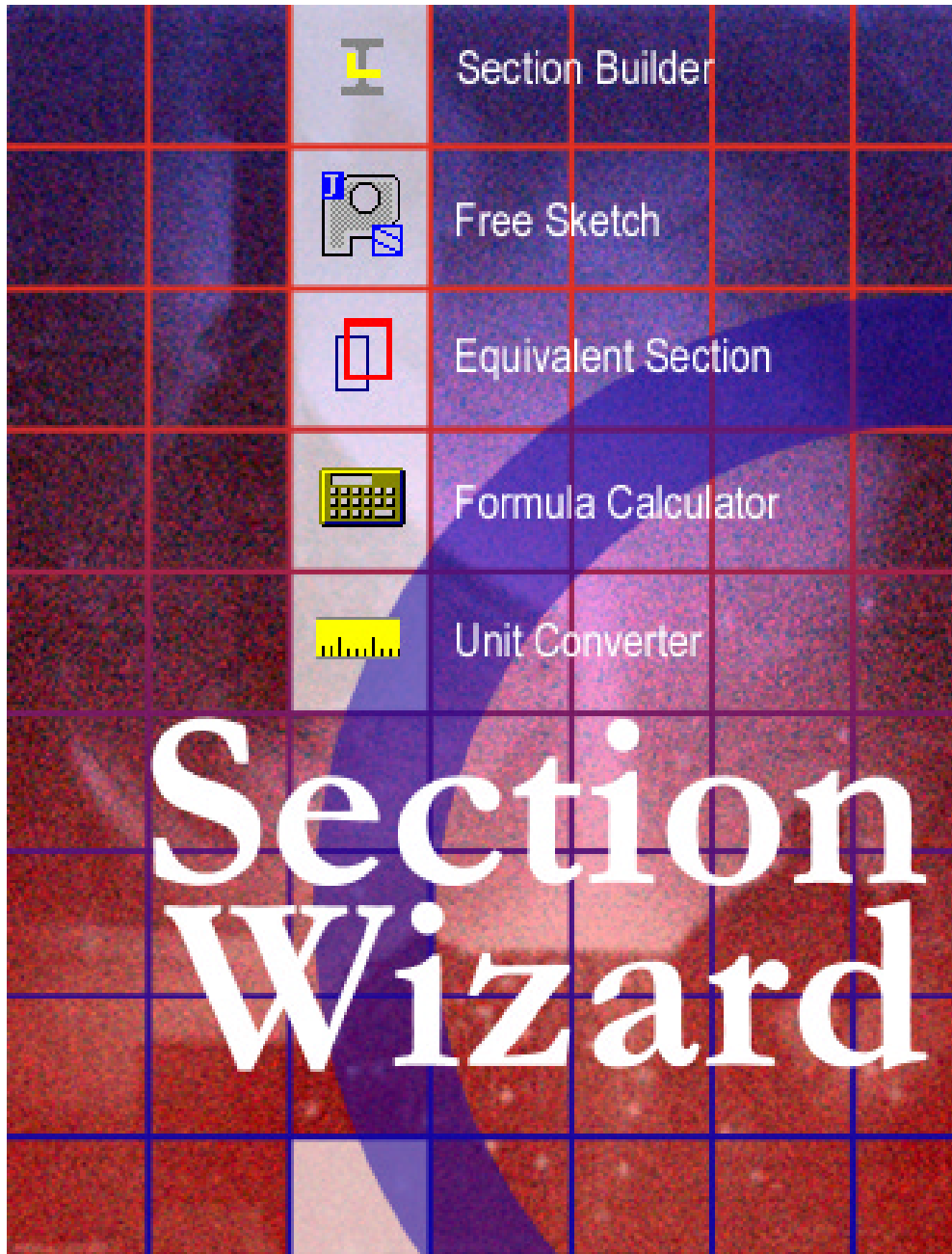


Section Wizard Tutorial



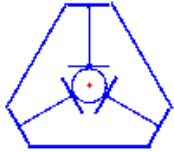
CONTENTS

Section Wizard Tutorial.....	1
Overview	3
Section Builder Tutorial	4
The section:-	4
The loading:-	4
Section Builder Tutorial - Steps	5
Free Sketch Tutorial	13
The section:-	13
The loading:-	13
Free Sketch Tutorial - Steps	14
Equivalent Section Tutorial.....	20

OVERVIEW

This help file contains a series of tutorials to help you become familiar with the use of the main *Section Wizard* modules.

There are tutorials for three main modules, select one to view:-



Section Builder

Create sections using standard shapes.



Free Sketch

Create sections using a drawn geometry.

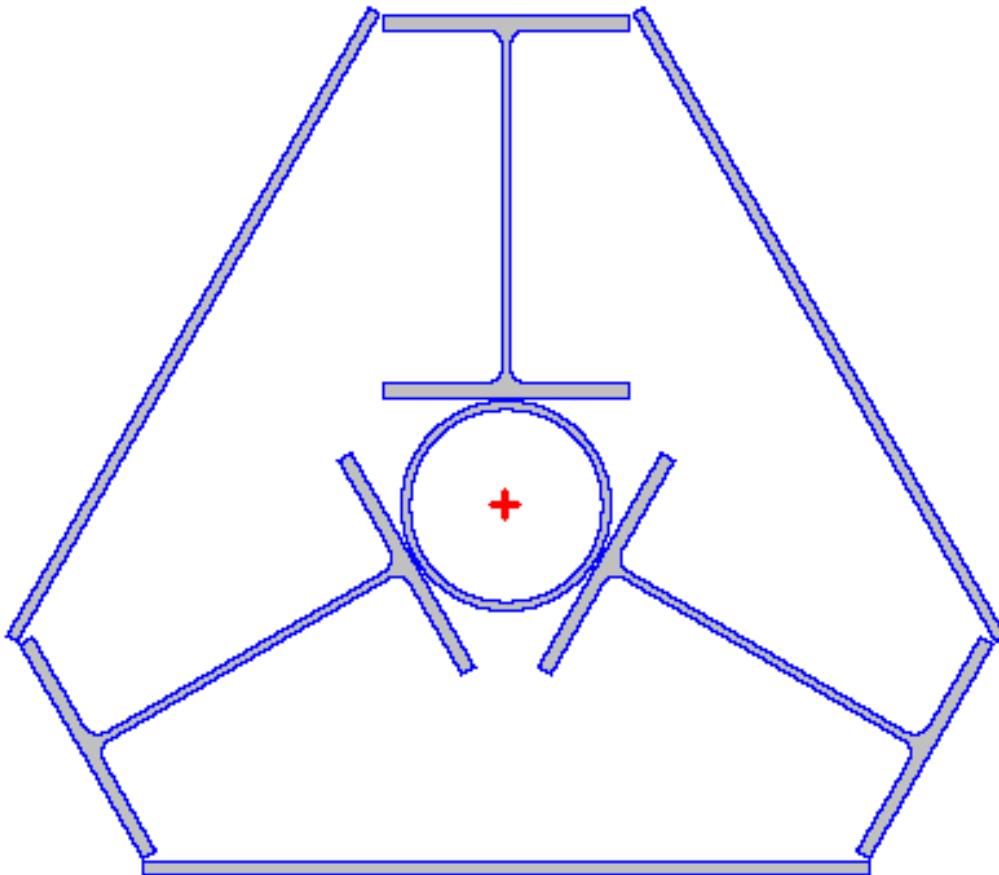
Equivalent Section (No tutorial in this version)

Note that the files used in these examples are installed in an Examples folder in your application installation folder.

