Lesson 17 Shell, Reorder, and Insert Mode

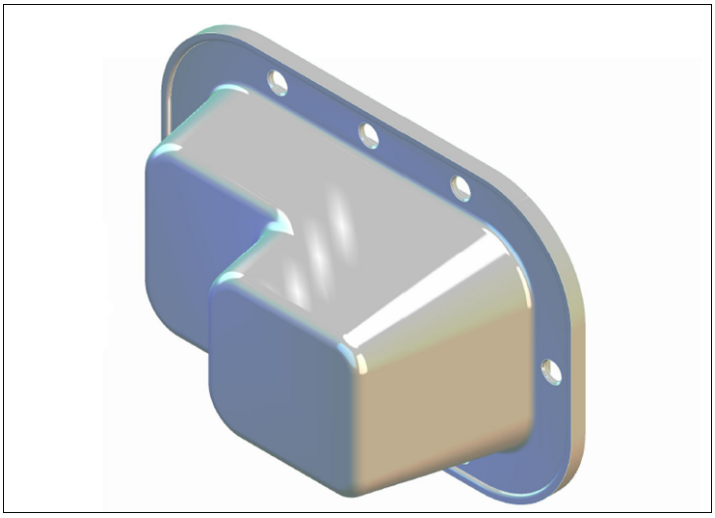


Figure 17.1 Oil Sink

OBJECTIVES

- Master the use of the **Shell Tool**
- Reorder features
- **Insert** a feature at a specific point in the design order
- Create a **Hole Pattern** using a **Table**
- **Render** the part using new **lights**
- Create a **3D PDF**
- **Detail** the part

REFERENCES AND RESOURCES

For **Resources** go to **www.cad-resources.com** > click on the PTC Creo Parametric 3.0 Book cover

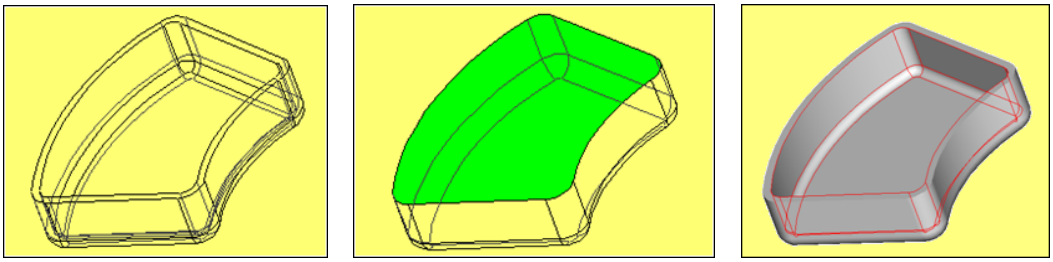
- Lesson Lecture
- Book Projects PDF
- Project Lectures
- Quick Reference Card
- Configuration Options

SHELL, REORDER, AND INSERT MODE

The **Shell Tool** removes a surface or surfaces from the solid and then hollows out the inside of the solid, leaving a shell of a specified wall thickness, as in the Oil Sink (Fig. 17.1). When Creo Parametric makes the shell, all the features that were added to the solid before you chose the Shell Tool are hollowed out. Therefore, the *order of feature creation* is very important when you use the Shell Tool. You can alter the feature creation order by using the **Reorder** option. Another method of placing a feature at a specific place in the feature/design creation order is to use the **Insert Mode** option.

Creating Shells

The Shell Tool [Figs. 17.2(a-c)] enables you to remove a surface or surfaces from the solid, then hollows out the inside of the solid, leaving a shell of a specified wall thickness. If you flip the thickness side by entering a negative value, dragging a handle, or using the  **Change thickness direction** icon, the shell thickness is added to the outside of the part. If you do not select a surface to remove, a "closed" shell is created, with the whole inside of the part hollowed out and no access to the inside. In this case, you can add the necessary cuts or holes to achieve the proper geometry at a later time.



Figures 17.2(a-c) Shell

When defining a shell, you can also select surfaces where you want to assign a different thickness. You can specify independent thickness values for each such surface. However, you cannot enter negative thickness values, or flip the thickness side, for these surfaces. The thickness side is determined by the default thickness of the shell. When Creo Parametric 3.0 makes the shell, all the features that were added to the solid before you started the Shell Tool are hollowed out. Therefore, the order of feature creation is very important when you use the Shell Tool. To access the Shell Tool, click  icon in the **Model** tab Engineering Group. The Thickness box lets you change the value for the default shell thickness. You can type the new value, or select a recently used value from the drop-down list.

In the graphics window, you can use the shortcut menu (press **RMB**) to access the following options:

- **Removed Surfaces** Activates the collector of surfaces. You can select any number of surfaces
- **Non Default Thickness** Activates the collector of surfaces with a different thickness
- **Exclude Surfaces** Activates the collector of excluded surfaces
- **Clear** Remove all references from the collector that is currently active
- **Flip** Change the shell side direction

The Shell Dashboard displays the following slide-up/down panels (tabs):

- **References** Contains the collector of references used in the Shell feature
- **Options** Contains the collector of Excluded surfaces
- **Properties** Contains the feature name and an icon to access feature information

The **References** slide-up/down panel (tab) contains the following elements:

- The **Removed surfaces** collector lets you select the surfaces to be removed. If you do not select any surfaces, a "closed" shell is created.
- The **Non-default thickness** collector lets you select surfaces where you want to assign a different thickness. For each surface included in this collector, you can specify an individual thickness value.

The **Properties** panel (tab) contains the Name text box 

, where you can type a custom name for the shell feature, to replace the automatically generated name. It also contains the  icon that you can click to display information about this feature in the Browser.

Reordering Features

You can move features forward or backward in the feature creation (regeneration) order list, thus changing the order in which features are regenerated [Figs. 17.3(a-b)]. You can reorder features in the Model Tree by dragging one or more features to a new location in the feature list. If you try to move a child feature to a higher position than its parent feature, the parent feature moves with the child feature in context, so that the parent/child relationship is maintained.

You can reorder multiple features in one operation, as long as these features appear in *consecutive* order. Feature reorder *cannot* occur under the following conditions:

- **Parents** Cannot be moved so that their regeneration occurs after the regeneration of their children
- **Children** Cannot be moved so that their regeneration occurs before the regeneration of their parents

You can select the features to be reordered by choosing an option:

- **Select** Select features to reorder by picking on the screen and/or from the Model Tree
- **Layer** Select all features from a layer by selecting the layer
- **Range** Specify the range of features by entering the regeneration numbers of the starting and ending features

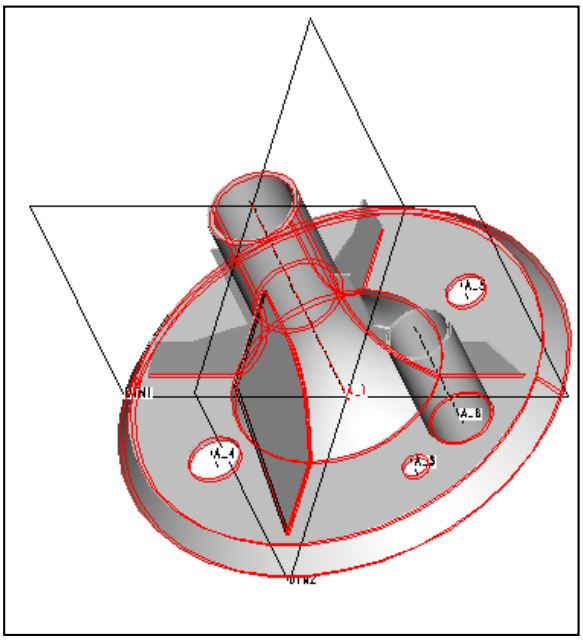


Figure 17.3(a) Reorder (Note the gusset closest to you)

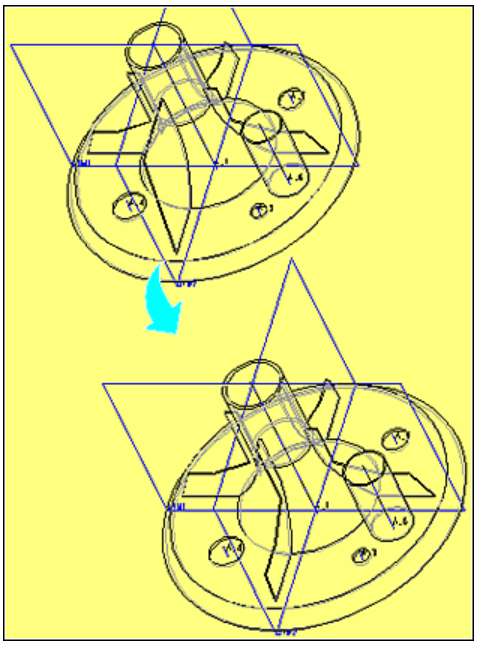


Figure 17.3(b) Reordered feature

Inserting Features

Normally, Creo Parametric adds a new feature after the last existing feature in the part, including suppressed features. Insert Mode allows you to add new features at any point in the feature sequence, except before the base feature or after the last feature. You can also insert features using the Model Tree. There is an arrow-shaped icon on the Model Tree that indicates where features will be inserted upon creation. By default, it is always at the end of the Model Tree. You may drag the location of the *arrow* higher or lower in the tree to insert features at a different point. When the *arrow* is dropped at a new location, the model is rolled backward or forward in response to the insertion *arrow* being moved higher or lower in the tree.

Lesson 17 STEPS

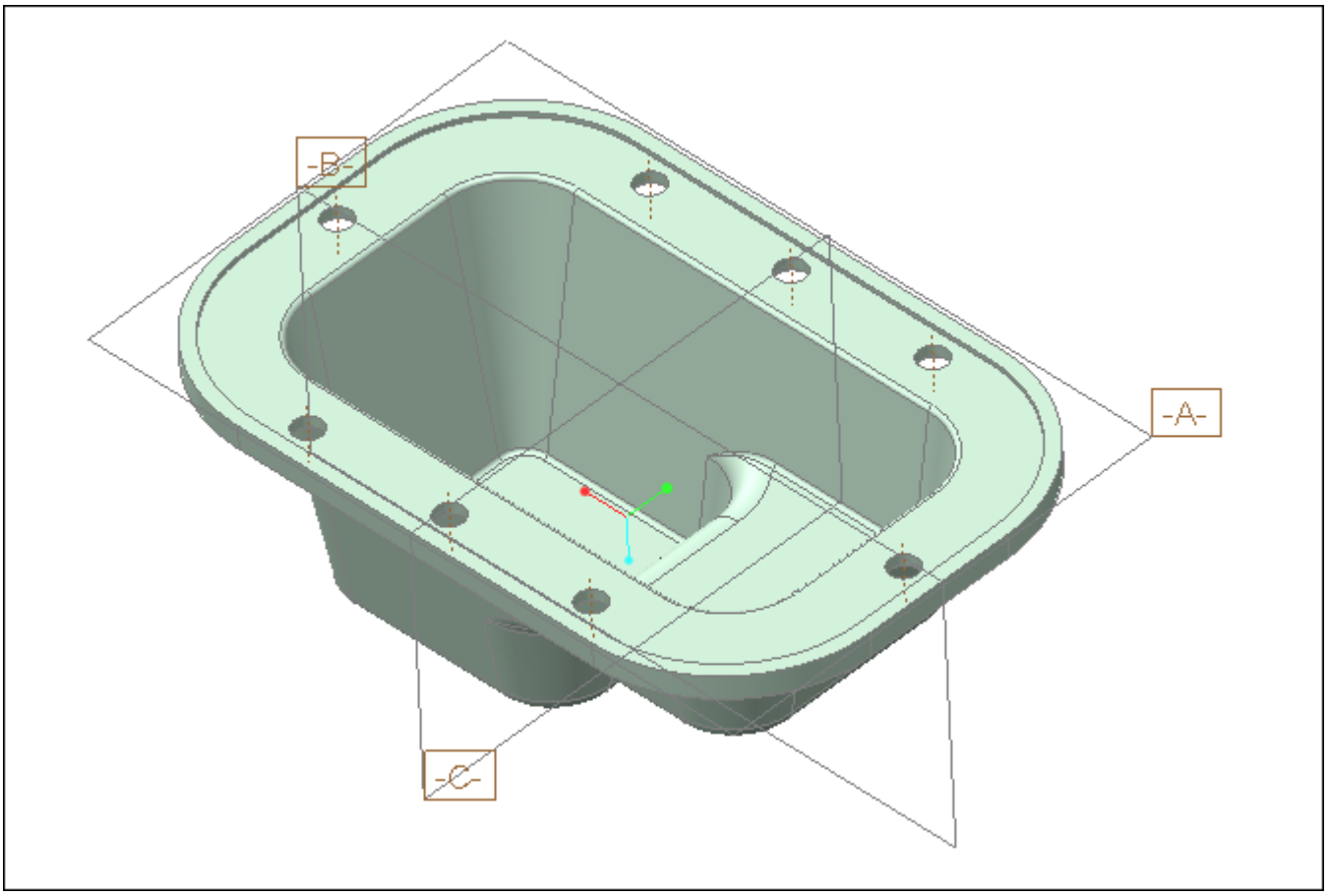


Figure 17.4 Oil Sink

Oil Sink

The Oil Sink (Fig. 17.4) requires the use of the **Shell Tool**. The shelling of a part should be done after the desired extrusions and most rounds have been modeled. This lesson part will have you create an extrusion, a cut, and a set of rounds. Some of the required rounds will be left off the part model on purpose.

Creo Parametric's **Insert Mode** option enables you to insert a set of features at an earlier stage in the design of the part. In other words, you can create a feature after or before a selected existing feature even if the whole model has been completed. You can also *move the order in which a feature was created* and therefore have subsequent features affect the reordered feature. A round created after a shell operation can be reordered to appear before the shell, to have the shell be affected by the round.

In this lesson, you will also insert a round or two before the existing shell feature using Insert Mode. The rounds will be shelled after the **Resume** option is picked, because the rounds now appear before the shell feature. The details shown in Figures 17.5(a) through (h) provide the design dimensions.

