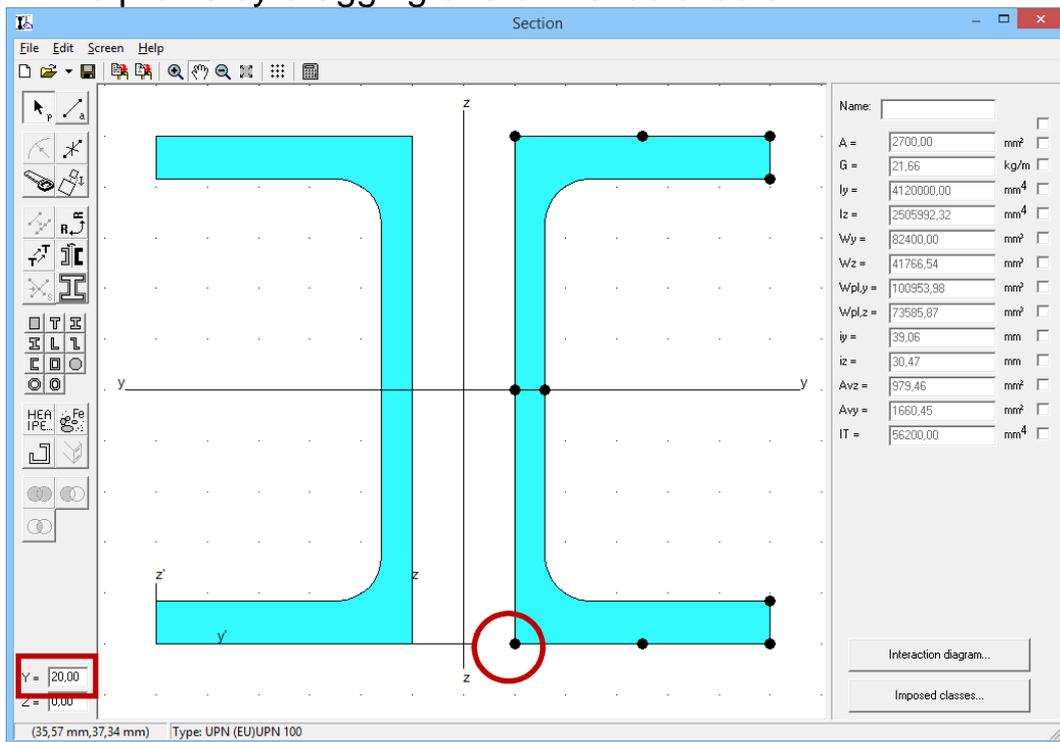
- Select the profile and click on the icon . The following window appears:



- Select 'left' and press 'OK'. Result:



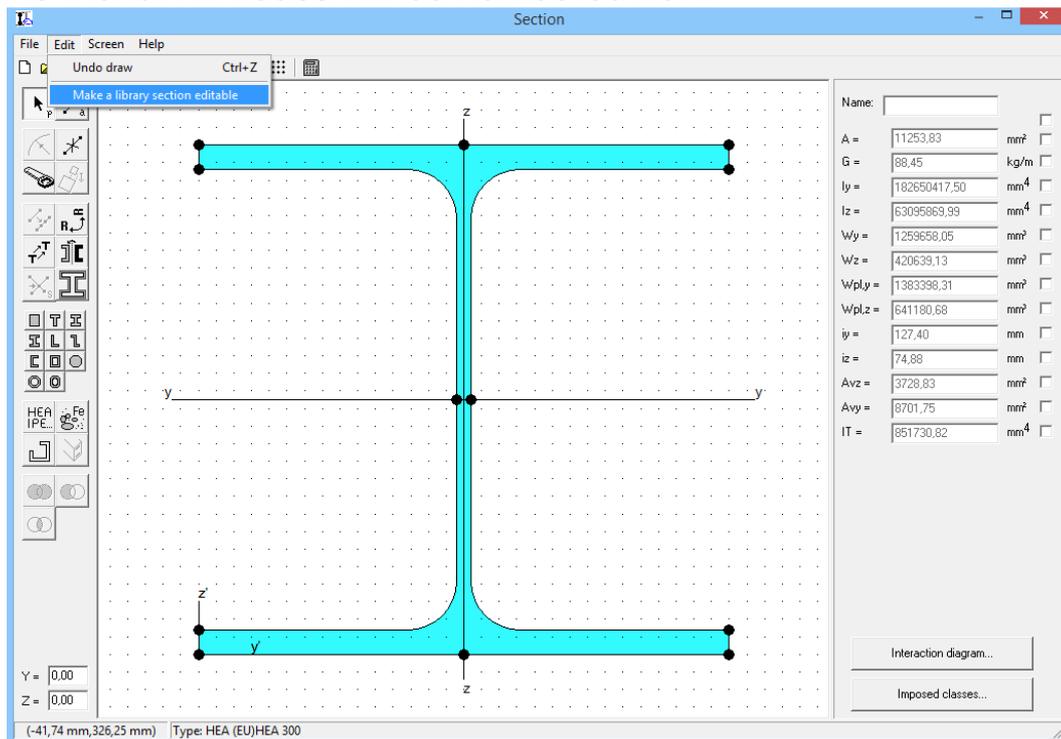
Now we can enter an offset in the two edit boxes at the bottom left. Or we can also shift the profile by dragging one of the 'bold' dots.



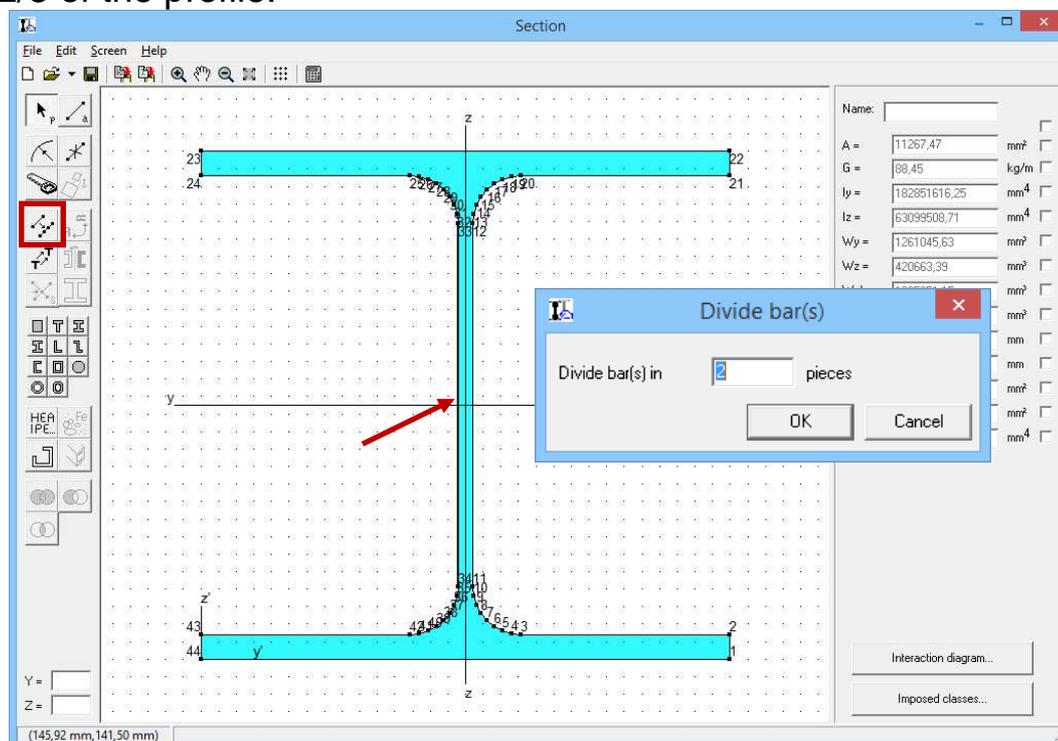
3.12 Example 12: Divide cross into two parts

