Chapter 3

Part Modeling - I

Leaning Objectives

After completing this chapter, you will be able to:

- *Understand the DesignModeler Workspace.*
- *Draw sketches.*
- *Convert sketches into 3D models.*
- *Understand views of the model in 3D space.*
- *Apply constraints and relations.*
- *Create new sketching planes.*

INTRODUCTION TO PART MODELING

For conducting an FEA analysis, a part model is mandatory. In ANSYS Workbench, the next step after defining the material properties is to define the geometry to which the material properties will be applied. Like most of the other CAD modeling packages, the part models created in ANSYS Workbench are parametric and feature based. The parametric nature of an application is defined as its ability to use the standard properties or parameters (dimensions) in determining the shape and size of the geometry. You can also modify the shape and size of the model using these parameters. Feature is defined as the smallest building block of the model that can be modified individually.

Most of the 3D models created using ANSYS Workbench are a combination of sketched and placed features. The placed features are created without drawing a sketch, but the sketched features require a sketch to be drawn first. Generally, the base feature of any 3D model is a sketched feature and is created using a sketch. Therefore, while creating any design, you first need to draw the sketch for the base feature. Once you have drawn the sketch, you can convert it into the base feature and then add the other sketched and placed features to it to complete the 3D model.

In general terms, a sketch is defined as the basic contour for the feature. For example consider the spanner shown in Figure 3-1. It is created using a single sketch drawn on the XY plane, as shown in Figure 3-2.

Figure 3-1 *Base feature of the spanner*

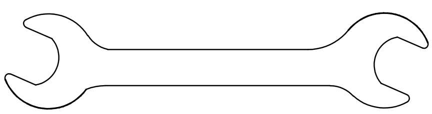


Figure 3-2 *Sketch for the base feature of the spanner*

INTRODUCTION TO DesignModeler WINDOW

In ANSYS Workbench, the part models and their sketches are defined in the **DesignModeler** window. You can define part models either by importing the CAD model created in some other CAD applications such as Pro/E, SolidWorks, and so on, or by creating the model in the **DesignModeler** window of ANSYS Workbench 14.0.

In any system, the **DesignModeler** window is associated with the **Geometry** cell. The **Geometry** cell can be added to any analysis project as a standalone component system or as a part of any mechanical analysis system. Figure 3-3 displays system **A** as a standalone component system. In system **B**, the standalone system **A** is used as a part of an analysis system. In analysis system **C**, the model will be defined in the **DesignModeler** window that is associated with the **Geometry** cell of the same system.

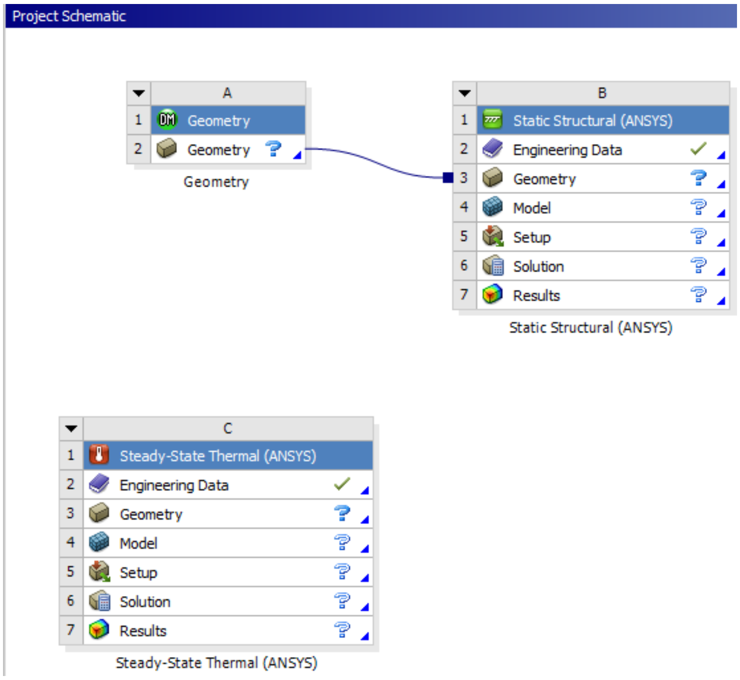


***Figure 3-3 Project Schematic** window displaying various standalone systems in it*

To open the **DesignModeler** window, double-click on the **Geometry** cell in an analysis system; the **DesignModeler** window along with the **ANSYS Workbench** dialog box will be invoked. Alternatively, right-click on the **Geometry** cell of any analysis system and choose the **New Geometry** option from the shortcut menu displayed.

The **ANSYS Workbench** dialog box is the startup dialog box that is displayed along with the **DesignModeler** window. When this dialog box is opened, you are prompted to specify the unit to be used for creating the models. Select the radio button corresponding to the desired unit of length. To always use the unit specified in this dialog box, select the **Always use selected unit** check box. However, you can also use the unit system specified for the project in the **Workbench** window by selecting the **Always use project unit** check box. Note that only one check box can be selected at a time. After specifying the units and your preferences, choose the **OK** button from the **ANSYS Workbench** dialog box; the **DesignModeler** window will be activated, as shown in Figure 3-4.

Note

*1. If you select the **Always use project unit** check box or the **Always use selected unit** check box then the **ANSYS Workbench** dialog box will not be displayed along with the **DesignModeler** window. However, if you want to display the **ANSYS Workbench** dialog box at the start of new **DesignModeler** session, choose **Tools > Options** from the Menu bar in the **DesignModeler*** *window; the **Options** dialog box will be displayed. Expand the **DesignModeler** node from the left pane of the **Options** dialog box and select the **Units** sub option; the corresponding options will be displayed in the right pane. Click on the **Display Units Pop-up Window** option from the right pane; a drop-down list will be displayed on the right of this option. Select the **Yes** option from this drop-down list.*

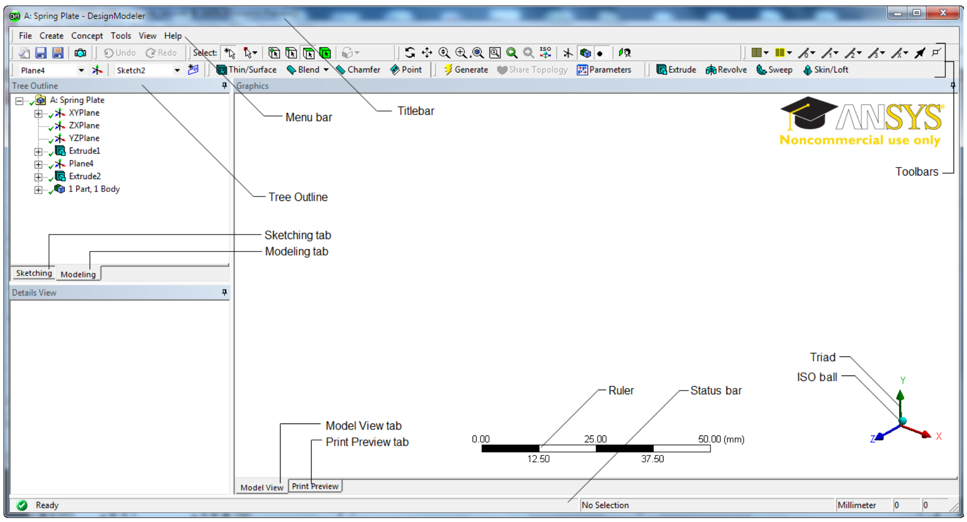


***Figure 3-4** The **DesignModeler** window*

The **DesignModeler** window can be used in two basic modes that are discussed next.

Sketching Mode

The **Sketching** mode is used to draw 2D sketches. Later on, these sketches can be converted into 3D models using the **Modeling** mode. To invoke the **Sketching** mode, choose the **Sketching** tab from the **DesignModeler** window, refer to Figure 3-4. The **Sketching** mode displays the **Sketching Toolboxes** window which contains five toolboxes. These toolboxes are used to create, modify, and dimension the sketches, refer to Figure 3-5.

Modeling Mode

The **Modeling** mode is used to generate the part model using the sketches drawn in the **Sketching** mode. By default, the **Modeling** mode is active when the **DesignModeler** window is invoked. If not, choose the **Modeling** tab from the **DesignModeler** window, refer to Figure 3-4. In the Modeling mode, the Tree Outline is displayed on the left of the Graphics window which contains three default planes. Apart from three default planes, the list of all operations that are used to create a model in the **Modeling** mode will be listed in the Tree Outline in the sequence they are performed, refer to Figure 3-6.

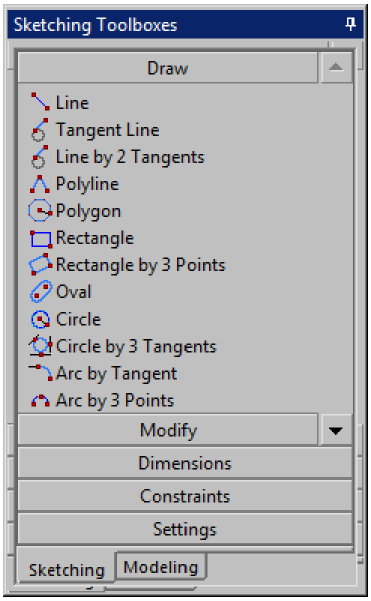


*Figure 3-5 The **Sketching Toolboxes** window*

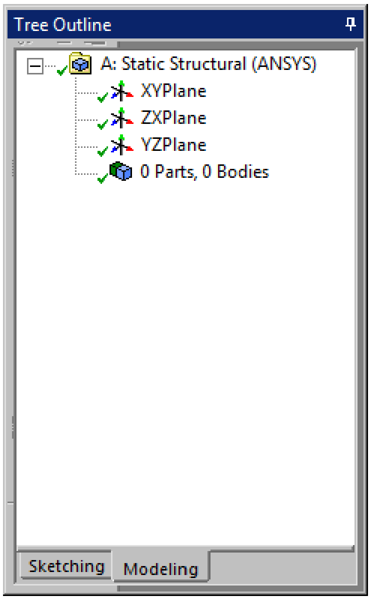


Figure 3-6 The Tree Outline

SCREEN COMPONENTS OF THE DesignModeler WINDOW

The various screen components of the default **DesignModeler** window and important terms related to this window are discussed next.

Tree Outline

In ANSYS Workbench three planes **XYPlane**, **YZPlane**, and **ZXPlane** corresponding to the XY, YZ, and ZX planes of cartesian coordinate system are created by default and are displayed in the Tree Outline. These planes are used as sketching planes for drawing the sketches to be used for generating the part model.

A sketch is a collection of 2D drawing entities which is used for generating the features of a part model. The sketch can be created on any one plane. However, a single plane can have multiple sketches associated with it. The plane for creating the sketch can be specified by selecting it from the Tree Outline. When you click on a plane in the Tree Outline, it is displayed in the **Graphics** window. Figure 3-7 shows the plane that will be displayed on selecting **XYPlane** from the Tree Outline.

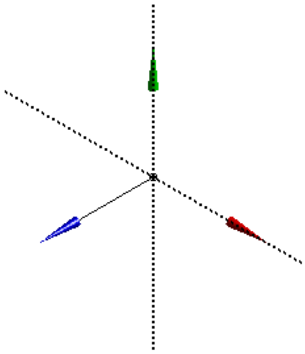


*Figure 3-7 The XY plane displayed when **XYPlane** is selected in the Tree Outline*

Note

1. The last sketching plane you worked on will act as the active plane for any future operation, unless and until you change the plane by selecting it from the Tree Outline.

*2. Apart from these three default planes, the user can also create new planes at the specified location and orientation in the **Graphics** window. The method of creating new planes will be discussed in detail in the later chapters.*

*3. To start a new sketch, choose the **New Sketch** tool from the **Active Plane/Sketch** toolbar. On doing so, a new sketch instance will be added under the desired plane in the Tree Outline. Alternatively, you can select the desired plane in the Tree Outline and then switch to the **Sketching** mode and draw a sketch. On doing so, a sketch instance will automatically be added under the selected plane.*

Details View Window

The **Details View** window located near the **Graphics** window is contextual in nature and changes its content according to the selection made in the Tree Outline. Figure 3-8 shows the **Details View** window which is displayed when the **Extrude1** is selected in the Tree Outline. You can also modify the selected entity by editing its parameters in the **Details View** window.

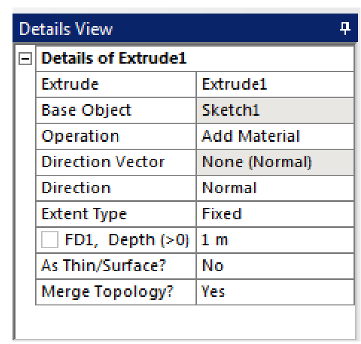


***Figure 3-8** The **Details View** window*

Model View/Print Preview

The **Model View** and the **Print Preview** tabs are located at the lower left corner of the **Graphics** window, refer to Figure 3-4. By default, the **Model View** tab is chosen in the **DesignModeler** window. Subsequently, the sketches and the part model are displayed in this interface. To preview the current view of the model, choose the **Print Preview** tab. After previewing the model, choose **File > Print** from the Menu bar to print it. Note that the option to print the model will be available only in the **Print Preview** mode.

Ruler

The Ruler is displayed at the bottom of the **Graphics** window, refer to Figure 3-9. The Ruler helps the user to visualize and compare the actual size of the model with the size of the model displayed. The number displayed at the end of each block in the Ruler represents the cumulative length of the blocks on the left of the number. The quantity shown in brackets at the extreme right of the ruler represents current unit of the length. To toggle the display of the Ruler in the **Graphics** window, choose **View > Ruler** from the Menu bar.

Triad

The Triad is displayed at the lower right corner of the **Graphics** window, refer to Figure 3-10. Triad helps the user to visualize the X, Y, and Z directions in the **Graphics** window. To toggle the display of the Triad in the **Graphics** window, choose **View > Triad** from the Menu bar. Move the cursor to any axis, its name will be displayed attached to the cursor. Moving the cursor in the negative direction of the three orthogonal axis system displays the temporary view of the axis and its name with a negative symbol (-). If you click on any axis system, the view will get oriented normal to the selected axis. Click on the ISO ball (cyan color) displayed at the center of the triad to orient the model in the Isometric view.

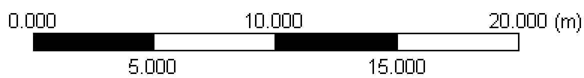


Figure 3-9 The Ruler

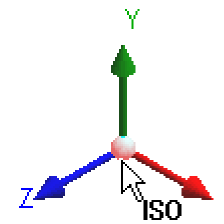


Figure 3-10 The Triad

Status Bar

The Status bar is located at the bottom of the screen, refer to Figure 3-4. The instructions for the currently active tool and its status are displayed on the left of the Status bar. The middle portion of the Status bar displays the information about the currently selected object. The right portion of the Status bar displays the current unit system and the coordinate value of the cursor location.

TUTORIALS

Tutorial 1

In this tutorial, you will create the I-section solid model shown in Figure 3-11. The dimensions of the model are shown in Figure 3-12. Save the project with the name *c03_ansWB_tut01* at the location *C:\ANSYS_WB\c03\Tut01* **(Expected time: 30 min)**

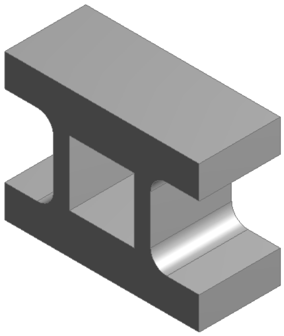


Figure 3-11 Model for Tutorial 1

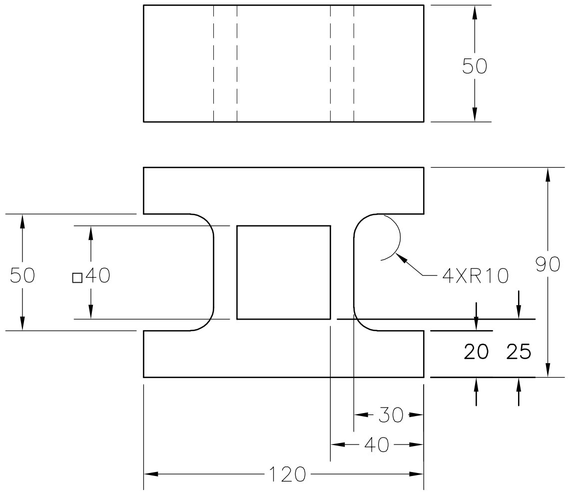


Figure 3-12 Dimensions of the I-section

The following steps are required to complete this tutorial:

a. Start ANSYS Workbench 14.0 and add the **Geometry** component system.
b. Create the sketch for the outer loop of the I-section.

c. Apply constraints to the sketch.
d. Generate dimensions and edit them to achieve the required size of sketch.
e. Create the base feature.
f. Create the second feature.
g. Create the blend feature.
h. Rotate the view of the model dynamically.
i. Save the project and exit the ANSYS Workbench session.

