Autodesk®
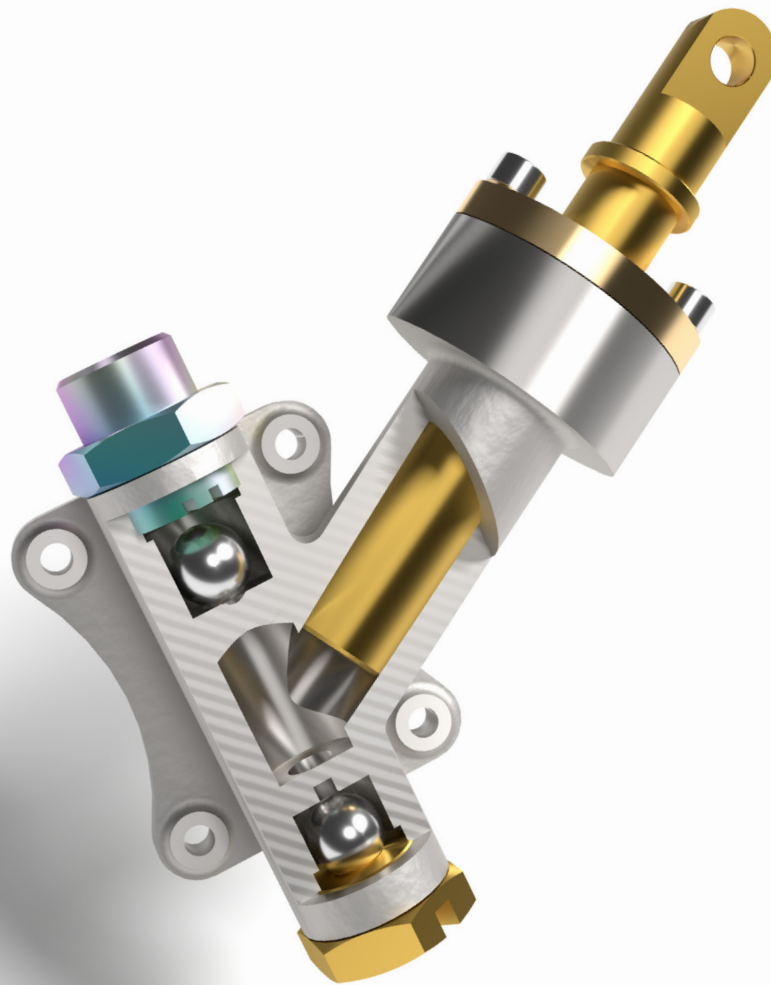# Inventor® 2023
Essentials Plus

Shawna Lockhart & Daniel T. Banach

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2 Sketching, Constraining, and Dimensioning

INTRODUCTION

Most 3D parts in Autodesk Inventor start from a 2D sketch. This chapter first provides a look at the application options for creating a part file and sketching. It then covers the three steps in creating a 2D parametric sketch: sketching a rough 2D outline of a part, applying geometric constraints, and then adding parametric dimensions. Lastly, you learn how to use 2D AutoCAD data in a sketch.

OBJECTIVES

After completing this chapter, you will be able to do the following:

- ☐ Change the part and sketch Application Options to meet your needs
- ☐ Sketch an outline of a part
- ☐ Create geometric constraints to a sketch to control design intent
- ☐ Use construction geometry to help constrain a sketch
- ☐ Dimension a sketch
- ☐ Change a dimension's value in a sketch
- ☐ Insert AutoCAD DWG data into a part's sketch

PART AND SKETCH APPLICATION OPTIONS

Before you start a new part, examine the part and sketch options in Autodesk Inventor that will affect how the part file will be created and how the sketching environment will look and act. While learning Autodesk Inventor, refer to these option settings to determine which ones work best for you—there are no right or wrong settings.

Part Options

You can customize Inventor Part options to your preferences. Click the File tab > Options button, and click on the Part tab, as shown in the following image. Descriptions of a couple of the most common Part options follow. For more information about the Application Options consult the help system. These settings are global—they will affect all active and new Inventor documents.
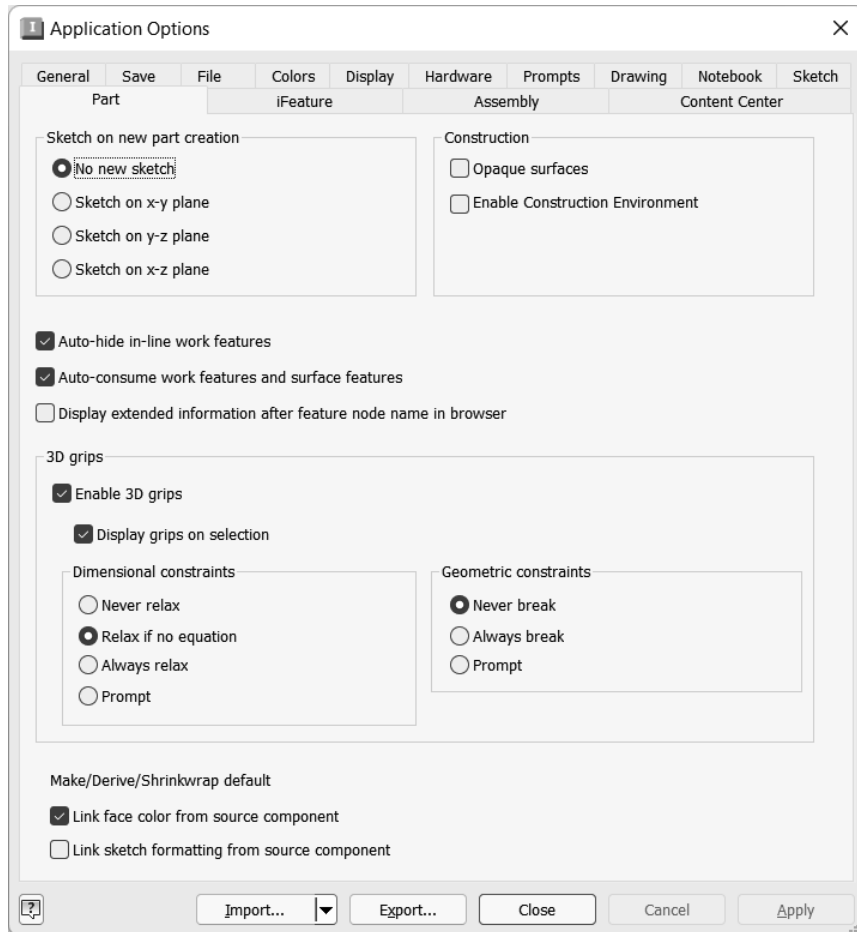

Figure 2.1

A common option that you may want to change is the first option: Sketch on New Part Creation. This option controls if and how a sketch is created when a part file is created.

No new sketch
When checked, Inventor does not set a sketch plane when you create a new part (this is the default setting).

Sketch on x-y plane
When checked, Inventor sets the x-y plane as the current sketch plane when you create a new part.

Sketch on y-z plane
When checked, Inventor sets the y-z plane as the current sketch plane when you create a new part.

Sketch on x-z plane
When checked, Inventor sets the x-z plane as the current sketch plane when you create a new part.

Sketch Options

Autodesk Inventor sketching options can be customized to your preferences. Click File tab > Options, and then click on the Sketch tab as shown in the following image. Descriptions of the most common Sketch options follow. For more information about the Application Options consult the help system. These settings are global and affect active and new Inventor documents.
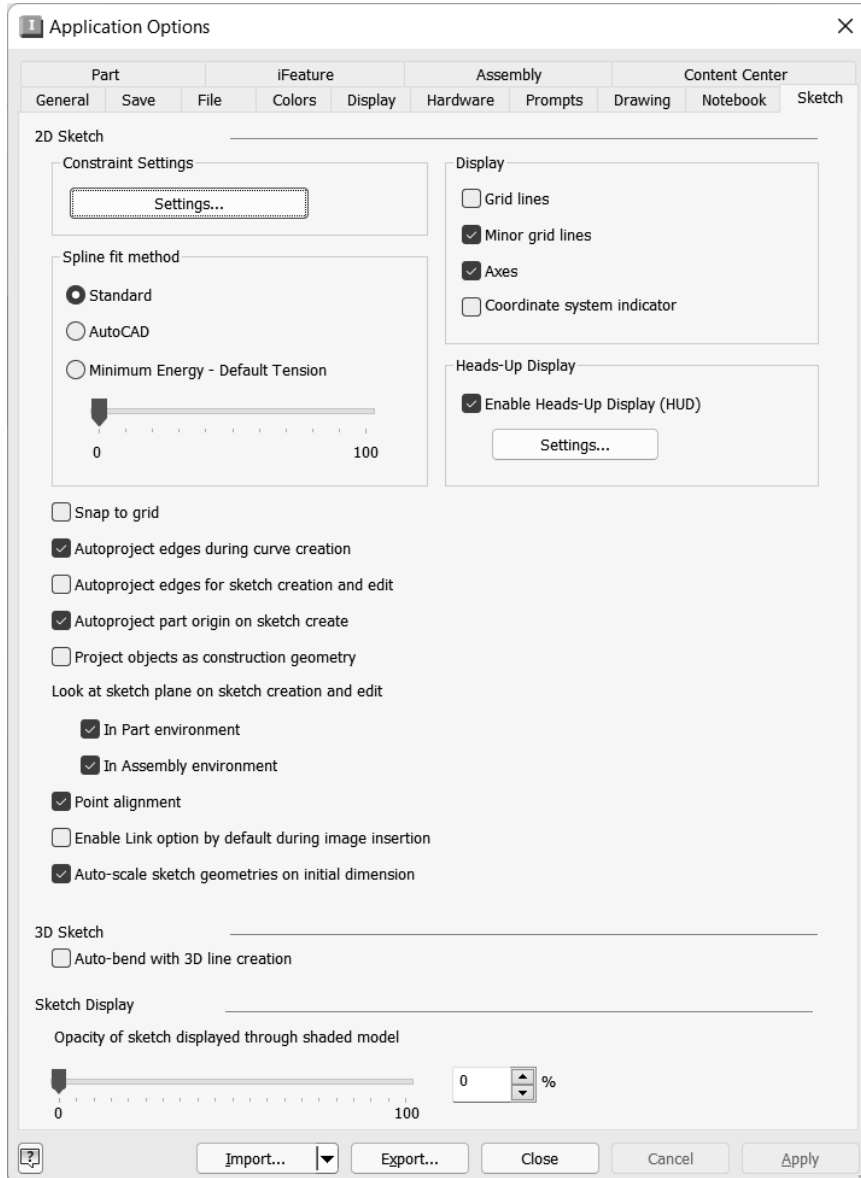

Figure 2.2

Following are descriptions of the common settings that you may want to change.

Constraint Settings

Click the Settings button to control how sketch constraints and dimensions behave.

Display

Grid lines

Toggles both minor and major grid lines on the screen on and off. To set the grid distance, open a part, or other file and click the Tools tab > Options panel > Document Settings command, and on the Sketch tab of the Document Settings dialog box, change the Snap Spacing and Grid Display.

Minor grid lines

Toggles the minor grid lines displayed on the screen on and off.

Axes

Toggles the lines that represent the X and Y-axis of the current sketch on and off.

Coordinate system indicator

Toggles the icon on and off that represents the X-, Y-, and Z-axes at the 0, 0, 0 coordinates of the current sketch.

Snap to Grid

When checked, endpoints of sketched objects snap to the intersections of the grid as the cursor moves over them.

Autoproject edges during curve creation

When checked and while sketching, place the cursor over an object and it will be projected onto the current sketch. You can also toggle Autoproject on and off while sketching by right-clicking and selecting Autoproject from the menu.

Autoproject edges for sketch creation and edit

When checked, and when a new sketch is created or edited, all the edges that define the plane are automatically projected as reference geometry.

Project objects as construction geometry

When checked, automatically projects selected geometry as construction geometry.

Look at sketch plane on sketch creation and edit

When checked, automatically changes the view orientation to look directly at the new or active sketch.

Autoproject part origin on sketch create

When checked, the parts origin point will automatically be projected when a new sketch is created. It is recommended to keep this setting on.

Point alignment

When checked, automatically infers alignment (horizontal and vertical) between endpoints of newly created geometry. No sketch constraint is applied. If this option is not checked, points can still be inferred; this technique is covered later in this chapter in the Inferred Points section.

Sketch Display

Change the opacity of a sketch that is displayed when the model is shaded.

UNITS

Autodesk Inventor uses a default unit of measurement for every part and assembly file. The default unit is set from the template file from which you created the part or assembly file. When specifying numbers in dialog boxes with no unit, the default unit will be used.

You can change the default unit in the active part or assembly document by clicking the Tools tab > Options panel > Document Settings button and click the Units tab as shown in Figure 2.3. The unit system values change for all the existing values in that file.
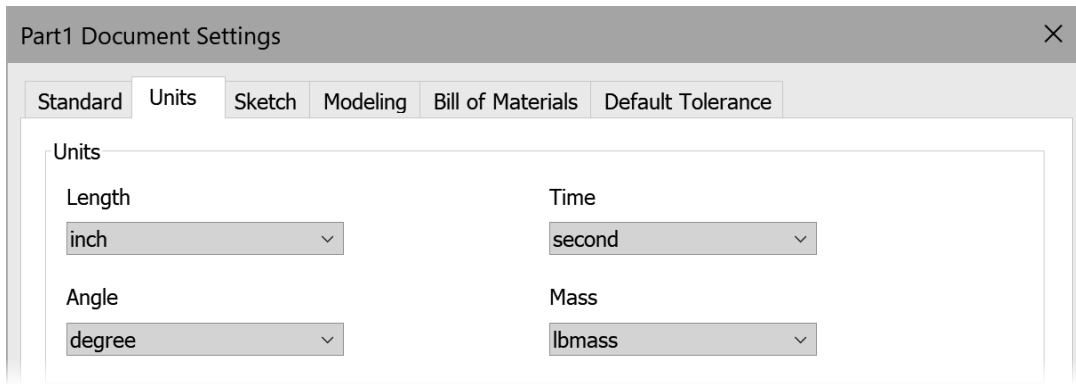
Part1 Document Settings   ✕

Standard | Units | Sketch | Modeling | Bill of Materials | Default Tolerance

Units

Length

inch ⌄

Angle

degree ⌄

Time

second ⌄

Mass

lbmass ⌄

Figure 2.3

TIP: In a drawing file, the appearance of dimensions is controlled by dimension styles. Drawing settings are covered in Chapter 5.

You can override the default unit for any value by entering the desired unit. If you were working in a metric file whose unit is set to mm, for example, and you placed a 20 mm horizontal dimension as shown in the following image on the left, and you edited the dimension to 1 in (adding the unit) as shown in the middle image, the dimension would appear on the screen in the default units which would be 25.4 as shown in the right image.

20

Edit Dimension : d0 ✕

20 mm   >   ✔

20

Edit Dimension : d0 ✕

1 in   >   ✔

25.4

Figure 2.4

When you edit a dimension, the overridden unit appears in the Edit Dimension dialog box. For the previous example when the 25.4 mm dimension is edited, 1 in is displayed in the Edit Dimension dialog box as shown in Figure 2.5.
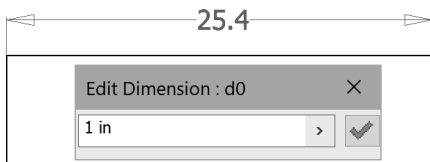
25.4

Edit Dimension : d0 ✕

1 in   >   ✔

Figure 2.5

TEMPLATES

Each new file is created from a template. You can modify existing templates or add your own templates. As you work, make note of the changes that you make to each file. You then create a new template file or modify an existing file that contains all the changes and save that file to your template directory, which by default in Windows 10 is *C:\Users\Public\Public Documents\ Autodesk\Inventor 2023\Templates*. You can also create a new subdirectory under the templates folder, and place any Autodesk Inventor file in this new directory. After adding an Inventor file, the new tab will appear, and it will be available as a template.

You can use one of two methods to share template files among many users. You can modify the location of templates by clicking the File tab > Options button > File tab, and modifying the Templates location as shown in the following image. The Templates location will need to be modified for each user who needs access to templates that are not stored in the local location.

Default templates

Configure Default Template...

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Templates\%LANGUAGE%\   🔍

Figure 2.6

You can also change the unit of measurement (inches or millimeters) for the default part and assembly template files and set the default drawing standard (ANSI, DIN, ISO, etc.) for the default drawing template by clicking Tools tab Application Options File Configure Default Template button as shown in the previous image. Then make the changes in the Configure Default Template dialog box as shown in the image.
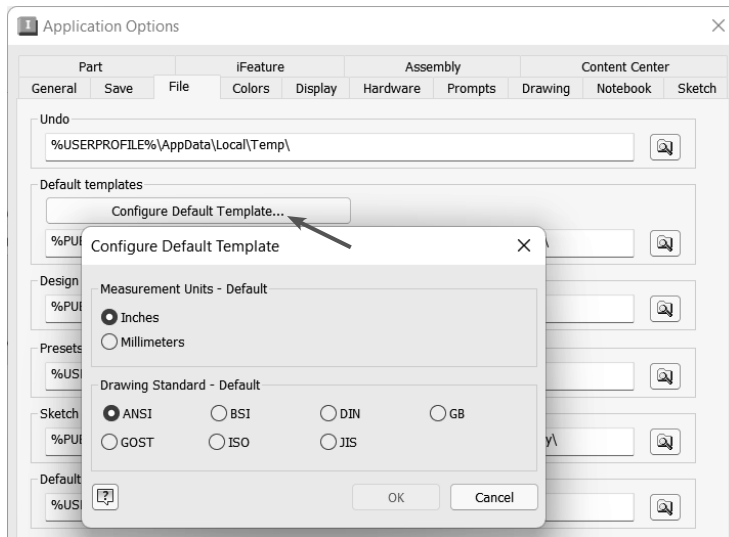

Figure 2.7

You can set the Templates location in each project file. This is useful if you need different templates for each project. While editing a project file, change the Templates location in the Folder Options area. The following image shows the default location in Windows 10. The Template location in the project file takes precedence over the Templates option in the Application Options, File tab.
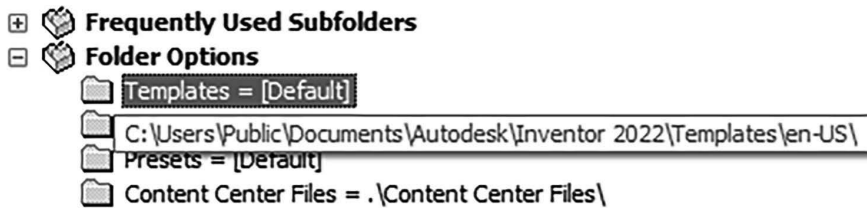

Figure 2.8

> **TIP:** Template files have file extensions that are identical to other files of the same type, but they are in the template directory. Template files should not be used as production files.

CREATING A PART FILE

The first step in creating a part is to start or create a new part file in an assembly. You can use the following methods to create a new part file:

- In the Quick Access toolbar click the down arrow on the New icon, and click Part as shown in the following image on the left. This creates a new part file based on the default unit as was discussed in the previous Templates section.

- Click the down arrow by New on the Home page and choose part (bottom)

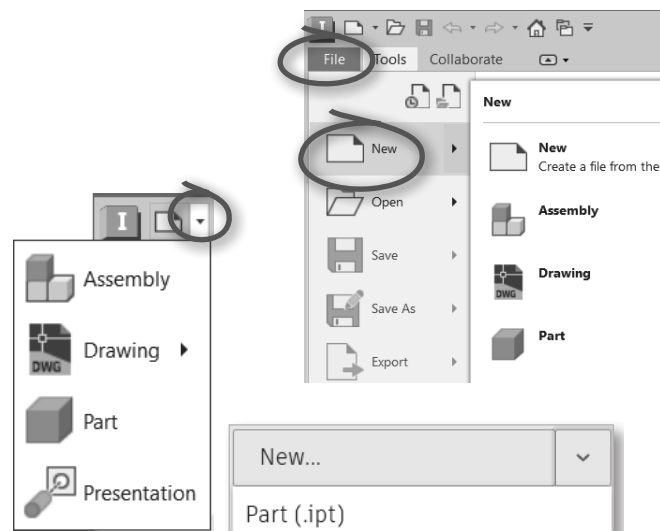- From the File tab click New > Part as shown at right.

