Autodesk Inventor 2009

Getting Started

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Contents

Introducing Autodesk Inventor

1

Welcome to Autodesk® Inventor™. This book explains the fundamental skills to start using Autodesk Inventor. In these chapters, the basic features are presented through examples and step-by-step procedures. The data files used in the procedures are installed with the Autodesk Inventor software.

Getting Started

Autodesk Inventor provides options during installation. The options selected determine what you see the first time you start Autodesk Inventor. If you indicate during installation that you are a new or returning Autodesk Inventor user, you are presented with the Open dialog box. If you indicated that you are transitioning to Autodesk Inventor from AutoCAD®, an empty part file is displayed (like opening a new DWG file during AutoCAD start-up). The main Autodesk Inventor Help page is displayed with slightly different selections depending on the selected install options. No matter how Autodesk Inventor was installed, you can tailor the start-up experience through settings in the Application Options dialog box to suit your needs. You can specify that Autodesk Inventor always starts with the Open dialog box or always starts in a new file. You can decide if you want to see Help on start-up or not (and which version of the main Help page to see).

Projects

Autodesk Inventor uses projects to represent a logical grouping of a complete design project. A project organizes your data by maintaining information about where design data is stored, where you can edit files, and maintains valid links between them. You use projects when you work in a team, work on multiple design projects, and share libraries among several design projects. See ProductName Utilities on page 273, for detailed information about setting up and using projects.

Data Files for Exercises

When you install Autodesk Inventor, a project called tutorial_files is created. Make this project active so that you can locate the data files that are used for some exercises in this book.

TRY IT: Make the tutorial_files project active

1. In Autodesk Inventor, on the Standard toolbar, click Files ➤ Projects.
2. On the Project Editor, Select Project pane, double-click the tutorial_files project to make it the active project.

   In the Edit Project pane, in Location, the path to the folder containing the tutorial data files is displayed. It is the folder where the files you create and edit while performing the exercises are saved.
3. Close the Project Editor dialog box.
4. Click File ➤ Open.

   The data files contained in the tutorial_files project are listed in the Open File dialog box.
5. Click a file to see a preview of it, and double-click a file to open it in Autodesk Inventor.

File Types

Once you activate a project, you can open an existing file or start a new file. Click New to see the New File dialog box with templates for a new part, assembly, presentation file, sheet metal part, weldment, or drawing. You can choose from several templates with predefined units.

Templates are stored in the *Autodesk\Inventor(version number)\Templates* directory or in the *English* or *Metric* subdirectories. Subdirectories in the *Templates* directory are displayed as tabs in the Open New File dialog box. You can create and save custom templates in the *Templates* directory.

A template can contain property information, such as part and project data, and drawing views. You can see information stored in a file by viewing its properties.

TRY IT: View the Properties dialog box

- With a file open, right-click a component in the browser or in the graphics window, and then choose Properties from the menu.
- Click the tabs to see properties.

Application Options

You can change the look and feel of Autodesk Inventor using settings on the Application Options dialog box. On the Standard toolbar, select Tools ➤ Application Options. Use the tabs on the Options dialog box to control the color and display of your Autodesk Inventor work environment, the behavior and settings of files, the default file locations, and a variety of multiple-user functions.

Application options remain in effect until you change them.

Document Settings

You can specify settings in individual files. On the Standard toolbar, select Tools ➤ Document Settings to display the Document Settings dialog box. Click the tabs to view and specify settings for the active document, such as indicating the active styles, units of measure, sketch and modeling preferences, bill of materials, and default tolerance.

Styles and Standards

You select a drafting standard when you install Autodesk Inventor, and it includes a default set of styles that control most objects used in documents, such as balloons, dimensions, text, layers, parts lists, symbols and leaders, materials, and lighting. Usually the default styles are enough to get you started, but you can use the Styles and Standards Editor to create, modify, and purge unused styles.

By default, actions such as creating or modifying styles affect only the current document. You can choose to save the style to the style library, a master library that contains definitions for all available styles associated with a drafting standard. Usually, the style library is managed by a CAD administrator. This practice ensures that the style definitions, used by all documents that use the drafting standard, are not accidentally replaced by a custom style.

Style libraries make it easy to share formatting conventions across projects because they contain the definitions of formatting objects. Using a style library, you can update a style for all documents, such as revising the arrow heads of dimensions, by editing the style and saving the revision to the master style library. All documents that use that drafting standard have access to the library and any new or changed styles that are added to it.

TRY IT: View the Styles and Standards Editor dialog box

1. In Autodesk Inventor, click File ➤ New and select the drawing template.
2. On the Standard toolbar, click Format ➤ Styles Editor.
3. On the Styles and Standards Editor dialog box, click Standard in the Style Type browser, and then double-click a listed standard.
4. Click the General tab to see the values controlled there, and then click the Available Styles tab to see the list of styles. As you click through the style type list, you may notice that most names are checked. If the check box is cleared, that style is not available for use in the current document.
5. In the left pane of the Styles and Standards Editor, click the Dimension style, and then double-click one of the dimension styles to display it in the right pane. Click through the tabs to see the settings for units, alternate units, text, tolerance, options, and notes and leaders. Click a different dimension style to see if any of the values differ.
6. In the top-right corner of the dialog box, click the Filter list and change the filter type. Notice how the list of available styles changes if you select All Styles, Local Styles (for the current document), or Active Standard.

   You may notice differences in the lists because the local styles may have had some unused styles purged to make the file size smaller.
7. Click Done. Any changed values are discarded.

   If you click Save to preserve changes, the changes are saved only in the current document.

Using Shortcut Keys and Command Aliases

Autodesk Inventor provides shortcut keys and command aliases to help you perform certain tasks more quickly. A command alias is an alphanumeric character or character sequence used to start a command. Define a shortcut by using any of the following keys or key combinations:

- A punctuation key (including ` - = [ ] \ ; ' , . /), or one of the following virtual keys: Home, End, Page Up, Page Down, Up Arrow, Down Arrow.
- A combination of the SHIFT key along with a numeric key (0-9), punctuation key, or one of the following virtual keys: Home, End, Page Up, Page Down, Up Arrow, Down Arrow.
- Any combination of SHIFT, CTRL, and ALT keys along with an alphanumeric character.

Remember that some shortcut keys and command aliases are active in specific environments only.

TRY IT: View a complete guide to shortcut keys and command aliases

1. Open Autodesk Inventor.
2. On the Standard menu, click Tools ➤ Customize ➤ Keyboard tab. For each category, there is a list of the command name and its associated shortcut or alias, if one exists.
3. Click through several categories to see the associated commands.

The following is a list of some of the commonly used shortcut keys and command aliases.

Key	Result
F1	Displays Help for the active command or dialog box.
F2	Pans the graphics window.
F3	Zooms in or out in the graphics window.
F4	Rotates objects in the graphics window.

Key	Result
F5	Returns to the previous view.
F6	Returns to isometric view.
B	Adds a balloon to a drawing.
C	Adds an assembly constraint.
D	Adds a dimension to a sketch or drawing.
DO	Adds an ordinate dimension to a drawing.
E	Extrudes a profile.
FC	Adds a feature control frame to a drawing.
H	Adds a hole feature.
L	Creates a line or arc.
P	Places a component in the current assembly.
R	Creates a revolved feature.
S	Creates a 2D sketch on a face or plane.
T	Tweaks a part in the current presentation file.
ESC	Quits a command.
DELETE	Deletes selected objects.
BACKSPACE	In the active Line tool, removes the last sketched segment.
ALT + drag mouse	In assemblies, applies a mate constraint.

Key	Result
	In a sketch, moves spline shape points.
SHIFT + right-click	Activates the Select tool menu.
SHIFT + Rotate tool	Automatically rotates model in graphics window. Click to quit.
CTRL + ENTER	Return to previous editing state.
CTRL + Y	Activates Redo (revokes the last Undo).
CTRL + Z	Activates Undo (revokes the last action).
Spacebar	When the 3D Rotate tool is active, switches between dynamic rotation and standard isometric and single plane views.

NOTE Click Help ➤ Shortcut Quick Reference to see the list of command names and associated shortcuts and aliases in the active environment.

Viewing Models

Use viewing tools to view a model:

- Use the ViewCube to orbit your 3D model and to switch between standard and isometric views.
- Use the SteeringWheels to access a variety of navigational tools.
- Select one of the viewing tools in the Standard toolbar to achieve a specific view.
- Right-click in the graphics window, and then select Isometric View from the menu. The view vector changes to the isometric orientation.
- Right-click in the graphics window, and then select Previous View from the menu. The view changes back to the previous view.
- Press F5 to return the model to the last view.

To rotate a view in 3D, use the Free Orbit or Constrained Orbit tool in the Standard toolbar to rotate a view around one of the coordinate axes.

Zoom Tools

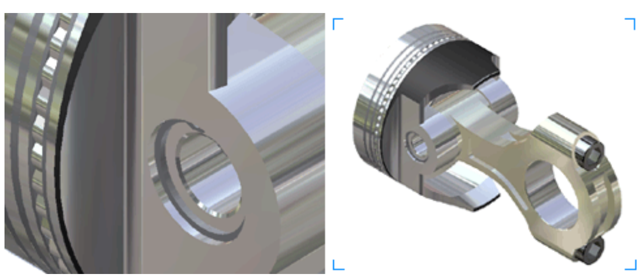
The zoom tools are located in the Standard toolbar and are also available from the SteeringWheels.

Zoom

Use the Zoom tool on the Standard toolbar to enlarge or reduce the image in the graphics window. Click the tool. In the graphics window, press the cursor as you move it up or down to zoom the view dynamically in or out. You can zoom the view while other tools are active.

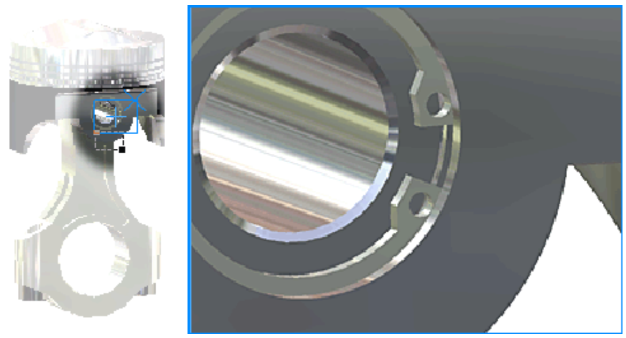
Zoom All

Use the Zoom All tool on the Standard toolbar to resize the image of a part or assembly so that all elements are displayed in the graphics window. You can zoom a drawing so that the active sheet fits within the graphics window.



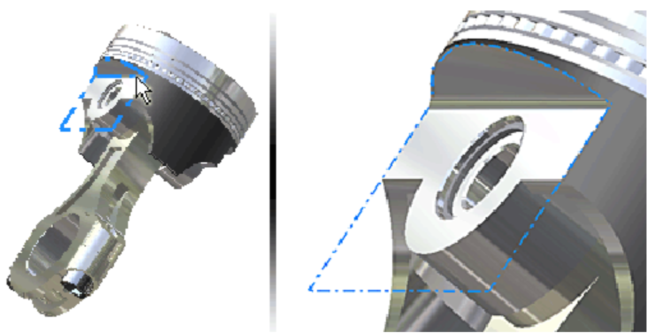
Zoom Window

Use the Zoom Window tool on the Standard toolbar to define an area of a part, assembly, or drawing to fill the graphics window.



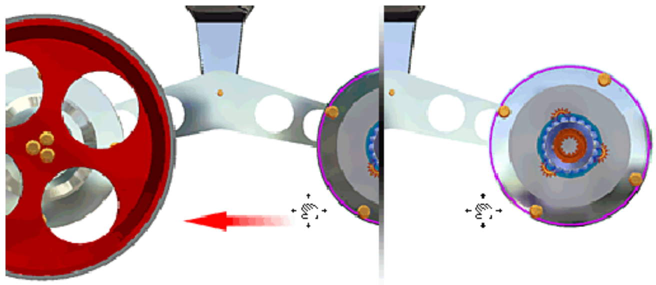
Zoom Selected

Use the Zoom Selected tool on the Standard toolbar to zoom a selected edge, feature, or other element to the size of the graphics window.



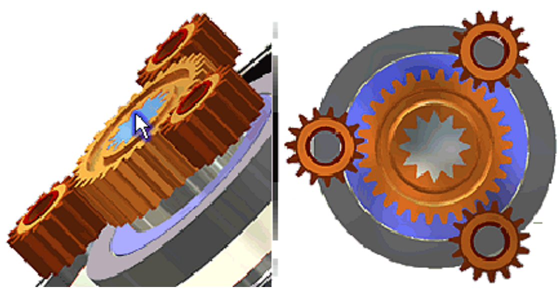
Pan

Use the Pan tool on the Standard toolbar to move the view in the graphics window in any direction planar to the screen. You can pan the view while other tools are active.



Look At

Use the Look At tool on the Standard toolbar to zoom and rotate the display in the graphics window. You can position a selected planar element parallel to the screen or position a selected edge or line horizontal to the screen.



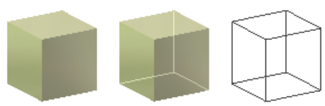
Rotate

Use the Orbit tools on the Standard toolbar to:

- Rotate a part or assembly in the graphics window.
- Display standard, isometric, and single plane projections of a part or assembly.
- Redefine the isometric view.

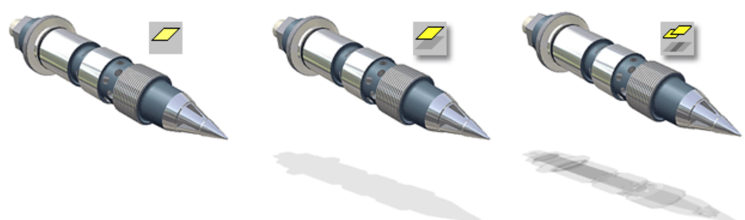
Shaded, Hidden Edge, and Wireframe Display

Use one of the Change Display tools to switch among the three display modes: Shaded, Hidden Edge, and Wireframe. You can apply display modes to part and assembly models, and to views in the Engineer's Notebook.



Ground Shadow Display

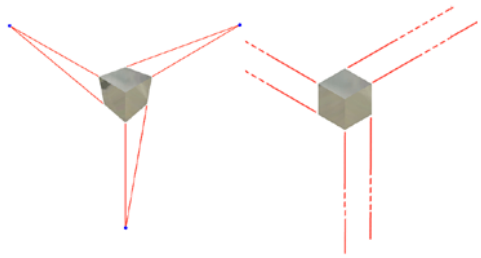
Use the Ground Shadow tool to cast a shadow on the plane beneath the model.



Orthographic and Perspective Camera Views

The Camera View tool has two settings: Orthographic Camera mode and Perspective Camera mode.

In Perspective Camera mode, part or assembly models are displayed in three-point perspective, a visual effect where parallel lines converge on a vanishing point. It is the way real objects are perceived by the human eye or by a camera.



The following chart shows how the other viewing tools behave and how they can be modified in each camera mode.

Zoom or Pan Type	Orthographic Camera mode	Perspective Camera mode	Keys/Commands
Camera Translation Pan	Yes	Yes	F2 Pan
Camera Pivot Pan	Yes	Yes	SHIFT+F2 Pan
Camera Position Zoom	Yes	Yes	F3 Zoom
Camera Position/Camera Target Point Zoom	No	Yes	SHIFT+F3 Zoom
Lens Focal Length Zoom	No	Yes	CTRL+F3 Zoom
Set Perspective Distortion			SHIFT+CTRL+F3 Zoom

Importing and Exporting Data

You can import Pro/ENGINEER®, Parasolid®, SolidWorks™, UGS NX, SAT, STEP, IGES, and AutoCAD and Autodesk® Mechanical Desktop® (DWG) files for use in Autodesk Inventor. You can export AAutodesk Inventor parts and assemblies to many file formats, including Pro/ENGINEER and Parasolid, and you can export Autodesk Inventor drawings as DXF™ or AutoCAD drawing (DWG) files.

The options for importing and saving AutoCAD files in Autodesk Inventor are:

- Selection of layers.
- Window selection of entities.
- Saving files in DWG format.
- Support for DXF files back to version 12.
- Creation of AutoCAD® Mechanical files, if AutoCAD Mechanical is installed.

NOTE Mechanical Desktop files can be linked to Autodesk Inventor assemblies without importing.

AutoCAD Files

When you open an AutoCAD file in Autodesk Inventor, you can specify the AutoCAD data to translate. You can select:

- Model space, a single layout in paper space, or 3D solids.
- One or more layers.

You can also place 2D translated data:

- On a sketch in a new or existing drawing.
- As a title block in a new drawing.
- As a sketched symbol in a new drawing.
- On a sketch in a new or existing part.

If you translate 3D solids, each solid becomes a part file containing an ASM solid body. Blocks are translated as sketched symbols.

When you import AutoCAD (DWG) drawings into a part sketch, a drawing, or a drawing sketch overlay, the converter takes the entities from the *XY* plane of model space and places them on the sketch. In a drawing, certain entities, such as splines, cannot be converted.

When you export Autodesk Inventor drawings to AutoCAD, the converter creates an editable AutoCAD drawing and places all data in paper space or model space in the DWG file. If the Autodesk Inventor drawing has multiple sheets, each is saved as a separate DWG file. The exported entities become AutoCAD entities, including dimensions.

You can open a *.dwg* file and then copy selected AutoCAD data to the clipboard and paste into a part, assembly, or drawing sketch. The data is imported at the cursor position.

Autodesk Mechanical Desktop Files

Autodesk Inventor can translate Autodesk Mechanical Desktop parts and assemblies so the design intent is preserved. You can import a Mechanical Desktop file as either an ASM body or a full conversion when Mechanical Desktop is installed and running on your system. Using the DWG/DXF File Import Wizard, you can import Mechanical Desktop data, including parts, assemblies, and drawings. The data is associative to Autodesk Inventor drawing views and annotations.

Features that are supported in Autodesk Inventor are converted. Unsupported features t are not translated. If Autodesk Inventor cannot translate a feature it skips that feature, places a note in the browser, and then completes the translation.

Files from Other Applications

You can open and change models created in Pro/ENGINEER, Parasolid, SolidWorks, and UGS NX. Autodesk Inventor translates assembly and part files, solids, multi-solids, surfaces, and more. After the import operation is complete, you can change the model as if it was originally created in Autodesk Inventor.

After changing a file, you can continue to use it as an Autodesk Inventor file or export it to other file formats, including Pro/ENGINEER and Parasolid.

SAT Files

SAT (*.*sat*) files contain nonparametric solids that can be Boolean solids or parametric solids with the relationships removed. You can use a SAT file in an assembly and add parametric features to the base solid.

When you import a SAT file that contains a single body, it produces an Autodesk Inventor part file with a single part. If it contains multiple bodies, it produces an assembly with multiple parts. Surface data in a SAT file is also supported.

STEP Files

STEP files are the international format developed to overcome some of the limitations of data conversion standards. Past efforts in developing standards have resulted in localized formats such as IGES (U.S.), VDAFS (Germany), or IDF (for circuit boards). Those standards do not address many developments in CAD systems. The STEP converter for Autodesk Inventor is designed for effective communication and reliable interchange with other CAD systems.

When you import a STEP (*.*stp*, *.*ste*, *.*step*) file, only 3D solid, part, surface, and assembly data are converted. Drafting, text, and wireframe data are not processed by the STEP converter. If a STEP file contains one part, it produces an Autodesk Inventor part file. If it contains assembly data, it produces an assembly with multiple parts.

IGES Files

IGES (*.*igs*, *.*ige*, *.*iges*) files are a standard in the United States. Many NC/CAM software packages require files in IGES format. Autodesk Inventor imports and exports IGES files, including wireframe data.

DWF Files

Design Web Format (DWF™) is a compressed, secure format used to publish CAD data. DWF files are fast to open and view, and can easily be shared by e-mail with customers, vendors, marketing, and others who do not have Autodesk Inventor installed. Use the DWF Publisher to publish an accurate visual representation of 2D and 3D data in one file. Download and install the free Autodesk Design Review viewer to view a DWF file.

Learning Autodesk Inventor

You can select a learning tool that suits your preferred learning style. You can get help for the current task, follow a workflow in a tutorial or Show Me animation, learn a new skill using a Skill Builder, or click through Help topics. You can gain 3D knowledge as you transition from 2D and watch animations of operations.

Be more productive with Autodesk software. Get trained at an Autodesk® Authorized Training Center (ATC®) with hands-on, instructor-led classes to help you get the most from your Autodesk products. Enhance your productivity with proven training from over 1,400 ATC sites in more than 75 countries. For more information about Autodesk Authorized Training Centers, contact *atc.program@autodesk.com* or visit the online ATC locator at *www.autodesk.com/atc*.

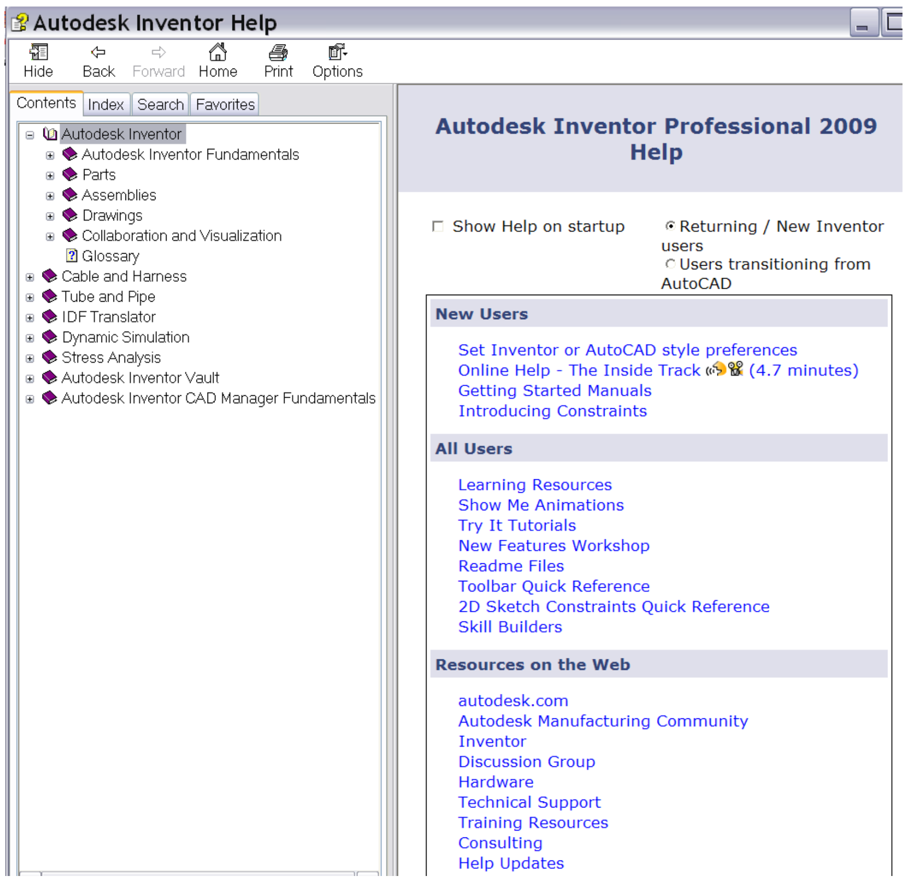
Using Technical Publications

Autodesk Inventor integrates software tools, knowledge, and interactive learning for assistance in specific work tasks and to increase your productivity. The complete set of technical publications includes:

- Printed Getting Started manual
- Help
- Help for the AutoCAD user
- Welcome modules
- Tutorials
- Show Me animations
- New Features Workshop to explore what's new in Autodesk Inventor
- User comments links
- Skill Builders

Help

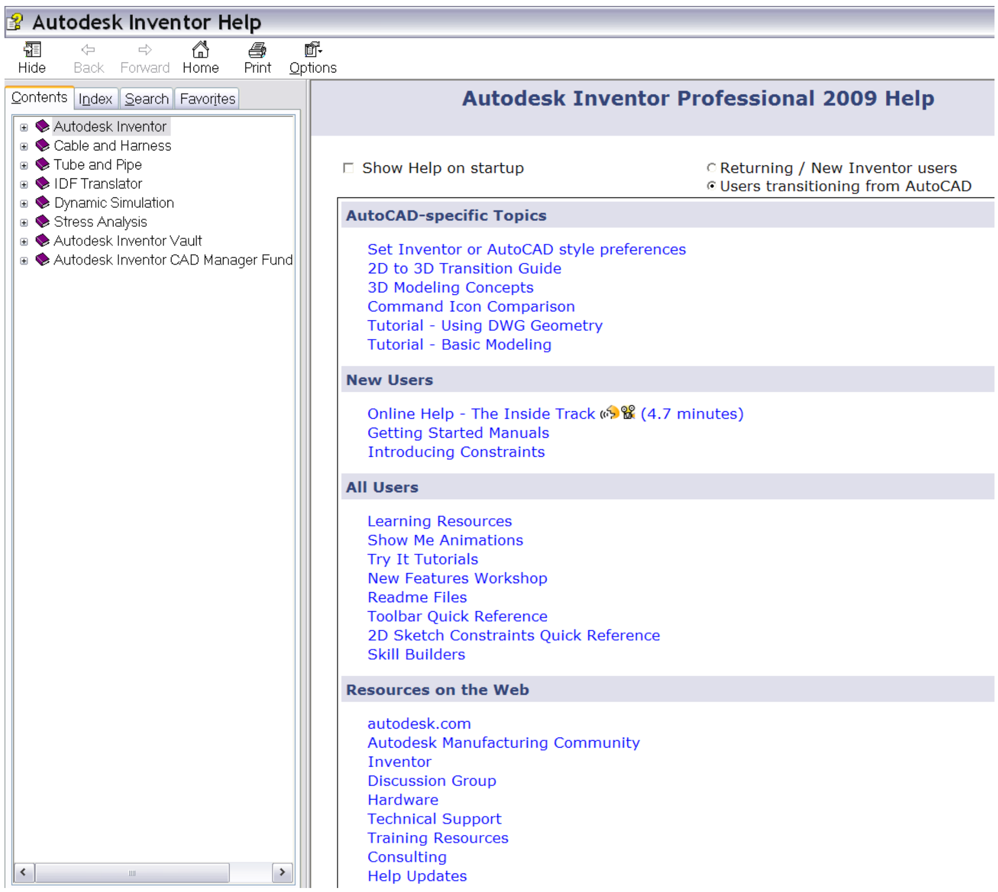
Click Help ➤ Help topics for easy access to the Help topics, Skill Builders, and Tutorials. You can also navigate through the Table of Contents or use the Index and Search functions.



When using Autodesk Inventor, click Help buttons on dialog boxes to retrieve a reference topic automatically that describes options for the dialog box.

Help for AutoCAD Users

In Autodesk Inventor, click Help. If you selected "Users transitioning from AutoCAD" as your preference during the installation, your Help home page opens with topics and tutorials that ease the transition from 2D to 3D. You can also navigate to them through the AutoCAD Topics section in the Table of Contents. There are explanations of the differences between designing in 2D and 3D, equivalents to AutoCAD commands, tutorials, and a workflow to explain everything from sketching to presentations.



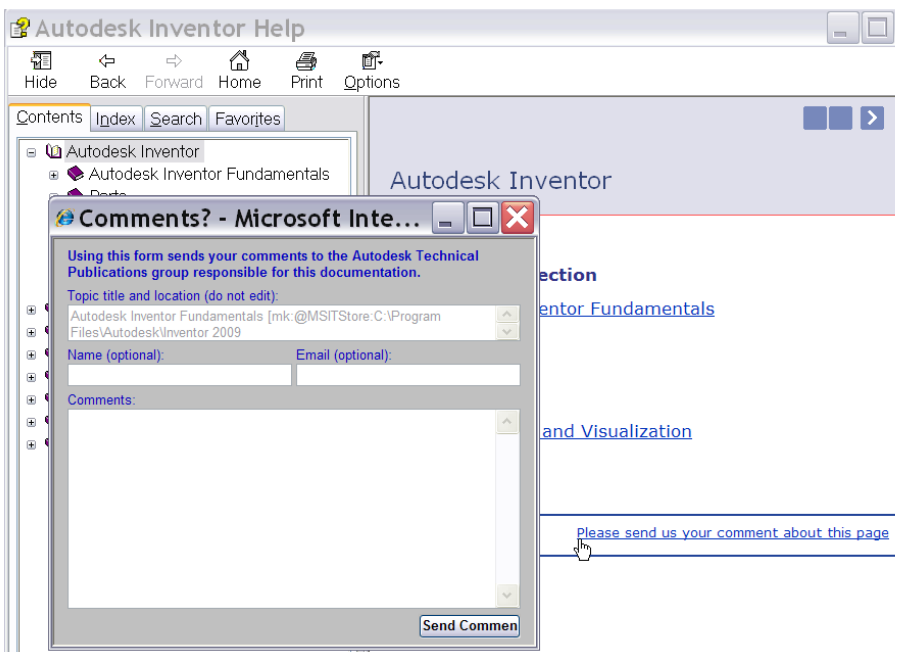
Tutorials and Show Me Animations

Online tutorials are step-by-step illustrated lessons that show you how to create and document your designs. You can access them from the Help home page or click Help ➤ Tutorials.

Show Me animations are videos that show step-by-step instructions how to complete an operation. You can access Show Me animations from the Standard toolbar, the Help home page, and in individual help topics.

Feedback Links

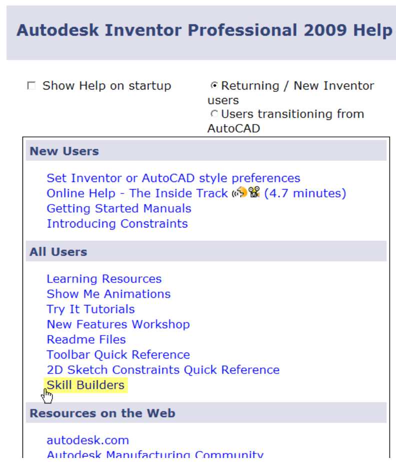
Click the Comments Link on a Help topic page to address specific topics, provide general feedback about the topic, and submit input about what you want and need from the Autodesk Inventor Technical Publications team.



Skill Builders

Autodesk Inventor provides extended learning through its Skill Builders learning modules. Skill Builders are posted throughout a release cycle on the Web to address customer needs and requests.

You can access them (if you have Internet access) by clicking the link on the Help home page.



Creating Sketches

2

In Autodesk® Inventor™, sketching is the first step in creating a part. This chapter gives you an overview of the sketch environment and the workflow for creating sketches.

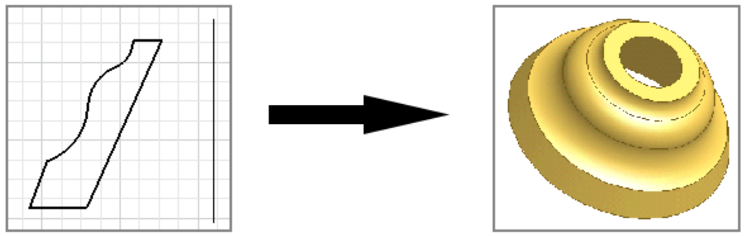
Understanding Sketches

Most parts start with a sketch. A sketch is the profile of a feature and any geometry (such as a sweep path or axis of rotation) required to create the feature.

All sketch geometry is created and edited in the sketch environment, using Sketch tools on the panel bar. You can control the sketch grid, and use sketch tools to draw lines, splines, circles, ellipses, arcs, rectangles, polygons, or points. You can fillet corners, extend or trim curves, and offset and project geometry from other features.

To start a sketch from scratch, open a new part file, select a Sketch tool, and then start sketching in the graphics window. As you sketch, constraints are automatically applied to the various sketch elements as you sketch. For example, if you sketch a line nearly horizontal, a horizontal constraint is implied. You can modify or delete any implied sketch constraint. You can also add constraints manually to any sketch element. To exit a given sketch tool right-click, and then select Done, or press ESC.

You create a 3D model from a sketch by extruding the profile or revolving it around an axis.



From sketch

to 3D model

The model you create in Autodesk Inventor is linked to its underlying sketches and sketch information. If you change a sketch, the model is automatically updated.

Sketch Environment

When you create or edit a sketch, you work in the sketch environment. The sketch environment consists of a sketch and sketch tools to control the sketch grid, and to draw lines, splines, circles, ellipses, arcs, rectangles, polygons, or points.

When you open a new part file, the sketch environment is active. The 2D Sketch button is selected, and the Sketch tools are available, along with a sketch plane on which to sketch. You can control the initial sketch setup by using template files or settings on the Sketch tab of the Application Options dialog box. Click Tools ➤ Application Options ➤ Sketch tab to customize the settings.

When you create a sketch, a sketch icon is displayed in the browser. When you create a feature from a sketch, a feature icon is displayed in the browser with the sketch icon nested under it. When you click a sketch icon in the browser, the sketch is highlighted in the graphics window.

After you create a model from a sketch, re-enter the sketch environment to make changes or start a new sketch for a new feature. In an existing part file, first activate the sketch in the browser. This action activates the tools in the sketch environment. You can create geometry for part features. The changes you make to a sketch are reflected in the model.

Sketch Coordinate System

When you start a new sketch, the sketch coordinate system is displayed as *X* and *Y* axes of the sketch grid. You can turn on the 3D indicator to display it at the sketch origin. (Click Tools ➤ Application Options ➤ Sketch tab. In the Display box, select the Coordinate System Indicator check box.) The default grid lies on the sketch plane.

You can reposition and change orientation of the sketch coordinate system to:

- Change the orientation of dimensions you create.
- Aid in precise input for sketch geometry.

TRY IT: Reposition the sketch origin in the coordinate system

1. Open a part file. In the browser, click the plus sign in front of a feature to expand the display.
2. In the expanded feature display, right-click the sketch, and then click Edit Coordinate System on the menu.

   In the graphics window, the axis icon is displayed for the highlighted sketch.
3. On the axis icon, click the red arrow to realign the *X* axis, the green arrow to realign the *Y* axis, or the blue centroid to relocate the origin.
4. Select one of the following methods to relocate the highlighted axis:
   - A feature vertex to move the coordinate system.
   - A feature edge to rotate the coordinate system.

   To flip the axis, right-click and select Flip axis from the menu.
5. Right-click, and then click Done to activate the new coordinate system.

   The sketch origin in the coordinate system is repositioned.

Using Model Edges as References for Sketches

While you sketch, you can:

- Automatically project edges of the part to the sketch plane as you sketch a curve.
- Create dimensions and constraints to edges of the part that do not lie on the sketch plane.
- Control the automatic projection of part edges to the sketch plane.

Workflow overview: Project part edges to a sketch plane

- Click the Project Geometry tool, and then select any part edge.
- Select an edge of the part while creating a dimension or constraint.

NOTE You can also use model edges as references for continuous loops or points.

Precise Values

In the sketch environment, you can enter relative *X* and *Y* distances from the last point selected. The tools for precise input are located on the Precise Input toolbar. It is only available when a sketch tool that requires placement of a point is activated. For example, you can use precise input to define a line, a sketch point, and a three-point arc, among others.

Enter precise values for geometry as you sketch. The Precise Input toolbar has X and Y fields. You can enter both values to define a point, or enter only X or Y to limit the placement of the point to a vertical or horizontal line.

TRY IT: Input precise values

1. In the sketch environment, click a Sketch tool to make it active.
2. On the standard toolbar, click View ➤ Toolbar ➤ Autodesk Inventor Precise Input.

   The toolbar is displayed in the graphics window.
3. Click a start point, or on the Precise Input dialog box, enter a value in the X field.

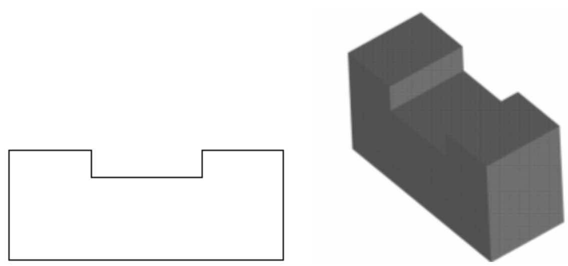
4 Press TAB to activate the Y field, and then enter a value.

5 Press ENTER to accept your input.
The sketch is drawn according to the input.

6 Right-click and select Done to end the sketch tool.

Creating Sketches

In this exercise, you create a part file, and then create sketch geometry using basic sketching techniques. The following illustrates a completed sketch and sketched feature.



Create Sketches

When you open a new part file, the Sketch environment is active.

The current grid setting provides a visual clue to the size of sketches. Use Application Options and Document Settings to define the grid display.

TRY IT: Modify the sketch grid display

1 Click Tools ➤ Application Options.

2 On the Sketch tab, define the grid line display. You can also select the Snap to Grid setting.

TRY IT: Modify the grid spacing

1 Click Tools ➤ Document Settings.

2 Select the Sketch tab and make adjustments, as needed.

TRY IT: Start a sketch

1 On the standard toolbar, click File ➤ New. On the Metric tab, double-click *Standard(mm).ipt*.

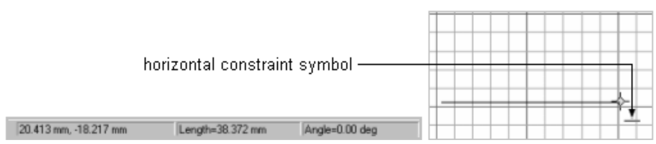
   The new part is listed in the browser, and the sketch environment is active.

2 On the 2D Sketch panel bar, click the Line tool. Click the left side of the graphics window to specify a first point, move the cursor to the right approximately 100 units, and then click to specify a second point.

   As you sketch, the position of the current point, length, and angle of the line are dynamically displayed in the lower right border of the graphics window.

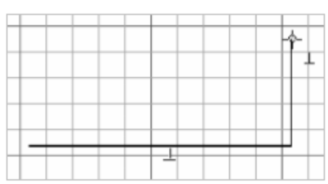
NOTE Use the Zoom tool to zoom out if a line of 100 units does not fit in the graphics window.

The position of the current line point is relative to the sketch 0,0 coordinates. The line angle is relative to the sketch *X* axis. Symbols to indicate implied constraints are displayed next to the current line point as you sketch.

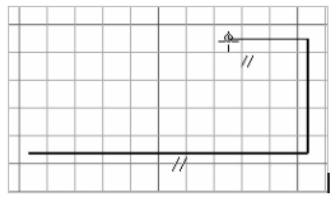


TRY IT: Complete the sketch

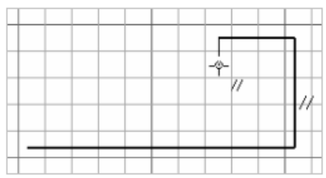
1 Move the cursor up approximately 40 units, and then click to create a perpendicular line.



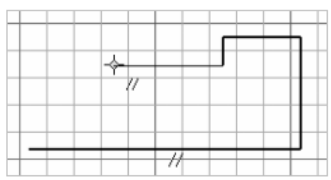
2 Move the cursor to the left and create a horizontal line of approximately 30 units. The parallel constraint symbol is displayed.



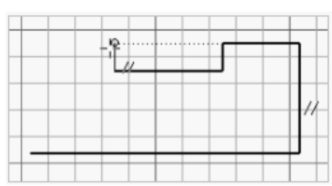
3 Move the cursor down and create a vertical line of approximately 10 units.



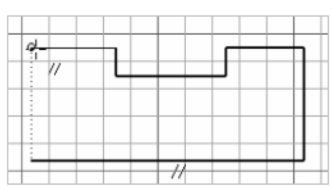
4 Move the cursor to the left to create a horizontal line of approximately 40 units.



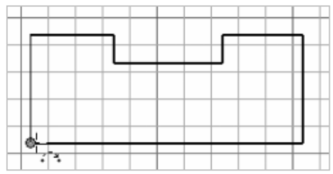
5 Move the cursor up until the parallel constraint symbol is displayed and a dotted line appears. Click to specify a point.



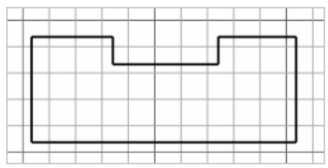
6 Move the cursor left until the parallel constraint symbol is displayed and a dotted line appears, and then click to specify a point.



7 Move the cursor down until it touches the first point you specified at the beginning of the exercise. When the coincident constraint symbol is displayed, click to close the sketch.



8 In the graphics background, right-click and select Done.

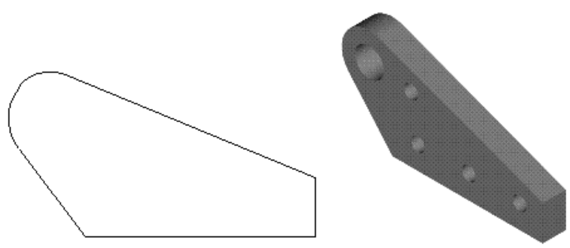


9 Right-click again and select Finish Sketch.

The sketch is completed. Do not save the file.

Create Profiles with Tangencies

In this exercise, you create a part file, and then use basic sketching techniques to create a simple profile. The profile consists of lines and tangential arcs.



TRY IT: Create a sketch

1 Click the New tool on the standard toolbar, select the Metric tab, and then double-click *Standard(mm).ipt*.

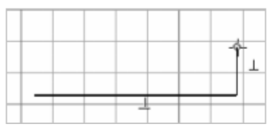
A new part and sketch are listed in the browser. The sketch environment is active.

2 On the standard toolbar, click View ➤ Toolbar ➤ Autodesk Inventor Precise Input to display the Precise Input toolbar.

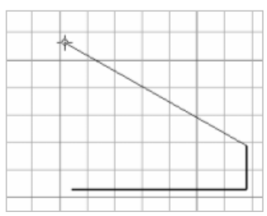
3 Click the Line tool on the panel bar or on the 2D Sketch panel toolbar. Click the center of the graphics window, and then enter *65* in the X field of the Precise Input toolbar. Move the cursor to the right to display the horizontal constraint symbol, and then click to create a 65-mm horizontal line.

4 On the Precise Input dialog box, click the Y field, and then enter *15*. Move the cursor to display the perpendicular constraint symbol, and then click the second point. A perpendicular line of 15 units is sketched.

NOTE Use the Zoom tool to zoom out and view the entire line if it is not visible in the graphics window.

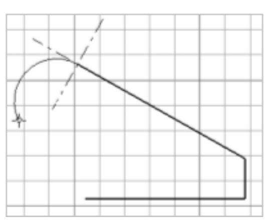


5 Move the cursor up and to the left, and then click to create a sloping line. The exact angle is not important.



TRY IT: Complete the sketch

1 Click the end of the line, hold, and drag the endpoint to create a tangent arc. Release the mouse button to place the endpoint of the arc.



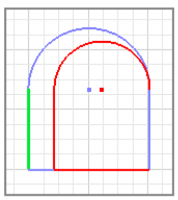
2 Move the cursor to the start point of the profile and click when the coincident constraint symbol is displayed.

3 In the graphics background, right-click, and select Done, and then right-click and select Finish Sketch.

The sketch is completed. Do not save the file.

Drag Sketch Geometry

After you create sketch geometry, and while it is unconstrained or underconstrained, you can drag it to resize it.



Drag to resize geometry

Tips for Sketching

- Start a line by dragging off a circle or an arc.
  Drag radially for a perpendicular line or drag tangentially for a tangent line.
- Start a line by dragging off the interior (not the endpoints) of another line and drag in a perpendicular direction.
  The new line is constrained perpendicular to the existing line.
- Create an arc by dragging off the end of a line.
  Return the pointer to the endpoint of the line to change the direction of an arc.
- Start a spline tangent to a line by dragging off the line.
  Select the endpoint of a line, and then drag it in the direction of tangency to end a spline tangent to a line.
- Create coincident constraints.

When you start a new line, arc, or circle from an existing line, Autodesk Inventor can infer a coincident constraint to the midpoint, endpoint, or interior of the line.

- Use SHIFT to drag.
  All drag features, except for a tangent spline, are also available by pressing and holding SHIFT while moving the cursor.
- Drag multiple lines, curves, or points at the same time.
  Select the geometry, and then drag the last item you selected.
- Switch between the Trim and Extend tools.
  Press SHIFT or select the other tool from the context menu to switch between Trim and Extend.

Constraining Sketches

Constraints limit changes and define the shape of a sketch. For example, if a line is horizontally constrained, dragging an endpoint changes the length of the line or moves it vertically, but does not affect its slope. You can place geometric constraints between two objects in the same sketch, or between a sketch and geometry projected from an existing feature or a different sketch.

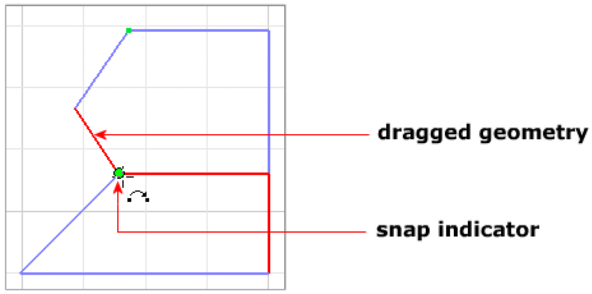
Constraints are automatically applied when you sketch. For example, if the horizontal or vertical symbol is displayed when you create a line, then the associated constraint is applied. Depending on how accurately you sketch, one or more constraints may be required to stabilize the sketch shape or position.

Although you can use unconstrained sketches, fully constrained sketches result in more predictable updates.

NOTE The term *constraints* is often used in Autodesk Inventor to refer to both geometric constraints and dimensions. Remember that dimensions and geometric constraints work together to create a sketch that meets design intent.

Add Constraints

Define your design intent by adding geometric constraints to the sketch. You can use autodimensioning to confirm whether a sketch is fully constrained and apply any needed constraints. You can also create constraints by inference by dragging geometry until the cursor brushes the geometry you want to constrain.



Constraint symbol appears when the dragged geometry touches the endpoints

To view or remove constraints, use the Show Constraints tool on the 2D Sketch panel toolbar. Alternatively, right-click in the graphics window, and then use options on the menu to show or hide all constraints at once. To delete a constraint, select a constraint symbol, right-click, and then select Delete.

Some geometric constraints work only with lines, while others work only with arcs, circles, or radial features.

Open Data Files for Exercises

The location of data files for the exercises in this manual are specified in the project called *tutorial_files*. This project must be activate so you have access to the required files. Once the project is active, you can open the tutorial files.

TRY IT: Activate the project file and open the tutorial file for an exercise

1. Close any open Autodesk Inventor files.
2. On the standard toolbar, click File ➤ Projects.
3. On the Project Editor, top pane, double-click the *tutorial_files* project to make it the active project. Click Done.
4. Click File ➤ Open.
5. On the Open dialog box, click the file *consketch.ipt* see a preview of it, and double-click it to open it.

   The file opens in Autodesk Inventor. You are ready to start the exercise.

Add Constraints to the First Sketch

In this exercise, you practice adding geometric constraints to an existing sketch containing three closed loops. In some cases, you can greatly reduce the number of dimensional constraints required on a sketch.

This exercise contains geometry that does not meet design criteria and requires additional geometric constraints to comply with the design intent.

TRY IT: Add constraints to the first sketch

1. On the standard toolbar, click the Look At tool, and then select any curve. The plan view is displayed.
2. Click the Zoom All tool on the standard toolbar to view the three loops.

3. In the browser, double-click Sketch1 to make it active.
4. On the standard toolbar, click the Zoom Window tool, and then draw a window around the sketch loop on the left.

   The sketch loop is centered on your screen.
5. Right-click the graphics window, and select Show All Constraints. The current sketch constraints are displayed.
6. Move the cursor over the constraint glyphs to highlight the sketch geometry that is constrained.