SECTION BUILDER TUTORIAL

The following steps show a number of the facilities available in *Section Builder* to create a complex composite section from profiles from standard steel suppliers and establish the stress pattern under a given loading regime.

THE SECTION:-




THE LOADING:-

Bending about major principal axis,	$M_u = 200 \text{ kNm}$
Bending about minor principal axis,	$M_v = 0 \text{ kNm}$
Axial Load,	$N = 100 \text{ kN (100000 N)}$

SECTION BUILDER TUTORIAL - STEPS

- 1) Launch *Section Builder* from:-
Start>Programs>Research Engineers>Section Wizard menu.

- 2) If the screen contains data from a previous file, start a new file by either clicking the icon  or selecting the menu option:-
File>New

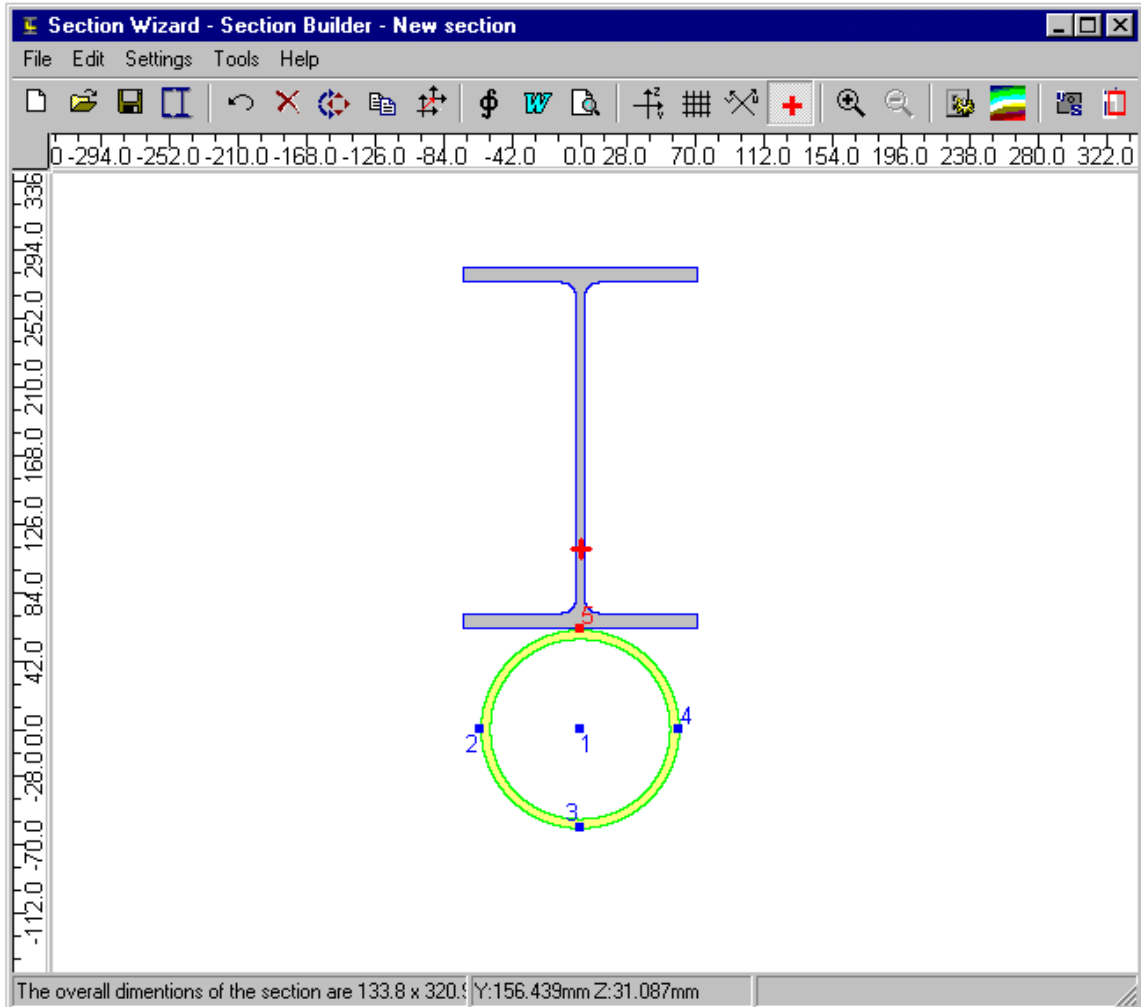
- 3) There are two parts to the Section Builder module:-
Section Element
Main Section Builder Window
The first step is to select a section from the **Section Element** window. Click on the Select Profile button.

- 4) Section Builder displays a list of sections from a number of available databases. Form the British Standard Sections, select the Circular Hollow Sections database. From the available sections select:-
 114x6.3 CHS
Click on the OK button to accept your selection.

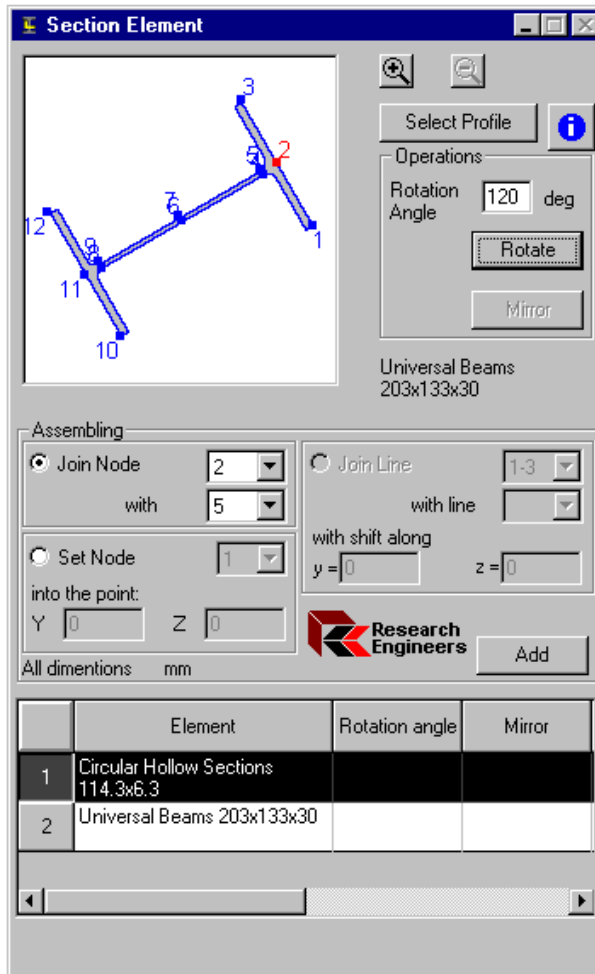
- 5) This section now has to be added into the composite section.
Simply click on the Add button.