Figure 2.9

TIP: The default unit for the part and assembly templates and the standard for the drawing template is set in the Application Options dialog box > File tab > Configure Default Template.

Creating a Part File from a Specified Template

You can also create a part file from a template that is not the default location by clicking the New file command from one of these areas:

- Quick Access toolbar as shown in the following image on the left.

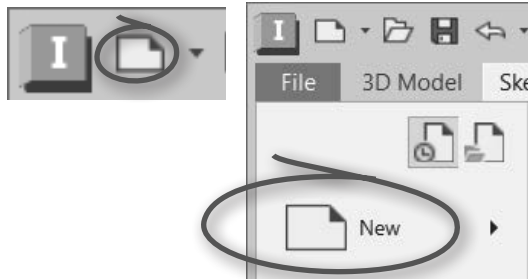- File tab, as shown in the middle image.

- Or press CTRL + N.

Figure 2.10

The Create New File dialog box appears. Then click the desired templates folder on the left side of the Create New File dialog box and then from the Part section on the right side of the dialog box click on the desired part template file, as shown in the following image.
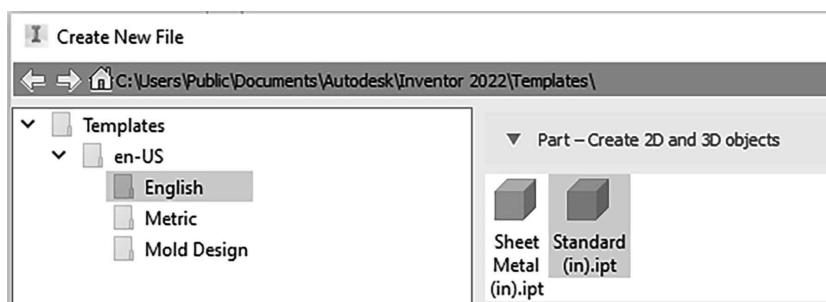

Figure 2.11

After starting a new part file using one of the previous methods, Autodesk Inventor's screen will change to reflect the part environment.

Sketches and Origin (Default) Planes

Before you start sketching, you select a plane on which to draw. A sketch is a plane on which 2D objects are sketched. You can use any planar part face or work plane to create a sketch. By default, when you create a new part file no sketch is created, and you will select an origin plane to sketch on. You can change the default plane on which you will create the sketch by selecting the File tab > Options and clicking on the Part tab. Select the sketch plane to which new parts should default.

Each time you create a new Autodesk Inventor part or assembly file, there are three planes (XY, YZ, and XZ), three axes (X, Y, and Z), and the center (origin) point at the intersection of the three planes and axes. You can use these default planes to create an active sketch. To see the planes, axes, or center point, expand the Origin entry in the browser by clicking on the left side of the text. You can then move the cursor over the names, and they will appear in the graphics window. The following image on the left and the middle image illustrate the default planes, axes, and center point with their visibility on. To leave the visibility of the planes or axes on, right-click in the browser while the cursor is over the name and click Visibility from the menu. When a plane is visible

you can display the plane's label by moving the cursor over a plane in the browser or in the graphics window as shown in the following image on the right.
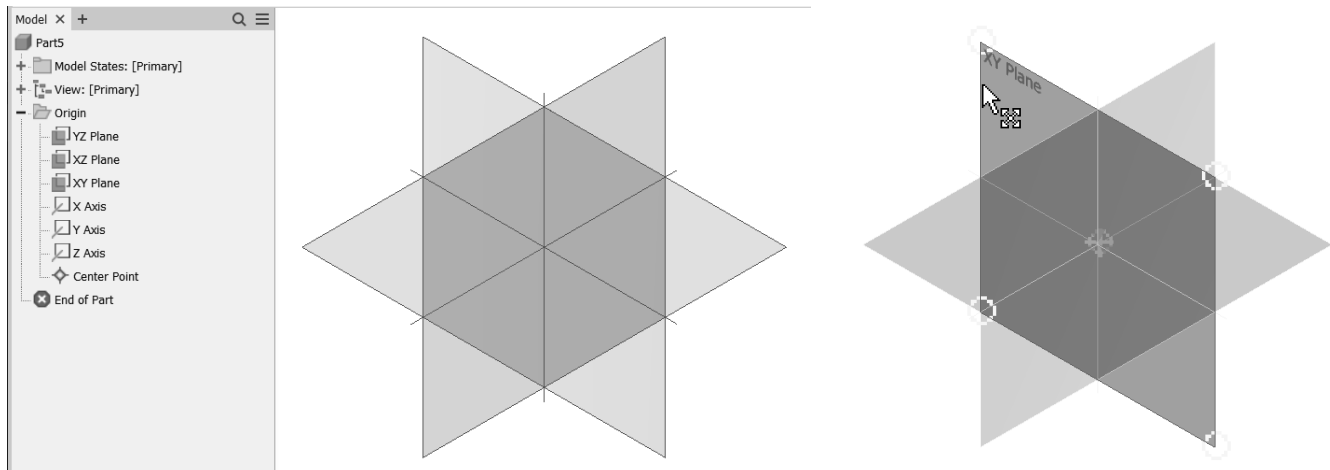

Figure 2.12

Origin 3D Indicator

When working in 3D, it is common to get your orientation turned around. By default, in the lower left corner of the graphics screen, there is an XYZ axis indicator that shows the default (world) coordinate system as shown in the following image on the left. The direction of these planes and axes cannot be changed. The arrows are color-coded:

- Red arrow = X axis

- Green arrow = Y axis

- Blue arrow = Z axis

In the Application Options dialog box > Display tab, you can turn the axis indicator and the axis labels on and off as shown in the following image on the right.
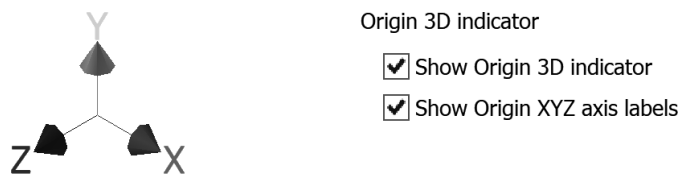

Origin 3D indicator

☑ Show Origin 3D indicator

☑ Show Origin XYZ axis labels

Figure 2.13

By default, Inventor will automatically project the origin point (0,0) when a new sketch is created in a part file. The origin point can be used to constrain a sketch to the 0, 0 point of the sketch. If desired, you can turn this option off by clicking the Tools tab > Application Options or from the File tab click Options > Sketch tab, and then uncheck Autoproject part origin on sketch create as displayed in the following image.

☑ Autoproject part origin on sketch create

Figure 2.14

New Sketch

By default, when you create a new part file no sketch is active. You can define a plane from the origin folder to be the default by selecting a default plane from the File tab > Options > Part tab. Issue the 2D Sketch command to create a new sketch on a planar part face or a work plane or to activate a non-active sketch in the part. When you are in a part file that does not have a sketch defined and when you start the 2D Sketch command, the origin planes will be displayed in the graphics window, and you can select one of these planes to create the sketch on.

To create a new sketch or make an existing sketch active, use one of these methods:

- Click the 3D Model tab > Sketch panel > Start 2D Sketch as shown in the following image on the left or from the Sketch tab > Sketch panel > Start 2D Sketch. Then click a planar face, a work plane, or an existing sketch in the browser.

- Press the S key (a keyboard shortcut) and click a planar face of a part, a work plane, or an existing sketch in the browser.

- While not in the middle of an operation, right-click in the graphics window, and select New Sketch from the marking menu as shown in the middle image. Then click a planar face, a work plane, or an existing sketch in the browser.

- While not in the middle of an operation, click a planar face of a part, a work plane, or an existing sketch in the browser. Then right-click in the graphics window, and click 2D Sketch from the mini-toolbar. The following image on the right shows the mini-toolbar after selecting an origin plane.

TIP: You can start the command first, and then select a plane *or* you can select a plane first and then start the command.
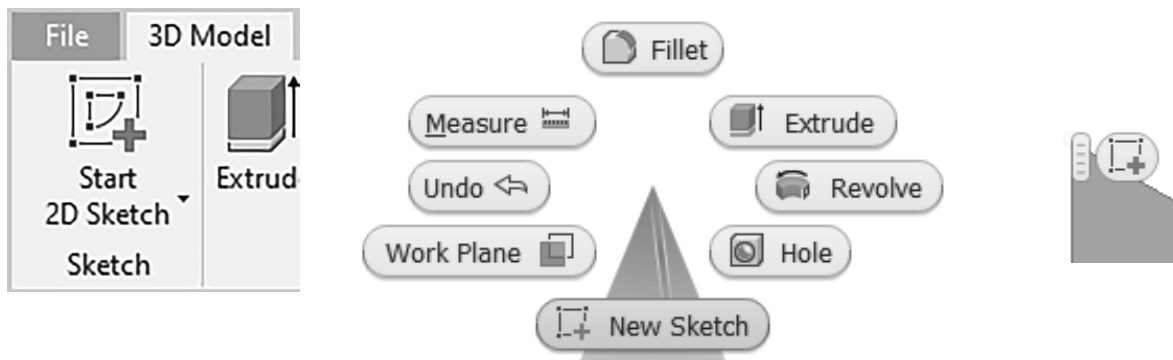

Figure 2.15

After creating a sketch, a Sketch entry will appear in the browser as shown in the following image, and a Sketch tab will appear in the ribbon. By default, after you have defined a sketch, the X and Y-axes will align automatically to this plane, and you can begin to sketch.
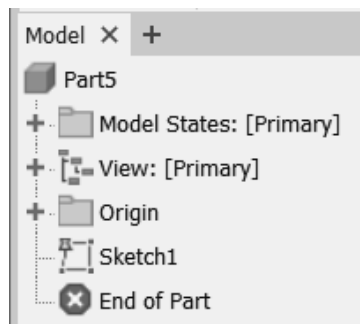

Figure 2.16

SKETCHING THE 2D OUTLINE OF THE PART

As stated at the beginning of this chapter, 3D parts usually start with a 2D sketch of the outline shape of the part. You can create a sketch with lines, arcs, circles, splines, or any combination of these elements. The next section will cover sketching strategies, commands, and techniques.

Sketching Overview

When deciding what outline to start with, analyze how the finished shape will look. Look for the 2-dimensional shape that best describes the part. When looking for this outline, try to look for a flat 2-dimensional shape that can be extruded or revolved to create a shape that other features can be added to, to create the finished part. It is usually easier to sketch 2-dimensional geometry than 3-dimensional geometry. As you gain modeling experience, you can reflect on how you created the model and think about other ways that you could have built it. There is usually more than one way to generate a given part.

When sketching, draw the geometry so that it is close to the desired shape and size— you do not need to be concerned about exact dimensional values. Even though Inventor allows islands in the sketch (closed objects that lie within another closed object) it is NOT recommended to sketch islands (when you extrude a sketch, island(s) may become voids in the solid). A better method is to place features, which make editing a part easier. For example, instead of sketching a circle inside a rectangle to represent a hole, extrude a rectangle and then place a hole feature.

The following guidelines will help you successfully generate sketches:

- Select a 2-dimensional outline that best represents the part. The 2D outline will be used to create the base feature. A base feature is the first feature. It is the feature other features will add material to or remove material from.

- Draw the geometry close to the finished size. If you want a 20-inch square, for example, do not draw a 200-inch square. Use dynamic input to define the size of the geometry. Dynamic input is covered in a later section in this chapter.

- Create the sketch proportional in size to the finished shape. When drawing the first object, verify its size in the lower-right corner of the status bar. Use this information as a guide.

- Draw the sketch geometry so it doesn't lie over other geometry, that is, a line on top of another line.

- Do not allow the sketch to have a gap; the geometry should start and end at a single point, just as the start and end points of a rectangle share the same point.

- Keep sketches simple. Leave out fillets and chamfers when possible. You can easily place them as features after making the sketch into a solid. The simpler the sketch, the fewer constraints and dimensions are required to constrain the model.

SKETCH COMMANDS

Before you start sketching the outline of the part, examine the 2D sketching commands that are available. After creating a sketch, the 2D sketch tab is current in the ribbon. The most frequently used commands will be explained throughout this chapter. Consult the help system for information about the remaining commands.
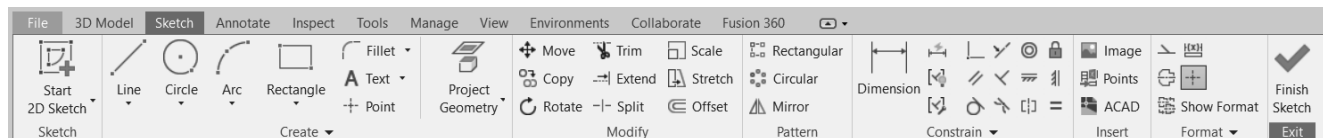
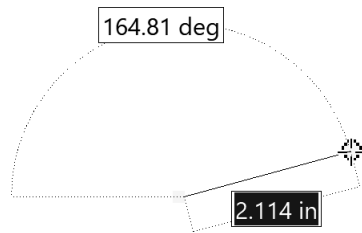

Figure 2.17

Using the Sketch Commands

After starting a new part, a sketch will automatically be active so that you can now use the sketch commands to draw the shape of the part. To start sketching, issue the sketch command that you need, click a point in the

graphics window, and follow the prompt on the lower-left corner of the status bar. The sections that follow will introduce techniques that you can use to create a sketch.

Dynamic Input in the sketch environment makes a Heads-Up Display (HUD), which shows information near the cursor for many sketching commands that helps you keep your eyes on the screen. While using the Line, Circle, Arc, Rectangle, or Point commands, you can enter values in the input fields. You can toggle between the value input fields by pressing the TAB key. The following image shows examples of entering Cartesian coordinates and Polar coordinates.
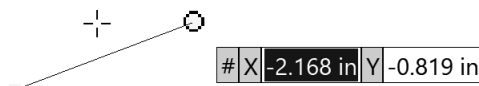
> **TIP:** If no data is entered in the input fields and you click in the graphics window to locate geometry, dimensions will NOT automatically be placed. You can manually place dimensions and constraints after the geometry is sketched.

Polar Coordinates   **Cartesian Coordinates**

Figure 2.18

Dimension Input

When defining lengths and angles for a second point, the dimensional values change as you move the cursor. Press TAB to move to the next input field or click in another cell. After entering a value and pressing the Tab key, the value will be locked and a lock icon will appear to the right of the value as shown in the following image. After a dimension's value is locked, the parametric dimension will be created after clicking a point or pressing the Enter key. You can change the value in an input field by either clicking in the field or pressing the Tab key until the field is highlighted and then typing in a new value.
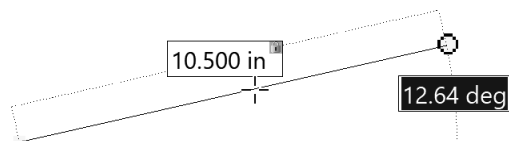
Figure 2.19

Line Command

The Line command is one of the most powerful commands that you will use to sketch. Not only can you draw lines with it, but you can also draw an arc from the endpoint of a line segment. To start sketching lines, click the Line command from the Sketch tab > Create panel as shown in the following image on the left, or right-click in a blank area in the graphics window and click Create Line from the marking menu as shown in the middle image, or press the L key on the keyboard. After starting the Line command, you will be prompted to click a first point, select a point in the graphics window, and then click a second point. The image on the right shows the line being created with the dynamic input as well as the horizontal constraint.
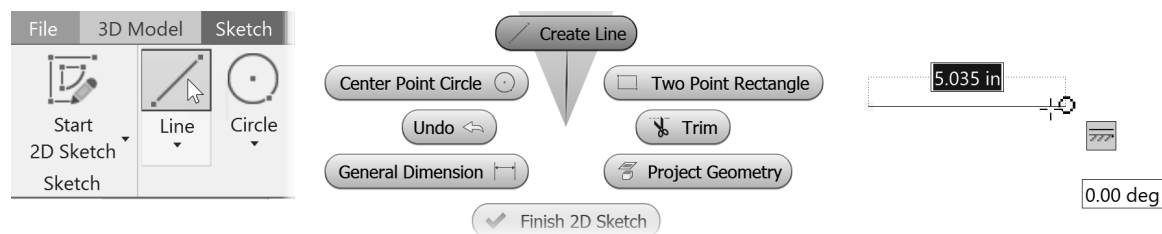
Figure 2.20

Drawing an Arc within the Line Command

You can continue drawing line segments, or you can sketch an arc from the endpoint. To make an arc from the line command, move the cursor over the endpoint of a line segment or arc until a small gray circle appears (Figure 2.21 left). Click the small circle and holding the left mouse button down, drag the cursor in the direction to define the arc.

Up to eight different arcs can be drawn, depending upon how you move the cursor. The arc will be tangent to the horizontal or vertical edges that are displayed from the selected endpoint.

Figure 2.21 right shows an arc that is tangent to the sketched line being drawn.

Figure 2.21

TIP: When sketching, look at the bottom-right of the status bar at the bottom of the screen to see the coordinates, length, and angle of the objects you are drawing.

The following image shows the status bar when a line is being drawn.

2.500 in, -0.001 in x=2.500 in y=-0.001 in

Figure 2.22

Object Tracking – Inferred Points

If the Point Alignment On option is checked from the Sketch tab of the Application Options, dashed lines will appear on the screen as you sketch. These dotted lines represent the endpoints; midpoints; and theoretical intersections of lines, arcs, and center points of arcs and circles that represent their horizontal, vertical, or perpendicular positions.

As the cursor gets close to these inferred points, it will snap to that location. If that is the point that you want, click that point; otherwise, continue to move the cursor until it reaches the desired location. When you select inferred points, no constraints (geometric rules such as horizontal, vertical, collinear, and so on) are applied from them. Using inferred points helps create more accurate sketches. The following image shows the inferred points from two midpoints that represent their horizontal and vertical position.
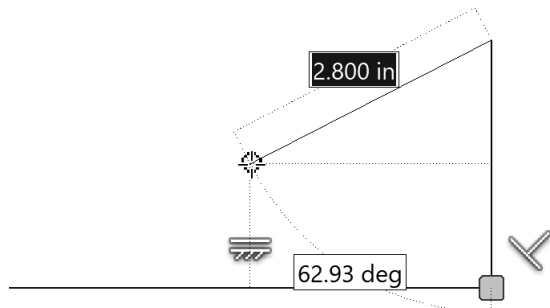
2.800 in

62.93 deg

Figure 2.23

Automatic Constraints

As you sketch, a small constraint symbol appears representing geometric constraint(s) that will be applied to the object. If you do not want a constraint to be applied, hold down the CTRL key when you click to create the geometry.

Figure 2.24 shows a line being drawn from the arc, tangent to the arc, and parallel to the angled line, and the dynamic input is also displayed. The symbol appears near the object from which the constraint is coming. Constraints will be covered in the next section.
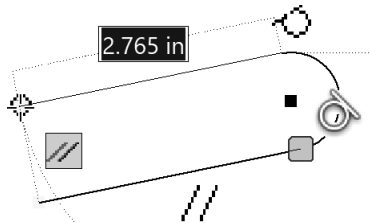
Figure 2.24

Scrubbing

As you sketch, you may prefer to apply a constraint different from the one that automatically appears on the screen. You may want a line to be perpendicular to a given line, for example, instead of being parallel to a different line. The technique to change the constraint is called scrubbing.

To place a different constraint while sketching, move the cursor so it touches (scrubs) the other object to which the constraint should be related. Move the cursor back to its original location, and the constraint symbol changes to reflect the new constraint. The same constraint symbol will also appear near the scrubbed object, representing that it is the object to which the constraint is matched. Continue sketching as normal.

