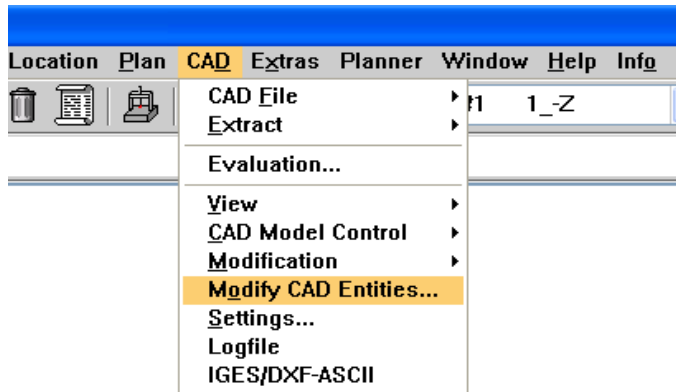


# Working with CAD

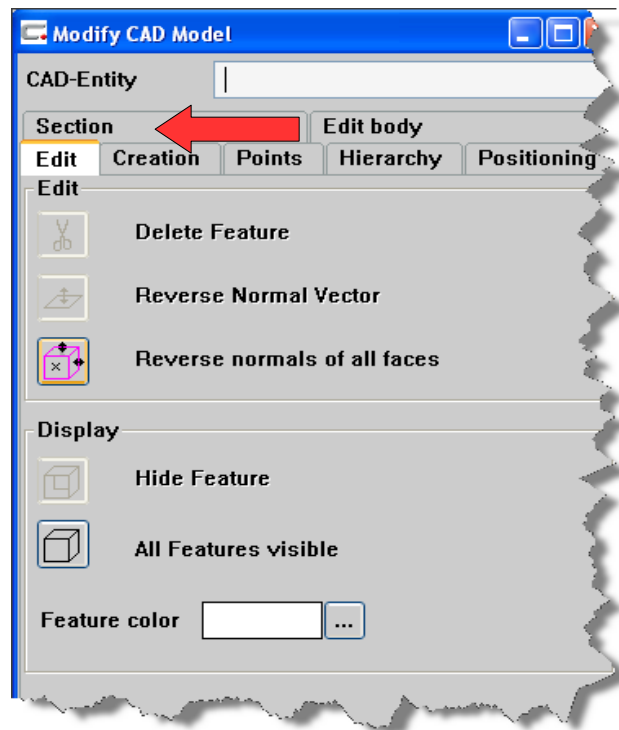
Exploring some of the seldom covered CAD options found within Calypso.

By. Phil Adair



Within Calypso's CAD option, there are several functions available to you which may be useful and are only normally covered when a “curve” is being created. However there may be aspects of it that are useful and of interest to everybody.

To access the functions, select CAD from the main menu and then select “Modify CAD Entities”.

