



# Chapter 2

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## ***Drawing Sketches for Solid Models***

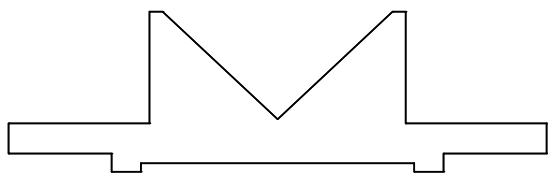
### **Learning Objectives**

**After completing this chapter, you will be able to:**

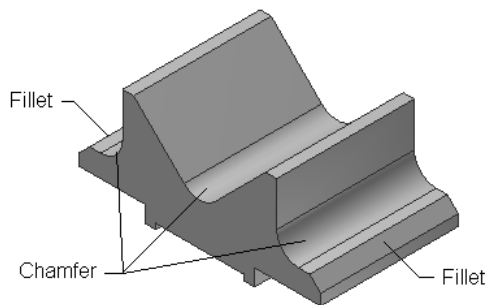
- *Start a new template file for drawing sketches.*
- *Set up the sketching environment.*
- *Understand various drawing display tools.*
- *Understand the sketcher environment in the Part module.*
- *Get acquainted with sketcher entities.*
- *Draw sketches using various sketcher entities.*
- *Delete sketched entities.*

## THE SKETCHING ENVIRONMENT

Most designs created in Autodesk Inventor consist of sketched and placed features. A sketch is a combination of 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 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

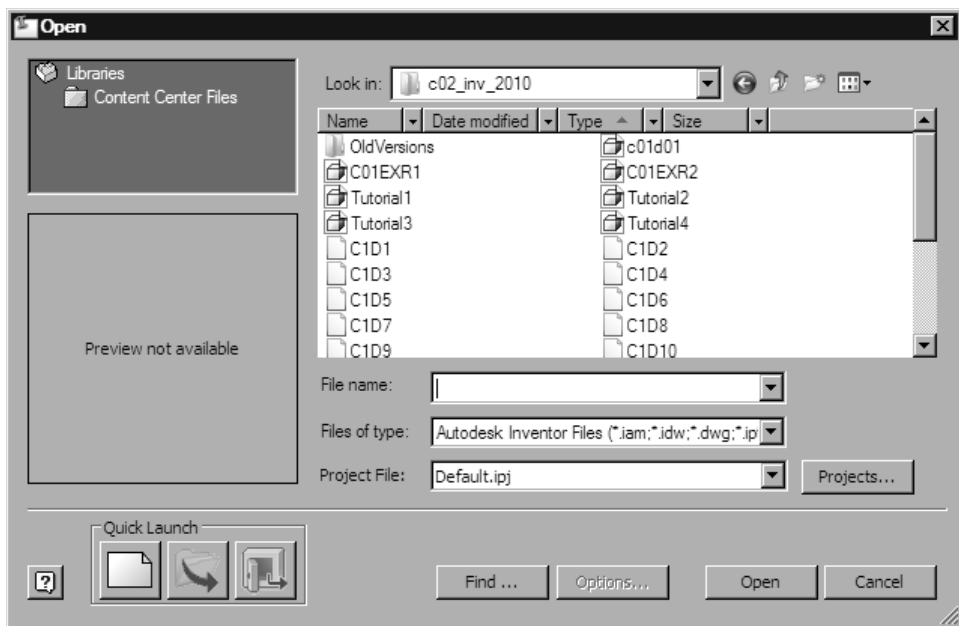
Once you have drawn the basic sketch, refer to Figure 2-1, you need to convert it into a solid model using solid modeling tools.

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 active. 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.

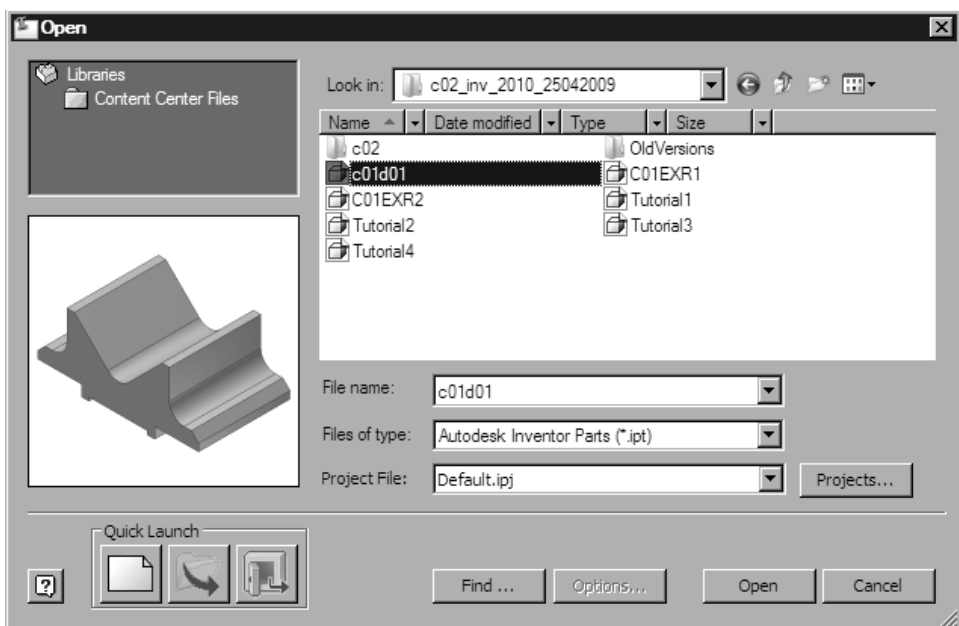
### The Open Dialog Box

When you start a new session of Autodesk Inventor, the initial screen will be displayed. Next, choose the **Open** button from the **Launch** panel of the **Get Started** tab in the **Ribbon**; the **Open** dialog box will be displayed, as shown in Figure 2-3.

The options given in the **Open** dialog box are used to create new files and open the existing files. You can browse and select the file to be opened from the list displayed in the dialog box. The preview of the selected file will be 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, AliasStudio, 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 from 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 information regarding the 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.

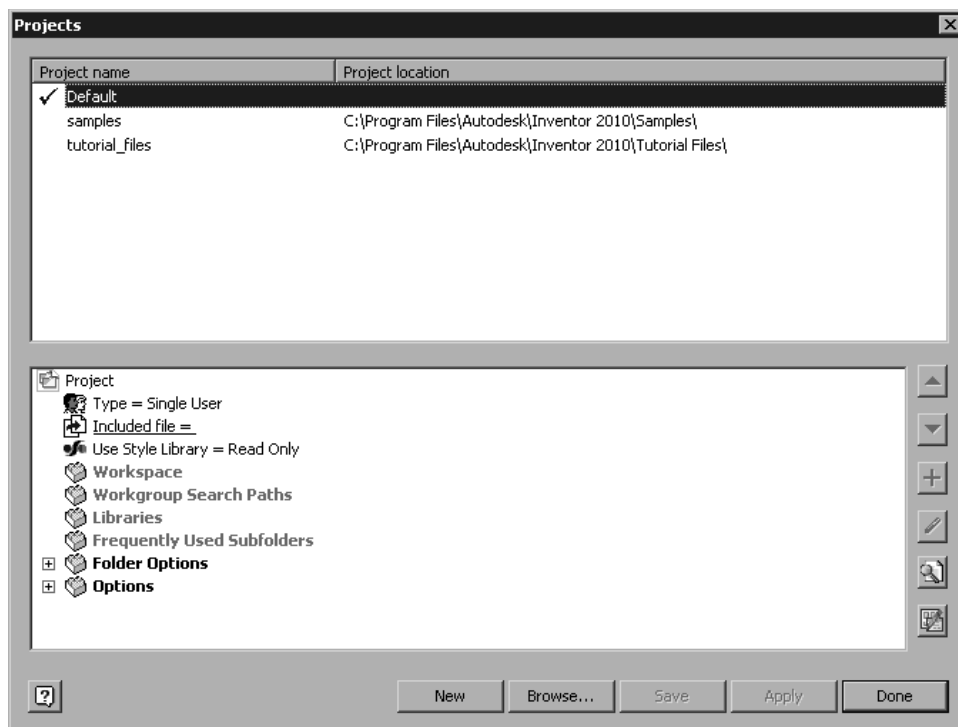


Figure 2-5 The **Projects** dialog box

The **Open** dialog box also allows you to start new files in Autodesk Inventor. When you start 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 from the **Quick Launch** area; 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. By default, the **Returning / New Inventor users** radio button is selected in this window. However, you can select the **Show Help on startup** check box in 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 the **New** button from the **Get Started** tab of the **Ribbon** or 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** button from the **Quick Access Toolbar**.