Starting ANSYS Workbench and Adding the Component System

First you need to start ANSYS Workbench and then add an analysis system to the **Project Schematic** window.

1. Choose **All Programs > ANSYS 14.0 > Workbench 14.0** from the Start menu to start the ANSYS Workbench session; the **Getting Started** window is displayed with the **Workbench** window.

2. Choose the **OK** button to close the **Getting Started** window.

Note

*1. If you do not want the **Getting Started** window to be displayed the next time you start a new session of ANSYS Workbench, clear the **Show Getting Started Message at Startup** check box located at the bottom of the **Getting Started** window and then close it.*

*2. If you want the **Getting Started** window to be displayed at the startup of a new ANSYS Workbench session, choose **Tools > Options** from the Menu bar; the **Options** dialog box will be displayed. Next, choose **Project Management** from the left pane of the **Options** dialog box, if not chosen by default. Scroll down in the right pane and select the **Show Getting Started Dialog** check box.*

3. Double-click on **Geometry** displayed under the **Component Systems** toolbox in the **Toolbox** window; the **Geometry** component system is added and displayed in the **Project Schematic** window, refer to Figure 3-13.

After the **Workbench** window is displayed and an analysis system is added to the **Project Schematic** window, the first step in any analysis is to define the geometry. There are two ways to define a geometry: by creating a new geometry and by importing an already created geometry from any solid modeling software. The **DesignModeler** window is used to create and edit geometries which are used in ANSYS Workbench. In ANSYS Workbench 14.0, a standalone system known as **Geometry** component system, is available to create geometries. Figure 3-13 shows a **Geometry** component system added to the **Project Schematic** window.

Now, you need to save the project.

4. In the **Workbench** window, choose the **Save** button from the **Standard** toolbar; the **Save As** dialog box is displayed.

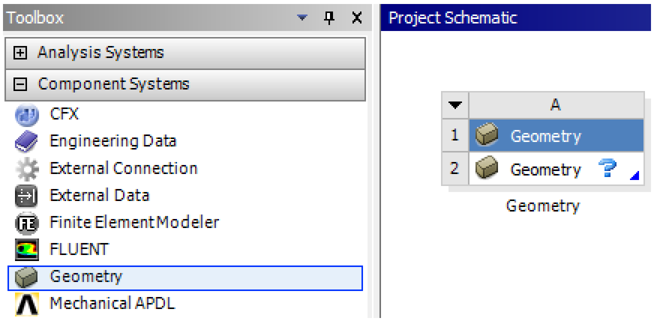


***Figure 3-13** The **Geometry** component system added to the **Project Schematic** window*

5. Browse to the location *C:\ANSYS_WB*.

6. Create a folder with the name **c03** in the *ANSYS_WB* folder and then then double-click on the newly created *c03* folder to open it. Next, create a sub folder with the name **Tut01** under the *c03* folder and then choose the **Open** button from the **Save As** dialog box.

7. Enter **c03_ansWB_tut01** in the **File name** edit box and choose the **Save** button from the **Save As** dialog box; the project is saved with the name specified.

Creating the Sketch

You now need to invoke the **DesignModeler** window to create the sketch.

1. Right-click on the **Geometry** cell of this component system to display a shortcut menu. Next, choose **New Geometry** from the shortcut menu; the **Starting DesignModeler** message is displayed in the status bar. After sometime, the **DesignModeler** window is displayed along with the ANSYS Workbench dialog box.

2. In the **ANSYS Workbench** dialog box, select the **Millimeter** radio button and accept the default settings for other check boxes. Next, choose the **OK** button to close the dialog box.

   Next, you need to select the plane in which you want to create the sketch for the tutorial.

3. Select **XYPlane** in the Tree Outline that is displayed in the left pane of the **DesignModeler** window.

The Tree Outline in the **DesignModeler** window lists all the operations that are carried out on the model. The Tree Outline for **Geometry** component system is shown in Figure 3-14. Note that the operations in the Tree Outline are listed in the sequence in which they were created.

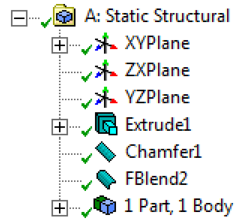


Figure 3-14 *The Tree Outline displayed in the* ***DesignModeler*** *window*

4. Choose the **Sketching** tab available at the bottom of the Tree Outline to display all the tools available for creating sketches.

   Since the XY plane has already been selected for drawing the sketch, as shown in Figure 3-15, you need to orient the plane normal to the viewing direction.

5. Right-click anywhere in the **Graphics** window and choose the **Look At** tool from the shortcut menu; the view is oriented as required, refer to Figure 3-16.

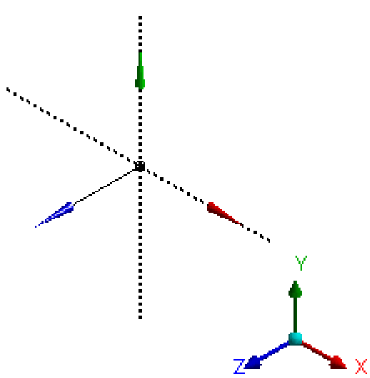


Figure 3-15 *The default Isometric view of the XY plane*

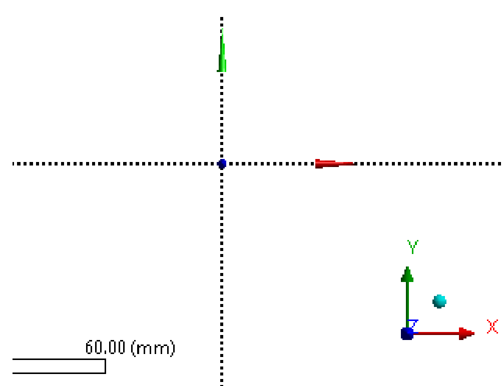


Figure 3-16 *XY plane after choosing* ***Look At*** *tool from the* ***Graphics*** *toolbar*

The **Look At** tool is used to orient the plane on which the sketch is drawn, normal to the viewing direction. You can invoke this tool from the **Graphics** toolbar, refer to Figure 3-17. Alternatively, invoke this tool from the shortcut menu which is displayed on right-clicking anywhere in the **Graphics** window.

Figure 3-17 *The* ***Graphics*** *toolbar*

Next, you need to create the sketch, refer to Figure 3-18. For ease of creating the sketch, the entities of the sketch have been numbered, refer to Figure 3-18.

6. From the **Draw** toolbox, choose the **Line** tool which is displayed by default; the Status bar displays the message **Line -- Click, or Press and Hold, for start of line**.

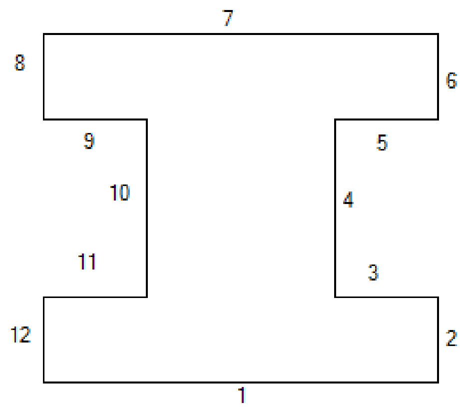


Figure 3-18 *Sketch for Tutorial 1*

A line is defined as the shortest distance between two points. A line is one of the basic sketching entities available in the **DesignModeler** window. To draw a line, choose the **Line** tool from the **Draw** toolbox in the **Sketching Toolboxes** window. On doing so, you will be prompted to specify the start point of the line. Click anywhere in the Graphics window to specify the start point; you will be prompted to specify the end point of the line. Next, click to specify the end point of the line; a line will be created.

After specifying the start point of the line, if you realize that the specified start point is wrong, right-click in the **Graphics** window and choose the **Back** option from the shortcut menu displayed. As a result, the specified start point will get nullified without exiting the **Line** tool.

7. Move the cursor close to the origin in the **Graphics** window and click to specify the start point of the line when the symbol of Coincident Point constraint (**P**) is displayed, refer to Figure 3-19.

After specifying the start point of the line, if you move the cursor horizontally, an H symbol will be displayed on the line. This symbol indicates that if you specify the end point of the line at the current location, a horizontal line will be drawn. Similarly while drawing a line close to an existing line, if C symbol is displayed, it indicates that the end point specified at the current location of the cursor will be coincident with the existing line.

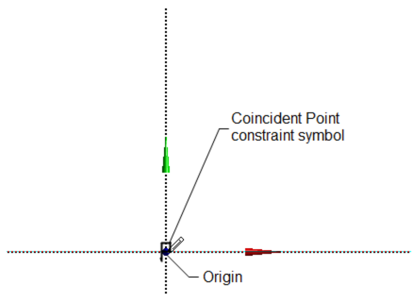


Figure 3-19 *Specifying the start point of the line*

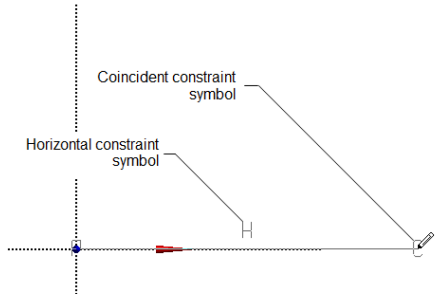


Figure 3-20 *Specifying the end point of the line*

Table 3-1 *List of auto constraints*

Auto Constraint Name	Symbol	Description
Horizontal	H	Makes the entity horizontal
Vertical	V	Makes the entity vertical
Parallel	//	Makes the entity parallel to another entity
Perpendicular	⊥	Makes the entity perpendicular to another entity
Tangent	T	Makes the entity tangent to another entity
Equal Radius	R	Makes the entity equal to another entity
Coincident	C	Makes the end point of an entity coincident with another entity
Coincident Point	P	Makes the end point of the current drawing entity coincident with a point.

8. Move the cursor toward right to some distance and click to specify the end point of line when the symbol of Horizontal constraint (**H**) is displayed, as shown in Figure 3-20. Line 1 is created and its blue color indicates that the line is fully constrained, refer to Figure 3-21.

9. Next, move the cursor close to the end point of line created in the last step; the symbol of Coincident Point constraint (**P**) is displayed. Click to specify the start point of the second line, refer to Figure 3-21.

10. Move the cursor vertically upward; the symbol of Vertical constraint (**V**) is displayed. Click to specify the end point of the line; line 2 is drawn, as shown in Figure 3-22.

11. Now, move the cursor close to the end point of line 2 created in the last step; the symbol of Coincident Point constraint (**P**) is displayed. Click at that point to specify the first point of line 3, as shown in Figure 3-23.

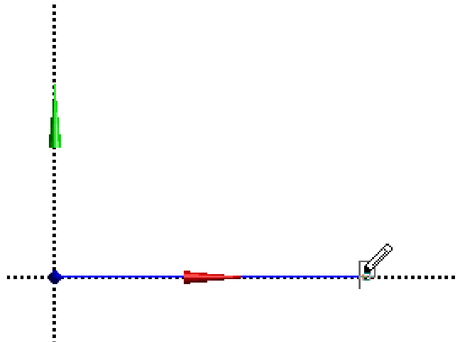


Figure 3-21 Specifying the startpoint of line 2

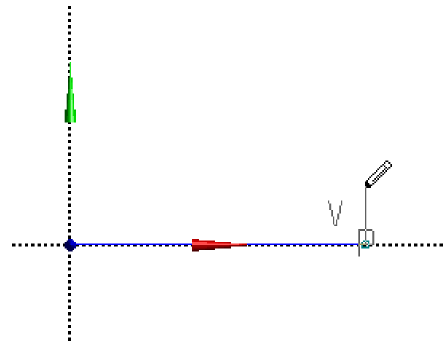


Figure 3-22 Specifying the endpoint of line 2

12. Move the cursor toward left to some distance; the symbol of Horizontal constraint (**H**) is displayed. Next, Click to specify the end point of line 3, as shown in Figure 3-24.

13. Move the cursor close to the end point of line3; the symbol of Coincident Point constraint (**P**) is displayed. Click at that point to specify the start point of line 4, as shown in Figure 3-25.

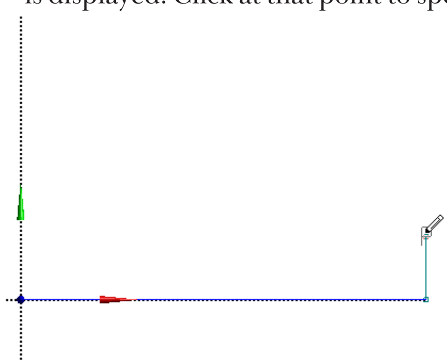


Figure 3-23 Specifying the startpoint of the line 3

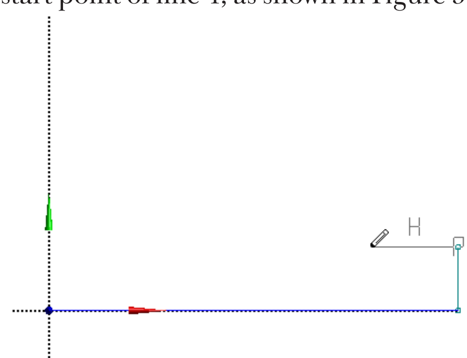


Figure 3-24 Specifying the endpoint of line 3

14. Move the cursor upward to some distance; the symbol of Vertical constraint (**V**) is displayed along the cursor. Click to specify the end point of line 4, refer to Figure 3-26.

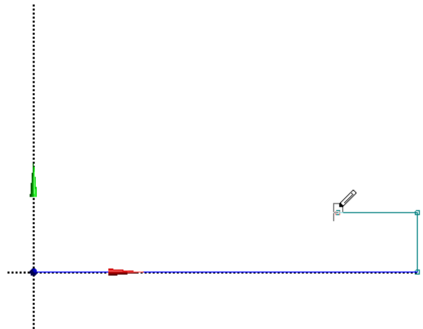


Figure 3-25 Specifying the startpoint of the line 4

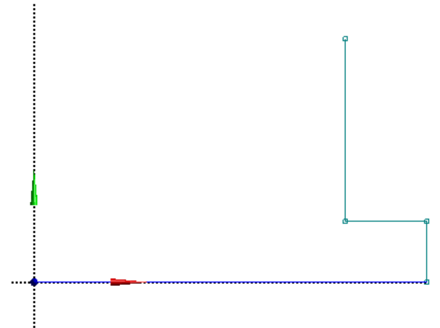


Figure 3-26 Specifying the endpoint of line 4

15. Similarly, draw all the lines specified in the sketch. The sketch drawn is just a representation of the final sketch to be drawn based on dimensions. The final sketch before applying the constraints and dimensions is shown in Figure 3-27.

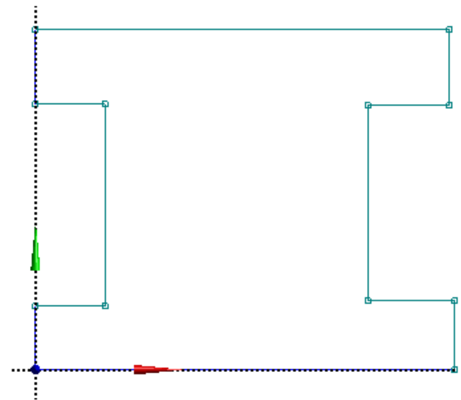


Figure 3-27 *The sketch drawn using automatic constraints*

Note

1. The sketch shown in Figure 3-27 is just for reference and is not created based on dimensions.

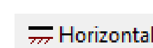
*2. While drawing the sketch, some of the constraints like Horizontal, Vertical and Coincident are automatically applied. You can also apply constraints manually after drawing the sketch, by choosing the desired constraint tool from the **Constraints** toolbox.*

Applying Constraints to the Sketch

After a sketch for the model is drawn, refer to Figure 3-27, you need to constrain the entities of the sketch to restrict some degrees of freedom.

1. Expand the **Constraints** toolbox in the **Sketching Toolboxes** window.

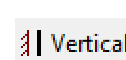
2. Choose the **Horizontal** tool from the **Constrains** toolbox; the **Horizontal -- Select line or ellipse for horizontal constraint** message is displayed in the Status bar indicating that the Horizontal constraints can be applied to entities of the sketch.



Horizontal constraint is used to make a line or the major axis of an ellipse horizontal.

3. Select lines 1, 3, 5, 7, 9, and 11 one by one to make them horizontal, if not already horizontal, refer to Figure 3-20.

4. Next, choose the **Vertical** tool from the **Constraints** toolbox; the **Vertical -- Select line or ellipse for vertical constraint** message is displayed in the Status bar indicating that you can now apply vertical constraints to entities.

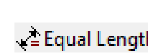


Vertical constraint is used to make a line or the major axis of an ellipse vertical.

5. Select the lines 2, 4, 6, 8, 10, and 12 one by one to apply vertical constraints to them; all these lines become vertical now, refer to Figure 3-28.

   The next step is to apply Equal Length constraints to lines which are equal in length.

6. Choose the **Equal Length** tool from the **Constraints** toolbox of the **Sketching Toolboxes** window. Use the down arrow available next to the **Settings**



toolbox tab to scroll down and view the **Equal Length** tool, if it is not visible by default; you are prompted to select the first line to apply Equal Length constraint.

