Autodesk's VEX® Robotics Curriculum

Unit 2: Introduction to Autodesk Inventor

Overview

In Unit 2: Introduction to Autodesk Inventor, you review the Autodesk® Inventor® user interface. The main components are the application menu, Quick Access toolbar, ribbon, browser, shortcut menus, and shortcut keys.

You also review the modeling process with Autodesk Inventor. This includes creating sketches, parts, assembling parts, and documenting parts and assemblies.

Unit Objectives

After completing Unit 2: Introduction to Autodesk Inventor, you will be able to:

- Review the Autodesk Inventor modeling process and the user interface.

Prerequisites and Resources

Before starting Unit 2: Introduction to Autodesk Inventor, you must have:

- A working knowledge of the Windows operating system.
- Completed Unit 1: Introduction to VEX and Robotics > Getting Started with Autodesk Inventor.

Key Terms and Definitions

The following key terms are used in Unit 2: Introduction to Autodesk Inventor.

Term	Definition
Assembly	Two or more components (parts or subassemblies) considered as a single model. An assembly typically includes multiple components positioned absolutely and relatively (as required) with constraints that define both size and position.
Browser	The graphical hierarchy showing relationships between geometric elements in parts, assemblies, and drawings. Icons represent sketches, features, constraints, or attributes for each model. Objects are shown in the browser in the order in which they were created. Objects may also be edited, renamed, added, deleted, copied, and moved to a different location in the browser.
Dimension	Parametric dimensions that control sketch size. When dimensions are changed, the sketch resizes. Dimensional constraints may be expressed as numeric constants, as variables in equations, or in parameter files.
Drawing	A 2D representation of a part or assembly. The drawing file type has an IDW extension.
Extrude	A feature created by adding depth to a sketched profile. Feature shape is controlled by profile shape, extrusion extent, and taper angle.

Term	Definition
Features	Parametric geometry that creates or modifies parts or assemblies. Relationships between features are defined by geometric and dimensional constraints. Types include sketched, placed, and duplicated features, work (construction) features, and assembly features. Features combine to build up a complex part or assembly model. Individual features may be modified as needed.
Graphics Window	The active modeling area in which sketches, constraints, features, parts, and assemblies are created and edited. In the graphics window, models can be rotated, zoomed in and out, and view characteristics such as color, material and light can be defined.
Loft	A sketched feature specified by tangency condition, termination, and order of sketches. Loft features blend two or more dissimilar sketch shapes on nonintersecting sketch planes.
Menu	An area in the Inventor application window where you select Autodesk Inventor tools.
Part	A group of faces that define a closed volume.
Rendered Image	A view of the part or assembly created in the Inventor Studio application.
Revolve	A solid feature created by revolving a profile around an axis.
Ribbon	The ribbon is composed of a series of panels, which are organized into tabs labeled by task. If you have part, assembly, and drawing files open at the same time, the ribbon changes depending on which window is active.
Sketch	A sketch consists of the sketch plane, a coordinate system, 2D curves, and the dimensions and constraints applied to the curves. A sketch may also incorporate construction geometry or reference geometry. Sketches are used to define feature profiles and paths.
Sweep	A feature created by moving a profile along a planar path. A sweep feature requires two sketches, a profile, and a path on intersecting planes.

Technical Overview

The following Autodesk Inventor tools are used in Unit 2: Introduction to Autodesk Inventor.

Icon	Name	Description
	Two Point Rectangle	Create a two-point rectangle.
	General Dimension	Add dimensions to a sketch. Dimensions control the size of a part. They can be expressed as numeric constants, as variables in an equation, or in parameter files.
	Fillet	Placed features that round off or cap interior or exterior corners or features of a part.
	Free Orbit	In a part or assembly, adds a rotate symbol and cursor to the view. You can rotate the view planar to the screen around the center mark, around a horizontal or vertical axis, or around the X and Y axes. Not used in drawings.
	Sketch	A sketch consists of the sketch plane, a coordinate system, 2D curves, and the dimensions and constraints applied to the curves.
	Line	Straight curve bounded by two endpoints. The Line tool on the Sketch toolbar chains line segments together and creates arcs tangent or perpendicular to existing curves.
	Colinear Constraint	A geometric constraint that causes two or more line segments or ellipse axes to lie along the same line. In an assembly, a colinear constraint is achieved with a mate constraint between two lines, edges, or axes.
	Equal Constraint	A geometric constraint that causes selected arcs and circles to have the same radius or selected lines to have the same length.
	Vertical Constraint	A geometric constraint that aligns selected geometry vertically with respect to the sketch axes. Geometry is typically a line or two points.
	Point, Center Point	Create both center points (default) and sketch points.
	Project Geometry	Project geometry (model edges, vertices, work axes, work points, or other sketch geometry) onto the active sketch plane as reference geometry.
	Center Point Circle	Create a circle from a center point and radius, or tangent to three lines.
	Zoom All	Zoom to a part or assembly so that all elements are displayed in the graphics window.

Icon	Name	Description
	Place Component	Specify one or more files to place as a component in an assembly.
	Constrain	Determine how components in the assembly fit together.
	Move Component	When you constrain assembly components to one another, you control their position. To move a component, either temporarily or permanently.
	Base View	The first view in a new drawing is a base view. Use the Base View button on the Drawing Views panel bar to add additional base views to a drawing.
	Projected View	Create a projected view with a first-angle or third-angle projection, depending on the drafting standard for the drawing.
	Auto Balloon	Create one or more item balloons used to identify components in drawing views.
	Parts List	In an assembly, a listing of components. Usually, a parts list is single level and consists of an item number or other designation, part name, and quantity.

Required Supplies and Software

The following software is used in Unit 2: Introduction to Autodesk Inventor.

Software
Autodesk® Inventor® Professional 2010

Academic Standards

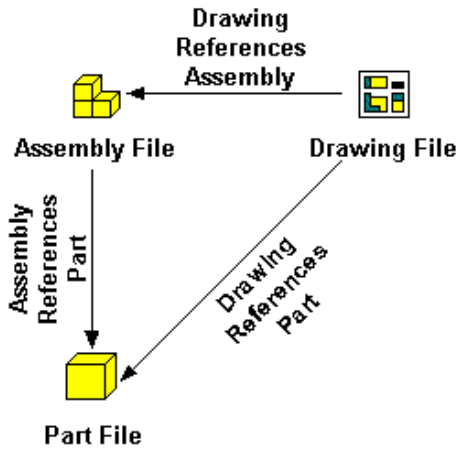
The following national academic standards are supported in Unit 2: Introduction to Autodesk Inventor.

Phase	Standard
Create	**Science (NSES)** ■ *Unifying Concepts and Processes*: Form and Function ■ *Physical Science:* Motions and Forces ■ *Science and Technology*: Abilities of Technological Design **Technology (ITEA)** ■ 5.8: The Attributes of Design ■ 5.9: Engineering Design ■ 6.12: Use and Maintain Technological Products and Systems **Mathematics (NCTM)** ■ *Numbers and Operations*: Understand numbers, ways of representing numbers, relationships among numbers, and number systems. ■ *Algebra Standard*: Understand patterns, relations, and functions. ■ *Geometry Standard*: Use visualization, spatial reasoning, and geometric modeling to solve problems. ■ *Measurement Standard*: Understand measurable attributes of objects and the units, systems, and processes of measurement.

Quick Start for Autodesk Inventor

In order to provide the greatest design flexibility and reuse, each part, assembly, and drawing is stored in a separate file. Each part file is a stand-alone entity that can be used in different assembly files and drawing files. When you make a change to the part, that change is evident in each assembly or drawing into which it is referenced. Assembly files can be referenced into other assembly files, into presentation files, and into drawing files.

The basic file references that exist in a typical 3D design are represented in the following illustration.

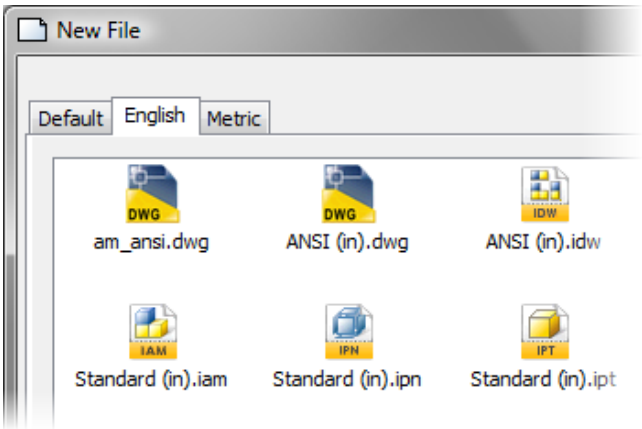


Using Template Files

Template files serve as the basis for all new files you create. By using template files, you control default settings such as units, snap spacing, and default tolerances in your new file.

Autodesk Inventor includes template files for each type of file. Template files are categorized into two main groups: English for English units (inches and feet), and Metric for metric units (millimeters and meters).

The New File dialog box has three tabs: Default, English, and Metric. The Default tab includes templates based on the default unit you select during installation, while the English and Metric tabs contain template files in their respective units.

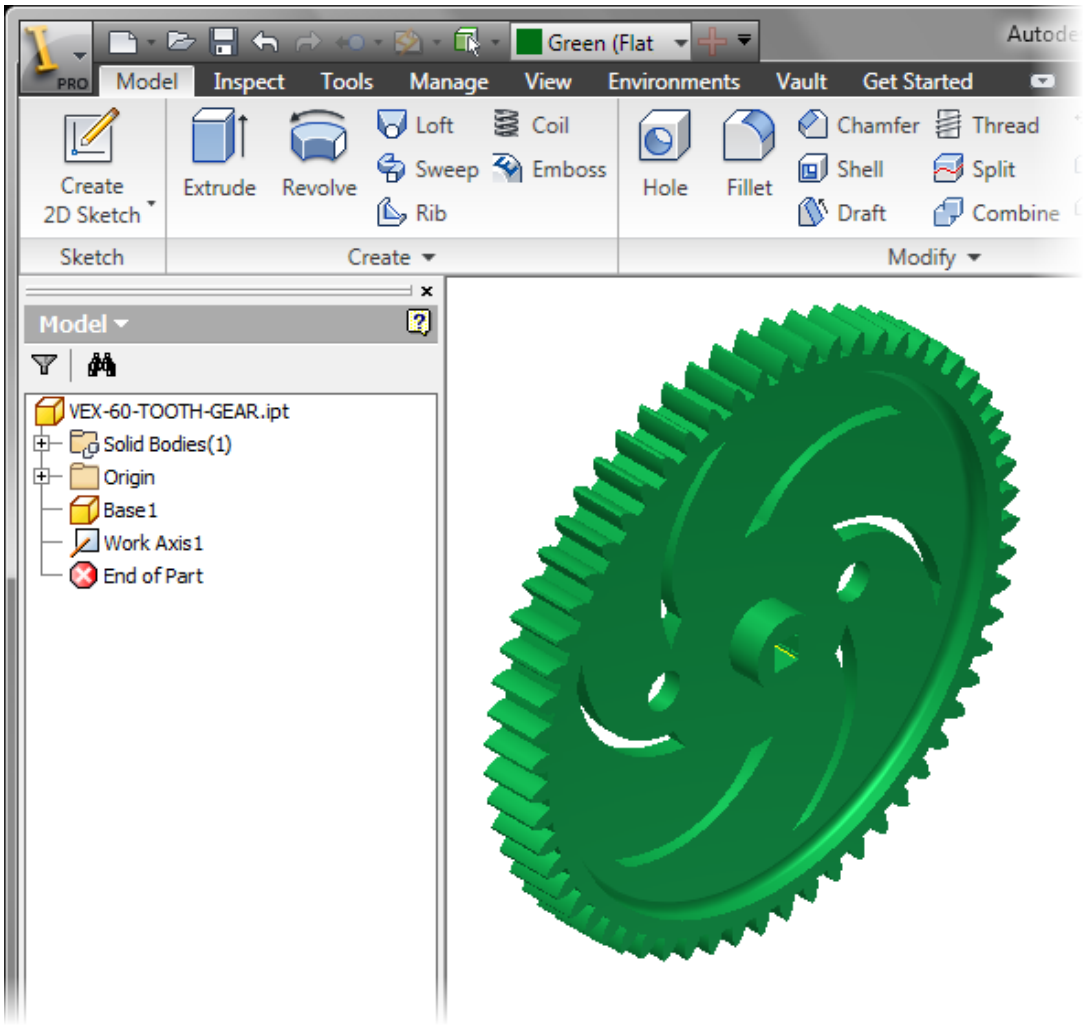


Part Modeling Environment

In the part modeling environment:

- You create and edit 3D part models.
- The interface adjusts automatically to present tools for your current task; for example, tools for sketching or tools to create 3D features.

The user interface in the part modeling environment is as shown.

