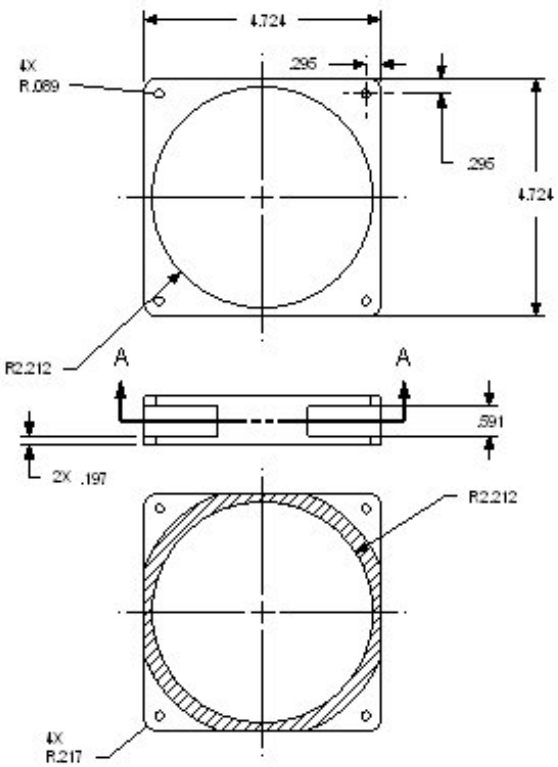
39-245
RAPID PROTOTYPE DESIGN

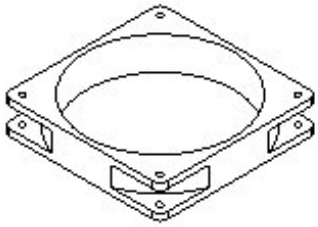
CARNEGIE MELLON UNIVERSITY

SPRING 2007

MAKING THE FAN HOUSING

Our goal is to make the following part:



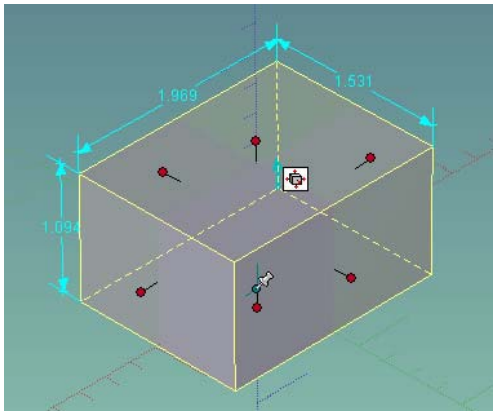


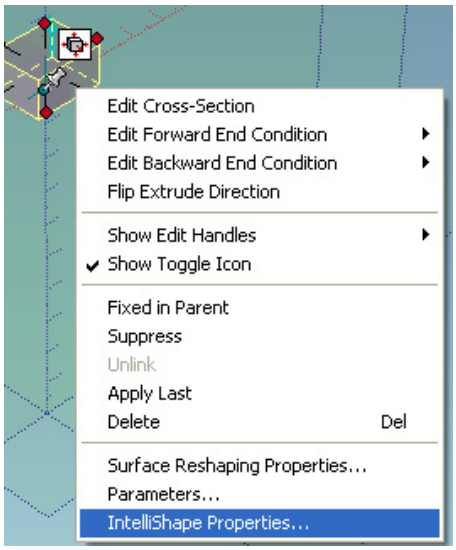
This part is made up of two plates joined by a cylinder with holes in the corners of both plates. The corners of the plates are rounded. We will start by making the bottom plate with the dimensions 4.724 x 4.724 x 0.197. Then we will make a cylinder with an inner radius of 2.212, an outer radius of 2.527 and a height of 0.591.

1.1 Making the bottom plate

Start a new scene and drag a block into the scene.

Select **View →Sizebox dimensions** from the menus. Click to select the block so it turns yellow, and you will see the dimensions.

