On the Features toolbar, click the Revolved Protrusion command.
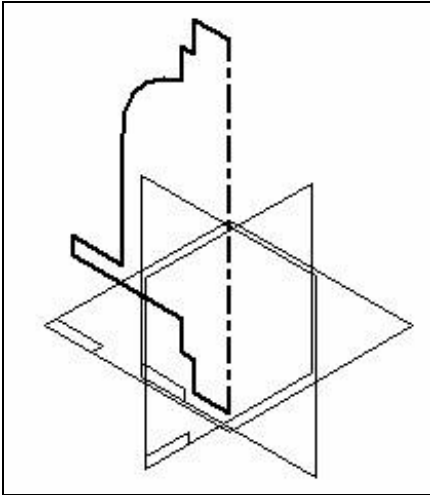

On the ribbon bar, click the Select from Sketch button.

In the part window, select the Sketch, and on the ribbon bar click the Accept button.

For the axis of revolution, select the vertical line shown dashed in the following illustration.
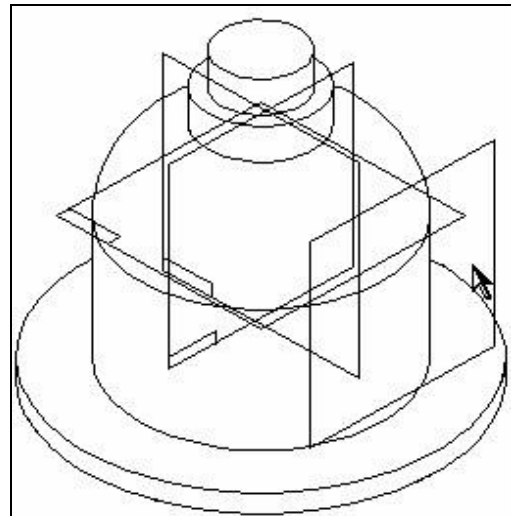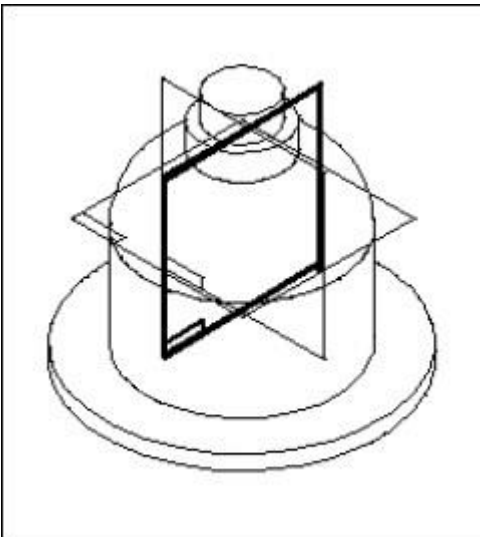
On the ribbon bar, click Revolve 360 Degrees.

On the Features toolbar, click the Protrusion command.

On the ribbon bar, click the Parallel Plane option.

Select the reference plane shown.

Note:
Throughout this activity, hidden lines and reference planes have been removed for clarity.

On the Protrusion ribbon bar, type 82.5 in the Distance box and press ENTER.

Move the cursor to the bottom right of the window, and click to define the location of the new reference plane.

On the Main toolbar, click the Fit command.

On the Draw toolbar, use the Line command to draw the profile shown. Use the Arc option, press A on the keyboard, within the Line command to draw the arc. Draw the profile with the same dimensional values and relationships shown in the following illustrations. Notice the vertical relationship between the midpoint of the vertical reference plane and the center of the profile arc.

Note:
Within the line command, you can either press A on the keyboard or click the arc option on the ribbon bar to enter arc mode. Once you place the arc, the command reverts back to line mode. When in arc mode, notice the intent zones available for arc placement.
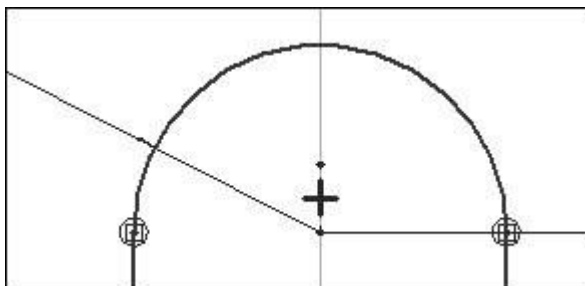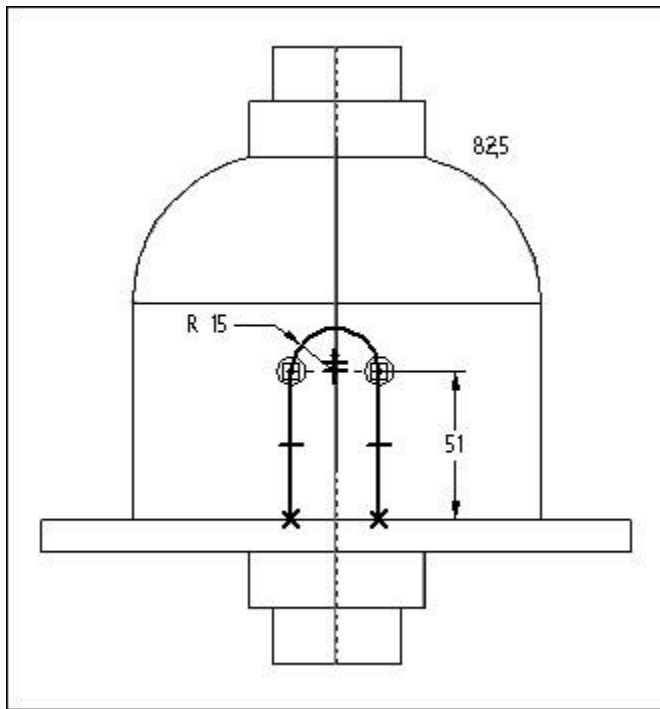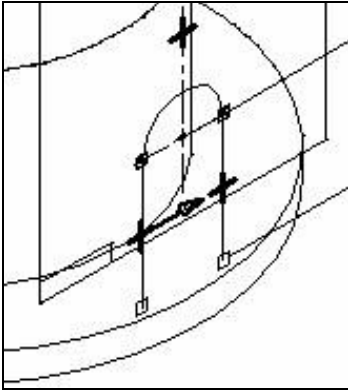