7. Move the cursor close to line 1; a rectangular box is attached to the cursor, which means that you can now select the line to apply Equal Length constraint. Select line 1; it turns yellow and you are prompted to select the second line.

8. Select 7; the Equal Length constraint is applied and both the lines become equal in length.

9. Similarly, apply Equal Length constraint between the lines 2 and 12, 6 and 8, and 2 and 6 to make these lines equal. In addition, make the lines 3, 5, 9, and 11 equal in length.

10. Next, apply Equal Length constraint to lines 4 and 10.

Applying Dimensions to the Sketch

Now you need to apply dimensions to the sketch. To apply dimensions to the sketch, you need to invoke the desired tool from the **Dimensions** toolbox tab.

1. Choose the **Dimensions** toolbox from the **Sketching Toolboxes** window to expand it.

2. Choose the **General** tool from the expanded **Dimensions** toolbox; the **Details View** window is displayed. Also, you are prompted to select first point or 2D edge for applying Horizontal dimension.

   The **General** tool is used to create dimensions depending upon the entities selected. After the **General** tool is invoked, select an entity from the **Graphics** window to dimension it,

3. Move the cursor close to line 1; a rectangle is attached to the cursor. Select the line 1; the cursor changes to pencil shape. Click to place dimension. As the line is horizontal, H1 is placed above the line 1.

Note

*In **DesignModeler**, the dimensions are placed as symbols. For example, when you invoke the **Horizontal** tool from the **Dimensions** toolbox and then dimension an entity, the dimension that would be placed is **H1**. The values of all the placed dimensions in the sketch can be changed by changing the corresponding dimensional values under the **Dimensions** node in the **Details View** window.*

4. Similarly, select the line 8 and place dimension V2 to the left of the line.

5. Select line 9, and place dimension H3 below the line.

6. Select line 10 and place dimension V4 below V2, as shown Figure 3-28.

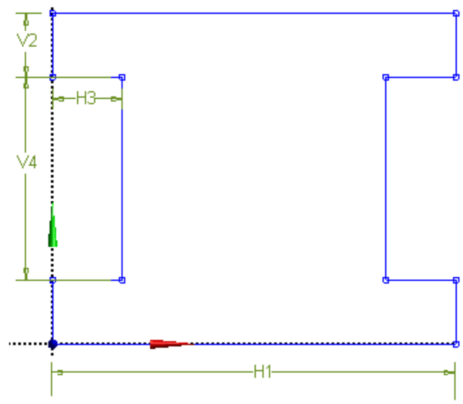


Figure 3-28 *Sketch after applying dimensions*

Note

1. When line 11 is selected for dimensioning, the ***Geometry - DesignModeler*** *warning message box is displayed, as shown in Figure 3-29. This message states that placing this dimension will over-constrain the sketch and you can edit and place this dimension as a reference. On choosing the* ***OK*** *button from the warning message box, the dimension of line 9 will turn red, indicating that the sketch is over-constrained, as shown in Figure 3-30.*

2. If you want to place the over-constrained dimension, place it on the sketch, as shown in Figure 3-31.

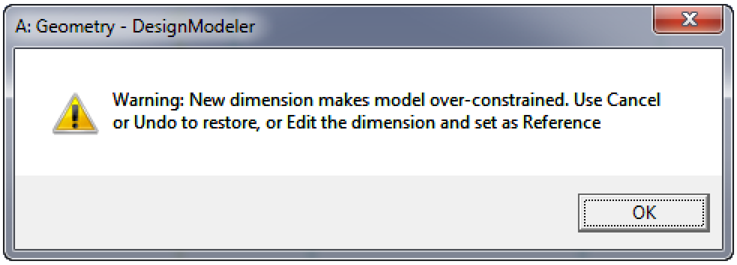


Figure 3-29 *Warning messege*

To undo changes, click anywhere in the **Graphics** window and then press CTRL+Z keys; over-constrained dimensions will vanish.

Tip. *By default, the name of the dimension is displayed on the dimension line. However, if you want to display the value of the dimension or both the name and value of the dimension on the dimension line, choose the* ***Display*** *tool from the* ***Dimensions*** *toolbox. On doing so, the* ***Name*** *and* ***Value*** *check boxes will be displayed on the right of the* ***Display*** *tool. Select the respective check box, the corresponding parameter will be displayed on the dimension lines in the* ***Graphics*** *window. Note that you cannot clear both the check boxes at the same time.*

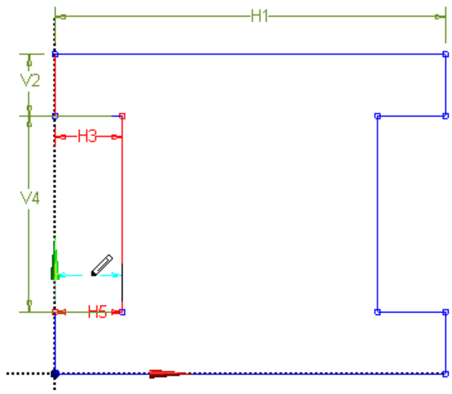


Figure 3-30 Over-constrained sketch

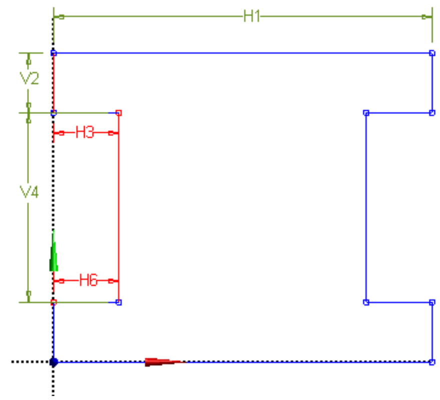


Figure 3-31 Placing dimension on sketch

Assigning Dimensional Values to the Sketch

After dimensions are assigned to the sketch, you need to change their values to get the exact dimensions.

1. In the Tree Outline, choose the **Modeling** tab to view the Tree Outline. On doing so, the **Details View** window is also activated with various nodes active in it.

2. Expand the Tree Outline, if it is not already expanded and then expand the **XYPlane** node to view **Sketch1**. Select **Sketch1**; the **Details View** window with corresponding parameters is displayed.

3. Expand the **Dimensions** node in the **Details View** window, if it is not already expanded.

   There are four dimensions present under this node : **H1**, **H3**, **V2**, and **V4**. These dimensions refer to the dimensions of lines 1, 9, 8, and 10, respectively.

4. To assign a different dimension, click on the edit box to the right of **H1**; the corresponding dimension is highlighted in yellow in the sketch. Enter **120** as the new dimension value and then press ENTER; the lengths of lines 1 and 7 are changed.

Note

1. As you specify the dimensional values to a dimension in the ***Details View*** *window, the length of the entity is changed and the length of the entities which are constrained along with this entity are also modified.*

2. While changing the dimensions, the complete sketch may not fit in the screen of your computer. To fit the sketch in the screen, choose the ***Look At*** *tool from the* ***Graphics*** *toolbar.*

5. Next, in the **Details View** window, click on the edit box next to **H3**; the corresponding dimension is highlighted in the sketch. Enter **30** as the new dimension value and then press ENTER; the length of lines 3, 5, 9, and 11 is changed.

6. Click on the edit box on the right of **V2**; the corresponding dimension is highlighted in yellow in the sketch. Enter **20** in the edit box and then press ENTER; the length of lines 2, 6, 8, and 12 is changed in the sketch.

7. Click on the edit box on the right of **V4**; the corresponding dimension is highlighted in yellow in the sketch. Enter **50** in the edit box and then press ENTER; lines 4 and 10 are modified.

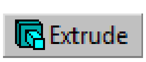
   You will notice that as soon as **H1**, **V2**, **H3**, and **V4** are modified in the **Details View** window, the whole sketch is modified. Therefore, changing the dimension of one entity will ensure that the dimensions of all other entities which are equal in length are also modified.

Extruding the Sketch

After creating the sketch of the outer profile of the I-section, you need to convert it into the base feature by using the **Extrude** tool. The **Extrude** tool is used to add or remove material from the specified sketch along a straight line in the specified direction.

1. Click on the ISO ball placed at the bottom right in the **Graphics** window; the view is changed to Isometric, as shown in Figure 3-32.

2. Choose the **Extrude** tool from the **Features** toolbar; the preview of the extruded feature with default values is displayed in the **Graphics** window, as shown in Figure 3-33. Also, **Extrude1** is added under the three default planes in the Tree Outline, as shown in Figure 3-34.



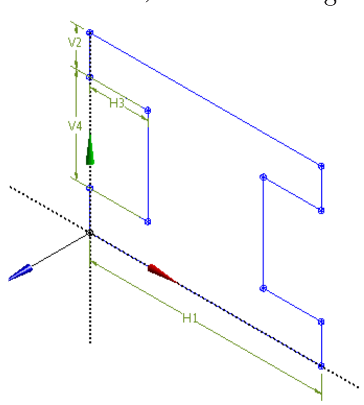


Figure 3-32 *The Isometric view of the sketch*

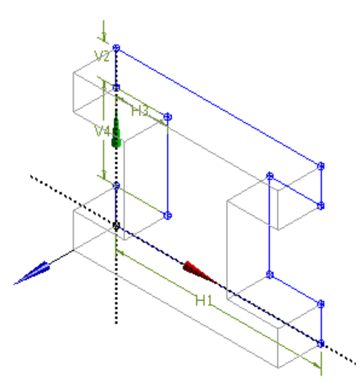


Figure 3-33 *The preview of feature after choosing the* ***Extrude*** *tool*

Note

1. The sketch plane on which you worked last acts as an active plane for any future operation, until you change the plane.

2. To remove material from a feature, you need to have the base feature. Therefore, you cannot use the ***Extrude*** *tool for material removal before you create the base feature.*

The default parameters used for generating the preview of the extruded feature are displayed in the **Details View** window. To get the required shape of the base feature, you need to edit parameters specified in each node of the **Details View** window. The **Both-Symmetric** option is used to add material on both the sides of the sketch with same depth of material.

3. Based on the design requirements, the material should be added normal and symmetrically on both sides of the sketch. To add material to the extrusion, click on **Direction**; a down arrow is displayed on the right of **Direction** in the **Details View** window. Next, click on the down arrow and select the **Both-Symmetric** option from the list displayed, as shown in Figure 3-35. The preview of the extruded feature is changed in the **Graphics** window and now it displays the same amount of material added on both sides of the sketch.

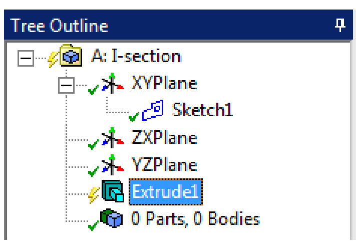


Figure 3-34 *The Tree Outline with* ***Extrude 1*** *added to it*

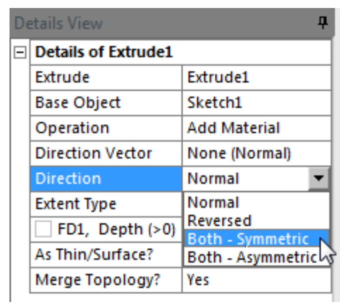


Figure 3-35 *Selecting the* ***Both-symmetric*** *option from the* ***Direction*** *drop-down list in the* ***Detail View*** *window*

The **Reversed** option in the **Direction** drop-down list is used to reverse the direction of material addition. The **Both-Asymmetric** option is used to add material on both the sides of sketch with different values of the depth of material addition.

Next, you need to specify the depth of material addition.

4. Enter **25** in the **FD1, Depth (>0)** edit box of the **Details View** window, refer to Figure 3-36.

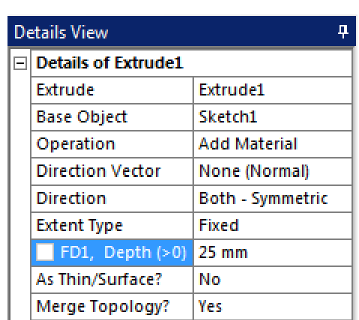


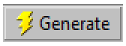
Figure 3-36 *Specifying the depth of material addition in the* ***Details View*** *window*

The total depth of material addition is 50 mm, but the material will be added symmetrically by the same depth on both the sides of the sketch. Therefore, 25 mm has been specified as the depth value.

After all the parameters in the **Details View** window are specified, the preview of the extrude feature is displayed in the **Graphics** window. To create the solid model, you need to generate the extrude feature. Also, notice the thunderbolt symbol displayed before **Extrude 1** in the Tree Outline, which indicates that the feature needs to be generated.

Next, you need to complete the extrusion process with the specified values.

5. Choose the **Generate** tool from the **Features** toolbar; the base feature is created with the specified settings, refer to Figure 3-37. Also, the thunderbolt symbol is changed to green tick mark indicating that the feature is updated.



The **Generate** tool is available in the **Features** toolbar and is used to update the model after any changes are made in it. This tool can also be invoked from the shortcut menu which is displayed on right-clicking anywhere in the **Graphics** window.

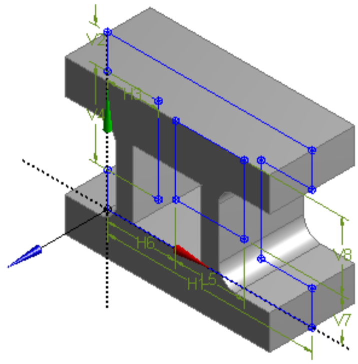


***Figure 3-37** Resulting base feature*

Tip. *Instead of choosing the **Generate** tool from the **Features** toolbar, you can press the F5 key to generate a feature.*

Creating the Second Feature

Next you need to create a rectangular cutout in the I-section, refer to Figure 3-11. To create a rectangular cutout, you will draw a sketch on the XY plane. Next, you will remove material by extruding he sketch.

1. Select **XYPlane** from the Tree Outline; the XY plane becomes the active plane.

2. Choose the **New Sketch** tool from the **ActivePlane/Sketch** toolbar; the new entry with the name **Sketch2** is added under the **XYPlane** node in the Tree Outline, refer to Figure 3-38.

The **New Sketch** tool is available in the **Active Plane/Sketch** toolbar located just above the **Graphics** window. This tool is used to create new sketches for the models.

Notice that the Sketch 1 and the dimensions are still displayed in the **Graphics** window. Since this sketch is not required, you can hide its display.

3. Right-click on **Sketch1** in the Tree Outline and then choose the **Hide Sketch** option from the shortcut menu displayed, refer to Figure 3-39.

Note

*If needed, you can display the sketch and its dimensions again. To do so, right-click on **Sketch1** in the Tree Outline and then choose the **Show Sketch** option from the shortcut menu displayed.*

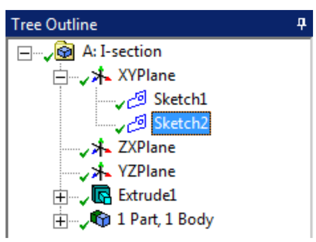


Figure 3-38 *The Tree Outline with **Sketch2** added to it*

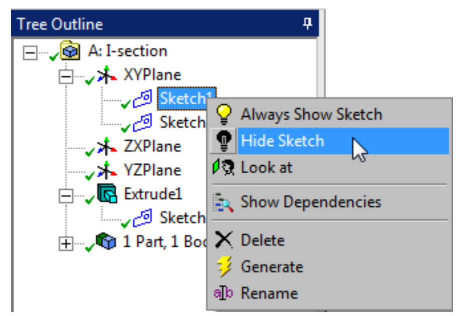


Figure 3-39 *Hiding **Sketch1** by selecting the **Hide Sketch** option from the shortcut menu*

4. Select **Sketch 2** from the Tree Outline and then choose the **Sketching** tab displayed at the bottom of the Tree Outline to invoke the **Sketching** mode.

5. Choose the **Look At** tool from the **Graphics** toolbar; the sketching plane is oriented normal to the viewing direction. Orienting the sketching plane enables you to easily draw the sketch.

6. Choose the **Rectangle** tool from the **Draw** toolbox; the cursor changes into the Draw cursor and you are prompted to specify the first corner point of the rectangle.

In the **DesignModeler** window, there are two methods of drawing rectangle: by specifying two diagonally opposite points of the rectangle and by specifying the three corners of the rectangle The rectangle created by using the first method can be either horizontal or vertical. The rectangle created by the second method is placed at an orientation specified by the user. You can use any of the two methods for drawing the rectangle using the **Draw** toolbox.

7. Specify the first corner point, refer to Figure 3-40; the preview of the rectangle is attached to the cursor and you are prompted to specify the diagonally opposite corner point of the rectangle.

8. Click to specify the second point of the rectangle, refer to Figure 3-40; the rectangle is created, refer to Figure 3-41.