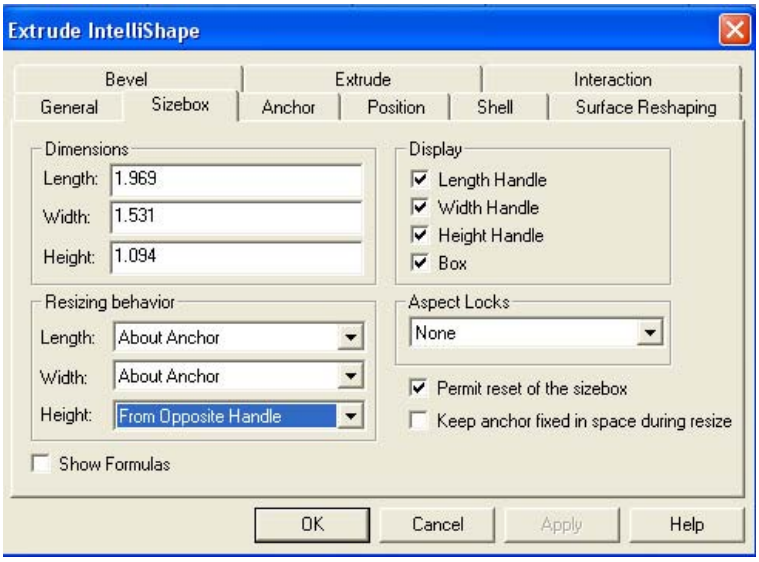




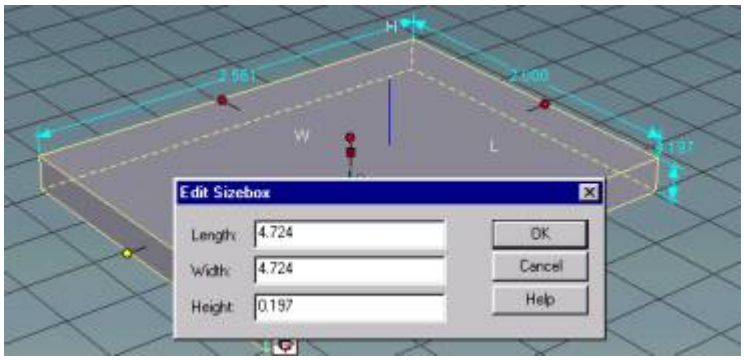
To change the resizing behavior, right click on a **face** of the block and select Intellishape Properties at the bottom of the menu

Select the **Sizebox** tab. We need to change the default resizing behavior for **Width** and **Height**. Pull down the menu next to **Width** and select **About anchor**. Do the same for **Length**

After you press OK, IronCAD will resize the width and length around the anchor (the pin in the blue/green dot) and resize the height from the opposite face.



Now, select the handle on the top face of the block and right click. Since we have selected this handle, IronCAD will resize the block relative to the opposite face. Since the opposite face is sitting on the *x-y* plane this is what we want.



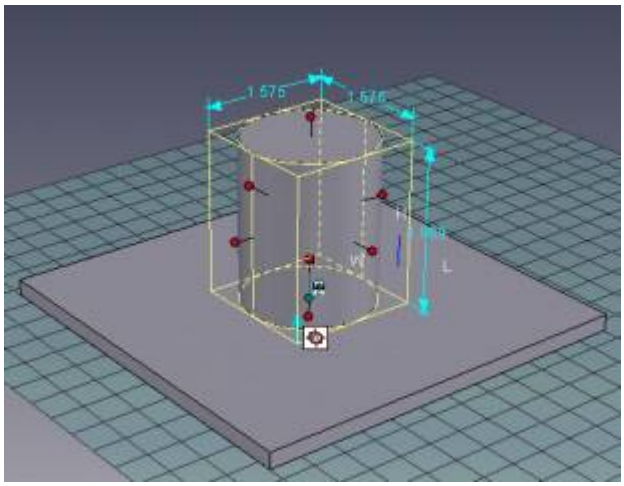
Now we need to enter the dimensions of the length, width, and height. Enter **4.724** for the width and length and **0.197** for the height.

Press the **fit scene** button if your part goes off the screen.

If your plate is no longer sitting with it's center point on 0,0,0, see the handout on floating parts.