On the ribbon bar, click Finish.

Move the cursor so that the arrow points as shown and click. This will add material to the inside of the profile.
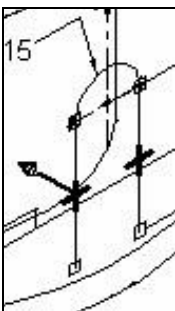
Note:
Notice the side step on the ribbon bar. When you use an open profile, you must specify the side of the profile to add material to.
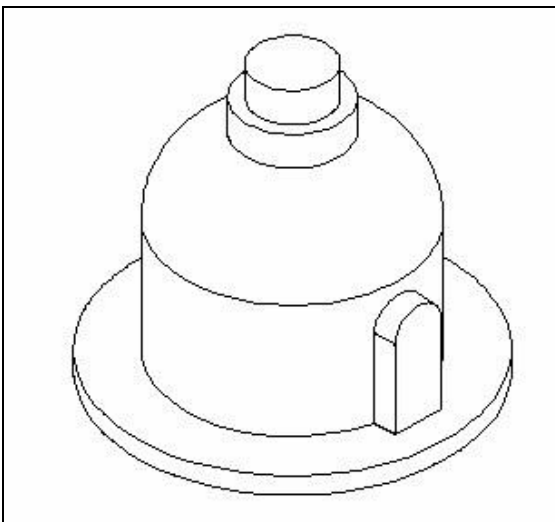
On the ribbon bar, click Through Next.

Move the cursor so that the arrow points as shown and click.
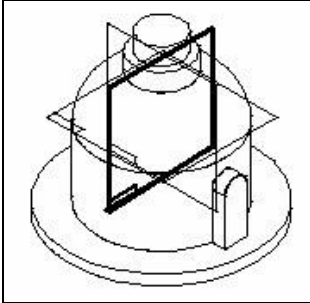
Click Finish to complete the protrusion.

5. In the next few steps, you will add a cutout to the part. You will draw a profile and then mirror the profile to remove material from both sides of the part.
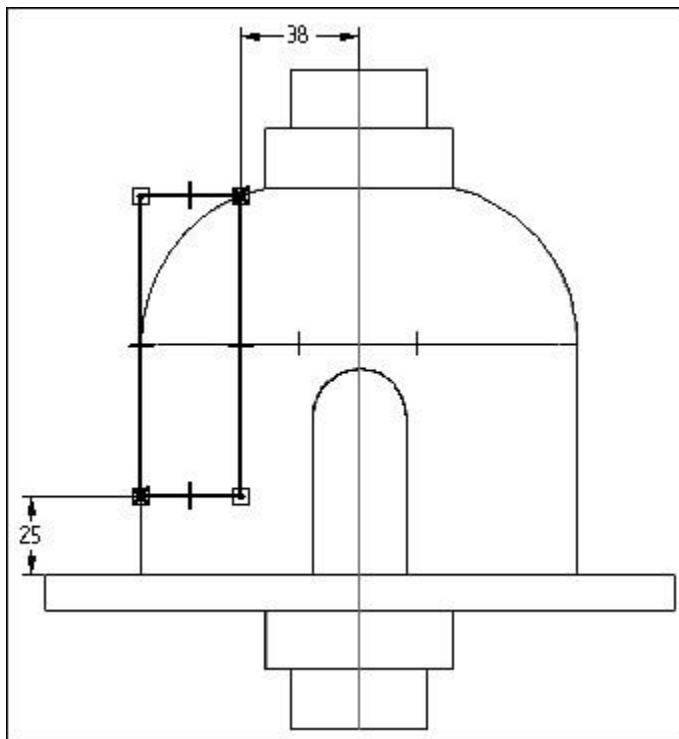
On the Features toolbar, click the Cutout command.
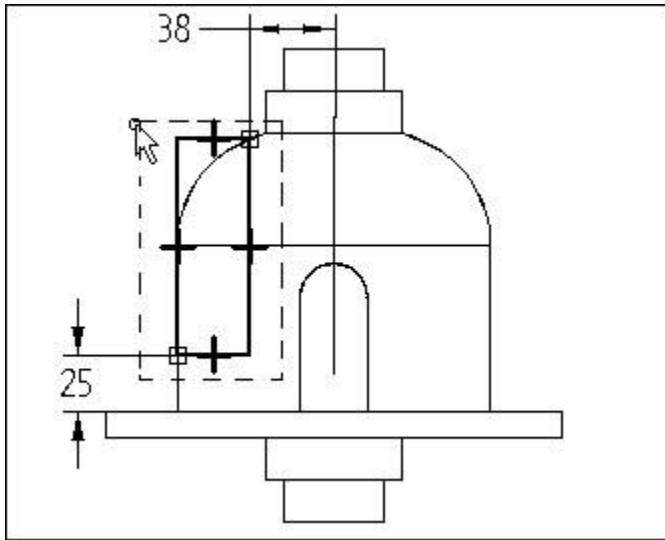

