# Lesson 18 Drafts, Suppress, and Text Extrusions

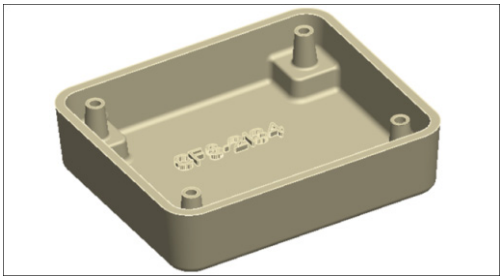


**Figure 18.1** Enclosure

## OBJECTIVES

- Create **Draft** features
- **Shell** a part
- **Suppress** features to decrease regeneration time
- **Resume** a set of suppressed features
- Create **Text** features on parts
- **Render** the part using **room scenes**

## REFERENCES AND RESOURCES

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

- Lesson 18 Lecture
- Book Projects PDF
- Project Lectures
- Creo Parametric Quick Reference Card
  http://www.cad-resources.com/Creo_Parametric_Qick_reference_cards.pdf
- Creo Parametric Configuration Options
  http://www.cad-resources.com/Creo_1.0_configoptions.pdf

## DRAFTS, SUPPRESS, AND TEXT EXTRUSIONS

The **Draft** feature adds a draft angle between surfaces. A wide range of parts incorporate drafts into their design. Casting, injection mold, and die parts normally have drafted surfaces. The ENCLOSURE in Figure 18.1 is a plastic injection-molded part.

Suppressing features by using the **Suppress** command temporarily removes them from regeneration. Suppressed features can be "unsuppressed" (**Resume**) at any time. It is sometimes convenient to suppress text extrusions and rounds to speed up regeneration of the model. Suppressing removes the item from regeneration and requires you to resume the item later.

**Hide** is another option. Creo Parametric allows you to hide and unhide some types of model entities. When you hide an item, Creo Parametric removes the item from the graphics window. The hidden item remains in the Model Tree list, and its icon dims to reveal its hidden status. When you unhide an item, its icon returns to normal display (undimmed) and the item is redisplayed in the graphics window. The hidden status of items is saved with the model. Unlike the suppression of items, hidden items are regenerated.

**Text** can be included in a sketch for extruded extrusions and cuts, trimming surfaces, and cosmetic features. To decrease regeneration time of the model, text can be suppressed after it has been created. Text can also be drafted.

## Drafts

The **Draft Tool** adds a draft angle between two individual surfaces or to a series of selected planar surfaces. During draft creation, remember the following:

- You can draft only the surfaces that are formed by tabulated cylinders or planes.
- The draft direction must be normal to the neutral plane if a draft surface is cylindrical.
- You cannot draft surfaces with fillets around the edge boundary. However, you can draft the surfaces first, and then fillet the edges.

The following table lists the terminology used in drafts.

| TERM | DEFINITION |
|---|---|
| **Draft surfaces** | Model surfaces selected for drafting. |
| **Draft Hinges** | Draft surfaces are pivoted about the intersection of the neutral plane with the draft surfaces. |
| **Pull direction** | Direction that is used to measure the draft angle. It is defined as normal to the reference plane. |
| **Draft angle** | Angle between the draft direction and the resulting drafted surfaces. If the draft surfaces are split, you can define two independent angles for each portion of the draft. |
| **Direction of rotation** | Direction that defines how draft surfaces are rotated with respect to the neutral plane or neutral curve. |
| **Split areas** | Areas of the draft surfaces to which you can apply different draft angles. Split object is also a choice. |

## Suppressing and Resuming Features

Suppressing a feature is similar to removing the feature from regeneration temporarily. You can "unsuppress" (**Resume**) suppressed features at any time. Features on a part can be suppressed to simplify the part model and decrease regeneration time. For example, while you work on one end of a shaft, it may be desirable to suppress features on the other end of the shaft. Similarly, while working on a complex assembly, you can suppress some of the features and components for which the detail is not essential to the current assembly process.

Unlike other features, the base feature cannot be suppressed. If you are not satisfied with your base feature, you can redefine the section of the feature, or you can delete it and start over again. Select feature(s) to suppress by: selecting it, picking on it from the Model Tree, specifying a *range*, entering its *feature number* or *identifier*, or using *layers*.

You can use **Suppress** and **Resume** to simplify the part before inserting features such as text extrusions. In addition, you may wish to suppress the text extrusion if there is other work to be done on the part. Text extrusions take time to regenerate, and increase the file size considerably.

## Text Extrusions

When you are modeling, **Text** can be included in a sketch for extruded extrusions and cuts, trimming surfaces, and cosmetic features. The characters that are in an extruded feature use the font **font3d** as the default. Other fonts are available.

# Lesson 18 STEPS

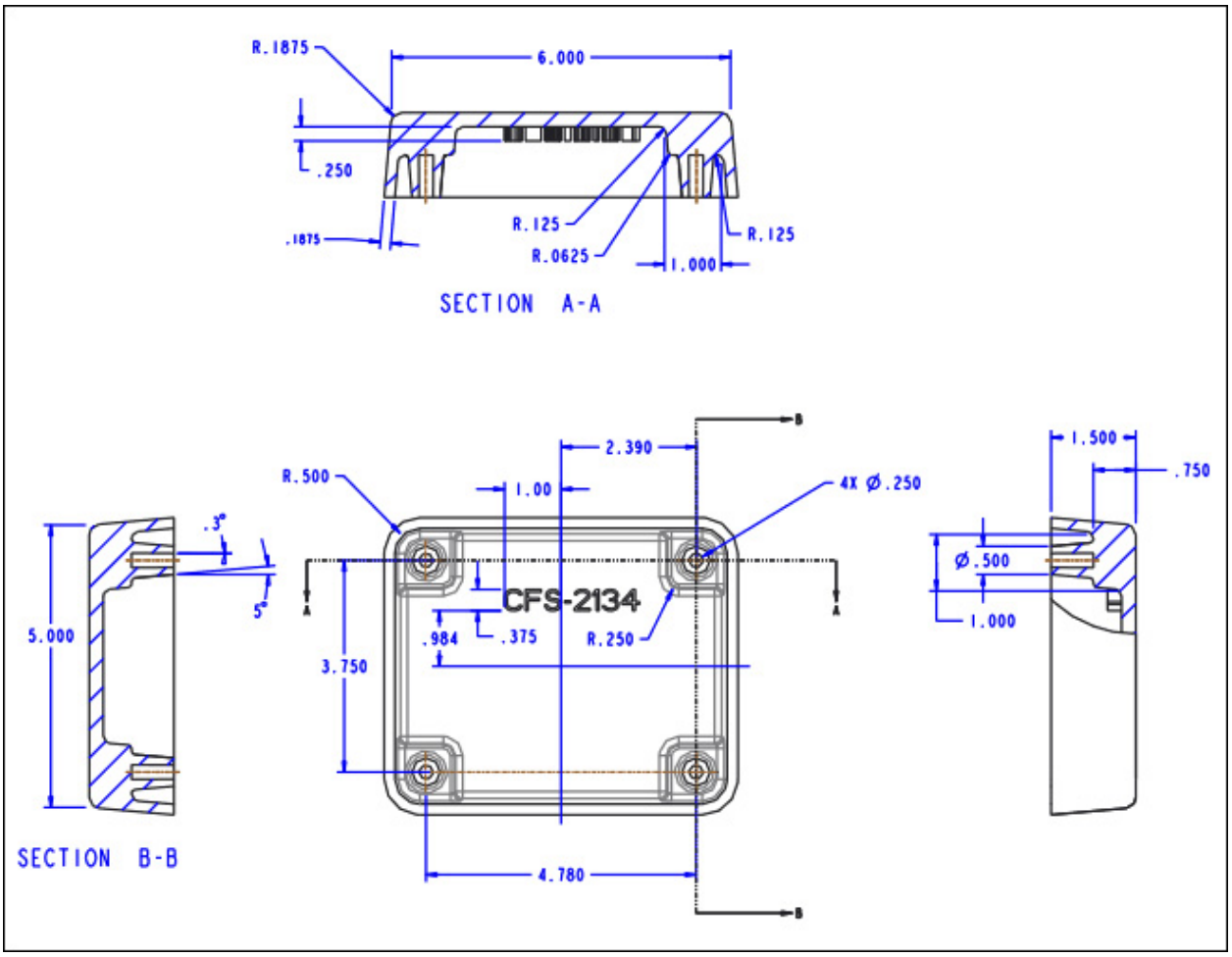
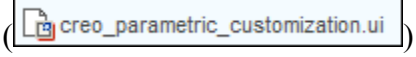



**Figure 18.2(a)** Enclosure

## Enclosure

The Enclosure is a plastic injection-molded part. A variety of drafts will be used in the design of this part. A *raised text extrusion* will be modeled on the inside of the Enclosure, as shown in Figure 18.1. The dimensions for the part are provided in Figures 18.2(a) through 18.2(e).

Click: **File > Manage Session > Select Working Directory >** select the working directory **> OK > Ctrl+N > enclosure > OK > File > Options > Configuration Editor > Import/Export > Import configuration file >** select your previously created and saved file (**clamp.pro**) **> Open > Find >** 1. Type keyword: ***default_dec_places*** **> Find Now >** 3. Set value: **3 > Enter > Close > OK > No > File > Prepare > Model Properties >** Units **change >** Units Manager **Inch lbm Second (Creo Parametric Default) > Close >** Material **change > fe20.mtl** (plastic) **>**  **> OK > Close >** *slowly* double-click on the default coordinate system name in the Model Tree-- **PRT_CSYS_DEF >** type **CSYS_ENCLOSURE > Enter > Ctrl+S > Enter > Files > Options > Customize Ribbon > Import/Export > Import customization file >** select your previously saved **.ui** file from Lesson 2 ( 

) [only available if you created a **.ui** file and saved it as instructed] **> Open > Import Mode Customizations > OK > LMB** to deselect

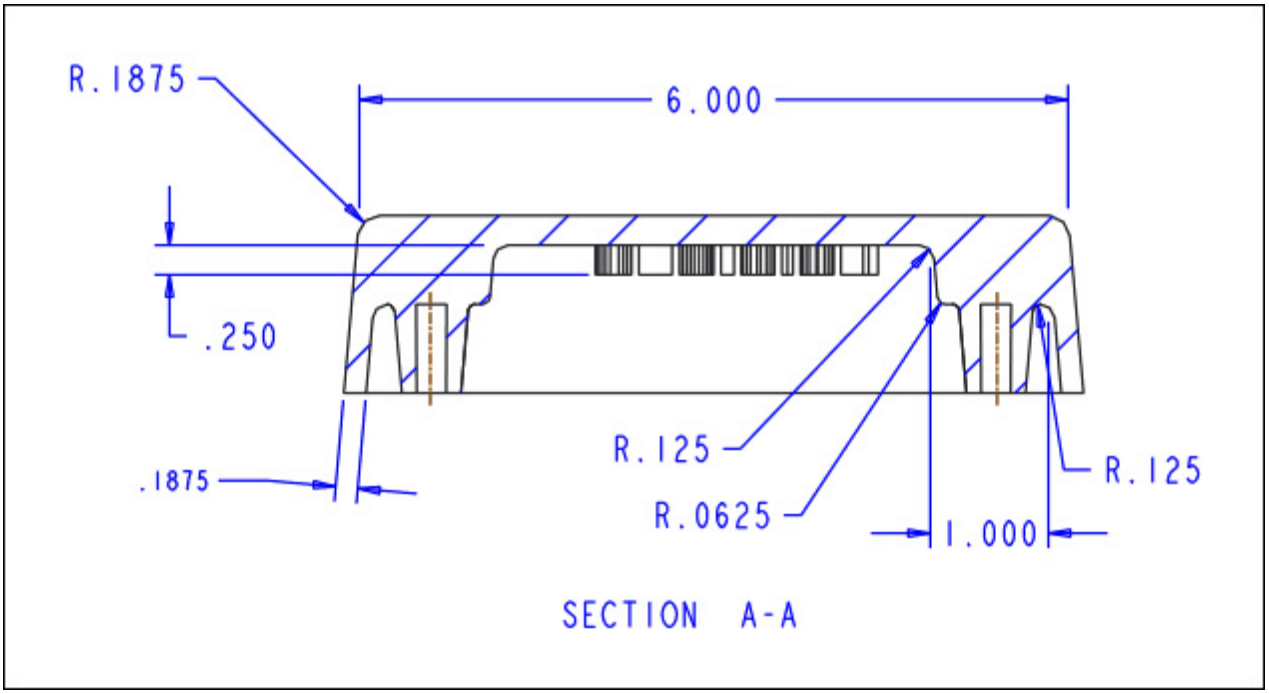



**Figure 18.2(b)** SECTION A-A (Top View)

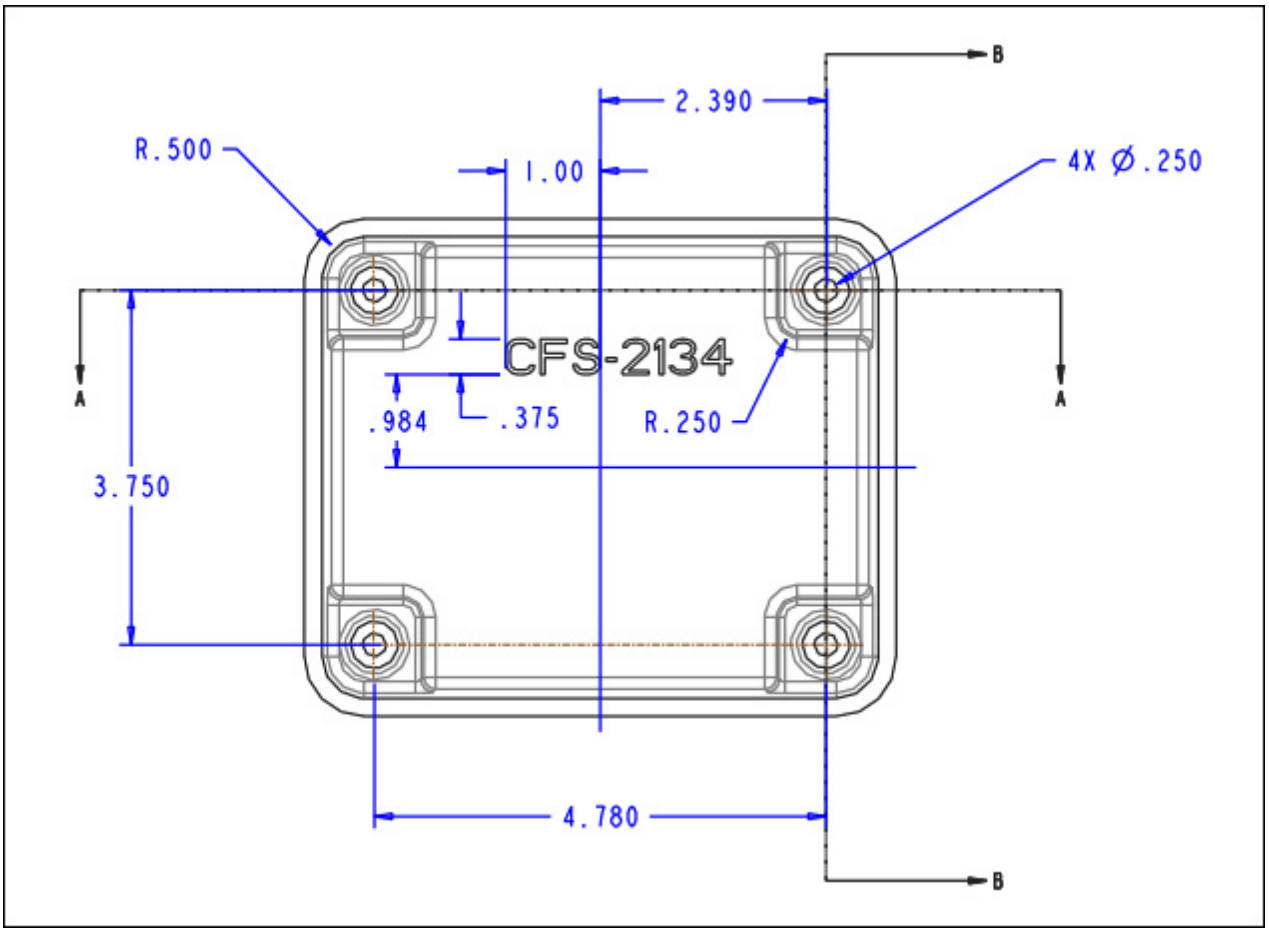



**Figure 18.2(c)** Front View

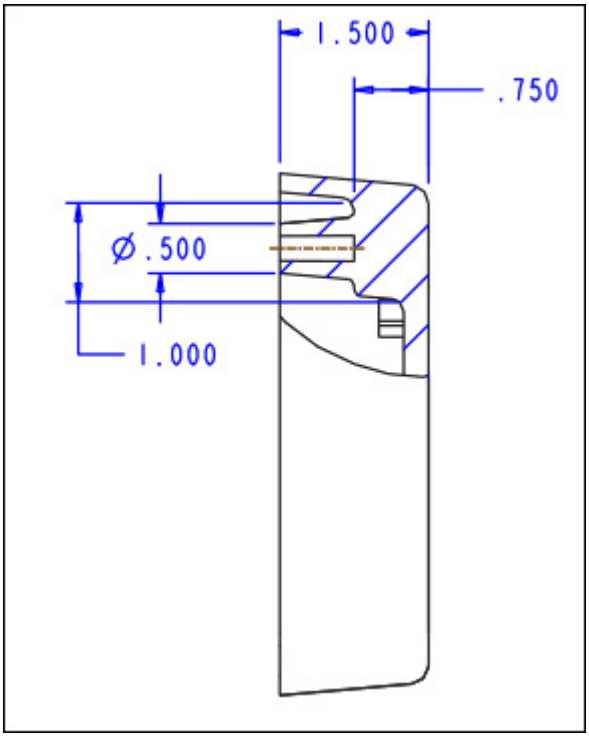



**Figure 18.2(d)** Right Side View

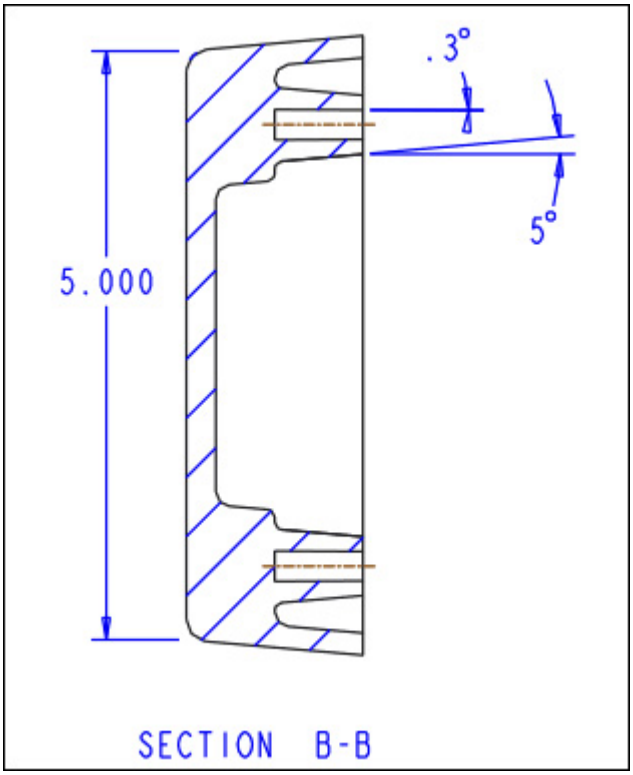



**Figure 18.2(e)** SECTION B-B (Left Side View)

Make the first extrusion **6.00** (width) **X 5.00** (height) **X 1.50** (depth), with **R.50** rounds. Add the fillets in the sketch instead of rounds after the first extrusion is complete. Sketch on datum **FRONT**. Center the first extrusion horizontally on datum **TOP** and vertically on datum **RIGHT**. Add constraints as needed to control your sketch geometry. Incorporate the draft angle into the first extrusion instead of adding a separate draft feature [Figs. 18.3(a-c)]

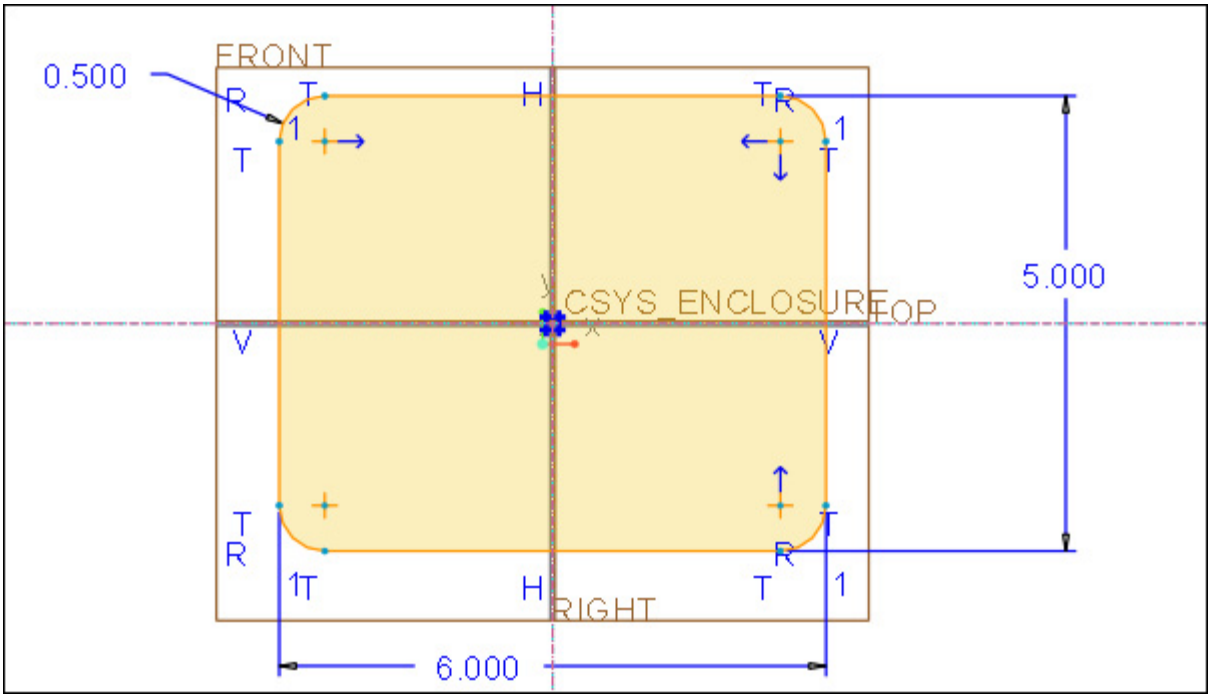



**Figure 18.3(a)** Sketch on the Front Datum

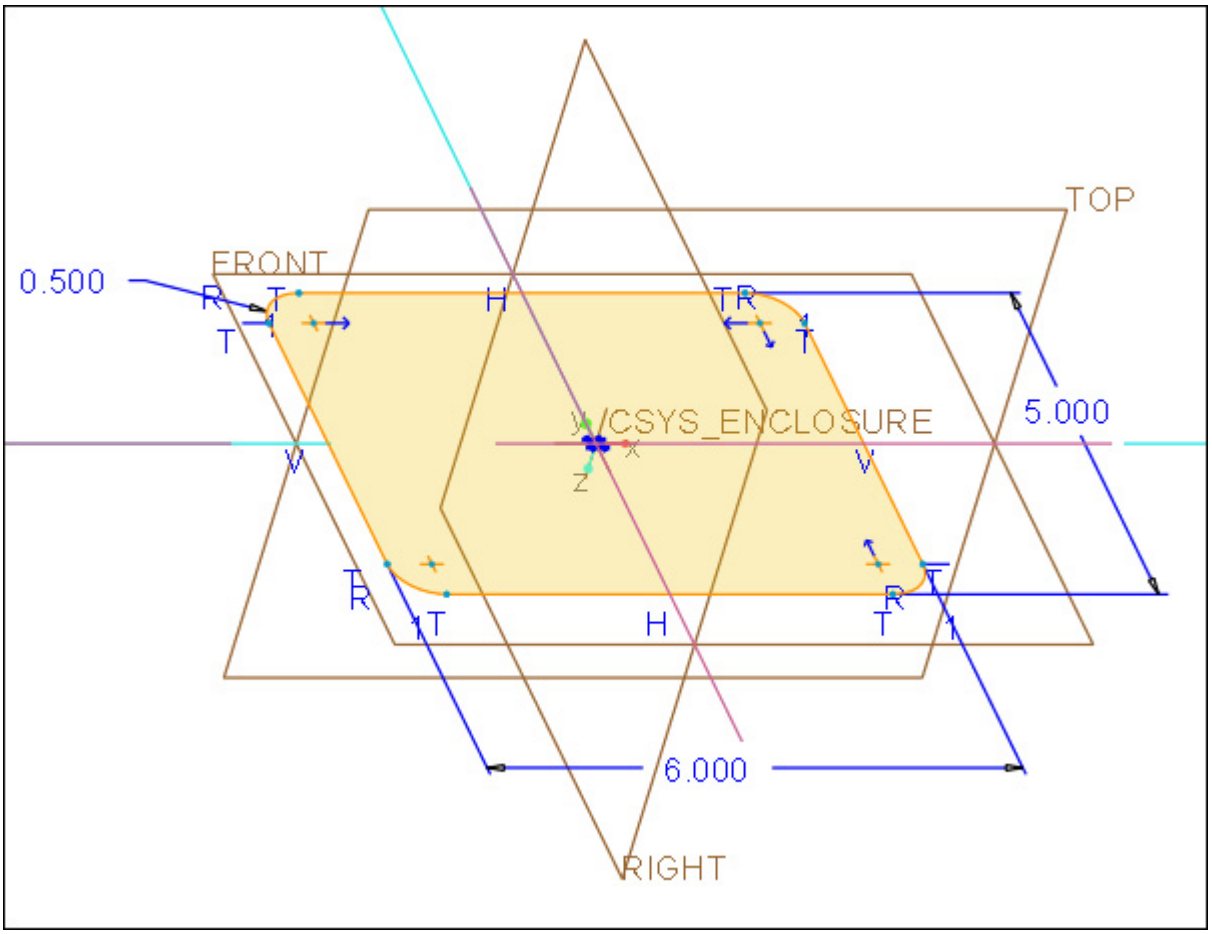



**Figure 18.3(b)** Dimensions

Click:  to compete the sketch > **Options** tab > 

> **5** > **Enter** [Fig. 18.3(c)] >  >  > **Ctrl+S** > **Enter** > **LMB** to deselect > **View** tab > **Appearance Gallery** > set a new color for your part

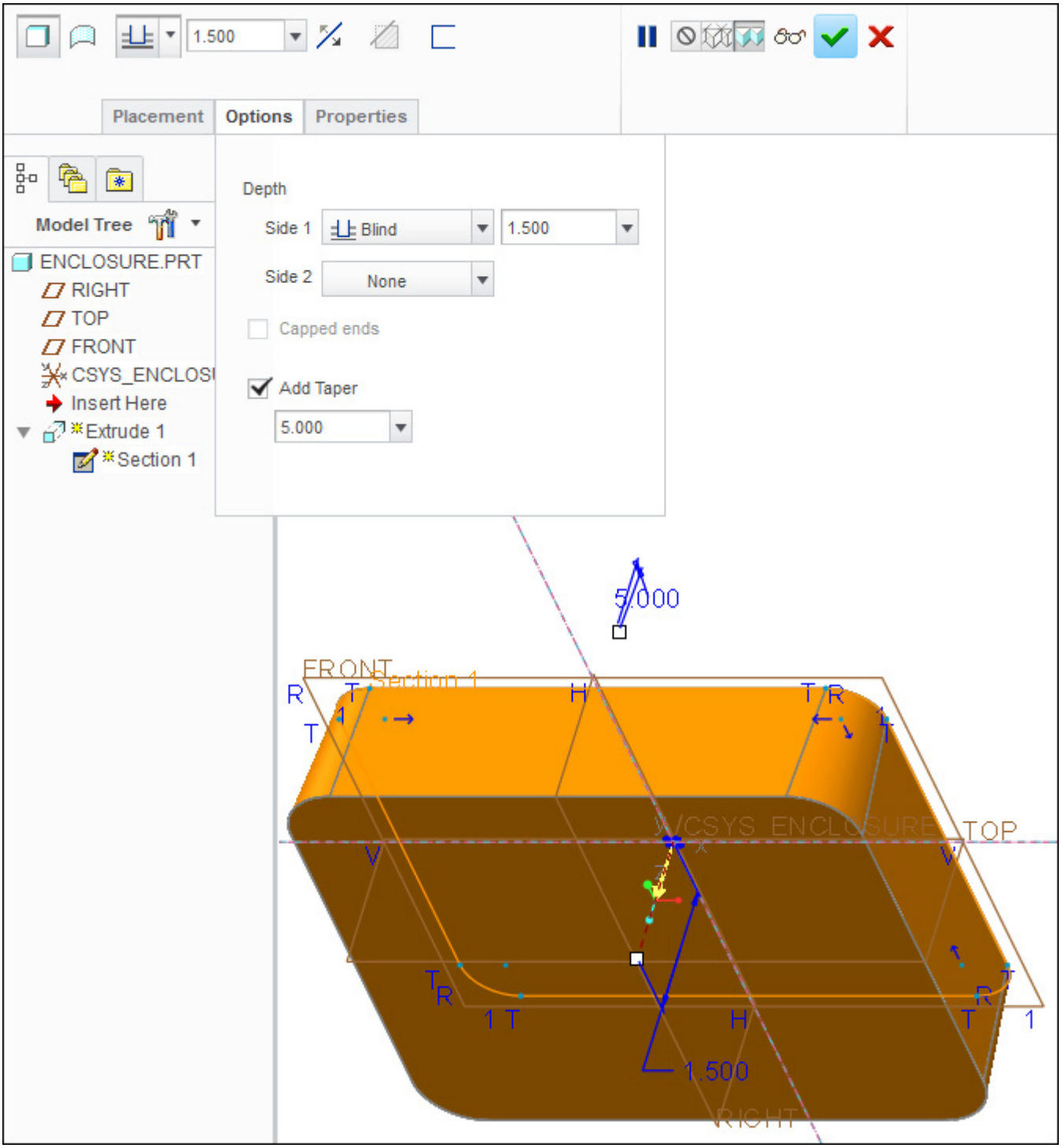



**Figure 18.3(c)** Adding a Taper

Click: **Model** tab > Shell > Thickness **.1875** > **Enter** > **References** tab > select the face to be removed [Figs. 18.4(a-b)]