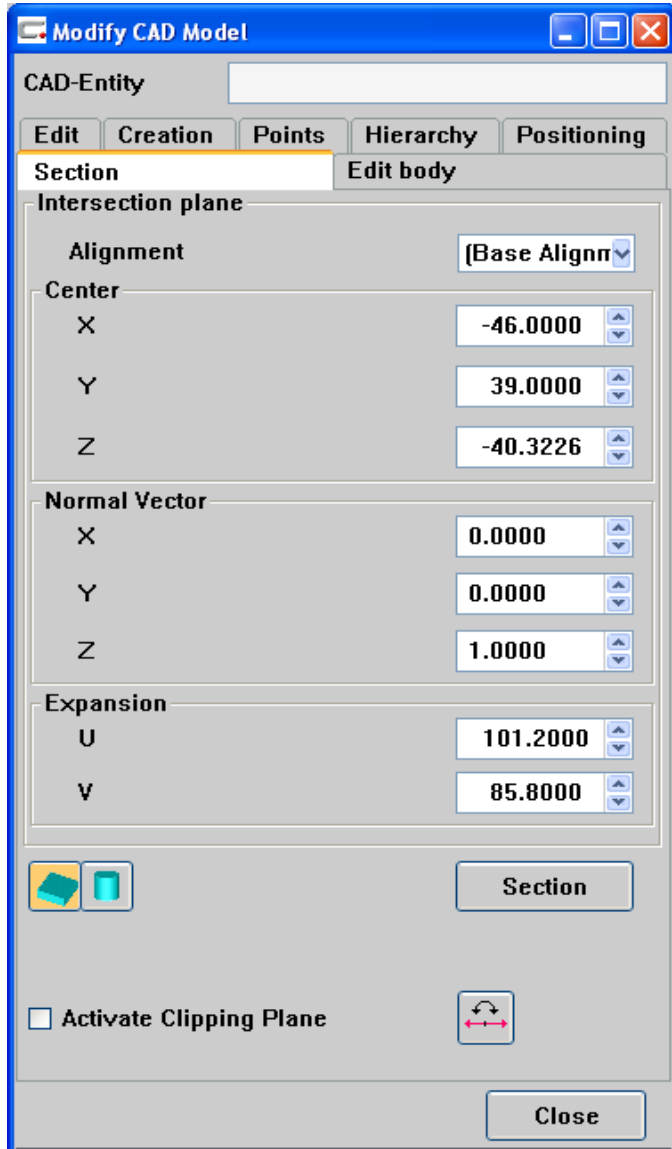


This will display the menu that allows control of various elements of the CAD.

The option that we are interested in is contained within the “Section” tab.

Click on the “Section” tab to open it.