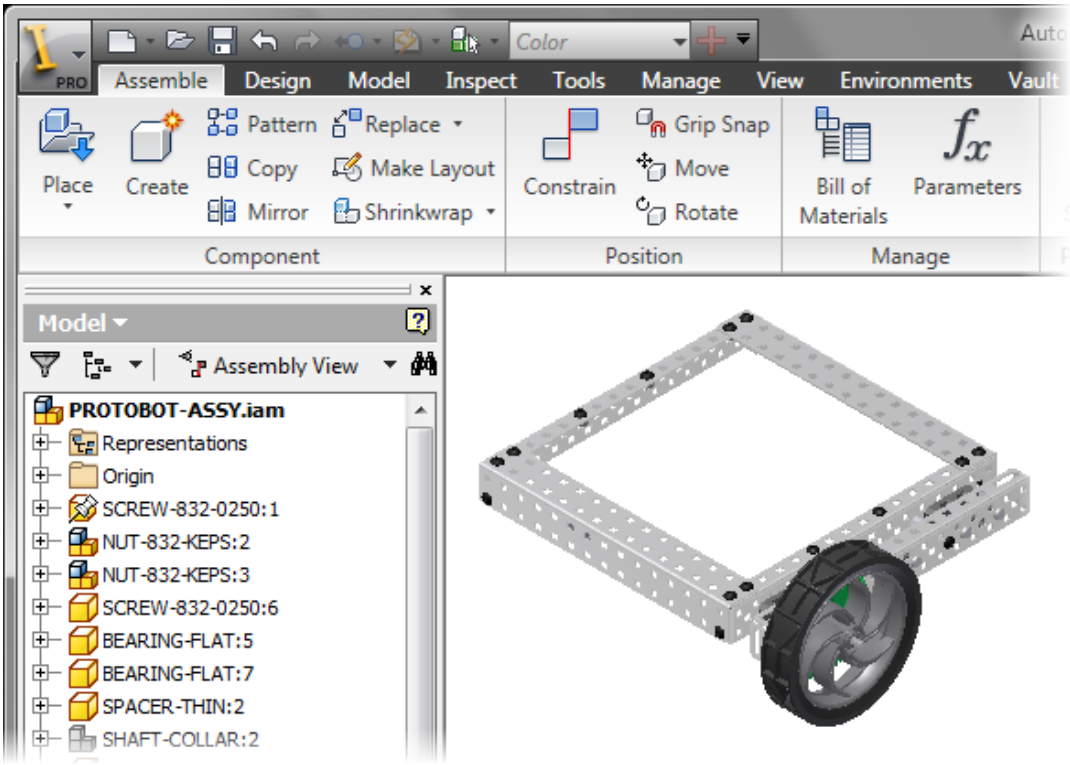


Assembly Modeling Environment

In the assembly modeling environment:

- You build and edit 3D assembly models. The components displayed in the system are references to external parts and subassemblies.
- You use assembly-specific tools to position and build relationships between components.
- You have access to a common set of viewing tools.

The user interface in the assembly modeling environment is as shown.

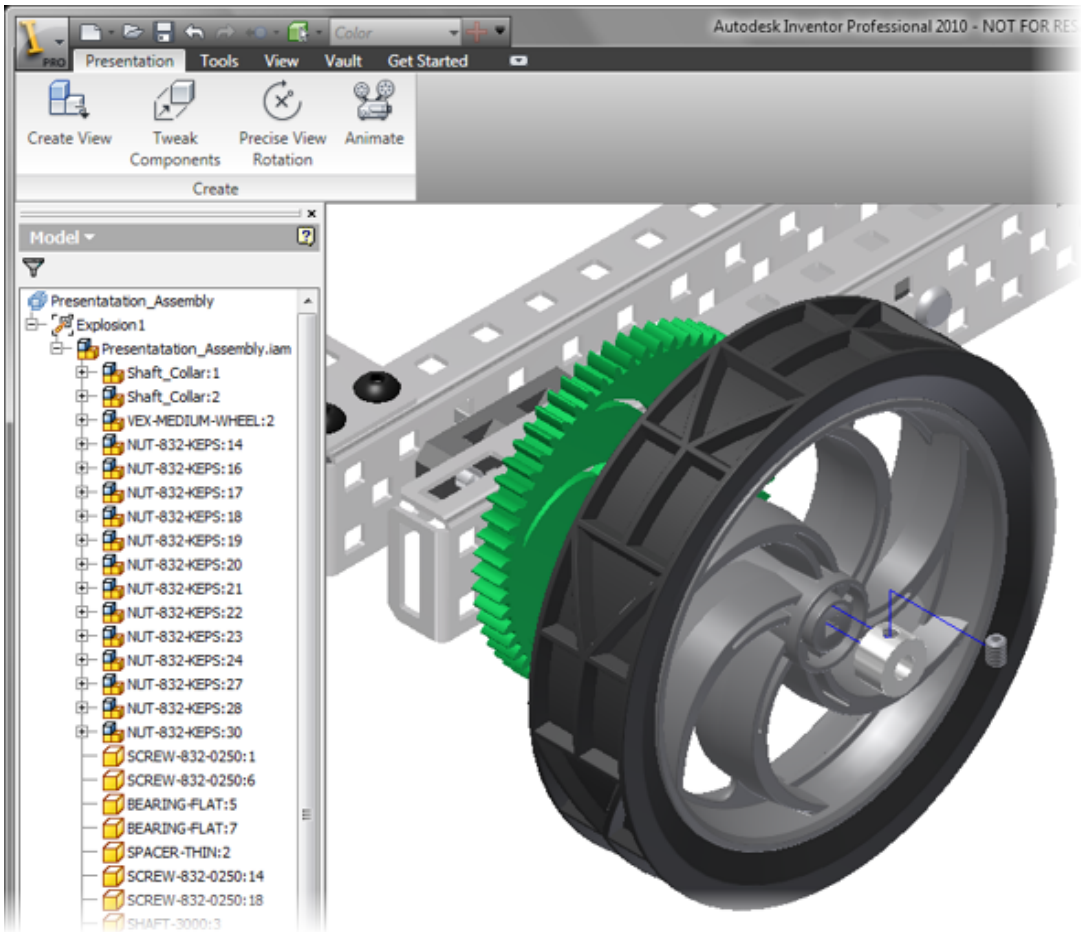


Presentation Environment

In the presentation environment:

- You create exploded assembly views.
- You can record an animation of an exploded view to help document your assembly.
- The presentation file references an existing assembly.
- A common set of viewing tools is available.

The user interface in the presentation environment is as shown.

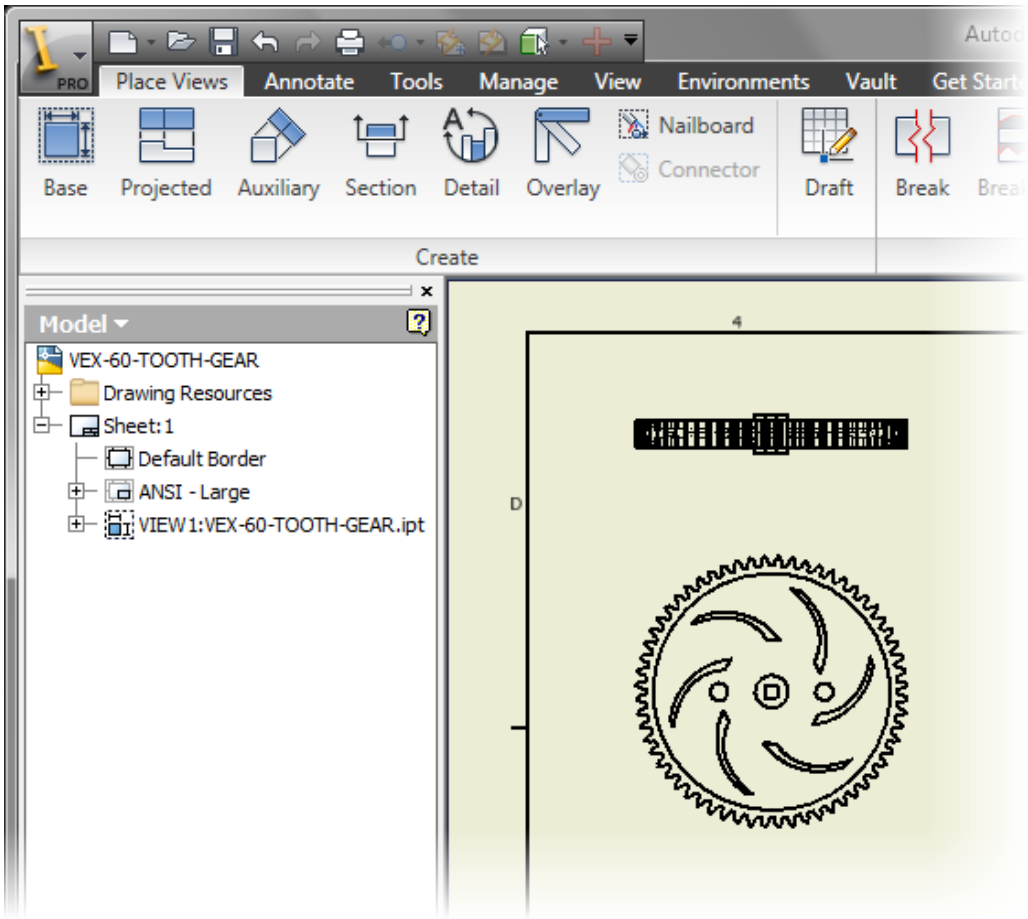


Drawing Environment

In the drawing environment:

- You create 2D drawings of parts and assemblies.
- A drawing file references one or more parts, assemblies, or presentation files. Changes to the part or assembly model update the associated drawing views and annotations.

The user interface in the drawing environment is as shown.



Context-Sensitive Tools

As you switch between environments or between tasks in a single environment, Autodesk Inventor displays the appropriate tools and information for the current task. The ribbon automatically presents tools for the current task, and the browser displays information on the active environment.

Overview of the Browser

The browser is one of the main interface components. It is context-sensitive with the environment you use. For example, when you work on an assembly you use the browser to present information specific to the assembly environment. While you use the part modeling environment, the browser displays information that is relevant to part modeling.

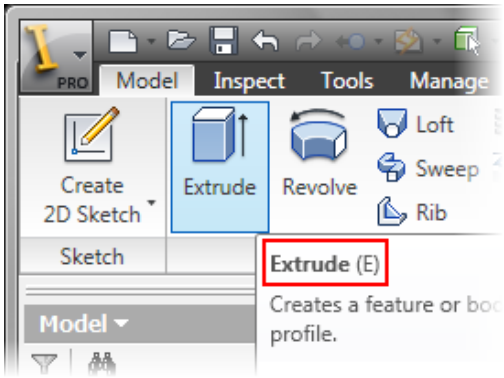
Overview of the Ribbon

The ribbon is your primary interface for accessing the tools available while you design. The context-sensitive design presents the relevant tools based on the current context of your design session. For example, when you switch from assembly modeling to part modeling, the ribbon switches automatically to display the correct tools for the context in which you work.

Keyboard Shortcuts

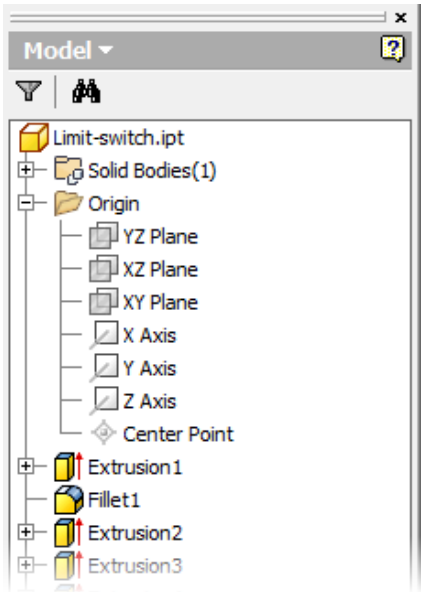
On the ribbon and menus, you can use keyboard shortcuts to access tools. For example, you can enter E for Extrude. Entering the keyboard shortcut is the same as clicking the tool on the ribbon.

Keyboard shortcuts are displayed when you hover the cursor over a tool icon on the ribbon.

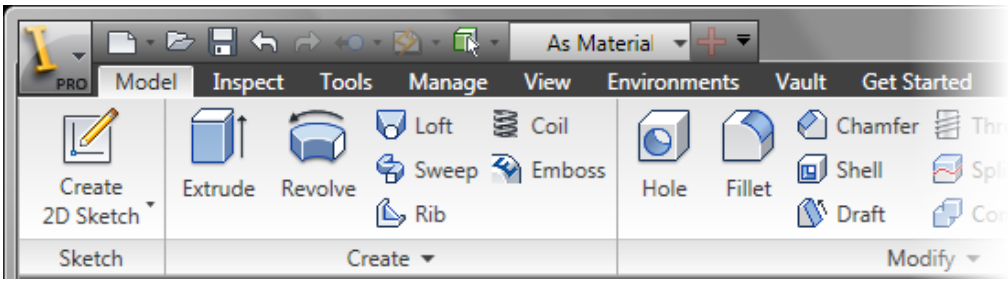


Part Modeling Environment

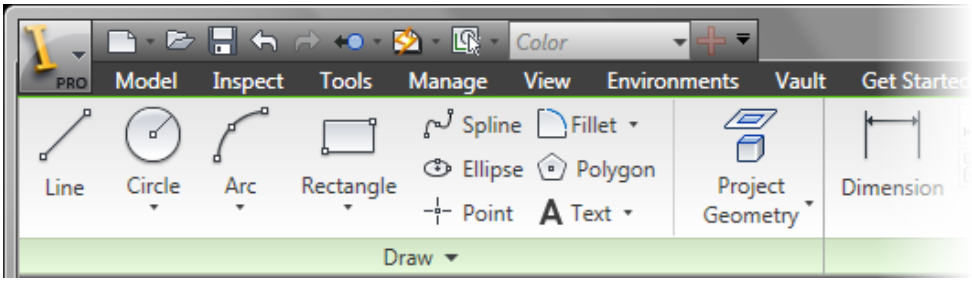
When you are in the part modeling environment, the browser displays all features you used to create the part. The features are listed in the order in which the model is created. The browser also displays the Origin folder at the top of the list, which contains the default *X*, *Y*, and *Z* planes, axes, and center point.



You use these tools on the ribbon to create parametric features on the part.



