



Chapter 8

Editing Features

Learning Objectives

After completing this chapter you will be able to:

- *Edit the Features.*
- *Edit the Sketches of the Sketched Feature.*
- *Edit the Sketch Plane.*
- *Edit using the Move/Size Features option.*
- *Cut, Copy, and Paste the Features and Sketches.*
- *Copy Features using the Drag and Drop Method.*
- *Delete the Features.*
- *Delete the Bodies.*
- *Suppress and Unsuppress the Features.*
- *Move/Copy Bodies.*
- *Reorder the Features.*
- *Roll Back the Model.*
- *Rename the Features.*
- *Create the Folders.*
- *Use What's Wrong? Functionality.*

EDITING THE FEATURES OF THE MODEL

Editing is one of the most important aspect of the product design cycle. Almost all the designs require editing either during their creation or after they are created. 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 8-1 shows a base plate with some holes.

Now, if you have to replace the four inner holes with four counterbore holes, all you need to do is to use one editing operation. Using the editing operation, you will change the drilled holes to the counterbore holes. For editing the holes, you have to select the hole feature and right-click to invoke the shortcut menu. Then you will have to choose the **Edit Definition** option from the shortcut menu to invoke the **Hole Definition** dialog box. Finally you have to set the new parameters and end feature creation. The drilled holes will be automatically replaced by the counterbore holes. Figure 8-2 shows the base plate with drilled holes modified to the counterbore holes.

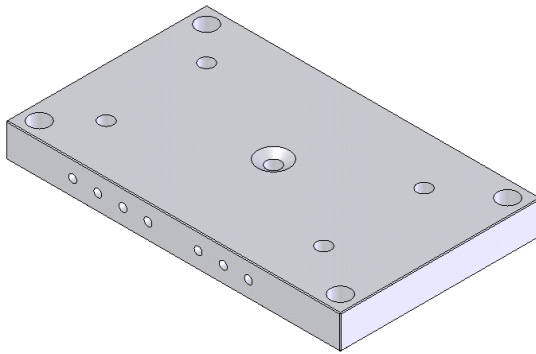


Figure 8-1 Base plate with drilled holes

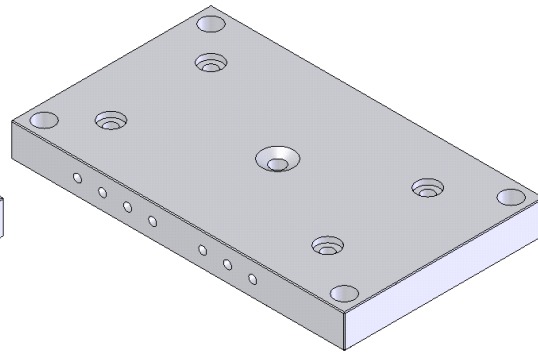
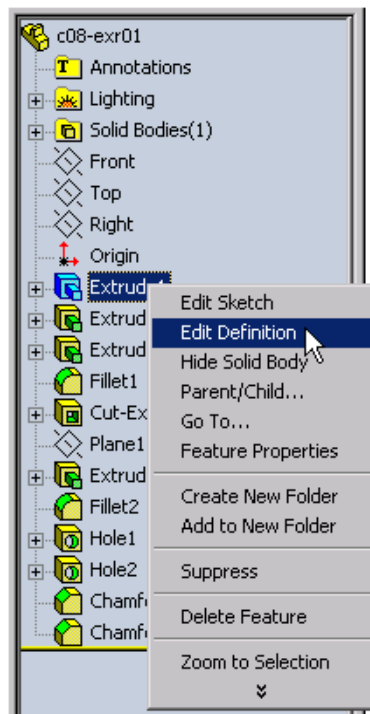


Figure 8-2 Modified base plate

Similarly, you can also edit reference geometry and the sketches of the sketched features. The feature created using the reference geometry also modify 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 resultant feature is automatically modified. In SolidWorks, you can perform editing tasks using various methods. All the methods of editing are discussed as under.

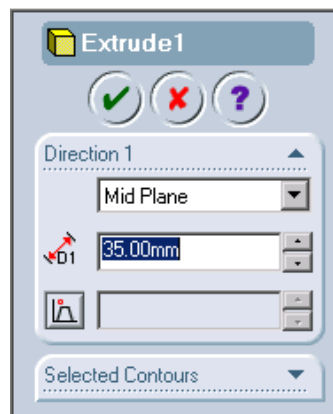
Editing Using Edit Definition Option

The editing using the **Edit Definition** option is the most commonly used method of editing in SolidWorks. To edit any feature of the model using this option, select that feature either from the **FeatureManager Design Tree** or from the drawing area. After selecting the feature, right-click to invoke the shortcut menu and choose the **Edit Definition** option from the shortcut menu as shown in Figure 8-3. Depending on the feature selected, the **PropertyManager** or dialog box will be invoked and you can modify the parameters of that feature. The



*Figure 8-3 Selecting the **Edit Definition** option from the shortcut menu*

PropertyManager will also have the sequence number of the feature. The **Extrude PropertyManager** is displayed in Figure 8-4 with sequence number 1. After editing the parameters, choose the **OK** button to complete feature creation. The feature will be modified automatically.



*Figure 8-4 The **Extrude PropertyManager***

Editing the Sketches of the Sketched Features

In SolidWorks, you can also edit the sketches of the sketched entities using the **Edit Sketch** option. To edit the sketch of a sketched feature select the feature either from the **FeatureManager Design Tree** or from the drawing area and right-click to invoke the shortcut menu. Choose the **Edit Sketch** option from the shortcut menu. When you choose this option, you will enter the sketching environment. Using the standard sketching tools, edit the sketch of the sketched feature. After editing the sketch, exit the sketching environment. To exit the sketching environment you can also select the **Rebuild** button from the **Standard** toolbar or choose **View > Rebuild** from the menu bar. Using this you will exit the sketching environment and the model will also be rebuilt. You can also choose CTRL+B to rebuild the model.



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

Changing the Sketch Plane of the 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 the left of that feature 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 the shortcut menu. Figure 8-5 shows the **Edit Sketch Plane** option being chosen from the shortcut menu. The **Sketch Plane PropertyManager** will be displayed as shown in Figure 8-6. The name of the current sketch plane is displayed in the **Sketch Plane/Face** display area. Now, select any other plane or face as the sketching plane. After selecting the new sketching plane, choose the **OK** button from 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 **Rebuild Errors** dialog box will be displayed. You, will have to undo the last step using the **Undo** button from the **Standard** toolbar, invoke the **Sketch Plane PropertyManager** again, and then select the appropriate plane. You will learn more about the **Rebuild Errors** dialog box later in this chapter.

Editing by Double-Clicking the Entities and Features

You can also edit a feature, a 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 an extrude feature, all the dimensions of the feature and the sketch of the feature are displayed in the drawing area. Remember that all the dimensions of the sketch will be displayed in black color and all the dimensions of the feature will be displayed in blue color. Move the cursor on the dimension to be modified, the dimension will be displayed enclosed in a red box. Double-click the dimension to invoke the **Modify** dialog box. Set the new value in the **Modify** dialog box and press the ENTER key from 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 have to rebuild the model using the **Rebuild** option. To rebuild the model, choose the **Rebuild** button from the **Standard** toolbar or choose **Edit > Rebuild** from the menu bar. You can also rebuild the model using CTRL+B from the keyboard.

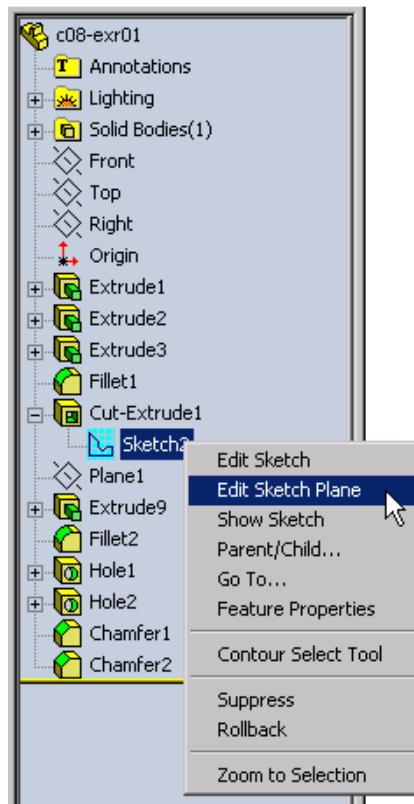


Figure 8-5 The *Edit Sketch Plane* option being selected

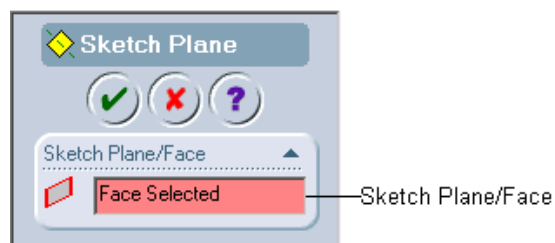


Figure 8-6 The *Sketch Plane PropertyManager*

Editing Using the Move/Size Features

Toolbar: Features > Move/Size Features



Using this option you can dynamically modify the feature and the sketch of the sketched feature without invoking the sketching environment. To edit the feature or sketch using this option, choose the **Move/Size Features** button from the **Features** toolbar. You will notice that the **Move/Size Features** button is chosen in the **Features** toolbar. 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 the sketch of that feature will also be highlighted in

green. You will be provided with the resize handle, rotate handle, and the move handle. Figure 8-7 shows the move handle, resize handle, and the rotate handle provided for the selected feature.

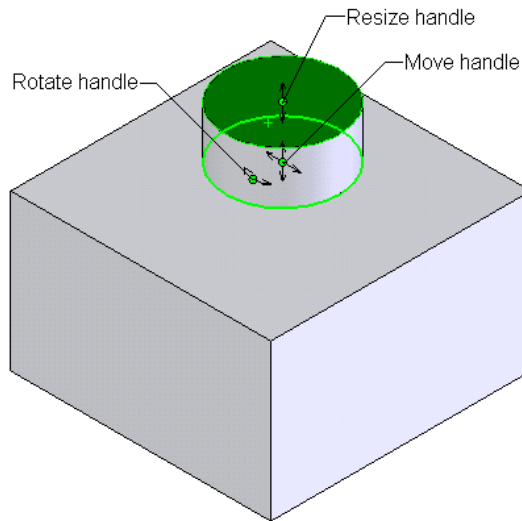


Figure 8-7 Feature selected with **Move/Size Features** button selected

To resize the feature, move the cursor to the resize handle; the select cursor will be replaced by the resize cursor and the name of the resize cursor will be 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 the feature. You will notice that the feature will be dynamically resized. Figure 8-8 shows the cursor being dragged to resize the feature. Figure 8-9 shows the resultant modified feature.

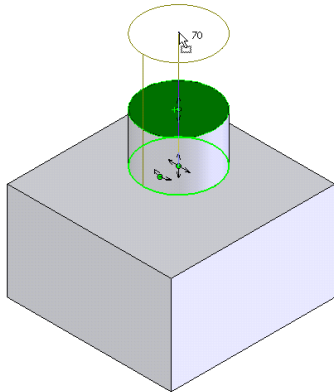


Figure 8-8 Dragging the resize handle to resize the feature

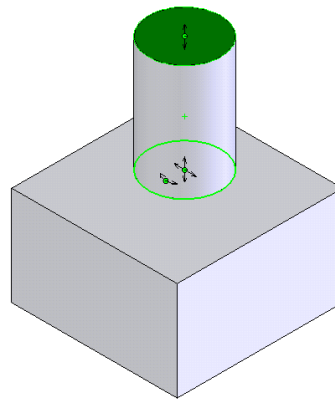


Figure 8-9 Resultant modified feature

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 and 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°. Release the left mouse button after rotating the feature to a specified location.

**Note**

If you rotate a sketched feature in which the sketch of the feature is fully defined or partially defined using relations and dimensions, the **Move Confirmation** dialog box will be displayed. The **Move Confirmation** dialog box is displayed in Figure 8-10. This dialog box informs you that the external constraints in the feature are being moved, do you want to delete those constraints or keep the constraints by recalculating or make them dangling. Those relations or dimensions that do not find the external reference after the placement are made dangling.

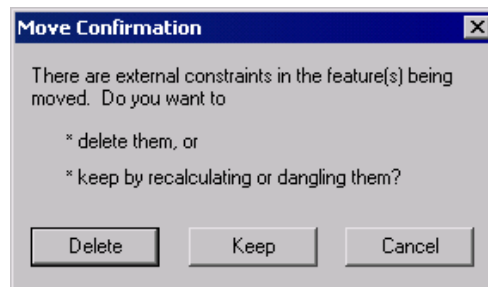


Figure 8-10 The **Move Confirmation** dialog box

After rotating the feature, if the **Move Confirmation** dialog box is invoked then you have to choose either the **Delete** button or the **Keep** button, based on the geometric and dimensional conditions. Figure 8-11 shows the preview of rotating the feature. Figure 8-12 shows the resultant rotated feature.