# Working with CAD

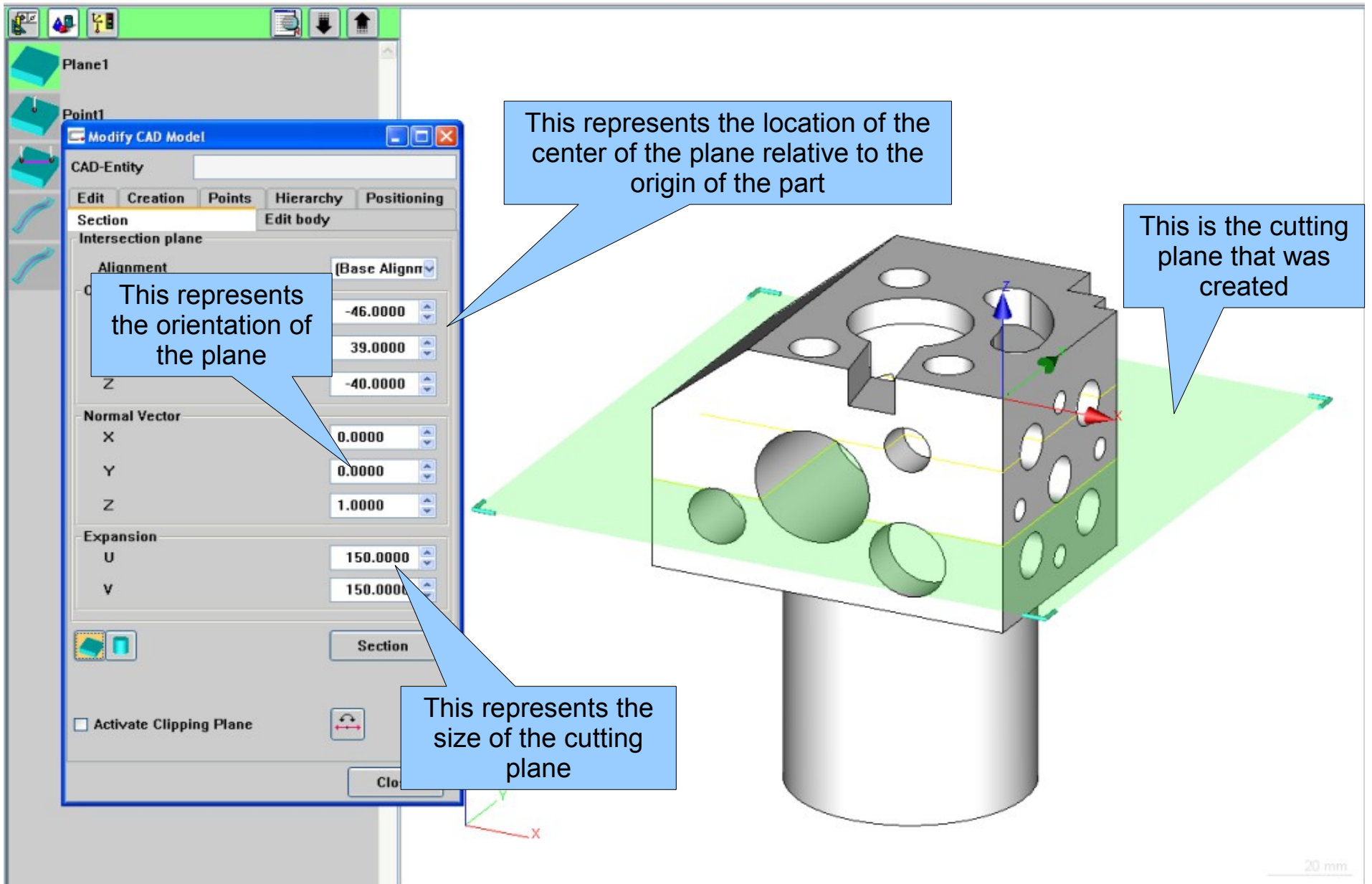


The “Section” option allows you to take a CAD model and create a plane that becomes a slice through the part.

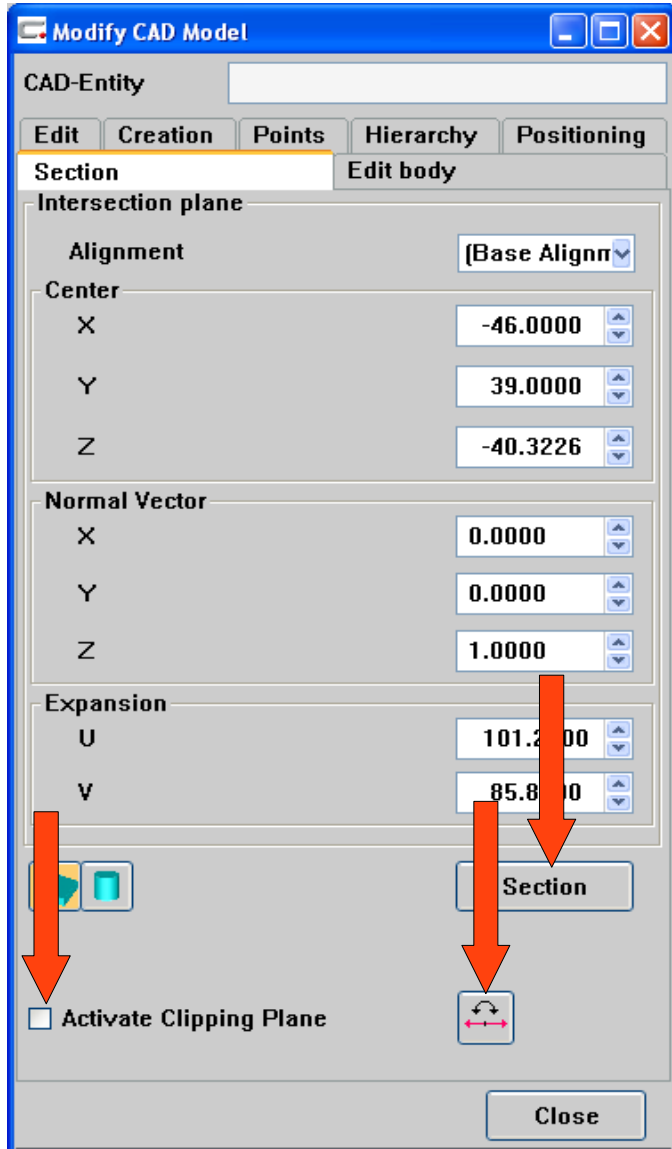
The location of the slice is controlled by the “Center” co-ordinates. This defines the location of the center of the plane relative to the origin of the part.

The Normal vector defines the orientation of the plane, remember that it is a slice that we wish to take through the CAD model. The vector could also be thought of as the normal vector to the slice.

The “Expansion” option is the size of the slice, in general examples the slice is bigger than the part, ensuring that all components of the part are included.



