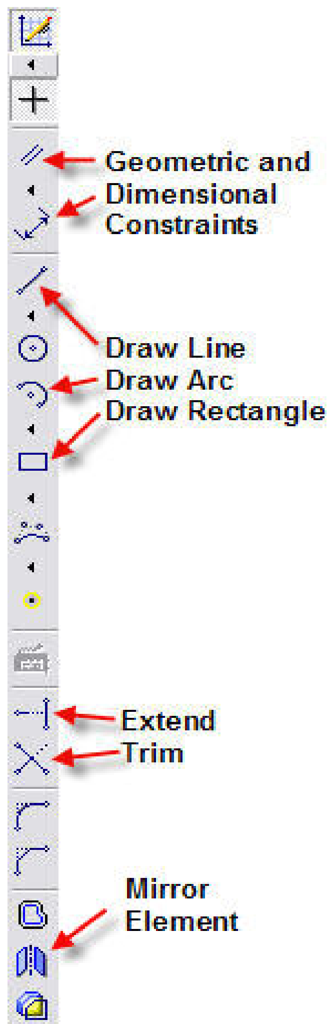
Chapter 1 - Designing the Any Angle Tool Vise - The Saddle Base

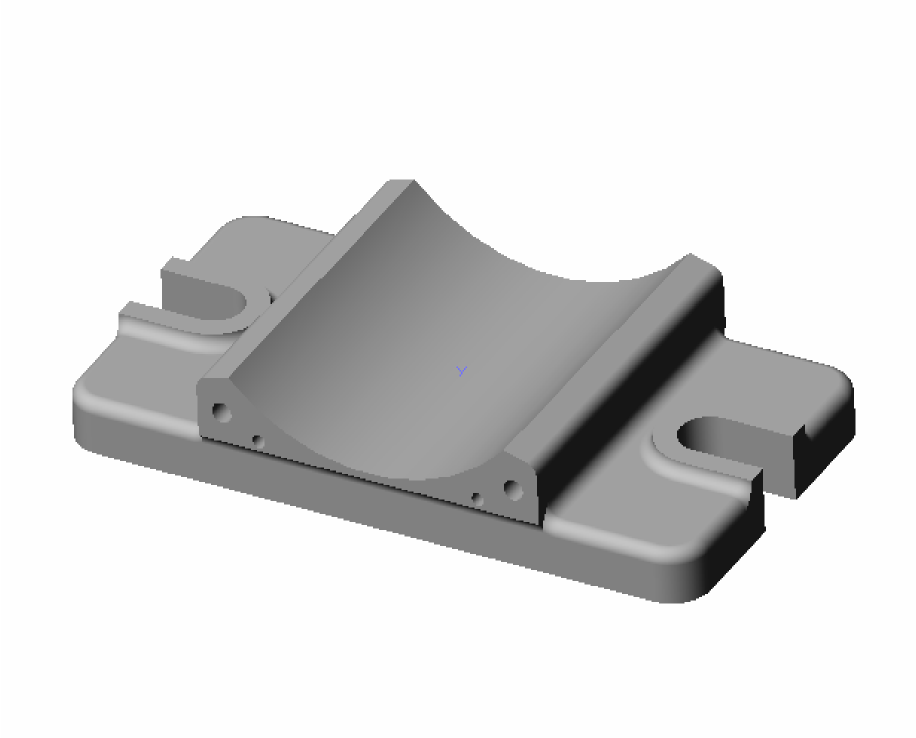
In this section of the design module, the following Workspaces and command sets will be introduced:

1. Sketcher Workspace
   - Draw Rectangle Command
   - Draw Line Command
   - Draw Arc command
   - Mirror Element command
   - Sketch Constraints both Dimensional and Geometric
   - Extend command
   - Trim command

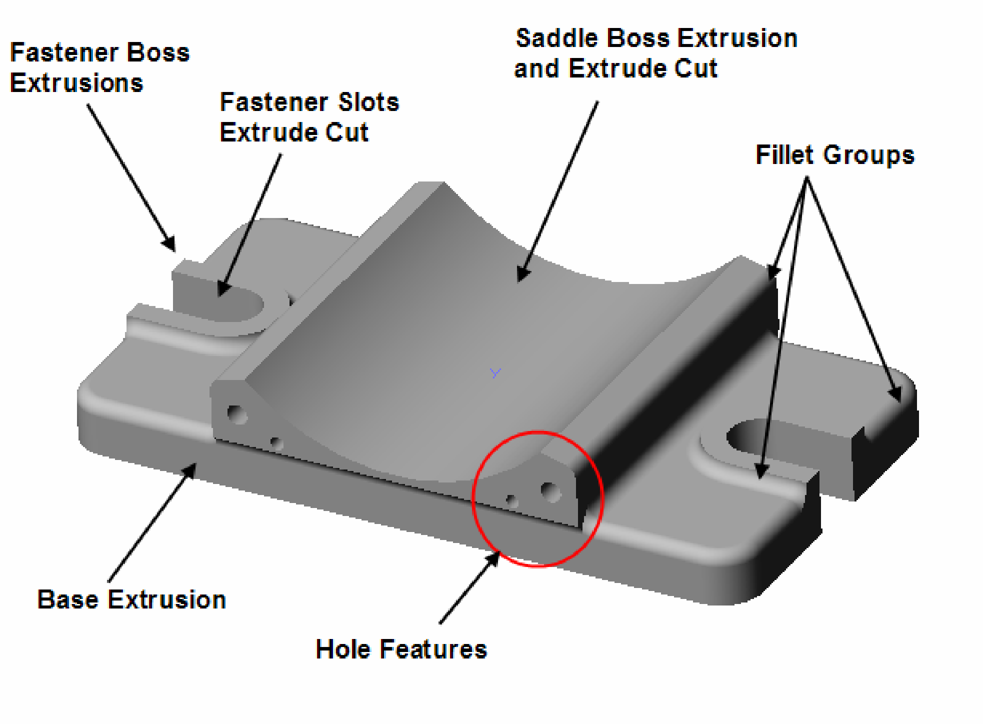


2. Part Design Workspace
    - Extrude Boss
    - Extrude Cut
    - Hole
    - Fillets
    - Mirror Element

The first part of the ‘Any Angle Tool Vise’ we’ll model is the base or ‘Saddle’. The picture below shows the finished part. Refer to Drawing 1- Saddle, for all construction dimensions.



Analyzing the part, we can see that it’s constructed of a combination of design features or elements.



In the real world, our design would start with an idea or concept for a tool that could be used to hold a work piece in a variety of positions, and progress through various design iterations until we arrived at a final concept. Then and only then would the real work of capturing, and detailing our design begin. In the case of the Any Angle Tool Vise we'll pretend that the design has reached the final stage and our job now is to create the models and drawings necessary to convey these ideas to a manufacturer. We'll refer to the drawings in the appendix for all design information, and begin planning a course of action for each piece that will allow us to construct them in the most logical manner possible.

We'll begin with Drawing 1, the Saddle for the Any Angle Tool Vise.

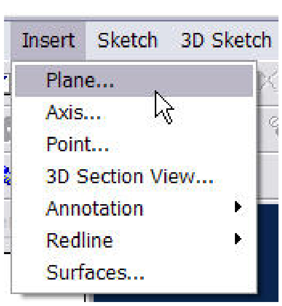
A typical course of action might evolve like this;

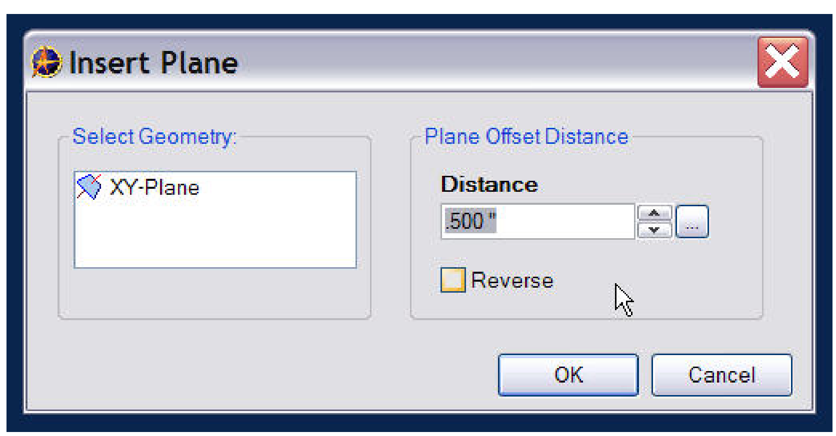
1. First, create the base from a simple rectangular sketch, and extruded to a specific depth, using the Extrude Boss' command.
2. Then, create the Saddle Boss by either combining using the 'Extrude Boss' command and the 'Extruded Cut', or…
3. …by using just the Extruded Boss' comand.
4. Next, create the Fastener Bosses using the 'Extrude Boss' command.
5. Then, create the slots by using the 'Extrude Cut' command.
6. Next, create the holes using the 'Hole' command, and duplicate them using the 'Mirror Element' command.
7. Lastly, apply the fillets groups to the base using the 'Fillet' command.

In this creation process, it's possible to use elements (faces, edges, verticies, etc.) of previously created features to anchor added sketch elements and part design features to, and in some cases it is preferable to do so. But in keeping with the best practices list for creating solids, there are several entities that should be constructed before sketching begins. These are the three reference planes shown in the figure below. The reference planes serve as geometric anchors for the following features:

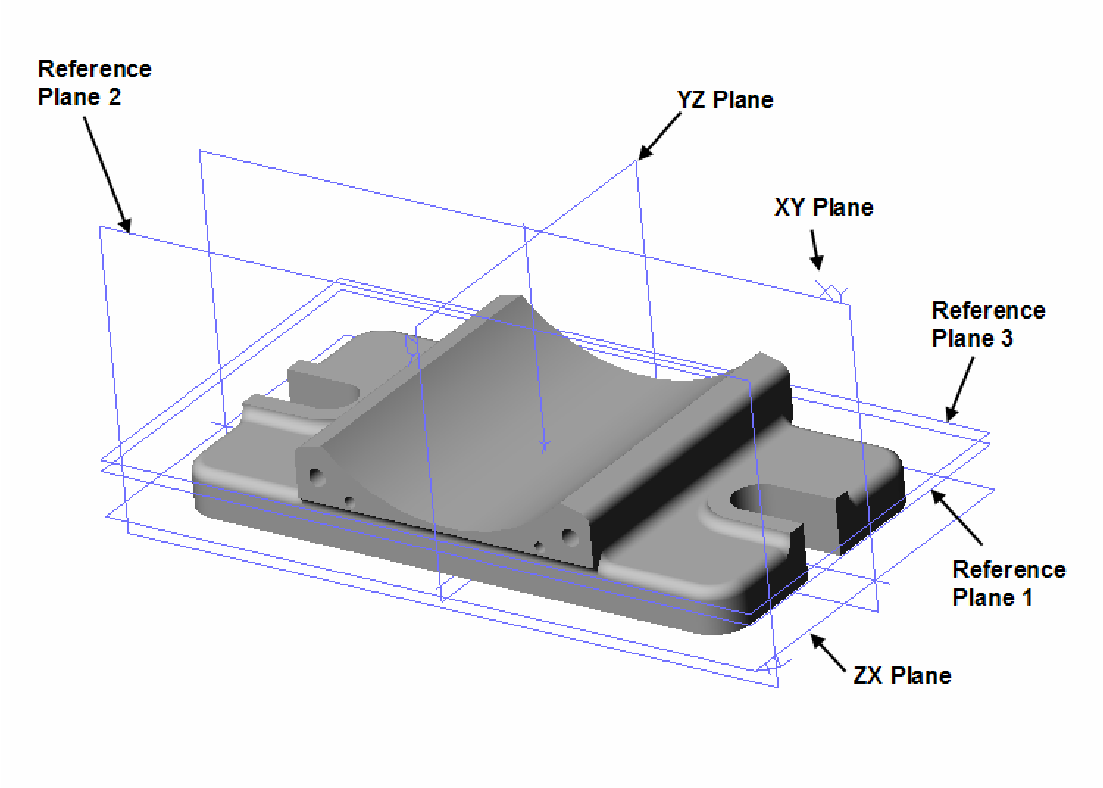
1. The Saddle Boss – Reference Plane 1 - **offset .5in from the XZ plane**
2. The Saddle Boss Cut Out – Reference Plane 2 – **offset 1.75in from the XY plane**
3. The Fastener Boss – Reference Plane 3 – **offset .625in from the XZ plane.**

They 'Planes' are created by using the 'Insert Plane > Offset' command.





After creating the reference planes, the basic sketches defining the part can be constructed.



Note: Planning for future modifications is an integral part of good design practice. This is why the closer all designers adhere to the 'Best Practices' guidelines, the easier it will for another designer be to make edits to the part. It is not an un-common occurrence in organizations that do not enforce a best practices approach in part design, to have designers completely re-build a part when changes are required. This is not only a waste of valuable time and resources; it can result in discrepancies between the new model and the previous model data that in turn re-defines the tooling data, etc. Parts should be thoroughly analyzed before work begins in order to establish the most efficient and logical design strategy. Jumping into a design without a good plan is a sure recipe for future problems.

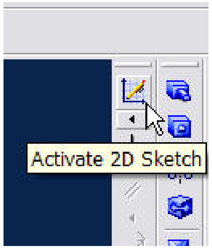
There are also several practices you'll want to make a habit of, if you haven't already done so. They are:

1. Save your work after each major design change
2. Constrain all design sketches completely (0 degrees of freedom), if possible before creating any 3D parts.
3. Analyze your sketches before creating any 3D parts.
4. Check your part using the 'Check Part' command after each major design change, using the High or Very High modifiers and correct any problems before moving on to the next design stage.
5. Save your work. Save it often.

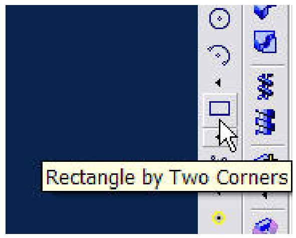
Now, lets get started creating the Any Angle Tool Vise Saddle.

The Saddle Base

Enter the sketcher workspace by clicking on the XZ plane in the Design Explorer panel, then click on the 'Sketch' icon.

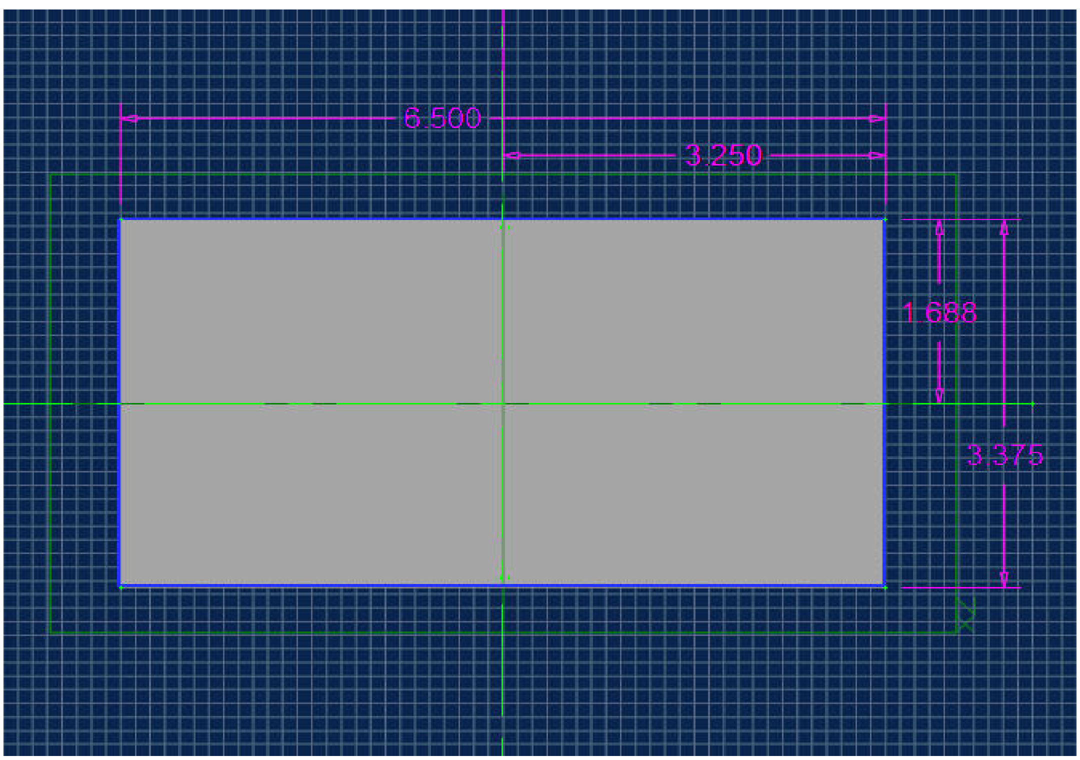


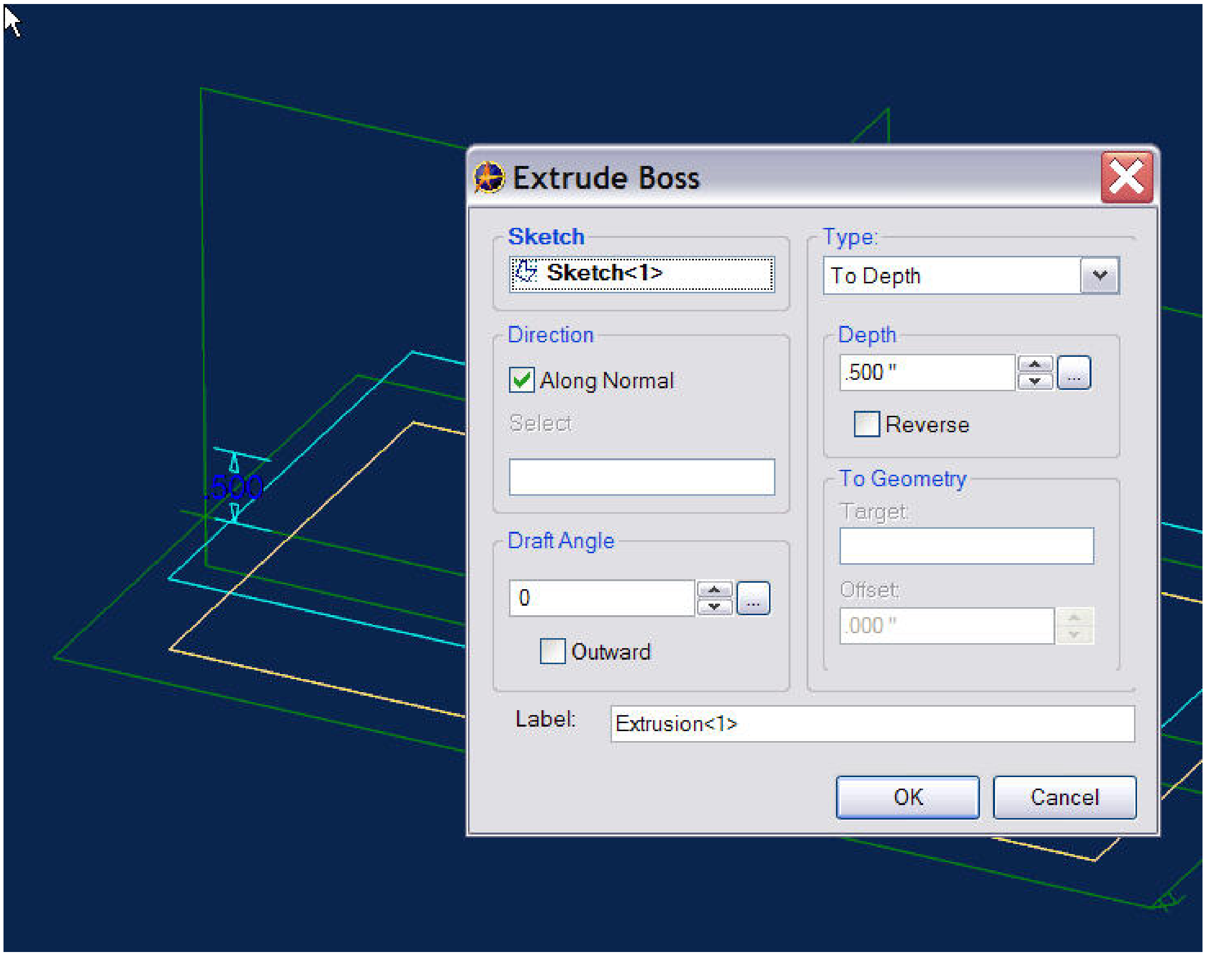
The 'Base' is developed from a simple rectangular sketch that defines the overall dimensions of the part, and is anchored to the geometric supports of the XZ plane, and the X and Z-axis. Click on the draw 'rectangle' icon.



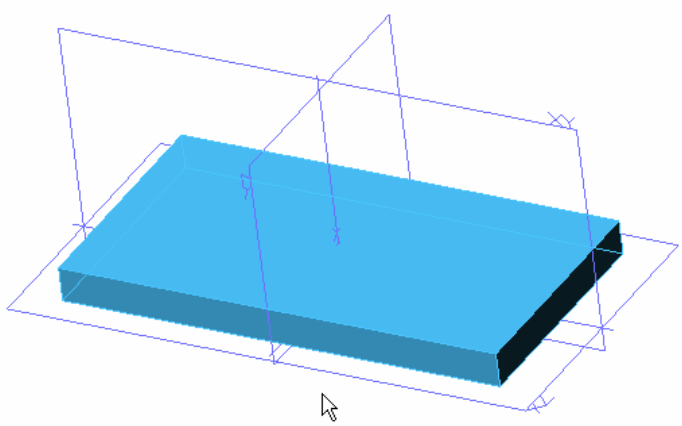
Position and dimension the sketch show below. Remember to check the DOF (degrees of freedom) indicator in the lower right hand corner of the screen to make sure the part is fully constrained. .

Extruded the sketch to the thickness **.5in** desired using the 'Extrude Boss to Depth' command.

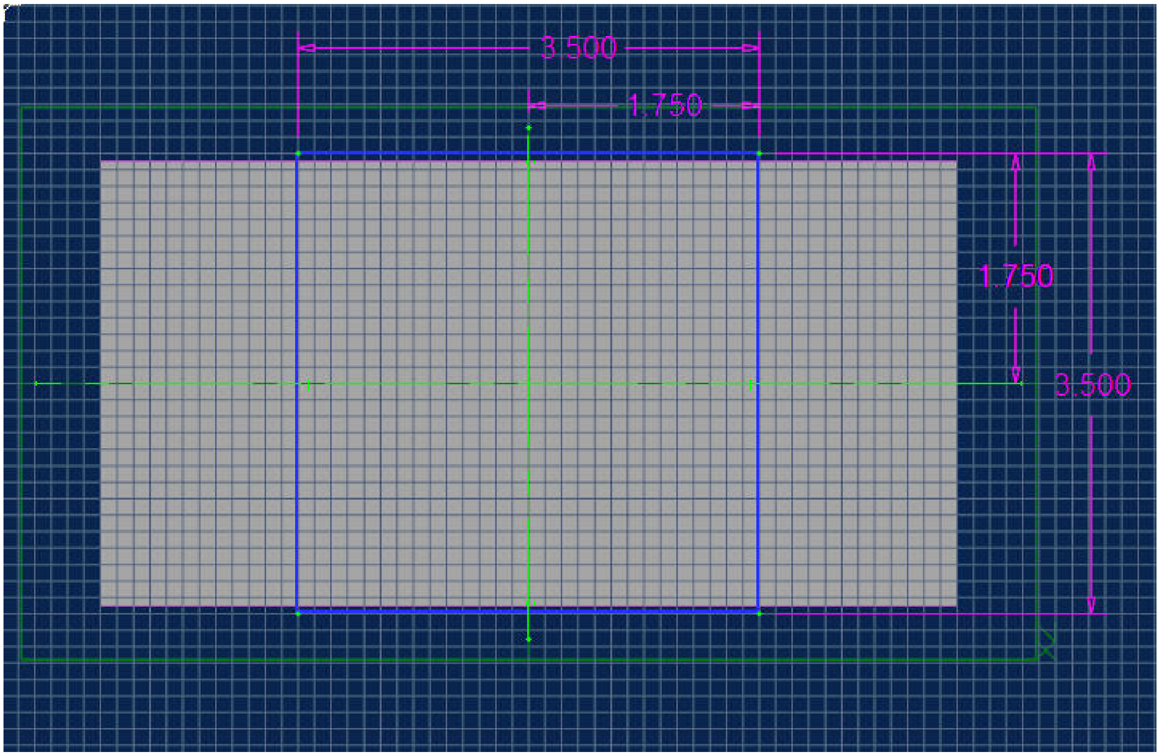




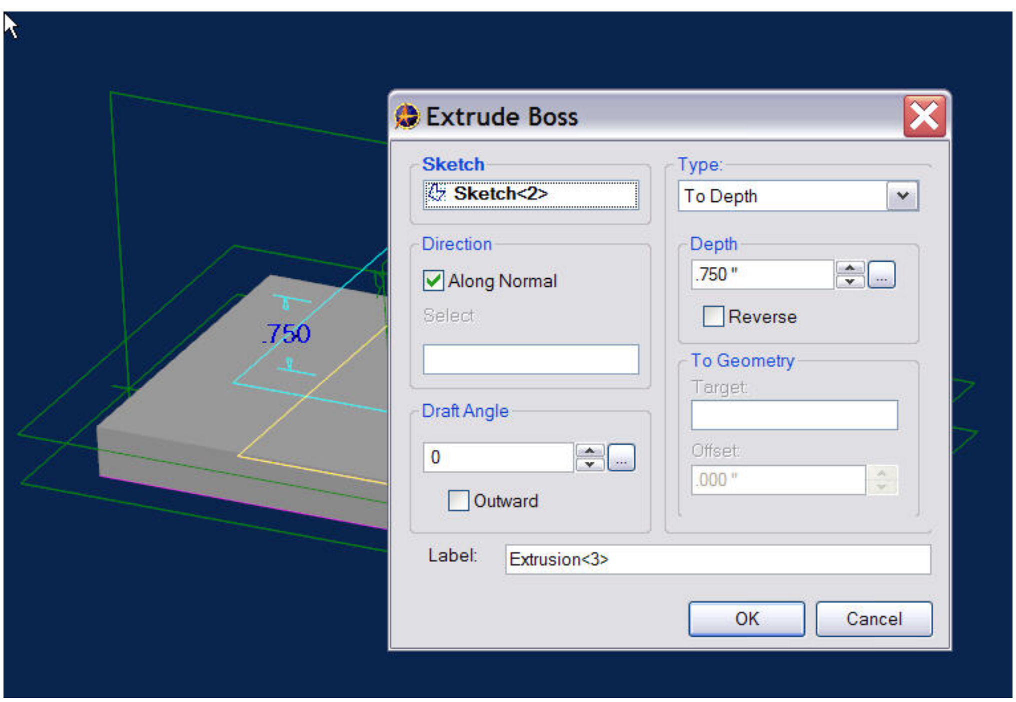
The next illustration shows the extruded base part.



The next step is to define the 'Saddle Boss'. In this example, it is a two-step process. The first step is to create an extruded boss. The second step is to remove material from the boss using the 'Extrude Cut' command.

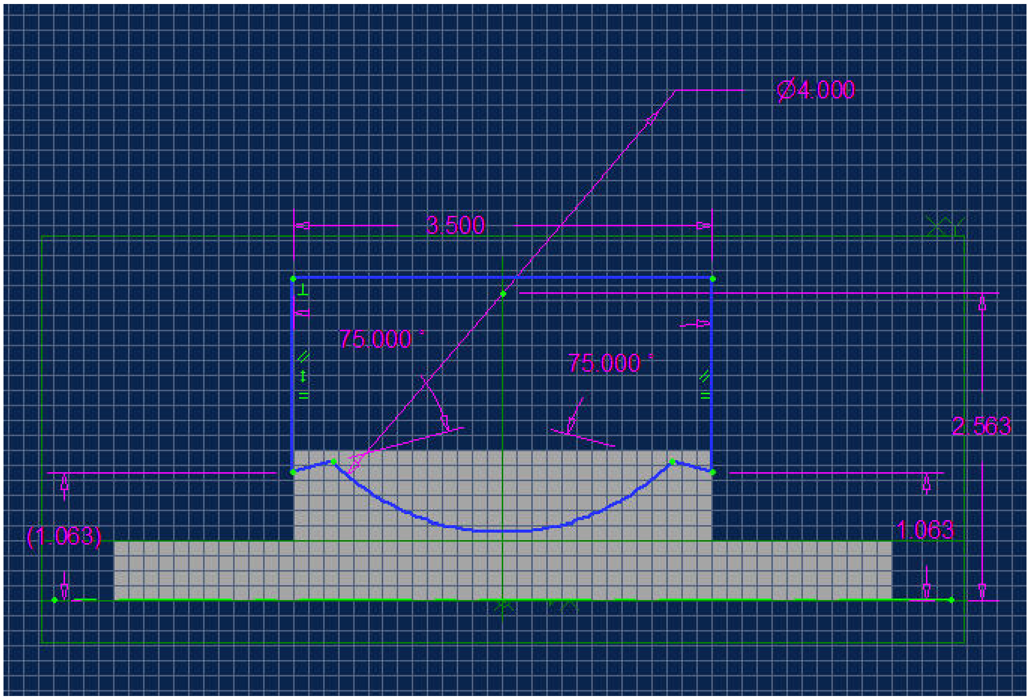


Using Reference Plane 1 as the anchor, create and constrain the sketch as shown above. Then extrude it to a height of .75 in.

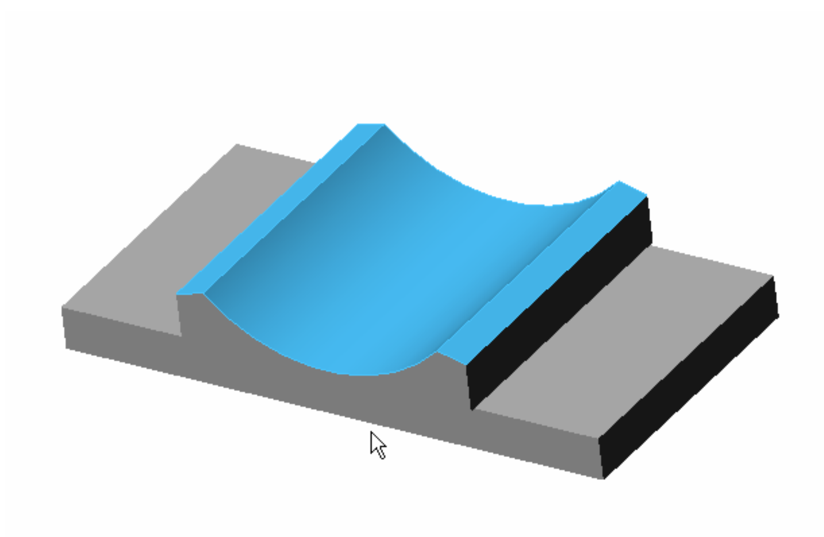


Extruding the pad to a height of .75in ensures it is high enough to contain the cut features we'll develop in the next step.

Now create the 'Saddle Boss Cut' feature. Select Reference Plane 3 and enter sketch mode. Create the sketch shown, and then use the 'Extrude Cut Through All' to remove the material from the Saddle Boss. Again, the height of the top line in the sketch is not critical, as long as it clears the profile of the Saddle Boss.

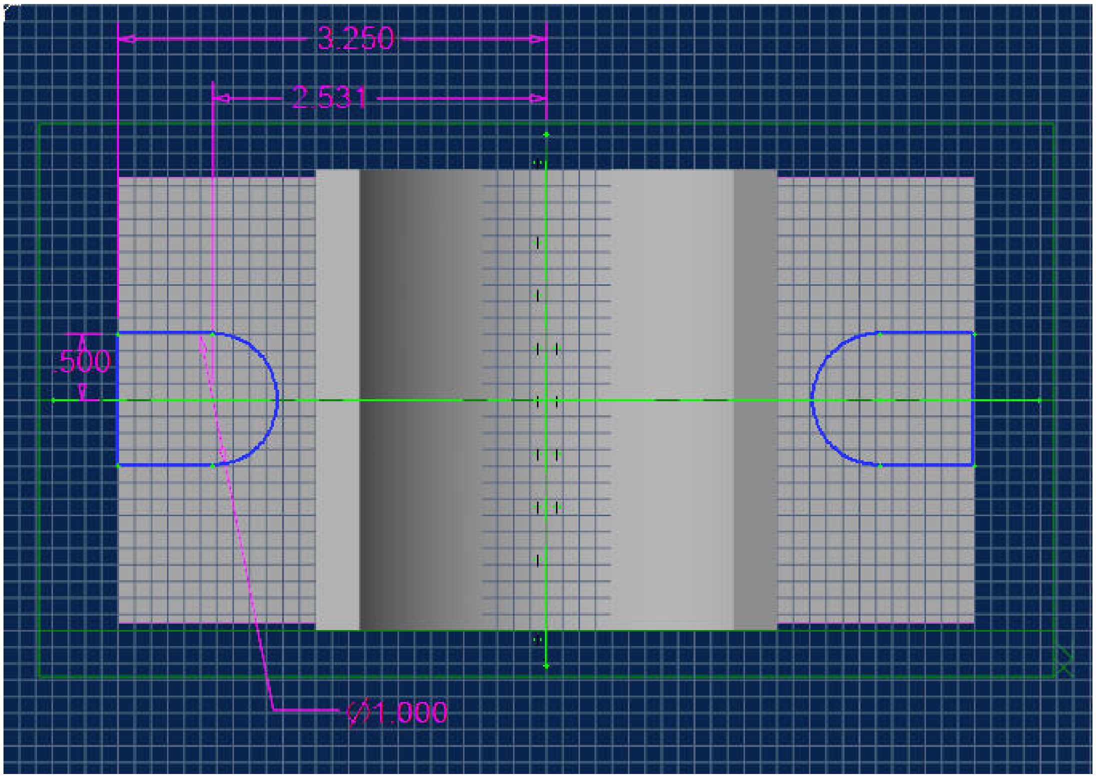


The part should now look like this.

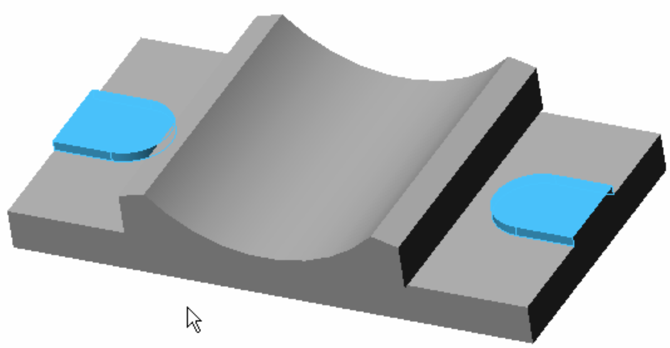


The Saddle Boss could be created using a single sketch that incorporates the radial saddle feature. The reasoning behind doing it as shown in this example is that by creating it using both the Extrude Boss and Extrude Cut commands the designer gains added control over the individual features in the part, resulting in easier edits in the future.

The next step will be to add the Fastener Bosses. Select Reference Plane 3 and enter sketch mode. Develop the sketch show in by drawing one boss sketch and then mirroring the entities using the X-axis as the symmetry axis. This could also be accomplished by extruding one pad boss and the mirroring the 'Feature' in the part design workspace. Always anchor the sketch geometry to common reference elements, and not to any underlying elements contained in the solid model. (See Section 3.x.x.x for explanation)



Use the ‘Extrude Boss to Geometry’ and select the top face of the pad base as the target geometry to create the two pads as shown in the illustration below.

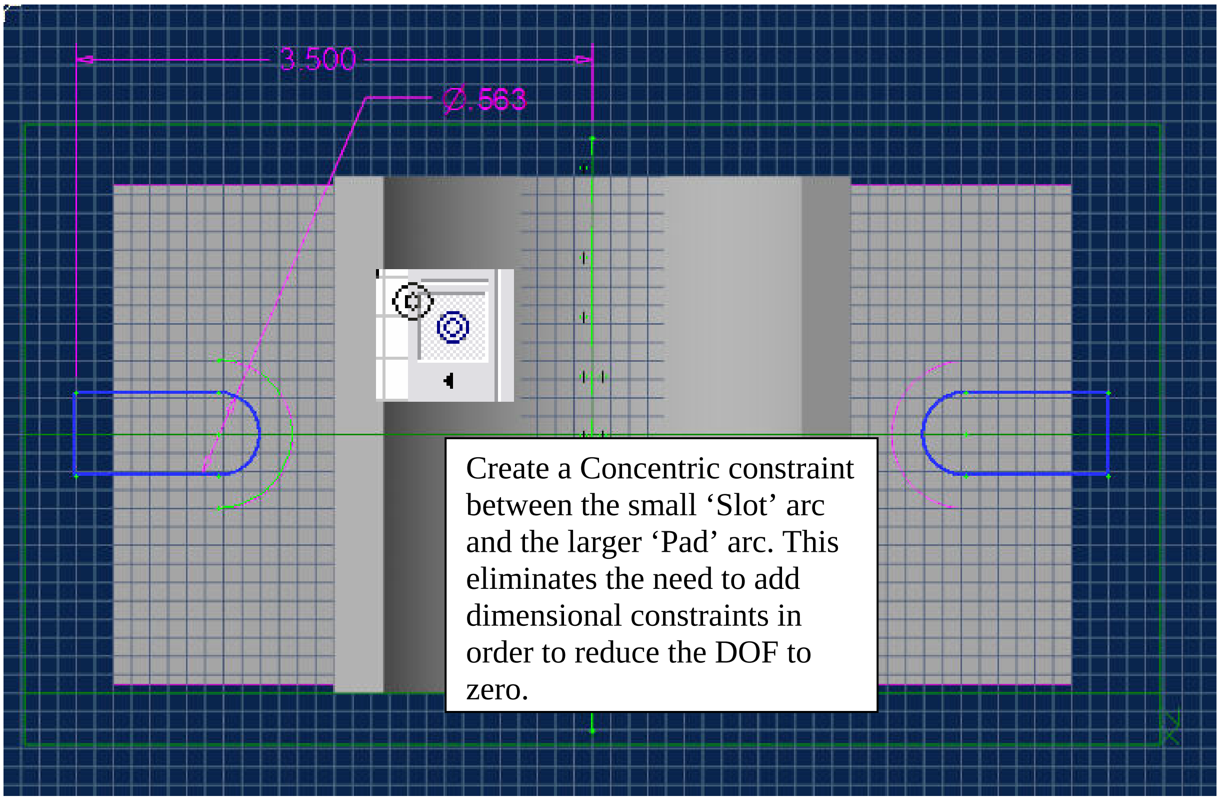


In this case, using the ‘Extrude to Geometry’ modifier allows the height of the pad to follow the top surface of the base part. If the base grows or shrinks, to the limit of Reference Plane 3 the pad height grows or shrinks in response.

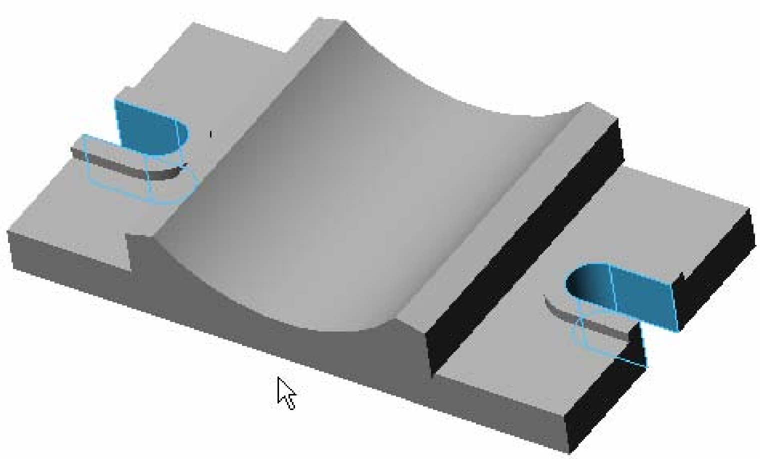
The alternative to this is to use the upper face of the base as the sketch anchor and extrude the pad upward by .125 in. Although this allows the pad to follow changes in the height of the base, it could result in problems if the width of the base is changed enough to fall inside the pad profiles.

The next step is to create the slots in the 'Fastener Bosses'. The procedure is the same as that used to create the fastener pads.

In this case, extend the end of the sketch past the end of the base profile and then use the 'Extrude Cut > Through All' command to create the slots. Extending the sketch past the end of the base profiles helps to ensure the complete removal of material from the vertical face of the base, and eliminates any possibility of problems in the event the draft angles are added to the base in the future.

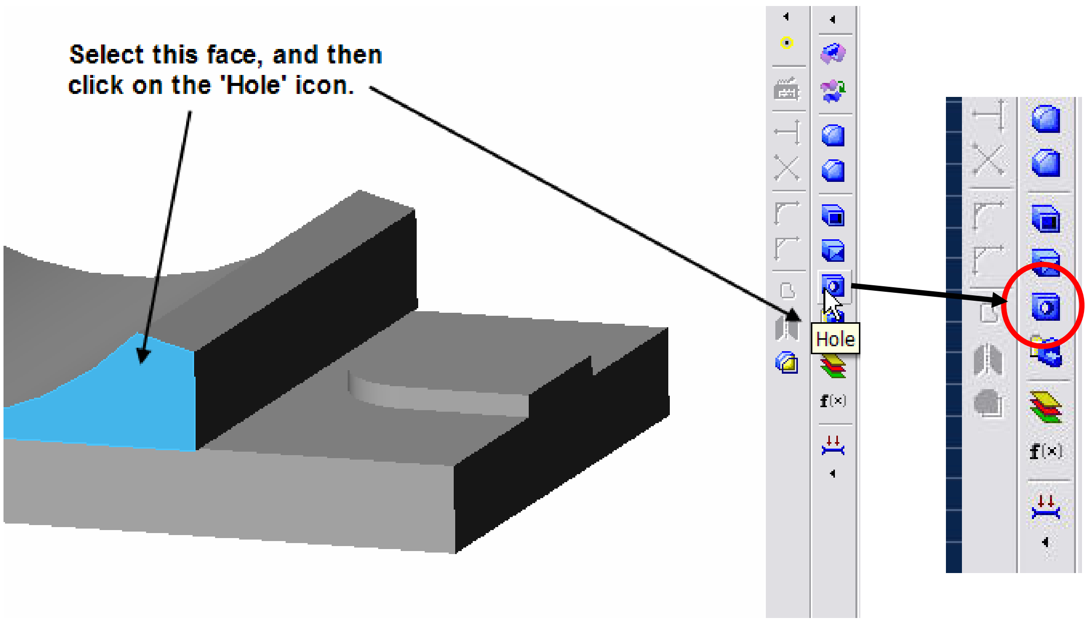


Your part should now look like this.

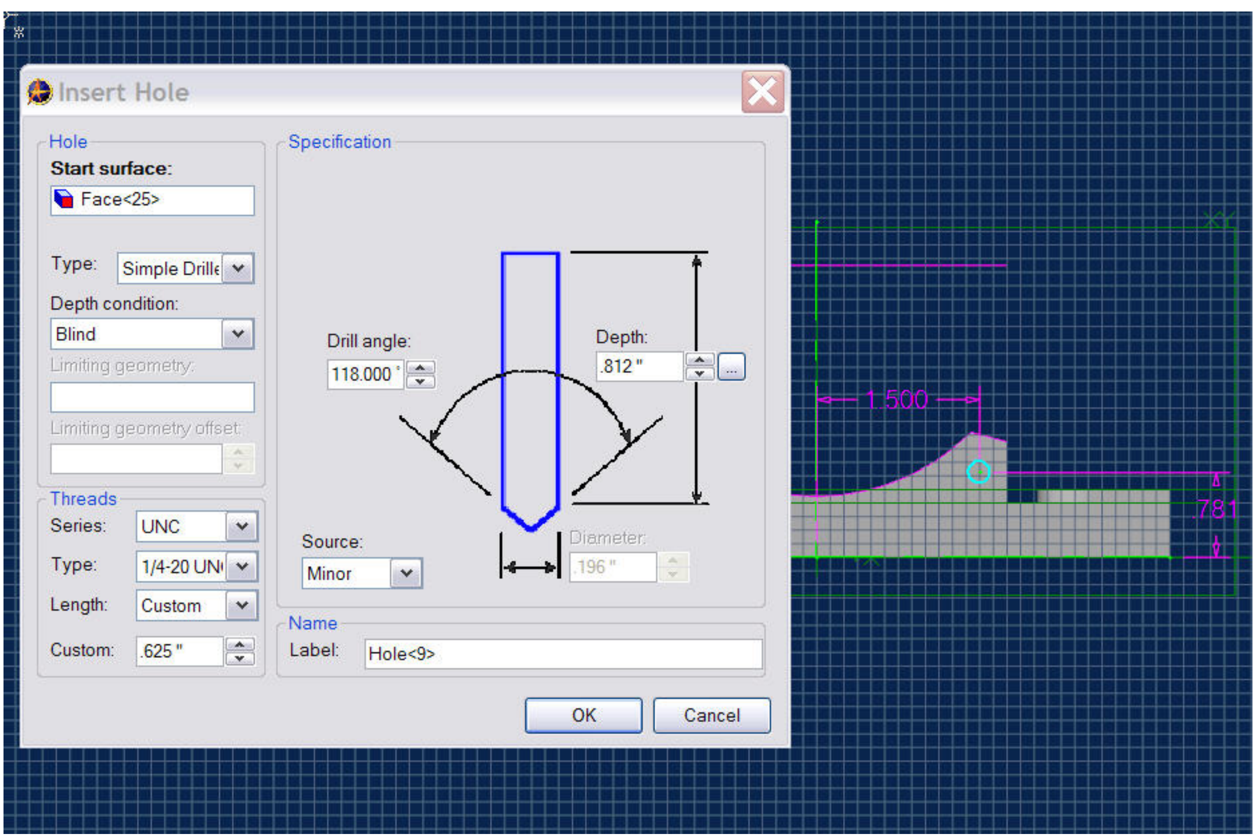


Now that the base design is complete, it's time to add 'Design Features' (holes), and 'Dress-up Features' (fillets) to the part. Remember, per the 'Best Practices' guidelines it's always preferable to add these features as late in the design as possible. This is to insure that changes to the design or dress-up features don't effect the basic design in un-wanted ways.

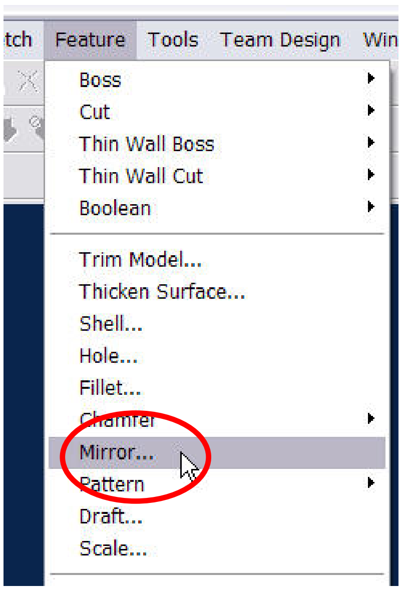
We'll add the 'Design Features' first. Select the face of the Saddle Boss and then click on the 'Hole' icon in the toolbar.

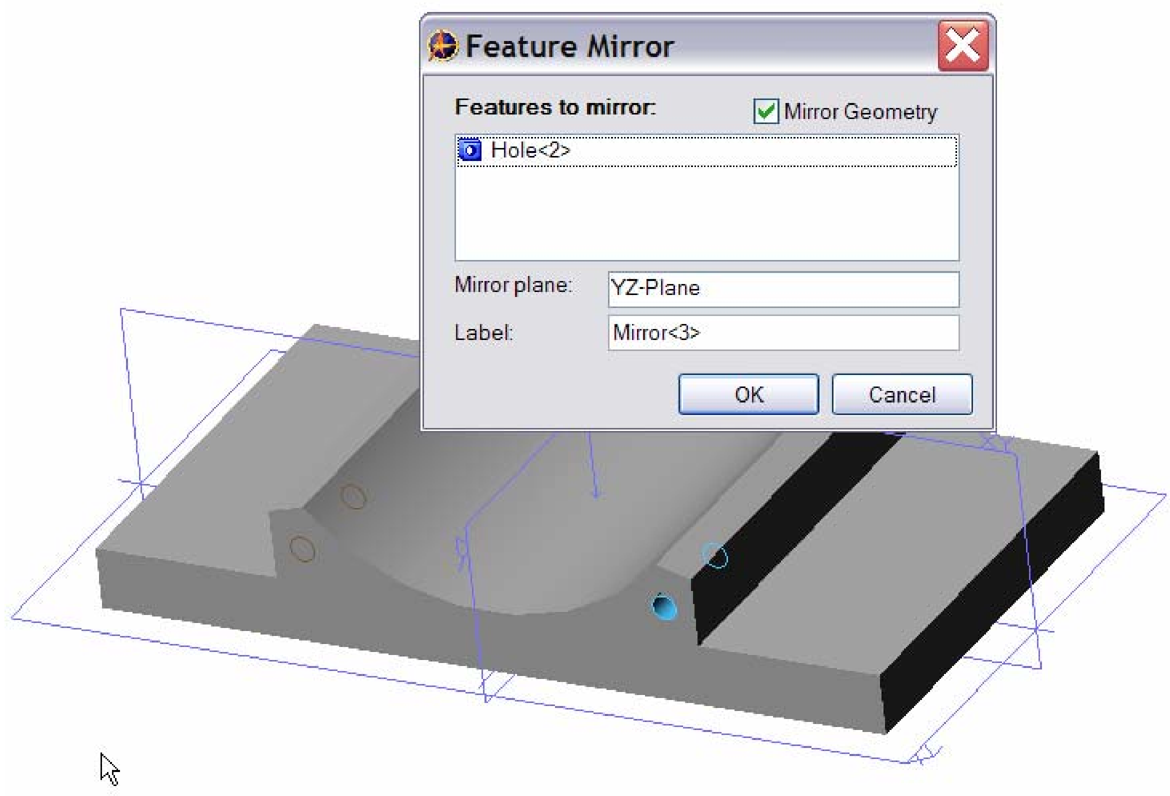


Define the desired 'Hole' parameters, and click 'OK'.

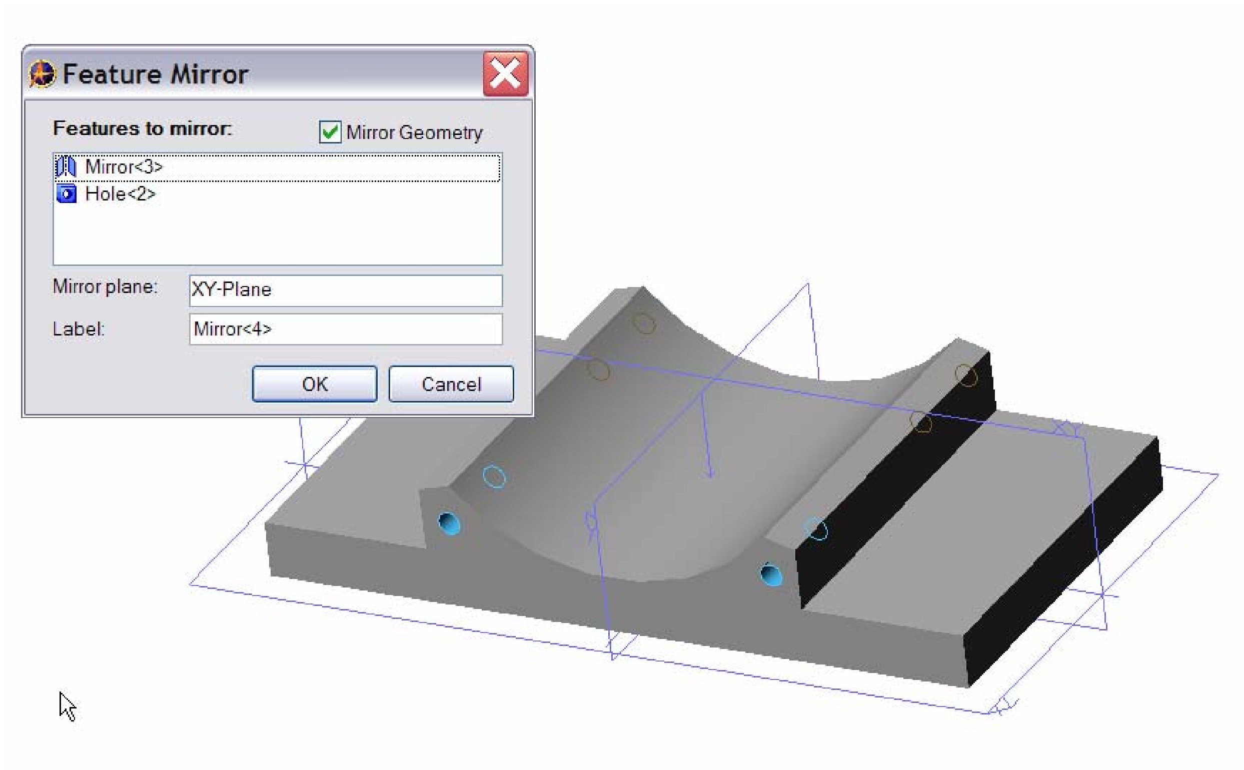


Next, select the ‘Sketch’ in the Design Explorer window, and locate the hole per the dimensions shown in the illustration. Now use the ‘Mirror’ feature command to create the second hole on the left hand side of the Saddle Boss.



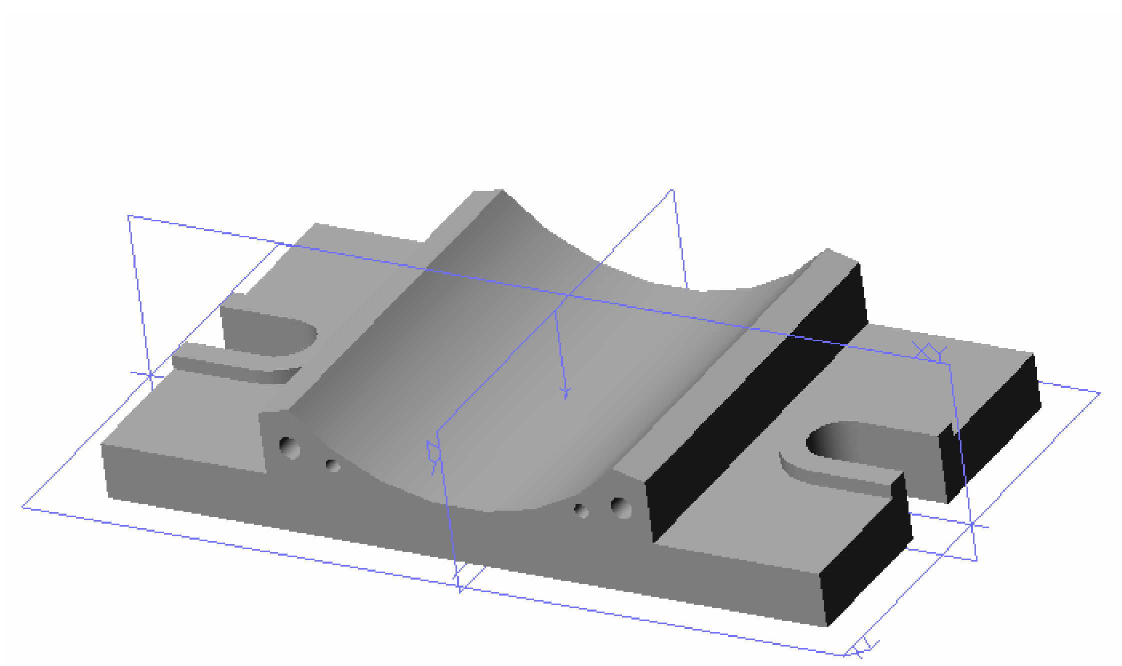


And, again to create the holes on the opposite side of the boss.

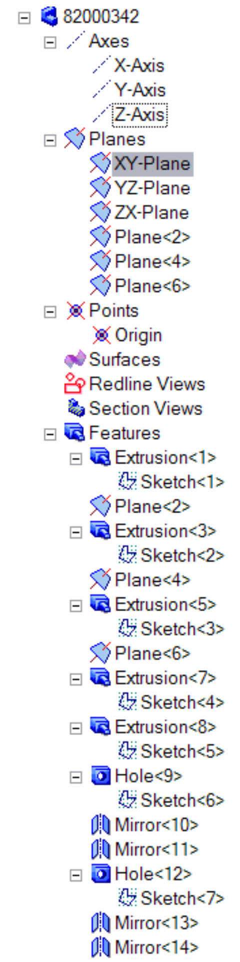


The ‘Mirror’ feature function should be used when the features in question will remain the same in relation to one another, i.e. hole diameter, counter bore, etc. Use this same methodology to create the remaining holes on the part.

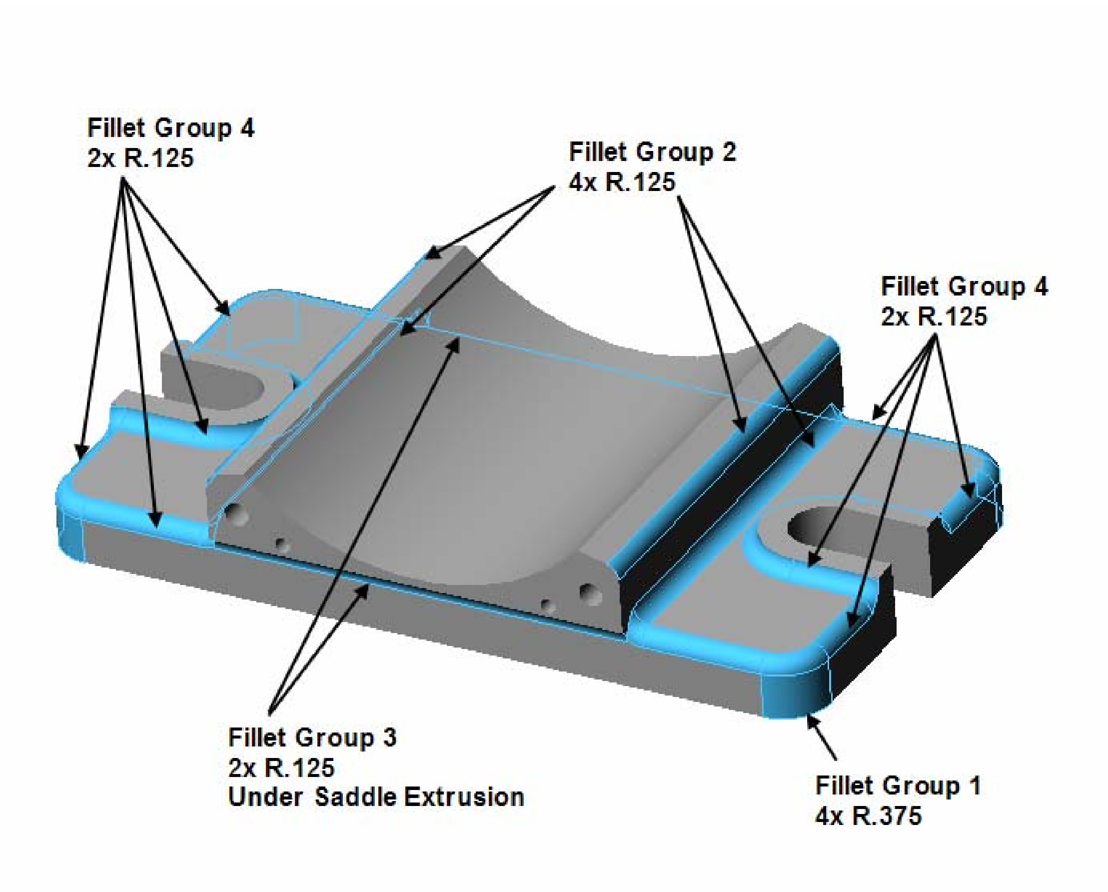
Your part should now look like this.



The design history should look like this when the operations are complete.



Fillets and chamfers can be the most frustrating dress-up features to add to a part. Usually this is because little thought is given to the order in which these features are added to the part. They should be added to the design as late in the process as possible in order to avoid problems with future edits. They should also be grouped whenever possible, in order to reduce the complexity of the design history tree (See Section 3.x.x.x). The fillets being added to the Any Angle Tool Vise Saddle part are simple fillets that lend themselves to such grouping. The picture below shows the way the fillets can be grouped and the order in which they should be applied.

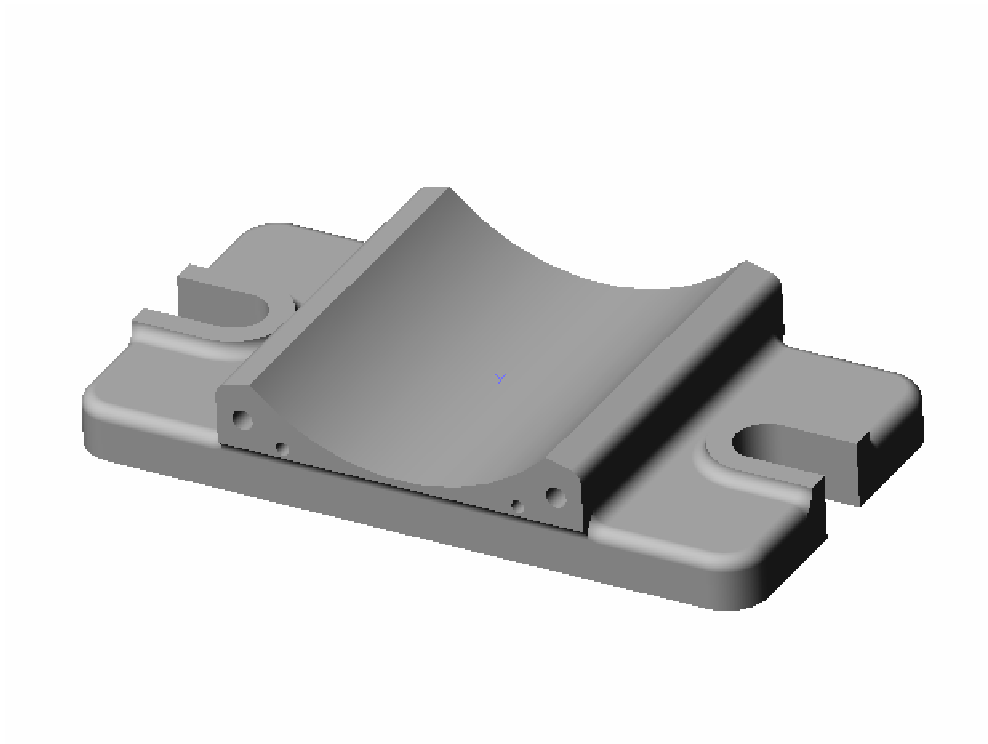


Fillet Group 1 is added first for two reasons; the fillets do not touch and they aid in the propagation of fillet group 4. The same can be said for Fillet Group 2. Fillet group 3 (front and rear of the saddle also aids in the propagation of Group 4. Fillet Group 4 is the last to be added.

After adding the fillets to the ‘Saddle’ save the part as 82000342. You’re ready to move on to the next part in the Any Angle Tool Vise Assembly.

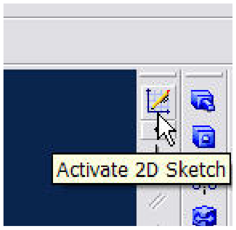
In creating the Any Angle Tool Vise Saddle it should be noted that the part is made from cast iron. This implies a casting operation, and a multi-staged machining operation. Whether the casting operation is rough, i.e. sand casting, or precise, i.e. an investment casting or the part is machined from a cast iron block will make some difference in the part cost, but one thing is certain; this will be an expensive part to produce. Design work always involves trade off’s between cost, feature content, weight, strength, manufacturability, and durability. As you go through each of the design modules for the Any Angle Tool Vise, look at each with an eye towards optimizing each in relation to the listed parameters. Also think about the possibility of utilizing off the shelf parts in place of some of the manufactured parts shown (the locking handles for instance).

Your part should now look like this. File it as 82000342.

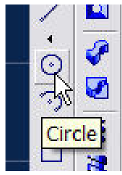


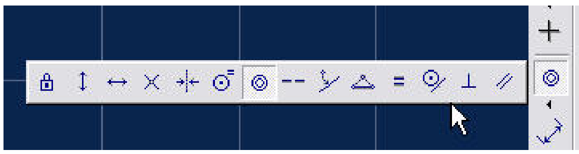
Chapter 2 -Designing the Any Angle Tool Vise – The Clamp Plug

In this section of the design module, we'll use the same Workspaces and command sets we used in the creation of the Saddle, with the addition of the Chamfer dress-up feature. Refer to Drawing 2 Clamp Plug, for all construction dimensions.

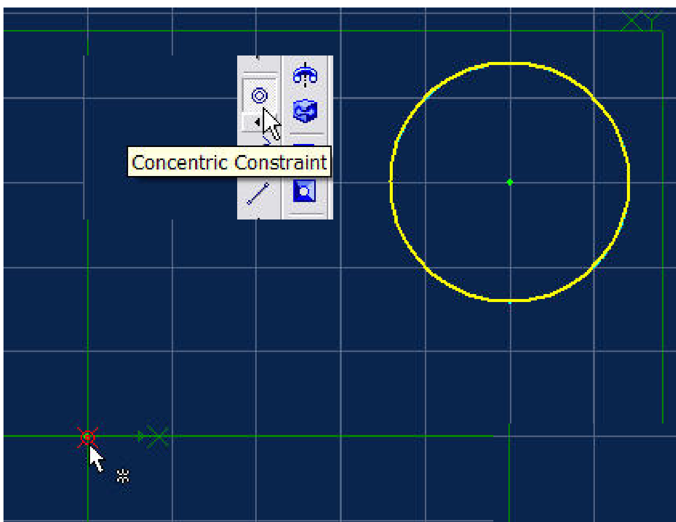
The first step is to extrude a simple .312 dia. cylinder using the XY plane as our geometric anchor. Click on the XY plane in either the graphics window or the Design History panel, and then click the 'Sketch'  icon.

Next, click on the 'Circle' icon and then click anywhere in the graphics window and drag a circle to any diameter.

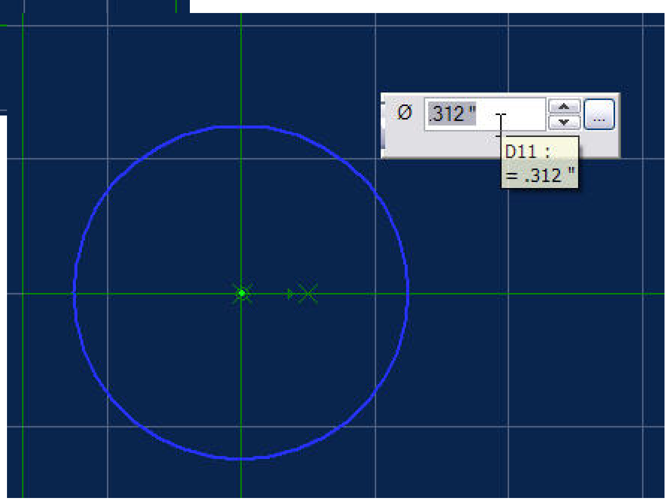




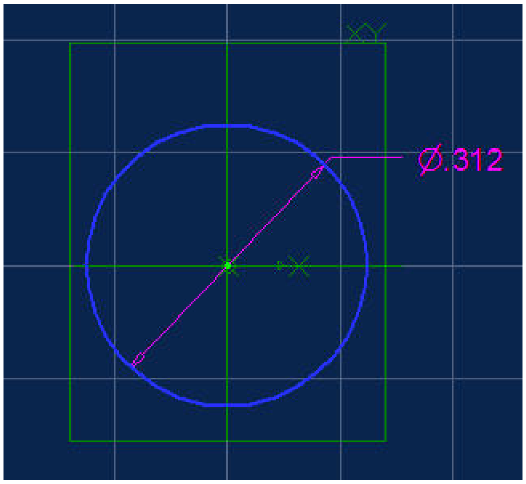
Now click on the 'Constraint' selection arrow and when the panel opens, select the 'Concentric' constraint icon.



Click on the 'Concentric' constraint icon, click on the circle you created, press the 'Shift' key, and click on the origin point (X0Y0).



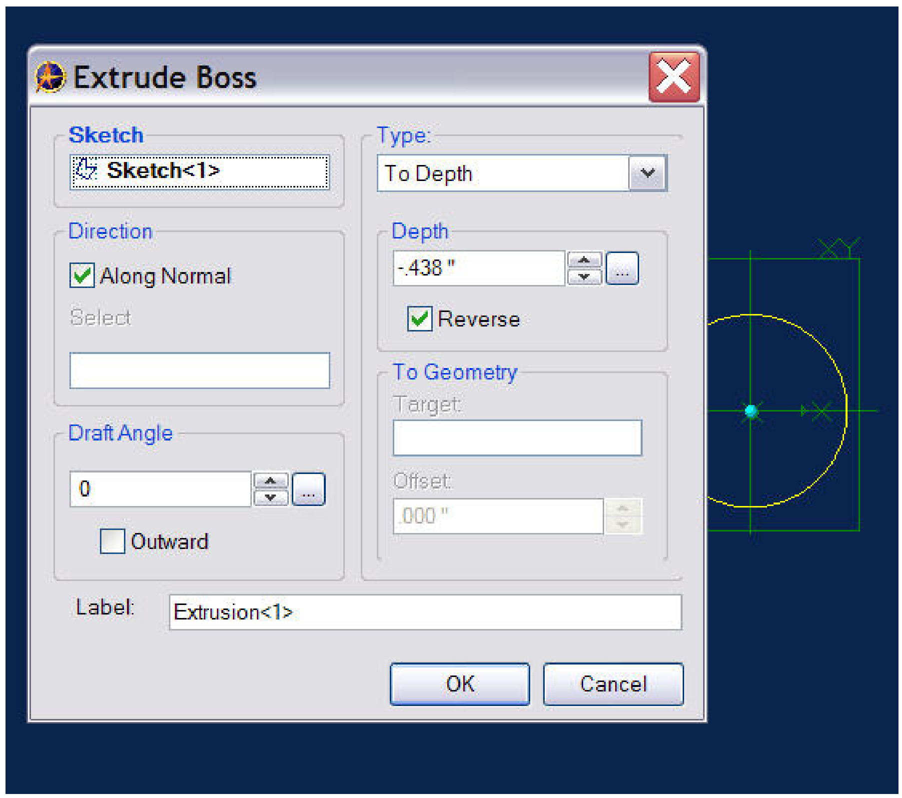
Click on the 'Dimension' icon, click on the circle, and enter .312 in the 'Text' box, and press the 'Enter' key.

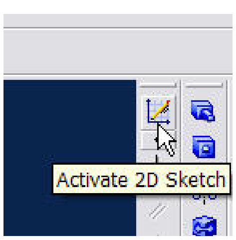


Your sketch should now look like this.

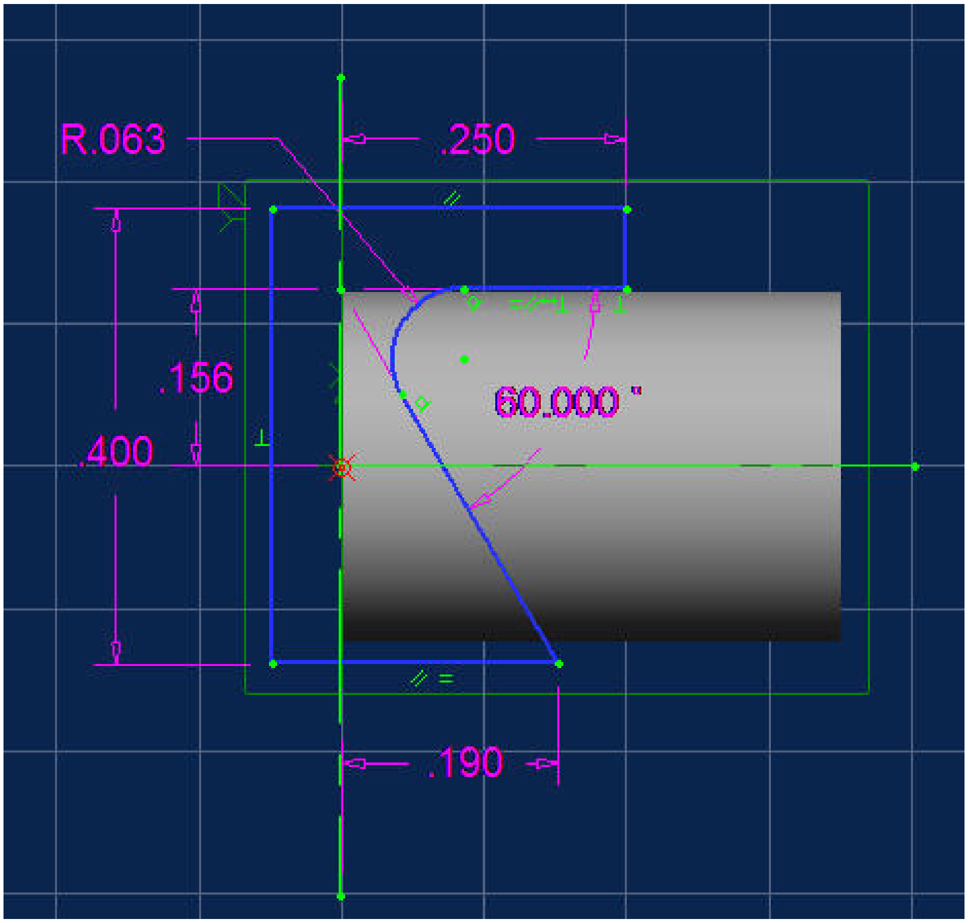
Click on the ‘Extrude Boss’ icon.

When the Extrude Boss’ panel opens, fill in the information shown and click OK.

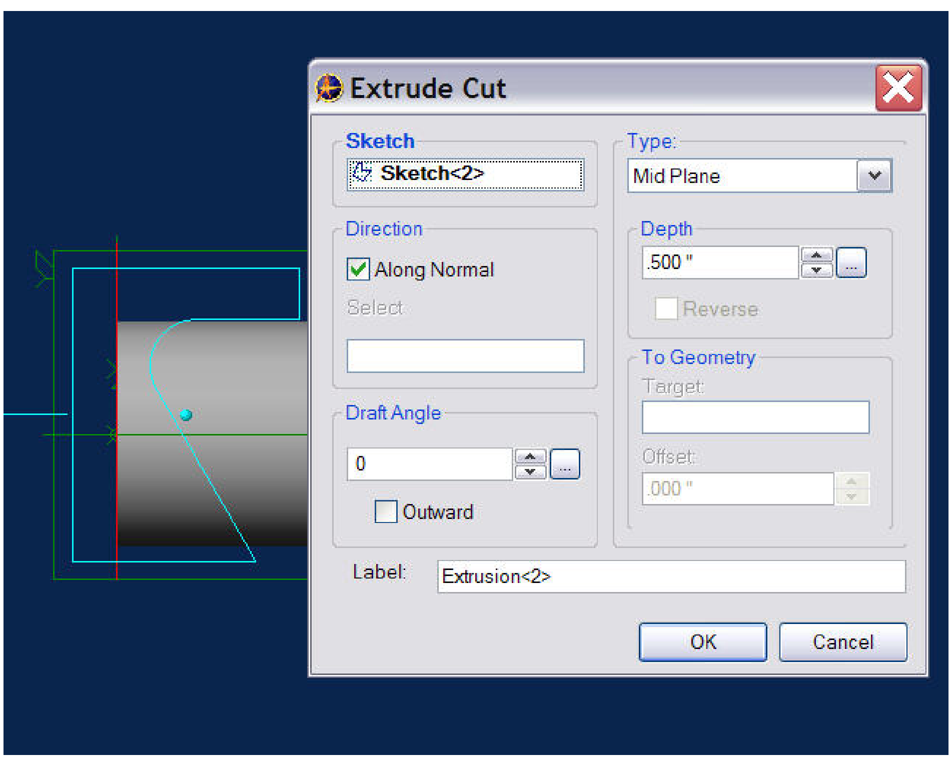




Click on the YZ plane in either the graphics window or the Design History panel, and then click the ‘Sketch’ icon.

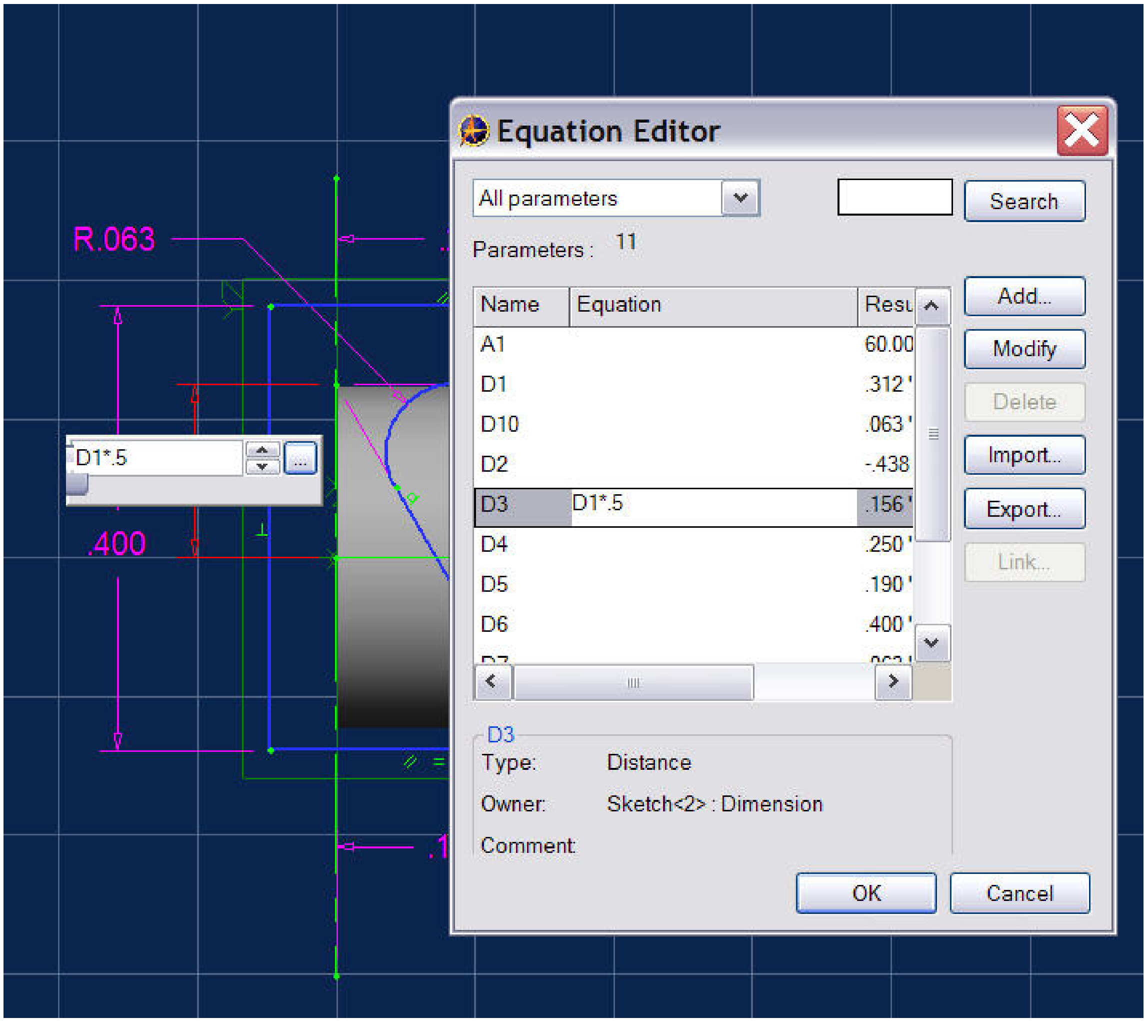


Create the sketch shown. The dimensions are only critical in the sense that the sketch profile should lie outside of the cylinder profile.



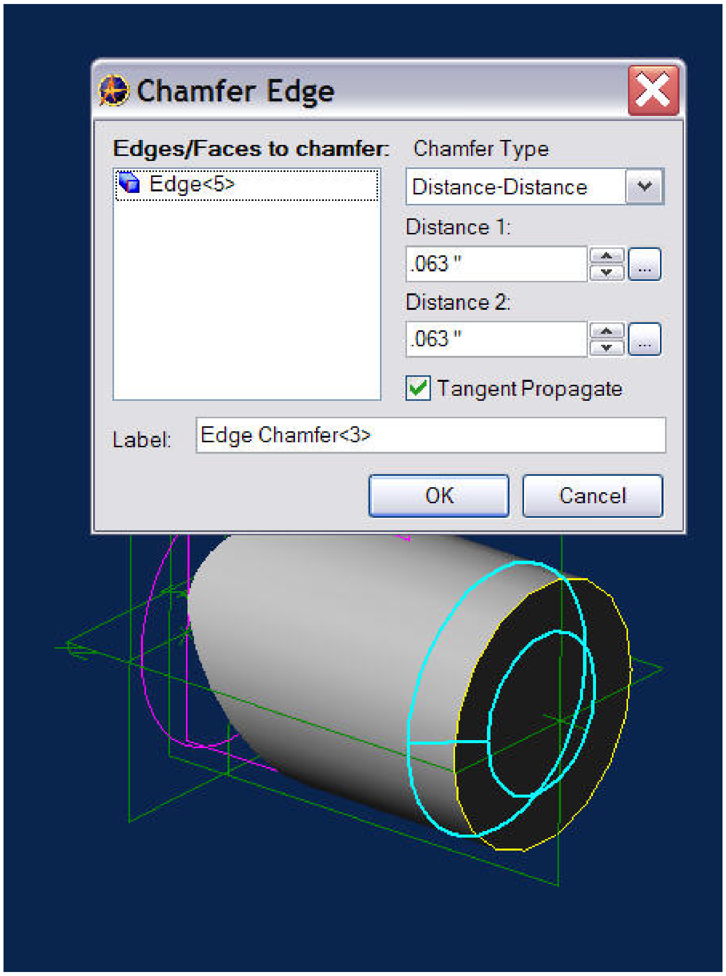
Use the Extrude Cut > Midline, and select .500” for the depth. The part should now look like this.

Another way to insure that the sketch .156 dimension) is tangent to the top of the cylinder is to open the dimension formula panel by clicking on the three ellipses button to the right of the dimension text field and tying the .156 dimension to half the diameter of the cylinder, D1*.5 =.156.

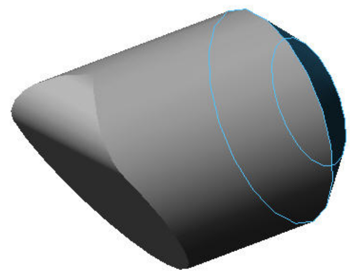


Using the equation editor to control the shape and size of your parts is a powerful Alibre design feature you will become more familiar with in a later section of this training manual. In Chapter XX, we look at how we can tie part parameters to an Excel spreadsheet and drive our designs and drawings through these links.

Add the chamfer to the end of the cylinder as shown, to complete the Clamp Plug part.



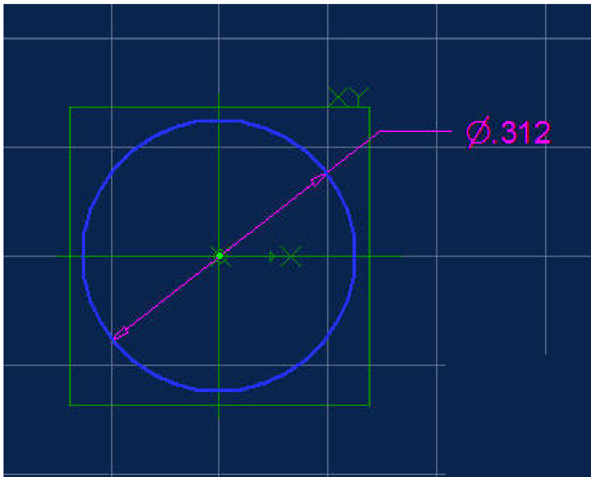
The finished Clamp Plug should look like this.



File this as Part Number 82000343.

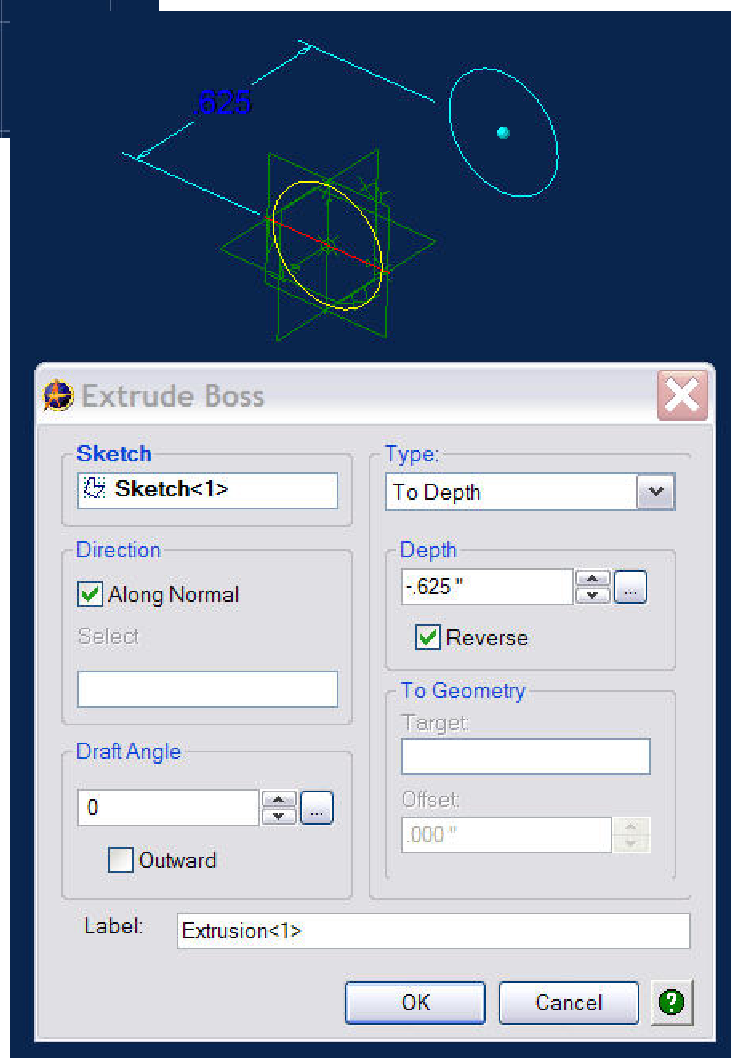
Designing the Any Angle Tool Vise – The Eccentric

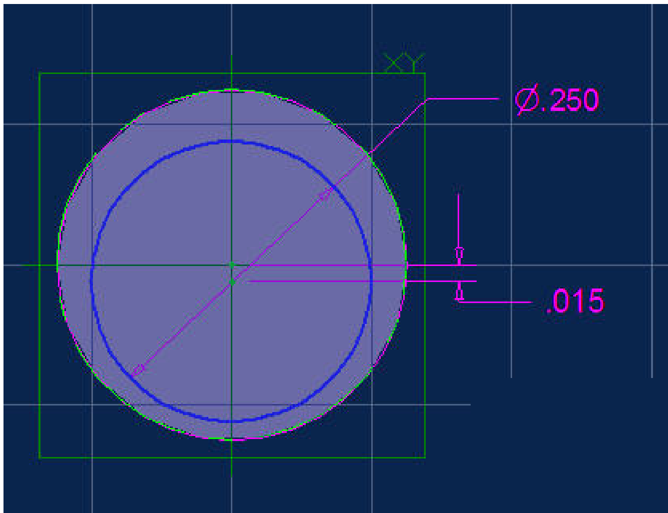
The first step in creating the Eccentric, is to extrude two simple cylinders using the XY plane as our geometric anchor and extruding in opposite directions from what could be called the mid-plane of the part. Create the sketch shown below keeping in mind the Best Practices rule for constraining circles.



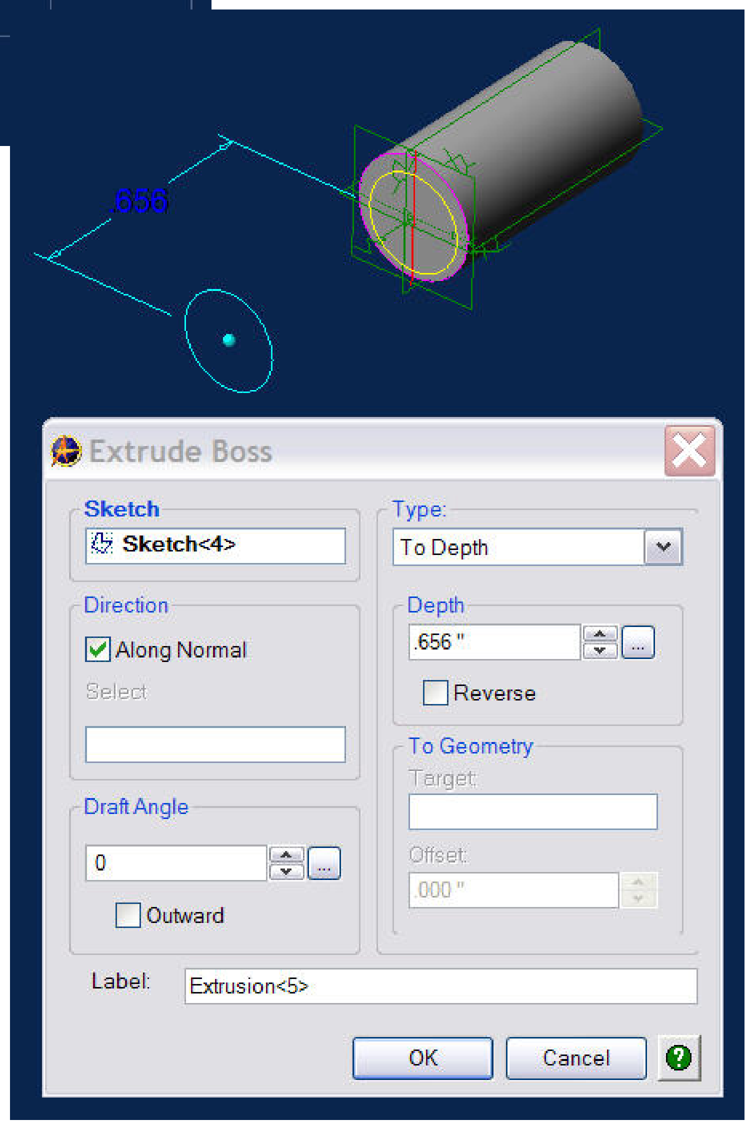
Click on the 'Extrude Boss' icon and

Extrude the circle .625" deep as shown.