The following image shows the top horizontal line being drawn with a parallel constraint that was scrubbed from the bottom horizontal line. Without scrubbing the bottom horizontal line, the applied constraint would have been perpendicular to the right vertical line.
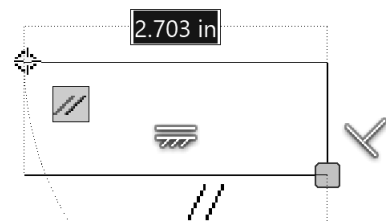
Figure 2.25

Deleting Objects

To delete objects first cancel the command that you are in by pressing the ESC key. Then select objects to delete, and either press the DELETE key or right-click and choose Delete as shown in Figure 2.26.
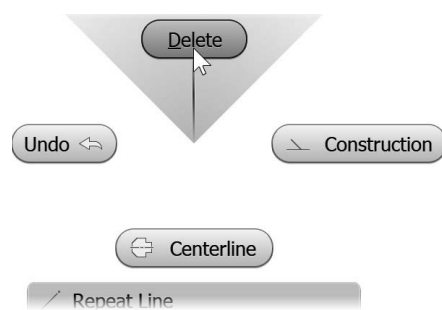
Figure 2.26

Common Sketch Commands

The following table lists common 2D sketch commands. Some commands are available by clicking the down arrow in the lower-right corner of the top command in the panel. Consult the help system for more information.

Command	Function
Center-point Circle	Creates a circle by clicking a center point for the circle and then a point on the circumference of the circle.
Tangent Circle	Creates a circle that will be tangent to three lines or edges by clicking the lines or edges.
Three-Point Arc	Creates an arc by clicking a start and endpoint and a point that will lie on the arc.
Tangent Arc	Creates an arc that is tangent to an existing line or arc by clicking the endpoint of a line or arc and then clicking a point for the other endpoint of the arc.
Center-Point Arc	Creates an arc by clicking a center point for the arc and then clicking a start and endpoint.
Two-Point Rectangle	Creates a rectangle by defining a point and then clicking another point to define the opposite side of the rectangle. The edges of the rectangle will be horizontal and vertical. If values were entered, dimensions will be placed on the rectangle.
Three-Point Rectangle	Creates a rectangle by clicking two points that will define an edge and then clicking a point to define the third corner. You can also type values to define the three points of the rectangle, and dimensions will be created that define the size of the rectangle.
Two-Point Center Rectangle	Creates a rectangle by defining a center point and another point to define the rectangle's size or type values for the center point and its X and Y values of the rectangle. The edges of the rectangle will be horizontal and vertical, and if values were entered, dimensions will be created.
Three-Point Center Rectangle	Creates a rectangle by defining a center point, a point to define the rectangle's starting point and its angle and another point size or type values for the center point, and size of the rectangle. The edges of the rectangle will be horizontal and vertical, and if values were entered, dimensions will be created.
Center to Center Slot	Creates a slot by defining the center-to-center distance, angle, and the diameter.
Overall Slot	Creates a slot by defining the overall distance, angle, and then the diameter.
Center Point Slot	Creates a slot by defining the center-to-center distance, angle, and the diameter.
Three Point Arc Slot	Creates an arc slot by defining a start point, end point, and an angle, a radius of the center of the slot and then the diameter of the slot.
Center Point Arc Slot	Creates an angled slot by defining a radius of the center of the slot and a starting angle, an ending angle and then the diameter of the slot.
Fillet	Creates a fillet between two nonparallel lines, two arcs, or a line and an arc at a specified radius. If you select two parallel lines, a fillet is created between them without specifying a radius. When the first fillet is created, a dimension will be created. If many fillets are placed in the same operation, you choose to either apply or not apply an equal constraint.
Chamfer	Creates a chamfer between lines. There are three options to create a chamfer: both sides equal distances, two defined distances, or a distance and an angle.
Polygon	Creates an inscribed or a circumscribed polygon with the number of faces that you specify. The polygon's shape is maintained as dimensions are added.
Mirror	Mirrors the selected objects about a centerline. A symmetry constraint will be applied to the mirrored objects.

Rectangular Pattern	Creates a rectangular array of a sketch with rows and columns that you specify.
Circular Pattern	Creates a circular array of a sketch with copies and spacing that you specify.
Offset	Creates a duplicate of the selected objects that are a given distance away. By default, an equal-distance constraint is applied to the offset objects.
Trim	Trims the selected object to the next object it finds. Click near the end of the object that you want trimmed. While using the Trim command, hold down the SHIFT key to extend objects. If desired, hold down the CTRL key to select boundary objects. While in the Trim command you can also hold down the left mouse button and move the cursor to dynamically trim geometry. While in the Dynamic mode you can hold down the Shift key to dynamically extend geometry.
Extend	Extends the selected object to the next object it finds. Click near the end of the object that you want extended. While using the Extend command, hold down the SHIFT key to trim objects. If desired, hold down the CTRL key to select boundary objects. While in the Extend command, you can also click and hold down the left mouse button and move the cursor to dynamically extend geometry. While in the Dynamic mode you can hold down the Shift key to dynamically trim geometry.

Selecting Objects

After sketching objects, you may need to move, rotate, or delete some or all the objects. To edit an object, it must be part of a selection set. There are multiple methods that you can use to place objects into a selection set.

- **CTRL or SHIFT Keys**. You can select objects individually by clicking on them. To manually select multiple individual objects, hold down the CTRL key or SHIFT key while clicking the objects. You can remove selected objects from a selection set by holding down the CTRL or SHIFT key and reselecting them. As you select objects, their color will change to show that they have been selected.

- **Window**. You can select multiple objects by defining a selection window. Not all commands allow you to use the selection window technique and only allow single selections. To define the window, click a starting point. With the left mouse button depressed, move the cursor to define the box. If you draw the selection window from left to right (solid lines), as shown in the following image on the left, only the objects that are fully enclosed in the window will be selected.

- **Crossing window**. If you draw the selection window from right to left (dashed lines), as shown in the following image on the right, a crossing window is used and all the objects that are fully enclosed in the selection window and the objects that are touched by the window will be selected.

- You can use a combination of the methods to create a selection set.

When you select an object, its color changes according to the color style that you are using. To remove all the objects from the selection set, click in a blank section of the graphics window.
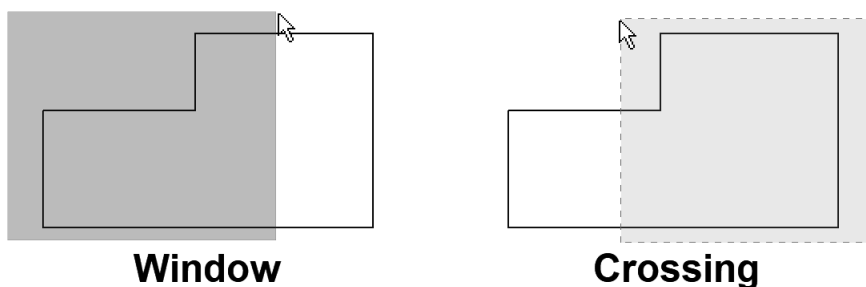

Window   **Crossing**

Figure 2.27

EXERCISE 2-1: CREATING A SKETCH WITH LINES

In this exercise, you create a new part file and 2D Sketch geometry using basic construction techniques. In this exercise, no dimensions will be created.

1   Click New from the Quick Access toolbar, click the English folder, and then double-click Standard (in).ipt or if inch is the default unit; from the left side of the Quick Access toolbar you can click the down arrow on the New icon and click Part.

2   Click Start 2D Sketch from the 3D Model tab > Sketch panel and then select the XY origin plane in the graphics window as shown in the following image.
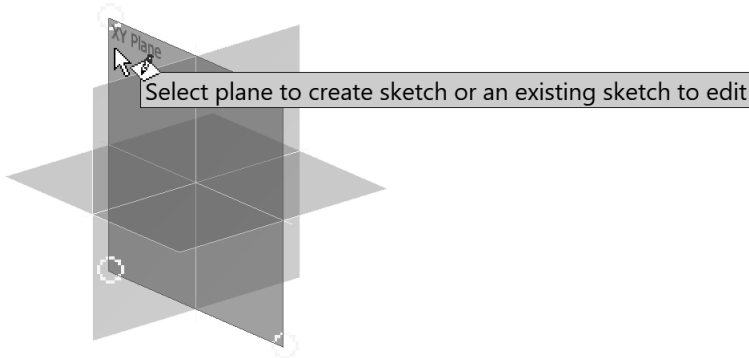
Figure 2.28

3   Start the Line command from the Sketch tab > Create panel.

4   Click on the origin point in the graphics window, move the cursor to the right about **4 inches**, and when the horizontal constraint symbol displays, click to specify a second point as shown in Figure 2.29. You may need to zoom back and pan the screen to see the entire line.
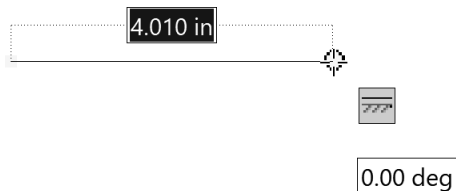

Figure 2.29

> **TIP:** Symbols indicate the geometric constraints. In the image above, the symbol indicates that the line is horizontal. When you create the first entity in a sketch, make it close to final size.

5   Move the cursor up until the perpendicular constraint symbol displays beside the first line and then click to create a perpendicular line that is approximately **2 inches** as shown in Figure 2.30 left.

6   Move the cursor to the left and create a horizontal line perpendicular to the vertical line that is approximately **1 inch**. The perpendicular constraint symbol is shown in Figure 2.30 right.
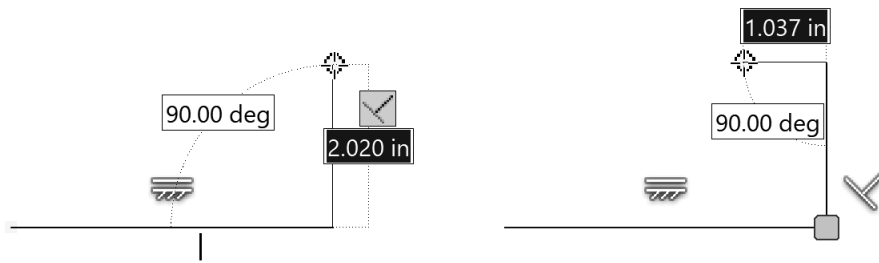

Figure 2.30

7   Move the cursor downward to create a line perpendicular to the top horizontal line and about **1 inch**.

8   Move the cursor left to create a line approximately **2 inches** long and perpendicular to the inside vertical line.

9   Move the cursor up and notice the perpendicular constraint symbol is displayed; to apply a parallel constraint instead, move (scrub) the cursor over the inside vertical line to create a relationship to it. Click when an *inferred line* (horizontal dotted line) appears from the top point as shown in Figure 2.31 left.

10   Move the cursor to the left until an inferred vertical line appears from the bottom-left point as shown in Figure 2.31 right and then click to locate the point.
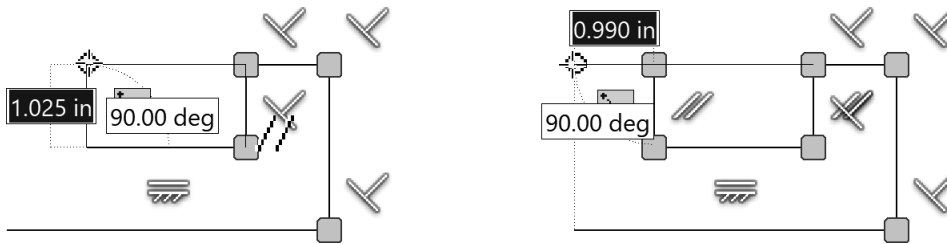

Figure 2.31

11   To close the profile right-click and choose Close from the menu.

12   Figure 2.32 left shows the sketch constraints and the image on the right without the constraints.

> **TIP:** To see the sketch constraints select the geometry using window or crossing window. Constraints will be covered in the next section in this chapter.

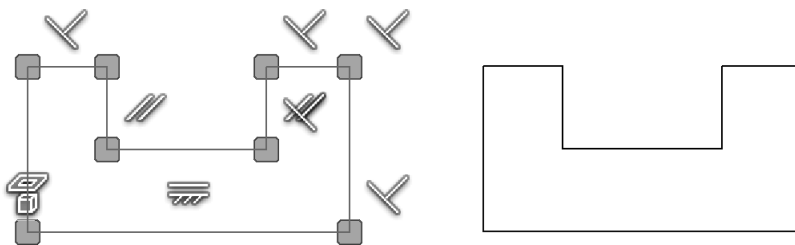**13**   Right-click in the graphics screen, and choose Finish 2D Sketch.

Figure 2.32

14   Close the file. Do not save changes.

End of exercise.

EXERCISE 2-2: CREATING A SKETCH WITH TANGENCIES

In this exercise, you create a new part file, and then you create a profile consisting of lines and tangent arcs.

1 Click the New command, and then double-click Standard (in).ipt; or if inch is the default unit, from the left side of the Quick Access toolbar you can click the down arrow of the New icon, and select Part.

2 Click the Start 2D Sketch command on the 3D Model tab > Sketch panel and then select the XY origin plane.

3 Start the Line command by right-clicking in a blank area in the graphics window and click Create Line from the marking menu.

4 Click on the projected origin point in the middle of the graphics window, and create a horizontal line to the right of the origin point and type **3** (inches will be assumed as the unit because the part file is based on the unit of inch) in the input field. Press the tab key and move the cursor until the horizontal constraint symbol appears and then click. If the second point of the line lies off the screen, roll the mouse wheel away from you to zoom out, hold down the mouse wheel, and drag to pan the view.

5 Create a perpendicular line, move the cursor up until the perpendicular constraint appears, type **1.5** in the input field as shown in the following image on the left, and then press enter.

6 In this step, you infer points, meaning that no sketch constraint is applied. Move the cursor to the intersection of the midpoints of the right-vertical line and bottom horizontal line. Dotted lines (inferred points) appear as shown in the image on the right, and then click to create the line. No dimension was created since a value was not entered.
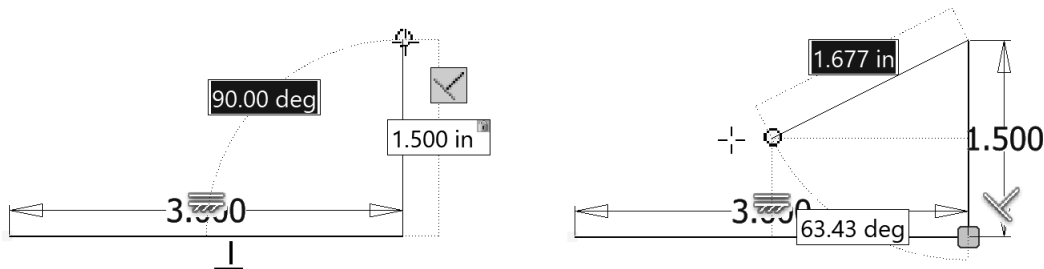

Figure 2.33

7 Next create a line parallel to the bottom line. If needed scrub the bottom line by moving the cursor over the bottom line (do NOT click), and then move the cursor up and to the left until the vertical inferred line and the constraints are displayed as shown in the following image on the left, and then click to create the line.

8 Next sketch an arc while in the line command. With the Line command active move the cursor over the left endpoint of the top horizontal line until the gray circle appears, click on the gray dot at the left end of the line, holding down the left mouse button drag the cursor to the left and then down to preview a tangent arc. Do not release the mouse button.

9 Move the cursor over the left endpoint of the first line segment until a coincident constraint (green circle) and the two tangent constraints at start and end points of the arc are displayed as shown in the following image on the right.
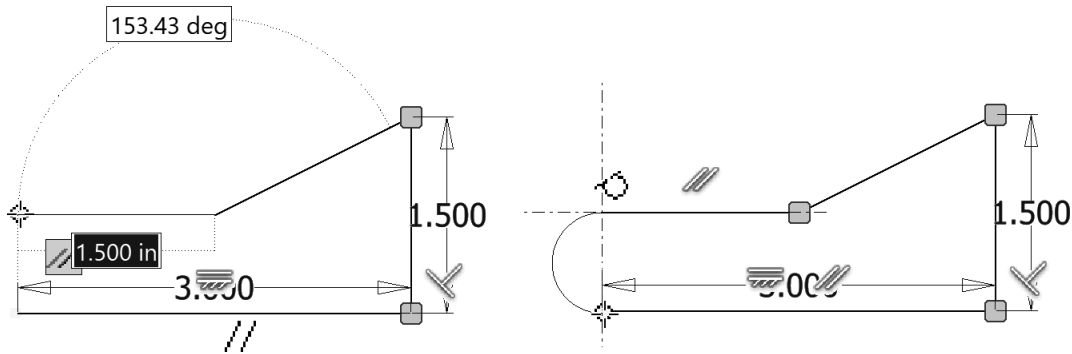
Figure 2.34

10 Release the mouse button to create the arc.

11 Right-click in the graphics window, and choose OK from the marking menu. Later in this chapter you will learn how to create dimensions.

12 Click Finish Sketch from the Sketch tab > Exit panel.

13 Close the file. Do not save changes.

End of exercise.

CONSTRAINING A SKETCH

After you draw a sketch, you may want to add geometric constraints to it to add design intent. Geometric constraints apply behavior to a specific object or create a relationship between two objects. An example of using a constraint is applying a vertical constraint to a line so that it will always be vertical. You can apply a parallel constraint between two lines to make them parallel to one another; then as you change the angle of one of the lines, the angle of the other line will change with it. You can apply a tangent constraint to a line and an arc or to two arcs.

When you add a constraint, the number of constraints or dimensions that are required to fully constrain the sketch will decrease. On the bottom-right corner of Autodesk Inventor, the number of constraints or dimensions will be displayed like what is shown in the following image. A fully constrained sketch is a sketch whose objects cannot move or stretch.

4 dimensions needed 1 1

Figure 2.35

Constrain to the Origin Point

When sketching, it is recommended to constrain a point on the sketch to the origin point with a coincident constraint or dimension a point on the sketch to the origin point so it cannot move. You could apply a fix constraint instead of using the origin point, but it is not recommended. When a sketch is constrained to the origin point, Inventor will change the color of constrained objects. If the sketch is not constrained to the origin point, objects are free to move in the sketch and the color of the objects will not change.

> **TIP:** Autodesk Inventor does not force you to fully constrain a sketch. However, it is recommended that you fully constrain a sketch, as this will allow you to better predict how future changes will affect the sketch and part.

Constraint Types

Autodesk Inventor has 12 geometric constraints that you can apply to sketch geometry. The following image shows the constraint types that can be applied from the Sketch tab > Constrain panel.

Figure 2.36

The following chart describes the geometric constraints.

Icon	Constraint	Function
	Coincident	A point is constrained to lie on another point or curve (line, arc, etc.).
	Collinear	Two selected lines will line up along a single line; if the first line moves, so will the second. The two lines do not have to be touching.
	Concentric	Arcs and/or circles will share the same center point.
	Fix	Applying a fix constraint to a point prevents the constrained point from moving. Multiple points in a sketch can be fixed. If you select a line segment, the angle of the line will be fixed and only its length can change.
	Parallel	Two selected lines will remain parallel to one another.
	Perpendicular	Lines will remain at 90° angles to one another.
	Horizontal	Line is positioned parallel to the X-axis, or a horizontal constraint can be applied between any two points in the sketch. The selected points will be aligned such that a line drawn between them will be parallel to the X-axis.
		Tip: Your keyboard's dash "-" key is a quick shortcut to start the Horizontal Constraint command.
	Vertical	Line is positioned parallel to the Y-axis, or a vertical constraint can be applied between any two points in the sketch. The selected points will be aligned such that a line drawn between them will be parallel to the Y-axis.
	Tangent	An arc, circle, or line will remain tangent to another arc or circle.
	Smooth (G2)	A spline and another spline, line, or arc that connect at an endpoint with a coincident constraint will represent a smooth G2 (continuous curvature) condition.
	Symmetry	Selected points defining the geometry are made symmetric about a selected line.
	Equal	If two arcs or circles are selected, they will have the same radius or diameter. If two lines are selected, they will become the same length. If one of the objects changes, so will the constrained object. If the Equal constraint is applied after one of the arcs, circles, or lines has been dimensioned, the second arc, circle, or line will take on the size of the first one. If you select multiple similar objects (lines, arcs, etc.) before selecting the command, the constraint is applied to all of them.

Adding Constraints

As stated previously, you can apply constraints while you sketch objects. You can also apply additional constraints after the sketch is drawn. However, Inventor will not allow you to **over-constrain** the sketch or add duplicate constraints. If you add a constraint that conflicts with another, you will be warned with the message, "Adding this constraint will over-constrain the sketch." For example, if you try to add a vertical constraint to a line that already has a horizontal constraint, you will be alerted.

