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# What's new in Inventor Fusion?

# 1

## Inventor Fusion 2012

### Sketch

Persistent sketch constraints (page 116) and dimensions (page 121).

- Add and remove constraints on a sketch.
- Constraints display on the sketch.
- Turn off the display of constraints with a status bar button.
- Pin dimensions and create new dimensions

Sketches burn thru the model making it easier to work with sketch geometry "inside" solids.

Reorient a sketch (page 108) to change its origin and rotation.

Place an image in a sketch.

- Control scale, rotation, and position on placement.
- Knock out the background for images without alpha transparency support.
- Toggle image overall opaqueness and calibrate the image by selecting two points to set a precise model unit size.
- If SketchBook Designer is installed on the same machine, edit the image in sketch book and return to Fusion to see the changes (curves not supported).
- Update the image manually in case the image was modified in an external application.

### Modeling

Validate command (page 269) for imported solids and surfaces.

- Repair common surface errors automatically.
- Set a tolerance and the level of check to perform.

Surface groups in the browser (page 267) now show an "Unstitched" group to more clearly represent the loose surfaces in a group.

Freeform modeling (page 294) commands. Formerly know as Alias Design for Inventor.

- Edit edge
- Symmetry definition
- Straighten edge
- Shape mode and interpolation mode
- Toggle symmetry and edge locking from status bar buttons
- Toggle curvature comb display for modified edges

Partial reblend support.

- When editing a model with blends, some blends fail. Fusion fails only the blends with issues> Other valid blends continue to update. You can finish the command allowing only some of the original blends to be maintained.

When creating a surface, you can create a component to contain the surface.

You can move and rotate sketches (page 108) using the modeling Move command.

When creating a circular array of components, control component orientation.

When creating a tangent work plane, align it to a face or edge, or define a set angle from the pick location.

Scale command (page 204) that can scale bodies, surfaces, sketches, and components.

Primitive commands (page 163) including box, sphere, torus, and cylinder.

Sculpt command (page 219) to modify solids using surface objects.

### Other

In canvas glyph appearance is more consistent with Autodesk Inventor.

Assemble command supports work points.

New "recover" command in the application menu allows you to recover data from Fusion files if they crash on open, like the Audit command in AutoCAD.

ShowMotion dialog box and ShowMotion data storage are improved.

Copy paste of surfaces and components uses a triad for precision placement.

Marking menu and context menu are improved.

Ribbon UI is improved.

### Companion Add-ins

Integration of Moldflow and Fusion (page 481)

Integration of Algor Simulation and Fusion (page 476)

Integration of AutoCAD and Fusion (page 475)

Integration of Inventor and Fusion (page 480)

# Essential videos

# 2

User Interface and Navigation

Creating Sketches

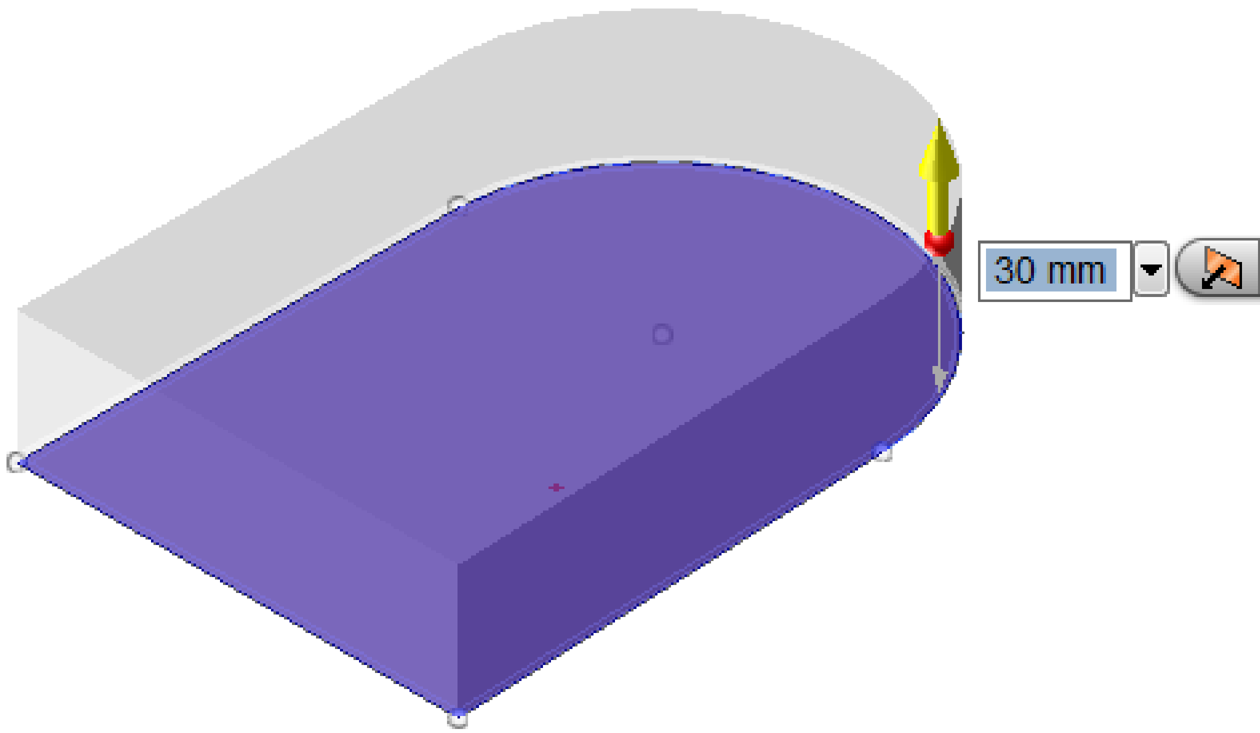
Creating Objects

Modifying Objects

Assemble Objects

# 3 Inventor Fusion User Interface

The user interface in Inventor Fusion comprises standard Windows features, such as the ribbon, application menu, and context menus. Commands in Fusion provide controls in the graphics window rather than dialog boxes, so you can focus on the modeling task.



# The Ribbon

## Display and Organize the Ribbon

When you create or open a file, the ribbon displays across the top of the application window. It provides a compact palette of all tools necessary to create your model.

You can minimize the display of the ribbon.

When a command is active, a temporary ribbon pane displays across the top of the application window. In this pane, you can input command options, range limits, and other settings.

**Ribbon Tabs and Panels**

The ribbon is organized into tabs, labeled by task. Each tab contains a series of panels.

Some ribbon panels display a drop-down arrow that indicates there are additional commands related to that panel. To access the additional commands, click the drop-down arrow

You can display or hide panels. Right-click anywhere inside the ribbon, and then click or clear the name of a panel.

**Float Panels**

You can drag a panel off a ribbon tab and into the drawing area, or onto another monitor. That panel floats where you place it and remains open until you return it to the ribbon, even if you switch ribbon tabs.

**Expand Panels**

An arrow at the bottom of a panel title indicates that you can expand the panel to display additional commands. By default, an expanded panel closes automatically when you click another panel. To keep a panel expanded, click the push pin icon in the bottom-left corner of the expanded panel.

**Minimize the ribbon using the Minimize button**

1. On the ribbon, to the right of the ribbon tabs, click the minimize button.

The minimize behavior cycles through options to minimize the ribbon:

- **Minimize to Tabs** Only tab titles display.
- **Minimize to Panels** Only tab and panel titles display.

- **Show Full Ribbon** Tabs and full panels display, including controls.

**Other methods to minimize the ribbon**

- Right-click the ribbon tab bar, click Minimize, and then click one of the minimize options.
- Double-click the name of the active ribbon tab.

**Turn off the display of a panel**

Right-click anywhere inside the ribbon. Under Panels, click or clear the name of a panel.

**Switch the display of panel titles**

Right-click the ribbon tab bar, and click Show Panel Titles.

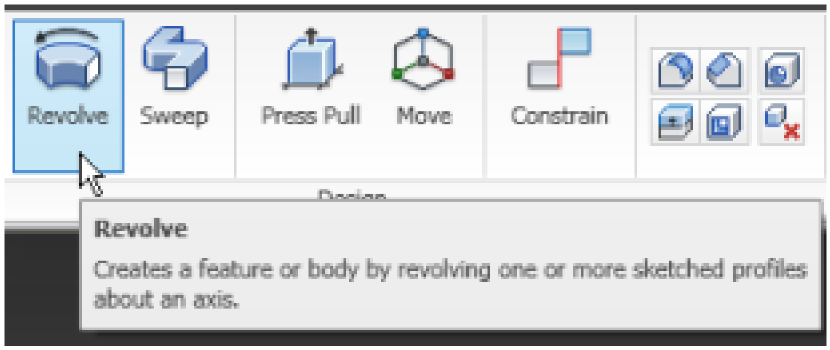
## Customize the Ribbon

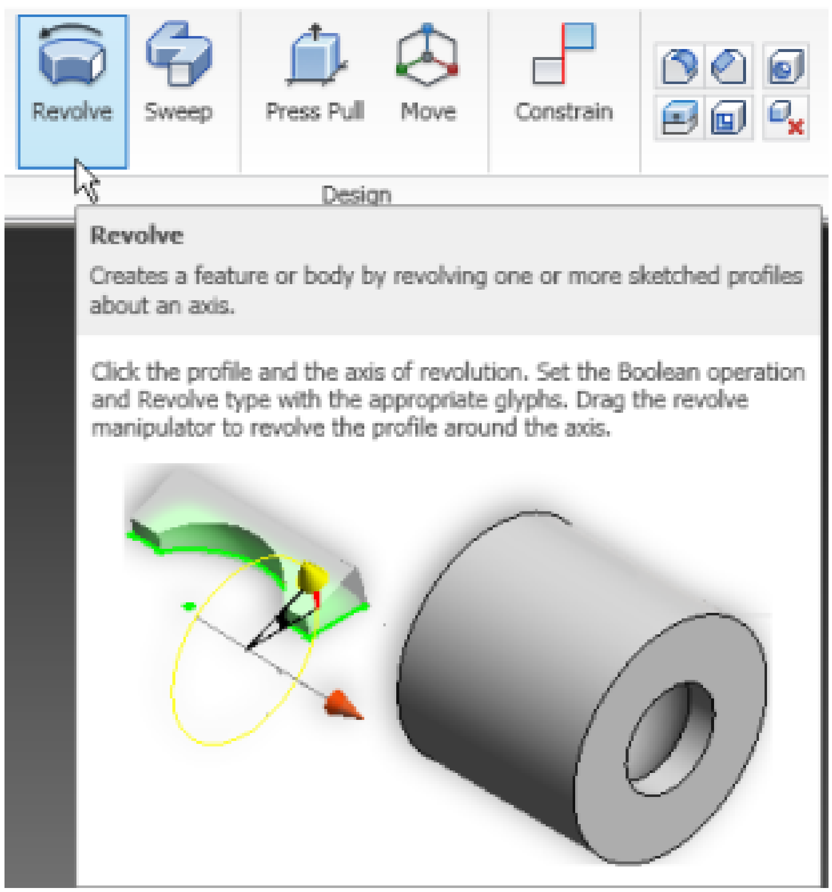
- Change the order of ribbon tabs. Click the tab you want to move, drag it to the appropriate position, and release.
- Change the order of ribbon panels on each tab. Click the panel to move, drag it to the appropriate position, and release.
- Hide panels on a tab. Right-click a tab, and choose which panels to display.
- Set the ribbon back to the default configuration. Right-click anywhere on the ribbon, and select Reset Ribbon.
- Change the color schemes of the ribbon. Right-click anywhere on the ribbon, and select Theme. There are three color themes:
  - Dark
  - Light
  - Black

## Enhanced tooltips

Most of the ribbon commands have enhanced (progressive) tooltips, which display information for interaction with commands. Initially, the name of the command and a short description of the command displays. If you continue to pause the cursor, the tooltip expands to display additional information.

Within some of the tooltips, a ToolClip provides an animation of a quick overview or sample usage of a ribbon command.



# Browser

The browser presents an organized view of the data in your design in a tree format, like the navigation pane in Windows Explorer. Context menus have tools and commands for the selected object type. Some are specific to the browser and some are also available in the graphics window.

Objects are cross-highlighted in the browser and graphics window. Selecting an object in the browser also highlights it in the graphics window, and selecting an object in the graphics window highlights it in the browser. Cross-highlighting provides visual feedback and simplifies selecting a particular object so you can edit it in the other environment.

### Browser items and controls

Some items have links to sections and topics with more information.

1. Turn the Favorites (page 13) browser on and off. The Favorites browser displays at the bottom of the browser pane.
2. Click the light bulb to turn visibility of an object on or off.
3. Toggle features On and Off.
4. The Named Views folder has default standard views. You can select New Named View from the context menu to save the current model orientation and zoom.
5. The Origin folder has the X-Y-Z coordinate system. Components and created features are positioned relative to the parent coordinate system.
6. Annotation Planes are created for dimensions that document the model.
7. A blue background indicates the active component.
8. Constraints position components relative to each other.
9. Child Components (page 372) can contain features and other components. This hierarchy organizes the model by grouping related features and components together.
10. A component can have one solid body (page 163) . A solid body usually has multiple features, and there can be gaps between features.
11. A component can have multiple surfaces (page 221). A surface can be independent, and multiple surfaces can be stitched together to form a quilt.
12. Sketches (page 104) contain geometry for creating features.

13 There are two types of features, created and placed. Created features, such as extrude and revolve, are new entities in the graphics window. Placed features, such as fillets and holes, modify existing features.

14 Objects can be mirrored and patterned (page 326) to organize the browser and capture information on how objects are related. For example, if a component has a row of holes, a pattern locates the holes and groups them in the browser.

15 Turn the Information Panel on and off to display information about a browser object.

16 The Information Panel displays when you pause the cursor over a browser object. Some objects, such as the Sketches folder, do not have information panels.

### Favorites

Click on Toggle Favorites to display the Favorites browser at the bottom of the model browser. It is minimized by default and you drag the arrow to increase the browser height.

You can drag copies of browser items, such as components and features, to Favorites. You can add the same item to multiple sub-folders to improve organization. For example, you can have separate folders for features, for components, and features for a specific component.

You can search the Favorites browser using tags. A search results folder is created that automatically updates as items are added or remove. There are two types of predefined tags- feature names, such as Fillet, and feature dimensions, such as Fillet Radius. Click on the Expand button to display the feature dimension fields. You can also create User Tags (page 455) to identify items.

### Browser Context Menus

The browser has tools for common tasks on the context menus. The context menus vary depending on the type of node.

### Light Bulb

**Isolate Component** Hides all other components.

**Unisolate Component** Is added to the context menu on the top browser node when a child component is isolated. It makes all components visible again.

### Component Context Menus

**New Component** Creates a child component. Child components are typically used for assemblies. For example, a cover is usually a child component.

**Activate Component** Sets which component you are working on. Features are created in the active component. If you accidentally create a feature in the wrong component, you can cut and paste it into the correct one.

**Save As** Creates a separate document for the selected child component.

**Make Independent** Breaks the link between a copied component and the original.

**Grounded** Fixes a component in its current location. For example, an imported assembly comes in as separate components. The components are not constrained, so you can drag them out of position. To keep the components in their original locations, select all of the components and set them to Grounded.

**Insert Component** Imports a file and adds it as a child component.

**Material (page 413)** Sets the material type and the visual appearance. Some materials, such as rubber and copper, have a visual style. You can override the visual style by changing the color or texture. If you change the visual appearance, Remove Appearance Override is added to the context menu.

**Dissolve Features** removes the individual feature nodes in the component but leaves the geometry.

### Solids and Surfaces context menus

**Validate (page 269)** Checks the solid or surfaces for issues and tries to repair bad surfaces.

**Material (page 413)** Sets the material type and the visual appearance. Some materials, such as rubber and copper, have a visual style. You can override the visual style by changing the color or texture. If you change the visual appearance, Remove Appearance Override is added to the context menu.

**New Group** (Surfaces Only) Creates a sub folder for organizing surfaces.

**Stitch (page 259)** (Surfaces Only) Starts the stitch command to combine individual surfaces into a quilt.

**Unstitch** (Surfaces Only) Starts the unstitch command to separate a quilt into the individual surfaces.

**Reverse Normal (page 267)** (Surfaces Only) Changes which side of the surface is positive. For example, the Z-axis of an XY plane is positive in one direction and negative in the other.

**Thicken (page 213)** (Surfaces Only) Offsets a surface and adds side faces to create a solid.

### Sketches

**Slice Graphics** Cuts the model graphics across the sketch plane and displays the internal components. Right-click on the sketch and select Slice Graphics again to display the entire model again.

### Assembly Constraints

**Suppress** Temporarily turns off the constraint . You can move the components. They return to the original position when the constraint is unsuppressed.

### Features

**Recognize As Revolve** Changes the feature type to Revolve. This works on features such as a cylindrical extrude, but does not work on a rectangular extrude.

**Dissolve**Removes the feature node from the browser but leaves the geometry in the model.

# Tools in the Graphics Window

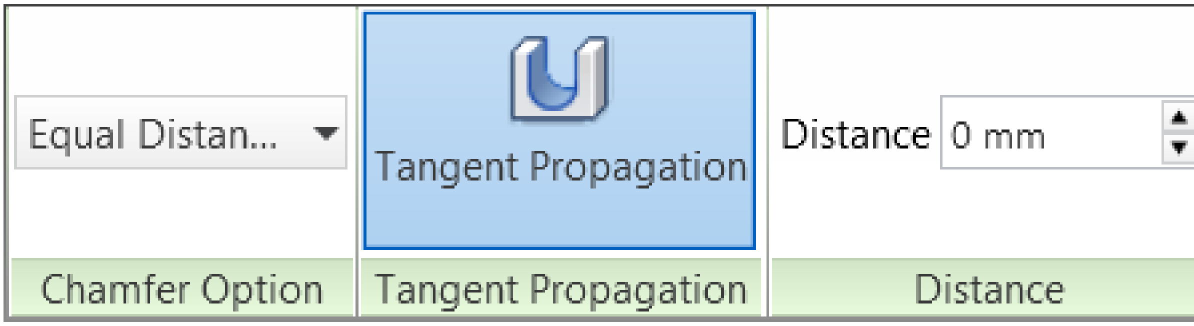
## Heads-Up Display

Inventor Fusion commands use a Heads-Up Display (HUD). The HUD provides controls in the graphics window instead of on dialog boxes. There are five types of controls:

- Command Ribbon
- Value Entry Fields
- Glyphs
- Mini-toolbars
- Manipulators

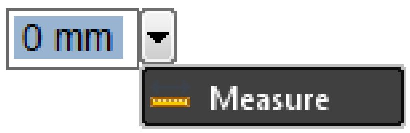
### Command Ribbon

Many commands display a custom ribbon that includes additional options. The command ribbon replaces the ribbon bar (page 8) while the command is active.




**Value Entry Fields**

The value entry fields provide several methods for setting the value for a command or entity.

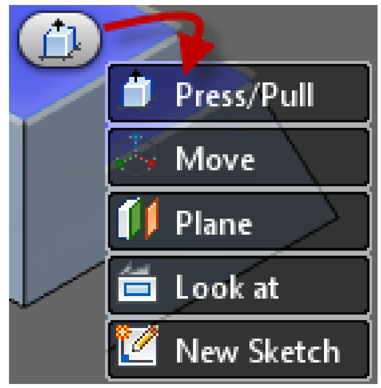



- Enter a value directly.
- Drag a manipulator to set the value.
- Measure other geometry.
- Enter a simple equation to calculate the value.

### Glyphs

Glyphs display next to the cursor to provide access to tools and commands. Pause the cursor over the glyph to display a list of options. The options display until you make a selection or move the cursor away from the glyph.

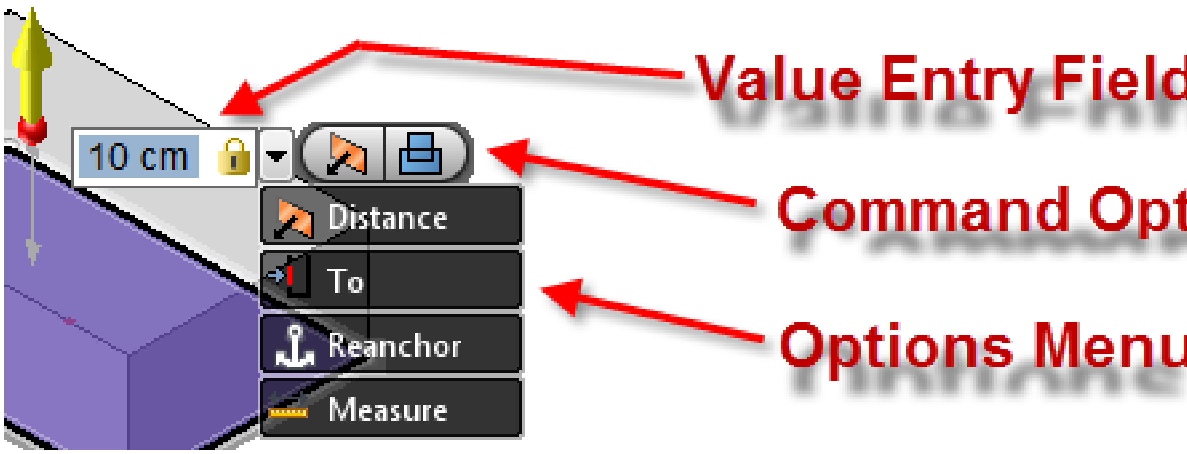

The Selection Tool glyph provides options for selecting geometry and objects.




The Command glyph provides a list of commands.

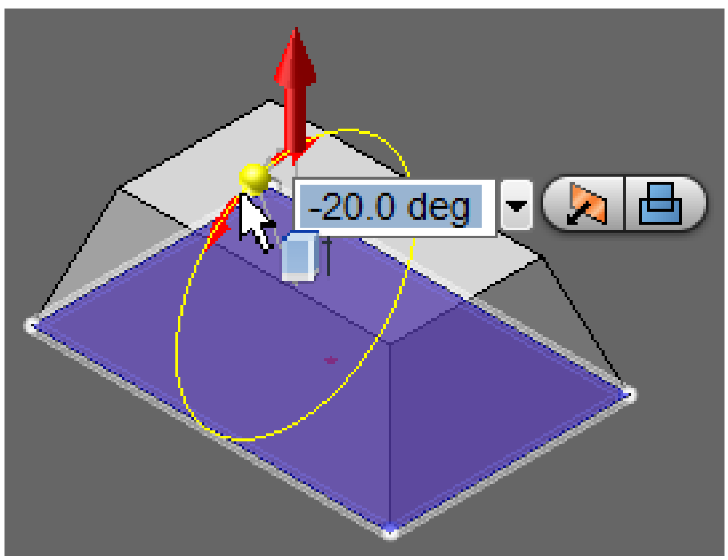
### Mini-toolbars

Mini-toolbars can have value entry fields, glyphs, and buttons. Most mini-toolbars display near the cursor when a command is active. Other mini-toolbars, such as the sketch control buttons, display in the same location.

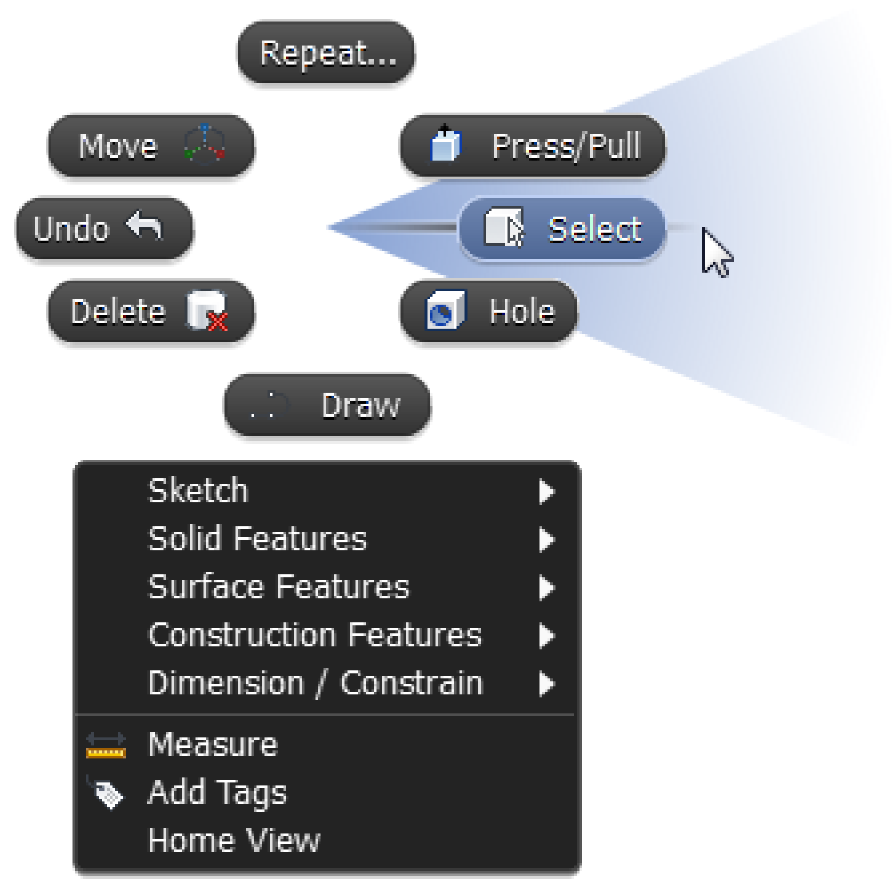



**Manipulators**

Manipulators are 3D tools that display while a command is active. Drag the manipulator to change a value or location. If the command has multiple manipulators, click on a manipulator to activate it or press Tab to cycle through them. Sketch and Feature commands use arrow, sphere, and ring manipulators; and Move uses a triad manipulator.



## Marking Menu




A marking menu is a radial menu of commonly used commands, along with a context menu. Each item, or node, is associated with a wedge of the circle. The node displays the command name and icon.

As you move the cursor from the center of the marking menu towards a node, the wedge displays on the screen. You can click anywhere in the wedge to select the item.

Close the marking menu by clicking in the center, or move the cursor away from the marking menu until the wedge highlighting disappears, and then click.

