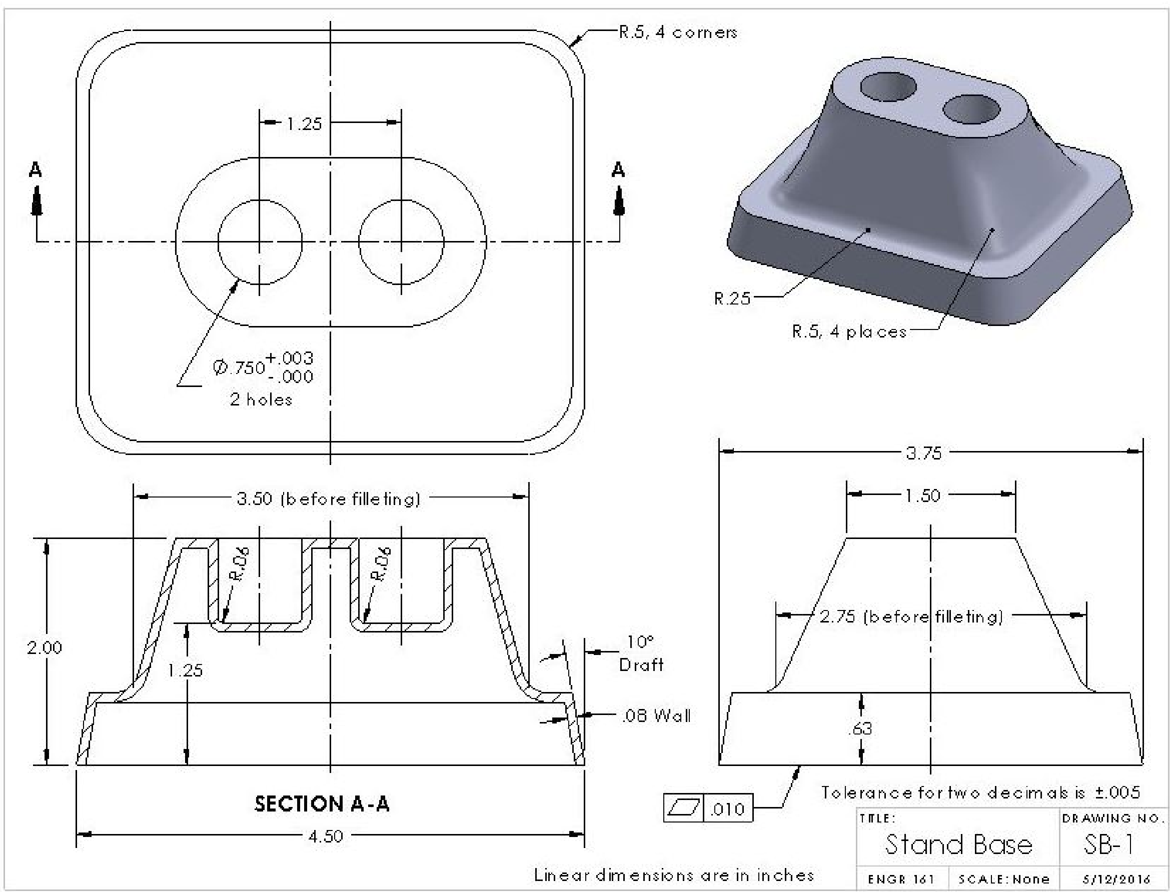
Stand Base

Updated 10/25/2018

Introduction

The basic geometry of some features is such that they cannot be created by extruding a single sketch using the **Extruded Boss/Base** tool. This is the case when the shape of the feature changes from one end to the other. To create such features in [1]**SolidWorks®**, a different sketch is made on each of a series of two or more stacked planes and then they are blended together to create the 3D feature using the **Lofted Boss/Base** tool from the Features group. Consider the *Stand Base* below. The bottom base portion can be formed by extruding a rectangle upward with a draft angle using **Extruded Boss/Base** (corner rounds are added later). However, the top portion starts out rectangular but ends up with a different shape on the very top. Using **Lofted Boss/Base** with two separate sketches is a good way to create features such as this.

Stand Base Drawing



Start a new Part and be sure that Units are set to **IPS**. Set Length decimal places to either **2** or **3** and Angle decimal places anywhere between **None** and **3**.

[1]**SolidWorks** is a registered trademark of Dessault Systémes SolidWorks Corporation. Screen shots from SolidWorks are used throughout this tutorial.