To add a constraint, follow these steps:

1 Click a constraint from the Constrain panel, or right-click in the graphics window and choose Create Constraint from the menu and select specific constraint from the menu (see iFigure 2.36.)

2 Click the object or objects then apply the constraint.

Showing Constraints

To display the geometric constraints that are applied to a sketch, do one of the following:

- Select the geometry in the graphics window by selecting individual objects or using a window or crossing selection (see "Selecting Objects" on page 49 if needed.)

- Click the Show Constraints command from the Status Bar as shown in Figure 2.37 left or from the Constrain panel as shown in the middle image.

- Right-click in a blank area in the graphics window and click Show All Constraints from the menu.

- Press the F8 key.

The constraints on the selected geometry will be displayed. The yellow squares represent coincident constraints; move the cursor over a yellow square to display the two coincident constraints for the point. The image on the right shows all the constraints in a sketch.
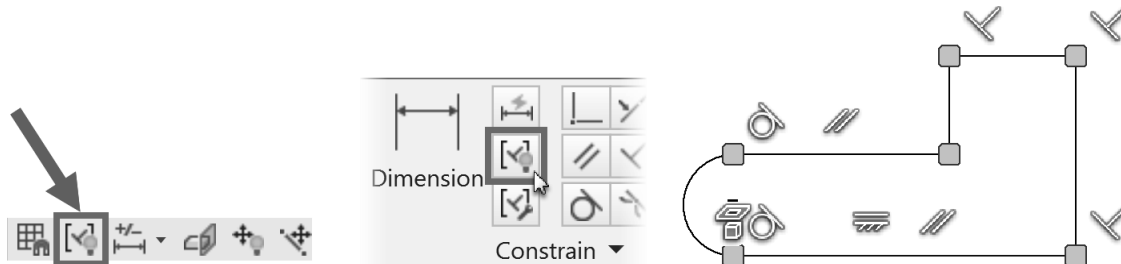

Figure 2.37

Modifying Constraint Size

You can set the Annotation Scale to modify the size of the constraint icons displayed on the screen. Use the Tools tab > Options panel > Application Options, and near the middle right of the General tab modify the Annotation Scale factor.

Figure 2.38 shows the Annotation Scale increased from 1.0 to 1.5. This setting also changes the size of the dimensions in a sketch. This change has no effect on the size of dimensions in a drawing.

Figure 2.38

Deleting Constraints

To delete the constraint(s), select a constraint or multiple constraints using one of the selection methods. Right-click and click Delete from the menu as shown in the following image on the left.

As an alternate method to deleting a constraint, you can press the Delete key once the constraint is selected.

- To delete all constraints except the coincident constraints, select or use the window or crossing selection technique, right-click and click Delete Constraints from the menu as shown in the image on the right.

Figure 2.39

Hiding Constraints

You can hide the display symbol for individual or all geometric constraints. To perform this task, do one of the following:

To hide a constraint:

- Move the cursor over a constraint, right-click and click Hide from the marking menu as shown in the following image on the left.

To hide all constraints, do one of the following:

- Move the cursor over a constraint, right-click and click Hide All Constraints from the marking menu as shown in the image on the left.
- Click Hide All Constraints on the Status Bar as shown in the middle image. This is the same icon you selected to Show All Constraints.
- Right-click in a blank area in the graphics window and click Hide All Constraints on the menu as shown in the following image on the right.
- Press the F9 key.

Figure 2.40

Construction Geometry

Construction geometry can help you create sketches that would otherwise be difficult to constrain. You can constrain and dimension construction geometry like normal geometry, but construction geometry will not be recognized as a profile edge in the part when you turn the sketch into a feature. Normal sketch geometry by default is visible in features created from the sketch.

Construction geometry can reduce the number of constraints and dimensions required to fully constrain a sketch, and can help define the sketch. For example, a construction circle that is tangent to the inside of a hexagon (drawn with individual lines and not the Polygon command) can drive the size of the hexagon. Without construction geometry, the hexagon would require six constraints and dimensions. With construction geometry, it would require only three constraints and dimensions; the circle would have tangent or coincident constraints applied to it and the hexagon.

To create construction geometry change the line style before or after you sketch the geometry using one of the following ways:

- After creating the sketch, select the geometry that you want to change and click the Construction icon on the Format panel as shown in Figure 2.41.
- Before sketching, click the Construction icon on the Format panel. All geometry created will be construction until the Construction command is deselected. Remember to click the Construction icon to turn it off afterward to avoid all you draw being construction geometry.
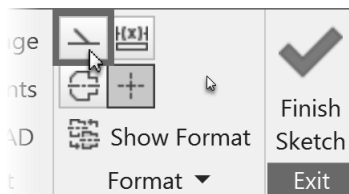
Figure 2.41

After turning the sketch into a feature, the construction geometry will be *consumed* with the sketch and is maintained in the sketch. When you edit a feature's sketch that has construction geometry, the construction geometry will reappear during editing and disappear when the part is updated.

You can add or delete construction geometry in a sketch just like normal sketch geometry can be. In the graphics window, construction geometry displays as a dashed line, lighter colored and thinner than normal geometry.

Figure 2.42 left shows a sketch with an angled construction line. The angled line has a coincident constraint applied to every endpoint that it touches. The image at right shows the sketch after it has been extruded. Notice that the construction line was not extruded.
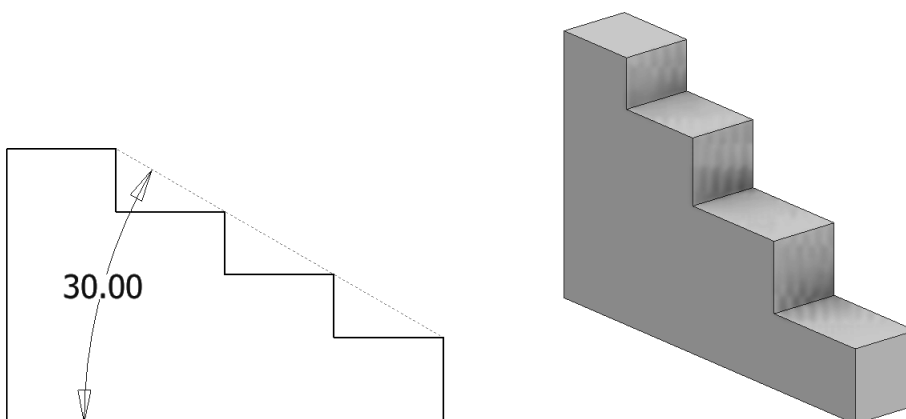

Figure 2.42

Number of Required Constraints or Dimensions

While constraining and dimensioning a sketch, there are multiple ways to determine the number of constraints or dimensions that are required to fully constrain the sketch. When you add a constraint or dimension, the number of constraints or dimensions needed to constrain the sketch decreases. A fully constrained sketch is a sketch whose geometry cannot move or stretch.

The bottom-right of the status bar shows the number of constraints or dimensions needed to fully constrain the sketch (Figure 2.43 left.) When no additional constraints or dimensions are needed to define the sketch, the message Fully Constrained appears at the bottom-right of the status bar (middle image), and in the browser, a pushpin icon appears to the left of the Sketch entry (right.)

7 dimensions needed   Fully Constrained   Sketch:

Figure 2.43

Degrees of Freedom

To see the areas in the sketch that are NOT constrained, you can display the degrees of freedom. While a sketch is active, click Show Degree of Freedom on the status bar, as shown in Figure 2.44 left, or right-click in a blank area of the graphics window and choose Show All Degrees of Freedom from the menu.

Lines and arcs with arrows will appear on your sketch as shown in the middle image. As you add constraints and dimensions to the sketch, degrees of freedom will disappear.

To remove the degree of freedom symbols from the screen, use the status bar to click Hide All Degrees of Freedom, as shown in Figure 2.44 right, or right-click in a blank area in the graphics window and choose Hide All Degrees of Freedom from the menu.
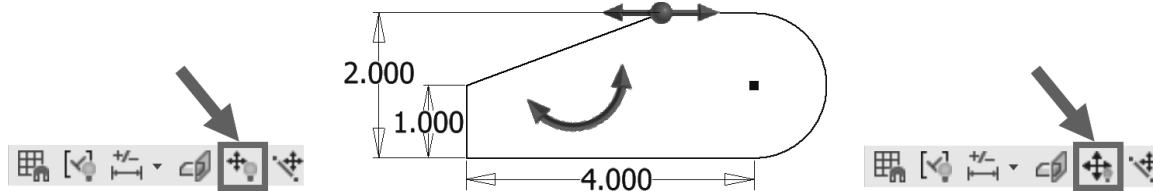
Figure 2.44

Dragging a Sketch

Another method to determine if an object is constrained is to try to drag it to a new location. While not in a command, click a point or an edge, or select multiple objects on the sketch. With the left mouse button depressed, drag it to a new location. If the geometry stretches, it is under constrained.

For example, if you draw a rectangle that has two horizontal and two vertical constraints applied to it and you drag a point on one of the corners, the size of the rectangle will change, but the lines will maintain their horizontal and vertical behaviors. If dimensions are set on the object, they will prevent the object from stretching.

EXERCISE 2-3: ADDING AND DISPLAYING CONSTRAINTS

In this exercise, you add geometric constraints to sketch geometry to control the shape of the sketch.

1   Click the New command, click the English folder, and double-click Standard (in).ipt.

2   Click the Start 2D Sketch command on the 3D Model tab > Sketch panel and then select the XY origin plane.

3   Sketch the geometry as shown in the following image, with an approximate size of **2 inches** in the X (horizontal) direction and **1 inch** in the Y (vertical) direction. Do not apply dimensions dynamically. Place the lower-left corner of the sketch on the origin point. Right-click in the graphics window, and then click OK. By starting the line at the origin point, that point is constrained to the origin with a coincident constraint.

4   Click Show All Constraints on the Status Bar, or press the F8 key. Your screen should resemble the following image.
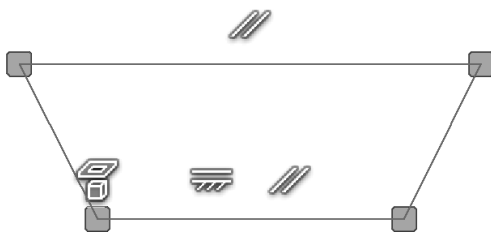
Figure 2.45

5   If another constraint appears, place the cursor over it, right-click, and then click Delete from the marking menu.

6   On the Constrain panel, click the Parallel constraint icon.

7   Select the two angled lines. Depending upon the order in which you sketched the lines, the angles may be opposite of the following image on the left. The constraints that are applied are previewed.

8   Press the ESC key twice to stop adding constraints.

9   The new constraints you just added are not displayed. Press the F8 key to refresh the visible constraints. Your screen should resemble the following image.
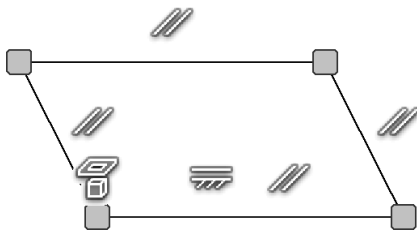
Figure 2.46

10   Select the top horizontal line in the sketch and drag the line. Notice how the sketch changes its size, but not its general shape. Try to drag the bottom horizontal line. The line cannot be dragged as it is constrained.

11   Select the endpoint on the bottom-right horizontal line, and drag the endpoint. The lines remain parallel due to the parallel constraints. Notice on the bottom-right of the Status Bar that 3 dimensions are needed to fully constrain the sketch.

12   On the Sketch tab > Constrain panel, click the Perpendicular constraint icon.

13 Select the bottom horizontal line and the angled line on the right side.

14 Even though it may appear that the rectangle is fully constrained, the height and length of the rectangle still need to be defined with dimensions. Notice on the bottom-right that the Status Bar is down to 2 dimensions to fully constrain the sketch.

15 Place the cursor over the icon for the parallel constraint on the right-vertical line, right-click, and click Delete from the marking menu as shown in the following image on the left. The parallel constraint that was applied to both angled lines is deleted and 3 dimensions would be needed to fully constrain the sketch.
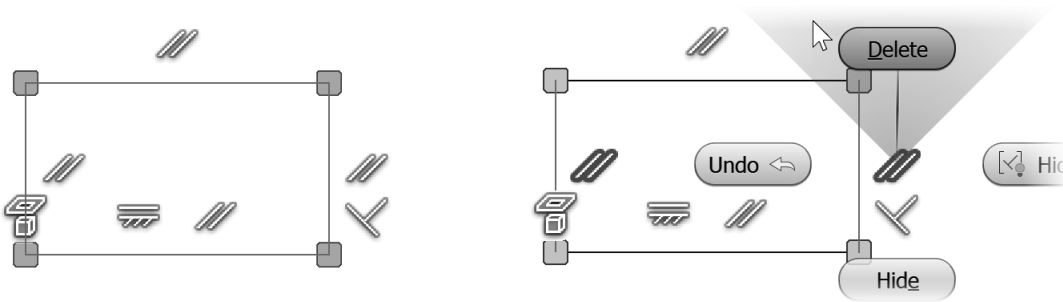
Figure 2.47

16 Click and drag the top-left corner of the rectangle to the left as shown in the following image. Notice that the line on the right does not move because it's perpendicular to the bottom horizontal line.
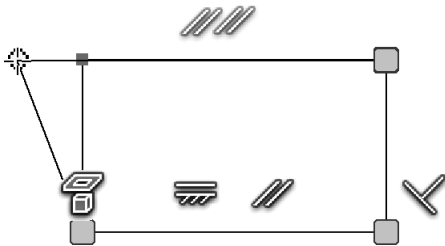
Figure 2.48

17 Click Hide All Constraints on the Status Bar, or press the F9 key.

18 Use a window (drawn left to right) to select the 4 lines and then right-click and choose Delete from the marking menu.

19 Use the Line command to sketch the geometry as shown in the following image with an approximate size of **3 inches** in the X direction and **1.5 inches** in the Y direction. Place the lower-left point of the sketch on the origin (projected center point). Do not apply dimensions dynamically. Right-click in the graphics window, and then click OK.
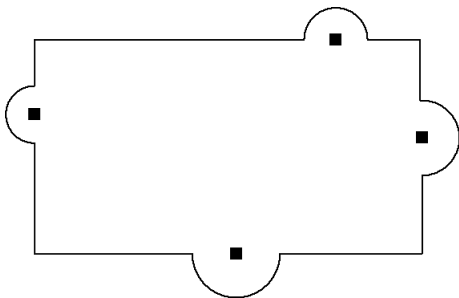
Figure 2.49

20 Inspect the constraints by dragging different points and edges.

21 Next make the arcs equal in size. From the Constrain panel, click the Equal constraint command or press the = key on the keyboard.

22 Select the arc on the left and then the bottom arc.

23 Select the arc on the left and then the arc on the right side.

24 Select the arc on the left and then the arc on the top.

25 Next align the line segments if necessary. From the Constrain panel, click the Collinear constraint command. **Note:** if the endpoints and center point of the arcs are aligned horizontally ⎵⎵⎵ or vertically when they were sketched, you will receive a message "Adding this constraint will over-constrain the sketch." If you see this message click Cancel in the dialog box for steps 22 a. b. c. and d.

    a. Select the two bottom-horizontal lines.

    b. Select the two top-horizontal lines.

    c. Select the two left-vertical lines.

    d. Select the two right-vertical lines.

26 To stop applying the collinear constraint, either right-click and click Cancel (ESC) from the marking menu or press the ESC key.

27 Next you will align the top and bottom arcs vertically. On the Constrain panel, click the Vertical constraint command.

28 Select the center point of the bottom arc and then click the center point of the top arc.

29 Next you will align the left and right arcs horizontally. On the Constrain panel, click the Horizontal constraint command.

30 Click the center point of the left arc, and then click the center point of the right arc.

31 To stop applying the constraints, right-click and click Cancel (ESC) from the marking menu, or press the ESC key.

32 If desired, you can move the arcs by clicking and dragging on them.

33 Display all the constraints by pressing the F8 key. Your screen should resemble the following image.
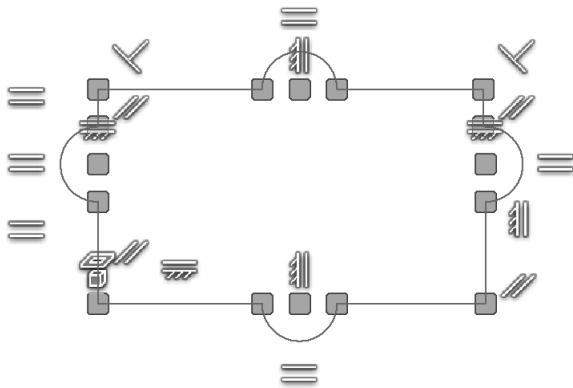

Figure 2.50

34 Hide all the constraints by pressing the F9 key.

35 Click on an endpoint in the sketch and drag the endpoint. Try dragging different points, and notice how the sketch changes.

36 Next delete geometry. as directed:

    a.   Press the ESC key twice to cancel any command.

    b.   Click a point above and to the left of the top arc and drag a window down and to the right so it encompasses the arc on the right as shown in the following image on the left.

    c.   Release the mouse button and press the Delete key on the keyboard.

37 Close the open line segments. Drag the open endpoints onto each other until your sketch resembles the following image on the right. A green circle will appear when the two endpoints are near each other; this applies a coincident constraint.

38 Note, you could have connected the two open endpoints by using the coincident constraint.
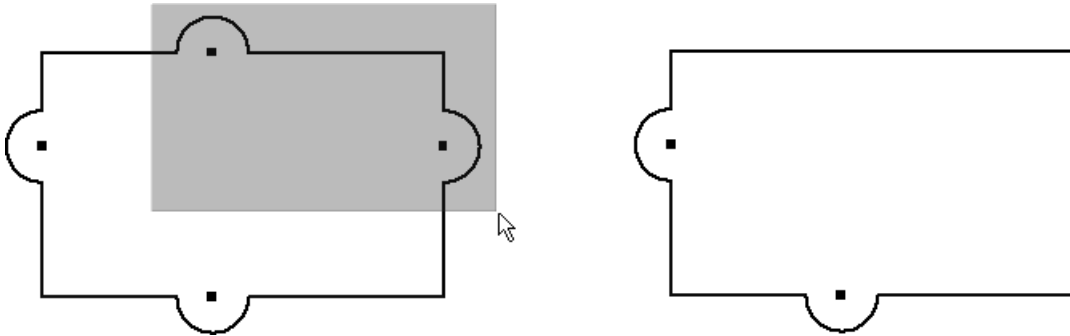
Figure 2.51

39 Next center the arcs in the middle of the sketch. On the Constrain panel, click the Vertical constraint command, click the center point on the bottom arc and then the midpoint of the top horizontal line as shown in the following image on the left.

On the Constrain panel, click the Horizontal constraint command, click the center point on the left arc and the midpoint of the right vertical line as shown in the following image on the right.
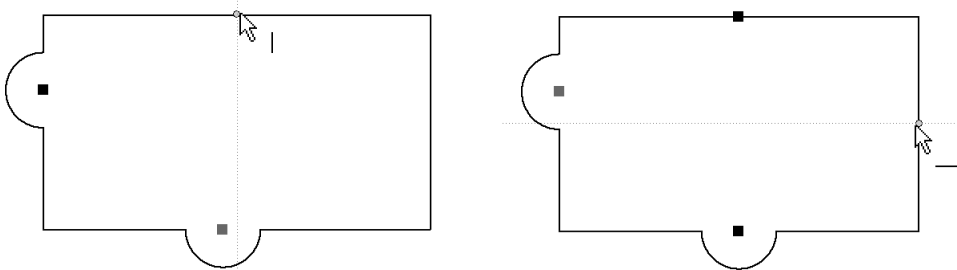
Figure 2.52

40 Click on different points and drag them, notice how the sketch changes shape, but the arcs are always centered as shown in the following image.
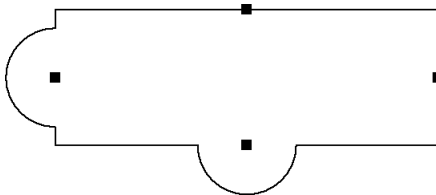
Figure 2.53

41 Close the file. Do not save changes.

Note: Dimensions are needed to fully constrain the sketch. Dimensions are covered in the next section.