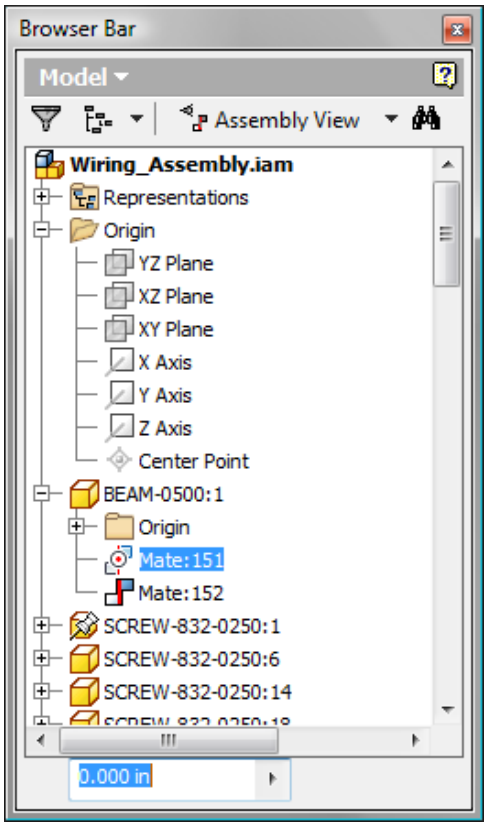
You use the sketch tools to create 2D parametric sketches, add dimensions, and constraints.



Assembly Modeling Environment

When you are in the assembly modeling environment, the browser displays all the parts you use in the assembly. It also lists the Origin folder containing the default *X*, *Y*, and *Z* planes, axes, and center point of the assembly.

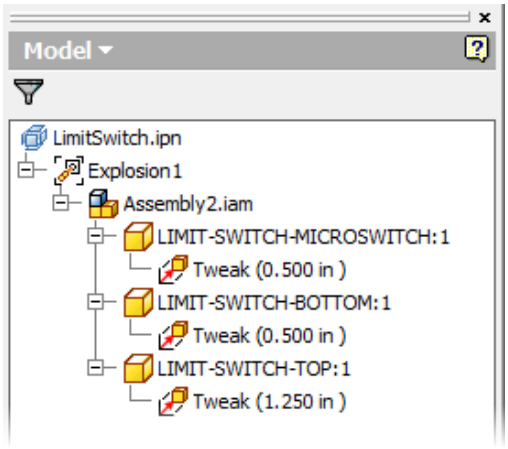
Nested under each part, you see the assembly constraints. If you select an assembly constraint, an edit box is displayed at the bottom of the browser, enabling you to edit the offset or angle value for the constraint.



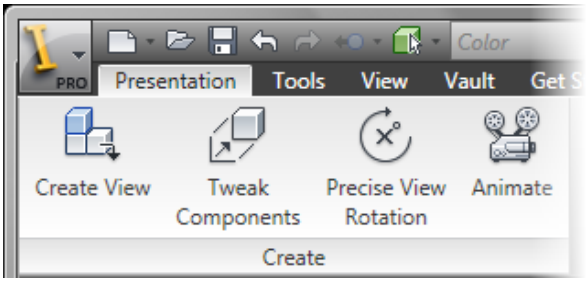
In the assembly environment, you can use the Modeling View option in the Assembly View drop-down list to display the part features nested under the parts instead of the assembly constraints. This is useful when performing part modeling functions in the context of the assembly.

Presentation Environment

When you are in the presentation environment, the browser displays the presentation views you create followed by the tweaks you use for the explosion. When you expand each tweak, you see the parts included in that tweak. You can also switch the browser mode from Tweak View to Sequence View or Assembly View.

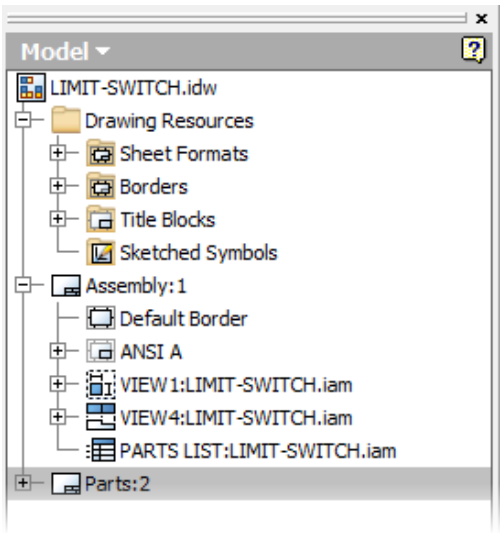


You use the tools on the ribbon to create presentation views and tweaks, and animate geometry in the presentation environment.

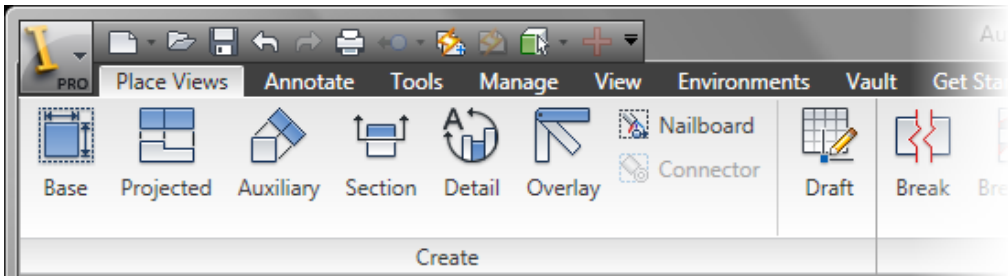


Drawing Environment

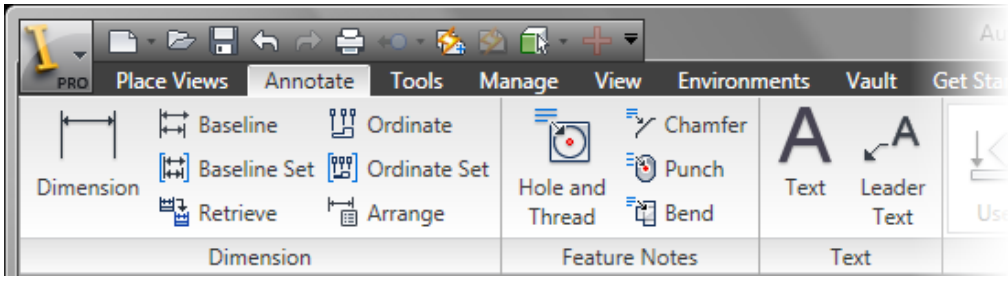
In the drawing environment, the browser displays the Drawing Resource folder containing sheet formats, borders, title blocks, and sketched symbols. It also displays each sheet in the drawing along with the views you create for each.



You use the tools on the ribbon to create drawing views on the sheet.



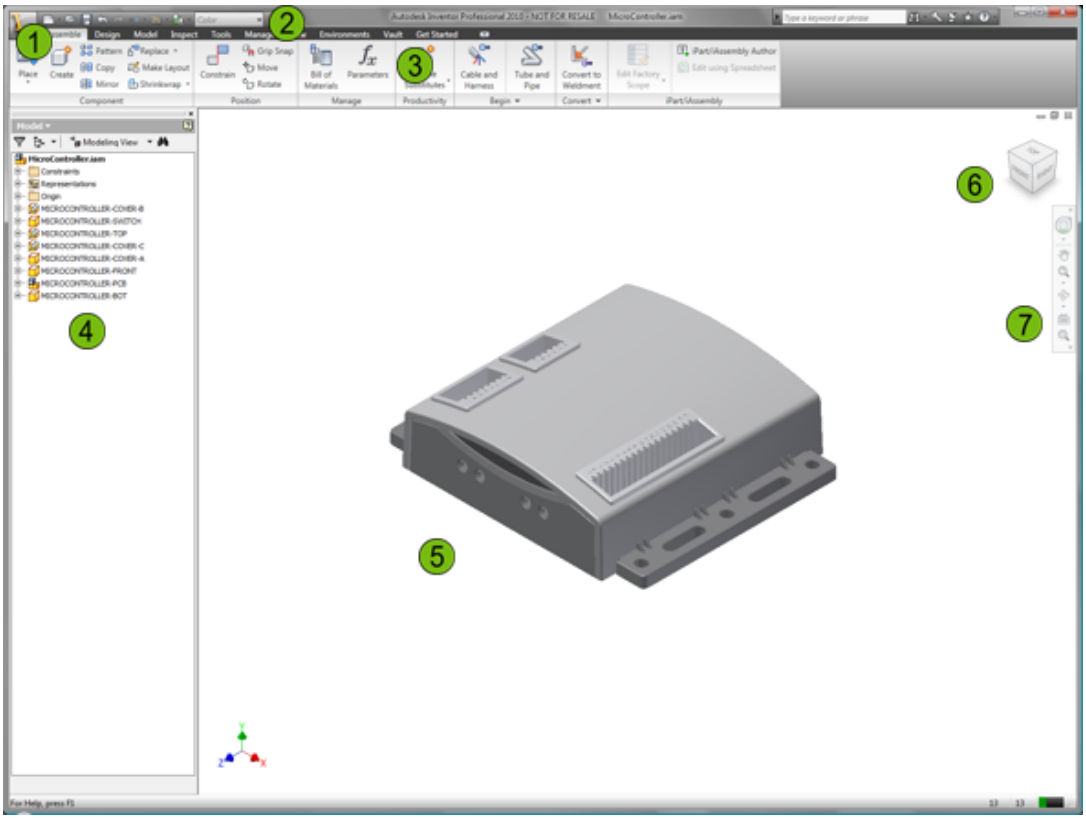
In the drawing environment. on the Annotate tab, you use the tools to add reference dimensions and other annotation objects.



Accessing Tools

The tools and commands you use are located in different areas of the user interface. In the exercises, you are given instructions to follow. You need to understand the basic areas where tools and commands are located so that you can follow the instructions.

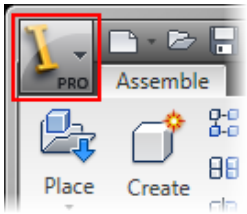
For your reference, the main areas of the Autodesk Inventor user interface are shown.



1. Application menu
2. Quick Access toolbar
3. Ribbon
4. Browser
5. Graphics window
6. ViewCube
7. Navigation toolbar

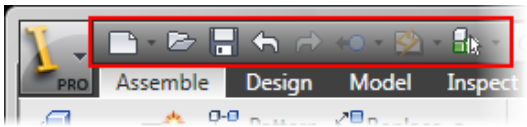
Application Menu

The Application menu is located at the top left of the Autodesk Inventor window.



Quick Access Toolbar

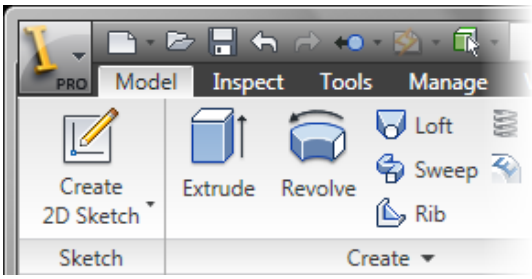
The Quick Access toolbar is located at the top of the Autodesk Inventor window.

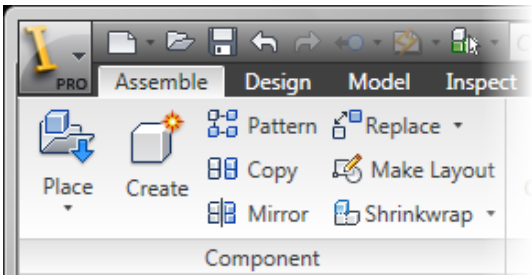


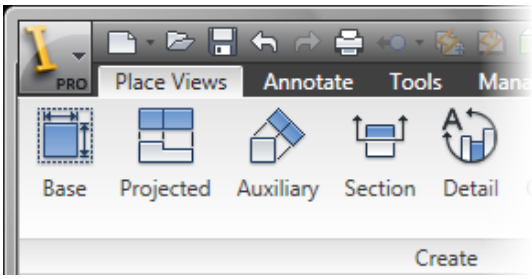
Ribbon

The ribbon is composed of a series of panels, which are organized into tabs labeled by task.

If you have part, assembly, and drawing files open at the same time, the ribbon changes depending on which window is active.

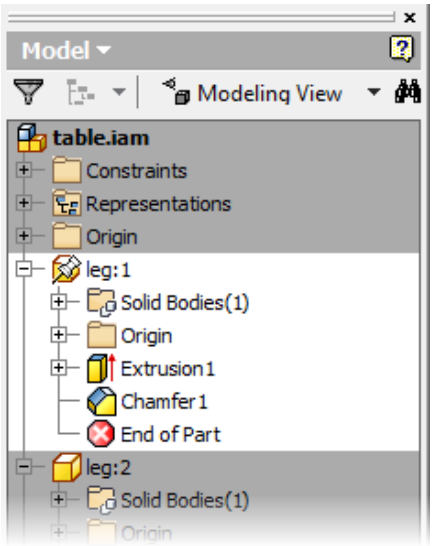






The Browser

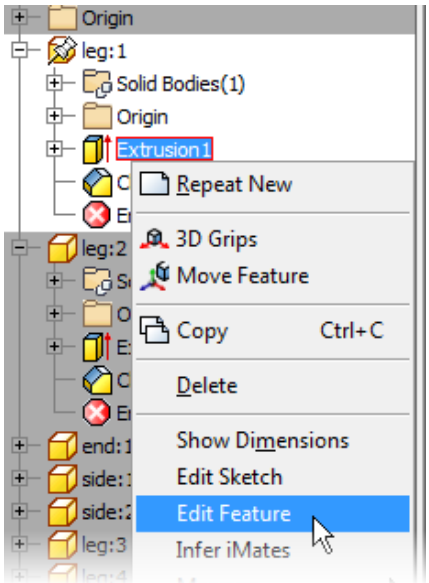
The browser is typically located on either the left or right side of the Autodesk Inventor window. The browser changes depending on the type of file you are editing. For example, for parts, the browser displays all of the features that were used to create the part. For assemblies, the browser displays all of the parts that make up the assembly. You frequently use the browser to access features or parts.

