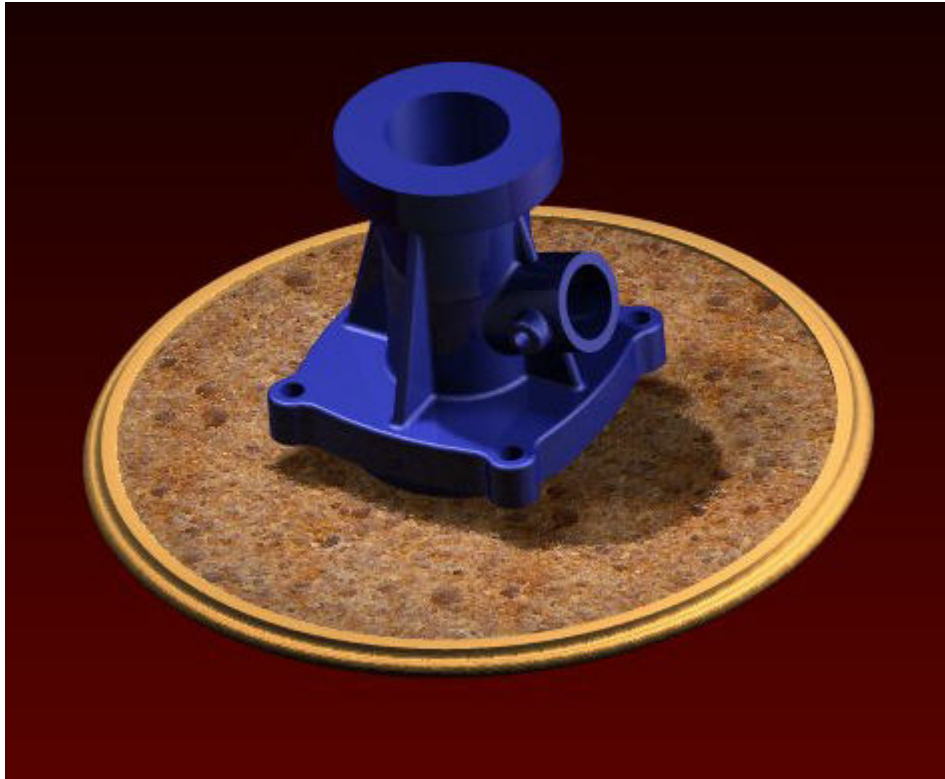


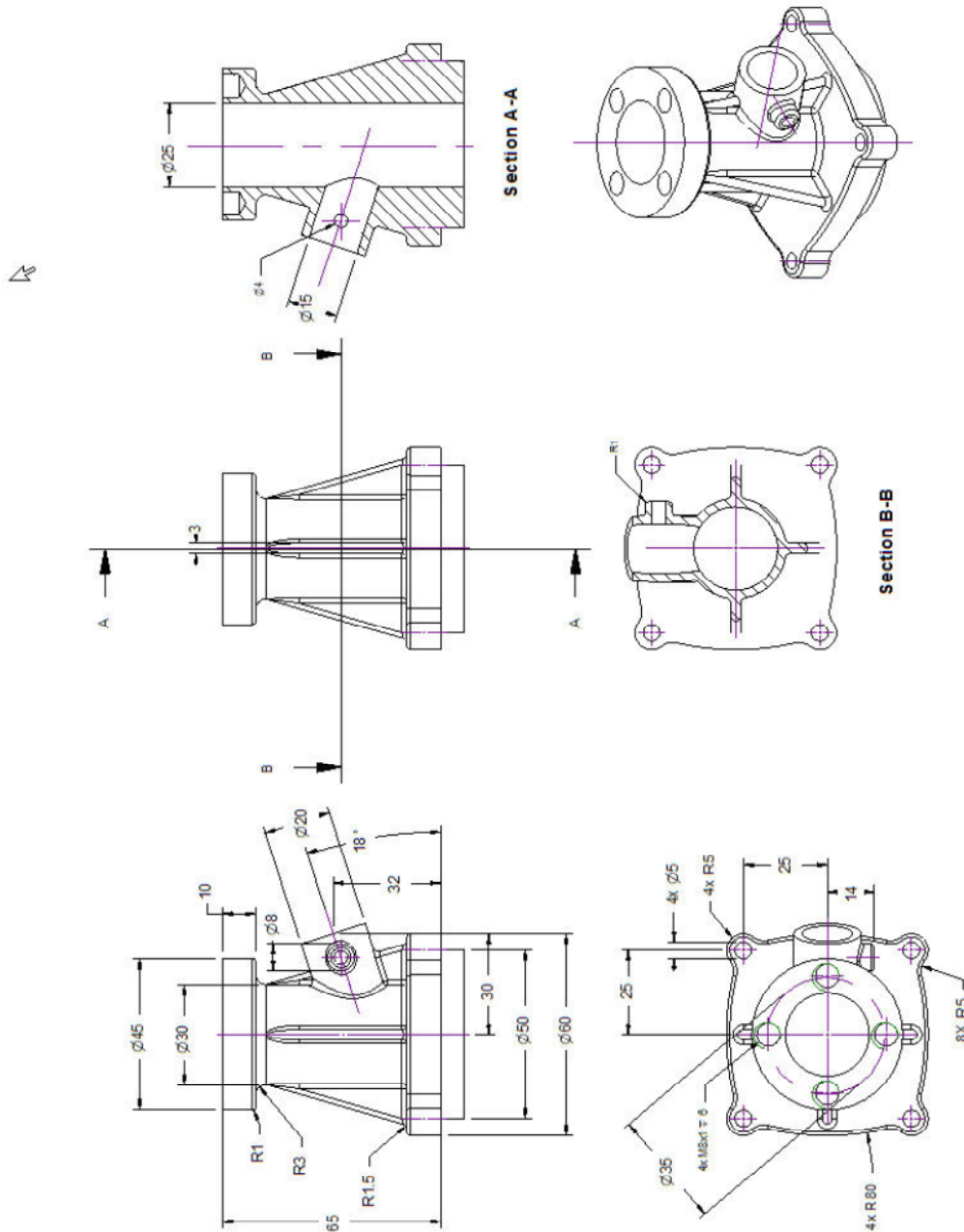
Inlet Manifold
An Alibre Design Tutorial
By
CIS



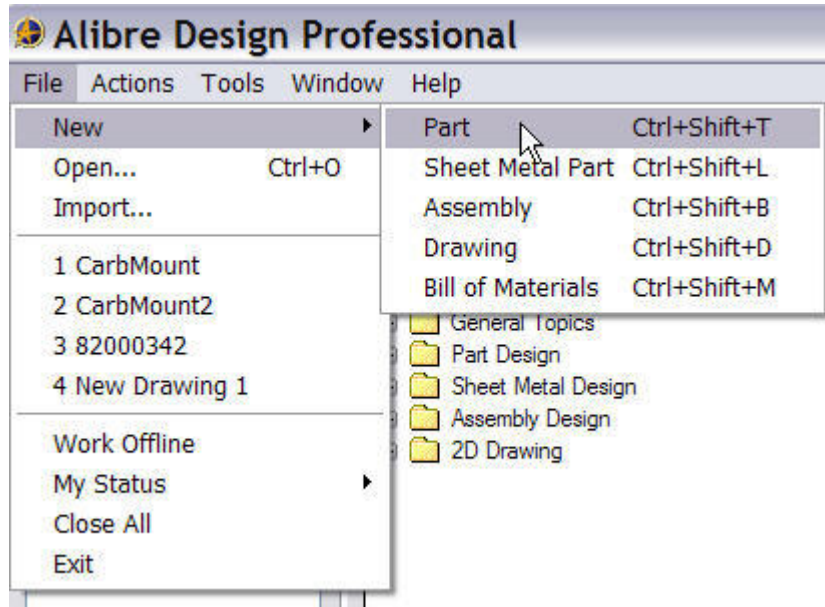
Written by Steve Carr
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This tutorial will introduce you to many of the tools, workbenches, and methods you will use in your daily work. The part you'll create is an aluminum casting. It should take less than two hours to complete. Many aspects of basic design work are touched on in this exercise, but no single tutorial can show-case all of the capabilities of Alibre. Please feel free to contact me with any corrections, or suggestions on ways to improve the material.

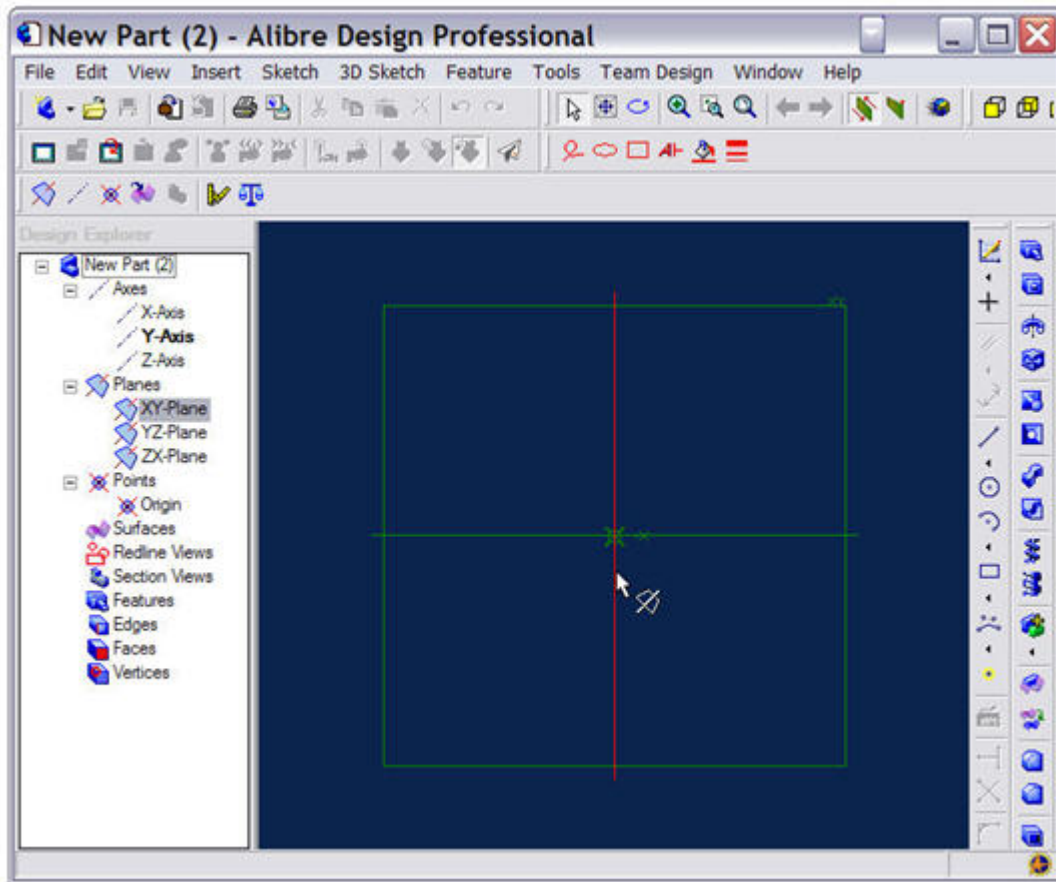
The drawing below will give you all the information required to create the Inlet Manifold and serve as a reference in the part design process.



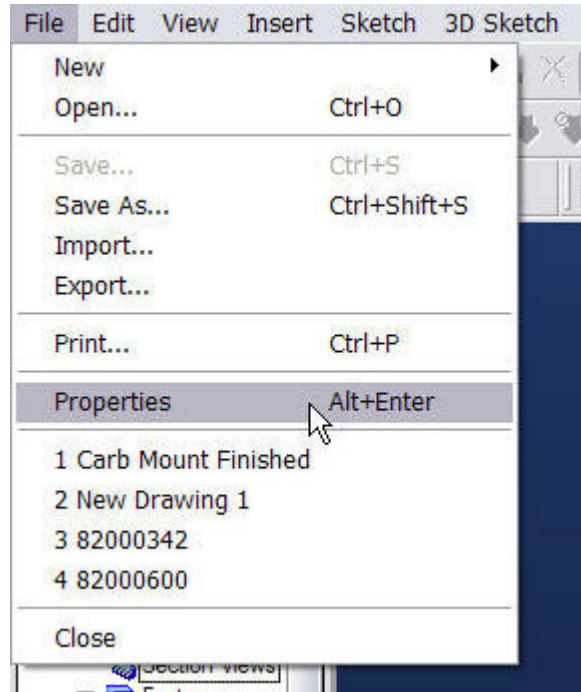
We need to set a few basic parameters in Alibre before we start the actual design process. In the Alibre Design Main panel, click on File>New>Part.



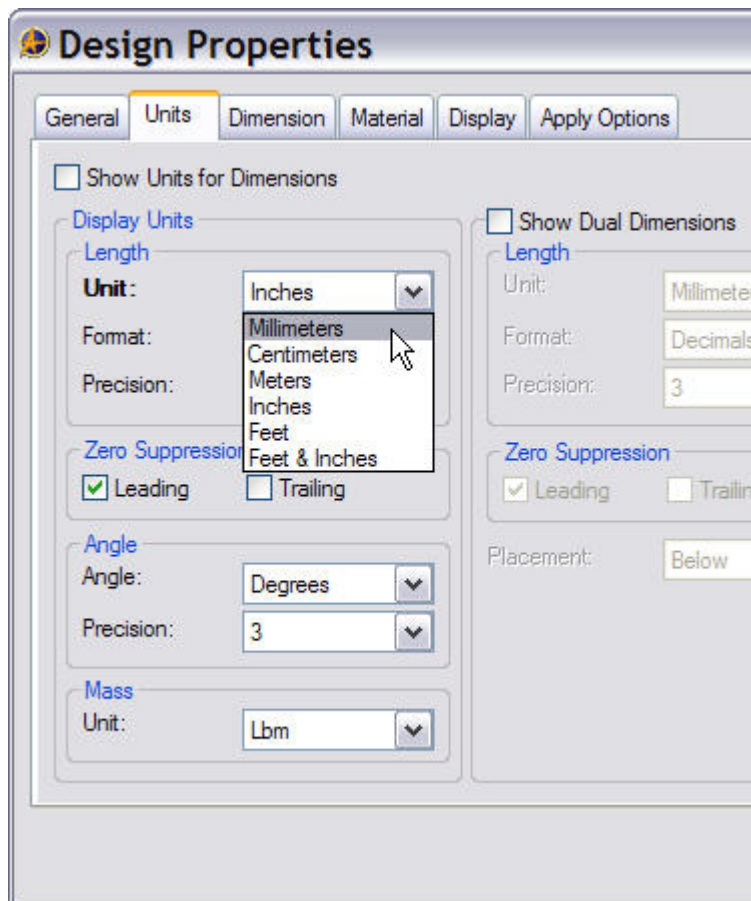
The New Part window opens.



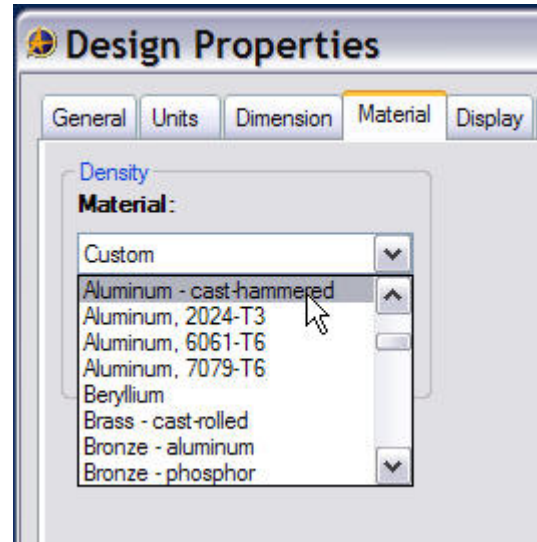
Under the File menu tab, select Properties.



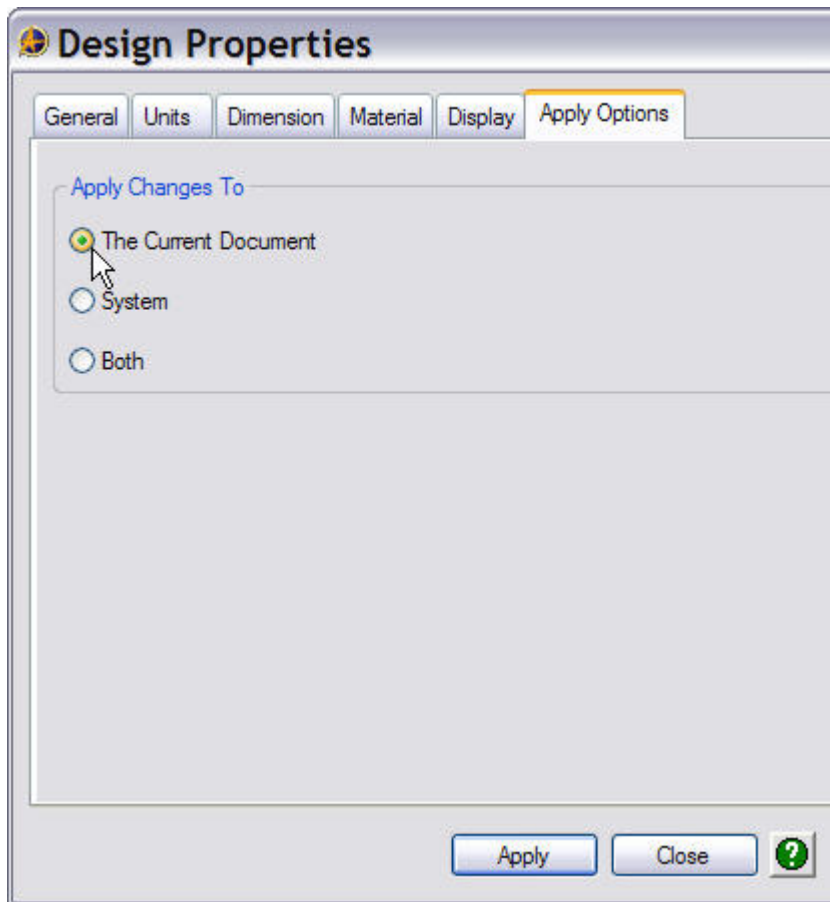
The Design Properties panel will open. Click on the 'Units' tab and select Millimeters.



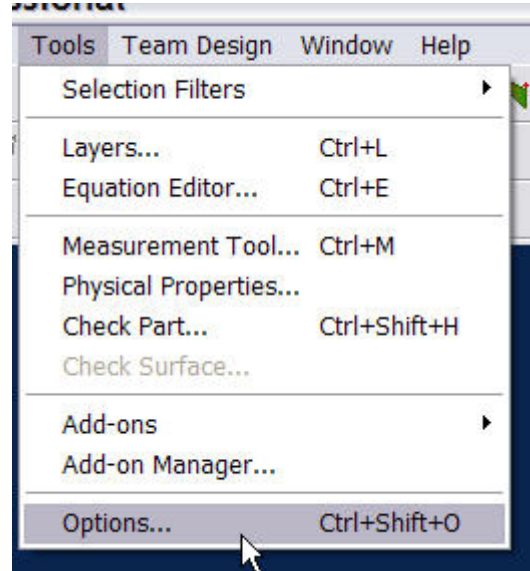
Under the 'Material' tab scroll down in the selection window and click on 'Aluminum – cast – hammered'. This will set the material properties for our model, allowing us to determine the mass, and volume of the part as well as the center of mass, surface area, and moments of inertia.