Select the reference plane shown.

Draw the profile shown. Notice the connect relationships between the top right and bottom left corners of the rectangle and the outer line of the revolved protrusion.

On the Draw toolbar, click the Select Tool. Select the profile by clicking to the top and left of the profile, holding, and dragging to the bottom and right of the profile.
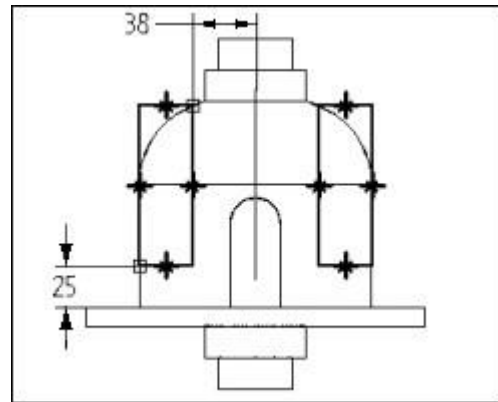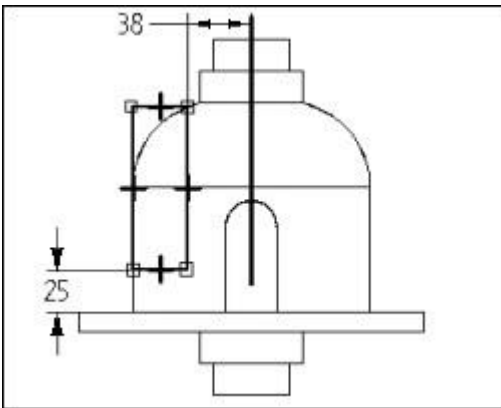
On the Draw toolbar, click the Mirror command. This command is on the same fly out as the Move command.

On the ribbon bar, make sure the Copy option is set. Select the vertical reference plane as the line to mirror about.
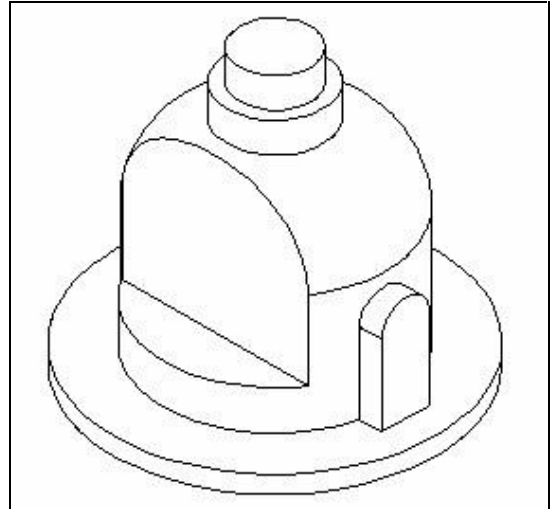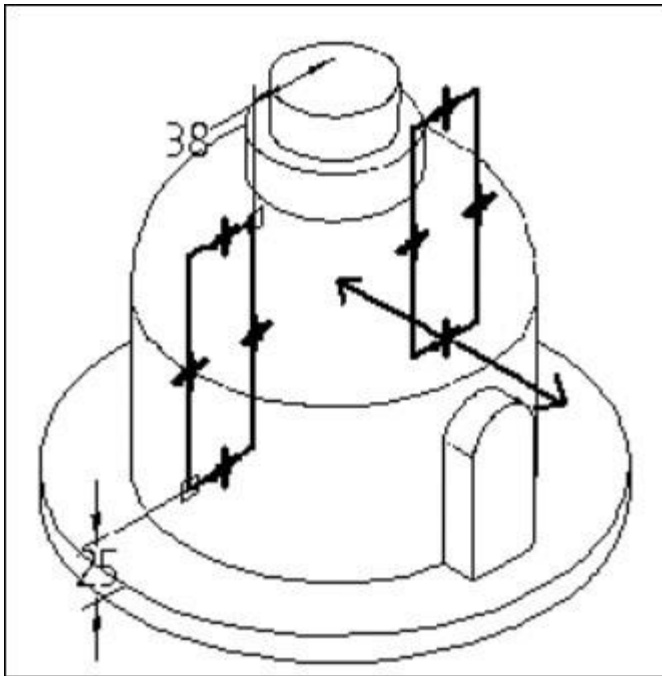

Click Finish to complete the profile.