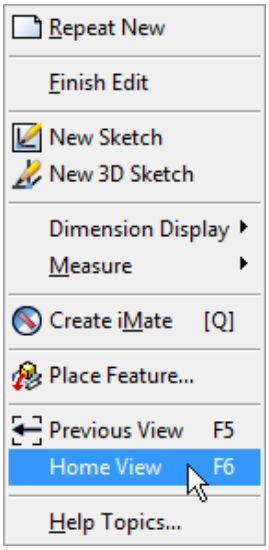


Shortcut Menus

You frequently access shortcut menus in the graphics area of the screen. Different menus are displayed depending on whether you right-click a part or right-click the graphic background (the blank area around the part). If you cannot find the command on the menu, make sure you are right-clicking in the correct area.

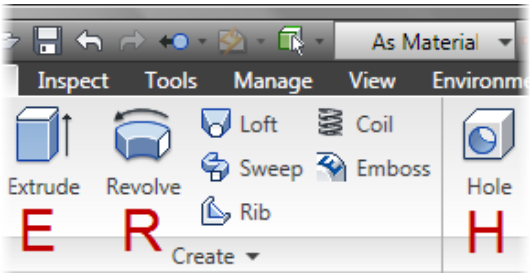
Some tools and commands are accessed through shortcut menus. You display a shortcut menu by clicking the right mouse button.

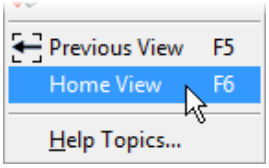




Shortcut Keys

Some of the commands can be accessed by pressing a key or combination of keys instead of clicking. When you are familiar with the shortcut key for a command, you may find it faster to press a key than to find the correct menu and click the icon. The shortcut key is listed next to the tool in the menu or panel bar. The following image shows three commands from the ribbon. To create an extrusion, either click Extrude or press E. To create a revolved feature, press R. To create a hole, press H.





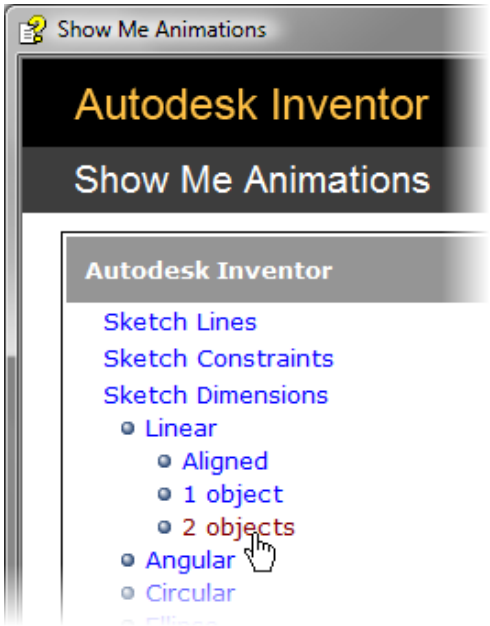
Exercise: Use the Design Support System

In this exercise, you access Autodesk Inventor software support using the Help System.

This exercise illustrates how you can access immediate support during the design process. This support includes Tutorials, Show Me Animations, and Help Topics.

Before starting the exercise, you can refer to a video demonstration of the workflow. To view the demonstration, navigate to the Video Demonstration folder for this exercise.

You can pause and scroll through the video to search for help on how to complete a specific section.

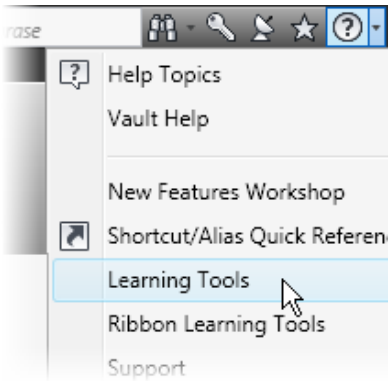


The completed exercise

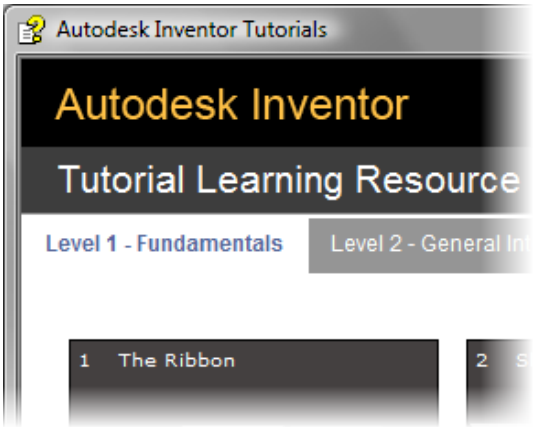
Access Autodesk Inventor Tutorials

Autodesk Inventor includes several tutorials to help you learn the features of the software. If you want to spend time on your own learning more about Autodesk Inventor, try some of the tutorials from the Level 1 - Fundamentals section.

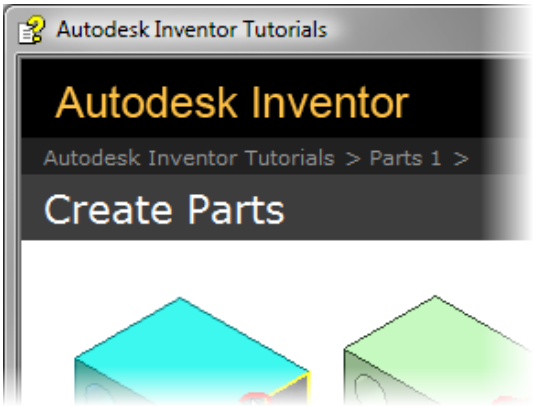
1. Start Autodesk Inventor.
2. If the Open dialog box or Help window is displayed, close them.
3. Click Help menu > Learning Tools > Tutorials.



The Autodesk Inventor Tutorials window is displayed.



4. On the Level 1 - Fundamentals tab, click Parts 1.
5. Review the first page.



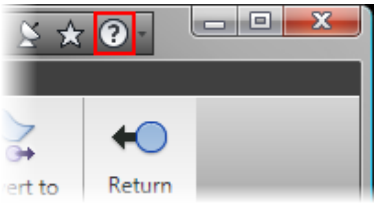
6. Click the Next button. Review the next four pages.

7. Close the Autodesk Inventor Tutorials windows.

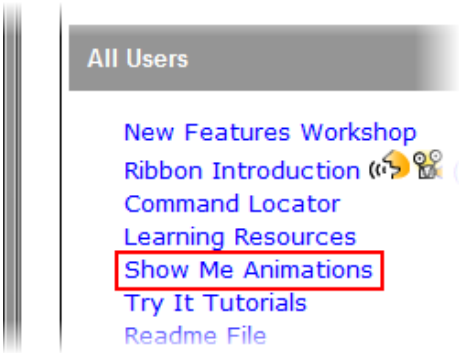
Access Show Me Animations

In a design session, you work in different environments such as part modeling and assembly modeling. There are tasks in these environments that you may not be familiar with and require immediate help. The Visual Syllabus displays a palette of design tasks that guides you through the task.

1. Click Help.

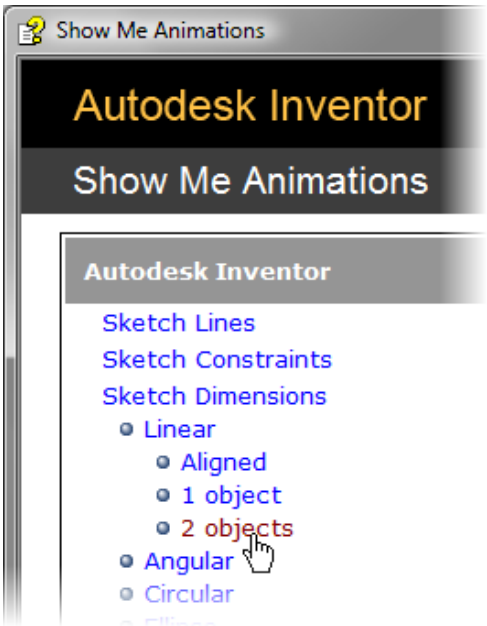


2. Under All Users, click Show Me Animations.



3. Under Autodesk Inventor, click Sketch Dimensions.

4. Click Linear > 2 objects.

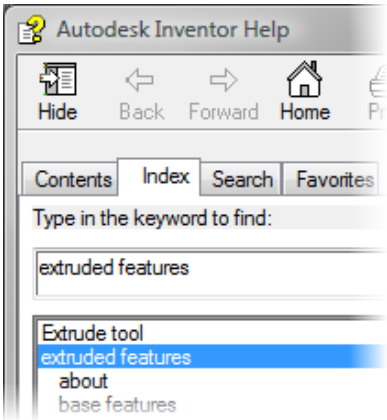


5. Review the animation.
6. Close the Show Me Animations and Autodesk Inventor Help windows.

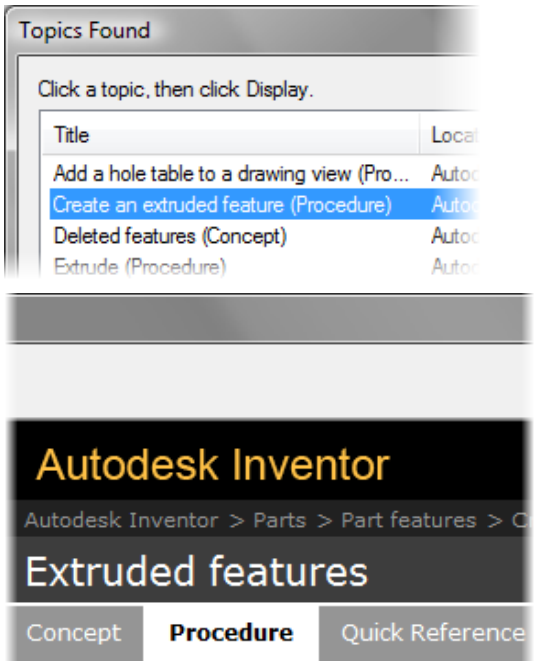
Access Help Topics

In this section of the exercise, you access Help using a key word and review an animated file.

1. Click Help.
2. If required, click Show.
3. Click the Index tab.
4. Enter **extrude**.
5. In the list of topics, double-click Extruded Features.



6. In Topics Found, make sure that Create an Extruded Feature (Procedure) is selected. Click Display.



7. Scroll down the page. Click Show Me How to Create a Basic Extrusion.

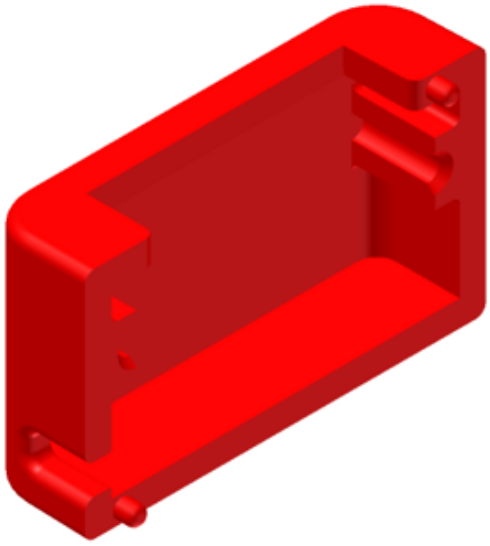
   The Show Me Animations window is displayed and the animation is played.

8. Review the animation. Use the navigation tools to repeat or advance the animation.

9. Close the Show Me Animations and Help windows.

Exercise: Build a Limit Switch

In this exercise, you create the top case of the limit switch from the VEX kit of parts. After you model the case, you change the material to plastic and determine the part's mass.



The completed exercise

Before starting the exercise, you can refer to a video demonstration of the workflow. To view the demonstration, navigate to the Video Demonstration folder for this exercise.

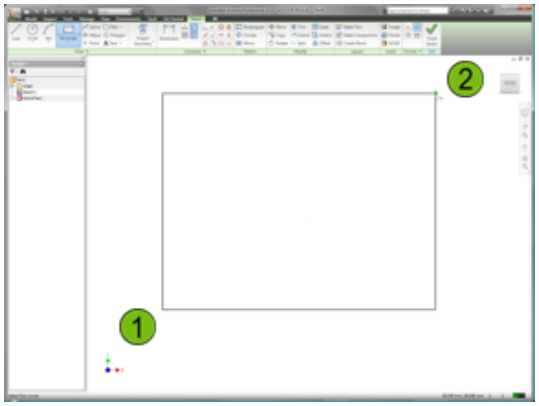
You can pause and scroll through the video to search for help on how to complete a specific section.

Create a Sketch for the Base

In this section of the exercise, you start a new part file and sketch the profile of the base.

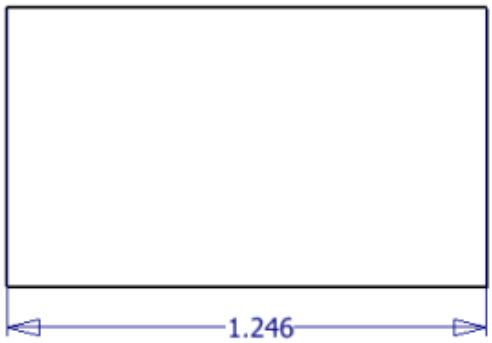
1. Make *IFI_Unit2.ipj* the active project.
2. On the Launch panel, click New.
3. Click the English tab. Double-click *Standard (in).ipt*.
4. On the Draw panel, click Two Point Rectangle.

5. To sketch the rectangle:
   - Click in the graphics window to set the first corner point (1).
   - Move the cursor diagonally. Click to set the second point (2).

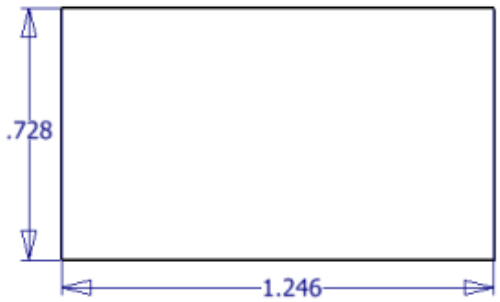


6. On the Constrain panel, click General Dimension.

7. To place the horizontal dimension:
   - Click the bottom line and drag to display the dimension.
   - Click to place the dimension.
   - Click the dimension value to open the Edit Dimension box.
   - Enter **1.246**.
   - Press ENTER or click the check mark.



