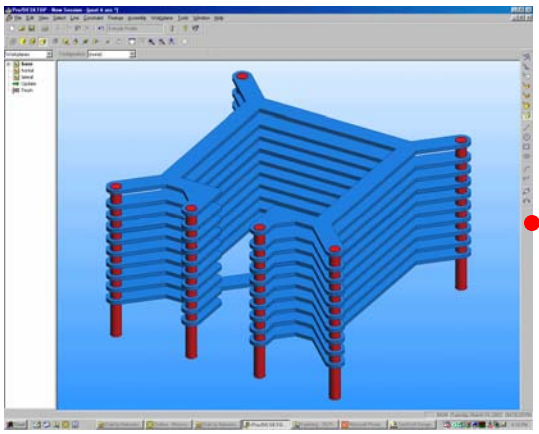
Pro/DESKTOP CAD/CAM Task

This section assumes the CNC manufacturing machine is the TEP milling machine, although the process is relevant to a wider range of machines.

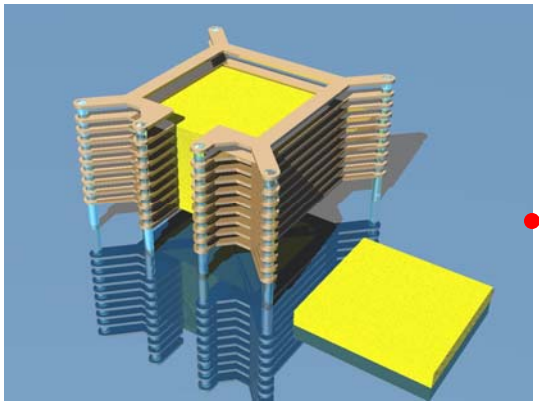
This CAD/CAM tutorial assumes some prior knowledge of Pro/DESKTOP to complete the design. A tutorial for novice users of Pro/DESKTOP called Architectural Solutions to CAD/CAM is available from the TEP website www.tep.org.uk.

Conventions: This tutorial uses abbreviations to explain protocol and functions within Pro/DESKTOP.

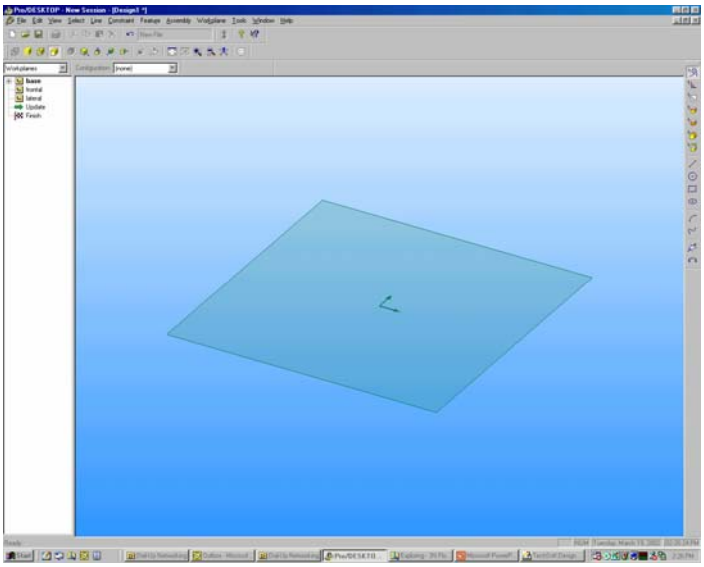
RC = Right Click
LC = left click
Tip: = short cut or quick key function



Post it holder target Design



Open a Design file in Pro/DESKTOP

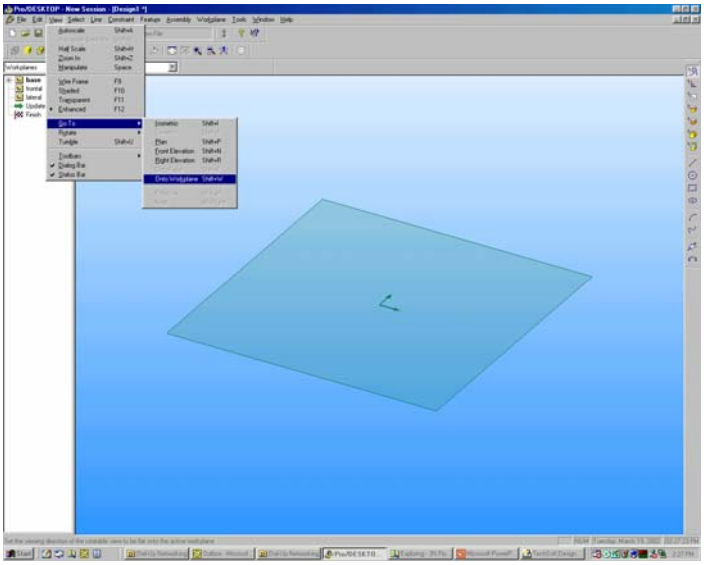


From the pull down menu select:

View - Go To – Onto Workplane

This will change the orientation of the Workplane so it appears flat to the screen.

Tip: A quick key option is Shift + W



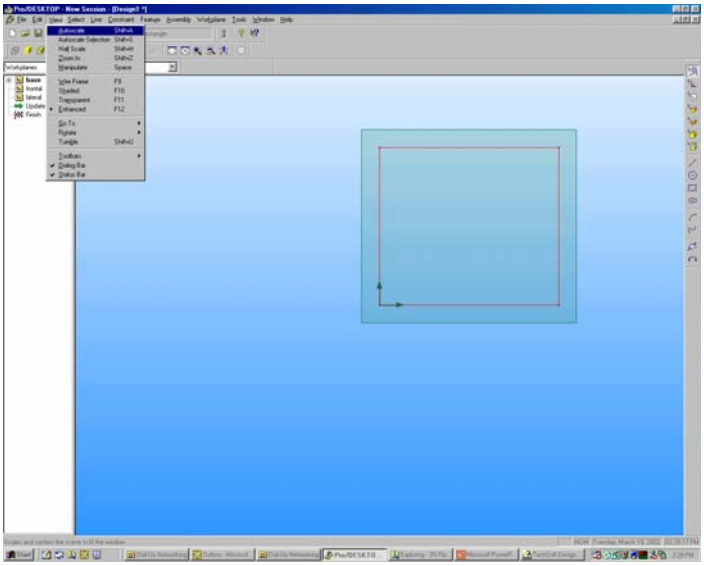
Draw a rectangle from the intersection of the axis at 0,0.

From the pull down menu select:

View – Autoscale

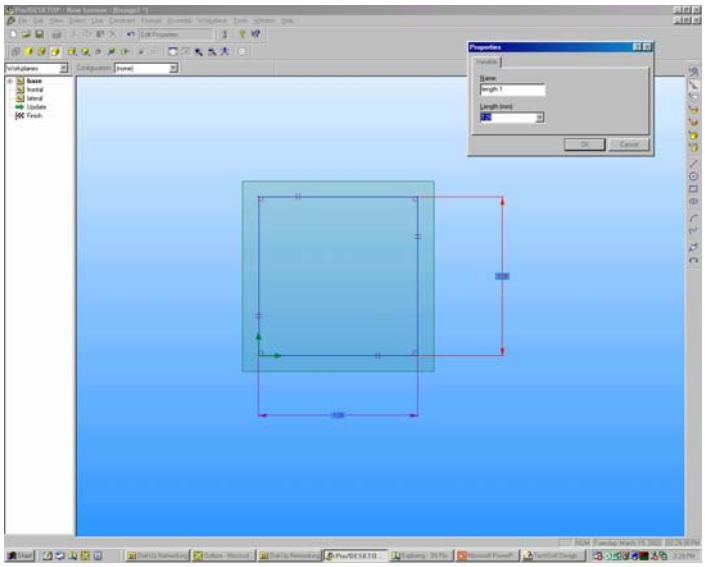
This will centralize the image on screen.

Tip: A quick key option is Shift + A
and Shift + H to reduce the image by half.