**NOTE** If you press **Esc** to close the marking menu when a command is active, you also close that command.

### Marking Menu Nodes

The marking menu consists of eight wedges. Each wedge represents a command/operation. These commands are the seven frequently used commands, plus a context menu that contains additional commands. The eight node positions use the same names as a compass


The default commands in the marking menu are:

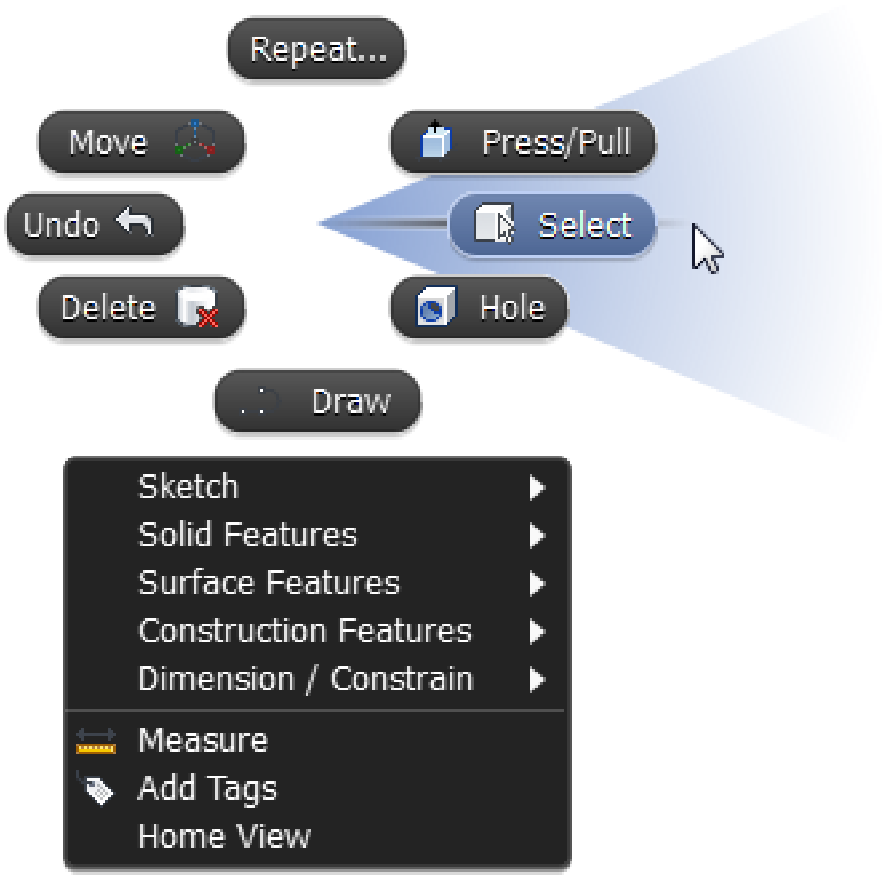
- **North** Repeat last command
- **Northeast** Press Pull
- **East** OK
- **Southeast** Hole
- **South** Draw
- **Southwest** Delete
- **West** Cancel
- **Northwest** Move

Most nodes are consistent to simplify selecting common items. Some marking menu nodes update based on the active environment and selection. For example, the North node is always Repeat while the East node changes to OK when a command is active.

### Use the marking menu

The two basic ways to select items from the marking menu are menu mode and marking mode.

**Menu mode** is the standard context menu behavior. Right-click in the graphics window and select a marking menu node or a context menu item.




1 Right-click in the graphics window. A radial menu displays, centered on the current cursor position.

2 Move the cursor towards a menu item. A wedge displays showing which item is active.

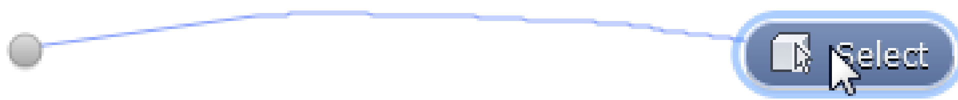
3 Click anywhere in the wedge to select the item.

**NOTE** You can use the Up and Down arrow keys on your keyboard to navigate the marking menu selections. Each node highlights as you move around the circle, but a wedge does not display. Press Enter to select the item.

4 To cancel the marking menu, perform one:
- Click in the center of the circle.

- Move the cursor away from the marking menu until the wedge highlighting disappears, and then click.
- Press **Esc**.

After you become familiar with the marking menu, **Marking mode**, also called gesture behavior, is an efficient way to select commands.




1. Press the right mouse button and immediately drag the cursor towards an item. A trail displays showing you cursor path, but the marking menu does not display.
2. Release the right mouse button to select the item. The node temporarily displays to indicate which item was selected.
3. If you pause the cursor, the marking menu displays with a highlighted wedge. Release the right mouse button to select the item.

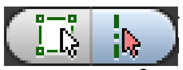
## Glyphs

Glyphs, or symbols, often display next to the cursor in the graphics window. Glyphs indicate that you can select a mode of operation, or that certain tools are available for use.

Pause over a glyph to display a list of options. Drag the cursor to the appropriate option and click to select it. If the cursor moves away from a glyph, the options are hidden.

**Frequently used glyphs**

| Glyph Icon | Glyph Title | Description |
|---|---|---|
|  | Selection Tool | Displays when you pause the cursor over an object. Provides options for selecting related geometry. For more details, see the Selection Tool (page 38) topic. |

| Glyph Icon | Glyph Title | Description |
|---|---|---|
|  | Command Option | Displays in the mini-toolbar. Provides options for a command. For example, lists directions for Extrude. |
|  | Context menu | Displays when an object is selected. Provides a list of commonly used commands. |
|  | Selections | Displays when a command is active. Represents the selections required in the command. Click a glyph to change the active selection.<br>**NOTE** The arrow changes from red to white when the selection is complete. In this image of the Revolve glyphs, the profile is selected, but not the axis. |

## Manipulators

Manipulators are 3D tools that display while a command is active. Drag the manipulator to change a value or location.

The active manipulator is yellow and inactive manipulators are red. Click on a manipulator to activate it, or press Tab to cycle through them.

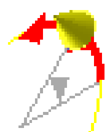
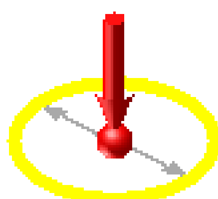
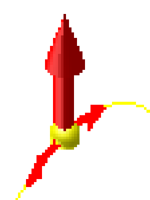
Sketch and Feature commands use arrow, sphere, and ring manipulators. Move uses a triad manipulator.

When using manipulators, it is not necessary to keep your cursor exactly on the 3D arrow. You can drag anywhere in the graphics window. Many manipulators can also snap to other geometry on your model. While a manipulator is active, you can move your cursor over other geometry to display prompts for snapping.

### Feature and Model Geometry Manipulators

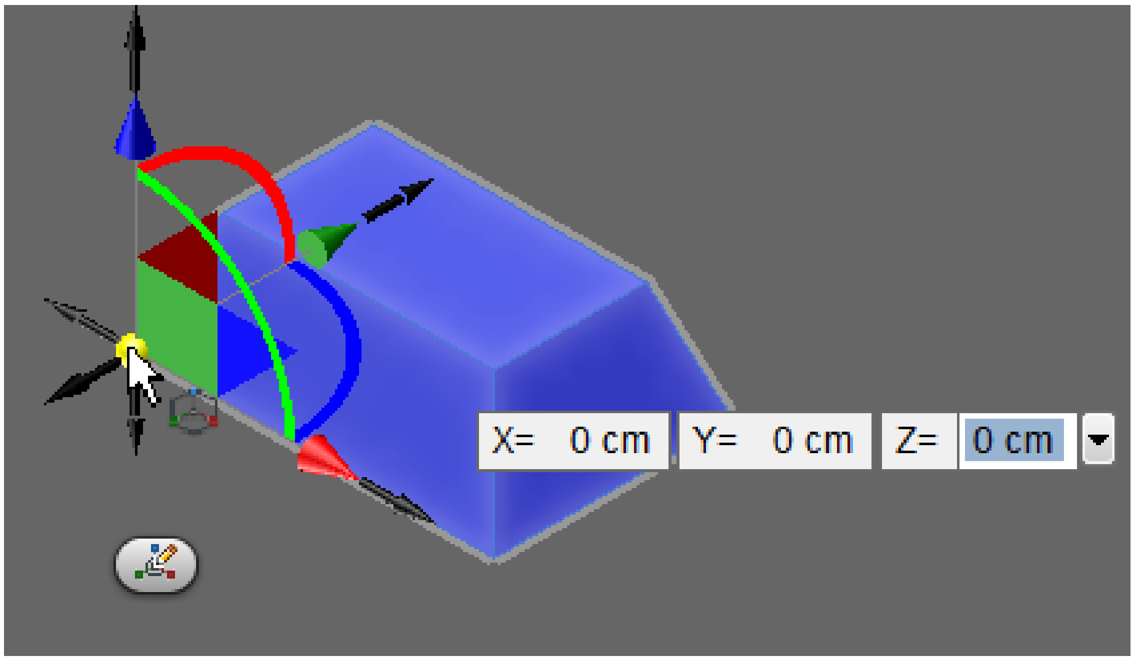
Use manipulators to create features and modify model geometry interactively. The active manipulator is yellow and inactive manipulators are red. Click on a manipulator to activate it, or press Tab to cycle through them.

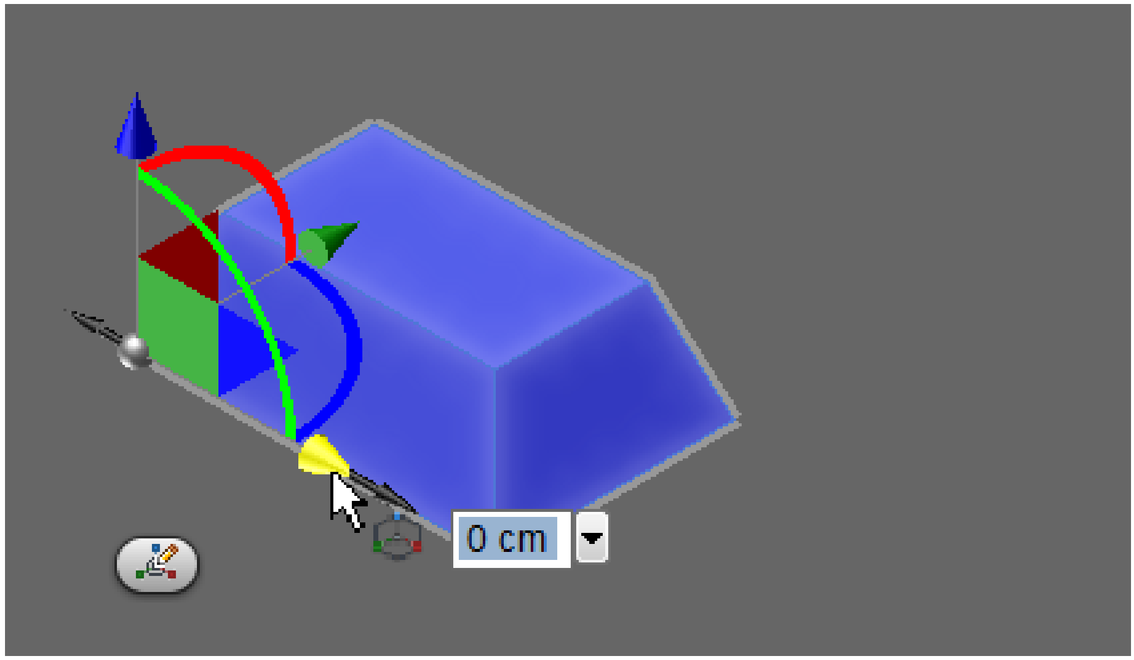
There are four basic manipulator types. The purpose of each manipulator can vary by command:

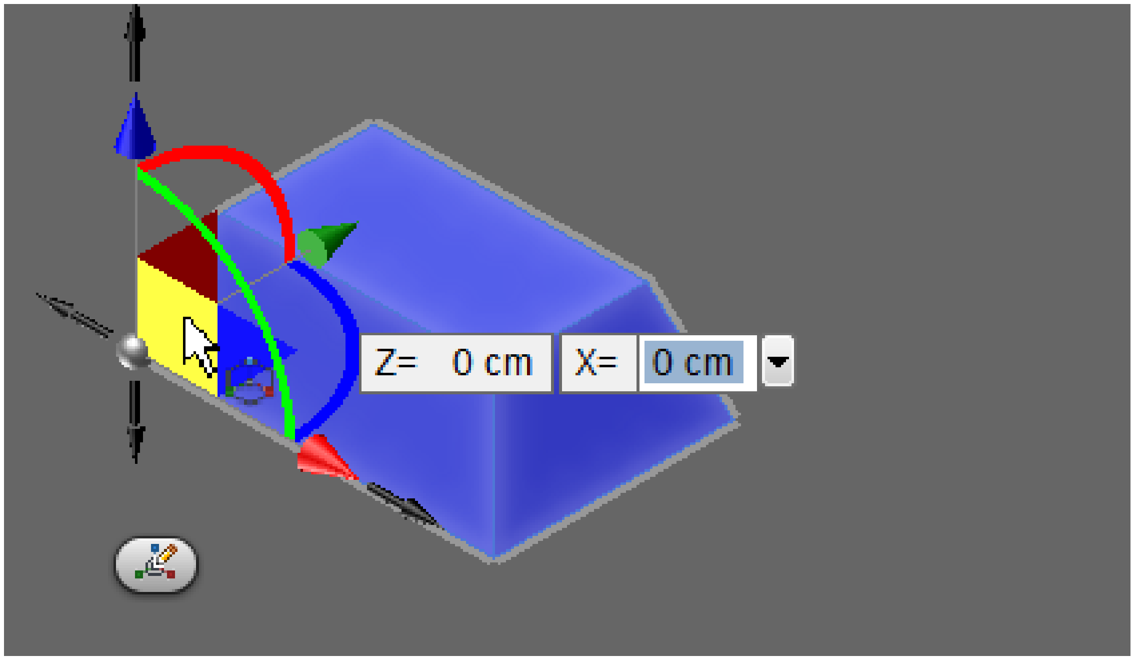
| **Manipulator Image** | **Type** | **Description** |
|---|---|---|
|  | Linear Arrow | Sets the distance and direction for a feature. |
|  | Radial Arrow | Sets the rotation angle for Revolve. |
|  | | Sets the diameter for hole features. |
|  | ere | In the Hole command, sets the hole center location. In the Extrude command, sets the taper angle. In the Sweep command, sweeps the selected profile along the path. |

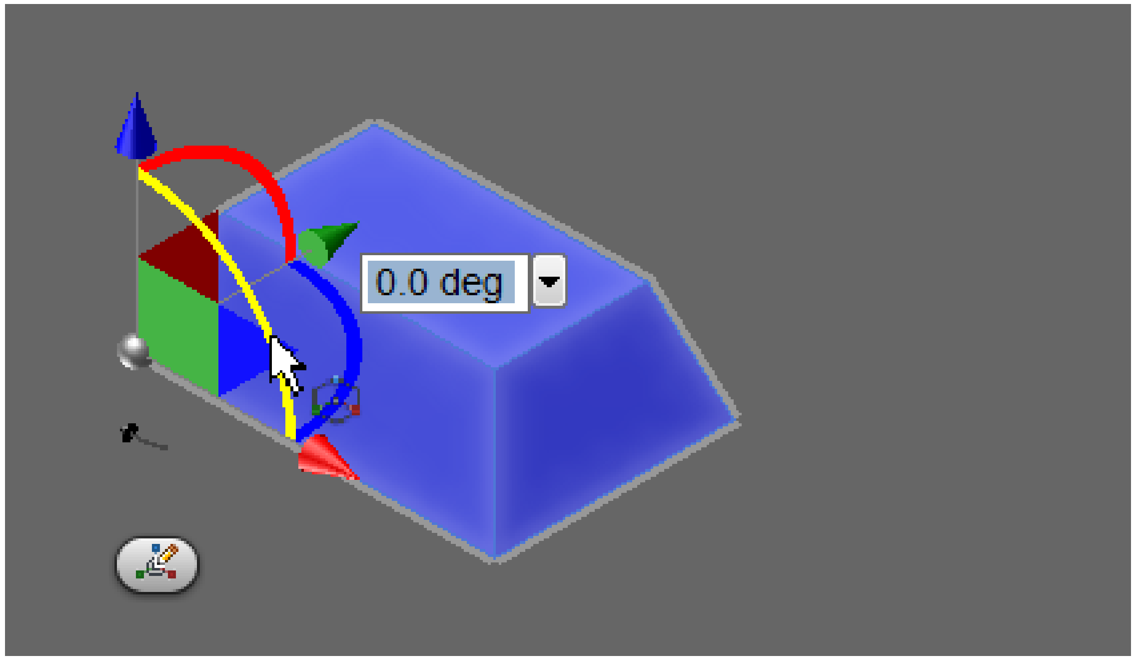
**Move Manipulator**

The triad provides constrained translation or rotation for an object. The triad has arrow, plane, arc, and sphere manipulators. You can translate using the arrows, planes, or sphere; and rotate using the arcs.




0 cm


Z= 0 cm
X= 0 cm




### Use Manipulators

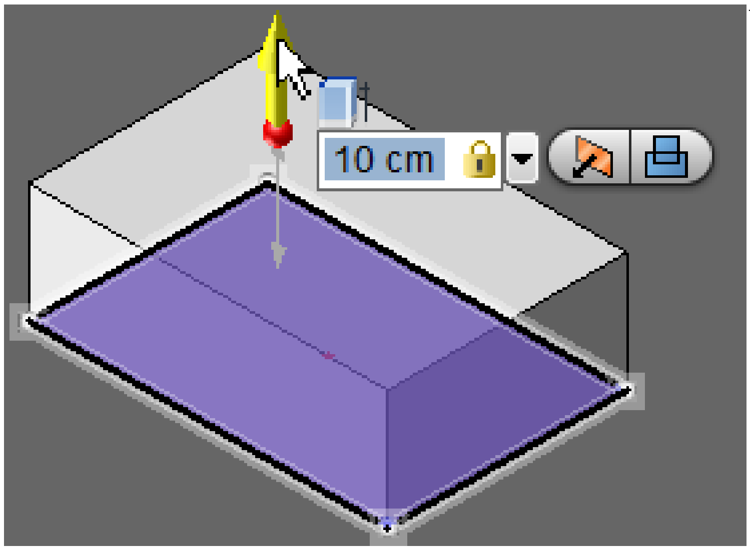
Drag the manipulator to complete (or preview) the operation. The value that is set by dragging the manipulator can also be set in the ribbon or heads-up-display (HUD).

The manipulator has priority when it is active. You can click and drag anywhere in the graphics window, so it is not necessary to position the cursor exactly over the manipulator.

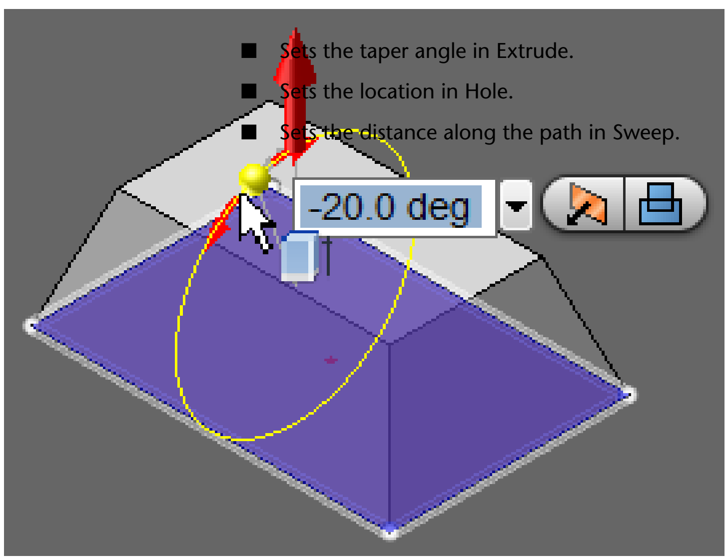
The active manipulator is yellow and inactive manipulators are red. Any manipulators that are not valid for a selection are disabled and display in gray. Click on a manipulator to activate it, or press Tab to cycle through them.

You can measure (page 54) geometry using the drop-down next to any HUD. You can enter simple arithmetic expressions and mix units when you enter values into the HUD.

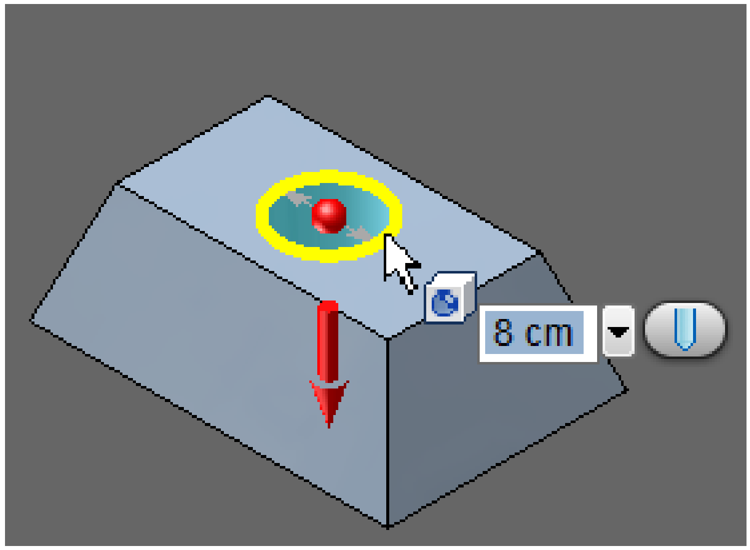
Many manipulators can also snap to other geometry on your model. Move your cursor over other geometry to display a prompt for snapping. When you snap a manipulator to other geometry, the manipulator becomes blue to indicate it is constrained.

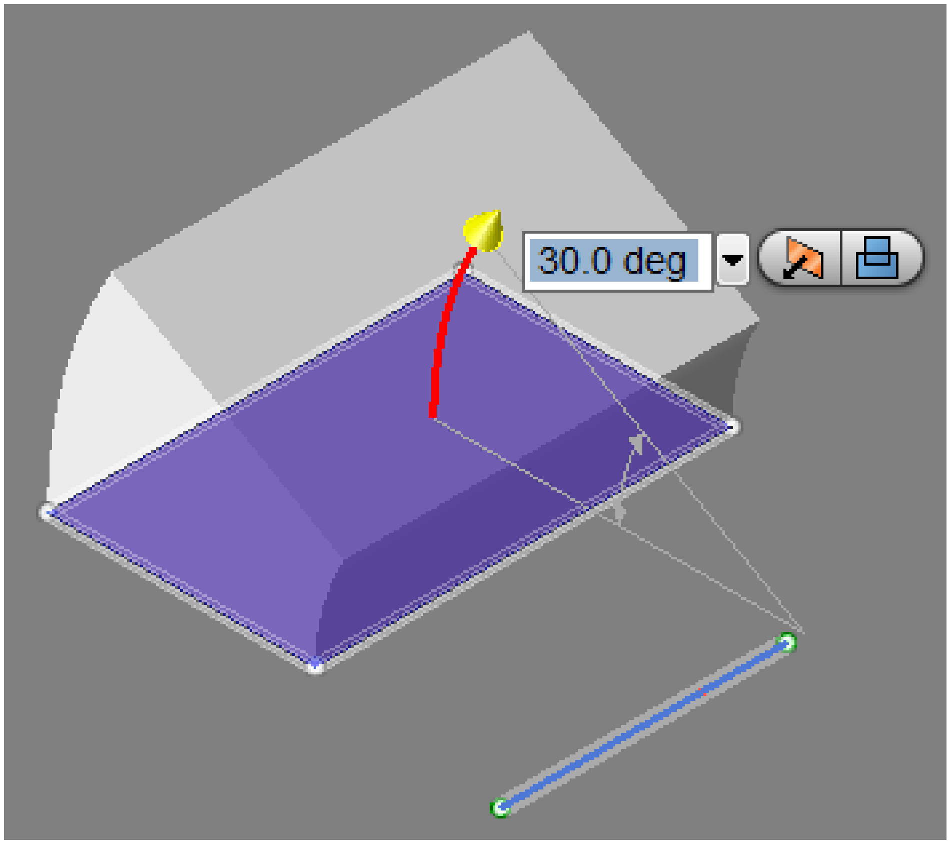



feature.




- Sets the taper angle in Extrude.
- Sets the location in Hole.
- Sets the distance along the path in Sweep.

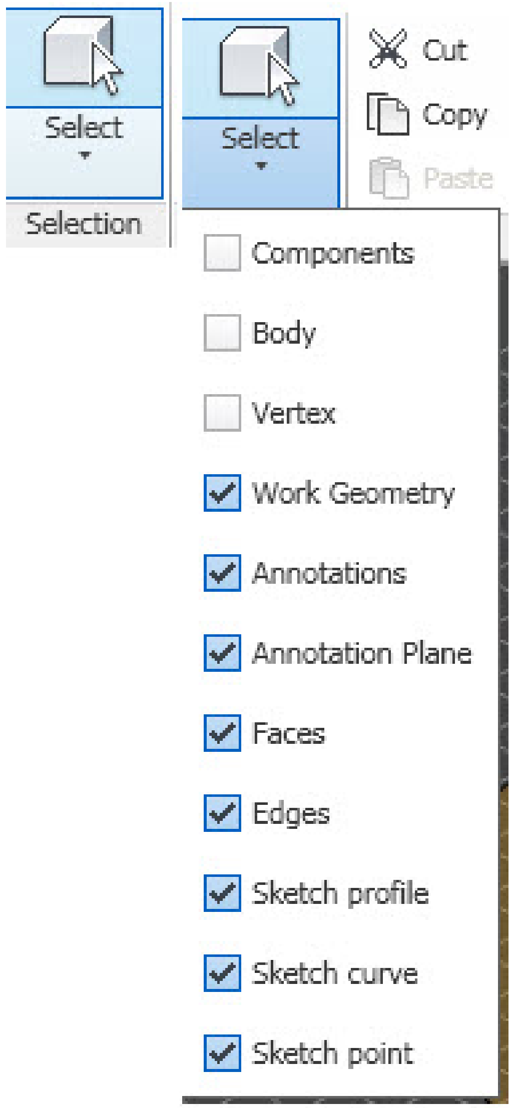

8 cm


30.0 deg

## Selection Filters

Selection filters limit the types of objects that you can select in the graphics window. Filters simplify the selection of specific items in a complex model.