Bottom Base

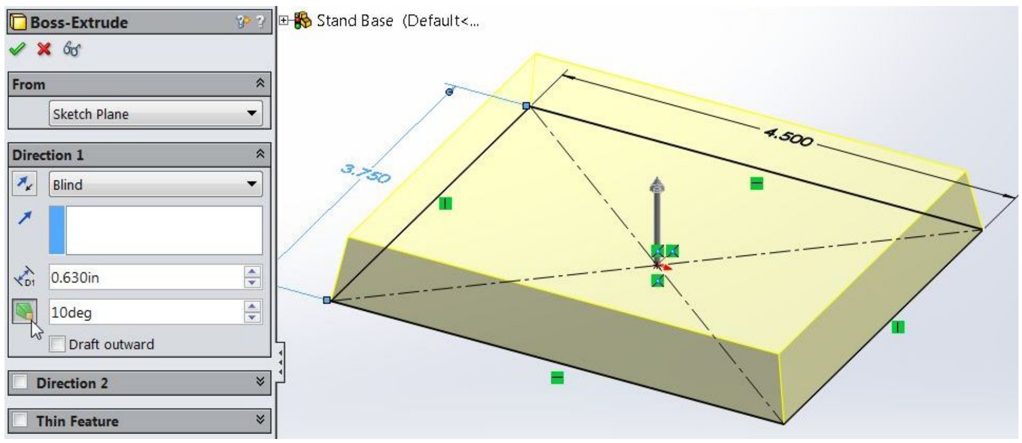
Select the **Top Plane** for the first sketch of the model. Drag a **Center Rectangle** out from the origin.

Add dimensions using **Smart Dimension**, making the rectangle **4.50** long and **3.75** high, as shown.

Click on **Extruded Boss/Base** from the Features group to open its Property Manager.

Set **D1** for a blind depth of **.63**.

Click on the **Draft On/Off** button to toggle it on.



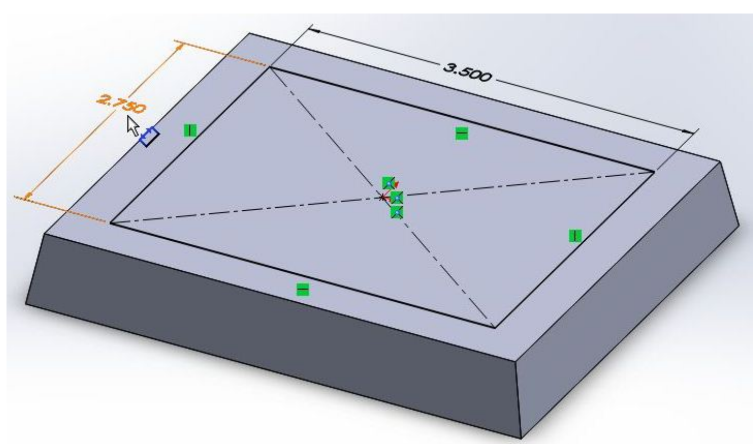
To the right of the draft angle window are up and down arrows. Click on the <u>up</u> arrow to index the angle up to **10 degrees** (or just type in the angle value). Click on the green **OK** check mark to accept the parameters.

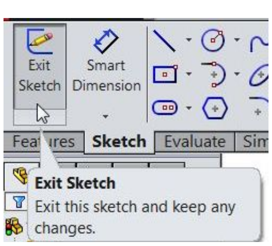
Draft angles are often applied to parts that are to be cast in a mold because perpendicular sides make it difficult to remove the pattern or part without damaging the mold.

Loft

The Lofted Boss/Base feature that sits on top of the bottom base requires **two sketches**, each on a separate surface or plane. The first sketch will be a **rectangle** sitting directly on the top of the existing base.

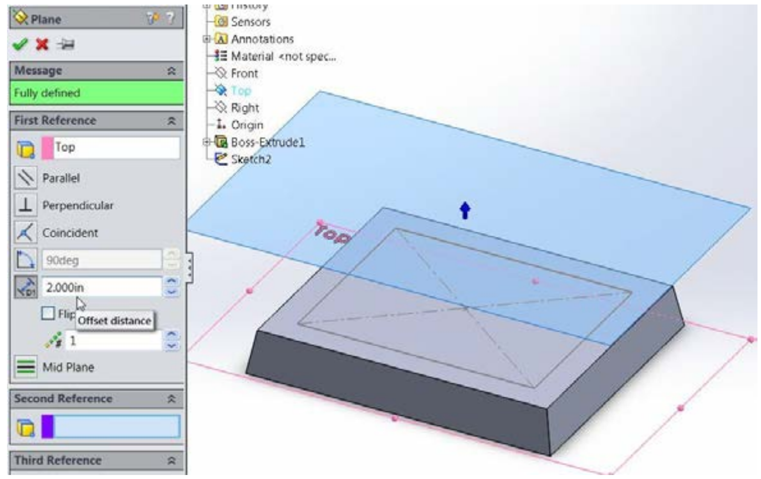
Click on the **top surface** of the newly-created base and sketch a **Center Rectangle** on it, centered at the sketch origin. Dimension it **3.50** wide and **2.75** high.