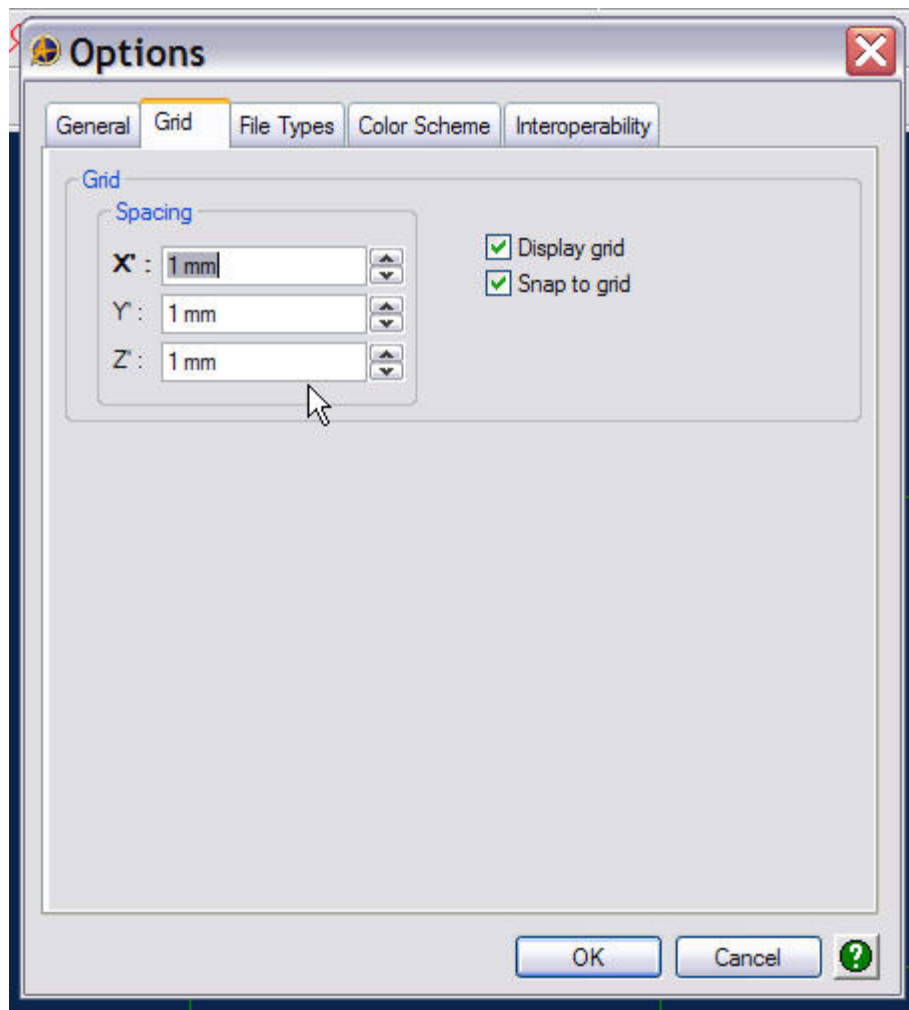
Under the Apply Options tab, check the radio button for 'The Current Document'.

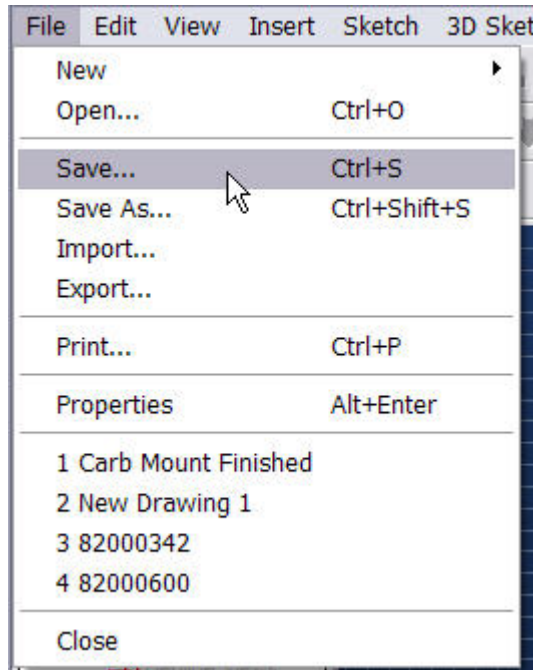


Go back to the main panel toolbar and locate 'Tools'. Under the 'Tools' menu tab, select 'Options'.

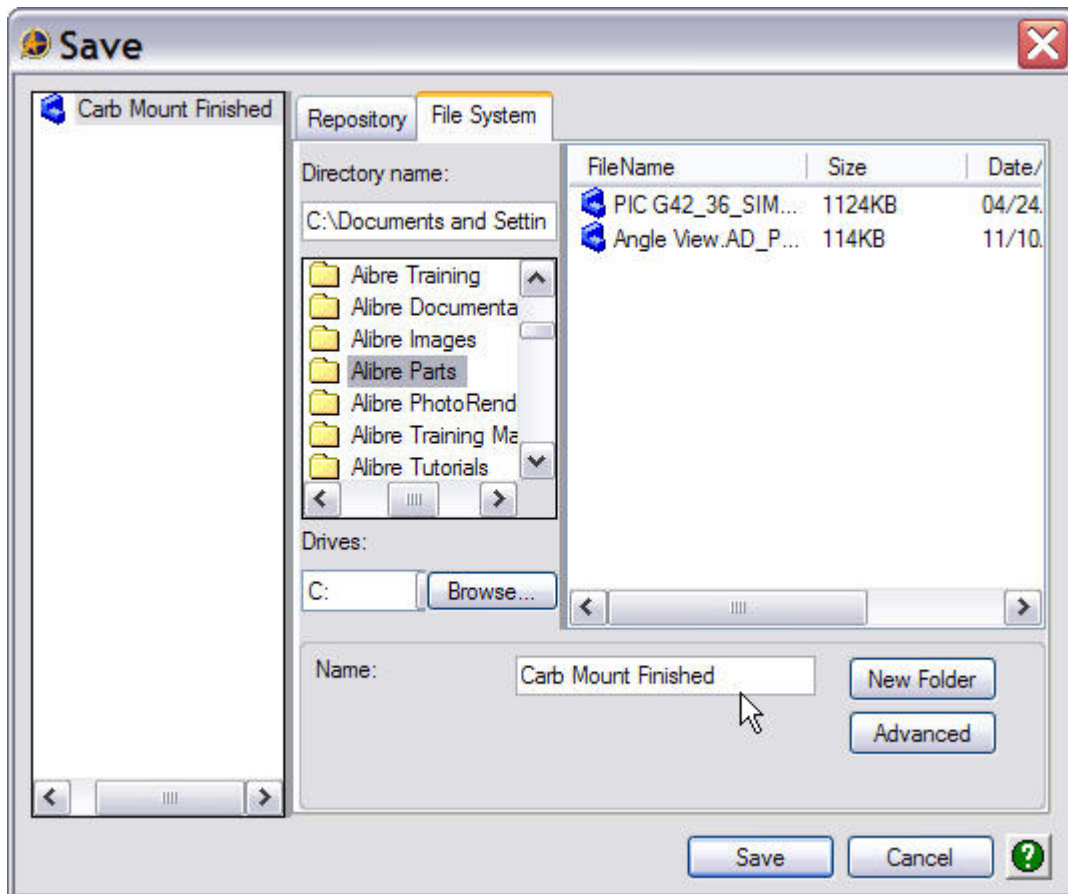


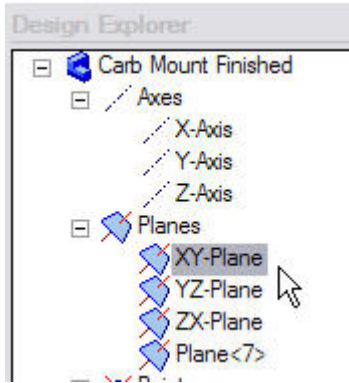
The 'Options' panel will open. Under the 'Grid' tab change the spacing settings to 1mm for X, Y, and Z. Make sure the 'Display grid', and 'Snap to grid' boxes are checked. Click OK to return to the New Part window.



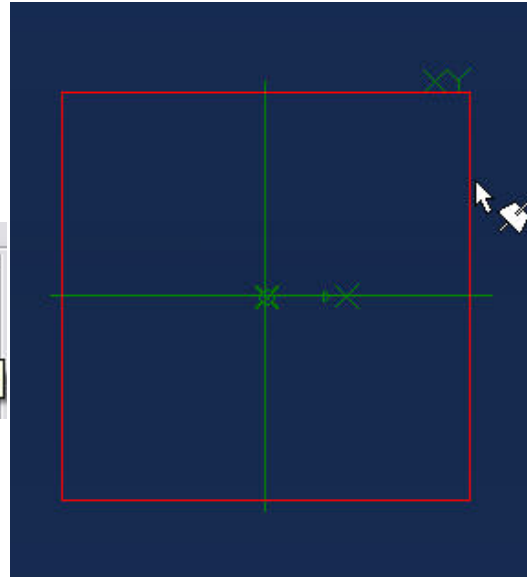
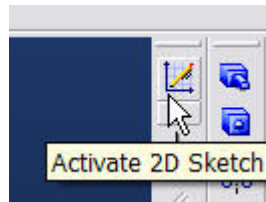


In the New Part window under the File menu tab, select 'Save'. Select a location to save the file, either in the Windows File System or in your local repository. Give the part a logical name, and click OK.

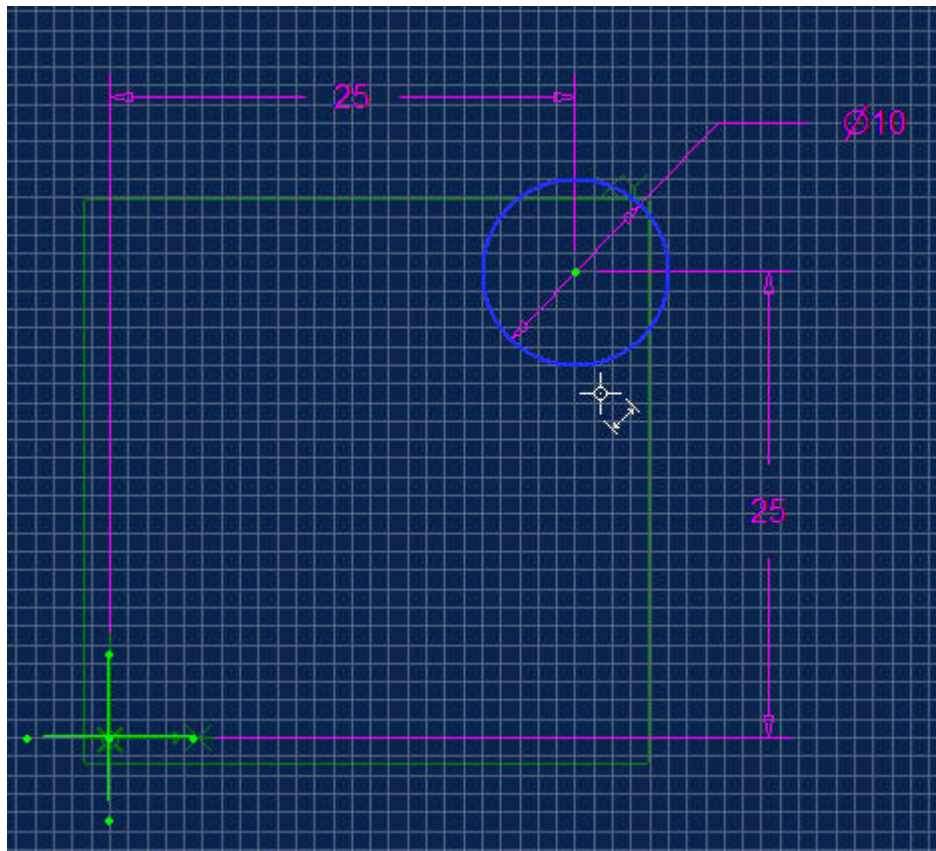




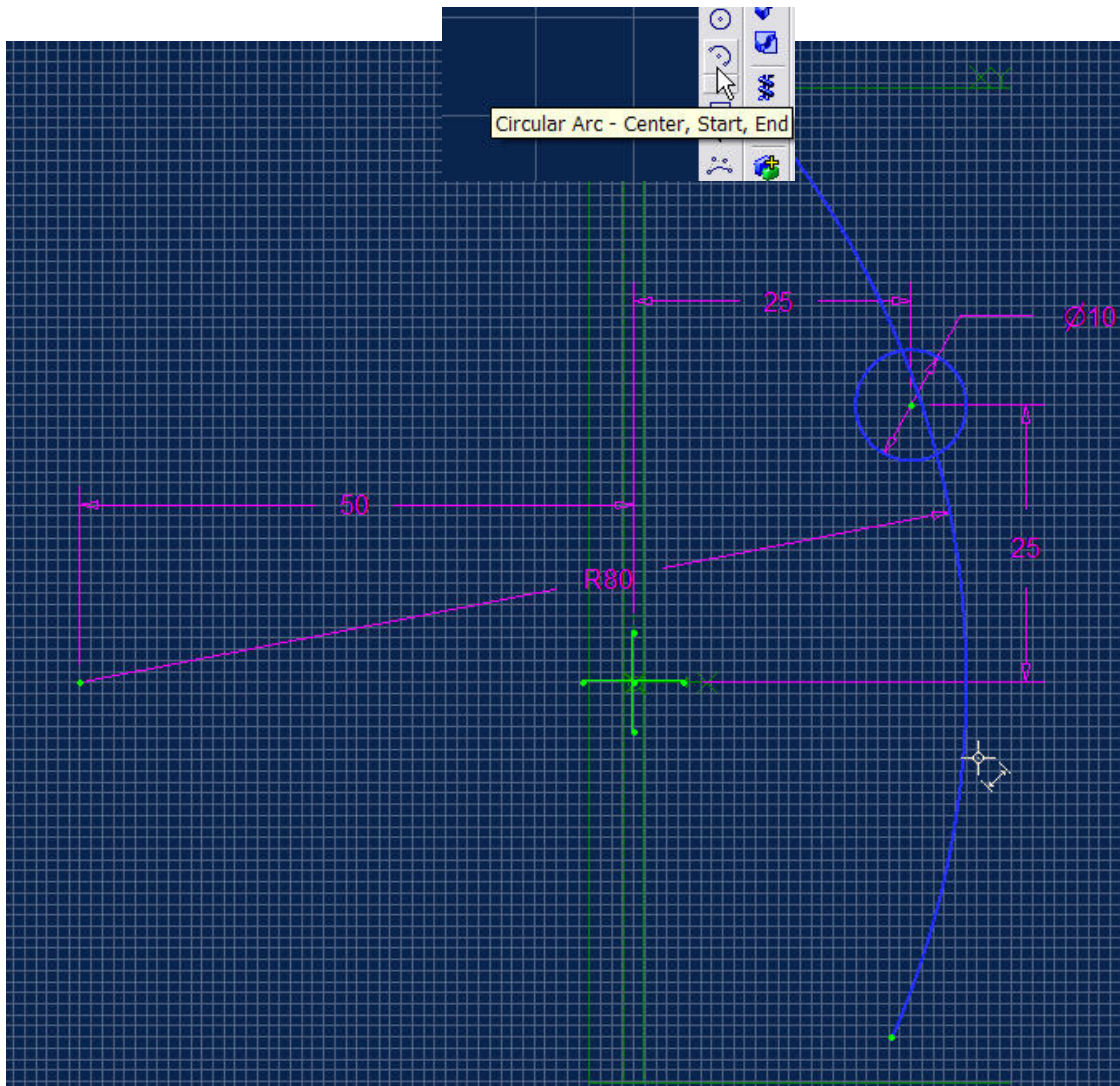
Click on the XY plane in the Design Explorer panel, or the XY plane in the Part workspace and then click the 'Activate 2D Sketch' icon. Now you're in 'Sketch' mode.



We'll start our design by drawing a circle, dimensioning it, and positioning it as shown below.

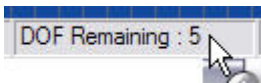


Next, we'll sketch an arc using the Circular Arc – Center, Start, End command, with a radius of 80 mm and position it as shown.

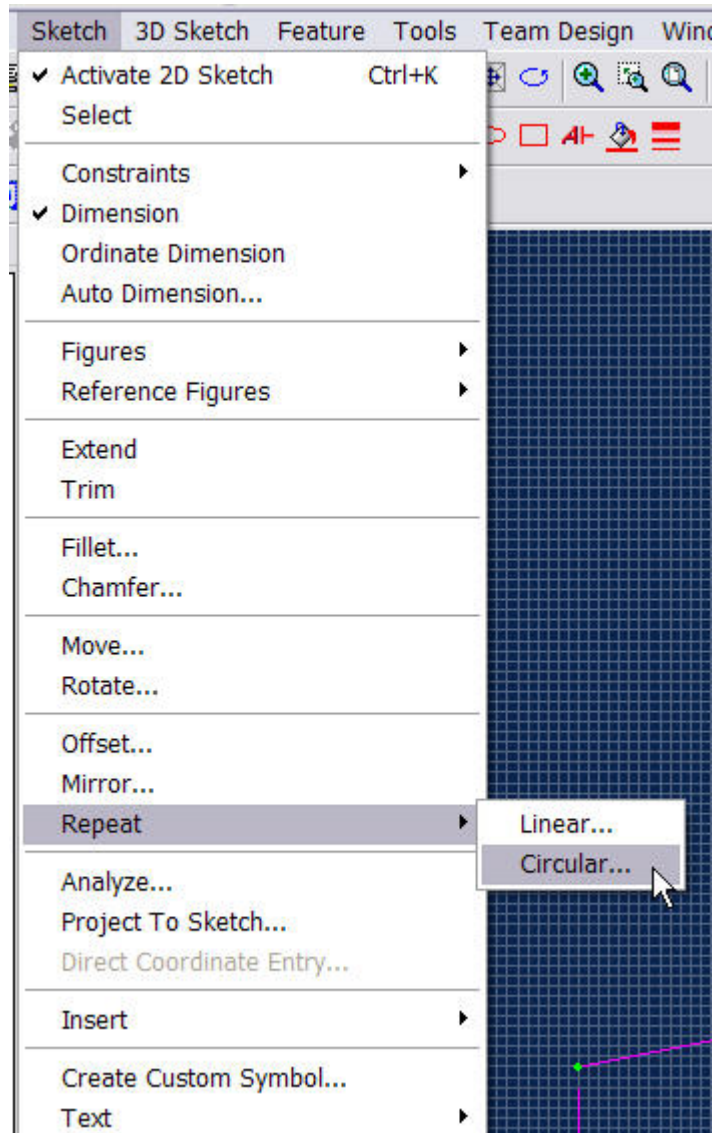


Make the arc large enough to extend past the circle and the X-axis, with an angle value of approximately 30-40 degrees.

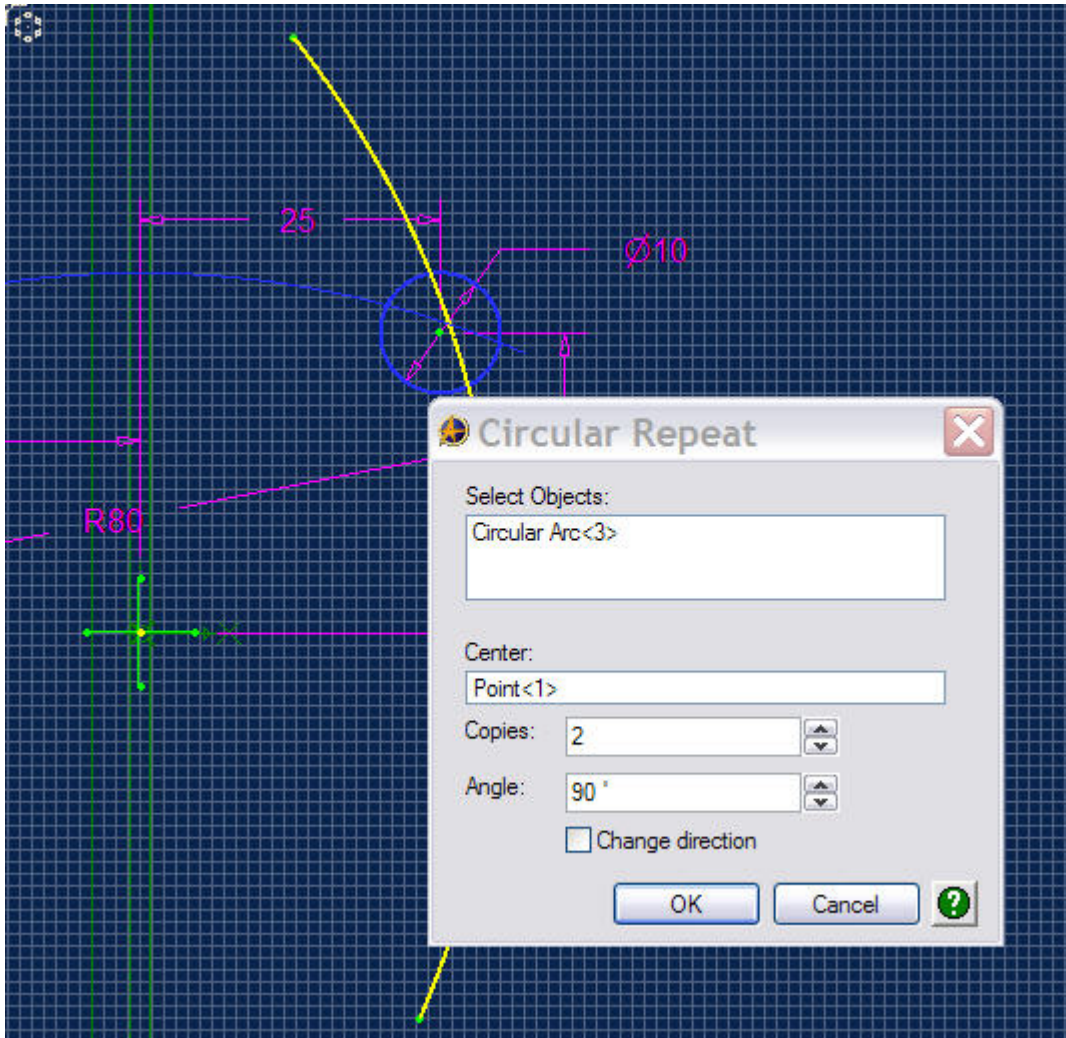
Dimensioning and constraining sketches is one of the most important aspects of good design work. You'll notice in the lower right hand corner of the Alibre main panel screen a running script labeled DOF (Degrees of Freedom). Reducing the DOF's to zero, through dimensional or geometric constraints, although 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.