# Working with CAD



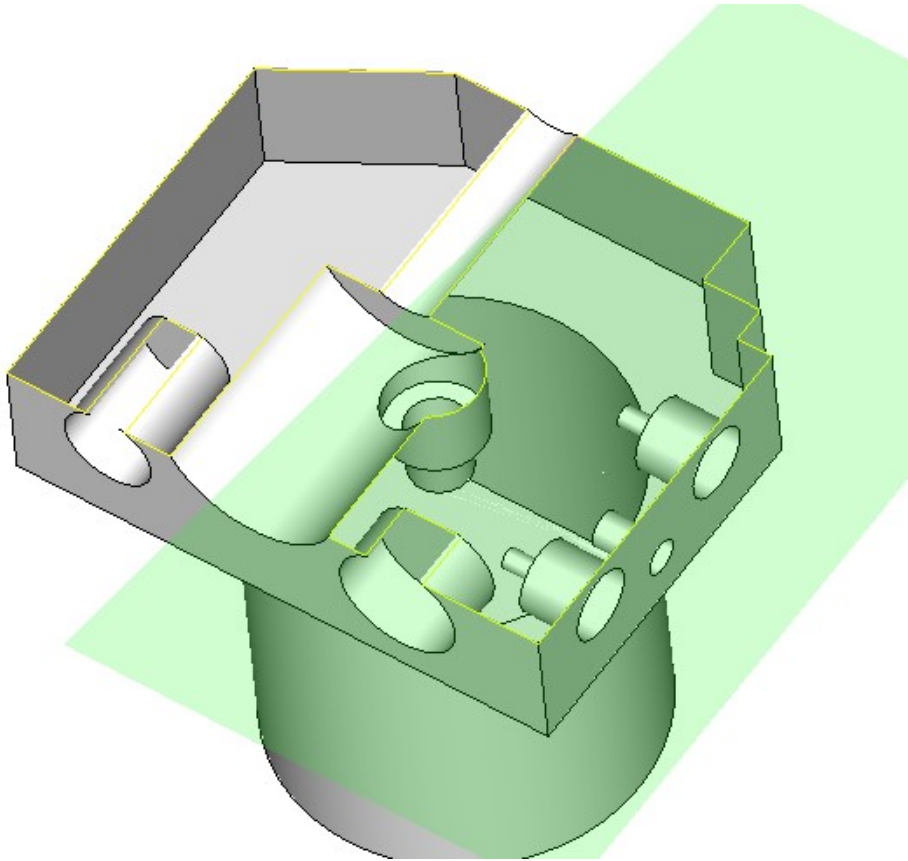
If we click on the “Section” button, a slice is made through the part at the location of the slice we created.

This slice is the fundamental method of creating a curve using this method but it also gives us access to a tool that could be useful for other users.