Suppose we want to halve an HEA 300, then we proceed as follows:

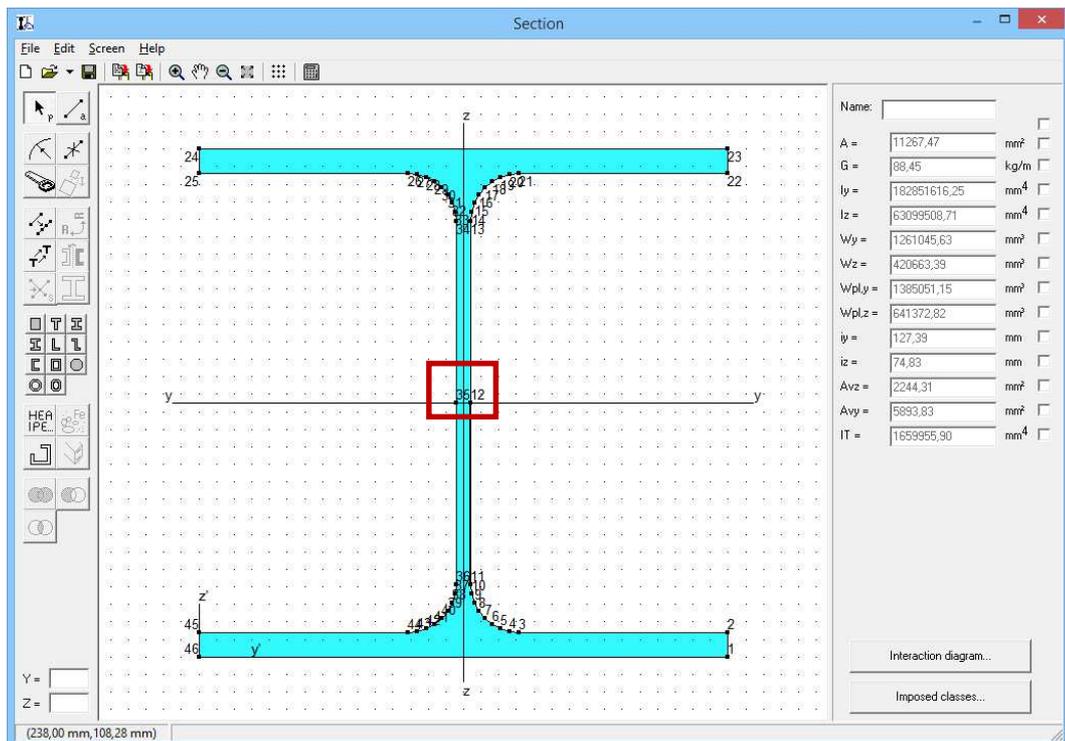
- Chose a HEA 300 from the library ^{HEA IPE...}
- Select the profile and click 'Edit' – 'Make a library section editable' from the menu. All nodes will be numbered now.



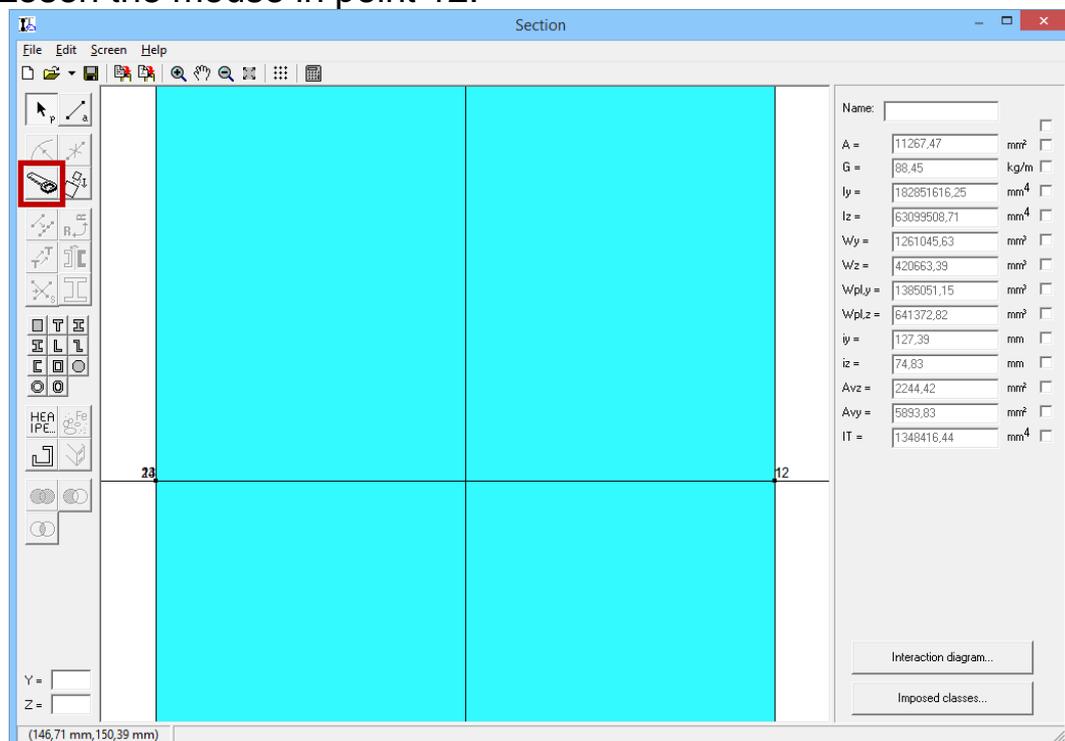
- Select the left line of the web and divide it in 2. If necessary, you can change the length of two pieces in case you want to keep for example 2/3 of the profile.



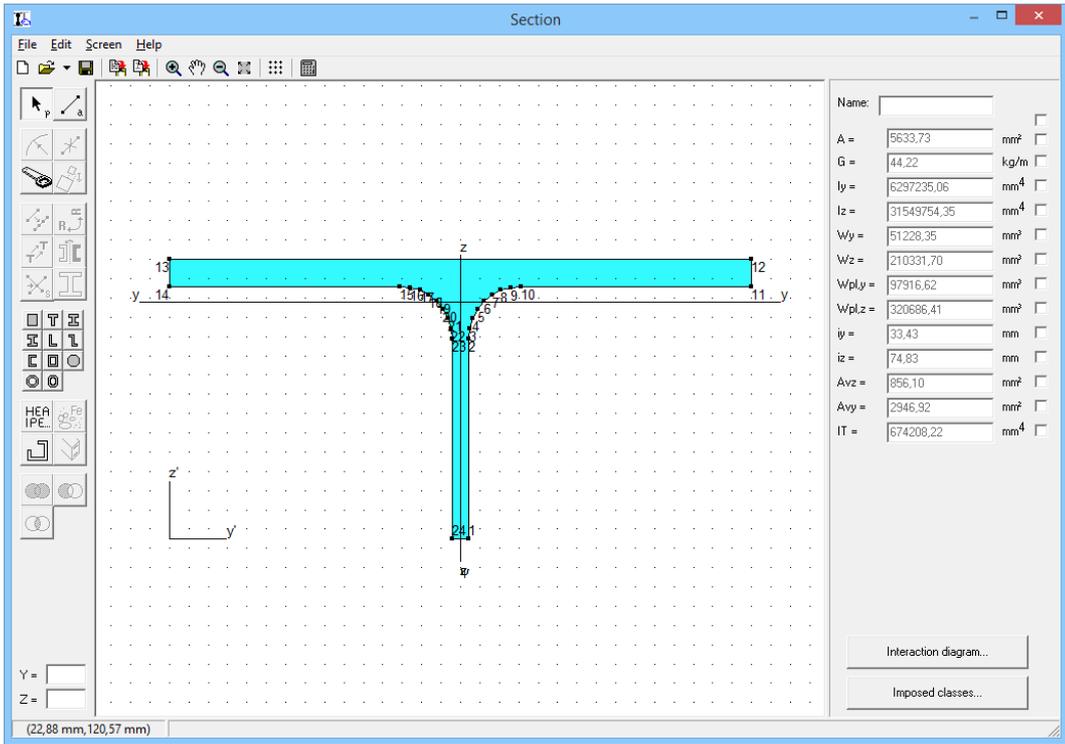
- Repeat the steps for the line on the right of the web. Now, two numbers will be shown in the middle of the cross section.



