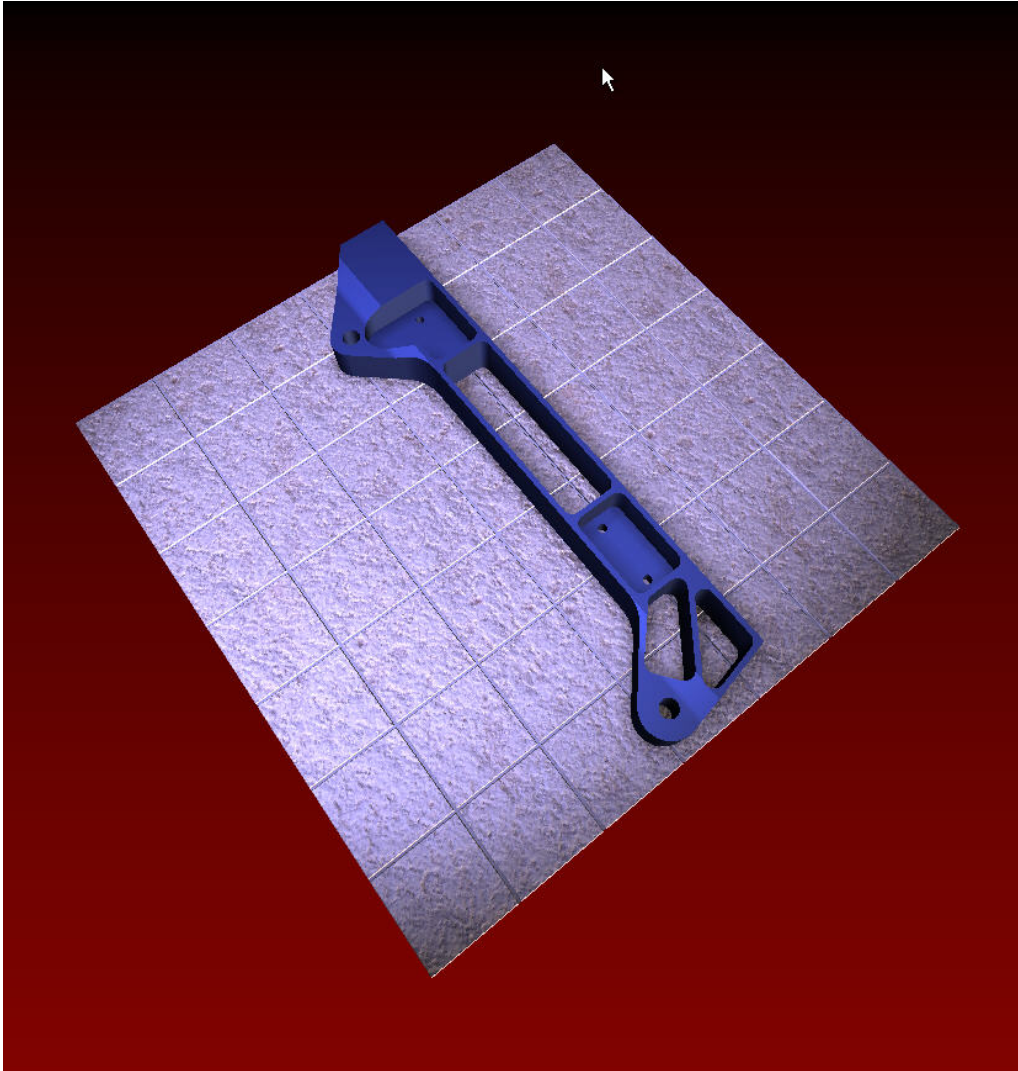
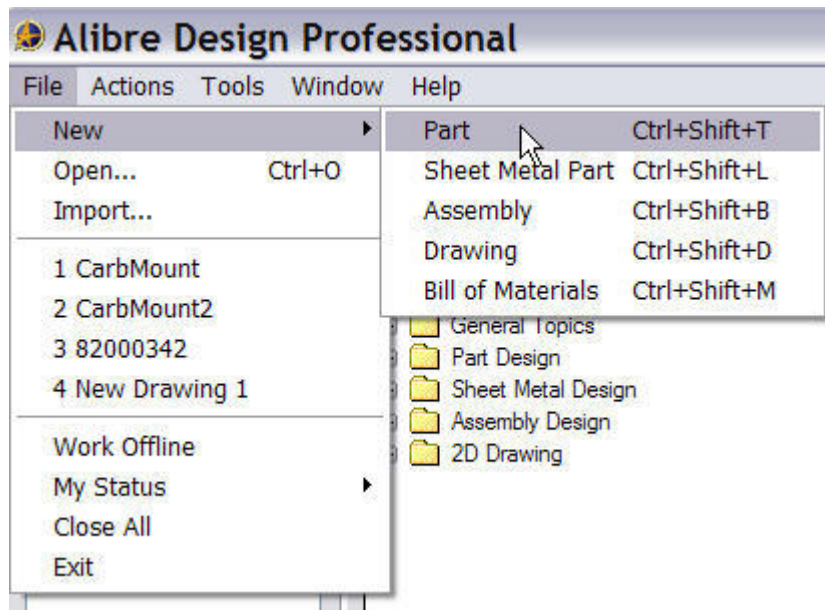


Motor Mount
An Alibre Design Tutorial
By
CDS

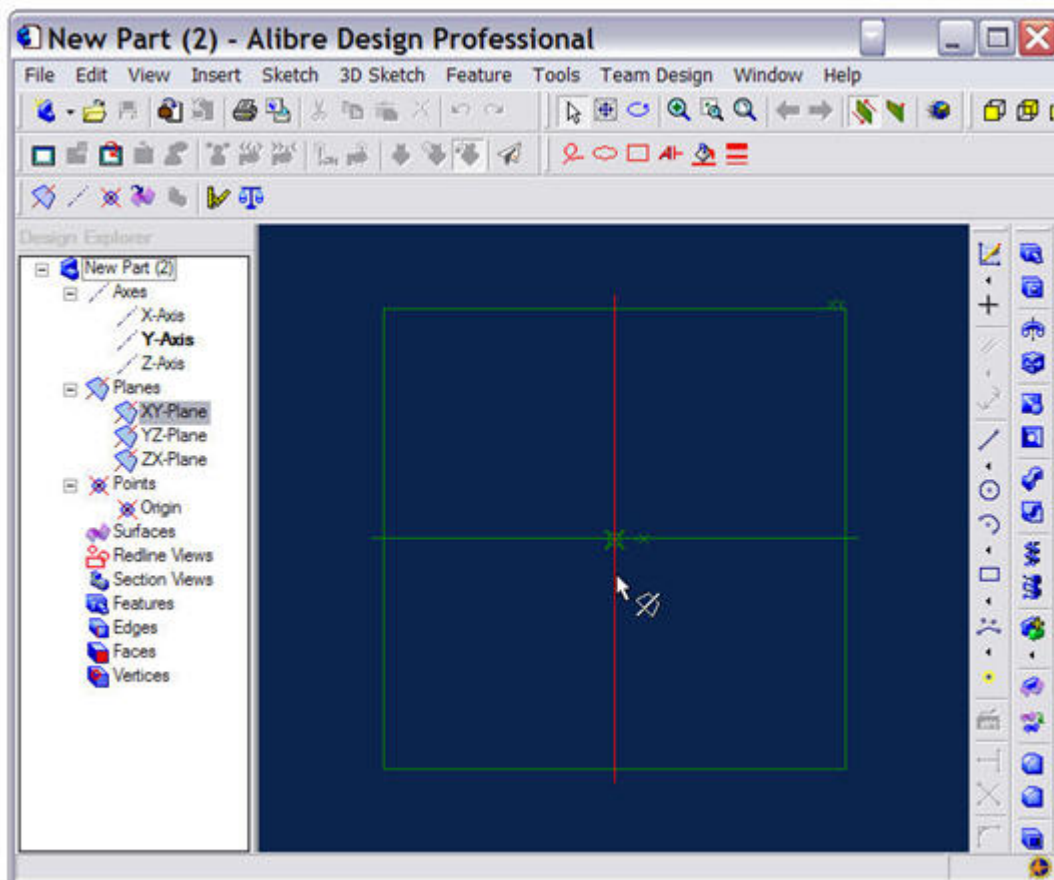


Written by Steve Carr
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@by CDS 04-15-06

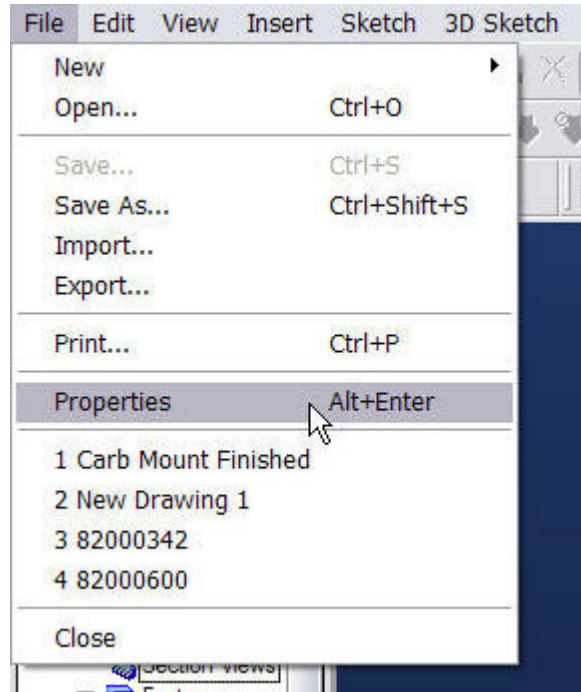
We need to set a few basic parameters in Alibre before we start the actual design process. In the Alibre Design Main panel, click on File>New>Part.



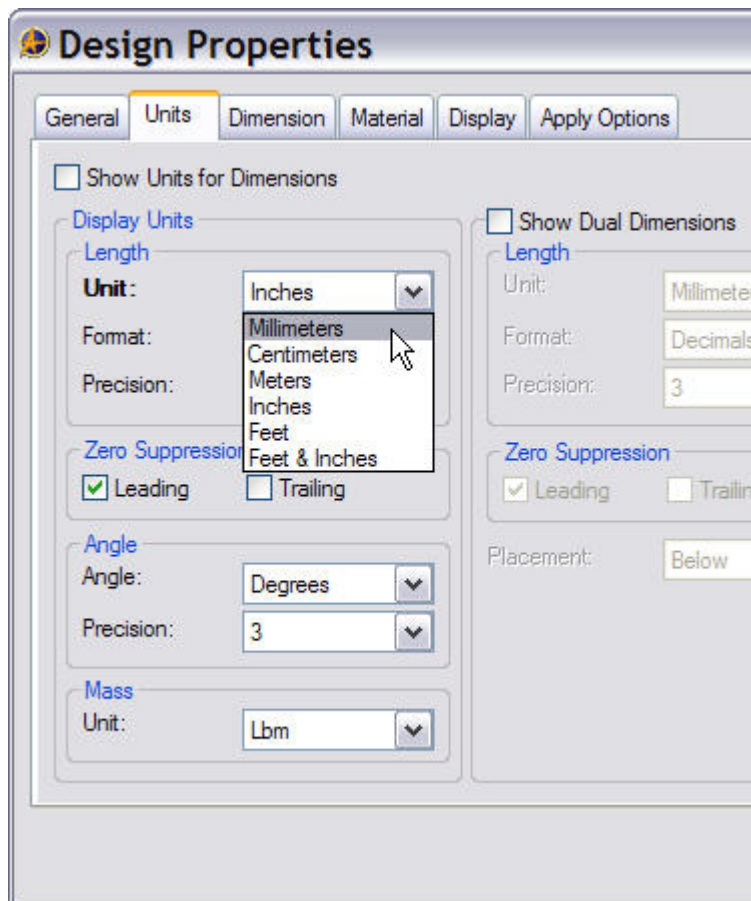
The New Part window opens.



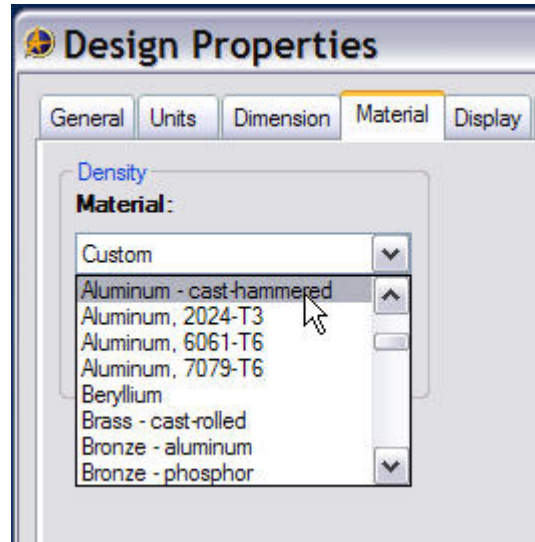
Under the File menu tab, select Properties.



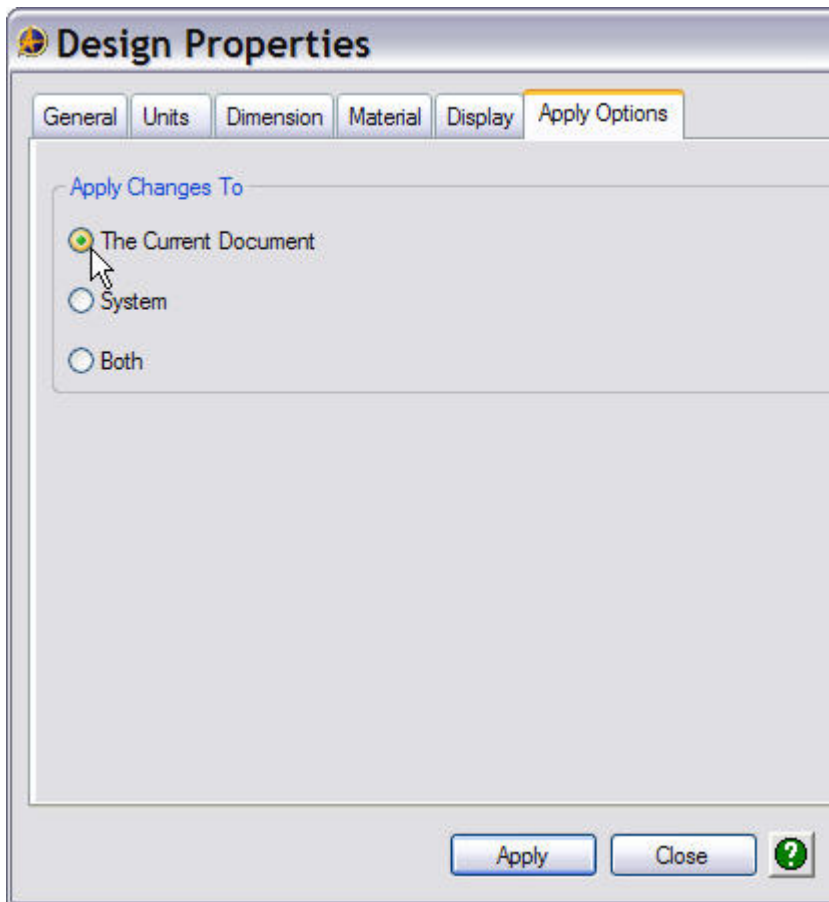
The Design Properties panel will open. Click on the 'Units' tab and select Millimeters.



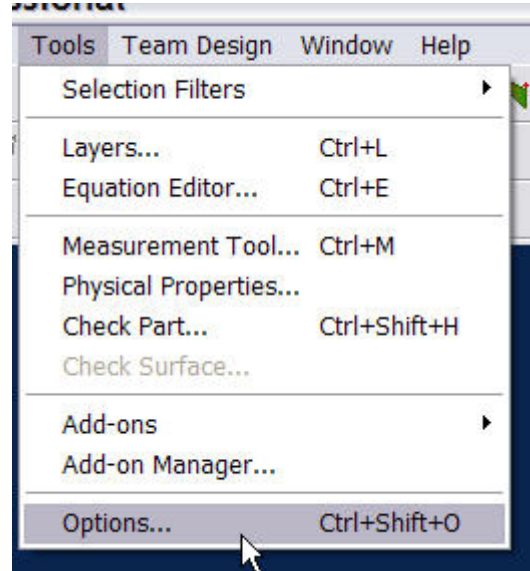
Under the 'Material' tab, scroll down in the selection window and click on 'Aluminum – cast – hammered'. This will set the material properties for our model, allowing us to determine the mass, and volume of the part as well as the center of mass, surface area, and moments of inertia.



