



Chapter 9

Editing Features

Learning Objectives

After completing this chapter, you will be able to:

- *Edit features.*
- *Edit sketches of the sketched features.*
- *Edit the sketch plane of the sketched features.*
- *Edit using the Move/Size Features option.*
- *Cut, copy, and paste features and sketches.*
- *Copy features using the drag and drop method.*
- *Delete features.*
- *Delete bodies.*
- *Suppress and unsuppress features.*
- *Move or copy bodies.*
- *Reorder features.*
- *Roll back the model.*
- *Rename features.*
- *Create folders.*
- *Use the What's Wrong functionality.*

EDITING FEATURES OF THE MODEL

Editing is one of the most important aspects of the product design cycle. Almost all designs require editing during or after their creation. As discussed earlier, SolidWorks is a feature-based parametric software. Therefore, the design created in SolidWorks is a combination of individual features integrated together to form a solid model. All these features can be edited individually. For example, Figure 9-1 shows a base plate with some holes.

Now, to replace the four inner holes with four counterbore holes, you need to perform one editing operation. Using the editing operation, you will change the drilled holes to the counterbore holes. For editing the holes, you need to select the hole feature and right-click to invoke the shortcut menu. Next, choose the **Edit Feature** option from the shortcut menu to invoke the **Hole Definition** dialog box. Finally, set the new parameters and end the feature creation. The drilled holes will be automatically replaced by counterbore holes. Figure 9-2 shows the base plate with drilled holes modified to the counterbore holes.

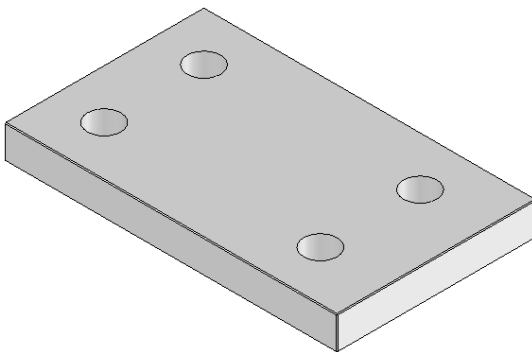


Figure 9-1 Base plate with drilled holes

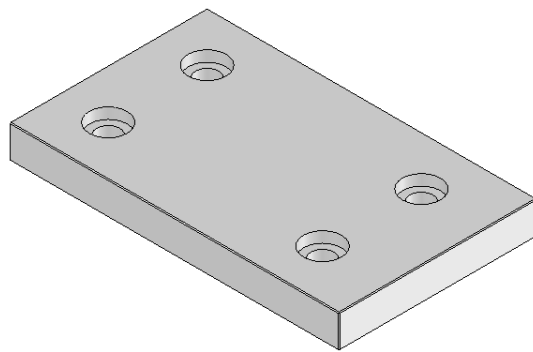
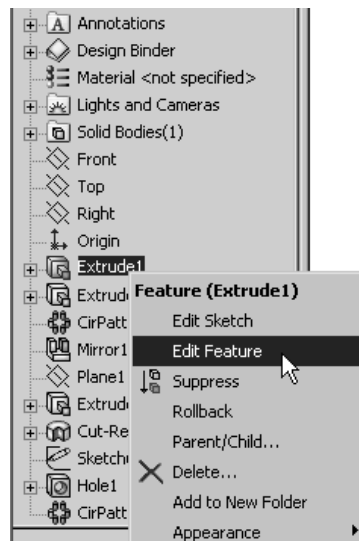


Figure 9-2 Modified base plate

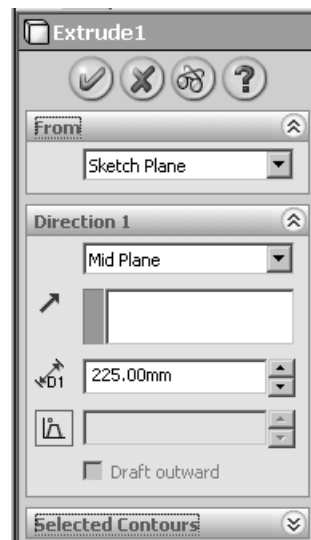
Similarly, you can also edit the reference geometry and the sketches of the sketched features. The feature created using the reference geometry also modifies automatically, when you modify the reference geometry. For example, if you create a feature on a plane at some angle and then edit the angle of the plane, the resulting feature is automatically modified. In SolidWorks, you can perform editing tasks using various methods, which are discussed next.

Editing Using the Edit Feature Option

Editing, using the **Edit Feature** option, is the most common method in SolidWorks. To edit any feature of the model using this option, select that feature from the **FeatureManager Design Tree** or from the drawing area. Next, right-click on it to invoke the shortcut menu and choose the **Edit Feature** option, as shown in Figure 9-3. Depending on the feature selected, the **PropertyManager** or dialog box will be invoked, and you can modify the parameters of that feature. The **PropertyManager** will also have the sequence number of the feature. The **Extrude PropertyManager** is displayed in Figure 9-4 with the sequence number 1. After editing



*Figure 9-3 Choosing the **Edit Feature** option from the shortcut menu*



*Figure 9-4 The **Extrude PropertyManager***

the parameters, choose the **OK** button to complete the feature creation. The feature will be modified automatically.

Editing Sketches of the Sketched Features

In SolidWorks, you can also edit the sketches of the sketched features using the **Edit Sketch**

option. Select the feature from the **FeatureManager Design Tree** or from the drawing area and right-click to invoke the shortcut menu. Choose the **Edit Sketch** option from it and you will enter the sketching environment. Using the standard sketching tools, edit the sketch of the sketched feature and exit the sketching environment. Choose CTRL+B to rebuild the model. You can also select the **Rebuild** button from the **Standard** toolbar to exit the sketching environment and rebuild the model.



Tip. You can also use the + sign available on the left of the sketched feature to expand the sketched feature in the **FeatureManager Design Tree**. The sketch icon will be displayed when you expand the sketched feature. Select the sketch icon and invoke the shortcut menu. Select the **Edit Sketch** option from it to enter the sketching environment to edit the sketch.

Changing the Sketch Plane of Sketches

You can also change the sketch plane of the sketches of the sketched features. To edit the sketch plane, expand the sketched feature by clicking on the + sign on its left in the **FeatureManager Design Tree**. Select the sketch icon in the **FeatureManager Design Tree**. Right-click to invoke the shortcut menu and choose the **Edit Sketch Plane** option from it. Figure 9-5 shows the **Edit Sketch Plane** option chosen from the shortcut menu.

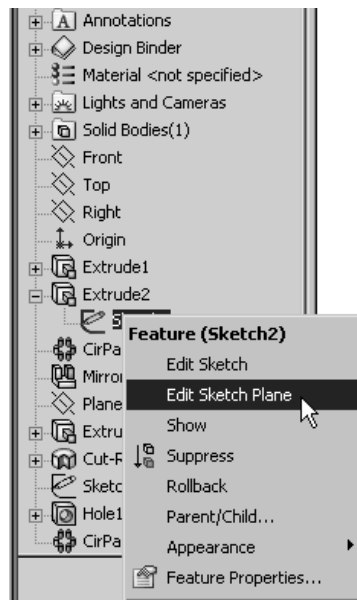


Figure 9-5 The Edit Sketch Plane option being chosen

The **Sketch Plane PropertyManager** will be displayed, as shown in Figure 9-6. The name of the current sketch plane is displayed in the **Sketch Plane/Face** selection box. Now, select any other plane or face as the sketching plane. Next, choose the **OK** button from the **Sketch Plane PropertyManager**.

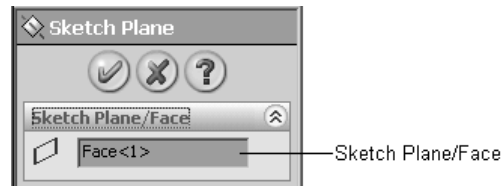


Figure 9-6 The Sketch Plane PropertyManager



Tip. If you select a sketch plane on which the relations and dimensions do not find any reference to be placed, the **What's Wrong** dialog box will be displayed. You will need to undo the last step using the **Undo** button from the **Standard** toolbar. After this, invoke the **Sketch Plane PropertyManager** again and select the appropriate plane. You will learn more about the **What's Wrong** dialog box later in this chapter.

Editing by Double-clicking on Entities and Features

You can also edit a feature, reference geometry, or a sketch by double-clicking the feature either from the **FeatureManager Design Tree** or from the drawing area. For example, if you double-click on the extrude feature, all its dimensions, and those of the sketch used to create it are displayed in the drawing area. Remember that the dimensions of the sketch will be displayed in black and the dimensions of the feature will be displayed in blue. Double-click the dimension that you need to modify; the **Modify** dialog box is invoked. Set the new value in the **Modify** dialog box and press the ENTER key on the keyboard or choose the **Save the current value and exit the dialog** button from the dialog box. You will notice that the value of the dimension is modified but the model is not modified relative to the modified value. Therefore, you need to rebuild the model using the **Rebuild** option. To rebuild the model, choose the **Rebuild** button from the **Standard** toolbar or choose CTRL+B on the keyboard.

Editing Using the Move/Size Features Tool

CommandManager: Features > Move/Size Features (Customize to Add)
Toolbar: Features > Move/Size Features (Customize to Add)



You can dynamically modify the feature and the sketch of the sketched feature without invoking the sketching environment by using this option. To edit the feature or sketch, choose the **Move/Size Features** button from the **Features CommandManager**.

You will notice that the **Move/Size Features** button is chosen in it. Select any face of the feature to modify. The selected face will be highlighted in green and if the selected feature is a sketched feature then its sketch will also be highlighted in green. You will be provided with the resize handle, rotate handle, and the move handle. Figure 9-7 shows the move handle, resize handle, and the rotate handle provided for the selected feature.

To resize the feature, move the cursor to the resize handle; the select cursor will be replaced by the resize cursor with its name displayed as the tooltip. Press and hold down the left mouse button at this location and drag the cursor to resize the feature. The feature will be resized with an increment of 10. Release the left mouse button when you have resized it. You will notice that

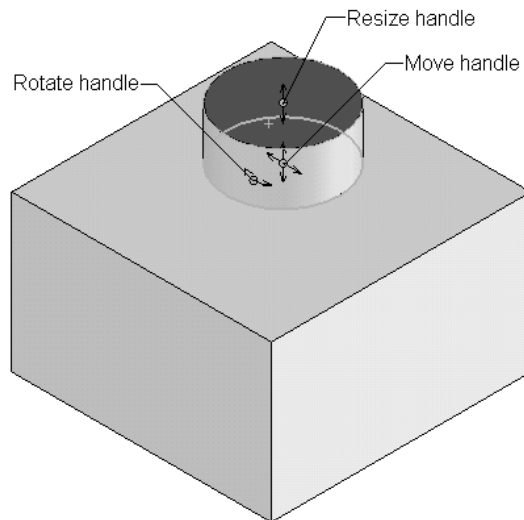


Figure 9-7 Feature selected with the **Move/Size Features** button selected

the feature will be dynamically resized. Figure 9-8 shows the cursor being dragged to resize the feature. Figure 9-9 shows the resulting modified feature.

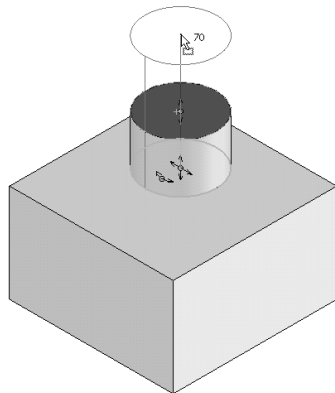


Figure 9-8 Dragging the resize handle to resize the feature

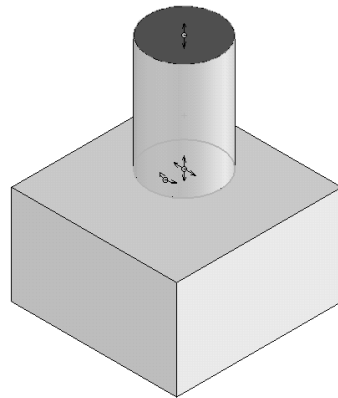


Figure 9-9 Resulting modified feature

The rotate handle is used to rotate the selected feature. To rotate the feature using the rotate handle, move the cursor to the rotate handle, press and hold down the left mouse button. Drag the cursor to rotate the feature. You can drag the feature clockwise or counterclockwise. The preview of the feature will be displayed in the drawing area. The feature will be rotated with an increment of 10-degrees. Release the left mouse button, after rotating the feature to a specified location.

**Note**

If you rotate a sketched feature whose sketch is fully or partially defined using relations and dimensions, the **Move Confirmation** dialog box will be displayed, as shown in Figure 9-10. This dialog box informs you that the external constraints in the feature are being moved, and prompts whether you want to delete those constraints or keep them by recalculating or make them dangling. The relations or dimensions that do not find the external reference after the placement are made dangling.

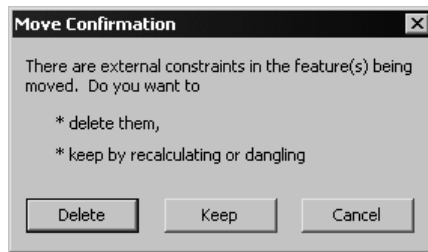


Figure 9-10 The **Move Confirmation** dialog box

After rotating the feature, if you invoke the **Move Confirmation** dialog box, you need to choose either the **Delete** or the **Keep** button, based on the geometric and dimensional conditions. Figure 9-11 shows the preview of the rotating feature. Figure 9-12 shows the resulting rotated feature.