- Zoom in  (a lot). Use the cutting tool  to divide the section. Click once in point 35, keep the mouse pressed in while you go to point 12. Loosen the mouse in point 12.



- Now click the button  to show the profile as large as possible on the screen.
- Remove the lowest part of the profile (DEL-button). Result:



4 Other functions

4.1 Edit results

When we use a composite section to calculate structures in eg. Diamonds, it may be useful to modify certain properties.

In the menu 'Screen' we click on 'Results', option 'Results editable' on. Note also that we can hide the results column by selecting 'None'.

The result column looks as follows:

Name:	<input type="text" value="SECTION 0"/>	<input type="checkbox"/>
A =	<input type="text" value="3400,82"/>	mm ² <input type="checkbox"/>
G =	<input type="text" value="26,72"/>	kg/m <input type="checkbox"/>
I _y =	<input type="text" value="8644139,96"/>	mm ⁴ <input type="checkbox"/>
I _z =	<input type="text" value="3175230,75"/>	mm ⁴ <input type="checkbox"/>
W _y =	<input type="text" value="144069,00"/>	mm ³ <input type="checkbox"/>
W _z =	<input type="text" value="52920,51"/>	mm ³ <input type="checkbox"/>
W _{pl,y} =	<input type="text" value="165221,32"/>	mm ³ <input type="checkbox"/>
W _{pl,z} =	<input type="text" value="80969,76"/>	mm ³ <input type="checkbox"/>
i _y =	<input type="text" value="50,42"/>	mm <input type="checkbox"/>
i _z =	<input type="text" value="30,56"/>	mm <input type="checkbox"/>
A _{vz} =	<input type="text" value="1096,32"/>	mm ² <input type="checkbox"/>
A _{vy} =	<input type="text" value="2760,25"/>	mm ² <input type="checkbox"/>
IT =	<input type="text" value="138408,45"/>	mm ⁴ <input type="checkbox"/>

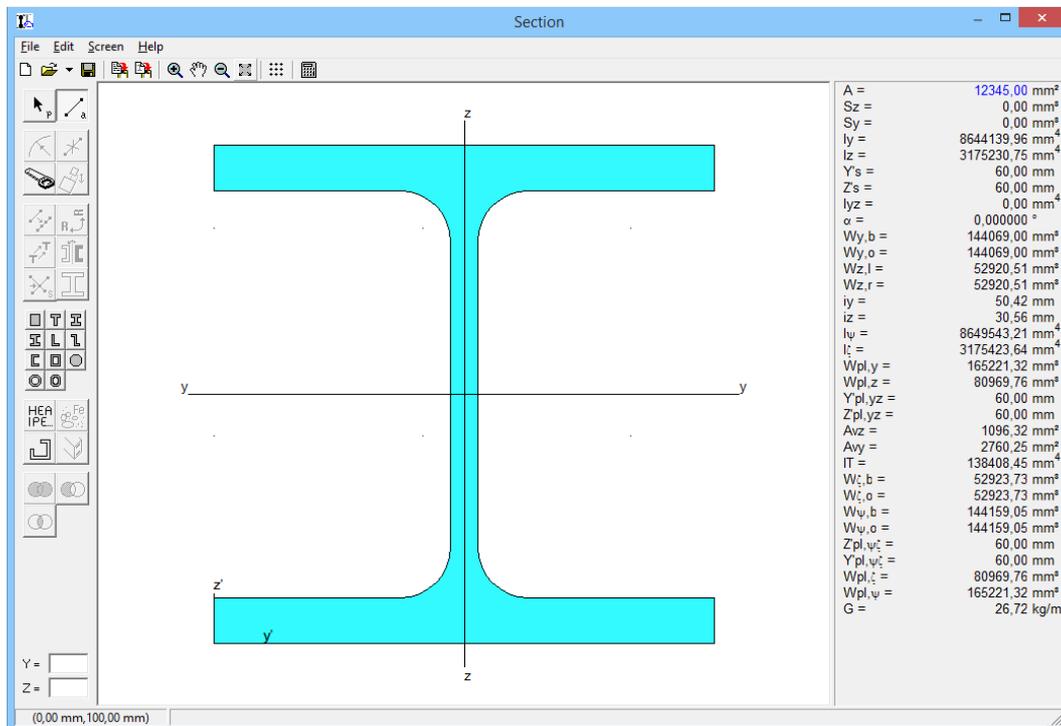
At the top we can give the profile a name. As long as we don't enter a name, the name of the first inserted element is displayed.

Next to the name, there's a check box. This is used to turn on or off ALL check boxes.

Next, we see the calculated values for a number of characteristics. At this time the values can't be changed. We wish now to give the area for example a personal value. Check the box behind the area. Now we can enter a value for the area.

Name:	<input type="text" value="SECTION 0"/>	<input type="checkbox"/>
A =	<input type="text" value="12345"/>	mm ² <input checked="" type="checkbox"/>

As long as we have ticked the box, the entered value will be displayed for the area. When you select 'Screen' – 'Results' – 'Results Ineditable' from the menu again, then the entered value for the area will still be shown. However, to indicate that it is a value entered, it is displayed **in blue**.



If we change the section now, the characteristics which will not be completed by the user are automatically recalculated. The characteristics that were entered by the user remain unchanged.

4.2 The button 'Imposed classes'

Using an elastic analysis (in Diamonds or PowerFrame) we determine the internal forces M, V, N , elastic stresses σ and the elastic deformations δ, φ in a steel structure.

During design, we check if the chosen profiles (HEA, IPE, ...) are capable of resisting the internal forces M, V, N .