End of exercise.

ADDING DIMENSIONS MANUALLY

The last step to constraining a sketch is to add dimensions that were not added dynamically. The dimensions you place will control the size of the sketch and can also appear in the part drawing views when they are generated. When placing dimensions, try to avoid having extension lines go through the sketch, as this will require more cleanup when drawing views are generated. Click near the side from which you anticipate the dimensions will originate in the drawing views.

All dimensions that you create are parametric as well as the dynamic dimensions that are placed automatically when sketching geometry. Parametric means that they will change the size of the geometry.

Scale Sketch

If the sketch is not constrained to the origin point and no dimension was dynamically added to the sketch when it was created, then the entire sketch will be uniformly scaled when the first dimension is added.

General Dimensioning

The General Dimension command can create linear, angle, radial, or diameter dimensions one at a time. The following image on the left shows an example of a dimensioned sketch. To start the General Dimension command, follow one of these techniques:

- Click the General Dimension command from the Sketch tab > Constrain panel as shown in the following image in the middle.
- Right-click in the graphics window and click General Dimension from the marking menu as shown in the image on the right.
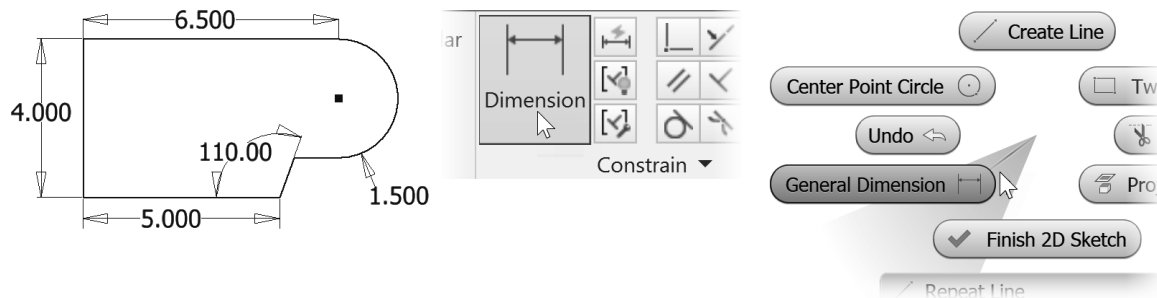- Press the shortcut key D.

Figure 2.54

When you place a linear dimension, the extension line of the dimension will snap automatically to the nearest endpoint of a selected line; when an arc or circle is selected, it will snap to its center point. To dimension to a tangent point of an arc or circle, see "Dimensioning to a Tangent of an Arc or Circle" later in this chapter.

 After you select the General Dimension command, follow these steps to place a dimension:

1. Select the geometry to be dimensioned.

2. After selecting the geometry, a preview image will appear attached to your cursor showing the type of dimension. If the dimension type is not what you want, right-click, and then select the correct style from the menu. After changing the dimension type, the dimension preview will change to reflect the new style.

3. Click to place the dimension.

4. Enter a value for the dimension.

The next sections cover how to dimension specific objects and how to create specific types of dimensioning with the Dimension command.

Dimensioning Lines

There are multiple techniques for dimensioning a line. Use the Dimension command and do one of the following:

- Click near two endpoints, move the cursor until the dimension is in the correct location, and click.
- To dimension the length of a line, click anywhere on the line; the two endpoints will be selected automatically. Move the cursor until the dimension is in the correct location and click.
- To dimension between two parallel lines, click one line and then the next, and then click a point to locate the dimension.
- To create a dimension whose extension lines are perpendicular to the line being dimensioned, click the line and then right-click. Click Aligned from the menu, and then click a point to place the dimension.

Dimensioning Angles

To create an angular dimension, use the General Dimension command, click two lines whose angle you want to define, move the cursor until the dimension is in the correct location. Place the dimension by clicking on a point.

Dimensioning Arcs and Circles

To dimension an arc or circle: start the General Dimension command, click on the circle's circumference, move the cursor until the dimension is in the correct location, and click. By default, when you dimension a circle, the default is a diameter dimension; when you dimension an arc, the result is a radius dimension. To change a radial dimension to diameter or a diameter to radial, right-click before you place the dimension and select the style from the Dimension Type menu.

To dimension the angle of the arc: start the Dimension command, click the arc's circumference, click the arc's center point, and then place the dimension. You can also click the arc's center point and then its circumference.

You can also add an arc length dimension this way. Start the dimension command, then click on the arc, right-click and choose Arc Length from the Dimension Type menu, then click a point to locate the dimension.
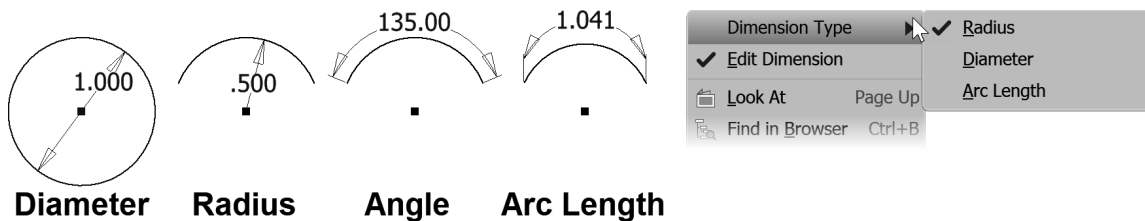

Diameter   Radius   Angle   Arc Length

Figure 2.55

Dimensioning to a Tangent of an Arc or Circle

To dimension to a tangent of an arc or circle, follow these steps:

1. Start the General Dimension command.
2. Select a line that is parallel to the tangent arc or circle that will be dimensioned, labeled (1) in the following image on the left.
3. Move the cursor over the arc or circle until the tangent constraint symbol appears, labeled (2) in the following image on the left.
4. Then move the cursor until the dimension is in the correct location and click to create the dimension, labeled (3) in the following image on the right.
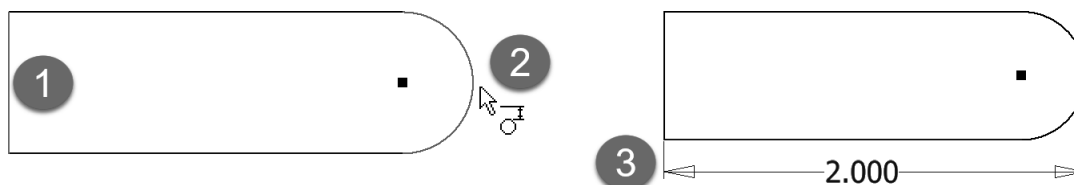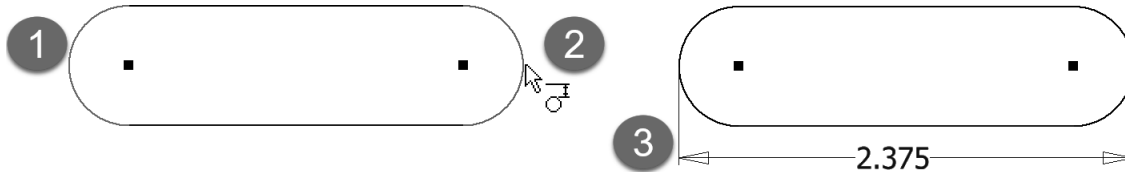

Figure 2.56

To dimension to two tangents, follow these steps:

1. Start the General Dimension command.

2. Select an arc or circle that includes one of the tangents to which it will be dimensioned. The following image illustrates an example of dimensioning a slot; the first selection is labeled (1).

3. Move the cursor over a second arc or circle until the tangent constraint symbol appears, as shown in the following image on the left, labeled (2).

4. Click to select the tangent point and then move the cursor until the dimension is in the correct location. Then click to create the dimension, labeled (3) in the following image on the right.

Figure 2.57

Entering and Editing a Dimension Value

After placing the dimension, you can change its value. Depending on your settings, the Edit Dimension dialog box may or may not appear automatically after you place the dimension. The Edit Dimension options can be set by any one of the following:

- Click the Tools tab > Options panel > Application Options. On the Sketch tab of the Application Options dialog box, from the Constrain Settings area, click the Settings button and then click the box next to "Edit dimension when created" as shown in Figure 2.58 left.

- Set the option by right-clicking in the graphics window while placing a dimension and click Edit Dimension from the menu as shown in Figure 2.58 right. This method will change the application option "Edit Dimension when created" as previously described.

If "Edit dimension when created" is checked, the Edit Dimension dialog box appears automatically after you place the dimension. Otherwise, the dimension will be placed with the default value.
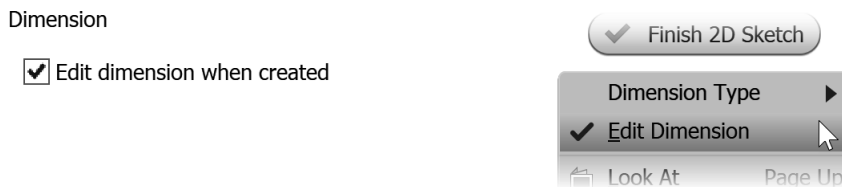

Figure 2.58

To edit a dimension that has already been created, double-click on the dimension, and the Edit Dimension dialog box will appear, as shown in Figure 2.59.

Enter the new value and unit for the dimension; then either press ENTER or click the checkmark in the Edit Dimension dialog box. If no unit is entered, the units that the file was created with will be used. Enter the exact value — do not round up or down. The accuracy of the dimension that is displayed in a sketch is set in the Document Setting. For example, if you want to enter 4 1/16 as a decimal, enter 4.0625 in not 4.06 in. You can set how many decimal places to display on the drawing later, but create your model with as much accuracy as possible.
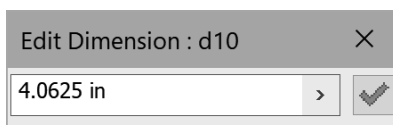

Figure 2.59

Fractions

Inventor allows you to enter a fraction when such a value is required. With the Length unit in the Tools tab > Options panel > Document Settings > Units tab set to any non-metric unit, as shown in Figure 2.60 left, when a fraction is entered, it will display as such in the graphics window and in the Edit Dimension dialog box. If the Length unit is set to a metric unit and a fraction is entered, the decimal equivalent will be displayed in the graphics window but the fraction will be shown in the Edit Dimension dialog box.

After entering a fraction, you can click on the right-faced arrow and set the type of dimension to display: Decimal, Fractional, or Architectural as shown in the middle of the following image.

When entering fractions do not use a dash to separate the fraction, just add a space. For example, enter 4 1/16, *not* 4-1/16, because Inventor interprets the — as part of an equation and would return the value 3.9375. Figure 2.60 right shows the fraction displayed in the graphics window.
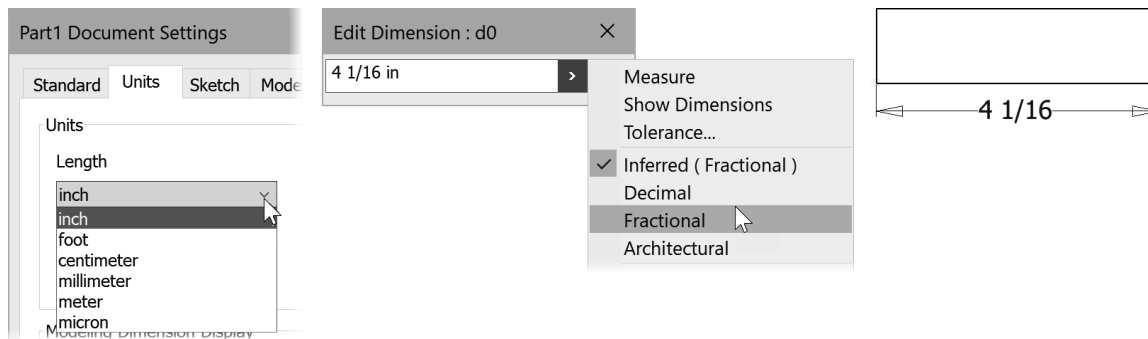
Figure 2.60

TIP: When placing dimensions, it is recommended that you place the smallest dimensions first. This will help prevent the geometry from flipping in the opposite direction.

Repositioning a Dimension

Once you place a dimension, you can reposition it, but the origin points cannot be moved. Follow these steps to reposition a dimension:

1. Exit the current operation either by pressing ESC twice or right-clicking and then clicking Cancel (ESC) from the marking menu.

2. Move the cursor over the dimension until the move symbol ⊞ appears as shown in Figure 2.61.

3. With the left mouse button depressed, move the dimension to a new location and release the button.
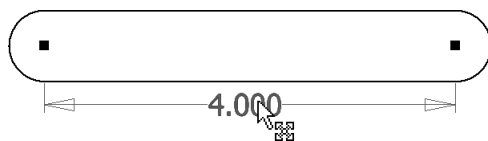

Figure 2.61

Fully Constrained Sketch

As described in the "Constraining the Sketch" section, as you add constraints and dimensions to a sketch, the number of required dimensions decreases. When no more constraints or dimensions are needed to constrain the sketch, instead of a number in "dimensions needed" on the bottom-right of the status bar Fully Constrained will show as in Figure 2.62 left. The icon to the left of the Sketch entry in the browser displays a push-pin when the sketch is fully constrained as shown in Figure 2.62 right.
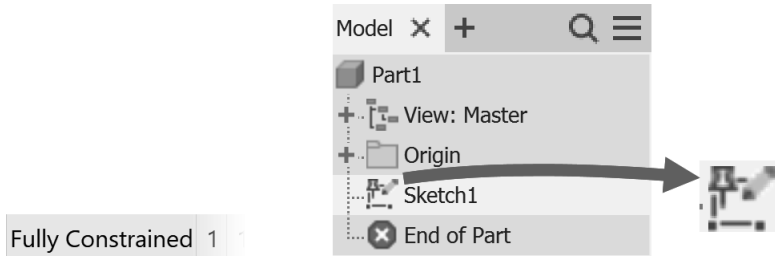
Figure 2.62

Over Constrained Sketch

As explained in the "Adding Constraints" section, Inventor does not allow over-constrained sketches or dupli-cate constraints. The same is true when adding dimensions. If you add a dimension that conflicts with another constraint or dimension, you will be warned that this dimension will over-constrain the sketch or that it already exists. You can either cancel the operation and no dimension will be placed, or accept the warning and a *driven* dimension will be created.

A *driven dimension* is a *reference* dimension. It is not a parametric dimension—it reflects the size of the points to which it is dimensioned. If the part changes, the driven dimension updates to show the new value. A driven dimension will appear with parentheses around the dimension's value—for example, (2.500). When you place a dimension that will over-constrain a sketch, a dialog box will appear like the following image.
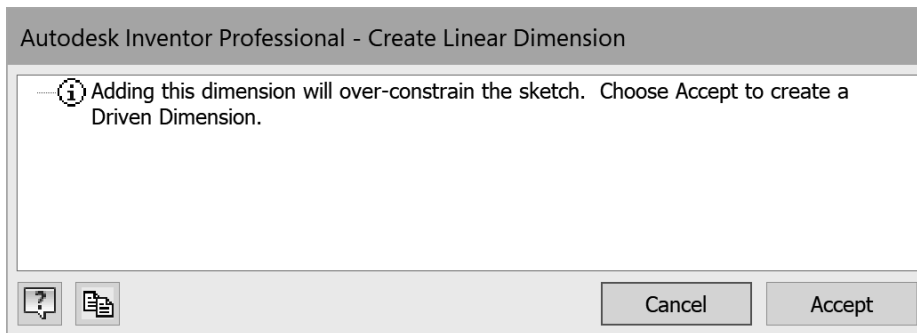
Figure 2.63

Relax Mode

When you try to place a constraint or add a dimension and receive the over-constrained message, if you want the constraint or dimension to take precedence, you can turn on relax mode. With relax mode on, when you reapply the constraint or add the dimension, the conflicting constraint will automatically be deleted, except for Coincident, Smooth, Tangent, Symmetry, Pattern, and Project constraints. A conflicting dimension will become a driven dimension. While in relax mode, if you are unable to add the new constraint or dimension, you may need to manually delete one of the Coincident, Smooth, Tangent, Symmetry, Pattern, and Project constraints.

Turn on relax mode by clicking the Relax Mode icon on the Status Bar as shown in Figure 2.64 left. You can also click Constrain Settings from the Sketch tab > Constrain panel, then use the Relax Mode tab to check Enable Relax Mode as shown in Figure 2.64 middle. When you apply a constraint or dimension that would over-constrain a sketch, a dialog box appears stating a constraint or dimension will be deleted to solve the con-flict as shown in Figure 2.64 right.

Another method to remove conflicting constraints or dimensions is to drag a point or edge while in Relax Mode and conflicting constraints or dimensions will be deleted.

TIP: When done constraining a sketch, turn off Relax Mode to return to normal editing. If you don't turn it off, sketch points and objects can be changed by accidentally dragging them.
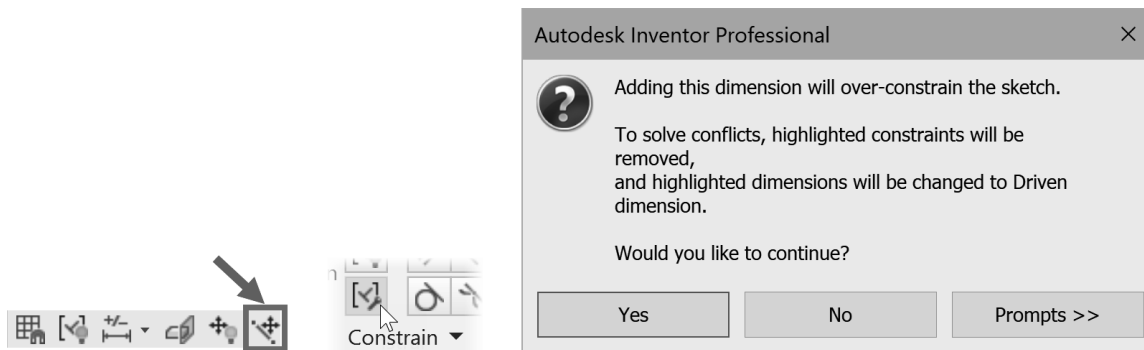
Figure 2.64

Figure 2.65 left shows a fully constrained rectangle with geometric constraints visible. With Relax Mode on a 120-degree angle dimension was placed. Clicking Yes in the warning dialog box caused the dimension to be created, and the perpendicular constraint in the lower-right corner deleted as shown in the middle image. The image at right shows the sketch after a perpendicular constraint was applied to the bottom-horizontal and left-vertical lines. Notice the 120-degree dimension was maintained.
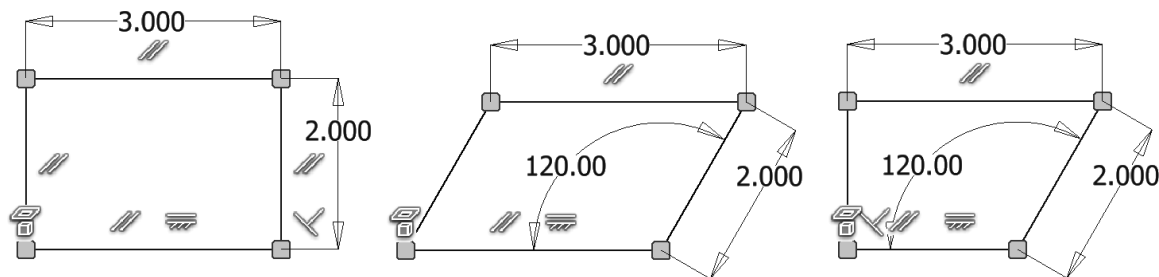
Figure 2.65

Driven Dimension

Driven dimensions can also be created using the Driven Dimension option. A driven dimension does NOT reduce the dimensions needed to constrain the sketch; it only reports the length of the object. You use this option to show reference dimensions.