### Note

When you invoke the **New File** dialog box by choosing the **New** button 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** button 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**, or **Metric** tabs. If you have installed Autodesk Inventor by selecting millimeter as the unit for measurement, the metric standard will be used on starting the standard template in the **Default** tab. However, if you have installed Autodesk Inventor by selecting inch as the unit for measurement, you need to select templates from the **Metric** tab, as shown in Figure 2-7. The templates that are available on choosing the **Metric** tab are discussed next.

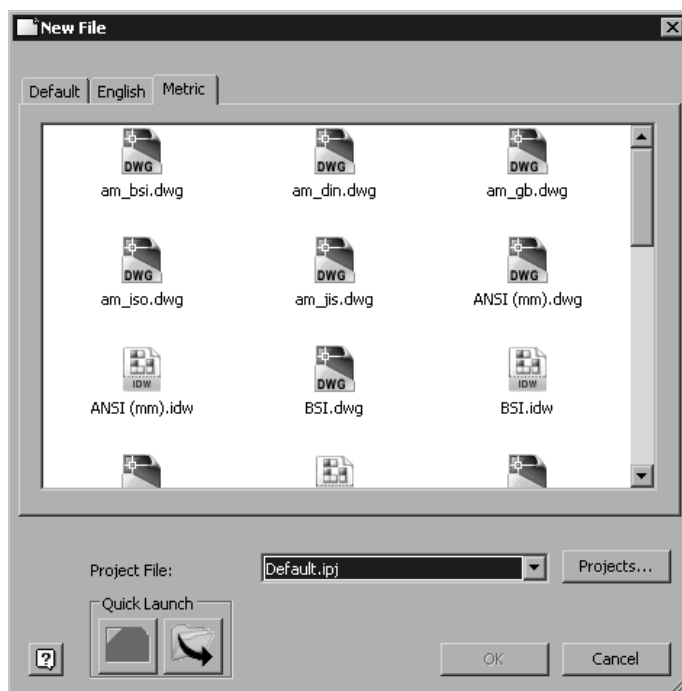


Figure 2-7 The *Metric* tab of the *New File* dialog box

### **.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.

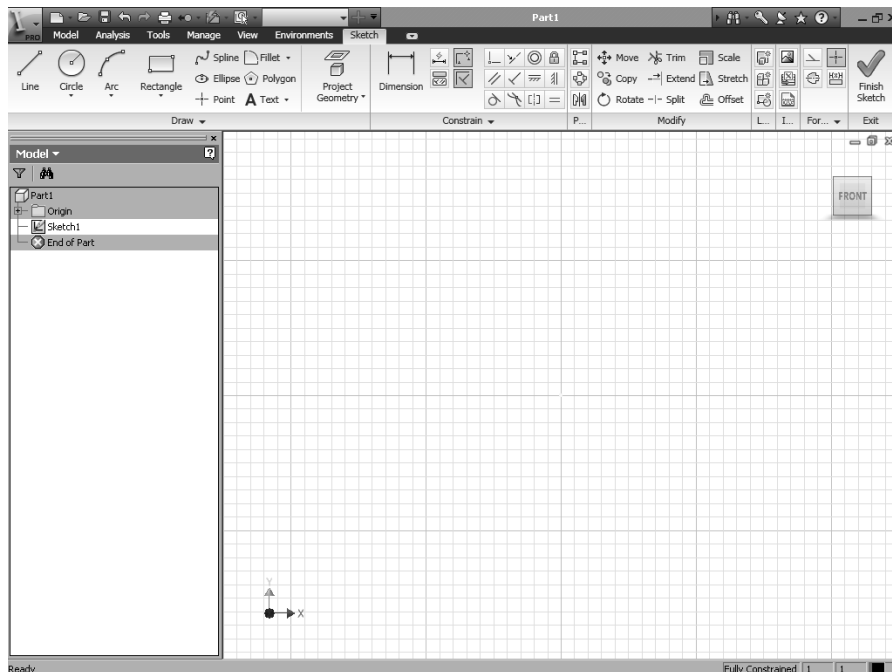


Figure 2-8 Initial screen appearance in the sketching environment

## SETTING UP THE SKETCHING ENVIRONMENT

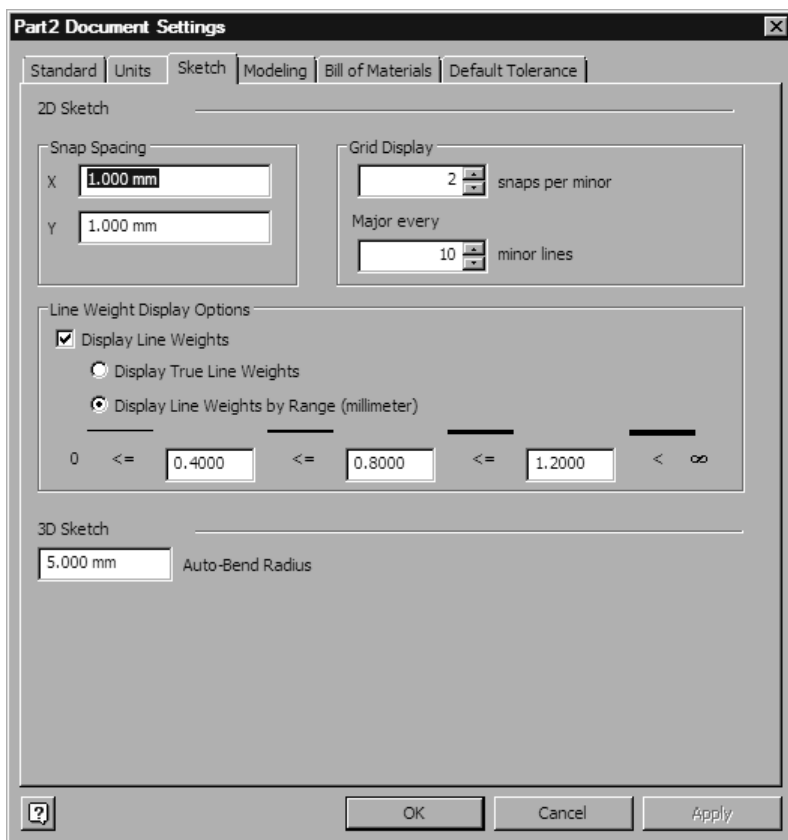
It is very important to first set up the sketcher environment. This has to be done before you start drawing a sketch. Setting up the sketcher environment includes modifying the grids of a drawing. It is unlikely that the designs you want to create consist of small dimensions. You

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 the 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 the precise location of 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** button from the **Options** panel of the **Tools** tab in the **Ribbon**; 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 provided 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, choose the **Application Options** button from the **Options** panel of the **Tools** tab in the **Ribbon**; 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 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 in the **Ribbon**. Some of the drawing display tools in Autodesk Inventor are discussed next. The rest of these tools will be discussed in later chapters.

## Zoom All

**Ribbon:** View > Navigate > Zoom All  
**Navigation Bar:** Zoom All



The **Zoom All** tool increases the drawing display area to include all the sketched entities in the current display.

## Zoom Window

**Ribbon:** View > Navigate > Zoom All > Zoom Window  
**Navigation Bar:** Zoom All > 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 All > Zoom  
**Navigation Bar:** Zoom All > Zoom



The **Zoom** tool is used to interactively zoom in and out of the drawing view. When you choose this button, the default cursor is replaced by the arrow 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 All > Zoom Selected  
**Navigation Bar:** Zoom All > Zoom Selected



When you choose the **Zoom Selected** button, you will be prompted to select an entity to zoom. The selected entity 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



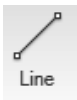
The **Pan** tool is used to drag the current view in the drawing window. This option is generally used to display the contents of the drawing that are outside the display area, without actually changing the magnification of the current drawing. It is similar to holding the drawing and dragging it across 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** button from the **Format** panel of the **Sketch** tab in the **Ribbon**. All entities drawn after choosing the **Construction** button will be the construction entities. Deselect this button by choosing it again to draw normal entities. The sketcher entities in Autodesk Inventor are discussed next.

