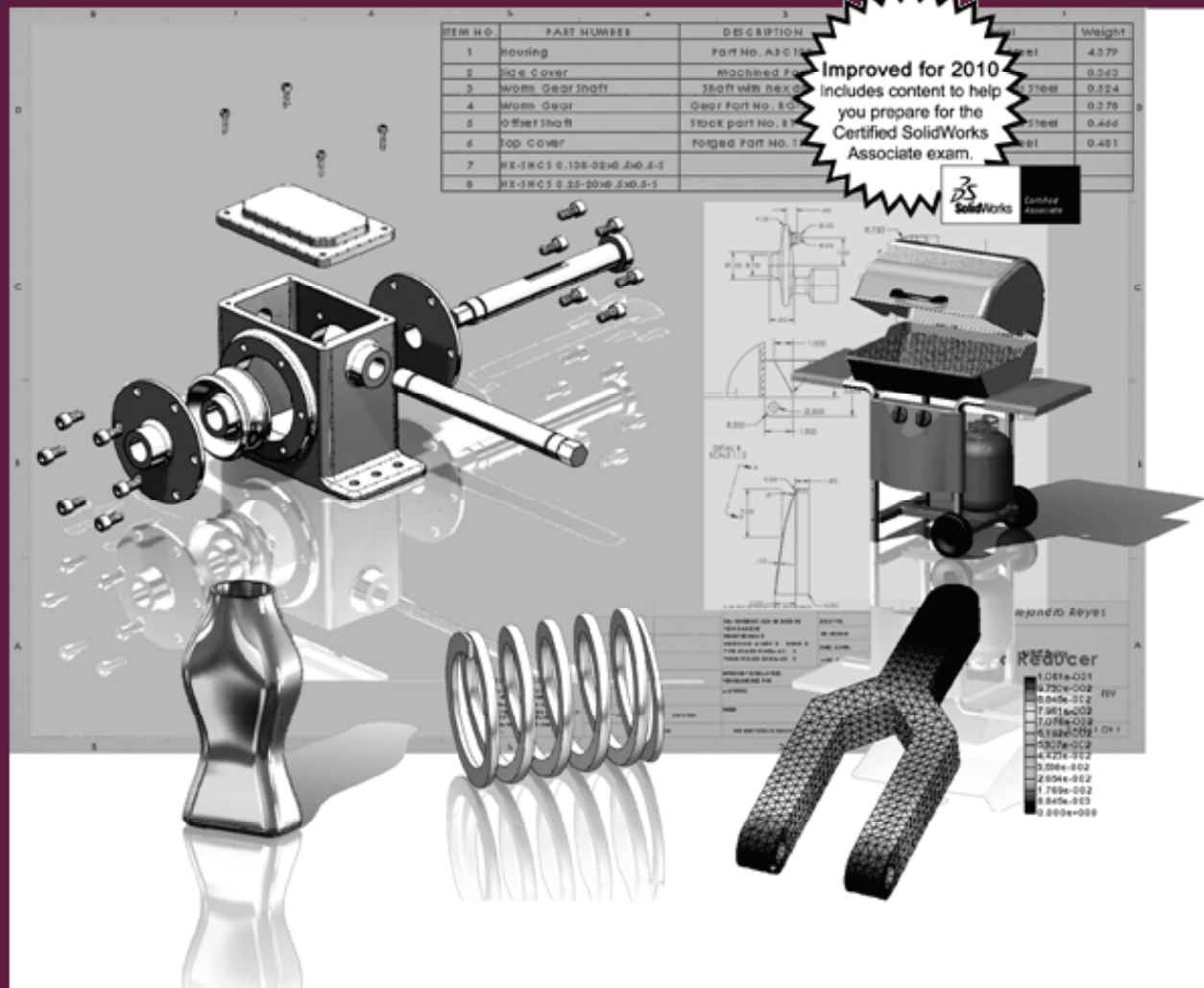


Beginner's Guide to SolidWorks 2010



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Certified SolidWorks Professional

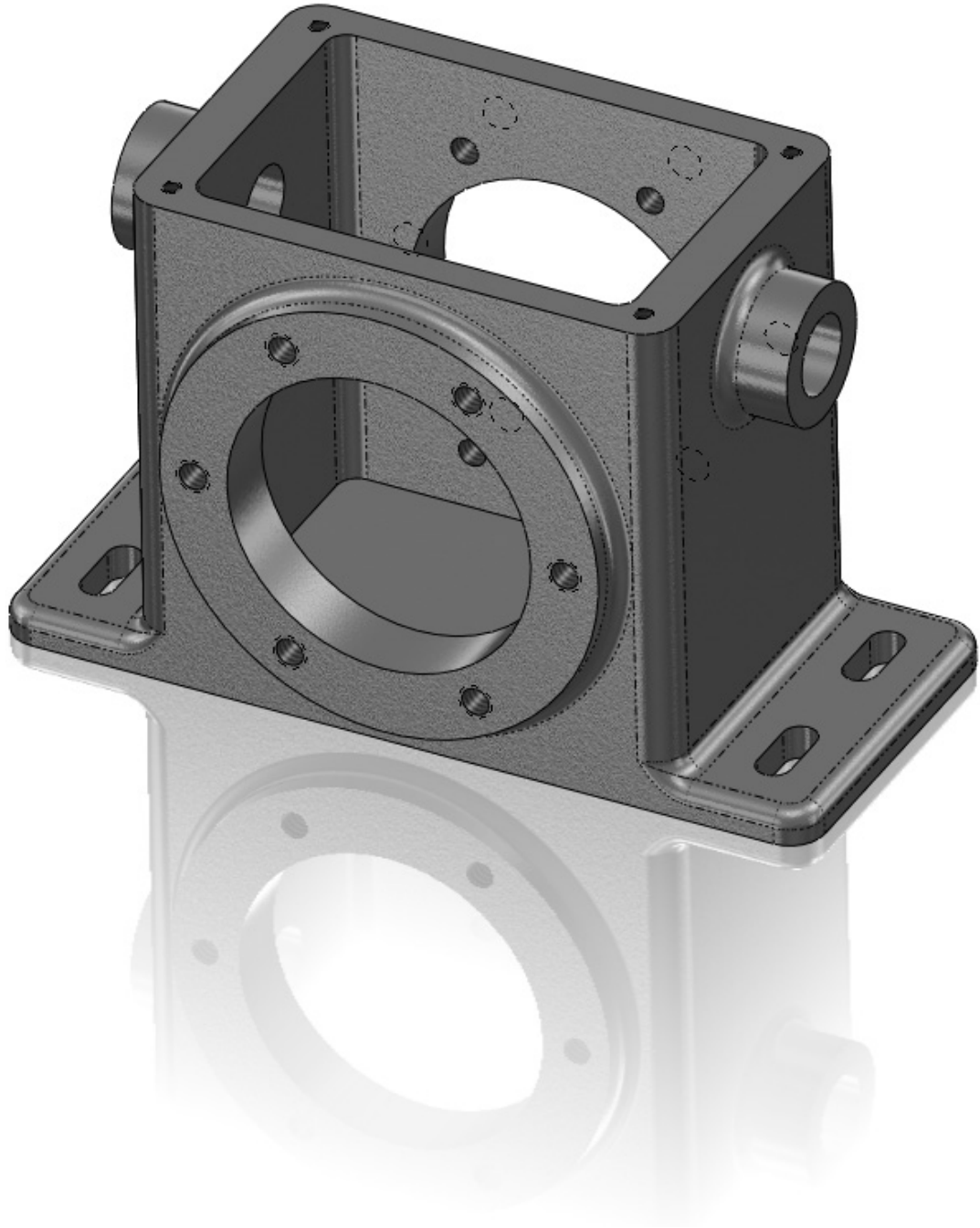


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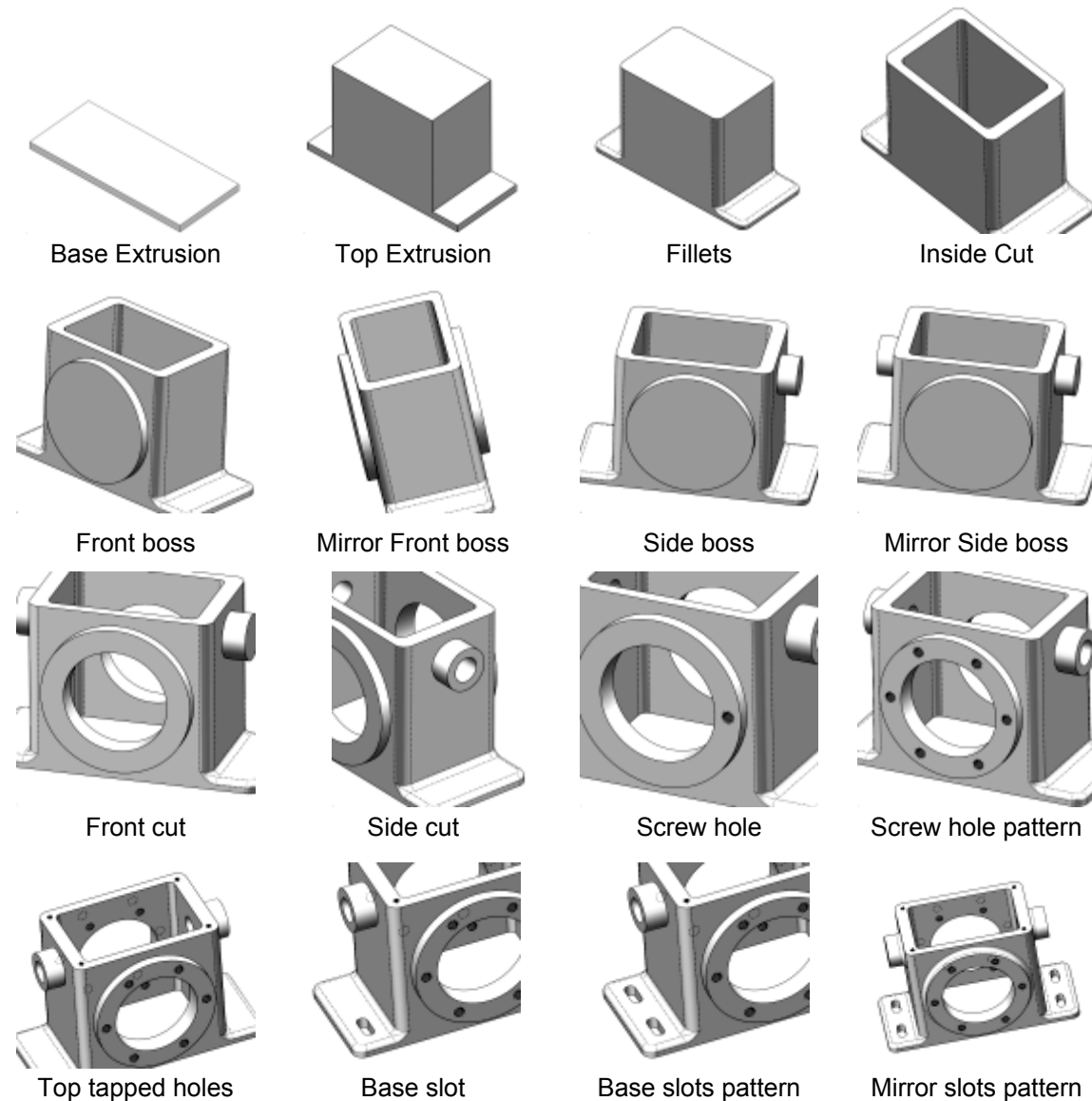
Schroff Development Corporation

The Housing

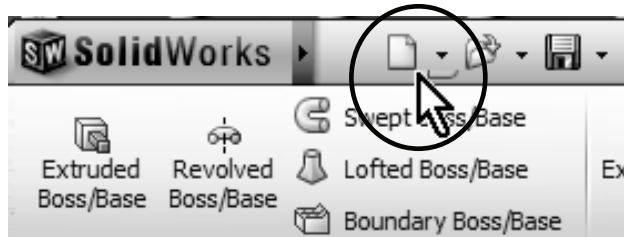


When we start a new design, we have to decide how we are going to model it. Remember that the parts will be made one feature or operation at a time. It takes a little practice to define the optimum feature sequence for any given part, but this is something that you will master once you learn to think of parts as a sequence of features or operations. To help you understand how to make the Housing part, we'll show a "roadmap" or sequence of features. The order of some of these features can be changed, but remember that we need to make some features before others. For example, we cannot round a corner if there are no corners to round! A sequence will be shown at the beginning of each part, and the dimensional details will be given as we progress.

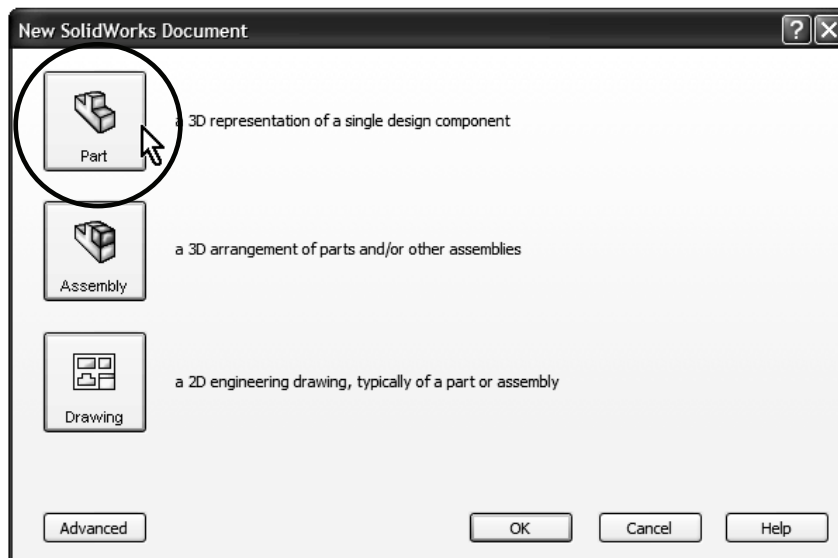
In this lesson we will cover the following tools and features: creating various sketch elements, geometric relations and dimensions, Extrusions, Cuts, Fillets, Mirror Features, Hole Wizard, Linear and Circular Patterns. For the Housing, we'll follow the following sequence of features:



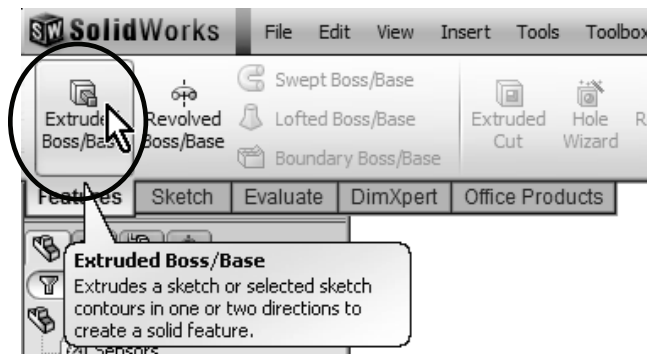
1. - The first thing we need to do after opening SolidWorks, is to make a **New Part** file. Go to the “New” document icon in the main toolbar and select it.



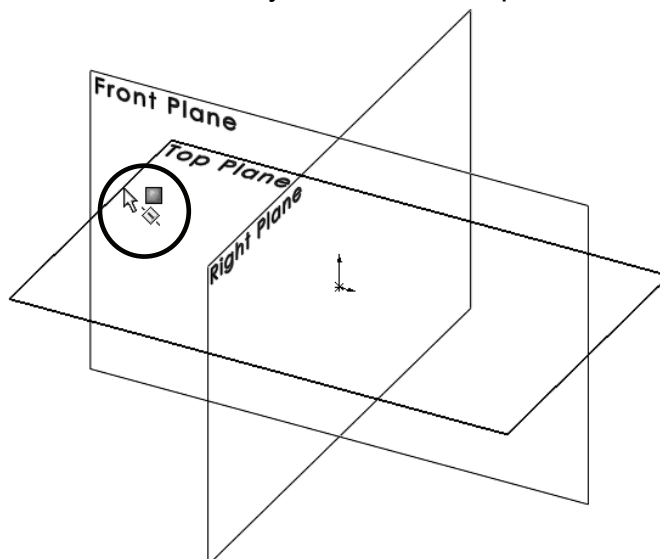
2. - We are now presented with the New Document dialog. If your screen is different than this, click the “Novice” button in the lower left corner. Now select the “Part” template, and click OK, this is where we tell SolidWorks that we want to create a Part file. Additional Part templates can be created, with different options and settings, including different units, dimensioning standards, materials, colors, etc. See the Appendix for information on how to make additional **templates** and change the document **units** to inches and/or millimeters. Using the “Advanced” option allows the user to choose from different custom templates when creating new documents.



3. - Now that we have a new Part file, we have to start modeling the part, and the first thing we need to do is to make the extrusion for the base of the Housing. The first feature is usually one that other features can be added to or one that can be used as a starting point for our model. Select the “Extruded Boss/Base” icon from the Command Manager’s Features tab (active by default). SolidWorks will automatically start a new **Sketch**, and we will be asked to select the plane in which we want to start working. Since this is the first feature of the part, we will be shown the three standard planes (Front, Top and Right). Remember the sketch is the 2D environment where we draw the profile before creating an extrusion, in other words, before we make it “3D”.



4. - For the Housing we’ll select the “Top Plane” to create the first sketch. We want to select the Top Plane because we are going to start modeling the part at the base of the Housing and build it up as was shown in the roadmap at the beginning of this chapter. Do not get too worried if you can’t figure out which plane to choose first when starting to model a part; at worst, what you thought would be a Front view may not be the front; this is for the most part irrelevant, as the user is able to choose the views at the time of detailing the part in the 2D drawing for manufacturing. Select the **Top Plane** from the screen using the left mouse button. Notice the plane is highlighted as we move the mouse to it. The view orientation will be automatically rotated to a Top View.

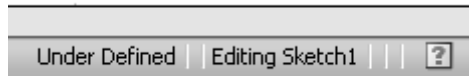


What we have just done is we created a new **Sketch** and are now in the sketch environment. This is where we will create the profiles that will be used to make Extrusions, Cuts, etc. SolidWorks gives us many indications, most of them graphical, to help us know when we are working in the Sketch environment.

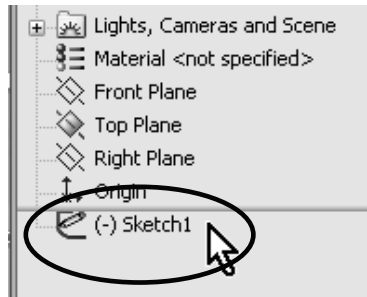
- a) The **Confirmation Corner** is activated in the upper right corner and displays the Sketch icon in transparent colors.



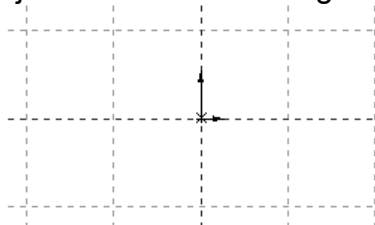
- b) The Status bar at the bottom shows "Editing Sketch" in the lower right corner.



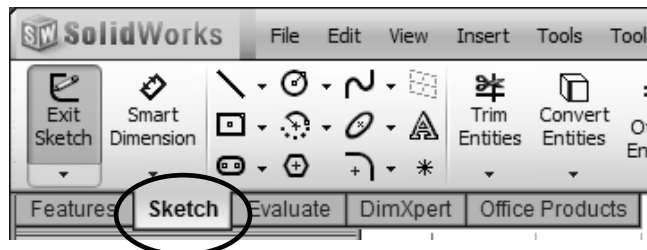
- c) In the Feature Manager "Sketch1" is added at the bottom just under "Origin".



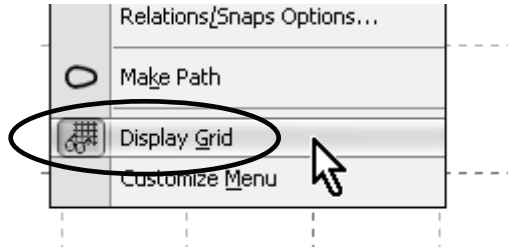
- d) The part's Origin is projected in red and the grid is visible.



- e) The Sketch tab is activated in the Command Manager displaying sketch tools.



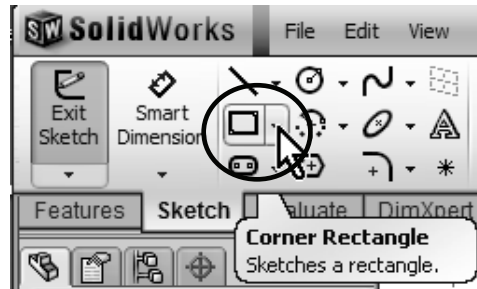
- f) If the option is activated, the Sketch Grid will be displayed. This can be easily turned on or off *while in the Sketch environment*, by Right-Mouse-Clicking in the graphics area and selecting the “Display Grid” option.



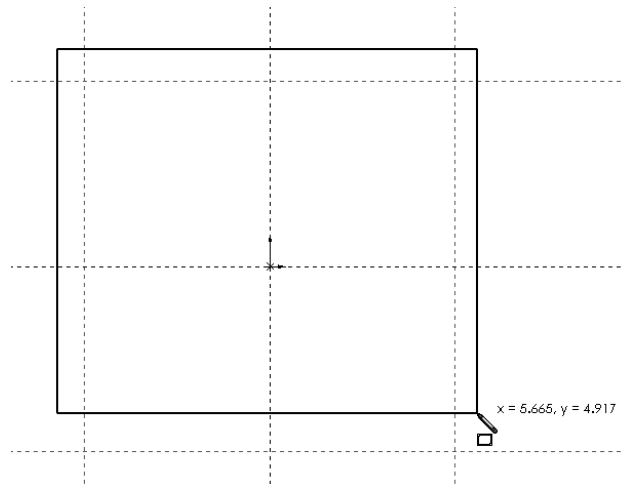
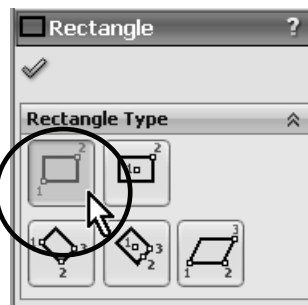
As the reader can see, SolidWorks gives us plenty of clues to help us know when we are working in a sketch.

5. - Notice that when we make the first sketch, SolidWorks rotates the view to match the plane that we selected. In this case, we are looking at the part from above. This is done only in the first sketch to help the user get oriented. In subsequent operations we have to rotate the view manually using the view orientation tools or the Middle Mouse Button to rotate the model.

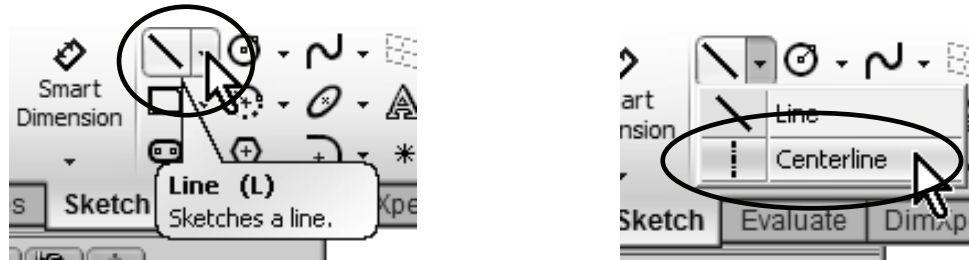
6. - The first thing we need to do is draw a rectangle and center it about the origin. Select the “**Rectangle**” tool from the “Sketch” tab or by making a Right Mouse Button click in the graphics area, from the pop up menu. Make sure we have the “Corner Rectangle” option selected in the Rectangle’s Property Manager.



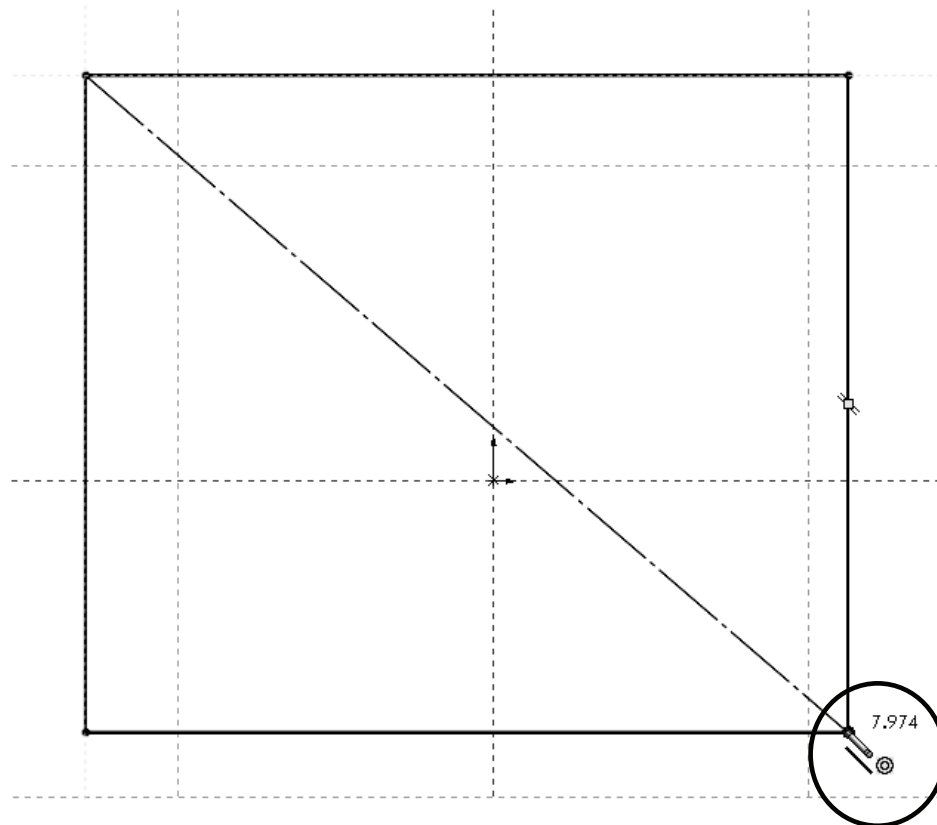
Click and drag in the graphics area to draw a Rectangle around the origin as shown. Left-Mouse-Click and drag to the opposite corner. Don't worry too much about the size; we'll dimension it in a later step.



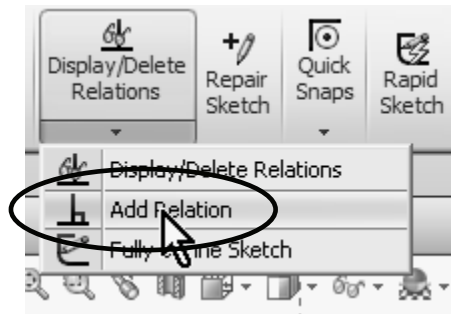
7. - Notice the lines are in green after finishing the rectangle. The color green means the lines are selected. You can unselect them by hitting the Escape (Esc) key, this will also de-select (turn off) the rectangle tool. We only need one rectangle in this sketch, so hit the "Esc" key. Now we will draw a "Centerline" from one corner of the rectangle to the opposite corner. The purpose of this line is to help us center the rectangle about the origin. We'll also learn a faster way to do this in the next few steps. From the "Sketch" tab select the Line command's drop down arrow, and select "Centerline" (Menu "Tools, Sketch Entities, Centerline").



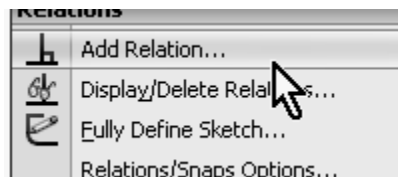
8. - SolidWorks indicates that we will start or finish a line at an existing entity with yellow icons; when we locate the cursor near an endpoint, line, edge, origin, etc. it will "snap" to it. Click in one corner of the rectangle, click in the opposite corner as shown and press the "Esc" key to complete the Centerline command.



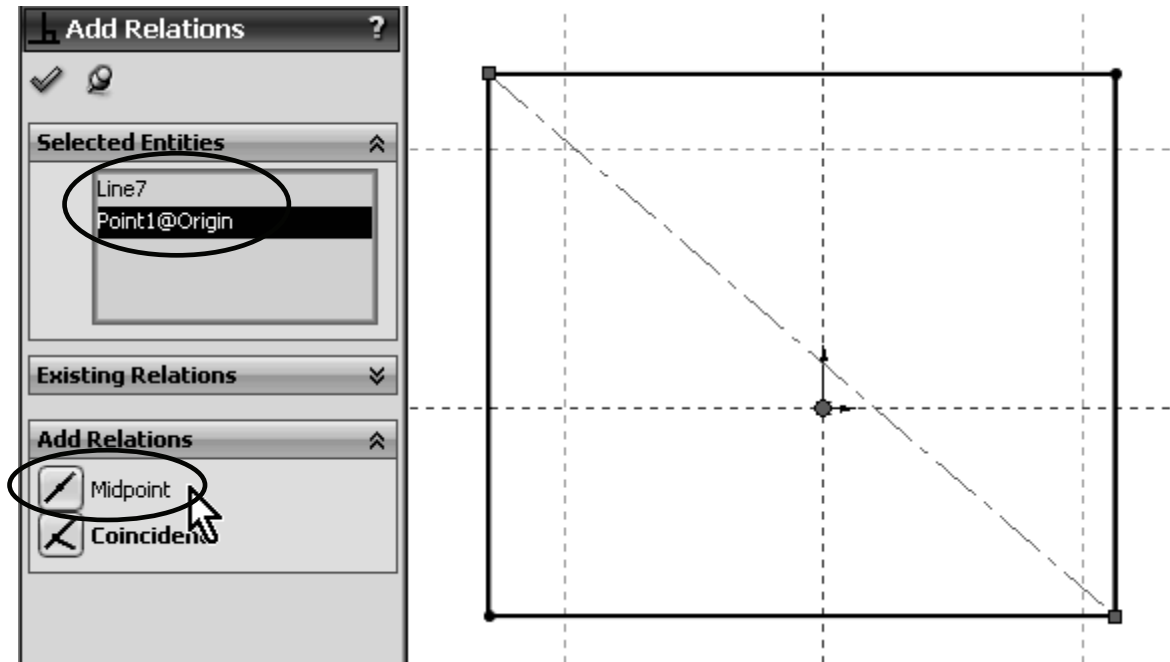
9. - Next we want to make the midpoint of the centerline coincident to the origin; for this step we will add a **“Midpoint”** geometric relation between the centerline we just drew and the part's origin. Select from the menu **“Tools, Relations, Add”** or the **“Add Relation”** icon from the **“Display/Delete Relations”** drop-down icon. By adding this relation, the centerline's mid point will be forced to coincide with the origin; this way the rectangle (and the part) will be centered about the origin. Centering the part about the origin and the model planes will be useful in future operations.