   In this example, you want the sloping lines in the sketch to be vertical, so you now add a vertical constraint.
7. Right-click the graphics window, and select Hide All Constraints.
8. Click the down arrow beside the Constraint tool in the 2D Sketch panel bar, and then click the Vertical constraint tool.

NOTE The cursor displays the constraint type. In this case, the vertical symbol is displayed.

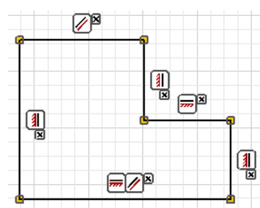
Click the three sloping lines (be sure that you do not select the midpoint of the lines).

Your sketch should look like the one in the following figure.

9 Right-click the graphics window, and select Done.

10 Right-click the graphics window, and select Show All Constraints.

11 All constraints display as shown in the following figure.



12 Right-click the graphics window, and select Hide All Constraints.

13 Click Return on the standard toolbar to exit the sketch.

Add Constraints to Existing Sketches

Constraints can be added to a sketch after it is created. In this procedure, you add constraints to the second sketch.

To redisplay all of the sketches, use the Zoom All tool on the standard toolbar.

TRY IT: Add constraints to a sketch

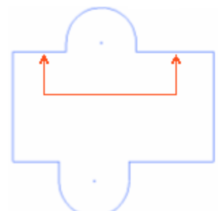
1 Double-click Sketch2 in the browser.

2 On the standard toolbar, click the Zoom Window tool, and then drag a window around the second sketch loop.

The second sketch loop is centered on your screen.

3 Click the arrow beside the Constraint tool on the panel bar or on the 2D Sketch panel toolbar to open the pop-up menu. Click the Collinear constraint tool. Click the horizontal lines at the top of the sketch.

Your sketch should look like the following figure. Note the collinear lines identified by the red arrows.

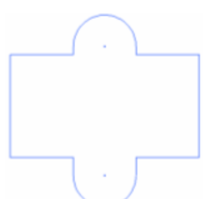


4 Press ESC to cancel the Collinear constraint tool. Drag the top-right horizontal line down and note how the sketch changes. This technique is known as constrained drag.

5 Click the down arrow beside the Constraint tool again, and then click the Equal constraint tool. Click the horizontal line at the lower left of the sketch and then click the horizontal line at the upper left.

Make the two horizontal lines on the right side equal to the line at the lower left.

Your sketch should look like the following figure.



6 Press ESC to cancel the Constraint tool. Drag the right vertical line and note how the sketch changes. With the equal constraint applied, the sketch retains its symmetry as you drag the vertical lines.

7 In the graphics background, right-click and select Done, and then right-click Finish Sketch to exit the sketch.

Delete and Add Constraints

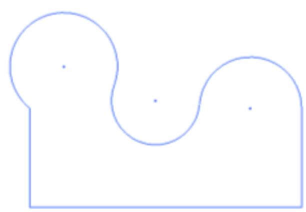
Constraints can be removed from sketches. Show constraints, and then use the Delete option on the context menu.

TRY IT: Delete a constraint and add a constraints

1 Activate Sketch3.

2 On the standard toolbar, click the Zoom Window tool, and then drag a window around the third sketch loop.

The third sketch loop is centered on your screen.



Click the Show Constraints tool on the panel bar or on the 2D Sketch panel toolbar. Pause the cursor over the vertical line at the left of the sketch. The constraints are displayed.

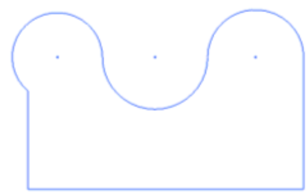
3 Move the cursor over the Equal constraint symbol, and then click to select it. Right-click, and then select Delete to remove the constraint.

4 Click the down arrow beside the Constraint tool on the panel bar or the 2D Sketch panel toolbar to open the pop-up menu. Click the Horizontal constraint tool.

5 Click the center point of the arc at the left of the sketch, and then click the center point of the arc in the center of the sketch.

Repeat this process for the third center point.

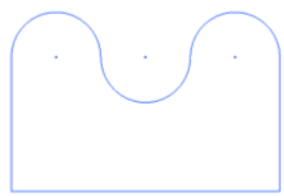
Your sketch should look like the following figure.



6 Apply a tangent constraint to the arc and line at the left side of the sketch.

7 Apply equal constraints to the radii of the three arcs.

Your sketch should look like the following figure.



8 In the graphics background, right-click and click Finish Sketch to exit the sketch.

Do not save the file.

Tips for Constraining Sketches

- Turn off automatic constraints. Press and hold CTRL while sketching.
- Infer a constraint. Move the cursor over other geometry while sketching to infer a constraint.
- Define dimensions with equations. Double-click a dimension to open the Edit Dimension dialog box. Click the reference geometry, and its dimension identifier appears on the dialog box. You can use the dimension identifier in a mathematical expression (for example, D1*2). Dimensions that are based on equations are marked with the *fx*: prefix.
- Override the units on a particular dimension. For example, in a part file set to metric dimensions, you can enter 1 inch on the Edit Dimension dialog box.

Dimensioning Sketches

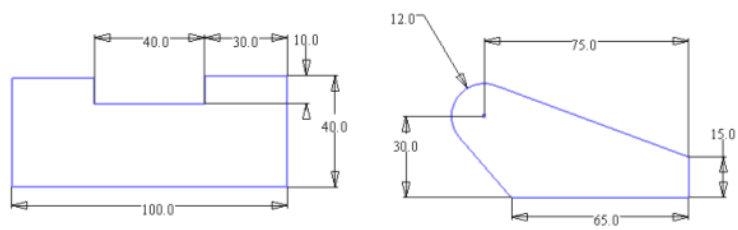
To retain design intent, sketch geometry generally requires dimensions in addition to geometric constraints to maintain size and position.

Geometric constraints, such as horizontal, vertical, or parallel can be applied while you sketch. Dimensions are typically added after your sketch geometry is in place.

In general, all dimensions within Autodesk Inventor are parametric. You can modify the dimension to change the size of the item dimensioned. You can also specify that a dimension be driven. The dimension reflects the size of the item but cannot be used to modify the size of the item.

When you add parametric dimensions to sketch geometry, you are applying constraints that control the size and position of objects in the sketch. The sketch is automatically updated when changes are made to the dimension values.

Examples of dimensioned sketches are shown in the following illustration.



To create dimensions, use the General Dimension tool on the panel bar or on the 2D Sketch panel toolbar. You select the sketch geometry you want to dimension, and then click to place the dimension.

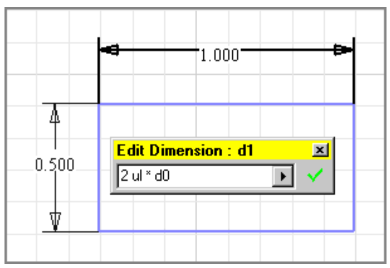
The selection of geometry and the placement of the dimension determine the kind of dimension that is created. For example, if you select the edge of one circle, a radial dimension is created. If you select the edges of two circles, a linear dimension is established between their center points.

Place Dimensions

Parametric dimensions define the size of your sketch. After you add a dimension, you cannot change the size of a line or curve by dragging it. In Autodesk Inventor, you cannot apply double dimensions to a sketch.

TRY IT: Create a parametric dimension

1. Create a sketch, or open an existing sketch.
2. In the Sketch environment, on the panel bar or on the 2D Sketch panel toolbar, click the General Dimension tool.
3. Select the sketch geometry you want to dimension, and then drag to a point to display the dimension.
4. Double-click the dimension to open the Edit Dimension box.
5. Enter a dimension value. You can enter numeric values or the parameter names associated with other dimensions or equations. Dimensions based on equations, as shown in the following image, are preceded by the *fx:* prefix.



Automatic Dimensions

You can also use the Auto Dimension tool on the panel bar or from the 2D Sketch panel toolbar to speed up the dimensioning process. You individually select sketch geometry such as lines, arcs, circles, and vertices and dimensions and constraints are automatically applied. If you do not individually select sketch geometry, all undimensioned sketched objects are automatically dimensioned. The Auto Dimension tool provides a fast and easy way to dimension sketches in a single step.

You can:

- Use Auto Dimension to fully dimension and constrain an entire sketch.
- Identify specific curves or the entire sketch for constraining.
- Create only dimensions, only constraints, or both.

- Use the Dimension tools to provide critical dimensions, and then use Auto Dimension to finish constraining the sketch.
- Use AutoDimension in complicated sketches when you are unsure which dimensions are missing to constrain the sketch fully.
- Remove automatic dimensions and constraints.

NOTE To ensure that your sketch is fully dimensioned, use the Project Geometry tool to project all reference geometry to the sketch before using the Auto Dimension tool.

You can define dimensions with other dimension values. The names of dimensions are parameters. When you edit a dimension, you can enter an equation that uses one or more parameters.

You can display sketch dimensions in one of three forms:

- Calculated value
- Parameter name
- Parameter name and calculated value

You can modify dimensions using the Edit Dimension box. To display the Edit Dimension box, click the dimension when it is placed, or double-click the dimension when the General Dimension tool is not active.

There are two ways to display the Edit Dimension box upon placement of a dimension:

- Click Tools ➤ Application Options ➤ Sketch tab, and turn on Edit Dimension when Created
- With General Dimension active, right-click in the graphics window and select Edit Dimension.

Dimension Types

In some cases, the dimension preview does not meet the design intent. You can change the dimension type by repositioning the dimension, or you can right-click, and then select the type from the menu. You can also control which type of linear dimension is applied by selecting and edge or a vertex.

For example, when you dimension an edge to a vertex, the dimension automatically aligns itself with the edge.

Diametric Dimensions

In the design process of creating a revolved part, you can add a centerline as the axis of rotation. If this centerline is used in a sketch dimension, it is placed as a diametric dimension by default.

Driven Dimensions

You can place driven dimensions with Autodesk Inventor, and you can change the dimension type of an existing dimension to driven. A driven dimension reflects the size of the geometry, but you cannot edit the dimension value. Use driven dimensions to display dimension values for reference purposes only.

Workflow overview: Apply a driven dimension

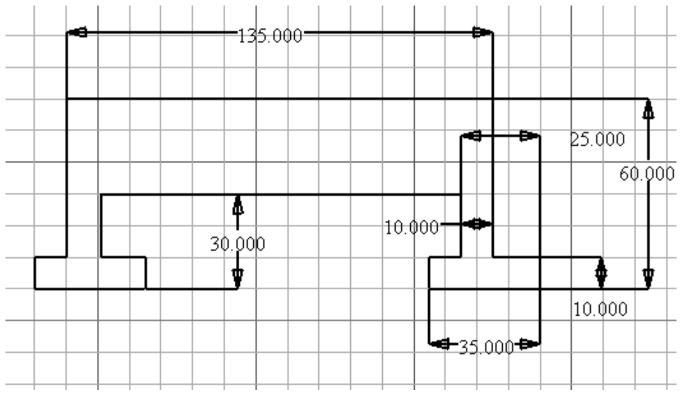
- For an existing dimension, select the dimension, and then select the Driven dimension button from the standard toolbar.
- To create driven dimensions on the fly, while the General Dimension tool is active, select the Driven dimension button from the standard toolbar.

Driven dimensions are displayed in parentheses.

You can also create driven dimensions automatically on constrained sketch objects. When you try to place a dimension on a constrained sketch object, a dialog box is displayed where you can choose to accept a driven dimension or cancel the placement.

Dimension Profiles

In this exercise, you add dimensional constraints to a sketch. The completed exercise is shown in the following figure.



TRY IT: Apply dimensions to linear objects

1 With the project tutorial_files active, open the file *dimsketch.ipt*.

The sketch geometry requires dimensional constraints to maintain its overall size. Geometric constraints were already applied to maintain the shape of the sketch.

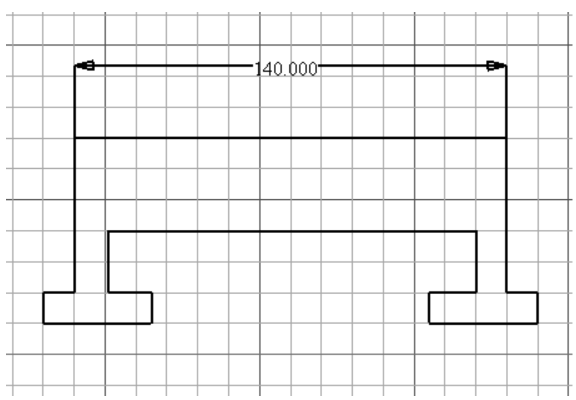
2 In the browser, double-click Sketch1 to make the sketch active.

3 Click the Look At tool on the standard toolbar, and then select any line to obtain a plan view of the sketch.

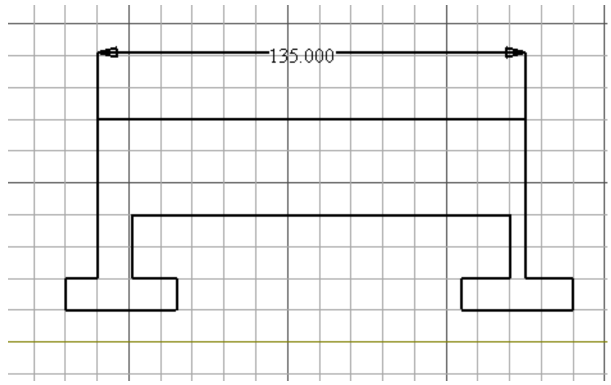
Click the Zoom All tool to view the entire sketch.

4 Click the General Dimension tool on the panel bar or on the 2D Sketch panel toolbar.

5 Click the top horizontal line of the sketch, and then place the dimension.



6 Click the dimension to display the Edit Dimension box. Enter *135* and press ENTER.

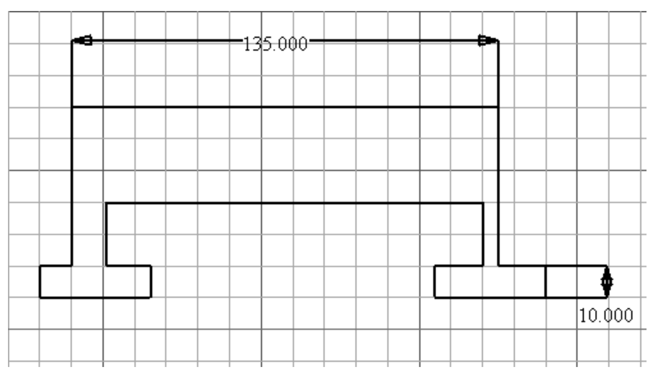


In this example, you clicked the dimension to display the dialog box. If you are placing many dimensions, you can display the Edit Dimension box automatically.

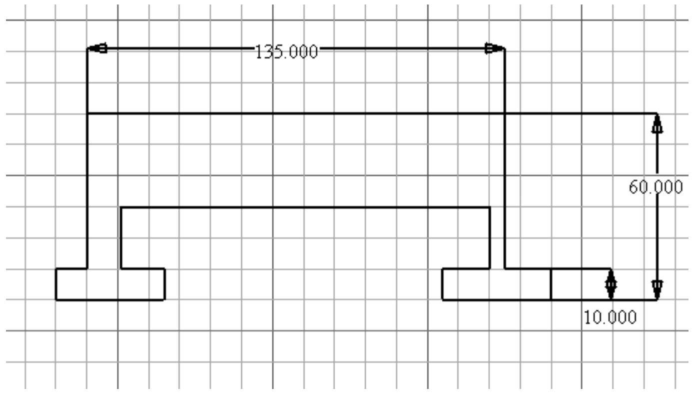
7 With the General Dimension tool active, right-click the graphics window background, and select Edit Dimension from the context menu.

8 Complete the dimensional constraints as follows:

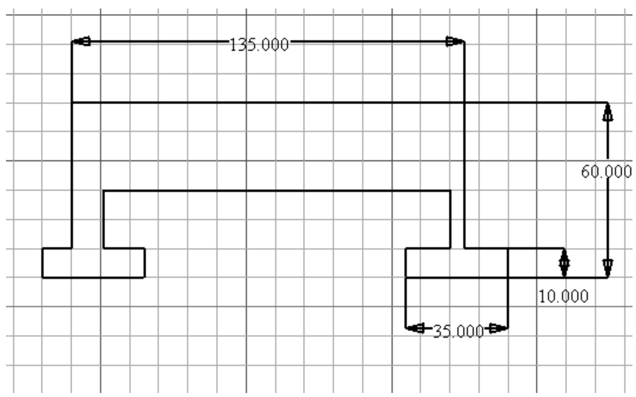
Add a dimension of *10*.



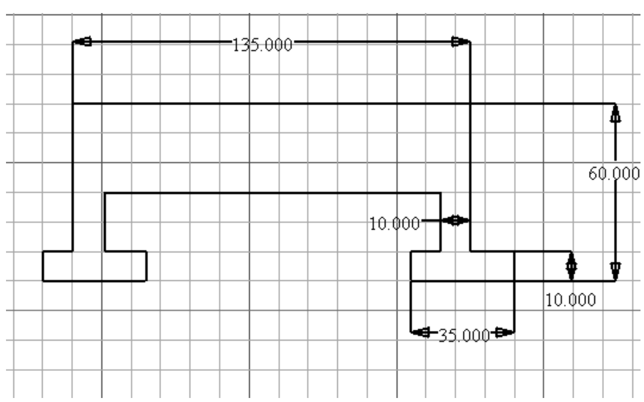
Add a dimension of *60*.



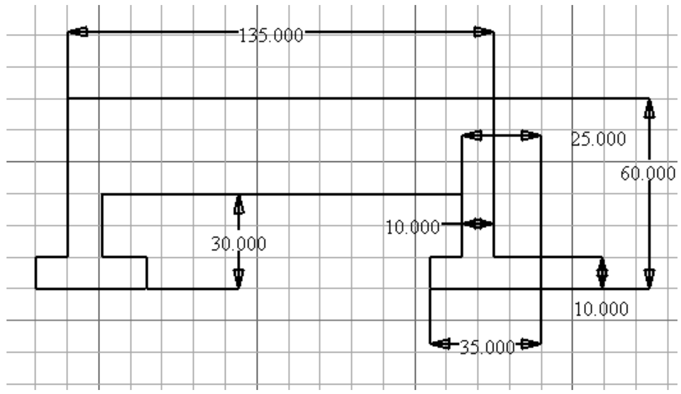
Add a dimension of *35*.



Add a dimension of *10*.



Add dimensions of *25* and *30*.



9 Right-click the graphics window and select Done from the context menu to exit the General Dimension tool.

Delete and Add Dimensions

Next, you remove the existing dimensions and use the Auto Dimension tool to quickly dimension the sketch.

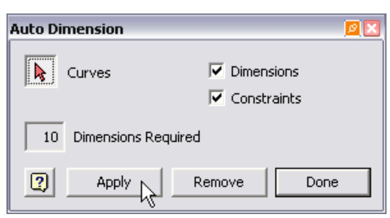
TRY IT: Remove dimensions and add dimensions to the sketch

1 Hold down the SHIFT key while you select each of the dimensions on your sketch.

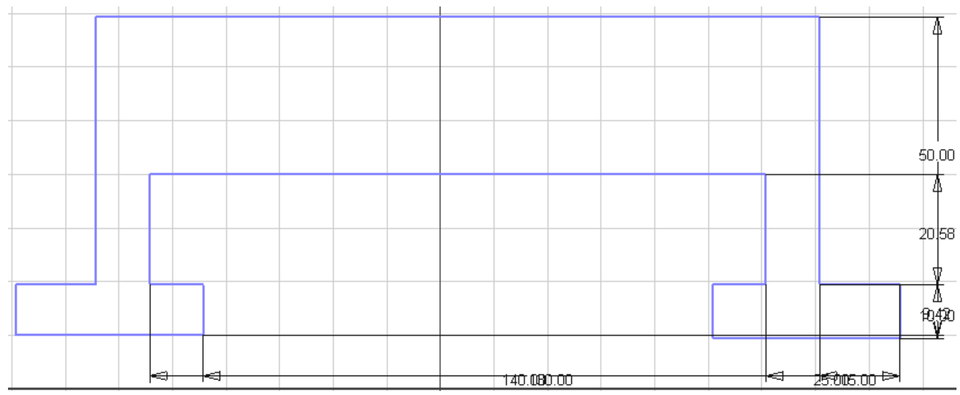
2 When all the dimensions are selected, press DELETE to remove them.

3 Click the Auto Dimension tool on the 2D Sketch panel bar.

4 When the Auto Dimension dialog box is displayed, click Apply to accept the default settings and begin to dimension the sketch.



The dimensions are applied to the sketch.

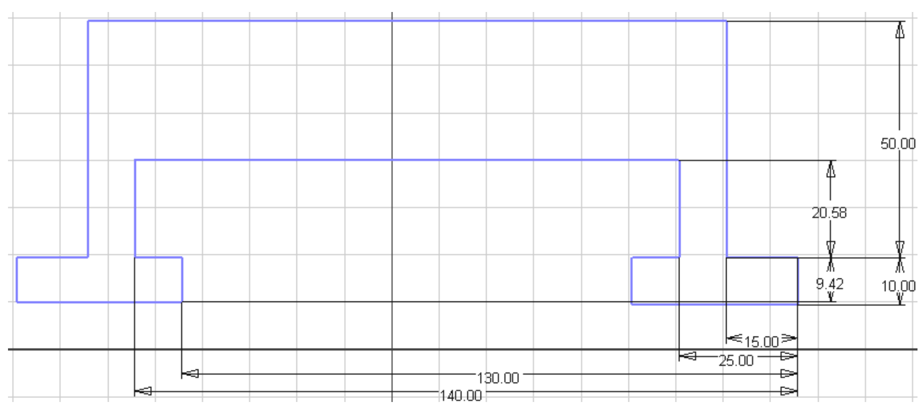


Notice that the Auto Dimension dialog box now indicates that two dimensions are required. This is due to two missing Fix constraints.

5 Click Done on the Auto Dimension dialog box.

6 In the sketch, select and reposition dimensions so they are easier to read.

Your dimensions should look like the following figure.



Close the file without saving changes.

Tips for Creating Dimensions

- Place critical dimensions using the General Dimension tool, and then use Auto Dimension to speed up the dimensioning process. For remaining objects to dimension, you may find it faster to automatically dimension all sketch geometry. You can then delete unwanted dimensions instead of selecting sketch geometry individually for automatic dimensioning.

- If Auto Dimension does not dimension your sketch appropriately, you can experiment with selecting some of the sketch geometry to control how automatic dimensions are applied.
- If you use automatic dimensions, you may find it easier to accept sketch default dimension values, and then edit them with correct values in an order (large to small) so you can control sketch behavior.
- Use geometric constraints when possible. For example, place a perpendicular constraint instead of using a dimension value of 90 degrees.
- Place large dimensions before small ones.
- Incorporate relationships between dimensions.
- Consider both dimensional and geometric constraints to meet the overall design intent.

Modifying Sketches

After you create sketch geometry, you can refine and adjust the proportions of the sketch by applying dimensions or geometric constraints. You can also drag any unconstrained or underconstrained geometry.

Patterning Sketches

You can use the Circular and Rectangular pattern tools on the Sketch toolbar to create patterns of your original sketch. The pattern geometry is fully constrained. These constraints are maintained as a group, If you remove the pattern constraints, all constraints to the pattern geometry are deleted.

Workflow overview: Create a circular sketch pattern

1. Use sketch tools to create the geometry to include in the pattern.
2. Click the Circular Pattern tool on the Sketch toolbar, and then select the sketch geometry to pattern.
3. On the Circular Pattern dialog box, click Axis, and then select the point, vertex, or work axis to use as the pattern axis.

4 In the Count box, specify the number of elements in the pattern.
5 In the Angle box, specify the angle to use for the circular pattern.
6 Optionally, click More, and then choose one or more options:
   - Click Suppress to select individual pattern elements to remove from the pattern. The geometry is suppressed.
   - Click Associative to specify that the pattern updates when changes are made to the part.
   - Click Fitted to specify that pattern elements are equally fitted within the specified angle. If not selected, the pattern spacing measures the angle between elements instead of the overall angle for the pattern.
7 Click OK to create the pattern.

Workflow overview: Create a rectangular sketch pattern

1 Use sketch tools to create the geometry to include in the pattern.
2 Click the Rectangular Pattern tool on the Sketch toolbar, and then select the sketch geometry to pattern.
3 Click the Direction 1 button, and then select geometry to define the first direction for the pattern.
4 In the Spacing box, specify the spacing between the elements.
5 Click the Direction 2 button, select geometry to define the second direction for the pattern, and then specify the Count and Spacing.
6 Optionally, click More, and then choose one or more options:
   - Click Suppress to select individual pattern elements to remove from the pattern. The geometry is suppressed.
   - Click Associative to specify that the pattern updates when changes are made to the part.
   - Click Fitted to specify that pattern elements are equally fitted within the specified angle. If not selected, the pattern spacing measures the angle between elements instead of the overall angle for the pattern.
7 Click OK to create the pattern.

Tips for editing sketch patterns

- You can modify the spacing between pattern elements, change the pattern count and direction, change the pattern calculation method, and suppress geometry in the sketch pattern. Right-click the sketch in the browser and select Edit Sketch. Then right-click a pattern member in the graphics window, and select Edit Pattern. On the pattern dialog box, revise values as needed.
- You can edit pattern dimensions. In the sketch, double-click the dimension to change, enter a new value on the Edit Dimension box, and then click the check mark. You can enter dimensions as equations, parameter names, or specific values.
- You can remove the associative relationship among pattern elements but the geometry becomes individual curves and pattern editing options are no longer available. Right-click a pattern member and then, on the pattern dialog box, click More. Clear the Associative check box and then click OK.
- Suppress one or more pattern elements to remove them from the pattern. Right-click the sketch in the browser, and select Edit Sketch. Right-click the pattern geometry to suppress and select Suppress Elements. Suppressed pattern elements are not included in profiles and do not appear in drawing sketches.
- Pattern elements, including dimensions and sketch geometry used to define axis and directions, can be deleted but not while the pattern is associative. If you want to retain the relationships among the pattern elements, but need to remove one or more instances, consider using Suppress to remove them.

Delete Sketches

If a sketch was used in a feature, you cannot delete the sketch. You can edit the feature sketch and delete sketch geometry, but the feature might not update properly. You may need to edit the sketch or feature to recover the feature.

Workflow overview: Delete a sketch

1. In the browser, select the sketch to delete.
2. Press DELETE or right-click and select Delete.

NOTE To delete individual sketch curves, edit the sketch, select the curve, and then press Delete.

You can remove dimensional constraints from a sketch, and allow the sketch to resize as needed. Parts with adaptive features resize in when they are constrained to fixed geometry.

Workflow overview: Delete sketch dimensions

1 Right-click the sketch in the browser and choose Edit Sketch.

2 Click the Select tool.

3 Right-click the dimension in the graphics window and select Delete.

NOTE If the sketch is part of a feature, click Update after you delete dimensions.

You can delete a sketch pattern or suppress selected elements of a sketch pattern.

Workflow overview: Delete sketch patterns

1 In the browser, select the sketch, right-click and then select Edit Sketch.

2 Select the sketch pattern geometry to delete.

3 Right-click and select an option. The selection options are determined by selected geometry:

- Non pattern geometry: Click Delete to remove the selected geometry.
- Pattern geometry: Select Delete Pattern to remove the entire pattern or Suppress Elements to suppress selected pattern geometry.
- Non pattern and pattern geometry: Select Delete to remove non pattern geometry, and select Delete Pattern to remove the entire pattern, or select Suppress Elements to suppress selected pattern geometry.

Learning about 3D Sketches

You are now familiar with sketching in 2D because it is much like sketching on paper. You can also sketch in 3D, connecting points on X, Y, and Z planes to create a three-dimensional shape. A 3D sketch provides a way to create a path for 3D sweep features, such as those used in wiring, cabling, and tubing.

When working in a 3D sketch, points can lie on any plane. Like 2D sketches, you can constrain sketch geometry to control its shape, add dimensions, and precisely position points relative to the last placed point.

One way to learn about sketching in 3D is to create a box.

TRY IT: Create a box and sketch 3D lines on X, Y, and Z planes

1. On the 2D Sketch panel bar, click the rectangle tool and create a rectangle and then enter *E* on your keyboard to use the shortcut to start the Extrude command.
2. On the Extrude dialog box, enter any distance and click OK.
3. Right-click in the graphics window and choose Isometric View. On the standard toolbar, click Display and choose Wireframe Display.

   Now you have a 3D “space” in which to visualize the sketch.
4. On the standard toolbar, click the arrow beside the Sketch button and select 3D Sketch. In the browser, a 3D sketch icon is added and the 3D Sketch panel bar is activated.
5. On the 3D Sketch panel bar, click the Line tool. Notice that the 3D triad displays with arrows to indicate the X, Y, and Z axes. The Autodesk Inventor Precise Input toolbar may display, but you can sketch without entering coordinates.

   Click anywhere in space to start the line. On the 3D triad, click a plane or arrow to change the sketch plane and then click to place another sketch point. As you change sketch planes, notice the sketch grid for that plane becomes active.

   Bends might be automatically added as you sketch the lines. This setting is controlled on the Sketch tab of the Application Options dialog box.
6. Continue to place points as needed, changing sketch planes as you go to make sure that you are sketching in all dimensions. When finished, right-click and select Done.
7. On the standard toolbar, click the Rotate tool and rotate the box in all directions.

   You can see that the 3D line has points on X, Y, and Z planes.

Now that you can see the line in the context of the box, start a new file and experiment with creating a 3D sketch without the box. Because the file opens with a 2D sketch active, click Return on the standard toolbar to close the sketch, and then click Sketch ➤ 3D Sketch.

Sketch some 3D lines and then use some of the other tools on the 3D Sketch panel bar:

- Use the General Dimension tool to dimension the lines.
- Use constraint tools to constrain 3D lines to other lines or points.
- Optionally, change the setting on the Sketch tab of the Application Options dialog box to add or remove automatic bends in 3D lines.

Working with Sketched Features

3

In this chapter, you learn about parametric part modeling and the process for creating sketched features on parts.

Parametric Part Modeling

A part model is a collection of features. Parametric modeling gives you the flexibility to adjust the parameters that control the size and shape of a model, and automatically see the effect of your modifications.

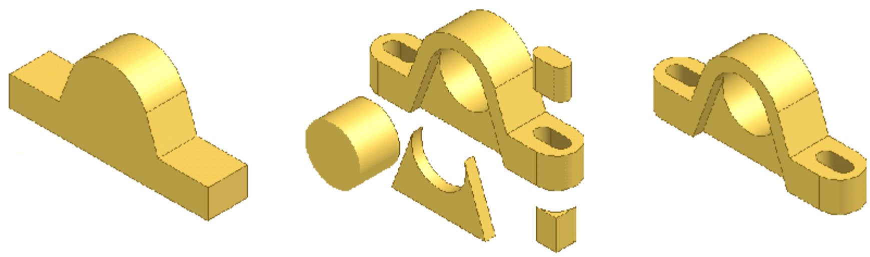
To create a 3D part model in Autodesk® Inventor™, you extrude sketch geometry, sweep or project sketch geometry along a path, or revolve sketch geometry around an axis. These models are often called solids because they enclose volume, unlike wireframe models which only define edges. The solid models in Autodesk Inventor are constructed of features.

You can also create surfaces with many of these operations. Use surfaces to define shapes or aspects of the part body. For example, a curved surface can be used as a termination plane for cuts in a housing.

You can edit the characteristics of a feature by returning to its underlying sketch or changing the values used in feature creation. For example, you can change the length of an extruded feature by entering a new value for the extent of the extrusion. You can also use equations to derive one dimension from another.

Using Autodesk Inventor, you can create several kinds of features, including sketched, placed, work, pattern, and library. Some features require that you create sketches or paths, while others do not. Some represent visible geometry, and some, such as work features, help you precisely position geometry on a part. You can edit a feature at any time.

Parent/child relationships exist between features, which means that one feature controls another. There can be multiple levels of parent/child relationships. A child feature is created after the parent feature, and cannot exist without a parent feature. For example, you can create a boss on a casting, and it may or may not have a hole drilled in it, depending on the application. The boss (the parent) can exist without the hole (the child), but the hole cannot exist without the boss.



base part   child features removed   finished part

Part Modeling Environment

The part modeling environment is active any time you create or edit a part. In the part modeling environment, create or modify features, define work features, create patterns, and combine features to create parts. Use the browser to edit sketches or features, show or hide features, create design notes, make features adaptive, and access properties.

Your first sketch for a part can be a simple shape. You can edit features after you add them, so you can develop your design quickly. Throughout the design process, add geometric and dimensional detail and constraints to improve your models. Evaluate design alternatives by changing relationships and constraints, or adding and deleting features.

The browser displays the part icon, with its features nested below. Surface features and work features are nested, or consumed by default. To edit a feature, right-click it in the browser or the graphics window. From the context menu, select Edit Feature to revise the feature creation parameters or Edit Sketch to revise the underlying sketch. To control nesting, or consumption of surface and work features for all features, set the option using the Tools ➤ Applications Options, Part tab. To override consumption on a per-feature basis, right click the feature in the browser, and then select Consume Inputs.

Workflows

Before you begin, analyze the part to determine which features to create, and the most efficient order in which to create them.

Answer these questions before you start to model your design:

- Are you creating a stand-alone part, a component in an assembly, or the first of a family of parts?
  Determine whether to create the part in a part file or within an assembly file, and whether you create constraints using fixed values or equations.
- Which view of the part best describes its basic shape?
  The most prominent feature in that view is usually the best feature to begin modeling. The first feature in your part is called the base feature.
- Which features require the use of work planes and work points to position the model geometry?
- What are the most important features of your part?
  Create these features early in the modeling process so that the dimensions of other features can be based on their dimension values.
- Which features of your part can be added with sketched features, and which features can be added with placed features?
- Based on these feature decisions, which features should be created first?

Base Features

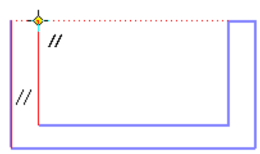
The first feature you create in a part is the base feature. The base feature is most commonly based on a sketch profile, and represents the most basic shape in the part. The base feature may also be an imported base solid (*.sat* or *.step* file format). You can also create a work feature as the base feature.

You create additional features to complete your part. Since these features are dependent on the base feature, good planning can dramatically reduce the time required to create a part. After you plan your strategy, decide how to create the base feature.

Workflow overview: Create a parametric solid model and associated drawings

1 Create a part in a part file (*.ipt*) or assembly (*.iam)* file. If you are working on a small assembly or it is early in the design process, consider creating your part in a part file.

2 Use tools on the Sketch toolbar or panel bar to sketch the basic shape of the base feature.



Geometric constraints define the shape of objects in your sketch.

3 Analyze your sketch geometry and, if necessary, choose the appropriate geometric constraint from the panel bar or from the Sketch toolbar.

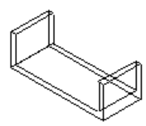
You can add or delete constraints later to modify the shape of the sketch.

Dimensions define the size of the objects in your sketch.

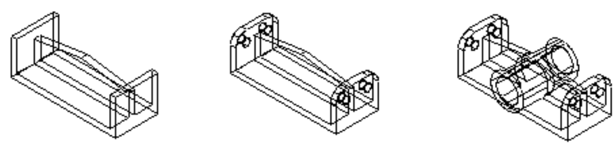
4 Click the General Dimension tool in the panel bar or from the Sketch toolbar and apply dimensions.

You can change the lengths of lines and the radii of arcs within the sketch at a later time.

5 Extrude, revolve, sweep, loft, or coil the parametric sketch to create the first, or base feature of the part.

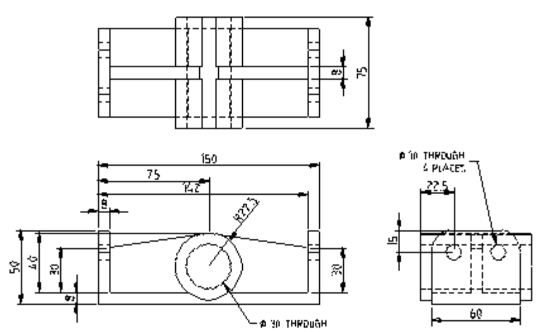


6 Repeat the process to create additional features, selecting join, cut, or intersect to complete the part.



7 Document the part in an Autodesk Inventor drawing file to create the desired annotated 2D drawing views.

Any time during the part modeling process, you can create a drawing file (*.idw*) and begin making a fabrication drawing of your part. Changes you make to your part are automatically reflected in drawing views of the part.



Adding Sketched Features

Sketched part features depend on sketch geometry. The first feature of a part, the base feature, is typically a sketched feature.

You can select a face on an existing part, and sketch on it. The sketch is displayed with the Cartesian grid defined. If you want to construct a feature on a curved surface, or at an angle to a surface, first construct a work plane.

Each of the following operations creates a solid extrusion from a sketch profile. Extrude, Revolve, Sweep, and Loft can also create surface extrusions:

Extrude	Projects a sketch profile along a straight path. Use to create surfaces as well as solids.
Revolve	Projects a sketch profile around an axis.
Sweep	Projects a sketch profile along a sketched path.
Loft	Constructs a feature with two or more sketch profiles sketched on multiple part faces or work planes. The model transitions from one shape to the next, and can follow a curved path.
Coil	Projects a sketch profile along a helical path.
Rib	Creates a rib or web extrusion from a 2D sketch.

The same procedure for creating a sketched base feature is used to create additional sketched features.

Extrude Features

Use the Extrude tool to create a feature by adding depth to an open or closed profile or a region.

- In the Assembly environment, the Extrude tool is available on the Assembly Panel bar when you are creating an assembly feature.
- In the Weldment environment, the Extrude tool is available on the Weldment Panel bar when you are creating a preparation or machining feature.
- In the Part environment, the Extrude tool is available on the Part Features panel bar when you are creating an extrusion for a single part.

Workflow overview: Create a parametric solid model and associated drawings

1 Start with a sketch, or select a profile or region that represents the cross section of the extruded feature you want to create. Open profiles cannot be used when creating extrusions as assembly features.

2.  Click the Extrude tool to display the Extrude dialog box.

   If there is only one profile in the sketch, it is automatically selected.

   If there are multiple profiles, on the Shape tab, click Profile, and then select the profile to extrude. Use Select Other to cycle through selectable geometry, and then click to select.

3. In Output, click either the Solid or Surface button.

   For base features, only Surface is available for open profiles. For assembly extrusions, only Solid is available.

4. Click the Join, Cut, or Intersect button.

   For assembly extrusions, only the Cut operation is available.

5. In Extents, on the drop-down menu, select the method to terminate the extrusion. Some methods are not available for base features.

   `Distance:` Enter the distance of the extrusion.

   `To Next:` Click the direction of the extrusion.

   NOTE To Next is not available for assembly extrusions.

   `To:` Click a sketch point, work point, model vertex, work plane, or the End termination plane.

   `From-To:` Click the Start and End termination planes.

   By default, the extrusion terminates on the maximum-distance plane

   `To and From-To extents:` Click Minimum Solution to terminate on the nearest-distance plan.

   `All:` Click the direction of the extrusion or to extrude equally in both directions.

   NOTE If termination options are ambiguous, such as on a cylinder or irregular surface, click the More tab, and then use Flip to specify direction

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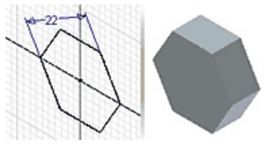
6. On the More tab, enter a Taper angle, if necessary.

   In the graphics window, an arrow shows the taper direction.

   Click OK.

   The sketch is extruded.

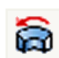


Close the file without saving.

Revolve Features

Use the Revolve tool on the Part Feature panel bar to create a feature by rotating one or more sketched profiles around an axis. The axis and the profile must be coplanar. If this is the first feature, it is the base feature.

Workflow overview: Create a revolved feature

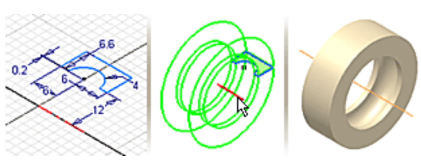
1. To begin, sketch a profile that represents the cross section of the revolved feature you want to create. Except for surfaces, profiles must be closed loops.
2.  Click the Revolve tool to display the Revolve dialog box.

   If there is only one profile in the sketch, it is automatically selected.

   If there are multiple profiles, on the Shape tab click Profile, and then select the profile to extrude.

   Use only unconsumed closed sketches in the active sketch plane.
3. Click Axis, and then select an axis from the active sketch plane.
4. Click Join, Cut, Intersect, or Surface. Surface outputs, along with cut and intersect operations, are not allowed as base features.
5. In Extents, select Angle or Full.
6. Click a direction button to revolve the feature in either direction or equally in both directions.

   Results are previewed on the model.

Sweep Features

Use the Sweep tool on the Part Features panel bar to create a feature by moving or sweeping one or more sketched profiles along a selected path. The path may be an open or closed loop, but must pierce the profile plane. Except for surfaces, profiles must be closed loops.

There are three ways to create sweeps. You can create a sweep surface by:

- Sweeping a profile along a path.

- Sweeping a profile along a path and guide rail. The guide rail controls scale and twist of the swept profile.

- Sweeping a profile along a path and guide rail. The guide rail controls scale and twist of the swept profile.

Workflow overview: Create a sweep feature along a path

1. To start, sketch a profile and then sketch a path on an intersecting plane.
2.  Click the Sweep tool.

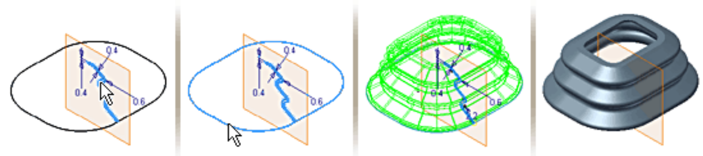
   If there is only one profile in the sketch, it is automatically highlighted.

   If there are multiple profiles, click Profile, and then select the profile to sweep.
3. Click Path, and select a 3d sketch or planar path sketch
4. From the Type list, select Path.
5. Click the orientation for the path.
   - Path holds the sweep profile constant to the path.
   - Parallel holds the sweep profile parallel to the original profile.
6. Enter a Taper angle, if desired.

   In the graphics window, a symbol shows taper direction.
7. Click Join, Cut, Intersect with another feature. Some methods are not available for base features.
8. Click Solid or Surface.

9 Click OK.

The sweep feature is created.



Loft Features

Use the Loft tool on the Part Features panel bar to blend or transition between the shapes of two or more profiles (called sections) on work planes or part faces. You can create a simple loft, a loft with rails (path), or a centerline loft. You can also select a point for one or both end sections of an open loft.

To use an existing face as the beginning or end of a loft, create a sketch on the face so the edges of the face are selectable for the loft. If using the loop of a planar or non planar face, select it directly without creating a sketch on the face.

Workflow overview: Create a loft or loft with rails feature

1 Sketch profiles on separate planes to represent cross sections of the loft feature.

2  Click the Loft tool to display the Loft dialog box.

3 On the Curves tab, in Output, click solid or surface.

4 Click in Sections and then click the profiles to loft in the sequence you want the shapes to blend. If you select multiple profiles on any plane, they must intersect.

NOTE If there is more than one loop in a sketch, first select the sketch, and then select the curve or loop.

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5 Optionally, click the Rails button to designate guide paths, and then click to add 2D or 3D curves for shape control. Sections (profiles) must intersect rails (guide paths). This option is not available when rail curves are specified.

6 If appropriate, click the Closed Loop check box to join the beginning and ending profiles of the loft.

7 If appropriate, click the Merge Tangent Faces check box so that an edge is not created between tangent faces.

8 In Operation, click Join, Cut, or Intersect.

9 On the Conditions tab, the start and end profiles are listed. Click each, and specify a boundary condition:

Free Condition	Apply no boundary conditions. It is the default.
Tangent Condition	If you selected a loop or the profile is in a separate sketch on the boundary of a face. Creates a loft tangent to the adjacent faces.
Direction Condition	Specify an angle measured relative to the section or rail plane. Set the angle and weight of the condition.
Smooth (G2) Condition	Creates a loft curvature continuous to the adjacent faces.

10 On the Transition tab, Automatic Mapping is selected by default. If desired, clear the check box to modify automatically created point sets or add or delete points.

- Click the point set row to modify, add, or delete.
- A default calculated map point is created for each profile sketch. Click position to specify a unitless value. Zero represents one end of the line; one represents the other end. Decimal values represent positions between ends.

11 Click OK to create the loft.

Coil Features

Use the Coil tool on the Part Features panel bar to create a helix-based feature. Use this feature to create coil springs and threads. If the coil is the first feature created, it is the base feature.

Workflow overview: Create a coil spring

1 To begin, sketch a profile that represents the cross section of the coil feature, and then use the Line tool or the Work Axis tool to create an axis of revolution for the coil.

2  Click the Coil tool. The Coil dialog box is displayed.

If there is only one profile in the sketch, it is automatically highlighted.

3 If there are multiple profiles, click Profile, and then select the profile.

4 Click Axis.

It can be at any orientation but cannot intersect the profile.

5 On the Coil Size tab, click the down arrow on the Type box, and then select one of the following types:

`Pitch and Revolution`

`Revolution and Height`

`Pitch and Height`

`Spiral`

Enter the Pitch, Height, Revolution, or Taper as appropriate. Taper is not available for a Spiral.

6 On the Coil Ends tab, choose one of the following methods to define the start and end of the coil, for example, to stand upright on a flat surface:

Flat	Create a transition in the pitch of the coil. Enter a Transition Angle and then a Flat Angle (up to 360 degrees).
Natural	End the coil without transition.

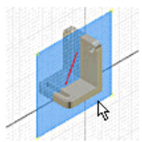
Rib and Web Features

Use the Rib tool to create ribs (thin-walled closed support shapes) and webs (thin-walled open support shapes).

Use the Zoom and Rotate tools to position the part so the face where the rib is located is visible.

Workflow overview: Set the sketch plane and create profile geometry for a rib

1. Create a work plane to use as the sketch plane.
2. On the Standard toolbar, click the 2D Sketch tool, and then click the work plane or a planar face to set the sketch plane.
3. Use the Look At tool to reorient the sketch.
4. Use tools on the 2D Sketch panel to create an open profile to represent the rib shape.



Workflow overview: Create a rib

1.  Click the Rib tool on the Part Features panel bar, and then click the profile, if it is not already selected.
2. Click the Direction button to set the direction of the rib.

   Pause the cursor over the open profile to see direction arrows that indicate if the rib extends parallel or perpendicular to your sketch geometry.
3. The Extend Profile check box is displayed if the ends of the profile do not intersect the part.

   The ends of the profile automatically extend. If you prefer, clear the check box to create a rib or web the exact length of your profile.
4. In the Thickness box, enter the rib thickness.

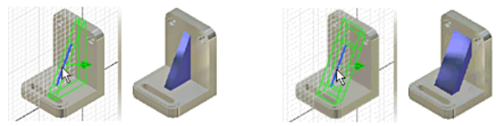
   Click a Flip button to specify the direction of the rib thickness.
5. Click one of the following buttons to set the depth of the rib:

    `To Next` Terminate the rib on the next face.

    `Finite` Enter a value in the box to set a depth.

6 Optionally, in the Taper box, enter a taper or draft value. The direction must be perpendicular to the sketch plane to apply a taper value.

7 Click OK to create the rib.



rib web

NOTE To create a rib or web network, sketch multiple intersecting or non-intersecting profiles on the sketch plane, and then follow the previous steps.

Modifying Features

There are several methods available to modify an existing feature. In the browser, right-click a feature, and then use one of three options on the menu:

Show Dimensions Displays the sketch dimensions so you can edit them.