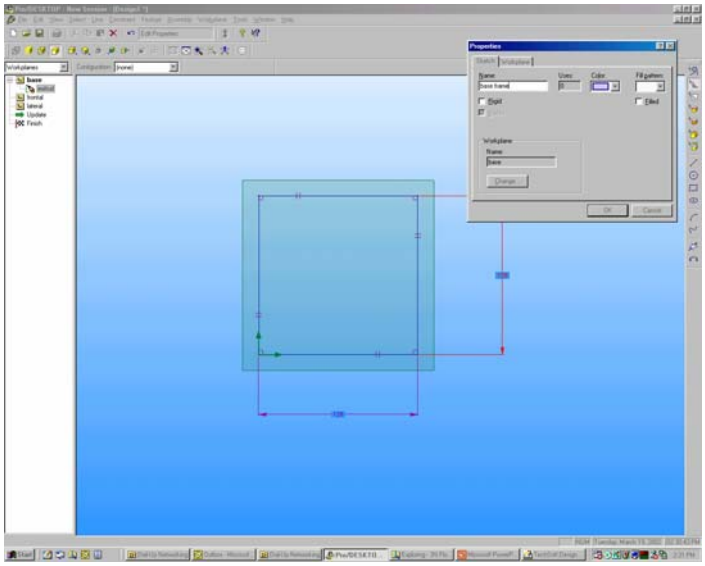


Create horizontal and vertical Sketch Dimensions and set the values to 128 mm.

SAVE YOUR DESIGN AS BASE.



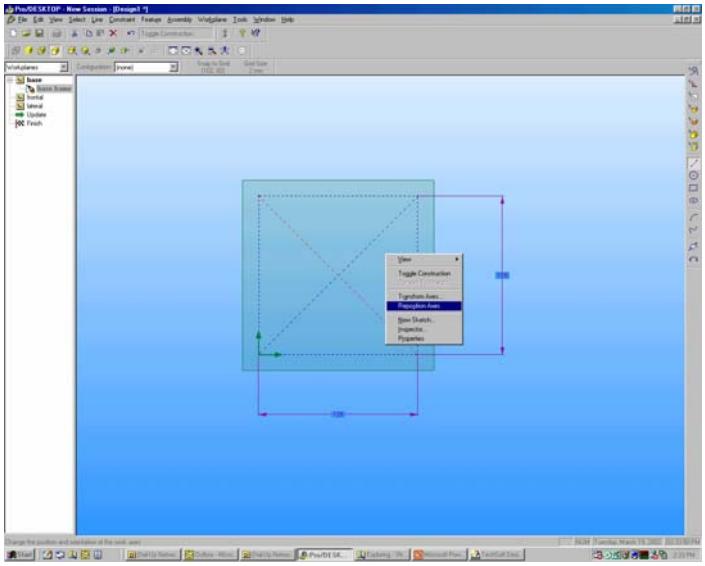
Change the name of the Sketch to Base Frame in the Browser.



Draw the diagonal lines as shown in the snapshot. This geometry must be converted into construction lines.

RC on the sketch name Base Frame in the Browser and choose Select Lines from the sub menu.
RC again and choose Toggle Construction to turn all the geometry into construction lines

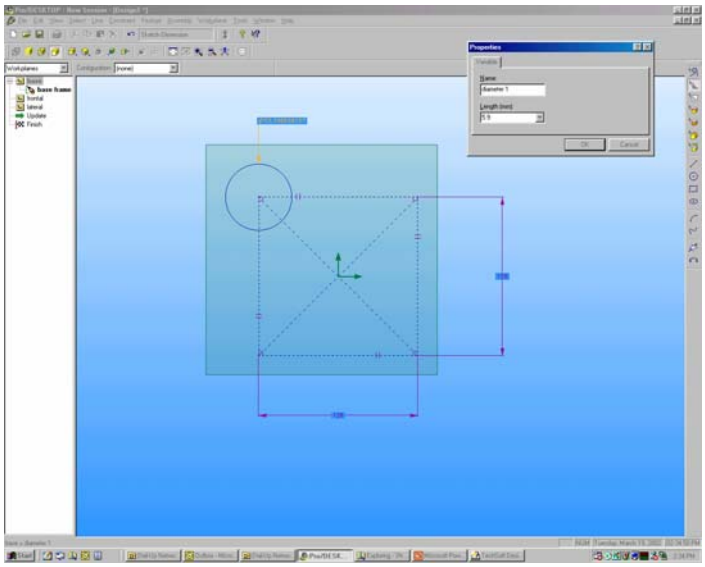
RC and select Reposition Axis from the sub menu then LC at the intersection of the diagonals.



Draw a circle in the position shown and set the diameter to 5.9 mm. (suitable for 6mm rod). If you are using rod of a different size make sure the hole is 0.1 mm smaller than the rod.
From the pull down menu select:

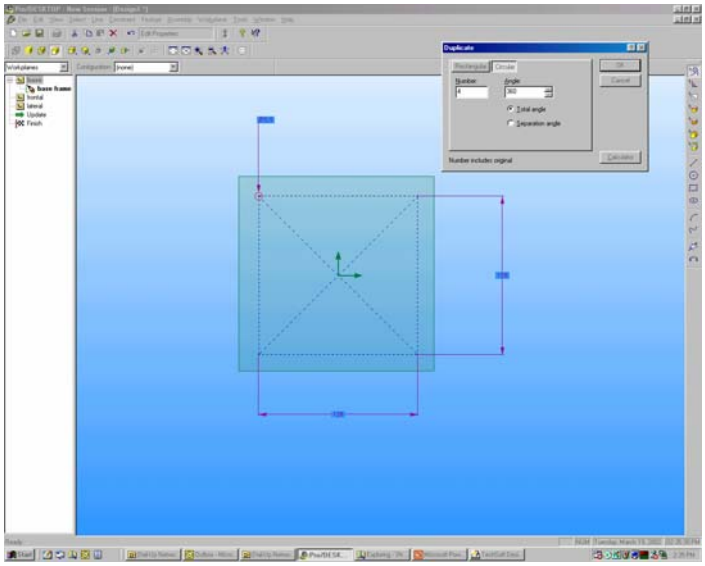
Edit – Duplicate

Select the Circular tab and set the Number to 4 and the Angle to 360.

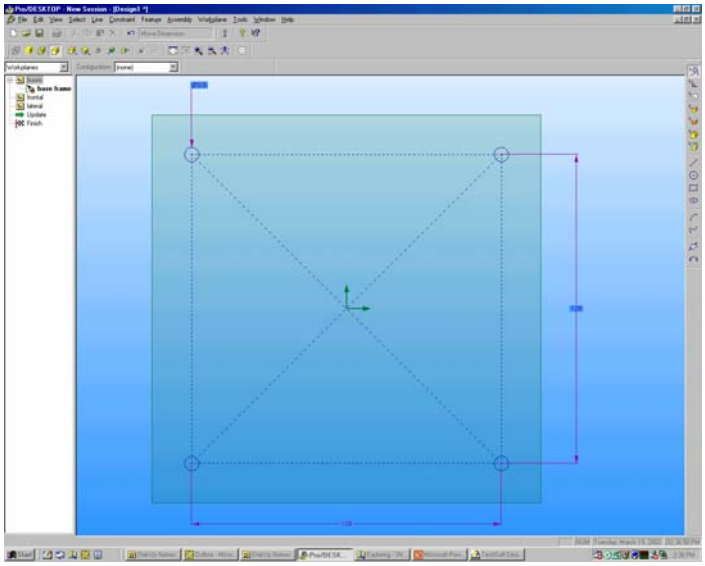


By repositioning the axis the center of the circular array was set to the intersection of the diagonal lines.

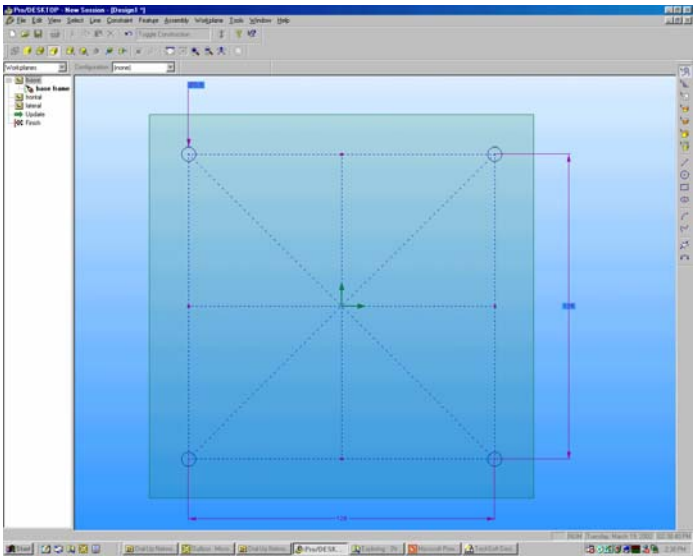
Pro/DESKTOP will duplicate the first circle 4 times around 360 degrees (90 degree gap).



This snapshot shows the circles in place at the intersection of each corner of the frame.



In the next sequence you will use the Offset Chain command to develop the frame. Draw a vertical and horizontal line as shown in the snapshot and toggle them into a construction line.