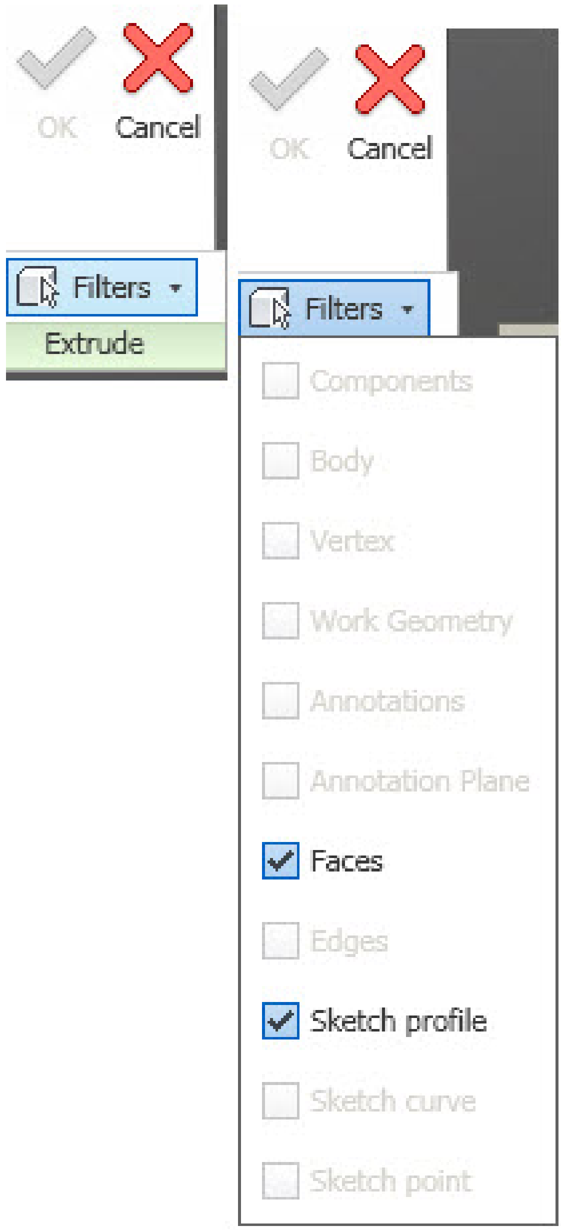
The selection filters control only selection in the graphics window. You can still select objects in the browser, and you can use the Selection Tool (page 38) to select related objects. For example, if the Face filter is turned off, you can select an edge on the model, and use the Selection Tool to select an adjacent face.

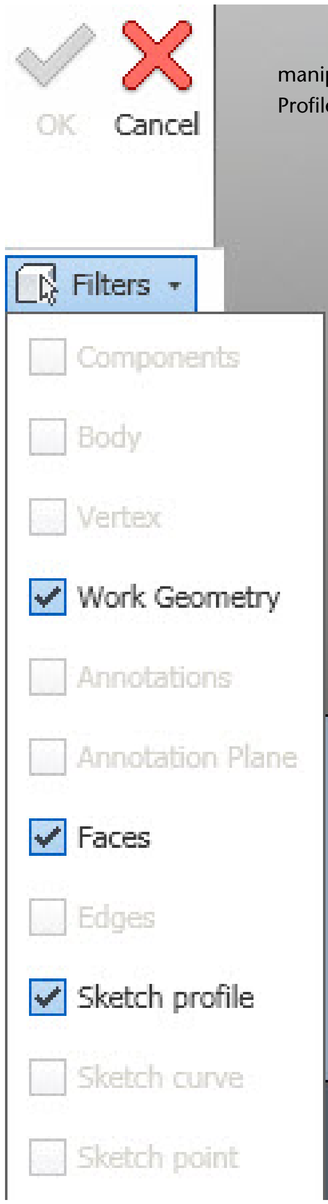



**Selection filters for commands**

Commands that require selections have custom filter lists. For example, you can select Faces or Sketch Profiles when you create an extrusion. You can deselect items on the filter list to simplify selection.

The selection filter list is available on the command ribbon. Expand the feature panel, and click Filters. Only the valid filters are available.


OK
Cancel
Filters
Extrude
OK
Cancel
Filters
Components
Body
Vertex
Work Geometry
Annotations
Annotation Plane
Faces
Edges
Sketch profile
Sketch curve
Sketch point




lter list updates for other modes in commands. For ple, in the To Face option for the extrusion distance manipulator, the options Work Geometry, Faces, and Sketch Profiles are available.

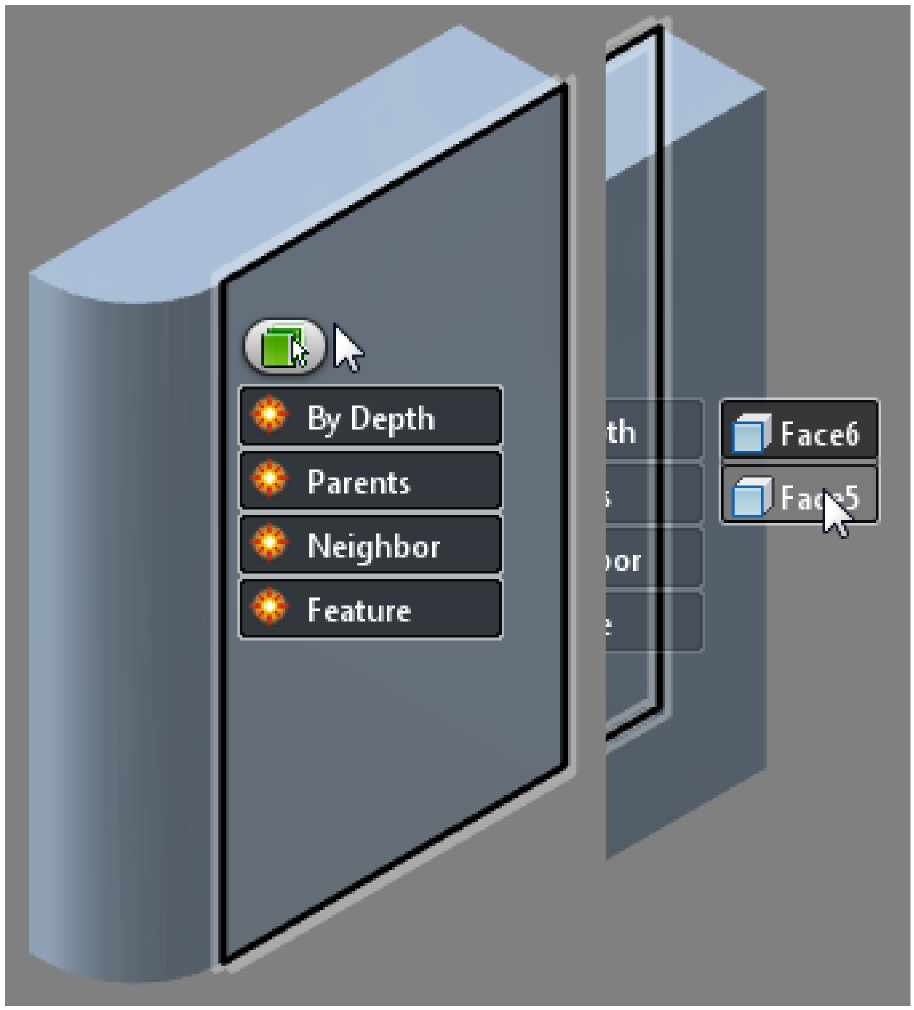
## Selection Tool

Use Selection for geometry that is hidden or difficult to select. The Selection glyph provides a list of options for selecting related geometry.

The selection options include:

- Hidden objects, such as faces.
- Parents of the root object, such as the solid and the component.
- Geometry that is next to the root object.

- Features that include the root object.

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**NOTE** Depending on the root object, some of the options do not have valid selections. For example, a sketch has no Feature selections.

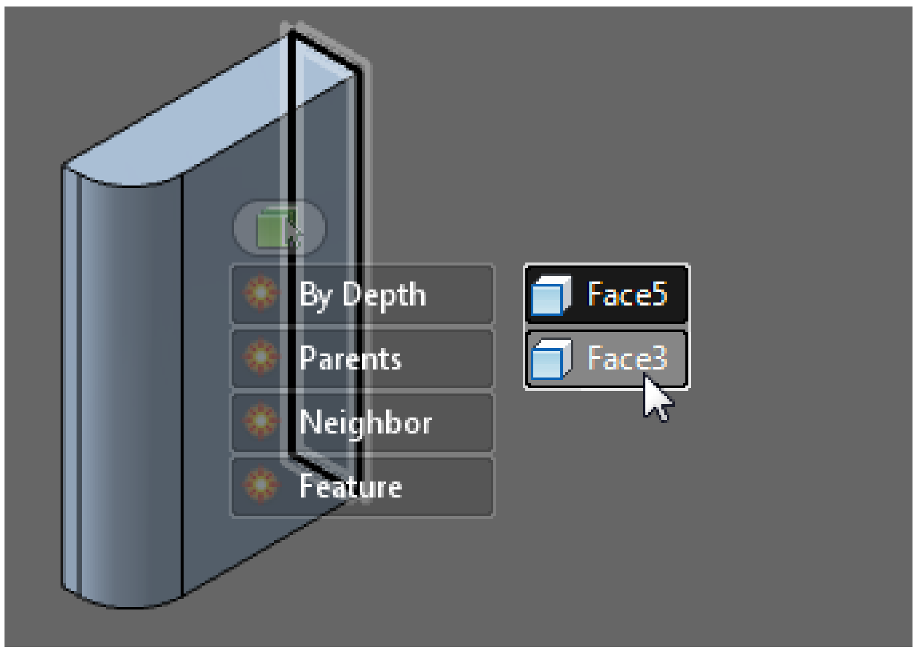
---

### Use the Selection command

The Selection glyph displays when you pause the cursor over a visible object. This object is called the root object. When you move the cursor to the glyph, a list displays with selection options.

**NOTE** When the cursor is over the selection glyph, use the mouse wheel to cycle through the By Depth selections. Click on the selection glyph to select the highlighted face or edge.

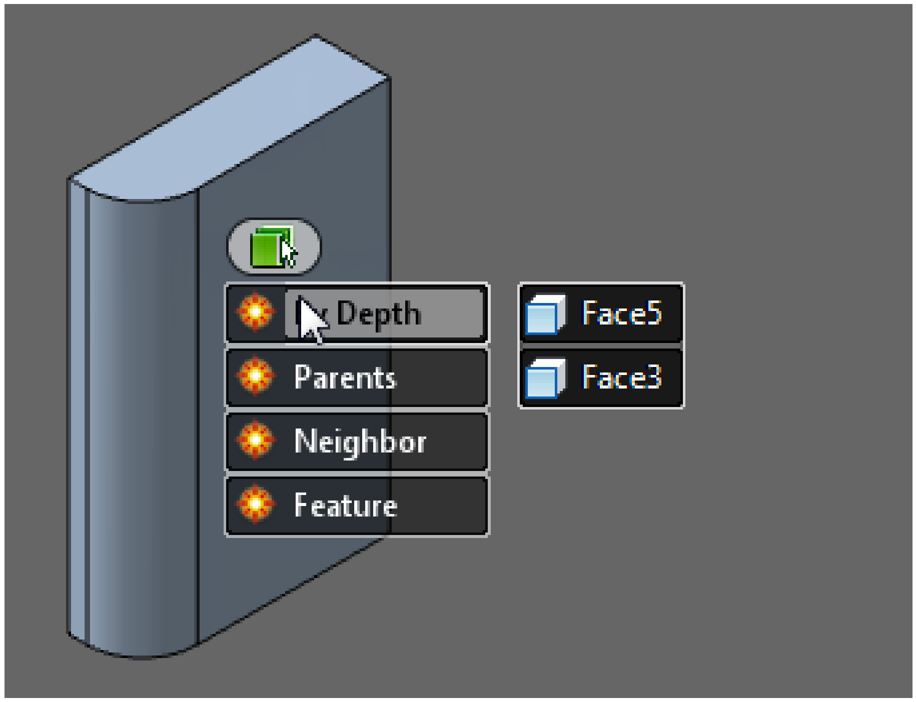
When you pause over any of the options, a selection list displays. The corresponding geometry in the model highlights when you move the cursor over the items in the list.




**NOTE** Click on a selection option to select the first item in the list. For example, clicking on Parents selects the Document.

### Select By Depth

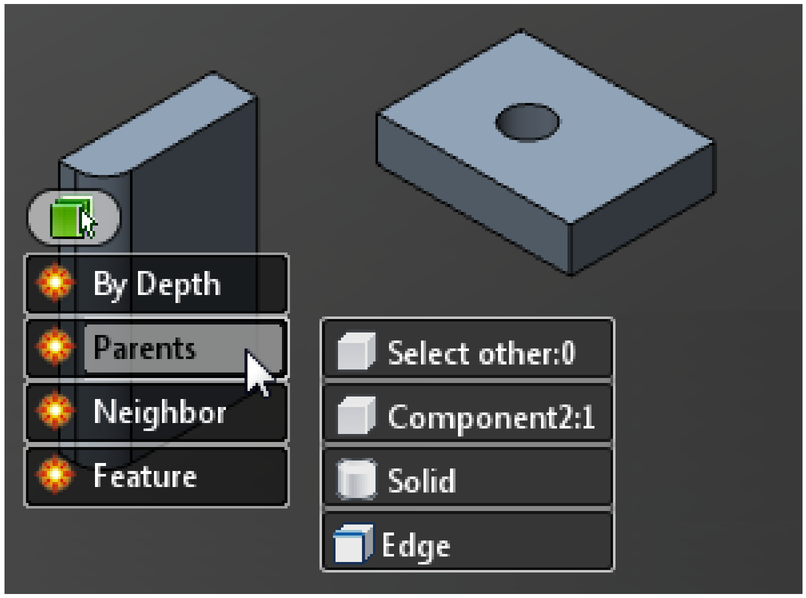
By Depth lists the faces, edges, work planes, work axes, work points, and profiles that are partially or completely hidden by the root object. The root object is the first item on the list, and the other items are in order by depth.




**Select Parents**

The Parents list displays the model structure from the document down to the root geometry. When you click on an item in the list, you select that level and everything below it. For example, Component selects the solid, surfaces, and work geometry, while Solid just selects the body.

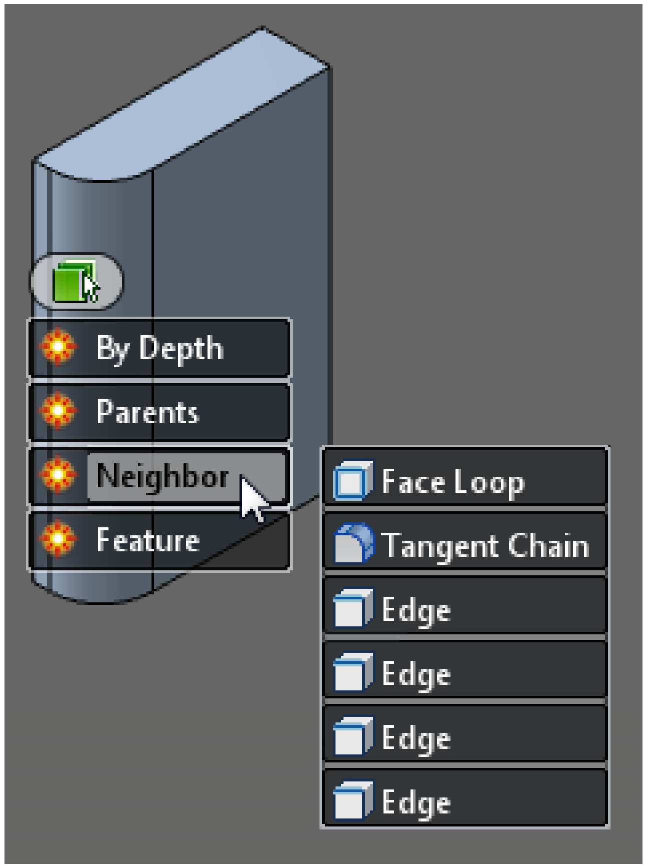
In the following example, there are two components in the file, and the cursor is paused over an edge. The first item is the document, followed by the component, the solid, and the edge.




**Select Neighbor**

The Neighbor list displays all geometry connected to the root geometry. This selection tool is useful when you want to select a small face or edge, or tangentially connected geometry.

- **Face Loop** All edges of the root face
- **Tangent Chain** For a root face, all faces that are tangent to the root face and each other. For a root edge, any edges that are tangent to the edge and each other. If there are not any tangent faces or edges, the root geometry is highlighted.
- **Edge** The edges of the root face.
- **Face** The faces next to the root edge.




**Select Feature**

The Feature list displays the features associated with the root geometry. The selection list differs depending on whether the cursor is over a face or an edge, and the adjacent geometry.

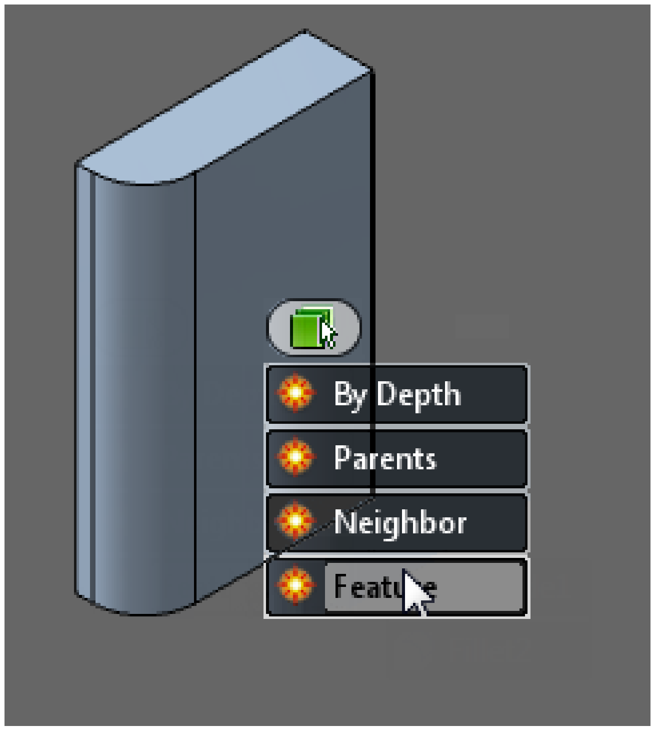
---

**NOTE** The root geometry loses the highlight when the cursor is moved to the Selection glyph. In the following examples, the root edge or face is highlighted to clarify the dependency between the root geometry and the selection list.

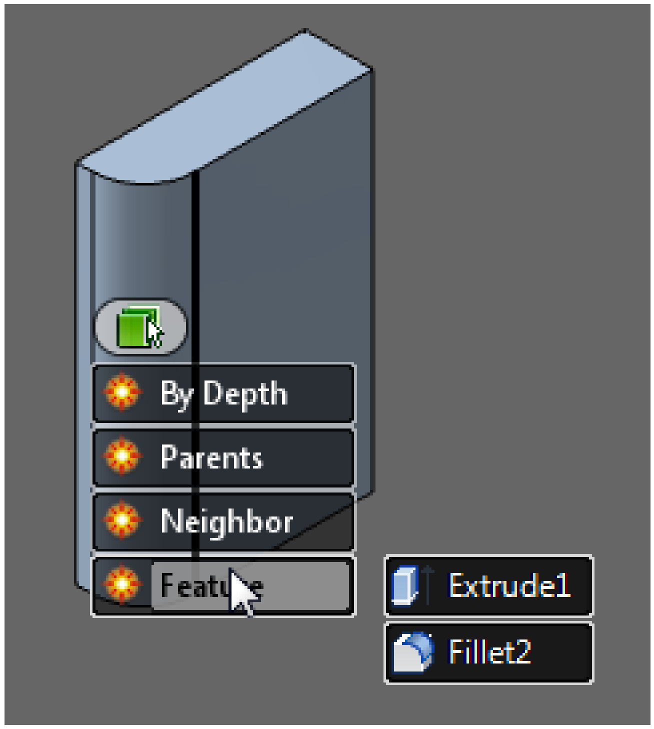
---

There are two selection cases for an edge, depending on whether the two features share the edge.

In the following image, the cursor paused over the edge on the right. Since that edge is not shared by two features, the fly out list does not display. You can still select the feature by double-clicking on the Feature option.




In this example, the cursor paused over the fillet edge. Since the edge is shared by Extrude1 and Fillet2, it has the standard highlighting and selection behavior.

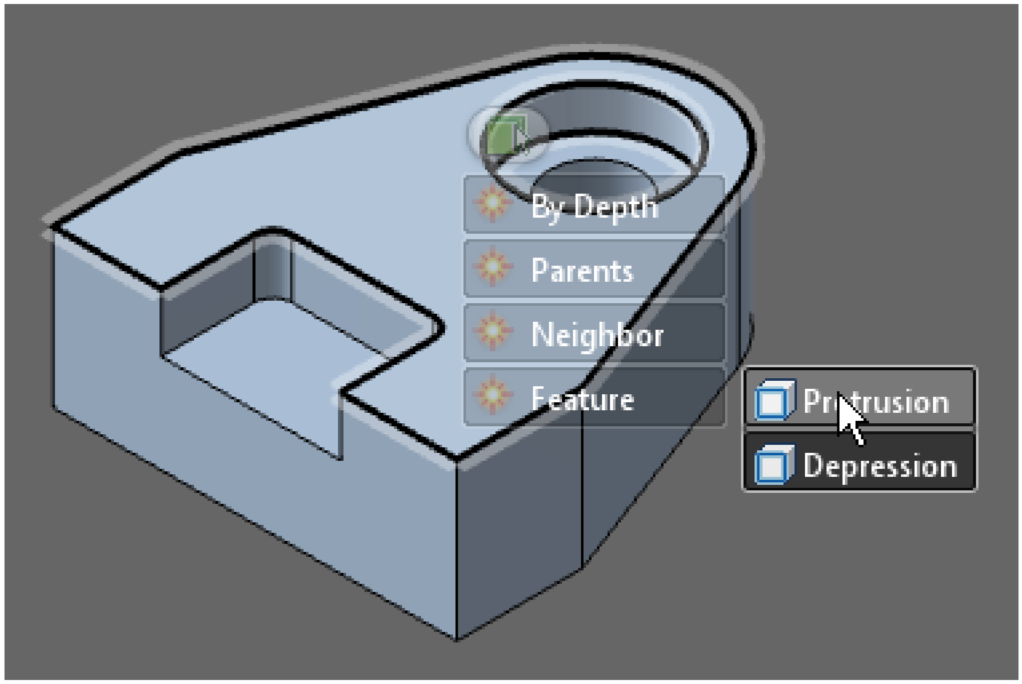



**Protrusion and Depression**

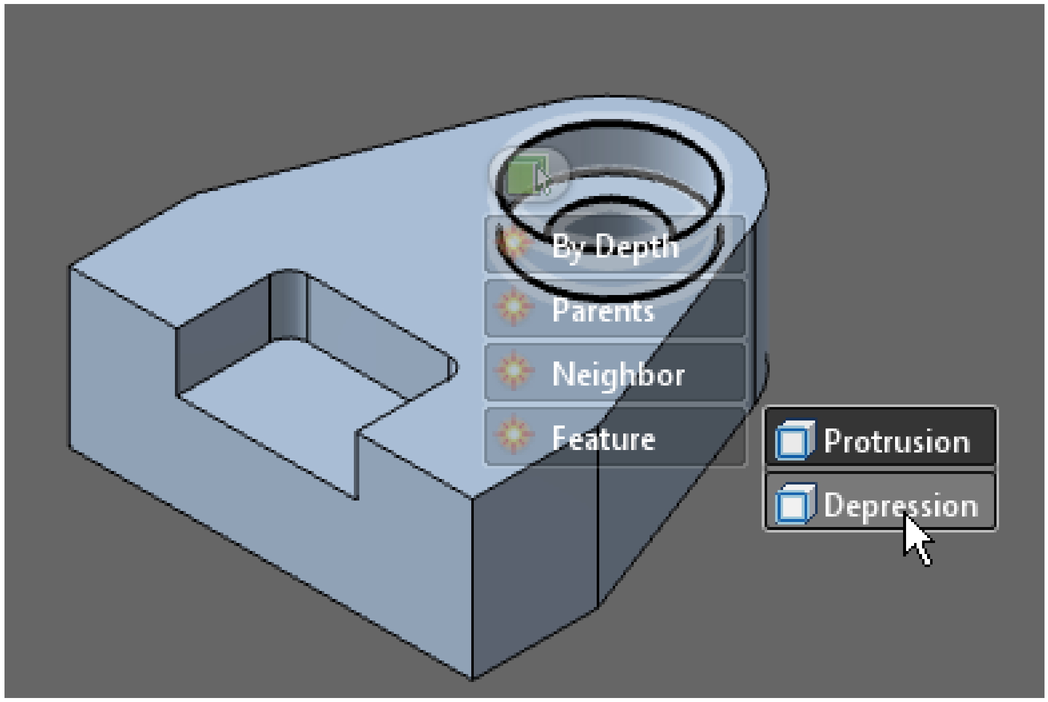
When the cursor pauses over a face, the Selection tool analyzes adjacent geometry to determine if the geometry cuts into or extends from the face. In addition to the parent feature, the list can also display Protrusion and Depression. Most faces without additional geometry will display the parent feature and Depression. In some cases, Protrusion is displayed instead of Depression.

In the images following, the cursor is paused on the cylindrical face. Both Protrusion and Depression are on the Feature selection list. Fusion analyzes adjacent faces and determines whether a set of faces is towards the center of the solid (Depression) or towards the outside (Protrusion).

The Protrusion selection set includes the cylinder and exterior faces.