## Drawing Lines

**Ribbon:** Sketch > Draw > Line  
**Toolbar:** 2D Sketch Panel > 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, you can drive the line to a new length or angle using parametric dimensions. You can also directly create the line of actual length and angle using the **Inventor Precise Input** toolbar. Both these methods of drawing the lines are discussed next.

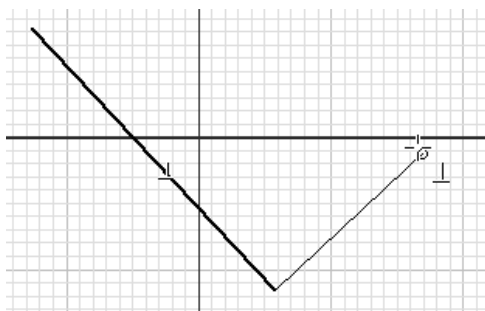
### Drawing Lines by Picking Points in the Drawing Window

This method is very convenient for drawing lines and is extensively used while sketching. When you invoke the **Line** tool from the **Draw** panel of the **Sketch** tab in the **Ribbon**, the cursor (that was initially an arrow) is replaced by crosshairs with a yellow circle at the intersection. Also, at the lower left corner of the Autodesk Inventor window, you are prompted to select the start point of the line or drag off the endpoint for the tangent arc. The point of intersection of the X and Y axes (black lines among the grid lines) is the origin point. If you move the cursor close to the origin, it snaps to the origin automatically. The coordinates of the cursor location in the drawing window are displayed on the lower right corner of the Autodesk Inventor window. To start drawing the line, specify a point anywhere in the drawing window; a rubber-band line

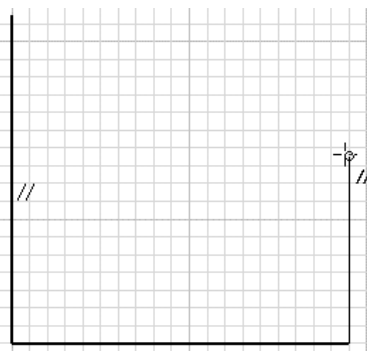
starts from that point. One end of this rubber-band line is fixed at the point you specified in the drawing window and the second end is attached to the yellow circle in the crosshairs.

Note that after specifying the start point of the line, the **Status Bar** of the Autodesk Inventor window displays three list boxes. The first list box displays the coordinates of the current location of the cursor; the second list box displays the current length of the line, and the third list box displays the current angle of the line. Taking the reference from these list boxes, you can move the cursor and specify the endpoint of the line. On doing so, 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 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, the valid constraints are automatically applied to the entities. Therefore, when you draw continuous lines, the horizontal, vertical, perpendicular, and parallel constraints are automatically applied to them. The symbol of the applied constraint is displayed on the line while drawing. You can exit the **Line** tool by pressing the ESC key. You can also right-click in the drawing window and choose **Done** from the shortcut menu. Figures 2-10 and 2-11 display the **Perpendicular Constraint** and **Parallel Constraint** being applied to the lines when they are being drawn.



**Figure 2-10** Drawing a line with the **Perpendicular Constraint**



**Figure 2-11** Drawing a line with 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** button from the **Options** panel of the **Tools** tab in the **Ribbon**; the **Application Options** dialog box will be displayed. Choose the **Colors** tab and then select the **Presentation** option from the **Color scheme** list box. 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 the entities, some constraints are applied automatically to the sketch. To turn this option off, press and hold the CTRL key and draw the entities.

## 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 on the right of the **Draw** panel in the **Sketch** tab; the **Draw** panel will expand. Choose the **Precise Input** button 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. Selecting the point at this stage closes the loop and you exit the current line chain.*

*To draw centerlines, first choose the **Centerline** button from the **Format** panel of the **Sketch** tab in the **Ribbon** and then create the line. Alternatively, select the required entities from the drawing window and then choose the **Centerline** button; 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 > Center Point Circle  
**Toolbar:** 2D Sketch Panel > Circle



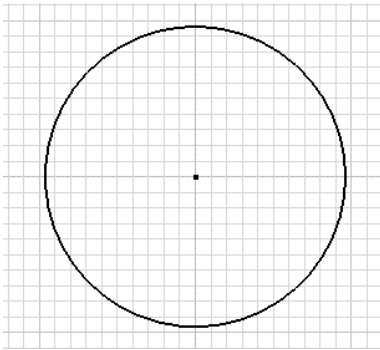
This is the default method of drawing circles. In this method, you need to define the center point and the radius of the circle. To draw this type of circle, choose the **Center Point Circle** button from the **Draw** panel of the **Sketch** tab; you will be prompted to select the center of the circle. 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. You can also specify the center and the radius using the **Inventor Precise Input** toolbar. Figure 2-12 shows a circle drawn by using the center and the radius.

### Drawing Circles Using Three Tangent Lines

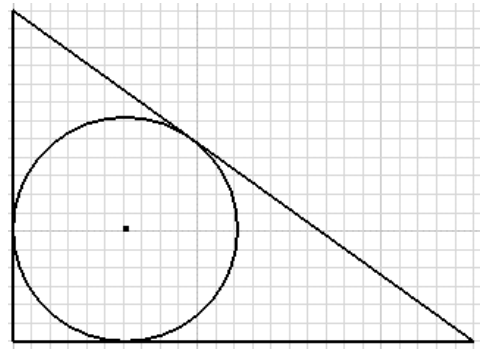
**Ribbon:** Sketch > Draw > Center Point Circle > Tangent Circle  
**Toolbar:** 2D Sketch Panel > Center Point Circle > Tangent Circle



This is the second method of drawing circles. This method draws a circle that is tangent to three selected lines. To invoke this option, choose the down arrow located to the bottom of the **Center Point Circle** button in the **Draw** panel of the **Sketch** tab and then choose the **Tangent Circle** button; you will be prompted to select the first, second, and third lines. As soon as you specify the three lines, a circle tangent to all the three specified lines will be drawn, as shown in Figure 2-13.



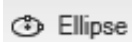
**Figure 2-12** Circle drawn using the center point and the radius of the circle



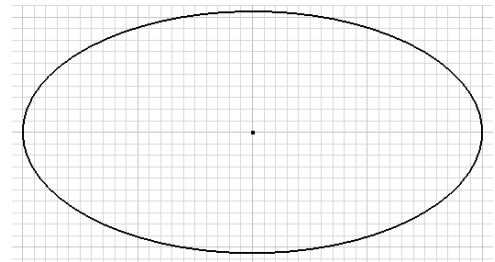
**Figure 2-13** Circle drawn using three tangent lines

## Drawing Ellipses

**Ribbon:** Sketch > Draw > Ellipse  
**Toolbar:** 2D Sketch Panel > Center Point Circle > Ellipse



To draw an ellipse, choose the **Ellipse** button from the **Draw** panel of the **Sketch** tab; 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 2-14 shows an ellipse.



**Figure 2-14** An ellipse drawn in the sketching environment

## Drawing Arcs

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

### Drawing an Arc Using Three Points

**Ribbon:** Sketch > Draw > Three Point Arc  
**Toolbar:** 2D Sketch Panel > Three Point Arc



This is the default method of drawing arcs. This method draws an arc using 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. Figure 2-15 shows an arc drawn using this method.

## Drawing an Arc Tangent to an Existing Entity

**Ribbon:** Sketch > Draw > Three Point Arc > Tangent Arc  
**Toolbar:** 2D Sketch Panel > Three Point Arc > Tangent Arc



This method draws an arc that is tangent to an existing open entity. The open entity can be an arc or a line. To invoke this method, choose the down arrow located on the bottom of the **Three Point Arc** button and then choose the **Tangent Arc** button. On doing so, you will be prompted to select the start point of the arc. The start point of the arc has to be the start point or the endpoint of an existing open entity. Once you specify the start point, a rubber-band arc starts from it. Note that this arc is tangent to the selected entity. Next, you will be prompted to specify the endpoint of the arc. It is very important to mention here that you cannot use the **Inventor Precise Input** toolbar to select the start point of this arc. However, you can use this toolbar to specify the endpoint of this arc. Figure 2-16 shows an arc drawn tangent to the line.

