



Chapter 2

Drawing Sketches for Solid Models

Learning Objectives

After completing this chapter, you will be able to:

- *Start a new template file to draw sketches.*
- *Set up the sketching environment.*
- *Use various drawing display tools.*
- *Understand the sketcher environment in the Part module.*
- *Get acquainted with sketcher entities.*
- *Specify the position of entities by using dynamic input.*
- *Draw sketches by using various sketcher entities.*
- *Delete sketched entities.*

THE SKETCHING ENVIRONMENT

Most designs created in Autodesk Inventor consist of sketched and placed features. A sketch is the combination of a number of two-dimensional (2D) entities such as lines, arcs, circles, and so on. The features such as extrude, revolve, and sweep that are created by using 2D sketches are known as sketched features. The features such as fillet, chamfer, thread, and shell that are created without using a sketch are known as placed features. In a design, the base feature or the first feature is always a sketched feature. For example, the sketch shown in Figure 2-1 is used to create a solid model, as shown in Figure 2-2. In this figure, the fillets and chamfers are the placed features.

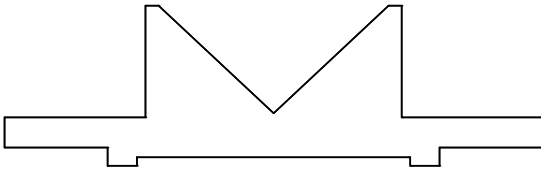


Figure 2-1 The basic sketch for the solid model

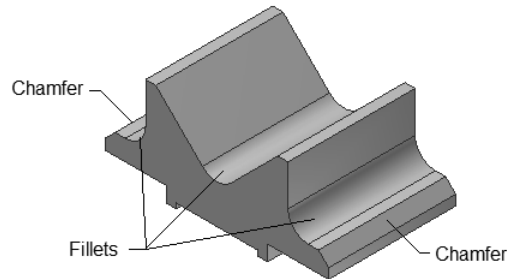


Figure 2-2 A solid model created using the sketched and placed features

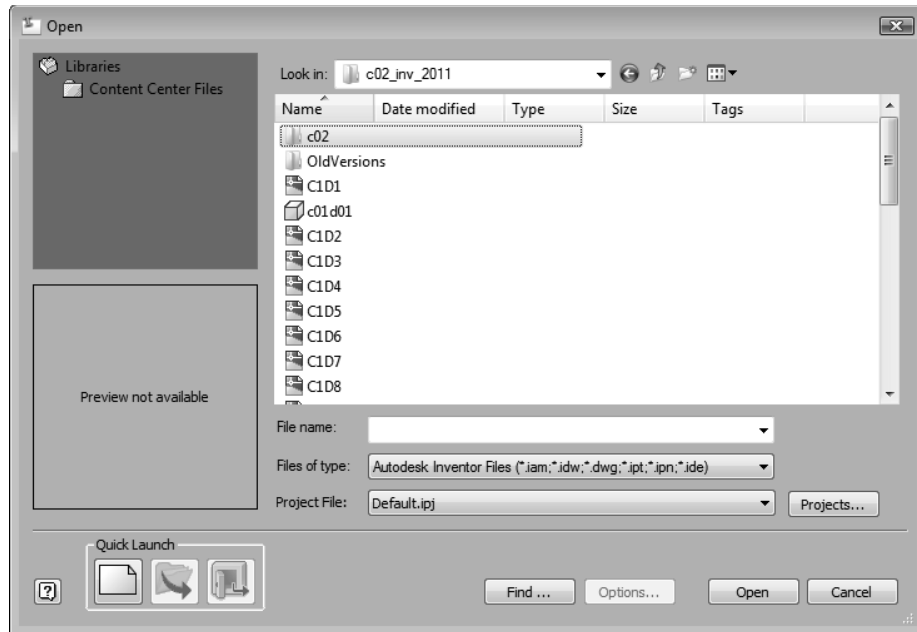
Once you have drawn the basic sketch, refer to Figure 2-1, you need to convert it into a solid model using solid modeling tools.

The sketching environment of Autodesk Inventor can be invoked at any time in the **Part** module or in the **Assembly** module. Unlike other solid modeling programs, here you just need to invoke the **Create 2D Sketch** tool and specify the plane to draw the sketch; the sketching environment will be activated. Also, when you start a new file in the **Part** module, first the sketching environment will be activated. You can draw a sketch in this environment and then proceed to the part modeling environment for converting the sketch into a solid model. The options in the sketching environment will be discussed later in this chapter.

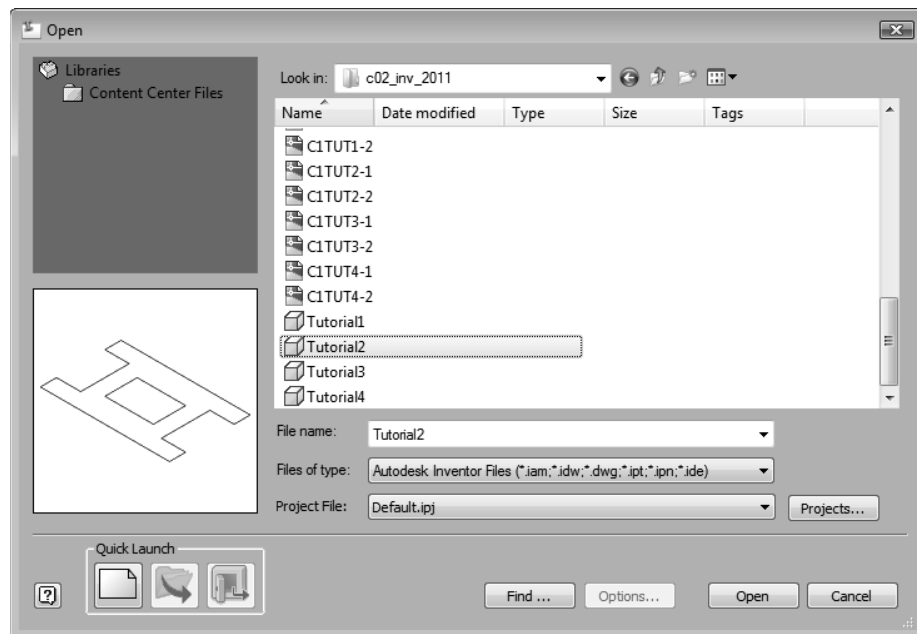
The Open Dialog Box

On starting a new session of Autodesk Inventor, the initial screen will be displayed. Now, choose the **Open** tool from the **Launch** panel of the **Get Started** tab; the **Open** dialog box will be displayed, as shown in Figure 2-3.

The options in the **Open** dialog box are used to create new files and open existing files. You can browse and select the file that you want to open from the list displayed in the dialog box. The preview of the selected file is displayed in the preview window located at the lower left portion of this dialog box, as shown in Figure 2-4. By default, you can open any file created in Autodesk Inventor. This is because by default, the **Files of type** drop-down list displays the **Autodesk Inventor Files (*.iam;*.idw;*.dwg;*.ipt;*.ipn, and *.ide)** option.



*Figure 2-3 The **Open** dialog box*



*Figure 2-4 The **Open** dialog box showing the preview of the selected file*

You can also open the files created in other solid modeling programs such as AutoCAD, Pro/ENGINEER, Alias, SolidWorks, NX, and so on, by selecting the respective options from the **Files of type** drop-down list.

In Autodesk Inventor, a project defines all the files related to a design project you are working on. You can create new projects or retrieve the previously created projects by choosing the **Projects** button available on the right of the **Project File** drop down list in the **Open** dialog box. When you choose the **Projects** button, the **Projects** dialog box will be displayed, as shown in Figure 2-5. All the project folders will be displayed in the upper half of the dialog box and the options regarding the selected project folder will be displayed in the lower half of the dialog box. To add another project folder to this list, choose the **New** button; the **Inventor project wizard** dialog box will be displayed. The **New Vault Project** radio button is selected by default in this dialog box. Choose the **Next** button from the **Inventor project wizard** dialog box. Specify the name of the project in the **Name** text box and the location in the **Project (Workspace) Folder** text box. You can also choose the **Browse for project location** button to specify the location of the project. Next, choose the **Finish** button. Once you have specified the project folder, it will be added in the upper part of the dialog box and its location will be displayed. When you select a project, the options related to it will be shown in the lower part of the dialog box. The **Projects** dialog box with various projects is displayed, refer to Figure 2-5.

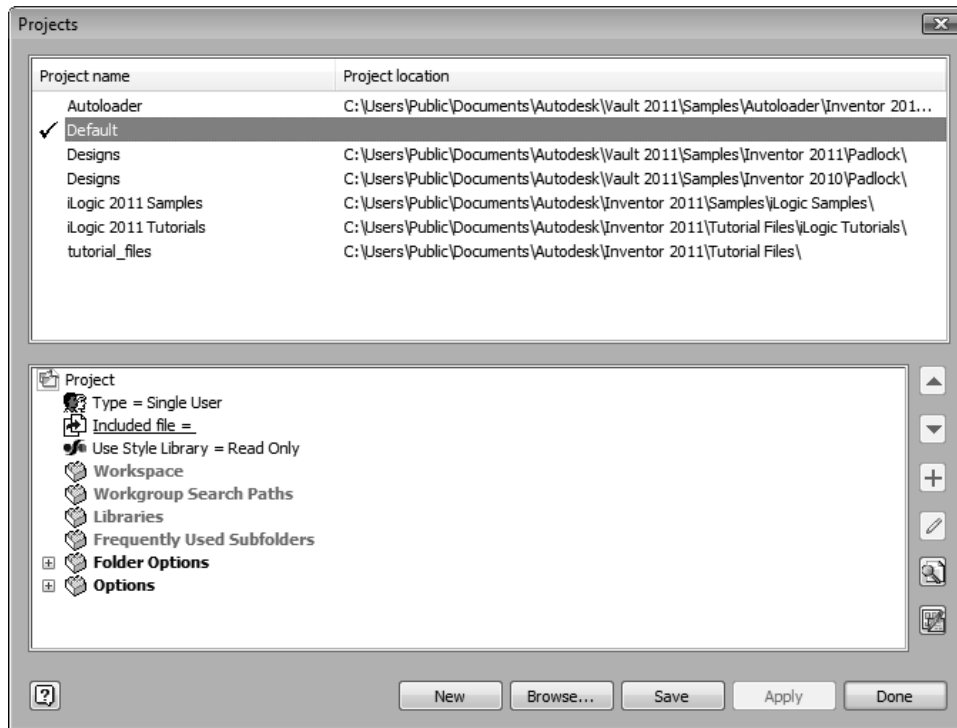


Figure 2-5 The Projects dialog box

On starting a new session in Autodesk Inventor, only the **Start a new file** button will be available in the **Quick Launch** area of the **Open** dialog box. Choose this button; the **New File** dialog box will be displayed, as shown in Figure 2-6. To exit the **Open** dialog box, choose the **Cancel** button.

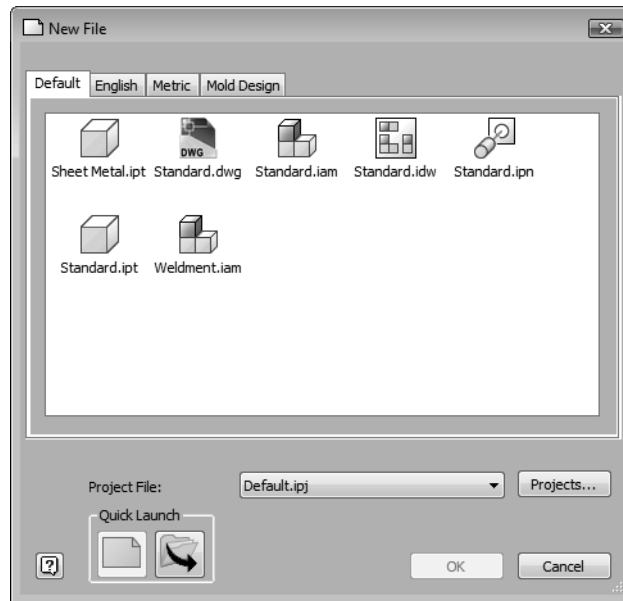


Figure 2-6 The New File dialog box

To view help about topics, press F1; the **Autodesk Inventor Help** window will be displayed. In this window, you will find help topics explaining how to use a particular tool or option of Autodesk Inventor. You can select the **Show Help on startup** check box available on the right of the **Autodesk Inventor Help** window to make this window appear whenever you start a new session. You can exit the **Autodesk Inventor Help** window by choosing the **Close** button.

Starting a New File



In Autodesk Inventor, you can start a new file by choosing **New** from the **Get Started** tab or by choosing the **New** option from the **Application Menu**. On doing so, the **New File** dialog box will be displayed, refer to Figure 2-6. Alternatively, you can start a new file by choosing the **New** tool from the **Quick Access Toolbar**.



Note

*If you invoke the **New File** dialog box by choosing the **New** tool from the **Quick Access Toolbar** or by choosing **New > New** from the **Application Menu**, only the **Open** button will be available in the **Quick Launch** area. You can choose the **Open** tool to invoke the **Open** dialog box and open an existing file in Autodesk Inventor.*

The options in the **New File** dialog box are used to select a template file for starting a design. You can select a template in the **Default**, **English**, **Metric**, or **Mould** tabs. To start a new metric part file, choose the **Metric** tab, as shown in Figure 2-7. The templates that are available on choosing the **Metric** tab are discussed next.