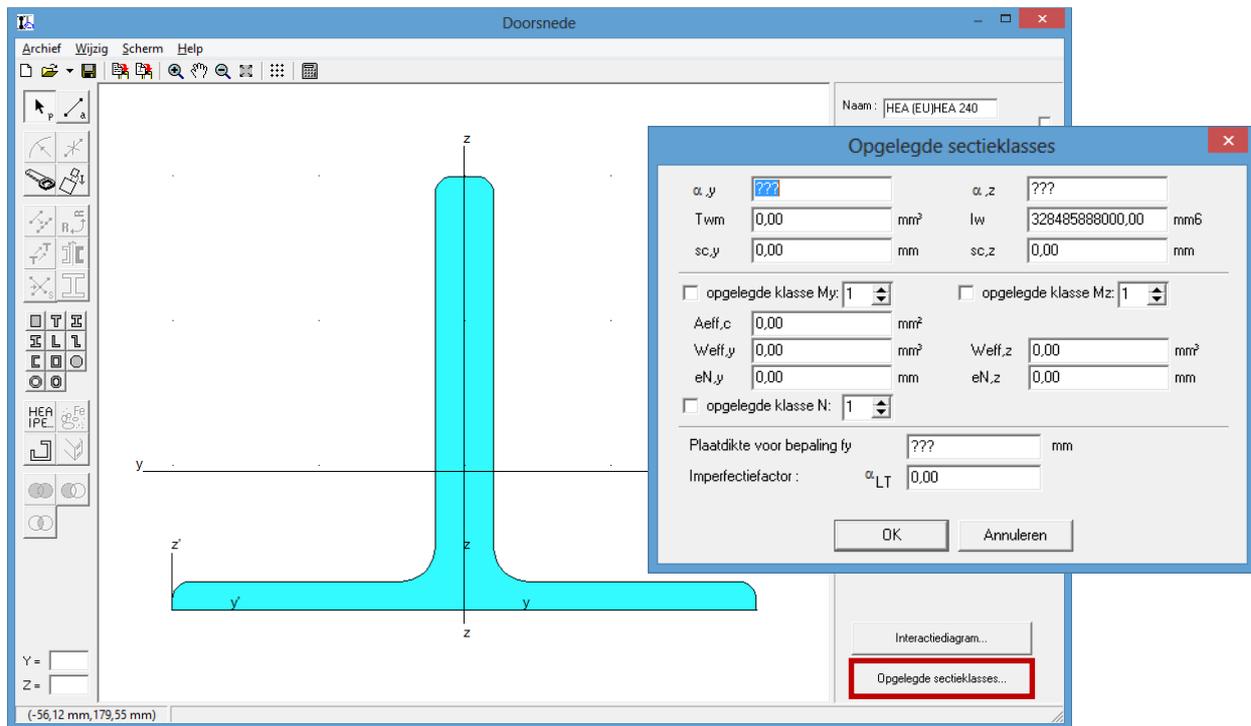
In EN 1993-1-1 this design consists of two parts:

- On one side strength (will the profile break or not?)
- On the other side stability (will the profile not buckle?)

To perform these calculations we need some additional parameters next to the section properties.

EN 1993-1-1 gives rules to calculate these additional parameters for default sections (I, H, T, L, O, □). No rules are provided for all other cases (+, △, HH, J L, combinations of basis shapes...).

If you do wish to design F_e these profiles (+, \triangle , HH, \perp L, combinations of basis shapes ...), then you should enter the parameters after the button 'Imposed section classes':



- The imperfection factors for buckling α_y en α_z around the y' and z' -axis (see EN 1993-1-1 Table 6.1)
- I_w is the warping constant (see Berekening van constructies, deel I, Vandepitte, p163 and 173)
- T_{wm} is the torsional resistant moment (see Berekening van constructies, deel I, Vandepitte, p138)
- sc_y and sc_z are the coordinates of the shear centre in relation to the xy-coordinate system (see Berekening van constructies, part I, chapter 2.2, Vandepitte, p108).
- The section class for axial force N and bending moments M_y and M_z (see EN 1993-1-1 Table 5.2). In case the cross section is of class 4 (thin sections), you should also enter the effective properties of the cross section (see EN 1993-1-5 §5.2.2.).
 - o $A_{eff,c}$ is the effectieve area
 - o $W_{eff,y}$ is the effective moment of resistance for bending around the axis y'
 - o $W_{eff,z}$ is the effective moment of resistance for bending around the axis z'
 - o $e_{N,y}$ and $e_{N,z}$ are the coordinates of the centre of gravity of the effective cross section in relation to the centre of gravity of the gross cross section.

- The plate thickness for the determination of the fielding strength. This is the thickness of the thickest element (flange or web) of the cross section (see EN 1993-1-1 Table 3.1).
- The imperfection factor for lateral torsional buckling α_{LT} (see EN 1993-1-1 Table 6.3)

Opmerkingen/tips:

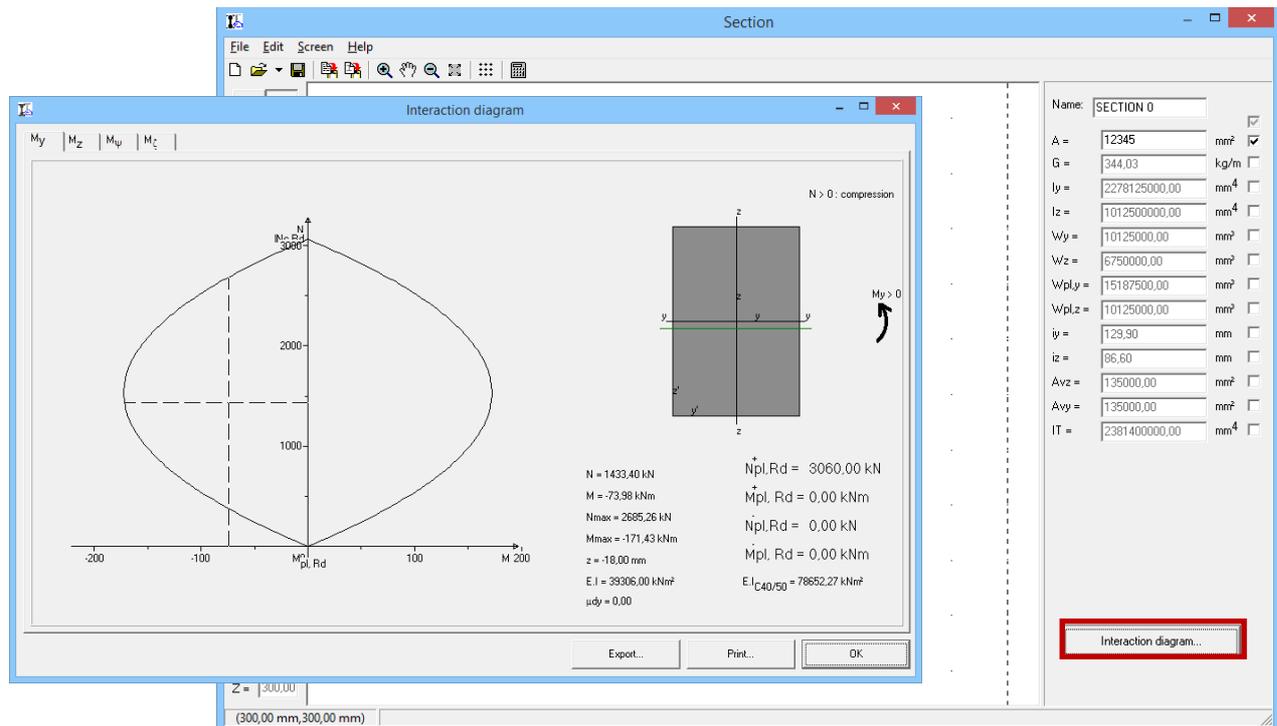
- *Berekening van constructies* by Daniël Vandepitte is a Dutch book that can be read online: <http://www.berekeningvanconstructies.be/>
- Think about the design before enter values:
 - o If you'd enter random values for all parameters, the results of the design will have no significance!
 - o Is it necessary to fill in/calculate all parameters?
For example: when a cross section is of class 1, 2 or 3, you may enter random numbers for the effective properties. Because effective properties are only used when a cross section is of class 4.
Or when a cross section is not loaded with torsion, a resistance check for torsion will not be required. So you can enter a random value for T_{wm} .
 - o Some profiles are simply not designed to take certain internal forces.
For example: THQ-profiles are designed to resist bending, not axial compression. Therefore take the necessary provisions in terms of modeling so no axial compression will be acting on this profile.
Open cross sections (H, T, L-sections) are not designed to take torsion, closed section are (O, \square)!
- If possible, try to approximate the profile by a default shape (I, H, T, L, O, \square). For these sections the properties are automatically calculated. For example, a double L-profile (as shown in the figure above) you can be approached by a T-shaped profile that you subsequently rotate over 180°.