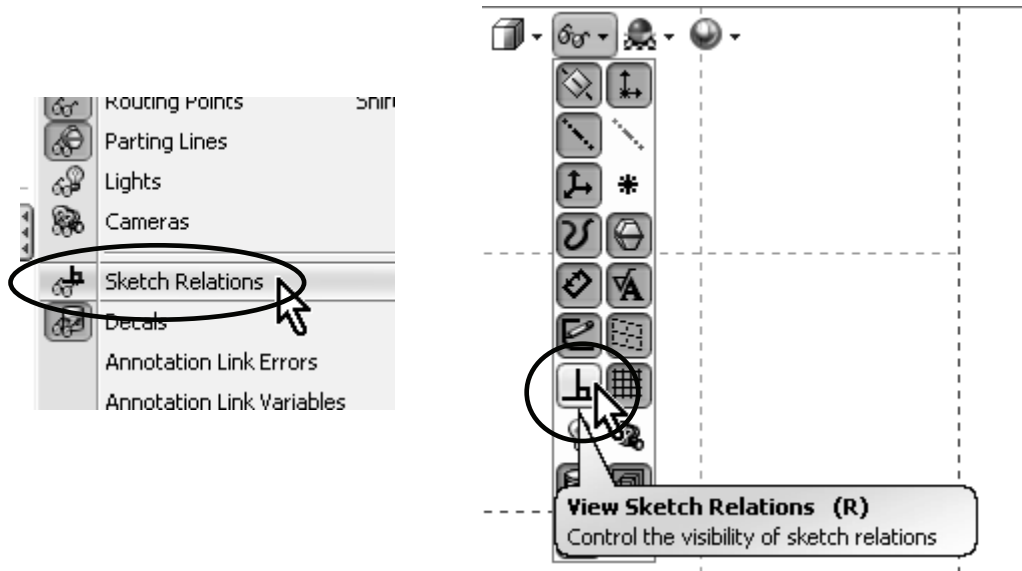
“Add Relation” can also be accessed through the right mouse button menu, or be configured to be available in the **Mouse Gestures** shortcuts.



10. - The “Add Relations” Property Manager is displayed. The **Property Manager** is the area where we will make our selections and choice of options for most commands. Select the previously made centerline and the part's origin by clicking on them in the graphics area (notice how they change to green and get listed under the “Selected Entities” box). Click on **“Midpoint”** under the “Add Relations” box to add the relation. Now the center of the line coincides with the Origin. Click on OK (the green checkmark) to finish the command. Click and drag one corner of the rectangle to see the effect of the relation. Notice the center of the line stays in the origin and the rectangle resizes symmetrically about the center.



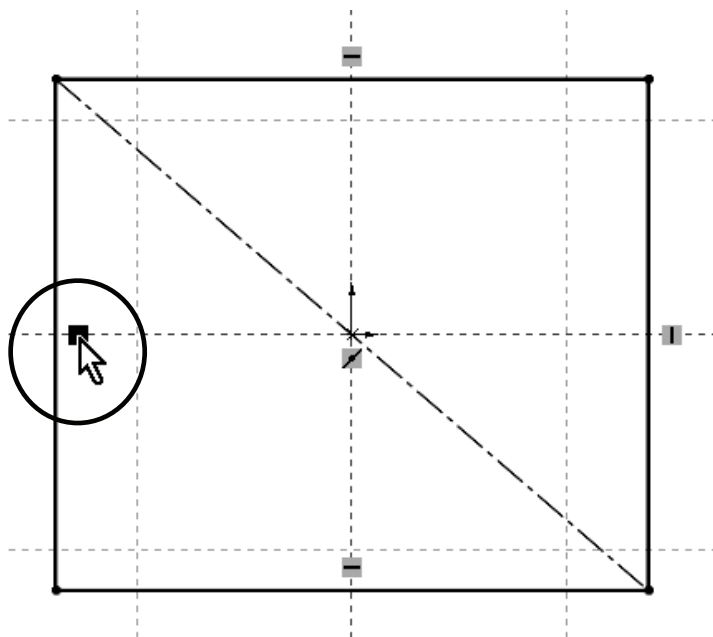
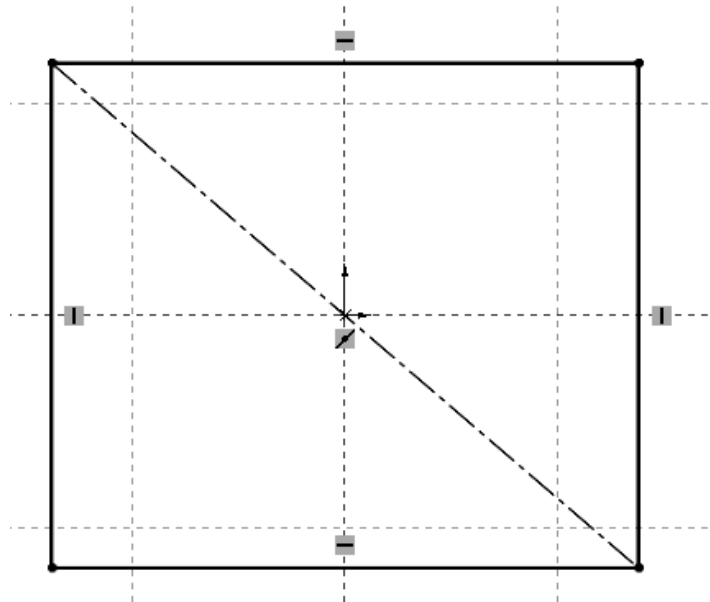
11. - We added a geometric relation manually, and we also added geometric relations automatically when we drew the rectangle and the centerline in the previous step. SolidWorks allows us to view the existing relations graphically between sketch elements. Go to the menu, **“View, Sketch Relations”** if not already activated, or from the “Hide/Show Items” drop down icon in the graphics area.







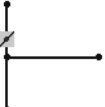



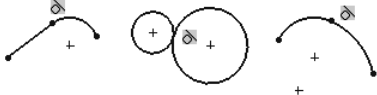
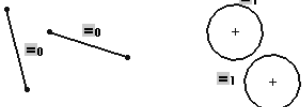

12. - Now we can see the geometric relations graphically represented by small blue icons next to each sketch element. Notice that when we move the mouse pointer over a geometric relation icon, the entity or entities that share the relation are highlighted.



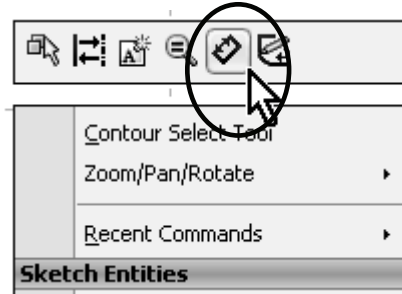
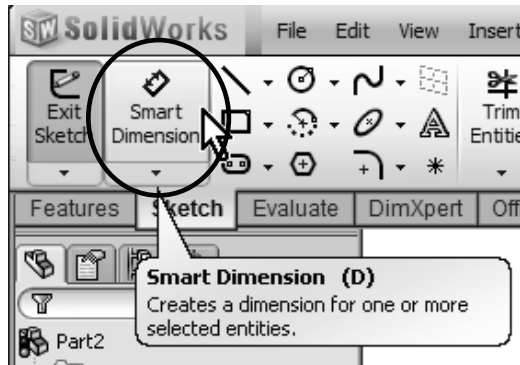
To delete a geometric relation select the relation icon in the screen and press the “Delete” key, or Right Mouse Click on the Geometric Relation icon and select “Delete”. (Do not delete any relations at this time!)



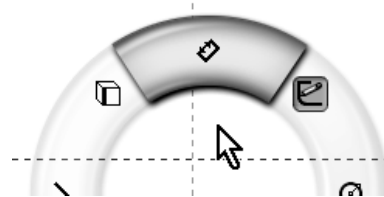
The **Sketch Origin**  defines the local Horizontal (short red arrow) and local Vertical (long red arrow) directions for the sketch, this is important because we may be looking at the part in a different orientation, and vertical may not necessarily mean “up” on the screen; since we are working in three dimensions, and we can view the part from any view orientation; it is a convenient way for us to know where “up” is in the sketch. In SolidWorks we have the following basic types of geometric relations between sketch entities:

	<p>Vertical Parallel to the sketch vertical direction (long red arrow in the origin)</p>
	<p>Horizontal Parallel to the sketch horizontal direction (short red arrow in the sketch origin)</p>
	<p>Coincident is when an endpoint touches another line, endpoint, model edge or circle.</p>
	<p>Midpoint is when a line endpoint coincides with the middle of another line or model edge. A Midpoint relation implies it is also Coincident.</p>
	<p>Parallel is when two or more lines or a line and a model edge have the same inclination.</p>
	<p>Perpendicular is when two lines are 90 degrees from each other, like vertical and horizontal lines. Note that the lines don't have to touch each other in order to be perpendicular.</p>
	<p>Concentric is when two arcs or circles share the same center. Concentric can be also between a point or line's endpoint and an arc or circle's center.</p>
	<p>Tangent is when a line and an arc or circle, or two arcs or circles are tangent to each other.</p>
	<p>Equal is when two or more lines are the same length, or two or more arcs or circles have the same diameter.</p>
	<p>Collinear is when two or more lines lie on the same line.</p>

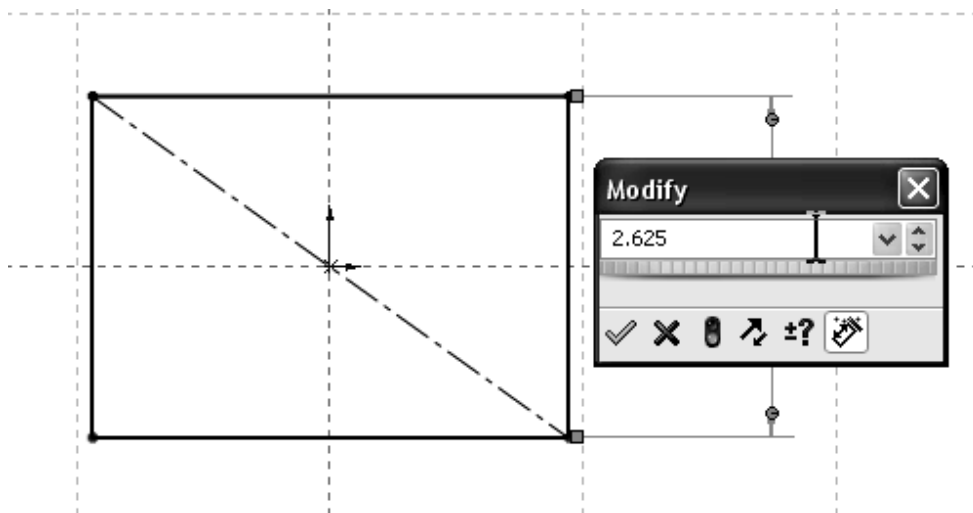
13. - Once we have added the geometric relation “Midpoint”, the next step is to dimension the rectangle. Turn off the geometric relations display in the menu “**View, Sketch Relations**” to avoid visual clutter in the screen if so desired. Click with the Right Mouse Button in the graphics area and select “**Smart Dimension**” or select the “Smart Dimension” icon from the Sketch Toolbar. Notice the cursor changes adding a small dimension icon next to it. This icon will indicate the user the Smart Dimension tool is selected.



Smart Dimension can also be activated from the Mouse Gestures (Right-Mouse-Click and drag up by default).

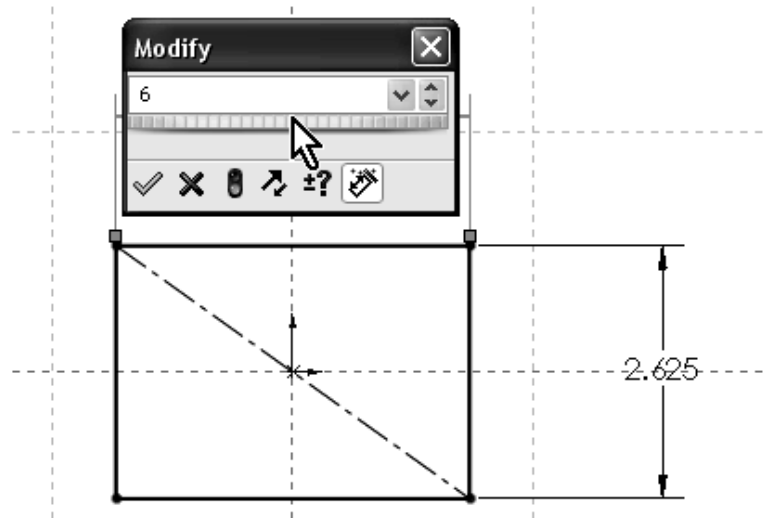


14. - Adding dimensions in SolidWorks is simple and straight forward. Click to select the right vertical line and then click just to the right to locate the dimension. SolidWorks will show the “Modify” dialog box, where we can add the 2.625” dimension. Repeat with the top horizontal line and add a 6” dimension. As soon as the dimension value is accepted, the geometry updates to reflect the correct size.






View the Appendix if you need to change the document's **units** from millimeters to inches or vice versa. You can also override the default units for one dimension by adding "in" or "mm" at the end of the value in the Modify dialog box.



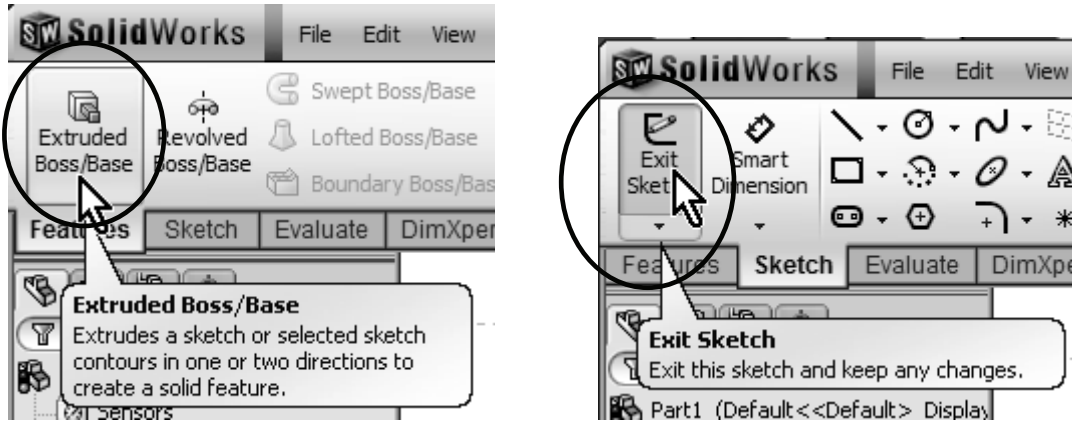
To change a dimension's value double click on it to show the "Modify" box.

After dimensioning the lines, notice the lines changed from Blue to Black. This is the way SolidWorks lets us know that the geometry is defined, meaning that we have added enough information (dimensions and/or geometric relations) to define the geometry in the sketch. The status bar also shows "Fully Defined" in the lower right corner. This is the preferred state before creating a feature, since there is no information missing and the geometry can be accurately described.

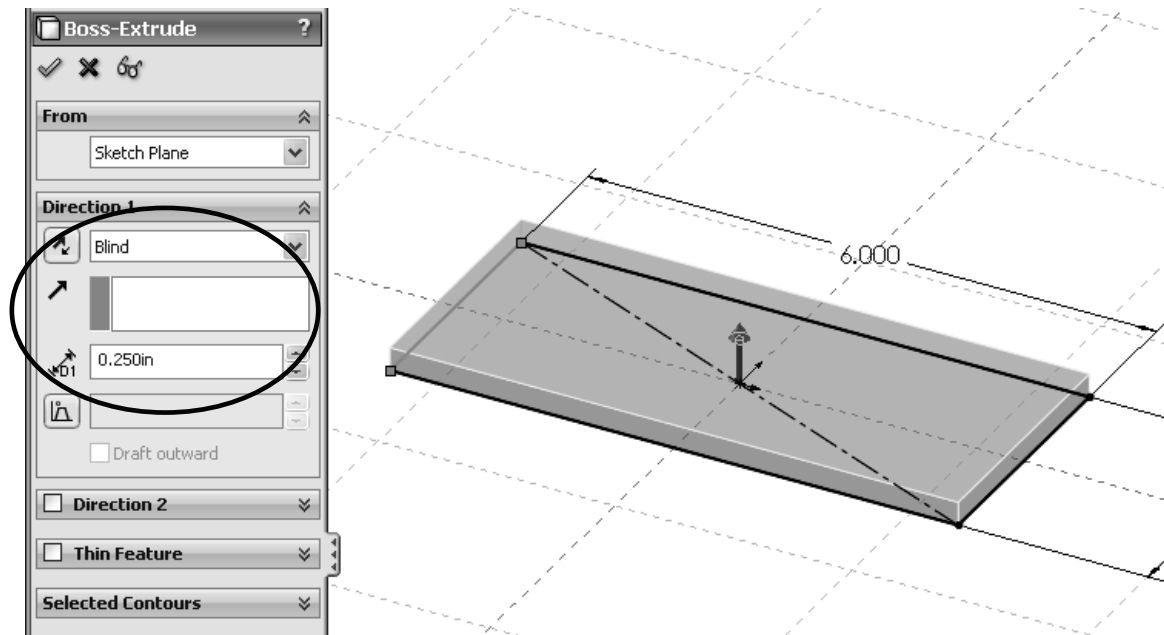
A sketch can be in one of several states; the three main ones are:

- **Under Defined:** (BLUE) Not enough dimensions and/or geometric relations have been provided to define the sketch. Sketch geometry is blue and lines/endpoints can be dragged with the left mouse button.
- **Fully Defined:** (BLACK) The Sketch has all the necessary dimensions and/or geometric relations to completely define it. This is the desired state. Fully defined geometry is black.
- **Over Defined:** (RED) Redundant and/or conflicting dimensions and/or geometric relations have been added to the sketch. If an over-defining dimension or relation is added, SolidWorks will immediately warn the user. If an over-defining geometric relation is added, delete it or use the menu, "**Edit, Undo**" or select the "Undo" icon . If an over-defined dimension is added, the user will be offered an option to cancel it.

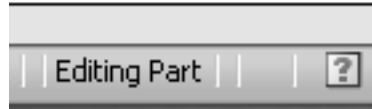
15. - Now that the sketch is fully defined, we will create the first feature of the housing; this is when we go from the 2D Sketch to a 3D feature. Click in the Features tab in the Command Manager and select the “**Extrude**” icon, or click in the “Exit Sketch” icon in the Sketch toolbar. In the second case SolidWorks remembers that we wanted to make an Extrusion in the first place, and displays the Extrude command’s property manager. Notice that the first time we create a feature in a new part, SolidWorks changes the view orientation to an Isometric view and gives us a preview of what the feature will look like when finished.



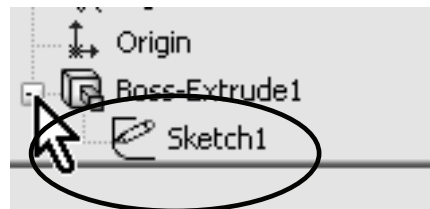
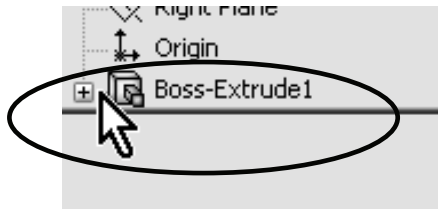
Select the options indicated in the “**Extrude**” command to make the extrusion 0.25” thick. To finish the command, select the OK button or press the “Enter” key.



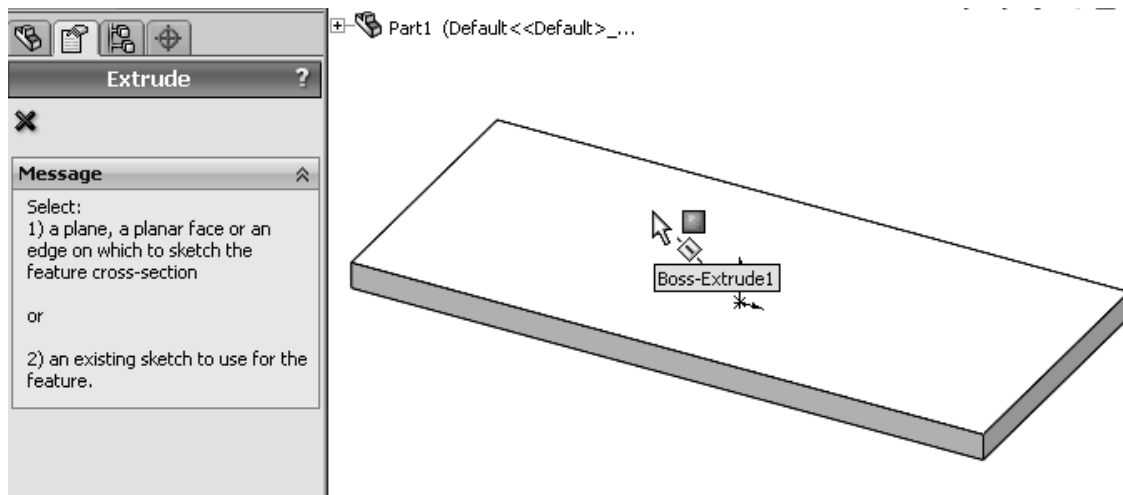
16. - After the first extrusion is completed, notice that “Extrude1” has been added to the Feature Manager. The confirmation corner is no longer active. The status bar now reads “Editing Part” to alert us that we are now editing the part and not a sketch.



Expanding the “Boss-Extrude1” feature in the Feature Manager by clicking on the “+” on the left side of it, we see that “Sketch1” has been absorbed by the “Boss-Extrude1” feature.

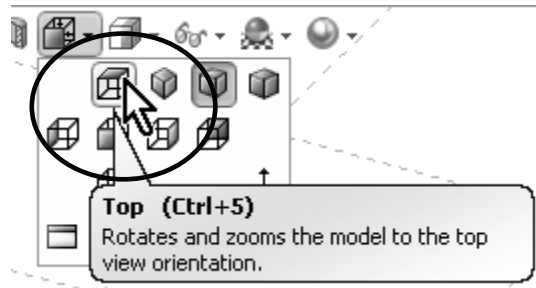


17. - The second feature will be similar to the first one but with different dimensions. To create the second extrusion, we need to make a new sketch. When we select the Extrude or Sketch icon, SolidWorks gives us a yellow message in the Property Manager asking us to select a Plane or a planar (flat) face. We'll select the top face of the previous extrusion for the next feature (If a Plane or flat face is pre-selected, the Sketch opens immediately in that Plane/face).

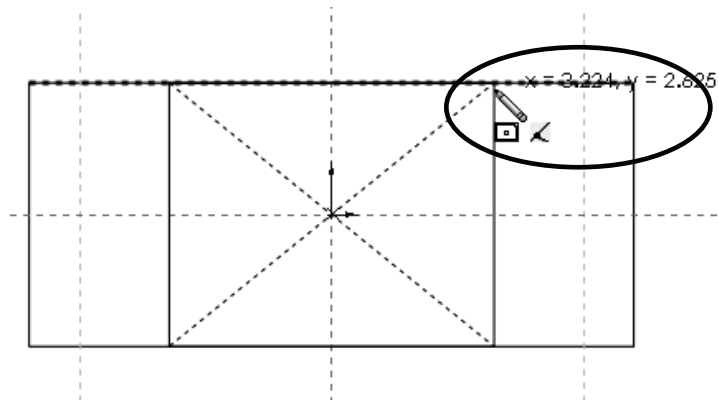
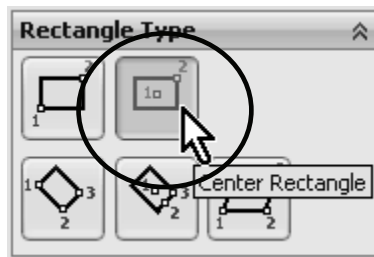
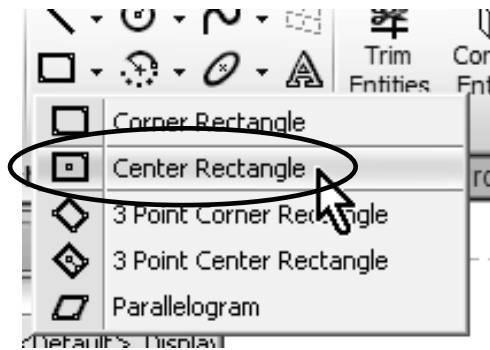


18. - Only in the first sketch of the part, the view is oriented to the sketch plane. To help us get oriented, we will switch to a **Top View** to see the part from the top. In SolidWorks the user is free to work in any orientation, as long as he/she is able to see what they are doing. Re-orienting the view helps new users get used

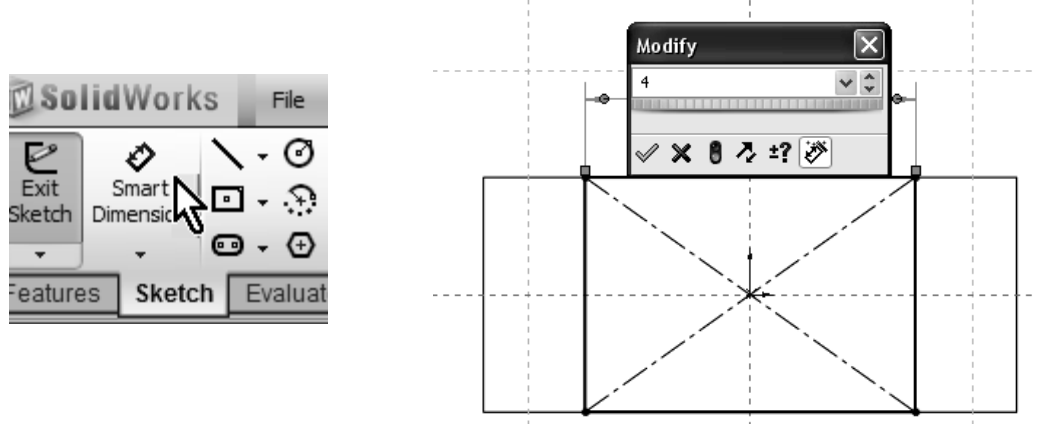
to 3D in a more familiar way by looking at it in 2D. Use the “View Orientation” icon, or the default shortcut “Ctrl+5”, as indicated in the tooltip.