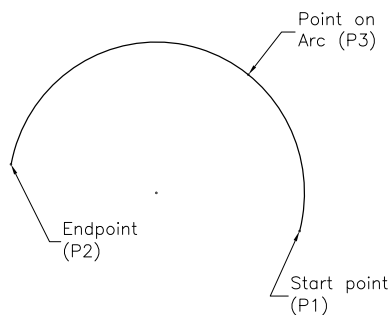


Figure 2-15 Drawing the three point arc

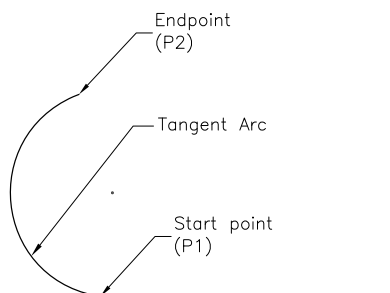


Figure 2-16 Drawing the tangent arc

## 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. If you draw the arc in continuation with the lines, you do not need to perform this step. Next, move the cursor back to the point that you selected; the yellow 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 Using the Center, Start, and Endpoint of the Arc

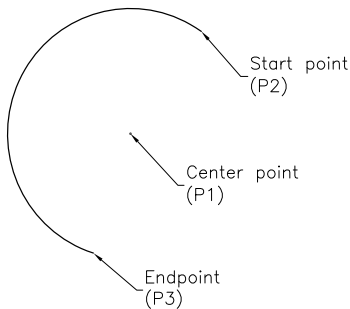
**Ribbon:** Sketch > Draw > Three Point Arc > Center Point Arc  
**Toolbar:** 2D Sketch Panel > Three Point Arc > Center Point Arc



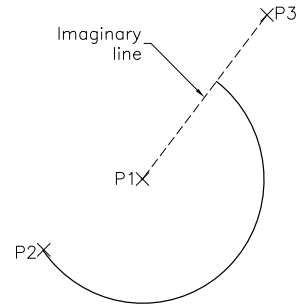
To invoke this method, choose the down arrow located on the bottom of the **Three Point Arc** button in the **Draw** panel and then choose the **Center Point Arc** button. This method allows you to draw an arc by specifying the center point, start point, and endpoint of the arc. On choosing this button, you will be prompted to specify



the center point of the arc. Once you specify the center, you will be prompted to specify the start point and then the endpoint of the arc, see Figure 2-17. You can also use the **Inventor Precise Input** toolbar to specify these three points of the arc. As you define the center point and the start point, the radius of the arc will be automatically defined. Therefore, the third point is just used to define the arc length. An imaginary line is drawn from cursor point 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 2-18.



*Figure 2-17 The center point arc*



*Figure 2-18 The imaginary line created while drawing the center point arc*

## Drawing Rectangles

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

### Drawing Rectangles Using Two opposite Corners

**Ribbon:** Sketch > Draw > Two Point Rectangle  
**Toolbar:** 2D Sketch Panel > Two Point Rectangle



This is the default method of drawing rectangles. This method draws a rectangle by specifying its two opposite corners. On choosing the **Two Point Rectangle** button, you will be prompted to specify the first corner of the rectangle. Once you specify the first corner, you are prompted to specify the opposite corner of the rectangle. Figure 2-19 shows a rectangle drawn using the **Two Point Rectangle** method.

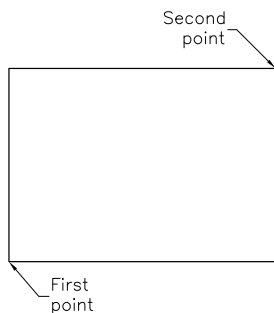
### Drawing Rectangles Using Three Points on a Rectangle

**Ribbon:** Sketch > Draw > Two Point Rectangle > Three Point Rectangle  
**Toolbar:** 2D Sketch Panel > Two Point Rectangle > Three Point Rectangle

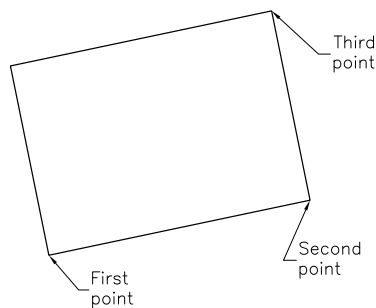


You can draw a rectangle by specifying three points of a rectangle. Therefore, to create a rectangle by using this method, click on the down arrow on the bottom of the **Two Point Rectangle** button in the **Draw** panel of the **Sketch** tab; a flyout will be displayed. Next, choose the **Three Point Rectangle** button from the flyout. 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. On invoking this tool, 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. Therefore, these points are used 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 three points for drawing the rectangle using the **Inventor Precise Input** toolbar. Figure 2-20 shows an inclined rectangle drawn using the **Three Point Rectangle** method.



**Figure 2-19** Drawing rectangle using two points



**Figure 2-20** 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 multisided 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 2-21, and you will be prompted to select the center of the polygon. The options in this dialog box are discussed next.



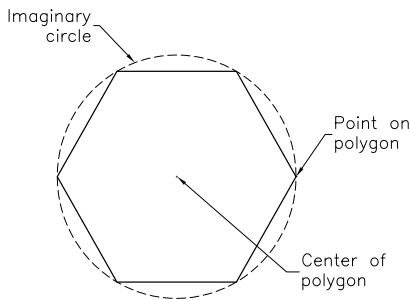
**Figure 2-21** The **Polygon** dialog box

### Inscribed

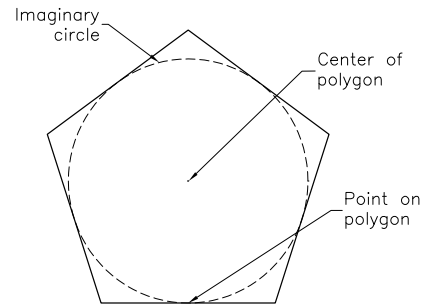
This is the first button in the **Polygon** dialog box and is chosen by default. This option is used to draw an inscribed polygon. An inscribed polygon is one that is drawn inside an imaginary circle such that its vertices touch the circle. Once you have specified the polygon center, you will be prompted 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 2-22.

### Circumscribed

This is the second button in the **Polygon** dialog box and is used to draw a circumscribed polygon. A circumscribed polygon is 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 2-23.



**Figure 2-22** Drawing a six-sided inscribed polygon



**Figure 2-23** 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. Therefore, the entire object is moved or stretched.*

## Placing Points

**Ribbon:** Sketch > Draw > Point  
**Toolbar:** 2D Sketch Panel > Point



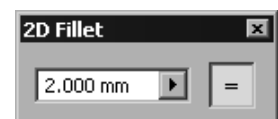
In Autodesk Inventor, you can place the sketched points or the hole centers in a sketch using the **Point** tool. To place a point, choose the **Point** button from the **Draw** panel of the **Sketch** tab in the **Ribbon**; you will be prompted to select the center point. Specifying the center point, a center 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  
**Toolbar:** 2D Sketch Panel > Fillet



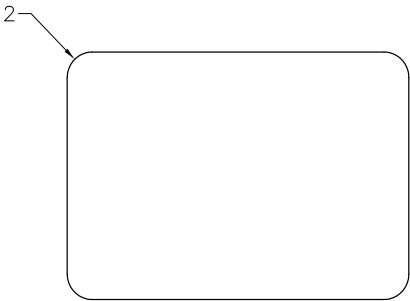
Filleting is defined as the process of rounding the sharp corners of a sketch. This is done to reduce the stress concentration in the model. Using the **Fillet** tool, you can round the corners of the sketch by creating an arc tangent to both the selected entities. The portions of the selected entities that comprise the sharp corners are trimmed when the fillet is created. When you invoke this tool, the **2D Fillet** dialog box will be displayed with the current fillet radius, as shown in Figure 2-24, and you will be prompted to select the lines or the arcs to be filleted. If you have already created some fillets, their



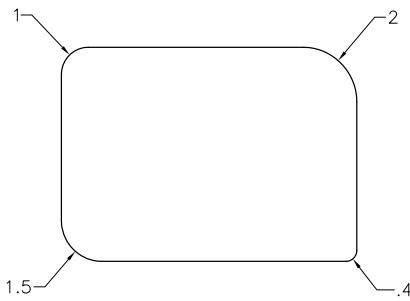
**Figure 2-24** The 2D Fillet dialog box

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 side of the edit box.