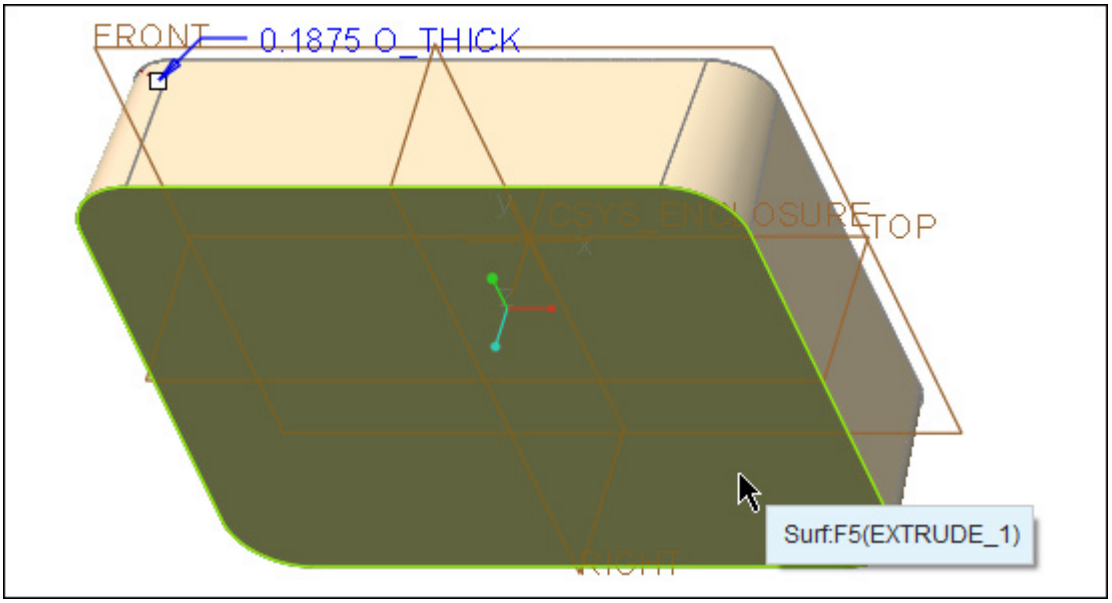



**Figure 18.4(a)** Select the Surface to Remove

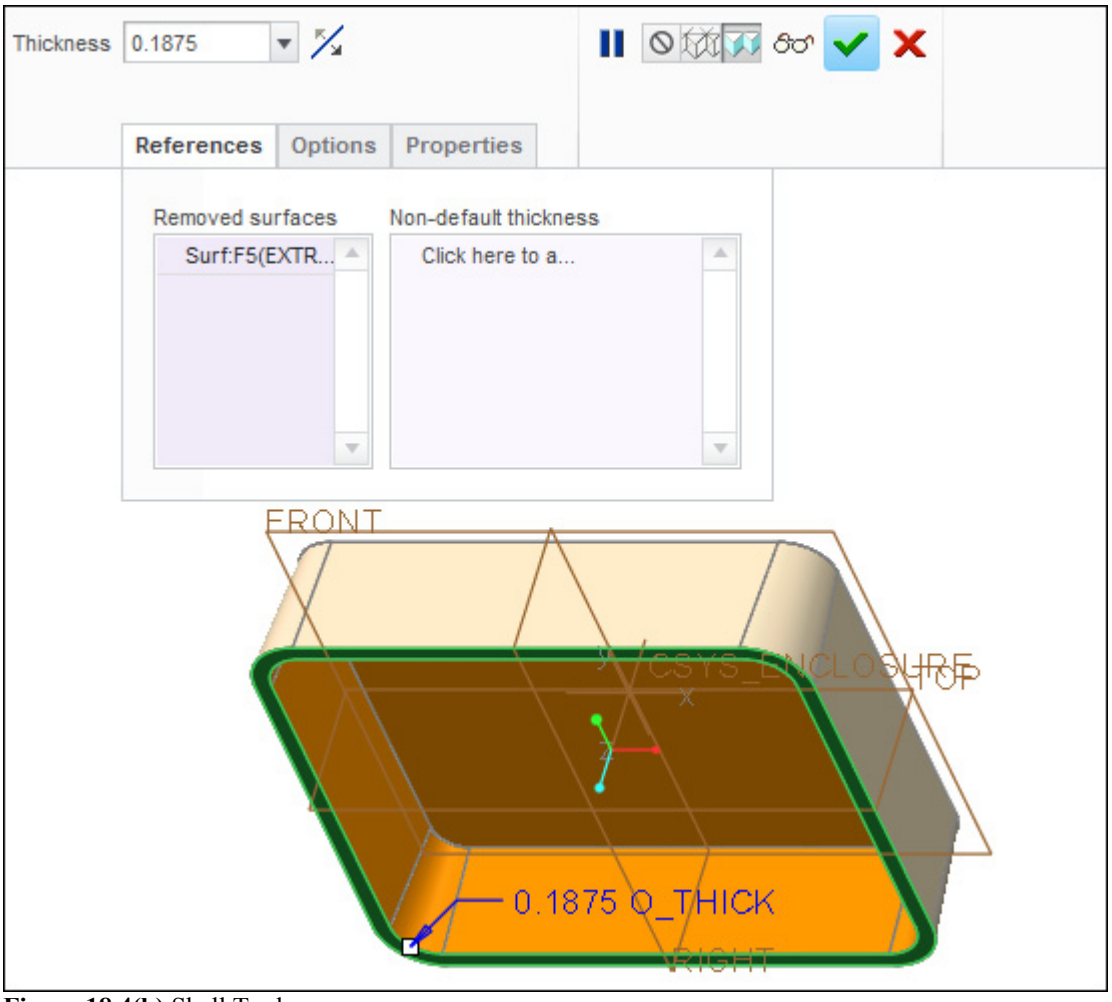
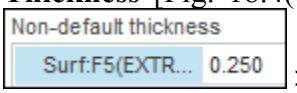



**Figure 18.4(b)** Shell Tool

Change the thickness of the enclosure to be **.25**, the walls will remain **.1875**. Press: **RMB > Non Default Thickness** [Fig. 18.4(c)] > select the face (*highlights*) [Fig. 18.4(d)] > type **.250** in the Dimension field




**> Enter** [Fig. 18.4(e)] >  **> LMB** to deselect **> Ctrl+D > Ctrl+S > Enter**

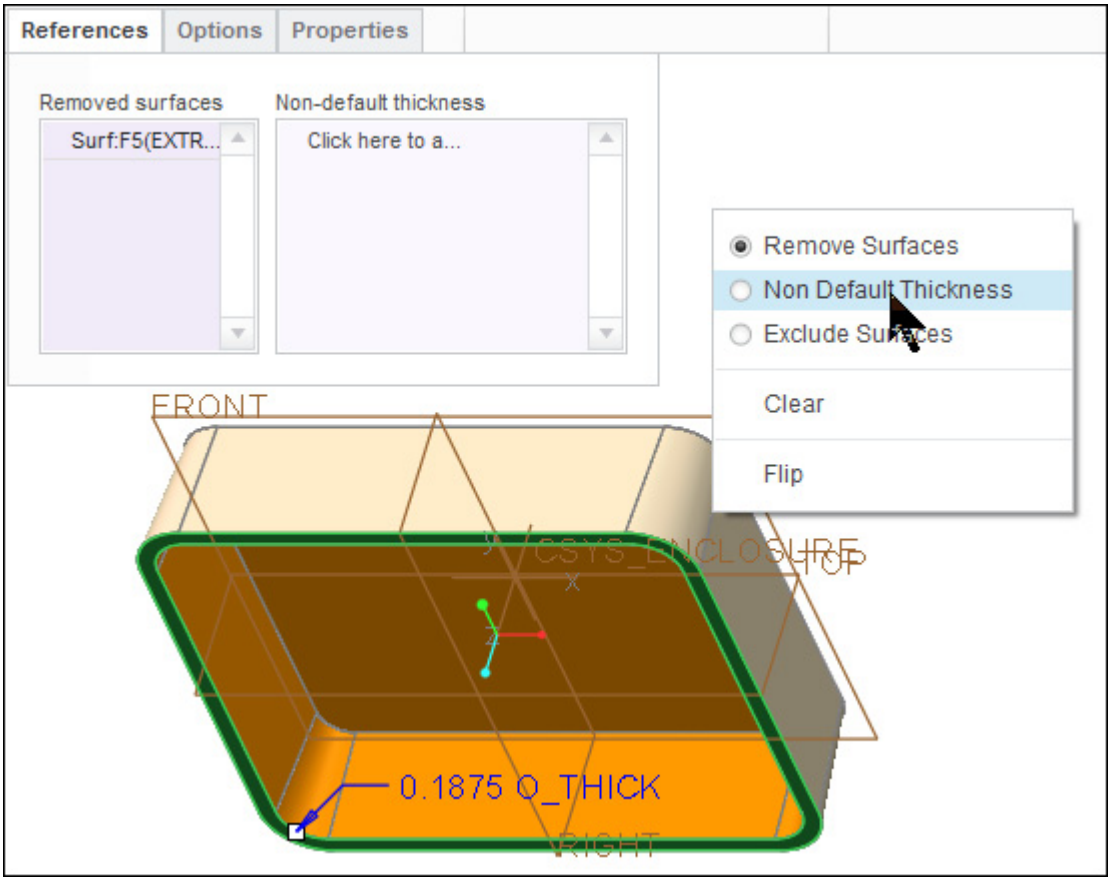



**Figure 18.4(c)** Non Default Thickness

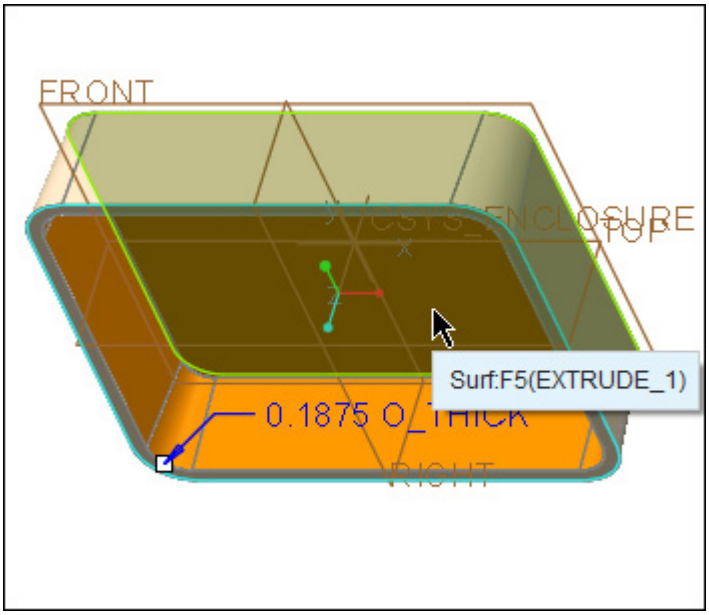



**Figure 18.4(d)** Select the Non Default Surface

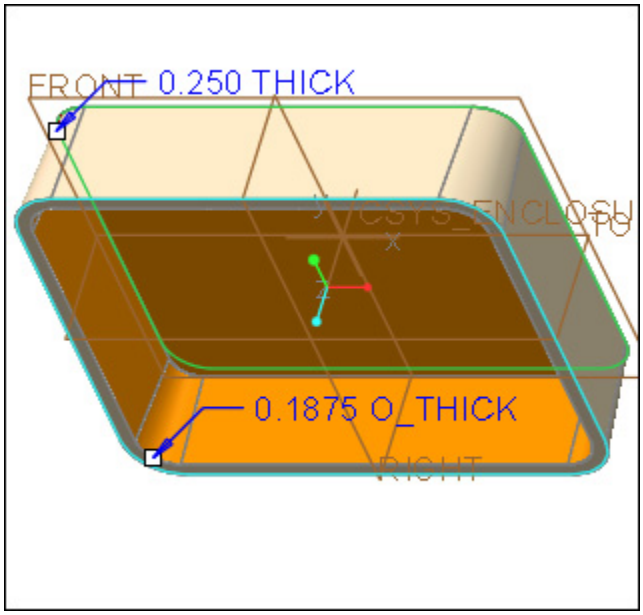



**Figure 18.4(e)** Shelled Extrusion

Create the raised pedestal-like extrusion. Click: Plane > select datum **FRONT** > drag the depth handle forward by **.75** [Fig. 18.5(a)] > **OK** (**DTM1** will be used to control the height of the pedestal.) > **LMB** to deselect > Extrude > **Placement** tab > **Define** > select the inside surface of the shell as the sketching plane > **Sketch** > press **RMB** > **References** > delete the datum references > select only the *edges (toggle with RMB until an edge is highlighted)* [Fig. 18.5(b)] > **Close**

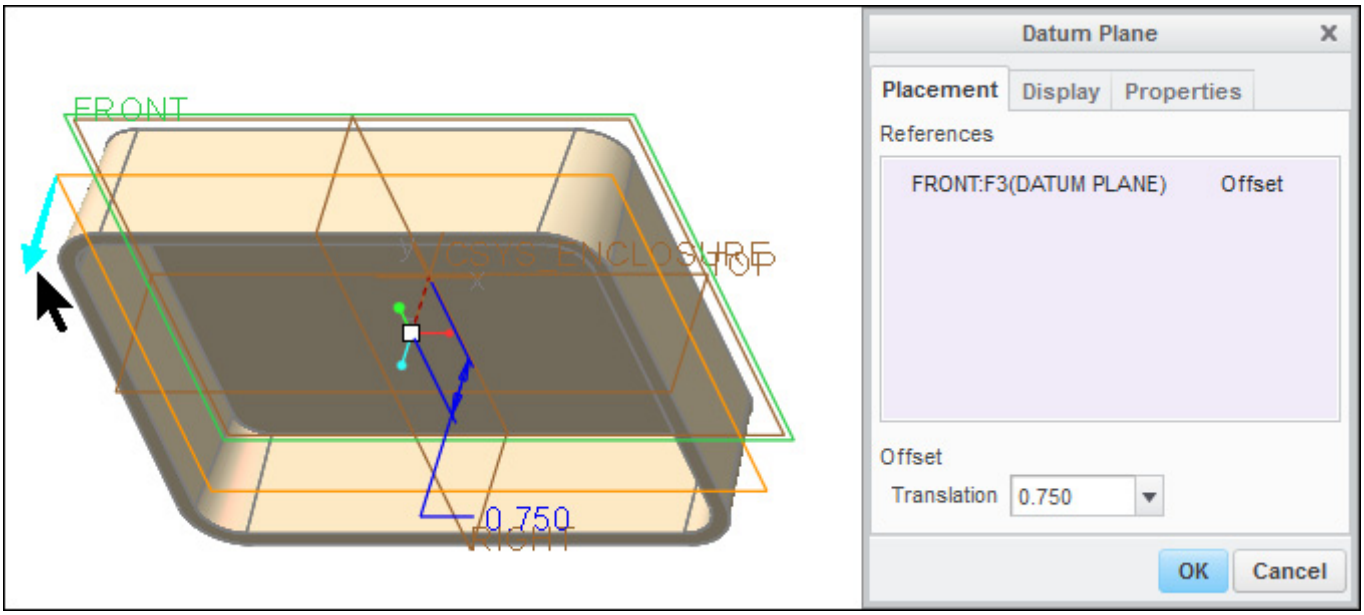



**Figure 18.5(a)** Offset Datum DTM1

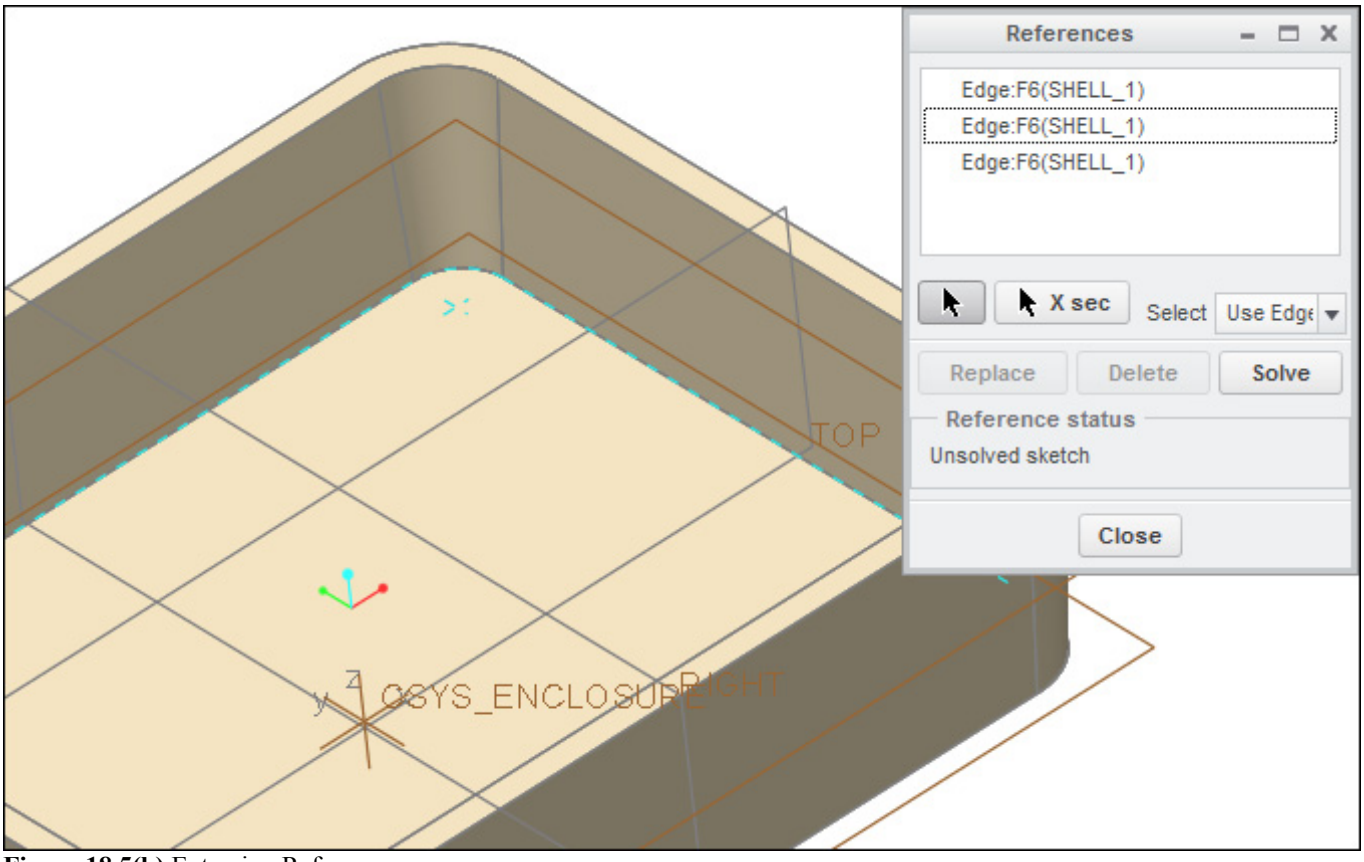



**Figure 18.5(b)** Extrusion References

Click:  > select an existing *internal shelled edge* to start the section [Fig. 18.5(c)] > **Close** > add four lines and a fillet > add and modify dimensions [Fig. 18.5(d)] >  [Fig. 18.5(e)]

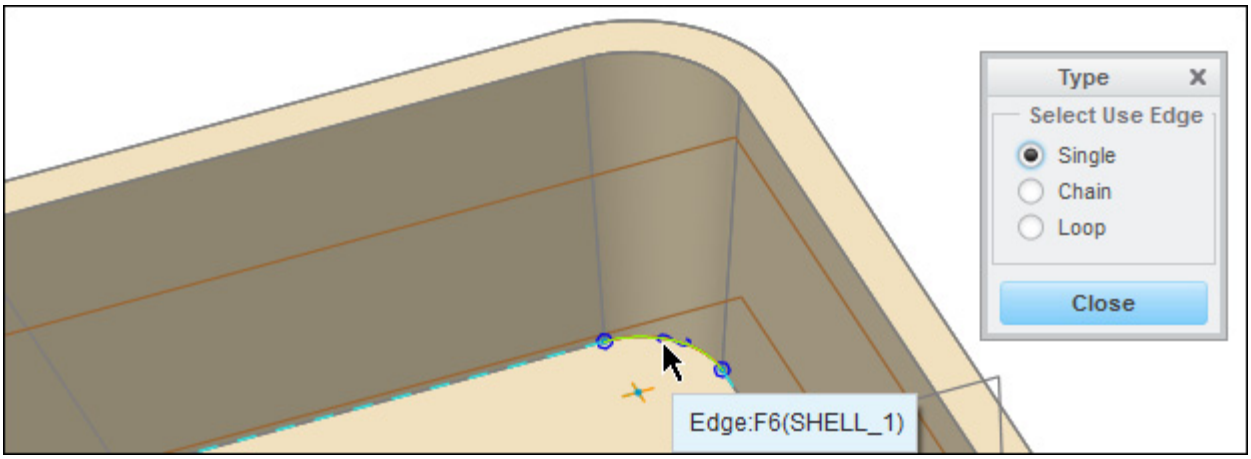



**Figure 18.5(c)** Create the First Entity using:  **Create entities by projecting curves or edges onto the sketch plane**

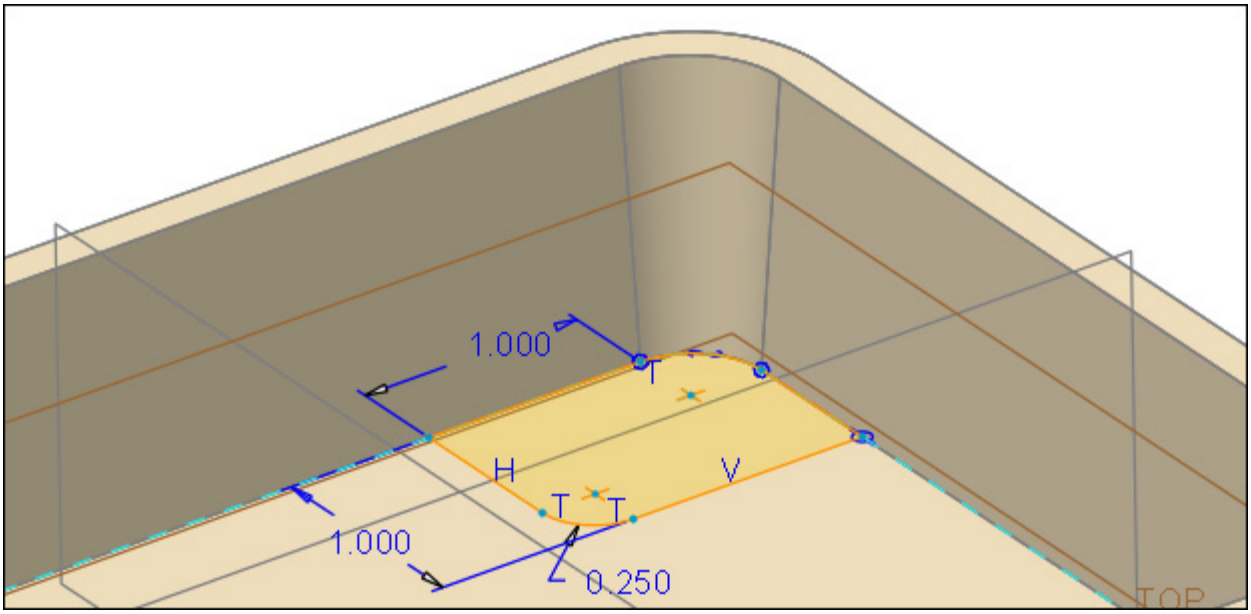



**Figure 18.5(d)** Section Sketch

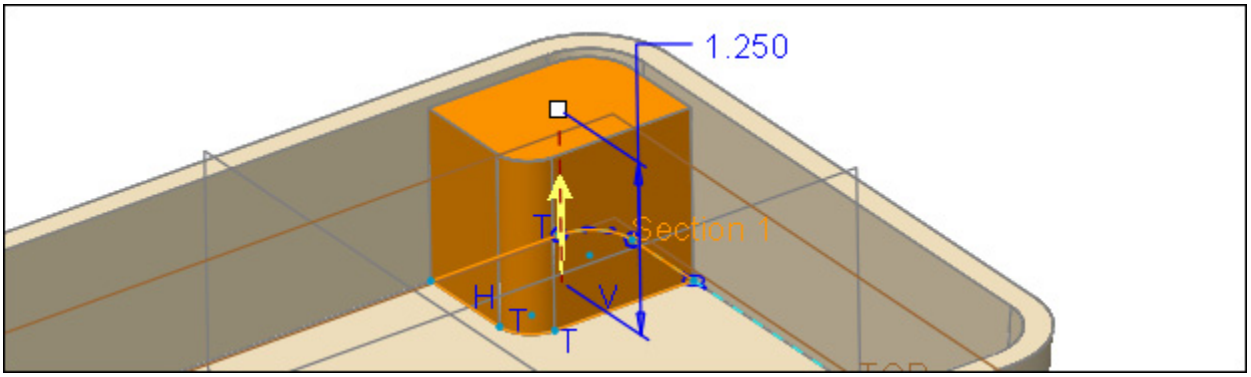



**Figure 18.5(e)** Previewed Extrusion Depth

Place your pointer over the depth drag handle [Fig. 18.5(f)] > press **RMB > To Selected** [Fig. 18.5(g)] > select **DTM1** [Fig. 18.5(h)] >  > press **RMB > Edit** [Fig. 18.5(i)] > **Ctrl+S > Enter > LMB**

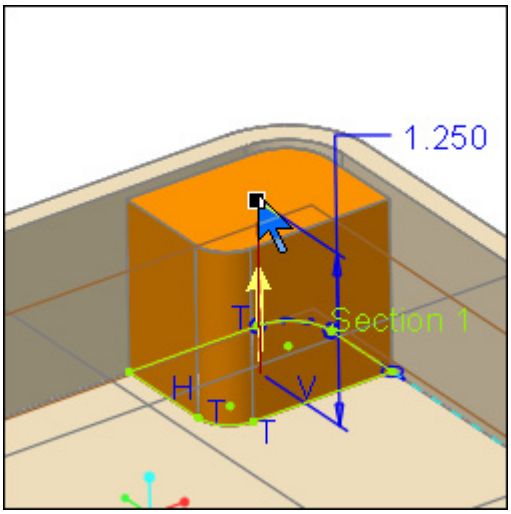



**Figure 18.5(f)** Place Pointer Over the Drag Handle

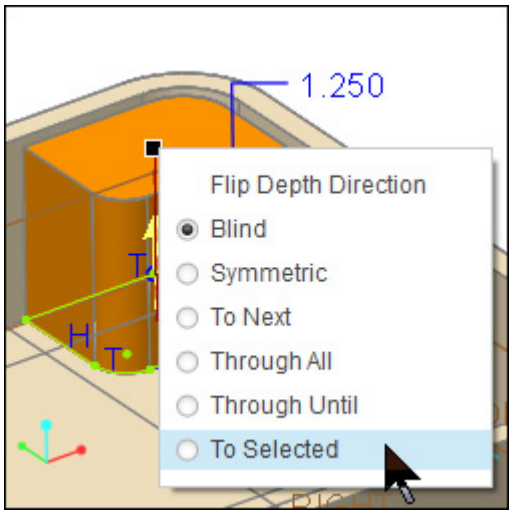



**Figure 18.5(g)** To Selected

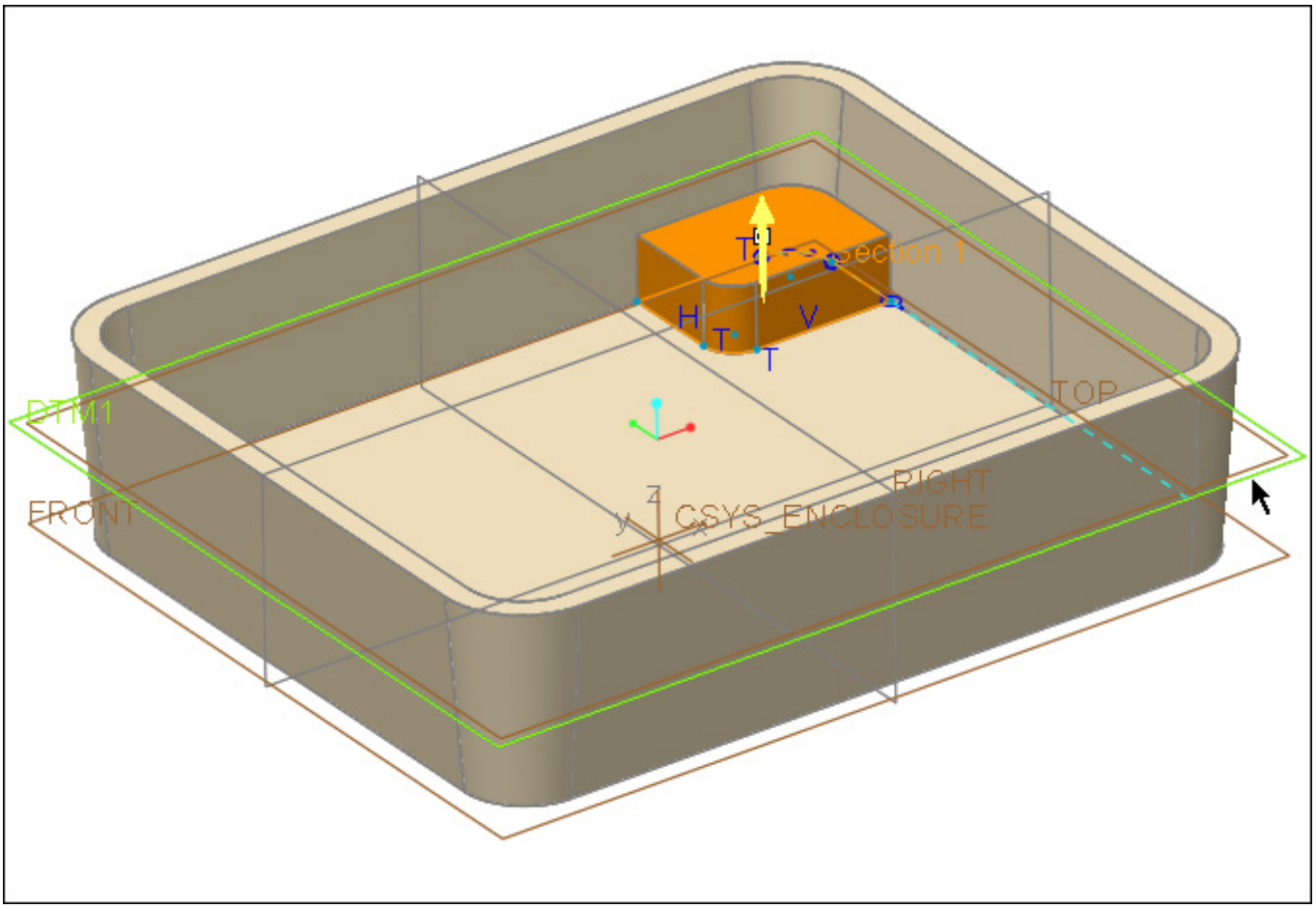



**Figure 18.5(h)** Select DTM1 to Establish the Depth

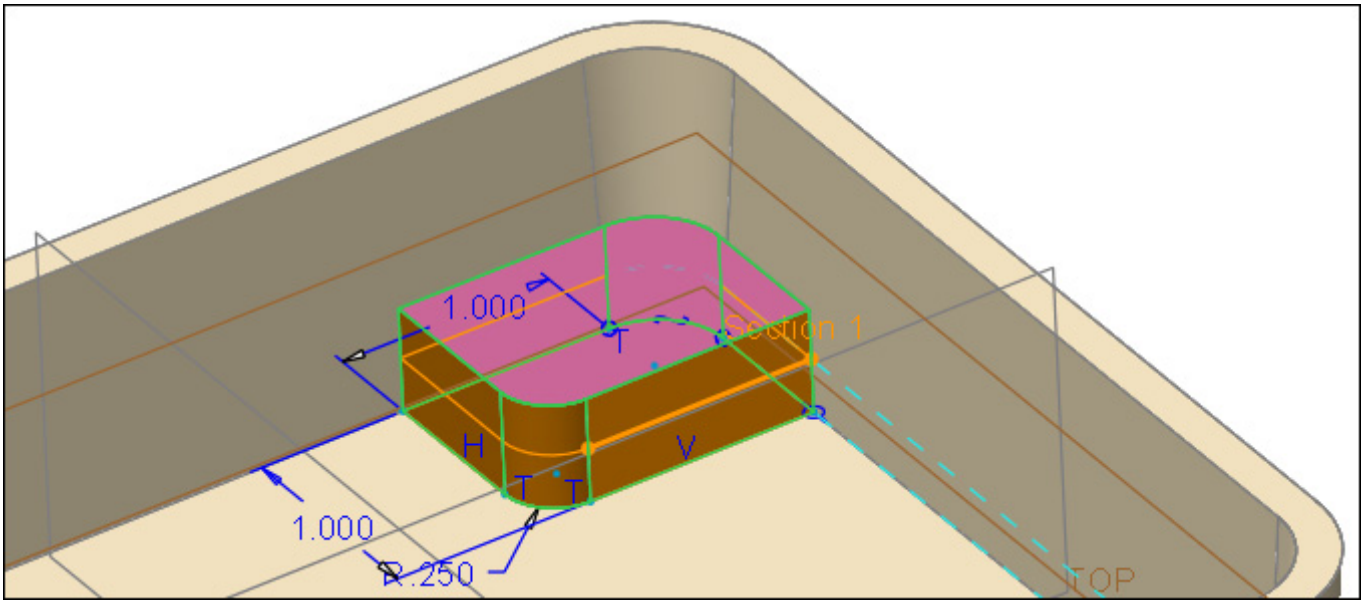



**Figure 18.5(i)** Completed Pedestal Extrusion

Click: Draft > **References** tab > select a vertical surface of the pedestal > press **RMB** > **Draft Hinges** > select **FRONT** > type **5** in Dimension field > **Enter** > **Reverse pull direction** (Fig. 18.6) >