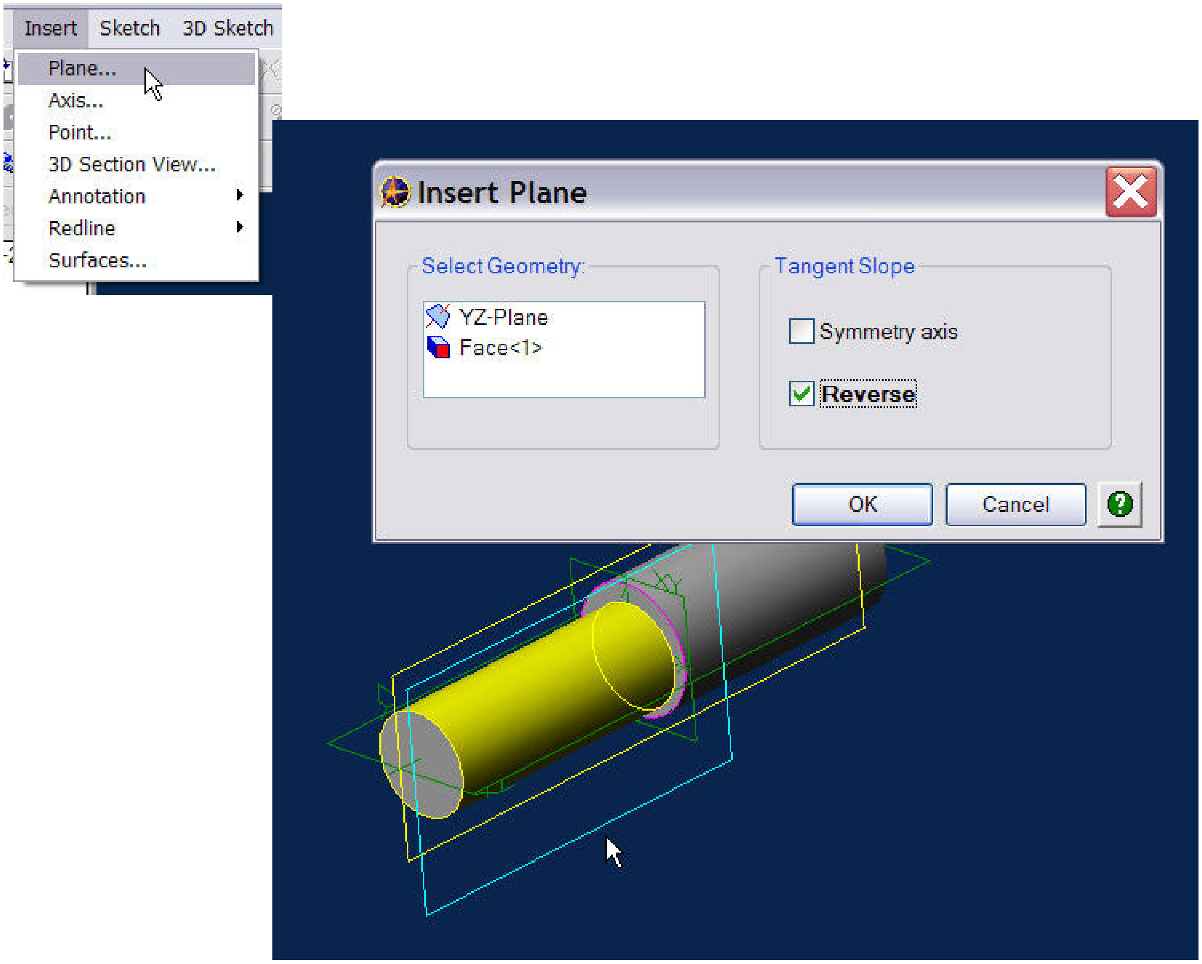




Now sketch a second circle as shown, using the XY plane as the anchor for the sketch and then extrude it .656 in. in the opposite direction of the first sketch.

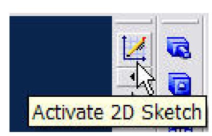


The next step is to create a simple hole through the smaller shaft. To do this we'll insert a plane tangent to the cylindrical surface of the smaller diameter shaft by using the Insert> Plane command and selecting the YZ plane as a reference and then using Shift > Click to select the face of the smaller cylinder. If necessary, click the Reverse checkbox to orient the plane as shown.



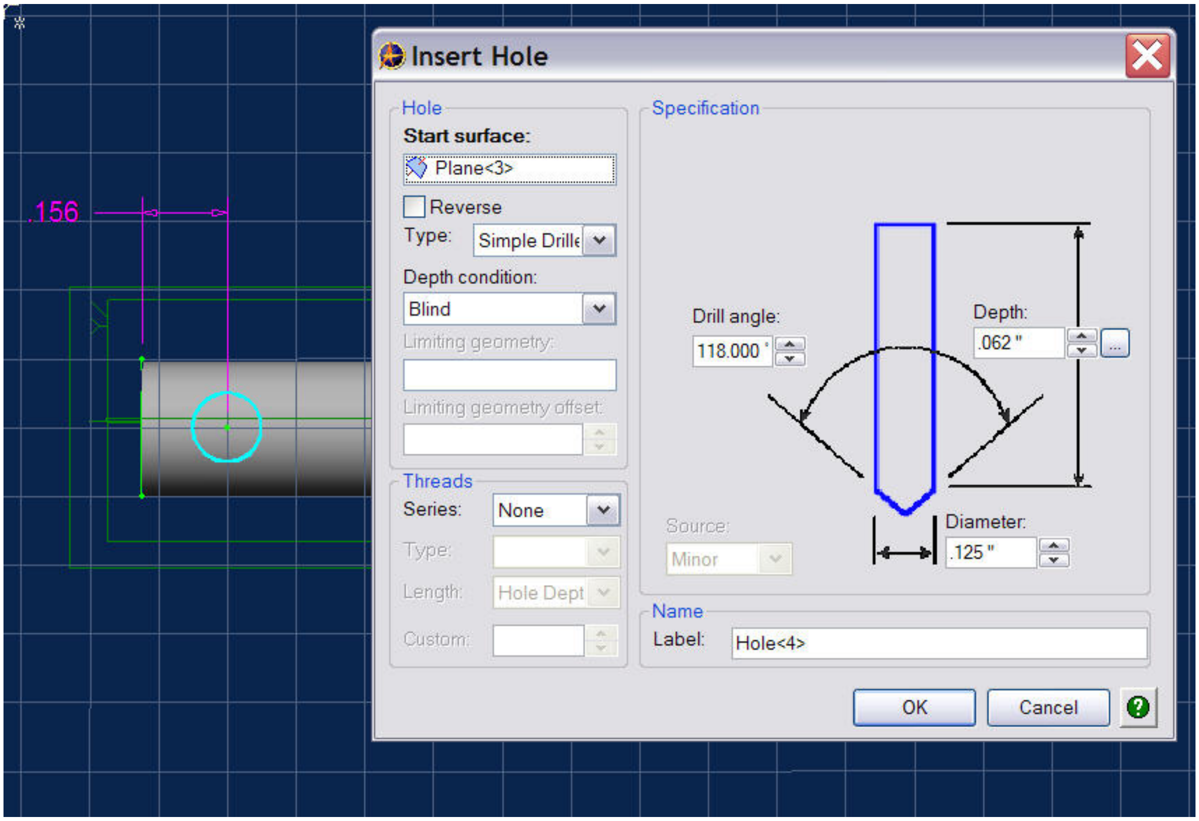
Alternately, you could use the YZ plane for the geometric anchor and simply sketch a .125 in diameter circle and extrude a cut using the Midplane modifier, but in this example we want to control the hole's parameters as a feature and not as a sketch entity.

Select the newly created plane and click the sketcher icon to enter sketch mode.

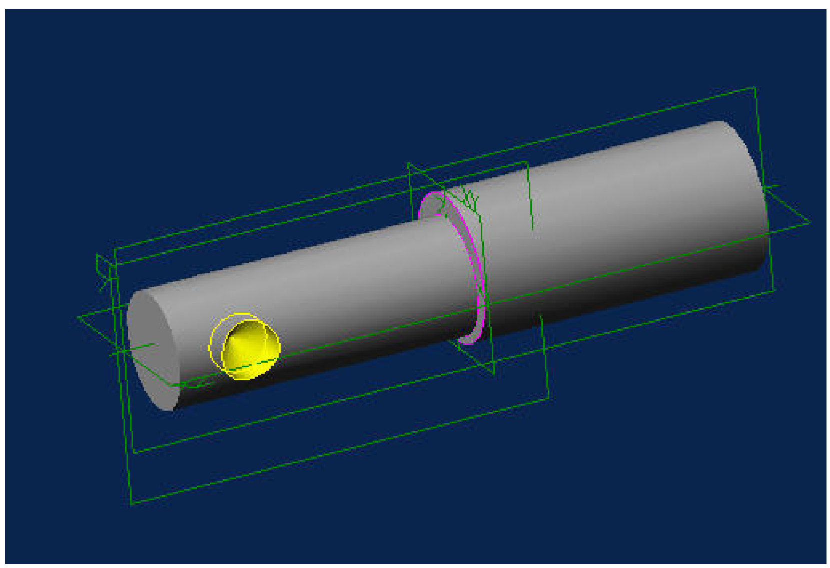


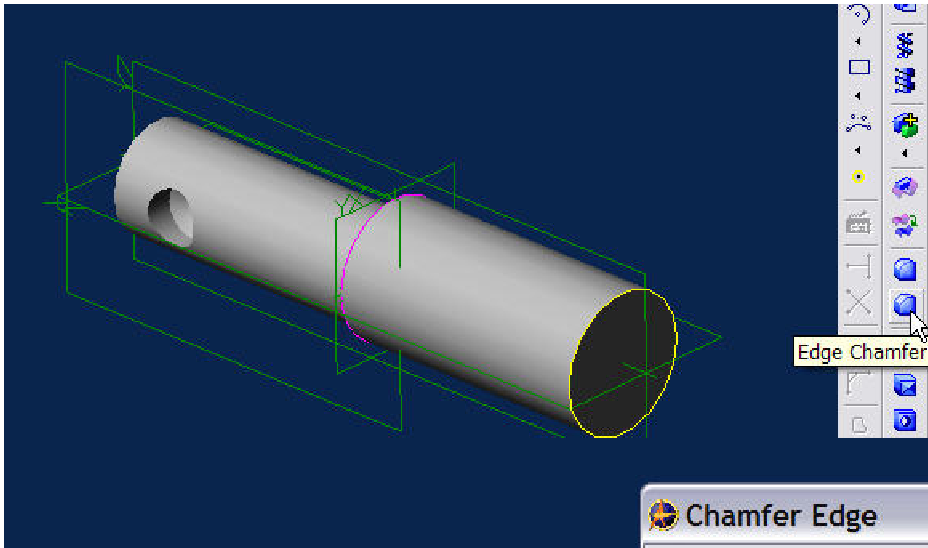
Click on the Hole icon 

to activate the Hole feature panel. Define and constrain the hole as shown on the next page.



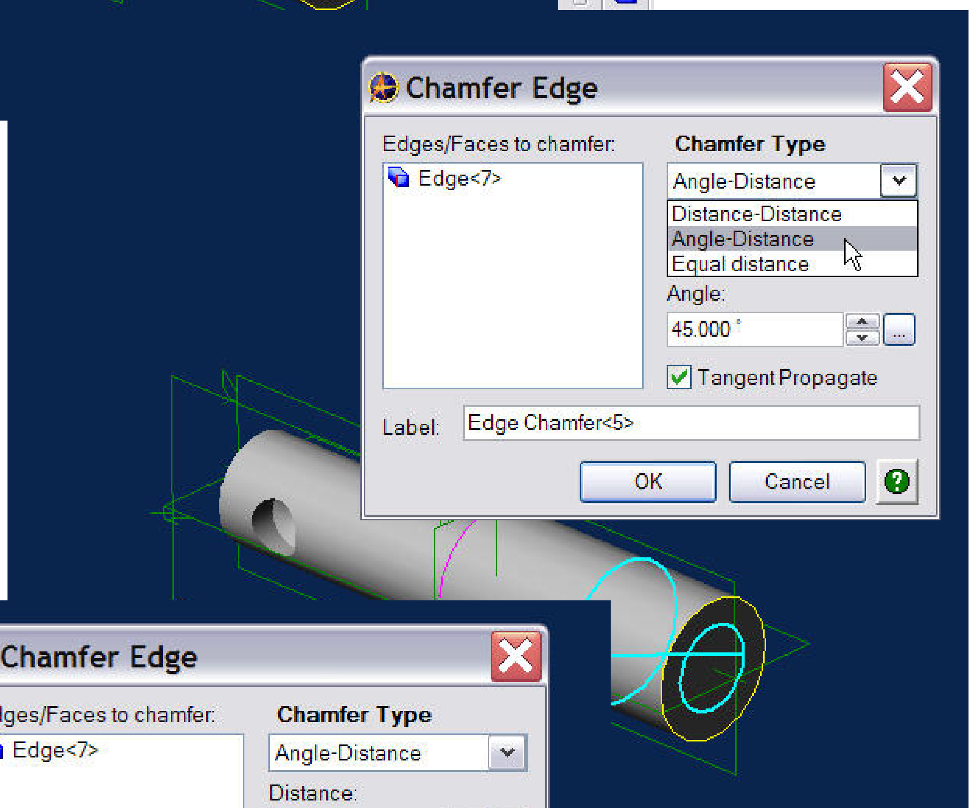
When finished your part should look like this.

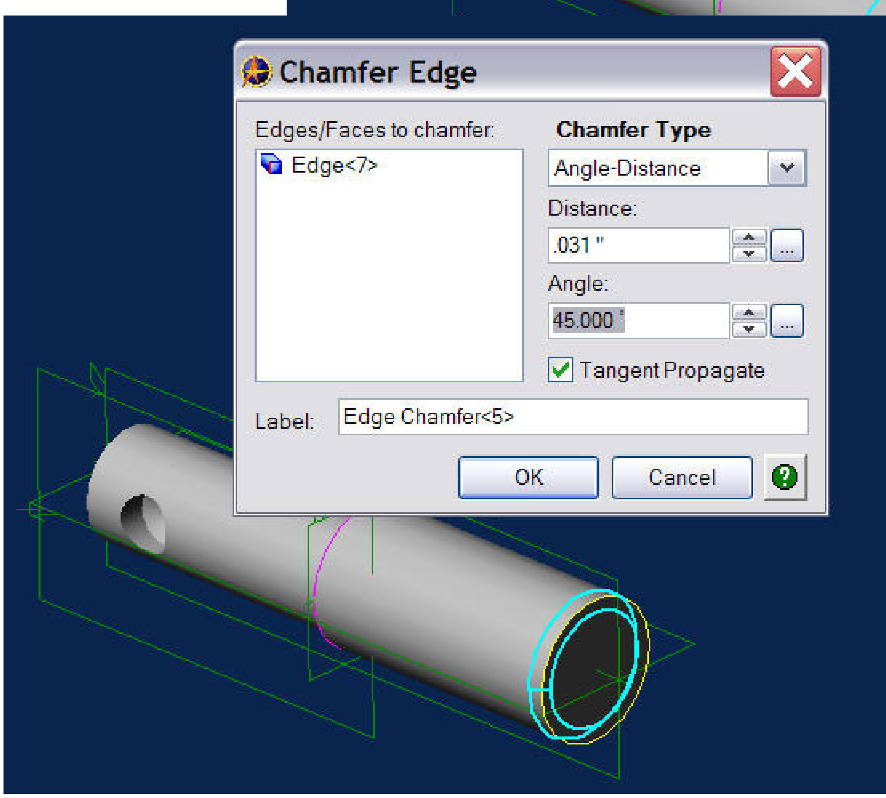




Next, chamfer the end of the large cylinder. Select the edge you want to apply the chamfer to and click on the Edge Chamfer icon.

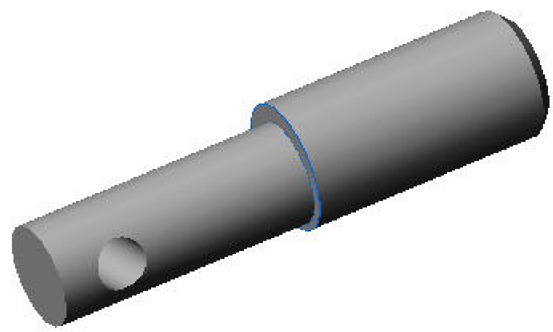
When the Chamfer Edge panel open, select Angle – Distance for the Chamfer Type.





Enter .031 in. for the distance value, and 45° for the Angle value, then click OK.

The finished Eccentric should look like the figure below.

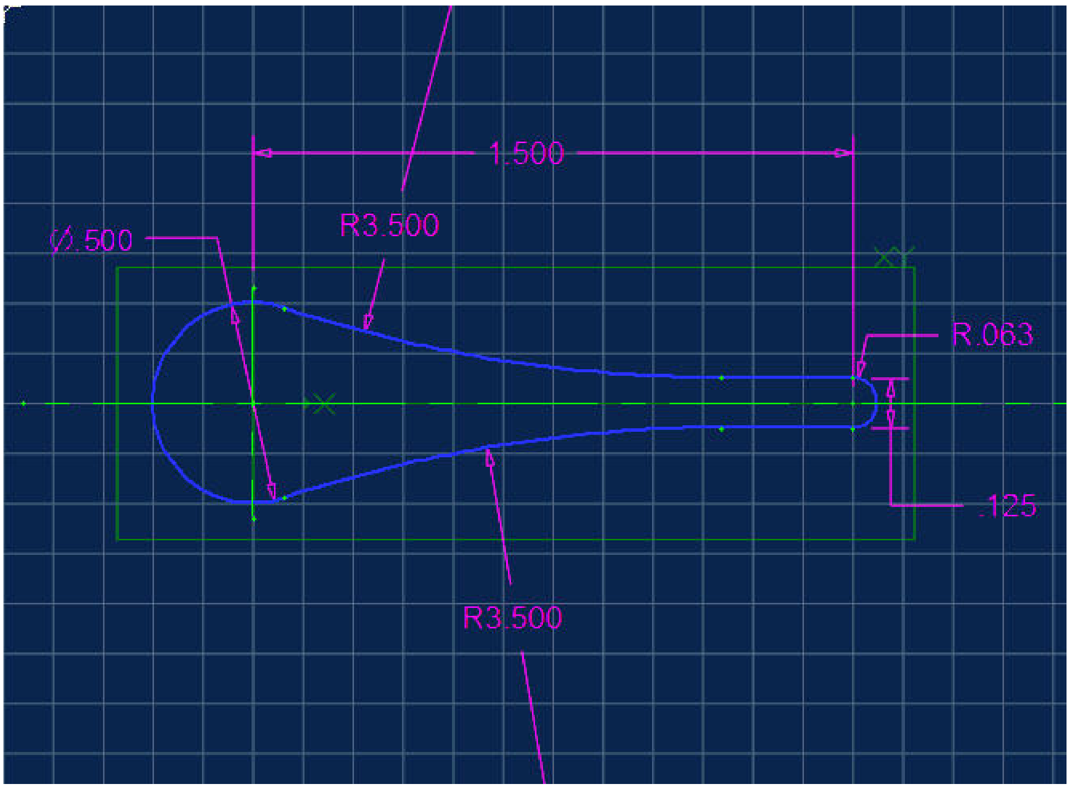


File this as Part Number 82000345.

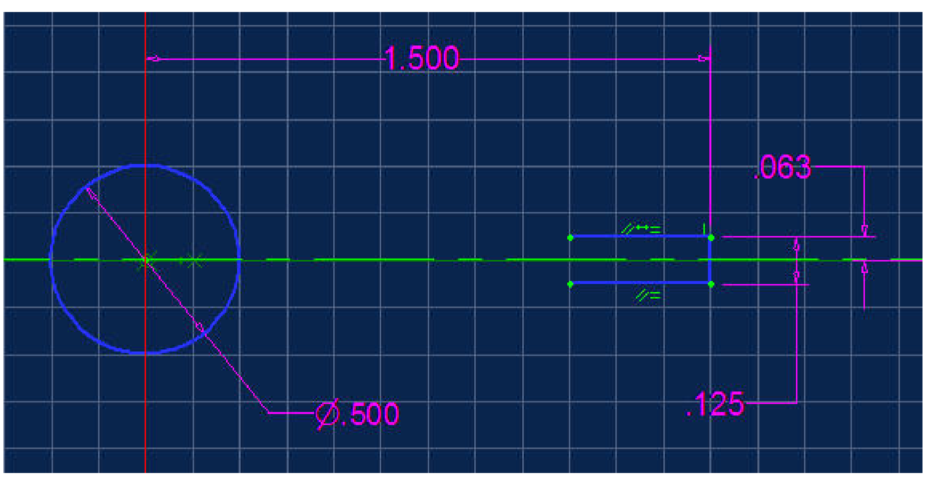
Designing the Any Angle Tool Vise – The Locking Handle

In this section of the design module we'll use the same Workspaces and command sets we used in the creation of the Saddle and Clamp Plug, with the addition of the use of the tangent constraint command to create the Locking Handle. Refer to the locking Handle drawing for all construction dimensions.

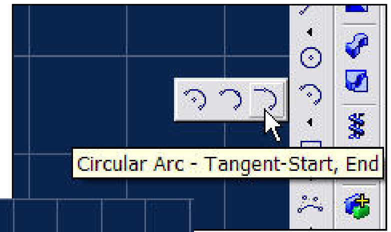
The first step is to create the sketch shown below.

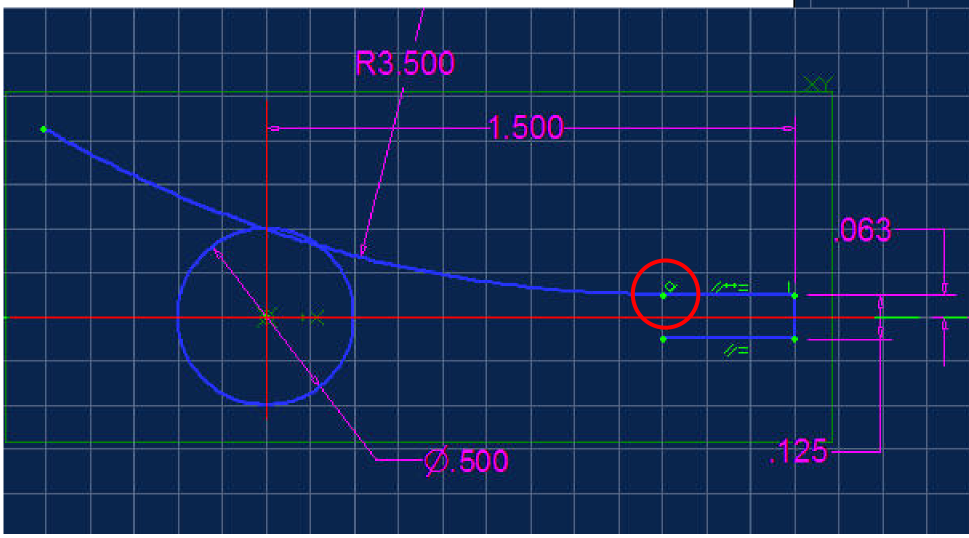


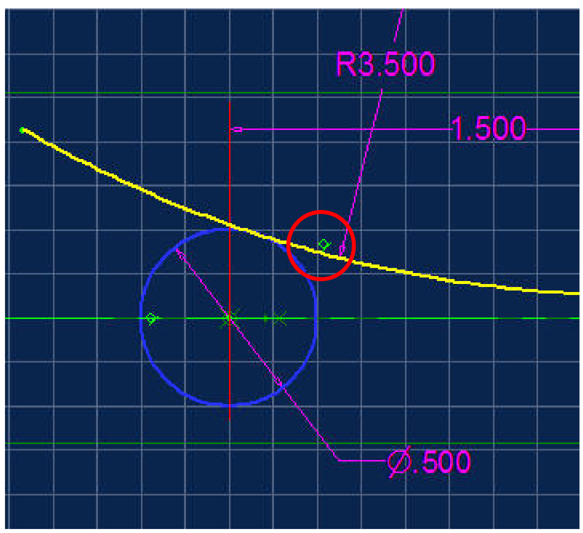
To begin, sketch (ø.500) circle and group of three lines, and constrain them per the dimensions shown.

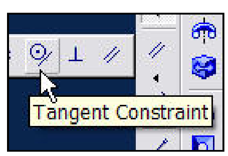


Sketch in the large arc shown below using the Tangent,-Start, End command. Dimension it as R3.500. Start the arc tangent to the horizontal line at the end of the line.



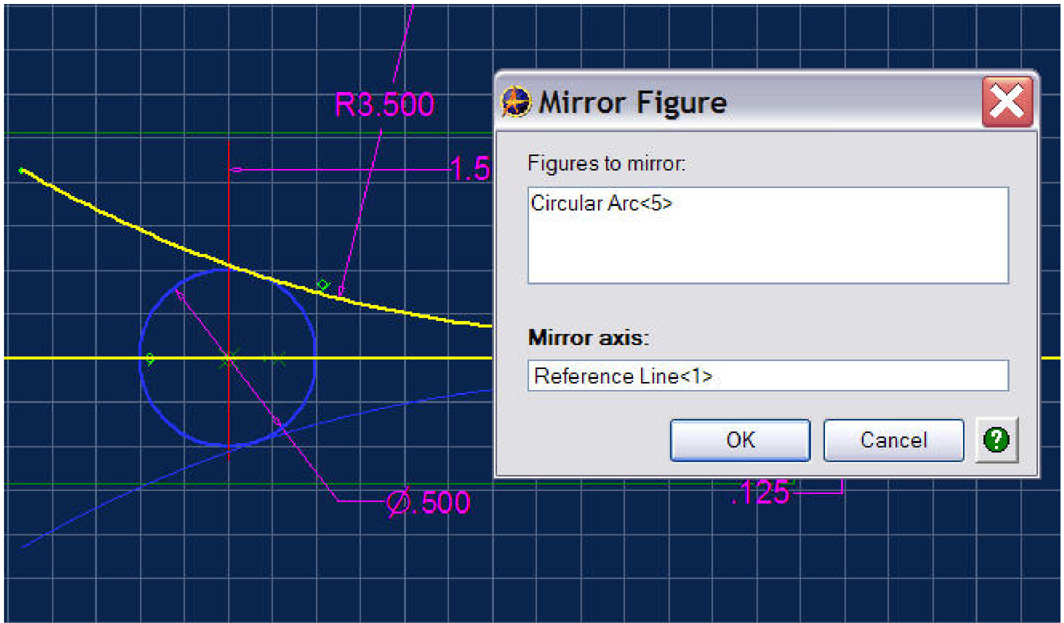


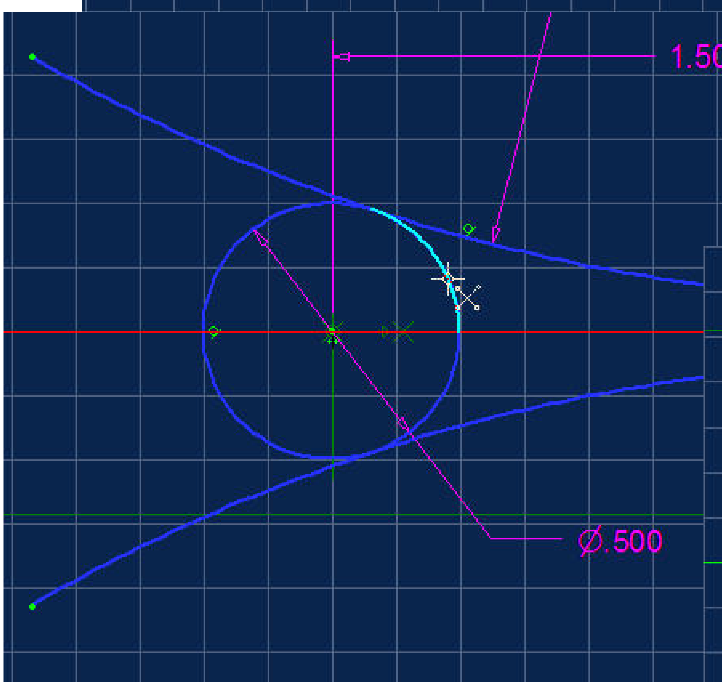


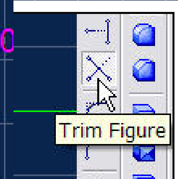
Now, apply a Tangent constraint 

between the large circle and the R3.500 arc as shown above.

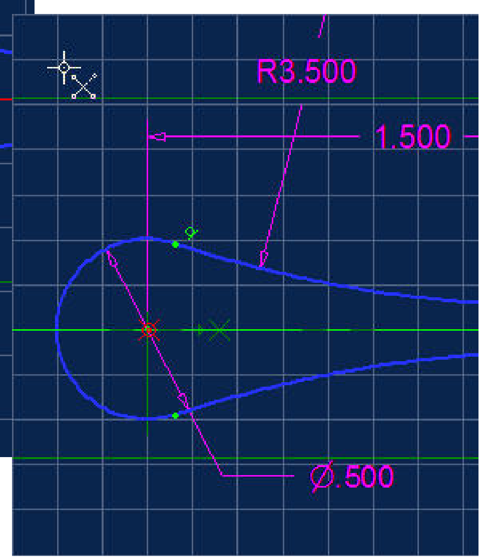
Use the Mirror element  command to copy the arc to the opposite side of the horizontal axis as shown below.

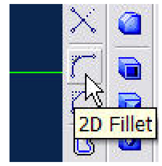




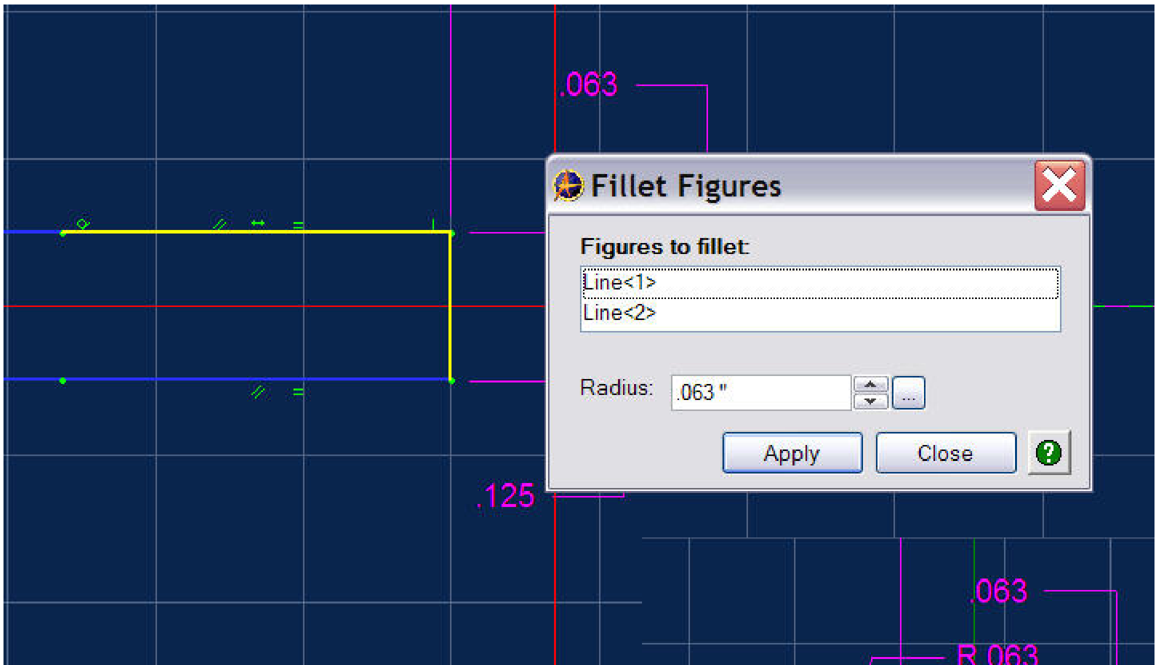


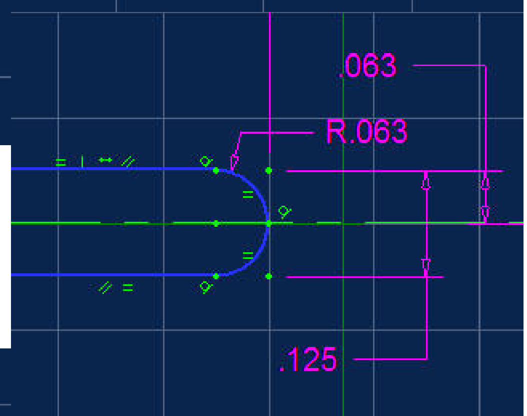
Use the Trim command to trim the arcs and circle to look like the sketch below



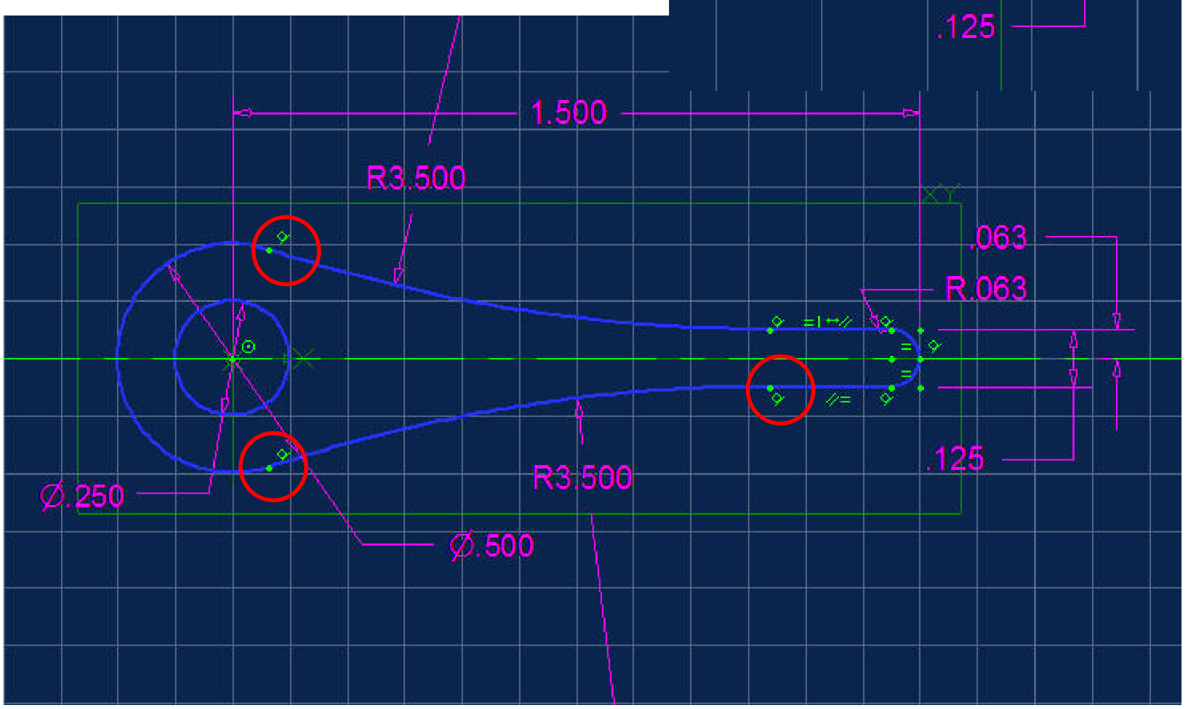
To complete the sketch. We'll use the 2D Fillet command  to add the radial elements to the end of the handle. Alternately, we could add these as 3D fillet features in the extruded model.

Select the elements to fillet, enter .0625" (.063" will appear in the Radius window), and click OK. Repeat for the other side of the handle.

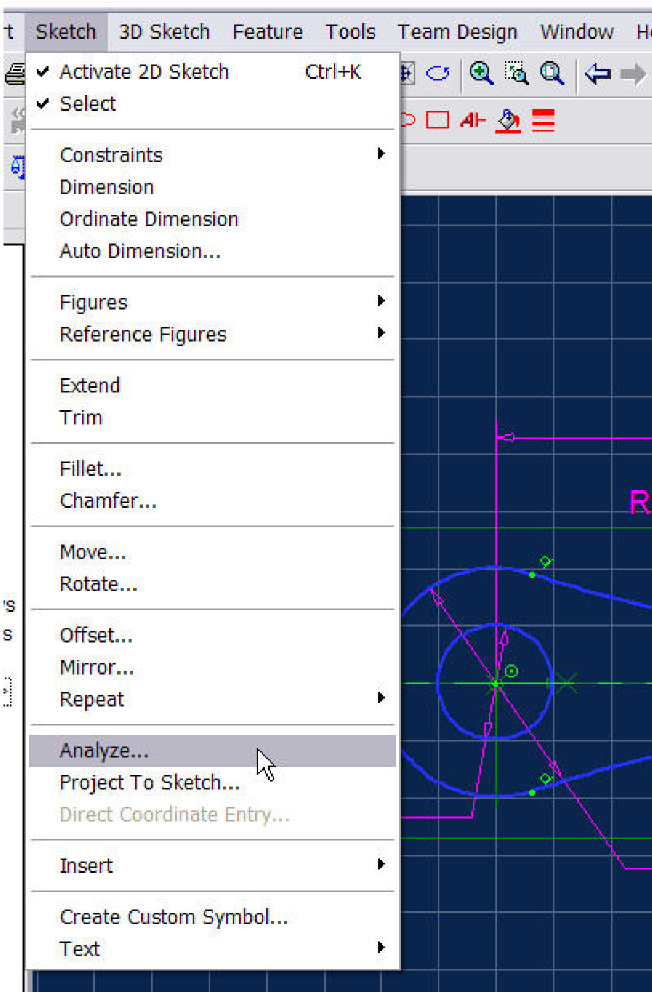


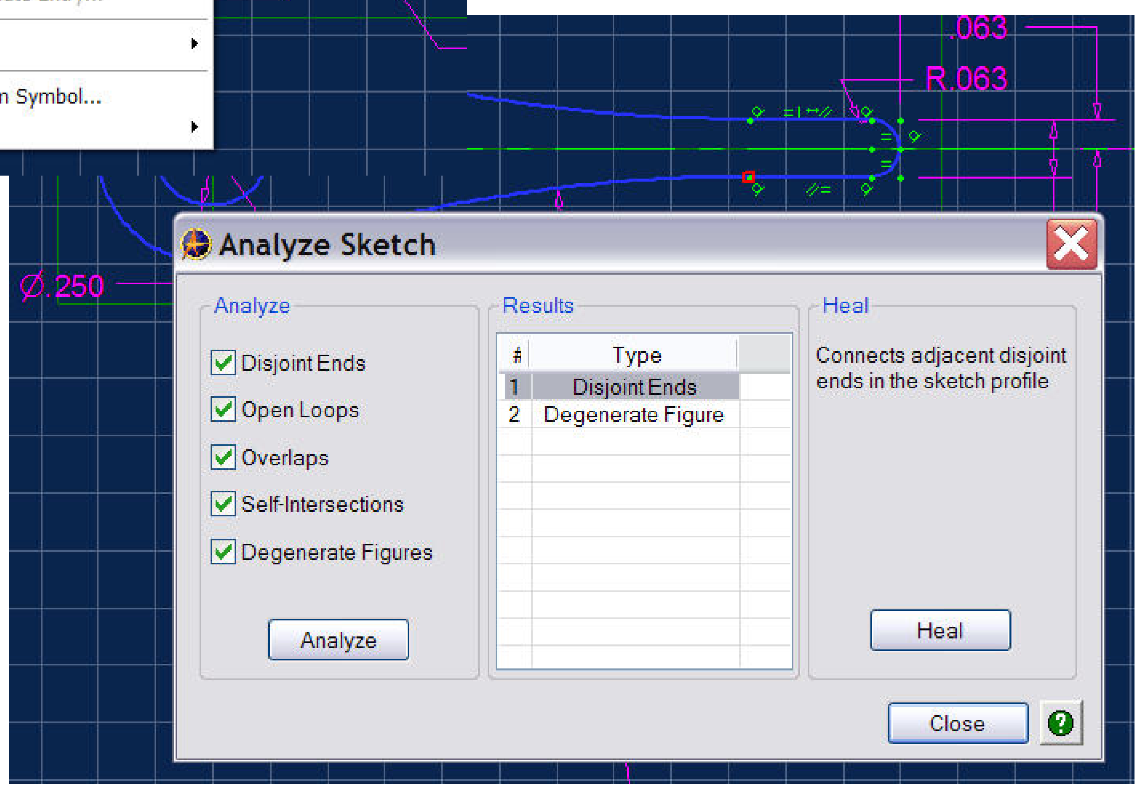


Add the ø.250" circle and tangent constraints highlighted below. The through hole in the handle could also be added as a 'Hole' feature in the extruded model.

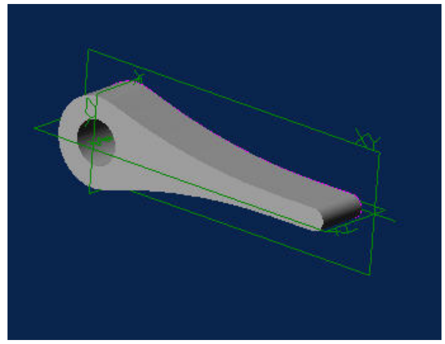


Analyze the sketch by selecting ‘Analyze’ from the Sketch menu. Correct any potential problems by using the ‘Heal’ command or by modifying the sketch.



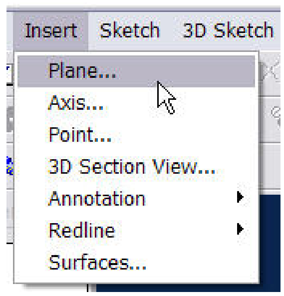


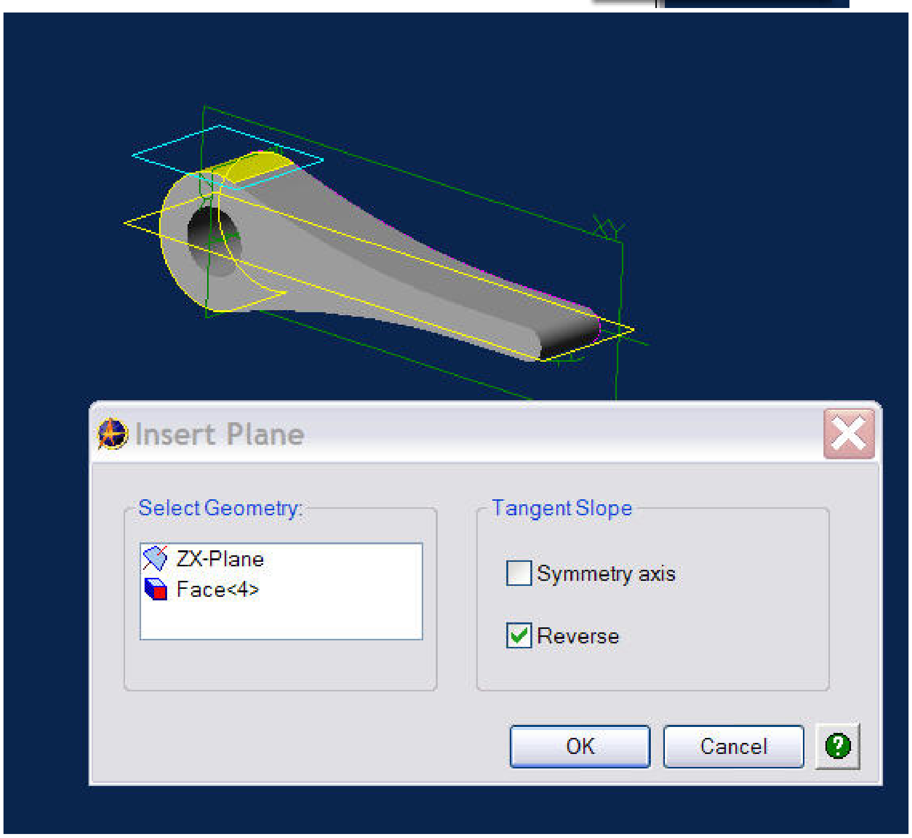
Extrude the sketch to a depth of .3125”. Your part should now look like this.



Now it’s time to add the locking pin hole. To create the locking hole we first need to create a plane tangent to the top surface of the large end of the handle using the same technique we used to create the mating hole in the ‘Eccentric’.

Click on the Insert>Plane command.

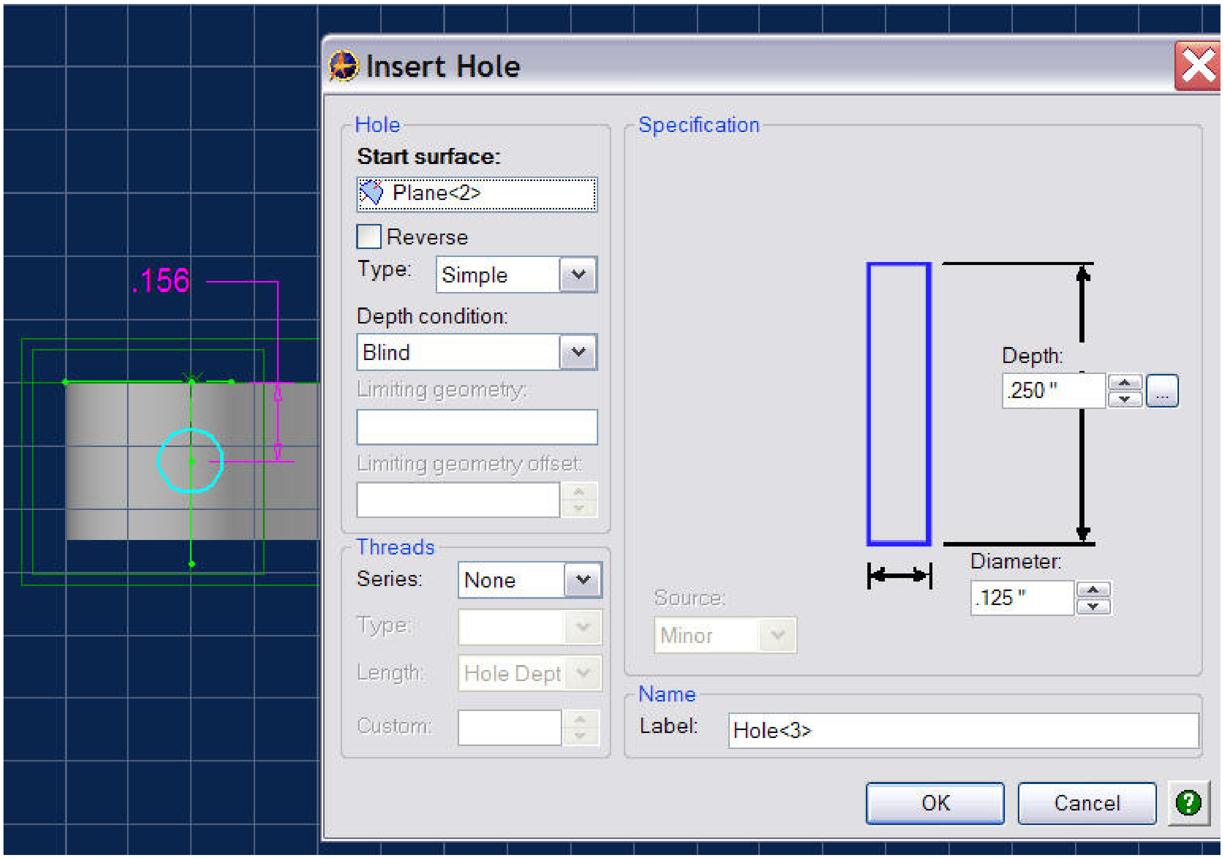


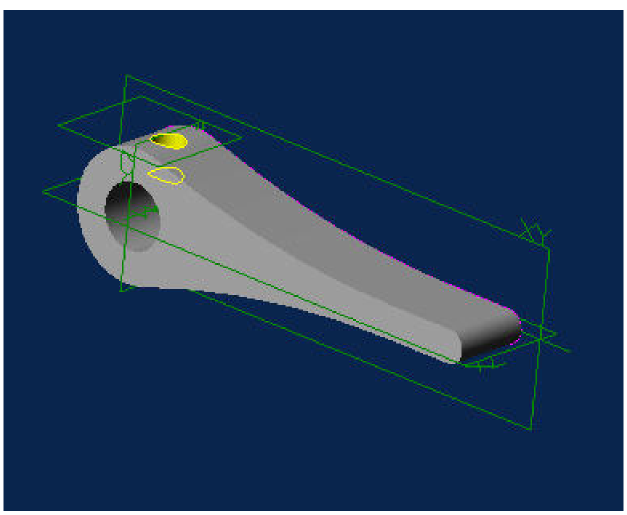


Select the ZX plane as the reference plane and then press the ‘Shift’ key and click on the top surface of the large end of the handle. Use the ‘Reverse’ selection box if necessary to position the plane on the top of the handle.

Next select the newly inserted plane and click the ‘Hole’ icon to return to ‘Sketch’ mode and open up the Hole dialog box.

Click on the handle. A circle appears. Adjust the 'Hole' parameters as shown and constrain the sketch using a dimensional constraint, .15625" from the face of the handle, and a Coincidence constraint (center of the circle to the Z axiz).

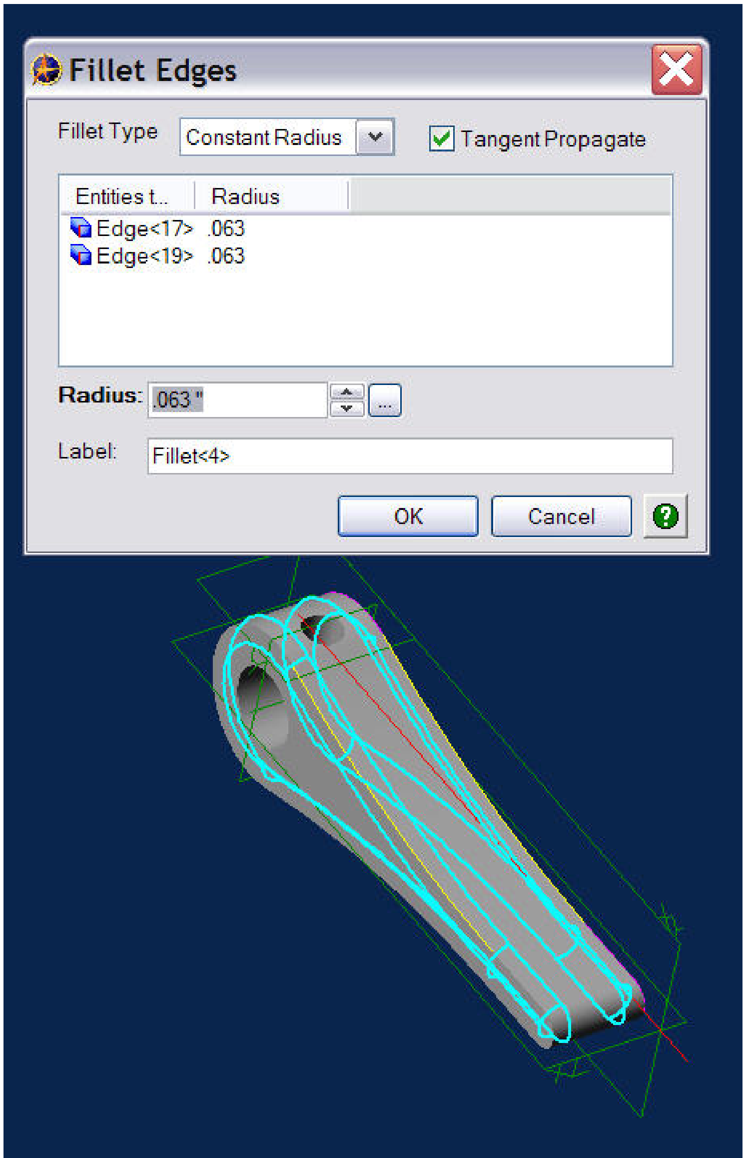


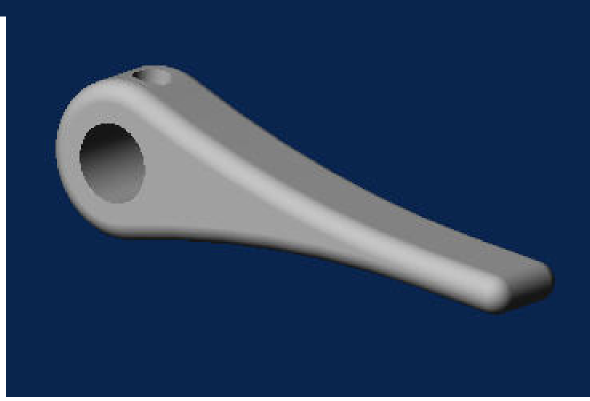


The part should now look like this.

The last step is to create the fillets. Click on the 3D Fillet command. 

Select an edge on both sides of the handle, change the default fillet radius to .0625 and click OK.

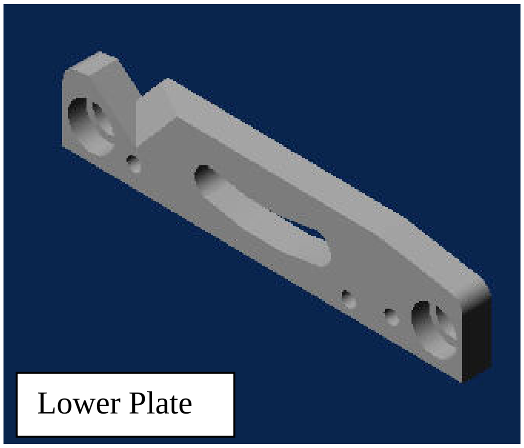




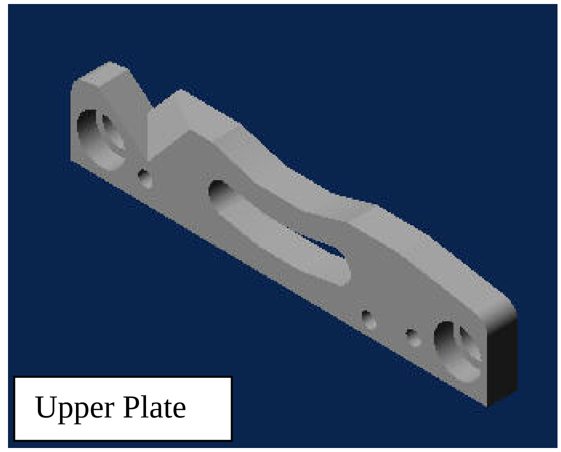
When finished the part should look like this. File it as part number 82000346

Designing the Any Angle Tool Vise – Upper and Lower Plates

Both the Upper and Lower Plates are made from a .25 in thick piece of bar stock, cut, notched, machined, surfaced, and drilled to the shape represented by the figures shown.



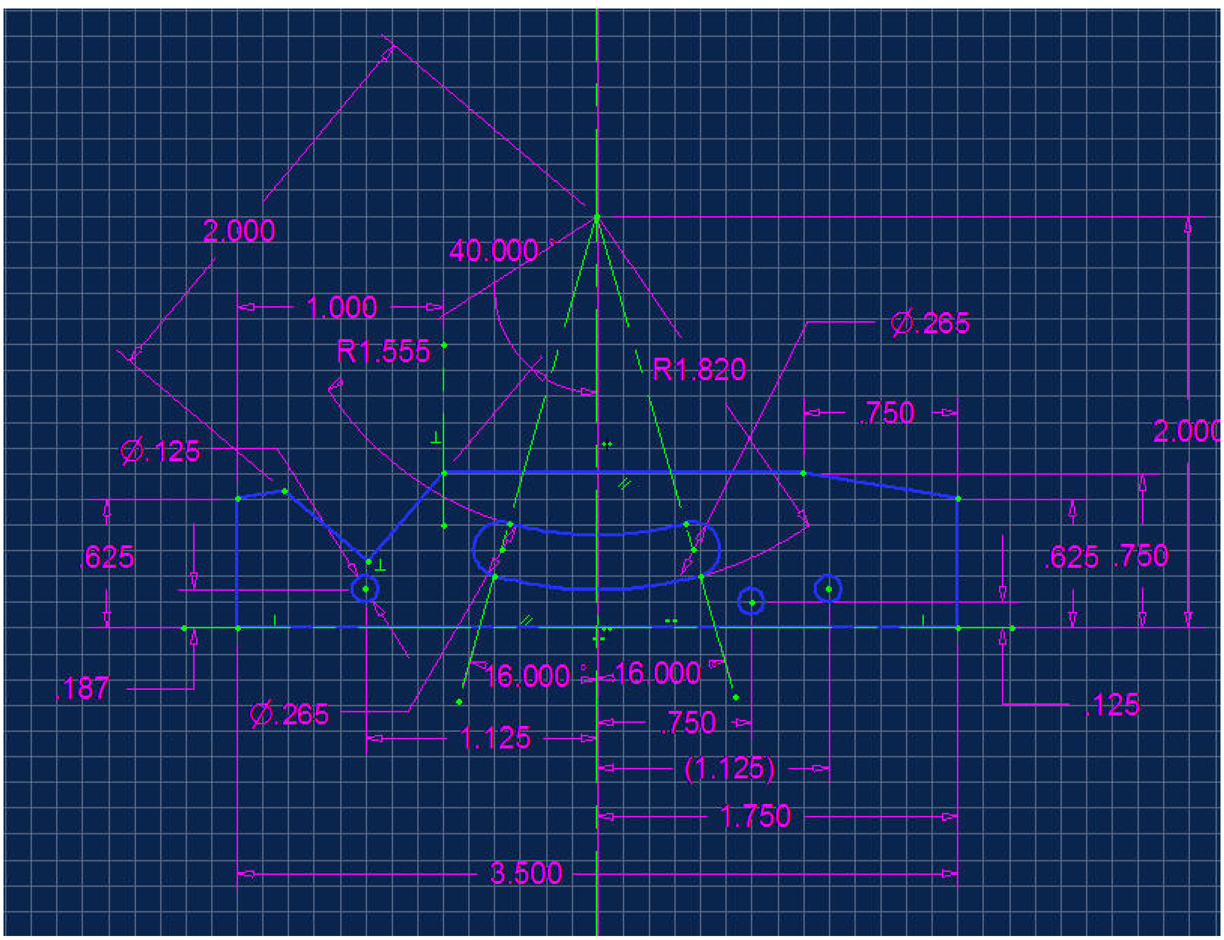
Lower Plate



Upper Plate

The difference between the two, consists of the radial cut out in the Upper Plate. We will model the Lower Plate and use a copy of it to create the Upper Plate.

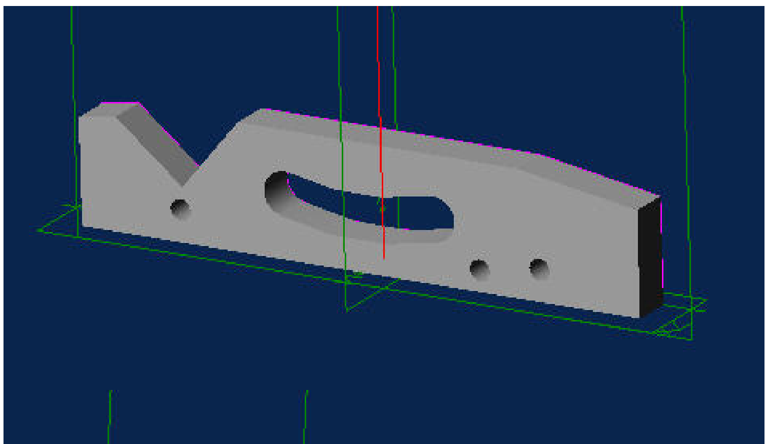
The first step in creating the Upper and Lower Plates, is to develop and dimension the sketch shown below. Refer to drawing XX for all required dimensions.



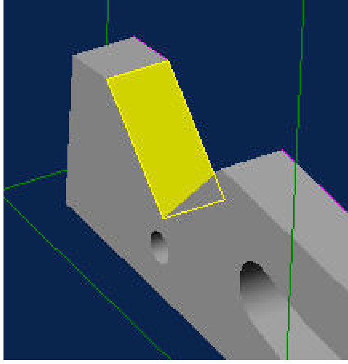
Although some dimensions may seem redundant or irrelevant, they are necessary to the complete stability of the extruded part. One quick method of reaching the zero DOF is to use the ‘Auto Dimension’ command under the “sketch’ tab after you’ve established and constrained the overall dimensional and geometric aspects of the part.

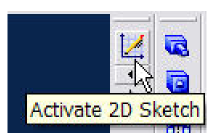
Caution is the keyword here. Save before you use the Auto Dimension command just in case you get more than you bargained for.

You’ll note in this case that I’ve elected to place the locking pin holes in the original sketch and not added as a dress-up feature later in the design. Per design intent, these holes are used solely as locating holes for drilling the receiver holes for the pins in the Saddle. Conceivably, their size would not change even if a resizing of the pin occurs, as they would be re-drilled at the time to fit. Creating them as a sketch feature does not compromise the design, or affect our ‘Best Practice’ rules in any but the most minor way. You are free to create them as dress-up features if you wish.

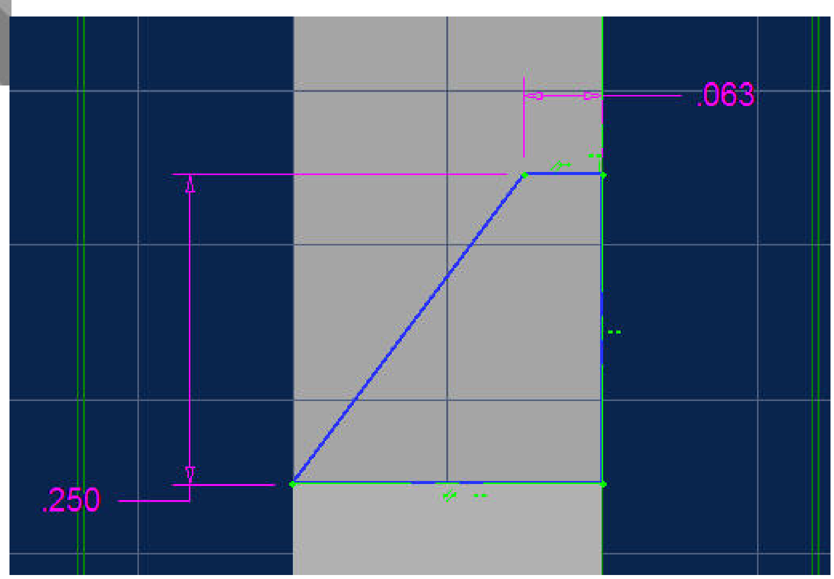


After the sketch is complete, analyze it, and then extrude it .25 in to form the plate. Your part should look like the one on the left.

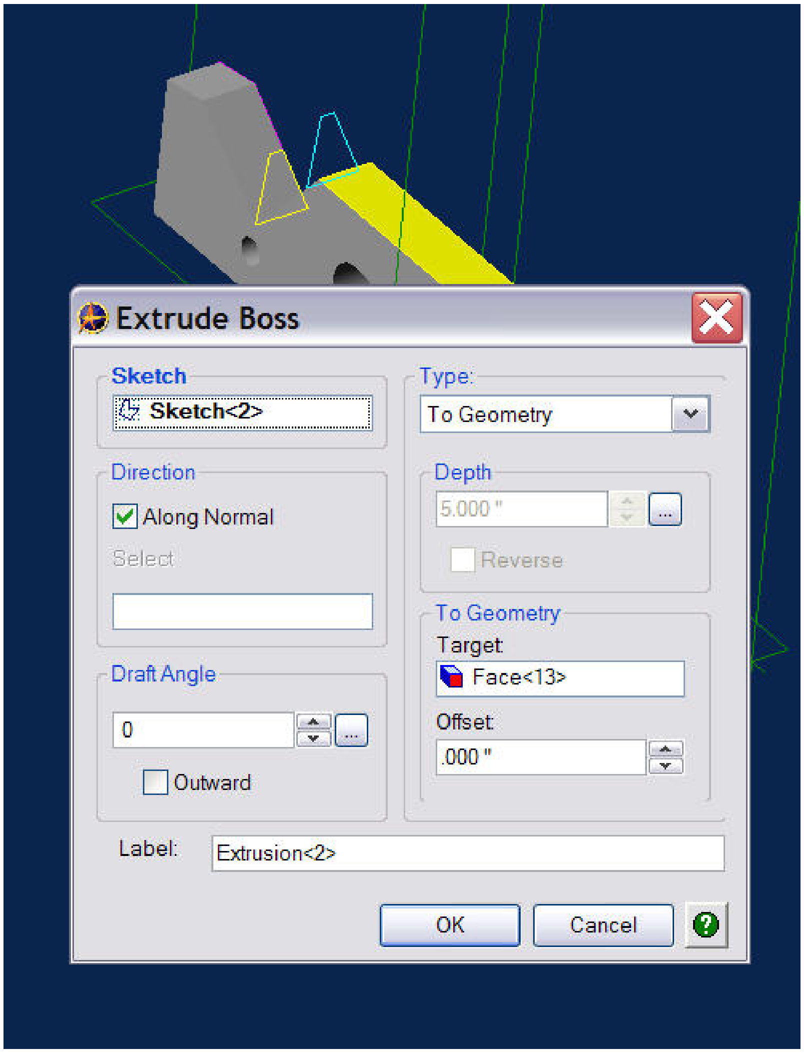


The next step is to create the machined angle guide as a separate extrusion. Click on the notch face shown, and then click on the 

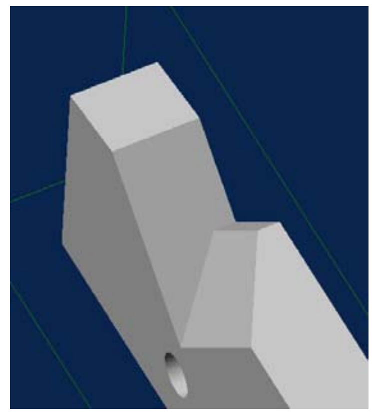
‘Sketcher’ icon.



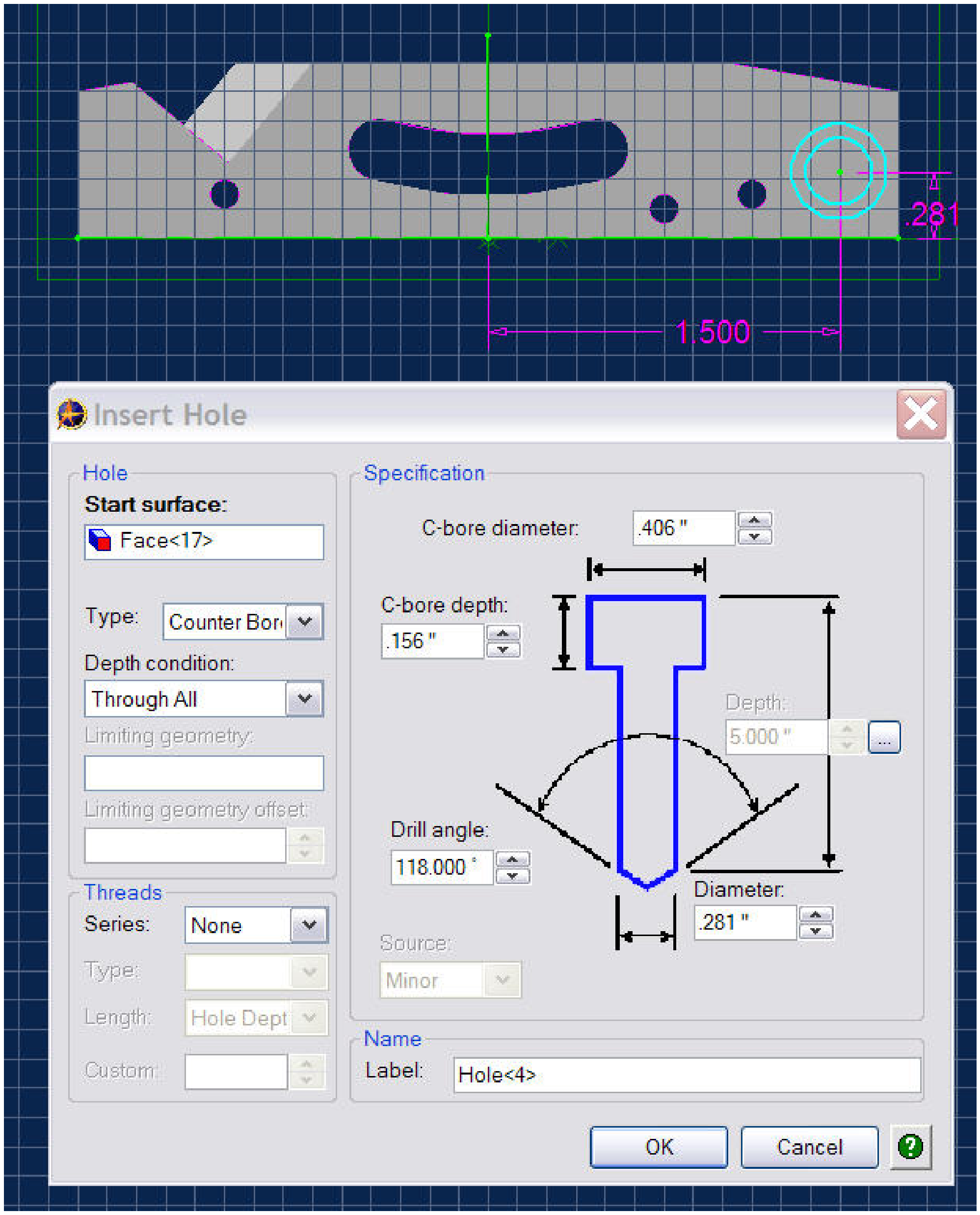
Create the sketch shown at the right. Constrain the bottom line of the sketch with the line describing the bottom of the notch, and the right vertical line with the right edge of the plate, using a collinear constraint for both.



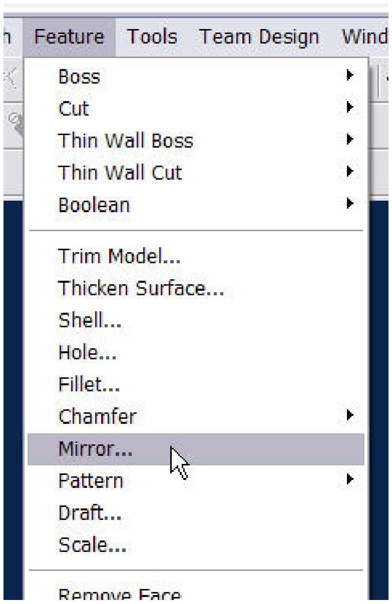
After the sketch is complete, click the ‘Extrude’ boss command and change ‘Depth’ to ‘To Geometry’ and select the top face of the Upper Plate as shown. The finished extrusion should look like that below.

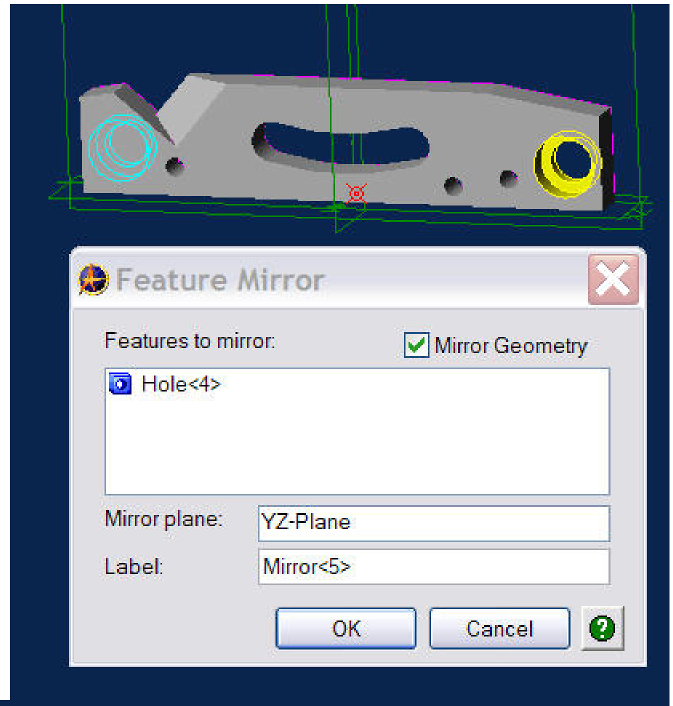


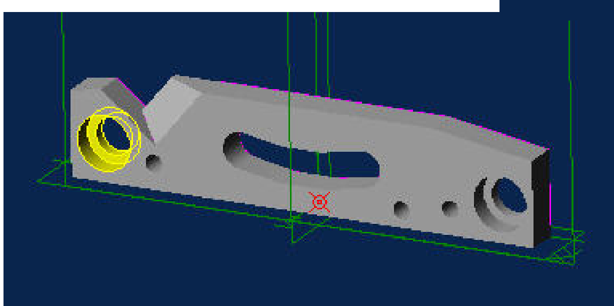
Next add the dress-up features, starting with the larger counter-bored holes and ending with the fillets. Click on the front face of the plate and then click on the 'Hole' icon to return to sketch mode and open the 'Insert Hole' panel. Click on the plate and modify the hole parameters as shown. Constrain the hole position per the dimensions shown



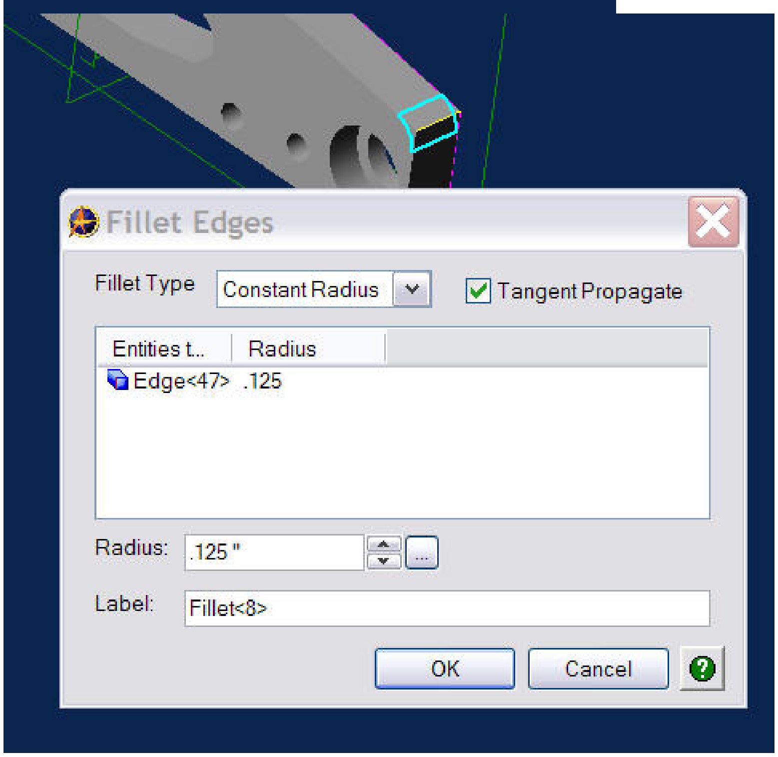
Use the ‘Mirror Features’ command
To duplicate the newly created hole to the other side of the plate.



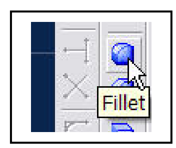




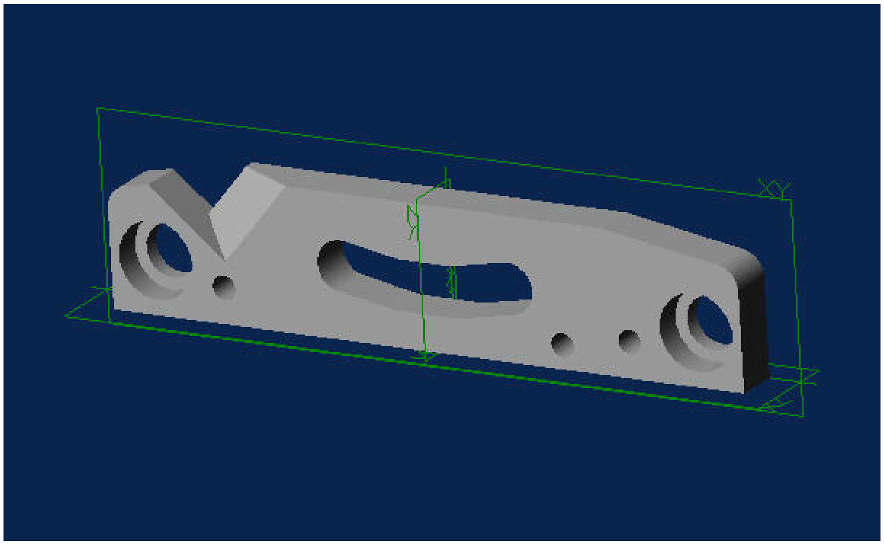
Your part should now look like this.



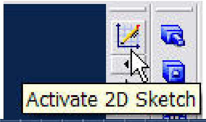
Add the Fillets by clicking on the ‘Fillet’



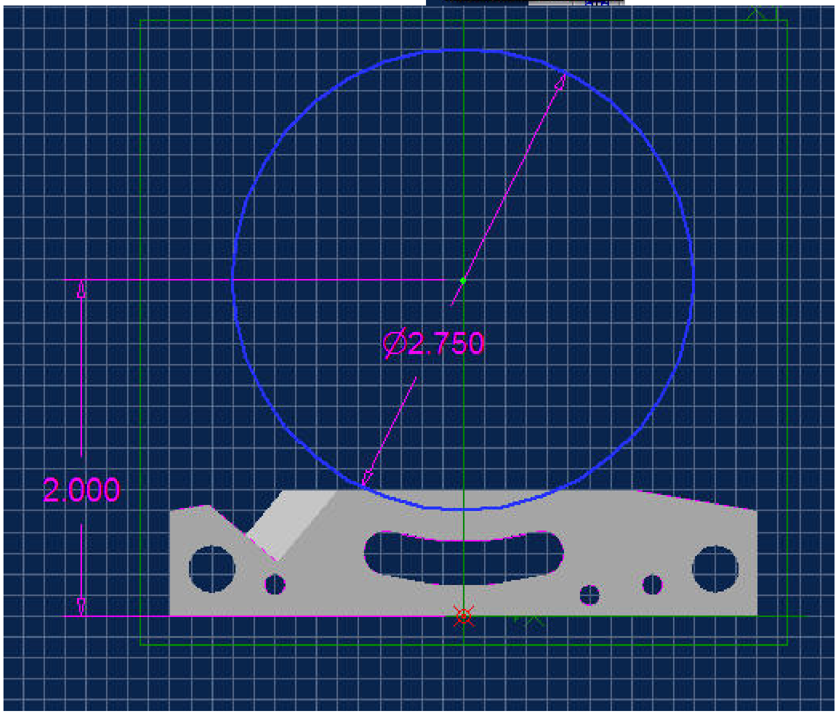
icon and selecting the two top edges, entering .125 in the Radius text box and clicking OK.



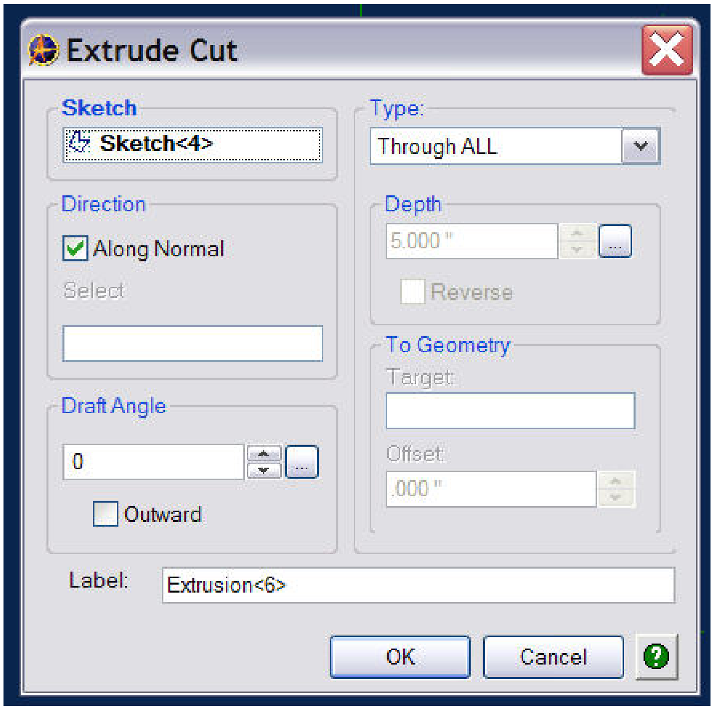
Your completed part should look like this. Save the part as 82000347 and then save it as 82000346 (for the Upper Plate model).

Make certain that the Upper Plate (82000346) part is open and active. Click on the XY plane, click on the Sketcher' icon 

and create, dimension, and constrain the circle shown below.

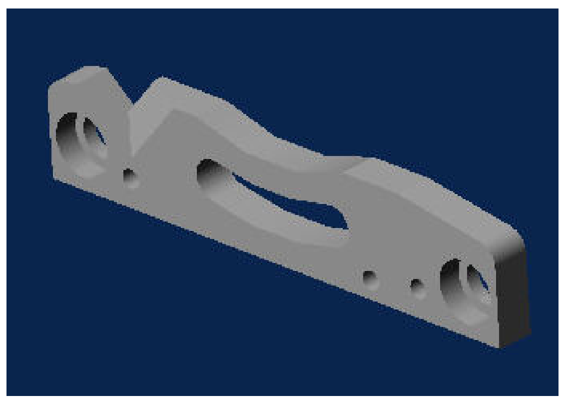


Click the 'Extrude Cut' icon. 



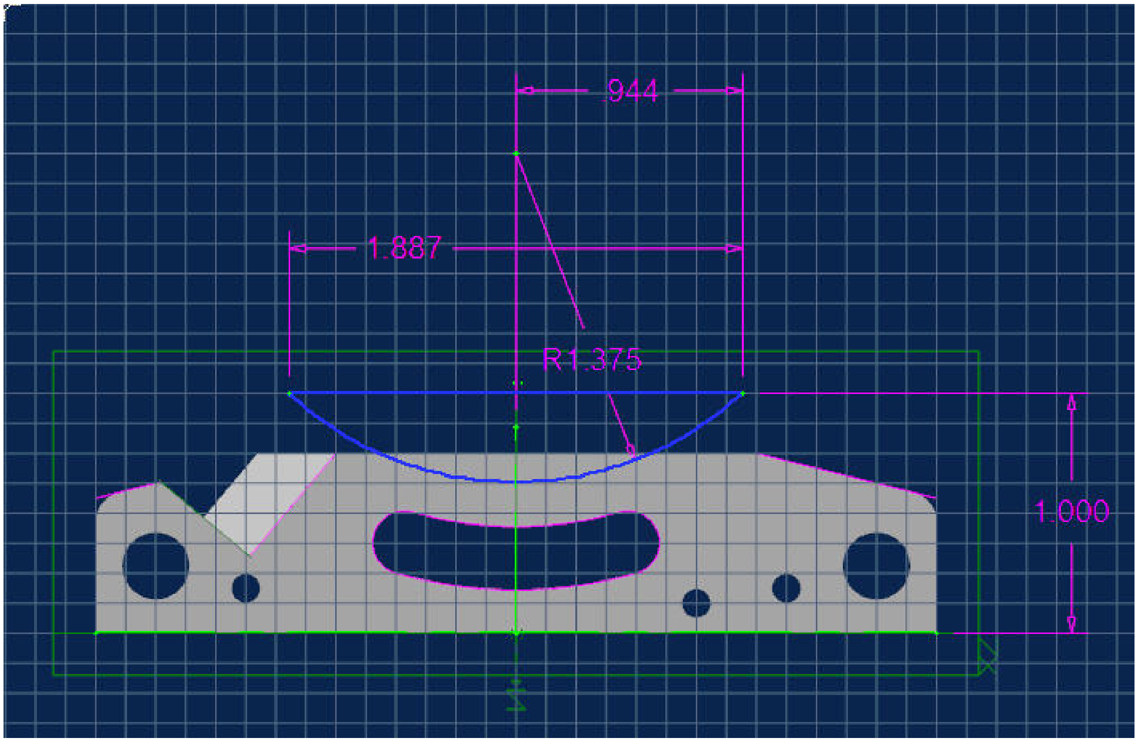
When the 'Extrude Cut' panel opens, change 'Depth' to 'Through All'.

Your part should look like this.
File it (as 82000346) and close it.

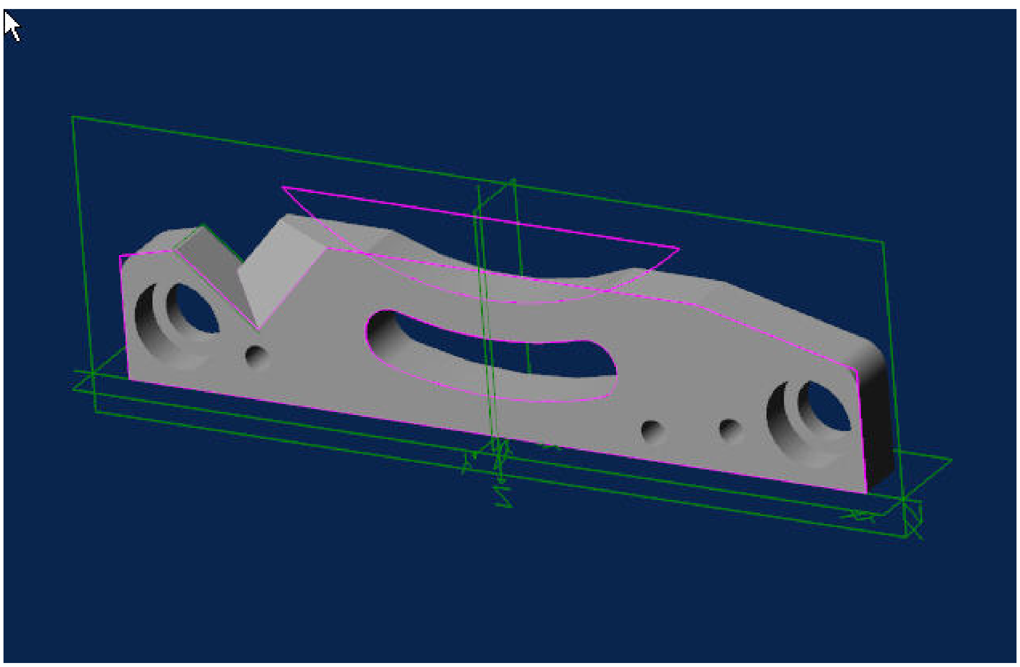


Designing the Any Angle Tool Vise – Upper Plate

The Upper Plate is a the same as the Lower Plate with one modification. With the Lower Plate model active, click File > Save as and save it as Part Number 8200346. Create the sketch shown below and extrude it 'Through All'.

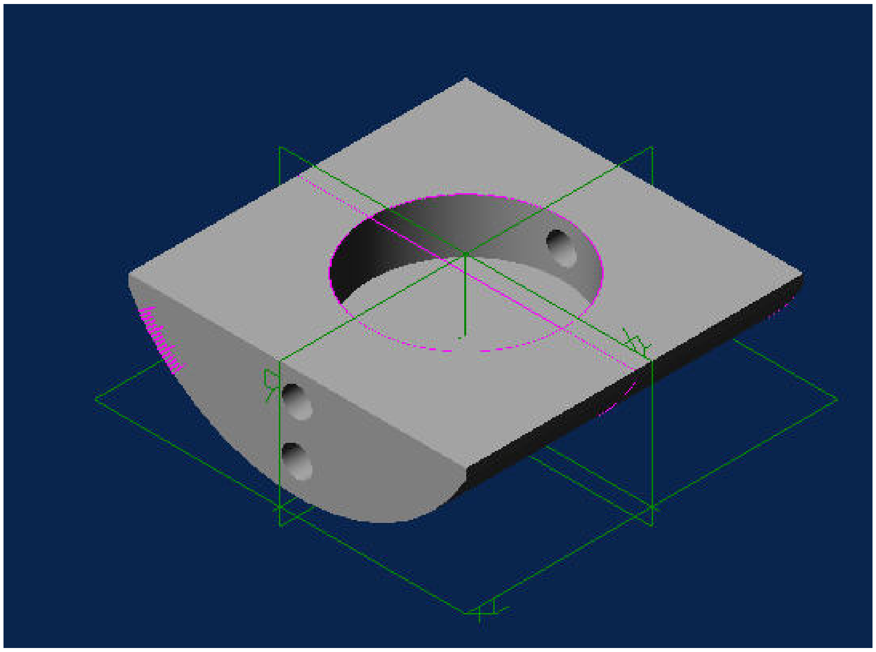


The part should now look like the part below. Save the model file.

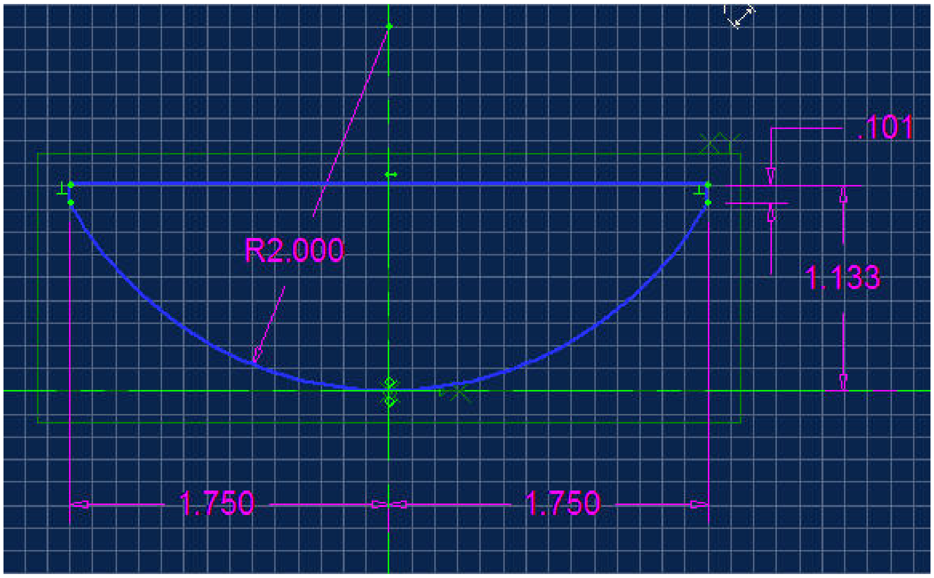


Chapter 6 - Designing the Any Angle Tool Vise – The Upper Compound Member

In this section of the design module, we'll expand our use of use the Workspaces and command sets we used previously, and we'll add the <u>Save, and Copy</u> sketch command that will allow us to transfer elements of one design to another, specifically the Angle markings, which will also be used in the Compound Center Member. Refer to Drawing X Upper Compound Member, for all construction dimensions.

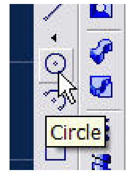


Create the sketch shown below, using the XY plane for the geometric support. Using the XY plane and extruding the shape using the Mid-plane modifier makes mirroring the hole features more efficient in that you don't have to create an offset symmetry plane to accomplish the task.