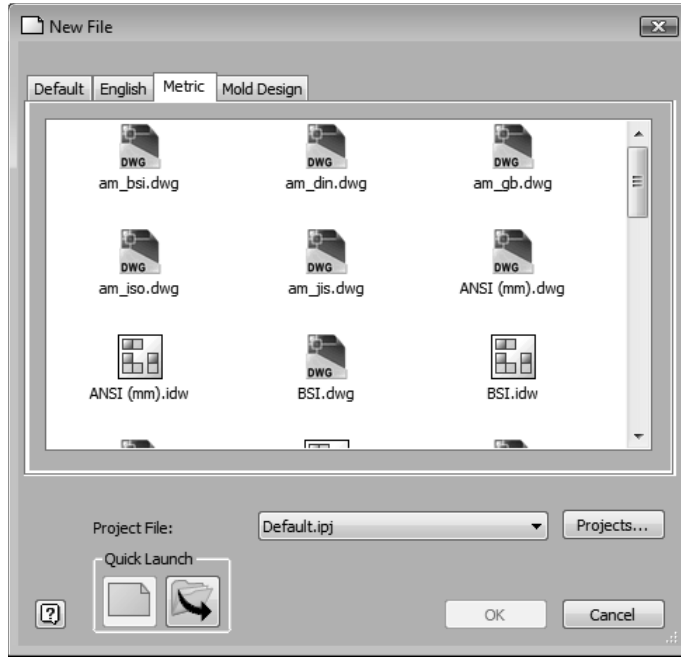


Figure 2-7 The Metric tab of the New File dialog box

.ipt Templates

Select any *.ipt* template to start a new part file for creating a solid model or a sheet metal component. When you start a new part file, the sketching environment will be automatically active and you can directly start drawing sketches.

.iam Templates

Select any *.iam* template to start a new assembly file for assembling various parts. Similarly, use the *Weldment.iam* template to weld two different components in the **Weldment** module.

.ipn Templates

Select any *.ipn* template to start a new presentation file for animating the assembly. The **Presentation** module marks the basic difference between Autodesk Inventor and other design tools. This module allows you to animate the assemblies created in the **Assembly** module. For example, you can create a presentation in the **Presentation** module that shows a Drill Press Vice assembly in motion.

.idw Templates

Select any *.idw* template to start a new drawing file for generating the drawing views. You can use the drawing templates of various standards that are provided in this tab, such as ANSI, ISO, DIN, GB, JIS, GOST, and BSI.

.dwg Templates

Select any *.dwg* template for creating AutoCAD drawing files. You can use the drawing templates of standards such as JIS, ISO, GB, DIN, BSI, and ANSI.

The **Project File** drop-down list in the **New File** dialog box displays the active project in which the new file has been started. The **Projects** dialog box can be invoked by choosing the **Projects** button from the **New File** dialog box.

INTRODUCTION TO THE SKETCHING ENVIRONMENT

The initial screen appearance in the sketching environment of a *Standard (mm).ipt* file is shown in Figure 2-8. By default, the **Ribbon** is placed at the top of the graphics window, refer to Figure 2-8. You can move this **Ribbon** anywhere in the graphics window. To do so, right-click on the **Ribbon**; a shortcut menu will be displayed. Select the **Undock Ribbon** option from the shortcut menu; the **Ribbon** will be undocked. Now you can drag the **Ribbon** anywhere in the graphics window. It is recommended to place (dock) the **Ribbon** at the top of the graphics window so that you can use the space efficiently. To do so, right-click on the **Ribbon** and choose **Docking Position > Top** from the shortcut menu. Alternatively, double-click on the title bar of the **Ribbon** to dock it.

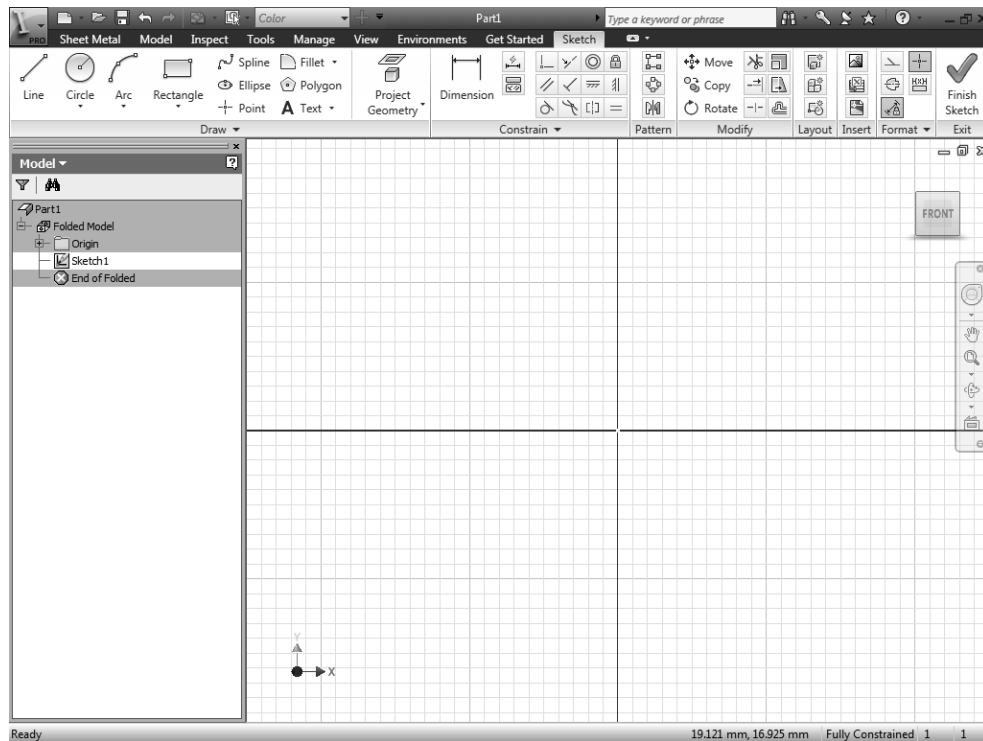


Figure 2-8 Initial screen appearance in the sketching environment

SETTING UP THE SKETCHING ENVIRONMENT

It is very important to first set up the sketcher environment. This has to be done before you start drawing a sketch. Setting up the sketcher environment includes modifying the grids of a drawing. It is unlikely that the designs that you want to create consist of small dimensions. You

will come across a number of designs that are large. Therefore, before starting a drawing, you need to modify the grid settings. These settings will depend on the dimensions of the design. The process of modifying the grid settings of a drawing is discussed next.

Modifying the Document Settings of a Sketch

Before sketching, you may need to modify the setting of the sketching environment as per your requirement. You can change the snapping distance, grid spacing, and various attributes related to line display of the sketching environment. You must have noticed that the drawing window in the sketching environment consists of a number of light and dark lines that are normal to each other. These normal lines are called grid lines. The grid lines help you locate an entity, thereby helping you to draw a sketch correctly or modify an existing sketch precisely.

You can modify the document settings of a sketch. To do so, choose the **Document Settings** tool from the **Options** panel of the **Tools** tab; the **Document Settings** dialog box will be displayed. In this dialog box, choose the **Sketch** tab to display the options related to the sketching environment, see Figure 2-9. The options under this tab are discussed next.

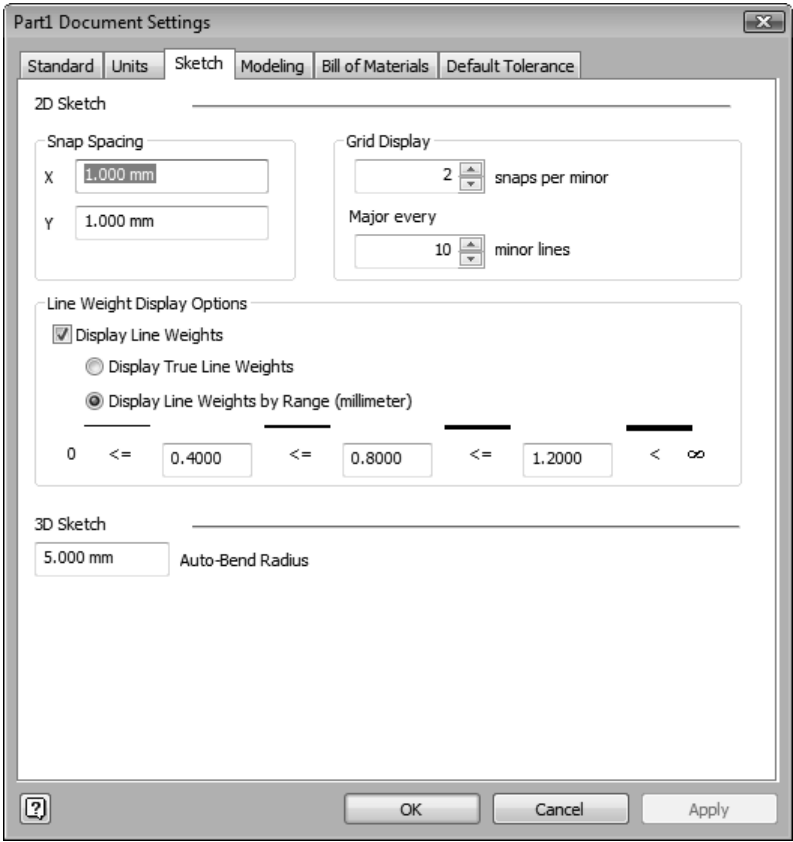


Figure 2-9 The *Sketch* tab of the *Document Settings* dialog box

Snap Spacing Area

The options under this area are used to specify the snap distances.

X

This edit box is used to specify the snap spacing in the X direction.

Y

This edit box is used to specify the snap spacing in the Y direction.

Grid Display Area

The options in this area are used to control the number of major and minor lines. The minor lines are the light lines that are displayed inside the dark gray lines. The dark gray lines are called the major lines.

snaps per minor

This spinner is used to specify the number of snap points between each minor line.

Major every minor lines

This spinner is used to specify the number of minor lines between two major lines.

Line Weight Display Options Area

The options in the **Line Weight Display Options** area allow you to control the line weight in the sketching environment. The **Display Line Weights** check box is selected by default and displays the sketches with the set line weights. If this check box is cleared, then the differences in the line weights will not be displayed in the sketch. The **Display True Line Weights** radio button, if selected, displays the line weights on screen as they would appear on paper when printed. The **Display Line Weights by Range (millimeter)** radio button, if selected, displays the line weights according to the values entered.



Note

You will have to increase the drawing display area after increasing the grid spacing. The options to do so are discussed next.



Tip. You can also turn off the display of the major and minor grid lines and the axes. To turn off the display of the grid line and the axes, choose the **Application Options** tool from the **Options** panel; the **Application Options** dialog box will be displayed. Next, choose the **Sketch** tab and clear the **Grid lines**, **Minor grid lines**, and **Axes** check boxes from the **Display** area.

UNDERSTANDING THE DRAWING DISPLAY TOOLS

The drawing display tools or navigation tools are an integral part of any design software. These tools are extensively used during the design process. These tools are available in the **Navigation Bar** located on the right in the graphics window and in the **Navigate** panel of the **View** tab. Some of the drawing display tools in Autodesk Inventor are discussed next. The rest of these tools will be discussed in the later chapters.

Zoom All

Ribbon: View > Navigate > Zoom drop-down > Zoom All
Navigation Bar: Zoom flyout > Zoom All



The **Zoom All** tool is used to increase the drawing display area to display all the sketched entities in the current display.

Zoom Window

Ribbon: View > Navigate > Zoom drop-down > Zoom Window
Navigation Bar: Zoom flyout > Zoom Window



The **Zoom Window** tool is used to define an area to be magnified and viewed in the current drawing. The area is defined using two diagonal points of a box (called window) in the drawing window. The area inscribed inside the window will be magnified and displayed on the screen.



Tip. The size of the dimension text always remains constant even if you magnify the area that includes some dimensions.

To switch to the previous view, right-click in the drawing window and then choose **Previous View** from the shortcut menu or press the F5 key. You can restore nine previous views in the current sketching environment by using this option.

Zoom

Ribbon: View > Navigate > Zoom drop-down > Zoom
Navigation Bar: Zoom flyout > Zoom



The **Zoom** tool is used to interactively zoom in and out of the drawing view. When you choose this tool, the default cursor is replaced by the zoom cursor. You can zoom into the drawing by pressing the left mouse button and dragging the cursor down. Similarly, you can zoom out of the drawing by pressing the left mouse button and then dragging the cursor up. You can exit this tool by choosing another tool or by pressing ESC. You can also choose **Done** from the shortcut menu, which is displayed on right-clicking. You can also zoom into the drawing by rolling the scroll wheel of the mouse in the reverse direction. Similarly, you can zoom out of the drawing by rolling the scroll wheel in the forward direction.



Tip. You need to increase the drawing display area by zooming out from the drawing using the **Zoom** tool after increasing the grid spacing.

Zoom Selected

Ribbon: View > Navigate > Zoom drop-down > Zoom Selected
Navigation Bar: Zoom flyout > Zoom Selected



When you choose the **Zoom Selected** tool, you will be prompted to select an entity to zoom. Select an entity from the drawing area; it will be magnified to maximum extent and placed at the center of the drawing window. This tool can also be invoked by pressing the END key.

Pan

Ribbon: View > Navigate > Pan
Navigation Bar: Pan



The **Pan** tool is used to drag the current view in the drawing window. This option is generally used to display the contents of the drawing that are outside the display area, without actually changing the magnification of the current drawing. It is similar to holding the drawing and dragging it across the drawing window. You can also invoke the **Pan** tool by pressing and holding the middle scroll wheel of the mouse.

SKETCHING ENTITIES

Getting acquainted with the sketching entities is an important part of learning Autodesk Inventor. A major part of the design is created using the sketched entities. Therefore, this section can be considered as one of the most important sections of the book. In Autodesk Inventor, the sketched entities are of two types: **Normal** and **Construction**. The normal entities are used to create a feature and become a part of it, but the construction entities are drawn just for reference and support, and cannot become a part of the feature. By default, all drawn entities are normal entities. To draw construction entities, choose the **Construction** tool from the **Format** panel of the **Sketch** tab. All entities drawn after choosing the **Construction** tool will be the construction entities. Deselect this tool by choosing it again to draw normal entities.