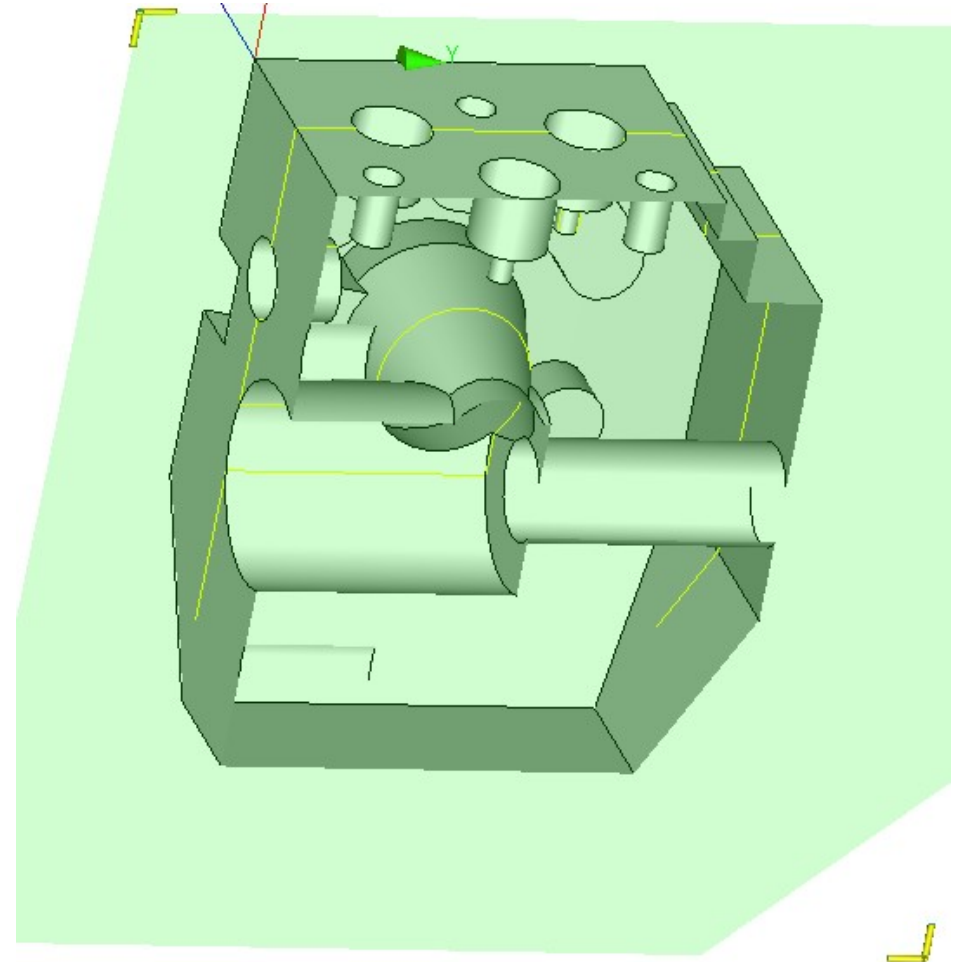
If we put a check in the “Activate Clipping Plane” box, what happens is that visually the top is taken off the part (see next page for view).

If we “flip” the view it replaces the view that was omitted with a view from the opposite direction. (see next page for view)