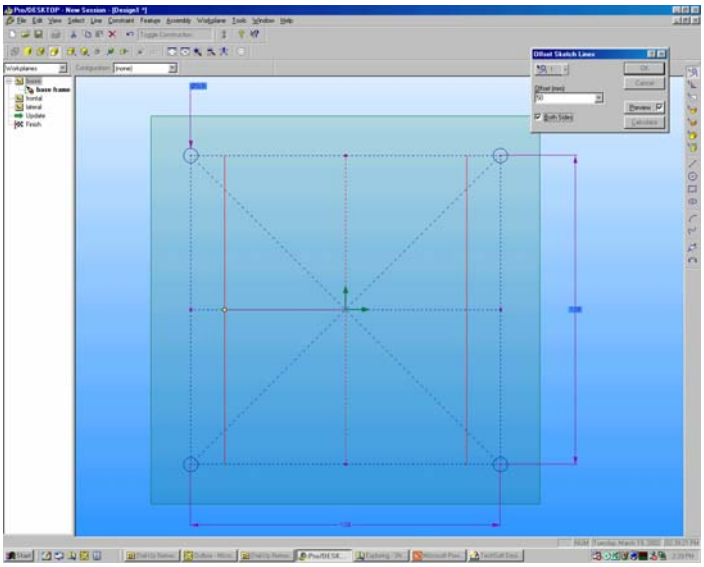
Choose the Select Lines tool from the Design Toolbar and pick the center vertical line.

From the Pull Down Menu select:

Line – Offset Chain

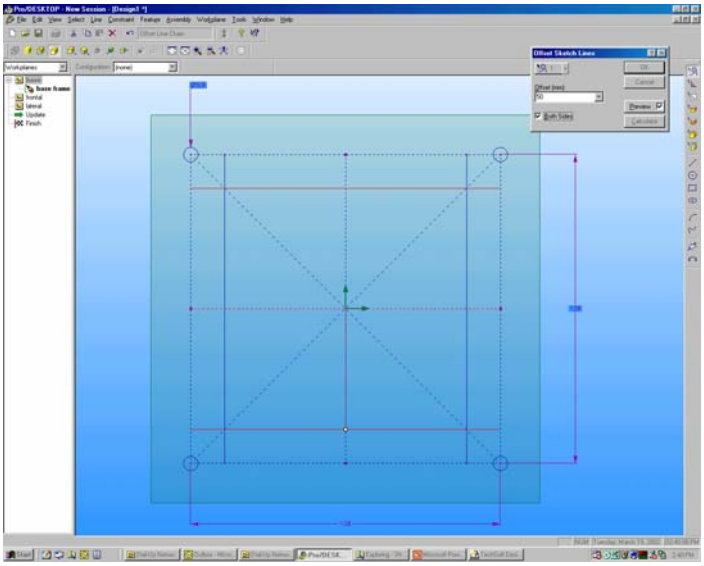
Set the Distance to 50 mm and pick the Both Sides tick box.

Two vertical lines are created each 50 mm from the center.

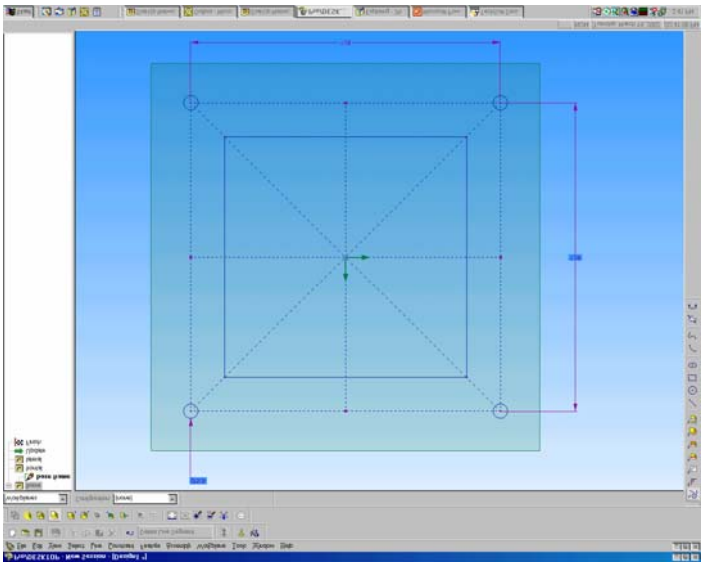


Repeat the process for the horizontal line to create the geometry shown in the snapshot.

SAVE YOUR DESIGN.

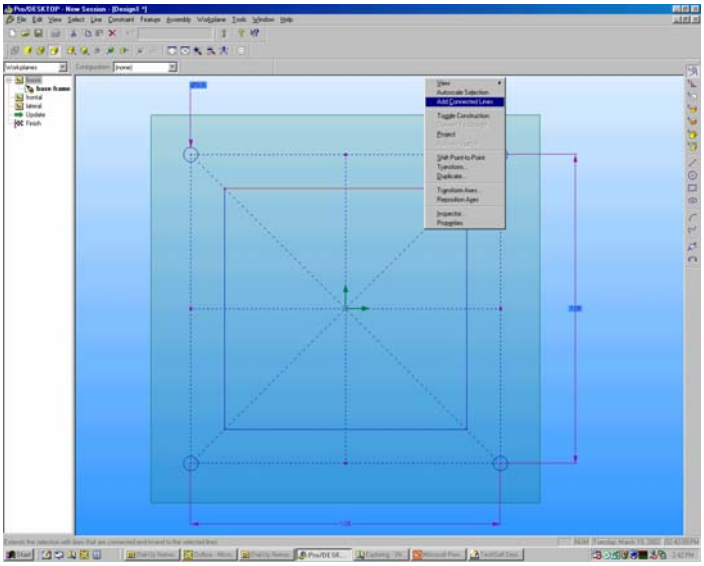


Choose the Delete Line Segment tool from the Design Toolbar and delete the lines until you have created a square as shown in the snapshot.



Choose the Select Line tool from the Design Toolbar and pick the horizontal line shown in the snapshot. RC and select Add Connected Lines from the sub menu.

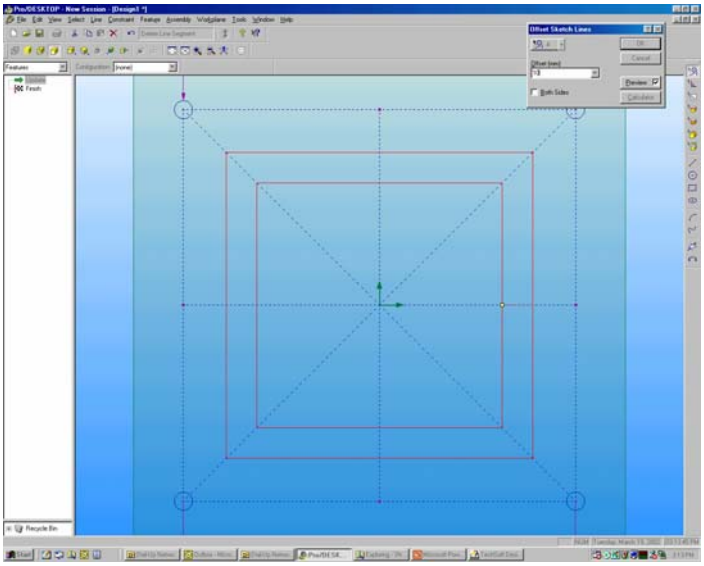
Pro/DESKTOP will connect all the lines that comprise the square.



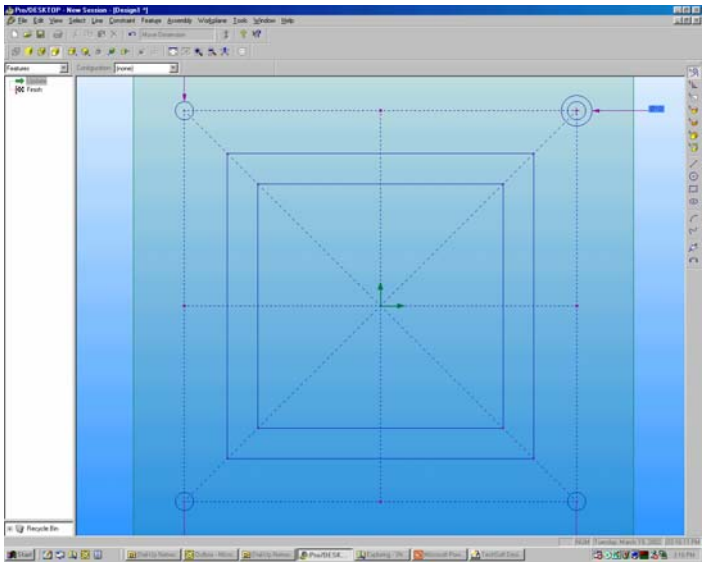
From the pull down menu select:

Line – Offset Chain

Set the Distance to 10 mm and ensure the new square appears inside the original. If the new square moves outside you may need to change the value to –10 mm.