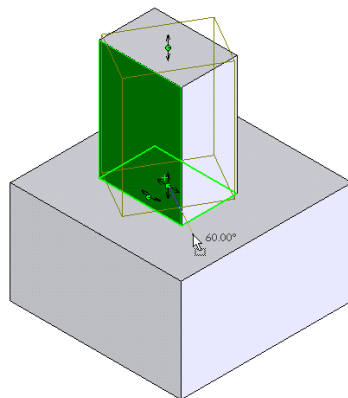


Figure 8-11 Preview of feature being rotated

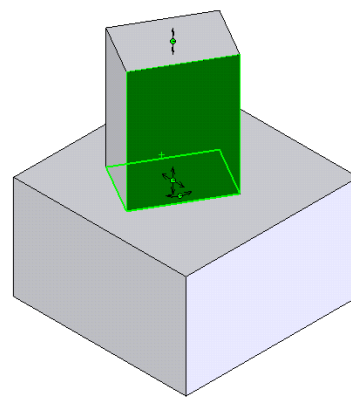


Figure 8-12 Resultant rotated feature



Tip. When you resize, move, or rotate a feature using the **Move/Size Features** tool, the feature(s) that has 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 you modify the parent 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 and drag the cursor to move the feature. Release the left mouse button after moving the feature to the required position. If the dimensions or relation of the feature are not able to be defined after moving the feature, the **Move Confirmation** dialog box will be displayed. Choose the options available in this dialog box. Using this option, you can also change the placement plane of the feature or the sketch plane of the feature. Figure 8-13 shows the feature being moved to another face. Figure 8-14 shows the resultant moved feature.

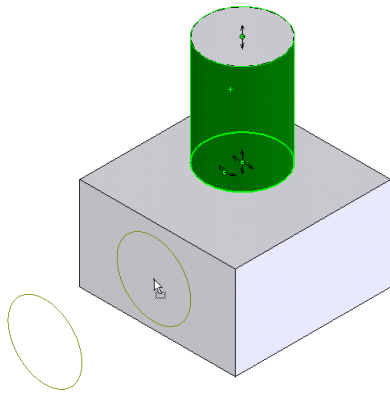


Figure 8-13 The feature being moved

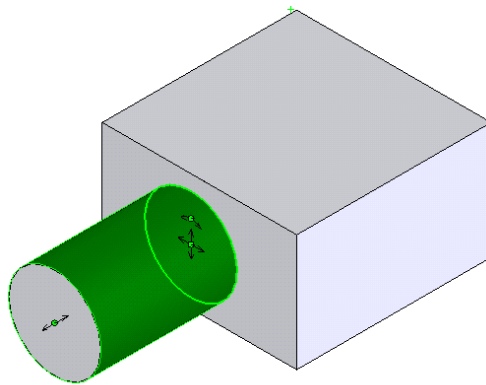


Figure 8-14 Resultant moved feature

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. A vertical line 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 8-15 shows the preview of the sketch being modified. Figure 8-16 shows the resultant modified model.



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

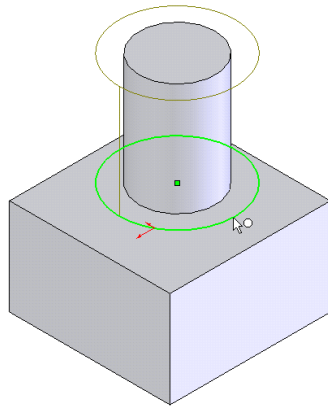


Figure 8-15 Preview of the sketch being modified

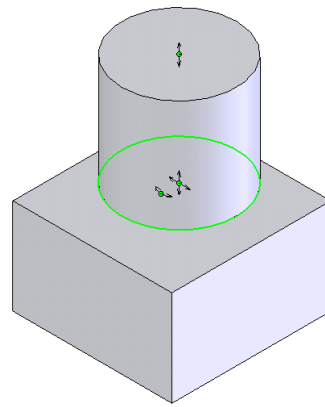


Figure 8-16 Resultant modified model

Editing the 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 have to modify will be displayed in transparent yellow temporary graphics as shown in Figure 8-17.

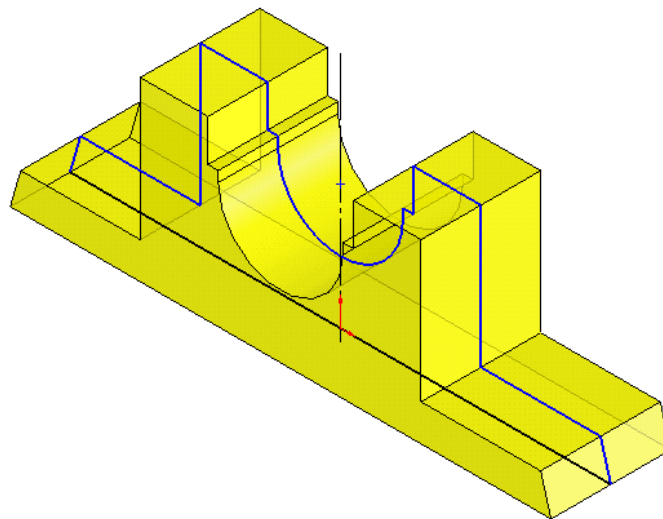


Figure 8-17 Preview of the sketch being modified

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 sketcher environment. Therefore, you can have a better understanding of the model, that how it will be displayed after modifying



Tip. If you want to modify the sketch by dragging and the sketch is fully or partially defined using dimensions, 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.

the sketch. Note that the temporary graphic will show the preview of the feature created by the section you add only if you drag one of the entity in the sketch. Otherwise, the new addition will not be display 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 sketcher environment. If the sketch of the model is dimensioned then you can modify the sketch by modifying the dimension. Figure 8-18 shows two circles added to the model and the length of the inclined line being increased. Figure 8-19 shows the resultant modified model.

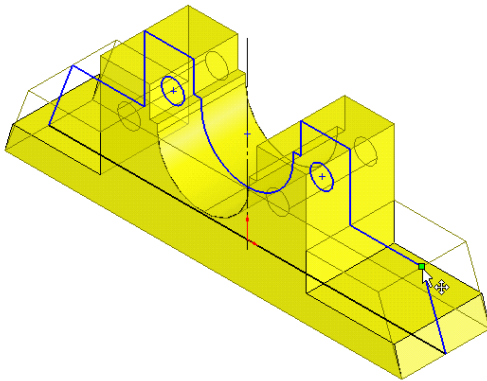


Figure 8-18 Sketch being modified

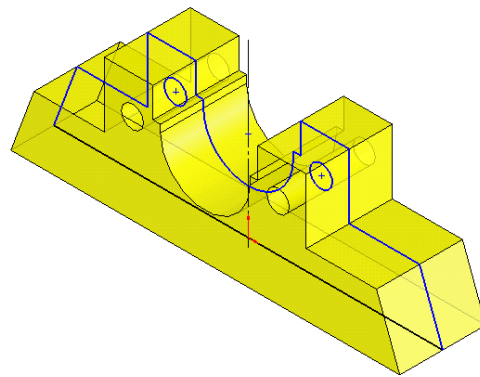


Figure 8-19 Resultant modified model

Editing the 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. 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 item, choose **Edit > Cut** from the menu bar or use the shortcut key, CTRL+X. The **Delete Confirmation** dialog box is displayed because when you cut the selected item, it is deleted from the document. Choose **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 from the keyboard. Sometimes the **Copy Confirmation** dialog box is displayed as shown in Figure 8-20. In that dialog box, you are prompted either 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.

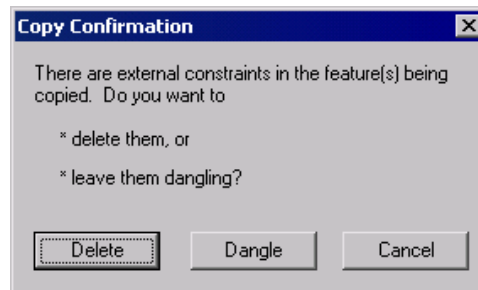


Figure 8-20 The *Copy Confirmation* dialog box



Tip. For pasting a selected sketch, you have 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 or edges as the references.

If you copy and paste an item, the selected item will remain at its position and a copy of that item 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 from 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 from the keyboard to paste. You can paste the selected item any number of times. If you select another item and copy on the clipboard, the last copied item will be deleted from the memory of the clipboard.

Cut, Copy, and Paste 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 have to copy a sketch created in the current document and paste it in a new document. You just need to select the sketch and use CTRL+C from 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 from 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 drag drop functionality of windows to copy and paste the item within the document. Press and hold down the CTRL key from the keyboard and 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 on that location. 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 8-21 shows the feature being dragged. Figure 8-22 shows the resultant pasted feature.

Dragging and Dropping Features from One Document to 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

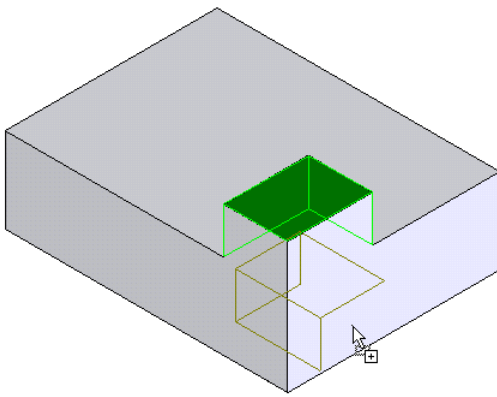


Figure 8-21 Feature being dragged

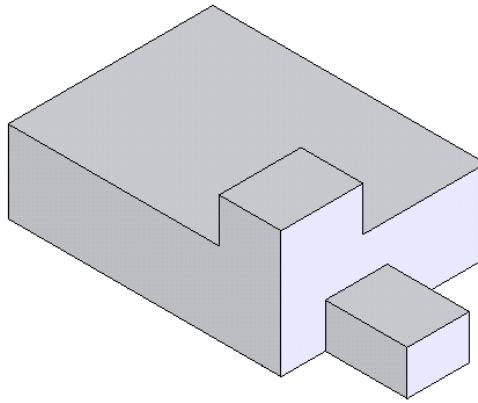


Figure 8-22 Resultant pasted feature

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 from the keyboard, select and drag the feature or sketch to the other document, and place it on the required reference. Figure 8-23 shows the fillet feature being dragged to be applied on the edge of the model in the second document.

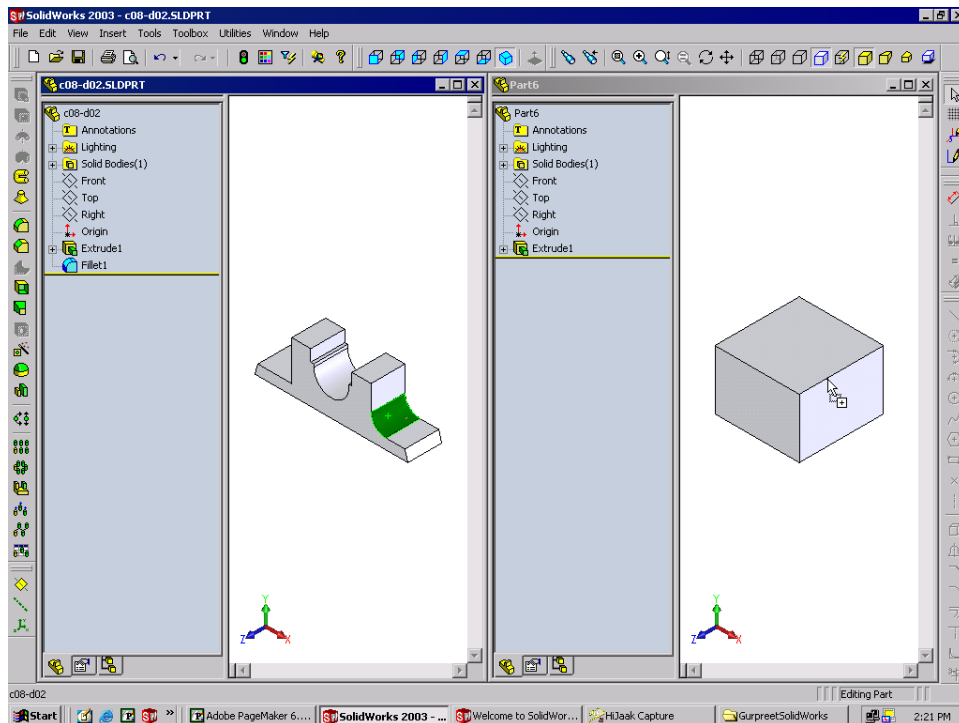


Figure 8-23 Fillet feature being dragged to be pasted in the second document

Deleting the Features

You can delete the unwanted features from the model by selecting the feature either from the **FeatureManager Design Tree** or from the drawing area. After selecting the feature to be deleted, choose the **DELETE** key from the keyboard or right-click to invoke the shortcut menu and choose the **Delete Feature** option. If you delete a sketched feature or a feature that acts as the parent feature for the other feature, the **Confirm Delete** dialog box is displayed as shown in Figure 8-24. This dialog box informs you that all the dependent features to this parent feature will also be deleted. Choose the **Yes** button if you want to delete all the 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. When you delete a feature, the absorbed feature related to that feature will not be deleted. For example, if you delete a boss feature the sketch feature is not deleted. If you need to delete the absorbed feature when you delete the parent feature select the **Also delete absorbed features** check box from the **Confirm Delete** dialog box.