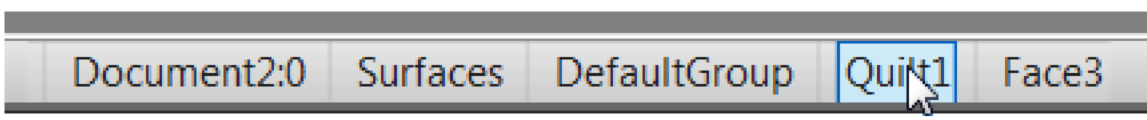

The Depression selection set includes the cylinder and the bottom face.




**Additional Selection Methods**

You can double-click a face to select all objects in that component, including child components.

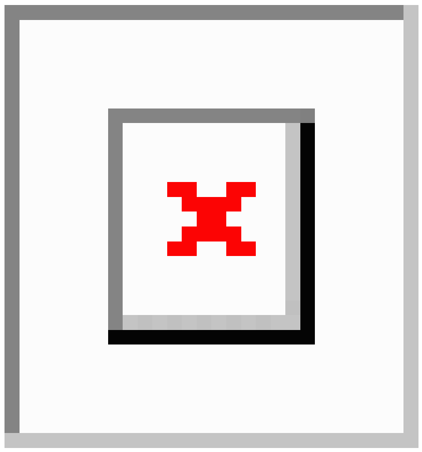
The status bar displays the parents of a selected object. Click a button to select that object.




## Triad

The Triad is a 3D tool with manipulators to move and rotate objects in the model. The manipulators can translate along one, two, or three axes, or rotate about an axis.

The active manipulator is yellow. Click on a manipulator to activate it, then click and drag in the graphics window to move the object. The triad has priority, so you can click and drag anywhere in the graphics window.

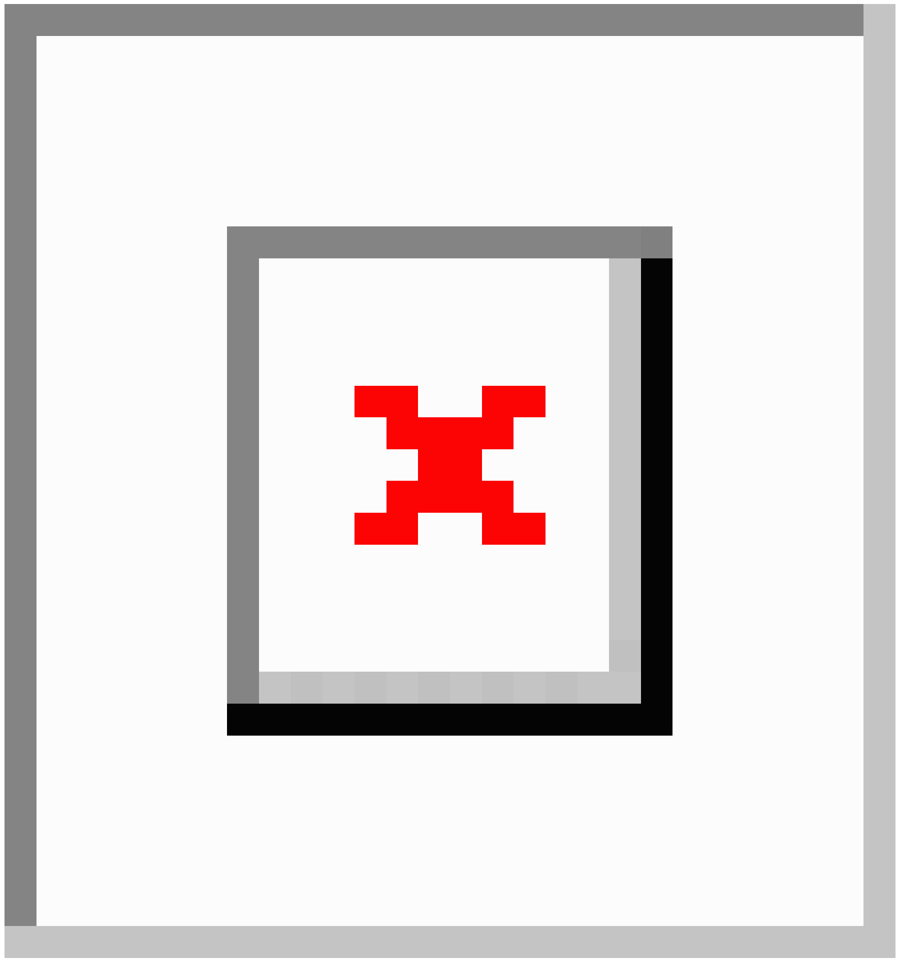


| Manipulator | Description |
|---|---|
| Arrowheads | Move along the axis. |
| Planes | Move on the plane. |
| Sphere | Move in three dimensions. |
| Arcs | Rotate around the axis. |

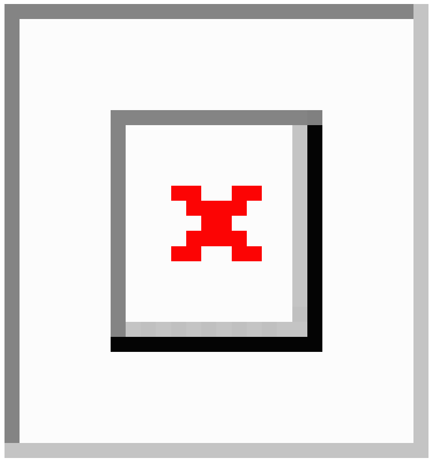
### Appearance of the Triad

The manipulator color indicates its status. The active manipulator is yellow. The inactive manipulators are red (X axis), blue (Y-axis), or green (Z-axis). Disabled manipulators are transparent.

Only the active manipulator displays during drag. In the images following, the active manipulator is yellow, and reference manipulators are transparent.

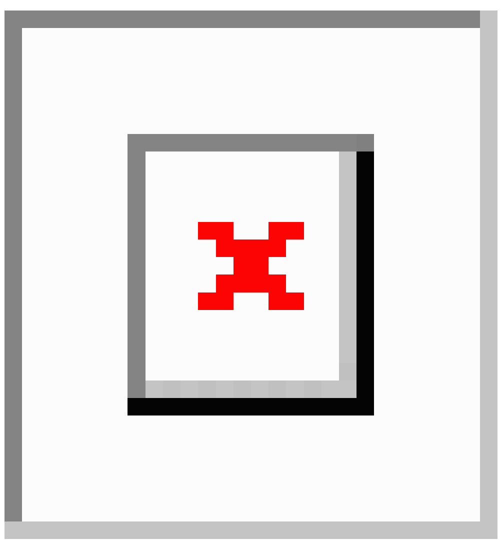


When the cursor moves away from the Triad, the arc and plane manipulators dim. If the cursor moves father away, the rotation and plane components become invisible.



### Reorient the Triad

The Triad Reorient glyph button displays near the triad origin. To change the position or orientation of the triad, click a manipulator, and then selecting geometry in the model. For example, you can click a plane manipulator, and then select a planar face or work plane. As you move the cursor, a dynamic preview shows how the triad moves for that selection. Click the Finish Reorient glyph to move the triad.



 **Show me how to reorient the triad**

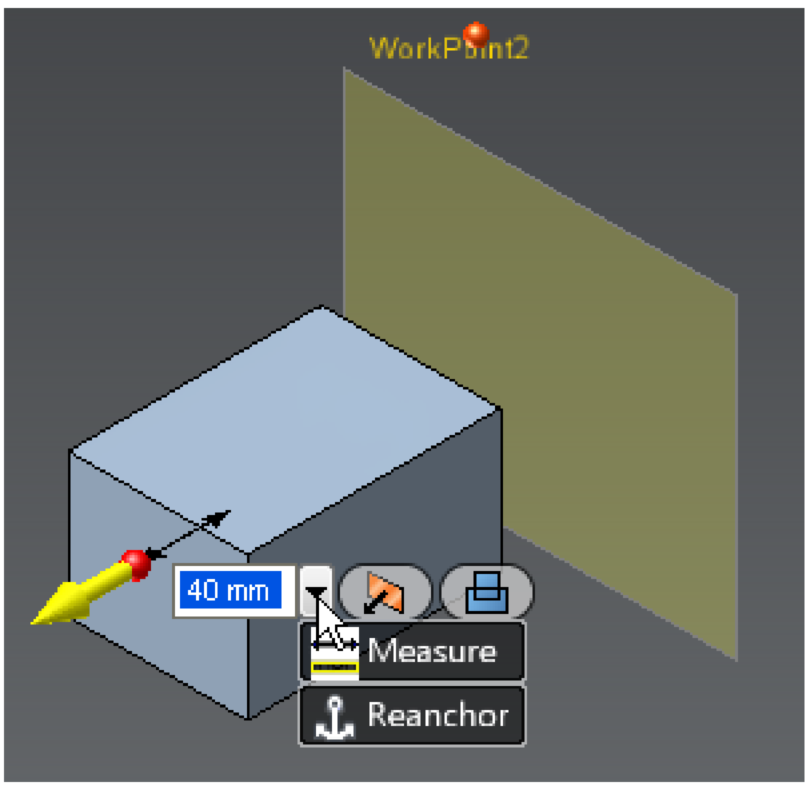
This video shows how to reorient the triad, and how the different orientations affect the changes to the model.

## Reanchor Command

When you use a linear manipulator, the distance is measured from the original position. The Reanchor command changes the origin reference so it displays the distance from the new location.

### Use Reanchor

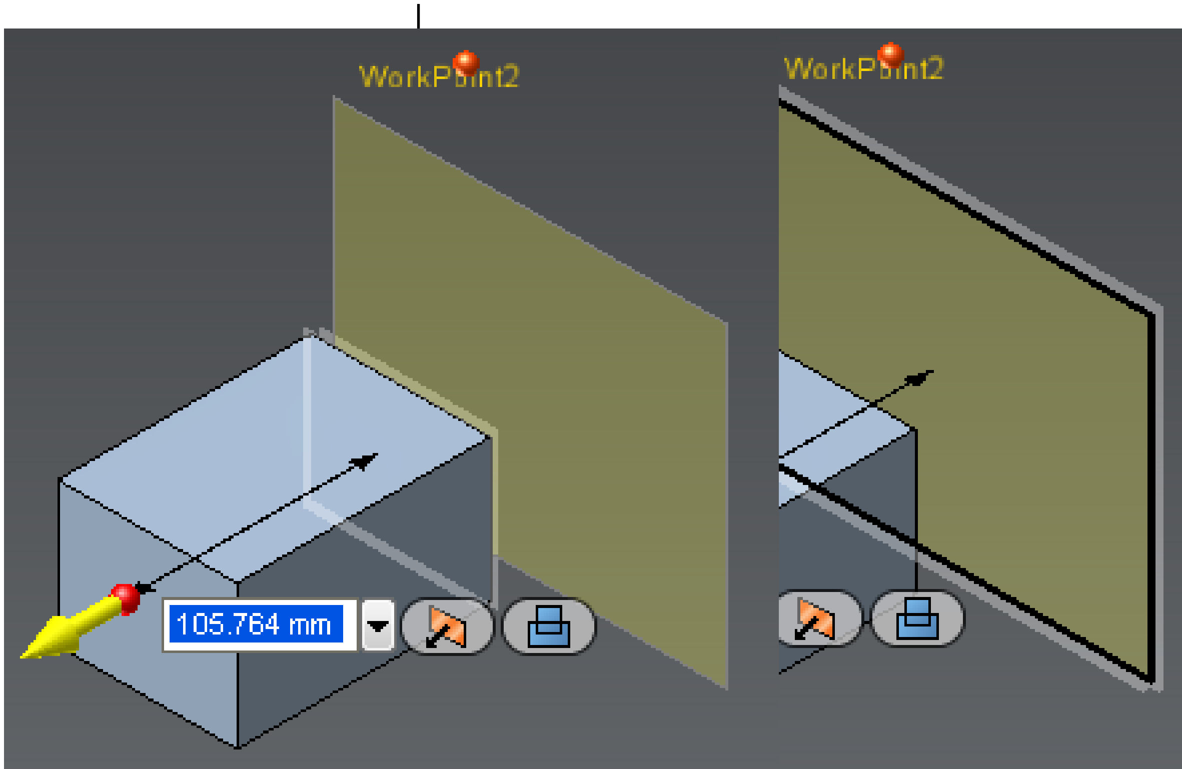
1 Next to the value entry field, pause the cursor over the drop-down arrow, and select Reanchor.



2 Select a point, planar face, or work plane. The face or work plane must be perpendicular to the manipulator axis.

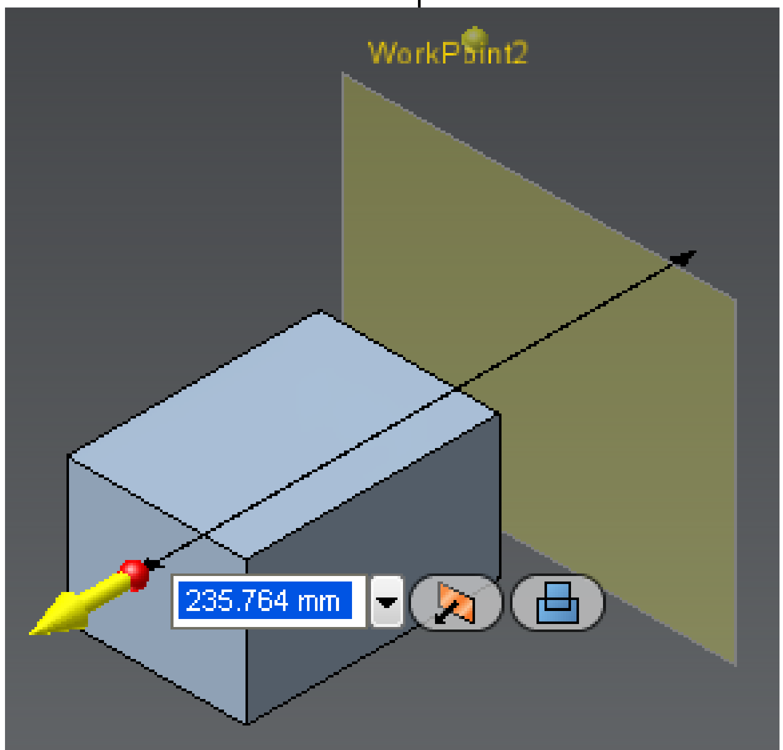
3 The dimension arrows connect to new reference and distance value updates.

In the images below, the manipulator is reanchored to the back face of the feature, the work plane, and the work point.



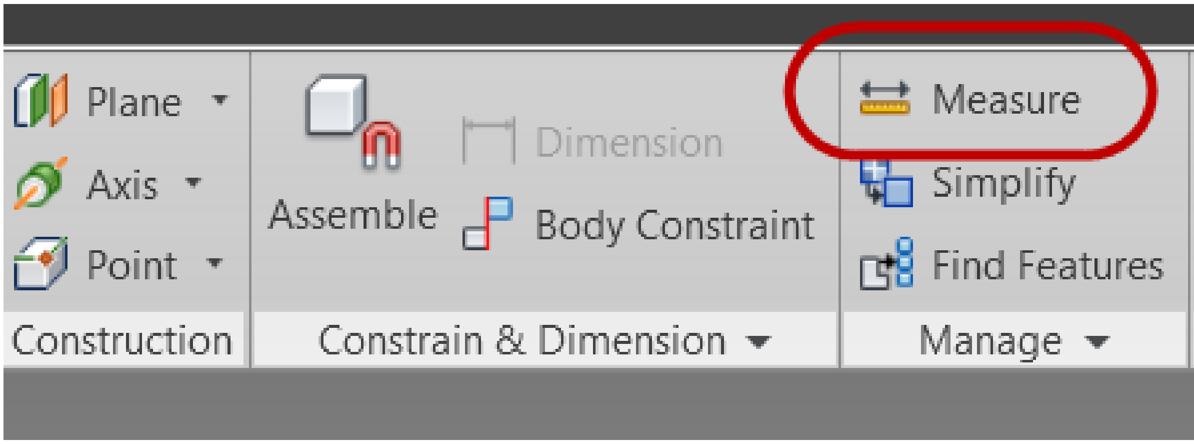

Linear manipulator reanchored to the perpendicular face.

Linear manipulator reanchored to the perpendicular workplane.

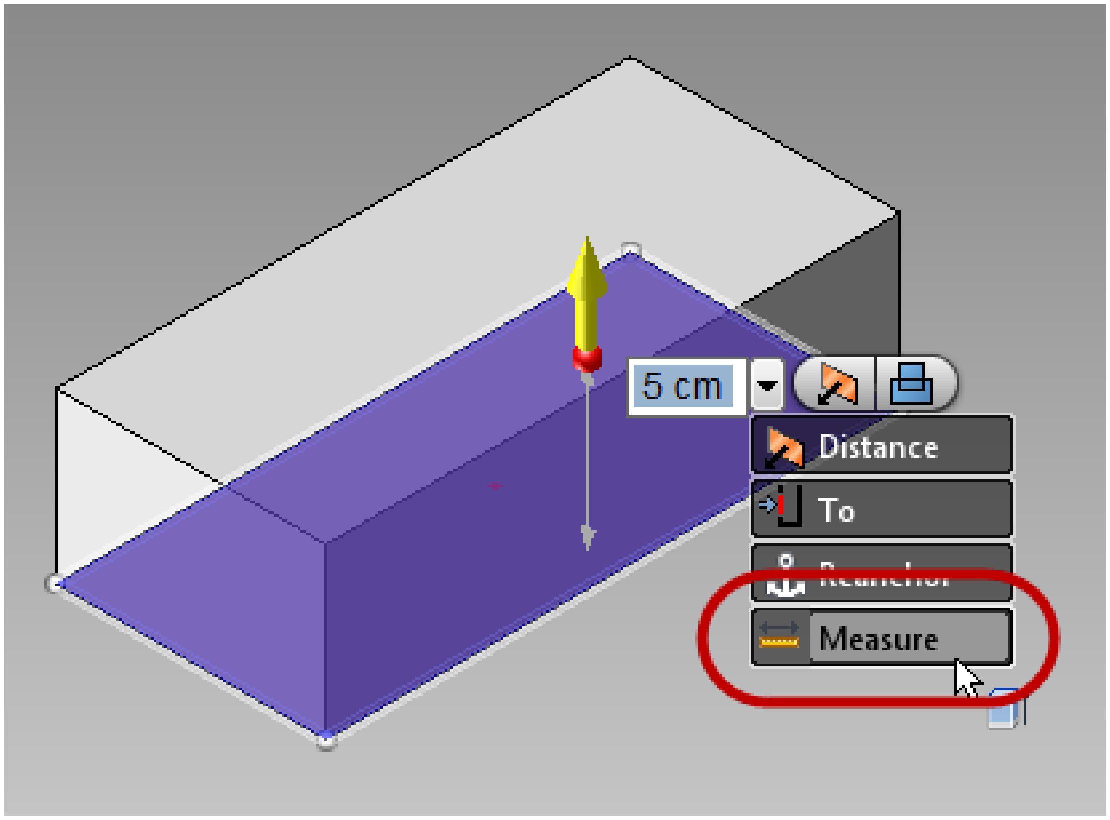


Linear manipulator reanchored to the workpoint.

## Measure Command

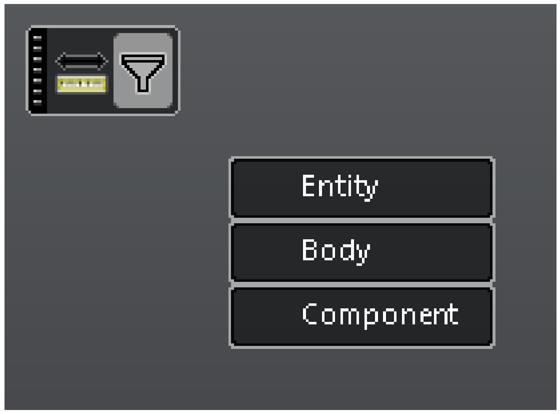



Measure provides geometry information (distance, angle, area, and so on), and Populates input boxes with measurement.



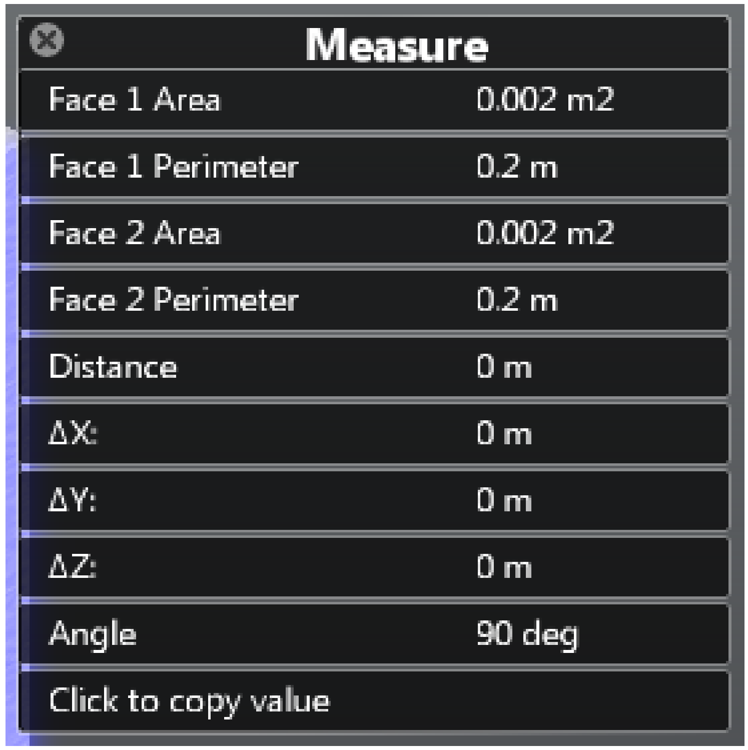

**Selection Support**

You can select objects in the browser or in the graphics window when using measure. You can use filters to control which geometry types are eligible for selection.




**Measure Dialog Box**

The Measure dialog box displays when you start the command, and persists until you terminate the command. The dialog box displays geometry information dependent on the selected entities.




To copy a value to the clipboard, click a row in the measure dialog box. Use `CTRL+V` to paste the value.

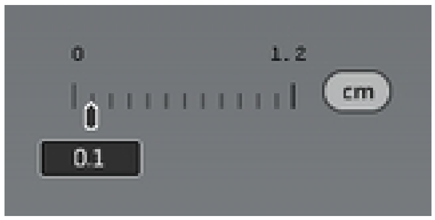
Click and drag the dialog box to reposition it.

**Sample Workflow**

1. Select a face to extrude.
2. On the value input flyout,, click Measure.
3. In the graphics area, select the objects to measure.
4. You can set the length of the extrusion equal to the length of the selected edge (2). In the Measure dialog box, click the Curve 2 Length row. The value is copied to the input box and the clipboard.

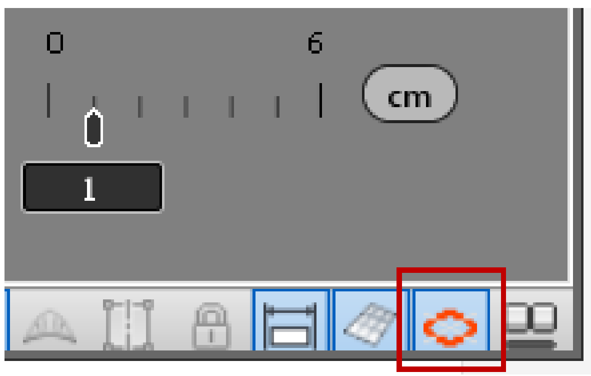
   The Measure dialog box closes, and the model updates accordingly.

## Snap Bar



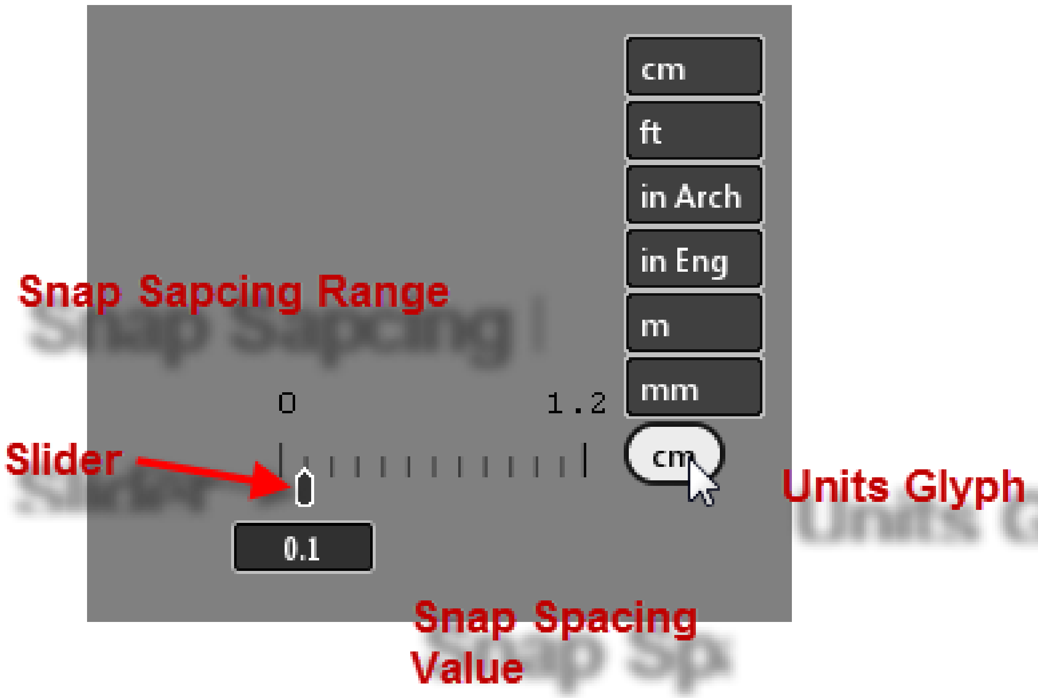

The Snap Bar displays in the lower right corner of the graphics window. It displays the document units, snap value range, and snap spacing. The snap range and spacing automatically update as you zoom in and out.

Snap spacing is used for sketching and modeling. You can turn snapping off by clicking the Snap Mode icon on the status bar. See the Status Bar topic for more information.