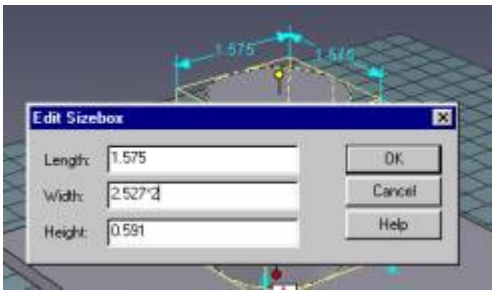
1.2 Adding the central cylinder

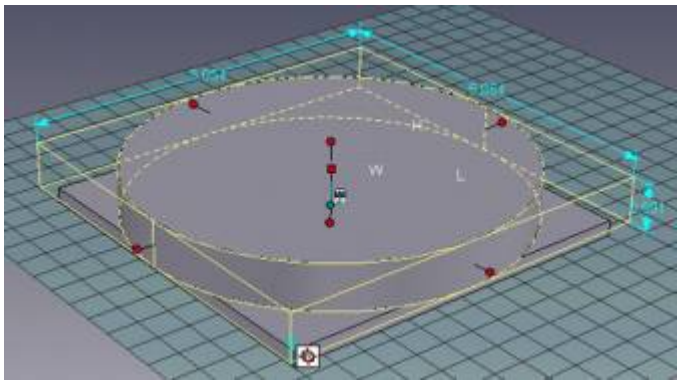
Go to the **Shapes** menu and select a cylinder. Drag it to the plate. As you drag it close to the center, the green dot will become brighter. Drop the cylinder when the dot is highlighted. If your cylinder isn't sitting on top of the plate in the middle, see the handout on floating parts.



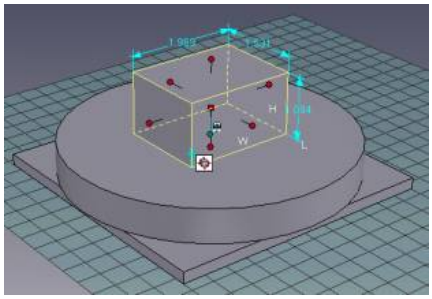
Notice that because earlier we selected the option to show the dimension box, the dimensions are shown for all selected parts now. You can toggle this option on and off depending on what you are working on.

Right click on the top handle to get the edit sizebox window. From the drawing, we know that the height of the cylinder is 0.591 and the outer radius is 2.527. In the sizebox, set the height to 0.591. Since the radius is 2.527, the width of the cylinder is twice that. IronCAD will do the arithmetic for you, so you can enter 2.527*2. As soon as you enter this, the length will change to the same value since the length and the width of a cylinder have to be the same.

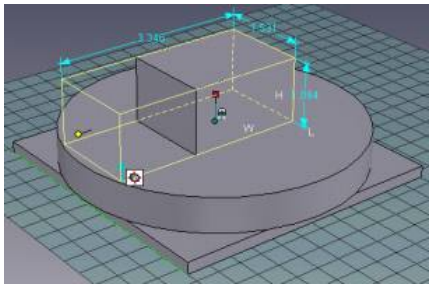




1.3 Adding the top plate

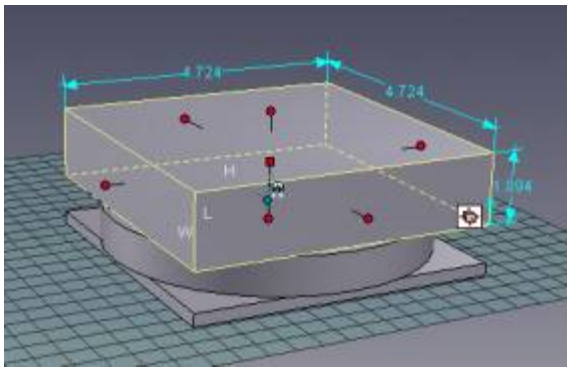


Select another **block** from the **Shapes** menu. Drag it to the top face of the cylinder and drop it when the green center dot is highlighted. We will use smartsnap to set its dimensions.

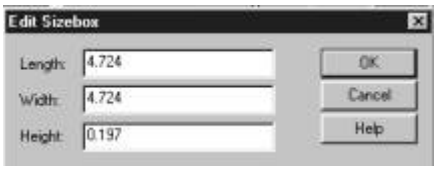


Holding the shift key down, pull on the left-hand handle of the plate. As you pull it toward the edge, you will first see the edge of the cylinder highlighted and then the edge of the bottom plate.

Release the mouse when the edge or face of the bottom plate is highlighted. The face of the new block is now co-planar with the face of the lower block. Do the same thing with each face. Use the **orbit** tool to see and change the other faces.