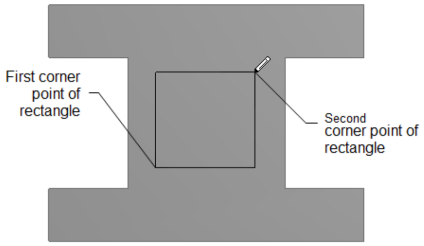


Figure 3-40 *Specifying the first and second corner points of the rectangle*

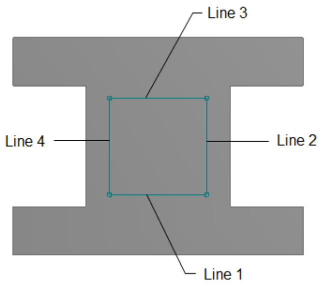


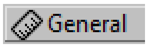
Figure 3-41 *The rectangle created*

9. Press the ESC key to exit the **Rectangle** tool.

   After creating the rectangle, you need to generate its dimensions so that you can specify the size of the rectangle and place it at the desired location.

10. Expand the **Dimensions** toolbox in the **Sketching Toolboxes** window.

11. Choose the **General** tool from the **Dimensions** toolbox, if not chosen by default.



12. Select Line 1; the preview of dimensional constraint attached to the cursor is displayed.

13. Click below the Line 1 to place the dimension; the dimension is generated and its value is displayed on the dimension line.

    Next, you need to generate a dimension between Line 4 and the Y axis.

14. Move the cursor near the lower end point of Line 4 and click when the cursor changes into the Point selection cursor.

15. Click on the Y axis; the preview of the dimension is attached to the cursor.

16. Move the cursor downward and click to place the dimension, refer to Figure 3-42.

17. Similarly, generate other two dimensions, refer to Figure 3-42.

    After the dimensions are placed on the sketch, you need to specify their exact values in the **Details View** window.

18. Edit the value of the first dimension **H6** to 40 in the **Details View** window; the length of the line is changed to 40 mm and is displayed in the **Graphics** window, refer to Figure 3-42.

19. Click on the edit box next to the second dimension **L5** and specify the dimensional value as 40; the dimension of the line is changed to 40, refer to Figure 3-42.

20. Similarly change the dimensional values of **V7** and **V8** dimensions to 25 and 40, respectively. Figure 3-42 shows the final sketch of the cutout feature.

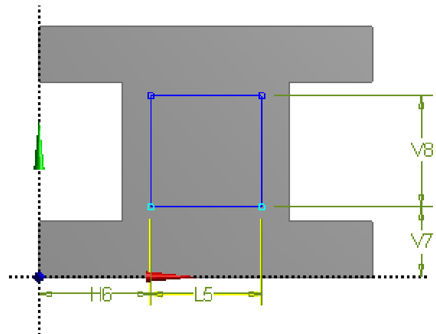


Figure 3-42 *Final sketch of the cutout feature*

Note
The name of the dimension displayed on the ***Details View*** *window may be different in your system.*

21. Choose the **Modeling** tab displayed at the bottom of the **Sketching Toolboxes** window to switch to the **Modeling** mode; the Tree Outline is displayed.

    After you exit the **Sketching** mode, the sketching plane still remains normal to the viewing direction. To proceed further with the feature creation operation, it is advised that you change the view of the sketching plane to Isometric view.

22. Right-click in the **Graphics** window, and then choose the **Isometric View** option from the shortcut menu displayed; the sketch is displayed in the Isometric view.

Tip. *You can also click on the ISO ball (cyan color) displayed on the Triad to change the view of the sketching plane to the Isometric view. The Triad is displayed at the lower right corner of the* ***Graphics*** *window, refer to Figure 3-43.*

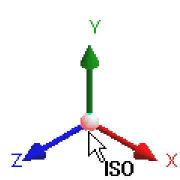


Figure 3-43 *The Triad with the ISO ball*

After the sketch is created, you need to cut the material from the sketch to create the cutout feature. You will use the **Extrude** tool to cut the material.

23. Select **Sketch2** from the Tree Outline and then choose the **Extrude** tool from the **Features** toolbar; the preview of the extruded feature with default values is displayed in the **Graphics** window. Also, a node for the extruded feature with the name **Extrude2** is added below the **Extrude1** node in the Tree Outline.

The default parameters used for generating the preview of the extruded feature are displayed in the **Details View** window. You need to edit values in the **Details View** window to get the desired shape of the cutout.

In this tutorial, material should be removed in the normal direction and symmetrically from both sides of the sketch.

24. Select the **Cut Material** option from the **Operation** drop-down list in the **Details View** window, refer to Figure 3-44; the preview of the material to be removed from the existing base feature is displayed in the **Graphics** window.

25. Select the **Both-Symmetric** option from the **Direction** drop-down list in the **Details View** window, refer to Figure 3-45; the same amount of material is removed from both sides of the sketch.

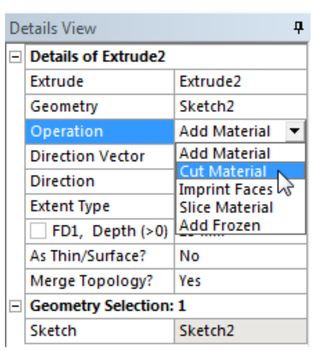


*Figure 3-44 Selecting the **Cut Material** option from the **Operation** drop-down list*

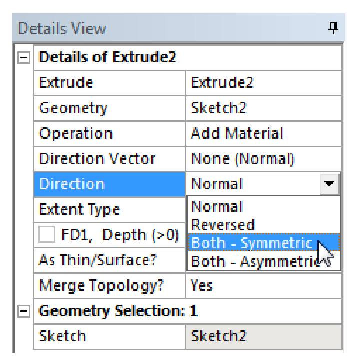


*Figure 3-45 Selecting the **Both-Symmetric** option from the **Direction** drop-down list*

Next, you need to select the method to specify the depth of material removal.

26. Select the **Through All** option from the **Extent Type** edit box in the **Details View** window, refer to Figure 3-46.

The **Through All** option is used to remove material throughout the model in the specified direction. Instead of selecting this option, you can also select the **Fixed** option and then specify the exact depth of material removal, as you did for the base feature.

Notice the yellow thunderbolt symbol displayed at the upper-right corner of the **Extrude 1** node in the Tree Outline, which indicates that the feature needs to be generated.

27. Next, to complete the process of material removal, choose the **Generate** tool from the **Features** toolbar; the cutout feature is created, as shown in Figure 3-47. Also, the yellow thunderbolt symbol is changed to green check mark, indicating that the feature is updated.

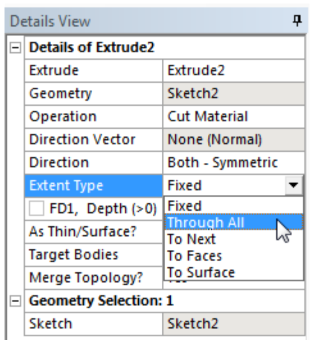


*Figure 3-46 Selecting the **Through All** option from the **Extent Type** drop-down list*

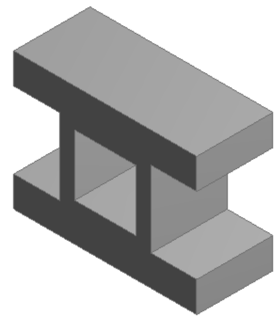


Figure 3-47 The model after creating the cutout

Note

In Figure 3-45, the display of plane has been turned off to see the cutout feature clearly.

In the Tree Outline, certain symbols are displayed on the upper right corner of each feature node. These symbols and their meanings are given in Table 3-2.

***Table 3-2** Various symbols and their meaning*

Symbol	Meaning
Green tick mark	The feature creation succeeded and can be used for further processes.
Yellow thunderbolt mark	The feature has been modified and it needs to be updated.
Yellow tick mark	The feature has been generated, but some warnings are associated with it.
Red exclamation mark	The feature has failed to generate and you may need to redefine the feature.
Blue cross mark	The feature is suppressed and has no influence on the final model.

Creating the Blend Feature

Next you need to create the blend feature (fillet) with the radius of 10 mm at the sharp corners of I-section, refer to Figure 3-11. You will create the fillet on the four edges of the model by using the **Fixed Radius** option from the **Blend** drop-down in the **Features** toolbar.

1. Choose the **Fixed Radius** tool from the **Blend** drop-down in the **Features** toolbar, refer to Figure 3-48; **FBlend1** is attached to the Tree Outline. Also, you are prompted to select the edges to be blended.

The **Blend** tool is used to remove sharp edges in a 3D model. The **Fixed Radius** option is used to create blends with a radius that is constant throughout the edge on which the blend is applied.

2. Press the CTRL key and select the four edges of the model, refer to Figure 3-49. You can use the tools available in the **Graphics** toolbar to rotate the model to view its respective edges.

Note

*To select the edges of a model without rotating it for generating the blend feature, change the display mode to wireframe. To change the display of the model to wireframe mode, choose the **Wireframe** option from the **View** menu. To change the display of the model back to the shaded mode, choose the **Shaded Exterior and Edges** or **Shaded Exterior** option from the **View** menu.*

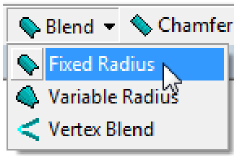


Figure 3-48 *Choosing the **Fixed Radius** option from the **Blend** drop-down*

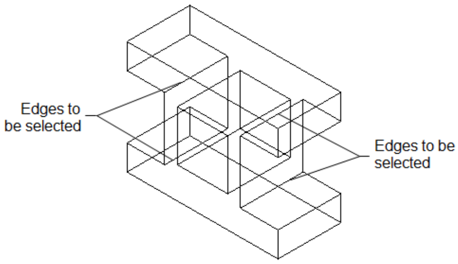
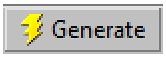


Figure 3-49 *Edges to be selected for creating the blend feature*

3. Choose the **Apply** button from the **Geometry** selection box in the **Details View** window to accept the selection of the edges to be blended, as shown in Figure 3-50.

4. Enter **10** in the **FD1, Radius (>0)** edit box to specify the radius of the blend feature.

5. Next, choose the **Generate** tool from the **Features** toolbar to finish creating the blend with the specified radius. The final model after creating the blend is shown in Figure 3-51.



Note
If you select a face for creating blend, all the edges of the selected face will be blended.

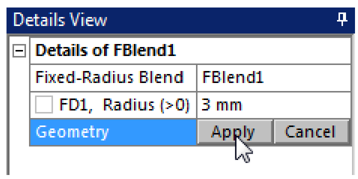


Figure 3-50 *Choosing the* ***Apply*** *button from the* ***Details View*** *window*

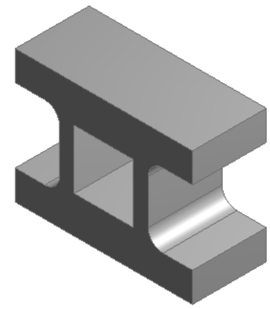


Figure 3-51 *The model after generating the blend feature*

Rotating the View of the Model Dynamically

You can rotate the view of the model dynamically in 3D space so that it can be viewed from all directions. This allows you to view all features clearly.

1. Choose the **Rotate** tool from the **Graphics** toolbar; the cursor changes into the Rotate cursor. Alternatively, right-click in the **Graphics** window and then choose the **Cursor Mode > Rotate** option from the shortcut menu displayed.

The **Rotate** tool is used to rotate the view of the model freely in the **Graphics** window. You can invoke this tool from the **Graphics** toolbar of the **DesignModeler** window. Like other tools of the **Graphics** toolbar, the **Rotate** tool is also a transparent tool, implying that the **Rotate** tool can be invoked even when you are using some other tools.

2. Next, press and hold the left mouse button and drag the cursor to rotate the model in 3D space.

3. Choose the **Rotate** tool from the **Graphics** toolbar to exit it.

Tip. *You can also rotate the view of the model without invoking the* ***Rotate*** *tool. To do so, press and hold the middle mouse button in the* ***Graphics*** *window and drag the cursor.*

Note
After rotating the model dynamically, you can restore the Isometric view again by clicking on the ISO ball (cyan color) in the Triad.

4. Exit the **DesignModeler** window; the **Workbench** window is displayed.

Note
You can close the ***DesignModeler*** *window even without saving the project. However, in this case, the model will be saved automatically but the project will remain unsaved. You need to save the project by choosing the* ***Save*** *button from the* ***Standard*** *toolbar of the* ***Workbench*** *window.*

*If the project is saved then its name is displayed on the title bar of the **Workbench** window. Otherwise it will display **Unsaved Project** on the title bar after closing the unsaved project. Also, the model data that was saved automatically will be lost forever.*

Apart from freely rotating the model, you can also view it using the standard orthographic projections. To view the model in the orthographic direction, move the cursor over the Triad on any axis, the name of the axis will be displayed attached to the cursor. Click on any axis; the view will be oriented normal to the selected axis. Move the cursor in the negative direction of the three orthogonal axes, the system will display the temporary view of the axis and its name with a negative symbol (-). You can also use these negative axes to orient the view of the model. Table 3-3 lists the various orthographic views and axes that need to be selected for achieving the corresponding view and the shortcut key to get it.

Table 3-3 *Various orthographic views and axes that are selected*

Orthographic Views	Triad	Shortcut Key (Num pad)
Right View	+X	3
Left View	-X	7
Top View	+Y	8
Bottom View	-Y	2
Front View	+Z	1
Back View	-Z	9
Default Isometric	ISO ball (cyan color)	5

Saving the Project and Exiting ANSYS Workbench

After visualizing the model and restoring the default Isometric view, you need to save the project and exit ANSYS Workbench.

1. Choose the **Save** button from the **Standard** toolbar; the project is saved.

2. Close the **Workbench** window to close the ANSYS Workbench session.

Note

1. Instead of creating the I-section with cutout as three separate features, you can create it as single feature. To do so, create the sketch as shown in Figure 3-52 and extrude it symmetrically by 25 mm on both sides. If the sketch consists of some closed loops inside the outer loop, they will automatically be subtracted from the outer loop while extruding.

2. In this tutorial, the model has been created as three separate features to explain the various tools and options available in the software. Also, creating the model consisting of various small features makes the editing work easier in case any changes are to be made in the design at later stages.

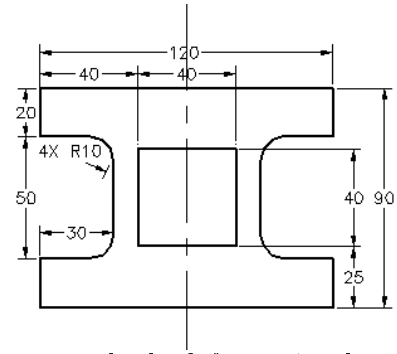


Figure 3-52 The sketch for creating the complete model as single feature

Tutorial 2

In this tutorial, you will create the solid model of the Spring Plate shown in Figure 3-53. The dimensions of the model are shown in Figure 3-54. Thickness of the Spring Plate is 2mm. Save the project with the name *c03_ansWB_tut02* at the location *C:\ANSYS_14\c03\Tut02*

(Expected time: 30 min)

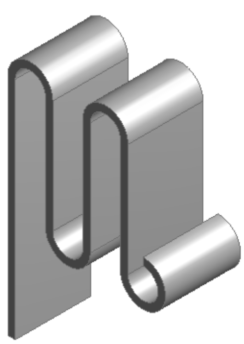


Figure 3-53 *Model for Tutorial 2*

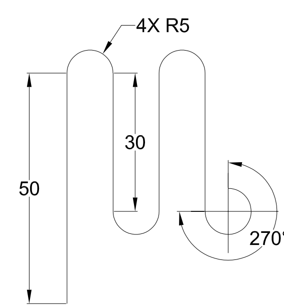


Figure 3-54 *Sketch and dimensions for Tutorial 2*

For the ease of creating the model, the sketch has been divided into small segments, as shown in Figure 3-55.

The following steps are required to complete this tutorial:

a. Start ANSYS Workbench and add the **Geometry** component.
b. Create the sketch.
c. Apply constraints to the sketch.
d. Apply dimension to the sketch.
e. Extrude the sketch.
f. Save the model.

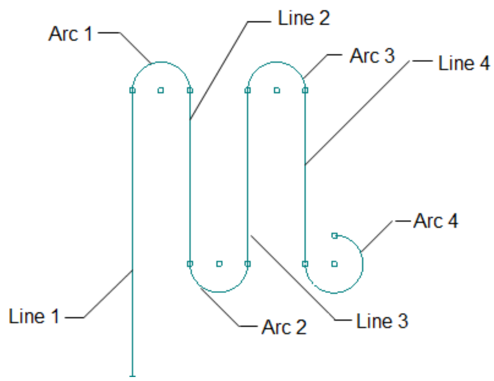


Figure 3-55 *Various entities of the sketch*

Starting ANSYS Workbench and Adding Geometry Component System

In this section, you need to start ANSYS Workbench and then add a component system to the project.

1. Choose **All Programs > ANSYS 14.0 > Workbench** from the start menu; the **Workbench** window is displayed.

Note

*If the **Getting Started** window is displayed along with the **Workbench** window, you need to close it by choosing the **OK** button.*

After invoking the **Workbench** window, you have to add appropriate analysis system or the component system to the **Project Schematic** window. In this tutorial you will create a solid model using the **Geometry** component system.