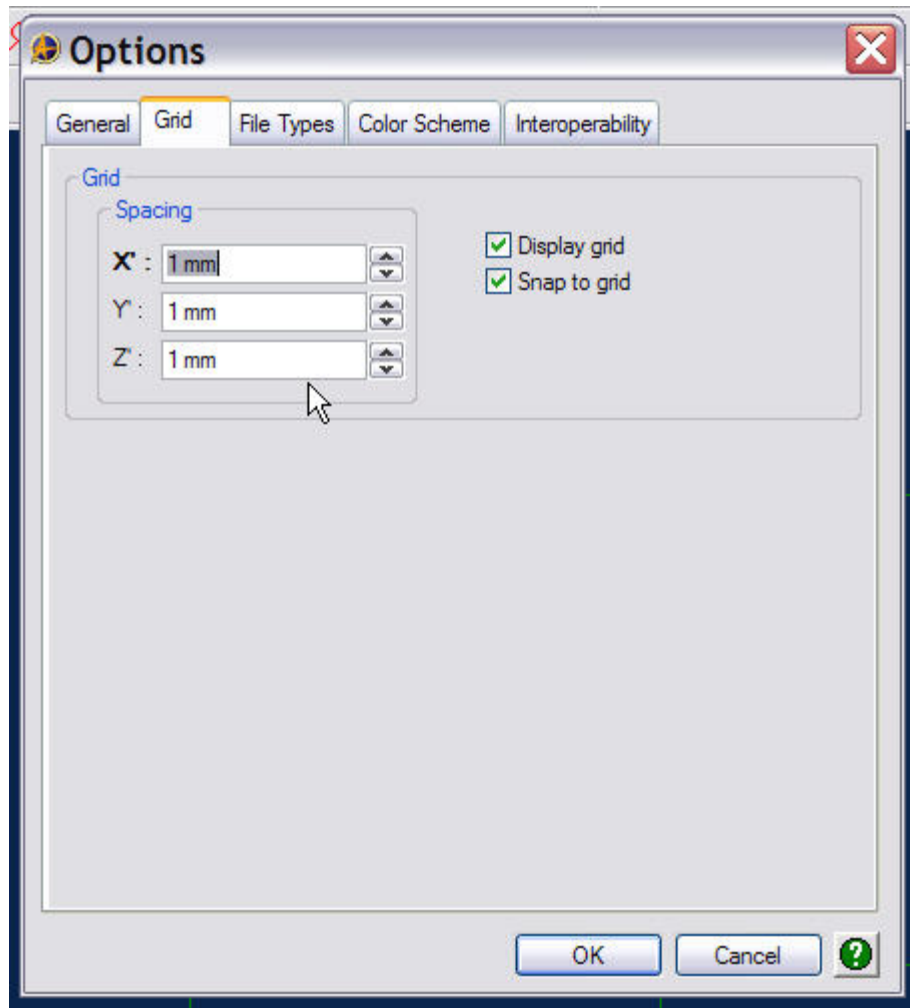
Under the Apply Options tab, check the radio button for 'The Current Document'.

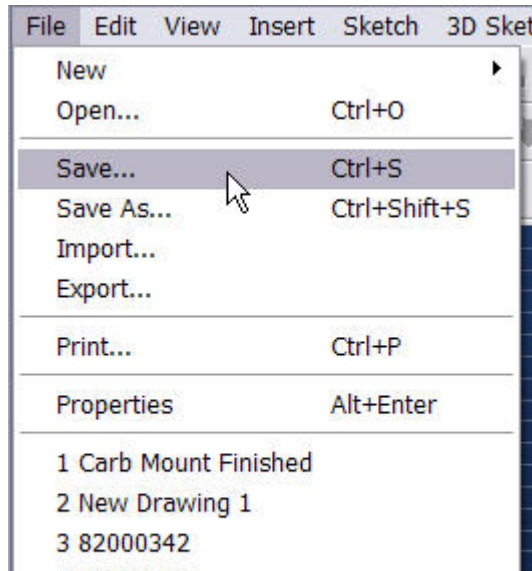


Go back to the main panel toolbar and locate 'Tools'. Under the 'Tools' menu tab, select 'Options'.

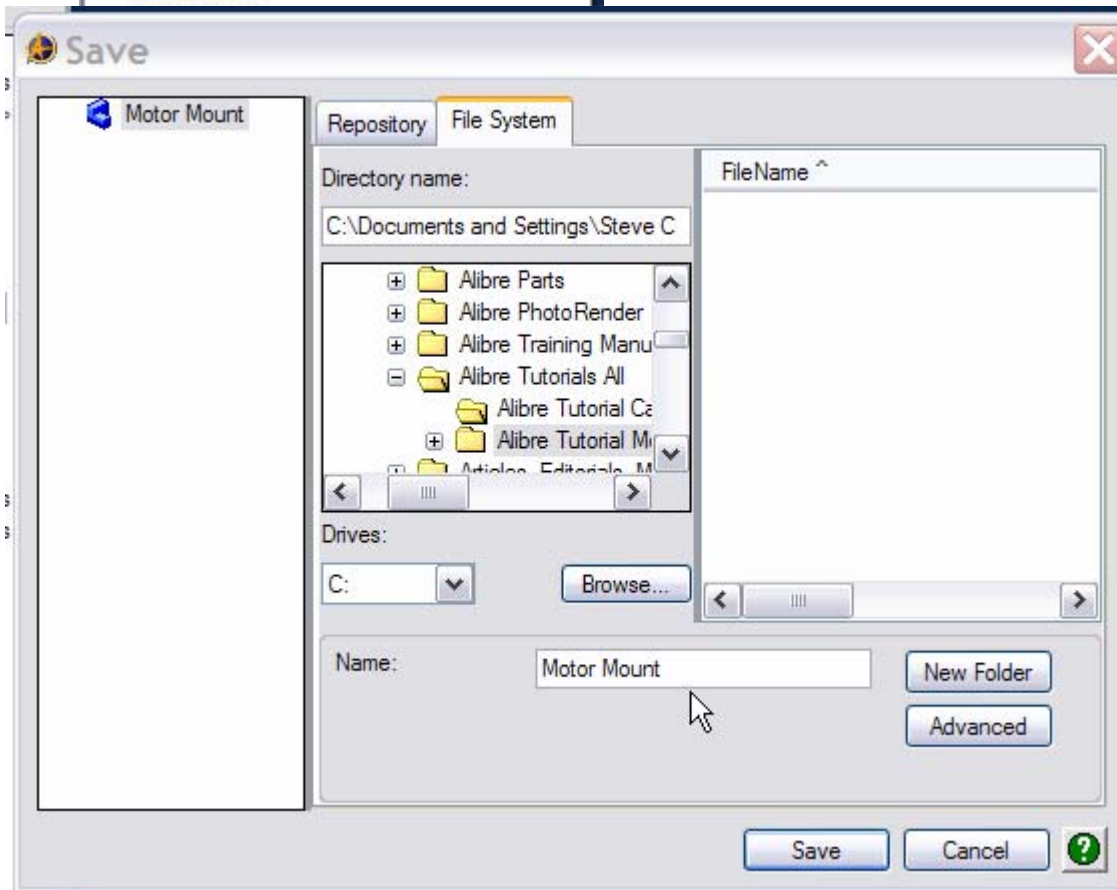


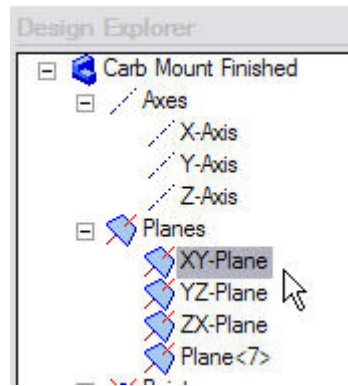
The 'Options' panel will open. Under the 'Grid' tab change the spacing settings to 1mm for X, Y, and Z. Make sure the 'Display grid', and 'Snap to grid' boxes are checked. Click OK to return to the New Part window.



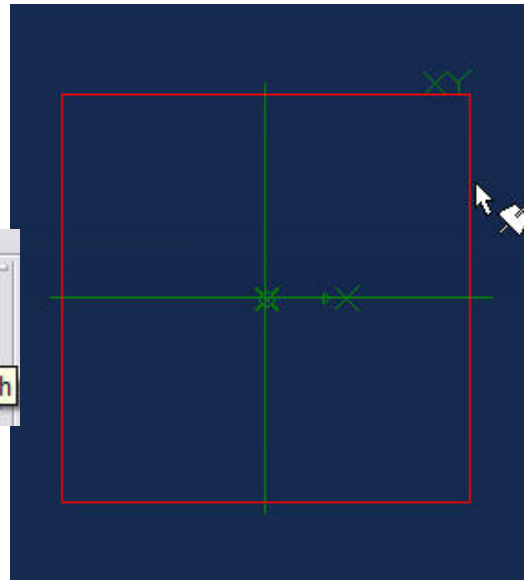
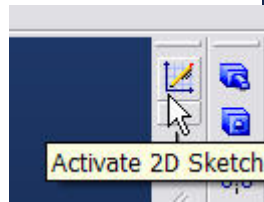


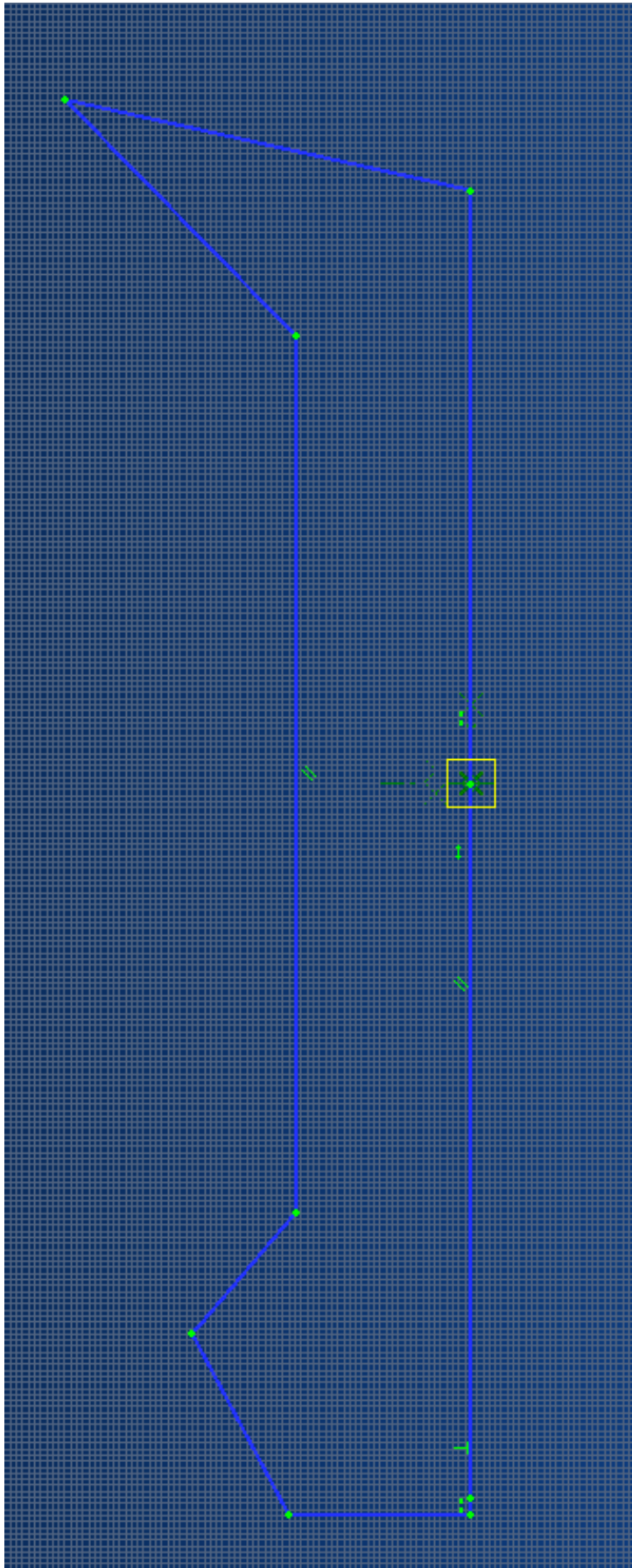
In the New Part window under the File menu tab, select 'Save'. Select a location to save the file, either in the Windows File System or in your local repository. Give the part a logical name, and click OK.





Click on the XY plane in the Design Explorer panel, or the XY plane in the Part workspace and then click the 'Activate 2D Sketch' icon. Now you're in 'Sketch' mode.

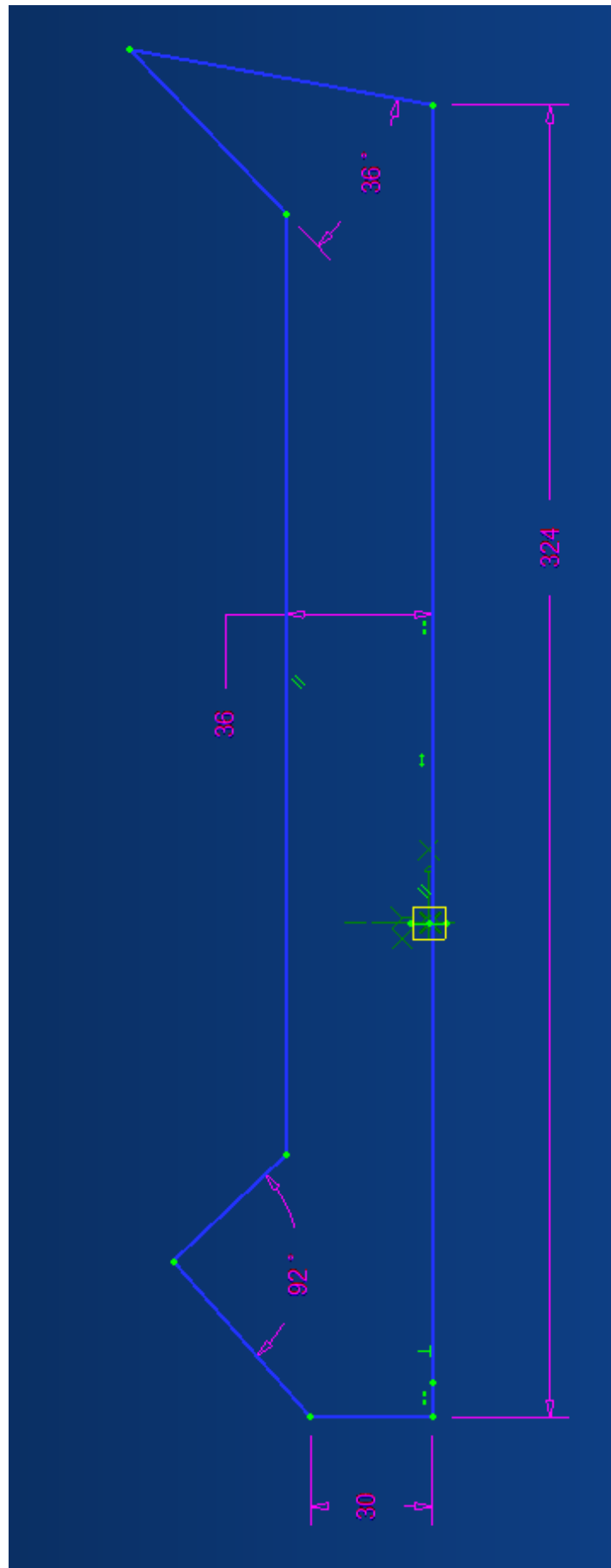
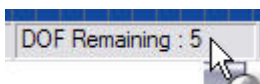


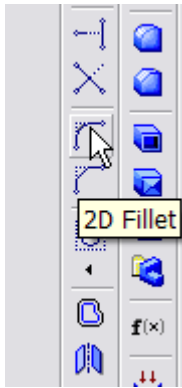


We'll start our design by drawing a sketch similar to the one at the left. The picture has been rotated 90 counter clockwise, which places the X-axis in the vertical orientation. The sketch only needs to be a close approximation to what we want the final part to look like. In most of the following pictures, the sketch grid has been turned off, simply for the sake of clarity. You should leave your grid turned on to aid you in your sketch work.

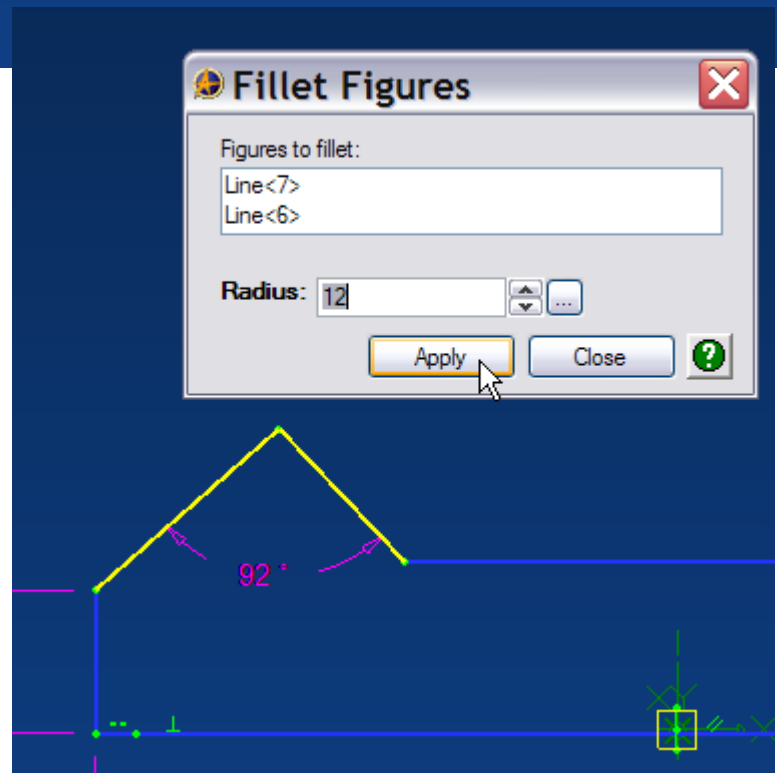
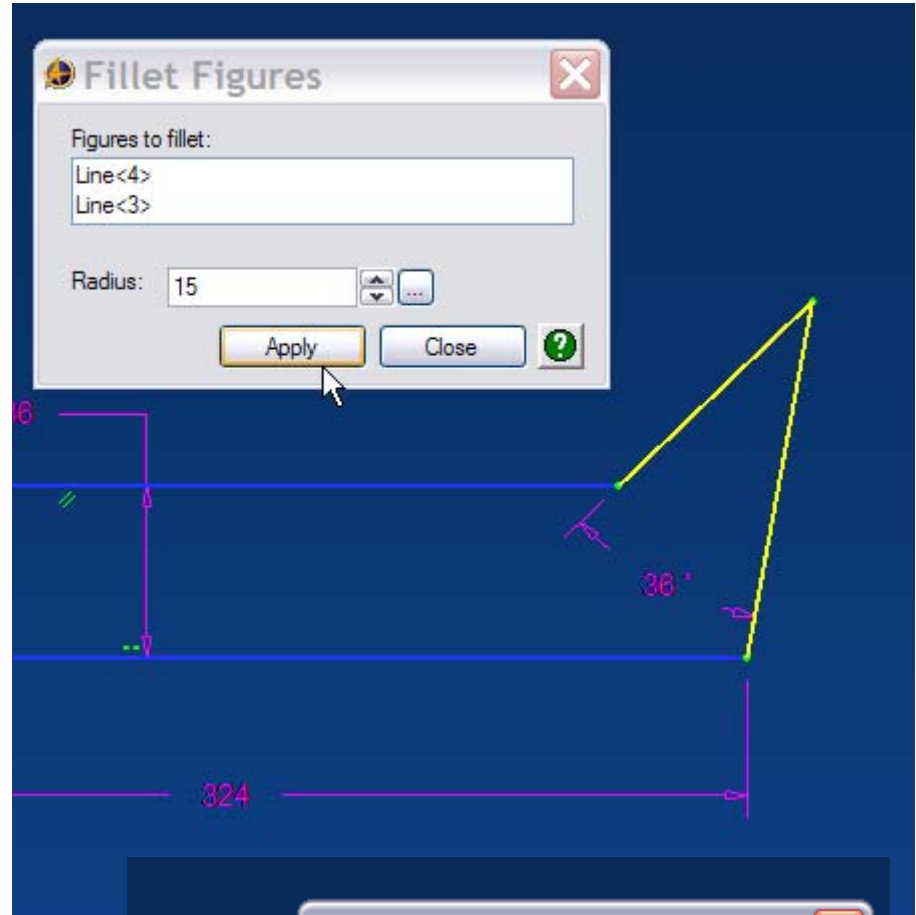
Now apply the five dimensions shown at the right. These dimensions will 'lock in' the overall shape of out part, and establish an important set of relationships.