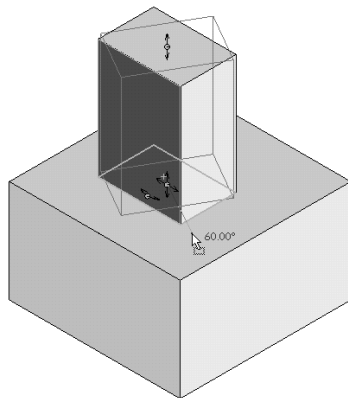


Figure 9-11 Preview of the feature being rotated

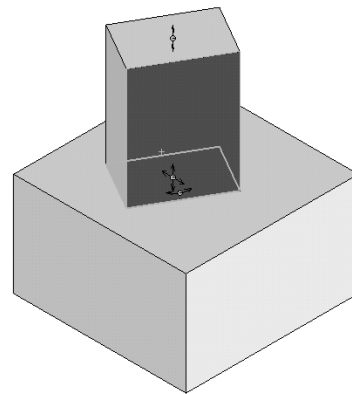


Figure 9-12 Resulting rotated feature

To move a feature using the move handle, select a face of the feature and move the cursor to the move handle. Press and hold down the left mouse button, drag the cursor to move the feature to the desired position and then release the left mouse button. If the dimensions or relations of the feature do not remain defined after moving, the **Move Confirmation** dialog box will be displayed. Choose the options available in this dialog box. Using these options, you can also change the placement plane or the sketch plane of the feature. Figure 9-13 shows the feature being moved to another face. Figure 9-14 shows the resulting moved feature.

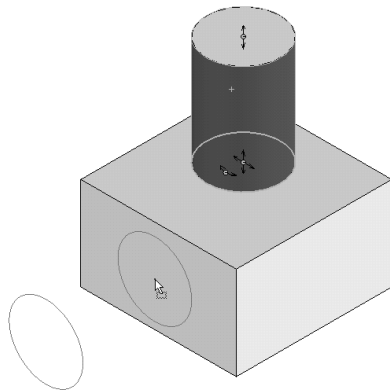


Figure 9-13 The feature being moved

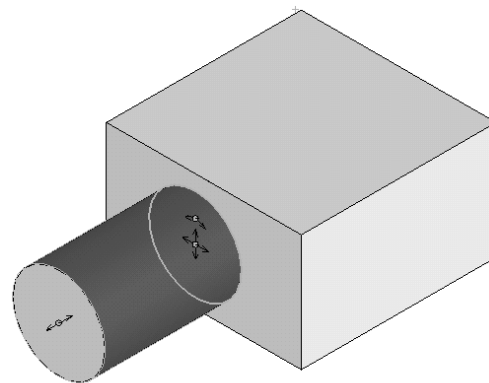


Figure 9-14 Resulting moved feature

You can also modify the sketches of the sketched feature using this option. To modify the sketch, move the cursor to the sketch highlighted in light green color. The symbol of the geometry, close to which the cursor is moved, appears on the right of the cursor. Press and hold down the left mouse button and drag the cursor. After modifying the sketch to the required size, release the left mouse button. Figure 9-15 shows the preview of the sketch being modified. Figure 9-16 shows the resulting modified model.

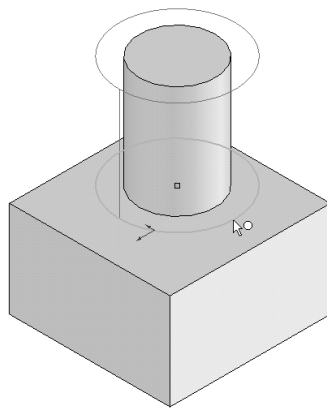


Figure 9-15 Preview of the sketch being modified

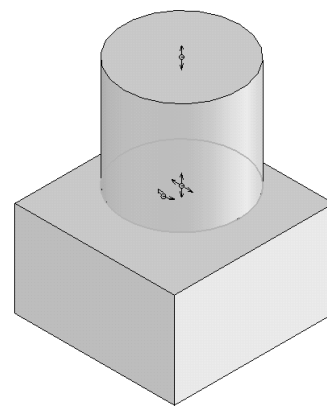


Figure 9-16 Resulting modified model

Editing Sketches With the Move/Size Features Tool Active

When the **Move/Size Features** tool is active, and you choose the **Edit Sketch** option to edit the sketch of the sketched feature, you will enter the sketching environment. The feature whose sketch you need to modify will be displayed in transparent yellow temporary graphics, as shown in Figure 9-17.

Now, modify the sketch according to the requirement. Consider a case in which two circles are added to the model and the height of the inclined line is increased. The model shown in temporary graphics will be modified dynamically in the sketching environment. Therefore, you



Tip. When you resize, move, or rotate a feature using the **Move/Size Features** tool, the feature(s) that has/have a relationship with that feature will also be modified. In other words, if a child-parent relationship is established between the features, the child features will also be modified when the parent feature are modified.

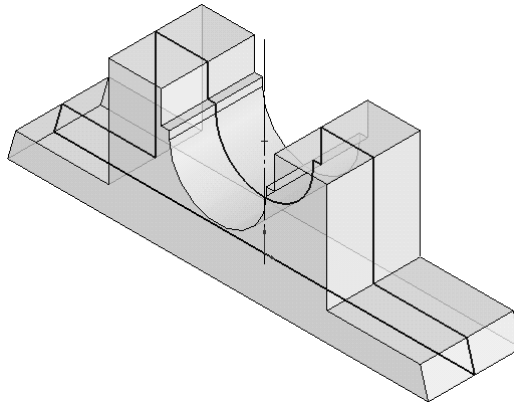


Figure 9-17 Preview of the sketch being modified



Tip. To clear the selected feature, click once anywhere in the drawing area, or choose the ESC key on the keyboard. You will notice that the **Move/Size Features** button is still chosen in the **Features** toolbar, indicating that the dynamic move/size features option is still active. To exit this option, choose the **Move/Size Features** button again.

can have a better understanding of how the model will be displayed after modifying the sketch. Note that the temporary graphics will show the preview of the feature created by the section you add only if you drag one of the entities in the sketch. Otherwise, the new addition will not be displayed as a preview in the temporary graphics. It is recommended that you invoke the **Modify/Size Features** button before modifying the sketch. After modifying the sketch, exit the sketching environment. If the sketch of the model is dimensioned, you can modify it by modifying the dimension. Figure 9-18 shows two circles added to the model and the length of the inclined line being increased. Figure 9-19 shows the resulting modified model in the sketching environment.



Tip. If you want to modify a fully or partially defined sketch by dragging, the **Override Dims on Drag/Move** option should be selected. To select this option, choose **Tools > Sketch Settings > Override Dims on Drag/Move** from the menu bar. If this option is not selected, you cannot move or drag a dimensioned sketched entity.

Editing Features and Sketches by Cut, Copy, and Paste

SolidWorks allows you to adapt the windows functionality of cut, copy, and paste to copy and paste the features and sketches. The method of using this functionality is the same as used in other windows-based applications. Select the feature or sketch to cut or copy. To cut the selected

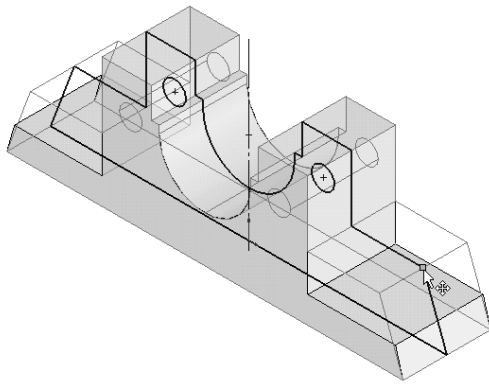


Figure 9-18 Sketch being modified

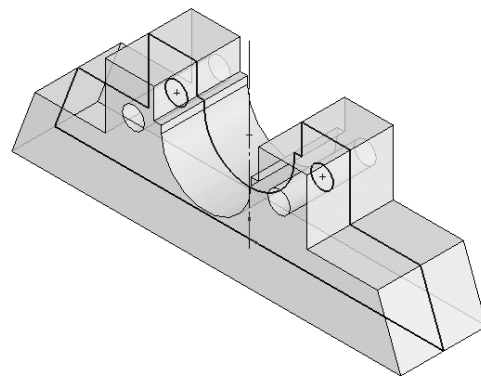


Figure 9-19 Resulting modified model in the sketching environment

item, choose **Edit > Cut** from the menu bar or use the shortcut key, CTRL+X. The **Confirm Delete** dialog box is displayed, because when you cut the selected item, it is deleted from the document. Choose the **Yes** button from this dialog box. You will learn more about deleting later in this chapter. After you cut an item, select the placement plane or placement reference where you want to place the feature. Choose **Edit > Paste** from the menu bar or use the shortcut key CTRL+V on the keyboard. Sometimes the **Copy Confirmation** dialog box is displayed, as shown in Figure 9-20, in which you are prompted to delete the external constraints or leave them dangling. This dialog box is displayed only when the item to paste has some external references in the form of relations and dimensions. The feature will be pasted on the selected reference.



Figure 9-20 The *Copy Confirmation* dialog box



Tip. For pasting a selected sketch, you need to select a plane or a planar face as the reference. For pasting a sketched feature, a simple hole, or a hole created using the hole wizard, you have to select a plane or a planar face as the reference. For pasting chamfers and fillets, you have to select an edge, edges, or face as the references.

If you copy and paste an item, the selected item will remain at its position and its copy will be pasted on the selected reference. To copy an item, select the feature or sketch. Choose **Edit > Copy** from the menu bar, or press CTRL+C on the keyboard. Select the reference where you want to paste the selected item and choose **Edit > Paste** from the menu bar, or press CTRL+V

on the keyboard to paste. You can paste the selected item any number of times. If you select another item and copy it on the clipboard, the last copied item will be deleted from the memory of the clipboard.

Cutting, Copying, and Pasting Features and Sketches from One Document to the Other

You can also cut or copy the features and sketches from one document and paste them in another document. For example, you need to copy a sketch created in the current document and paste it in a new document. You need to select the sketch and use CTRL+C on the keyboard to copy the item to the clipboard. Create a new document in the **Part** mode and select the plane on which you want to paste the sketch. Press CTRL+V on the keyboard to paste the sketch on the selected plane. Using the same procedure, you can also copy features from one document to the other.

Copying Features Using Drag and Drop

SolidWorks also provides you with the drag and drop functionality of windows to copy and paste the item within the document. Press and hold down the CTRL key on the keyboard. Next, select and drag that item from the drawing area or from the **FeatureManager Design Tree**. Drag the cursor to a location where you want to paste the item and release the left mouse button. If the item to be pasted is defined using the dimensions or relations, the **Copy Confirmation** dialog box will be displayed to delete or make those constraints dangle. Figure 9-21 shows the feature being dragged. Figure 9-22 shows the resulting pasted feature.

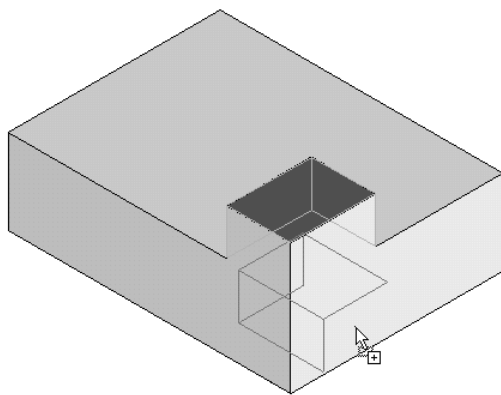


Figure 9-21 Feature being dragged

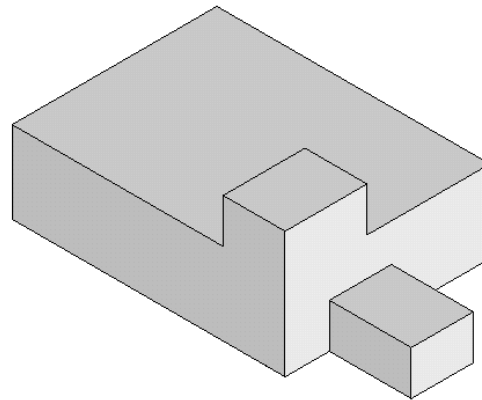


Figure 9-22 Resulting pasted feature

Dragging and Dropping Features from One Document to the Other

You can also drag and drop features and sketches from one document to the other. For pasting the items from one document to the other, you should open both the documents in the SolidWorks session. Choose **Windows > Tile Vertical/Tile Horizontal** from the menu bar. Display both the documents at the same time in the SolidWorks window. Press and hold down the CTRL key on the keyboard. Select and drag the feature or sketch to the other document, and place it on the

required reference. Figure 9-23 shows the fillet feature being dragged to be applied on the edge of the model in the second document.

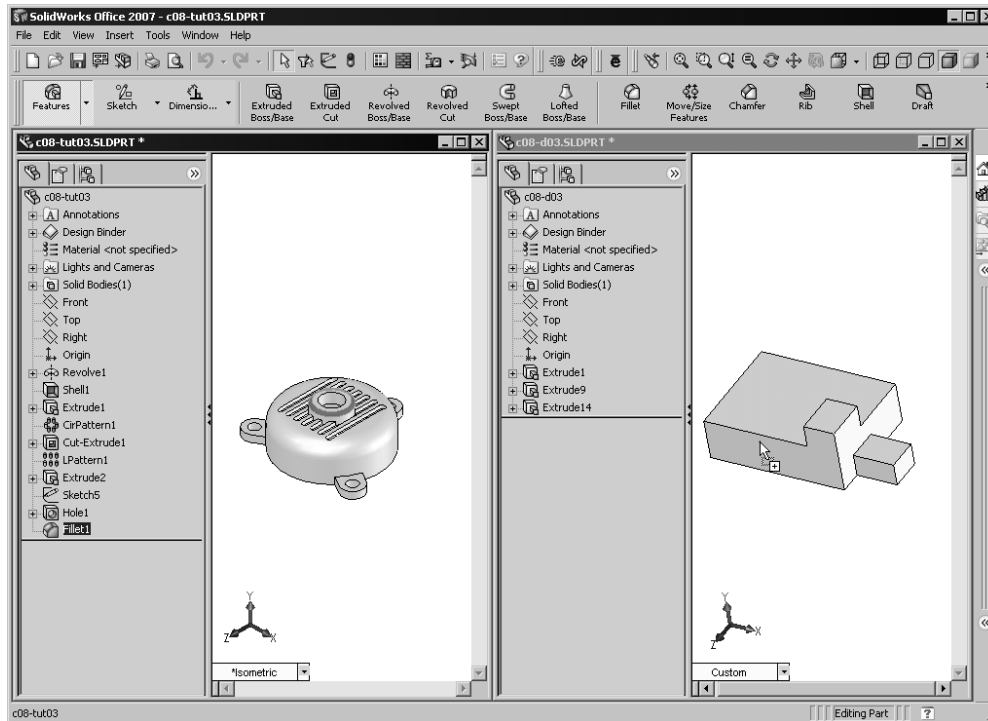


Figure 9-23 Fillet feature being dragged to be pasted in the second document

Deleting Features

You can delete the unwanted features from the model by selecting the feature from the **FeatureManager Design Tree** or from the drawing area. After selecting the feature to be deleted, choose the **DELETE** key on the keyboard, or right-click to invoke the shortcut menu and choose the **Feature** option from the **Feature** area. When you delete a feature, the **Confirm Delete** dialog box is displayed, as shown in Figure 9-24. The features that are dependent on the feature to be deleted are also displayed in the **Confirm Delete** dialog box, which informs you that all the dependent features to the parent feature will also be deleted. You are provided with the **Also delete all child features** check box. If this check box is selected, all the child features related to the parent feature are also deleted. When you delete a sketched feature, the sketches related to it will not be deleted. These sketches are known as absorbed features. To delete the absorbed features along with the parent feature, select the **Also delete absorbed features** check box from the **Confirm Delete** dialog box. Choose the **Yes** button to delete the selected features, or else choose the **No** button to cancel the delete operation. You can also delete a selected feature by choosing **Edit > Delete** from the menu bar.