- 6) To add the first of the I sections onto the composite section, click on the Select Profile button. Once again select the British Standard Sections, but this time select the Universal Beams database. From the available list select:-
 203x133x30 UB
Click on the OK button to accept your selection.

- 7) To add the I section onto the composite section the appropriate assembling method needs to be selected. The first section to be added will be the top I section. This is to be joined to the top of the CHS. Select the Assembling option **Join Node**. The first node is from the I section. Set this to 2, noting that it is coloured red on the preview window. The second node is from the CHS. Set this to 5, noting that it is coloured red on the main Section Builder window. To combine the sections, click on the Add button.



- 8) We now need to add the same section on again, but rotating it through 120 degrees. Below the Select Profiles button, set a rotation angle of 120 degrees and click on the Rotate button. The preview window should show:-



- 9) This time we need to assemble the section onto the composite section using the **Set Node** option. Select this option and set the node number to 2, noting that it becomes coloured red on the preview window. Set the co-ordinate that it is to link to as:-

$$Y = -49.40675$$

$$Z = -28.525$$

Then click on the Add button.

- 10) To add the final I section onto the composite section, change the rotation angle of the selected I section to 240 degrees and click on the Rotate button.

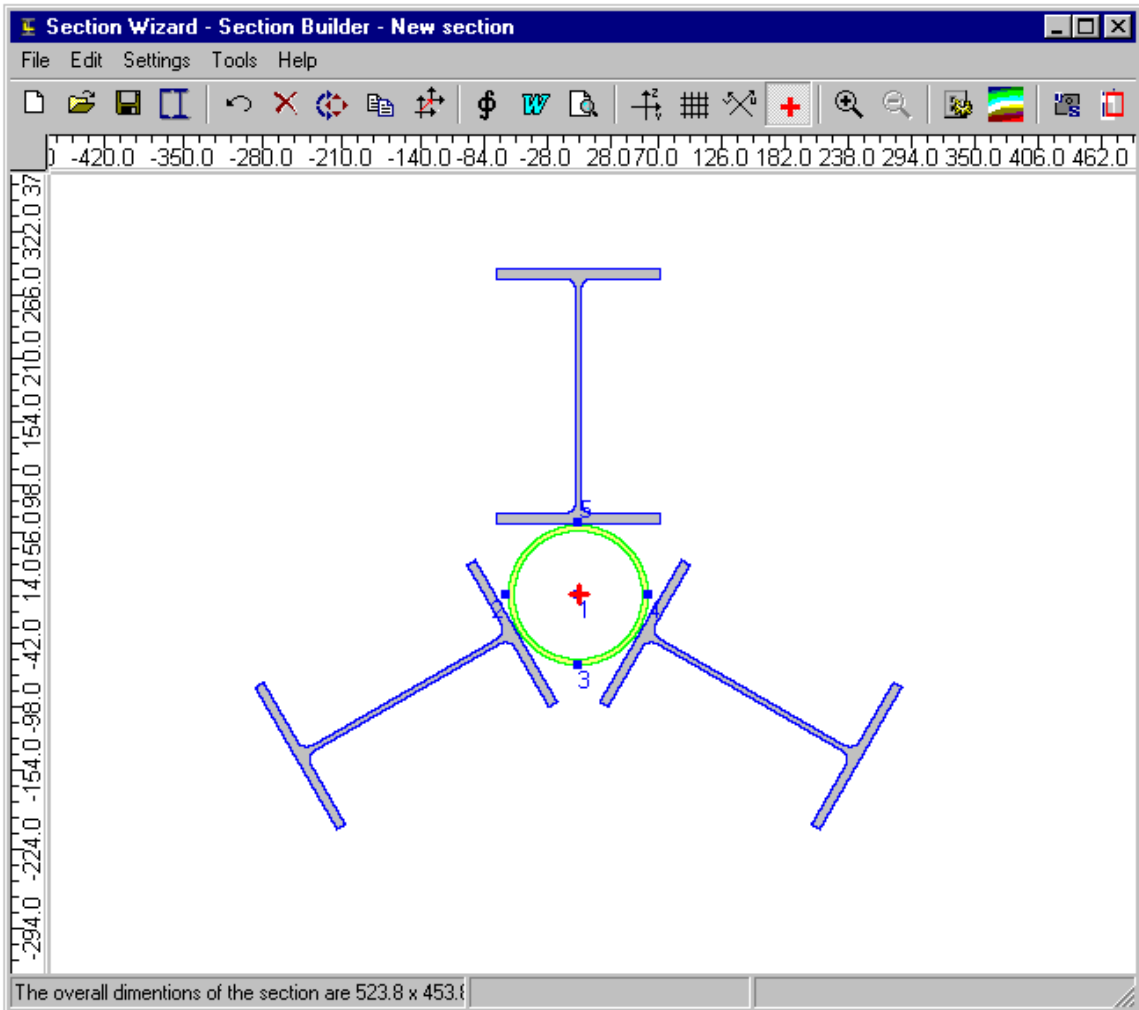
Maintain the option to Set Node with the selected node as node number 2, but change the co-ordinate to :-

$$Y = 49.40675$$

$$Z = -28.525$$

Then click on the Add button.