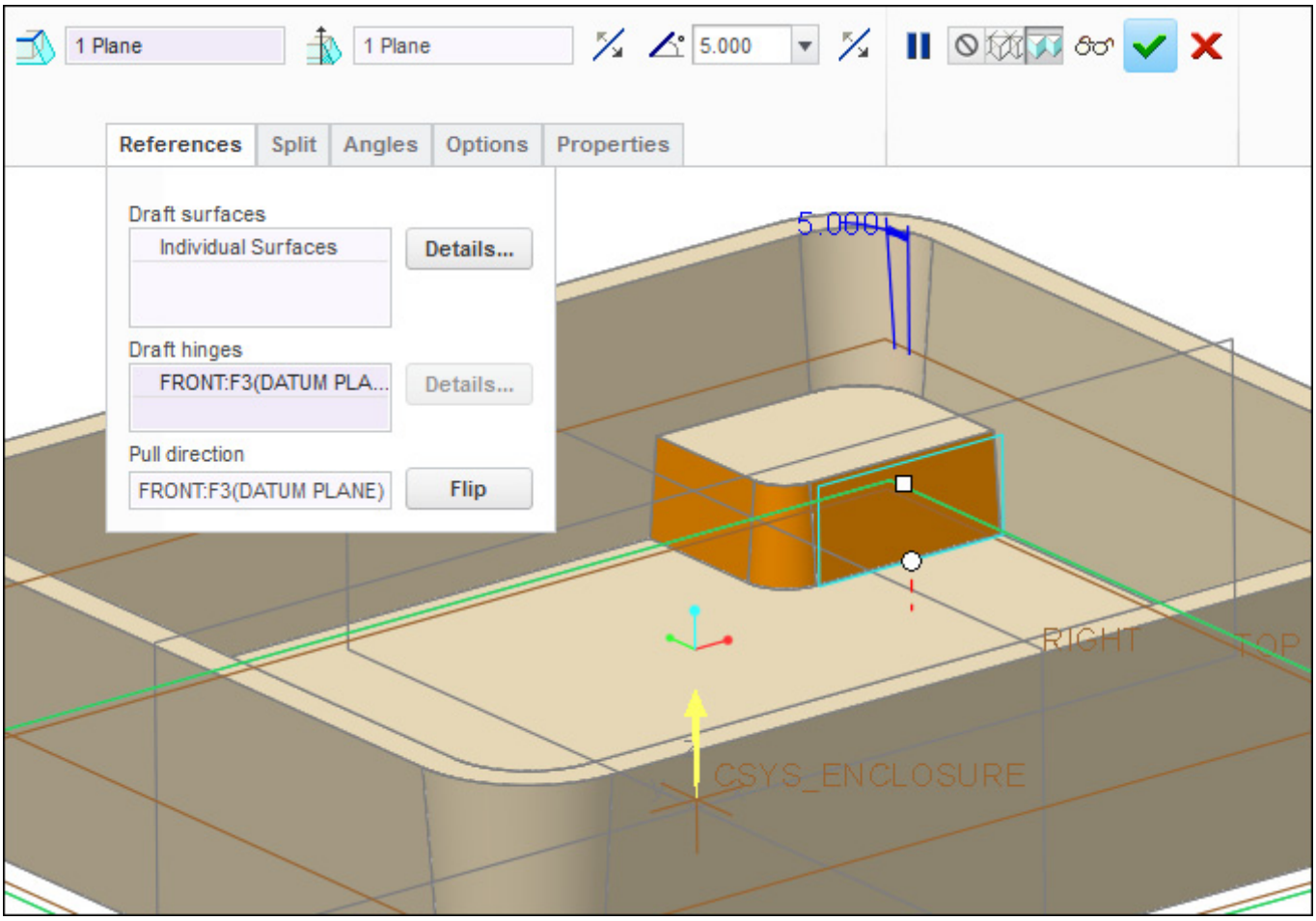



**Figure 18.6** Draft Three Lateral Surfaces of the Pedestal

Model the circular extrusion. Click: Extrude > select the top surface of the pedestal as the sketching plane > keep the default references > the section consists of one circle [Fig. 18.7(a)] > ✔ > rotate the model [Fig. 18.7(b)] > place your pointer over the depth drag handle > press **RMB > To Selected** [Fig. 18.7(c)]

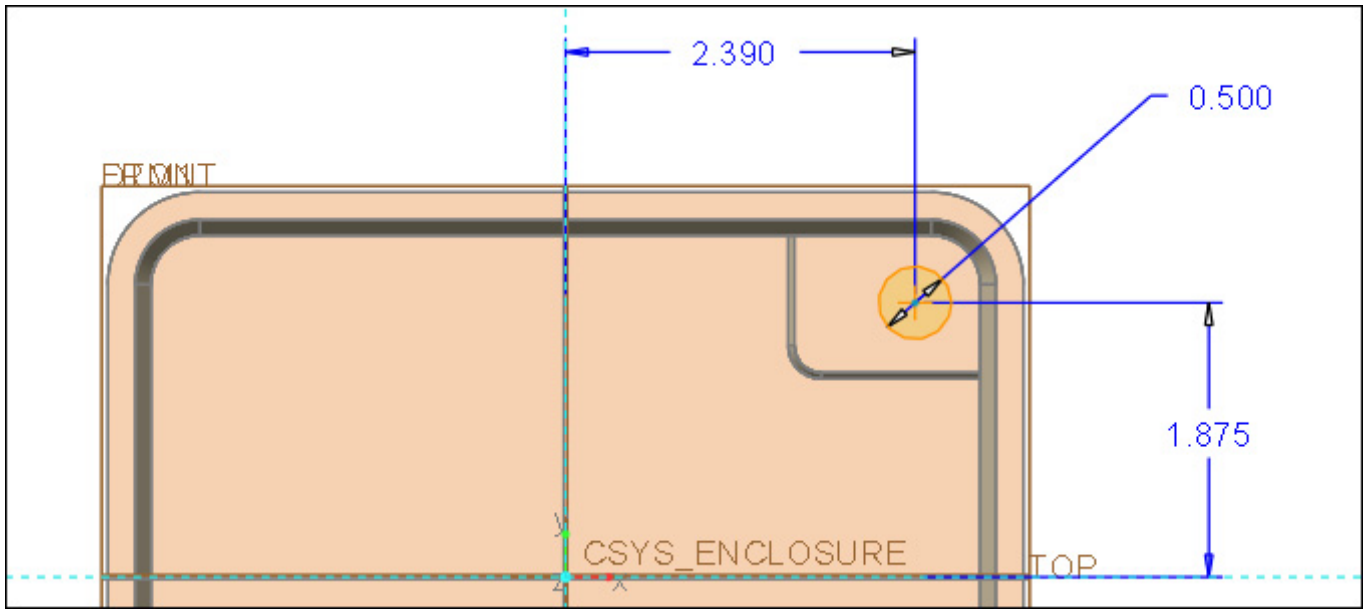



**Figure 18.7(a)** Section Sketch for the Circular Extrusion

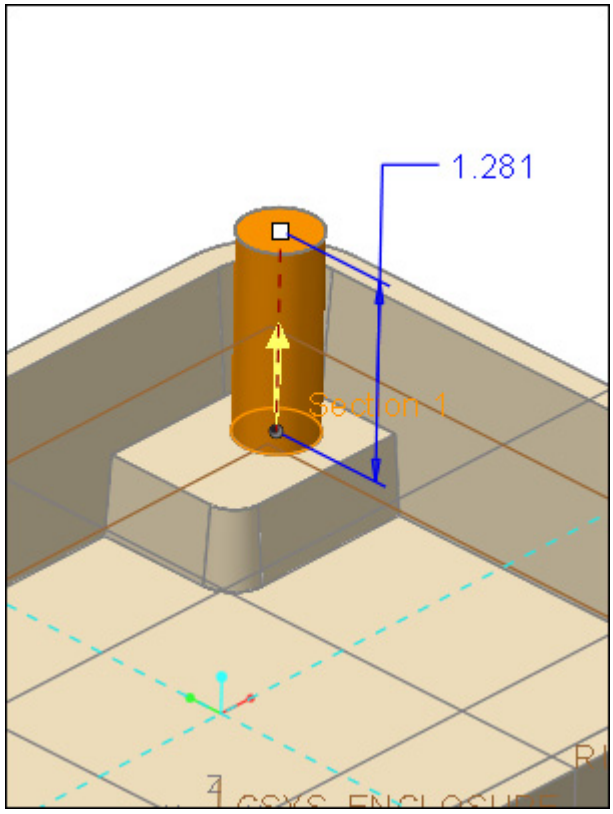



**Figure 18.7(b)** Extrusion Depth

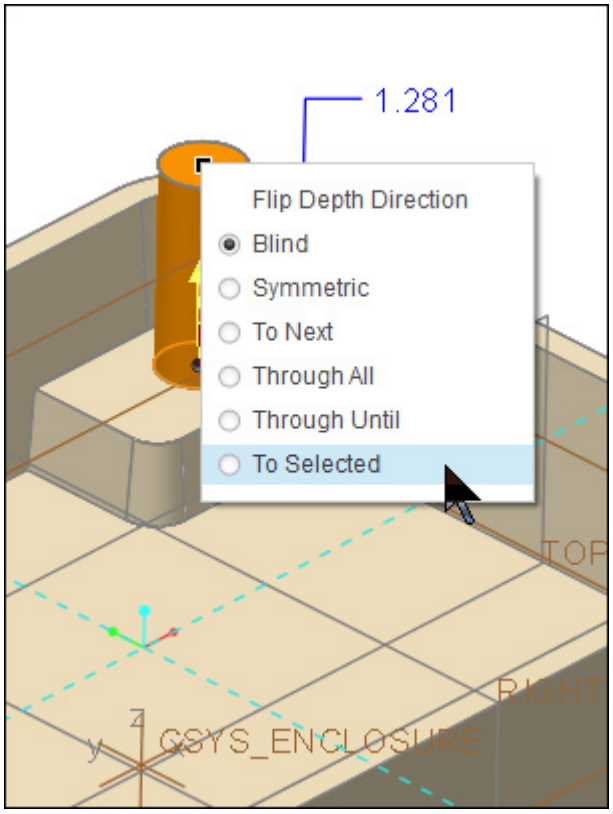



**Figure 18.7(c)** Depth Options

Select the surface [Fig. 18.7(d)] >  [Fig. 18.7(e)] > **LMB** to deselect > **Ctrl+S** > **OK**

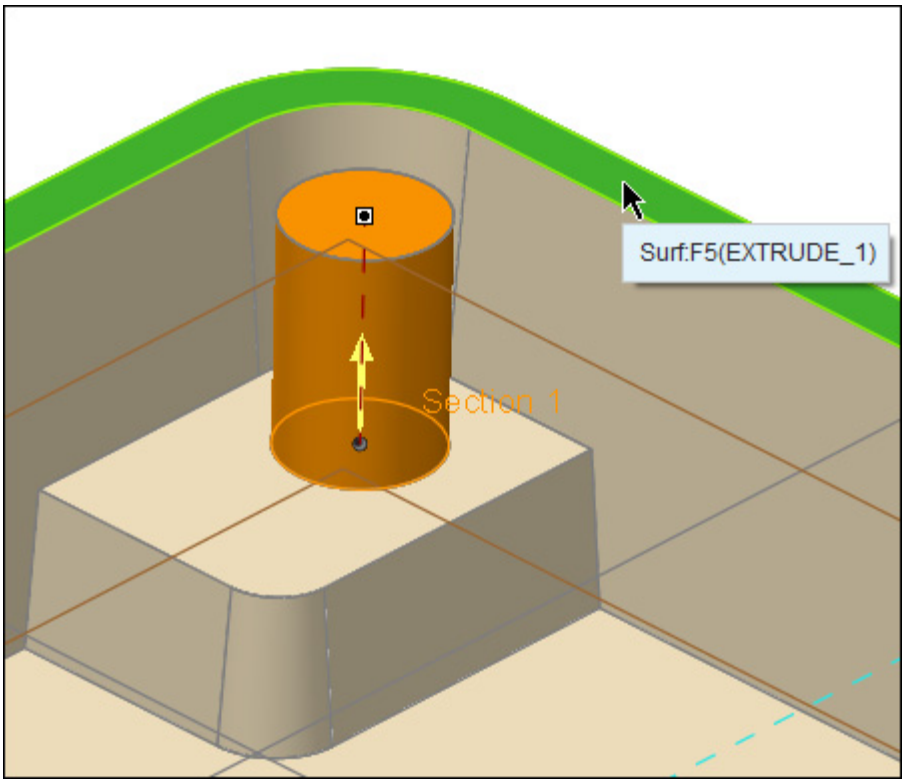



**Figure 18.7(d)** Select Surface to Extrude To

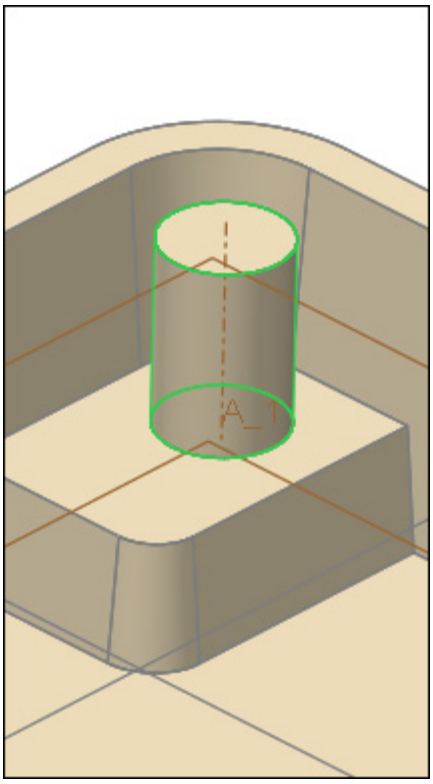



**Figure 18.7(e)** Circular Extrusion

The circular feature looks correct, but there seems to be a problem with the pedestal.

Select the pedestal extrusion in the Graphics Window > press **RMB > Edit Definition** [Fig. 18.8(a)]

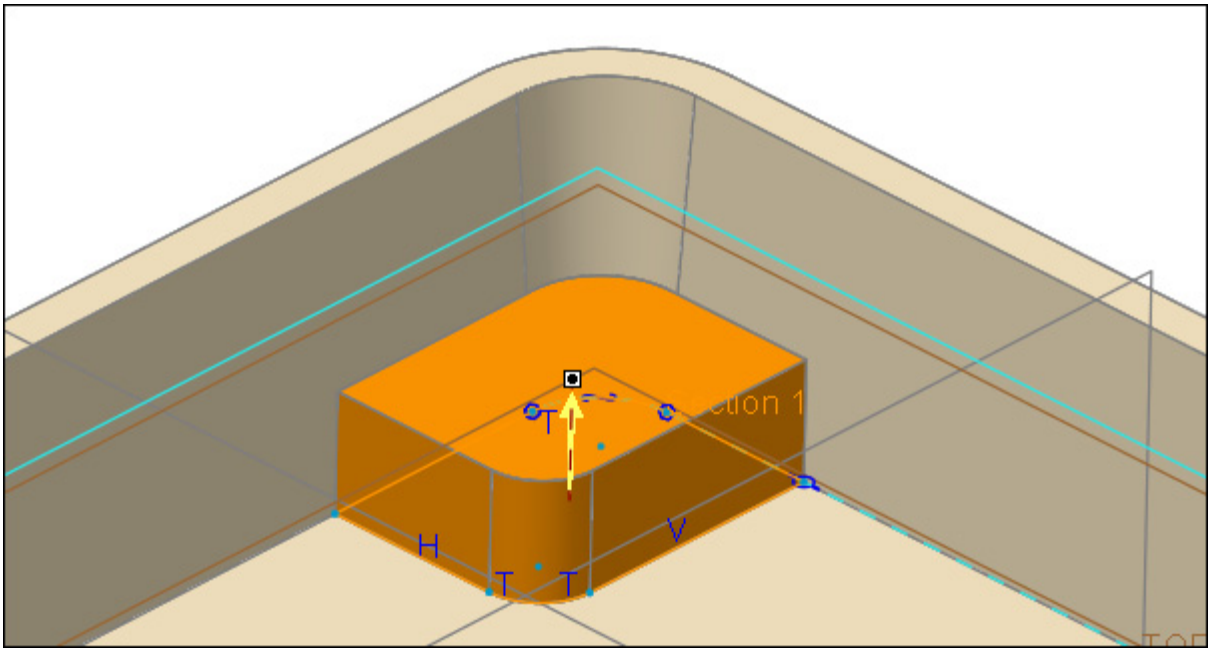



**Figure 18.8(a)** Redefining the Pedestal

Press: **RMB > Edit Internal Sketch** [Fig. 18.8(b)] > A dimension is referencing the end of the arc instead of the edge. Select this dimension. > press **RMB > Delete** > create a new defining dimension and modify the new dimension to **1.000** [Fig. 18.8(c)] >  >  > [Fig. 18.8(d)] > **LMB** to deselect > **Ctrl+S** > **Enter**

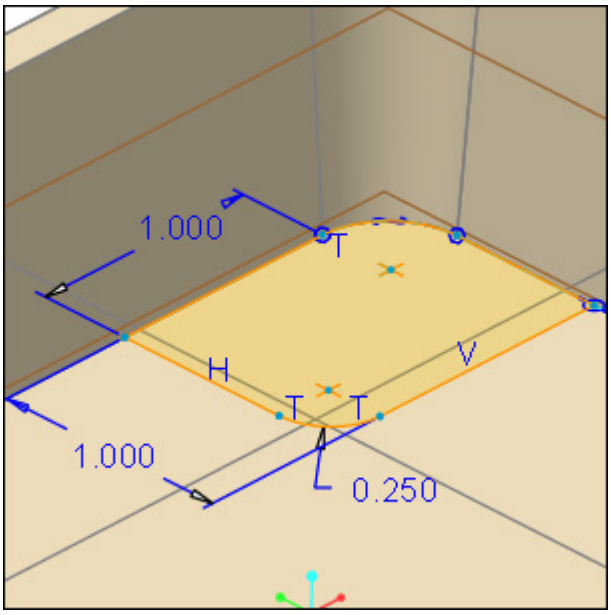



**Figure 18.8(b)** Original Dimensioning Scheme

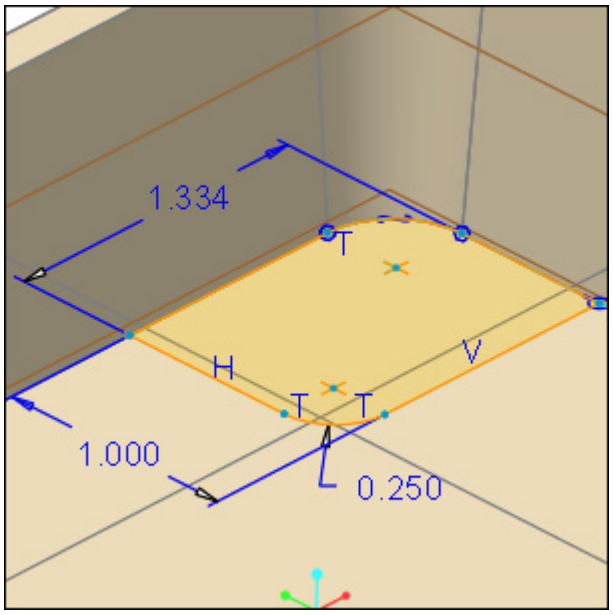



**Figure 18.8(c)** New Defining Dimension

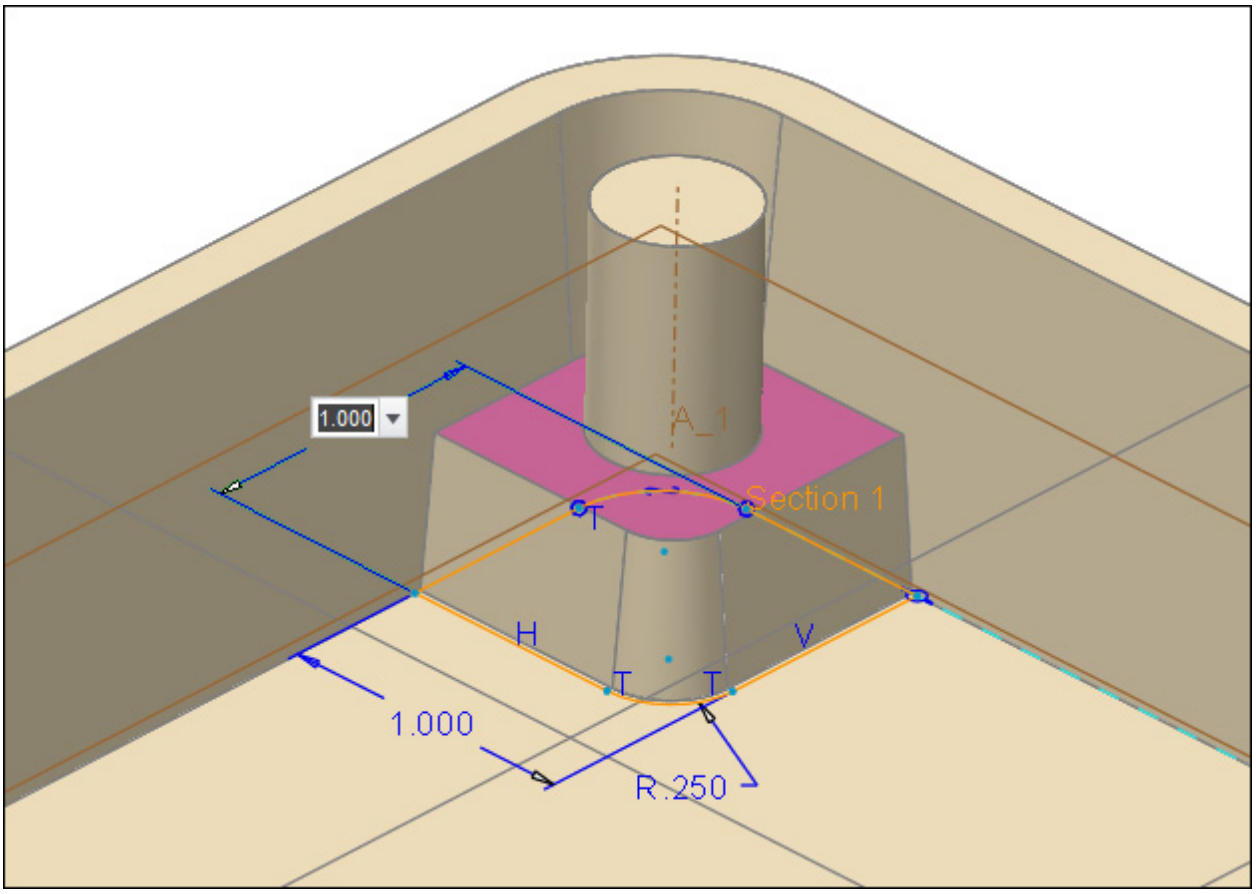



**Figure 18.8(d)** Redefined Pedestal

Click: Draft (Draft the circular extrusion at **5°**. Use the top surface of the circular extrusion as the draft hinge [Fig. 18.9(a)]) > > > press **RMB** > **Edit** > select the **5** degree dimension > press **RMB** > **Properties** > **Display** tab > **Flip Arrows** > **Properties** tab > **Move** > select a new position > Decimal Places **0** > **Enter** > **OK** [Fig. 18.9(b)] > press **RMB** > **Exit Edit Dimension** > **LMB** to deselect

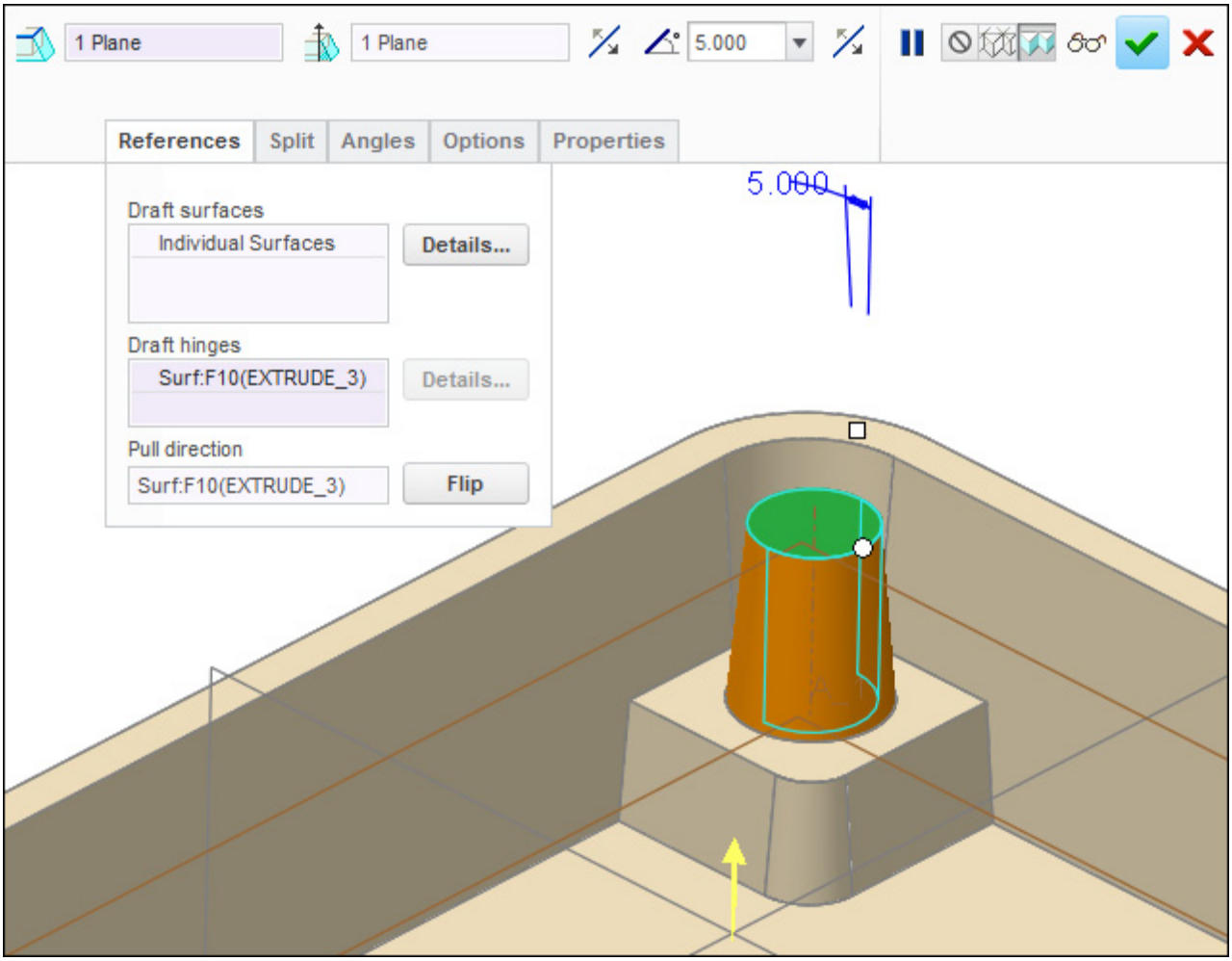



**Figure 18.9(a)** Draft the Circular Extrusion

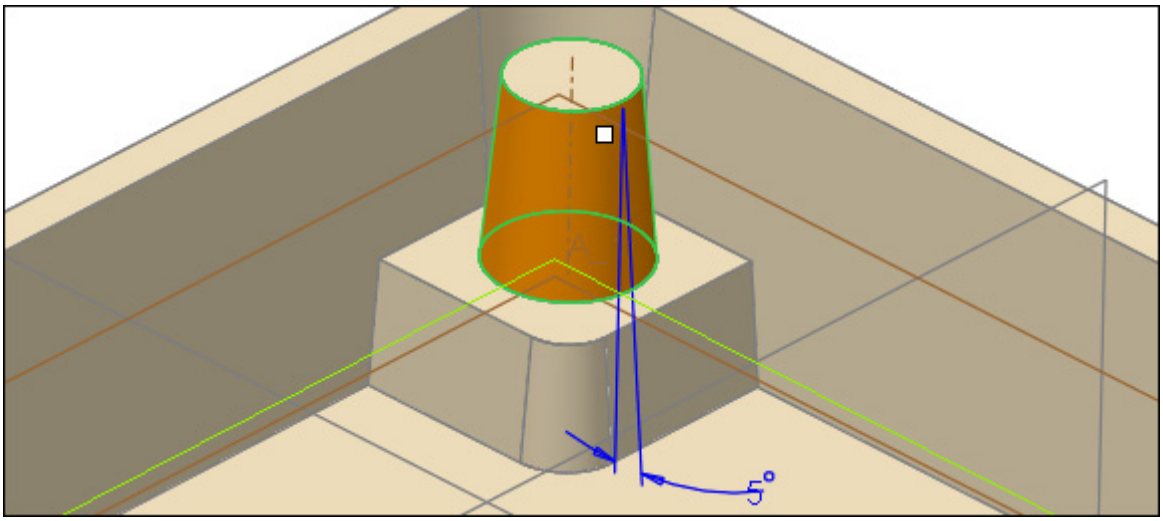



**Figure 18.9(b)** Edit the Dimensions Properties

Create a **.250** diameter coaxial hole on the upper surface of the circular extrusion [Fig. 18.10(a)]. Use "To Selected" to establish the hole's depth to the top surface of the pedestal [Fig. 18.10(b)].