SPECIFYING THE POSITION OF ENTITIES DYNAMICALLY BY USING THE DYNAMIC INPUT



With this release of Autodesk Inventor, you can specify the position of sketching entities dynamically by using the Pointer Input and the Dimension Input. The Pointer Input is displayed when you invoke the sketching tools such as **Line**, **Rectangle**, **Arc**, and it displays the coordinates of the current location of the cursor. As you move the cursor, the coordinates change dynamically. When you specify the first point, the Pointer Input is displayed. The Pointer Input is displayed in the form of Cartesian Coordinates (X and Y). If you specify the second point or the subsequent points of entities, the Dimension Input will be displayed. The Dimension Input is displayed in the form of polar coordinates (Length and Angle).

To specify the position of sketching entities dynamically, invoke the required sketching tool and then move the cursor in the drawing window; the location of the cursor will be displayed in the cartesian coordinate in the Pointer Input. Press the TAB key and enter the X and Y coordinate values in the Pointer Input to specify the first point; you will be prompted to specify the endpoint or second point of the entity. Alternatively, you can specify the first point of the entity by clicking in the drawing window. On doing so, the Pointer Input will be modified to the Dimension Input and the polar coordinate input fields will be displayed. To specify the endpoint or second point of the entity, enter the length and angle values in the input fields. To toggle between the length and angle input fields, use the TAB key. If you specify input values by using the Dimension Input and then use the TAB key, lock icons will be displayed on the right of the input fields. The lock icons indicates that the values defined are constrained. Figure 2-10 shows the Pointer Input of a line and Figure 2-11 shows the Dimension Input of the endpoint of a line of length 20 mm at an angle of 45-degree.

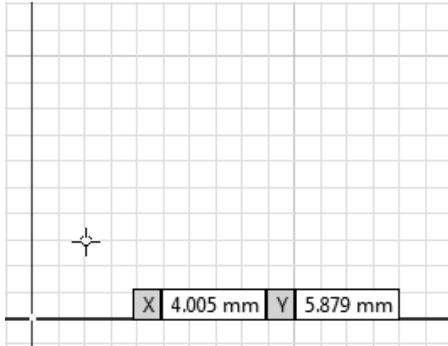


Figure 2-10 Pointer input of a line

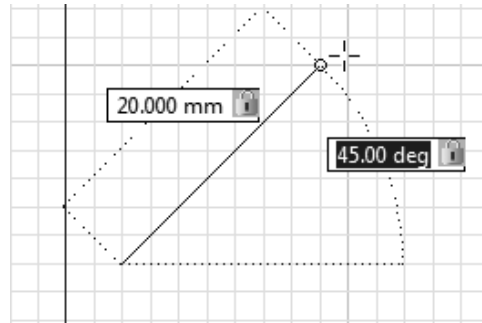


Figure 2-11 Dimension input of the endpoint of a line of length 20 mm at 45-degree

If some sketched entities exist in the drawing window and you start creating new entities in the drawing window, an appropriate constraint symbol will be displayed near the cursor. You can control the display of the Pointer Input and Dimension Input by using the **Application Options** dialog box. This dialog box can be invoked by choosing the **Application Options** tool from the **Options** panel. To control the display of Pointer Input and Dimension Input, choose the **Sketch** tab in the **Application Options** dialog box. Clear the **Enable the Heads-Up Display (HUD)** check box from the **Sketch** tab and choose the **OK** button from this dialog box. As a result, the display of Pointer Input and Dimension Input will be turned off and now you cannot enter the input values of the entities dynamically.

The sketcher entities in Autodesk Inventor are discussed next.

Drawing Lines

Ribbon: Sketch > Draw > Line
Toolbar: 2D Sketch Panel > Line drop-down > Line



Lines are the basic and one of the most important entities in the sketching environment. As mentioned earlier, you can draw either normal lines or construction lines. A line is defined as the shortest distance between two points. The two points are the start point and the endpoint of the line. Therefore, to draw a line, you need to define these two points. The parametric nature of Autodesk Inventor allows you to draw the initial line of any length or at any angle by just picking the points on the screen. After drawing the line, you can drive it to a new length or angle by using parametric dimensions. You can also create the line of actual length and angle directly by using the **Inventor Precise Input** toolbar. Both these methods of drawing the lines are discussed next.

Drawing Lines by Picking Points in the Drawing Window

This is a very convenient method to draw lines and is used extensively while sketching. When you invoke the **Line** tool from the **Draw** panel, the cursor (which was initially an arrow) is replaced by crosshairs with a yellow circle at the intersection. Also, you are prompted to select the start point of the line or drag off the endpoint for the tangent arc. In addition, the coordinates of the current location of the cursor are displayed in the Pointer Input and also at



the lower right corner of the Autodesk Inventor window. The point of intersection of the X and Y axes (black lines among grid lines) is the origin point. If you move the cursor close to the origin, it will snap to the origin automatically. To draw a line, specify a point anywhere in the drawing window; the Pointer Input will display both length and angle values as zero. Move the cursor; a rubber-band line will start from the specified point and the length and angle values will change accordingly in the Pointer Input. One end of this rubber-band line is fixed at the point specified in the drawing window and the other end is attached to the yellow circle in crosshairs. As you move the cursor after specifying the start point of the line, the Pointer Input will display the length and angle of the current location of the line. Click at the required position in the drawing window. Alternatively, enter the required length and angle values in the Pointer Input to specify the endpoint of the line. You can use the TAB key to toggle between the length and angle values in the Pointer Input. After specifying the endpoint of the line, a line is drawn and a new rubber-band line starts. The start point of the new rubber-band line is the endpoint of the last line and you are again prompted to specify the endpoint of the line. You can continue specifying the endpoints to draw continuous lines.

When you draw entities in Autodesk Inventor, valid constraints are applied automatically to entities. Therefore, when you draw continuous lines, the horizontal, vertical, perpendicular, and parallel constraints are applied automatically to them. The symbol of the applied constraint is displayed on the line while drawing it. You can exit the **Line** tool by pressing the ESC key. You can also right-click in the drawing window, and then choose **Done** from the shortcut menu displayed. Figures 2-12 and 2-13 display the **Perpendicular Constraint** and **Parallel Constraint** being applied to the lines while they are being drawn.

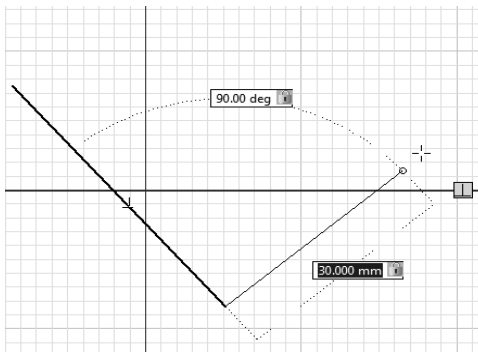


Figure 2-12 Drawing a line using the **Perpendicular Constraint**

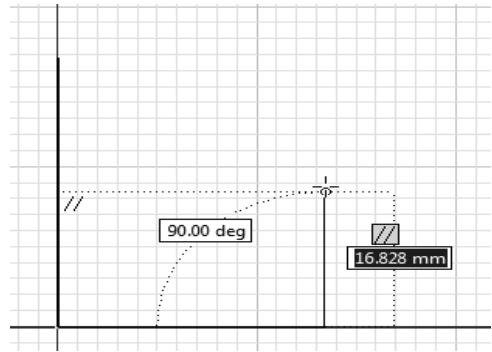


Figure 2-13 Drawing a line using the **Parallel Constraint**



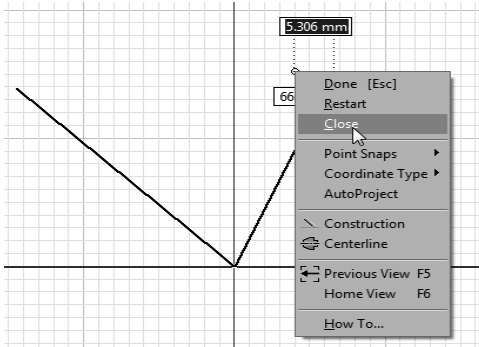
Note

The default screen appearance in the sketcher environment has been modified for clarity. To change the screen appearance, choose the **Application Options** tool from the **Options** panel; the **Application Options** dialog box will be displayed. Choose the **Colors** tab and then select the **Presentation** option from the **Color scheme** list box. Select **1 Color** from the **Background** drop-down list. Next, choose the **Apply** button from the **Application Options** dialog box to change the default appearance of the screen in the sketcher environment.



Tip. While drawing a sketch, constraints are displayed dynamically and are applied to it. To turn off the display of these constraints, press and hold the CTRL key and draw the sketch.

With this release of Autodesk Inventor, you can close a sketch that has two or more than two lines. To do so, create two or more than two continuous lines and then right-click in the drawing window; a shortcut menu will be displayed. Choose the **Close** option from the shortcut menu; a line joining the endpoint of the current line and the start point of the first line will be created and the sketch will be closed. Figure 2-14 shows the **Close** option being chosen from the shortcut menu to close the sketch and Figure 2-15 shows the closed sketch created.



*Figure 2-14 Choosing the **Close** option from the shortcut menu*

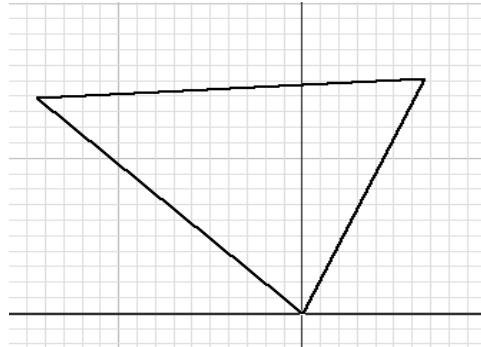


Figure 2-15 Closed sketch created

Drawing Lines by Specifying Exact Values

This is another method of drawing lines in Autodesk Inventor. In this method, you use the **Inventor Precise Input** toolbar to define the coordinates of the start point and the endpoint of lines. To display the **Inventor Precise Input** toolbar for the line, first invoke the **Line** tool. Next, click on the down arrow displayed at bottom of the **Draw** panel in the **Sketch** tab; the **Draw** panel will expand. Choose the **Precise Input** tool from this panel. As mentioned earlier, the origin of the drawing lies at the intersection of the X and Y axes. The X and Y coordinates of this point are 0, 0. You can take the reference of this point to draw lines. There are two methods to define the coordinates using this toolbar. Both the methods are discussed next.

Specifying Coordinates with respect to the Origin

The system of defining the coordinates with respect to the origin of the drawing is termed as the **absolute coordinate system**. By default, the origin lies at the intersection of the X and Y axes. All the points in this system are defined with respect to this origin. To define the points, you can use the following four methods.

Defining the Absolute X and Y Coordinates. In this method, you will define the X and Y coordinates of the new point with respect to the origin. To invoke this method, select the **Indicate a point location by typing X and Y values** option from the drop-down list in the **Inventor Precise Input** toolbar. The exact X and Y coordinates of the point can be entered in the **X** and **Y** edit boxes provided in this toolbar.

Defining the Absolute X Coordinate and the Angle from the X Axis. In this method, you will define the absolute X coordinate of a point with respect to the origin and the angle that this line makes with the positive X axis. The angle will be measured in the counterclockwise direction from the positive X axis. To invoke this method, select the **Specify a point using X coordinate and angle from X axis** option from the drop-down list. The X coordinate of the new point and the angle can be defined in the respective edit boxes in the **Inventor Precise Input** toolbar.

Defining the Absolute Y Coordinate and the Angle from the X Axis. In this method, you will define the absolute Y coordinate of a point with respect to the origin and the angle that this line makes with the positive X axis. To invoke this method, select the **Specify a point using Y coordinate and angle from X axis** option from the drop-down list. The Y coordinate of the new point and the angle can be defined in the respective edit boxes in the **Inventor Precise Input** toolbar.

Specifying the Distance from the Origin and the Angle from the X Axis. In this method, you will define the distance of the point from the origin and the angle that this line makes with the X axis. To invoke this method, select the **Specify a point using distance from the origin and angle from X axis** option from the drop-down list. The distance and the angle can be defined in the respective edit boxes.

Specifying Coordinates with respect to the Last Point

This system of specifying the coordinates with respect to the previous point is termed as the **relative coordinate system**. Note that this system of defining the points cannot be used for specifying the first point (the start point of the line). All absolute coordinate methods for specifying a point with respect to the origin can also be used with respect to the last specified point by choosing the **Precise Delta** button along with the respective method. This button will be available only after you specify the start point of the first line.



Note

While drawing continuous lines, when you move the cursor close to the start point of the first line, the yellow circle changes to green and the cursor snaps to the start point. On selecting the point at this stage the loop will be closed and you will exit the current line chain.

*To draw centerlines, first choose the **Centerline** tool from the **Format** panel and then create the line. Alternatively, select the required entities from the drawing window and then choose the **Centerline** tool; the selected entities will become centerlines.*

Restarting a Line

To restart a line, right-click and choose **Restart** from the shortcut menu. The start point of the line is canceled and you are prompted to select the start point of the line.

Drawing Circles

In Autodesk Inventor, you can draw circles using two methods. You can draw a circle by defining the center and the radius of the circle or draw a circle that is tangent to three specified lines. Both these methods of drawing the circle are discussed next.



Drawing Circles by Specifying the Center Point and Radius

Ribbon:

Sketch > Draw > Circle drop-down > Circle Center Point

Toolbar:

2D Sketch Panel > Circle drop-down > Center Point Circle



This is the default method of drawing circles. In this method, you need to define the center point and radius of a circle. To draw a circle using this method, choose the **Circle Center Point** tool from the **Draw** panel, see Figure 2-16; you will be prompted to select the center of the circle. Specify the center point of the circle in the drawing window; you will be prompted to specify a point on the circle. Click at the required location in the drawing window to specify a point on the circumference of the circle. This point will define the radius of the circle. Alternatively, enter the required value in the Pointer Input to specify the diameter of the circle. You can also specify the center and the radius using the **Inventor Precise Input** toolbar. Figure 2-17 shows a circle drawn by using the center and the radius.