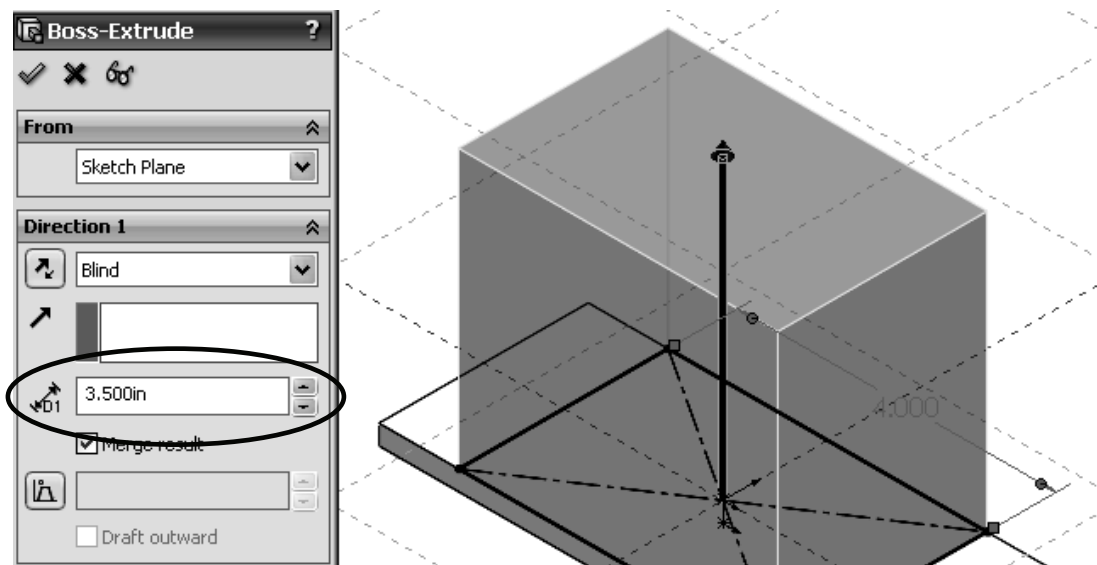
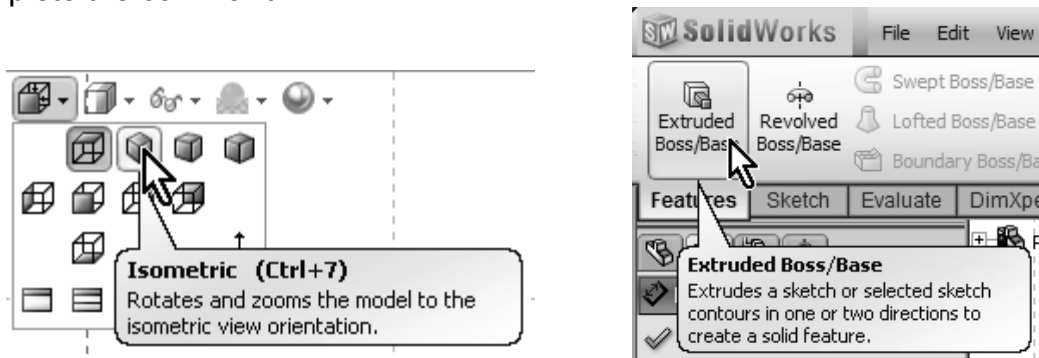
19. - For the second extrusion, we'll use the “Center Rectangle” command. Click in the Rectangle's tool drop down menu in the Sketch tab, and select “Center Rectangle”; if you selected the rectangle as before, you can change the rectangle type to “Center Rectangle” from the Rectangle's Property Manager. To make the rectangle, click in the Origin first, then on the top edge of the first extrusion (or the bottom, it really doesn't matter) to finish it. Notice the yellow Coincident icon as the pointer is in the origin and then on the model edge. By doing it this way, we automatically add coincident relations to the origin and the top edge. The “Center Rectangle” command saves us from adding the centerlines and midpoint relations making the rectangle centered about the origin in a single operation.




20. - Select the “Smart Dimension” tool (toolbar, right mouse menu or mouse gesture), and dimension the rectangle 4” wide by selecting the top line as shown. Adding this dimension will fully define the sketch.



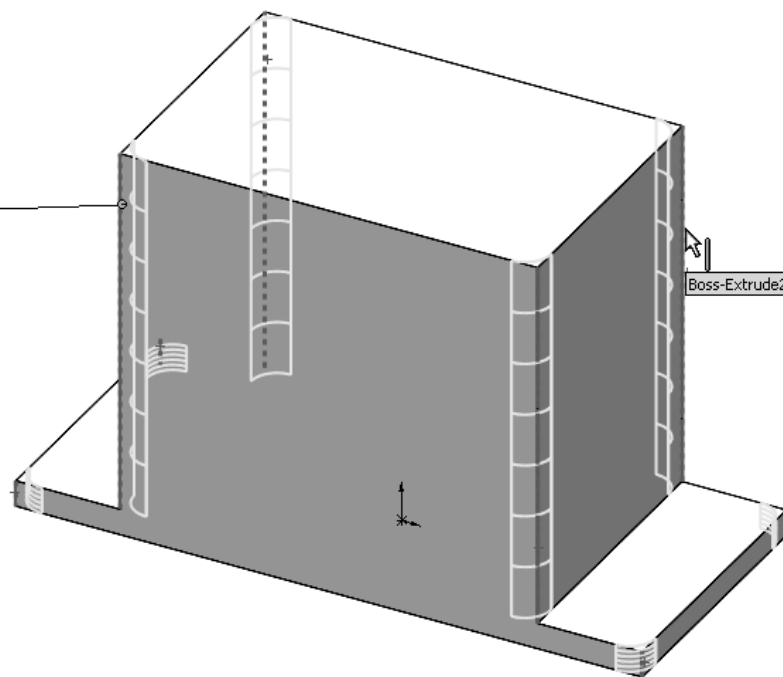
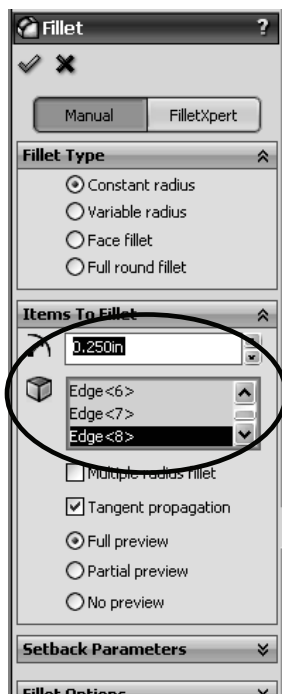
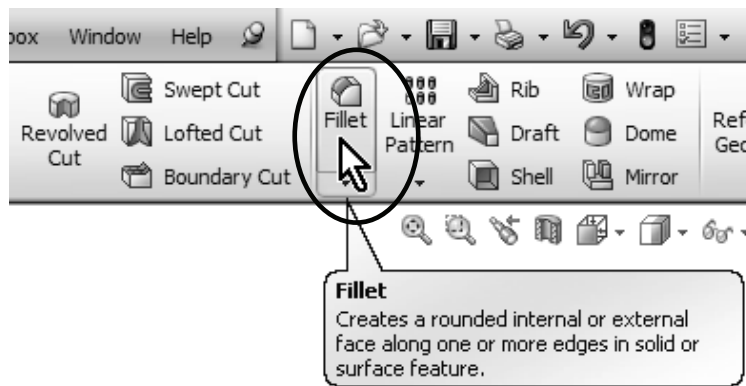
21. - We are now ready to make the second extruded feature. Select the “Extrude” command in the “Features” tab of the Command Manager (or “Exit Sketch” if you initially selected “Extrude”) and make the extrusion 3.5”. From the Standard Views icon, select the **Isometric** view (or the “Ctrl+7” shortcut or Mouse Gesture) to see the preview of the second extrusion. Click on OK to complete the command.



22. - The next step is to round the edges of the two extrusions. To do this, we will select the **Fillet** command. The Fillet is what's called an applied feature; we don't need a sketch to create it, and it's applied directly to the solid model. Select the "Fillet" icon from the Features Tab of the Command Manager. By default, "Constant Radius" type is selected. We'll change the radius to 0.25" and select the corners indicated in the preview. SolidWorks highlights the model edges when we place the cursor on top of them to let us know that we'll select them if we click on them. If an edge to be selected is not visible, rotate the model using the menu **"View, Modify, Rotate"**.  Rotate Click and drag in the graphics area to rotate. Another way to do this is by holding down the middle mouse button (scroll wheel), and dragging in the graphics area or using the arrow keys. Click OK when all eight edges are selected to complete the command.

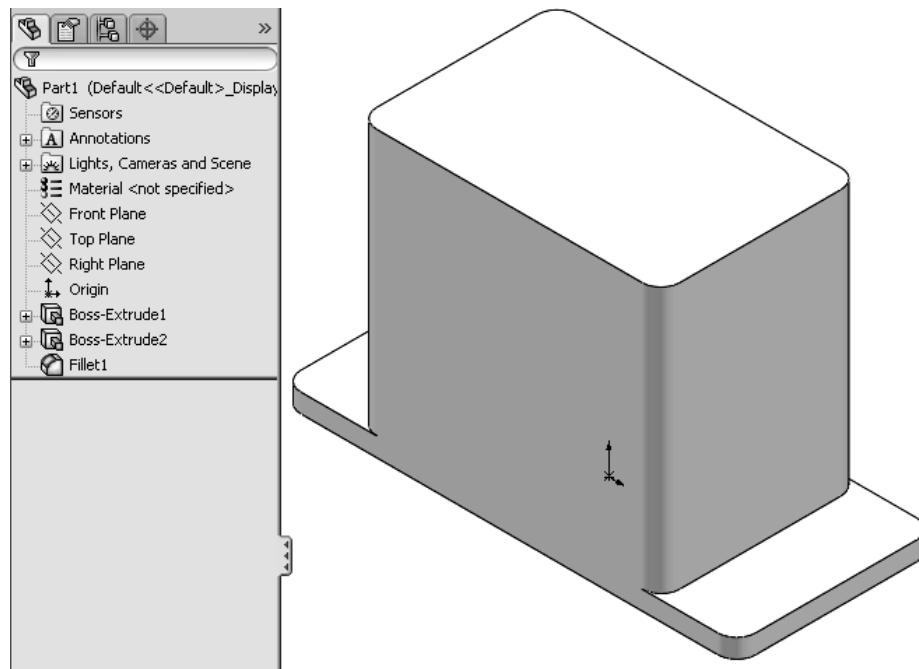
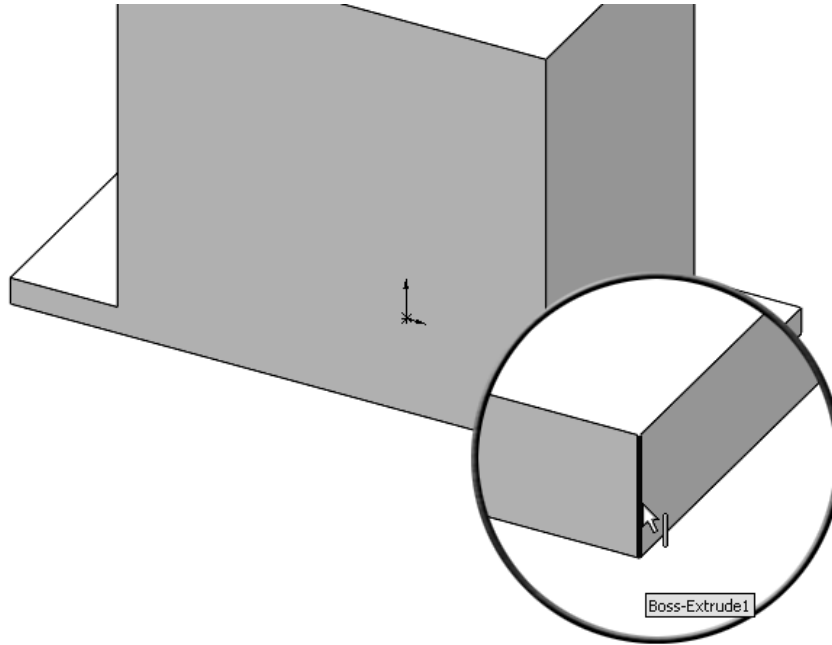


If an edge or face is mistakenly selected, simply click on it again to de-select it.





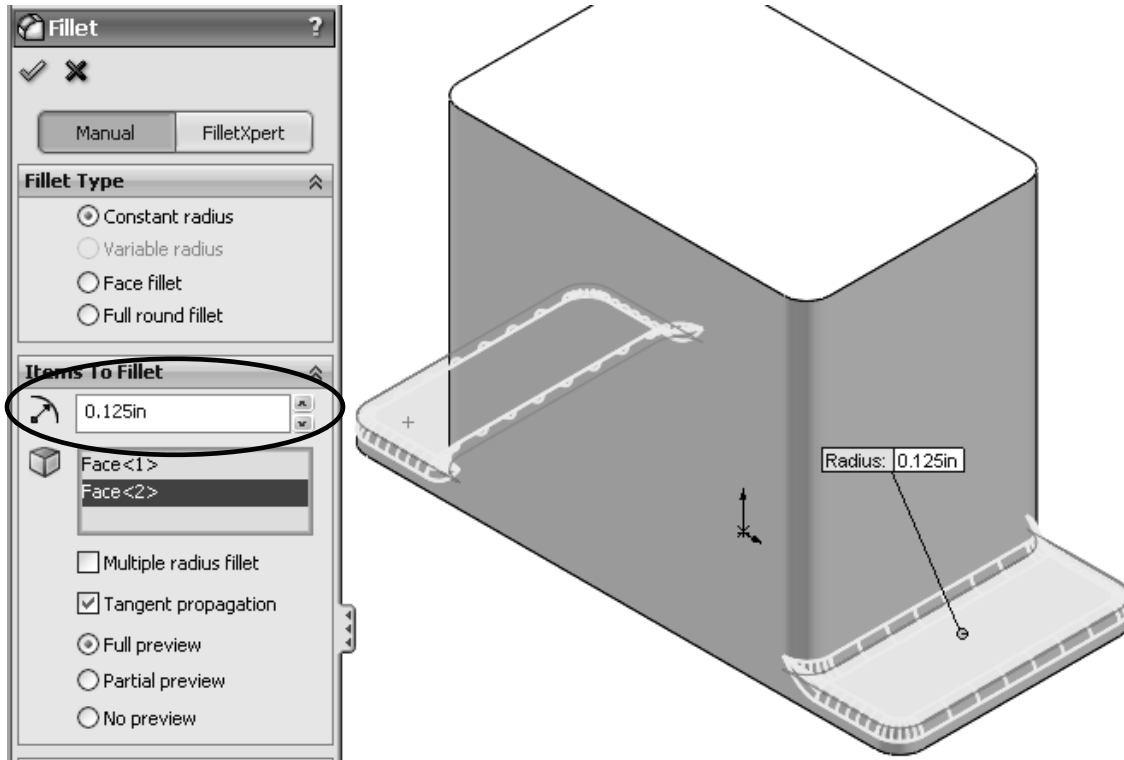
A handy feature is a “**Magnifying Glass**” to selectively zoom in on one area of the model. To activate it, use the default shortcut “G” in the keyboard. To make multiple selections with it, hold down the “Ctrl” key, otherwise, the Magnifying Glass will turn off after making one selection or after pressing “G” again. Scrolling with the mouse wheel will zoom inside the Magnifying Glass for more or less magnification.



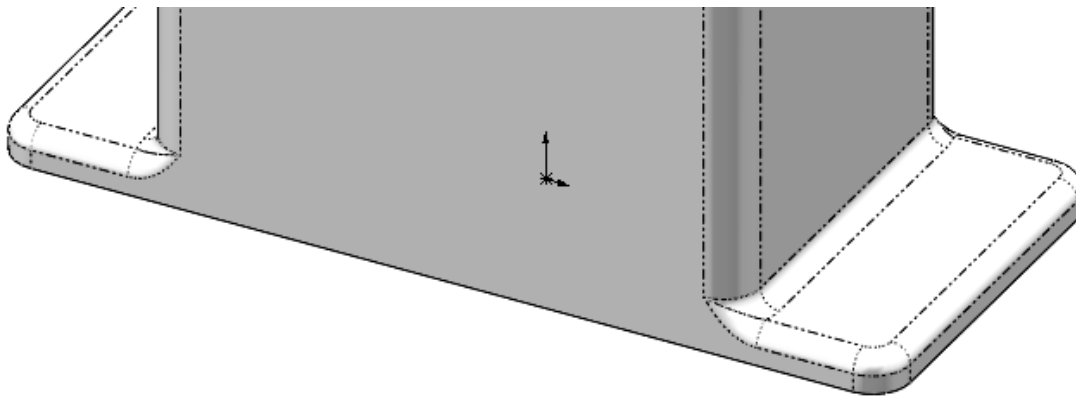
23. - Repeat the fillet command to add a 0.125” radius fillet at the base of the Housing.



By selecting a face SolidWorks rounds all the edges of that face.



Notice how the new fillets blend nicely with the existing fillets.



We can change the appearance of tangent edges (The edges where two tangent faces meet) by selecting the menu, “**View, Display**” and selecting the display option desired: **Visible**, as **Phantom** or **Removed**. Explore the different options to find the one you feel more comfortable with. In this book Phantom lines are used for clarity.

24. - We will now remove material from the model using the “**Extruded Cut**” command. Switch to a **Top View** using the View Orientation Toolbar, Mouse Gesture or shortcut Ctrl+5.



Select “**Extruded Cut**” from the Features tab; you will be asked to select a face or planar face just as with the “**Extruded Boss**”. Select the top most face to create a new Sketch, and using the “**Corner Rectangle**” tool from the now visible

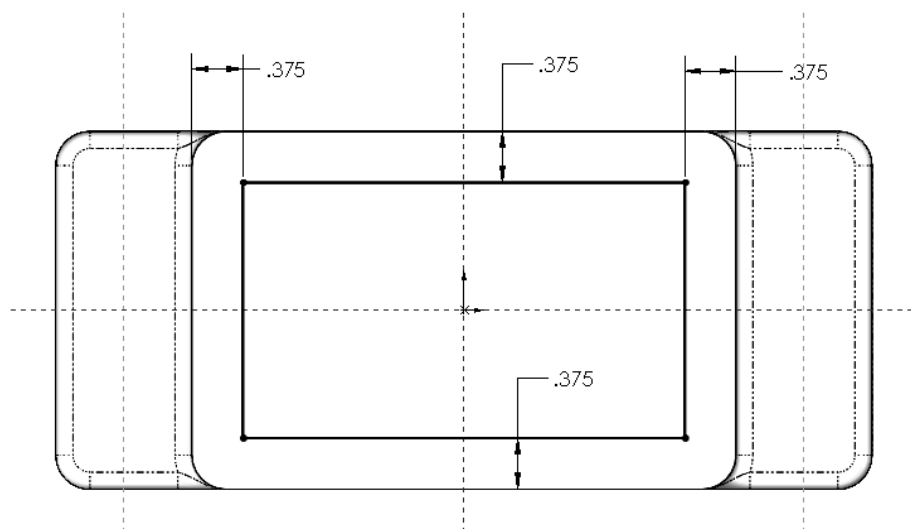
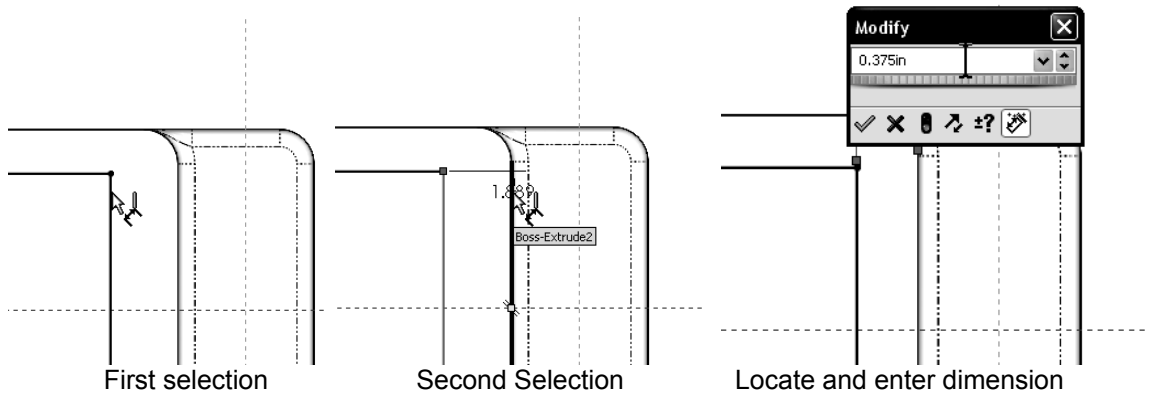


“Sketch” toolbar, draw a rectangle *inside* the top face.

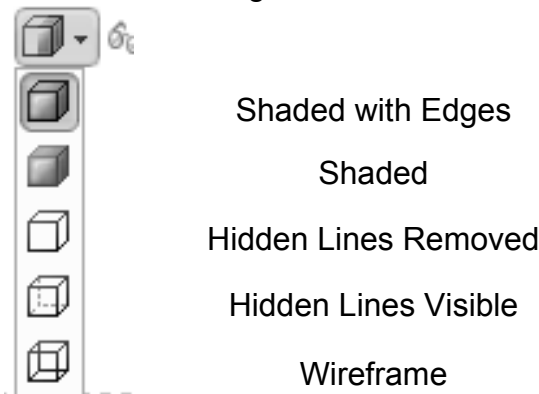


Extruded Cut
Cuts a solid model by extruding a sketched profile in one or two directions.

To add the dimensions select the “**Smart Dimension**” tool; we can add dimensions from sketch geometry to model edges simply by selecting them. Select a Sketch line, click on a model edge parallel to it, and finally click to locate the dimension in the screen. You will be asked to enter the dimension as before. Repeat to add the rest of the dimensions.



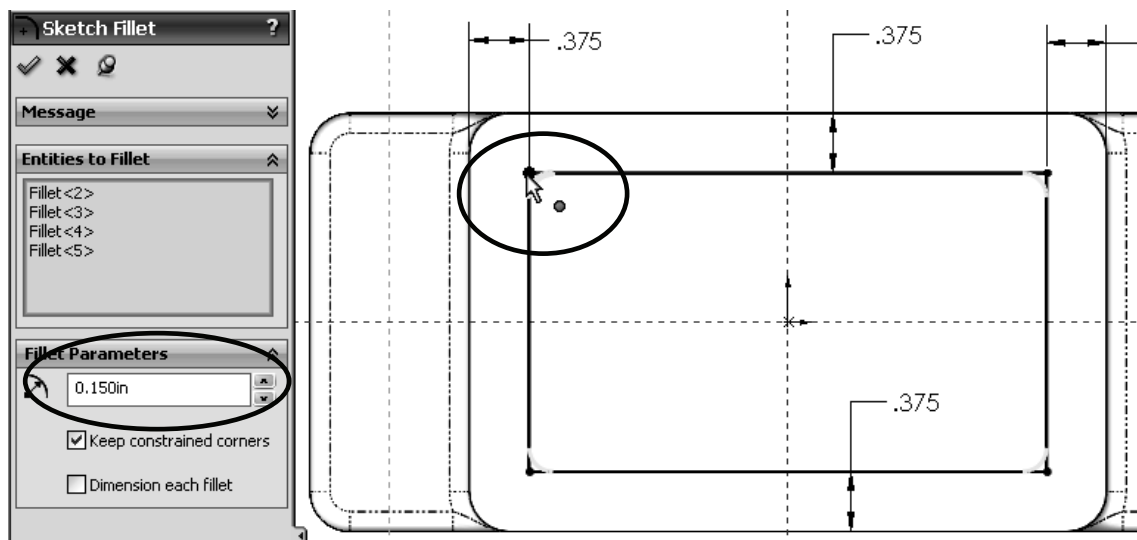
25. - If needed, switch to a “Hidden Lines Removed” mode from the **View Style** icon to view the model without shading to facilitate visualization.



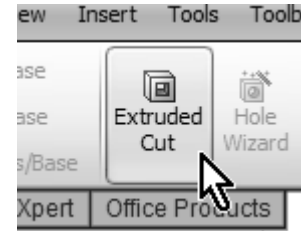
26. - In this feature, we will round the corners in the sketch using a **Sketch Fillet**. We can add the fillets as applied features like before, but in this step we chose to show you how to round the corners in the Sketch *before* making the “Extruded Cut” feature. Select the “**Sketch Fillet**” icon from the Sketch Toolbar.



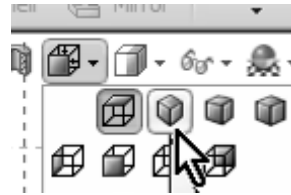
Set the fillet radius to 0.150”, and click on the corners of the sketch lines as indicated to round them. Notice the preview in the screen. After clicking on all 4 corners, click OK to finish the Sketch Fillet command. Notice that only one dimension is added. The reason is that SolidWorks adds an equal relation from each fillet to the dimensioned one.



27. - Now we select the **Extruded Cut** icon from the “Features” tab in the Command Manager to remove the material. Opposite to the Boss Extrude feature that adds material, the Cut feature, as its name implies, removes material from the model.



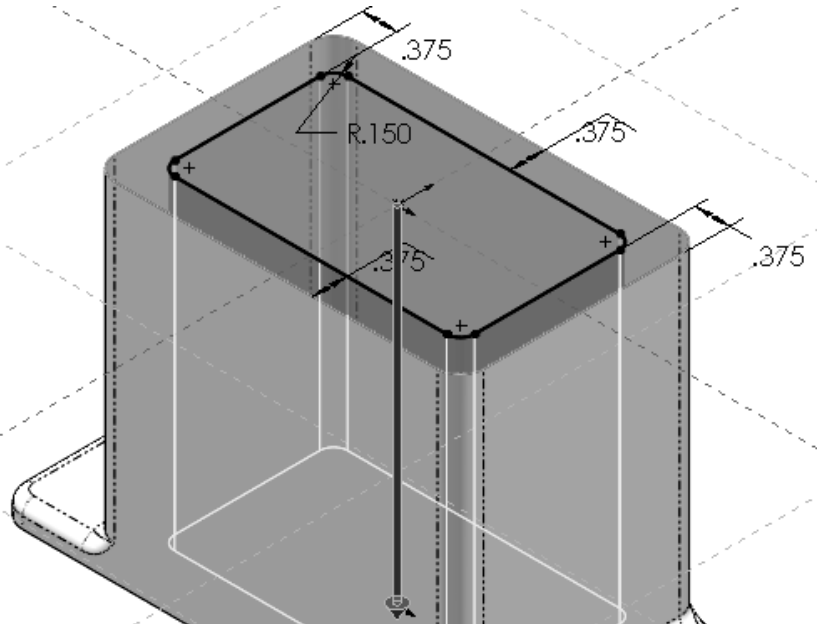
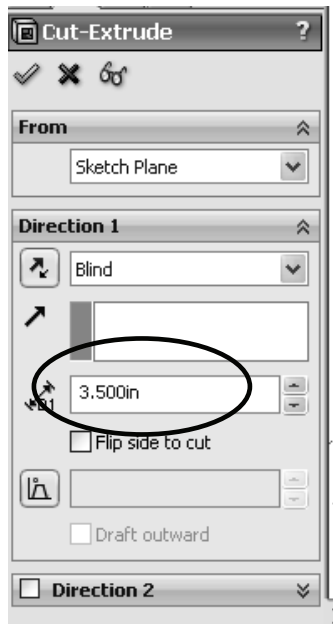
Change the view to Isometric....



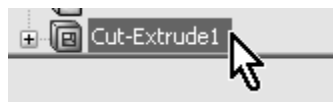
...and **Shaded with Edges** for better visualization.



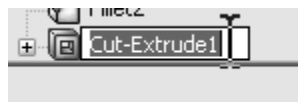
Make the cut 3.5” deep and click on OK to finish the cut.



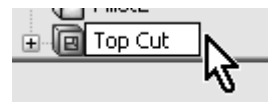
Features can be **Renamed** in the Feature Manager for easier identification. To rename a feature slowly double-clicking the feature, or select and press F2 and type a new name (just like renaming files in Windows Explorer!).



Select Feature,



Click again...(or F2)



Type a new name

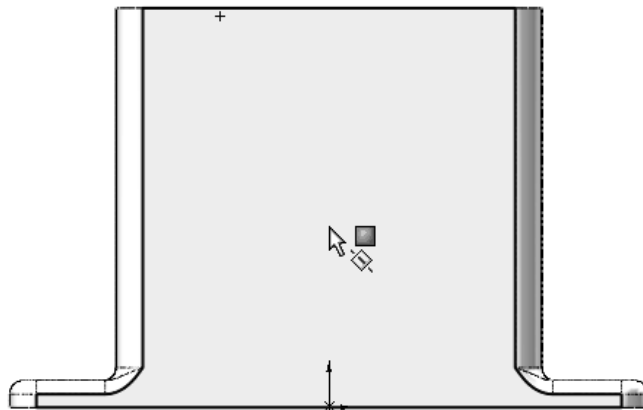
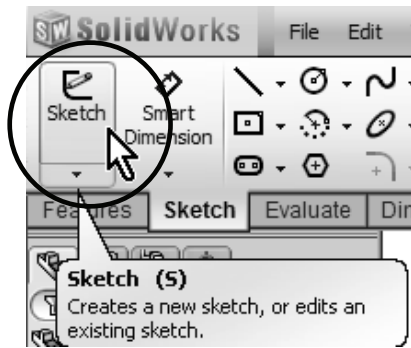


Finish

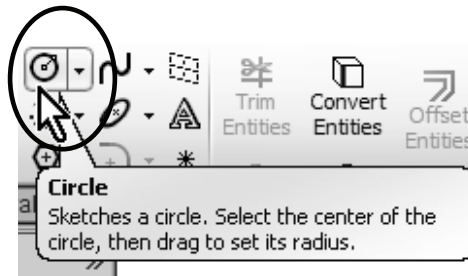
28. - In the next step we will add a simple round boss to the front of the Housing. Switch to a **Front View** using the “View Orientation” toolbar, Mouse Gestures or shortcut Ctrl+1.



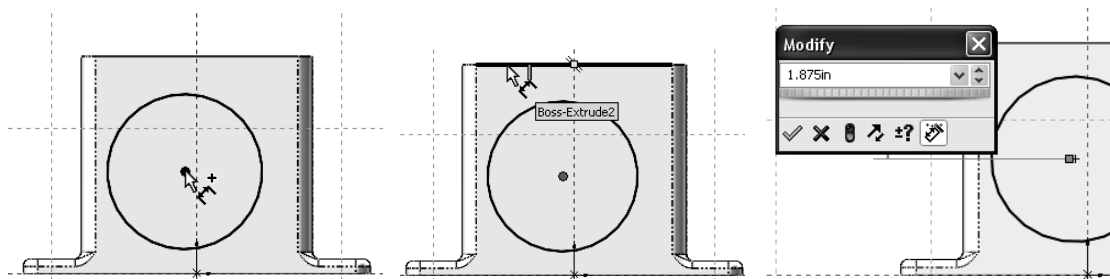
Select the “Sketch” icon from the “Sketch” tab in the Command Manager and click in the front face, or, the reverse order: select the face first, and then click in the “Sketch” icon. Notice that SolidWorks highlights the entire flat face before selecting it to add a sketch.