### Elements of the Snap Bar

- **Snap Spacing Range** Displays the range of snap spacing values. The range updates automatically, depending on the model size and zoom level.
- **Snap Spacing Value** Displays the current snap spacing value. The snap spacing automatically updates as you zoom in and out. Enter a value to override the automatic snap spacing.
- **Slider** Shows the position of the current snap value on the scale. Drag the slider to override the automatic snap spacing.
- **Units Glyph** Displays the current document units. Pause the cursor over the glyph to display the units list. Select from the list to change the units.




**Dynamic Snap Spacing**

The Snap Bar dynamically updates the snap range and value based on the model size and current zoom level. The snap range becomes small when you zoom in, and large when you zoom out. The default snap spacing value is the first division of the snap range.

**Override the Snap Spacing**

To override the snap spacing value. drag the slider, or enter a value. The snap range still updates when you zoom, but the snap spacing does not change unless the value is outside the range. If your snap spacing is too large or too small for the zoom level, the value temporarily updates.

For example, enter a value of 25 when the range was 0-100. If you zoom in until the range changes to 0-20, the snap spacing is greater than the range, and the value changes to 5. When you zoom out so that the range includes 25, the snap spacing changes back to the original value.

**Set the Units**

The units system is set for each document. Pause the cursor over units glyph to display the units list. Select from the list to change the units system.

There are two Inch unit systems:

- **In Arch (Inch Architecture)** The snap range and spacing values adjust in fractional inch increments.
- **In Eng (Inch Engineering)** The snap range and spacing values adjust in decimal inch increments.

# Commands in the Application Window

The application window displays commands such as the application button, the Quick Access toolbar, and the status bars.

## Application Menu

Click the 

application button to access commands to create, open, and export a file.

### Access Common commands

On the Application menu, access commands to:

- Create a file.
- Open and save files.
- Print the current view.
- Publish to an image file.
- Close files.
- Display the Welcome Screen.
- Options to set the type of render mode.

### Command Reference

The following commands are accessible from the Application menu:

#### New

Creates a new Fusion .dwg file. The new file is set as the active file.

#### Open

Open an existing file in Fusion. Available types include:

- Alias (*.wire)
- CATIA V5 (*.CATPart, *.CATProduct)
- DWG (*.dwg)
- SAT (*.sat)
- IGES (*.igs, *.iges)
- Inventor (*.iam, *.ipt)
- Parasolid Text (*.x_t)
- Parasolid Binary (*.x_b)
- Pro/ENGINEER Granite (*.g)
- Pro/ENGINEER (*.prt, *.asm)
- Rhino (*.3dm)
- Solidworks (*.sldprt, *.sldasm)
- STEP (*.stp, *.step)

#### Save, Save as

Save the active file (Save) or save the active file with a different name or file type (Save as). Available types include:

- CATIA V5 (*.CATPart, *.CATProduct)
- DWF (*.dwf)
- DWG (*.dwg)
- AutoCAD 2009 (*.dwg)
- STL (*.stl)
- SAT (*.sat)
- IGES (*.igs, *.iges)

- Parasolid Text (*.x_t)
- Parasolid Binary (*.x_b)
- Pro/ENGINEER Granite (*.g)
- STEP (*.stp, *.step)

### Print

Use Print Setup, Print Preview, and Print to set options and print any portion of your model. Only the portion of the model that is displayed in the graphics window prints.

### Publish

Use Publish to save the current view to an image file. Click the Options button in the Publish to Image dialog box to configure settings to the image. Available file typs include:

- PNG
- JPEG
- GIF
- Windows Bitmap
- TIFF
- EMF
- EXIF
- WMF

### Recover

Attempts to repair then open a damaged dwg file.

### Close and Close All

Use Close to close the active file or Close All to close all files in the current session of Fusion. You are prompted to save any files with unsaved changes.

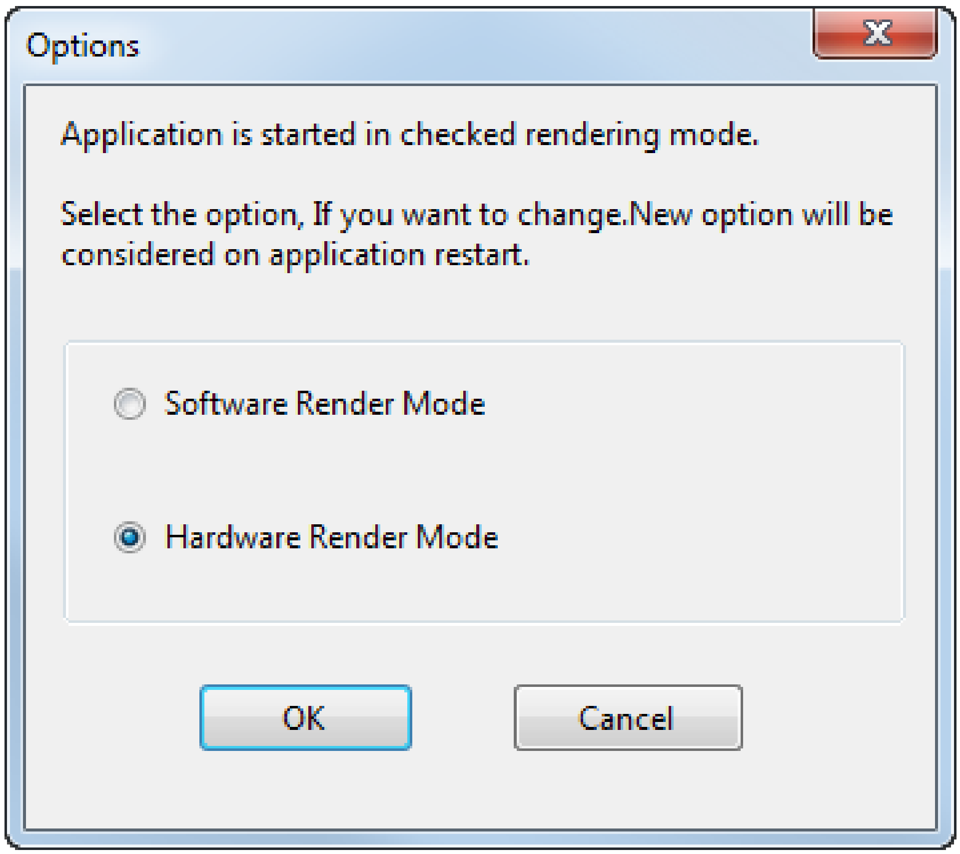
### Welcome Screen

Displays the Fusion Welcome Screen.

**Options**

Set the type of render mode.

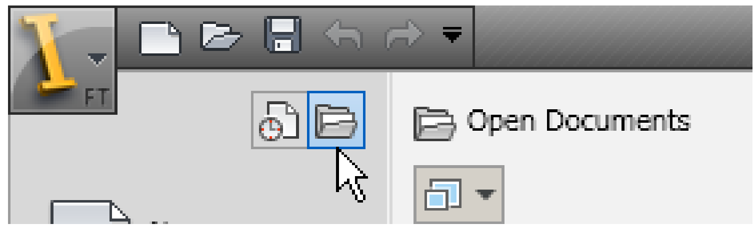
- Software Render Mode is an application based rendering with dependencies on the machine operating system. Compare your machine specifications with the recommended machine specifications for the application.
- Hardware Render Mode is the rendering provided by the machine's graphic card. The graphic card specifications, feature level, RAM, and so on, determine the rendering performance.



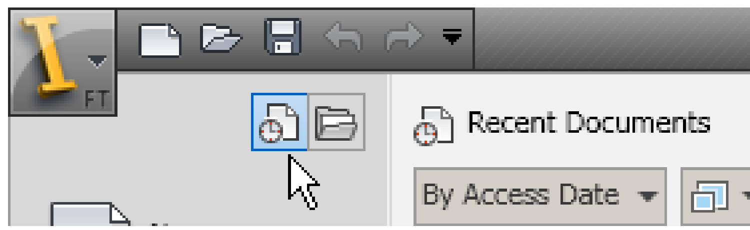
## Documents List

The documents list in the Application menu lists either recently opened documents or currently open documents.

To view open documents, click Open Documents .




To view recent documents, click Recent Documents.

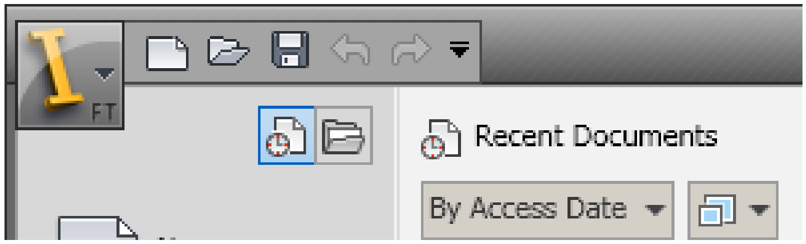



### Open Documents

Files that are open display in the Open Documents list with the most recently opened file at the top. To make a file active, click the file in the list.


### Recent Documents

Displays a list of recently used files, with the most recently used file at the top.




**Pin Files**

To keep a file listed, regardless of files that you save later, use the push pin button to the right. The file displays at the bottom of the list until you turn off the push pin button.

**Sort and Group Options**

At the top of the Recent Documents list, use the By Ordered list drop-down menu to group files by:

- File name
- File size
- File type
- Date the files were last modified.

### Document Information

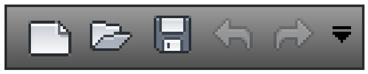
Displays file information in the Recent Documents and Open Documents lists.

Pause the cursor over a file in either of the lists to display the following information:

- Path where the file is stored
- Date the file was last modified
- Version of the product used to create the file
- Name of the person who last saved the file
- Name of the person who is currently editing the file

# Quick Access Toolbar

Display frequently used commands with the Quick Access toolbar (QAT).



### View Undo and Redo History

The QAT displays options to undo and redo changes to your file.

### Add Commands and Controls

- Add unlimited commands to the QAT. Commands that extend past the maximum length of the toolbar display in a drop-down menu.
- Add a ribbon command to the QAT. Right-click the command on the ribbon, and click Add to Quick Access toolbar. Commands you add to the QAT are positioned to the right of the default commands.

### Move the Quick Access Toolbar

Place the QAT either above or below the ribbon using the Customization button.

1. On Quick Access toolbar, click the drop-down arrow.

2. On the Customize menu, click Show Above the Ribbon, or, Show Below the Ribbon to display the Quick Access toolbar above or below the ribbon. A check mark next to a command name indicates it is displayed on the Quick Access toolbar.

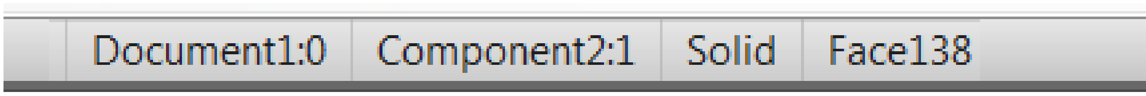
### Add and remove commands on the Quick Access toolbar

1. On the Quick Access toolbar, click the drop-down arrow.

2. On the Customize menu, click the command name to display on the Quick Access toolbar. A check mark next to a command name indicates it is displayed on the Quick Access toolbar.

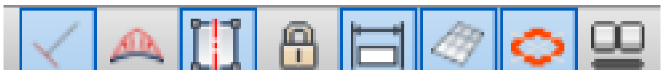
## Status Bar




The status bar displays across the bottom of the active window.

In a command that requires you to perform an action to continue, a message displays on the far bottom left of the status bar. It indicates the next step to proceed with the active command.

Information about the currently selected objects is displayed in the center of the status bar.



There are a series of icons on the far bottom right of the status bar. These icons provide quick access to turn options on/off. These options from left to right are:

- Constraint: Controls the display of sketch constraints while in sketch mode.
- Curvature Comb: Controls the display of the curvature comb while editing an edge using the Edit Edge command.
- Symmetry Mode: Controls the symmetry lock when using the Edit Edge command.
- Lock Mode: Controls the edge lock when using the Edit Edge command.
- Precise Input: Controls the display of value input controls when creature sketch geometry or feature geometry.
- Sketch Grid Mode: Controls the display of the sketch grid when in sketch mode.
- Snap Mode: Controls the snap behavior when creating sketch geometry or feature geometry.
- Document Selector: Displays the document selector to change the active document.

# Navigation commands

Navigation commands change the orientation and view of your model.

Using the View Cube, Steering Wheels, and commands in the Navigation panel, you can:

- Increase or decrease the magnification of objects.
- Rotate the view of the model.
- Create a view that defines an area of a model as the Home view and with the View Cube.
- Use preset views to restore known viewpoints of a model with the View Cube.

## View Cube

### ViewCube Overview

The ViewCube command provides the means to switch between standard and isometric views of your model. The ViewCube displays in one of the corners of the window over the model in an inactive state.

While the ViewCube command is inactive, it provides visual feedback about the current viewpoint of the model as view changes occur. When you position the cursor over the ViewCube command, it becomes active. You can switch to one of the available preset views, roll the current view, or change to the Home view of the model.


**Appearance of the ViewCube**

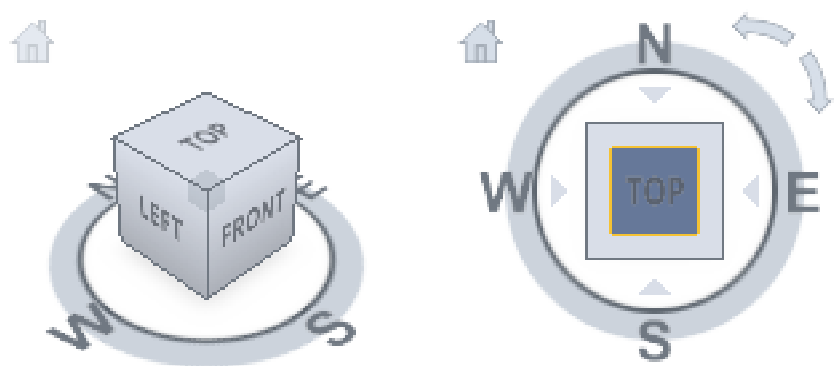
The ViewCube command displays in one of two states: inactive and active. When the ViewCube command is inactive, it appears partially transparent by default so that it does not obscure the view of the model. When active, it is opaque and could obscure the view of the objects in the current view of the model.

In addition to controlling the inactive opacity level of the ViewCube command, you can also control the following properties for the ViewCube command:

- Size
- Position
- Default orientation
- Compass display

**Compass in the ViewCube**

The compass displays below the ViewCube, and indicates where the direction North is defined for the model. You can click a cardinal direction letter on the compass to rotate the model. You can click and drag one of the cardinal direction letters or the compass ring to rotate the model interactively around the center of the view.




## Reorient Views of Models with the View Cube

View Cube reorients the current view of a model.

To reorient the view of a model with the View Cube, click predefined areas to set a preset view current. Click and drag to change the view angle of the model, and define and restore the Home view.

## ViewCube Menu

The ViewCube menu provides options to:

- Restore and define the Home view of a model.
- Switch between view projection modes.
- Change the interactive behavior and appearance of the ViewCube.

The options on the ViewCube menu are as follows:

- **Go Home** restores the Home view saved with the model.
- **Orthographic** switches the current view to orthographic projection.
- **Perspective** switches the current view to perspective projection.
- **Perspective with Ortho Faces** switches the current view to perspective projection unless the current view aligns with a face view defined on the ViewCube.
- **Lock to Selection** uses the selected objects to define the center of the view when a view orientation change occurs with the ViewCube.

  **NOTE** Click Home on the ViewCube, to return to the Home view, even if Lock to Current Selection is selected.

- **Set Current View as Home** defines the Home view of the model based on the current view.
- **Restore Default Home** resets the Home view of the model to its default orientation.
- **Set Current View as Front** defines the Front view of the model based on the current view.
- **Reset Front** resets the Front view of the model to its default orientation.
- **Properties** displays the dialog box so you can adjust the appearance and behavior of the ViewCube.
- **Help Topics** Opens the Help system and displays the topic for the ViewCube.

**Display the View Cube menu**

- Right-click the compass, Home icon, or the main area of the View Cube.
- Click the context menu button located near the View Cube.

## SteeringWheels

SteeringWheels are tracking (they follow your cursor) menus that provide access to 2D and 3D navigation commands from a single command.

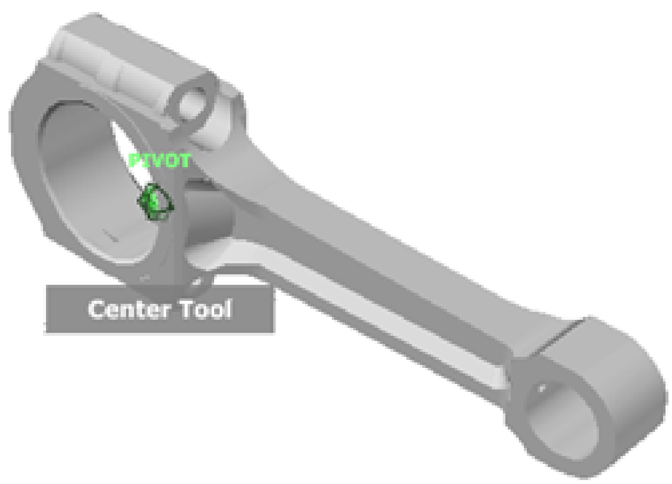
### Navigation commands

Each wheel is divided into different wedges. Each wedge contains a navigation command that you can use to reorient the current view of a model. The navigation commands that are available depend on the active wheel.

### Center Command

Points to the center of the current view of a model. To find the center, drag the cursor over the model. A sphere indicates that the point below the cursor is the center of the current view of the model. When you release the mouse button, the model is centered on the sphere.

**Note:** If a center point on a model cannot be identified, then a circle with a diagonal line displays instead of the sphere, indicating that the operation cannot be performed.



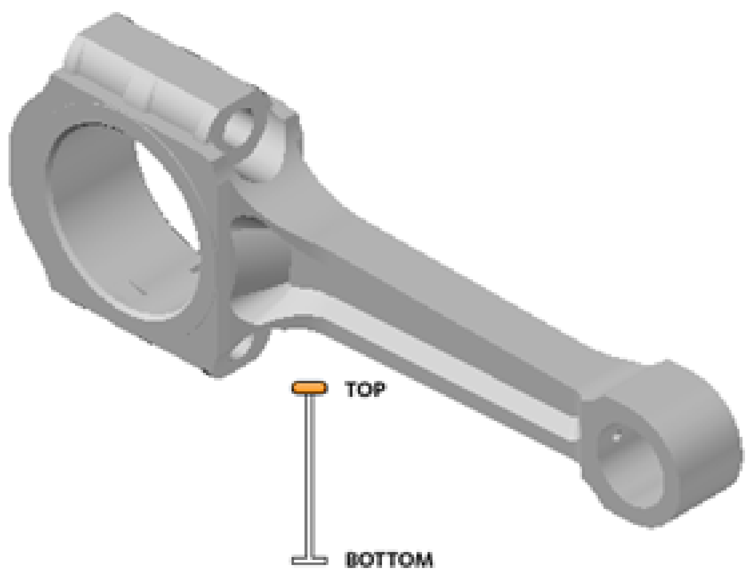

The point defined by the Center command provides a focal point for the Zoom command and a pivot point for the Orbit command.

**Note:** To zoom from the Full Navigation wheels from your defined center point, hold down CTRL before zooming.

## Up - Down Command

Unlike the Pan command, the UP/Down command adjusts the height of the current viewpoint along the Z axis of the model. To adjust the vertical elevation of the current view, drag up or down. As you drag, the current elevation and the allowed range of motion displays on a graphical element called the Vertical Distance indicator.

The Vertical Distance indicator has two marks that show the highest (Top) and lowest (Bottom) elevation possible for the view. While changing the elevation with the Vertical Distance indicator, the bright orange indicator shows the current elevation. The dim orange indicator shows the previous elevation.




**Display a Full Navigation or Tour Building wheel**

1 Click and hold down the Up/Down wedge. The Vertical Distance indicator displays.

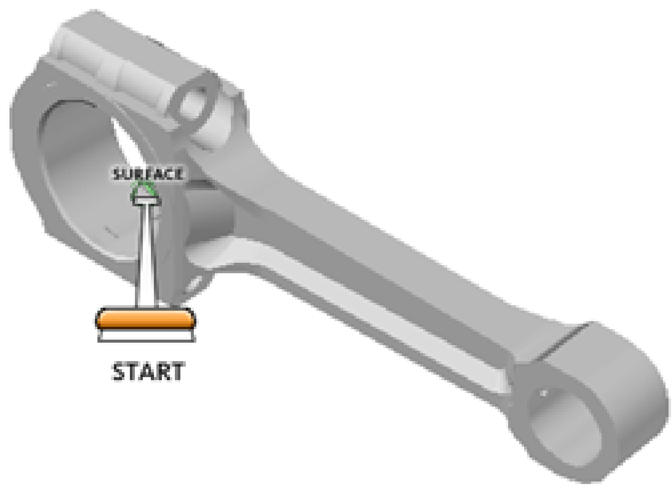
2 Drag up or down to change the elevation of the view.

3 Release the button on your pointing device to return to the wheel.

## Forward Command

The Forward command changes the magnification of the model. It increases or decreases the distance between the current point of view and the pivot point. The position of the pivot point limits the distance that you can move forward or backward.

**Note:** In orthographic views, the Forward command is limited to the distance between the current position and the pivot point. In perspective views, it is not limited, allowing you to move the cursor through the pivot point.




To adjust the distance between the current point of view and the pivot point, use the Drag Distance indicator. The Drag Distance indicator has two marks on it that show the start and destination distances from the current point of view. The orange position indicator shows the current traveled distance. Slide the indicator forward or backwards to decrease or increase the distance towards the pivot point.

**Display the big Tour Building wheel**

1 Click and hold down the Forward wedge within the scope of the model. The Drag Distance indicator displays.

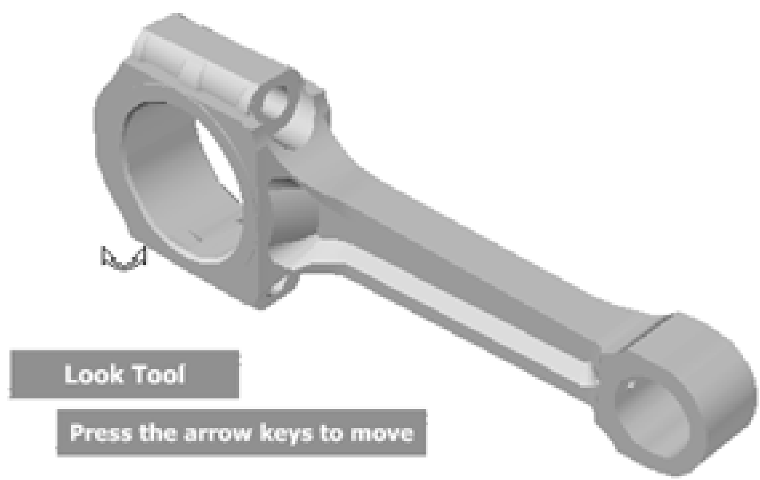
   **Note:** If you click the Forward wedge once, the model moves forward 50% of the distance between the current location and the pivot point.

2 Drag the cursor up or down to change the distance from which you view the model.
3 Release the button on your pointing device to return to the wheel.

## Look Command

With the Look command, you can rotate the current view vertically and horizontally. When rotating the view, your line of sight rotates about the current eye position, like turning your head. The Look command is comparable to you standing in a fixed location, and looking up or down while turning your head left or right.

When using the Look command, adjust the view of the model by dragging the cursor. As you drag, the cursor changes to the Look cursor and the model rotates around the location of the current view.




**Walk through Models**

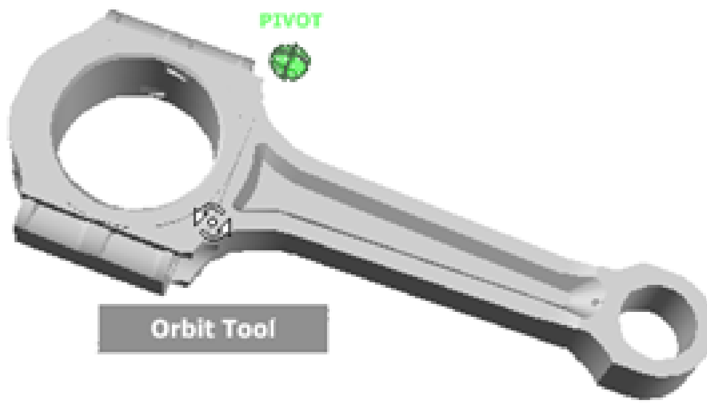
When using the Look command from the big Full Navigation wheel, you can walk through a model by using the arrow keys on the keyboard. Use the Properties dialog box for the steering wheels to adjust the walk command.

**Invert Vertical Axis**

When you drag the cursor upward, the target point of the view lowers; dragging the cursor downward raises the target point of the view. Use the properties dialog box for the steering wheels to invert the vertical axis for the Look command.

## Orbit Command

Orbit changes the orientation of a model. The cursor changes to the Orbit cursor. You drag the cursor to rotate the model around a pivot point, while the view remains fixed.




### Specify the Pivot Point

Orbit uses the pivot point as the base point when rotating the model. You can specify the pivot point in the following ways:

- **Default pivot point** When you open a model, the target point of the current view is the pivot point.
- **Select objects** Before you click Orbit, select objects. The pivot point is based on the center of the extents of the selected objects.
- **Center command** Specify a point on the model as the pivot point for orbiting with the Center command (page 71).
- **CTRL+Click and drag** Press and hold down**CTRL**before you click the Orbit wedge, or while the Orbit command is active. Then drag to the point on the model to use as the pivot point. This option is available only when using the big and mini Full Navigation wheels or the mini View Object wheel. **Note:** While the Orbit command is active, press and hold **CTRL** at any time to move the pivot point.

### Maintain Up Direction

To control whether the up direction is maintained for the Orbit command, use the properties dialog box for the Steering Wheels.