You can create as many fillets of similar or different radii using the same sequence of the **Fillet** tool. If the **Equal** button in the **2D Fillet** dialog box is chosen, the dimension of the fillet is placed only on the first fillet and not on the other fillets created using the same sequence of this tool. On modifying the dimension of the first fillet, all the fillet instances are modified. To show dimensions on all fillet instances, clear the **Equal** button before creating fillets. Displaying the dimensions on all the instances of fillets, makes them independent and you can modify their dimensions individually by double-clicking on them. You can fillet two parallel or perpendicular lines (Figures 2-25 and 2-26), intersecting lines or arcs, non-intersecting lines or arcs, and a line and an arc.



**Figure 2-25** Rectangle filleted using the same radius with the **Equal** button chosen



**Figure 2-26** Rectangle filleted using different radii with the **Equal** button cleared

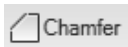
**Creating Chamfers**

**Ribbon:**

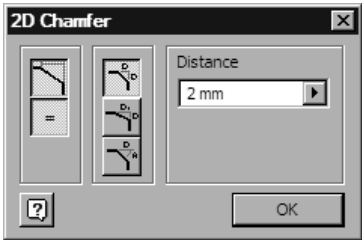
Sketch > Draw > Fillet > Chamfer

**Toolbar:**

2D Sketch Panel > Fillet > 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 the sketched entities, choose the down arrow located on the right of the **Fillet** tool in the **Draw** panel of the **Sketch** tab and then choose the **Chamfer** button; the **2D Chamfer** dialog box will be displayed, as shown in Figure 2-27, and you will be prompted to select the lines to be chamfered. The options in this dialog box are discussed next.



**Figure 2-27** The **2D Chamfer** dialog box

## 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.

## 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, the two dimensions of the same value will be shown in the sketch, as shown in Figure 2-28.

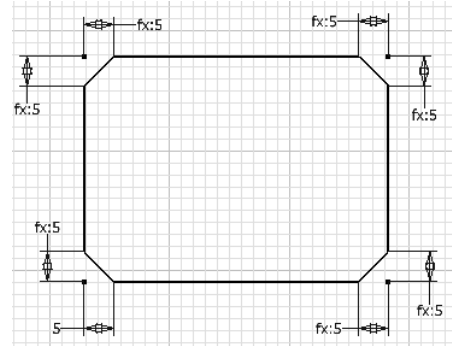


Figure 2-28 Chamfer with dimension values

## 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 of **Distance2** edit box is measured along the edge selected next. Figure 2-29 shows a chamfer created by using the Unequal Distances method.

## Distance and Angle



The Distance and Angle button is chosen to create a chamfer using a distance and an angle. The distance is 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 2-30.

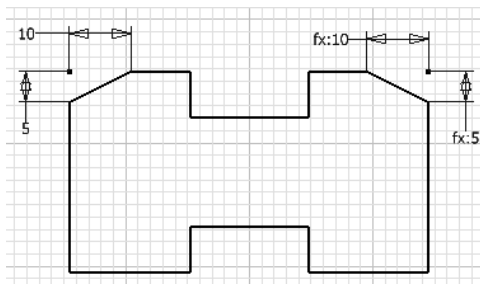


Figure 2-29 The unequal distances chamfer

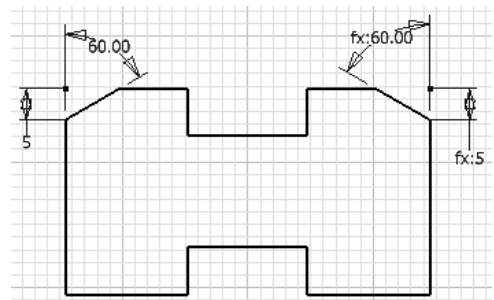


Figure 2-30 Distance and angle chamfer



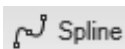
**Tip.** If multiple chamfers are created with same values, the dimension value is displayed only at the first instance. At the remaining chamfers, the dimension will be displayed as fx of the value, which means the function of the original value.

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.

## Drawing Splines

**Ribbon:** Sketch > Draw > Spline

**Toolbar:** 2D Sketch Panel > Line > Spline



To draw a spline, choose the **Spline** button from the **Draw** panel of the **Sketch** tab in the **Ribbon**; 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 2-31. You can drag these square and diamond points to modify the shape of the spline.

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, then hold the left mouse button and drag it. A construction line will be drawn that displays the possible tangent directions for the spline. Drag the mouse in the required direction to draw the tangent spline. Figure 2-31 shows a spline drawn by specifying different points and Figure 2-32 shows a spline drawn tangent to an existing line.

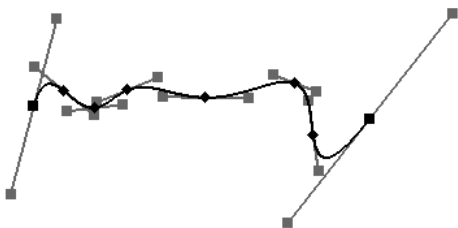


Figure 2-31 A spline drawn

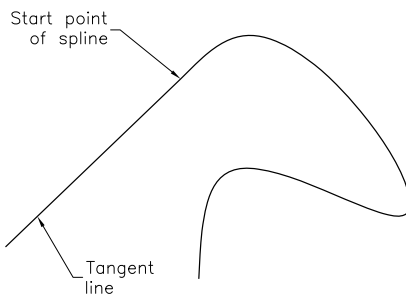


Figure 2-32 A spline drawn tangent to a line



**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.

## 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 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 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 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 it. If not, it will be added to it.

## TUTORIALS

Although Autodesk Inventor is parametric in nature, in this chapter you will use the **Inventor Precise Input** toolbar to draw objects. This is to make you comfortable with the various drawing options in Autodesk Inventor. From the next chapter onwards, you will use the parametric nature of Autodesk Inventor for sizing or drawing the entities as per the desired dimension values.

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

## Tutorial 1

In this tutorial, you will draw the sketch for the model shown in Figure 2-33. The sketch is shown in Figure 2-34. Do not dimension it, as the dimensions are given only for reference.

(Expected time: 30 min)

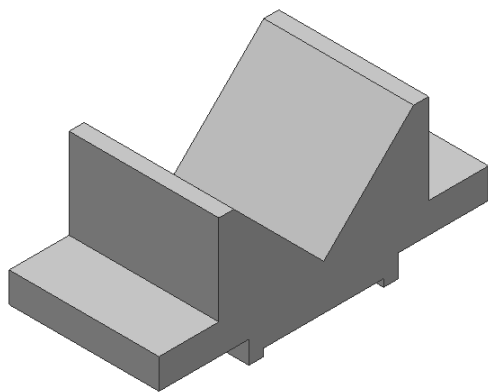


Figure 2-33 Model for Tutorial 1

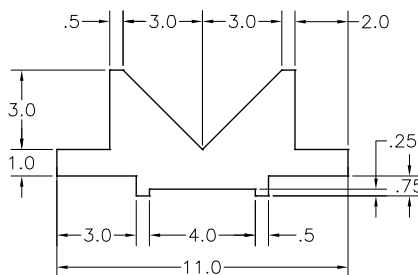



Figure 2-34 Sketch of the model

The following steps are required to complete this tutorial:

- Start a new metric standard part file and then invoke the **Sketching** environment.
- 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. You can also choose **Start > Programs > Autodesk > Autodesk Inventor 2010 > Autodesk Inventor Professional 2010** from the taskbar; a new session of Autodesk Inventor is started.
- Choose the **New** button from the **Launch** panel of the **Get Started** tab in the **Ribbon**; 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

If you have installed Autodesk Inventor by selecting millimeter as the measurement unit, you can also open a standard metric template by selecting **Standard.ipt** from the **Default** tab.



## Drawing the Sketch

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

1. Choose the **Line** button in the **Draw** panel of the **Sketch** tab in the **Ribbon**. Next, click the down arrow available on the right of the **Draw** panel in the **Sketch** tab and choose the **Precise Input** button to display the **Inventor Precise Input** toolbar. 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. It is enabled only when you invoke a sketching tool. Because all the initial settings are configured, you can start drawing the sketch.