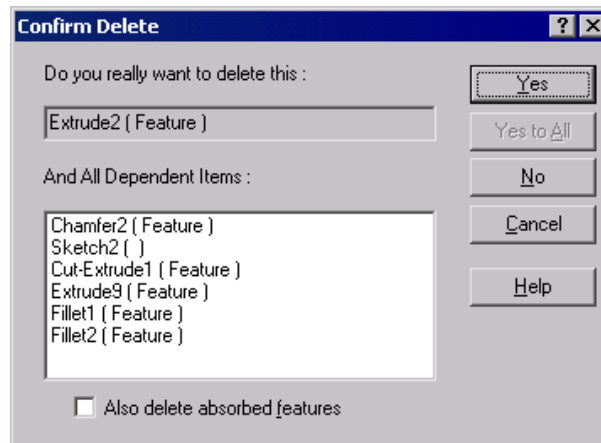


Figure 8-24 The *Confirm Delete* dialog box

Deleting the Bodies

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



As discussed earlier, the multi-body 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. You will learn more about surface bodies in later chapters. To delete a body, choose the **Delete Solid/Surface** button from the menu bar or choose **Insert > Features > Delete Body** from the menu bar. The **Delete Body PropertyManager** is invoked as shown in Figure 8-25. You are prompted to select the solid and/or surface bodies to be deleted. Select the body or bodies to be deleted from the drawing area or from the **Solid Bodies** folder available in the **FeatureManager Design Tree** flyout. The selected body is displayed in green and the edges of the selected body are highlighted in yellow. The name of the selected body is displayed in



Figure 8-25 The *Delete Body PropertyManager*

the **Bodies to Delete** display area. 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 Body** option from the shortcut menu. The **Delete Body PropertyManager** will be displayed. Choose the **OK** button from the **Delete Body PropertyManager** to delete the body.

Suppressing the Features

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



Sometimes, you do not want a feature to be displayed in the model or in the drawing views of that model. 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 anytime resume the feature by unsuppressing it. When you suppress a feature, the features that are dependent on the suppressed feature 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** toolbar or right-click and choose the **Suppress** option from the shortcut menu. The suppressed feature will be removed from the model and the icon of the feature will be displayed in gray in the **FeatureManager Design Tree**.



Note

You can work with different configurations in a single document of SolidWorks. Therefore, you have to specify in which configuration you are suppressing a feature. This is specified when you choose the suppress option from the menu bar. You will learn more about configurations later in this book.

Unsuppressing the Features

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



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 **Unsuppressed** button from the **Features** toolbar or choose **Edit > Unsuppressed** from the menu bar. 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 have to unsuppress all the features independently. Resuming the parent feature and the dependent features using a single option is discussed next.

Unsuppressing Features With Dependents

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



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 > This Configuration** from the **Features** toolbar. You will observe that the dependent suppressed features are also unsuppressed.

Hiding the Bodies

While working in the multibody environment, you can also hide the bodies. The hidden body is not displayed in the model and assembly, and drawing views. The bodies that are dependent on the hidden body do not hide. To hide a body, expand the **Solid Bodies** folder available in the **FeatureManager Design Tree** and select the body to hide. Right-click to invoke the shortcut menu and choose the **Hide Solid Body** option from the shortcut menu. 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 show the hidden body.

Move/Copy Bodies

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



In SolidWorks you can also move or copy the bodies. To move or copy the bodies choose the **Move/Copy Bodies** button from the **Features** toolbar or choose **Insert > Features > Move/Copy Bodies** from the menu bar. The **Move/Copy Body PropertyManager** will be displayed as shown in Figure 8-26. The confirmation corner is also displayed in the drawing area. You will observe a filled square at the document origin. The various options available in the **Move/Copy Body PropertyManager** are discussed next.

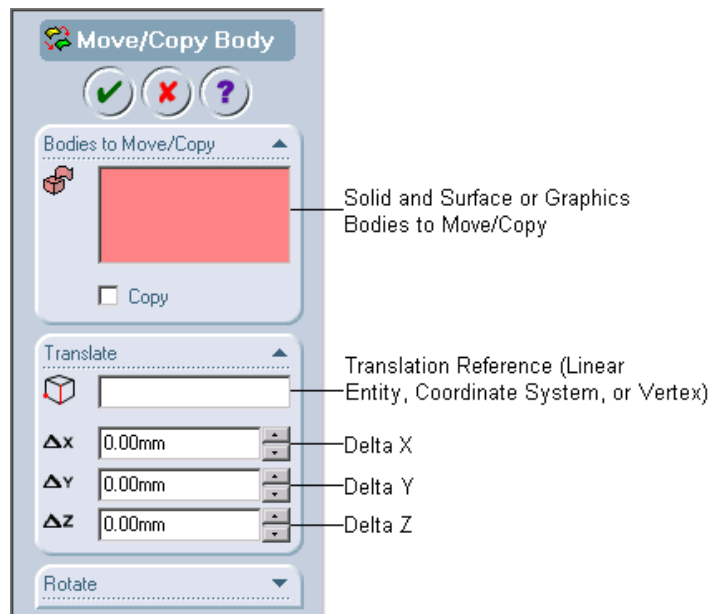


Figure 8-26 The *Move/Copy Body PropertyManager*

Bodies to Move/Copy

The **Bodies to Move/Copy** rollout is used to define the body to copy or move. When you invoke the **Move/Copy Body PropertyManager**, you are prompted to select the bodies to move/copy and set the options. Move the cursor to the body to be selected. The cursor will be replaced by the body selection cursor and the edges of the body are highlighted in red. The name of the body is also displayed in the tooltip. Use the left mouse button to select the body. The selected body is highlighted in green and the edges of the body are displayed in yellow. The name of body is displayed in the **Solid and Surface or Graphics Bodies to Move/Copy** display area. You can also select the body from the **Solid Bodies** folder after invoking the **FeatureManager Design Tree** flyout.

Copy

Since the **Copy** check box is cleared by default, you can move the selected body only after setting the options. 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 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)** display area. 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 translation reference, the **Delta X**, **Delta Y**, and **Delta Z** spinners are replaced by the **To Vertex** display area. The selection mode in the **To Vertex** display area is active. Select the second translation reference. You will observe the preview of the body move between two selected vertices. The placement of the body also depends on the sequence of selection of vertices. Therefore, you have to be very careful while selecting the two vertices. Figure 8-27 shows the sequence for the selection of references. Figure 8-28 shows the resultant copied body.

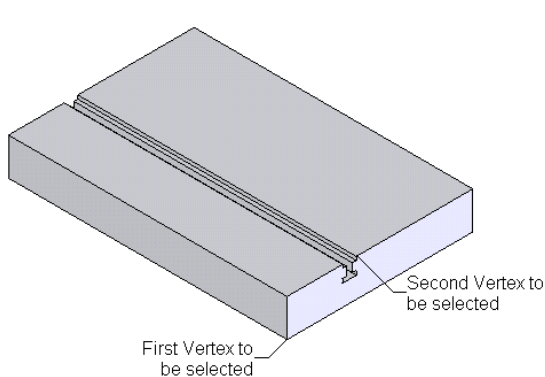


Figure 8-27 Sequence of selection

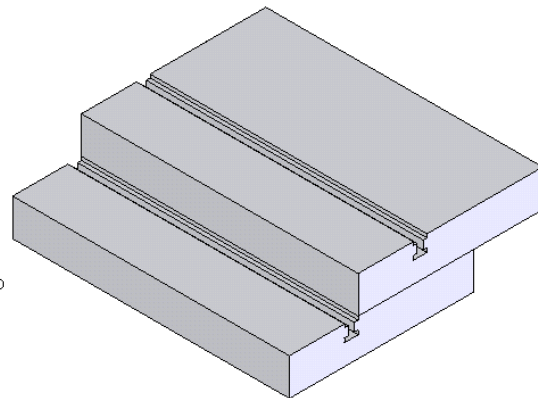


Figure 8-28 Resultant copied feature

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 8-29.

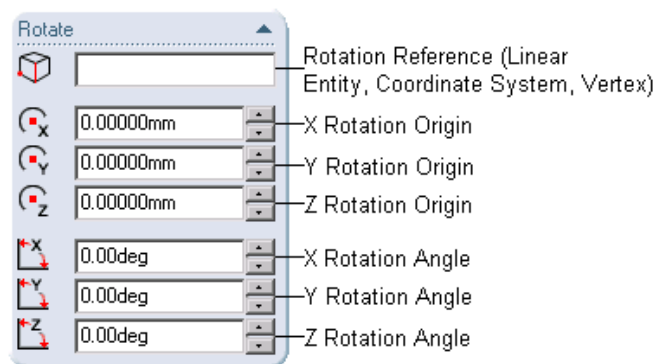


Figure 8-29 The **Rotate** rollout

As discussed earlier, a filled square is placed at the origin when you invoke the **Move/Copy PropertyManager**. This filled square is clearly visible if 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, **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 x-axis. The **Y Rotation Angle** spinner is used to rotate or copy the body along y-axis. The **Z Rotation Angle** spinner is used to rotate or copy the body along z-axis.

To rotate or copy the selected body along an edge, click once in the **Rotation Reference (Linear Entity, Coordinate System, Vertex)** display area 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 have to specify the axis along which you want to rotate the body.

Reordering the 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 the features of the model 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 the feature to the required position. When you drag the feature to reorder, the bend arrow pointer  is displayed, that 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 message dialog box will be displayed as shown in Figure 8-30. Choose **OK** from this dialog box.

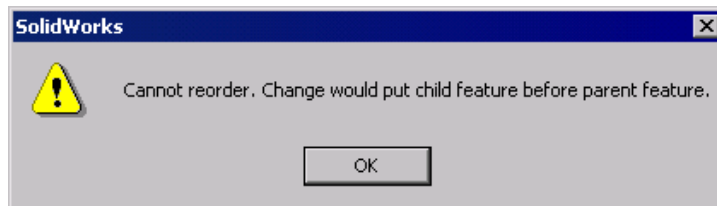


Figure 8-30 The SolidWorks warning message dialog box

Consider a case in which you have created a rectangular block and a pattern of through holes are created on the base feature as shown in Figure 8-31. Now, if you create a shell feature and remove the top face, the front face, and the right face of the model, the model will appear as shown in Figure 8-32. But this was not the desired result. This, is the reason 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 8-33.

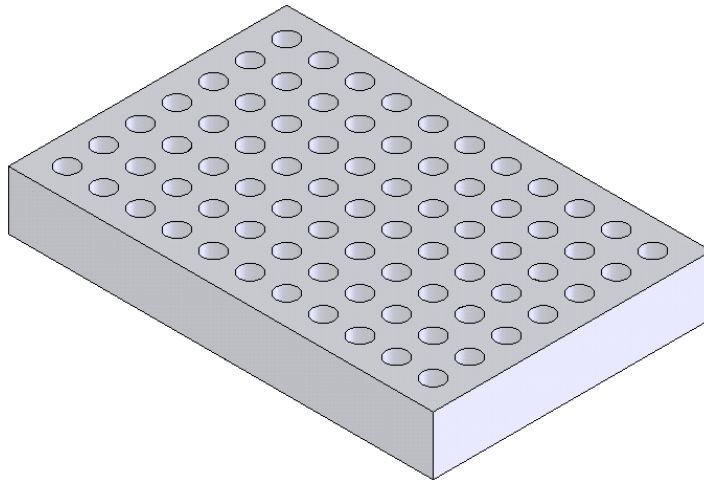


Figure 8-31 Model created creating a pattern of the holes on the base feature

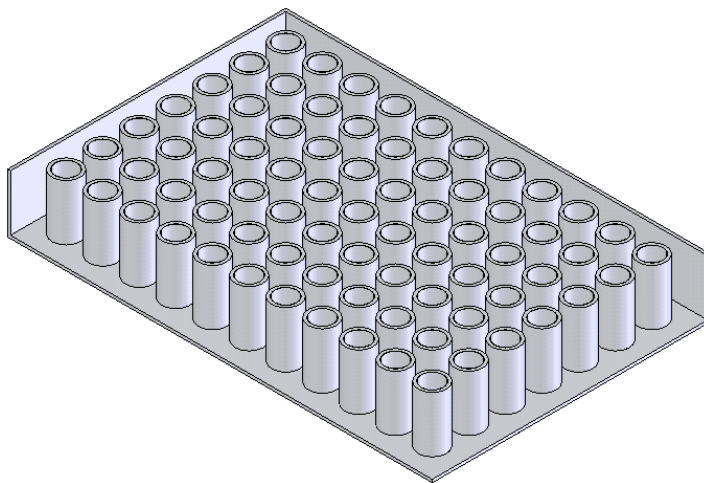


Figure 8-32 Shell feature added to the model

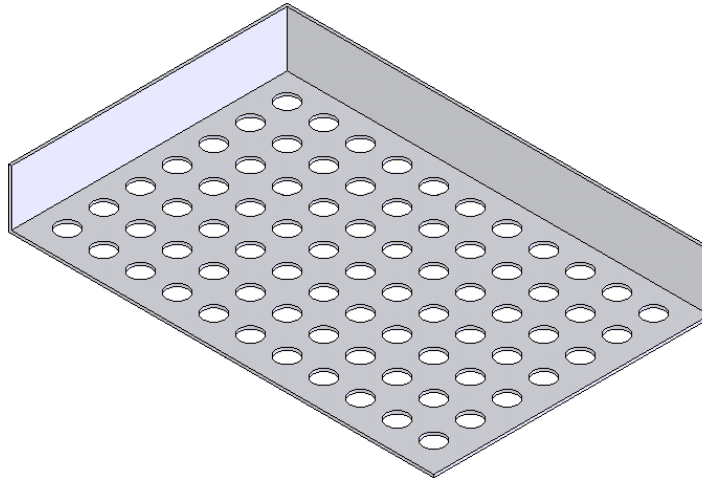


Figure 8-33 Model after reordering the features.