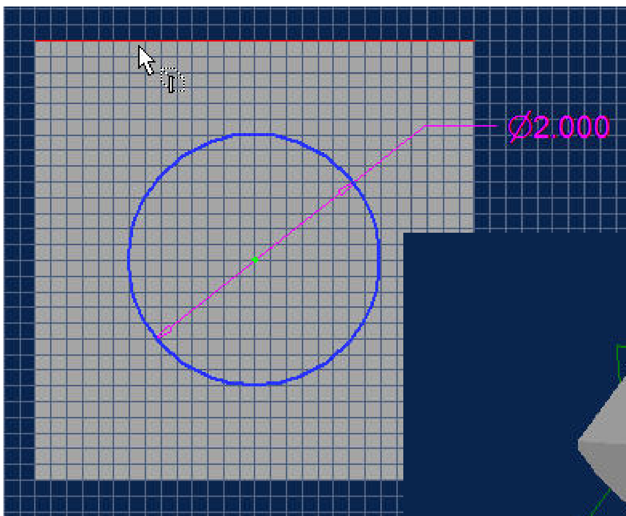


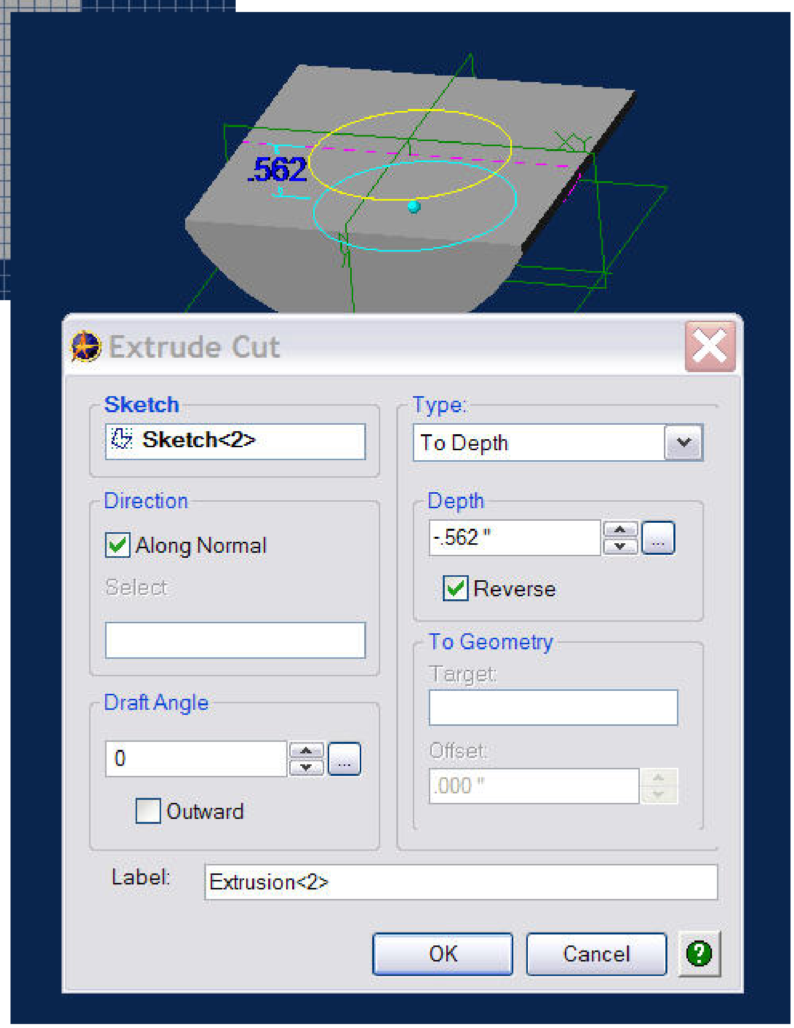
Once you've extruded the basic shape, the remaining features should be added in the following order:

- Large 'Hole' in the top.
- Two smaller Holes' in the front and then Mirror those features to the back of the part.
- The Angle markings – we'll construct a sketch of these markings for use in this design as well as in the design of the Compound Center Member part we'll create in the next exercise.

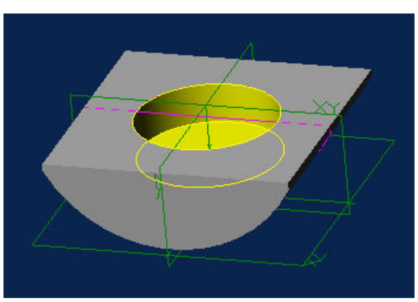
Click the top surface of the block and then click the 'Circle' icon, 

and sketch the circle shown below, using a 'Concentric' constraint to constrain it to the origin point. Extrude it to a depth of .562" as shown.



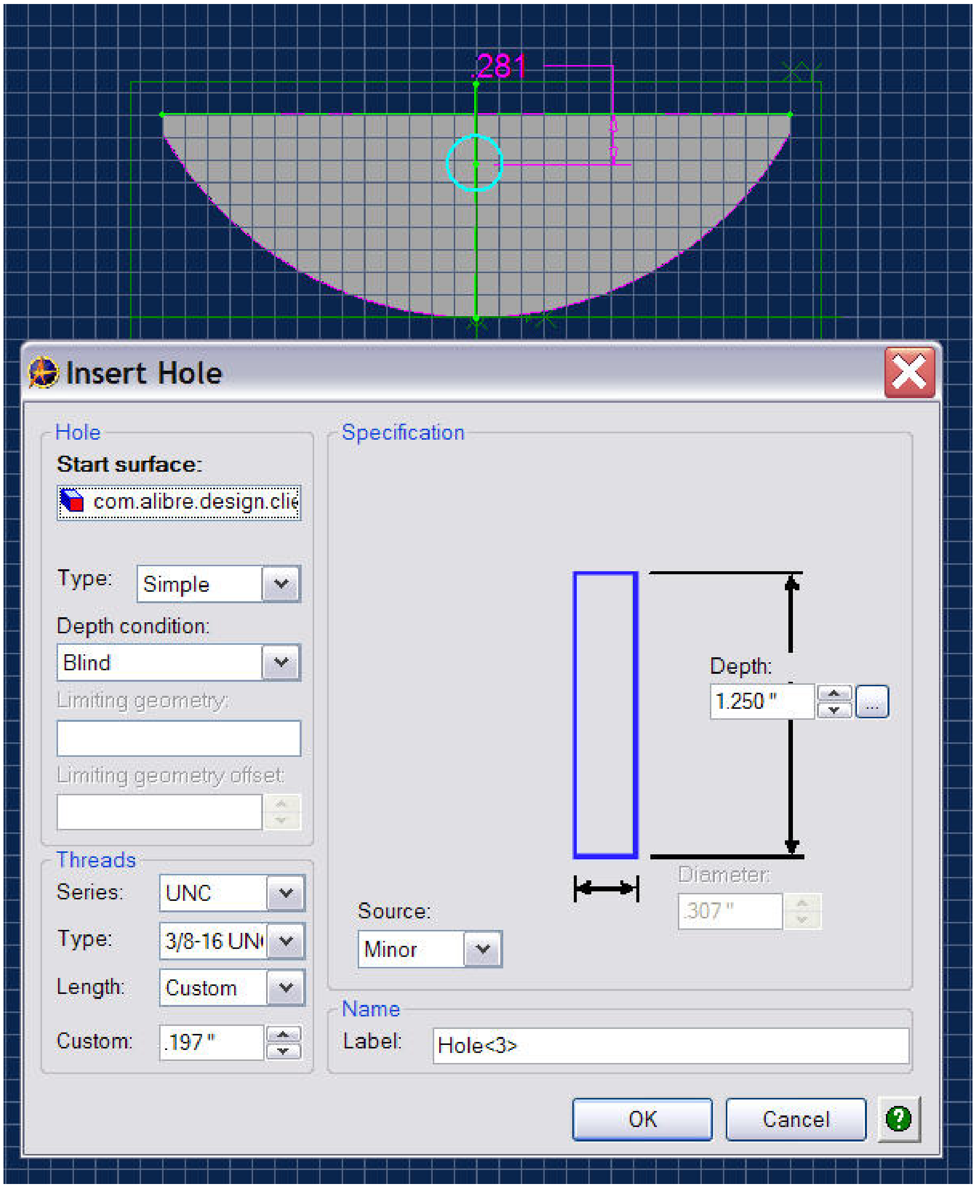


The part should now look like this.

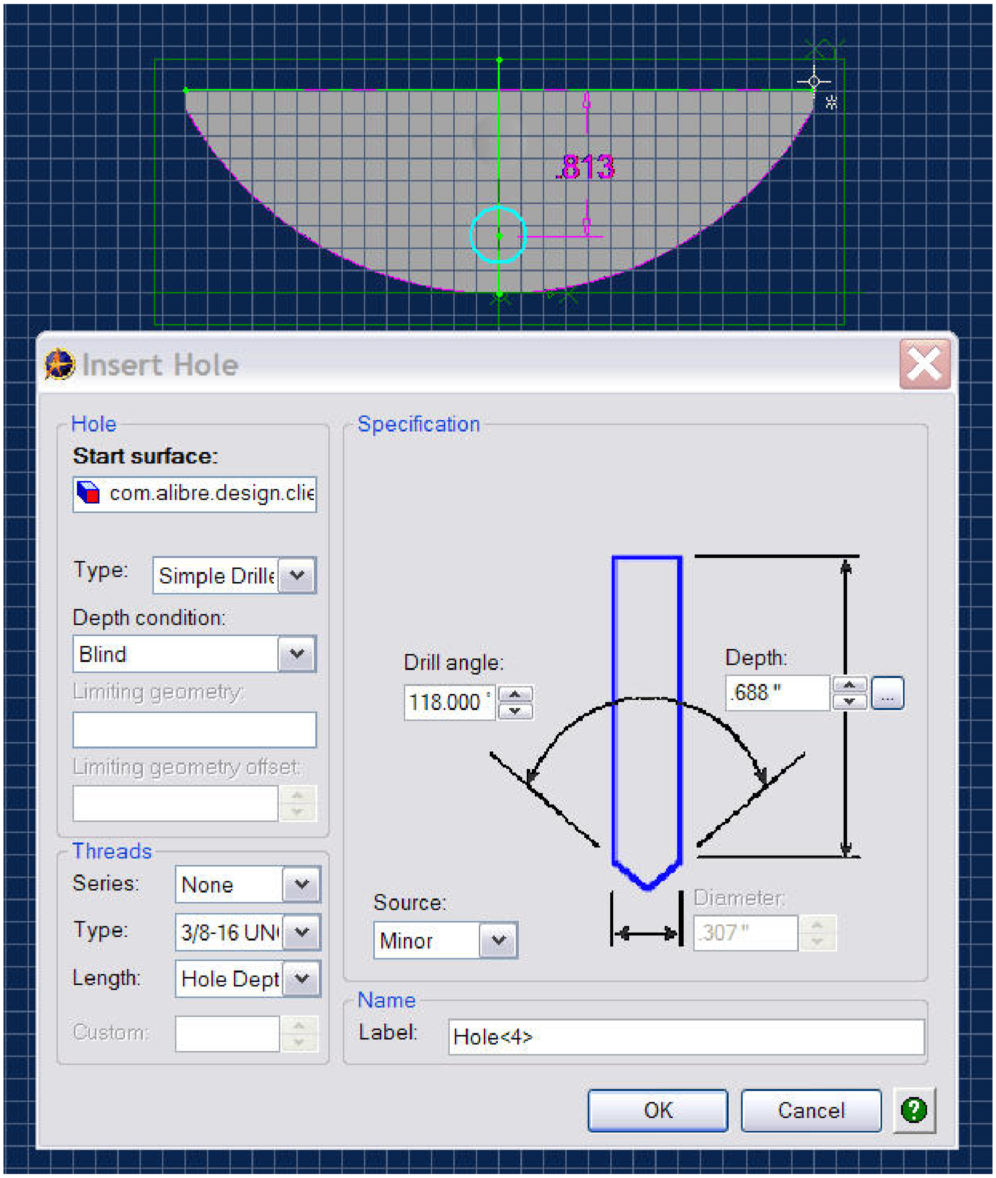


Click on either face of the block, click on the ‘Hole’ 

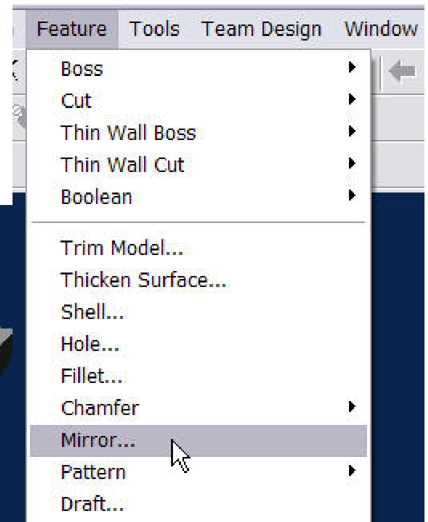
icon, and enter the values shown in the ‘Insert Hole’ panel, then click OK.

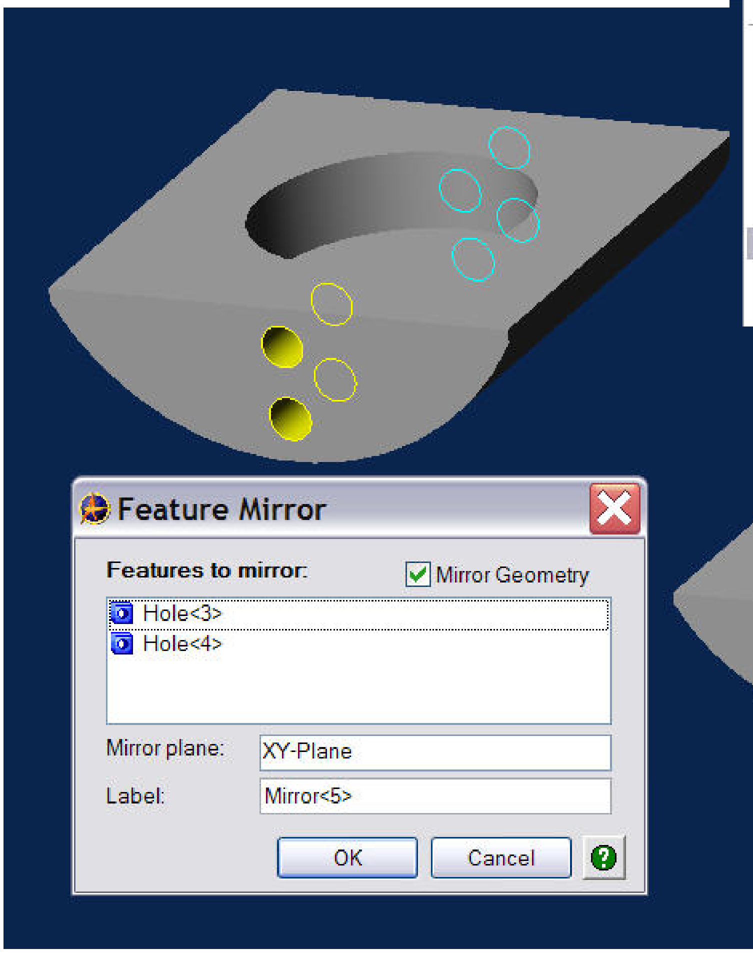


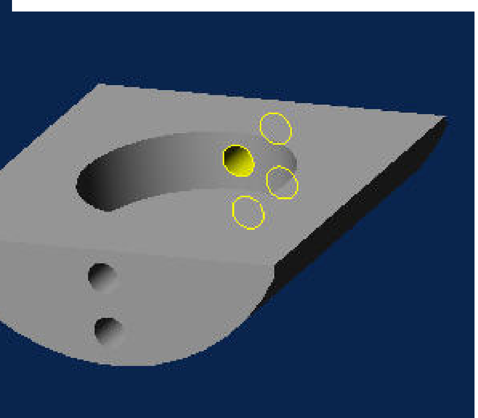
Repeat this process to create the second hole as shown below.



Click on the Feature tab in the top toolbar and select 'Mirror'. When the Feature Mirror panel opens select the two holes, you just created as the 'Features to mirror' and the XY plane as the Mirror plane. Click OK.

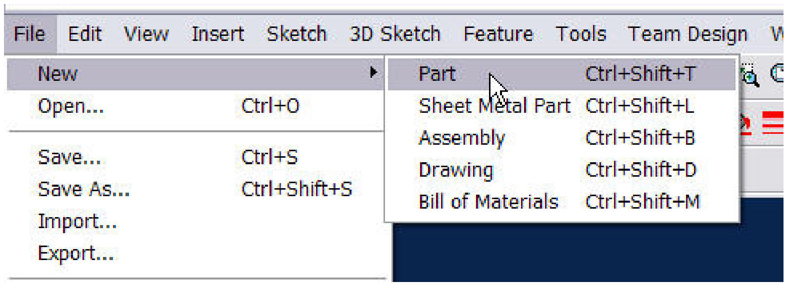


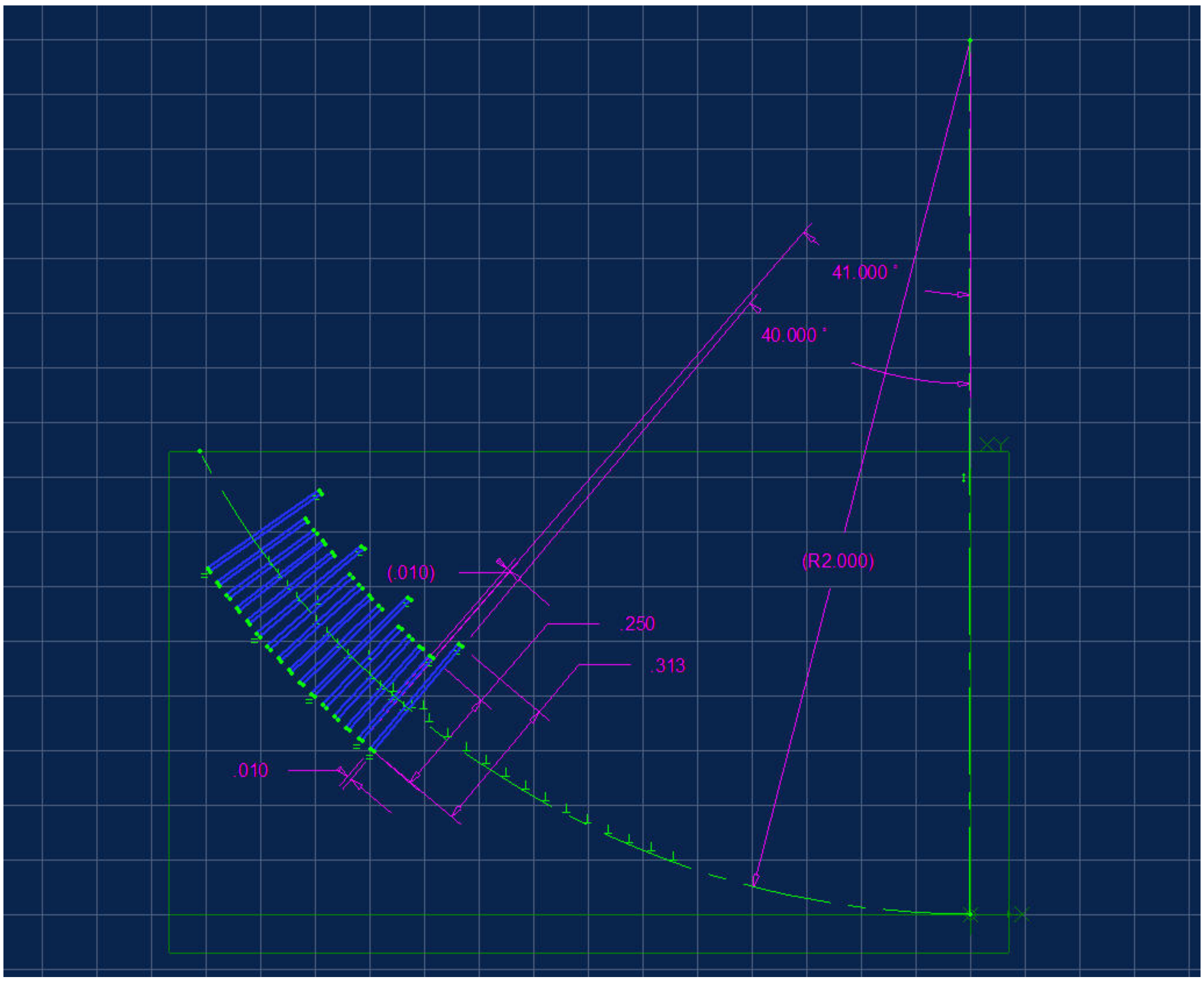




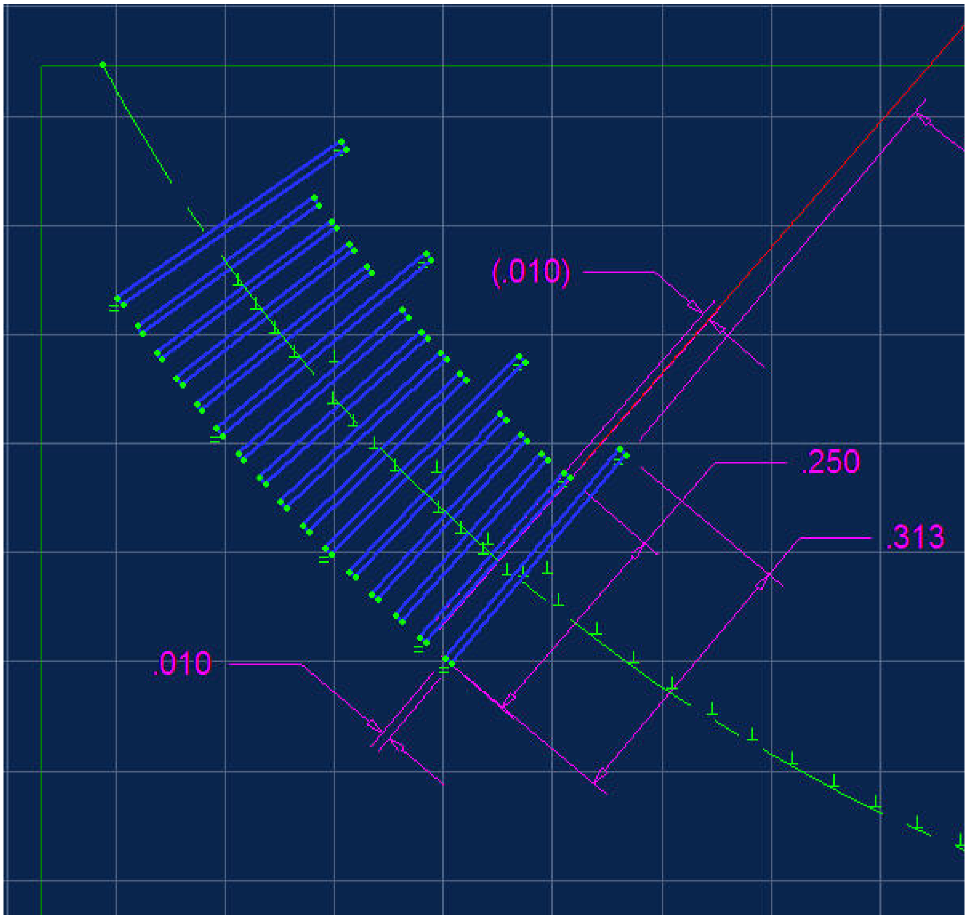
Your part should now look like this.

Open a new part and enter the Sketch mode by selecting the XY plane as the anchor for the geometry. Create the sketch shown below and on the next page. Use the Sketch > Repeat > Circular command to duplicate the degree markings. Duplicating a group of sketch features to simplify this procedure (one long line and four short lines rotated every 5 degrees for three copies.), will save you a lot of time and frustration.





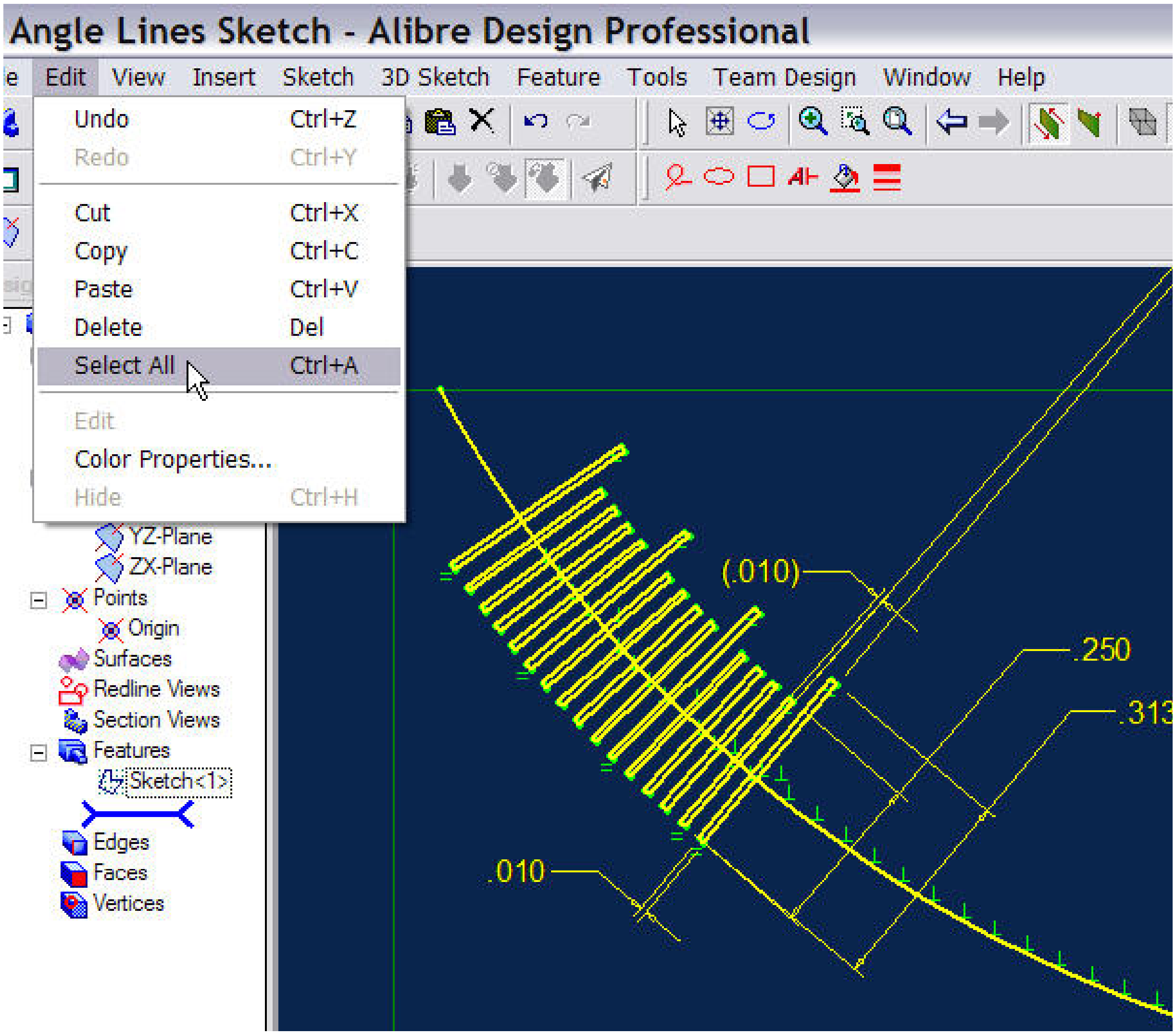
In the case of the degree markings sketch, solving the sketch to reduce the DOF to zero is involves a degree of dimensioning that if approached manually would be un-necessarily complex. If required, use the Sketch >Auto Dimension command, but save before you do so, just in case the results prove to be a little overwhelming.



Once you've completed the sketch, file IT under a reference number or name you'll remember. You'll be using it again when you create the Compound Center Member in Chapter 7.

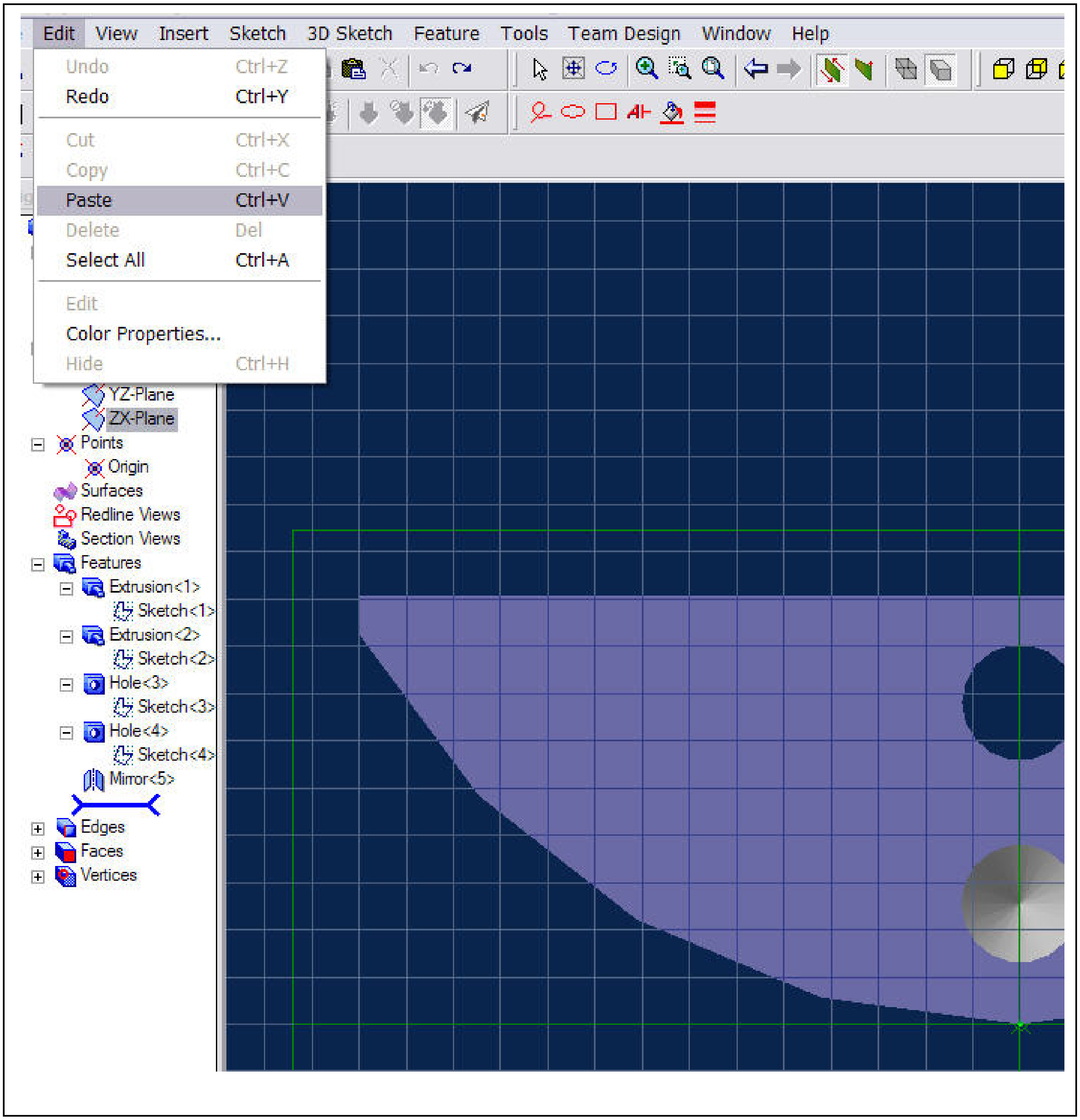
The ability to save sketches as parts and use them over an over again can be a real design time saver. If you use material with a specific cross-section (ex. I-beam, Channel, T-Bar, etc.) you can create your own library of sections, saving many hours of repetitious and time-consuming sketch work.

The sketch we created for our angle markings, could have been turned into a boss feature, saved as a library part, and used in a Boolean Subtract operation with the same results. Pick the method that is most useful to you, but always keep in mind that in Alibre, there are usually several paths to the same destination.



Select the sketch entities you just created by using the 'Edit> Select All' command and the selecting 'Copy'. Make sure your in Sketcher mode or you won't be able to select the sketch entities.

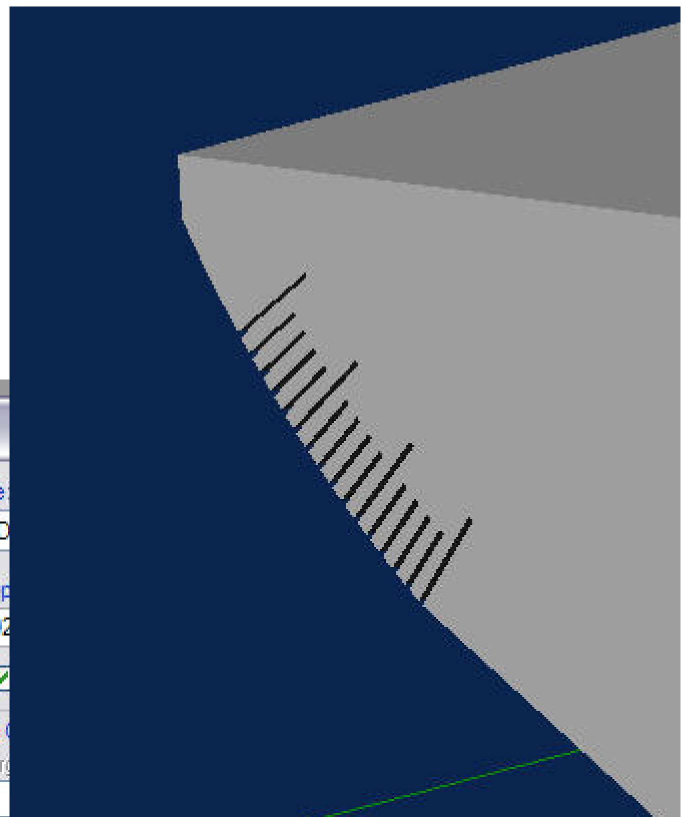
Return to the Upper Compound Member window, click on the front face of the block, click on the Sketcher icon to enter Sketch mode, and under Edit, select 'Paste'. Click on the origin point for the Upper Compound Member to place the sketch in the proper location.

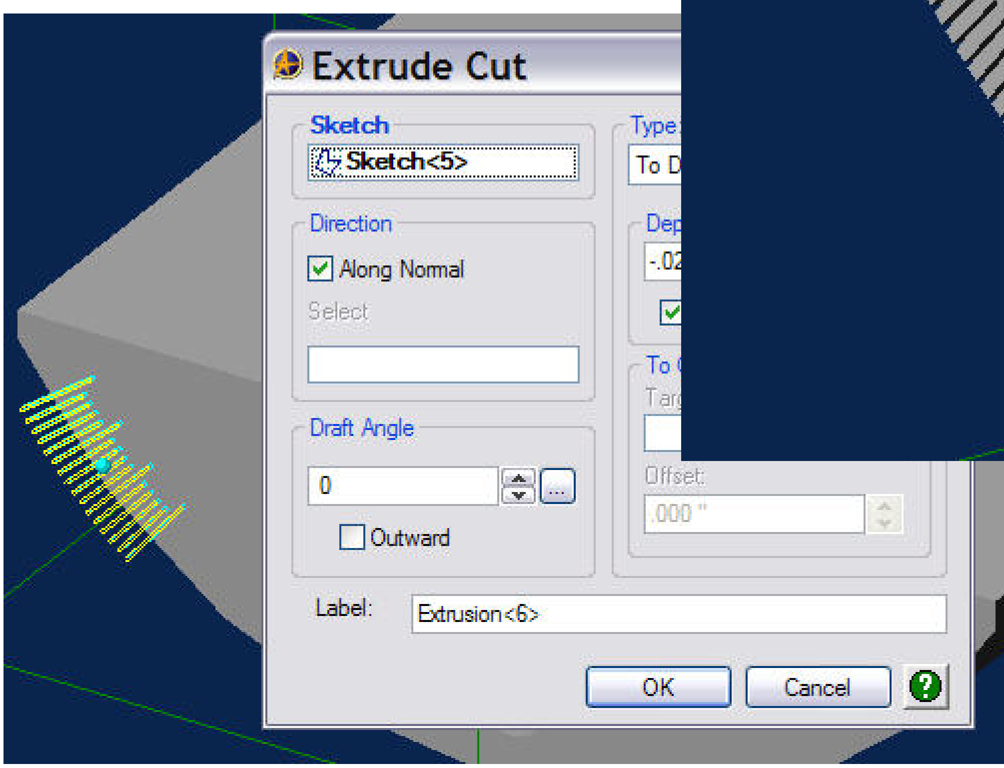


Now execute an Extrude Cut to Depth, making the depth .020 in.

The part should now look like this.

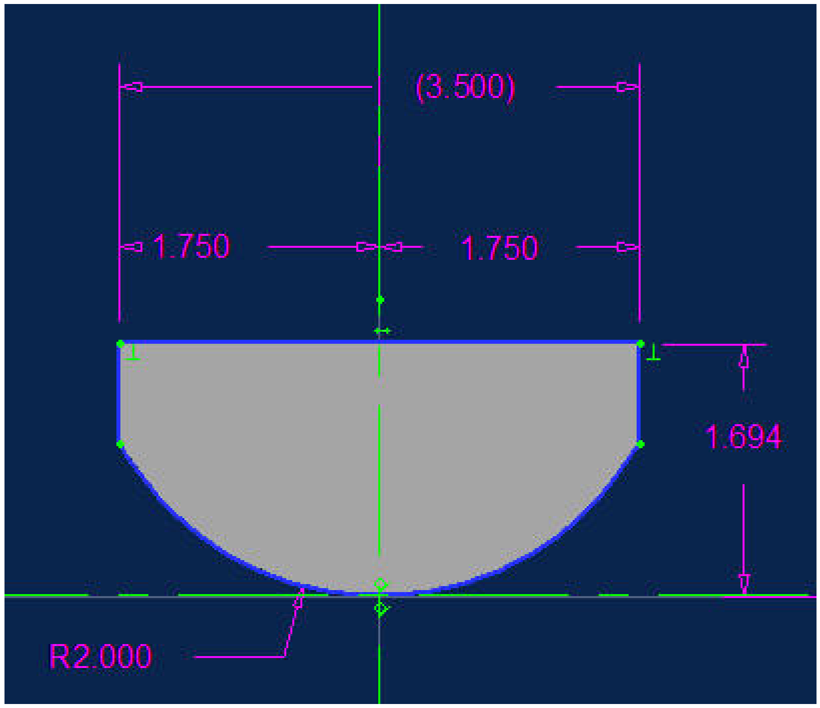
File this part as Part Number 82000348.



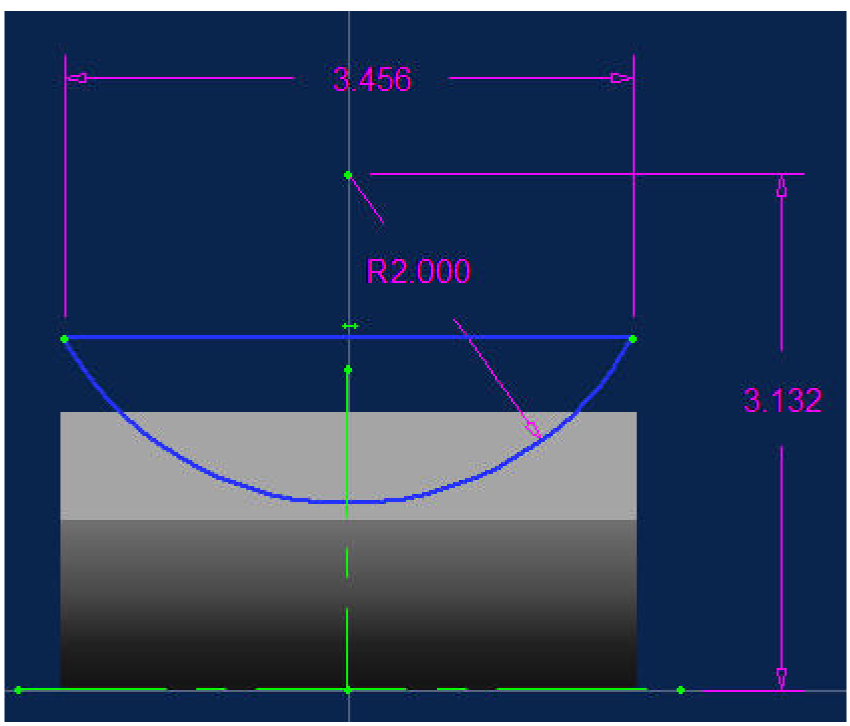


Chapter 7 - Designing the Any Angle Tool Vise – The Compound Center Member

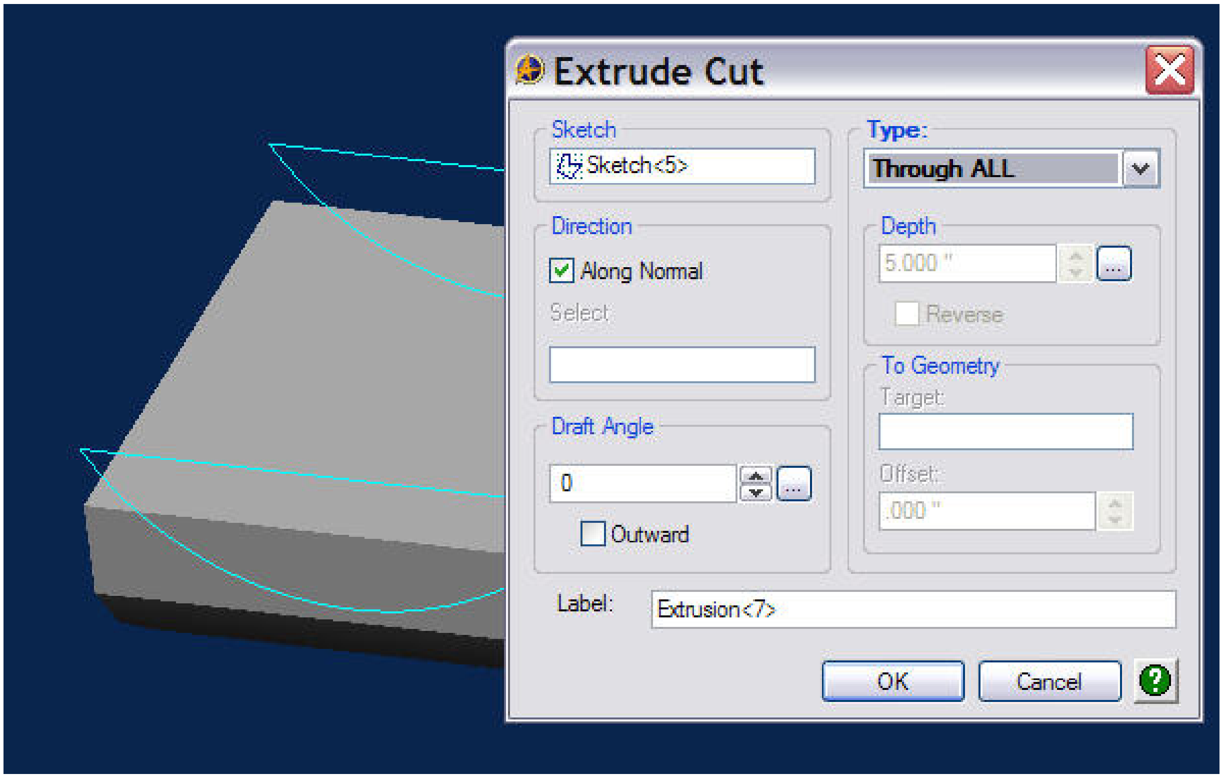
The first step is to extrude the basic shape of the part using the XY plane as the geometric anchor for the sketch shown below. Use the Mid-plane modifier to extrude the part symmetrically about the XY plane. Doing this will make the next operations much easier.



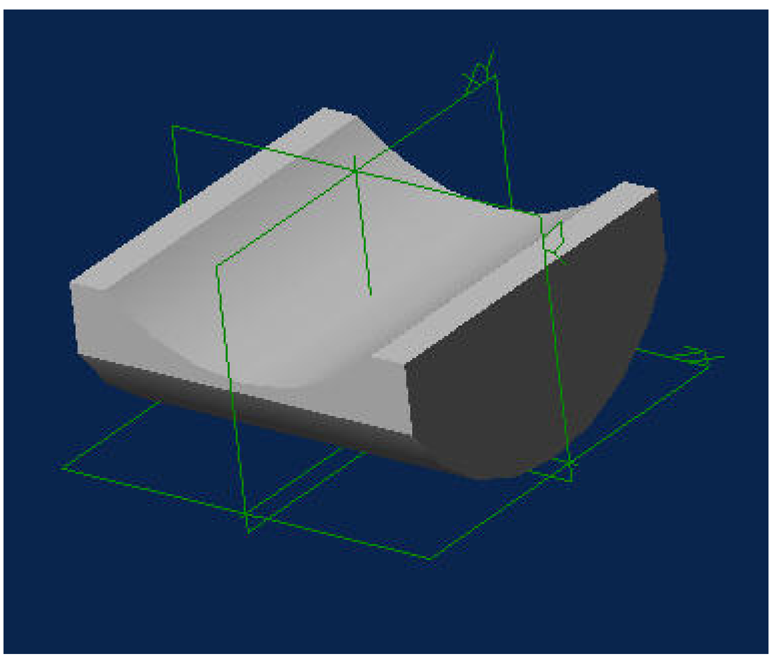
In conjunction with the Mid-Plane extrusion modifier, remove the material for the top curved slider bed in the same manner as used on the Saddle base. Create the sketch shown.



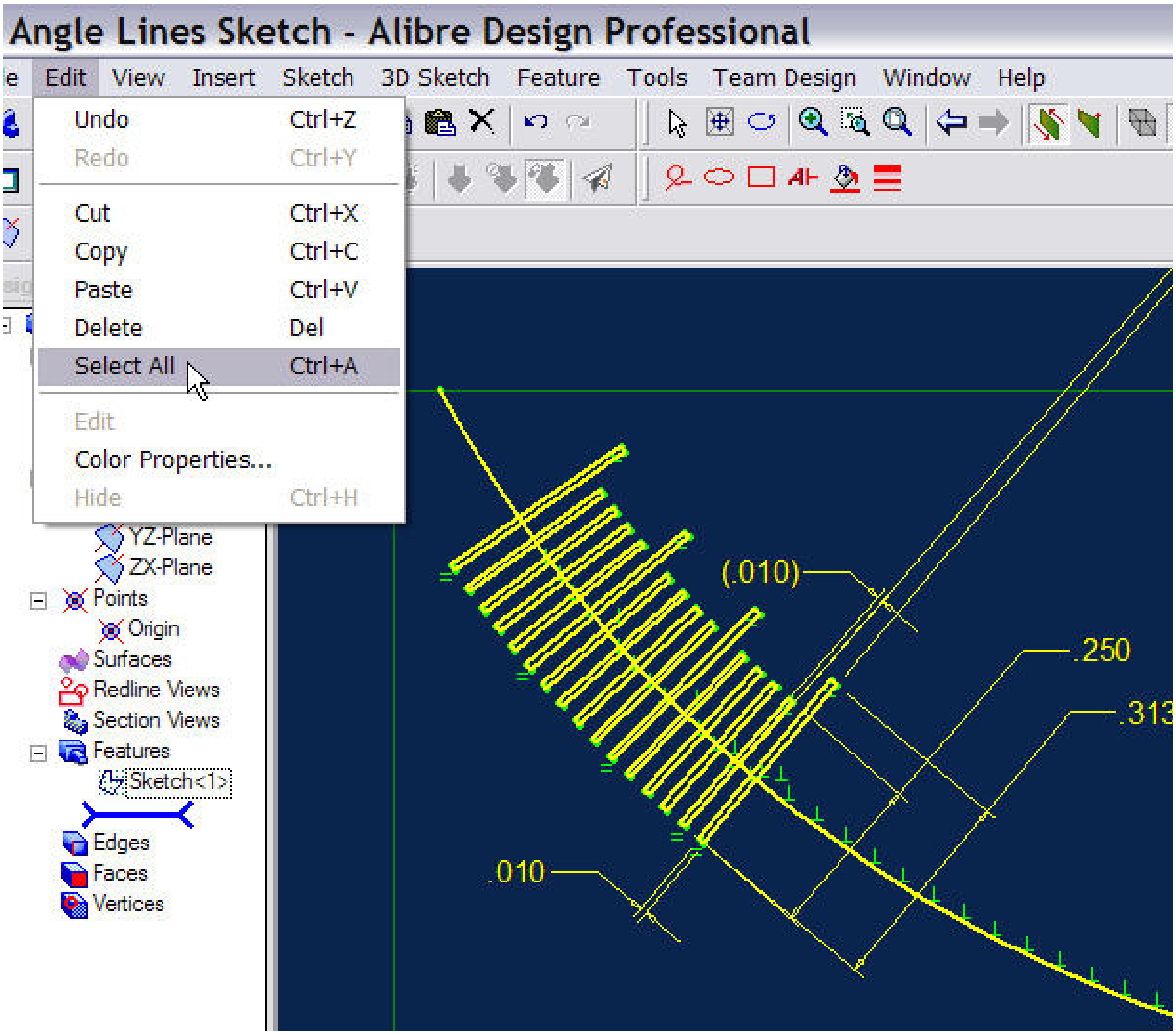
Use the ‘Extrude Cut’ command with the ‘Through All’ modifier



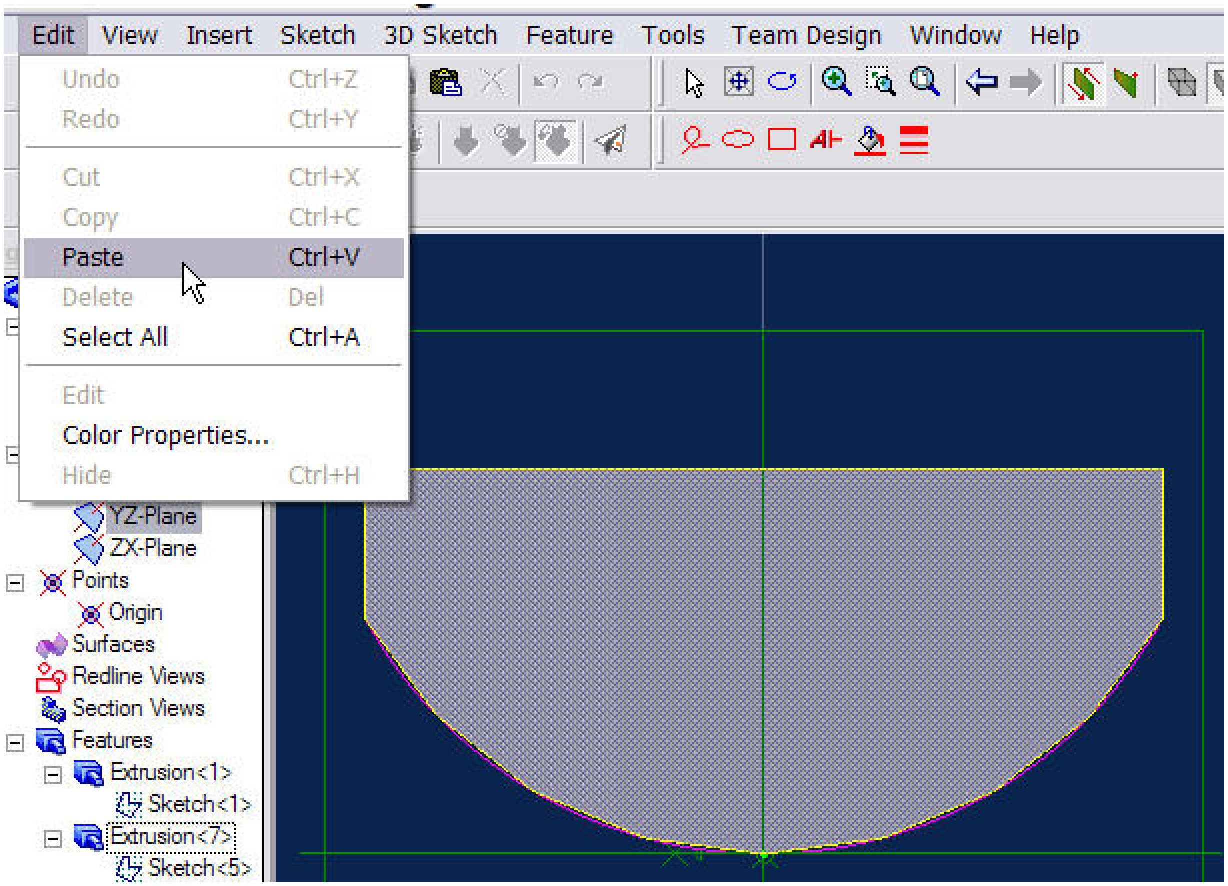
Your part should now look like this.

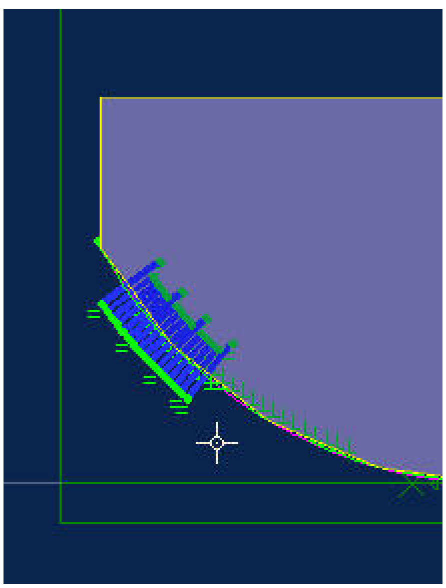


Open the Angle Lines sketch you created in the previous chapter and copy it.



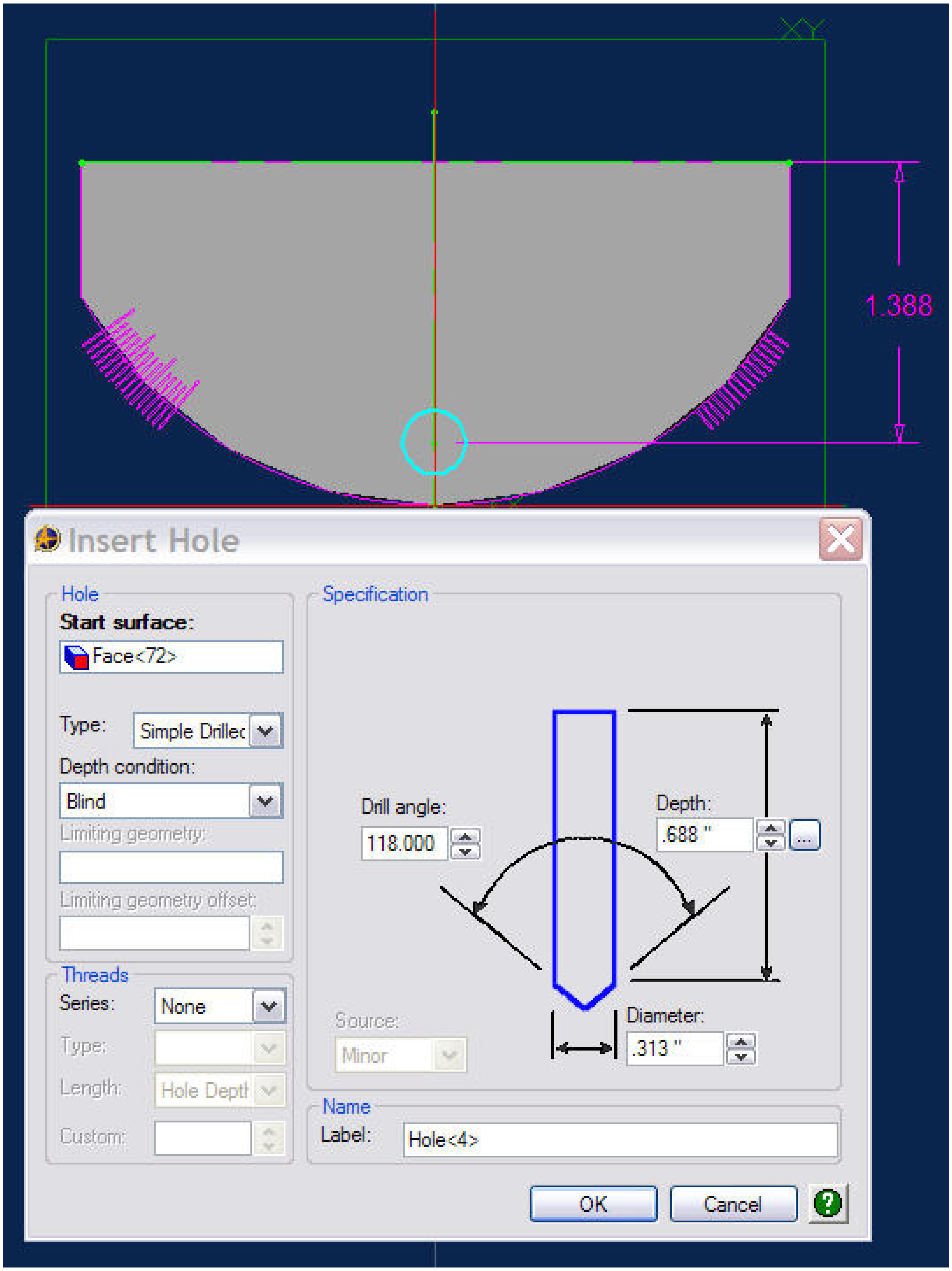
Click on either of the flat faces of the part and paste the sketch on to the part, and it extrude as you did for the Upper Compound Member.



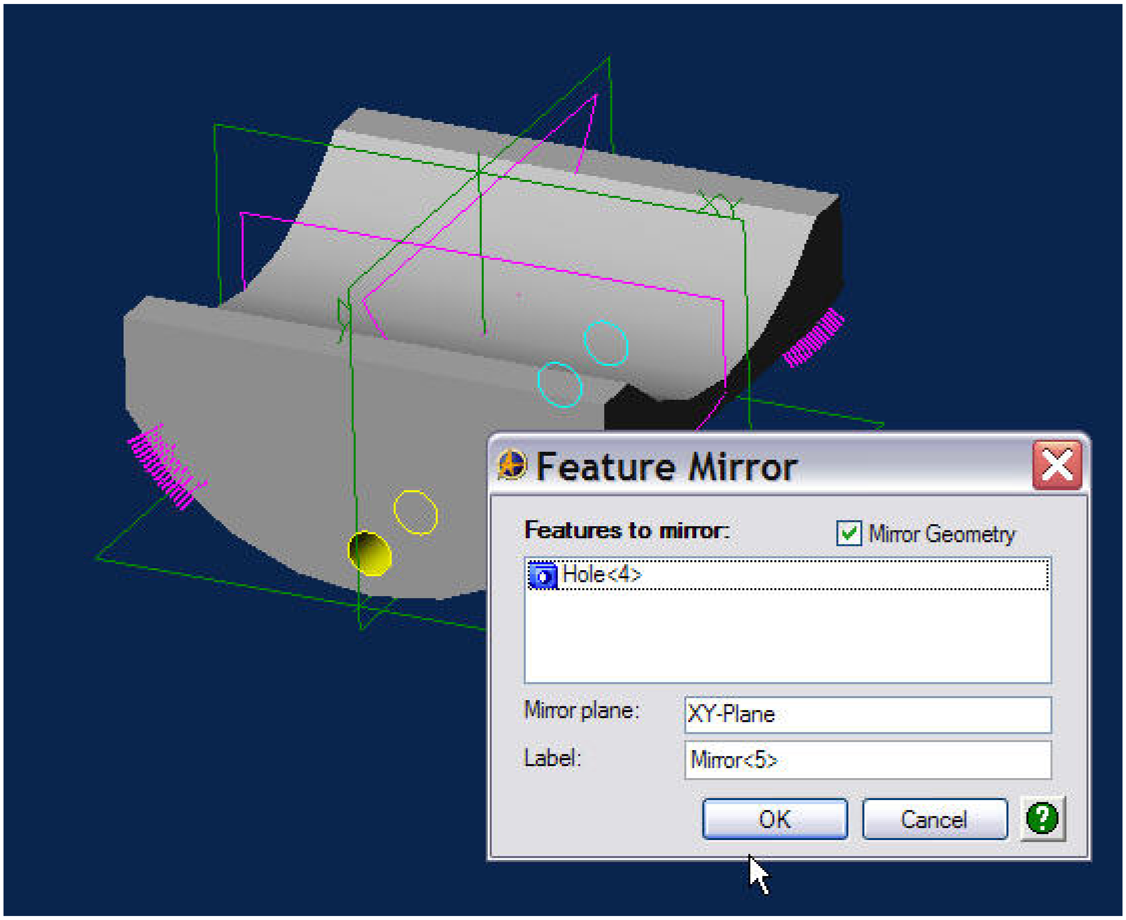


Extrude the Angle line sketch to a depth of .020 in, just as you did in the Upper Compound Member.

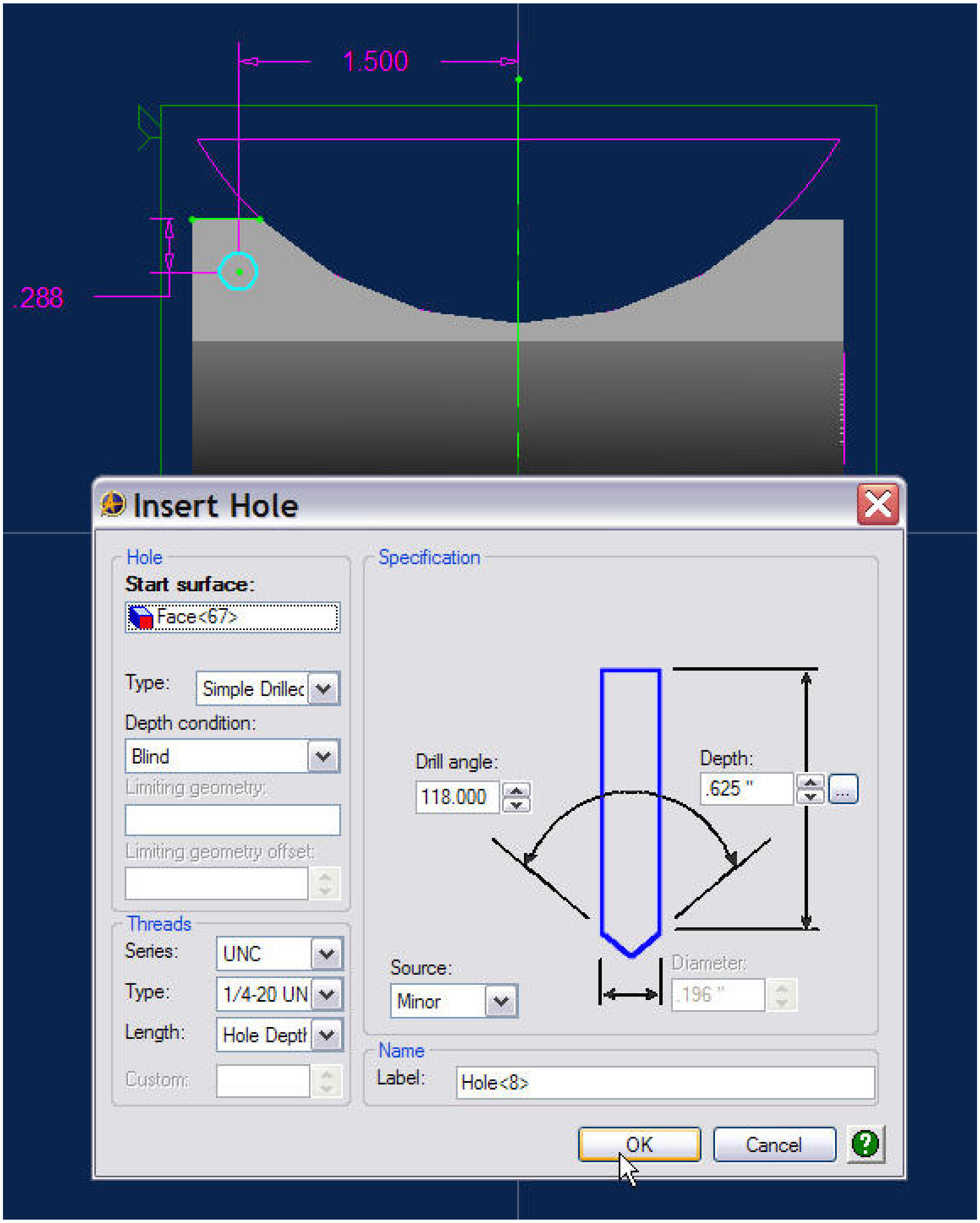
Insert the hole for the Eccentric shaft per the sketch below.



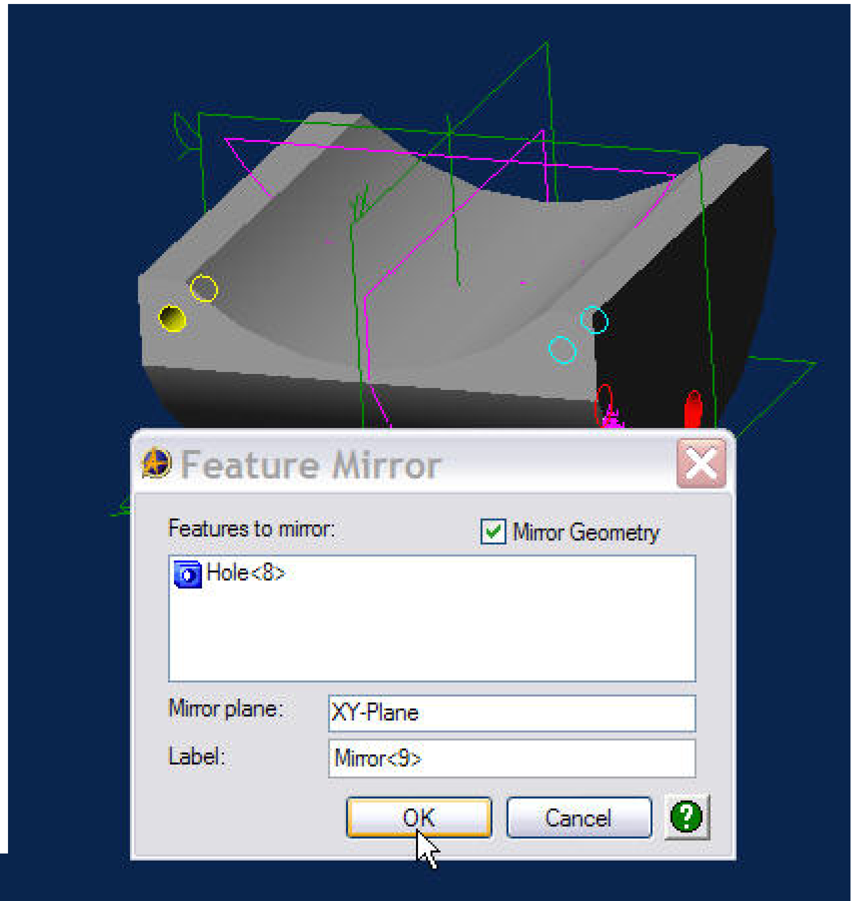
Mirror it using the XY plane as the Mirror plane.

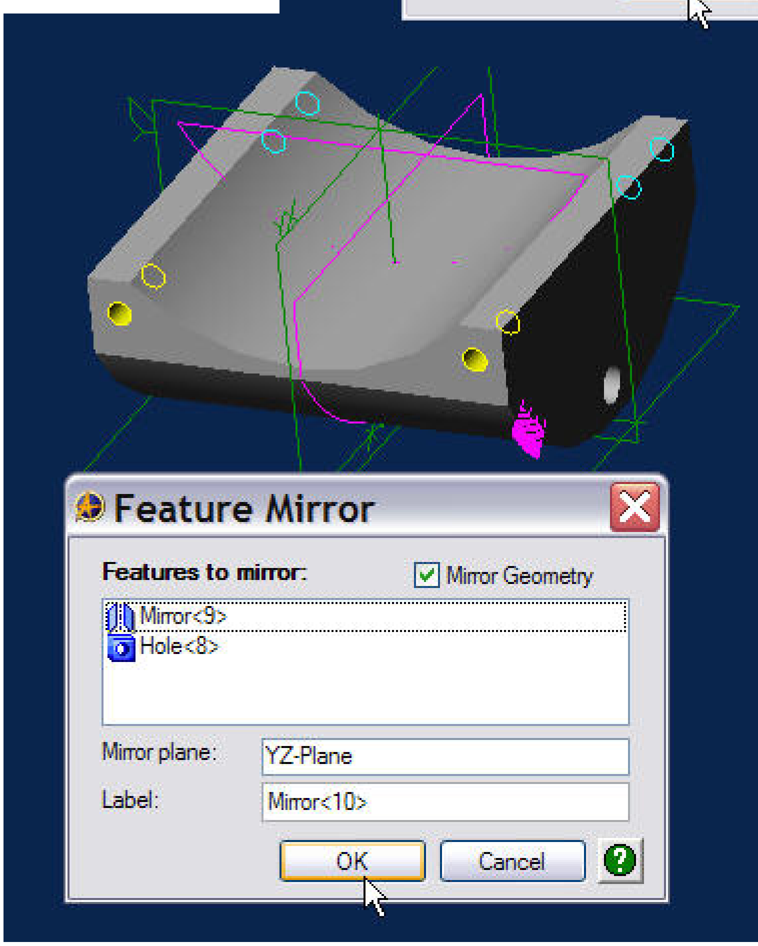


Insert the first hole for the Upper Plate(s) using the sketch shown below.

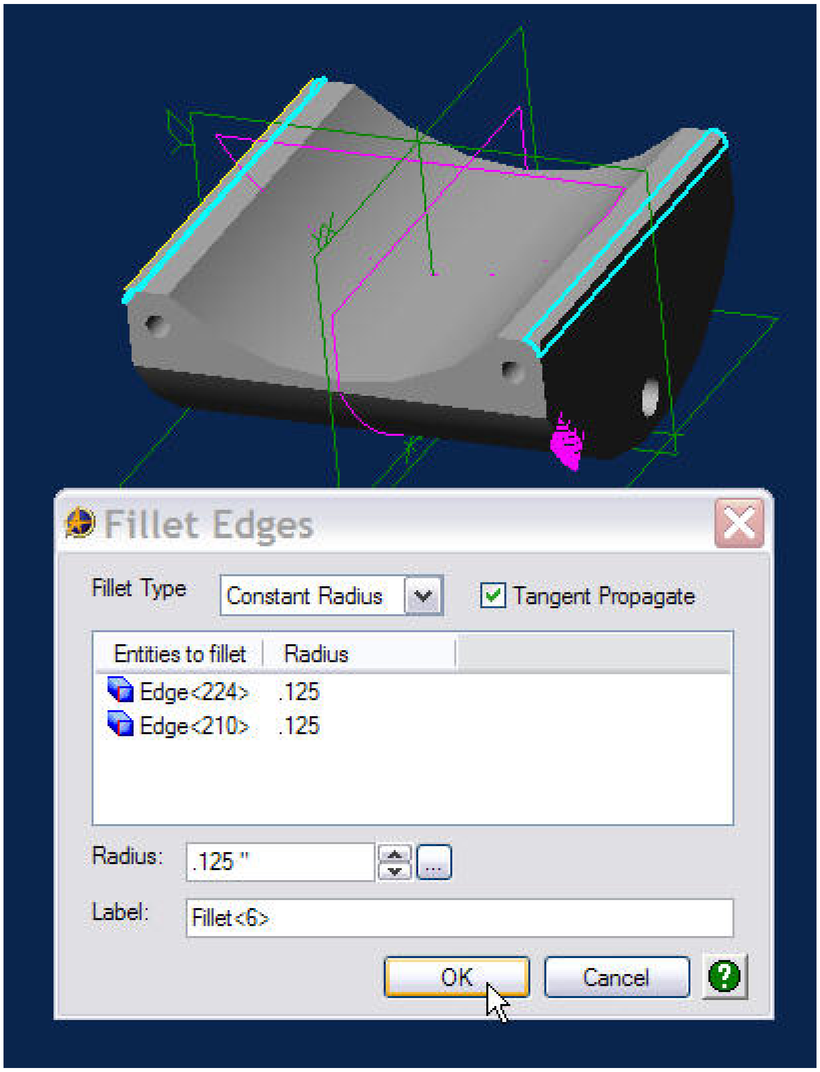


Mirror the first hole, using the XY plane as the Mirror plane.



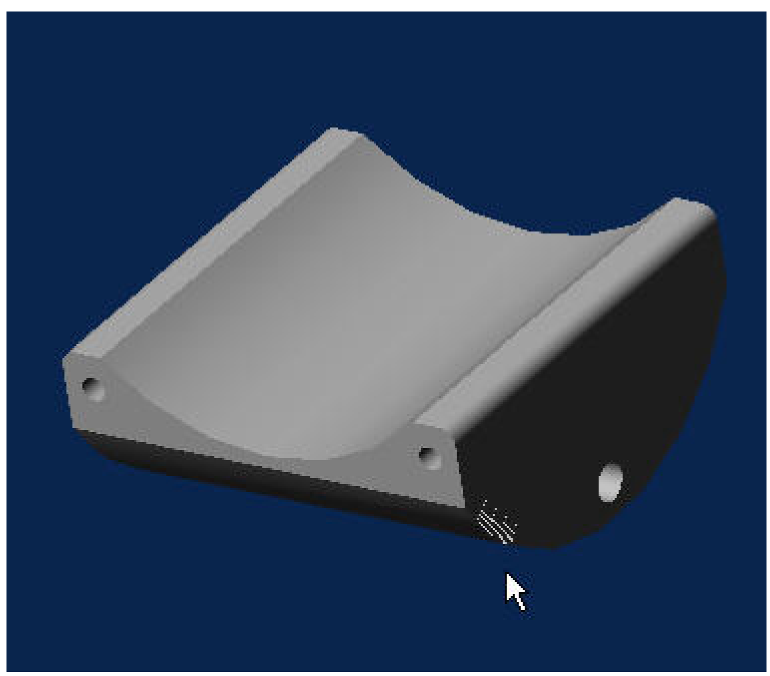


Then mirror the two resulting holes using the YZ plane as the Mirror plane.



Finally, create the fillets.

The finished part should look like this. File the part as Part number 82000349.

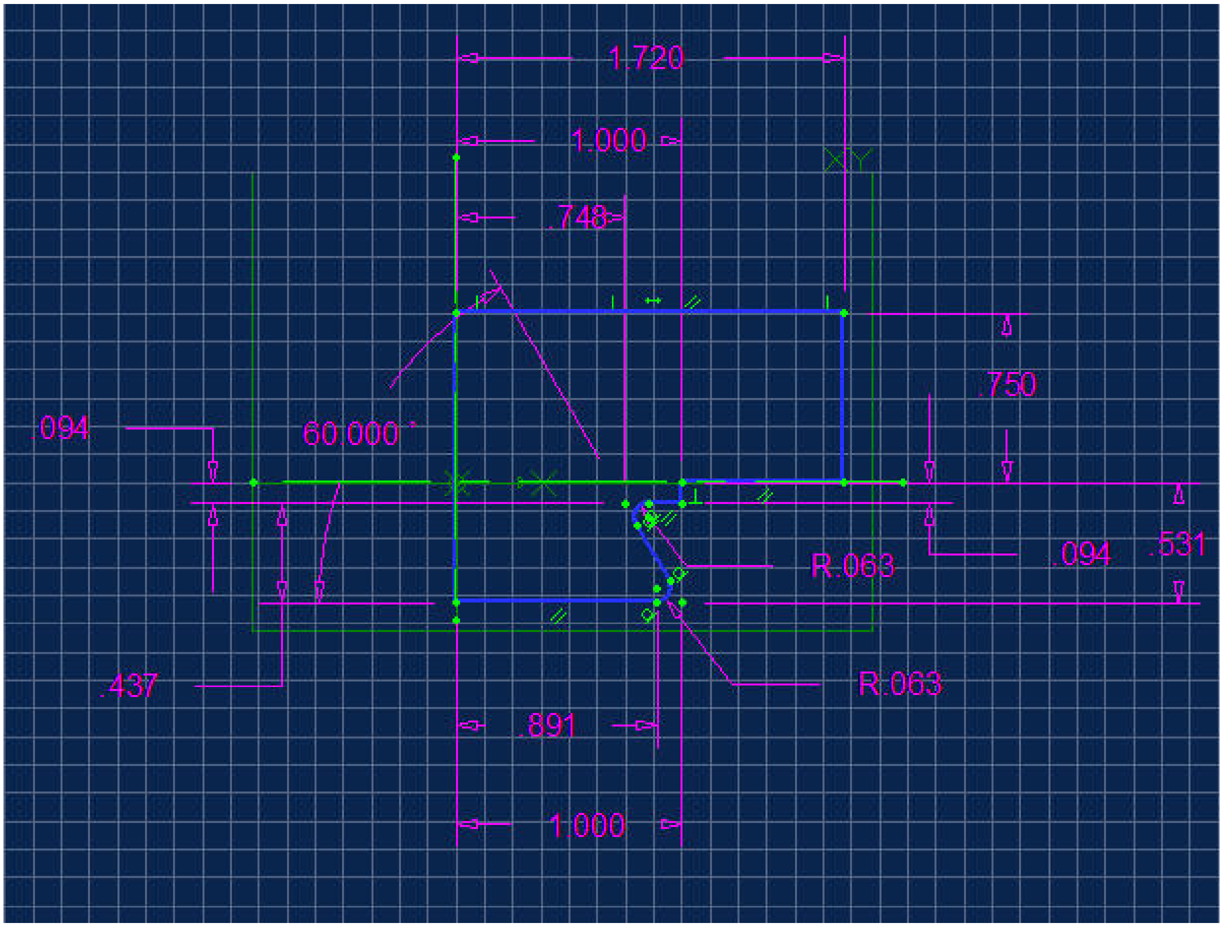


Chapter 8 - Designing the Any Angle Tool Vise – The Compound Tool Holder

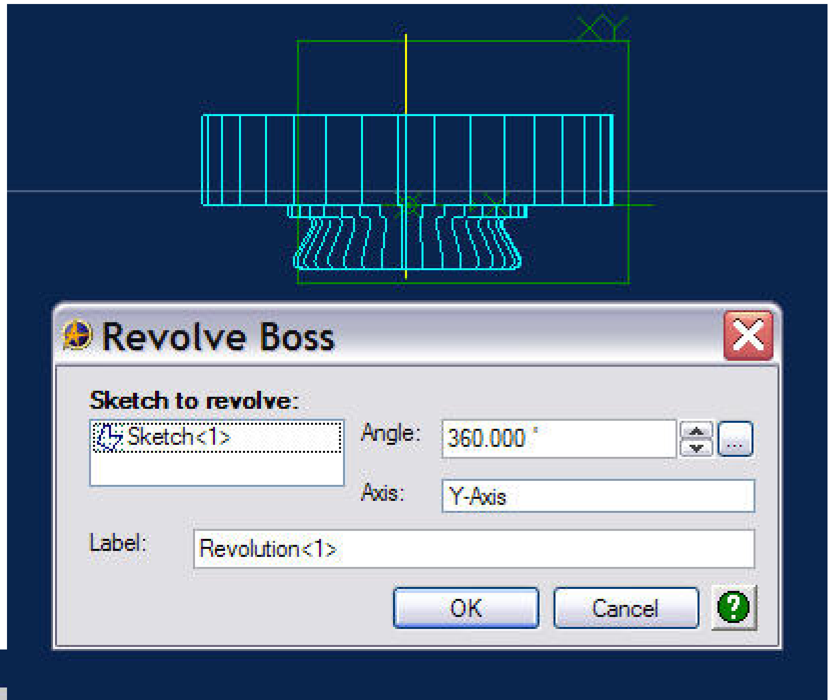
The Upper Tool Holder is made from 3.500 in CRS round stock, turned to the finished 3.440 diameter, machined to include the holder slot, counter-bored, and tapped, and simple drilled and tapped holes. It is a simple, but relatively expensive part to manufacture, requiring multiple machining set-ups and if the part is to be mass-produced, a checking fixture or fixtures of a precision even more expensive to produce. A large investment in time on the part of the quality control department would also be necessary to certify the part.

In creating the sketch below, I chose to include the .063 R fillet radii in the basic sketch instead of adding them as dress-up features later in the design process. The reasoning behind this is based on the assumption that these particular design features will not change even if certain features in the mating parts do. If changes do occur in these parts, a re-evaluation and possible re-design of the Compound Tool Holder would necessitate a revision of the basic sketch, at which point a change in fillet radii would be easy to complete.

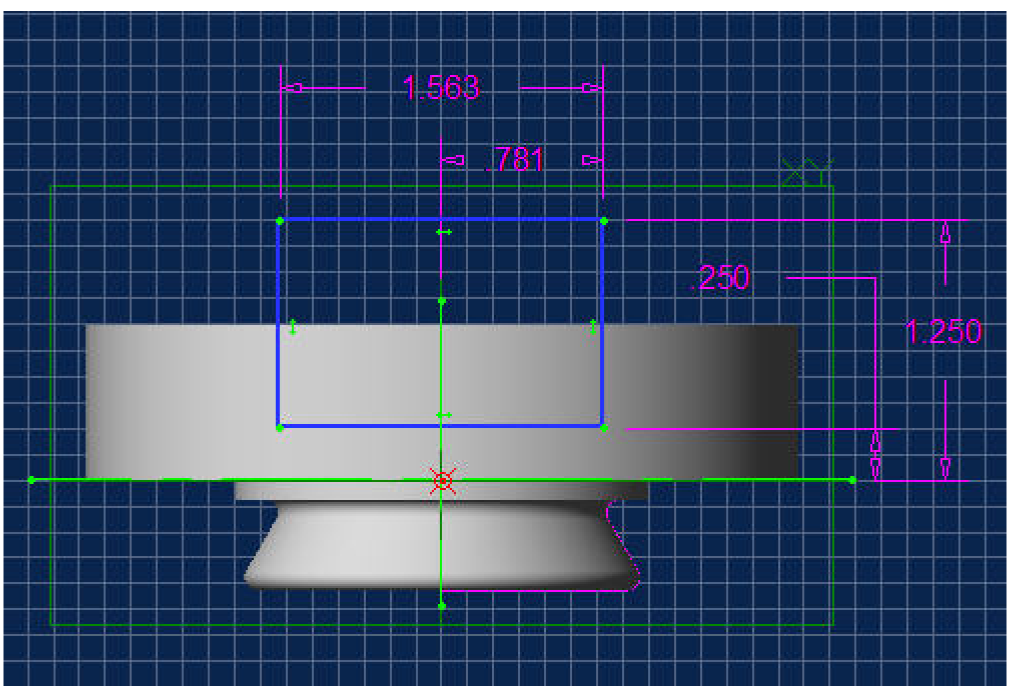
Create and constrain the sketch shown below using the XY plane as your geometric anchor.

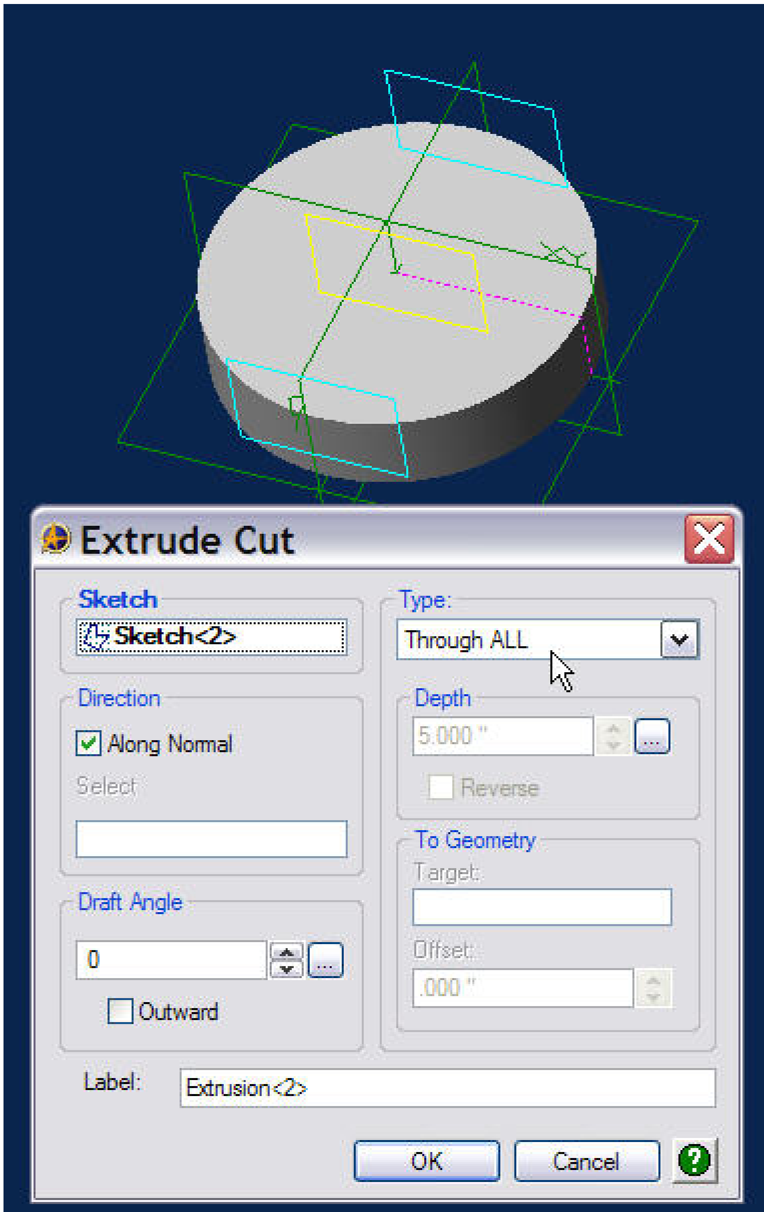


Click on the 'Revolve Boss' command and rotate the sketch around the Y axis to create the raw part shown.



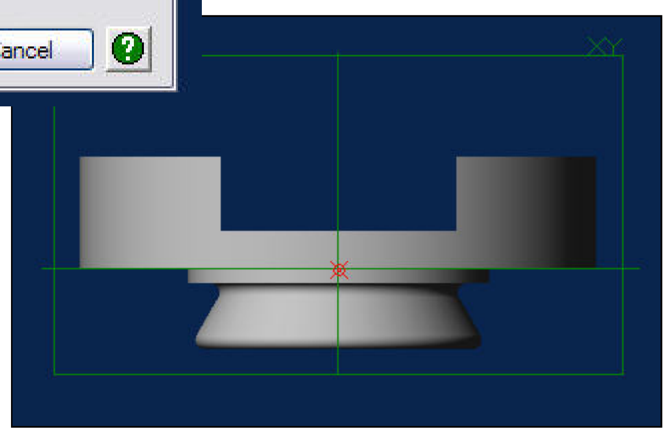
Next, we'll create the holder slot in the tool holder. Create the sketch shown below, again using the XY plane as our geometric anchor and extruding the shape using the Mid-plane command.





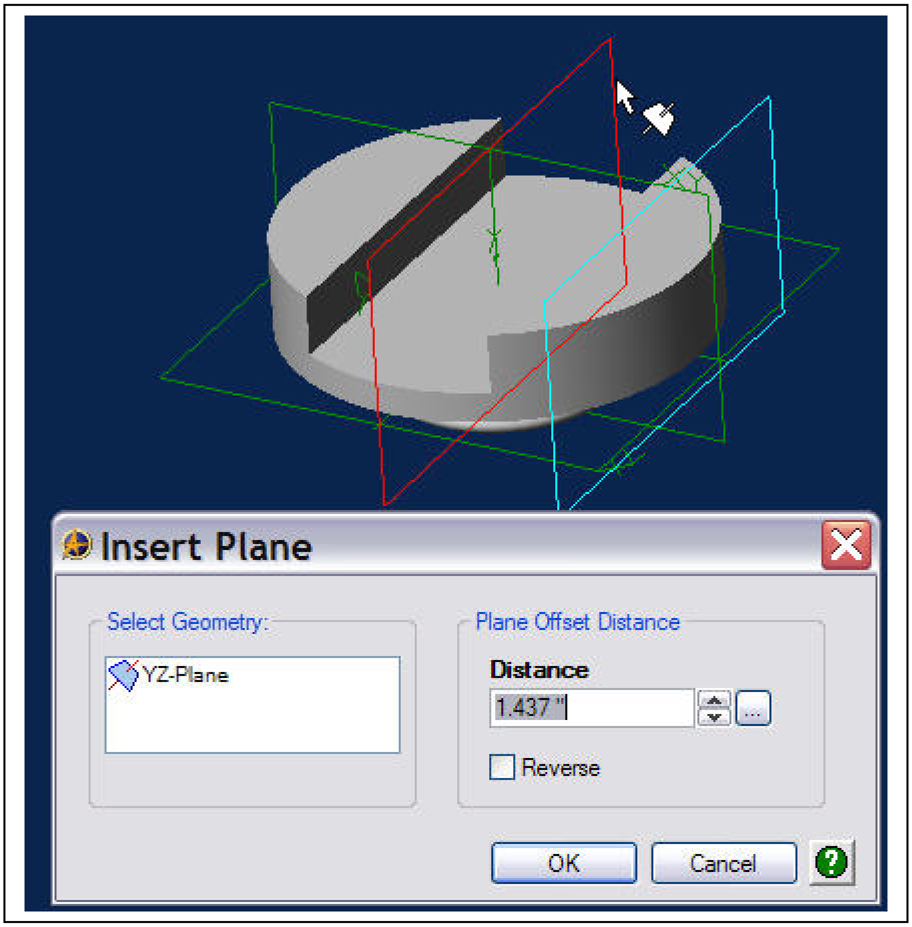
Use the ‘Through All’ modifier when performing the cut.

The part should now look like this.



he elements of symmetry and patterning should always be considered when designing parts. Utilizing an approach that considers these can save a lot of design time up-front and make editing of the part much easier in the future. Look for ways to optimize your use of both. Extruding the cut from the Mid-plane creates an element of symmetry that could make future modifications of the Tool Holder easier.

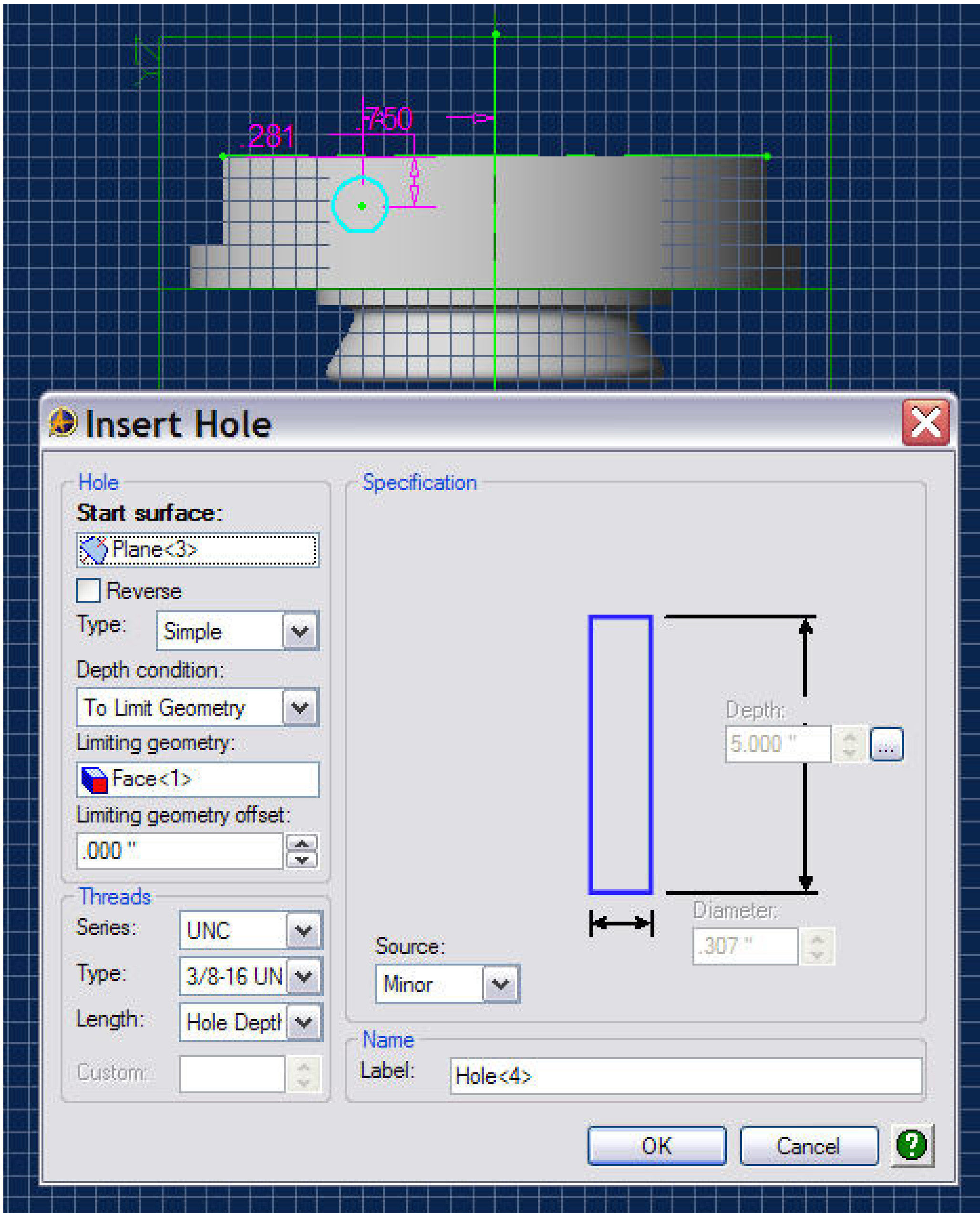
The next step in creating the Compound Tool Holder will be to insert the plane that will serve as the anchor for the four drilled, tapped, and counter bored holes used to hold the clamp screws. Click on Insert > Plane, Select the YZ plane, and enter 1.4375 as the distance. Click the 'Reverse' box if necessary to create the plane shown below.