On the ribbon bar, click the Through All option.

Position the cursor so that arrows point from both sides of the profile as shown and click. This adds material symmetrically to both sides of the profile.

Click Finish to complete the cutout.

6. In the next few steps, you will edit the cutout you placed in the previous step by adding a fillet to the profile.

On the Features toolbar, click the Select Tool command. Highlight the cutout feature by moving the mouse over it and click to select it.
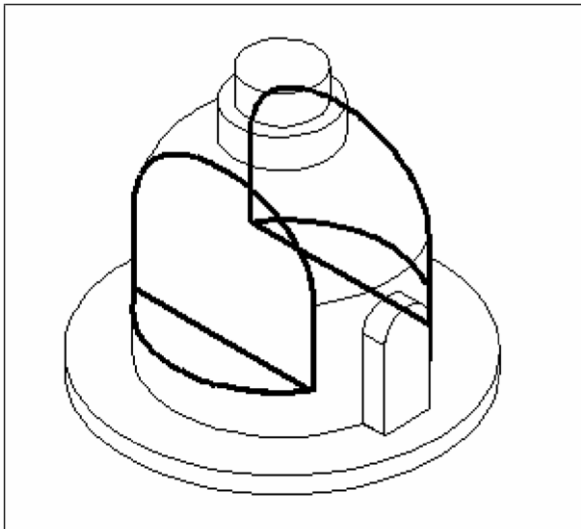
❑   On the ribbon bar, click Edit Profile.

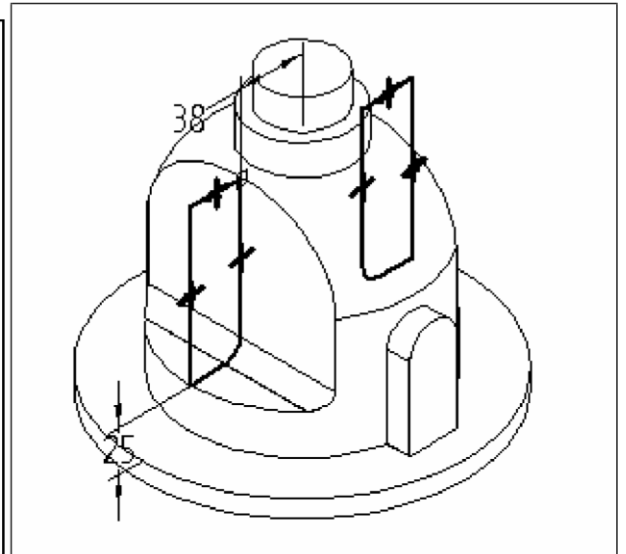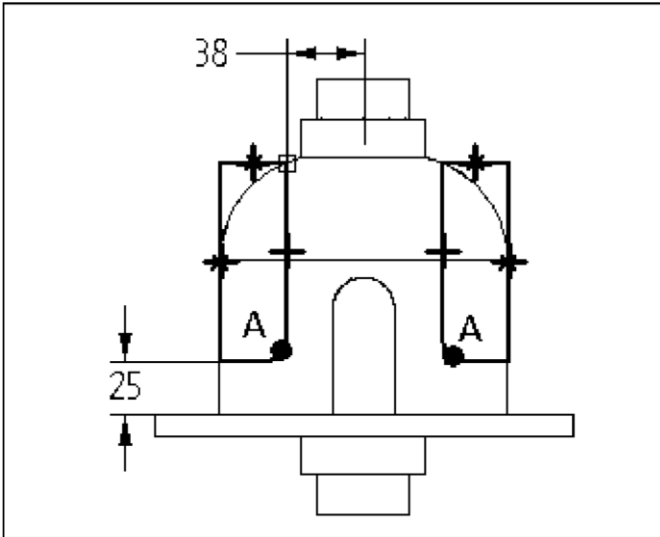On the Main toolbar, click the Fit command.

On the Draw toolbar, click the Fillet command.

On the ribbon bar, type 10 in the Radius box and press ENTER.
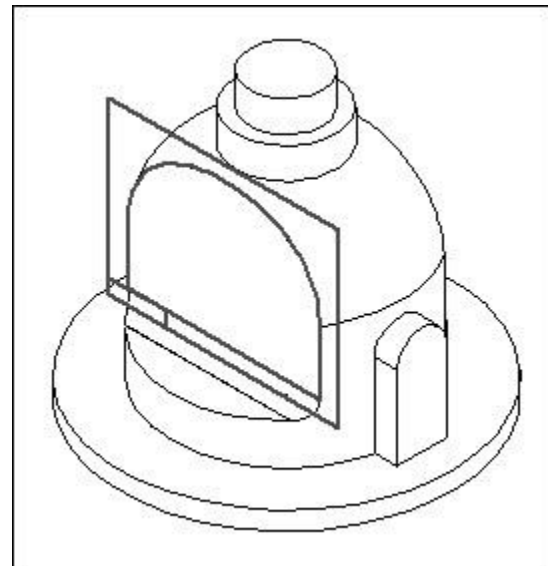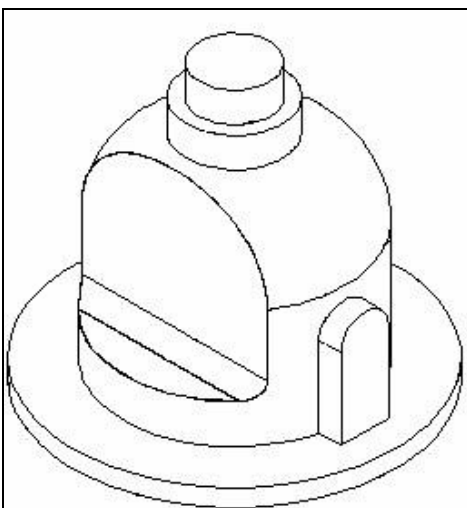