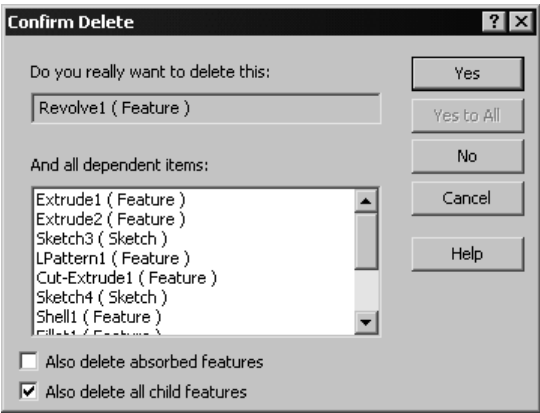


Figure 9-24 The Confirm Delete dialog box

Deleting Bodies

CommandManager:	Features > Delete Solid/Surface	(Customize to Add)
Menu:	Insert > Features > Delete Body	
Toolbar:	Features > Delete Solid/Surface	(Customize to Add)



As discussed earlier, the multibody environment is supported in SolidWorks. Therefore, you can create multiple disjoint bodies in SolidWorks. You can also delete the unwanted bodies. The bodies to be deleted can be solid bodies or surface bodies. To delete a body, choose **Insert > Features > Delete Body** from the menu bar. You can invoke this tool by selecting the **Delete Solid/Surface** button from the **Features CommandManager** after customizing the **CommandManager**. The **Delete Body PropertyManager** is invoked, as shown in Figure 9-25. You are prompted to select the solid and/or surface bodies to be deleted.

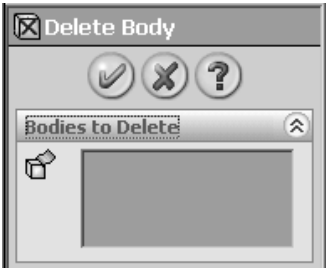


Figure 9-25 The Delete Body PropertyManager

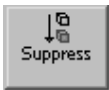
Select the body or bodies to be deleted from the drawing area or from the **Solid Bodies** folder available in the **FeatureManager Design Tree**, which is displayed in the drawing area. The selected body is displayed in green and its name is displayed in the **Bodies to Delete** selection box. Choose the **OK** button from the **Delete Body PropertyManager**. A new item with the name **Body-Delete1** appears in the **FeatureManager Design Tree**. This item stores the deleted bodies. Therefore, at any point of your design cycle, you can delete or suppress this item to resume the deleted body back in your design. You will learn more about suppressing the feature later in this chapter.



Tip. You can also choose the **Delete Body** option from the shortcut menu. To delete a body using the shortcut menu, select the body and right-click to invoke the shortcut menu. Choose the **Delete** option from the **Body** area of the shortcut menu; the **Delete Body PropertyManager** will be displayed. Choose the **OK** button from the **Delete Body PropertyManager** to delete the body.

Suppressing Features

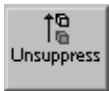
CommandManager:	Features > Suppress (Customize to Add)
Menu:	Edit > Suppress > This Configuration
Toolbar:	Features > Suppress (Customize to Add)



Sometimes, you do not want a feature to be displayed in the model or in its drawing views. Instead of deleting those features, they can be suppressed. When you suppress a feature, it is neither visible in the model nor in the drawing views. If you create an assembly using that model, the suppressed feature will not be displayed even in the assembly. You can resume the feature anytime by unsuppressing it. When you suppress a feature, the features that are dependent on it are also suppressed. To suppress a feature, select it from the **FeatureManager Design Tree** or from the drawing area. Choose the **Suppress** button from the **Features CommandManager** after customizing it, or right-click and choose the **Suppress** option from the shortcut menu. The suppressed feature will be removed from the display of the model, and the icon of the feature will be displayed in gray in the **FeatureManager Design Tree**. When you suppress a feature, then all the features dependent on that feature are also suppressed.

Unsuppressing the Suppressed Features

CommandManager:	Features > Unsuppress (Customize to Add)
Menu:	Edit > Unsuppress > This Configuration
Toolbar:	Features > Unsuppress (Customize to Add)



The suppressed features can be unsuppressed using this option. To resume the suppressed feature, select the suppressed feature from the **FeatureManager Design Tree** and choose the **Unsuppress** button from the **Features CommandManager**. You can also choose this option from the shortcut menu, after selecting the suppressed feature. As discussed earlier, when you suppress a feature, the dependent features are also suppressed. But when you resume a suppressed feature, the dependent features remain suppressed. Therefore, you need to unsuppress all the features independently. The method of resuming the parent feature and the dependent features using a single option is discussed next.

Unsuppressing Features With Dependents

CommandManager:	Features > Unsuppress with Dependents (Customize to Add)
Menu:	Edit > Unsuppress with Dependents > This Configuration
Toolbar:	Features > Unsuppress with Dependents (Customize to Add)



Using this option, you can resume the suppressed feature along with the dependents of the suppressed parent feature. To resume the suppressed feature using this option, select the suppressed feature from the **FeatureManager Design Tree**. Choose **Unsuppress with Dependents** button from the **Features** toolbar. You will observe that the dependent suppressed features are also unsuppressed.

Hiding Bodies

While working in the multibody environment, you can also hide the bodies. The hidden body

is not displayed in the model, assembly, or in the drawing views. The display of the dependent bodies is not turned off when you hide a body. To hide a body, expand the **Solid Bodies** folder available in the **FeatureManager Design Tree**, and select the body to be hidden. Right-click to invoke the shortcut menu and choose the **Hide Solid Body** option from it. The selected body will disappear from the drawing area. The icon of the hidden body is displayed in wireframe in the **Solid Bodies** folder. Select the hidden body from that folder and choose the **Show Solid Body** option from the shortcut menu to turn on the display of the hidden body.

Moving and Copying Bodies

CommandManager:	Features > Move/Copy Bodies (Customize to Add)
Menu:	Insert > Features > Move/Copy Bodies
Toolbar:	Features > Move/Copy Bodies (Customize to Add)



In SolidWorks, you can also move or copy the bodies by choosing the **Move/Copy Bodies** button from the **Features CommandManager** after customizing it, or choosing **Insert > Features > Move/Copy Bodies** from the menu bar. The **Move/Copy Body PropertyManager** will be displayed. By default, it shows the **Mates Settings** rollout. Choose the **Translate/Rotate** button below the **Options** rollout; the **Translate** and **Rotate** rollouts will be displayed, as shown in Figure 9-26.

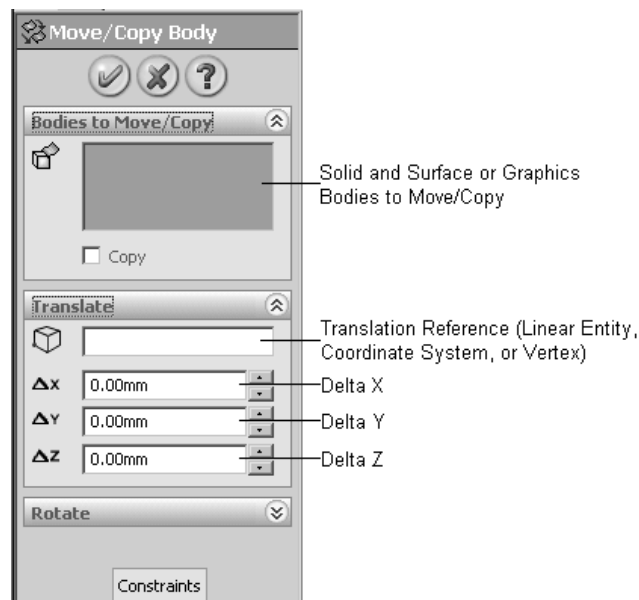


Figure 9-26 The Move/Copy Body PropertyManager

You will notice a 3D triad is displayed at the centroid of the selected body. It is used to dynamically rotate or move the selected body. The three arrows of this triad are the X, Y, and Z axes along which the body can be moved. The triad also has three rings around which the body can be rotated, see Figure 9-27.

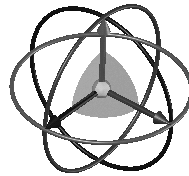


Figure 9-27 3D triad to move and rotate the body

To move a body dynamically, move the cursor close to any one of the arrows of the triad; the arrow is highlighted and the select cursor is replaced by the move cursor. Press and hold the left mouse button and drag the cursor to move the body. You can also move the cursor on the plane between two arrows to move the body in that plane.

To rotate the body, move the cursor over one of the rings; the ring will be highlighted and the cursor will be replaced by the rotate cursor. Press and hold the left mouse button and drag the cursor to rotate the body.

You can also move the body freely in 3D space. To do so, move the cursor on the spherical ball where the three arrows meet. The cursor is replaced by the move cursor. Drag it to move the body to a desired location in the 3D space.

Various options available in the **Move/Copy Body PropertyManager** are discussed next.

Bodies to Move/Copy

The **Bodies to Move/Copy** rollout is used to define the body to copy or move. On invoking the **Move/Copy Body PropertyManager**, you are prompted to select the bodies to move/copy and set the options. Move the cursor on the body to be selected; the cursor will be replaced by the body selection cursor and the edges of the body will be highlighted in red. The name of the body is also displayed in the tooltip. Select the body; it will be highlighted in green. The name of body is displayed in the **Solid and Surface or Graphics Bodies to Move/Copy** selection box. You can also select the body from the **Solid Bodies** folder after invoking the **FeatureManager Design Tree** flyout.

Copy

The **Copy** check box is cleared by default. If you select the **Copy** check box, you can create the copies of the selected body. When you select this check box, the **Number of Copies** spinner is displayed below the **Copy** check box. Set the number of copies using this spinner.

Translate

The **Translate** rollout available in the **Move/Copy Body PropertyManager** is used to define the translational parameters to move the selected body. Set the value of the destination in the **Delta X**, **Delta Y**, and **Delta Z** spinners. When you set the value, the preview of the moved body is displayed in temporary graphics in the drawing area. You can also move or copy the selected body between points. To move or copy a body by specifying two points, select the **Translation Reference (Linear Entity, Coordinate System, Vertex)** selection box. The selection mode in

this area is active. Select the vertex from which you want the translation to start. When you select the first vertex as the translation reference, the **Delta X**, **Delta Y**, and **Delta Z** spinners are replaced by the **To Vertex** selection box. The selection mode in the **To Vertex** selection box is active. Select the second translation reference. You will observe the preview of the body move between the two selected vertices. The placement of the body also depends on the sequence of selection of the vertices. Therefore, you need to be very careful, while selecting the two vertices. Figure 9-28 shows the sequence for the selection of references and Figure 9-29 shows the resulting copied body.

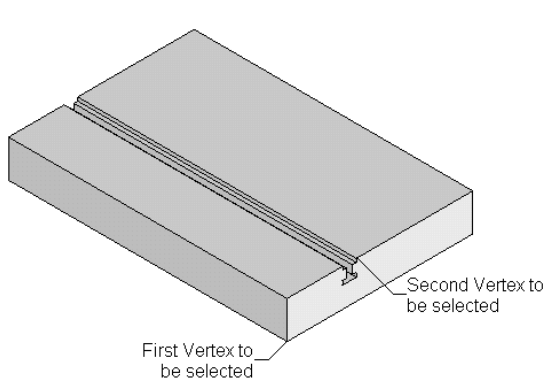


Figure 9-28 Sequence of selection

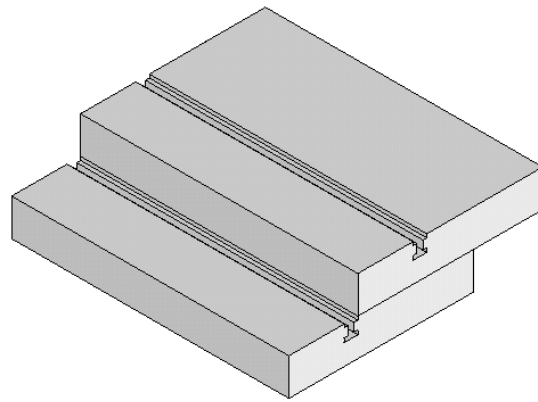


Figure 9-29 Resulting copied body

Rotate

The **Rotate** spinner available in the **Move/Copy Body PropertyManager** is used to define the parameters to rotate the body. To open this rollout, click once on the black arrow provided at the right of this rollout. The **Rotate** rollout is shown in Figure 9-30.