- Change the dimensions of a feature sketch.
- Change, add, or delete constraints

Edit Sketch Activates the sketch so it is available for edit.

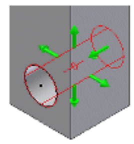
- Select a different profile for the feature

After you modify a part sketch, exit the sketch and the part is updated automatically.

Edit Feature Opens the dialog box for that feature.

- Choose a different method to terminate the feature
- Choose whether the feature joins, cuts, or intersects another feature.

3D Grips Uses grip handles to drag a feature or face, or snap to other geometry to resize a feature. Arrows indicate the drag direction. The feature preview shows the expected results before you commit to the change.



Creating and Editing Placed Features

4

In this chapter, you learn about placing and editing part features. Exercises step you through creation of holes, fillets, chamfers, threads, shells, circular and rectangular patterns, mirror features and analyzing faces.

Adding Placed Features

Placed features are common engineering features that do not require a sketch when you create them with Autodesk® Inventor™. When you create these features, you usually provide only the location and a few dimensions. The standard placed features are shell, fillet, chamfer, face draft, hole, and thread.

Here are some of the tools for placed features, located on the Part Features panel bar:

Fillet	Places a fillet or round on selected edges.
Chamfer	Breaks sharp edges. Removes material from an outside edge and can add material to an inside edge.
Hole	Places a specified hole in a part, optionally with thread.
Thread	Creates regular and tapered external and internal threads on cylindrical or conical faces.
Shell	Produces a hollow part with a wall thickness you define.
Rectangular Pattern	Creates a rectangular pattern of features.
Circular Pattern	Creates a circular pattern of features.
Mirror Feature	Creates a mirror image across a plane.

Dialog boxes define values for placed features, such as the Hole dialog box in the following illustration.

Hole Features

With Autodesk Inventor, you can create different types of holes:

- Drill
- Counterbore
- Countersink
- Spotface

You can specify hole depth using one of three termination options: Distance, Through All, and To.

Use the Drill Point option to set flat or angle drill points.

Holes also can be classified as simple hole, clearance hold, tapped hole, or taper tapped hole. However, you cannot create a taper tapped hole of counterbore type.

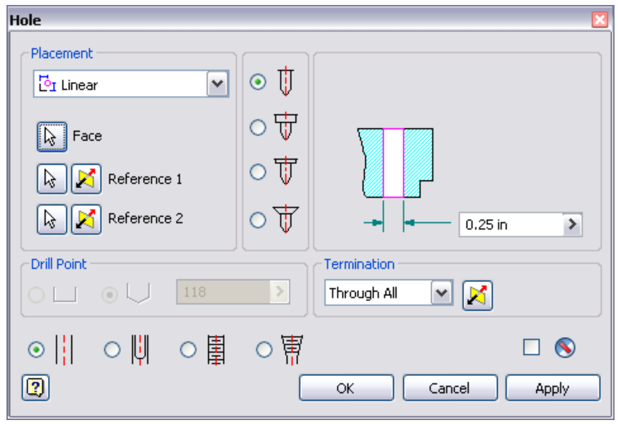
When you create a tapped hole or taper tapped hole, the tap data is stored with the hole, and the threads are displayed when any isometric view is active.

TRY IT: Create a hole feature on a part

1. With the project *tutorial_files* active, open the file *Upper_Plate.ipt*.
2. On the Part Features panel bar, click the Hole tool.
3. On the Holes dialog box, Placement, select Linear from the drop-down list.
4. Click the Face button, and then in the graphics window, click the face where you want to place the hole
5. Click an edge of the face to specify Reference 1, and then click another edge of the face to specify Reference 2.

   The reference dimensions from each edge are displayed. You can double-click each dimension and enter changes to define placement of the hole.
6. Select the first hole type, Drilled, and then enter a diameter of *.25* in.

7 In Termination, select Through All.



8 Click OK.

The hole you defined is placed on the face.

Close the file without saving or save the file under a different name to preserve the original data file.

You can specify hole depth using one of three termination options: Distance, Through All, and To.

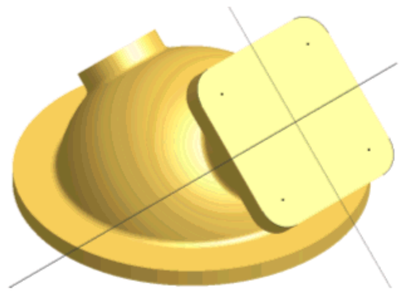
TRY IT: Place a hole feature using arc centers

1 With the project *tutorial_files* active, open file *hole.ipt*.

The part looks like the following figure.

2 Click the Sketch tool on the Standard toolbar, and then click the rectangular face.

The edges of the face and arc centers are projected onto the new sketch, allowing you to position the hole features.



3 In the graphics background, right-click, and then click Finish Sketch to close the sketch.

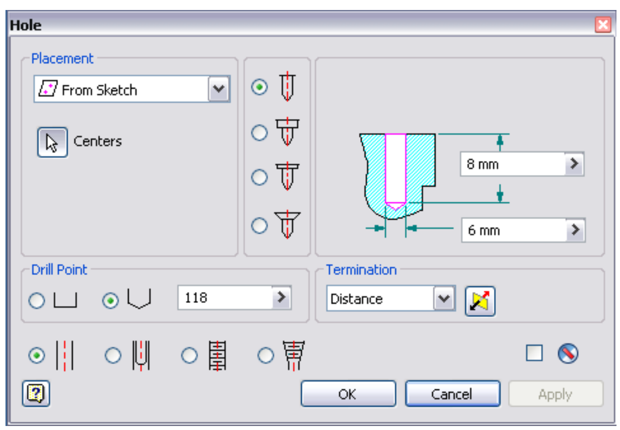
4 Click the Hole tool in the Part Features panel bar to display the Holes dialog box.

In the preview window, edit the value of the hole diameter to read *6 mm.*

5 Click the four arc centers.

6 In Termination, select Distance.

7 In the preview window, edit the value of the hole diameter to read *6 mm.*



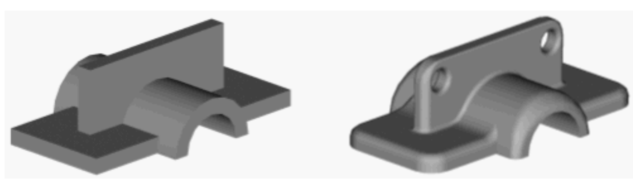
8 Click OK.

The hole feature is created, and is added to the browser. Notice that one feature defines all four holes.

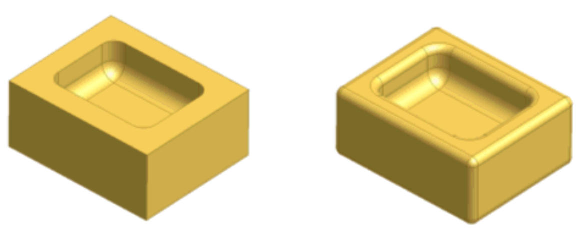
Close the file without saving or save the file with a new name to preserve the original data file.

Fillet Features

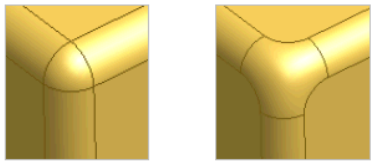
Fillet features add fillets or rounds to one or more part edges, between two faces or face sets, or between three adjacent faces or face sets. Fillets add material to interior edges to create a smooth transition from one face to another. Rounds remove material from exterior edges. You can create constant-radius and variable-radius edge fillets and edge fillets of different sizes in a single operation. For edge fillets, tangent (G1) or smooth (G2) continuity can be applied to adjacent faces. You specify values in a dialog box and select the edges to create the fillet.



For edge fillets you can use the All Fillets and All Rounds selection modes to apply fillets to multiple edges as shown in the following figure.



The corner style can be set to either rolling ball or blend.

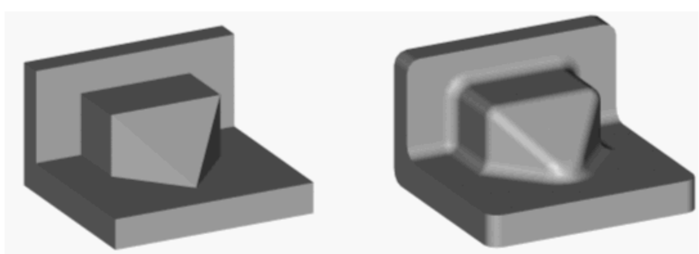


When you create variable radius edge fillets and rounds, you choose between a smooth blend from one radius to another and a straight blend between radii. The method you choose depends on your part design and the way adjacent part features blend into the edge.

You can also specify points between the start and endpoints of a selected edge, and then define their relative distances from the start point and their radii. This technique provides flexibility when creating variable radius edge fillets and rounds.

This illustration shows smooth and straight transitions on variable-radius rounds.

You can model special fillet applications where more than three edges converge. Choose a different radius for each converging edge, if needed.



To find the radius of an existing fillet, right-click the feature in the browser, and then choose Show Dimensions. The fillet radius is displayed on your part.

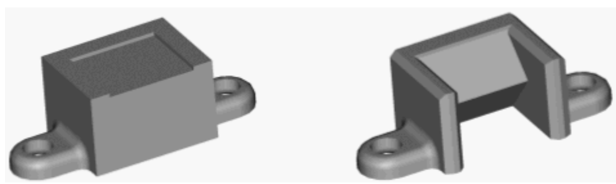
For information on face fillets and full round fillets, refer to "fillet features" in the Help index.

Chamfer Features

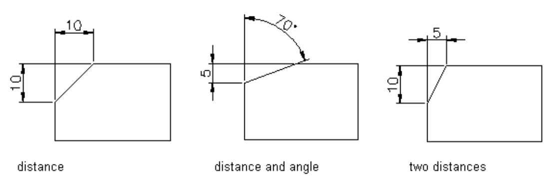
Chamfers are like fillets, except that the edge is beveled rather than rounded. When you create a chamfer on an interior edge, material is added to your model. When you create a chamfer on an exterior edge, material is cut away from your model.

When you create a chamfer, you can specify one of three operations:

- Distance
- Distance and Angle
- Two Distances



A distance chamfer creates a face at an equal distance along the two faces that meet at the selected edge. A Distance and Angle chamfer is established at a distance from the edge and at an angle from a selected face. A two distances chamfer creates a face at different offset distances from the edge.



Chamfers and Fillets

In this exercise, you add chamfer and edge fillet features to complete the shaft socket bracket model.

The completed model is shown in the following figure.

TRY IT: Add a chamfer

1. With the project *tutorial_files* active, open the file *chamfillet.ipt*.

   The file contains a model of a shaft socket bracket.

2. Click Chamfer from the Part Features panel bar.
3. On the Chamfer dialog box, click the Edges button, and then select the four vertical edges of the base.

NOTE You may need to rotate the model to select the appropriate edges. Press F6 to return to the default isometric view.

---

4. In Distance, enter *10 mm*, and then click OK.

   The chamfers are added to the model and the browser

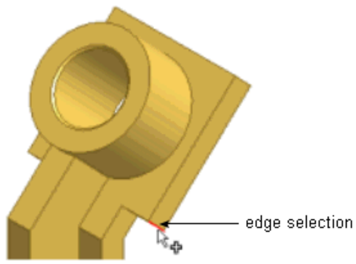
Next, you add equal distance chamfers to top-hole edges.

5 Click Chamfer, and then select the top edge of each of the three holes in the part.

6 On the Chamfer dialog box, change the distance to *1 mm*, and then click OK.

Next, you add different distance chamfers to complete the basic shape of the socket support.

7 Click Chamfer, and then click the Two Distances button. Select the edge shown in the following figure.



8 Enter the following values:

`Distance 1:` 14 mm

`Distance 2:` 18 mm

Click the Direction button to see how the preview changes when the distances are switched.

9 Click the Direction button again to return to the original settings, and then click OK to create the chamfer feature.

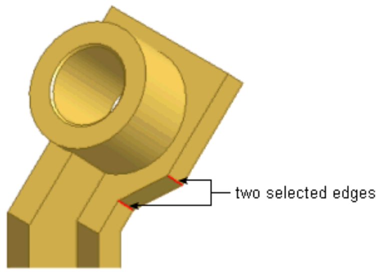
10 Repeat this process to add the same size chamfer to the other side of the part.

Your part should now look like the following figure.

Next, you add fillets to complete the final shape of the part.

TRY IT: Add fillets to a part

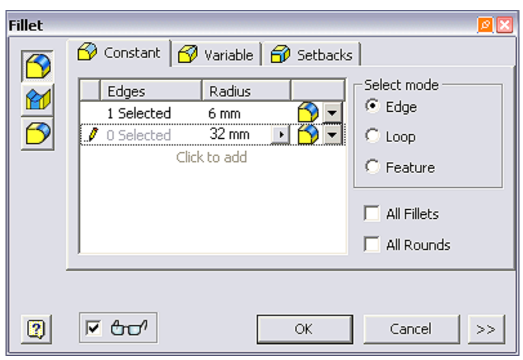
1. Click Fillet from the Part Features panel bar and ensure that the Edge Fillet button is selected. Select the two edges shown in the following figure.

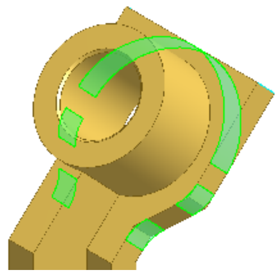


2. Rotate the part, and then select the same two edges on the other side. In the Fillet dialog box, on the Constant tab, change the radius to *16 mm*.

3. Under the edges and radius text, click the line that reads Click to Add. For the next set of edges, select the two vertical edges at the corners at the top of the part.

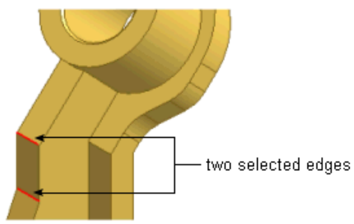
4. Change the radius for the fillet to *32 mm*. When your dialog box and preview look like the following figures, click OK.





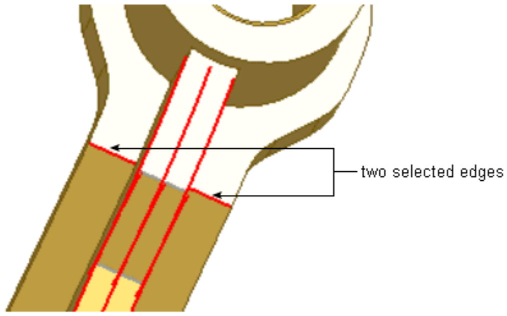
The fillet feature is added to the part and to the browser.

5 Click Fillet, and then select the two horizontal edges on the front of the rib, as shown in the following figure.



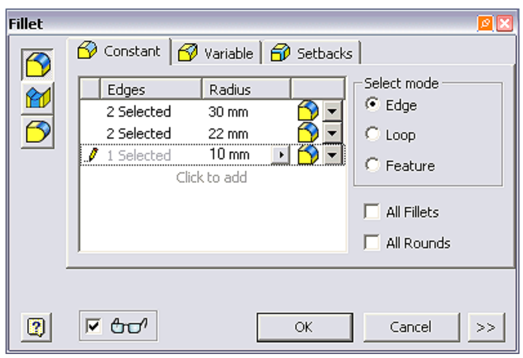
6 On the Fillet dialog box, enter *30 mm* for the radius.

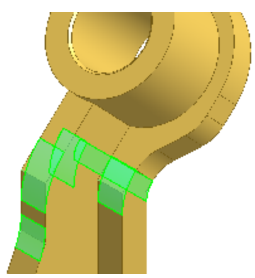
7 To add another set of edges, click the Click to Add text, and then select the two horizontal edges shown in the following figure.



8 On the Fillet dialog box, change the radius for the second selection set to *22 mm*. Click the Click to Add text to create a third set of edges.

9 Rotate the model and select the horizontal edge on the back face directly opposite the second selection set. Enter *10 mm* for the radius. When your dialog box and preview look like the following figures, click OK.

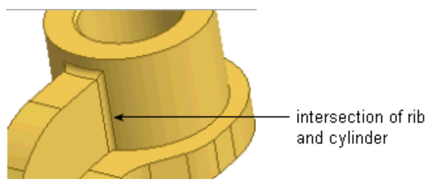




Fillet preview

Fillet feature added to part

10 Click Fillet, and then select the three edges where the rib meets the cylinder at the top of the part. Change the radius to *2 mm*, and then click OK.

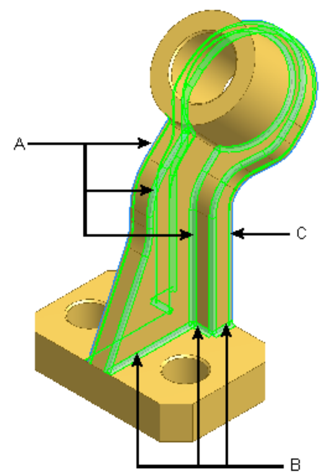


11 Click Fillet. Select the two front edges of the rib, and then select the back edge of the rib (A). These edges are added to the selection set.

12 Select the three edges on each side where the base meets the other features (B).

13 On the Fillet dialog box, select the Loop option in the Select Mode section. Select anywhere on the back edge of the part above the base (C). Notice how the Loop option automatically selected additional edges.

14 Verify that the fillet radius is set to *2 mm*. When your preview looks like the following figure, click OK. The fillet fails, and an error box is displayed.



15 In the Error box, click Edit.

16 On the Fillet dialog box, select the Edge selection mode. Press SHIFT while you select the six edges where the base meets the other features of the part. When these edges are removed from the selection set, click OK.

17 Add a *2 mm* fillet to the edges where the base meets the other features of the part. Notice how the fillets from Fillet 4 connect all the edges so only one selection point is required on each side.

The completed part looks like the following figure.

Close the file without saving or save the file with a new name to preserve the original data file.

Tips for Working with Fillets

- To edit a fillet, right-click the fillet name in the browser and select Edit Feature.
- To edit only the dimensional value of a fillet, double-click the fillet name in the browser. In the Edit Dimension box, change the value of the fillet.
- Alternately, you can change the select priority to Feature Priority, and then double-click a fillet on the part to display the dimensions.
- After editing a fillet, click the Update tool to update the part.

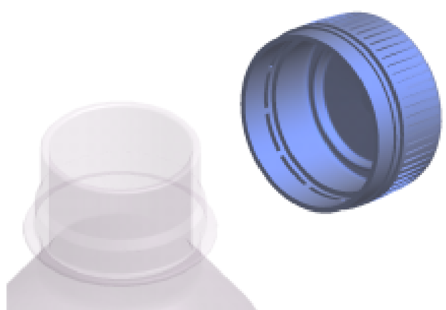
Thread Features

In this exercise, use the Thread tool to create a thread feature on mating faces of a plastic bottle and cap.

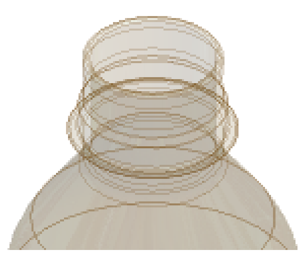
TRY IT: Add threads

1 With the project *tutorial_files* active, open the file *threads.iam*. The file contains a model of a plastic bottle and cap.

2 Use Zoom Window to zoom in on the bottle top and cap.



3 Use Zoom Window to zoom in on the bottle top and cap.

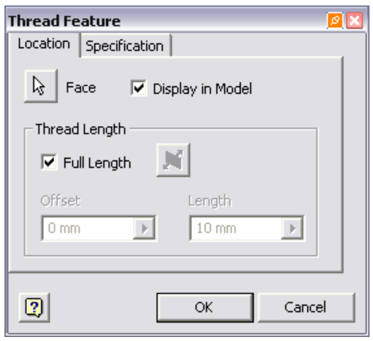


4 In the graphics window or browser, select the cap, and then right-click and clear the check mark on Visibility in the context menu.

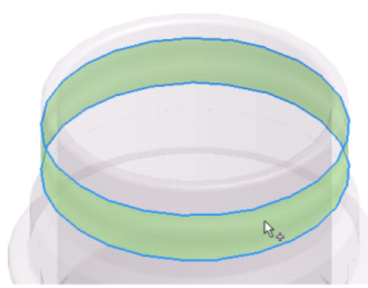
5 In the graphics window or browser, double-click the bottle to activate editing mode.

6 Click the Thread tool on the Part Features panel bar.

7 On the Location tab, enter settings to match the following figure.

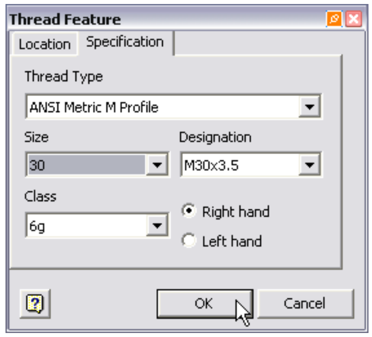


8 Select the split surface as shown in the following figure.



Notice how the thread is previewed on the model.

9 Select the Specification tab, adjust settings as necessary to match the following figure, and then click OK.



The Thread feature is created, as shown in the following figure, and is added to the browser.

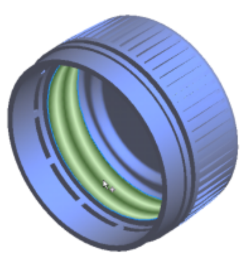
NOTE You can temporarily change the part color to see the threads more easily. On the Standard toolbar, click the arrow on the Styles box and choose a different color.

---

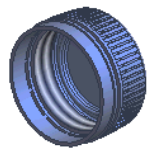
10 Click the Return button to exit edit mode for the bottle, and then turn off visibility for the bottle.

11 In the browser, double-click cap:1 to activate editing mode.

12 Repeat steps 5 through 8 and select the inside surface of the cap as shown in the following figure.



The thread is completed, as shown in the following figure.



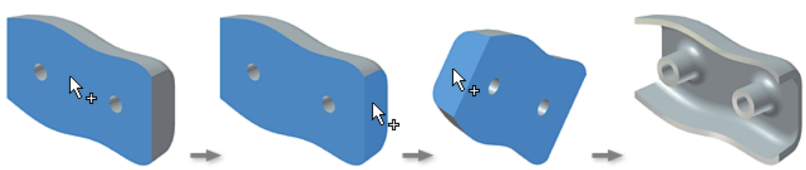
13 Double-click the assembly in the browser, turn on visibility of the bottle, and then restore the Isometric view.

Your completed model should look like the following figure.

Close the file without saving or save the file with a new name to preserve the original data file.

Shell Features

The Shell tool creates a hollow cavity in a part with walls of a specific thickness. It removes material from a part by offsetting existing faces to create new ones on the inside, outside, or both sides of the part. Use the shell feature to create multisided parts like casings or enclosures. A part can have multiple shell features.



When you start the shelling process, you specify which part faces to remove or offset, and you can specify a unique wall thickness for each face on a part.

Use the Shell tool on the Part Features panel bar to remove material from a part interior, creating a cavity with walls of a specified thickness. By default, Autodesk Inventor provides a precise shell feature. If a precise solution does not exist and approximation is enabled, an approximation is attempted.

Start with a single feature, a part, or a part in an assembly.

Workflow overview: Create a shell feature

1. For this exercise, create a simple block or cube.
2.  After you extrude the sketch profile, click the Shell tool.
3. In the graphics window, select the face or faces to remove.
4. On the Shell dialog box, click one of the three direction buttons, Inside, Outside, or Both, to specify the direction of the shell from the surface of the selected face.
5. Enter a value for the face thickness.
6. Click OK.

This time, create a shell feature with varying shell thicknesses.

Workflow overview: Create a shell feature with varying thicknesses

1. Select the shell feature in the browser, and then press the Delete key.
2. Click the Shell tool, and then select the faces to remove.
3. On the Shell dialog box, click a direction button (Inside, Outside, or Both) to specify the direction of the shell from the surface of the selected face.
4. Enter a value for the face thickness.
5. Click the More button in the Shell dialog box.
6. Select Click to Add, and then select a face and enter a specific shell thickness for it.

   In Unique Face Thickness, enter a value that is different from the value for the main shell thickness.
7. Click OK to create the shell.

   Close the file without saving or save the file with a new name to preserve the original data file.

Creating Pattern Features

Many designs call for the repetitive use of one or more features on a single part. Single features or groups of features can be duplicated and arranged in patterns. A pattern feature is a rectangular, circular, or mirrored duplication of features or groups of features. Individual occurrences in a pattern can be suppressed, as necessary. An example of a pattern feature is a rectangular pattern of identical holes cut from a calculator case.

NOTE Features that you can pattern include part features, surface features, and assembly features.

The pattern tools require reference geometry to define the pattern. You can create patterns using the Rectangular Pattern, Circular Pattern, and Mirror Feature tools. You can set the number of occurrences in the pattern, the angular spacing between occurrences, and the direction of the repetition.

Pattern creation methods include:

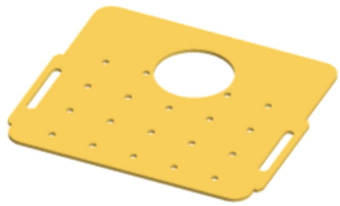
Identical	All occurrences use an identical termination.
Adjust to Model	The termination of each occurrence is calculated individually.
Optimized	Creates a copy and reproduces faces instead of features. Optimizes patterns for faster calculation.

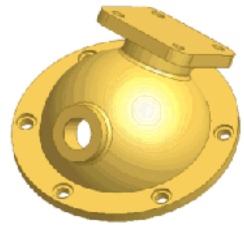
You can suppress components in a component pattern without removing them from the assembly. It makes it easy to replace parts and to create unique members in assemblies.

Rectangular Patterns

Features can be duplicated and arrayed in a rectangular or circular pattern. In the first part of this exercise, you create a single hole and then use it to add a rectangular pattern of holes to a plastic cover plate. You also complete an exercise that uses a circular pattern.

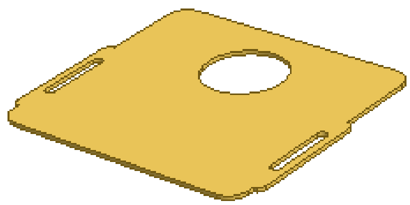
The following is an illustration of the completed exercises.





TRY IT: Create a hole feature

1 With the project *tutorial_files* active, open the file *recpattern.ipt*.



2 On the Part Features panel bar, click the Hole tool.

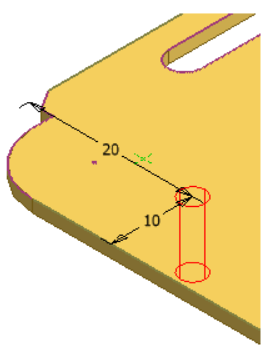
3 On the Hole dialog box, in the Placement box, select Linear. Click the Face button, and then select the top face of the part.

4 In the Hole dialog box, click the Reference1 button.

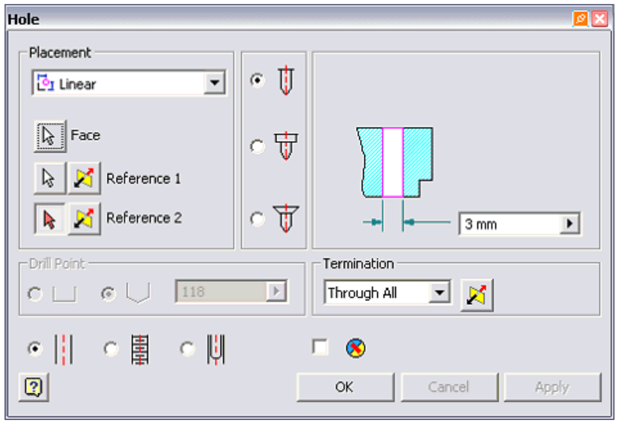
5 In the graphics window, click the leftmost edge of the part for Reference 1, and then the bottom edge for Reference 2.

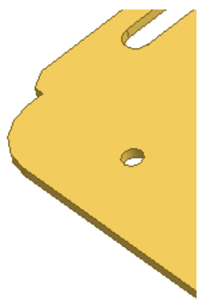
Dimensions from the part edges to the hole center are displayed.

6 Edit the dimensions to *20 mm* from the leftmost edge, and *10 mm* above the bottom edge, as shown in the following illustration.



7 On the Holes dialog box, Termination, select Through All, and verify that the hole diameter is *3 mm*.





8 Click OK to create the hole in the part according to the specifications you entered.

Add Hole Patterns

Use the hole feature you just created to create a hole pattern.

TRY IT: Create a hole pattern from a hole feature

1 On the Part Features panel bar, click Rectangular Pattern.

2 In the graphics window, click the hole feature.

3 On the Rectangular Pattern dialog box, click the Direction 1 button, and then click the bottom horizontal edge of the part.

Click the Flip button to change the direction, if needed.

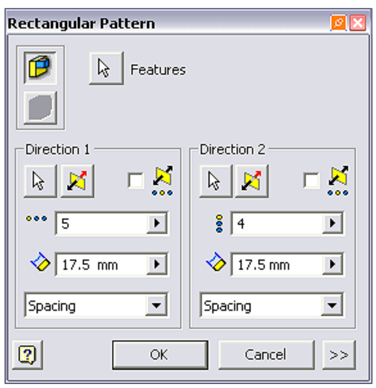
4 Verify that Spacing is selected in the drop-down list, and then in the Column Count field enter *5*, and in Column Spacing enter *17.5 mm*.

A preview of the pattern is displayed in the graphics window for Direction 1.

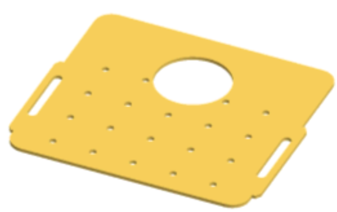
5 Click the Direction 2 Select button, and then click the leftmost vertical edge of the part.

6 Verify that Spacing is selected in the drop-down list, and then in the Column Count field enter *4*, and in Column Spacing enter *17.5 mm*.

In the graphics window, the preview of the pattern includes the occurrences in Direction 2.



7 Click OK to create a rectangle hole pattern.



In the next portion of this exercise, you suppress pattern occurrences.

Suppress Pattern Occurrences

A review of the design intent for the part shows that two unneeded occurrences were added. You can suppress all or individual occurrences in a pattern.

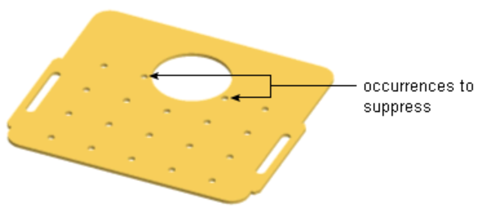
TRY IT: Suppress pattern occurrences

1. In the browser, Expand Rectangular Pattern1 to display the occurrences.

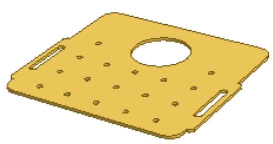
   Point to the occurrences. As the cursor points to each occurrence, it is highlighted in the graphics window.

2. Highlight the occurrence that did not execute. Press CTRL as you right-click the occurrence, and then click Suppress on the menu.

3. Suppress the two occurrences shown in the following figure.



The occurrences are suppressed, and your part looks like the following figure.



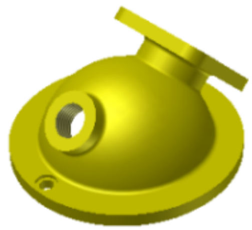
Close the file without saving or save the file with a new name to preserve the original data file.

Circular Patterns

In a previous exercise, you added hole features to a cylinder head for a face valve pump. In this exercise, you create a circular pattern using the counterbored hole.

TRY IT: Create a circular pattern

1 With the project *tutorial_files* active, open file *circpattern.ipt*.

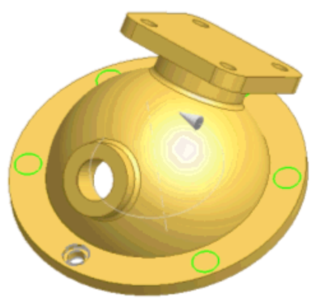


2 On the Part Features panel bar, click the Circular Pattern tool.

3 On the lower flange of the part, click the counterbored hole feature.

4 On the Circular Pattern dialog box, click the Rotation Axis button, and then in the browser, click Work Axis1.

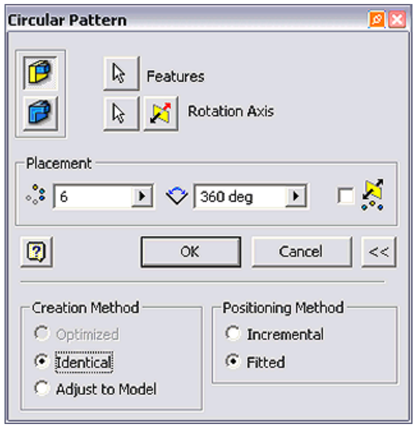
A preview of the pattern is displayed.



5 In Placement ➤ Count, verify that the value is *6*.

In this example, you can enter an incremental value of *60* or a fitted value of *360* for the positioning method.

6 Click the More button. In Positioning Method, verify that Fitted is selected.



7 Click OK to create a circular pattern.

Close the file without saving or save the file with a new name to preserve the original data file.

Mirror Features

You can mirror part, surface, and assembly features to create and maintain symmetry. By using a mirror feature, you can also reduce the amount of time required to create a model. You can mirror individual solid features, work features, surface features, or the entire solid. A mirror of the entire solid allows mirroring of complex features such as shells included in the solid.

Workflow overview: Mirror a part

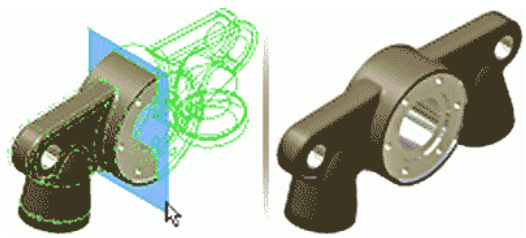
1 Create a part body to mirror. Create a work plane to serve as a mirror plane or, if you prefer, use a planar face as the mirror plane.

2  On the Part Features panel bar, click the Mirror Feature tool.

3 On the Mirror Pattern dialog box, click the Mirror Entire Solid button.

4 Click the Mirror Plane button, and then select a work plane or planar face.

5 Click OK.

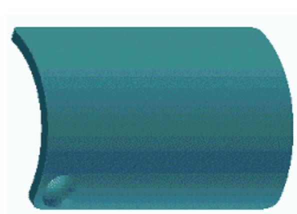


Patterns Along Paths

Use the Rectangular Pattern tool to create patterns of features that are placed along a 3D path. You can choose a surface or model body edge, work axis, line, arc, planar face, work plane, spline, or trimmed ellipse as the path for creating patterns.

Workflow overview: Create a rectangular pattern along a path

1 Create a part with a feature to pattern. Create a 3D path to use for patterning. If desired, you can use the edges of the feature to indicate direction for the path.



2 Click the Rectangular Pattern tool.

3 In the Rectangular Pattern dialog box, select the Pattern Individual Features option.

4 Click the Features button and in the graphics window or in the model browser, select features to arrange in a pattern.

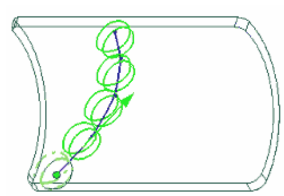
5 Click the Path selection button, and then select the path. Click Flip to change the column direction, if appropriate.

6 Enter the count (number of features) for the column, and then click the drop-down arrow to specify pattern length. Select one of the following options:

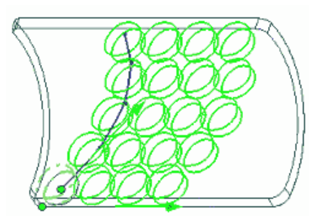
`Spacing:` Enter distance between features.

`Total distance:` Enter distance of the column.

`Curve length:` Length of selected curve is automatically entered.



7 To create a pattern with multiple rows, click Direction 2, and then set the row direction, count, and spacing, distance, or curve length.

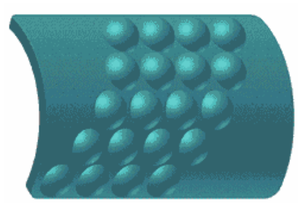


8 If appropriate, click the More button to set a start point for one or both rows, set Termination Method, and Orientation Method:

- If appropriate, click Start, and then click a point on the path to indicate the start of one or both columns. If path is a closed loop, a start point is required.

- Under Compute, select Optimized to create optimized pattern, Identical to create identical features, or Adjust to Model to terminate features when encounter a face.
- Under Orientation, select Identical to orient all features the same as the first selected, or Direction 1 or Direction 2 to specify which path controls the rotation of pattern features.

9 Click OK.



Suppress Pattern Occurrences

You can temporarily suppress the display of one or more solid or surface features in a pattern. You can also hide all or some work features. Features remain suppressed until you restore them.

Workflow overview: Control visibility of solid features

- To suppress all occurrences, select the pattern icon in the browser, right-click, and then select Suppress.
- To suppress an individual occurrence, expand the pattern icon in the browser, select the occurrence, right-click, and then select Suppress.
- To restore all occurrences, select the pattern icon in the browser, right-click, and then select Unsuppress Features.
- To restore individual occurrences, expand the pattern icon in the browser, select the occurrence, right-click, and then select Unsuppress.

NOTE Occurrences that were suppressed individually must be restored individually.

Analyzing Parts

Analyzing solids and surfaces provides information for validating the geometric quality before manufacturing. You can save several different analyses of the same or different types for a specific model. For example, you can define several ways to analyze a particular set of surfaces on the same model.

Once an analysis is applied, an Analysis folder is created in the browser and the analysis is placed in the folder. Each saved analysis is added to the browser in the order it is created. In the browser, the name and visibility of the active analysis is displayed along with the analysis folder name. For example, Analysis: Zebra1 (On).

You can use the Analysis folder in the browser to change the visibility of the active analysis and create new analyses. Expand the Analysis folder to view and manage all other saved analyses. You can switch the active analysis or edit, copy, and delete any saved analysis in the list.

The types of analysis you can perform include:

Zebra	Analyzes surface continuity by projecting parallel lines onto the model. Results show how light reflects off the surface to help you identify areas where the surface quality must be improved.
Draft	Evaluates if a model has adequate draft between a part and mold (based on the pull direction) and can be manufactured by casting. A spectrum shows draft angle changes within a specified range.
Curvature comb	Provides a visual analysis of the curvature and overall smoothness of model faces, surfaces, sketch curves, and edges.
Gaussian curvature	Evaluates areas of high and low surface curvature using a color gradient display on part surfaces. The gradient display is a visual indication of surface curvature that uses the Gaussian curvature analysis calculation.
Cross section	Provides a basic graphic view of the part at one section or detailed information and corresponding graphic about multiple sections of the interior of solid parts. It also analyzes whether the part adheres to the minimum and maximum wall thickness.

Workflow overview: Create and use analyses

1 Open a part file or double-click a part in an assembly.

2 Click Tools ➤ Analysis or click the arrow on the Analysis Visibility tool, and then select the type of analysis to create.

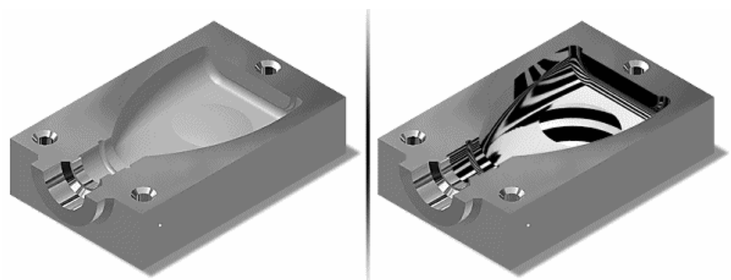
3 On the setup dialog box for the analysis, adjust the analysis settings as needed.

4 Apply the analysis.

5 Change visibility on the active analysis as needed.

6 Use the Model browser to edit, copy, delete, and rename saved analyses.

7 Optionally, create additional analyses for selected models and switch between them as needed.



Create Zebra Analyses

The zebra analysis evaluates faces by projecting parallel lines onto the model. Results show curvatures on the face to help identify areas that may be flat (stripes are parallel) or that are not continuously tangent (stripes are jagged where the curvature is not constant).

The active analysis is marked as On, for example, Analysis: Zebra1 (On). You can create an analysis based on an existing analysis. You specify the direction that shows the most contrast between stripes to indicate transition between surfaces, stripe thickness by relative proportion of black to white, and opacity of stripes.

Workflow overview: Create a Zebra Analysis

1 Open a part file or double-click a part in an assembly to in-place edit the part.

2 Click the arrow by the Analysis Visibility tool, and then select New Zebra Analysis from the list.

> **NOTE** After the initial analysis is saved, you can right-click the Analysis folder in the Model browser, and then select New Zebra Analysis from the context menu.

3 If appropriate, double-click the name to enter a custom name.

4 Specify horizontal, vertical, or radial direction.

5 Specify stripe thickness and density.

6 Specify the opacity of stripes.

7 Specify the display quality.

8 Select All, Faces, or Quilts and then select the appropriate geometry.

9 Click OK to analyze. If you prefer, click Apply to analyze and keep the dialog box open and continue to change settings and analyze as needed.

Create Draft Analyses

The draft analysis evaluates model faces for adequate draft between a part and mold based on the pull direction. Results show as a range of colors on the model by a specified angle range.

You can create an analysis based on an existing one. You specify the degree range to analyze for draft or draft angle, and select if draft analysis results are displayed in a gradient or in discrete color bands.

Workflow overview: Create a draft analysis

1 Open a part file or double-click a part in an assembly to in-place edit the part.

2 Click the arrow by the Analysis Visibility tool, and then select New Draft Analysis from the list.

3 If appropriate, enter a custom name.

4 Click New. If appropriate, double-click the name to enter a custom name.

5 Specify the degree range (relative to the pull direction) to analyze for draft angle.

6 Select Gradient to display results in a gradient rather than stripes.

7 Select All, Faces, Quilts, and then select the appropriate geometry.

8 Click an edge, axis, or planar face to specify pull direction or click Flip to reverse the direction.

9 Click OK.

For details on creating other types of analyses, refer to the Autodesk Inventor Help index under analysis.

Creating and Editing Work Features

5

In this chapter, you learn about creating and editing work features.

Defining Work Features

Work features are abstract construction geometry that you can use when other geometry is insufficient for creating and positioning new features. To fix position and shape, constrain features to work features.

Work features include work planes, work axes, and work points. The proper orientation and constraint conditions are inferred from the geometry you select and the order in which you select it.

The work feature tools provide on-screen prompts to help you with selection and placement. You can:

- Create and use work features in the part, assembly, sheet metal, and 3D sketch environments.
- Use and refer to work features in the drawing environment.
- Project work features into a 2D sketch.
- Create in-line work features to help you define a 3D sketch or position a part or assembly feature.
- Make work features adaptive.
- Turn the visibility of work features on or off.
- Drag to resize work planes and work axes.

Work Planes

A work plane is a flat plane extending infinitely in all directions along one plane. A work plane is similar to the default origin *YZ*, *XZ*, and *XY* planes. However, you create the work plane as needed, using existing features, planes, axes, or points to locate the work plane.

Use a work plane to:

- Create a sketch plane when no part face is available to create 2D sketched features.
- Create work axes and work points.
- Provide a termination reference for an extrusion.
- Provide a reference for assembly constraints.
- Provide a reference for drawing dimensions.
- Provide reference for a 3D sketch.
- Project into a 2D sketch to create curves for profile geometry or reference.

The following illustrations show some of the methods you can use to define a work plane.

Work Axes

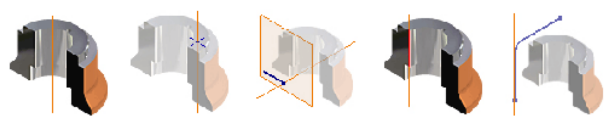
A work axis is a straight vector extending infinitely in two directions. A work axis is similar to the default origin *X*, *Y*, and *Z* axes, however, you create the work axis as needed, using existing features, planes, or points to locate the work axis.

Use a work axis to:

- Create work planes and work points.
- Project into a 2D sketch to create curves for profile geometry or reference.
- Provide a line of rotation for a revolved feature.

- Provide a reference for assembly constraints.
- Provide a reference for drawing dimensions.
- Provide reference for a 3D sketch.
- Provide reference for a circular pattern.
- Create lines of symmetry.

The following illustrations show some of the methods you can use to define a work axis.



Work Points

A work point is a point that exists relative to, and is dependent on, features or work features. A work point is similar to the default origin center point, however, you create the work point as needed, using existing features, planes, or axes to locate the work point.

Use a work point to:

- To create work planes and work axes.
- Project into a 2D sketch to create a reference point.
- Provide a reference for assembly constraints.
- Provide a reference for drawing dimensions.
- Provide a reference for a 3D sketch.
- Define coordinate systems.

The following illustrations show some of the methods you can use to define a work point.

Grounded Work Points

A grounded work point, like all work points, depends on an associated feature to determine its location. A grounded work point uses features or work features to initiate the grounded work point tool, but its position is then fixed in space and not dependent on, or associated to, those or other features.

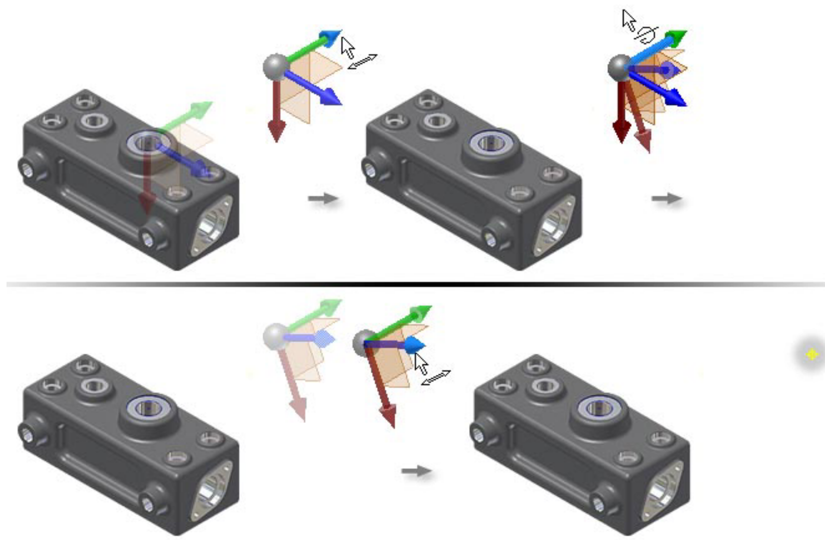
You can use a grounded work point in the same ways that you would a work point. However, the grounded work point remains fixed in space regardless of changes to model geometry. You can move a grounded work point with the 3D Move/Rotate tool.

Workflow overview: Define a grounded work point

1 On the Part Features panel bar, click the arrow on the Work Point tool and click Grounded Work Point.

2 Select a vertex, sketch point, or work point to initiate the 3D Move/Rotate tool. A pushpin cursor symbol indicates the selection is grounded.

3 The 3D Move/Rotate tool axes are aligned with the principal axes of the part. Click or drag an axis or center of the tool, and then enter values in the 3D Move/ Rotate dialog box, and click Apply.

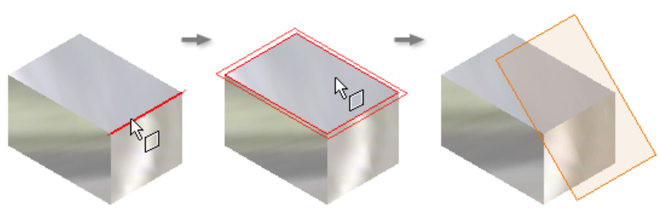


4 Continue to revise the position of the work point. When finished, click OK.

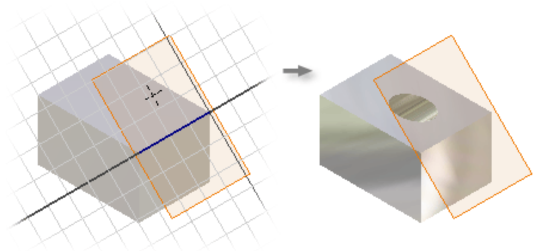
Modifying Work Features

Other than the grounded work point, all work features are associated to the features or geometry used to create them. If you modify or delete the locating geometry, the work feature changes accordingly. Conversely, any feature or geometry that is dependent on a work feature for its definition is also affected by changes to the work feature. Both scenarios are shown in the following illustrations.