Dimensioning and constraining sketches is one of the most important aspects of good design work. You'll notice in the lower right hand corner of the Alibre main panel screen a running script labeled DOF (Degrees of Freedom). Reducing the DOF's to zero, through dimensional or geometric constraints, although not always necessary, is good design practice. You can produce parts and features with sketches that have unresolved DOF's, but my advice is to reduce the DOF's to zero through use of dimensions or geometric constraints to insure the stability of the model data. If you find that you can't reduce the DOF's by manual insertion of the dimensions and constraints, use the Auto Dimension command under the Sketch menu tab to complete the task. In this case, we have a lot more dimensioning work to complete, so ignore this for the present.

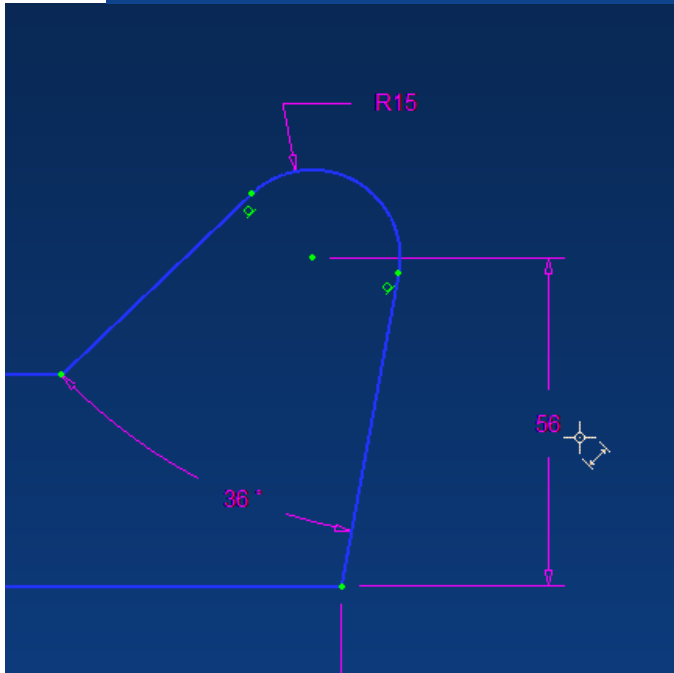
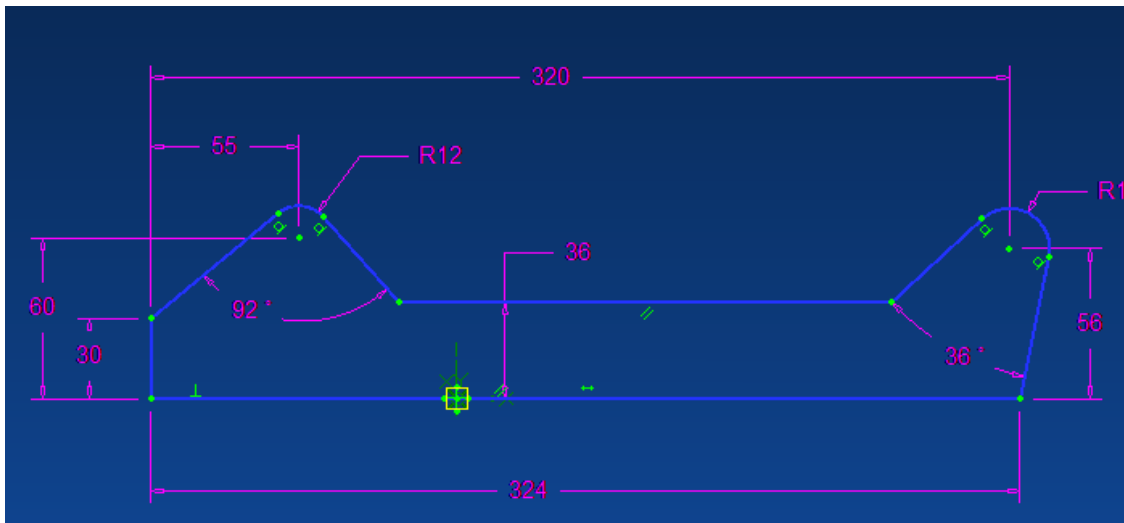




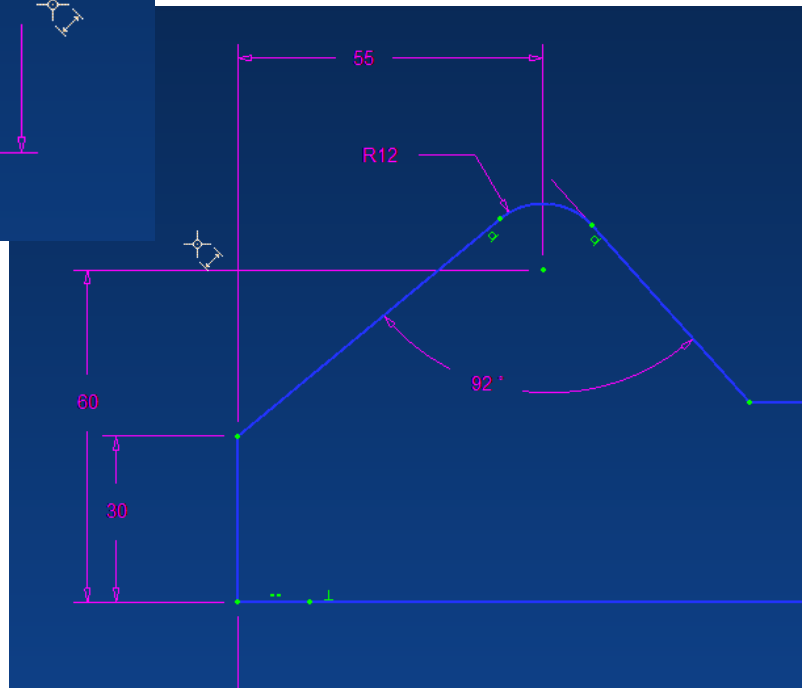
Select the 2D Fillet command and Insert two fillets as shown in the pictures at the right.

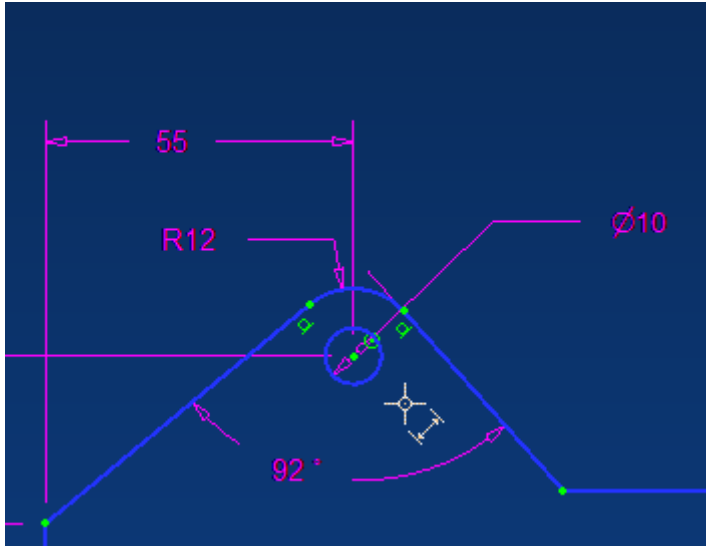


Dimension them as shown. When complete your sketch should look like the picture at the right, and the one below.

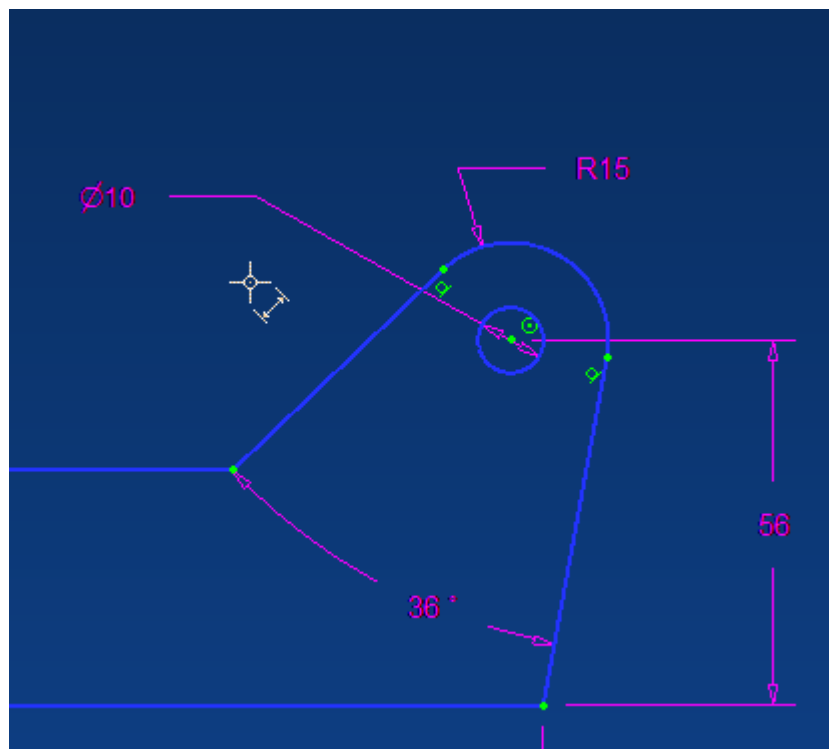


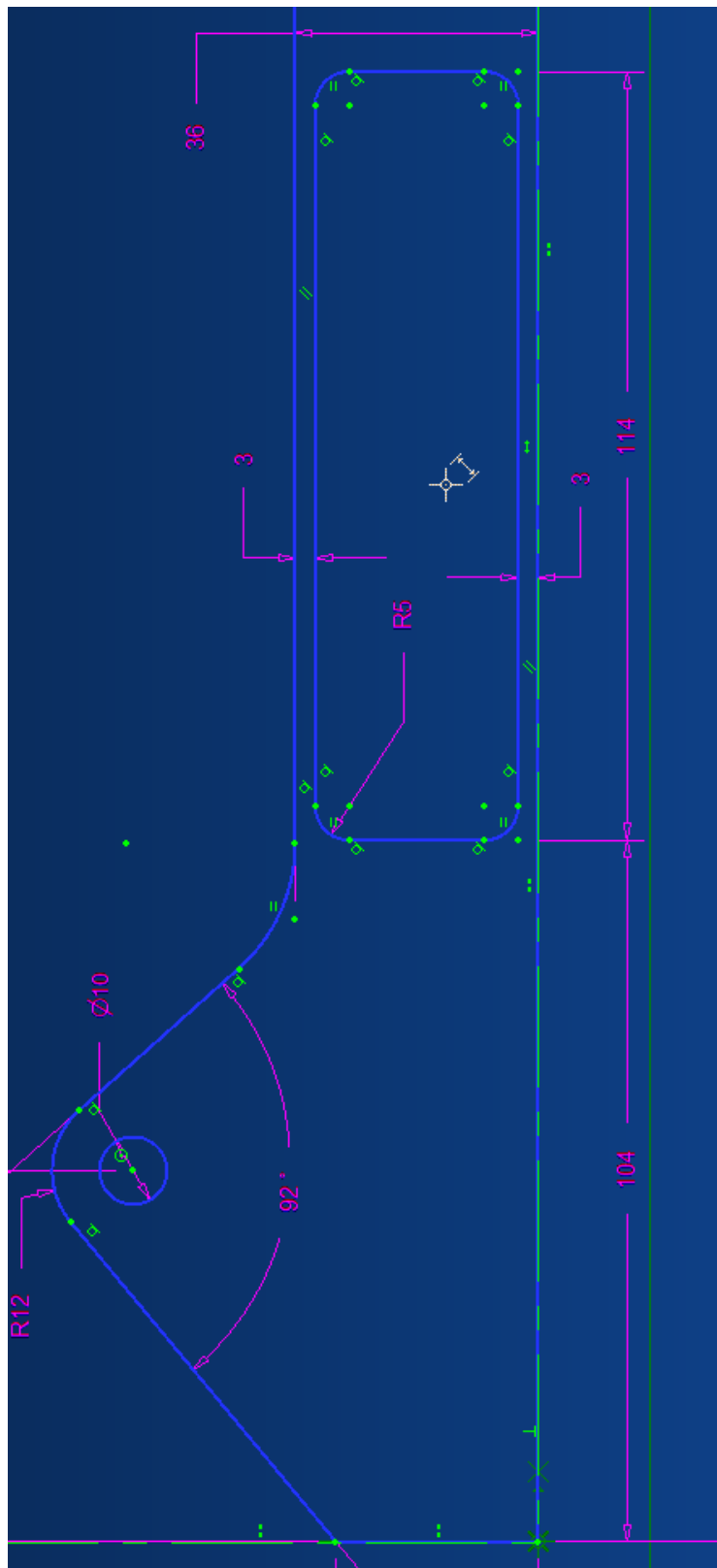
Add the 320mm and 55mm horizontal linear dimensions before adding the 56mm and 60mm vertical dimensions. Adding the dimensions in this order further constrains the overall shape of the sketch. Try it the other way to see what the results are.





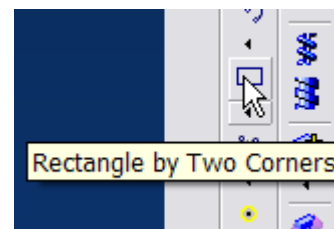
Insert the two 10mm diameter holes as show in the pictures. You could insert these holes in the finished model as a 'hole feature' using the 'Hole' command. This is considered a best practice rule when working with solid features. Exceptions to this rule include inserting holes, fillets, or chamfers in the sketch when such features constitute a basic, unchanging feature of the part, i.e. not a dress-up feature. In this case, we'll treat all holes and fillets as basic features of the casting.