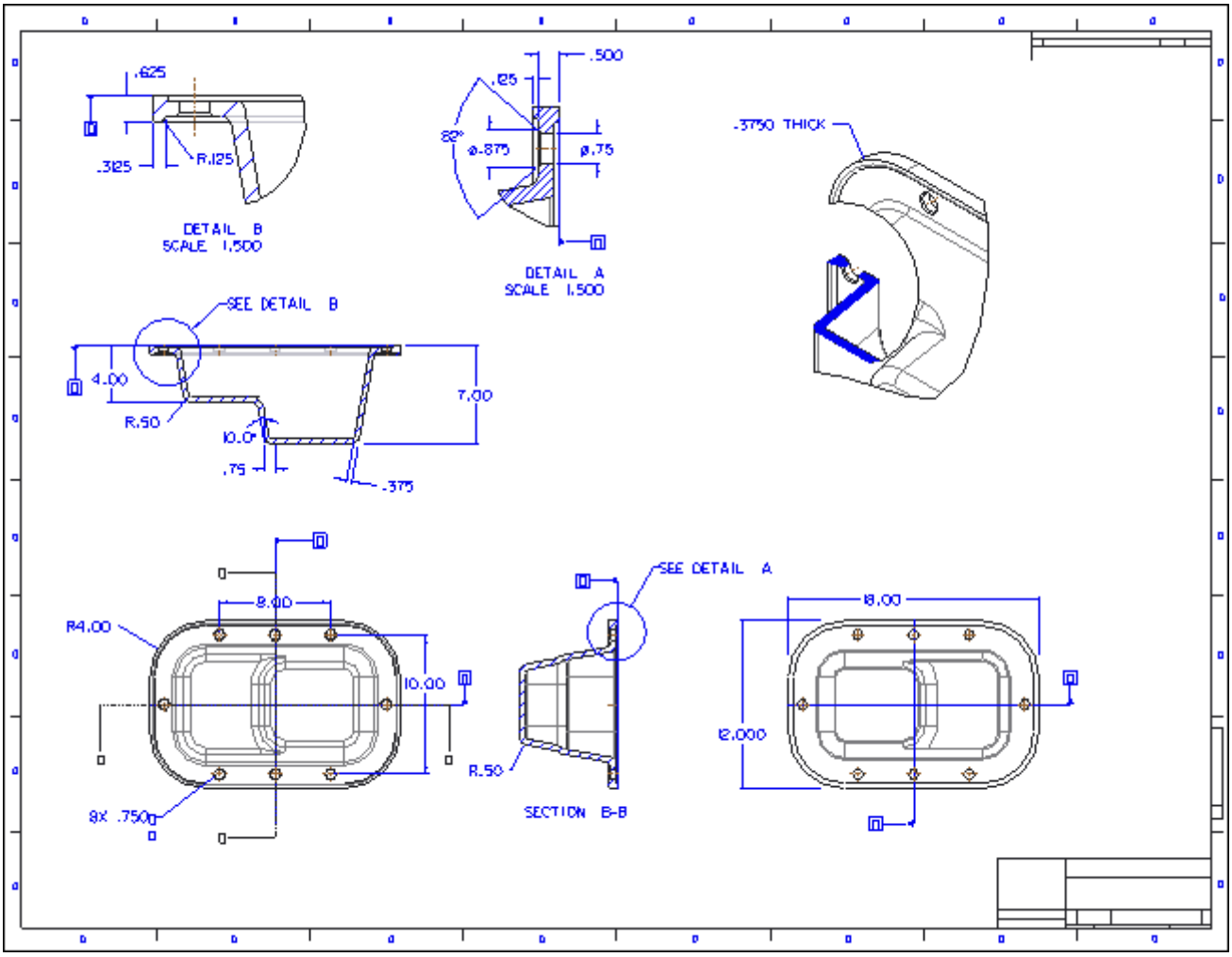


Figure 17.5(a) Oil Sink Detail Drawing

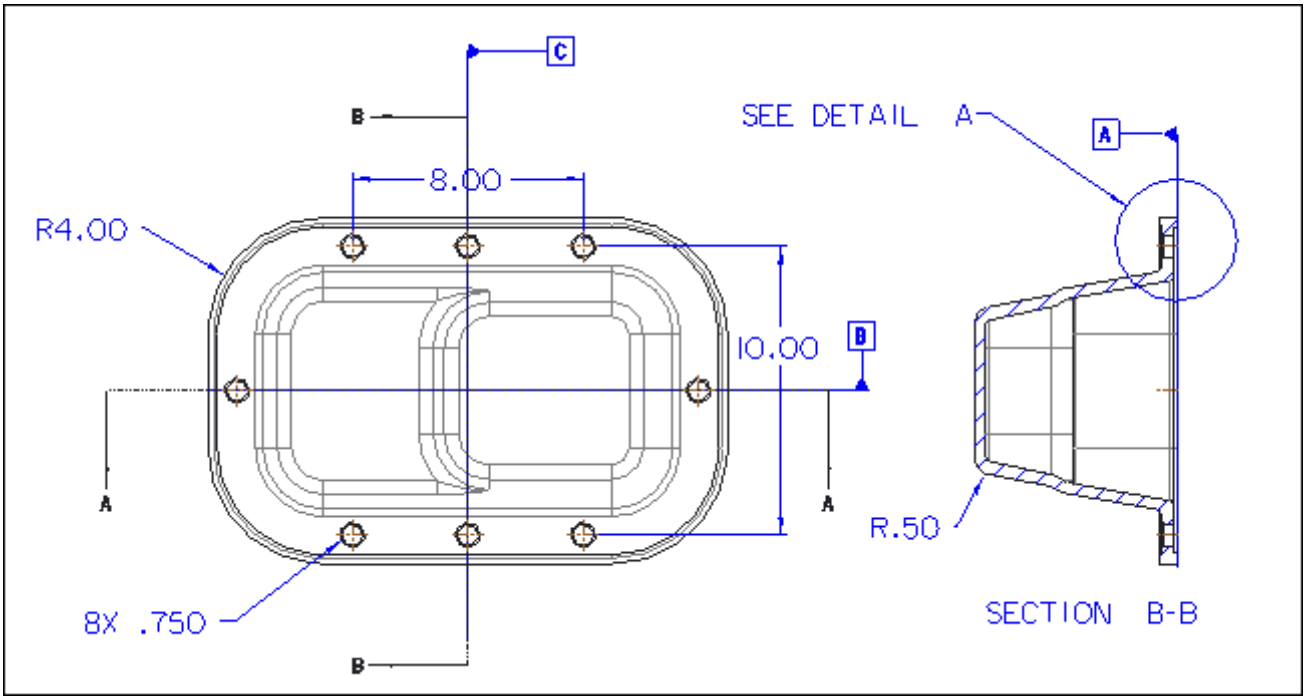


Figure 17.5(b) Oil Sink Front and Right Side Views

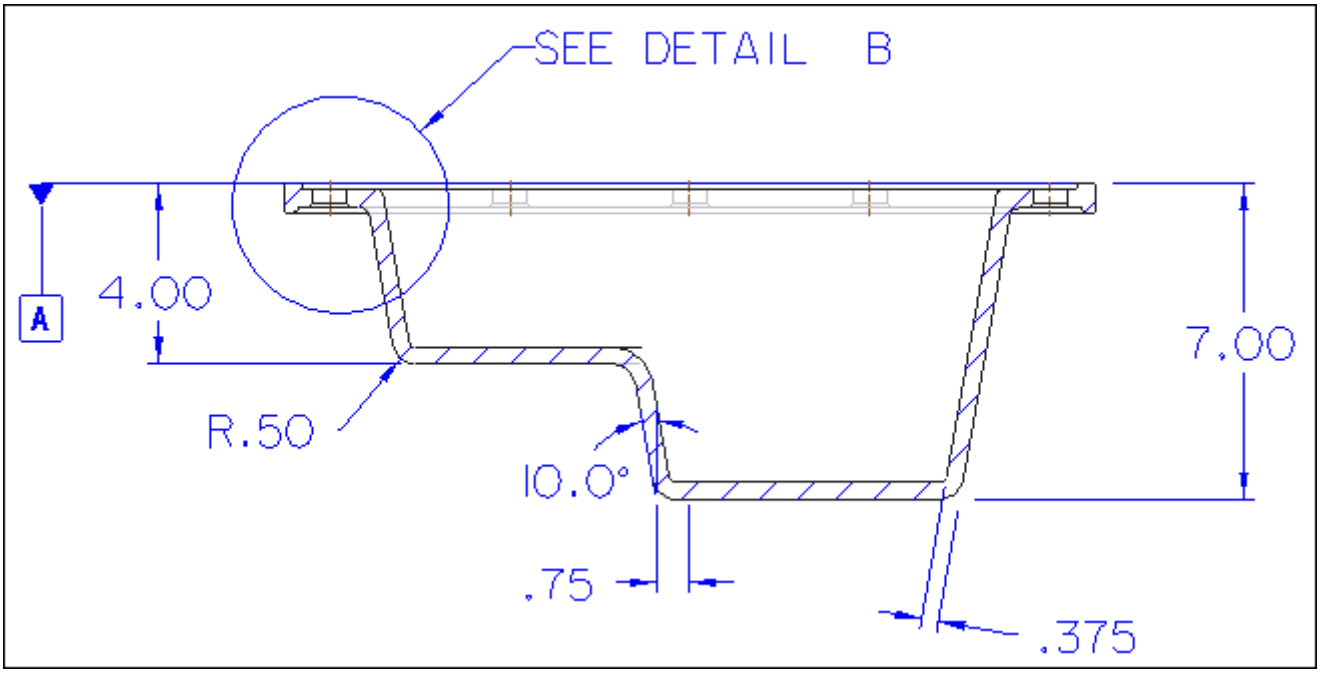


Figure 17.5(c) Oil Sink Section A-A

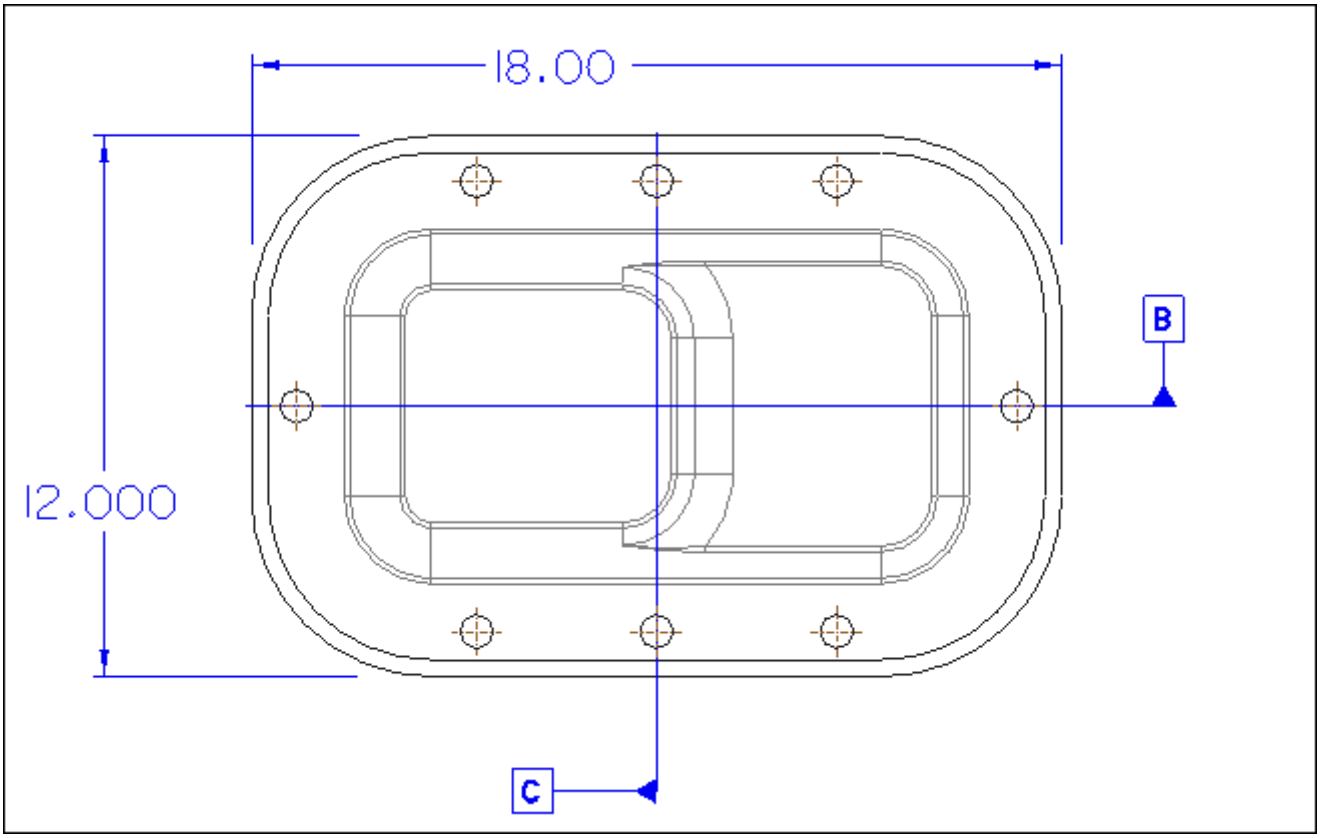


Figure 17.5(d) Oil Sink Back View

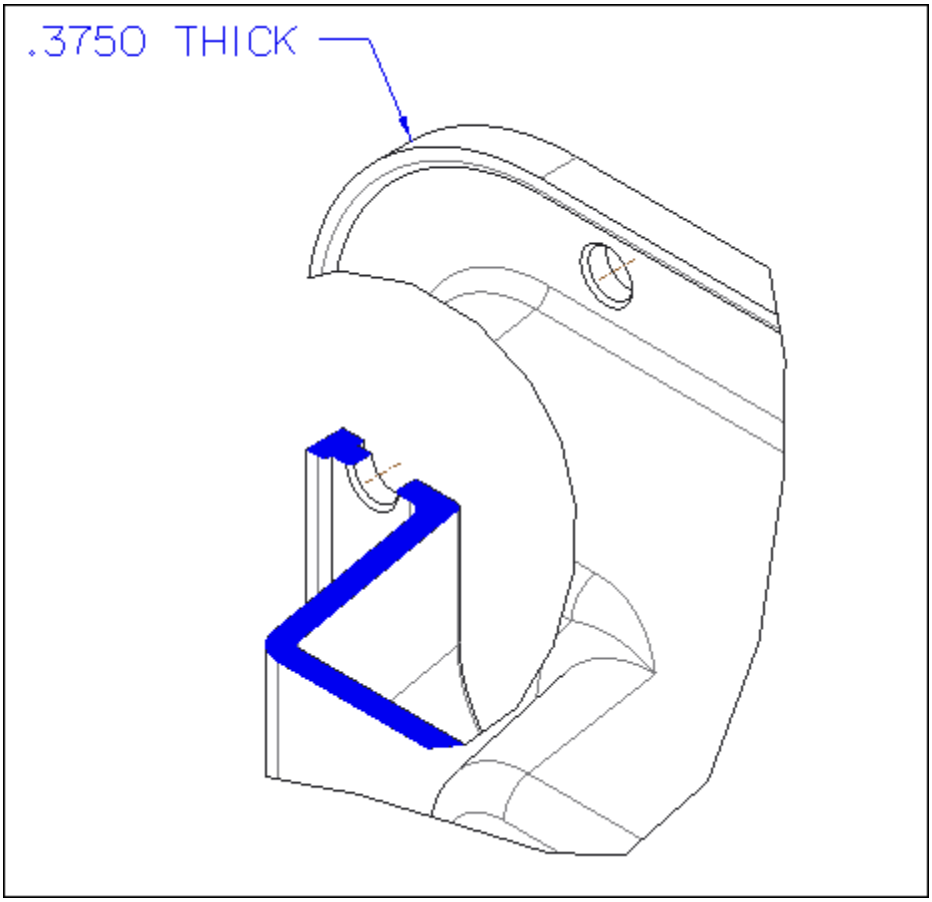


Figure 17.5(e) Oil Sink Cutaway View

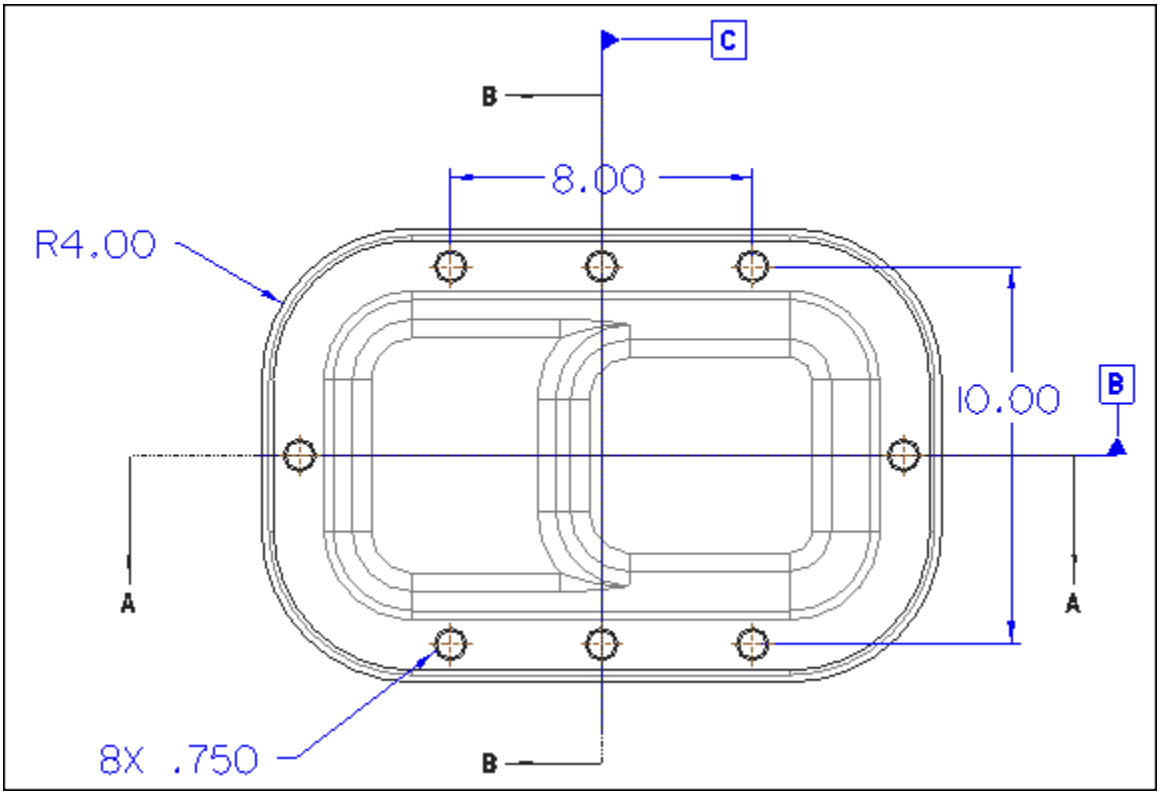


Figure 17.5(f) Oil Sink Front View Dimensions

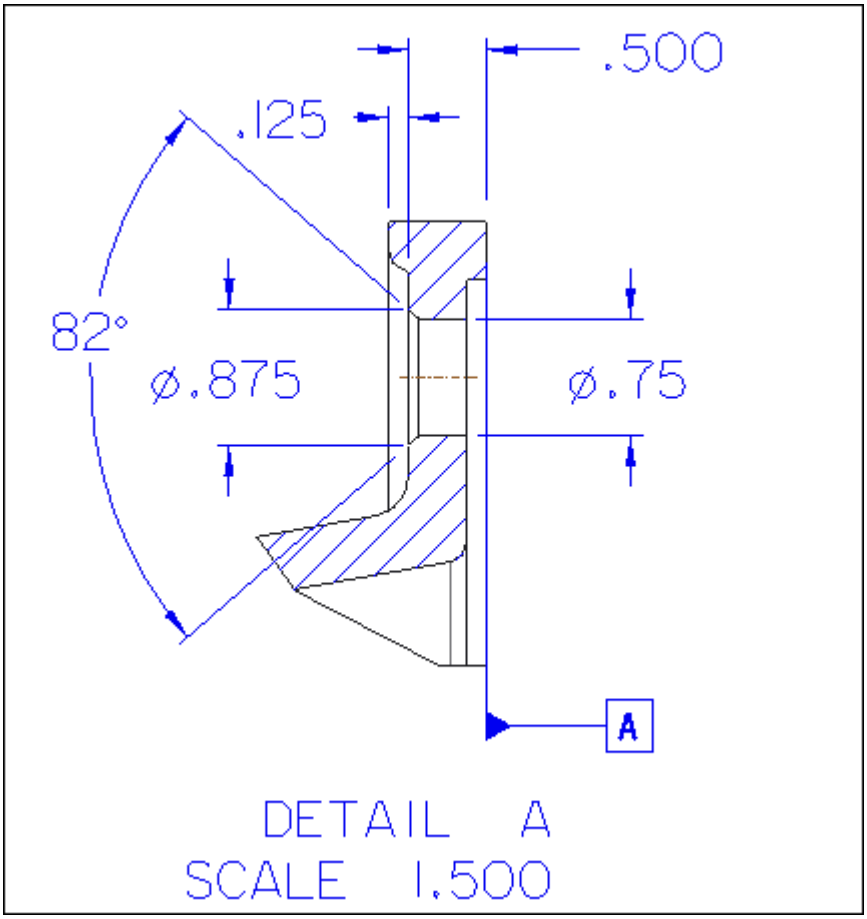


Figure 17.5(g) Oil Sink DETAIL A

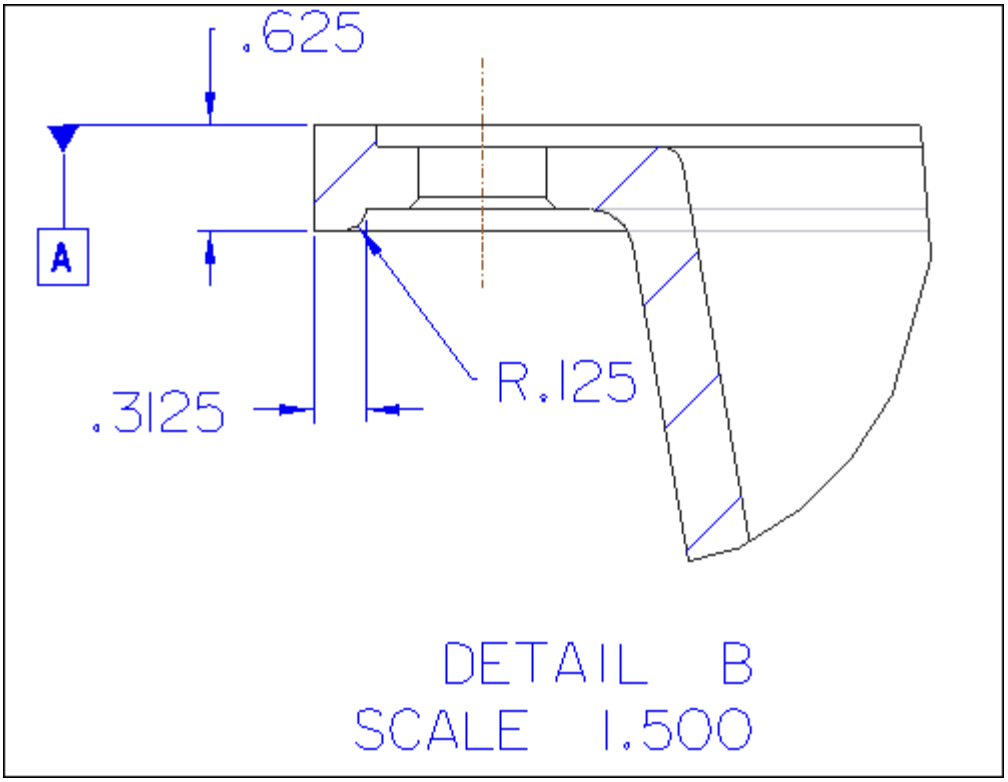


Figure 17.5(h) Oil Sink DETAIL B

Click: **File > Manage Session > Select Working Directory >** select the working directory **> OK > Ctrl+N >** Name **oil_sink >** Use default template **> OK > File > Prepare > Model Properties** (set the material and units):

- **Material** = steel.mtl
- **Units** = Inch lbm Second

Set Datum -A- and **Rename** the default datum planes and coordinate system (Fig. 17.6):

- Datum TOP = **B**
- Datum FRONT = **A**
- Datum RIGHT = **C**
- Coordinate System = **OIL_SINK**

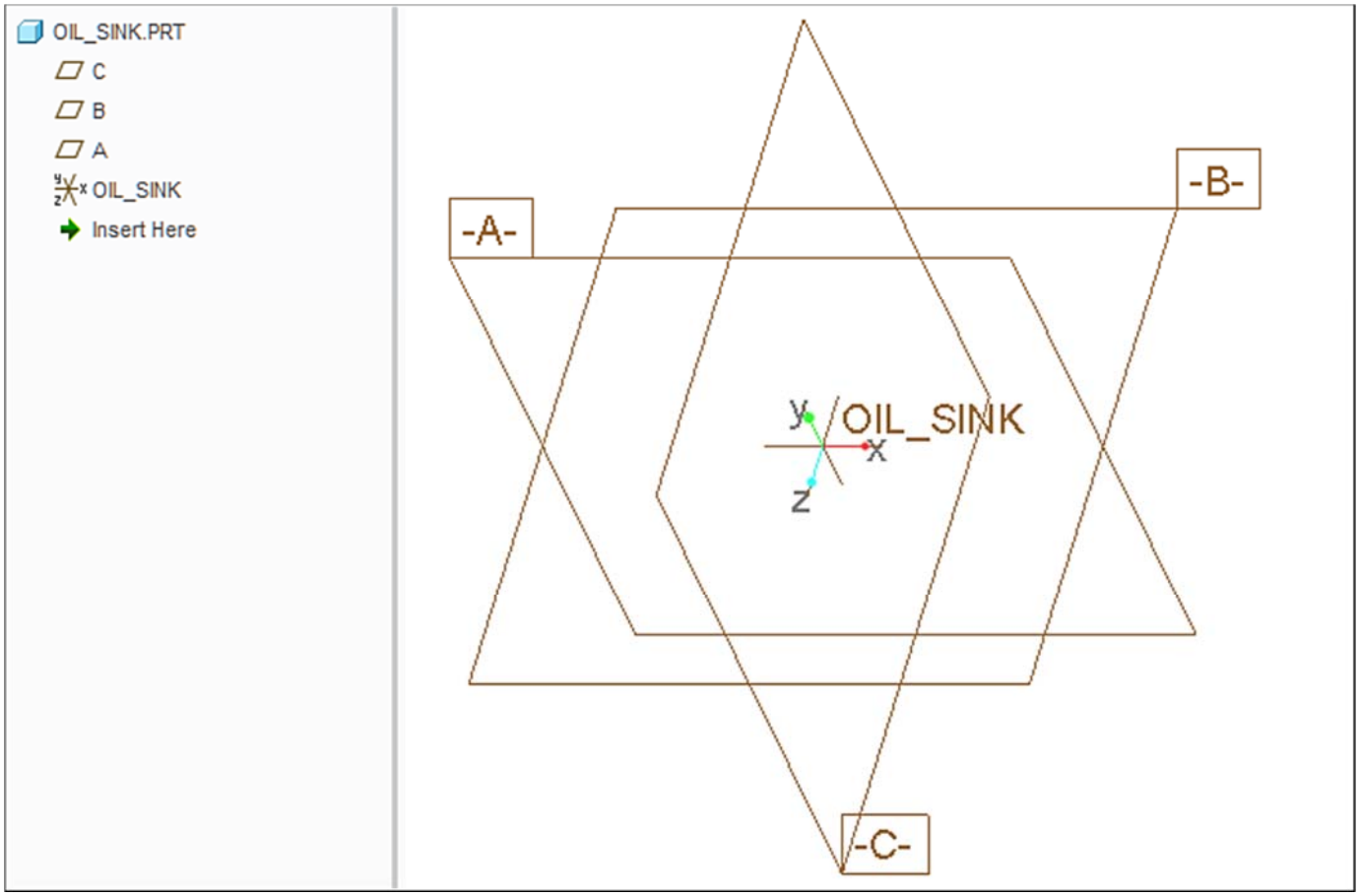


Figure 17.6 Set Datums and Renamed Coordinate System

Make the first extrusion **.50** (thickness) **X 13.00** (height) **X 18.00** (length), with **R4.00** rounds (add the fillets to the sketch). Sketch on datum plane **A**, and center the first extrusion horizontally on datum **B** and vertically on datum **C** [Figs. 17.7(a-b)].

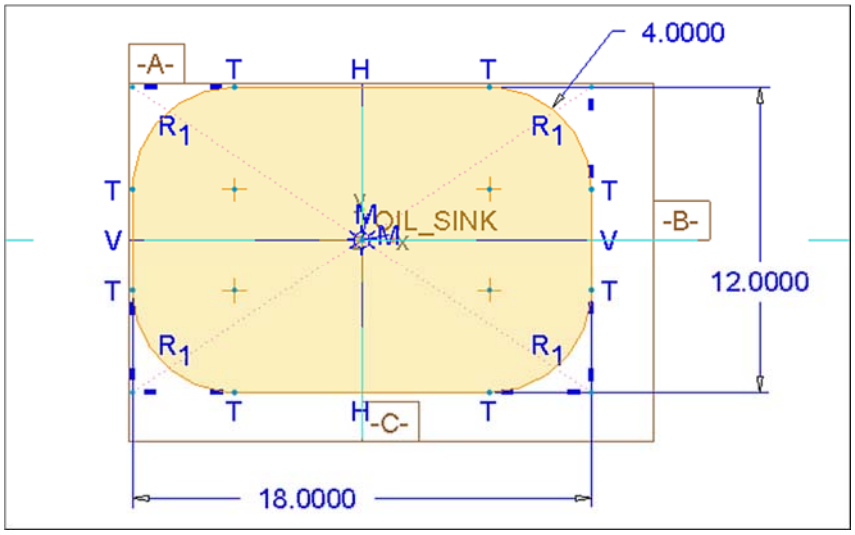


Figure 17.7(a) Dimensions for the First Extrusion

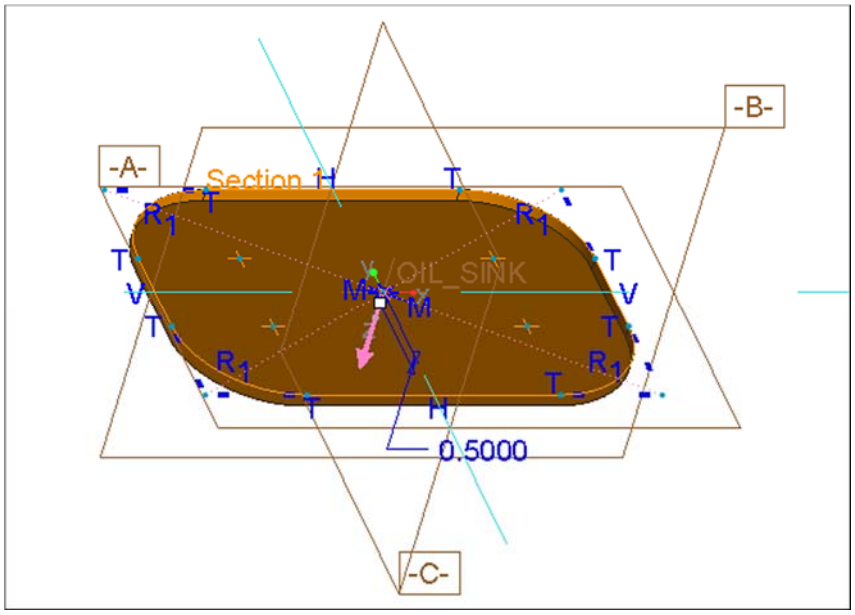


Figure 17.7(b) Standard Orientation

Make the second extrusion offset from the edge of the first extrusion **-3.00**, with a height of **7.00** [Figs. 17.8(a-b)]. Sketch on the top surface of the first extrusion; then, create the cut [Figs. 17.9(a-b)].

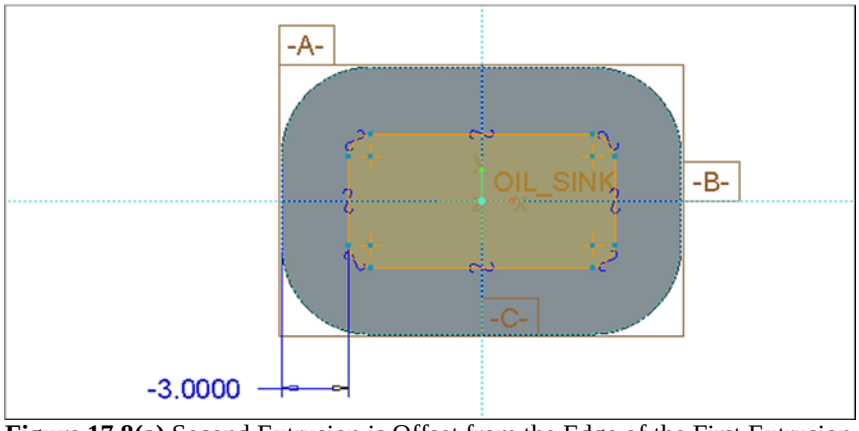


Figure 17.8(a) Second Extrusion is Offset from the Edge of the First Extrusion

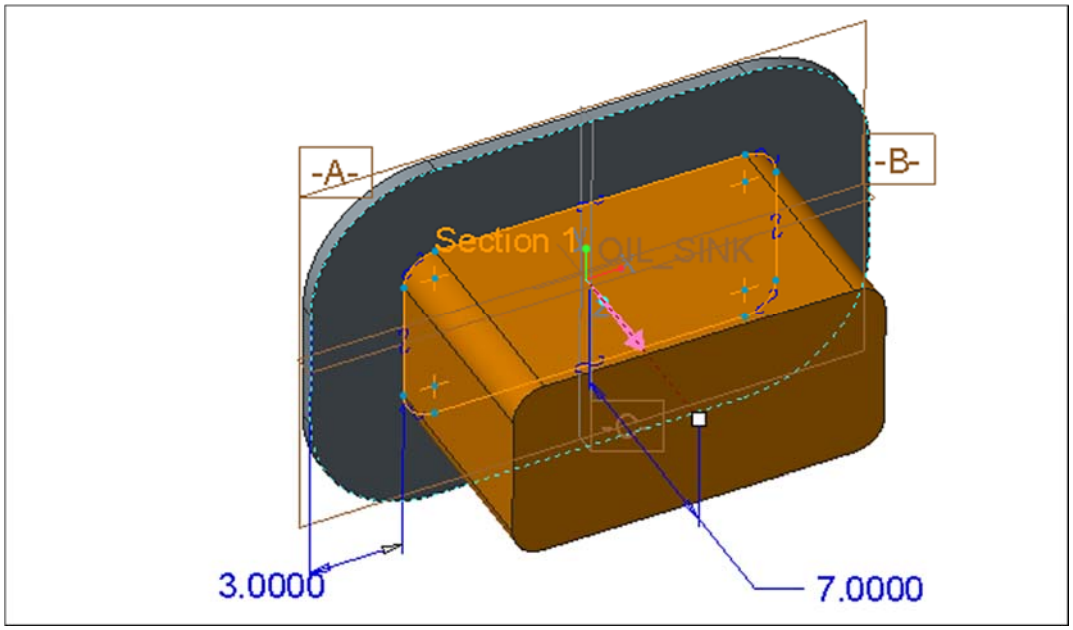


Figure 17.8(b) Second Extrusion

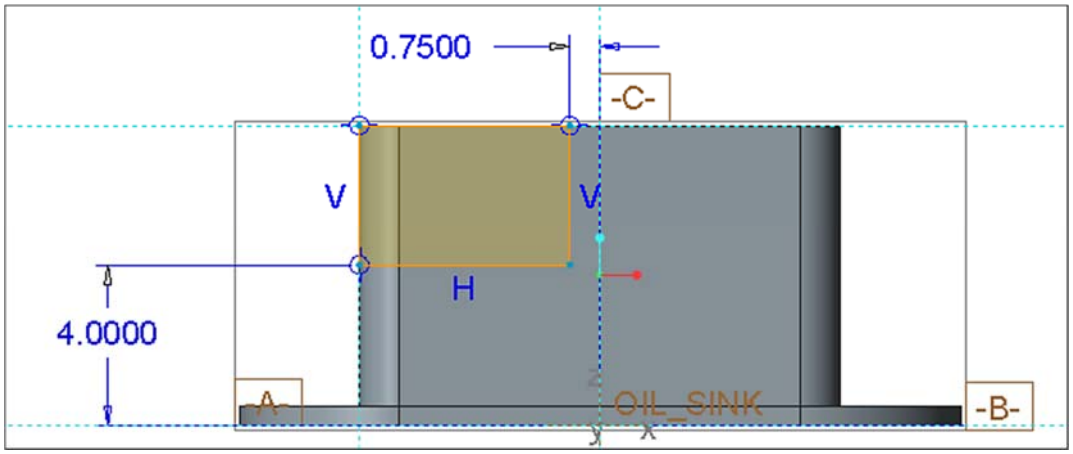


Figure 17.9(a) Dimensions for the Cut

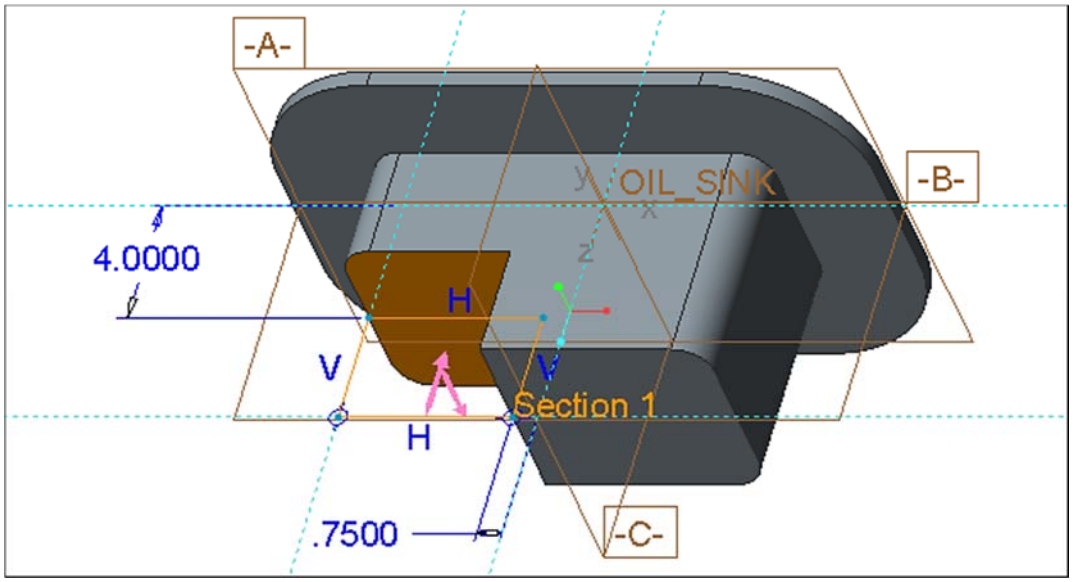


Figure 17.9(b) Standard Orientation of the Cut

Add the **R1.50** rounds [Figs. 17.10(a-b)]. Draft all vertical surfaces of the second extrusion **10** degrees. Use the top surface as the Draft hinge [Figs. 17.11(a-b)]. Change the model color as desired.