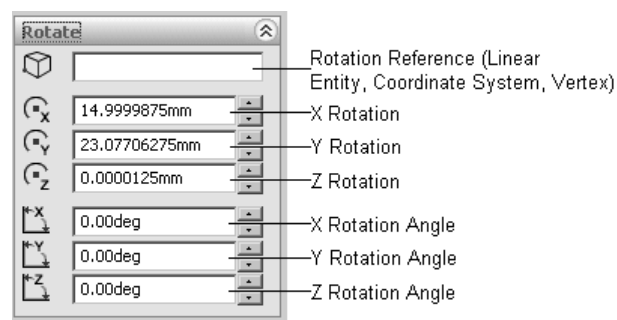


Figure 9-30 The **Rotate** rollout

As discussed earlier, a filled square is placed at the origin when you invoke the **Move/Copy PropertyManager**. It is clearly visible, when you hide the origin by choosing **View > Origin** from the menu bar. It indicates the origin along which the selected body will be rotated. You can adjust the position of this temporary moveable origin using the **X Rotation Origin** spinner, **Y**

Rotation Origin spinner, and **Z Rotation Origin** spinner. The **X Rotation Angle** spinner is used to set the value of the angular increment to rotate or copy the body along the X axis, the **Y Rotation Angle** spinner is used to rotate or copy the body along the Y axis, and the **Z Rotation Angle** spinner is used to rotate or copy the body along the Z axis.

To rotate or copy the selected body along an edge, click once in the **Rotation Reference (Linear Entity, Coordinate System, Vertex)** selection box to invoke the selection. Select the edge along which you want to rotate the selected body. When you select an edge, all the other spinners disappear from the rollout, and the **Angle** spinner is invoked in the **Rotate** rollout. Set the value of the angular increment in this spinner.

Instead of selecting an edge, you can also select a vertex along which the body will rotate or copy. Then you need to specify the axis along which you want to rotate it.

Constraint

In SolidWorks, you can apply mates between the multiple bodies to place them at an appropriate location. Choose the **Constraint** button from the **Move/Copy Body PropertyManager**; the **Bodies to Move** and the **Mate Settings** rollouts are displayed, as shown in Figure 9-31. You need to select the body that you need to move. The **Mate Setting** rollout helps you to position the selected body by applying mates. You will learn more about mates in later chapters.

Reordering Features

Reordering the features is defined as a process of changing the position of the features created in the model. Sometimes, after creating a model, it may be required to change the order in which its features were created. For reordering the features, the features are dragged and placed before or after another feature in the **FeatureManager Design Tree**.

For reordering the features, select the feature in the **FeatureManager Design Tree**, and drag it to the required position. When you drag the feature to reorder them, the bend arrow pointer is displayed, which suggests that feature dragging is possible. If you drag the child feature above the parent feature, the reorder error pointer will be displayed. If you drag a child feature above a parent feature, the **SolidWorks** warning box will be displayed, as shown in Figure 9-32. Choose **OK** from this warning box.

Consider a case, in which you have created a rectangular block and a pattern of through holes created on the base feature, as shown in Figure 9-33. Now, if you create a shell feature and remove the top face, the front face, and the right face of the model, it will appear, as shown in Figure 9-34.

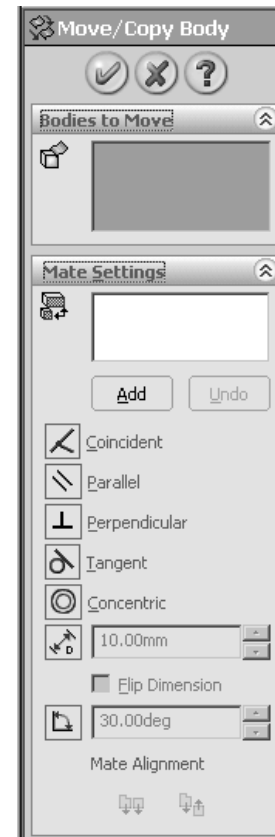


Figure 9-31 The **PropertyManager** after choosing the **Constraints** button

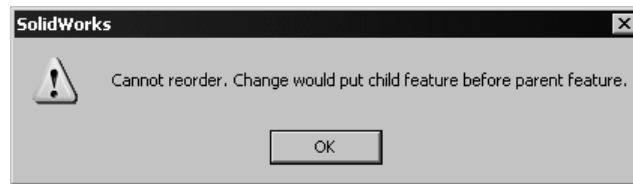


Figure 9-32 The SolidWorks warning box

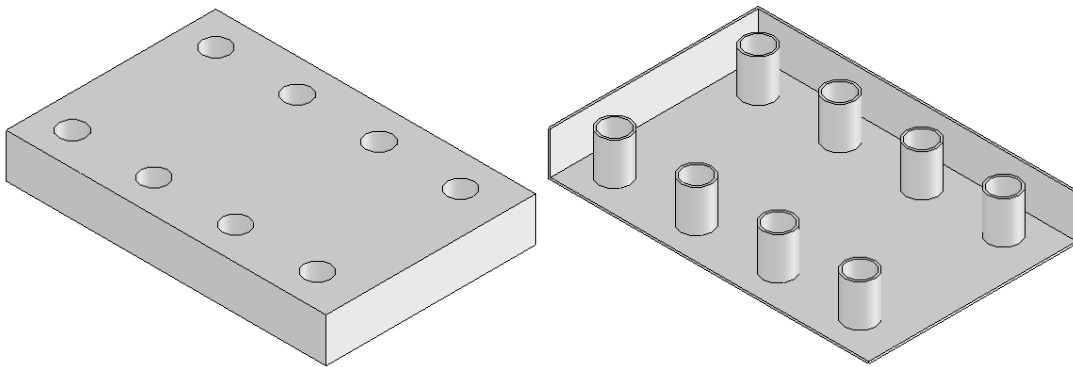


Figure 9-33 Model created with a pattern of through holes on the base feature

Figure 9-34 Shell feature added to the model

But this was not the desired result. Therefore, you need to reorder the shell feature before the holes. Select the shell feature in the **FeatureManager Design Tree** and drag it above the holes. All the features will be automatically adjusted in the new order, as shown in Figure 9-35.

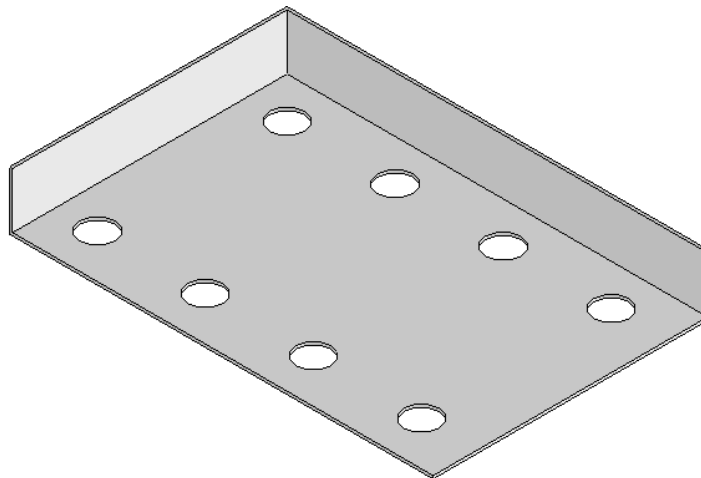


Figure 9-35 Model after reordering the features

Rolling Back the Model

Rolling back the model is defined as a process, in which you rollback the model to an earlier stage. When you rollback a feature or features, those features are suppressed, and you can add new features when the model is in the rollback state. The newly added features are added before the features that are rolled back. While working with a multifeatured model, if you want to edit a feature that was created at the starting of the design cycle of the model, it is recommended that you should rollback the model up to that feature. This is because after each editing operation, the time of regeneration will be minimized. Rolling back is done using the **Rollback Bar** available in the **FeatureManager Design Tree**, as shown in Figure 9-36.

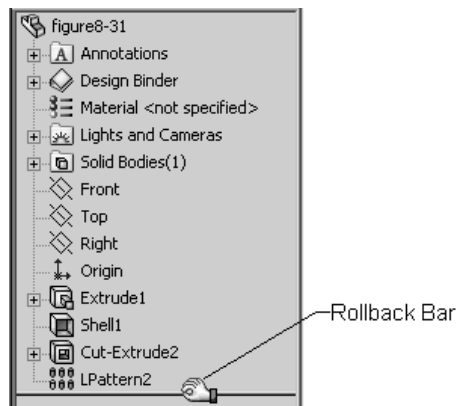


Figure 9-36 The rollback bar of the **FeatureManager Design Tree**

Using the select tool, select the **Rollback Bar**; after selection it will be changed to blue and the select cursor will be replaced by the hand pointer. Drag the hand pointer to the feature up to which you want to rollback the model, and then release it. To resume or roll the model, drag the **Rollback Bar** to the last feature of the model. You can also rollback using the menu bar. Select the feature up to which you want to rollback the model and choose **Edit > Rollback** from the menu bar. You will have to customize the **Edit** menu to add this option.



Tip. If you want to rollback the model to the previous step, choose **Edit > Roll to Previous** from the menu bar. To rollback the entire model to its original position, choose **Edit > Roll to End** from the menu bar.

You can also choose the **Roll Forward**, **Roll Previous**, **Roll End** options from the shortcut menus invoked by selecting the features placed below the **Rollback Bar**. These options are used to control the roll and rollback of the model.

You can also rollback the model using the keyboard, by selecting the **Rollback Bar** and using the **CTRL+ALT+Up Arrow** key to roll forward. Use the **CTRL+ALT+Down Arrow** key to roll backward.

Renaming Features

The names of the features are displayed in the **FeatureManager Design Tree**. By default, the

naming of the features is done according to the sequence in which they are created. You can also rename the features, according to your convenience, by selecting the feature from the **FeatureManager Design Tree** and then clicking once on the selected feature. An edit box will be displayed in the **FeatureManager Design Tree**. Enter the name of the feature and press the ENTER key or click anywhere on the screen.

Creating Folders in the FeatureManager Design Tree

You can also add the folders in the **FeatureManager Design Tree** and the features displayed in the **FeatureManager Design Tree** are added in the folder. This is done to reduce the length of the **FeatureManager**. Consider a case, in which the base of the model consists of more than one feature. You can add a folder named Base Feature, and add all the features used to create the base in that folder. To add a folder in the **FeatureManager Design Tree**, select any feature, right-click to invoke the shortcut menu, and choose the **Create New Folder** option. A new folder is created above the selected feature. Specify the name of the folder and click anywhere on the screen. Now, you can drag and drop the features to the newly created folder. You can also rename the folder by selecting it and then clicking on it once. Now, enter the name in the edit box and press the ENTER key.

To add the selected feature in a new folder, choose **Add to New Folder** from the shortcut menu. A new folder will be created in the **FeatureManager Design Tree**, and the selected feature will be added to the newly created folder. You can also delete the folder by selecting the folder, and invoking the shortcut menu, and selecting the **Delete** option from it. Using the options in this shortcut menu, you can also rollback and suppress the features available in the selected folder.

What's Wrong Functionality

Sometimes, a model may not be rebuilt properly after you modify a sketch or a feature because of the errors resulting from the modification. Therefore, you are provided with a **What's Wrong** dialog box shown in Figure 9-37. The possible errors in the feature are displayed in this dialog box along with their detailed description.

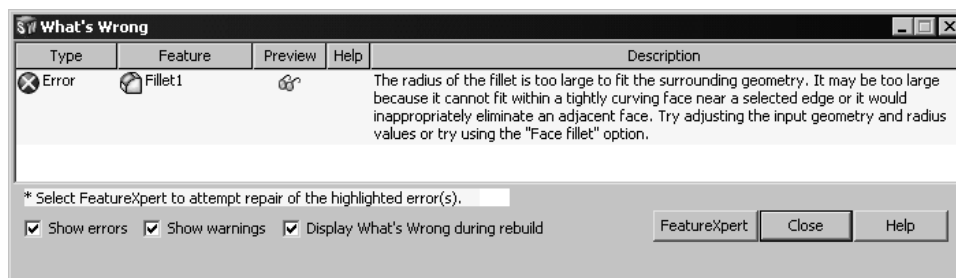


Figure 9-37 The *What's Wrong* dialog box

The **Show errors** check box is selected by default to display the errors in the **What's Wrong** dialog box. The **Show warnings** check box is selected by default to display the warnings messages. The **Display What's Wrong during rebuild** check box is selected by default and is used to display the errors at every rebuild of the model unless the error is fixed. After reading the

description of the errors from this dialog box, choose the **Close** button to exit it. The errors are also displayed in the **FeatureManager Design Tree**. The **FeatureManager Design Tree**, with an error in a feature, is displayed in Figure 9-38.

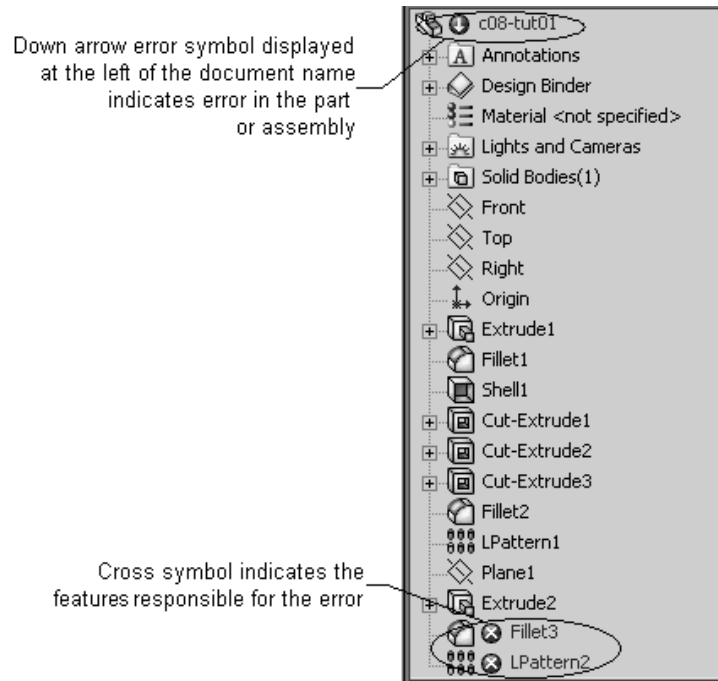


Figure 9-38 The **FeatureManager Design Tree** with a feature having errors

If there is an error in a model or in an assembly, the down arrow symbol appears on the left of the name of the model or assembly in the **FeatureManager Design Tree**. If a feature has an error, then the cross symbol appears on the left of the feature in the **FeatureManager Design Tree**. If there is an error in the child feature, the error symbol appears on the left of the parent feature and also on the name of the document in the **FeatureManager Design Tree**. If a warning message appears for a feature, then a triangle with an exclamation mark appears on the left of that feature in the **FeatureManager Design Tree**.