Select Driven Dimension from the Sketch tab > Format panel, as shown in Figure 2.66 left. With the Driven Dimension icon highlighted, use the Dimension command as usual to create a driven dimension. (Don't forget to turn it off when you are finished.) Driven dimensions are represented in the sketch with parentheses around the value; parametric dimensions do not use parentheses. Without the Driven Dimension option active, regular parametric dimensions are created, which is the default.

The Driven Dimension option can also be used to change an existing dimension to either a driven or back to normal by selecting it and clicking Driven Dimension. Figure 2.66 right shows an example of a 5.250 driven dimension referencing the overall length of the sketch. The three parametric dimensions control the length of the sketch. If a parametric dimension value changes, the driven dimension updates to reflect the change in overall length.
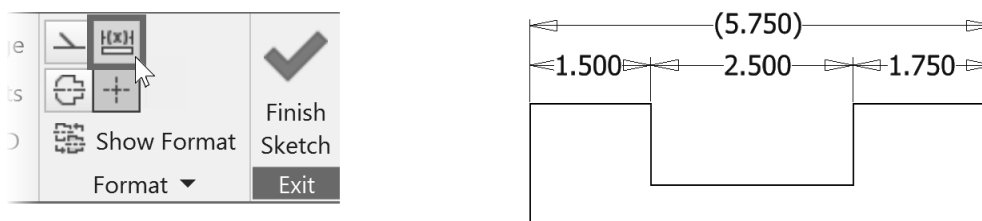
Figure 2.66

TIP: Avoid over using driven dimensions as they do not parametrically control the size of the sketch. They are only used for reference.

EXERCISE 2-4: CONSTRAINING AND DIMENSIONING A SKETCH

In this exercise, you add dimensional constraints to a sketch. Note: this exercise assumes that the "Edit dimension when created" and "Autoproject part origin on sketch create" options are checked in the Application Options dialog box under the Sketch tab. Experiment with Autodesk Inventor's color schemes to see how the sketch objects change color when they are constrained.

1   Click New, choose the English folder. Double-click Standard (in).ipt to select it.

2   Click Start 2D Sketch from the 3D Model tab > Sketch panel. Select the XY origin sketch plane.

3   Use the Line command from the Sketch tab > Create panel. Draw a line starting from the origin point, then move the cursor to the right, type **5** in the distance input. Press Tab and click a point when the horizontal constraint shows below the degrees input field as shown in Figure 2.67 left.

4   Next place an angled line and a dynamic dimension to define the angle. Press Tab and type **150** for the angle input, press the Tab key, and then click a point to the upper right as shown in Figure 2.67 right. The distance should be about 2 inches but the dimension is not needed to define this sketch.
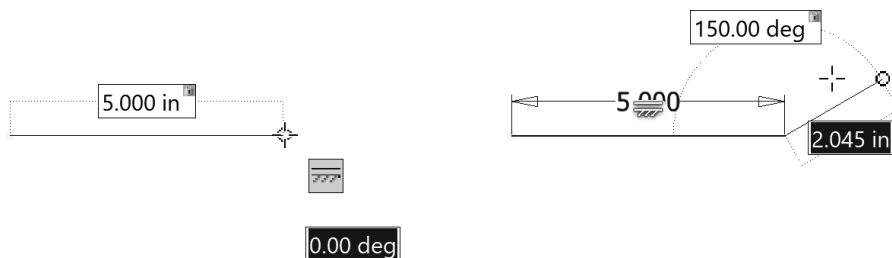

Figure 2.67

5   Sketch the geometry shown in the following image. When sketching, ensure that a perpendicular constraint is **not** applied between the two angled lines. If needed, hold down CTRL while sketching the top angled line to prevent the sketch constraint being applied. Make the arc tangent to both adjacent lines.
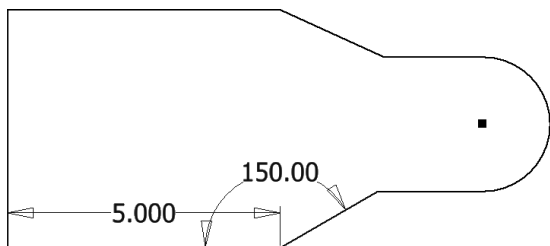

Figure 2.68

6   Horizontal Constraint command from the Sketch tab > Constrain panel to add a horizontal constraint between the midpoint of the left vertical line and the center of the arc as shown in Figure 2.69 left.

7   Use Vertical Constraint from the Sketch tab > Constrain panel and add a vertical constraint between the endpoints of the angled lines nearest to the right side of the sketch as shown in Figure 2.69 right.
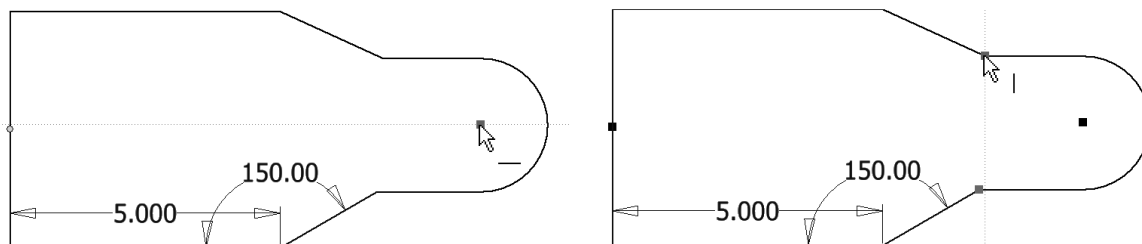

Figure 2.69

8   Press the = key and select the two angled lines to add an equal constraint to them.

9   Click General Dimension from the Sketch tab > Constrain panel and add a radial dimension to the arc. Move the cursor to position the dimension near the lower right corner of the sketch, then click a point and enter the value **1.5.** If the Edit Dimension dialog doesn't appear, double-click the dimension and change it to 1.5. Click the checkmark to finish.

> **TIP:** To show the Edit Dimension dialog box during dimension placement, right-click in the graphics window and click Edit Dimension from the menu.

10   While still in the General Dimension command, add a vertical dimension to the vertical line, click a point to locate the dimension to its left, enter **5**, and click the checkmark. When complete, your sketch should resemble the following image. Notice on the Status Bar bottom right, 1 dimension is required to fully constrain the sketch.
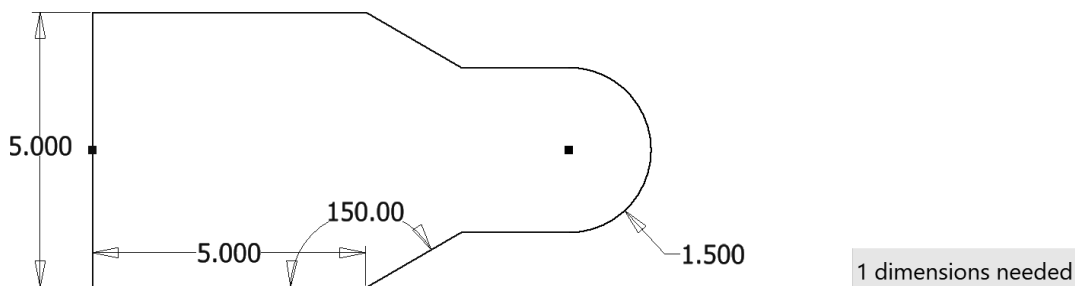

Figure 2.70

11   Add a horizontal dimension by selecting the left vertical line and the center point of the arc (or on top of the arc). Locate the dimension above the sketch. Enter **10** for the value as shown in Figure 2.71.
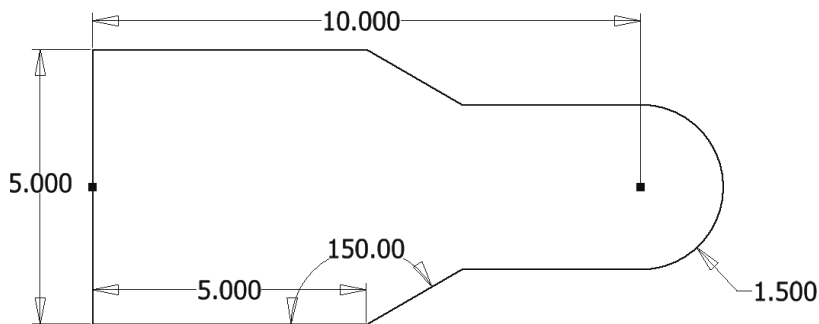

Figure 2.71

12   Press ESC twice to end the command. The Status Bar dimensions required section should display "Fully Constrained." In the browser, the icon to the left of the sketch should show a pushpin.

13   Try clicking and dragging different points of the sketch. The points do not change as it is fully constrained and dimensioned.

14   Click the Relax Mode icon in the Status Bar to turn it on as shown in Figure 2.72.

Figure 2.72

15 Click and drag different points on the sketch. The dimensions will change to reflect their new value.

16 Use Undo from the Quick Access toolbar to return the sketch to the values shown in Figure 2.71.

17 While still in Relax Mode, use General Dimension to add an overall horizontal dimension by clicking the vertical line (not an endpoint) and moving the cursor near the right tangent point of the arc until the *glyph* of a dimension with a circle appears ⌀, as shown in Figure 2.73 left.

- Locate the dimension above the 10.000 dimension.
- Click Yes to allow the highlighted dimension to be changed to a driven dimension.
- Accept the default length of **11.500** by clicking the green check mark in the Edit Dimension dialog.
- Notice that a perpendicular constraint was removed,

18 Select the 10.000 dimension and toggle it to a driven dimension by clicking Driven Dimension from the Sketch Tab > Format panel.

19 Add a perpendicular constraint between the left vertical line and the top horizontal line. Your sketch should resemble Figure 2.73 right.

Note that another option is to change the 10.000 dimension to a driven dimension and then add the 11.500 dimension.
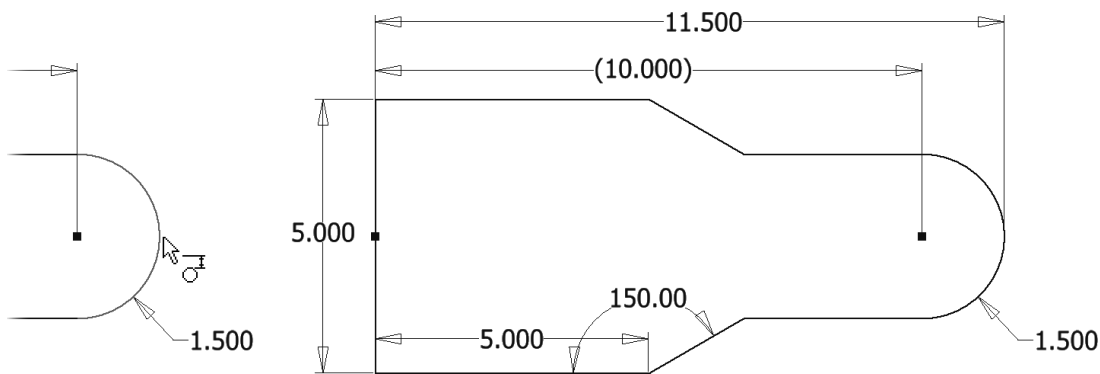

Figure 2.73

20 Click Relax Mode from the Status Bar to turn it off as shown in Figure 2.74.

Figure 2.74

21 Edit some sketch dimensions by double-clicking on the dimension's value and typing in a new one. Press ENTER on the keyboard or click the green check mark in the Edit Dimension dialog box when done. Examine how the sketch changes. The arc should remain in the middle of the vertical line. Notice how the driven dimension changes when the horizontal, angle or radial dimension values change.

22 Delete the horizontal constraint between the center of the arc and the midpoint of the vertical line by selecting the center point of the arc, select the horizontal constraint above the center point, right-click and click Delete from the menu as shown in the following image.
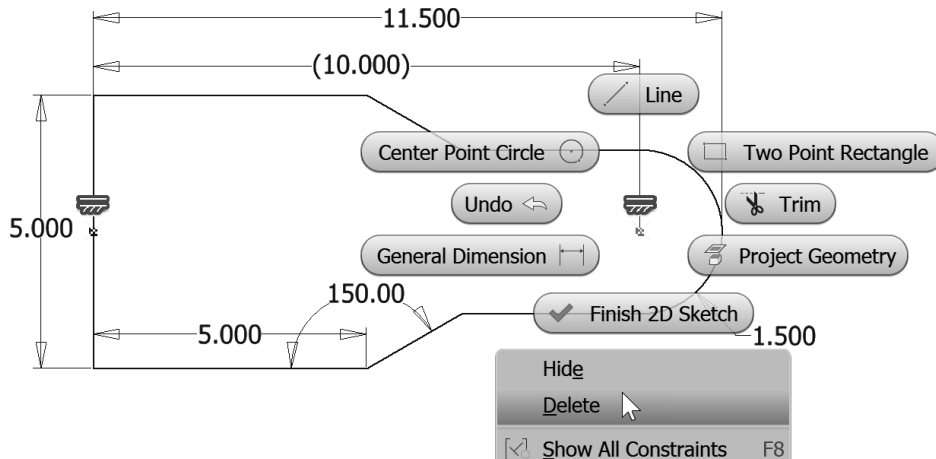

Figure 2.75

23 Prove the arc can move by clicking and dragging on the center point of the arc.

24 Display the visibility of all the constraints by using window to select all of the geometry.

25 Click in an empty area of the graphics window so the constraints disappear.

26 From the Status Bar, click Show All Constraints, as shown in Figure 2.76.

Figure 2.76

27 Practice deleting and adding other constraints.

28 Practice adding and deleting dimensions.

29 Close the file. Do not save changes.

End of exercise.

MEASURE COMMAND

The measure command assists in analyzing sketch, part, and assembly models by measuring position, length, angle, radius, diameter, perimeter, and area.

The Measure command is not a replacement for dimensions; it is an additional tool to give you more information. You can measure distances, angles, and loops, and you can perform area calculations. You can start the measure command first and then select the geometry, or select the geometry and then start a measure command.

The Measure command is located on the Inspect tab > Measure panel as shown in Figure 2.77 left. If desired, you can add it to the Quick Access toolbar by clicking the down arrow to the right of the Quick Access toolbar and choosing Measure from the menu as shown in Figure 2.77 middle image.

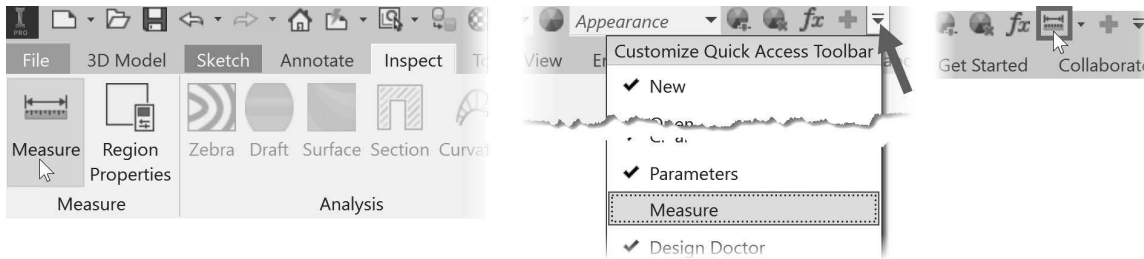Once the Measure command is added you can access it as shown in Figure 2.77 right.

Figure 2.77

Position and Distance Between Two Points

Select a point and the X, Y, Z position data will be displayed in the Measure tool panel as shown in the following image on the left. Select two points and the position data of both points will be displayed as well as the distance between two points as shown in the following image on the right.
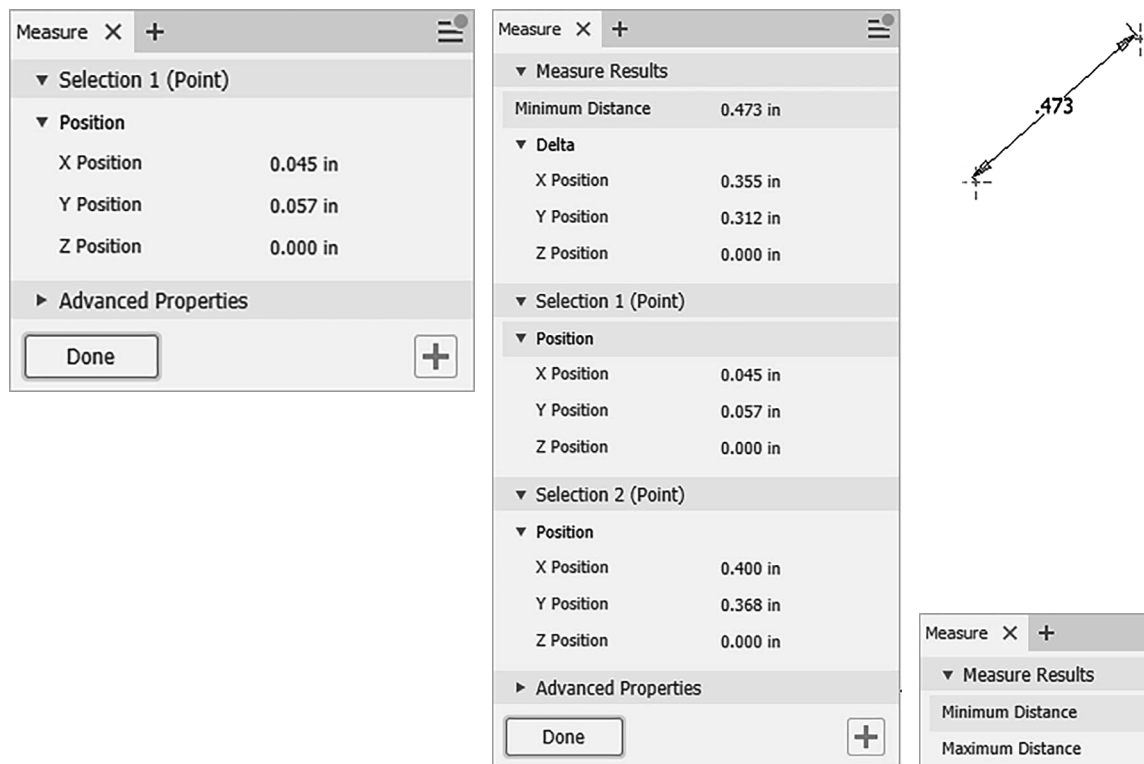
Figure 2.78

Length, Radius, Diameter

Measures the distance for an edge or line, diameter or radius of a cylindrical face or circle, distance between two points, and distance between two components in an assembly (covered in Chapter 6 Assemblies).

Angle

Measures the angle by selecting two edges, two lines, an arc, select two points and then hold down the Shift key and select a third point, or select a line or an edge and a planar face.

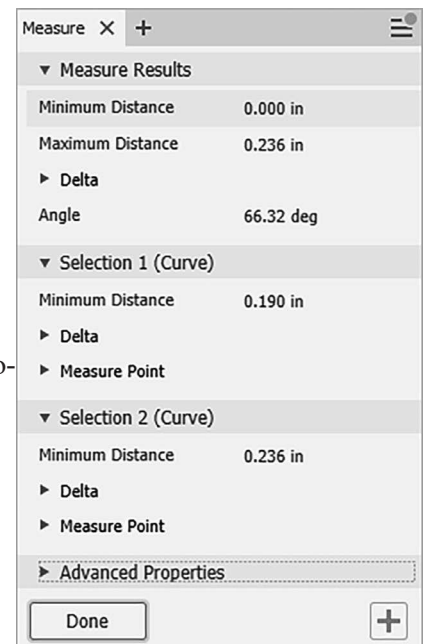Figure 2.79 shows the results after measuring two edges.

Figure 2.79

Loop

Start the Measure command and then move the cursor over an edge and let the cursor stay still for two seconds and then click Curve Loop from the toolbar as shown in the following image on the left. The loop distance will appear in the Measure property panel as shown in the following image on the right.
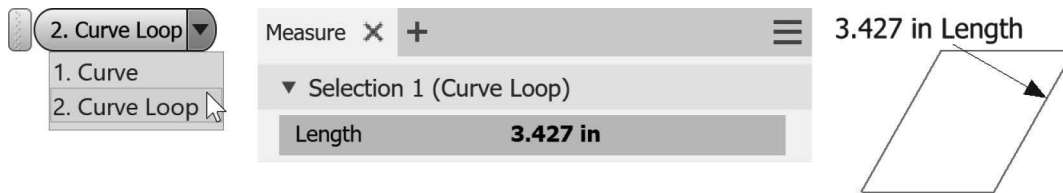
Figure 2.80

Measure Perimeter and Area

Measure the area by selecting inside an enclosed face of a 3D solid or surface. To get area data of a closed 2D Profile, you can also use the Region Properties command that is covered next.