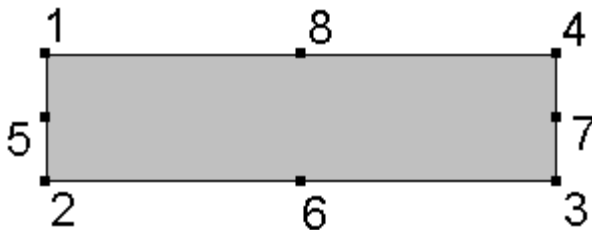
11) The composite section should now look like:-



12) To add the plate sections, click on the Select Profile button. This time select the option to define a plate. Enter the values:-

- Plate Thickness 80 mm
- Plate Width 390.0 mm
- Material Steel

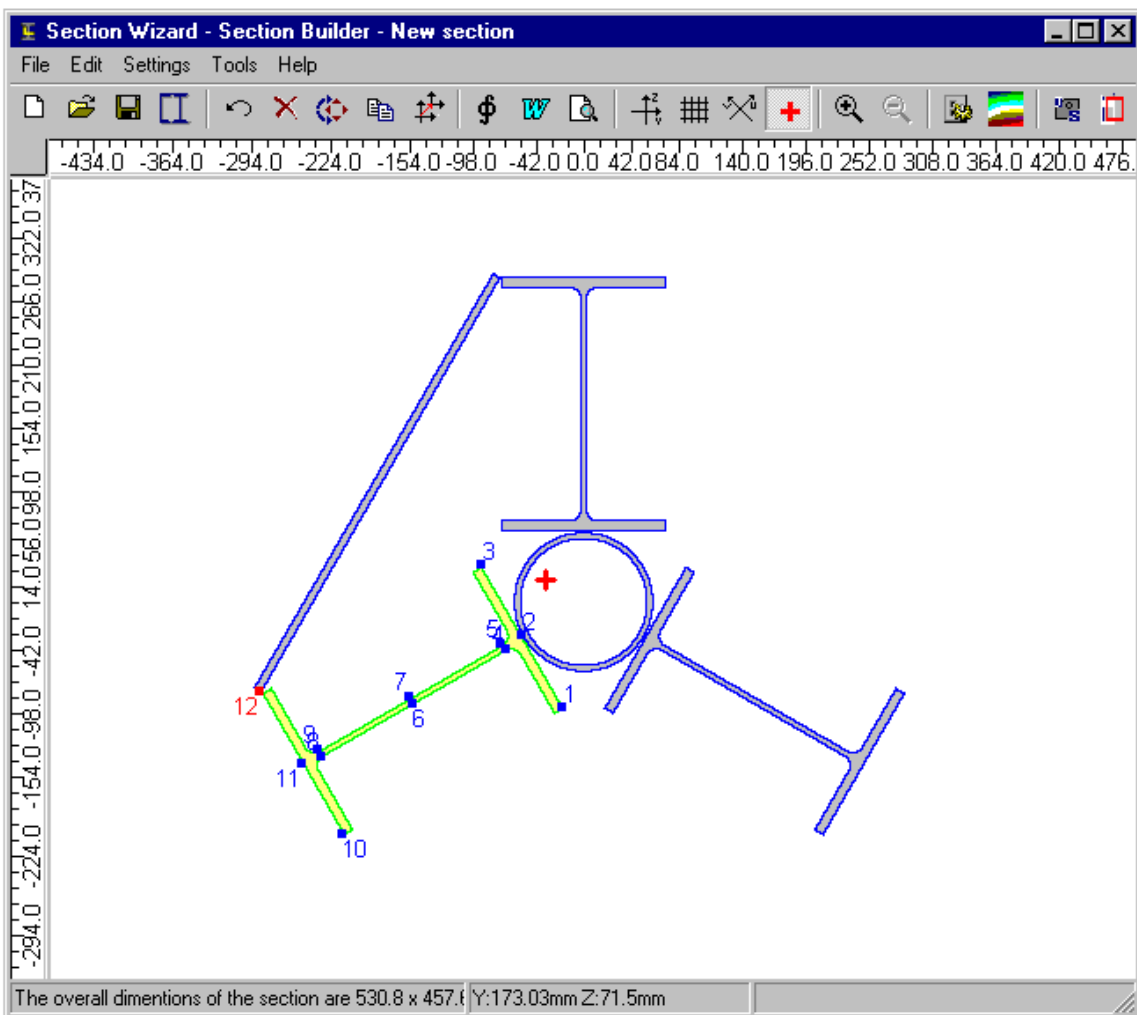
Then click on the OK button. Note that as the plate is rather thin, the node numbers overprint. Recall that the nodes on a plate are thus:-



Rotate the plate by 240 degrees.

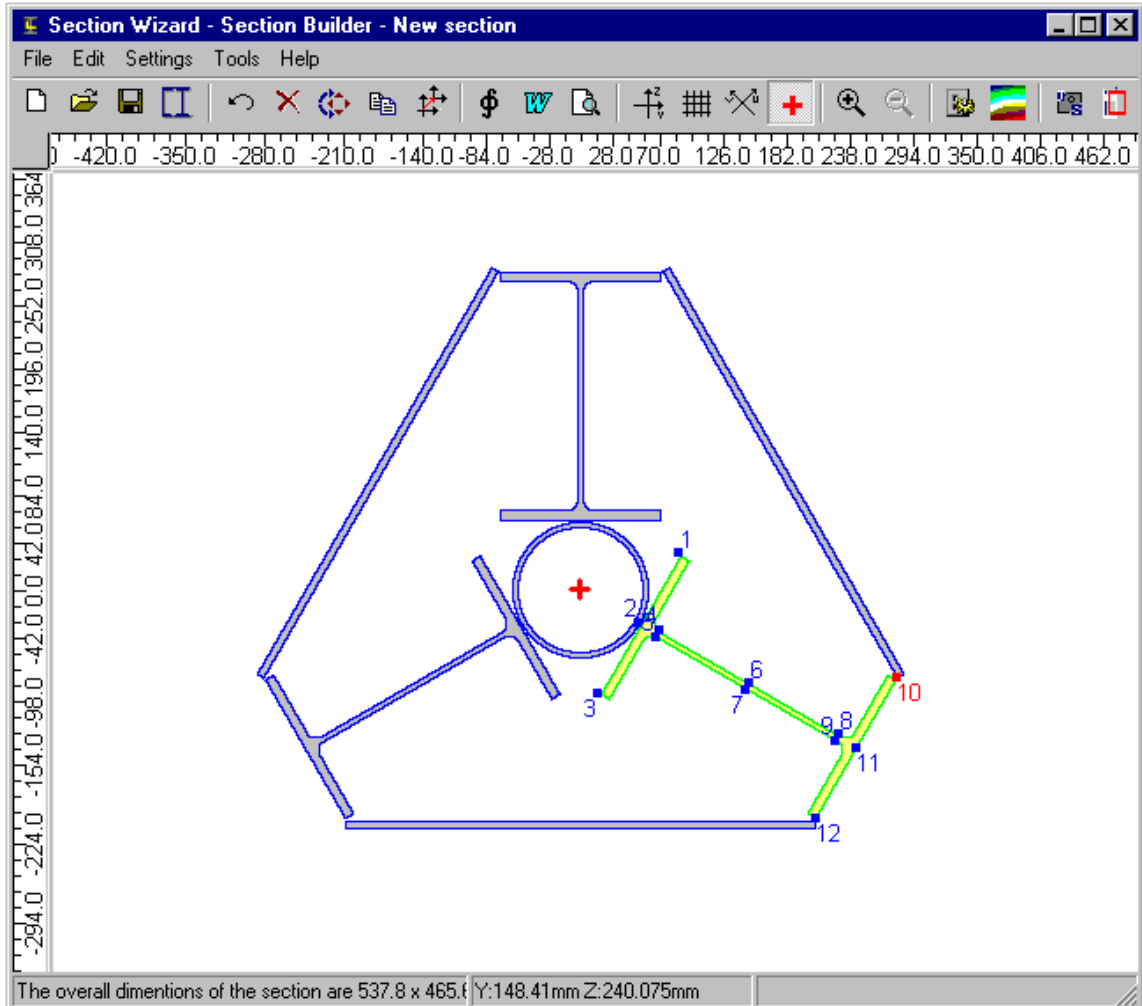
To combine this plate into the composite, in the Assembling option, select the **Join Node** option. As we now have a number of sections already in our composite section, then we need to select the section that we want to connect to. Do this by either clicking on the lower left I section in the main Section Builder Window or in the Section Element window click on element 2 in the list of section at the bottom of the dialog box. Either way, the I section in the lower left corner of the composite section should become coloured yellow and indicate the locations of its node points.