Exit this sketch. It is important to **Exit** the current sketch because the upcoming **Reference Geometry** tool is not available if a sketch is still in edit mode.

A **new plane** is needed for the second sketch of the loft. From the Features group, click on the drop down arrow under **Reference Geometry** and select **Plane**.

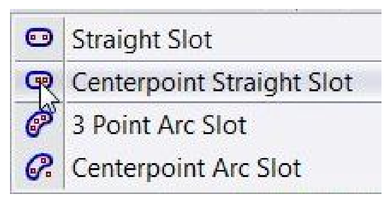


The Design Tree will have moved into the Graphics Area to make room for the **Plane** Property Manager. Click on the ▸ symbol by the Design Tree to expand it, if needed, and then select the **Top** plane from the Design Tree list as the **First Reference**. (Be careful to note that the reference is the **Top Plane**, not the **top surface** of the base.)

For the **Offset distance**, enter **2**. Click the **OK** check mark to accept.

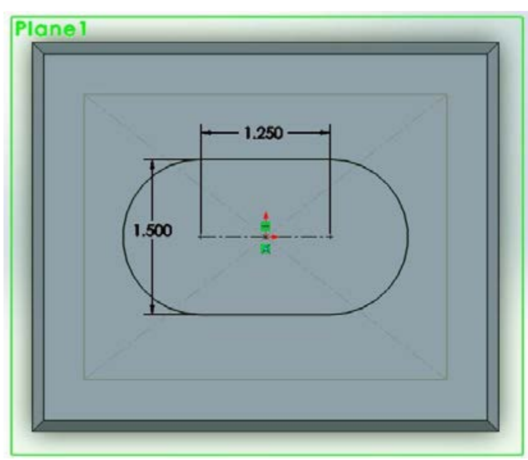
Click on the new plane to select it as a sketch plane and switch to **Top** view.

From the Sketch group, select the **Centerpoint Straight Slot** tool. Notice that there are several choices of **Slot** tools from a **drop-down list**, so make sure you select the correct one.



In the **Slot** Property Manager, make sure that the ☑ **Add dimensions** box is checked.

To create the shape that this slot tool makes, click on the sketch origin and drag horizontally to the right or left, not going past the edges of the base. You will notice a horizontal centerline being extended out to the right and left as you drag out the length of the slot. Release the mouse button, and then move up some (to add width to the slot) and click. The as-sketched length and width dimensions will be automatically added.

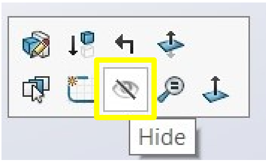


Press **Esc** to cancel the slot tool. Double-click on each dimension and edit them to these values: <u>Length</u> (distance between arc centers) is **1.25** and slot <u>width</u> is **1.50**.

Exit the sketch. The next step is performing the **Loft** operation which <u>will not work</u> if a sketch is still open.

The **Lofted Boss/Base** Feature tool uses sketches on two or more planes and blends them together with a smooth 3D transition. If more shape control is needed in the transition region, then a **Guide Curve** can be used (the loft on this *Stand Base* is a simple one and does not need a guide curve).

Switch to **Trimetric** view.