8. To place the vertical dimension:
   - Click the left vertical line and drag to display the dimension.
   - Click to place the dimension.
   - Double-click the dimension value to open the Edit Dimension box.
   - Enter **0.728**.
   - Click the check mark.



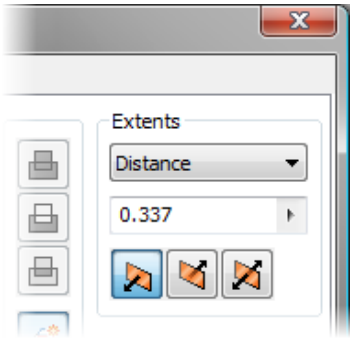
9. Press F6 to view the default Home view.
10. Press ESC to exit the General Dimension tool.
11. Right-click in the graphics window. Click Finish Sketch.

Extrude the Sketch

In this section of the exercise, you extrude the sketch to create the base. You then add fillets to two edges.

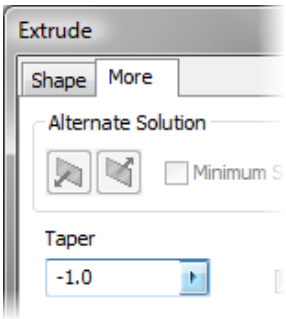
1. On the Create panel, click Extrude.

2. For Distance, enter **0.337**

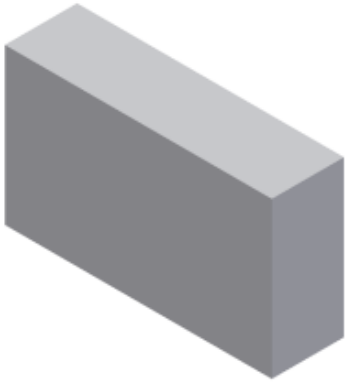


3. Click the More tab.

4. For Taper, enter **-1.0**

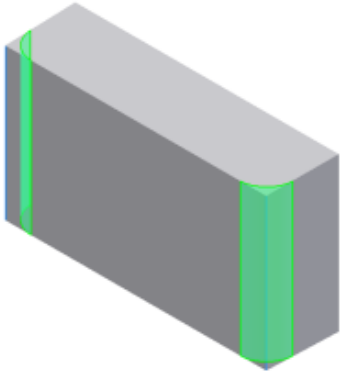


5. Click OK.

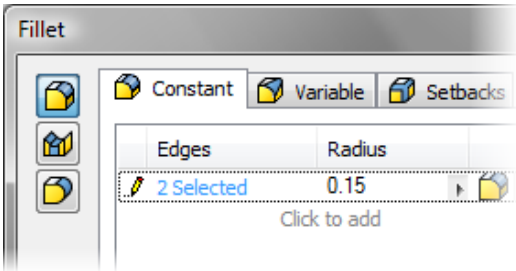


6. On the Modify panel, click Fillet.

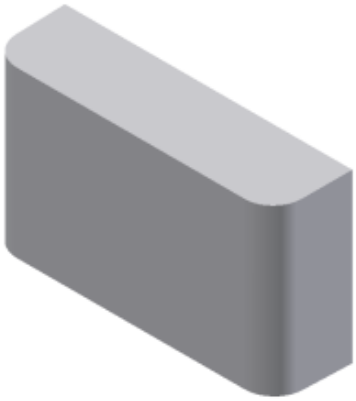
7. Select the two edges as shown.



8. For Radius, enter **0.15**



9. Click OK.



10. Save the file as *my_limitswitch_top.ipt*.

Create a Sketch for the Cutout

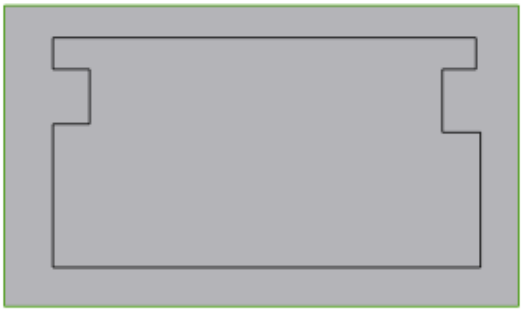
In this section of the exercise, you extrude the cutout on the back of the part.

1. On the ViewCube, click the top-right corner as shown.

2. On the ViewCube, click Back.
3. On the Sketch panel, click Create 2D Sketch.

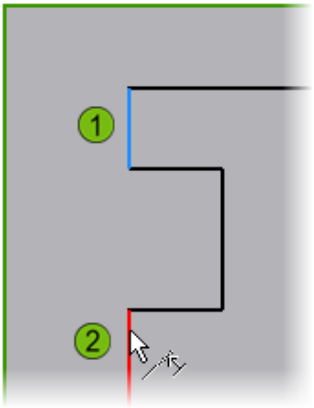
4. Select the back face of the part.
5. On the Draw panel, click Line.

6. Create the sketch as shown. Make sure that all the lines are vertical or horizontal.

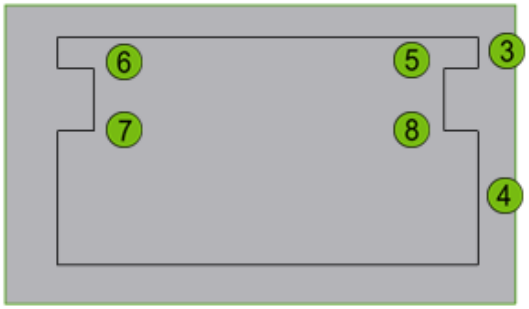


7. On the Constrain panel, click Colinear.

8. Select the line (1) as shown. Select the next line (2).

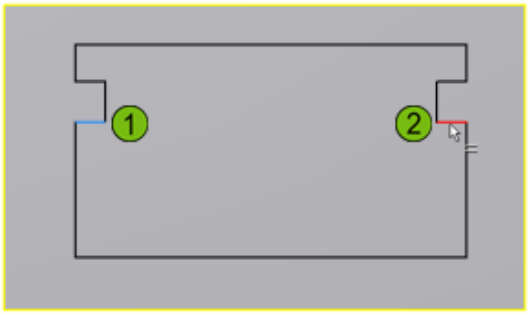


9. Repeat the workflow for:
   - The vertical lines 3 and 4.
   - The short horizontal lines 5 and 6.
   - The short horizontal lines 7 and 8.



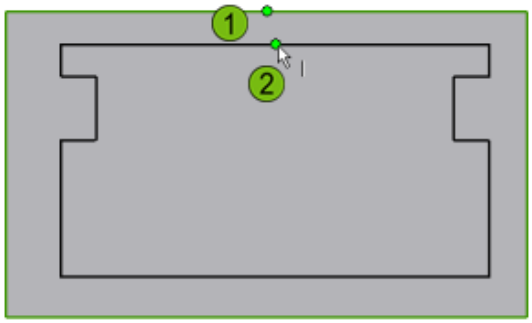
10. On the Constrain panel, click Equal.

11. Select the short horizontal lines 1 and 2.



12. On the Constrain panel, click Vertical.

13. Move the cursor over the center of the top edge of the part. Click when the large green dot is displayed (1). Click the midpoint on the sketch as shown (2).

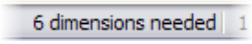


Extrude the Cutout

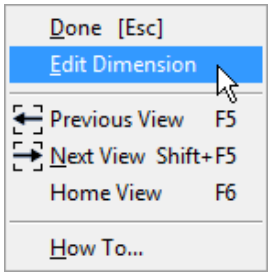
In this section of the exercise, you fully constrain the sketch by adding dimensions. You then extrude the sketch to make the cutout on the back of the part.

1. On the Constrain panel, click General Dimension.

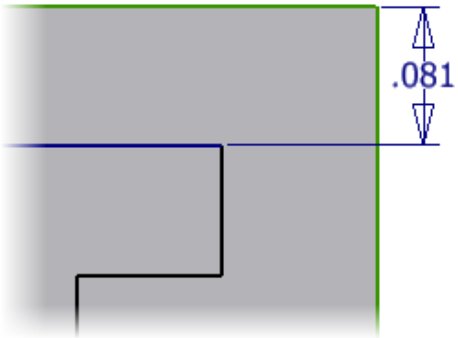
2. Look at the lower right of the graphics window. To fully constrain the sketch, six dimensions are needed.



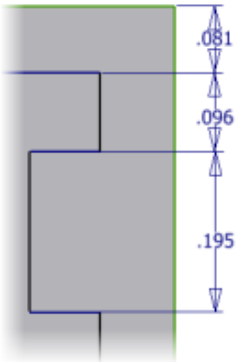
3. Right-click in the graphics window. Click Edit Dimension.



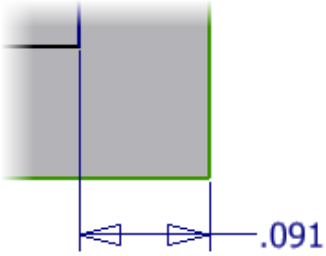
4. To place the first dimension:
   - Click the top edge of the part and top edge of the sketch to place the dimension. Now the Edit Dimension dialog box is displayed without clicking the dimension.
   - Enter **0.081**. Click the check mark.



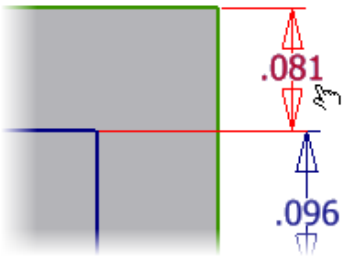
5. Add **0.096** and **0.195** dimensions as shown.



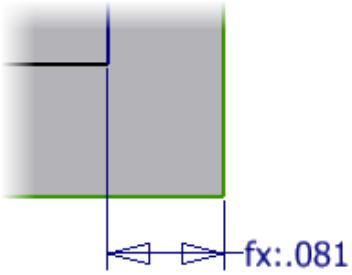
6. Place the dimension as shown. Do not click the check mark.



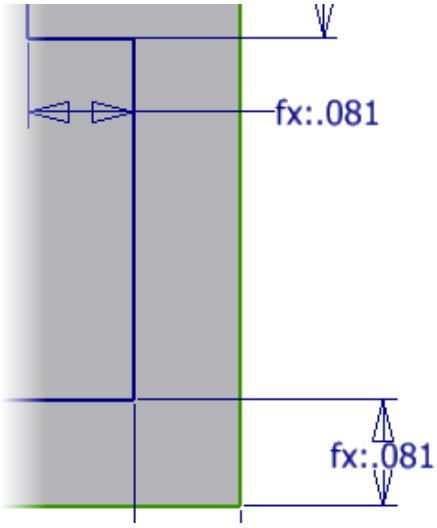
7. Move the cursor over the first 0.081 dimension. Click the dimension when the hand symbol is displayed.



8. Click the check mark. The new dimension has an fx: prefix to indicate it is a function of another dimension.



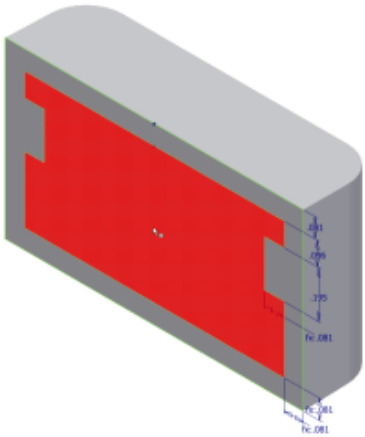
9. Repeat this workflow to place two more dimensions. The sketch is now fully constrained.



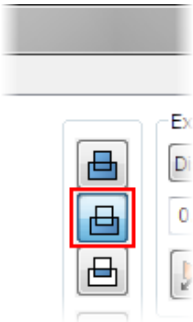
10. Press ESC to cancel the General Dimension tool.

11. On the ViewCube, click the top-right corner.

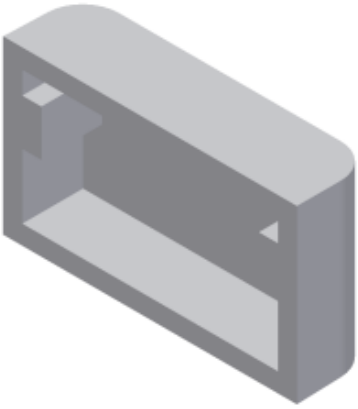
12. Press E to start the Extrude tool.
13. Select inside the sketch as the profile.



14. For Operation, click Cut.



15. For Distance, enter **0.258**.
16. Click OK.



17. Save the file.

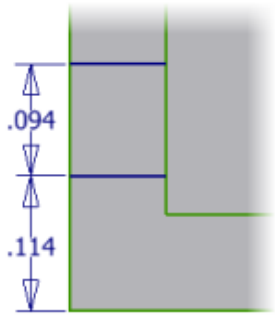
Extrude the Notch

In this section of the exercise, you extrude the notch on the wall of the part. The notch provides access for wiring to the limit switch microswitch.

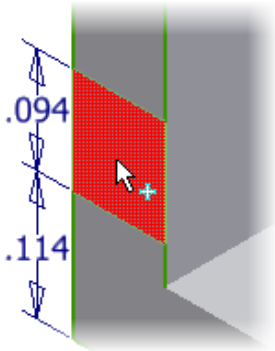
1. On the ViewCube, click Back.
2. On the Sketch panel, click Create 2D Sketch.

3. Select the face of the part. Make sure you select toward the outside edge as shown.

4. Create and dimension a sketch as shown. Make sure that the lines are horizontal.



5. On the ViewCube, click the top-right corner.
6. Press E to start the Extrude tool.
7. Select the sketch as the profile.



8. To create the extrusion:
   - For Operation, click Cut.
   - For Distance, enter **0.185**.
   - Click OK.

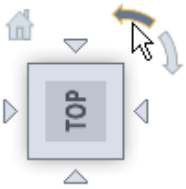
9. Save the file.