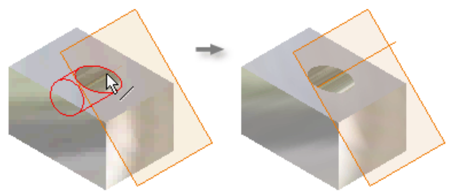
The work plane was created at a 45 degree angle to the top face.



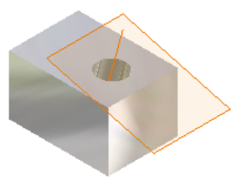
The hole was created from a sketch on the work plane, making the hole dependent on the work plane.



A work axis was added to the hole, making the work axis dependent on the hole.



If the angle of the plane is modified to 15 degrees, the hole and work axis adjust accordingly.



Using Projects to Organize Data

6

In this chapter, you learn how projects help you organize and manage your data. You learn to plan and set up your projects based on your design needs.

Key Terms

Term	Description
active project	The project that Autodesk® Inventor™ automatically defaults to when opening, saving, or editing components. There can be only one active project in an Autodesk Inventor session. Can be a project you have specified or the default installed project.
default project	A blank project installed with Autodesk Inventor. In the absence of a defined project, the default project is active.
editable locations	Folders where parts, drawings, assemblies, and presentations you create are stored. In a project, editable locations are the workspace and workgroup search paths. Use only one editable location per project.

Term	Description
frequently used subfolders	Named subfolders of project folders (including libraries) that are frequently accessed. Folders are not used to resolve references or store references. They are listed on file access dialogs so that you can easily locate the folders. The path to frequently used subfolders always begins with the name of the project location.
libraries	Libraries are project locations containing read-only files that are referenced, but not edited. Within a project, each library must have a unique name. Within a library, give each file a unique name.
options	Project settings that define such things as the number of old file versions to keep and whether unique file names are being used. Release ID (such as a project version) and Owner (such as who controls the released project) are intended for use with released projects.
project	An XML file with an *.ipj* extension that defines the folders that contain the files and folders of external data that the project files reference.
proxy file	When you place an Autodesk® Mechanical Desktop® part as a component in an assembly, Autodesk Inventor creates a proxy file that contains links to the Mechanical Desktop file and the translated data. Typically, the original Mechanical Desktop file is stored in a library and the translated proxy files are stored in a proxy library. The proxy library has the same name as the Mechanical Desktop library except that it is preceded by an underscore (_) character.

Term	Description
referenced file	A file used in the current design. A referenced file may be editable or it may be read-only, as in the case of library parts.
relative path	In projects, paths are relative to the location of the projects file (*.ipj*). Autodesk Inventor uses relative paths to locate referenced files.
root folder	A top-level folder defined as a library, workspace, or workgroup in a project. Data files in subfolders nested under the root folder are located by a relative path from the root folder. Store all files that belong to the current project in a root folder or one of its subfolders.
single user project	Data file locations are located in the workspace of the project. Files within the project are accessed by one user and one Autodesk Inventor session at a time. Files are not checked in or out.
styles library	A folder defined in the project Folder Options that specifies where the style definitions used in the project are located. The default is the Design Data folder.
templates folder	A folder defined in the project Folder Options that specifies where the templates used to create new files for the project are located. The default is the Templates folder created during setup.
workspace	A root folder or subfolder that contains your personal copy of design files. For single-user and vault projects, the workspace

Term	Description
	should be the only defined editable location.

Learn About Projects

A project represents a logical grouping of a complete design or product, including its model files, drawings, presentations, and design notes. Project information is stored in XML files with the *.ipj* extension that specify where you edit files, how many versions are retained when you save a file, where referenced data is stored, and other settings.

The project identifies the root folders where files are stored and the hierarchy of the project design. Data files can be stored directly in these root folders or in their subfolders.

You can create two project types: single user and, if you have installed Autodesk® Vault, a vault project. Single User projects are for designers working alone. When Autodesk Vault is installed, you use Vault projects to collaborate on projects with multiple designers. Common files are stored in a vault and never accessed directly, and each designer has a personal project that defines where the files are copied for viewing and editing. The vault also maintains version history of files as well as additional attributes.

Default Project

When you first start Autodesk Inventor, a default project is automatically active. The default project does not define an editable location, but you can use it to create designs immediately, and save files anywhere without regard to projects and file management. Generally, you use the default project for experimentation only, not actual design work. It is more difficult to migrate your files to a project when the design gets complex than it is to set up a project before you start designing.

Set an Active Project

In an Autodesk Inventor work session, only one project can be active. The active project specifies the options and paths to folders containing design files and libraries for the session. When you work on a different design project, you must make its project active before you can create or edit data files.

TRY IT: Make a project active

1. Verify that all Inventor files are closed.
2. Click File ➤ Projects, or on the Microsoft® Windows® Start menu, click Programs ➤ Inventor (release number) ➤ Tools ➤ Project Editor.
3. On the Projects dialog box, top pane, the existing projects are listed. Double-click a project to make it the active project. A check mark indicates the active project.

   The lower pane of the Projects dialog box shows information about the selected project in the top pane.

How Referenced Files are Found

When Autodesk Inventor searches for a file, it looks in the locations defined by the active project for the stored file name and relative path.

File references are stored according to the following rules:

- File references are stored as a relative path from the project root folder. If the file is in a library, the library name is also stored.
- If the referenced file is not in a project root folder, but the file is stored in the same folder or a subfolder of the file that references it, the reference is stored as a relative path.

To avoid file resolution problems, projects always use relative paths rather than absolute paths. This allows the project or its root locations to be easily moved or copied without breaking or needing to update references.

Always save new files in the workspace defined for the active project or one of its subfolders.

Except for library files, a file can be moved to a different non library location specified in the project, as long as the relative path is maintained.

Library locations contain library components that are referenced by another file, but not edited. Inventor searches for a library reference in the library location named in the project.

A project searches for nonliterary files in the editable locations. For best results, specify only one editable location in each project.

Because other design groups may also use the same library parts, library locations may be specified in multiple projects. It is a good practice to make library locations and the files in them read-only.

Setting Up Projects

Set the project type when you create or edit a project. The type determines where files can be edited and saved, who has access to files, and check-in and check-out behavior.

It is a good practice to set up your file structure and understand who can access the file data before you create a project.

Project Types

You can create these project types:

- Single user
- Vault (Autodesk Vault must be installed to use this project).

When multiple designers must have access to data, we recommend that you install Autodesk Vault. Its extensive data management capabilities let you keep all versions of a file and give you the ability to search and query design data.

Use single-user projects if you work alone or if no one else needs access to your files.

NOTE In the Project wizard, Semi-isolated and Shared project types are unavailable by default. It is recommended that you use Autodesk Vault to manage multi-user projects. If you have a requirement to create legacy project types, click Tools ➤ Application Options ➤ General tab. Select the Enable creation of legacy project types check box. Consult the online Help for more information about those project types.

Single-user Projects

Use single-user projects for individual designers:

- All design files are in one folder (the workspace) and its subdirectories, except for files referenced from libraries.
- Store the project file (*.ipj*) in the workspace (root) folder and specify .\ as the workspace.
- The file check-out status is not available in the browser.

Typical Single User project setup

Type	Single User
Workgroup Location	None
Location	One workspace defined at .\.
Included file	None
Libraries	One or more may be defined.
Frequently Used Subfolders	One or more may be defined, each specifying a subfolder of the workspace or one of the libraries.
Folder Options	Defines folders containing project-specific Styles, Templates, and Content Center Files.
Options	Old versions to keep on save = 1 Using unique file names = Yes Name = "Project Name" Shortcut = "Project Name" Owner = owner of the project Release ID = version of the released data

Vault Projects

To use the vault project, Autodesk Vault software must be installed. A different dialog box opens so that you can create a Vault project. Characteristics of a vault project include:

- Designers never view or work directly on the vaulted version of a file.
- Each designer uses a project file that defines a personal workspace where Autodesk Vault copies the vaulted files for viewing and editing.
- Changes to files made by other designers, and checked back into the vault, are not visible until you refresh your files to get the latest versions in your workspace.
- Autodesk Vault maintains copies of all of the previous checked-in versions of data files, and stores additions about edit history, file properties, and file dependencies in its database.
- You can set up queries on file properties, track file references, and retrieve past configurations.

For a vault project, you must have a workspace located at a path relative to the project file folder (such as .\ or .\workspace) and no other editable locations.

Typical Vault project setup	
Type	Vault
Location	One workspace defined at .\.
Workgroup locations	None
Included file	None
Libraries	One or more defined.
Frequently Used Subfolders	One or more may be defined, each specifying a subfolder of the workspace or one of the libraries.

Folder Options	Defines the folders containing project-specific Styles, Templates, and Content Center files.
Vault Options	Values are typically set in Autodesk Vault. Virtual folder = virtual folder within the vault database that maps to the root folder for the project. Publish folder = specifies where Streamline data is published.
Options	Old versions to keep on save = 1 Using Unique File Names - Yes Owner = team leader or blank Release ID = version of the released data

NOTE For more information about creating and using Vault projects, see the manual called *Autodesk Vault (version) Managing Your Data* in your product box. The manual is also available in PDF format on your product DVD at *ais (version)\dsk1\docs*.

Set Up Folder Structures

A typical project might have parts and assemblies unique to the project, standard components unique to your company, and off-the-shelf components such as fasteners, fittings, or electrical components.

To help Autodesk Inventor locate referenced files, it is a good idea to set up subfolders under your project workspace or workgroup folder. You can keep all your design files for a project in the subfolders.

As a design project grows, you often must move data. For example, you might change drives, move to a different server, add designers, or share data that was previously unshared. When you maintain a simple folder structure, it is easier to accommodate increased design complexity, more designers, and distributed data.

Use these guidelines to create a folder structure for project files:

- Follow your company standards and naming conventions for the project folders.
- If you plan to edit files from existing designs, copy them to a subfolder of the workspace.

- If you intend to reference released design files, copy them to a library folder, or define a library in your project that locates the root folder of the released project. If the released project also references libraries, include them in the project or use Pack and Go to flatten the file structure into a single folder.
- Keep the subfolder structure relatively flat and do not store files that are unrelated to the project under the root folder. Avoid storing more than one hundred files in a single folder.

NOTE Set the project option Using Unique File Names to Yes. Avoid using duplicate names, even when naming files in different paths, so that you do not confuse locations or documents, or overwrite files.

If you set the “Using Unique File Names” option in the project file, Autodesk Inventor tries to locate a file within the project folder structure, even if moved to a different location or renamed. If the file cannot be located, the Resolve Link dialog lets you search for it manually. To find duplicate file names in project folders, use the Find Duplicate Files button on the Project Editor.

Because references are stored as relative paths from project folders, if you change the folder structure, move, or rename folders, you are likely to break file references, unless you set the Using Unique File Names option to Yes.

Creating Projects

The default projects folder location is *My Documents/Inventor*, but you can change to a different location.

The Project Wizard creates a workspace in the same folder as the project file. If you change that setting, keep the workspace as subfolders of the folder containing the project file.

This table summarizes recommendations for each project type.

	Single User	Vault
Project type	Single User	Vault
Included file	None	None

Workspace locations	One defined at .\	One defined at .\
Workgroup locations	None	None
Libraries	One or more	One or more *not* nested under workspace

TRY IT: Create a project with the Project Editor

1 Close all documents in Autodesk Inventor.

When files are open, the active project is read-only. The only exception is that you can add libraries without closing all files.

2 In Inventor, click File ➤ Projects to activate the Project Editor, or

outside Autodesk Inventor, click the Microsoft Windows Start menu ➤ Programs ➤ Autodesk ➤ Autodesk Inventor (version) ➤ Tools ➤ Project Editor.

3 When the Project Editor opens, click the New button to start the Inventor project wizard.

In the Inventor project wizard, the default is a new single user project, which creates a workspace.

You must install Autodesk Vault to use the New Vault Project option.

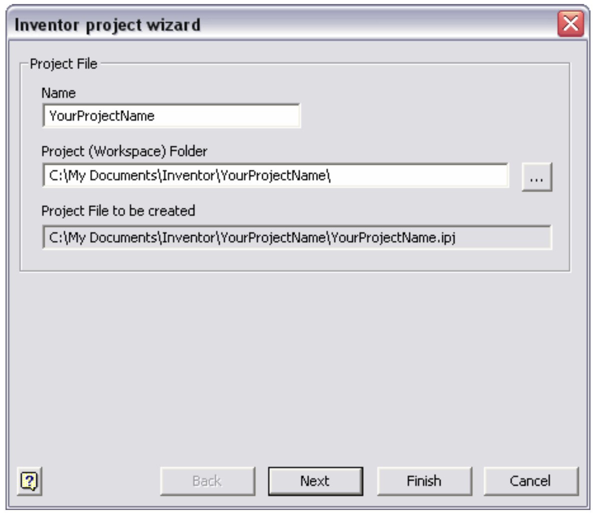
4 In the Project File box of the project wizard, specify required project information.

- In Name, enter the project name or accept the default.
  The Project wizard creates a subfolder with that name as a subfolder of the projects folder.

  If you browse to a different location, Autodesk Inventor uses the name of the folder you locate, or creates a new folder only if the folder name does not exist.

- In the project folder (the location of the *.ipj* at .\), a workspace is created for single-user and vault projects.

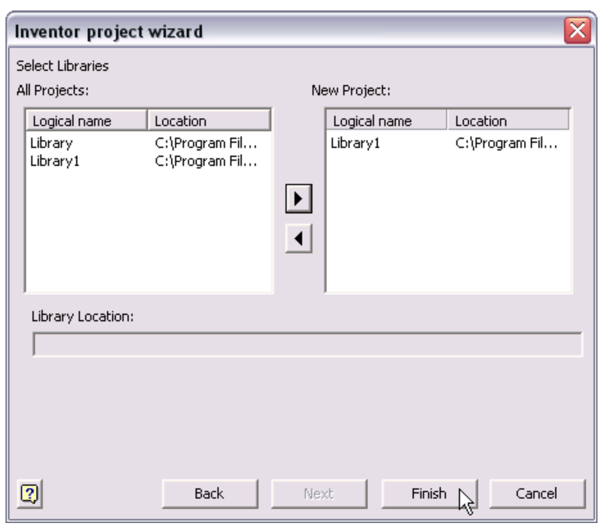
- In Project File to be created, the path and project file name is displayed.



5 Click Next to specify libraries.

New projects often use the same libraries as existing projects. In the left pane, libraries are collected from all project files in the projects list.

- In the right pane, click the right arrow to add a library location to the New Project.
- Click the left arrow to remove a library location from the New Project.



The Library location box shows the location of a library selected in the left or right panes.

6 Click Finish.

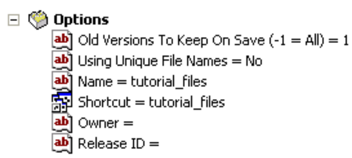
   Once the project is created, double-click it in the Project Editor, and then customize it by setting options. In the next sections, customize the project you just made by following the procedures.

Set Project Options

The project type, the default workspace, and library names and locations are set in the project wizard. Set the remaining project options to suit your design environment, and then specify project search paths.

If useful, review the option definitions and the setups for each project type discussed earlier in this chapter:

- "Single-user Projects"
- "Vault Projects"



Workspace

A workspace contains files that you edit. They are not accessible by other designers and changes you save to the workspace are not visible to other designers. The workspace may be a network location, but your files open faster if you specify a folder on your local hard disk.

You can specify a workspace in single-user projects. In vault projects, the workspace is required and is the only editable location. It must be in a subfolder or the same folder as the folder containing the project (*.ipj*) file.

When setting the workspace path:

- Specify only one location in a workspace, preferably at the root location containing the project file or in a subfolder of the project file. The recommended location is .\.
  In a single user project, the workspace should be the only location in your project, except for library locations.
- For best results, use Autodesk Vault to check out rather than manually copy files. Check out automatically sets the correct relative path so that Autodesk Inventor can resolve your file. If you manually copy, the file is not marked as checked out and the risk of overwriting edits is increased.
- Workspace is the first location searched for non library references.

Workflow overview: Set the workspace search path

1. On the Project Editor dialog box, click the Expand button to show the Workspace path.
2. If no workspace is defined, right-click Workspace, and then click Add Path or expand the existing workspace and click Edit.
3. Accept the path shown, edit the path, or browse to and select a workspace folder.
4. Click Save.

Library Locations

A library is a storage location for read-only parts or other files that will not be edited. You specify one or more library locations when you create a project. Libraries can be used in multiple projects.

NOTE When naming libraries, remember that files using a library part store the library name and relative path from the library folder. If you later need to rename the library, all the library references will be broken.

Libraries can be created implicitly (when linking to a Mechanical Desktop part) or explicitly (you create the search path in the project).

Keep in mind:

- The location of the Content Center library is defined in the Project Editor Folder Options.

- A library creates an association between a library name and the folder for the current project.
- The relative path stored in the referencing file is relative only to the library folder, not to any other project locations. Only the named library is searched when resolving a library reference.

Workflow overview: Set library locations

1. In the Edit Project pane of the Project Editor, right-click Libraries, and then select an option:

Add path	Enter a folder name and path in the box or browse to the location.
Paths from file	Browse to, select a project (*.ipj*) file, and then click Open. The local search paths from the selected file are added to the current project.
Paste path	Paste a copied path into the box.
Delete section paths	Delete all paths.

2. Click Save.
3. If appropriate, right-click a library location, and then select menu options.

Library Locations for Mechanical Desktop Parts

Mechanical Desktop parts are stored in libraries. Mechanical Desktop parts are linked to Autodesk Inventor assemblies through a proxy file that contains linking information, so that the assembly component updates when the part is edited in the Mechanical Desktop.

Before you add Mechanical Desktop parts to an assembly, set a library location in the project for both the folder containing the Mechanical Desktop part files and a corresponding proxy folder. The library names are the same except that an underscore character (_) precedes the proxy library.

Because Mechanical Desktop files can contain more than one part, multiple Autodesk Inventor parts might be retrieved from a single file.

About proxy files:

- A proxy file is created when you place a Mechanical Desktop part as a component in an assembly. Parts retrieved from a Mechanical Desktop file are stored in a corresponding proxy library location.

- You cannot open or edit a proxy file.
- Proxy files are updated only when you open or save the assembly that uses the corresponding Mechanical Desktop file.
- Proxy files contain the Mechanical Desktop design data for a single part, translated to the Inventor data format.
- Design properties (iProperties) are stored in proxy files, and are lost if you lose the proxy file.

Workflow overview: Set Library locations for Mechanical Desktop parts and proxy files

1. Verify that all Inventor files are closed.
2. In the Project Editor, double-click a project to activate it.
3. In the lower pane, right-click Libraries, and then click Add Path on the menu. Add a new named path for the Mechanical Desktop file.
4. Right-click the newly created library location, and then click Add Proxy Path on the menu.
5. Continue to define libraries as needed, assigning a descriptive name to each library.

Library Locations for iParts and iAssemblies

If you plan to store iPart or iAssembly factories on a network and share them with others, add the factories as libraries. You need to specify a proxy path.

If no proxy path is used, when you insert an iPart or iAssembly in an assembly, a subfolder is automatically created in the same location as the factory. The subfolder has the same name of the factory and stores the members.

When you use a proxy path, the subfolder is created in the proxy path and the members are created in the subfolder in the proxy path.

Keep these guidelines in mind:

- Store iParts and iAssemblies in a named library and the corresponding published members in the associated proxy library.
- You cannot change the location or name of a proxy library.

- You cannot open or edit a proxy file.
- iPart and iAssembly Factory proxy files are catalog elements from the factory, published with a specified set of parameters.
- Update proxy files by opening or saving the assembly that uses the corresponding factory member file.
- Do not make an iPart or iAssembly proxy library folder read-only, because the factory has to create new members there.

Workflow overview: Set library locations for factories and proxy files

1. Verify that all Inventor files are closed.
2. In the project editor, double-click a project to activate it.
3. In the lower pane, right-click Libraries, and then click Add Path on the menu. Add a new named path for the factory location.
4. Right-click the new library location, and then click Add Proxy Path on the menu.
5. Continue to define library locations as needed, assigning a descriptive name to each library.

Content Center Files

When you select a component from the Autodesk Inventor Content Center, it is created from parameters stored in a Content Center library. If the file is not a custom file, it will be saved in a folder specified in the Project Editor Folder Options.

The Content Center library also generates custom parts that have a standard cross section, such as structural steel shapes. You can create an infinite variety of parts by specifying other parameters such as length. You need to specify the file name and save location for all such parts. If you intend to make other edits, such as insert bolt holes or other cutouts, place the custom files in a subfolder of your workspace location.

Other Types of Libraries In Projects

Most projects use libraries such as third party components, company collections of commonly used parts, Mechanical Desktop parts, iAssemblies and iParts. However, you may find that your organization has other component files that you would like to reference but do not intend to edit.

Because library references include the library name, and only that library location is searched to resolve a library reference, library file names need only be unique within that library.

Library examples include:

- Another design set as a library.
  You can use project libraries as a way to subdivide projects among divisions, geographic boundaries, small teams, or subcontractors.

  A design set library can be a single part or a set of complete assemblies of a released design, an assembly or component purchased from a third party and that you cannot change, or a snapshot of another team's design project that is in development.
- Components that your organization commonly uses across design projects, such as preferred components readily stocked or provided by preferred suppliers.
- Components that you have designed in the past and do not need to recreate for successive projects.

Workflow overview: Define a set of files as a library

1. Place the files in a folder under a common root folder.
2. In the Project Editor, right-click Libraries.
3. Browse to the desired top-level folder.
4. Press ENTER to set the path.

   When a parent file is opened in a project, only the library named in the reference is searched.

Avoid Duplicate File Names

In general, it is not a good idea for different files to have the same file name, even if they reside in different folders. Inventor uses search rules to resolve references and duplicate names can lead to difficulty in locating a file. Avoid using duplicate file names and set the project option Using Unique File Names to Yes.

Libraries are an exception to this rule. Often the files come from third-party vendors that have their own file naming schemes. As a result, different libraries can have files with the same name, but files within a library should have unique names.

For simplicity and clarity, when you have a choice, avoid using duplicate file names for library parts and regular parts.

Library references differ from editable references because the library name associated with the library location is stored as part of the reference.

Creating and Opening Files In Projects

To open the files associated with a project, you must first make the project active. Close any open Inventor files.

TRY IT: Create or open a file in a project

1. Click the Open tool on the Inventor toolbar.
2. On the Open dialog box left pane, click the Projects button and then double-click the project you want to make active. The active project is identified with a check mark.
3. In the What to Do pane, click the New button to list file templates. Select a template to create a new file, and then click OK.

   When you save the new file, browse to the desired subfolder (if any) of the Workspace and specify a unique file name.
4. On the Inventor toolbar, click the Open tool. In the Open dialog box, in the Locations pane, project categories are listed in a browser tree. A selected location is highlighted and marked with a blue folder. Its files are listed in the Look In pane. Double-click a category or subfolder to change the file list.

NOTE If you specified Frequently Used Subfolders in the project options, they are also listed in the Locations pane.

5 Double-click a file to open it.

Managing Assemblies

7

This chapter introduces assembly modeling. You will learn about the assembly environment, assembly browser and working in the assembly environment.

Assembly Environment

In Autodesk® Inventor™, you place components that act as a single functional unit into an assembly document. Constraints define the relative position these components occupy with respect to each other.

When you create or open an assembly file (*.iam*), you are in the assembly environment. Assembly tools manipulate whole subassemblies and assemblies. You can group parts that function together as a single unit and then insert the subassembly into another assembly.

When you open a part file (*.ipt*), you are in the part environment. Part tools manipulate sketches and features, which combine to make parts. You insert parts into assemblies and constrain them in positions they will occupy when the assembly is manufactured.

You can insert parts into an assembly or use sketch and part tools to create parts in the context of an assembly. When you do this, all other components in the assembly are visible.

To complete a model, you can create assembly features that affect multiple components, such as holes that pass through multiple parts. Assembly features often describe specific manufacturing processes such as post-machining.

The assembly browser is a convenient way to activate components you want to edit, edit sketches, features, and constraints, turn component visibility on and off, and other tasks.

Assembly Design Strategies

Traditionally, designers and engineers create a layout, design the parts, and then bring everything together in an assembly. With Autodesk Inventor, you can create an assembly at any point in the design process instead of at the end. For a clean sheet design, you start with an empty assembly and create the parts as you develop the design. To revise an assembly, you create the new parts in-place so they mate with existing parts. This design methodology supports top-down, bottom-up, and middle-out design strategies.

The optimal order in which you create parts and subassemblies depends upon your answers to the following questions:

- Can you modify an existing assembly or do you have to start a new one?
- Can you break the larger assembly down into subassemblies?
- Can you use existing parts or iFeatures?
- Which constraints drive the functionality of the design?

Changes you make to external components are automatically reflected in your assembly models, and the drawings you use to document them.

Bottom-Up Assembly Design

When you design from the bottom up, you place existing parts and subassemblies into an assembly file, and position components with assembly constraints, such as mate and flush. If possible, place components in the order in which they would be assembled in manufacturing.

Unless component parts are built from adaptive features in their part files, they might not fit the requirements of an assembly design. You can place such a part in an assembly, and then make the part adaptive in the assembly context. The part is resized in the current design when you constrain its features to other components.

If you want all underconstrained features to adapt when positioned with assembly constraints, designate a subassembly as adaptive. When a part in the subassembly is constrained to fixed geometry, its features are resized as needed.

Top-Down Assembly Design

When you design from the top down, you begin with design criteria and create components that meet those criteria. Designers list known parameters and may create an engineering layout (a 2D design that evolves throughout the design process).

A layout can include contextual items such as the walls and floor where an assembly will stand, machinery that feeds into or receives output from the assembly design, and other fixed data. Other criteria such as mechanistic characteristics may also be included in the layout. You can sketch the layout in a part file, then place it in the assembly file. You develop sketches into features as the design evolves.

The final assembly is a collection of interrelated parts that are uniquely designed to solve the current design problem.

Middle-Out Assembly Design

Most assembly modeling combines the strategies of bottom-up and top-down design. Some requirements are known and some standard components are used, but new designs must also be produced to meet specific objectives.

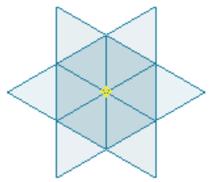
Usually, you begin with some existing components and design other parts as required. You analyze the design intent, then insert or create the grounded (base) component. As you develop the assembly, you place components or create new ones in place, as required.

Assembly Coordinate System

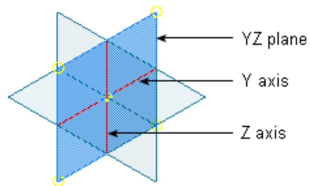
A new assembly file has three default work planes and work axes. The point of intersection of the work axes is the origin of the assembly coordinate system.

In the browser, the default work planes, work axes, and the center points are listed under the Origin icon. These features are initially hidden in the graphics window, but you can right-click them in the browser and select Visibility to display them. You can constrain components to the work planes and the origin.

In the following illustration, visibility has been activated for the assembly default work planes, axes, and center point, with the default isometric view.



Each default workplane is coplanar with its respective axes. For example, the YZ plane is coplanar with the *Y* axis and the *Z* axis.



Assembly Constraints

Assembly constraints are applied to components to define positional relationships in the assembly. For example, you can force two planes on separate parts to mate, or specify that a hole and a bolt always remain concentric. These constraints bind your assembly model together and tell Autodesk Inventor how to adjust the model as its component definitions change over time.

Assembly Analysis

After you create your assemblies, you can calculate mass properties and check for part interference. Properly constrained assemblies can be animated through a range of motion, so you can check for design problems.

Storing Data Files In Projects

Projects point to locations where data files for a design project are stored. In most cases, you will be working with a design team that houses data files in a main location. Individual designers create projects that specify a private workspace.

Projects manage component locations by specifying such things as:

- The master location of files (the workgroup), when you work in a design team.
- A private workspace specified by each designer, where files are created and edited.
- Libraries of standard and custom components.
- Locations of templates and style libraries.
- The names of frequently used subfolders, to make file location faster.

In addition to locations, projects also set preferences for other options such as whether you use unique file names in a project file structure (which can help locate files more easily), how many versions of a file to keep, and release information.

Working with the Assembly Browser

The Assembly browser shows the hierarchy of components in the assembly, as well as their relationships and dependencies. Each occurrence of a component is represented by a unique name. From the browser, you can select a component for editing, move components between assembly levels, control component status, rename components, edit assembly constraints, and manage representations.

In-Place Activation

You can only edit components or features in the active assembly. Double-click a subassembly or component occurrence in the browser to activate it, or right-click the occurrence in the browser, and then select Edit. In the browser, all components not associated with the active component are shaded.

The following actions can be performed on the first-level children of the active assembly:

- Delete a component.
- Display the degrees of freedom of a component.
- Designate a component as adaptive.

- Designate a component as grounded.
- Edit or delete the assembly constraints between first-level components.

The features of an activated part can be edited in the assembly environment. When you activate a part, you are working in the part environment.

Double-click a parent or top-level assembly to reactivate it.

Visibility of Components

Controlling component visibility is critical to managing large assemblies. You may need some components only for context, or the part you need may be obscured by other components. Assembly files open and update faster when the visibility of nonessential components is turned off.

The visibility of any component in the active assembly can be changed, even if the component is nested many layers deep in the assembly hierarchy.

Workflow overview: Change the visibility of a component

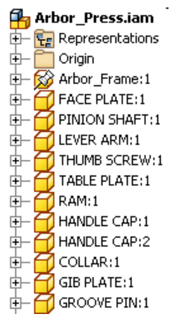
1. Expand the browser until the component occurrence is visible.
2. Right-click the occurrence, and then clear the check mark on Visibility.

Combinations of visible components can be stored in design view representations, described later in this chapter.

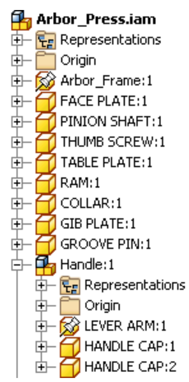
Assembly Structures

The structure of an assembly is the organization of the components. Grouping parts into subassemblies simplifies the browser. Subassemblies can also reflect manufacturing processes. With Autodesk Inventor, you can change the contents of subassemblies or create new ones at any point during the design process and over the life span of a product.

The top level of an assembly structure can consist of parts and subassemblies. Each subassembly can consist of parts and more subassemblies. Moving a component (a part or subassembly) into a subassembly is *demoting*. Moving a component out of a subassembly is *promoting*.



assembly with flat structure



assembly after restructure

Components restructured as a group maintain constraints between them. Constraints to components outside the group are lost.

Restructure Assemblies

In the browser, components are initially listed in the order in which they were placed in the assembly. You can rearrange components by dragging them to a new position in the browser or by using the context menu. Collapse subassemblies at the same level in the browser to ensure that the dragged components remain at the same assembly level. Moving components in the browser does not affect their position in the graphics window.

You can also create a new subassembly by selecting a group of components. The subassembly name is entered into the browser and the related components are nested under it.

Workflow overview: Create a new subassembly containing selected components

1. Start with an open assembly.
2. Select components from the assembly browser or in the graphics window.
3. Right-click, and then select Component ➤ Demote. The Create In-Place Component dialog box displays.
4. Enter a file name for the new assembly, select a new template if necessary, and then click OK.

   A new subassembly is created and populated with the selected components.

Promote is not available if the selected component is a child of the top-level assembly.

Workflow overview: Promote components within the assembly hierarchy

1. Select components from the assembly browser or in the graphics window.
2. Right-click, and then select Component ➤ Promote.

   The selected components are moved to the parent assembly.

NOTE For more information, see Changing File Structure on page 290.

Browser Display

Display controls are located in the Browser toolbar:

Browser Filters	Lists browser filters that limit and organize what is shown in the Assembly browser. The filters can be turned off and on, and multiple filters can be applied to the browser at the same time.
Design View Representations	Lists recently created Design View representations. Also gives access to activating and creating new Design View representations by selecting the Other... option. Design View representations preserve an assembly display configuration that captures component visibility, sketch and feature visibility, color and style characteristics, zoom magnification, and viewing angle.

Assembly View	Nests assembly constraint symbols below both constrained components. Part features are hidden. Selecting this button disables Modeling View.
Modeling View	Places assembly constraint symbols in a folder at the top of the browser tree. Part features are nested below parts, just as they are in part files. Selecting this button disables Assembly View.

Graphics Window Display

The physical appearance of a part in a shaded view of an assembly is initially determined by the material or color style assigned in the part file.

Parts use the color style defined by the material applied in the part file. The color style can be overridden either in the part file or in an assembly file. A color style override in the part file becomes the default or As Material color of the part in all assemblies. A color style override of a part or subassembly in an assembly is local to that assembly only.

Some examples of color overrides are:

- Changing the color style of adjacent parts to provide contrast
- Assigning a semitransparent color style to a component for better visualization
- Grouping components based on similar functionality or origin, such as all hydraulic components, all components from a specific vendor, or all critical failure parts

Component color styles can be repeatedly changed and saved in separate assembly design view representations. You can specify a design view representation when creating a drawing so that only needed components are visible.

Autodesk Inventor includes a range of standard materials and color styles, as well as tools to create custom color styles and material definitions.

To define a color or to modify the characteristics of a defined color, such as its brightness, intensity, or opacity, click Format ➤ Styles Editor ➤ Colors.

Producing Bills of Materials

You can create a bill of materials (BOM) for an assembly. A bill of materials is a table that contains information about parts in an assembly, such as quantities, names, costs, vendors, and all of the other information someone manufacturing the assembly might need.

Bill of materials information is automatically collected from iProperties. You can modify values on the bill of materials by changing the design properties on the Properties dialog box or on the Bill of Materials dialog box.

Within a drawing, you can create a similar table called a parts list. The parts list uses data in the assembly bill of materials.

Tips for Working with Assemblies

- Turn off visibility of nonessential components so you can access parts you need and update graphics faster.
- Create design view representations that highlight specific design problems or assembly subsystems, and apply them when opening the assembly model.
- Turn off part adaptivity after you size a component, to speed up solutions and prevent accidental changes.
- Assign different colors to components. Select colors from the Color list on the Standard toolbar.
- Click a component in the browser to highlight it in the graphics window.
- Right-click a component in the browser and use Find in Window to locate a component in a complex assembly.
- Use color to identify component groups. Using attributes, find components in specific subsystems or from specific vendors and color code them in named representations.

Placing, Moving, and Constraining Components

8

In this chapter, you learn how to place and constrain components, and to edit constraints using the Edit Constraints dialog box.

Placing Components In Assemblies

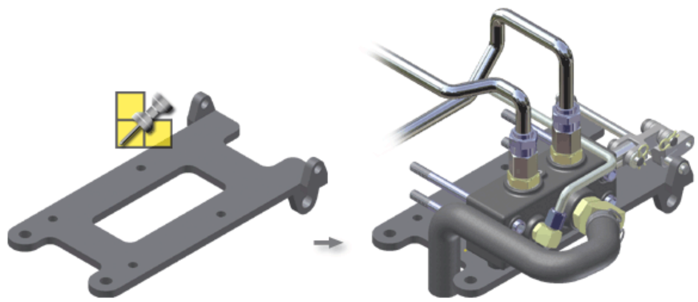
In the assembly environment, you can add existing parts and subassemblies to create assemblies or you can create new parts and subassemblies in-place.

A component (a part or subassembly) can be an unconsumed sketch, a part, a surface, or any mixture of both.

When you create a new component in-place, you can sketch on one of the assembly origin planes, click in empty space to set the sketch plane to the current camera plane, or constrain a sketch to the face of an existing component.

When a component is active, the rest of the assembly is dimmed in the browser and graphics window. Only one component can be active at a time.

When you place the first component in an assembly, choose a fundamental part or subassembly (such as a frame or base plate). Except for the first placed component, all placed components are unconstrained and ungrounded. You add the constraints you need.

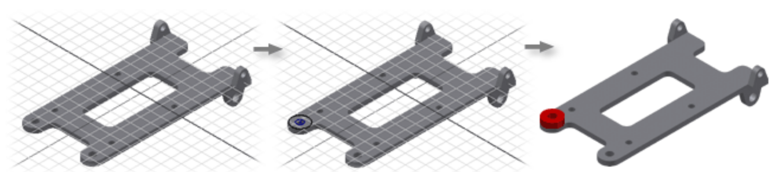


The first component placed in an assembly is automatically grounded (all degrees of freedom are removed). Its origin and coordinate axes are aligned with the origin and coordinate axes of the assembly. It is a good practice to place assembly components in the order in which they would be assembled in manufacturing.

Click in the graphics window to place additional ungrounded occurrences of the first component in the assembly. To finish placing the first component, right-click, and then select Done.

To place subsequent components from external files, continue to click in the graphics window to place additional occurrences, and then right-click and select Done.

Use the Create In-Place Component tool to create a component in the assembly context. The created component is nested under the active main assembly or subassembly in the browser. If a sketch profile for the in-place component uses projected loops from other components within the assembly, then that sketch profile is associatively tied to the projecting components.



Drag Components into Assemblies

You can place multiple components in an assembly file in a single operation by dragging them into the graphics window. You can drag components to an open assembly window from the following locations:

- From an open folder in Windows Internet Explorer®. Use this technique to quickly populate a new assembly with components.
- From an open Inventor part file. Drag the top-level icon from the part browser to the assembly graphics window.
- From an open Inventor assembly file. Drag parts, subassemblies, or the top-level assembly from the browser to the assembly graphics window.

You must drop the files over the graphics window where the assembly model is displayed. A single occurrence of each component is placed in the assembly file. The dropped components appear at the bottom of the browser in the receiving assembly.

Simplify Assemblies

In an assembly, you can employ several techniques to make it easier to work in the graphics window and to removed unneeded components from the display or memory. Techniques include:

- Turn visibility on and off
- Turn enable status on or off
- Suppress unneeded components

Sometimes a component obscures a component you need to work on. You can turn its visibility off or turn off visibility for all components you are not working on to simplify the graphics window.

Enabled components are fully loaded in the assembly and are available for any operation within the assembly environment. A component that is not enabled is selectable in the browser, but is not available for operations in the graphics window. You can in-place edit a component that is not enabled, which automatically switches the component to Enabled. Components that are not enabled consume fewer computer resources than enabled components, giving better performance in large assemblies.

If you are working in shaded mode, components that are not enabled are nearly transparent in the graphics window. In wireframe mode, they are displayed in a distinct color in the graphics window. An icon in the Assembly browser identifies the component as not enabled.

Parts and subassemblies that are required only for context, or components that do not require editing, are good candidates to be designated as not enabled. To set a component to not enabled, right-click the component in the browser, and then clear the check mark next to Enabled.

You can also suppress unneeded components. When you suppress a component, it is not loaded into computer memory, which makes editing the assembly much faster. In the browser, a suppressed component is indicated by strikethrough text. In a large assembly, consider, for example, suppressing interior components that are not needed in current operations.

Grounded Components

Grounded components are fixed in position, relative to the assembly coordinate system. A grounded component will not move when you apply assembly constraints. The first component placed or created in an assembly is automatically grounded, so that subsequent parts may be placed and constrained in relation to it. You can remove the grounded status of a component, including the first component.

To restore the degrees of freedom (unground) of a component, right-click the component occurrence in the graphics window or the Assembly browser, and then clear the check mark beside Grounded. Grounded components are displayed with a pin icon in the assembly browser.

There is no limit to the number of grounded components you can have in an assembly, but most assemblies have only one. Grounded components are appropriate for fixed objects in assemblies because their position is absolute (relative to the assembly coordinate origin), and all degrees of freedom are removed.

Other Sources of Components

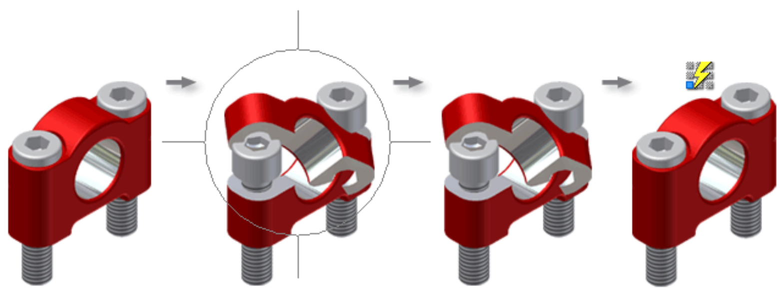
Most assembly components are parts and subassemblies you have previously created with Autodesk® Inventor™ or are generated by the Component Center, a library of standard parts installed with Autodesk Inventor.

You can use the DWG/DWF™ File Wizard to import parts and assemblies from the Autodesk® Mechanical Desktop®. You should migrate Mechanical Desktop files to the latest version before you translate to Autodesk Inventor. You have an opportunity to fix any errors before the translation.

Autodesk Inventor can also place components created in other CAD systems that have been saved as SAT files (ACIS) or IGES files, or exported through a STEP translation process. Imported SAT, STEP, and IGES files do not retain the parametric information used to create their features. You can add parametric features to these files, but you cannot edit their existing features.

Moving and Rotating Components

When constraining assembly components, you may need to temporarily move or rotate a constrained component to enhance the visibility of other components, or position a component to facilitate constraining. Rotating or moving a component temporarily suspends a component's constraints. The next assembly update restores the position of the component as determined by its constraints.



If a component is not grounded, or is unconstrained or underconstrained, you can click it and drag it to a new location in the assembly graphics window.

Constraining Components

After you place or create components in an assembly file, use assembly constraints to establish the orientation of the components in the assembly and to simulate mechanical relationships between components. For example, you can mate two planes, specify that cylindrical features on two parts remain concentric, or constrain a spherical face on one component to remain tangent to a planar face on another component.

Each time you update the assembly, the assembly constraints are enforced.

- You can make some parts adaptive. Autodesk Inventor allows adaptive part features to change size, shape, and position based on the applied assembly constraints.
- Assembly constraints remove degrees of freedom from components, positioning them relative to one another. As you modify component geometry, assembly constraints ensure that the assembly stays together, following the rules you have applied.
- The correct application of assembly constraints also permits interference checking, collision and contact dynamics and analysis, and mass property calculations. When you apply constraints properly, you can drive the value of an essential constraint and view the movement of components in the assembly.

Place Constraints

In Autodesk Inventor, four types of 3D assembly constraints define positional relationships between components: mate, angle, tangent, and insert. Each type of constraint has multiple solutions defined by the direction of a vector normal to the component. The constraint solution is previewed to show the orientation of the affected components before you apply the constraint.

In addition, motion and transitional constraints simulate intended movement.

- Motion constraints specify the intended motion between components. Because they operate only on open degrees of freedom, they do not conflict with positional constraints, resize adaptive parts, or move grounded components.
- A transitional constraint specifies the intended relationship between (typically) a cylindrical part face and a contiguous set of faces on another part, such as a cam in a slot. A transitional constraint maintains contact between the faces as you slide the component along open degrees of freedom.

Use the Place Constraint dialog box to control the type, solution, and offset for the constraint.

- Use the Selection buttons to specify the geometry to be constrained. The selection buttons are color cued to the corresponding geometry in the graphics window.

- Use the Predictive Offset and Orientation button with Mate, Flush, and Angle constraints. When turned on, it gives the offset value for the current location for the selections you are constraining. It also changes the orientation to a flush constraint if you have it set to mate, then pick two faces with the vectors pointing in the same direction, and visa versa.

The dialog box remains open as you place constraints, so you can place multiple constraints of all types.

In the following workflow, the Constraint tool on the Assembly toolbar is used to place a tangent constraint between assembly components. A tangent constraint positions faces, planes, cylinders, spheres, cones, and ruled splines tangent to one another.

Workflow overview: Place a tangent constraint in an assembly

1. To begin, place the components to constrain in an assembly file.
2. On the Assembly panel, click the Constraint tool.
3. On the Place Constraint dialog box, Assembly tab, in Type, click the Tangent button.
4. The First Selection button is already active. Select a face, curve, or plane for your first selection.
5. The Second Selection button is activated after you pick the first selection. Select the geometry that will be tangent to the first.
6. If applicable, select Inside or Outside to specify the tangency position.
7. Enter an offset value, if applicable.
8. If Show Preview is selected, observe the effects of the applied constraint. If either component is adaptive, constraints are not previewed.
9. Click Apply to continue to place constraints or click OK to create the constraint and close the dialog box.

NOTE Availability of objects for selection differs, depending upon the particular constraint tool you select in the Place Constraint dialog box.

If other components obscure the required geometry, do one of the following:

- Temporarily turn off the visibility of the foreground objects before you place a constraint.

- On the Place Constraint dialog box, select Pick Part First. Click the component you want to constrain. Clear the check box to restore the ability to select all components.
  Selectable geometry is limited to features on the selected component.
- Point the cursor to the required geometry. Right-click, and then choose Select Other.
  Click the arrows in the Select Other box to cycle through the underlying face, curve, and point selections.

  Click the green center button to accept the highlighted selection.

If you find it difficult to select faces, edges, or points, you can adjust the Locate Tolerance option to change selection priority. Click Tools ➤ Application Options ➤ General tab.

Workflow overview: Edit constraints

1. In the browser, right-click a constraint you previously placed.

   The Edit Constraint dialog box is displayed.
2. On the Edit Constraint dialog box, specify a new constraint type (Mate, Angle, Tangent or Insert).
3. Enter a distance to offset constrained components from one another.

   If you apply an angular constraint, enter the angle between the two sets of geometry. You can enter positive or negative values. The default value is zero.

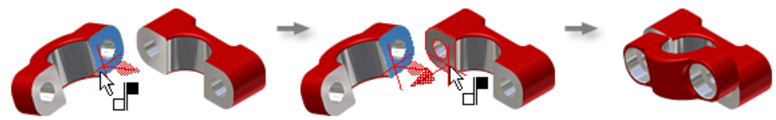
   If Show Preview is selected in the Constraint dialog box, the position of the components is adjusted to match the offset or angle value.
4. Apply the constraint through the Constraint dialog box or the context menu.

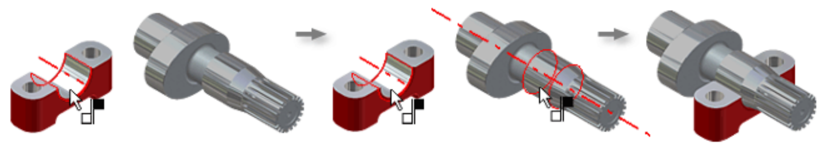
   The dialog box remains open, and you can apply as many assembly constraints as required.

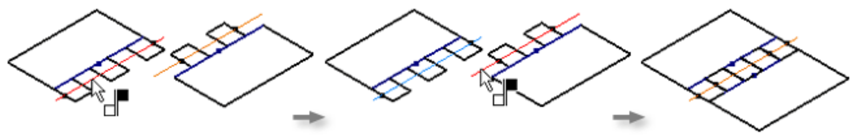
The following figures show before and after examples of applied assembly constraints.