- 13) Add the plate section into the composite by setting Node 4 on the plate to combine with Node 12 on the I section and click on the Add button. The composite section should now look like:-



- 14) Rotate the plate to 0.0 degrees and add it onto the composite section by selecting Node 1 on the plate to connect to Node 10 on the lower left I section and click on the Add button.


- 15) For the final plate addition. Select the lower right I section using either method in item 12 above. Rotate the plate to 120 degrees and join Node 1 on the plate with Node 10 on the lower right I section. The final composite section should look like this:-



- 16) Save the section by selecting the menu option:-
File>Save
and calling the file Tutorial 1.

17) To get a stress plot for the given loading regime select the menu option:-

Tools>Stress Contour

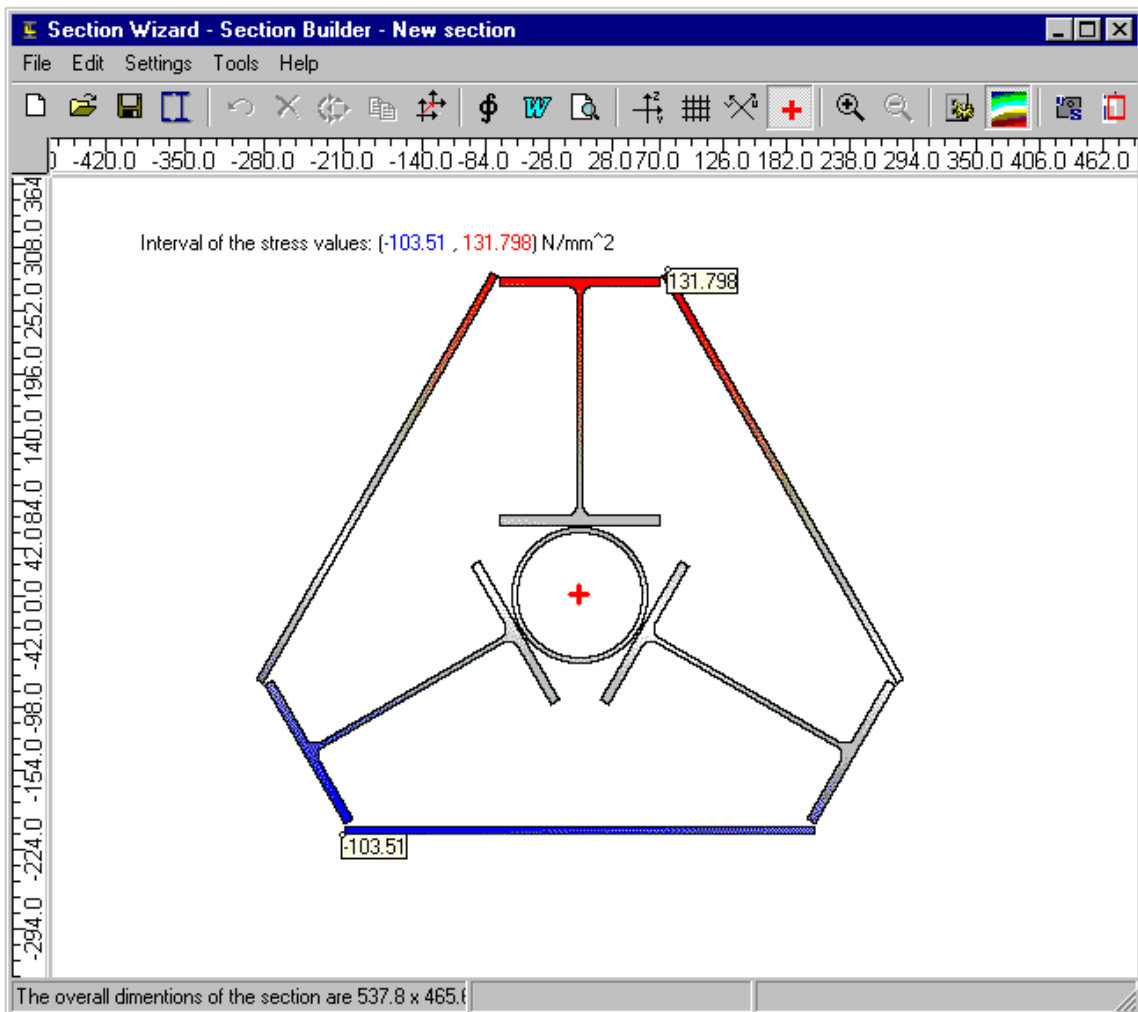
or click on the  icon. Enter the following values of applied force into the dialog box and clicking on the OK button:-


$$M_u = 200 \text{ kNm}$$

$$M_v = 0 \text{ kNm}$$

$$N = 100 \text{ kN}$$

You should then get the following picture.