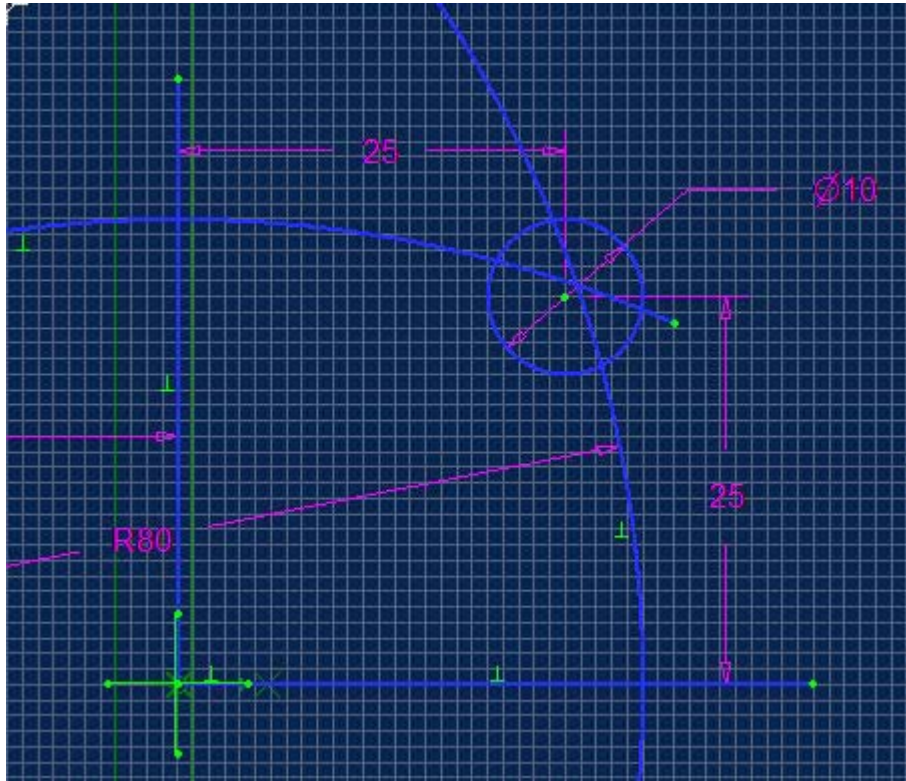
Next, under the Sketch menu tab select Repeat>Circular.



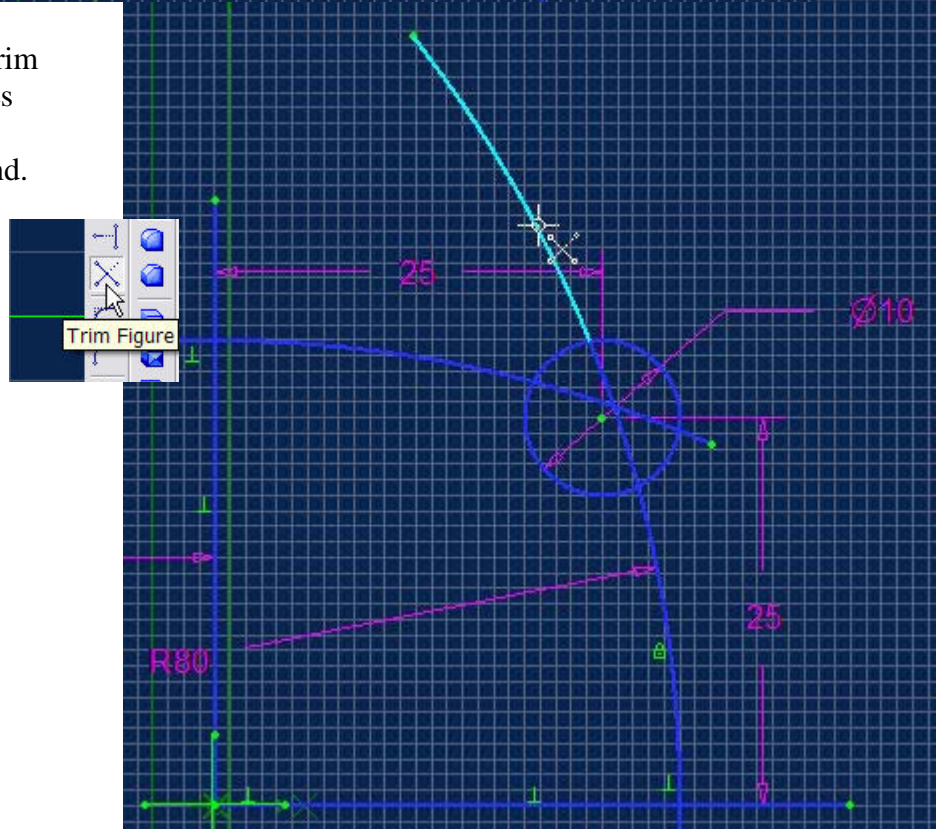
When the 'Circular Repeat' panel opens, click on the circular arc you created and click on the origin point as the Center, select 2 for the number of copies and an angle value of 90°. Click OK.



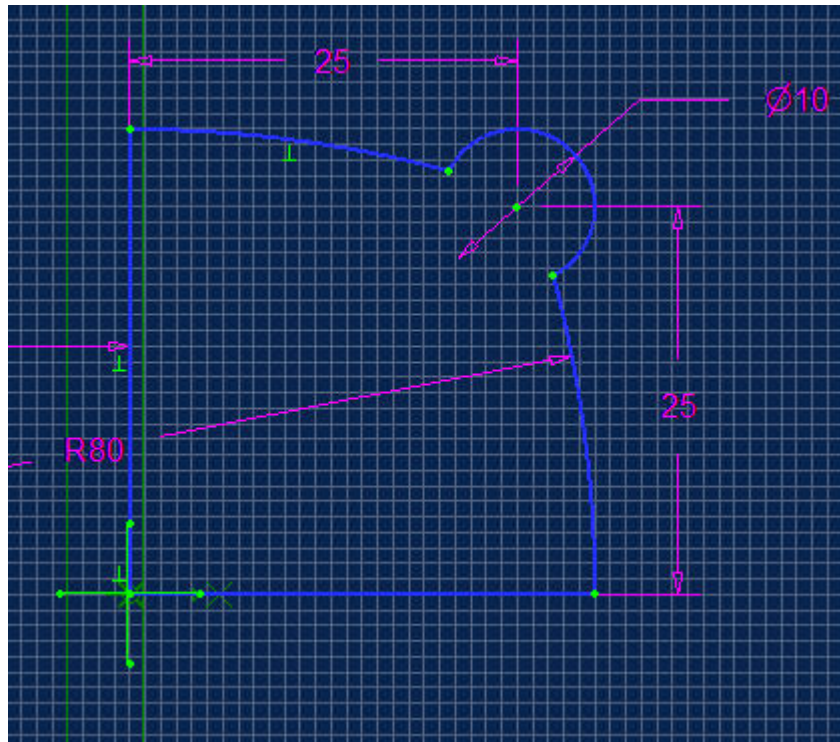
Draw two lines as shown, one vertical and one horizontal, both long enough to cross the two arcs. Your sketch should now look like this.



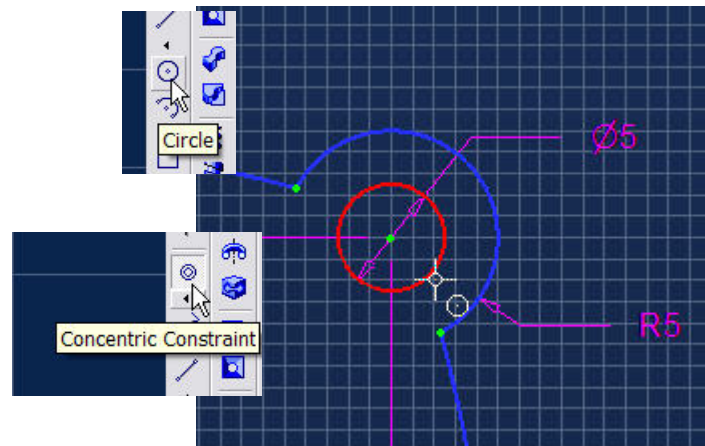
You'll need to trim the arcs and lines using the 'Trim Figure' command.



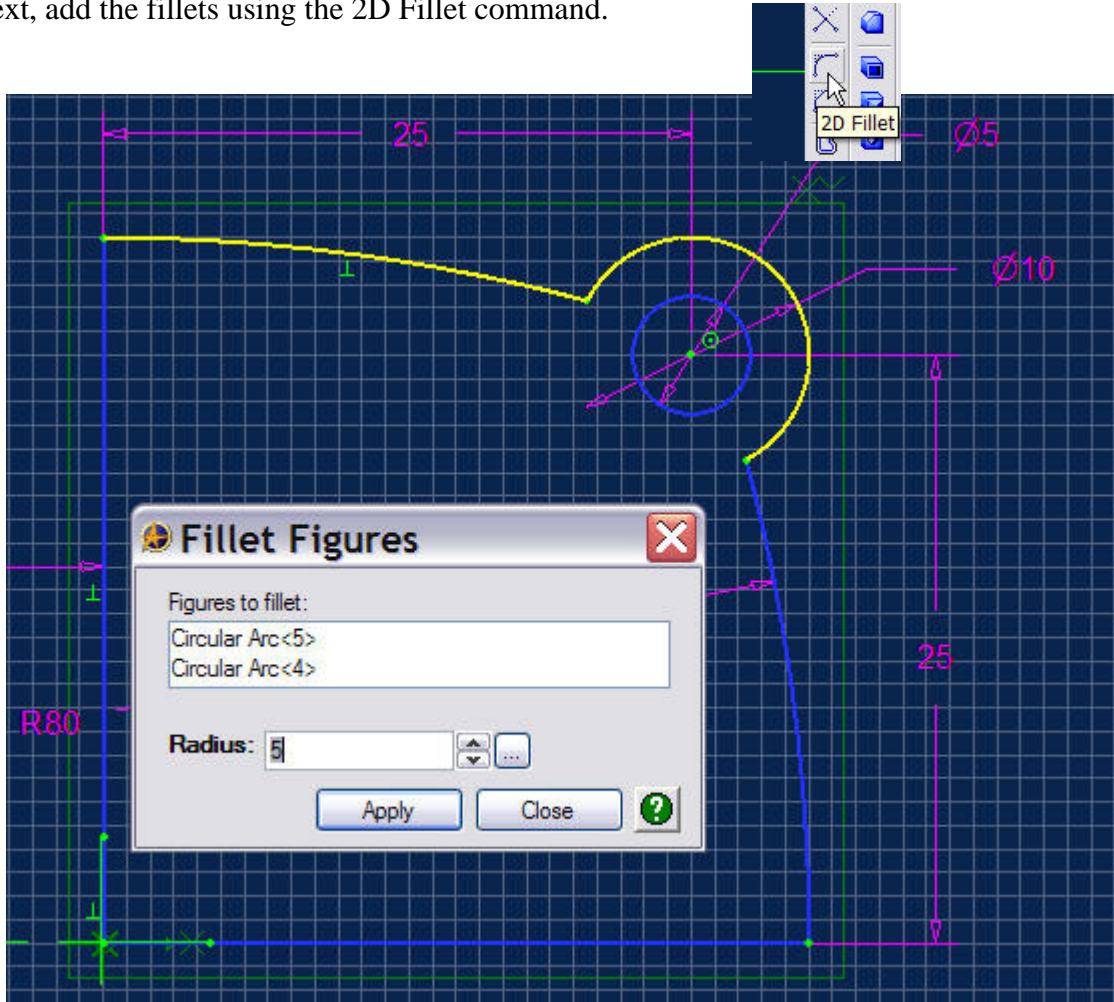
Continue trimming the sketch until it looks like this.



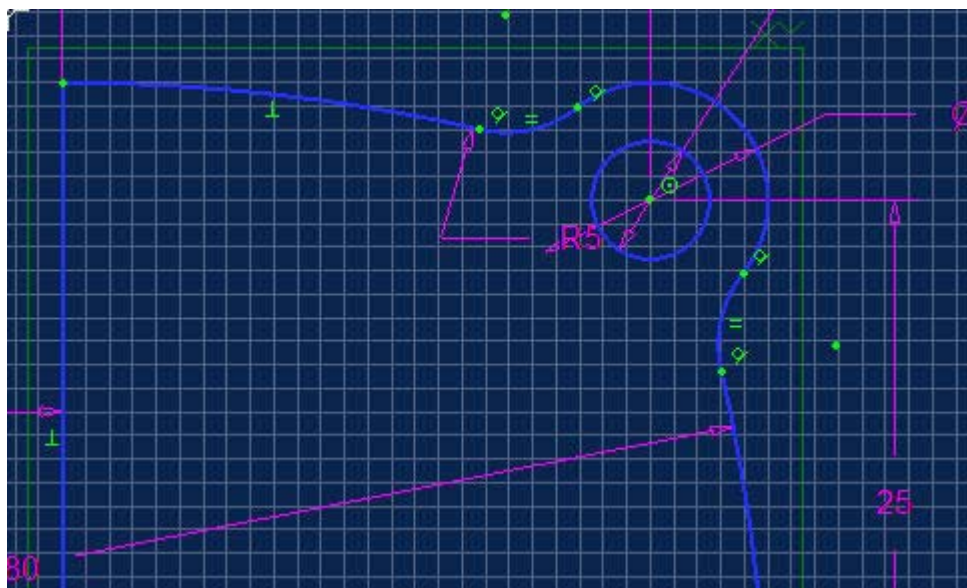
Although the best design practice is to add fillets and holes as dress-up features as late in the design process as possible, we'll add both to our base sketch at this point in order to simplify the part creation process. Add a circle of 5 mm diameter and position it concentric to the 5 mm radius circle with a concentric constraint as shown.



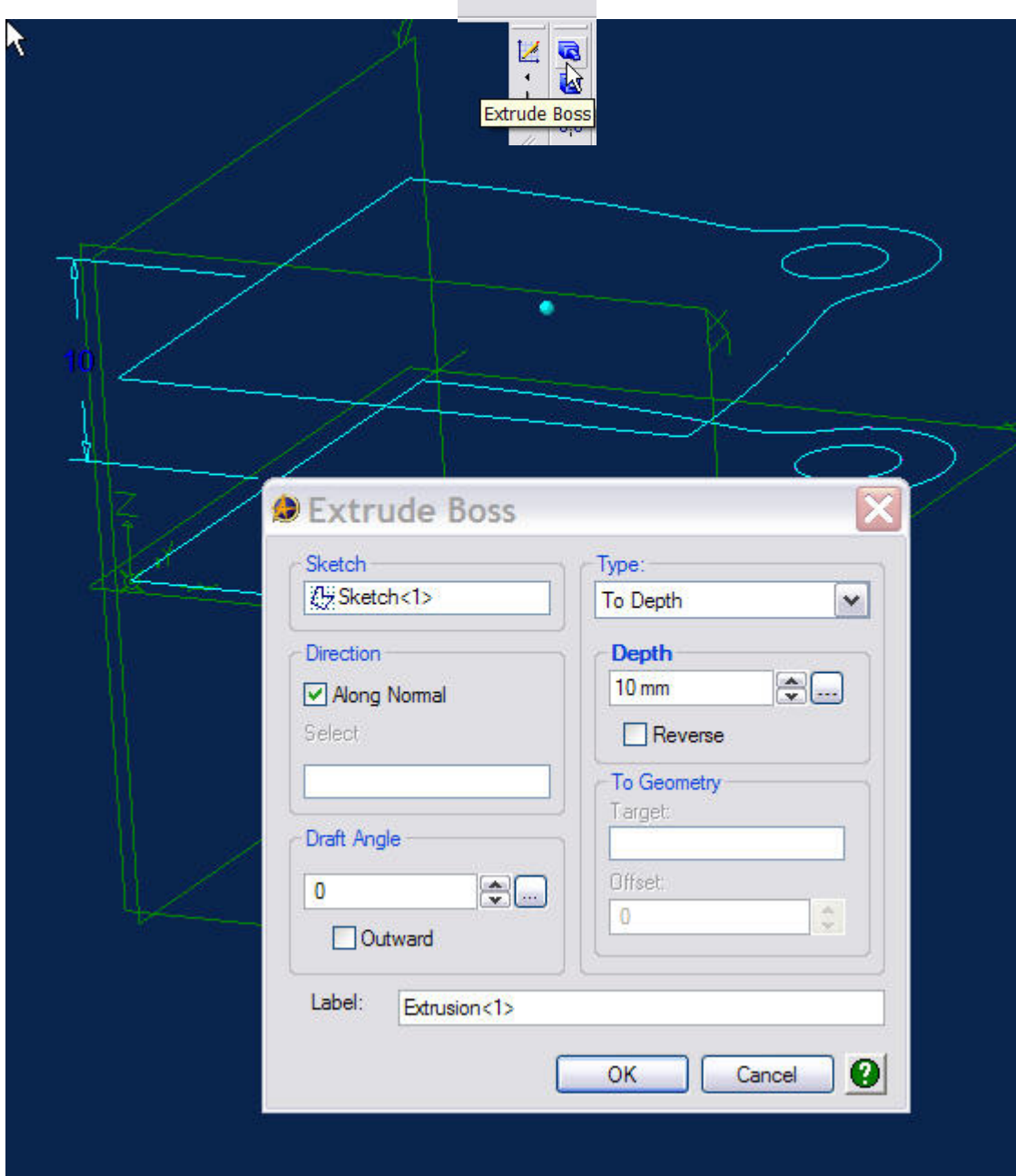
Next, add the fillets using the 2D Fillet command.



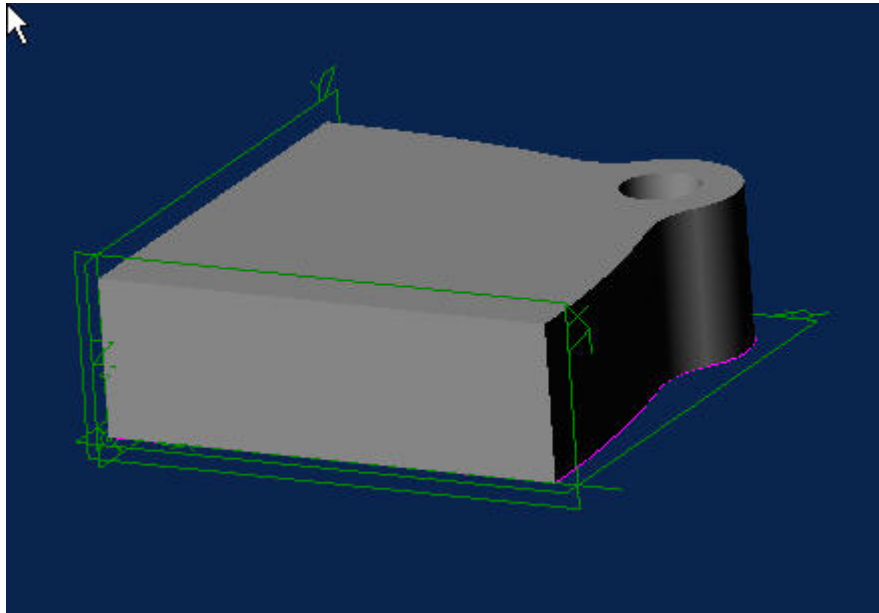
Your sketch should look like this when finished



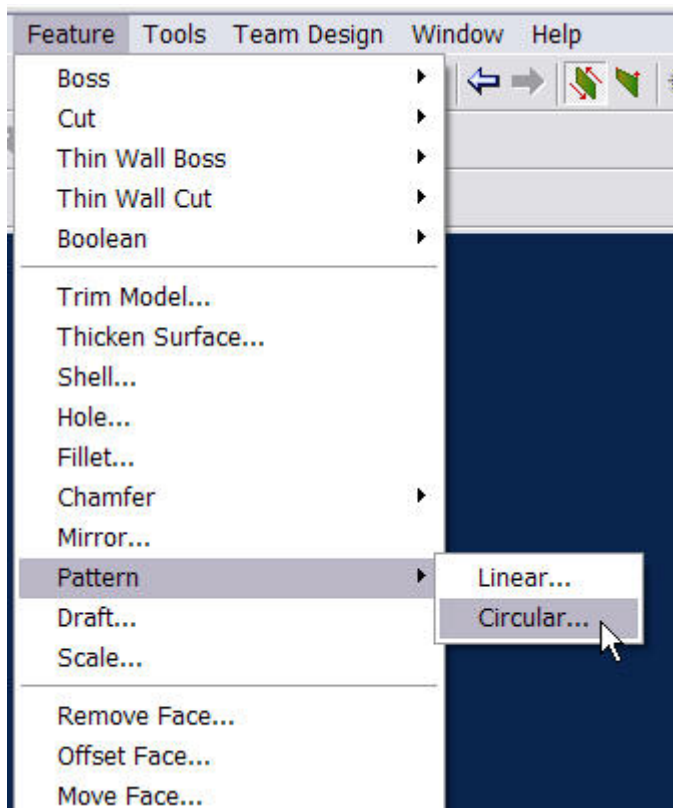
Next click on the Extrude Boss icon and extrude the sketch to a depth of 10 mm in the positive Z direction as shown.