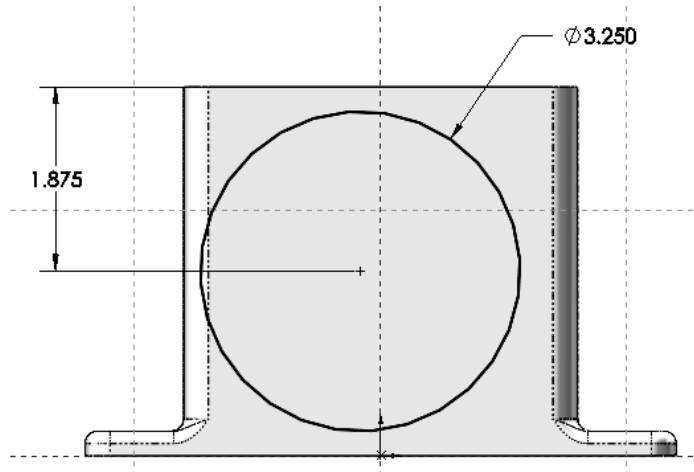


29. - Once we have the sketch created, from the “Sketch” tab of the Command Manager select the “Circle” tool. Draw a circle approximately as shown; click near the middle of the part to locate the center of the circle, and click again to set its size (You can also click and drag from the center to draw the circle).

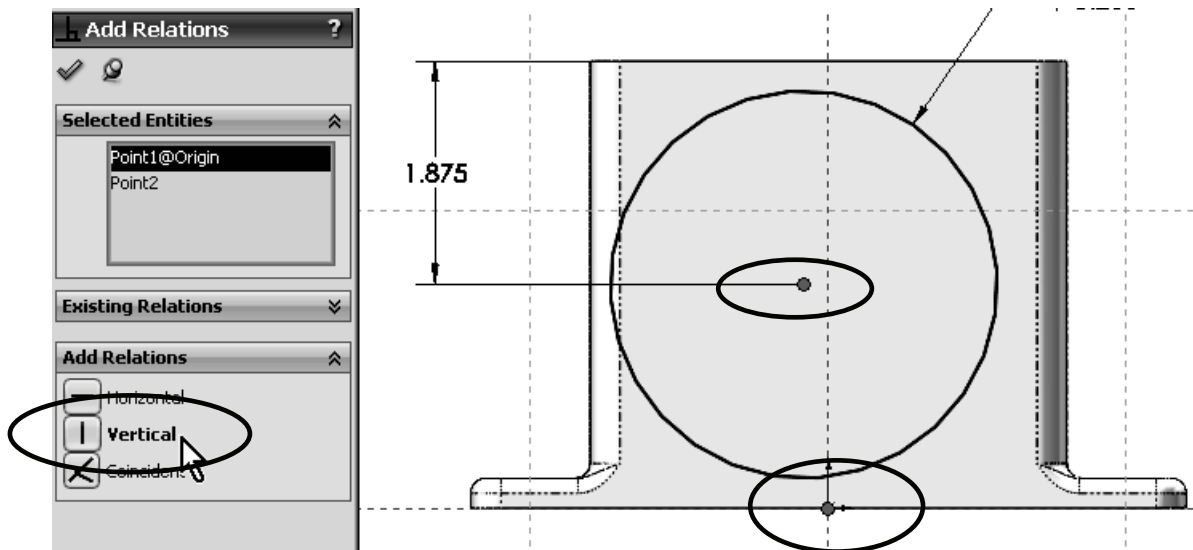
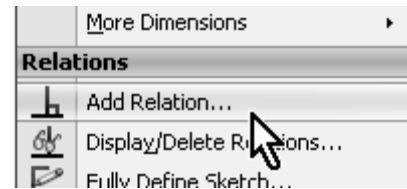


To dimension the circle's location, select the “**Smart Dimension**” tool, and click on either the center of the circle or its perimeter, then the top edge of the housing and finally locate the dimension and enter the value of 1.875”. For the Diameter, simply select the circle, and then locate the dimension as shown.



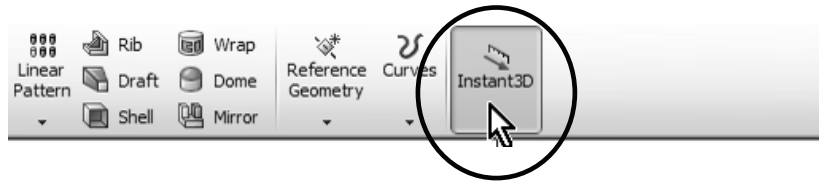


30. - To locate the circle horizontally centered in the part, we will add a **Vertical Relation** between the center of the circle and the part's origin. SolidWorks allows us to align sketch elements to each other or to existing model geometry (edges, faces, vertices, planes, origin, etc.). From the Right Mouse button menu, select “**Add Relation**” (or the menu “**Tools, Relations, Add**”), Select the circle's center (Not the perimeter!) and the origin. Click on “**Vertical**” to add the relation and OK to finish. By adding this relation our sketch is now fully defined.

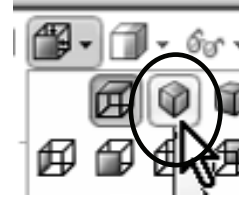


31. - After adding the **Vertical** geometric relation exit the sketch. We'll use a time saving feature called “**Instant 3D**” to make the extrusion. It should be active by default in the “**Features**” toolbar in the Command Manager, otherwise simply click to activate it.

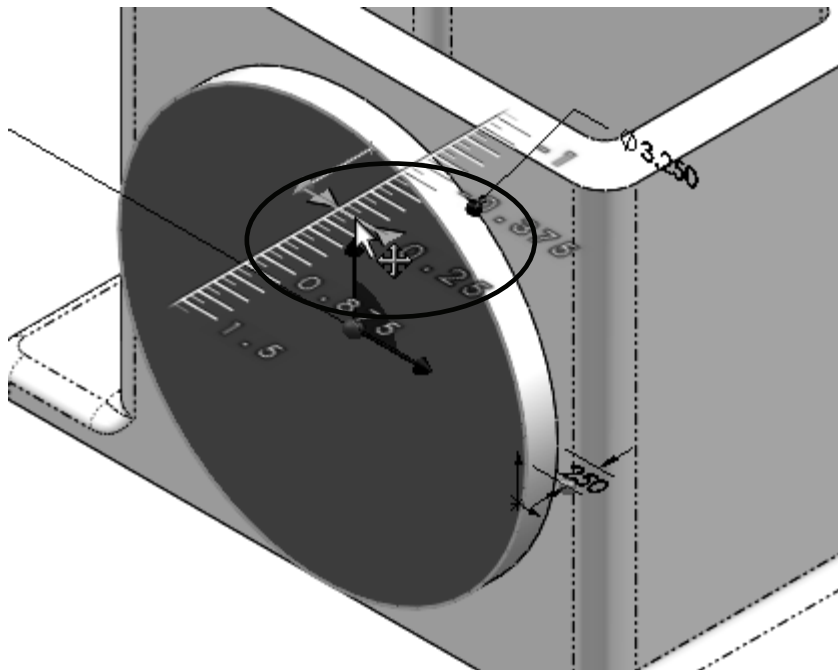




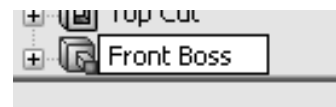
To make the extrusion, switch to an isometric view using the “View Orientation” Toolbar, select the circle of the previous sketch. Take note that now we are editing the part, we left the sketch editing environment. Once the sketch circle is selected, click and drag on the blue arrow, this is the handle to extrude the sketch. You will immediately see a dynamic ruler that will show the size of the extrusion as you drag it. Make sure to extrude it 0.250”. When you release the handle, a new extrusion will be created. To modify this extrusion, simply select a face of this extrusion and drag on the handle to the new size.



You can control the size with more precision by dragging the handle over the ruler’s marks; this way the handle will snap to them.



Rename this extrusion as “Front Boss” by slowly double-clicking the feature’s name in the Feature Manager.



32. - The next step is to create an identical extrusion on the other side of the Housing. To make it we’ll use the “**Mirror**” command. This will make an identical 3D copy of the extrusion we just made. Switch to an Isometric view to help us visualize the Mirror’s preview and make sure we are getting what we want.

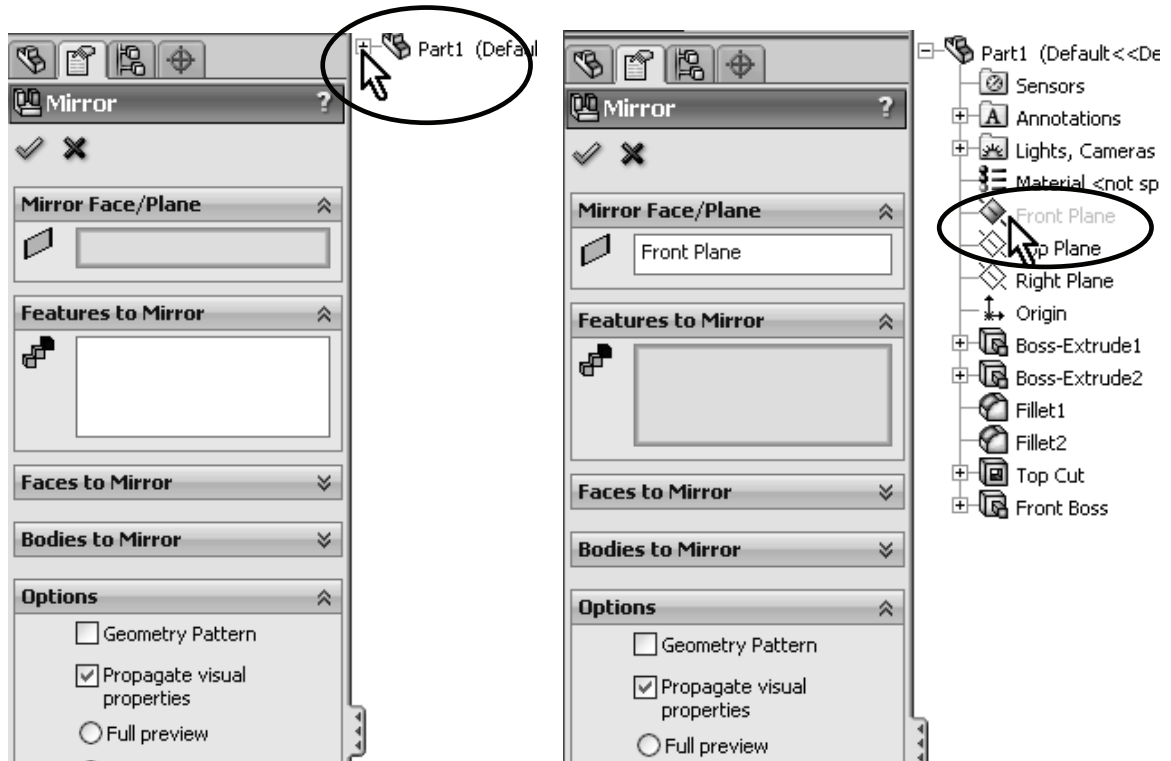
33. - Select the “**Mirror**” icon from the Features tab in the Command Manager.

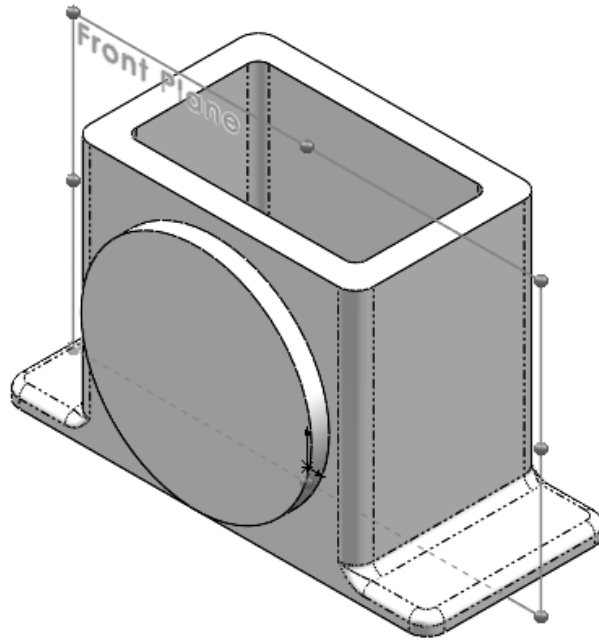


34. - From the Mirror's Property Manager, we have to make two selections. The first one is the Mirror Face or Plane and the second is the feature(s) we want to make a mirror from. The face or plane that will be used to mirror the feature has to be in the middle between the original feature and the desired mirrored copy. Making the first extrusion centered about the origin causes the Front plane to be in the middle of the part, making it the best option for a Mirror Plane.



To select the Front plane (make sure the “Mirror Face/Plane” selection box is highlighted, this means it is the active selection box), click on the “+” sign next to part's name to reveal a **fly-out Feature Manager**, from where we can select the Front Plane.



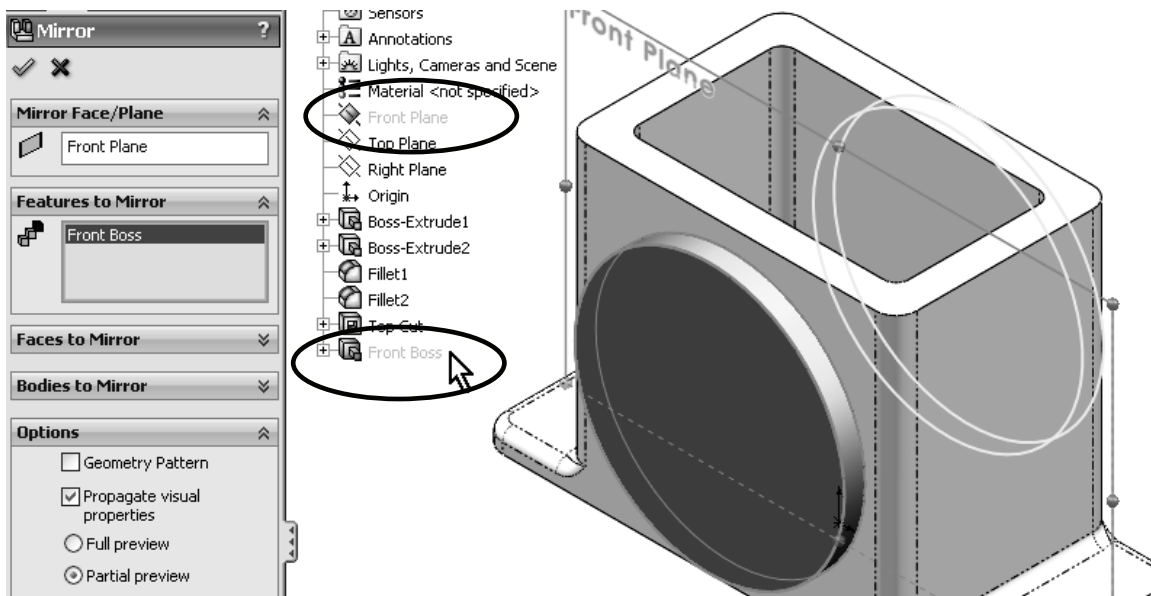


35. - After selecting the Front Plane from the fly-out Feature Manager, SolidWorks automatically activates the “Features to Mirror” selection box (now highlighted) and is ready for us to select the feature(s) we want to mirror. If the “Front Boss” extrusion was pre-selected, it will be listed automatically; otherwise we will select it either from the Feature Manager or in the graphics area.

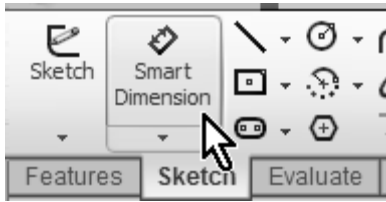


When selecting features from the graphics area, be sure to select a face that belongs to the feature. When traversing the Feature Manager, notice how SolidWorks highlights the features in the screen before selecting it.

Notice the preview and click OK. Rotate the view to inspect the mirrored feature.

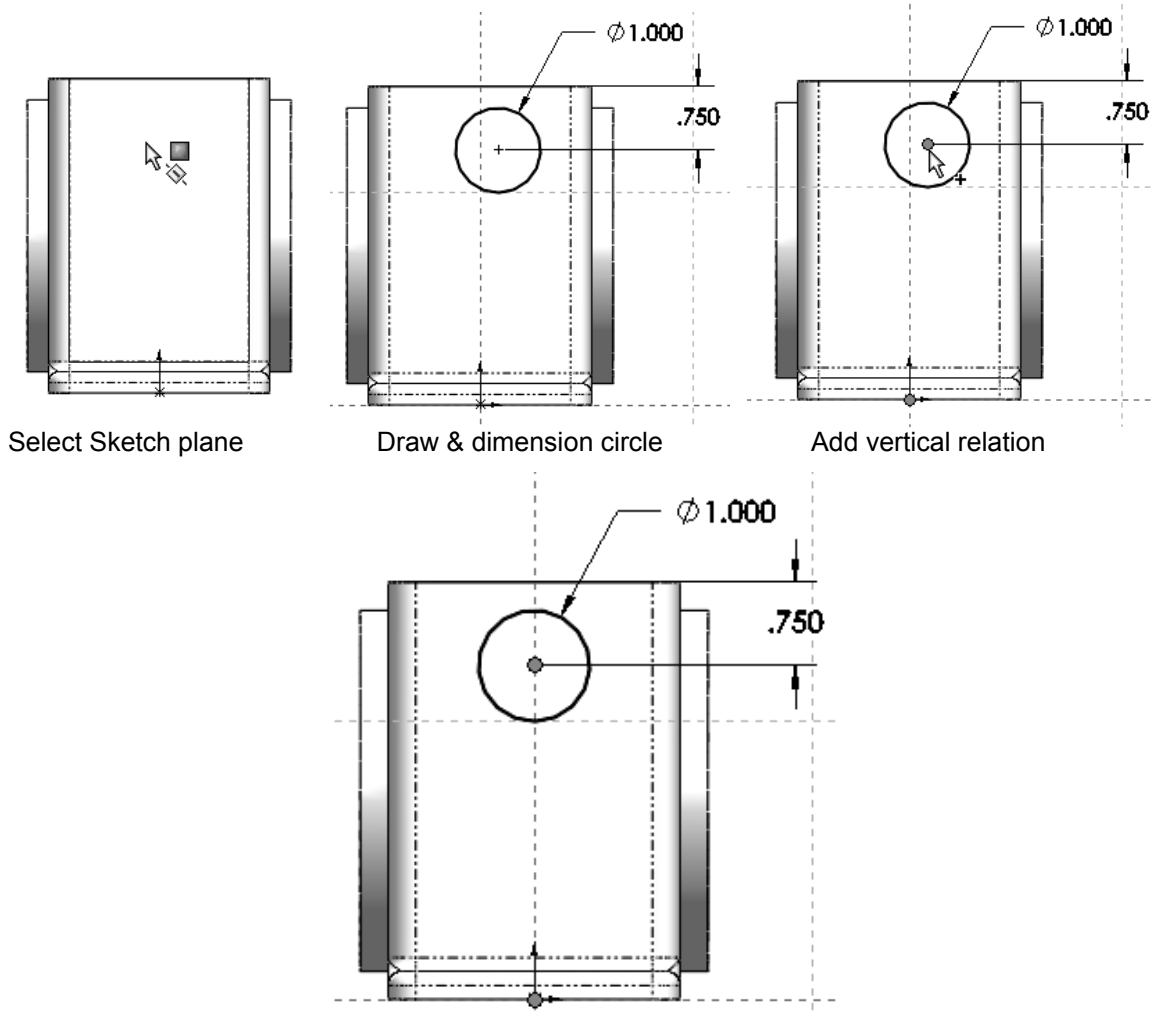
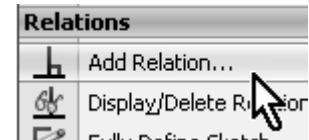


36. - In the next step we'll add the small boss at the right side of the Housing. Switch to a **Right View**, using the View Orientation toolbar (Shortcut Ctrl+4), and select the **"Sketch"** icon from the Command Manager's Sketch tab.



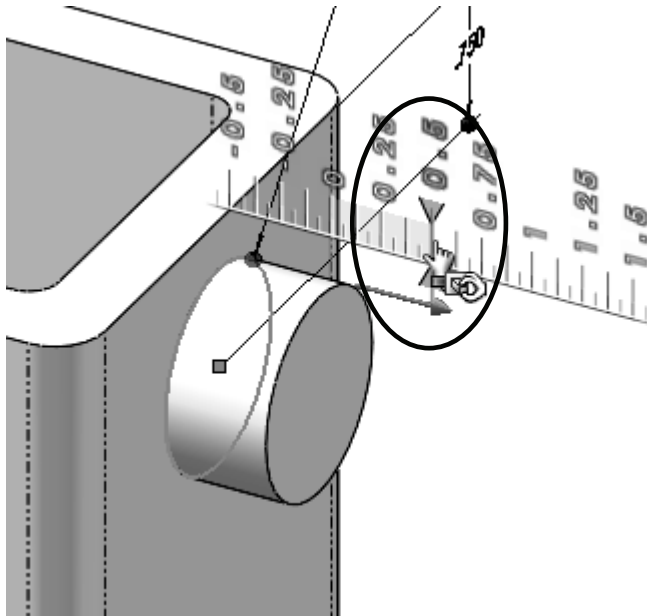
Select the rightmost face to create the Sketch, draw a circle using the "Circle" tool and add the dimensions shown. Just as we did with the front cylindrical boss, we'll add a **Vertical Relation** between the center of the circle and the part's origin.

From the Right Mouse click menu, select **"Add Relation"**, select the center of the circle and the origin, and click on "Vertical" in the "Add Relations" box.

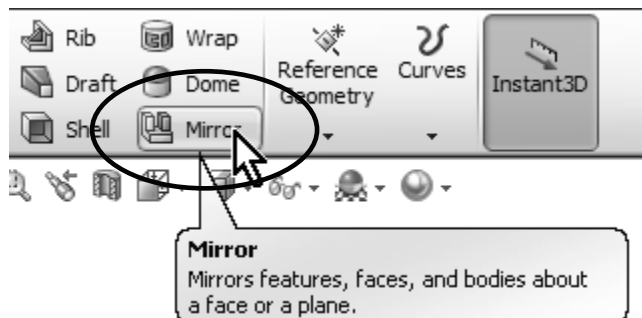


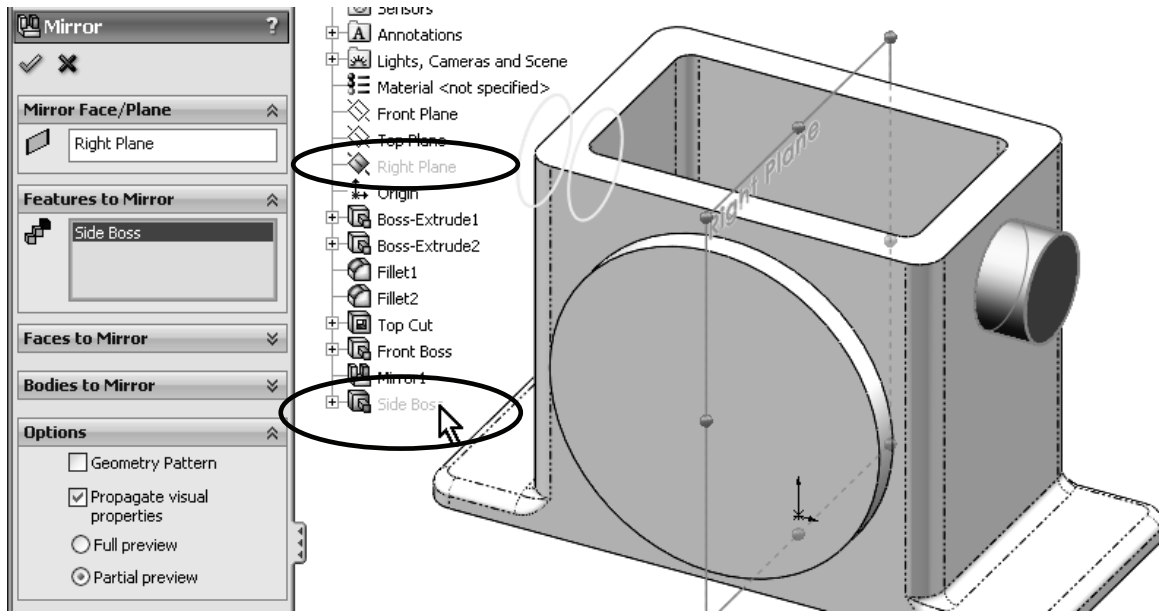
When adding relations, if you accidentally select more entities than needed, you can unselect them in the screen, or select them in the selection box and remove them using the "Delete" key.

37. - Now we are ready to extrude the sketch to make the side boss. We'll use the "Instant 3D" function as we did in the previous extrusion. Exit the Sketch by selecting the "Exit Sketch" icon in the Command Manager's Sketch tab and change to an Isometric view. In the graphics area select the circle of the sketch we just drew, and click-and-drag the arrow along the ruler markers to make the extrusion 0.5" long. Rename this extrusion "Side Boss" in the Feature Manager.

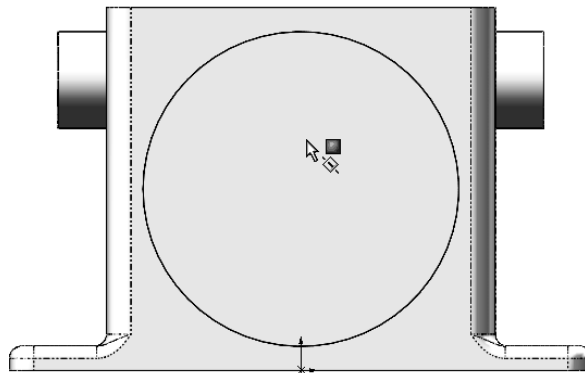


38. - Just like we did with the front circular boss, we'll mirror this extrusion about the **Right Plane** (which is also in the middle of the part). Select the "Mirror" command from the "Features" tab in the Command Manager, and using the fly-out Feature Manager select the "Right Plane" as the "Mirror Face/Plane" and the "Side Boss" extrusion in the "Features to Mirror" selection box to complete the Mirror command.

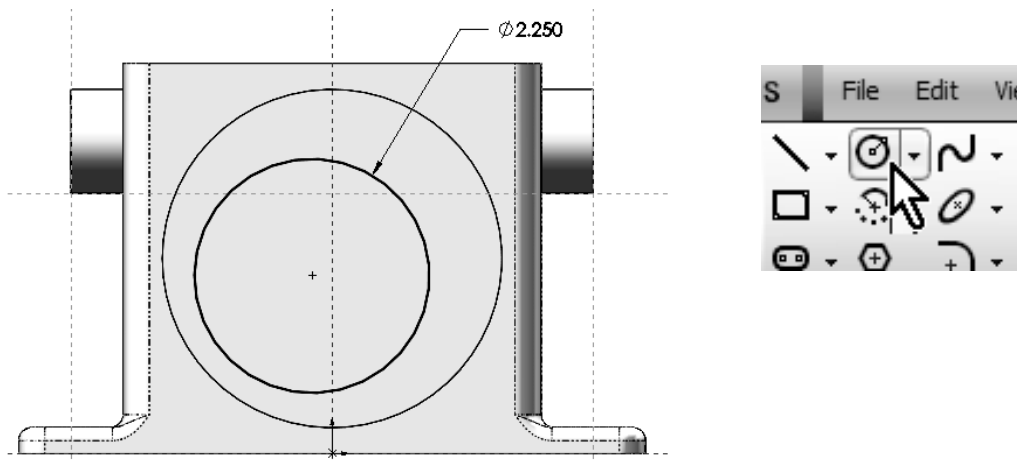




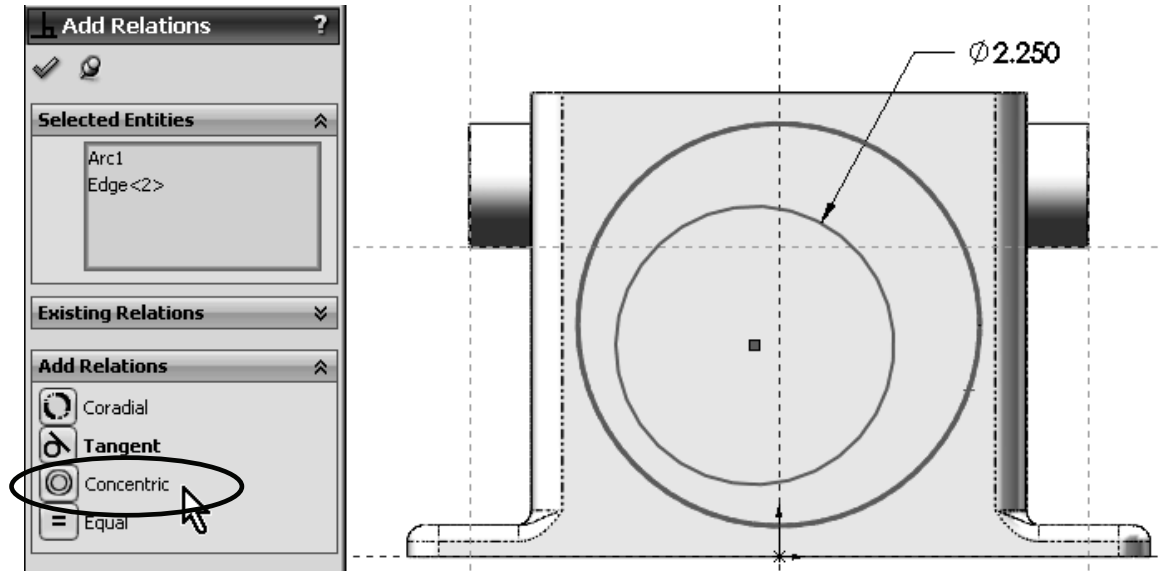
39. - We'll now make the circular cut in the front of the Housing. Change to a "Front View" for easier visualization. Select the "Sketch" icon from the Sketch tab and click in the round front face of the part.