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

2. Specify the start point of the sketch as **0** 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. Enter **-3** and **3** in the **X** and **Y** edit boxes, respectively, of the **Inventor Precise Input** toolbar and press ENTER to define the endpoint of the line. As a result, 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. The reason for the line being small is that the dimensions of the sketch are very small and the drawing display area is large. Therefore, you need to modify the drawing display area using the drawing display tools. To modify the drawing display area, you can use the **Zoom** tool.

4. Click on the down arrow available below the **Zoom All** button in the **Navigation Bar**; a flyout is displayed. Choose the **Zoom** option from the flyout displayed; the drawing cursor is changed to a normal 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 once you feel 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 line creation resumes and you are prompted to specify the endpoint of the next line.

7. The coordinates of the remaining points 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

8. After specifying all points, right-click to display the shortcut menu. In this menu, choose **Done** or press ESC to exit the **Line** tool.

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



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

9. The final sketch for Tutorial 1 is shown in Figure 2-35.

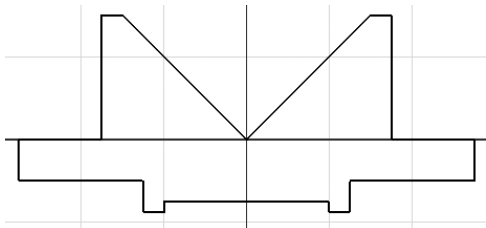


Figure 2-35 Final sketch for Tutorial 1



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

## Saving the Sketch

Remember that you cannot save a sketch in the sketching environment. This is because, in Autodesk Inventor, the sketching environment is just a part of the **Part** module. This environment is used only for drawing the sketches of the 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 of the **Sketch** tab in the **Ribbon**; the sketching environment is closed and the part modeling environment is invoked. Notice that the **Model** tab is activated in place of the **Sketch** tab. The **Model** tab provides options for creating features. The options under this tab will be discussed in later chapters.
2. Choose the **Save** button from the **Quick Access Toolbar**; the **Save As** dialog box is displayed.

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

3. Create a new folder with the name *Inventor\_2010* in the C drive of your computer. In this folder, create a folder with the name *c02*. Browse and select *C:\Inventor\_2010\c02*, as shown in Figure 2-36.

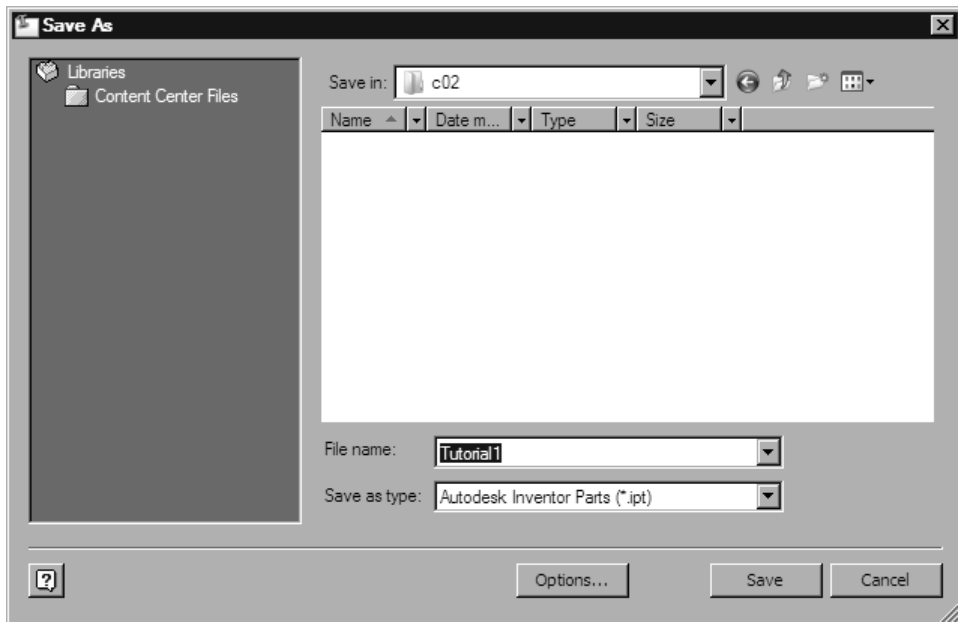
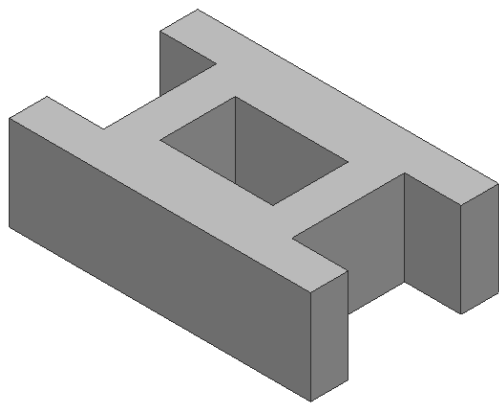


Figure 2-36 The *Save As* dialog box

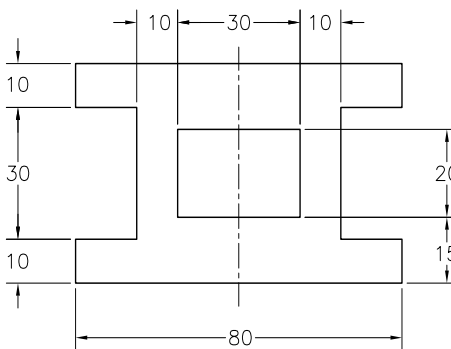
4. Enter the file name as *Tutorial1* in the **File name** edit box, refer to Figure 2-36 and then choose the **Save** button from the **Save As** dialog box to save the sketch.
5. Choose **Close > Close** from the **Application Menu** to close this file.

## Tutorial 2

In this tutorial, you will draw the sketch for the model shown in Figure 2-37. The sketch is shown in Figure 2-38. Do not dimension it. The solid model and dimensions are for reference only. **(Expected time: 30 min)**



**Figure 2-37** Model for Tutorial 2



**Figure 2-38** Dimensioned sketch for Tutorial 2

The following steps are required to complete this tutorial:

- a. Start a new metric standard part file.
- b. Draw the outer loop by specifying the coordinates of points in the **Inventor Precise Input** toolbar.
- c. Draw the inner closed loop using the **Inventor Precise Input** toolbar, refer to Figure 2-39.
- d. Save the sketch with the name *Tutorial2.ipt* and close the file.

## Starting a New File

1. Choose the **New** button from the **Launch** panel of the **Get Started** tab in the **Ribbon** to display the **New File** dialog box. Choose the **Metric** tab from the dialog box and then double-click on the **Standard (mm).ipt** icon to start a standard metric template.

## Drawing the Sketch

As evident in Figure 2-38, 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 in the model automatically when you extrude the sketch. This reduces the time and effort required in creating the inner cavity as another feature. Therefore, for this tutorial you can draw both the loops together.

The **Inventor Precise Input** toolbar was invoked in the previous tutorial and therefore, it is available on the screen.

1. Choose the **Line** button from the **Sketch** tab; you are prompted to specify the start point of the line or drag off an endpoint to create a tangent arc. The points and their coordinates, which you need to enter in the **Inventor Precise Input** toolbar, are given below:



Points	Coordinates (X, Y)
1	-40,-25
2	40,-25
3	40,-15
4	25,-15
5	25,15
6	40,15
7	40,25
8	-40,25
9	-40,15
10	-25,15
11	-25,-15
12	-40,-15
13	-40,-25

2. After specifying all these points, right-click to display the shortcut menu. Choose **Done** from the shortcut menu to exit the **Line** tool.

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

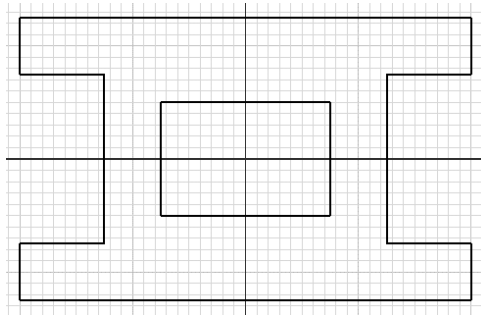
3. Choose the **Two Point Rectangle** button from the **Draw** panel of the **Sketch** tab; you are prompted to specify the first corner of the rectangle. Enter the first corner's **X** and **Y** coordinates as **-15** and **-10** in their respective edit boxes in the **Inventor Precise Input** toolbar. Press ENTER; you are prompted to specify the opposite corner of the rectangle.
4. Enter the coordinates of the other corner of the rectangle as **15** and **10**. Next, right-click and choose **Done** from the shortcut menu displayed; the sketch is created in Figure 2-39.