Your part should now look like this.

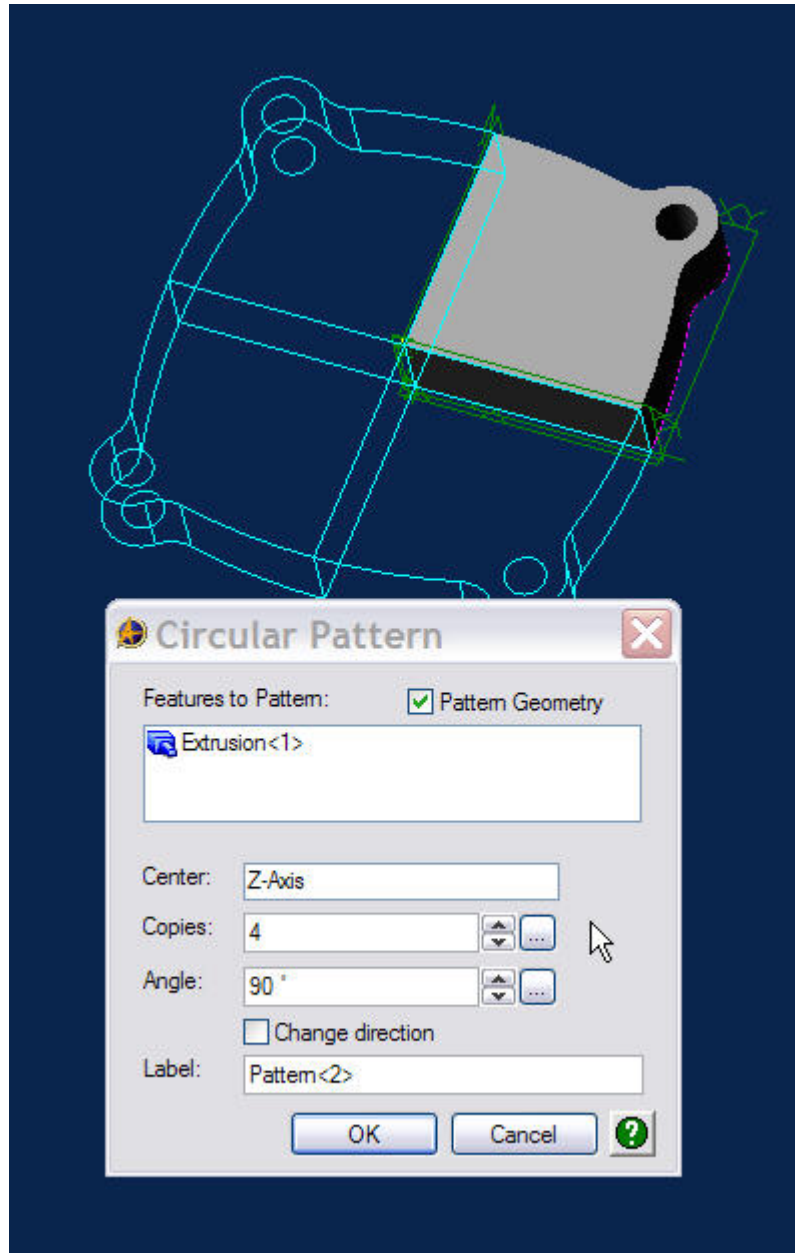


This is the basic building block of the base part.

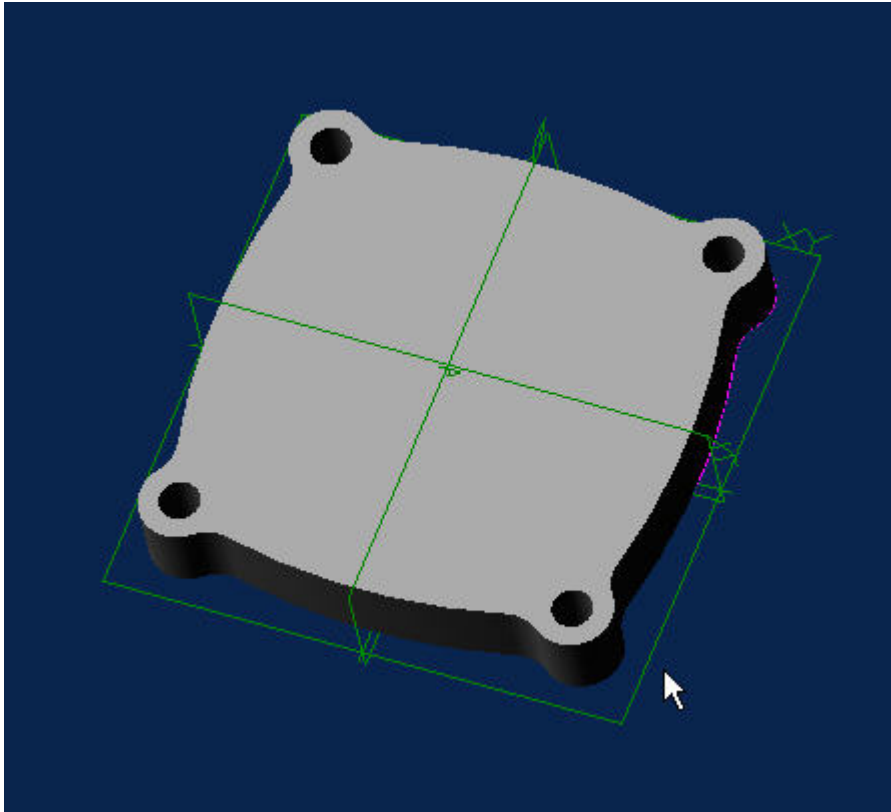


Now we need to duplicate this feature three more times in order to complete the base. Under the 'Feature' tab select 'Pattern>Circular'.

When the Circular Pattern panel opens, click on the extrusion you just created in either the Part workspace or in the Design Explorer panel. Select the Z-axis from the Part workspace or Design Explorer panel; enter 4 for the number of copies, and an angle value of 90°. Click the Change direction box if necessary and then click OK.

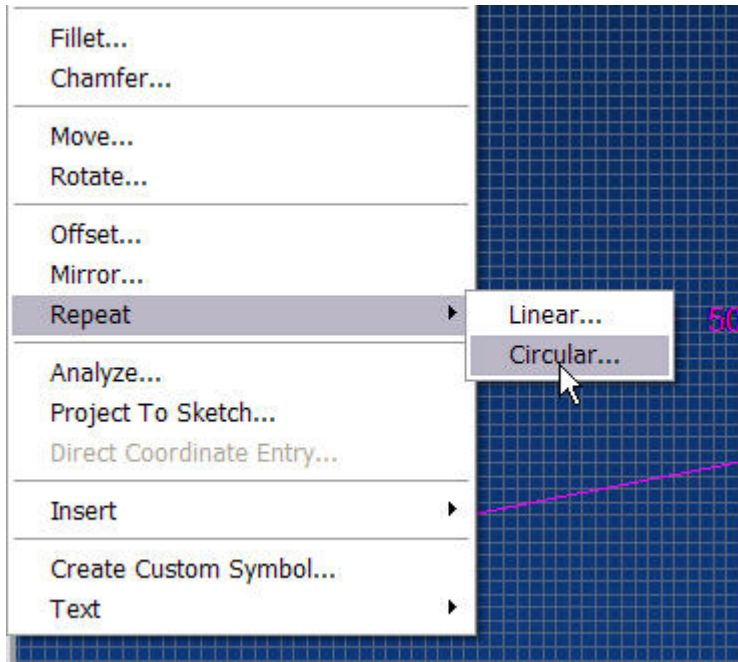


Your part should now look like this.



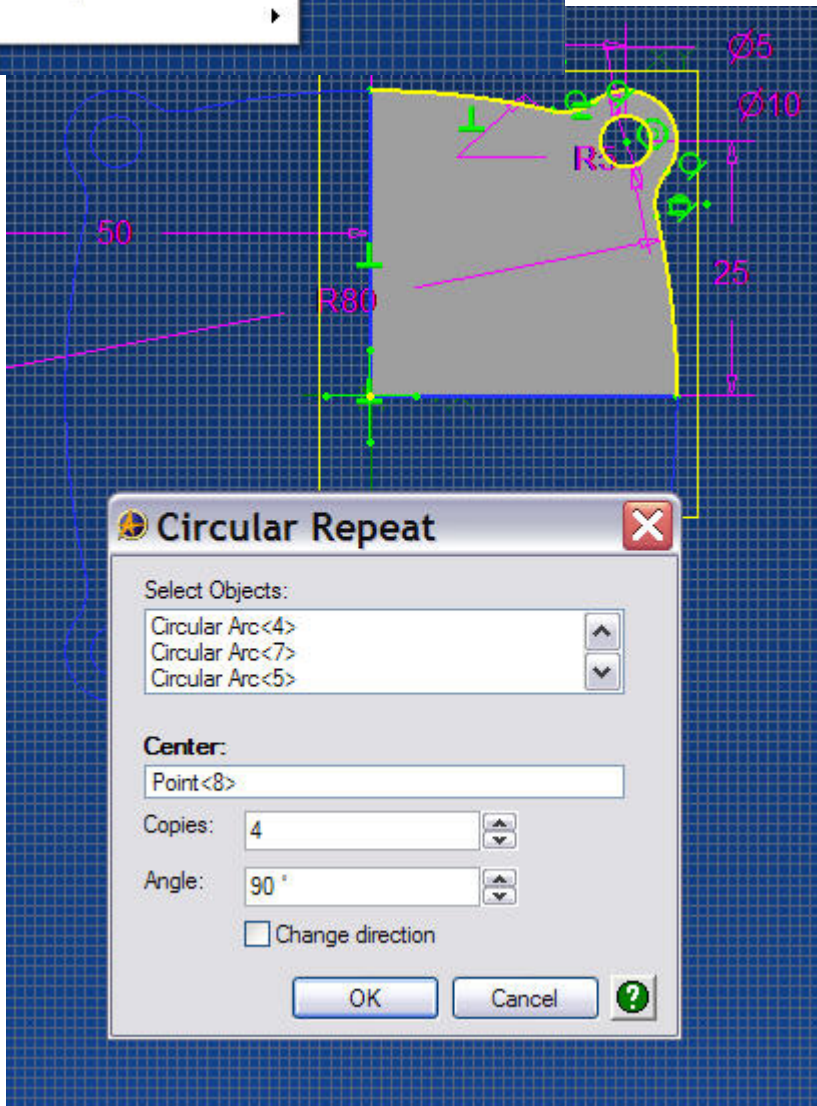
Another approach open to us is to rotate the sketch entities as shown on the next page, by using the Sketch>Repeat>Circular command, and then extrude this as one solid part. In using a quarter of the base design to generate our extrusion, and then rotating the feature makes future revisions much more efficient. Changes made to the base part will propagate in the copies when the part is regenerated. It's also easier to constrain the quarter part sketch, and reduce its DOF's to zero than it is for the more complex sketch.

It's a good idea to determine what elements of your design might lend themselves to symmetry or patterning operations. The simpler the design history is, the easier it will be to change or revise the part in the future.

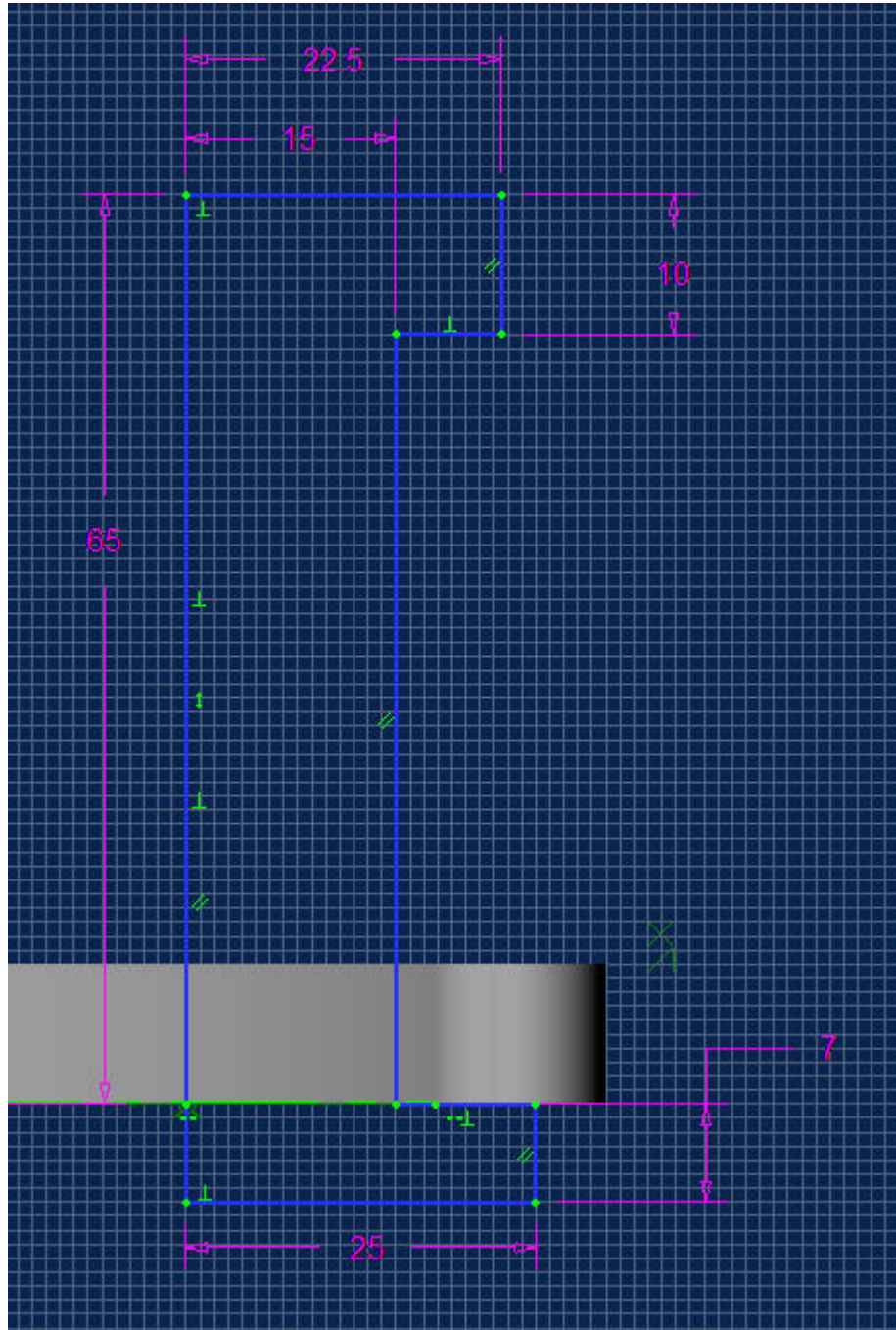


How we might have used the Sketch>Repeat>Circular command to generate the base sketch.

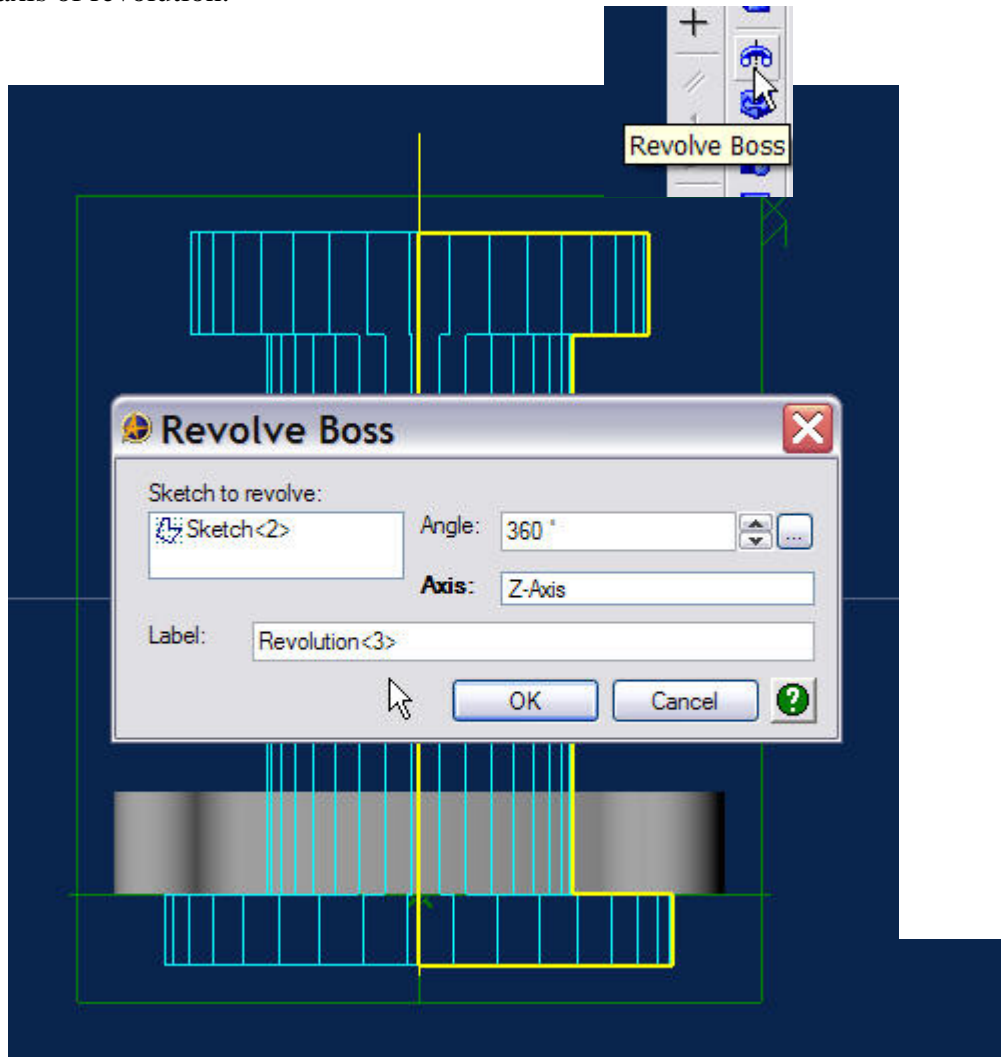
Using sketch patterns in lieu of feature patterns can be useful in many cases, but both options should be considered on a case-by-case basis with an eye towards future revisions.



The next step in the design process is to create the main cylindrical boss. Select the ZX plane in either the Part workspace or the Design Explorer panel and then click on the Activate 2D Sketch icon. Create the sketch below, dimensioning, and constraining it as shown.

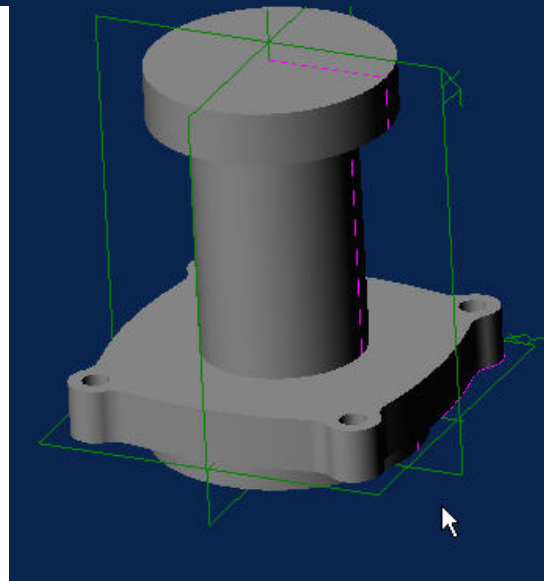


Click on the Revolve Boss icon, and fill in the information as shown, using the 'Z' axis as your axis of revolution.