4.3 The button 'interaction diagram'

The interaction diagram shows which combination of moment M and normal force N you can take around a particular axis. There are four charts:

- bending moment around the y-axis M_y + normal force N
- bending moment around the z-axis M_z + normal force N
- bending moment around the ψ -axis M_ψ + normal force N
- bending moment around the ζ -axis M_ζ + normal force N

For a double symmetric profile the chart with M_y is the same as with $M_{y'}$, and M_z the same as $M_{z'}$.



The interaction diagram is only available for concrete sections. This diagram doesn't depend on any standard and is based on a concrete section without reinforcement steel.

4.4 Import and export

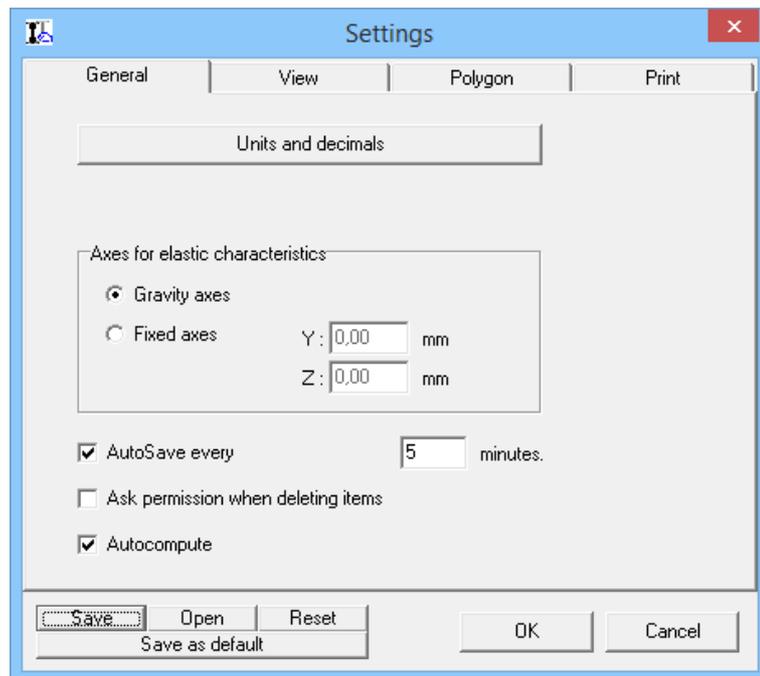
With the export function we can store our composite section on the hard disk. This is not the same as saving the file. When saving a file not only the section is written away, but also eg. the settings.

Sections which have been exported, can later be imported into another composite section. This way we don't need to draw common composite sections over and over again.

Even sections that are just saved can be imported.

4.5 Settings

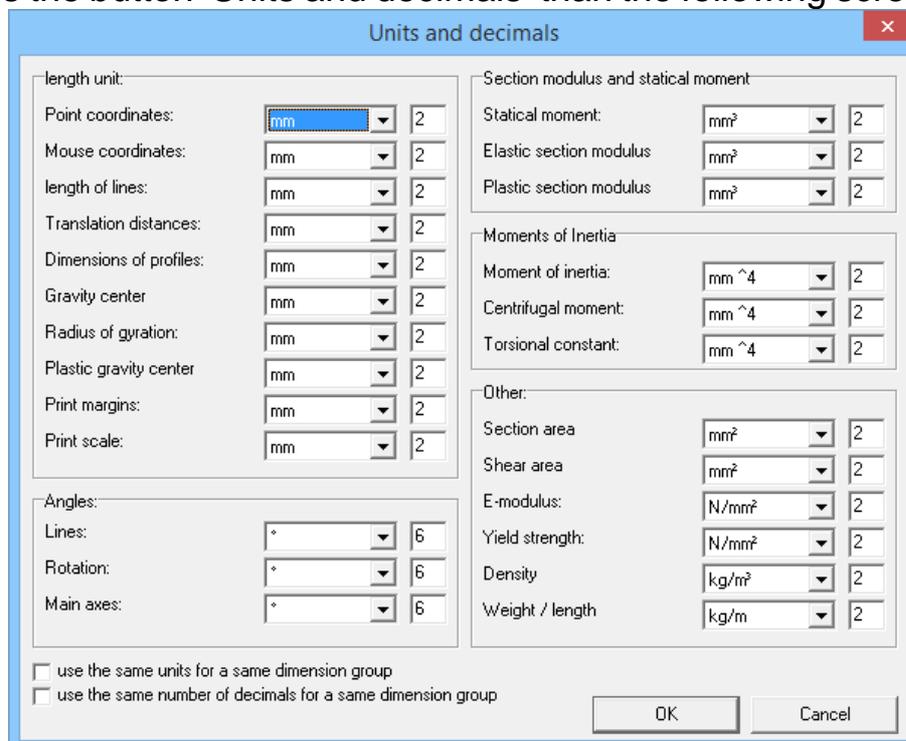
In the menu 'Screen' there's an item 'Settings'. When we click on it, we see the following dialog box:



This dialog box has different tab pages.

4.5.1 Tap page ‘General’

- Press the button ‘Units and decimals’ than the following screen appears:

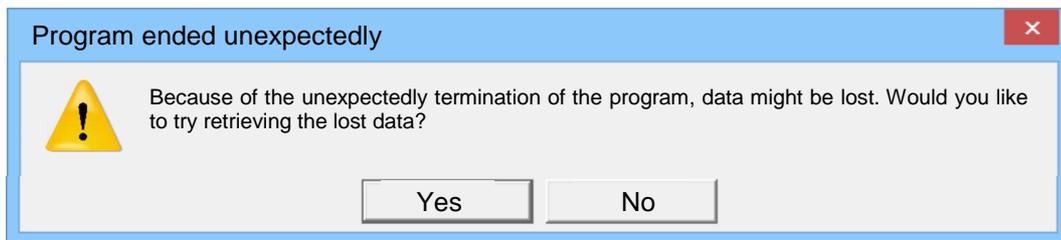


In this dialog window we can enter the units and number of decimals for all values used in the program. We see that different units are possible for entering a distance. For example it can be handy to use a smaller number of decimals on the mouse coordinates than on the point coordinates.

However when we'd like to change all units for length to inches, that would mean we'd have to select 'inch' instead of 'mm' 10 times. But we can do this in one step by selection the option 'use the same units for a same dimension group'. The same counts for the option 'use the same number of decimals for the same dimension group'.

We click 'OK' to store the units and number of decimals and return back to the 'Settings'-dialog.