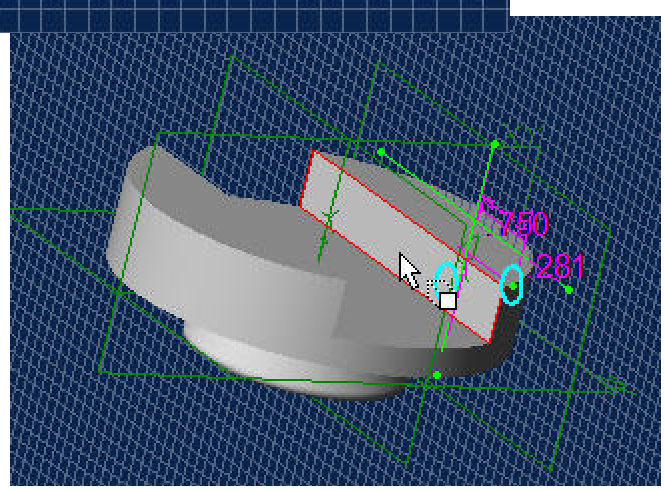


Select the newly inserted plane, and click the 'Hole' icon.

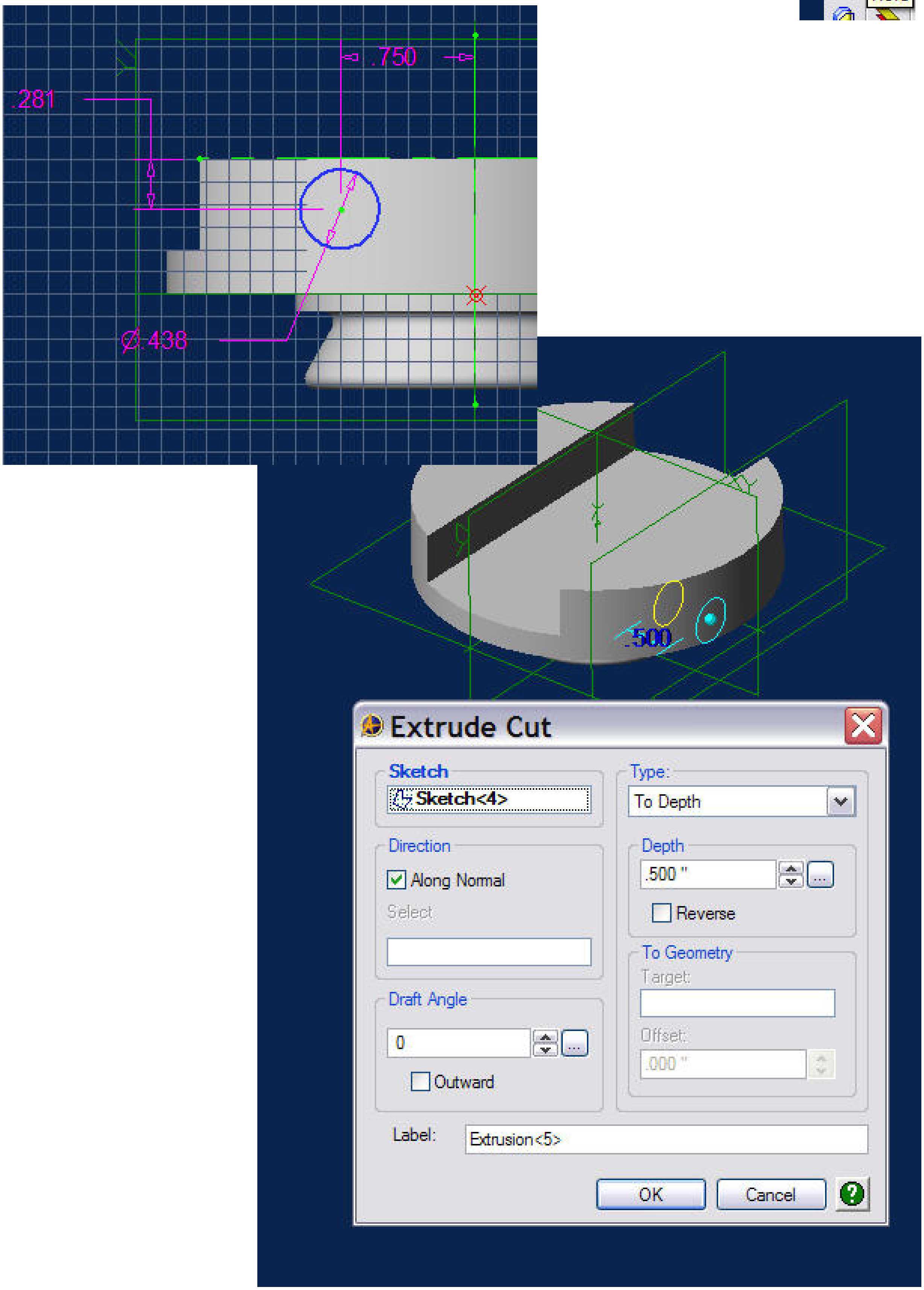
Alibre will drop into sketch mode.

Change the hole parameters to those shown on the next page and select the near inner face of the extruded holder slot as the geometrical limit for the hole and click OK.

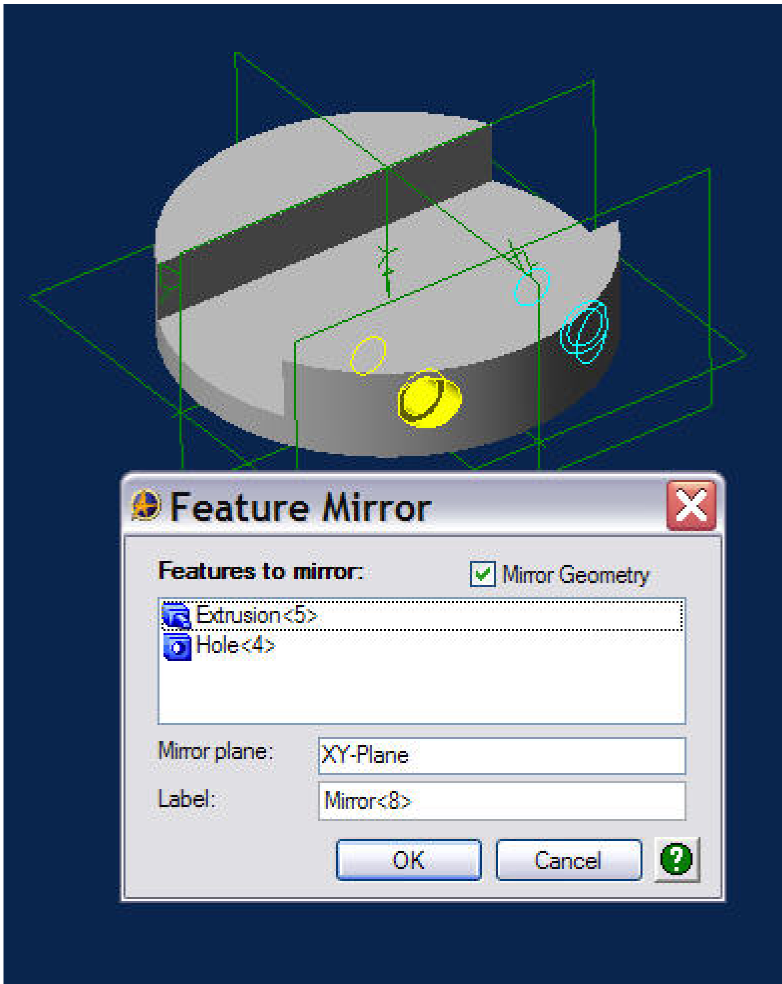


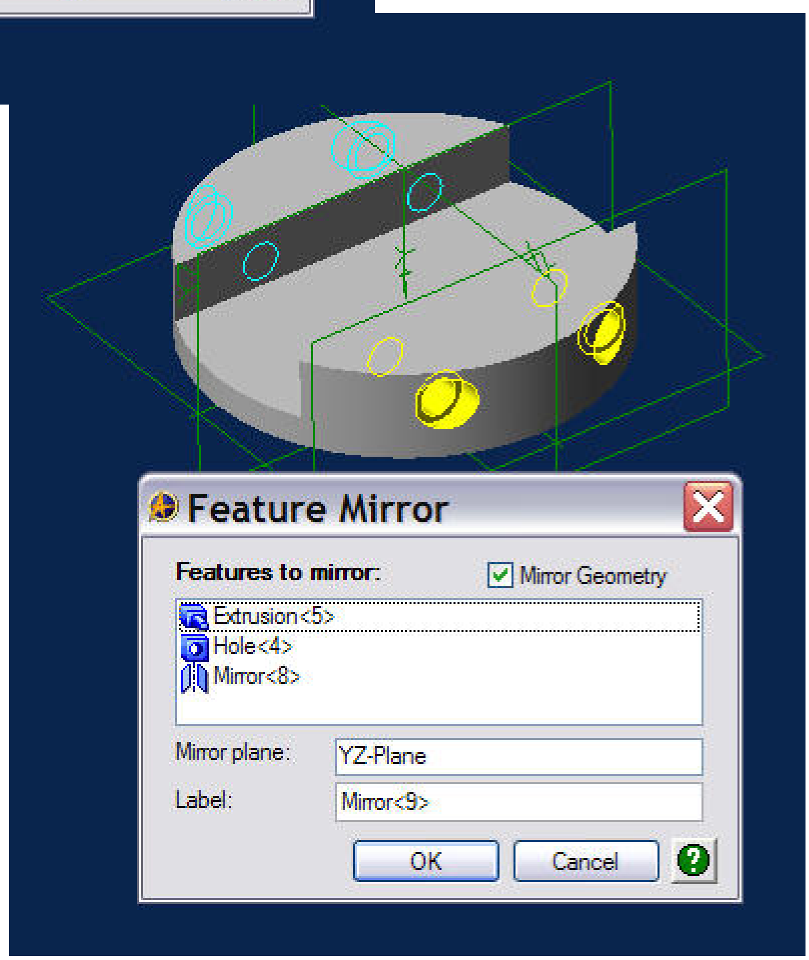


Select the plane you used to create the previous hole and click on the 'Hole' icon. Create the sketch shown and then extrude it to a depth of .500 in to create the counter bore feature of this hole.

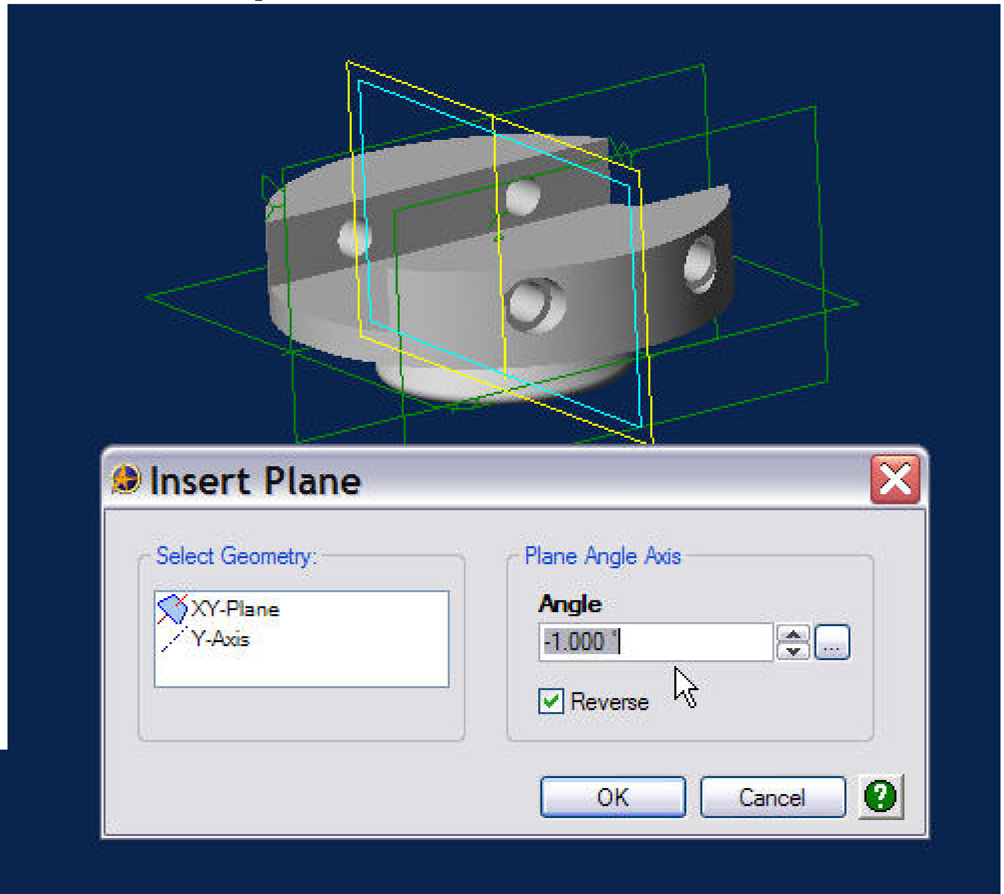


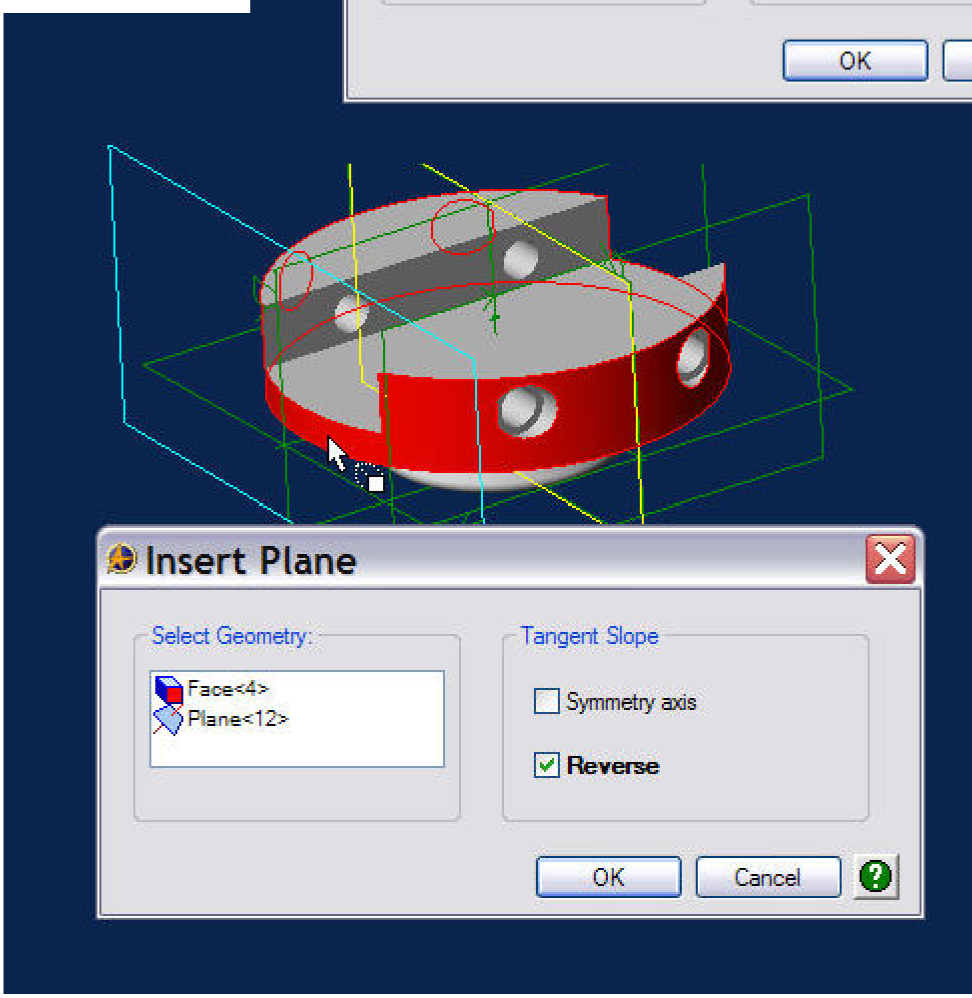
Use the mirror command to create the remaining holes.





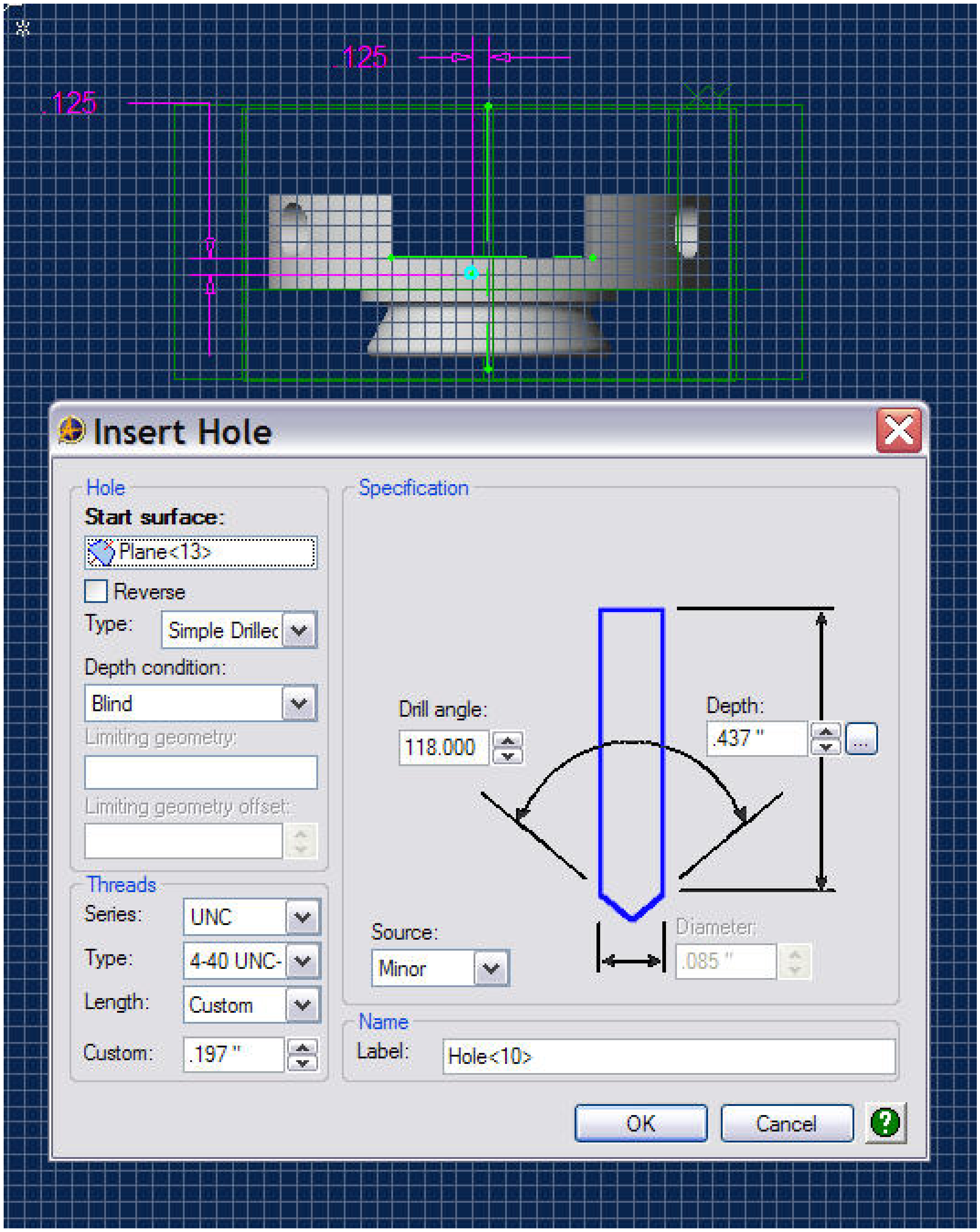
We'll now insert two more planes, the first as a reference for the second, in order to create the holes for the attachment of the protractor scale. Use the Insert Plane command and select the XY plane as the reference plane and the Y-axis as the axis of revolution. Enter -1.000° in the Angle text box and click OK.





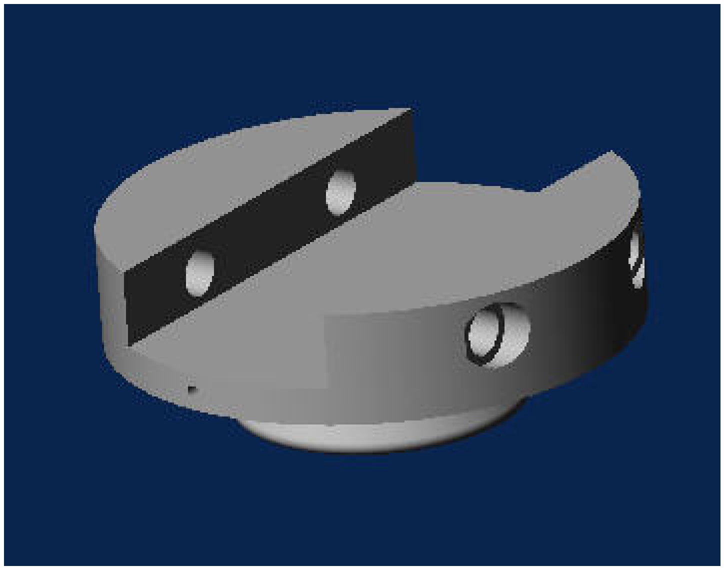
Next select the plane you just inserted, use the Shift key and select the outer face of the tool holder, select Reverse if necessary, and click OK.

Select this plane and click the Hole icon.



Change the hole parameters to those shown and constrain it per the dimensions shown and click OK. Mirror the hole feature about the XY plane to complete the creation of the Compound Tool Holder.

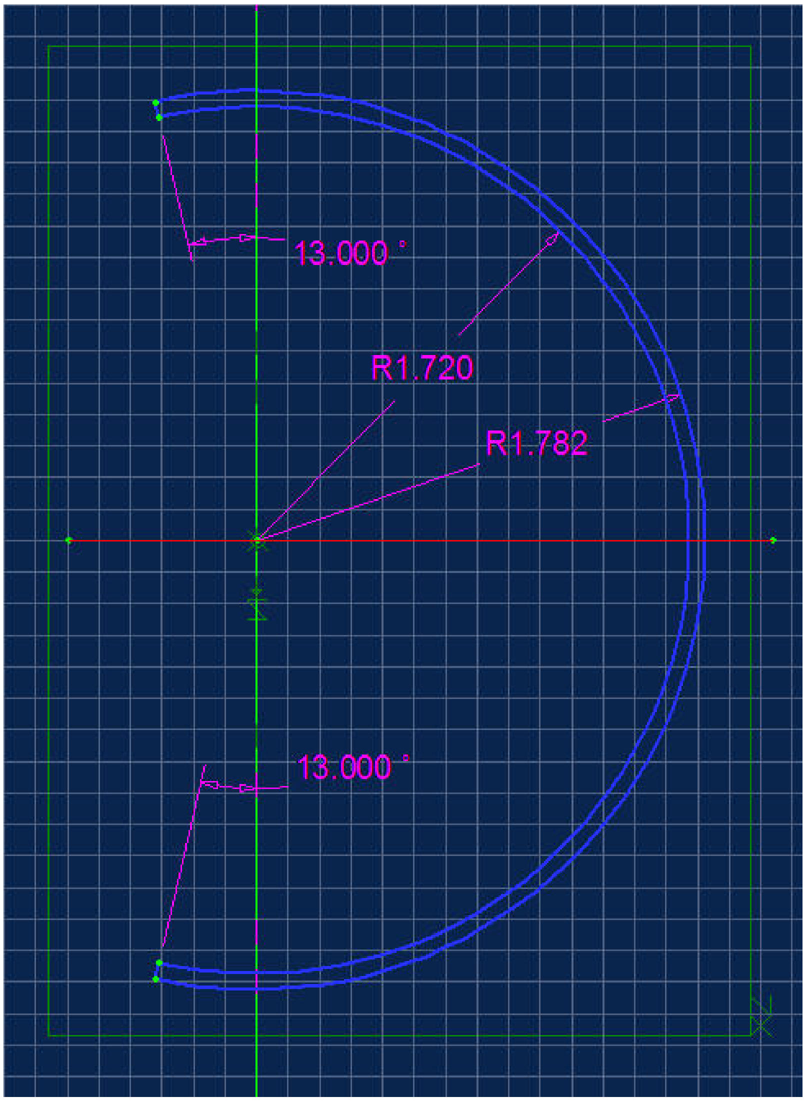
Your part should now look like this. File it as part number 82000350.



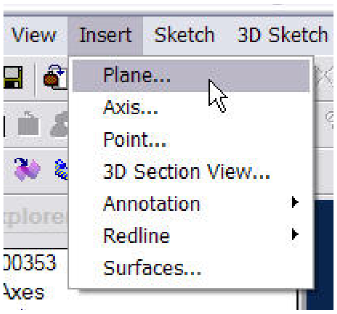
Chapter 9 - Designing the Any Angle Tool Vise – The Protractor Scale

The Protractor Scale is made from aluminum strip, .25in wide by 6.192in long (developed length). The inside radius of the scale is taken from the outside radius of the Compound Tool Holder (see Chapter YY, drawing XX Compound Tool Holder). For our purposes the Protractor Scale will be modeled as a partial tube.

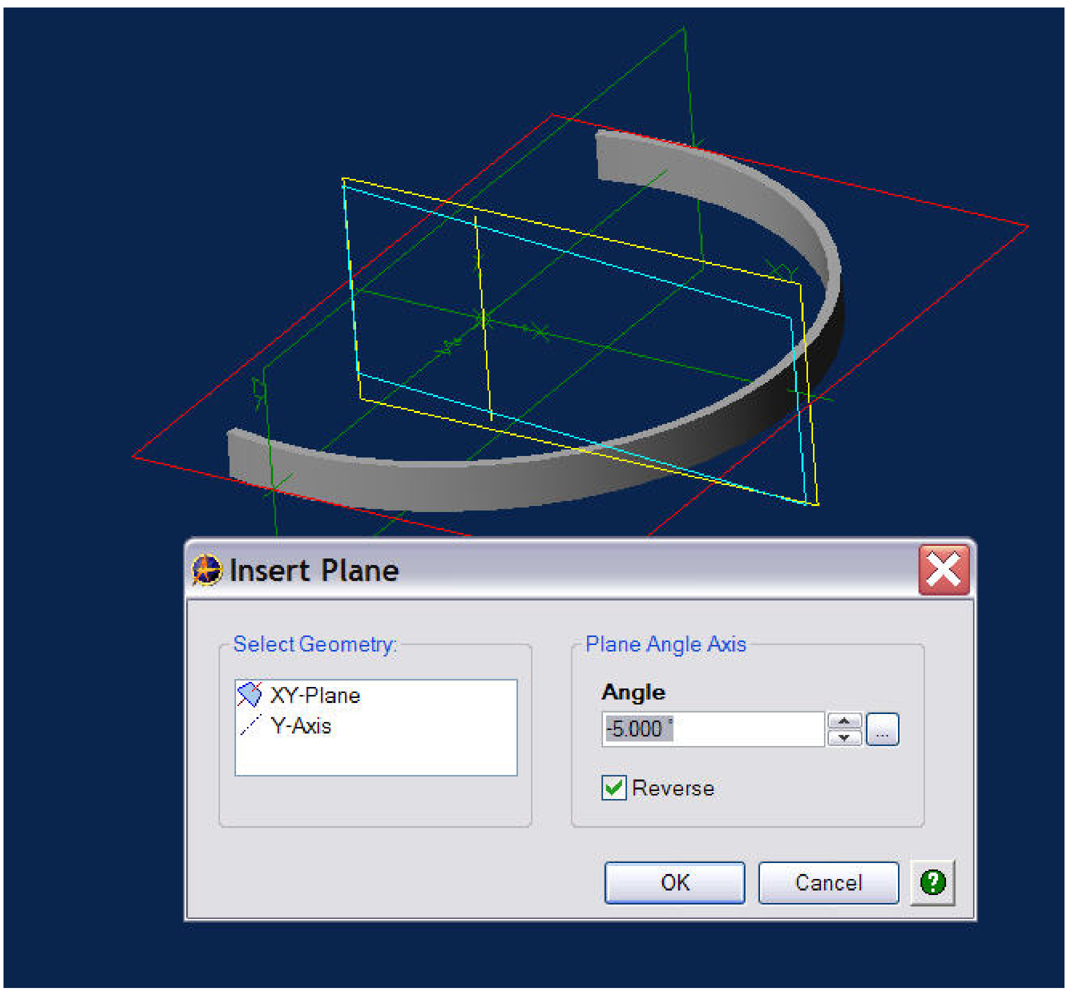
Create and constrain the sketch shown below using the ZX plane as your geometric anchor. Extrude the sketch .25in to create the protractor band.



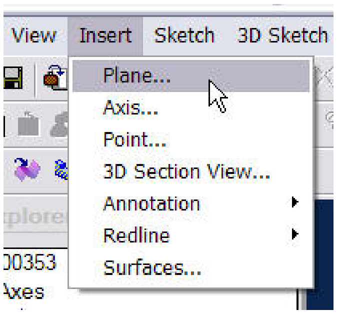
Your part should now look like the one below. The next step is to add the rounded ends to the band, but to do this we'll have to add two planes, the first based on the XY plane but rotated 5 degrees, and the second based on this newly created plane but tangent to the outer surface of the protractor band. Create the first plane by clicking on the Insert menu and selecting 'Plane.



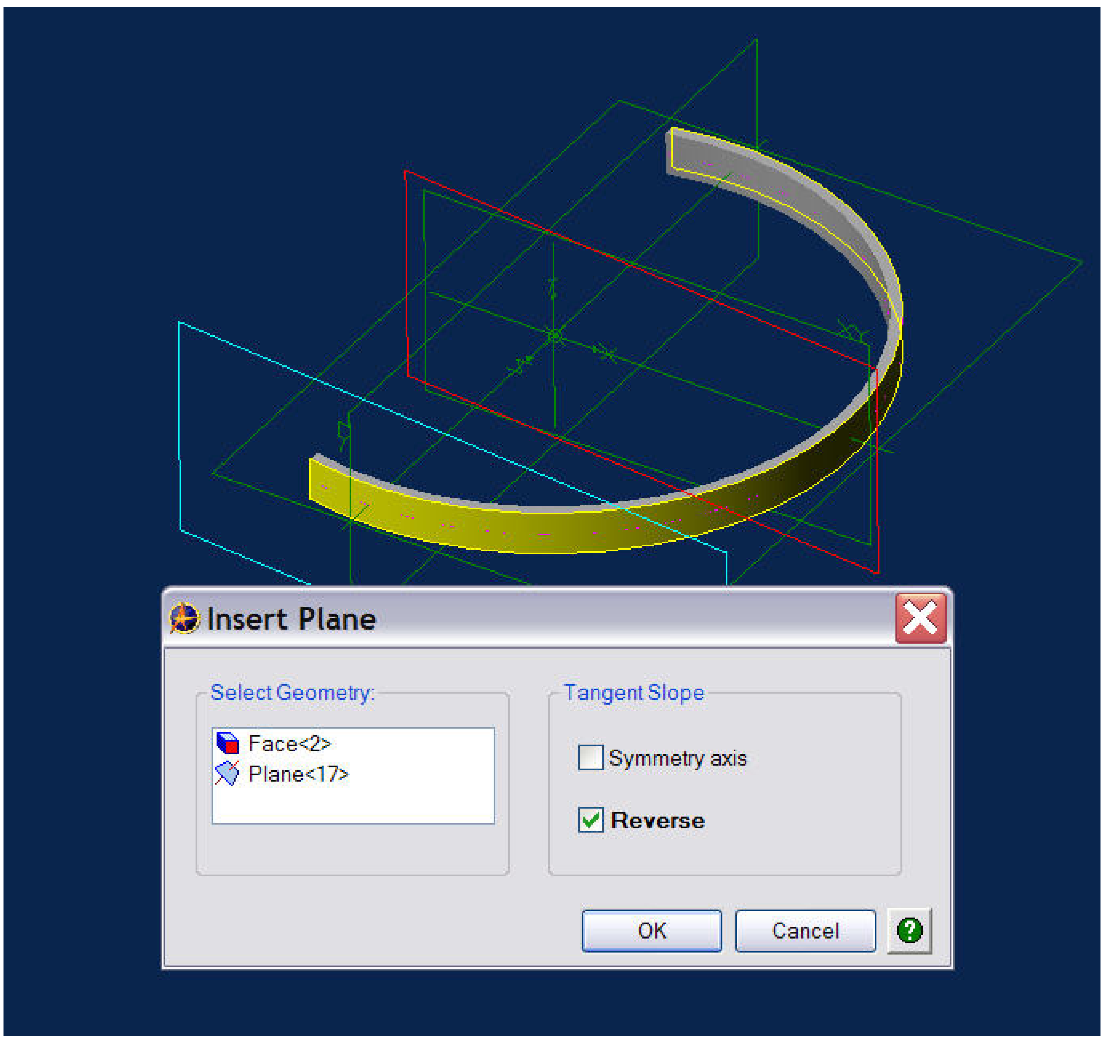
As your selected geometry, click the XY plane and then the Y axis. Enter 5.000 in the Angle field, select 'Reverse' and click OK.



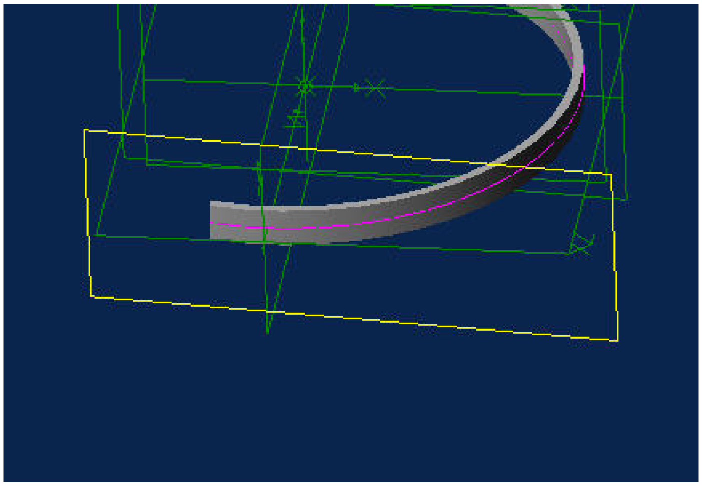
Next we'll create the plane that will be used for the basis for the sketch that will define the rounded ends of the protractor scale. Again click on the 'Insert' menu tab and select 'Plane'.

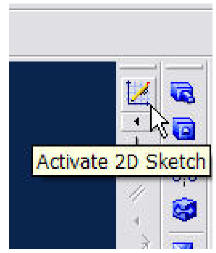


As the selected geometry select the newly created plane, then use the 'Shift' key and select the outer surface of the protractor band (highlighted in yellow), click the 'Reverse' check box and then click OK.

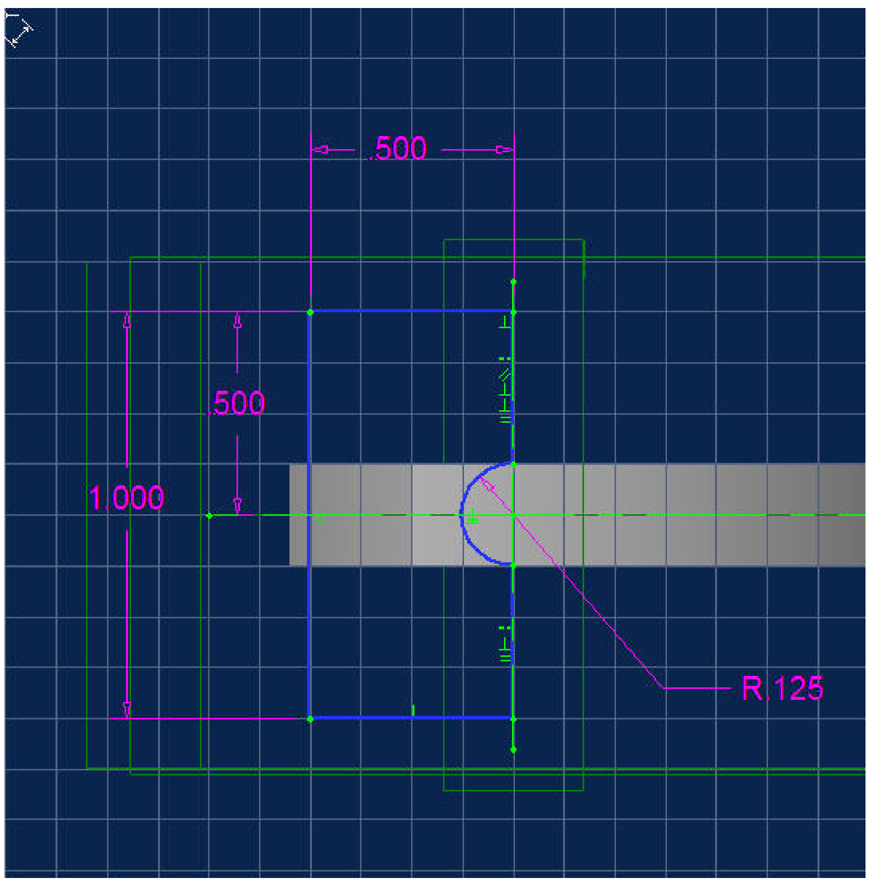


To create the rounded ends of the band, click on the plane you just created and then click on the 'Sketcher' icon.

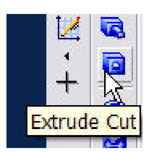




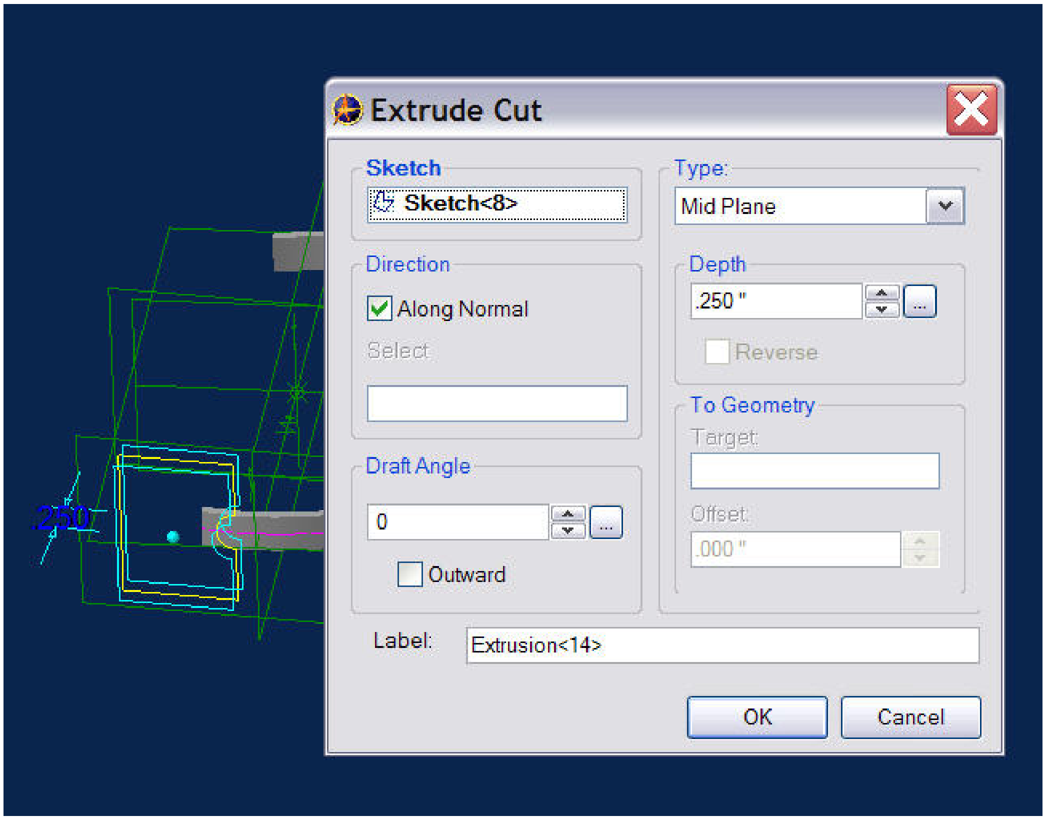
Sketch and constrain the profile shown below.



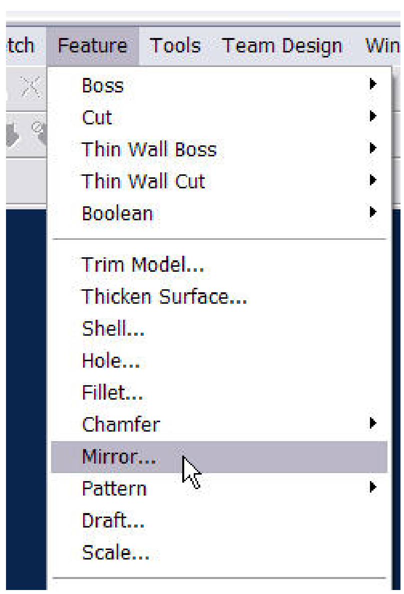
Click on the 'Extrude Cut' icon.

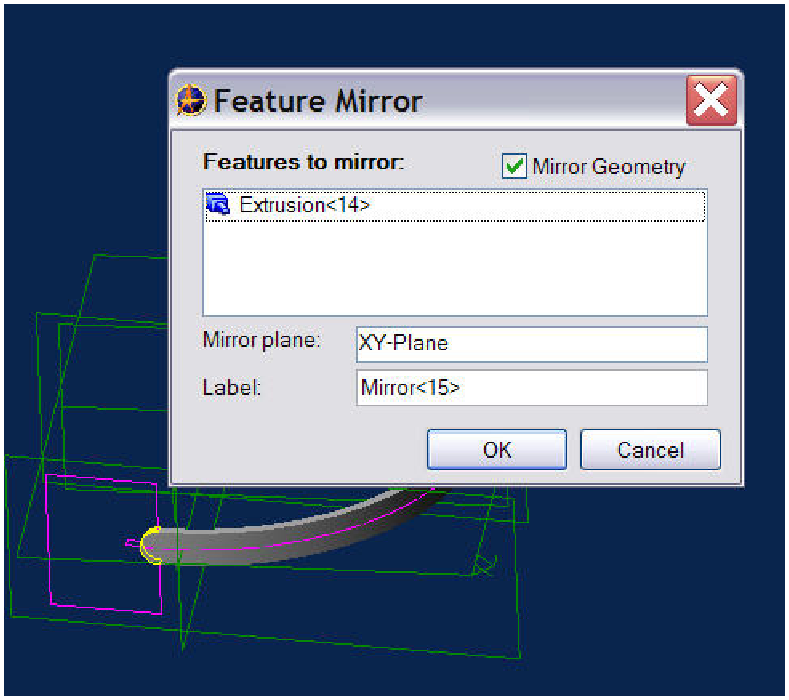


In the ‘Type’ scroll down menu select ‘Mid Plane’. Enter .250 in. in the depth text block, make sure the ‘Along Normal’ check box is checked, then click OK.



Click on the ‘Feature’ menu tab and select ‘Mirror’.

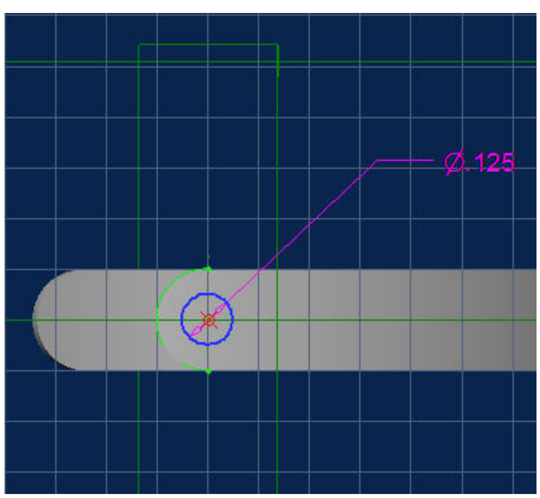


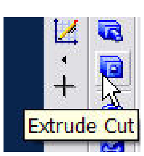


When the Mirror' feature dialogue box opens, fill in the 'Features to mirror' field by clicking on the Extrusion in the Design Explorer panel that corresponds to the feature you're going to mirror. To fill in the 'Mirror plane' field click the XY plane (either in the Work Area or in the Design Explorer panel), then click OK.

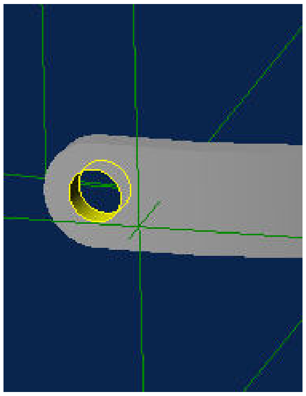
The elements of symmetry and patterning should always be considered when designing parts. Utilizing an approach that considers these can save a lot of design time up-front and make editing of the part much easier in the future. Look for ways to optimize the use of both.

The next step in creating the Protractor Scale will be to insert the punched holes used to fasten the scale to the Compound Tool Holder. Select the plane used to create the rounded band ends, and create the sketch below, using the 'Concentric' constraint to locate it in the center of the rounded end.



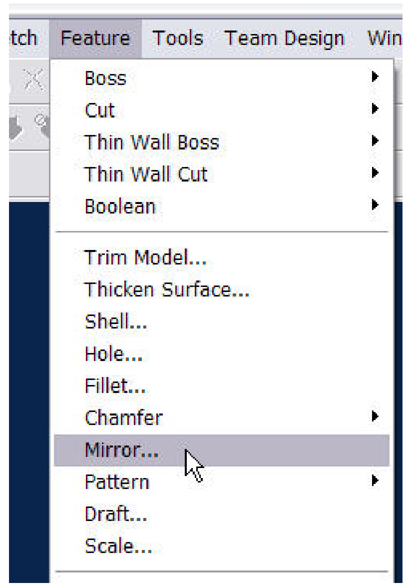


Click on the 'Extrude Cut' icon, select '

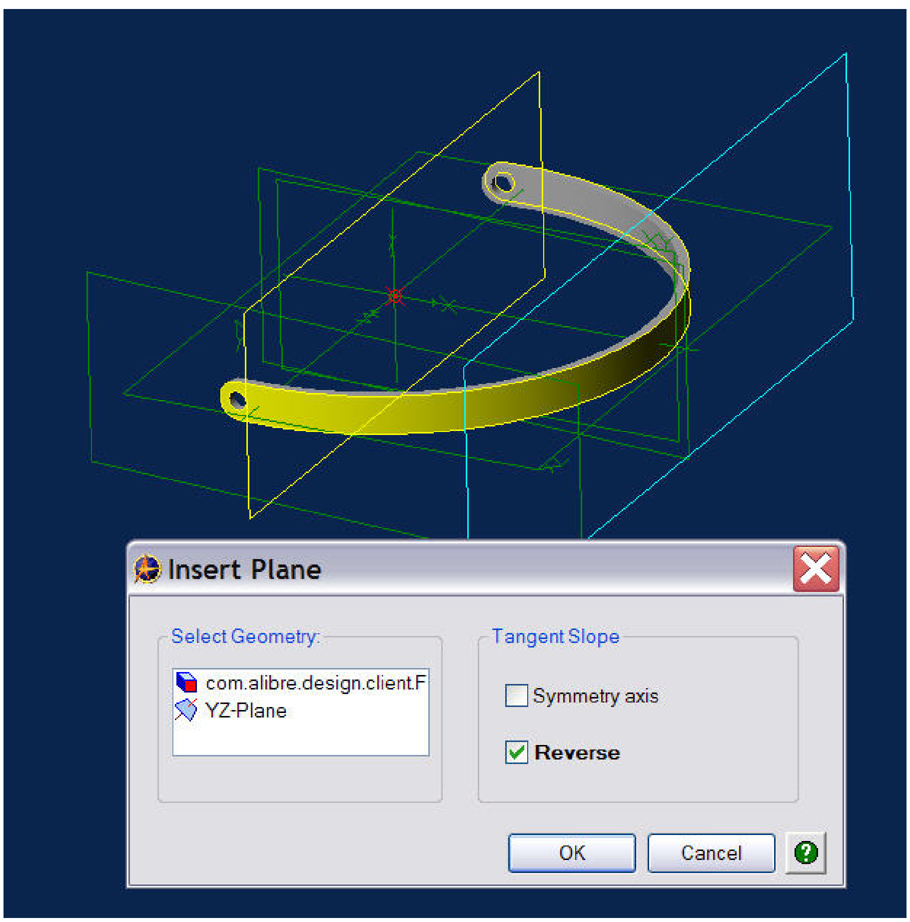


The part should now look like the figure at the left.

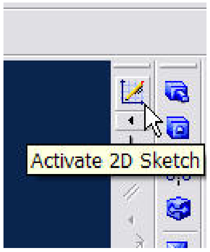
Again use the 'Mirror' feature to create the hole in the other side.



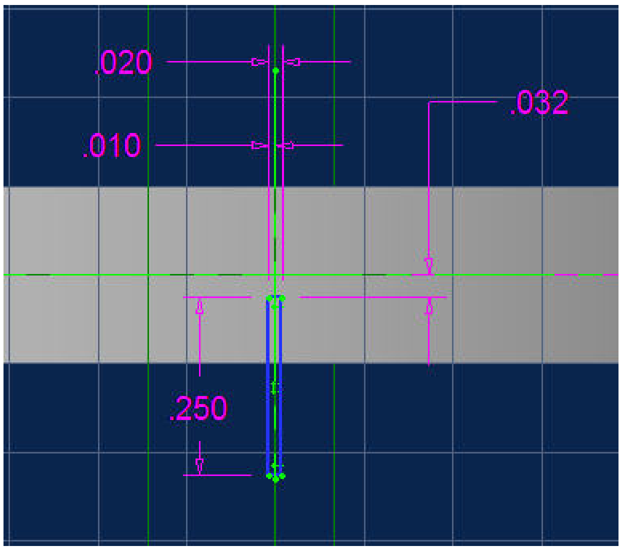
You could stop here, and use the band as an acceptable representation of the protractor scale, but adding the angle markings will give you added practice in creating new planes and using circular patterns.



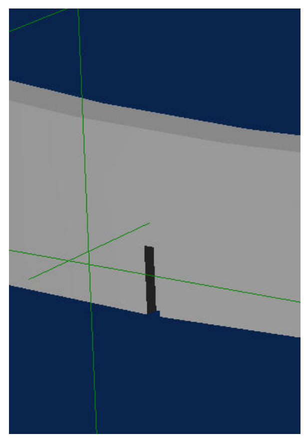
Click on Insert > Plane, Select the YZ plane, using Shift >click, select the face of the tool holder and click the Reverse box if necessary to create the plane shown. This plane will be the anchor for the 0 degree mark on the scale.



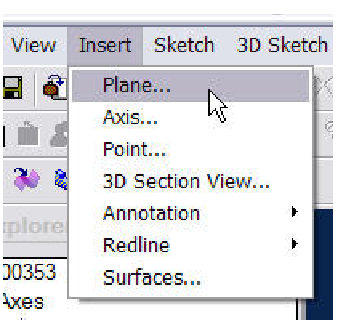
Select the newly inserted plane, and click the Sketch icon. Alibre drops into sketch mode.



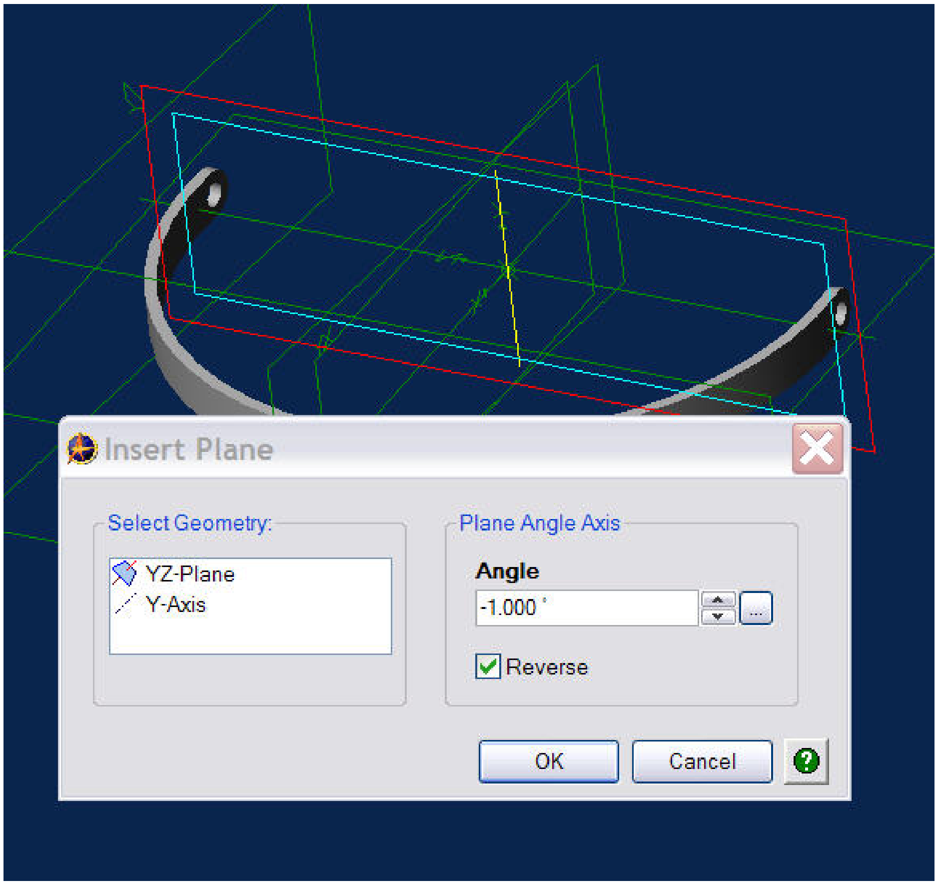
Create and constrain the sketch shown, then click on the 'Extrude Cut' icon and edit the depth to .020mm, select 'Reverse' if necessary.



Your part should now look like this.

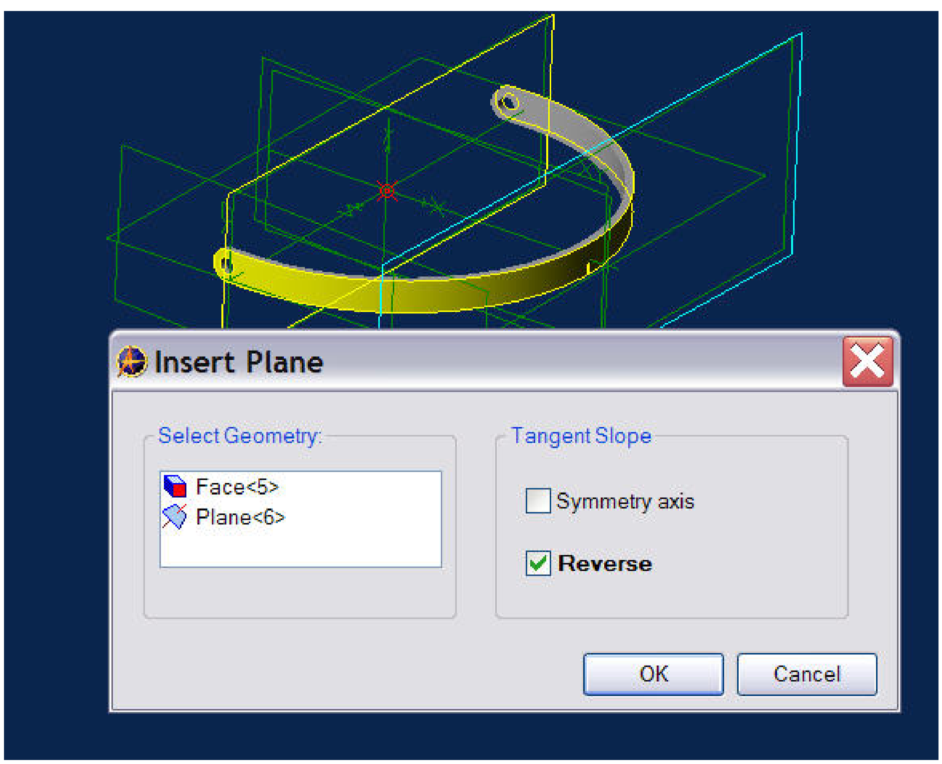


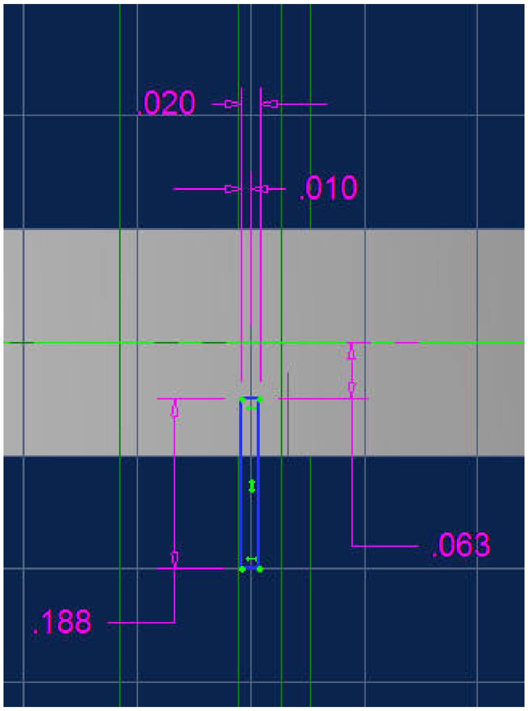
We'll now insert two more planes, the first as a reference for the second, in order to create the cut for the 1 degree mark on the protractor scale. Click on Insert > Plane.



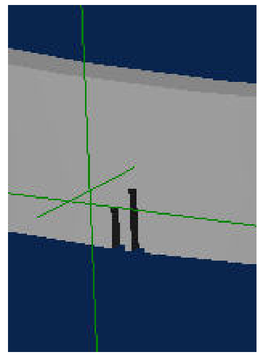
Select the YZ plane, and the Y axis. Enter -1.000 degrees and select 'Reverse' if necessary to insert a plane at 1 degree clockwise rotation from the YZ plane about the y axis.

Click on Insert > Plane again, Select the newly created plane and using Shift >click, select the face of the protractor scale and click the Reverse box if necessary to create the plane shown. This plane will be the anchor for the 1 degree mark on the scale.



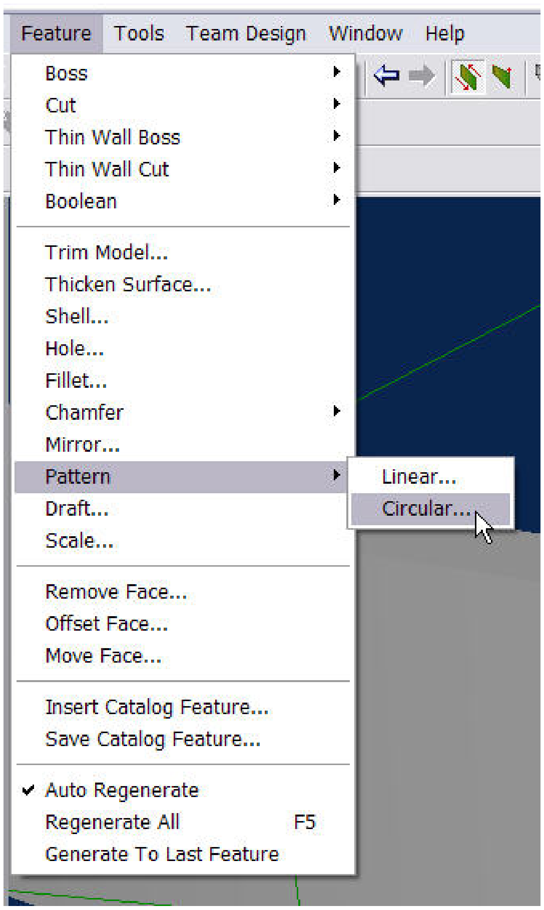


Create and constrain the sketch shown, then click on the 'Extrude Cut' icon and edit the depth to .020mm, select 'Reverse' if necessary

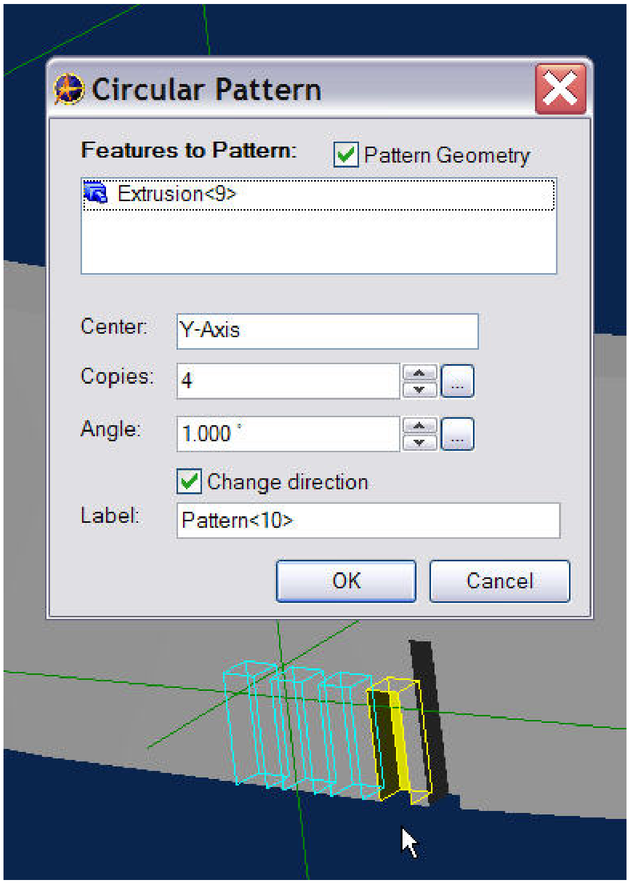


Your part should now look like this

You have now created the basic features for all remaining degree marker lines on the protractor scale.

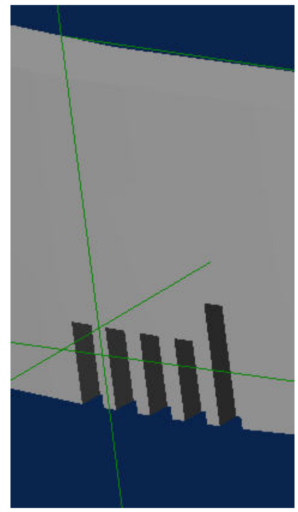


Click on the 'Feature' tab in the top Menu bar, select 'Pattern' then select Circular.

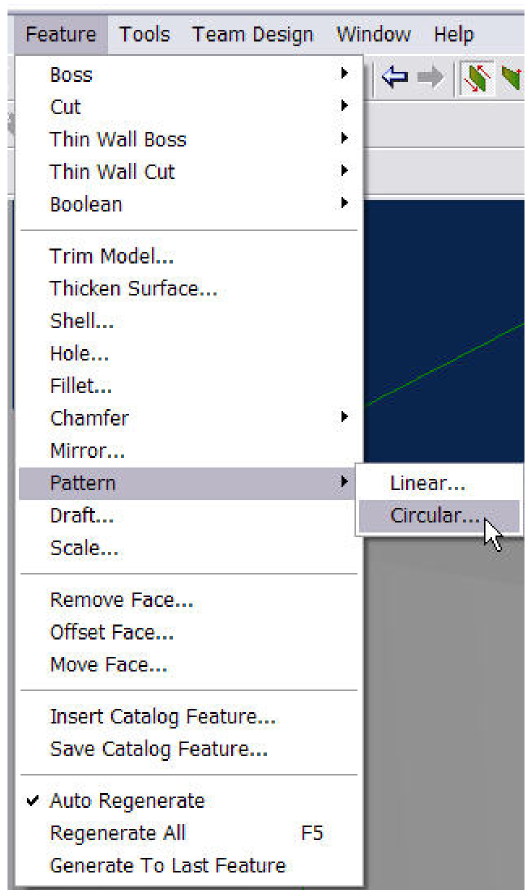


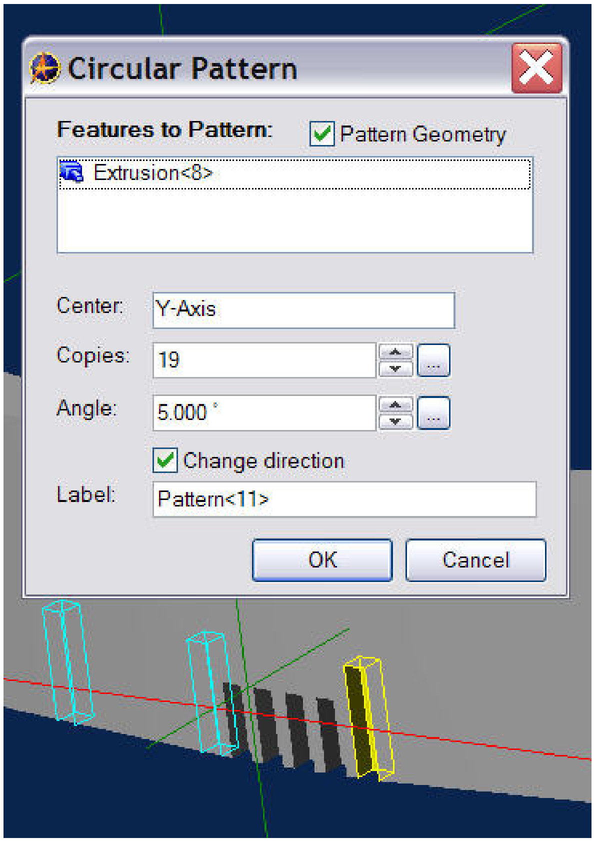
When the ‘Circular Pattern’ window opens, select the 1 degree mark feature (in this example Extrusion 9) as the ‘Feature to Pattern’, select he ‘Y axis’ as the center, enter 4 for the number of ‘Copies’, 1.000 degree as the ‘Angle’, check the ‘Change direction’ box if necessary, then click OK.

Your part should now look like this.



Again, select the ‘Circular Pattern’ option from the ‘Feature’ drop down menu.

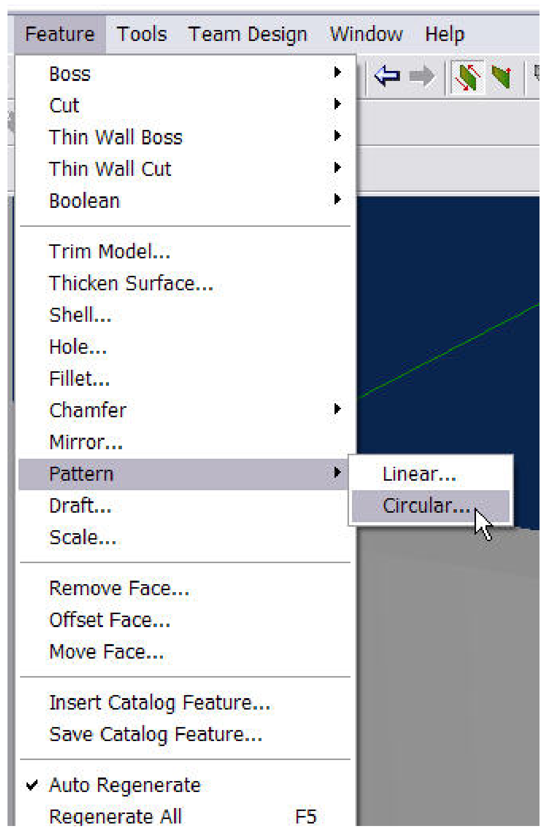


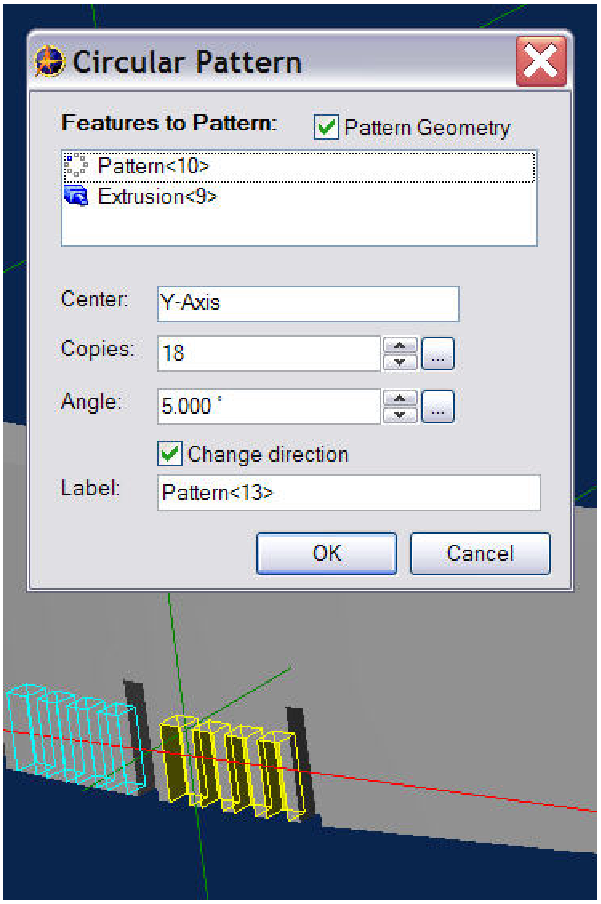


When the ‘Circular Pattern’ window opens, select the 0 degree mark feature (in this example Extrusion 8) as the ‘Feature to Pattern’, select he ‘Y axis’ as the center, enter 19 for the number of ‘Copies’, 5.000 degrees as the ‘Angle’, check the ‘Change direction’ box if necessary, then click OK. This will create the 5 degree marks from 0 to 90 degrees on the scale.

The next step is identical to last with the exception of the feature you select to pattern. In this case it is the pattern of four 1 degree marks you created by selecting Feature 9.

Again, select the Circular Pattern option from the ‘Feature’ drop down menu.

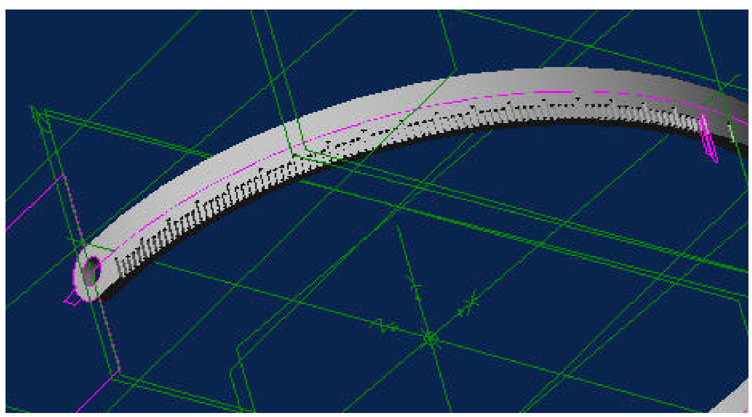




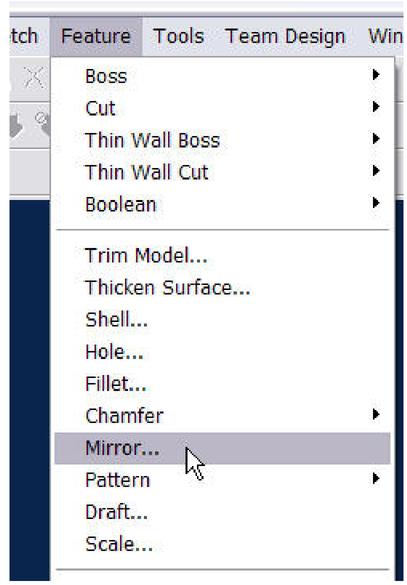
When the ‘Circular Pattern’ window opens, select the pattern that describes the four 1degree marks as the ‘Feature to Pattern’. Note that the Extrusion associated with this particular pattern is automatically selected. Next, select he ‘Y axis’ as the center, enter 18 for the number of ‘Copies’, 5.000 degrees as the ‘Angle’, check the ‘Change direction’ box if necessary, then click OK.

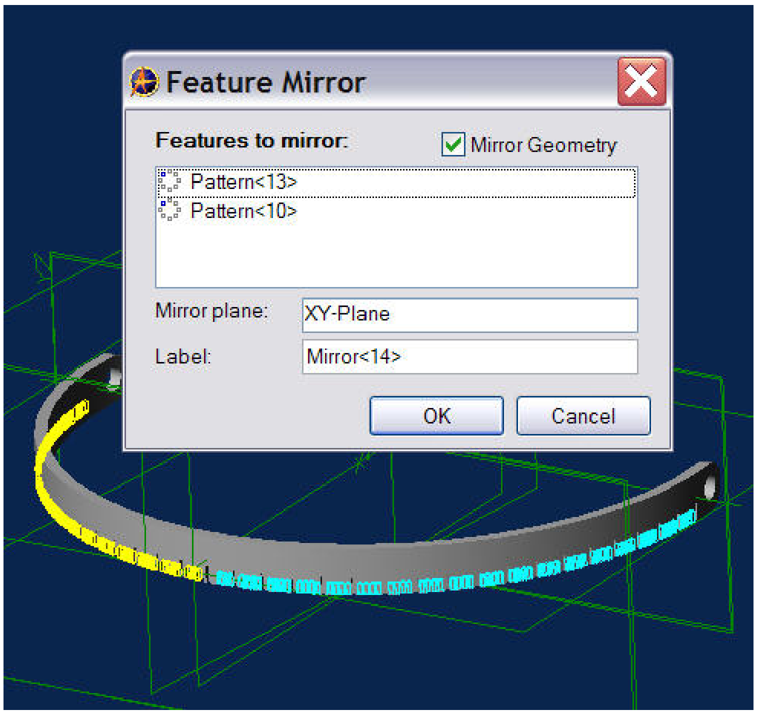
This will create the 1 degree marks between 0 and 90 degrees on the scale.

Your part should now look like this.



To finish the protractor scale part, select ‘Mirror’ command under the ‘Feature’ tab on the toolbar. Select the patterns describing the major and minor degree marks (in this case Patterns 10 and 13), select the XY plane as the Mirror plane, and click OK.





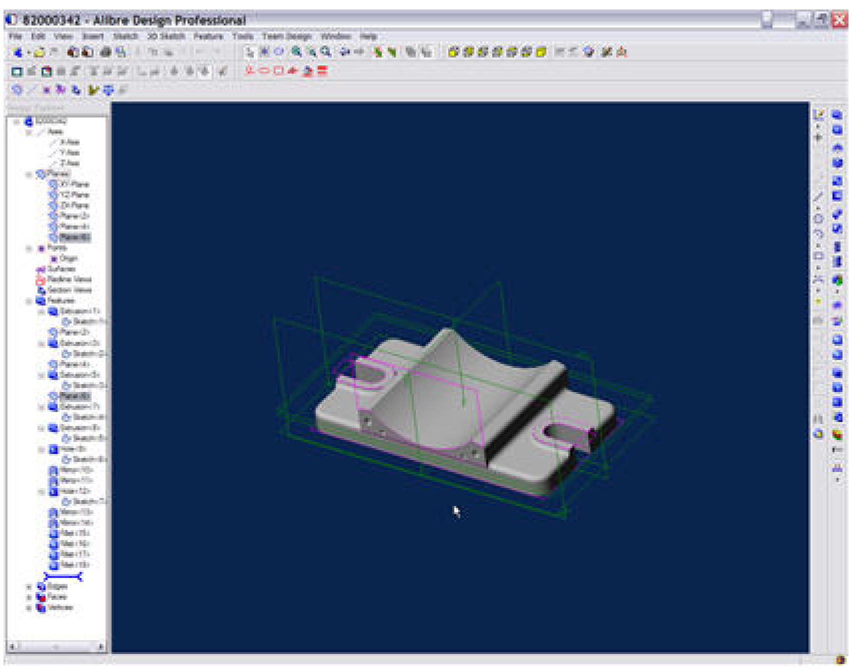
Your part should now look like this. File it as part number 82000350.

Chapter 10 - Documenting the Any Angle Tool Vise – Creating Drawings

Alibre Design contains a 2D drawing application that could be used as a standalone drawing tool, comparable in functionality to many other drafting tools and packages on the market today. You could, use the drawing package to document your design without ever touching on the 3D solid modeling functionality; the only question would be why would you want to do anything as silly as that? The drawing package contained in Alibre is connected directly to Alibre's 3D modeling environment, and the part or parts you create there generate most if not all of the 2D information you need to document your design(s) fully, including dimensions and B.O.M.'s. Changes in the 3D model are instantly reflected in the drawing, and changing a driving dimension in the drawing will be reflected in the 3D model. Use of the drawing package will drastically reduce your documentation time and provide you with a means of developing a set of drafting standards for your company or a way to duplicate those you already have in a very short time. Inherent in the Alibre Design drafting package, is the ability to import several different types of 2D drawing data including AutoCAD DXF and DWG files as well as STEP and IGES files. As with all conversions, some follow-up modification of the imported data might be necessary.

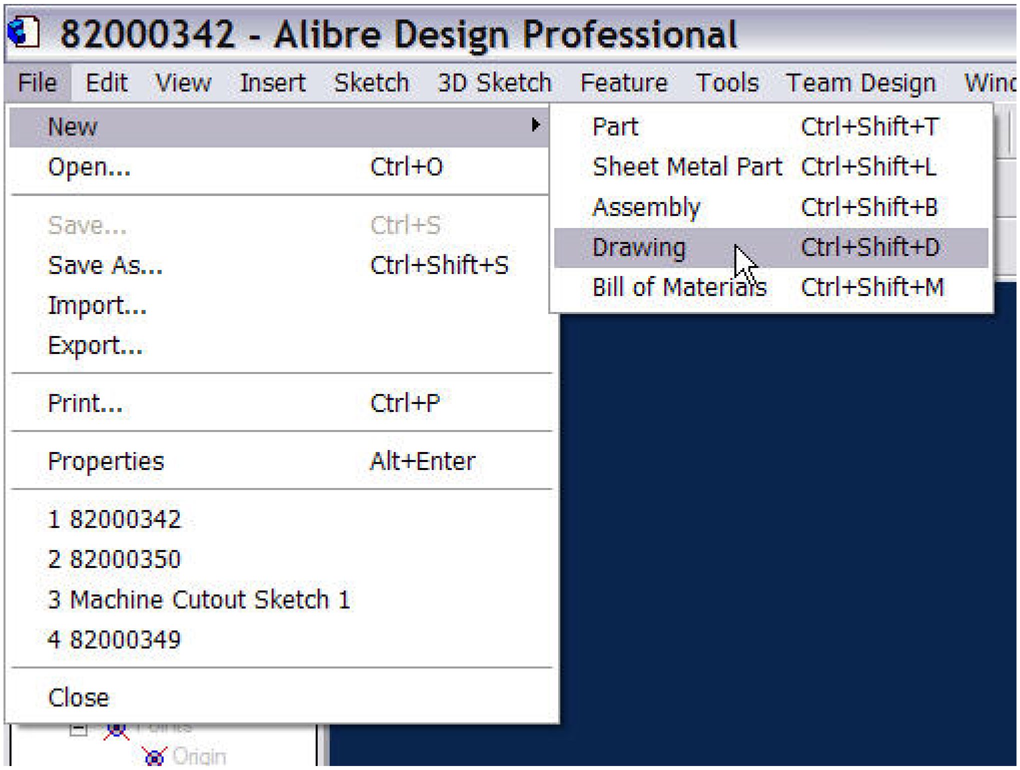
All of the drawings you used as reference for the parts you created in the previous exercises were created and detailed using the Alibe drafting package. In this exercise we'll document the Saddle base, dimension it, detail it, and get it ready for final release, all while exploring the various aspects of the drafting package. As with all other areas of the Alibre design package, practice and constant use will result in increased efficiency, so don't hesitate to push Alibre to its limits, which are constantly expanding. Let's get started.

Open the Saddle Base part you created in Chapter 1.

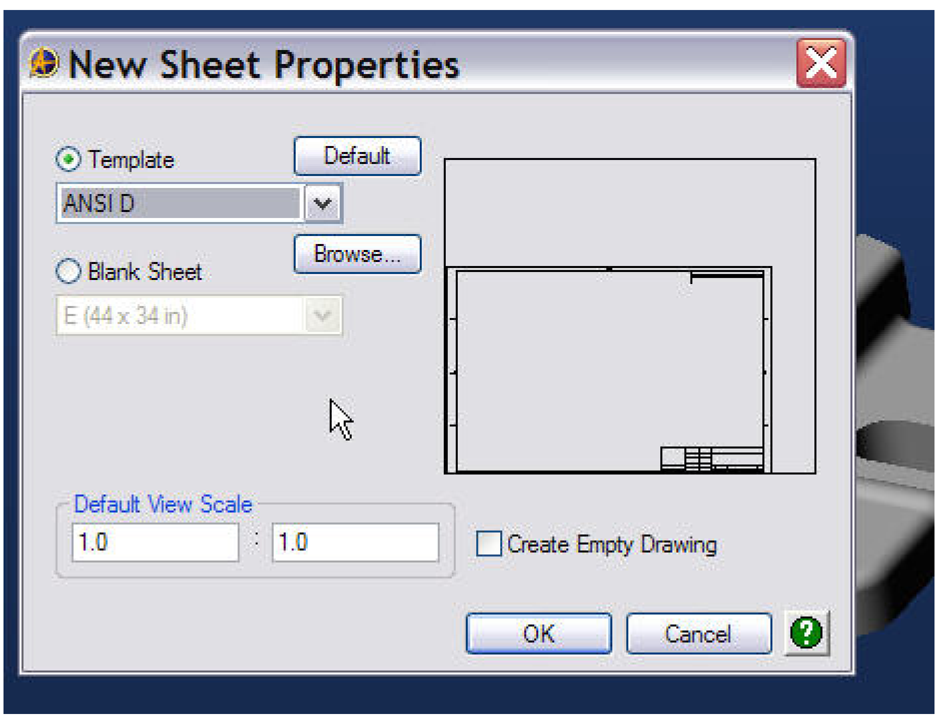


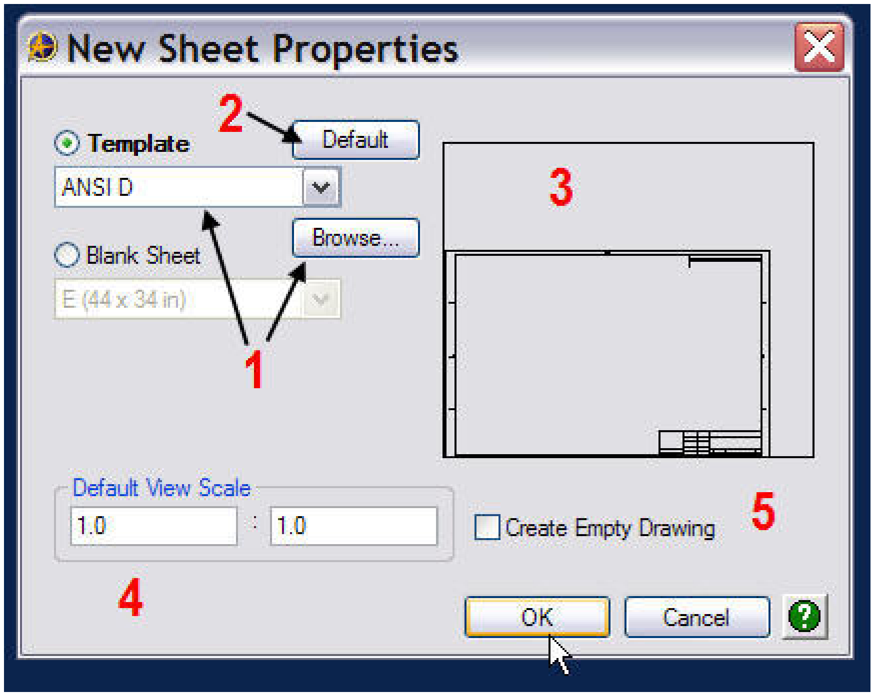
Although it isn't necessary to have a part active in order to create a drawing, we'll open the Saddle base part to show how the two environments relate to one another. The alternative to activating a part before you open a new drawing will be shown later in this exercise.

Click on File, select New>Drawing.



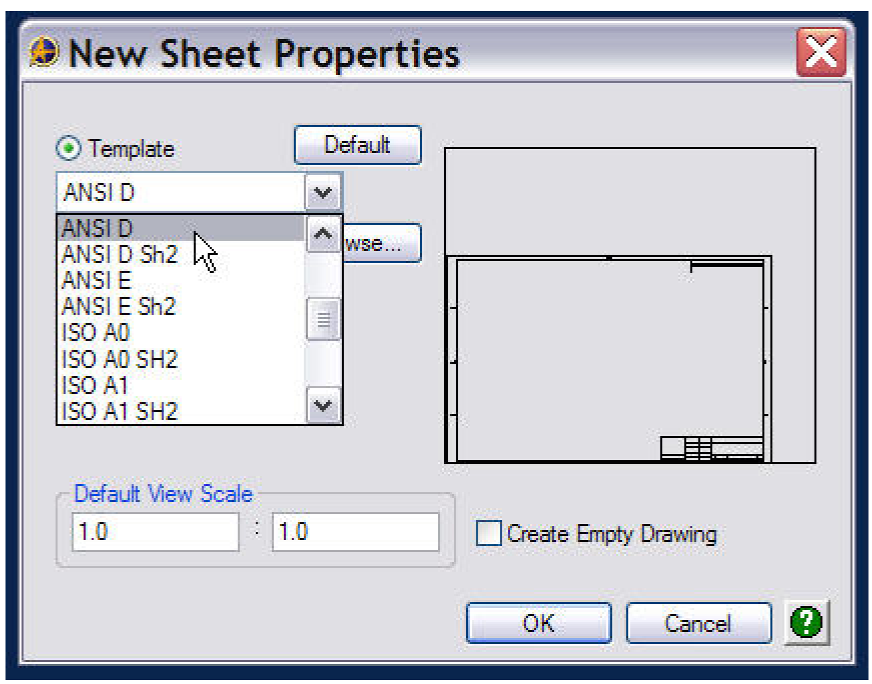
The New Sheet Properties panel will open.

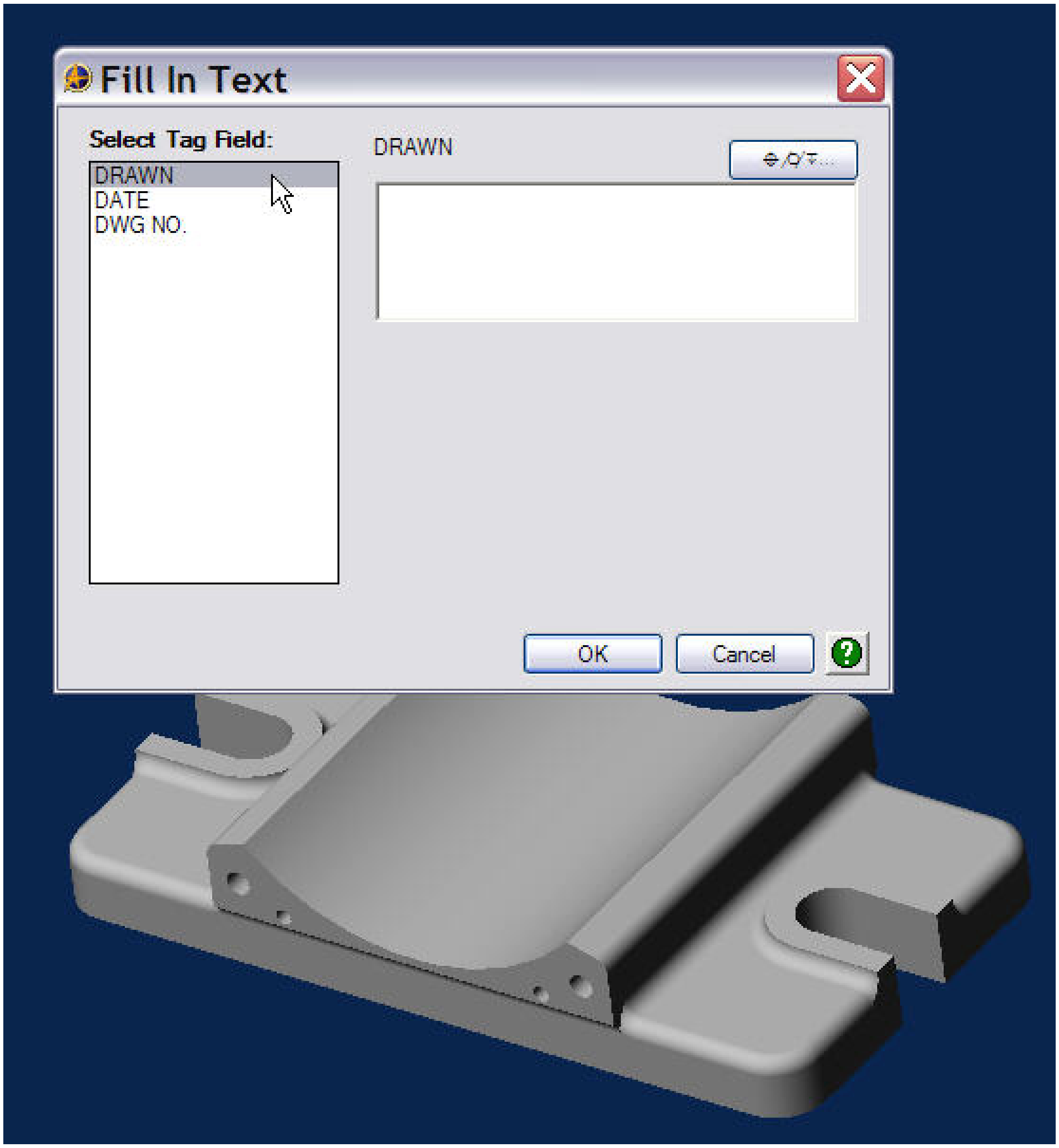




The New Sheet Properties Panel is where you'll select many aspects of the new drawing.
1. You can select a pre-existing Template (in either of two styles, ANSI, or ISO and a range of sizes from A to E or A4 to A0), or insert a custom template of your own creation (Click Browse and select your template form the appropriate file or repository location.), or insert a blank sheet.
2. Reset your custom selection to Default and select from the standard templates.
3. See your selection in the 'Preview' window before committing to your selection.
4. Set the View 'Scale'.
5. Create an 'Empty' drawing, will create a blank drawing.

For Saddle Base, we'll select an ANSI D as the drawing template.