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 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 8-34.

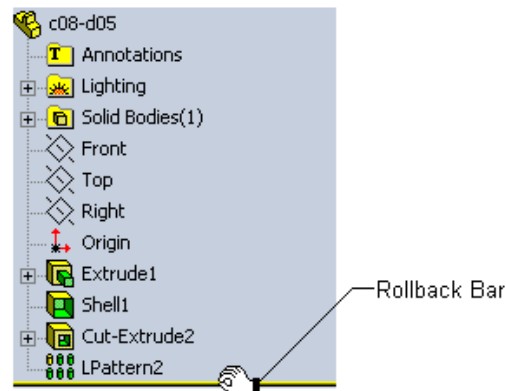



Figure 8-34 The rollback bar of the FeatureManager Design Tree

Using the select tool select the **Rollback Bar**; after selection it will be changed to blue color 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 release the pointer. 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.



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

You can also choose **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.

Renaming the Features

The name of the features is displayed in the **FeatureManager Design Tree**. By default the naming of the features is done according to the sequence in which the features are created. You can also rename the features according to your convenience. To rename a feature, select the feature from the **FeatureManager Design Tree** and click 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 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 and right-click to invoke the shortcut menu. Choose the **Create New Folder** option from the shortcut menu. A new folder is created above the selected feature. 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 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. Select the **Delete** option from the shortcut menu. Using the options in this shortcut menu you can also rollback and suppress the features available in the selected folder.

What's Wrong? Functionality

When you modify a sketch or a feature and rebuild the model, sometimes the model does not rebuild properly because of the geometric errors resulting the modification. Therefore, you are provided with a **Rebuild Errors** dialog box as shown in Figure 8-35. The possible error in the feature are displayed in this dialog box.

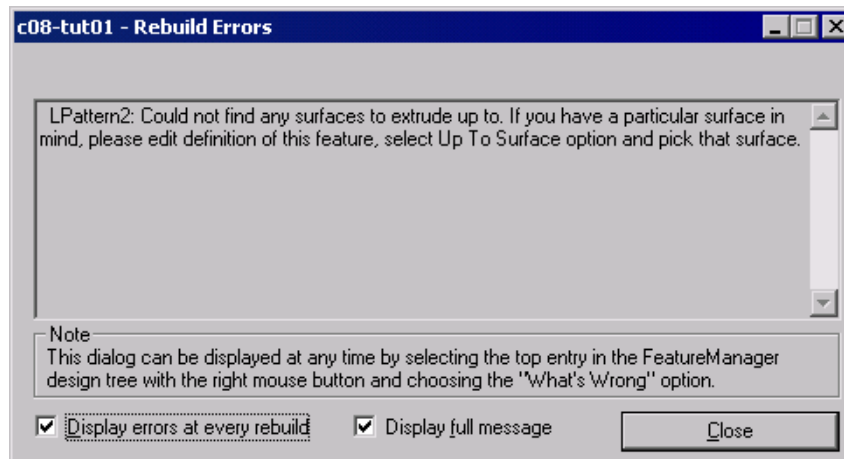


Figure 8-35 The Rebuild Errors dialog box

The **Display errors at every 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. The **Display full message** check box is used to display the complete error message in the **Rebuild Errors** dialog box. After reading the error from this dialog box, choose the **Close** button to exit the dialog box. The errors are also displayed in the **FeatureManager design Tree**. The **FeatureManager Design Tree** with an error in a feature is displayed in Figure 8-36.

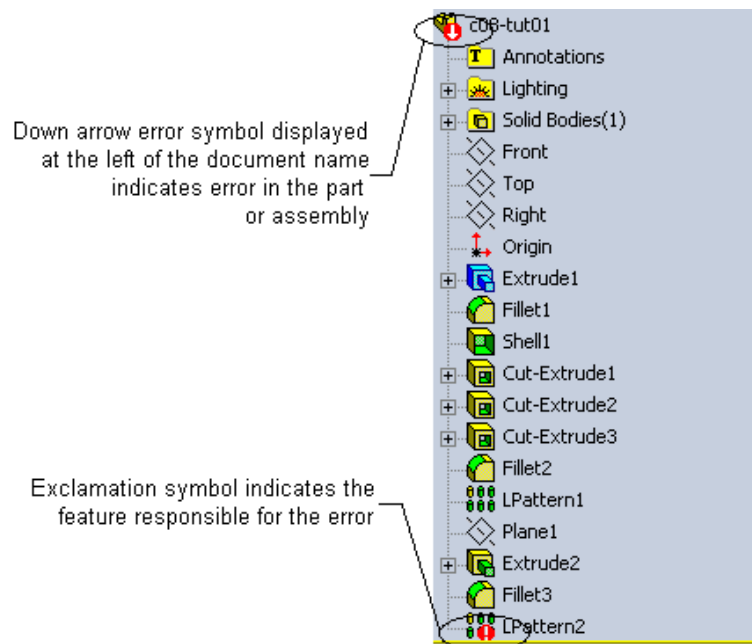


Figure 8-36 The FeatureManager Design Tree with a feature having errors



Tip. You can also invoke the **Rebuild Errors** dialog box by selecting the feature having errors from the **FeatureManager Design Tree** and choose the **What's Wrong?** option from the shortcut menu. To view all the errors in a model, select the name of the document from the top of the **FeatureManager Design Tree** and invoke the **Rebuild Errors** dialog box. All the errors in the model are displayed in the **Rebuild Errors** dialog box.

While creating the features, if there are some geometrical constraints in creating the feature, the **Rebuild Errors** dialog box is displayed.

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 8-37. After creating some of the features of the model you will dynamically modify the model and then undo the modification. The dimensions of the model are shown in Figure 8-38. **(Expected time: 30 min)**

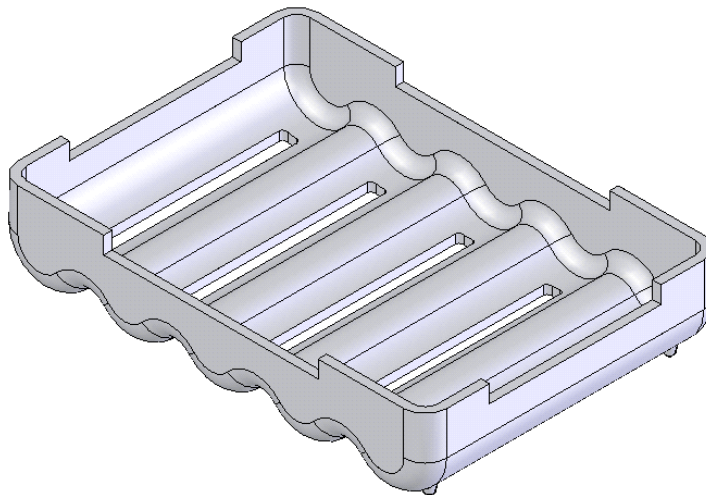


Figure 8-37 Model for Tutorial 1

The steps to be followed to create the model are discussed next:

- Create the base feature of the model by extruding the profile to a given distance, refer to Figures 8-39 and 8-40.
- Add the fillets to the base feature, refer to Figures 8-41 and 8-42.
- Add the shell feature to the model and remove the top face of the base feature, refer to Figures 8-43 and 8-44.
- Dynamically modify the model, refer to Figures 8-45 through 8-47.
- Create the cuts on the sides of the model, refer to Figure 8-48.

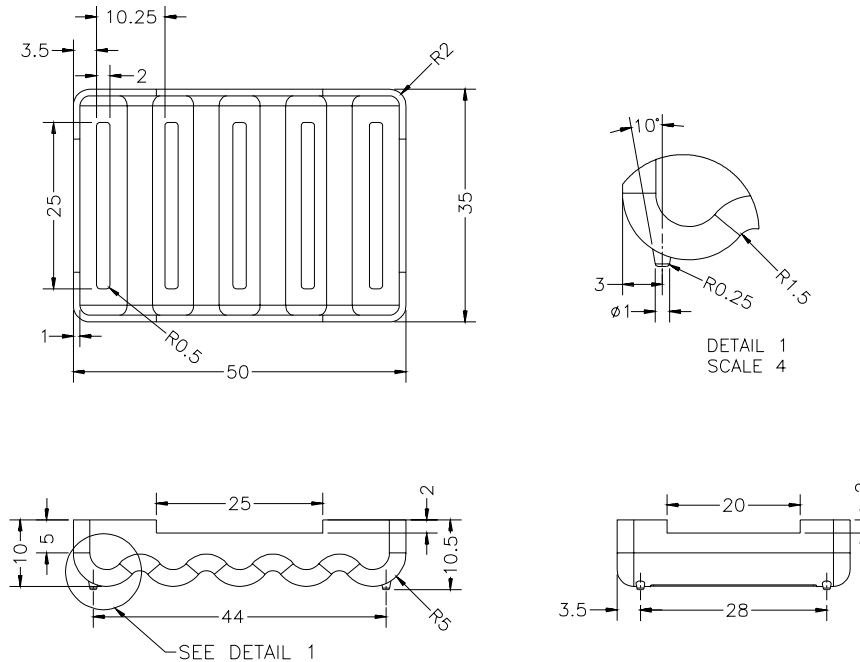


Figure 8-38 Views and dimensions of the model for Tutorial 1

- f. Create the slots on the lower part of the base and add the fillet to the slots feature, refer to Figure 8-48.
- g. Pattern the slots and the fillet feature, refer to Figure 8-48.
- h. Create a plane at an offset distance from the **Top** plane.
- i. Create the standoffs of the soap case using the extrude and fillet tool and pattern the same, refer to Figure 8-49.

Creating the Base Feature

First, you will create the base feature of the model by extruding a sketch using the **Mid Plane** option created on the **Front** plane.

1. Start SolidWorks and open a new part document from the **Template** tab of the **New SolidWorks Document** dialog box.
2. Draw the sketch of the base feature on the **Front** plane and add the required relations and dimensions to the sketch as shown in Figure 8-39.

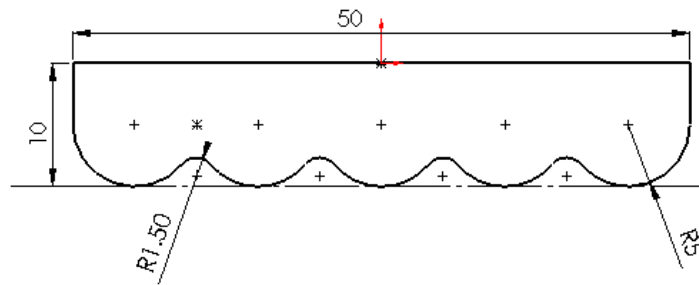


Figure 8-39 Sketch for the base feature

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

The base feature of the model is shown in Figure 8-40.

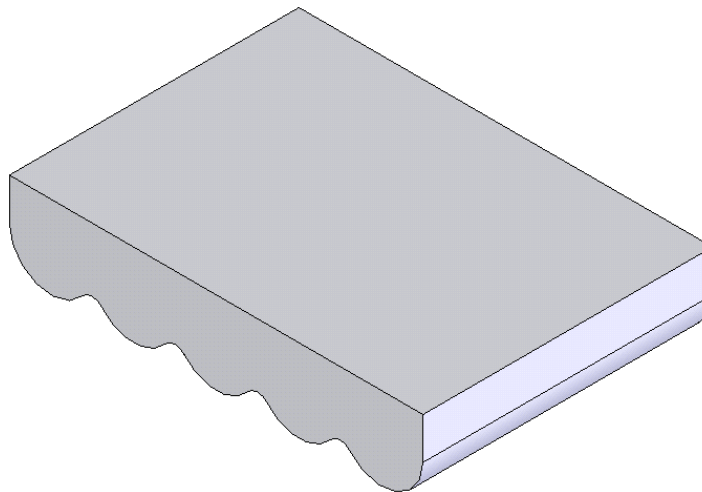


Figure 8-40 Base feature of the model

Filleting the Edges of the Base Feature

After creating the base feature you will fillet the lower edges of the base feature.