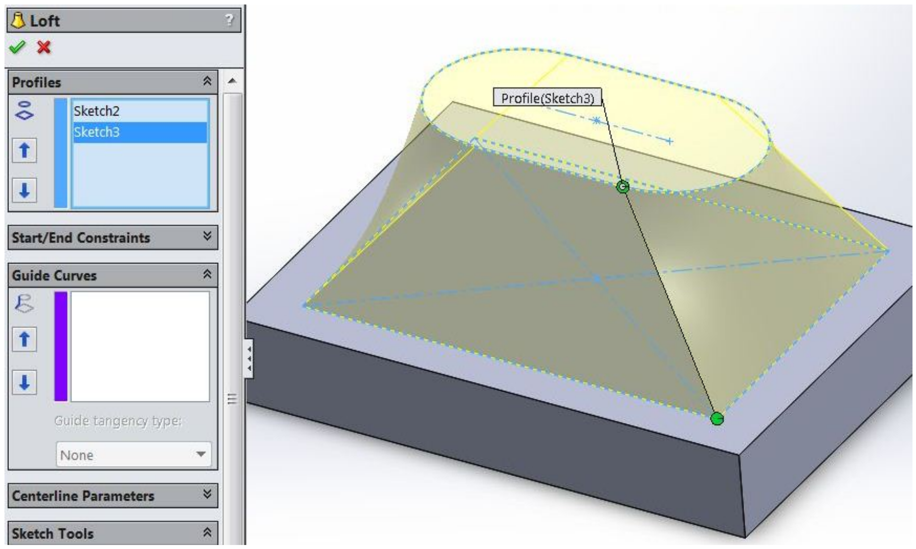


Click on the reference plane in the Graphics Area and select the **Hide/Show** icon (the eyeball) from the pop-up menu, to hide the plane from view.

From the Features group, select the **Lofted Boss/Base** tool.

In the **Loft** Property Manager, the **Profiles** window should be highlighted in blue which means that it awaits your input.

Click on a corner of the loft's bottom sketch (the rectangle) and then move up above that point and click on the sketch that you made on the reference plane. As you click on each of the two sketches you should see them listed in the **Profiles** window.



Note that the locations where you click on the sketches should be roughly in line with each other along the loft. If you were to pick points on opposite sides, the loft would twist. In any case, the loft preview shows you what it will look like and you can change selection points if needed.

If the preview looks okay, click on the **OK** check mark to complete the **Lofted Bass/Base**.

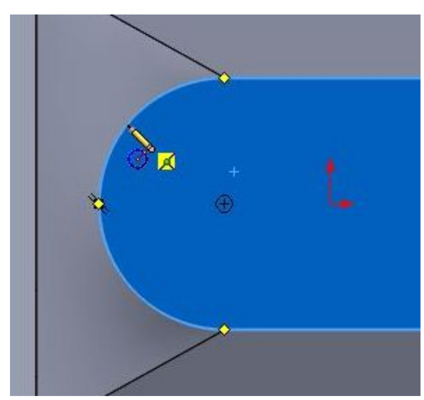
(If the loft looks **too tall**, go back and check your **Reference Plane** and see if maybe the offset distance is wrong or if you accidentally selected the **top surface** of the base as the First Reference, instead of the **Top Plane**.)

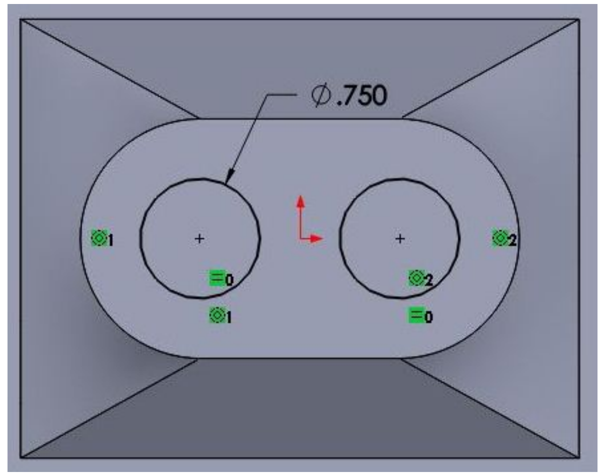
Pockets

For the pair of circular pockets, click on the very **top surface** of the loft to select it to sketch on.

Switch to **Normal To** (or **Top**) view.

Select the **Circle** Sketch tool and move the cursor over one of the arcs to cause the arc's center point to appear, allowing you to locate it. Snap to the center point, click and drag out a circle somewhat smaller than the arc. Use the same method to sketch a similar circle at the center of the other arc.





Add a **Smart Dimension** of **.75** for the diameter of <u>one</u> circle.

Press **Esc** to quit **Smart Dimension**.

Click on one circle then hold down **Ctrl** and click on the other circle, so that both circles are selected. Add the **Equal** relation.