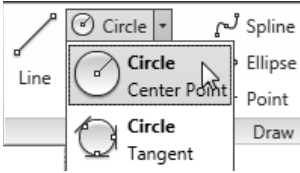


Figure 2-16 Tools in the Circle drop-down

Drawing Circles by Specifying Three Tangent Lines

Ribbon:

Sketch > Draw > Circle drop-down > Circle Tangent

Toolbar:

2D Sketch Panel > Circle drop-down > Tangent Circle



This is the second method of drawing circles and is used to draw a circle tangent to three selected lines. To draw a circle using this method, choose the **Circle Tangent** tool from the **Draw** panel, see Figure 2-16; you will be prompted to select the first, second, and third line, sequentially. As soon as you specify the third line, a circle tangent to all the three specified lines will be drawn, as shown in Figure 2-18.

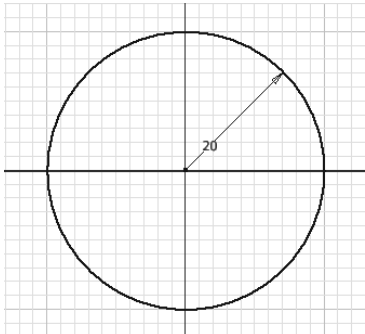


Figure 2-17 Circle drawn using the center point and the radius of the circle

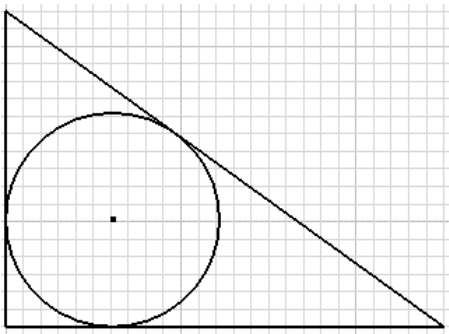


Figure 2-18 Circle drawn using three tangent lines

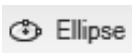
Drawing Ellipses

Ribbon:

Sketch > Draw > Ellipse

Toolbar:

2D Sketch Panel > Circle drop-down > Ellipse



To draw an ellipse, choose the **Ellipse** tool from the **Draw** panel; you will be prompted to specify the center of the ellipse. Select a point to specify the center of the ellipse; you will be prompted to specify the first axis point. Specify a point

to define the first axis of the ellipse; you will be prompted to select a point on the ellipse. Select a point on the ellipse; the ellipse will be created. You can also specify these points using the **Inventor Precise Input** toolbar. However, remember that you cannot use the relative options for defining the points of the ellipse. Therefore, if you use the **Inventor Precise Input** toolbar for drawing the ellipse, all the values will be specified from the origin. However, you can redefine the origin by choosing the **Precise Redefine** button and placing it at the point that you want to define as the origin. Figure 2-19 shows an ellipse.

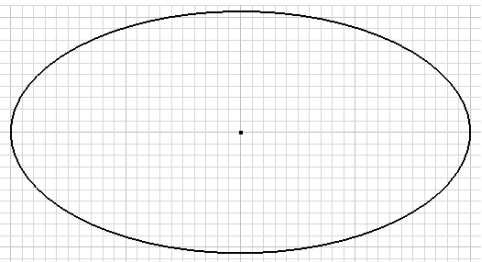


Figure 2-19 An ellipse drawn in the sketching environment

Drawing Arcs

Autodesk Inventor provides three methods for drawing arcs, which are discussed next.



Drawing an Arc by Specifying Three Points

Ribbon:	Sketch > Draw > Arc drop-down > Arc Three Point
Toolbar:	2D Sketch Panel > Arc drop-down > Three Point Arc



This is the default method of drawing arcs. To create an arc with three points, choose the **Arc Three Point** tool from the **Draw** panel, see Figure 2-20, and then specify three points. The first point is the start point of the arc, the second point is the endpoint of the arc, and the third point is a point on the arc. You can define these points by specifying them in the drawing window or by using the **Inventor Precise Input** toolbar. You can also use the Pointer Input for specifying the second and the third point of the arc. Figure 2-21 shows an arc drawn using this method.

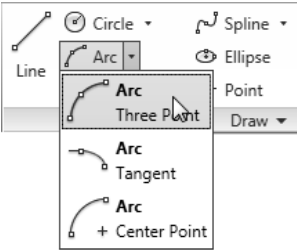


Figure 2-20 Tools in the Arc drop-down

Drawing an Arc Tangent to an Existing Entity

Ribbon:	Sketch > Draw > Arc drop-down > Arc Tangent
Toolbar:	2D Sketch Panel > Arc drop-down > Tangent Arc



This method is used to draw an arc that is tangent to an existing open entity. The open entity can be an arc or a line. To draw an arc using this method, choose the **Arc Tangent** tool from the **Draw** panel (see Figure 2-20); you will be prompted to select the start point of the arc. The start point of the arc must be the start point or endpoint of an existing open entity. Once you specify the start point, a rubber-band arc will start from it. Note that this arc is tangent to the selected entity. Now, you will be prompted to specify the endpoint of the arc. Click on the drawing window to specify the endpoint of the arc. Alternatively, enter the radius and the angle values in the Pointer Input to specify the endpoint of the arc. Here, it is very important to mention that the **Inventor Precise Input**

toolbar or the Pointer Input cannot be used to select the start point of this arc. However, you can use this toolbar to specify the endpoint of this arc. Figure 2-22 shows an arc drawn tangent to the line.

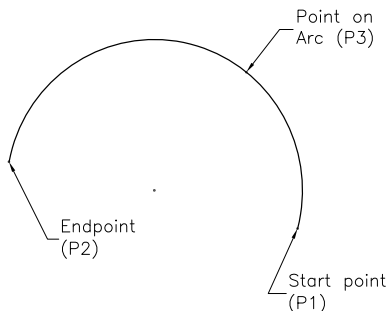


Figure 2-21 Drawing the three points arc

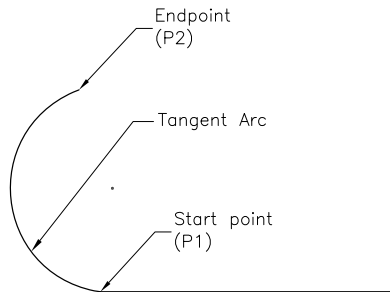


Figure 2-22 Drawing the tangent arc

Drawing Tangent/Normal Arcs Using the Line Tool

You can also draw a tangent or a normal arc when you are inside the **Line** tool. At least a line or an arc should be drawn before drawing an arc using this method. To draw an arc using the **Line** tool, draw a line or an arc and then invoke the **Line** tool. When you are prompted to select the start point of the line, move the cursor close to the point from where you want to start the tangent or normal arc; the yellow circle in the cursor turns green. Select the point at this stage; the green circle in the cursor turns gray. Press the left mouse button and drag the mouse; four construction lines appear at the start point displaying the normal and tangent directions. If you drag along the tangent direction, a tangent arc is drawn. But if you drag along the normal direction, an arc normal to the selected entity is drawn.

Drawing an Arc by specifying the Center, Start, and Endpoint

Ribbon: Sketch > Draw > Arc drop-down > Arc Center Point
Toolbar: 2D Sketch Panel > Arc drop-down > Center Point Arc



This method is used to draw an arc by specifying the center point, start point, and endpoint of the arc. To draw an arc using this method, choose the **Arc Center Point** tool from the **Draw** panel (see Figure 2-20). On doing so, you will be prompted to specify the center point of the arc. Once you specify the center point of the arc, you will be prompted to specify the start point and then the endpoint of the arc, see Figure 2-23. You can also specify the start point and endpoint of the arc by using the Pointer Input. In case of start point, you need to specify the radius and angle of the arc from the center point. Whereas, in case of endpoint, you need to specify the arc length in terms of angle value. You can use the TAB key to toggle between the input values of the Pointer Input. As you define the center point and the start point, the radius of the arc will be defined automatically. So, the third point is just used to define the arc length. An imaginary line is drawn from the cursor to the center of the arc. The point at which the arc intersects the imaginary line will then be taken as the endpoint of the arc, see Figure 2-24. You can also use the **Inventor Precise Input** toolbar to specify these three points of the arc.

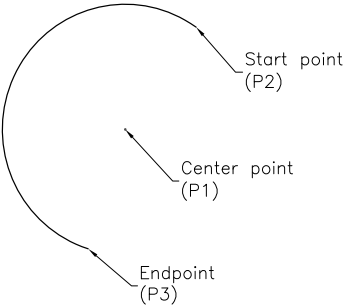


Figure 2-23 The arc created by specifying the center point, start point, and endpoint

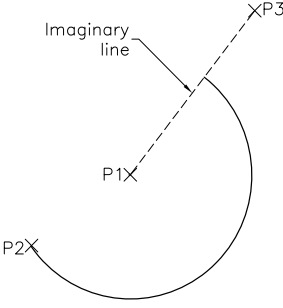


Figure 2-24 The imaginary line created while drawing the center point arc

Drawing Rectangles

In Autodesk Inventor, rectangles can be drawn by using two methods that are discussed next.



Drawing Rectangles by Specifying Two Opposite Corners

Ribbon: Sketch > Draw > Rectangle drop-down > Rectangle Two Point
Toolbar: 2D Sketch Panel > Rectangle drop-down > Two Point Rectangle



This is the default method used to draw a rectangle by specifying its two opposite corners. To draw a rectangle by using this method, choose the **Rectangle Two Point** tool from the **Draw** panel, see Figure 2-25; you will be prompted to specify the first corner of the rectangle and the Pointer Input will be displayed. Click at the required location to specify the first corner of the rectangle. Once you specify the first corner, you will be prompted to specify the opposite corner of the rectangle and the Pointer Input will be modified. Click to specify the second corner or enter the length and height of the rectangle in the Pointer Input. Figure 2-26 shows a rectangle drawn using the **Rectangle Two Point** tool.

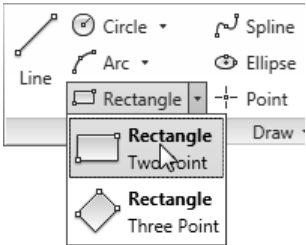


Figure 2-25 Tools in the **Rectangle** drop-down

Drawing Rectangles by Specifying Three Points on a Rectangle

Ribbon: Sketch > Draw > Rectangle drop-down > Rectangle Three Point
Toolbar: 2D Sketch Panel > Rectangle drop-down > Three Point Rectangle



You can draw a rectangle by specifying its three points. In this method, the first two points are used to define the length and angle of one of the sides of the rectangle and the third point is used to define the length of the other side. To create a rectangle by using this method, choose the **Rectangle Three Point** tool from the **Draw** panel of the **Sketch** tab, see Figure 2-25; you will be prompted to specify the first corner of the rectangle. Once you specify it, you will be prompted to specify the second corner of the rectangle. Both these corners are along the same direction. As a result, you can use these points to define the

length of one side of the rectangle. After specifying the second corner, you will be prompted to specify the third corner. This corner is used to define the length of the other side of the rectangle. Note that if you specify the second corner at a certain angle, then the resultant rectangle will also be inclined. You can also specify the first, second, and third points of the rectangle by using the Pointer Input. In case of second point, you need to specify the length and angle of rectangle in the input value fields of the Pointer Input. Whereas, in case of endpoint, you need to specify the height of the rectangle. You can use the TAB key to toggle between the input values of the Pointer Input. You can also specify the three points for drawing the rectangle using the **Inventor Precise Input** toolbar. Figure 2-27 shows an inclined rectangle drawn by using the **Three Point Rectangle** tool.

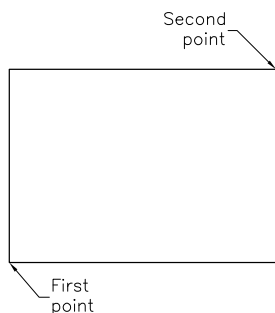


Figure 2-26 Drawing a rectangle using two points

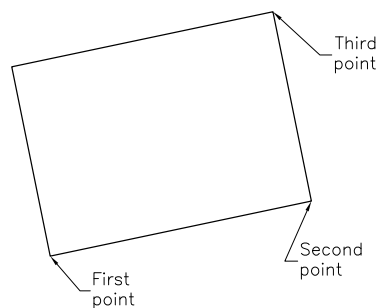


Figure 2-27 Drawing the three-point rectangle at an angle

Drawing Polygons

Ribbon: Sketch > Draw > Polygon
Toolbar: 2D Sketch Panel > Polygon



The polygons drawn in Autodesk Inventor are regular polygons. A regular polygon is a multi-sided geometric figure in which the length of all sides and the angle between them are the same. In Autodesk Inventor, you can draw a polygon with the number of sides ranging from 3 to 120. When you invoke the **Polygon** tool, the **Polygon** dialog box will be displayed, as shown in Figure 2-28, and you will be prompted to select the center of the polygon. The options in this dialog box are discussed next.

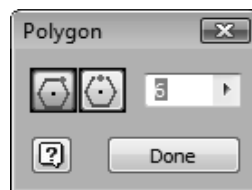


Figure 2-28 The **Polygon** dialog box

Inscribed

This is the first button in the **Polygon** dialog box and is chosen by default. This option is used to draw an inscribed polygon. An inscribed polygon is the one that is drawn inside an imaginary circle such that its vertices touch the circle. Once you have specified the polygon center, you will be prompted to specify a point on the polygon. In case of an inscribed polygon, the point on the polygon specifies one of its vertices, see Figure 2-29.