Place fillets on the bottom inside corners labeled A of the two profiles as shown.

❑ Click Finish to complete the profile.

Click Finish to complete the edit.

On the Features toolbar, click the Cutout command.

Select the face of the part as shown.

Draw the following profile. Use the Distance Between command to place the three dimensions and edit their values to those shown. Be sure the bottom horizontal line is collinear with the horizontal model edge and the top right and left corners of the rectangle are not connected to the model.
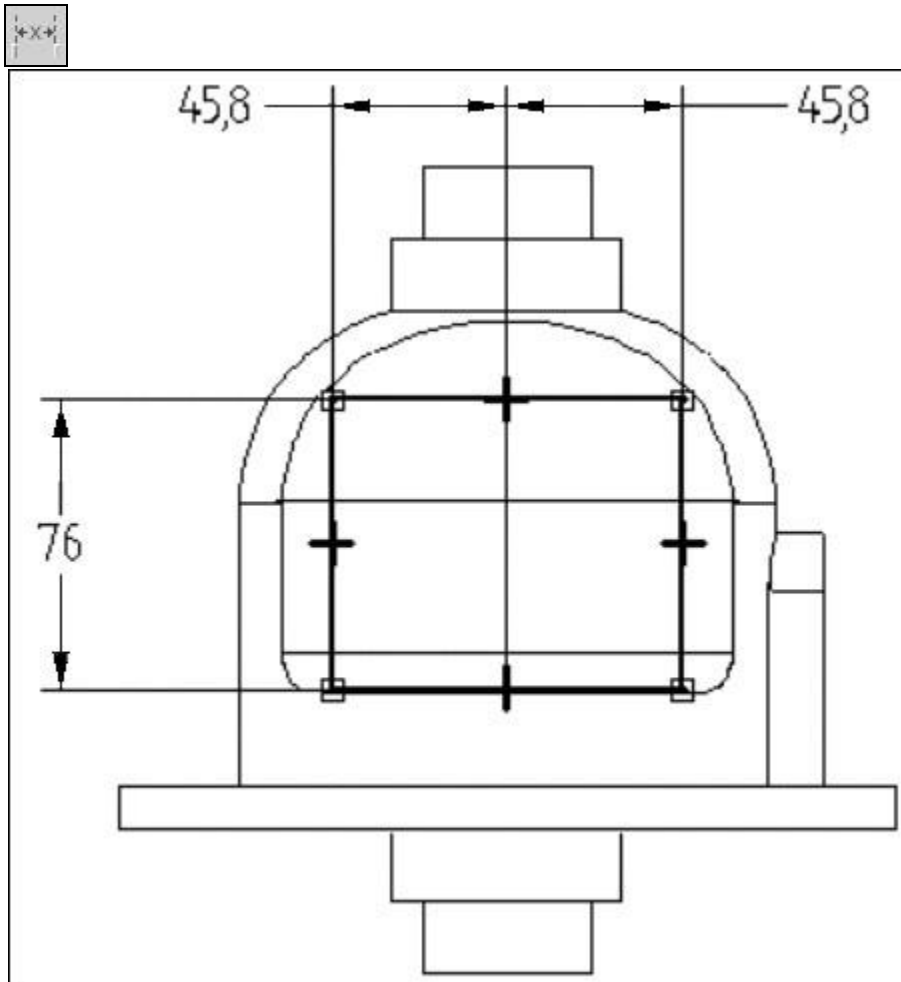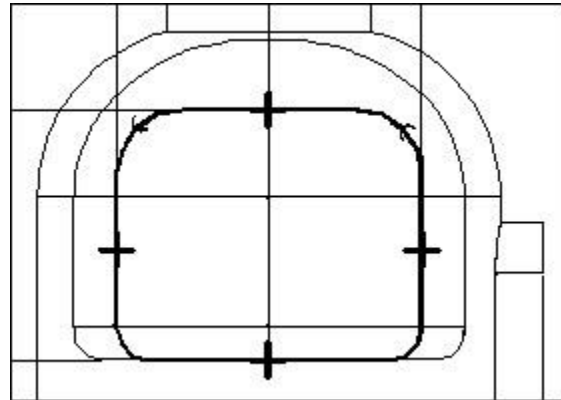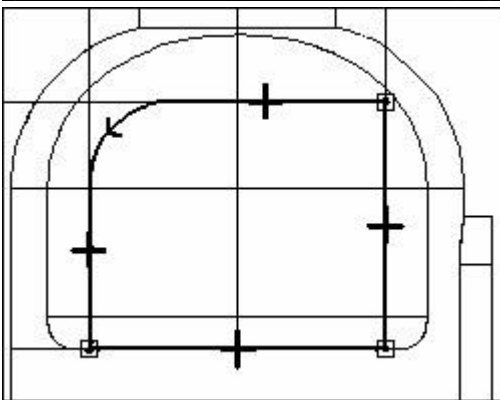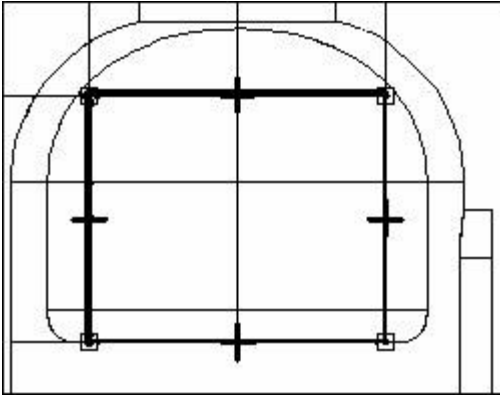