Extrude the Microswitch Opening

In this section of the exercise, you extrude the opening on the wall of the part. The opening is where the limit switch microswitch is located.

1. On the ViewCube, click Top. The part should be displayed as shown.

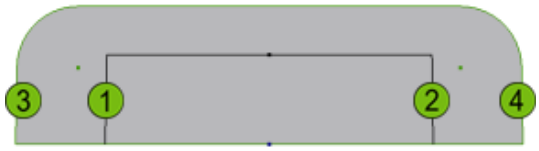
If necessary, click the rotation arrow on the ViewCube to position the part correctly.



2. On the Sketch panel, click Create 2D Sketch.

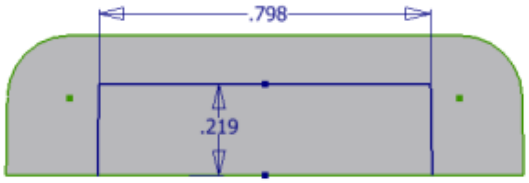
3. Select the top face of the part.

4. Sketch three lines as shown. When you sketch the shorter lines (1 and 2) exaggerate their angle to prevent a vertical, parallel or perpendicular constraint from being applied. You apply parallel constraints after you create the sketch.

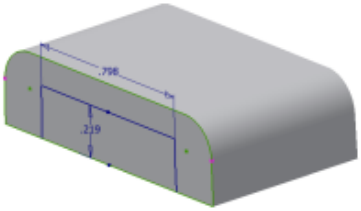


5. Apply parallel constraints between each of the short lines (1 and 2) and the adjacent part edges (3 and 4).
6. Apply a vertical constraint between the midpoint of the horizontal line and one of the horizontal edges so that the sketch is centered on the face.
7. Add **0.219** and **0.798** dimensions to the sketch as shown.

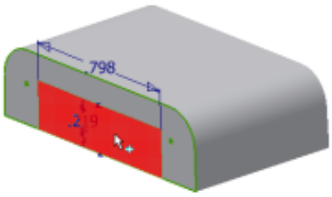
   Tip: Press D to start the General Dimension tool.



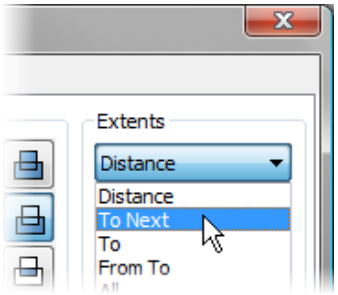
8. On the ViewCube, click the top-right corner.



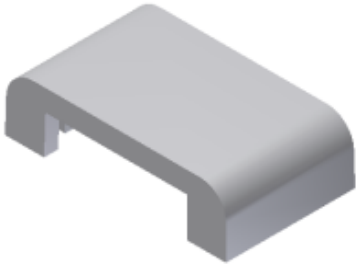
9. Press E to start the Extrude tool.
10. Select the sketch as the profile.



11. To create the extrusion:
    - For Operation, click Cut.
    - For Distance, select To Next.



12. Click OK.



13. Save the file.

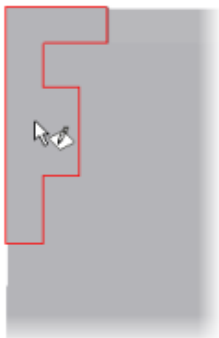
Create the Holes

In this section of the exercise, you create two holes. These holes are used to assemble the limit switch.

1. On the ViewCube, click the bottom left edge.

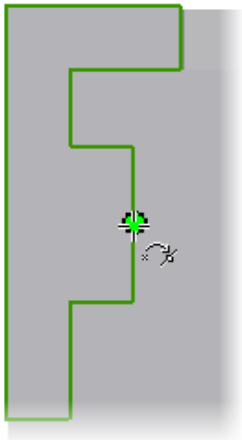
2. On the ViewCube, click Back.
3. On the Sketch panel, click Create 2D Sketch.

4. Select the face of the part as shown.



5. On the Draw panel, click Point.

6. Move the cursor over the midpoint of the vertical line. When the midpoint is displayed, click to place the point.

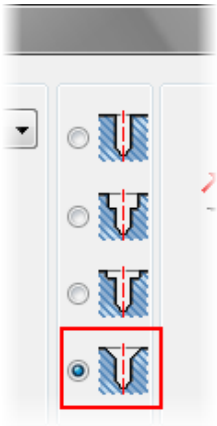


7. On the Draw panel, click Project Geometry.

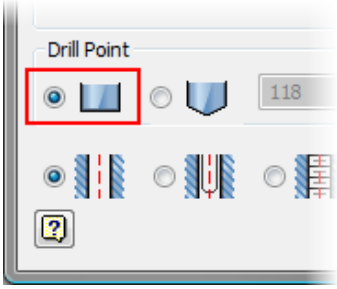
8. Select the line on the right side of the sketch as shown.

9. On the Draw panel, click Point. Place a center point on the midpoint of the line.

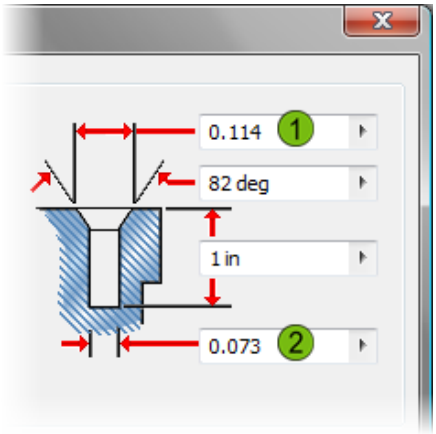
10. On the ViewCube, click the top right corner.
11. Press H to start the Hole tool. The center points are selected automatically.
12. Select Countersink as the hole type.



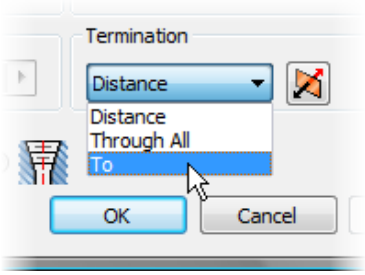
13. For Drill Point, select Flat.



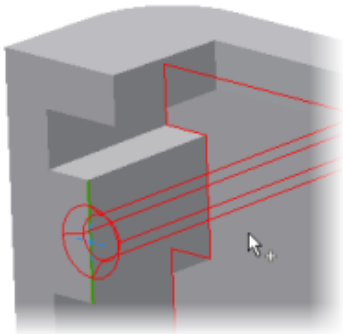
14. To set the hole dimensions:
    - For Countersink Diameter (1), enter **0.114**.
    - For Hole Diameter (2), enter **0.073**.



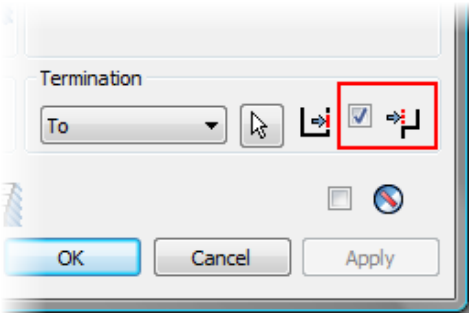
15. For Termination, select To.



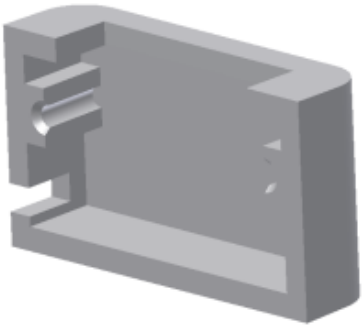
16. Select the bottom inside face of the part.



17. Select the Check to Terminate Feature on the Extended Face check box.



18. Click OK.



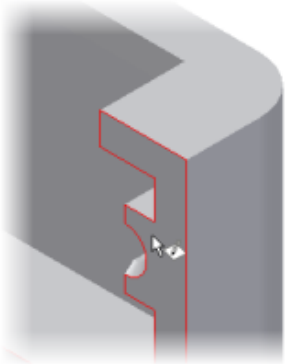
19. Save the file.

Create Two Locating Pins

In this section of the exercise, you extrude two locating pins.

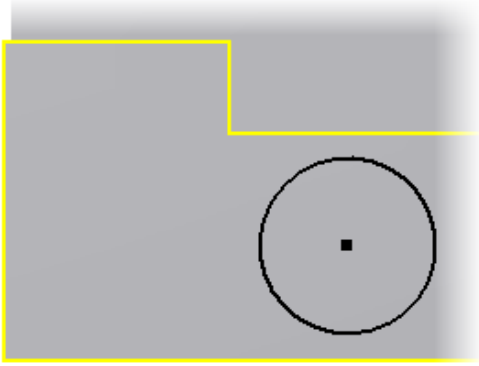
1. On the Sketch panel, click Create 2D Sketch.

2. Select the face on the right side of the part as shown.

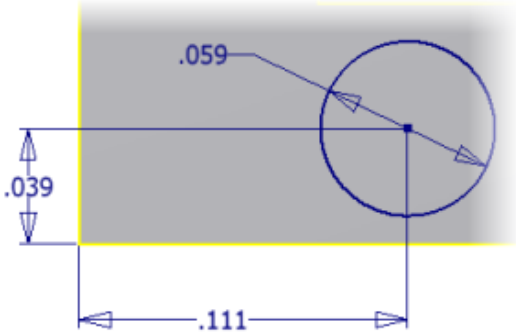


3. On the ViewCube, click Back.
4. On the Draw panel, click Circle.

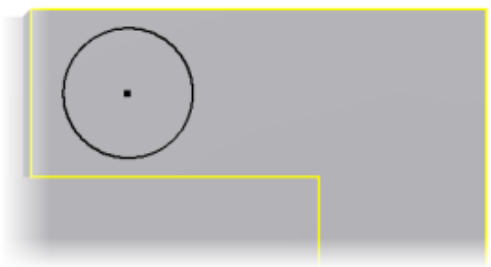
5. Create a circle at the bottom left of the part.



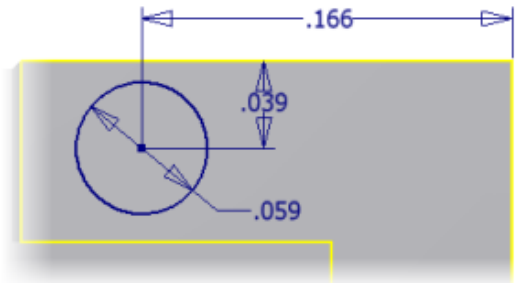
6. Dimension the circle as shown.



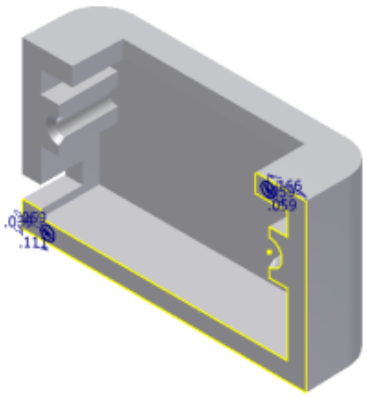
7. Create a circle at the top right of the part.



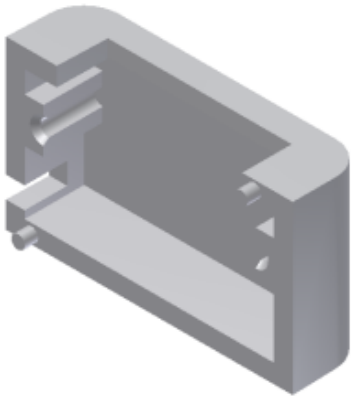
8. Dimension the circle as shown.



9. On the ViewCube, click the top-right corner. The part should be displayed as shown.



10. Press E to start the Extrude tool.
11. Select the circles as the profiles.
12. To create the extrusion:
    - For Distance, enter **0.059**
    - Click OK.



13. Save the file.

Add Fillets to the Part

In this section of the exercise, you add fillets to the part.

1. On the Modify panel, click Fillet.

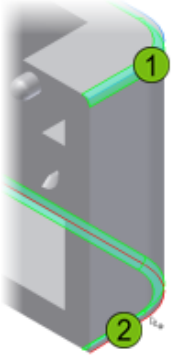
2. To create the fillets:
    - Select the edges of the two locating pins, 1 and 2.
    - For Radius, enter **0.025**.

3. Click OK.

4. On the Navigation toolbar, click Zoom All.

5. On the Modify panel, click Fillet.

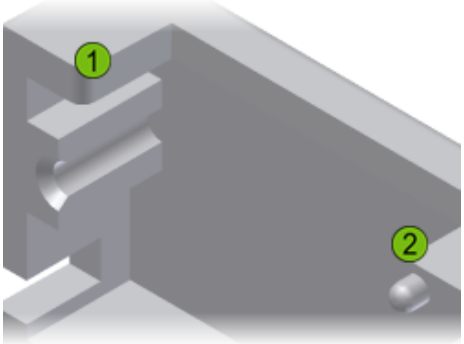
6. To create the fillets:
   - Select the edges. (1 and 2)
   - For Radius, enter **0.031**.



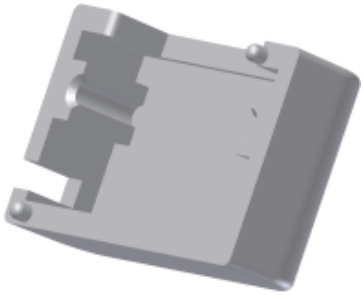
7. Click OK.

8. Create **0.030** fillets on the short vertical edges. (1 and 2)

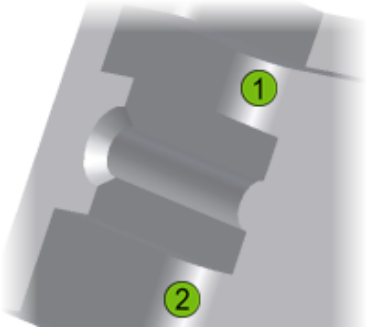
   Tip: If you select the wrong edges, deselect the edges by holding down the SHIFT key and selecting the edge.