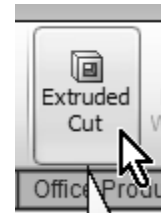
Draw a circle using the "Circle" tool and dimension it 2.250" diameter.



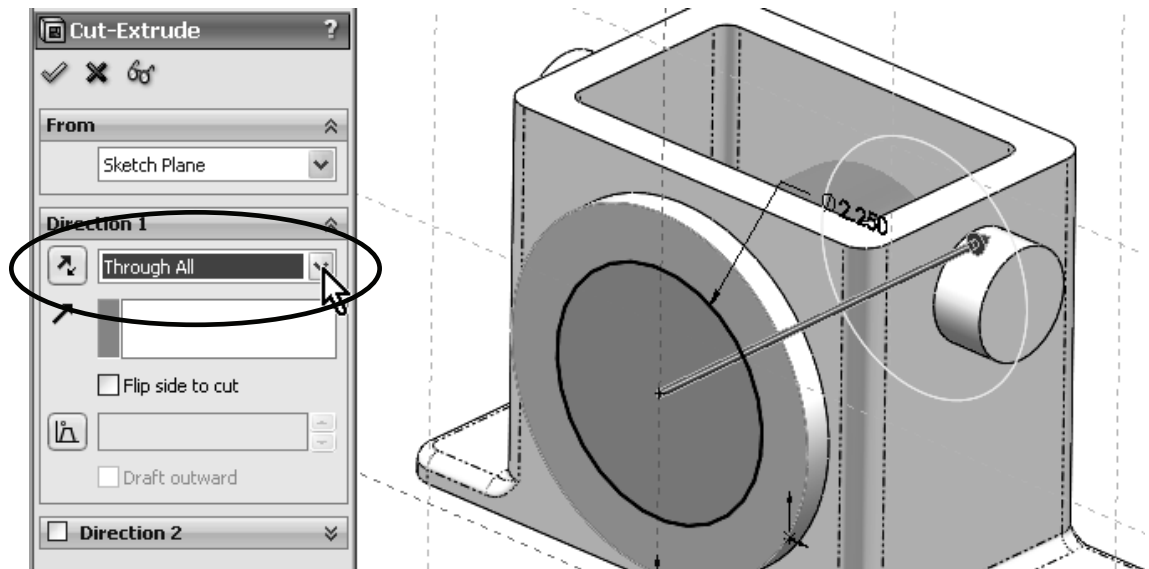
To locate the circle in the center of the circular face, we'll add a **"Concentric Relation"**. Select the **"Add Relation"** icon from the Right Mouse button menu, and select the circle we just drew and the edge of the circular face. Click **"Concentric"** to add the relation and center the circle. Click OK to finish the command.

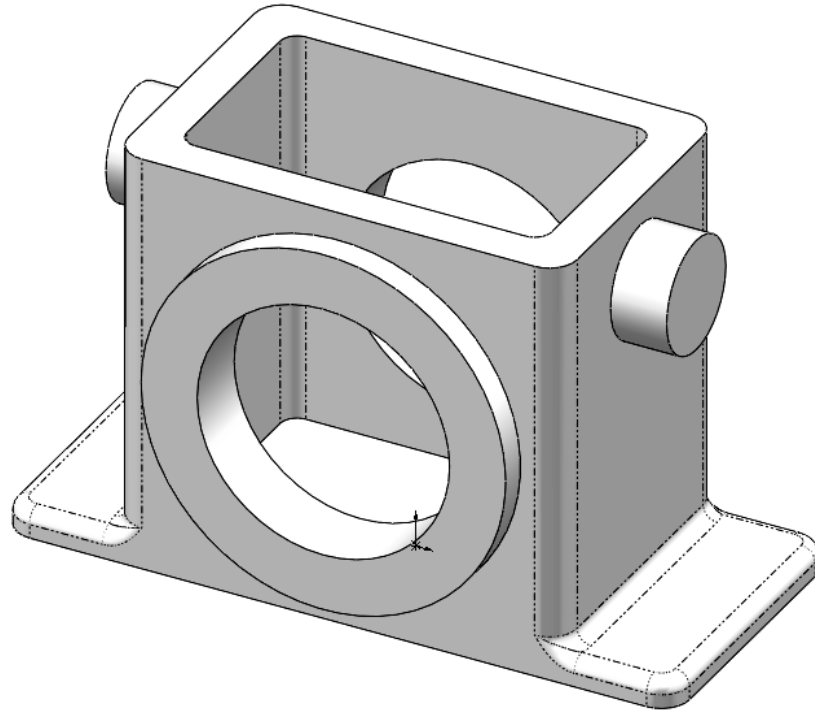


40. - Now that the circle is concentric with the boss, we'll make the cut. Select the **"Extruded Cut"** command and switch to an isometric view for better visualization.

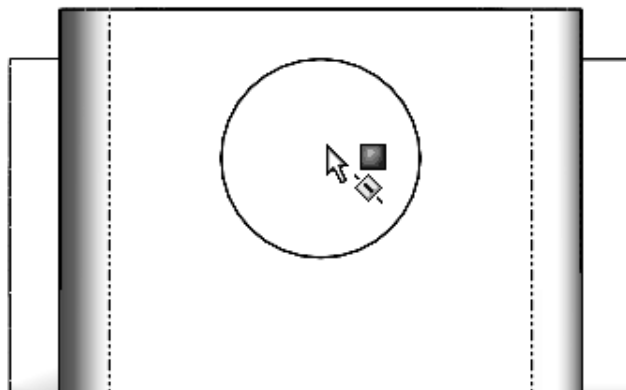


From the **"Extruded Cut"** properties select the **"Through All"** option; this way the cut will go through the entire part regardless of its size. In other words, *if we ever make the Housing wider, the cut will still go through the part.*

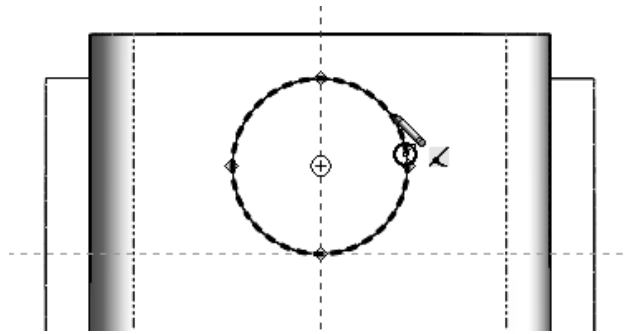




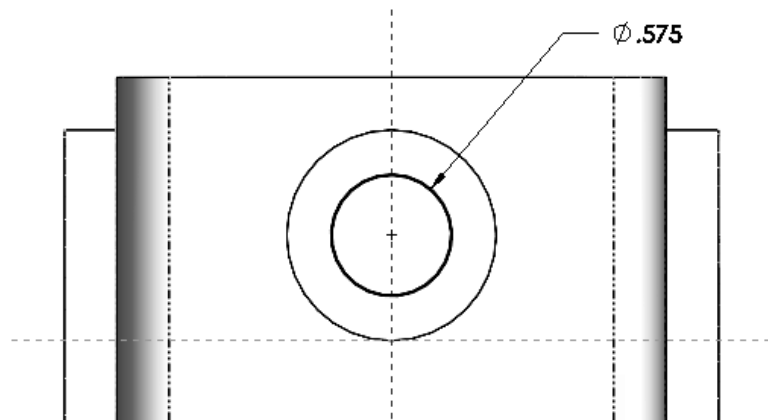
41. - We will now make a hole in the boss added in step 37 for a shaft. Switch to a “**Right View**” and create a sketch on the small circular boss’ face by selecting the “**Sketch**” icon and then the circular face to locate it.



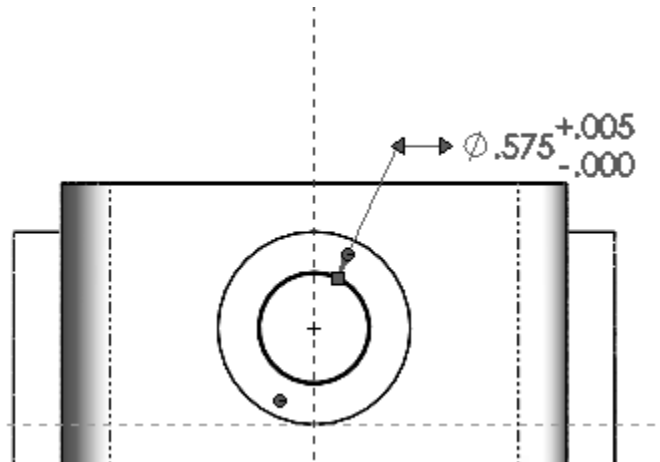
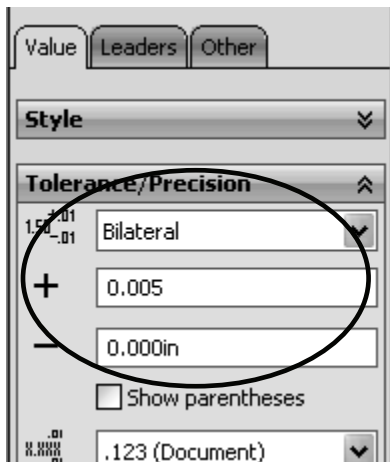
We know that we want the hole to be concentric with the boss. In order to do this we can draw the circle and add a concentric relation as we did before; however, this is a two step process. Instead, we will do it in one step as follows: Select the “**Circle**” tool icon from the “**Sketch**” toolbar and *before* drawing the circle, move the cursor and rest it on top of the circular edge as shown, the center of the circular edge is revealed in a fraction of a second. **DO NOT CLICK ON THE EDGE.** This highlight works only if you have a drawing tool active (Line, Circle, Arc, etc.). This technique can be used to reveal any circular edge’s center.



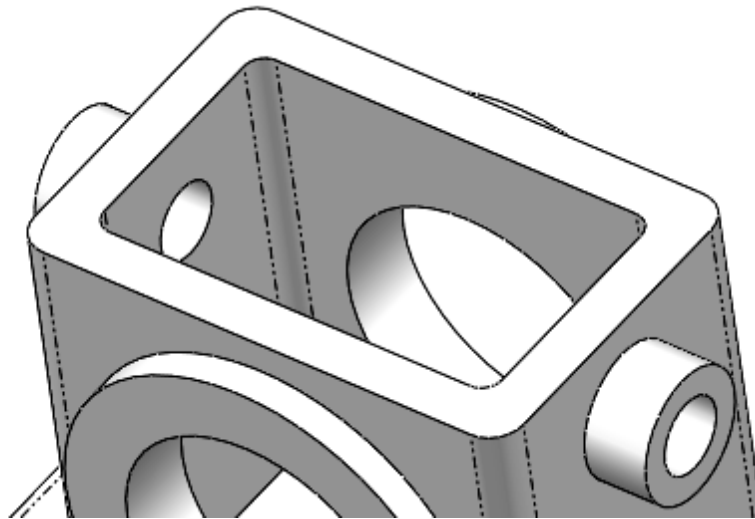
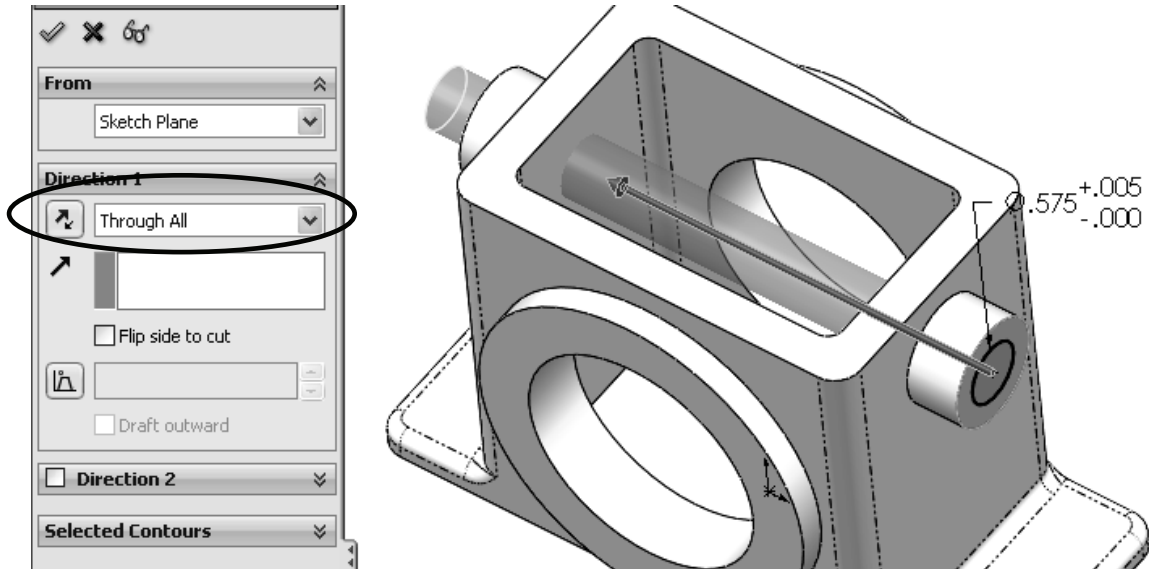
42. - Once the circular edge's center is revealed, click in the center to start drawing the circle and automatically capture a concentric relation with the boss. Finish the circle and dimension it 0.575" diameter. Now the sketch is fully defined.



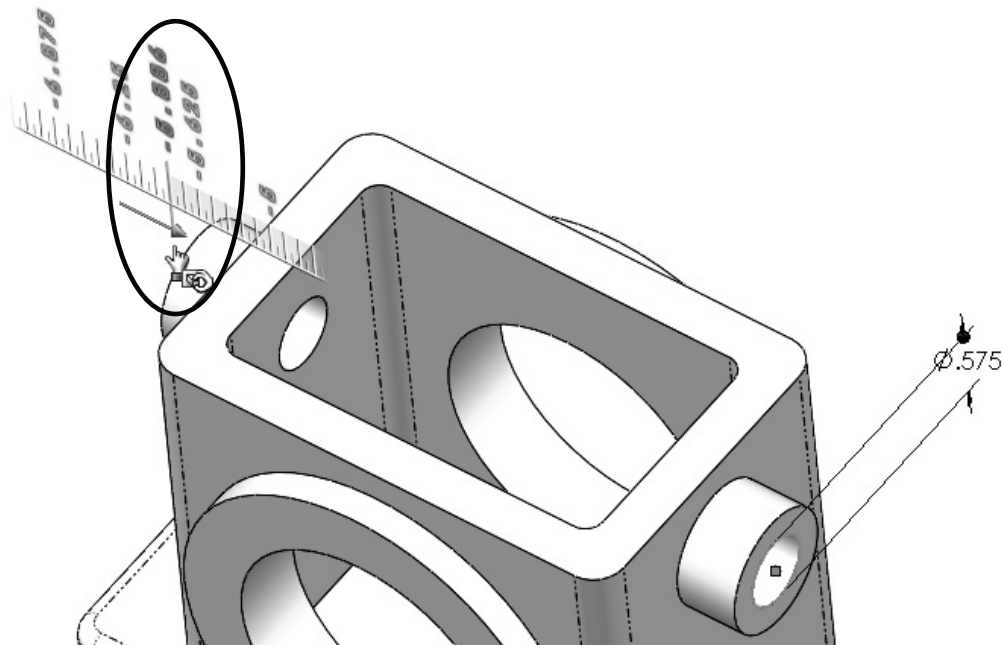
43. - Since this hole will be used for a shaft, we need to add a bilateral tolerance to the dimension. Select the 0.575" dimension in the graphics area, and from the dimension's Property Manager, under "Tolerance/Precision" select "Bilateral". Now we can add the tolerances. Notice that the dimension changes immediately in the graphics area. This tolerance will be transferred to the Housing's detail drawing later on. If needed, tolerances can also be added later in the detail drawing.



44. - Now we can make a Cut with the “Through All” option using the “**Extruded Cut**” command. Rename the Feature “Shaft hole”.

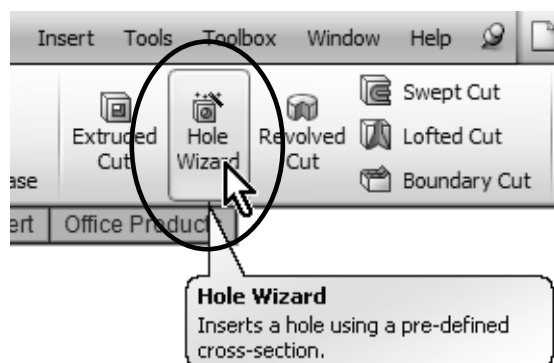


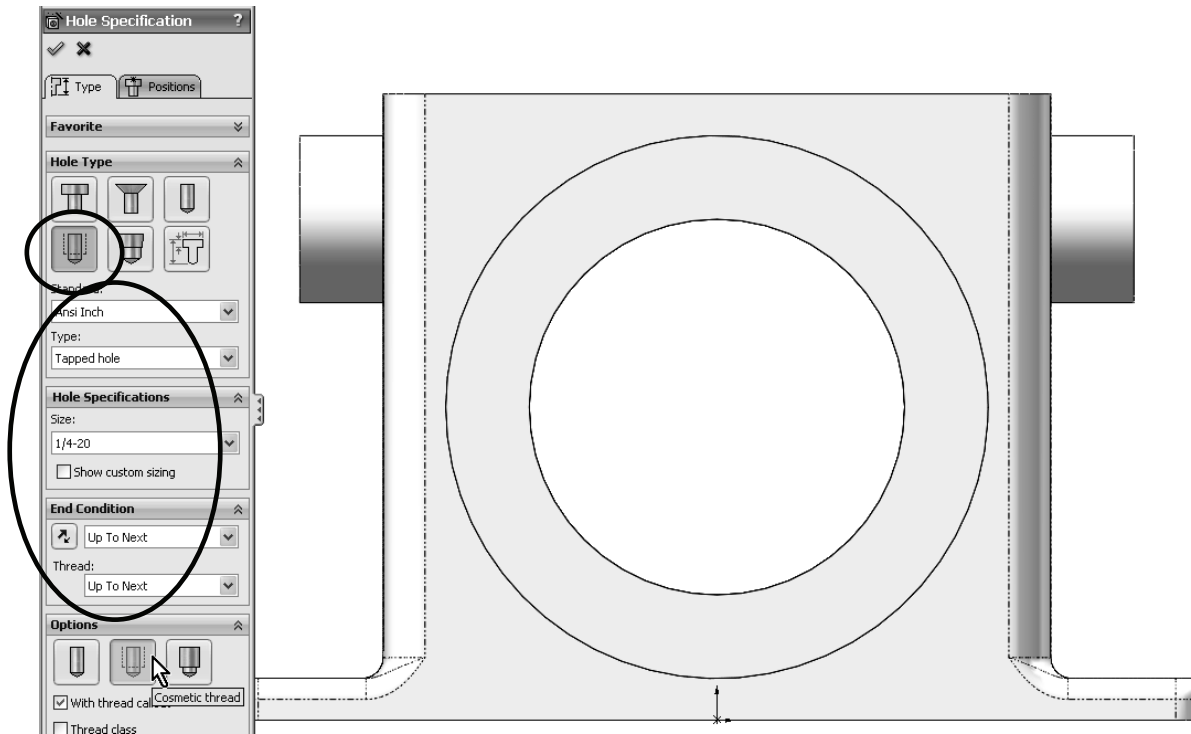
45. - Another way to do this cut is using the “Instant 3D” feature. To use “Instant 3D” to make a cut, click in the “**Exit Sketch**” icon, change to an Isometric view for clarity, and just like we did for the Extrusion, select the sketch circle and click-and-drag the handle *into* the part. You’ll see how the part is cut as the arrow is dragged. The only disadvantage to making the cut using this technique is that the “Through All” option is not available while dragging.



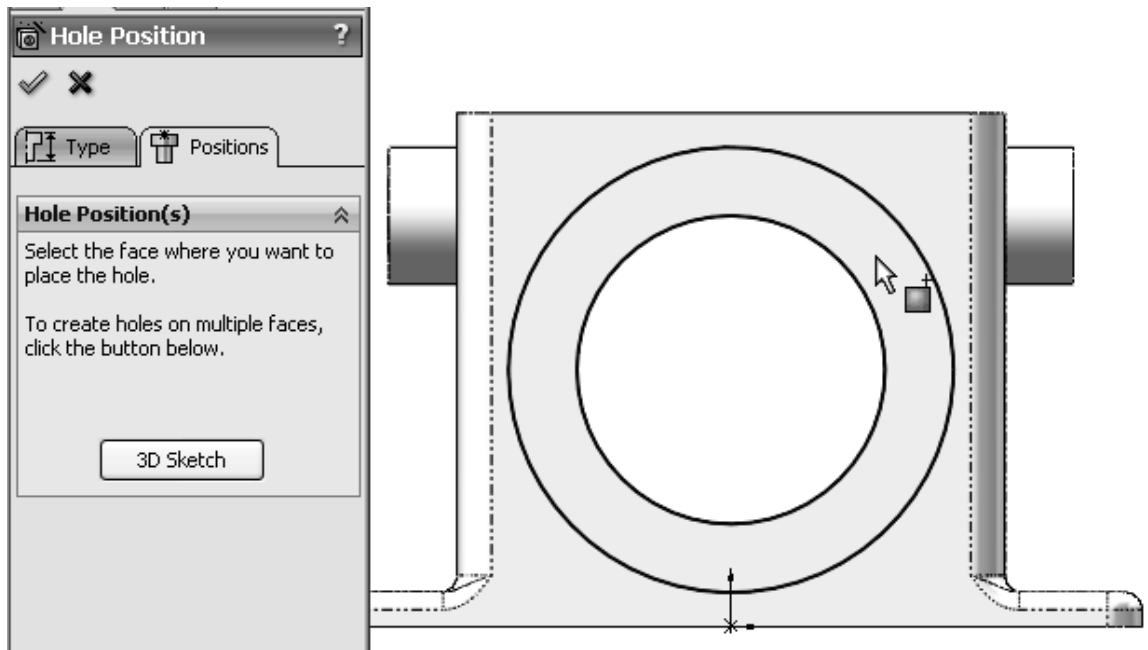
46. - For the next feature we'll make a 1/4"-20 tapped hole in the front face. SolidWorks provides us with a tool to automate the creation of simple, Countersunk and Counterbore holes, tap and Pipe taps by selecting a fastener size, depth and location. The "**Hole Wizard**" command is a two step process: in the first step we define the hole's type and size, and in the second step we define the location of the hole(s). To add the tapped hole, switch to a "Front View". The Hole Wizard is a special type of feature that uses 2 sketches that are automatically created, so there is no need to add a sketch first; it works very much like an applied feature.

Change to a Front View for clarity, and select the "Hole Wizard" icon from the "**Features**" tab in the Command Manager. The first thing we'll do is to define the hole's type and size. Select "**Tap**" for "Hole Type", "ANSI Inch" for Standard, "Tapped Hole" for Screw type and "1/4-20" for size from the drop down selection list. Change the "End Condition" to "Up to Next", this will make the tapped hole's depth up to the next face where it makes a complete round hole. At the bottom activate the button to add **Cosmetic Threads**.

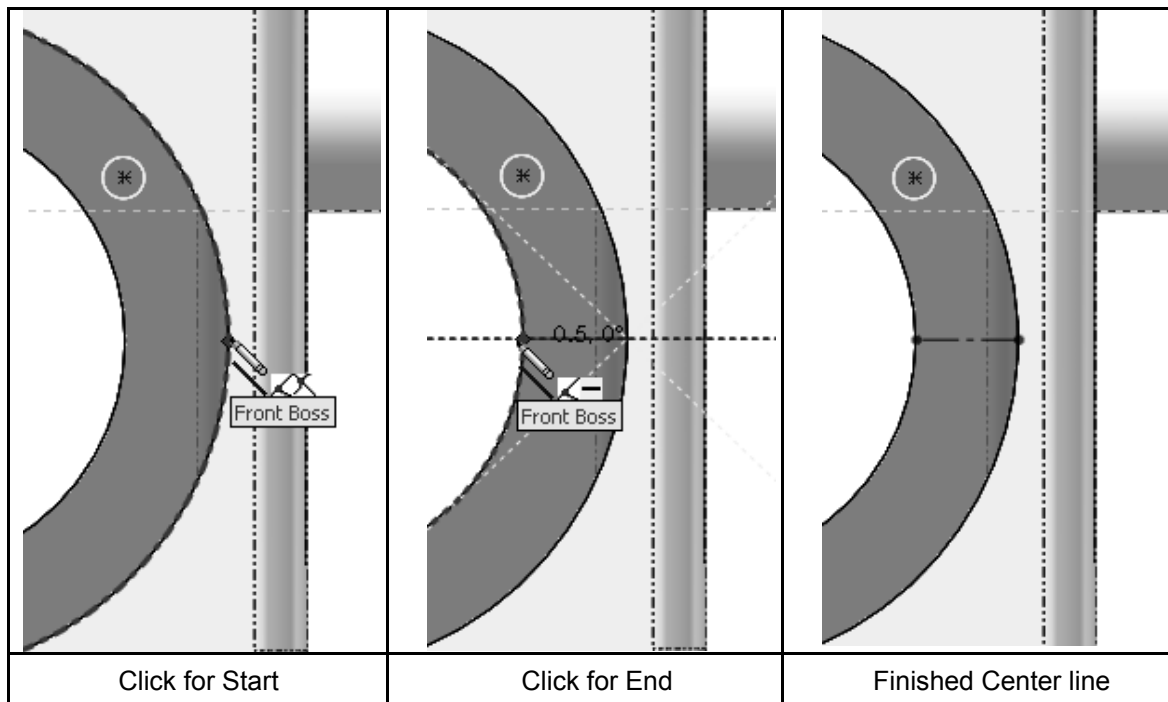
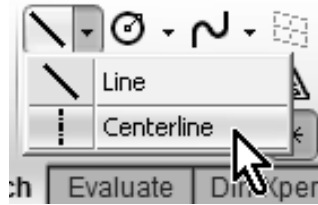




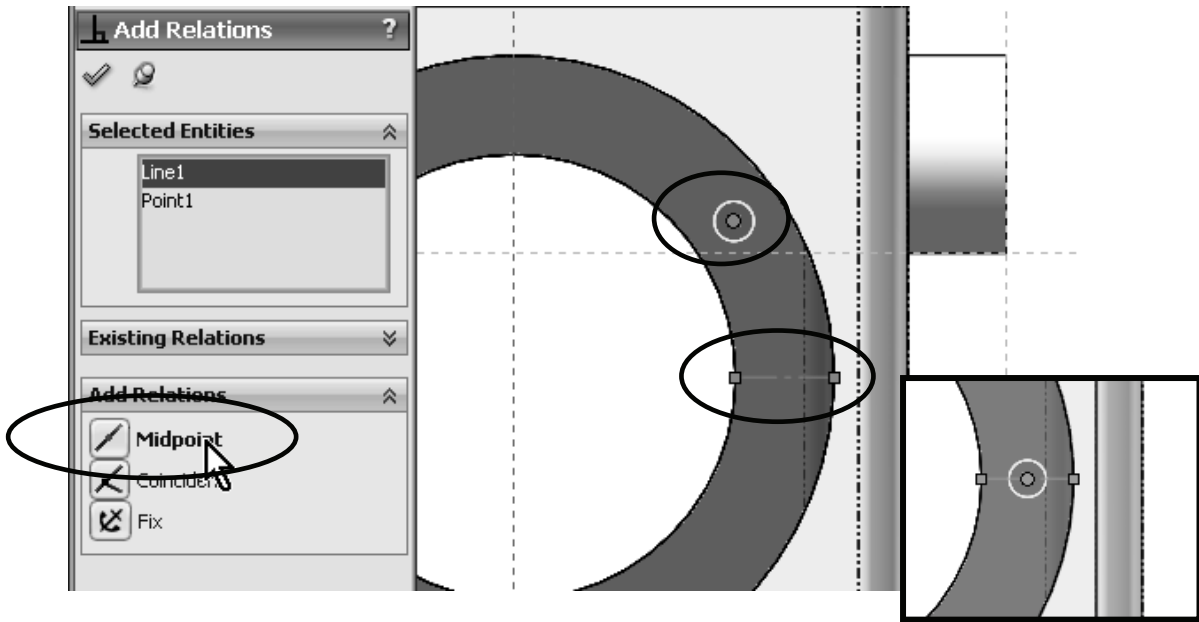
47. - The second step of the “Hole Wizard” is to define the hole’s location. After selecting the “Positions” tab at the top of the properties, SolidWorks will ask to select a flat face to locate the hole(s). Select the round face at the front of the part.



SolidWorks will add a “**Sketch Point**” where we select the face, and the Sketch Point tool will be automatically activated; this is in case we want to add more than one point in the face. For this exercise we will only make one hole. This point will define the location of the hole, and in order to precisely locate it, we can use regular Sketch tools and relations (Take notice that we are working in a Sketch). We want this hole to be located in the middle of the flat face, to locate it select the “**Centerline**” tool from the drop-down menu in the “Line” command. Start drawing the centerline at the right quadrant of the *outer circular edge*, and finish it in the same quadrant of the *inner circular edge*. The quadrants will be activated after selecting the Centerline tool, and touching (not clicking!) the circular edge. Hit the “Esc” key once to end the “Centerline” command.