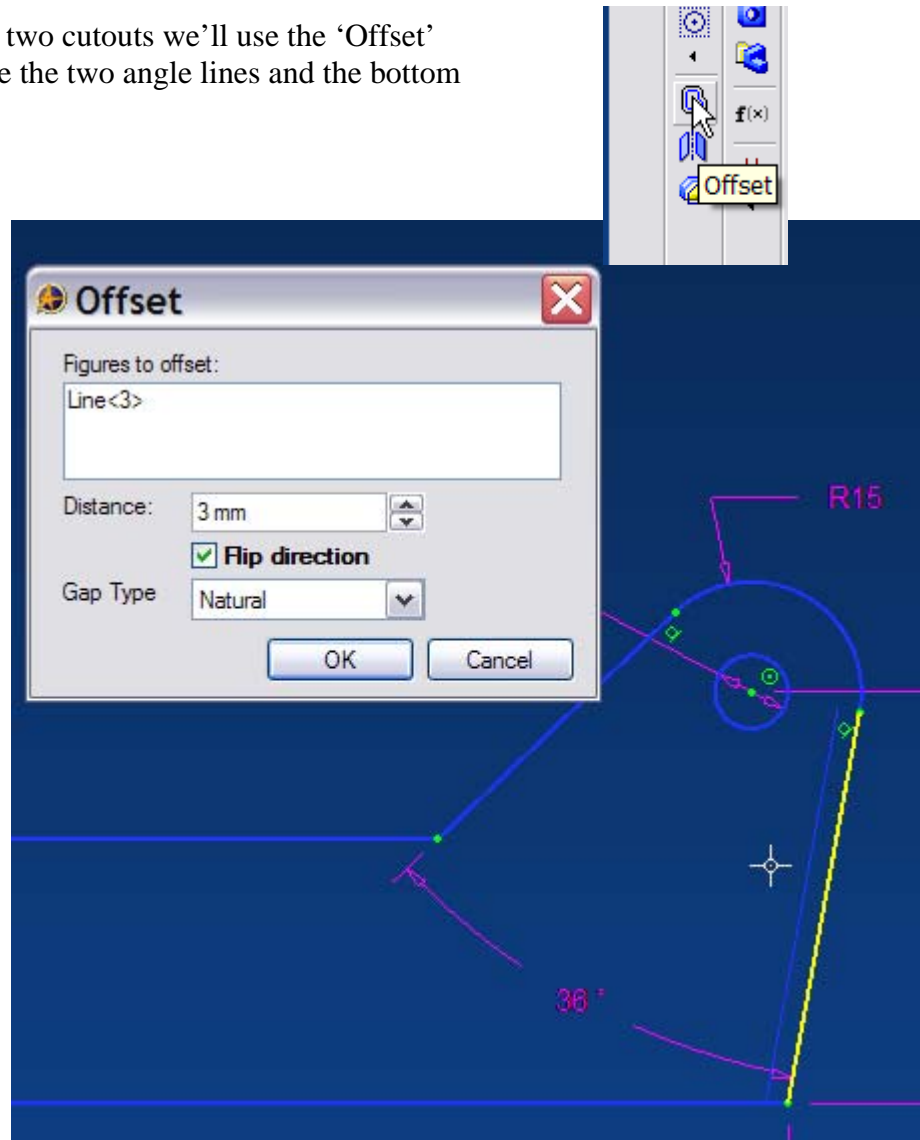


Any sketch created within the boundaries of the outer sketch will propagate as a cutout when the sketch is extruded. To create the rectangular cutout in the motor mount, sketch and dimension the shape shown at the left.

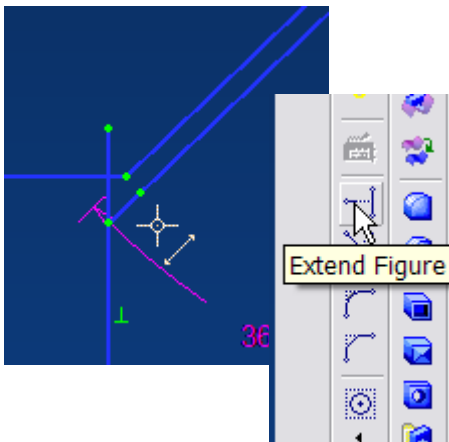
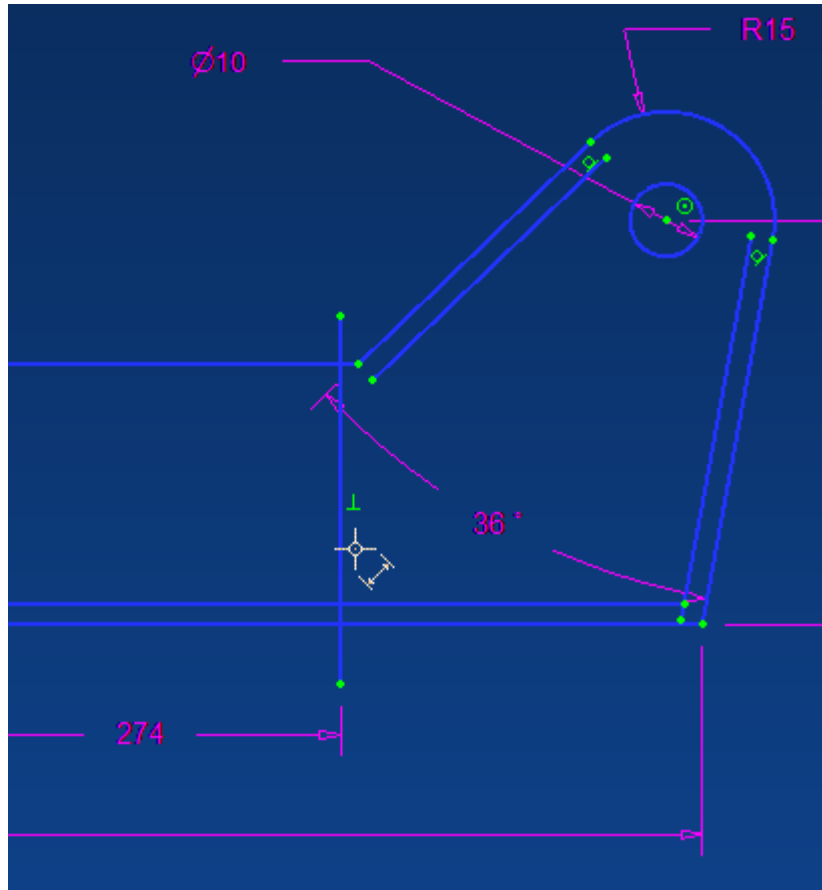
To make the task a little easier, use the Rectangle by Two Corners tool to create the initial sketch, then add the fillets and dimension it.



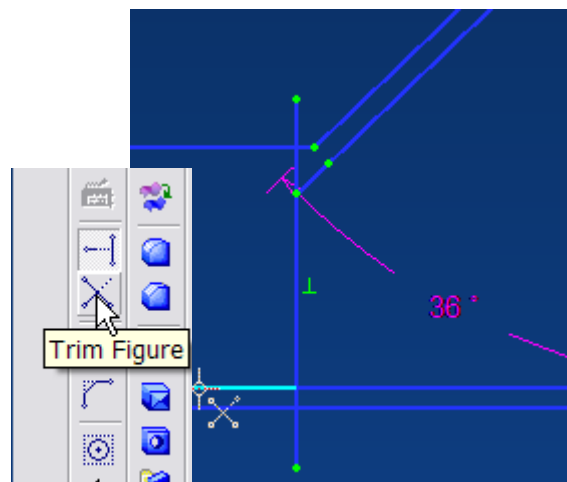
To create the next two cutouts we'll use the 'Offset' command to create the two angle lines and the bottom horizontal line.

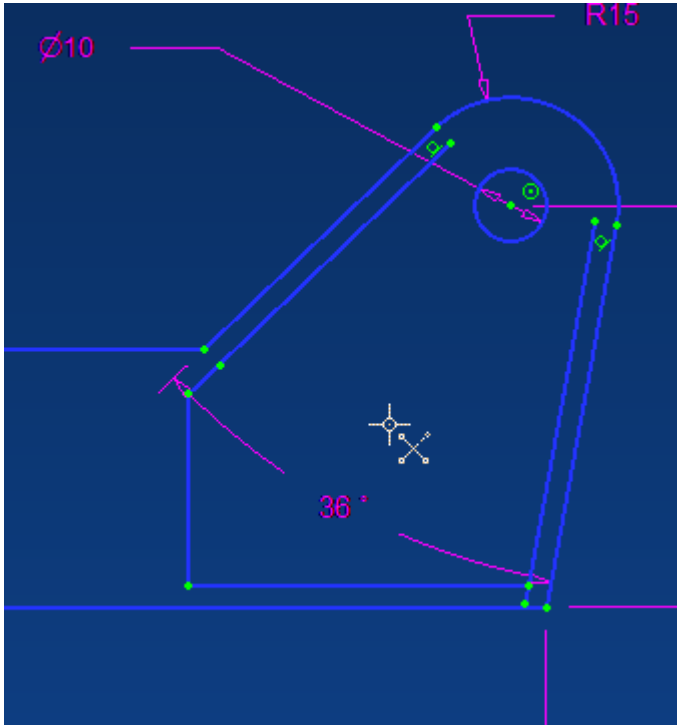


Now add and dimension the vertical line shown in the sketch below. The 274mm dimension is taken from the far left vertical sketch line.

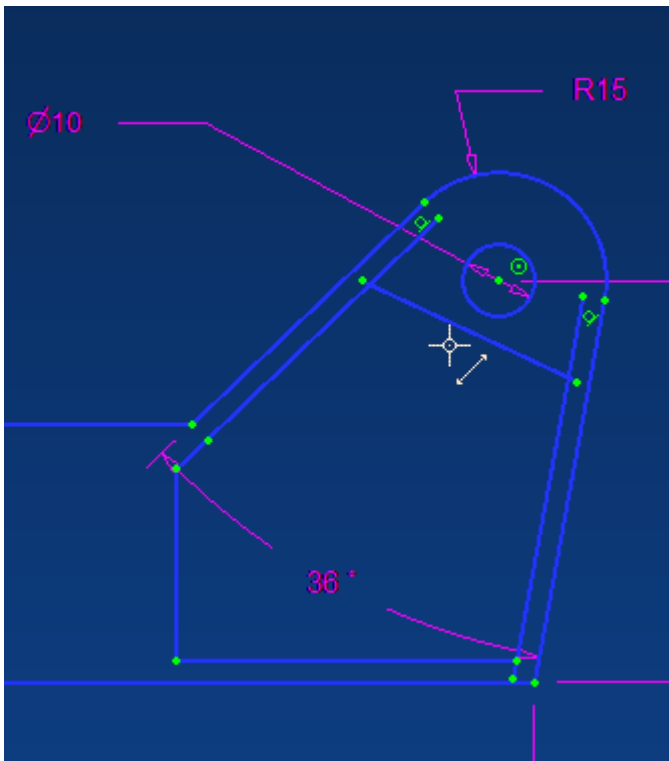


Now use the Extend Figure and Trim Figure commands to clean up your sketch.



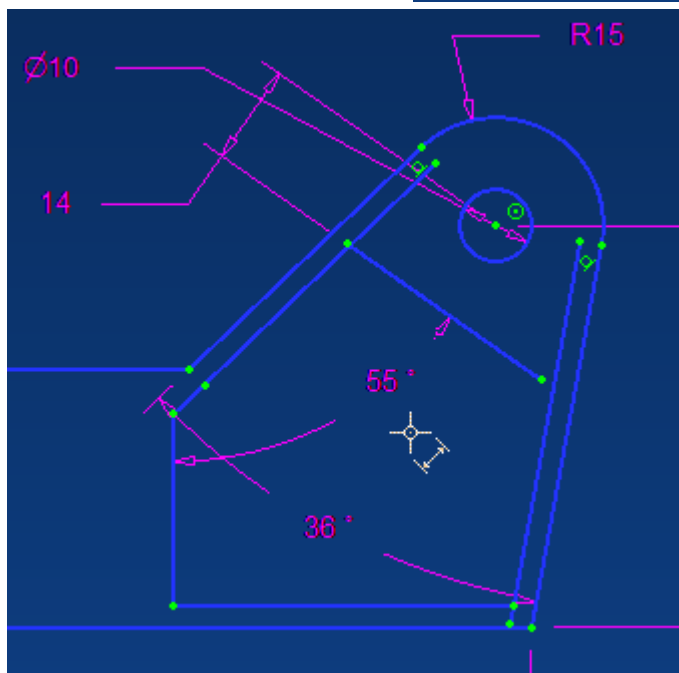
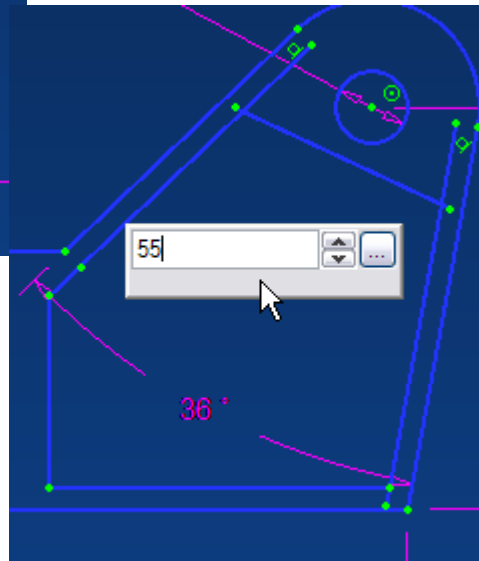
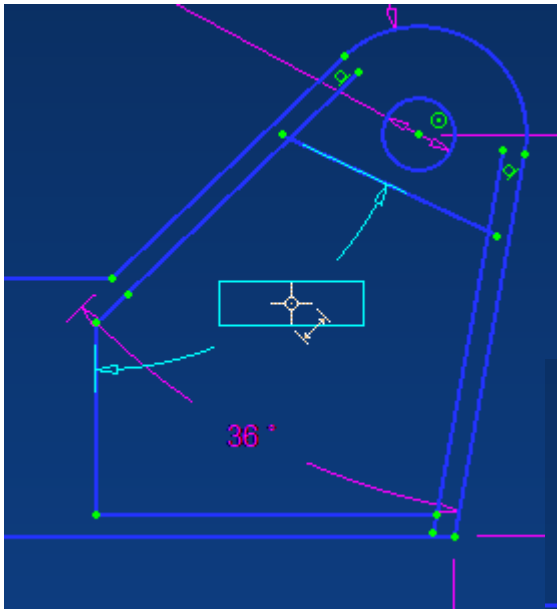


When finished the results should look like this,

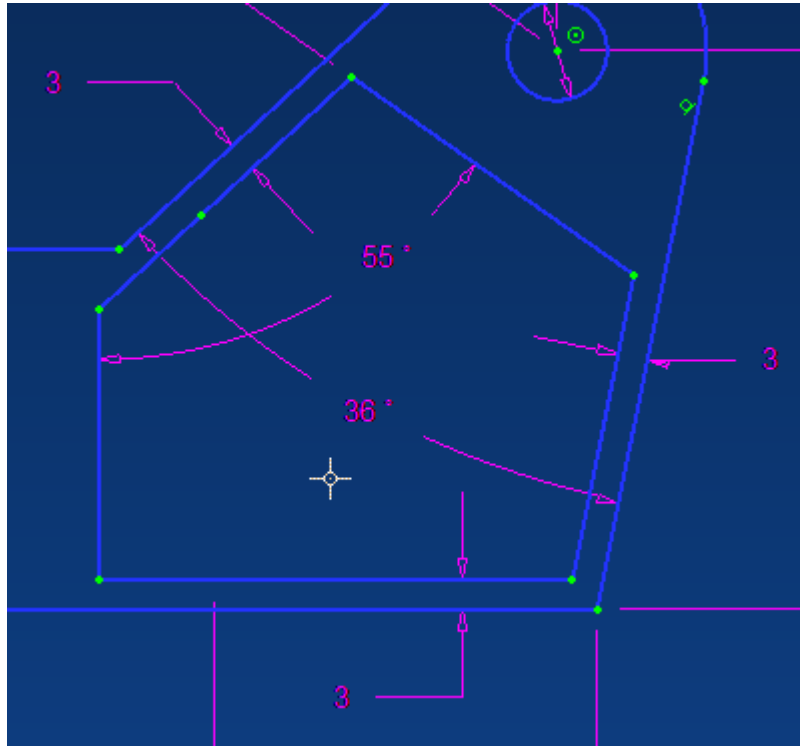


Add the angle line shown at the left.

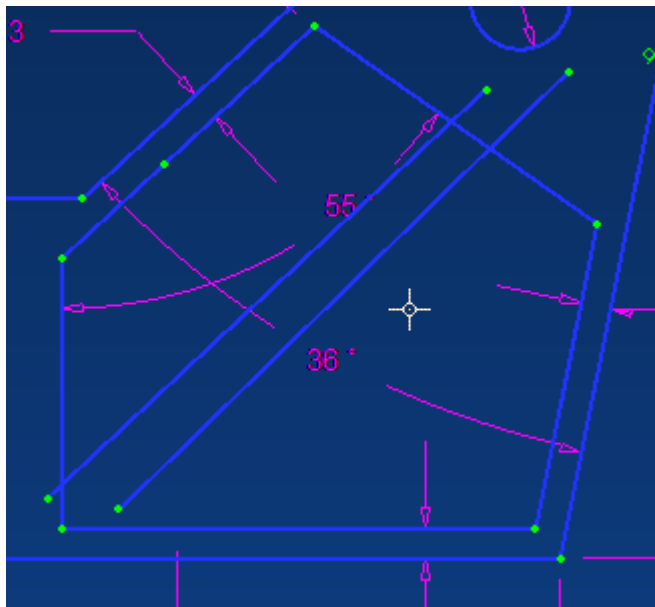
Dimension the line as shown.