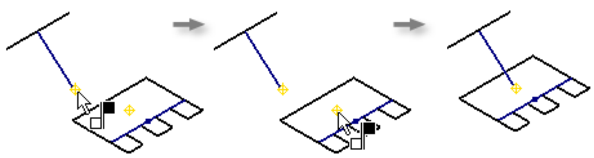
Mate Constraint

The mate constraint makes a set of geometry on one component coincident with geometry on another component.



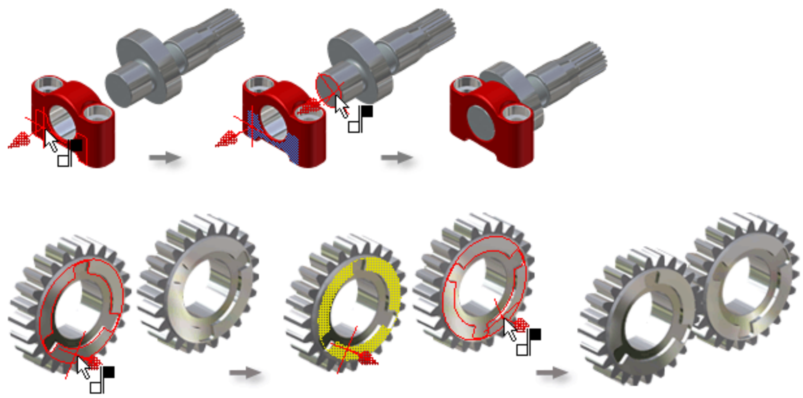






Mate Type - Mate Solution	Use the mate constraint with the mate solution to make two planes face each other and make them coplanar, make two lines collinear, or place a point on a curve or plane.
Mate Type - Flush Solution	Use the mate constraint with the flush solution to align two components so that the selected planes face the same direction, or have their surface normals pointing

in the same direction. Faces are the only geometry that can be selected for this constraint.



Angle Constraint

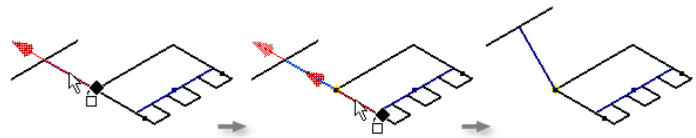
The angle constraint specifies an angle between planes or lines on two components.

Angle Type Specifies an angle between planes, axes, or lines on two components. The two sets of geometry need not be of the same type. For example, you can define an angle constraint between an axis and a plane. Constraints of this type are often used to drive assembly motion.

Angle Solution Orients the surface normal of a selected plane or the direction of the axis described by a selected line. When you select a face or line, an arrow shows the default direction of the solution.

Direct Angle Applies the right-hand rule. Some cases, such as zero or 180°, may flip in the opposite direction.

Undirected Angle Applies either right-hand or left-hand rule. The left-hand rule is applied automatically if the solved position more closely resembles the last-calculated position. This is the default behavior.

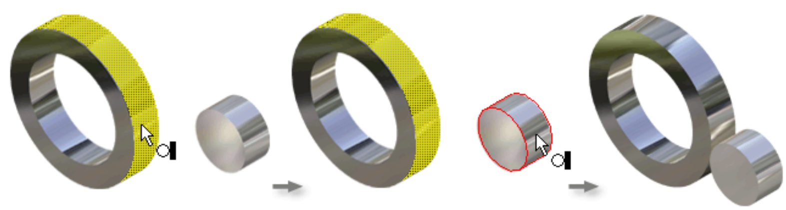


Tangent Constraint

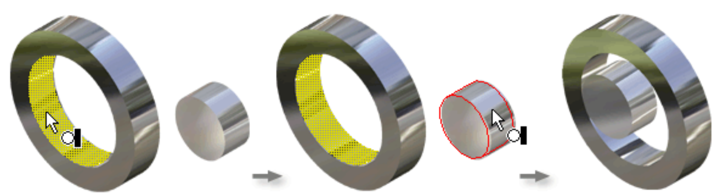
The tangent constraint causes surfaces of planes, cylinders, spheres, or cones to contact at the point of tangency.

Tangent Type At least one surface must be non planar. Surfaces defined by spline curves cannot be used in a tangent constraint. Tangency may be inside or outside a curve, depending on the direction of the selected surface normal.

Outside Solution Positions the first selected part outside the second selected part at their tangent point. Outside tangency is the default solution.



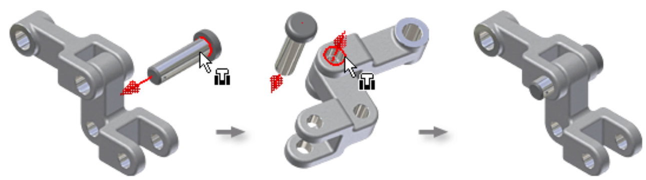
Inside Solution Positions the first selected part inside the second selected part at their tangent point.



Insert Constraint

The insert constraint causes a circular edge on one component to be concentric and coplanar with a circular edge on another component. The offset value for an insert constraint is the distance between the two faces containing the circular edges. For example, you can use this constraint to place a pin or a capscrew in a hole.

Solutions	Specifies the direction of the face normal for the planes containing the circular edges. An arrow indicates the normal direction. The opposed solution has the two planes facing each other, as in a mate constraint. An aligned solution has the normals pointing in the same direction.



Motion Constraints

You can add motion constraints to components in an assembly to animate the motion of gears, pulleys, rack and pinions, and other devices. By applying motion constraints between two or more components, you can drive one component and cause the others to move accordingly.

Two types of motion constraints are possible:

Rotation	Use to apply motion constraints to wheels, pulleys, and gears.
Translation	Apply motion constraints to rack and pinion or wheel and rail components. These constraints are bidirectional and accept a specified ratio or distance.

Motion constraints do not maintain positional relationships between components.

Fully constrain assembly components before you apply motion constraints. Then, suppress constraints that restrict the motion of the components you want to animate. To return components to their original positions, unsuppress any suppressed constraints.

iMates

An iMate is a constraint that is saved with a component to tell it how to connect with other components in an assembly. When you insert a component with an iMate, it snaps into place with another component with a matching iMate. The component can be replaced by another component while preserving these intelligent iMate constraints. Components with iMates speed accurate placement and replacement of components in assemblies.

Autodesk Inventor can infer iMates based on a special algorithm that places the constraint in a location likely to be the most useful. For example, you can infer iMates on a closed-loop circular edge for extruded, revolved, and hole features.

An iMate is usually an insert or mate constraint, but can be any constraint type that is useful for quickly positioning components. You may want to consider which other components may be substituted for the present component and which constraint strategy is most useful for an iMate.

A composite iMate is a collection of individual iMates into a single entity. Parts drawn from standard libraries snap together quickly with composite iMates. Visual and audio cues are provided to assist in the placement of components with iMates.

Viewing Constraints

The assembly browser provides two schemes for showing assembly constraints. You select Position View or Assembly View on the assembly browser toolbar to switch between the two schemes.

Modeling View	Shows each constraint under component occurrences in the browser. Constraints are listed under both constrained components.
Assembly View	Shows all assembly constraints collected into a folder labeled Constraints, located immediately below the top-level assembly. Each constraint is listed only once, in the order of placement.

When you hover the cursor over an assembly constraint in the browser, the constrained components are temporarily highlighted in the graphics window. Selecting the constraint in the Assembly browser highlights the geometry in the graphics window until you click again in the graphics window or the browser.

Editing Constraints

You can edit assembly constraints two ways.

Workflow overview: Edit constraint values by selecting in the browser

1. In the assembly browser, select an assembly constraint.

   The offset or angle value is displayed in the edit box at the bottom of the browser.

2. Enter a new value in the edit box, open the drop-down list to select recent values, or use the Measure tool to find a value.

Workflow overview: Edit constraint values in the Edit Constraints dialog box

1. Right-click a constraint in the assembly browser, and then select Edit from the menu, or double-click a constraint in the assembly browser.
2. On the Edit Constraint dialog box, edit any of the constraint parameters displayed.

You can change the selected geometry for one or both components, change the solution, and revise the offset, angle, or depth value of the constraint. Under certain conditions, the constraint type can be changed without losing the current selections. For example, you can change a mate constraint between two planar surfaces to an angle constraint. The OK button is not available if you select a new constraint type that cannot be applied.

Tips for Managing Assembly Constraints

- Start constraining components by mating planar faces. Add tangent, angular, and flush constraints later.
- Apply constraints after features are stable. Avoid constraints between features that might be removed later in the design process.

- Drag components to check translational degrees of freedom. You can see how a component is constrained.
- Create component iMates for repeated use. Using iMates, you can define placement information on parts and assemblies to use repeatedly.

Creating Assemblies

9

In this chapter, you learn how to create parts and assemblies in place, about adaptive parts, patterns, assembly features, and other procedures for managing assemblies.

Creating Assembly Components

Assembly modeling combines the strategies of placing existing components in an assembly, and creating other components in place within the context of the assembly. In a typical modeling process, some component designs are known and some standard components are used, but new designs must also be created to meet specific objectives.

Parts In Place

Use the Create Component tool to create a component in an assembly. When creating an in-place component, you sketch on the face of an existing assembly component or a work plane, a sketch plane in the camera view of the main assembly, or you can place the sketch plane normal to the view with the origin at a selected point. On the Create In-Place Component dialog box, you can choose an option to automatically constrain the sketch plane to the selected face or work plane.

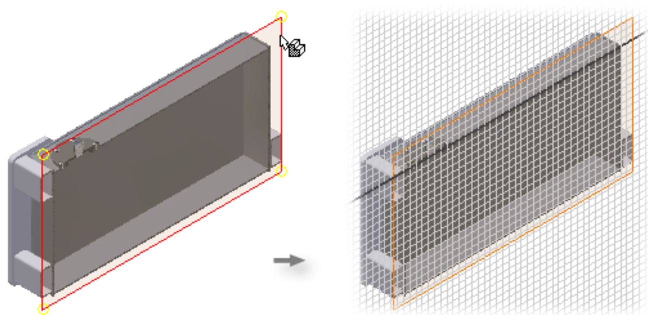
When you specify the location for the sketch, the new part immediately becomes active, and the browser, panel bar, and toolbars switch to the part environment. The Sketch tools are available to create the first sketch of your new part. Edges and features of existing components can be selected as reference geometry for sketching.

Most components are created in relation to existing components in the assembly. Optionally, you can click the graphics window background to define the current view orientation as the *XY* plane. If the *YZ* or *XZ* plane is the default sketch plane, you must reorient the view to see the sketch geometry.

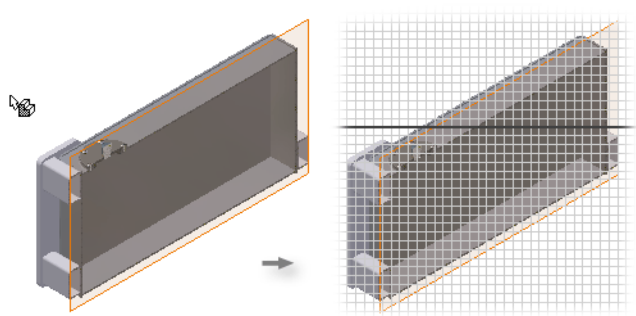
After you create the base feature of your new part, define additional sketches based on the active part or other parts in the assembly. When defining a new sketch, click a planar face of the active part or another part to define the sketch plane on that face. You can also click a planar face and drag the sketch away from the face to automatically create the sketch plane on an offset work plane.

When you create a sketch plane on a face of another component, an adaptive work plane is created and the active sketch plane is placed on it. The adaptive work plane moves as necessary to reflect any changes in the component on which it is based. When the work plane adapts, your sketch moves with it. Features based on the sketch then adapt to match its new position.

In the following illustration, the grid shows the sketch plane for an in-place part created on the plane used for a section view.



The following illustration shows the sketch plane for an in-place part created by clicking in the graphics window.



Workflow overview: Set a default sketch plane to create a component in place

1 Click Tools ➤ Application Options ➤ Part tab.
2 In the Sketch on New Part Creation box, select a sketch plane for the default.
3 Click OK.
4 Double-click the assembly name in the browser to return to the assembly.
5 In the browser header, click the arrow and select Assembly View.

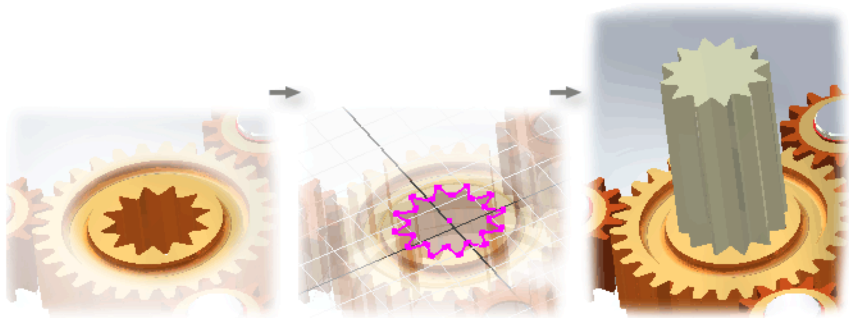
In the assembly view of the browser, assembly constraints are nested below the component with which they are associated. (In Model View, all constraints are collected in a single folder at the top of the browser).

If you selected the Constrain Sketch Plane to Selected Face or Plane option when you created your new part, a flush constraint appears in the assembly browser, and it can be deleted at any time. No flush constraint is generated if you create a sketch by clicking in the graphics window.

Projected Edges and Features

Parts created in place often need to match one or more features on existing components. Holes may be required to be concentric, or the outer edges of the new part must match those of an existing component. Faces, edges, and features on any visible component can be projected onto the current sketch. You can then use the projected points and curves to form sketch profiles or paths, or you can combine projected geometry with new sketch geometry if necessary.

The following illustration shows how edges from an adjacent part are projected into the sketch for an in-place part.



Projected geometry remains associated with the part from which it was projected and automatically updates to match changes in the original part's geometry.

When you project geometry from an existing component onto a new sketch it becomes reference geometry. You can use reference geometry to create an adaptive matching part that automatically updates to reflect any modifications to the outer boundary of the component from which the geometry was projected.

Use the Project Geometry tool on the Sketch toolbar to project faces, edges, and features onto the current sketch.

Projected geometry is positioned normal to the current sketch plane. If the selected edge lies in a plane that is not parallel to the sketch plane, the projected geometry is different from the original sketch. Reorient the view to the sketch to see a true view of the projected geometry.

Subassemblies In Place

When you create a subassembly in place, you define an empty group of components. The new subassembly automatically becomes the active assembly, and you can start to populate it with placed and in-place components. When you reactivate the parent assembly, the subassembly is treated as a single unit in the parent assembly.

Optionally, you can select components at the same assembly level in the browser, right-click, and then select Component ➤ Demote to place them into a new subassembly. You are asked to specify a new file name, template, location, and default bill of materials structure. You can then move components between assembly levels by dragging components in the browser.

Subassemblies can be nested many layers deep in a large assembly. By planning and building subassemblies, you can efficiently manage the construction of very large assemblies. Additionally, you can create subassemblies that match the intended manufacturing scheme to facilitate creating your assembly documentation.

Guidelines for Selecting Subassembly Components

When designing a subassembly for modeling, select:

- Component groups that repeat in an assembly.
- Combinations of standard parts that are common to many assemblies.
- Components that combine to perform a common function in an assembly.

When designing a subassembly for documentation purposes, select components that match your intended manufacturing scheme.

As you change the active assembly, the appearance of components in the graphics window changes. If you are working with a shaded display, the active subassembly is shaded and all other components appear translucent. If you are working with a wireframe display, all components other than the active subassembly are shaded light gray.

Any placed or new in-place components become part of the active assembly or subassembly. Double-click the parent assembly in the browser to make it the active assembly.

Creating Component Patterns

Components can be arranged in a rectangular or circular pattern in an assembly. Using component patterns can increase productivity and efficiently match your design intent. Typically, you may need to place multiple bolts to fasten one component to another or place multiple parts or subassemblies into a complex assembly.

As with feature patterns, you can create a rectangular pattern by specifying column and row spacing, or a circular pattern by specifying the number of components and the angle between them.

In addition, you can create associative component patterns of parts or subassemblies by selecting an existing pattern. For example, you could create a component pattern of a nut and bolt by selecting an existing bolt hole pattern. Edits to the bolt hole pattern control the location and number of bolts and nuts.

Associative component patterns:

- Include and retain constraints of the original component. If the original component is constrained, then the component pattern is constrained.
- Are associative to a part feature such as a pattern of bolt holes.
- Contain individual elements that can be suppressed for display or functional purposes.

Workflow overview: Create an associative component pattern

1. Place a component in an assembly file.
2. Constrain the position of the component relative to a feature pattern.
3. Click the Pattern Component button, and then select the Associative tab.
4. From either the browser or in the graphics window, select the placed component.
5. On the Associative tab, click the selection arrow, and then select an occurrence of a feature in a pattern from the graphics window.
6. Click OK.

   The placed component is patterned relative to the placement and spacing of the feature pattern. Changes made to the feature pattern automatically update the number and spacing of the components.

Workflow overview: Create a rectangular component pattern

1. Place a component in an assembly file.
2. Click the Pattern Component button, and then select the Rectangular tab.
3. From either the browser or in the graphics window, select the placed component.
4. On the Rectangular tab, click the Column Direction selection arrow and then select an edge or work axis from the graphics window. Click flip to the column direction, if necessary.

5. Enter the number of components to be created in the column and the spacing between each.
6. On the Rectangular tab, click the Row Direction selection arrow, and then select an edge or work axis from the graphics window, enter the number of components in the row, and the distance between the components.

   Click flip to the row direction, if necessary.
7. Click OK.

Workflow overview: Create a circular component pattern

1. Place a component in an assembly file.
2. Click the Pattern Component button, and then select the Circular tab.
3. From either the browser or in the graphics window, select the placed component.
4. On the Circular tab, click the Axis Direction selection arrow, and then select an edge or work axis from the graphics window. Click flip to the axis direction, if necessary.
5. Enter the number of components to be created in the circular pattern and the angular spacing between each.
6. Click OK.

Independent Instances

You can also make one or more component pattern elements independent of a pattern. When you make an element independent:

- The selected pattern element is suppressed.
- A copy of each component contained within the element is placed in the same position and orientation as the suppressed element.
- The new components are listed at the beginning of the assembly browser.
- Replacement components obey the rules for component replacement.

Workflow overview: Make a pattern element independent of a pattern

1 Expand the pattern in the browser.

2 Right-click an element other than the source component, and then select Independent.

   The element is suppressed and a copy of the components it contains is added to the browser.

NOTE To create a new component based on another component, save a copy with a different name and place the copy in the assembly.

You can restore an independent element to the pattern at any time by right-clicking it in the browser, and then clearing the check mark on Independent. The copied components created when the element was made independent are not automatically deleted from your model.

Creating Assembly Features

Assembly features are similar to part features except that they are created in the assembly environment, can affect multiple parts, and are saved in the assembly file.

Assembly features include chamfers, fillets, sweeps, revolved features, extrusions, holes, move face, rectangular feature pattern, circular feature pattern, and mirror. They also include the work features and sketches used to create them. The workflow and dialog boxes are the same as for part features, but some operations are not available (such as creating a surface for extruded and revolved features).

You can edit, add to, suppress, or delete assembly features. You can also roll back the state of the assembly features and add or remove components that participate in the feature.

Use Assembly Features

Assembly features describe processes that are applied after a model is assembled. Use assembly features to:

■ Define a single logical feature that spans multiple parts, such as an extrusion cut through multiple connecting plates.

- Describe a specific manufacturing process, such as match drilling or post-machining.

Components can be constrained to assembly features. You cannot, however, place a constraint between an assembly feature on one part and the same assembly feature on another part.

You can roll back the state of assembly features to view the effect of each assembly feature on the model or to place additional assembly features in the desired context. When rolled back, newly created assembly features are added above the End of Features (EOF) symbol in the browser.

Using Work Features in Assemblies

In the assembly environment, work features help you construct and position components and check for clearance in an assembly. You can also use work planes to help you generate section views of your assemblies. Create work planes and axes between parts in an assembly by selecting an edge or point on each part. These work features are associated with both parts and adjust accordingly as the assembly is modified.

By default, all work geometry is initially visible. You can turn visibility off and on for all work features at once. This is important in the assembly environment, where the display of work features from individual parts can quickly clutter the graphics window.

Workflow overview: Control visibility of work features in an assembly

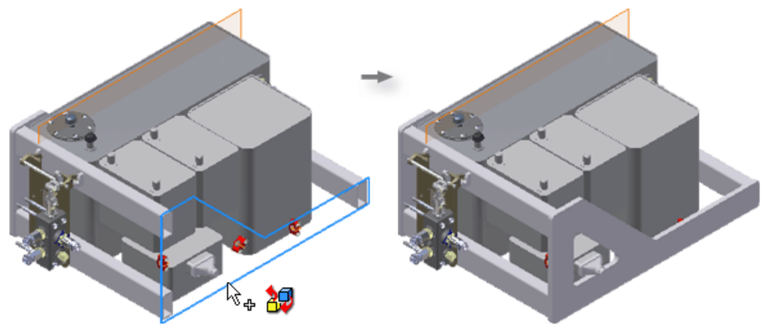
1. On the Standard toolbar, click View ➤ Object Visibility.
2. On the menu, turn off or on the work features by type or select All Work Features.

   This overrides the visibility setting for individual work features of that type in the assembly and in each part in the assembly. Although the work feature visibility in the assembly is suppressed, individual visibility control remains turned on.

Replacing Components

You may need to replace a component in an assembly as the design evolves. A simple representation of a component may be used during the design concept stage, which is replaced by the actual part or subassembly when detailed design is required. Parts from one vendor may be replaced with similar parts from another supplier.

In the following illustration, the Replace Component tool is used to replace a simple sketched representation with the actual part.



When you replace a component in an assembly, the new component is positioned with its origin coincident with the origin of the component it replaces. All assembly constraints from the original component are lost. You must place new assembly constraints to eliminate degrees of freedom of the new component

If the replacement part has an iMate constraint, and the part to which it must be constrained has a matching iMate, the parts snap together automatically with all constraints intact.

If the part you are replacing is an ancestor of the original part (a copy of the part that contains edits), then constraints are not lost during component replacement.

An assembly can be converted to an iAssembly, a special assembly that may contain many members with different characteristics such as length or diameter, constraint offsets, or other variations. When you use a member of an iAssembly as a component, you can exchange one member for another in the active assembly. To switch one member for another of an iAssembly nested deeply in the browser hierarchy, you must first make the immediate parent assembly active.

Mirroring Assemblies

The Mirror Component tool is useful for designing symmetrical parts. Use it to create a mirror of a source assembly and its components across a mirror plane. You create half of the assembly, and then mirror it to create the second half. The mirrored components are exact copies, positioned relative to the mirror plane.

You can either save a new assembly file with mirrored components and open it in a new window, or reuse components and add the mirrored components to the existing assembly file.

TRY IT: Mirror assembly components

1. Open the assembly you want to mirror.
2. On the Assembly panel bar, click the Mirror Components tool.
3. In the graphics window or the assembly browser, select all of the components of the assembly. Select the assembly, or a parent subassembly to automatically select all of the children.

   The assembly and its components are listed in the Mirror Components dialog box browser.
4. On the Mirror Components dialog box, click Mirror Plane, and then select the plane in the graphics window or the assembly browser.
5. Click the status button of a component to change its selection status as needed.

	Status	Description
	Mirrored	Creates a mirrored instance in the current or a new assembly file.
	Reused	Creates a new instance in the current or a new assembly file.
	Excluded	Subassembly or part is not included in the mirror operation.
	Mixed Reused/Excluded	Indicates that a subassembly contains components with reused and excluded status, or that a reused subassembly is not complete.

6. To change the orientation of a reused component, right-click the component, and select a Symmetric Plane.

7 Click the More button to select preview options and specify handling of content library components:

- To enable the mirrored state for library components, clear the Reuse Content Library Components check box.
  By default, only instances of the library parts are created in the current or new assembly file.
- To display status of mirrored components in the ghost color in the graphics window, In Preview Components, select check boxes.

8 Click OK.

9 On the Mirror Copy: File Name dialog box, review the copied files and make changes as needed:

- To edit the file name, click in the New Name box.
  To search in listed file names, right-click in the New Name column.
  To find and replace a string, click Replace.
- To change the location from Source Path to Workspace or User Path, right-click in the File Location box. If you select User Path, click in the File Location box and set the path.
  Keep the default location so the file can be located when you reopen the assembly.

10 In the Naming Scheme box:

- Select the Prefix check box, and then enter a prefix, if appropriate.
- To increment numbered files, select the Increment check box.
- Accept the default suffix (_MIR) or enter a different suffix. Clear the check box to remove the suffix.

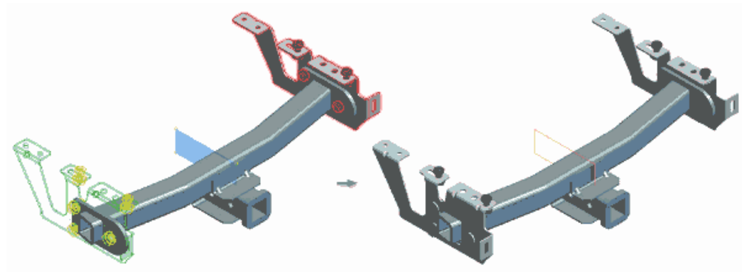
NOTE If you remove the suffix, give the file a unique name to avoid overwriting the original file.

11 Click Apply to update the file names, or click Revert to return to the original values.

12 In the Component Destination box, choose one of the following:

- To place components in the current assembly or a new assembly file, click Insert in Assembly.
- To open a new assembly file, click Open in New Window.

13 If you need to change status or select new components, click Return to Selection. Otherwise, click OK to accept and close the dialog box.



Copying Assemblies

Use the Copy Component tool to create a copy of a source assembly or its components.

You can either create a new assembly file and open it in a new window, or add copied components to an existing assembly file. Each copied component creates a new file. You can reuse components instead of copying them.

The resulting copied components are not associative, and are not updated if the original components are modified.

TRY IT: Mirror assembly components

1 On the Assembly panel bar, click the Copy Components tool.

2 In the graphics window or the assembly browser, select components to copy. The selected components are listed in the Copy Components dialog box browser.

   To automatically select all of the children, select the parent assembly or a subassembly.

3 Click the status button of a component to change its component status as needed:

	Status	Description
	Copied	Creates a copy of the component. Each copied component is saved in a new file.

	Status	Description
	Reused	Creates a new instance in the current or new assembly file.
	Excluded	Subassembly or part is not included in the copy operation.
	Mixed Reused/Excluded	Indicates that a subassembly contains components with reused and excluded status, or that a reused subassembly is not complete.

4 To enable copying of library components, click the More button and clear the Reuse Content Library Components check box.

5 Choose OK to open the Copy Components: File Name dialog box.

6 On the Copy Components: File Name dialog box, review the copied files and make changes as needed: to search in listed file names

- To edit the file name, click in the New Name box. To search in listed file names, right-click in the New Name column, and choose Find. To find and replace a string, choose Replace.
- To change the location from Source Path to Workspace or User Path, right-click in the File Location box. If you select User Path, click in the File Location box and set the path.
- Keep the default location so the file can be located when you reopen the assembly.

7 In the Naming Scheme box:

- Select the Prefix check box, and then enter a prefix, if appropriate.
- To increment numbered files, select the Increment check box.
- Accept the default suffix (_CPY) or enter a different suffix if appropriate.
- To remove the suffix, clear the check box.
  If you remove the suffix, give the file a unique name to avoid overwriting the original file.

8 To update the filenames, click Apply. To return to the original values, click Revert.

9 On the Component Destination box, choose one of the following:

- To place components in the current assembly file, click Insert in Assembly.
- To open a new assembly file, click Open in New Window.

10 To change status or select new components, click Return to Selection. Otherwise, click OK.

10 Analyzing Assemblies

In this chapter, you learn to analyze assembly components for interference by simulating the motion of the assembly components.

Checking for Interference

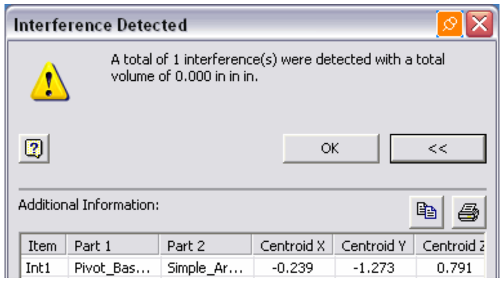
In the physical assembly built from your design, two or more components cannot occupy the same space at the same time. To check for such errors, Autodesk® Inventor™ can analyze assemblies for interference.

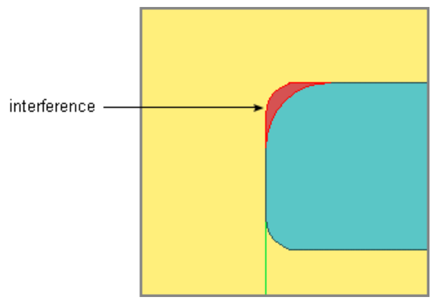
The Analyze Interference tool checks for interference between sets of components and among the components in a set. If an interference exists, Autodesk Inventor displays the interference as a solid and displays the volume and centroid in a dialog box. You can then modify or move the components to eliminate the interference.

Analysis takes longer for complex assemblies. An efficient strategy is to analyze only a few components at a time, such as those in close proximity to one another. It is a good idea to analyze, reposition, and redesign components on a regular basis rather than analyzing the complete assembly all at once.

To check interference within a set, select all components in the set. All parts in the set are analyzed against each other, and interferences are shown in red.

To speed up the process, you can select only the components that you want to check. For example, if you revise a part in an assembly, you can limit the interference check to the components affected by the change. The volume and location data are displayed when you click More in the dialog box, as shown in the following illustration.





NOTE Creating components in place, using faces of adjacent components as sketch planes, and projecting geometry from other component faces for use in sketches reduces the chance of interference between parts.

Workflow overview: Analyze interference between parts

1. Activate the assembly that you want to analyze. Interference analysis is only available in the assembly environment.
2. Click Tools ➤ Analyze Interference.
3. Select the two sets of components to be analyzed.
4. Click OK. The Interference Detected dialog box is displayed.
5. Expand the dialog box to see a detailed analysis report in table form. The report table can be copied to the clipboard or printed.

Checking for Degrees of Freedom

Each unconstrained component in an assembly has six degrees of freedom (DOF). It can move along or rotate about each of the X, Y, and Z axes. The ability to move along X, Y, and Z axes is called translational freedom. The ability to rotate around the axes is called rotational freedom.

Whenever you apply a constraint to a component in an assembly, you remove one or more degrees of freedom. A component is fully constrained when all degrees of freedom (DOF) have been removed.

Autodesk Inventor does not require you to completely constrain any component in an assembly. You can save time by removing only critical DOF for your model. There are situations in which you do not remove DOF. For example, do not remove DOF so that Inventor can correctly interpret the design intent when the assembly is animated, or to leave yourself design flexibility for a later phase in the design process.

To show the DOF symbol for all components in an assembly, select Degrees of Freedom from the View menu.

The straight lines with arrows at one end represent the translational degrees of freedom along the *X, Y,* and *Z* axes. The arcs represent rotational degrees of freedom about each axis.

Unconstrained Drag

You can move unconstrained components by dragging them in the graphics window.

Partially constrained components sometimes need to be moved or rotated to facilitate constraint placement. Use the Move Component and Rotate Component tools to temporarily release all assembly constraints so you can reorient a component. Any assembly constraints that have been placed are reapplied to the assembly as soon as you click Update.

Constrained Drag

Dragging a single constrained component causes other components in the assembly to move, according to the relationships defined by their assembly constraints. This technique is very useful in determining the suitability of assembly constraints placed on a component.

Drag a component after applying an assembly constraint to quickly gauge the effects of the constraint. Grounded components cannot be moved in this manner.

Careful planning and placement of assembly constraints is the key to obtaining proper assembly motion. Apply as many assembly constraints as needed to position, or in the case of an adaptive part, size your component. Temporarily suppress assembly constraints that interfere with assembly motion.

Constraint Drivers

Dragging a small component in a large assembly, or dragging a component about an axis of rotation can be difficult. Autodesk Inventor provides a unique tool to drive the value of an assembly constraint. You can specify movement range and step size, determine movement cycling, and set a pause time between steps. Mate and angle constraints between faces are common choices for driven constraints.

Assembly motion can be halted if interference is detected between components. Refine the increment value and drive the constraint to determine a precise constraint value where interference occurs. When interference is detected, the motion stops and the interfering components are highlighted in the browser and graphics window.

Adaptive parts can be resized to match the varying assembly constraint. Adaptive features and parts are presented earlier in this manual.

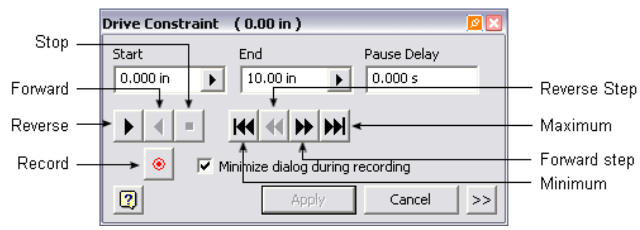
The motion can be recorded as an AVI file using any code available on your computer.

Drive Constraints

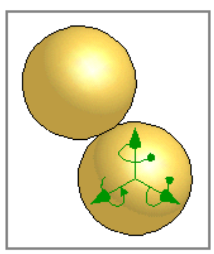
After you constrain a component, you can animate mechanical movement by changing the value of the constraint. The Drive Constraint tool repositions a part by stepping through a range of constraint values. You can rotate a component, for example, by driving an angular constraint from zero to 360 degrees. The Drive Constraint tool is limited to one constraint. You can drive additional constraints by using the Parameters tool to create algebraic relationships between constraints.

Drive constraint used to simulate a clock. Minute and hour hands are constrained to the dial. Drive constraint rotates the minute hand. Parameters tool defines the hour hand position as a function of minute hand position.

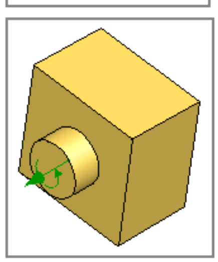
Use the Drive Constraint tool on the context menu to simulate mechanical motion by driving a constraint through a sequence of steps. Right-click the constraint in the browser and then enter information in the Drive Constraint dialog box to define the drive constraint and to control motion.



Constraints may limit the motion of parts. Depending on the geometry, degrees of freedom are removed or restricted. For example, if you apply a tangent constraint to two spheres, all six degrees of freedom remain, but you can't translate one of the spheres in just one direction.



Tangent constraint applied to two spheres. All six degrees of freedom remain, but they are restricted.



Tangent constraint applied to cylinder and hole. Cylinder and hole are the same size, so only two degrees of freedom remain.

Animating Assembly Components

Mechanical assemblies are rarely static. By animating the movement of constrained assemblies with Autodesk Inventor, you can examine your model throughout its range of motion. Use Inventor assembly animation to visually check for component interference and examine mechanism movement to improve your designs.

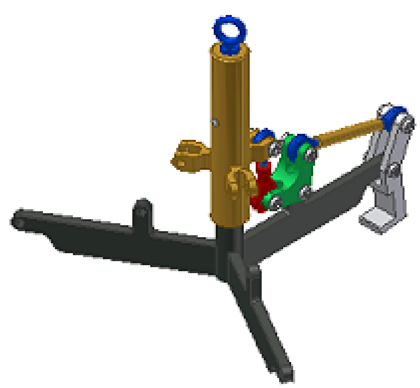
In this two-part exercise, you first constrain a component in a lift fixture assembly. You examine degrees of freedom as constraints are applied, and examine the motion of the assembly by dragging a strategic component in the graphics window.

In the second part of the exercise, you replace a simplified representation of a component in an assembly, define an angle constraint for a pivot, and then animate the assembly using the unique constraint driving capabilities of Inventor to check where component interference occurs.

You can view degrees of freedom for a part in the Properties dialog box available from the right-click menu in the browser. In the Properties dialog box, on the Occurrence tab, you can turn the Degrees of Freedom option on or off. The Degrees of Freedom option is also located in the View menu.

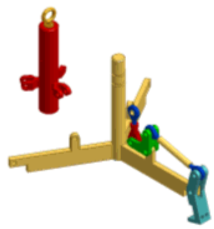
This exercise demonstrates how to properly constrain an assembly for motion analysis.

The completed exercises are shown in the following figure.

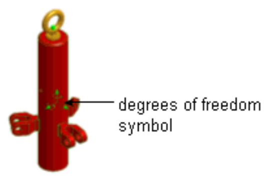


TRY IT: Remove a degrees of freedom constraint

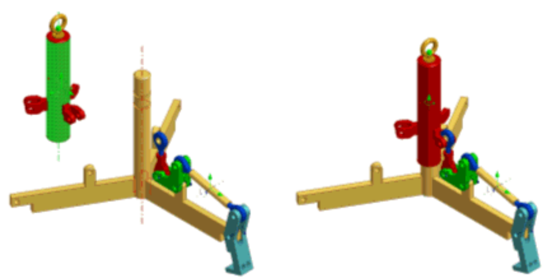
1 With the project *tutorial_files* active, open the file *remDOFs.iam*. The assembly should look like the following figure.



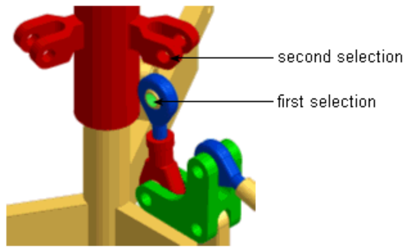
2 Click View ➤ Degrees of Freedom. The *NewSleeve.ipt* part is unconstrained, so all six degrees of freedom are available.



3 Click the Constraint tool in the panel bar or from the Assembly toolbar. Place a mate constraint between the major axis of *NewSleeve.ipt* and the axis through the cylinder feature of *NewSpyder.ipt*. This constraint removes two translational degrees of freedom and two rotational degrees of freedom from the sleeve.

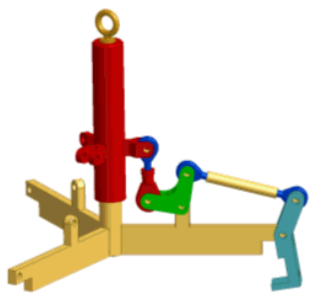


4 Remove the last rotational degree of freedom from the sleeve. Place a mate constraint between the axis (not a hole center) through the open bolt hole of the *NewAdjust_Link.iam*, and the axis through the bolt hole in one of the sleeve tabs. If required, zoom in or use the Select Other tool to select the axes.



5 The sleeve is now constrained to move only along the axis of the spider. Click View ➤ Degrees of Freedom to hide the DOF symbols.

6 Use the Rotate and Zoom tools to orient your view of the assembly as shown in the following figure.



7 Slowly drag the *NewLiftRing.ipt*. All components with constraints that are linked to the dragged component move in response, while honoring their own assembly constraints.

Close the file without saving or save the file with a new name to preserve the original data file.

Selecting Components

When working in an assembly, you often need to select a set of components for a common operation, such as turning off visibility or verifying which components are underconstrained. You may need to select components by size, location, relation to other components, or other criteria.

You can select components using one of several methods, and then apply one of several options, such as invert the selected order or revert to the previous selection set. You can isolate the selection set by turning off visibility of all components that are not selected.

Before you try the following exercises, open an assembly and click the Select button on the Standard toolbar, and then select the priority mode:

Part Priority	Selects parts or assemblies instead of features, faces or edges.
Component Priority	Selects only the first-level components of the edited assembly.
Feature priority	Selects features (including work features) on any part in the assembly.
Select Faces and Edges	Selects faces or individual curves that define faces on any part in the assembly.
Select Sketch Features	Selects sketch geometry used to create features or sketches or the individual curves that define sketches.
Select Visible Only	Includes only visible components in a selection set. Is applied to all selection methods.

The priority selection mode makes it easier to select the geometry you want to include in a selection set. Particularly in complex assemblies, narrowing the range of selections can help you select the correct object.

In addition to selecting a particular type of geometry, you can also base selections on criteria such as components constrained to a selected component, components of a particular size, or offset by a distance, or on a specified side of a plane.

In this example, you select components constrained to one or more components.

TRY IT: Select by constraints

1. In the graphics window or the browser, select one or more components.
2. On the Assembly Standard toolbar, click Select ➤ Constrained To. All components constrained to the preselected component are highlighted in the graphics window and in the browser.

After you isolate the selection set, you may notice that some components you expect to be included are not visible. This is a quick way to see which components are not constrained to the first-selected component.

In the next example, highlight components relative to the size of the selected component.

TRY IT: Select by component size

1. On the Assembly Standard toolbar, click Select ➤ Component Size.
2. If not preselected, use the Select tool in the Select by Size box to select a component.

   Selections are contained in a virtual box called a bounding box. Its size is determined by the outermost extremities of the selected component.
3. The size is shown, determined by the bounding box of the selected component. Click At Most or At Least to specify the relative size to select, and then click the green arrow.

   Selected components are highlighted in the graphics window and the browser.

   You can highlight components contained within a bounding box of the selected component, plus and offset distance.

TRY IT: Select by offset distance

1. On the Assembly Standard toolbar, click Select ➤ Component Offset.
2. If not preselected, use the Select tool in the Select by Offset box to select a component.

   Selections are contained in a virtual box called a bounding box, whose size is determined by the outermost extremities of the selected component.
3. The offset distance is shown, determined by the bounding box of the selected component. If desired, click a bounding box face and drag to resize. Select the check box to also include components partially contained in the bounding box, and then click the green arrow.

   Selected components are highlighted in the graphics window and the browser.

Try using some of the other selection methods, including:

- Select All Occurrences
- Sphere Offset
- Select by Plane
- External Components
- Internal Components

- All in Camera
- Visible Filter

Using Design Accelerator

11

In this chapter, you learn about Design Accelerator and how to work with generators and calculators.

What is Design Accelerator

Design Accelerator represents an important component of Functional Design, providing engineering calculation, and decision support to identify standard components or create standards-based geometry. Design Accelerator tools simplify the design process, by automating these selections and geometry creation, improving initial design quality by validating against design requirements, and increasing standardization by selecting the same components for the same tasks.

Design Accelerator provides a set of generators and calculators that can create mechanically correct components automatically by entering simple or detailed mechanical attributes. For example, you can use the bolted connection generator to insert a bolted connection at once by offering to select the right parts, select holes, and assemble components together.

To insert components using Design Accelerator generators and calculators, you must be within the assembly environment. Open the Design Accelerator panel bar. The available generators and calculators commands are displayed. Generators and calculators are grouped according to functional areas. For example, all welds are grouped. The panel bar displays the last calculator or generator used for the group.

NOTE Before you start working with any generator or calculator, your assembly must be saved.

To open generator or calculator, click the appropriate command. Generators are opened on the Design tab, calculators on the Calculation tab, by default. In the Design tab, you specify the placement options and select the type of components you want to insert. In the Calculation tab, you enter the calculation values. For Bolted Connection Generator and for some calculators, the Fatigue Calculation is also available to specify values for fatigue calculation.

NOTE Only one generator/calculator can be open at a time - when one generator/calculator is opened and you want to open a new one, the first one is automatically closed.

If you want to perform the calculation, click the Calculate button.

When you want to insert designed components and calculations to Autodesk Inventor assembly, click OK.

What operations can I perform within Design Accelerator?

When you open the Design Accelerator panel bar (click the arrow in the Assembly panel bar), a set of tools is displayed. They are divided into three groups:

- Component Generators
- Mechanical Calculators
- Engineer's Handbook

Work with Generators

Using Component generators you design desired components. You can also perform calculations. Some generators use Content Center components, for example Bolted Connection Generator, or Clevis Pin generator. Connect to the Content Center to insert components using such generators. Without being connected to Content Center, you can perform various calculations of the components.

You can use the following Design Accelerator generators:

- Bolted Connection
- Shaft

- Involute Splines
- Parallel Splines
- Key Connection
- Disc Cam
- Linear Cam
- Spur Gears
- Bevel Gears
- Worm Gears
- Bearing
- V-Belts
- Synchronous Belts
- Roller Chains
- Clevis Pin
- Joint Pin
- Radial Pin
- Secure Pin

Workflow overview: Insert components using Design Accelerator generators/calculators

1. On the Assembly panel bar, open the Design Accelerator panel bar, and click the appropriate generator or calculator.
2. On the Design tab, select the type of components you want to insert and complete the placement options.
3. On the Calculation tab, enter the calculation values.
4. Click Calculate to perform the calculation and verify results.
5. Click OK to insert component or calculations to the Autodesk® Inventor™ assembly.

Work with Bolted Connections

Use the Bolted Connection generator to design and check pretensed bolted connections loaded with axial or tangential force. The purpose of the design calculation is to select an appropriate bolted connection after specifying the required working load. The strength calculation performs a check of bolted connection (for example, pressure in the thread and bolt stress during joint tightening and operation).

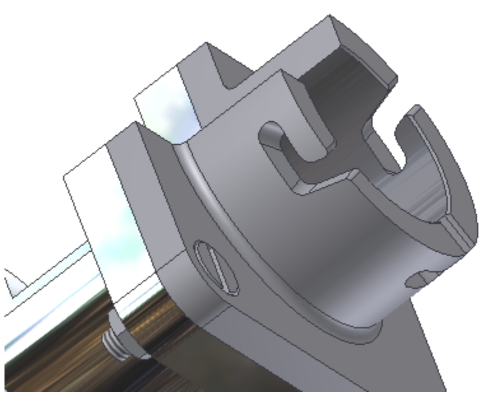
When you can create and design bolted connections, you can enter them as one group into Autodesk Inventor.

The generator supports two ways of designing bolted connections. The first uses an aperture you have already created in the Inventor model and, on the basis of the parameters of the aperture, selects the correct dimensions for the components in the bolted connection. The second method designs the whole connection, including the aperture which is then created during entry (or drilling) into the selected connection.

You can use either method to select fastener components directly from Content Center. You can display the standard components dialog box to add more information (for example, material values, information from external databases, or other company information).

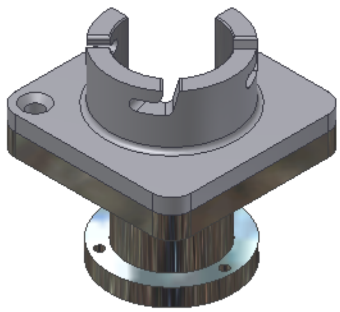
In this two-part exercise, you create a bolted connection. You select the bolted connection components and the bolted connection placement. In the second part of the exercise, you change the type of component of the already inserted bolted connection.

The completed exercises are shown in the following figure.

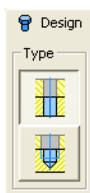


TRY IT: Insert a bolted connection

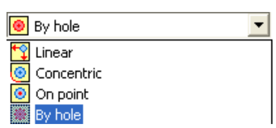
1. With the *tutorial_files* project active, open *Bolted_connection.iam*. The assembly should look like the following figure.



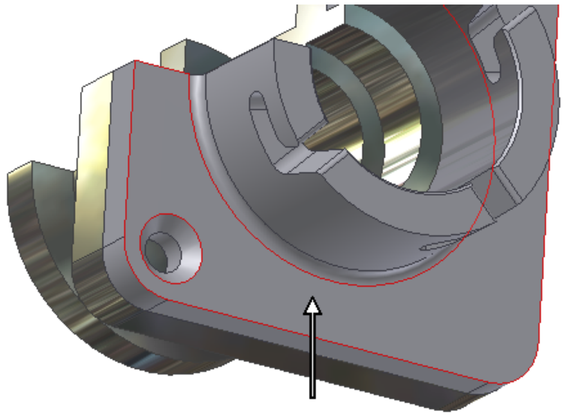
2. On the Assembly panel bar, click the Bolted Connection tool.
3. On the Design tab, Type section, select Through All.