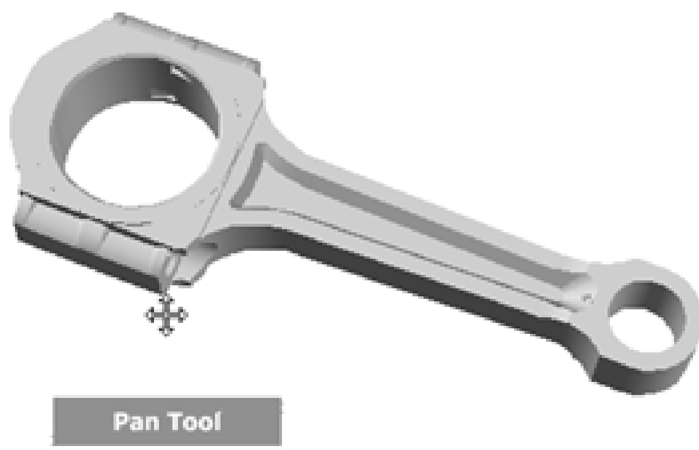
When the up direction is maintained, orbiting is constrained along the *XY* axis, and in the *Z* direction. If you drag horizontally, the camera moves parallel to the *XY* plane. If you drag vertically, the camera moves along the *Z* axis.

If the up direction is not maintained, you can roll the model using the roll ring, which is centered around the pivot point.

## Pan Command

When the Pan command is active, the Pan cursor (a four-sided arrow) displays. Drag the pointing device to move the model in the same direction. For example, drag upward to move the model up. Drag downward to move the model down.

In a 3D context, primarily when using 3D Steering Wheels, Pan dollies the camera left and right. In a 2D context, Pan scrolls the view. If you use Pan with an active view on a sheet, Pan scrolls the sheet view, not the active view on the sheet.




**Tip:** If the cursor reaches the edge of the screen, you can continue panning by dragging further to force it to wrap around the screen.

## Rewind Command

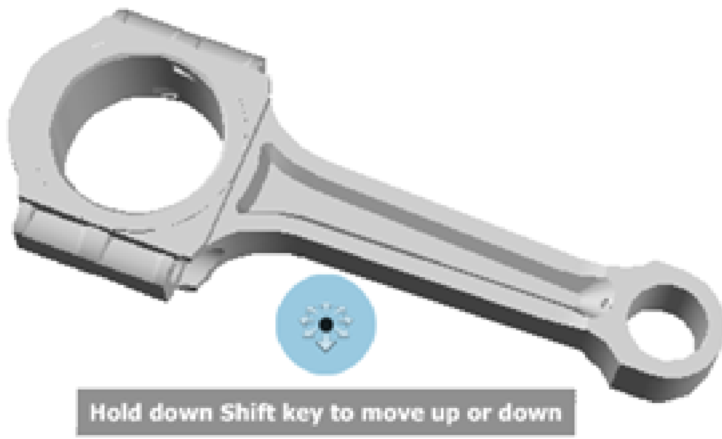
Retrieves previous views from the history of navigation commands that is saved as you reorient the view of a model.

The navigation history holds a representation of the previous views of the model, along with a thumbnail image. A separate navigation history is maintained for each window, until you close the window. Rewind navigation history is view-specific, and is not saved between sessions.

On the wheel, click Rewind and hold to display the Rewind History panel. Then scroll through the views in the navigation history. To restore a previous view, drag the bracket to the left in the Rewind History panel.


## Walk Command

With the Walk command, you can navigate through a model as if you were walking through it. After you start Walk, the Center Circle icon displays near the center of the view. The cursor display changes to a series of arrows. To walk through the model, drag in any direction.




### Constrain the Walk Angle

When walking through a model, you can constrain the movement angle to the ground plane. If the Constrain Walk Angle to Ground Plane option is enabled, you can freely walk around while maintaining a constant camera viewpoint elevation. If the walk angle is not constrained, you fly in the direction you are looking. Use the Properties dialog box for the SteeringWheels to constrain the movement angle to the ground plane for the Walk command.

### Movement Speed

As you walk or fly through a model, you can control the movement speed. The distance the Cursor moves from the Center Circle icon and the current movement speed setting control the movement speed. You can adjust the movement speed setting permanently or temporarily during the Walk command. To adjust the movement speed permanently, use the Properties dialog box for the SteeringWheels or the **<** and **>** keys when the Walk command is active. To increase movement speed temporarily, press and hold the + (plus) key while using the Walk command.

### Change the Elevation

As you use the Walk command, adjust the camera elevation by holding down `Shift`. It temporarily activates the Up/Down command. With the Up/Down command active, drag up or down to adjust the elevation of the camera. You can also use the `up arrow` and `down arrow` and keys as you walk to adjust the height of the view.

## Zoom Command

Zoom changes the magnification of a model. The following are mouse click and key combinations to control the behavior of the Zoom command:

- **Click** Click Zoom on a wheel to zoom the current view in by a factor of 25 percent. If you are using the Full Navigation wheel, requires that in the Properties dialog box for the SteeringWheels, incremental zoom is enabled.
- **SHIFT+click** If you hold down **SHIFT**key before you click Zoom on a wheel, the current view zooms out by a factor of 25 percent. Zooming is performed from the current location of the cursor, and not the current pivot point. If you are using the Full Navigation wheel, requires that in the Properties dialog box for the SteeringWheels, incremental zoom is enabled.
- **CTRL+click** If you hold down **CTRL**before you click Zoom on a wheel, the current view zooms in by a factor of 25 percent. Zooming is performed from the current pivot point, and not the location of the cursor.
- **Click and drag** To adjust the magnification of the model, click Zoom, hold down the button on your pointing device, and drag up and down.
- **CTRL+click and drag** When using the Full Navigation wheels or the mini View Object wheel, to control the target point used by Zoom, hold down CTRL. The Zoom command uses the location of the previous pivot point defined by the Zoom, Orbit, or Center command.
- **SHIFT+click and drag** When using the Full Navigation wheels or the mini View Object wheel, to zoom in to an area of the model, drag a rectangular window around the area. Hold down SHIFT, and then click and drag a window around the area. **Note** Hold down**CTRL**and**SHIFT**to zoom in to an area of a model using a center-based window instead of one defined by opposite corners.
- **Mouse wheel** When a wheel is displayed, to zoom the view of the model in or out, scroll the mouse wheel up or down.

**Note:** When you use Zoom from the Full Navigation wheel or the View Object wheel, the point in the view where you click to zoom is the center point for all Orbit operations. It remains the center point until you either use Zoom again, or use the Center command. If you press**CTRL**before you click the Zoom wedge, the Center point does not change.


**Zoom Constraints**

When changing the magnification of a model with Zoom, you cannot zoom in any closer than the focus point, or out any further past the extents of the model. The center point set by the Center command controls the direction you can zoom in and out.

**Note:** Unlike the Zoom command on the big View Object wheel, Zoom on the mini View Object and Full Navigation wheels are not constrained.

## Navigation Wheels

Wheels are available in two sizes: big and mini. The big wheel is larger than the cursor. A label is on each wedge in the wheel. The mini wheel is about the same size as the cursor. Labels are not displayed on the wheel wedges. The 2D Navigation wheel is only available in a big version.

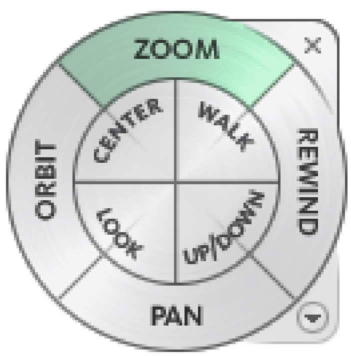
## 2D Navigation Wheel

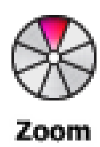
Provides basic commands for 2D navigation. It is useful when you do not have a pointing device with a scroll wheel.


- **Pan** Repositions the model on the screen.
- **Zoom** Adjusts the magnification of the current view.
- **Rewind** Restores recent views. Press the left mouse button to display thumbnails of the view history, then drag left or right to select a view.

## Full Navigation Wheels

The Full Navigation wheels (big and mini) combine the 3D navigation commands found on the View Object and Tour Building wheels. You can view individual objects, and walk through and around a model. The big and mini Full Navigation wheels are optimized for experienced 3D users.




**Note:** When one of the Full Navigation wheels displays, you can press and hold the middle mouse button to pan, and scroll the wheel button to zoom in and out. To orbit the model, hold down `SHIFT` while you press and hold the middle mouse button.

**Big Full Navigation Wheel Options**

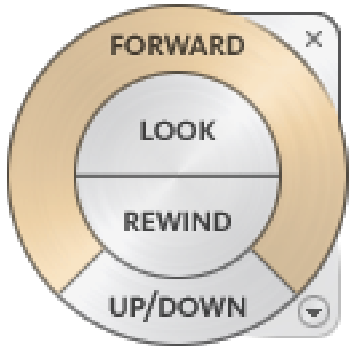
- **Zoom** adjusts the magnification of the current view.
- **Rewind** restores the most recent view. Move backward or forward by clicking and dragging left or right.
- **Pan** repositions the current view by panning.
- **Orbit** rotates the current view around a fixed pivot point.
- **Center** specifies a point on a model to adjust the center of the current view, or change the target point for navigation commands.
- **Walk** simulates walking through a model.
- **Look** swivels the current view.
- **Up/Down** slides the current view of a model along the Z axis of the model.

**Mini Full Navigation Wheel Options**

- **Zoom (Top wedge)** adjusts the magnification of the current view.
- **Walk (Upper right wedge)** simulates walking through a model.
- **Rewind (Right wedge)** restores the most recent view. Move backward or forward by clicking and dragging left or right.
- **Up/Down (Lower right wedge)** slides the current view of a model along the Z axis of the model.
- **Pan (Bottom wedge)** repositions the current view by panning.
- **Look (Lower left wedge)** swivels the current view.
- **Orbit (Left wedge)** rotates the current view around a fixed pivot point.
- **Center (Upper left wedge)** specifies a point on a model to adjust the center of the current view, or change the target point for navigation commands.

## Tour Building Wheels

With the Tour Building wheels (big and mini), you can move through a model, such as a building, assembly line, ship, or oil rig. You can also walk through and navigate around a model. The big Tour Building wheel is optimized for new 3D users while the mini Tour Building wheel is optimized for experienced 3D users.





**Options on Big Tour Building Wheel Wedges**

- **Forward** Adjusts the distance between the current point of view and the defined pivot point of the model. Clicking once moves forward half the distance as far as the object you clicked.
- **Look** Swivels the current view.
- **Rewind** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Up/Down command** Slides the current view of a model along the *Z* axis of the model.

**Options on Mini Tour Building Wheel Wedges**

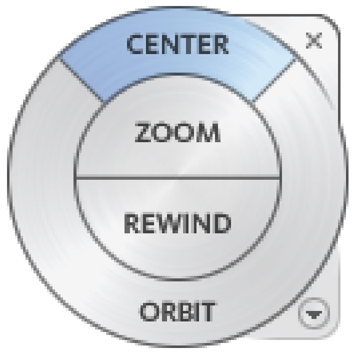
- **Walk (Top wedge)** Simulates walking through a model.
- **Rewind (Right wedge)** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Up/Down (Bottom wedge)** Slides the current view of a model along the *Z* axis of the model.
- **Look (Left wedge)** Swivels the current view.

When the mini wheel is displayed:

- To pan, press and hold the middle mouse button.
- To zoom in and out, scroll the wheel button.
- To orbit the model, hold the Shift key while pressing and holding the middle mouse button.

## View Object Wheels

With the View Object wheels (big and mini), you can view individual objects or features in a model. The big View Object wheel is optimized for new 3D users while the mini View Object wheel is optimized for experienced 3D users.





**Options on the Big View Object Wheel**

- **Center** specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation commands.
- **Zoom** adjusts the magnification of the current view.
- **Rewind** restores the most recent view orientation. You can move backward or forward by clicking and dragging left or right.
- **Orbit** rotates the current view around a fixed pivot point.

**Options on the Mini View Object Wheel**

- **Zoom (Top wedge)** adjusts the magnification of the current view.
- **Rewind (Right wedge)** restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Pan (Bottom wedge)** repositions the current view by panning.
- **Orbit (Left wedge)** rotates the current view around a fixed pivot point.

When the mini wheel is displayed:

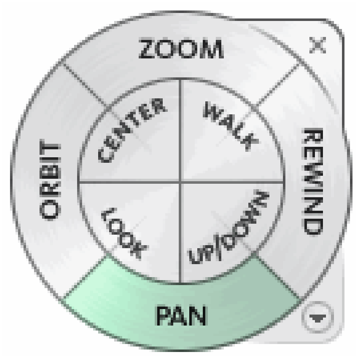
- To pan, press and hold the middle mouse button.
- To zoom in and out, scroll the wheel button.
- To orbit the model, hold down `Shift` while you press and hold the middle mouse button.

## SteeringWheels Overview

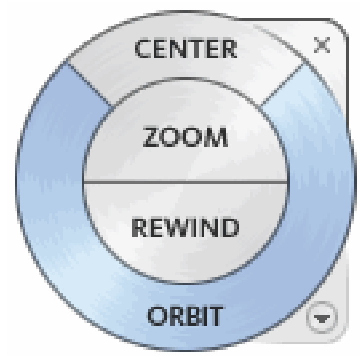
SteeringWheels, also known as wheels, combine many of the common navigation commands into a single interface. Wheels are specific to the context from which you view a model.

The following are the full and mini wheels available:

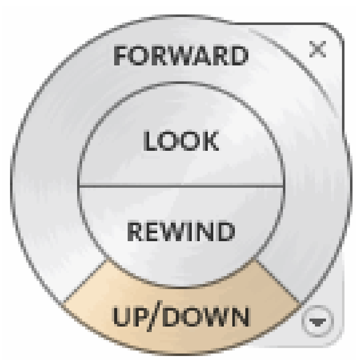
**Full Wheels**




Full Navigation Wheel



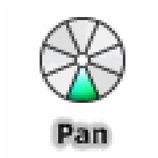

View Object Wheel




Tour Building Wheel


2D Wheel

### Mini Wheels




Mini Full Navigation Wheel


Mini View Object Wheel


Mini Tour Building Wheel

### Display and Use Wheels

The primary mode of interaction is to press and drag a wedge of a wheel. After a wheel displays, click and hold on a wedge to activate the navigation command. Drag to reorient the current view. To return to the wheel, release the button.

### Appearance of the Wheels

You can control the appearance of the wheels:

- **Adjust the size**. Wheels (except the 2D Navigation wheel) are available in two different styles: big and mini. The size of a wheel controls how large or small the wedges and labels appear on the wheel.
- **Adjust the opacity**. the opacity level controls the visibility of the objects in the model behind the wheel

### Wheel tooltip, command Messages, and command Cursor Text

As you move the cursor over the buttons on a wheel, a tooltip appears below the wheel and identifies the action to perform if you click the wedge or button.

Similar to tooltips, command messages and cursor text display when you use one of the navigation commands from a wheel. Command messages display when a navigation command is active. They provide basic instructions about using the command. Command cursor text displays the name of the active navigation command near the cursor. Disabling command messages and cursor text affects only the messages that display when using the mini wheels or the big Full Navigation wheel.

## Wheel Menu

Use the Wheel menu to switch between the big and mini wheels that are available. Go to the Home view, change the preferences of the current wheel, and control the behavior of the Orbit, Look, and Walk 3D navigation commands. The menu items available on the Wheel menu are dependent on the current wheel and program.

The Wheel menu has the following options:

- **Mini View Object Wheel** Displays the mini View Object wheel.
- **Mini Tour Building Wheel** Displays the mini Tour Building wheel.
- **Mini Full Navigation Wheel** Displays the mini Full Navigation wheel.

- **Full Navigation Wheel** Displays the big Full Navigation wheel.
- **Basic Wheels** Displays the big View Object or Tour Building wheel.

- **Go Home** Goes to the Home view saved with the model.
- **Fit to Window** Resizes and centers the current view to display all objects.

- **Restore Original Center** Restores the center point of the view to the extents of the model.
- **Level Camera** Rotates the current view so it is relative to the *XY* ground plane.
- **Increase Walk Speed** Increases the walk speed used for the Walk command by two times.
- **Decrease Walk Speed** Decreases the walk speed used for the Walk command by one half.

- **Orient to View** Orients the camera to match the view angle of the selected view (a plan, elevation, section, or 3D view).
- **Orient to a Plane** Adapts the view according to a specific plane.

- **Save View** Saves the current view orientation with a unique name.

  **Note:** Use Save View only to save a 3D view with a unique name when you are viewing the default 3D view. If you are viewing a previously saved orthographic 3D view or a perspective (camera) 3D view, the view is saved with the new orientation. You are not prompted to supply a unique name.
- **Increase/Decrease Focal Length** Acts as a zoom lens on the model, because it changes the focal length of the camera in a perspective view.
- **Move Crop Boundary** Moves the position of the crop boundary around in a perspective view.
- **Re-center Crop Boundary** Repositions the crop boundary to the center of the perspective view.

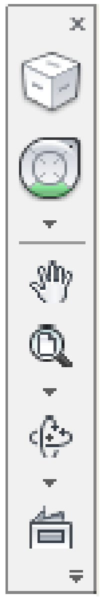
- **Help** Displays the Help system and displays the topic about the wheels.
- **Properties** Displays the dialog box where you can adjust the preferences for the wheels.

- **Close Wheel** Closes the wheel.

## Navigation Bar

The navigation bar floats over and along one of the sides of the window of the current model. It provides access to both unified and product-specific navigation commands.

To start a navigation command, click a button on the navigation bar, or select a command from the list that displays on a button drop down.



**Unified navigation commands** (such as Autodesk View Cube and Steering Wheels) are common across many Autodesk products. They are located on the two ends of the navigation bar.

- **View Cube** Indicates the current orientation of a model, and can reorient the current view of a model.
- **Steering Wheels** Collection of wheels that offer rapid switching between specialized navigation commands.

**Product-specific navigation commands** are unique to a product. They are located in the center of the navigation bar.

- **Pan** Moves the view parallel to the screen.
- **Zoom** A set of commands that increase or decrease the magnification of the current view.
- **Orbit** A set of commands that rotate the current view.
- **Look At** Views faces of a model from a selected plane.

### Reposition and Reorient the Navigation Bar

To adjust the position and orientation of the navigation bar:

- Link it to the View Cube. It displays above or below the View Cube in a vertical orientation.

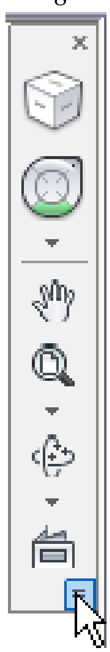
- Dock it when the View Cube is not displayed.
- When not linked or docked, use the grip handles to position it freely along any edge of the current window.

  If a side of the window is not long enough to show the entire navigation bar, it is truncated to fit. When truncated, a More Controls button displays and replaces the Customize button. Click the More Controls button to see more navigation commands.

You specify how the navigation bar can be repositioned from the Customize menu.

**Display of Commands on the Navigation Bar**

To control which unified and product-specific navigation commands display on the navigation bar, click the drop-down arrow at the bottom right of the navigation bar.



Click the Customize button, and then click the navigation commands to display on the navigation bar. You cannot change the position of the navigation commands on the navigation bar.

# Function Keys

Some mouse shortcuts and keys are reserved for specific purposes in Inventor Fusion, as shown in the following table:

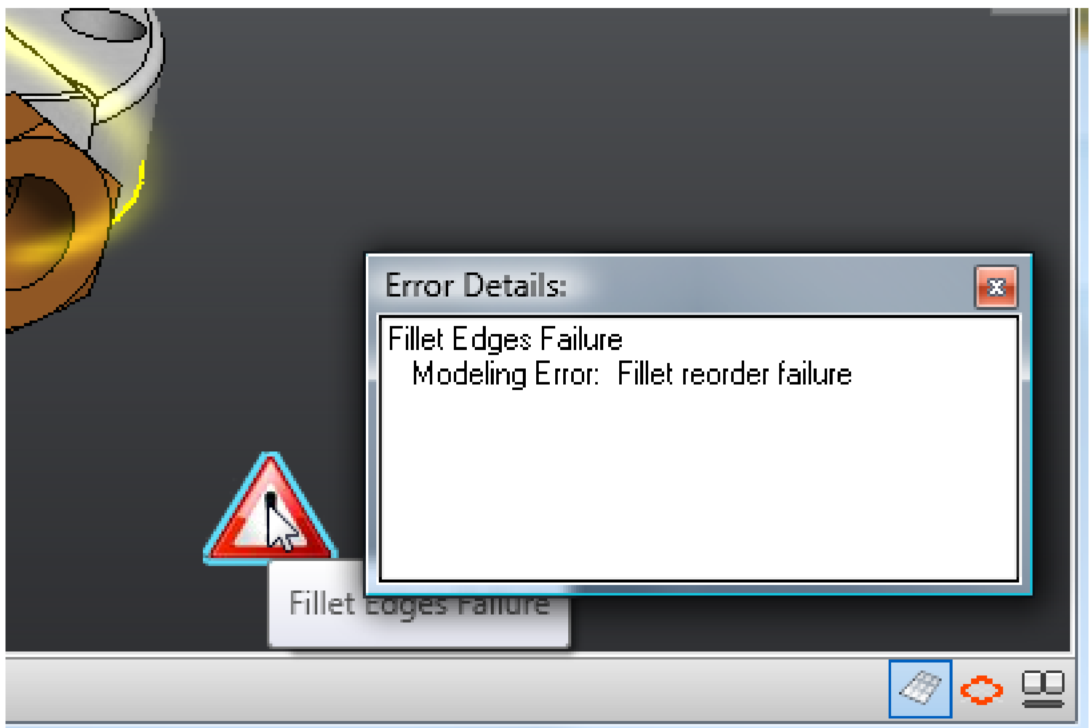
| **Function Key** | **Behavior** |
|---|---|
| F2 | Pan |
| F3 | Zoom |
| Shift+F3 | Zoom window |
| F4 | Orbit |
| F6 | Zoom all |
| F7 | Slice graphics. Requires that you first select a work plane or sketch profile, and then press F7. |
| F10 | Toggle shortcut key tips in the Application menu and Quick Access toolbar. |
| Middle mouse button | Pan |
| Mouse wheel | Zoom |
| `SHIFT`+Middle Mouse button | Orbit |

# Error Handling

When an error condition occurs, the Error Glyph displays in the lower right-hand corner of the application window:


Pause the cursor over the glyph to display a tooltip with a brief description of the error:


Click the error glyph to display a dialog box with more information.




The error glyph disappears when the condition is corrected, or the command is canceled. In this case, the fillet radius was too large. You can either make the radius smaller or cancel the command.

If an error occurs outside of a command, the error glyph continues to display until a command is started.

# Modeling in Fusion

# 4

A wide range of modeling techniques are available to create and edit model geometry and components in Inventor Fusion designs.

**Simple design in Fusion**

Key elements of the design:

- The Solid item in the browser contains the (single) solid body for a component. All Fusion components are limited to a single solid body.
- The Surface Bodies node in the browser. A component can own 0 or more surface bodies, surface quilts, and surface groups. This example has two surface quilts.
- The Sketches folder in the browser contains all the sketches for a component. This example has a single sketch, which is invisible. The light bulb icon shows the visibility state for many items in a Fusion design Sketches folder in the browser.
- The Feature nodes in the browser represent geometry which was created using a feature command in Fusion. In this example, the design has two features: Extrude1, and Hole2. The nodes underneath Feature nodes show individual body faces that the feature created.
- The component body shows the geometry of the component in the graphic area (solid and surfaces). You perform most operations on this representation of the component

In addition, an Inventor Fusion design can contain:

- Work geometry
- Annotation planes and annotations
- Named views
- Child components
- Favorites folders and search folders

### Complex design in Fusion

Example of a more complex design:

Key elements of the design:

1 More surface groups and surface bodies
2 Work axis
3 Child component (Component2)
4 Second-level of component (Component3)
5 Annotation plane

## Create Models in Fusion

A Fusion model is made up of features that correspond to the physical shape of a part. Typical model features include extrusions, holes, fillets, and chamfers.

This video demonstrates the creation of a solid body in Fusion.

**Show me how to create a solid body**

This video demonstrates how to create a solid body, and additionally a hole and fillet.

### Overview of modeling in Fusion

A feature can be a solid or a surface. A solid has all of the faces for a shape. For example, a solid cylinder has the outside cylindrical face and the ends. A surface is a face that does not have a thickness, or has multiple faces with openings. For example, a cylindrical surface just has the outside cylindrical face without the ends.

Fusion uses direct modeling instead of the parametric modeling used in Inventor. As a result, the concept of a feature differs in Fusion:

- Direct modeling features are independent while parametric features have relationships. You can change Inventor Fusion features without impacting other features.
- Direct modeling features do not have a history. If you create a feature, like an extrusion, from a sketch, the feature does not update if you change the sketch later.
- Direct modeling allows you to copy and paste features between different faces or even components. For example, I can copy a hole and paste it on a different face in a component, or paste it on a solid in a different

component. Inventor Fusion sees a feature as a collection of faces. It does not consider where it is located, as long as it can create the shape again.

- Inventor Fusion does not have a history, parameters, or relationships. You cannot create a model that automatically updates several features by changing a driving parameter. While this fact makes it less powerful than an Inventor part. You do not have to be a parametric modeling expert to create a model.
- Inventor Fusion features can be edited directly. You can change the size or shape of a feature by using Move or Push-Pull. You do not have to worry about cascading feature failures when you edit a feature. For example, if you change the size or shape of a feature with a hole, the hole is automatically deleted if it does not intersect any more. If you want to keep the hole, you can move it to a different location on the feature before you change it.

Use features to create solid bodies in Fusion. Some features, such as extrude and revolve, require 2D sketches to create the feature. The primitive feature commands combine sketching and feature creation into one operation to simplify modeling.

### Feature Command UI

The HUD is the primary command interface. The HUD for a feature includes value entry fields, manipulators to change the size dynamically, and glyphs for selecting options. Most feature commands have a command ribbon in addition to the HUD.

Related topics:

- Sketching: Sketch (page 104)
- Feature creation: Features
- Primitives: Primitive commands (page 163)

## Browser

The browser presents an organized view of the data in your design in a tree format, like the navigation pane in Windows Explorer. Context menus have tools and commands for the selected object type. Some are specific to the browser and some are also available in the graphics window.