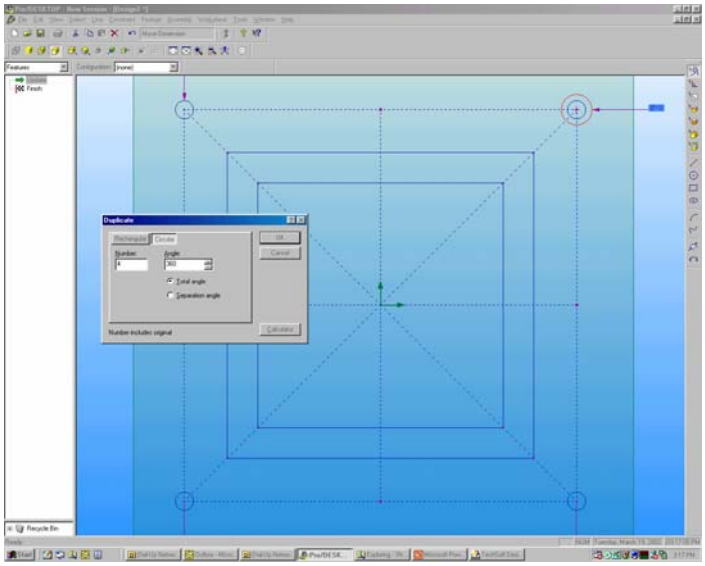
Create a circle as shown in the snapshot and set the diameter to 10 mm.



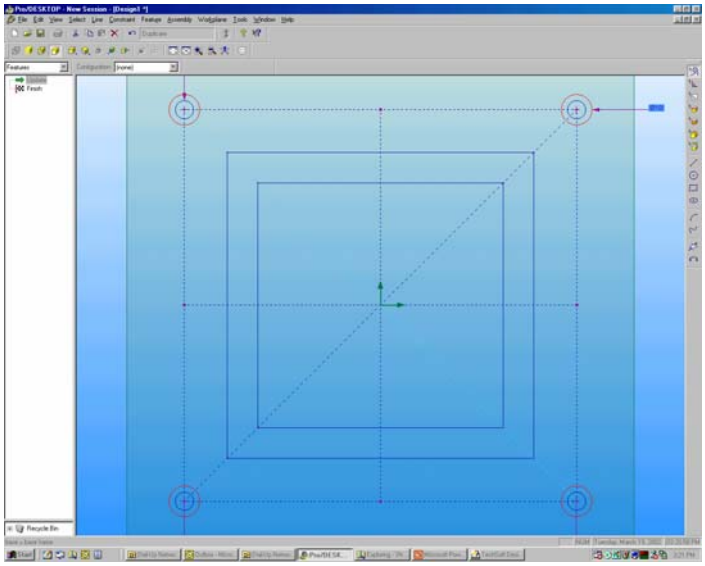
From the pull down menu select:

Edit – Duplicate

Select the Circular tab and set the Number to 4 and the Angle to 360.



A new circular array of circles appears, these will form the end of the frame that holds the support columns in place.

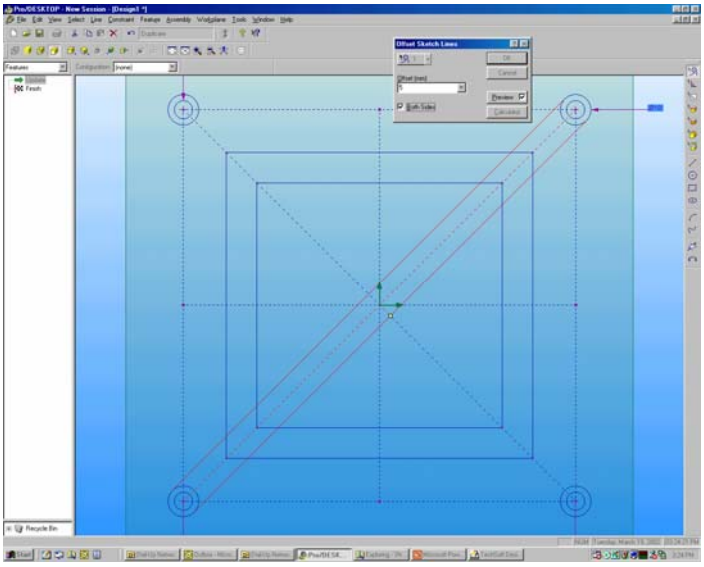


Create a diagonal line from the center of the two circles as shown in the snapshot.

From the Pull Down Menu select:

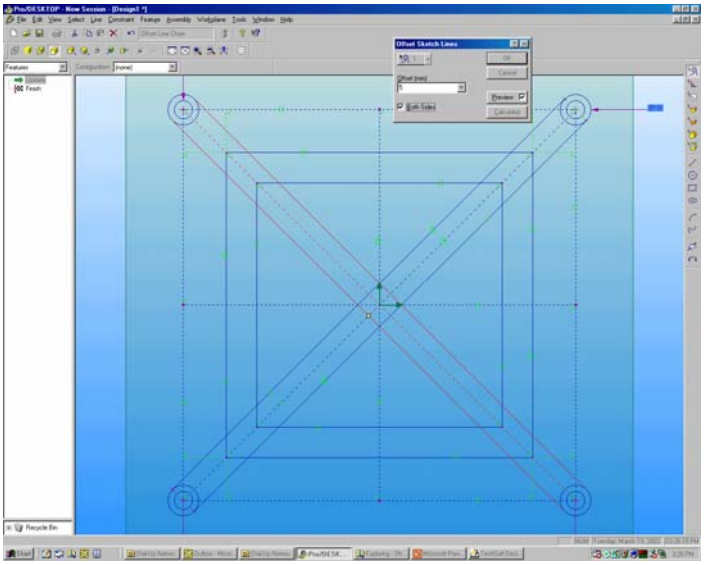
Line – Offset Chain

Set the Distance to 5 mm and pick the Both Sides tick box.

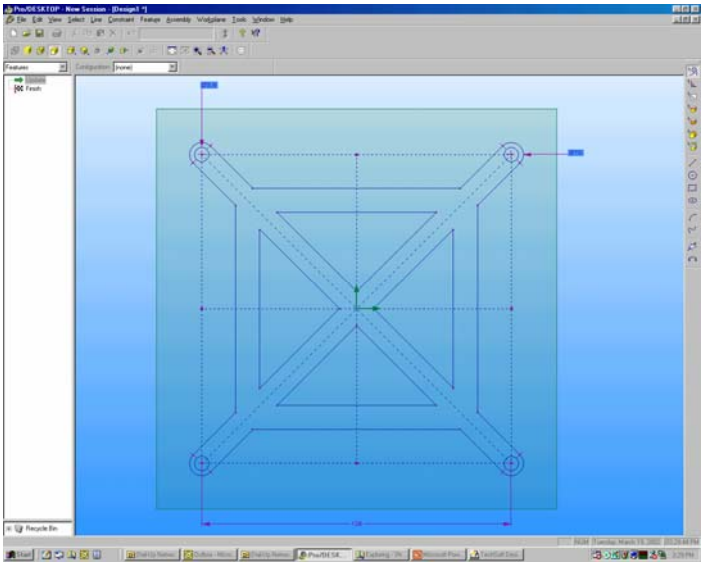


Repeat the sequence for the remaining diagonal line to create a triangulated frame as shown in the snapshot.

SAVE YOUR DESIGN.



Choose the Delete Line Segment tool from the Design Toolbar and delete the lines until you have created the frame as shown in the snapshot.