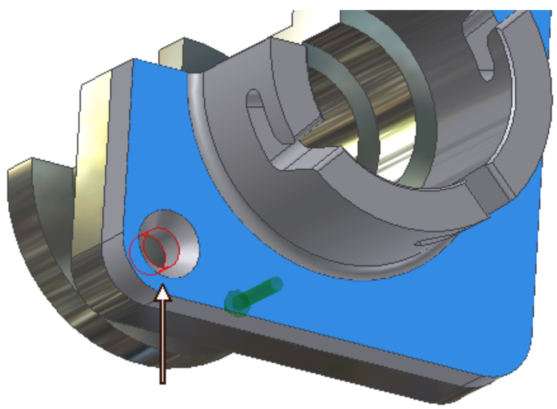
4. In our assembly, we have a hole where we want to insert the bolted connection. In the Placement selection list, select By hole.



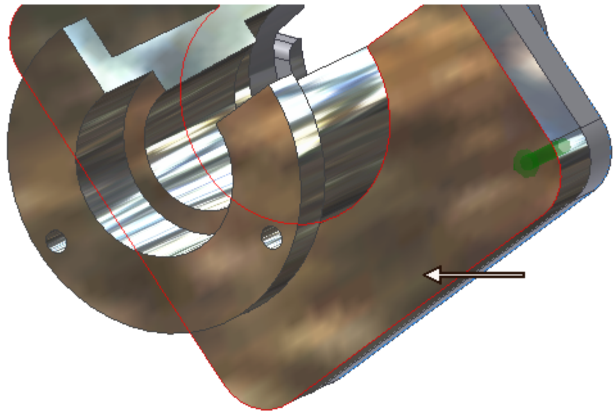
5. Click the Start Plane button. In the assembly, select the start plane as shown in the following figure.



6 Select Existing Hole. In the assembly, select the hole as shown in the following figure.



7 Select Termination. In the assembly, select the termination plane as shown in the following figure.



8 Specify the thread type and dimension as follows:

`As Thread:` ISO Metric profile

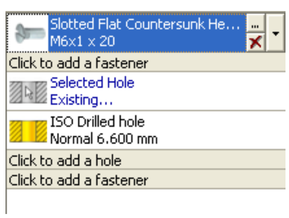
`Insert Diameter:` 6mm

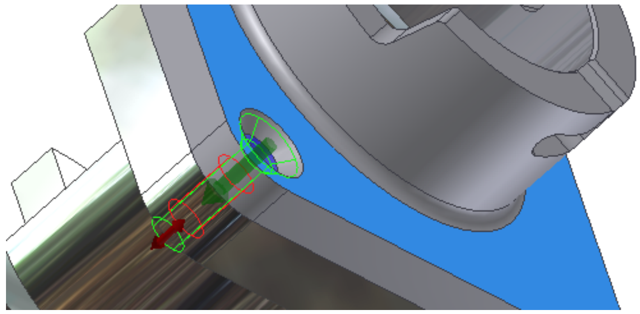
9 Begin populating the bolted connection. Select Click to add a fastener.

A filtered list of available fastener content (from Content Center) displays. It is based upon previously set Thread settings (Standard and Thread size). You can narrow the displayed list of fasteners by selecting a standard.

10 In the displayed list of bolts, select Slotted Flat Countersunk Hex bolt.

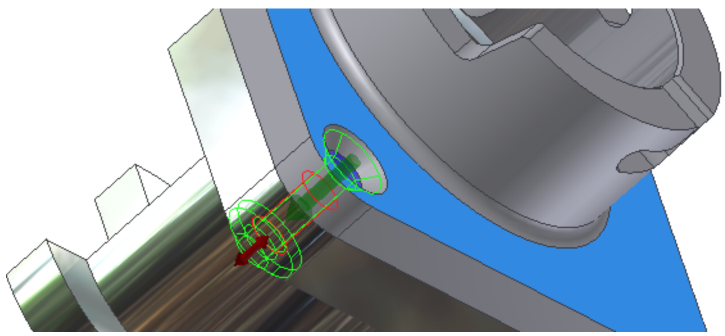
The selected bolt displays in the Design tab of the bolted connection generator. In the Inventor assembly, the bolt preview is created.





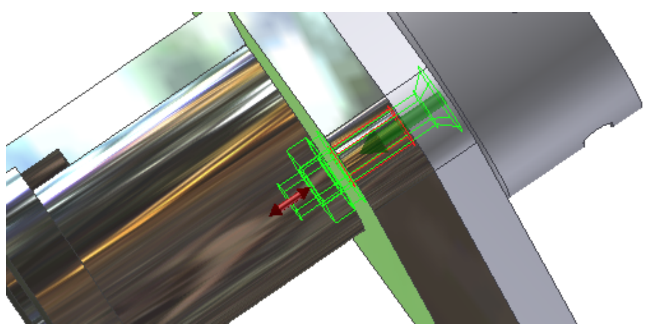
11 Select Click to add a fastener. In the washer selection list, select Plain washer (metric).

The selected washer displays in the Design tab of the bolted connection generator. In the Inventor assembly, the washer preview is created.

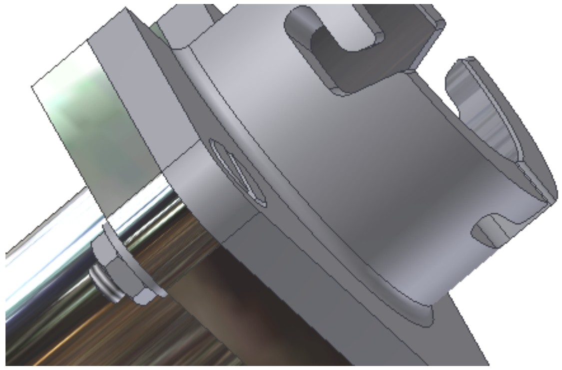


12 Select Click to add a fastener. In the nut selection list, select Hex jam Nut.

The selected nut is displayed in the Design tab of the bolted connection generator. In the Inventor assembly, the nut preview is created.

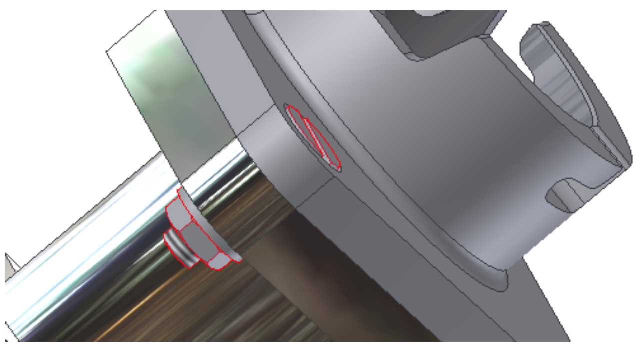


13 Click OK.



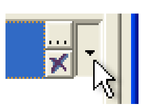
Now, when the bolted connection is inserted, it is easy to change any component with the bolted connection. We will change the type of washer.

14 In the file *Bolted_connection.iam* assembly, select the inserted bolted connection.

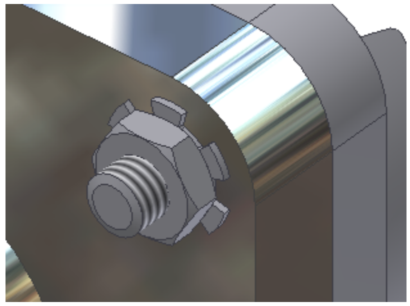


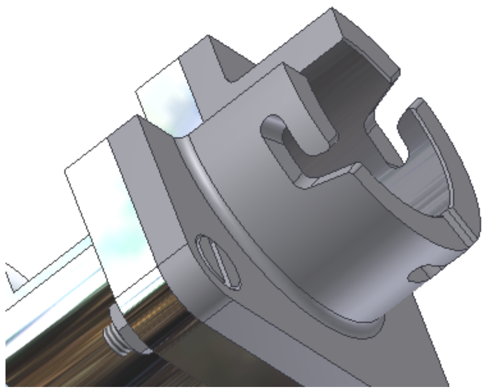
15 Right-click and select Edit Using Design Accelerator. The Bolted Connection generator displays.

16 Click the arrow at the end of the Plain washer (Metric) edit field to display the washer selection list.



17 In the selection list, select the External Tooth Lock Washer and click OK to update the bolted connection assembly.





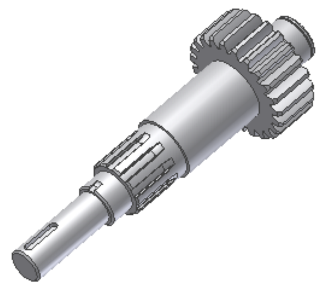
Insert All Components At Once

It is also possible to insert all types of components (component, feature, calculation) at one time.

In the following exercise, you insert key and shaft groove using the Key Connection generator.

TRY IT: Insert Key connection

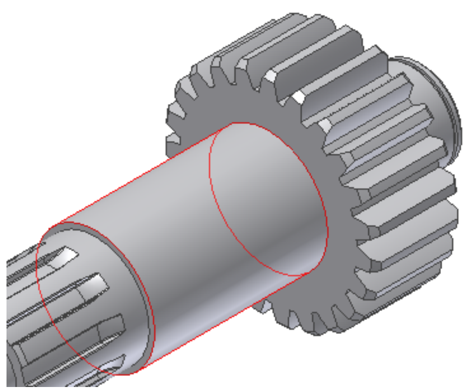
1 With the project *tutorial_files* active, open the file *bearing.iam*. The assembly should look like the following figure.



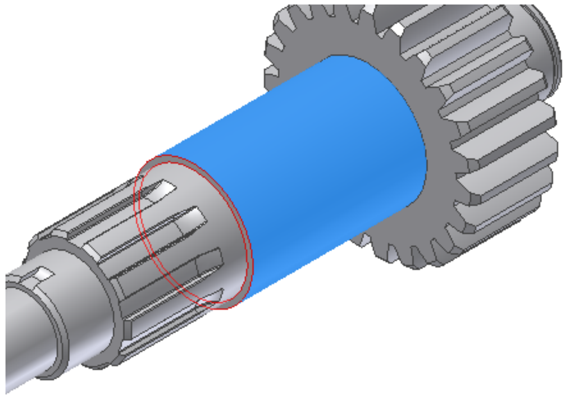
2 On the Design Accelerator Assembly panel bar, click the arrow next to Shaft item and select Key Connection.

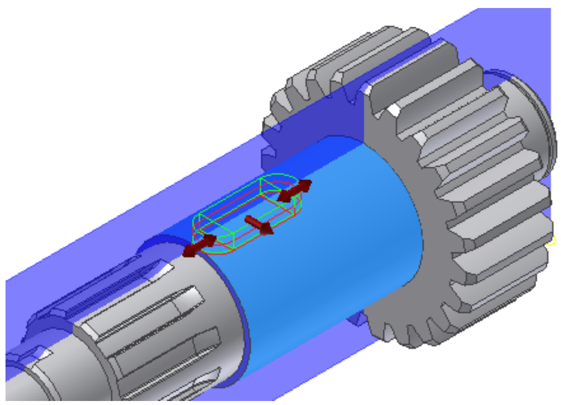
The Key Generator displays.

3 On the Design tab, first specify the placement of the shaft groove. Make sure that Create New is selected in the Shaft Groove selection list. Click the shaft element to select reference for shaft groove insertion as shown in the following figure.



4 Select the second reference as shown in the following figure. In the assembly, the shaft groove preview is presented. Click the grips in the preview to change the shaft groove length, position, and angle (rotation).



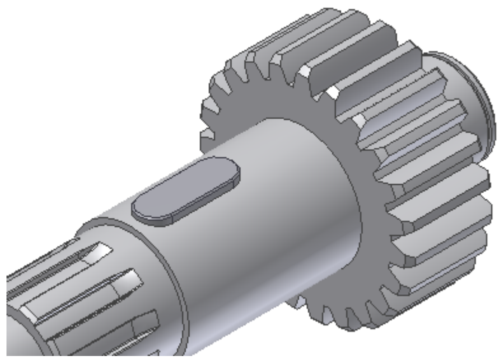


5 According to the selected shaft element, the shaft diameter value was inserted to the Key Generator and edit field became disabled. Edit the Key Length by either selecting the value from the selection list or use the preview grip in the assembly.

6 On the Design tab, Select objects to Generate, click the Hub Groove button to disable hub groove insertion.

Only the key and shaft groove icons are enabled to insert key and shaft groove.

7 Click OK.



Work with Calculators

Design Accelerator offers a set of tools to perform mechanical calculations of selected components. Engineering calculators use standard mechanical formulas and physical theories in design and validation of mechanical systems. You specify calculation criteria and calculators perform the calculation and displays report about a calculation. If the calculation doesn't indicate the calculation compliance, error message is displayed advising you what values need to be updated. Using Design Accelerator calculators you do not insert components into the Autodesk Inventor, only calculation is inserted when you click OK.

The following calculators are available:

- Plain Bearing
- Plug and Groove Weld
- Butt Weld
- Spot Weld
- Filled Weld (Connection Plane Load)
- Filled Weld (Spatial Load)
- Butt Solder Joint
- Bevel Solder Joint
- Lap Solder Joint

- Step Tube Solder Joint
- Step Solder Joint
- Separated Hub Joint
- Slotted Hub Joint
- Cone Joint
- Tolerance
- Limits and Fits
- Press Fit
- Power Screw
- Beam and Column
- Plate
- Shoe Drum Brake
- Disc Brake
- Cone Brake
- Band Drum Brake

TRY IT: Calculate separated hub joint

1. On the Design Accelerator panel bar, click Separated Hub Joint.
2. On the Calculation tab, enter calculation parameters.
3. Click Calculate to perform the calculation. Results are displayed in the Results area.
4. If the calculation results are satisfactory, click OK to insert separated hub joint calculation to the assembly.

Author User Parts

The Component Authoring tool enables the authoring of custom content for use with AutoDrop and content-based Design Accelerator generators. Guided step-by-step prompts prepare components with iMates and properties necessary for efficient use of functionally smart content ready for publishing to the Content Center Library.

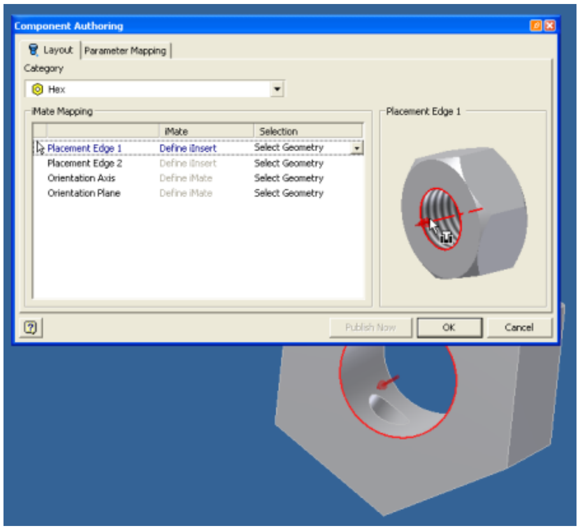
During the authoring of user parts, you specify:

- Content Category
- Part placement (iMates)
- Category parameters
- Parameter and iPart table mapping

You can author and publish all standard parts:

- Bearings
- Bolts
- Clevis Pins
- Cotter Pins
- Retaining Rings (Circlips)
- Keys
- Nuts
- Washers

Use the Component Authoring tool to prepare an iPart or a normal part for publishing to the Content Center Library. In the Component Authoring dialog box, items listed in red are required settings for publishing.



TRY IT: Author user parts

1. Open a custom iPart in Autodesk Inventor.
2. Click File ➤ Component Authoring.
3. Select the part category. The Category selection list displays the list of available publishing categories. Once you select a Category, the graphics and selection prompts change depending upon the Category of component selected.
4. Create the iMates following the tooltips and graphical guide for the specific component.
5. Move to the Parameter Mapping Tab where you map the parameters to the related column name in the iPart table. Map all required parameters in the list.
6. When authoring is successful, click OK.
7. Click the Publish Now button to start the Publish Part command.
8. Continue working with the Content Center Publish tool to publish parts to the Content Center.

NOTE Since you selected a category early on in the process, the Content Center displays this category for publishing. For example, a bolt can now be published directly into the Bolt category, an existing subcategory of Bolt, or you can create a category under Bolt.

NOTE You can rename the iMates as needed. These new names populate the iMate list in both the panel browser and the Component Authoring dialog under the iMate list.

Set File Names

The Design Accelerator Settings dialog box is available to change or redefine the Display name and the File name for Design Accelerator generators and calculators.

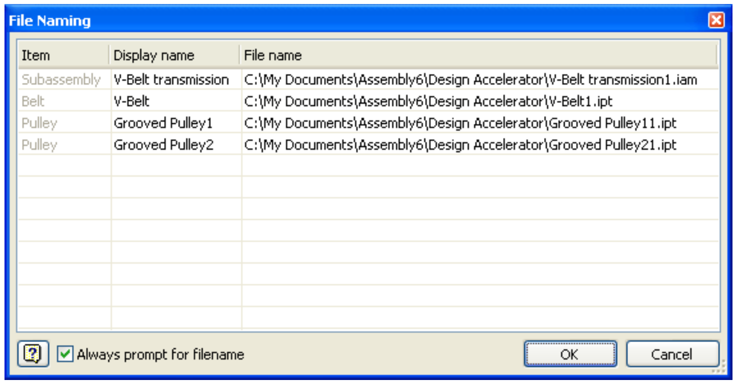
Use the File Naming dialog to specify the name of the connection and newly created components lets you change the filename and location on disk.

When you work in any generator or calculator, V-Belts Generator foe example, you click the File Naming icon in the right upper corner. The File Naming icon is available within all tabs (Design, Calculation, Fatigue Calculation).

Double-click the row to enter the Displayed name, V-Belt transmission1, for example. The name can contain letters, numbers, or any symbol.

To specify the file name location, click the button next to the selected row. The button is available only for one selected row at a time. In the selected folder, the component is stored.

When you select the Always prompt for filename option, you are prompted to specify file name and display name every time you insert a new DAcc component into the assembly. During edit, the dialog is displayed only if you add a new component, a new pulley using V-Belts Generator, for example.



NOTE In the File Naming dialog, you can only edit items in white edit fields.

TRY IT: Set file names

1 In the Design Accelerator Generators and Calculators, click the file Naming icon.

2 Double-click the Display name to insert the display name.

3 Click the button next to the edit filed to specify the folder where component is stored.

4 Click OK to confirm the settings and close the File Naming dialog box.

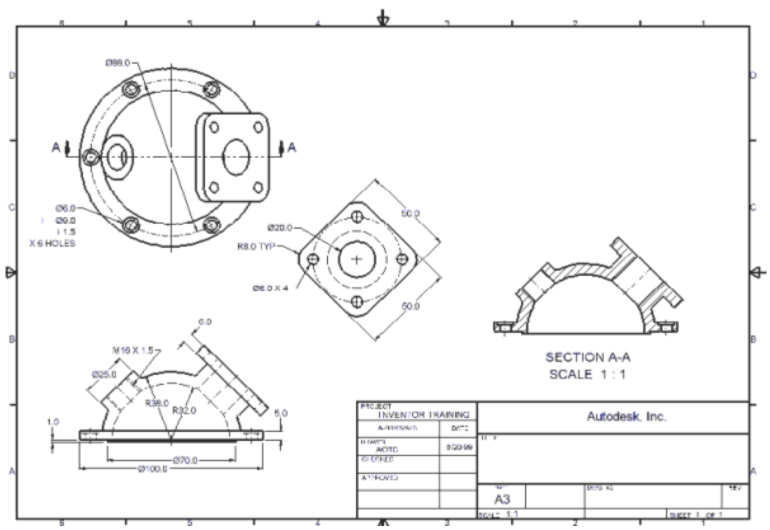
Setting Up Drawings

12

In this chapter, you learn about setting up drawings, using drawing styles, and using drawing resources such as sheet layouts, title blocks, and borders.

Creating Drawings

After you create a model, you can create a drawing to document your design. In a drawing, you place views of a model on one or more drawing sheets. Then you add dimensions and other drawing annotations to document the model.



You can change the alignment, label, line style, scale, and displayed dimensions in any view. You can also edit your part by changing the parametric model dimensions from within the drawing file, if when you installed Autodesk®

Inventor™. Set the option to allow drawing dimensions to resize the model. Similarly, your drawing file automatically updates with any changes saved in the model file.

Autodesk Inventor comes with standard templates to use as the starting point for your drawings. Template files have the standard drawing extension (*.idw*, *.dwg*). Autodesk Inventor stores template files in the *Autodesk\Inventor (version number)\Templates* folder. You can also create your own templates, specifying unique characteristics, and save them in the Templates folder.

NOTE When you select New Drawing from the drop-down menu next to the New button, Autodesk Inventor looks for a file named *Standard.idw* or *Standard.dwg* in the *Autodesk\Inventor (version number)\Templates* folder. The setting specified in the Default Drawing File Type option in the Drawing Tab in Application Options controls the default drawing type used (*.idw* or *.dwg*) when creating a drawing using the New Drawing button in the Standard toolbar.

You start with a drawing template when you create a drawing.

Workflow overview: Create a drawing

1. Click the New button on the Standard toolbar, and then choose a drawing template from the Default, English, or Metric tab.

   The default drafting standards are based on the settings you specified when you installed Autodesk Inventor. The default drawing is a blank sheet with a border and title block. The English and Metric tabs contain the templates for those units of measure.

2. On the Drawing Views panel bar, click Base View.

3. On the Drawing View dialog box, click the Browse button beside the File box to locate a part or assembly. If you already have a model open, it is used by default for the view.

4. Accept the default scale, label, and other settings. A preview of the view is attached to the cursor. Click a point on the drawing sheet to place the view and close the dialog box.

   If the view is not positioned as you would like it, click its dotted line boundary and drag to a new location.

Autodesk Inventor maintains links between components and drawings, so you can create a drawing at any time during the creation of a component. By default, the drawing updates automatically when you edit the component. However, it is a good idea to wait until a component design is nearly complete before you create a drawing. Edit the drawing details (to add or delete dimensions or views, or to change the locations of notes and balloons) to reflect the revisions.

Sometimes it is more efficient to create a quick 2D drawing using a sheet sketch or draft view than it is to design a solid model. With Autodesk Inventor, you can create 2D parametric drawing views, which you can also use as sketches for 3D modeling.

NOTE You can directly open AutoCAD® DWG (*.dwg*) files in Autodesk Inventor using the open command and then view, plot, and measure the file contents. The AutoCAD objects remain as AutoCAD objects in Autodesk Inventor, and display exactly as they do in AutoCAD. In addition, all the AutoCAD data is available for copy and paste. You can open an AutoCAD DWG file in Autodesk Inventor, and then copy and paste AutoCAD entities into any Autodesk Inventor sketch.

Edit Model Dimensions in Drawings

In addition to model changes updating the drawing, you can also revise parts and assemblies by changing model dimensions in a drawing. This two-way associativity helps ensure that documentation represents the latest version of a component.

NOTE When you installed Autodesk Inventor, you specified if you wanted drawing dimensions to update models. To change this setting, re-install Autodesk Inventor.

To view and edit model dimensions in a drawing, use the Retrieve Dimension command on the Drawing Annotations panel bar. Autodesk Inventor updates all instances of the part to reflect your changes.

Whenever you revise a part in the drawing environment, check any assemblies where the part is used to confirm there are no interferences.

Formatting Drawings with Styles

Use styles to format objects in Autodesk Inventor documents. Assembly and part modeling styles format color, lighting, and materials. Drawing styles format dimensions, hole tables, hole notes, text, parts lists, tables, balloons, and many other attributes associated with drawing annotations.

When you install Autodesk Inventor, you specify a drafting standard. Each drafting standard comes with a default set of styles that are enough to get you started. You can add and edit styles as needed.

All styles associated with a drafting standard are stored in a style library. You can customize the style library and link it to a project file (*.ipj*). All files included in the project then use the same styles for formatting.

If you use style libraries on projects, share styles among designers. Documents are uniformly formatted, and updates are easy. When you update the main style definition in the library, all documents that use the style library can update their formatting.

NOTE Usually, when you create or edit a style in a document, it remains in the document. If you want to include the style in the style library, click Format ➤ Save Styles to Style Library. When you save a style to the library, you replace the master definition of the style with the new version. Use caution because it can affect other documents that reference the style library and use the style for formatting.

Use Styles In Templates

Legacy documents were formatted using styles stored in template files. You can still use templates with styles and reference a style library as well. To avoid making your file size larger than necessary, to use Format ➤ Purge Styles to remove unused styles from the document. Only styles that are not in use can be purged, so you do not risk losing formatting you need.

Some styles reference other styles as substyles. One example is the dimension style, which uses the text style to format dimension text. If you change a text style, it affects a dimension style that references the text style. If you purge a style that references another style, confirm if you also want to purge the referenced style.

Styles and templates each have their uses and advantages:

- Templates are a good place to store information that stays the same, such as title blocks, borders, default views, sheet sizes, and so on.
- Style libraries are a good way to control formatting. If a style is stored in a template, it is only available to future documents created with the template, so previously created documents must be manually updated. With style libraries, a style definition is available in any document simply by refreshing the library.

Share Styles Between Documents

You can share styles between documents in two ways:

- Use Format ➤ Save Styles to Style Library to save a new or edited style to the style library. Then it is available for use in any document.
- Use the Styles Editor Import/Export tool to select one or more styles and export them at once. The same process is used to import styles.

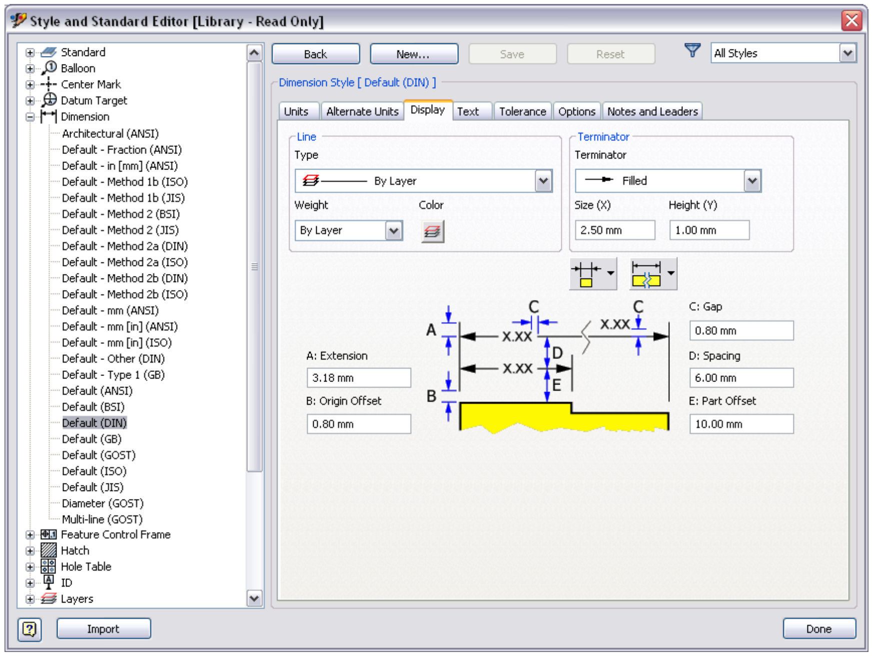
Use Styles Available In Drafting Standards

Each drafting standard has a complete set of styles. Before you create views of your model, review the available styles to become familiar with the formatting settings. If necessary, you can adjust values and save them to the style library.

All formatting for styles is controlled within the Styles and Standards Editor. Create or open a drawing document and then review the styles.

TRY IT: Examine the available styles for the drafting standard

1. With an *.idw* file open, click Format ➤ Style and Standard Editor.
2. On Style and Standard Editor, select All Styles from the Filter Styles dropdown in the top right corner.
3. On the left pane of the Style and Standard Editor dialog box, review the list of object included in the style.
4. Expand the Standard item. All available standard styles are displayed. Click a style name to display the Standard window.
5. On the Standard window, review the settings and values associated with the standard style.
6. On the left pane, expand the Dimension item and click a dimension style. Notice that dimension style settings are edited on seven tabs. Click the Display tab, for example, to see preferences for how dimensions are represented in the graphics window and on the drawing sheet.



NOTE Some styles are used on several tabs. For example, the Dimension Text tab specifies appearance of text used in dimensions. The formatting originates in the Text style, accessed in the browser pane. When a style references another style for some of its formatting, the referenced style is called a substyle.

Create Styles

You can create a style by modifying an existing style. The changed style is saved in the current document and is not available to other documents until it is saved to the style library.

TRY IT: Create a style in the current document

1. On the Style and Standard Editor dialog box, click a style you want to use as the basis for the new style. For example, click Leader in the browser pane and then select a leader style to display its attributes in the window.

2. Click New. On the New Style Name dialog box, accept the default "Copy of ...." name or give it a unique name.

Select the Add to Standard check box so that the style is listed among available styles for the standard. You can check later by clicking the Standard style and then clicking the Available Styles tab. Your new style is listed and its check box is selected.

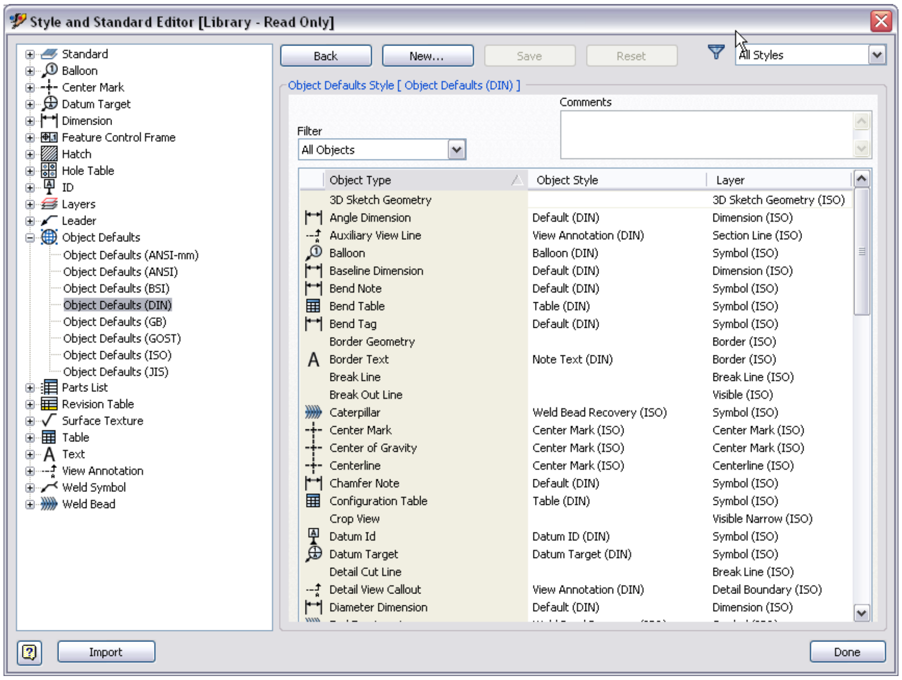
3. The new style name is listed in the browser pane under Leader. Select the name and change values as desired.
4. Click Save to save the new style in the current document, and then click Done.

Object Defaults Styles and Layers

The Object Defaults style assigns a style to all objects created in a drawing. It lists the object type, the style on which it is based, and its associated layer. The Object Defaults style is referenced in the Standard style.

Use the Object Defaults list to customize the default object style and the layer on which the object resides. For example, for balloons, you can change the default layer from Symbol to Dimension. Any previously created balloons that use the default layer are moved to the Dimension layer. New balloons are also placed to the Dimension layer.

Consider using layers as a grouping mechanism, like AutoCAD. Click the column headings to sort the columns. For example, all objects that use the Symbol layer are grouped, so you can see if you want to change any.



Using Drawing Resources

You can modify the drawing border and title block to comply with your company specifications.

NOTE Always save customized settings to drawing resources in the template. Otherwise, they are available only in the current document.

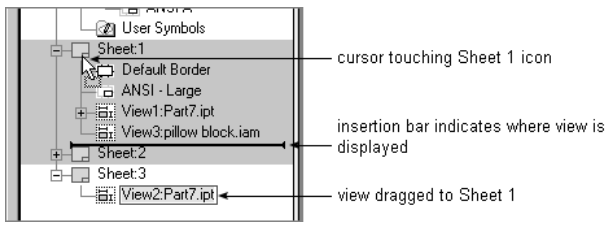
The first folder at the top of the browser is Drawing Resources. You can expand Drawing Resources to show the sheet formats, borders, title blocks, and sketched symbols that are available to use in the drawing. You can customize, add to, or delete items from Drawing Resources.

Sheet Layouts

When a new drawing is created, it automatically has at least one sheet. You can change the default sheet size to a standard or custom sheet size, and specify its orientation.

You can insert borders, title blocks, and views onto the sheet. Available borders and title blocks are listed in the Drawing Resources folder in the browser. Icons in the browser represent the sheet and all its component elements.

You can add multiple sheets to a drawing. Use the browser to move views between sheets. Only one sheet is active at a time. Inactive sheets are dimmed in the browser.

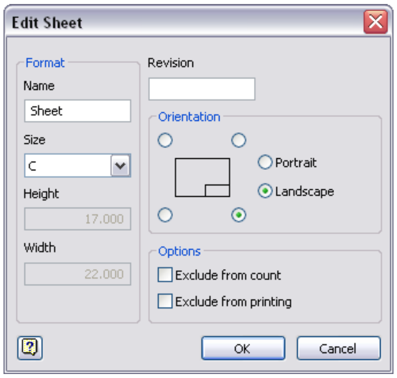


Edit Default Sheets

Edit the default sheet to modify the following information:

- Sheet name
- Sheet size
- Sheet revision
- Orientation
- Count attribute
- Print attribute

To edit the sheet attributes, right-click the sheet in the browser, and then select Edit Sheet. The Edit Sheet dialog box is displayed.



Format Sheets

You can create a sheet with a predefined layout of border, title block, and views by using a sheet format from Drawing Resources ➤ Sheet Formats. Right-click the sheet resource, and then select New Sheet. The format corresponds to a standard sheet size with an appropriate title block and border.

If the format you choose contains one or more views, the Select Component dialog box is displayed when you create a sheet. Use the Browse button to specify the model to document. Default views of the model are then created automatically.

Sketch Overlays

You can create a sketch overlay sheet to add graphics or text to your drawing without affecting drawing views. You can redline a drawing, for example, by working on the sketch overlay.

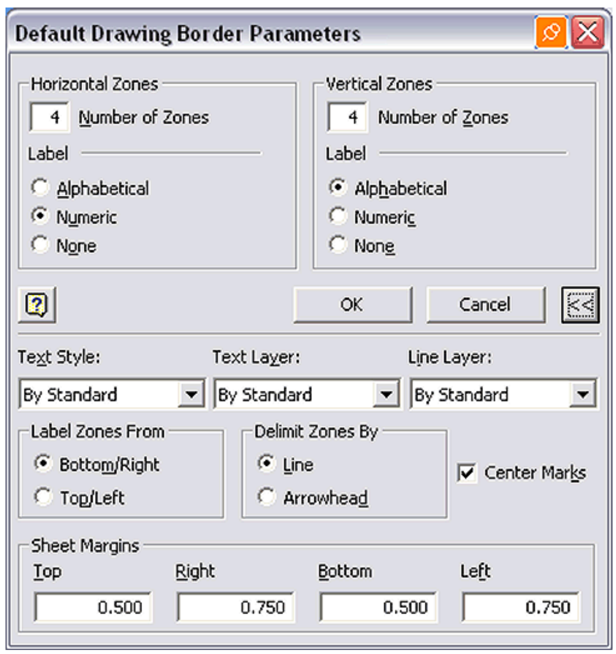
Drawing Borders

The Default Border is parametric. It automatically adjusts its size and labeling if the sheet is resized. When you insert a border, the Default Drawing Border Parameters dialog box is displayed. The default setting for the number of horizontal and vertical zones depends on the current sheet size.

Click the More button to modify the text, zone layout, and sheet margins.

You can create and save custom borders in the current drawing. Unlike the Default Border, custom borders are not parametric and do not resize when a sheet is resized. Once a custom zone border is inserted, right-click the border and select Edit Definition or Edit Instance. Make changes and save it according to the option selected (to the instance or the definition). If it exists within the Drawing Resources folder under Borders, you can right-click and select Edit.

To insert a border, expand Borders in the browser, right-click the border you need, and select Insert Drawing Border. If you select the Default Border, the Default Drawing Border Parameters dialog box is displayed.



To insert a custom border, expand Borders in the browser and then click Define New Border. Use tools on the Sketch panel bar to create the border, and then right-click in the sketch window and select Save Border. Enter a name for the new border and then click Save. You cannot save a custom border in a template.

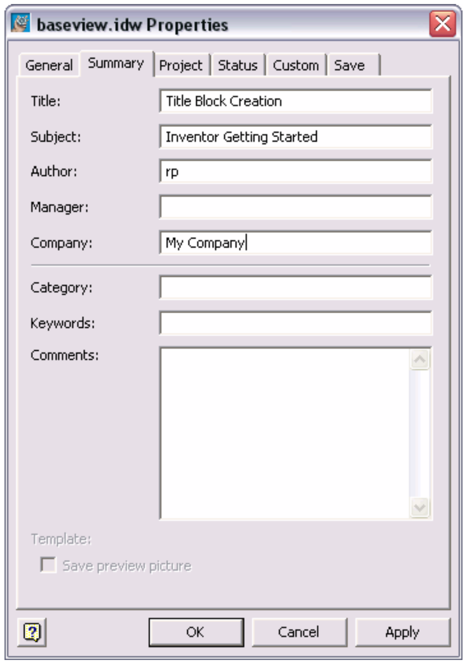
Title Blocks

The title blocks in an Autodesk Inventor drawing reflect information about the drawing, the sheet, and the design properties. As this information changes, the title block is automatically updated to display the current information.

Examples of the information that can be displayed in a title block include:

- Title
- Author
- Part number
- Creation date
- Revision number
- Sheet size
- Number of sheets
- Approved by

The information in a title block is referred to as a property field. Use the Drawing Properties dialog box to enter most of the information for your title block.



The standard drawing templates contain title block formats that you can customize and use. You can also create your own title block formats.

Workflow overview: Define a new title block

1. With an *.idw* file open, click Format ➤ Define New Title Block. The current sheet becomes an active sketch plane, and the Drawing Sketch panel bar is activated.
2. Use the tools on the Drawing Sketch panel bar to draw the title block. Define and use a grid to accurately sketch the lines for the title block.
3. On the Sketch toolbar, click the Text tool and then click in the title box to indicate the insertion point.
4. On the Format Text dialog box, click Type, and then select a property type from the list. Click Property, and then select a specific property. Specify other setting as needed, and then click OK.
5. Right-click in the graphics window, and then click Save Title Block. Enter the name for the new title block, and click save.

NOTE The new title block is added to the Drawing Resources folder in the drawing browser.

Align Title Blocks

Position a title block in any of the four corners of your drawing sheet. You can set the default position for title blocks using the Title Block insertion control in the Drawing tab of the Options dialog box. To access the Options dialog box, click Tools ➤ Application Options.

Autodesk Inventor uses that point to position the title block in the specified corner of the sheet.

You can also edit the position of the title block on any drawing sheet in the Edit Sheet dialog box. In the browser, right-click the sheet you want to modify and select Edit Sheet.

Edit Title Blocks

You can edit a title block and save the changes in the drawing. All sheets in the drawing using that title block is updated. When you select a title block for editing, the Drawing Sketch panel bar is activated, and you can add or modify geometry and text fields.

The tools for sketching in the drawing environment are the same tools used for sketching part profiles. Tools to place text fields are unique to the drawing environment.

Select these tools from the Drawing Sketch panel bar.

Tips for Creating Drawings

- Use a style library to ensure uniform formatting in drawing documents. Use either the default style library provided with each drawing standard or customize as needed. Save needed custom styles in the style library so that all documents can access them.
- Purge unused styles from legacy templates to avoid increasing your file size.
  You cannot purge styles that are in use.

- Use drawing formats with predefined views.
  To make sheet formats available to new drawings, create them in a template file that you use to create new drawings. Define a sheet for each sheet type you use.
- Use the Select filters.
  In addition to the Edge, Feature, and Part filters, you can specify various drawing elements for the Select filter.
- Drawing formats override units of measure.
  If components in an assembly have different units, the drawing format overrides them. The model dimensions have consistent units in the drawing environment.
- Use templates to maintain drawing standards and ensure consistent title blocks and borders among drawings. Add other information in templates that does not change, such as company logos.
- Use the default border to ensure that the border resizes if the sheet size changes.
- Use formats to save the sheet size, title block, border, and views on the sheet for quick layouts.
  You cannot save section, auxiliary, or detail views in a format.
- Select multiple objects within a drawing or within a drawing view using a selection or containing window.
- Move a drawing view by clicking and dragging the border.

Creating Drawing Views 13

In this chapter, you learn about the types of drawing views you can create using Autodesk ® Inventor™.

Drawing Views

Drawing views are referenced from, and associative with, external assembly or part files. You can produce multiview drawings of principal orthographic views and auxiliary, detail, section, and isometric views. You can also create views from assembly representations such as design views, positional, and level of detail, and presentation views. Autodesk Inventor calculates and displays hidden lines as required.

The first view in any drawing is a base view. This view is the source for subsequent views, such as projected and auxiliary views. A base view sets the scale for dependent views, except detail views. A base view also sets the display style for dependent projected orthographic views.

For a part model, the first view is usually a standard view such as a front or right-side view.

NOTE A draft view is a special view in a drawing that does not contain a representation of a 3D model. A draft view has one or more associated sketches. You can place a draft view and construct a drawing without an associated model. You can also use a draft view to provide detail that is missing in a model.

Drawing View Types

With Autodesk Inventor, you create and manipulate a variety of views using tools on the Drawing Views panel bar. Click the Base View button on the

Drawing View toolbar to create a base view and set the options on the Drawing View dialog box. Use the base view to create a projected, auxiliary, overlay, section, and detail views.

You can also create an isometric view using the projected view tool. When placing a projected view, move the preview to change the orientation of the projected view to an isometric view.

The following types of drawing views are available:

projected view	Projects from the base view to a desired location. The orientation of the projected view determines the relationship of the projected view to the base view. Use this tool to create an isometric view.
auxiliary view	Projects from an edge or line in a base view. The resulting view is aligned with its base view.
section view	Creates a full, half, offset, or aligned section view from a base, projected, auxiliary, detail, or broken view. Creates a view projection line for an auxiliary or partial view. A section view is aligned with its parent view.
detail view	Creates and places a detailed drawing view of a specified portion of a base, projected, auxiliary, break-out, or broken view. The view is created without an alignment to the base view.
overlay view	Overlays use positional representations to show an assembly in multiple positions in a single view. Each overlay can reference a design view representation independent of the parent view.
draft view	Creates a blank view with the sketch environment activated for drafting. You can import AutoCAD® data into a draft view, and you can copy a draft view and paste it into the same or another drawing.

Use the following operations to change drawing views:

break	Create a break in a view if the component view exceeds the length of the drawing, or contains large areas of nondescript geometry, like the center portion of a shaft.
break out	Removes a defined area of material to expose obscured parts or features in an existing drawing view. The parent view must be associated to a sketch that contains the profile defining the break out boundary.

crop	Provides control over the view boundary in an existing drawing view. Set the type of boundary (rectangular or circular) and specify crop boundaries.
slice	Produces a zero-depth section from an existing drawing view. The Slice operation is performed in a selected target view.

Base Views

The first view in a new drawing is a base view. Use the Base View button on the Drawing Views panel bar to create additional base views as needed.

Projected Views

After you create a base view, you can create projected views with a first-angle or third-angle projection, depending on the active drafting standard. Projected views can be orthographic or isometric. You can create multiple views with a single activation of the tool.

Orthographic projections are aligned to the base view and inherit scale and display settings. If the base view is moved, view alignment is maintained. If the scale of the base view is modified, the scale of the projected view changes.

NOTE Isometric projections are not aligned to the base view. They default to the same scale as the base view, but do not update if you change the scale of the base view.

The position of the cursor relative to the base view determines the orientation of the projected view. A view preview is displayed as you move the cursor. Click a point on the sheet to place the view. You can continue to place views until you right-click, and then select Create.

Editing Views

After you create a view, you can edit it. If the view is a base view, changes to the view parameters are reflected in the dependent views. You can remove the association between dependent views and base views by editing the dependent view. Then set independent scale, style, and alignment for the derived views.

To edit view parameters, select the view, right-click, and then select Edit View to open the Drawing View dialog box.

Creating Multiview Drawings

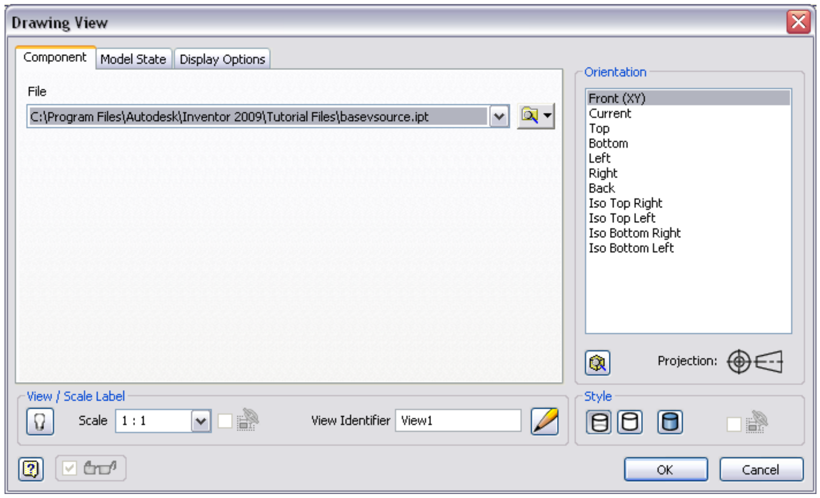
A multiview drawing contains a set of single plane orthographic views which are used to display an object through one view plane per projection. For example, a first angle projection is one view in a multiview projection set.

Base Views

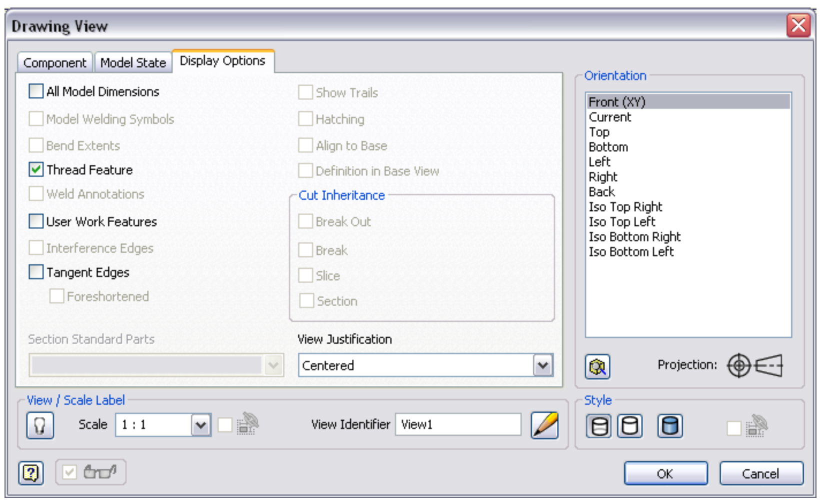
In this exercise, you create a base view, and then project views to create a multiview orthographic drawing. Finally, you add an isometric view to the drawing.