# Working with CAD



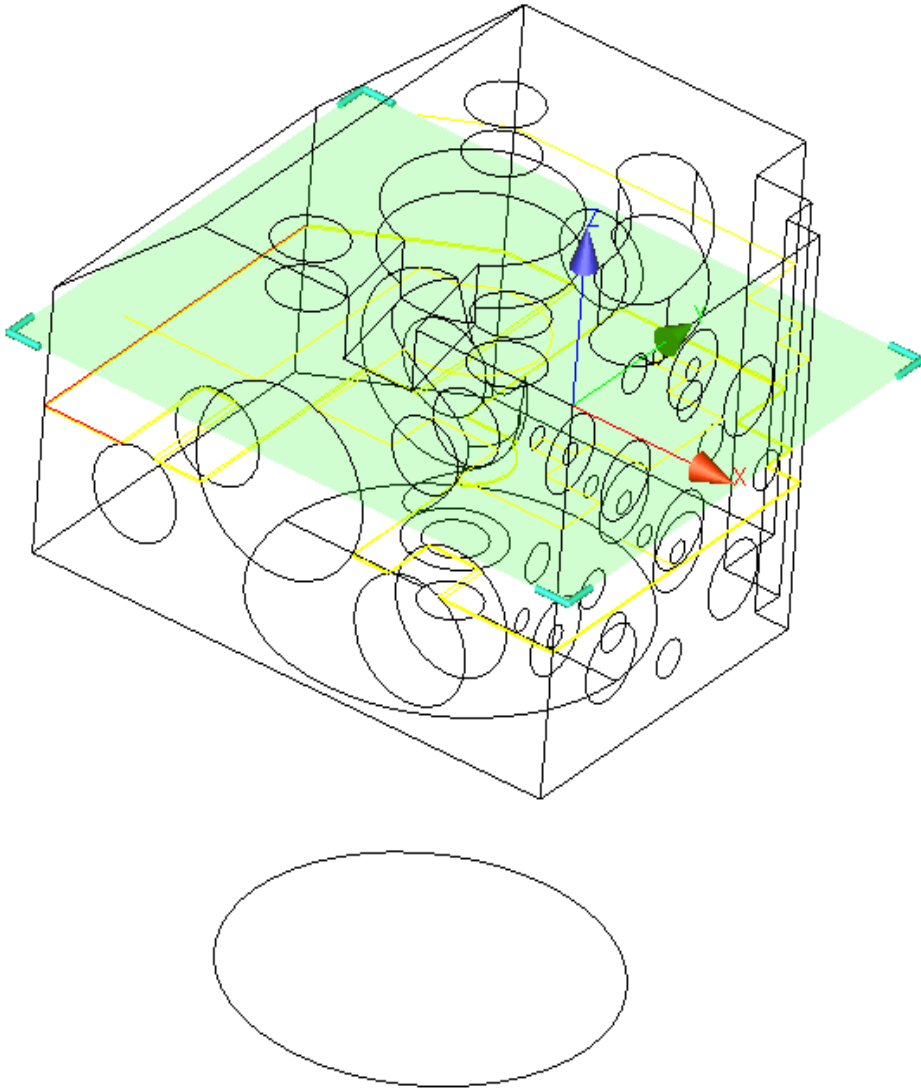
This is prior to flipping



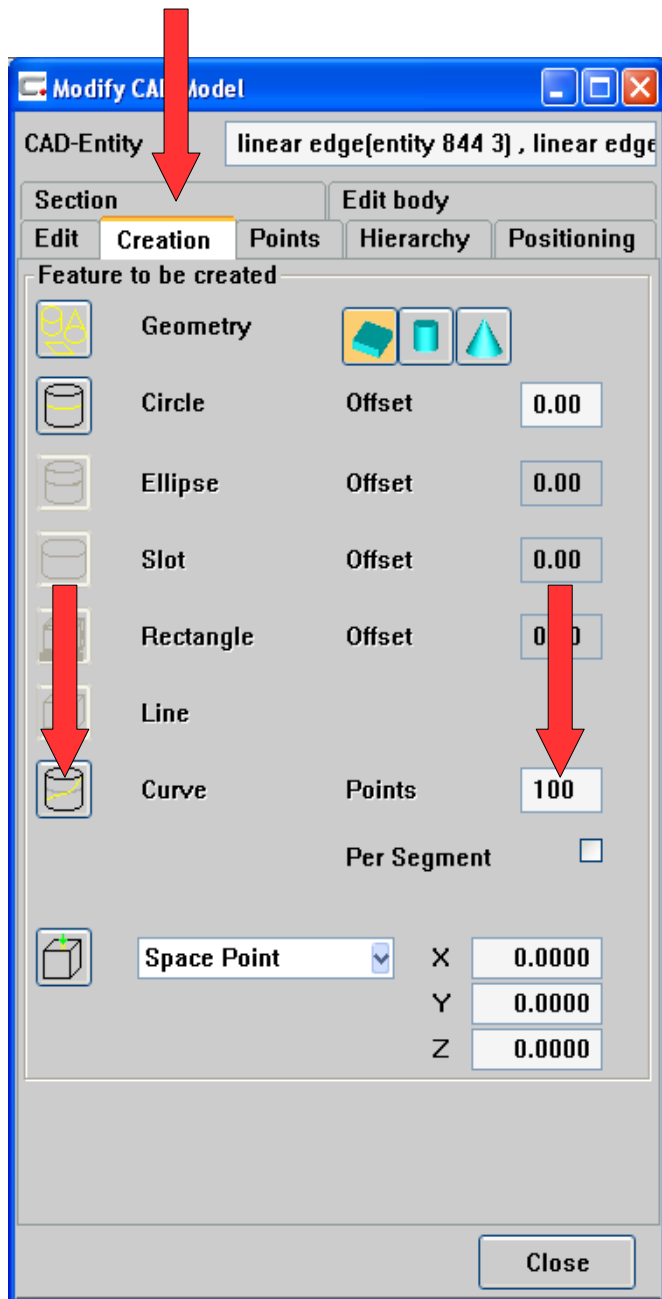
This is after flipping

# Working with CAD

This capability makes it very useful for capturing images for reports or just seeing what the part looked like internally for wall sections, depth of holes etc.



# Working with CAD



For those of you with “Curve” you have one more step to create a curve.

After you have created the slice, go to wireframe mode and then select the aspects of the slice that you wished to create a curve from.

If there are more than one line, hold down Ctrl. and click on subsequent lines.

Once the lines have been selected, go to the “Creation” tab, select the number of points, and click on “Curve”, a 2D Curve is created in the Feature List.