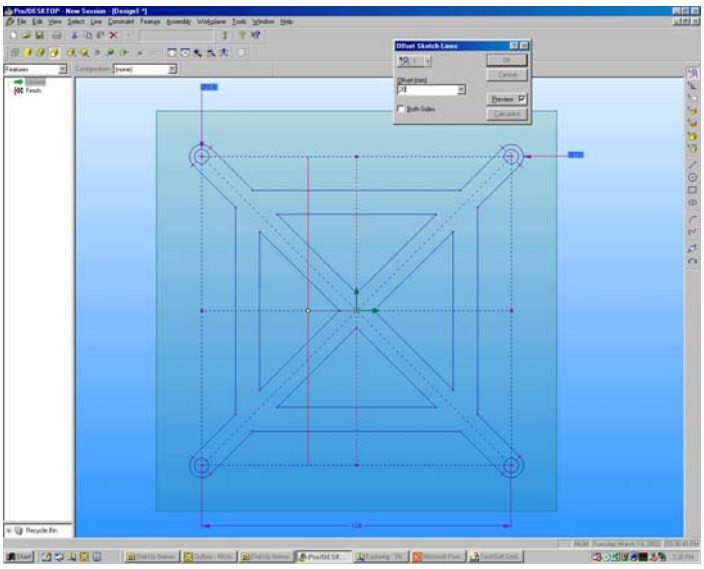


Select the center vertical line.

From the Pull Down Menu select:

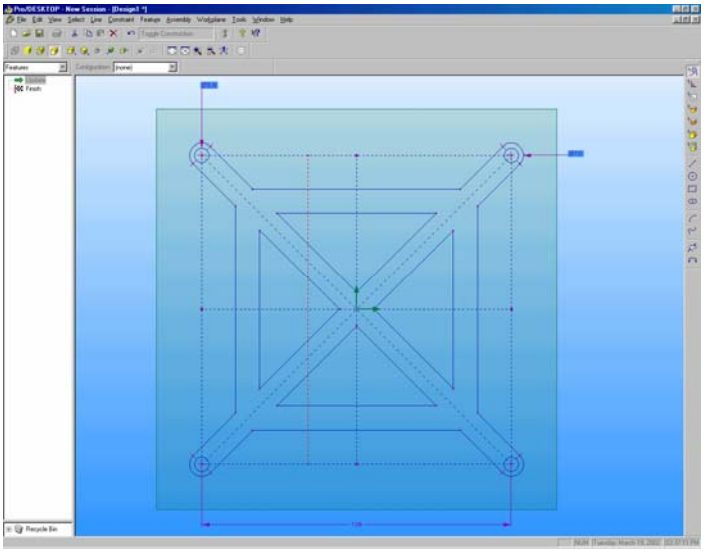
Line – Offset Chain

Set the Distance to 20 mm.



Toggle the line to a construction line.

Draw two concentric circles at the intersection of the construction lines and set the diameters to 5.9 mm (0.1 mm smaller than the 6 mm rod size) and 10 mm respectively.

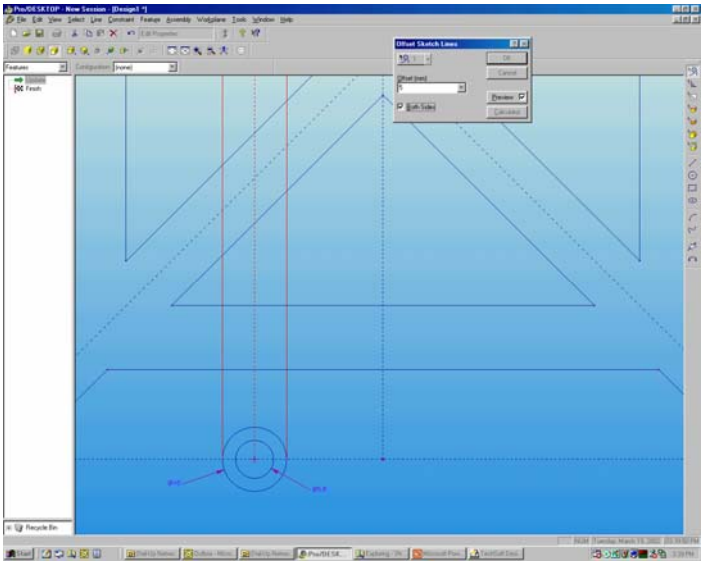


Zoom in to capture the detail shown in the snapshot.

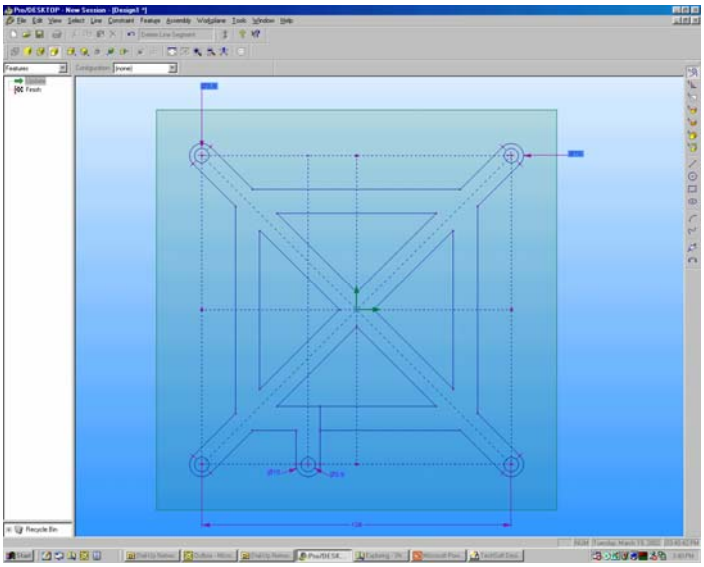
From the Pull Down Menu select:

Line – Offset Chain

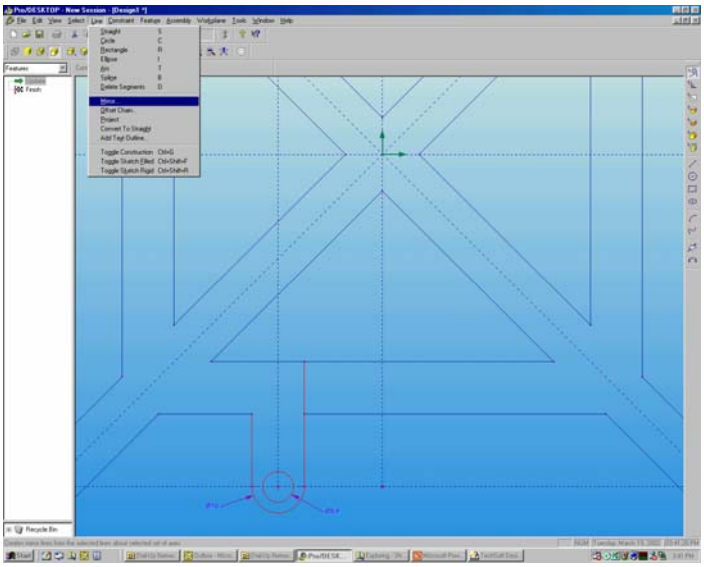
Set the Distance to 5 mm and pick the Both Sides tick box.



Select Delete Line Segment and modify the geometry until it is the same as the snapshot.



Zoom in to capture the detail shown in the snapshot. Choose the Select Line tool from the Design Toolbar and pick the lines and circles shown as red in the snapshot.

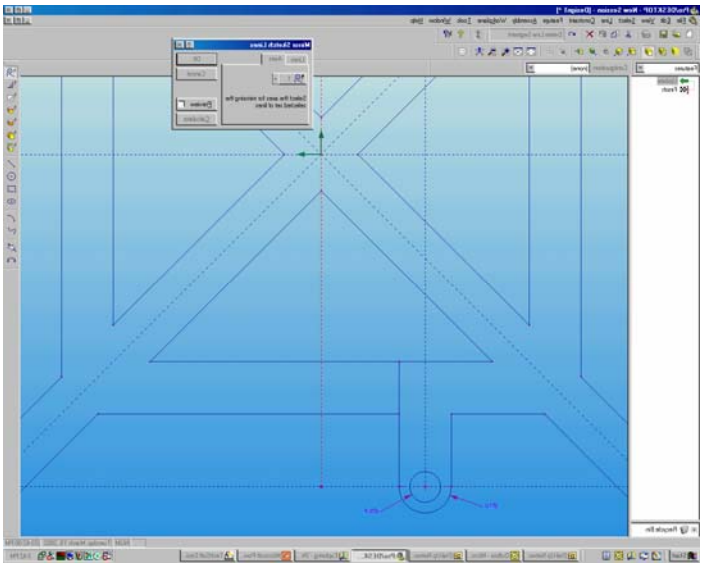


From the Pull Down Menu select:

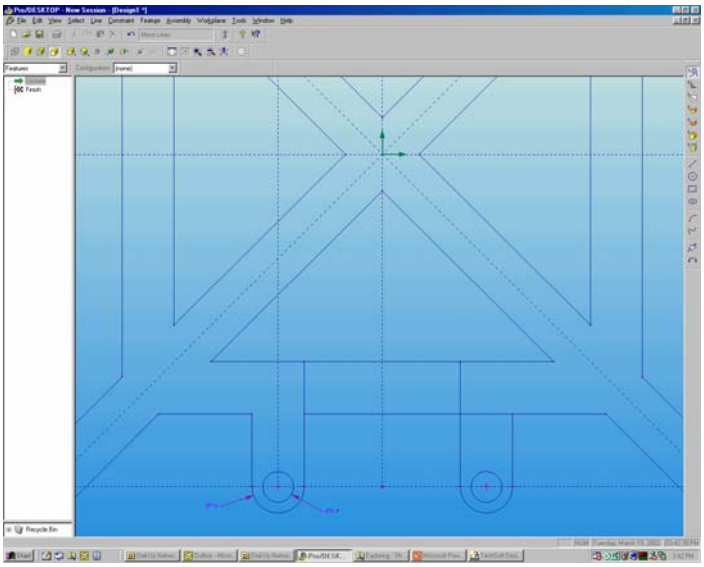
Line – Mirror

Change the tab to Axes and select the center toggle construction line shown in the snapshot.