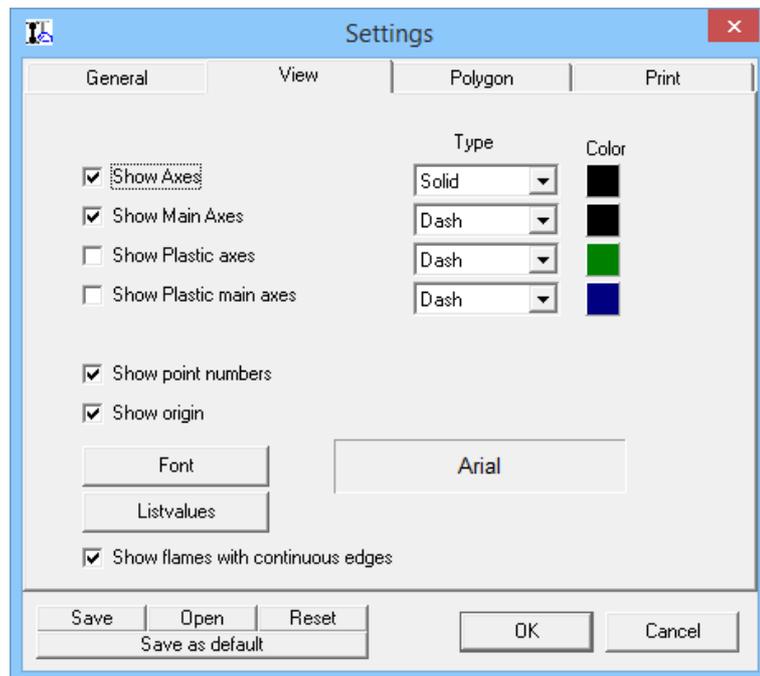
- Calculating the elastic characteristics depends **on which axes we use**. We can calculate our characteristics around the gravity axis, but we can also impose the axes around which the characteristics are to be calculated.
- We see a check box to enable or disable **the automatic saving**. The time interval between two successive save operations can be entered. With automatic storage, however, the section is not written to the file itself, but to a temporary file. When the program unexpectedly breaks the following message will appear at the restarting of the program:



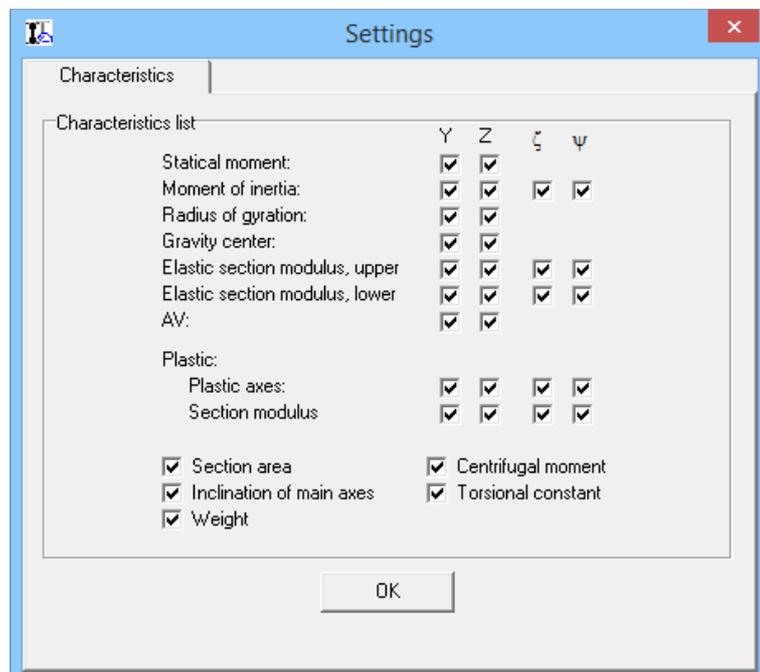
If we select 'Yes' the last saved section will be loaded. If we click 'No' the last saved section will be deleted from the hard disk.

- We can also set whether we'd like **a confirmation when removing elements**.
- We can also set if Section Utility should automatically calculate the characteristics. When the latter is off the construction is calculated by pressing the icon  in the icon bar.
- At the bottom there are 4 buttons. One to save the settings to a file and one to retrieve the saved settings. The 'Reset' button to reset all values to default again. And de 'Save as Default' command that saves the settings as default.

4.5.2 Tap page 'View'



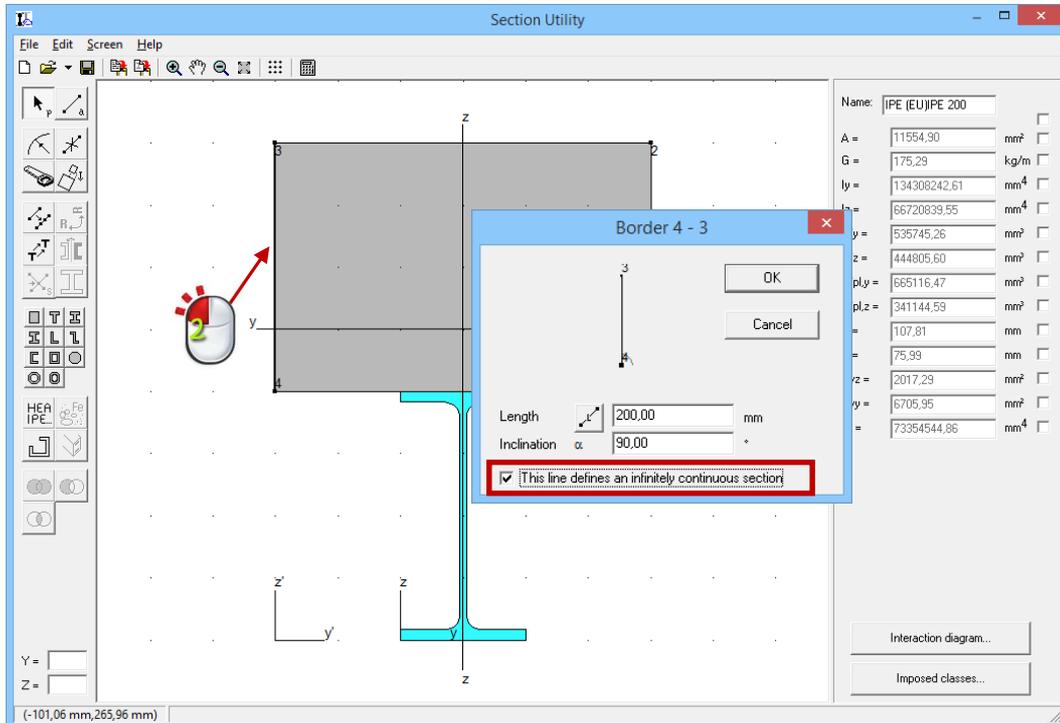
- For all axes, we can define here if **they should be shown**, in which **line type** and in which **color**.
- Also the **point numbers** and the **origin** can be shown.
- By clicking on the button '**Font**' the font of the resultswindow and the calculation note can be changed.
- With the button 'Listvalues' we enter the tab page '**listvalues**'.