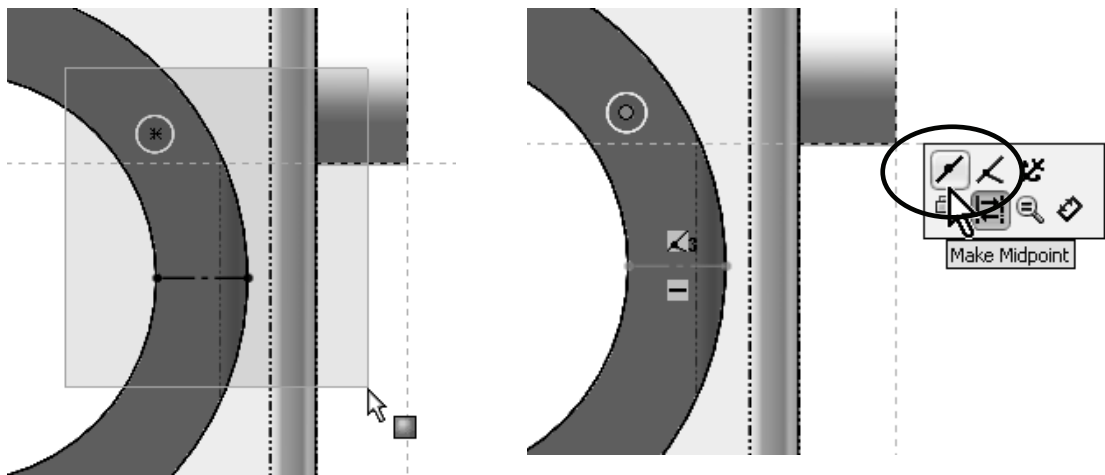
The idea behind this technique is to make sure the hole is centered in the circular face. We'll now add a relation using the “**Add Relation**” command; select the pre-existing point and the centerline we just made and click on “**Midpoint**”, this way the point (the center of the hole) will be located exactly in the middle of the centerline. Now the “**Point**” is located in the middle of the Centerline.



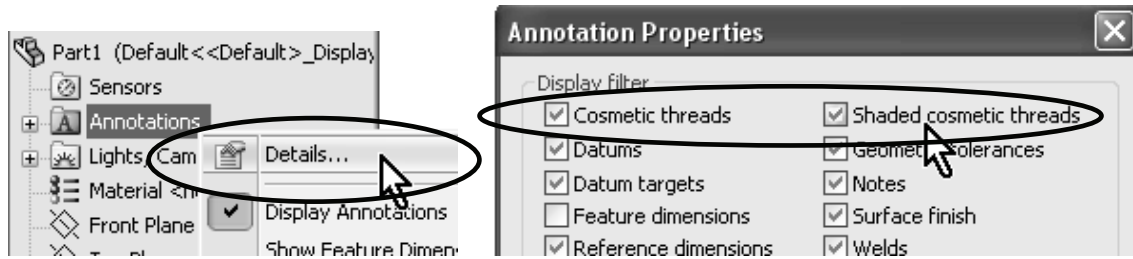
Click OK to close the “Add Relations” dialog, and then click OK again to finish the Hole Wizard.



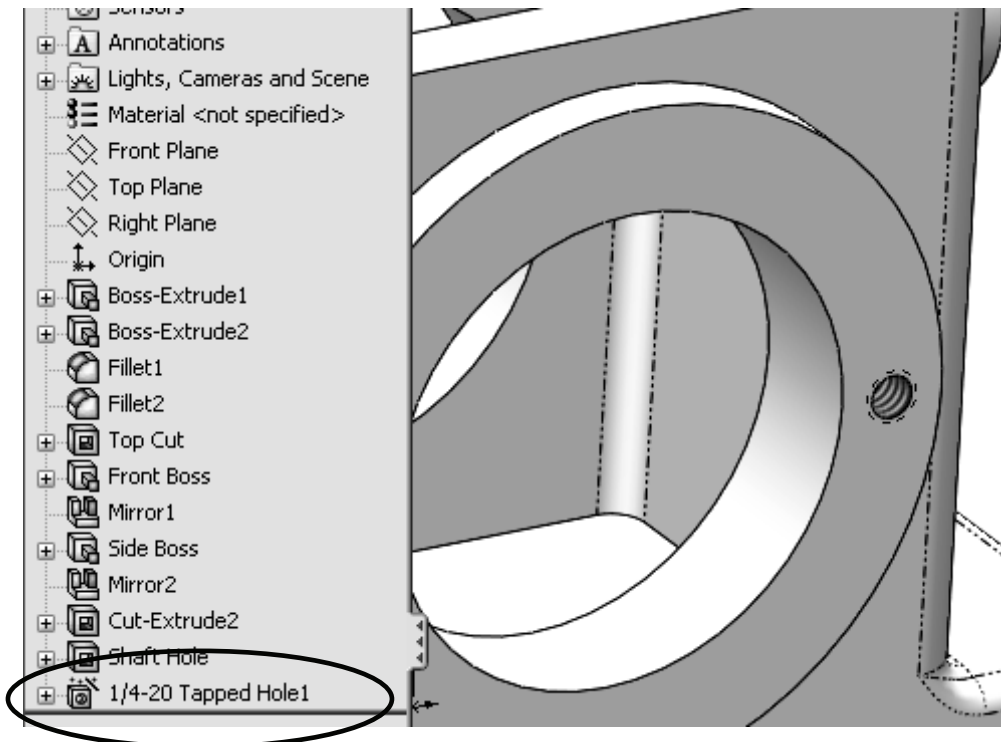
A quicker way to add this relation is to Window-Select the “**Point**” and the “**Centerline**”, and from the pop up toolbar select “**Midpoint**” to add the relation. This approach works well when we pre-select the entities to relate.



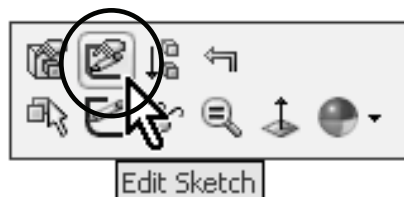
In order to view the “Cosmetic Threads” (for a nicer look), Right-Mouse-Click in the “Annotations” folder at the top of the Feature Manager, select Details and activate the options “Cosmetic Threads” and “Shaded Cosmetic Threads”. This is just for show, as there are no real threads modeled. It can be done, but it’s mostly unnecessary in this type of design.

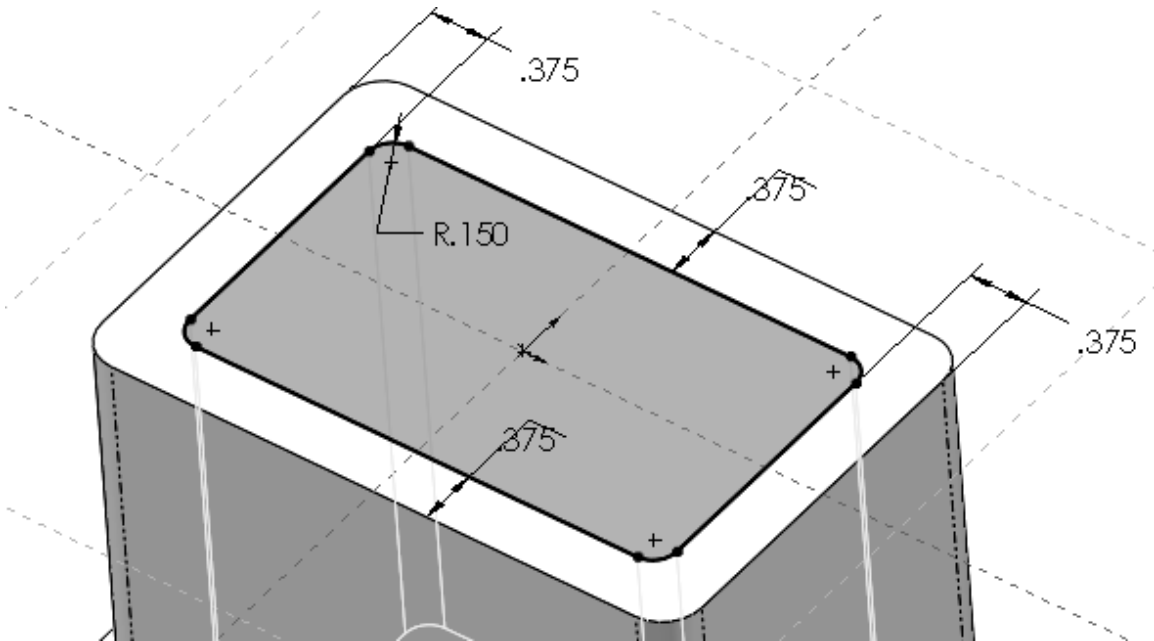
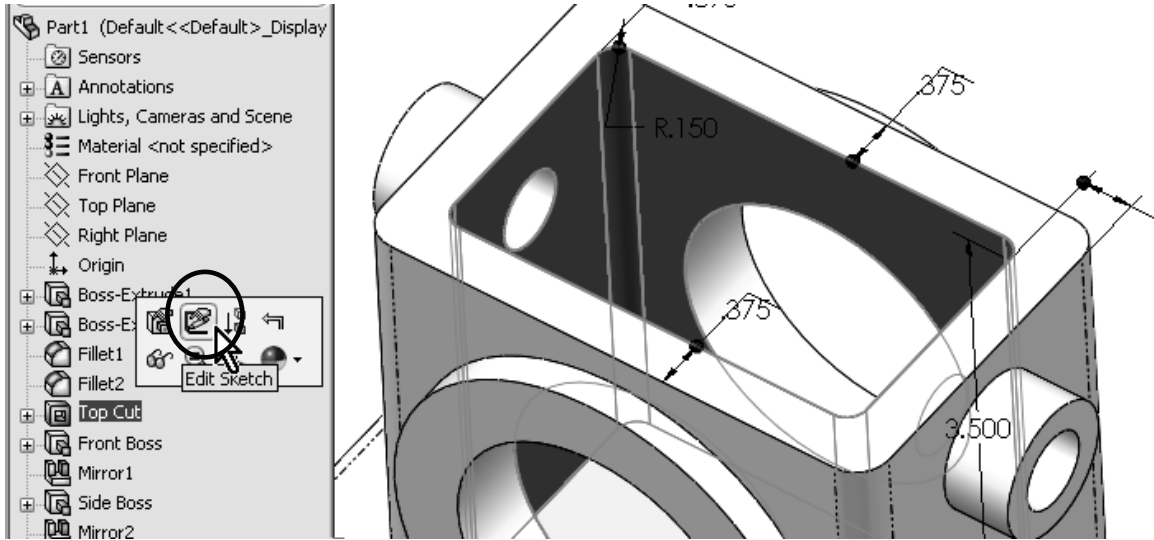


This is the finished 1/4"-20 Tapped Hole with Cosmetic Threads.

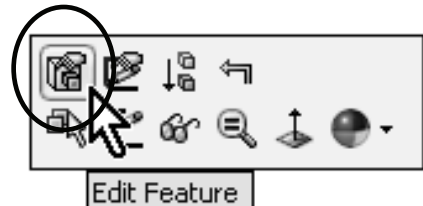


48. - After making the Tapped hole, we realize that we want the walls of the Housing to be thinner, and need to make a change to our design. In order to do this, we find the feature that we want to modify in the Feature Manager ("Top Cut") or in the graphics area, and select it. From the pop up toolbar, select the **"Edit Sketch"** icon. This will allow us to go back to the original sketch and make changes to it. Notice the selected feature is highlighted in the screen.





Selecting the “Edit Feature” icon will show the feature’s command, this is where we can change the cut’s depth and other parameters.

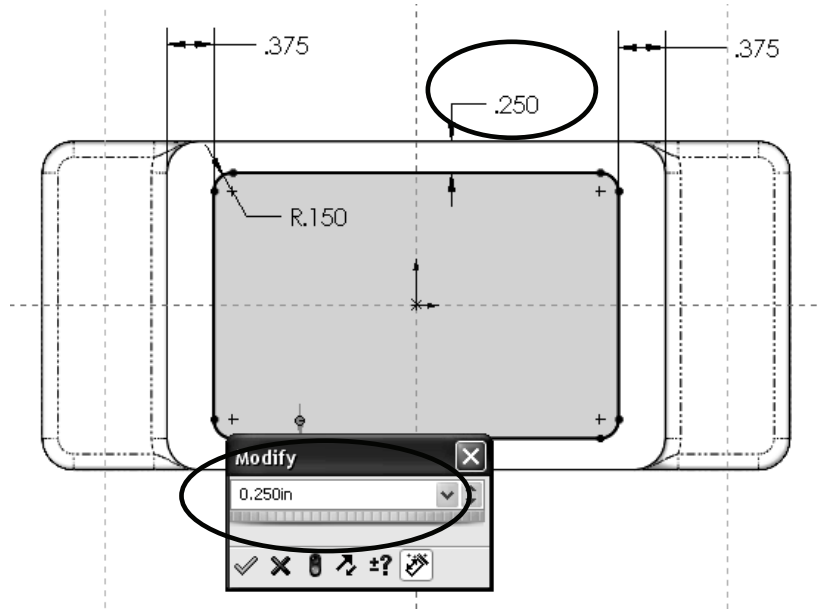


If we select the feature with a Right-Mouse-Click, we will see the pop up toolbar along with an options menu. The most commonly used commands are already in the pop up toolbar



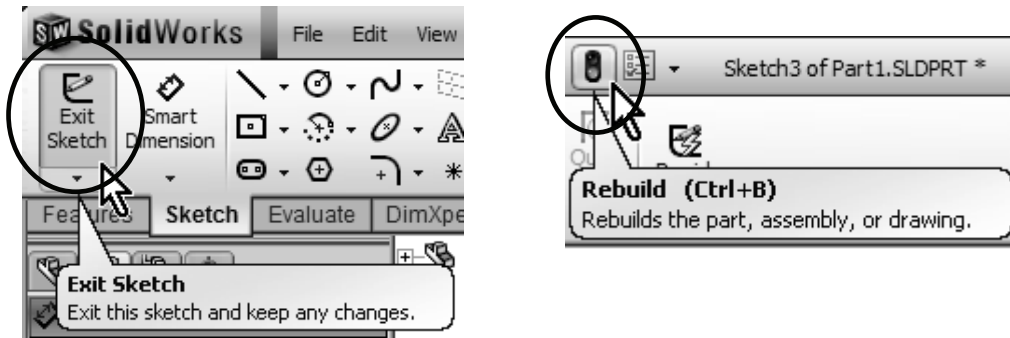
If “Instant 3D” is activated, selecting a feature will show its dimensions on the screen.

49. - What we just did was to go back to editing the feature's Sketch, just like when we first created it. Switch to a top view if needed for visualization. To change a dimension's value, double click on it to display the "Modify" box. Change the two dimensions indicated from 0.375" to 0.25" as shown.



To move a dimension just click-and-drag it.

50. - After changing the dimensions we cannot make another "Cut Extrude" because we had already made a cut; what we have to do is to "Exit Sketch" or "Rebuild" (Shortcut Ctrl+B) to update the model with the new dimension values.

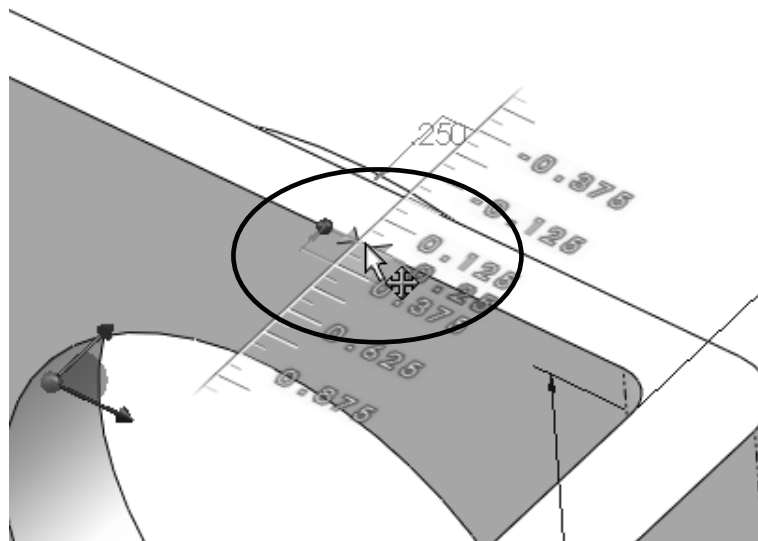
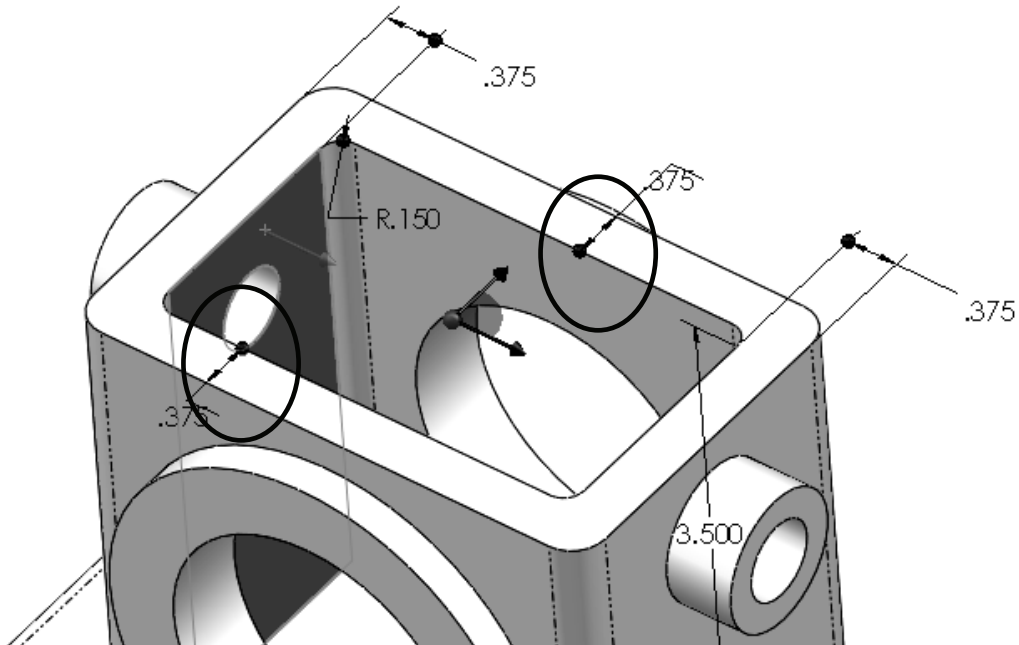


There is no real purpose to this dimensional change but to show the reader how to change an existing feature's sketch if needed, including dimensions, geometric relations and add or remove sketch geometry.

Another way we can make this change is using the "Instant3D" functionality. The way it works is very simple: We select the feature that we want to modify either in the Feature Manager or the graphics area (in this case one of the inside faces which were made with the

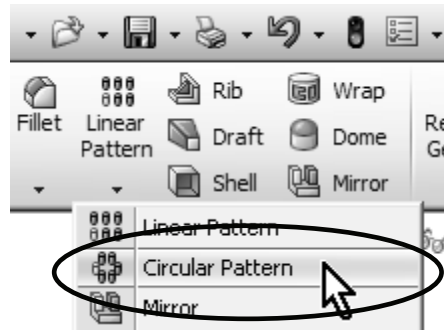


Cut Extrude) and click-and-drag the **blue dots** at each of the dimensions that need to be modified until we get the desired value, without having to edit the sketch. Dragging the mouse pointer over the ruler markers will give you values in exact increments. Depending on the speed of your PC and the feature being modified Instant3D may be slow, as the solid model is dynamically updated.

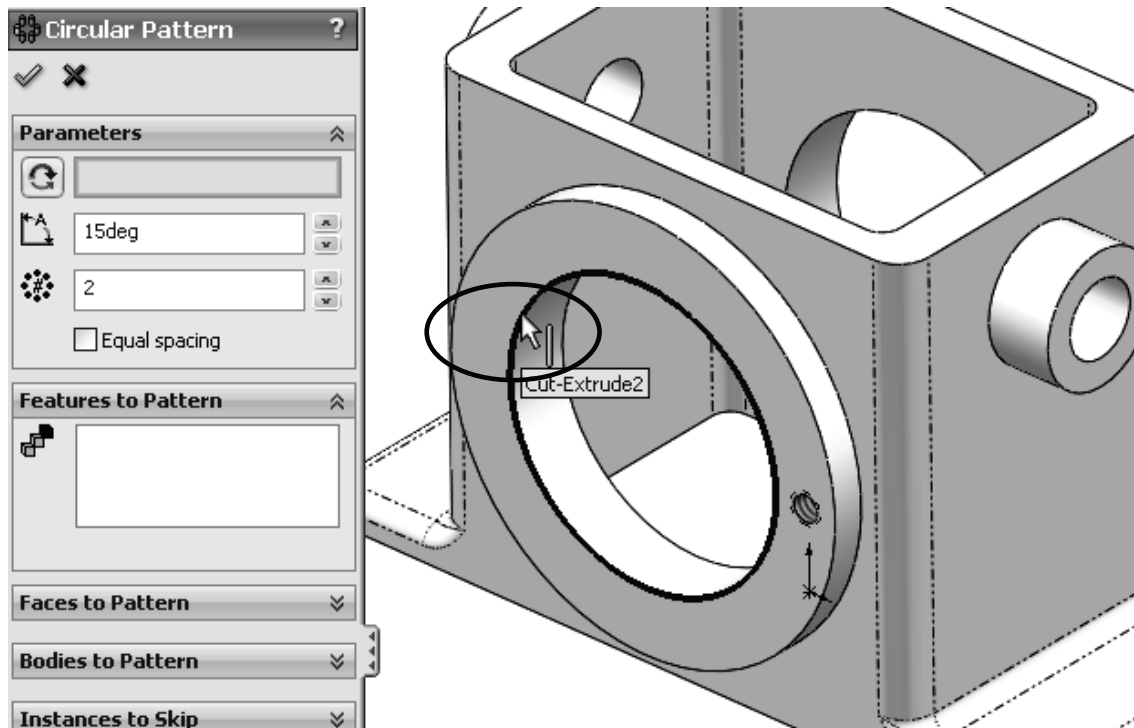


A third option to change the dimensions is to select the value and type a new one. If Instant3D is not active, the Modify box will be displayed, and after making all the dimensional changes needed the user needs to select the Rebuild icon to update the model.

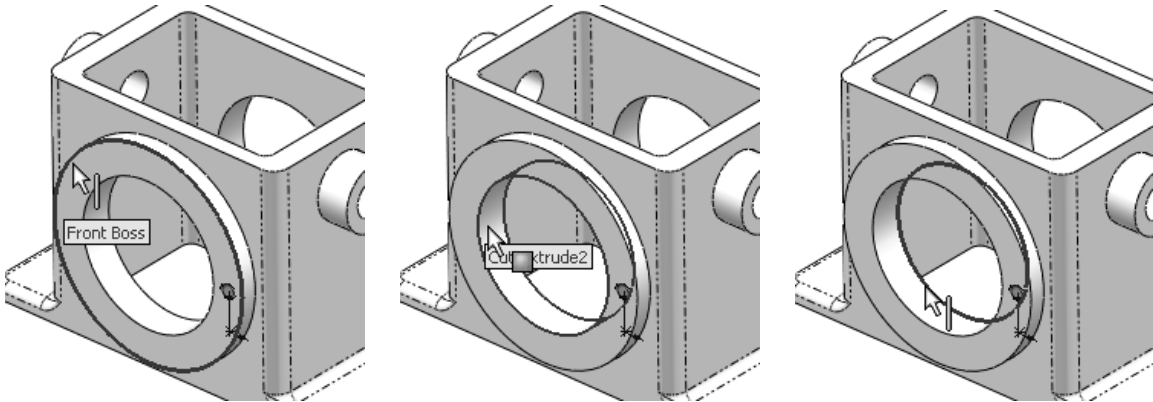
51. - Now we will add more tapped holes to complete the flange mounting holes. We'll use the first hole as a "seed" to make copies of it using the "**Circular Pattern**" command. In the "Features" tab, select the drop-down list below the "**Linear Pattern**" to reveal the drop down menu and select "**Circular Pattern**". Notice that commands are grouped by similar functionality. Optionally use the menu "**Insert, Pattern/Mirror, Circular Pattern**".



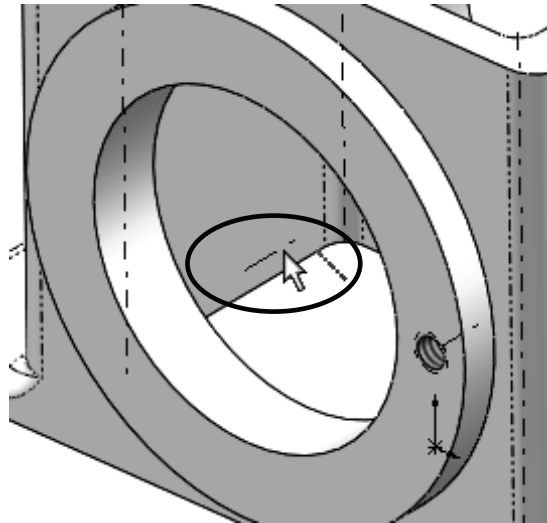
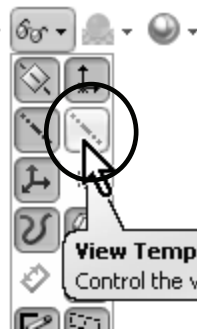
52. - The "**Circular Pattern**" needs a circular edge, a cylindrical surface or an axis as a reference for the direction of the circular pattern. Click inside the Parameters selection box to activate it, and then select the edge indicated for the direction of the pattern.



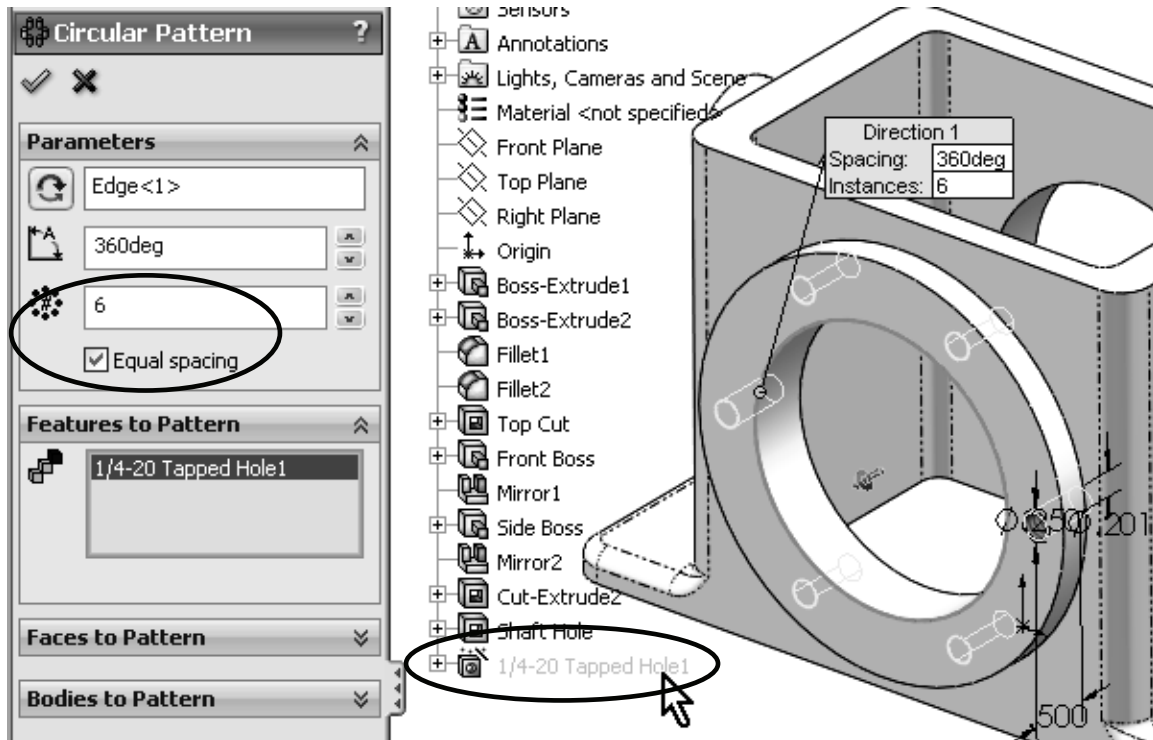
53. - Any circular edges or cylindrical faces that share the same axis can be used for a direction, as shown in the following images.