TRY IT: Create a base view

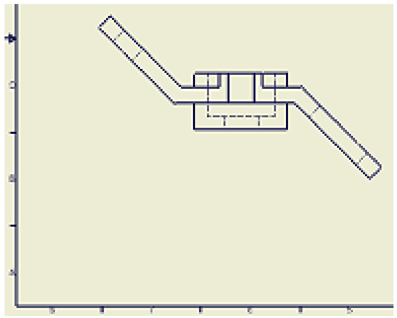
1. With the project *tutorial_files* active, open the file *baseview-2.idw*. The drawing file contains a single sheet with a border and title block.
2. Click the Base View tool on the Drawing Views panel bar. The Drawing View dialog box is displayed.
3. Click the Browse button, and then double-click *basevsource.ipt* to use it as the view source.
4. Verify that Front is selected in the View list. Set the Scale to 1.



5 Click the Display Options tab, and then verify that All Model Dimensions is not selected.



6 Position the view preview in the lower left corner of the sheet, in Zone B7. Click the sheet to place the view.

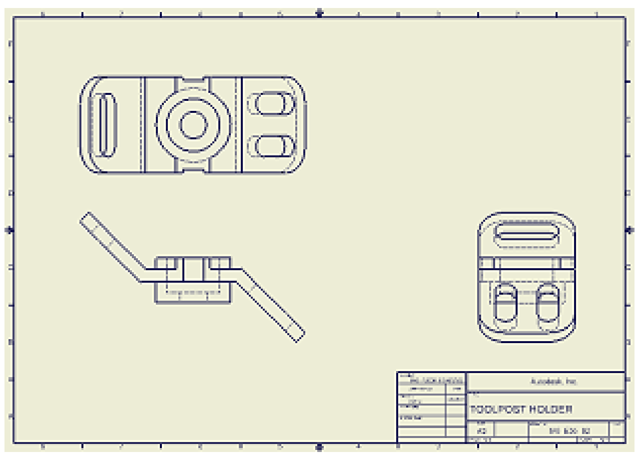


7 Click the Projected Views tool in the Drawing Views Panel.

   Click the base view and move the cursor vertically to a point above the base view. Click the sheet in Zone E6 to place the top view.

8 Move the cursor to the right of the base view. Click the sheet in Zone C2 to place the right-side view.

9 Right-click, and then select Create from the context menu.



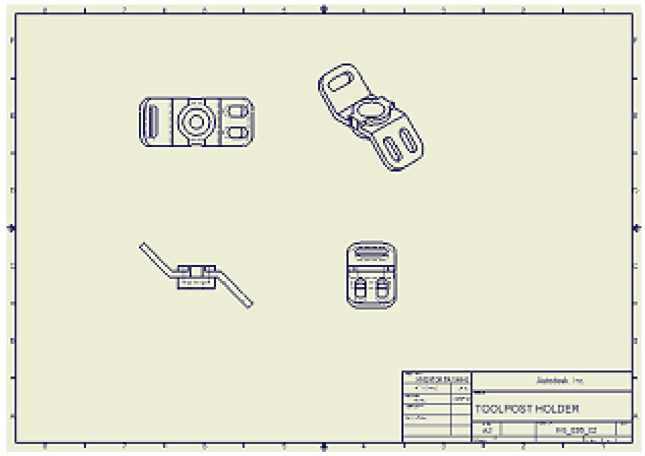
   Now create an isometric view.

10 Click the Projected View tool in the panel bar or from the Drawing Views panel bar.

   Click the base view and move the cursor above the right-side view. Click the sheet in Zone E3 to place the isometric view.

11 Right-click the sheet, and then choose Create.

   Your drawing should look like the following illustration.



Section Views

Autodesk Inventor can create a full, half, offset, or removed section view from a base view. The crosshatching, section line, and labels are placed automatically.

You can also use the Section Views tool to create a view projection line for an auxiliary or partial view. By default, a section view is aligned to its base view.

Press and hold CTRL as you position the section view to place it without alignment.

The section line arrowheads on the base view automatically orient to reflect the position of the section view relative to the base view. You can reverse the direction by dragging the view, or by editing it later.

You can also display multi segmented section lines by clicking the section line and disabling the Show Entire Line option in the right-click context menu.

Edit section view labels at any time by right-clicking the section view and choosing Edit View.

Defining Section Views

Use the Section Views tool to define a projection line for a section view. You can hover the cursor over view geometry to infer the position or orientation of the cutting line. The cutting line can consist of a single straight segment or multiple segments.

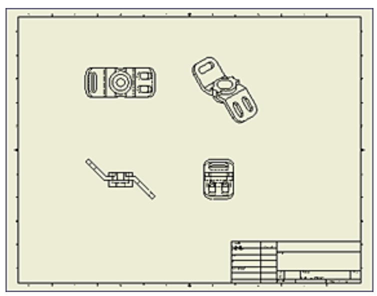
When you have defined the view projection line, the Section View dialog box is displayed.

NOTE You can use the CTRL key to prevent constraining the view projection line.

In this exercise, you create section, detail, and auxiliary views.

TRY IT: Create a section view

1 With the project *tutorial_files* active, open the file *sectionview.idw*. The drawing contains orthographic views and an isometric view.

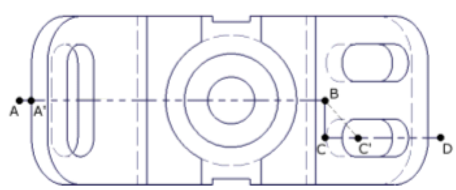


2 Click the Zoom Window tool on the Standard toolbar, and then create a window around the top view.

3 Click the Section View tool in the Drawing Views panel.

4 Click inside the top view.

Place the cursor over the midpoint of the left edge of the part (A'). Move the cursor to extend the projection line away from the part (A), and then click to place the start point of the section line.



5 Drag horizontally past the center of the part (B), and then click to define the first segment of the section line.

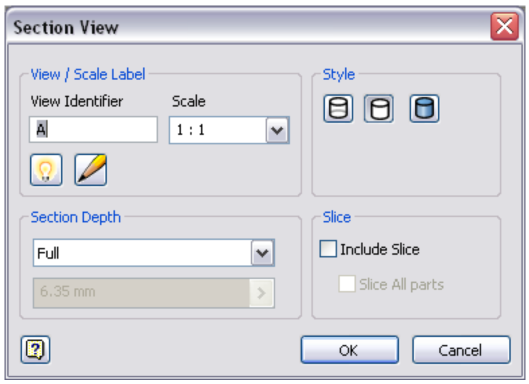
6 Drag the cursor to create an inferred constraint along the slot center (C').

Next, drag horizontally until a perpendicular constraint appears (C), and then click to define the second segment of the section line.

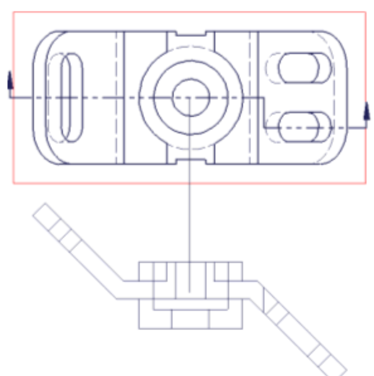
7 Drag horizontally to the right of the part (D), and then click to create the last segment of the section line.

Right-click, and then select Continue.

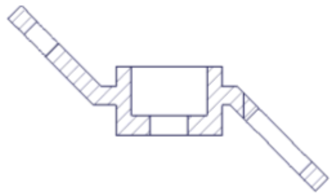
The projection line is defined, and the Section View dialog box is displayed.



8 Zoom out. Drag the section preview down to Zone D6, and then click to place the view.



9 The section view is placed in the drawing.



NOTE Press F5 to return to the previous view after zooming in to place the cutting plane.

Auxiliary Views

With Autodesk Inventor, you can create and place a full auxiliary view of a selected view. The auxiliary view is projected from and aligned with a selected edge or line in the base view. The selected edge or line in the base view defines the projection direction.

Auxiliary views are labeled, and display a projection line to the base view.

Use the Auxiliary View tool to create views aligned to non orthogonal geometry in a selected view. When you select the base view, the Auxiliary View dialog box is displayed. You can set the view Label, Scale, and Display options.

NOTE To create a partial auxiliary view, select the objects to remove from the auxiliary view, right-click, and then set Visibility off. Or, use the Section View tool to place a projection line that excludes the geometry you do not want in the view.

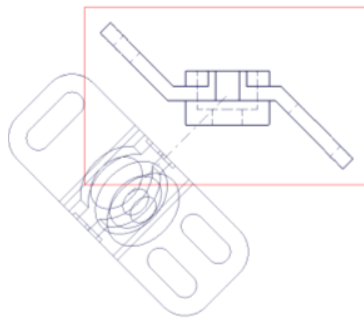
An auxiliary view is required to document the features on the inclined face.

TRY IT: Create an auxiliary view

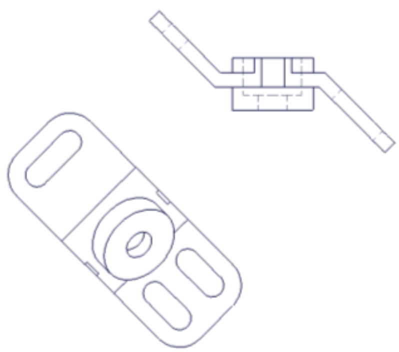
1 Click the front view.

2 Click the Auxiliary View tool on the Drawing Views panel bar to open the Auxiliary View dialog box.

3 Select the edge that defines the auxiliary projection.



4 Move the preview down and to the left. Click the sheet in Zone B7 to place the auxiliary view.



Detail Views

With Autodesk Inventor, you can create and place a detail view of a specified area of a drawing view. A detail view is created without alignment to its parent view.

By default, the scale of the detail view is double the scale of the parent view, but you can specify any scale.

Autodesk Inventor labels the detail view and the area it is derived from on its parent view. Either a circular or rectangular fence can be set for the detail.

Use the Detail View tool to define a detailed view of a selected area of any view. Specify the area to detail, and then drag the detail view to any location. When you select the parent view, the Detail View dialog box is displayed. You can set the detail label, scale, and view display options.

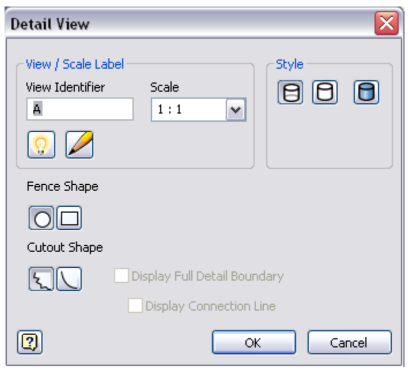
The center point of the fence positions the detail, and the fence determines the extent of the viewed detail. Right-click to select fence shape, click the center point of the detail, and then click a point to set the fence for the detail.

Next, you create a detail view to show a portion of the parent view at an enlarged scale.

TRY IT: Create a detail view

1 Zoom in on the top view.

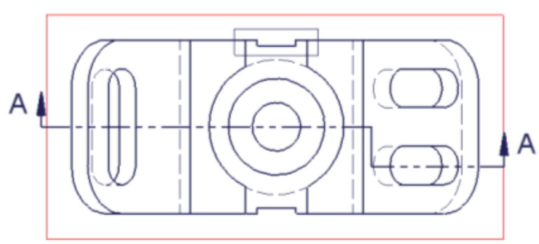
2 Click the Detail View tool in the panel bar or from the Drawing Management toolbar. Select the top view to open the Detail View dialog box.



3 Select Rectangular Fence Shape.

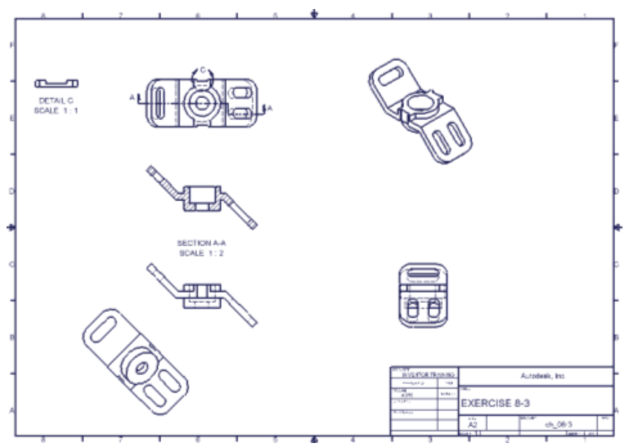
4 Select Smooth Cutout Shape.

5 Click in the top view to set the center of the detail and then drag to size the fence.



6 Zoom out enough to drag the preview to the left of the top view, and then click.

The view is placed. If necessary, click the view boundary and adjust its position.



Close the file without saving or save the file with a new name to preserve the original data file.

Break Views

You can create breaks in existing base, projected, section, detail, and auxiliary views. You select the existing view, define the appearance of the break, and then specify the location of the break lines in the view. The broken view retains the scale of the original view.

Use the Break tool to modify a view of a long component that cannot be scaled to fit the drawing sheet without obscuring important details. Select the view, and then place the break lines to specify the portion of the view to remove.

When you select the view to break, the Break dialog box is displayed. You can set the break style, orientation, gap, symbol size, and number of symbols displayed in the break lines.

Select the Rectangular or Structural style to define the general appearance of the break lines in your view. Then use the Orientation controls to specify the direction of the break lines. Use the Gap control to set the distance between the remaining segments of the view after it was broken. Adjust the value in the Symbols field to control the number of break symbols displayed each break line. You can set the symbol size in proportion to the gap size by using the slider control. Finally, click in the drawing view to specify the location of each break line.

Draft Views

A draft view is a special view in a drawing that does not require a representation of a 3D model. A draft view has one or more associated sketches. You can place a draft view and construct a drawing without an associated model. You can also use a draft view to provide detail that is missing in a model.

When you import an AutoCAD file to an Autodesk Inventor drawing, the data is placed in a draft view. Dimensions, text, and other annotations are placed on the drawing sheet and geometry is placed in the associated sketch.

Modifying Views and Sections

You can constrain the relative positions of two views. One of the views acts as the base view. If the base view moves, the aligned view also moves. If a view is aligned vertically, the position of that view relative to the base view is constrained to points along the *Y* axis of the base view. Horizontal alignment constrains the position to points along the *X* axis of the base view.

An in-position alignment establishes the relative angular position between the view and the base view. An auxiliary view is an example of in-position alignment. The view is constrained to maintain the alignment as the base view is repositioned.

To remove the constraints between views, select the view from which to remove the alignment. Right-click, and then select Alignment ➤ Break.

You can restore broken alignments to views. Select the view to align, right-click, and then select Alignment ➤ Horizontal, Vertical, or In Position. Click the base view to set the alignment.

NOTE To place a section view without an alignment constraint, press and hold the CTRL key as you place the view.

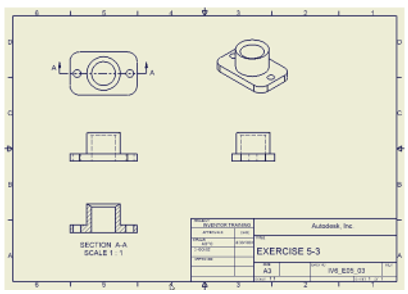
Delete Views

You can delete views that are no longer necessary. If you delete a base view, dependent projected and auxiliary views can either be deleted or retained. Section and detail views require a base view and cannot be retained.

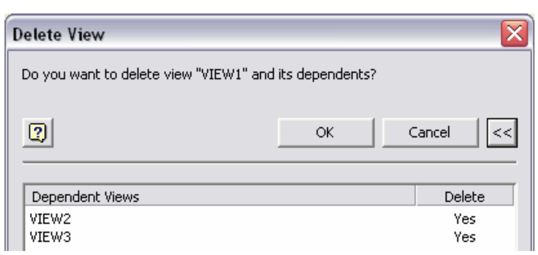
To delete a view, select the view, right-click, and then select Delete. In the Delete View dialog box, click the More button (>>) to select the dependent views to retain.

TRY IT: Delete a base view

1. With the project *tutorial_files* active, open the file *delbasev.idw*. The drawing contains three orthographic views, an isometric view, and a section view.



2. In the browser, right-click View1: view1-4.ipt, and then choose Delete to show the Delete View dialog box.
3. Click the More button to expand the dialog box. In the Dependent Views box, highlight View2, and then click Yes in the Delete column to switch it to No.
4. Repeat for View3.
5. Click OK to delete the base view and retain the two dependent views.



Align Views

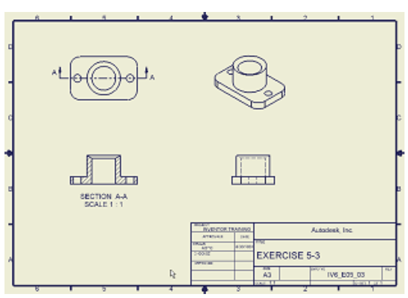
Alignment is the constraint relationship between a dependent view and its parent view. An aligned view can be moved only within its constraints. If the parent view is moved, the aligned view moves to maintain its alignment.

Most dependent views are created with an alignment, but you can add, change, or remove alignment relationships. There are four possible alignment relationships between a dependent view and its parent view: Vertical, Horizontal, In Position and Break.

TRY IT: Align views

1. Select the projected view in zone B2, right-click, and then select Alignment ➤ Break.
2. Select this view again, right-click, and then select Alignment ➤ Horizontal.
3. Select the section view as the base view.
4. Select the section view, and then drag the view vertically to the location previously occupied by the front view.

   The right-side view remains aligned to the section view. A view direction indicator is added to the original, projecting base view, and a view label is added to the projected view. It ensures that an indication of the projected view orientation and view direction is retained, regardless of where you move the view.
5. Right-click the isometric view and select Alignment ➤ In Position.
6. Select the section view as the base view.
7. Move the section view, and notice that the isometric view now moves with the section view.



Edit Hatch Patterns

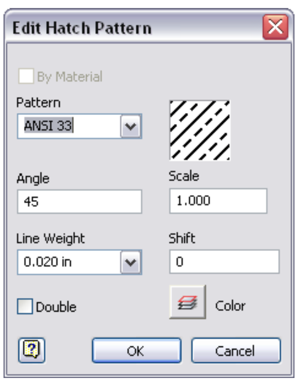
You can apply double-hatching, and you can modify the following aspects of a section view hatch pattern:

- Pattern
- Angle
- Line weight
- Scale
- Shift

In the following steps, you edit the section view hatch pattern to represent the material as bronze using the ANSI 33 hatch pattern.

TRY IT: Modify a hatch pattern

1. Right-click the hatch pattern in the section view, and then choose Modify Hatch. The Modify Hatch Pattern dialog box is displayed.
2. Select ANSI 33 from the Pattern list, and then click OK.



Close the file without saving or save the file with a new name to preserve the original data file.

Rotate Views

You can rotate views by edge or by angle. Views rotate as rigid bodies, including any sketches. When a view is rotated, annotations maintain their associativity to the view and model geometry. Depending upon the drawing standard used, additional information may be provided in the View label indicating that the view is rotated out of its normal position.

NOTE When you rotate a view, if a section view cutting plane line is not updated, you can edit the section line as you would edit a sketch, including constraints.

Move Views

You can move a view by clicking and dragging the red border. You can move multiple views with a crossing selection window. Specify a crossing selection window by clicking, dragging your mouse from right to left, and then clicking again. Views that are fully or partly within its borders are selected. To move the selected views, click and drag one of the red borders.

Viewing Multiple Positions of Assemblies

Overlay drawing views use positional representations to show an assembly in multiple positions in a single view. Overlays are available for unbroken base, projected, and auxiliary views. Each overlay can reference a design view representation independent of the parent view.

In the drawing browser, overlays are shown as child nodes to the parent view, displayed as "PosRepName: ViewNumber: ModelName." Right-click an overlay node to open the model file using the positional representation set by the overlay.

Add dimensions between overlay views to show the distance or angle a component has moved from its position in another representation. Drawing dimensions automatically update if the model position changes.

Some guidelines for using overlay views include:

- In the assembly, create design view representations that include only components of interest. In the overlay view, specify such a design view representation.

- In the assembly, create as many positional representations as are needed to show different positions.
- To change to a different positional representation for an overlay view, delete the overlay and specify a new positional representation when creating an overlay.

Tips for Creating Drawing Views

- Create nonaligned section views.
  Press and hold CTRL while placing section views to break the alignment.
- Move views between sheets.
  Click a view in the browser and drag it to another sheet. The cursor must be on a sheet name or icon to enable drop.
- Copy views or sheets between drawings.
  Right-click the view or sheet, and then select Copy. Paste it into the other drawing.
- Redline drawings.
  Use sketch overlay to redline drawings without affecting the drawing views or annotations.
- Use the context menus for quick access to editing operations and common commands.

Annotating Drawings

14

In this chapter, you learn about annotating drawings using dimensions, center marks, centerlines, hole tables and hole notes, parts lists and tables, and leader text.

Annotation Tools

Drawing annotations provide additional information to drawing views to complete documentation of a component. In Autodesk® Inventor™, styles define annotations, according to the active drawing standard. Each standard has a default set of available styles, which can be customized as needed.

NOTE Legacy documents usually had styles defined in a template. When using style libraries, import styles as needed from template-based documents to the library. Then purge all unused styles to avoid making the file size larger. You cannot purge styles that are in use.

The following are some of the tools on the Drawing Annotation toolbar:

Annotation Tool	Description
General Dimension	Adds drawing dimensions to a view.
Baseline Dimension and Baseline Dimension Set	Adds baseline drawing dimensions to a view to create a dimension set. Baseline dimensions add multiple dimensions to drawing views in an automated fashion.
Ordinate Dimension Set, and Ordinate Dimension	Adds two types of ordinate dimensions to your drawings. Individual ordinate dimensions provide support for importing AutoCAD® drawings containing ordinate dimensions.

Annotation Tool	Description
Hole/Thread Notes	Adds hole and thread notes to features created using the Hole feature or Thread feature tools in parts.
Bend or Punch Notes	Adds bend or punch notes to drawing views of sheet metal parts.
Chamfer Note	Adds a chamfer note to a drawing view.
Center Mark	Automatically sizes center mark extension lines to fit the geometry. You can copy and paste center marks.
Centerlines	Autodesk Inventor supports three types of centerlines: bisector, centered pattern, and axial.
Symbols	Adds symbols for surface texture, welding, feature control frames, and feature identifiers. You can create leaders for symbols.
Datum Identifier Symbol and Datum Target leaders	Creates one or more datum target symbols and leaders. The active drafting standard determine the color, target size, line attributes, and measurement units of the symbol.
Text or Leader Text	Both Text and Leader Text use formats such as font type, bold, and special symbols. Leader Text attached to geometry is associative, and moves with the drawing view.
Table	Inserts a table in the drawing. You can create a generic, configuration, or bend table.
Balloons	Adds balloons to individual parts or all parts at once. You can add balloons to a custom part after it is added to the parts list.
Parts list	Inserts a parts list into a drawing.
Hole table	Adds a hole table to a drawing view.

Annotation Tool	Description
Caterpillar	Adds a caterpillar annotation to geometry in a drawing view. The annotation is not associated with weldments in the model.
End Fill	Adds a 2D end fill annotation to geometry in a drawing view. The weld bead style determines the size and formatting.
Revision table	Places a revision table on a drawing sheet.
User Defined Symbols	Adds sketched symbols to a drawing or a template. User-defined symbols may include bitmaps, AutoCAD geometry, and 2D geometry created with sketch tools.
Retrieve Dimensions	Selects the model dimensions to display in a drawing view.

Using Styles to Format Annotations

In Setting Up Drawings on page 203, you learned about Style and Standard Editor. Default styles associated with the drafting standard are available in the Style and Standard Editor, where they can be edited, and new styles created, as necessary.

Styles control the annotation formatting. Using the Style and Standard Editor, you can see that some styles refer to other styles. For example, the settings in the text style specify the dimension text. Whenever the text style changes, all styles that reference it also update.

Working with Tables

Tables in drawings are useful for data that can be presented in tabular format. They are formatted by styles, and can be customized to suit many purposes. Hole Tables are formatted by the Hole Table style, Parts Lists are formatted by the Parts List style and Configuration and General tables are formatted by the Table style.

Hole Tables

Hole tables in drawings show the size and location of some or all of the hole features in a model. Hole tables eliminate the need to add notations for each hole feature in a model.

In addition to drilled, counterbored, and countersunk holes, you can add center marks, iFeatures, holes in patterns, and extruded cuts to a hole table.

The format for hole tables is set in the hole table style. You can specify the title, text style, heading, line format, default column settings, and the defaults for row merging, hole tags, tag order, and view filter.

If you want to include extruded cuts and iFeatures in hole tables, edit the Hole Table style and select them. Click Format ➤ Style and Standard Editor. In the Hole Table style, click the Options tab. In the Default Filters (View) box, select Circular Cuts and Centermarks.

General and Configuration Tables

Use the Table tool on the Drawing Annotation panel bar to add a general, bend, or configuration table to a drawing. The Table style sets the default format of tables.

You can create a general table with a default number of rows and columns, or customize its size. The general table can reference external data from *.xls, .xlsx* or *.csv* files or you can enter any other type of data you need.

General and configuration tables are versatile. You can sort data, export data to an external file, change the table layout, add or remove rows, and change the format of a row or column.

In drawings of iParts and iAssemblies, configuration table rows represent the members of the factory. You can specify the columns to include in the configuration table, such as exclusion status and values that are different among members.

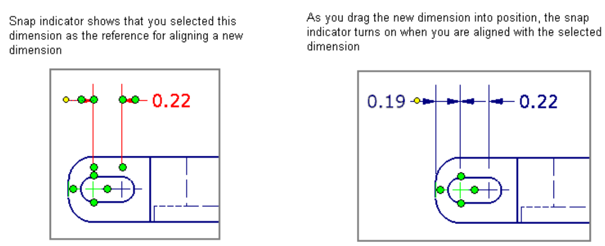
Parts Lists

To create a parts list in a drawing in Autodesk Inventor, use the Parts List tool. Its default formatting is set in the Parts List style associated with the active drafting standard.

You can generate a parametric parts list for an assembly. The properties for each part or subassembly are displayed in the parts list. You can specify the items you want in the list, such as part number, description, and revision level. You can edit a parts list.

Creating Dimensions In Drawings

The tools you use to create drawing dimensions are different from the tools used for model dimensions. When you select a feature or relationship between features to dimension, Autodesk Inventor creates a horizontal, vertical, or aligned dimension, depending on the direction you move the cursor. Snap is activated to help place a dimension at a standard distance from the view and to align dimensions with each other.



Place Dimensions

You can use two types of dimensions to document your design in a drawing: model dimensions and drawing dimensions.

Model Dimensions

Model dimensions define the sizes of features. If you change a model dimension in a drawing, the source component updates to match. Model dimensions are also referred to as a bidirectional or driving dimensions.

Only model dimensions parallel to the view plane are available in a view. If, when installing Autodesk Inventor, you select the option, Modify a Model

Dimension from a Drawing, you can edit a model dimension and the source component also updates.

Use the Retrieve Dimension tool to display model dimensions. After you select the dimension to retrieve, right-click a dimension to delete or edit. You can drag dimensions to adjust their positions.

When you place a view, you can choose to display model dimensions. Usually, model dimensions are in the first, or base view in a drawing. In subsequent projected views, only those model dimensions not shown in the base view are displayed. If it is necessary to move a model dimension from one view to another, right-click the dimension in the first view and select Move. Click the second view to move the dimension. As an alternative, you can add a drawing dimension to the second view.

NOTE If you choose to change the model dimensions in the drawing, make only minor changes to single dimensions. If there are significant changes, or if you need to modify dimensions that are referred to by other dimensions, open the part and edit the sketch or feature there.

To avoid accidentally modifying a standard part, you can prevent the editing of driven dimensions in read-only parts that are referenced to the drawing file.

If you change the size of a part that is used multiple times in an assembly or is used in multiple assemblies, all occurrences of the part are resized. Check other assemblies to see if the changed size causes interference.

Drawing Dimensions

Drawing dimensions are unidirectional. If the part size changes, the drawing dimension updates. However, changing a drawing dimension does not affect the size of a part, unless you specified differently when you installed Autodesk Inventor. Usually, drawing dimensions are used to document, but not to control, the size of a feature.

You use the same tools to place drawing dimensions as sketch dimensions. Linear, angular, radial, and diameter dimensions are all placed by selecting points, lines, arcs, circles, or ellipses, and then positioning the dimension. Constraints are inferred to other features as you place drawing dimensions.

Autodesk Inventor displays symbols that indicate the type of dimension being placed. Visual clues are also used to position dimensions at fixed intervals from the object.

Change Dimensions

Dimensions styles control the appearance of dimensions in drawings. When you apply the style to a dimension, it takes on the characteristics defined in the style. Within the dimension style, the text style is referenced as a substyle to format dimension text. Alternate units, the preferred display style, tolerance, leaders (which reference the leader style) and text are also specified in the dimension style.

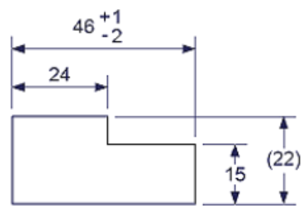
After you place a dimension, you can right-click the dimension and use options on the context menu. You can change:

- Options for arrow head position and appearance and if a leader is created.
- Set precision.
- Open the Edit Dimension dialog box to edit dimension text, change precision and tolerance. Specify the dimension as an inspection and set its appearance, or override the model value.
- Open the Format Text dialog box to add or change parameters, and modify text attributes such as justification, position, font, spacing and other settings.
- Hide dimension value or extension line.
- Edit first and second arrowheads.
- Create a style or edit the dimension style.

Controlling Dimension Styles

A dimension style is a named group of dimension settings that determine the appearance of a dimension to which it is applied. Autodesk Inventor controls drawing dimensions through the Style and Standard Editor. The available dimension styles are listed in the browser pane of the dialog box. You can click a dimension style to view and edit its settings.

This figure shows several dimensions that were modified by dimension styles.



Use dimension styles to control dimension text, arrowheads, dimension lines, and extension lines. A dimension style is provided for each drafting standard, but you can create new styles to suit your own annotation requirements.

These examples show a dimension that uses the default ISO dimension style, and one with custom style settings applied.

default ISO dimension style

custom dimension style

TRY IT: View dimension styles in the Styles and Standards Editor dialog box

1. Open an existing drawing or create a drawing.
2. Choose Format ➤ Style and Standard Editor.
3. Expand Dimension in the browser pane and then double-click a dimension style. Click the tabs to see how attributes are defined, and then click Done to close the dialog box.

You can change the settings for the default dimension style, or you can create your own variation of the dimension style and save it with a new name. You can apply a dimension style to any number of drawing dimensions.

TRY IT: Override dimension style settings

1. In the drawing, right-click a dimension, and then choose New Dimension Style.
2. In the New Dimension Style dialog box, click a new dimension style to apply it, and then click OK.

NOTE If you apply a dimension style to a dimension, overrides on that dimension are lost.

Copy Dimension Styles among Drawings

The Style Library Manager provides a convenient way to copy dimension styles (and other styles) from one drawing to another.

Close Autodesk Inventor before using the Style Library Manager.

TRY IT: Access the Style Library Manager

1. On your desktop, click Start ➤ Programs ➤ Autodesk ➤ Autodesk Inventor [version] ➤ Tools ➤ Style Library Manager.
2. In the Style Library Manager dialog box, double-click the Dimension style. All dimension styles available in the current document are listed.

   If you prefer, click the Browse button in the Style Library 1 pane and locate a different style library.
3. In Style Library 2 pane, click the Create New Library button or browse to an existing library. Accept the default or give the new library a name and click OK.
4. Select the styles from Library 1 to add to Library 2 and click the right arrow button to add the styles. Optionally, you can click buttons to Show All Styles, Show Mismatched Styles (style names are the same but definitions differ in the two libraries), or Unique Styles (definition exists in one library, but not the other).
5. Click Exit.

NOTE Do not create a style in an existing library unless you have authority to do so. You could replace an existing definition that could affect the formatting of other documents.

Placing Center Marks and Centerlines

Autodesk Inventor has simplified the tasks of placing center marks and centerlines. There are four tools to assist you:

- Center Mark

- Centerline
- Centerline Bisector
- Centered Pattern

Add center marks and centerlines before adding drawing dimensions. You can dimension to the ends of the center marks and centerlines and maintain correct gaps.

You can add center marks to extruded circular cut features and include these cuts in a hole table. Add the center marks to the hole table style so they are recognized in the drawing.

TRY IT: Add center marks, circular cuts, and hole features to the hole table style.

1. Open a drawing file.
2. Click Format ➤ Style and Standard Editor.
3. In the browser pane of the Style and Standard Editor dialog box, expand Hole Table and double-click the style to edit.
4. Click the Options tab. In the Default Filters (View) box, in the Included Features category, select hole features, circular cuts, and center marks.
5. Click Done and then click Yes to save edits.

Now, hole features, circular cuts, and center marks can be selected for inclusion in a hole table.

In addition to manually placed centerlines, you may be able to use automated centerlines in a drawing view. When you set up a drawing, use options in document Settings to define the default criteria for adding automated centerlines. If you want to use them in all drawings, set them in drawing templates. You specify the types of features to receive centerlines and if the geometry is normal or parallel projection, as well as thresholds to exclude circular features smaller or larger than a specified radius and smaller than a minimum angle.

Adding Notes and Leader Text

Use the Text tool to add general notes to a drawing. General notes are not attached to any view, symbol, or other object in the drawing.

Use the Leader Text tool to add notes to elements in a drawing. If you attach the leader line to geometry in a view, the note is moved or deleted when the view is moved or deleted.

The Format Text dialog box is used to enter text and to set the text parameters.

Using Hole and Thread Notes

Hole and thread notes document both internal and external hole features or threaded objects. These notes typically contain information necessary to manufacture a threaded feature:

- Hole diameter and depth
- Thread size and depth
- Counterbore or countersink size
- Quantity, especially for hole patterns

Use the Hole/Thread Notes tool to add hole notes and thread information to holes and threaded features in drawing views.

Autodesk Inventor captures the information used when creating holes and threaded features on a part. This information accurately generates the hole or thread notes in drawing views. If you modify a hole or threaded feature, the hole or thread note is automatically updated.

Hole and thread notes are generated according to the current drafting standard. Right-click a hole note, and then choose Text from the context menu to change its format and choice of parameters.

A thread is always considered right hand unless otherwise specified. A left-hand thread is always labeled LH on a drawing.

Thread Representations

Autodesk Inventor drawings present threads using the simplified method. Visible external threads display in side views, sections views, and shaded views.

In a drawing view, you can add hole and thread notes to features created using the hole feature or thread feature tools in parts. In addition, a hole note may be added to extruded cuts (except midplane extrusions), iFeatures, holes in patterns, and sheet metal flat patterns.

In section views, the hole must either be displayed in its face normal position or seen as a profile.

You can also annotate holes in isometric views.

Working with Title Blocks

Title block information that you typically enter when you complete a drawing is obtained from the drawing properties. Right-click the drawing name in the drawing browser and select iProperties. You enter information in the Properties dialog box, and the values are displayed in the corresponding locations in the title block.

The Drawing Properties dialog box has six tabs for entering information:

- General
- Summary
- Project
- Status
- Custom
- Save

While some of the title block information is entered when you start the drawing, approvals and approval dates are not entered until the drawing is complete.

Other information displayed in the title block is derived from the operating system, the drawing, and the sheet.

NOTE The date format is set by the system Regional Settings application, located in the Control Panel folder.

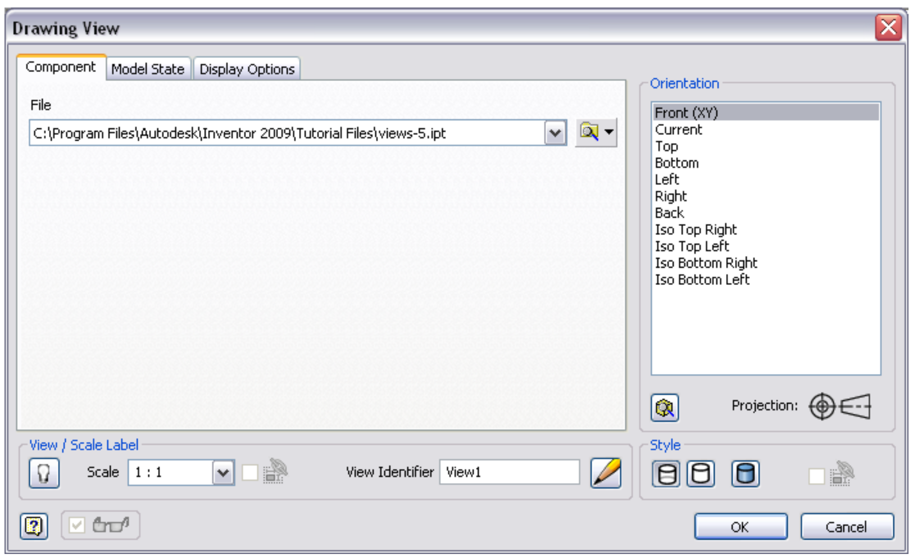
Working with Dimensions and Annotations

In this exercise, you create drawing views, edit a view, and then add dimensions and annotations to a drawing of a clamp that is used to hold a work piece in position during machining operations.

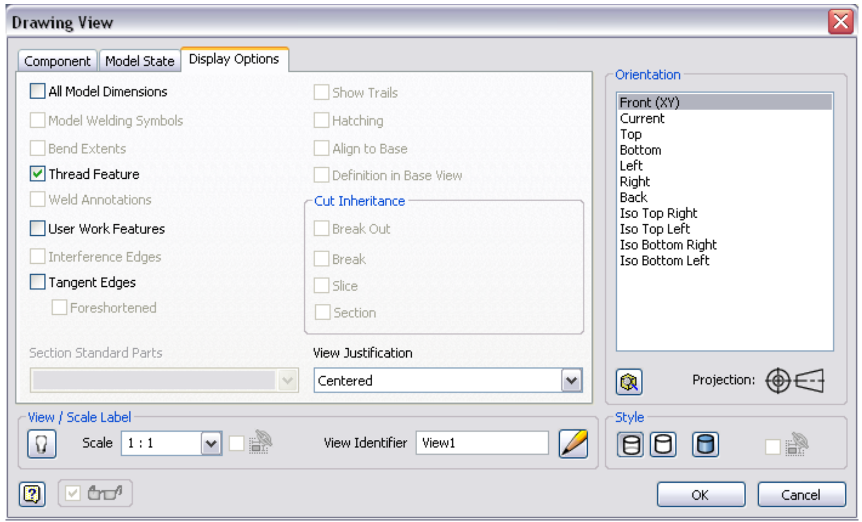
Both model dimensions and drawing dimensions are used to document feature size.

TRY IT: Add views to a drawing

1. With the project tutorial_files active, open the file *dimsannot-5.idw*. The drawing file contains a single sheet with a border and title block.
2. Click the Base View tool in the panel bar or from the Drawing Views panel bar. The Drawing View dialog box is displayed.
3. Click the Browse button, and then double-click *views-5.ipt* to use it as the source for the view.
4. On the Component tab, verify that Front is selected in the View list. Set the Scale to 1:1.



5. Click the Display Options tab. Make sure that All Model Dimensions is not selected. Select the Tangent Edges check box.



6 Position the view preview in the lower left corner of the sheet (in Zone C6). Click the sheet to place the view.

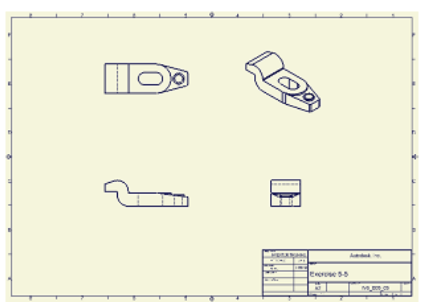
7 Click the Projected View tool in the panel bar or from the Drawing Views panel bar.

Click the base view and move the cursor vertically to a point above the base view. Click the sheet in Zone E6 to place the top view.

8 Move the cursor horizontally to the right of the base view. Click the sheet in Zone C3 to place the right-side view.

9 Move the cursor above the right-side view. Click the sheet in Zone E3 to place the isometric view.

10 Right-click the sheet, and then select Create.

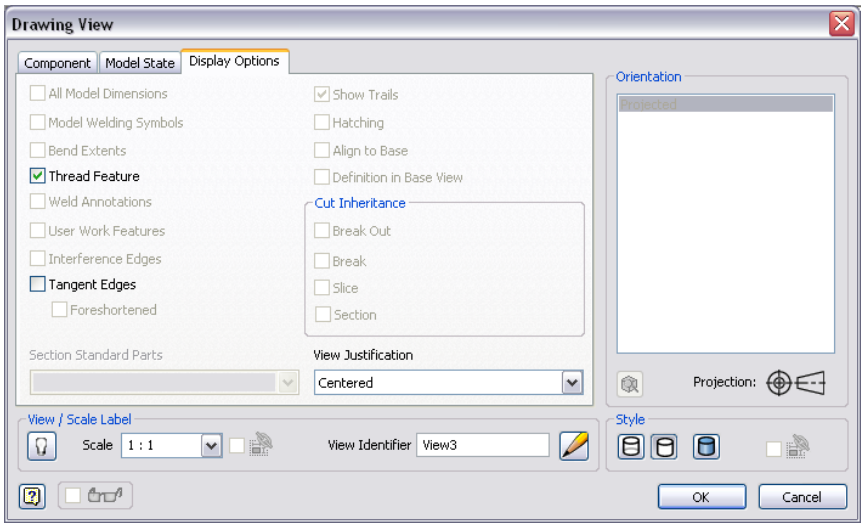


Turn Off Tangent Edge Displays

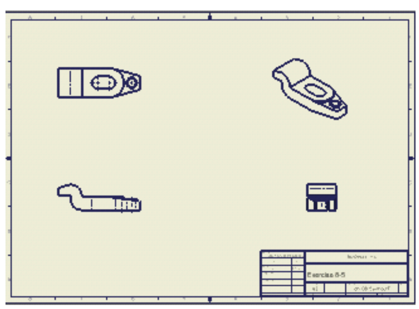
Turn off the display of tangent edges in the isometric view.

TRY IT: Modify a drawing view

1. Right-click the isometric view, and then select Edit View.
2. In the Drawing View dialog box, click the Options tab, and then clear the check mark from Tangent Edges. Click OK.



The following are orthographic and isometric views of the clamp.



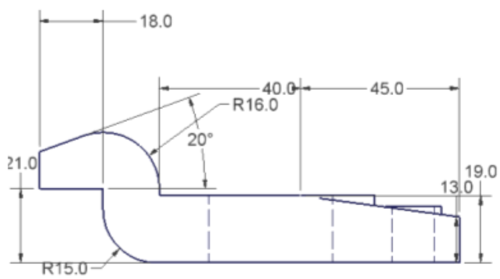
Add Model Dimensions

Next, you add model and drawing dimensions to the views using the Retrieve Dimensions command. Some model dimensions are removed, while others are repositioned.

TRY IT: Add model dimensions

1 Zoom in on the front view.

2 Right-click the front view, and then choose Retrieve Dimensions. In the Retrieve Dimensions dialog box, click the Select Dimensions tool. The model dimensions that are planar to the view are displayed.



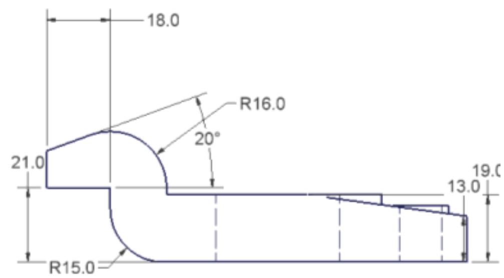
3 Select each of the dimensions except for the 45.0 horizontal dimension and the 40.0 horizontal dimension.

4 **NOTE** If you prefer, click and then drag a window around the model to select all of the dimensions in the view. You can then delete the dimensions you do not need.

Click Apply. Each of the selected dimensions are displayed. The dimensions that were not selected are hidden.

NOTE If you accidentally selected a dimension, hold down the CTRL key and reselect it to remove it from the selection set.

5 Click Cancel to exit dialog box.



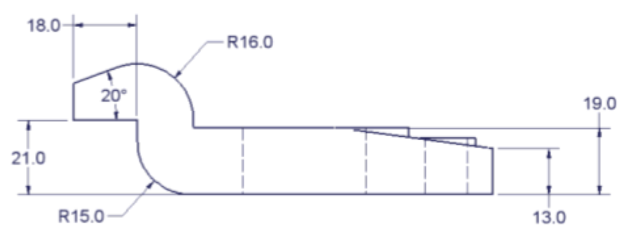
Reposition Model Dimensions

To reposition dimension text, click a dimension text object, and then drag it into position. The dimension are highlighted when it is a preset distance from the model.

Radial dimensions can be repositioned by selecting the handle at the end of the leader.

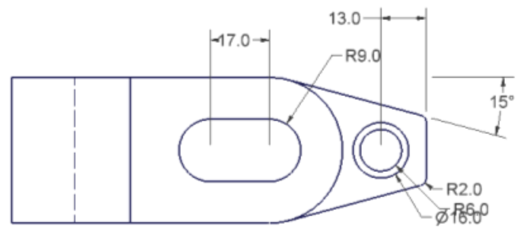
TRY IT: Reposition radial dimensions

1 Drag the dimensions until they appear as illustrated in the following figure.



2 Pan to display the top view, right-click, and then choose Done.

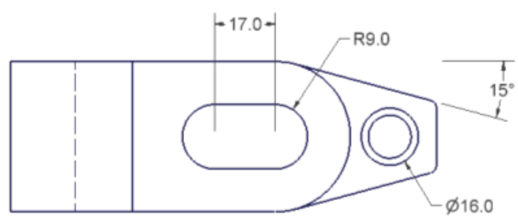
3 Right-click the top view, and then choose Retrieve Dimensions. In the Retrieve Dimensions dialog box, click the Select Dimension tool. The model dimensions that are planar to the view are displayed.



4 Select each of the dimensions except the 13.0 horizontal dimension, and the R6.0 and R2.0 radial dimensions.

5 Click Apply. Each of the selected dimensions are displayed. The dimensions that were not selected are hidden. Click Cancel to exit dialog box.

6 Drag the remaining dimensions until they appear as shown in the following figure.



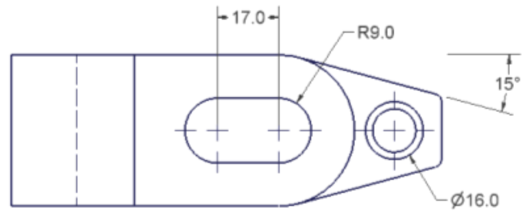
Add Centerlines and Center Marks

Centerlines and center marks are added to aid in the placement of drawing dimensions.

TRY IT: Add centerlines and center marks

1 On the Drawing Annotation panel bar, click the Center Mark tool in the panel bar or from the Drawing Annotation toolbar.

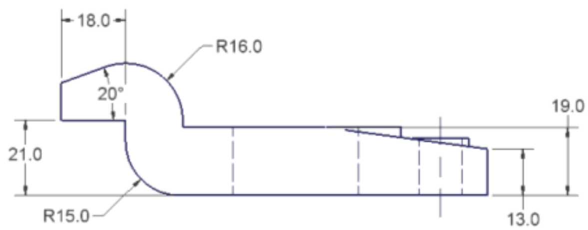
2 Click the outer circle of the boss and the two arcs of the slot.



3 Pan to display the front view.

4 Click the arrow beside Center Mark and then click the Centerline Bisector tool.

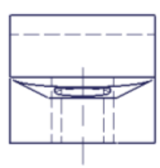
5 Select the two hidden lines that represent the drilled hole through the boss.



The bisecting centerline is added.

6 Pan to display the right-side view.

7 Select the two hidden lines that represent the drilled hole through the boss.



The bisecting centerline is added.