- 18) To View the values of this composite section click on the  icon. This should show you values similar to the following:-

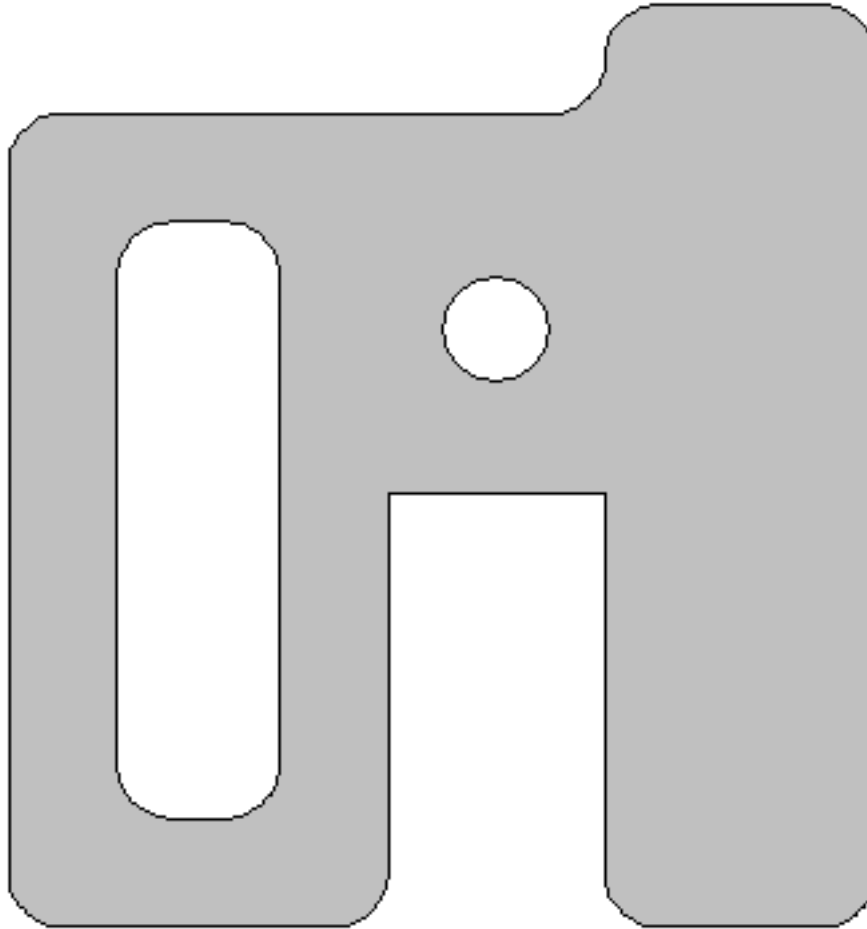
Basic geometry (reduced)			
Parameter	Value	Units	
A	Sectional area	229.608	cm ²
α	Angle of principal inertia axes	-12.241	deg
I_y	Inertia moment about centroidal Y1-axis parallel with Y-axis	43529.161	cm ⁴
I_z	Inertia moment about centroidal Z1-axis parallel with Z-axis	43522.303	cm ⁴
i_y	Radius of inertia about Y1-axis	13.769	cm
i_z	Radius of inertia about Z1-axis	13.768	cm
W_{u+}	Maximum resisting moment about U-axis	1854.163	cm ³
W_{u-}	Minimum resisting moment about U-axis	1569.329	cm ³
W_{v+}	Maximum resisting moment about V-axis	1755.242	cm ³
W_{v-}	Minimum resisting moment about V-axis	1567.647	cm ³
I_u	Maximum inertia moment	43529.499	cm ⁴
I_v	Minimum inertia moment	43521.965	cm ⁴
i_u	Maximum radius of inertia	13.769	cm
i_v	Minimum radius of inertia	13.768	cm
a_{u+}	Radius of gyration along positive direction of Y(U)-axis	8.075	cm
a_{u-}	Radius of gyration along negative direction of Y(U)-axis	6.835	cm
a_{v+}	Radius of gyration along positive direction of Z(V)-axis	7.645	cm
a_{v-}	Radius of gyration along negative direction of Z(V)-axis	6.827	cm
y_m	Coordinate of the center of gravity along Y-axis	-0.000350659	cm
z_m	Coordinate of the center of gravity along Z-axis	0.001	cm

This concludes Tutorial 1

FREE SKETCH TUTORIAL

The following steps show a number of the facilities available in *Free Sketch* to create a complex composite section and establish the stress pattern under a given loading regime.

THE SECTION:-




THE LOADING:-


Bending about major


principal axis	$M_u = 200 \text{ kNm}$
Bending about minor principal axis	$M_v = 0 \text{ kNm}$
Axial Load	$N = 100 \text{ kN}$


FREE SKETCH TUTORIAL - STEPS


- 1) Launch *Free Sketch* from:-
Start>Programs>Research Engineers>Section Wizard menu.

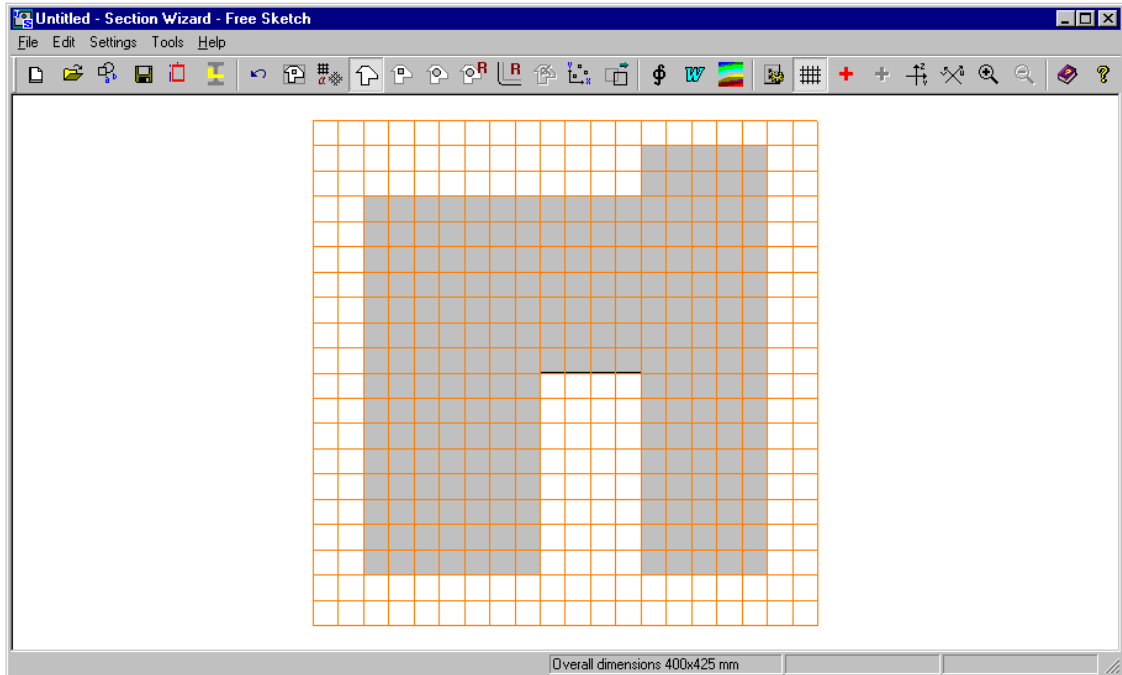
- 2) If the screen contains data from a previous file, start a new file by either clicking the icon  or selecting the menu option:-
File>New

- 3) Define the size of the workspace that will be required. Click on the Overall Dimensions button  and set the size to 500 mm x 500 mm



- 4) The next stage is to set a snapable grid of 25 mm in either direction that is aligned to the boundary. To do this, first click on the Grid Settings button  and set the Y and Z values to 25mm and the angle to 0 degrees.