1. Invoke the **Fillet PropertyManager**, rotate the model, and select the edges of the base feature as shown in Figure 8-41.

2. Set the value of the radius spinner to **2.5** and choose the **OK** button from the **Fillet PropertyManager**.

The model after filleting the edges of the model is shown in Figure 8-42.

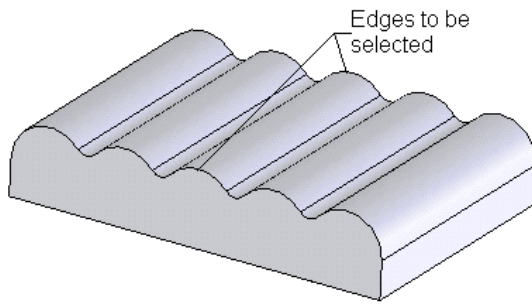


Figure 8-41 Edges to be selected

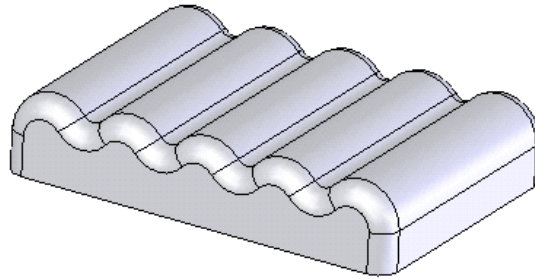


Figure 8-42 Fillet added to the model

Shelling the Model

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

1. Orient the model in the isometric view and invoke the **Shell PropertyManager**.
2. Select the top planar face of the model as shown in Figure 8-43.
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 8-44.

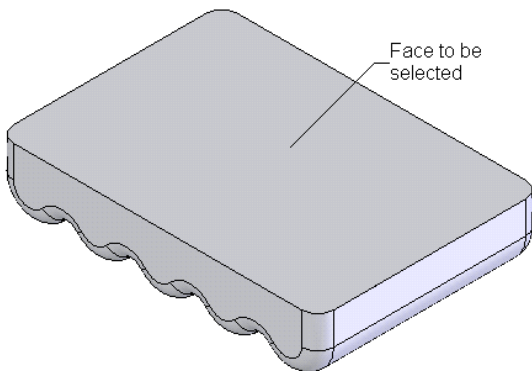


Figure 8-43 Face to be selected

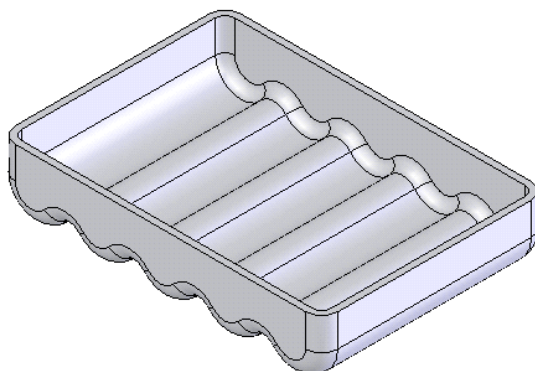



Figure 8-44 Shell feature added to the model

Dynamically Editing the Features

After creating and shelling 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** toolbar to invoke the dynamic dragging tool. 
2. Select the right planar face of the base feature from the drawing area. 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 8-45.

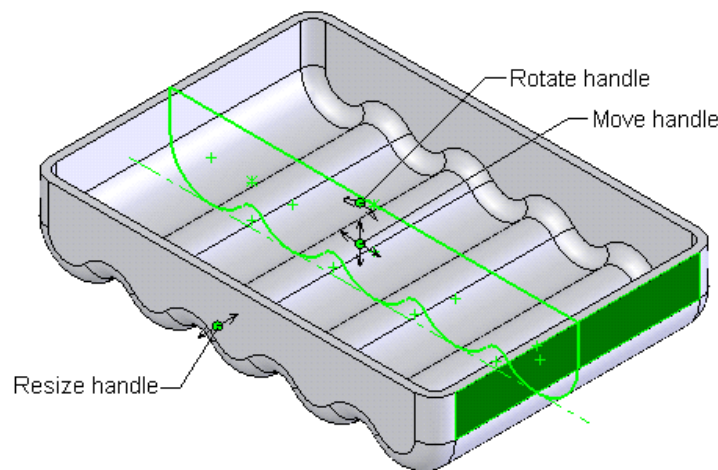


Figure 8-45 Editing handles for editing the base feature



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

*If you want to edit the sketches using the **Move/Size Features** tool, then you need to select the **Override Dimension** option by choosing **Tools > Sketch Settings > Override Dims on Drag/Move** from the menu bar. If a check mark is displayed at the left of this option in the menu bar, then the option is already selected.*

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

The preview of the resized feature and the dimension of the feature is displayed in the drawing area. As you drag the cursor the preview and the dimension updates automatically.

4. Release the left mouse button after dragging the feature to some distance.

Figure 8-46 shows the preview of the dragged feature. Figure 8-47 shows the edited feature.

5. Select the **Move/Size Features** button again to disable the dynamic editing tool and click

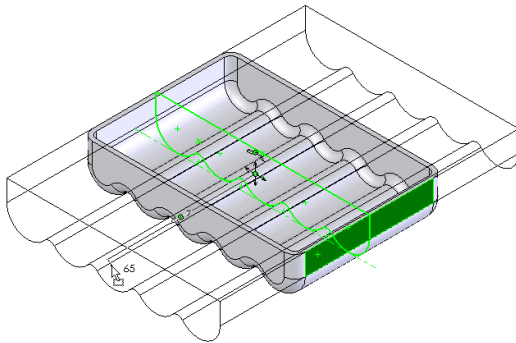


Figure 8-46 Dragging the **Resize** handle

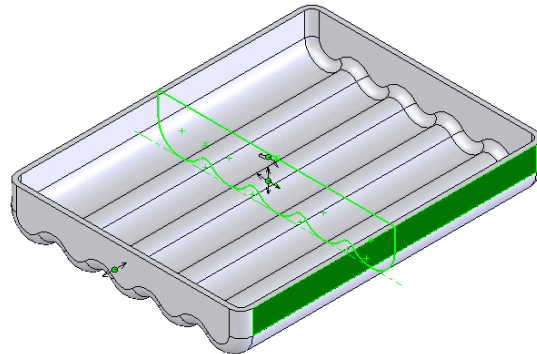


Figure 8-47 Resultant edited feature

anywhere in the drawing area to clear all the selections from the selection set.

You have edited the model by dynamically dragging, the feature needs to have the required dimension of 35mm. Therefore, now you will again edit the feature to retain the depth of the feature.

6. Double-click the base feature in the **FeatureManager Design Tree** or from the drawing area. All the dimensions of the feature are displayed in the drawing area.
7. Double-click the dimension that reflects the depth of the base feature.
8. The **Modify** dialog box will be displayed. Set the value of the **Dimension** spinner to **35** and press the ENTER key from the keyboard.
9. Choose the **Rebuild** button from the **Standard** toolbar or choose CTRL+B from the keyboard to rebuild the model.

Using the **Cut-Extrude**, **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 8-48.

Creating the Standoffs of the Model

After creating all the features of the model, you need to create the standoffs of the model. It is created by extruding a sketch created 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 10.5mm from the **Top** plane. You need to select the **Reverse direction** check box from the **Plane PropertyManager**.
2. Select the newly created sketching plane and create the sketch of the standoff and apply the required relations and dimensions. The sketch consists of a circle of diameter 1mm. For more dimensions refer to Figure 8-38.

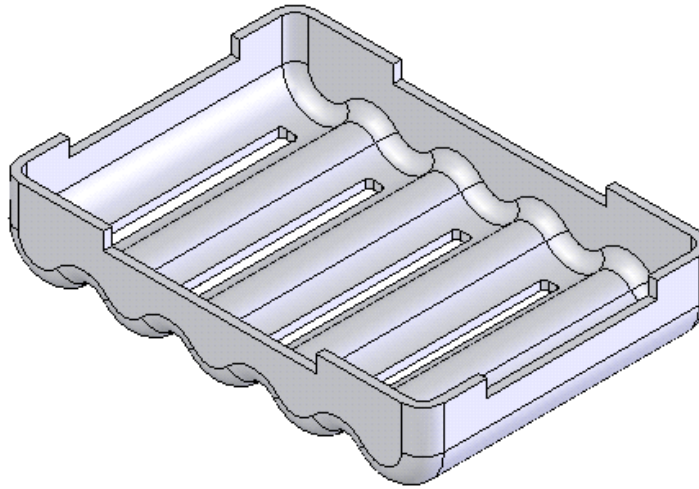


Figure 8-48 Model after creating other features

3. Extrude the sketch using the **Up To Next** option with an outward draft angle of 10-degree. Hide the newly created plane.
4. Rotate the model and add a fillet of radius 0.25 to the extruded feature.

The rotated and zoom view of complete standoff is displayed in Figure 8-49.

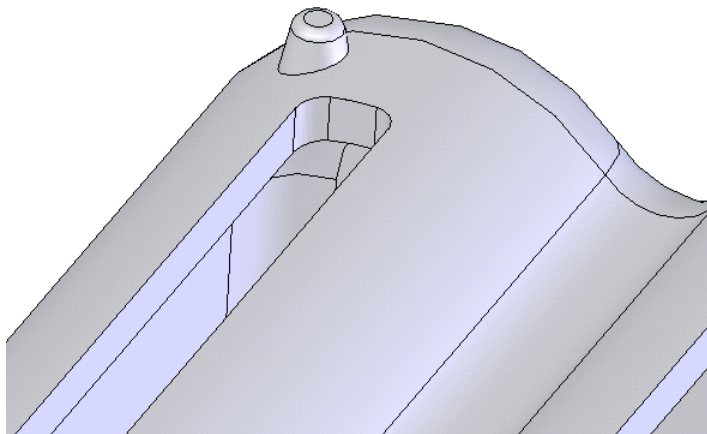


Figure 8-49 Rotated and zoom view of the model

5. Pattern the extruded and filleted feature using the **Linear Pattern** tool. The isometric view of the final model is shown in Figure 8-50. The **FeatureManager Design Tree** of the model is shown in Figure 8-51.

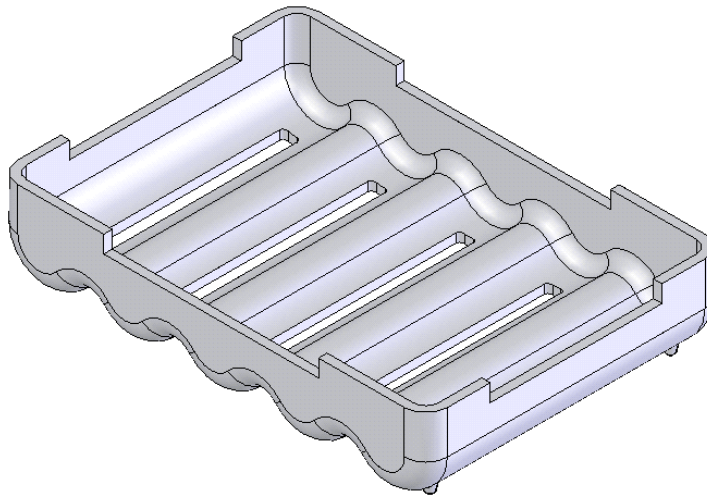


Figure 8-50 Final model

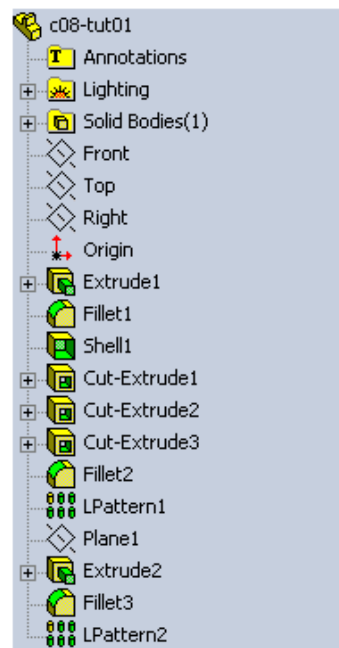


Figure 8-51 The FeatureManager Design Tree of the model

Saving the Model

Next, 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\c08\c08-tut01.SLDPRT.