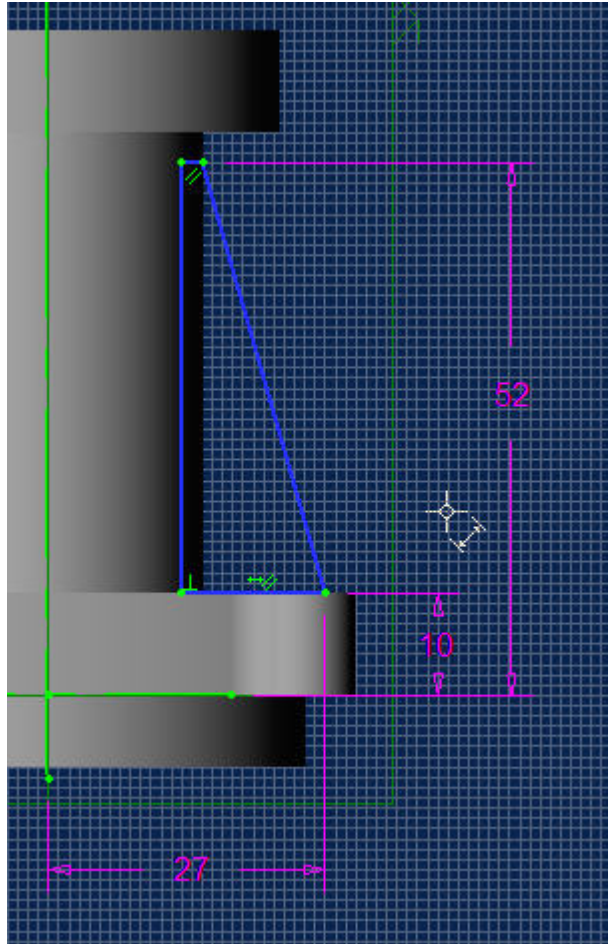


Click OK.

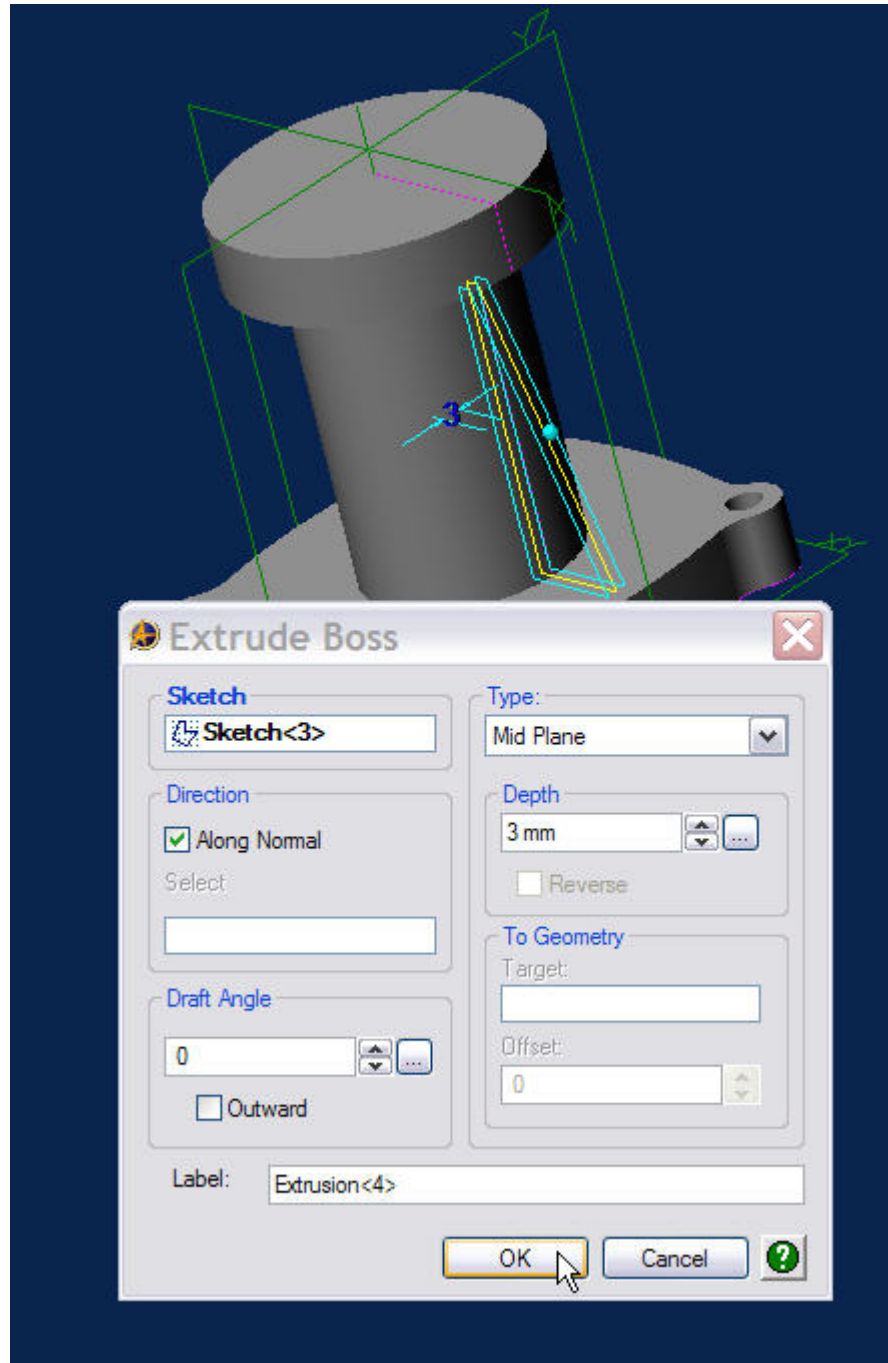
Your part should now look like this.

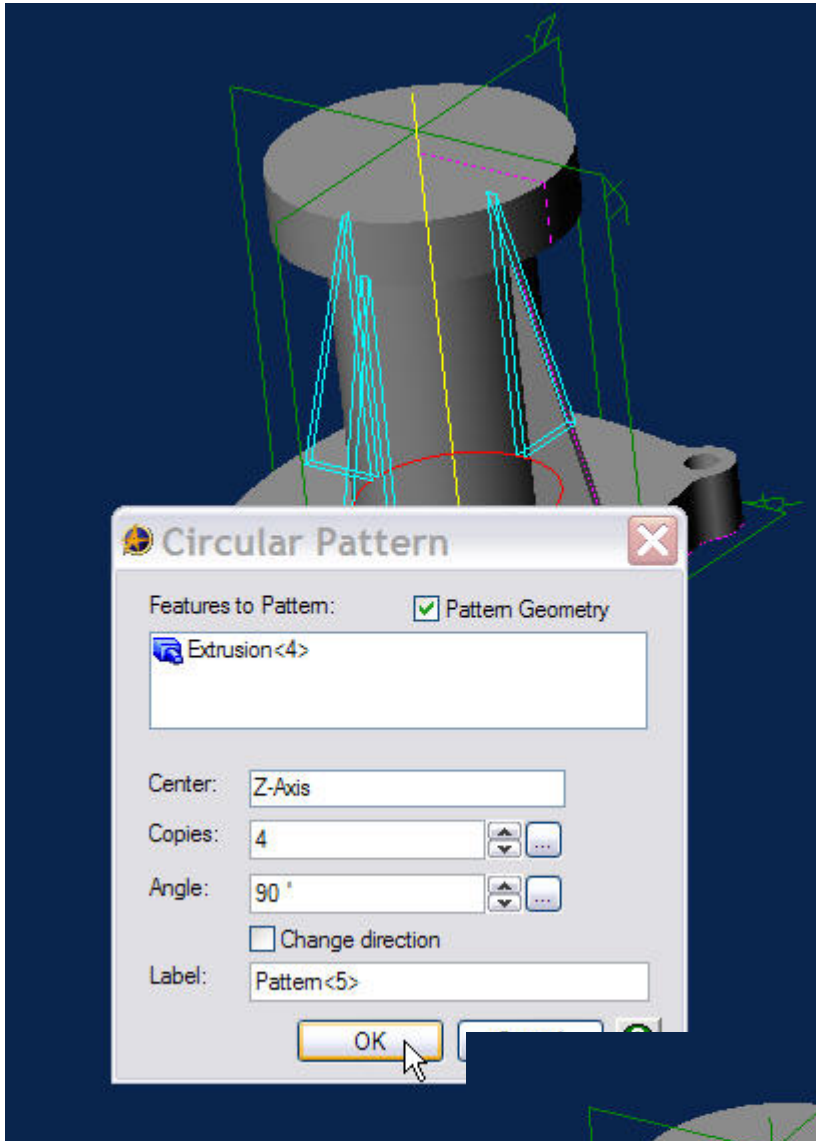


Now we'll create one of the stiffener ribs and use the 'Pattern>Circular' command to create the other three. Select the XZ plane and click the 'Activate 2D Sketch' icon to enter sketch mode. Create the sketch shown at the right. Make sure your sketch 'penetrates' the cylinder envelope. This will avoid any problems you might otherwise encounter because of the vertical rib element not being coincident with the profile of the cylinder after it's extruded.



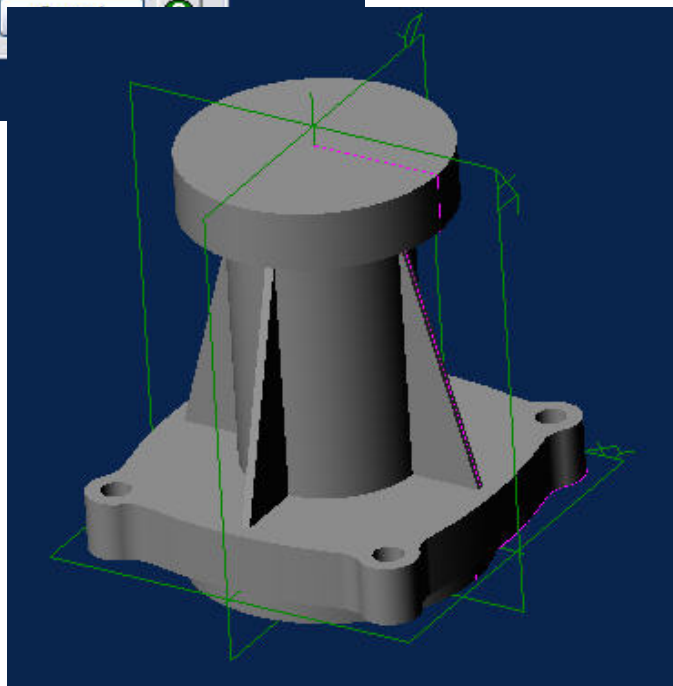
Click the 'Extrude Boss' icon and enter the information shown, set 'Type' to 'Mid Plane', and 'Depth' to 3 mm. Click OK. The reason for using the 'Mid Plane' modifier should be obvious. It maintains the symmetry of the rib around the XZ plane and also ensures a good interface between the rib and the cylinder.



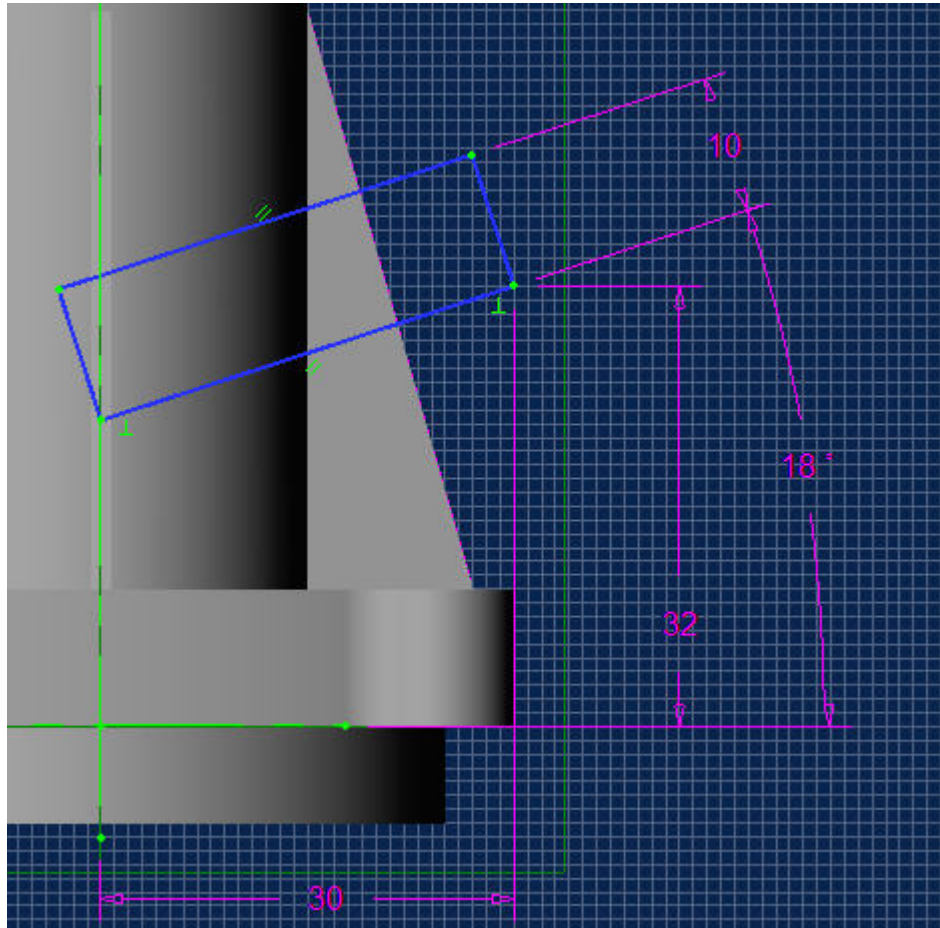


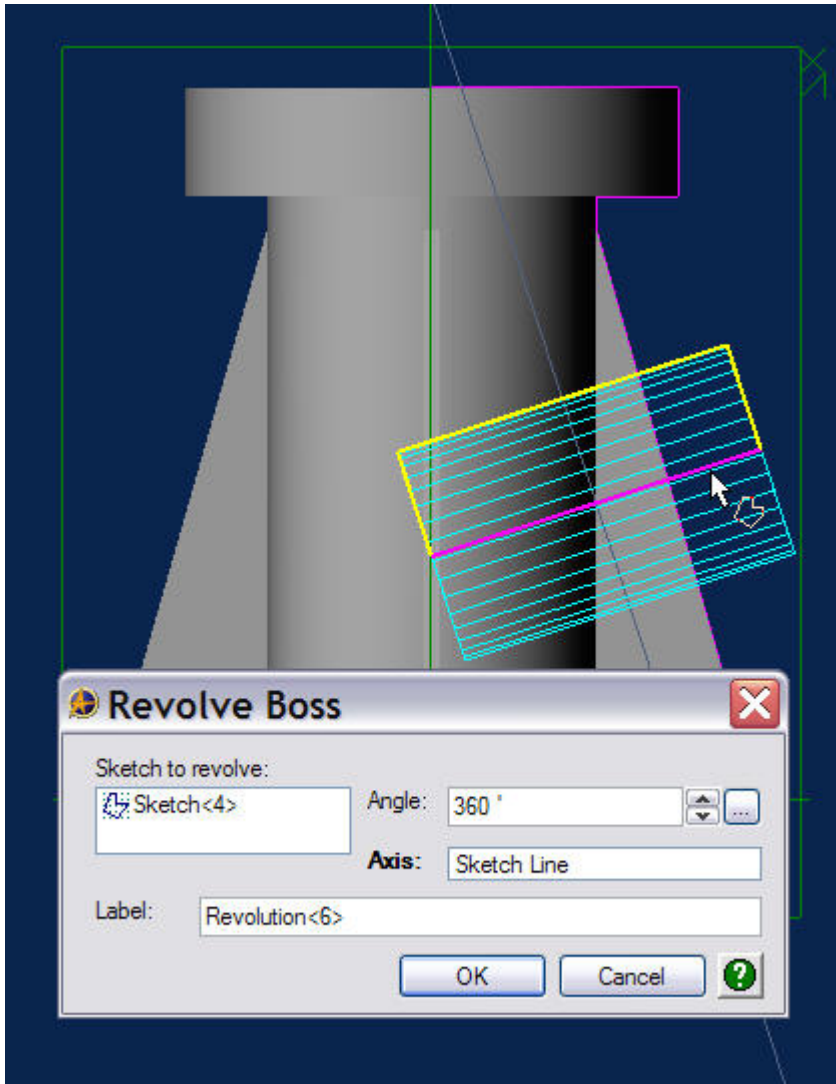
Under the 'Features' menu on the toolbar, select 'Pattern>Circular. Click' on the 'Fin' feature to select it as the feature to pattern. Select the Z-axis as the center. Enter 4 for the number of copies and an angle value of 90°. Click OK.

Your part should now look like this.



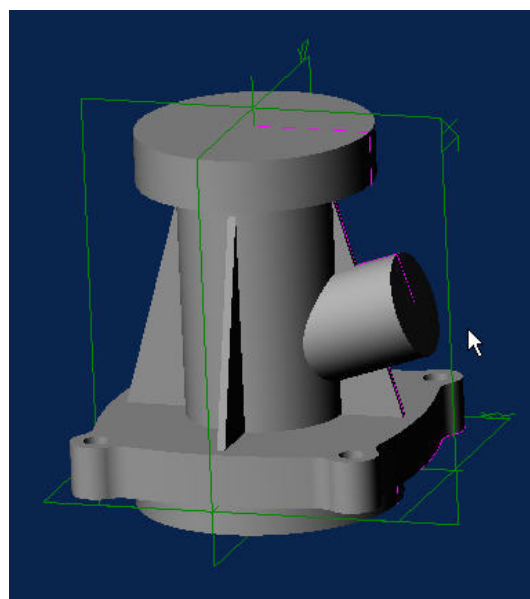
The next step is to create the angled cylindrical boss and the smaller boss attached to it. Select the XZ plane and click on the 'Activate 2D Sketch' icon to enter sketch mode. Create, dimension, and constrain the sketch shown below.



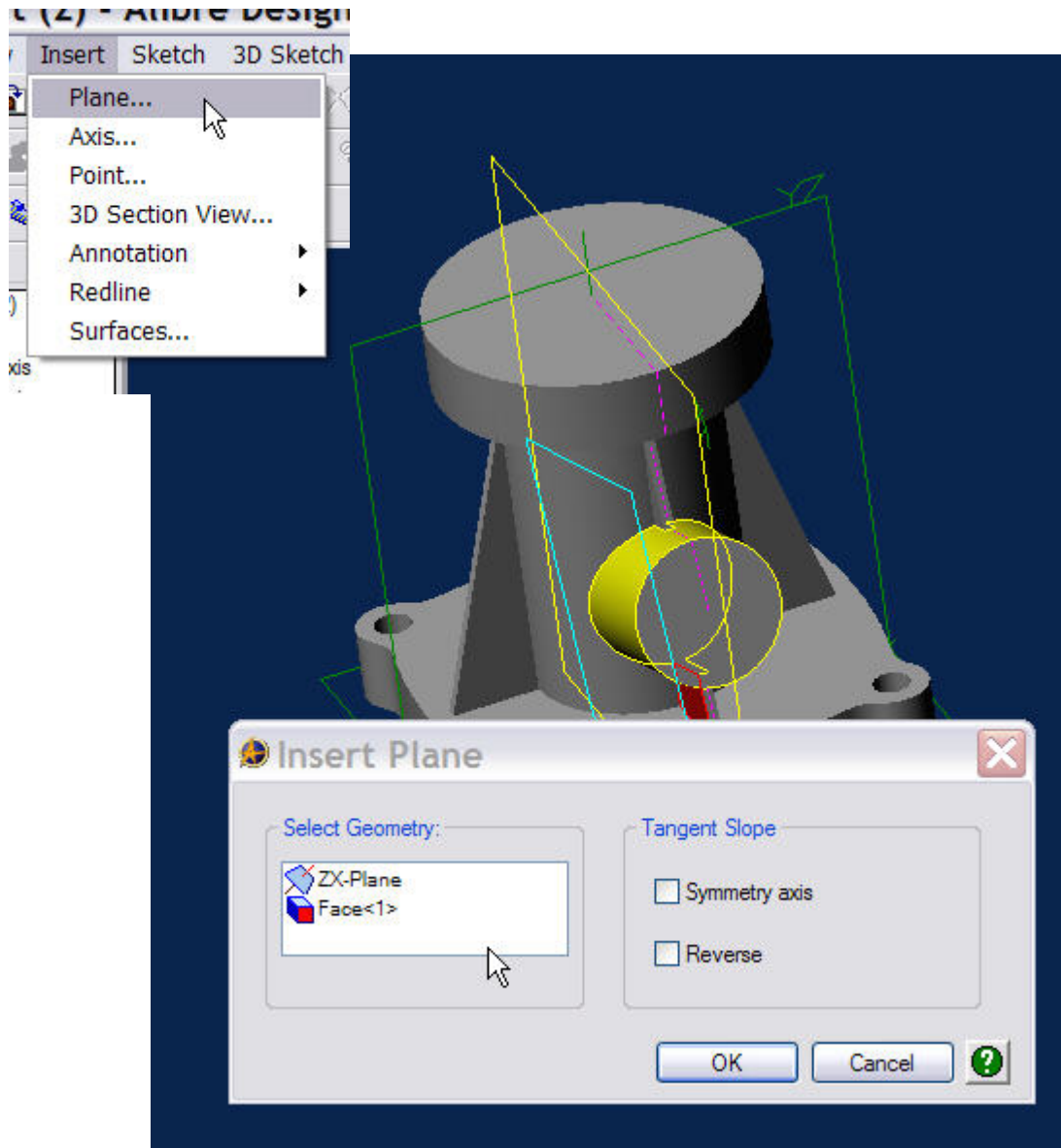


Click the 'Revolve Boss' icon and enter the information shown. Use the dimensioned 18° sketch line as the axis of revolution. Click OK.