Tip. You can also invoke the **What's Wrong** dialog box by selecting the feature having errors from the **FeatureManager Design Tree** and choosing the **What's Wrong?** option from the shortcut menu.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 9-39. After creating some of its features, you will dynamically modify it, and then undo the modification. The dimensions of the model are shown in Figure 9-40. **(Expected time: 30 min)**

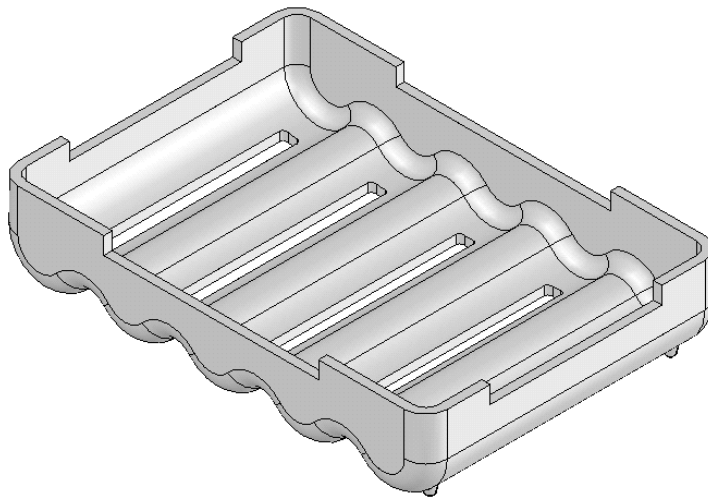


Figure 9-39 Model for Tutorial 1

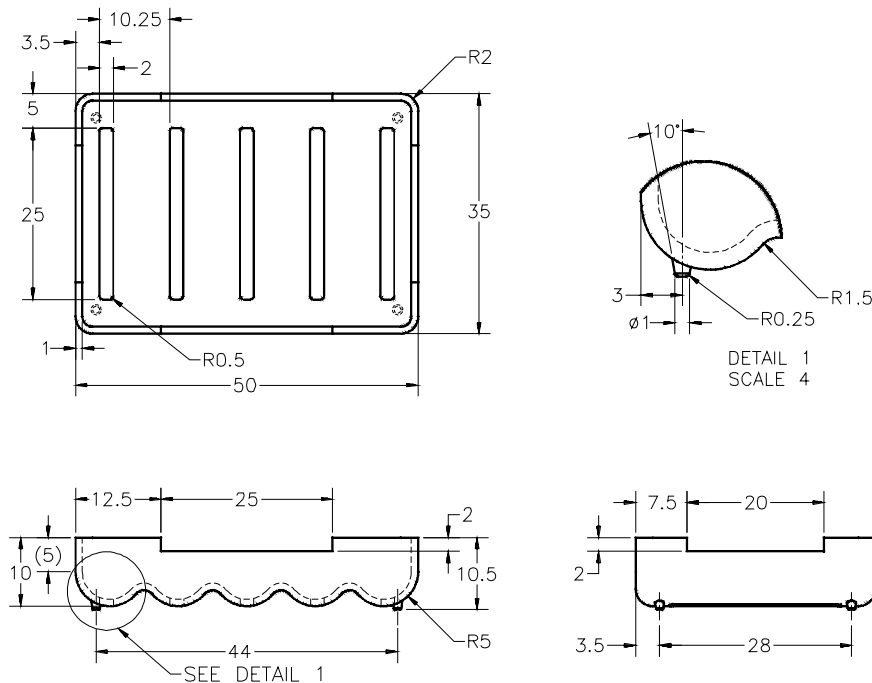


Figure 9-40 Views and dimensions of the model for Tutorial 1

The steps to be followed to create the model are listed next.

- Create the base feature of the model by extruding the profile to a given distance, refer to Figures 9-41 and 9-42.
- Add the fillets to the base feature, refer to Figures 9-43 and 9-44.
- Add the shell feature to the model and remove the top face of the base feature, refer to Figures 9-45 and 9-46.
- Dynamically modify the model, refer to Figures 9-47 through 9-48.
- Create the cuts on the sides of the model, refer to Figure 9-50.
- Create the slots on the lower part of the base and add the fillet to the slots feature, refer to Figure 9-50.
- Pattern the slots and the fillet feature, refer to Figure 9-50.
- Create a plane at an offset distance from the **Top Plane**.
- Create the standoffs using the extrude and **Fillet** tool, and pattern the standoffs, refer to Figure 9-51.

Creating the Base Feature

You will draw the sketch of the base feature on the **Front Plane** and extrude it using the **Mid Plane** option.

- Start SolidWorks and start a new SolidWorks Part document using the **New SolidWorks Document** dialog box.

2. Invoke the **Extruded Boss/Base** tool and draw the sketch of the base feature on the **Front Plane**. Add the required relations and dimensions to the sketch, as shown in Figure 9-41.

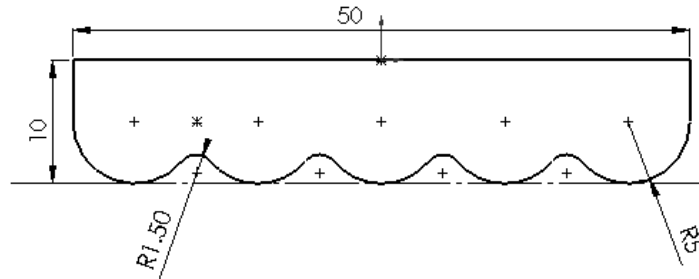


Figure 9-41 Sketch for the base feature

3. Extrude the sketch to a distance of 35 mm using the **Mid Plane** option.

The base feature of the model is shown in Figure 9-42.

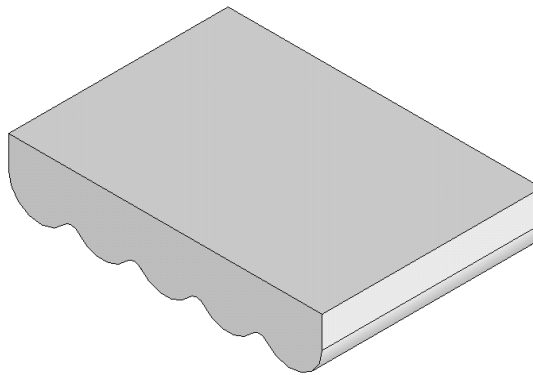


Figure 9-42 Base feature of the model

Adding Fillet to the Base Feature

After creating the base feature, you will fillet its lower edges.

1. Invoke the **Fillet PropertyManager**, rotate the model, and select the edges of the base feature, as shown in Figure 9-43.
2. Set the value of the **Radius** spinner to **2.5** and choose the **OK** button from the **Fillet PropertyManager**.

The model, after adding fillet to its edges, is shown in Figure 9-44.

Adding Shell to the Model

After creating the fillet feature, you need to shell the model using the **Shell** tool.

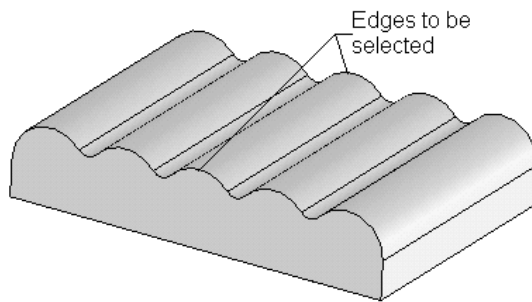


Figure 9-43 Edges to be selected

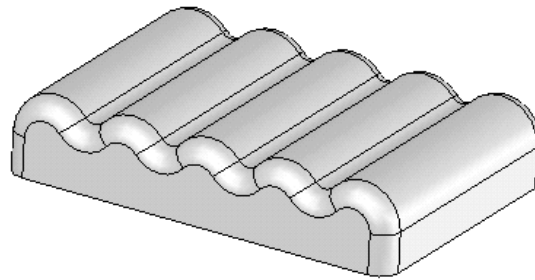


Figure 9-44 Fillet added to the model

1. Orient the model in the isometric view and invoke the **Shell1 PropertyManager**.
2. Select the top planar face of the model, as shown in Figure 9-45.
3. Set the value of the **Thickness** spinner to **1**, and choose the **OK** button from the **Shell PropertyManager**.

The model, after adding the shell feature, is shown in Figure 9-46.

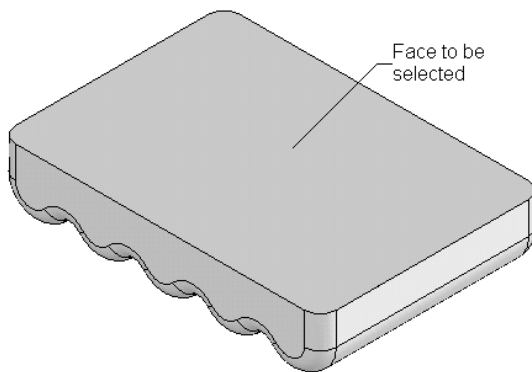


Figure 9-45 Face to be selected

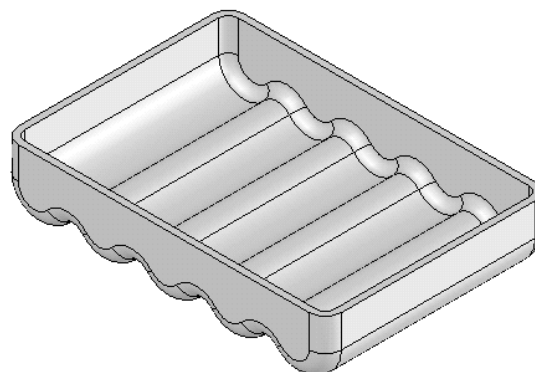


Figure 9-46 Shell feature added to the model

Dynamically Editing the Features

After creating and adding the shell to the base of the model you will learn how to edit the features dynamically using the **Move/Size Features** tool.

1. Choose the **Move/Size Features** button from the **Features CommandManager** after customizing it to invoke the dynamic dragging tool.
2. Select the right planar face of the base feature from the drawing area, refer to Figure 9-47. The selected face will be highlighted in green. The sketch of the selected feature is also



displayed in the drawing area, along with various editing handles, as shown in Figure 9-47.

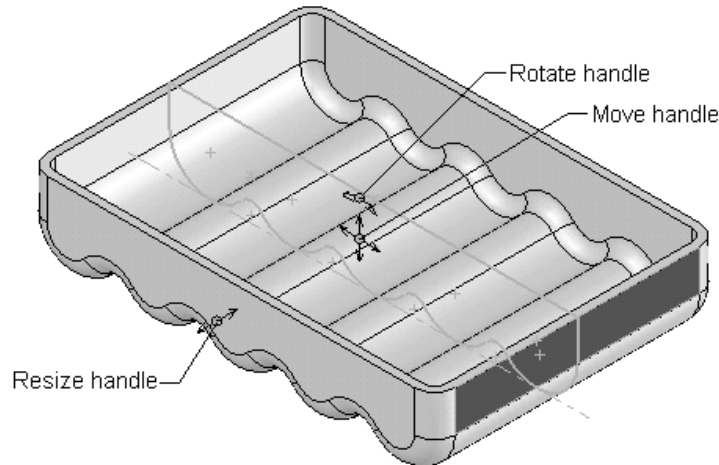


Figure 9-47 Editing handles for editing the base feature



Tip. You can also select the feature to be edited from the **FeatureManager Design Tree**.

To edit the sketches using the **Move/Size Features** tool, you need to select the **Override Dimension** option by choosing **Tools > Sketch Settings > Override Dims on Drag/Move** from the menu bar. The check mark displayed on the left of this option in the menu bar means that the option is already chosen.

3. Select the **Resize** handle from the drawing area and drag the cursor to resize the feature.

The preview of the resized feature and its dimensions are displayed in the drawing area. As you drag the cursor, the preview and the dimensions update automatically.

4. Release the left mouse button after dragging the feature to some distance. Figure 9-48 shows the preview of the dragged feature and Figure 9-49 shows the edited feature.
5. Select the **Move/Size Features** button again to disable the dynamic editing tool and click anywhere in the drawing area to clear all the selections from the selection set.

You have edited the model by dynamically dragging, but the original depth value of the feature was 35 mm. To bring the base feature back to its original size, you need to edit the feature.

6. Double-click on the base feature in the **FeatureManager Design Tree** or from the drawing area. All dimensions of the feature are displayed in it.
7. Double-click on the dimension that reflects the depth of the base feature that appears in blue.

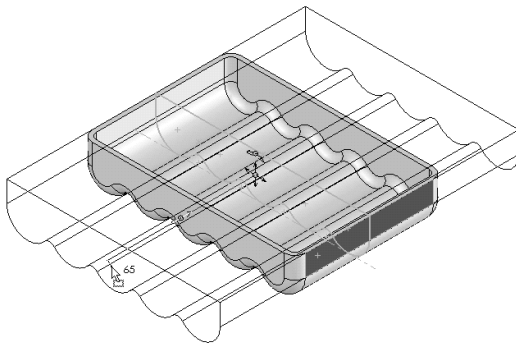


Figure 9-48 Dragging the **Resize** handle

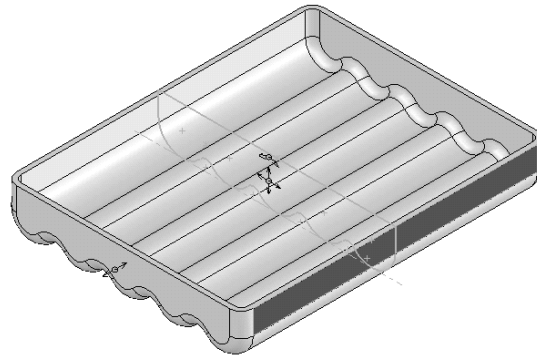


Figure 9-49 Resulting edited feature

8. The **Modify** dialog box will be displayed. Set the value of the **Dimension** spinner to **35** and press the ENTER key on the keyboard.
9. Choose the **Rebuild** button from the **Standard** toolbar or CTRL+B on the keyboard to rebuild the model.
10. Using the **Extruded Cut**, **Fillet**, and **Linear Pattern** tools, create the remaining features of the model. The model, after creating the features using these tools, is displayed in Figure 9-50.