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

Tutorial 2

In this tutorial you will create the model shown in Figure 8-52. After creating the model you will edit the model with **Move/Size Features** option selected. The views and dimensions of the model are shown in Figure 8-53. **(Expected time: 45 min)**

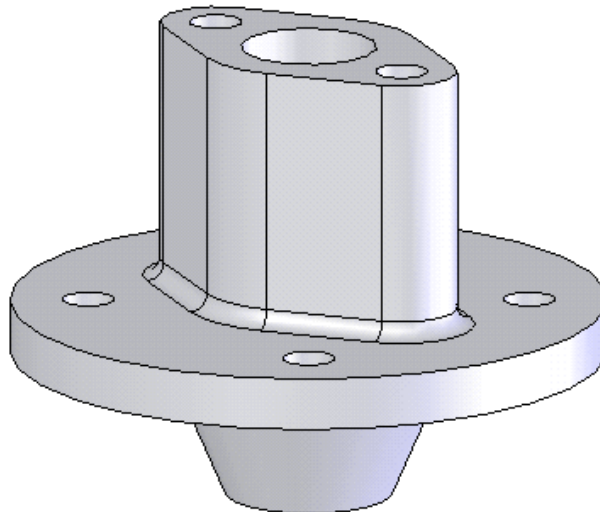


Figure 8-52 The dimetric view of the model for Tutorial 1

The steps to be followed to complete this tutorial are discussed next:

- a. Create the base feature of the model by revolving the sketch along the central axis of the sketch, refer to Figures 8-54 and 8-55.
- b. Create the sketch of the second feature on the top face of the base feature and extrude it to the given distance, refer to Figures 8-56 and 8-57.
- c. Create the revolve cut feature, refer to Figures 8-58 and 8-59.
- d. Create the holes using the **Simple Hole** tool and pattern the hole using the **Circular Pattern** tool, refer to Figure 8-59.
- e. Create a drilled hole feature using the **Hole Wizard** tool, refer to Figure 8-59.
- f. Mirror the hole feature along the **Right** plane, refer to Figure 8-59.
- g. Create the fillet feature, refer to Figure 8-59.

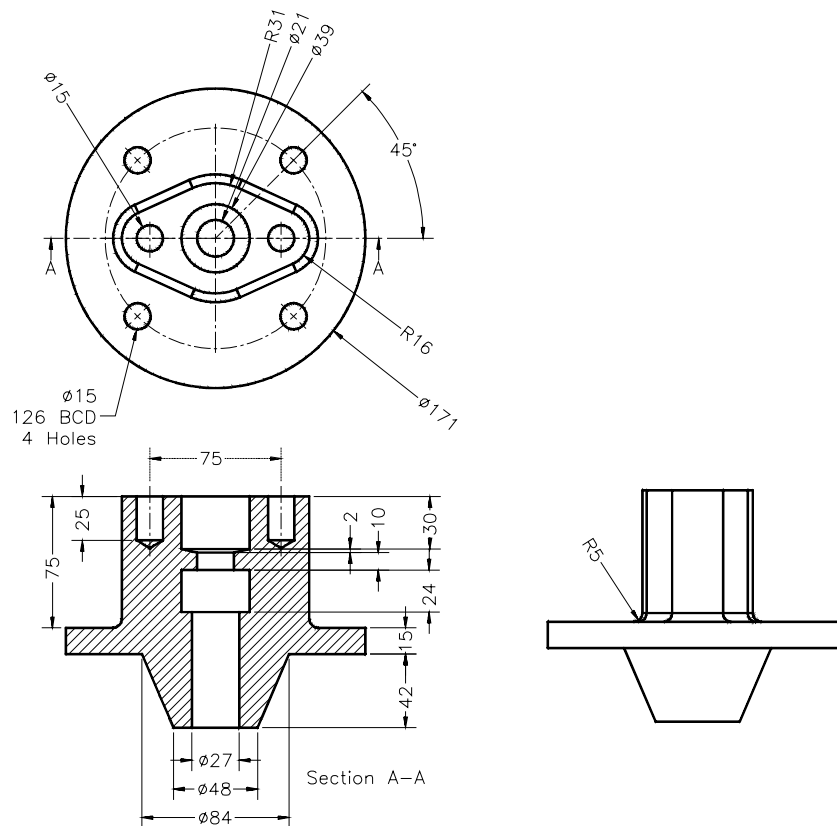


Figure 8-53 Views and dimensions of the model for Tutorial 1

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 SolidWorks and open a new part document from the **Template** tab of the **New SolidWorks Document** dialog box.
2. Draw the sketch of the base feature on the **Front** plane and add the required relations and dimensions to the sketch as shown in Figure 8-54.
3. Invoke the **Revolve PropertyManager**.

Since the default value in the **Angle** spinner is the same as required, therefore you do not need to set any parameters in this **PropertyManager**.

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

The base feature created after revolving the sketch is shown in Figure 8-55.

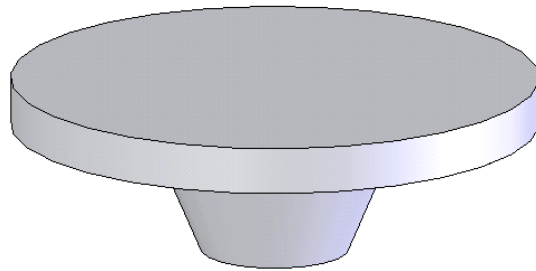
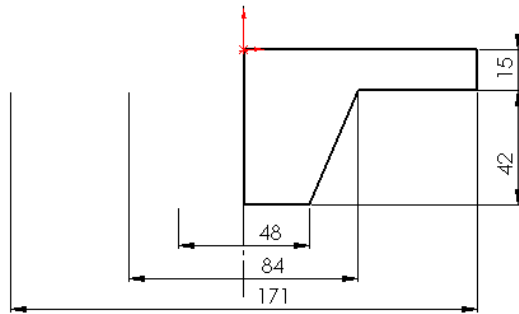


Figure 8-54 Sketch for the base feature

Figure 8-55 Dimetric view of the base feature

Creating the Second Feature

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

1. Select the top planar face of the base feature and invoke the sketching environment.
2. Create the sketch of the second feature and apply the required relations and dimensions to the sketch as shown in Figure 8-56. Make sure the sketch is symmetrical about the axis. If it is not, the sketch will not be properly mirrored.

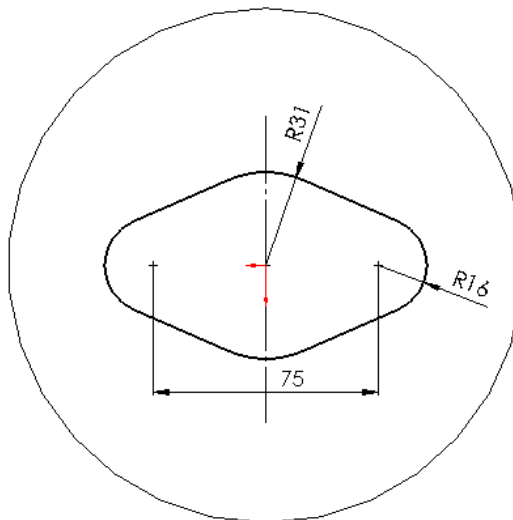


Figure 8-56 Sketch for the second feature

3. Extrude the sketch to a distance of 75mm.

The isometric view of the model after creating the second feature is displayed in Figure 8-57. Make sure the sketch is symmetrical about the axis. If it is not, the sketch will not be properly mirrored.

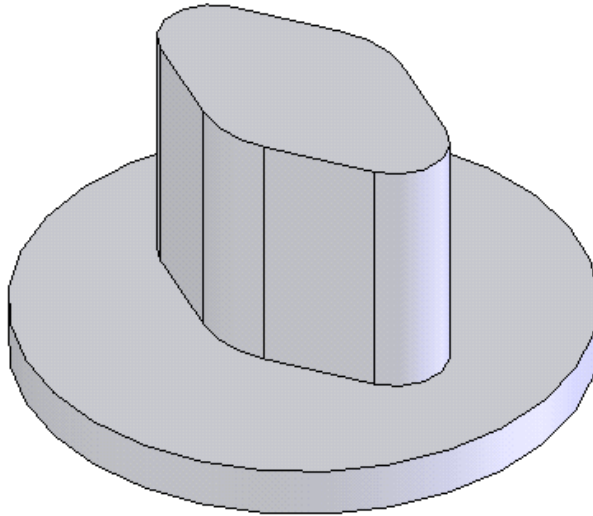


Figure 8-57 Second feature added to the model

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. Select the **Front** plane and invoke the sketching environment.
2. Create the sketch for the revolve cut feature and apply the required relations and dimensions to the sketch as shown in Figure 8-58.
3. Invoke the **Cut-Extrude PropertyManager** and create a revolved cut feature by specifying an angle on 360-degree.

Creating the Remaining Features

1. Create the other features of the model by referring to Figure 8-53.

The isometric view of the model after creating all the other features is displayed in Figure 8-59.

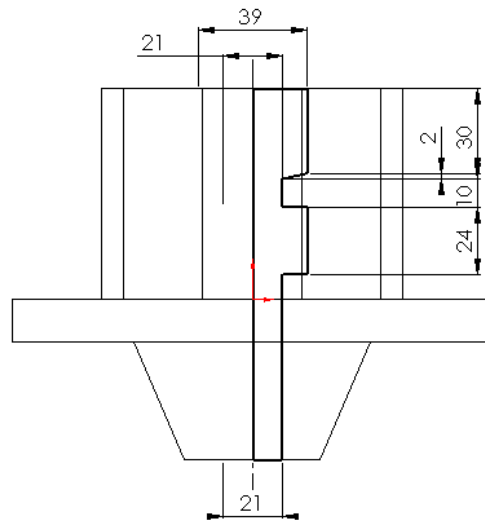


Figure 8-58 Sketch for the revolve cut feature

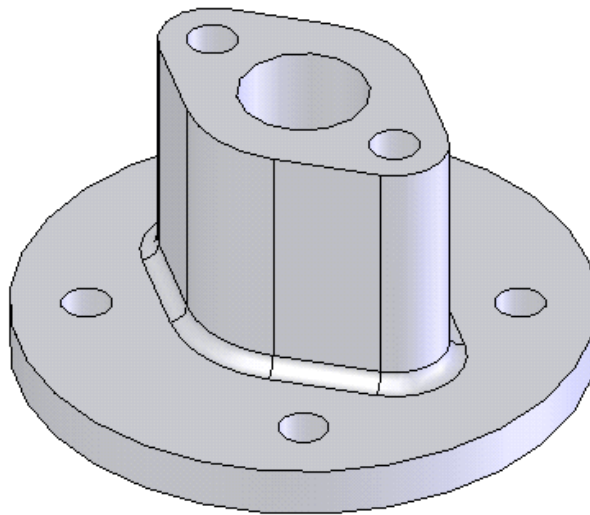


Figure 8-59 Final model

Editing the Sketch of the Model With the Move/Size Features Option Active

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

1. Choose the **Move/Size Features** button from the **Features** toolbar to invoke the **Move/Size Features** option.



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.

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 8-60.

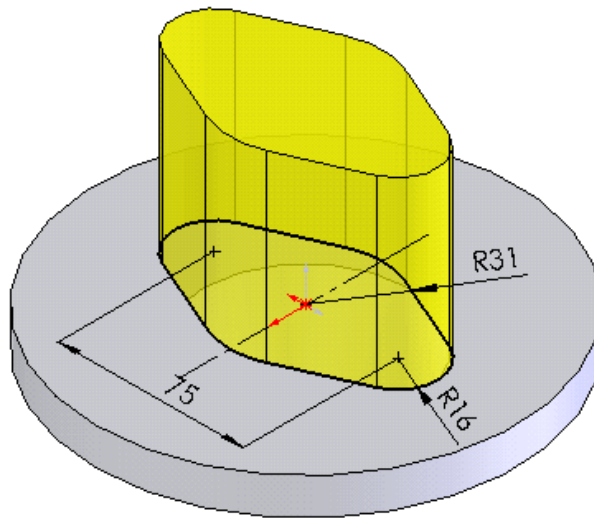


Figure 8-60 Sketch and the preview of the second feature

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

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

Figure 8-61 shows the sketch being dragged. Figure 8-62 shows the final dragged sketch.

After modifying the sketch by dragging, now you will retain the previous dimensions.

5. Double-click the linear dimension and enter **75** in the **Modify** edit box and press the ENTER key.

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

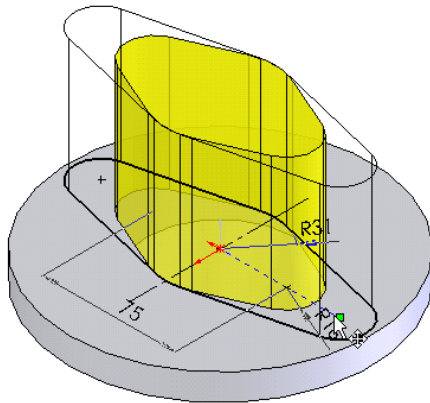


Figure 8-61 Dragging the sketch

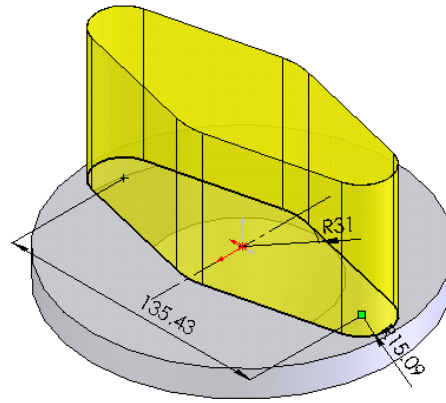


Figure 8-62 Sketch after dragging

6. Use CTRL+B from the keyboard to rebuild the model.

The **FeatureManager Design Tree** of the model is shown in Figure 8-63.