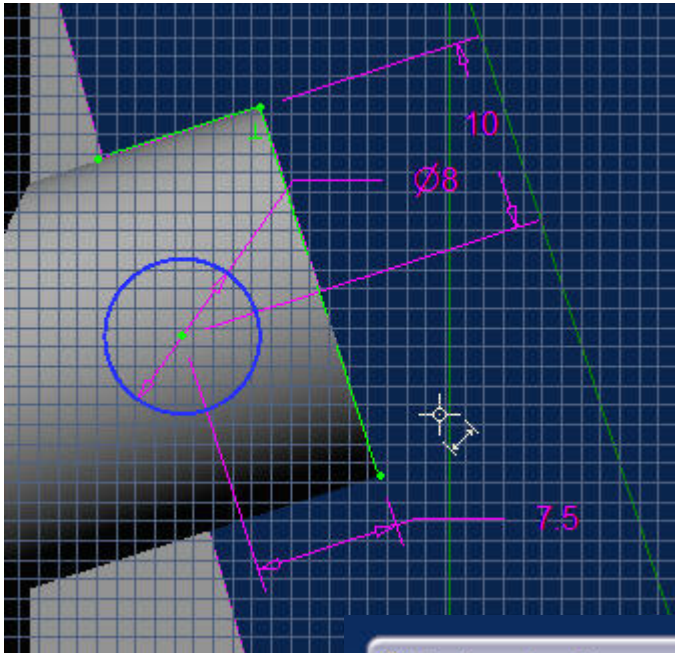
Your part should now look like this.



To create the small cylindrical boss on the side of the larger one, we'll first need to insert a plane tangent to the larger cylinder. Under the 'Insert' menu tab, select 'Plane'.

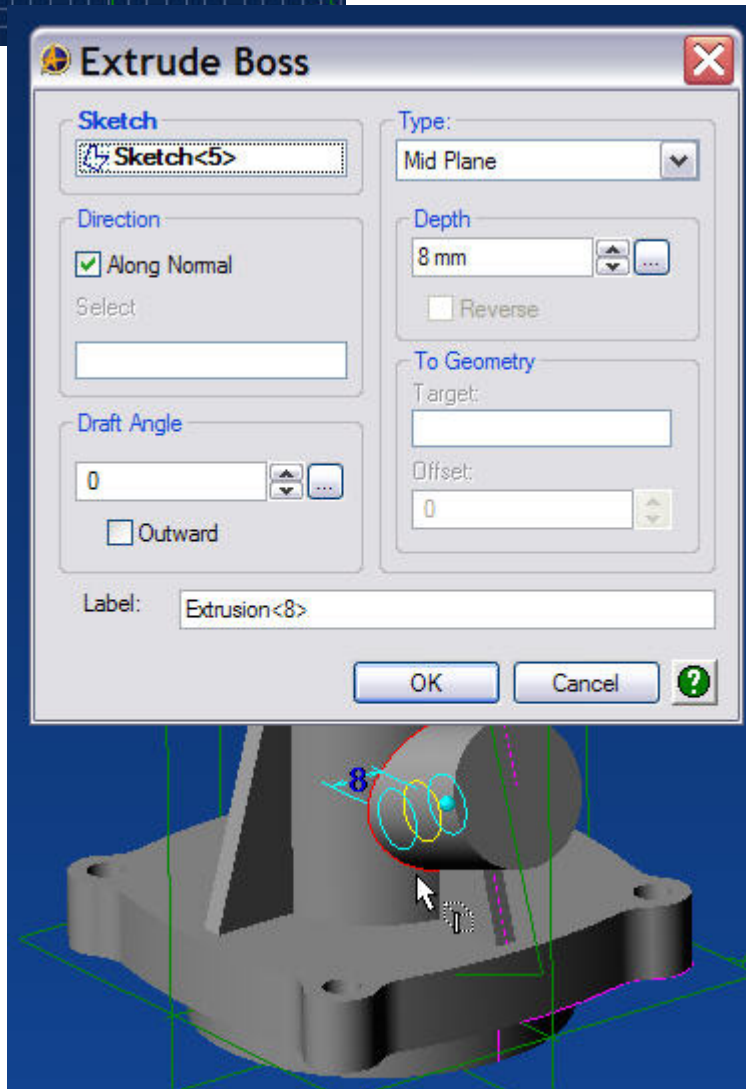
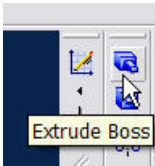


The 'Insert Plane' panel will open. Click on the ZX plane and then click on the surface of the larger cylinder. Click 'Reverse' if necessary and then click OK. You now have the geometric anchor for the small cylinder



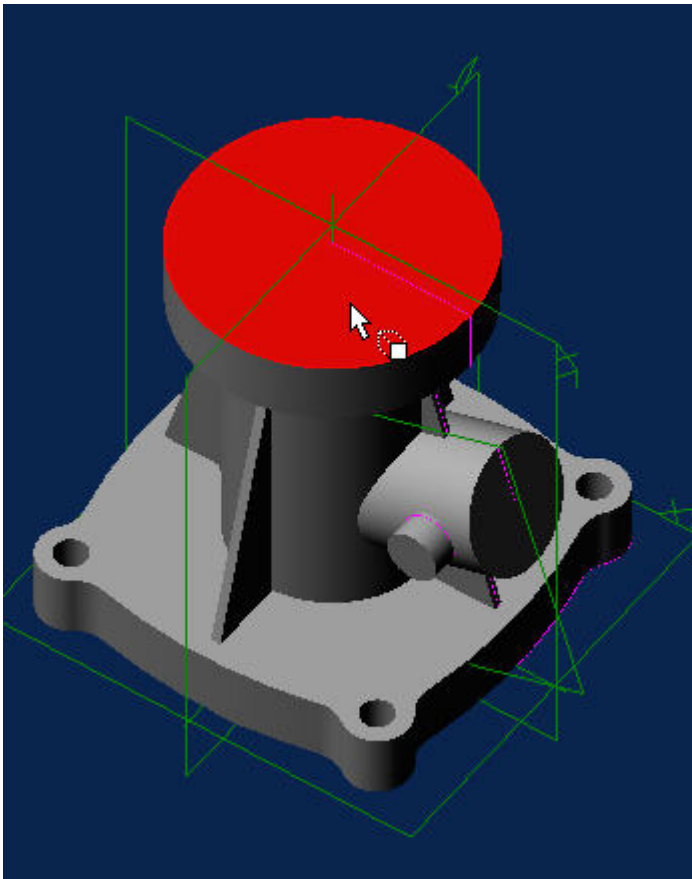
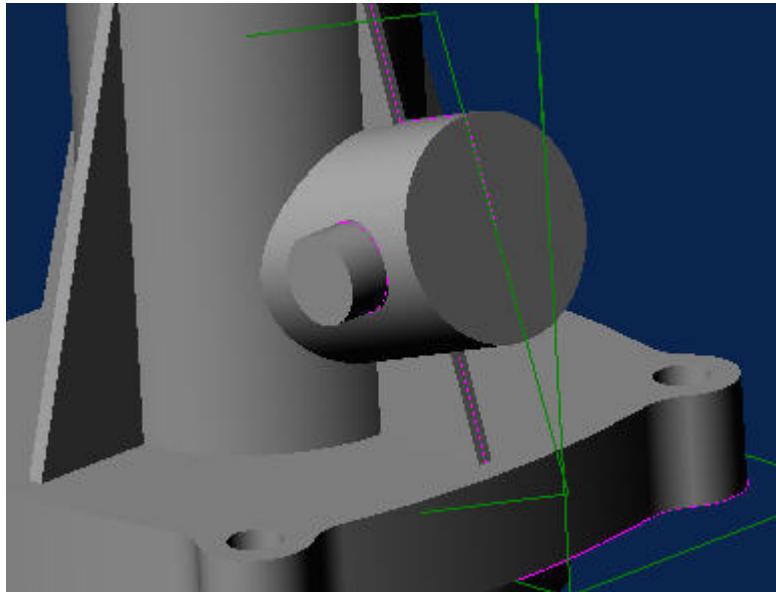
Click on the new plane and click on the 'Activate 2D Sketch' icon to enter sketch mode. Create the sketch shown at the left using a reference line if necessary for the 10 mm dimension.

Hint : Use Project to Sketch to create the front cylinder face reference line and two points.

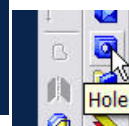


Click on the 'Extrude Boss' icon and enter the values shown, then click OK.

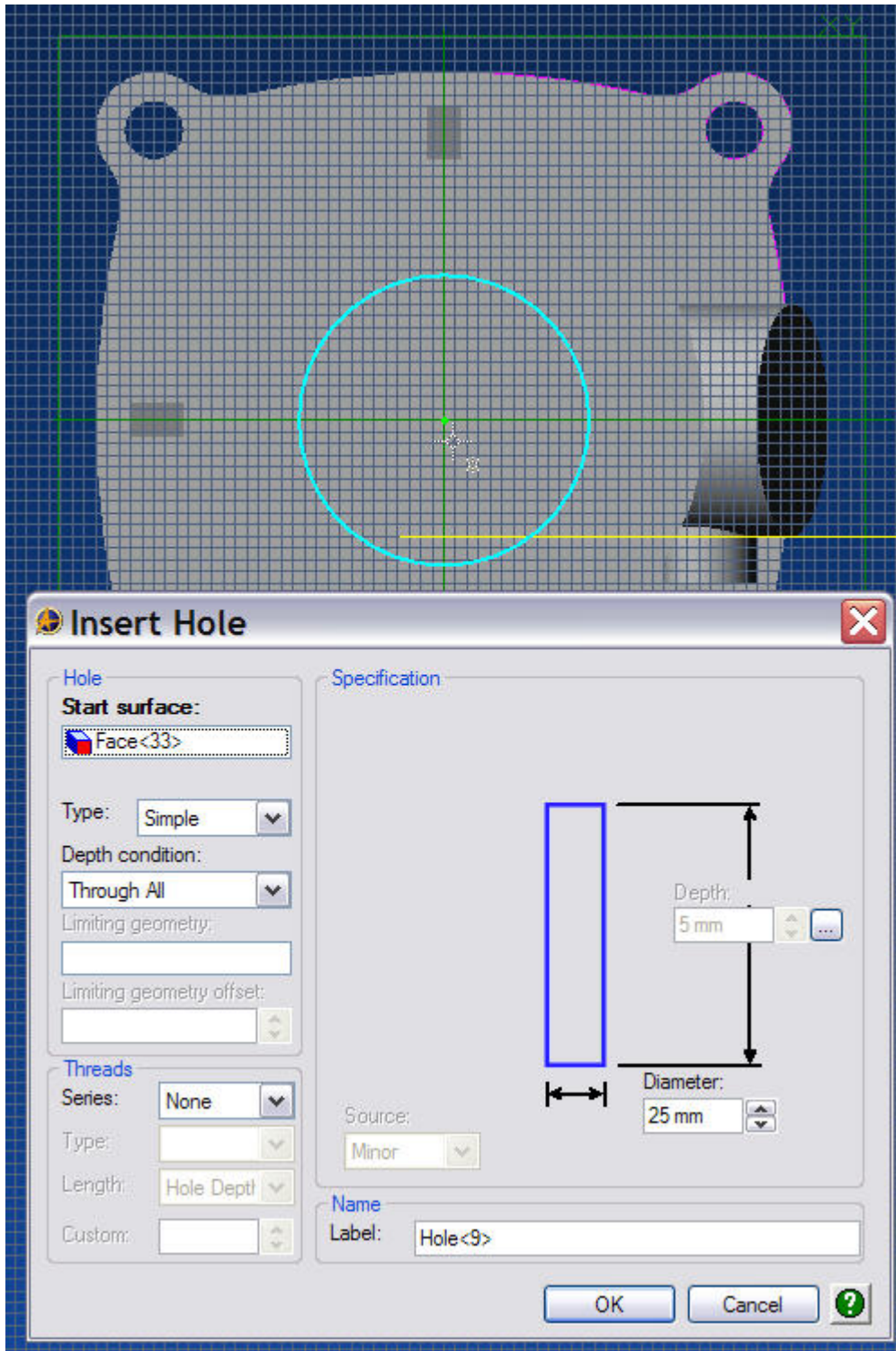
Your part should now look like this.



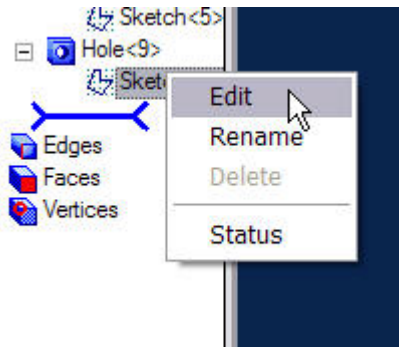
Now we need to add the holes to our part. Click on the top face of the large cylindrical boss as shown in the figure to the left. Click on the 'Hole' icon.



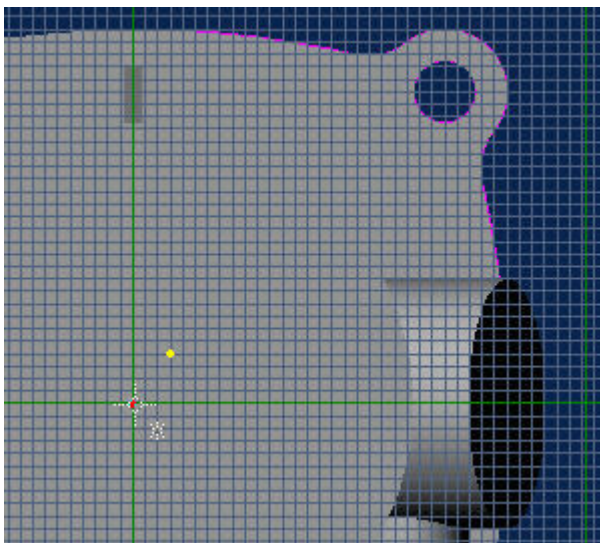
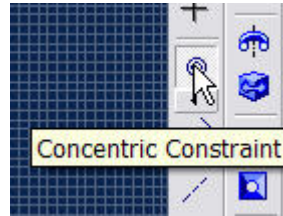
The 'Insert Hole' panel will open. Select the options and fill in the information shown, then click OK.



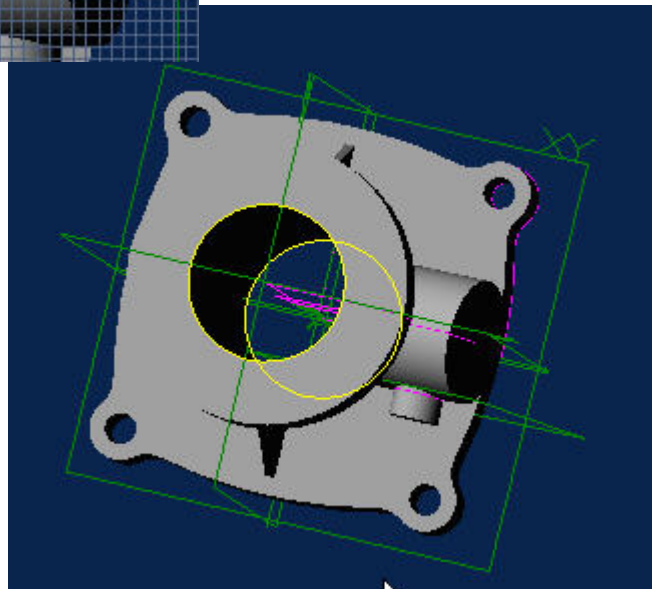
It might be necessary to locate the hole after it's created. To do this, click on the 'Hole' 'Sketch' in the Design Explorer panel, right click and select 'Edit'.



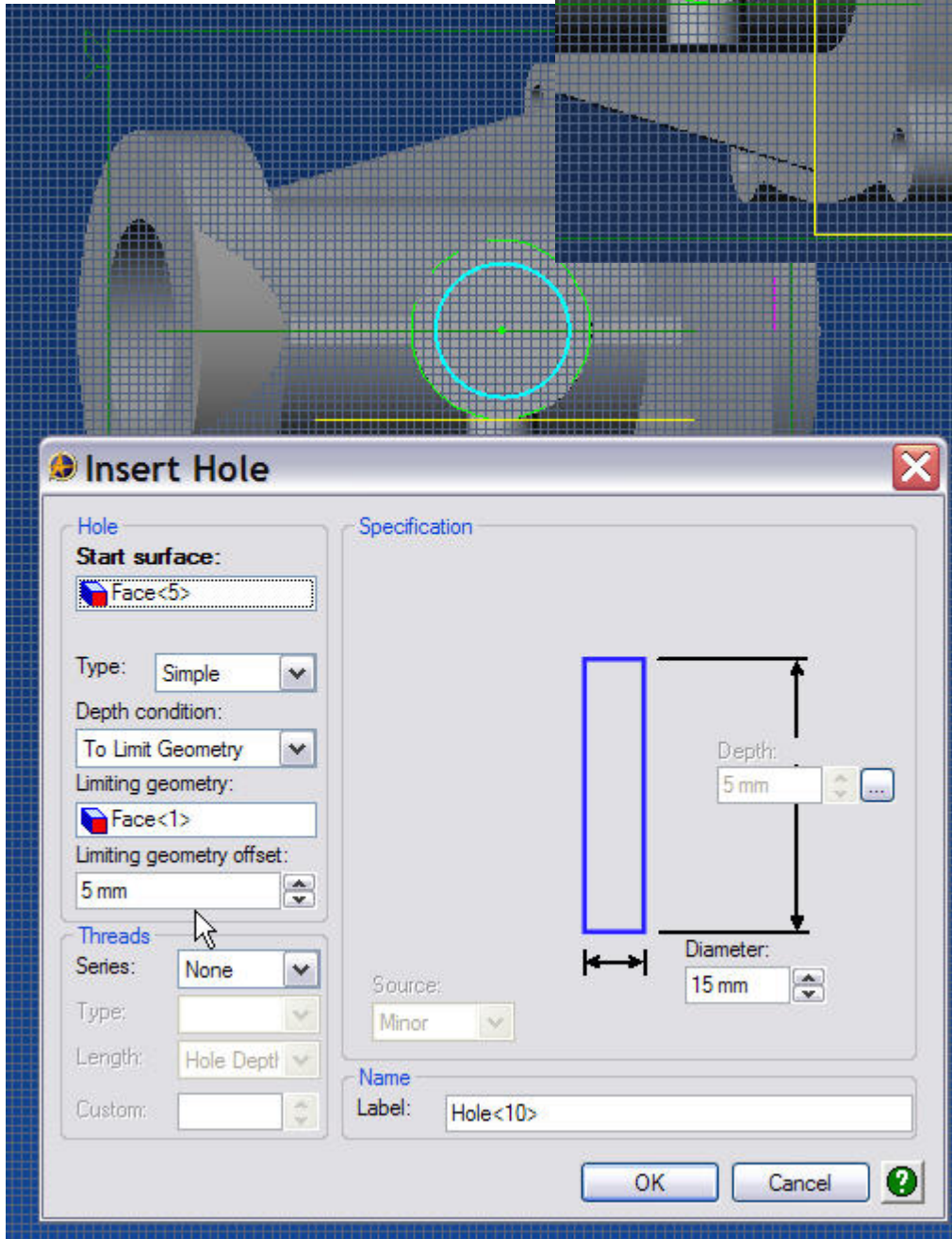
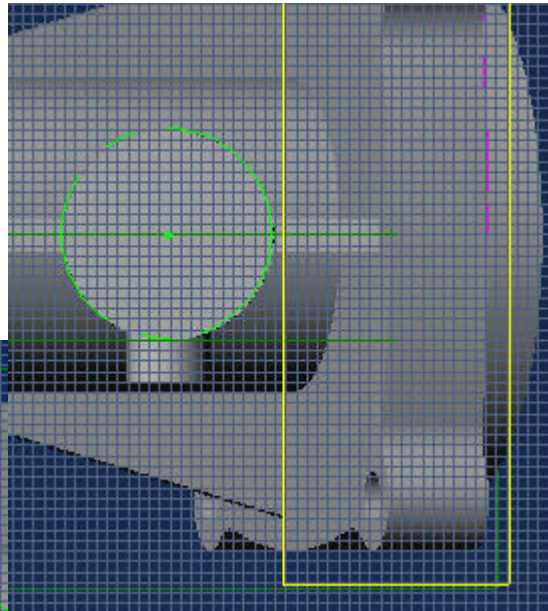
Apply a concentric constraint between the center point of the hole and the center point of the boss.