Objects are cross-highlighted in the browser and graphics window. Selecting an object in the browser also highlights it in the graphics window, and selecting an object in the graphics window highlights it in the browser. Cross-highlighting provides visual feedback and simplifies selecting a particular object so you can edit it in the other environment.

### Browser items and controls

Some items have links to sections and topics with more information.

1. Turn the Favorites (page 13) browser on and off. The Favorites browser displays at the bottom of the browser pane.
2. Click the light bulb to turn visibility of an object on or off.
3. Toggle features On and Off.
4. The Named Views folder has default standard views. You can select New Named View from the context menu to save the current model orientation and zoom.
5. The Origin folder has the X-Y-Z coordinate system. Components and created features are positioned relative to the parent coordinate system.
6. Annotation Planes are created for dimensions that document the model.
7. A blue background indicates the active component.
8. Constraints position components relative to each other.
9. Child Components (page 372) can contain features and other components. This hierarchy organizes the model by grouping related features and components together.
10. A component can have one solid body (page 163) . A solid body usually has multiple features, and there can be gaps between features.
11. A component can have multiple surfaces (page 221). A surface can be independent, and multiple surfaces can be stitched together to form a quilt.
12. Sketches (page 104) contain geometry for creating features.
13. There are two types of features, created and placed. Created features, such as extrude and revolve, are new entities in the graphics window. Placed features, such as fillets and holes, modify existing features.
14. Objects can be mirrored and patterned (page 326) to organize the browser and capture information on how objects are related. For example, if a component has a row of holes, a pattern locates the holes and groups them in the browser.
15. Turn the Information Panel on and off to display information about a browser object.

**16** The Information Panel displays when you pause the cursor over a browser object. Some objects, such as the Sketches folder, do not have information panels.

### Favorites

Click on Toggle Favorites to display the Favorites browser at the bottom of the model browser. It is minimized by default and you drag the arrow to increase the browser height.

You can drag copies of browser items, such as components and features, to Favorites. You can add the same item to multiple sub-folders to improve organization. For example, you can have separate folders for features, for components, and features for a specific component.

You can search the Favorites browser using tags. A search results folder is created that automatically updates as items are added or remove. There are two types of predefined tags- feature names, such as Fillet, and feature dimensions, such as Fillet Radius. Click on the Expand button to display the feature dimension fields. You can also create User Tags (page 455) to identify items.

### Browser Context Menus

The browser has tools for common tasks on the context menus. The context menus vary depending on the type of node.

### Light Bulb

**Isolate Component** Hides all other components.

**Unisolate Component** Is added to the context menu on the top browser node when a child component is isolated. It makes all components visible again.

### Component Context Menus

**New Component** Creates a child component. Child components are typically used for assemblies. For example, a cover is usually a child component.

**Activate Component** Sets which component you are working on. Features are created in the active component. If you accidentally create a feature in the wrong component, you can cut and paste it into the correct one.

**Save As** Creates a separate document for the selected child component.

**Make Independent** Breaks the link between a copied component and the original.

**Grounded** Fixes a component in its current location. For example, an imported assembly comes in as separate components. The components are not constrained, so you can drag them out of position. To keep the components in their original locations, select all of the components and set them to Grounded.

**Insert Component** Imports a file and adds it as a child component.

**Material (page 413)** Sets the material type and the visual appearance. Some materials, such as rubber and copper, have a visual style. You can override the visual style by changing the color or texture. If you change the visual appearance, Remove Appearance Override is added to the context menu.

**Dissolve Features** removes the individual feature nodes in the component but leaves the geometry.

### Solids and Surfaces context menus

**Validate (page 269)** Checks the solid or surfaces for issues and tries to repair bad surfaces.

**Material (page 413)** Sets the material type and the visual appearance. Some materials, such as rubber and copper, have a visual style. You can override the visual style by changing the color or texture. If you change the visual appearance, Remove Appearance Override is added to the context menu.

**New Group** (Surfaces Only) Creates a sub folder for organizing surfaces.

**Stitch (page 259)** (Surfaces Only) Starts the stitch command to combine individual surfaces into a quilt.

**Unstitch** (Surfaces Only) Starts the unstitch command to separate a quilt into the individual surfaces.

**Reverse Normal (page 267)** (Surfaces Only) Changes which side of the surface is positive. For example, the Z-axis of an XY plane is positive in one direction and negative in the other.

**Thicken (page 213)** (Surfaces Only) Offsets a surface and adds side faces to create a solid.

### Sketches

**Slice Graphics** Cuts the model graphics across the sketch plane and displays the internal components. Right-click on the sketch and select Slice Graphics again to display the entire model again.

### Assembly Constraints

**Suppress** Temporarily turns off the constraint . You can move the components. They return to the original position when the constraint is unsuppressed.

### Features

**Recognize As Revolve** Changes the feature type to Revolve. This works on features such as a cylindrical extrude, but does not work on a rectangular extrude.

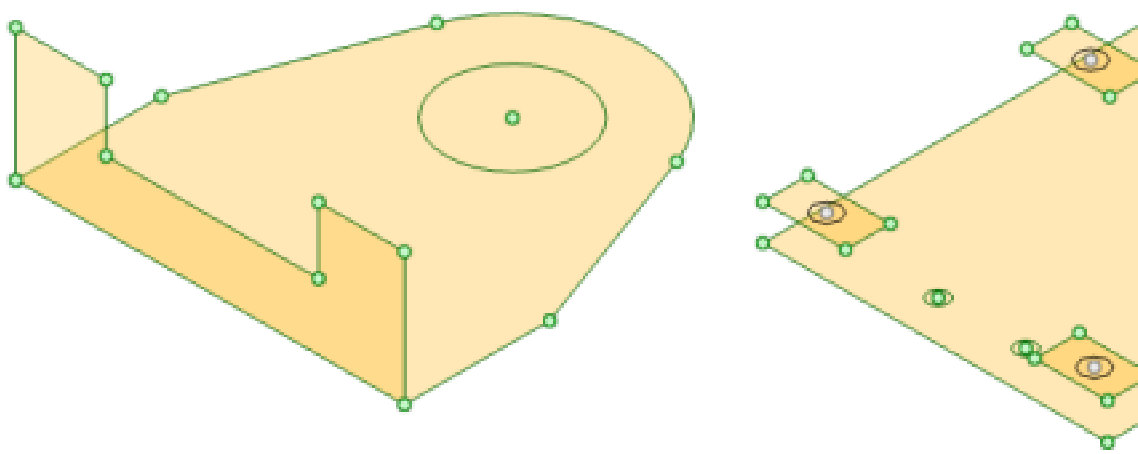
**Dissolve**Removes the feature node from the browser but leaves the geometry in the model.

## Sketch Environment

Sketches are used to create some features, like extrusions, and to lay out component details. This section explains the various commands to create and edit sketches.

## Plan and create sketches

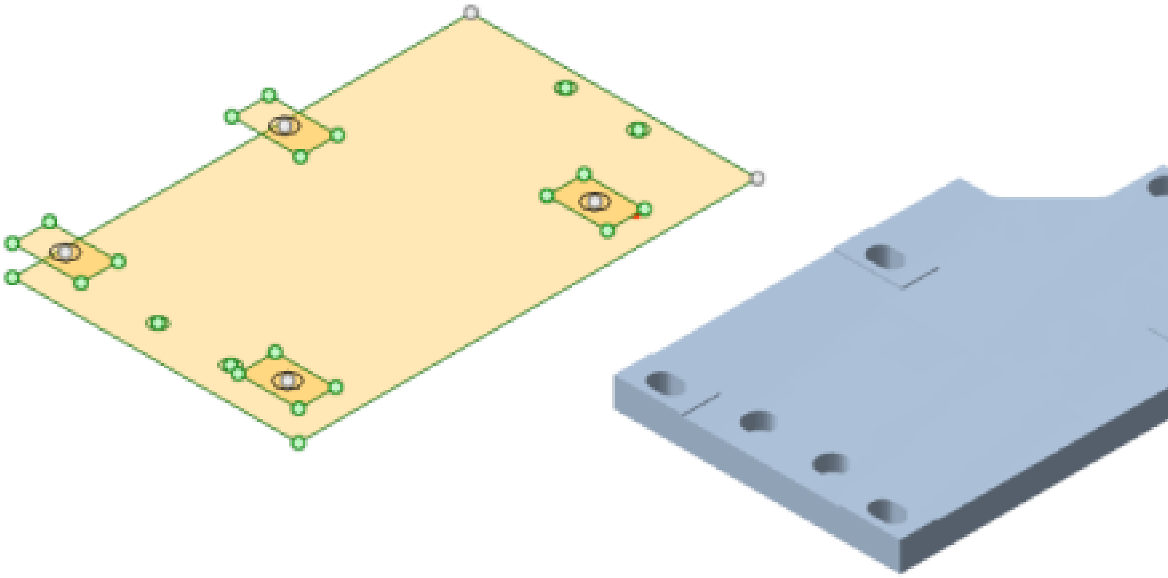
Define your design and create features with sketches. A layout sketch contains the references between important features. Detail sketches have the exact shape and size of features. Inventor Fusion provides several methods for creating sketches.



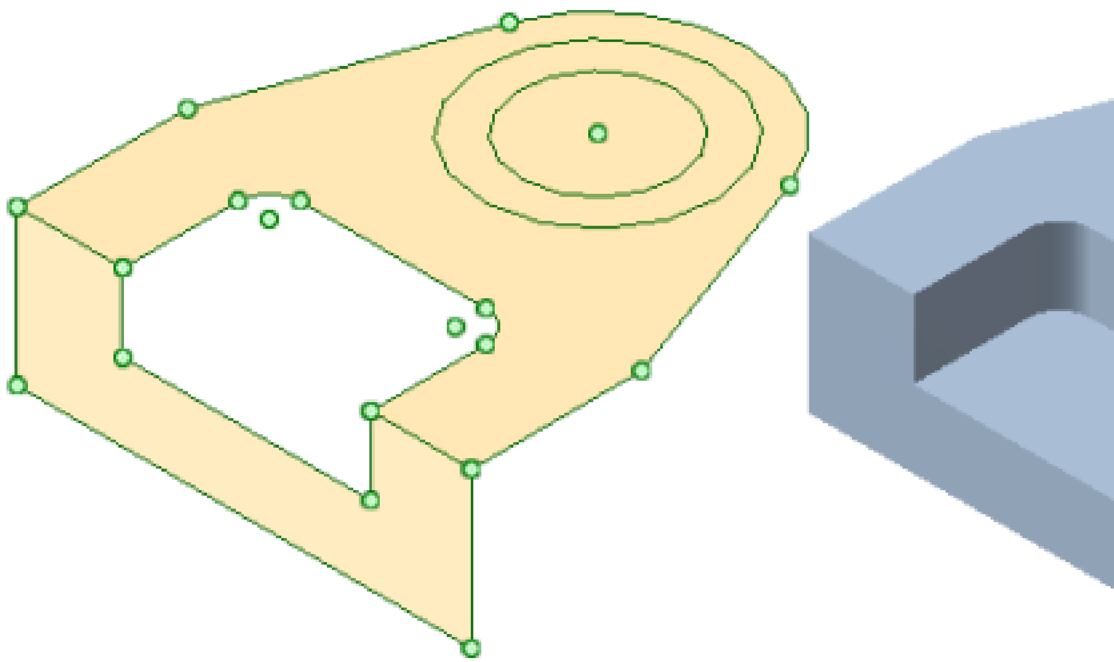
### Plan sketches

There can be several layout sketches for a component. Most components are attached to other components. One layout sketch usually contains details, like the size of a mounting base and the mounting hole locations. It sometimes contains sketch points and lines to locate other features relative to the mounting. Layout sketches usually do not have details like fillets.

In the image below, one layout sketch has the overall size and mounting holes for the plate. The second layout sketch has the height and mounting details for a component.



Detail sketches contain the exact profile for a feature. In the image below, the sketches contain the details for the model profiles.



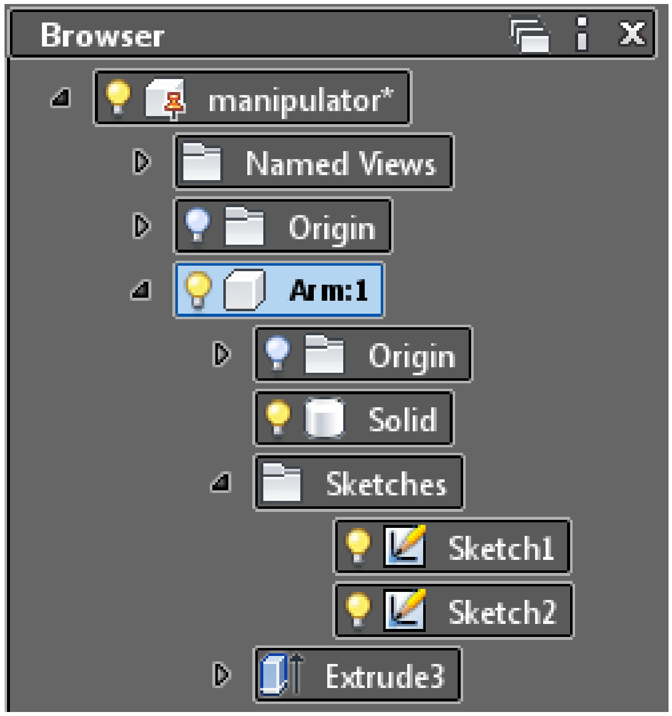
Inventor Fusion is a direct modeler, so it does not link features to sketches. Therefore, changes to sketches and features do not affect each other.

In many cases, a detail sketch is disposable. After you create the feature, you can delete the sketch. Keep a detail sketch if you want to reuse it, or use it for future reference.

Since layout sketches capture design intent, it is common to keep them. If you update the model at a later date, it is helpful to refer to original layout sketch. Some companies create new layout sketches to capture changes at each stage of the design process.

### Create sketches

Sketches are created in the active component (page 372). In the browser, the sketch is added to the Sketches folder under the component. There can be multiple sketches in a component, but only one is active at a time.




You can use New Sketch to create a sketch, or you can start a sketch command and create the sketch as the first step. Starting a sketch command first simplifies the workflow.

If there is already a visible sketch on the face or plane, or if you select sketch geometry, that sketch is edited. No new sketch is created. The direct modeling workflow does not link a sketch to a particular feature. It simplifies the model to have just one sketch per plane or face.

## Work with Sketches

If you start New Sketch and select sketch geometry or a plane that already has a sketch, that sketch is activates, and a new sketch is not created.

#### Methods to edit a sketch

- In the browser, right-click a sketch, and select Edit Sketch. The parent component is also activated.

■ In the graphics window, right-click and select a sketch tool from the context menu.

### The Mini-toolbar

A glyph is displayed when a sketch is active. Pause cursor on the glyph to display a mini toolbar. It has buttons for Stop Sketch, Reorient, and Look At.


Stop Sketch exits the active sketch.

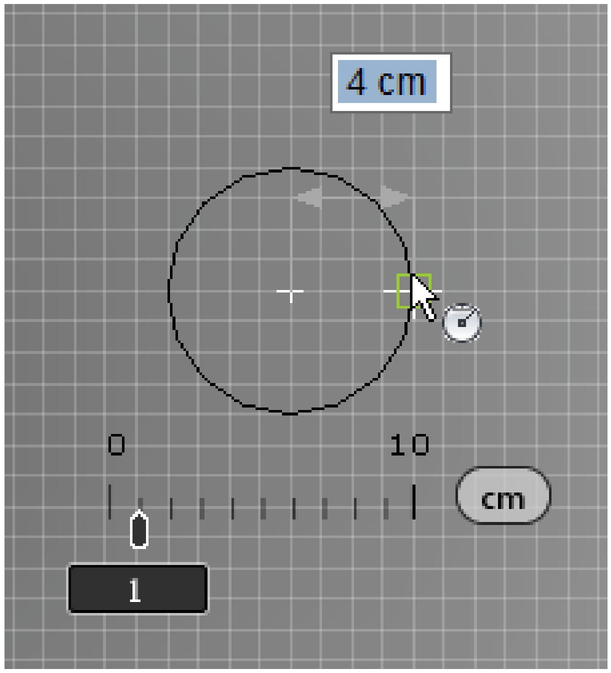
Reorient displays the Move triad so you can realign it by selecting geometry, a work feature, or a sketch entity.

Look At Sketch Grid changes the camera position so the sketch is parallel to the screen.

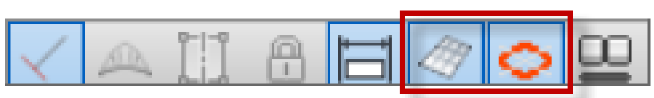
### Sketch Grid and Snap Bar

The sketch grid displays when a sketch is active. The grid spacing automatically updates as you zoom in and out, so the snap distance is appropriate for the zoom factor. Precision increases as you zoom in, and prevents the spacing from being too small when you zoom out.

The grid spacing displays on the snap bar in the lower right-hand corner. If you prefer a static value, you can manually override the grid spacing. The override is maintained unless the grid is not visible when you zoom in. The spacing is temporarily changed, and the original override value is used when you zoom out again.

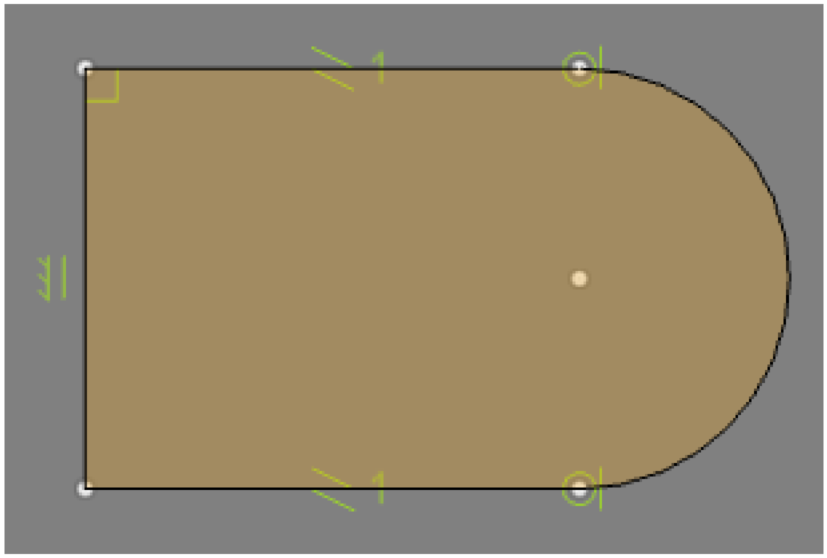



The status bar contains buttons for Sketch Grid Mode, which turns visibility on and off, and Snap Mode, which turns snapping on and off. Snapping is disabled when the sketch grid is not visible.



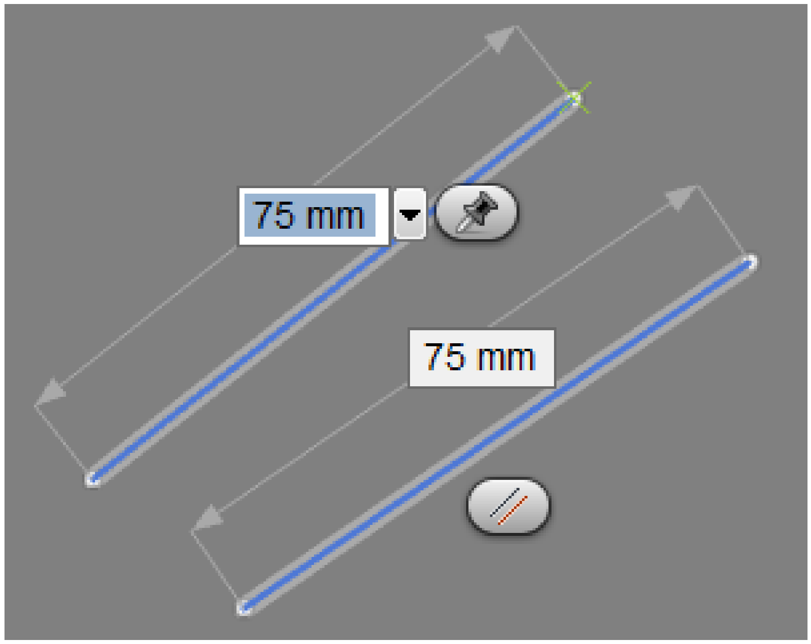
### Constraints

Sketch constraints (page 116) are created automatically (inferred), as you sketch. Design intent and relationships between sketch entities are captured.



If you do not want to create constraints automatically, press **CTRL** while sketching. Coincident constraints on points are still created, but relationship constraints, like horizontal or perpendicular, are not created

You can add constraints to existing sketch entities. When under-constrained entities are selected, a constraint glyph displays. You can click the glyph to apply that constraint, or pause over the glyph to display a list of valid constraints.




The Constraint button on the status bar turns the constraint visibility on and off. Constraints are still inferred when constraint visibility is off.

### Dimensions

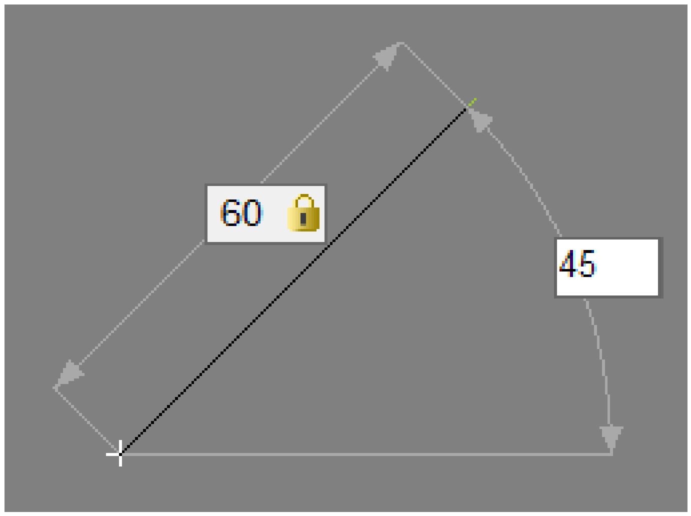
Fusion has two types of dimensions (page 121): transient and persistent. Transient dimensions do not control the size of the entity. Persistent dimensions control the size of an entity, so you cannot change the size by dragging or changing related entities.

Transient dimensions display only when you click an entity. They report the current size of an entity. You can change the size of an entity by dragging or changing related entities.

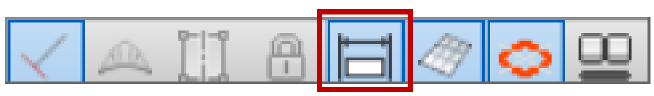
Persistent dimensions display when a sketch is active. You can change only the size of an entity by clicking on the dimension and entering a new value or turning persistent off.

### Precise Input

Precise Input provides value entry boxes so you can specify the entity size. By default, value entry boxes display as you create sketch entities. The primary value highlights, and you can type a new value. If you want to enter a secondary value, such as the angle of a line, you can press Tab to highlight that box. If you enter a value and Tab to the next box, that value locks. It does not update as you move the cursor or enter a value in the other box.




The Precise Input button on the status bar turns the value entry boxes on and off.



#### Lock Geometry

Lock Geometry (page 124) is a method to fix geometry without adding constraints and persistent dimensions. Locked geometry displays in green.

#### Reorient Sketch

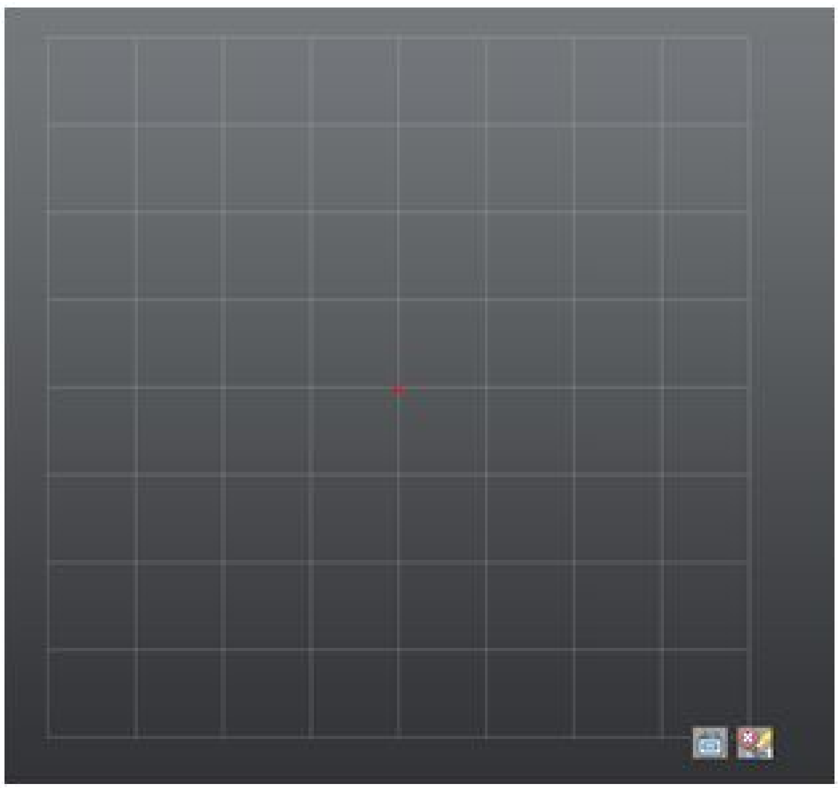
1 Click Reorient Sketch. The valid triad controls display.
2 Click a manipulator to move or rotate the sketch.
   - The arrow manipulators align the X or Y axis with a linear object.
   - The arc manipulator rotates the sketch.
   - The ball manipulator moves the sketch to the selected point.
3 Press OK to accept the changes

### Move the Sketch

1. On the ribbon Solid or Surfaces panel, click Move.
2. In the browser, or in the graphics window, select a sketch.
3. Use the triad to move and rotate the sketch plane.

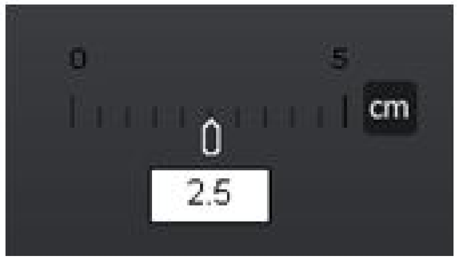
## Sketch Grids

When a sketch plane is activated, a sketch grid is drawn that encompasses all the sketch entities on the plane. The camera updates automatically to look at the sketch plane. If there are no entities on the plane, the grid assumes a default size.