Figure 2.81

TIPS: Copy measured data to the clipboard by right-clicking in the cell containing the data and choosing Copy from the menu. ■ To restart measure selection, click anywhere in the graphics window.

Set Precision and Dual Units

To set the precision and display the results in dual units, click the right facing arrow to the left of the Advanced Settings area as shown in Figure 2.82.
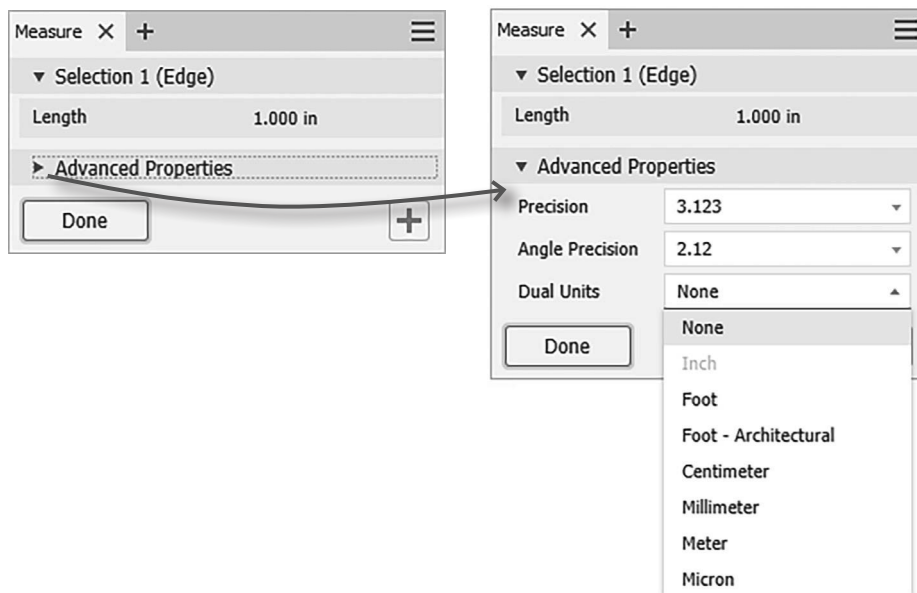
Figure 2.82

REGION PROPERTIES

The Region Properties command located on the Inspect tab > Measure panel can be used to determine the area, perimeter, and moments of inertia of a closed 2D sketch while in Sketch mode (Figure 2.83.)
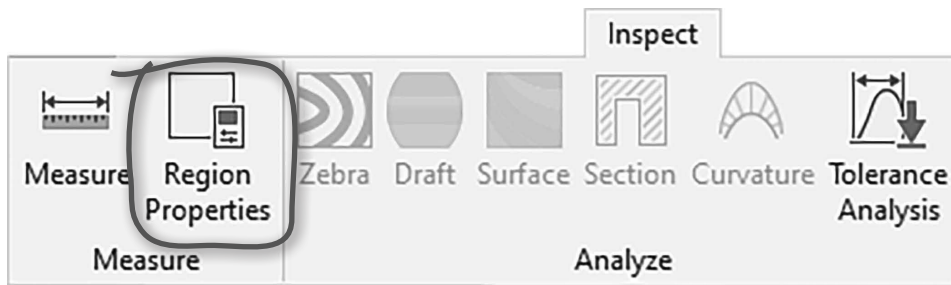
Figure 2.83

Measurements are taken from the sketch coordinate system (0,0). After selecting inside a closed region, click the Calculate button. The properties can be displayed in the document's default units or dual units of your choice. The following image shows the region properties of a closed profile that has a void consisting of a slot.
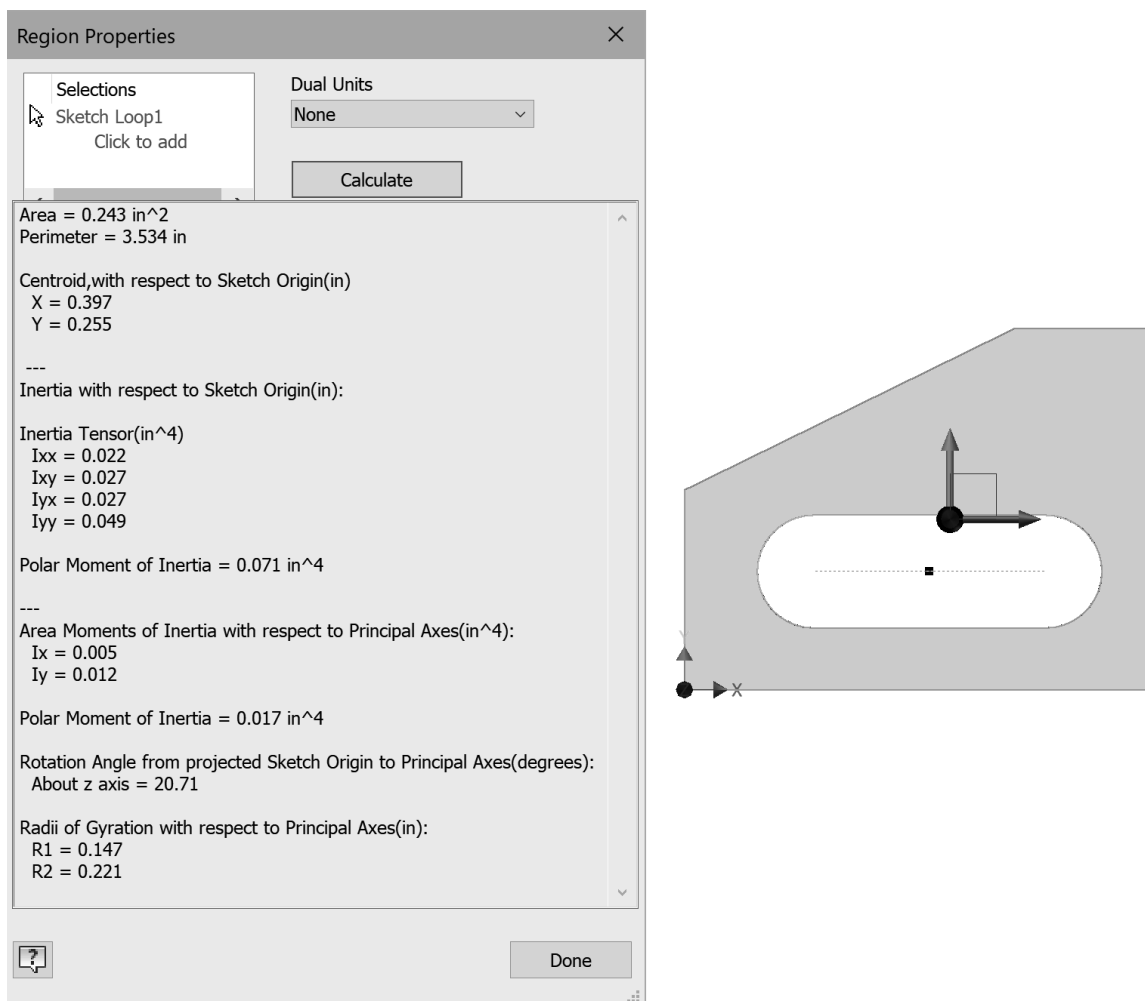
Region Properties

Selections
Sketch Loop1
Click to add

Dual Units
None

Calculate

Area = 0.243 in^2
Perimeter = 3.534 in

Centroid,with respect to Sketch Origin(in)
  X = 0.397
  Y = 0.255

Inertia with respect to Sketch Origin(in):

Inertia Tensor(in^4)
  Ixx = 0.022
  Ixy = 0.027
  Iyx = 0.027
  Iyy = 0.049

Polar Moment of Inertia = 0.071 in^4

Area Moments of Inertia with respect to Principal Axes(in^4):
  Ix = 0.005
  Iy = 0.012

Polar Moment of Inertia = 0.017 in^4

Rotation Angle from projected Sketch Origin to Principal Axes(degrees):
  About z axis = 20.71

Radii of Gyration with respect to Principal Axes(in):
  R1 = 0.147
  R2 = 0.221

Done

Figure 2.84

INSERTING AN AUTOCAD FILE

You may have legacy AutoCAD files or receive AutoCAD files that you need to convert into Inventor parts. In this section, you learn how to insert AutoCAD data into a sketch. When importing a 2D DWG file into Autodesk Inventor, you can either copy the contents from the DWG file to the clipboard via Autodesk Inventor or AutoCAD and paste the contents into Autodesk Inventor, or use an import wizard that guides you through the process. In this section, you learn how to insert AutoCAD 2D data into a sketch in an Inventor part file.

> **TIP:** AutoCAD does not need to be installed to import AutoCAD geometry into Autodesk Inventor.

Inserting 2D AutoCAD Data into a Sketch

This section describes how to insert AutoCAD 2D data into the active sketch in a part or drawing. To insert AutoCAD data into the active sketch, follow these steps:

1. Start a new Inventor part file or open an existing part file.

2. Create a new sketch or make an existing sketch active.

3. Click the Insert AutoCAD File command on the Sketch tab > Insert panel as shown in the following image.
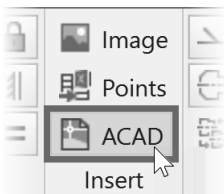

Figure 2.85

4. The Open dialog box will appear. Browse to and either double-click the desired DWG file or click on the DWG file and then click Open.
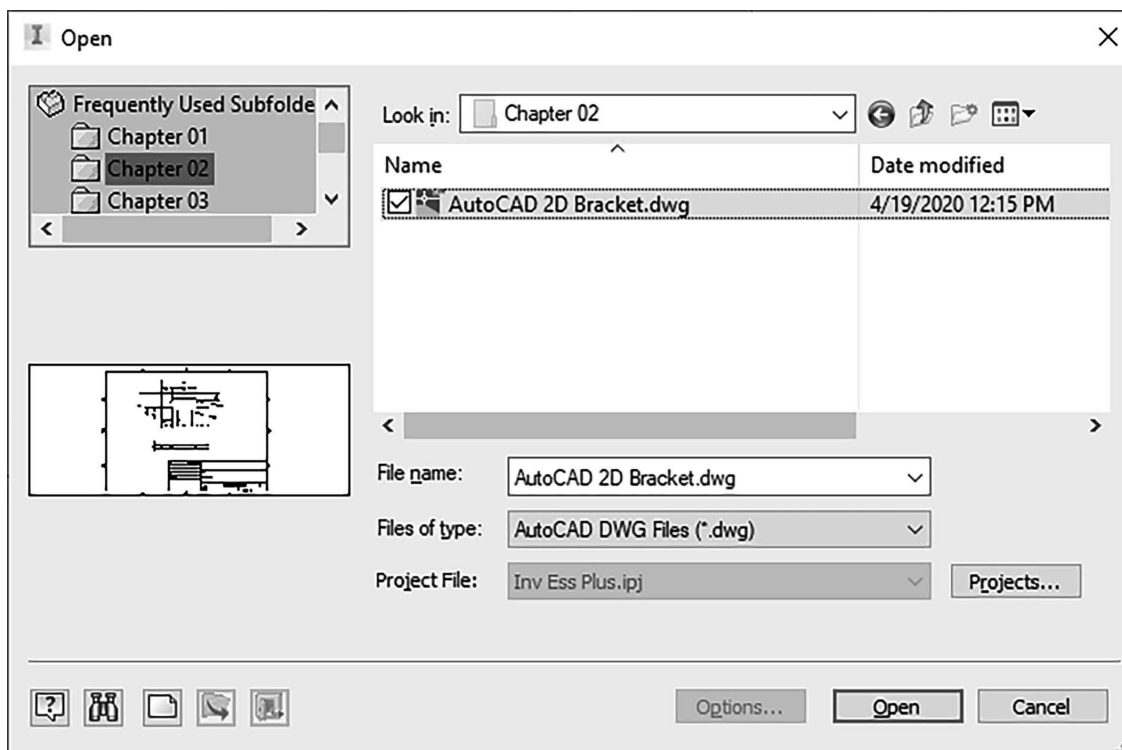

Figure 2.86

5. The Layers and Objects Import Options dialog box appears. In the Selective import section in the upper-left corner of the dialog box, uncheck the layer names you do not want data imported from as shown in the following image.

6. To select specific objects to insert, uncheck the All option, and then select the desired data in the preview window. In the preview window, you can zoom and pan as needed.

7. You can change the background color of the preview image by clicking the black or white icon at the top-right corner of the dialog box.
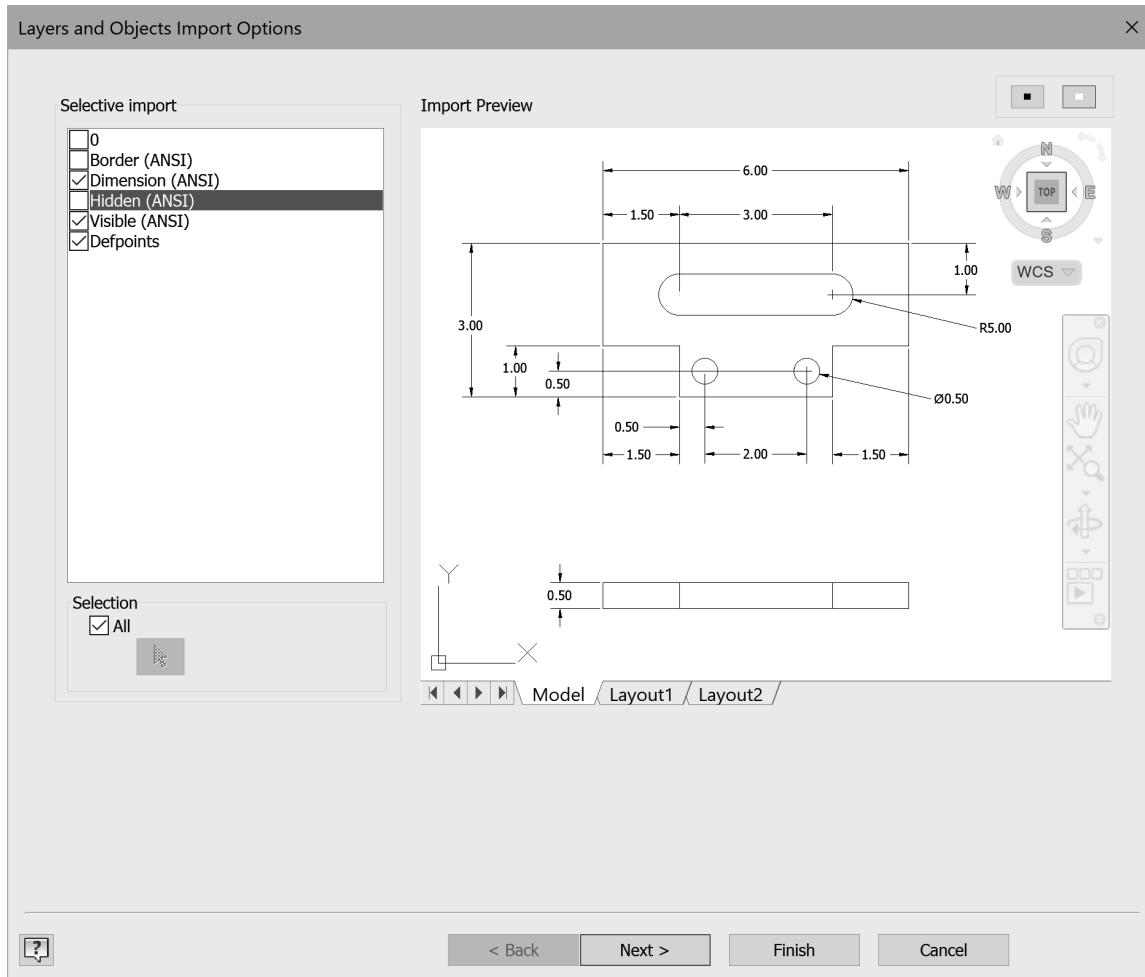

Figure 2.87

8. Click the Next button to go to the next step. In the Import Destination Options dialog box specify the units in which the data was created as shown in the following image.

9. Check the options to Constrain End Points and Apply geometric constraints as shown in Figure 2.88. The Apply geometric constraints option will add sketch constraints to geometry that is parallel, perpendicular and tangent.
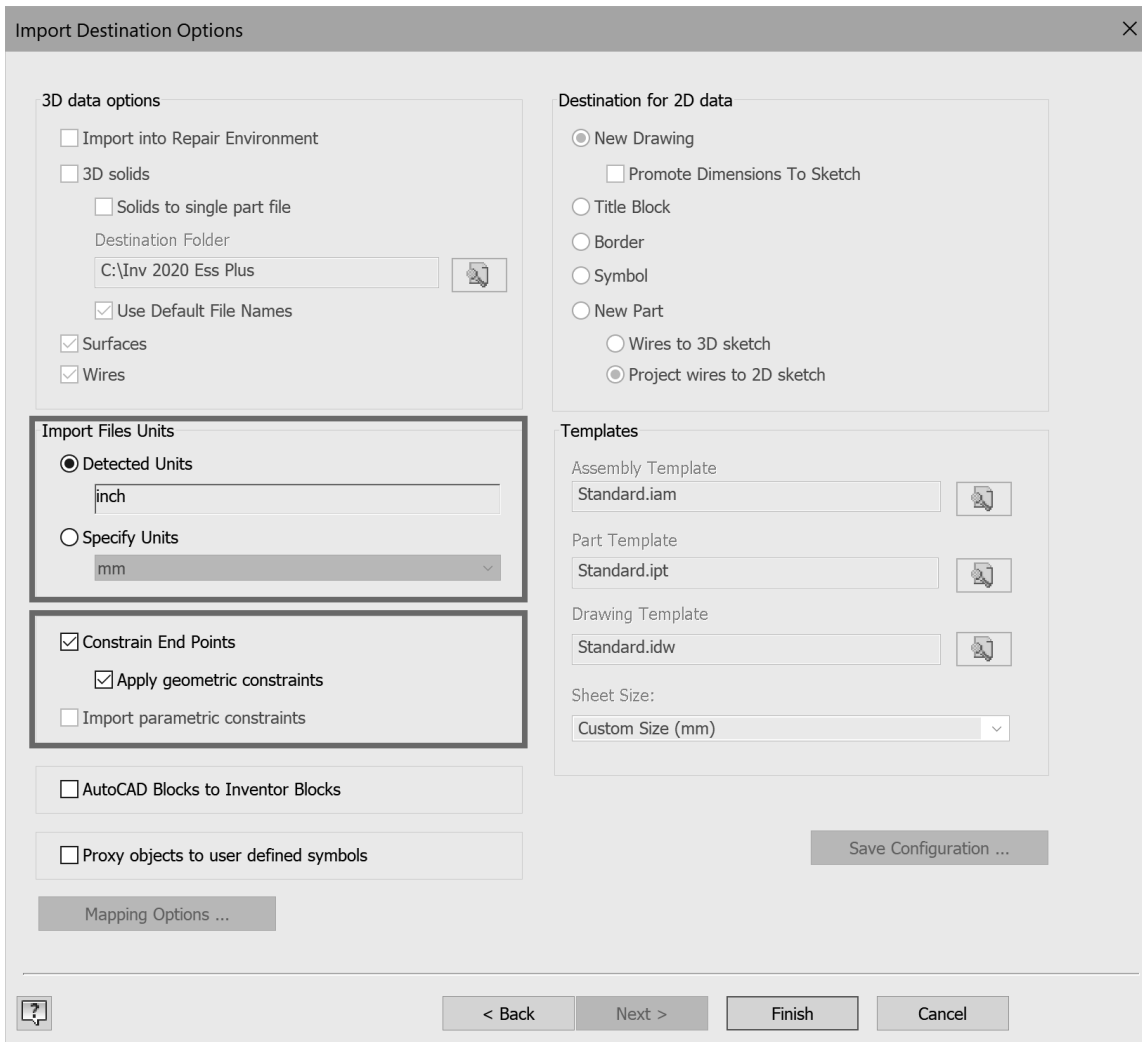
Figure 2.88

Figure 2.88

10. To import the data, click Finish.

11. Use the Zoom All command or double-click the wheel to see all the geometry.

12. Delete unnecessary geometry, constraints, and dimensions.

13. Add geometry (if needed), constraints, and dimensions to fully constrain the sketch.

Insert AutoCAD File with Associativity

If you want to import AutoCAD data as an underlay, which allows you to project the imported geometry onto a sketch, use the Import command on the Manage tab > Insert panel. If the imported AutoCAD data changes, you can update the geometry by clicking the Update command on the Quick Access toolbar.