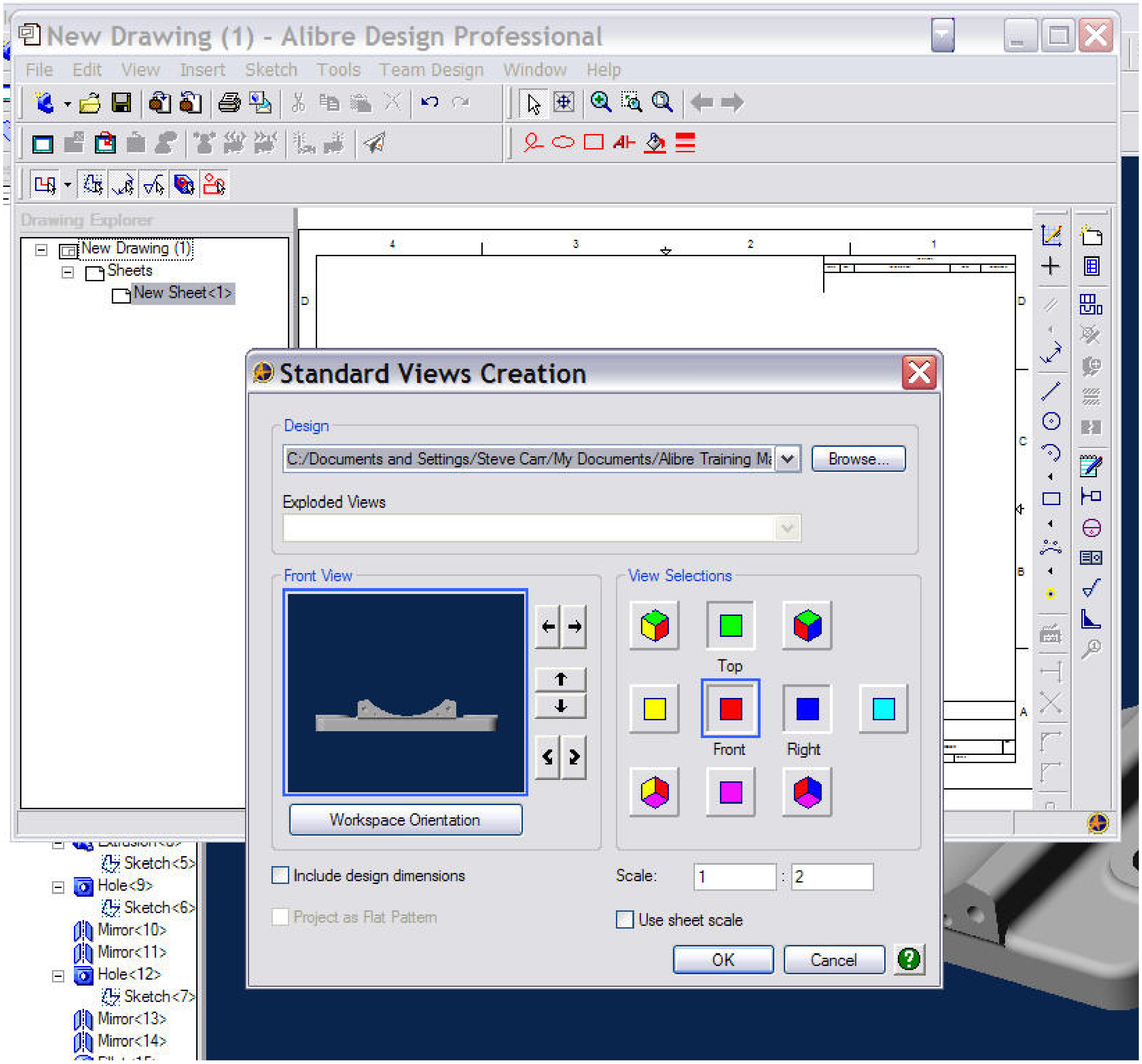




When we click OK to create the new drawing the ‘Fill In Text’ panel opens. Here you can fill in the Drawn by text, the date, and a drawing number. For now just click ‘Cancel’. We’ll fill in all the drawing text later.

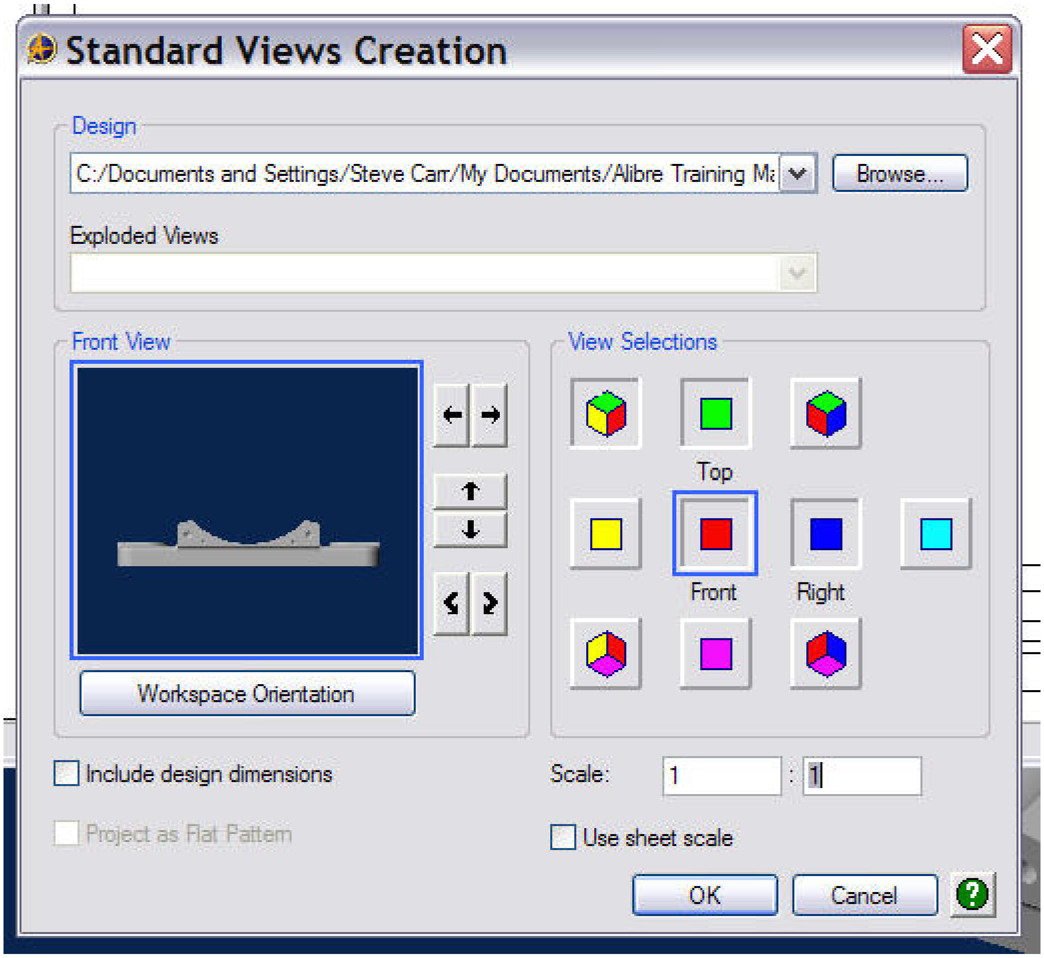
Note: Creating, placing, and editing text in the Alibre drawing environment is very similar to the processes used in many word processing and presentation programs. Notes, callouts, geometric tolerance and finish symbols, etc., can be added at anytime during the documentation process.

The Standard Views Creation panel opens. Here we'll select the views that will be included in our drawing. Like text, views can be added or deleted at any time in the documentation process.



In selecting the views to use in any drawing, a designer has to make a determination as to the information she's trying to convey. It is usually standard practice to make the Front view the one that will show the greater portion of that information

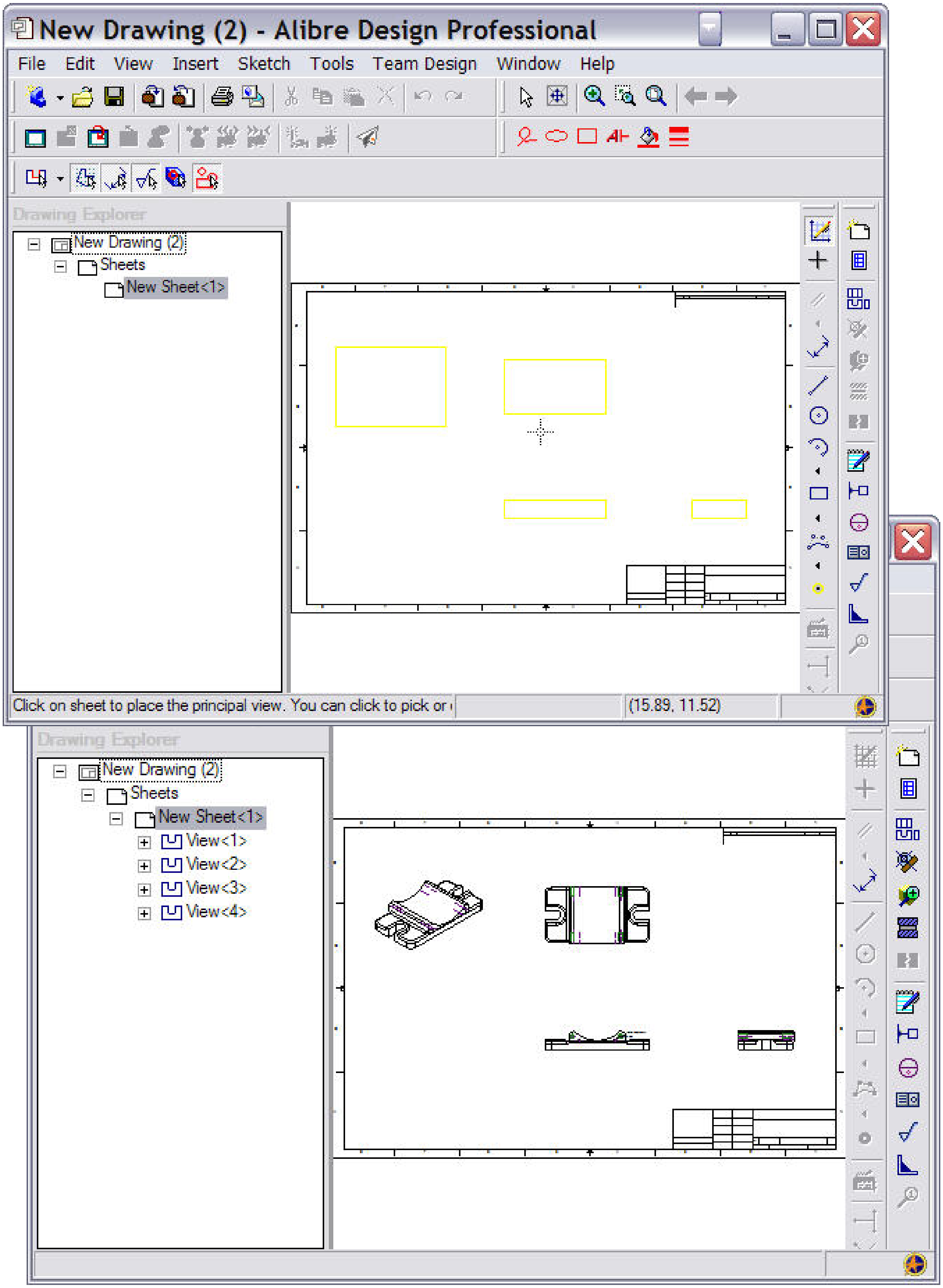
. In the case of the Saddle Base the view we'll make our Front view the one showing the holes and 2.00 in. radial feature, as viewed when looking directly at the XY plane. If this view isn't present in the 'Front View' window, use the orientation arrows at the right of the window to rotate the Saddle Base into the correct orientation. Select the other views you want to include in the



drawing using the View Selection panel to the right of the Front View window. We'll select the Top and Right views and the isometric view button in the upper left hand corner.
You can also elect to include any exploded view (for assemblies only), or include design dimensions in the views when they're created. Leave this option blank for the moment.

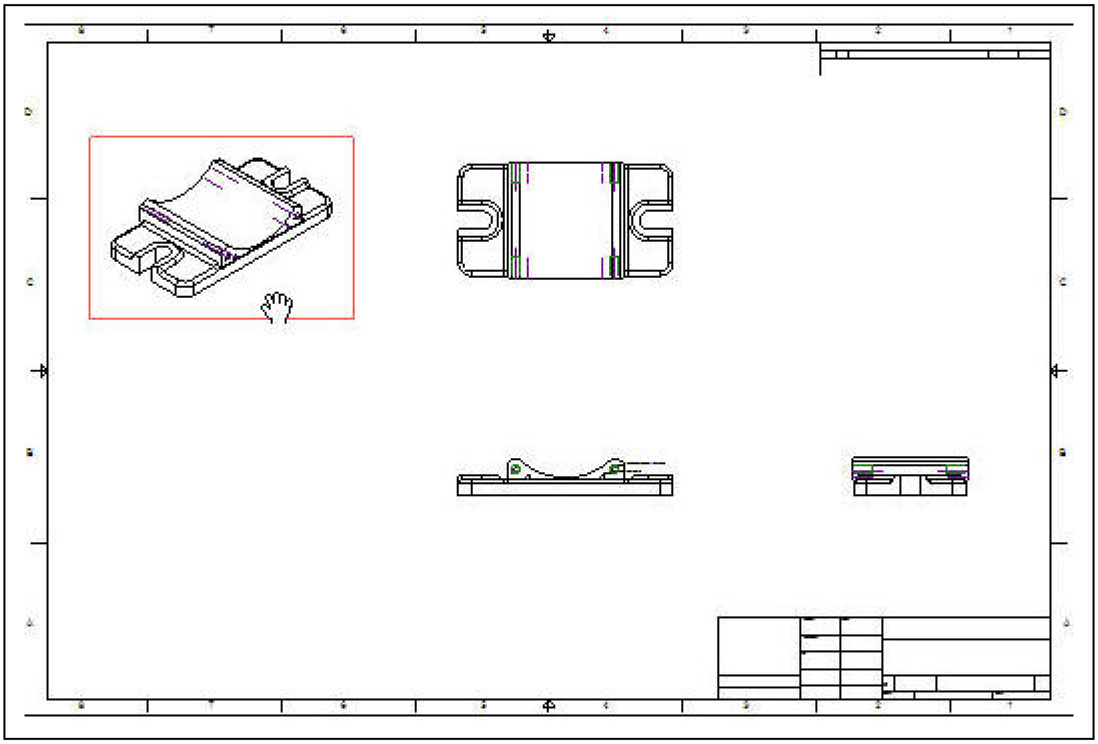
Even if the Front View is in the correct orientation, it would still be a good idea if you cycled through the various alternatives just to familiarize yourself with the process. Now that you've selected the views you want to use, make sure the scale is set to the desired value and then click OK.

The selected views will appear as 'View Frames, here shown as pale yellow rectangles.



Place the view approximately where you want them and click to terminate the command.

If the location of the views needs to be changed, it's easy to do. Move the cursor over a view until its frame highlights (see the isometric view below). The cursor will turn into a hand and at this point, you can drag the view to a new location. The isometric view will move independently of all the other views.



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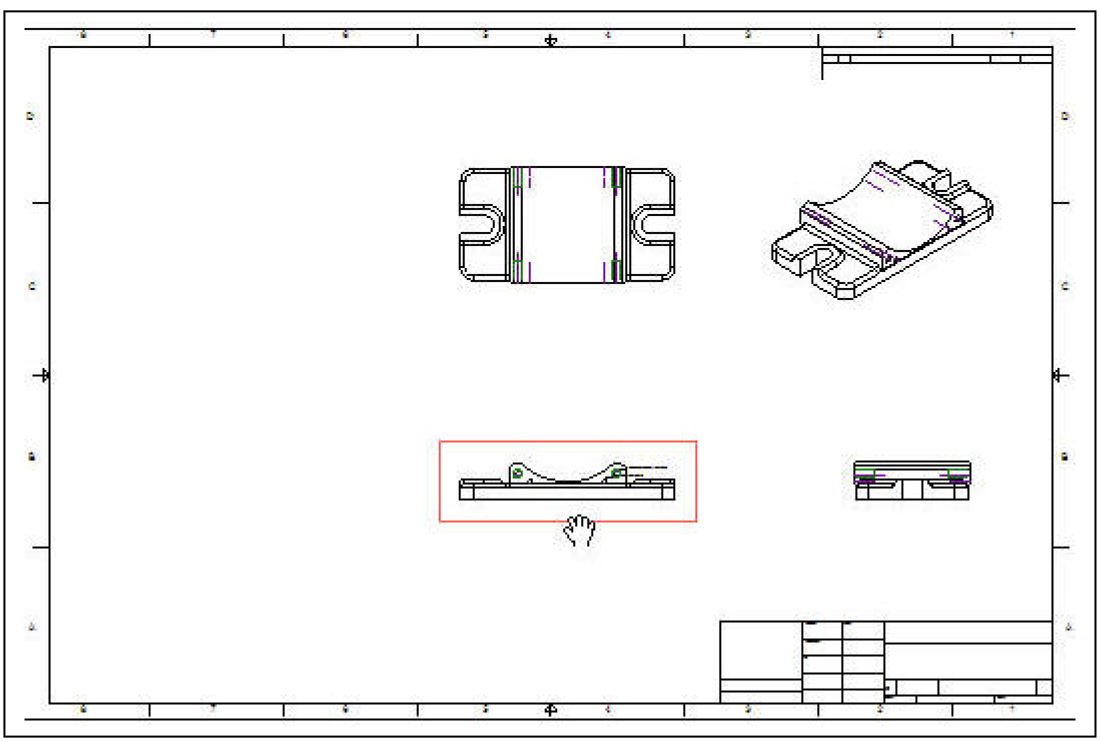
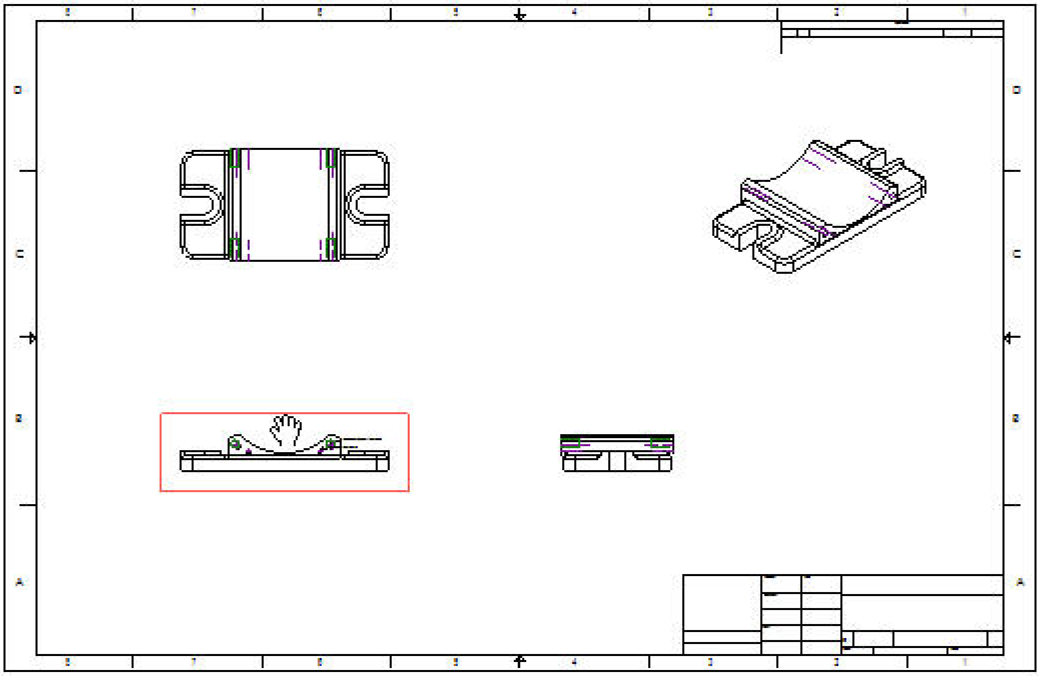
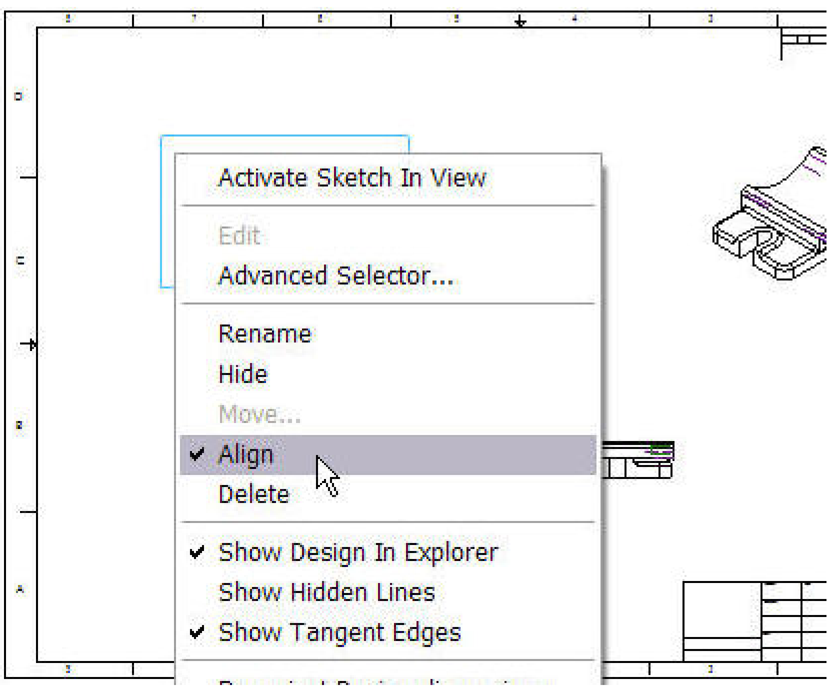
The Front view is the master view and moving it will move the top and right side views.

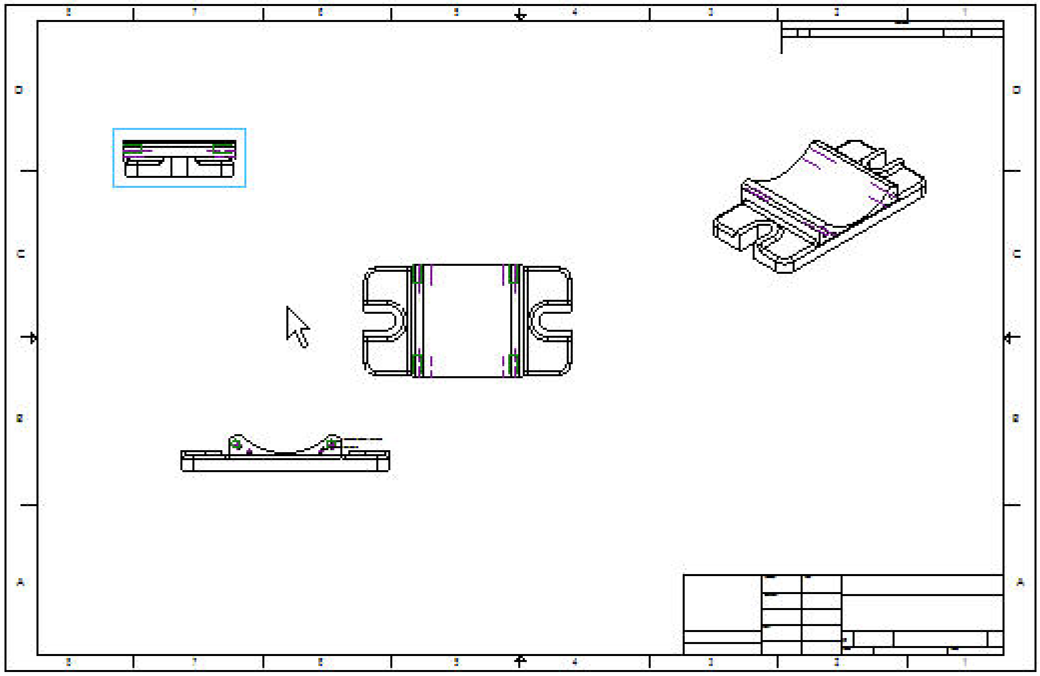




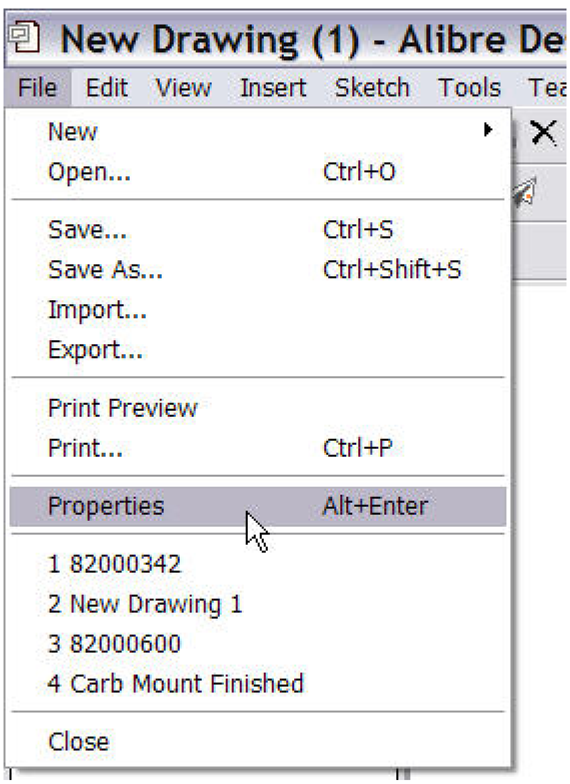
You can see in the picture above that moving the Front view has moved the Top and Right views to a new location. Both the Top, and Right views can be moved independently, but there are restrictions as to how they can be moved. The default arrangement in all new drawings only allows the Top view (and/or Bottom view) to be moved vertically in relation to the Front view, and the Right view (or Left view) can only be moved horizontally. Both will maintain their alignment to the Front view. You can over ride this alignment restriction by right clicking on a view and clearing the check mark next to the Align command.



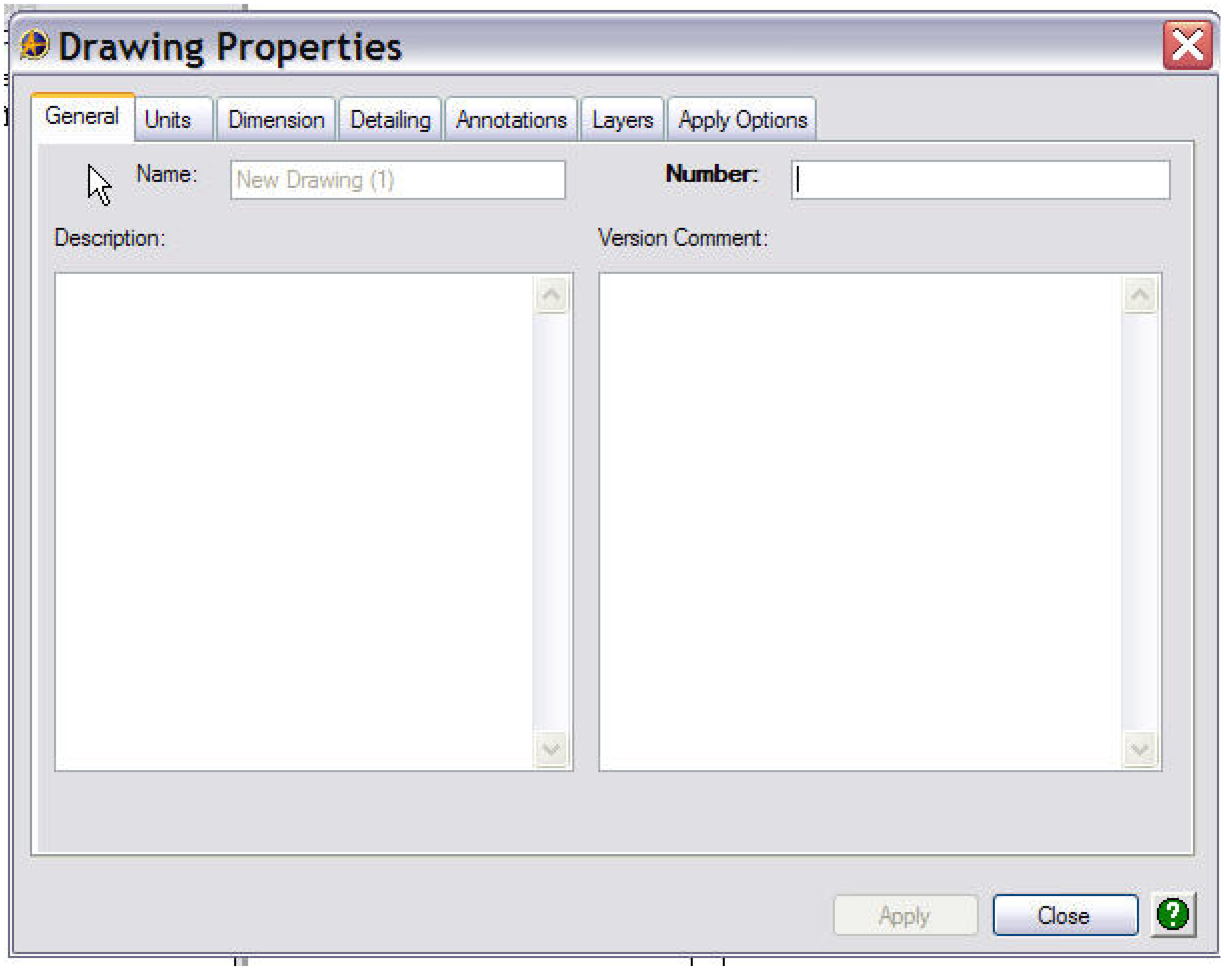
Now you can move the view wherever you'd like it.



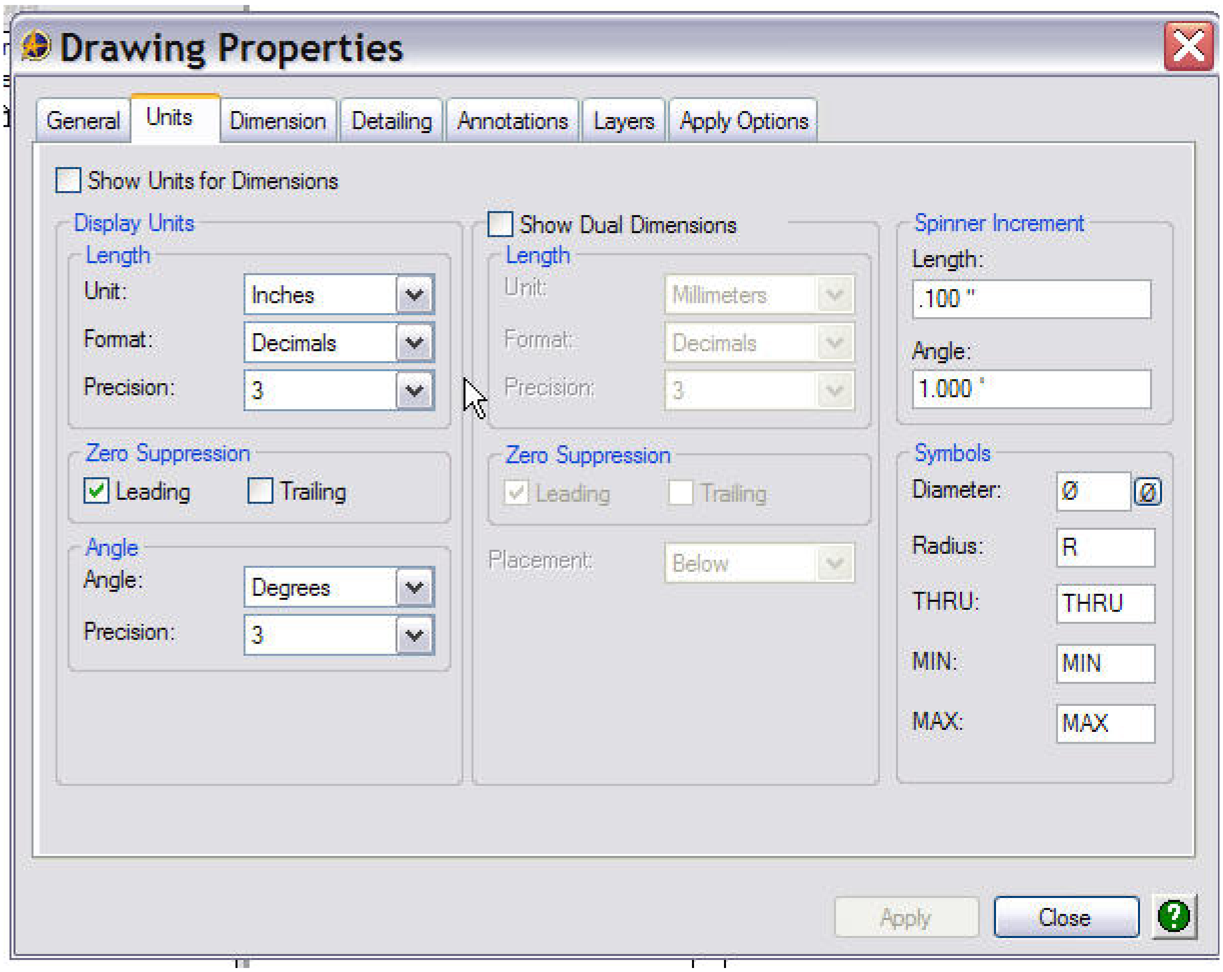
Before we get any further along in the detailing process, this is a good place to point out some of the other tools available that will allow you to customize the way your drawing is detailed. These properties can be set as defaults for your custom templates, allowing you to create drawing templates and detailing standards, i.e., text size and font, line widths, and styles, layering standards, etc. for use by everyone in your company. Click on the Properties tab under the file menu.



The Drawing Properties panel will open. Here you can set properties that will determine how you'll detail your drawing. Under the ‘General’ tab, you can enter the drawing number, a description of the part and version comments, all of which can make tracking changes more efficient when future changes are necessary or when your boss asks you why and when a specific change was made.



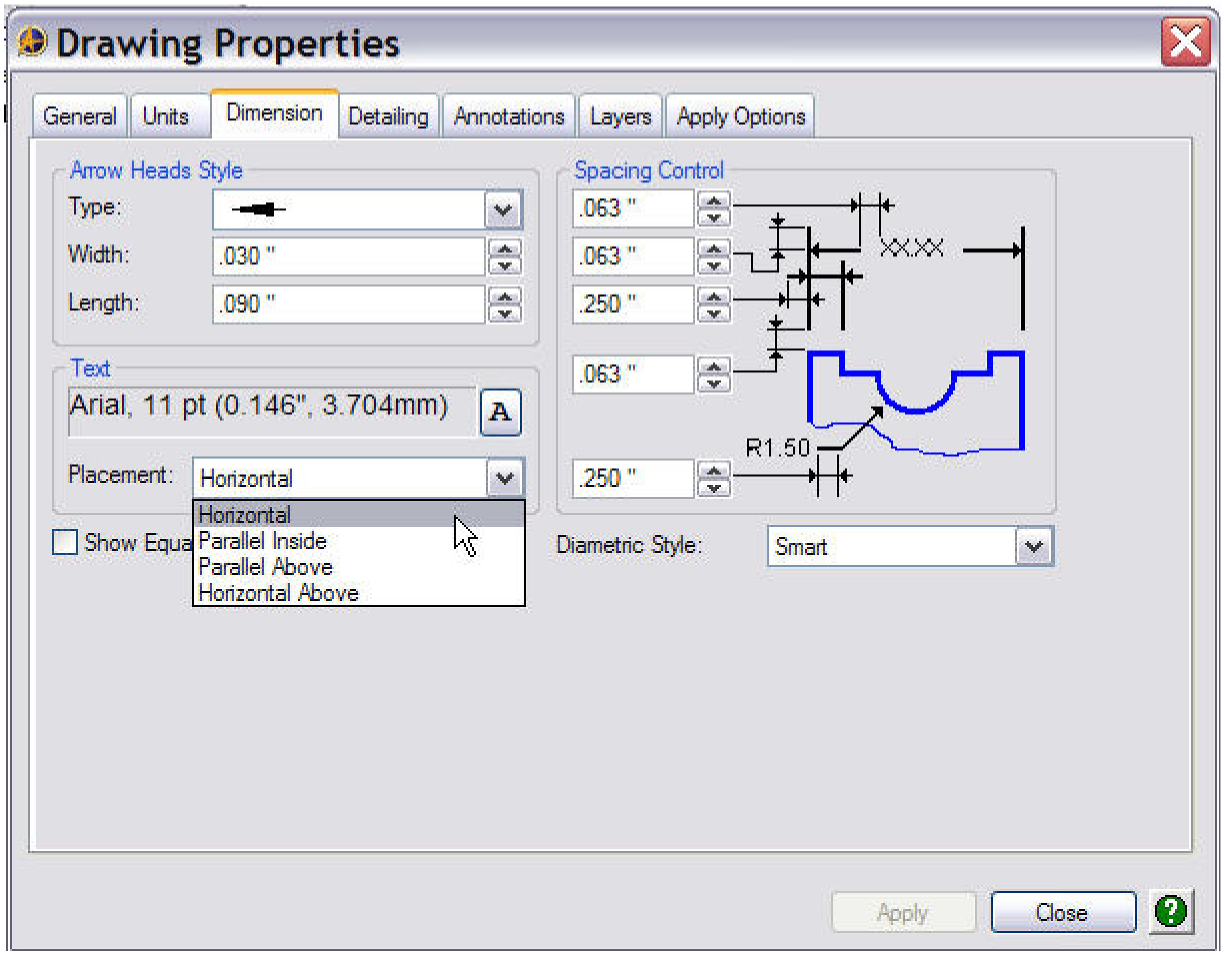
Under the ‘Units’ tab, you can select the Units your drawing will use when displaying dimensions, the format and precision level, zero suppression, dual dimensions and symbols specific to certain dimensional callouts, i.e. Diameter, Radius, etc., as well as the settings for the Spinner Increment.



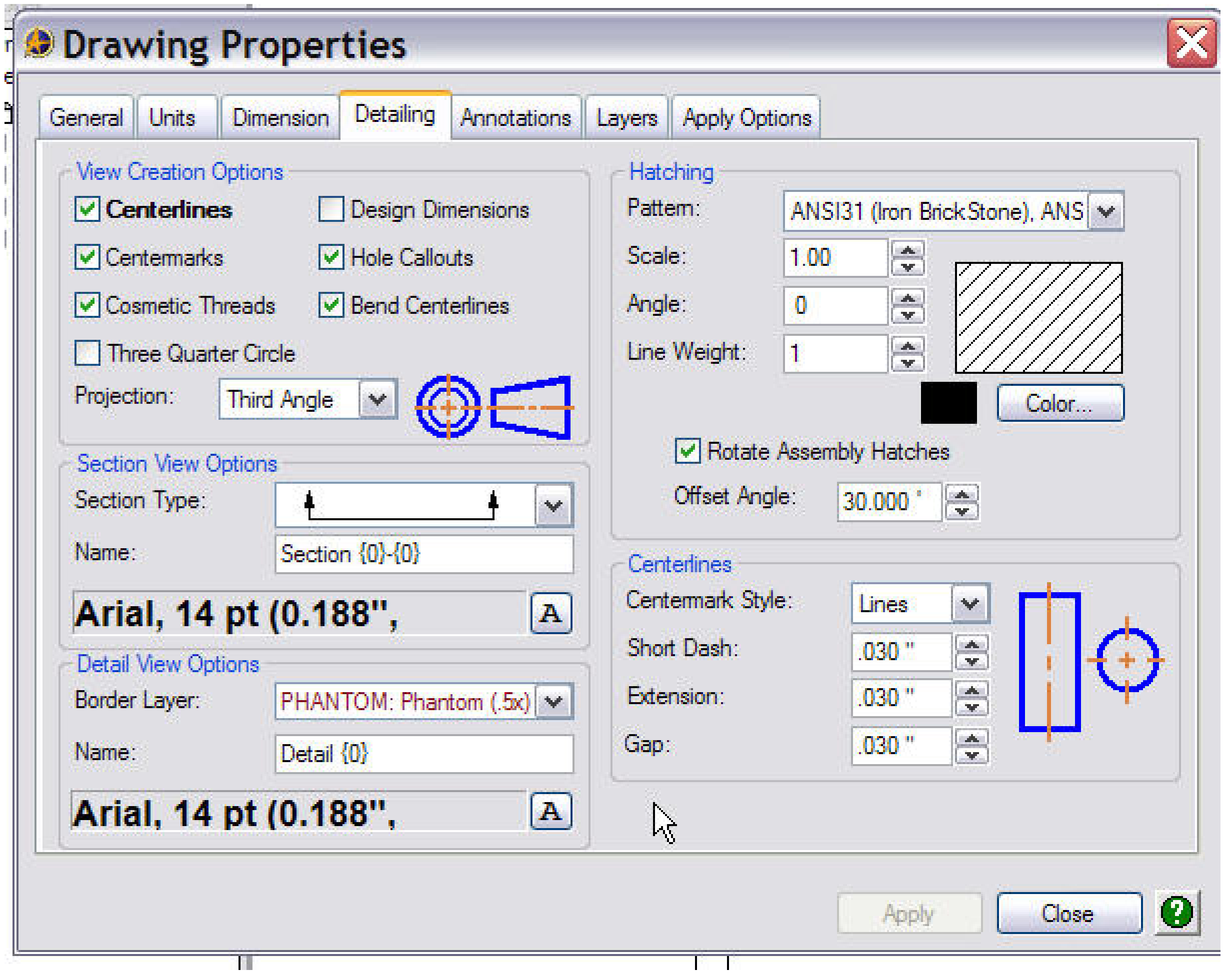
Spinner Increments are associated with the dimensional control box that appears whenever you place a dimension on a drawing.

The arrows to the right of the text value window control the incremental value (Spinner Increments) by which you can increase or decrease the value. In most cases, you’ll never see the Spinner box as almost all the dimensions you’ll be dealing with are governed by the dimensional value associated with the model feature selected. However, you may need to add non-associative dimensions to drawing figures and this is where you might encounter this particular item.

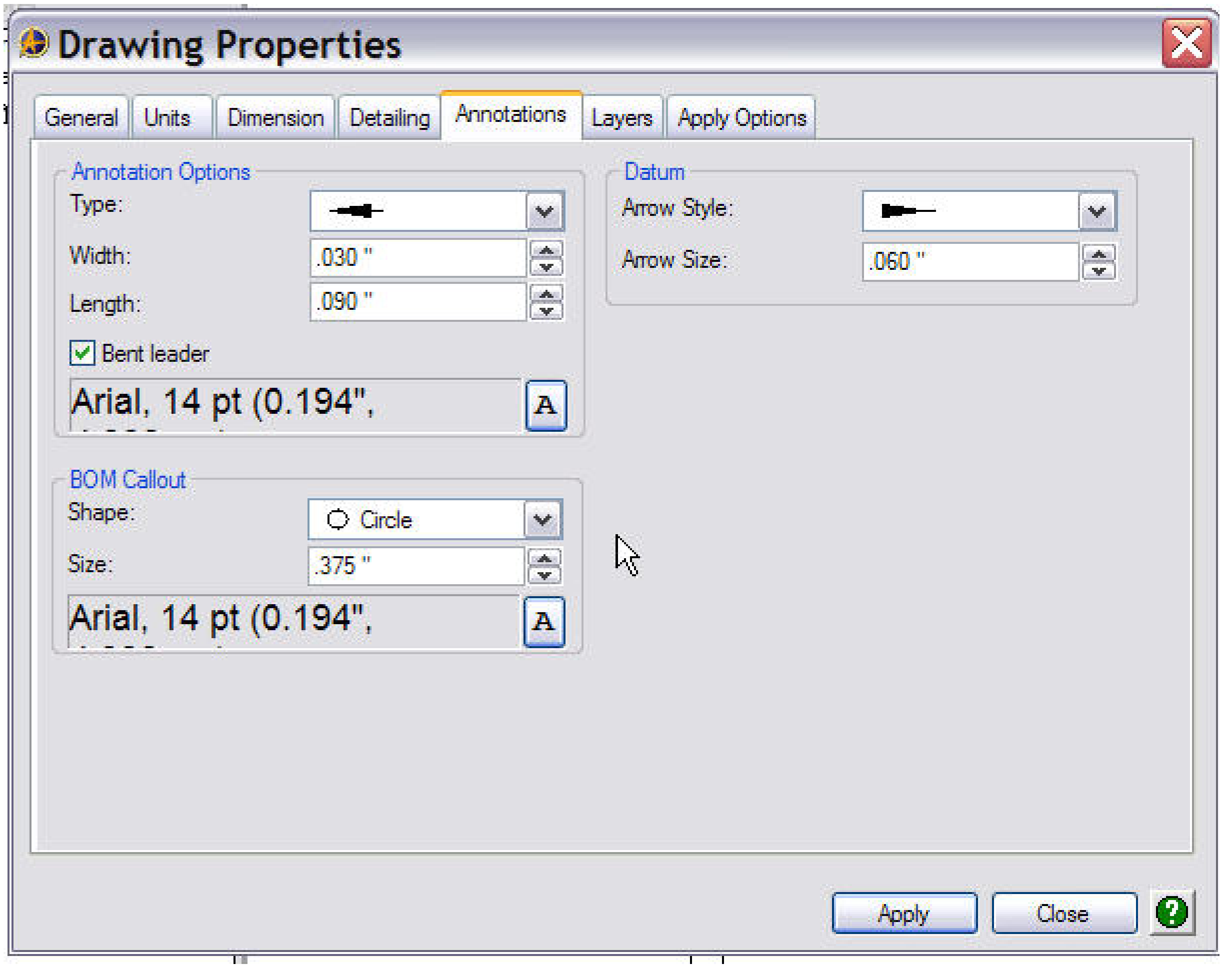
Under the 'Dimension' tab you can select your arrowhead style, and properties; the text font, size, and associated characteristics including how it will be placed in relation to the dimension, and the spacing characteristics of the dimensions.



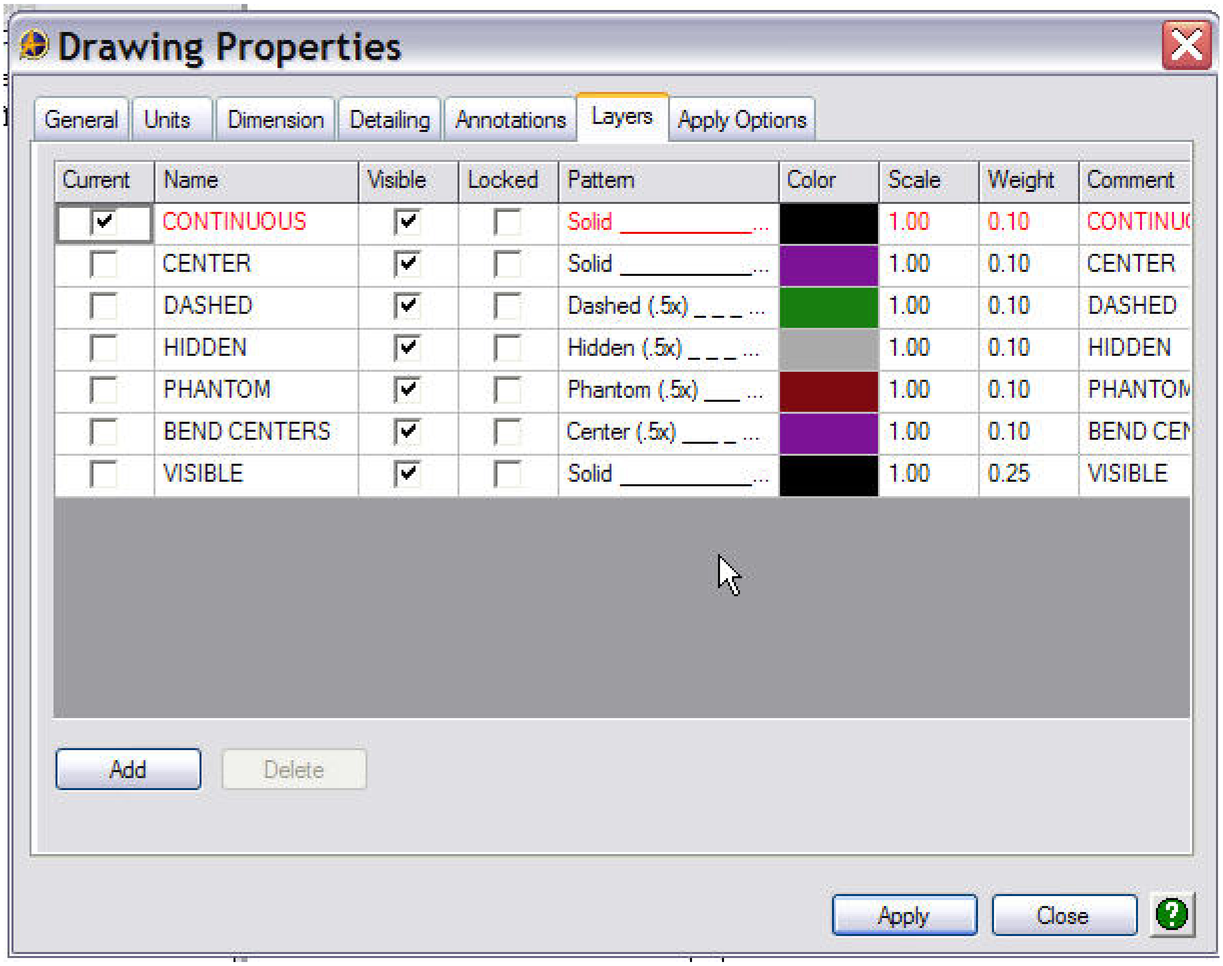
Under the ‘Detailing’ tab, you can select your view creation options, and hatching and centerline styles.



Under the ‘Annotations’ tab you can select the arrowhead styles, BOM callout shapes, and Datum arrow styles and sizes associated with drawing ‘Callouts’.



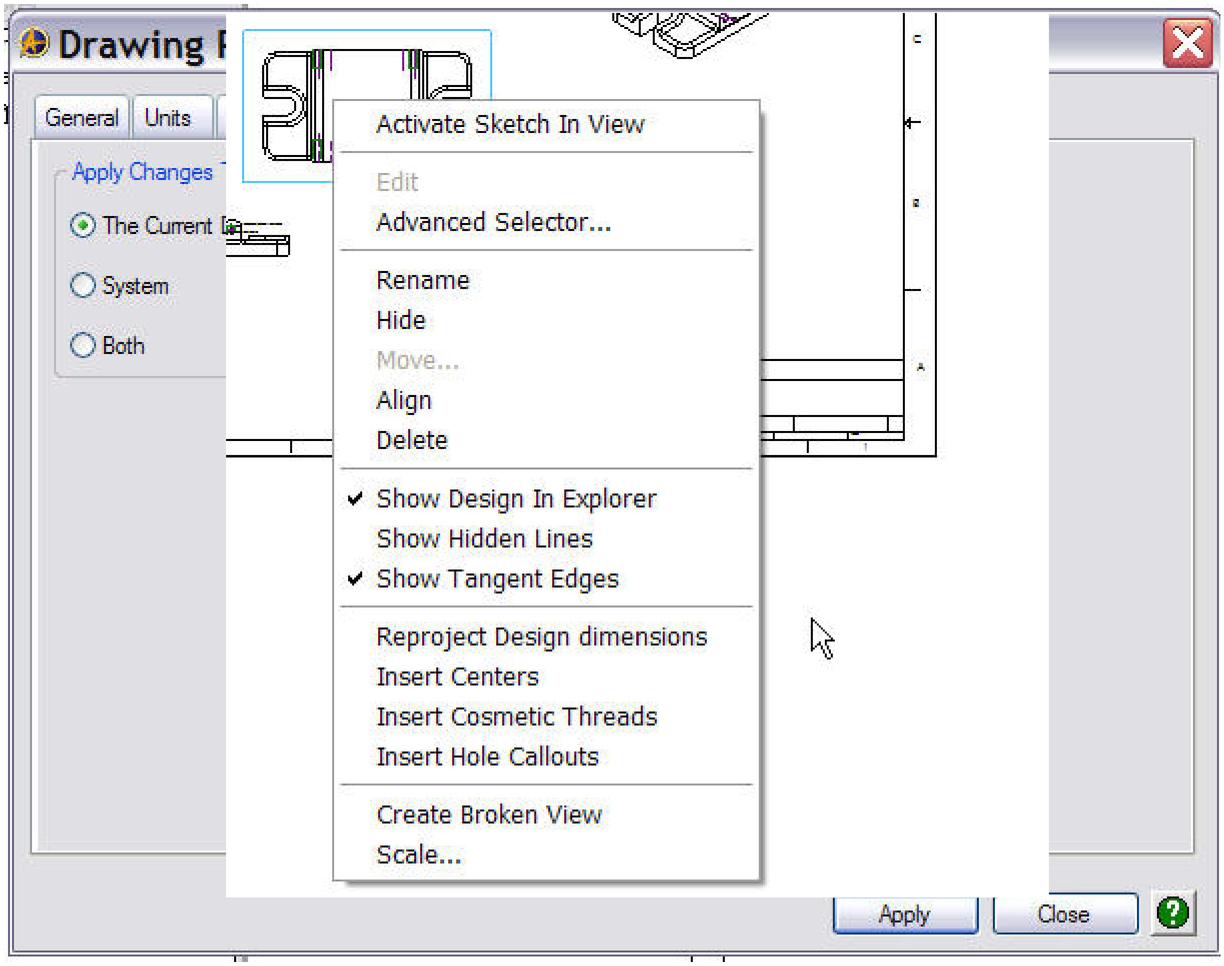
Under the ‘Layers’ tab you can set active layers, create new layers, set line names, visibility, and lock properties, patterns, colors, scale and weight properties. If your company uses a standard layering schema, it can be created here and associated to your drawing templates.



In the past, companies used layering standards as a way to control various aspects of their design and detailing processes. In the case of the 3D aspects of parametric modeling applications such as Alibre Design, these can be viewed as a holdover from the days when wire frame and surface modeling were the only CAD tools available. Designers used layers as a way to highlight and separate (no this isn’t a hair product commercial) various aspects of their designs, as well as track changes and alternative design solutions. This was often both confusing and cumbersome and depended on the user’s fastidiousness in following the layering standards.

With the advent of parametric based CAD tools, especially those that included a history tree as part of the design process, layering became much less critical, except in the detailing phase of the design process. Here layers are as important as ever, allowing users to control and customize layer characteristics to suit their specific needs. Different types of lines can occupy separate layers as can, text notes, callouts, geometrical tolerance symbols, etc.

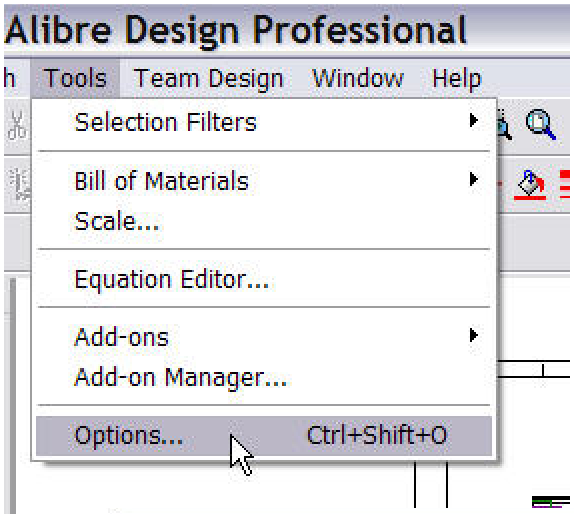
Under the 'Apply Options' tab you can select whether the options you've selected are to be applied to solely to the 'Current Document', to the 'System' or to 'Both'.



Right clicking on a view, will open a menu panel that will allow you to make changes that will only affect the selected view.

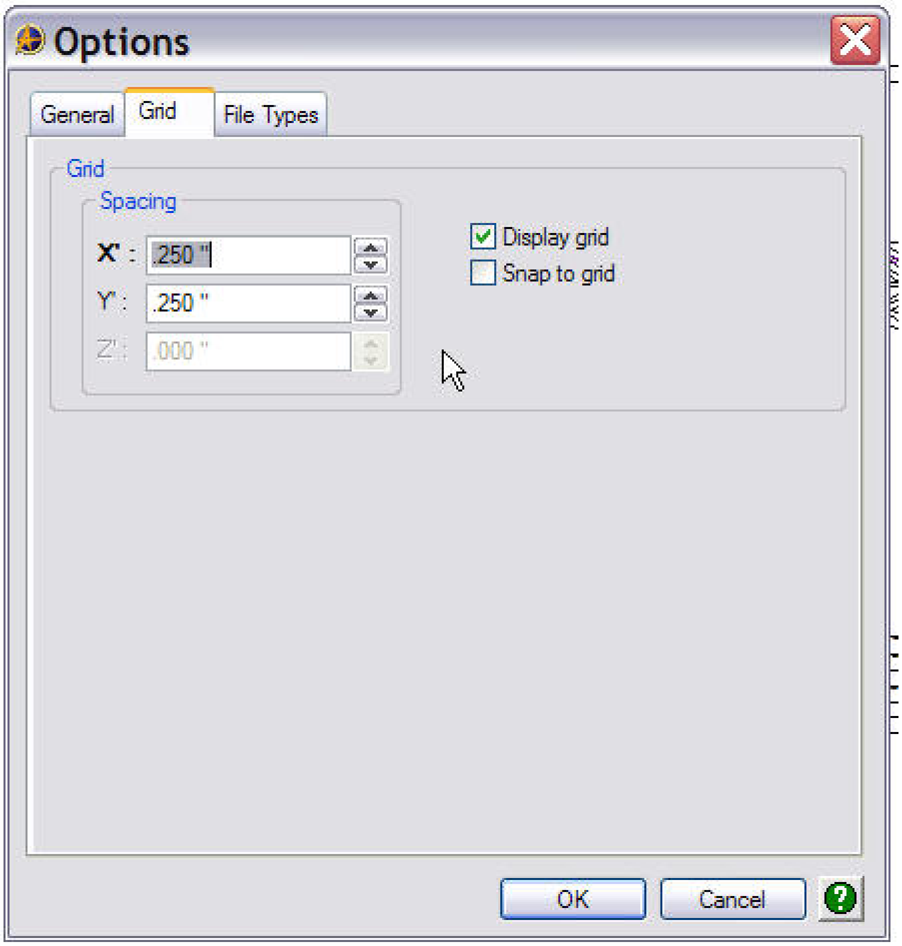
You can rename a view, hide it, change it's alignment properties, delete it, show and hide hidden, and tangent lines, re-project design dimensions (driving dimensions), insert centers (for holes, circles and radii, insert cosmetic threads, insert hole callouts, create broken views and change the view scale. It should be noted, that it is only possible to change the view scale in the front, and isometric views, and that when the scale is changed in the 'Front' view all other views, excluding the isometric view will follow suit.

If you modify a Design Dimension, the change will update the associated feature in the model database. You won't find this capability in non-parametric design applications. It is a very powerful feature and adds to a designer's efficiency by making it possible to make changes without the necessity of switching back to the modeling environment.

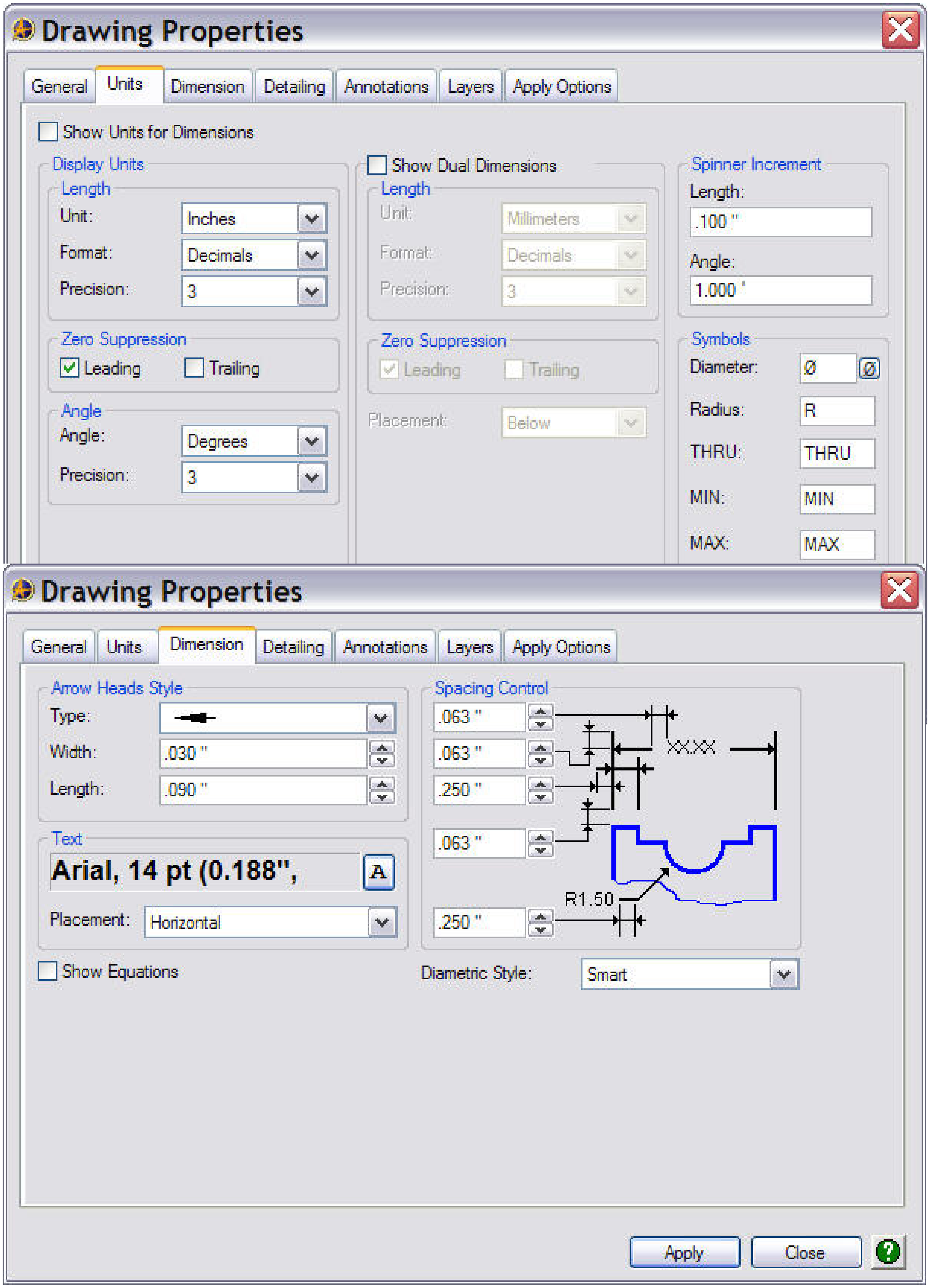


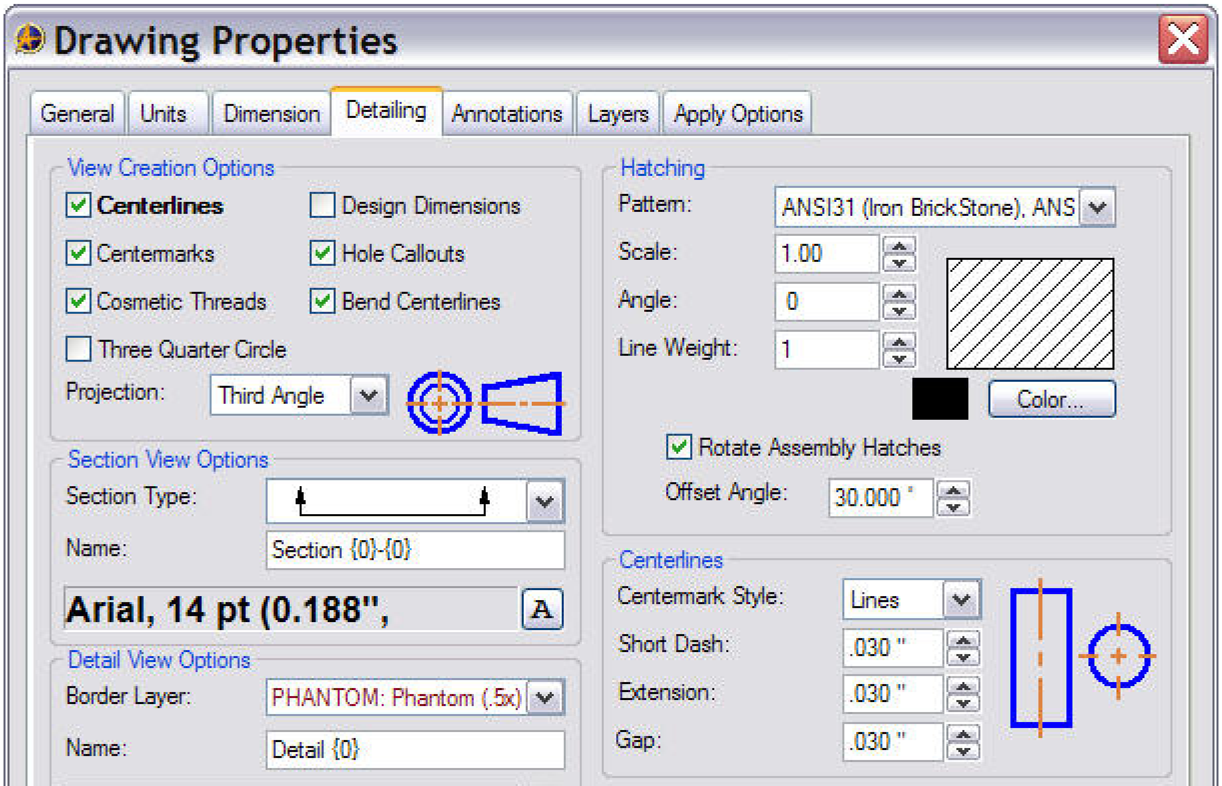
You can make changes to other drawing functions under the ‘Option’ panel, reached by opening the ‘Tools’ tab on the toolbar and selecting ‘Options’.

Under the ‘Grid’ tab, you can change grid spacing options as well as whether the grid should be displayed, or whether the cursor should snap to grid points.

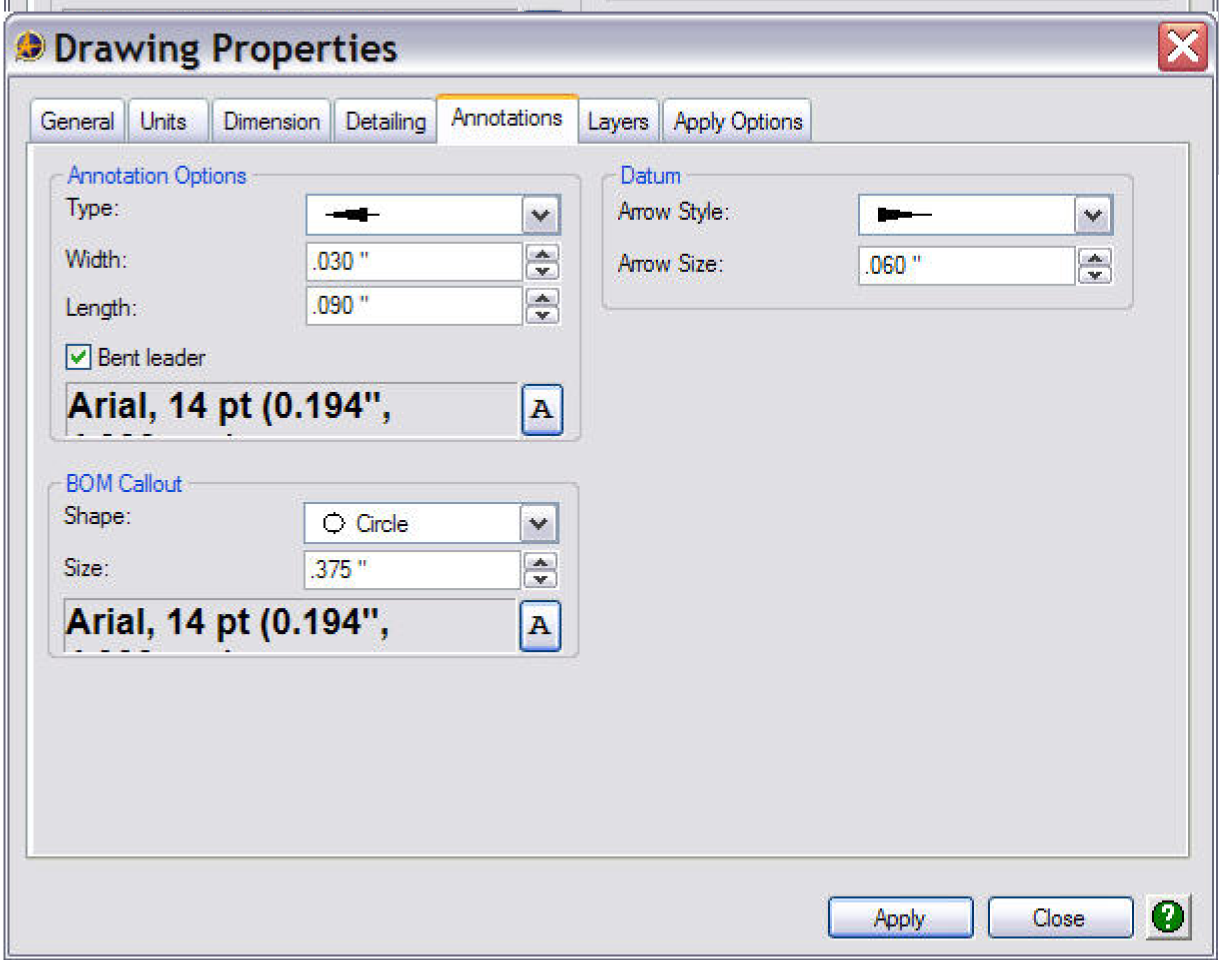


For the Saddle Base drawing set the following properties;

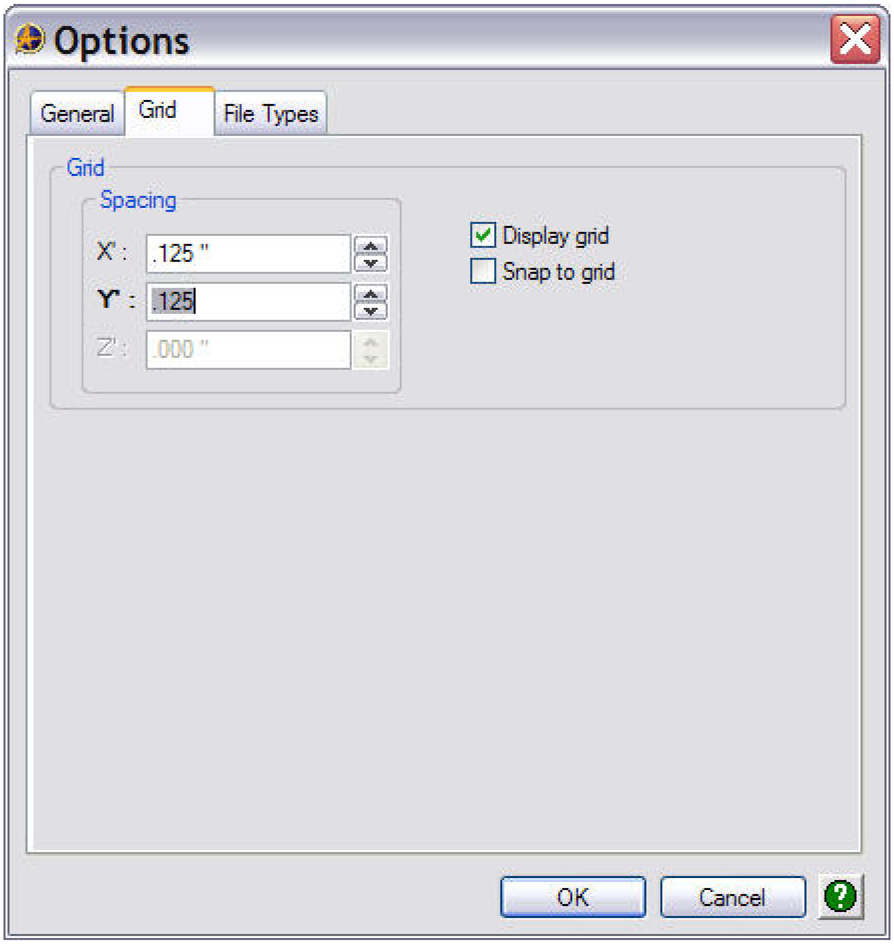




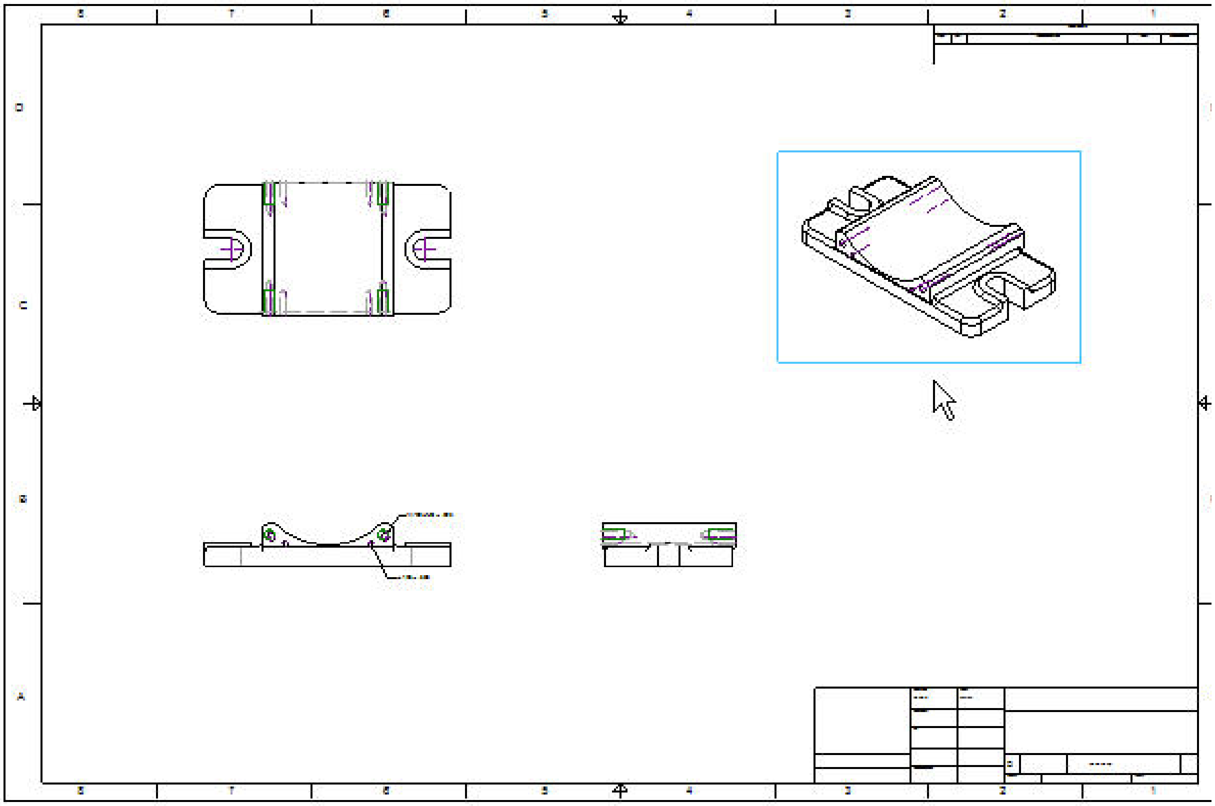
Drawing Properties
General
Units
Dimension
Detailing
Annotations
Layers
Apply Options
View Creation Options
Centerlines
Design Dimensions
Centermarks
Hole Callouts
Cosmetic Threads
Bend Centerlines
Three Quarter Circle
Projection:
Third Angle
Section View Options
Section Type:
Name:
Section {0}-{0}
Arial, 14 pt (0.188",
Detail View Options
Border Layer:
PHANTOM: Phantom (.5x)
Name:
Detail {0}
Hatching
Pattern:
ANSI31 (Iron BrickStone), ANS
Scale:
1.00
Angle:
0
Line Weight:
1
Color...
Rotate Assembly Hatches
Offset Angle:
30.000 °
Centerlines
Centermark Style:
Lines
Short Dash:
.030 "
Extension:
.030 "
Gap:
.030 "



Drawing Properties
General
Units
Dimension
Detailing
Annotations
Layers
Apply Options
Annotation Options
Type:
Width:
.030 "
Length:
.090 "
Bent leader
Arial, 14 pt (0.194",
BOM Callout
Shape:
Circle
Size:
.375 "
Arial, 14 pt (0.194",
Datum
Arrow Style:
Arrow Size:
.060 "
Apply
Close

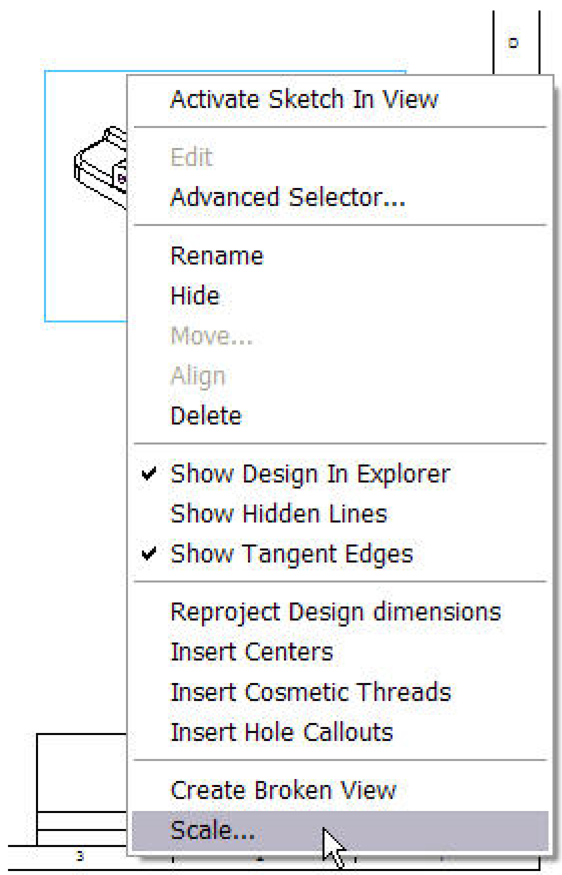


Now, back to the Saddle Base drawing.

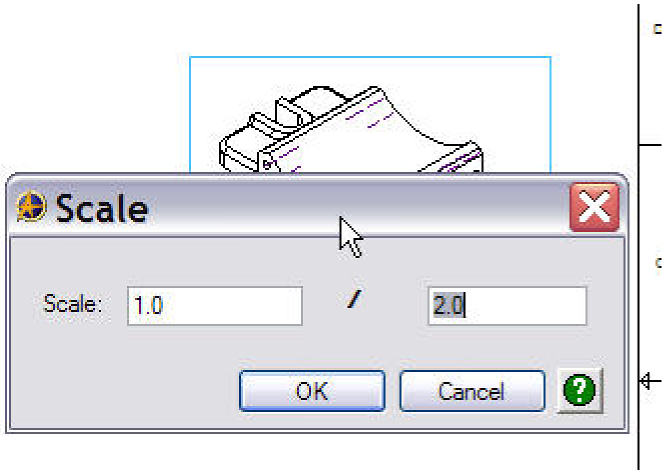


We need to make a few minor adjustments. The isometric view is sitting in the drawing area reserved for revision information, drawing notes and bills of material. It is also somewhat larger than required for what is in most cases simply an informational view. We'll change the scale and location to fall in line with most common detailing requirements.

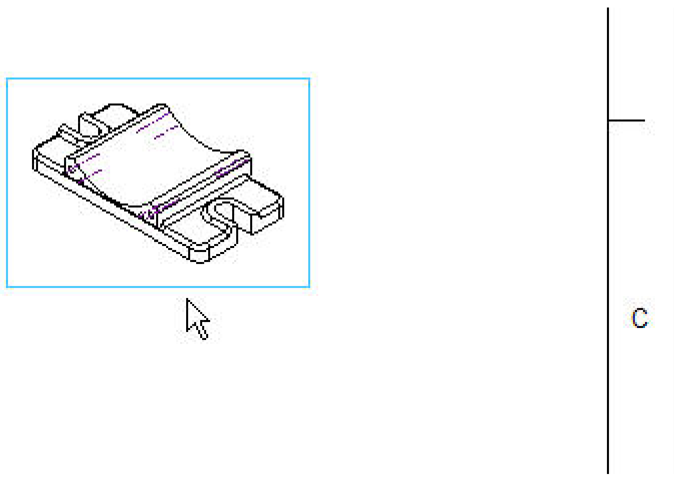
Make the isometric view active and then right click on it to open the view options menu. Select 'Scale'.



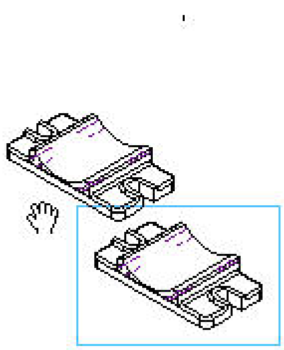
The 'Scale' panel will open. Change the view scale to 1.0/2.0 and click OK.



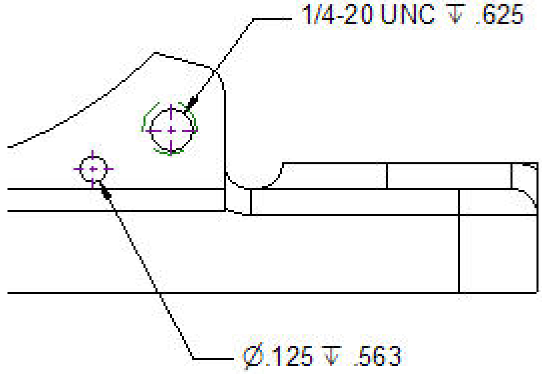
The view scale has changed.

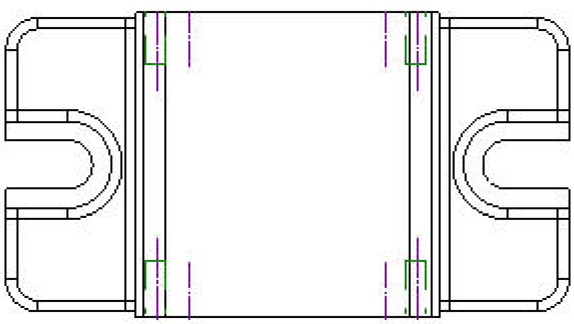


Now drag and drop it to a location outside the revision column area.

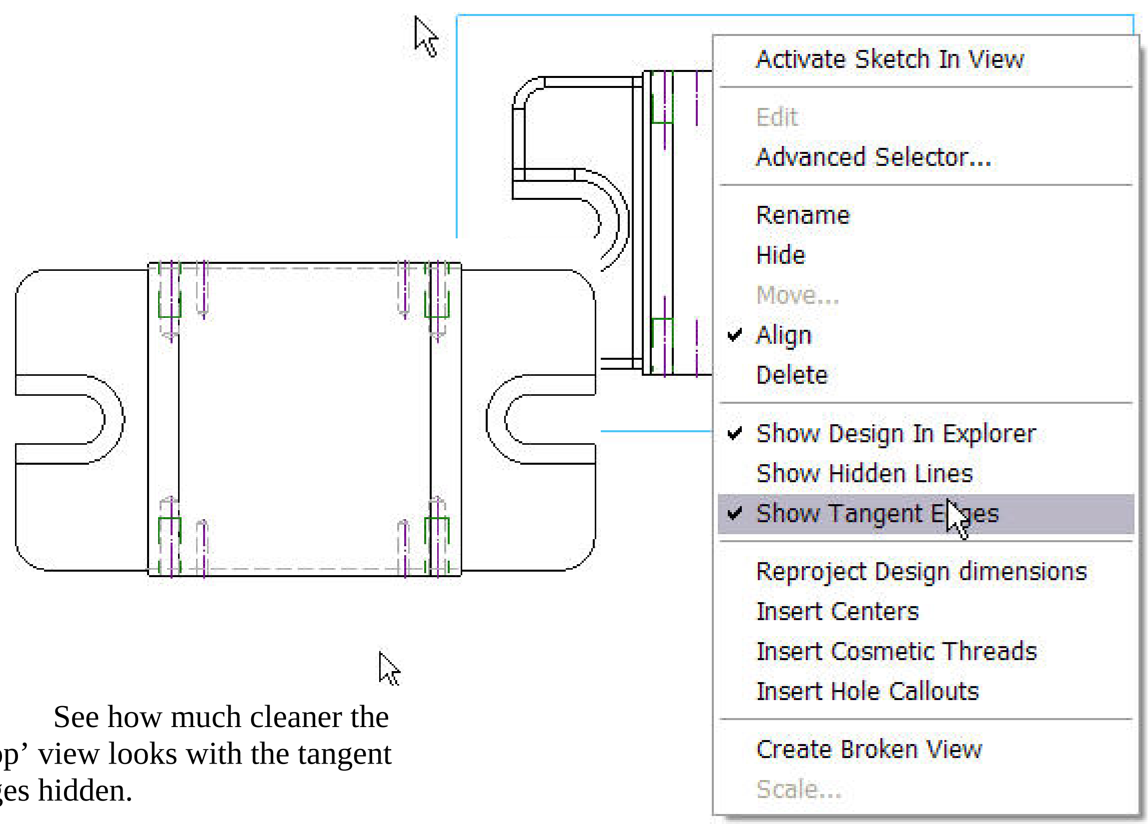


Note in the ‘Front’ view that the information for the holes you created in the model database are automatically displayed. We’ll edit the callouts later in this exercise.



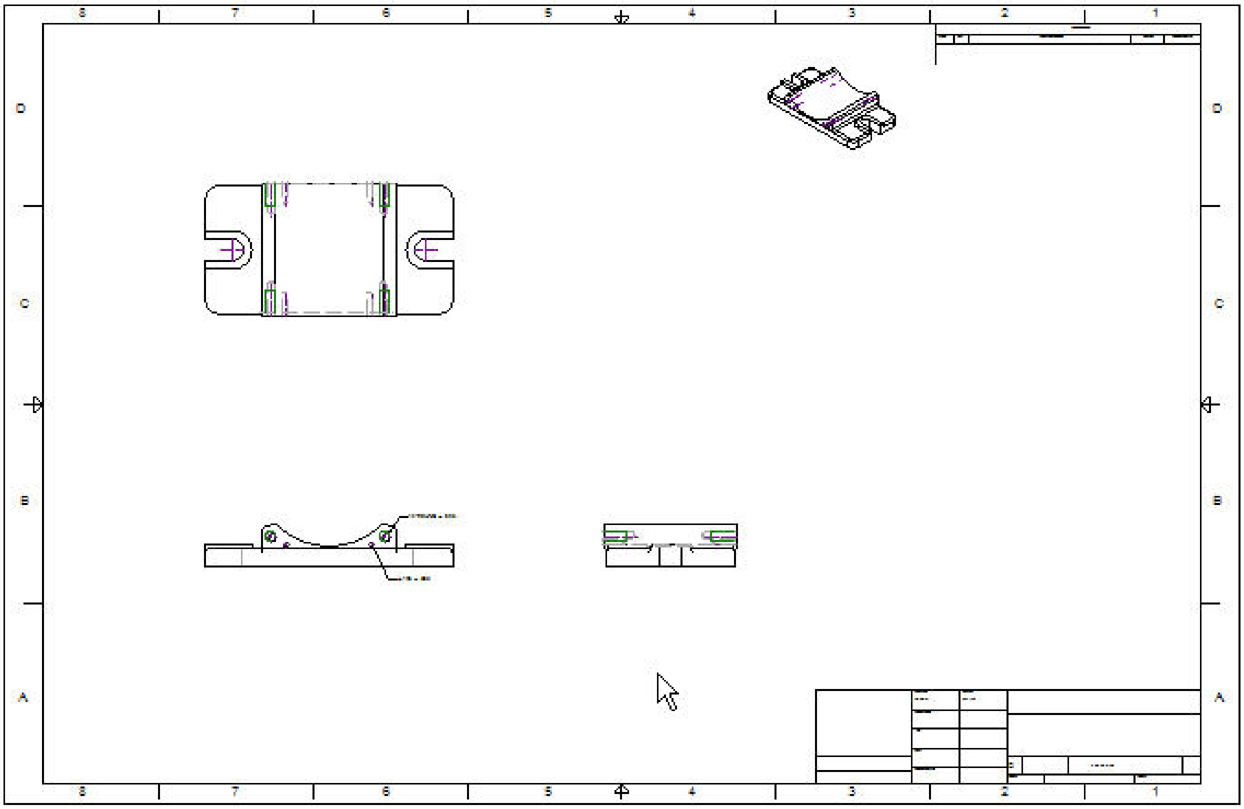


Note also that in all views, tangent lines are displayed, giving the views a cluttered look. It’s easy to hide these lines by right clicking on the view and un-checking the ‘Show Tangent Edges’ menu tab.



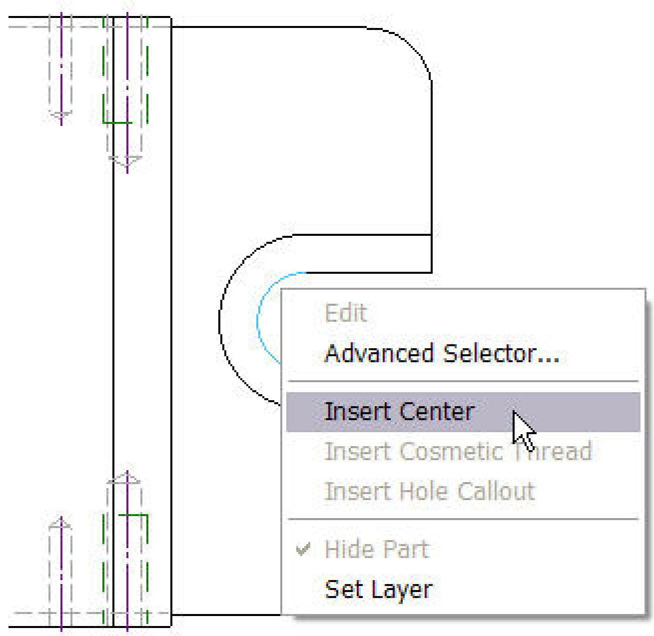
See how much cleaner the ‘Top’ view looks with the tangent edges hidden.

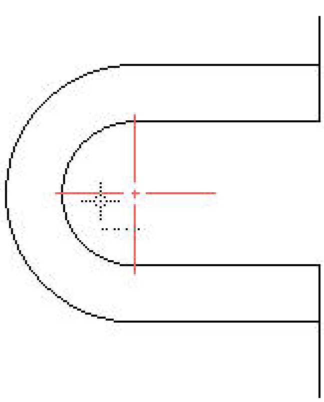
Hide the tangent edges in all views except the isometric view. The drawing should look similar to this.



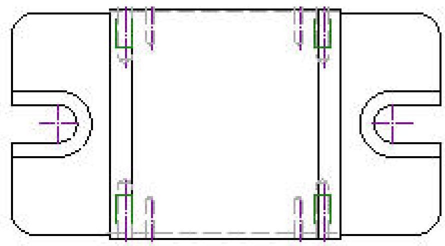
Next, we'll create the center marks for each of the major radial features in
our drawing.

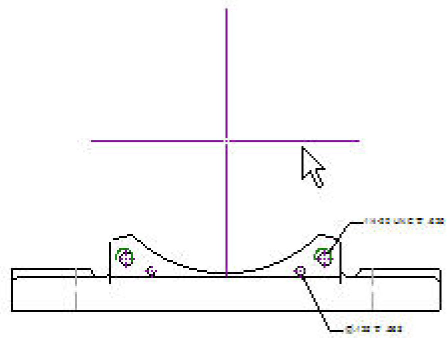
In the 'Top' view click on the radial feature of the right slot, right click and select 'Insert Center'. The 'Center' mark is created. Do this for the left hand slot and for the large radial feature in the 'Front' view.