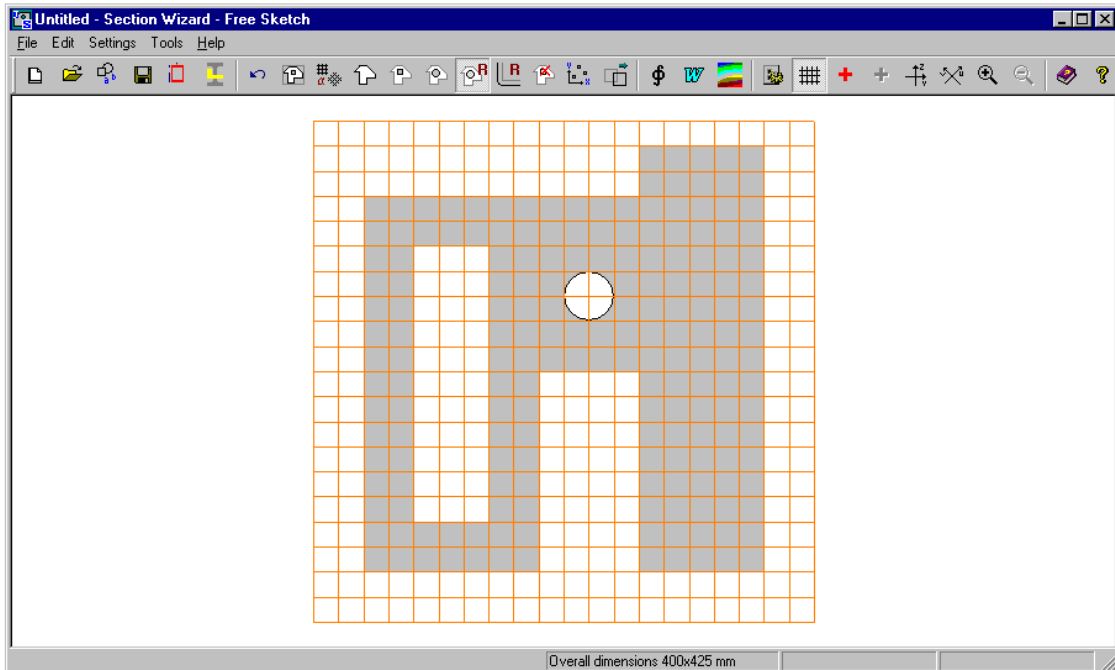
To ensure that the snap is turned on, click on the Preferences button  and turn to the Misc tab. In the group box titled 'Other' ensure that the option 'Snap to Grid' is turned on. Click in the check box to display a tick if necessary.


- 5) To define the boundary click on the External Boundary button , then click on the grid 2 units to the right and 2 units up from the bottom left corner of the workspace. Continue to click around the workspace either clockwise or anti-clockwise on the grid locations where the boundary is to change direction. Notice how the area is defined with each additional click. To complete the boundary double click on the last point. The screen should look somewhat like this:-




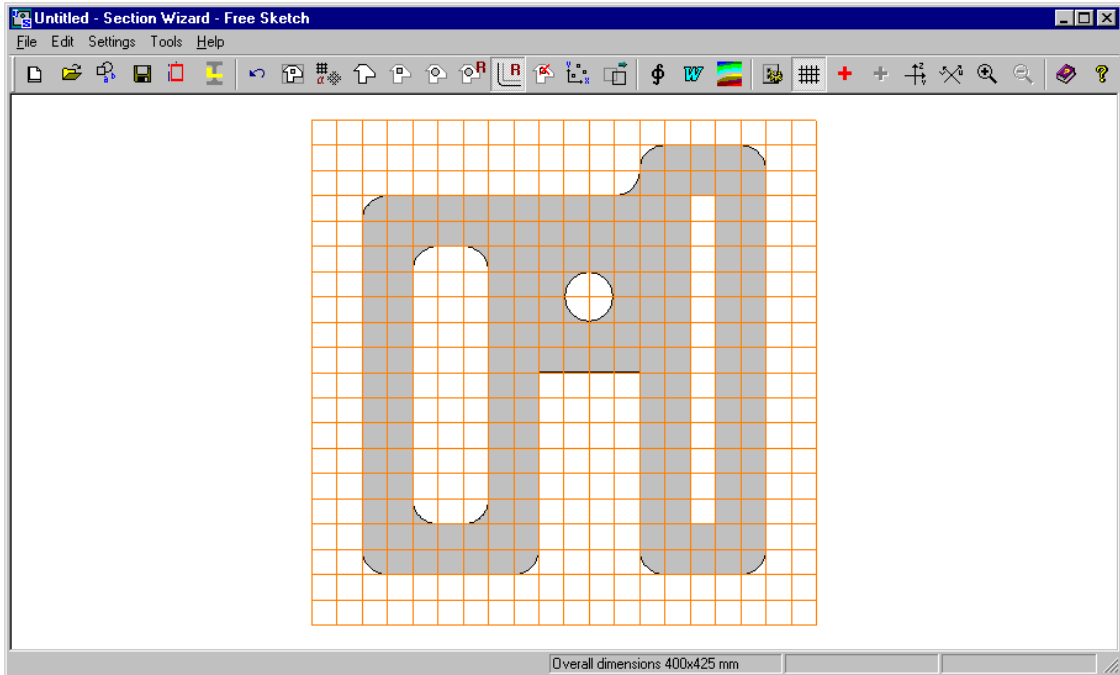
Do not worry if you have created a node in the wrong location. To edit the boundary, simply move the cursor over the boundary outline and note when the cursor changes to a double square. Clicking here will move the selected node to a new location and will continue to add further nodes into the boundary until you double click once again.

- 6) There are two types of internal boundaries that can be defined, polygonal and circular. The polygonal holes are created similar to the external boundary, clicking systematically either in a clockwise or anti-clockwise direction. Form the rectangular opening by clicking on the Polygon Hole button . Form the circular hole by clicking on the Create Circular Hole with Fixed Radius button , set the radius to 25mm and click on the grid intersection to give the following diagram:-




If the circular hole is located in the wrong place, click on the Delete Hole button  and click within the circle. Click on the Create Circular Hole with Fixed radius and position it in the correct location. Note that all boundaries are independent and whilst they may touch, they must not cross one another.