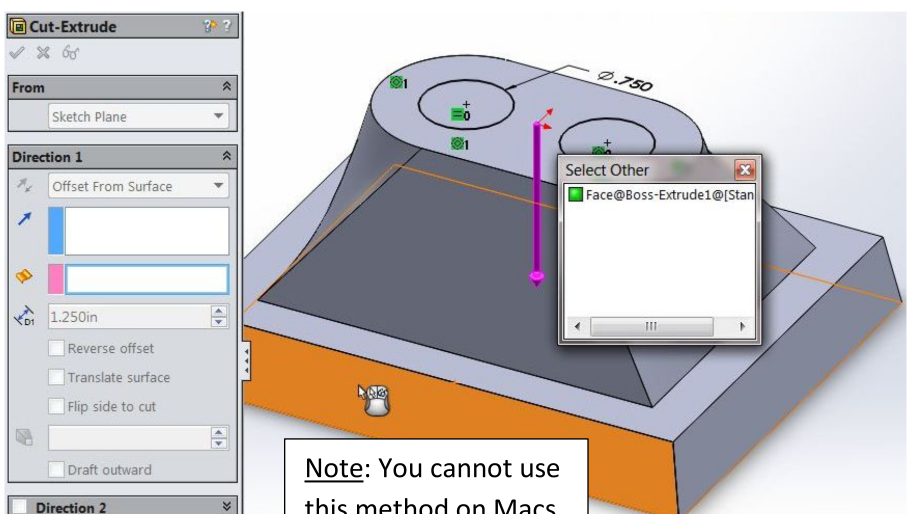
Next, click on one circle then hold down **Ctrl** and click on the arc that surrounds it and then add the **Concentric** relation. Do the same for the other circle and arc. The sketch should now be fully defined.

(Mac users must use **Add Relation** under **Display/Delete Relations** in the Sketch tool group.)

Switch to **Trimetric** view for better visibility. Select the **Extruded Cut** tool from the Features group.

In the **Cut-Extrude** Property Manager under Direction 1, select **Offset From Surface** as the end condition. This is a departure from the usual method of specifying a blind depth.

On the *Stand Base* drawing, instead of a pocket depth, a dimension of **1.25** is shown between the bottom of the base and the bottom of the pockets. The design intent of this model is for the 1.25 distance to remain the same even if later modifications to the model change the overall height. For example, if the height were increased from 2” to 2.25”, the pockets would get .25” deeper but the offset from the base would remain 1.25”. This is where the **Offset From Surface** option becomes useful because it maintains the desired dimension even if the model is changed later.



<u>Note</u>: You cannot use this method on Macs.

Enter **1.25** as the offset distance D1.

Now you must specify the face or plane that the offset is to be referenced from. Although you could rotate the part to select the bottom face, another method allows you to select the desired face without rotating. Right-click on the part and choose **Select Other** then move the mouse around until you see the **outline** of the *bottom* of the base light up, as shown here. Note that the face that the mouse actually sits on gets highlighted too, but the **outline** of the bottom represents the **other** face that we desire to choose.

Click (left button) to select that bottom face and then click the **OK** check mark to finish the procedure.

Shell

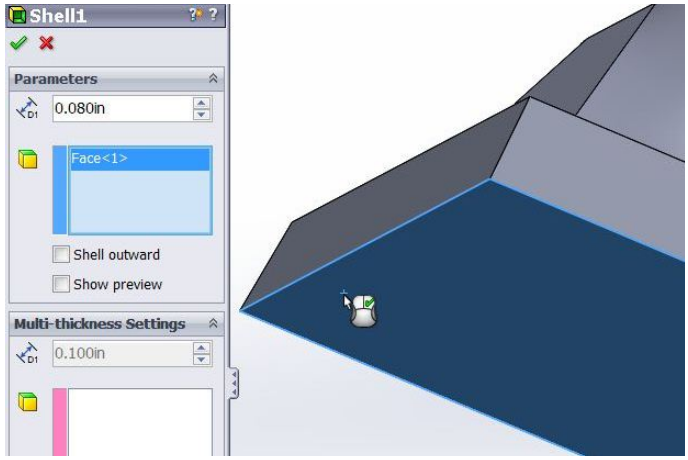
The **Shell** Feature tool offers a convenient way to hollow out a solid part. The usual method for using this tool is to specify a wall thickness and then select a surface to be removed, so that only the remaining outside shell of the part remains. On this *Stand Base*, the **bottom** surface is removed.

From the Features group, select the **Shell** tool.

In the **Shell** Property Manager, enter **.08** for the **Thickness** D1.

Click on the bottom face of the part to list it in the **Faces to Remove** window.

Click the green **OK** check mark to finish.



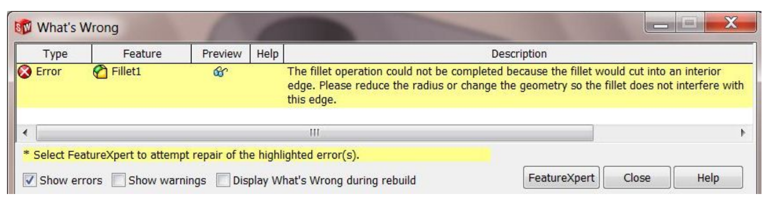
Fillets and Rounds

Notice that several of the edges on the *Stand Base* are rounded.

Use the **Fillet** tool to add the **R.5** rounds to the four corners of the bottom base.