Creating the Standoffs

Now, you need to create the standoffs for the model. It is created by extruding a sketch drawn on a sketch plane at an offset distance from the **Top Plane**. You also need to specify a draft angle, while creating this feature.

1. Create a reference plane at an offset distance of 10.5 mm from the **Top Plane**. You need to select the **Reverse direction** check box from the **Plane PropertyManager**.
2. Select the newly created sketching plane, draw the sketch of the standoff, and apply the required relations and dimensions. The sketch consists of a circle of diameter 1 mm. For more dimensions, refer to Figure 9-40.
3. Extrude the sketch using the **Up To Next** option with an outward draft angle of 10-degrees. Hide the newly created plane.

This creates the standoffs of the model.

4. Rotate the model and add a fillet of radius 0.25 to the base of the standoff.

The rotated and zoom view of the complete standoff is displayed in Figure 9-51.

5. Pattern the filleted standoff feature using the **Linear Pattern** tool. The isometric view of the final model is shown in Figure 9-52.

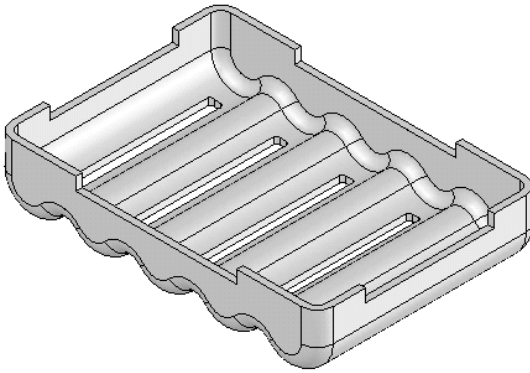


Figure 9-50 Model after creating other features

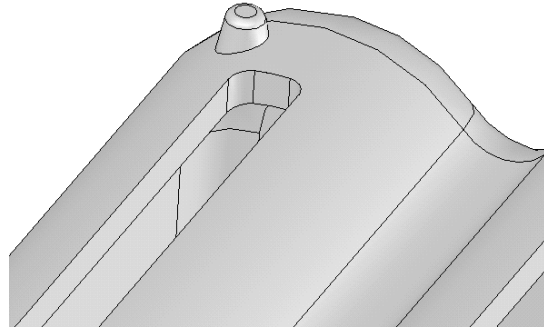


Figure 9-51 Rotated and zoom view of the model to show the standoff

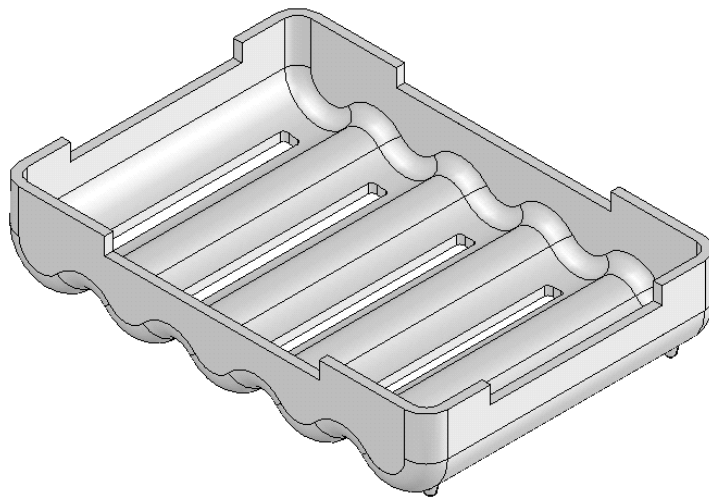


Figure 9-52 Final model

Saving the Model

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below.

`\\My Documents\\SolidWorks\\c09\\c09tut1.sldprt`

2. Choose **File > Close** from the menu bar to close the document.

Tutorial 2

In this tutorial, you will create the model, shown in Figure 9-53, and then edit it with the **Move/Size Features** option selected. The views and dimensions of the model are shown in the same figure.
(Expected time: 45 min)

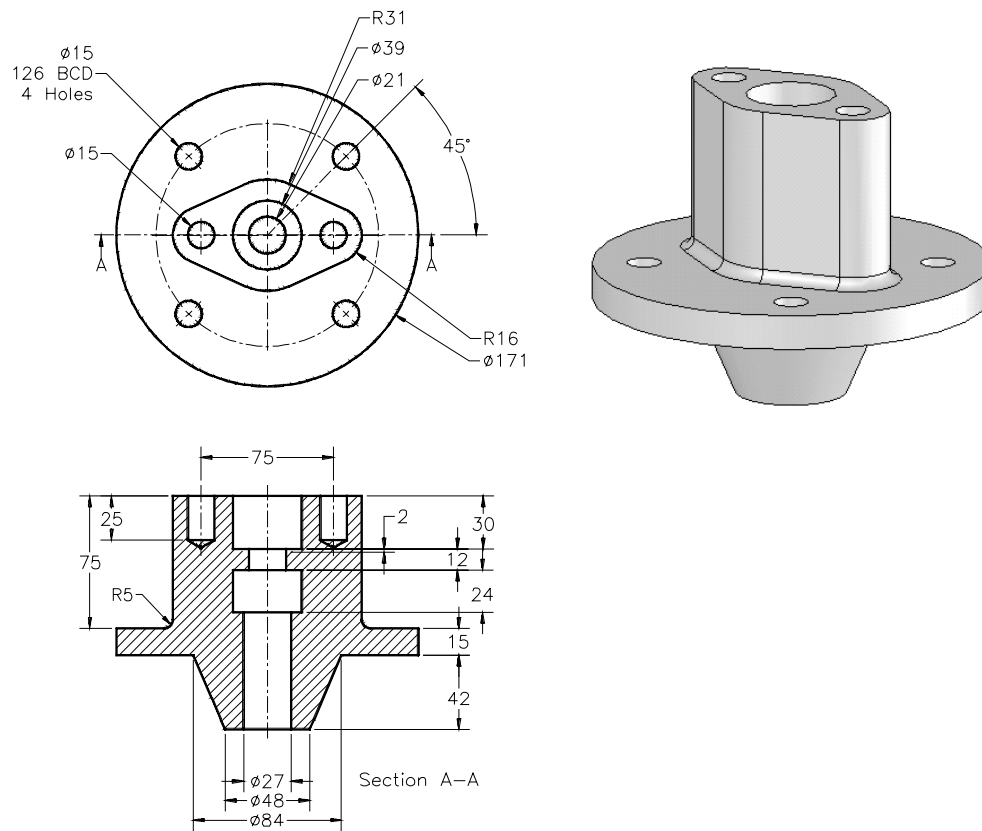


Figure 9-53 Views and dimensions of the model for Tutorial 2

The steps to be followed to complete this tutorial are listed next.

- Create the base feature of the model by revolving the sketch along the central axis of the sketch, refer to Figures 9-54 and 9-55.
- Draw the sketch of the second feature on the top face of the base feature and extrude it to a given dimension, refer to Figures 9-56 and 9-57.
- Create the revolve cut feature, refer to Figures 9-58 and 9-59.
- Create the hole using the **Simple Hole** tool, and then pattern it using the **Circular Pattern** tool, refer to Figure 9-59.
- Create a drilled hole feature using the **Hole Wizard** tool, refer to Figure 9-59.
- Mirror the hole feature along the **Right Plane**, refer to Figure 9-59.
- Apply the fillet, refer to Figure 9-59.
- Edit the model using the **Move/Size Features** tool, refer to Figures 9-60 through 9-62.

Creating the Base Feature

First, you will create the base feature of the model by revolving the sketch created on the **Front Plane**.

1. Start a new SolidWorks Part document using the **New SolidWorks Document** dialog box.
2. Invoke the **Revolved Boss/Base** tool and draw the sketch of the base feature on the **Front Plane**. Add the required relations and dimensions to the sketch, as shown in Figure 9-54.
3. Exit the sketching environment.

You do not need to set any parameters in the **Revolve PropertyManager** because the default value in the **Angle** spinner is 360-degrees, as required.

4. Choose the **OK** button from the **Revolve PropertyManager**.

The base feature created, after revolving the sketch, is shown in Figure 9-55.

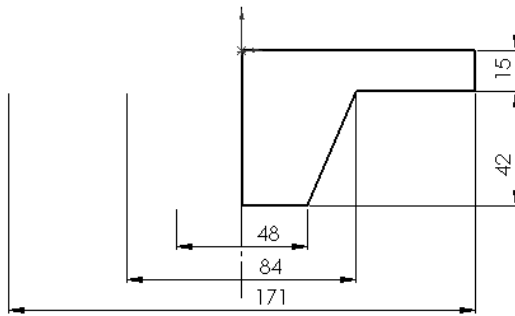


Figure 9-54 Sketch for the base feature

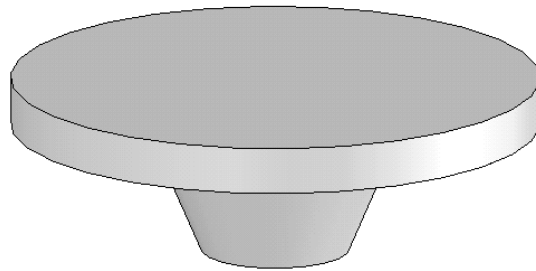


Figure 9-55 Dimetric view of the base feature

Creating the Second Feature

The second feature of this model is an extruded feature. It is created by extruding a sketch created on the top planar face of the base feature.

1. Invoke the **Extruded Boss/Base** tool and select the top planar face of the base feature as the sketching plane.
2. Draw the sketch of the second feature and apply the required relations and dimensions to it, as shown in Figure 9-56. Make sure that the sketch is symmetrical about the centerline. If it is not, the sketch will not be properly mirrored.
3. Extrude the sketch to a distance of 75 mm.

The isometric view of the model, after creating the second feature, is displayed in Figure 9-57.

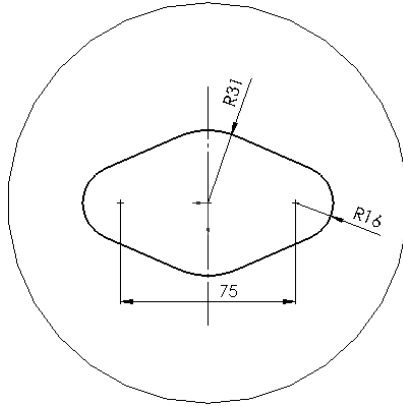


Figure 9-56 Sketch for the second feature

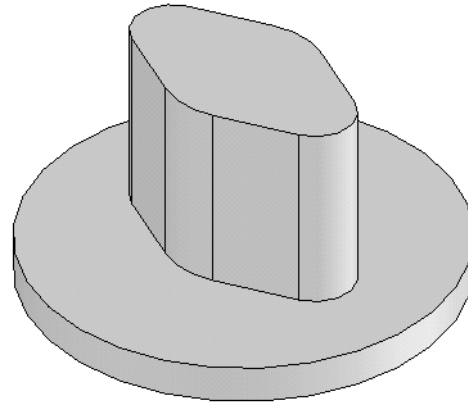


Figure 9-57 Second feature added to the model

Creating the Third Feature

The third feature of the model is created by revolving a sketch using the cut option. The sketch for this feature will be created on the **Front Plane**.

1. Invoke the **Revolved Cut** tool and select the **Front Plane** as the sketching plane.
2. Draw the sketch for the revolved cut feature and apply the required relations and dimensions to it, as shown in Figure 9-58.
3. Exit the sketching environment, and create a revolved cut feature with a default angle value of 360-degrees.

Creating the Remaining Features

1. Create the other features of the model by referring to Figure 9-53 and using the **Simple Hole**, **Hole Wizard**, and **Mirror** tools.

The isometric view of the model, after creating all the other features, is displayed in Figure 9-59.

Editing the Sketch of the Model With the Move/Size Features tool

After creating the model, you will edit the sketch of the second feature with the **Move/Size Features** tool. Therefore, before proceeding further you need to invoke this option.

1. Choose the **Move/Size Features** button from the **Features CommandManager** to invoke the **Move/Size Features** tool.
2. Select the second feature from the drawing area, and right-click to invoke the shortcut menu.
3. Choose the **Edit Sketch** option from the shortcut menu.



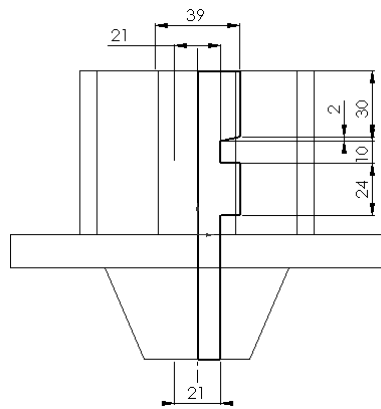


Figure 9-58 Sketch for the revolve cut feature

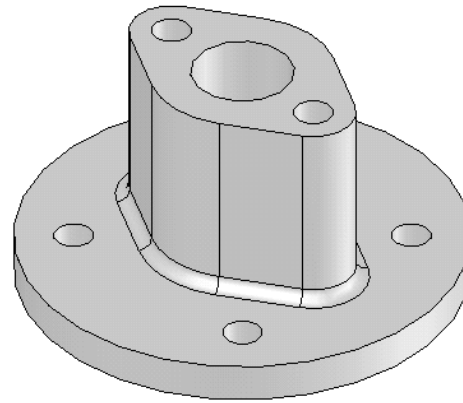


Figure 9-59 Isometric view of the final model

The sketch of the second feature with the preview of the feature in temporary graphics is displayed in the drawing area, as shown in Figure 9-60.