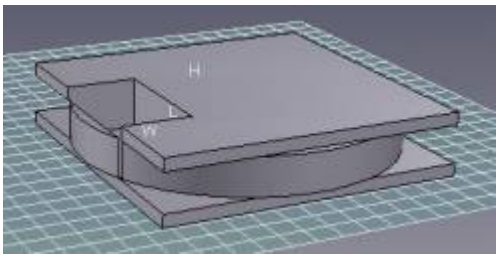


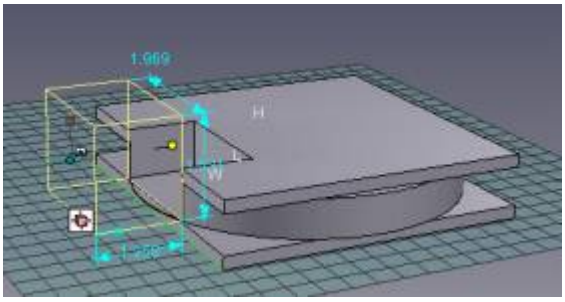
Now all we have to do is change the height of the top plate. Select the top handle. Right click and change the height to 0.197.



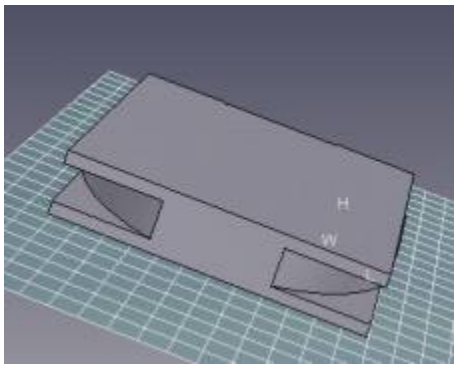
1.4 Removing the cylinder overlap

Next, we will get rid of the material of the cylinder that's hanging out over the edges using a negative block (H-block). Go the **Shapes** menu and select an H block. Drag it to the left hand end of your part. Drop it when the green dot at the midpoint of the edge is highlighted. (The screen dump of this doesn't show anything interesting. You will just have to do it.) When you drop the H-block, IronCAD will take a chunk out of the part. This is not what we want!

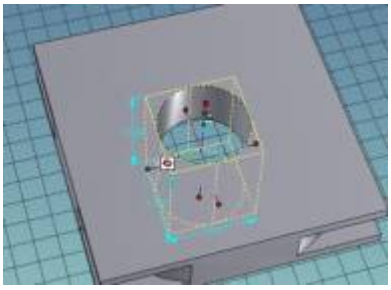




Select the H-block as an intellishape. (If you can't select it, select the scene browser and select the H-block from the tree.) Drag the left hand face of the H-block to the left. Drag the bottom face so that it extends beyond the bottom of the part. Now use Smart Snap again. Press the shift key, and drag the right-hand handle toward the edge of the part. When the edge of the lower plate is highlighted, release the mouse. The negative block has subtracted the overhang. Perform the same operation for all the sides.



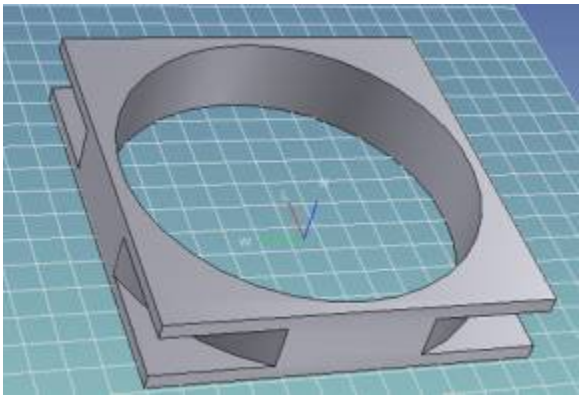
1.5 Making the hole in the center of the part



To make the hole for the center of the cylinder, go to the **Shapes** catalog to the right of the drawing area and select the H-cylinder. Drag it and drop it on the center of the top plate. Be sure the green dot that marks the center of the top face is highlighted when you drop the shape.

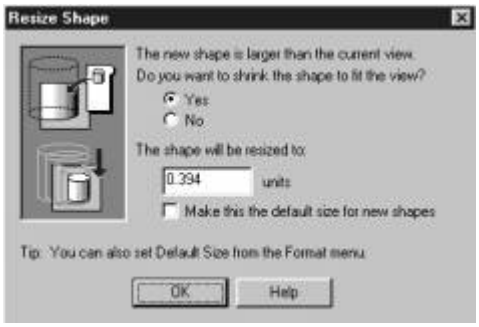
The inner radius of the cylinder is 2.212. So, select the top handle and right click to get the size box. Enter **2.212*2** for the width of the H-cylinder.