Circumscribed

This is the second button in the **Polygon** dialog box and is used to draw a circumscribed polygon. A circumscribed polygon is the one that is drawn outside an imaginary circle such that its edges are tangent to the imaginary circle. In case of a circumscribed polygon, the point on the polygon is the midpoint of one of the polygon edges, see Figure 2-30.

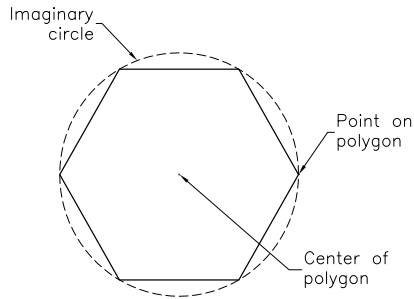


Figure 2-29 Drawing a six-sided inscribed polygon

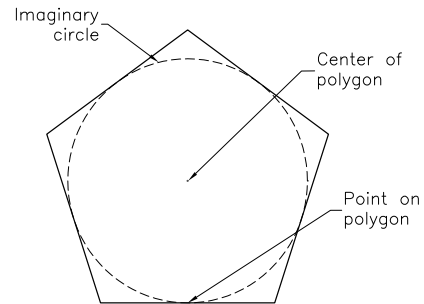


Figure 2-30 Drawing a five-sided circumscribed polygon

Number of Sides

This edit box is used to specify the number of sides of the polygon. The default value is 6. You can enter any value ranging from 3 to 120 in this edit box.

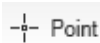


Note

The rectangles and polygons are a combination of individual lines. All the lines can be separately selected or deleted. However, when you select one of the lines and drag, the entire rectangle or polygon will be considered as a single entity. As a result, the entire object will be moved or stretched.

Placing Points

Ribbon:	Sketch > Draw > Point
Toolbar:	2D Sketch Panel > Point, Center Point



In Autodesk Inventor, you can place the sketched points in a sketch using the **Point** tool. To place a point, choose the **Point** tool from the **Draw** panel; you will be prompted to select the center point. Specify the center point; a point will be placed. You can specify the location of a point in the sketch by picking a point from the graphics window or by entering the value in the **Inventor Precise Input** toolbar.

Creating Fillets

Ribbon:	Sketch > Draw > Fillet/Chamfer drop-down > Fillet
Toolbar:	2D Sketch Panel > Fillet/Chamfer drop-down > Fillet



Filleting is defined as the process of rounding the sharp corners of a sketch. This is done to reduce the stress concentration in the model. Using the **Fillet** tool, you can round the corners of the sketch by creating an arc tangent to both the selected

entities. The portions of the selected entities that comprise the sharp corners are trimmed when the fillet is created. When you invoke this tool from the **Fillet/Chamfer** drop-down, refer to Figure 2-31, the **2D Fillet** dialog box will be displayed with the default fillet radius, as shown in Figure 2-32, and you will be prompted to select the lines or the arcs to be filleted. If you have already created some fillets, their radius values will be stored as preset values. You can select these preset values from the list that is displayed when you choose the arrow provided on the right of the edit box.

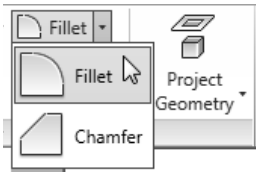


Figure 2-31 Tools in the **Fillet/Chamfer** drop-down



Figure 2-32 The **2D Fillet** dialog box

You can create any number of fillets of similar or dissimilar radii. If the **Equal** button in the **2D Fillet** dialog box is chosen, the dimension of the fillet will be placed only on the first fillet and not on the other fillets created by using the same sequence, see Figure 2-33. On modifying the dimension of the first fillet, all instances of fillet will be modified. To create fillets of independent radii values, deactivate the **Equal** button before creating fillets. The fillets thus created will show individual dimensions, see Figure 2-34. As a result, you can modify the dimension of one fillet without affecting the other. You can fillet two parallel or perpendicular lines, intersecting lines or arcs, non-intersecting lines or arcs, and a line and an arc.

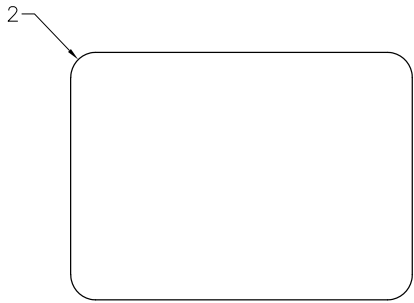


Figure 2-33 Rectangle filleted using the same radius with the **Equal** button chosen

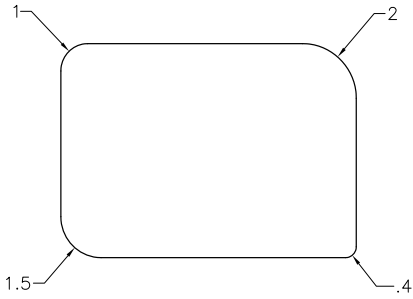
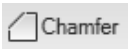


Figure 2-34 Rectangle filleted using different radii with the **Equal** button deactivated

Creating Chamfers

Ribbon:	Sketch > Draw > Fillet/Chamfer drop-down > Chamfer
Toolbar:	2D Sketch Panel > Fillet/Chamfer drop-down > Chamfer



Chamfering is defined as the process of beveling the sharp corners of a sketch. This is the second method of reducing stress concentration. To chamfer sketched entities, choose the **Chamfer** tool from the **Draw** panel (see Figure 2-31); the **2D Chamfer** dialog box will be displayed, as shown in Figure 2-35. Also, you will be prompted to select

the lines to be chamfered. Select the lines; the chamfer will be created. The options in the **2D chamfer** dialog box are discussed next.

Create Dimensions



The **Create Dimensions** button is chosen to show the dimensions of the chamfer on the sketch. When you chamfer two lines, the dimensions of the chamfer are shown in the sketch. If you choose this button again, the chamfer dimensions will not be displayed in the sketch when you create another chamfer.

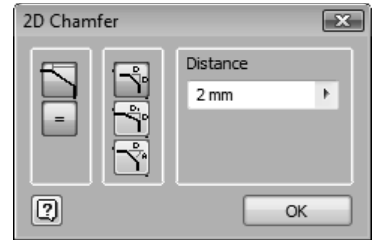


Figure 2-35 The 2D Chamfer dialog box

Equal



The **Equal** button is chosen to create multiple chamfers with the same parameters. This button is enabled only if the **Create Dimensions** button is chosen.

Equal Distance



The **Equal Distance** button is chosen to create an equal distance chamfer. The distance of the vertex along the two selected edges is the same. As a result, a 45-degree chamfer is created using this method. The distance value is specified in the **Distance** edit box. If the **Create Dimension** button is chosen, two dimensions of the same value will be shown in the sketch, as shown in Figure 2-36.

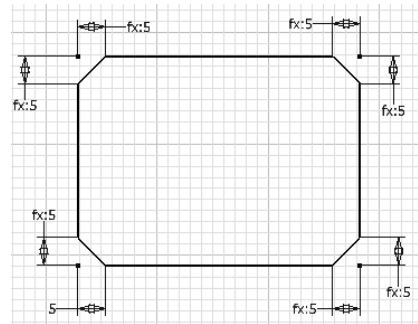


Figure 2-36 Chamfer with dimension values

Unequal Distances



The **Unequal Distances** button is chosen to create a chamfer with two different distances. The distance values are specified in the **Distance1** and **Distance2** edit boxes. The distance value specified in the **Distance1** edit box is measured along the edge selected first. Similarly, the value in the **Distance2** edit box is measured along the edge selected next. Figure 2-37 shows a chamfer created by using the **Unequal Distances** method.

Distance and Angle



The **Distance and Angle** button is chosen to create a chamfer by specifying a distance and an angle. On choosing this button, the distance needs to be specified in the **Distance** edit box and the angle in the **Angle** edit box. The specified angle is measured from the first edge selected to chamfer, see Figure 2-38.



Tip. If multiple chamfers are created with same values, the dimension value is displayed only at the first instance. At the remaining chamfers, the dimension will be displayed as *fx* of the value, which means the function of the original value.

You can also select the vertex to create a fillet or chamfer. The two entities forming the selected vertex will be filleted or chamfered using the current parameters.

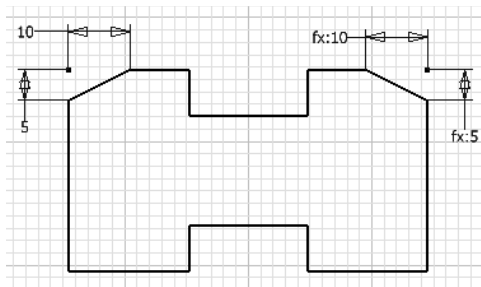


Figure 2-37 The unequal distances chamfer created

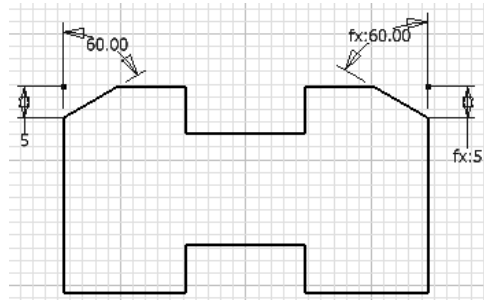
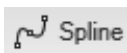


Figure 2-38 The distance and angle chamfer created

Drawing Splines

Ribbon: Sketch > Draw > Spline drop-down > Spline
Toolbar: 2D Sketch Panel > Line drop-down > Spline



To draw a spline, choose the **Spline** tool from the **Draw** panel, see Figure 2-39; you will be prompted to specify the first point of the spline. Specify the start point; you will be prompted to specify the next point of the spline. This process will continue until you terminate the spline creation. To end the spline at the current point, double-click in the drawing window or right-click to display the shortcut menu and choose **Create**. Note that if you choose **Done** from the shortcut menu, the spline will not be drawn. You can also end the spline creation by pressing the ENTER key. Note that after creating a spline, the square and diamond points will be displayed on the spline along with the tangent handles, as shown in Figure 2-40. You can drag these square and diamond points to modify the shape of the spline.

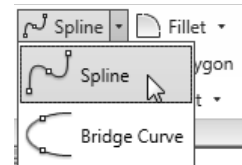


Figure 2-39 Tools in the **Spline** drop-down

You can undo the last drawn spline segment while drawing a spline. This can be done by choosing the **Back** option from the shortcut menu that is displayed when you right-click.

You can also draw a spline tangent to an existing entity. To draw the tangent spline, select the point where the spline should be tangent. Next, hold the left mouse button and drag it; a construction line will be drawn, which displays the possible tangent directions for the spline. Drag the mouse in the required direction to draw the tangent spline and release the left mouse button. Figure 2-40 shows a spline drawn by specifying different points and Figure 2-41 shows a spline drawn tangent to an existing line.



Tip. Autodesk Inventor allows you to invoke the last used tool by right-clicking anywhere in the drawing window. For example, create a line using the **Line** tool. If you want to create another line, right-click anywhere in the drawing window and choose the **Repeat Line** option from the shortcut menu. Alternatively, you can invoke the last used tool by pressing the SPACEBAR key.

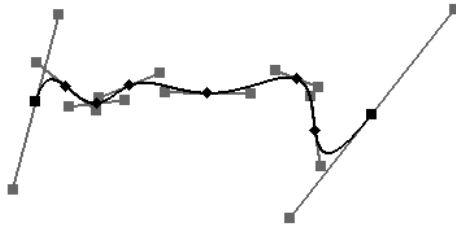


Figure 2-40 A spline drawn by specifying different points

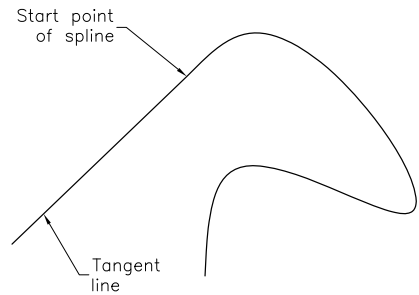


Figure 2-41 A spline drawn tangent to a line

Creating a Smooth Curve between Two Existing Curves

Ribbon: Sketch > Draw > Spline drop-down > Bridge Curve



In Autodesk Inventor, you can create a smooth (G2) continuous curve between two existing curves. The existing curves can be arcs, lines, splines, or projected curves.

To create a smooth curve, choose the **Bridge Curve** tool from the **Draw** panel (see Figure 2-39); you will be prompted to select the curves one after the other. Select the two curves; a smooth G2 continuous curve, known as bridge curve, will be created between the selected curves. The profile of the bridge curve depends on the position where two existing curves are selected. Figure 2-42 shows the point of selection of two curves and the resulting bridge curve. Figure 2-43 shows different points of selection of the same curve and the resulting bridge curve.

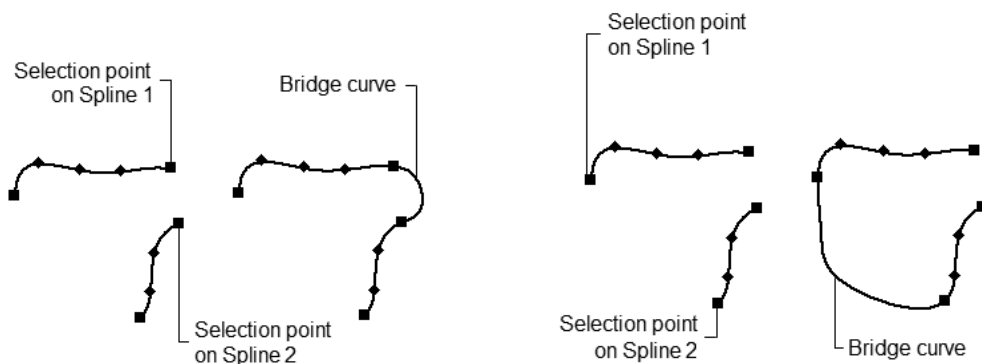


Figure 2-42 Bridge curve created between curves at the selection points

Figure 2-43 Bridge curve created between curves at different selection points

DELETING SKETCHED ENTITIES

To delete the sketched entity, first ensure that no drawing tool is active. If it is, press the ESC key. Now, select the entity you want to delete using the left mouse button and then right-click to display the shortcut menu. Choose the **Delete** option from this shortcut menu. You can also press the DELETE key to delete the selected entities. To delete more than one entity, you can use a window or a crossing. Deleting entities using these methods are discussed next.