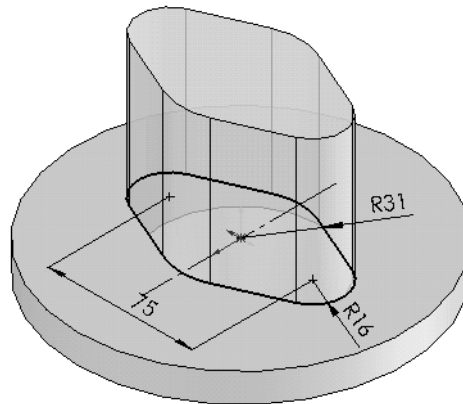


Figure 9-60 Sketch and the preview of the second feature

Now, you will modify the sketch by dragging the sketched entities. You will observe that the preview of the second feature displayed in temporary graphics is also modified.

4. Choose the **Tools > Sketch Settings > Override Dims on Drag/Move** option from the menu bar, if it is not selected.
5. Select the center point of the right arc of the sketch and drag the cursor toward the right. As you drag the cursor, the preview of the feature is also modified dynamically. Release the left mouse button. The new dimensions appear on the sketch. Continue this process until the value of the distance between the center of the right and left arcs shows a value close to 135.

Figure 9-61 shows the sketch being dragged. Figure 9-62 shows the final dragged sketch.

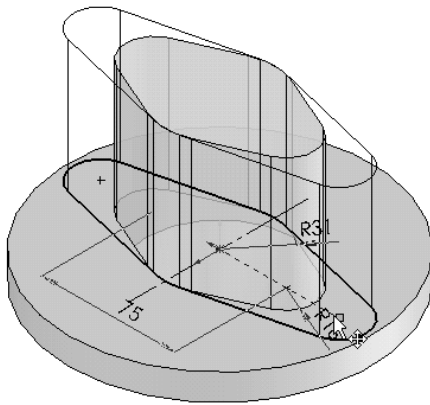


Figure 9-61 Dragging the center point of the right arc

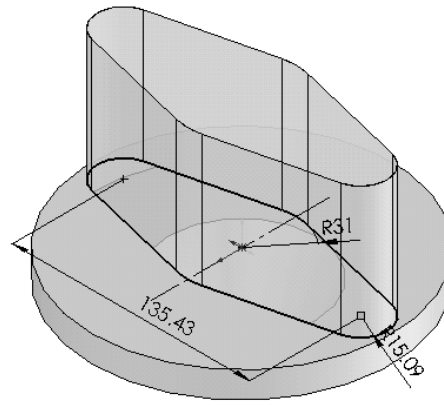


Figure 9-62 Sketch after dragging

After modifying the sketch by dragging, you need to change the modified dimensions back to the original dimensions.

6. Double-click the linear dimension value between the center points of the right and left arcs, enter **75** in the **Modify** edit box, and press the ENTER key.

Similarly, edit the other two dimensions using the **Modify** edit box, if necessary.

7. Use CTRL+B on the keyboard to rebuild the model.

Saving the Model

Now, you need to save the model.

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below.

\\My Documents\\SolidWorks\\c09\\c09tut2.sldprt.

2. Choose **File > Close** from the menu bar to close the document.

Tutorial 3

In this tutorial, you will create the model shown in Figure 9-63. While creating it, you will also perform some editing operations on it. The views and dimensions of the model are displayed in Figure 9-64.

(Expected time: 45min)

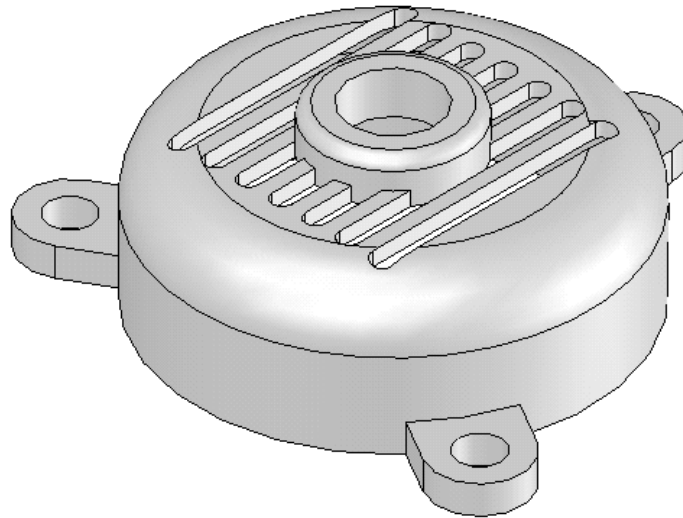


Figure 9-63 Model for Tutorial 3

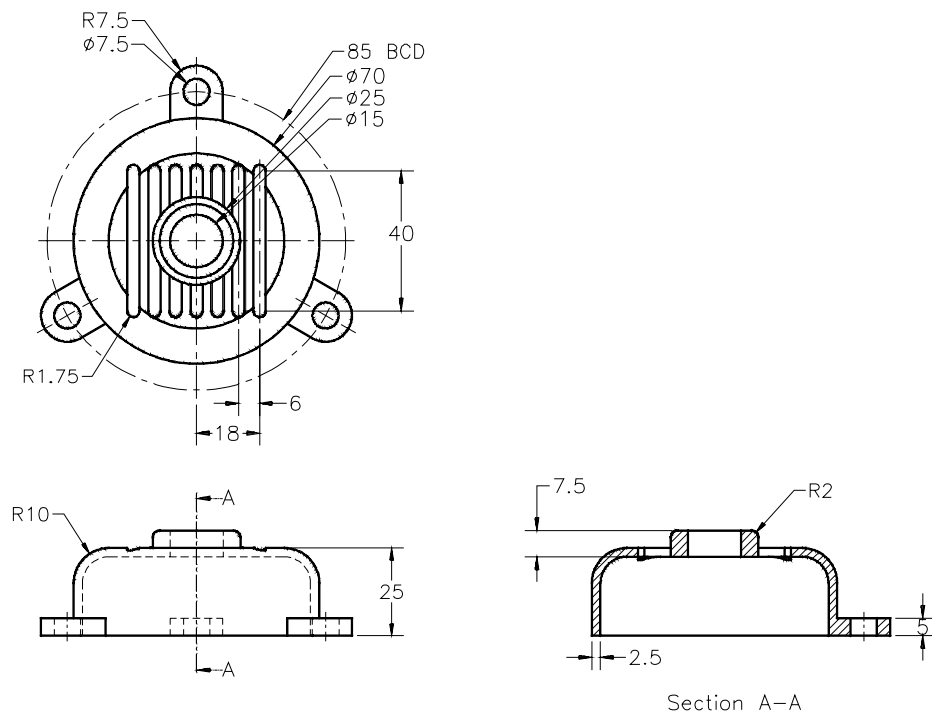


Figure 9-64 Views and dimensions of the model for Tutorial 3

The steps to be followed to complete this tutorial are listed next.

- Create a base feature of the model by revolving the sketch drawn on the **Front Plane**, refer to Figures 9-65 and 9-66.
- Shell the model using the **Shell** tool, refer to Figure 9-67.
- Draw the sketch on the **Top Plane** and extrude it to the given distance, refer to Figure 9-68.
- Pattern the extrude feature using the **Circular Pattern** tool, refer to Figure 9-69.
- Edit the circular pattern, refer to Figure 9-70.
- Create a cut feature on the top planar face of the base feature, refer to Figure 9-71.
- Pattern the cut feature using the **Linear Pattern** tool, refer to Figure 9-72.
- Unsuppress the suppressed features and create the remaining features of the model, refer to Figures 9-73 and 9-74.

Creating the Base Feature

First, you need to create the base feature of the model by revolving the sketch created on the **Front Plane**.

- Start a new SolidWorks Part document using the **New SolidWorks Document** dialog box.
- Invoke the **Revolved Boss/Base** tool and draw the sketch of the base feature on the **Front Plane**. Add the required relations and dimensions to it, as shown in Figure 9-65.
- Exit the sketching environment and create the base feature of the model, as shown in Figure 9-66.

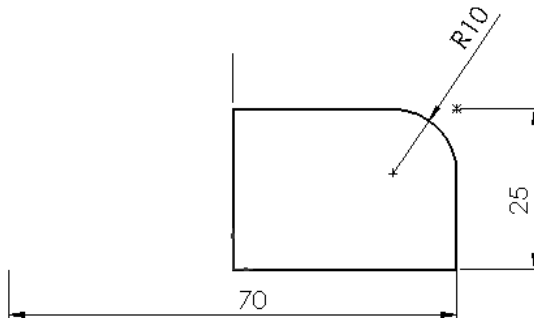


Figure 9-65 Sketch for the base feature

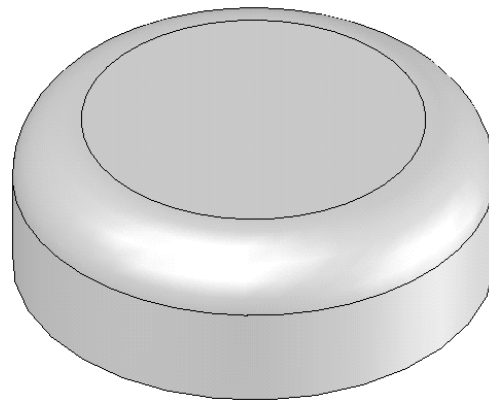


Figure 9-66 Base feature of the model

Shelling the Base Feature

After creating the base feature, you need to shell the model using the **Shell** tool. You will also remove the bottom face of the base feature, leaving behind a thin walled model.

- Invoke the **Shell1 PropertyManager** and set the value of the **Thickness** spinner to **2.5**.
- Rotate the model and select its bottom face to remove it.

3. Choose the **OK** button from the **Shell1 PropertyManager**. The model, after adding the shell feature, is displayed in Figure 9-67.

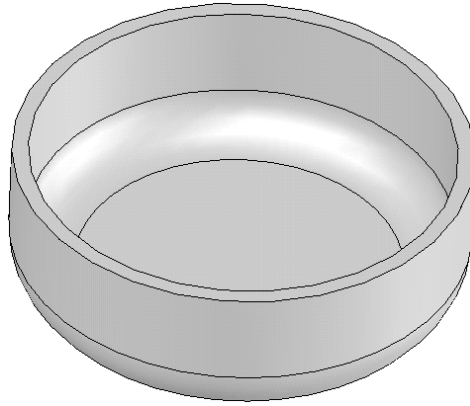


Figure 9-67 Shell feature added to the model

Creating the Third Feature

After adding the shell feature to the model, you need to create its third feature, which is an extruded feature. The sketch for this feature will be drawn on the **Top Plane**.

1. Invoke the **Extruded Boss/Base** tool and select the **Top Plane** as the sketching plane.
2. Orient the model in the top view.
3. Draw the sketch of the third feature and add the required relations and dimensions to the sketch, as shown in Figure 9-68.

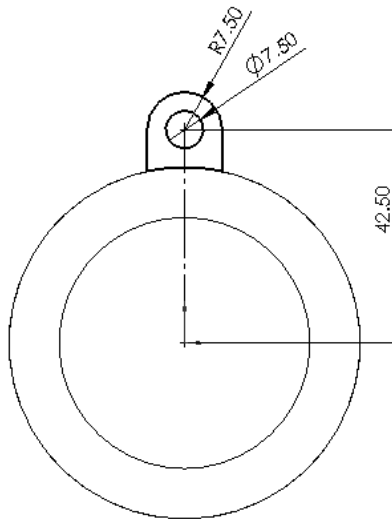


Figure 9-68 Sketch of the third feature

4. Exit the sketching environment and extrude the sketch to a depth of 5 mm.

Patterning the Third Feature

You need to pattern the third feature after creating it. This feature will be patterned using the **Circular Pattern** tool. Before proceeding further you need to display the temporary axes. The temporary axis of the base feature will be selected as the central axis to create the circular pattern.

1. Choose **View > Temporary Axes** from the menu bar to display the temporary axes.
2. Invoke the **Circular Pattern PropertyManager** and select the temporary axis of the base feature as the central axis.
3. Select the third feature, created earlier from the drawing area, if not selected in the **Features to Pattern** selection box. The preview of the pattern feature is displayed in the drawing area.
4. Set the value of the **Number of Instances** spinner to **5** and choose **OK** from the **Circular Pattern PropertyManager**.
5. Choose **View > Temporary Axes** from the menu bar to remove the temporary axes from the current display.

The model, after creating the pattern feature, is displayed in Figure 9-69.

Editing the Pattern Feature

The pattern created is not the same as required, refer to Figure 9-63. As a result, you need to edit it.

1. Select **CirPattern1** from the **FeatureManager Design Tree** or any one of the pattern instances other than the parent instance from the drawing area. Right-click and choose the **Edit Feature** option from the shortcut menu.

The **CirPattern1 PropertyManager** is displayed. Presently, the number of instances in the pattern feature is 5, but the required number of instances is 3. Therefore, you will edit the number of instances.

2. Set the value of the **Number of Instances** spinner to **3** and choose the **OK** button from the **FeatureManager Design Tree**.

The model, after editing the features, is shown in Figure 9-70.

Suppressing the Features

As discussed earlier, sometimes you may need to suppress some features to reduce the complications in the model. The suppressed features are actually not deleted; their display is turned off. When you suppress a feature, the child features associated with that feature are also suppressed.

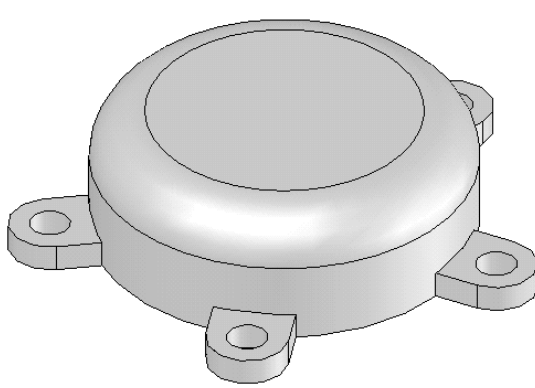


Figure 9-69 Pattern feature added to the model

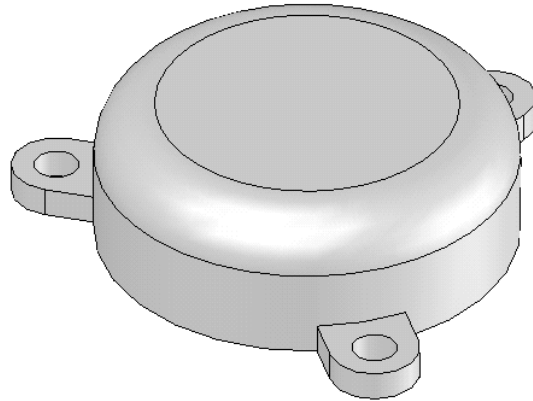


Figure 9-70 The edited pattern feature

1. Select the **Extrude1** feature, which is the third feature of the model, from the **FeatureManager Design Tree**. Right-click and choose the **Suppress** option from the shortcut menu.