Oops – Error Message:

The radius is too large and would cut through the shell wall, although it would work if the part were solid.



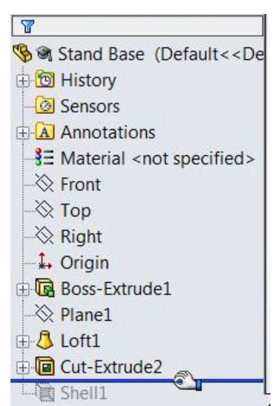
If the rounds had been added **before** the **Shell** operation, then the shell would have followed the rounds along the inside and that would have worked.

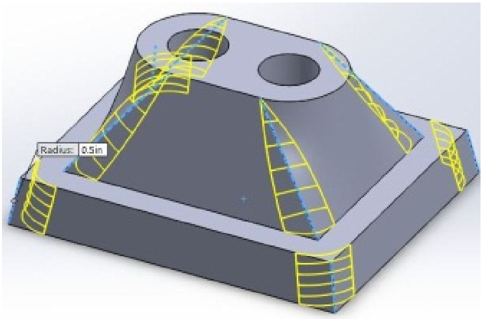
We made this error intentionally so that a convenient feature of **SolidWorks** could be demonstrated: the **Rollback Bar**. Instead of deleting the shell then creating the rounds and re-doing the shell, you can just go backwards in the Design Tree list to the point where you need to be.

Click the **X** in the **Fillet** Property Manager to cancel it.

Now simply click on the line (the Rollback Bar) below **Shell1** in the Design Tree and drag it up **above Shell1**.

You have now backed up in your model creation process to the point just prior to doing the shell.

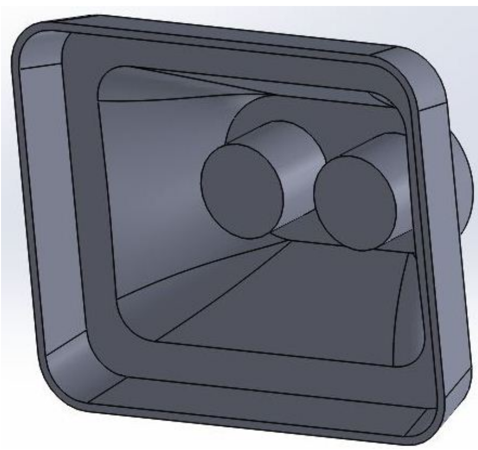


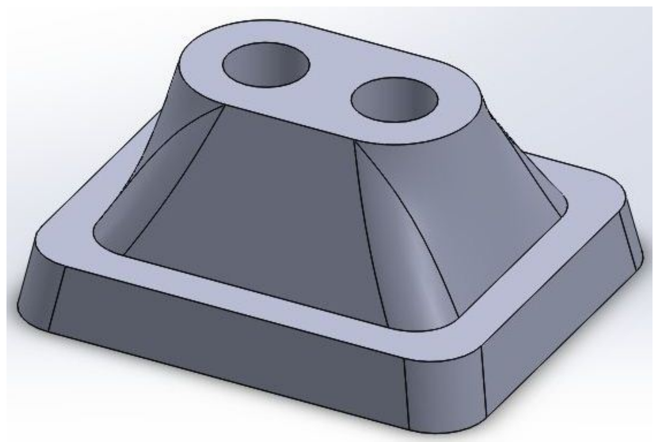


Use **Fillet** now to create the **R.5** rounds at the four base corners **and**, while you're at it, at the four edges of the loft, which are also **R.5**, (eight places total) with **one** use of the **Fillet** tool.

Return the Rollback Bar down to the end of the Design Tree list.

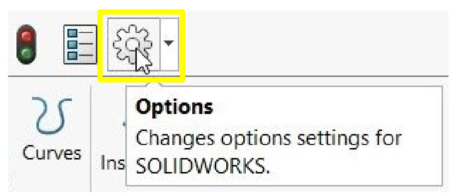
Rotate the part to see how the shell follows the contour of the rounds on the inside.