Deleting Entities by Using a Window

A window is defined as a box created by pressing and holding the left mouse button and dragging the cursor from the left to the right in the drawing window. The window has a property that all the entities that lie completely inside the window will be selected. The box defined by the window consists of continuous lines. All the selected entities will be displayed in cyan color. After selecting the entities, right-click and choose **Delete** from the shortcut menu or press the DELETE key to delete all the selected entities.

Deleting Entities by Using a Crossing

A crossing is defined as a box created by pressing and holding down the left mouse button and dragging the cursor from the right to the left in the drawing window. The crossing has a property that all entities that lie completely or partially inside the crossing or the entities that touch the crossing will be selected. The box defined by the crossing consists of dashed lines. Once the entities are selected, right-click and choose **Delete** from the shortcut menu.



Tip. You can add or remove an entity from the selection set by pressing the **SHIFT** or the **CTRL** key and then selecting the entity by using the left mouse button. If the entity is already in the current selection set, it will be removed from the selection set. If not, it will be added to set.

FINISHING A SKETCH

After creating the required sketch, you need to save it. But before you save the sketch, you need to finish the sketch and come out of the sketching environment. To do so, choose the **Finish Sketch** tool from the **Exit** panel of the **Sketch** tab; the sketch will be finished and you will switch to the **Home** view. This view enables you to view and create the modeling features with ease. After switching to the **Modeling** environment, you can save the document.



TUTORIALS

Although Autodesk Inventor is parametric in nature, in this chapter you will use the **Inventor Precise Input** toolbar and the dynamic input method to draw objects. This is to make you comfortable with various drawing options in Autodesk Inventor. From the next chapter onward, you will use the parametric feature of Autodesk Inventor to size or draw the entities as per the desired dimension values.

Although the sketches for the tutorials in this chapter are to be drawn on the other sketching planes, you will draw them on the default XY plane. In the later chapters, you will learn how to change a sketching plane.

Tutorial 1

In this tutorial, you will draw the sketch of the model shown in Figure 2-44. The sketch to be drawn is shown in Figure 2-45. Do not dimension it, as the dimensions are given only for reference.
(Expected time: 30 min)

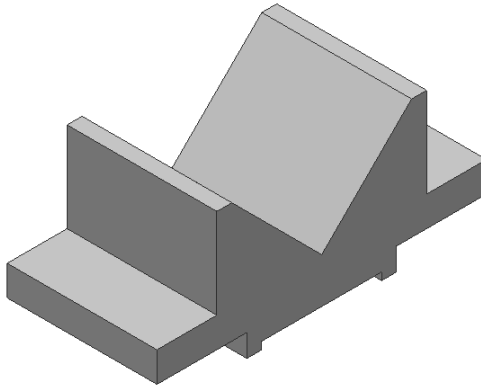


Figure 2-44 Model for Tutorial 1

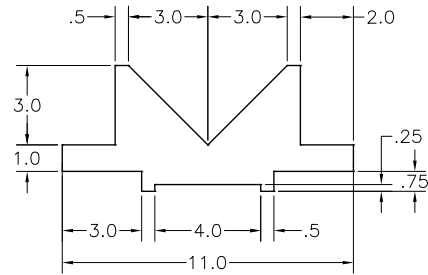



Figure 2-45 Sketch of the model

The following steps are required to complete this tutorial:

- Start a new Autodesk Inventor session and then start a new metric part file.
- Invoke the **Line** tool and draw the sketch by specifying the coordinates of the points in the **Inventor Precise Input** toolbar.
- Save the sketch with the name *Tutorial1* and close the file.

Starting Autodesk Inventor

- Start Autodesk Inventor by double-clicking on its shortcut icon on the desktop of your computer. Alternatively, choose **Start > Programs > Autodesk > Autodesk Inventor 2011 > Autodesk Inventor Professional 2011** from the taskbar; a new session of Autodesk Inventor is started.
- Choose the **New** tool from the **Launch** panel of the **Get Started** tab; the **New File** dialog box is displayed. 
- Choose the **Metric** tab and then double-click on the **Standard (mm).ipt** icon to start a standard metric template; a new metric standard part file is started in the **Sketching** environment.

The major and minor grid lines are also displayed in the drawing window, along with the X and Y axes.



Note

You can also open a standard metric template by selecting **Standard.ipt** from the **Default** tab of the **New File** dialog box.

Drawing the Sketch

As mentioned earlier, Autodesk Inventor is parametric in nature. Therefore, you can draw the sketch from any point in the drawing window. However, it is recommended that you initially use the **Inventor Precise Input** toolbar to specify the points. Once you are conversant with this design tool, you can specify the points directly in the drawing window.

1. Choose the **Line** tool from the **Draw** panel in the **Sketch** tab. Next, choose **Precise Input** from the **Draw** panel in the **Sketch** tab; the **Inventor Precise Input** toolbar is displayed. Double-click on the title bar of this toolbar to dock it. If you want, you can also leave this toolbar floating on the screen.



Initially, the **Precise Input** button is not enabled. This button is enabled only when you invoke a sketching tool. Since all initial settings are configured, you can now start drawing the sketch.

When you invoke the **Line** tool, the cursor is replaced by the drawing cursor that has a yellow circle at the intersection of crosshairs. This circle is used to snap to the points in the drawing window.

2. Specify **0** as the start point of the sketch in both the **X** and **Y** edit boxes of the **Inventor Precise Input** toolbar and then press ENTER; you are prompted to specify the endpoint of the line.
3. In the **Inventor Precise Input** toolbar, enter **-3** and **3** in the **X** and **Y** edit boxes, respectively and then press ENTER to define the endpoint of the line. On doing so, the first line of the sketch is drawn and you are prompted to select the endpoint of the line or drag off to create a tangent arc.

You will notice that the line is very small because the dimensions of the sketch are very small and the drawing display area is large. Therefore, you need to modify the drawing display area by using the drawing display tools. To modify this area, you can use the **Zoom** tool.

4. Choose the **Zoom** tool from the **Navigation Bar**; the drawing cursor is changed to an arrow cursor.
5. Move the cursor to the top of the drawing window, press and hold the left mouse button and then drag the cursor downward. Stop dragging the cursor once you notice that the display is adjusted.
6. Right-click to display the shortcut menu, and then choose **Done** to exit the **Zoom** tool.



You will notice that the creation of line is resumed and you are prompted to specify the endpoint of the next line.

7. The coordinates of the remaining points (see Figure 2-46) in the sketch are given next.

Point	Coordinates (X, Y)
3	-3.5,3
4	-3.5,0
5	-5.5,0
6	-5.5,-1
7	-2.5,-1
8	-2.5,-1.75
9	-2,-1.75
10	-2,-1.5
11	2,-1.5
12	2,-1.75
13	2.5,-1.75
14	2.5,-1
15	5.5,-1
16	5.5,0
17	3.5,0
18	3.5,3
19	3,3
20	0,0



Tip. You can use the **TAB** key to switch from the **X** edit box to the **Y** edit box and vice versa in the **Inventor Precise Input** toolbar.

8. After specifying all points, right-click to display the shortcut menu. Choose **Done** from the shortcut menu or press **ESC** to exit the **Line** tool. The final sketch for Tutorial 1 is shown in Figure 2-46. However, the points in the final sketch in this figure have been numbered for reference only.

While specifying various points, you will notice that some of the constraints are applied automatically to the lines. These constraints help you reduce the number of dimensions to be specified to complete the sketch.

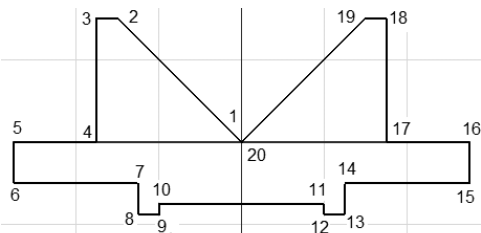


Figure 2-46 Final sketch for Tutorial 1



Note The method of applying additional constraints and using them to fully constrain the sketch will be discussed in Chapter 3.

Saving the Sketch

Remember that you cannot save a sketch in the sketching environment. This is because the sketching environment is just a part of the **Part** module in Autodesk Inventor. This environment is used only for drawing the sketches of features. Therefore, you need to exit the sketching environment to save the sketch for further use. The sketches in the **Part** module are saved in the *.ipt* format.

1. Choose the **Finish Sketch** button from the **Exit** panel; the **Sketching** environment is closed and you will switch to the **Home** view of the part modeling environment. Also, notice that the **Model** tab is activated in place of the **Sketch** tab. The options in the **Model** tab are used to create features. The options under this tab will be discussed in later chapters.
2. Choose **Save** from the **Quick Access Toolbar**; the **Save As** dialog box is displayed.

Whenever you invoke the **Save As** dialog box for the first time, all files are saved in the *My Documents* folder, by default.

3. Create a new folder with the name *Inventor_2011* in the C drive of your computer. In this folder, create a folder with the name *c02*.
4. Enter *Tutorial1* as the file name in the **File name** edit box, refer to Figure 2-47, and then choose the **Save** button from the **Save As** dialog box to save the sketch.

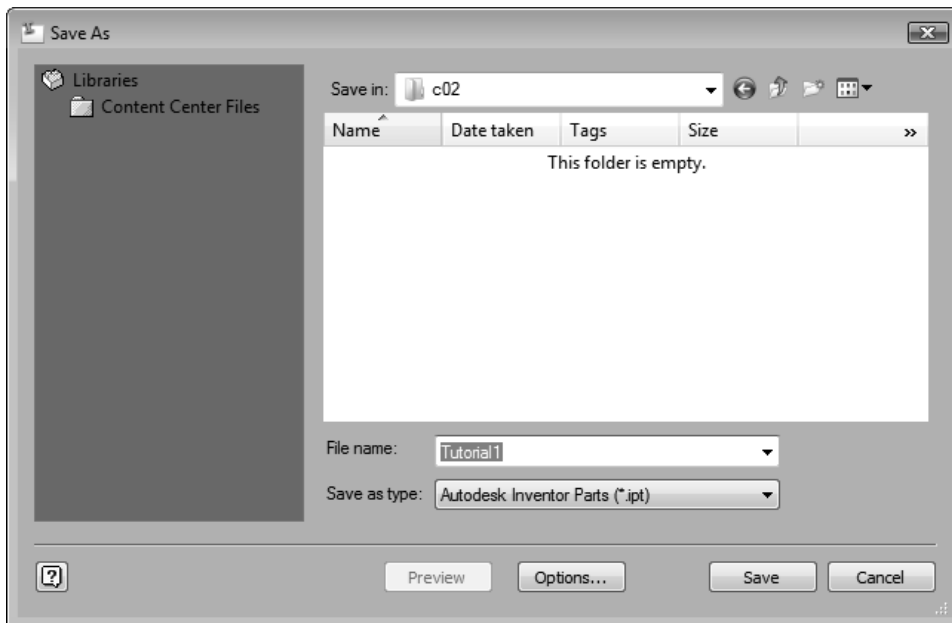


Figure 2-47 The **Save As** dialog box

5. Choose **Close > Close** from the **Application Menu** to close this file.

Tutorial 2

In this tutorial, you will draw the sketch for the model shown in Figure 2-48. The sketch to be drawn is shown in Figure 2-49. Do not dimension it, as the dimensions are given only for reference.
(Expected time: 30 min)

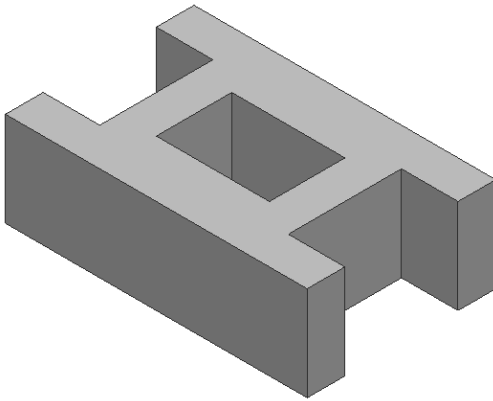


Figure 2-48 Model for Tutorial 2

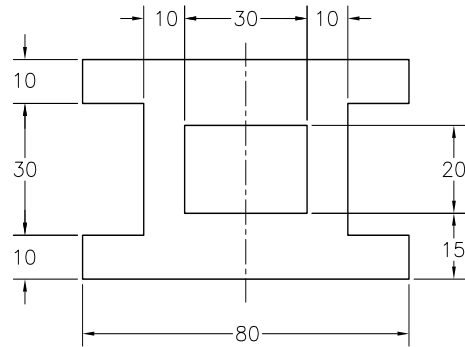


Figure 2-49 Dimensioned sketch for Tutorial 2

The following steps are required to complete this tutorial:

- Start a new metric standard part file.
- Draw the outer loop by specifying the length and angle lines by using the dynamic input method.
- Draw the inner closed loop by using the dynamic input method, refer to Figure 2-51.
- Save the sketch with the name *Tutorial2.ipt* and close the file.


Starting a New File

- Choose the **New** tool from **Launch** panel of the **Get Started** tab to display the **New File** dialog box. In this dialog box, choose the **Metric** tab and then double-click on the **Standard (mm).ipt** icon to start a standard metric template.

Drawing the Sketch


As evident from Figure 2-49, the sketch consists of two nested loops: inner loop, and outer loop. While extruding a nested loop, the inner loop can be subtracted from the outer loop. In this way, a cavity will be created automatically in the model when you extrude the sketch. This reduces the time and effort required in creating the inner cavity as another feature. Therefore, you can draw both the loops together for this tutorial.