- 7) Finally to smooth the edges of the external and polygonal internal click on the Smooth button . Set the radius to 25mm and click on the corners of the boundaries to create the completed section:-



- 8) To get a stress plot for the given loading regime select the menu option:-
Tools>Stress Contour

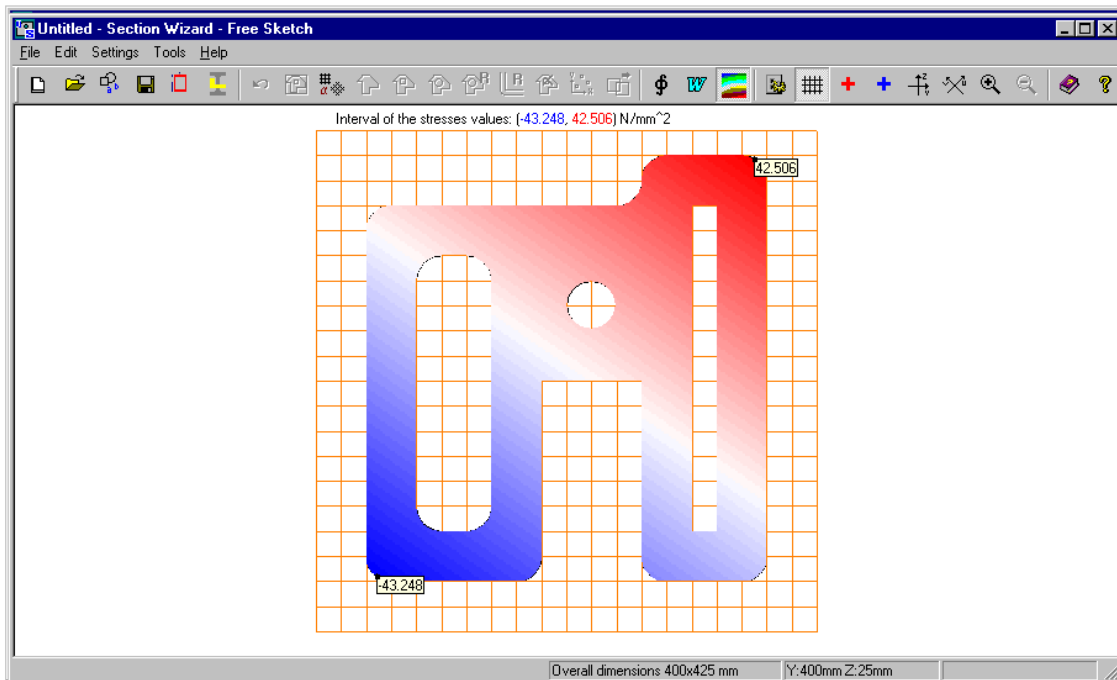
or click on the  icon. Enter the following values of applied force into the dialog box and clicking on the OK button:-


$$M_u = 200 \text{ kNm}$$

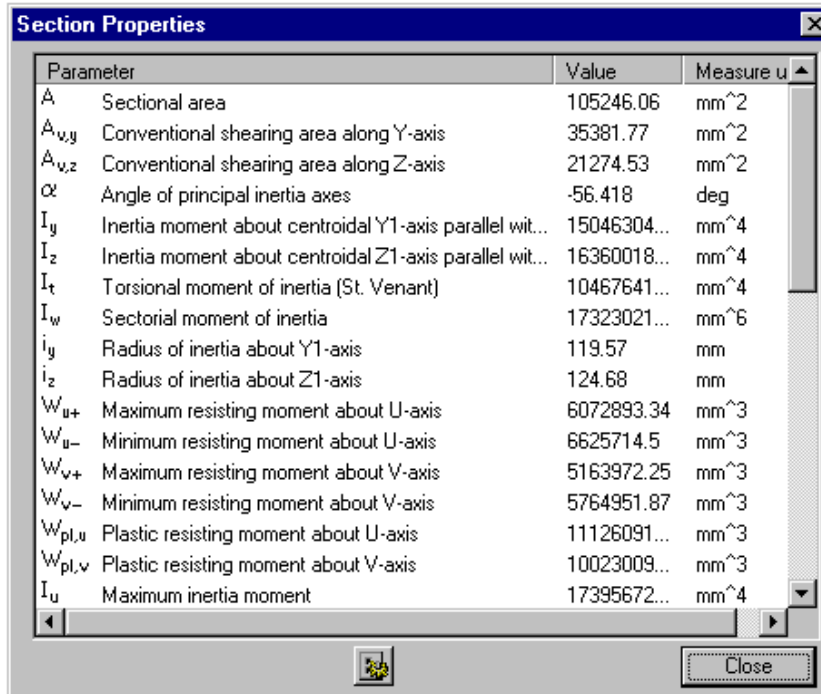
$$M_v = 1000 \text{ kNm}$$

$$N = 100000 \text{ N}$$

You should then get the following picture.



- 9) To View the values of this composite section click on the  icon. Enter the poissons ratio (ν) of 3 and click on the OK button. This should show you values similar to the following:-



Parameter	Value	Measure u
A	Sectional area	105246.06 mm ²
A _{v,y}	Conventional shearing area along Y-axis	35381.77 mm ²
A _{v,z}	Conventional shearing area along Z-axis	21274.53 mm ²
α	Angle of principal inertia axes	-56.418 deg
I _y	Inertia moment about centroidal Y1-axis parallel wit...	15046304... mm ⁴
I _z	Inertia moment about centroidal Z1-axis parallel wit...	16360018... mm ⁴
I _t	Torsional moment of inertia (St. Venant)	10467641... mm ⁴
I _w	Sectorial moment of inertia	17323021... mm ⁶
i _y	Radius of inertia about Y1-axis	119.57 mm
i _z	Radius of inertia about Z1-axis	124.68 mm
W _{u+}	Maximum resisting moment about U-axis	6072893.34 mm ³
W _{u-}	Minimum resisting moment about U-axis	6625714.5 mm ³
W _{v+}	Maximum resisting moment about V-axis	5163972.25 mm ³
W _{v-}	Minimum resisting moment about V-axis	5764951.87 mm ³
W _{pl,u}	Plastic resisting moment about U-axis	11126091... mm ³
W _{pl,v}	Plastic resisting moment about V-axis	10023009... mm ³
I _u	Maximum inertia moment	17395672... mm ⁴

This concludes Tutorial 2.

EQUIVALENT SECTION TUTORIAL

This tutorial has not been included with this version of the program.