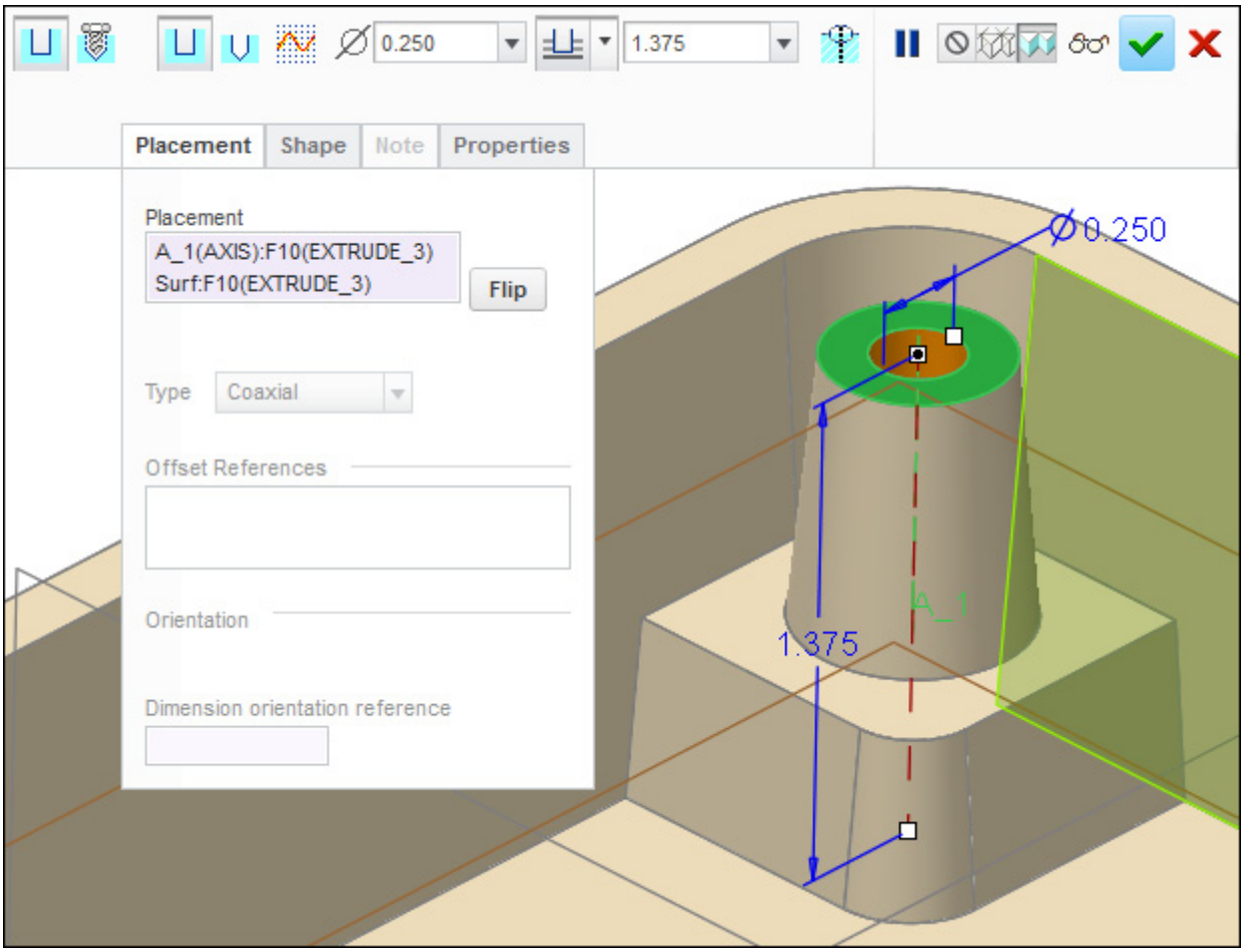



**Figure 18.10(a)** Coaxial Hole

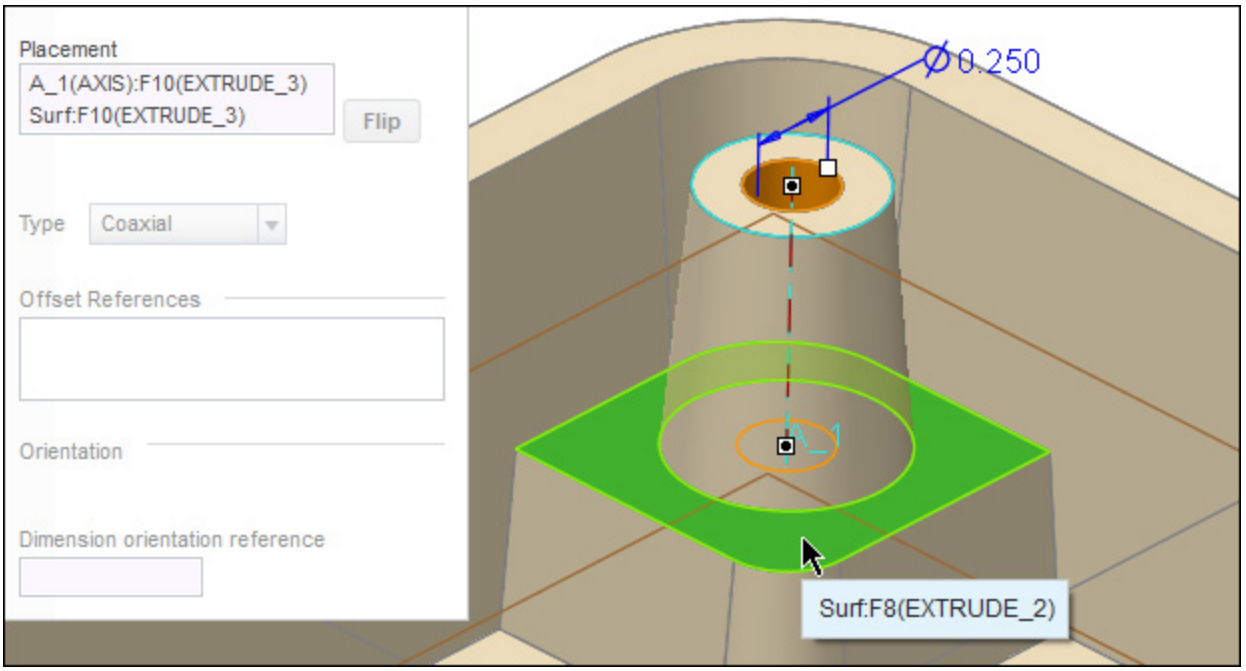



**Figure 18.10(b)** Hole Depth to Selected Surface

Next, add an internal draft of **.3°** to the coaxial hole (select the top surface of the cylinder as the draft hinge) (Fig. 18.11).

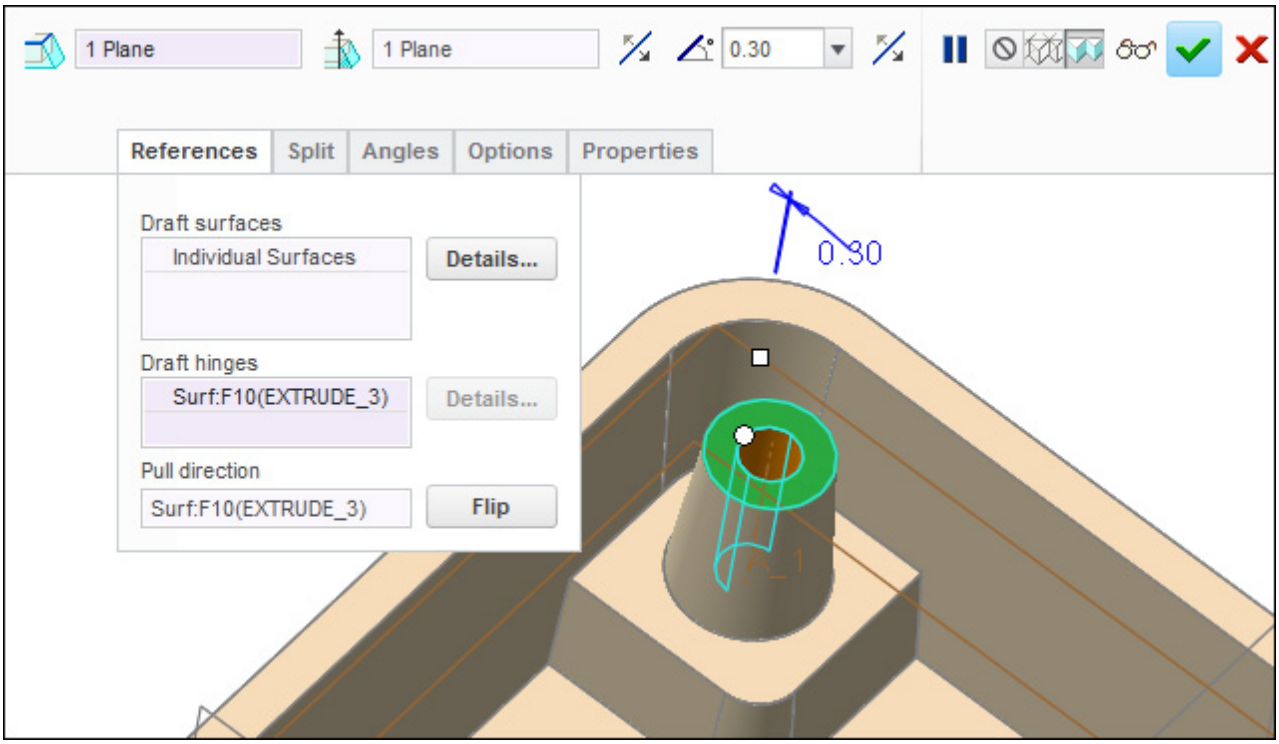



**Figure 18.11** Draft the Coaxial Hole

Create the **.0625** [Fig. 18.12(a)] and the **.125** rounds [Fig. 18.12(b)].

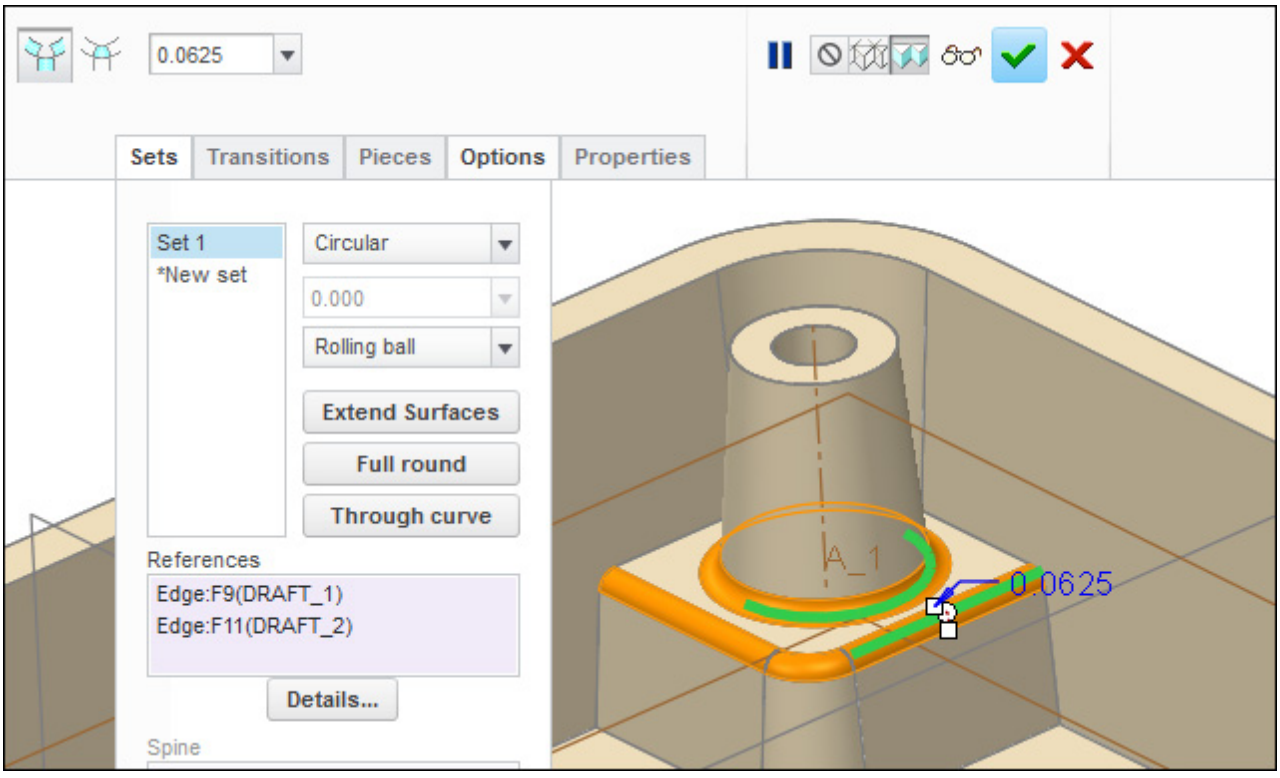



**Figure 18.12(a)** Round Set 1 (**R.0625**)

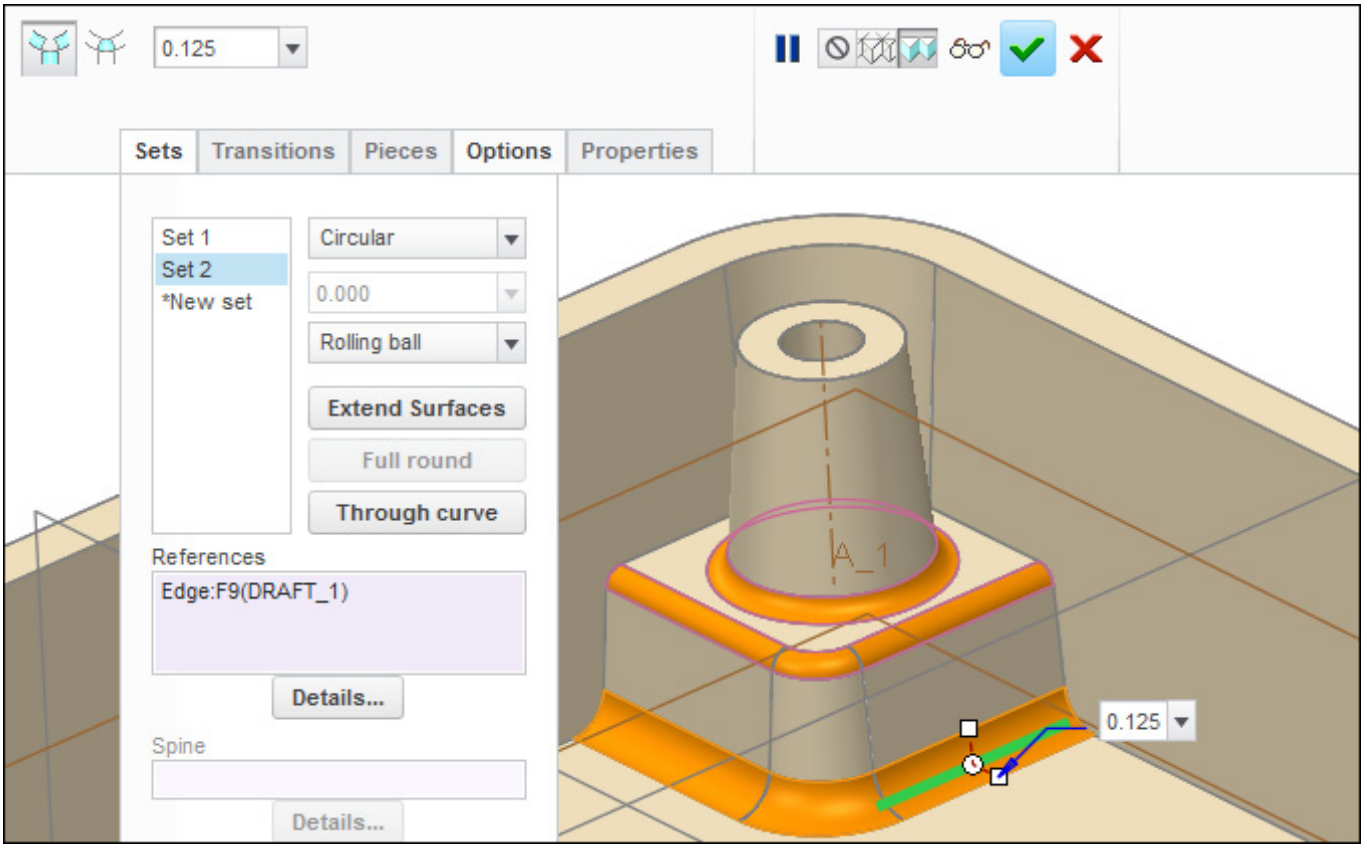



**Figure 18.12(b)** Round Set 2 (**R.125**) pedestal

Group the extrusions, the hole and the rounds. Select features **> RMB > Group** [Fig. 18.13(a)] **>** click on the group name **>** type **PEDESTAL > Enter** Group PEDESTAL [Fig. 18.13(b)] **> LMB** to deselect

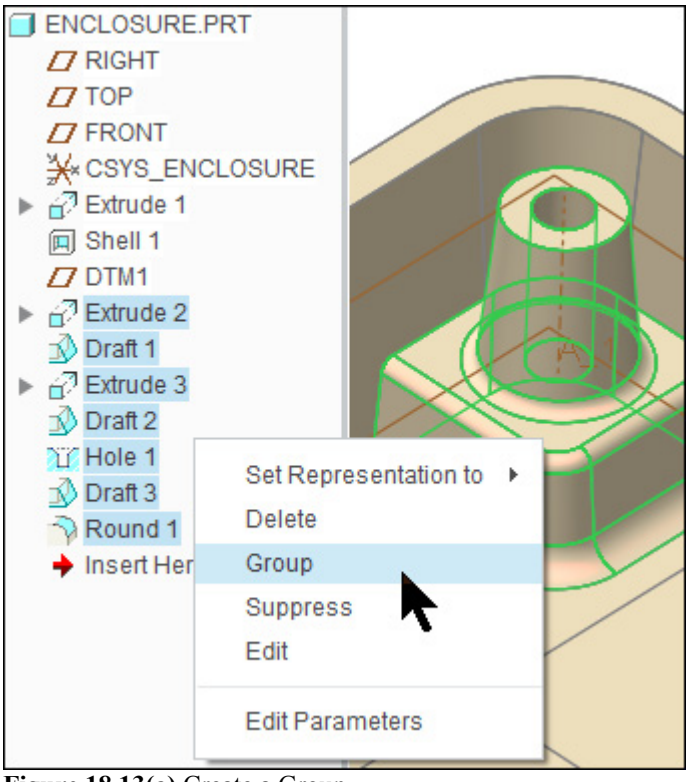



**Figure 18.13(a)** Create a Group

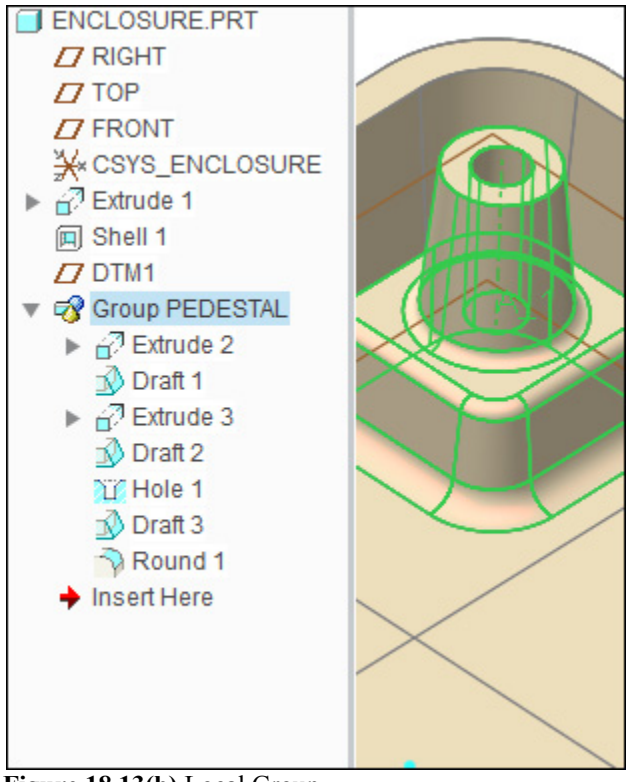



**Figure 18.13(b)** Local Group

Create three identical grouped features. From the Model Tree, select: Group **PEDESTAL** > Mirror > select datum **RIGHT** [Fig. 18.14(a)] > ✔ > with the **Ctrl** key pressed, select Group **PEDESTAL** and Group **COPIED_GROUP** from the Model Tree > release the **Ctrl** key > Mirror > select datum **TOP** [Fig. 18.14(b)] > ✔ **> Ctrl+S > MMB > File > Manage File > Delete Old Versions > Enter**

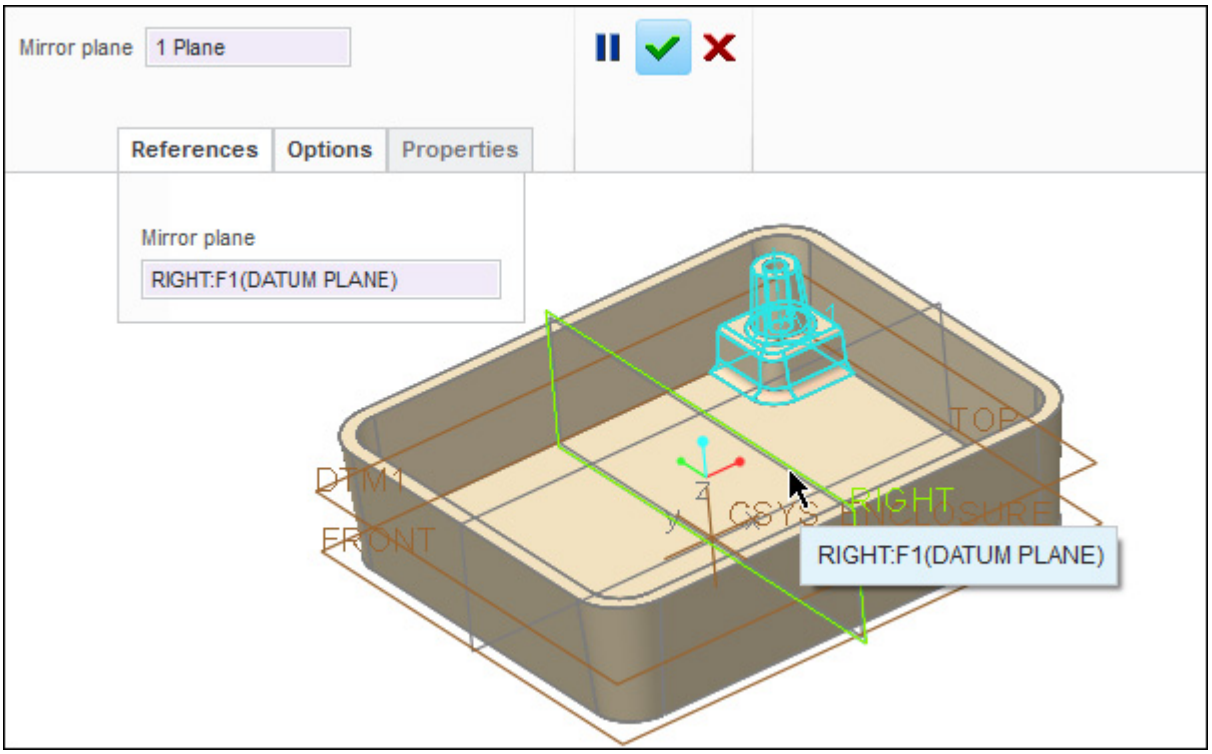



**Figure 18.14(a)** Group Copied and Mirrored about Datum RIGHT

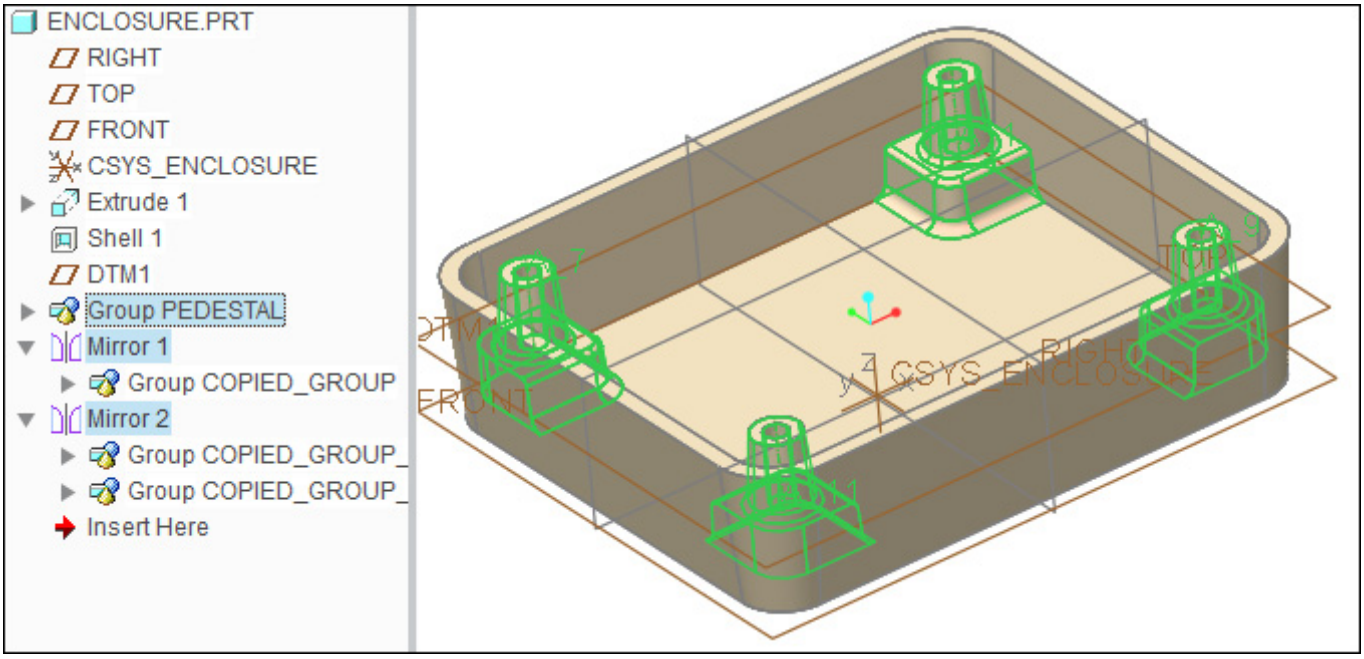



**Figure 18.14(b)** Groups Copied and Mirrored about Datum TOP

Create the internal round, click: Round > type **.125** > select the inside of the shelled wall as the first reference [Fig. 18.15(a)] > **Sets** tab > press and hold the **Ctrl** key > select the top surface of the pedestal as the second reference [Fig. 18.15(b)] > release the **Ctrl** key > > **LMB** to deselect

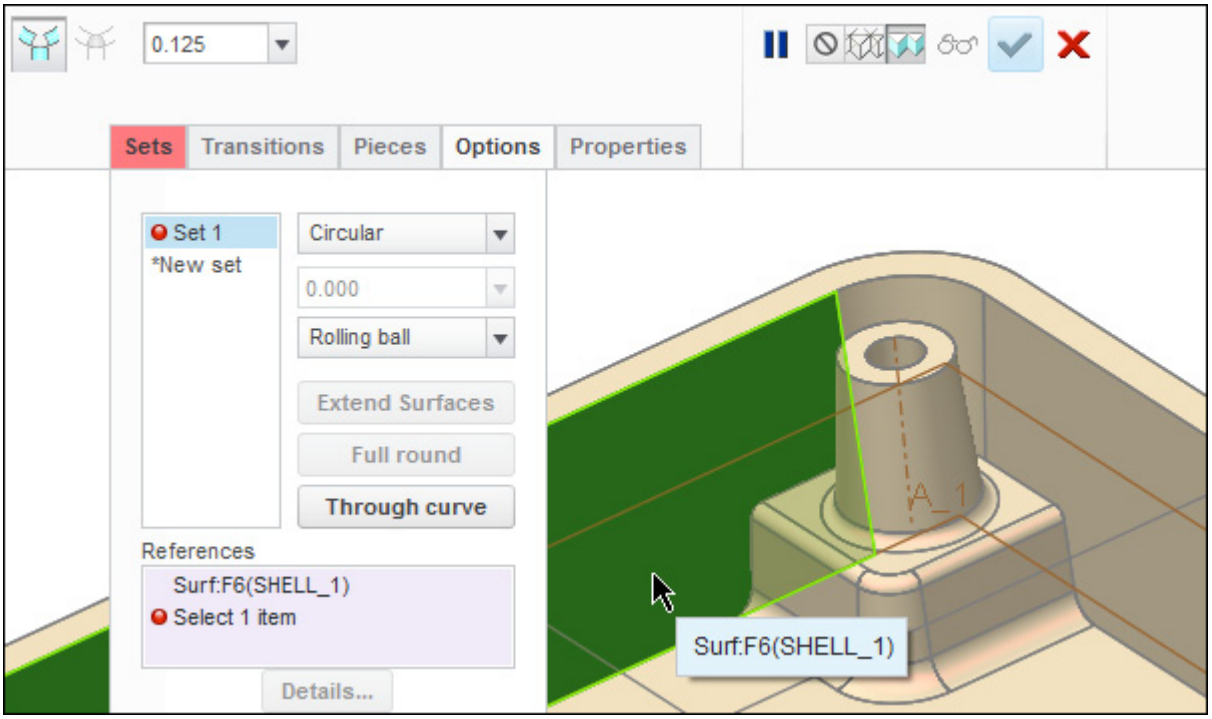



**Figure 18.15(a)** Select First Reference

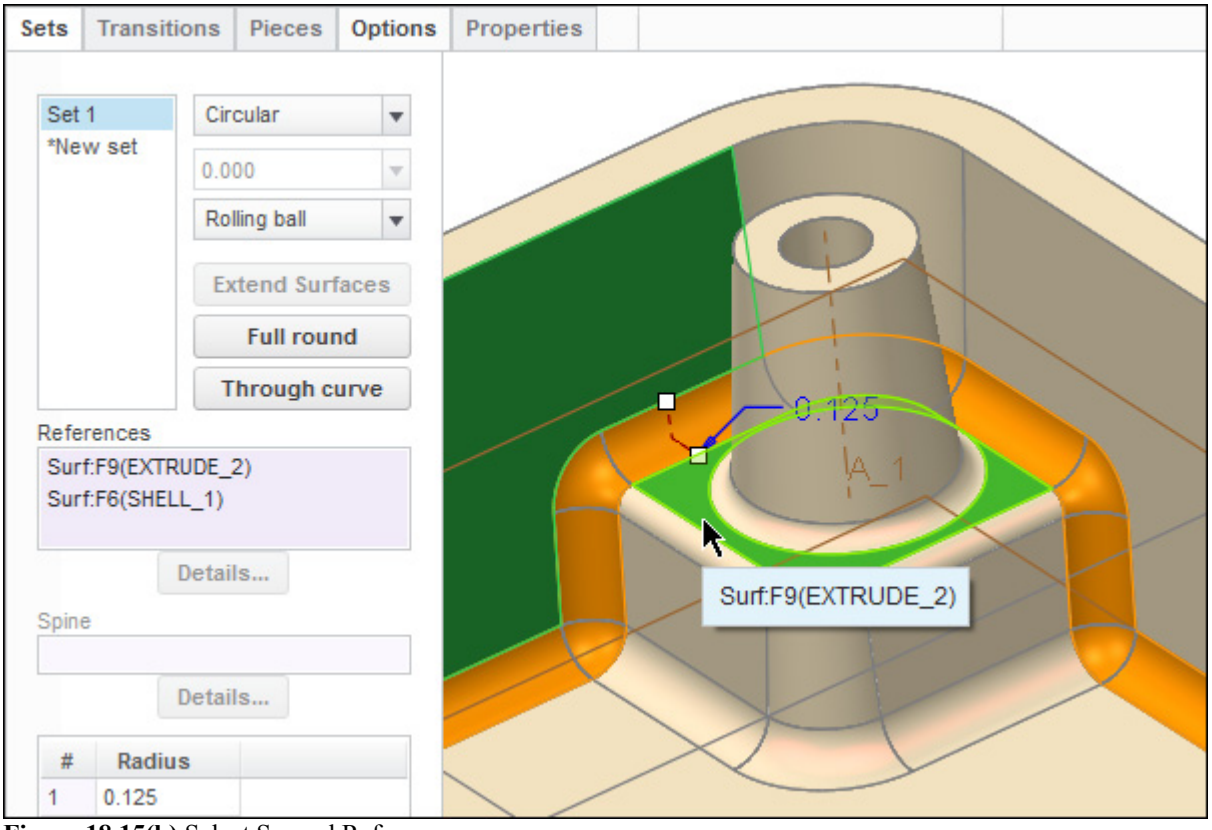



**Figure 18.15(b)** Select Second Reference

Click: Plane > References: select axis **A_1** [Fig. 18.16(a)] > press and hold the **Ctrl** key > References: select axis **A_7** *(your id's may be different)* [Fig. 18.16(b)] > release the **Ctrl** key > **OK** (creates DTM2)