After you press enter, your part should look like:



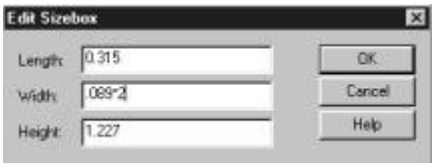
1.6 Adding the holes in the corners

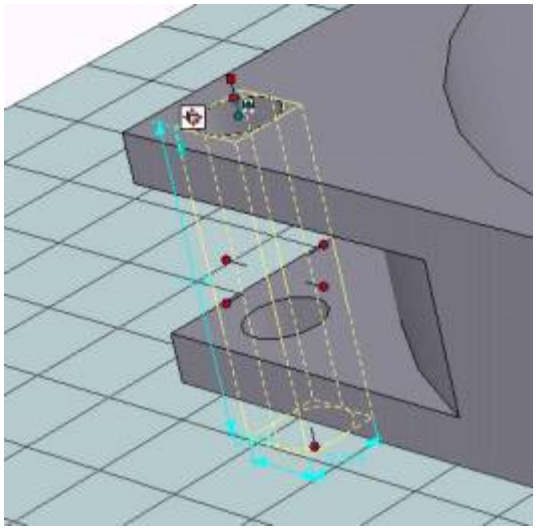
From the **Shapes** palette, drag an H-cylinder (negative cylinder) onto the top face of the corner. You may get the dialog box on the right. If you do, say OK. This will make the cylinder about the right size for the current view.



Use the IntelliShape handles on the H-cylinder to make it go all the way through both plates. If you can't click on the H-cylinder, use the Scene Browser to find it.

To size the hole, right click on the top handle. Select **Edit Sizebox**. The radius of the hole is 0.089 inches, so the width is **0.089*2**. Remember that for cylinders, you only need to specify either the length or the width since they have the same value. The height of the negative cylinder doesn't matter as long as it extends beyond both plates.

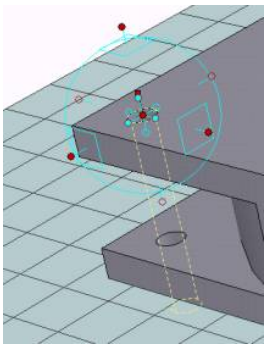


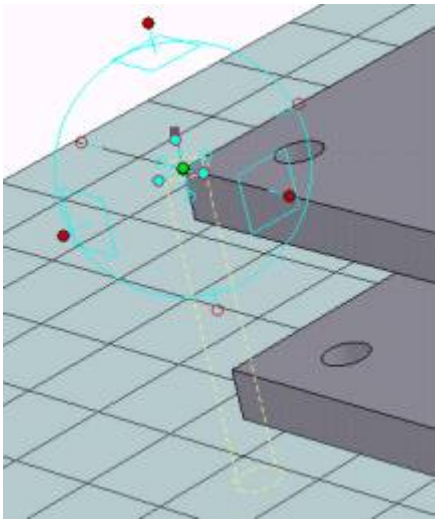


1.7 Locating the hole

Now that the hole is the right size, we will get it in the right location -- with its center 0.295 inches from each edge. With the hole selected, click on the triball tool button.

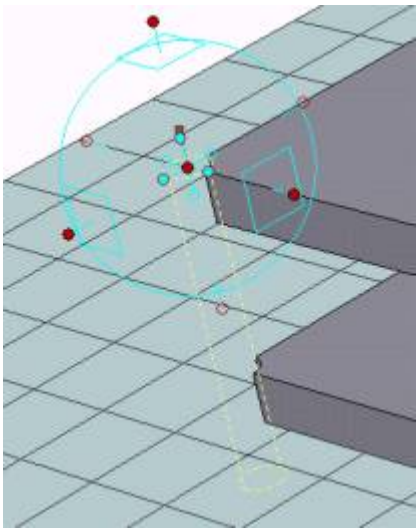
Strategy: We are going to move the center of the hole to the corner of the plate. This is easy using smart snap. Then we will use the triball to move the center of the cylinder to exactly 0.295 inches from the edge.