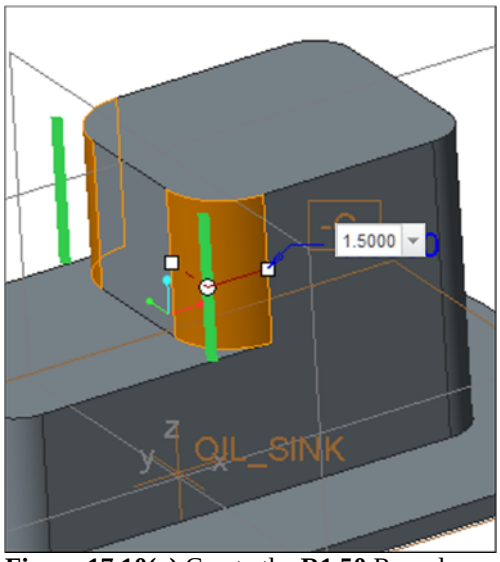


Figure 17.10(a) Create the **R1.50** Rounds

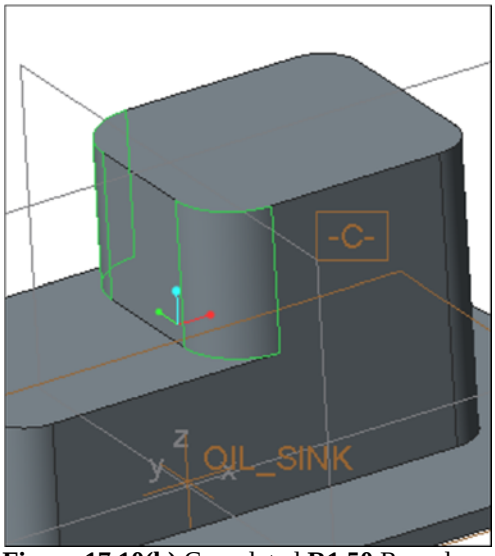


Figure 17.10(b) Completed **R1.50** Rounds

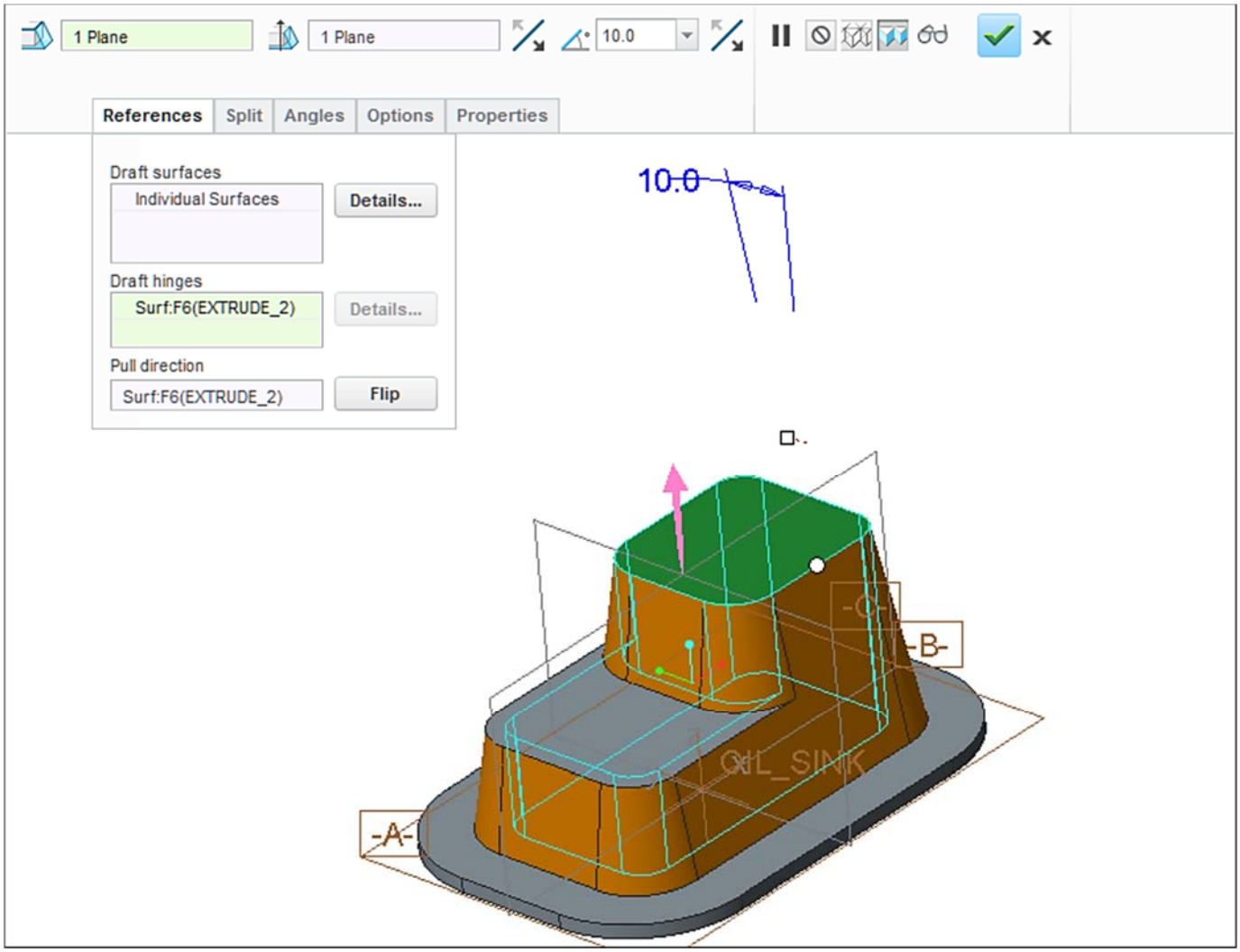


Figure 17.11(a) Draft References

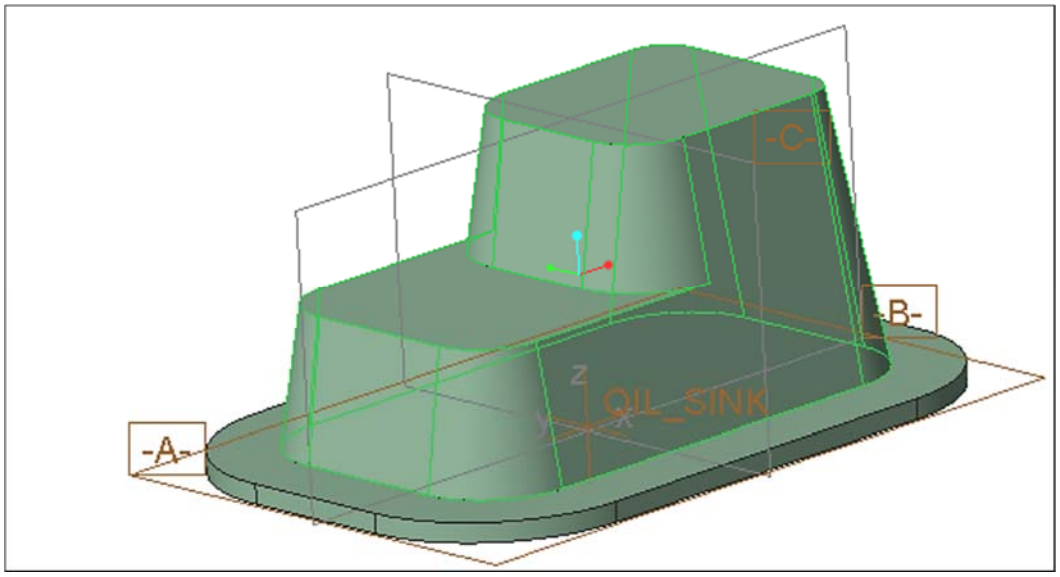


Figure 17.11(b) Drafted Sides. New Color.

Click: Shell > Thickness **.375** > **Enter** > spin the model > **References** tab > Removed surfaces-- select the bottom surface of the part [Fig. 17.12(a)**]** > **MMB** > **LMB** > **Ctrl+R** > **Ctrl+S** [Fig. 17.12(b)]

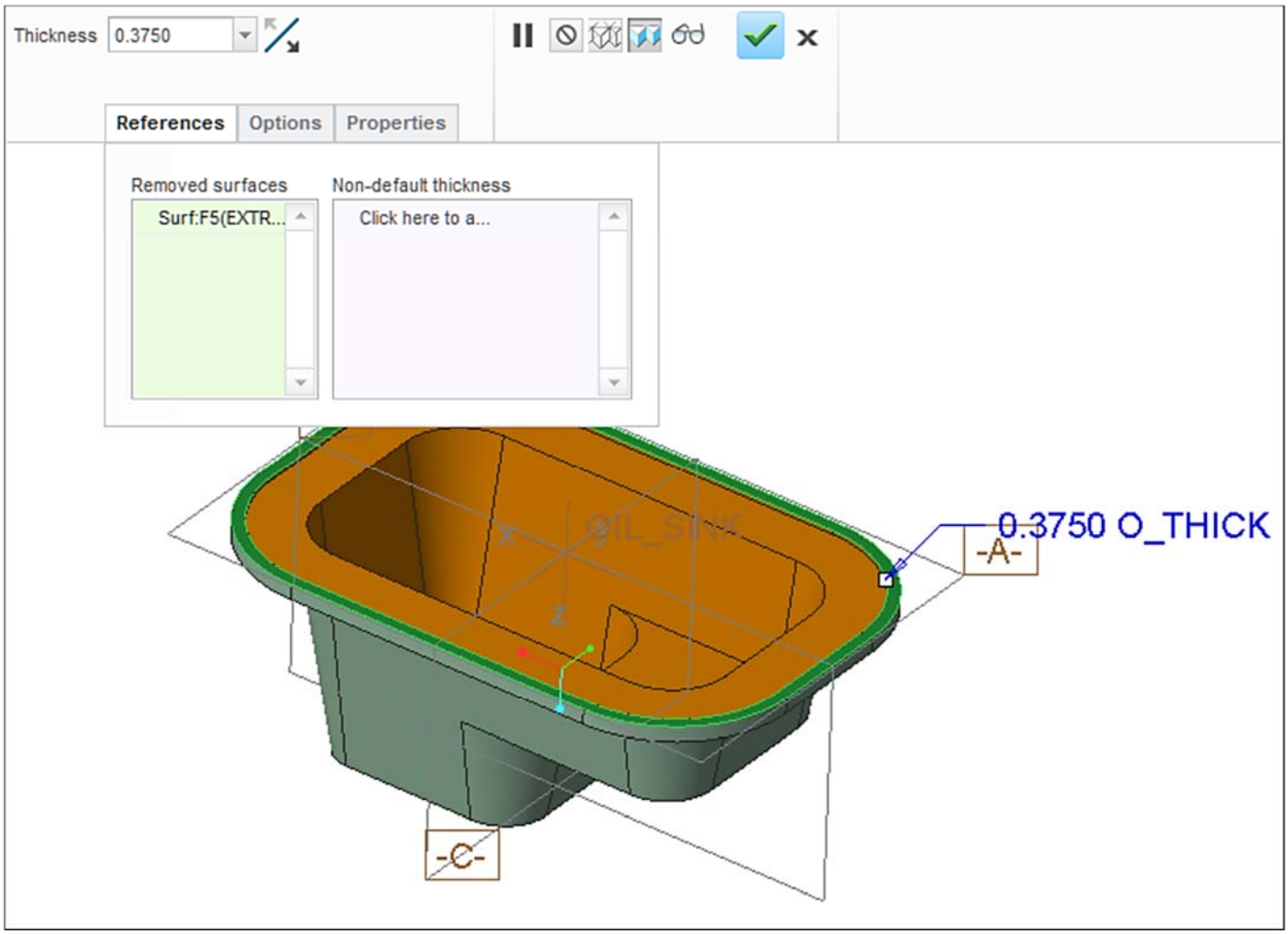


Figure 17.12(a) Shell Tool

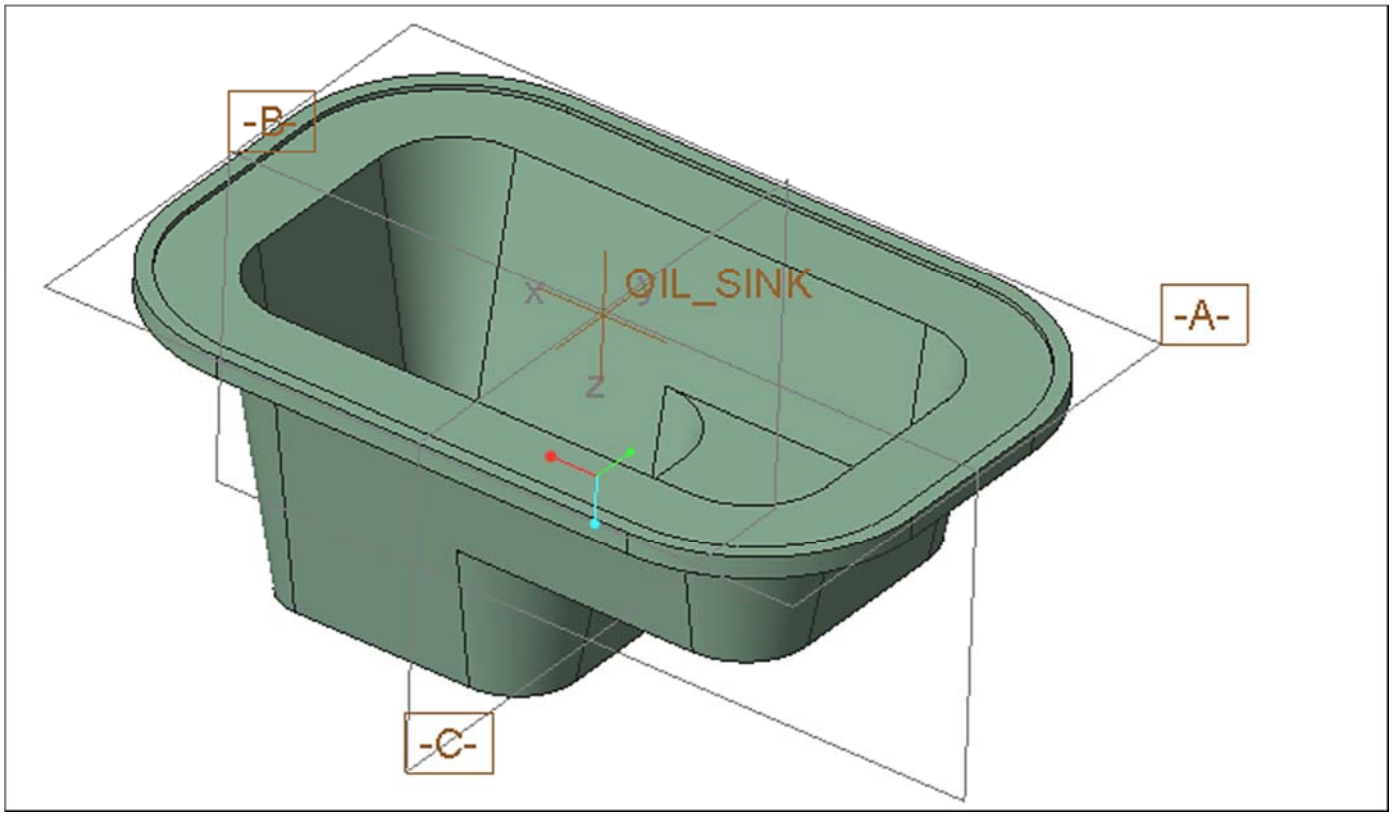


Figure 17.12(b) Shelled Part

The next feature you need to create is a *"lip"* around the part using an extrusion. Press: **Ctrl+D** > sketch on the top surface of the first extrusion > sketch two closed loops [Fig. 17.13(a)]. Use the edge of the first extrusion for the first loop and then create an offset edge (**-.3125**) for the second loop [Fig. 17.13(b)]. > The depth of the lip extrusion is **.125** [Figs. 17.13(c-d)].

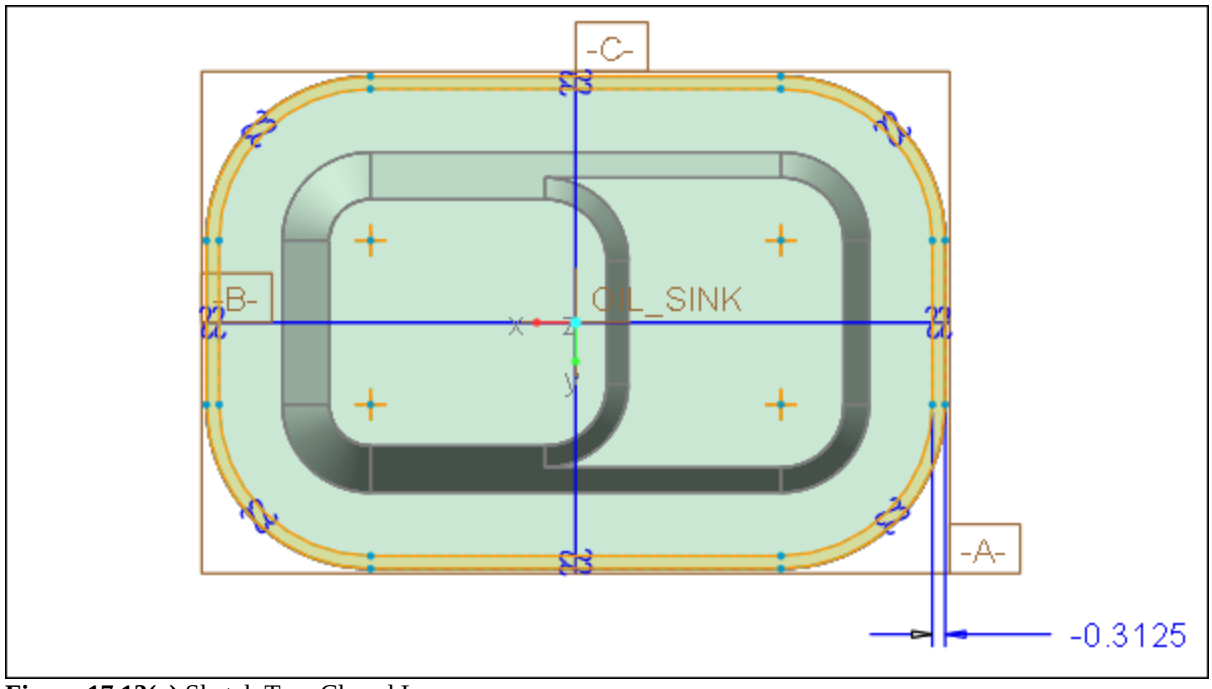


Figure 17.13(a) Sketch Two Closed Loops

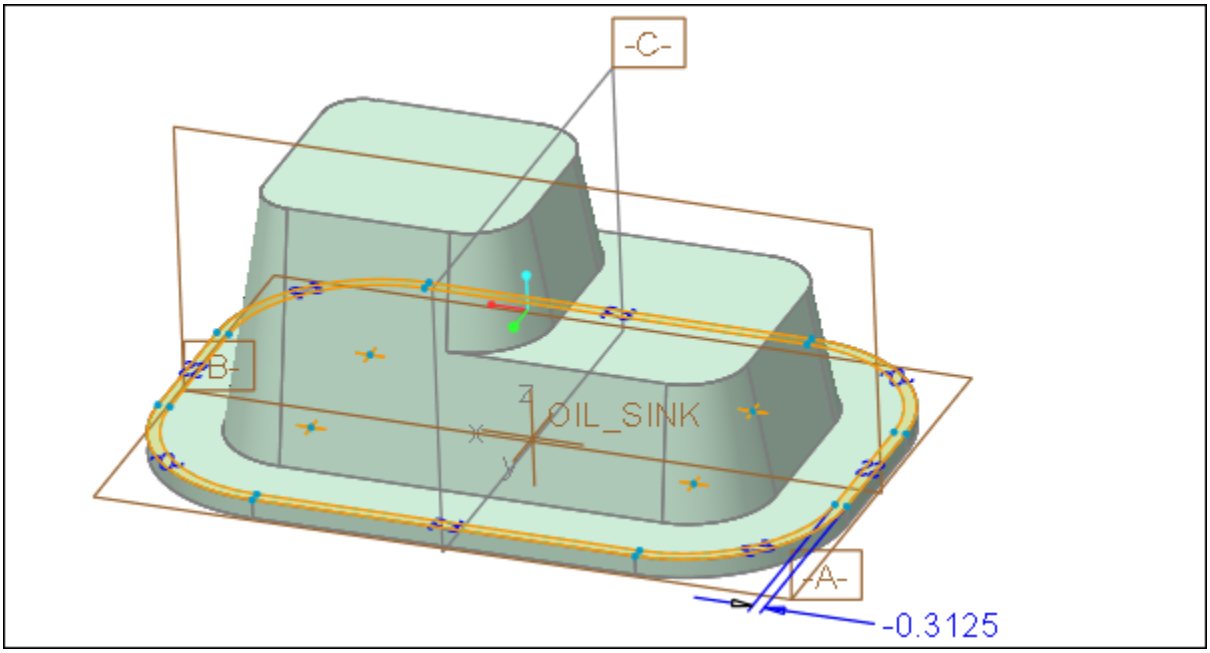


Figure 17.13(b) 3D View of the Sketch

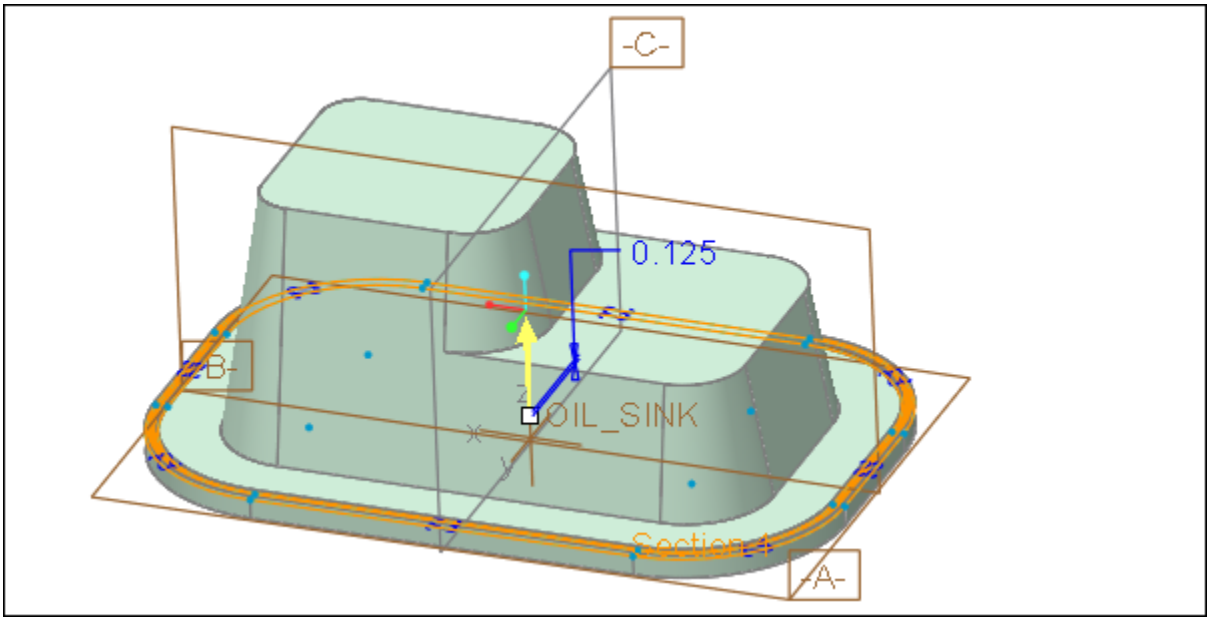


Figure 17.13(c) Depth **.125**

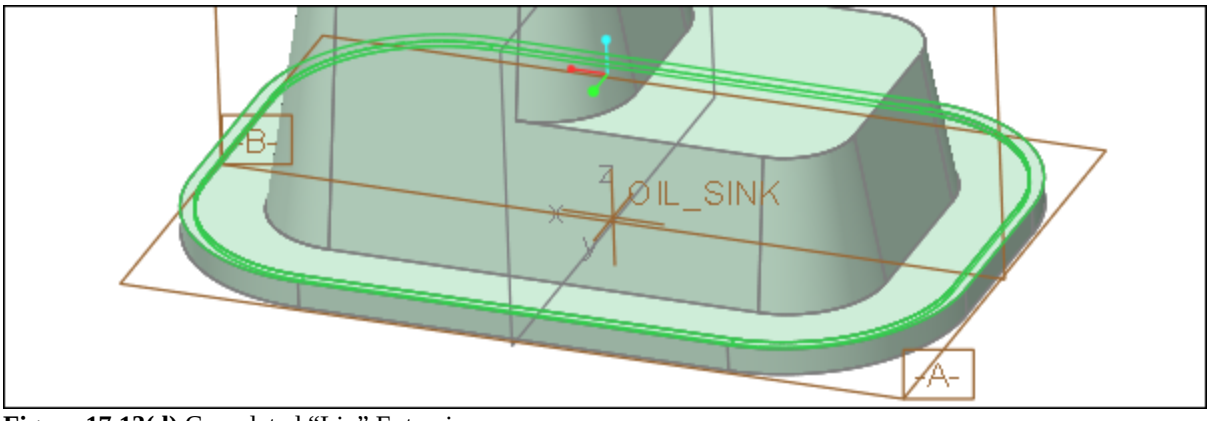


Figure 17.13(d) Completed "Lip" Extrusion

Add the following rounds with one round feature: **R.125** round on the inside of the *"lip"* [Fig. 17.14(a)] > spin the model > **R.125** round to the inside edge [Fig. 17.14(b)] > spin the model > **R.250** round between the first two extrusions [Fig. 17.14(c)] >  > **Ctrl+S** > **LMB**

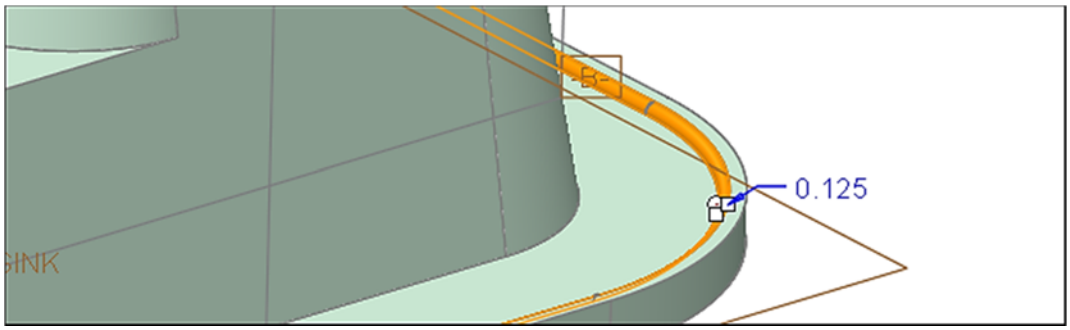


Figure 17.14(a) Set 1 Round **R.125**

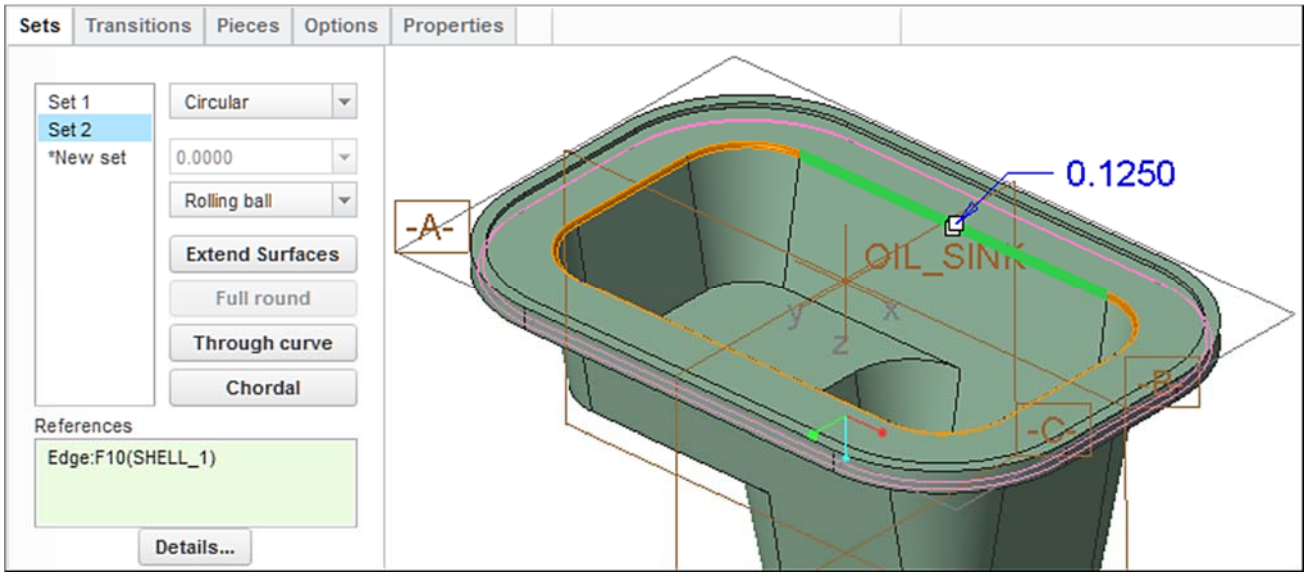


Figure 17.14(b) Set 2 Round **R.125**

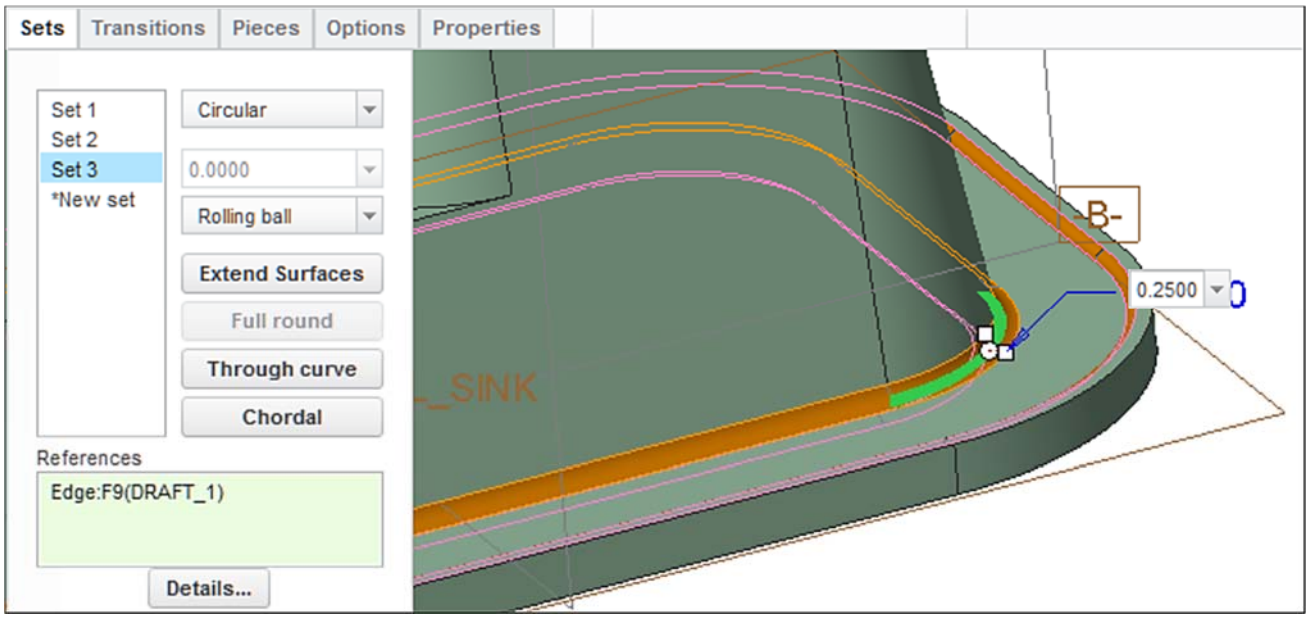


Figure 17.14(c) Set 3 Round **R.250**

The countersunk holes will be added next, click: Hole > spin the part > **Drill to intersect with all surfaces** > change the diameter to **.750** > **Enter** > **Placement** tab > select a location on the surface for the hole placement [Figs. 17.15(a-c)]