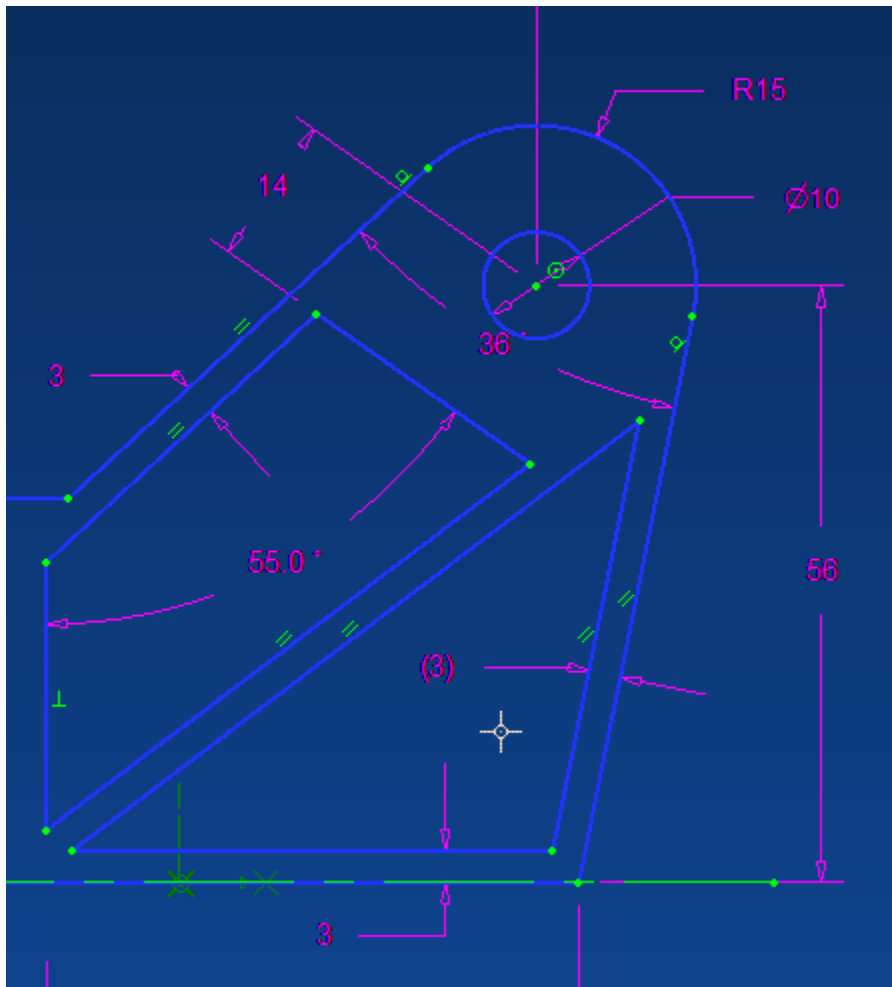
Trim the lines until you have the shape shown below.



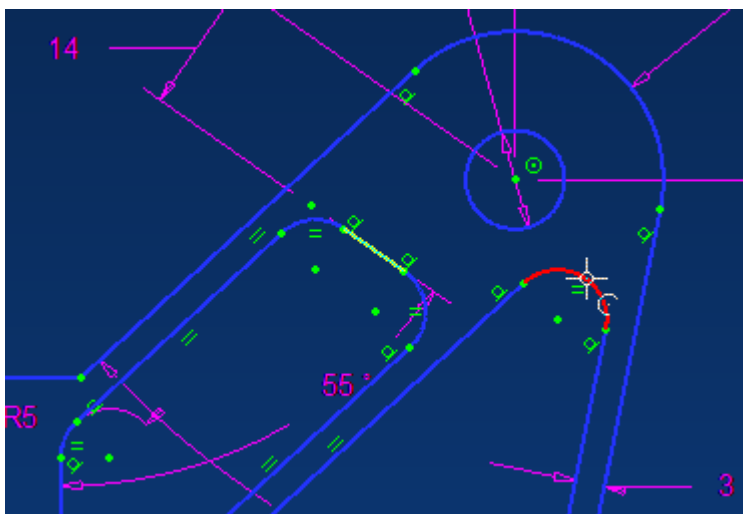
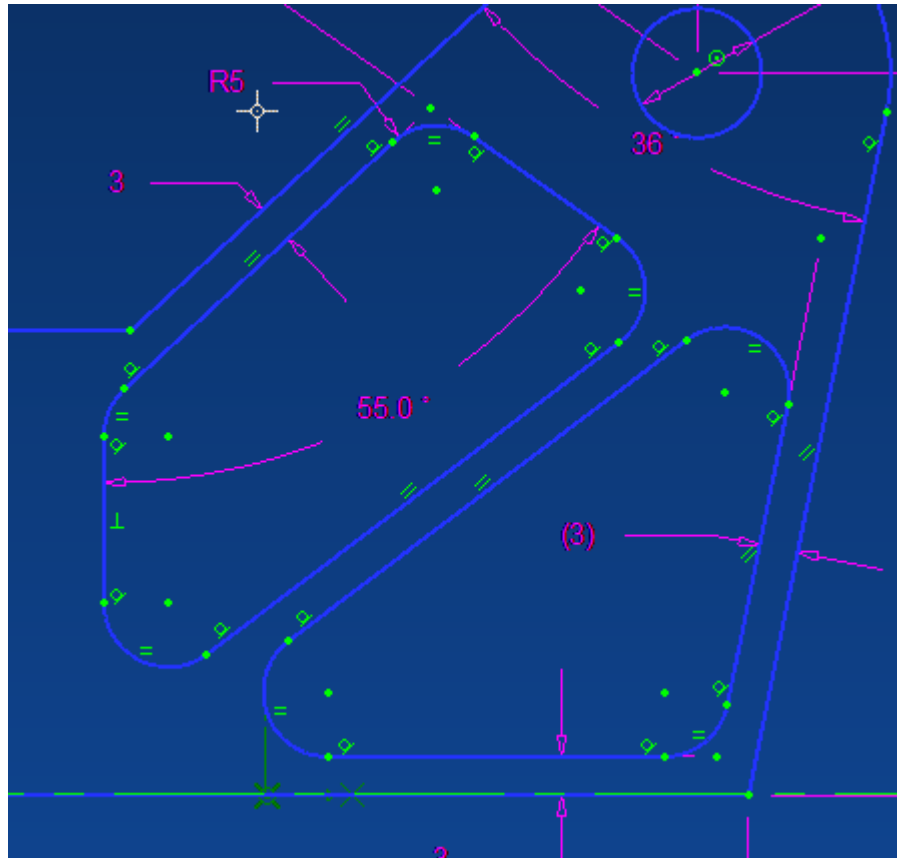
Insert two angled lines as shown.



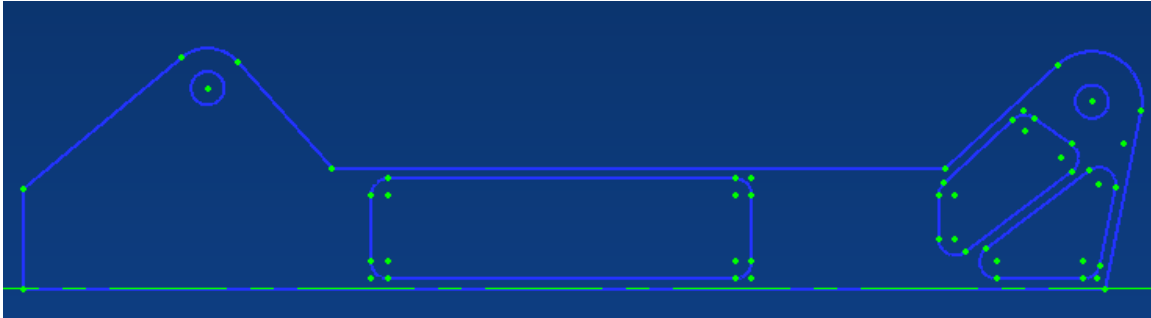
Extend and Trim the lines, and insert the parallel constraints as shown.



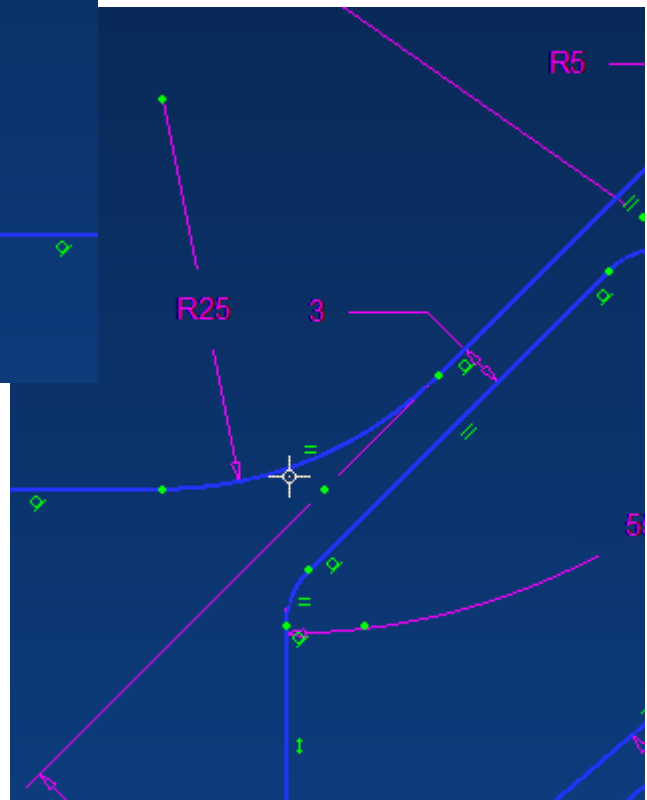
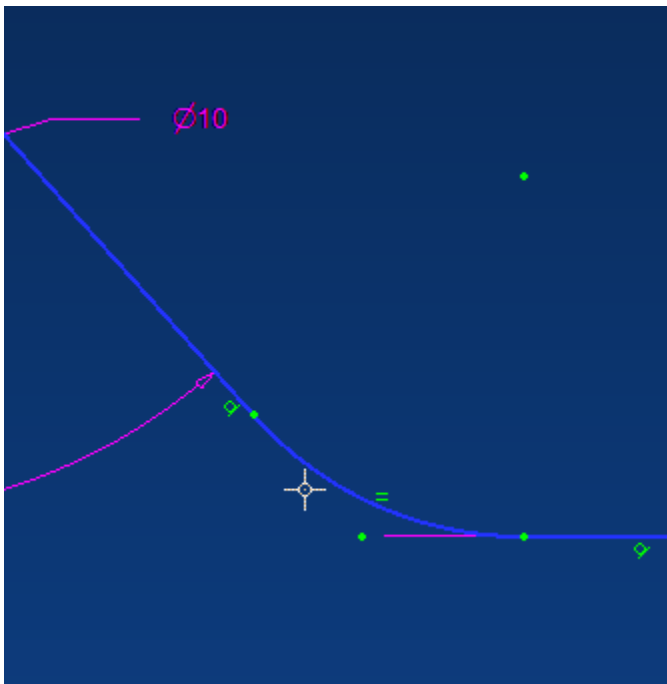
Next, insert the seven, 5mm corner fillets shown.



It might be necessary to add a Tangent constraint between the line and arc shown to bring the cutout figures into correct alignment. To do this, open the constraint tool, select Tangent Constraint, click on it to make it active, and then click on the line and then the arc.



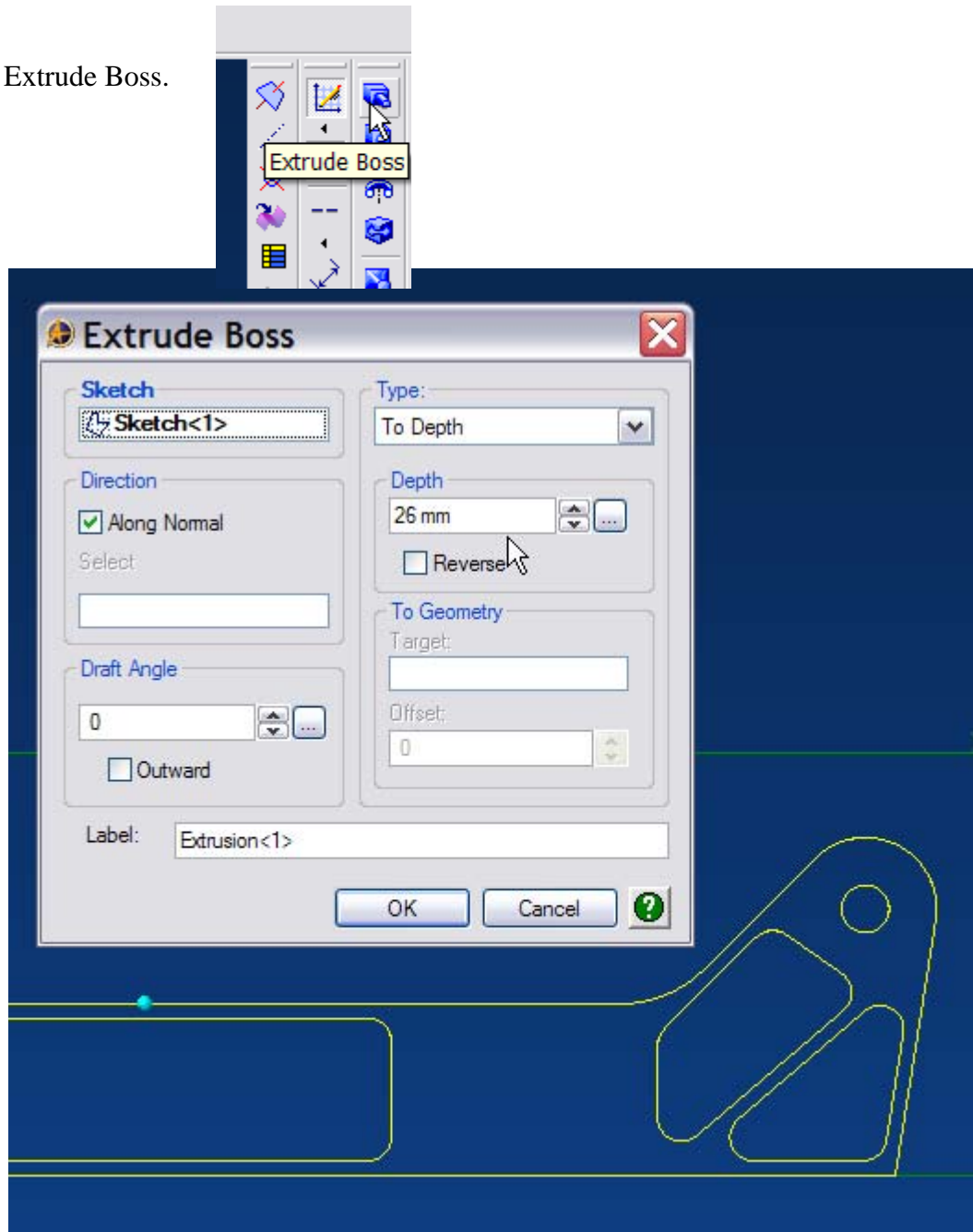
Your sketch should now look like this. (The dimensions and constraints have been hidden to clarify the sketch.)



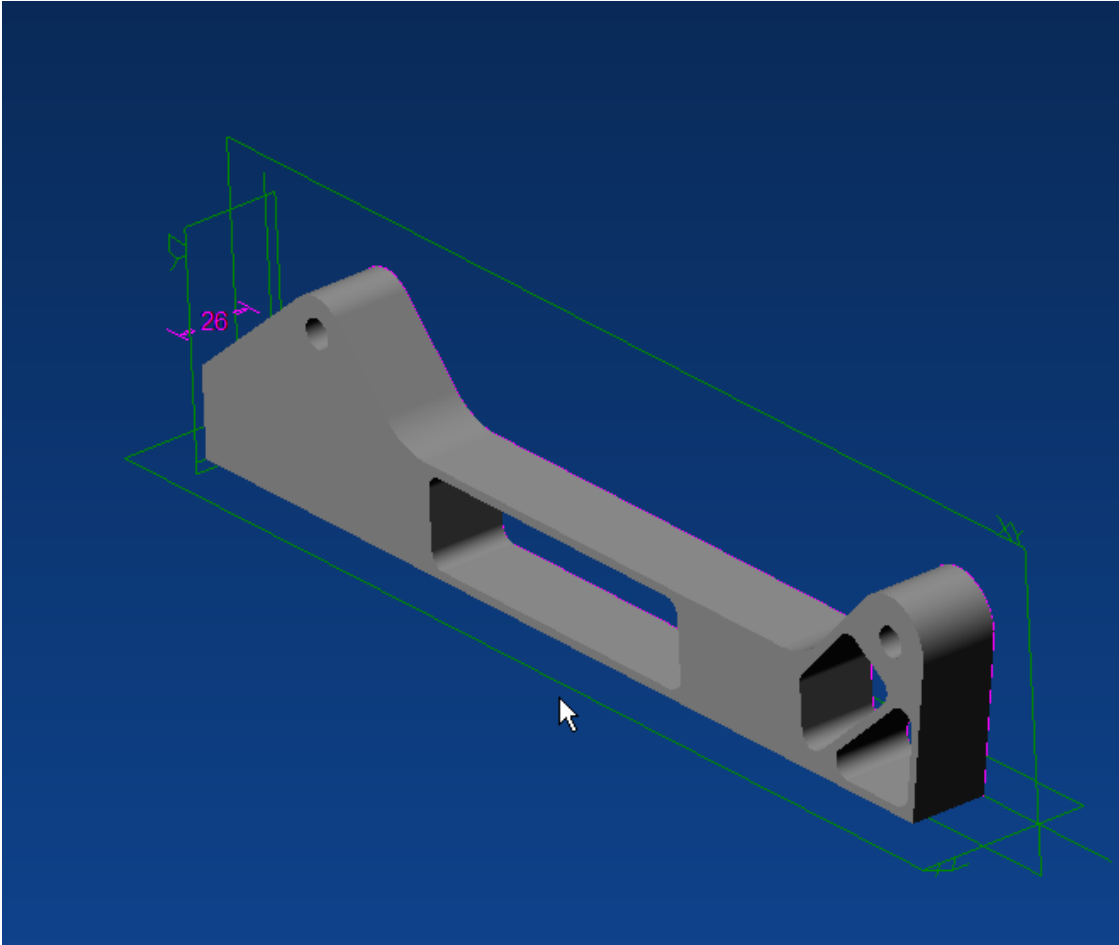
Now add the 25mm fillets. Again, we could create these as a dress up features after we extrude the basic motor mount shape, but for this case, we can consider them basic features of the part, not subject to change, so we'll include them as part of the sketch.

It's now time to extrude our sketch and create the basic, motor mount shape. It would be a good idea at this point to use the Sketch Analyze tool to check your sketch for problems before you extrude it.

Select Extrude Boss.