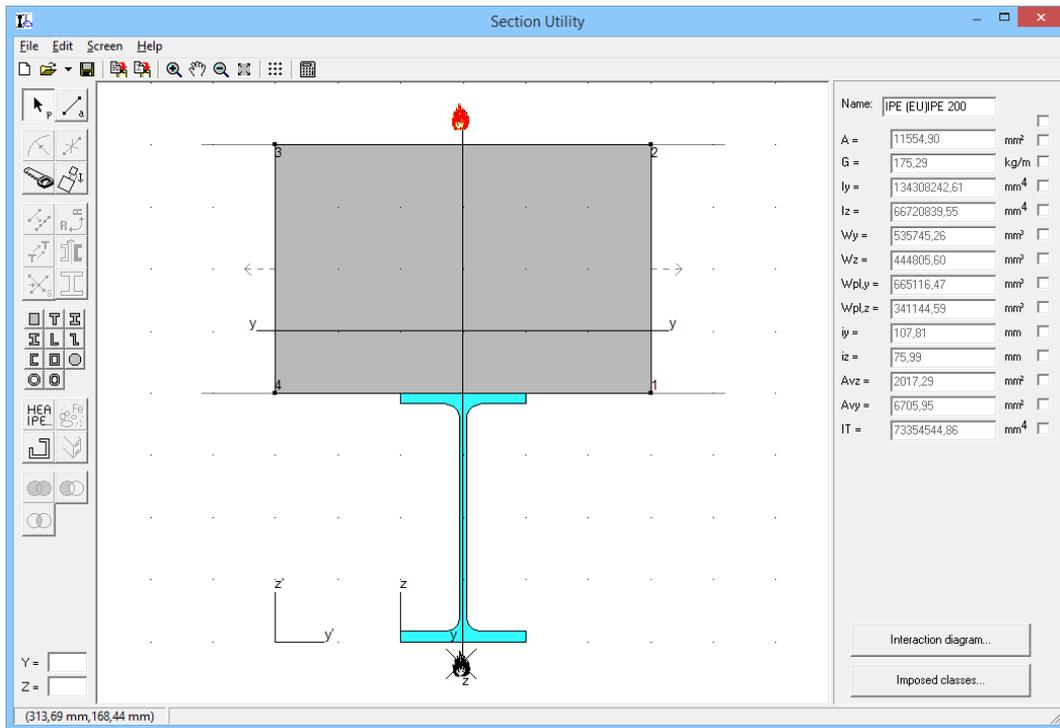
- Here we can set for all characteristics if they should be shown or not.
- The option 'Show flames with contiuous edges' is used in fire calculations (only possible in PowerFrame for the moment).

Assume a floor slab of 20cm lies on an H-profile. The floor slab can be considered as continuous. This can be entered by defining the vertical borders as continuous.

- Double click on one of the borders.
- Select the option 'This side defines an infinitely continuous section'

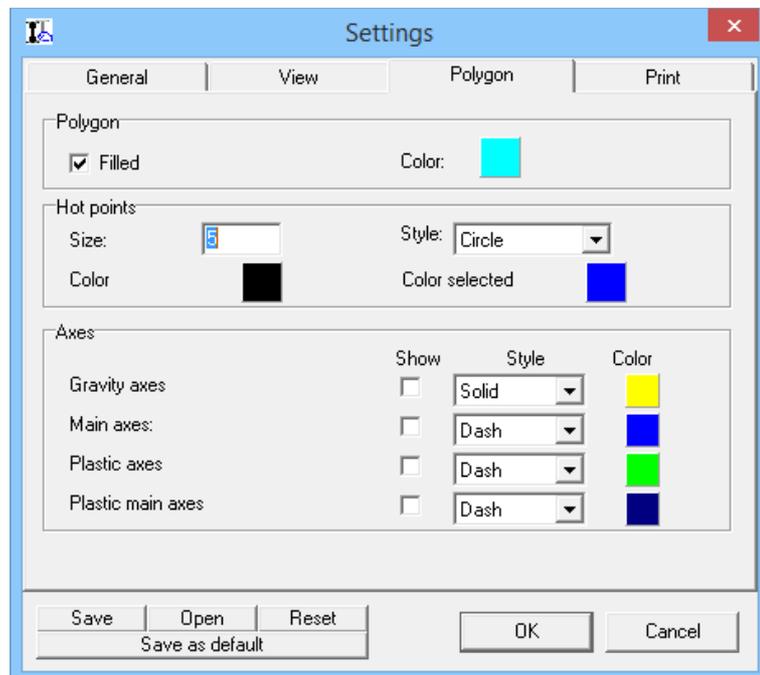


- Repeat the steps for the vertical line on the other side.
- The continuity of the plate edges is indicated by arrows. Also two flame icons appear . By clicking on these icons you can specify where the fire is and where not. In the figure below the fire is at the level of the floor, but not at the level of the H-profile.



It's recommended to always show the flame icons.

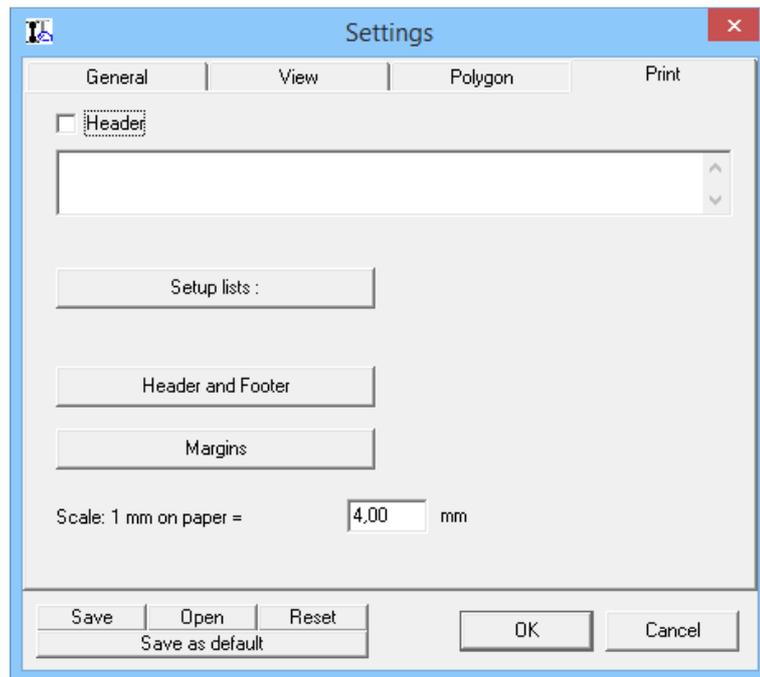
4.5.3 Tap page 'Polygon'



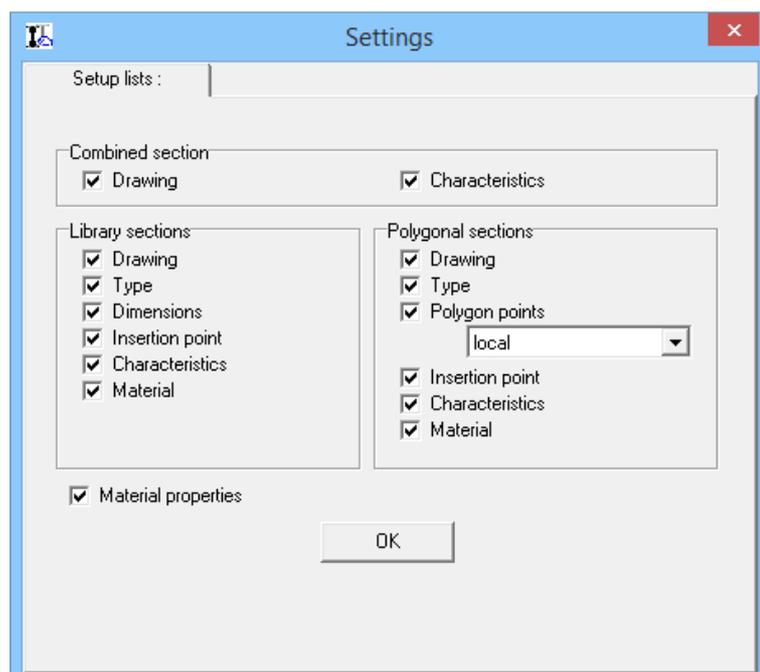
On this tab we can determine the color of the polygon, this is the color the polygon before a material assigned to it. We can also determine whether the polygon should be filled or not.