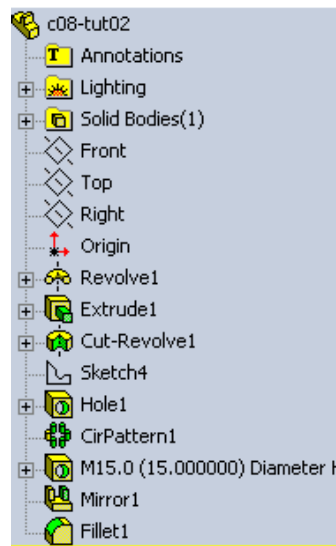


Figure 8-60 The FeatureManager Design Tree

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\\c08\\c08-tut02.SLDPRT.

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

Tutorial 3

In this tutorial you will create the model shown in Figure 8-64. While creating the model you will also perform some editing operations to the model. The views and dimensions of the model are displayed in Figure 8-65. (Expected time: 45min)

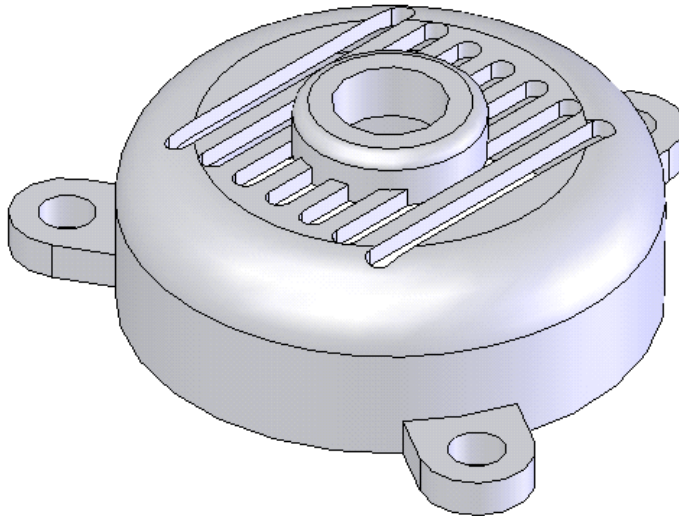


Figure 8-64 Model for Tutorial 3

The steps to be followed to complete this tutorial are discussed next:

- Create a base feature of the model by revolving the sketch created on the **Front** plane, refer to Figures 8-66 and 8-67.
- Shell the model using the **Shell** tool, refer to Figure 8-68
- Create the sketch on the **Top** plane and extrude it to the given distance, refer to Figure 8-69.
- Pattern the extrude feature using the **Circular Pattern** tool, refer to Figure 8-70.
- Edit the circular pattern, refer to Figure 8-71.
- Create a cut feature on the top planar face of the base feature, refer to Figure 8-72.
- Pattern the cut feature using the **Linear Pattern** tool, refer to Figure 8-73.
- Unsuppress the suppressed features and create the remaining features of the model. refer to Figures 8-74 and 8-75.

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 SolidWorks and open a new part document from the **Template** tab of the **New SolidWorks Document** dialog box.

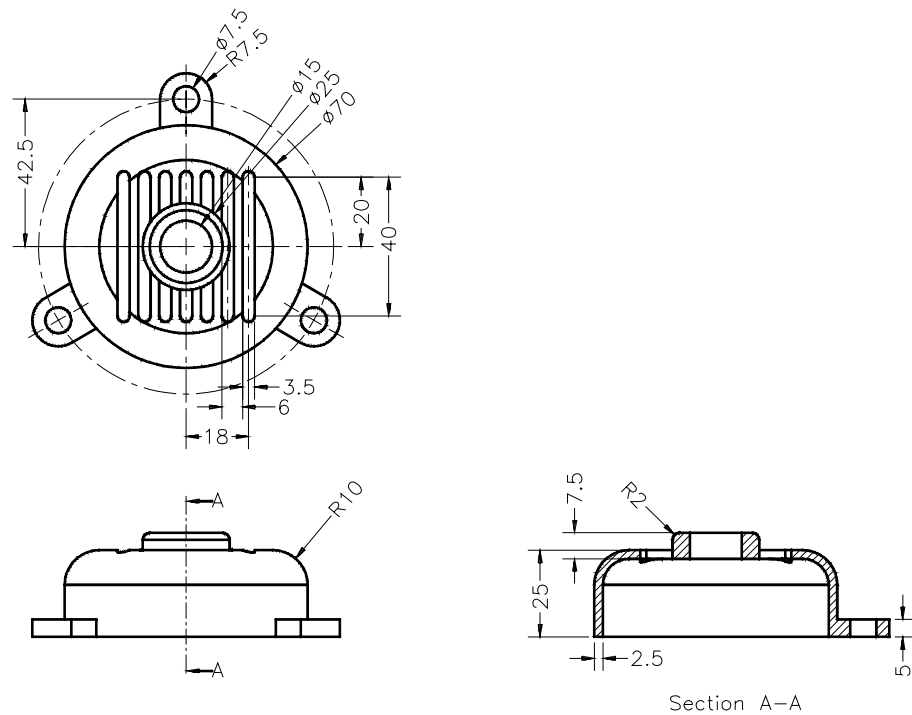


Figure 8-65 Views and dimensions of the model for Tutorial 3

2. Draw the sketch of the base feature on the **Front** plane and add the required relations and dimensions to the sketch as shown in Figure 8-66.
3. Using the **Revolve** tool create the base feature of the model. The base feature of the model is shown in Figure 8-67.

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.

1. Invoke the **Shell PropertyManager** and set the value of the **Thickness** spinner to **2.5** mm.
2. Rotate the model and select the bottom face of the model to remove.

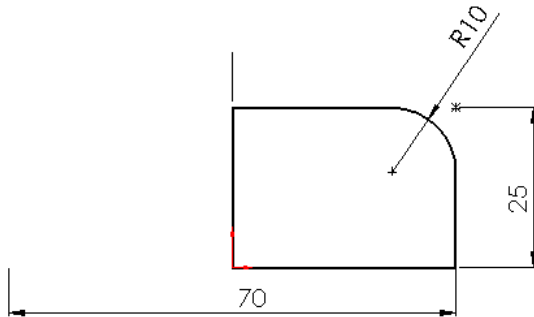


Figure 8-66 Sketch for the base feature

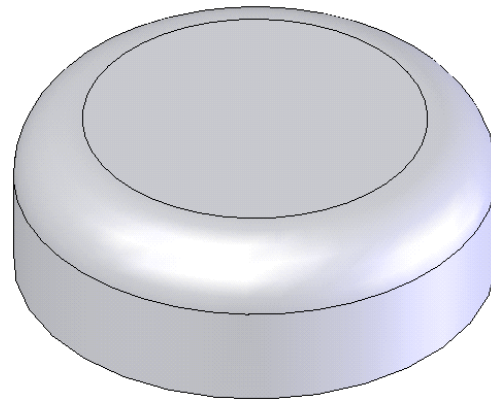


Figure 8-67 Base feature of the model

3. Choose the **OK** button from the **Shell PropertyManager**. The shelled model is displayed in Figure 8-68.

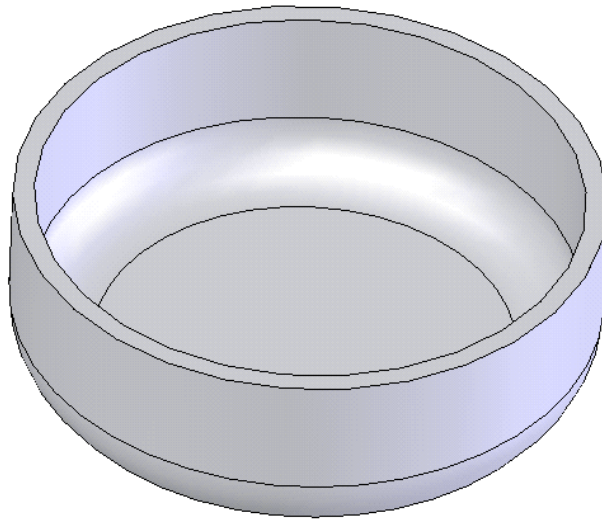


Figure 8-68 Shell feature added to the model

Creating the Third Feature

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

1. Select the **Top** plane from the **FeatureManager Design Tree** and invoke the sketching environment.
2. Orient the model in the top view using the **View** toolbar.

3. Create the sketch of the third feature and add the required relations and dimensions to the sketch as shown in Figure 8-69.

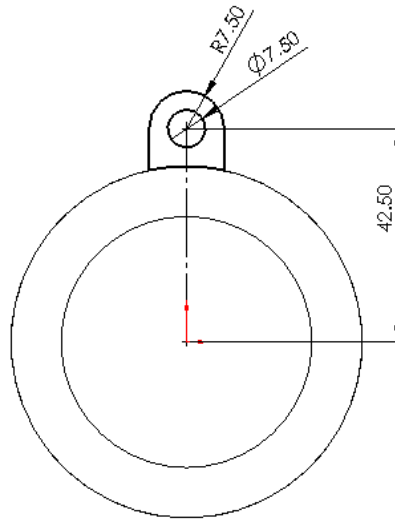


Figure 8-69 Sketch of the third feature

4. Extrude the sketch to the a depth of 5mm.

Patterning the Third Feature

After creating the third feature of the model, you need to pattern this feature. This feature will be patterned using the **Circular Pattern** tool. Before proceeding further you need to display the temporary axis. The central temporary axis of the model will be selected as pattern axis.

1. Choose **View > Temporary Axes** from the menu bar to display the temporary axis.
2. Invoke the **Circular Pattern PropertyManager** and select the central axis of the model as the pattern axis.
3. Select the third feature created earlier from the drawing area. 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 axis from the current display.

The model after creating the pattern feature is displayed in Figure 8-70.

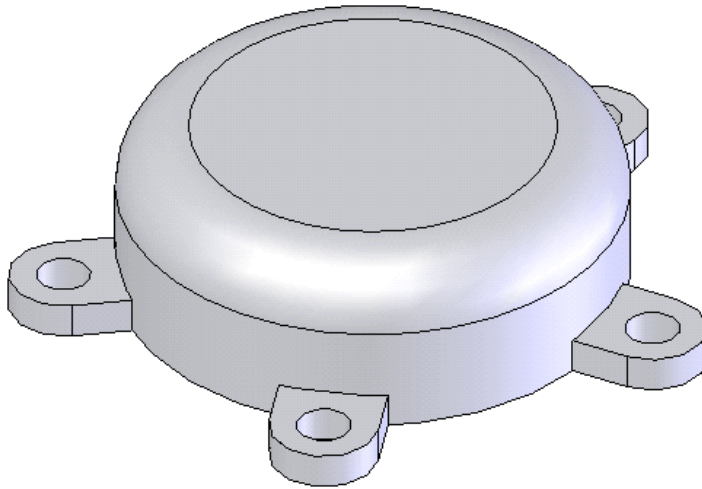


Figure 8-70 Pattern feature added to the model

Editing the Pattern Feature

The pattern created is not the same as required, refer to Figure 8-65. As a result you need to edit the pattern feature.

1. Select **CirPattern1** from the **FeatureManager Design Tree** or select any one of the pattern instance other than the parent instance from the drawing area. Right-click and choose the **Edit Definition** 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 8-71.

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 just turned off. When you suppress a feature, the child features associated with that feature are also suppressed.

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.

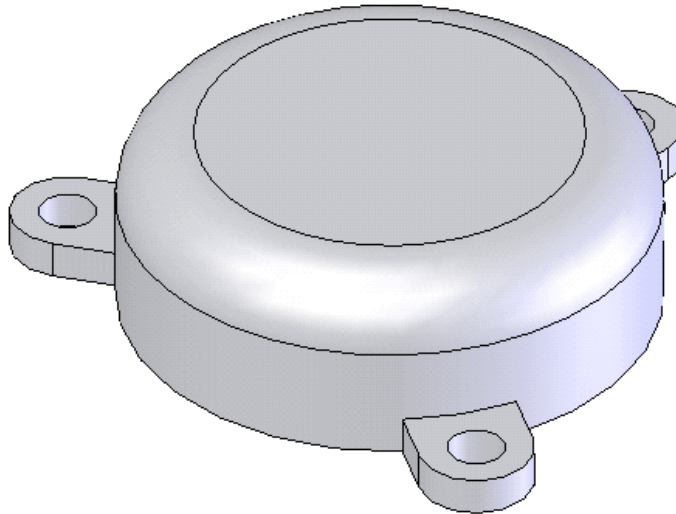


Figure 8-71 The edited pattern feature

The circular pattern feature is the child feature of the extrude feature, therefore, it is also suppressed. Both the 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 the features 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 created on the top planar face of the base feature.

1. Select the top planar face of the base feature as the sketching plane and invoke the sketching environment.
2. Create the sketch of the cut feature and add the required relations and dimensions to the sketch as shown in Figure 8-72.
3. Invoke the **Cut-Extrude PropertyManager** and extrude the sketch using the **Through All** option.
4. Choose the **OK** button from the **Cut-Extrude PropertyManager**.