2. Right-click in the **Project Schematic** window and choose **New Component Systems > Geometry** from the shortcut menu displayed; the **Geometry** component system is added to the **Project Schematic** window, as shown in Figure 3-56.

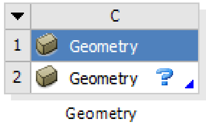


Figure 3-56 *The **Geometry** component system added to the project*

By adding the **Geometry** component system to the **Project Schematic** window, you can have a stand-alone system for the model to be analyzed. When any change is made on the geometry using the **Geometry** component system, the changes are displayed in all analysis systems with which the geometry is shared.

Note

*A component system can also be added by dragging it from the **toolbox** and then dropping it in the **Project Schematic** window or by double-clicking on the **Geometry** option displayed under the **Component Systems** toolbox in the **Toolbox** window. But in this tutorial, you will use the method mentioned in Step 2.*

3. Double-click on the name field of the **Geometry** component system and enter **Spring Plate** to rename it.

Note
Once the **Geometry** *component system is added to the* ***Project Schematic*** *window, you can rename it when the name field gets highlighted at the bottom of the component system in blue.*

4. Choose the **Save** button from the **Standard** toolbar; the **Save As** dialog box is displayed.

5. Browse to the location *C:\ANSYS_WB\c03* and then create a sub folder with the name **Tut02** in the *c03* folder.

6. Enter **c03_ansWB_Tut02** in the **File name** edit box in the **Save As** dialog box and then choose the **Save** button in it; the project is saved with the specified name.

Creating the Sketch

After the component system is added to the project, you need to create the sketch for the Spring Plate model. To do so, invoke the **DesignModeler** window and perform the following steps to complete the sketch.

1. Right-click on the **Geometry** cell of the **Spring Plate** component system; a shortcut menu is displayed.

2. Choose the **New Geometry** option from the shortcut menu; the **DesignModeler** window along with the **ANSYS Workbench** dialog box is displayed.

3. Select the **Millimeter** radio button to specify the unit of length and then choose the **OK** button; the dialog box is closed and you are directed to the **DesignModeler** window to proceed with the modeling.

The **Millimeter** radio button is selected to specify the unit of length as millimeter.

4. In the Tree Outline, expand the **A: Spring Plate** node, if not already expanded; the components of the Tree Outline are displayed.

5. Select **XYPlane** from the Tree Outline; the XY plane becomes the active plane.

6. Choose the **New Sketch** tool from the **Active Plane/Sketch** toolbar; the new entry with the name **Sketch1** is added under the **XYPlane** node in the Tree Outline, refer to Figure 3-57.

7. Right-click on **Sketch1** node under the **XYPlane** node to display a shortcut menu.

8. Choose the **Look At** tool from the shortcut menu displayed; the sketching plane is oriented normal to the viewing direction.

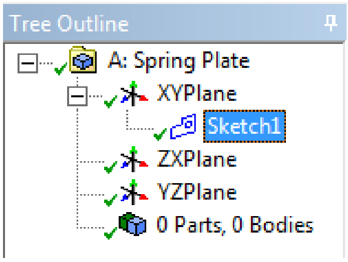


Figure 3-57 *The Tree Outline with* ***Sketch1*** *added to it*

9. Choose the **Sketching** tab from the bottom of the Tree Outline to switch to the **Sketching** mode.

10. Next, click on the **Draw** toolbox to expand it, if it is not already expanded.

11. In the **Draw** toolbox, choose the **Line** tool; you are prompted to specify the start point of the line. Also, the cursor is replaced with Draw cursor.

12. Move the cursor close to the origin; the symbol of Coincident constraint (P) is displayed, as shown in Figure 3-58.

13. Click on the origin to specify the start point of the line; you are prompted to specify the end point of the line.

14. Move the cursor upward along the axis such that the symbol of Vertical constraint (**V**) is displayed along the path of the cursor. Move the cursor to some distance along the Y axis and then click to specify the end point of line, as shown in Figure 3-59. The line is created and is displayed in blue color indicating that it is fully constrained.

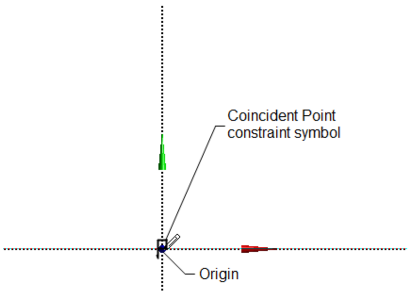


Figure 3-58 *Specifying the start point of the line*

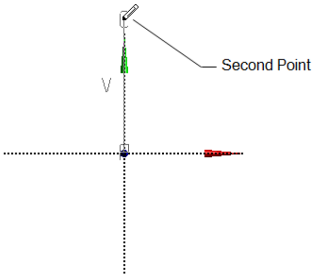


Figure 3-59 *Specifying the end point of the line*

After Line 1 is created, you now need to create the arc, Arc 1 (refer to Figure 3-55 for naming conventions used in this tutorial).

15. Now, invoke the **Arc by Tangent** tool from the **Draw** toolbox.

The **Arc by Tangent** tool is used to create an arc tangent to a line. Choose the **Arc by Tangent** tool from the **Draw** toolbox and specify a point on any existing sketched entity, the arc to be created will maintain tangency with the specified point. Also, the symbol of Tangent constraint (**T**) is displayed when you move the cursor close to the line. Next, specify the second point to define the end point of the arc.

16. Move the cursor close to end point of Line 1; the symbol of Point Coincident (P) is displayed. Next, click to specify the start point for arc 1, as shown in Figure 3-60.

17. Move the cursor toward right to some distance and click to specify the end point of arc 1; the arc is created, as shown in Figure 3-61.

Note

*In case the arc is created in a direction opposite to the desired direction, right-click to display a shortcut menu and then choose **Reverse** from it.*

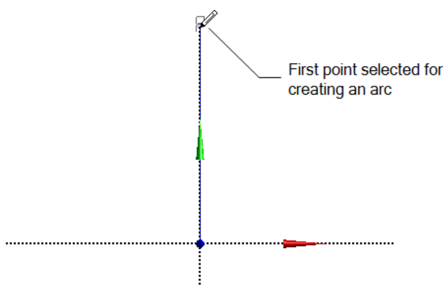


Figure 3-60 *Specifying the start point of the arc*

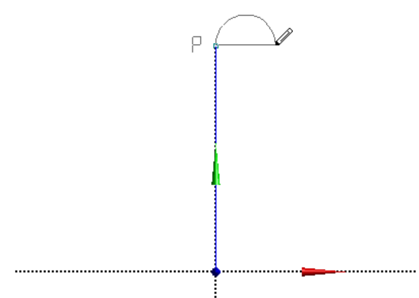


Figure 3-61 *Specifying the endpoint of the arc*

18. Now you need to create a line tangent to the arc 1. Invoke the **Tangent Line** tool from the **Draw** toolbox and move the cursor close to the end point of arc 1; the symbols of Tangent constraint (**T**) and the Point Coincident constraint (**P**) are displayed. Click to specify the start point of the line 2, as shown in Figure 3-62.

19. Specify the end point for line 2, as shown in Figure 3-63.

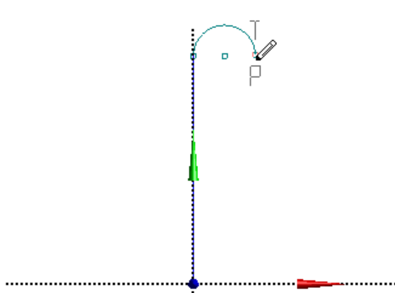


Figure 3-62 *Specifying the startpoint of the line*

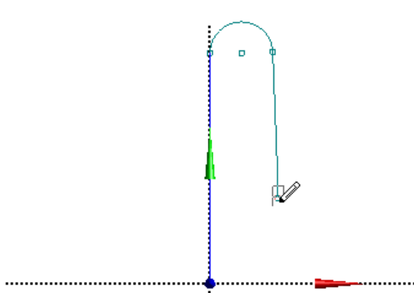


Figure 3-63 *Specifying the endpoint of the line*

You can create a tangent line by using the **Tangent Line** and **Line by 2 Tangents** tools. To create a tangent line using the **Tangent Line** tool, choose this tool from the **Draw** toolbox; the cursor will change to the Draw cursor. Next, select a curved sketch; the preview of the line will be displayed attached to the Draw cursor. Next, specify the end point of the line to define the length of the line, refer to Figure 3-61. You can also create lines by using the **Line by 2 Tangents** tool. To do so, invoke this tool from the **Draw** toolbox and select two existing curve sketched entities; the tangent line will be created. In case more than one tangent locations are available on the selected entities, the tangent line will be generated using the tangent location nearest to the point of selection.

Note

The lines created using the ***Tangent Line*** *tool may not always be vertical. They can be made vertical by applying Vertical constraints.*

Now, you need to create an arc.

20. Invoke the **Arc by Tangent** tool from the **Draw** toolbox of the **Sketching** tab; you are prompted to specify the start point of the arc 2.

21. Move the cursor close to the end point of line 2; the symbol of Coincident constraint (**P**) sign is displayed. Click to specify the start point of arc 2, as shown in Figure 3-64.

22. Move the cursor to the right till the symbol of Equal Radius constraint (R) is displayed. Then click to specify the end point of arc 2; arc 2 is created, as shown in Figure 3-65.

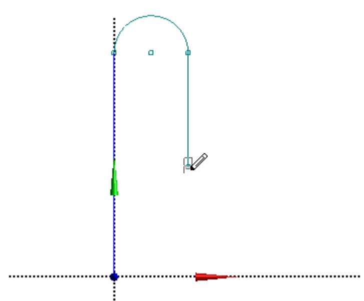


Figure 3-64 *Specifying the start point of the arc*

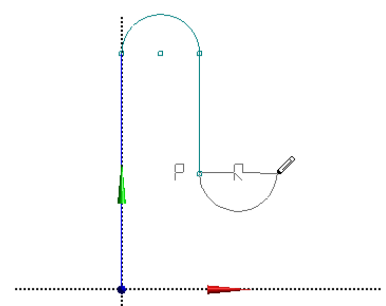


Figure 3-65 *Specifying the end point of the arc*

Now you need to draw the line 5.

23. Invoke the **Tangent Line** tool from the **Draw** toolbox; you are prompted to specify the start point of line. Move the cursor close to the end point of arc 2 and click to specify the start point of line 3 when the symbols of Tangent constraint (T) and Point Coincident constraint (P) are displayed, as shown in Figure 3-66.

24. Move the cursor upward to some distance. While moving the cursor, the symbol of Vertical constraint (**V**) is displayed along with the preview of the line. Click to specify the end point of line 3, as shown in Figure 3-67.

25. Next, invoke the **Arc by Tangent** tool; you are prompted to select a 2D edge or end point of a line.

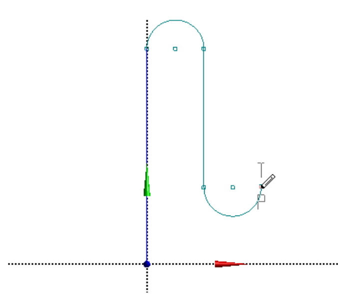


Figure 3-66 *Specifying the start point of the line 3*

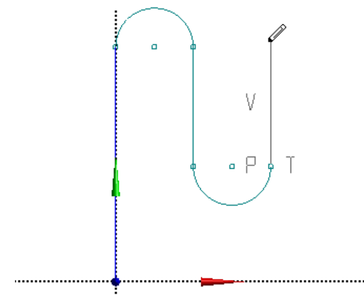


Figure 3-67 *Specifying the endpoint of the line 3*

26. Select the end point of line 3, as shown in Figure 3-68; you are prompted to specify the end point of arc 4.

27. Move the cursor toward right till the symbol of Equal Radius constraint (R) is displayed. Click to specify the end point of arc 4, as shown in Figure 3-69.

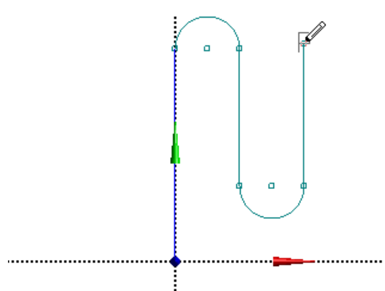


Figure 3-68 *Specifying the startpoint of the arc 4*

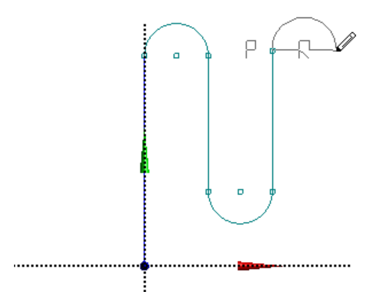


Figure 3-69 *Specifying the endpoint of the arc 4*

Next, you need to draw the line 4 and arc 4 to finish the sketch.

28. Invoke the **Tangent Line** tool from the **Draw** toolbox; you are prompted to specify the start point of the line.

29. Move the cursor close to end point of arc 3; the symbols of Tangent constraint (T) and the Point Coincident constraint (P) are displayed attached to the cursor.

30. Select the end point of arc 3 to specify it as the start point of line 4, as shown in Figure 3-70. Also, you are prompted to specify the end point of line 4.

31. Move the cursor downward; the preview of the line will be displayed. Also, the symbol of Vertical constraint (V) gets attached to the preview. After moving the cursor to some distance, click to specify the end point of line; line 4 is drawn, as shown in Figure 3-71.

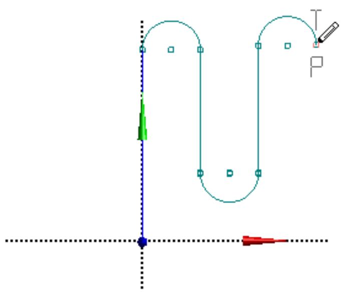


Figure 3-70 *Specifying the start point of the line*

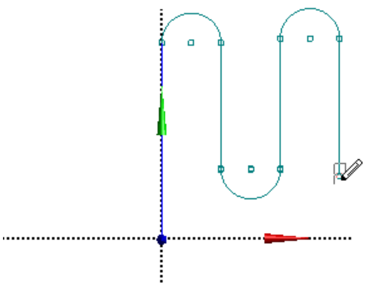


Figure 3-71 *Specifying the endpoint of the line*

Now, you need to draw the arc 4 to complete the sketch. Note that the arc to be drawn should be made with an angle of 270 degrees.

32. Invoke the **Arc by Tangent** tool from the **Draw** toolbox; you are prompted to select an edge or end point of line to specify the start point of arc. Select the end point of line 4 as the start point of arc 4, as shown in Figure 3-72.

33. Next, move the cursor toward right and then create an arc similar to the one shown in Figure 3-73.

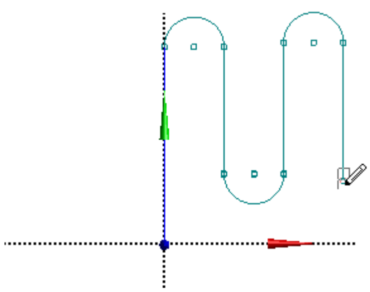


Figure 3-72 *Specifying the start point of arc 4*

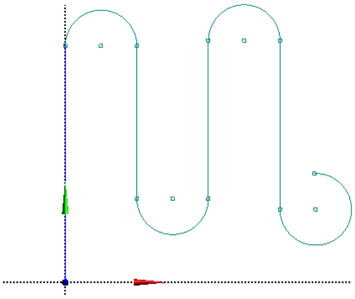


Figure 3-73 *Final sketch after the arc 4 is drawn*

Applying Constraints to the Sketch

After the sketch is drawn, you need to apply constraints to the entities.

1. Expand the **Constraints** toolbox from the **Sketching Toolboxes** window.

2. Choose the **Equal Length** tool from the **Constraints** toolbox; you are prompted to select the first line on which Equal Length constraint is to be applied.

The **Equal Length** tool is used to force two linear entities to maintain the same length.

3. Select line 2; you are prompted to select second line for equal constraint.

4. Select line 3; line 2 and line 3 become equal in length.

5. As the **Equal Length** tool is still active, select line 3 and then line 4; both the lines become equal in length.

   Now you need to apply the Equal Radius constraint between all the arcs.

6. Choose the **Equal Radius** tool from the **Constraints** toolbox; you are prompted to select the first arc or circle to apply the constraint.

The **Equal Radius** tool is used to force two circular entities to become equi-radius.

7. Select arc 1; you are prompted to select the second arc to apply the Equal Radius constraint.

8. Select arc 2; arcs 1 and 2 become equal in radius.

9. Similarly, apply the Equal Radius constraint between arcs 2 and 3 and then between arcs 3 and 4; all the arcs become equal in radius.

Assigning Dimensions to the Sketch

After the sketch is drawn and the constraints are applied, you need to assign dimensions to the entities.

1. Choose the **General** tool from the **Dimensions** toolbox; you are prompted to select a 2D edge or line that you want to dimension.

2. Select line 1; the shape of the cursor changes to Draw cursor. Also, the preview of the dimension is attached to the cursor.