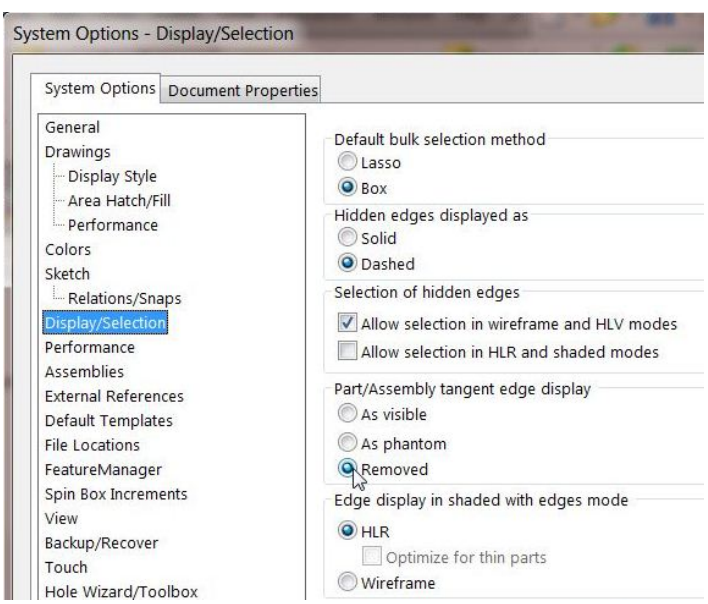


If the **tangent edges** of fillets and rounds are **visible** as lines as in the above illustrations, turn the display of them **off** by using the **Options** tool. (Sometimes the display of tangent edges as solid or phantom lines enhances the understanding of a model's geometry, but in this case they cause unnecessary clutter.)

Click on the **Options** tool in the Menu Bar at the top of the screen.



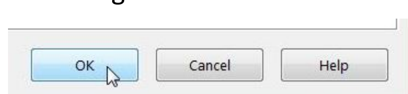
Select the **System Options** tab and click on **Display/Selection** in the list.



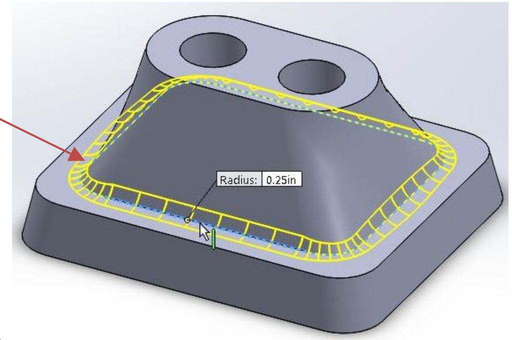
Now look in the right-hand portion of the box and find the **Part/Assembly tangent edge display** section.

Click on the ⦿ **Removed** button.

At the bottom, click on **OK** to close the dialog box.



Add the **R.25** fillet to the edge along the bottom of the loft.



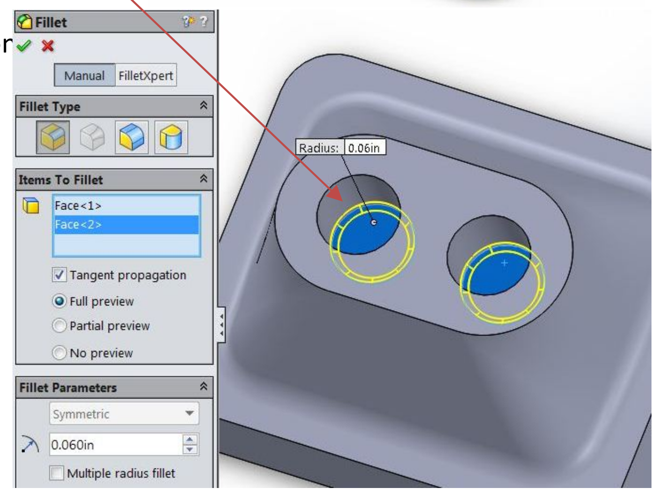
Rotate the view a little bit, about as shown here, so that you can see the edges at the bottom of the pockets.

Select the **Fillet** tool again.

Set the radius value to **0.06** and click on

OK ✓

Switch to **Trimetric** View.