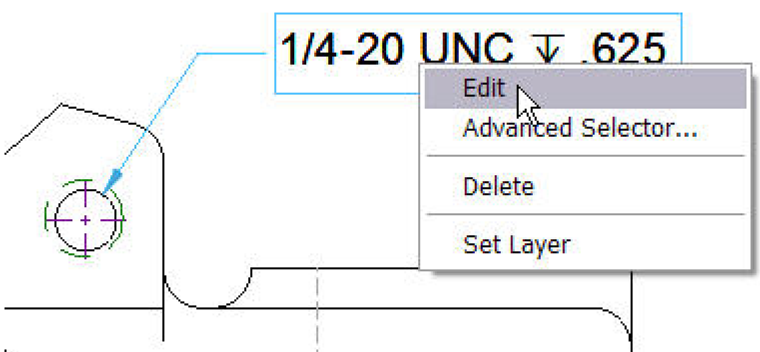




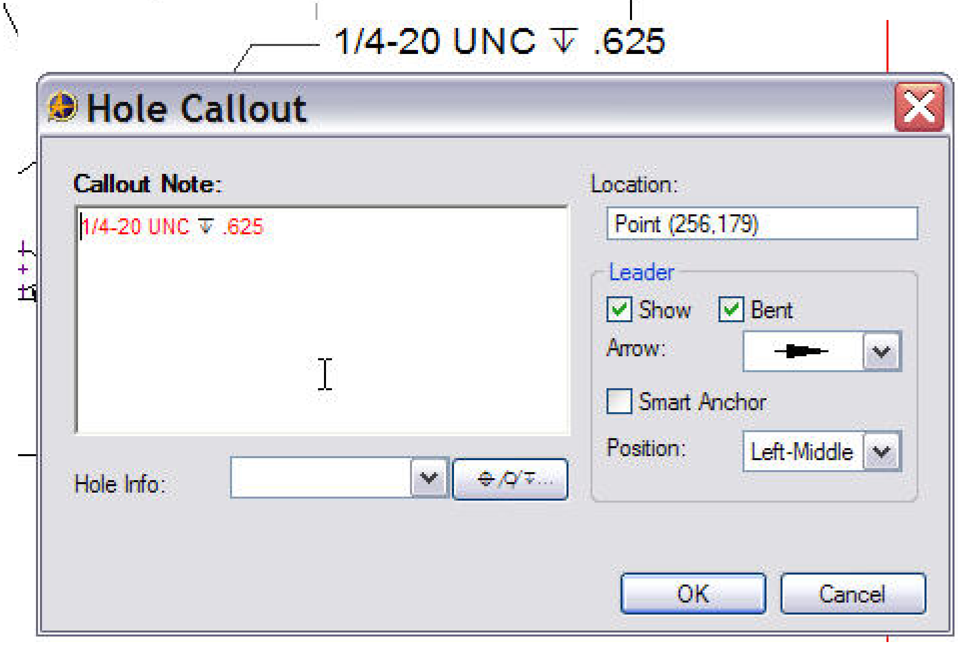
The two views should now look like this, with three center marks visible. We're now ready to dimension the different views.



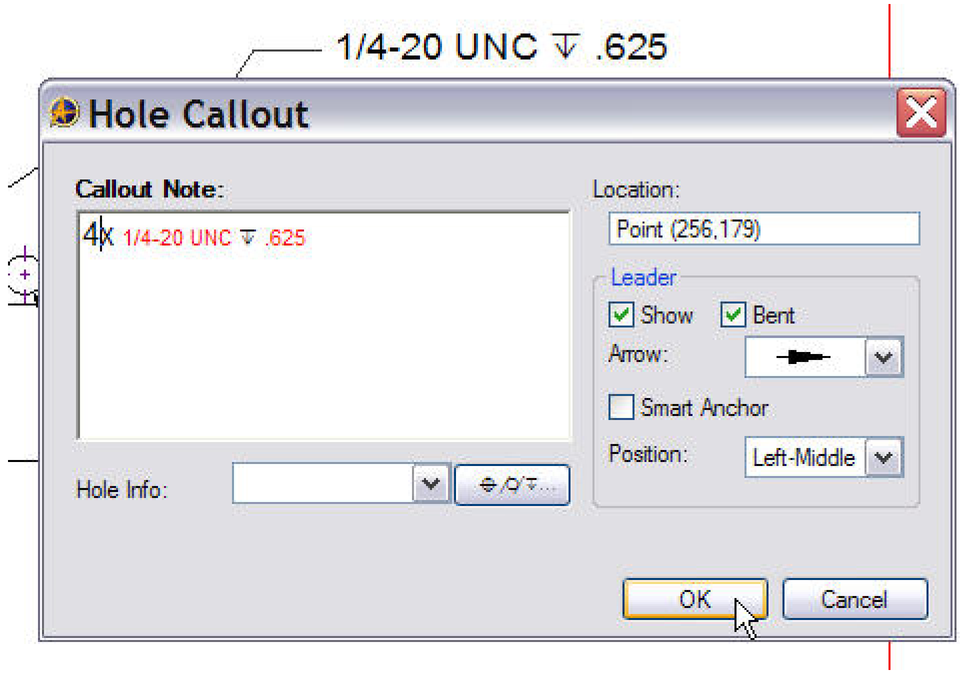




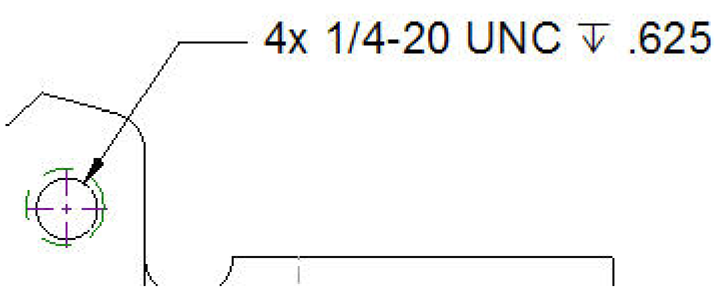
As mentioned previously, the 'Front' view should contain the bulk of dimensional information describing our part, so we'll begin our dimensioning here.



Let's start by editing the hole callout information. Place your cursor over the selected callout, right click, and select 'Edit'. The 'Hole Callout' edit panel will open.

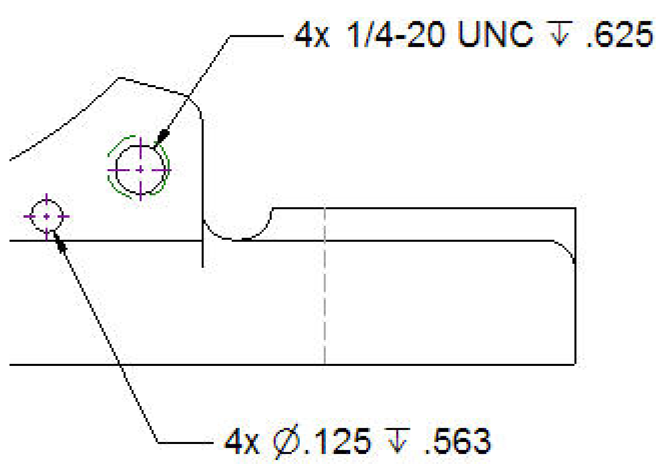


Position the cursor in front of the original text and enter 4x, then click ‘OK’.

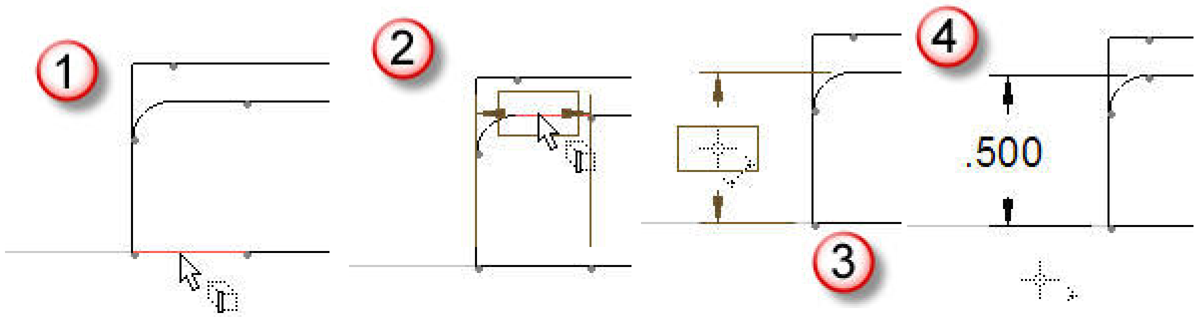


The callout should now look like this. Repeat this process for the smaller hole.

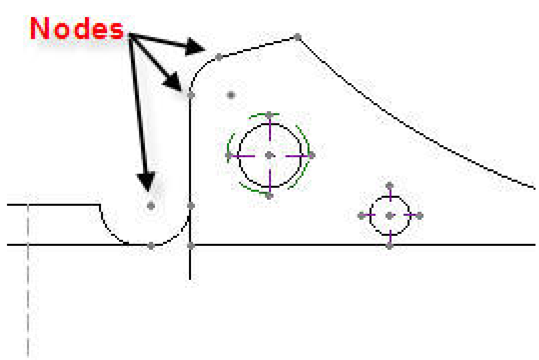
Your drawing should look like this.



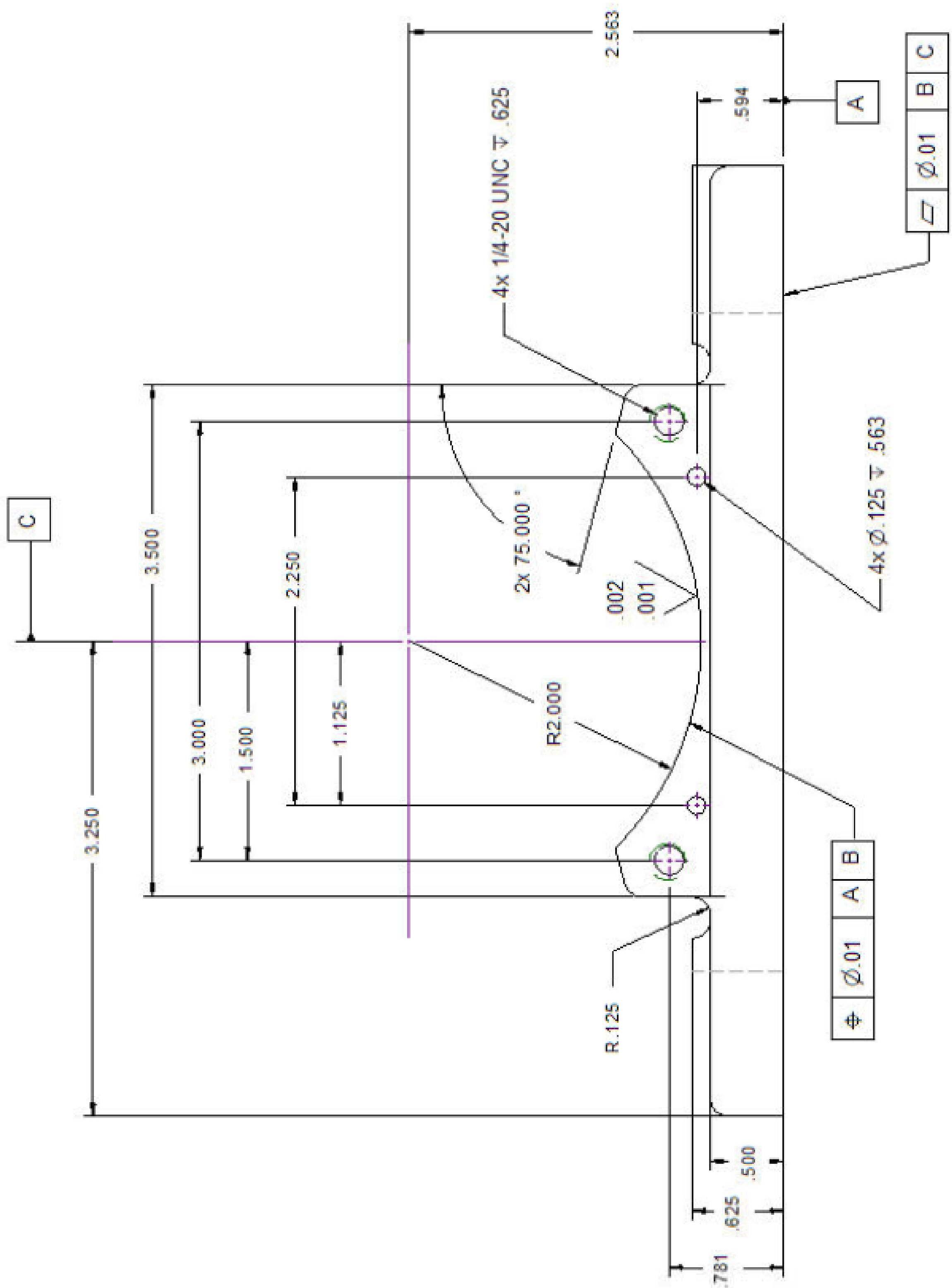
Make the front view active. Click on the ‘Dimension’ icon Linear dimensions are added by clicking on the selected features, and then positioning the dimension as shown in the pictures below.

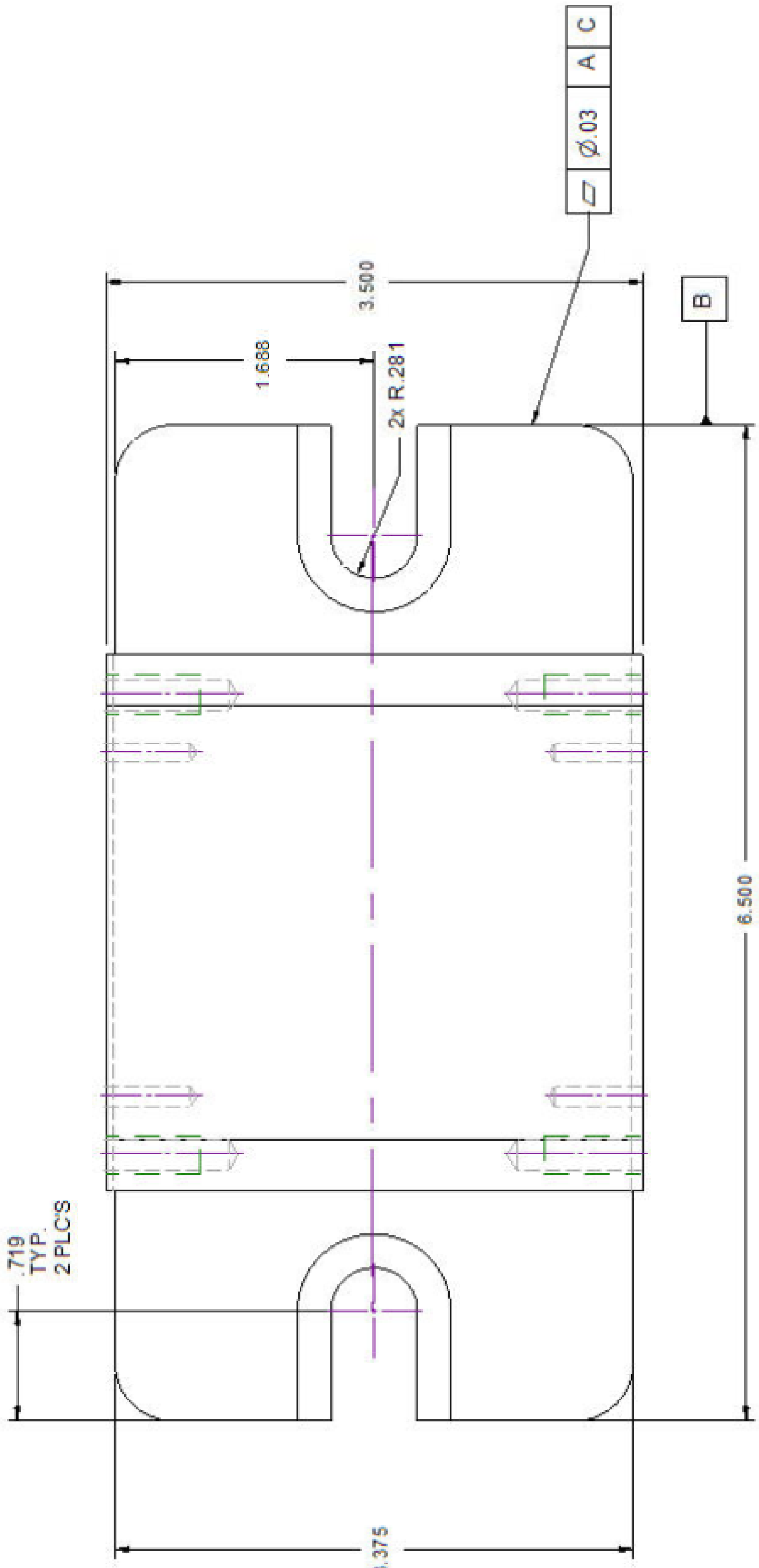


This applies to both horizontal and vertical dimensions. Dimensioning to the center of a circle or radial feature may involve a little finagling, depending on the features involved. Make the ‘Front’ view active. Click on the ‘Dimension’ icon, and select a circular or radial feature in the view, then click escape. Feature ‘nodes’ or ‘end points’ should now be visible for all elements in the view, and available for picking for dimensioning. You can now dimension to the center of a hole and not to the centerline entities.

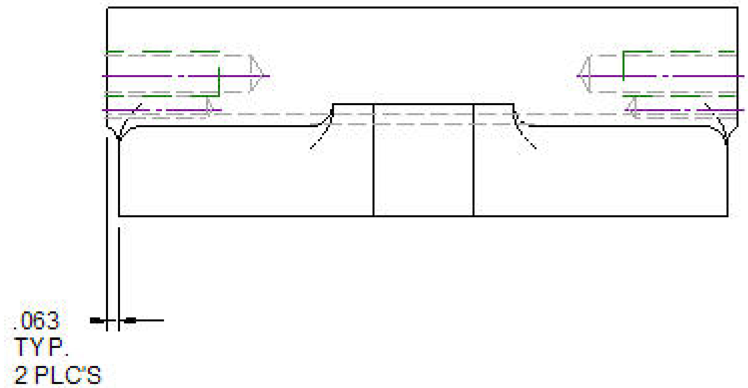


Continue dimensioning until the Saddle drawing looks like the pictures on the next pages. Your placement of dimensions may differ from that shown but the values should be the same.



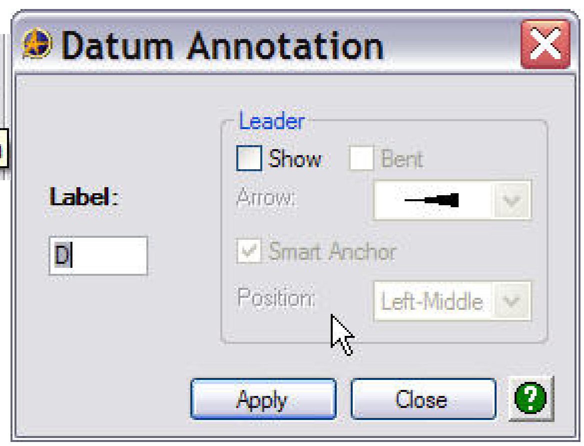


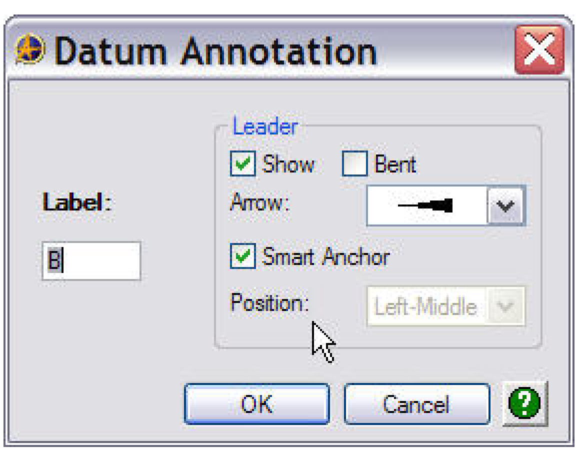
Ø.03
A
C
3.500
1.688
2x R.281
B
6.500
.719
TYP.
2 PLC'S
3.375



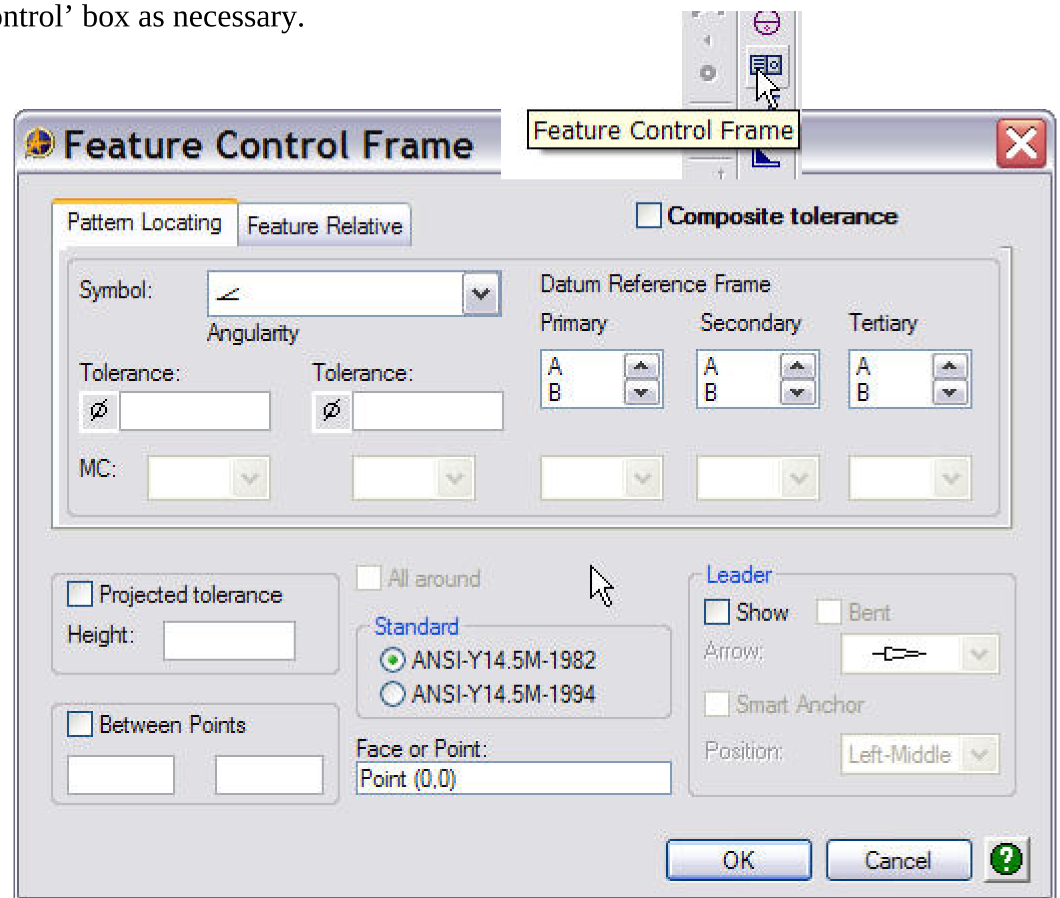
Inserting ‘Datum’s’ and ‘Feature Control Symbols’ is easily accomplished in the Alibre Design detailing package...

For Datum’s, click the ‘Datum’ icon, fill in the appropriate values, and select the desired options in the ‘Datum Annotation’ panel. When finished click ‘Apply and then position the Datum as desired. It is worthwhile to note that the Datum designation will advance one letter automatically as you insert them. You can edit any options after the initial datum creation by simply double clicking on the datum. The Datum Annotation panel will open, where you can edit the properties. When finished, click ‘OK’.

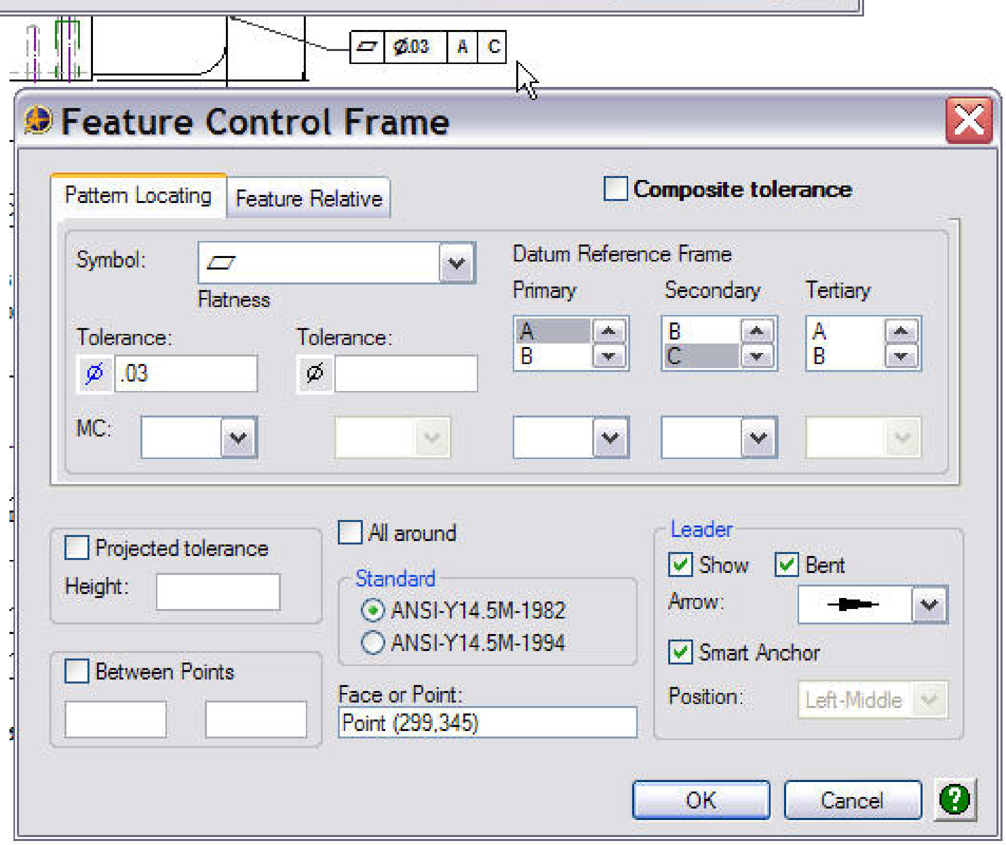




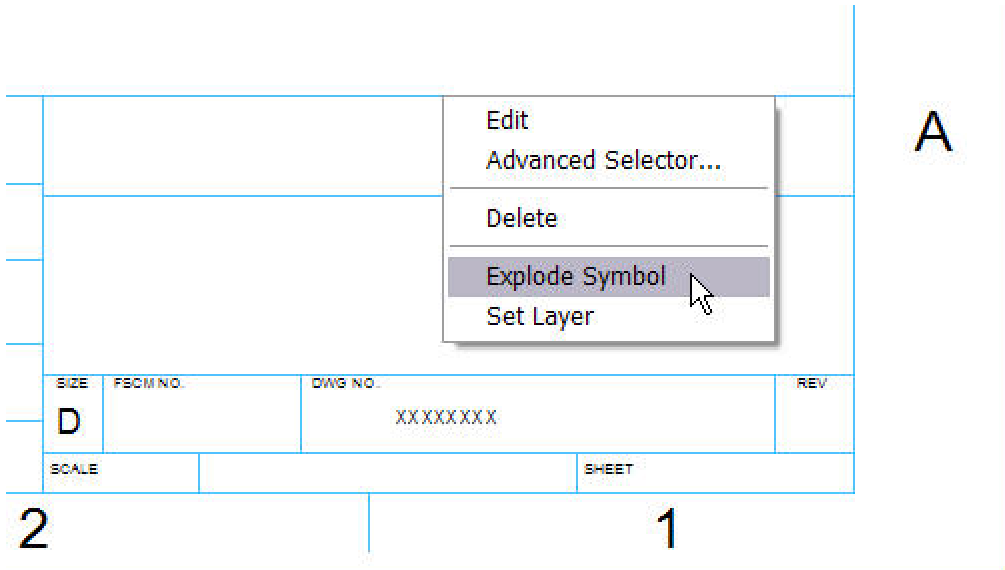
The same holds true for the Feature Control Symbols. Use the icon to open the 'Feature Control Frame' panel. Fill in the required information, click 'OK' and position the 'Feature Control' box as necessary.



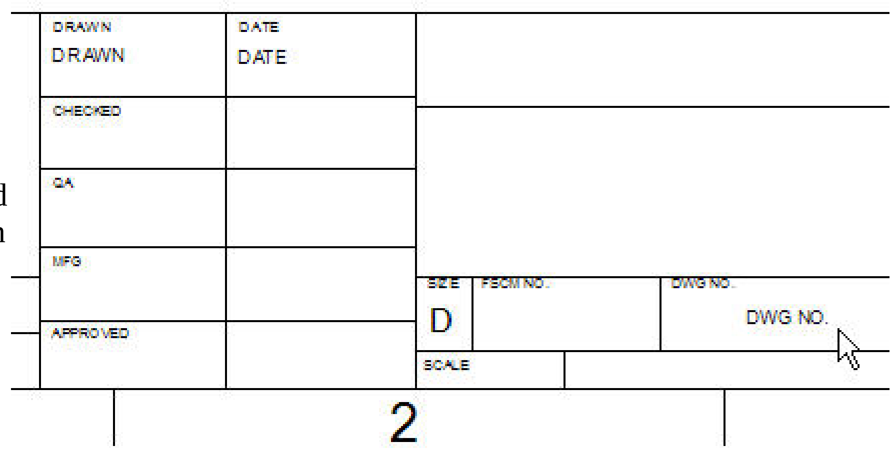
Editing is accomplished in the same way as for datum symbols. Double click on the feature control symbol, make your edits, and then click 'OK'.

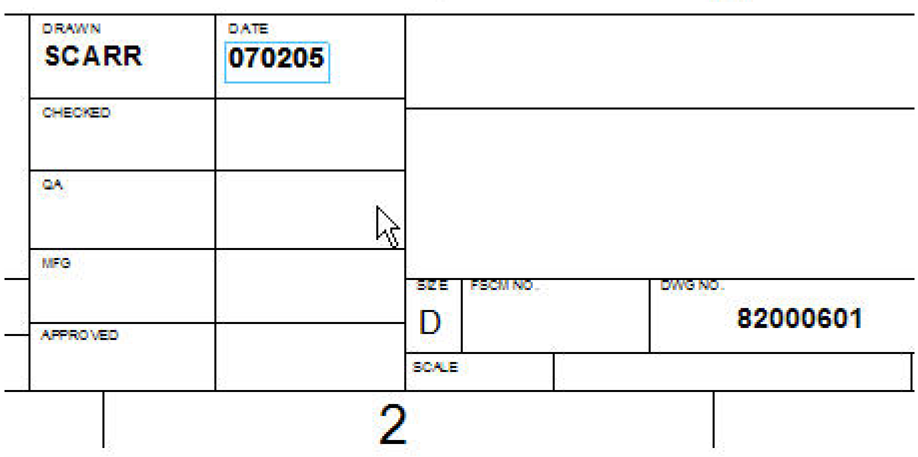


To edit the Title block, or other drawing frame text click on the drawing frame and select 'Explode Symbol'.



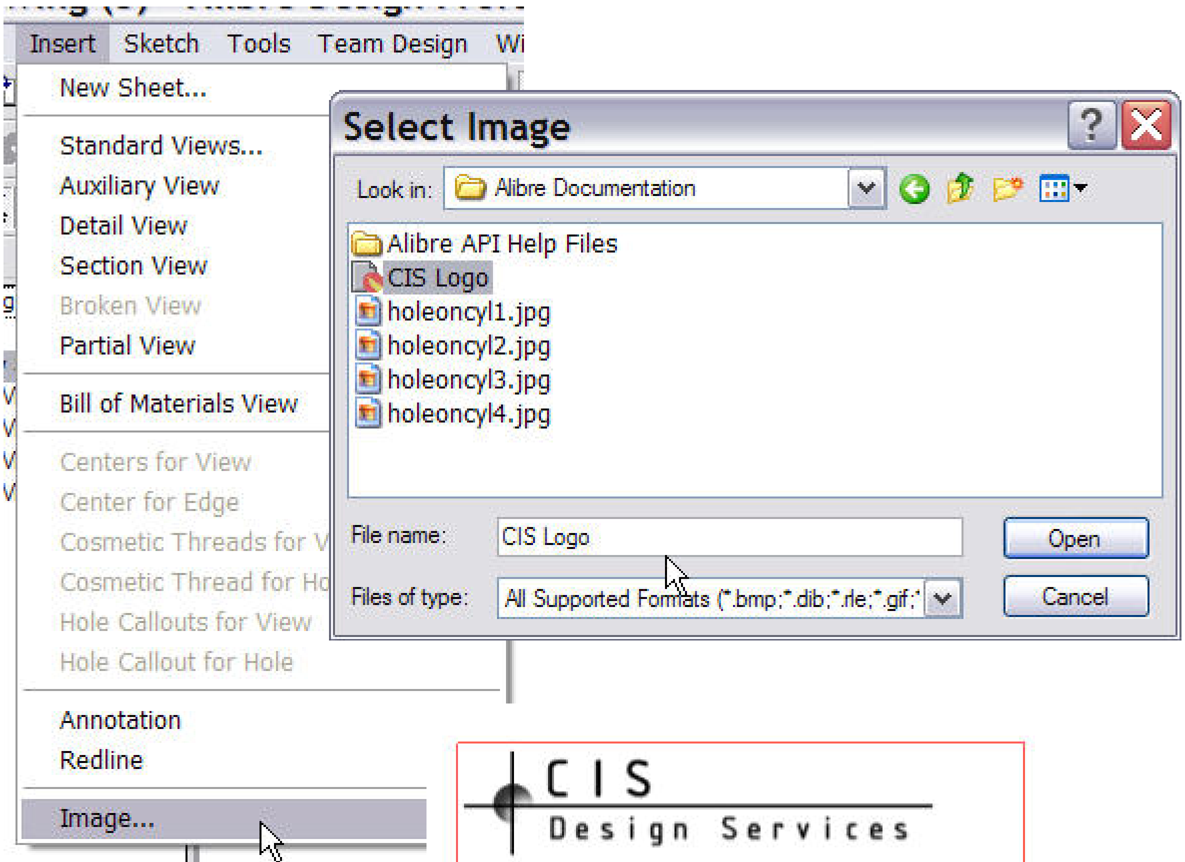
Note that when you explode the drawing frame all the text entities entered when you initially activated the drawing template return to their original designations.



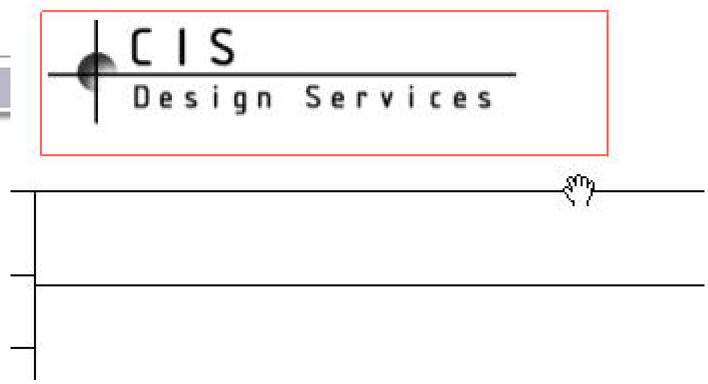


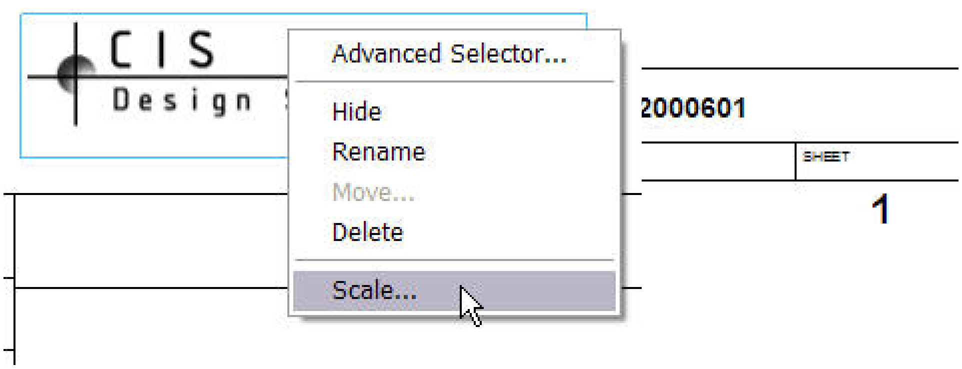
Edit the draw frame text just like you would any other text, by double clicking on it and then editing the text, setting font and, alignment properties.

You can insert images, such as company logos, in the drawing by clicking on the ‘Insert’ tab in the top toolbar and selecting ‘Image’. The ‘Select Image’ panel will open. Browse to the desired image file and click ‘Open’.

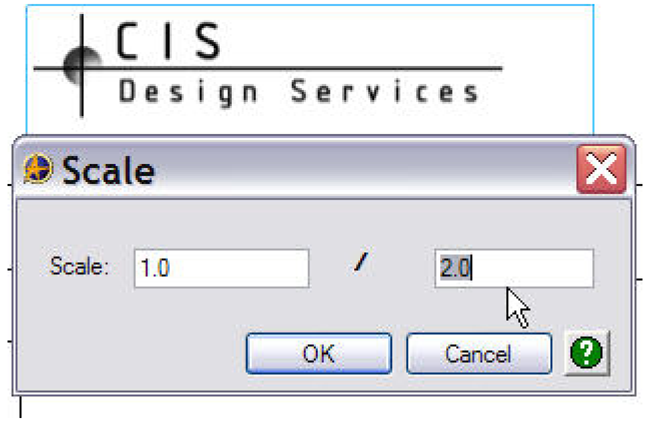


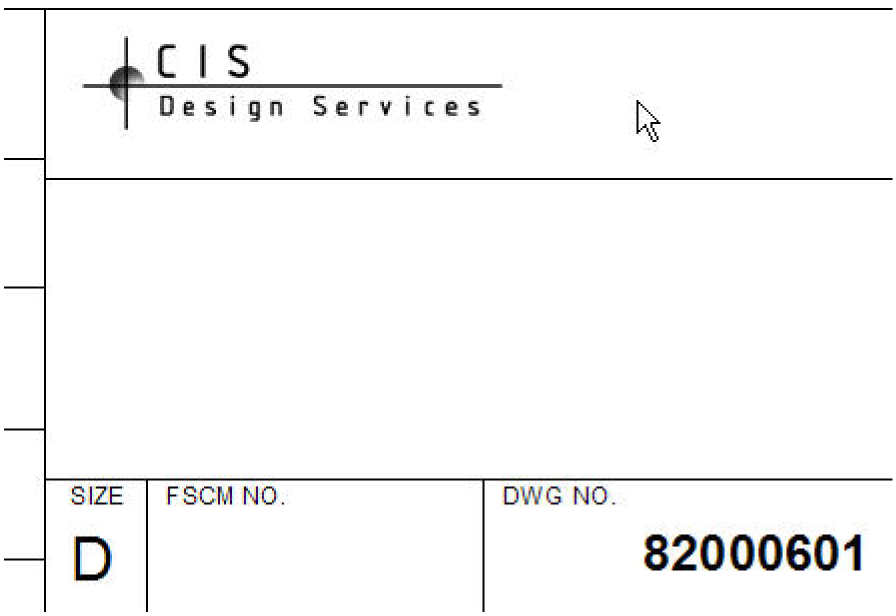
You can drag and drop the image where you need it. By right clicking on the image, you can also change its scale, rename it, hide it, or delete it.



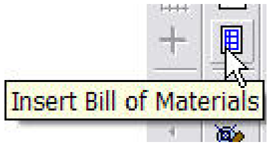


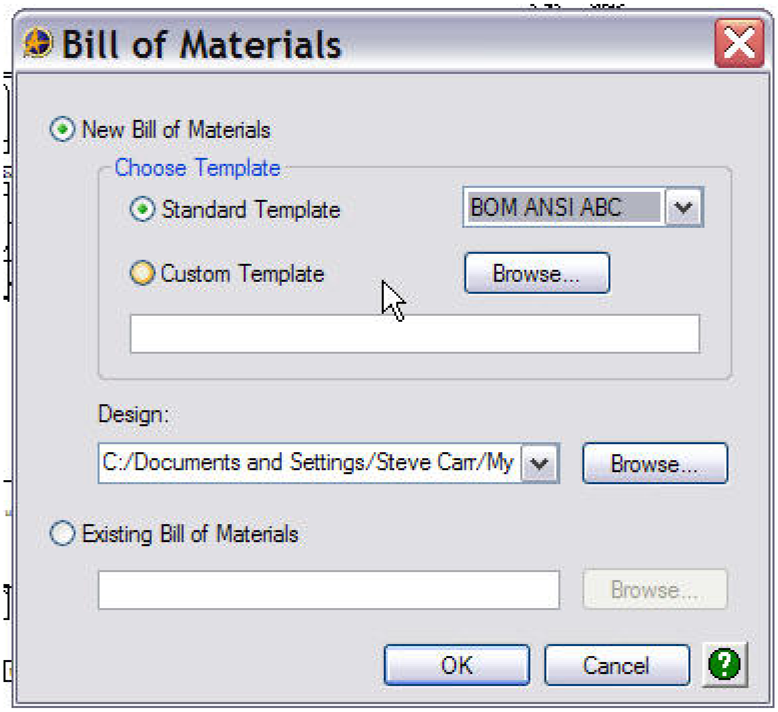
In this case, I've elected to change the scale of the image to get a better fit in the title block. The results are shown below.





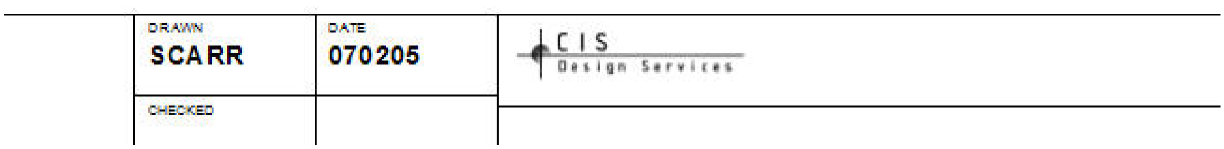
The next thing we need to do is insert a Bill of Material. Click on the ‘Insert Bill of Materials’ icon.





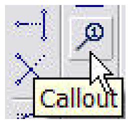
The ‘Bill of Materials’ panel opens. Select the appropriate template, and then click ‘OK’.

Item Number	Quantity	Part Number	Part Name	Revision	Comment
1	1	82000432	82000342		

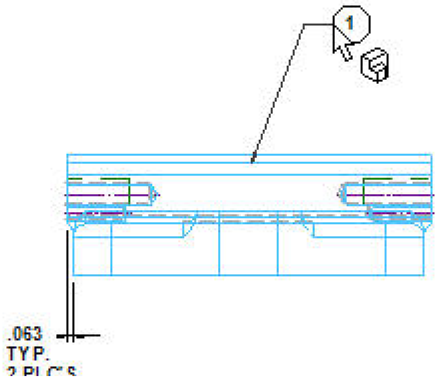


The Bill of Material will appear. Move it to the desired position and click once to place it. Save your file.

Add an item callout to the drawing by clicking on the 'Callout' icon

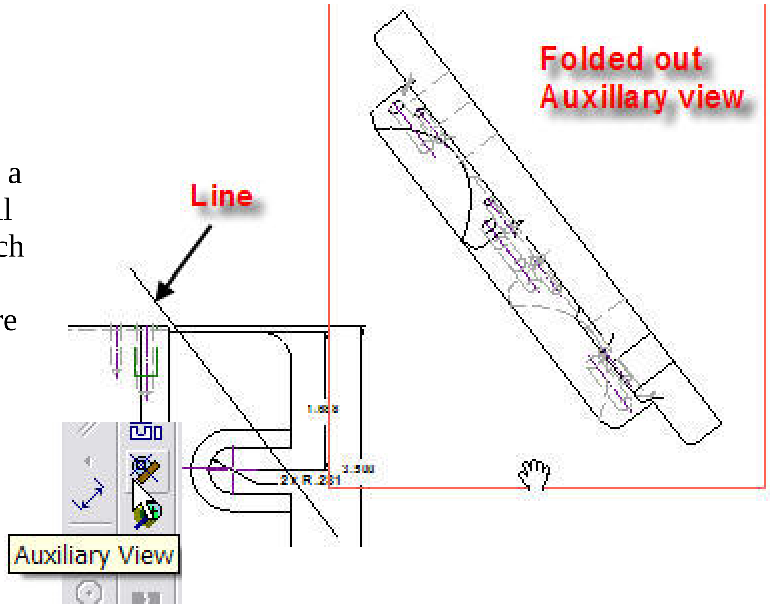


Click on the Saddle Base in any view and place the call out where desired. Click to place it. Item callouts automatically follow the items called out in the Bill of Material.



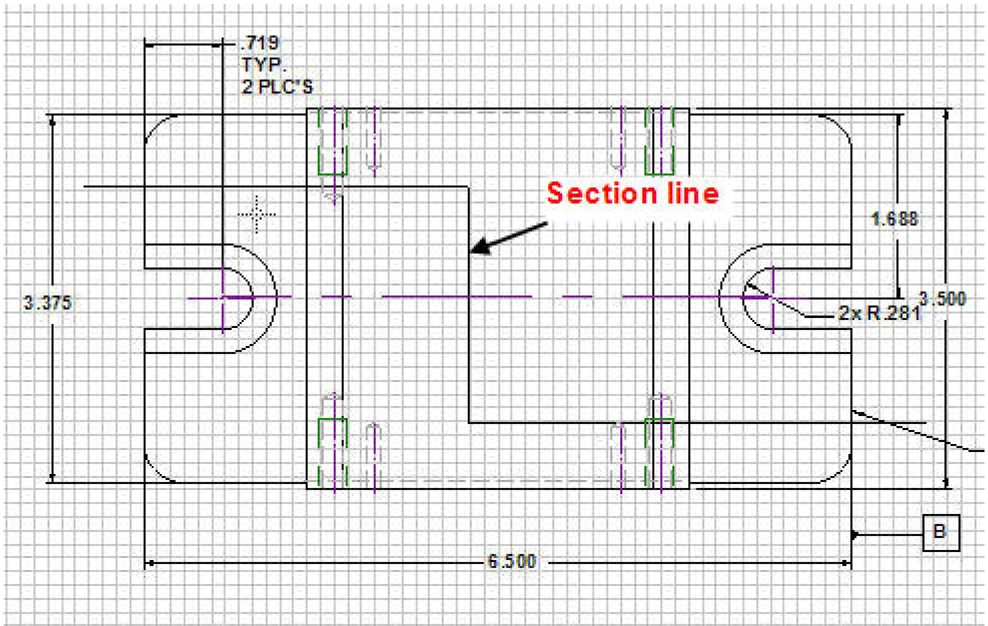
Although not required for detailing the Saddle Base, we'll insert a few of the other types of views available to us.

The first one we'll create is an 'Auxiliary' view. To generate an auxiliary view you'll need a straight-line entity that will act as the plane about which the view will be folded. This can be a model feature or as shown here a line drawn in 2D. Click the 'Auxiliary' view icon, and then click the line. Position the view and click to place it.

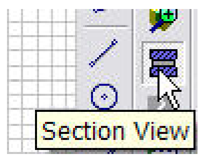


Successive auxiliary views can be created starting with the primary auxiliary view and folding the views out as required.

The next view we'll create is a 'Section View'. This view also requires one or a series of straight-line entities to define the cutting plane and direction. In this case, I've drawn a multi-section line to describe the cutting plane(s).

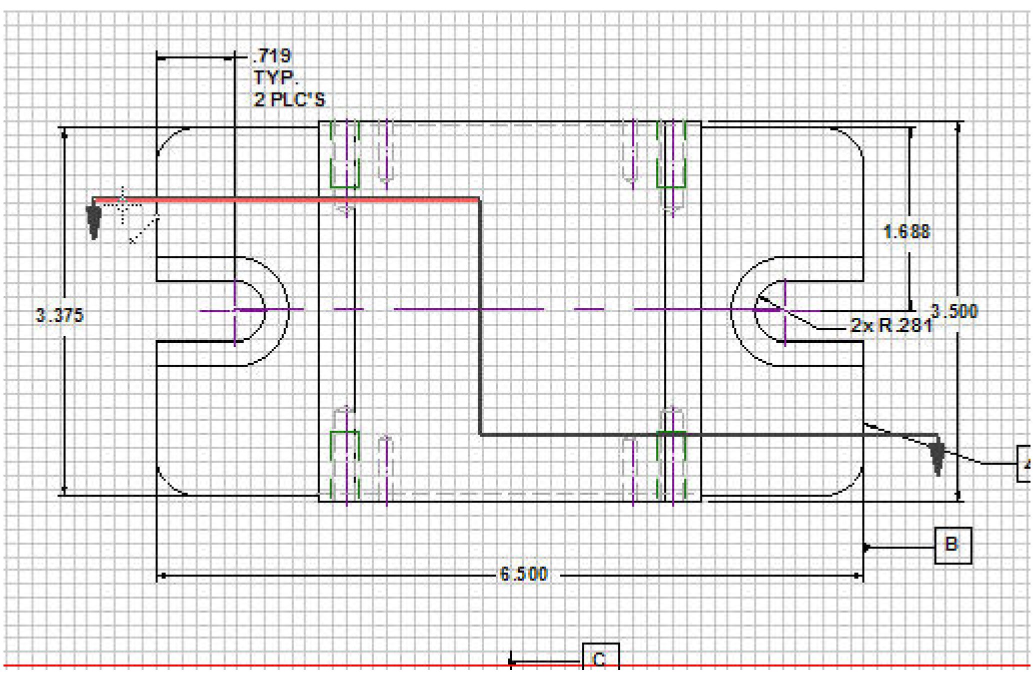


Click the 'Section View' icon.

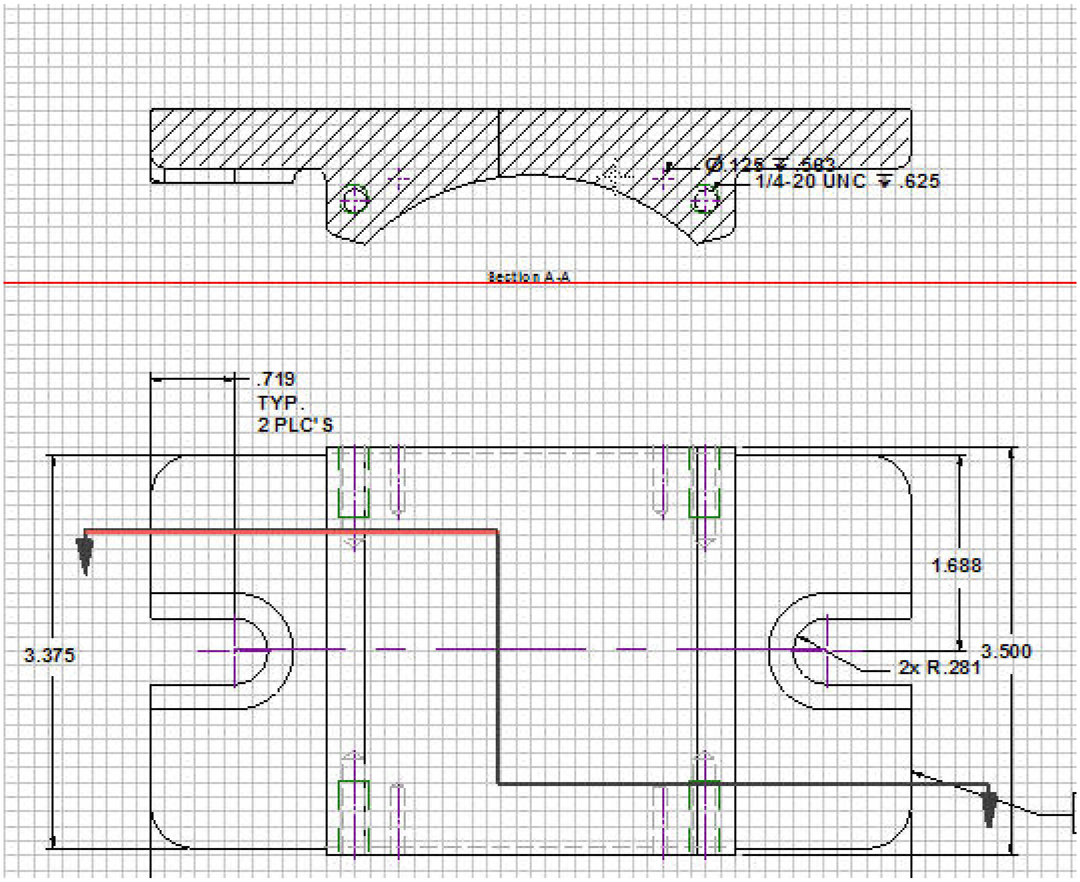


Then, select the section line.

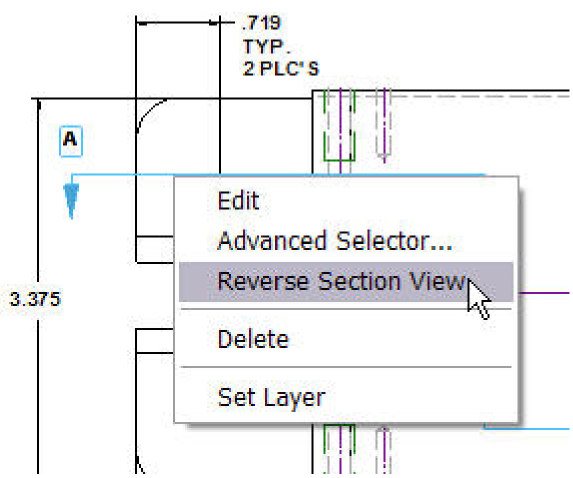
Section view arrows will appear.



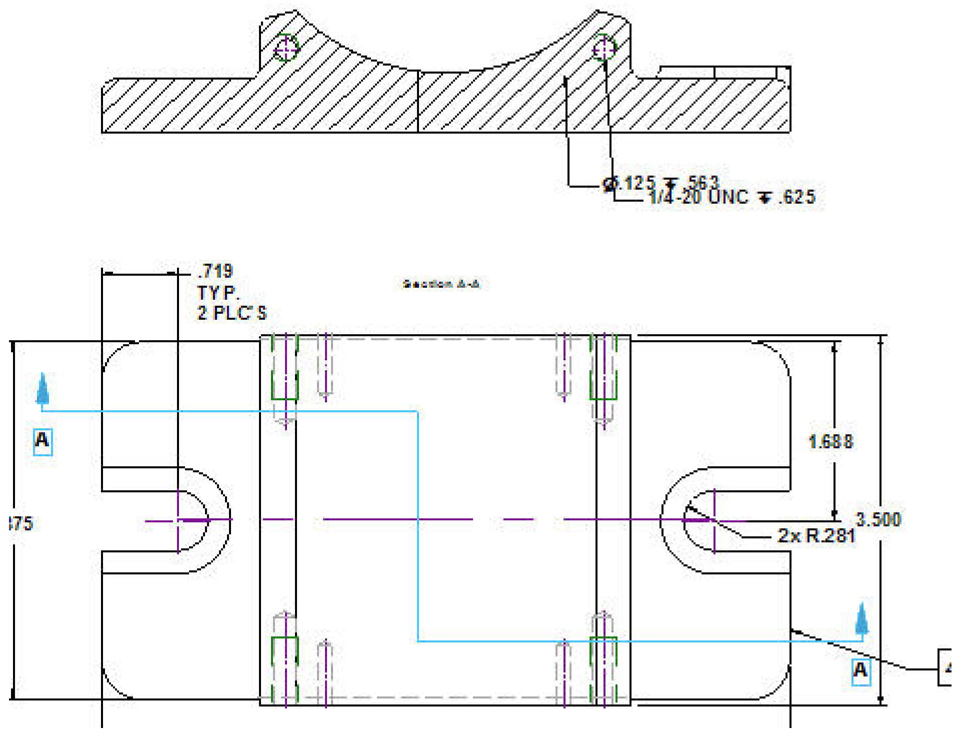
Click on the line and position the view. Click to place it.



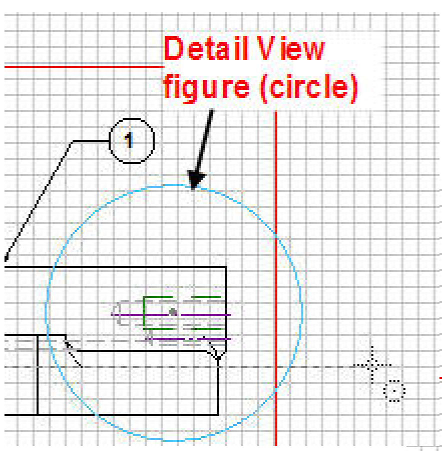
To change the direction of the section view, right click on the section arrow and select 'Reverse Section View'.

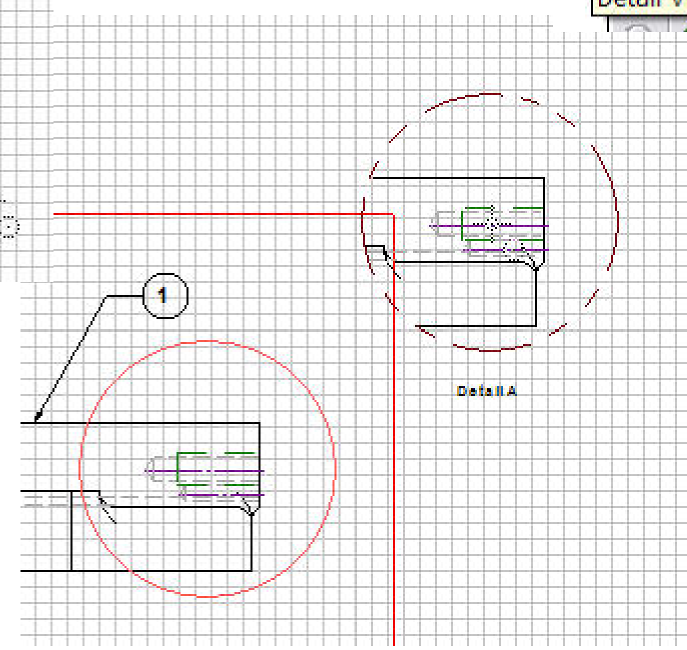


The view direction has been reversed. Position it and click to place it.

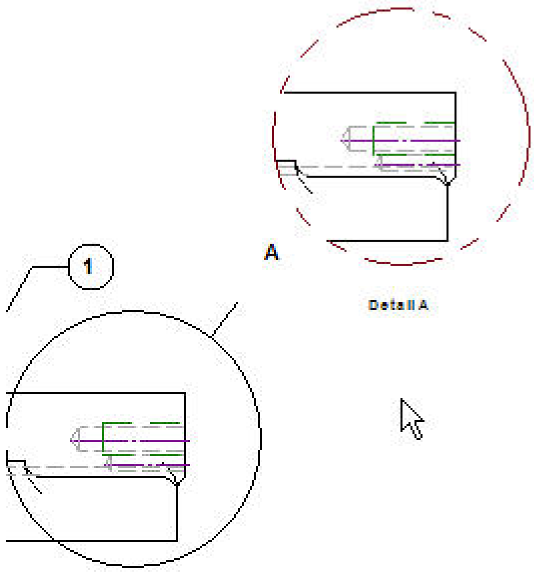


The next view type we'll create is a 'Detail View'. Before creating a detail view, we need to create a circle that will delineate the center and limits of the view. After creating the view element, click on the 'Detail View' icon, and then click the circle. Position the view as desired, and click to place it.

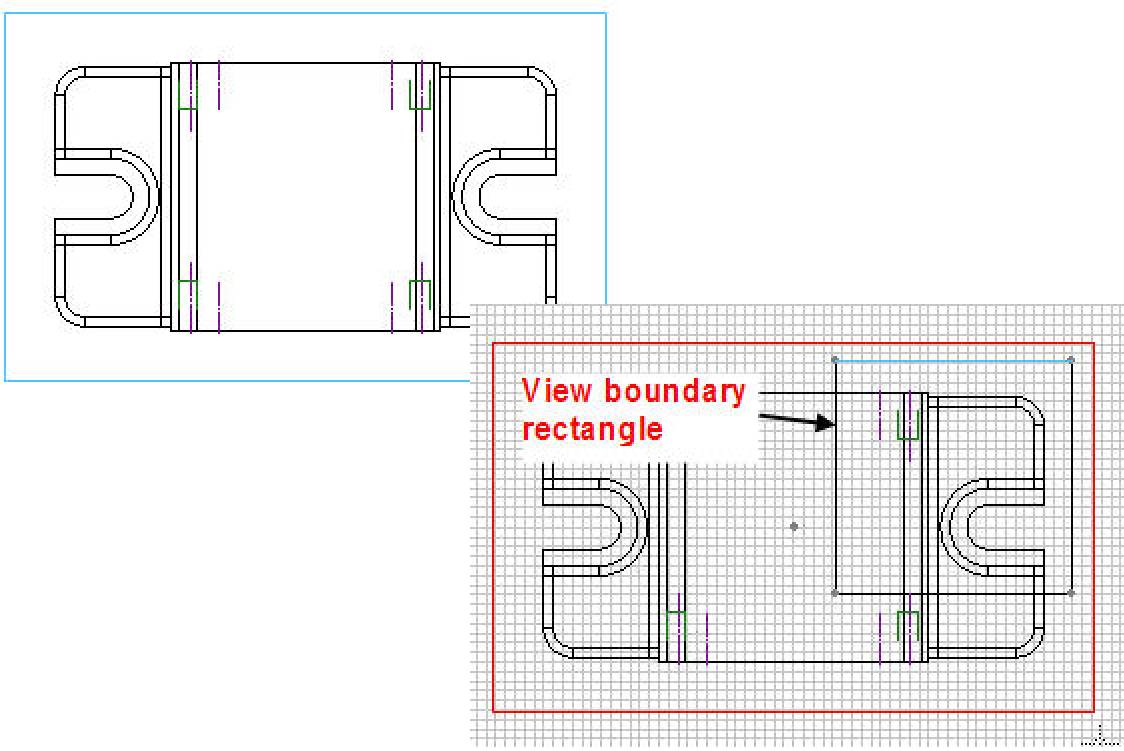


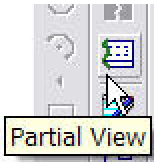


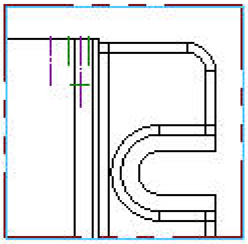
As with the other views, right clicking on the detail view will allow you to change many of its properties, including the view scale if desired.



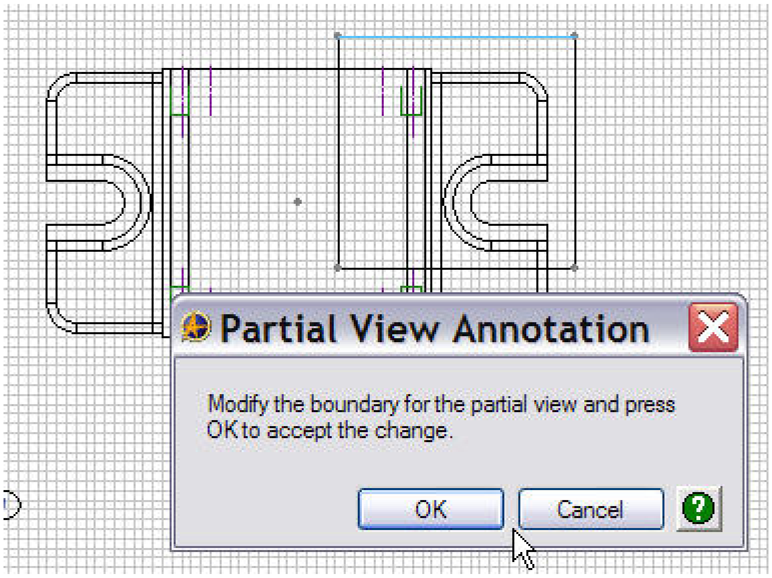
The last view type we'll create is the 'Partial View'. As in the case of the 'Detail View', a boundary must be defined before the view can de created. A duplicate 'top' view has been create for this example. Click the view to make it active and then sketch a boundary rectangle around the area of interest.



Click the ‘Partial View’ icon,  and then click the rectangle.



The ‘Partial View’ is created. You can edit the boundries of the view if you need to by right clicking on the view, and selecting ‘Edit’. The ‘Partial View Annotation’ panel will open. Make the desired changes and click ‘OK’.

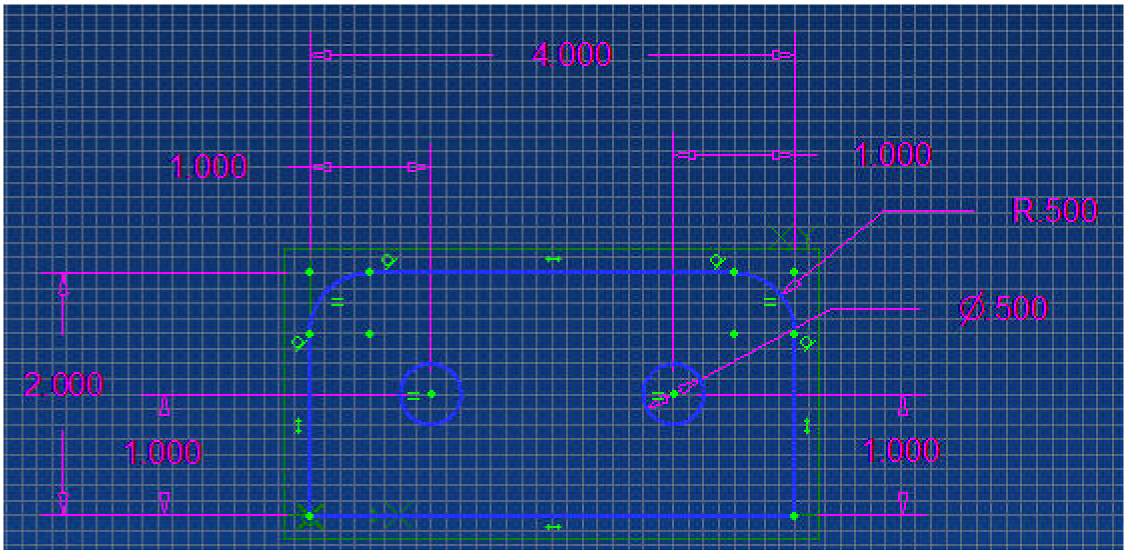


Although we haven’t exhausted the functionalities of the Alibre detailing application, this completes the drawing section of the Any Angle Vise Tool. For practice detail some or all of the other parts that make up the vise, using the original drawings as a guide and adding any other features you think necessary to complete them.

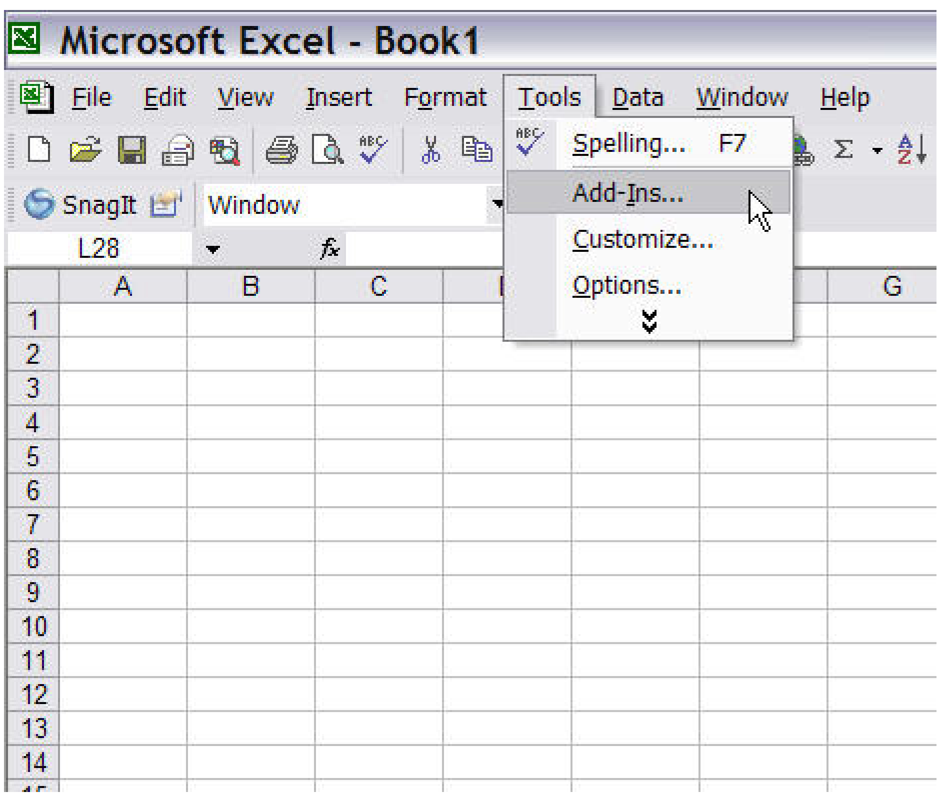
The next section deals with ‘Assembly’ practices. After completing the assembly of the vise tool, you’ll have a chance to create an assembly drawing, complete with an exploded view and BOM.

Chapter 11 –Using Alibre with Microsoft Excel to Drive Designs

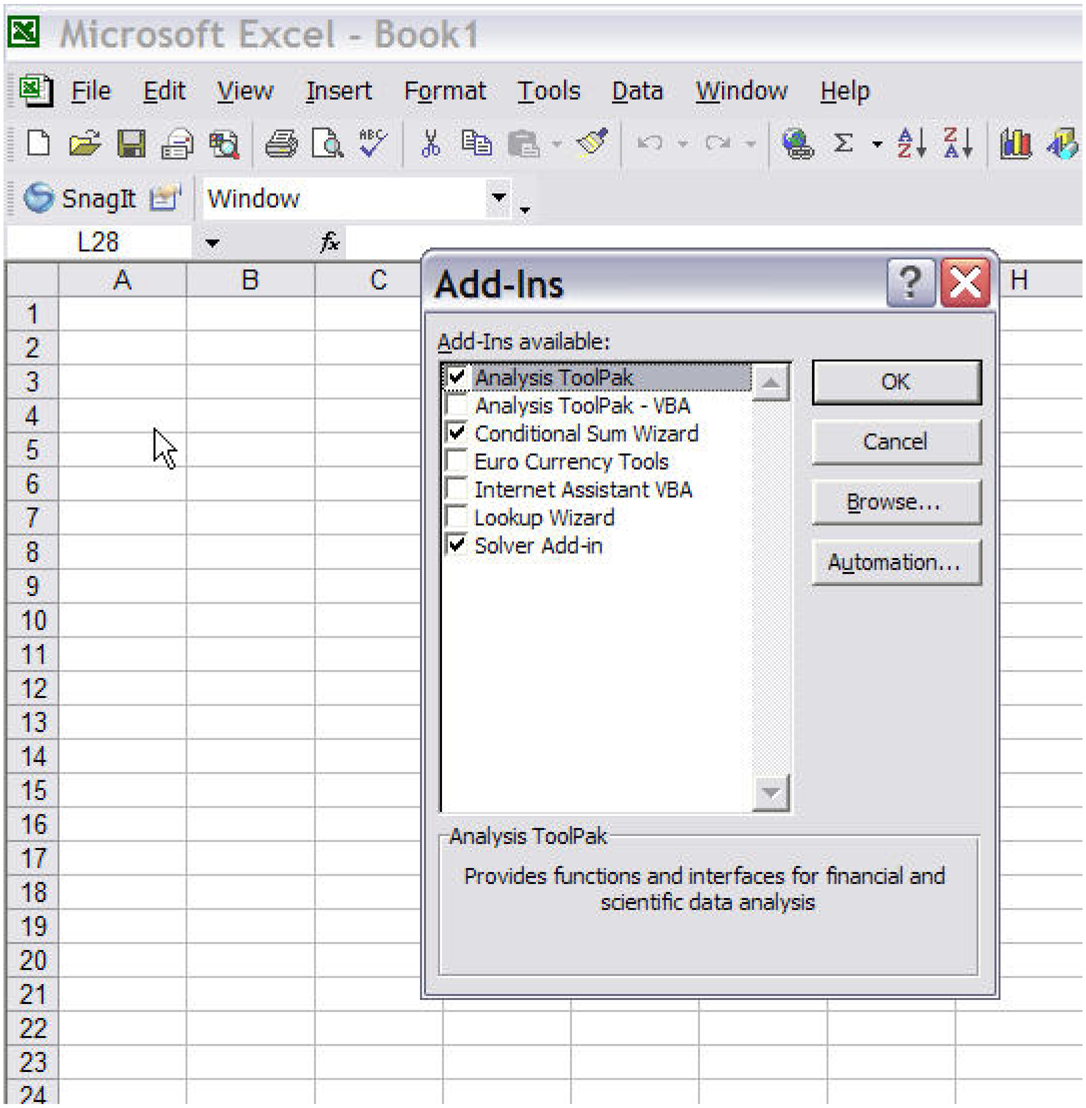
In chapter 2 it was stated that it was possible to use Alibre Design in conjunction with Microsoft Excel@ to drive design parameters within a model database. This section of the training manual will illustrate how this very powerful connection can be used to make your work easier and more efficient. Create the sketch below and file it under a logical name. There's no need to extrude it at



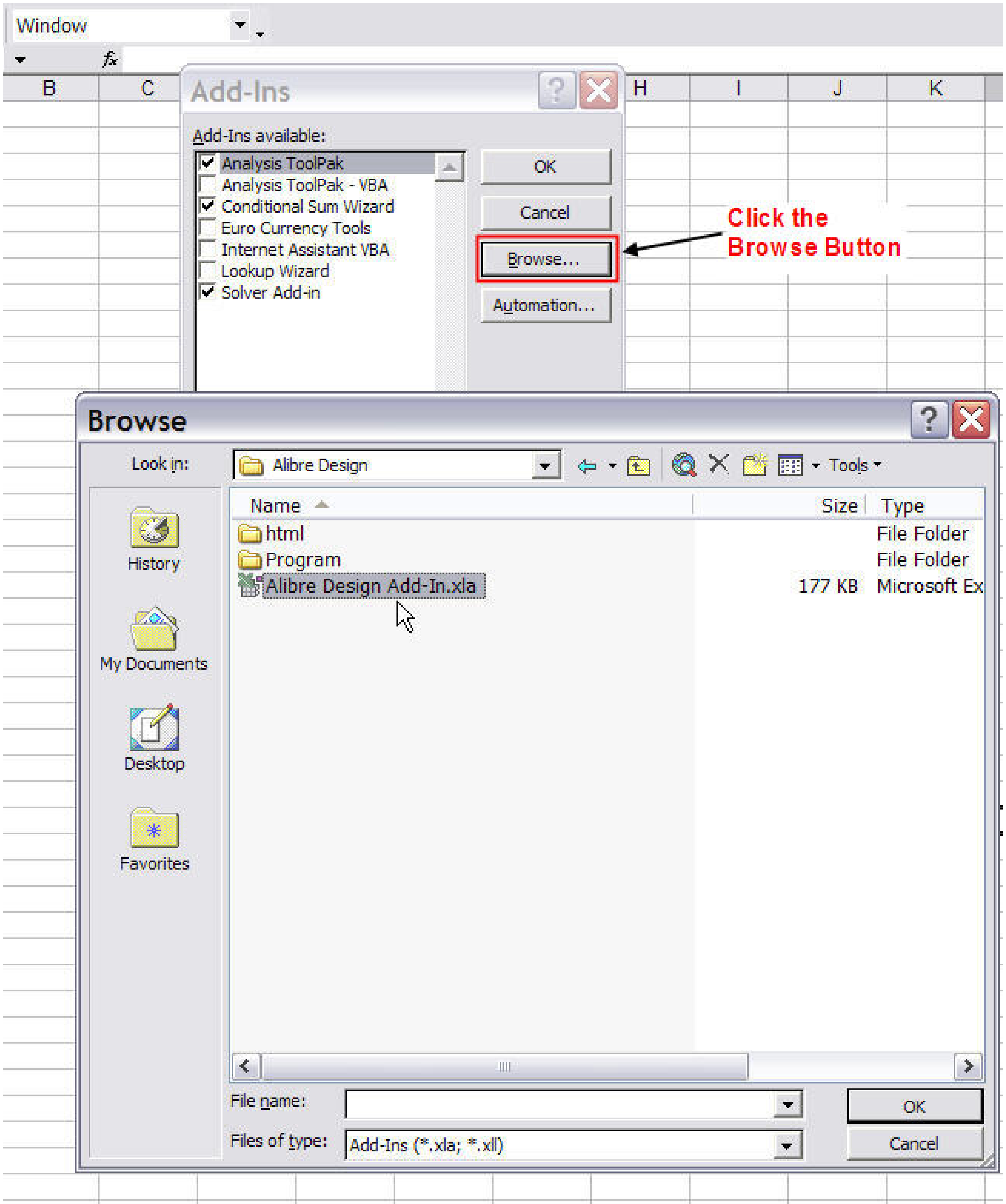
the present time. Now open Microsoft Excel. Alibre Design comes with an Excel extension that you may need to install before you can take advantage of this application. To check to see if the Add-On is already installed, click the 'Tools' tab on the top menu bar in Excel and scroll down and select 'Add-Ins'.



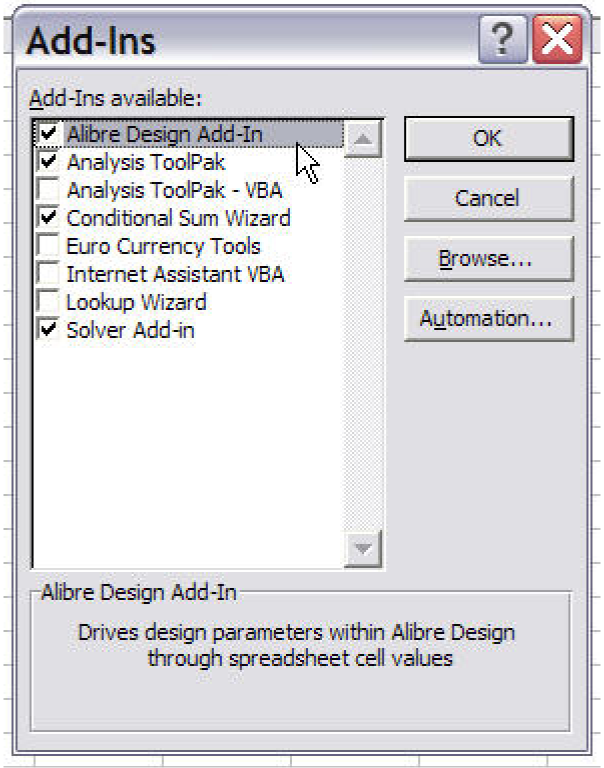
The ‘Add-Ins’ panel will open. If the Alibre Add-In isn’t present it will be necessary to load it.



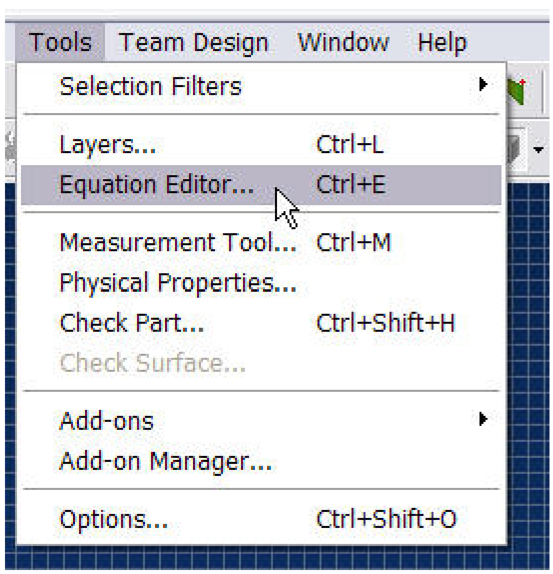
Click on the 'Browse' button and locate the Alibre Design Add-In.xls file in your Alibre Design program folder. Click on it and then click OK.

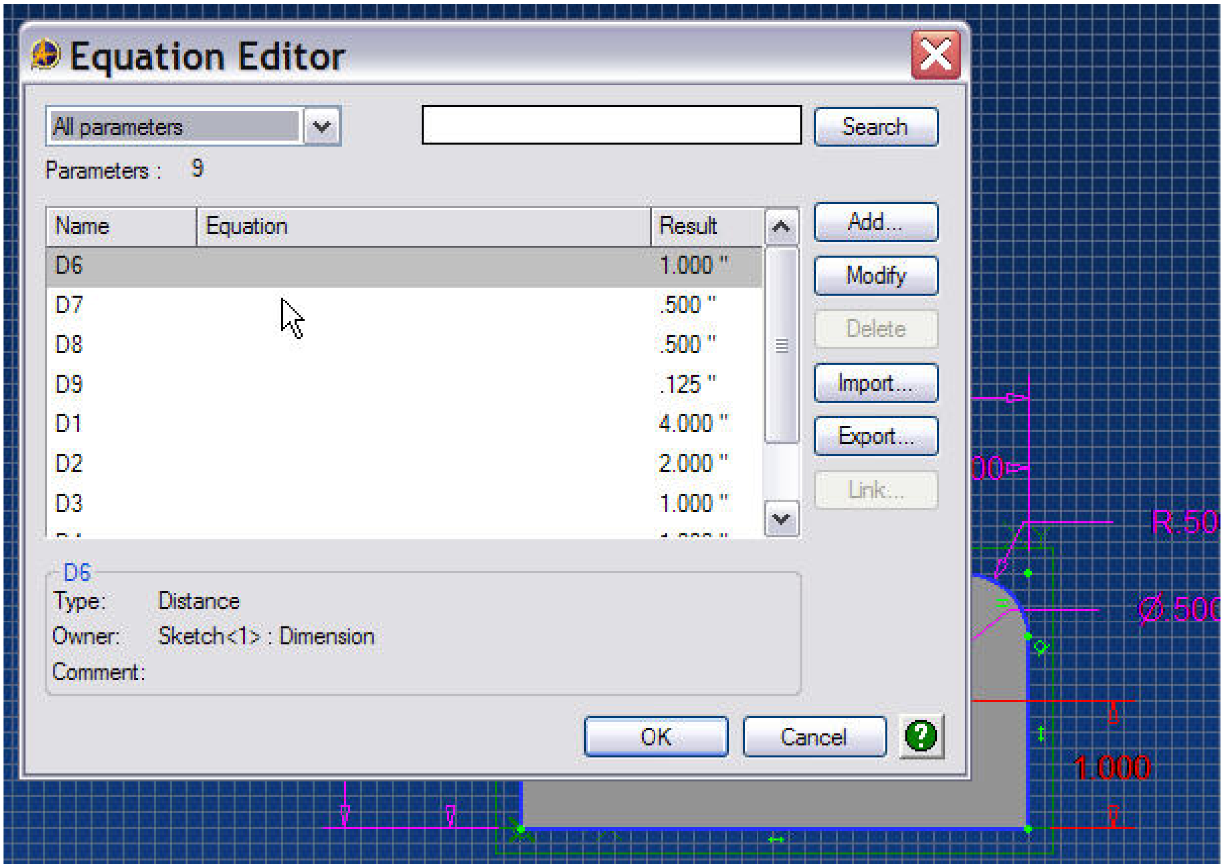


The Alibre Design Add-In should now appear in the 'Add-Ins' Panel. Click OK to add it to the Tools menu, and close the panel.



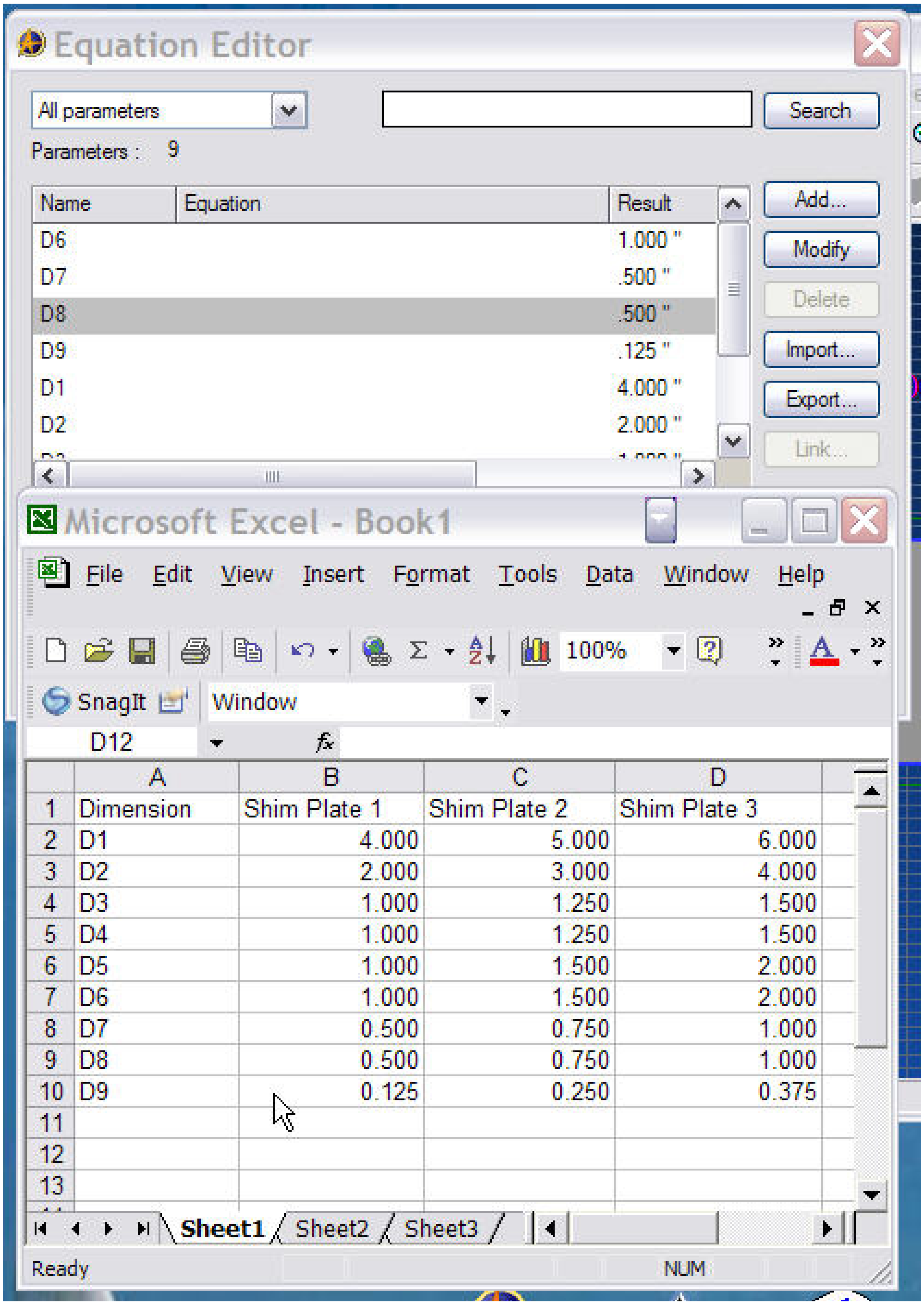
In the Alibre Design window, click on the Tools tab and select 'Equation Editor'.



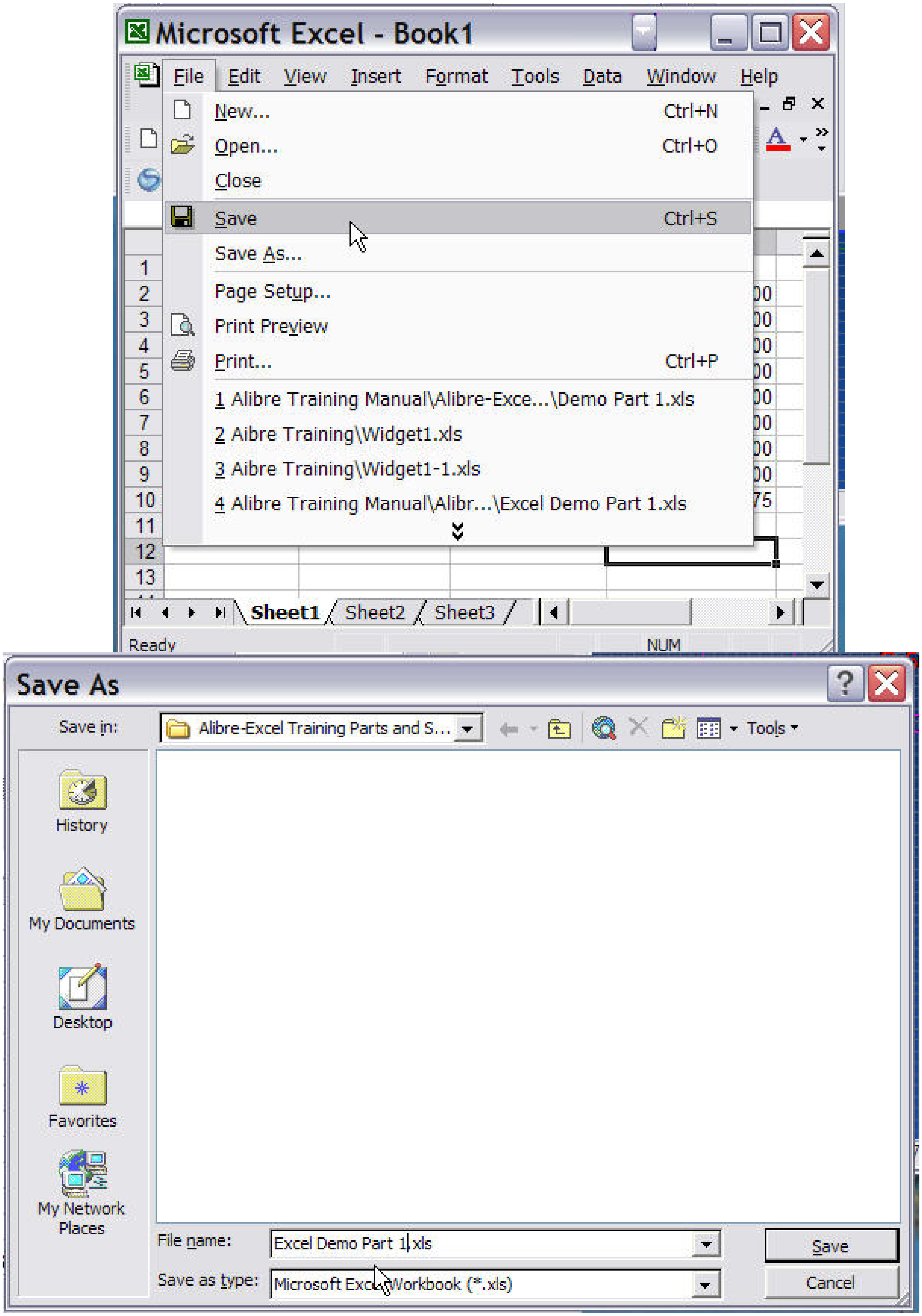


The ‘Equation Editor’ panel will open. You only need this panel open in order to insure the information you’ll enter in the Excel spread sheet matches what’s found in the model database.