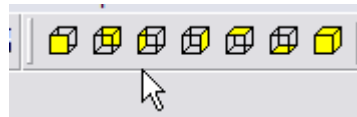


Select the sketch if it is not already selected, then select To Depth as Type, and enter 26mm as the Depth, then click OK.



Your part should now look like this.

Change the view orientation
in the position shown below.



until the motor mount is



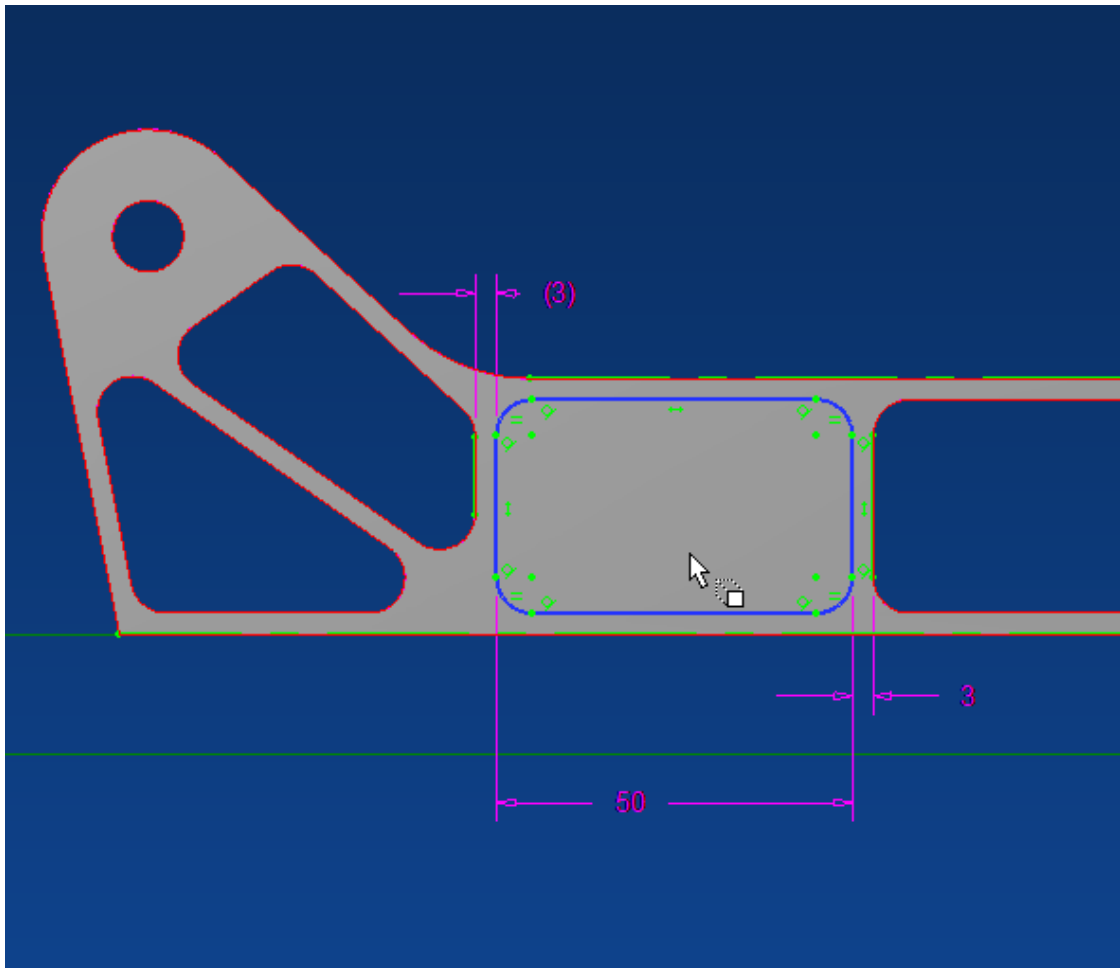
Select the face of the motor mount,

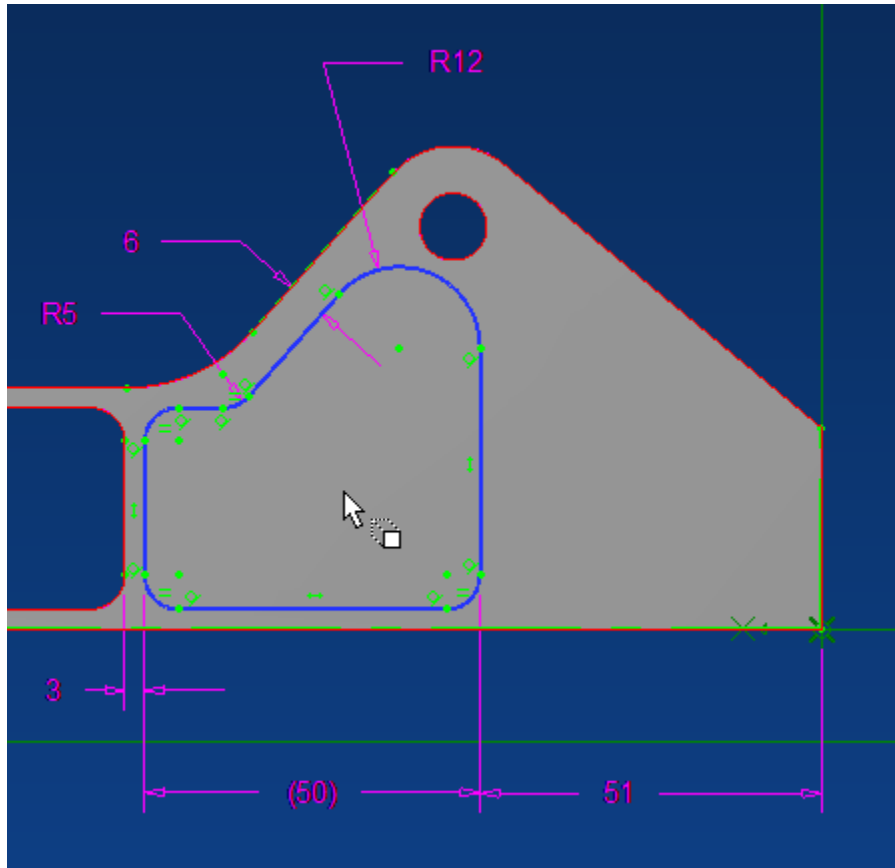


then select Activate 2D Sketch.

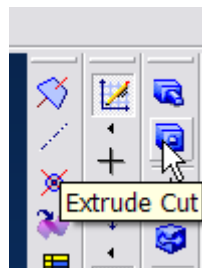


Now we need to add the two 18mm deep pockets. We'll add both pockets at the same time. Create and dimension the sketches shown in the following two pictures. Use the Rectangle by Two Corners tool for the left hand pocket.

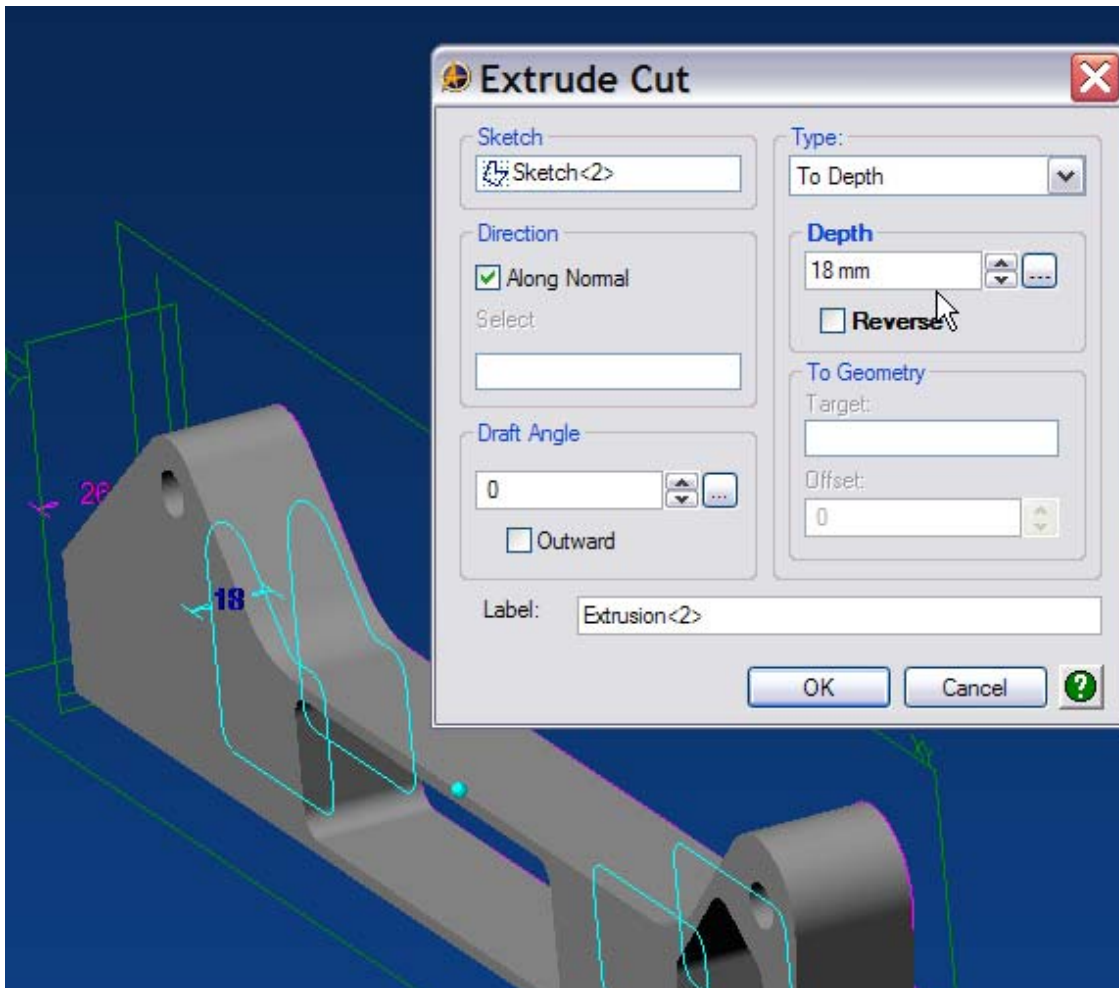


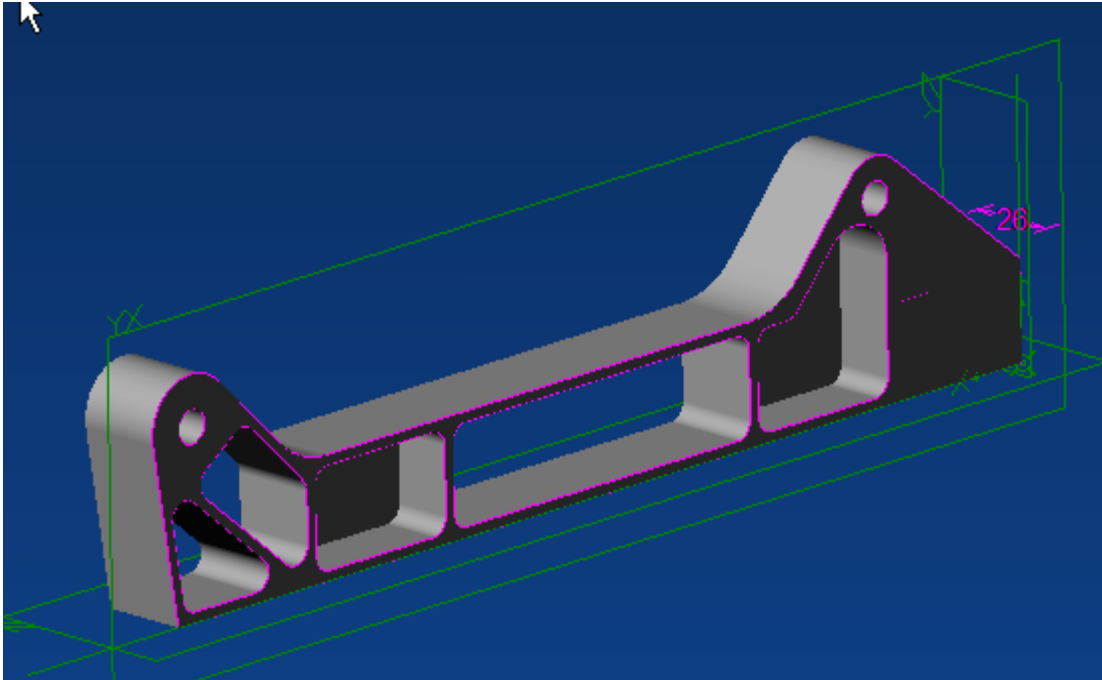


Click on Extrude Cut.



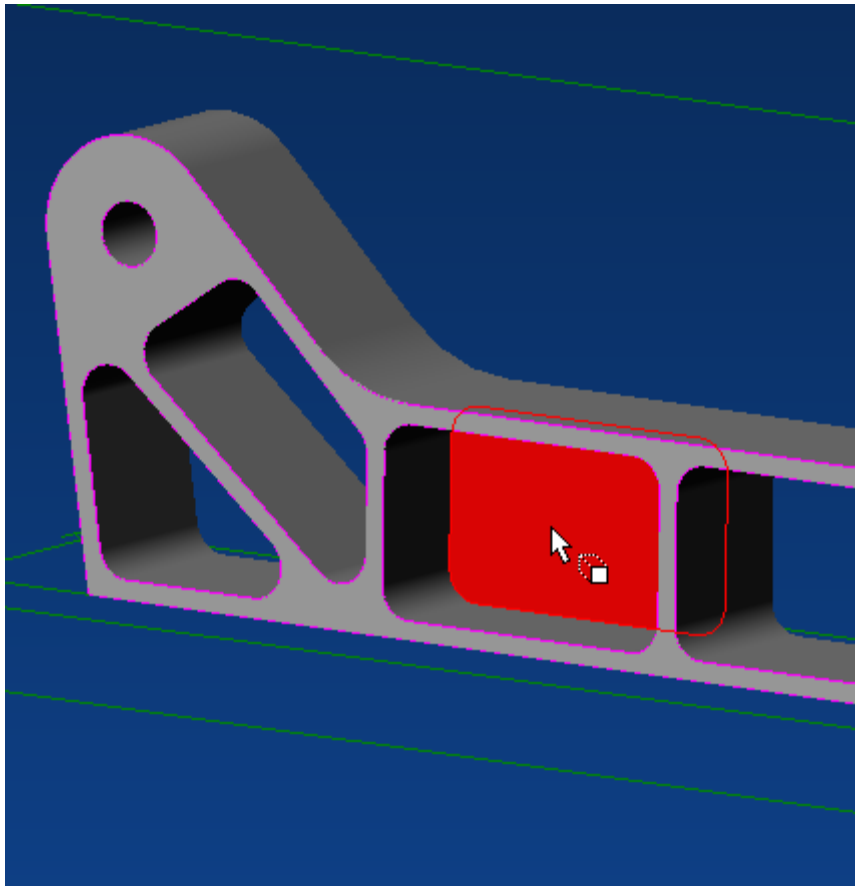
Select the newly created sketch if it is not already selected, select To Depth as the Type, insert 18mm as the Depth, and click OK. (Part rotated to clarify direction of depth cut.)

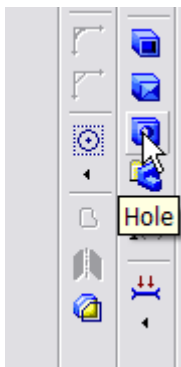




Your part should now look like this.

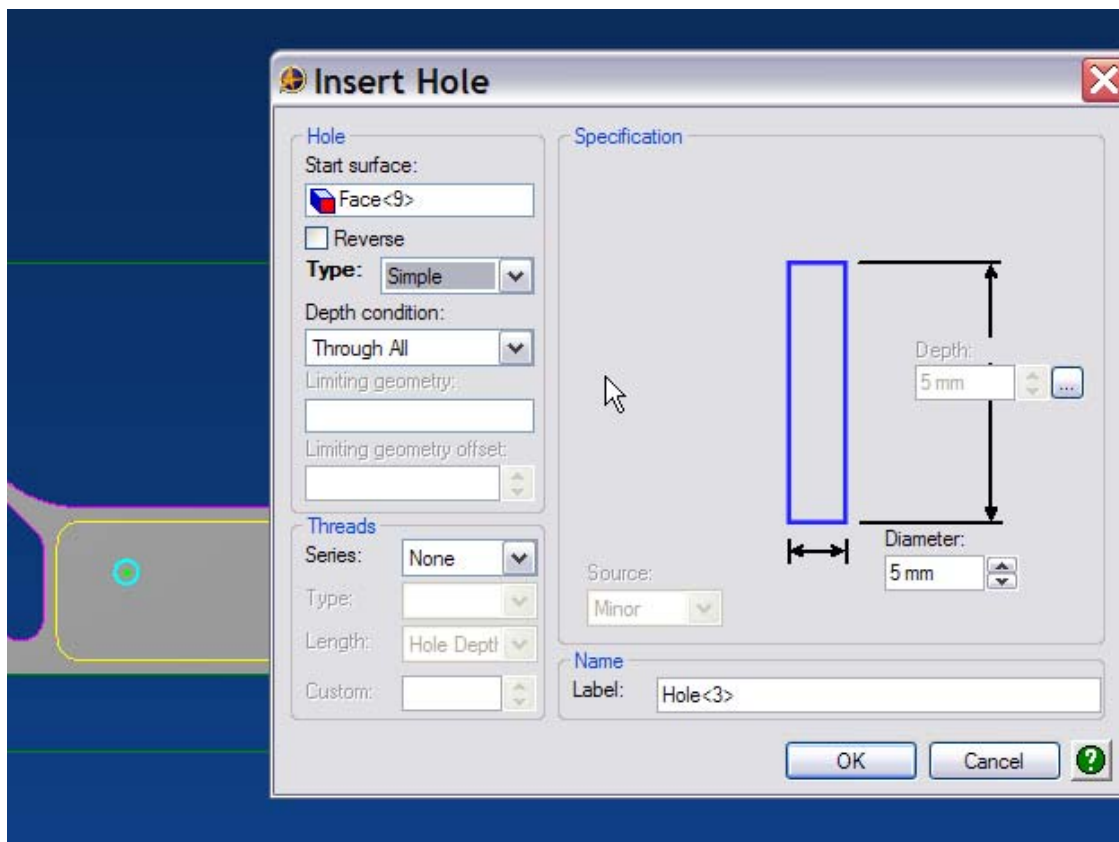
Now we need to add the two sets of holes. Select the bottom face of the first pocket. then select Activate 2D Sketch.