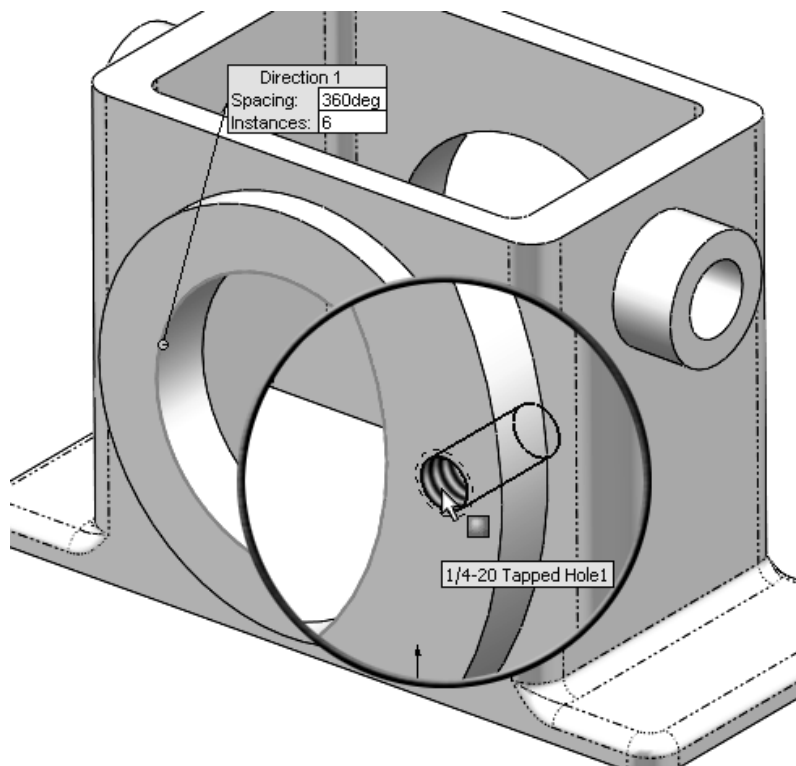
Another option for direction is a temporary axis. Every cylindrical surface has a "Temporary Axis" that runs through its center. To view them select the menu **"View, Temporary Axes"** or in the "Hide/Show Items" toolbar.



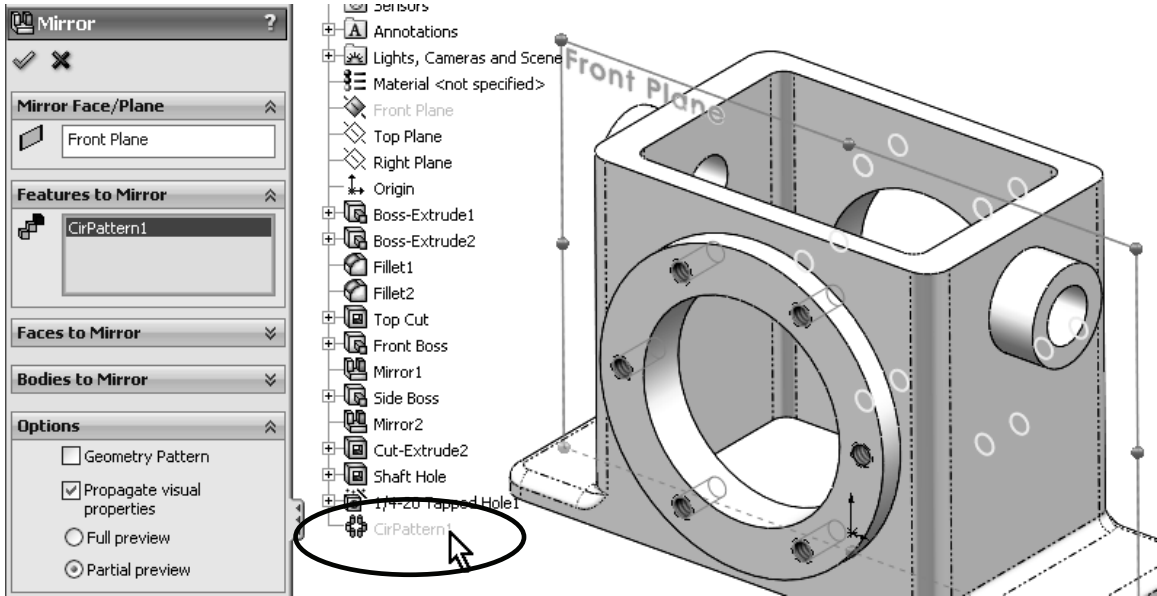
54. - Now click inside the "Features to Pattern" selection box to activate it (Notice it gets highlighted). Select the "1/4-20 Tapped Hole1" feature from the fly-out Feature Manager; change the number of copies to six (notice this value includes the original), and make sure the "Equal spacing" option is selected to equally space the copies in 360 degrees. Notice the preview in the graphics area and click OK to finish the command.



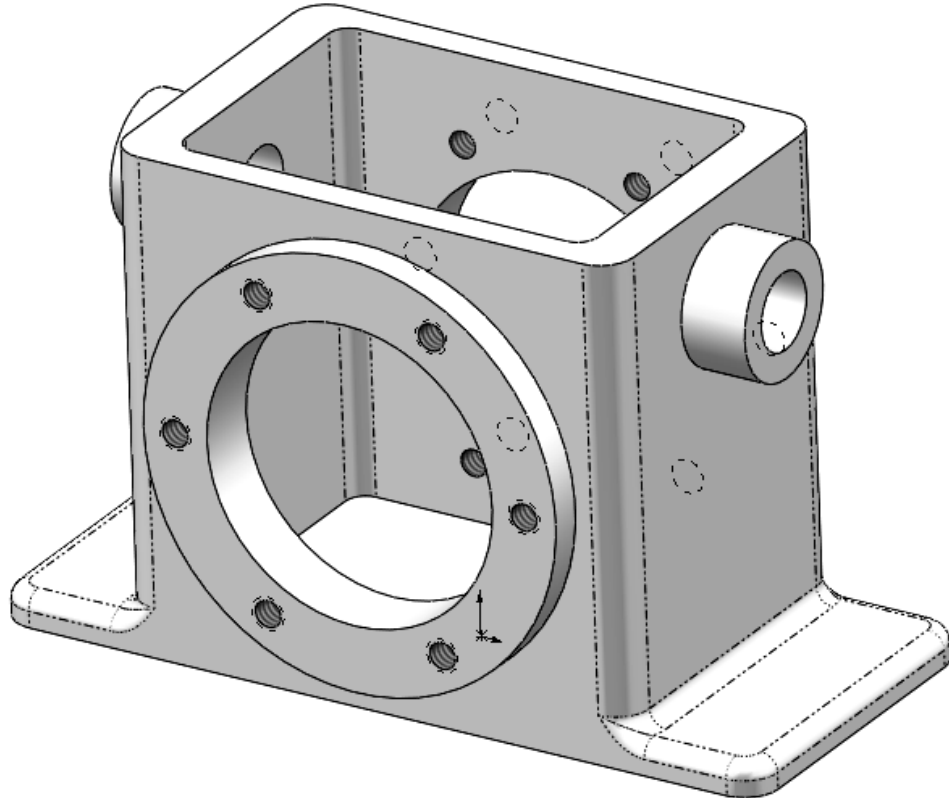
The feature to be patterned can also be selected from the graphics area; in this case a face of the feature needs to be selected. Sometimes a face can be difficult to select because it may be small, like this hole. In this case, we can use the “**Magnifying Glass**” (Shortcut “G”) to make selection easier.



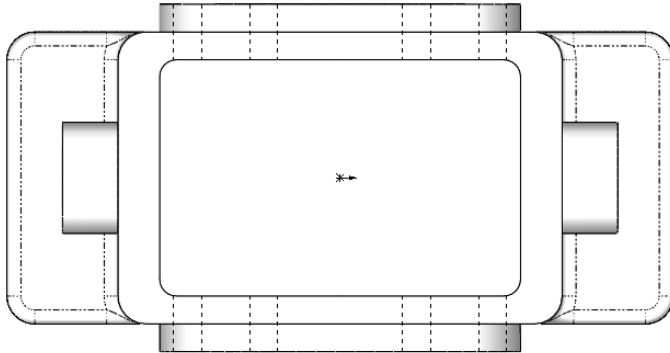
55. - Since we need to have the same six holes in the other side of the Housing, we will use the “Mirror” command to copy the Circular Pattern about the “Front Plane”. Make this mirror about the Front Plane and mirror the “CircPattern1” feature created in the previous step.



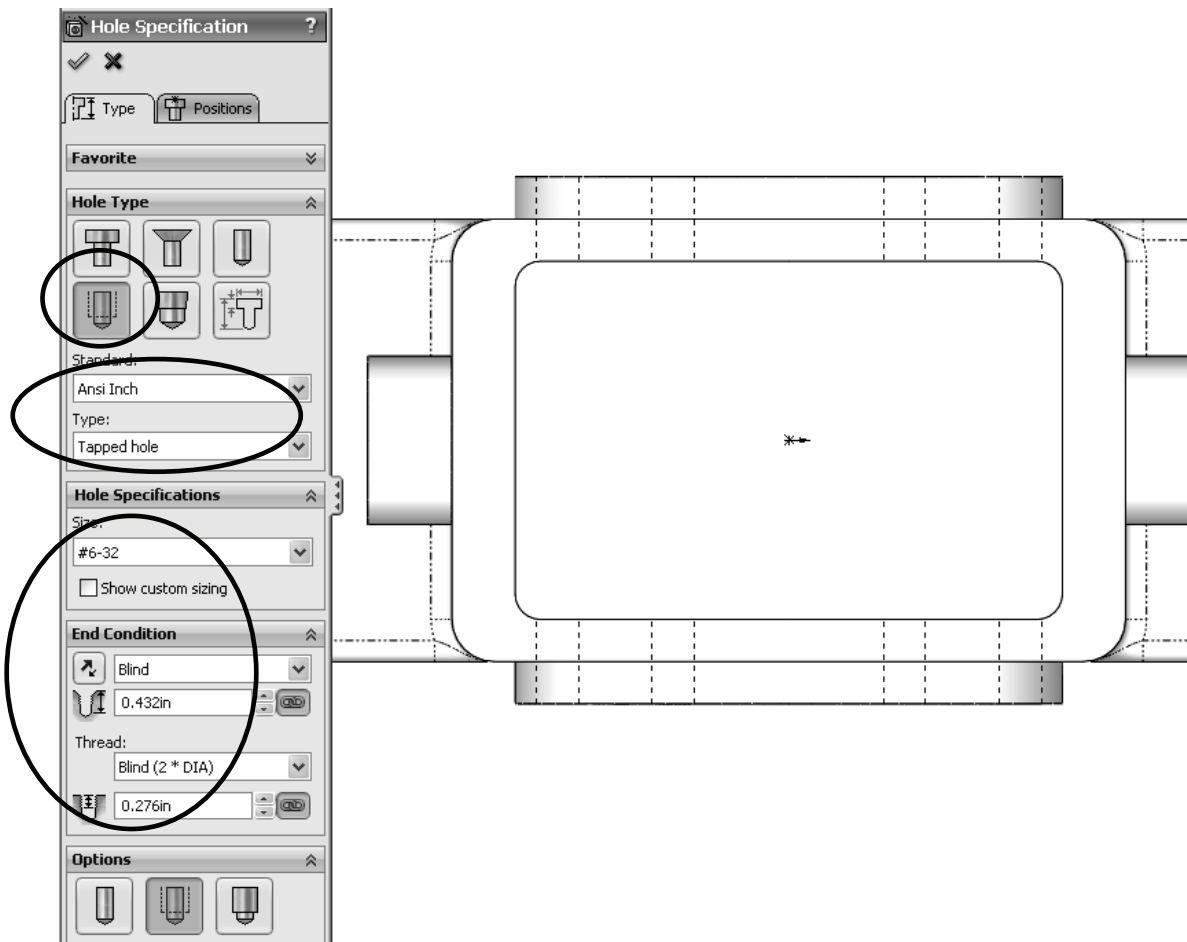
After the mirror, your part should look like this:



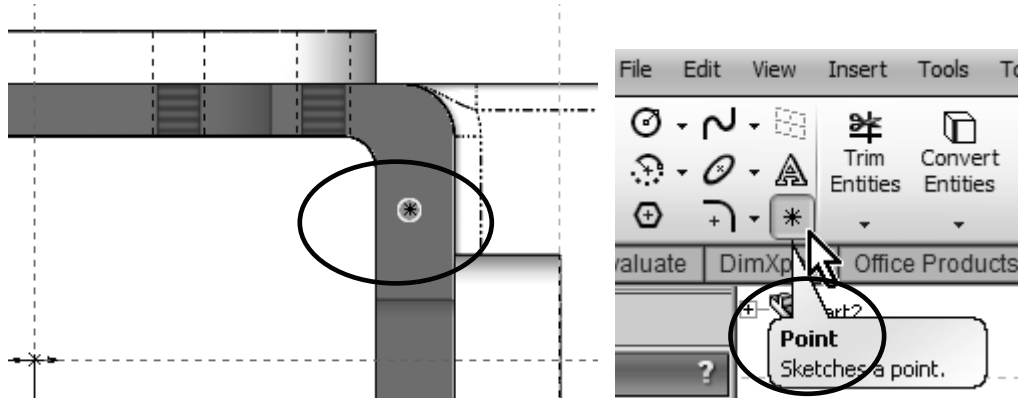
56. - We will now add four #6-32 tapped holes to the topmost face using the **Hole Wizard**. Switch to a Top View (Ctrl+5 or Mouse Gestures) for visibility, and select the "Hole Wizard" icon.



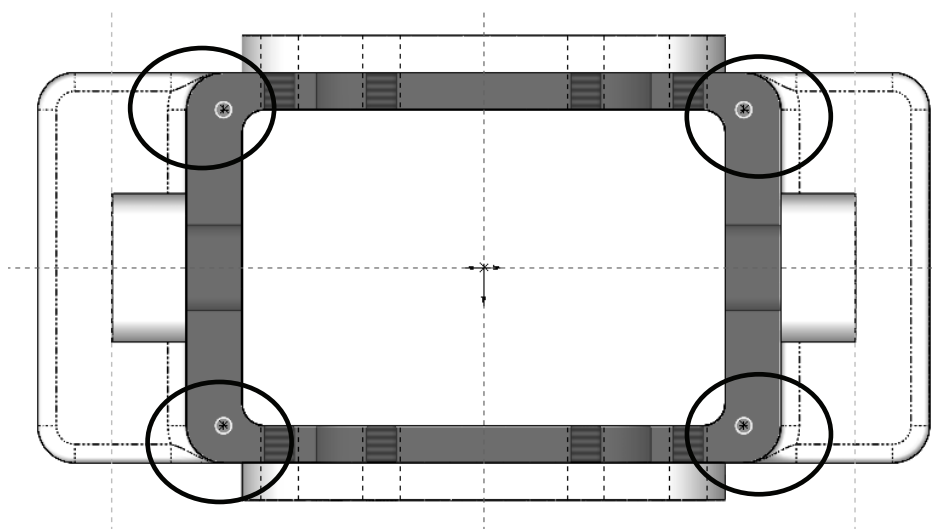
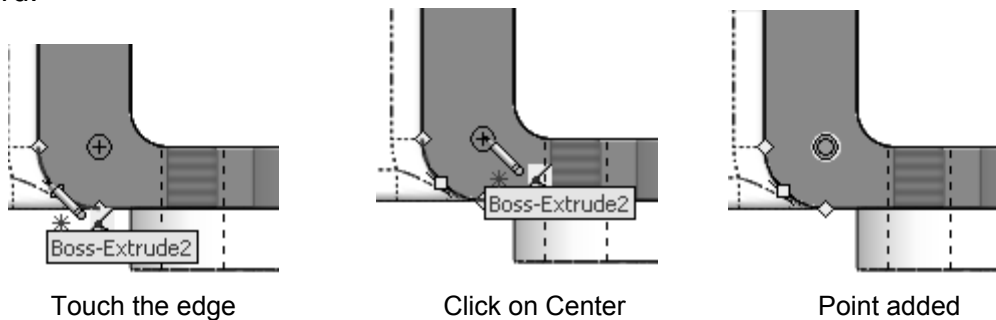
57. - In the Hole Wizard's Property Manager, select the "Tap" Hole Specification icon, and select the options shown for a #6-32 Tapped Hole. The "Blind" condition tells SolidWorks to make the hole an exact depth.

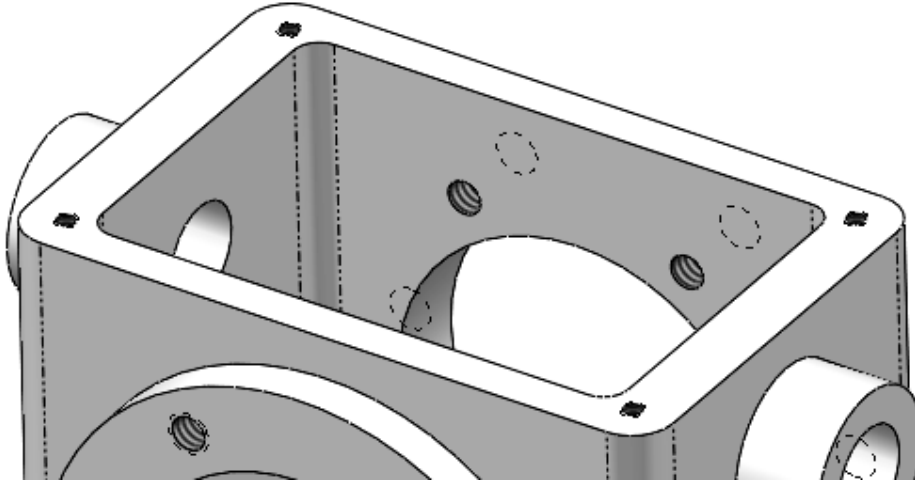


58. - Click in the “Positions” tab to define the hole’s location. Select the top face where we need to add the holes, notice that when we select the face a “**Sketch Point**” is added, and now we are editing a sketch. Notice the Sketch Toolbar is automatically activated and the “Point” tool is pre-selected.

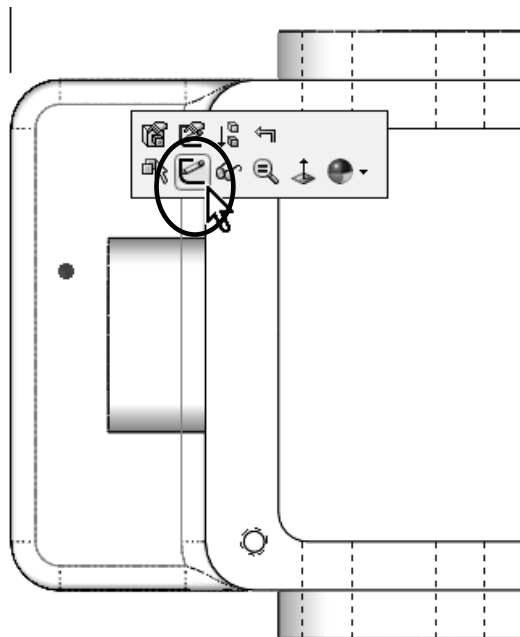


59. - With the “**Point**” tool active, touch each of the round corner edges to reveal their centers. Then click in their centers to add a point in each one; this way we’ll make the points concentric to each corner fillet’s center. If you have a point where we first selected the face, turn off the Point tool, select the point and delete it. Your model should look like the next image. Click OK to finish the Hole Wizard.

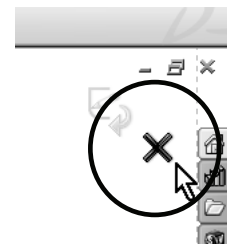




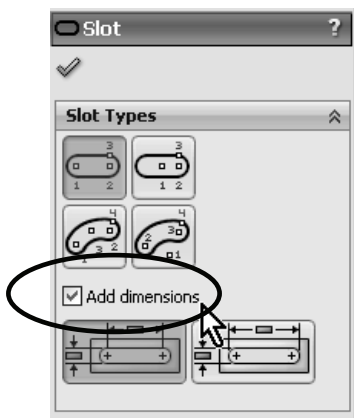
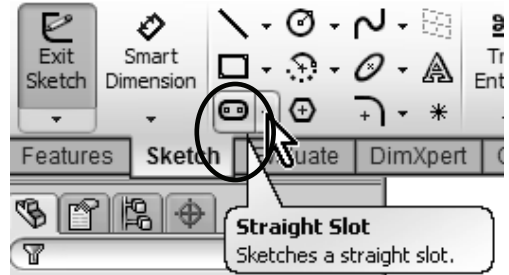
60. - We are now ready to make the slots at the base of the housing. For this task it will be easier to switch to a “Top View”. To add a new sketch, we can select the “Sketch” command and click on the selected face as before, but in this case we’ll learn how to use the pop up toolbar. Simply select the face, and from the pop up toolbar, select “Sketch”. Notice there are two similar icons, the one on top is “**Edit Sketch**” to modify the sketch of the feature we selected, and the one below is “**Sketch**” to create a new sketch on the selected flat face.



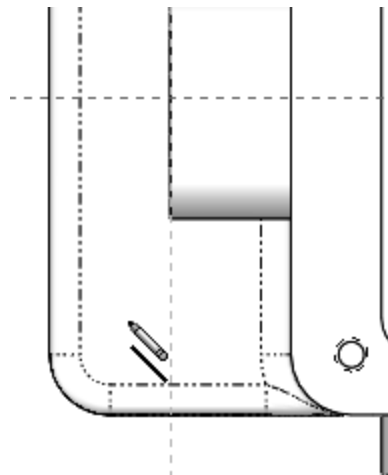
If “Edit Sketch” is selected instead of “Sketch”, simply click on the red “X” in the transparent confirmation corner in the upper right corner of the graphics area to cancel any changes made to the sketch and go back to editing the model.



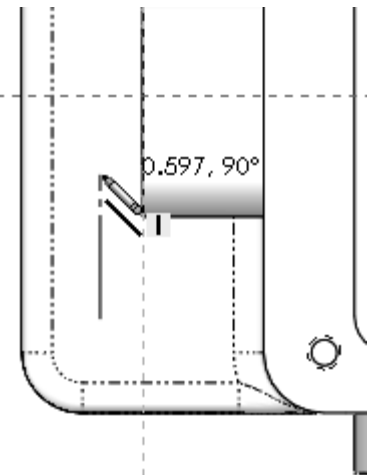
To make the slot, we'll use the **"Straight Slot"** command from the "Sketch" tab in the Command Manager. This tool will create a slot by first drawing the centerline and then defining the width of the slot. Select the **"Straight Slot"** icon and activate the "Add Dimensions" option, it will automatically add dimensions when we finish. The process is click to locate the center of one arc, click to locate the second, and click a third time to define the width. When finished, double click the dimensions to change them and make the slot 0.375" long and 0.250" wide.



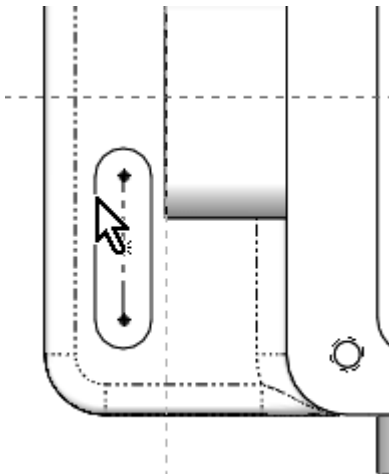
Add dimension option



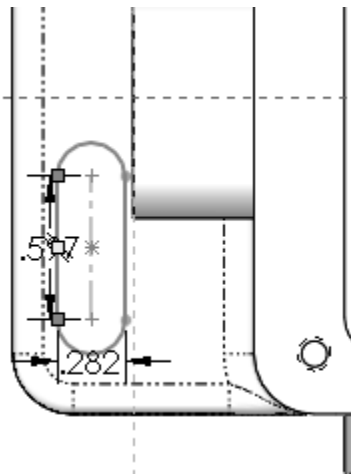
Locate first center



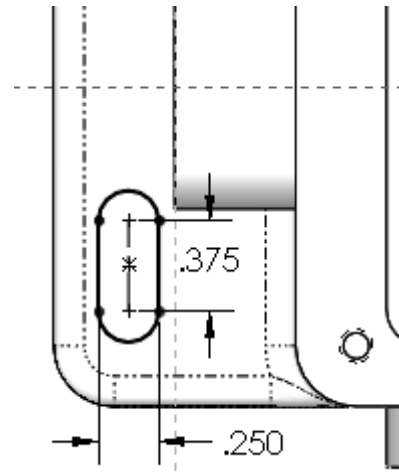
Locate second Center



Define slot width



Automatic dimensions



Corrected dimensions



The "Slot" command has more options, including arc slots and overall slot length dimension.

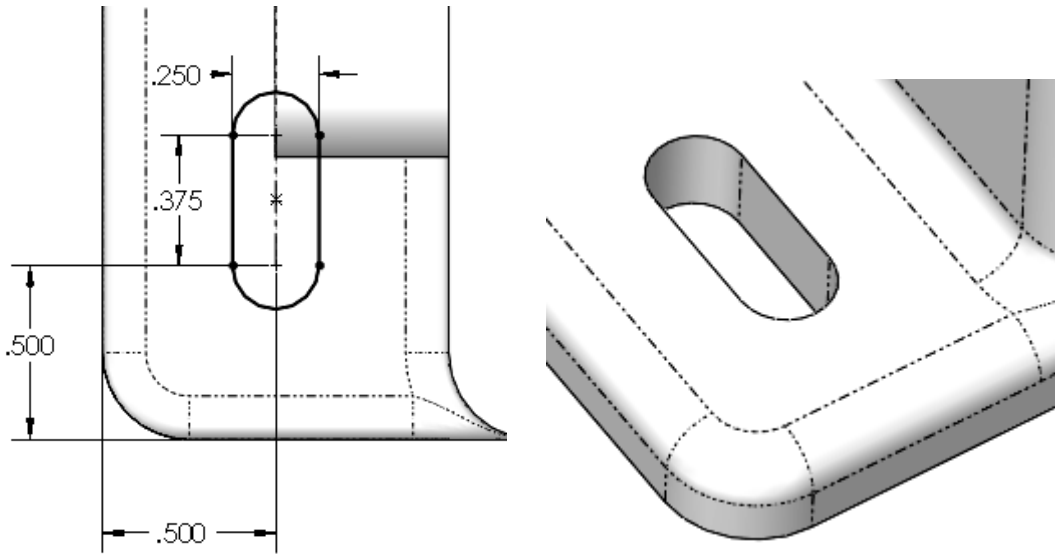


To enable auto dimension while adding sketch elements select the menu **“Tools, Options”**. In the **“System Options”** tab select the **“Sketch”** section and activate:

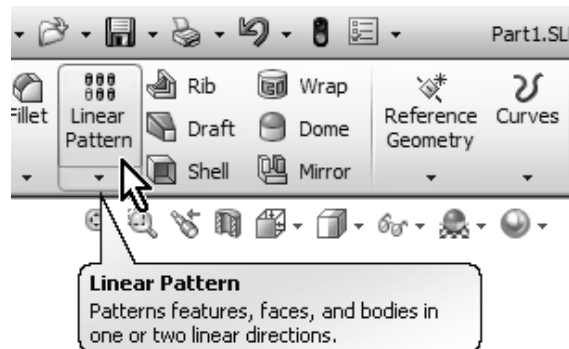
Enable on screen numeric input on entity creation

This way when any sketch entity is created it will be automatically dimensioned and works with lines, rectangles, circles and arcs.

After the slot is defined, locate the bottom of the slot by adding two 0.5” dimensions from the lower and left edges of the base as shown. Finish the slot by making an **“Extruded Cut”** using the **“Through All”** option. Rename this feature **“Slot”**.



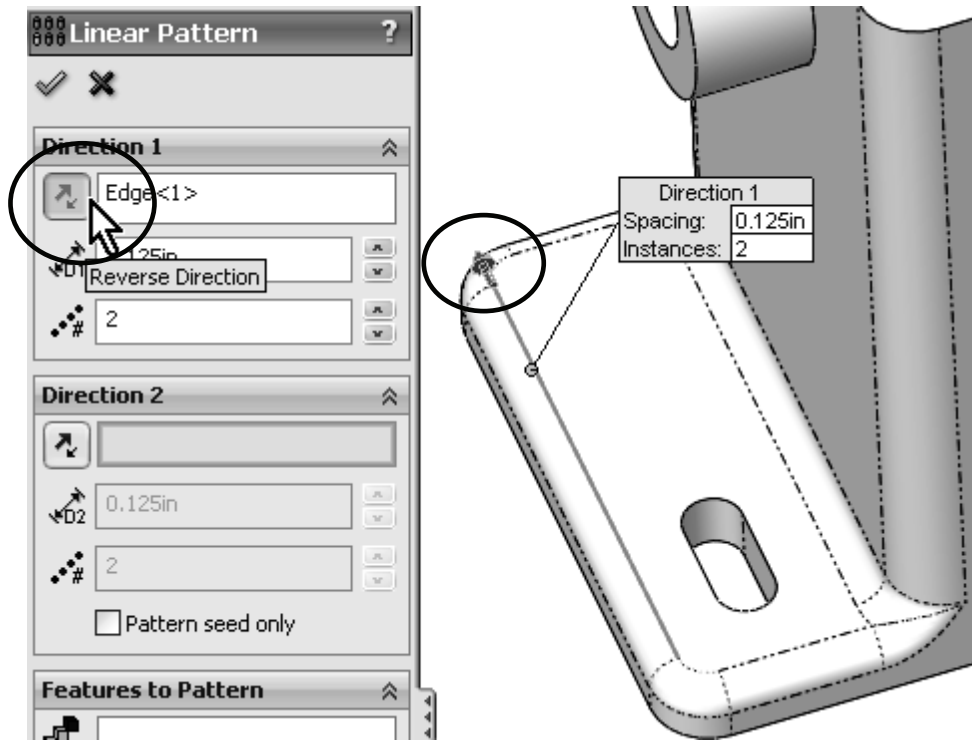
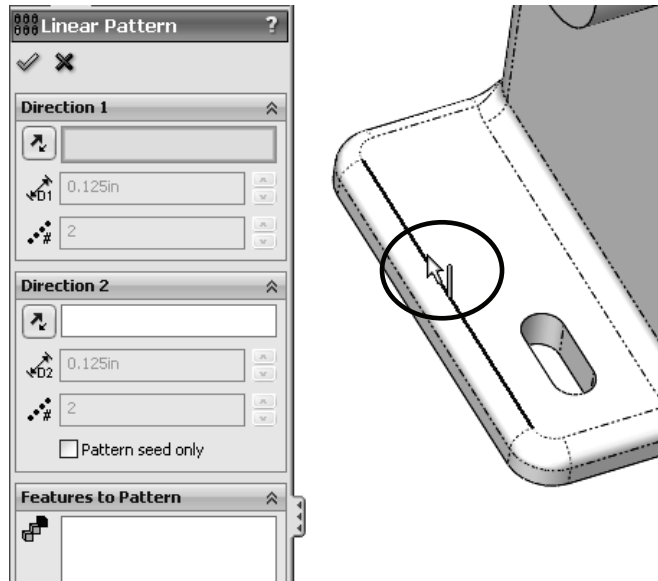
61. - We will now create a **“Linear Pattern”** of the slot. A linear pattern allows us to make copies of one or more features along one or two directions (usually along model edges). Select the **“Linear Pattern”** icon from the **“Features”** tab in the Command Manager.