The size, color and style of the fixed suspension points can be changed here. The axes of the individual sections may also be displayed, and for each axis the line type and color can be set.

4.5.4 Tap page 'Print'

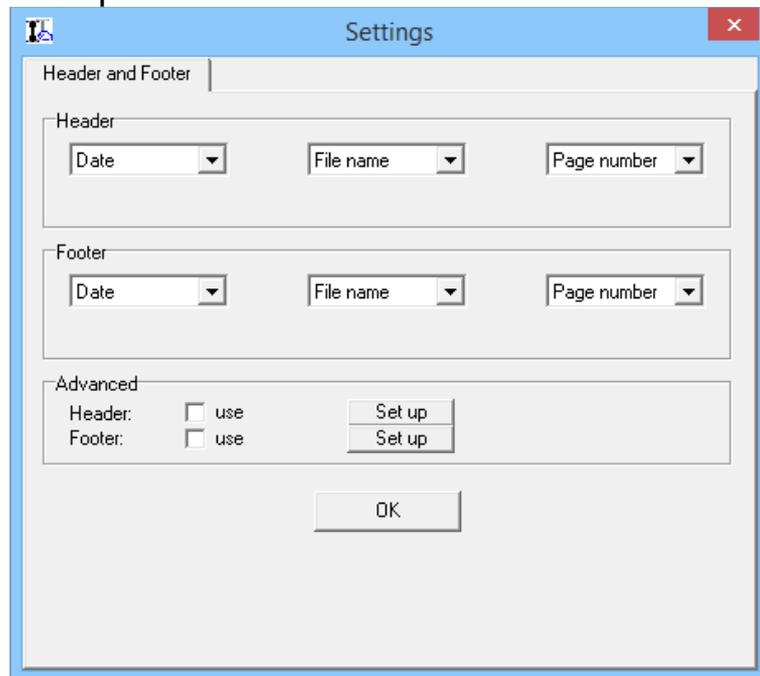


- On top of the first page we can put a **header**. The text of the header as to be filled in in the text field.
- When we click the '**Setup lists**' button this dialog box appears:



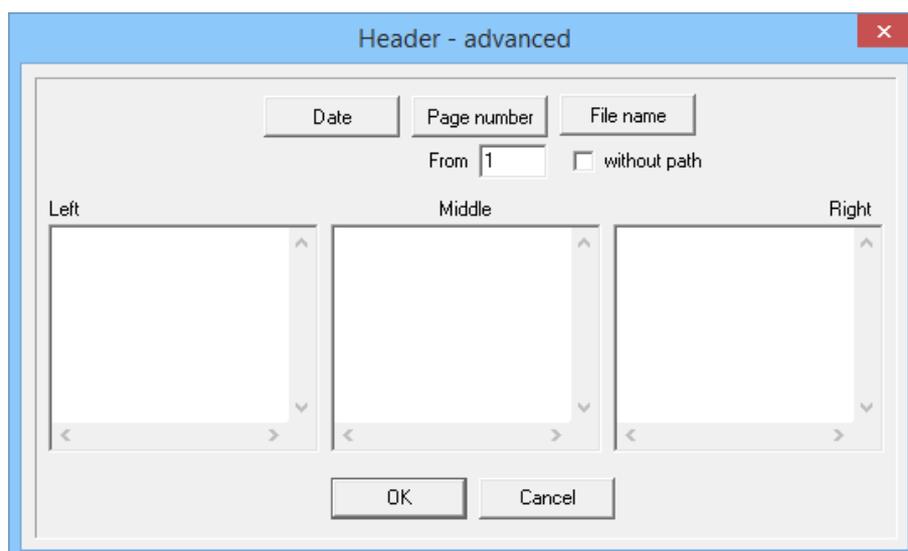
- o Indicate here which data should be printed and which shouldn't.

- The characteristics of the used material can be printed by selection the option 'Material properties'.
- There's also a button to set the header and footer. Click on it and this dialog shows up:



The pagenumbers, date, file name or a text chosen by the user can be used as header or footer.

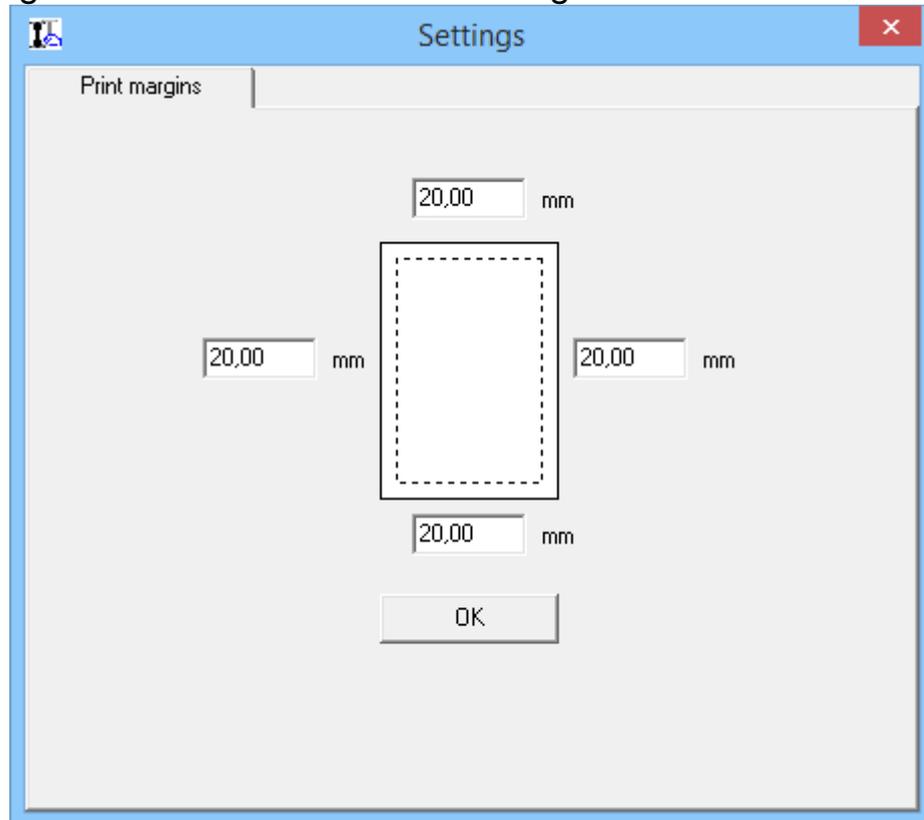
We can also make use of the advanced header and footer. This can consist of multiple rules. To do so, select the corresponding check box and click on 'Set up'. The following dialog window appears:



Now different lines can be entered.

- The date on which the note was printed can be added by clicking on the button 'Date'.

- Also the pagenumber can be inserted. The number of the first page can be adapted as well.
 - The file name, with or without path can also be added.
- The **margins** top, bottom, left and right can be entered by clicking on the 'Margins' button. The tab for the margins is as follows:



- Finally **the scale** should also be set.

4.6 Printing

We can print the image in the drawing field by selection the item 'File' – 'Print preview'. Then we get to see the print preview. On top of this window there's an icon of a printer to print the preview.

The drawing can also be printed directly by clicking on 'File' – 'Print' in the menu. Although it's recommended to check the print preview first.

It is also possible to print a calculation note.

The calculation note can be printed directly from the menu 'File' – 'Print' – 'Calculation note'. Also here it's recommended to take a print preview first.

On the next pages you'll find an example of a calculation note.