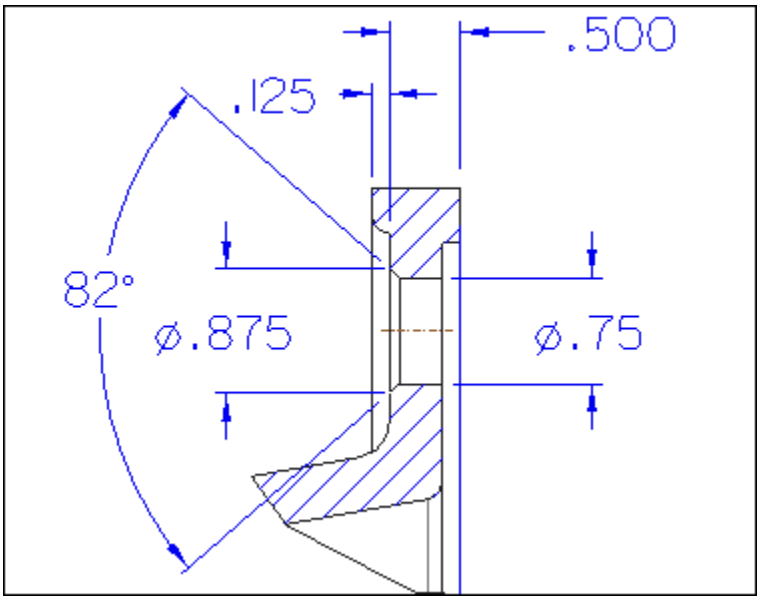


Figure 17.15(a) Hole Dimensions

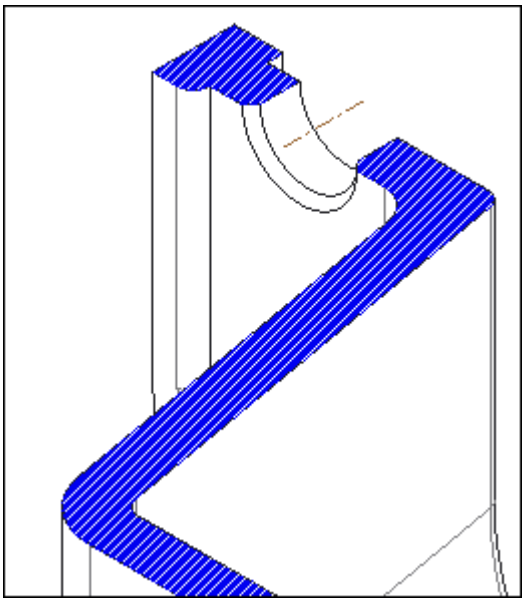


Figure 17.15(b) X-Section of Hole

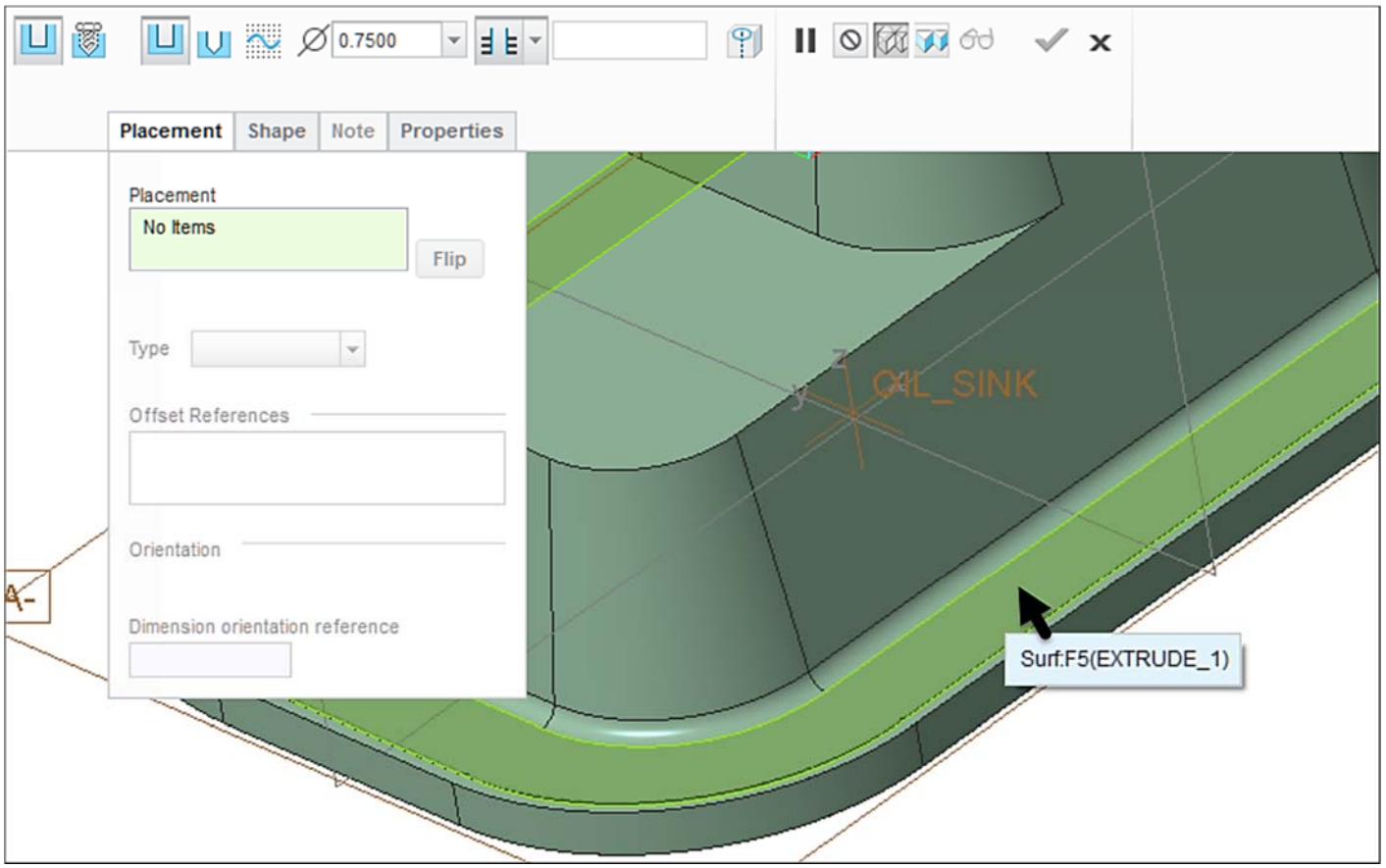


Figure 17.15(c) Hole Placement View Orientation

Place the pointer on a drag handle *(green)* > press and hold down the **LMB** > move the pointer to datum **C** (highlights) > release the **LMB** > similarly, move the other drag handle to datum **B** [Figs. 17.15(d-e)]

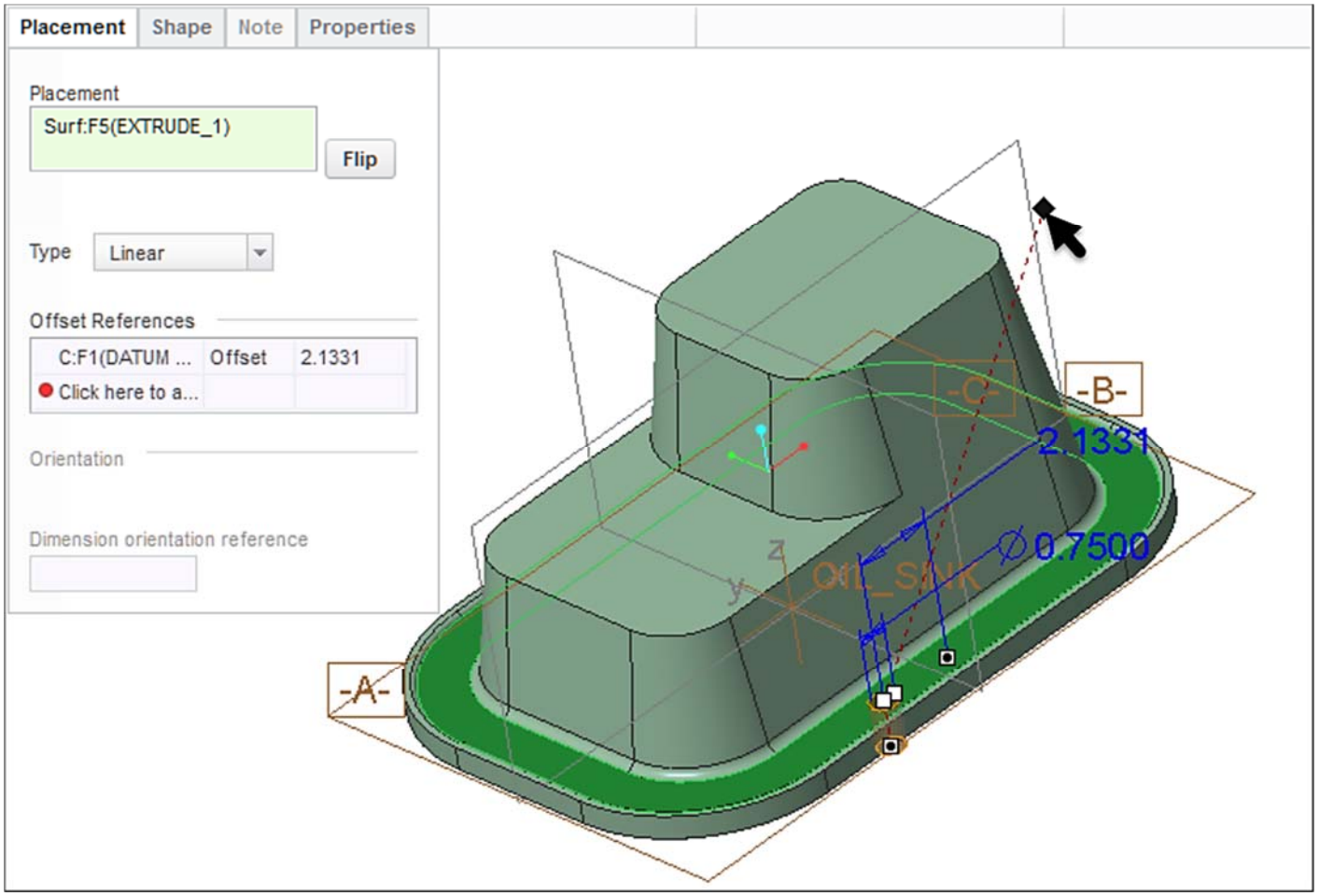


Figure 17.15(d) Hole Offset Reference to Datum **C** Established. Drag Remaining Position Handle to Datum **B**

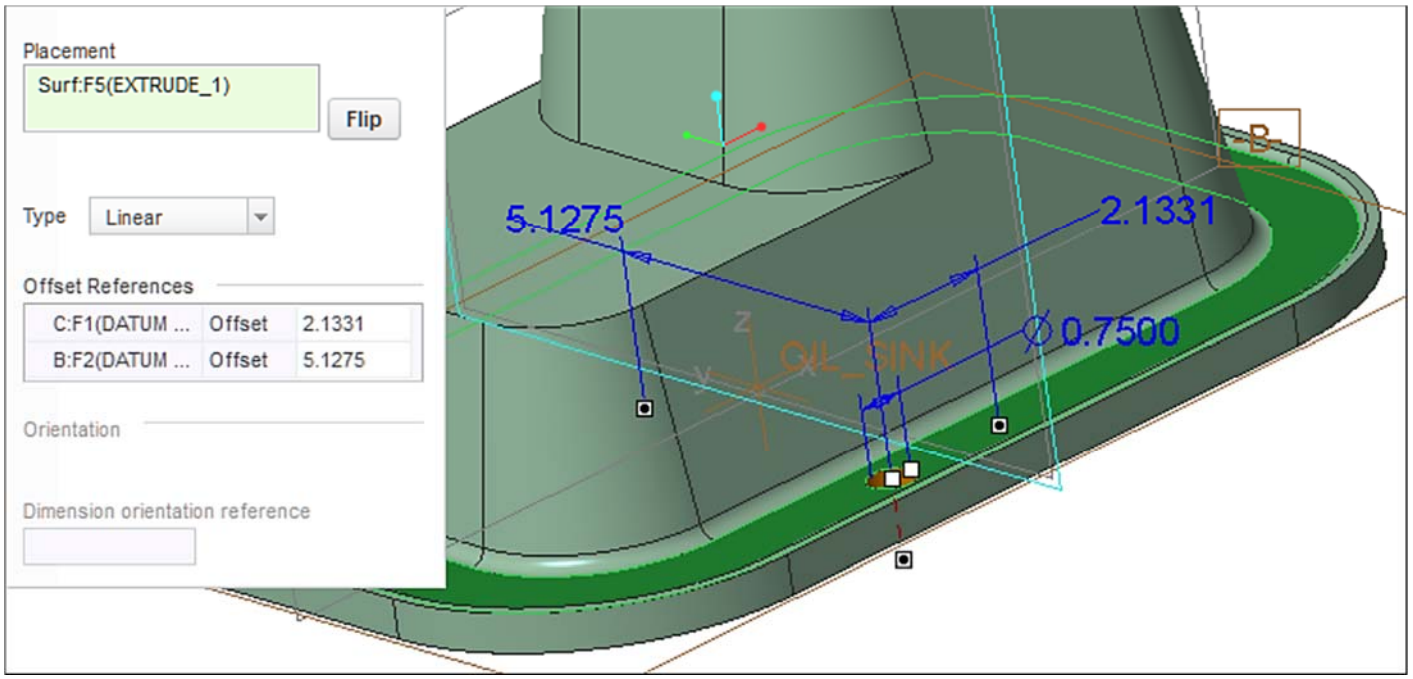


Figure 17.15(e) Initial Offset References Dimensions

Modify the values to be **4.00** from datum **C** and **5.00** from datum **B** [Fig. 17.15(f)]

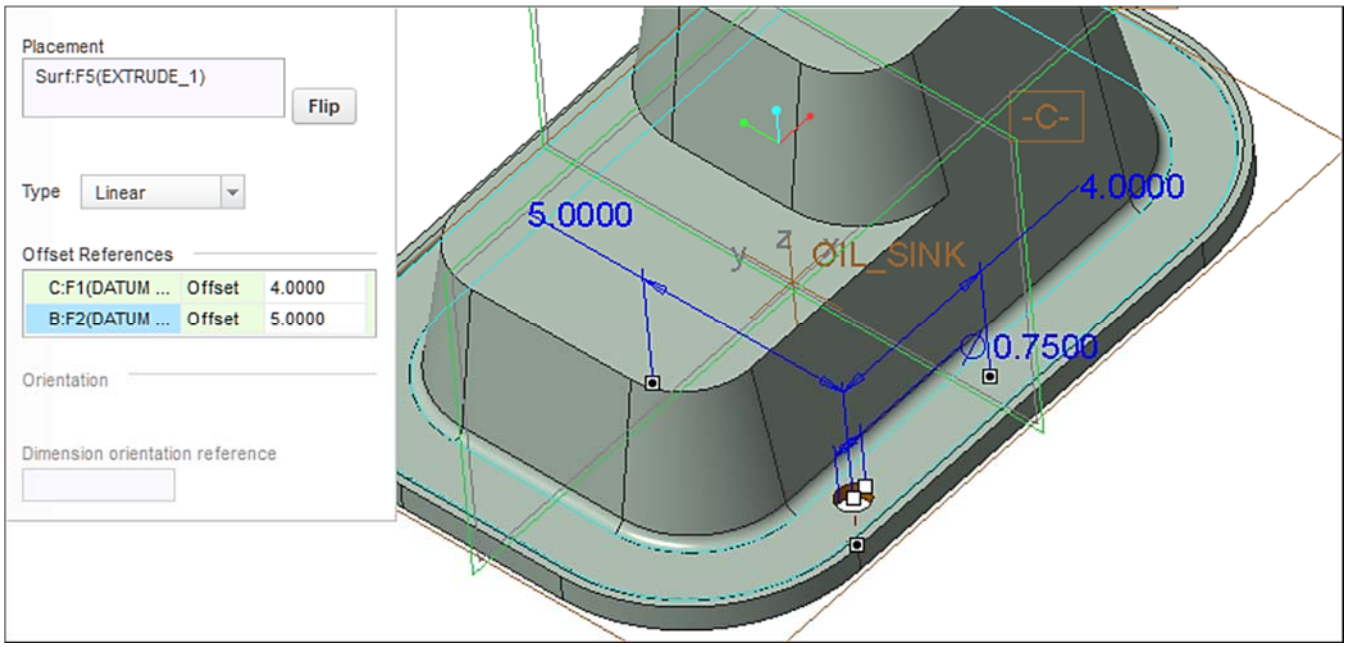


Figure 17.15(f) Offset References Dimensions (design dimensions)

Set: 

> **Shape** tab [Fig. 17.15(g)]

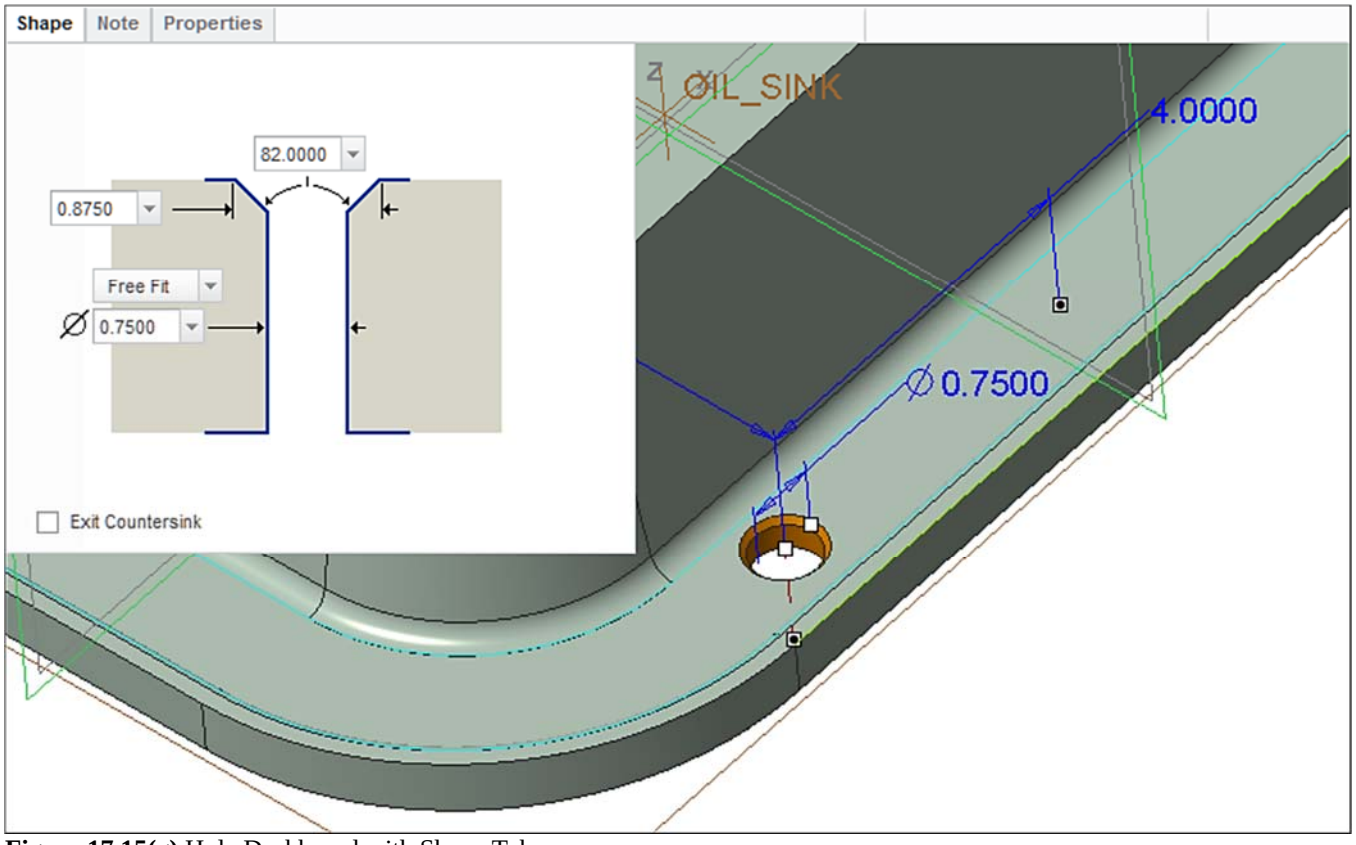
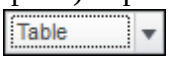


Figure 17.15(g) Hole Dashboard with Shape Tab open

Click:  > **Ctrl+S** > check the Navigator settings, click: **Settings** > **Tree Filters** > toggle *on* all Display options > **OK** > next to the Hole feature in the Model Tree, pick  (to expand) > pick on the Note [Fig. 17.15(h)] > select the Hole feature > press **RMB** > **Pattern** > 

> 

[Fig. 17.16(a)]

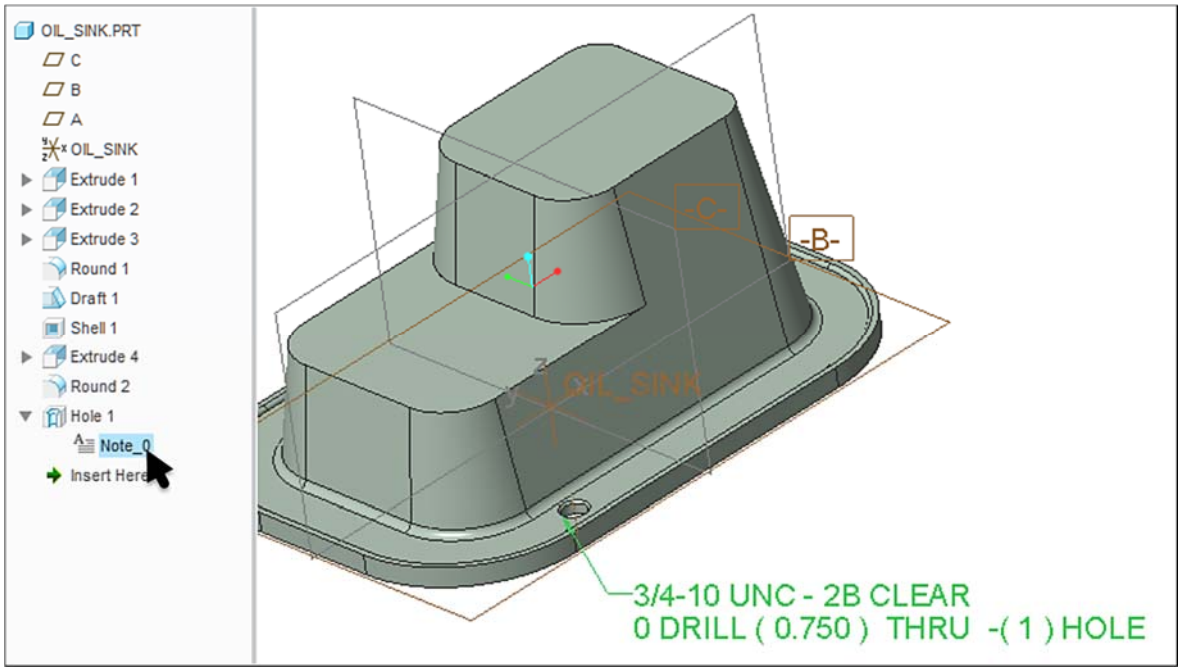


Figure 17.15(h) Completed Countersunk Hole *(your Hole id and Note id may be different)*

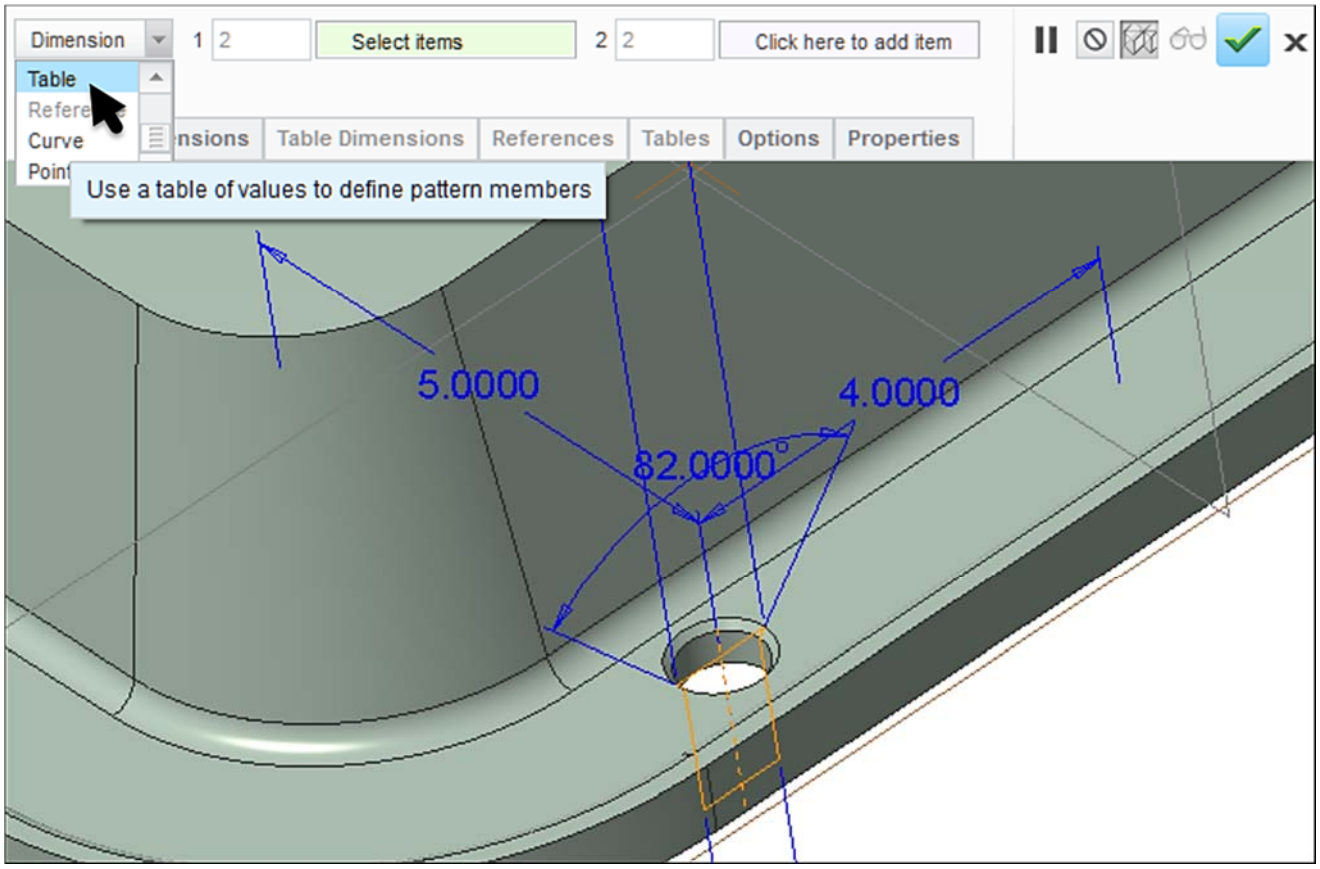


Figure 17.16(a) Pattern Members Defined by Table

Click: **Table Dimensions** tab > with the **Ctrl** key pressed, select the **4.00** dimension and then the **5.00** dimension [Fig. 17.16(b)] > **Edit** [Fig. 17.16(c)] > add the information [Figs. 17.16(d-f)]