3. Click on the left of line 1 to place the dimension, as shown in Figure 3-74.

4. Place all other dimensions to their respective places, as shown in Figure 3-75.

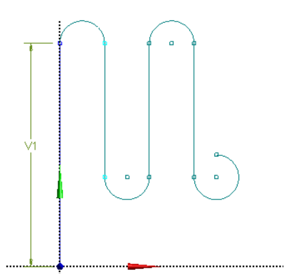


Figure 3-74 *Placing the dimension of line 1*

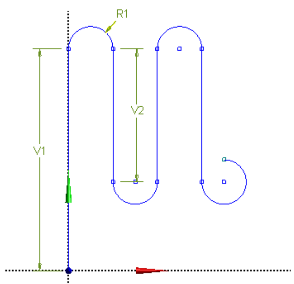


Figure 3-75 *Sketch after all the dimensions are placed*

You need to place only three dimensions in the sketch, remaining dimensions will be applied automatically as the Equal Radius and Equal Length constraints have been applied to them earlier in this tutorial. After the dimensions are placed, you need to assign values to each of them.

5. Choose the **Modeling** tab located at the bottom of the **Sketching Toolboxes** window; the **Modeling** mode becomes active.

6. Expand the **XYPlane** node in the Tree Outline, if not already expanded.

7. Select **Sketch1** under the expanded **XYPlane** node; the corresponding **Details View** window is displayed.

8. In the **Details View** window, expand the **Dimensions** node if not already expanded.

The **Dimensions** node in the **Details View** window displays the dimensions that are placed on the sketch in the **Graphics** window.

9. Click on the **R1** edit box in the **Details View** window and enter **5**; the radius of all arcs is changed to 5 mm instantaneously.

Note
When you click on any edit box under the ***Dimensions*** *node in the* ***Details View*** *window, the corresponding dimension in the* ***Graphics*** *window is highlighted in yellow color.*

10. Next, click on the **V1** edit box and then enter **50**; the length of Line1 is changed.

11. Similarly, modify dimension **V2** to 30.

Figure 3-76 shows the **Dimensions** node of the **Details View** window.

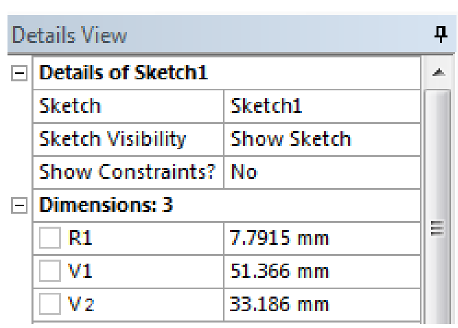


Figure 3-76 *The* ***Dimensions*** *node in the* ***Details View*** *window*

Creating the Extrude Feature

After the sketch is fully constrained and dimensions are applied, you now need to add material to the sketch. This is done by using the **Extrude** tool.

1. Choose the **Extrude** tool from the **Features** toolbar; **Extrude1** is attached to the Tree Outline. Also, the preview of extrusion is displayed in the **Graphics** window.

2. Click on the ISO ball available on the bottom right corner of the **Graphics** window; the view is changed to isometric. Figure 3-77 shows the ISO ball with the Triad and Figure 3-78 shows the Isometric view of the model.

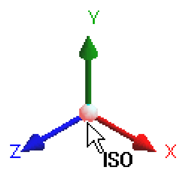


Figure 3-77 The Triad with the ISO ball

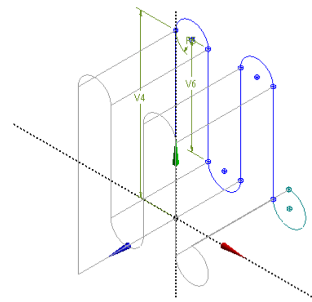


Figure 3-78 The Isometric view of the model

Now, you need to set the parameters for extrusion. The parameters of extrusion are set in the **Details View** window.

3. Select **Geometry** in the **Details View** window to display the **Apply** and **Cancel** buttons, if not already displayed.

4. Choose the **Apply** button in the **Geometry** selection box to specify the sketch as the sketch to be extruded.

5. In the **Operations** drop-down list in the **Details View** window, select **Add Material** if it is not already selected.

6. In the **Directions** drop-down list, select **Both - Symmetric**; the material is added to both sides of the plane.

7. In the **FDI, Depth (>0)** edit box, enter **10** as the depth of extrusion.

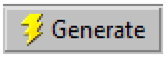
8. In the **As Thin/Surface?** drop-down list, select **Yes**; the **FD2, Inward Thickness (>0)** and **FD3, Outward Thickness (>0)** edit boxes are activated.

The **Thin/Surface** tool is used to create surface out of sketches or create shell features in models.

9. Click on the **FD2, Inward Thickness (>0)** edit box and enter **1** as the value of thickness in the inward direction.

10. Click on the **FD2, Outward Thickness (>0)** edit box and enter **1** as the value of thickness in the outward direction.

11. After specifying all the parameters in the **Details View** window, choose the **Generate** tool from the **Features** toolbar; the geometry is extruded, as shown in Figure 3-79.



The final model for Tutorial 2 is shown in Figure 3-80. The axes and the sketch have been turned off for a better visibility.

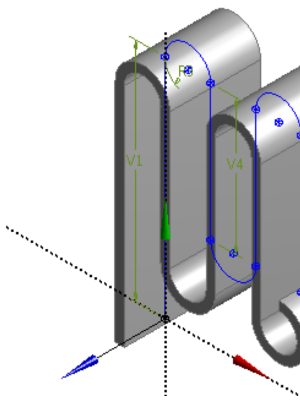


Figure 3-79 *The generated feature*

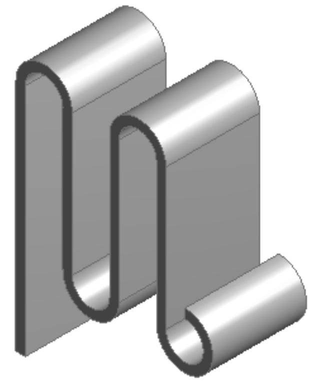


Figure 3-80 *The final model for Tutorial 3*

12. Close the **DesignModeler** window; the **Workbench** window is displayed.

Saving the Model

After the model is created, you now need to save your work.

1. Choose the **Save** button from the **Standard** toolbar to save the model.

2. Exit the **Workbench** window.

Tutorial 3

In this tutorial, you will create the solid model of the clamp shown in Figure 3-81. The sketch of the model and its dimensions are shown in Figure 3-82. Save the project with the name *c03_ansWB_tut03* at the location *C:\ANSYS_WB\c03\Tut03*. **(Expected time: 30 min)**

Figure 3-81 *Model for Tutorial 3*

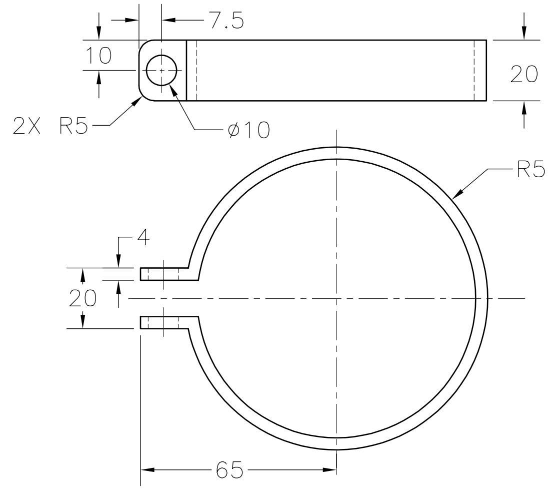


Figure 3-82 *Dimensions of the clamp*

The following steps are required to complete this tutorial:

a. Start ANSYS Workbench 14.0.
b. Add the **Geometry** component system to the project.
c. Start **DesignModeler** window and specify unit system.
d. Draw the sketch for the base feature on the XYPlane.
e. Create the base feature.
f. Create the circular cutout.
g. Create the blend feature.
h. Save the project and exit the ANSYS Workbench session.

Starting ANSYS Workbench and Adding Geometry Component System

First, you need to start ANSYS Workbench 14.0 and then add a component system to the project.

1. Choose **All Programs > ANSYS 14.0 > Workbench 14.0** from the Start menu; the **Workbench** window along with the **Getting Started** window is displayed.

2. Choose the **OK** button in the **Getting Started** window to close it.

   After invoking the **Workbench** window, you have to add appropriate analysis system or a component system to the **Project Schematic** window. In this tutorial, you will create a solid model using the **Geometry** component system.

3. Right-click in the **Project Schematic** window and choose **New Component Systems > Geometry** from the shortcut menu displayed, as shown in Figure 3-83; the **Geometry** component system is added to the project and is displayed in the **Project Schematic** window.

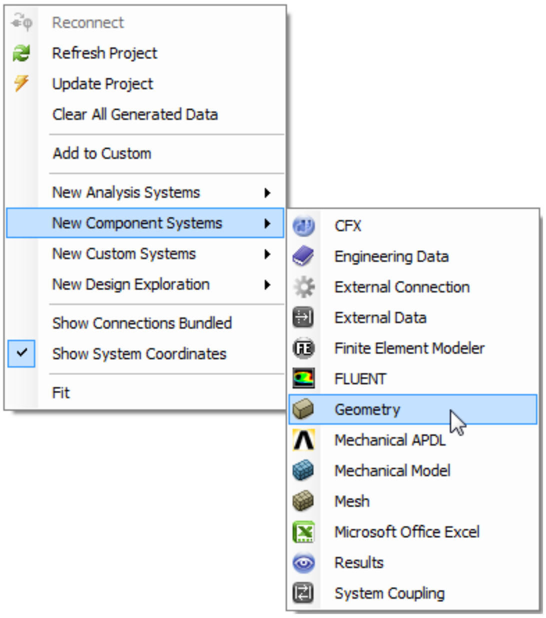


*Figure 3-83 Choosing the **Geometry** option from the shortcut menu displayed on choosing the **New Component System** option*

4. Once the **Geometry** component system is added to the **Project Schematic** window, the name field at the bottom of the component system is highlighted in blue. If it is not highlighted, double-click on the name field and enter **Clamp**, refer to Figure 3-84. The component system is renamed as **Clamp**.

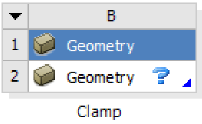


*Figure 3-84 The **Geometry** component system added to the project*

The blue question mark on the right of the **Geometry** cell indicates that an immediate action is required for this cell and the user cannot proceed further without fixing this cell.

5. In the **Workbench** window, choose the **Save** button from the **Standard** toolbar; the **Save As** dialog box is displayed.

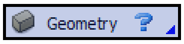
6. Browse to the location *C:\ANSYS_WB\c03* and then create a sub folder with the name **Tut03** in the *c03* folder and then choose the **Open** button from the **Save As** dialog box.

7. Enter **c03_ansWB_Tut03** in the **File name** edit box in the **Save As** dialog box and then choose the **Save** button in it; the project is saved with the specified name.

Starting DesignModeler Window and Specifying Unit System

To define the geometry, you need to start the **DesignModeler** window associated with this cell.

1. Double-click on the **Geometry** cell in the **Clamp** component system; the **DesignModeler** window along with the **ANSYS Workbench** dialog box is invoked.



The **ANSYS Workbench** dialog box is used to specify the unit system for creating the model.

Note
*You can also invoke the **DesignModeler** window by choosing the **New Geometry** option from the shortcut menu that is displayed by right-clicking on the **Geometry** cell of an analysis system or component system.*

2. Choose the **Millimeter** radio button and then choose the **OK** button from the **ANSYS Workbench** dialog box to accept the specified unit system.

Drawing the Sketch for the Base Feature

You need to create the sketch for the base feature on the XY plane which is the default plane. Therefore, you need not specify the plane.

1. Choose the **Sketching** tab displayed in the lower left corner of the Tree Outline to invoke the **Sketching** mode.

Note
*1. To select a plane other than the default plane (XY), select it from the **New Plane** drop-down list in the **Active Plane/Sketch** toolbar.*

*2. To insert a sketch instance or create a new sketch on a plane other than the default plane, you can right-click on the plane node in the Tree Outline to display a shortcut menu. Next, choose **Insert Sketch Instance** from it; a sketch instance will be displayed under the desired node.*

Now, you need to orient the sketching plane normal to the viewing direction, so that you can easily draw the sketch on the specified plane.

2. Choose the **Look At** tool from the **Graphics** toolbar to orient the model normal to the viewing direction.

Note
*You can also orient a plane normal to the viewing direction by choosing the **Look At** tool from the shortcut menu displayed on right-clicking on the sketch instance.*

3. Choose the **Circle** tool from the **Draw** toolbox; you are prompted to specify the center of the circle.

In **DesignModeler**, you can draw circles by using two different methods. In the first method, specify the center point of the circle and then define its radius. In the second method, you need to specify the three existing drawing entities in the **Graphics** window with which the new circle to be created must maintain the tangency relation. This type of circle is known as tri-tangent circle. You can choose any of the two methods for drawing circles.

4. Move the cursor close to the origin in the **Graphics** window and click; the symbol of Coincident Point constraint (**P**) is displayed. After specifying the center point of the circle, the preview of the circle is displayed attached to the Draw cursor. Also, you are prompted to specify the radius of the circle.

5. Move the cursor away from the center and click; a circle is created, as shown in Figure 3-85.

6. Press the ESC key to exit the **Circle** tool.

   After creating the first entity of any sketch, it is better to generate its dimensions first. This gives you a fair idea about the graphics space required to complete the sketch. Also, it helps you decide the comparative size of other sketched entities to complete the outer profile. Now, you will generate the radius dimension of the circle that you created in the previous step and change its value to 50mm.

7. Expand the **Dimensions** toolbox in the **Sketching Toolboxes** window.

8. Choose the **Radius** tool from the **Dimensions** toolbox; you are prompted to select the entity to place the dimension.

The **Radius** tool is used to generate dimensions for circles, arcs, or ellipses. When you select an arc for dimensioning, the radius dimension is generated, and when you select an ellipse, the major and minor dimensions are generated.

9. Move the cursor over the circle and select it; the preview of dimension is attached with the cursor.

10. Move the cursor away from the circle and click to place the dimension. The dimension is generated and its name is displayed on the dimension line.

    Other details of the dimension are displayed in the **Details View** window.

11. In the **Details View** window, click in the edit box displayed on the right of the dimension name (**R1**) under the **Dimensions: 1** node.

Note

The name of the dimension displayed on the dimension line can be different in different systems. To avoid such confusion and to facilitate the proper explanation, refer to the corresponding screen captures of the dimensions.

12. Enter **50** in this edit box and press the ENTER key; the radius of circle changes to 50 mm and is displayed in the **Graphics** window, refer to Figure 3-86.

Now, you need to complete the remaining part of sketch for the base feature.

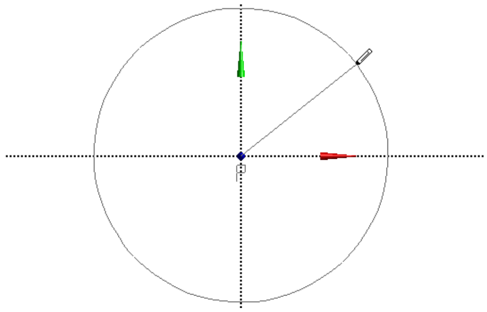


Figure 3-85 *Specifying the radius of the circle*

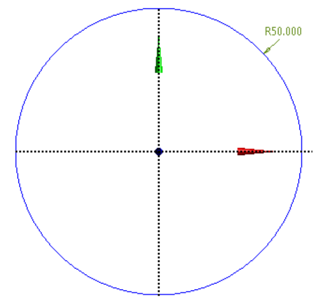
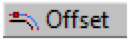


Figure 3-86 *The circle after editing the dimension value*

13. Click on the **Modify** toolbox in the **Sketching Toolboxes** window; the **Modify** toolbox expands.

The **Modify** toolbox contains various tools such as **Fillet**, **Chamfer**, **Trim**, **Extend**, **Split**, and so on. These tools are used to edit sketched entities in the **Sketching** mode.



14. Scroll down the **Modify** toolbox to display other tools, refer Figure 3-87. Next, choose the **Offset** tool from the **Modify** toolbox; you are prompted to select the line or arc to offset.

The **Offset** tool is used to draw multiple parallel lines, parallel polylines, concentric circles, concentric curves, concentric arcs, and so on. When you choose the **Offset** tool from the **Modify** toolbox, you will be prompted to select the entities to be offset. The entities selected for offsetting must be connected end to end and should form open or closed profile.

15. Select the circle and right-click in the **Graphics** window; a shortcut menu is displayed.

16. Choose the **End selection / Place offset** option from the shortcut menu; the preview of the entity to be offset is displayed attached to the cursor.

Note
If you have made a wrong selection by mistake, then choose the ***Clear Selection*** *option from the shortcut menu and select again the correct entity to be offset.*

17. Move the cursor inside the circle and click to specify an offset distance, refer to Figure 3-88.

18. Right-click in the **Graphics** window and choose the **End** option from the shortcut menu displayed; the **Offset** tool is deactivated.