## Saving the Sketch

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

1. Choose the **Return** button from the **Quick Access Toolbar** to exit the sketching environment.



**Figure 2-39** Completed sketch for Tutorial 2

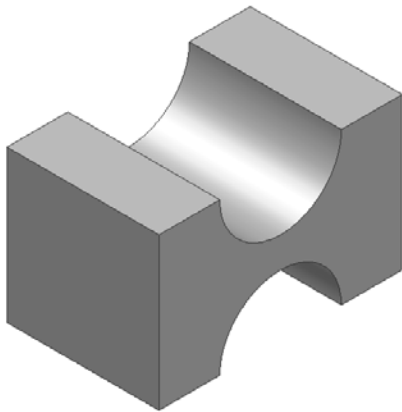
2. Choose the **Save** button and save the sketch with the name *Tutorial2* at location given below:

*C:\Inventor\_2010\c02*

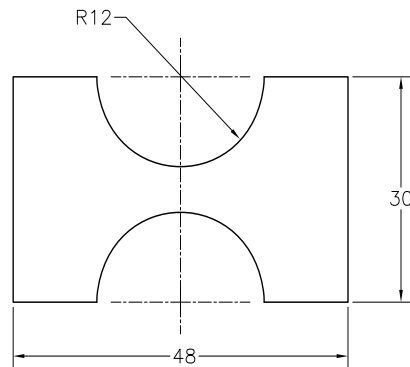
3. Choose **Close > Close** from the **Application Menu** to close this file.

### Tutorial 3

In this tutorial, you will draw the sketch of the model shown in Figure 2-40. The sketch of the model is shown in Figure 2-41. Do not dimension the sketch as these dimensions are only for reference.  
(Expected time: 30 min)



**Figure 2-40** Model for Tutorial 3



**Figure 2-41** 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 using the **Arc** and **Line** tools, refer to Figure 2-43.
- c. Save the sketch with the name *Tutorial3* and close the file.

## Starting a New File


1. Choose the **New** button 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 the endpoint of the arc. Therefore, you will use the **Center Point Arc** tool to draw this arc.

1. Click the down arrow at the bottom of the **Three Point Arc** in the **Draw** panel of the **Sketch** tab; a flyout is displayed. Choose the **Center Point Arc** button from this flyout; you are prompted to specify the center of the arc. 
2. Enter the coordinates of the center of the arc as **0,15** in the **Inventor Precise Input** toolbar; you are prompted to specify the start point of the arc.
3. Specify the start point as **-12,15** in the **Inventor Precise Input** toolbar.

Next, you need to define the endpoint of the arc. The arc drawn when you specify the endpoint, can be in the clockwise or counterclockwise direction. However, you need to define the arc in the counterclockwise direction. To do so, you need to move the mouse to a small distance in the counterclockwise direction. By doing this, you can define the direction in which the arc will be drawn.

4. Move the mouse from the start point of the arc to a small distance in the counterclockwise direction. Now, enter the coordinates of the endpoint of the arc as **12,15** in the **Inventor Precise Input** toolbar. As a result, the upper arc is drawn.
5. Next, you need to draw the lines in the sketch. Choose the **Line** button from the **Draw** panel of the **Sketch** tab; you are prompted to specify the start point of the line. 

It is evident from Figure 2-41 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.

It is recommended that you select the start point of the line in the drawing window. When you invoke any sketching tool, a yellow circle appears on the cursor. If you move the cursor close to any endpoint, the yellow circle at the end of the cursor automatically snaps to the endpoint and turns green. You will also notice that the symbol of the coincident constraint is displayed. This symbol suggests that the coincident constraint will be automatically applied to the endpoint of the arc and the start point of the line.

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.

Because it is easier to define the points by using relative coordinates, therefore, it is recommended that you use the **Precise Delta** button in the **Inventor Precise Input** toolbar to draw the 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 will be placed at the origin (0,0). Enter the start point of the line as **-12, 15** in the **Inventor Precise Input** toolbar. Next, enter the endpoint of the line as **-12, 0** in the **Inventor Precise Input** toolbar.

8. Enter the second point as **0, -30** and the third point as **12, 0** 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 from 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, because of the triad, you will not be able to view the gray circle. 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, therefore, 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.

You will notice that as you move the cursor, an arc normal to the last line is drawn. Remember that you need to drag the mouse upward only through a small distance and then drag it toward right without releasing the left mouse button. Note that the point at which you release the left mouse button will be taken as the endpoint of the arc. Therefore, you need to be very careful in specifying the endpoint.

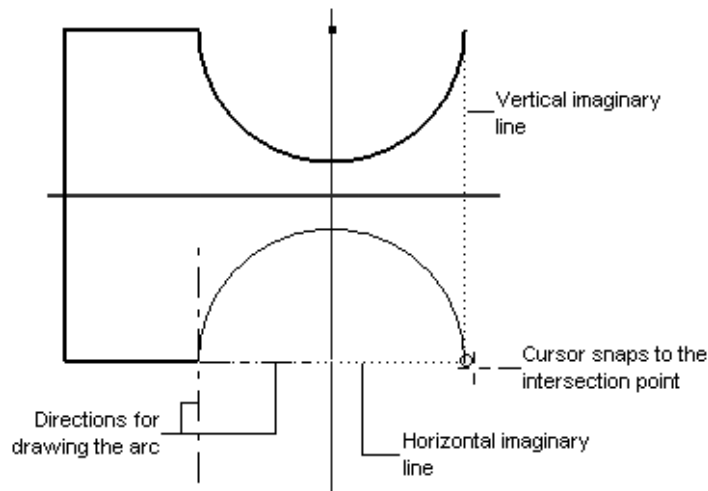
While drawing the arc by dragging the cursor, you cannot use the **Inventor Precise Input** toolbar. This is because the point where you release the left mouse button will be taken as the endpoint of the arc. Therefore, it is very difficult to define the endpoint of the arc precisely. To solve this problem, 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, imagine 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, which is essentially the endpoint of the lower arc. The temporary tracking option draws these imaginary lines for you and removes them after you have selected the point.

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, since 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 2-42. This point is the endpoint of the lower arc. The cursor will automatically snap to the point where both the imaginary lines intersect. Do not release the left mouse button until this entire process is completed.



**Figure 2-42** Use of the temporary tracking option to draw an arc



#### Note

*In Figure 2-42, the major and minor grid lines and triad are not displayed for a better display of the sketch and the imaginary lines.*

12. When the cursor snaps to the intersection point of the imaginary lines, release the mouse button to complete the lower arc.
13. Enter the coordinates of the next point as **12,0** in the edit boxes in the **Inventor Precise Input** toolbar.

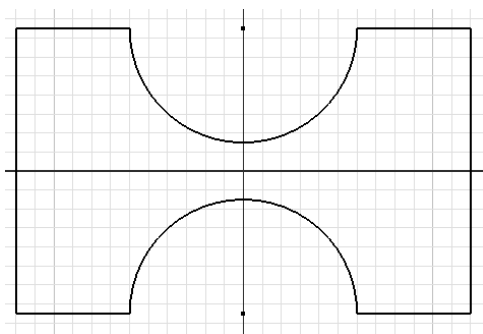
14. For the next point, you can enter the coordinates in the **Inventor Precise Input** toolbar or use the temporary tracking option. To use the **temporary tracking** option, move the cursor close to the endpoint of the upper arc. Once the cursor snaps to this point and turns green, move it horizontally 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 have to press the left mouse button only 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. Right-click to display the shortcut menu and choose **Done** to exit the **Line** tool.
16. The final sketch for Tutorial 3 is shown in Figure 2-43.



*Figure 2-43 Final sketch for Tutorial 3*

## Saving the Sketch

1. Choose the **Return** button from the **Quick Access Toolbar** to exit the sketching environment.
2. Choose the **Save** button and save the sketch with the name *Tutorial3* at location given below:  
  
*C:\Inventor\_2010\c02*
3. Choose **Close > Close** from the **Application Menu** to close this file.

## Tutorial 4

In this tutorial, you will draw the basic contour of a revolved solid model shown in Figure 2-44. The contour that you draw for creating this revolved solid is shown in Figure 2-45. Do not dimension the sketch as these dimensions are for reference only.