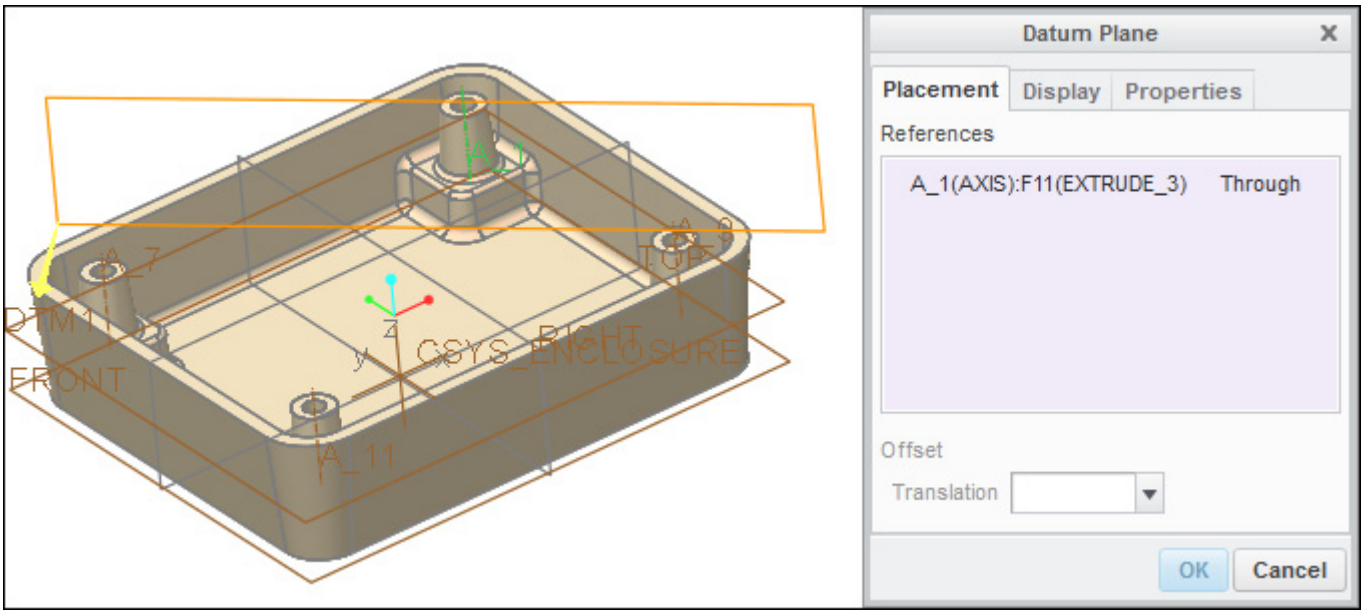



**Figure 18.16(a)** Select Axis A_2 *(your id's may be different)*

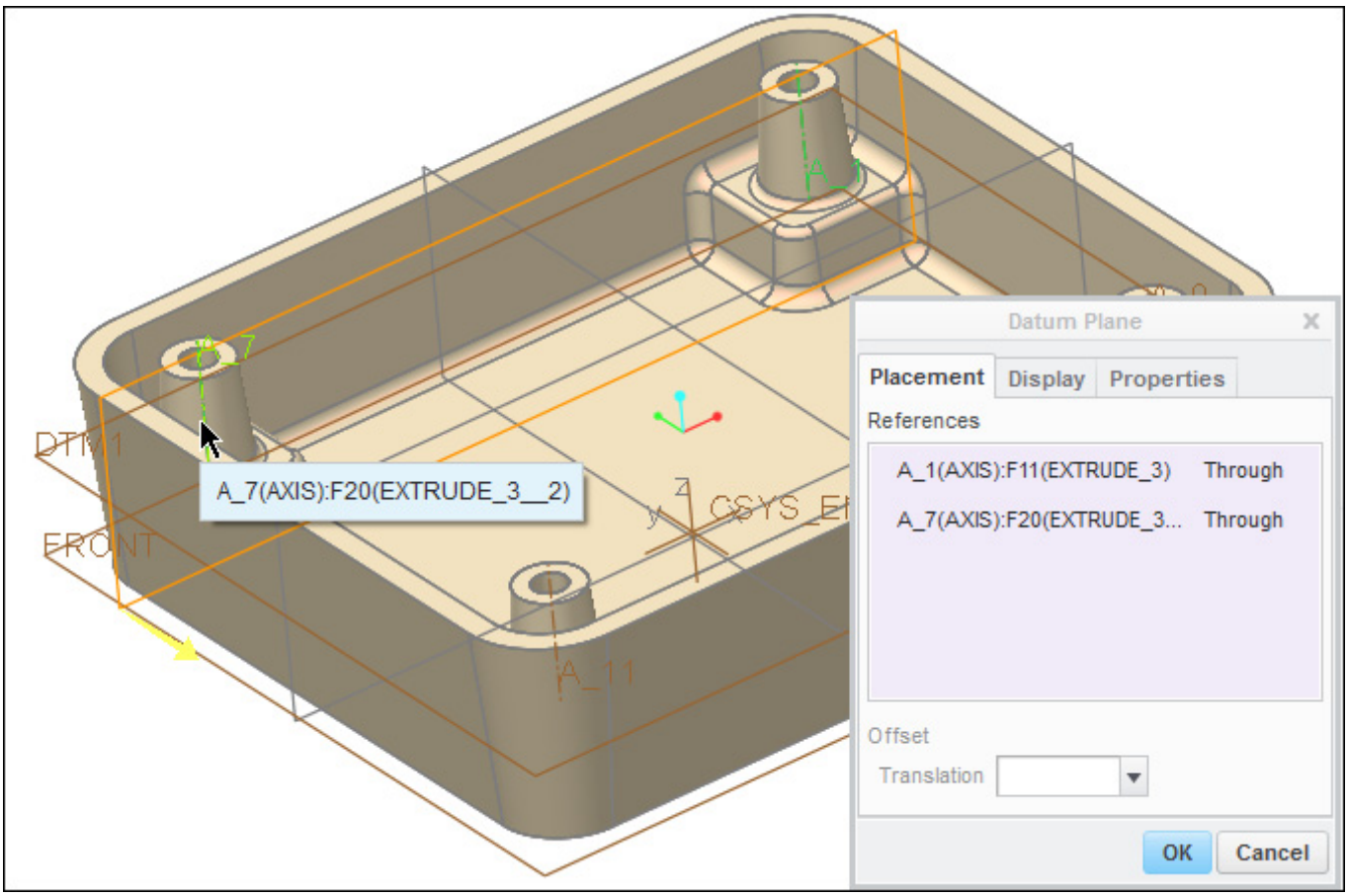



**Figure 18.16(b)** Select Axis A_6 *(your id's may be different)*

Create a cross section through the part. Click:  **Open the view manager > Xsec** tab **> New >** type name **A > Enter > Planar > Single > Done > Plane >** select **DTM2 >** select **A >** press **RMB > Visibility** [Fig. 18.17(a)] > press **RMB > Set Active** [Fig. 18.17(b)] > select **No Cross Section >** press **RMB > Set Active >** select **A >** press **RMB > Visibility** (uncheck) **> Close > Ctrl+S > OK**

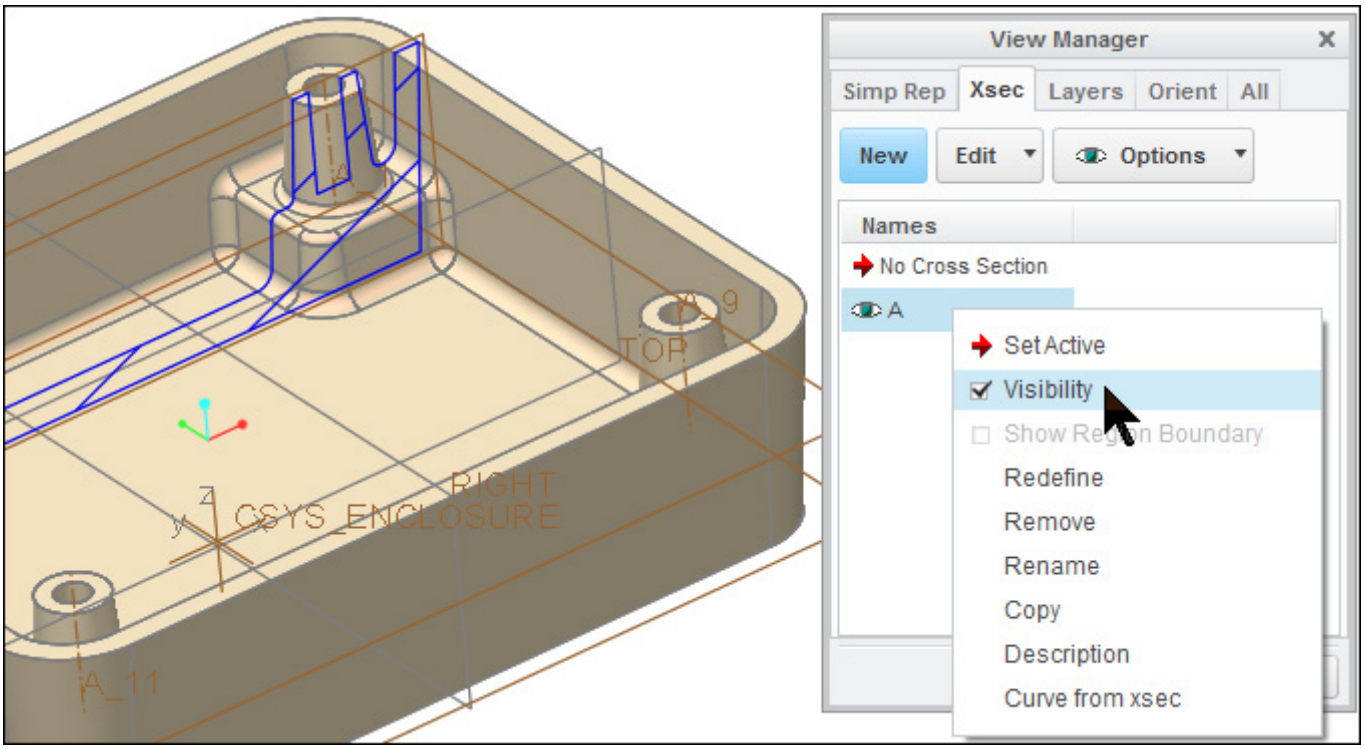



**Figure 18.17(a)** Show X-Section

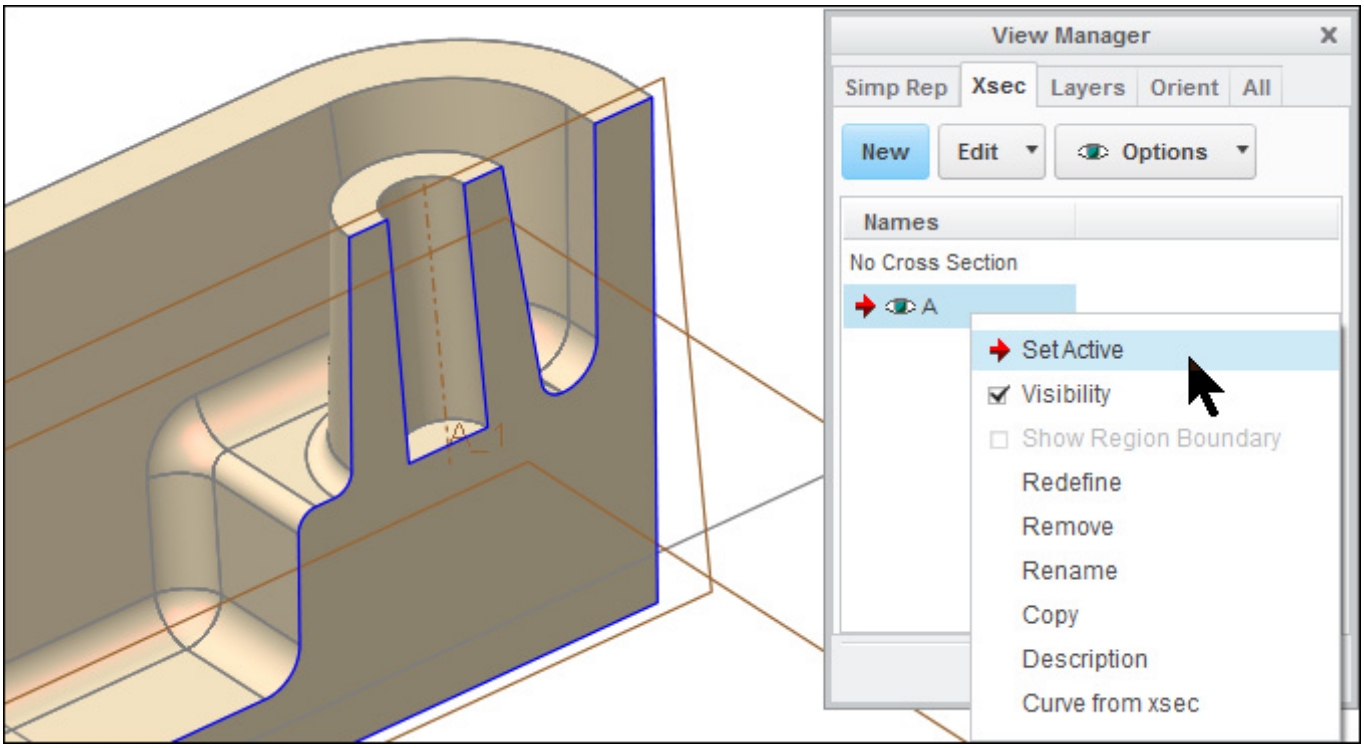



**Figure 18.17(b)** Set Active X-Section A

Before creating the text extrusion, **Suppress** all the features after the shell command. Expand the Model Tree to include the feature number and status. Click: **Settings > Tree Filters >** toggle on all options (Fig. 18.18) **> Apply > OK**

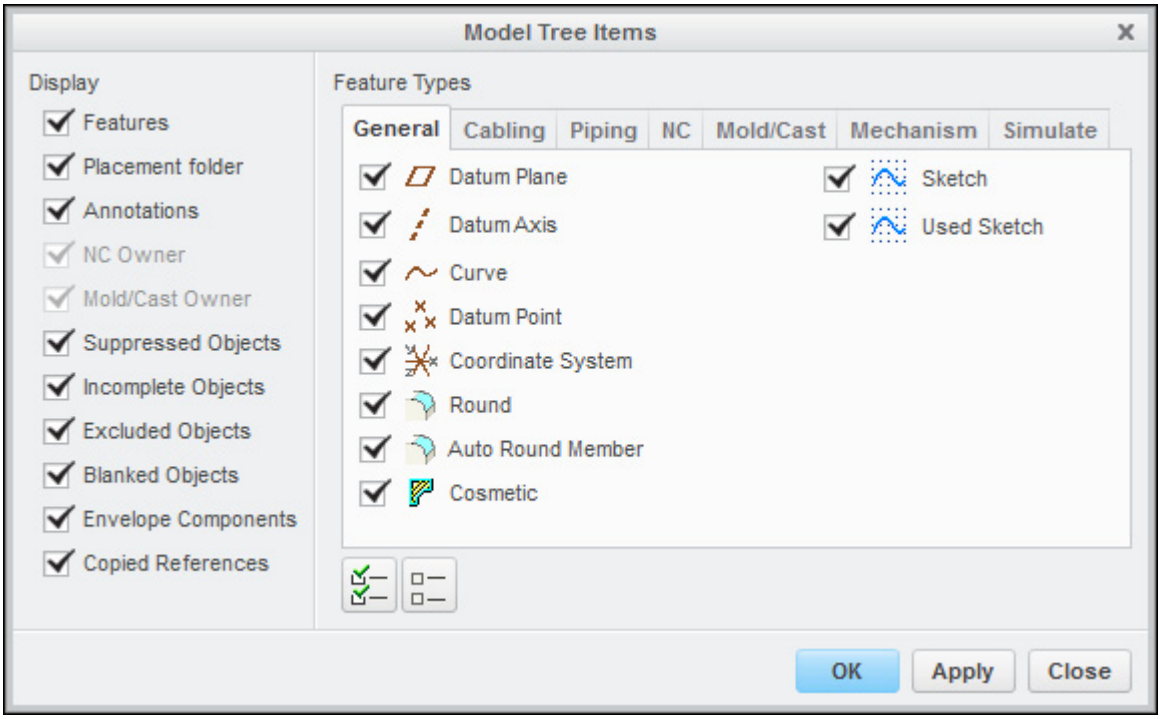



**Figure 18.18** Model Tree Items Dialog Box

Click on **Group PEDESTAL** in the Model Tree **>** press and hold the **Shift** key **>** click **DTM2** in the Model Tree (Fig. 18.19(a) **>** press **RMB > Suppress** (Fig. 18.19(b) **> OK** (Fig. 18.19(c) **> LMB** to deselect **> Ctrl+S > Enter**

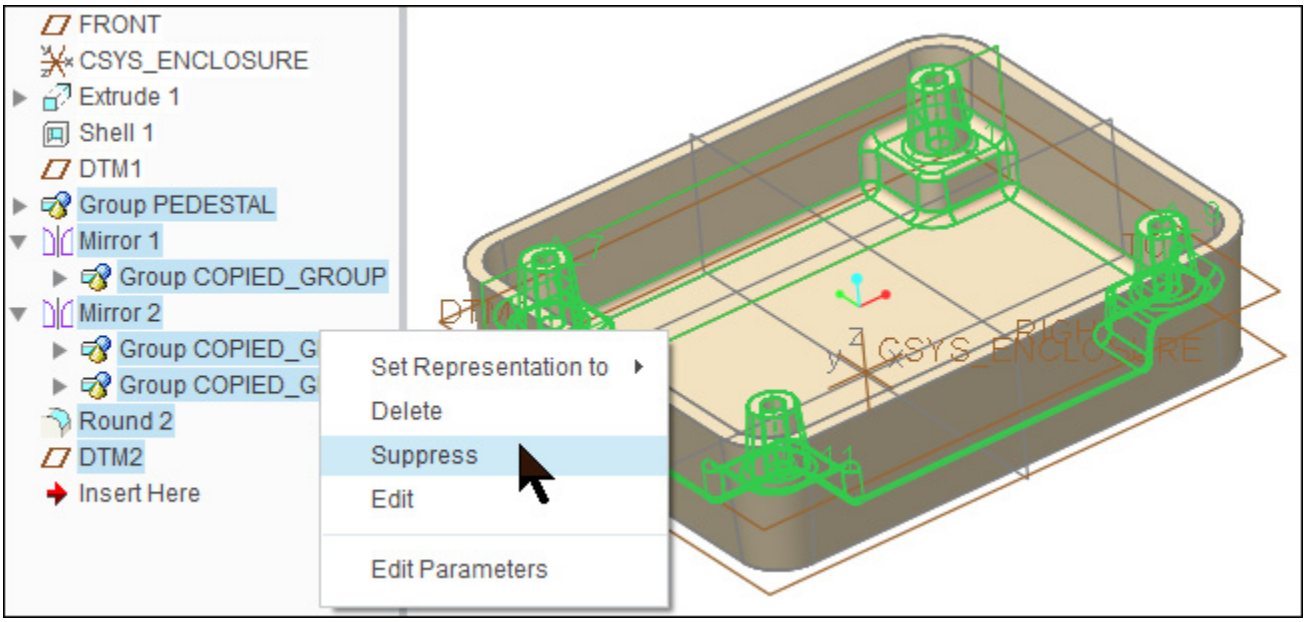



**Figure 18.19(a)** Select the Features in the Model Tree to be Suppressed

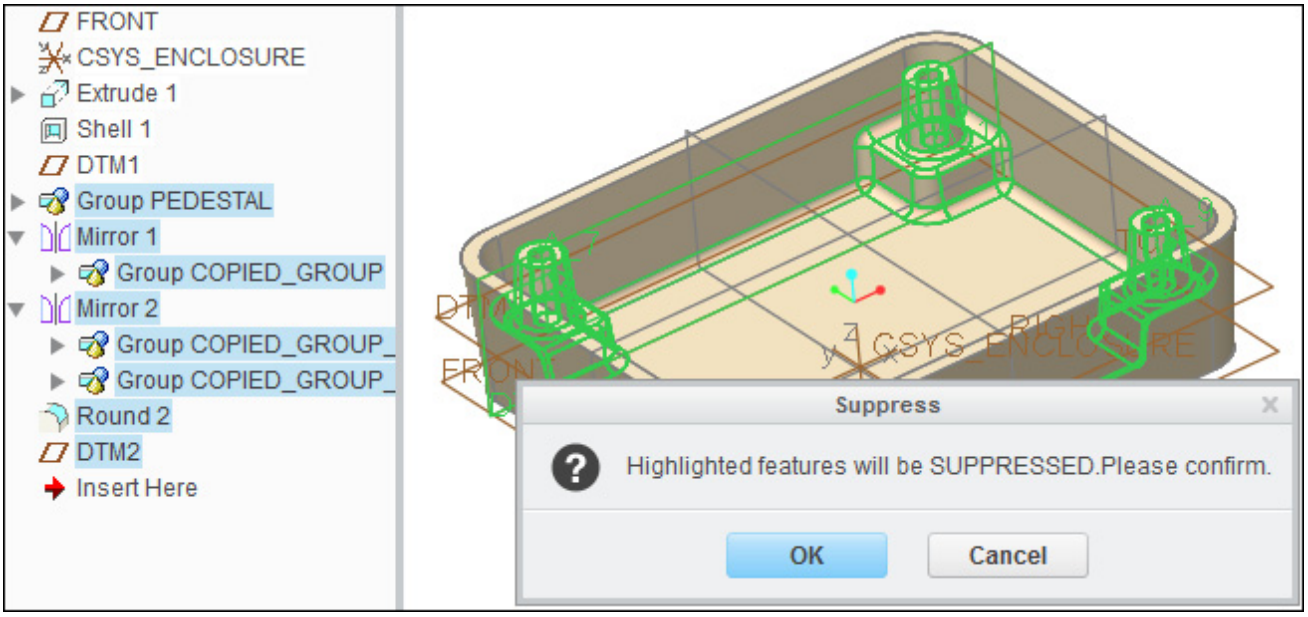



**Figure 18.19(b)** Highlighted Features

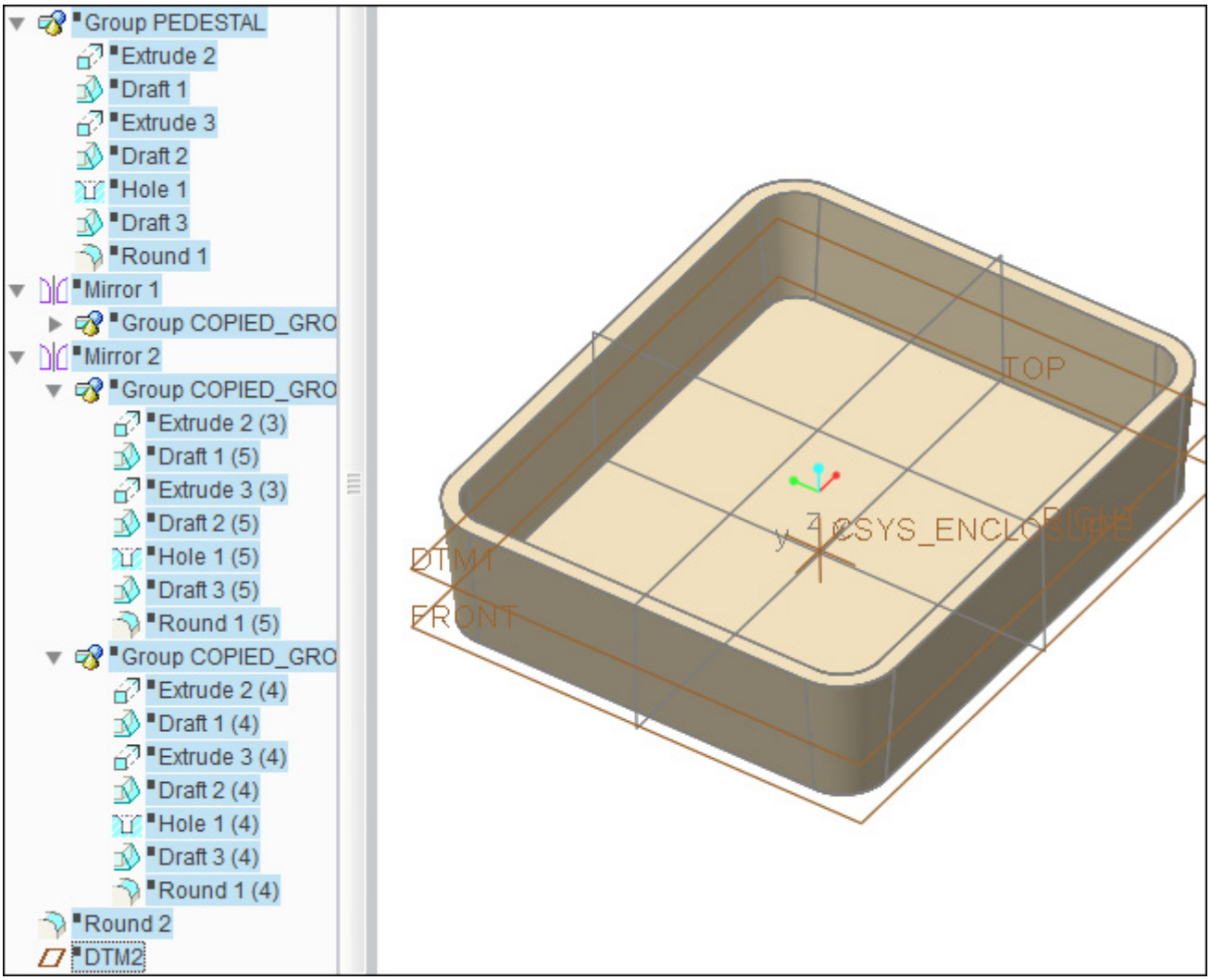



**Figure 18.19(c)** Suppressed Features

The regeneration time for your model will now be shorter. Next, add the text extrusion.

Press: **Ctrl+D >** Extrude **>** press **RMB > Define Internal Sketch >** Sketch Plane- Plane: select the *inside surface of the enclosure* for the sketching plane [Fig. 18.20(a)] **> Sketch >**  **>** Text **>** select *start point* of line to determine text starting position [Fig. 18.20(b)] **>** select *second point* of line to determine text height and orientation [Fig. 18.20(c)]

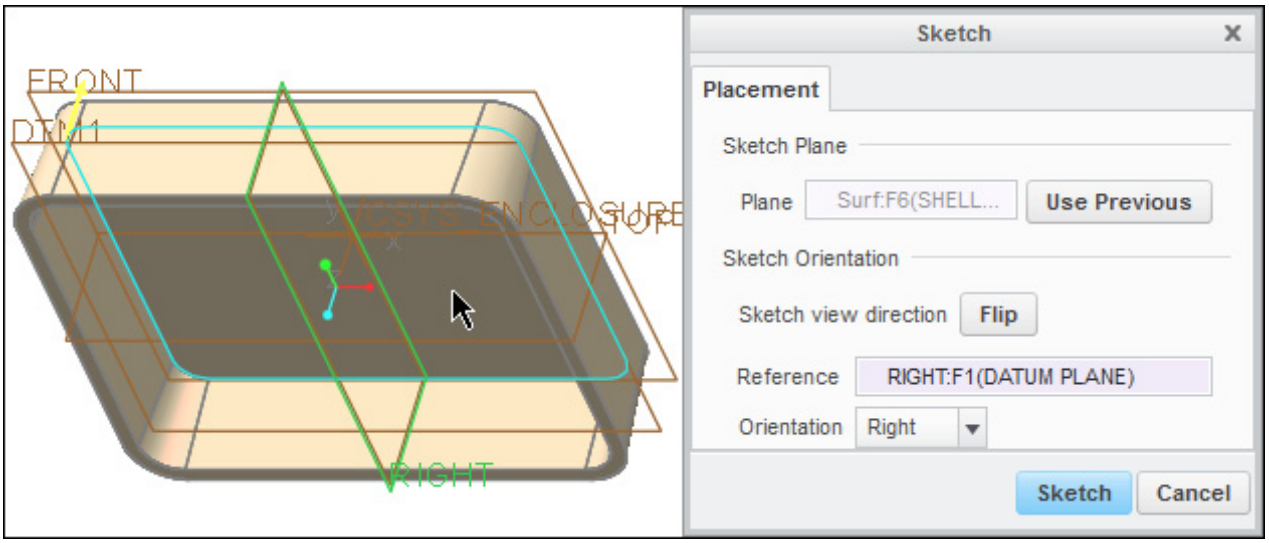



**Figure 18.20(a)** Sketching Plane, Inside Surface

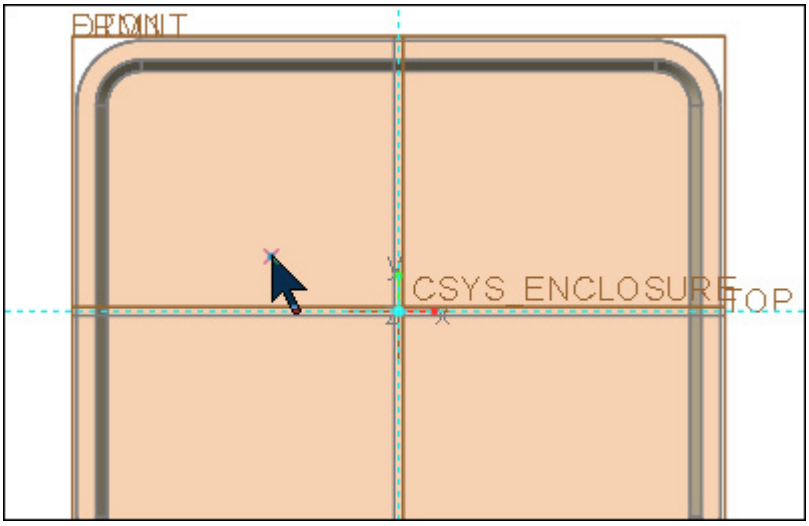



**Figure 18.20(b)** Pick First Point to Determine the Starting Point of the Lettering

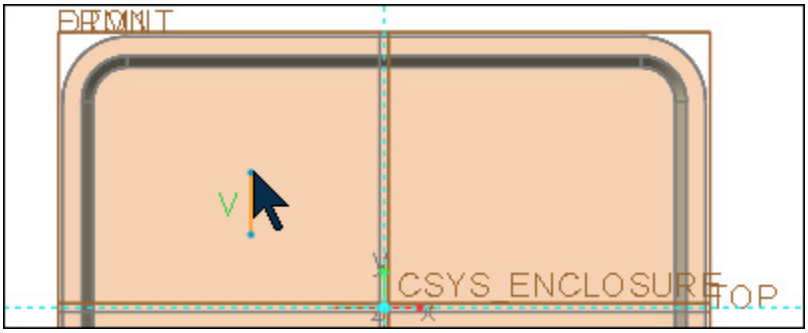



**Figure 18.20(c)** Pick Second Point to Determine the Height of the Lettering

Click:  > 

all off > **CFS-2134** in Text line field [Fig. 18.20(d)] > **OK** > **MMB** > window-in (select) the sketch to capture all dimensions > press **RMB** > **Modify** > modify the dimensions [Fig. 18.20(e)] >  from Modify Dimensions dialog box >  > 