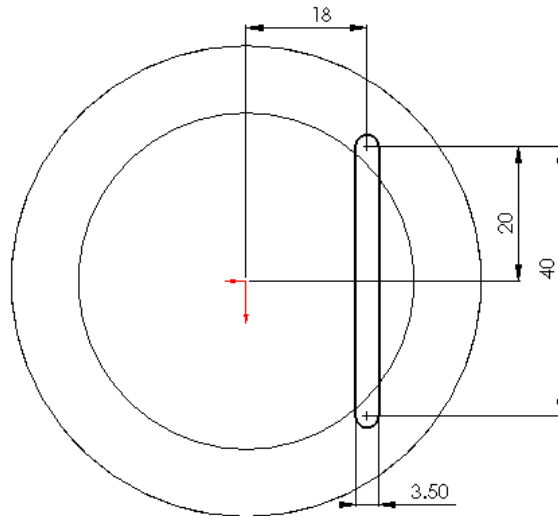


Figure 8-72 Sketch for the cut feature

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 8-73.

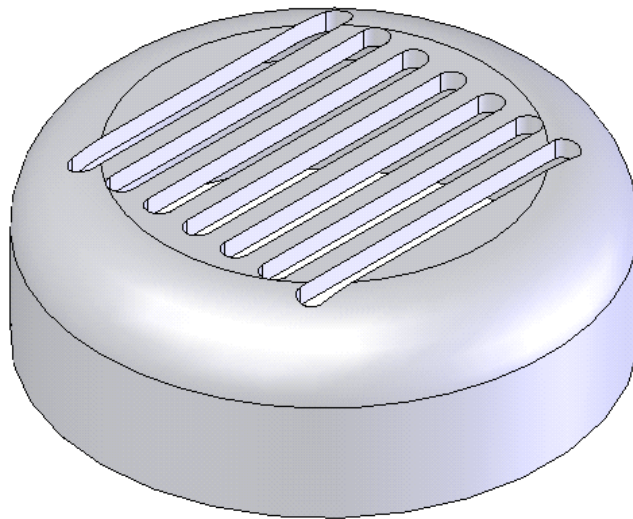


Figure 8-73 Model after patterning the cut feature

6. Create the other features of the model. For dimensions, refer to Figure 8-65. The model after creating the other features is shown in Figure 8-74.

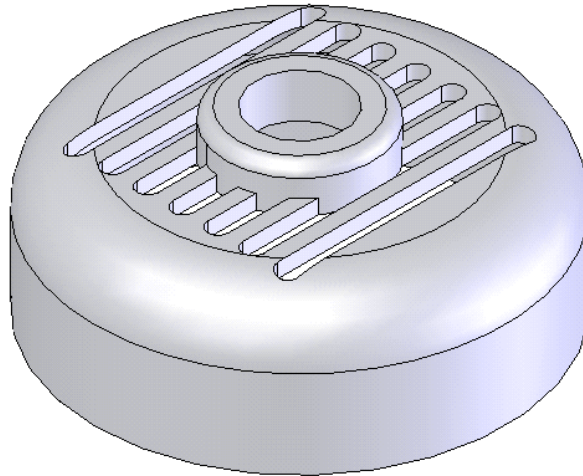


Figure 8-74 Model after creating other features.

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 from the keyboard and select the suppressed features from the **FeatureManager Design Tree**.



Note

If you select only the parent suppressed feature and unsuppress it then 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 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.*

If you unsuppress the child feature then 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 8-75. The **FeatureManager Design Tree** of the model is shown in Figure 8-76.

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\\c08\\c08-tut03.SLDPRT.`

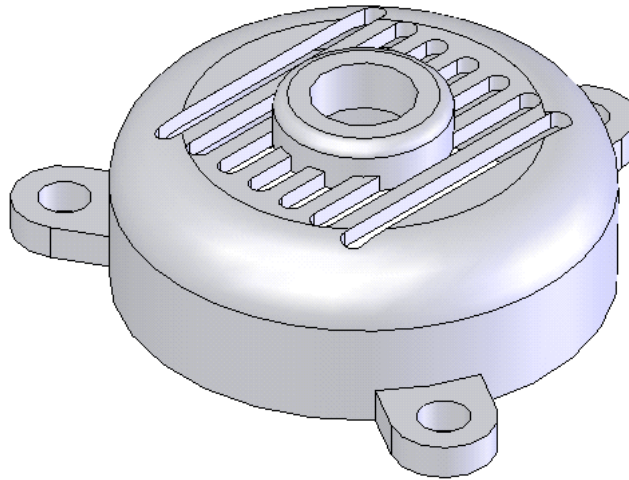


Figure 8-75 The final model

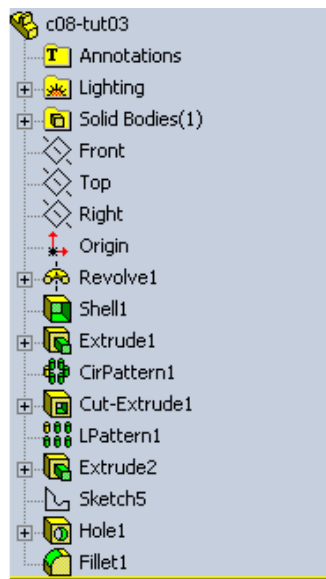


Figure 8-76 The FeatureManager Design Tree

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

SELF-EVALUATION TEST

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

1. You cannot edit the sketch of a sketched feature. (T/F)

2. The **Edit Definition** 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. Using the **Move/ Size Features** option you cannot edit the sketches. (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. _____ key is used from the keyboard to copy a feature or a sketch.
4. The _____ key is used from the keyboard to cut a feature or sketch.
5. When _____ tool is active then 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 the feature to the required position. (T/F)
9. The **Modify** dialog box is invoked using a single click on the dimension to modify. (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 shown in section in Figure 8-77. The dimensions of the model are also given in the same figure. The complete model is shown in Figure 8-78.

(Expected time: 45 min)

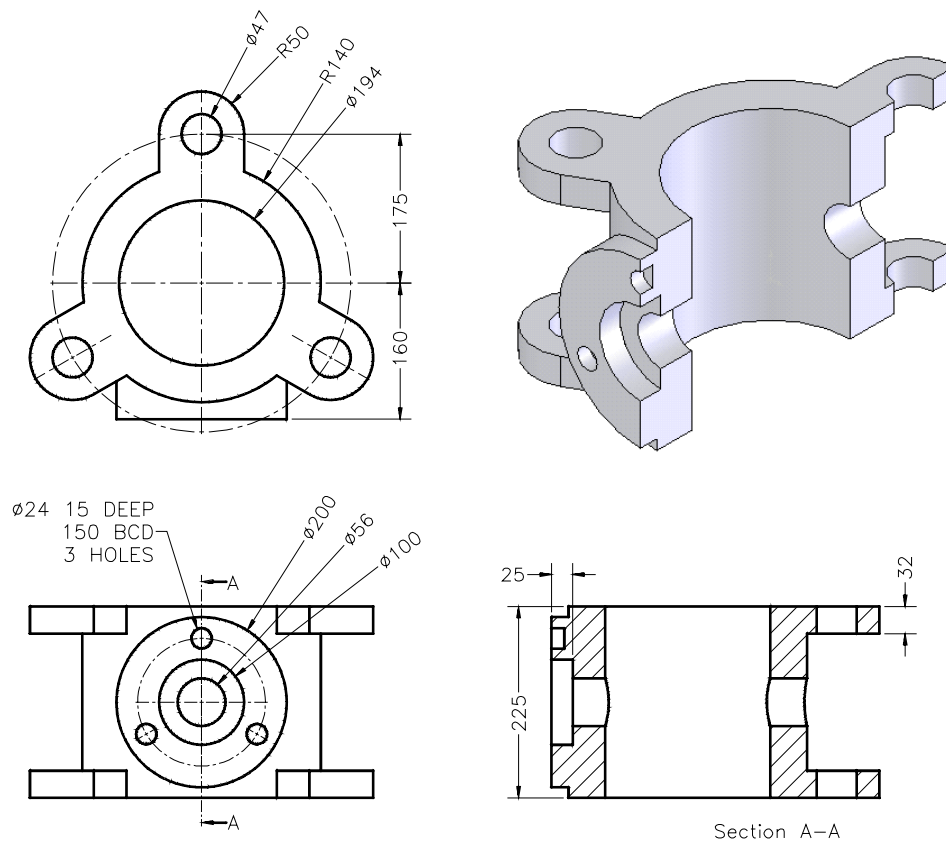


Figure 8-77 Views and dimensions of the model for Exercise 1

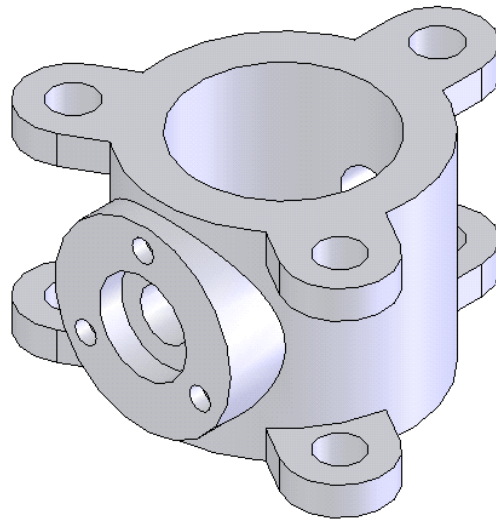


Figure 8-78 Model for Exercise 1

Exercise 2

Create the model shown in Figure 8-79. The dimensions of the model are shown in Figure 8-80.
(Expected time: 30 min)

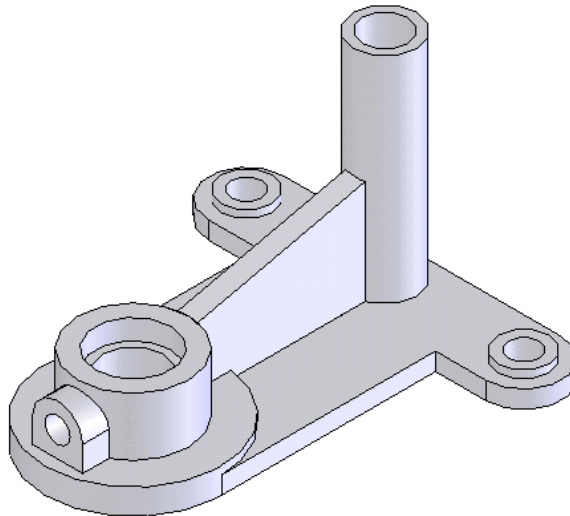


Figure 8-79 Model for Exercise 2

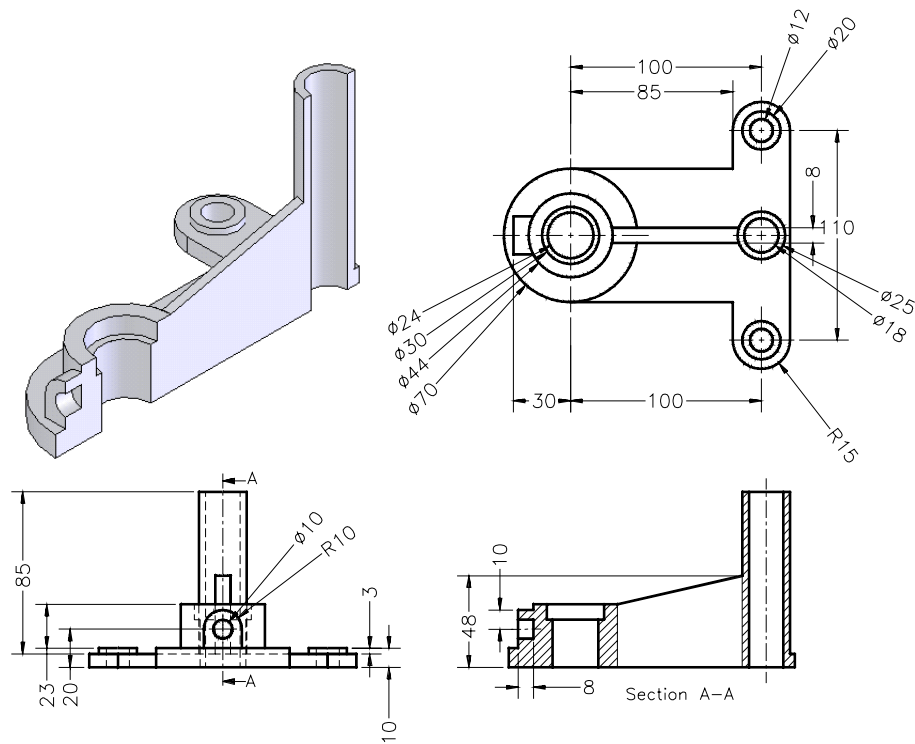


Figure 8-80 Views and dimensions of the model for Exercise 2

Answer 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