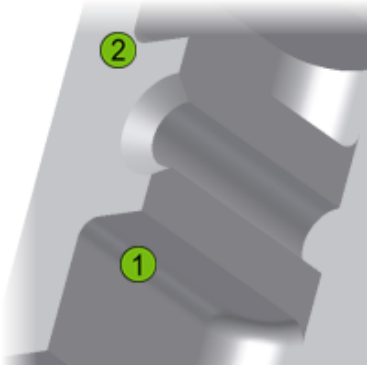
9. Rotate the part to view the inside edges.



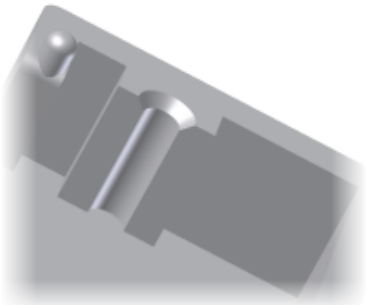
10. Create **0.060** fillets on the edges. (1 and 2)



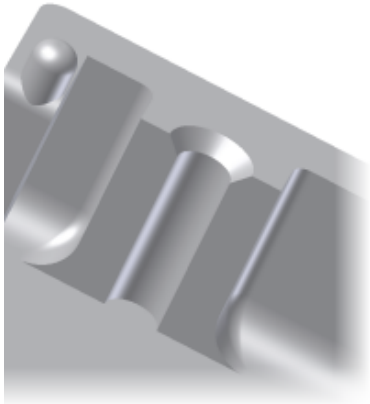
11. Create **0.021** fillets on the edges. (1 and 2)



12. Rotate the part to view the inside face on the opposite side of the part.



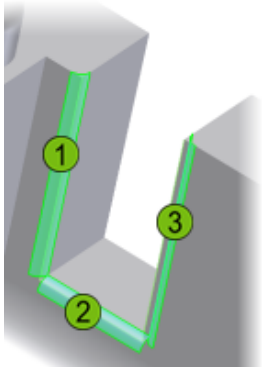
13. Repeat the workflow from the previous steps and add **0.060** and **0.021** fillets to the equivalent edges on the part.



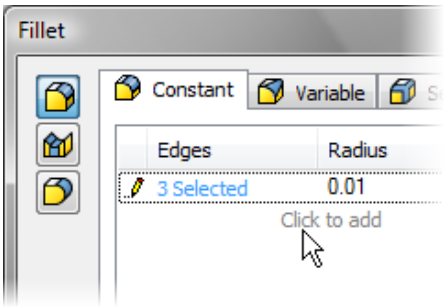
14. Rotate the part to view the cutout.

15. On the Modify panel, click Fillet.

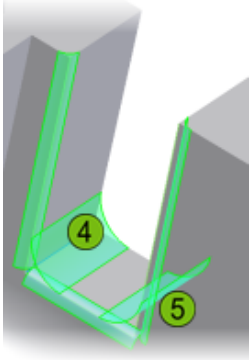
16. To select the first edge set:
    - Select edges 1, 2, and 3.
    - For radius, enter **0.01**.



17. In the Fillet dialog box, click Click to Add.



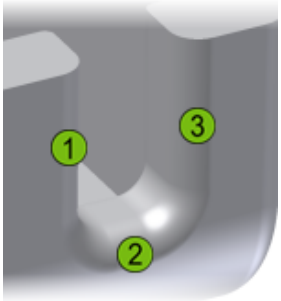
18. To select the second edge set:
    - Select edges 4 and 5.
    - For radius, enter **0.03**.



19. Click OK.

20. Rotate the part to view the outside of the cutout.

21. Create **0.03** fillets on the edges. (1, 2, and 3)



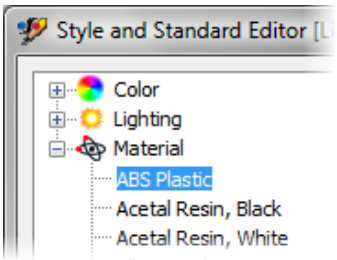
22. Save the file.

Change the Material of the Limit Switch Top

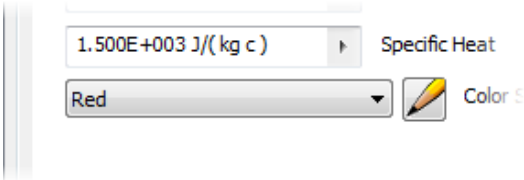
In this section of the exercise, you change the material of the limit switch top to ABS plastic.

1. Press F6 to view the default Home view.
2. On the Manage tab, Styles and Standards panel, click Styles Editor.

3. Expand Material. Click ABS Plastic.



4. For color, select Red.

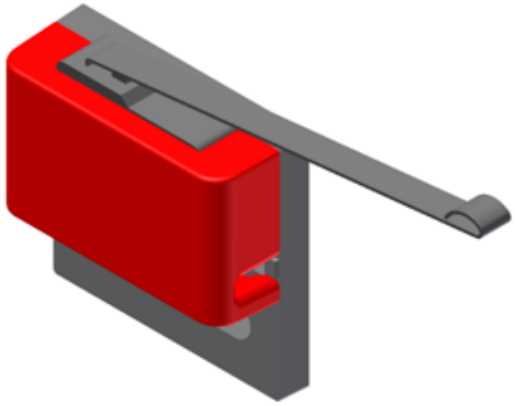


5. Click Save.
6. Click Done.
7. In the browser, right-click *my_limit_switch.ipt*. Click iProperties.
8. Click the Physical tab.
9. For Material, select ABS Plastic.
10. Click Apply. Note the updated properties of the part, such as Mass, Area, and Volume.
11. Click Close.

12. Save the file.
13. Close the file.

Exercise: Assemble a Limit Switch

In this exercise, you assemble the three parts of the limit switch.



The completed exercise

Before starting the exercise, you can refer to a video demonstration of the workflow. To view the demonstration, navigate to the Video Demonstration folder for this exercise.

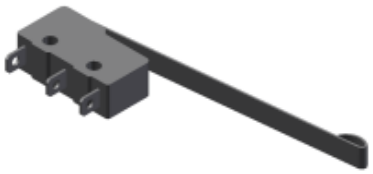
You can pause and scroll through the video to search for help on how to complete a specific section.

Place the Microswitch in the Assembly

In this section of the exercise, you create a new assembly file, and then place two parts of the limit switch in the assembly.

1. Make *IFI_Unit2.ipj* the active project.
2. On the Launch panel, click New.
3. Click the English tab. Double-click *Standard (in).iam*.
4. On the Component panel, click Place.

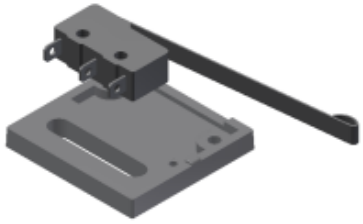
5. To place the first part:
   - Select *LIMIT-SWITCH-MICROSWITCH.ipt*.
   - Click Open.
   - Right-click in the graphics window. Click Done.



Note: The first part in the assembly is grounded. All degrees of freedom are removed. In the browser, the part has a thumbtack icon to indicate it is grounded.

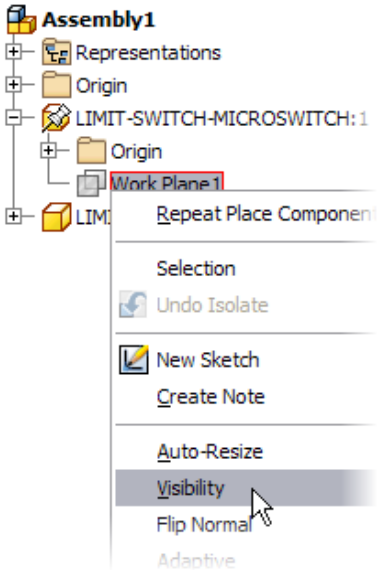
6. On the Component panel, click Place.

7. To place the second part:
   - Select *LIMIT-SWITCH-BOTTOM.ipt*.
   - Click Open.
   - Click to place an occurrence of the component.
   - Right-click in the graphics window. Click Done.



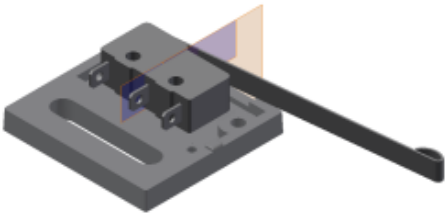
8. In the browser, expand the LIMIT-SWITCH-MICROSWITCH:1 listing.

9. Right-click Work Plane1. Click Visibility to turn on visibility.



10. Repeat this workflow for LIMIT-SWITCH-BOTTOM:1.
11. On the Position panel, click Constrain.

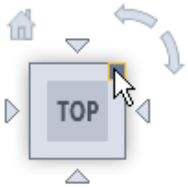
12. To place the constraint:
    - Select the edge of the workplane on the LIMIT-SWITCH-MICROSWITCH.
    - Select the edge of the workplane on the LIMIT-SWITCH-BOTTOM.
    - Click OK.



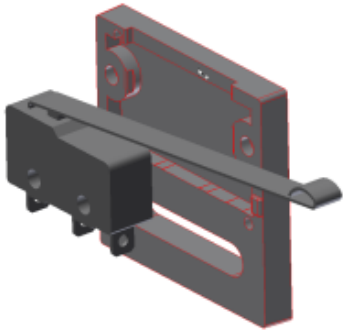
13. Using the same workflow from the previous steps, turn off the visibility of the work planes.
14. Save the file as *my_limitswitch.iam*.

Assemble the Parts

1. On the ViewCube, click Top.
2. On the ViewCube, click the top right corner.

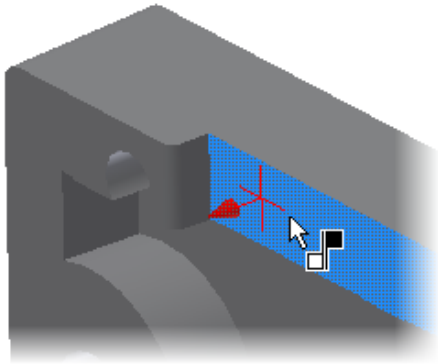


3. Drag the LIMIT-SWITCH-BOTTOM away from the assembly.

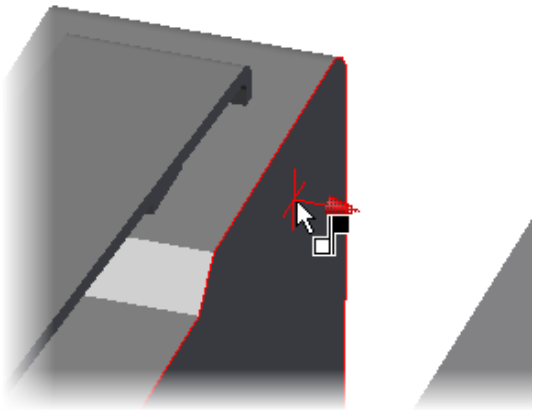


4. On the Position panel, click Constrain.

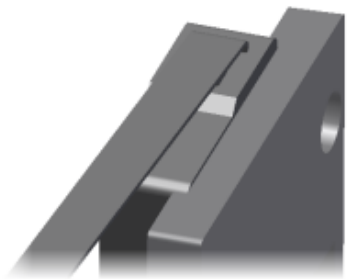
5. Select the face of the LIMIT-SWITCH-BOTTOM.



6. Rotate the assembly. Select the face of the LIMIT-SWITCH-MICROSWITCH.



7. Click OK.

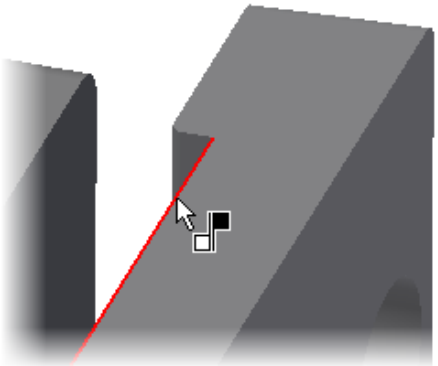


8. Drag the LIMIT-SWITCH-BOTTOM. It is free to move up and down.
9. On the Position panel, click Move.

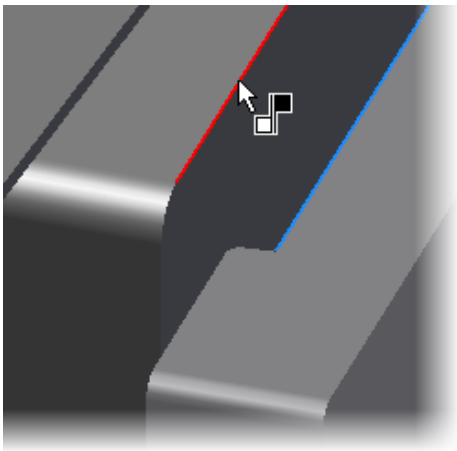
10. Drag the LIMIT-SWITCH-BOTTOM away from the assembly.

11. On the Position panel, click Constrain.

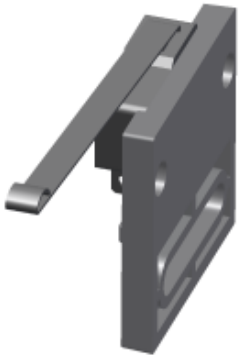
12. Select the edge of LIMIT-SWITCH-BOTTOM. Make sure the edge is highlighted, not the face.



13. Select the edge of LIMIT-SWITCH-BOTTOM.



14. Click OK.



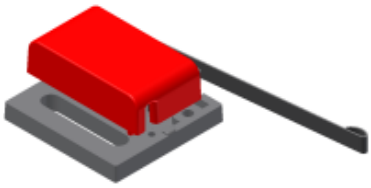
15. Drag the LIMIT-SWITCH-BOTTOM. It cannot move. All degrees of freedom have been removed.
16. Save the file.