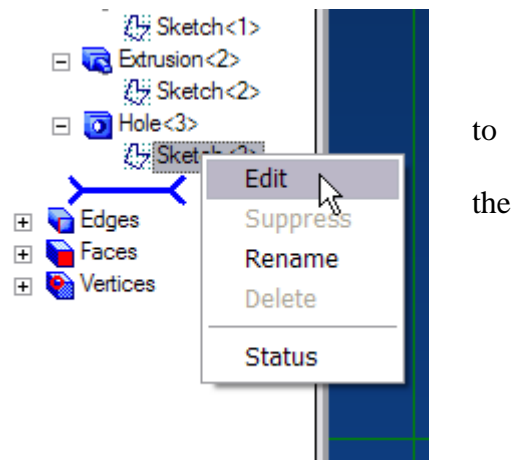


Click on the Hole icon.

The Insert Hole Panel opens. Set the diameter to 5mm, the Type as Simple, the Depth condition as Through All, and then click OK.

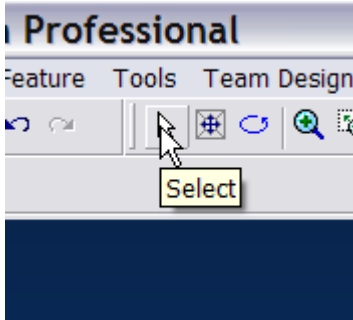


The hole appears in the pocket. We now need to properly locate the hole. In the Design Explorer panel select the sketch associated to hole, right click, and select Edit.

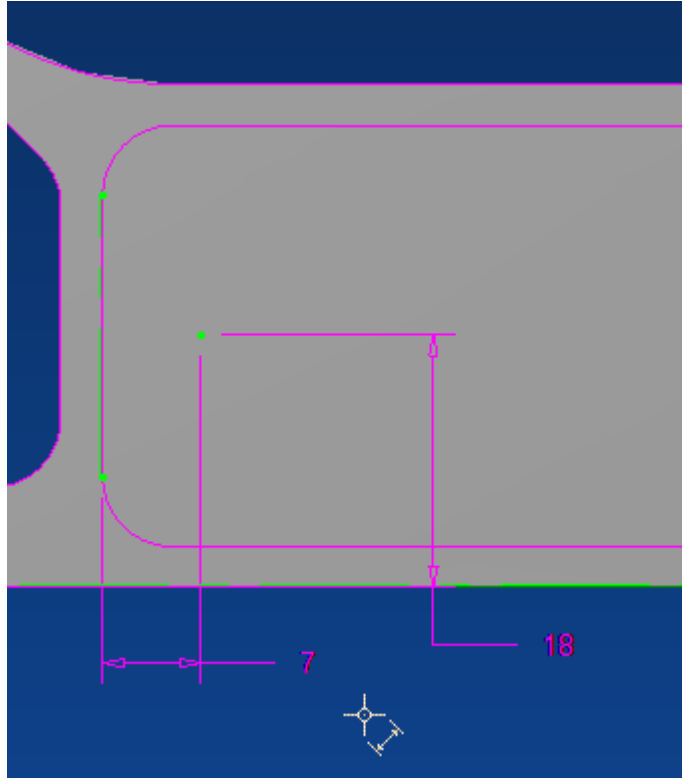


to
the

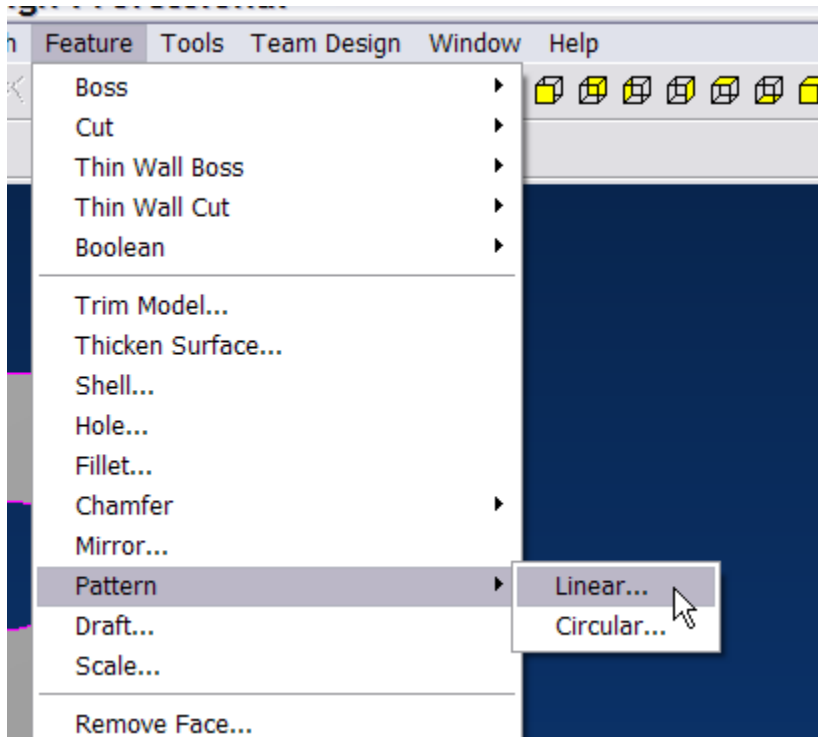
Locate the Hole center per the following dimensions. When finished, click the Select arrow to end the Edit process.



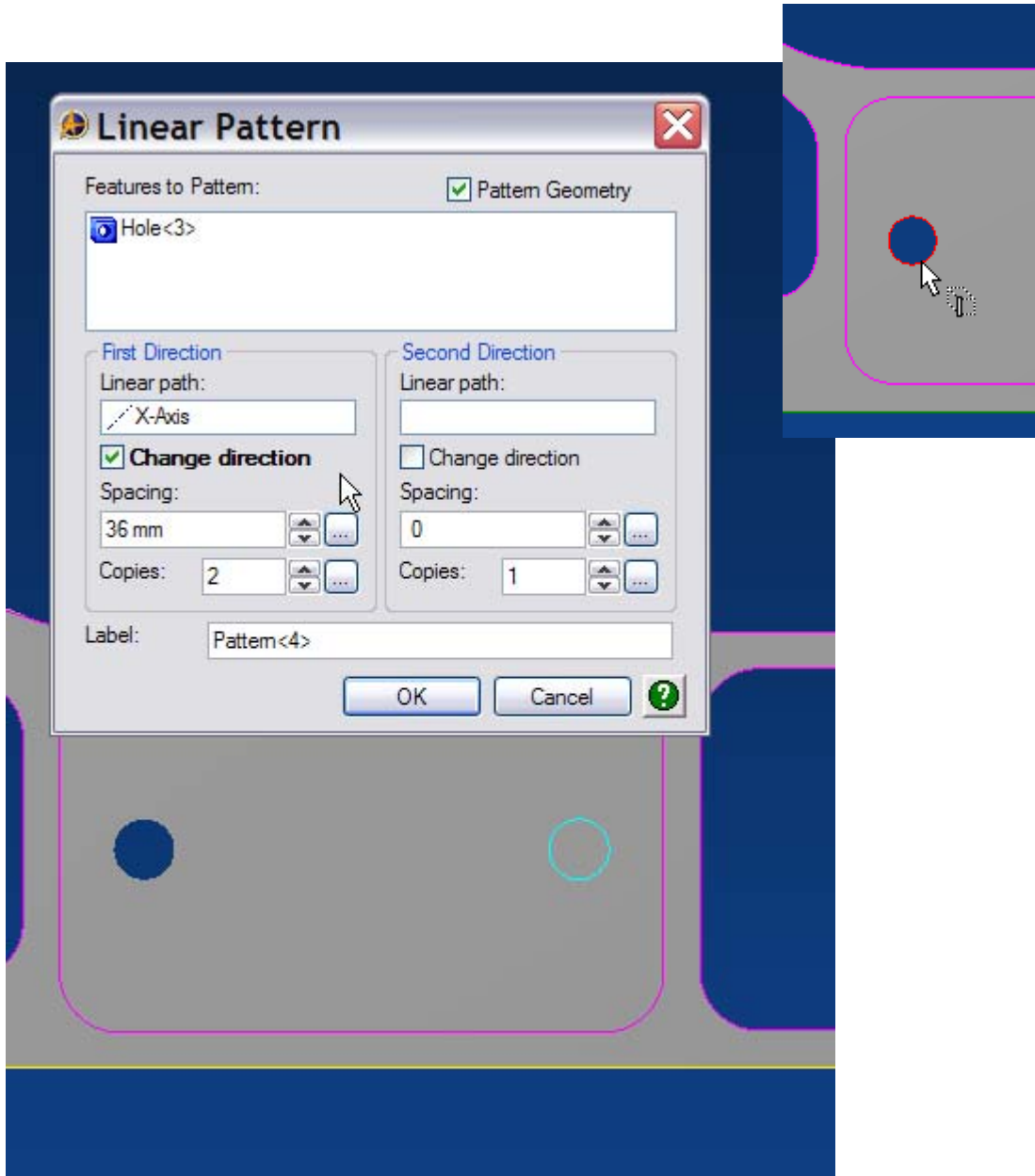
The hole is now located in the correct position.



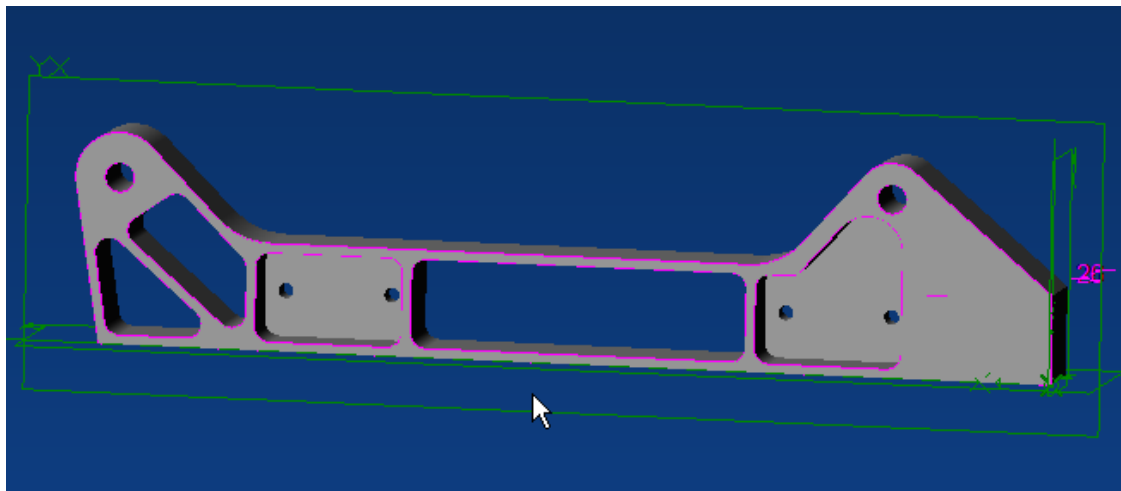
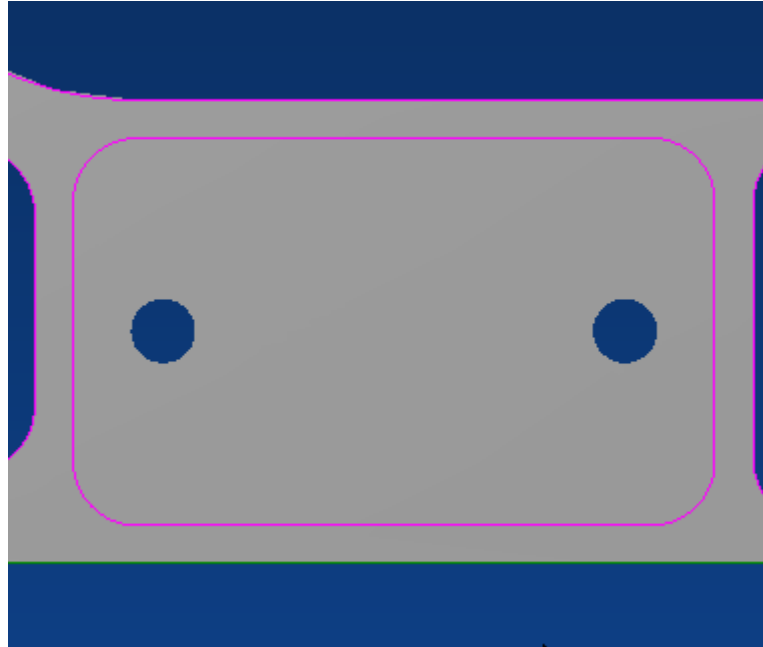
Per the drawing, we need to locate the next hole 7mm from the opposite inside wall of the pocket, and 18mm from the bottom edge. In this part of the exercise, we'll use the Feature >Pattern>Linear command to locate the second hole instead.



The Linear Pattern panel will open. Select the hole as the Feature to pattern. You can do this from the Design Explorer panel or by selecting it directly from the model. Make sure Pattern Geometry is selected, click on the X axis either in the model window or select it from the Design Explorer, enter 36mm as the Spacing, and select copies as 2. A hole outline will appear. If necessary click Change direction, then click OK.

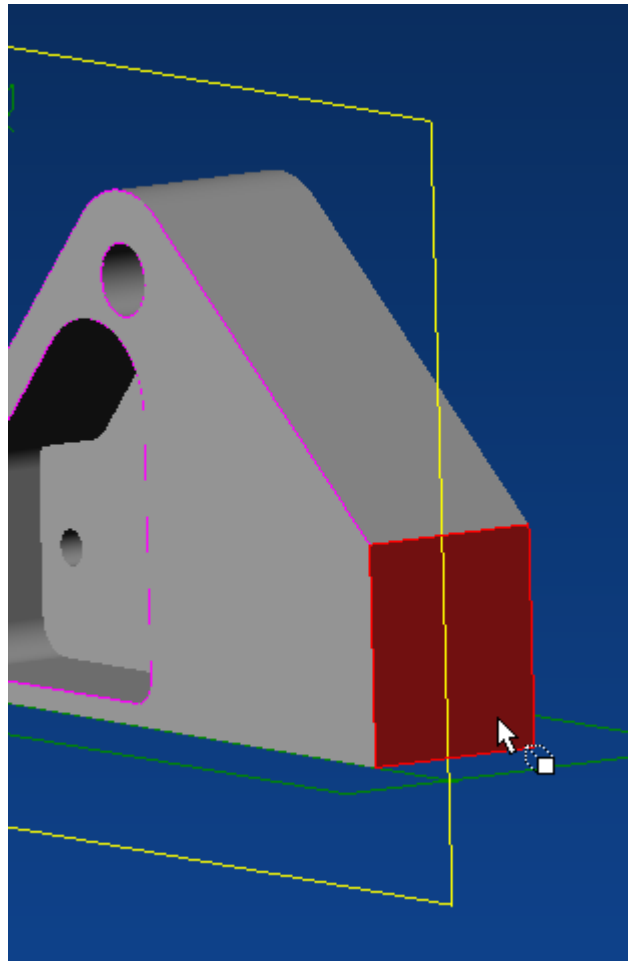


The new hole appears.