> **LMB** in the Graphics Window > 

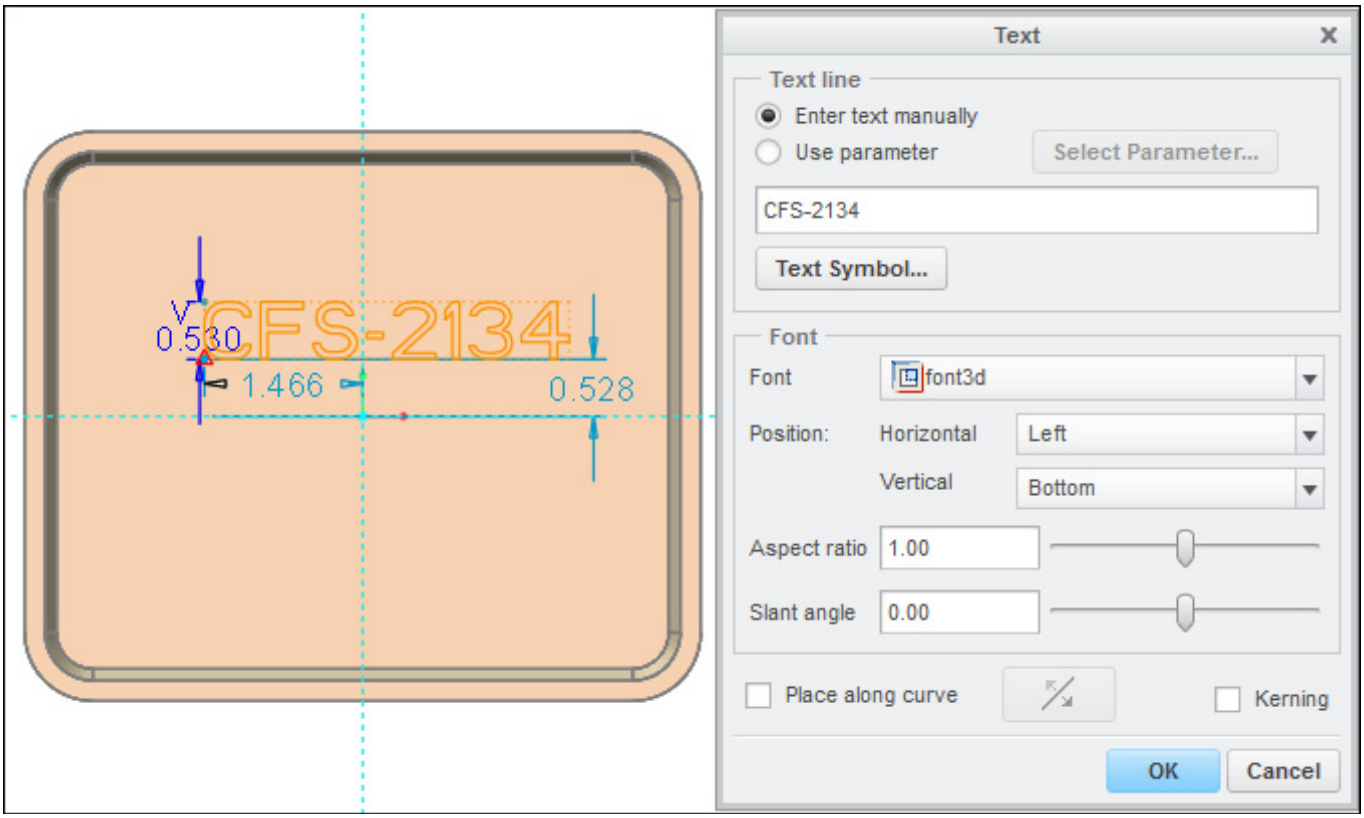



**Figure 18.20(d)** Type the Text "**CFS-2134**" (case sensitive)

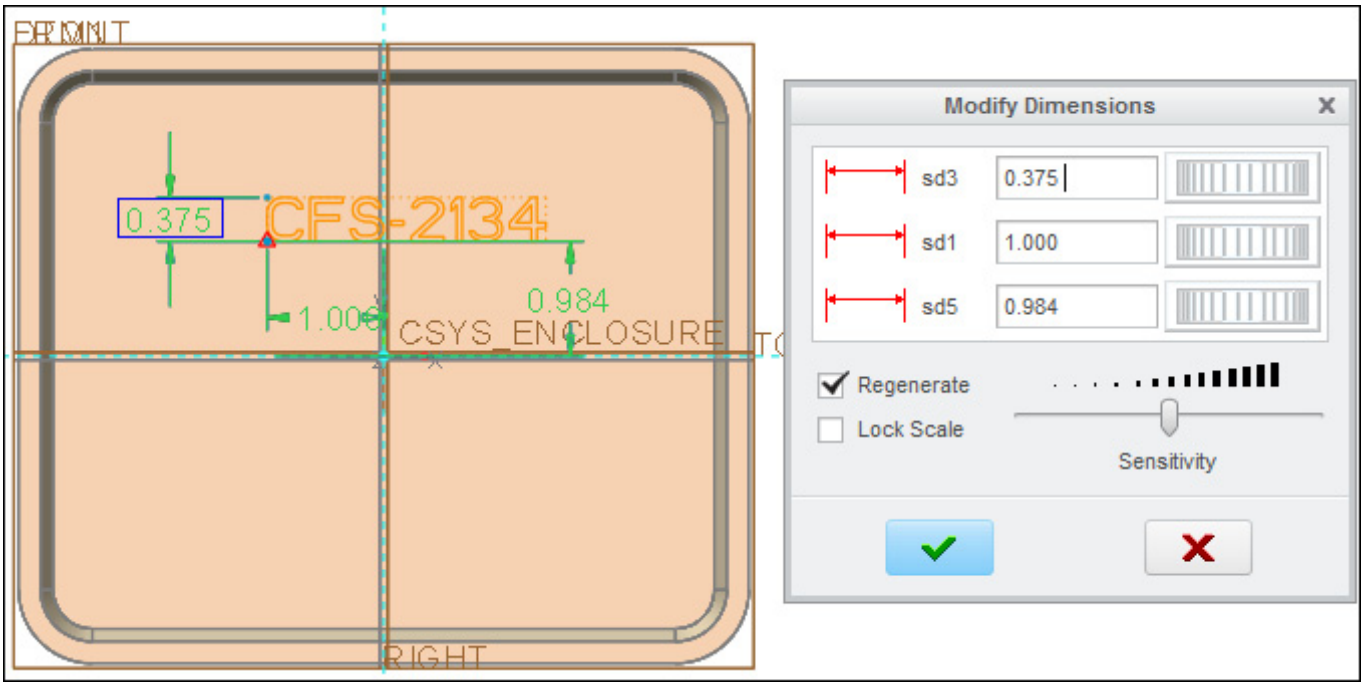



**Figure 18.20(e)** Modified Dimensions [dimensions are slightly different than those shown in Figure 18.2(c)]

Click: **MMB** to spin the model [Fig. 18.20(f)] **>** double-click on the height dimension and modify it to **.0625** **> Enter** [Fig. 18.20(g)] **>**  [Fig. 18.20(h)] **> Ctrl+S > Enter**

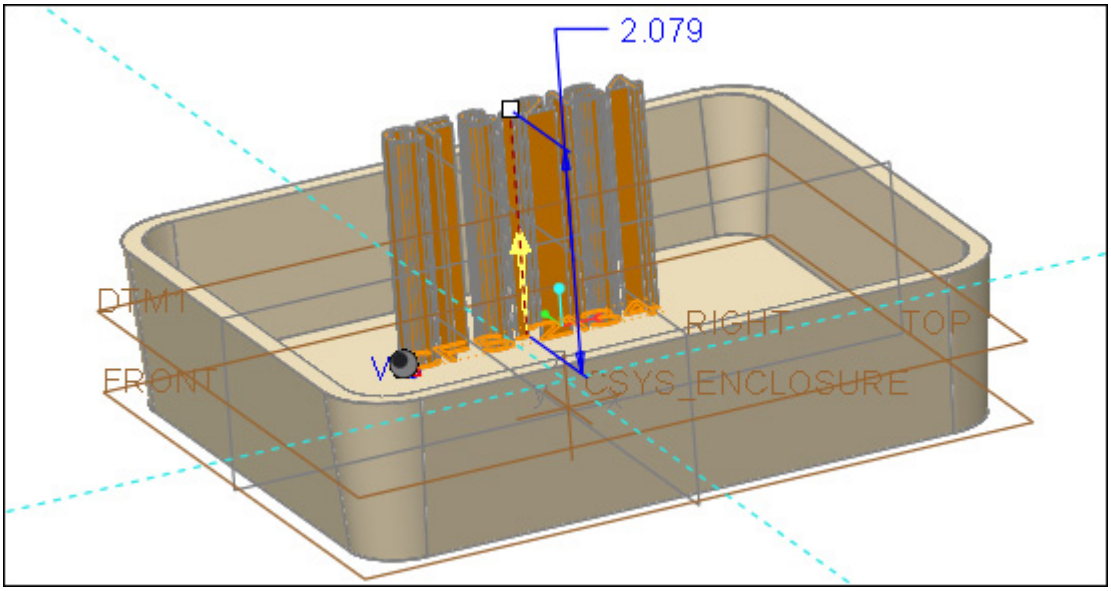



**Figure 18.20(f)** Dynamic Preview

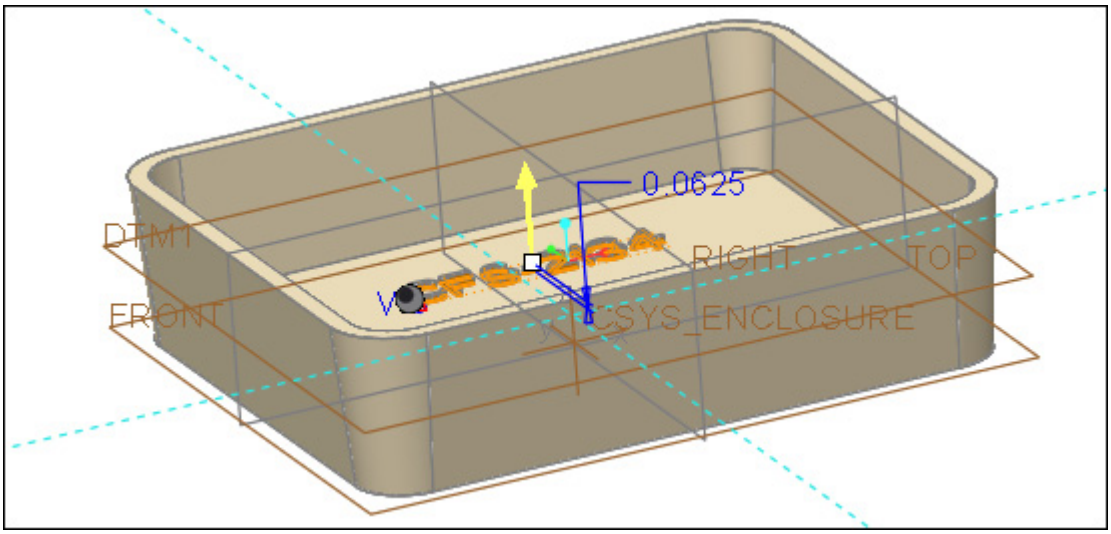



**Figure 18.20(g)** Modified Depth

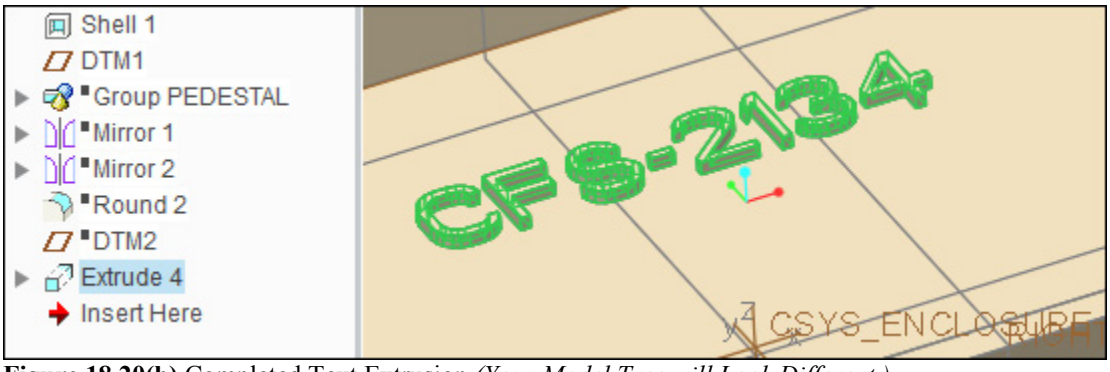



**Figure 18.20(h)** Completed Text Extrusion *(Your Model Tree will Look Different.)*

Click: **LMB** to deselect **>** select all of the suppressed features in the Model Tree **> RMB > Resume >** [Figs. 18.21(a-b)] **> Ctrl+D > Ctrl+R > Ctrl+S > Enter** [Fig. 18.21(c)] **> LMB** to deselect

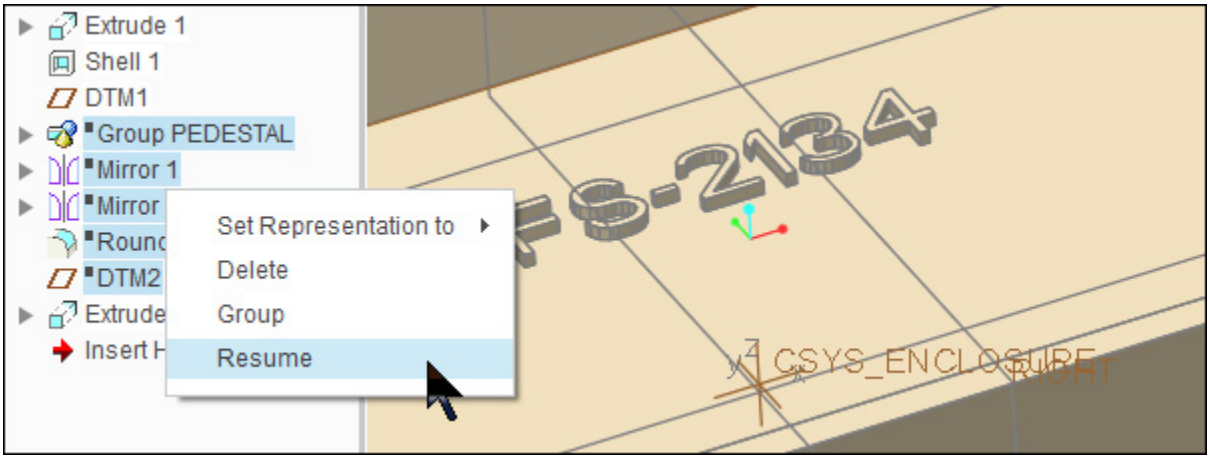



**Figure 18.21(a)** Resume

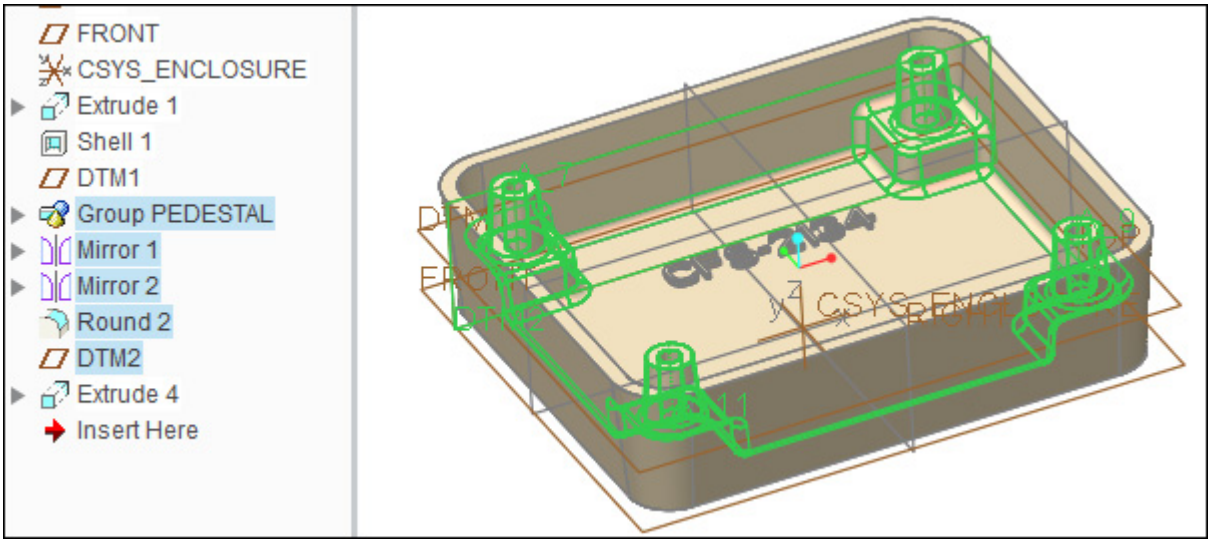



**Figure 18.21(b)** Suppressed Features Resumed

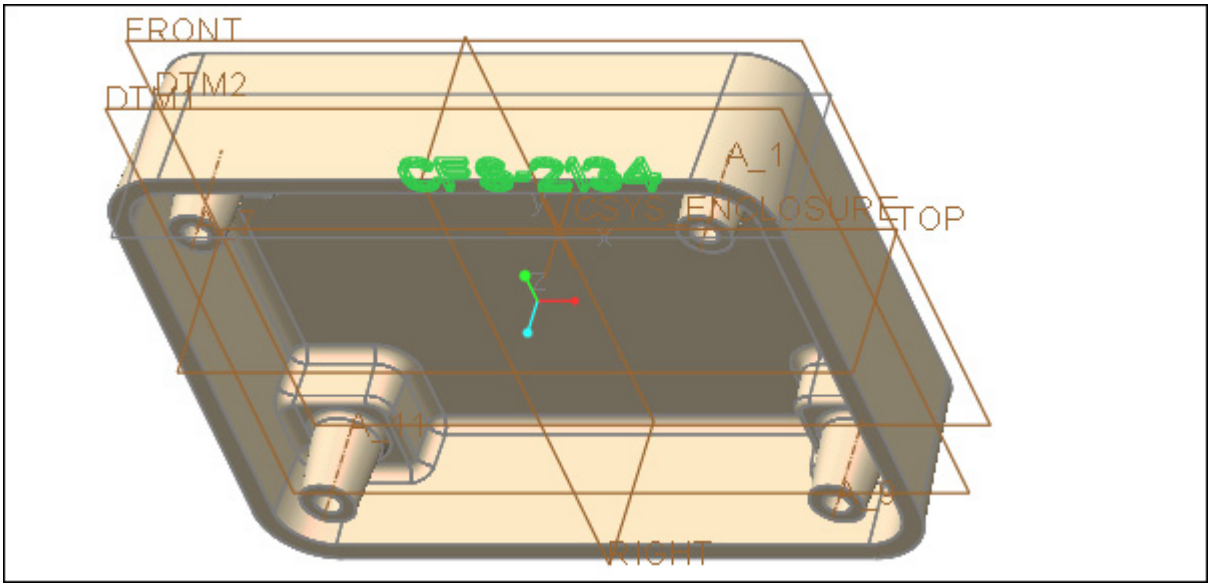
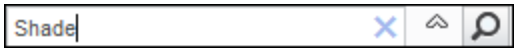


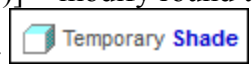

**Figure 18.21(c)** Standard Orientation

Click: **MMB** to spin the part >  > 

> 

> select the top surface > select on the parts' edge [Fig. 18.22(a)] > press and hold **Shift** key > select the surface again [Fig. 18.22(b)] *(Top surface edges will highlight, as the loop was selected. Note that this method was not necessary for this round since all the edges were tangent, but this was used to demonstrate another process of selection)* > press **RMB > Round Edges** [Fig. 18.22(c)] > modify round to **.1875** [Fig. 18.22d)] > **Enter** >  > **LMB** > 

> 

[Fig. 18.22(e)]

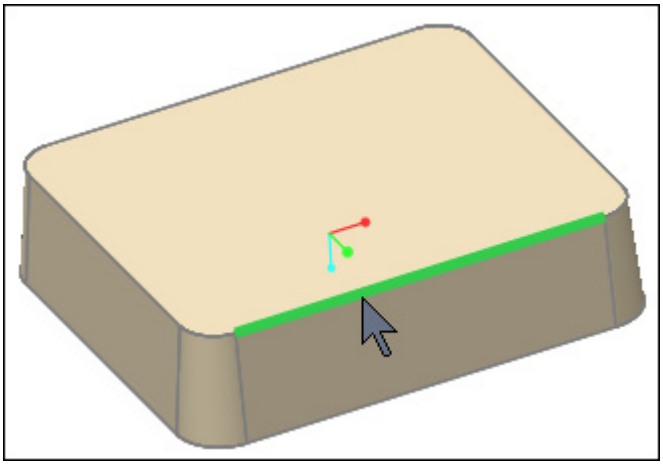


**Figure 18.22(a)** Select the Edge

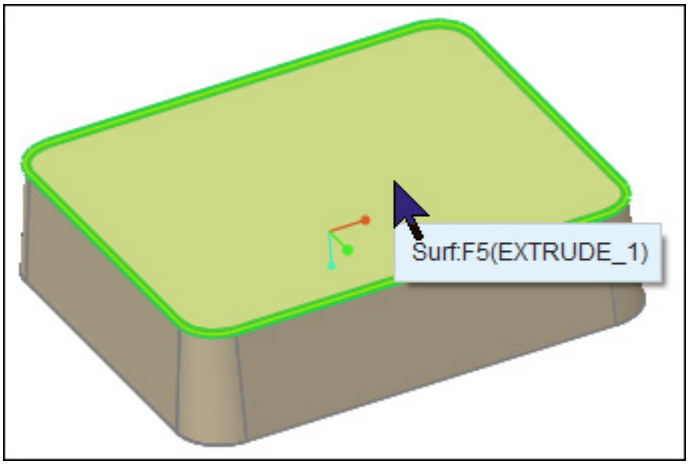



**Figure 18.22(b)** Shift + Select the Surface

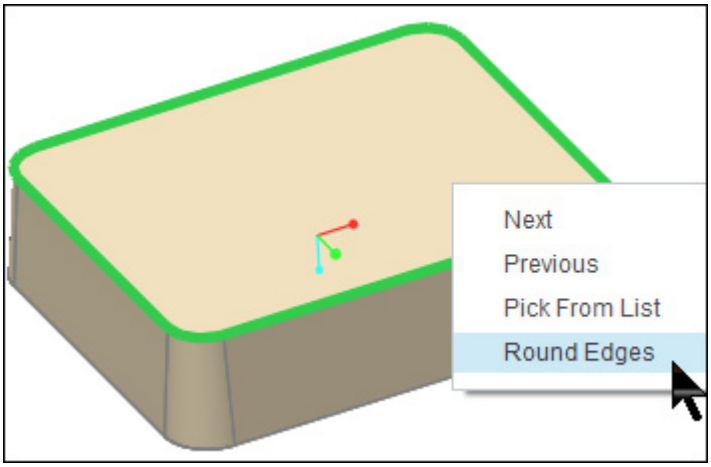



**Figure 18.22(c)** Round Edges

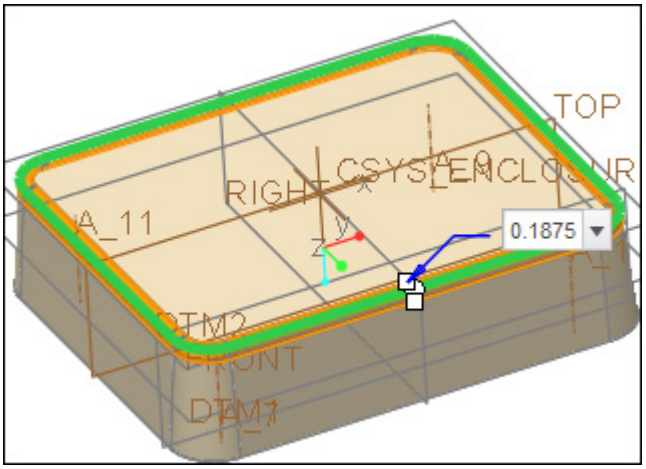



**Figure 18.22(d)** Modify to **.1875**

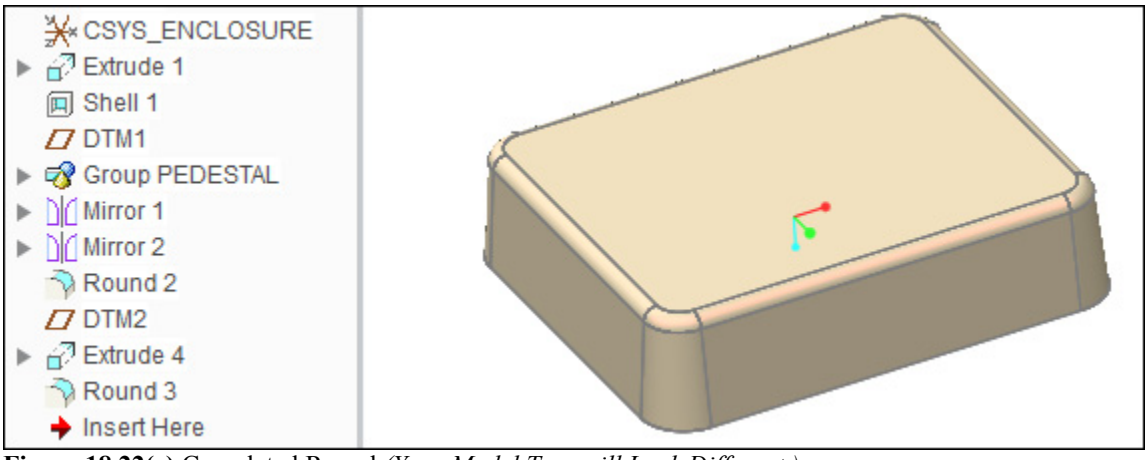



**Figure 18.22(e)** Completed Round *(Your Model Tree will Look Different.)*

Press: **Ctrl+D** > select the text extrusion in the Model Tree > **View** tab > Orientation > Orient Mode > Orientation > Orient Type > Velocity [Fig. 18.23(a)] > hold down the **MMB** and move the cursor about the screen to orbit the model [Fig. 18.23(b)] > release the **RMB** > press **RMB > Exit Orient Mode**

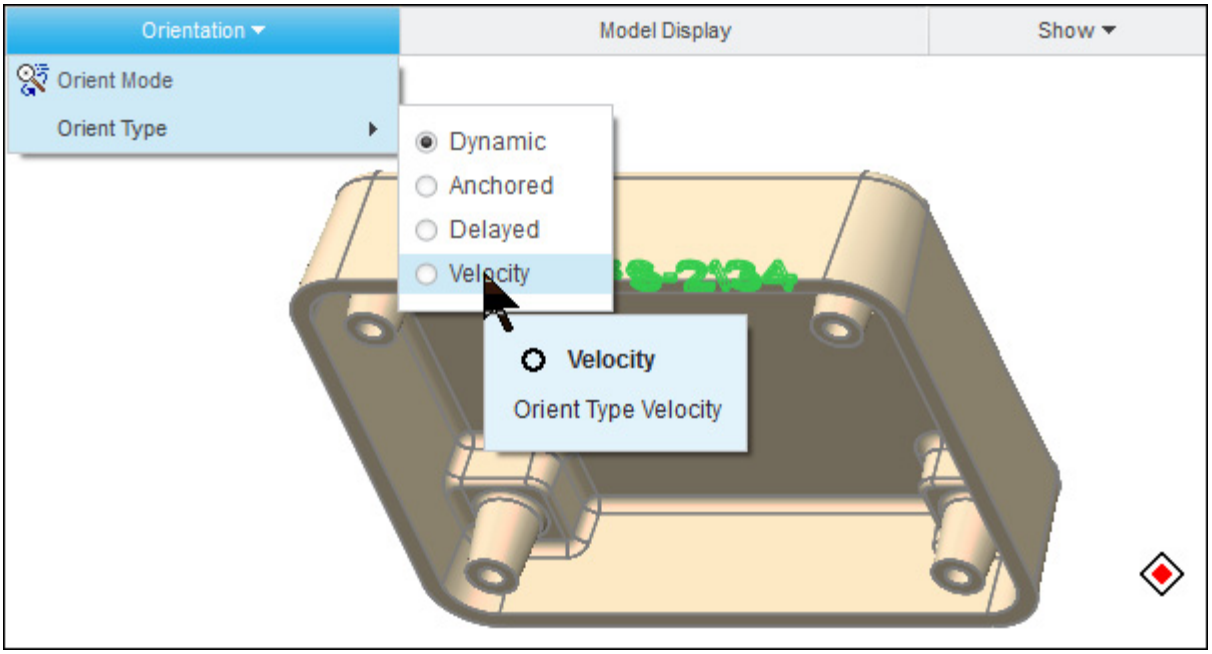



**Figure 18.23(a)** Orient Mode Velocity

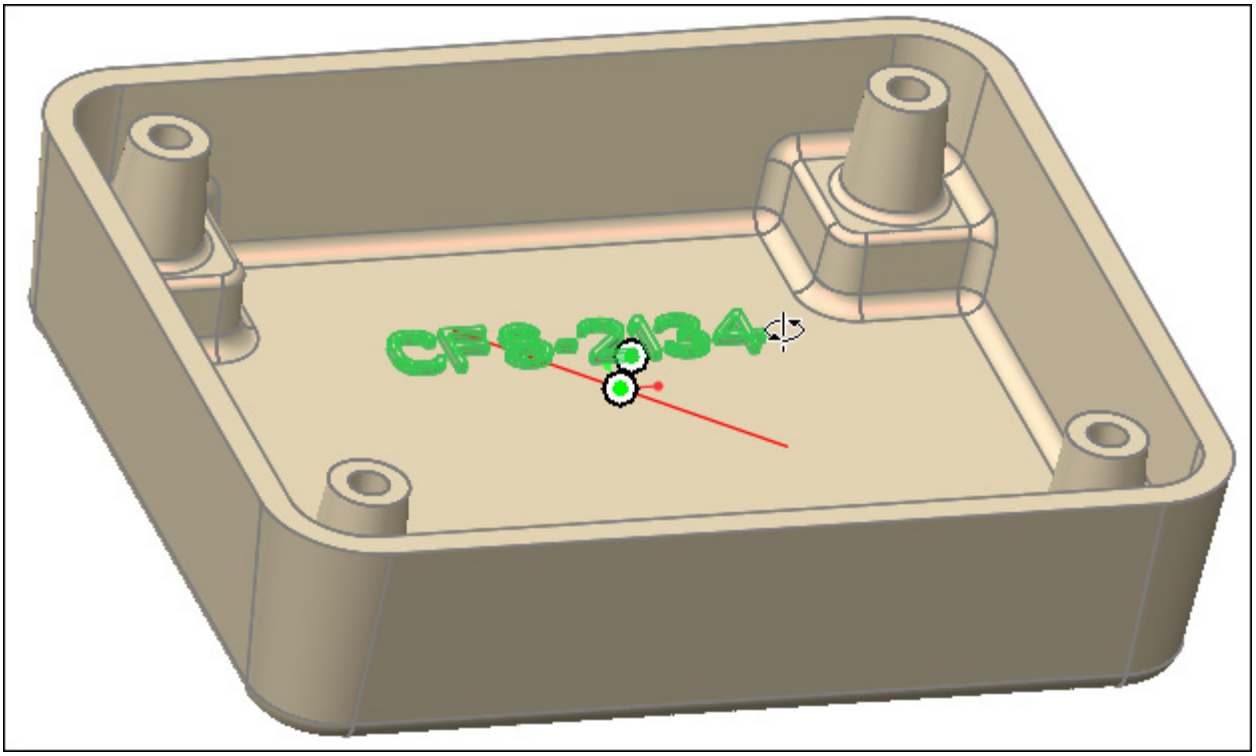


**Figure 18.23(b)** Spinning the Model in Real Time

With the text extrusion selected, press: **RMB > Edit > Model** tab [Fig. 18.24(a)] **>** drag the handle to **.250** [Fig. 18.24(b)] **> Ctrl+G** Regenerate **> Ctrl+D > Ctrl+S > Enter > File > Manage File > Delete Old Versions > Enter** [Fig. 18.24(c)] **> LMB** to deselect