On the Draw toolbar, click the Fillet command.

On the Fillet ribbon bar in the Radius box, type 19 and press ENTER.

Move the mouse to the intersection of the top and left lines of the profile. When both lines highlight, click.

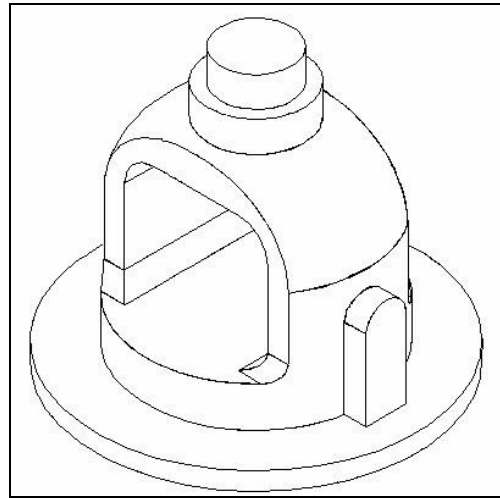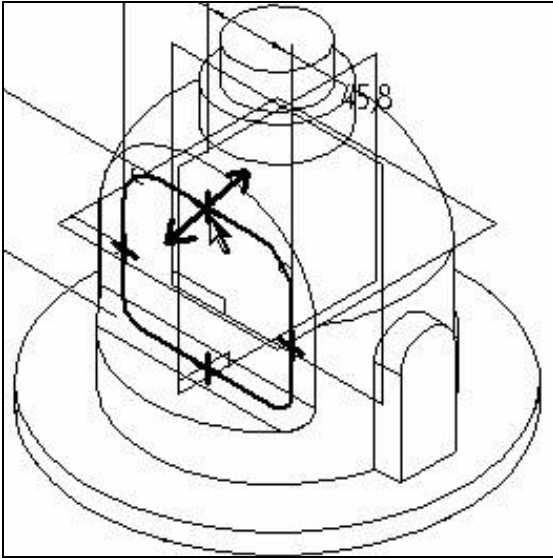On the top right corner of the profile, place another 19 mm fillet.

On the ribbon bar, type 10 in the Radius box, and place fillets on the bottom corners of the profile as shown.

Click Finish to complete the profile.

On the ribbon bar, click the Through All button.

Position the cursor so that arrows point from both sides of the profile as shown and click.
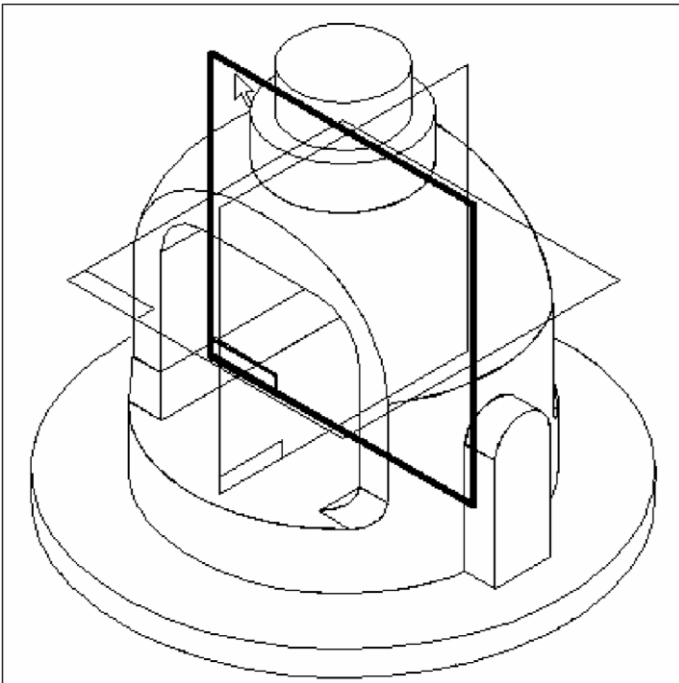
Click Finish to complete the feature.

9. In the next few steps you will add a revolved cutout to the part. To create this cutout, you will include and offset an existing part edge.