After editing the sketch, if you want to exit the current selection and select other entity to offset, choose the **Clear Selection** option from the shortcut menu.

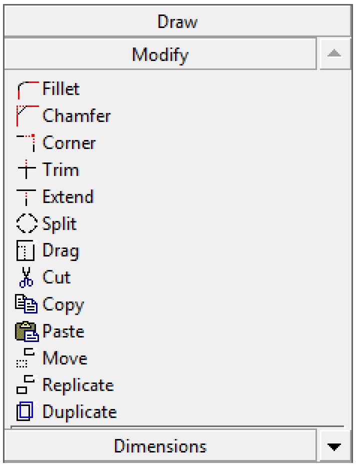


Figure 3-87 *Tools in the **Modify** toolbox*

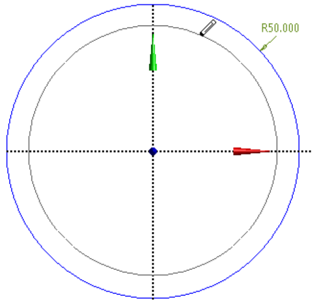
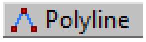


Figure 3-88 *Specifying the offset distance*

After the circular entities are created, you need to create the linear entities.

19. Expand the **Draw** toolbox and invoke the **Polyline** tool; you are prompted to specify the start point of the line.



You need to define the start point and end point of the line, each time you want to create a line using the **Line** tool. But if you want to create a continuous connected line where the start point of the next line is automatically defined as the end point of the previous line, choose the **Polyline** tool from the **Draw** toolbox. Specify the start and end points of the first line; the first line will be created and the preview of another line whose start point is the end point of the first line will be attached to the Draw cursor. Specify the third point; the second line will be created and the preview of the third line whose start point will be the end point of the second line will be displayed attached to the Draw cursor. Keep on specifying the points to create continuous lines. To stop creating the polyline and exit the **Polyline** tool, right-click in the **Graphics** window and choose the **Open End** option from the shortcut menu.

20. Move the cursor near the circumference of the outer circle and click when the symbol of Coincident constraint (C) is displayed, refer to Figure 3-89.

21. Move the cursor toward left and click to draw a horizontal polyline.

22. Draw the vertical second entity of the polyline and then draw the horizontal third entity of the polyline. Make sure that the end point of the third entity is coincident with the inner circle, refer to Figure 3-90.

23. After drawing the three entities of the polyline, right-click in the **Graphics** window and choose the **Open End** option from the shortcut menu displayed.

24. Press the ESC key to exit the **Ployline** tool.

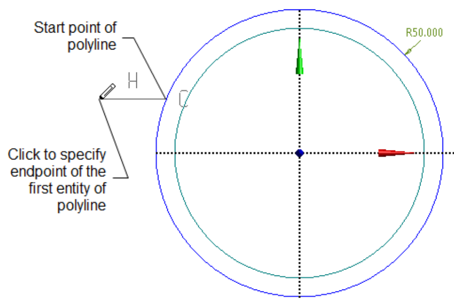


Figure 3-89 *Creating the first entity of the polyline*

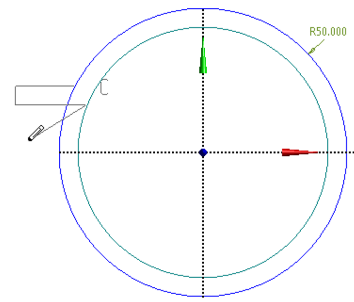


Figure 3-90 *Sketch after creating the third entity of the polyline*

Next, you need to create a similar sketch on the other side of the X axis such that this sketch becomes the mirror copy of the sketch already created.

25. Expand the **Modify** toolbox and choose the **Replicate** tool; you are prompted to select points or edges to replicate.

The **Replicate** tool is used to copy entities from an existing sketch and paste them wherever required.

26. Select the three entities of the polyline created in the previous steps and right-click in the **Graphics** window; a shortcut menu is displayed, as shown in Figure 3-91.

27. Choose the **End / Use Plane Origin as Handle** option from the shortcut menu, refer to Figure 3-91; the preview of the entities to be replicated along with the paste handle is displayed, refer to Figure 3-92.

The paste handle is used to set a reference point while replicating entities. This reference point is used while placing the entities to be replicated. To replicate the entities, select the entities and then right-click to display a shortcut menu. This shortcut menu contains options such as **Clear Selection**, **End/Set Paste Handle**, **End/Use Plane Origin as Handle**, and **End/Use Default Paste Handle**.

The **Clear Selection** option is used to deselect the entities that were selected to replicate earlier. The **End / Set Paste Handle** option is used to specify the paste handle by specifying a point in the **Graphics** window. The **End / Use Plane Origin as Handle** option is used to specify the origin of the sketching plane as the paste handle. The **End / Use Default Paste Handle** option is used to specify a system specified point of the selected entity as the paste handle.

Since the entities to be replicated are the mirror copies of the selected entities, you have to flip them about the X axis.

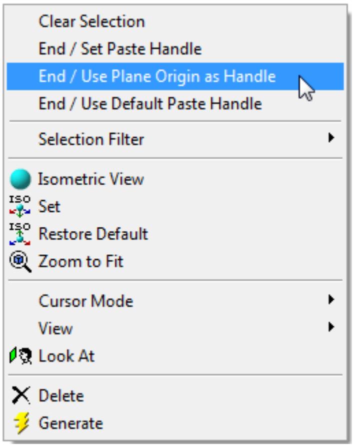


Figure 3-91 *The shortcut menu displayed while the **Replicate** tool is active*

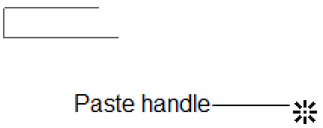


Figure 3-92 *Preview of the entities to be replicated and the paste handle*

28. Next, select the origin; you are prompted to specify the location to paste the entities, as shown in Figure 3-93.

29. Right-click in the **Graphics** window and choose the **Flip Vertical** option from the shortcut menu displayed, refer to Figure 3-92; the preview of the flipped entity is displayed, as shown in Figure 3-93 displayed.

After specifying the location of paste handle, instead of replicating the entities directly, you can rotate them by the desired angle, scale them by desired scale factor, and flip them along the horizontal and vertical directions. Place the entities at the desired locations, by using the options from the shortcut menu.

To rotate the selected entities before replicating them, enter the required angle of rotation in the **r** edit box, displayed on the right of the **Replicate** tool. Right-click in the **Graphics** window to display the shortcut menu, refer to Figure 3-94. Next, choose the **Rotate by r Degrees** or **Rotate by -r Degrees** option from the shortcut menu.

To scale the selected entities before replicating them, enter the required value for scale factor in the **f** edit box, displayed on the right of the **Replicate** tool. Next, choose the **Scale by factor f** or **Scale by factor 1/f** option from the shortcut menu according to the requirement, refer to Figure 3-94.

30. Move the paste handle to the origin and click when the symbol of Coincident Point constraint (P) is displayed, refer to Figure 3-95; the mirror copies of the selected entities are replicated at their required locations, refer to Figure 3-96.

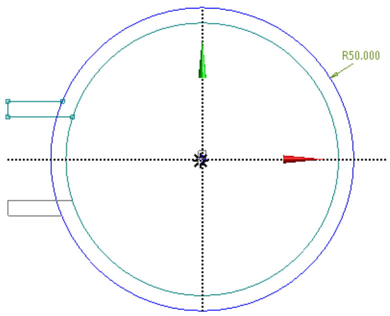


Figure 3-93 Specifying the location of paste handle

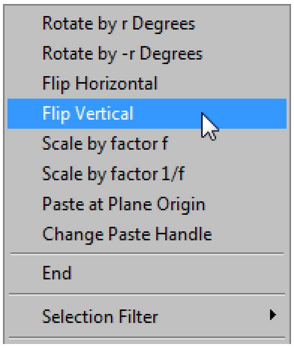


Figure 3-94 Specifying the option to flip the entities vertically

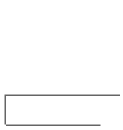


Figure 3-95 Preview of the entities to be replicated after flipping them vertically

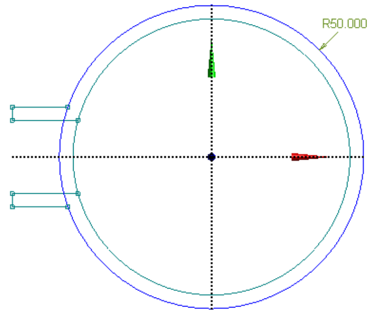


Figure 3-96 The sketch after replicating the selected entities

31. Press the ESC key to exit the **Replicate** tool.

Next, you need to trim the unwanted portion of the sketch using the **Trim** tool from the **Modify** toolbox.

32. Choose the **Trim** tool from the **Modify** toolbox; you are prompted to select edges to trim.

The **Trim** tool is used to trim the objects that extend beyond a required point of intersection. While creating a sketch, there are a number of places where you need to remove the unwanted and extending edges. Choose the **Trim** tool from the **Modify** toolbox; the Draw cursor will be displayed and you will be prompted to select the edges to be trimmed. Select the sketched entity to be trimmed; the selected sketched entity is trimmed to its nearest point of intersection with any other sketched entity or axis.

33. Select the **Ignore axis** check box displayed on the right of the **Trim** tool and click on the segments one by one, marked in Figure 3-97; the selected segments is trimmed and you get the sketch shown in Figure 3-98.

The **Ignore axis** check box is selected to ignore the intersection of the segment of circles with the X axis while trimming.

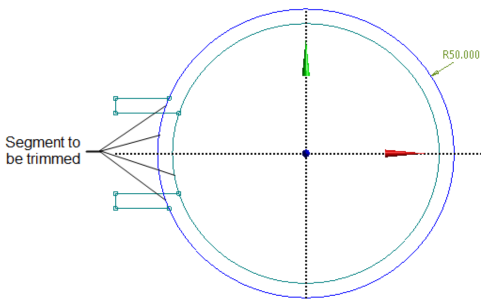


Figure 3-97 *Segments of the circle to be trimmed*

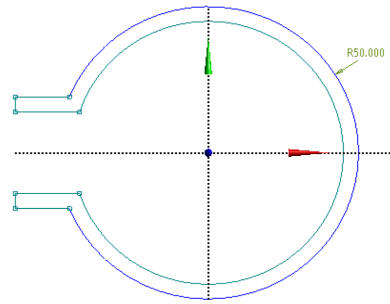
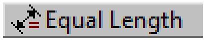


Figure 3-98 *Sketch after trimming the entities*

Applying Geometric Constraints and Dimensions to the Sketch

The entities of the sketch should be fully specified in terms of size, shape, orientation, and location. This is achieved by setting geometric constraints and dimensions.

Geometric constraints are the logical operations that are performed to add relationship (such as tangent or perpendicular) between the sketched entities, planes, axes, edges, or vertices. The constraints applied to the sketched entities are used to capture the design intent. By using constraints in a sketch, you can reduce the number of dimensions that are required in that sketch. The geometric constraints are applied using the tools available in the **Constraints** toolbox.



1. Expand the **Constraints** toolbox and choose the **Equal Length** tool from it; you are prompted to select lines to apply the constraints.

2. Select any one of the two vertical lines from the sketch; you are prompted to select the lines to apply the Equal Length constraint.

3. Select the second vertical line from the sketch; the Equal Length constraint is applied to the two vertical lines and you are prompted to select the first line for applying the Equal Length constraint.

4. Select the top most horizontal line of the sketch and then select the bottom most horizontal line of the sketch to make them equal in length.

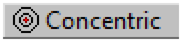
5. Select the **Symmetric** tool from the **Constraints** toolbox; you are prompted to specify the axis of symmetry. Select the X axis; you are prompted to select a point or edge to apply the Symmetric constraint.

The **Symmetric** tool is used to make entities symmetric about a centerline. After this tool is invoked, select a centerline and then select the entities which are to be made symmetric.

6. Select the horizontal line from the sketch that is just above the X axis; you are prompted to select the second point or edge to apply the Symmetric constraint.

7. Select the horizontal line from the sketch that is just below the X axis; the two horizontal lines become symmetric about the X axis.

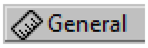
8. Choose the **Concentric** tool from the **Constraints** toolbox and select the two circular arcs from the sketch; the selected arcs become concentric.



The **Concentric** tool is used to force two circular entities share the same center.

Next, you need to generate the dimensions and edit their values to get the sketch of desired size.

9. Expand the **Dimensions** toolbox from the **Sketching Toolboxes** windows and then choose the **General** tool.



10. Generate all dimensions shown in Figure 3-99 and edit the value of dimensions in the **Details View** window, refer to Figure 3-100.

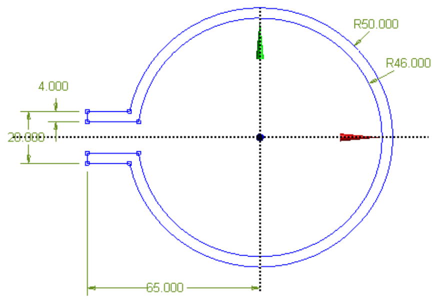


Figure 3-99 *Dimensions to be generated for the sketch of base feature*

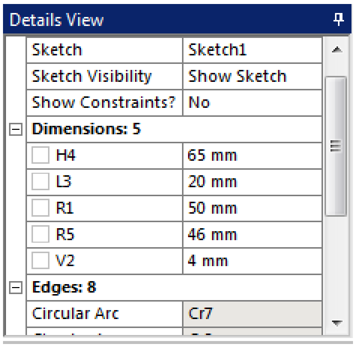


Figure 3-100 *Value of dimensions in the* ***Details View*** *window*

Note

The names of the dimensions displayed in the ***Details View*** *window can be different in your system.*

After applying the required geometric constraints and generating the dimensions, the color of the sketch will change to blue indicating that the sketch is fully constrained and is ready to be used for feature creation operations.

After completing the sketch, you need to exit the **Sketching** mode.

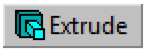
11. Choose the **Modeling** tab located at the bottom of the **Sketching Toolboxes** window; the **Sketching** mode is activated and the Tree Outline is displayed. Also, **Sketch 1** is added under the **XYPlane** node.

    After exiting the **Sketching** mode, the sketching plane is still normal to the viewing direction. To proceed further with the feature creation operation, it is advised to change the view of the sketching plane to Isometric view.

12. Right-click in the **Graphics** window, and then choose the **Isometric View** option from the shortcut menu displayed; the sketch is displayed in Isometric view.

Creating the Base Feature

Next, you need to create the base feature using the **Extrude** tool from the **Features** toolbar.



1. Choose the **Extrude** tool from the **Features** toolbar; the preview of the extruded feature with the default values is displayed in the **Graphics** window. Also, a node for the extruded feature with the name **Extrude 1** is added below the three default planes in the Tree Outline.

The default parameters used for generating the preview of the extruded feature are displayed in the **Details View** window. To get the required shape of the base feature, you need to edit the values in the **Details View** window.

As per the requirement of this tutorial, the material should be added normal to and symmetrically on both sides of the sketch.

2. Select the **Both-Symmetric** option from the **Direction** drop-down list.

3. Enter **10** in the **FD1, Depth (>0)** edit box of the **Details View** window.

   The complete depth of material addition is 20mm, but the material will be added symmetrically by the same depth on both the sides of the sketch. Therefore, 10mm is specified as the depth value.

4. Choose the **Generate** tool from the **DesignModeler** toolbar; the base feature is created with the specified settings, refer to Figure 3-101.

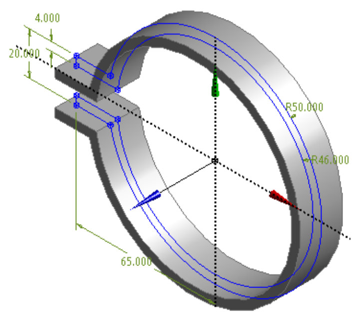


Figure 3-101 *The base feature*

By default, the sketch is displayed only when the plane on which it is created is the active plane. Since the XY plane is the current active plane, the sketch and the dimensions of the *Sketch1* are still displayed in the **Graphics** window. You can hide the sketch as the sketch and its dimensions are not needed now.

5. Right-click on the **Sketch1** in the Tree Outline and choose the **Hide Sketch** option from the shortcut menu displayed.

Note

If needed, you can again display the sketch and its dimensions. To do so, right-click on the ***Sketch1*** *in the Tree Outline and choose the* ***Show Sketch*** *option from the shortcut menu displayed.*

Creating the Circular Cutout

Next, you need to create the circular cutout on the two rectangular flanges of the base feature. The sketch for this feature should be created on the rectangular flange. As the three default planes do not pass through the surface on which the cutout has to be created, these planes cannot be used for drawing the sketch. Therefore, you have to define a new plane on the top flat face of the rectangular flange and draw the sketch for the circular cutout.