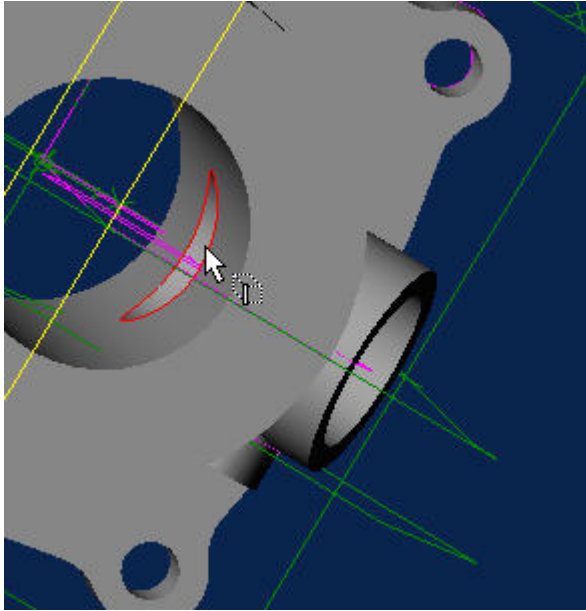


Your part should now look like this.

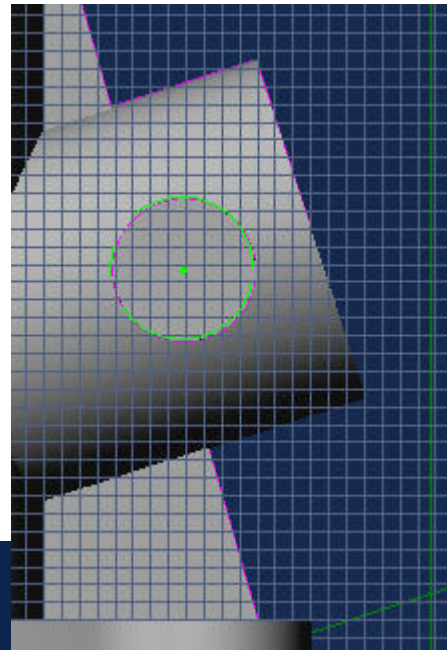


Repeat the steps you used to create the hole in the large boss to create the hole in the medium boss as shown below. Use a large enough offset (Limiting geometry offset) so your extruded cut extends past the inside surface of the large hole.

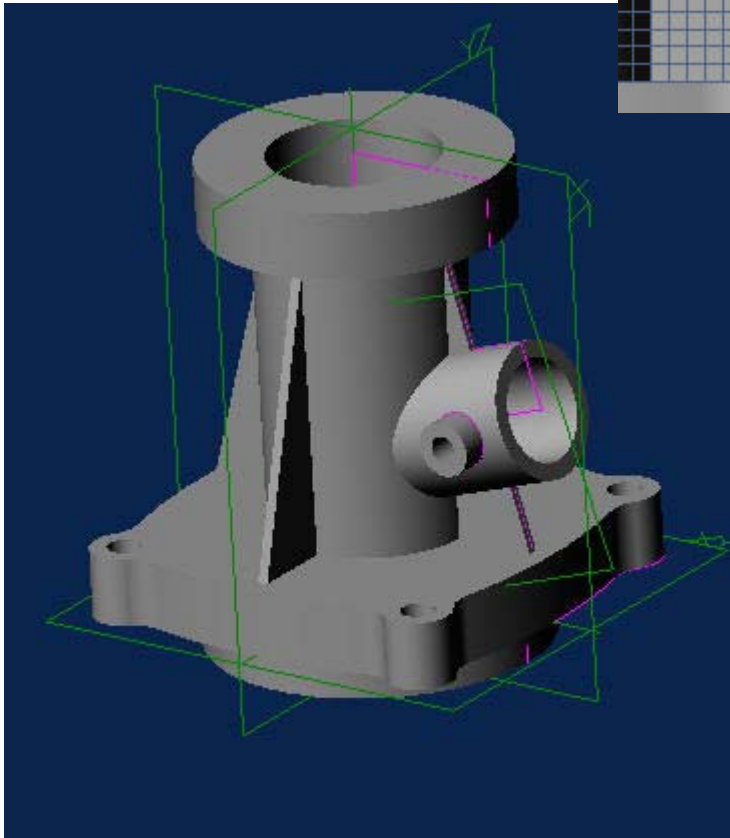




This is how the part should look after the cut is complete.

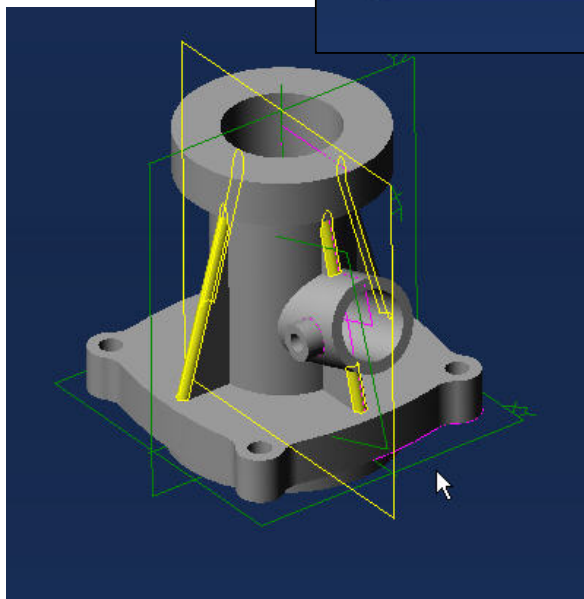
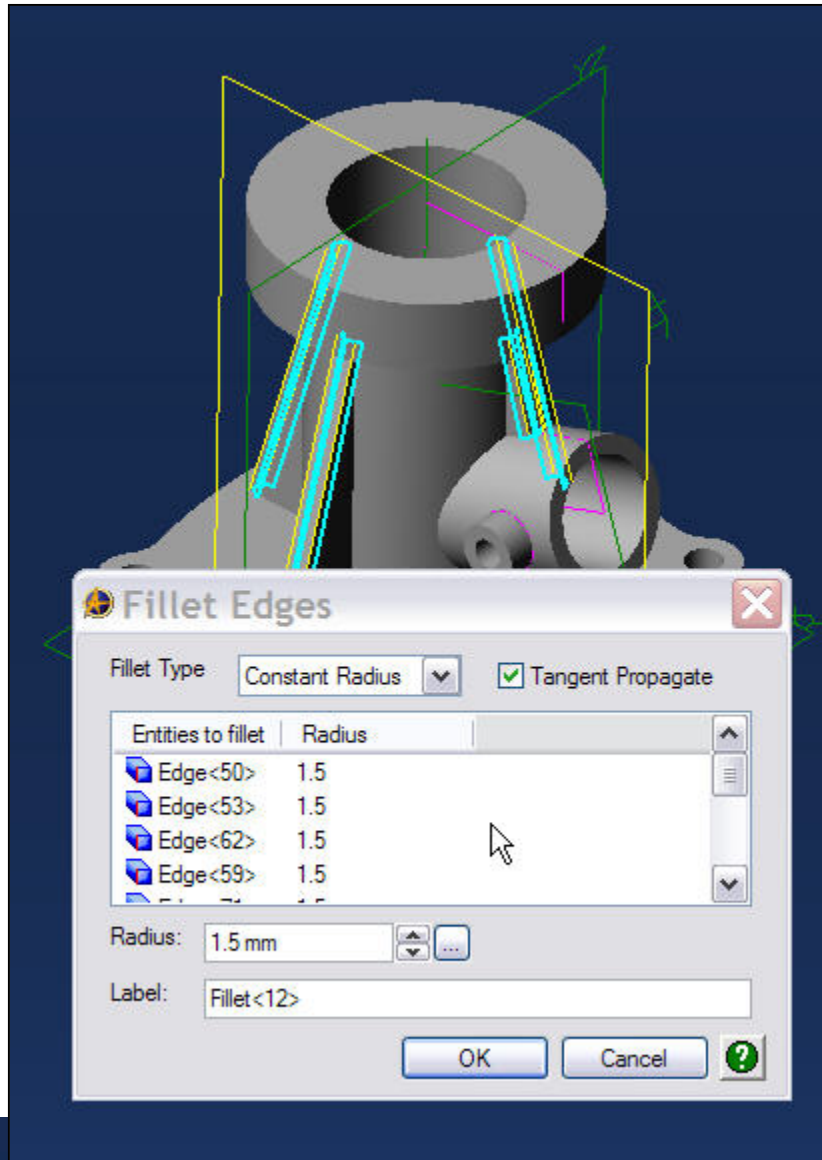


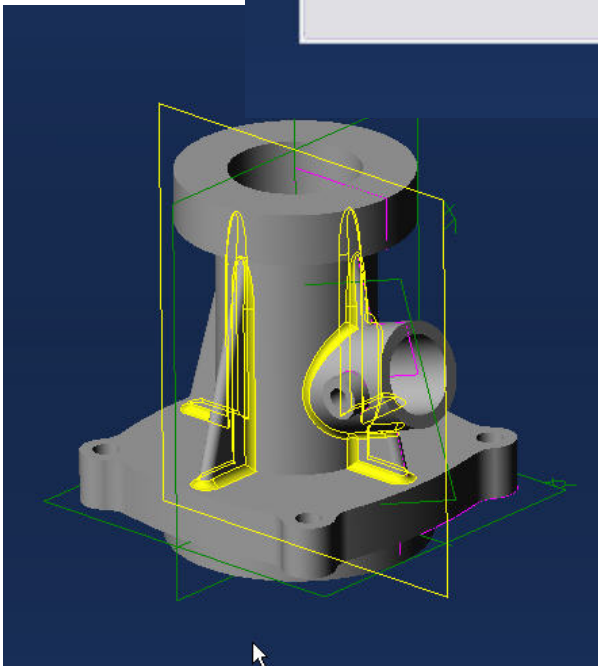
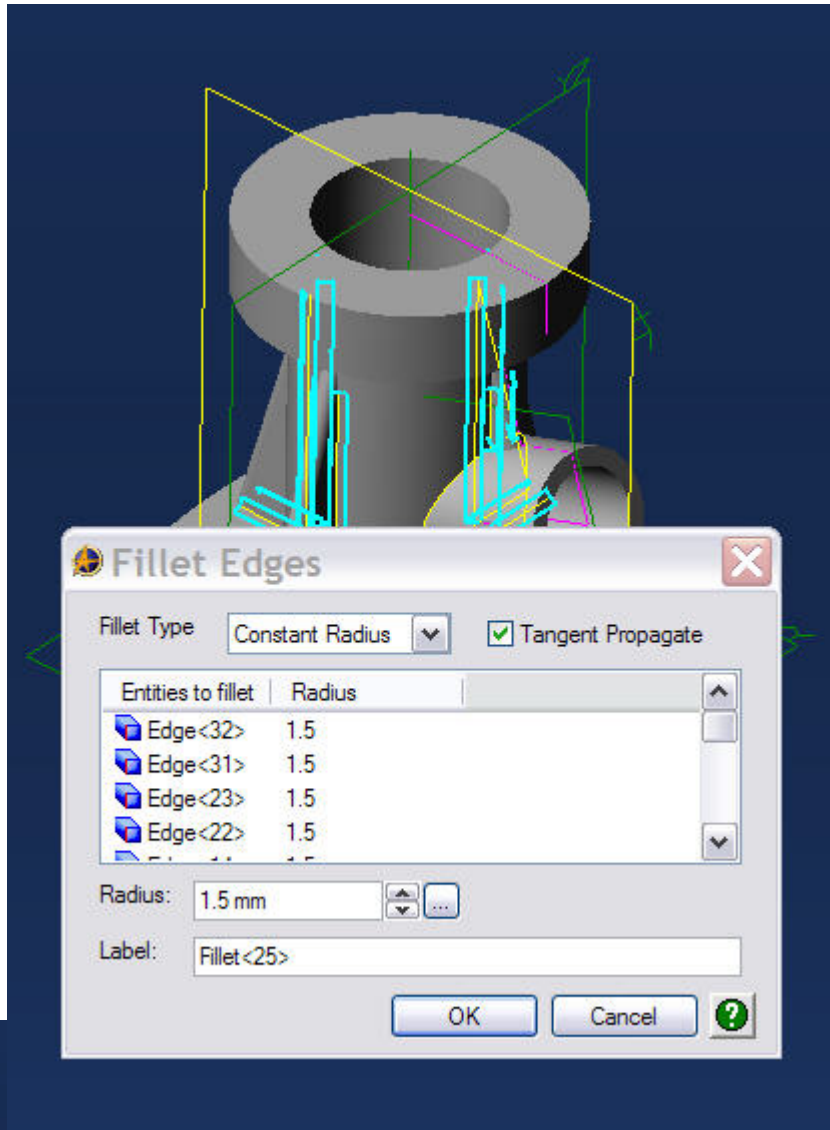
Use the same process for the hole in the small boss.

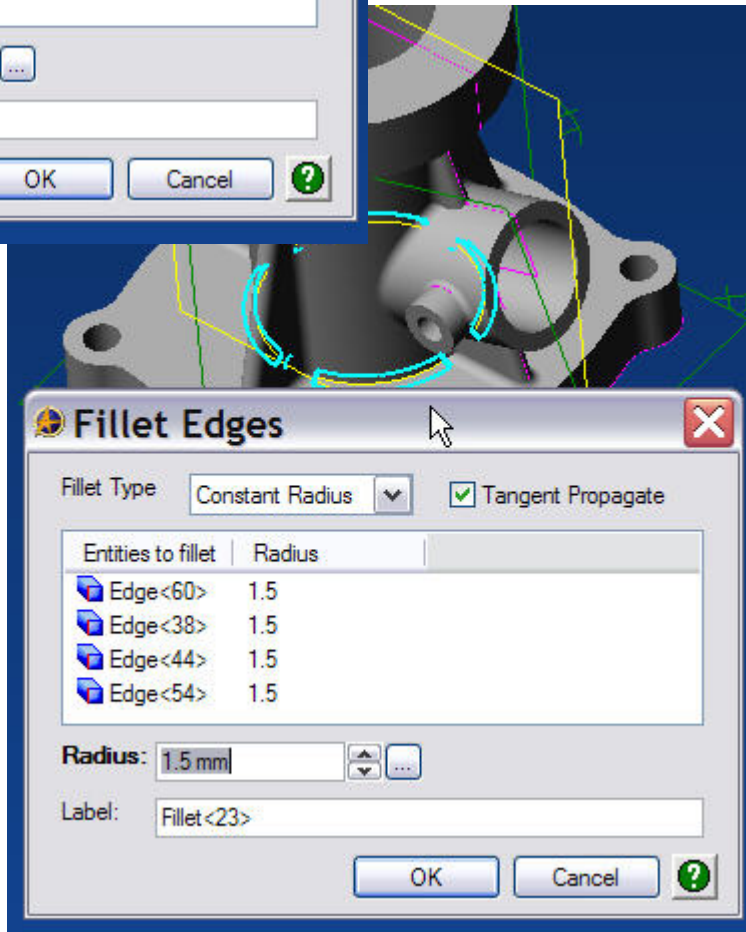
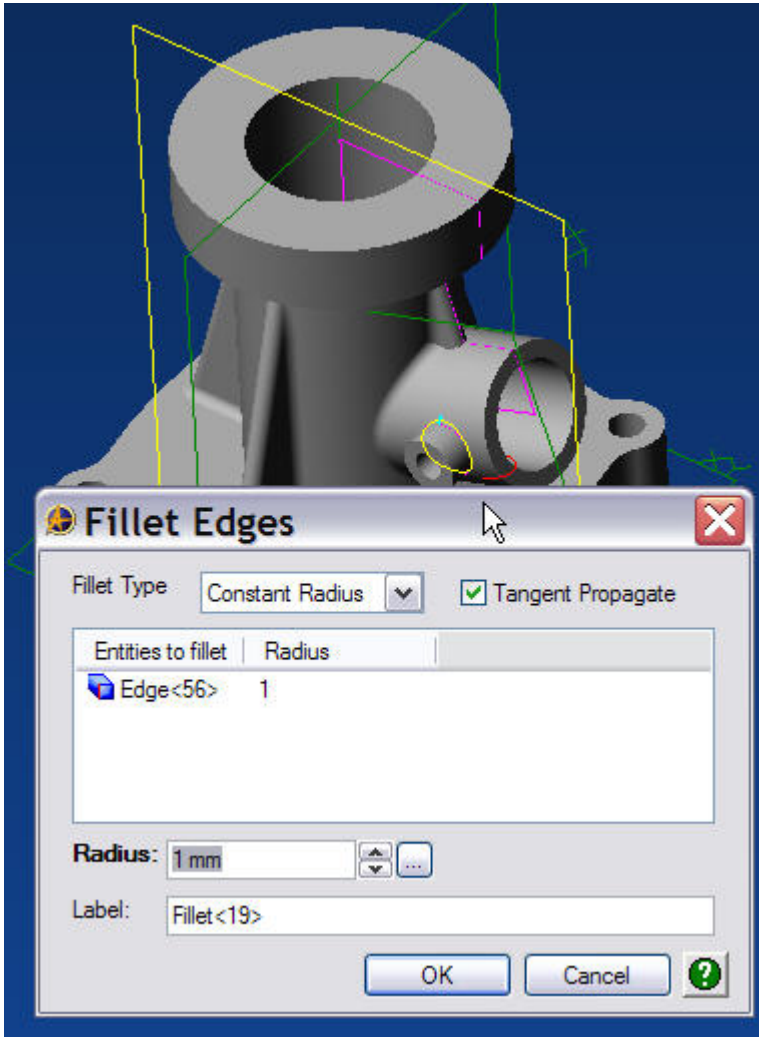


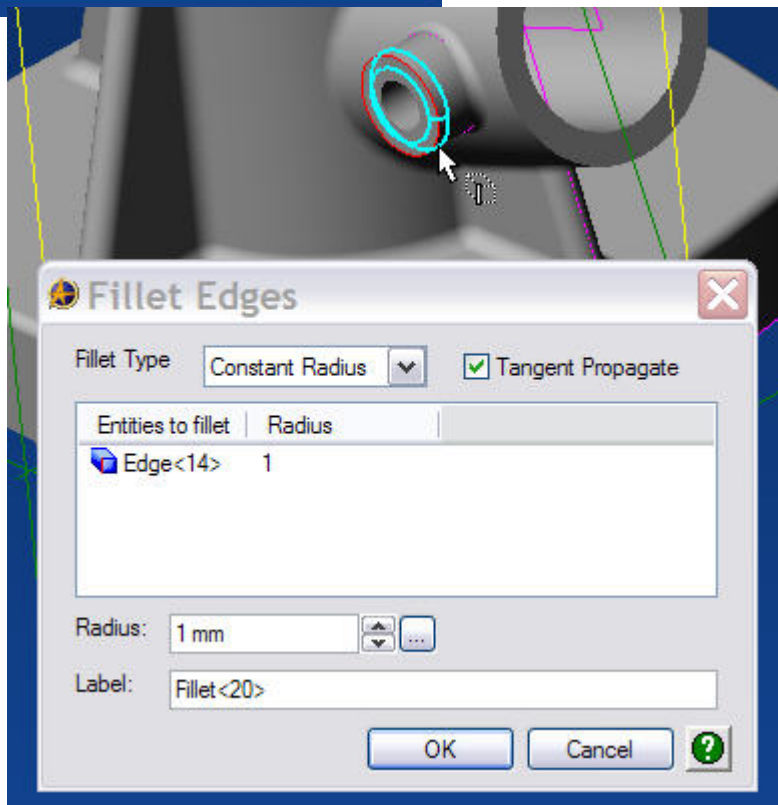
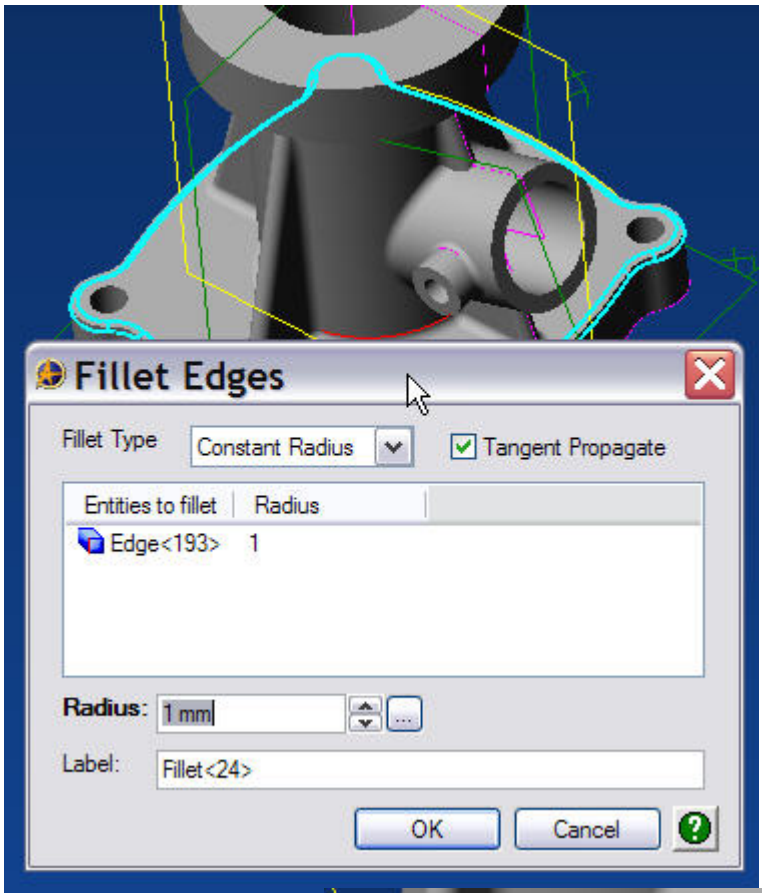
Your part should now look like this.