The **Inventor Precise Input** toolbar was invoked in the previous tutorial; and therefore, it is available on the screen. You need to close it before drawing the sketch.

1. Close the **Inventor Precise Input** toolbar and then choose the **Line** tool from the **Draw** panel; you are prompted to specify the start point of the line or drag off an endpoint to create a tangent arc. Also, the coordinates of the current location of the cursor are displayed in the Pointer Input. 
2. Move the cursor and click when the cursor displays -40 and -25 in the Pointer Input or enter the X and Y coordinate values in the Pointer Input. As you specify the first point, you are prompted to specify the endpoint of the line and the Pointer Input is changed to Dimension Input.
3. Enter **80** in the length input field of the Dimension Input and press the TAB key; the angle input field becomes active. Enter **0** and then press ENTER; the first line is created and the Dimension Input is displayed again.
4. Move the cursor up and then enter **10** in the length input field of the Dimension Input. Next, press the TAB key and enter **90** in the angle input field and then press ENTER; the second line is created and the Dimension Input is displayed again.
5. Move the cursor toward left and then enter **15** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the third line is created and the Dimension Input is displayed again.
6. Move the cursor up and then enter **30** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the fourth line is created and the Dimension Input is displayed again.
7. Move the cursor toward right and then enter **15** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the fifth line is created and the Dimension Input is displayed again.
8. Move the cursor up and then enter **10** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the sixth line is created and the Dimension Input is displayed again.
9. Move the cursor toward left and then enter **80** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the seventh line is created and the Dimension Input is displayed again.
10. Move the cursor down and then enter **10** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the eighth line is created and the Dimension Input is displayed again.
11. Move the cursor toward right and then enter **15** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the ninth line is created and the Dimension Input is displayed again.

12. Move the cursor down and then enter **30** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the tenth line is created and the Dimension Input is displayed again.
13. Move the cursor toward left and then enter **15** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the eleventh line is created and the Dimension Input is displayed again.
14. Move the cursor down and then enter **10** in the length input field. Next, press the TAB key and enter **90** in the angle input field. Then, press ENTER; the twelfth line is created and the Dimension Input is displayed again.
15. Right-click to display the shortcut menu. Choose **Done** from the shortcut menu to exit the **Line** tool. The sketch of the outer loop is shown in Figure 2-50. You need to arrange the position of dimensions in the sketch by dragging them to view the sketch clearly.

Next, you need to draw the inner loop. You can draw the loop by using the **Two Point Rectangle** tool.

16. Choose the **Rectangle Two Point** tool from **Sketch > Draw > Rectangle** drop-down; you are prompted to specify the first corner of the rectangle. 
17. Press the TAB key and then enter **-15** and **-10** in the Pointer Input. Alternatively, move the cursor and click when the cursor displays **-15** and **-10** in the Pointer Input. As you click to specify the first corner, you are prompted to specify the opposite corner of the rectangle and the Pointer Input is changed to Dimension Input.
18. Move the cursor diagonally upward in the right direction and enter **30** and **20** in the horizontal and vertical input fields of the Dimension Input, respectively. Press ENTER and right-click; a shortcut menu is displayed.
19. Choose **Done** from the shortcut menu; the sketch is created, as shown in Figure 2-51.

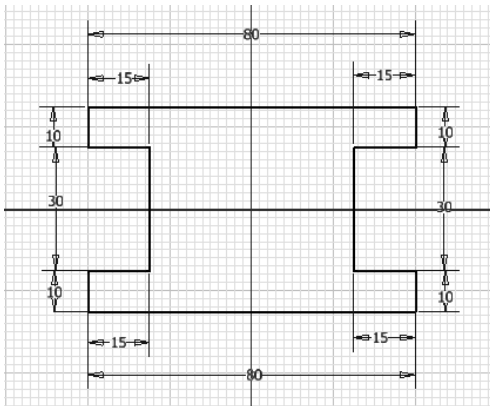


Figure 2-50 Sketch of the outer loop

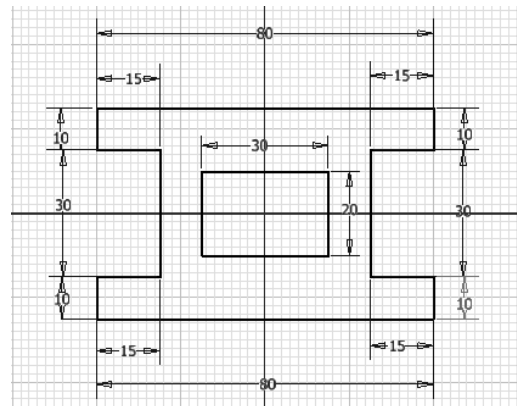


Figure 2-51 Completed sketch for Tutorial 2

**Note**

The angular dimensions have not been shown in Figures 2-50 and 2-51 for clarity of sketches. You can control the display of the linear and angular dimensions of the sketch by using the **Sketch** tab of the **Application Options** dialog box. This dialog box is displayed when you choose the **Sketch** tab of the **Application Options** tool from the **Options** panel. Alternatively, choose **Options** from the **Application Menu** to invoke the **Application Options** dialog box. In the **Sketch** tab of the **Application Options** dialog box, choose the **Settings** button; the **Heads-Up Display Settings** dialog box is displayed. Clear the **Create dimensions when inputting dimension value** check box in the **Persistent Dimensions** area of the **Heads-Up Display Settings** dialog box and then choose **OK**. Next, choose the **Apply** and **Close** buttons from the **Application Options** dialog box to turn off the visibility of the dimension during sketching.

Saving the Sketch

Next, you need to save the sketch. As mentioned earlier, you cannot save the sketch in the sketching environment. First, you need to exit the sketching environment and then save it.

1. Choose the **Finish Sketch** button from the **Exit** panel; the sketching environment is closed and you will switch to the **Home** view of the part modeling environment.
2. Choose the **Save** tool and save the sketch with the name Tutorial2 at the location given below:

C:\Inventor_2011\c02

3. Choose **Close > Close** from the **Application Menu** to close this file.

Tutorial 3

In this tutorial, you will draw the sketch for the model shown in Figure 2-52. The sketch of the model is shown in Figure 2-53. Do not dimension the sketch as the dimensions are given only for reference.

(Expected time: 30 min)

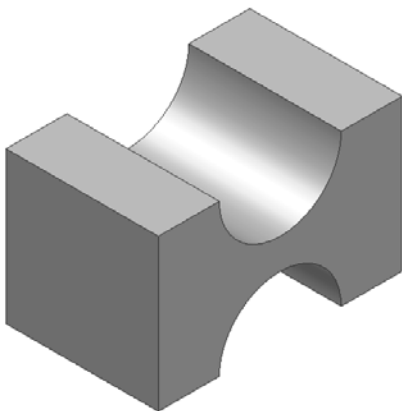


Figure 2-52 Model for Tutorial 3

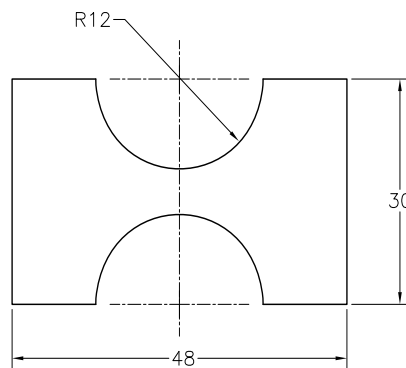


Figure 2-53 Sketch for Tutorial 3

The following steps are required to complete this tutorial:

- a. Start a new metric standard part file.
- b. Draw the sketch by using the **Arc** and **Line** tools, refer to Figure 2-55.
- c. Save the sketch with the name *Tutorial3* and close the file.

Starting a New File

1. Choose the **New** tool from the **Launch** panel of the **Get Started** tab to invoke the **New File** dialog box.
2. Start a new metric standard part file by double-clicking on **Standard (mm).ipt** in the **Metric** tab.

Drawing the Sketch

The upper arc of the sketch can be drawn by specifying the center point, start point, and endpoint of the arc. Therefore, you need to use the **Center Point Arc** tool to draw this arc.

1. Choose the **Arc Three Point** tool from **Sketch > Draw > Arc** drop-down; you are prompted to specify the center of the arc.
2. Enter **0, 15** in the **Inventor Precise Input** toolbar and press ENTER; you are prompted to specify the start point of the arc.
3. Enter **-12,15** in the **Inventor Precise Input** toolbar and press ENTER.



Next, you need to define the endpoint of the arc. The arc drawn after you specify the endpoint, can be in the clockwise or counterclockwise direction.

4. Move the mouse from the start point of the arc to a small distance in the counterclockwise direction. Now, enter **12,15** as the endpoint in the **Inventor Precise Input** toolbar and press ENTER; the upper arc is drawn.

Next, you need to draw lines in the sketch.

5. Choose the **Line** tool from the **Draw** panel; you are prompted to specify the start point of the line.



It is evident from Figure 2-53 that the lines start from the endpoints of the arc. You can specify the coordinates of the start point of the arc in the **Inventor Precise Input** toolbar or select the start point of the line in the drawing window.

6. Move the cursor close to the start point of the arc; the yellow circle snaps to the endpoint of the arc and turns green. When the yellow circle turns green, it indicates that the cursor has snapped to the endpoint of the arc. Press the left mouse button to select this point as the start point of the line.

As it is easier to define the points by using relative coordinates, it is recommended that you use the **Precise Delta** button in the **Inventor Precise Input** toolbar to draw lines. Choose the **Precise Delta** button, if it is not already chosen.

7. If the **Precise Delta** button is not chosen automatically, you need to specify the start point of the line and then choose this button. Next, press the ESC key to exit the **Line** tool. Now, invoke the **Line** tool again; a triad is placed at the origin (0,0). Enter **-12, 15** as the start point of the line in the **Inventor Precise Input** toolbar. Next, enter **-12, 0** as the endpoint of the line in the **Inventor Precise Input** toolbar.
8. Enter **0, -30** as the second point and **12, 0** as the third point in the **Inventor Precise Input** toolbar. As mentioned earlier, Autodesk Inventor provides you with the option to draw tangent or normal arcs while drawing lines. This is done by dragging the cursor from the point where you want to start the arc. You can directly draw it within the **Line** tool.
9. Move the cursor close to the endpoint of the last line until the yellow circle snaps to that point. When the yellow circle snaps to the endpoint, it turns gray. However, you will not be able to view the gray circle because of the triad. Now, press and hold the left mouse button and drag the mouse through a small distance in the upward direction.

You will notice that four imaginary lines are displayed, showing the four directions in which you can draw the arc.

10. As you need to draw the arc normal to the line, drag the cursor vertically upward in the direction of the vertical imaginary line to a small distance and then drag the cursor toward the right. While drawing the arc by dragging the cursor, you cannot use the **Inventor Precise Input** toolbar.

To specify the endpoint of the arc precisely, you can use the temporary tracking option. The temporary tracking option allows you to select a point by using two different points. For example, in this case, the right endpoint of the lower arc has to be vertically in the same line as that of the right endpoint of the upper arc, and horizontally in the same line as that of the start point of the lower arc. Now, assume a vertical imaginary line drawn from the endpoint of the upper arc and a horizontal imaginary line drawn from the start point of the lower arc. Both these imaginary lines intersect at a point that is essentially the endpoint of the lower arc. The temporary tracking option is used to draw these imaginary lines and to remove them after the point has been selected.

11. With the left mouse button pressed to define the endpoint of the arc, drag the cursor close to the right endpoint of the upper arc; the cursor snaps to the endpoint of the arc and turns green. Now, move the cursor vertically downward.

You will notice that a vertical imaginary line appears at the right endpoint of the upper arc. You do not need to snap to the horizontal point, because this point was automatically selected when you started drawing the lower arc. As you move the cursor downward, you

will notice a point where both the vertical and horizontal imaginary lines intersect each other; see Figure 2-54. This point is the endpoint of the lower arc. The cursor automatically snaps to the point where both the imaginary lines intersect. Do not release the left mouse button until the entire process is completed.

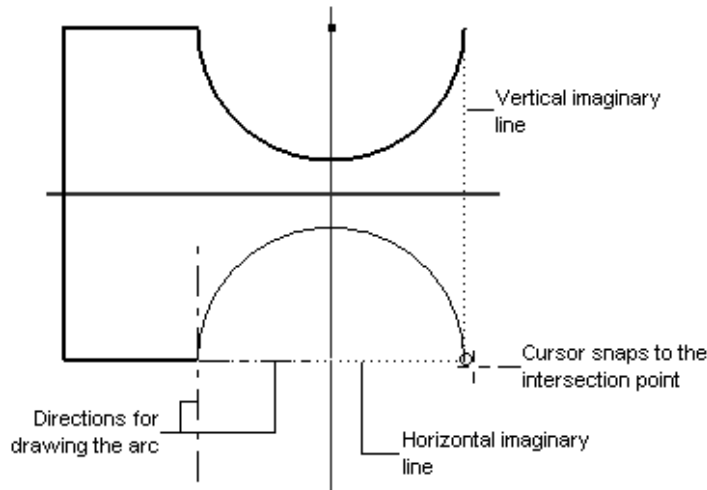


Figure 2-54 Use of the temporary tracking option to draw an arc



Note

In Figure 2-54, the major and minor grid lines and triad have not been displayed for a better display of the sketch and the imaginary lines.

12. When the cursor snaps to the intersection of the imaginary lines, release the mouse button to complete the lower arc.
13. Enter **12,0** as the coordinates of the next point in the **Inventor Precise Input** toolbar.
14. For the next point, you can enter coordinates in the **Inventor Precise Input** toolbar or use the temporary tracking option. To use this option, move the cursor close to the endpoint of the upper arc. Once the cursor snaps to this point and turns green, move it horizontally toward the right. As a result, a horizontal imaginary line is drawn. Using the left mouse button, select the point at which the vertical line becomes perpendicular to the horizontal imaginary line. This point is the endpoint of the right vertical line.