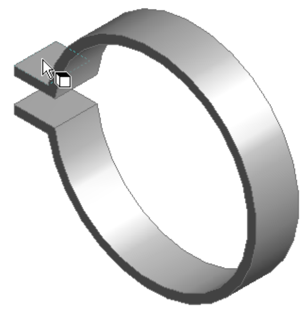


Figure 3-102 *Selecting the flat face for defining the Sketching plane*

1. Select the top face of the rectangular flange, refer to Figure 3-102.

You can use the tools available in the **Select** toolbar to select an edge, face, vertex, and so on in a geometry, refer to Figure 3-103. For example, the **Edge** tool is used to select an edge, the **Face** tool is used to select a face, and so on. Alternatively, right-click in the **Graphics** window to display a shortcut menu and then choose the desired tool from the **Selection Filter** cascading menu, as shown in Figure 3-104.

2. Choose the **New Plane** tool from the **Active Plane/Sketch** toolbar; **Plane4** is added to the Tree Outline.

3. Choose the **Generate** tool available in the **Features** toolbar to generate the new plane.

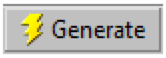


Figure 3-103 *The* ***Select*** *toolbar*

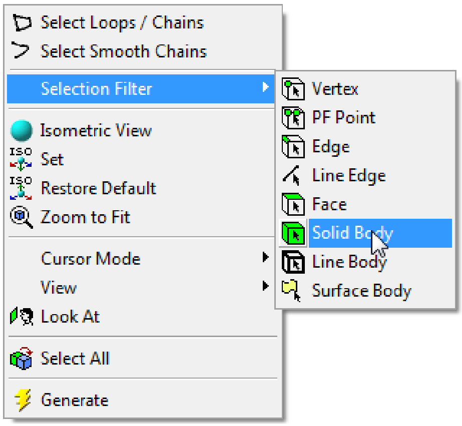


Figure 3-104 *Choosing a selection mode from the* ***Selection Filter*** *cascading menu*

4. Choose the **Sketching** tab available under the Tree Outline; the **Sketching** mode is invoked.

5. Choose the **Look At** tool from the **Graphics** toolbar.

The **Look At** tool is used to orient the view normal to the screen.

6. Choose the **Circle** tool from the **Draw** toolbox and draw a circle as shown in Figure 3-105.

7. Expand the **Dimensions** toolbox. The **General** tool is chosen by default in this toolbox.

8. Generate the dimensions of the circle and specify their values in the **Details View** window, refer to Figure 3-106. The sketch gets fully-defined and is ready to be used for feature creation.

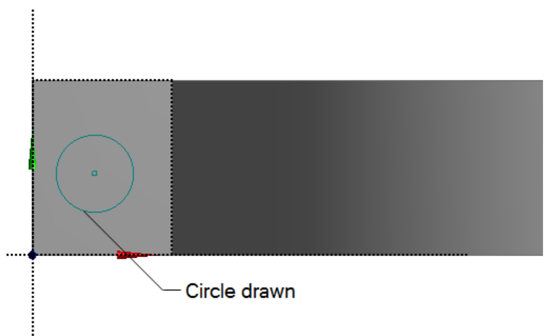


Figure 3-105 *Creating circle on the defined sketching plane*

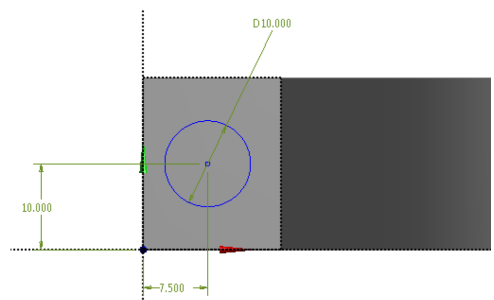


Figure 3-106 *Generating the dimensions of the sketch for cutout feature*

9. Invoke the **Modeling** mode by choosing the **Modeling** tab displayed below the **Sketching Toolboxes** window. The sketch of the cutout feature is created on Plane4 and is displayed as **Sketch2** in the Tree Outline, as shown in Figure 3-107.

10. Change the view to isometric by clicking on the ISO ball (cyan color) of the Triad, displayed at the bottom right corner of the **Graphics** window.

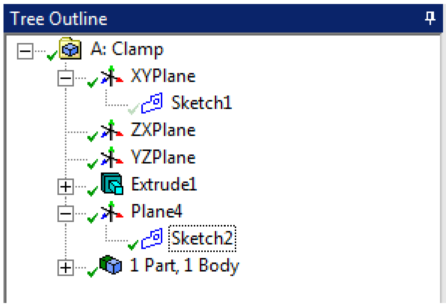
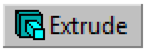


Figure 3-107 *The new plane and sketch added in the Tree Outline*

After drawing the sketch, you need to remove the material from the base feature using the **Extrude** tool.

11. Choose the **Extrude** tool from the **Features** toolbar; the preview of the extruded feature with the default values is displayed in the **Graphics** window.



Also, the Tree Outline is activated and **Extrude2** is added below **Extrude1** in the Tree Outline.

12. Select the **Cut Material** option from the **Operation** drop-down list.

The **Cut Material** option is used to create cutouts, holes, and so on in an existing components.

As per the design requirements, the material should be removed starting from the flat face on which the sketch is created and up to the bottom most face of the second rectangular flange.

13. Select the **To Surface** option from the **Extent Type** drop-down list, refer to Figure 3-108; the **Target Faces** selection box is added in the **Details View** window.

To extrude a sketch to a desired face of an existing model, choose the **To Surface** option.

14. Click on the **Target Faces** selection box; the **Apply** and **Cancel** buttons are displayed in the **Target Faces** selection box and you are prompted to select faces to create extrude.

15. Select the bottom face of the second flange, refer to Figure 3-109; the material is removed up to the specified surface.

Note

While selecting the target face, you need to rotate the view of the model. The process of dynamically rotating the model has been discussed in detail in the previous tutorial.

16. Choose the **Apply** button from the **Target Face** selection box to accept the specified face.

17. Choose the **Generate** tool from the **Features** toolbar; the cutout feature is created, refer to Figure 3-110.

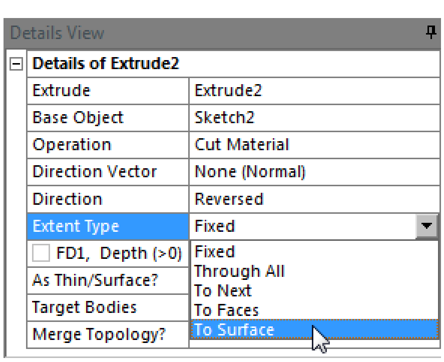


Figure 3-108** Selecting the **To Surface** option from the **Extent Type drop-down list *in the **Details View** window*

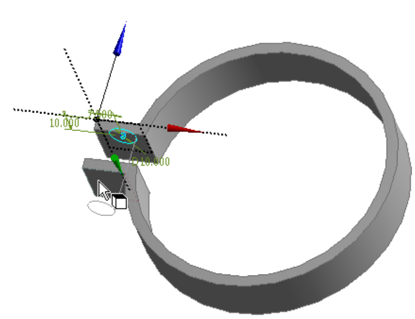
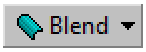


***Figure 3-109** Specifying the face up to which the material will be removed*

Creating the Blend Feature

To remove the sharp edges from the clamp, you need to create fillets of radius 5 mm at the vertical edges of the clamp. The fillet will be created on four vertical edges of the model using the **Fixed Radius** option of the **Blend** tool.



1. Choose the **Fixed Radius** tool from the **Blend** drop-down in the **Features** toolbar; you are prompted to select edges to blend.

2. Press the CTRL key and select the four vertical edges of the model, refer to Figure 3-111.

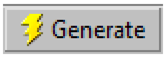
Note

1. In Figures 3-110 and 3-111, the display of planes has been turned off for better visualization. As per the need, you can turn on or off the display of planes by choosing the ***Display Plane*** *button from the* ***Graphics*** *toolbar.*

2. To facilitate the selection of edges without rotating the model and for generating the blend feature, change the display mode to wireframe. The procedure to change the display mode has been discussed in the previous tutorial.

3. Click on the **Geometry** selection box in the **Details View** window; the **Apply** and **Cancel** buttons are displayed. Next, choose the **Apply** button from the **Details View** window to accept the selected of edges to be blended.

4. Enter **5** in the **FD1 Radius (>0)** edit box as the radius of the edit box.



5. Choose the **Generate** tool from the **Features** toolbar; the blend feature is created. Figure 3-112 shows the final model.

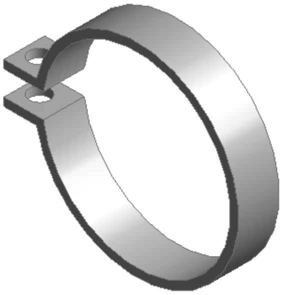


Figure 3-110 *Model after creating the circular cutout*

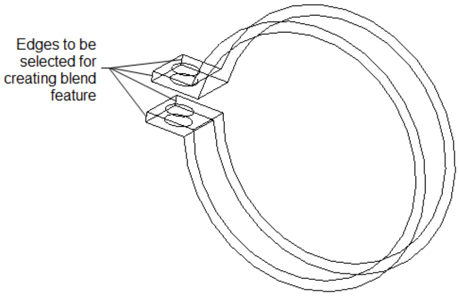


Figure 3-111 *Edges to be selected for creating the blend feature*

Figure 3-112 The final model

> **Tip.** *Even after creating a feature, you can modify some of its parameters. To do so, select the feature that you need to modify from the Tree Outline; the parameters of the selected feature will be displayed in the* ***Details View*** *window. The parameters that cannot be edited will be grayed out and the remaining parameters can be edited. Change the value of the required parameters and then choose the* ***Generate*** *button from the* ***Features*** *toolbar.*

6. Close the **DesignModeler** window; the **Workbench** window is displayed.

Saving the Project and Exiting ANSYS Workbench

After visualizing the model and restoring the default Isometric view, you need to save the project and exit ANSYS Workbench. This saved project will be used in later chapters for analysis.

1. In the **Workbench** window, choose the **Save** button from the **Standard** toolbar; the project is saved with the name *c03_ansWB_tut03*.

2. Choose **File > Exit** from the **Workbench** window to exit the ANSYS Workbench session.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. In the **DesignModeler** window, the P symbol represents the Coincident Point constraint. (T/F)

2. The **Extrude** tool can be invoked from the **Create** menu of the Menu bar. (T/F)

3. In the **DesignModeler** window, none of the constraints are automatically applied while drawing a sketch. (T/F)

4. The **Horizontal** tool in the **Constraints** toolbox can be used to make a linear entity horizontal. (T/F)

5. You can change the view type to isometric by using the ISO ball present in the Triad. (T/F)

6. You can switch to the **Modeling** mode by choosing the **Modeling** tab available at the bottom of the **Sketching Toolboxes** window. (T/F)

7. The **Arc by Tangent** tool can be invoked from the __________ toolbox.

8. The __________ tool is used to make two entities equal in length.

9. You can invoke the **Offset** tool from the _________ toolbox.

10. You can hide or show a sketch anytime by using the ____________________.

Review Questions

Answer the following questions:

1. The options in the **Details View** window are contextual in nature. (T/F)

2. You can create patterns of entities by using the **Replicate** tool available in the **Modify** toolbox. (T/F)

3. You can change the direction of extrusion by using the options in the **Direction** drop-down list in the **Details View** window. (T/F)

4. On choosing any tool from the **Draw** toolbox, the normal arrow cursor changes to Draw cursor. (T/F)

5. Like other tools in the **Graphic**s toolbar, the **Rotate** tool is also a transparent tool. (T/F)

6. You can create line segments tangent to arcs by using the ______________ tool from the **Draw** toolbox.

7. In the **DesignModeler** window, user actions are recorded in the ___________________ window.

8. You can switch to the **Sketching** mode by choosing the **Sketching** tab available at the bottom of the **Tree Outline** window.

9. In the **DesignModeler** window, you can change the dimension of the entities by specifying the new values in the ____________ window.

10. In **DesignModeler**, the Vertical constraint is represented by _____ symbol.

EXERCISES

Exercise 1

Create the model shown in Figure 3-113. The dimensions of the model are shown in Figure 3-114. **(Expected time: 30 min)**

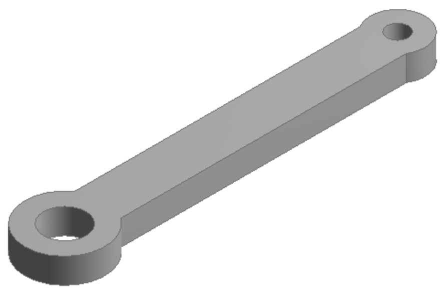


Figure 3-113 Model for Exercise 1

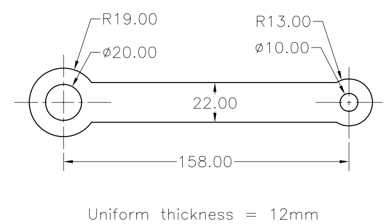


Figure 3-114 Dimensions of the model for Exercise 1

Exercise 2

Create the model shown in Figure 3-115. The dimensions of the model are shown in Figure 3-116. **(Expected time: 45 min)**

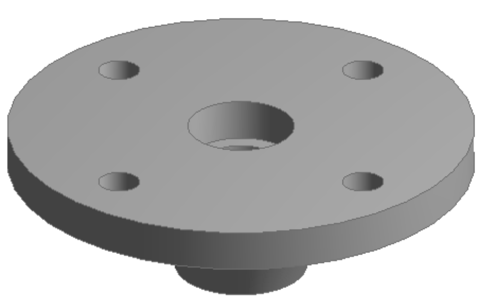


Figure 3-115 Model for Exercise 2

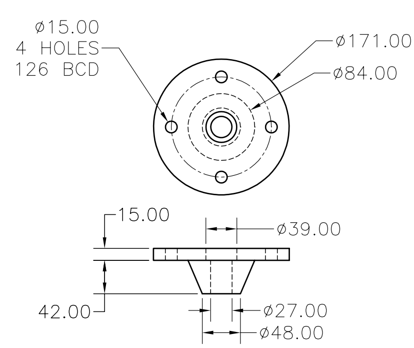


Figure 3-116 Dimensions for Exercise 2

Exercise 3

Create the model shown in Figure 3-117. The dimensions of the model are shown in Figure 3-118. **(Expected time: 45 min)**

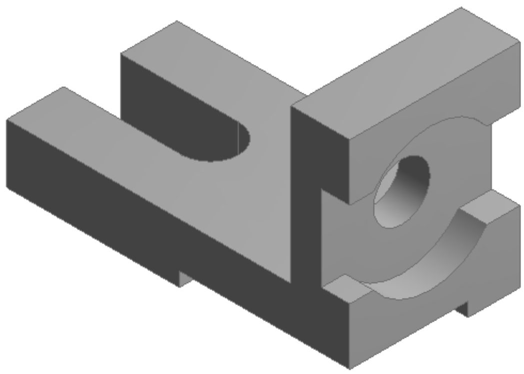


Figure 3-117 *Model for Exercise 3*

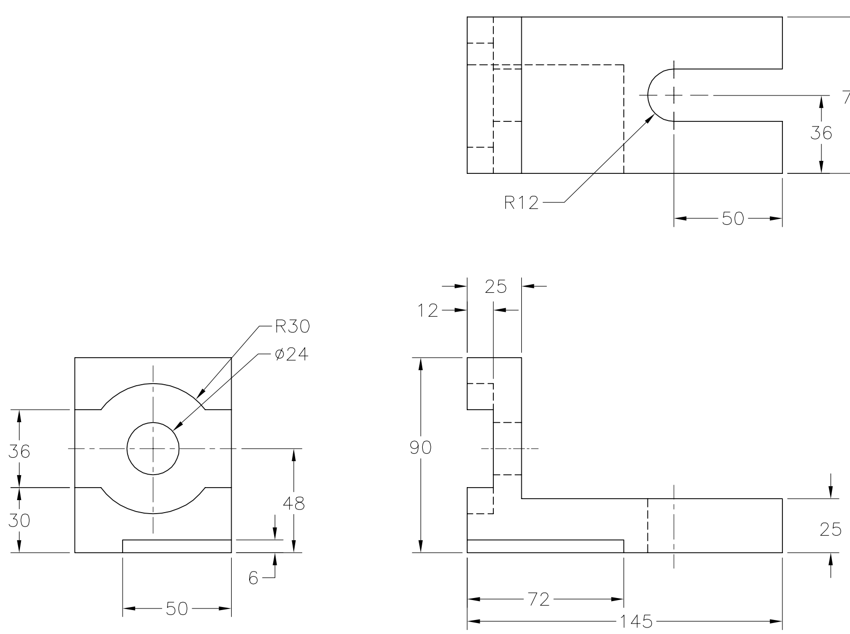


Figure 3-118 *Dimensions of the model for Exercise 3*

Answers to Self-Evaluation Test

1. T, **2.** F, **3.** F, **4.** T, **5.** T, **6.** T, **7. Draw**, **8. Equal Length**, **9. Modify**, **10.** Tree Outline