Add Drawing Dimensions

Drawing dimensions are added to complete the documentation of the model.

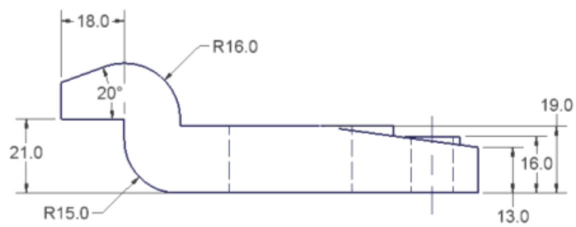
TRY IT: Add drawing dimensions

1 Pan to display the front view.

2 Click the General Dimension tool on the Drawing Annotation panel bar.

3 Click the right endpoint of the bottom edge, and then click the right endpoint of the top of the boss.

4 Move the cursor to the right and place the 16.0 dimension between the 13.0 and 19.0 vertical dimensions, as shown in the following figure.



5 Pan to display the top view.

6 Use the General Dimension tool to add the 13.0, 45.0, and 40.0 horizontal dimensions as shown in the following figure.

NOTE To align a dimension when dragging it, move the cursor over an existing dimension and acquire an alignment point. Move the cursor back to the dimension being placed. The dotted line indicates an alignment inference. Click to place the dimension.

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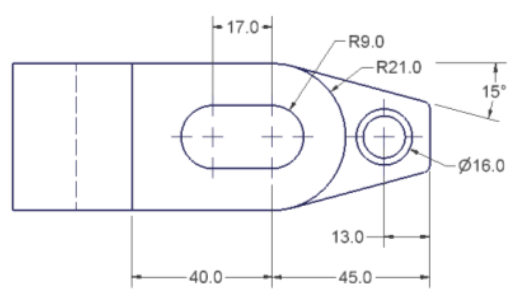
7 Use the General Dimension tool to add the R21.0 radial dimension, right-click, and then choose Done.

8 Drag the 16.0 dimension to a position that avoids crossing the extension lines.



The drawing dimensions are added.

Format Dimensions

The dimensions can be formatted to add additional information, to adjust precision, or to add tolerances.

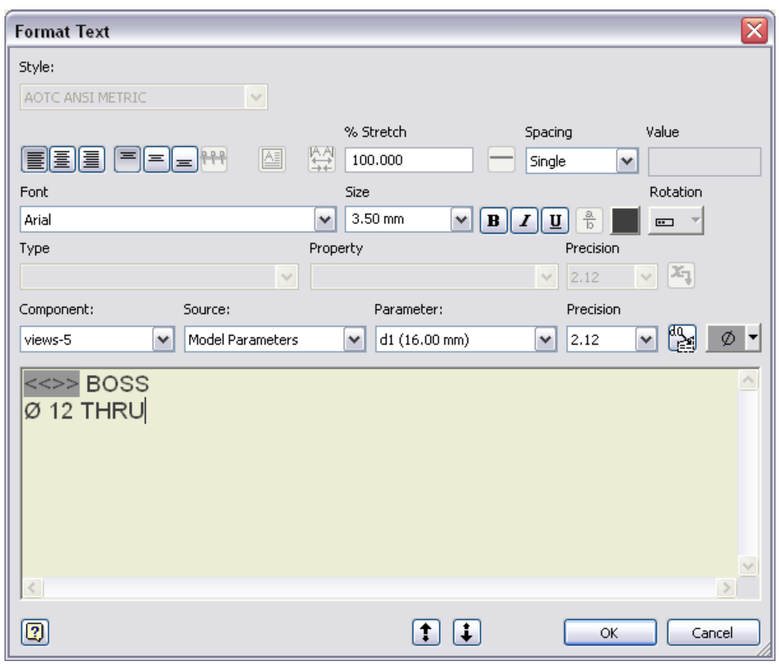
TRY IT: Format dimensions in a drawing

1. Right-click the 15° dimension, and then choose Text.
2. In the Format Text dialog box, enter *TYP,* and then click OK.
3. Right-click the 16.00 dimension, and then select Text.
4. In the Format Text dialog box at the insertion point, press the space bar, and then enter *BOSS.* Press ENTER.

   Select the Diameter symbol from the symbol list in the dialog box.

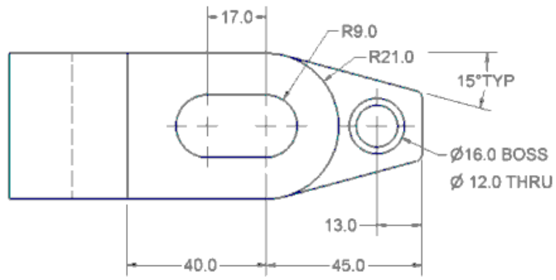
   Select Arial from the font drop-down list.

   Press the space bar, and then enter *12.0 THRU*.



Click OK.

The formatted dimensions are displayed.

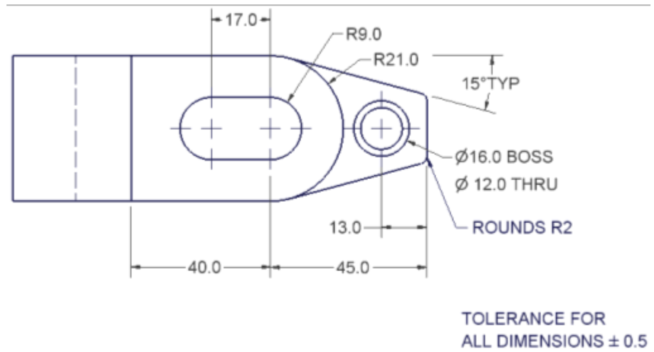


Add Notes and Leader Text

In the following steps, you add a general note and use leader text to document the round.

TRY IT: Add a note and leader text to a drawing

1. Click the Text tool in the panel bar or from the Drawing Annotation toolbar.
2. Click a point below and to the right of the top view.
3. Enter *TOLERANCE FOR*, and then press ENTER.
4. On the next line, enter *ALL DIMENSIONS* (space).
5. Select the tolerance icon from the symbol list. Enter *0.5*. Click OK. Right-click, and then choose Done.
6. Click the Leader Text tool in the panel bar or from the Drawing Annotation toolbar.
7. Select the bottom arc on the right end to define the leader start point.
8. Click a point below and to the right to define the end of the leader, right-click, and then select Continue.
9. Enter *ROUNDS R2*. Click OK.



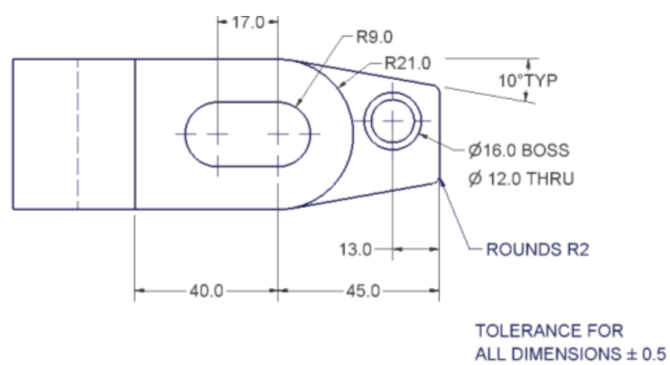
Edit Model Dimensions

If, when you installed Autodesk Inventor, you set the option to allow drawing dimensions to resize the model, when a model dimension is edited, the part model is updated along with the drawing views.

TRY IT: Edit a model dimension in a drawing

1. Right-click the 15° dimension, and then choose Edit Model Dimension.
2. In the Edit Dimension dialog box, enter 10-deg for the new dimension, and then press ENTER.

   The model and drawing are updated.
3. Click the 10° dimension, and then drag to position it correctly. Reposition any other dimensions that were moved.



Notice how the position of the boss was affected by the change to the model dimension.

WARNING Modifying a model dimension directly affects your model. Autodesk Inventor automatically updates the part file with the changes you make.

Complete Title Blocks

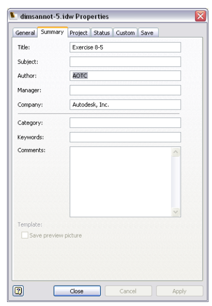
The drawing properties are used to complete the title block information.

TRY IT: Complete a title block

1 From the File menu, select iProperties.

The Properties dialog box is displayed.

2 On the Summary tab, in Author, enter your name.

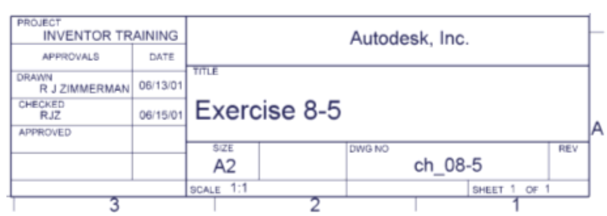


3 Click the Status tab and select the current date from the Checked Date list.

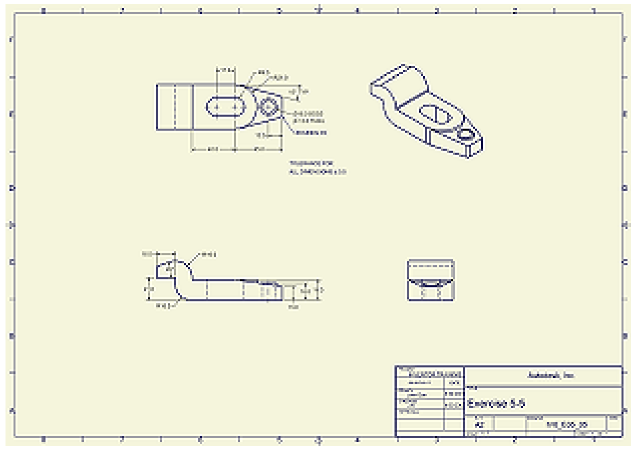
4 In Checked By, enter your initials.

5 Click OK.

The title block is updated.



The drawing is complete.



Save the file.

Printing Drawing Sheets

Autodesk Inventor uses any Microsoft® Windows® configured printer to print a copy of your design documentation. Most large-format plotters can be configured as Windows system printers. In the Print Drawing dialog box, you can control the following:

- Printer selection
- Print range if you have a multisheet drawing
- Scale
- Print all colors in black and white
- Rotate by 90 degrees
- Remove object line weights
- Number of copies

From the Print Drawing dialog box, you can display a preview of the plot based on the selected printer and the current settings.

If the drawing is too big to print on one sheet, select the Tiling Enabled check box. This option is only available when the scale is set to Model 1:1. Registration marks are printed on page corners to allow alignment of printed pages. Page identifiers contain the drawing and sheet name and a table cell number to help keep pages in order.

Plotting Multiple Sheets

Use the Multi-Sheet Plot wizard to plot multiple drawing sheets that include drawings of various sizes. You access the wizard from the Start menu on your computer task bar. Click Programs ➤ Autodesk ➤ Autodesk Inventor [version] ➤ Autdesk Multi-Sheet Plot. You can:

- Set up the printer to use
- Set layout preferences
- Specify the project that contains the drawings you want to plot
- Select the drawing files to plot
- Preview the generated composite (which automatically arranges drawings for efficient use of paper)
- Plot now, print to a file, or schedule a plot job in the Task Scheduler.

If sheets are larger than the plot area, you can change page size or remove files from the print list.

Tips for Annotating Drawings

- Use text parameters to display drawing properties and other information in the title block.
- Use the cursor symbols as cues to place and align dimensions.
- Drag the dimension text and dimension handles to reposition dimensions.
- Edit model dimensions (not drawing dimensions) to update the model.
- Include general tables and customized style overrides in the drawing template so they are available in all drawings based on that template.

Using Content Center

15

This chapter provides basic information and concepts about Content Center and Content Center libraries. For more detailed information, see Help in Autodesk® Inventor™ and Autodesk® data management server.

About Content Center

Content Center is a tool used for accessing and maintaining the Content Center library. You use Content Center to:

- Find a part in a Content Center library.
- Insert a Content Center library part in an assembly.
- Edit Content Center library parts placed in assembly.
- Customize, add, or remove Content Center library parts.
- Publish parts, iParts, or features in the Content Center library.

Content Center can be installed as a part of Autodesk Inventor.

Set and Manage Permissions

Editor and Administrator accounts are required for users who access the Content Center libraries installed on a central server and must edit the Content Center libraries or perform administrator tasks. Use Autodesk® Server Console to create user accounts for anyone requiring these permissions. Communicate user account information and the Log In procedure to each member in the team.

Content Center Library

The Autodesk Inventor Content Center library provides Inventor parts (fasteners, steel shapes, shaft parts) and features to insert in assemblies.

Libraries can be either local, or in a shared environment accessed from a central server. The Content Center library data are accessed in the Content Center. See the Help for more information about the library configuration.

The basic component in the Content Center library is a family (part family or feature family). A family contains related content (members) based on the same underlying templates. A family is composed of parts with the same shape but with different sizes. A family member is a part or feature with a specific size. The family member is the lowest level of the hierarchy.

Families are arranged in categories and subcategories in the Content Center library. A category is a logical grouping of part types. For example, studs and hex head bolts are functionally related, and nested under the Bolts category. A category can contain subcategories and can contain families. A family cannot be subcategorized.

Two types of parts are included in the Content Center library: standard parts and custom parts. Standard parts (fasteners, shaft parts) have all part parameters defined as exact values in the table of parameters. Custom parts (steel shapes, rivets) have a parameter arbitrarily set in the defined range of values.

Content Center Library Data

The Content Center library contains required data to create part files for Content Center library parts. The data are:

- Parametric *.ipt* files which provide the graphics for Content Center library parts.
- Values of part parameters.
- Descriptions for parts including family properties such as family name, description, standard, standard organization.
- Preview pictures displayed in the Content Center.

Parametric *.ipt* files, description texts, and preview pictures are common for all sizes of one part family. The Content Center library usually contains several sets of parameter values for one part family. Every set of parameters defines one member of the part family.

Working with Content Center

You use the Content Center dialog box to navigate in the Content Center library hierarchy. You expand categories in the Category Listing panel, double-click items in the List panel, or use navigation buttons in the toolbar, such as Back, Forward, and Up one Category Level.

Content Center Environments

There are two distinct environments:

- Consumer environment for locating and placing a library part or feature into a document.
- Editor environment for editing a Content Center library.

If the Content Center libraries are installed on a remote server, you must be assigned Editor permissions in Autodesk Server Console to perform any of the editing tasks. You must also be logged into your Content Center server. If you work in a single-user environment, you automatically have permissions to perform editing tasks and you log into the Autodesk server on your local drive.

The Content Center Consumer and Editor environments include methods for locating the content including:

- Browsing capabilities.
- Advanced and Quick search tools to find parts that correspond to appropriate parameters.
- Filters for excluding families you do not use.
- Favorites for creating your own personal folder structure with shortcuts to Members, specific Families or Categories.
- History option to display the last 20 used parts in the Category and Family pane.

Consumer Environment

Use the Content Center Consumer environment to use library parts in the design process. The following commands are available:

Open from Content Center	Opens a family *.ipt* file.
Place from Content Center	Places a feature or a part into an open assembly file.
Place Feature from Content Center	Places a feature into an open part file.
Change size	Changes the size of a part placed in an assembly.
Replace from Content Center	Replaces an existing part in the assembly with a part from the Content Center library.

TRY IT: Navigate in the Content Center library and place a part

1. To begin, open an assembly file.
2. Click Place from Content Center on the Assembly panel. The Place from Content Center dialog box is displayed.
3. To select the viewing mode, click the Thumbnails View, List View, or Detailed View tool button.

   NOTE The List View mode is faster, but the Thumbnails View mode is more descriptive.

4. In the panel on the right, browse in the library until you get to the appropriate part family:

   Double-click a category to display its descendant categories or part families.

   Click Back to return to the previously displayed list.

   Click Forward to display the forward list.

   Click Up one Category button to get up one level in the category structure.

5. Select the family and click OK.
6. Select the family member to place in the Select or Table View tab of the Family dialog box.
7. Click OK in the Family dialog box.

8 Use the typical placement operation to place the part in assembly. Add constraints as needed to position the part to other geometry.

Editor Environment

Use the Content Center Editor environment to modify library parts by using one of the following commands:

- Create, Delete, and Edit Categories.
- Copy a read-only category or family to a read/write library for editing purposes.
- Use the Save Copy As command to create a copy of a family in a read/write library.
- Rename library parts and families.
- Edit family properties and family tables and then publish changes to a read-write library.

TRY IT: Customize libraries

1 Connect to the data server and have editing permissions to edit Content Center libraries.

   You can only modify library data in read/write libraries. If desired, create a read-write library using Autodesk Server Console.

2 Click Tools ➤ Content Center Editor. The Content Center Editor dialog box displays.

3 To edit a part family that is contained in a read-only library, copy the family to a read/write enabled library: Right-click the family, select Copy To, and select a read/write library from the list.

   After the family is copied, it can be edited. Edits are saved to the read/write library.

4 Right-click the family and select Family Table. The Family Table dialog box displays.

5 Edit the family data. Use context or toolbar commands to work with the data.

6 Click OK.

7 On the Content Center Editor dialog box, click Done.

Tips for Using Content Center

- Use Search to find a part in the Content Center library. You can search for parts with a specified string in the Part Number or Description property, or you can specify conditions for part family or category parameters.
- Use Content Center Favorites to store frequently used parts or part families. You can create a folder structure in Favorites and order favorite items as you need. You can create additional Favorites groups.
- Remove all Content Center libraries you do not use. If you remove libraries from Content Center configuration, the amount of displayed data reduces and the performance increases.

Using the Publish Tool

The Publish tool uses a separate dialog box. You use the Publish tool to:

- Publish parts, iParts, or features.
- Define and map family properties.
- Select the library to publish to.

If you publish a part or iPart in Content Center library, the published part is saved as a part family to a selected category. Before you publish a part, map the part parameters to the category parameters.

For more information on publishing to the Content Center library, refer to the Help in Autodesk Inventor.

NOTE You must have Content Center Editor permissions to publish a part or feature.

Managing Administrative Tasks

You add and remove libraries to and from the active Content Center configuration, using the Configure Content Center Libraries dialog box. The configuration of Content Center libraries is saved in the active project.

Libraries must be configured on the Autodesk server before you configure Content Center libraries in the Inventor project. Perform all administrator tasks using Autodesk Server Console.

- Create a new read/write or read-only library.
- Detach a library from the server.
- Attach or reattach a library.
- Delete a library from the computer.
- Create user accounts and permission. Set up Editor accounts for users who edit or publish libraries and access the Content Center libraries from a central server.

Autodesk Inventor Utilities

16

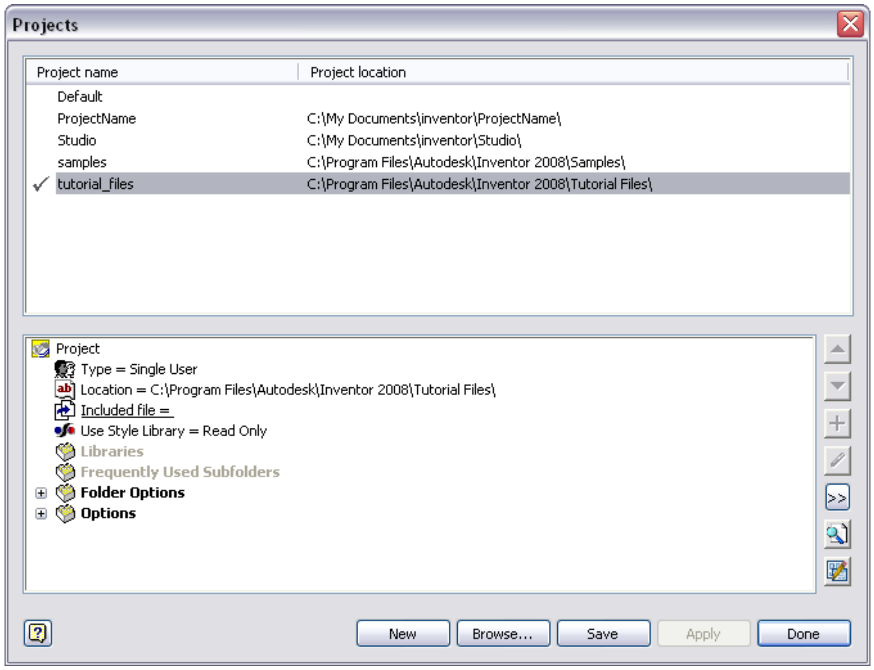
In this chapter, you learn how to edit a project in Autodesk® Inventor™, resolve missing file links, search rules for library and non library files, and old versions of files. You learn how to copy, move, rename and delete data, and change the file structure in a project.

Editing Projects

After you create a project, you can use the Project Editor to change some of its options, add or delete file locations, or change its name. To edit the active project or change which project is active, first close all open Autodesk Inventor files.

In general, avoid creating multiple editable locations. File resolutions problems greatly increase when you use a complicated file structure.

The active project is marked read-only once you open files. An exception is that you can add libraries when files are open.



Keep in mind:

- To add a single folder to a project path, right-click Libraries or Frequently Used Subfolders, and then select Add Path. Browse to the individual folder and add it to the project.
- To add an editable location for each immediate subfolder of a folder, right-click a search path, and then select Add Paths from Directory. Browse to the root folder and add it to the project.
- To change the order in which locations are listed, select a location, and then use the Move Up or Move Down arrow on the right side of the Project Editor.
- If you delete a path, Autodesk Inventor is unable to find referenced files from that path.
- To add a Content Center library, click the Configure Content Center Libraries button on the right side of the Project Editor and use Data Management Solutions tools. You must be logged in to Data Management.

TRY IT: Edit a project

1 Verify that all Autodesk Inventor files are closed.

2 Use one of these methods to start editing:

- Click File ➤ Project.
- On the Microsoft® Windows® Start menu, click Programs ➤ Autodesk Inventor ➤ Tools ➤ Project Editor.
- On the Microsoft Windows Start menu, click Autodesk ➤ Autodesk Inventor ➤ Tools ➤ Project Editor.
- In Microsoft® Windows® Explorer, right-click an *.ipj* file, and then click Edit.

3 In the Project Editor, double-click a project name in the top pane to make it the active project. A check mark indicates the active project.

4 In the lower pane, right-click the type of path to edit, and then select an option from the menu:

Add path	Browse to the folder you want to add. Add a custom name for the folder, if desired.
Add paths from file	Browse to another project file. The paths from the file are added to the current project file. Use only to add libraries.
Add paths from directory	Browse to any folder on any disk drive where you have files in its subfolders. A path is automatically generated for each subfolder. For best results during file resolution, avoid nesting paths. Use only for libraries.
Paste paths	Paste a path from the Clipboard into the selected section of the project. Use only for libraries.
Delete section paths	Deletes all of the paths from the selected section of the project.

5 If you prefer, select the type of path to edit, and use the Add or Edit button on the right side of the dialog box.

6 Right-click other options to make changes. You can, for example, add frequently used subfolders, rename the project, expand and change folder options, or change the project owner and release ID.

7 Click Save, and then click Close.

NOTE To review definitions of all project options, click the Help button on the Project Editor dialog box.

Legacy Project Types

In the Project wizard, semi-isolated and shared project types are unavailable by default. Autodesk® Vault is the recommended solution for managing multi-user projects. If, however, you have legacy projects and have a requirement to create and use them, consider the following points:

- Click Tools ➤ Application Options ➤ General table. Select the check box for Enable creation of legacy project types. The legacy project types can then be created with the Project wizard.
- Use the Help topics to learn how to set options, set up workspaces and workgroups. Multi-user projects have additional restrictions and capabilities from single-user projects.
- Do not use Design Assistant Manager to change files that are checked out in a semi-isolated or shared project. Always make sure that all files are checked in to the shared storage location.

Resolving File Links

When one Autodesk Inventor file references another file, the relative path from the first project location containing the referenced file, the file name and library name (if it is in a library location) is saved in the referencing file.

Autodesk Inventor uses this information to locate the referenced file the next time the source file is opened. The first file found that matches validation conditions and reference information stored in the source document is loaded. The location must be in a defined project path to be resolved automatically.

NOTE An exception is if no project location is defined, such as when using the default project or when a source file and its referenced files are both copied outside a project.

Autodesk Inventor cannot find a file if:

- The file is no longer located in a location defined in the active project.

- The Using Unique File Names option is No, and the file was renamed, moved to a different subfolder, or one of the project subfolders was renamed.
- A library was renamed or its location was removed from the project.
- The file was moved from one library to another or from an editable location.
- The file was moved from one library subfolder to another.
- The data set was taken off site without the shared libraries. If it is acceptable, select the Skip All options when the Resolve Link.
- A network location defined by a project becomes unavailable.

When a file cannot be located, the Resolve Link dialog box opens automatically, showing the location and file name from the last save.

In the Resolve Link dialog box, choose an option:

Browse	Browse to a new location for the part, and open it.
Skip	Loads the assembly without the missing component file.
Skip All	Loads the assembly without trying to resolve any missing files.
Cancel	Cancels file loading and closes the dialog box.

Sometimes an entire set of files is missing. For example, the library name changed or a subfolder was moved or renamed. Because many files would be missing for the same reason, Autodesk Inventor automatically attempts to find other unresolved files that were originally in the named library or folder by searching in the new location you specify in the Resolve Link dialog box.

In the same session, if other files in the same project have unresolved references to the same path, you can select the check box to search for other unresolved references using the location.

To resolve the file correctly in the future:

- Move the file to a location in the active project. If you move the file to a subfolder of a location, use the Resolve Link dialog box once to establish the correct subfolder path to the new location of the file.
- Save the referencing file to save the updated information.

Search for Library and Non Library Files

Autodesk Inventor searches for referenced files in library locations in the order listed in the Project Editor and then the workspace.

If a referenced file is contained in multiple project locations, the reference uses the relative path from the first one found, and stores the path in the reference. If the project location is a library, the name of the library is also stored in the reference.

If the referenced file is not found in a project location, the relative path from the referencing file is used in the reference.

If the referenced file is also not located in the folder or a subfolder of the referencing file, the absolute path is stored.

A message warns you if the file is not in a location specified in the project. Confirm its location each time you open the referencing file until you relocate the file in the project location.

Autodesk Inventor uses a different search procedure for resolving library and non library references.

Search for Library References

Library references use the following rules:

- A reference to a file in a library location includes the library name in the reference.
- If a reference includes a library name,Autodesk Inventor searches *only* in that library location for the file.
- If the source file is in a library, the referenced file is presumed to be in the same library unless the reference identifies a different library.

NOTE Avoid using duplicate file names, even for files in different directories. Set the project option Using Unique File Names to Yes. The Resolve Link dialog box opens only if the file cannot be located in any project location.

If no library location is defined in the project,Autodesk Inventor searches for the referenced file relative to the source file.

Search for Non Library Locations

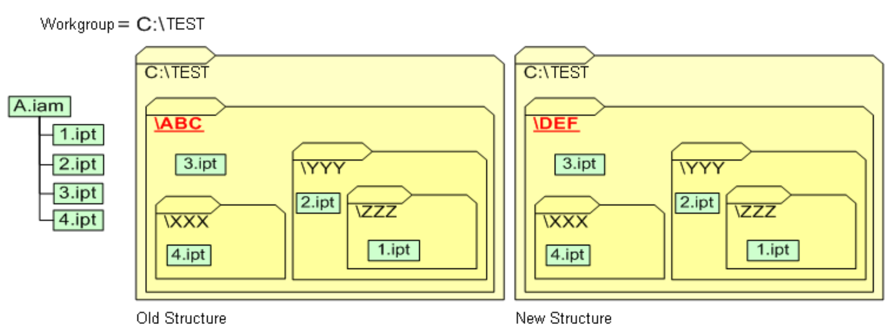
In non library locations, Autodesk Inventor appends the relative path stored in the reference to the project location and looks for a file at the resulting full path. If no file is found, the file name stored in the referenced file is appended to the project location folder path and Autodesk Inventor looks there.

Use Substitution Rules to Find Missing Files

In the Resolve Link dialog box, you can create a substitution rule to search for missing files. Click the More button (>>) to see the current substitution rule or modify it, if necessary, to create a substitution rule:

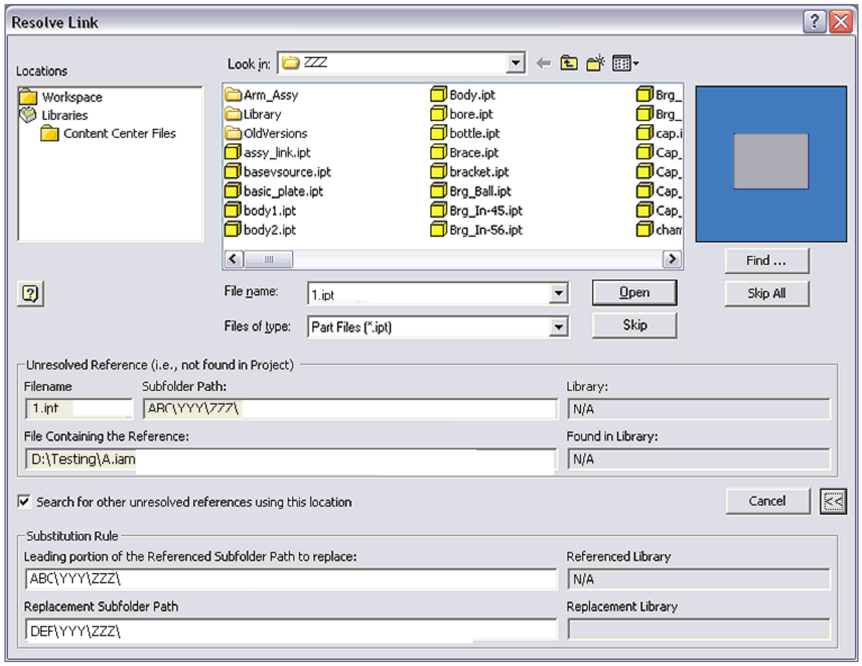
- Select the check box for Search for other unresolved references using this location.
- Specify a path location. You can edit the substitution rule, usually by deleting a tail portion of the path common to both the original and resolved path.
- Browse to the new library location. You might need to remove both the source and replacement folder paths.

In the following example, use Microsoft Windows Explorer to rename the folder from *ABC* to *DEF*.

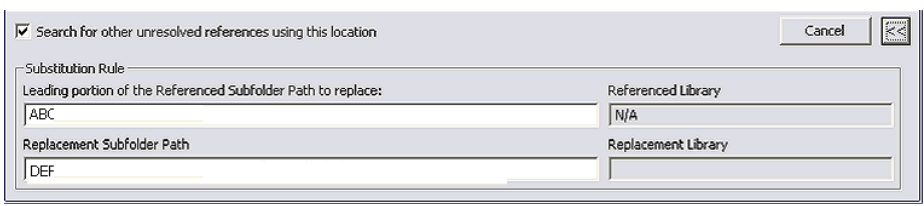


When the assembly *A.iam* is opened, *1.ipt* is expected to be in a location specified in the active project. Because it is in a renamed directory, you browse to the new directory.

After you browse to *1.ipt*, the dialog box shows the following:



Using the new subfolder path, select the check box, and edit both paths to remove the *\yyy\zzz* tail as shown in the following image to locate all parts.



When you click Open, you indicate that the path is correct. Autodesk Inventor attempts to find part *2.ipt,* and each of the other referenced parts, it automatically substitutes DEF for the ABC subfolder portion of the relative path.

If you renamed a library, but kept the file at the same relative path location, you remove both the leading portion of the referenced subfolder path to replace and the replacement subfolder path. In this case, the replacement library contains the new library name. If the original reference was to a library, the referenced library box contains its name. You can use these boxes to repair references to:

- A renamed library.
- Files moved from one library to another.
- Files moved from a library to an editable location.
- Files moved from an editable location to a library.

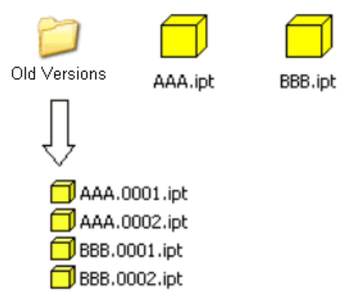
NOTE *Search for other unresolved references using this location* is automatically selected. Clear the check box to avoid creating a substitution rule.

Keeping Old File Versions

When you save a file, the previous version is moved to an automatically created folder called *OldVersions*.

The previous versions of an Autodesk Inventor file use this naming convention.

<file name>.<version>.<file type>



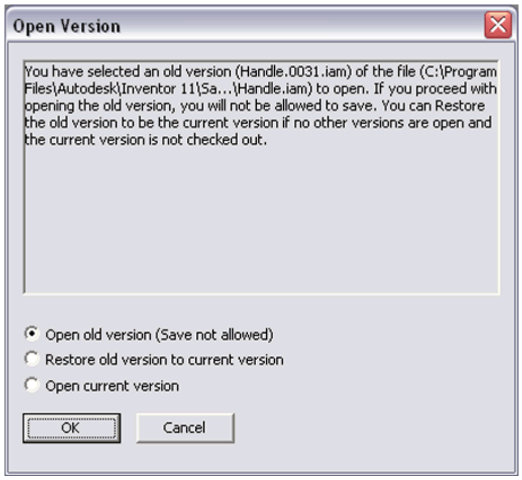
Other designers who opened the same file in an Autodesk Inventor work session continue to use the older version until they refresh the file, or close and reopen it.

You can always retrieve a previous version of a file. But it is important to note that while you can retrieve an old version of an assembly file, it may not include edits to all of its various referenced files. Autodesk Vault can restore any checked-in version of a referenced file.

To restore an old version, do not move the file from the *OldVersions* folder and rename it. Instead, restore the old version with Autodesk Inventor

TRY IT: Restore an old version of a file

1. Click File ➤ Open.
2. Browse to the file you want to restore from the *OldVersions* directory. The Open Version dialog box is displayed.



3. Select one of the options in the Open Version dialog box:

Open old version	Opens the old version of the file. Because the current file still exists, you cannot save the opened version. Use Save Copy As to save a copy.
Restore old version to current version	Restores the selected old version as the current version. The current version is not lost, but it becomes a file in the Old Versions folder.
Open current version	Opens the current version of the file.

4. Click OK.

   Every time you save a file, a new globally unique version ID (GUID) is assigned to the file. Each Autodesk Inventor session remembers the ID version it is accessing. If you try to reopen the file at the normal location and the version ID has changed, Autodesk Inventor searches the *OldVersions* folder and then opens the appropriate version of the file. The version of the opened file is used from then on to access additional information.

NOTE To save memory, Autodesk Inventor loads only the portion of a file that is needed for an operation. Additional information is loaded as needed. Therefore, do *not* delete an Autodesk Inventor file if there is a chance that someone else is using the file in an active Autodesk Inventor session.

You can set the number of versions to keep when you create or edit a project.

Each time a file is saved, the previous version is moved to its *OldVersions* folder. When the folder contains the maximum number of versions, and a newer file is moved into the folder, the oldest version is automatically purged from the *OldVersions* folder and deleted, if it is not open in an Autodesk Inventor session.

Moving, Copying, and Archiving Design Files

You can safely copy files to new locations. Do not replace existing files in the process.

If you use Microsoft Windows Explorer to rename files in a project, you break references to the files. It can also happen if you move files or folders or rename folders. When a file is not located, the Resolve Link dialog box automatically opens. Update the references to the new location.

NOTE Open the files immediately after you move them and resolve the links. You can avoid forgetting where you moved the files or confusing another user who tries to open the files without knowing the new location. Do not move files when anyone has the file open.

To avoid breaking links or losing data:

- In all cases, close all Autodesk Inventor sessions before you move or copy files.
- Back up your files. Create a zip file or use Pack and Go to package the entire data set so you can restore data if necessary.
- Use the Where Used tool in Design Assistant to determine where a file is used in other designs. For example, drawing *D.idw* might have an indirect reference to *P.ipt* (if, for example, it has a dimension or annotation to a face or edge or P). If you rename *P.ipt*, use the Where Used tool to locate *S.iam*, and then use Where Used again to find *D.idw*. Using this recursive

search, you preserve the references, along with annotations and dimensions in the indirect references.

- Use Design Assistant to move, copy, or rename files and repair the references from referencing files at the same time.

After you copy or move files, open them in Autodesk Inventor to verify that all of the links are correct before you give them to a vendor or other designer to use.

Zip Files

You can use zip files to move data, archive, or copy data sets for vendors. Avoid using nested folders in your project.

TRY IT: Move or copy Autodesk Inventor data using zip files

1. Copy each project location folder to a zip file, including subfolder paths and files.

   Do the same for the Styles Folder, Template Folder, and Content Center Files folder. If you share these folders across multiple projects, you may get files that are not used in your project.

2. Name each zip file with the location name of the project.
3. Include each zip file and the project in another zip file.

   NOTE If your project has thread or clearance data, also include the *Threads.xls* and *Clearance.xls* file from the Design Data folder. You may need to restore the file to retrieve thread data for the project, but be sure to preserve the original file so other projects that reference the file are not damaged.

4. The recipient unzips each project location folder into a new separate folder and resets the project so that each location goes to the new destination folder.

If you prefer, you can use a temporary root folder.

Temporary Root Folders

You can move data, archive, or copy data sets for vendors. When you have no nested folders in your project, you can easily archive or give data sets to vendors.

You can move data, archive, or copy data sets for vendors. When you have no nested folders in your project, you can easily archive or give data sets to vendors.

TRY IT: Move or copy data using a temporary root folder

1. Create a top-level (root) folder.
2. Create a subfolder for each project location (with the identical name of the location named in the project).
3. Copy the contents of each project location to its corresponding subfolder. If you can easily identify the library files you used, you can copy only those files.
4. Create a subfolder for each of the folder locations, such as .*Styles,* .*Content Center* and .*Templates*, and copy the corresponding folders from the source project. If you can easily identify the content center files used by your project, you can copy only those files.

   It is a good idea to copy the *Threads.xls* and *Clearance.xls* file from the Design Data folder location to the Styles folder as a precaution.
5. Copy your project file to the new top-level (root) folder.
6. Edit the project copy so that all locations are relative to the top-level (root) folder. Use the format .*subfolderName*\\.
7. Zip the contents of the root folder, specifying the Recursive and Preserve Subfolder Path options.
8. The recipient unzips to an empty folder and the project is ready to use.

   To reference your styles folder or merge the *Threads.xls* and *Clearance.xls* files, you may have to reset the Design Data folder location on the File tab of the Application Options dialog box.

You can use Pack and Go to package an Autodesk Inventor file and all or a selection of its referenced files under a single folder, even though the files are stored in multiple network locations. You can also include files that reference the selected Autodesk Inventor file. When you package a file, the files are copied to the specified location without changing the contents of any of the source files.

All referenced files must be resolvable using the current project (*.ipj*) file. If not, it is important to either open the correct *.ipj* file and make it current either in Autodesk Inventor or the standalone project editor, or to browse to it in the Pack and Go dialog box in the Project File field.

Pack and Go

Pack and Go is a tool that packages an Autodesk Inventor file and all of its referenced files in a single location. All files that reference the selected Autodesk Inventor file from a selected project or folder can also be included in the package.

Use Pack and Go to archive a file structure, copy a complete set of files while retaining links to referenced files, or isolate a group of files for design experimentation.

TRY IT: Move or copy Autodesk Inventor data using Pack and Go

1. Use Microsoft Windows Explorer or a Design Assistant session started outside Autodesk Inventor to find the file to package.
2. Select the file, right-click, and then select Pack and Go.
3. On the Pack and Go dialog box, specify the Destination Folder for the package.
4. Set the path and packaging options.
5. Make sure that the path in the Project File field identifies the appropriate project file for the selected file. If not, use the Browse button to locate it.
6. Click Search Now to search for referenced files. The total files found and required disk space are displayed when the search is complete.
7. Click Start to begin packaging the files. As the files are packaged, the status is shown in the Progress box. A new project file and a log file are created in the destination folder reflecting the structure chosen in the packaging options.

   The log file is replaced each time you package an Autodesk Inventor file to the same destination.

If the Missing File dialog box is displayed, click the Set Project button. Select the project to use for resolving referenced file locations. Click Open, and then click Start to begin the search.

You can click Cancel on the Find Missing File dialog box to cancel the action and display the Pack and Go dialog box without referenced files.

You can use Design Assistant to copy an entire assembly file (*.iam*), including the referenced drawing file (*.idw*). Close all Autodesk Inventor files and use Windows Explorer to open the Design Assistant.

Design Assistant Manager

Using Design Assistant, the Design Assistant Manager can maintain links between Autodesk Inventor files. You can change the file relationships four ways: rename files, revise files, replace files, and create product configurations.

NOTE You cannot manage links for released files and read-only files.

TRY IT: Copy assembly and referenced drawing files using Design Assistant

1. In Windows Explorer, right-click the *.iam* file (for example, *test1.iam*), and then click Design Assistant.
2. In the left panel, click the Manage button (if it is not already selected).
3. Highlight the entry for the *.iam* file (*test1.iam*) in the upper pane.
4. Select the Drawing Files check box in the bottom right pane, and then select Find Files.

   A message box is displayed, indicating if files were found. The list of files found is displayed in the lower part of the right pane.
5. Right-click in the Action column for the *.iam* file (*test1.iam*) in the upper pane and then click Copy.
6. Right-click in the Name column for the *.iam* file (*test1.iam*) in the upper pane, and then click Change Name. Enter the new name for the Assembly (for example, *test2.iam*). Enter a new path to specify a new location, if desired.
7. Repeat steps 5 and 6 for the *.idw* file. Use the same name you used for the assembly file, but use an extension of *.idw* (for example, *test2.idw*).
8. On the File menu, click Save to save the settings.

All changes are saved and new files (*test2.iam* and *test2.idw*) are created.

NOTE The newly created (or copied) drawing file (*test2.idw)* is referenced only to the newly created (or copied) assembly file (*test2.iam)*. All changes made in the original assembly file (*test1.iam*) is reflected only in the copied *test2.idw* that references it.

Occasionally, annotations for a referenced file of a subassembly may not be visible in a drawing view after you use Design Assistant to move or copy. If so, open the drawing file in Autodesk Inventor and use the Resolve Link dialog box to restore the annotations.

Move and Copy Files Between Projects

You can temporarily rename the original files or move them out of the project locations (so they are not found). Then open a top-level assembly or drawing and use the Resolve Link dialog box to change the references to copies of the referenced files. After the referencing files are saved, the original files (from which the copies were made) can be restored and renamed to their original names.

If you copy files from one project to another, remember:

- If the file you copy to another project has a library reference, you must define the same library in the destination project. It can be to the same UNC location as the original.
- If the file you copy to another project has a non library reference, it resolves to a file in the editable locations in the destination project. You can copy the entire contents of a workspace (including subfolders) to another project workspace. The files resolve to the copies rather than to the source files.
- You can copy an entire reference hierarchy in the same way, but preserve the original subfolder structure from the project root location for the copied files.

To copy a workspace folder to another location, follow these guidelines:

- The folders must include the project (*.ipj*).
- The project must contain only one editable location.
- All library locations must be accessible from the destination location (the folder where the copies reside).

You can move or copy the folder containing the project file, browse to and activate the copied project file in the project editor, and then use the design files immediately after you copy them.

If one or more of the previous conditions are not met, edit the destination project (*.ipj*) to specify the new paths for each of the copied folders.

Consider using Pack and Go when copying an entire project. You can create a zipped copy on a CD-ROM, for example, and send it to a customer, vendor, or client. You can include only those library files that are referenced instead of an entire library, if appropriate. Pack and Go can also create a read-only library copy so you can continue to change the project data, but the recipient can use the copy as a library.

Deleting Files

Deleting a file permanently removes it from your system. Because there is no recovery, follow these guidelines:

- Verify that the file or files you intend to delete are not referenced or open in Autodesk Inventor. If the file is open when you delete it, additional data cannot be retrieved from the file and the open file cannot be saved. The person using the file permanently loses any in-memory edits that were not saved.
- Back up your data before you delete the file. Copy the file to another location or create a zip file so that you can recover it if necessary.
- Use The Where Used tool in Design Assistant to find out where the file is being referenced by other Autodesk Inventor files (including drawings, parts, assemblies, and presentations).

When you are certain it is safe to delete the file, use Microsoft Windows Explorer to delete the file.

TRY IT: Delete a file using Windows Explorer

1. In Windows Explorer, browse to the folder where the file is located.
2. Right-click the file name, and then select Delete.
3. Click Yes to confirm deletion.

The deleted file is temporarily placed in the Recycle bin and can be restored to its original location if necessary. When you empty your Recycle bin, the file is permanently lost.

Changing File Structure

Projects often grow over time and the file structure must change to accommodate the complexity. You can more easily change the file structure if you plan it before you start the project, laying it out so that the data is portable.

Using the project option *Using Unique File Names* = *Yes*, you can restructure folders or move data files without breaking references.

NOTE Before you open files after moving them, click Tools ➤ Application Options. On the Save tab, select the Reference Resolution Changes check box. Open all files and save them to update the references to the new location.

TRY IT: Use Windows Explorer to restructure files

1. Define the subfolder structure you need before you create or edit any files. Create the subfolders immediately after creating a project so that new saved files are located in the correct subfolder.
2. Create subfolders under the project home folder for personal workspace, such as:
   - Create a subfolder named *Components* where you store shared subcomponents, including assemblies.
   - Create a subfolder named *SubAssemblies* and under it, a subfolder for each main subassembly. Put the assemblies and parts unique to each in the subfolder.
   - For custom content library parts such as structural steel shapes, create a subfolder named for it (for example, Structural Steel) and save the generated parts in it.
   - Create folders for drawings and presentations.
   - Place the top-level (main) assembly or assemblies in the root workspace folder.

- If needed, create a subfolder named *Tube_Pipe_Content*. Create a library named *Tube_Pipe Content*. Configure the tube and pipe library to place standard tube and pipe components in that library.

3 Add the paths to the new subfolders of the project as Frequently Used Subfolders. They are listed in the Locations box of the file Open dialog box.

4 For safekeeping, make a copy of all of the data files, before you move them to a new directory or delete old folders. After they safely move to a new folder, delete the copy.

TRY IT: Use the Resolve Link dialog box to restructure files

1. Create needed subfolders.
2. Open the referencing files in Autodesk Inventor.
3. In the Resolve Link dialog box, navigate to the new locations to restore links.
4. Save the referencing file with the new locations.

Within an Autodesk Inventor session, the Resolve Link dialog retains the information you specify regarding the source and destination folder paths and library names of files that were moved.

If you open other files with broken references to the same folders/libraries, Autodesk Inventor remembers the folder/library name and tries that location before opening the Resolve Link dialog box again. By default, a map is created with the complete folder path.

If you move a folder that has many subfolders and files, you can edit the fields in the dialog so that the source and destination paths of the moved folder are shown. Using a substitution rule, Autodesk Inventor attempts to resolve the file references without opening the Resolve Link dialog box for each of the subfolders below it.

About Autodesk Vault

Autodesk Vault is a workgroup data management system integrated with Autodesk Inventor. It enables fast, accurate sharing of design data across a project team. The vault is a file management and version control system for all engineering and related data, providing design team members with a central and secure collaborative environment.

Autodesk Vault is the preferred data management system for Autodesk Inventor. Its capabilities extend beyond the data management scope of projects.

After you install Autodesk Vault, use the Project Wizard to create a vault project. Specify a personal workspace where you create and edit files. In addition, you also specify the vault server, its name and the virtual folder stored on the server. (These values are set using Autodesk Vault.)

For more information about using Autodesk Vault, see *Managing Your Data*. The guide is installed in *Program Files* > *Autodesk* > *Vault 2009* > *Help*.

Index

D

E

F

G

H

I

L

M

N

S

T

V

W

Z