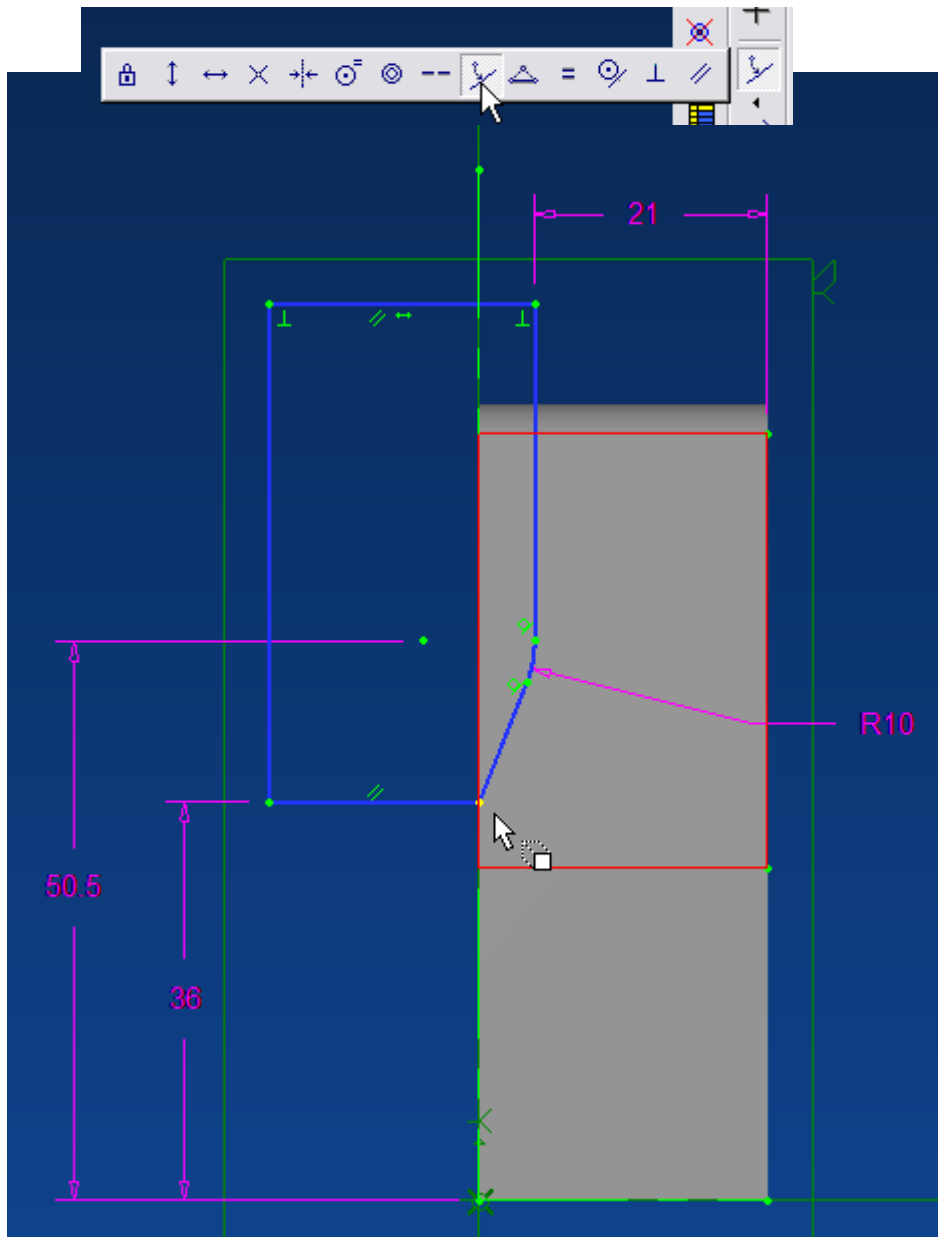


Repeat this process for the second set of holes. When complete your part should look like this.

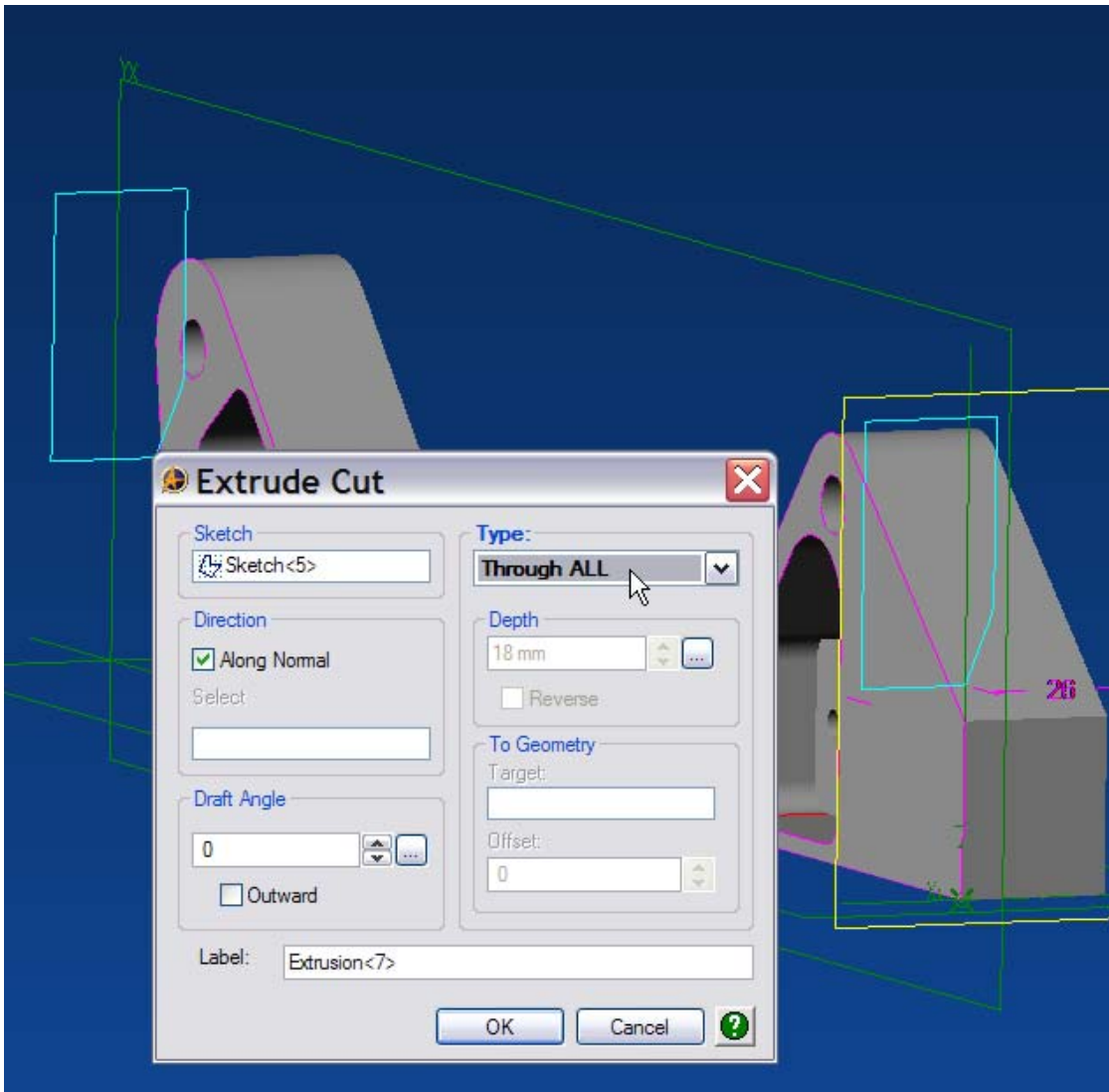
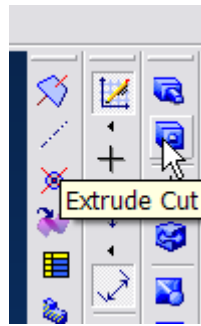
The final step in the design process is to create a cut on one face of the motor mount. Select the flat face shown as your sketch base, and then select Activate 2D Sketch.



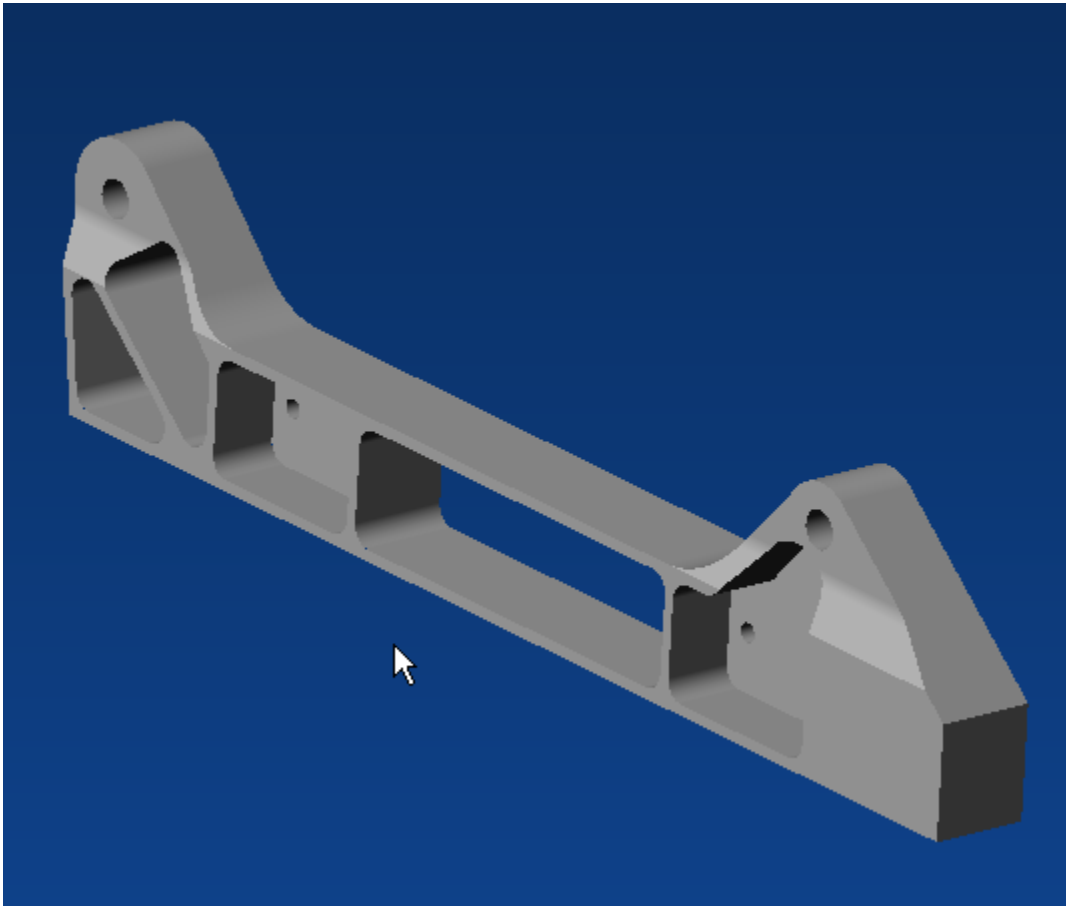
Create and dimension the sketch shown below. Constrain the point indicated to the line representing the front edge of the motor mount with a Coincidence constraint.



Select Extrude Cut.



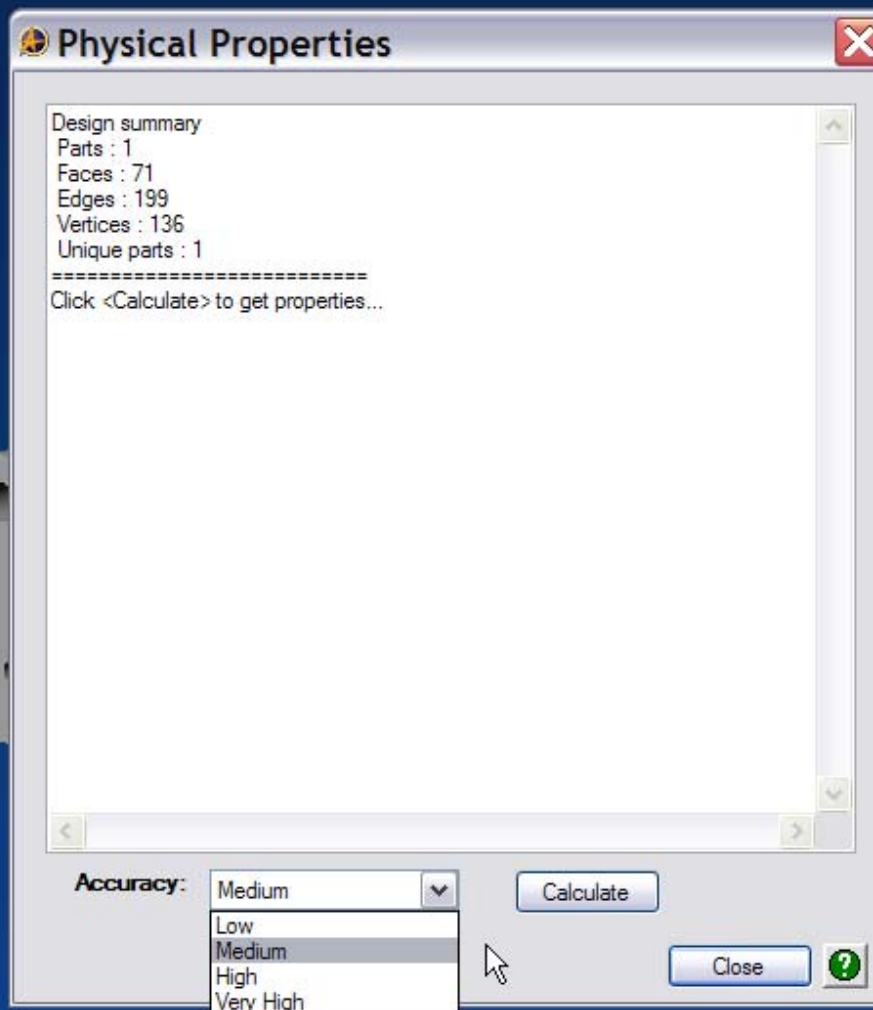
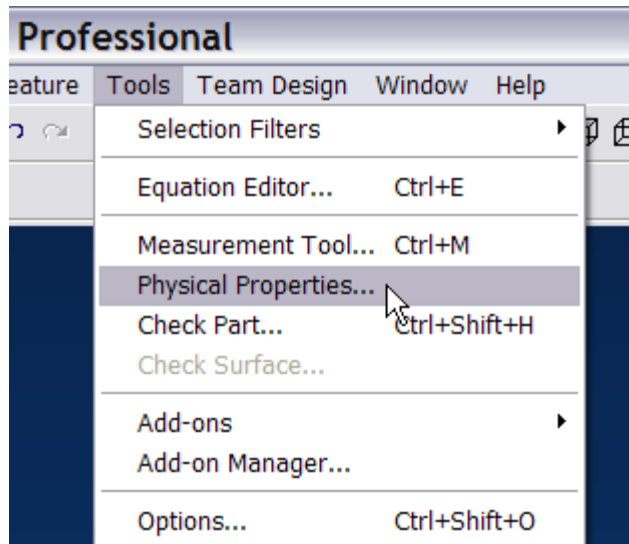
In the Extrude Cut panel, select the sketch if it's not already selected, select Through All as Type, and then click OK.



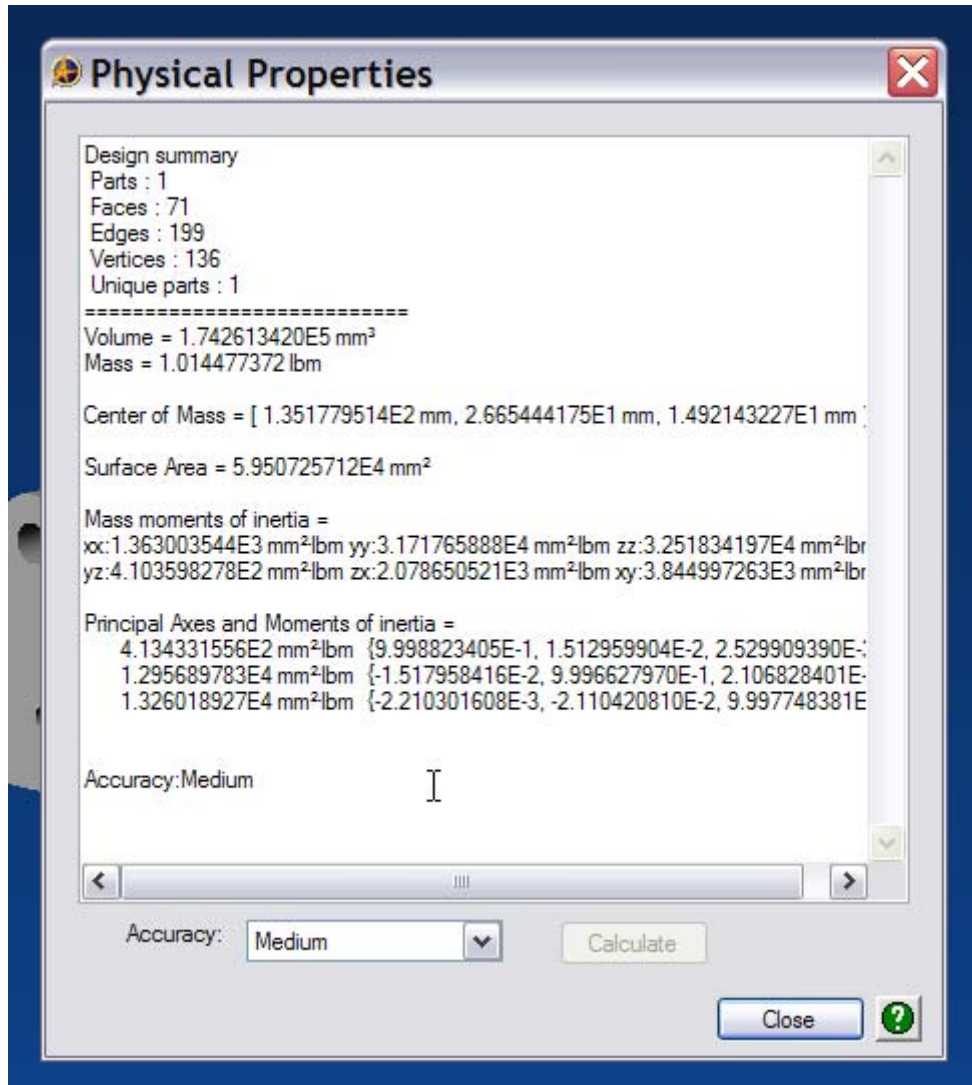
Your part should now look like this.

Because we specified the type of material from which the motor mount is fabricated, we can retrieve specific physical properties from our finished model. Click on Tools>Physical Properties.

In the Physical Properties Panel, select the level of Accuracy desired, and then click Calculate.



The Physical Properties are returned.



This concludes the Motor Mount Tutorial. I hope you've enjoyed creating the part, and hope you learned a few things along the way. Look for other tutorials in the future.