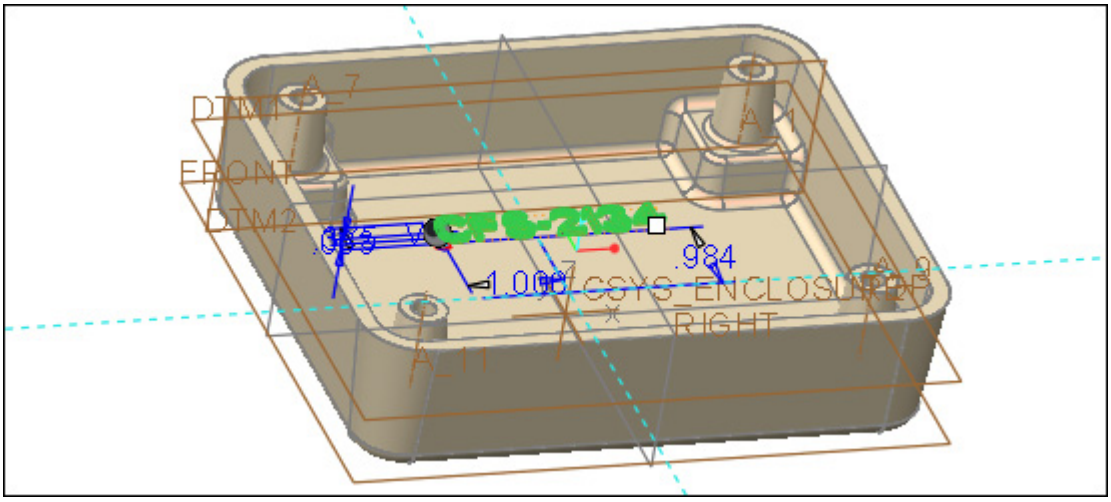



**Figure 18.24(a)** Edit the Text Extrusion

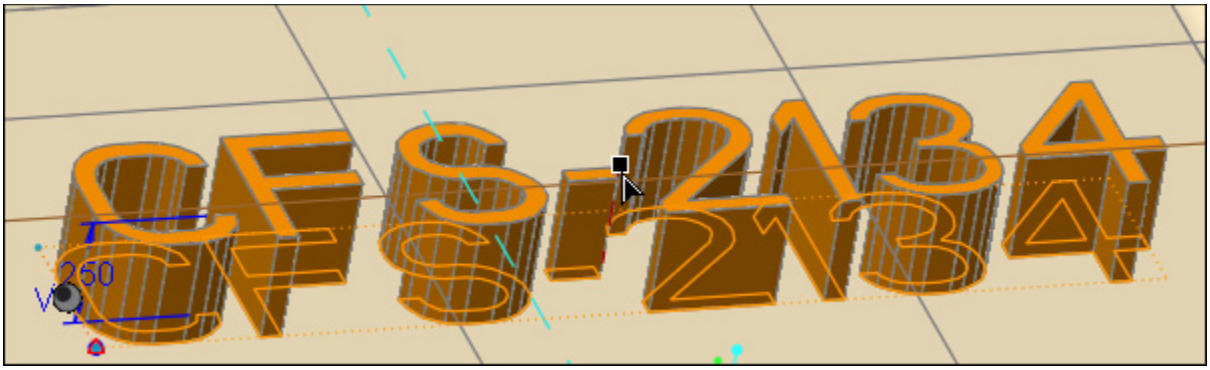



**Figure 18.24(b)** Drag the White Drag Handle to **.250**

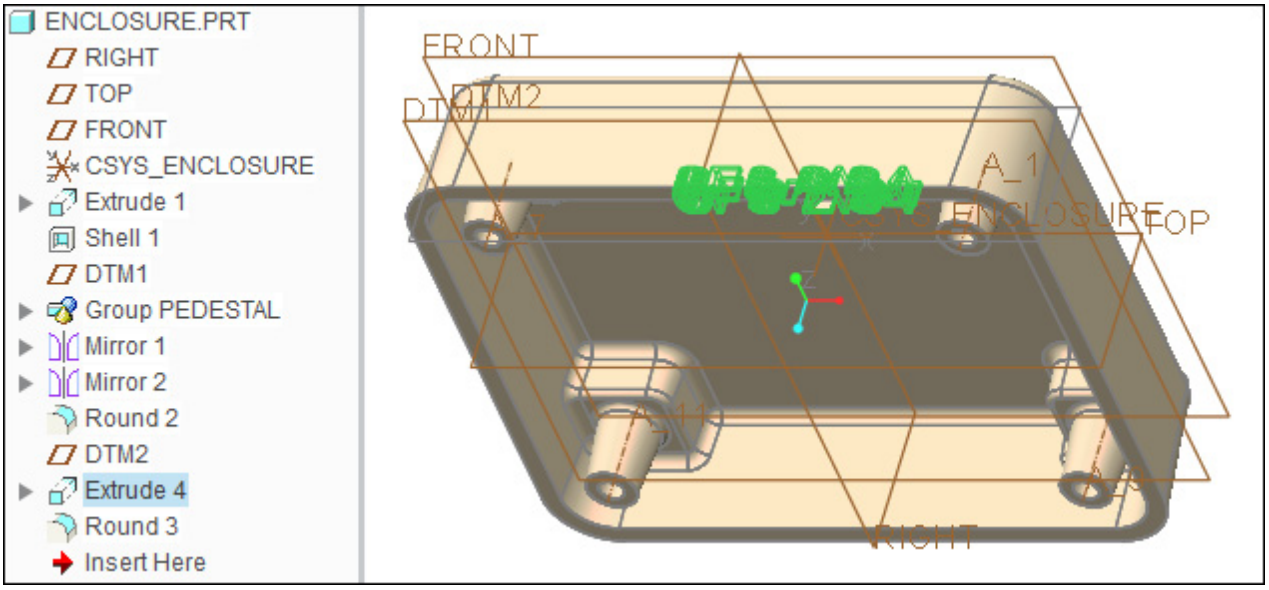



**Figure 18.24(c)** Completed Enclosure

Click: **File > Prepare > ModelCHECK Geometry Check** [Fig. 18.25(a)] **> OK >** [Figs. 18.25(b-c)] **> Ctrl+S > Enter >** close the Browser

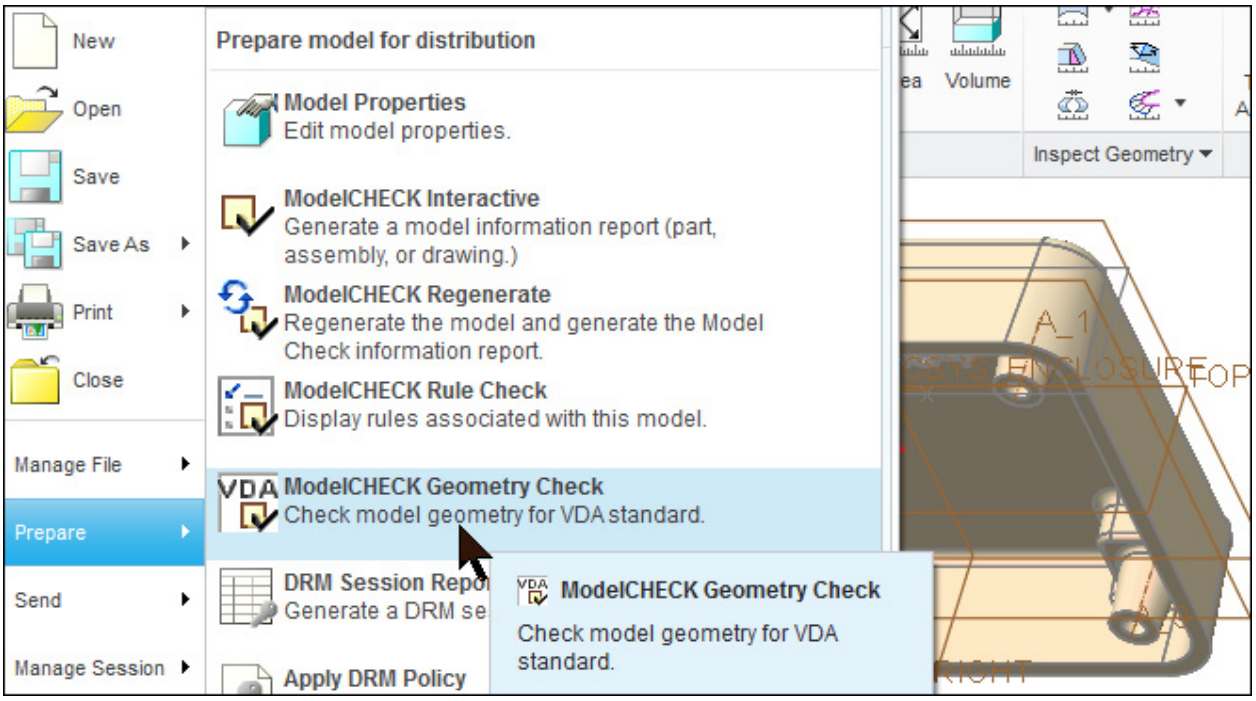



**Figure 18.25(a)** ModelCHECK Geometry Check

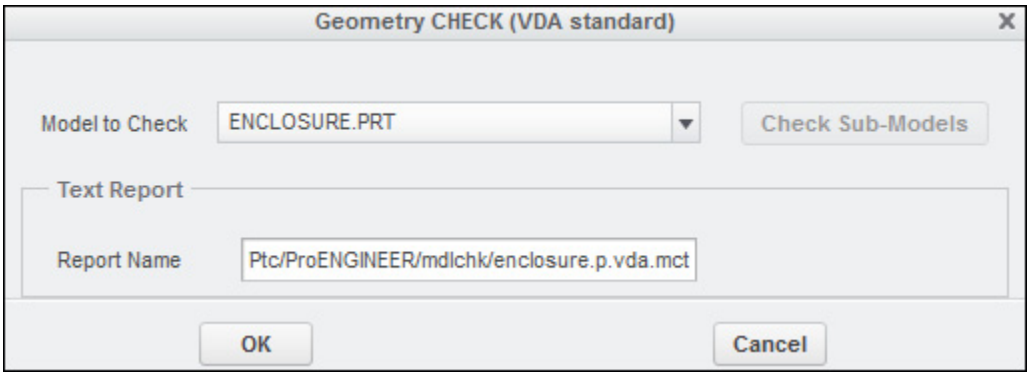



**Figure 18.25(b)** Geometry Check


**Figure 18.25(c)** ModelCHECK

Click: **File > Options > Model Display > Isometric > OK > No >**  **Shading > View** tab >  off >  off >  close **> Render** tab **> Render Setup >** Renderer  **> PhotoRender >** change settings as shown [Fig. 18.26(a)] **> Close > Ctrl+D**

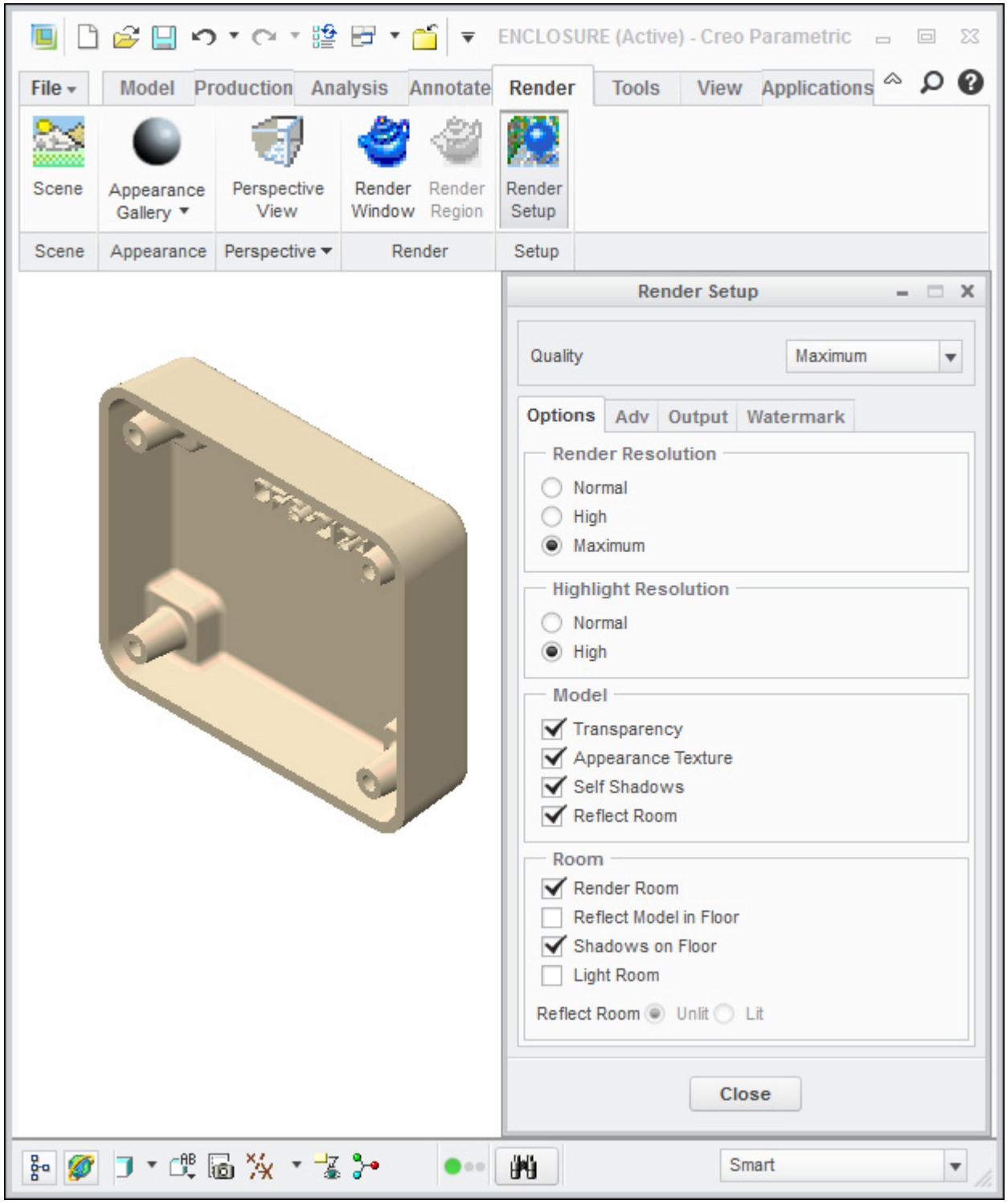



**Figure 18.26(a)** Render Setup

Click: **Scene > OK** (if needed) **> Lights** tab > **Add new spotlight >** Enable Shadows **>** Show Light **>** Name **Color for lighting** [Fig. 18.26(b)] > adjust the slide bars in the Color Editor

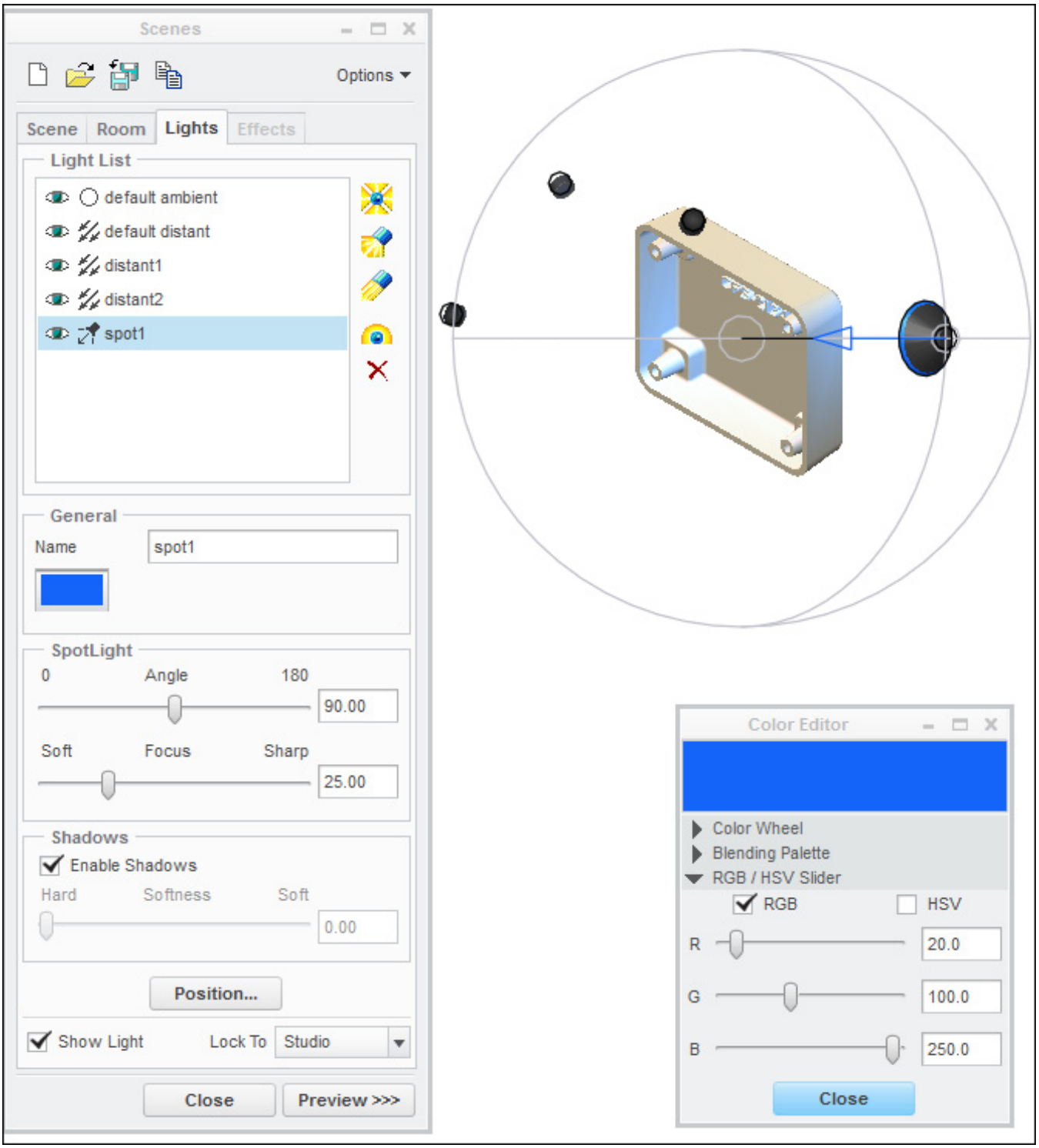



**Figure 18.26(b)** Adjust the Spot Light RGB Color Values

Click: **Close** Color Editor > select the new spot light and change its position and adjust the cone [Fig. 18.26(c)] > select the large arrow and change the focus [Fig. 18.26(d)]

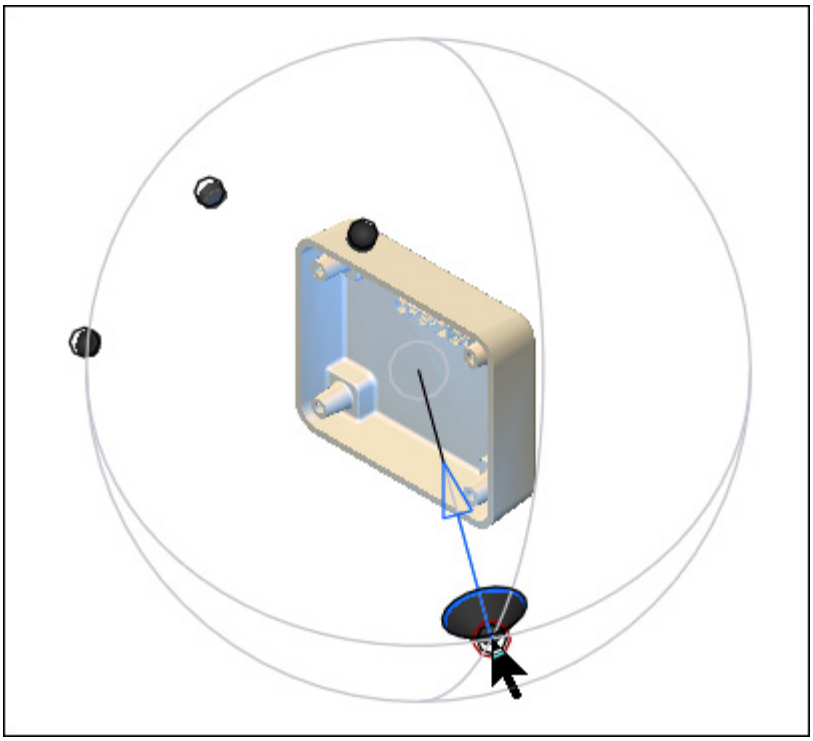


**Figure 18.26(c)** Change Spot Light Position

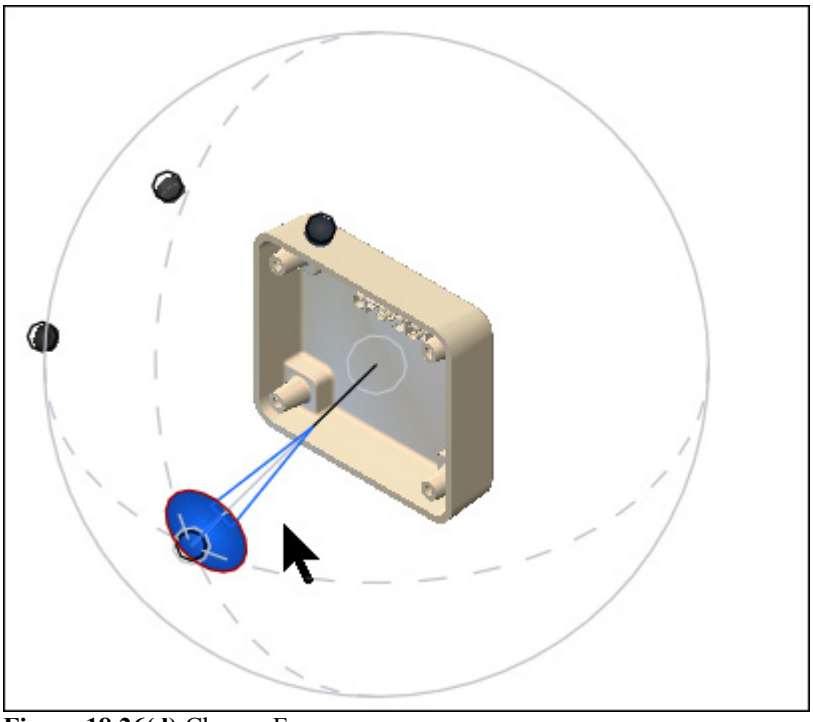


**Figure 18.26(d)** Change Focus

Click: **Add new distance light** > Enable Shadows > Show Light > Lock To Room > Name **Color for lighting** > adjust the slide bars in the Color Editor to the RGB values you desire > **Close** Color Editor > select the new distance light and change its position [Fig. 18.26(e)] > **Ctrl+D**

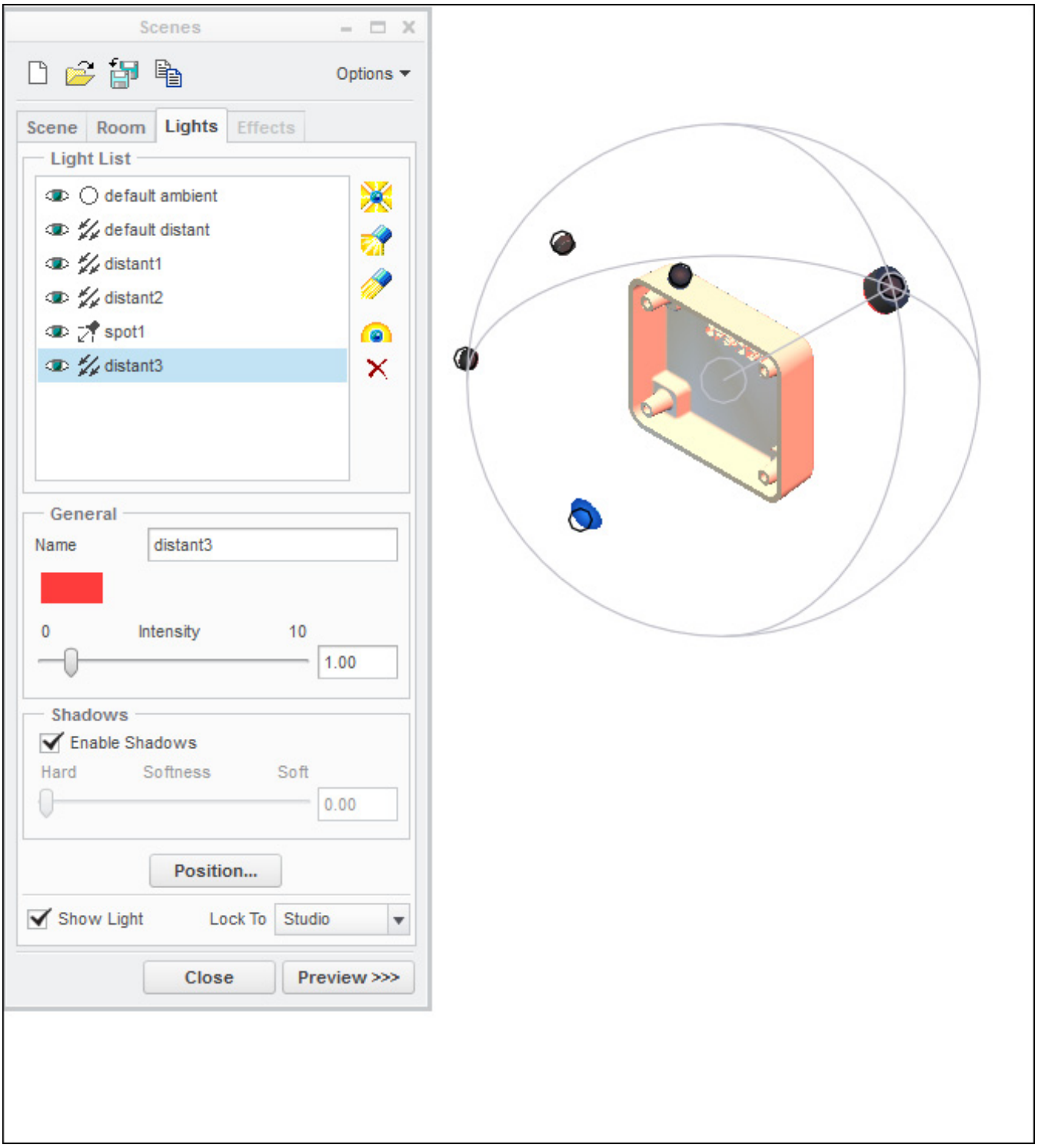



**Figure 18.26(e)** New Distance Light

Click: **Room** tab **>** **Wall1: enhanced-realism-wall1** [Fig. 18.26(f)] **>** Room Appearances **default wall appearance** [Fig. 18.26(g)] **> Close** Room Appearance Editor **>** repeat the process and change the walls and the floor settings **> Preview** [Figs. 18.26(h-i)]