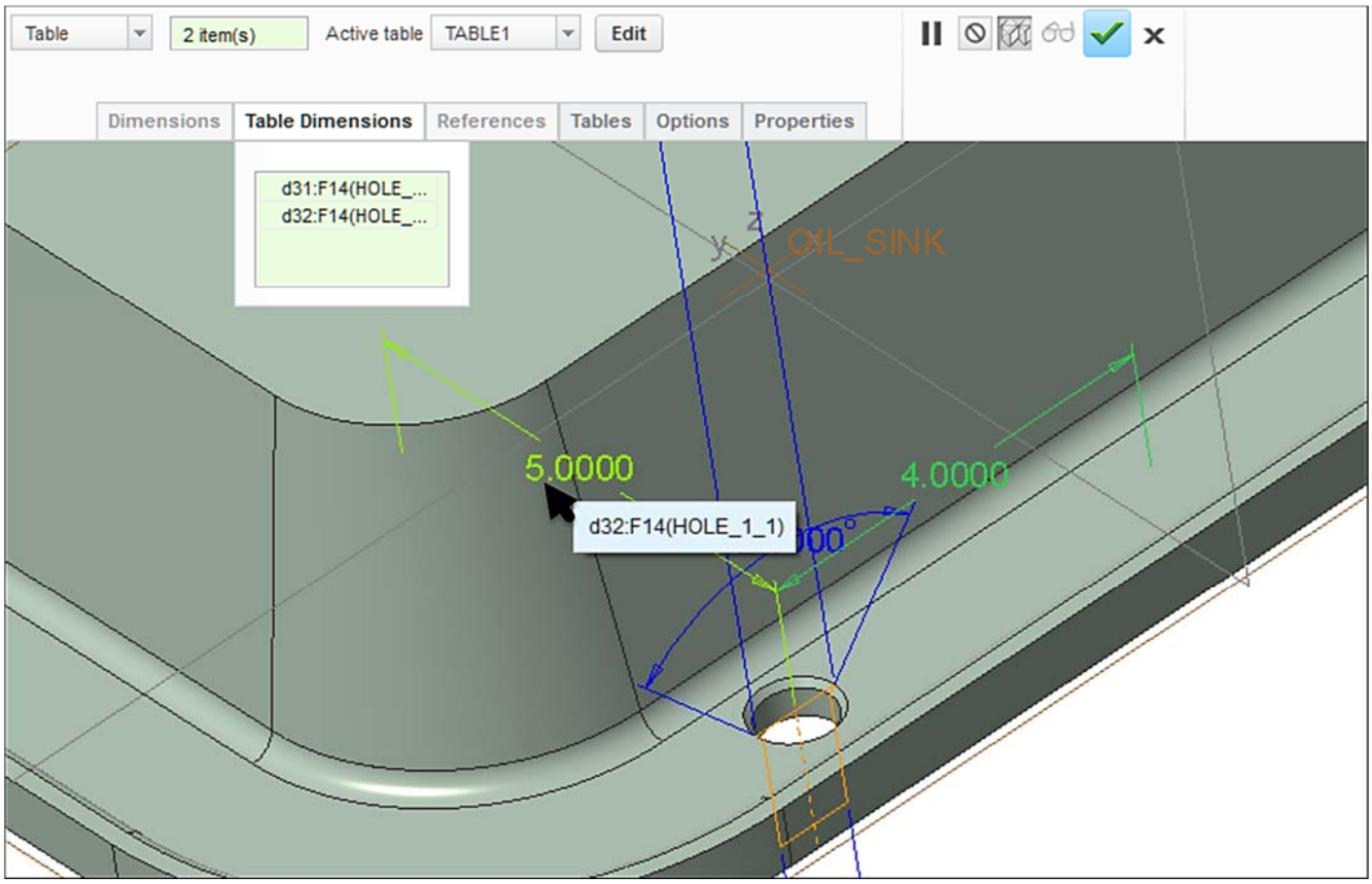


Figure 17.16(b) Table Dimensions Tab with the **4.00** and the **5.00** Dimensions Added to the Table *(your display may be different)*

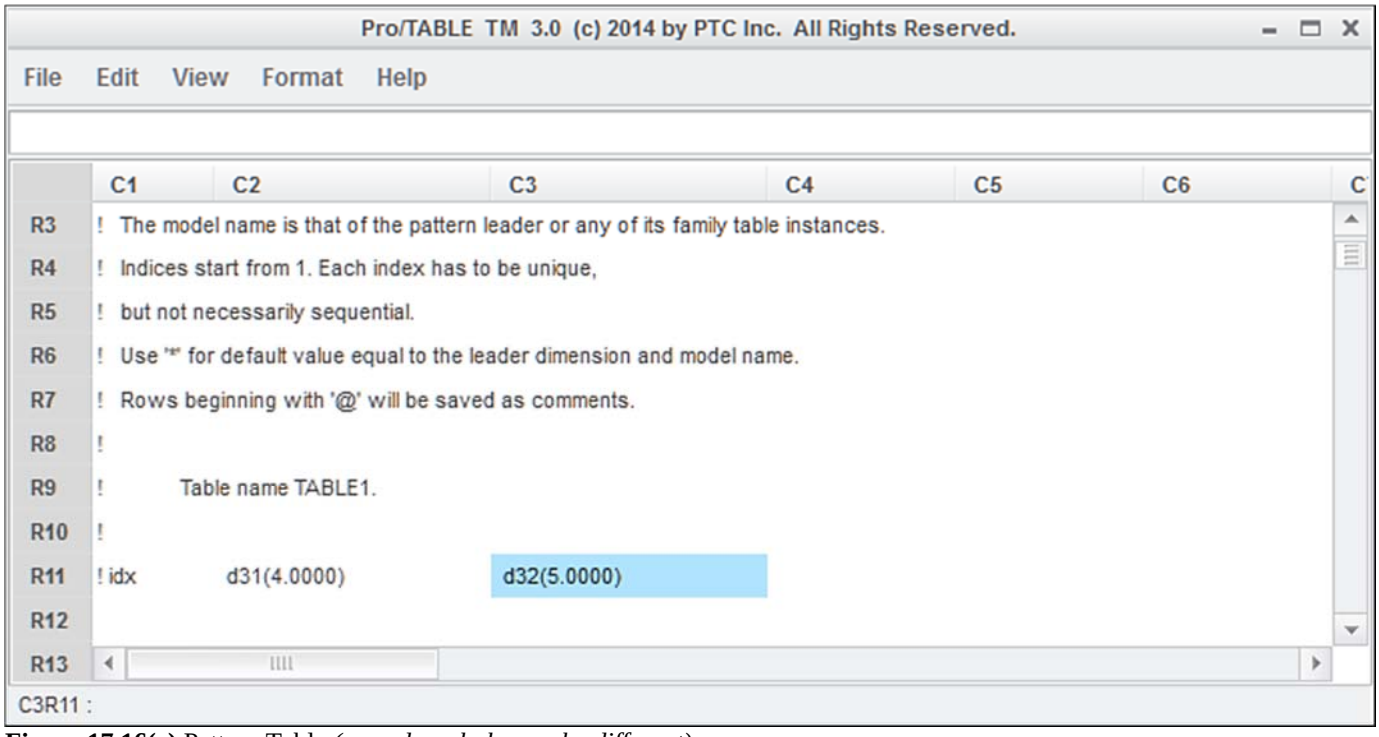


Figure 17.16(c) Pattern Table *(your d symbols may be different)*

	C1	C2	C3
R10	!		
R11	! idx	d31(4.0000)	d32(5.0000)
R12	1		
R13	2		
R14	3		
R15	4		
R16	5		
R17	6		
R18	7		

Figure 17.16(d) Add numbers 1-7

R10	!		
R11	! idx	d31(4.0000)	d32(5.0000)
R12	1	*	
R13	2	0	
R14	3	-4	
R15	4	0	
R16	5	-4	
R17	6	8	
R18	7	-8	
R19			

Figure 17.16(e) Add Values in the Second Column (***** means identical to parent value)

R11	! idx	d31(4.0000)		d32(5.0000)
R12	1	*		-5
R13	2	0	*	
R14	3	-4	*	
R15	4	0		-5
R16	5	-4		-5
R17	6	8		0
R18	7	-8		0

Figure 17.16(f) Add Values in the Third Column

From the Pro/TABLE window, click: **File** > **Exit** [Fig. 17.16(g)]

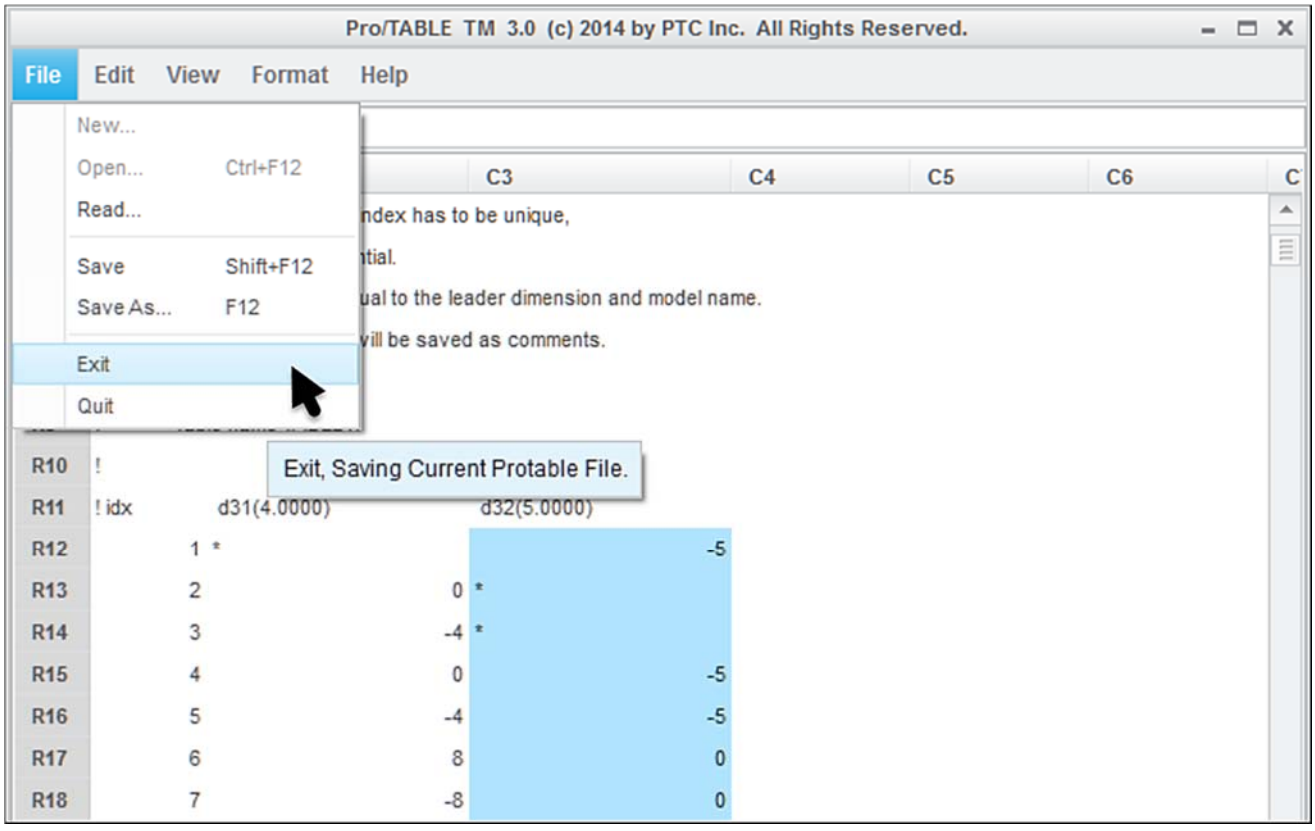


Figure 17.16(g) Completed Table

Click: [Fig. 17.16(h)] > **Ctrl+S** [Fig. 17.16(i)] > in the Graphics Window, **LMB** to deselect

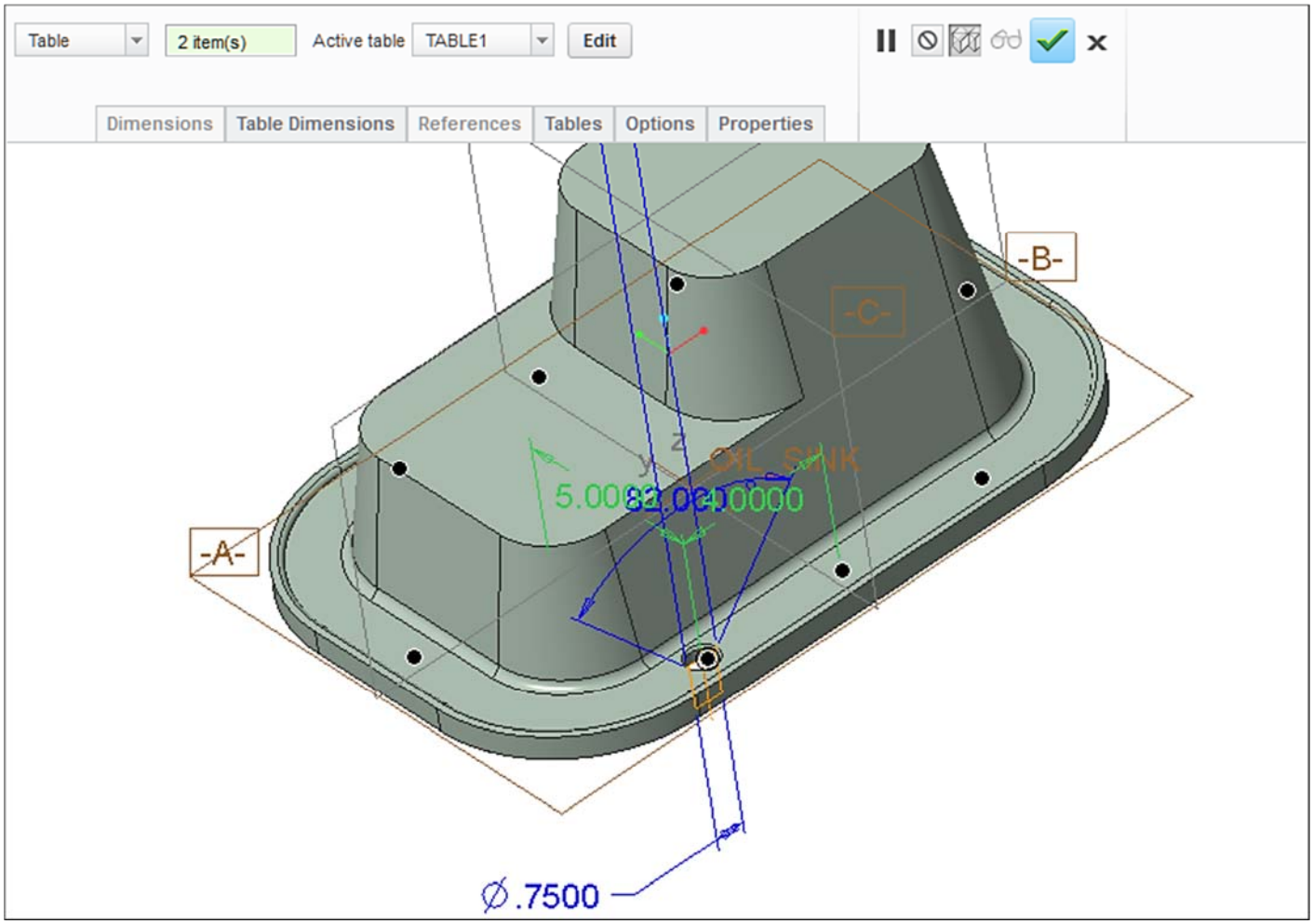


Figure 17.16(h) Previewed Pattern

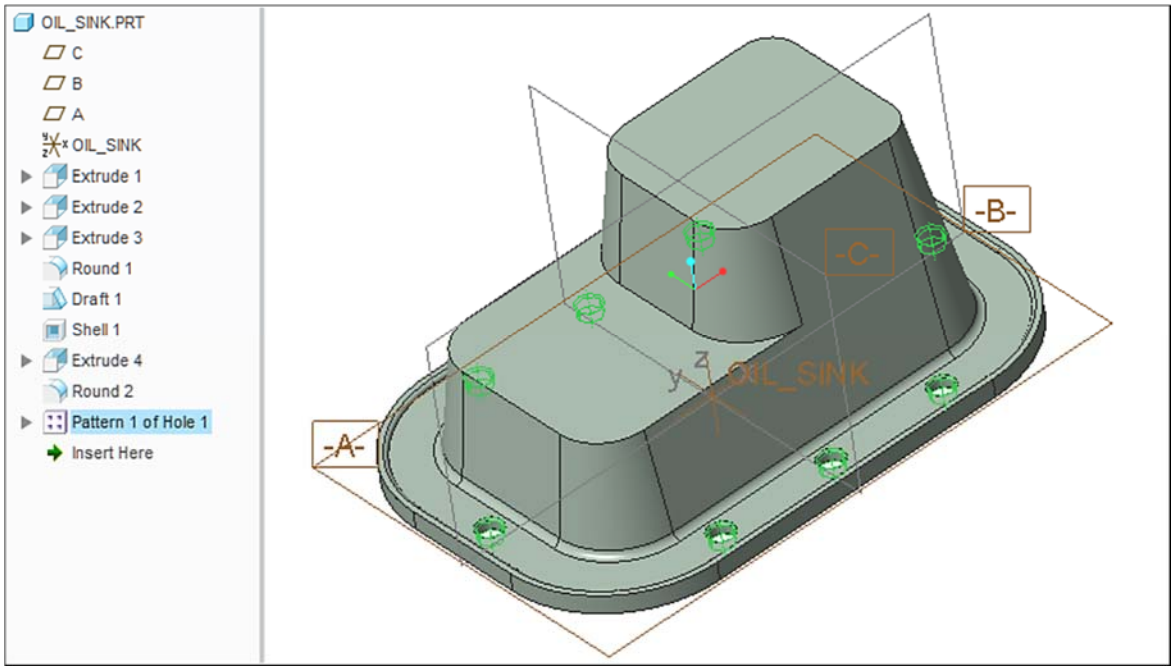


Figure 17.16(i) Completed Pattern

The next series of *features will be created purposely at the wrong stage* in this project. You will now create the **R.50** round. Because the design intent is to have a constant thickness for the part, the round should have been created before the shell. The reorder capability will be used to change the position of this round in the design sequence. Using the Model Tree, you can pick and drag the round to a new location in the feature list.

Select the top edge of the part > press **RMB** > **Round Edges** [Fig. 17.17(a)] > move a drag handle to **.50** or double-click on the dimension and *type* **.50** [Fig. 17.17(b)] > **Enter** >  [Fig. 17.17(c)] > spin the part > select the inner surface [Fig. 17.17(d)] (Notice that the rounds did not propagate to the internal edges.) > in the Graphics Window, **LMB** to deselect > **Ctrl+S**

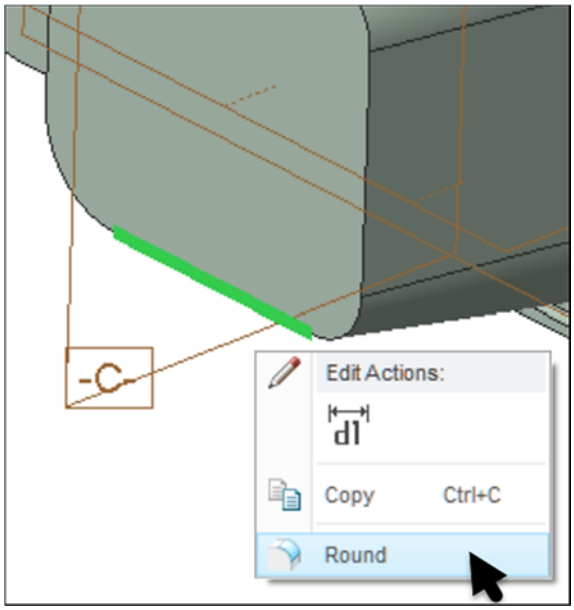


Figure 17.17(a) Add a **R.50** Round

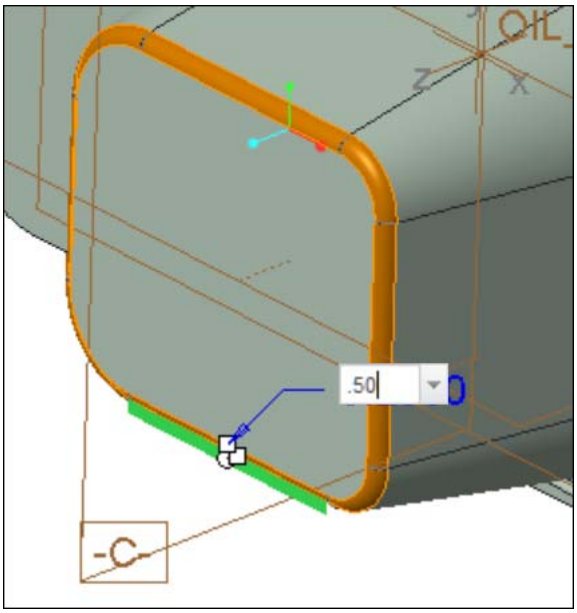


Figure 17.17(b) Move a Drag Handle to **.50** or *type* **.50** > Enter

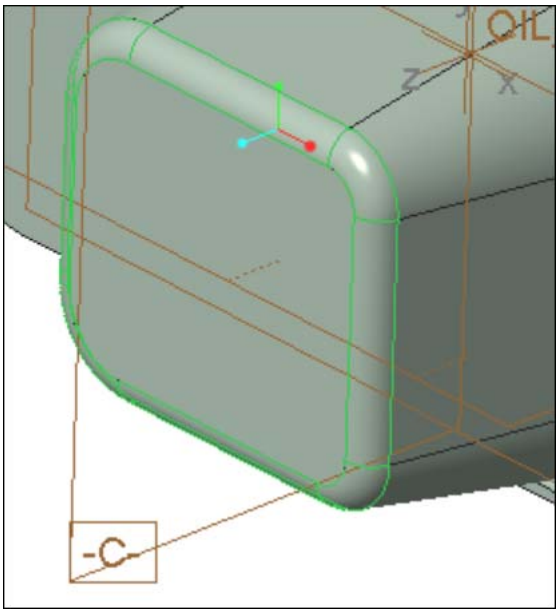


Figure 17.17(c) Completed Round

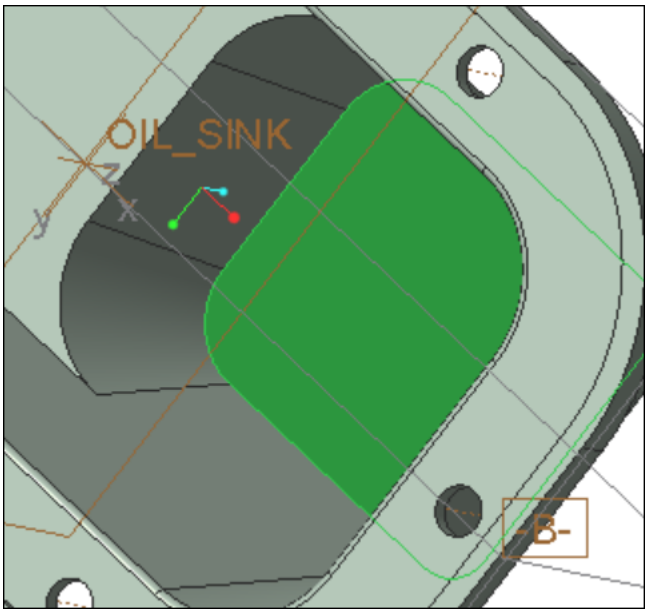


Figure 17.17(d) Interior is not Rounded

Reorder the round to appear before the Shell. Click: on the last **Round** in the Model Tree [Fig. 17.18(a)] > press **LMB** and drag **Round 3** to a position before/above the **Shell 1** feature [Fig. 17.18(b)] > release the **LMB** to drop the Round feature [Fig. 17.18(c)] > **Ctrl+S**

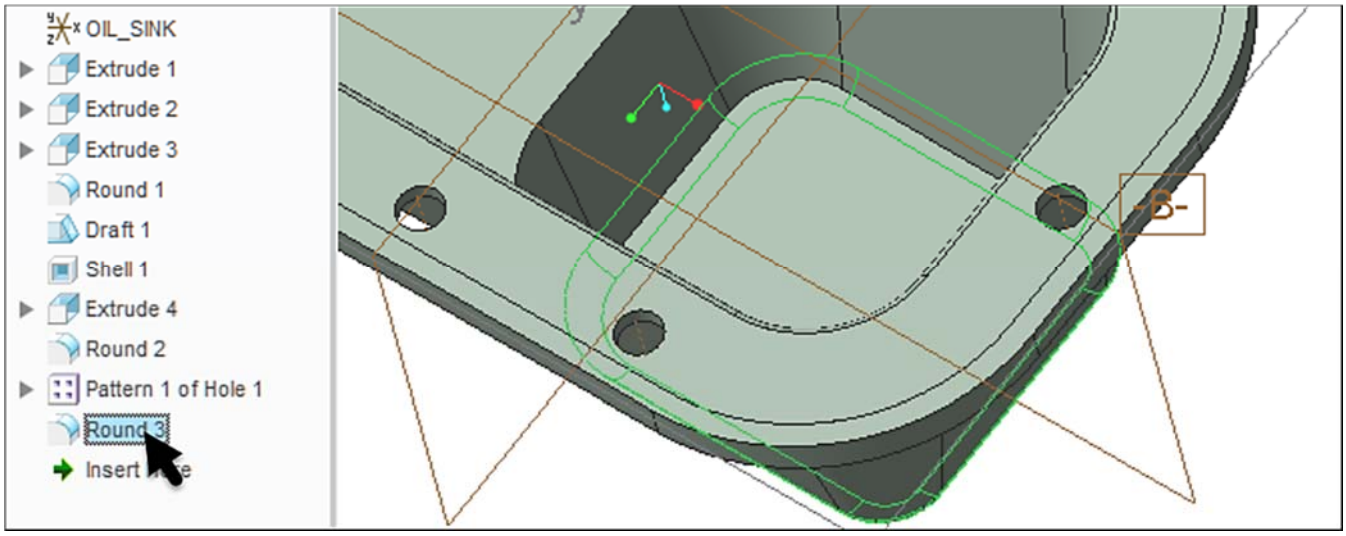


Figure 17.18(a) Click on the last Round in the Model Tree *(your Model Tree may look different)*