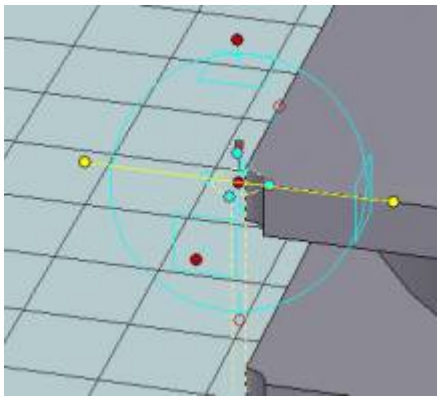




Select the center of the triball. *With the shift key depressed*, move the cursor until the corner point is highlighted.

Release the mouse. You will see a small cylinder removed from the top and bottom corners.

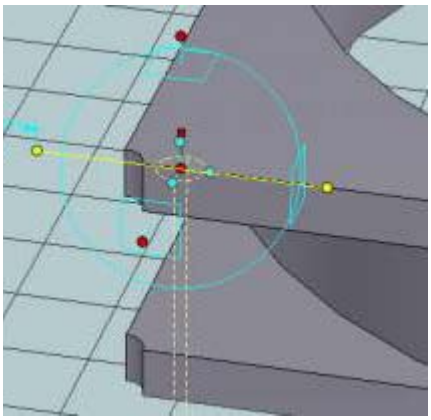




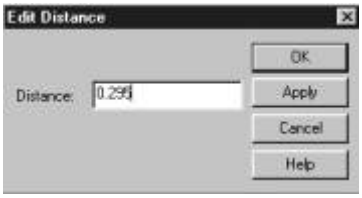
Right click on the triball axis parallel to the *x-y* plane. It will turn yellow, indicating that it is locked, that is, that the cylinder can only move along this axis.

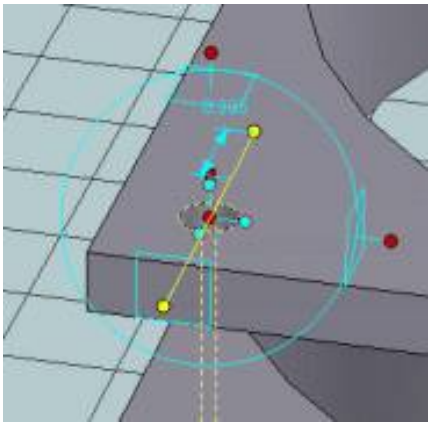
Grab one of the highlighted yellow handles and move it to the right. You will see a dimension showing you how far you have moved the part. We want to move it 0.295 inches from the edge.

Rather than trying to position it with the mouse, release the mouse and right click on the dimension. Select Edit Value.



In the dialog box, enter 0.295:





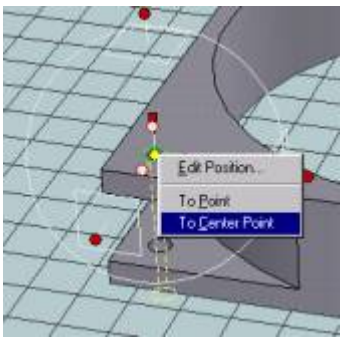
The hole will be positioned on the edge of the plate, 0.295 inches from the corner. Select and lock the other axis. Using the same technique, move the hole to 0.295 inches from the corner.

1.8 Copying the hole

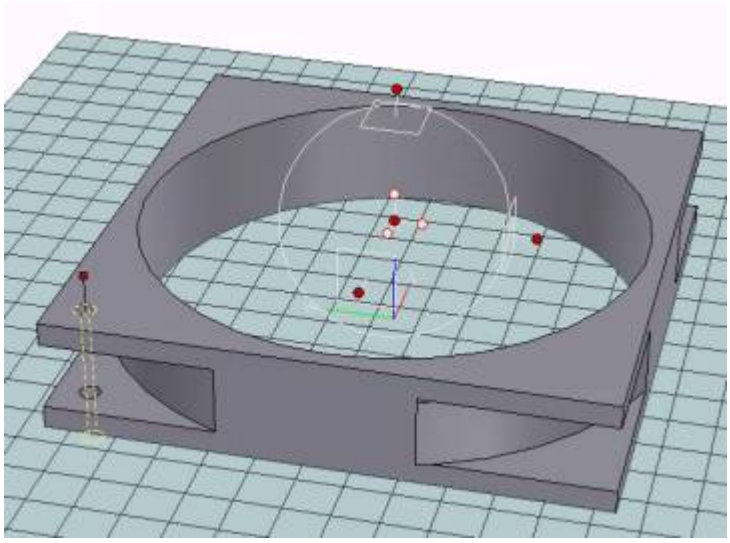
Now we are going to use the triball to copy the hole to the other corners. If one axis of the triball is still yellow (locked), click somewhere in the background to unlock it.