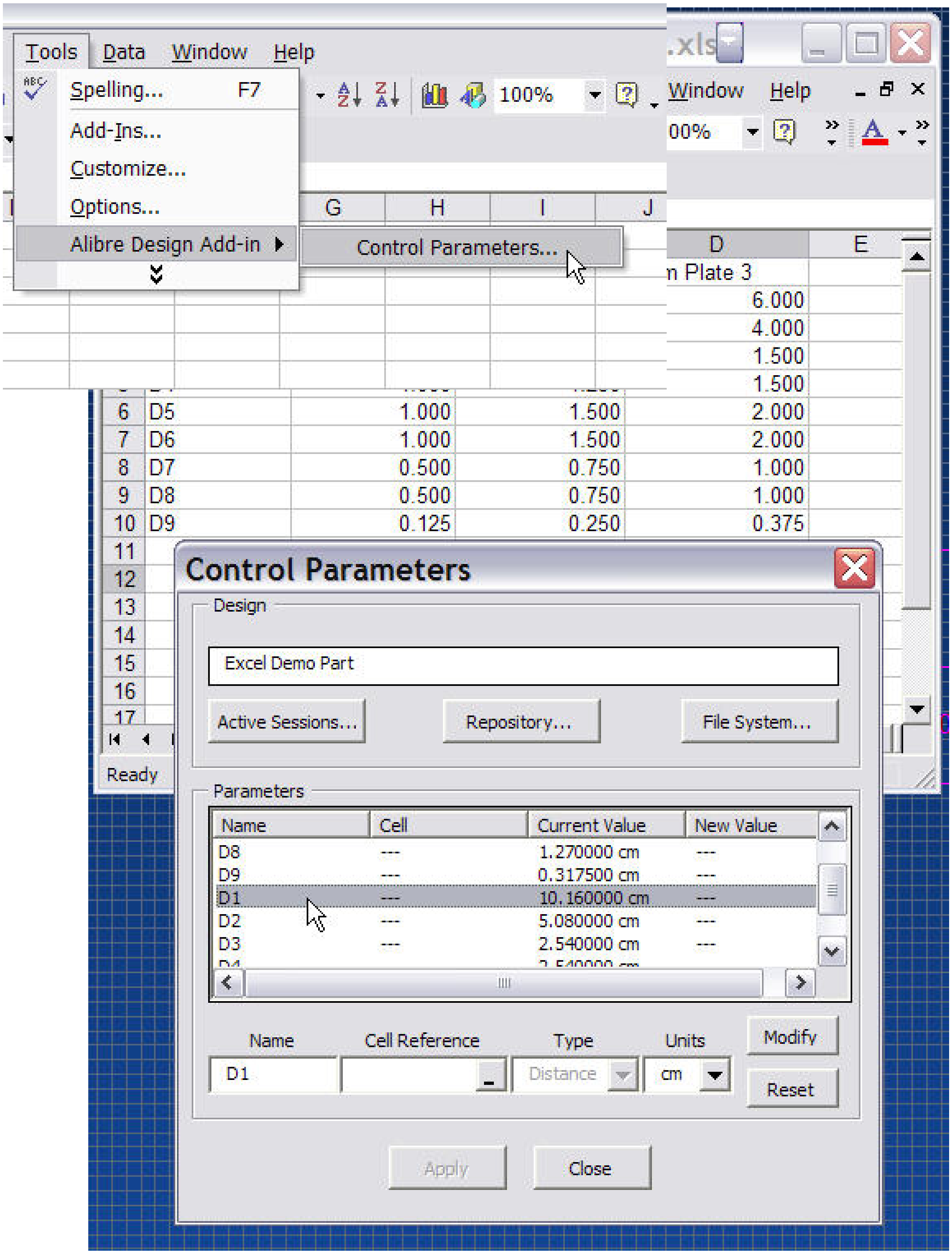
Open a new Excel spread sheet and enter the information from the Equation Editor panel as shown below. In this example, two alternate Shim Plate dimension lists have been created. These will be used to generate alternate plate models later in the exercise.



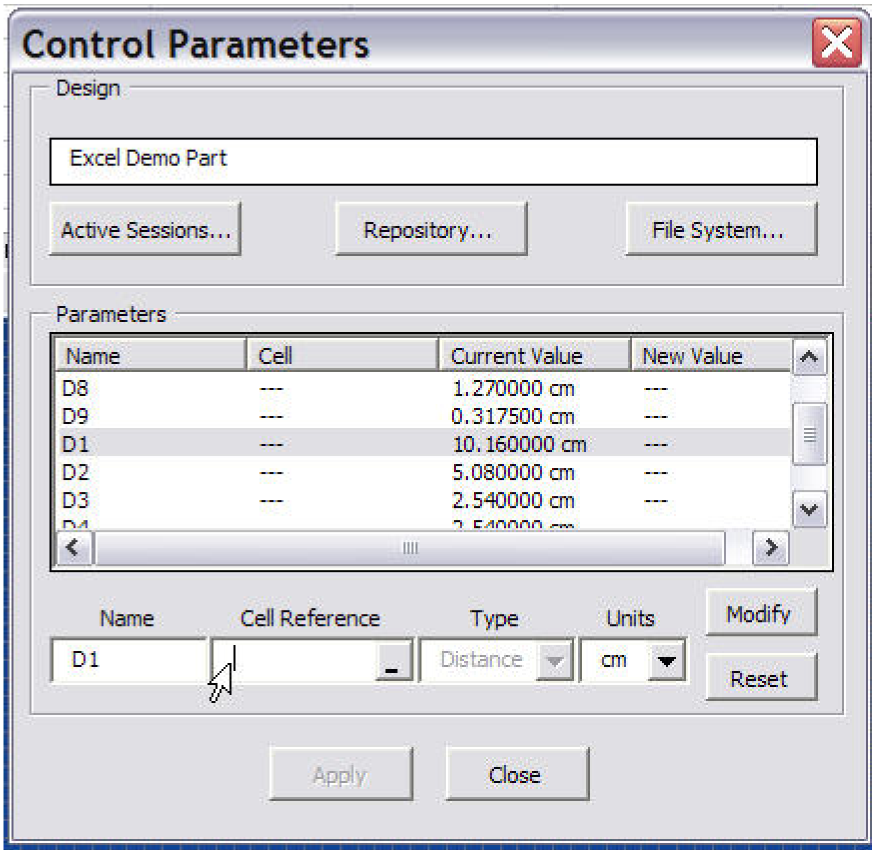
Save the Excel spread sheet file under a logical name in the same place you filed the part sketch you created.



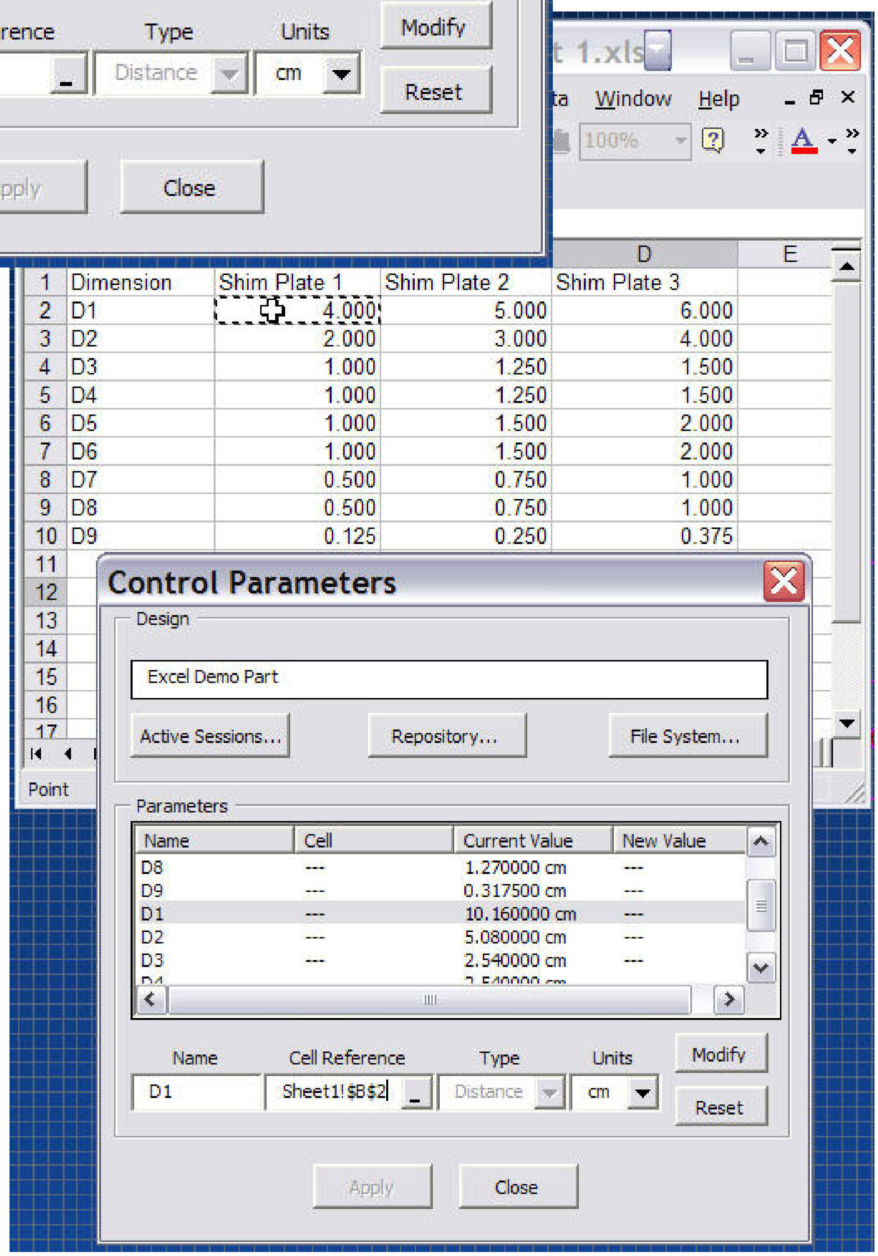
Now you'll need to tie the part and spread sheet together. Click on the 'Tools' tab in the Excel panel, scroll down and over to select the Alibre Design Add-In Control Parameters command. The 'Control Parameters' panel will open.



Note that the dimensions in the Parameters panel are all in centimeters. This is the default setting in Alibre. Since our part was created in inches it will be necessary to modify the dimensions to reflect this. Click on dimension D1 in the Parameters panel. It will highlight.

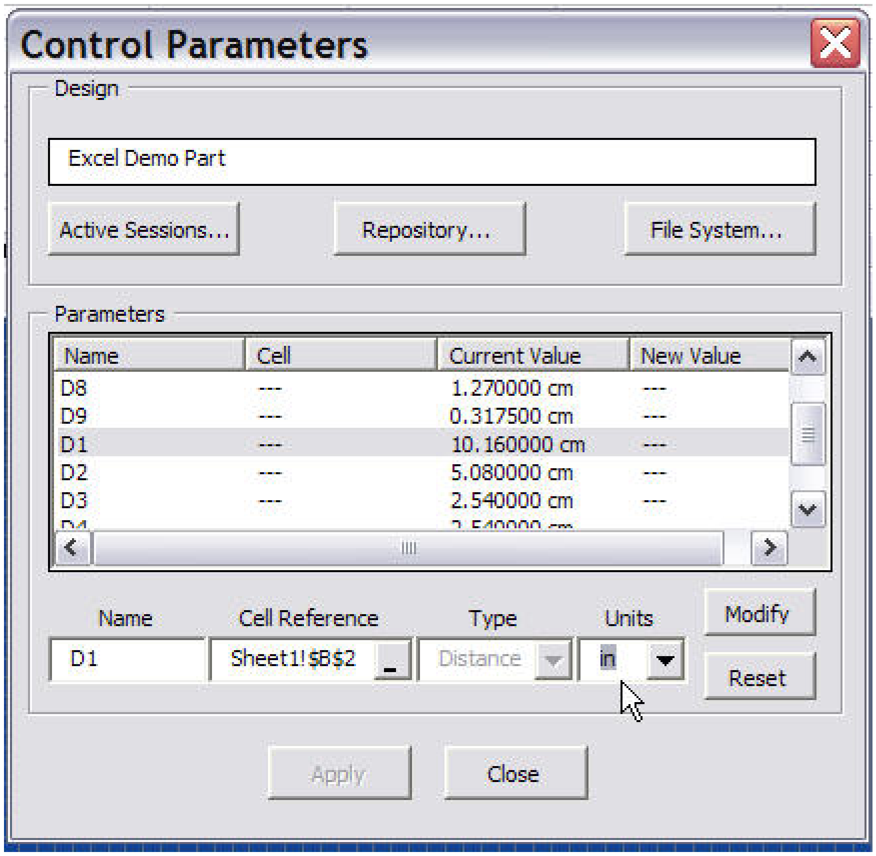


Next, click in the 'Cell Reference' text block to activate it, then click the 4.000 dimension in the Excel spread sheet.

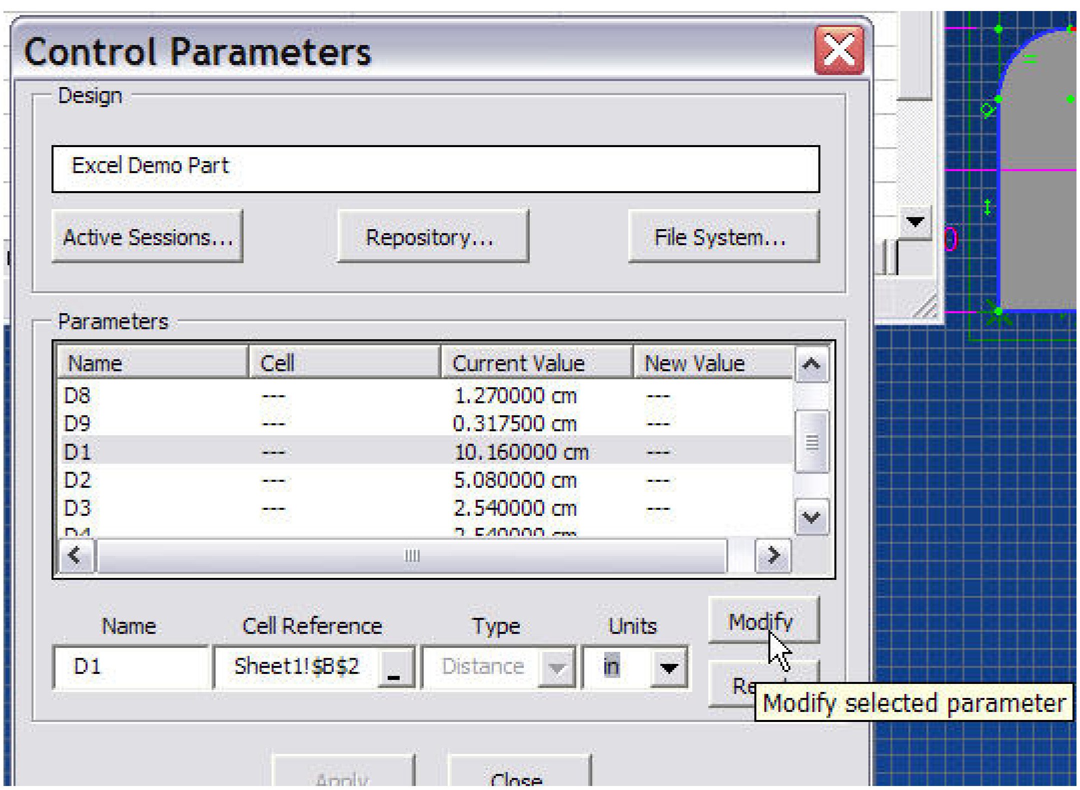


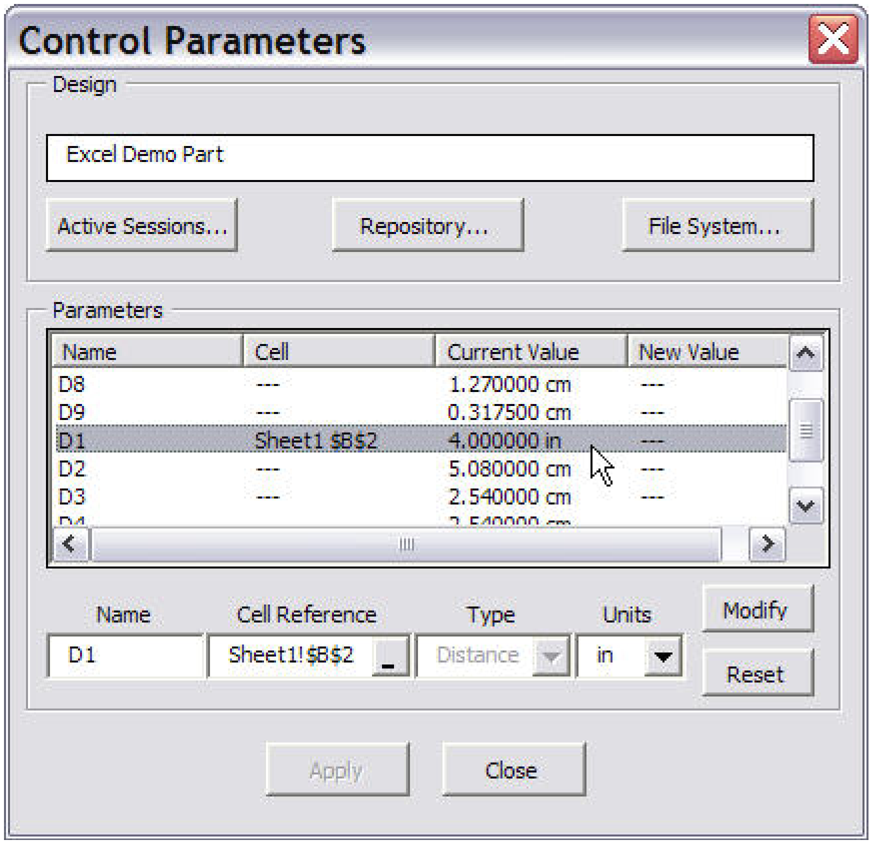
The Excel 'Cell' information will appear in the Cell Reference text block.

Click the arrow on the right side of the 'Units' text box and select inches.



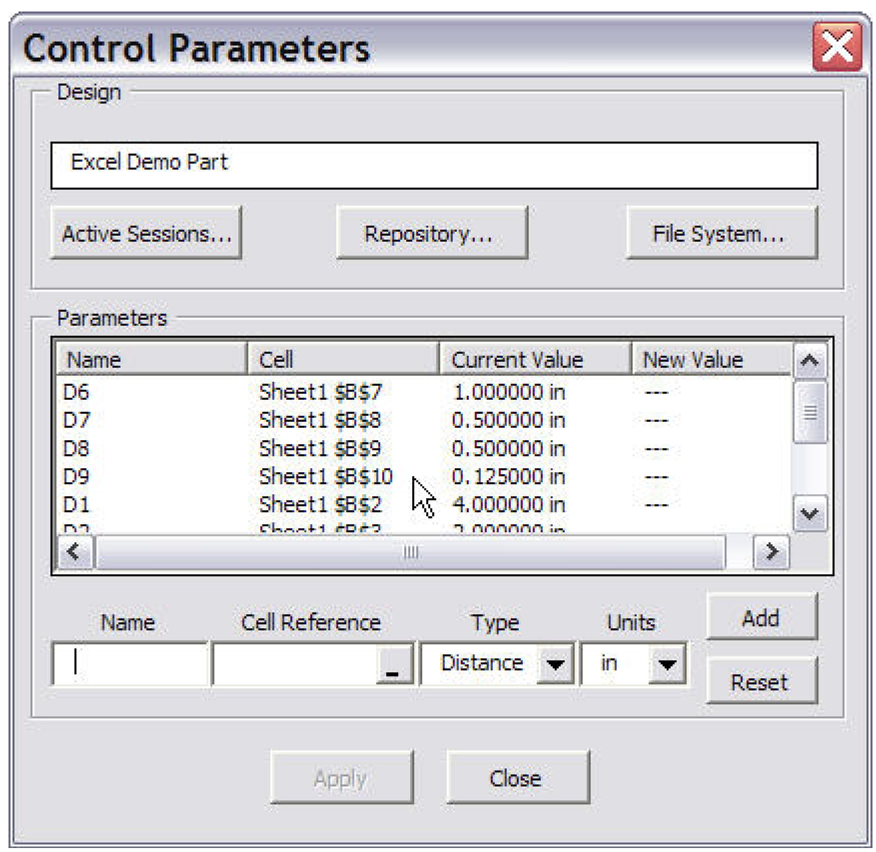
Then click the 'Modify' button.

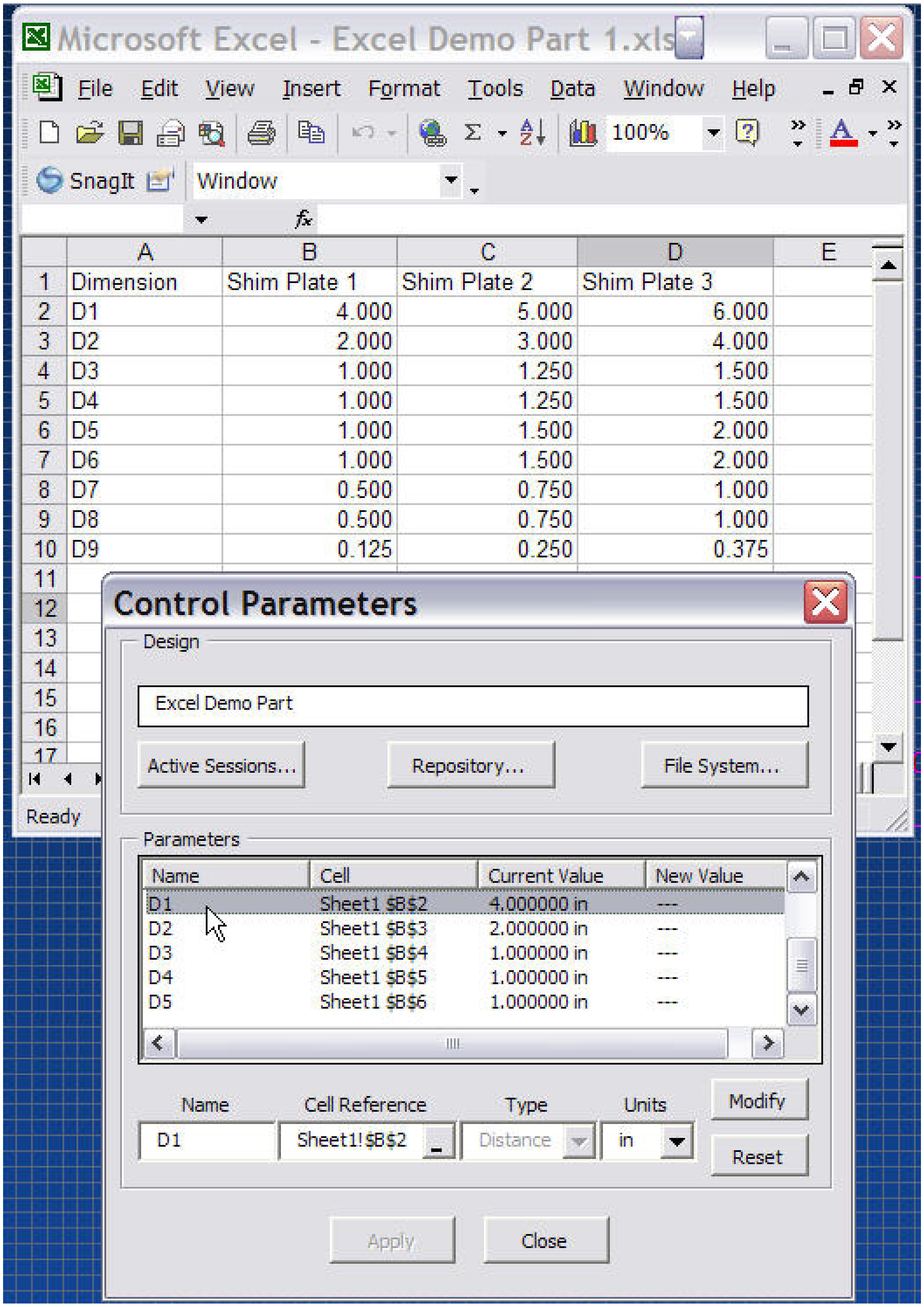




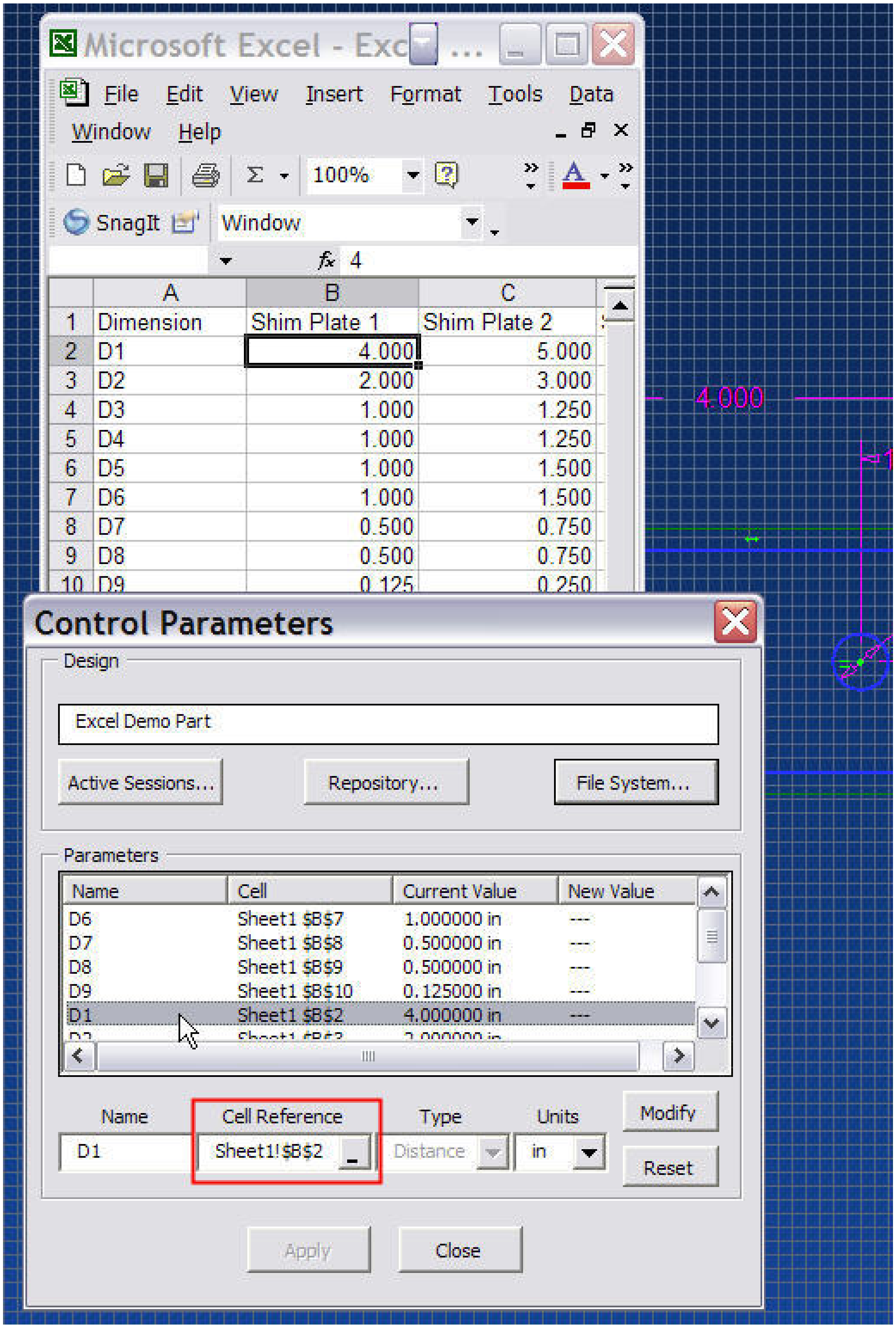
The D1 dimension has been modified.

Continue to do this with the rest of the dimensions, until they look like the ones shown here.

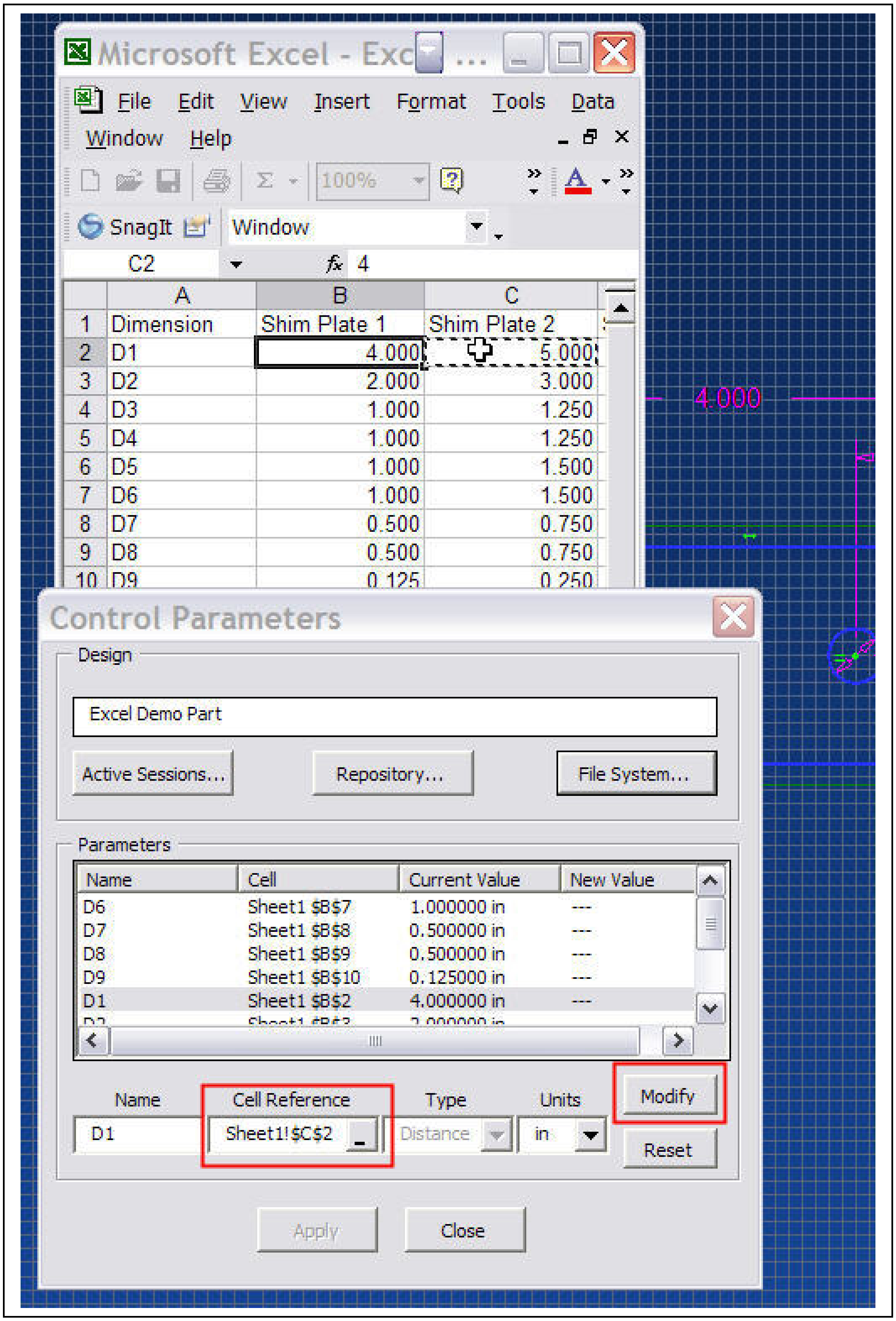



Microsoft Excel - Excel Demo Part 1.xls
File Edit View Insert Format Tools Data Window Help
100%
SnagIt Window
Dimension Shim Plate 1 Shim Plate 2 Shim Plate 3
D1 4.000 5.000 6.000
D2 2.000 3.000 4.000
D3 1.000 1.250 1.500
D4 1.000 1.250 1.500
D5 1.000 1.500 2.000
D6 1.000 1.500 2.000
D7 0.500 0.750 1.000
D8 0.500 0.750 1.000
D9 0.125 0.250 0.375
Ready
Control Parameters
Design
Excel Demo Part
Active Sessions...
Repository...
File System...
Parameters
Name Cell Current Value New Value
D1 Sheet1 B2 4.000000 in ---
D2 Sheet1 B3 2.000000 in ---
D3 Sheet1 B4 1.000000 in ---
D4 Sheet1 B5 1.000000 in ---
D5 Sheet1 B6 1.000000 in ---
Name
Cell Reference
Type
Units
D1
Sheet1!B2
Distance
in
Modify
Reset
Apply
Close

To see how Excel can drive design dimensions in Alibre, click the D1 dimension again. Click in the 'Cell Reference' block to make it active.

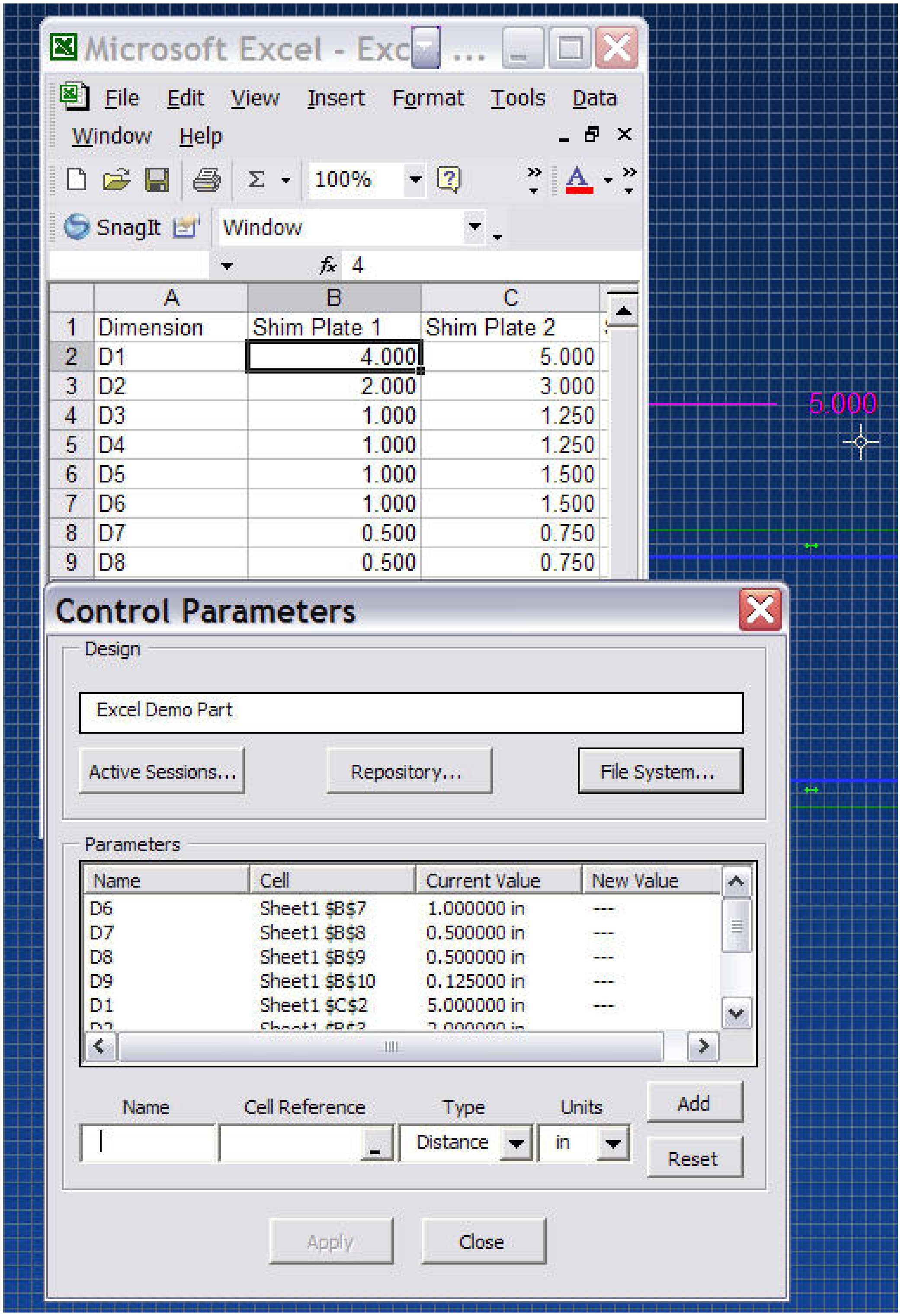


Click the 5.000 dimension cell in the Excel spreadsheet

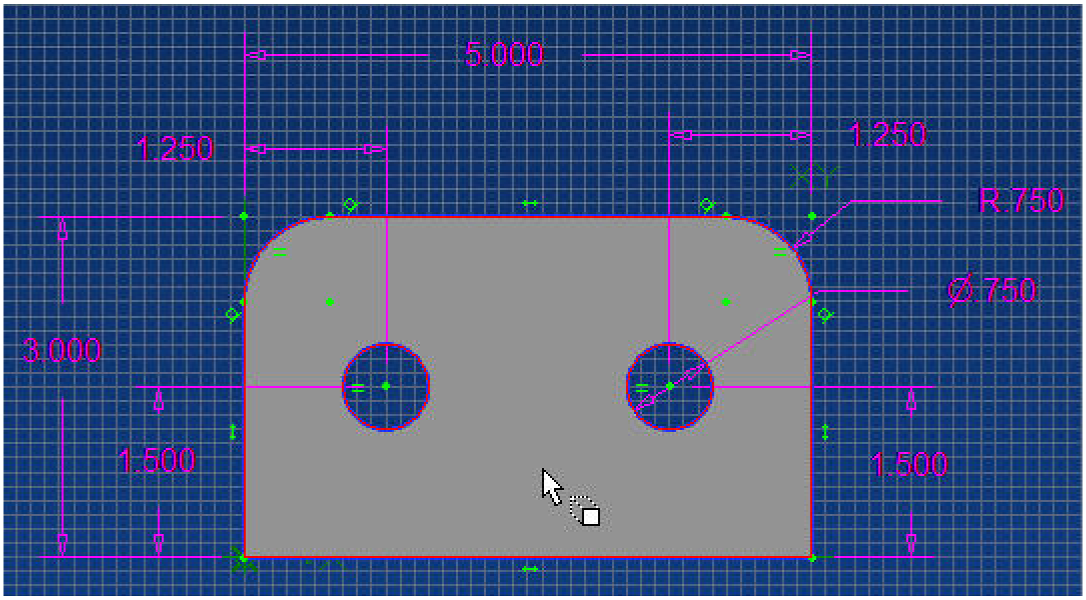


The Cell Reference information changes to reflect the 5.000 cell. Click Modify.

The 4.00 inch dimension is now 5.000 inches. Change the other dimensions per the information in the Shim Plate 2 dimensions column and save the spreadsheet.



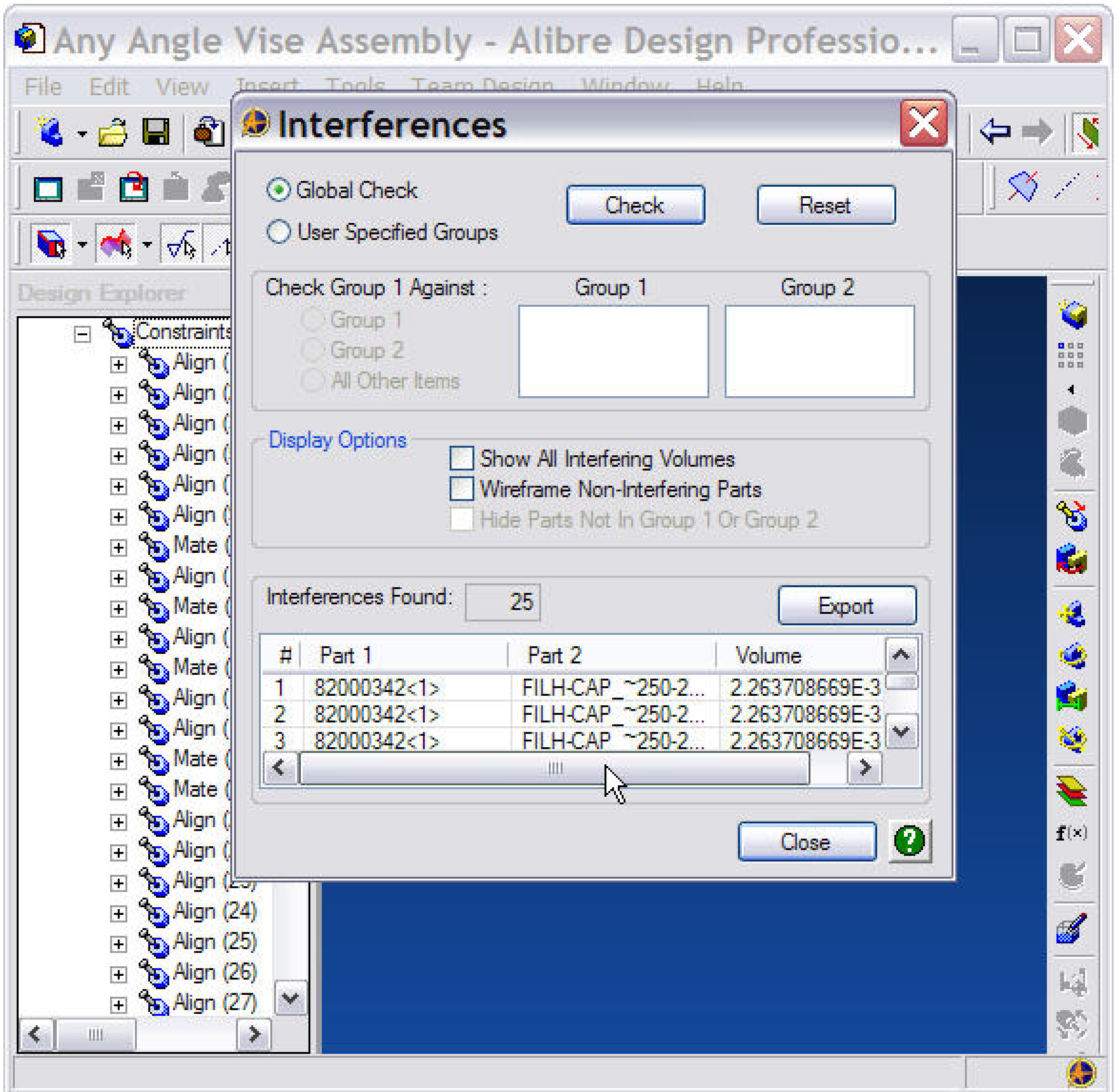
Save the modified part under a logical name, (not the same one you used for Shim Plate 1). The sketch should now look like this.



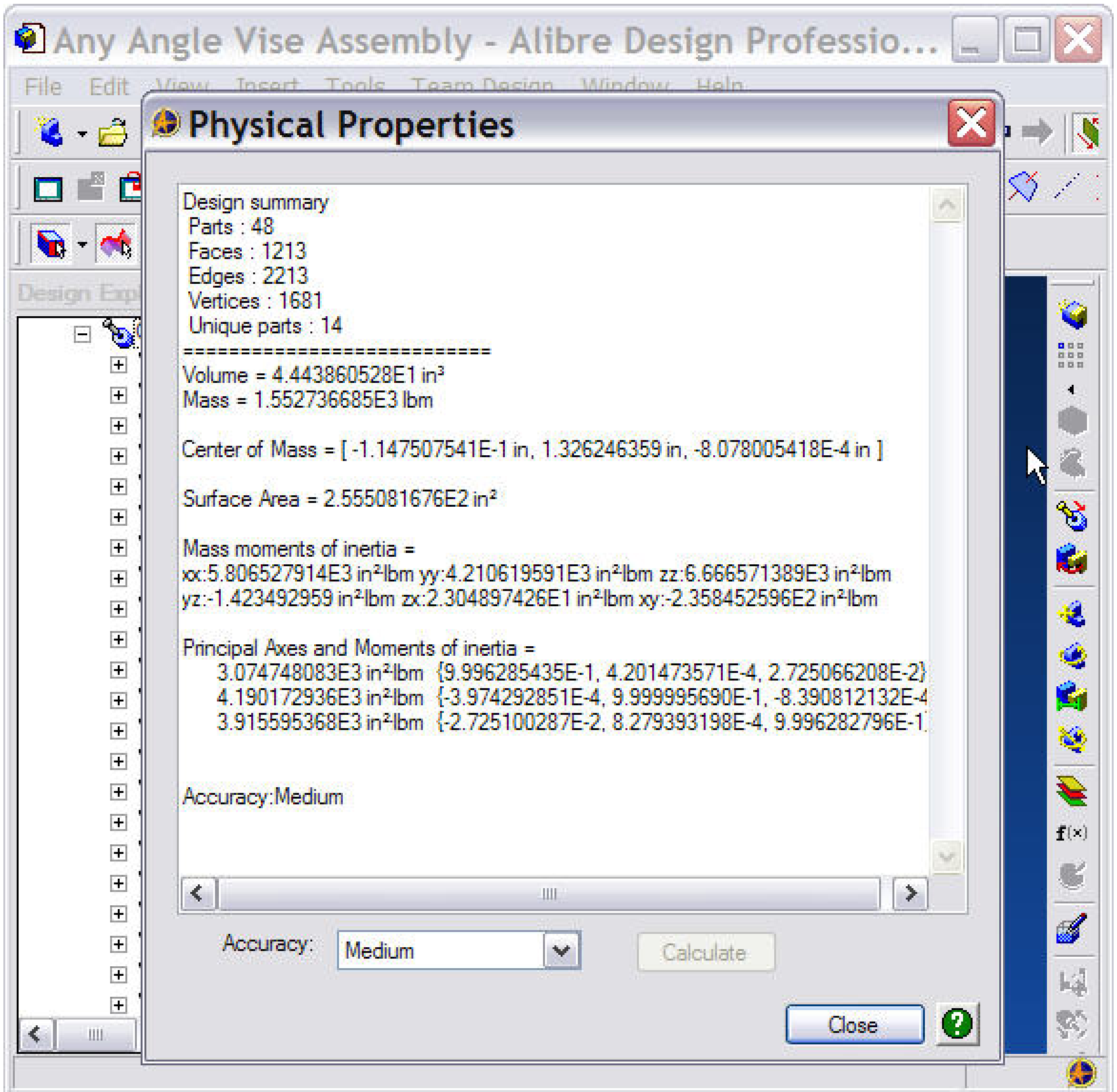
You can use this Alibre-Excel connectivity tool to drive Distance, Angle, Count and Scale parameters in your design, eliminating a great deal of repetitious design work. You owe it to yourself to become intimately familiar with this very powerful tool.

Chapter 12-Creating Assemblies in Alibre

Assembly Parts are such an important part of the design verification process that their value can't be overstated. In an assembly part, interference conditions show up that might otherwise go undetected in a simple 2D layout.

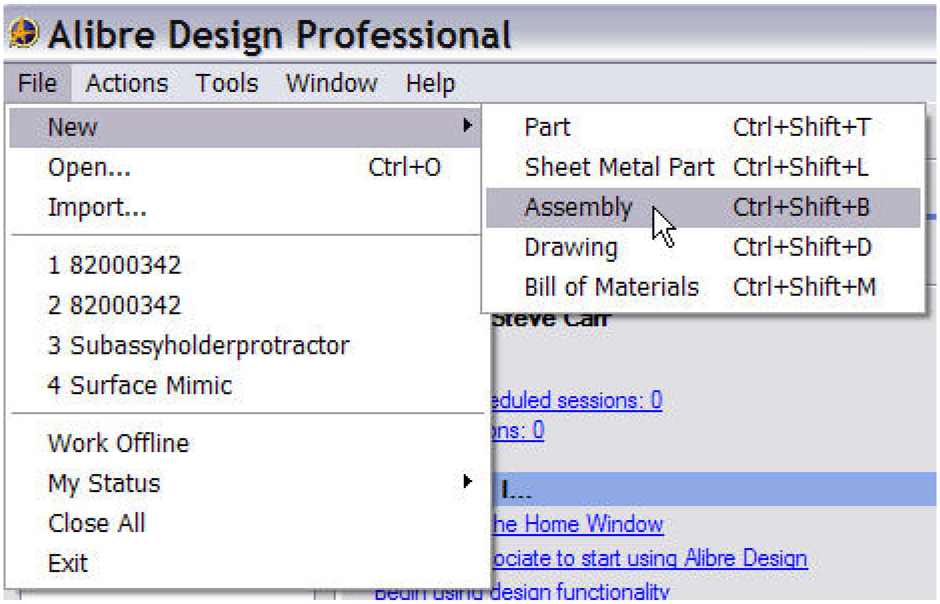


It much simpler to check the Physical properties of an assembled part, and with a better degree of accuracy than any manual calculation process could deliver, and do this in a matter of minutes, not hours or days.

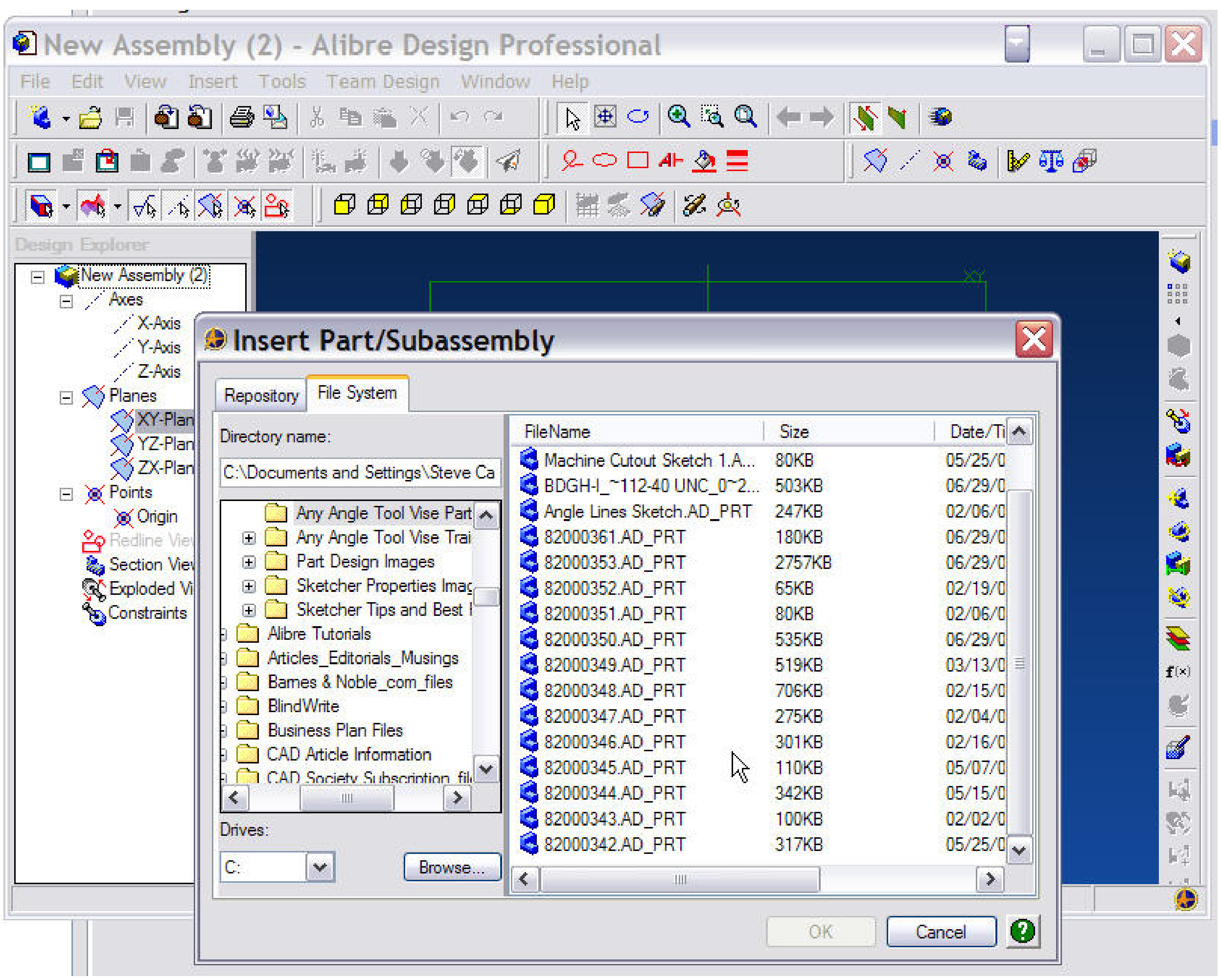


How you structure your assemblies will depend on how your company treats assembly data in its overall data management process, but it more than likely will be a hierarchal structure, including single parts, as well as sub-assemblies, purchased parts, and standard parts. Alibre is able to incorporate data from disparate sources, and in a number of different formats. For more information on importing data from outside sources, see XXXXXXXXXX.

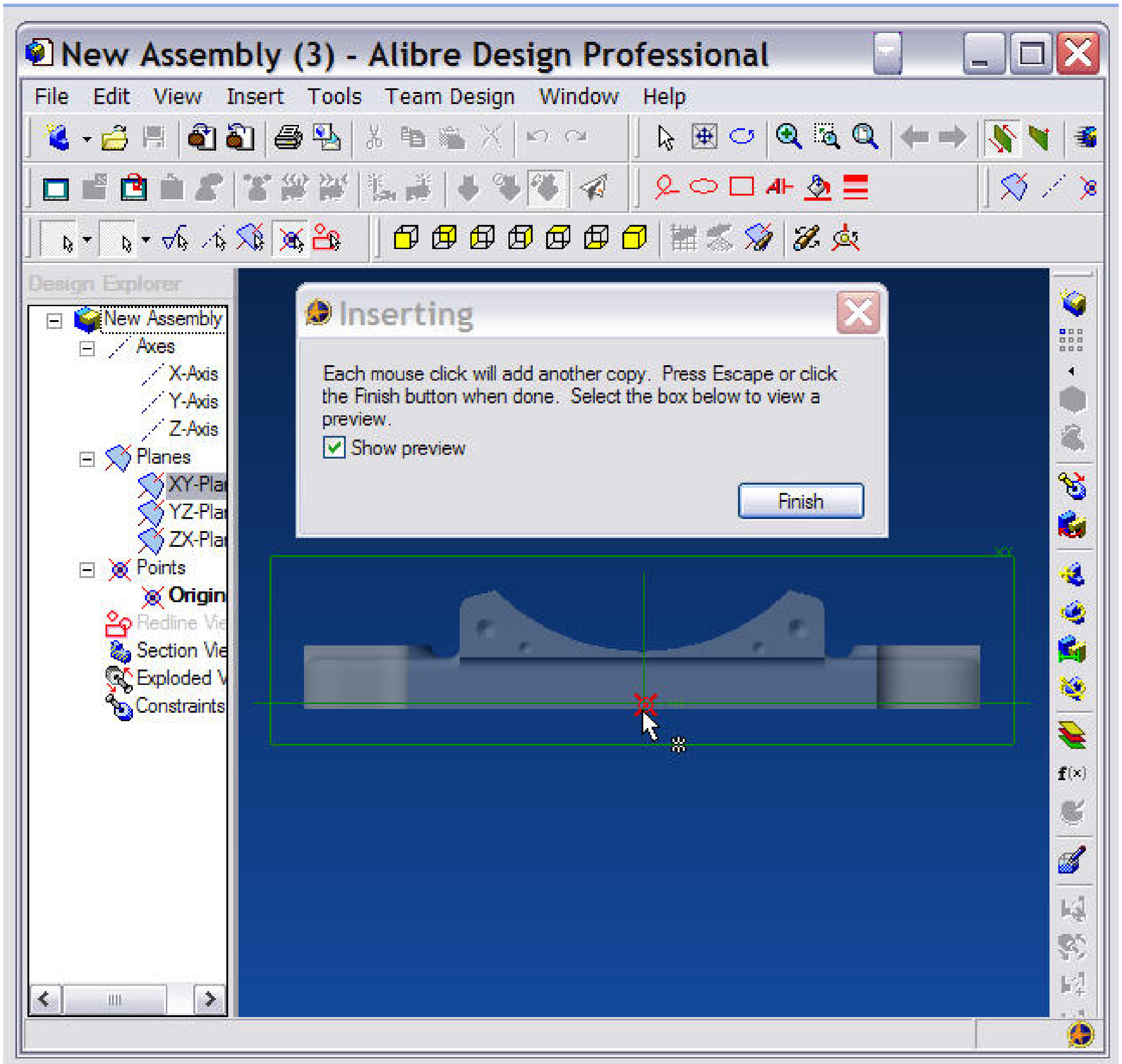
To get started with the assembly of the Any Angle Tool Vise, click on the File tab and select New>Assembly.



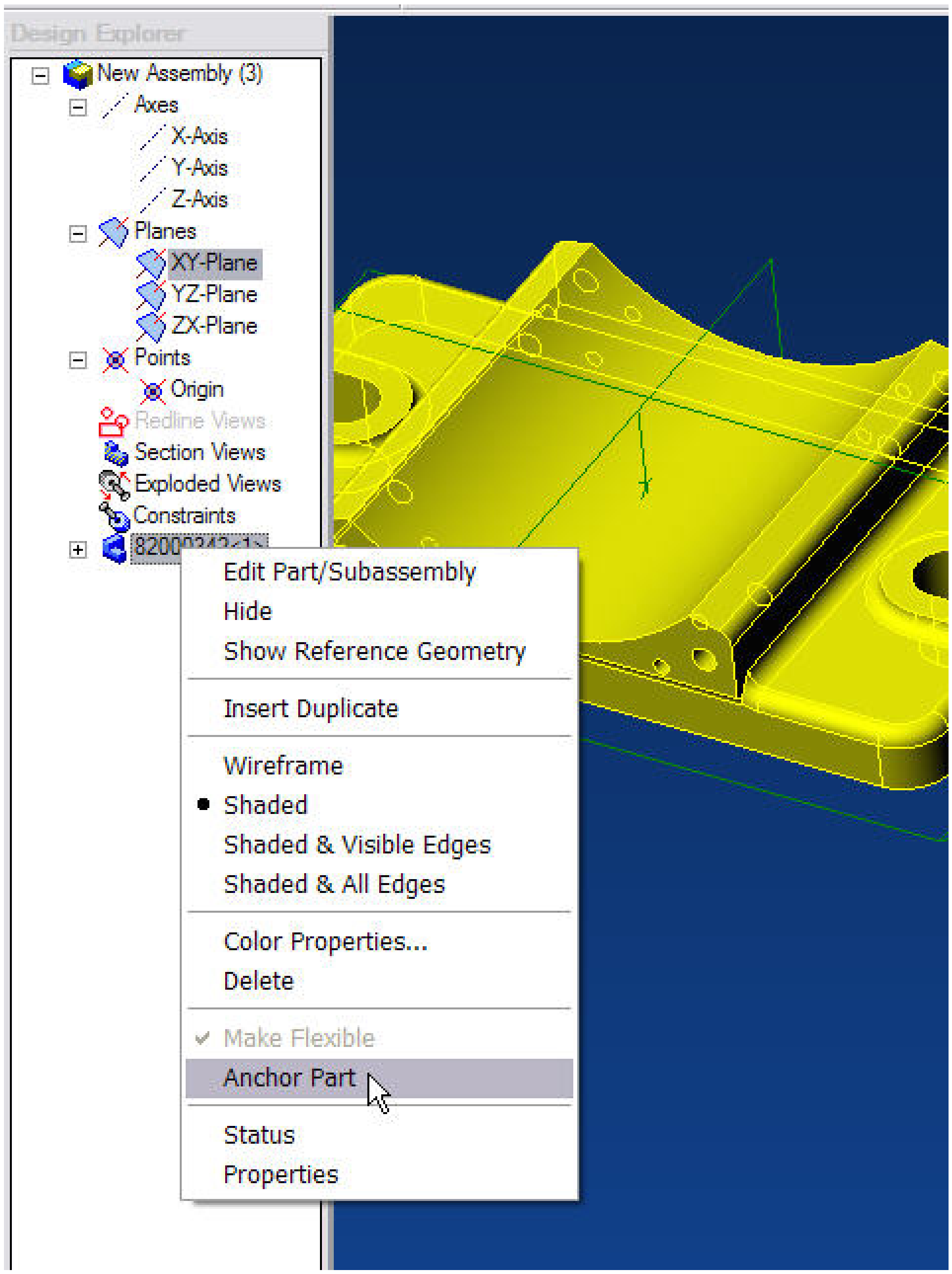
A new assembly work space panel opens and you're asked to select a part to insert.



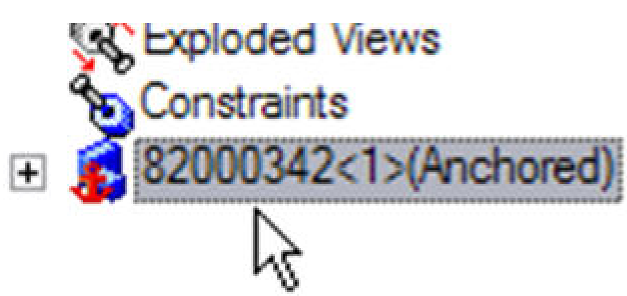
The logical approach in developing an assembly part is to start with the primary 'base' part, in this case the Saddle Base. Find the part number for the Saddle Base and click on it.
An 'Inserting' information panel will appear, advising you that each mouse click will add an additional copy of the part you're inserting. Click once on the 'Origin' point and then click the 'Finish' button on the information panel or hit the 'Escape' key on your keyboard to terminate the insert process.



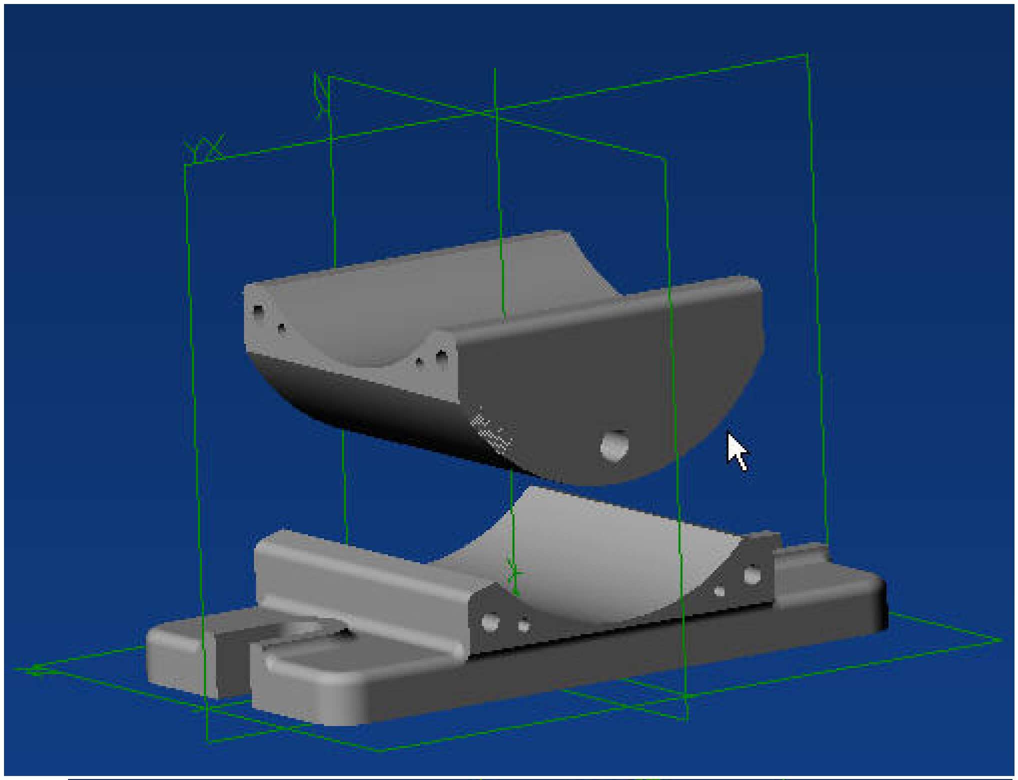
After adding the base part you'll want to anchor it in position. This is done to provide a fixed part that the other parts of the assembly will be added to, and to preventunwanted movement when adding constraints. Select the base part in the Design Explorer panel, right click and select 'Anchor Part'.



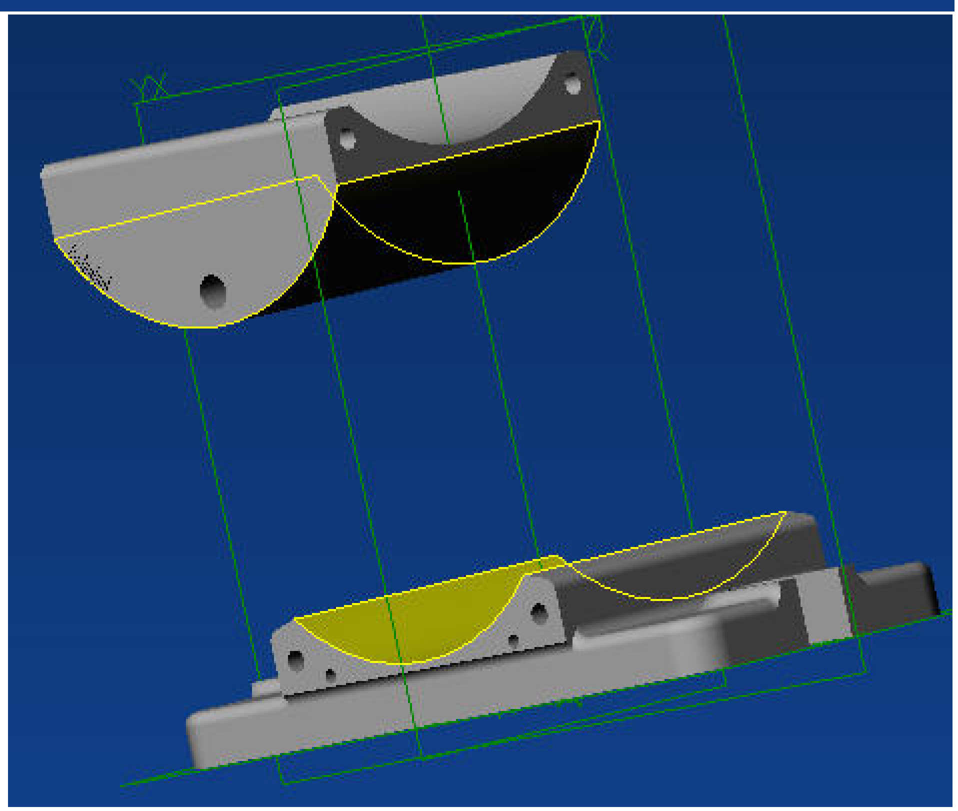
An 'Anchor' constraint symbol appears next to the part as well as the (Anchored) callout.



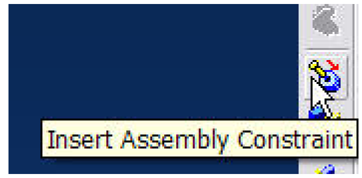
Next add a copy of the ‘Compound Center Member’ again using the ‘Origin’ point to initially locate it. Using the ‘Move Part’ command, separate the Compound Center member from the Saddle Base far enough so that your ability to select the curved surfaces of both parts in unimpaired.

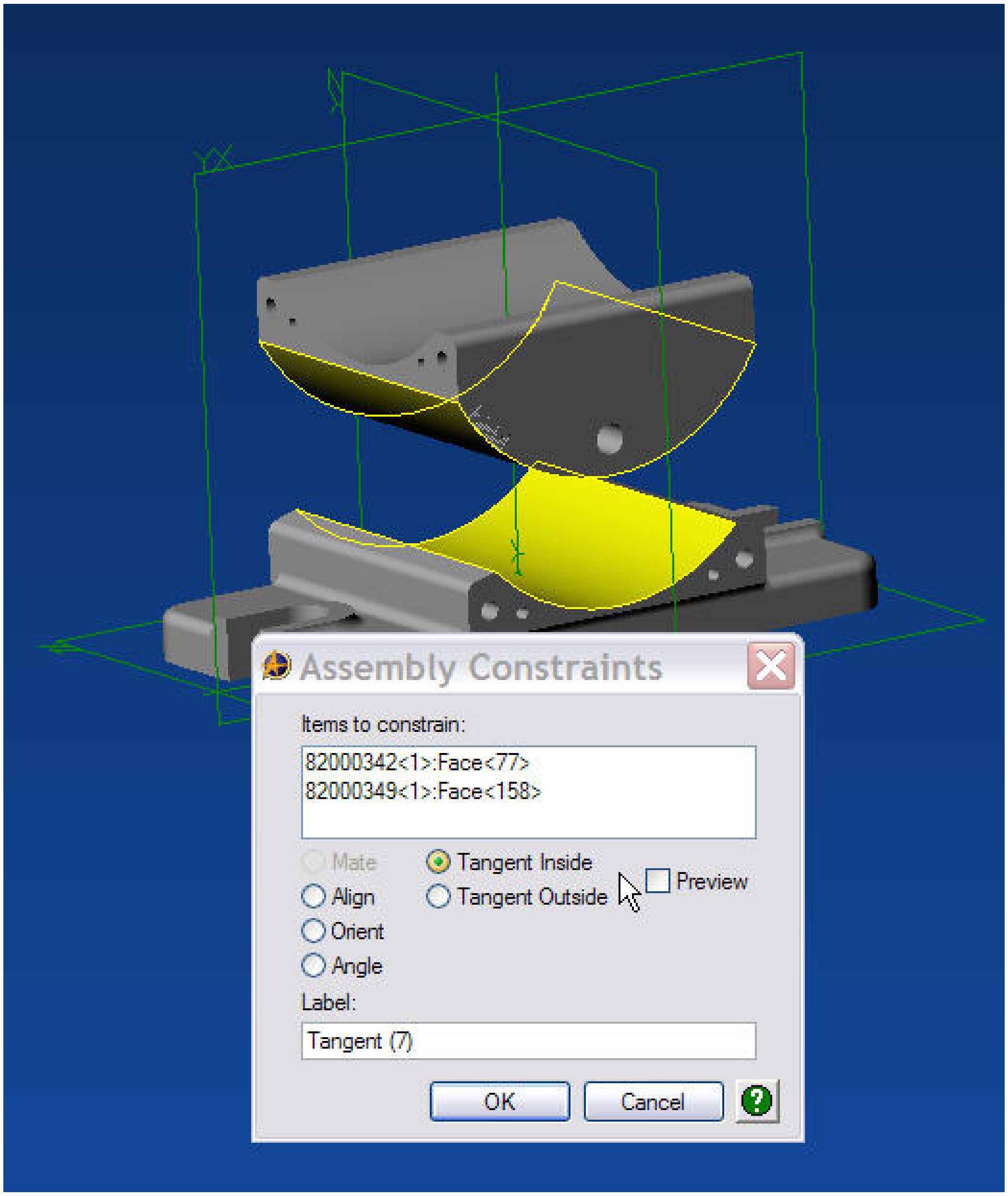


Select the two curved surfaces in the following order; the Base surface first and then the Compound Center Member surface. This established a ‘direction’ for the constraint you’ll apply in the next steps.



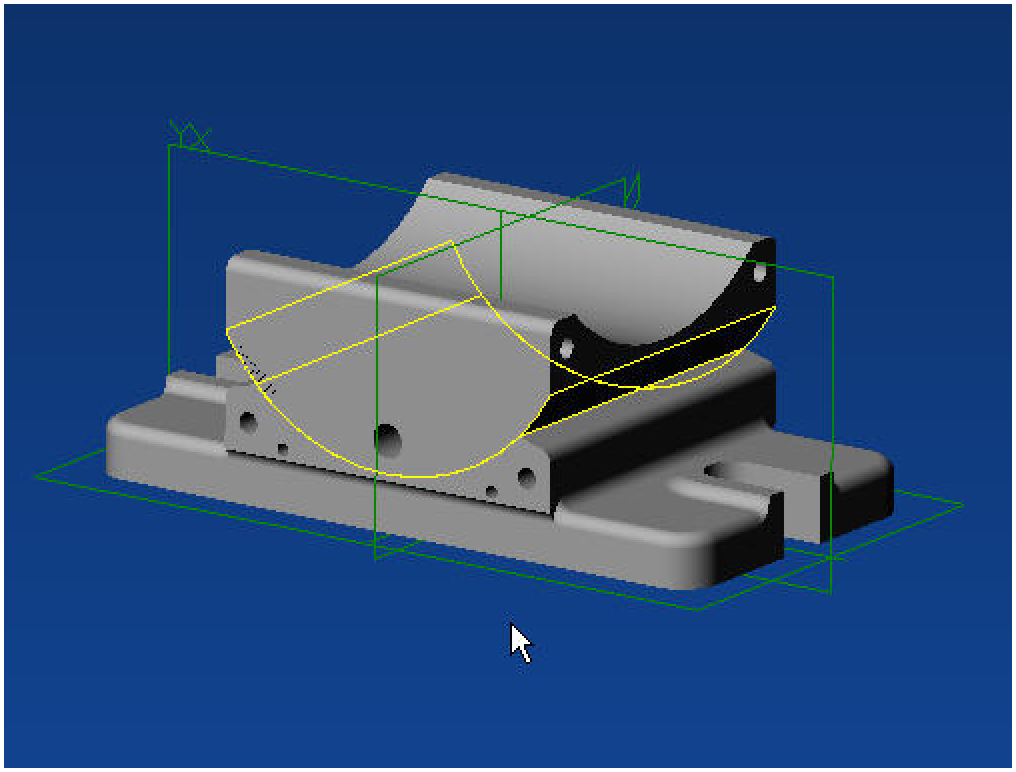
Click on the Insert Assembly Constraint icon.





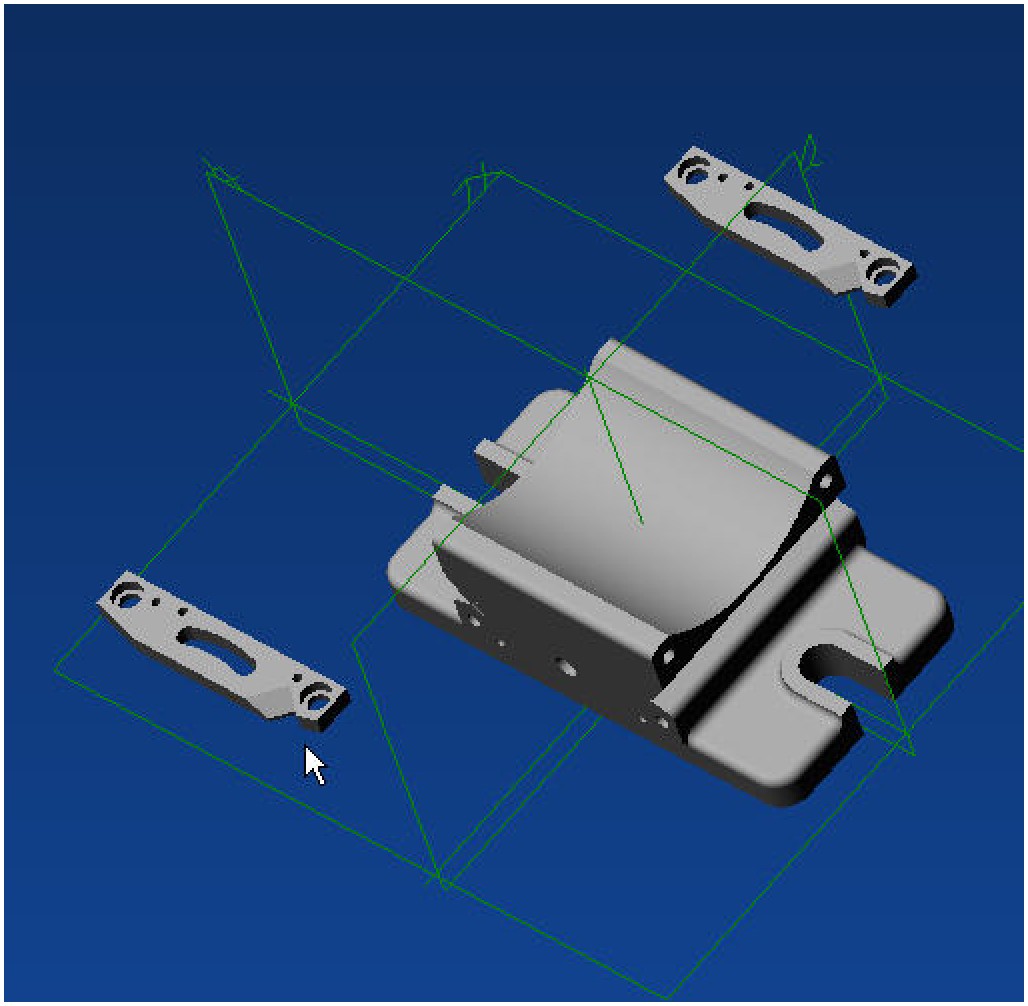
The Assembly Constraint panel will open. The parts you selected should appear in the correct order. Select the 'Tangent Inside' option and then click OK.

Your assembly part should now look like this.

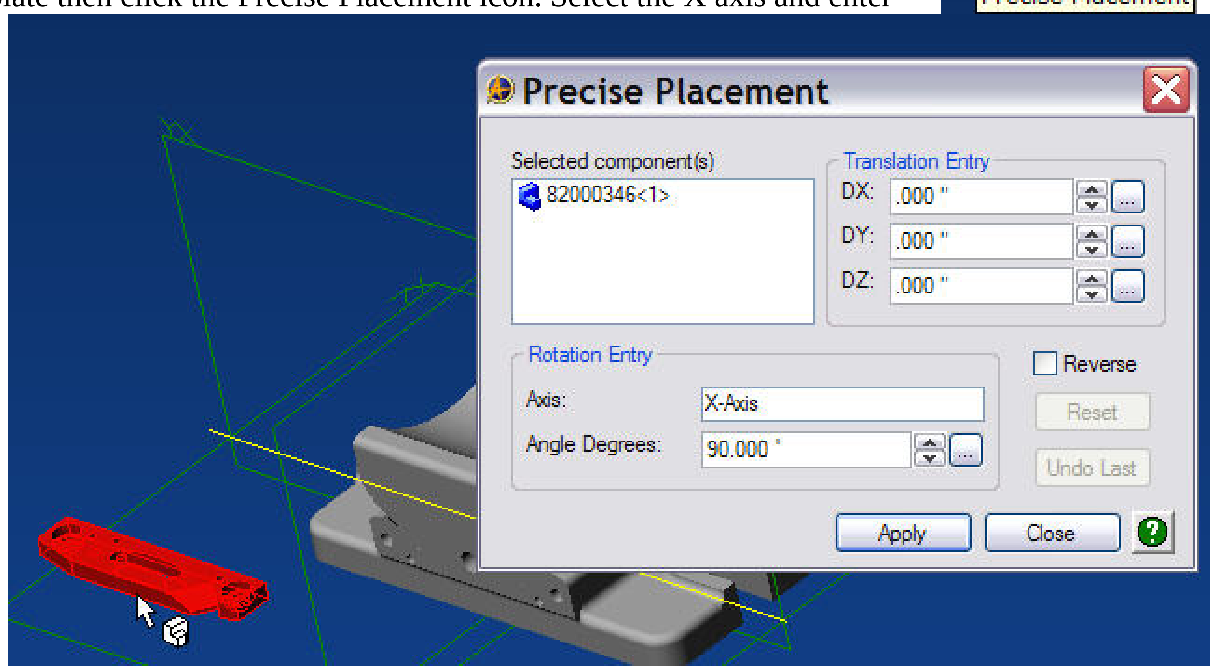


When you constructed your parts, you used the origin of the base coordinate system inside Alibre Design, to position and constrain your sketches and parts. This can provide many benefits in creating the assembly part in that it can reduce the number and types of constraints necessary to build you part. An example is the two parts you just assembled. It took one constraint to connect them because the original sketches and parts were constrained to the same original start point. Always try to use techniques like this to simplify the assembly process whenever possible. This is another reason that designing in part space is preferable to designing in assembly space.

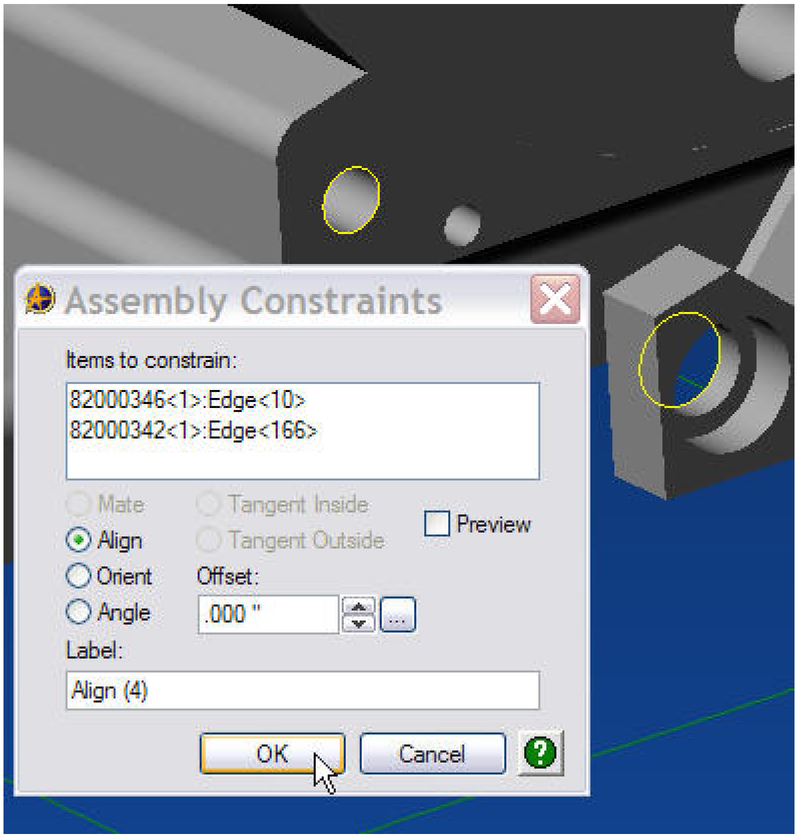
Let's add two instances of the Lower Plates.



To simplify the assembly process, we'll rotate the plates into positions that will allow us to more easily constrain them. Click on the indicated plate then click the Precise Placement icon. Select the X axis and enter



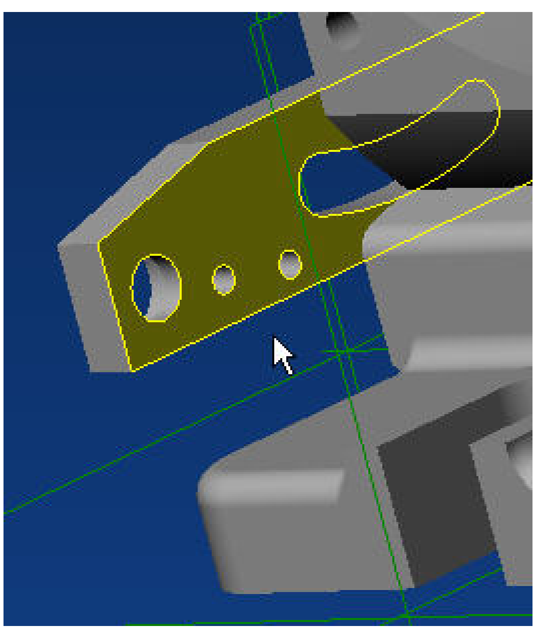
90 degrees in the Rotation Entry boxes. Click Apply.
Use this method to rotate the plates into the correct alignment with the respective sides of the Saddle Base. Although it is possible to constrain the plates without moving them into position, it may take several failed attempts to align them correctly. Taking the time to initially position them may save you wasted effort and reduce your level of frustration.

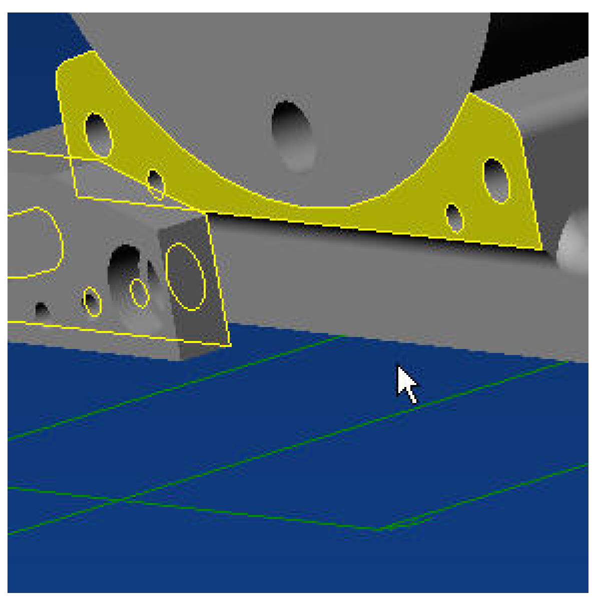


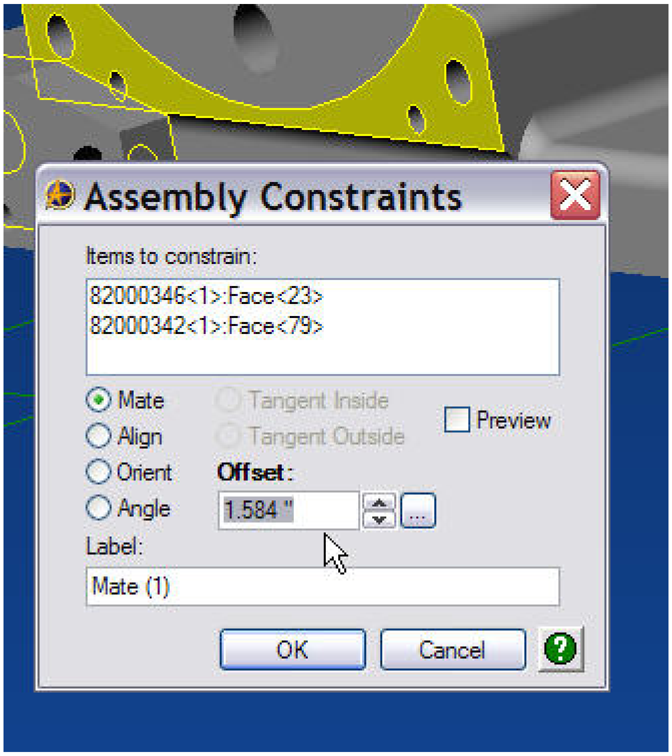
When the plates are in the correct relative position, we can constrain them to the Saddle Base.

Select the edge of the hole on the backside of the Plate, and the edge of the hole on the front side of the Saddle Base. Click on the Insert Assembly Constraint icon and whem the panel opens, select ‘Align’ as the constraint and .000” as the offset. Click OK. Repeat this process for corresponding right hand holes.

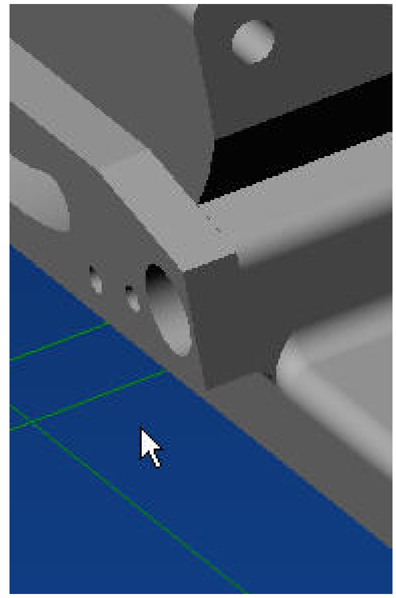
Now click on the rear face of the Plate and the front face of the Saddle Base. Click on the Insert Assembly Constrain Icon.





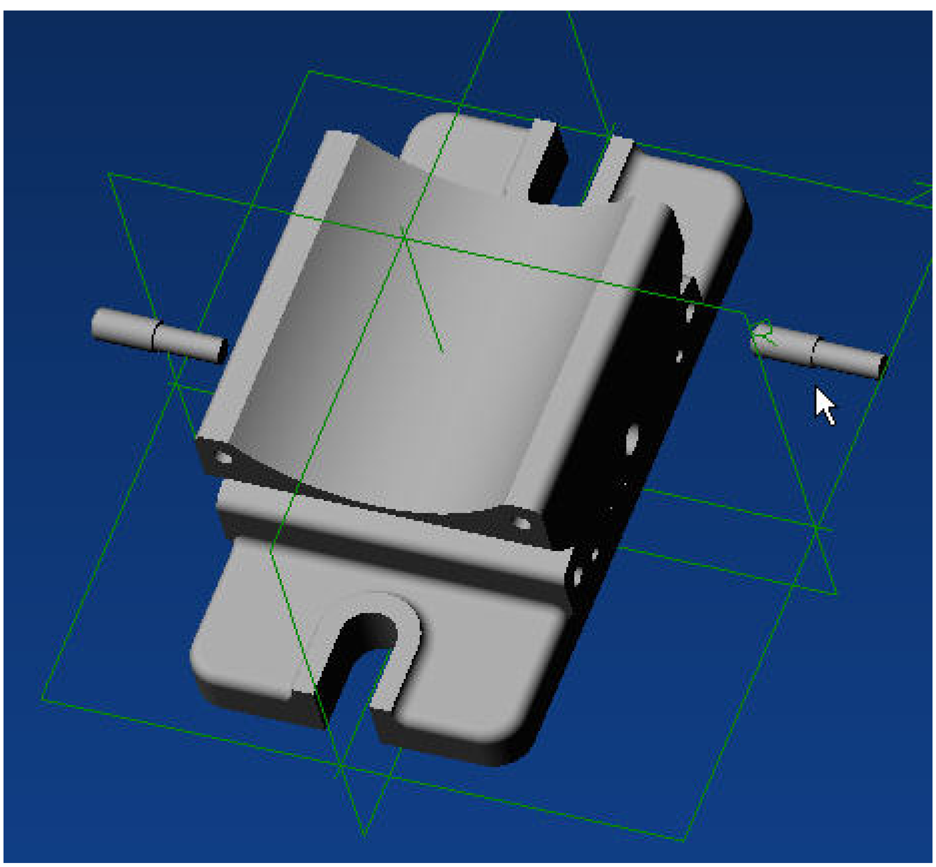


Select ‘Mate’ as the constraint type and enter .000” in the Offset box. Click OK. Your part should now look like this.

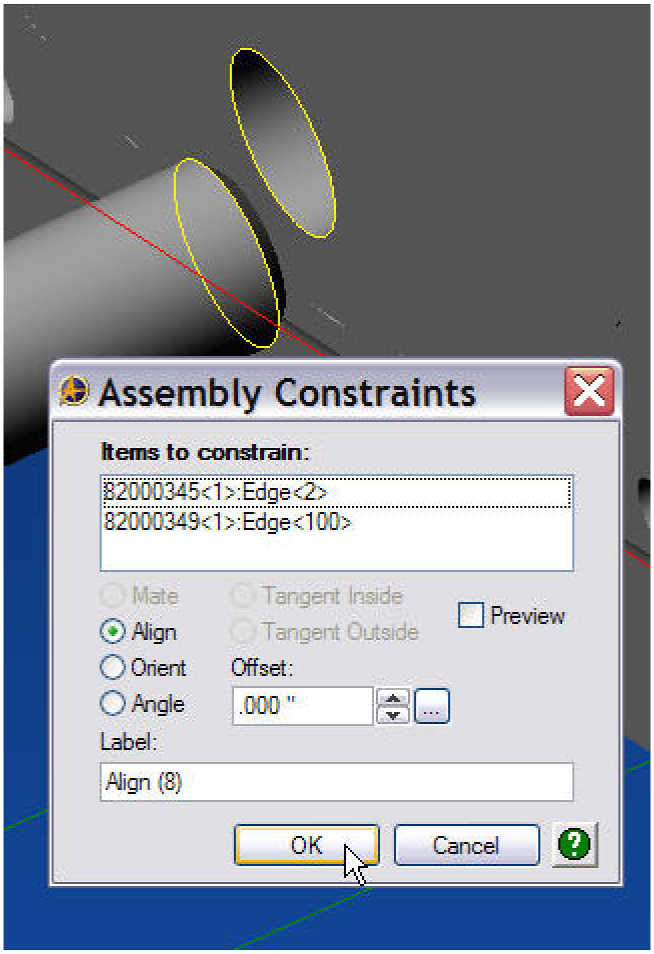


Mate was selected because you want the two parts to remain in the chosen constrained condition regardless of how other parts in the assembly move.