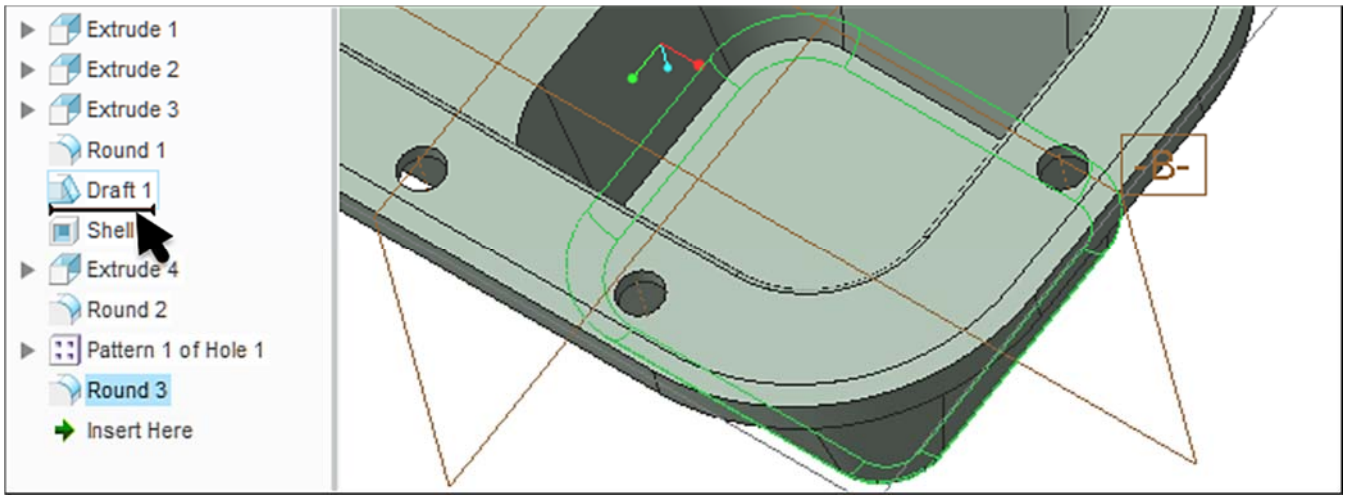


Figure 17.18(b) Move Cursor above the Shell Feature *(your Model Tree may look different)*

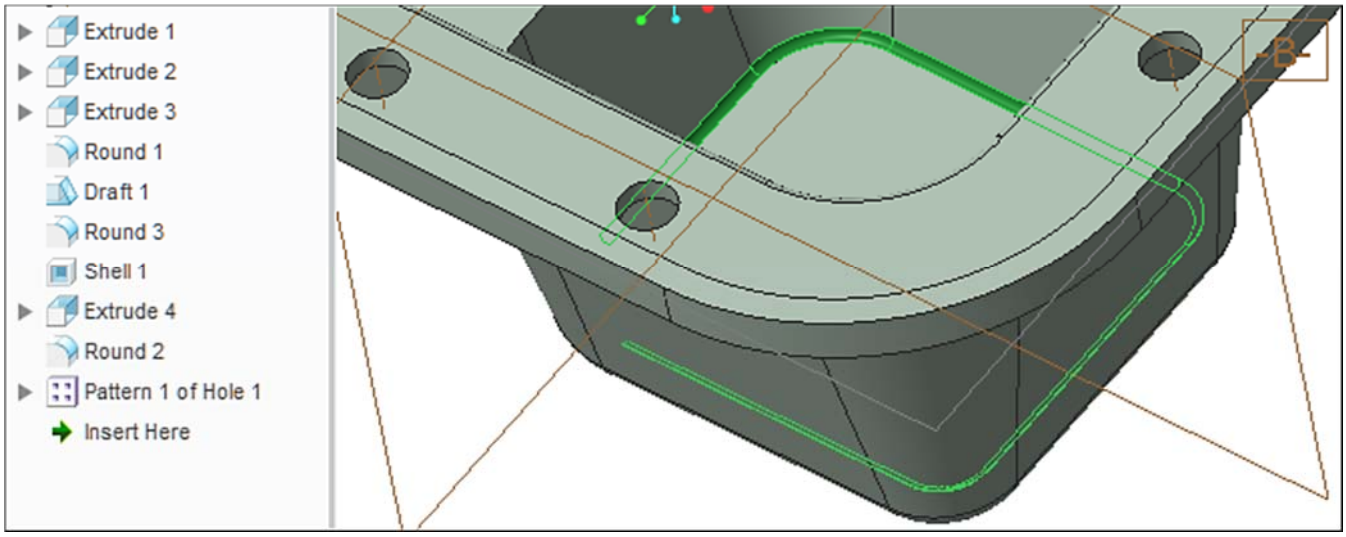
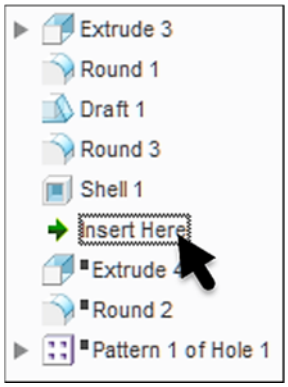


Figure 17.18(c) Reordered Round Shows on the Inside of the Part (**.50** minus the shell thickness)

You can also insert new features using the Model Tree. The arrow-shaped icon Insert Here in the Model Tree indicates where features will be inserted upon creation and is by default at the end (or bottom) of the Model Tree.

By dragging the location of the insert node higher, so that its position is before existing features, you can insert a new feature at that stage of the model history. When the *insert node* is dropped at a new location, the model is rolled backward (suppressed) or forward in response to the insertion node being moved higher or lower. The Model Tree displays a small square (■) next to the features that are not active (suppressed).



The previous round was created at the wrong stage in the design sequence and then reordered. To eliminate the reordering of a feature, the remaining **R.50** rounds will be created using Insert Mode with the Model Tree.

Insert Mode allows you to insert a feature at a previous stage of the design sequence. This is like going back into the past and doing something you wish you had done before--not possible with life, but with Creo Parametric 3.0 less of a problem. Add the additional **R.50** rounds.

(Your Model Tree may look different.)

Spin the model > **Ctrl+R** > in the Model Tree, place the pointer on Insert Here [Fig. 17.19(a)] > press and hold down the **LMB** > move (drag) the pointer to a position before/above the **Shell 1** feature > release the **LMB** to drop Insert Here [Fig. 17.19(b)]

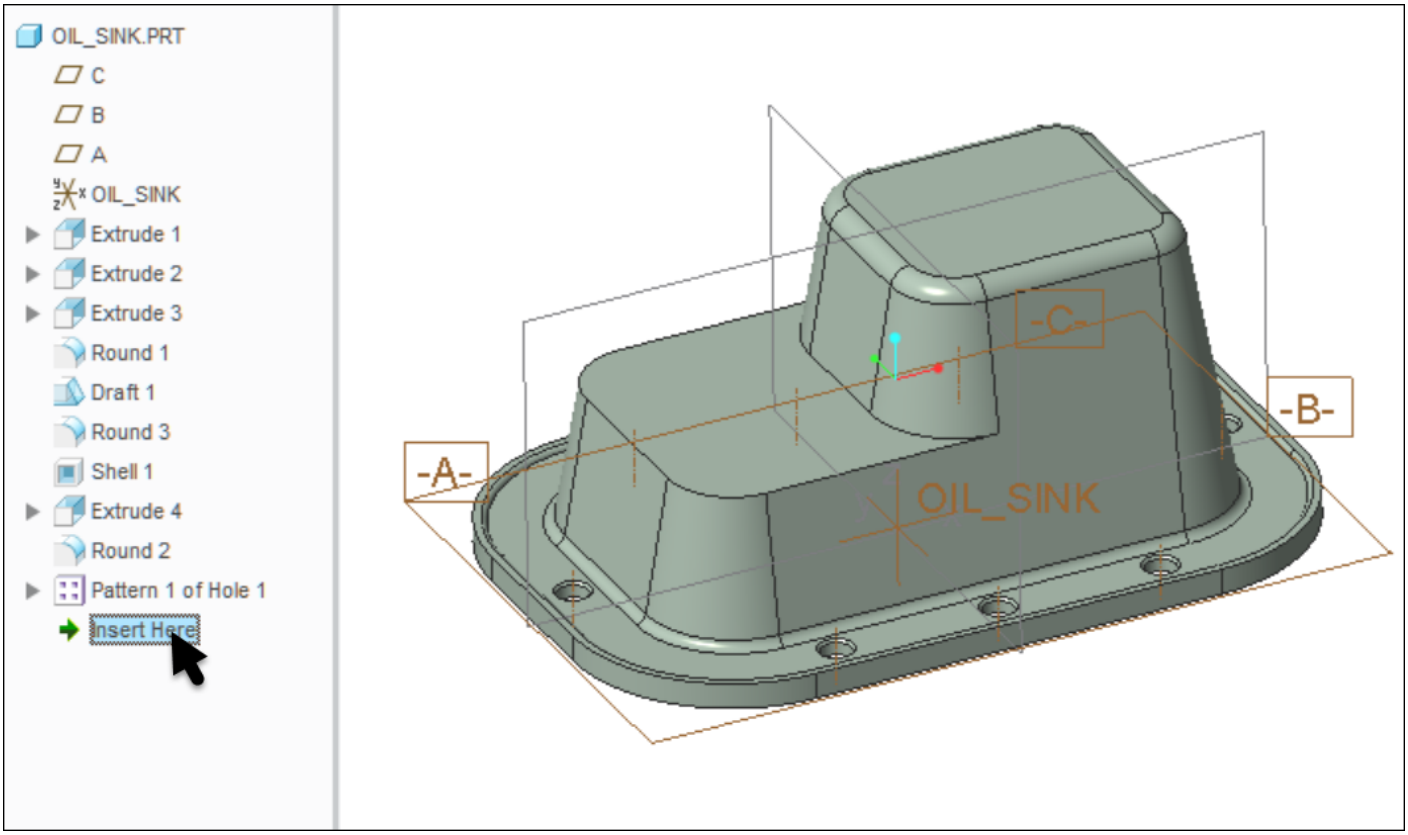


Figure 17.19(a) Place Pointer on Insert Here *(your Model Tree may Look Different)*

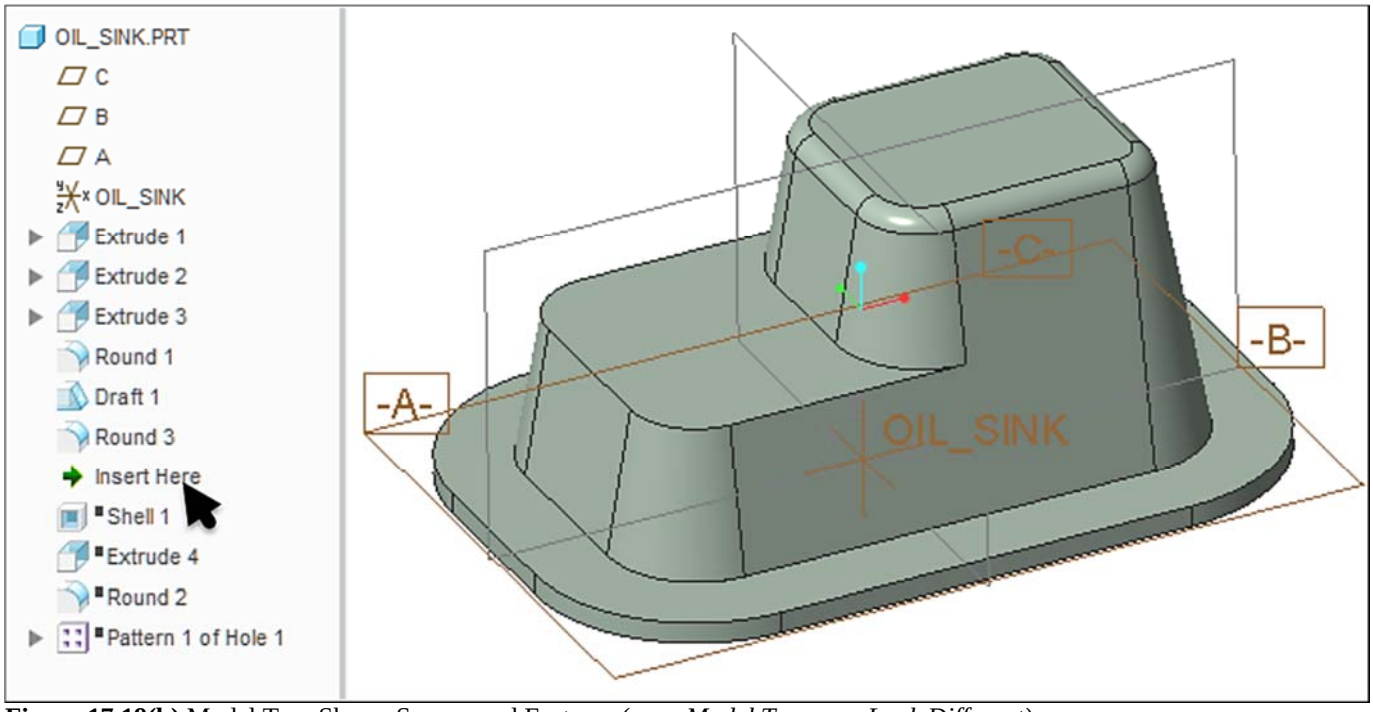


Figure 17.19(b) Model Tree Shows Suppressed Features (*your Model Tree may Look Different)*

Create two separate round features. Click: **Round** > select the upper edge > Radius **.50** > **Enter** [Fig. 17.20(a)] > > **Ctrl+C** > **Ctrl+V** > select the front edge [Fig. 17.20(b)] > > press the **Ctrl** key > select the previously created round features from the Model Tree > release the **Ctrl** key [Fig. 17.20(c)]

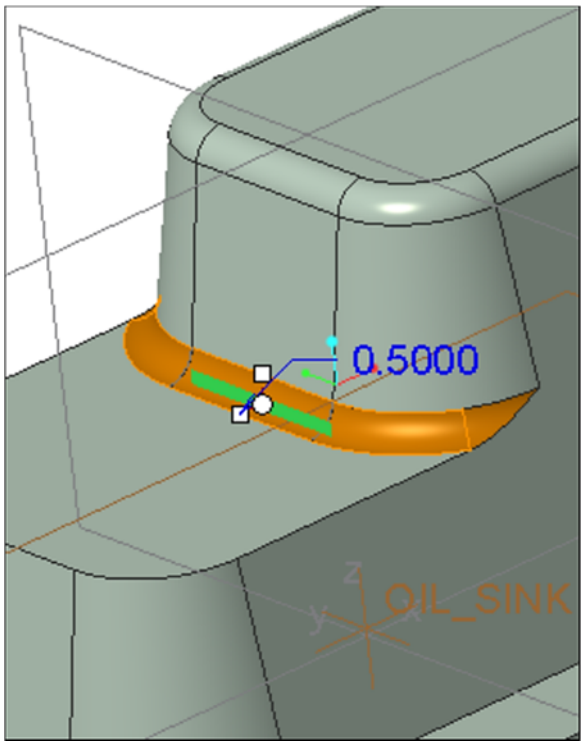


Figure 17.20(a) First Round

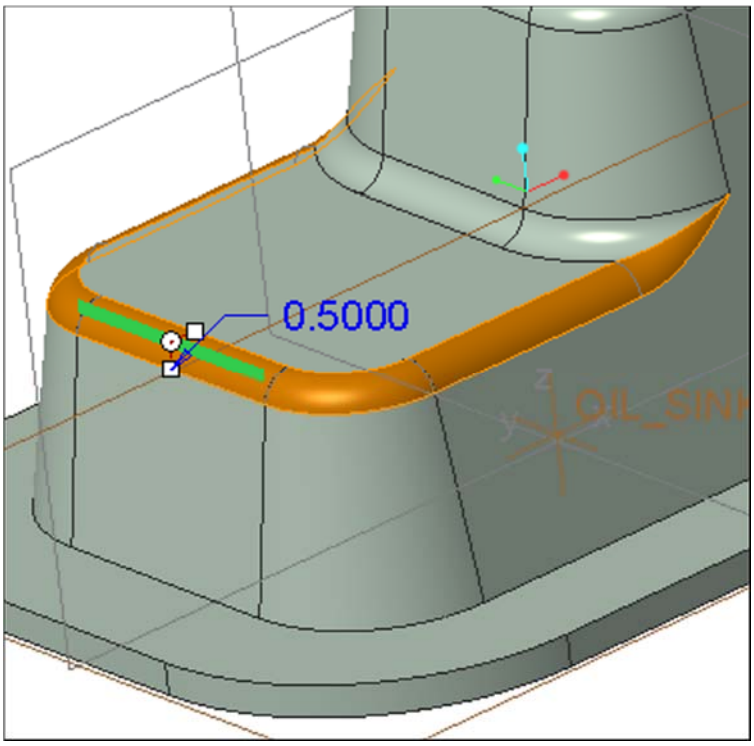


Figure 17.20(b) Second Round

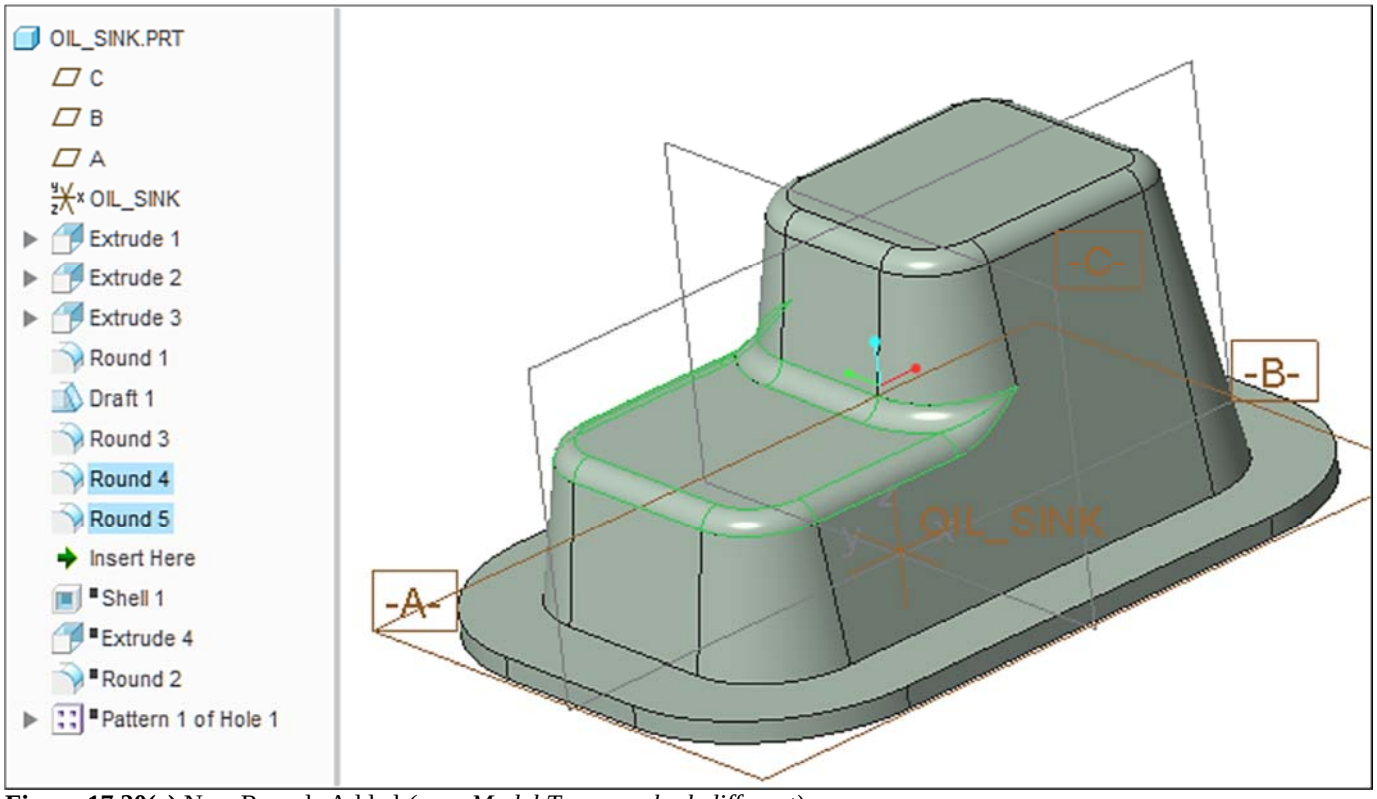


Figure 17.20(c) New Rounds Added *(your Model Tree may look different)*

Rotate the model > in the Model Tree, place the pointer on Insert Here > press and hold down the **LMB** [Fig. 17.21(a)]

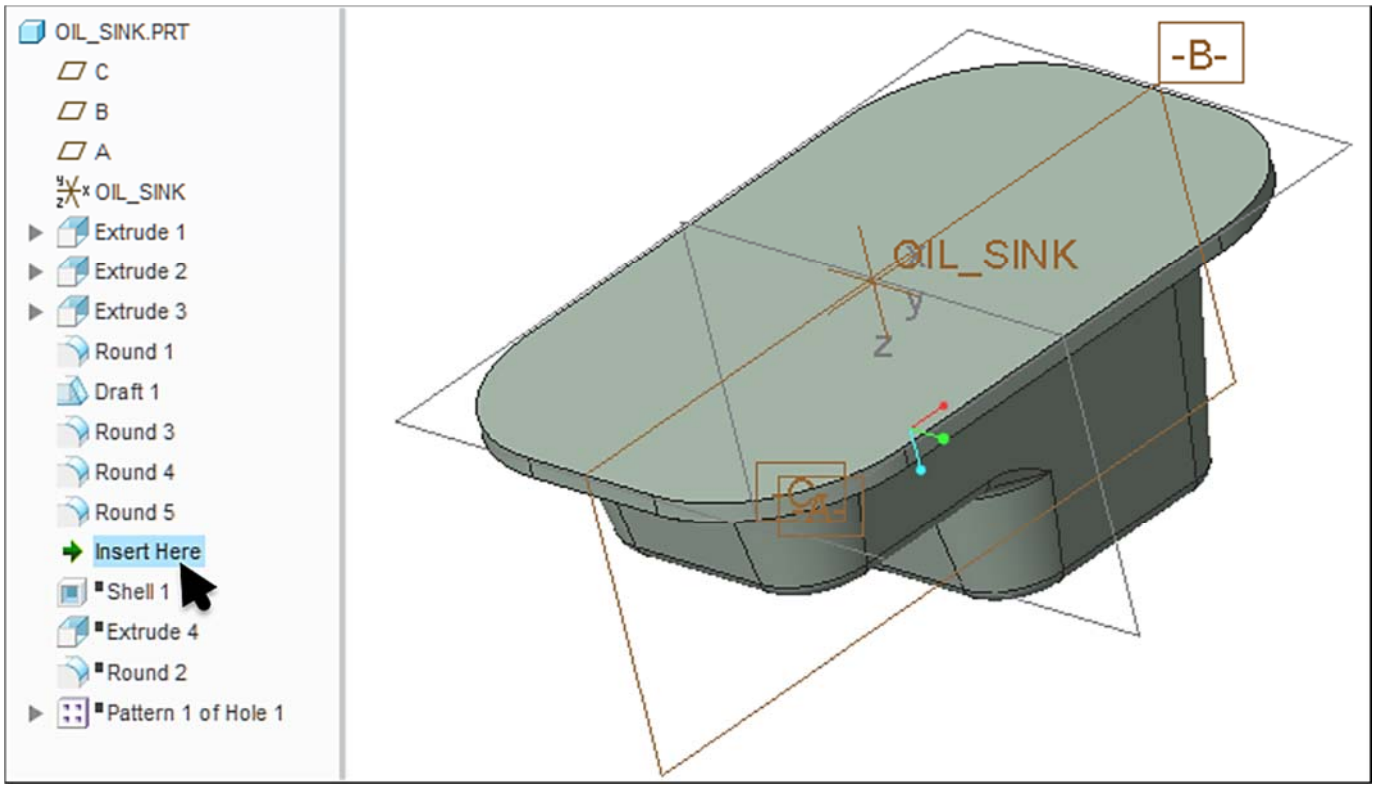


Figure 17.21(a) Rotate the Model *(your Model Tree may Look Different)*

Drag the pointer to the bottom of the Model Tree list > release the **LMB** to drop Insert Here [Fig. 17.21(b)] > select the propagated internal round surfaces in the model [Fig. 17.21(c)] > set the model display as **Isometric > Ctrl+D > Ctrl+S > File > Manage File > Delete Old Versions > Enter**

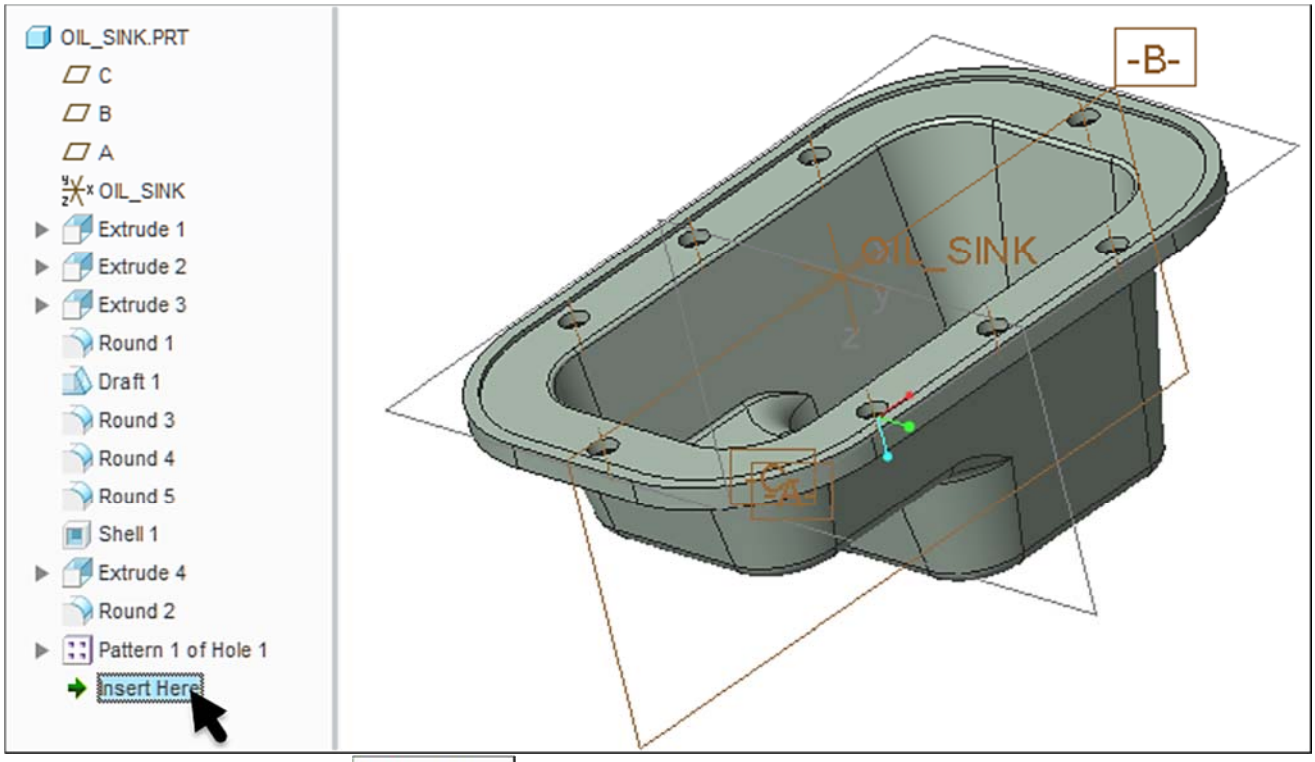


Figure 17.21(b) Drag and Drop Insert Here, All Features are Resumed *(your Model Tree will look different)*