Strategy: We are going to move the triball to the center of the fan housing. We will then make 3 linked copies of the hole, each at 90 degrees from the original. The holes should be linked because if the diameter of one hole changes, they should all change.

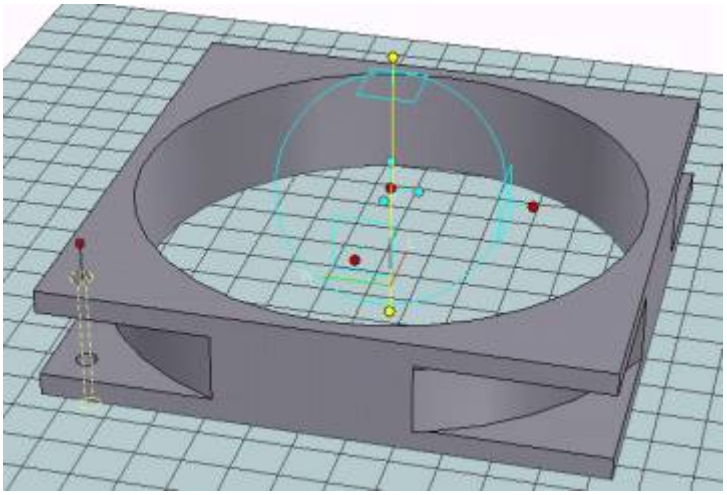
Zoom out to see the entire part. Press the space bar once. The triball should turn white. This allows us to move the triball without moving the part. Right click on the center of the triball and select **To Center Point**.



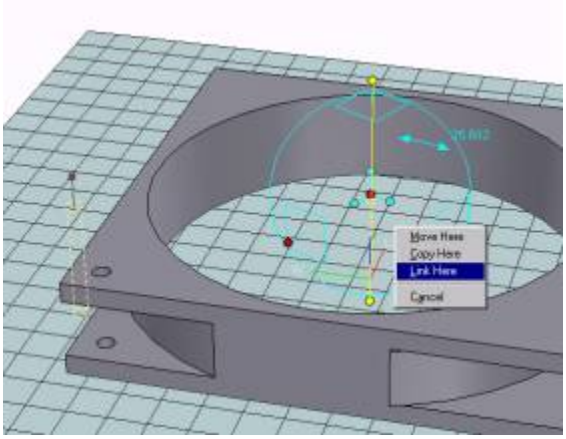
Move the cursor toward the center of the part. When the top circle is highlighted (green), drop the triball. Notice that the hole is still selected, but its triball is now centered on the entire part, not on the hole.



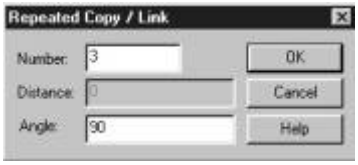
Press the space bar so that the triball and the part work together. The triball should turn cyan (blue). We want to move the copies through an angle -- that is, we want to rotate rather than translate. We want to rotate about the z axis, so lock the z axis by clicking on its handle. Your scene should look something like:



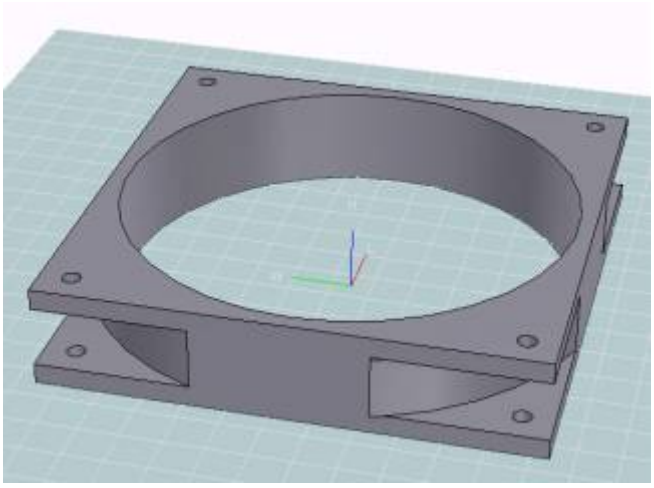
Right click within the triball -- but not on any handles or objects. As you move the cursor, the hole will rotate. Release the mouse anywhere. You will get the pop-up menu. Select **Link Here**.