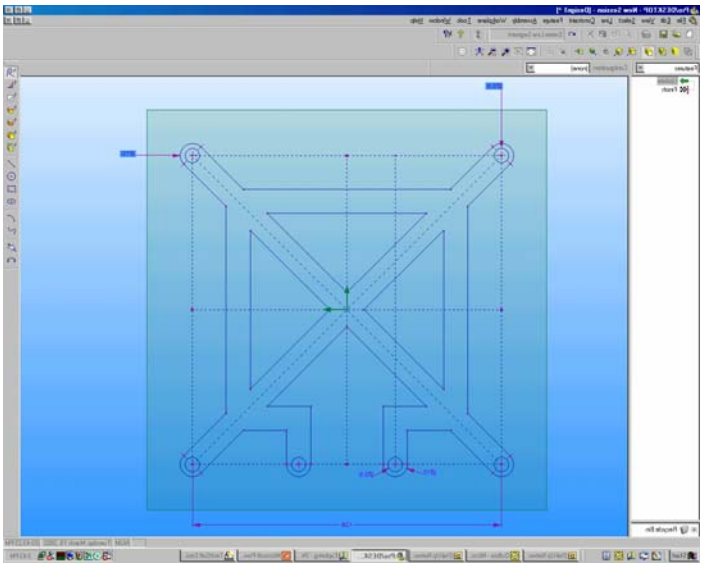
Click on OK to complete the command and mirror the geometry about the center line.



Select Delete Line Segment and modify the geometry until it is the same as the snapshot.

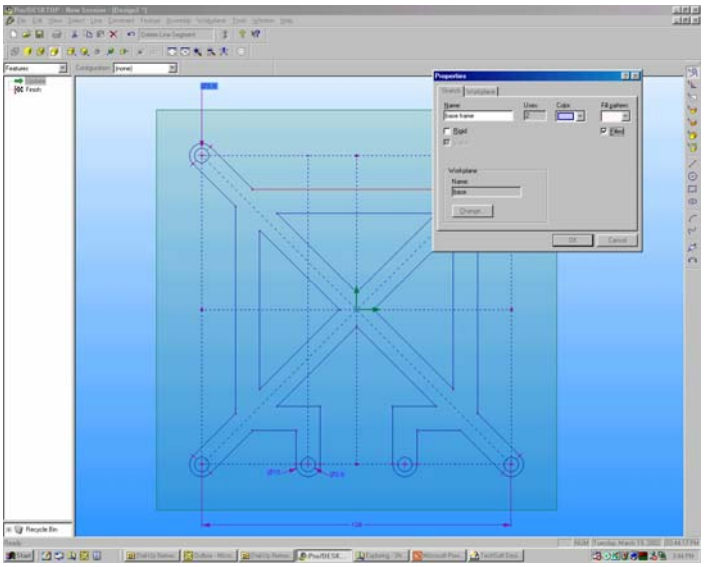


This snapshot represents the completed frame. In the next sequence you will modify this frame to create the second component of the design.



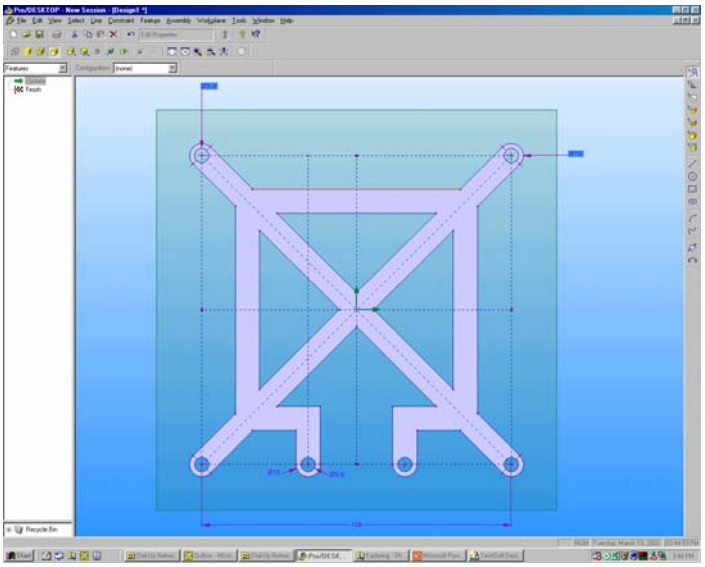
Before you turn this two-dimensional frame into a three-dimensional solid model it is wise to perform a simple check to ensure the geometry is valid and free of any errors.

Pick one line as shown in the snapshot. RC and select Properties from the sub menu.



Click in the fill tick box, if the geometry is valid the profile will fill with a colour.

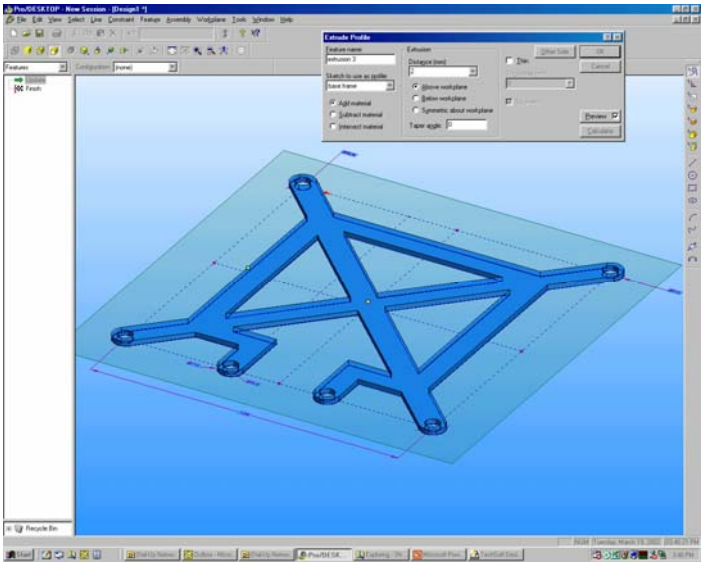
Its well worth doing this check before you try to extrude the profile into a three-dimensional shape. Return to a 3D view by selecting Shift – T.



From the Pull Down Menu select:

Feature - Extrude

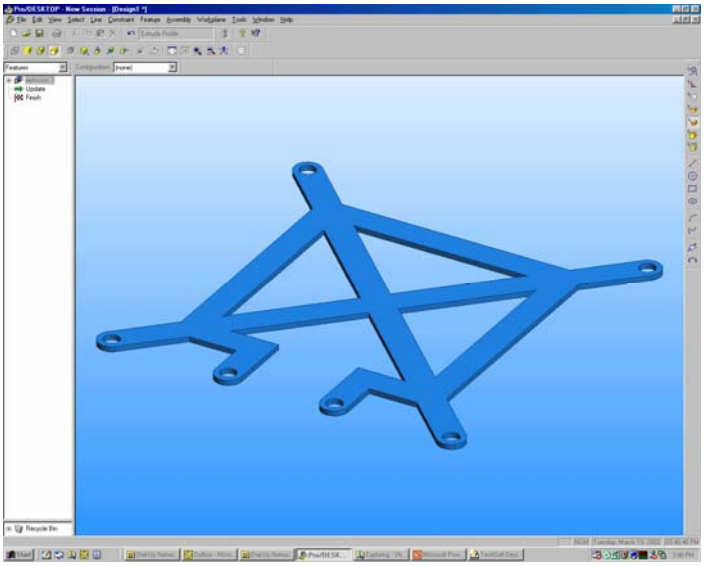
Set the values to Add Material – Above Workplane – 2 mm



The snapshot shows the completed solid model of the base of the post it holder design.