Note

While using the temporary tracking option to draw lines, you do not need to press the left mouse button and drag it. You need to press the left mouse button once to select the endpoint of the line after you get the intersection point of the imaginary lines.

15. Complete the sketch by snapping to the endpoint of the upper arc as the endpoint of the line. Next, right-click to display the shortcut menu, and then choose **Done** from it to exit the **Line** tool.
16. The final sketch for Tutorial 3 is shown in Figure 2-55.

Saving the Sketch

1. Choose the **Finish Sketch** button from the **Exit** panel; the sketching environment is closed and you will switch to the **Home** view of the part modeling environment.
2. Choose the **Save** button and save the sketch with the name *Tutorial3* at the location given below:

C:\Inventor_2011\c02

3. Choose **Close > Close** from the **Application Menu** to close this file.

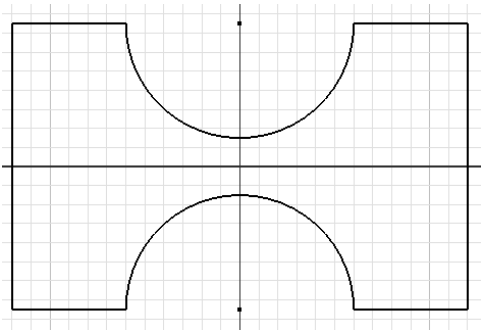


Figure 2-55 Final sketch for Tutorial 3

Tutorial 4

In this tutorial, you will draw the basic contour of the revolved solid model shown in Figure 2-56. The contour that you will draw for creating this revolved solid is shown in Figure 2-57. Do not dimension the sketch as the dimensions are given only for reference.

(Expected time: 30 min)

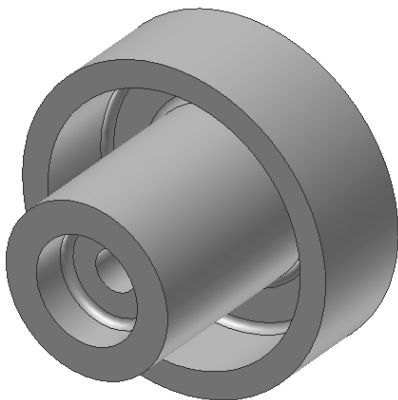


Figure 2-56 Revolved model for Tutorial 4

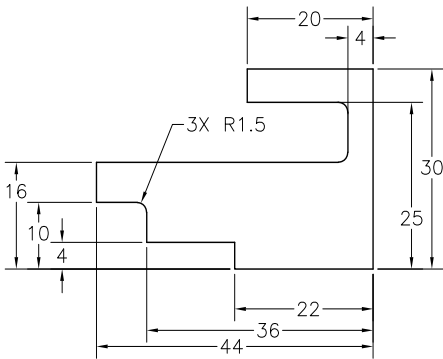


Figure 2-57 Sketch for the revolved model

The following steps are required to complete this tutorial:

- a. Start a new metric standard part file.
- b. Invoke the **Line** tool and draw the sketch by specifying the coordinates of points in the **Inventor Precise Input** toolbar, refer to Figure 2-57.
- c. Save the sketch with the name *Tutorial4* and close the file.

Starting a New File

- 1. Choose the **New** tool from the **Quick Access Toolbar** to display the **New File** dialog box.
- 2. Choose the **Metric** tab to display the standard metric templates. Double-click on **Standard (mm).ipt** to start a new metric part file.

Drawing the Sketch

- 1. Choose the **Line** tool from the **Draw** panel; you are prompted to specify the start point of the line. Enter **22,0** as the coordinates of the start point in the **Inventor Precise Input** toolbar and press ENTER; you are prompted to specify the endpoint of the line. You can specify the coordinates of the next point relative to the previous point, as it becomes easier to define points.



If the same session of Autodesk Inventor is used, the **Precise Delta** button is chosen automatically in the **Inventor Precise Input** toolbar. However, if you use a new session, you need to choose this button to invoke this toolbar.

- 2. Choose the **Precise Delta** button, if it is not chosen automatically and then press the ESC key. Invoke the **Line** tool and enter **22, 0** as the start point of the line in the **Inventor Precise Input** toolbar. Next, enter the following coordinates of the remaining points in the **Inventor Precise Input** toolbar.



Point	Coordinates (X, Y)
2	0,30
3	-20,0
4	0,-5
5	16,0
6	0,-9
7	-40,0
8	0,-6
9	8,0
10	0,-6
11	14,0
12	0,-4
13	22,0

- Right-click to display the shortcut menu, and then choose **Done** from it to complete the sketch. The sketch should look similar to the one shown in Figure 2-58. For your reference, the lines in the sketch are numbered.

You will create the arcs at the end of lines 4 and 5, 5 and 6, and 8 and 9 by using the **Fillet** tool. This tool draws the arcs at the point of intersection of the lines and removes sharp corners.

- Choose the **Fillet** tool from **Sketch > Draw > Fillet/Chamfer** drop-down; the **2D Fillet** dialog box is displayed with some default fillet radius. Enter **1.5** in the **Radius** edit box of this dialog box. Do not press ENTER.
- Select line 4 and then line 5, refer to Figure 2-58; a fillet is created between these lines and the radius of the fillet is displayed in the sketch.
- Similarly, select lines 5 and 6 and then lines 8 and 9 to create a fillet between these lines. Right-click, and then choose **Done** from the shortcut menu to exit the **Fillet** tool after creating all fillets.

As all lines are filleted with the same radius value, the radius of the fillet is not displayed on other fillets. This completes the sketch. The final sketch for this tutorial after filleting is shown in Figure 2-59.

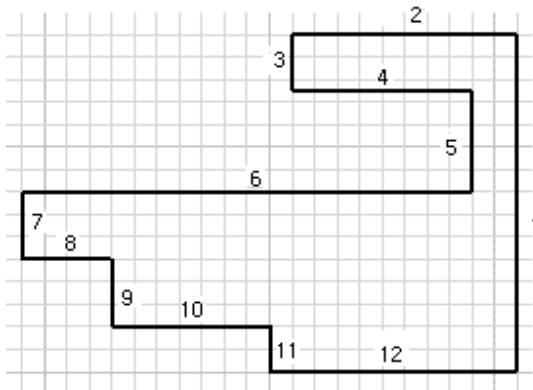


Figure 2-58 Sketch after drawing the lines

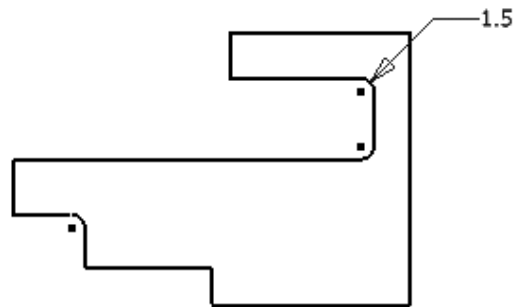


Figure 2-59 Final sketch after filleting



Note

In Figures 2-58 and 2-59, the display of axes has been turned off for a better visibility of the lines of the sketch.

Saving the Sketch

1. Choose the **Finish Sketch** button from the **Exit** panel.
2. Choose the **Save** button and save this sketch with the name *Tutorial4* at the location given below:

C:\Inventor_2011\c02

3. Choose **Close > Close** from the **Application Menu** to close this file.

Self-Evaluation Test

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

1. Most designs created in Autodesk Inventor are a combination of sketched features and placed features. (T/F)
2. Whenever you start a new file in the **Part** module, the sketching environment is invoked by default. (T/F)
3. You cannot turn off the display of grid lines. (T/F)
4. You cannot draw an arc within the **Line** tool. (T/F)
5. In Autodesk Inventor, the two types of sketching entities that can be drawn are _____ and _____.
6. In the sketching environment, the _____ tool is used to place a sketch point or a center point.
7. Filleting is defined as the process of _____ the sharp corners and sharp edges of models.
8. You can also delete sketched entities by pressing the _____ key.
9. In Autodesk Inventor, rectangles are drawn as the combination of _____ entities.
10. You can undo the last drawn spline segment when you are still inside the spline drawing option by choosing _____ from the shortcut menu displayed.

Review Questions

Answer the following questions:

1. In most designs, generally the first feature or the base feature is the placed feature. (T/F)
2. You can invoke the options related to sheet metal parts from the **.ipt** file. (T/F)
3. You can change the current project directory and the project files by choosing **Projects** from the **Open** dialog box. (T/F)
4. You can specify the position of entities dynamically by using the Dynamic Input. (T/F)
5. In Autodesk Inventor, you can save a file in the sketching environment. (T/F)
6. In Autodesk Inventor, you can start a new file by using the **Open** dialog box. (T/F)
7. Which of the following tools in the **Tools** tab is used to invoke additional toolbars?
 - (a) **Application Options**
 - (b) **Customize**
 - (c) **Document Setting**
 - (d) None of these
8. Which of the following drawing display options is used to interactively zoom in and out a drawing?
 - (a) **Zoom All**
 - (b) **Pan**
 - (c) **Zoom**
 - (d) **Zoom Window**
9. Which of the following keys is used to restore the previous view?
 - (a) F5
 - (b) F6
 - (c) F7
 - (d) F4
10. Which of the following drawing display options prompts you to select an entity whose magnification has to be increased?
 - (a) **Zoom**
 - (b) **Pan**
 - (c) **Zoom Selected**
 - (d) None of these

Exercises

Exercise 1

Draw the basic sketch of the model shown in Figure 2-60. The sketch to be drawn is shown in Figure 2-61. Do not dimension it, as the dimensions are given only for reference.

(Expected time: 30 min)

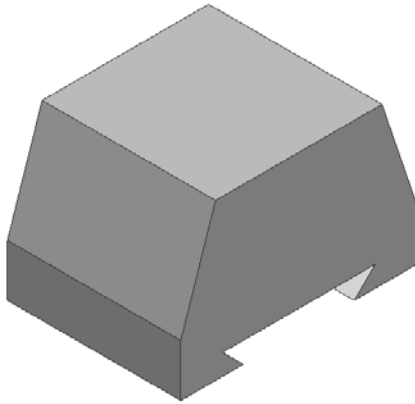


Figure 2-60 Model for Exercise 1

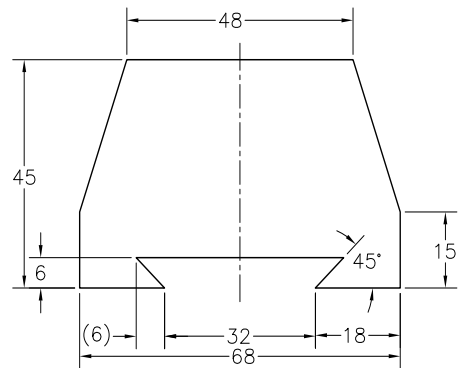


Figure 2-61 Sketch for Exercise 1

Exercise 2

Draw the basic sketch of the model shown in Figure 2-62. The sketch to be drawn is shown in Figure 2-63. Do not dimension it, as the dimensions are given only for reference.

(Expected time: 45 min)

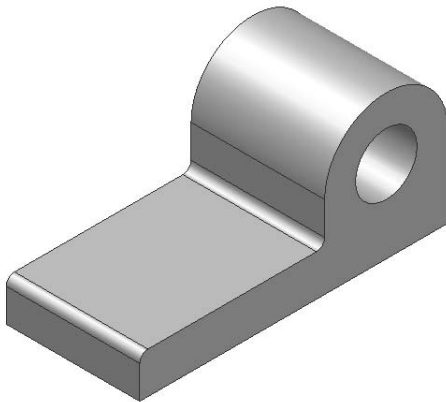


Figure 2-62 Model for Exercise 2

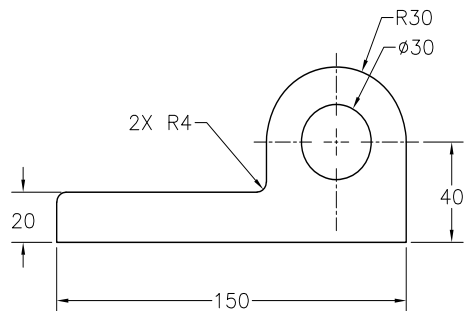


Figure 2-63 Sketch for Exercise 2

Exercise 3

Draw the sketch of the model shown in Figure 2-64. The sketch to be drawn is shown in Figure 2-65. Do not dimension it, as the dimensions are given only for reference.

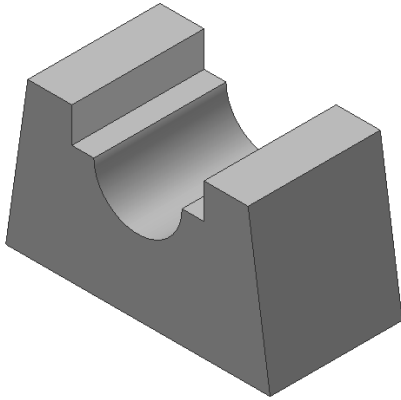


Figure 2-64 Model for Exercise 3

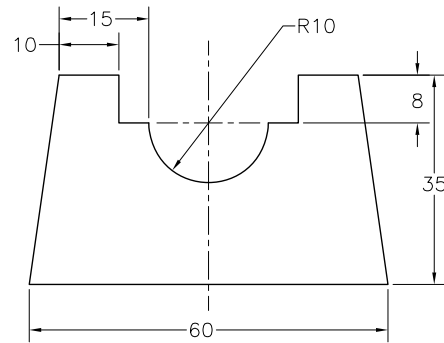


Figure 2-65 Sketch for Exercise 3

(Expected time: 45 min)

Answers to Self-Evaluation Test

1. T, 2. T, 3. F, 4. F, 5. normal, construction, 6. Point, 7. rounding, 8. DELETE, 9. individual, 10. Back