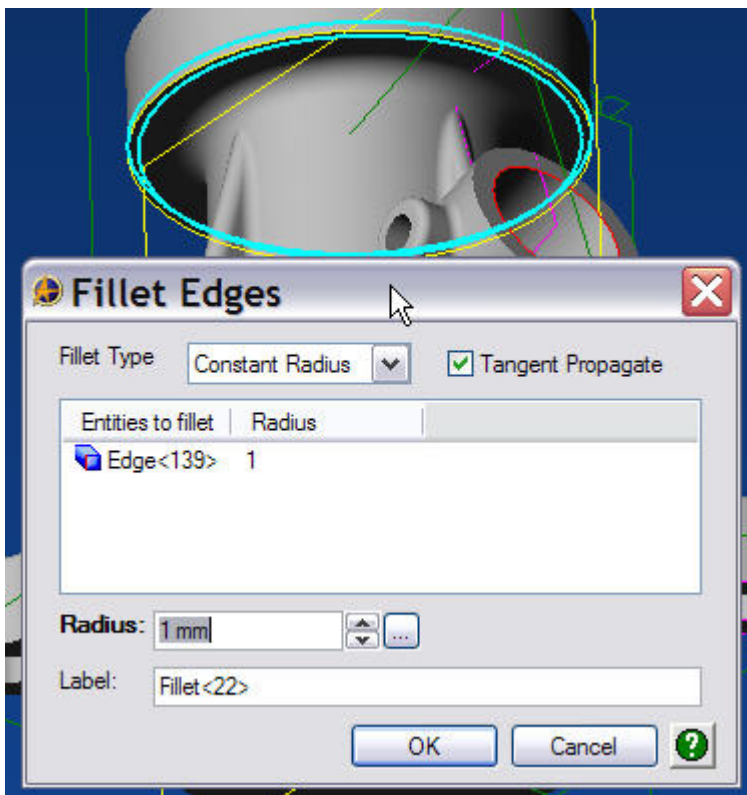
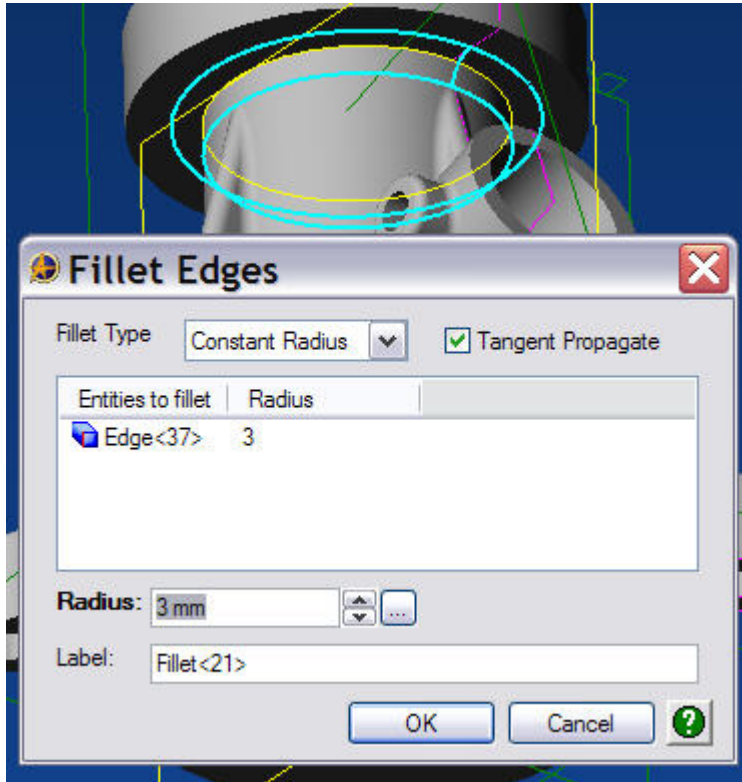
The last thing you'll need to do to finish the part, is to add the fillets and top holes. Fillets can be added in groups. This is best accomplished if the fillets do not touch one another. Something else to keep in mind, is the fillets should be added in a way that helps to support the other fillets you create. Add the fillets in the order shown in the following pictures.



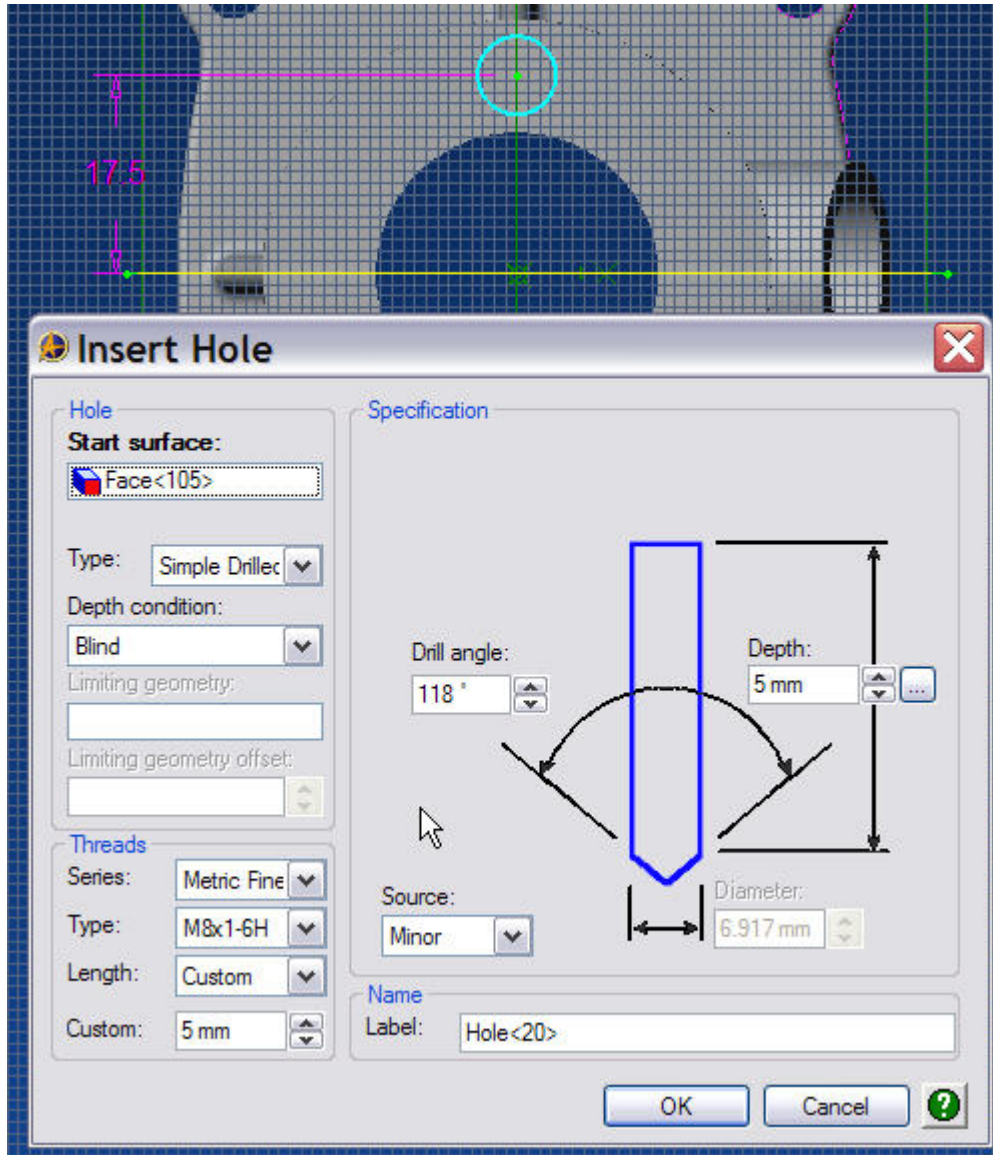






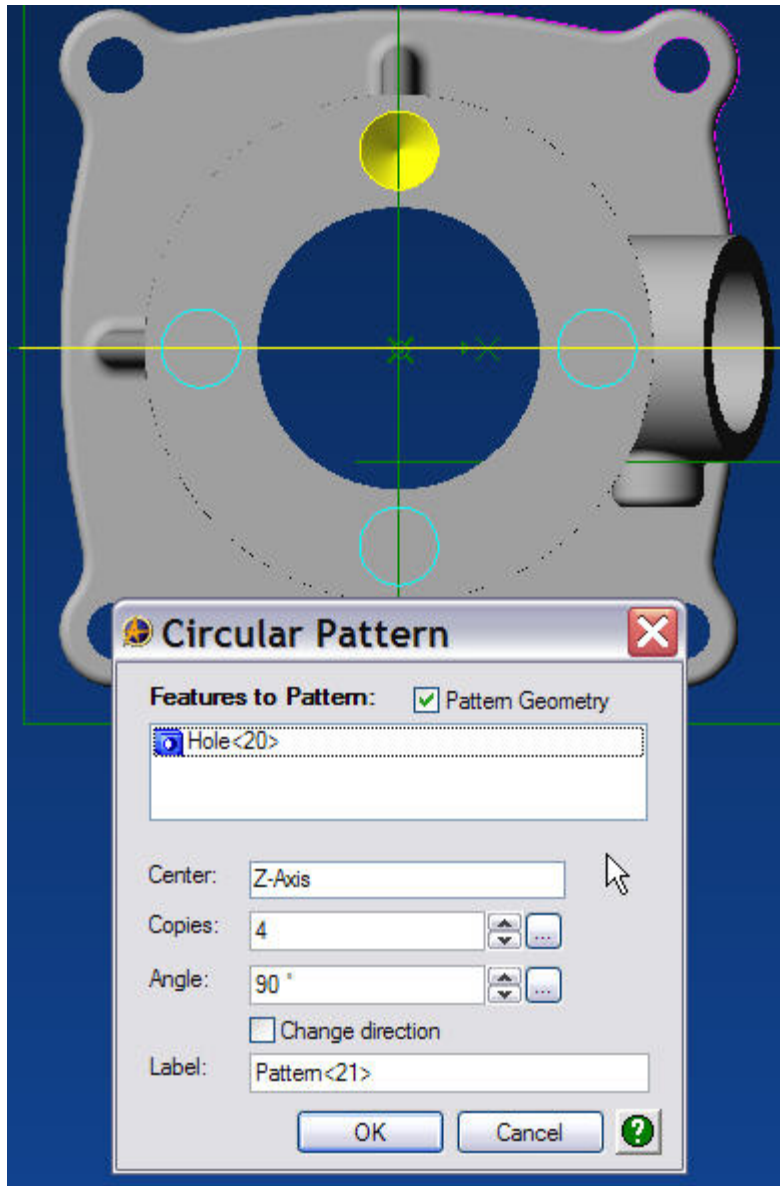


Now it's time to add the four drilled and tapped holes in the top flange of the casting.

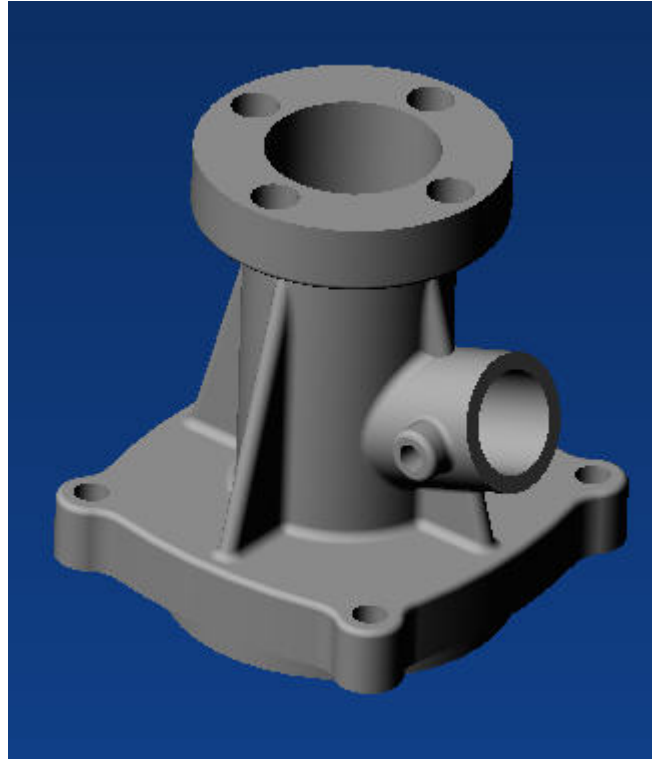


Click on the top flange and then click on the 'Hole' icon. Insert one machined hole in the top flange with the information shown above.

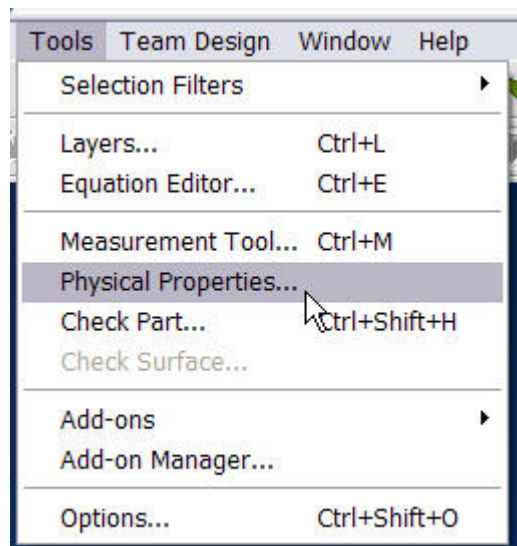
Then use the 'Circular Pattern' command to create the 3 other holes.



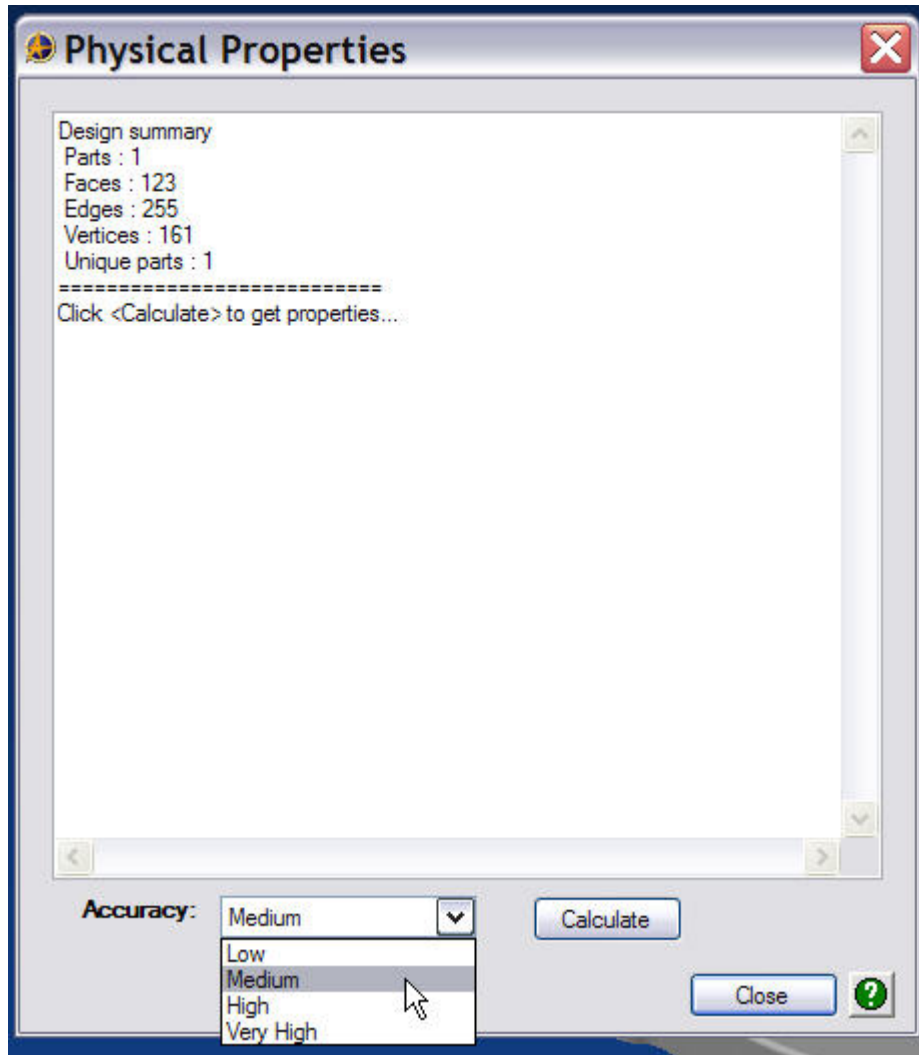
Congratulations you've completed the Inlet Manifold Tutorial and hopefully your part looks like the one below. If it doesn't, the model file is available at www.carrinfo.com/InletMaifold.stp. It's an Alibre STEP file and contains the complete history tree for the model.



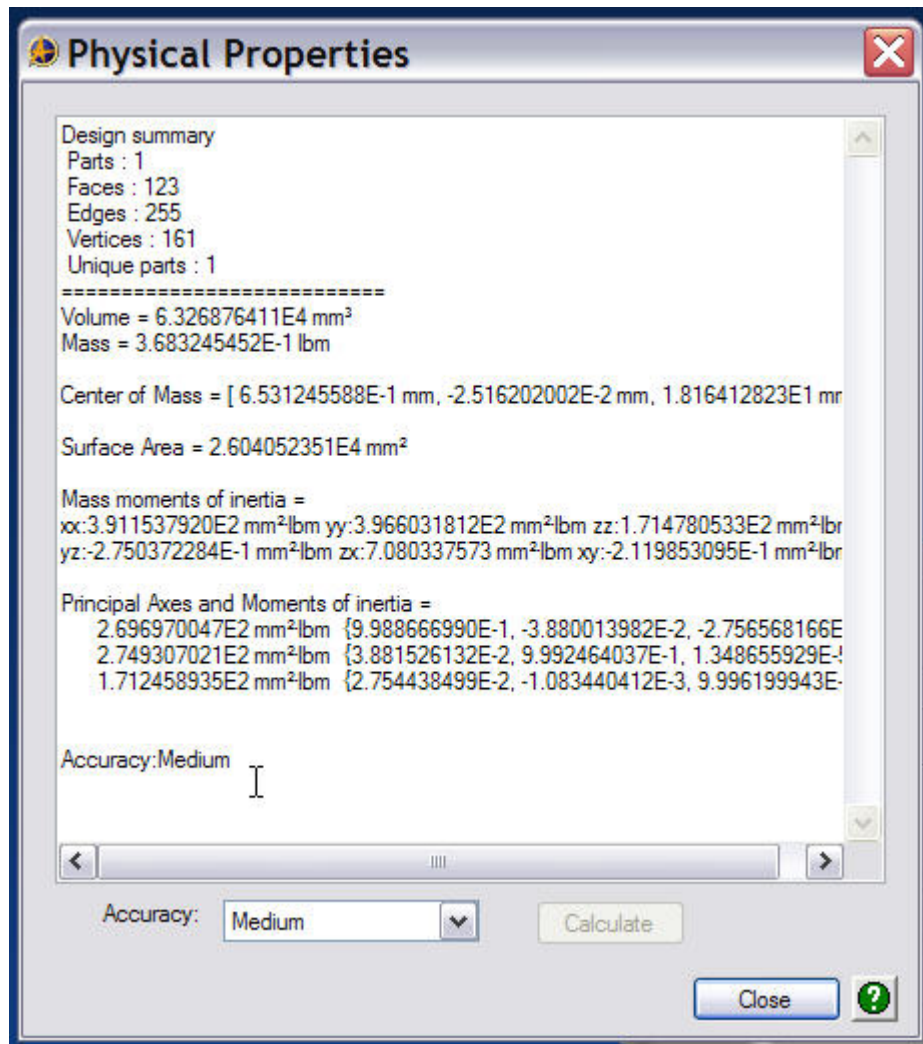
As an added feature of this tutorial, you'll remember at the beginning of the exercise we assigned a material to our part (Aluminum cast-hammered). To see another powerful feature of the Alibre Design application go to the 'Tools' tab on the top menu bar and select 'Physical Properties'.



The 'Physical Properties' panel will open. Select the level of accuracy you want to use and then click 'Calculate'



The results will be displayed in the panel. Try doing that with a surface model!



This ends the 'Inlet Manifold' tutorial.

Look for more tutorials in the future, and a complete training manual covering Alibre part design, drawing and BOM creation, assembly design, sheet metal parts, and how to drive designs through Excel spread sheet design tables.