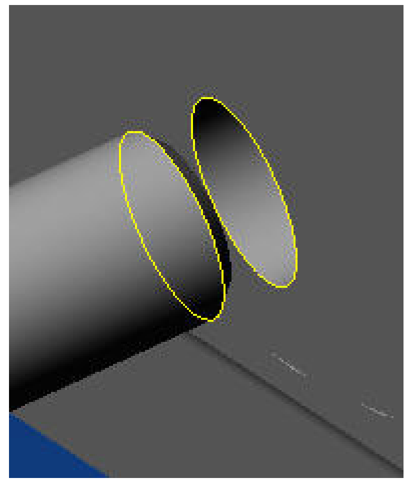
Repeat this process for the second plate. After you’re finished hide the two plates by right clicking on the and selecting ‘Hide’. This will make inserting and constraining the two eccentrics much easier.



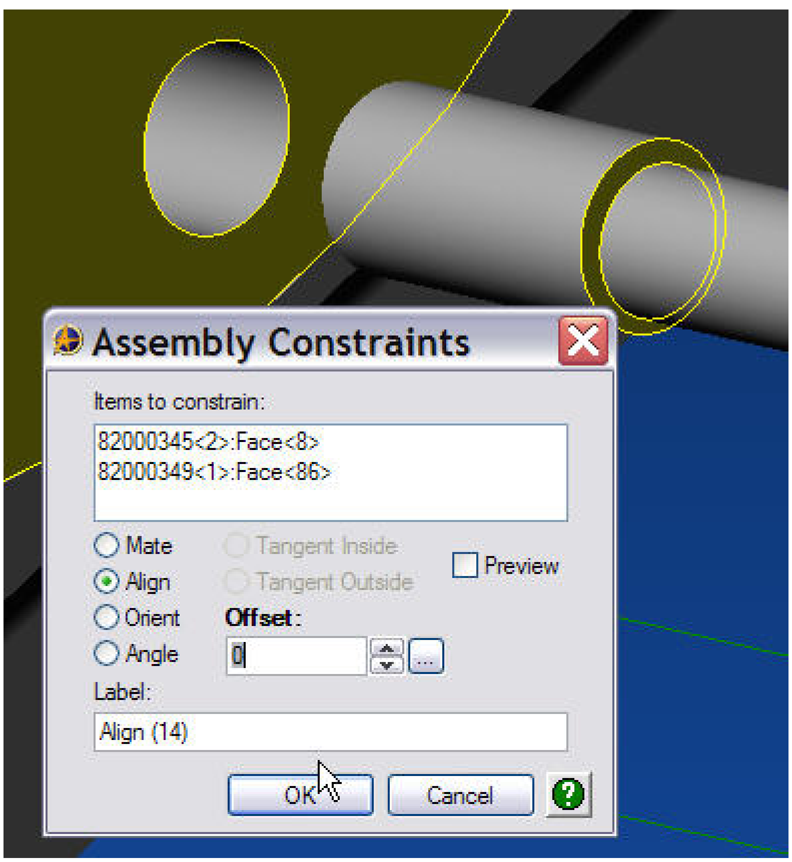
Now add two instance of the Eccentric (PN 82000643) Rotate the instances into an initial alignment with the holes in the Compound Center Member.



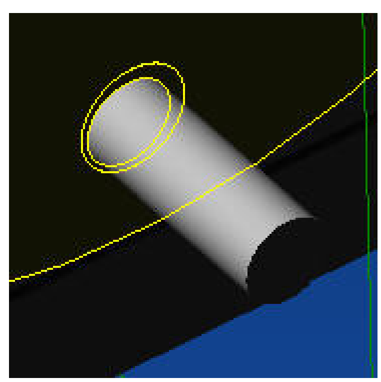
Select the circular edge of the large eccentric cylinder and circular edge of the hole on the Compound Center Member. Click the Insert Assembly Constraint icon and in the Assembly Constraint panel,



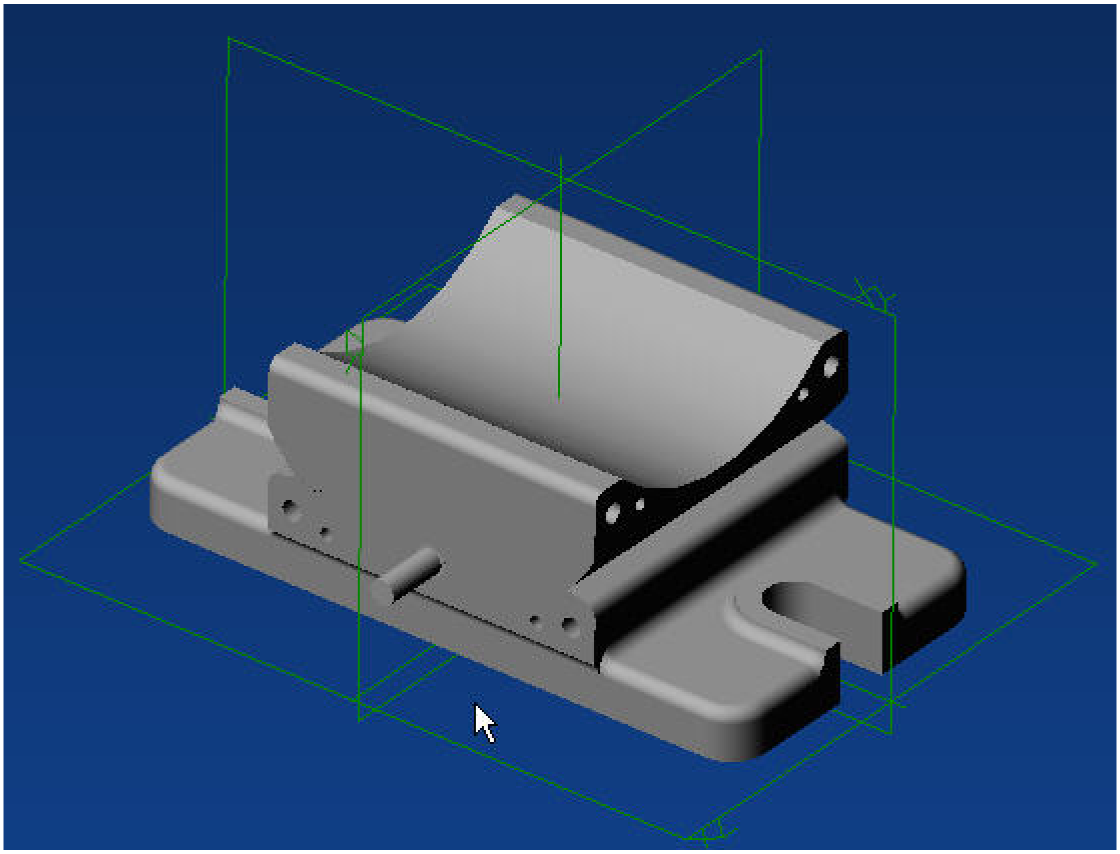
select Align, with an Offset of .000” . Your part should now look like this.



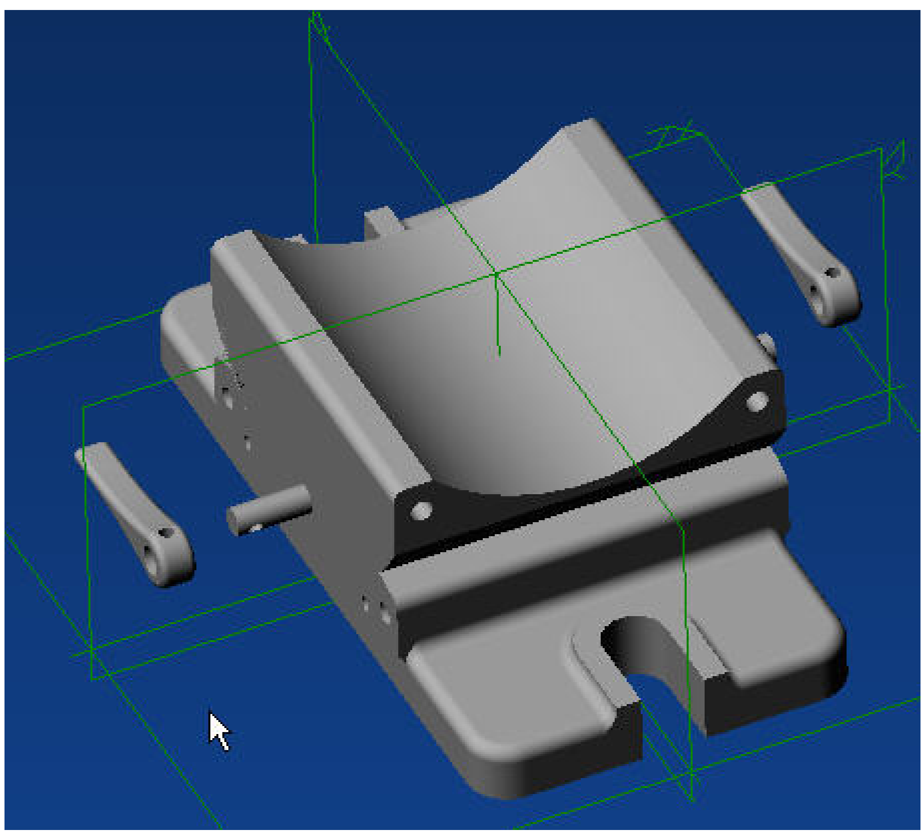
Now select the face of the large eccentric cylinder and the face of the Compound Center Member, click the Assembly Constraint icon, and again select ‘Align’, with an Offset of .000”. Your part should now look like this.



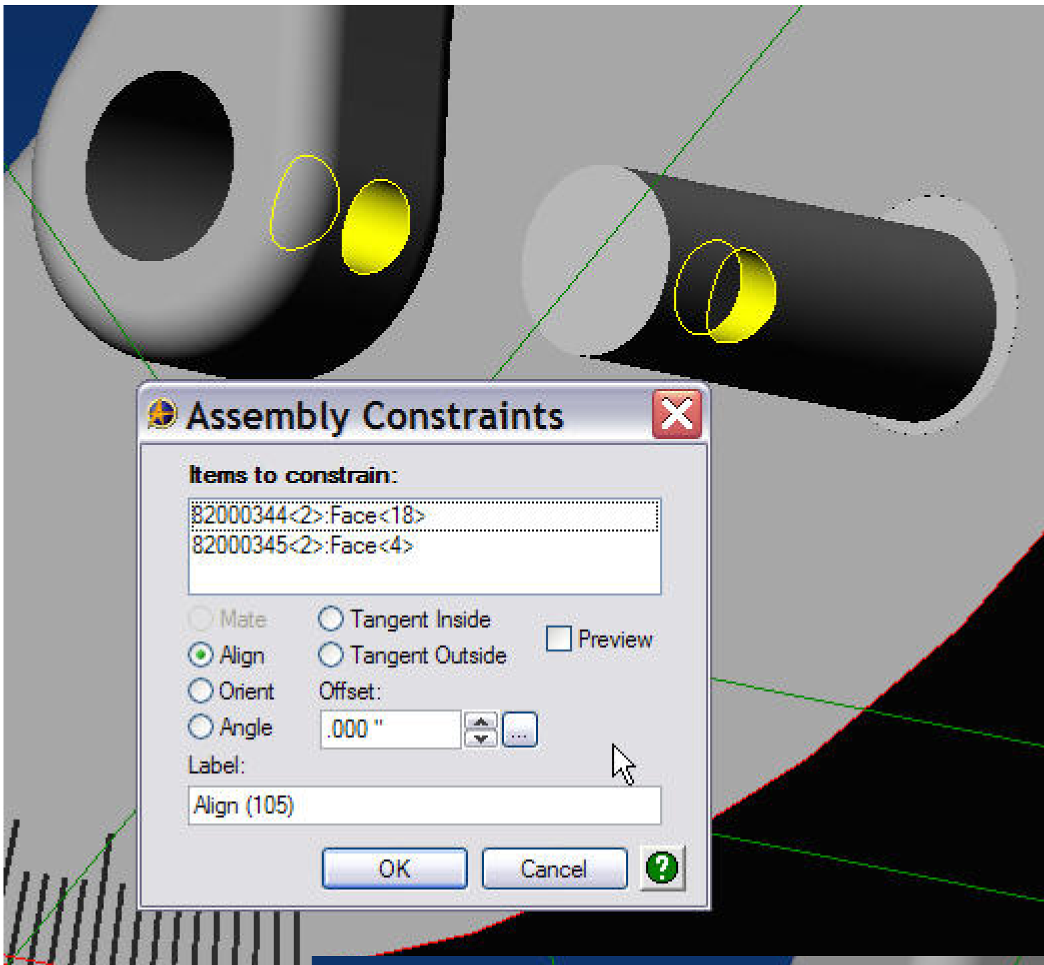
Repeat this process for the other eccentric.



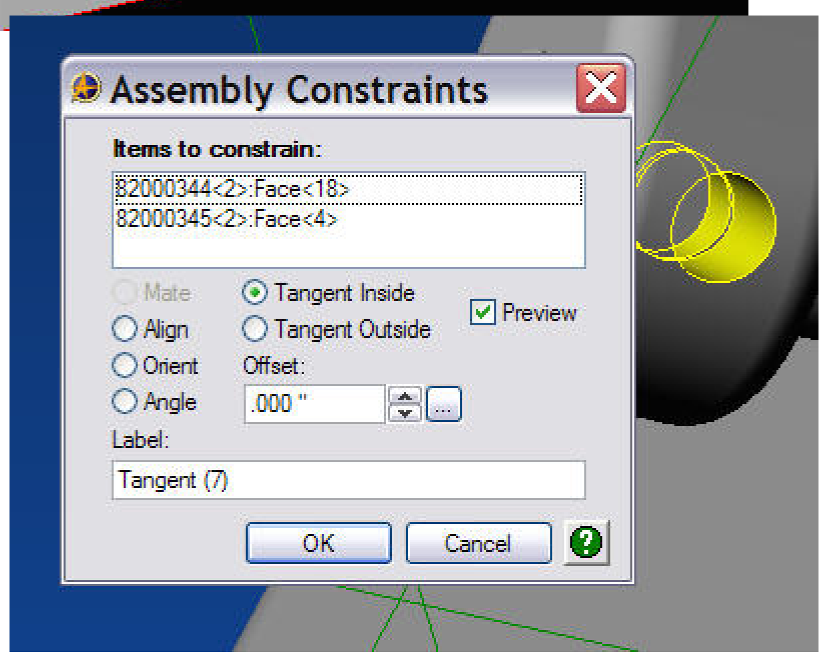
Insert two instance of the Locking Handle and move and rotate them into a reasonable position in relation to the eccentric and the Compound Center Member.

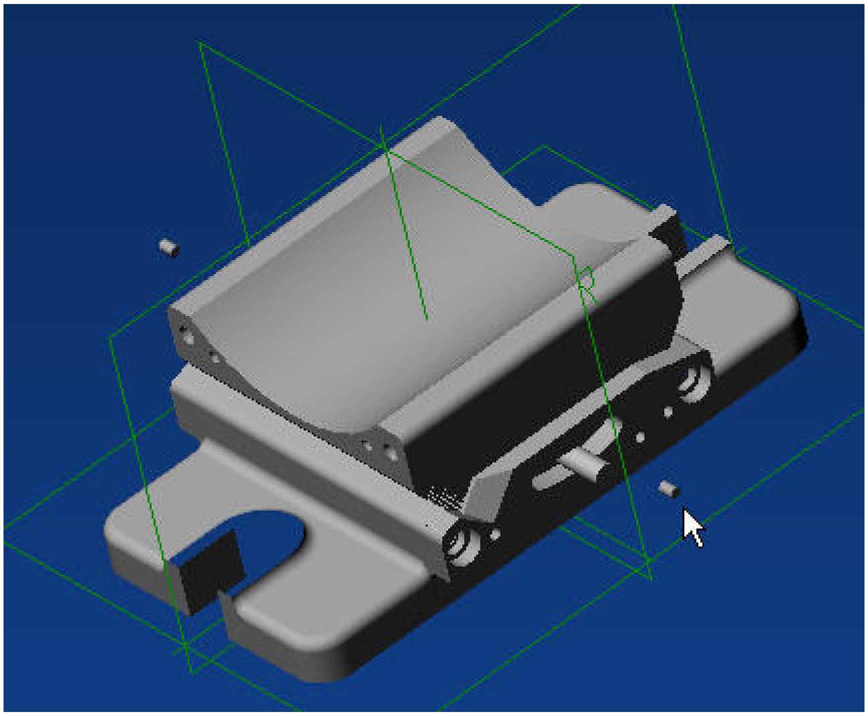


In the this example either the 'Align' constraint or the 'Tangent Inside' constraint will deliver essentially the same results. Which one to use is a personal choice. There is often more than one way to achieve a desired result in Alibre. To become familiar with the differences and similarities in constraint commands, you'll need to experiment them.



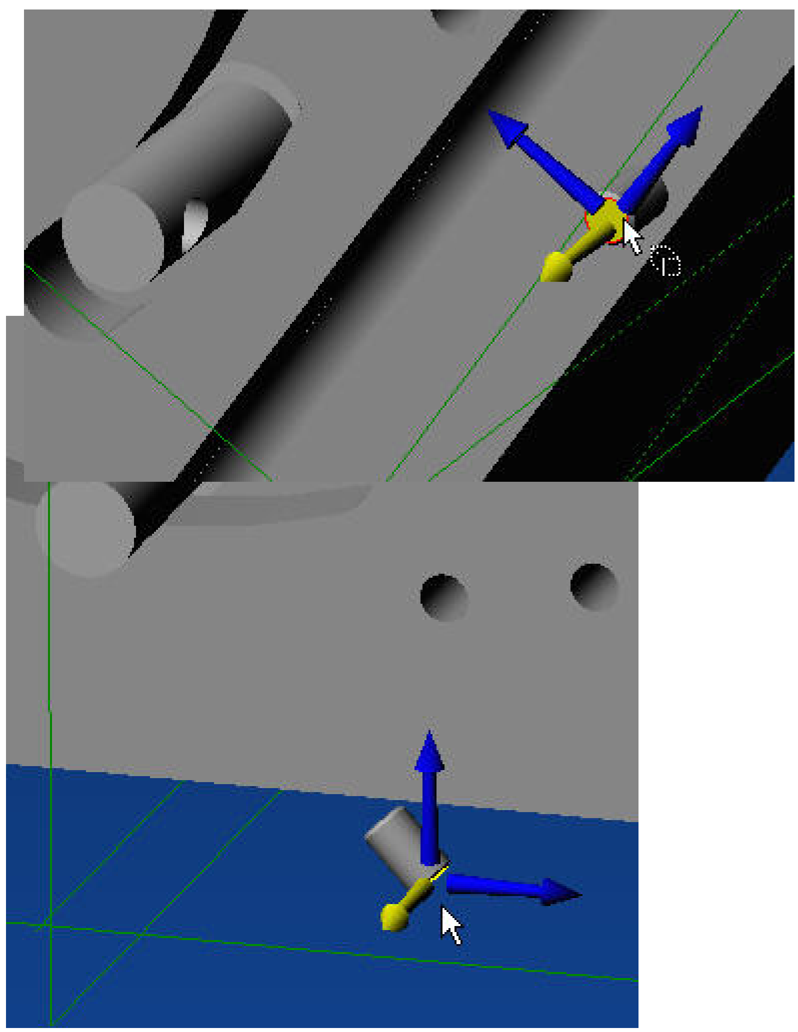
Repeat the process for the other Handle and Eccentric. After you're finished, un-hide the Lower Plates by right clicking on them in the Design Explorer tree and selecting 'Hide' again.



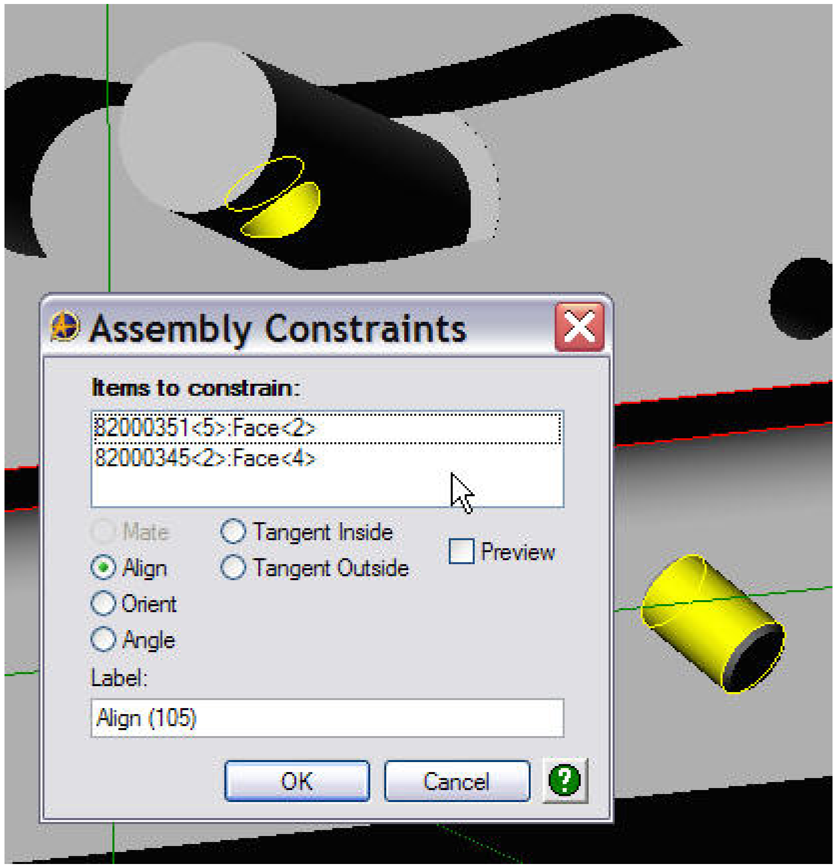


The next part we'll insert are the press pins (PN 82000351) that hold the handle and eccentric together. If desired, this combination of items handle, eccentric and press pin, could be created as a sub-assembly and inserted in our assembly part instead of inserting multiple instance of each part.

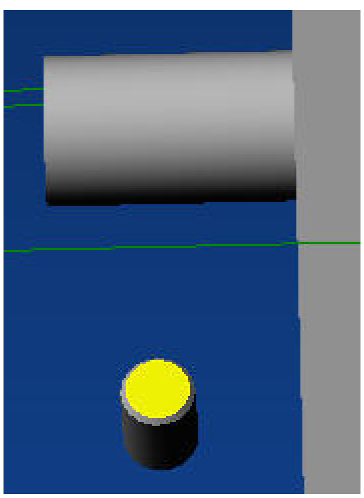
Insert two copies of the press pin and using the 'Rotate Part' command position them in rough proximity to their final position.

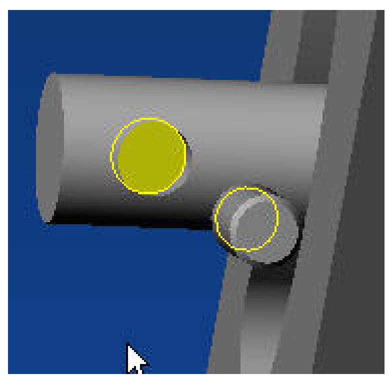


Next select the outer cylindrical face of the press pin and the inner cylindrical face of the hole in the eccentric. Click on the 'Insert Assembly Constraint' icon, and select align. Click OK.

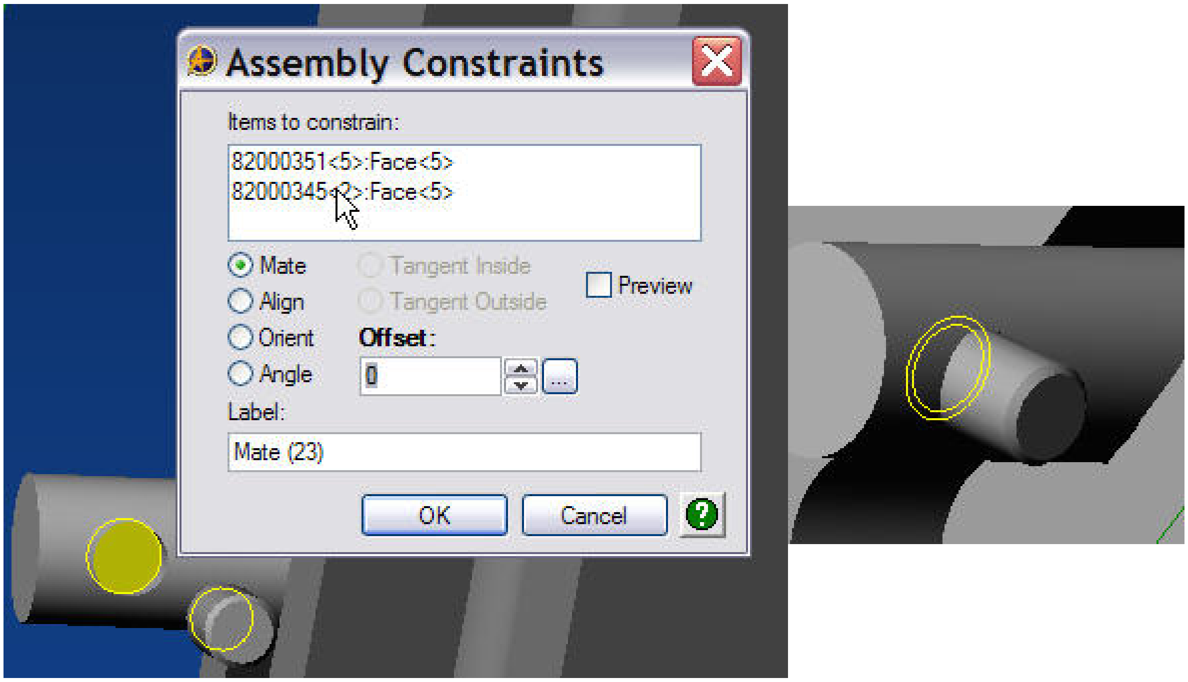


Next, click on the face of the press pin that will be inserted in the eccentric, then click on the bottom face of the hole in the eccentric.

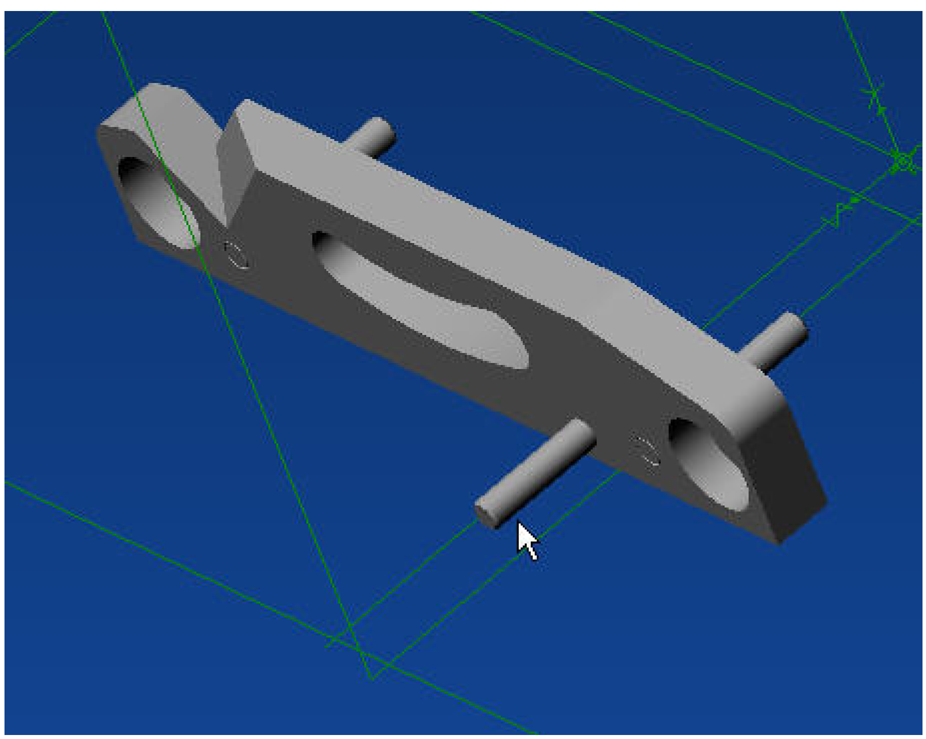




Click on the ‘Insert Assembly Constraint’ icon and select ‘Mate’ with an .000” Offset. Click OK. Your part should look like this. Repeat the process for the other handle.

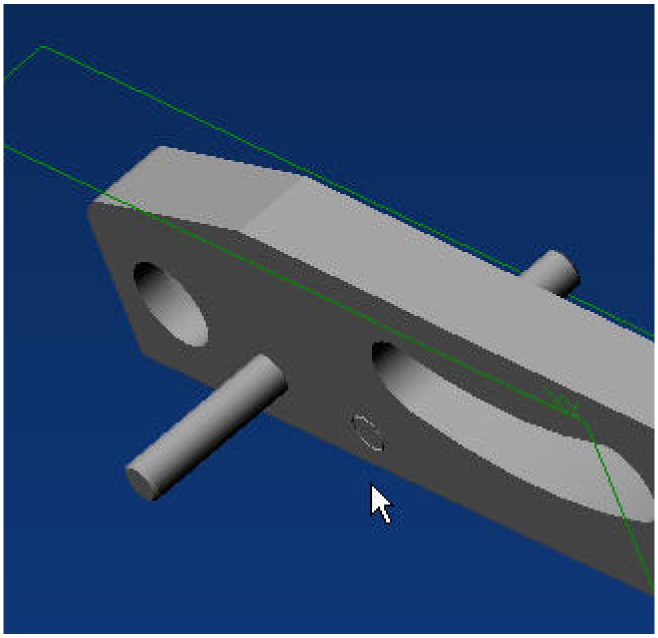


There are two other handle, eccentric, and press pin, combinations in our assembly and the same procedure used here will apply to these also. In looking at the total assembly, you can see that there are duplicate features that could be treated as sub-assemblies (the handle, eccentric, press pin is one) thus making the creation of the final assembly somewhat more efficient. Whether you use this approach or not depends what your documentation process requires or how you want to break your assembly down. In this chapter, I’ve approached this from the aspect that all parts of the assembly will be documented on one drawing. This also gives you added practice in positioning and constraining multiple copies of the same or similar parts, making you more familiar with the assembly process.

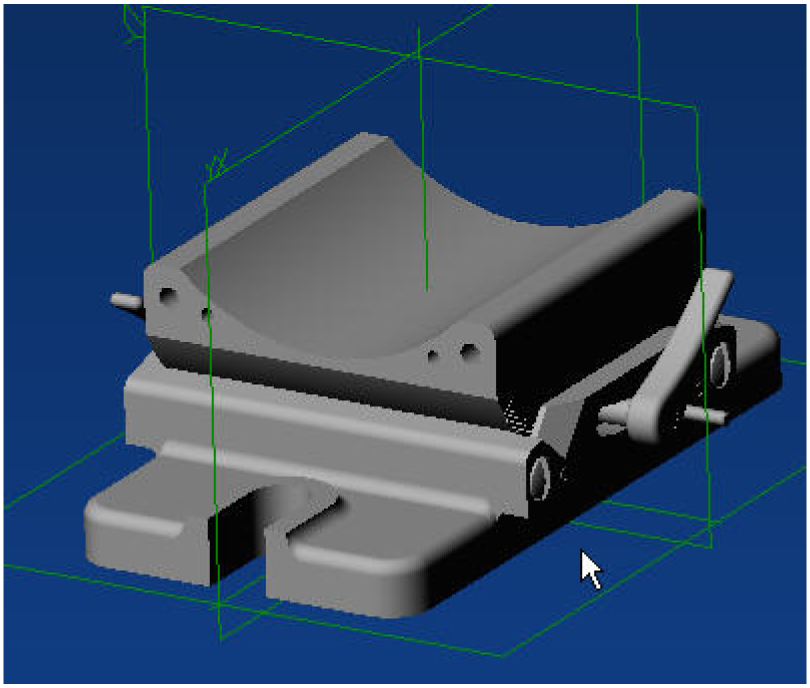


The next parts to add, are the press pins used in the lower plate as positioning pins and as a stop pin for the handle. The base and Compound Center Member of the assembly have been hidden to give you a clearer picture of how the three pins are assembled to the plate.

Essentially, each pin is located using the rough positioning technique we've used for all the other parts in the assembly, plus two alignment constraints, one using the circular edge of the pin and the circular edge of the hole in the plate, and the other using a face of the pin, and a face of the plate. The position of each in relation to one another determining whether the front or rear faces of each part is used in the process.

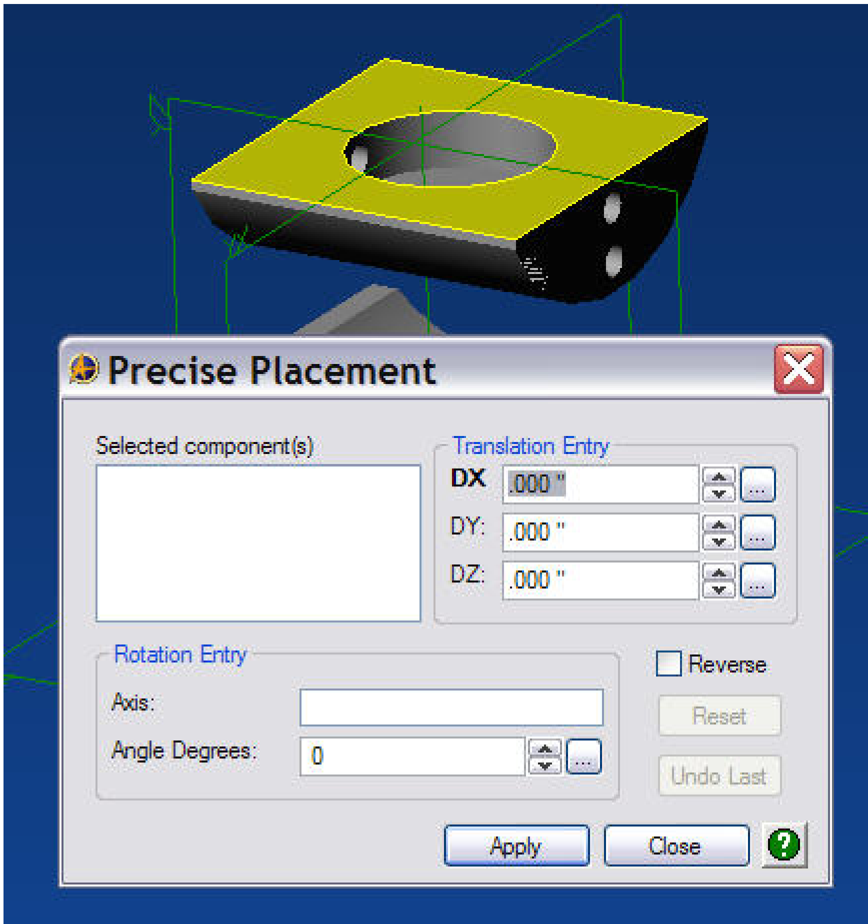


Again, either the 'Align' constraint or the 'Tangent Inside' constraint will deliver essentially the same results. Which one to use is a personal choice. Repeat the process for the other pins in the assembly.



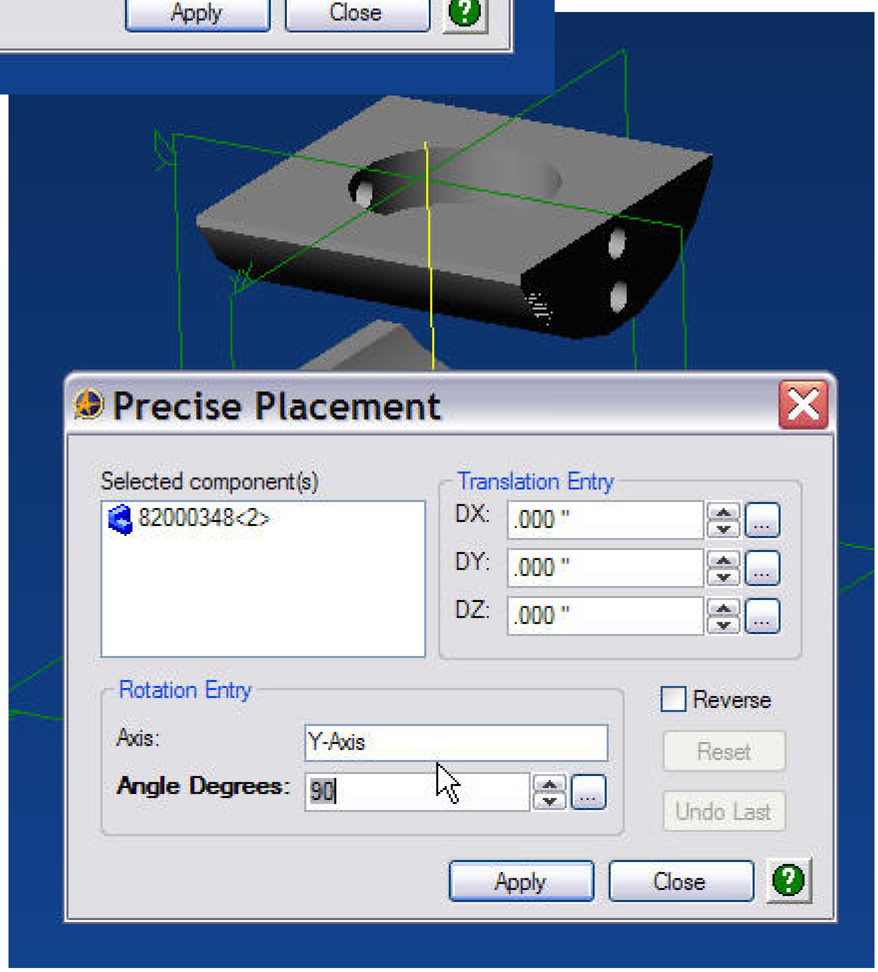
Your assembly should now look like this.

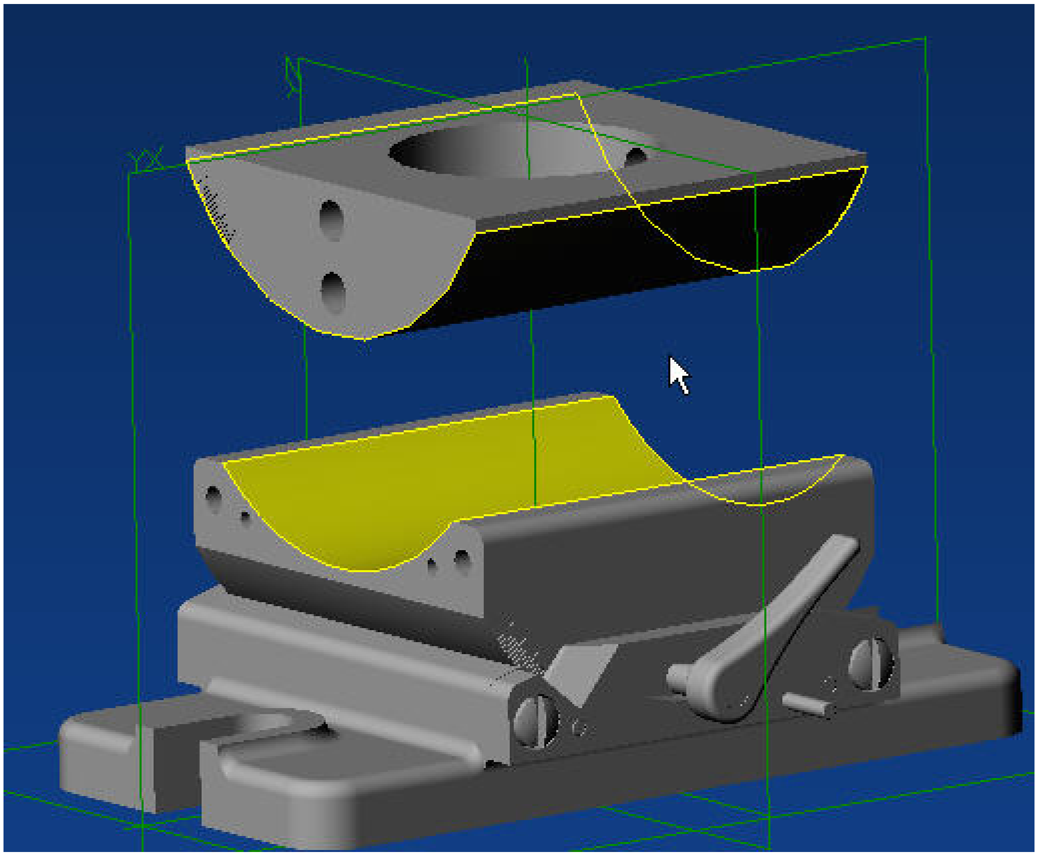
Anchor the Compound Center Member before proceeding.

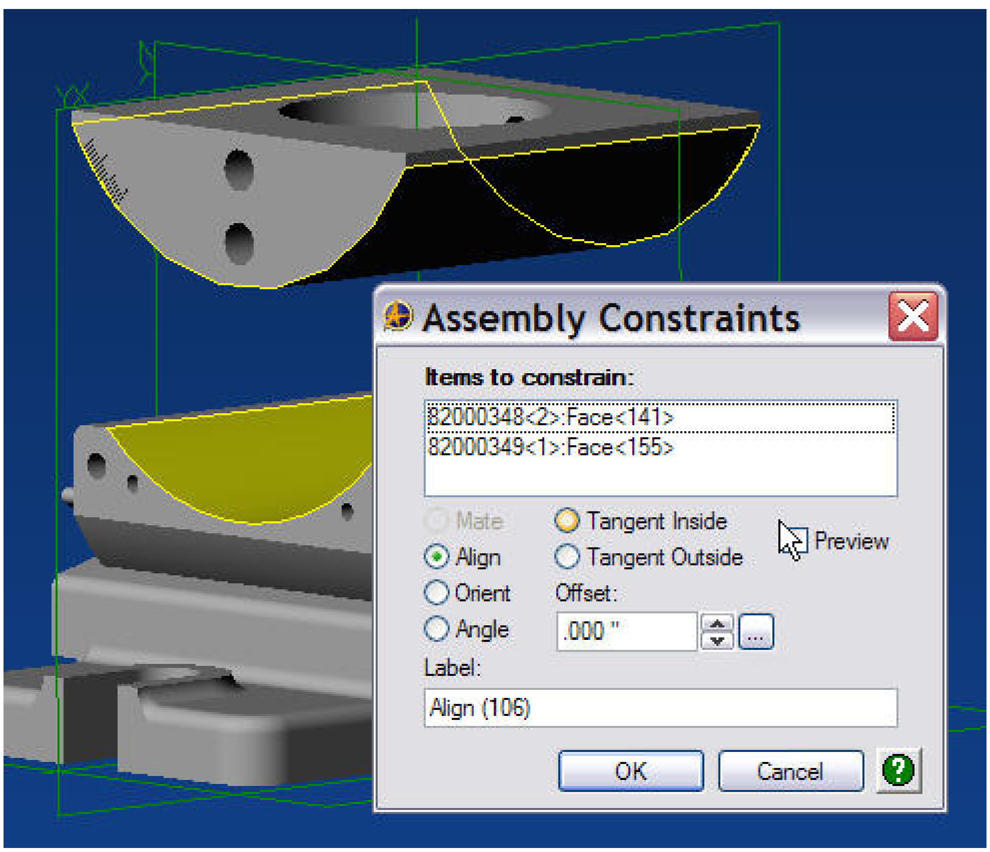


Now we'll add the 'Upper Compound Member' and position it and constrain it to the 'Compound Center Member' just as we did the 'Compound Center Member' to the 'Saddle Base'.

Use the Precise Placement command to position the part.

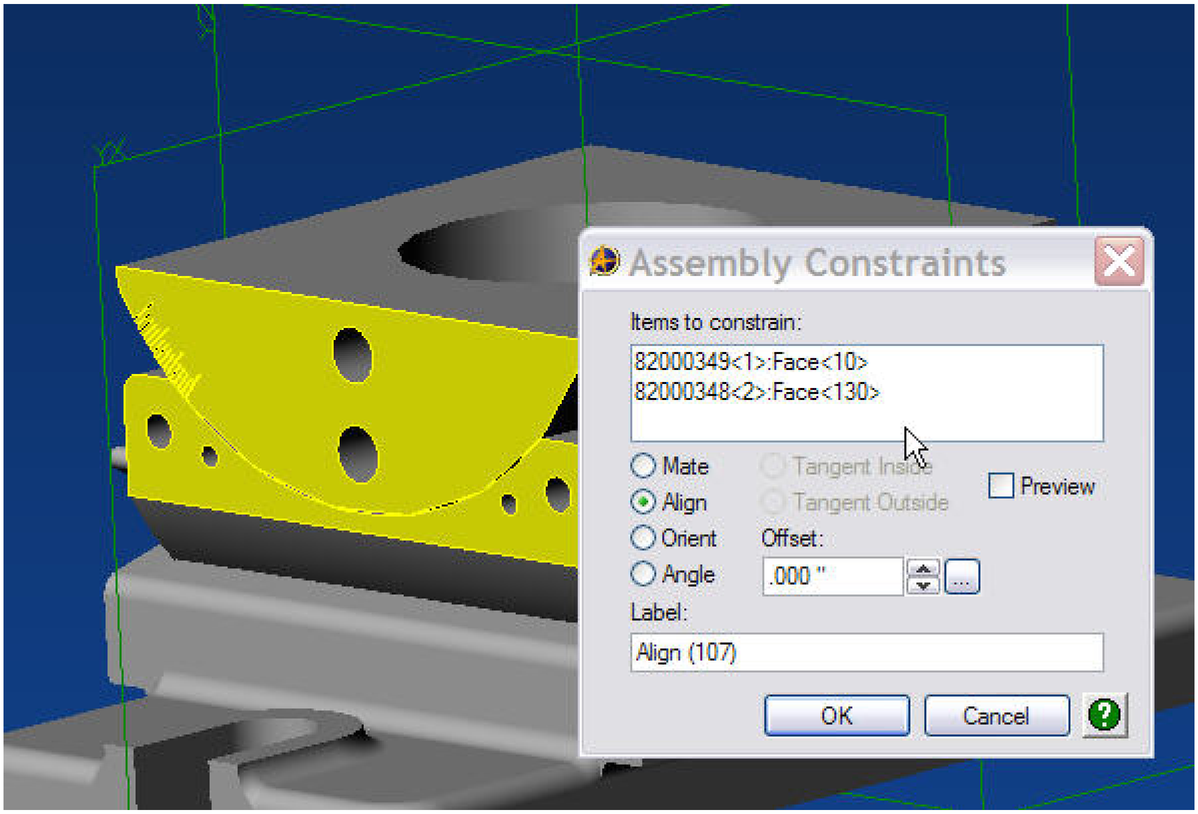




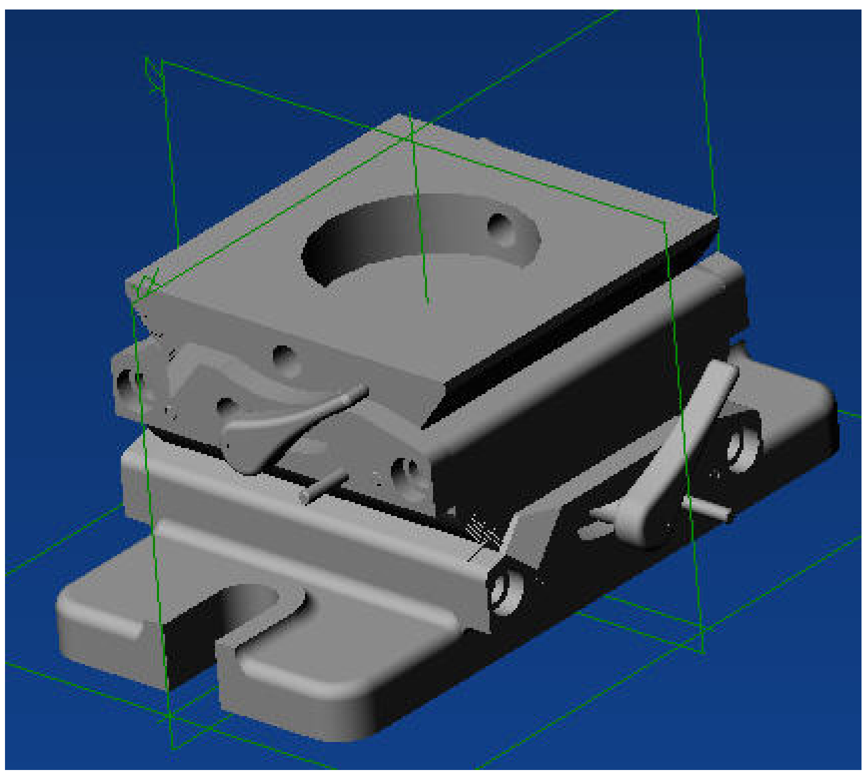


Select the lower cylindrical face of the Upper Compound Member, and the Upper Cylindrical face of the Compound Center Member. Click the ‘Insert Assembly Constraint’ icon and select Align, .000” Offset. Click OK.

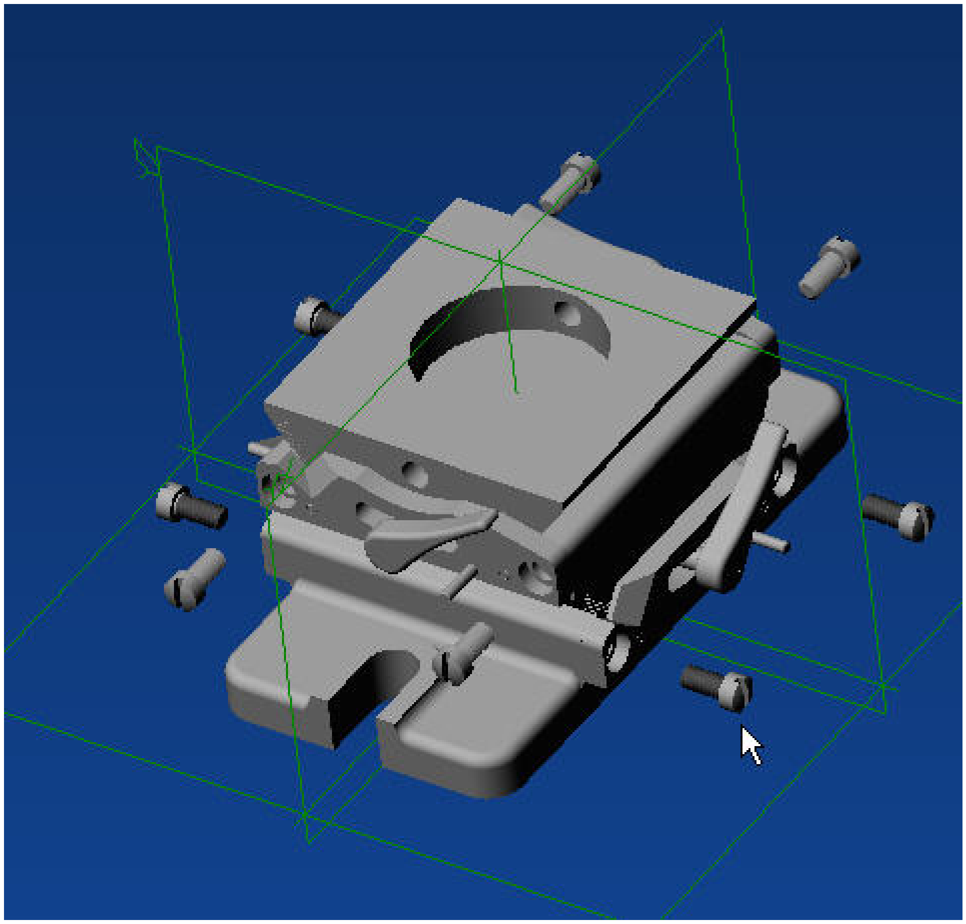
Select the front face of the ‘Upper Compound Member’ and the front face of the Center Compound Member. Click the ‘Insert Assembly Constraint’ icon and select Align, .000” Offset. Click OK.



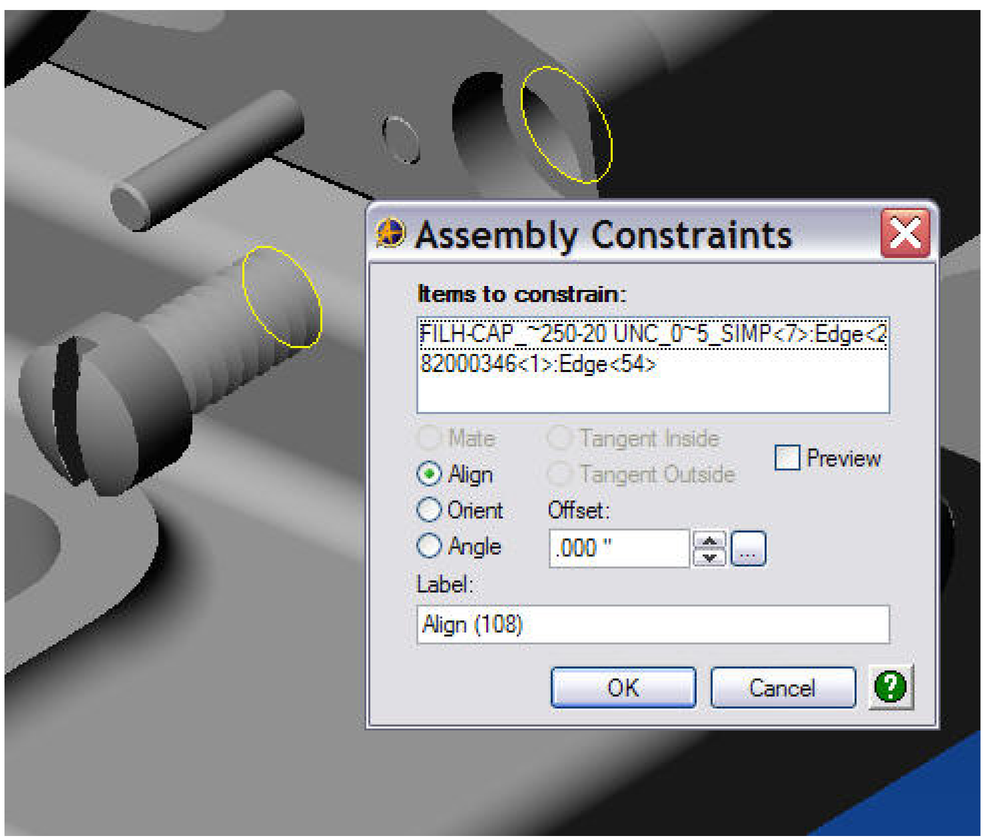
Add the Upper Plates, Eccentrics, Handles, and Pins using the same techniques you used for each part earlier in this exercise. Your Assembly should now look like this.

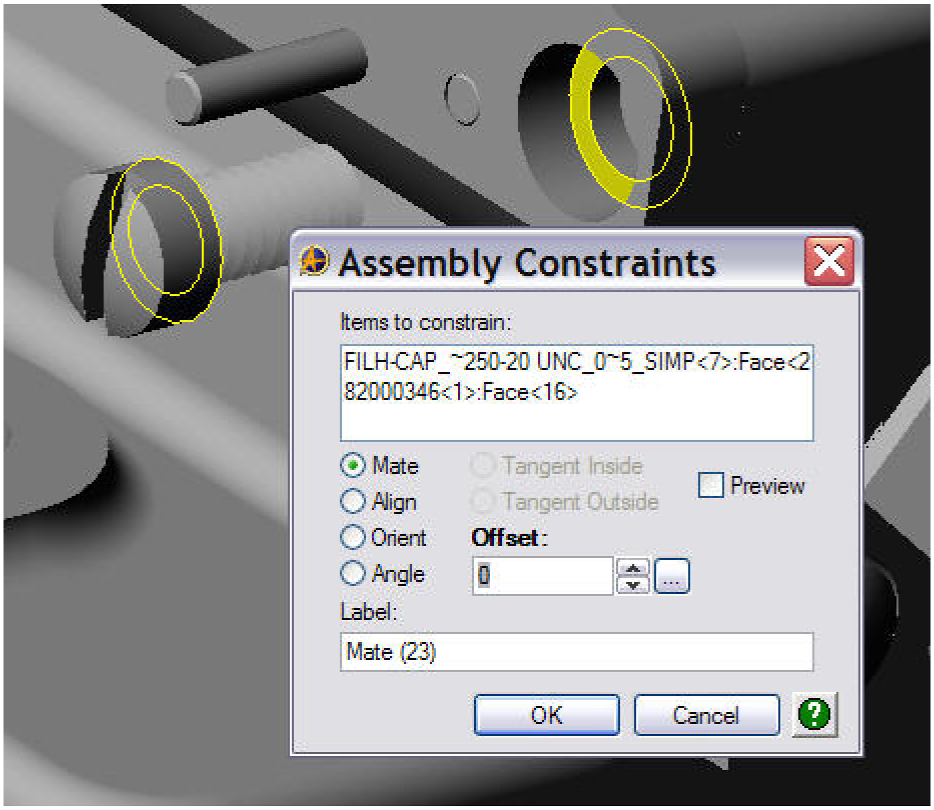


Now add eight copies of a .25 x 20 UNC x .625 L Fillister Head Cap Screws and constrain them using an a combination of a an Align and Mate constraint as shown.

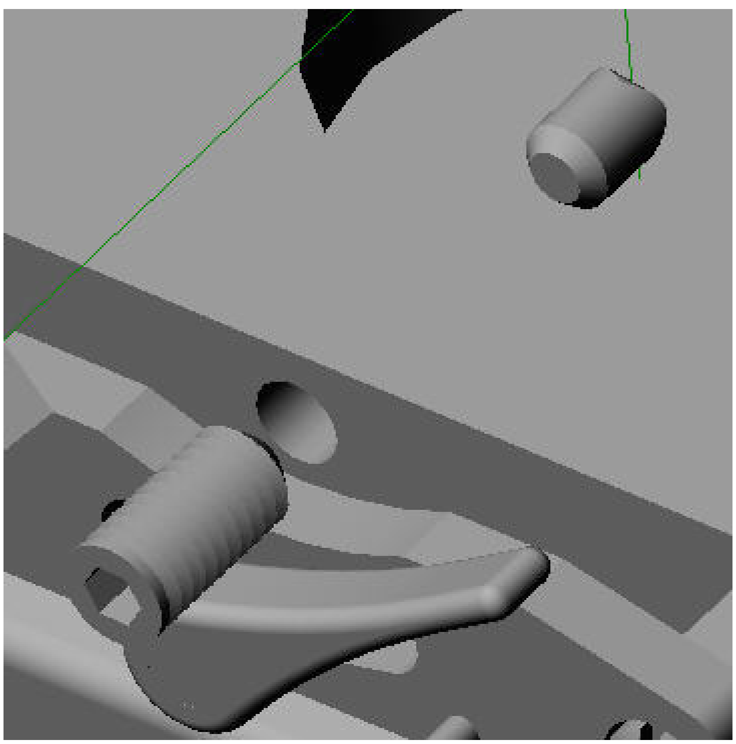


Align a circular edge of the screw with the corresponding circular edge of the small hole in the Lower and Upper Plates.



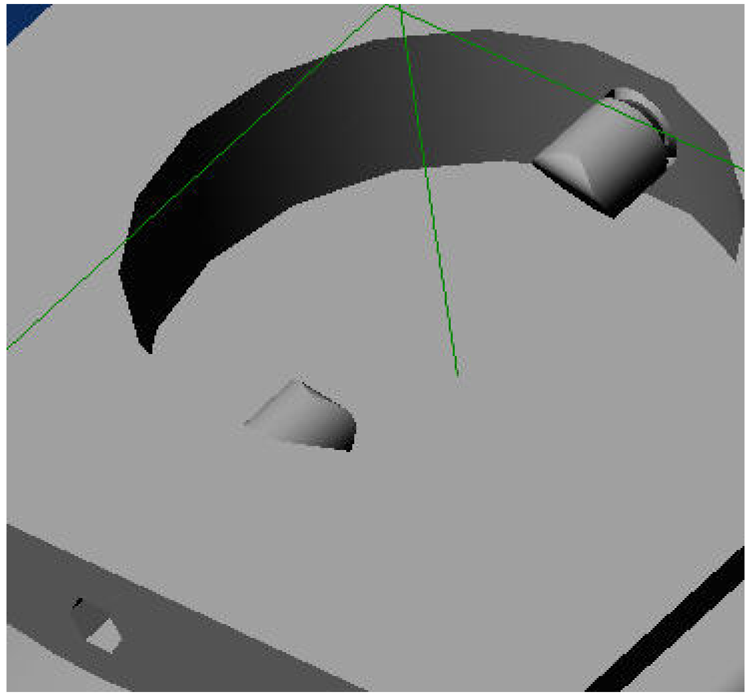


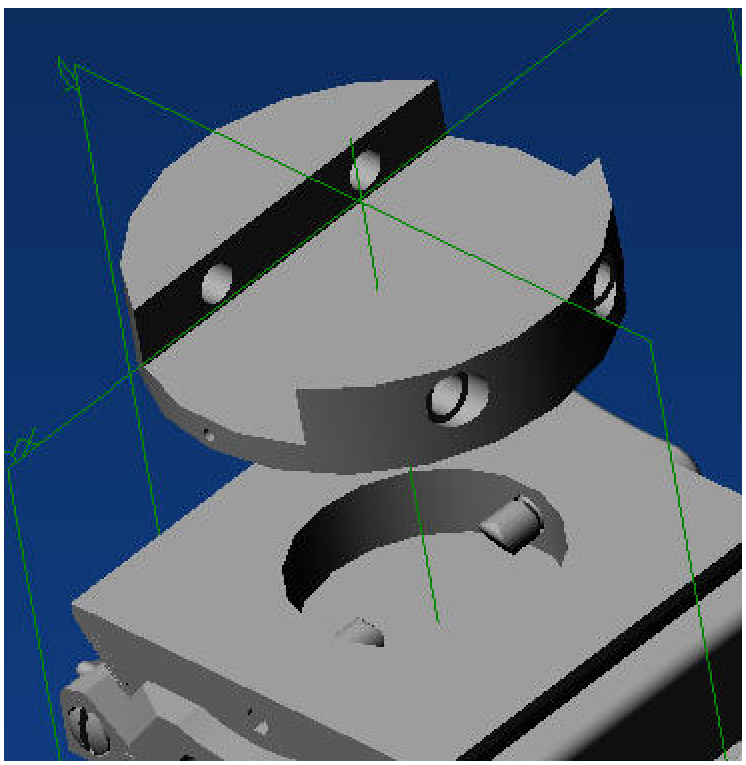
Use a Mate constraint between the inner clamping face of the screw and the bottom face of the large hole in the Lower and Upper plates.



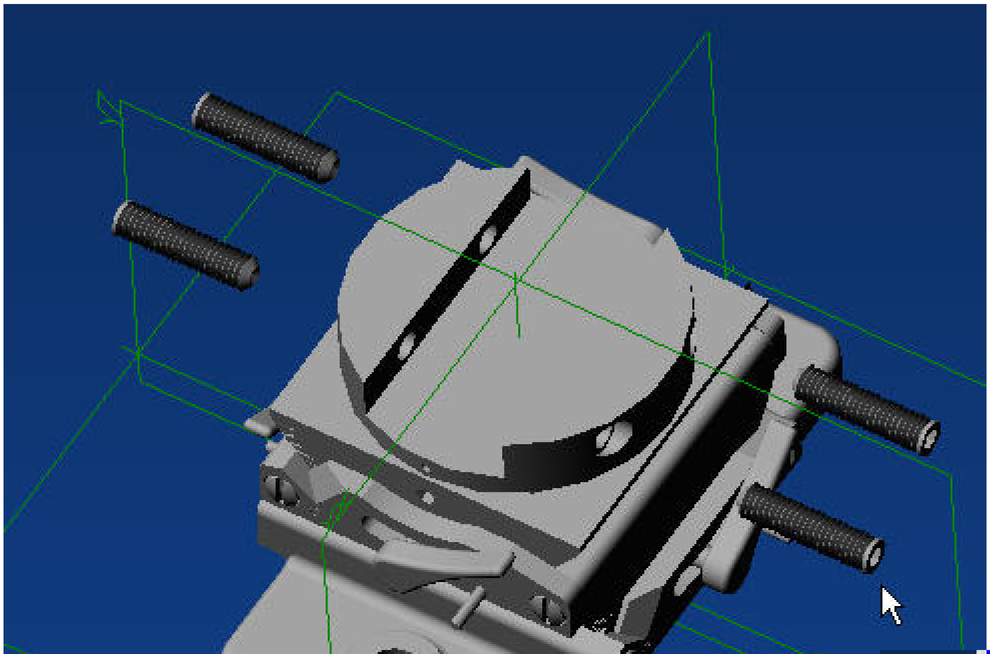
Add two copies of a .375 – 16UNC -2A x .562L Hex Flat Point Set Screw and the Clamp Plug. Use Align and Mate constraints to assemble them in the Upper Compound Member.

Your part should look like this.

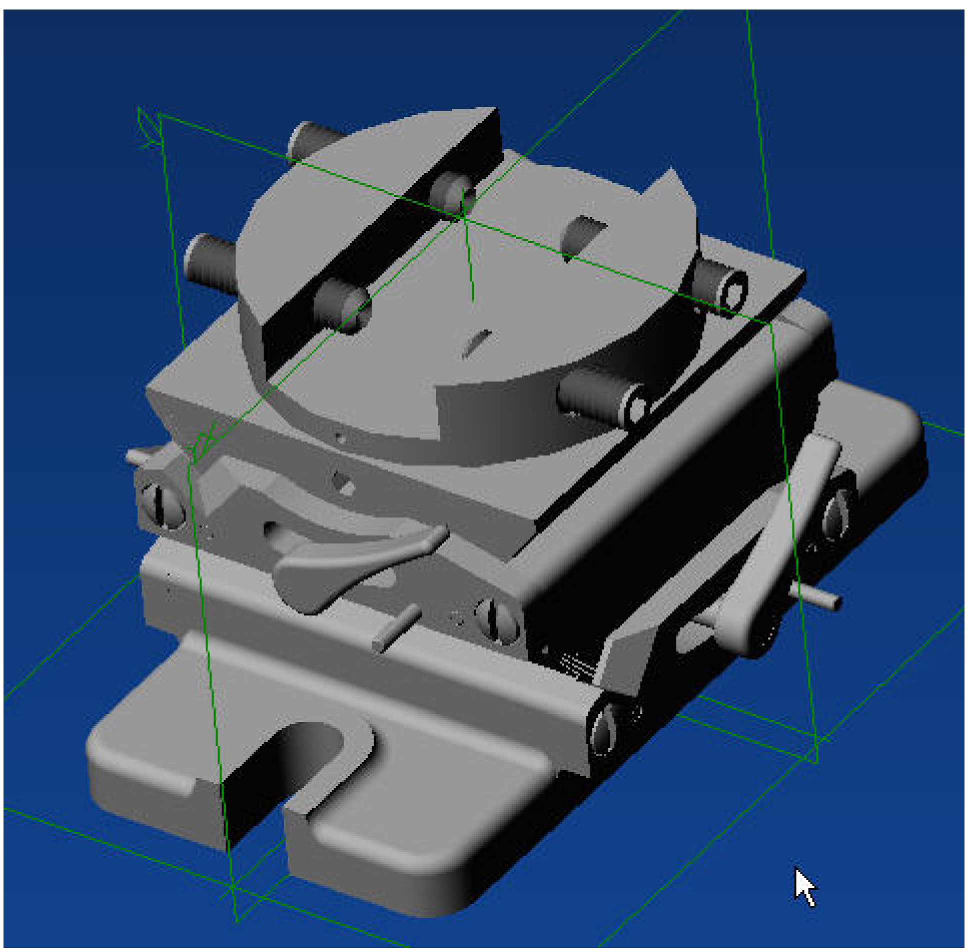




Add the Compound Tool Holder and constrain it using a combination of an Align and Mate constraint.



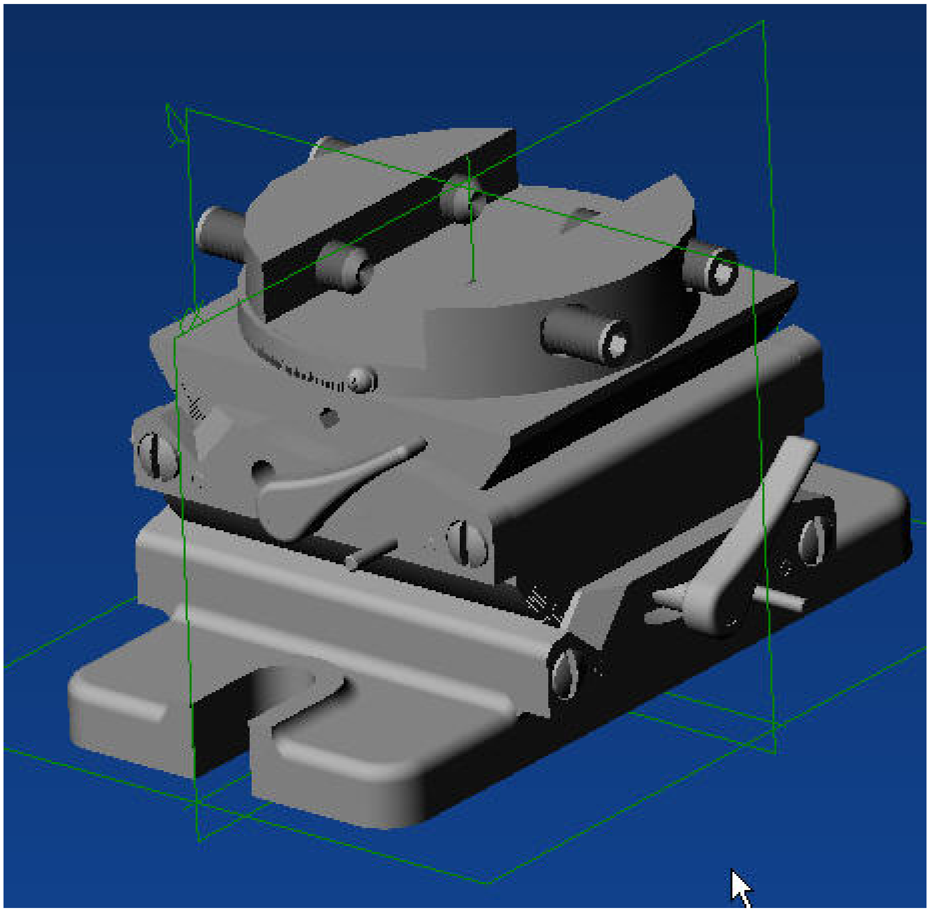
Add four copies of a .375-16 UNC-2A x 1.5L Socket Cup point Set Screw. Align them using a circular edge of each screw and the circular edge of the small inner hole in the Compound Tool Holder. Move them into position manually using the Move Part icon.



Your assembly should now look like this.

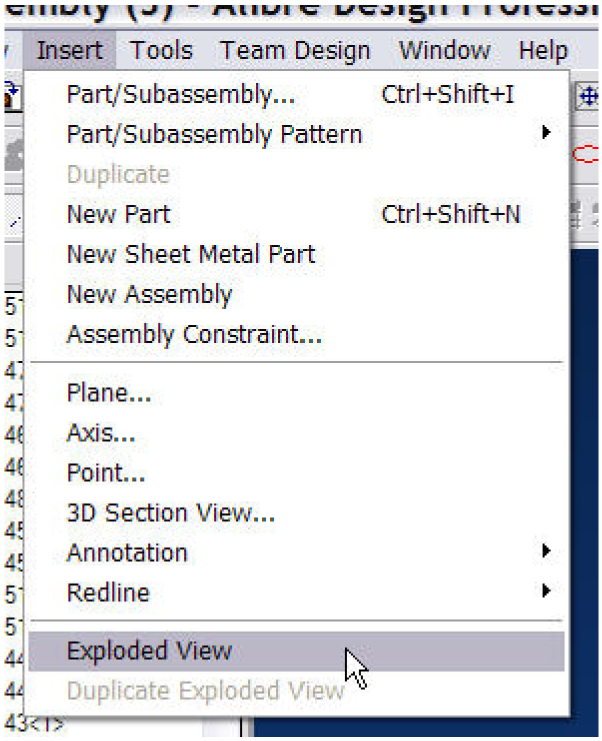
The last three parts that must be added are the Protractor Scale and the two #4-40 UNC-2A x .25L Round Head Machine screws that fasten it to the Compound Tool Holder.

Insert the Protractor Scale and align it using an Inside Tangent constraint between the inner face of the Scale and the outer face of the Compound Tool Holder, and two Align constraints. Create the first Align constraint between the inner faces of the hole in the Scale and the hole in the Compound Tool Holder, and the second between the bottom surface of the Scale and the bottom surface of the Compound Tool Holder.

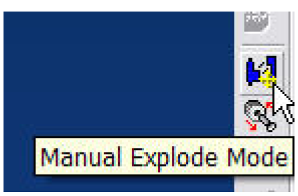


The completed Any Angle Tool Vise Assembly Reference.

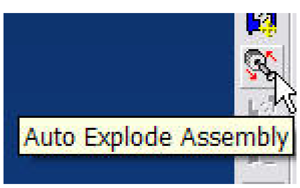
Now we'll add and exploded view of our Assembly for use in creating an exploded Assembly drawing.



Under the Insert tab on the upper toolbar select Exploded View. Then select either Manual Explode or Auto Explode from the Right hand tool bar.

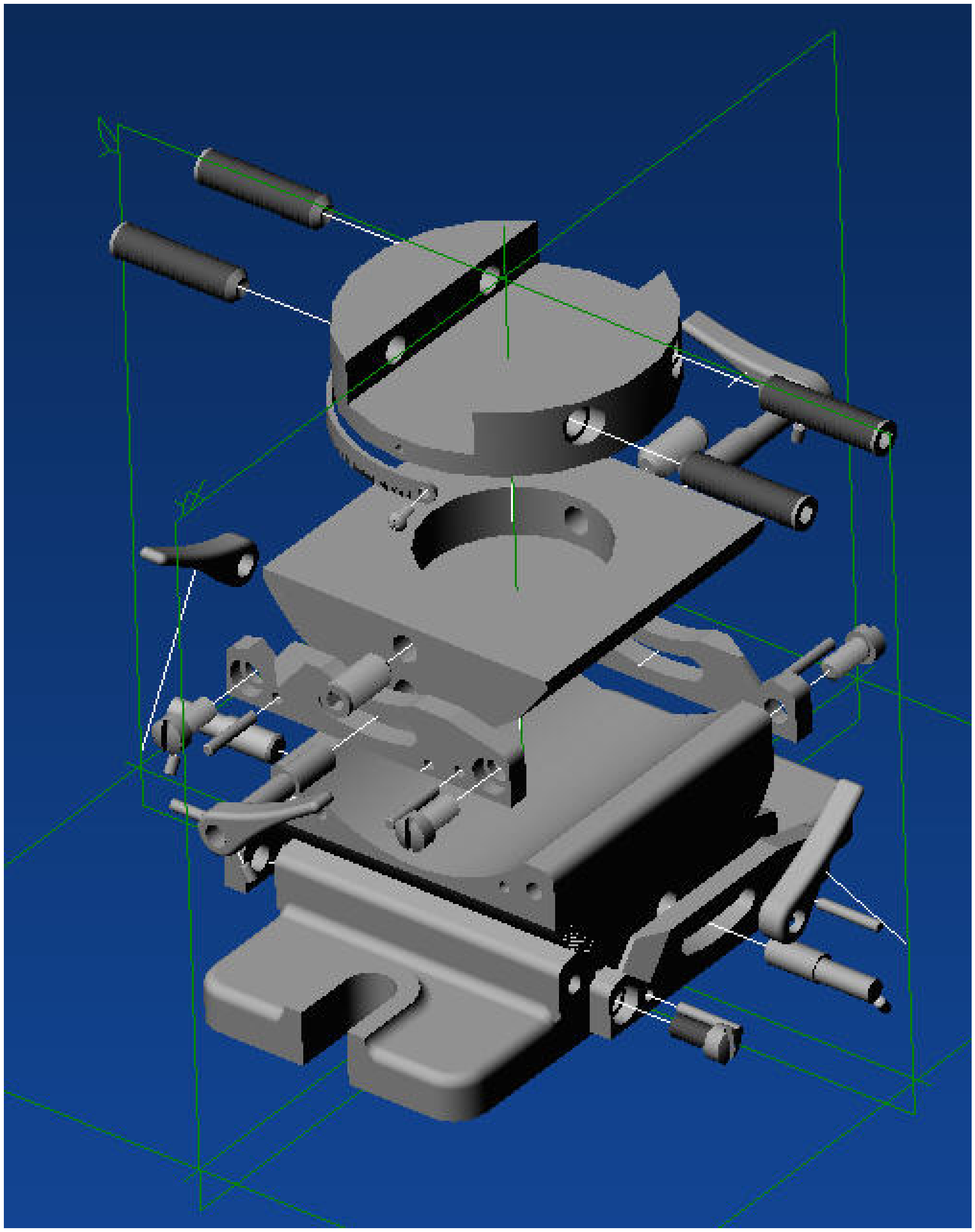


Using Manual Explode Mode allows you to move the individual parts of an assembly to any position you require.

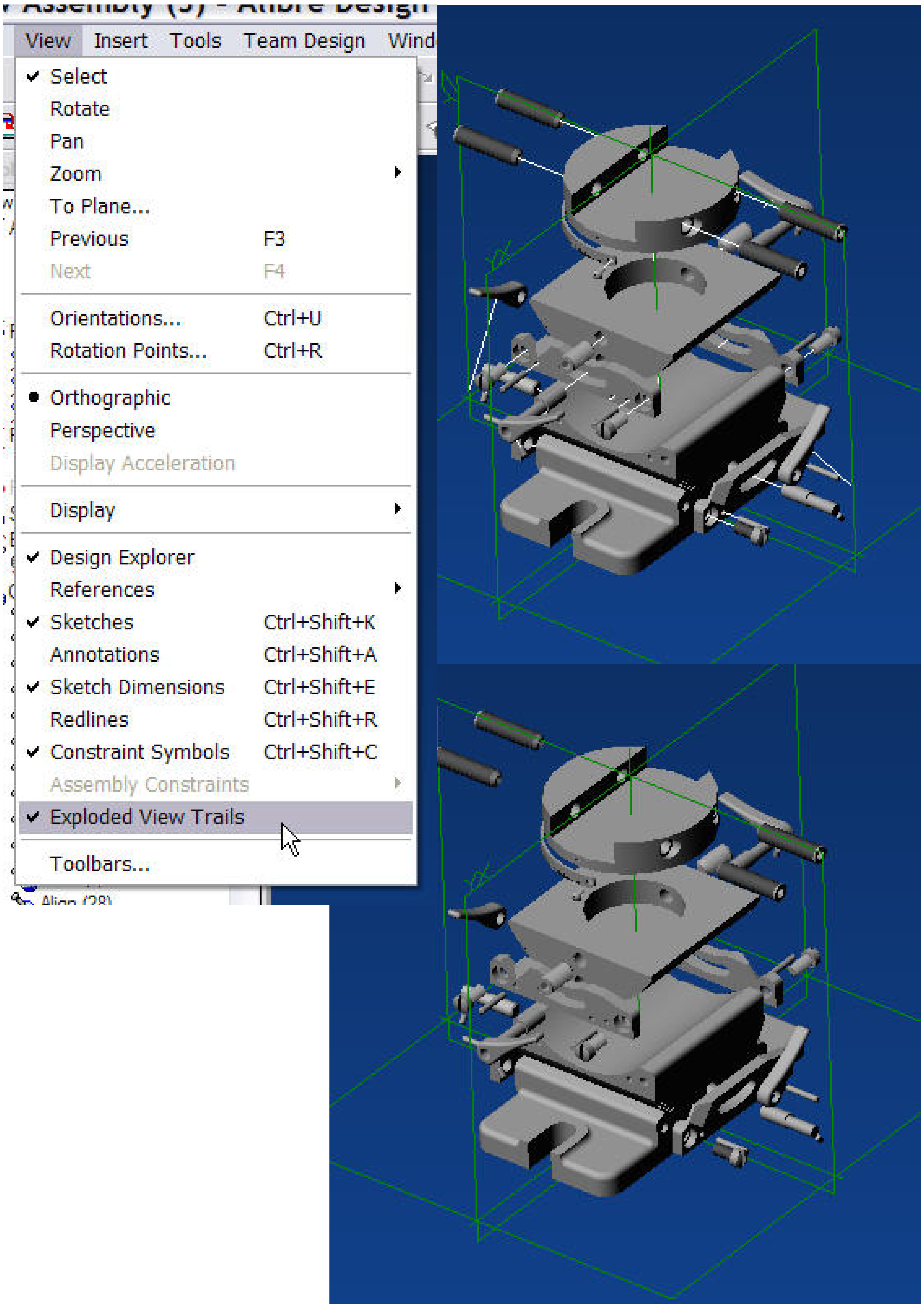


Using Auto Explode Assembly separates the part

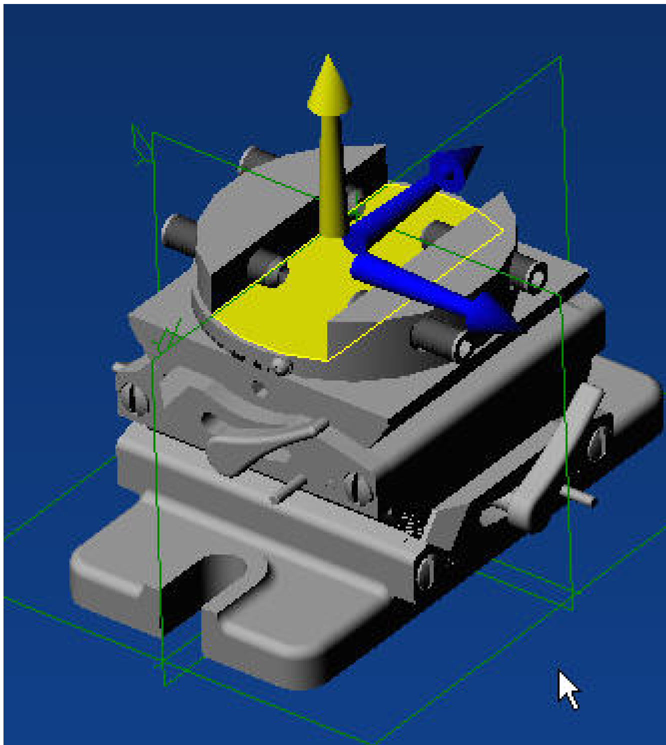
This is what the Any Angle Tool Vise looks like after an Auto Explode. Note the ‘Explode Trails, which show as white lines and describe the path of the ‘Explode’.



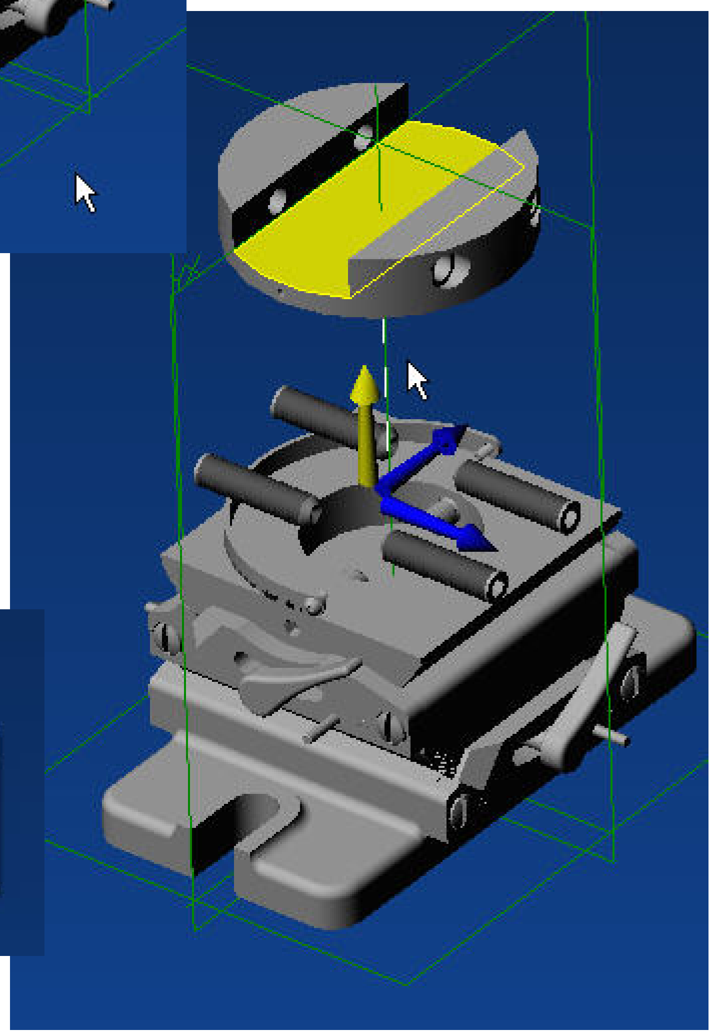
You can hide these if you'd like by selecting View, and un-checking 'Explode View Trails'.



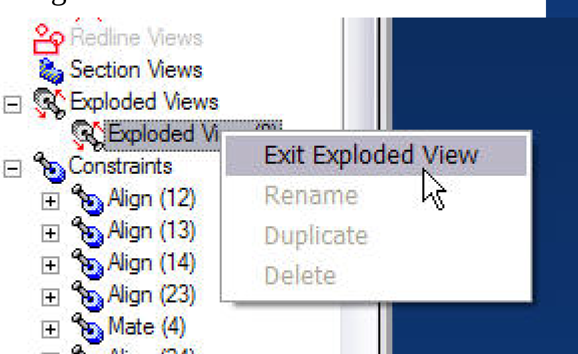
Using ‘Manual Explode Mode’, you move the individual parts of the assembly by clicking on the Manual Explode Mode icon and then selecting the part you want to move as shown below.

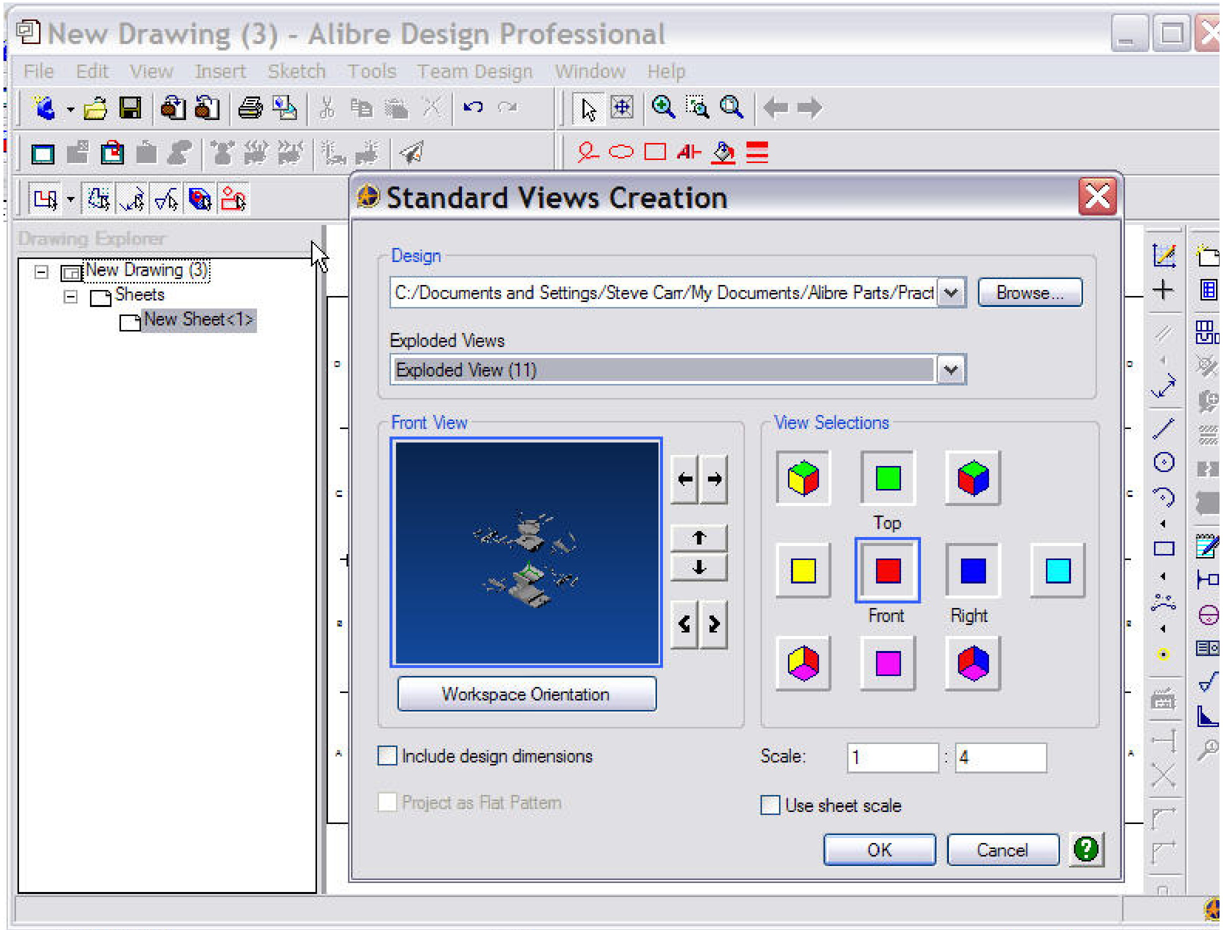


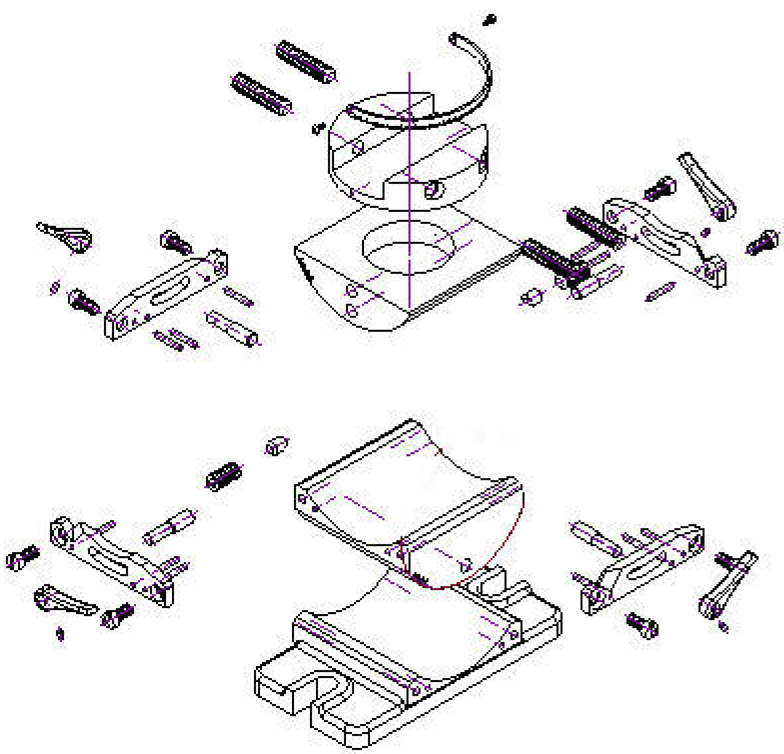
Select the axis arrow along which you wish to move the part and drag the part to the desired position with your mouse. Note that ‘Explode View Trails are created in this mode just as they are in ‘Automatic’ mode.



After your done moving the parts in either mode, simply save the file and then click on the ‘Exploded View’ header in the Design Explorer panel and select ‘Exit Exploded View’. This collapses your assembly to its original state.







Creating an Exploded view in a drawing simply requires you to select the view during the drawing view creation phase.

This is an example of an Exploded view in a drawing.

You can detail an Exploded View in the same manner as any other view i.n your drawing.