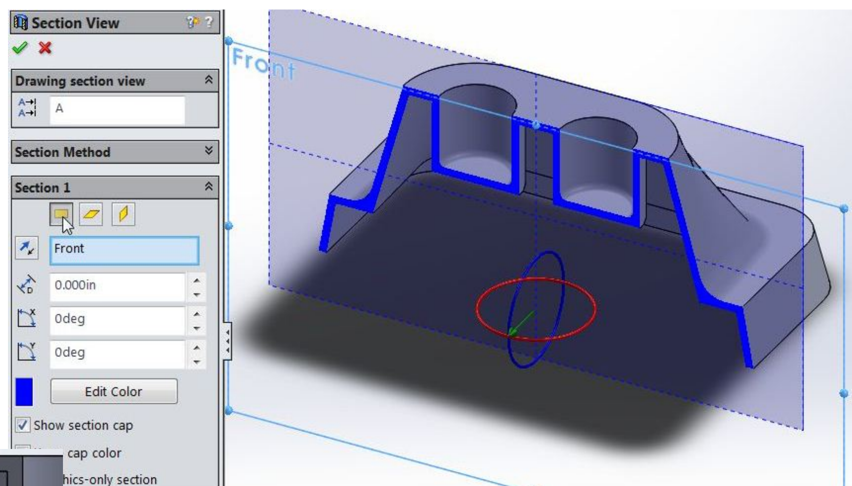
Section View

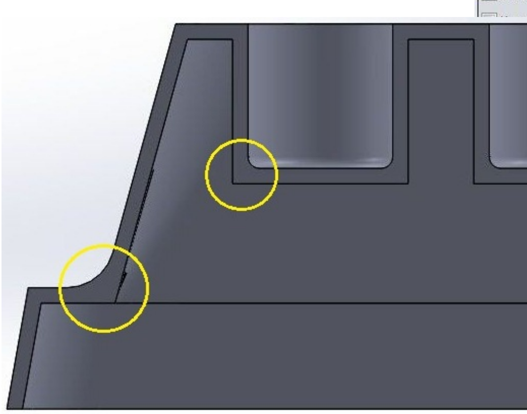
Click on the **Section View** tool in the Heads-Up View toolbar at the top of the Graphics Area.

In the Property Manager there are three buttons for selecting cutting planes for the section. Select the **Front** plane.

Click the **OK** check mark to accept the section and then switch to **Normal To** view.

Zoom to Fit (shortcut: **F** key).

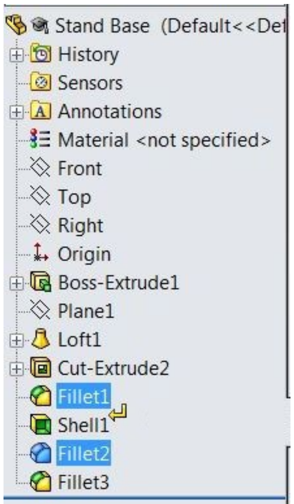




Now that a cross section of the shell is visible, notice that the fillets that were recently added (the ones at the bottom edge of the loft and at the bottoms of the pockets), are not followed by the shell contour on the inside surface of the model. Sharp corners remain at those locations inside the model.

If these fillets had been created **before** the shell, then the shell would have followed their shapes.

Again, this tutorial purposely proceeded in the wrong order so that we could demonstrate another useful **SolidWorks** technique: **rearranging** items in the Design Tree.

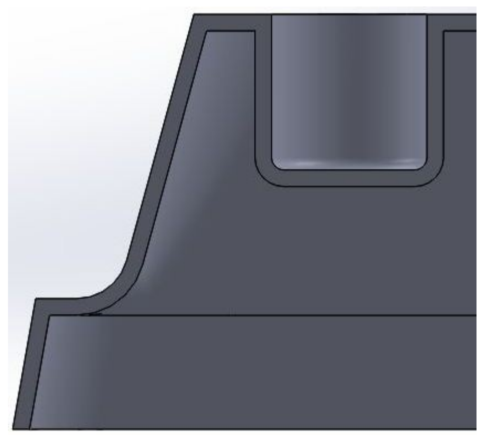


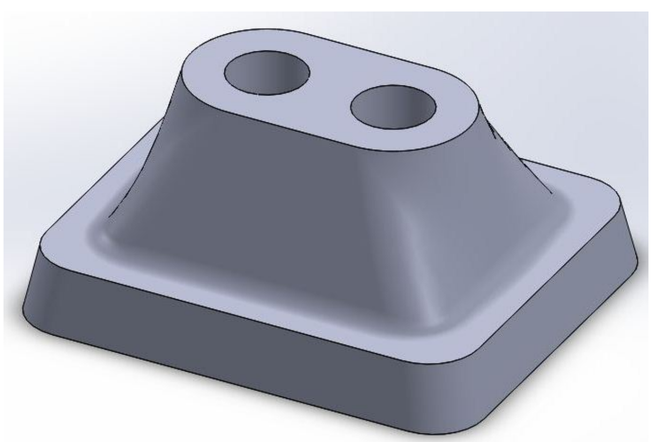
It is not necessary to use the rollback bar or to re-do the shell or fillets to correct the situation.

In the Design Tree, first simply click and drag **Fillet2** up above **Shell1** to relocate it to that spot in the list. A bent arrow appears as a location pointer.

Repeat the procedure to relocate **Fillet3** above **Shell1** in the list.

Notice that the geometry of the model automatically updates to match the revised sequence in the Design Tree, and the inner shell surface now follows the contours of the outer surface.





Click on the **Section View** tool to toggle it off and switch to **Trimetric** View for the final display of the model.

End of Stand Base Tutorial