The second component varies slightly from this original design so we can use a copy of the base to produce the shelf.

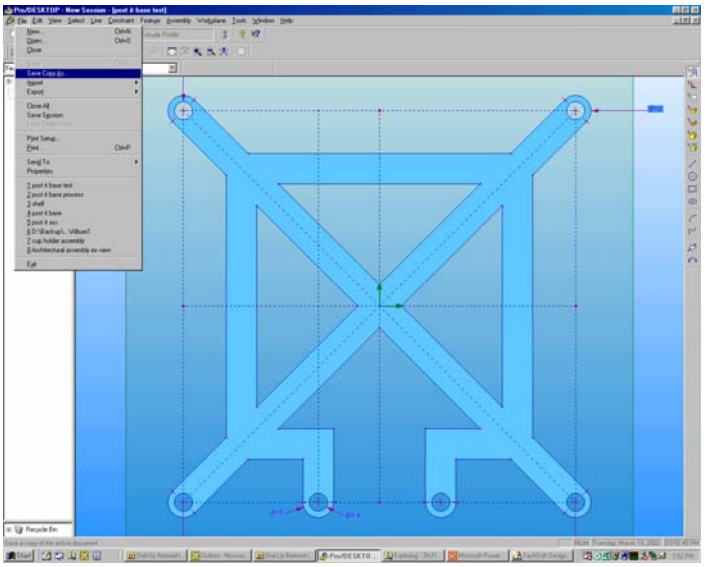
SAVE THE FILE.



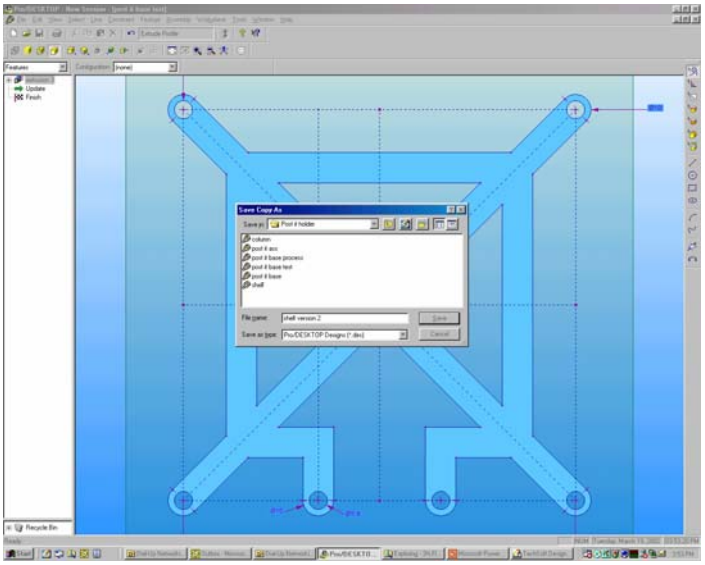
Form the Pull Down Menu select:

File – Save Copy As

The second component shares the same external profile as the base. In order to minimise any further drawing you can modify the original design by deleting the diagonal features.



SAVE THE FILE AS SHELF

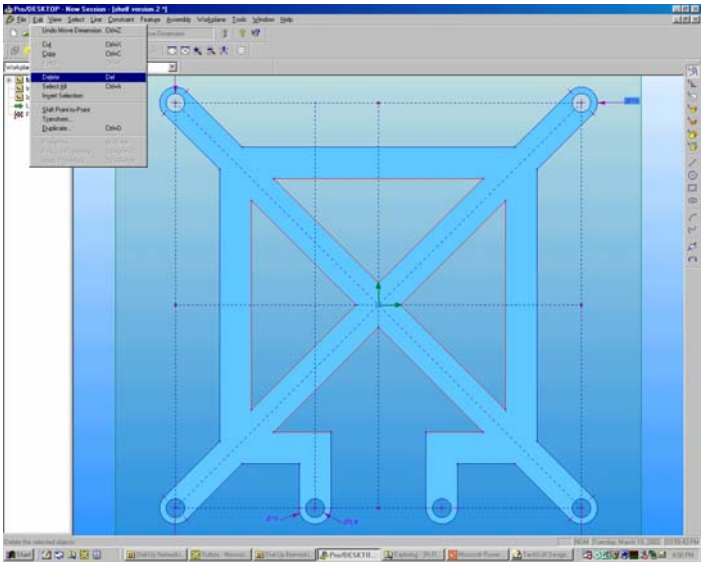


Close the file called BASE.

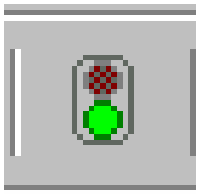
Open the file called SHELF.

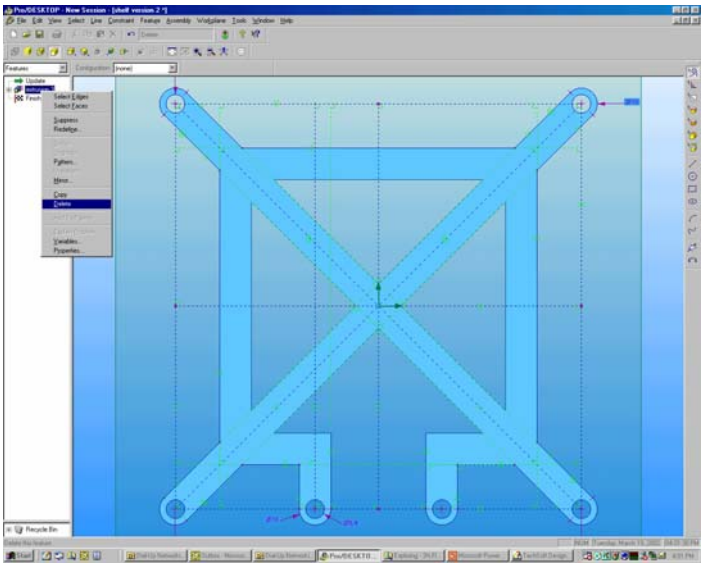
Change the Browser to Components.

In the next sequence you will delete the extruded feature of the solid model.

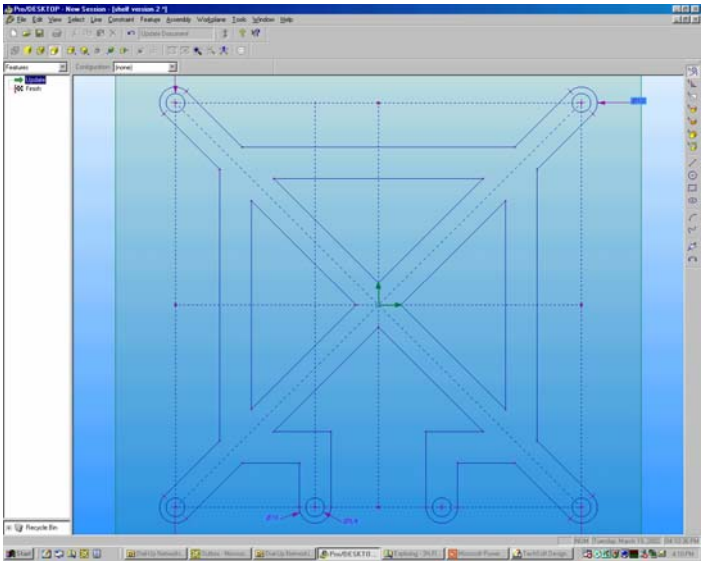


Move the cursor over the word extrusion 1 in the browser. RC and select Delete from the sub menu. Click on the Update button from the top of the screen to regenerate the design.





The previous sequence has destroyed the solid model of the frame but the two - dimensional lines still remain.

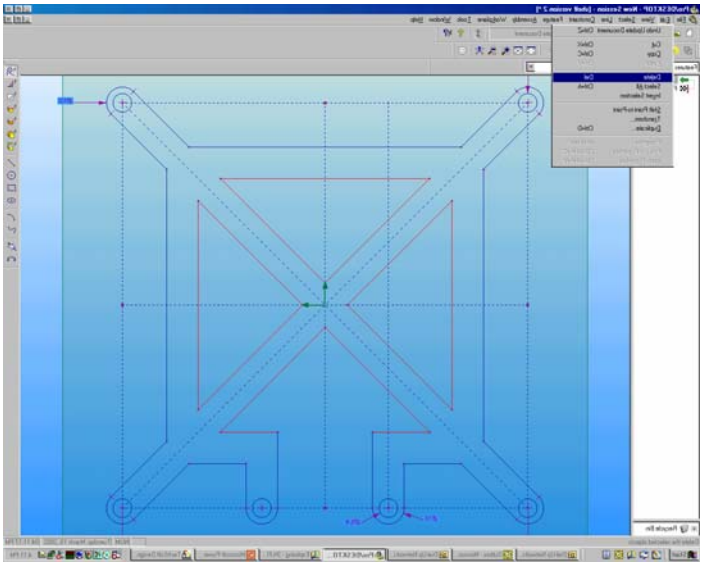


Choose the Select Lines tool from the Design toolbar and drag a window around the four triangles shown in the snapshot.