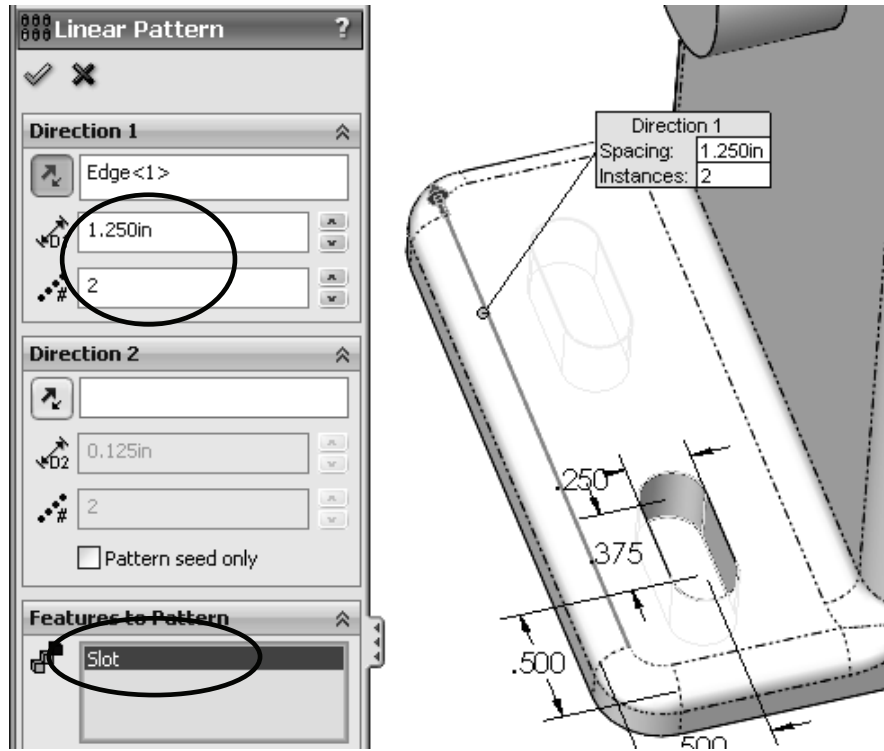
Note: Using the Mirror command keeps the design intent better, but we chose to show the user how to use the Linear Pattern function instead.

62. - In the Linear Pattern's Property Manager, the "Direction 1" selection box is active; select the edge indicated for the direction of the copies. The copies will follow this direction. Any linear edge can be used as long as it is in the desired direction of the pattern.

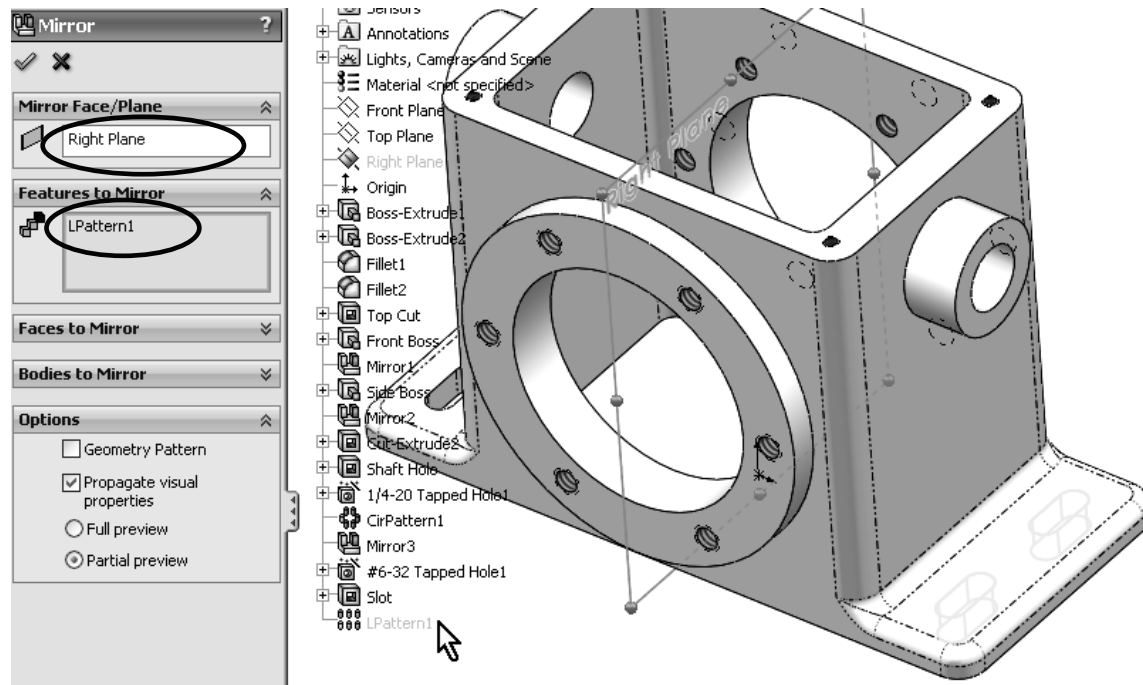
Once the edge is selected, a grey arrow indicates the direction that the copies will be made. If the Direction Arrow in the graphics area is pointing in the wrong direction, click on the **"Reverse Direction"** button next to the "Direction 1" selection box.



63. - Now click inside the "Features to Pattern" selection box to activate it and select the slot feature either from the fly-out Feature Manager or the graphics area. Change the spacing between the copies to 1.25" and total copies to 2. This value includes the original just like in the Circular Pattern. Click OK to finish the command.



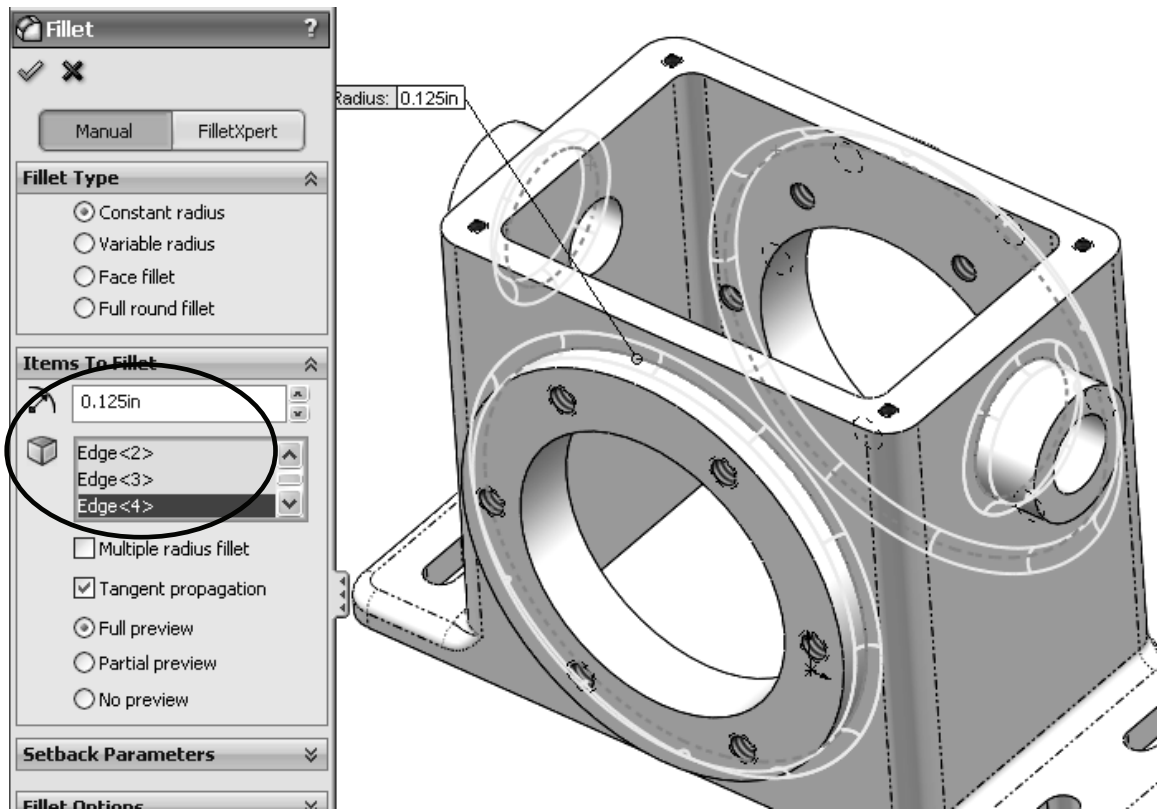
64. - We need the slots on both sides, so we'll copy the previous linear pattern to the other side of the Housing using the **"Mirror"** command about the Right Plane. Click on the "Mirror" icon in the Features tab of the Command Manager; select the "Right Plane" as the mirror plane and the Linear Pattern in the "Features to Mirror" selection box to copy the slots.

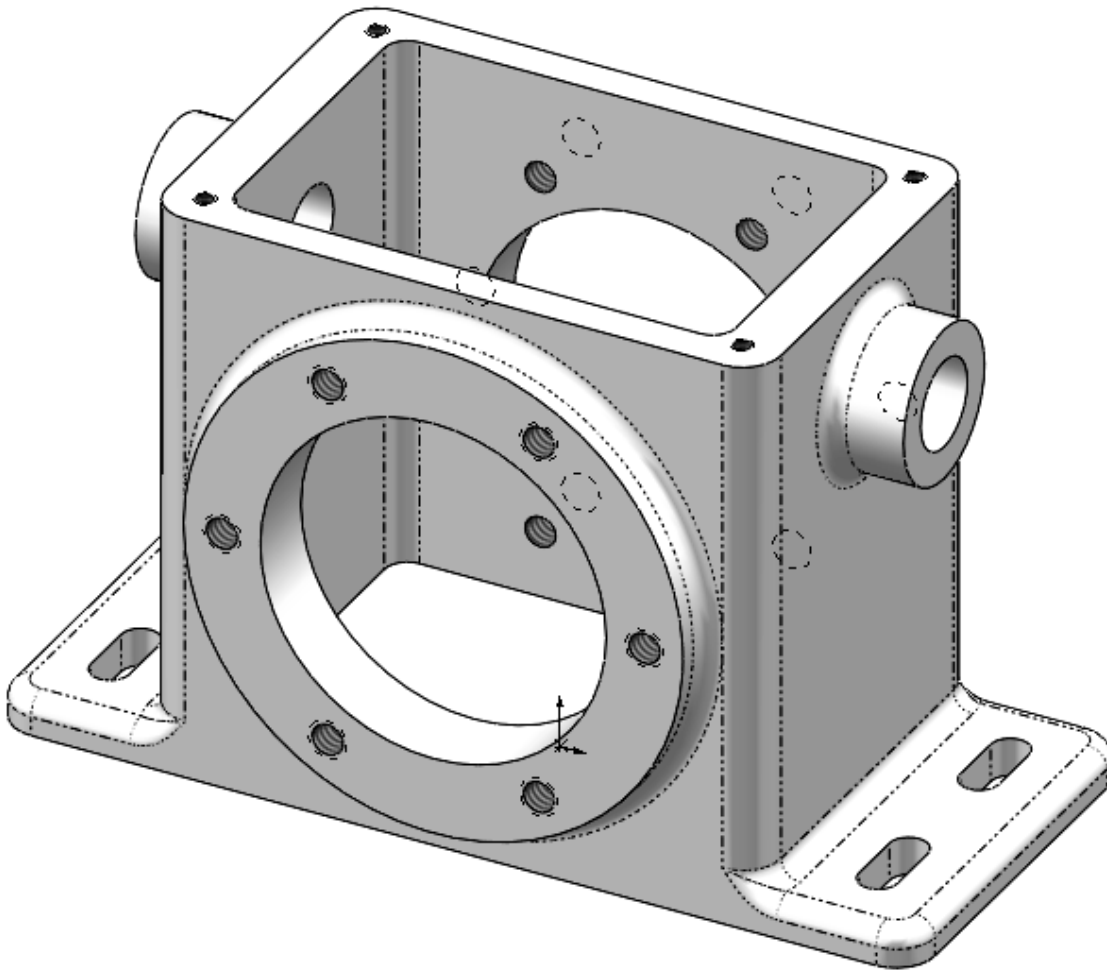




Selecting the Linear Pattern feature for the mirror also includes the “seed” feature (Slot).

65. - Using the “**Fillet**” command from the “Features” tab, add a 0.125” radius fillet to the edges indicated as a finishing touch. Rotate the model using the middle mouse button and/or change the display style to “Hidden Lines Visible” mode to make selection easier.





66. - Now that the model is finished, we can easily determine its physical properties, such as **Weight, Volume, Center of Mass and Moments of Inertia**. SolidWorks includes a built in materials library with many different metals and alloys, plastics, woods, composite materials and others like air, glass and water.

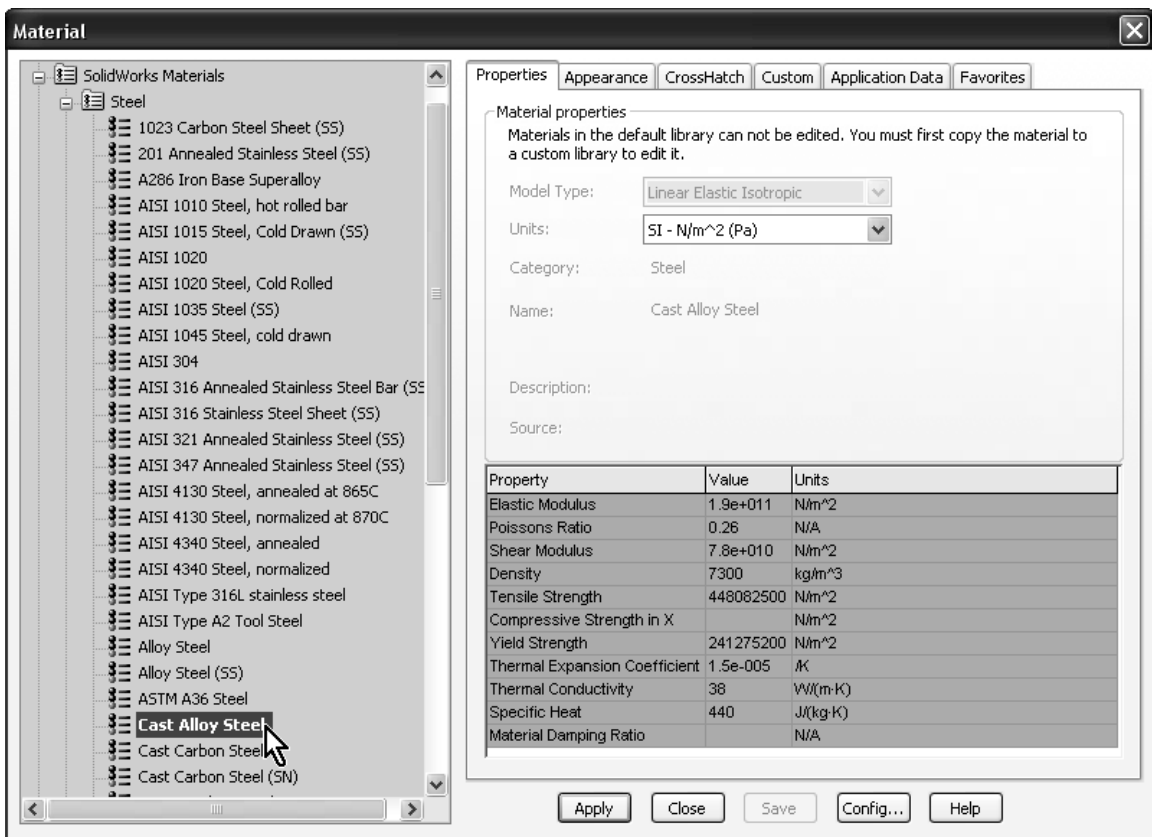
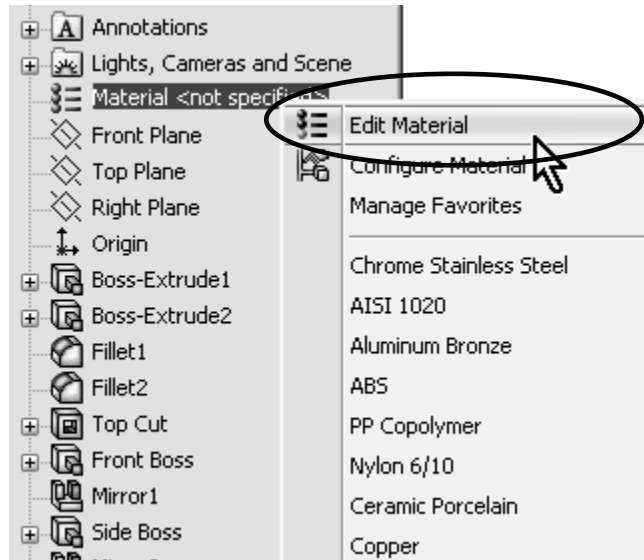
The library includes mechanical and thermal properties such as:

- Mass density
- Elastic and Shear modulus
- Tensile, Compressive and Yield strengths
- Poisson's ratio
- Thermal expansion coefficient
- Thermal conductivity
- Specific heat

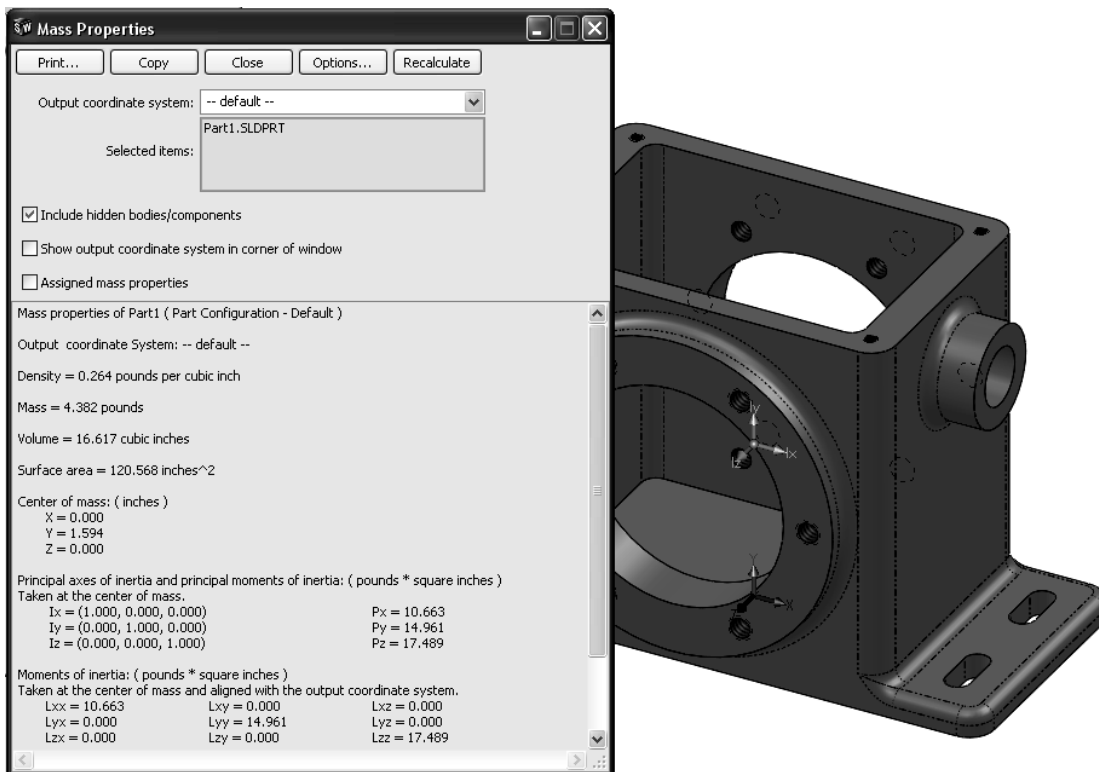
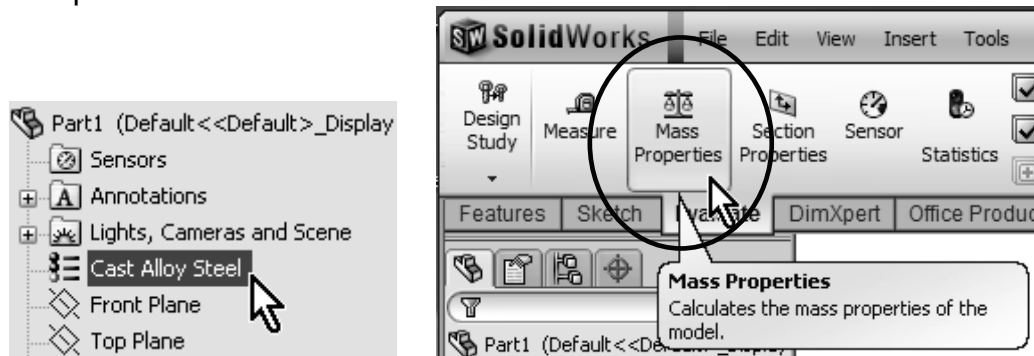


These properties are used by SolidWorks to determine a part's weight, or determine with SimulationXpress, the built in structural analysis software, if a component will fail under a given set of loading conditions.

To assign a material to a component, Right-Mouse-Click in the “Material” icon at the top of the Feature Manager, and either select “Edit Material” or pick one of the materials listed. This list can be changed in the Favorites tab in the Materials library. For this part select “Cast Alloy Steel” from the “Steel” library. Click on “Apply” to accept the material and Close the library.



Now the Feature Manager reads “Cast Alloy Steel” instead of “Material”. Select “**Mass Properties**” from the “Evaluate” tab in the Command Manager. We’ll see the Density (provided by the material selection), Mass (Calculated from the volume and density), Volume, Surface Area and Center of Mass coordinates relative to the origin (also indicated by a magenta triad in the graphics area), Principal Axes of inertia and Moments of inertia about the Center of Mass and the part’s origin all listed in a new window, where we can copy the text for later use in reports.

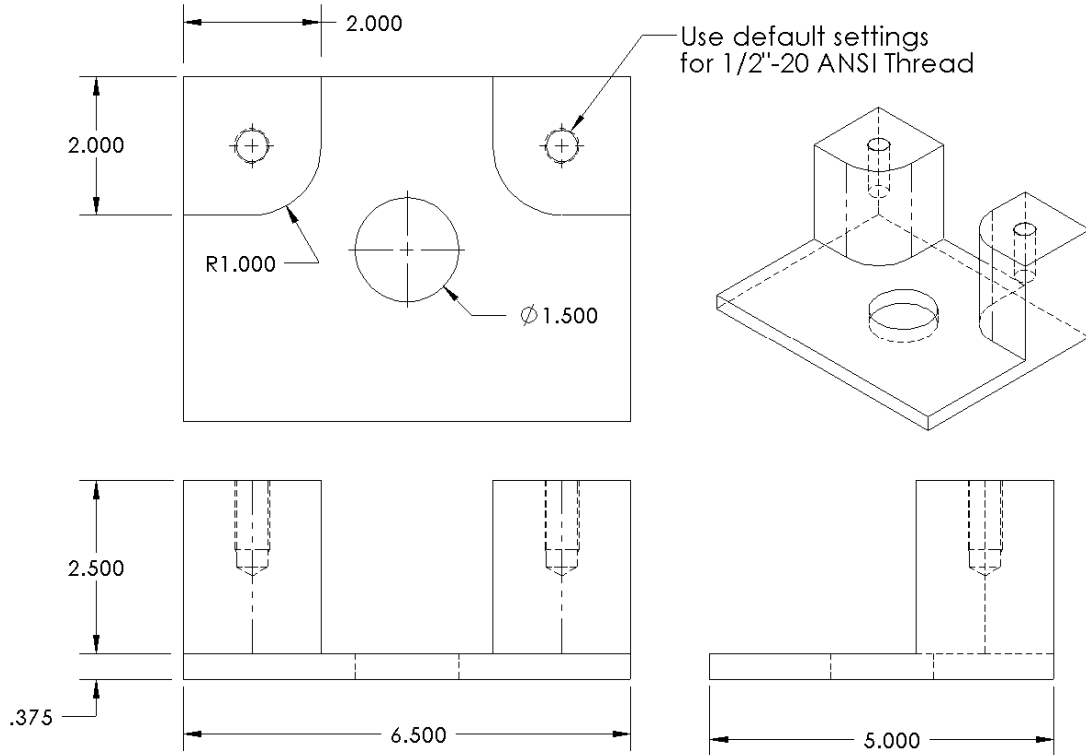


Mass properties are referenced to the origin by default, but they can also be referenced to a user defined coordinate system by selecting one from the “Output coordinate system” drop down list. In the “Options” button we can change the cubic units in which we want the results to be displayed. The default option uses the document’s units.

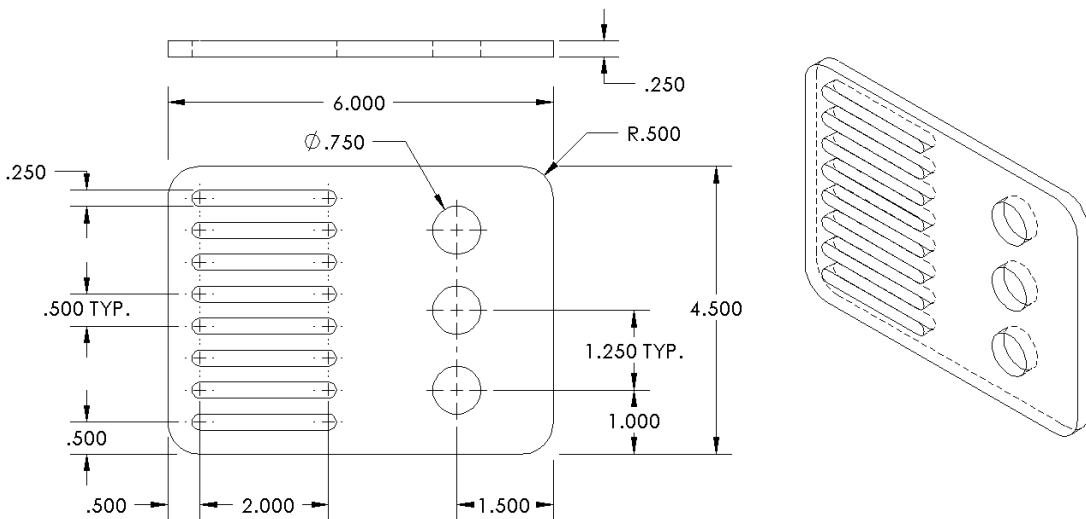
Save the finished part as “Housing” and close the file.

Exercises: Build the following parts using the knowledge acquired in this lesson. Try to use the most efficient method to complete the model.

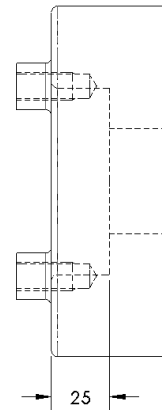
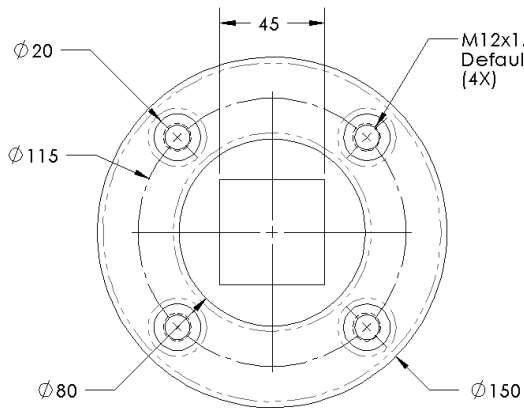
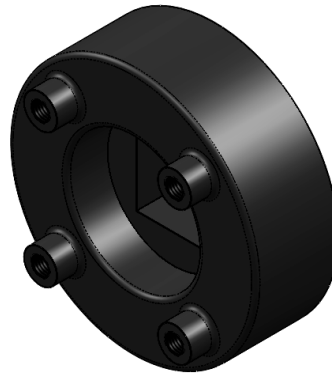
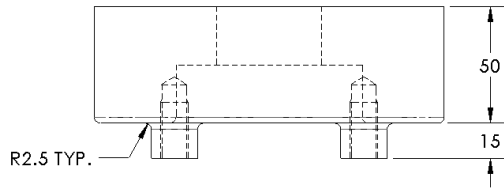
Exercise 1 Units: inches
Material: AISI 1020 Steel



Exercise 2 Units: inches
Material: Al 6061 Alloy



Exercise 3 Units: mm
Material: Copper



Exercise 4 Units: mm
Material: ABS Plastic