The circular pattern feature is the child feature of the extrude feature. Therefore, it is also suppressed. Both features are not displayed in the drawing area. The **Extrude1** and the **CirPattern1** features are displayed in gray in the **FeatureManager Design Tree**, indicating that both of them are suppressed.

Creating the Cut Feature

The next feature that you are going to create is a cut feature. The sketch for this feature will be drawn on the top planar face of the base feature.

1. Invoke the **Extruded Cut** tool and select the top planar face of the base feature as the sketching plane.
2. Draw the sketch of the cut feature and add the required relations and dimensions to the sketch, as shown in Figure 9-71.
3. Exit the sketching environment and specify the end condition as **Through All** from the **Cut-Extrude PropertyManager**.
4. Choose the **OK** button from the **Cut-Extrude PropertyManager**.
5. Now, using the **Linear Pattern** tool, create a linear pattern of the cut feature. You can select the dimension 18 as the directional reference. The model, after creating the linear pattern, is shown in Figure 9-72.
6. Create the other features of the model. For dimensions, refer to Figure 9-64. The model, after creating the other features, is shown in Figure 9-73.

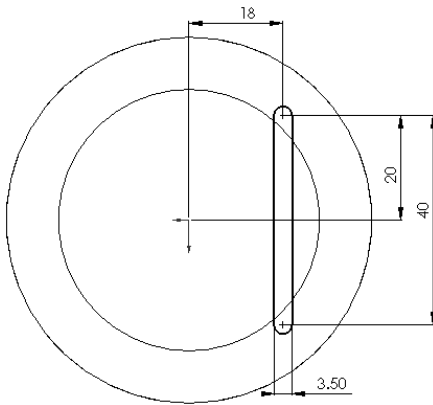


Figure 9-71 Sketch for the cut feature

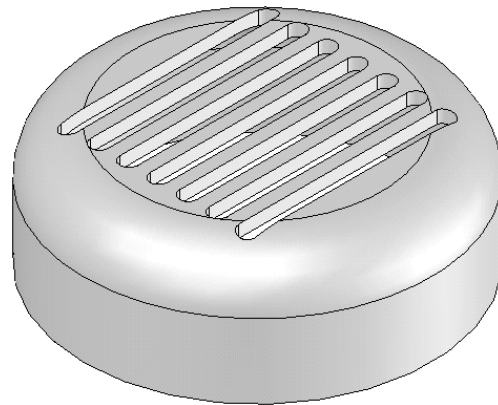


Figure 9-72 Model after patterning the cut feature

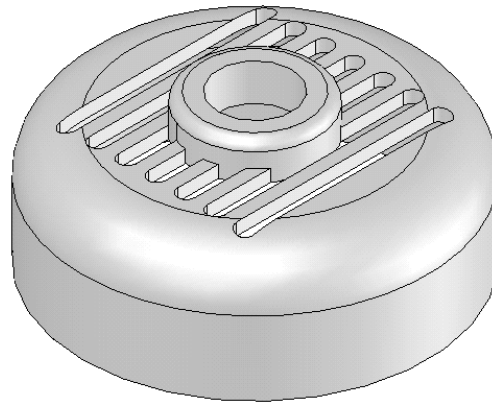


Figure 9-73 Model after creating other features.

Unsuppressing the Features

After completing the model, you need to unsuppress the features that you suppressed earlier.

1. Press and hold down the CTRL key on the keyboard, and select all the suppressed features from the **FeatureManager Design Tree**.



Note

On selecting only the parent suppressed feature and unsuppressing it, the child feature will not be unsuppressed. Therefore, you have to select the parent feature and the suppressed child features.

*Instead of selecting all the parent and child suppressed features from the **FeatureManager Design Tree**, select only the parent feature, and choose **Edit > Unsuppress with Dependents > All Configurations** from the menu bar. You will learn more about the configurations in the later chapters.*

On unsuppressing the child feature, the parent feature will be unsuppressed automatically.

2. Right-click and choose the **Unsuppress** option from the shortcut menu.

The suppressed features will be restored in the model. The final model, after unsuppressing the features, is shown in Figure 9-74.

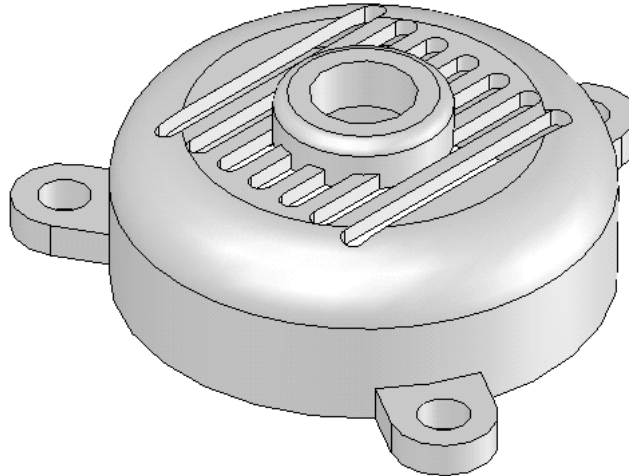


Figure 9-74 The final model

Saving the Model

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below.

`\\My Documents\\SolidWorks\\c09\\c09tut3.sldprt`

2. Choose **File > Close** from the menu bar to close the document.

SELF-EVALUATION TEST

Answer the following questions and then compare your answers with those given at the end of this chapter:

1. You cannot edit the sketch of a sketched feature. (T/F)
2. The **Edit Feature** option is used to edit any feature. (T/F)
3. You cannot rename the feature in the **FeatureManager Design Tree**. (T/F)
4. You cannot edit the sketch plane of the sketch of a sketched feature. (T/F)
5. You cannot edit the sketches using the **Move/ Size Features** option. (T/F)
6. The _____ dialog box is displayed when you edit a dimension.

7. The process of changing the position of a feature in the **FeatureManager Design Tree** is known as _____.
8. Using _____ **PropertyManager** you can delete the bodies.
9. The _____ **PropertyManager** is used to move or copy the bodies.
10. The _____ dialog box is displayed when there is any error in a feature.

REVIEW QUESTIONS

Answer the following questions:

1. The _____ **PropertyManager** is invoked to delete a body.
2. You can rotate a body using _____ **PropertyManager**.
3. The _____ key is used to copy a feature or a sketch.
4. The _____ key is used to cut a feature or a sketch.
5. When _____ tool is active, the preview of the feature is displayed in temporary graphics while editing the sketches.
6. The _____ **PropertyManager** is displayed to edit the sketch plane of a sketch.
7. To add the selected feature in a new folder, choose **Add to New Folder** from the shortcut menu. (T/F)
8. For reordering the features, select the feature in the **FeatureManager Design Tree** and drag it to the required position. (T/F)
9. The **Modify** dialog box is invoked, using a single click on the dimension to modify it. (T/F)
10. If you want to modify the sketch by dragging the fully or partially defined sketch, the **Override Dims on Drag/Move** option should be selected. (T/F)

EXERCISES

Exercise 1

Create the model that is sectioned and shown in Figure 9-75. The other views and dimensions of the model are also given in the same figure. The complete model is shown in Figure 9-76.

(Expected time: 45 min)

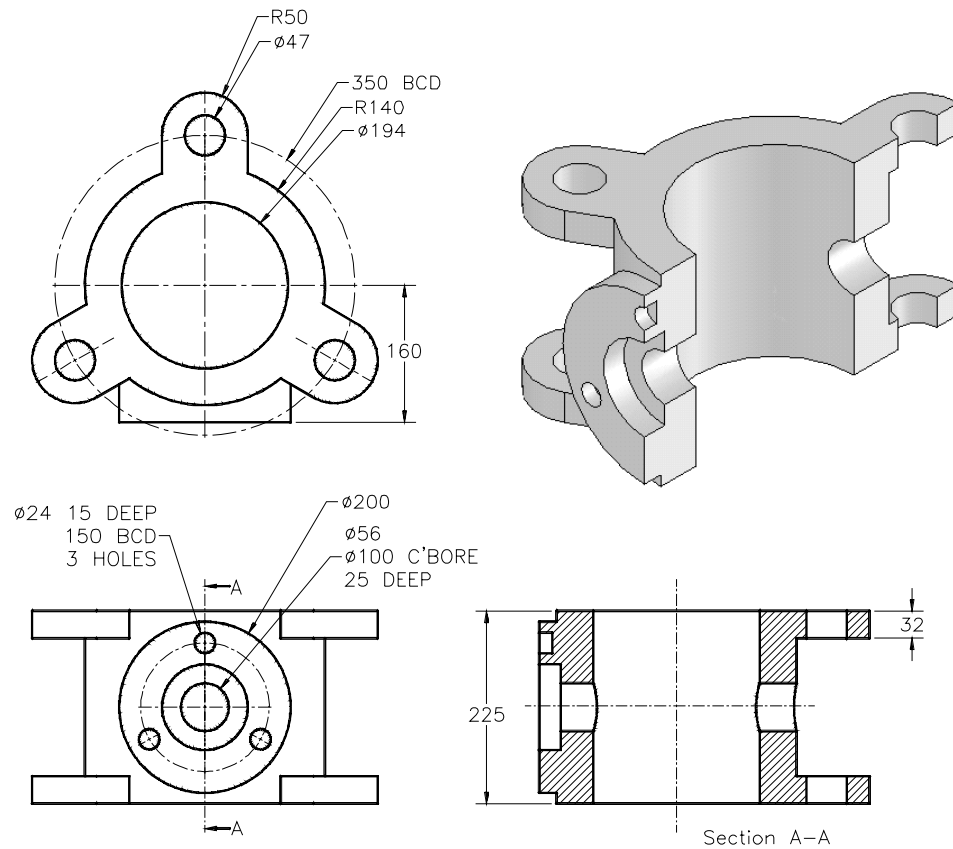


Figure 9-75 Views and dimensions of the model for Exercise 1

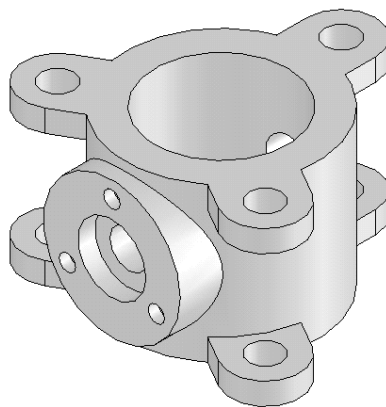


Figure 9-76 Model for Exercise 1

Exercise 2

Create the model shown in Figure 9-77. Its dimensions are shown in Figure 9-78.

(Expected time: 30 min)

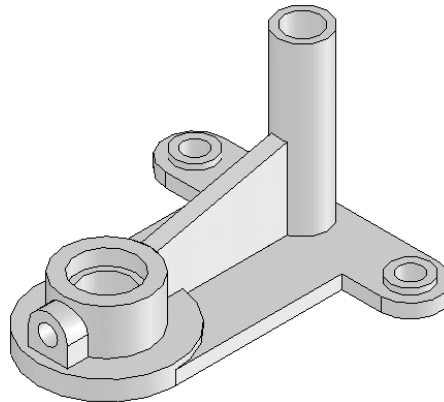


Figure 9-77 Model for Exercise 2

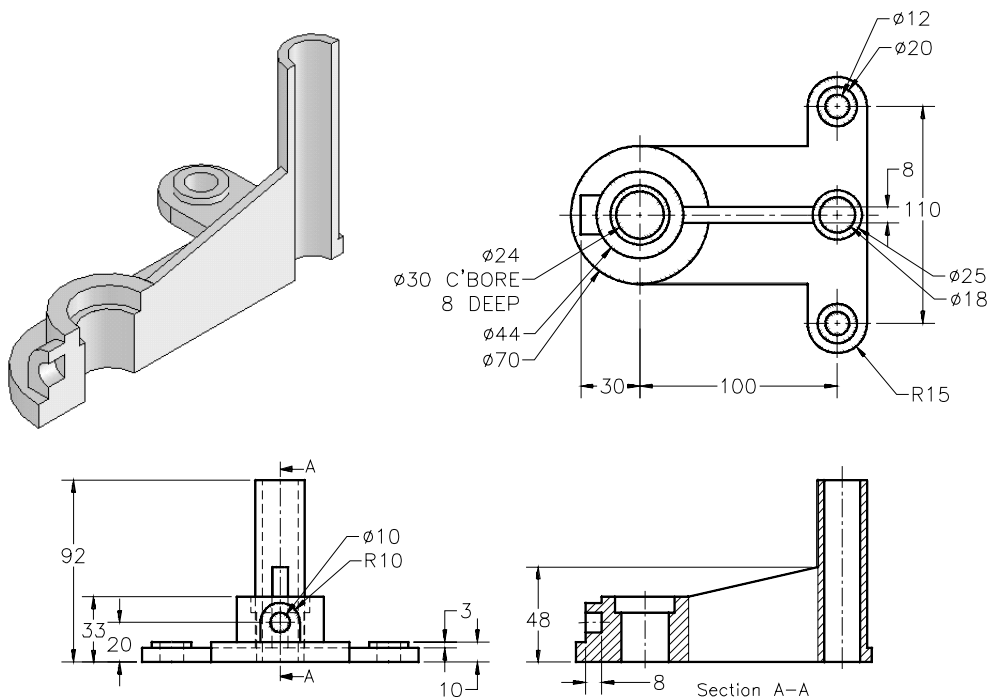


Figure 9-78 Views and dimensions of the model for Exercise 2

Answers to Self-Evaluation Test

1. F, 2. T, 3. F, 4. F, 5. F, 6. Modify, 7. Reordering, 8. Delete Body, 9. Move/Copy Body, 10. Rebuild Errors