From the Pull Down Menu select:

Edit – Delete

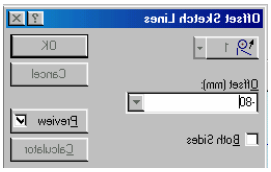
The selected geometry is erased.



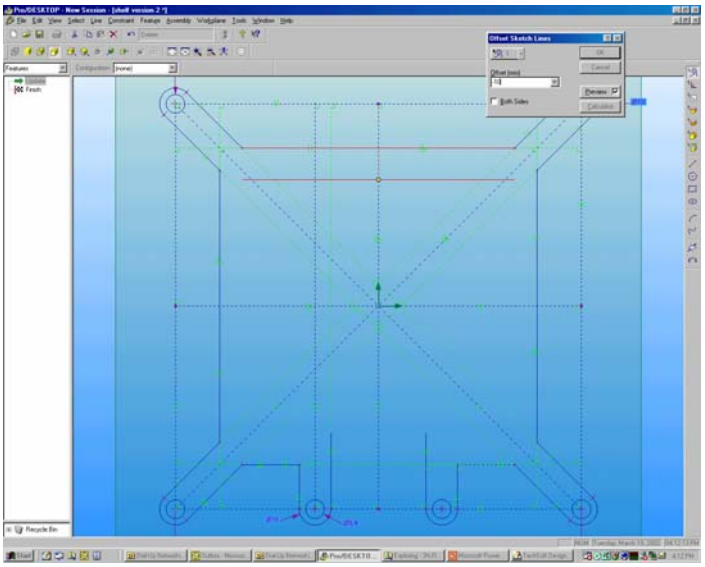
Click on the top horizontal line as shown in the snapshot.
From the pull down menu select:

Line – Offset Chain

Set the Distance to -10 mm.



Repeat the process for the left and right vertical lines.

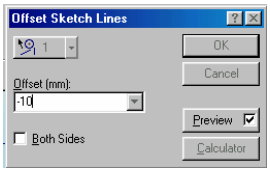


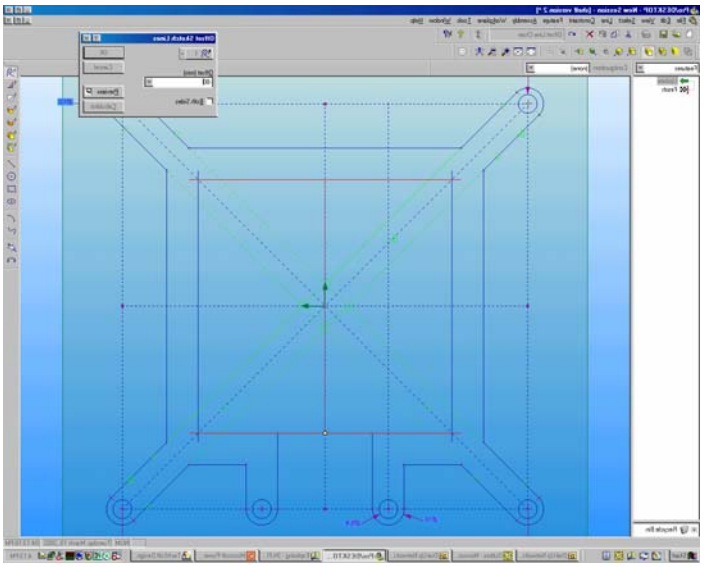
Click on the second horizontal line as shown in the snapshot.

From the pull down menu select:

Line – Offset Chain

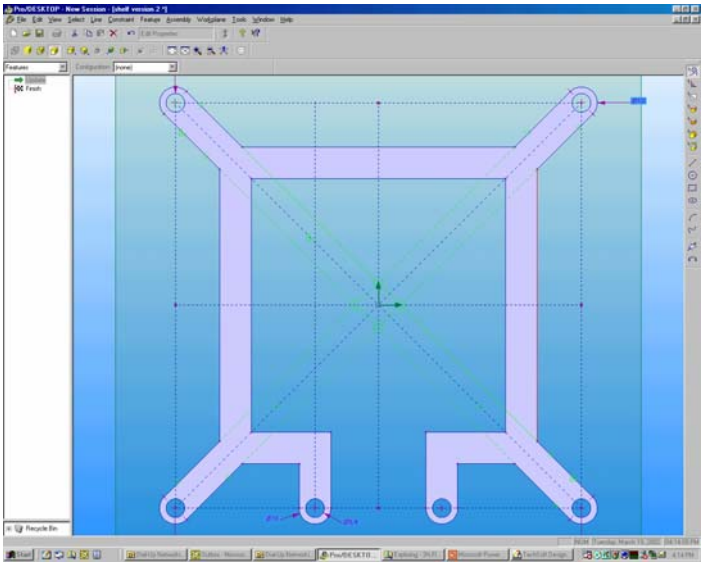
Set the Distance to -80 mm.





Use the Delete Line Segment tool to erase the overlapping lines until the profile looks like the snapshot.

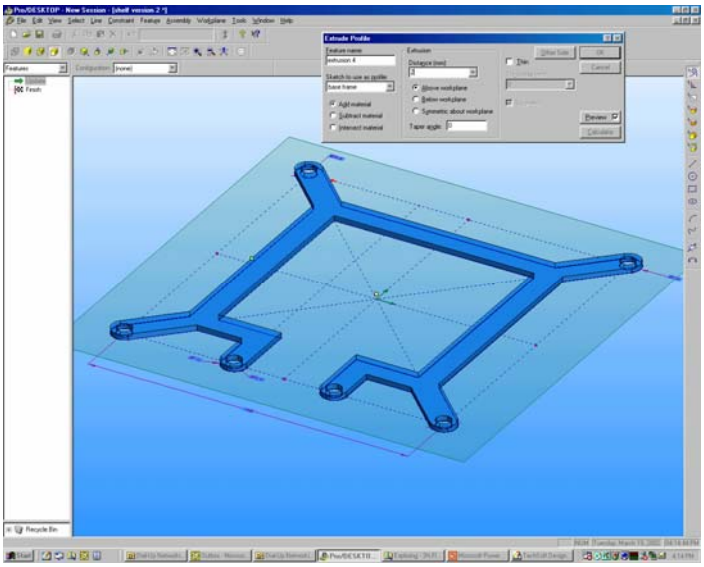
Repeat the procedure for checking the validity of the profile using the fill command.



From the Pull Down Menu select:

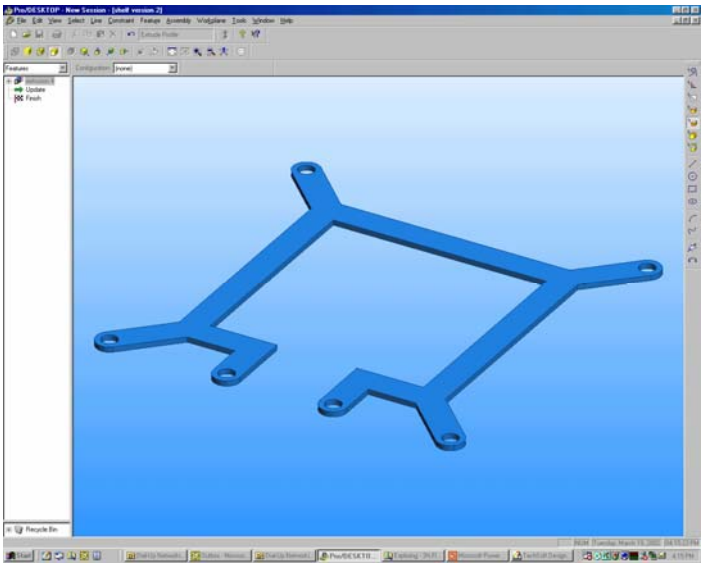
Feature - Extrude

Set the values to Add Material – Above Workplane – 2 mm



The snapshot shows the completed solid model.

SAVE THE FILE AS SHELF.



The snapshot shows an assembly of the Post it holder. The design comprises of one base element and nine shelf elements.