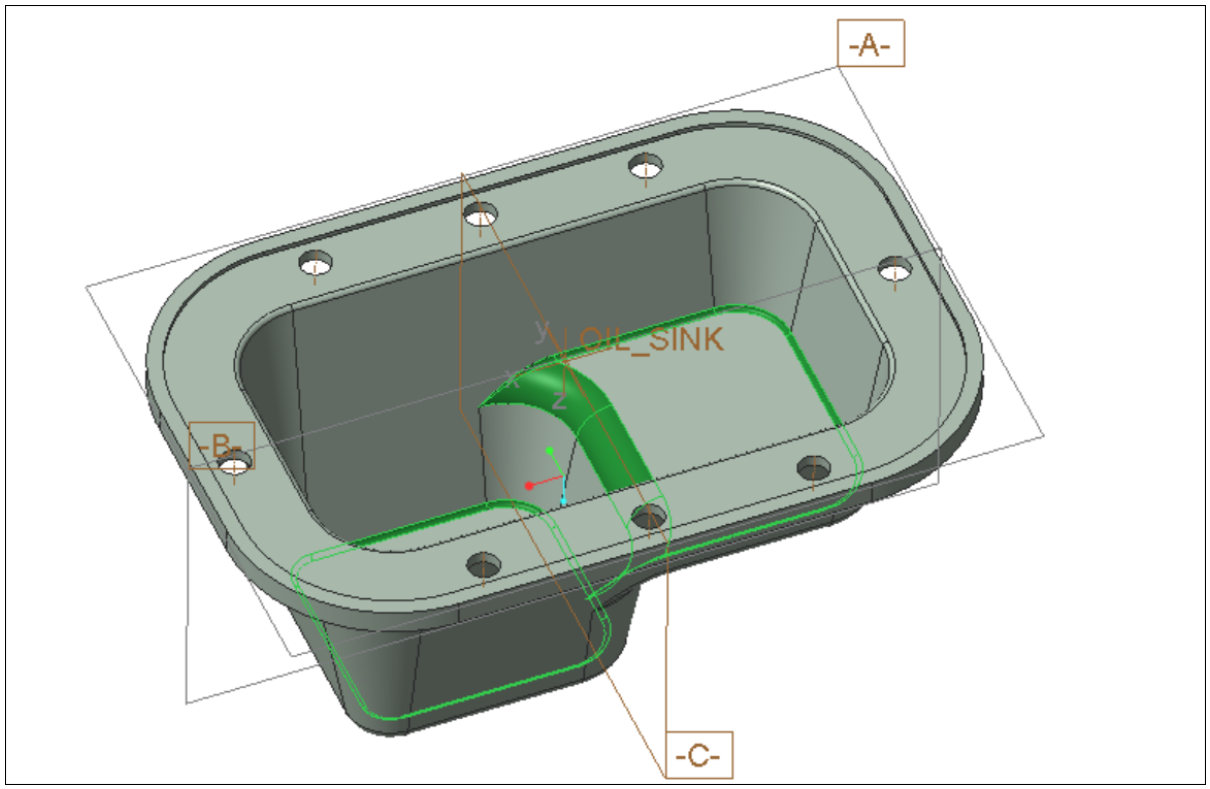
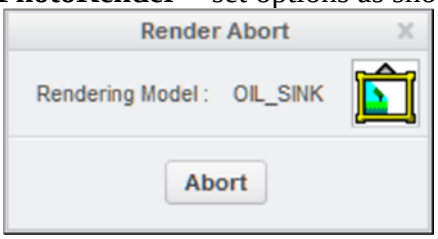
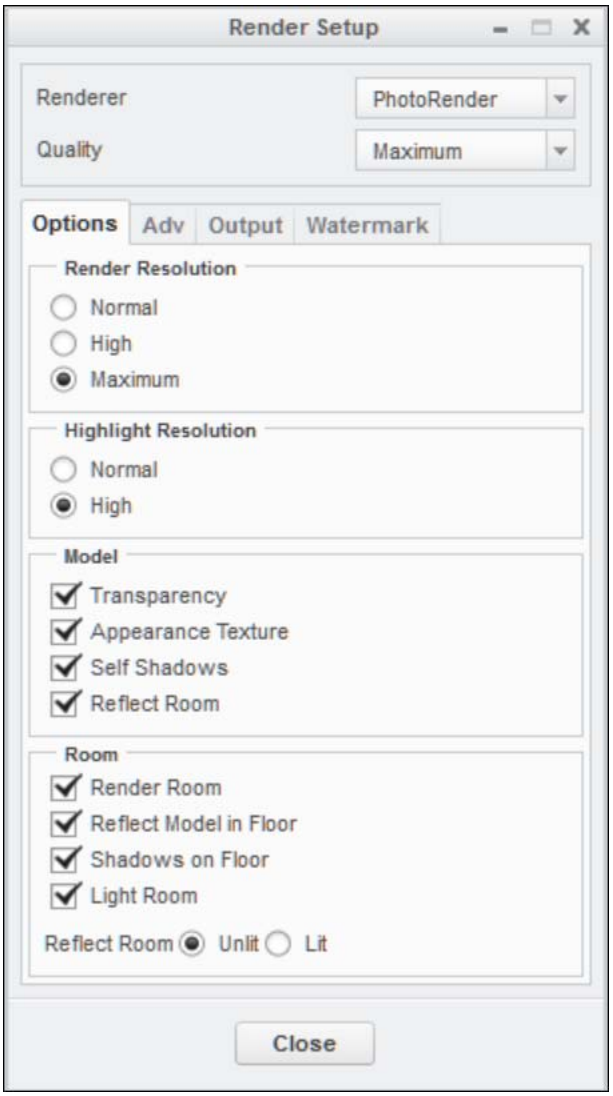


Figure 17.21(c) Propagated Internal Rounds

Hide datum **A, B and C** >  Model Tree *off* > **View** tab > **Display Style** > **Shading** >   *off* > hide the datums in the Model Tree > **Render** tab > **Render Setup** > Renderer  > **PhotoRender** > set options as shown [Fig. 17.22(a)] > **RENDER Window** >



> **OK** > **Ctrl+R** > **Ctrl+D**



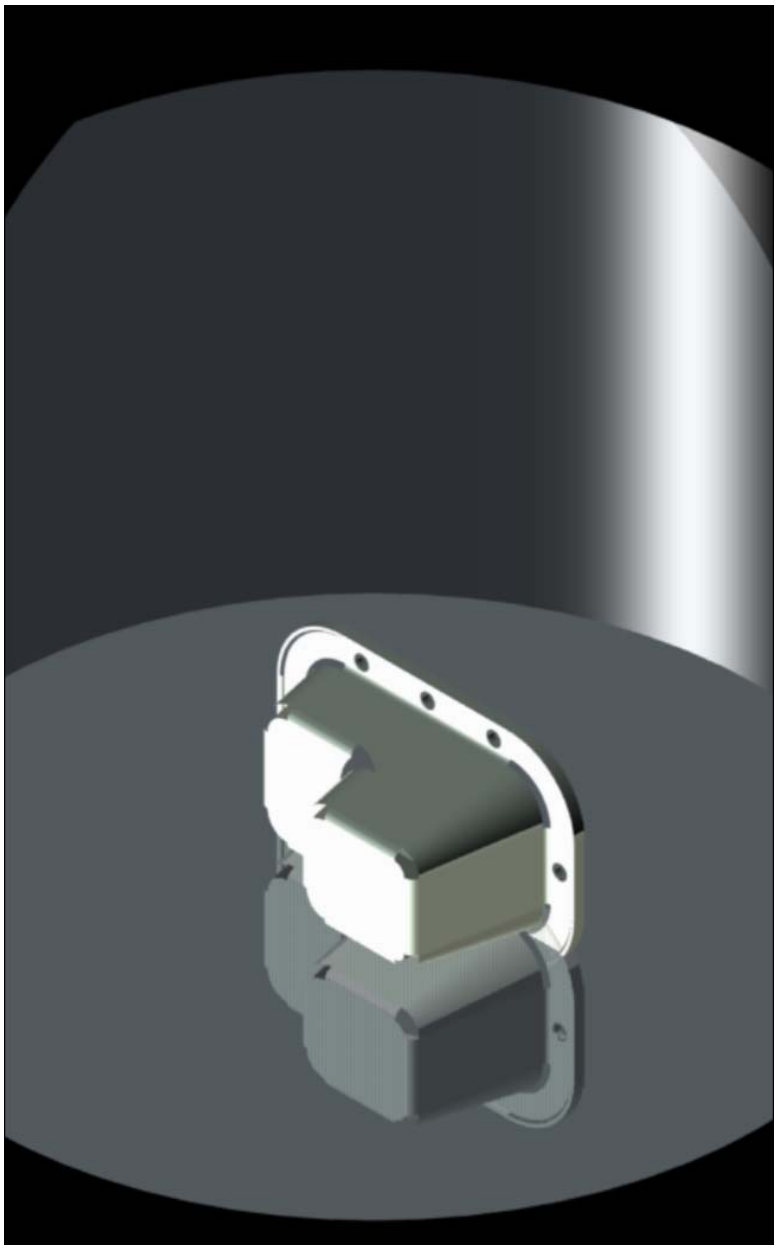


Figure 17.22(a) Render

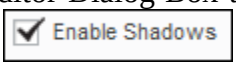
Click: **Scene** > **Lights** tab >  **Add new spotlight** > Name  **Color for lighting** > adjust the slide bars in the Color Editor Dialog Box to the RGB values you desire [Fig. 17.22(b)] > **OK** (from the Color Editor Dialog Box) >  > 

Figure 17.22(b) New Spot Light *(your display may appear differently)*

Click: **Add new distance light** > Name **Color for lighting** > adjust the slide bars in the Color Editor Dialog Box to the RGB values you desire > **OK** (from the Color Editor Dialog Box) > move the light to a new position > Enable Shadows > Show Light [Fig. 17.22(c)]

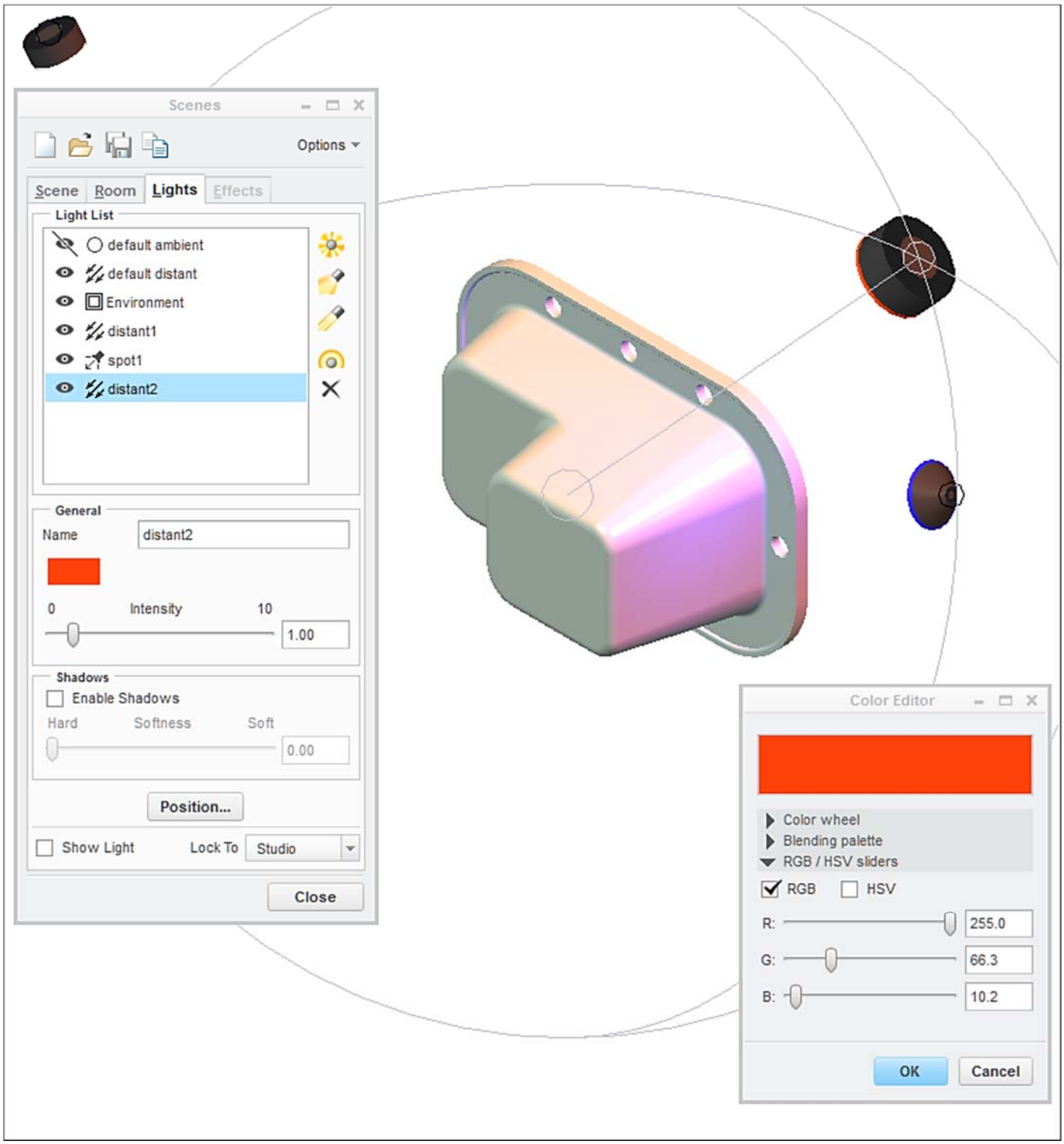


Figure 17.22(c) New Distant Light *(your display may appear differently)*

Click: **Add new skylight** > Name **Color for lighting** > adjust the slide bars in the Color Editor Dialog Box to the RGB values you desire > **OK** (from the Color Editor Dialog Box) > move the skylight to a new position > Enable Shadows > Show Light [Fig. 17.22(d)] > **Close** the Scenes Dialog Box

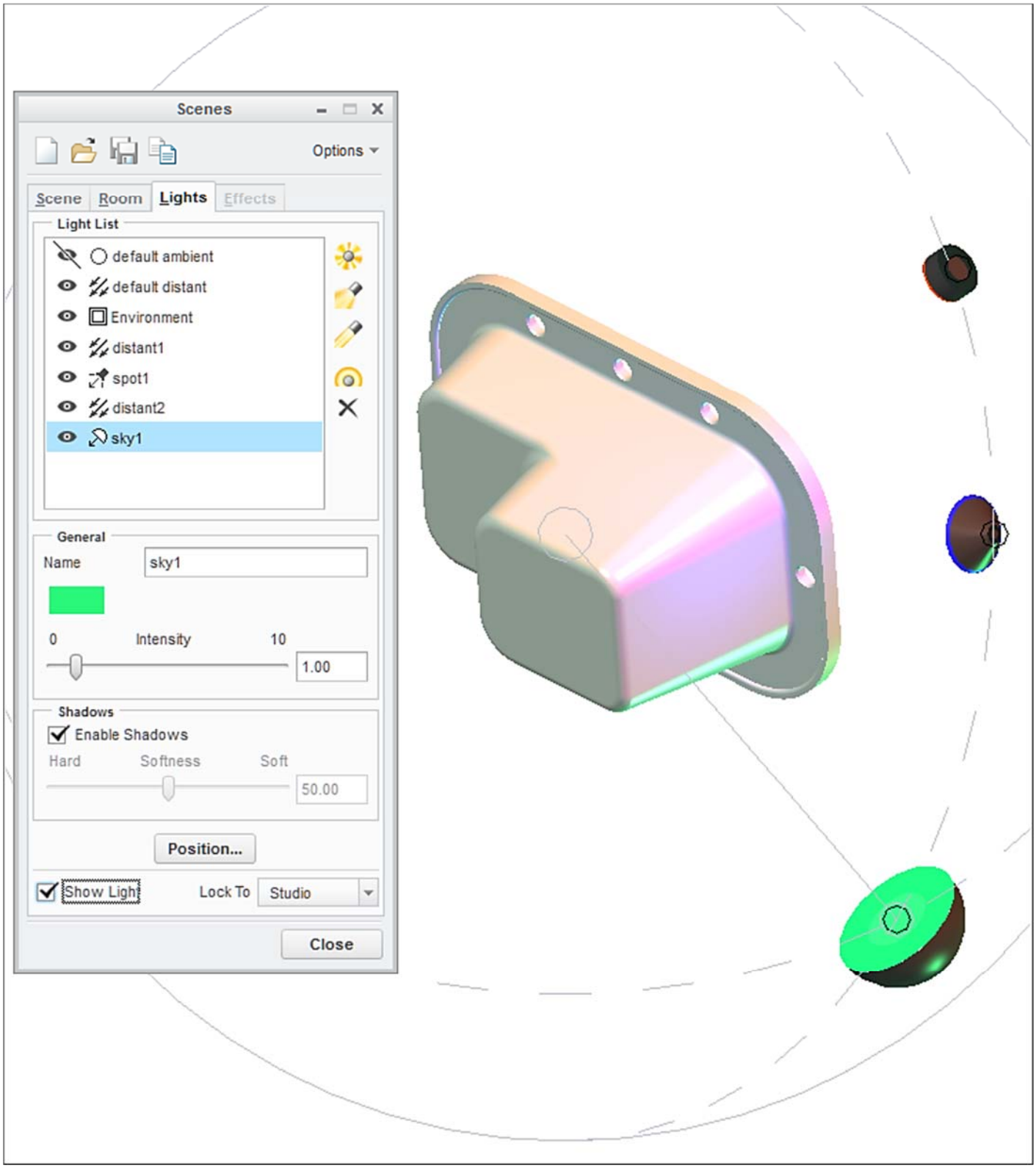


Figure 17.22(d) New Sky Light *(your display may appear differently)*

Shading With Reflections [Fig. 17.22(e)] > **Ctrl+S**

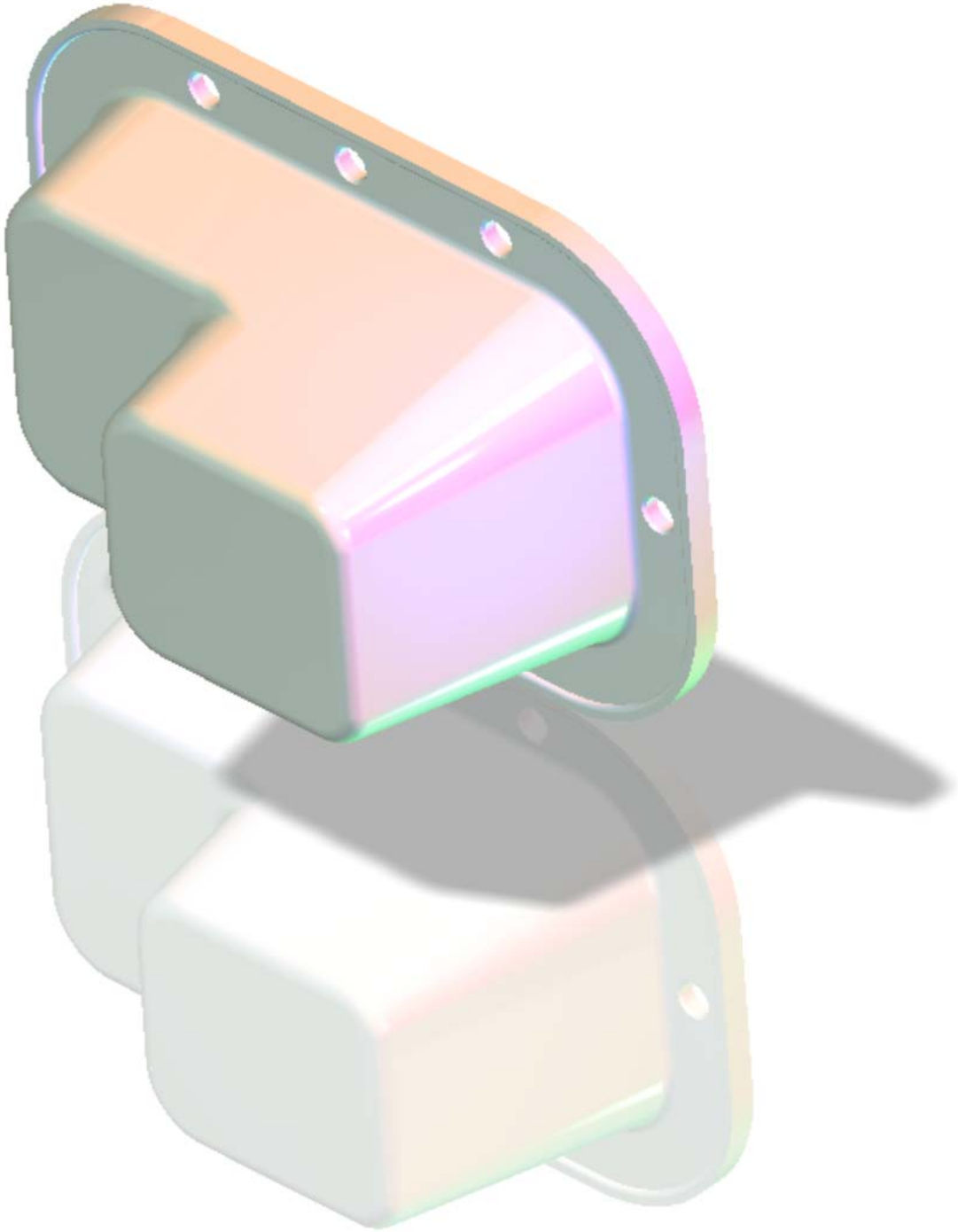


Figure 17.22(e) Shading With Reflections *(the quality of your graphics card and graphics settings may prevent this display)*

Click: **File** > **Save As** > **Save a Copy** > Type **PDF U3D (*.pdf)** [Fig. 17.23(a)] > **OK** [Fig. 17.23(b)] > **OK**

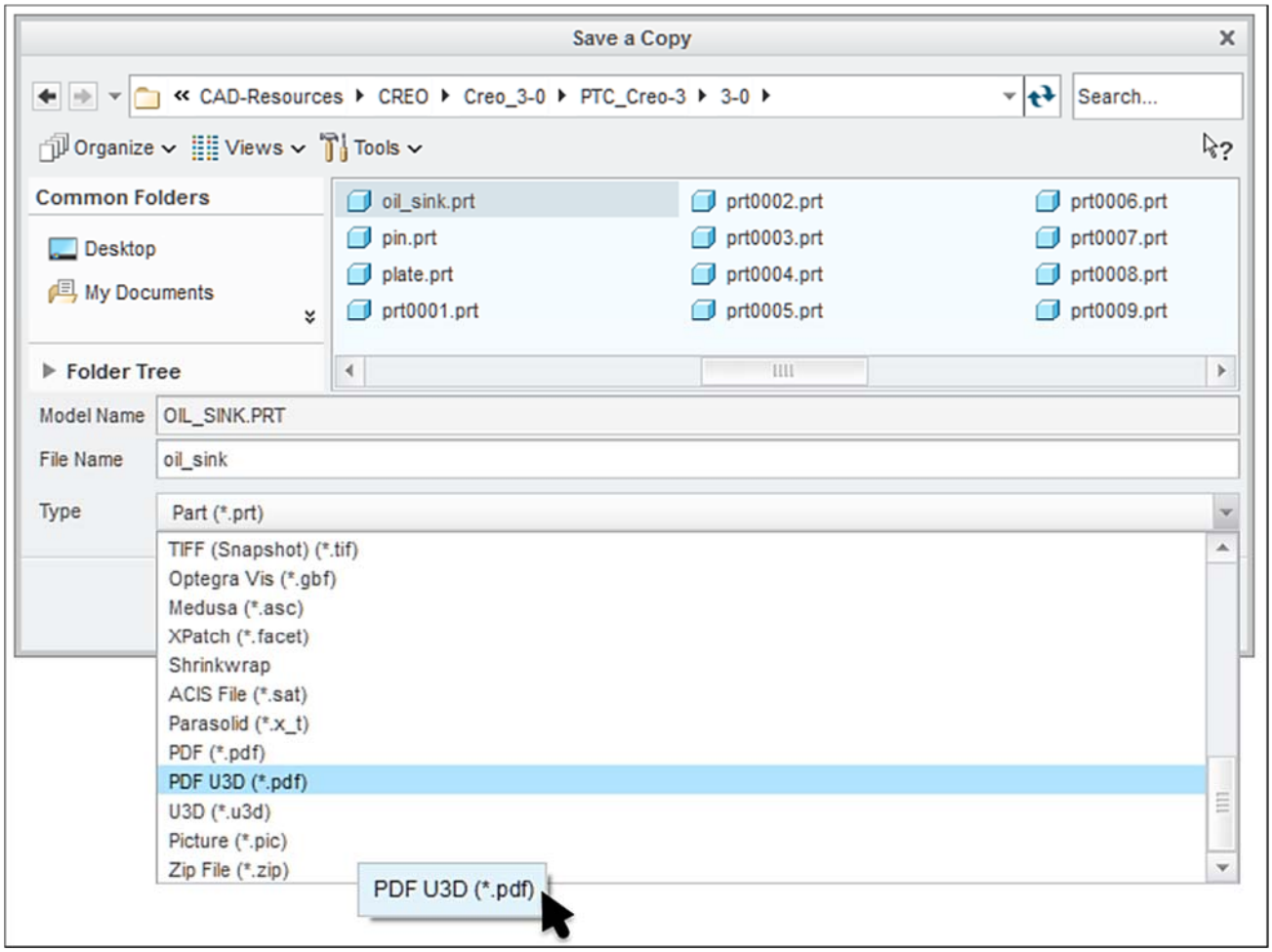


Figure 17.23(a) PDF U3D (*.pdf)

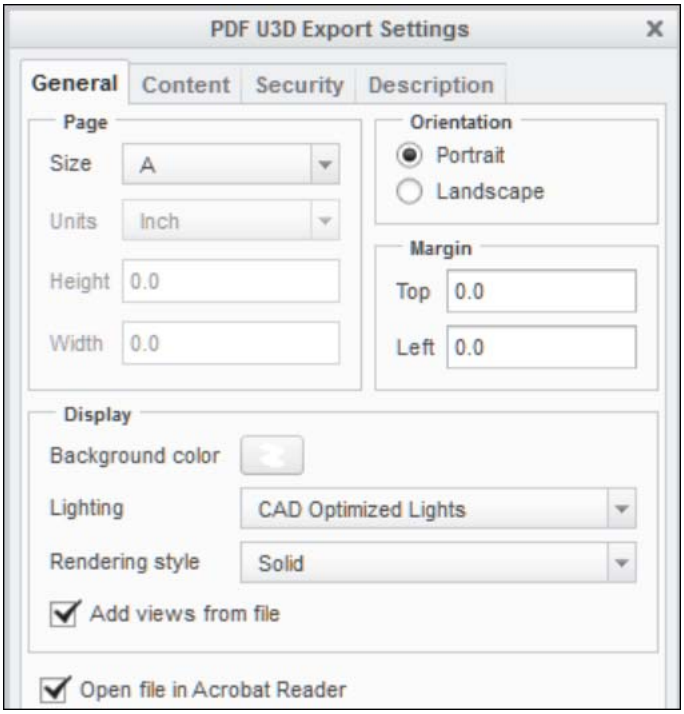


Figure 17.23(b) PDF U3D Export Settings

If needed- Click to activate > double-click on the model in the PDF [Fig. 17.23(c)]