(Expected time: 30 min)

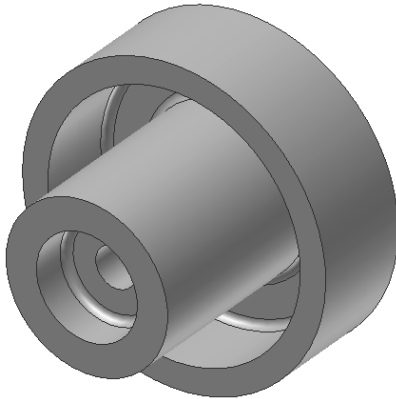


Figure 2-44 Revolved model for Tutorial 4

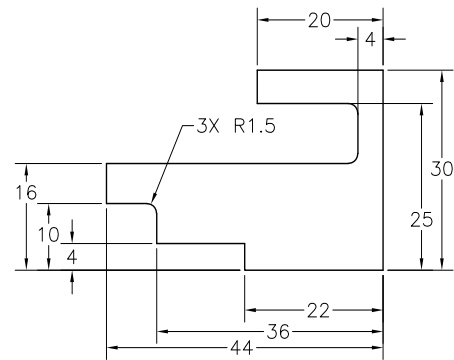


Figure 2-45 Sketch for the revolved model

The following steps are required to complete this tutorial:

- Start a new metric standard part file.
- Invoke the **Line** tool and draw the sketch by specifying the coordinates of points in the **Inventor Precise Input** toolbar; refer to Figure 2-47.
- Save the sketch with the name *Tutorial4* and close the file.

### Starting a New File

- Choose the **New** button from the **Quick Access** toolbar to display the **New File** dialog box.
- 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

- Choose the **Line** button from the **Draw** panel of the **Sketch** tab; 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; 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 this makes defining the points easier. The **Precise Delta** button in the **Inventor Precise Input** toolbar is chosen automatically if you use the same session of Autodesk Inventor. However, if you use a new session, you need to choose this button to invoke this tool.



2. Choose the **Precise Delta** button if it is not chosen and then press the ESC key. Next, invoke the **Line** tool and enter the start point of the line as **22, 0** 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

3. Right-click to display the shortcut menu and choose **Done** to complete the sketch. The sketch should look similar to the one shown in Figure 2-46. For your reference, the lines in the sketch are numbered.

The arcs at the end of lines 4 and 5, 5 and 6, and, 8 and 9 will be created using the **Fillet** tool. This tool will draw the arcs at the point of intersection of the lines and remove the sharp corners.

4. Choose the **Fillet** button from the **Draw** panel of the **Sketch** tab; 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.



5. Select line 4 and then line 5, refer to Figure 2-46; the fillet is created between these lines and the fillet radius is displayed in the sketch.
6. Similarly, select lines 5 and 6, and then lines 8 and 9 to create the fillet between these lines. Right-click and choose **Done** to exit the **Fillet** tool after creating all the fillets.

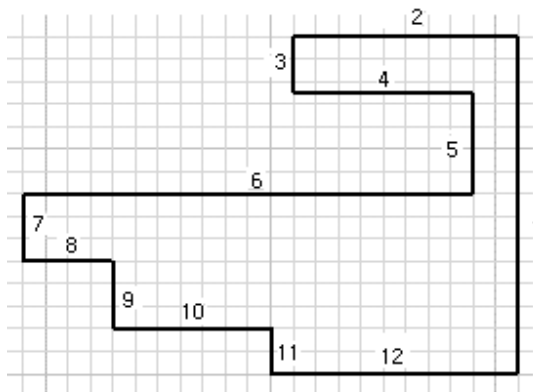
As all lines were filleted with the same radius value, the fillet radius is not displayed on other fillets. This completes the sketch. The final sketch for this tutorial, after filleting, is shown in Figure 2-47.



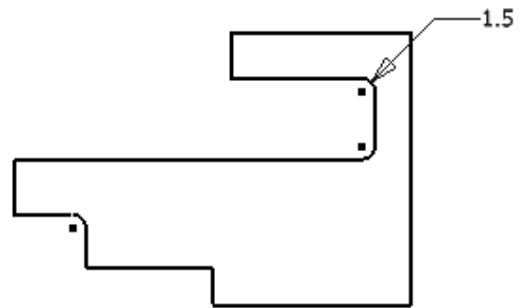
**Note**  
*In Figures 2-46 and 2-47, the display of axes has been turned off for clarity in displaying the lines of the sketch.*

**Saving the Sketch**

1. Choose the **Finish Sketch** button from the **Exit** panel of the **Sketch** tab.



**Figure 2-46** Sketch after drawing the lines



**Figure 2-47** Final sketch after filleting

2. Choose the **Save** button and save this sketch with the name *Tutorial4* at the location given below:

*C:\Inventor\_2010\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 active by default. (T/F)
3. You cannot turn off the display of grid lines. (T/F)
4. You cannot draw an arc from within the **Line** tool. (T/F)
5. The two types of sketching entities that can be drawn in Autodesk Inventor 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. Rectangles in Autodesk Inventor are drawn as a 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 on right-clicking.

## Review Questions

Answer the following questions:

1. In most designs, generally the first feature or the base feature has to be 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 cannot control the display of grid lines. (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 buttons in the **Tools** tab is used to invoke additional toolbars?
  - (a) **Application Options**
  - (b) **Customize**
  - (c) **Document Setting**
  - (d) None of the above
8. Which of the following drawing display options is used to interactively zoom in and out the 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 the above

## Exercises

### Exercise 1

Draw the basic 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 for reference only.

(Expected time: 30 min)

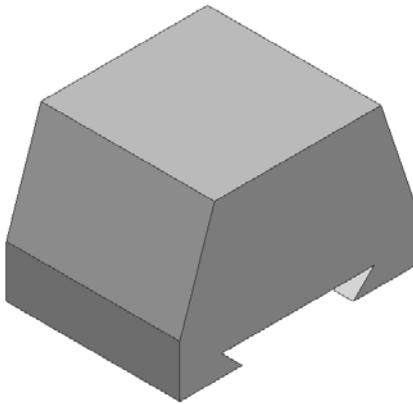


Figure 2-48 Model for Exercise 1

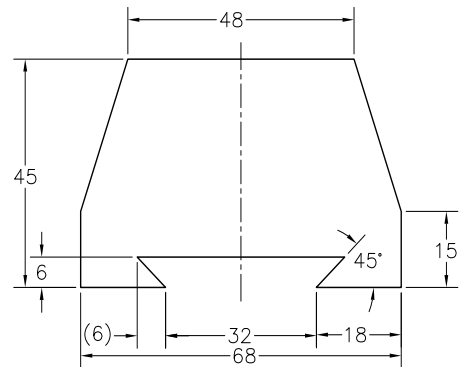


Figure 2-49 Sketch for Exercise 1

### Exercise 2

Draw the basic sketch for the model shown in Figure 2-50. The sketch to be drawn is shown in Figure 2-51. Do not dimension it, as these dimensions are just for reference.

(Expected time: 45 min)

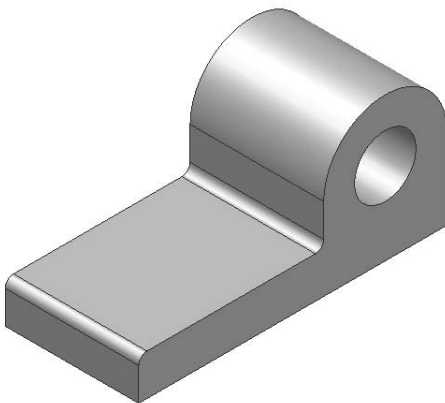


Figure 2-50 Model for Exercise 2

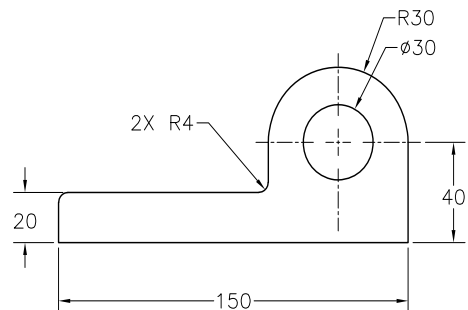


Figure 2-51 Sketch for Exercise 2

**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