OPEN OTHER FILE TYPES

Inventor can open parts and assemblies exported from other CAD systems. When files from other CAD systems are opened in Inventor, they will be imported as solids or surface models depending upon the original file and the components will NOT have feature history and an assembly will NOT have any assembly constraints.

You can add features to imported parts and edit the geometry by using Inventor's Direct Edit command. For files that are imported as an assembly, you can add assembly constraints.

To open file types such as DXF, Alias, Catia, IDF Board Files, IGES, JT, Parasolid, PRO/E, SAT, STEP, Solid-Works, and Unigraphics NX, click the File tab > Open or click Open on the Quick Access toolbar.

You can also select the Import CAD Files option using the Import DWG command from the File tab > Open > Import CAD Files; this command will import AutoCAD data into a new drawing, title block, border, symbol or part file without having to first create a new file.

In the Open dialog box, choose the desired file format from the Files of type list. See the help system for more information about the different file types.
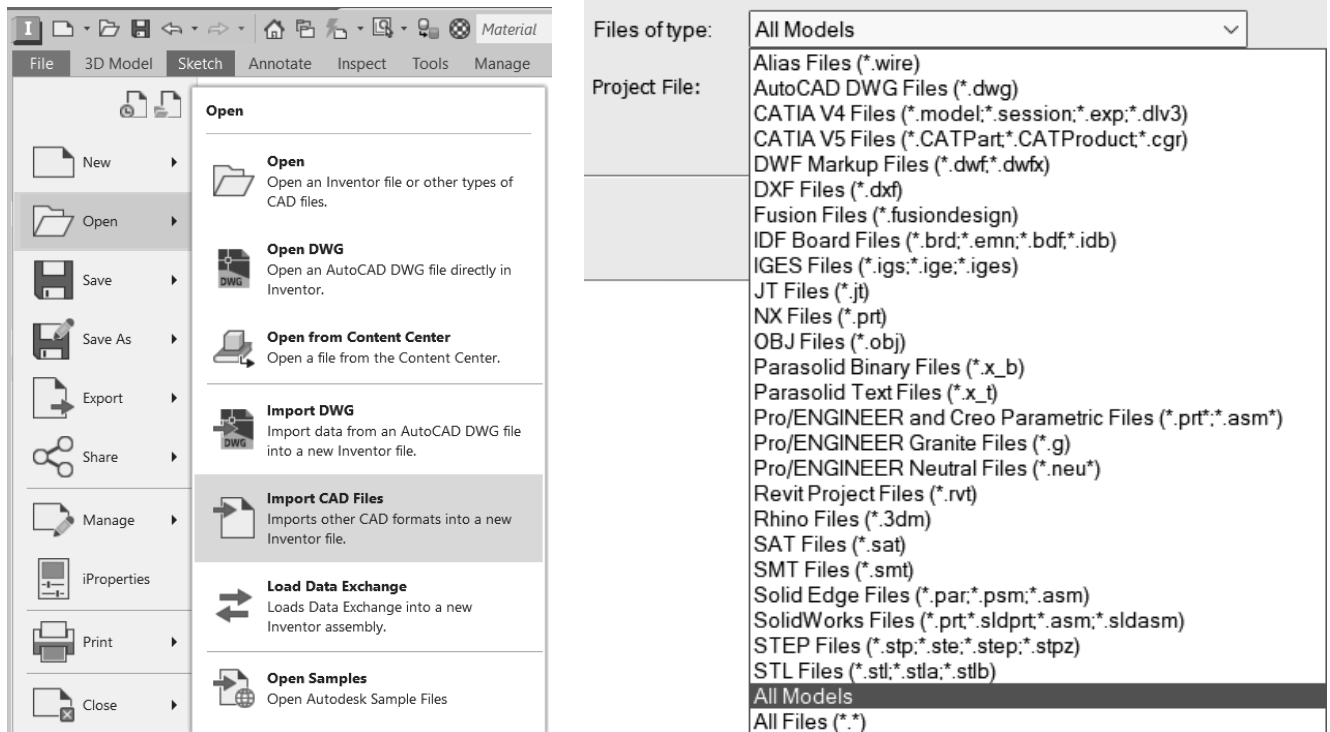

Figure 2.89

EXERCISE 2-5: INSERTING AUTOCAD DATA

In this exercise, you insert AutoCAD data into a sketch and add constraints to fully constrain the sketch.

1 Use the New command, English tab, and choose Standard (in).ipt, or if inch is already the default unit, from the left side of the Quick Access toolbar you can click the down arrow of the New icon, and select Part.

2 Click Start 2D Sketch from the 3D Model tab > Sketch panel. Select the XY origin sketch plane.

3 Click ACAD from the Sketch tab > Insert panel.

4 From the Frequently Used Subfolder area (dialog box upper-left) click the Chapter 02 subfolder and then in the file area double-click to choose the file *AutoCAD 2D Bracket.dwg*.
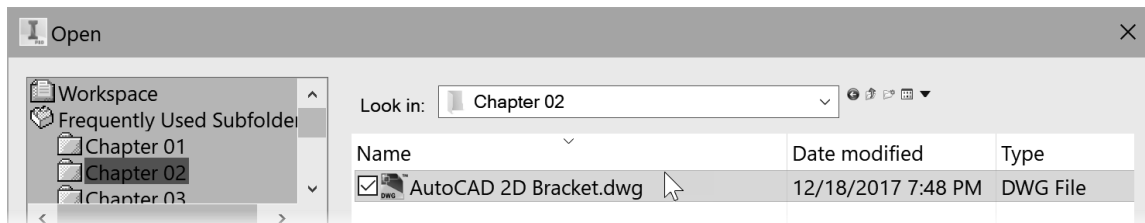

Figure 2.90

5   From the Layers and Objects Import Options dialog box *uncheck* the layers that are not needed. In the upper left corner of the dialog box *uncheck* layers 0, Border (ANSI) and Hidden (ANSI) as shown in Figure 2.91 (1).

6   In the Selection area near the bottom left corner of the dialog box uncheck All, labeled (2) in the following image.

7   Select the geometry and dimensions to insert, use the window selection–click a point above and to the left of the geometry labeled (3) and then click a point below and to the right of the geometry labeled (4). Note that if you click and drag you can draw an irregular shape around the geometry.
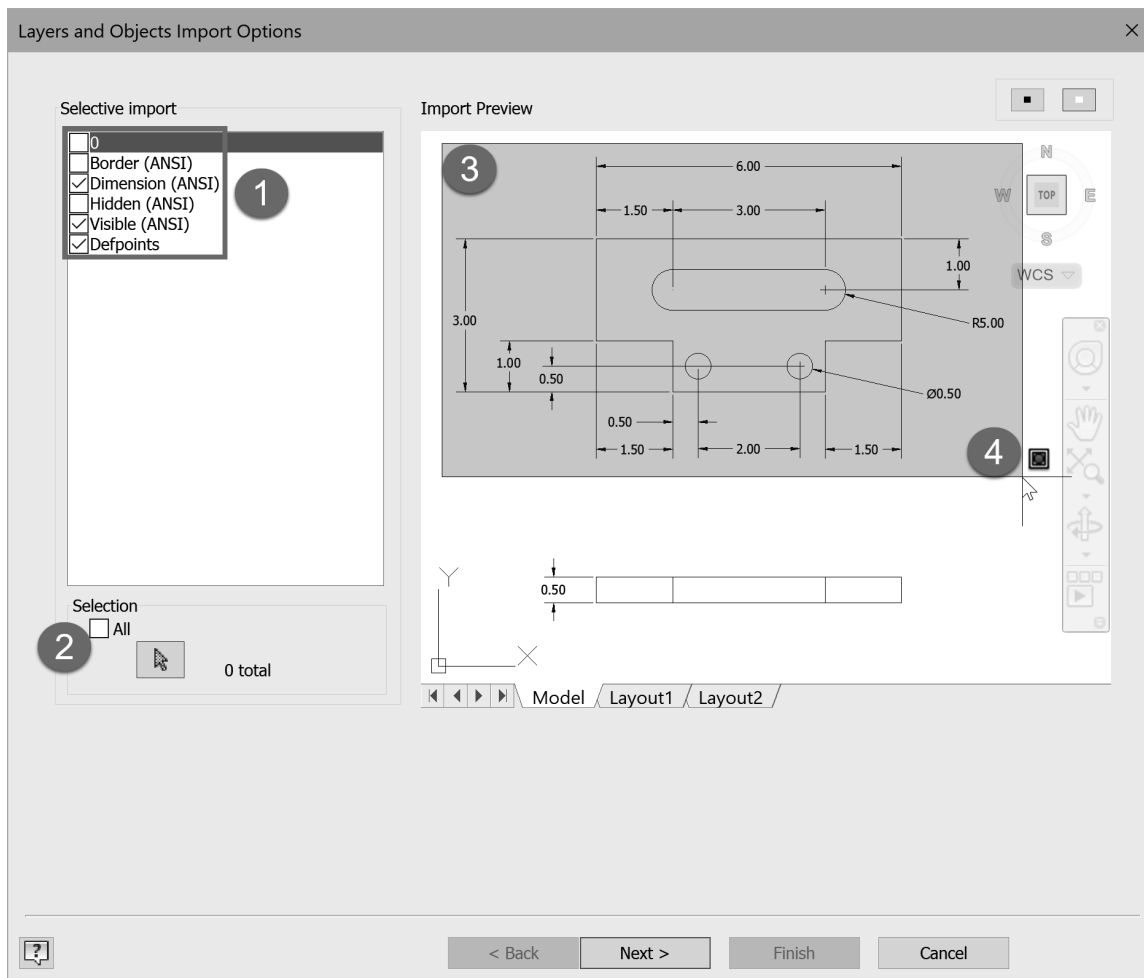

Figure 2.91

8   In the Selection area of the dialog box, verify that 27 total objects are selected as shown in Figure 2.92. If not, reselect all the data in the top view.

Figure 2.92

9   Click Next from the bottom of the dialog box.

10 In the Import Destination Options dialog ensure inch is set as the detected unit.

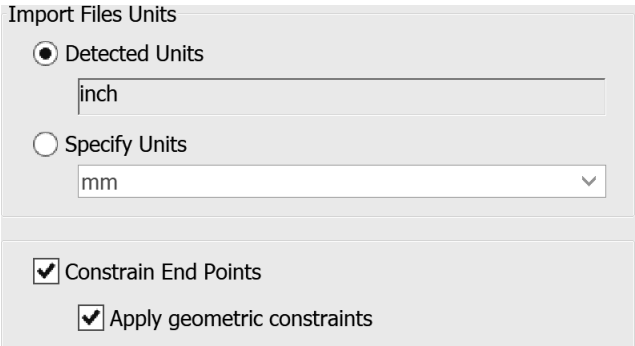**11** Check the Constrain End Points and the Apply geometric constraints options as shown in Figure 2.93.

Figure 2.93

12 Click Finish from the bottom of the dialog box.

13 If needed, display all the geometry by double-clicking the mouse wheel.

14 Apply a horizontal constraint between the center points of the two circles labeled (1) in Figure 2.94.

15 Apply a collinear constraint between the two middle horizontal lines labeled (2).

16 Press the ESC key twice to cancel the command.

17 The sketch is free to move. To constrain the sketch to the origin, drag the lower-left corner of the sketch labeled (3) in the following image to the origin point of the sketch (0,0). Or you could add a coincident constraint between the origin point and the left point on the bottom line.

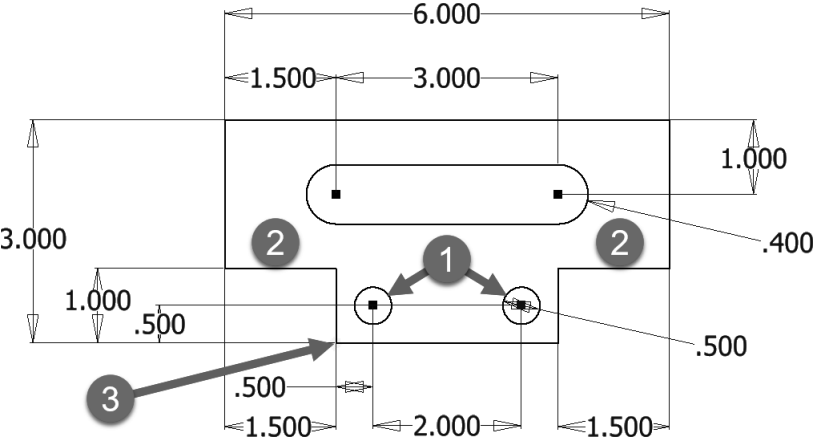**Note** that you may need to zoom out to see the origin below and to the left of the inserted geometry.

Figure 2.94

18 The lower-right of the Status Bar should show "1 dimensions needed" to constrain the sketch.

19 Drag the top-right endpoint of the upper horizontal line up. The sketch will be rotated slightly as shown in Figure 2.95 left.

20 Apply a horizontal constraint to the lower horizontal line, and this will fully constrain the sketch as shown in Figure 2.95 right. Reposition the dimensions as needed.
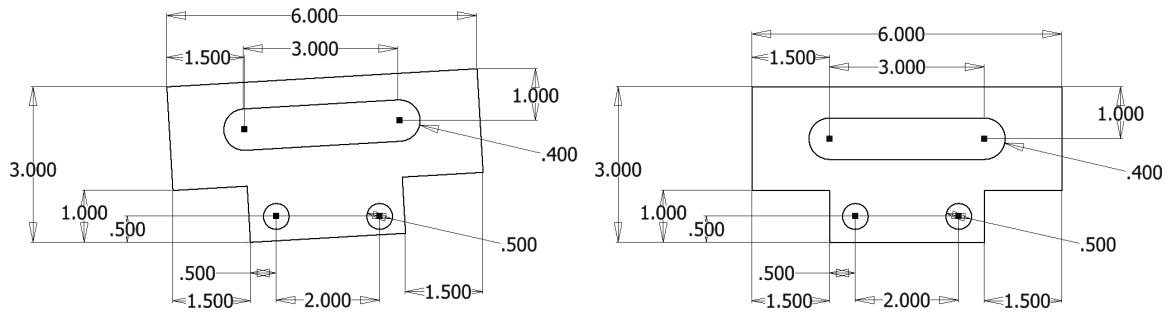
Figure 2.95

21 Press the F8 key to see all constraints.

22 Press the F9 key to hide all constraints.

23 The AutoCAD dimensions on the sketch are now parametric and can be edited. Practice editing the values of the dimensions by double-clicking on a dimension's value and enter a new value.

24 Close the file. Do not save changes.

End of exercise.

APPLYING YOUR SKILLS

Skills Exercise 2-1

Start a new part file based on the Standard (in).ipt, choose the XY sketch plane. Create the sketch and add geometric and dimensional constraints to fully control the size and shape for the part below.

- Assume that the top and bottom horizontal lines are collinear.
- The center points of the arcs are aligned vertically.
- The sketch is symmetric about the left and right sides.
- The bottom angled lines should be coincident with the center point of the lower arc (if the arc is drawn via the line command, the center point of the arc will automatically be coincident with the line it was drawn from).
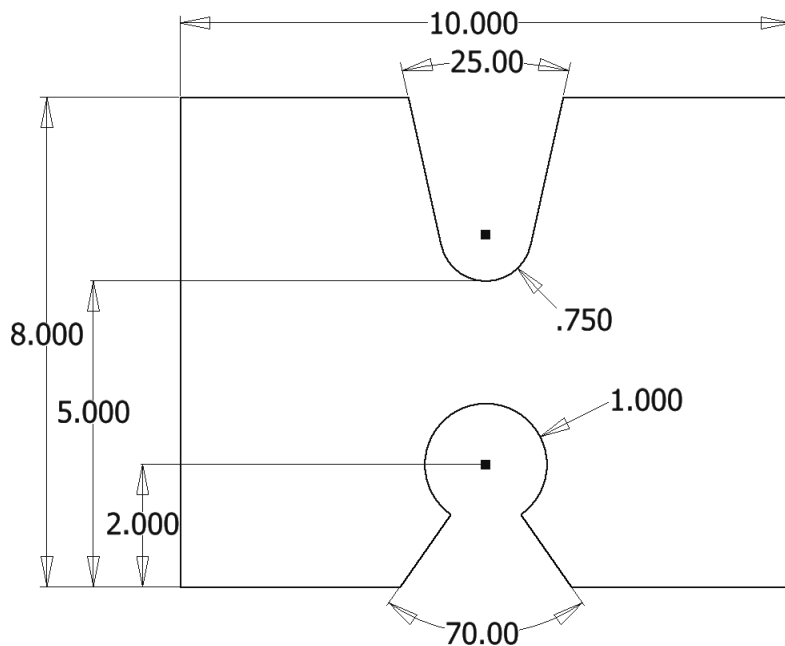
When done, close the file and do not save the changes.

Figure 2.96

Skills Exercise 2-2

Start a new part file based on the Standard (in).ipt template. Create a fully constrained sketch on the XY plane using lines and arcs as shown in the following image. First sketch the two circles and align their center points horizontally. Then create the two lines and place a vertical constraint between the line endpoints on both ends. Add the dimensions. Check that your sketch is fully constrained. When done, close the file and do not save the changes.
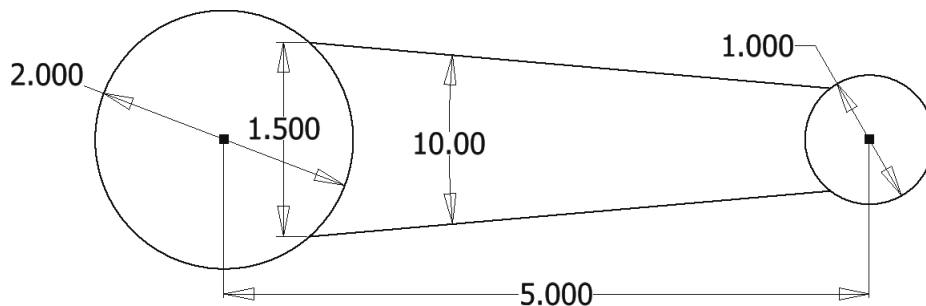
Figure 2.97

CHECKING YOUR SKILLS

Use these questions to test your knowledge of the material in this chapter.

1. True__ False__ While sketching, by default, geometric constraints are not applied to the sketch.

2. True__ False__ When you sketch and a point is inferred, a constraint is applied to represent that relationship.

3. True__ False__ It is recommended to never fully constrain a sketch.

4. True__ False__ When working on a millimeter part, you cannot input inch units.

5. True__ False__ After a sketch is fully constrained, you cannot change a dimension's value.

6. True__ False__ A driven dimension is another name for a parametric dimension.

7. True__ False__ Dimensions placed dynamically are not parametric.

8. True__ False__ You can only import 2D AutoCAD data into Autodesk Inventor.

9. Explain how to draw an arc while using the Line command.

10. Explain how to remove a geometric constraint from a sketch.

11. Explain how to change a vertical dimension to an aligned dimension while placing the dimension.

12. Explain how to create a dimension that is tangent to two arcs.

13. True__ False__ AutoCAD needs to be installed to insert AutoCAD geometry.

14. True__ False__ When a sketch is extruded that contains construction geometry, the construction geometry is deleted.

15. Explain how to change the unit type in a part file.

16. Explain where you would turn on Relax Mode.

17. True__ False__ When a pushpin appears in the Sketch entry in the browser, the sketch is fully constrained.

18. True__ False__ By default an arc length dimension can only be a driven dimension.

19. Explain how to draw a rectangle that is centered on the origin point.

20. True__ False__ When creating the first 2D sketch, you must select an origin plane to sketch on.