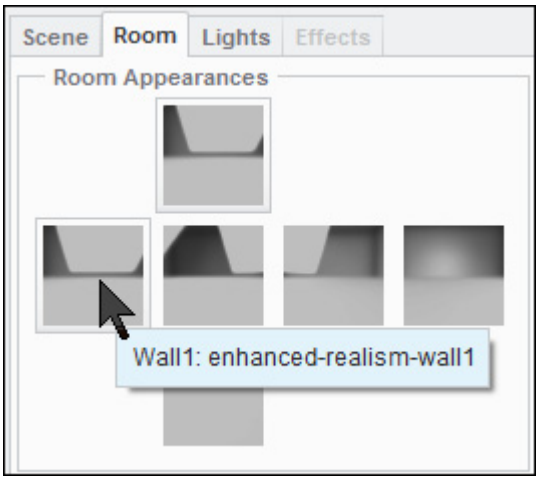



**Figure 18.26(f)** New Wall Selection

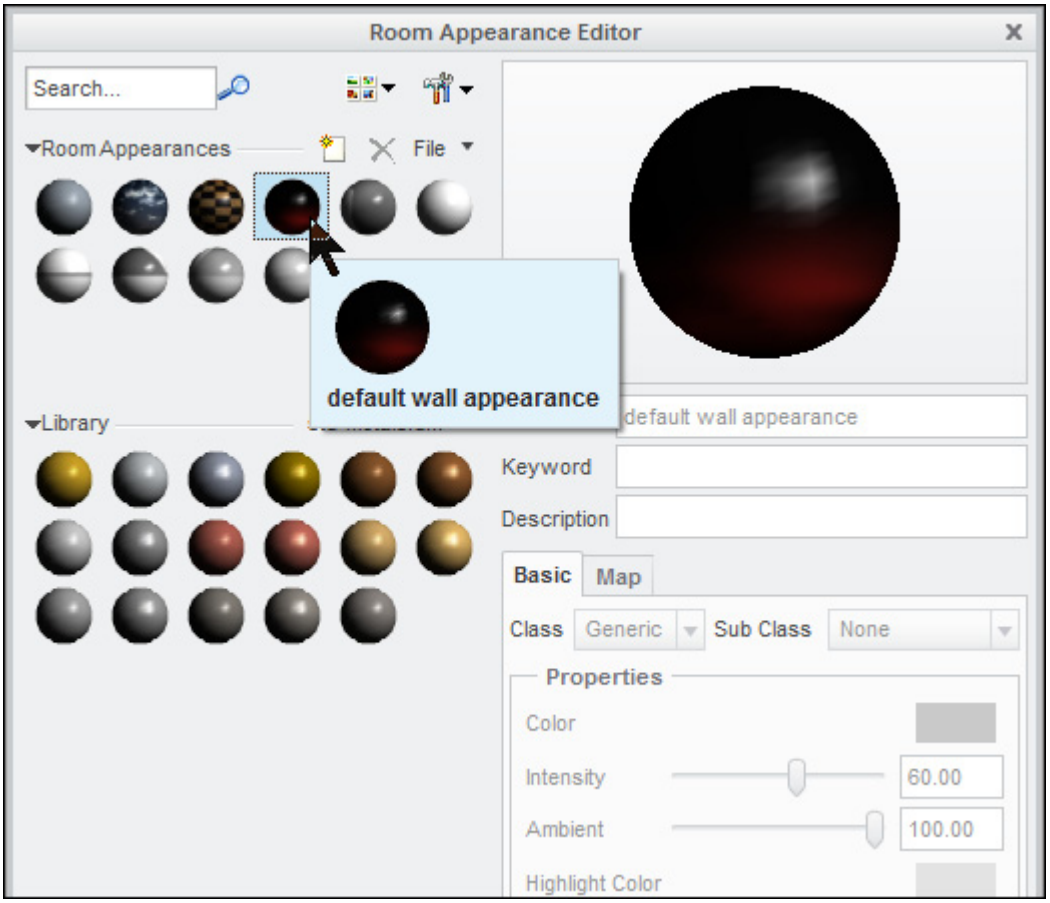



**Figure 18.26(g)** Room Appearance Editor

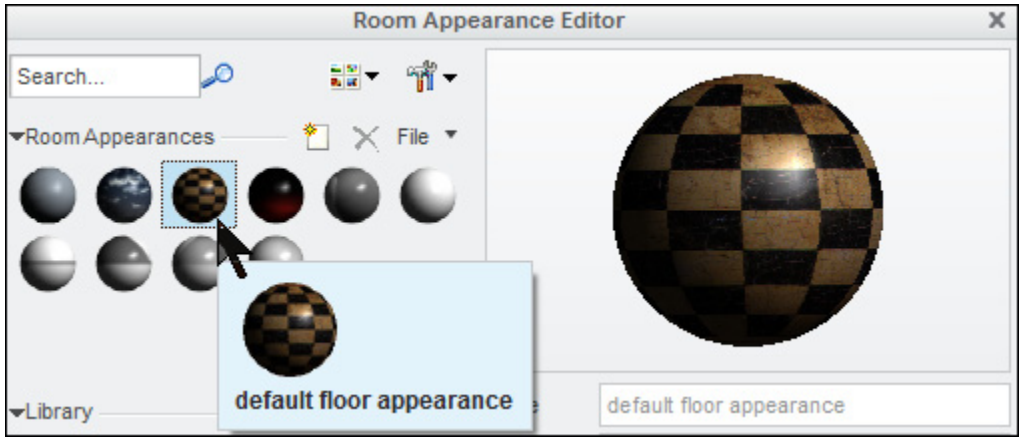



**Figure 18.26(h)** Room Appearance Editor

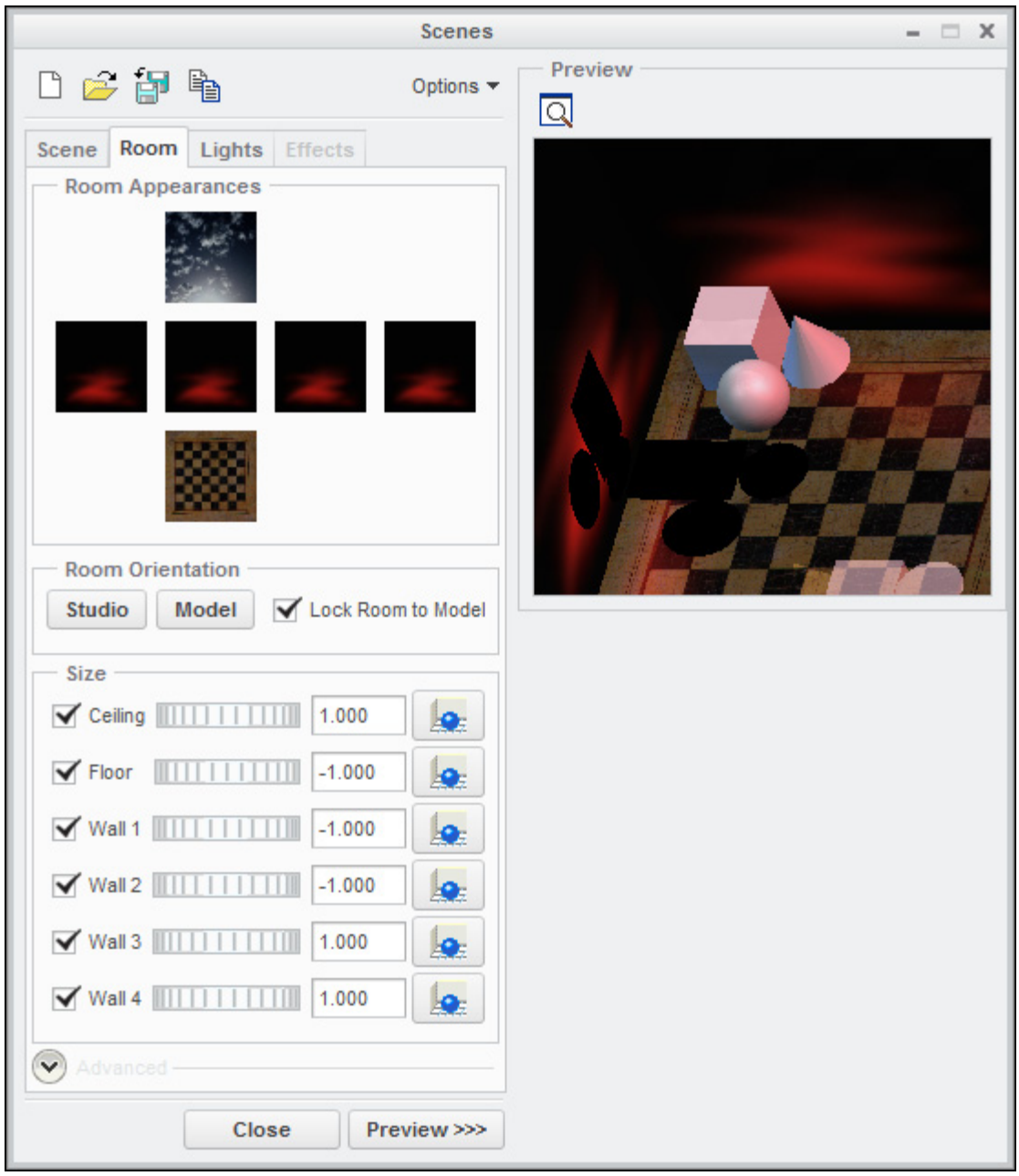



**Figure 18.26(i)** New Wall and Floor Appearance Selections with Preview

Click: **Close** [Fig. 18.26(j)] **> Render Window > Ctrl+S > Enter**

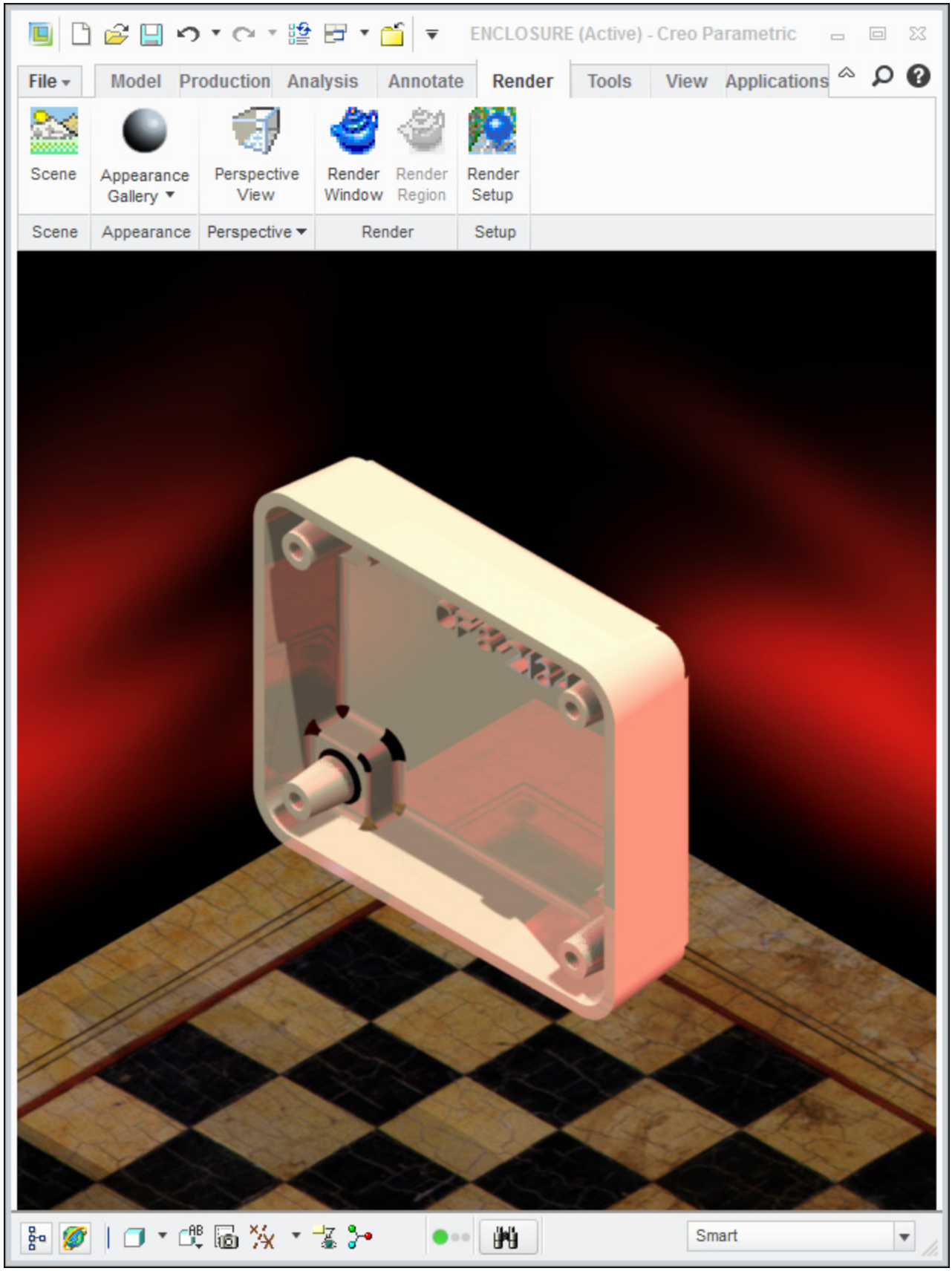



**Figure 18.26(j)** Render Window

Click: > Drawing > Name **enclosure** > **OK** > Template **d_drawing** > **OK** > **Layout** tab > Sheet Setup > Sheet 1 Format D Size > > **Browse** > **d.frm** > **Open** > **OK** > **Ctrl+S** > **OK** > off > (Select All) all off > **LMB** in the Graphics Window > **Ctrl+R** > select the front view > press **RMB** > **Insert Projection View** > select on the left of the front view > press **RMB** > **Lock View Movement** (uncheck) > move the views as required > select **left_4** view from the Drawing Tree > press **RMB** > **Properties** (Fig. 18.27) > **View Display** > **Follow Environment** > **Hidden** > **OK** > **LMB** to deselect > complete the detail [see Fig. 18.2(a)] > **Ctrl+S** > **Enter** > **File** > **Save As** > **Save a Copy** > 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**

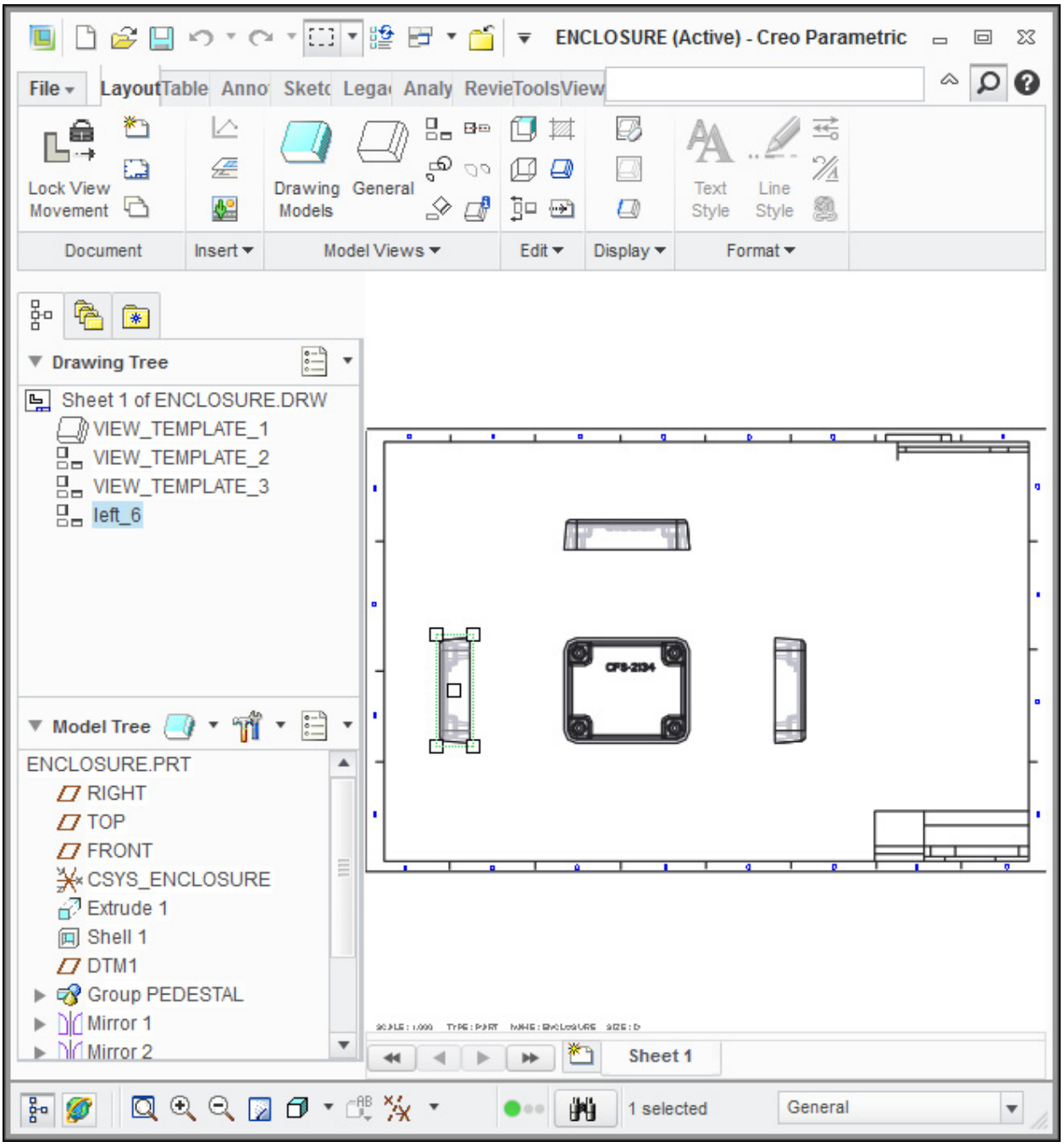



**Figure 18.27** Drawing Views

If your system has the Flexible Modeling application, complete the following commands.

Click: **Flexible Modeling** tab > spin the model > **Move/Rotate** > **Move With Dragger** > select the top face of the part [Fig. 18.28(a)] > select the vertical axis arrow [Fig. 18.28(b)] > move vertically > [Fig. 18.28(c)] > 

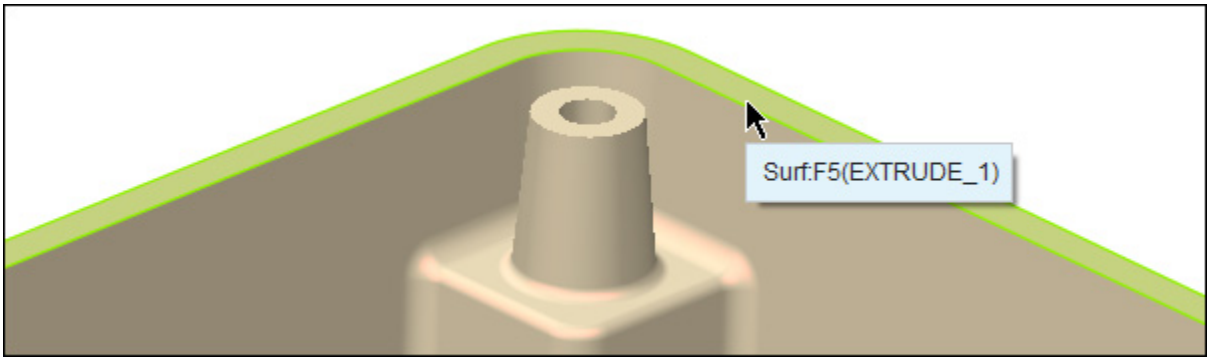



**Figure 18.28(a)** Select the Surface

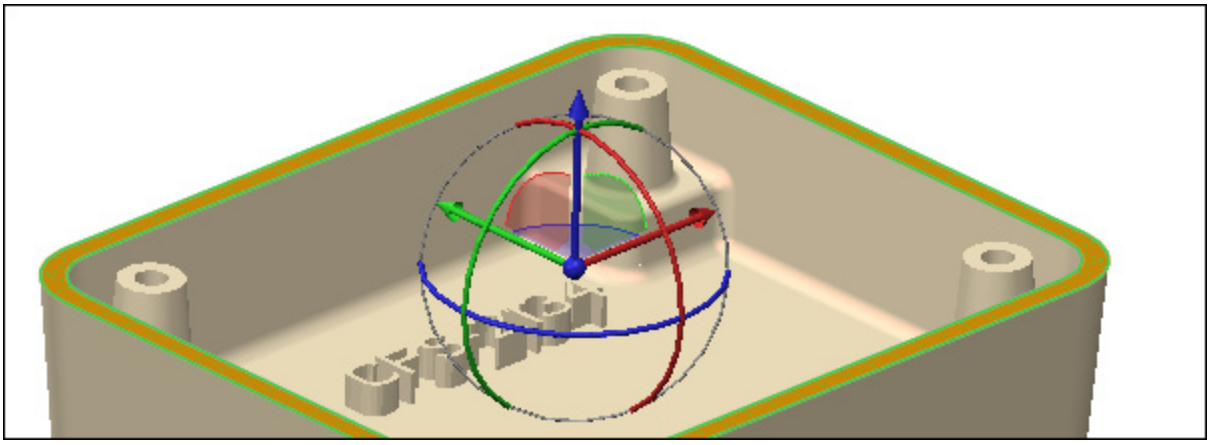


**Figure 18.28(b)** 3D Dragger Displays

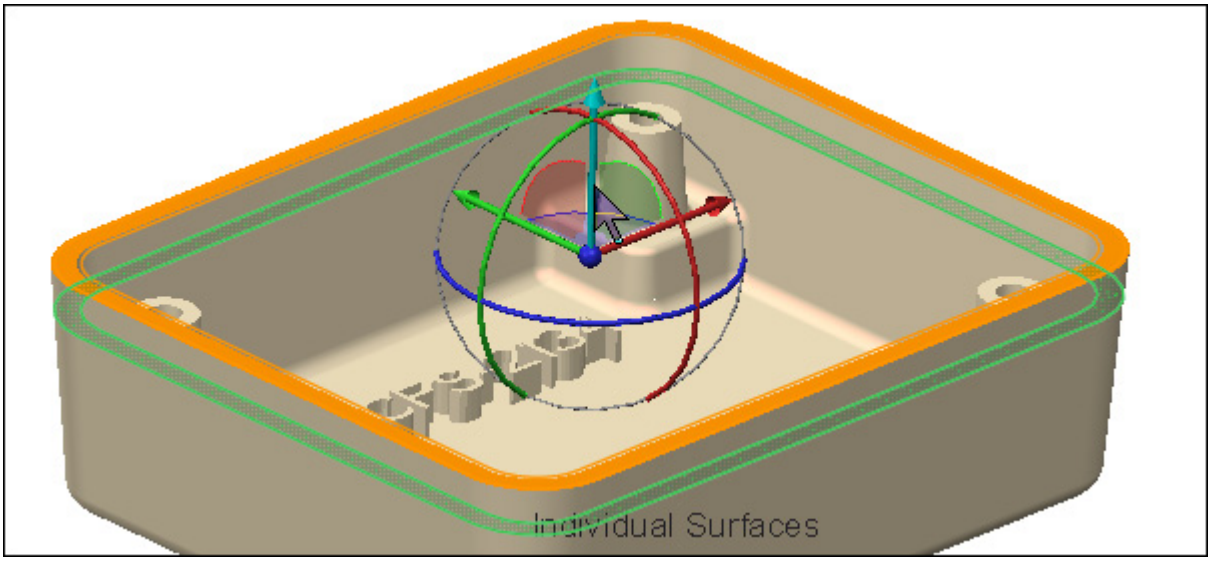



**Figure 18.28(c)** Drag the Vertical Axis

Click: **Move/Rotate** > **Move with Dragger** > select the top face of the lettering [Fig. 18.29(a)] > **3D Dragger** displays [Fig. 18.29(b)] > rotate the red arc > [Fig. 18.29(c)] > 

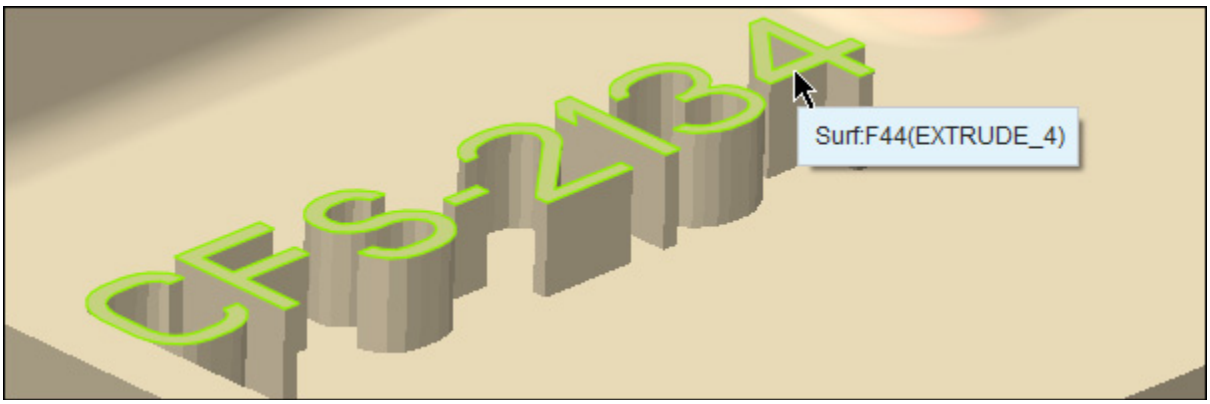



**Figure 18.29(a)** Select the Surface

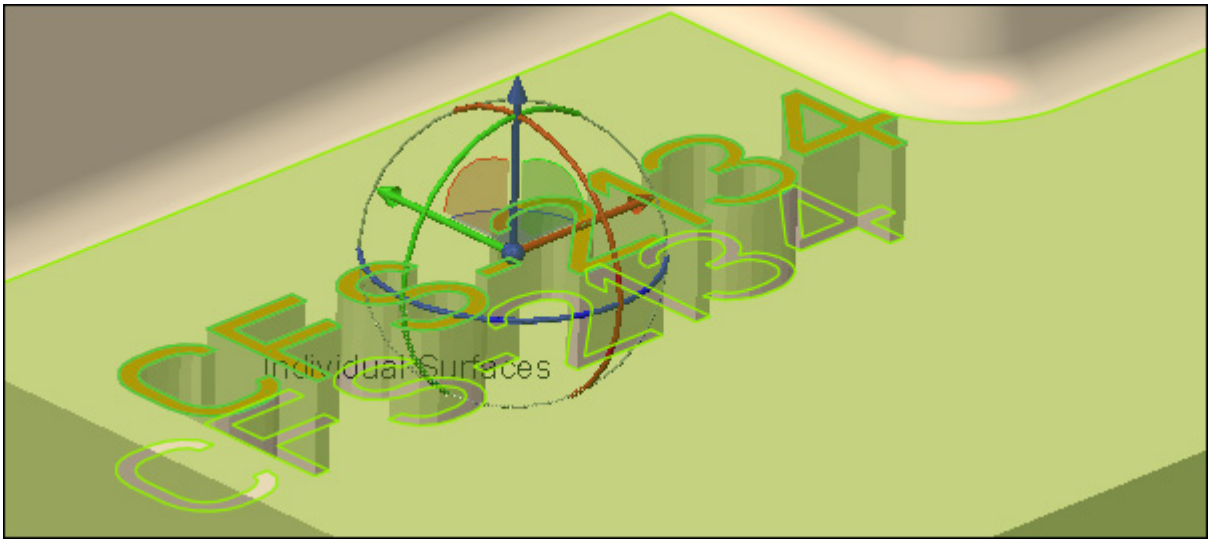



**Figure 18.29(b)** 3D Dragger Displays

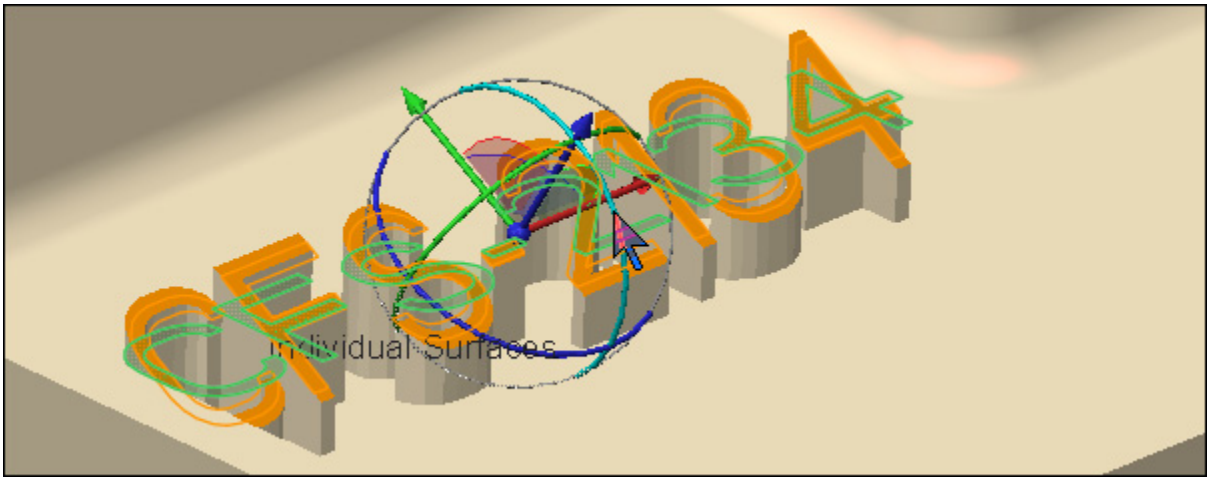



**Figure 18.29(c)** Drag the Vertical Axis

Click:  open [Fig. 18.30(a)] > **Ctrl+D**

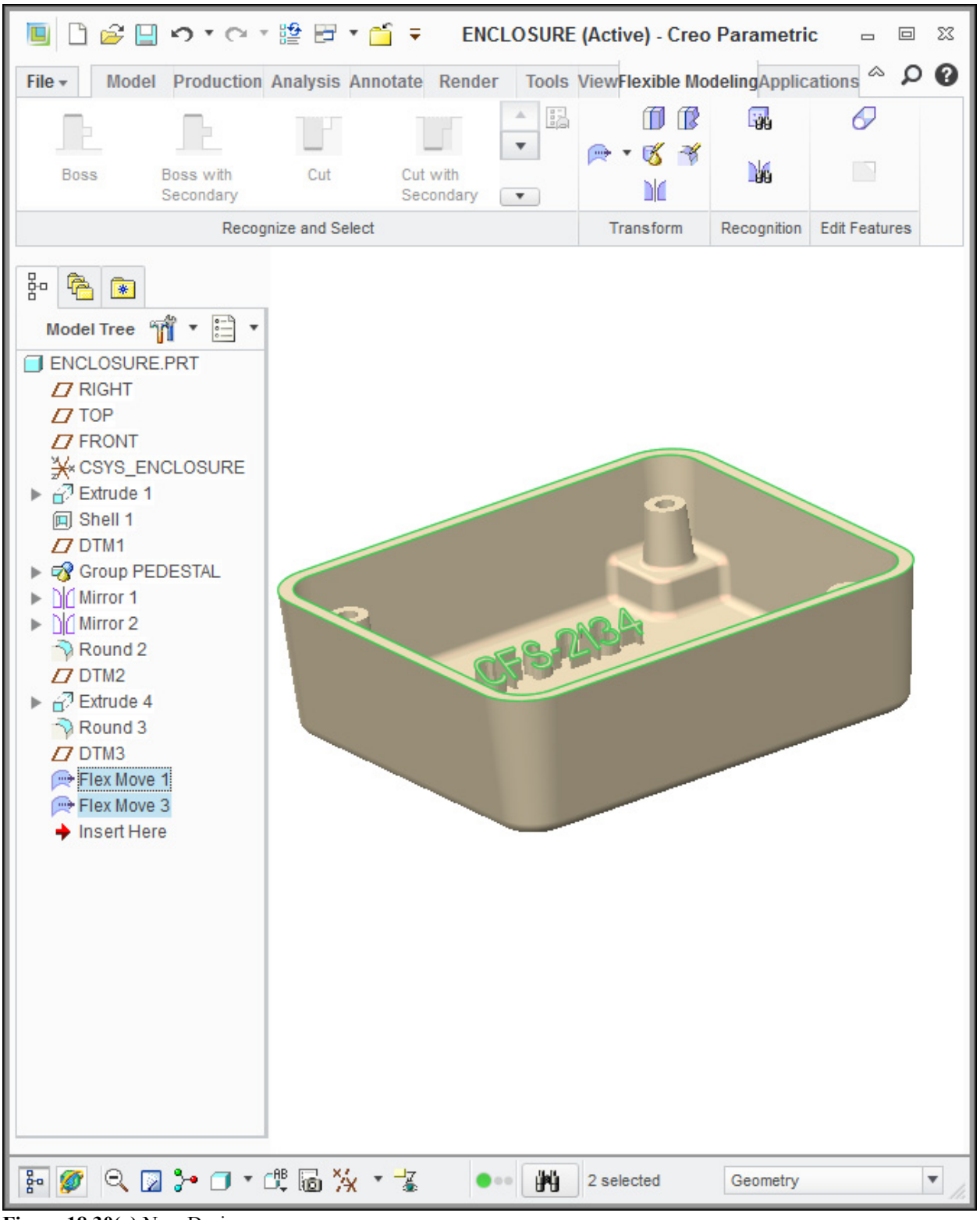



**Figure 18.30(a)** New Design

Click:  **Shading With Reflections** [Fig. 18.30(b)] **> File > Save As > Save a Copy > enclosure_one_off > OK > File > Close > File > Exit > Yes**

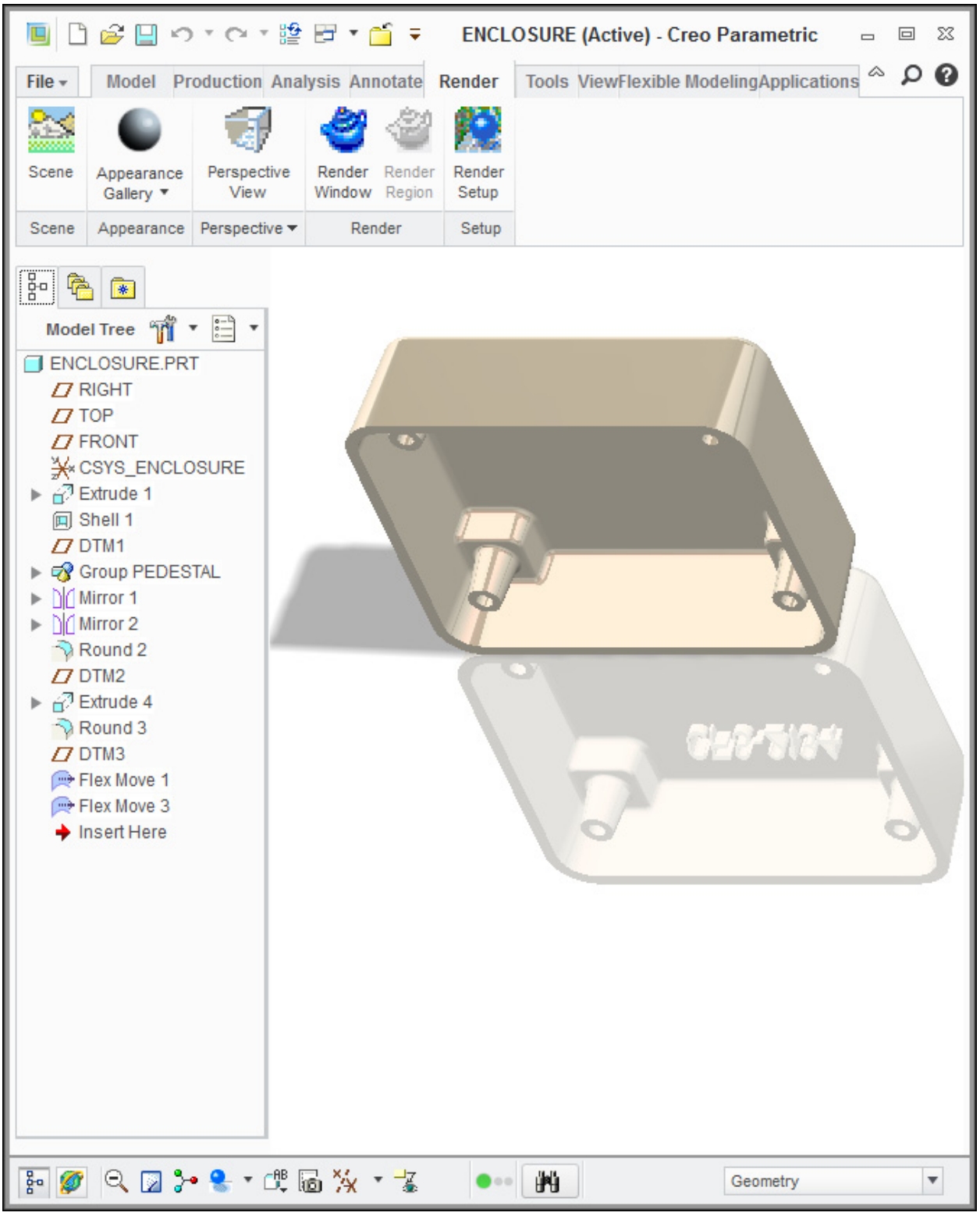



**Figure 18.30(b)** New Design *(the quality of your graphics card and graphics settings may prevent this display)*

For more part projects, see ***www.cad-resources.com*** **>** *click on the image of your book cover.*