Because you were rotating when the link option was selected, this time you will be able to enter an angle rather than a distance. You want 3 copies at 90 degrees.



When you press OK, you will have four holes in the corners of the part!



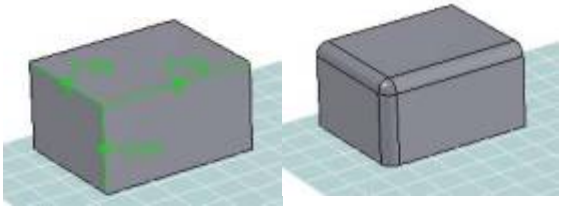
1.9 Blends

Next we need to bevel the corners of the part. (If you look back on page 1, you will see that the outer corners of the part are rounded.) Select the tool button for Blended Edges.

A new toolbar will appear. This toolbar lets you select whether you want to bevel a vertex, an edge or a face. The options in the toolbar also let you enter the type of blend and the radius of the blend. You can also select a variable blend and specify the start and end blend radius. To see how these options work, let's start with a practice part. Drag a block into your drawing area, away from the fan housing. In the first option, select **Face/Edge/Vertex**. In the second option, select **constant**, and in the third option, select **0.158**. (These are probably the defaults.)

Move your mouse around the block. Your cursor should change shape. The cursor will look like a dot when you are close to a **vertex**, the point at which several lines and faces meet. The cursor will look like a line when you are close to an edge, the **line** where two faces meet, and it will look like a sheet of paper

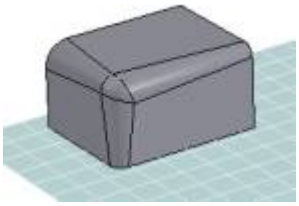
when you are over a **face**, one of the surfaces of the part. (Again, since the cursor doesn't show up in the screen dumps, I can't show you this.)



First, select a vertex. Notice that IronCAD shows you where the blends will be applied.

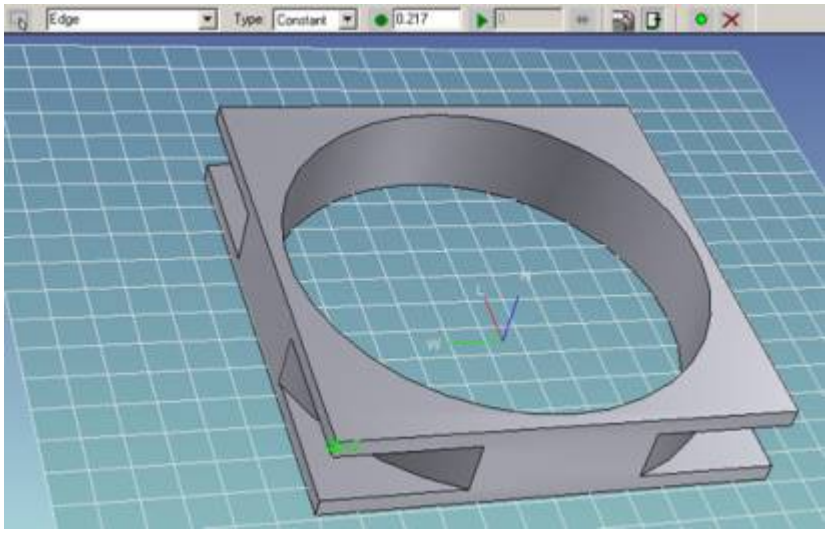
If you want IronCAD to make the indicated blends, press the green **go** button. If you don't want IronCAD to make the blend, press the red **X** button.

After you make this blend, press the undo button, and then try the other options to see how blend works when you select edges and faces. Notice that the blend toolbar disappears after the blend is created. You need to select the blend tool again to get it back.



Finally, change the type of blend from **Constant** to **Variable**. You will now be able to enter a start and end radius for the blend. Press the **go** button again. Here is a variable blend on a vertex:

Now we are ready to blend the corners of the fan housing. Press the **blend** tool button. From the drawing, we know that the blend radius of the corners is 0.217. Enter this value for the blend radius and check to be sure that the blend radius is constant. Select one of the edges (lines) at the corner of the part. (I changed the order of the lab because it's easier to position the holes before the blends are made. Your part should have holes in the corners.)



Press the **go** button. Work your way around the part, applying the blend to each of the eight corners.