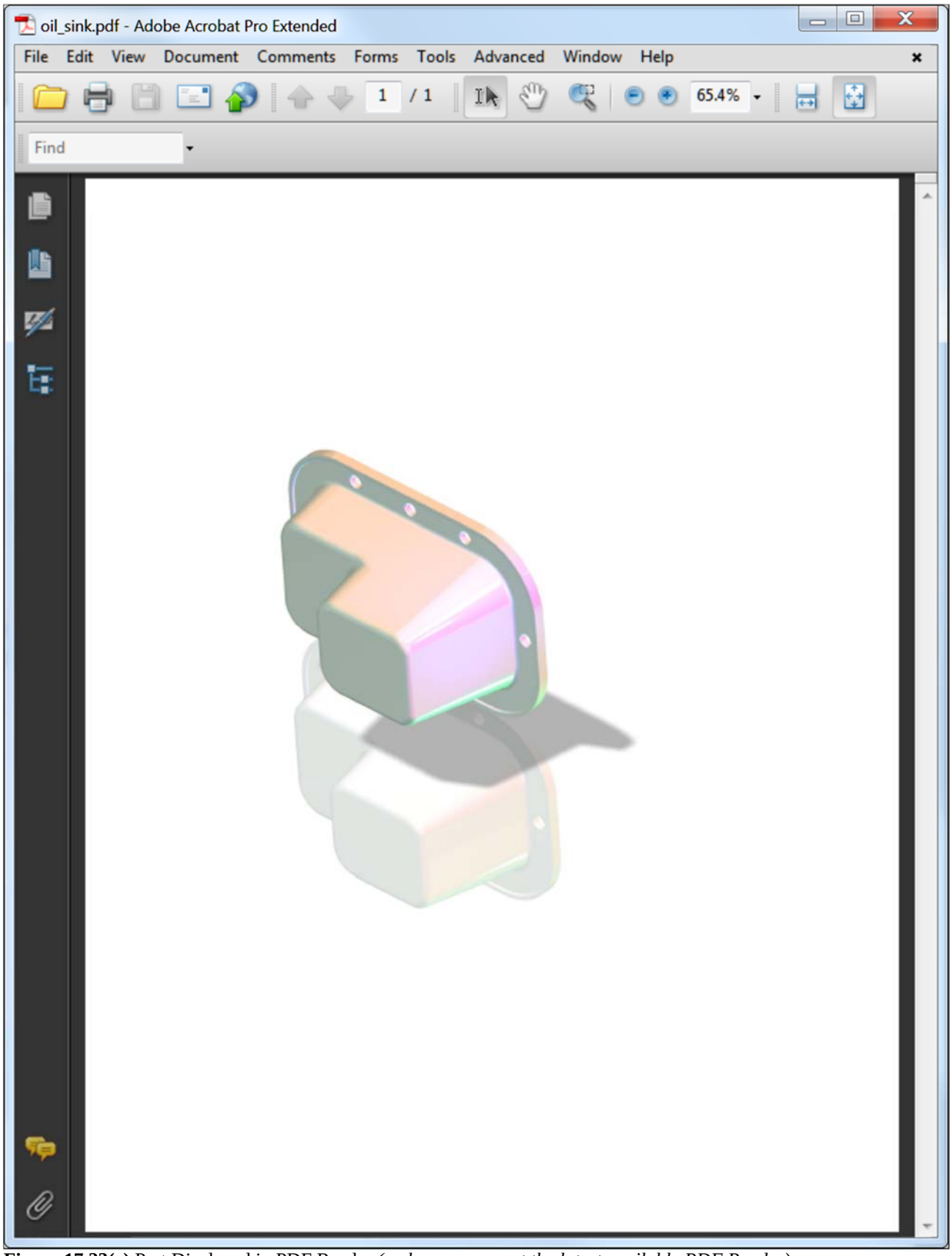


Figure 17.23(c) Part Displayed in PDF Reader *(make sure you get the latest available PDF Reader)*

Click: **View > Navigation Panels > Show Navigation Pane >** **Toggle Model Tree** [Fig. 17.23(d)]

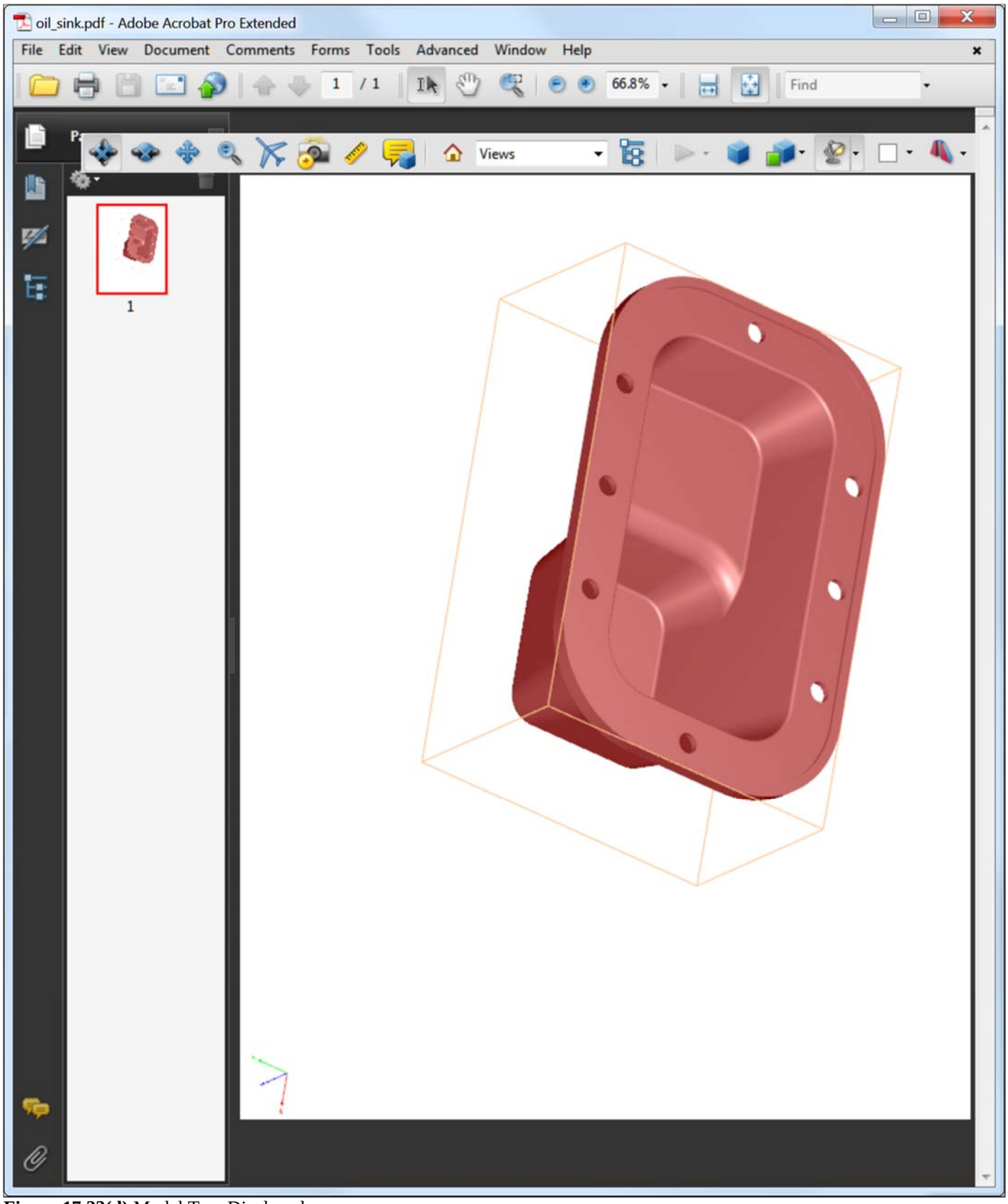


Figure 17.23(d) Model Tree Displayed

Click: **Use Perspective Projection** > *off* > **Background Color** (select a different color) > experiment with different options > **Toggle Cross Section > Y-Axis > Save Section View** [Fig. 17.23(e)] > **File > Properties** > fill in the information as desired > **OK > File > Save As > oil_sink > Save > Yes > Exit > File > Exit** > email/upload as required

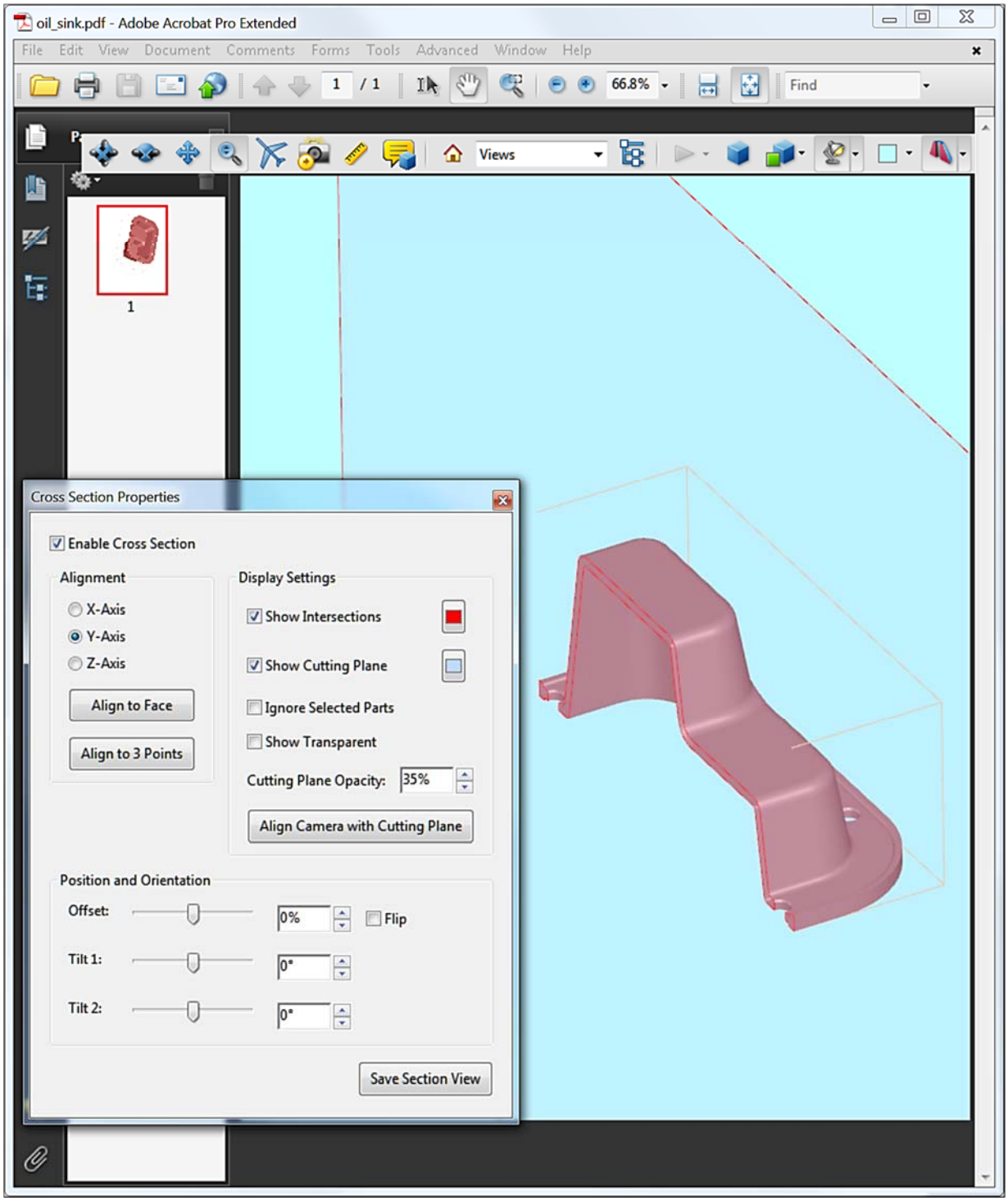
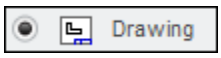


Figure 17.23(e) Cross Section

Click:  > 

> Name **oil_sink** (do not use a space in the name) > 

> **OK** > **OK** > double-click on the **SIZE: C** tag in the lower left corner of the Graphics Window 

> Sheet 1 **C Size** >  > **Browse** > **d.frm** > **Open** > **Preview** > **Open** > **OK** > double-click on the **SCALE:** tag in the lower left corner of the Graphics Window > *type* **.50** > **Enter** > from the Ribbon, 

off > rearrange the views [Fig. 17.24(a)] > **View** tab >   all *off* > in the Model Tree, press the **Ctrl** key and select datums **A**, **B**, and **C** > **RMB** > **Unhide in Model** > **LMB** > **File** > **Prepare** > **Drawing Properties** > Detail Options **change** > Option: *type* **gtol** > **Enter** > Value:  > **std_asme** > **Add/Change** > **Apply** > **Close** > **Close** > **Ctrl+S** > **OK**

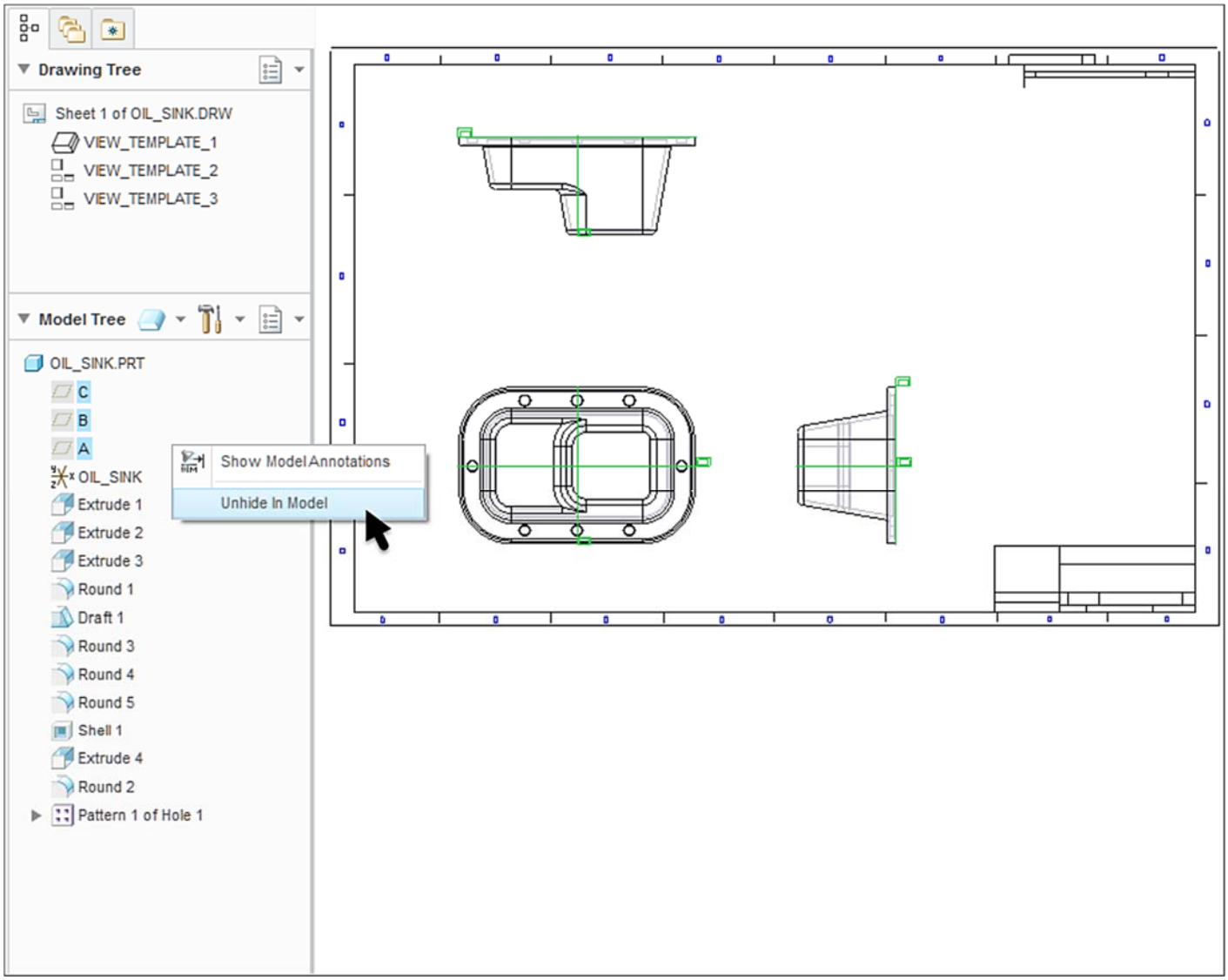


Figure 17.24(a) Oil Sink Drawing

Click: **Annotate** tab > from the Ribbon, **Show Model Annotations** > select the front view > hold down the **Ctrl** key > select the top view > release the **Ctrl** key > tab > (select all) > **Apply** > tab > (select all) > **Apply** > (close the dialog box) > in the Drawing Tree, expand Annotations [Fig. 17.24(b)] > **Layout** tab > select the right view > press **RMB** > **Delete** > rearrange the remaining two views to fit the sheet > Change the sheet size, add views and sections to completely describe the part. Erase, delete, and reposition the axes and annotations as per ASME standards [Figs. 17.24(c-d)]. Use additional sheets as needed.

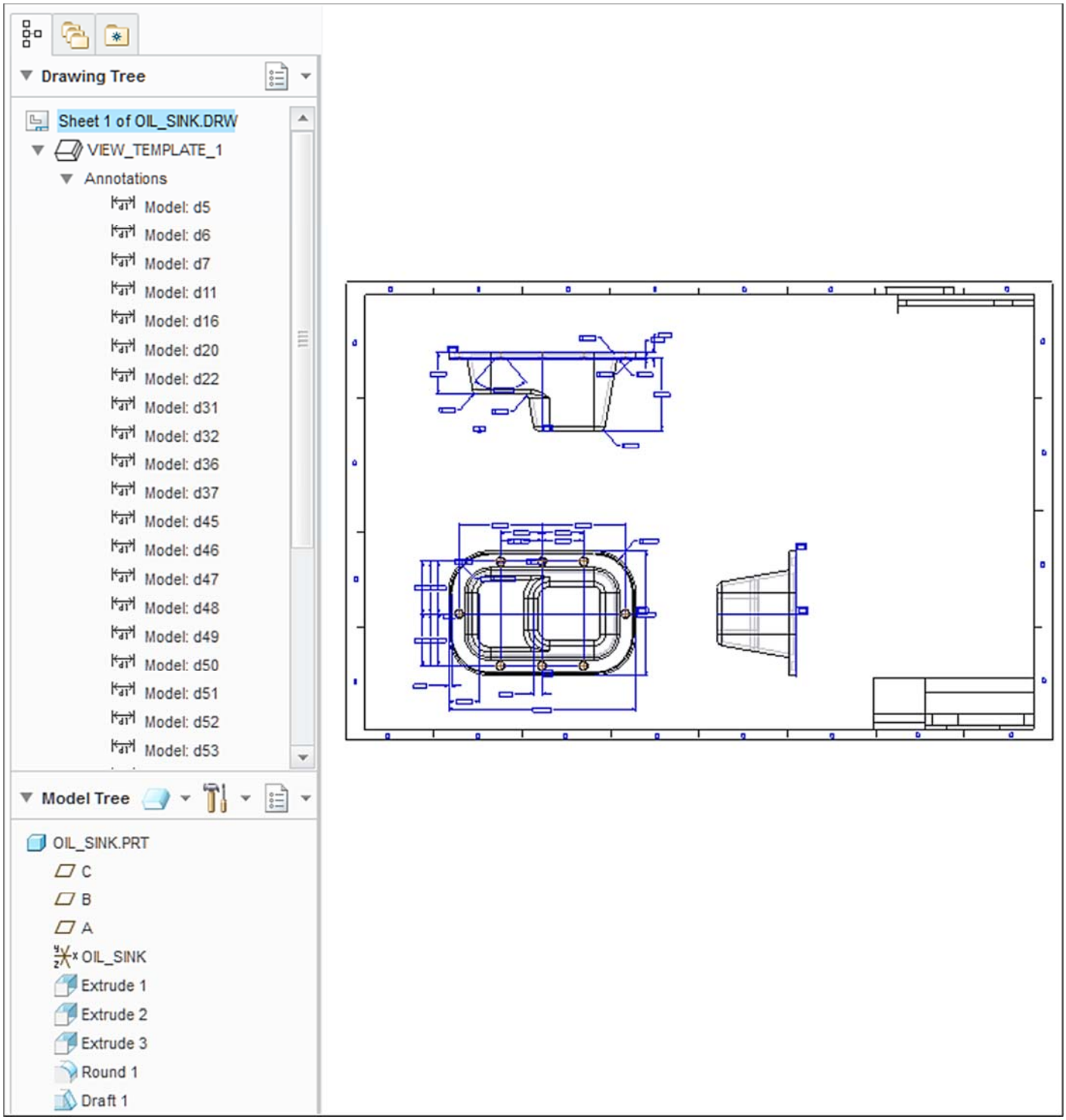


Figure 17.24(b) Drawing Annotations

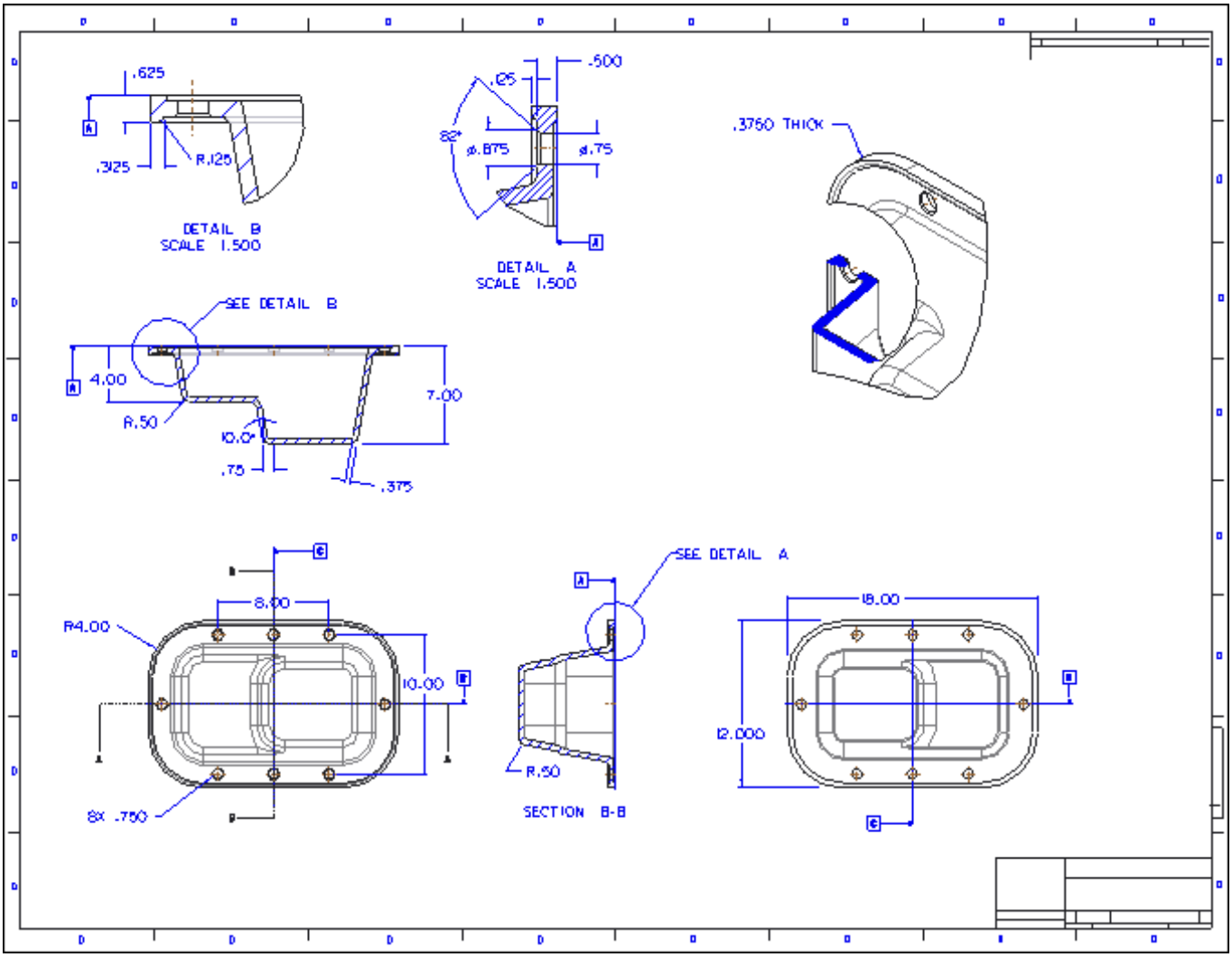


Figure 17.24(c) Oil Sink Detail Drawing, Sheet 1

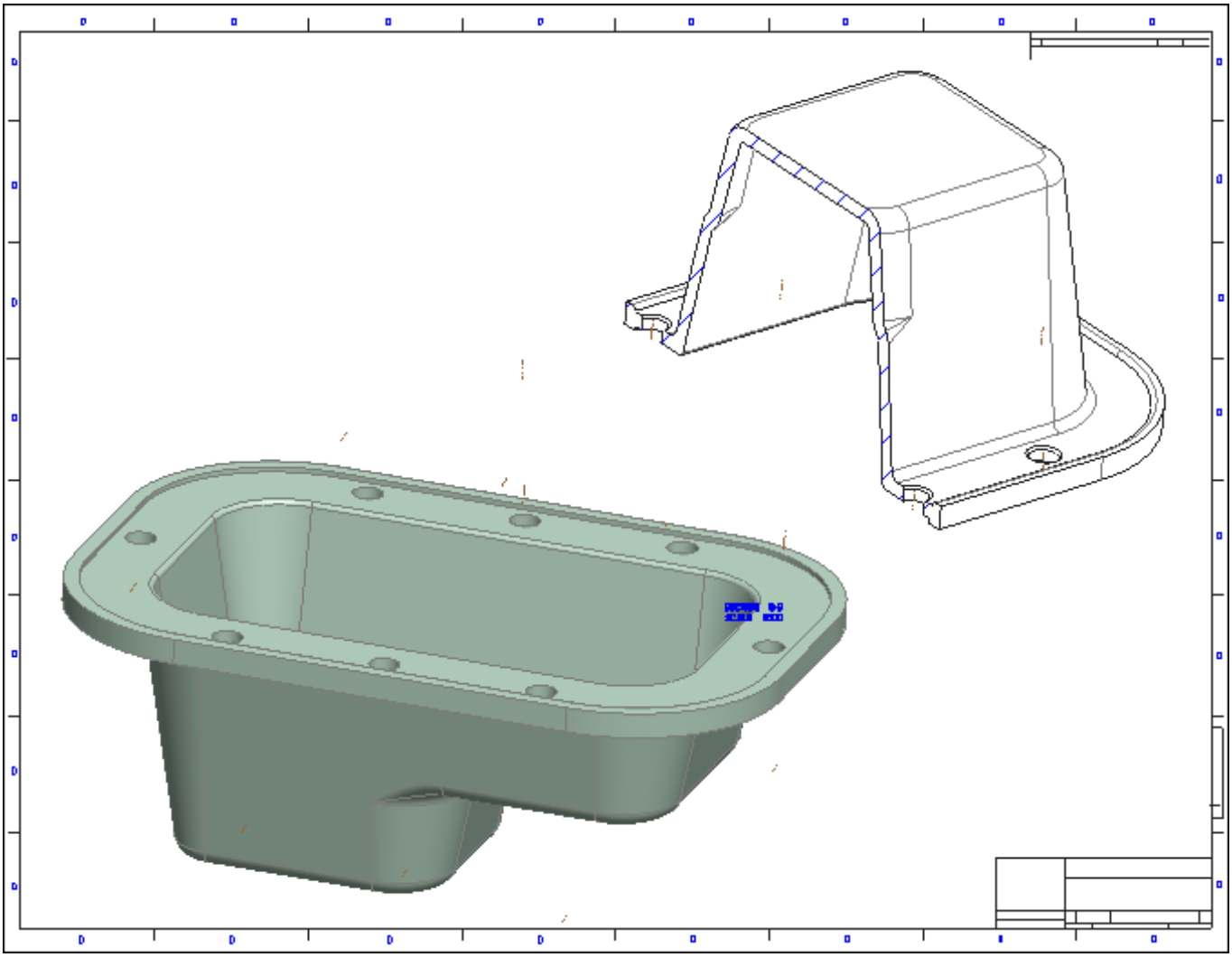


Figure 17.24(d) Oil Sink Detail Drawing, Sheet 2- Additional Views

Press: **Ctrl+S > File > Manage File > Delete Old Versions > Enter > File > Save As >** Type  **> Zip File (*.zip) > OK > upload** the zip file to your course interface or attach to an email and send to your instructor and/or yourself **> File > Close > File > Close > File > Exit > Yes**

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