On the Features toolbar, click the Revolved Cutout command.
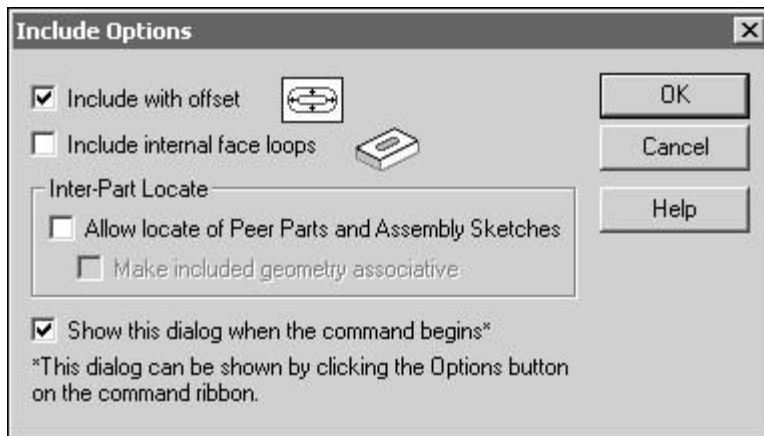

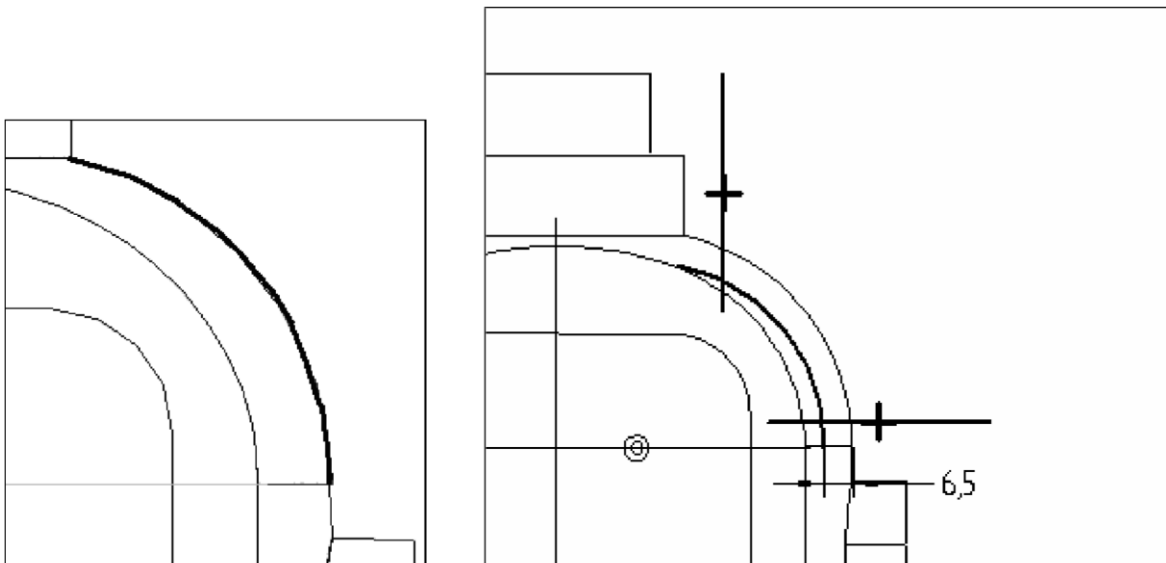
❏   Select the reference plane as shown.

On the Draw toolbar, click the Include command.

On the Include Options dialog box, set the Include with Offset option and click OK.

Select the arc shown, and on the ribbon bar click the Accept button.

❑   Type an offset distance of 6.5 mm and press ENTER.

Click inside the arc to accept the offset. Notice that the system places a dimension between the offset element and the arc from which it is offset.

Draw a horizontal line and a vertical line as shown.

On the Draw toolbar, select the Trim command.