Place the Top of the Limit Switch in the Assembly

In this section of the exercise, you place an occurrence of the top part of the limit switch in the assembly.

1. On the ViewCube, click Home.
2. On the Component panel, click Place.

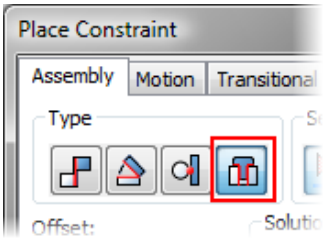
3. To place the third part:
   - Select *LIMIT-SWITCH-TOP.ipt*.
   - Click Open.
   - Click to place an occurrence of the component.
   - Right-click in the graphics window. Click Done.



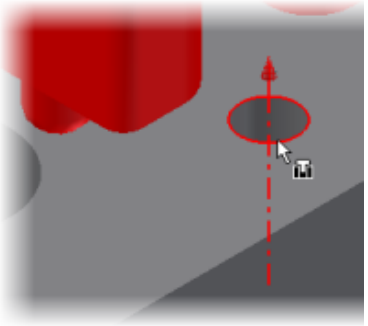
4. Zoom into the area around the locating pin and hole. If the hole is not visible, drag LIMIT-SWITCH-TOP so that the hole is visible.

5. On the Position panel, click Constrain.

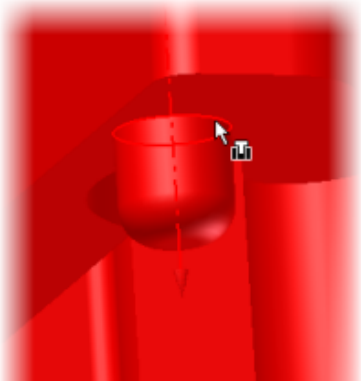
6. Under Type, click Insert.



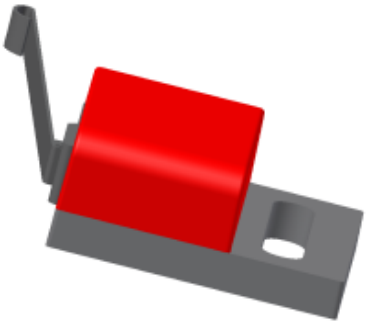
7. Select the edge of the locating hole.



8. Rotate the assembly. Select the edge of the locating pin.

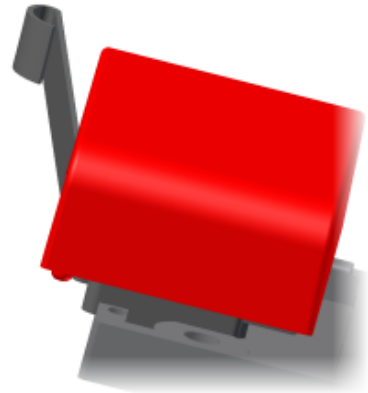


9. Click OK.
10. Rotate the assembly to view the opposite side of LIMIT-SWITCH-TOP.



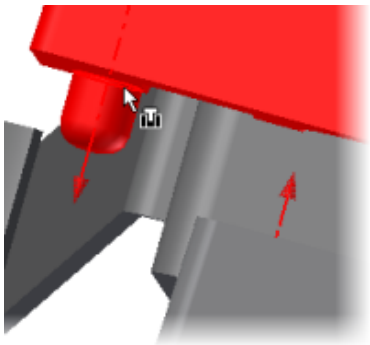
11. On the Position panel, click Move.

12. Drag the LIMIT-SWITCH-TOP away from the assembly.



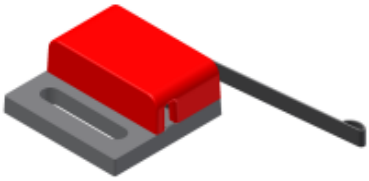
13. On the Position panel, click Constrain.

14. Place an insert constraint between the hole and the locating pin.



15. Click OK.

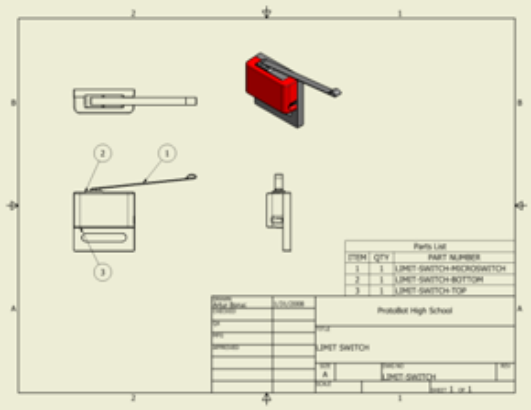
16. On the ViewCube, click Home.



17. Save the file.
18. Close the file.

Exercise: Create a Drawing of the Limit Switch

In this exercise, you create a drawing of the limit switch. You also add balloons and a parts list to the drawing.



The completed exercise

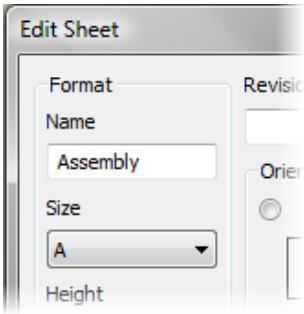
Before starting the exercise, you can refer to a video demonstration of the workflow. To view the demonstration, navigate to the Video Demonstration folder for this exercise.

You can pause and scroll through the video to search for help on how to complete a specific section.

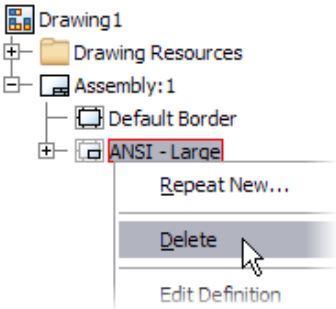
Create a New Drawing

In this section of the exercise, you create a new drawing file, rename the sheet, and insert a different title block.

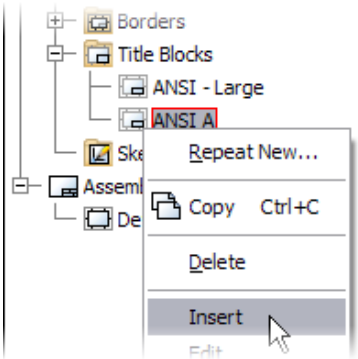
1. Make *IFI_Unit2.ipj* the active project.
2. On the Launch panel, click New.
3. Click the English tab. Double-click *ANSI (in).idw*.
4. In the browser, right-click Sheet:1. Click Edit Sheet.
5. In the Edit Sheet dialog box:
   - For Name, enter **Assembly**.
   - From the Size list, select A.
   - Click OK.



6. In the browser, right-click ANSI - Large. Click Delete.



7. In the browser, expand Drawing Resources > Title Blocks. Right-click ANSI A. Click Insert.



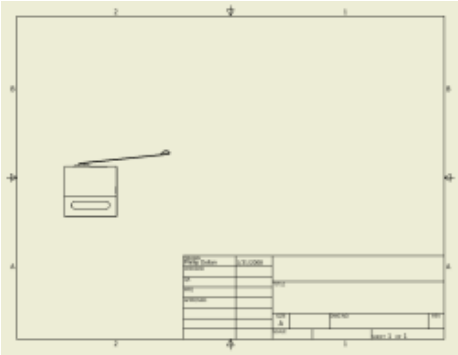
8. Save the file as *my_limitswitch.idw*.

Create the Drawing Views

In this section of the exercise, you create four views of the assembly.

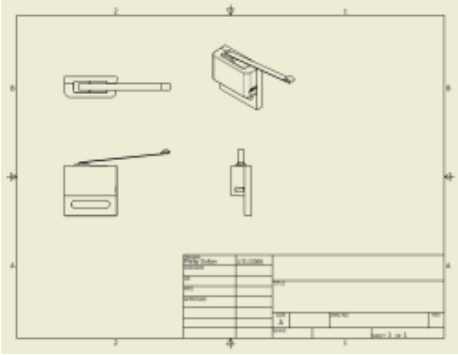
1. On the Create panel, click Base.

2. To create the base view:
   - Click Open an Existing File.
   - Select *LIMIT-SWITCH.iam*.
   - Click Open.
   - Under Orientation, click Top.
   - Click to place the view.

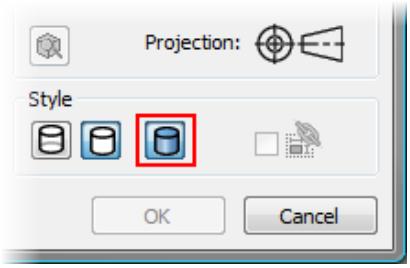


3. On the Create panel, click Projected.

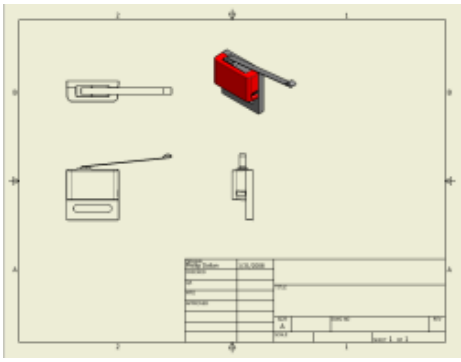
4. To create the projected views:
   - Select the base view. Move the preview upward.
   - Click to place the top view.
   - Move the cursor to the right of the base view.
   - Click to place the right side view.
   - Move the cursor up and to the right of the base view.
   - Click to place an isometric view.
   - Right-click in the graphics window. Click Create.



5. Move the cursor over the base view to display the red dotted line. Right-click the drawing. Click Edit View.
6. On the Display Options tab, select the Tangent Edges check box to turn on tangent edges display.
7. Click OK. The edges of the fillets are displayed.
8. Move the cursor over the isometric view to display the red dotted line. Right-click in the view. Click Edit View.
9. Under Style, click Shaded.



10. Click OK.

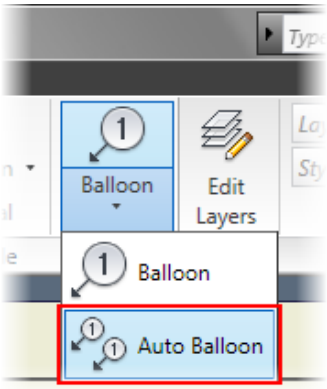


11. Save the file.

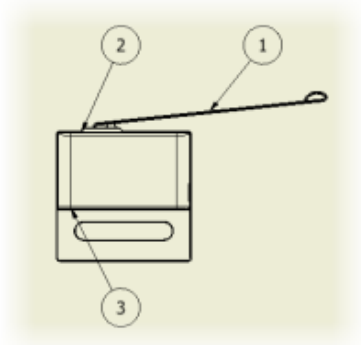
Add Balloons and a Parts List

In this section of the exercise, you add balloons and a parts list.

1. On the Annotate tab, Table panel, click Auto Balloon.

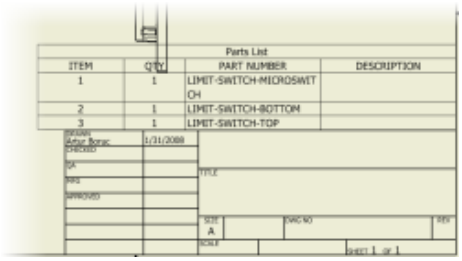


2. To add the balloons:
   - Select the front view.
   - Right-click the front view. Click Select All.
   - Right-click in the graphics window. Click Continue.
   - In the Auto Balloon dialog box, under Placement, click Around.
   - Click to place the balloons.
   - Click OK.
   - If a dialog box is displayed, click OK.

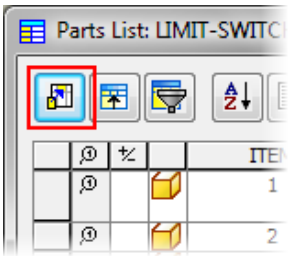


3. On the Table panel, click Parts List.

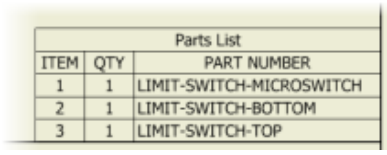
4. To create the parts list:
   - Select the front view.
   - Click OK.
   - Click to place the parts list over the title block.



5. Right-click the parts list. Click Edit Parts List.
6. Click Column Chooser.



7. To delete the Description column:
   - Under Selected Properties, select Description.
   - Click Remove.
   - Click OK.
8. To edit the column width:
   - Right-click the ITEM column header. Click Column Width.
   - For Column Width, enter **0.5**.
   - Click OK.
9. Repeat this workflow to set the column width for QTY to **0.5** and PART NUMBER to **2.5**.
10. Click OK to close the Parts List dialog box. If necessary, drag the parts list to its original location.



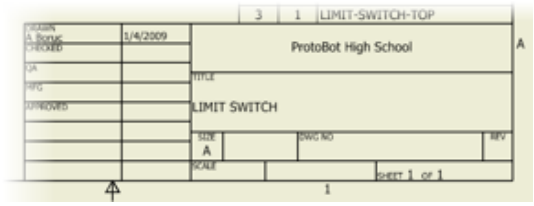
Parts List		
ITEM	QTY	PART NUMBER
1	1	LIMIT-SWITCH-MICROSWITCH
2	1	LIMIT-SWITCH-BOTTOM
3	1	LIMIT-SWITCH-TOP

11. If necessary, move the drawing views away from the parts list.
12. Save the file.

Edit the Properties

In this section of the exercise, you edit the drawing properties.

1. In the browser, right-click *my_limitswitch.idw*. Click iProperties.
2. To edit the Summary Properties:
   - Click the Summary tab.
   - For Title, enter **LIMIT SWITCH**.
   - For Author, enter your name.
   - For Company, enter your school name.
   - Click OK.



3. Save the file.
4. Close the file.