Trim away the lines and arc to produce the following profile shape. If you make a mistake, click Undo on the Main toolbar and repeat the step.
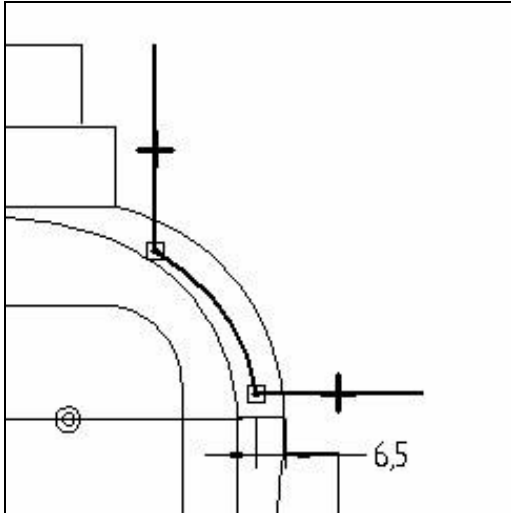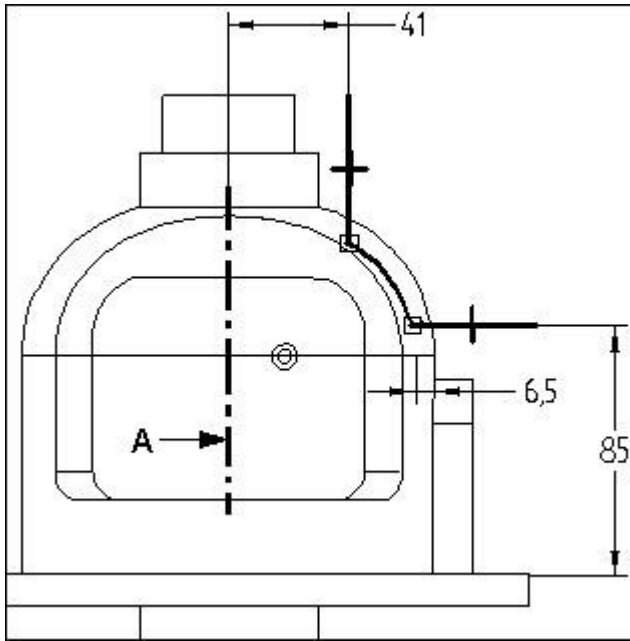
On the Draw toolbar, select the Distance Between command, and place dimensions as shown. Edit the values of the dimensions to the values shown below.
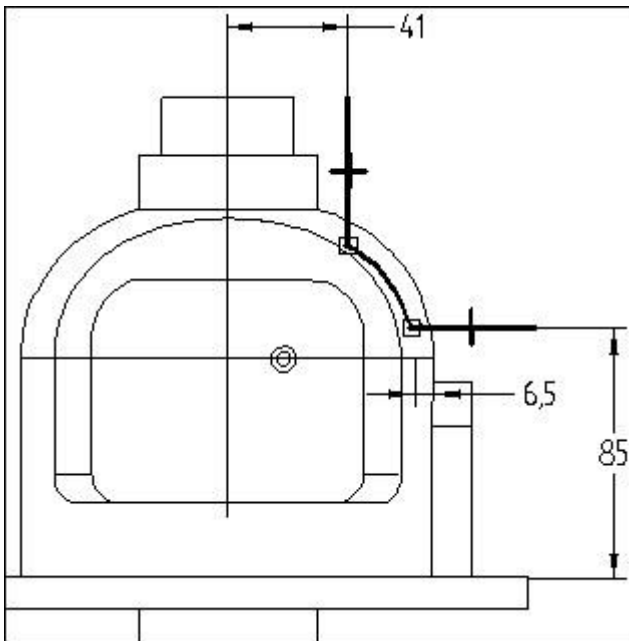
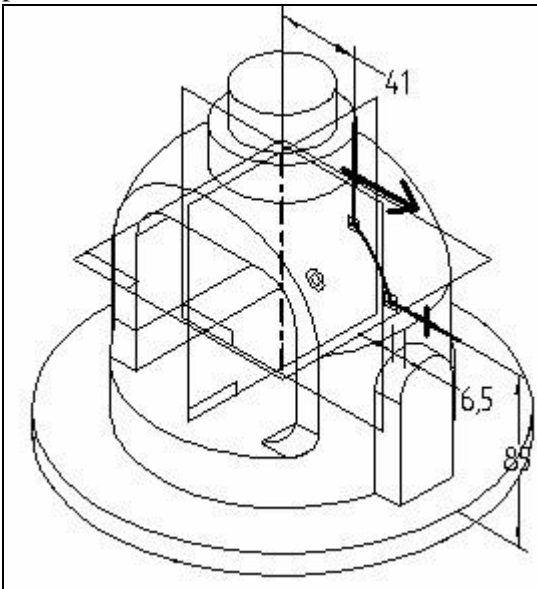On the Draw toolbar, click the Axis of Revolution command.

To define the axis of revolution, select the reference plane labeled A.

Click Finish to complete the profile.

To define the direction of material removal, position the cursor so that the arrow points to the outside of the part, and click.
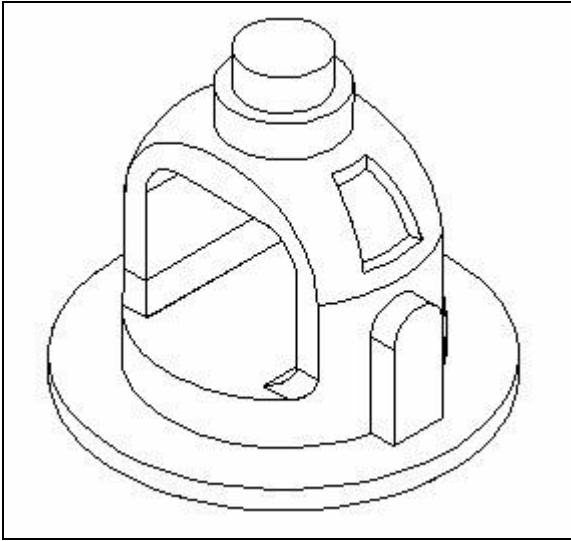

On the ribbon bar, click the Symmetric Revolve button.

On the ribbon bar, in the Angle box, type 30 degrees, and then press ENTER.

Click Finish to complete the revolved cutout.

10. Save and close the file. This completes this activity.

Lisaks

Luua loodud detaili joonised "Drafis"

Mõelda, mille näitamiseks kasutada vaateid ja kus on otstarbekad lõiked.