The sketch grid has two icons associated with it:

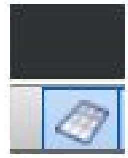
- **Look-At** If you perform any camera operations that change the orientation of the plane, changes the camera to look at the sketch plane.
- **Stop Sketch** Deactivates the sketch, and reverts to the select mode.

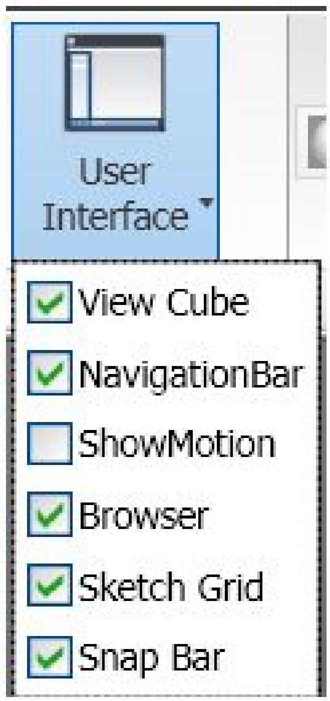



The grid provides precise snap points defined by its spacing. A red rectangle indicates a snap point.


Snap points are available throughout the sketch plane. They are not restricted to the visible Grid.

You can hide the Grid in two ways:

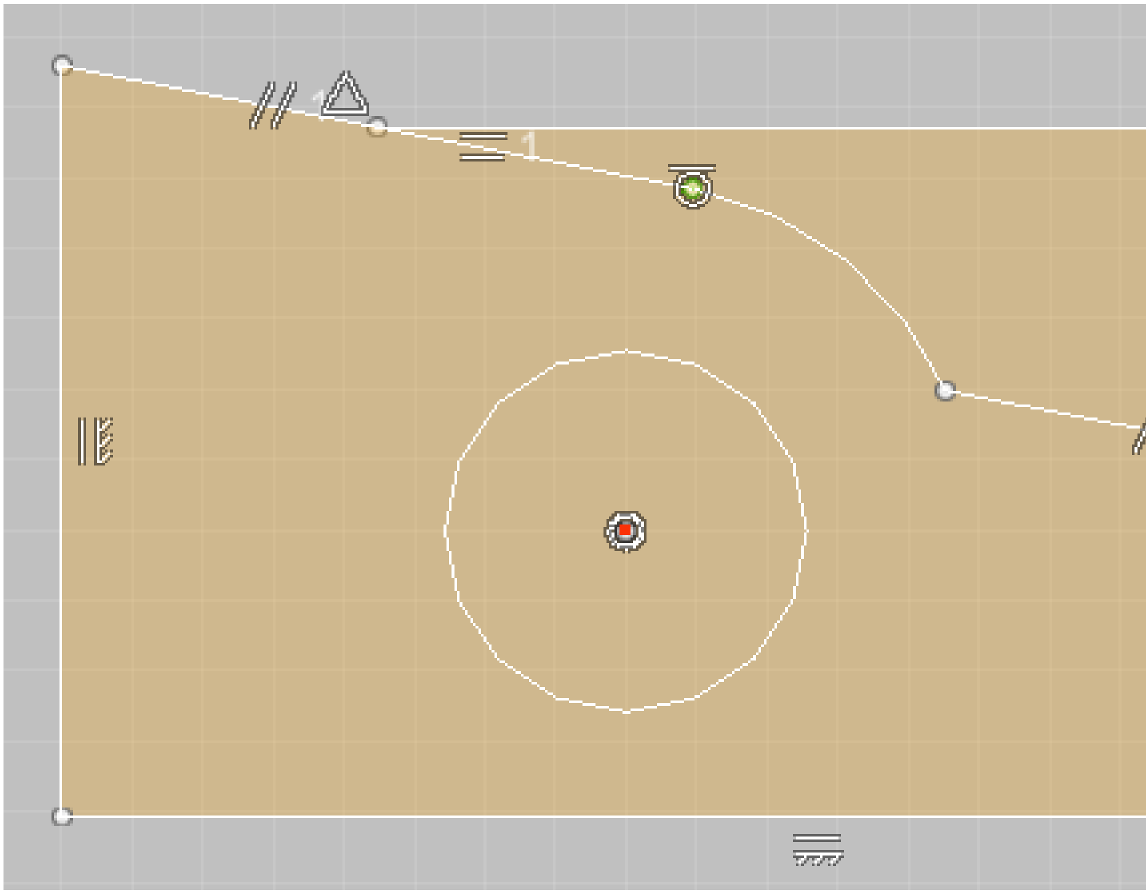
- Click the Grid icon at the bottom right of the toolbar. 
- On the View Tab, in the User Interface drop-down menu, clear the Sketch grid check box.




The snap bar at the bottom-right of the canvas controls the spacing of the sketch grid. If the snap bar is set to 0, the Grid assumes a default spacing value.

## Sketch Constraints

Constraints control the geometric relationships, such as parallel or tangent, between sketch entities. Sketch constraints are visible only while editing the sketch.

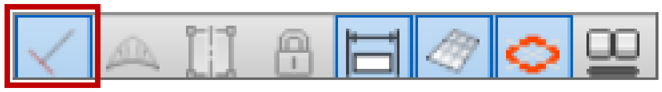


Constraints are automatically created or inferred, while sketching. Constraint symbols dynamically display as you sketch, and the cursor snaps to that position. For example, if a tangent constraint displays while sketching a line, the line preview stays tangent as you move the cursor in that direction.

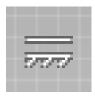
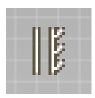
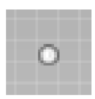
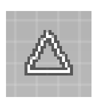
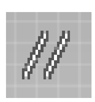
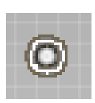
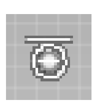
To disable inferred constraints, press **CTRL** while sketching. Coincident constraints on points are still created, but relationship constraints like horizontal or perpendicular are not created. This behavior is useful when creating a complex sketch to avoid creating unnecessary constraints.

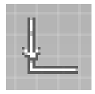
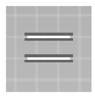
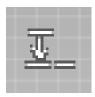
Add constraints to existing geometry by dragging an entity to infer a constraint, or by selecting entities to add a constraint manually.

The Constraint button on the status bar turns the constraint visibility on and off. Constraints are still inferred when constraint visibility is off.



**Constraint types**

| Constraint | Glyph | Description |
|---|---|---|
| Horizontal |  | Line parallel to horizontal sketch axis |
| Vertical |  | Line parallel to vertical sketch axis |
| Coincident |  | Point to point or snap<br>Point to entity |
| Midpoint |  | Point to midpoint of line |
| Parallel |  | |
| Concentric |  | |
| Tangent |  | |

| Constraint | Glyph | Description |
|---|---|---|
| Perpendicular |  | Inferred to circles, arcs<br>Manually added to circles, arcs, lines |
| Equal |  | Manually added only |
| Collinear |  | Manually added only |

### Inferred Constraint Priority

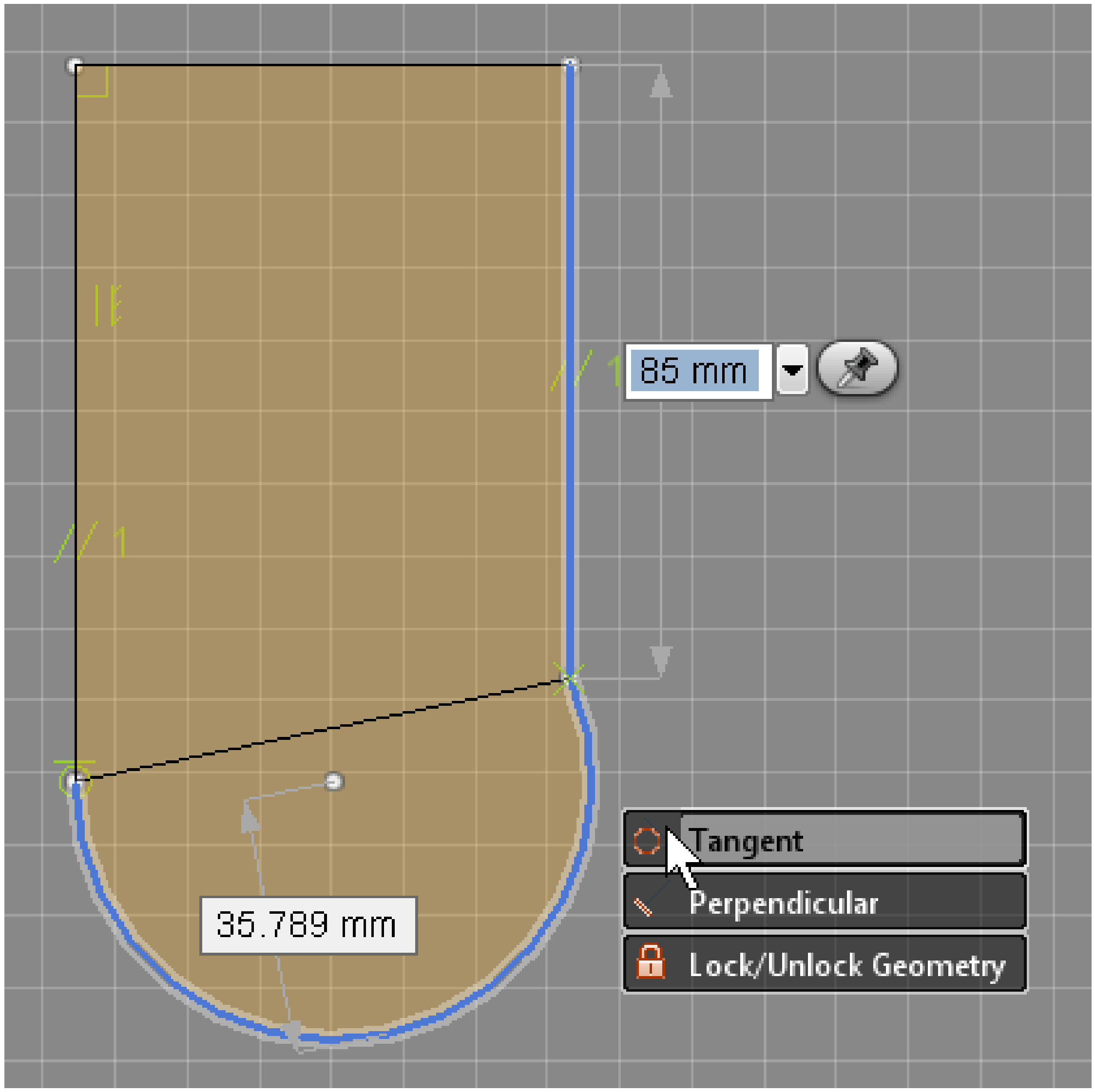
When multiple constraints can be inferred, the constraint with the highest priority displays. This list shows the priority of inferred constraints:

1. Coincident
2. Concentric
3. Tangent
4. Perpendicular
5. Horizontal
6. Vertical
7. Parallel

The equal and collinear constraints cannot be inferred. You can manually add these constraints after creating the entities.

### Add Constraints Manually

When you select two entities, a constraint glyph displays with the highest priority constraint. Click the glyph to apply that constraint, or pause the cursor over the glyph to display a list of valid constraints.

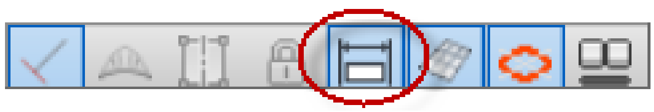

85 mm
35.789 mm
Tangent
Perpendicular
Lock/Unlock Geometry

**Show me how to use sketch constraints**

## Sketch Dimensions

Sketch dimensions define the size of entities. You can enter dimensions as you create entities using the Heads Up Display (HUD), or freeform sketch and change the sizes later.

The Precise Input button on the status bar turns off the HUD. Dimensions cannot be created or edited when Precise Input is off.



There are two types of sketch dimensions: transient and persistent. Transient dimensions are created by default. The key difference is that transient dimensions display only the current size, while persistent dimensions control the size.


Transient Dimension


Persistent Dimension

Click the pushpin to switch dimensions between transient and persistent.

### Transient dimensions

You can select several entities to display the transient dimensions. If you edit the transient dimension for one entity, the other entities do not change. This technique controls sketch updates without adding constraints or making dimensions persistent.

Fusion does not store transient dimensions. Transient dimensions display only when you click an entity.

To edit a transient dimension, enter a value in the edit box, or drag the entity. If you drag the entity or add a sketch constraint, the new size displays the next time you click it.

You can select several entities to display the transient dimensions. If you edit the transient dimension for one entity, the other entities do not change. This technique controls sketch updates without adding constraints or making dimensions persistent.

### Persistent Dimensions

Inventor Fusion uses direct modeling, so there are no links between sketches and features. This means that changing a sketch dimension does not update the feature. As a result, most dimensions can be transient. Persistent dimensions are useful in certain situations.

**Persistent dimensions:**

- Display when the sketch is active. It is not necessary to click an entity to see the size.

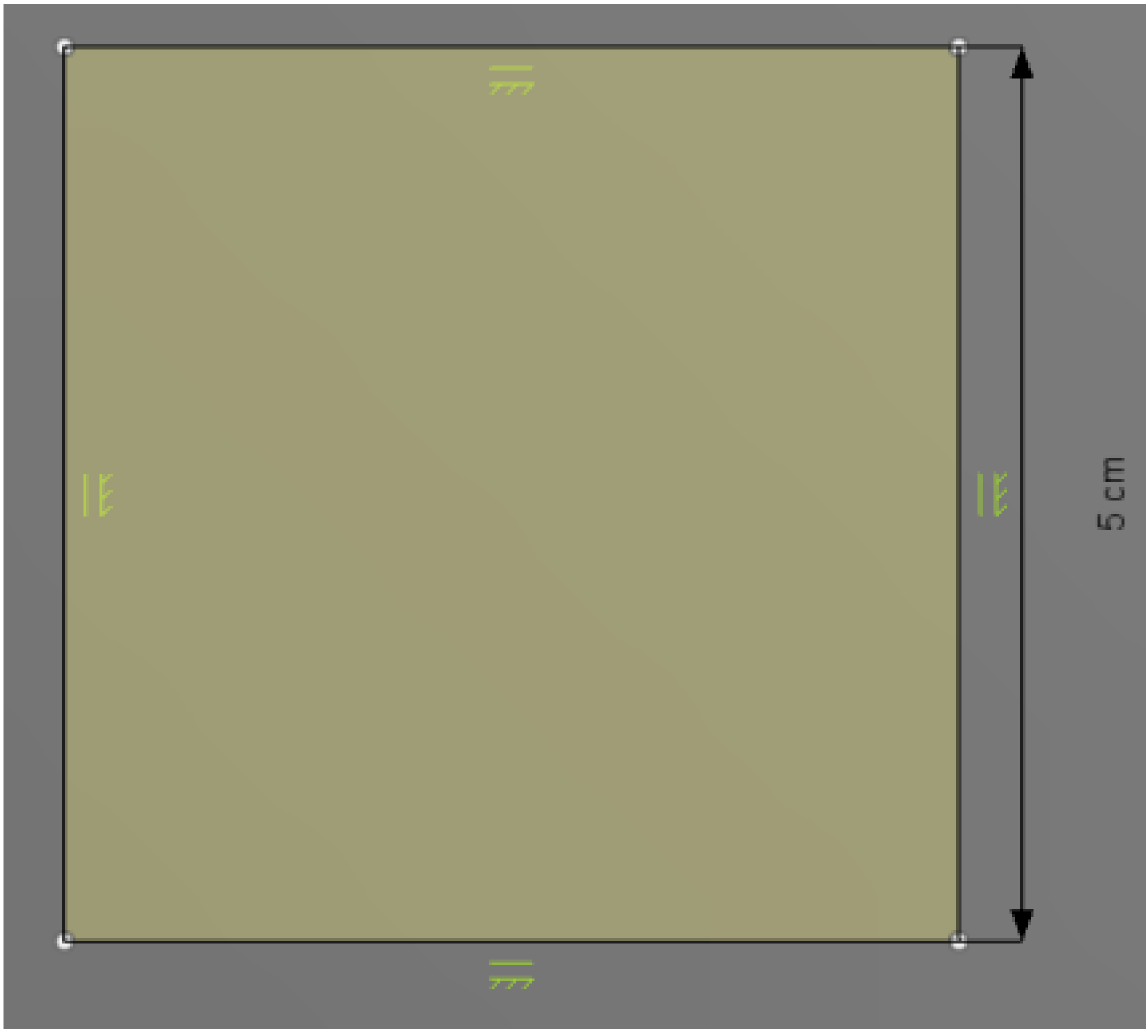
  **NOTE** Persistent dimensions are not visible when Precise Input is turned off.

- Prevent accidental changes to entity size. You can maintain the size of some dimensions while editing others.
- Can be changed when the entity is locked. Entities with transient dimensions must be unlocked to edit them.
- Capture useful design intent in sketches, such as the location or size of key features. If you change features after creation, you can refer to the sketch and see the original dimensions.

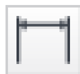
Fusion stores persistent dimensions. Persistent dimensions display when a sketch is active. You cannot drag the entity to change the size.

Sketch constraints (page 116) can change the position and size of entities. An error displays if the constraint cannot update an entity with a persistent dimension.

The vertical persistent dimension displays whenever the sketch is active. The horizontal transient dimension does not display until you click the line.


5 cm

### Dimensions between sketch entities



Creates dimensions between sketch entities. The dimension is the shortest distance between the selections.

1. Select two entities, and a dimension is automatically created. For linear dimensions, the dimension is the shortest distance between the entities.
2. Double-click the dimension to edit it. The sketch dimension and transient dimensions for the entities are active.

---

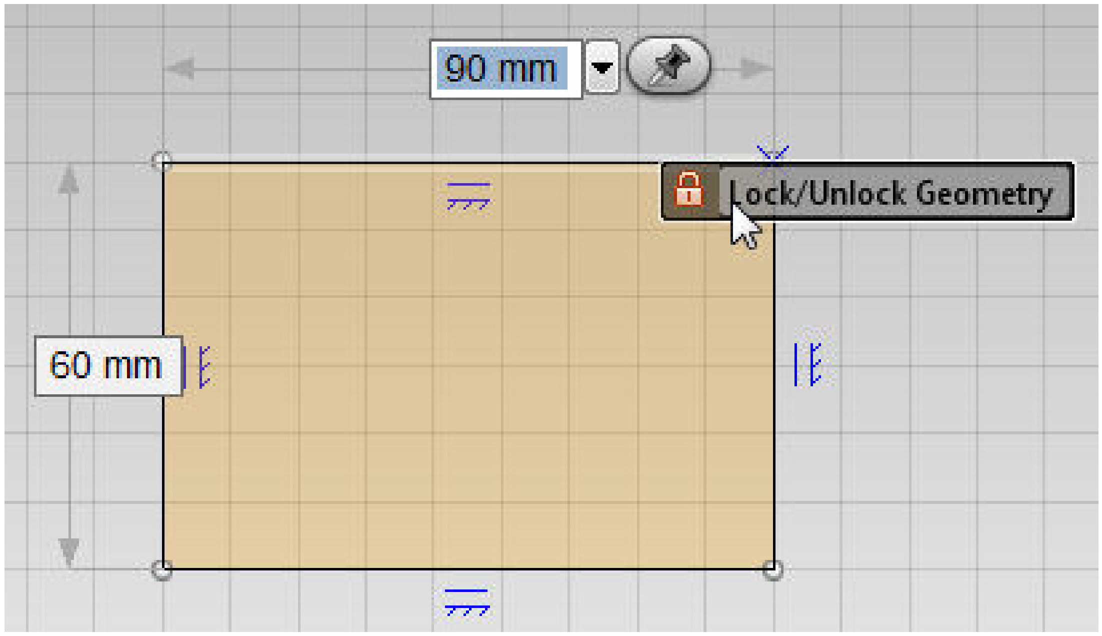
**NOTE** For some entities, such as arcs, you cannot place a dimension between the curve and another entity. If you select the entity, a radius dimension is created. You can create a dimension between the center point and other entities.

---

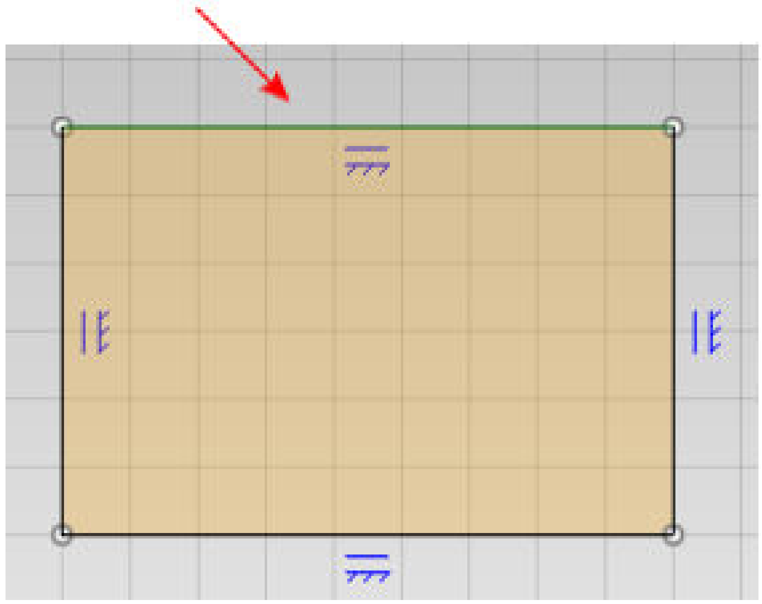
 **Show me how to create and edit sketch dimensions**

## Lock Sketch Geometry

Locks or unlocks a sketch entity. Select the entity, move the mouse over the lock icon, then select Lock/Unlock Geometry. To unlock it, select the icon again.




When an entity is locked, it displays in green. Edges you create are unlocked by default. Projected geometry (page 156) is locked by default.



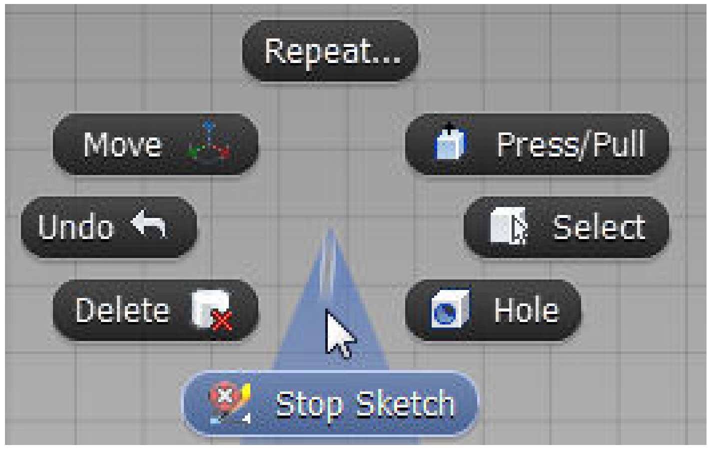
Locking a curve does not automatically lock its endpoints or vertices. In the previous image, the line is locked, but its endpoints are not locked. You can move the endpoints, but you cannot move or rotate the line. This state reduces the edit operations you can perform on the curve and on curves or points connected to it.

Locking geometry does not eliminate any inferred sketch constraints. You cannot edit entities in a way that breaks those constraints.

## Stop a Sketch

To exit the sketch environment, do one of the following:

- On the sketch mini toolbar, click the Stop Sketch glyph  .
- On the marking menu, click Stop Sketch.

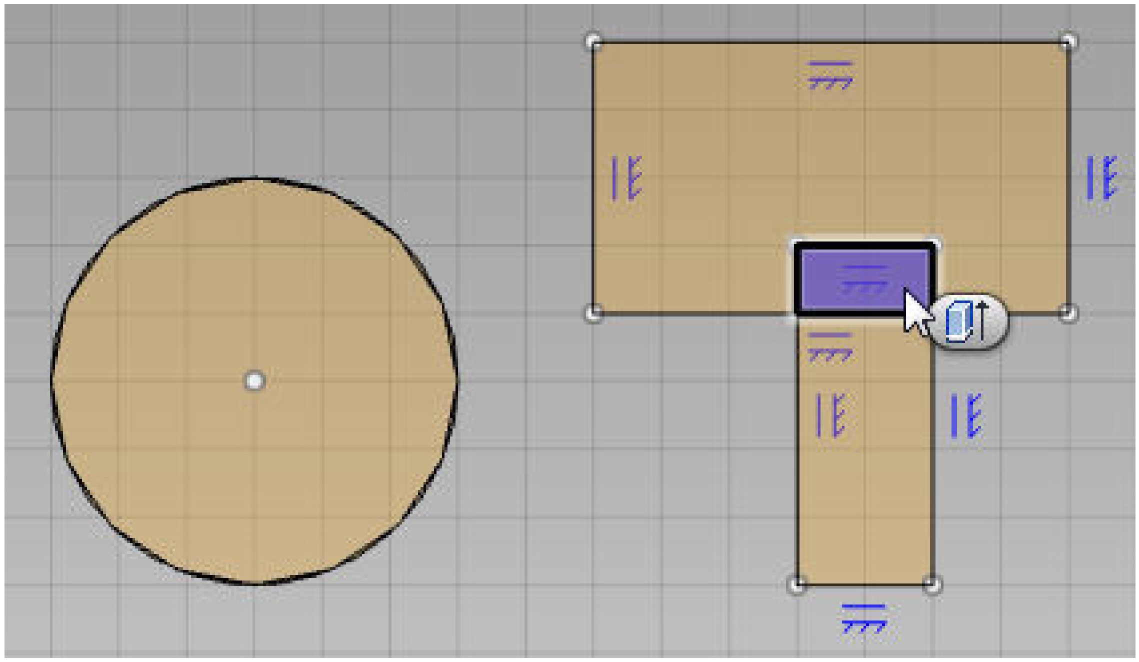



## Sketch Profiles

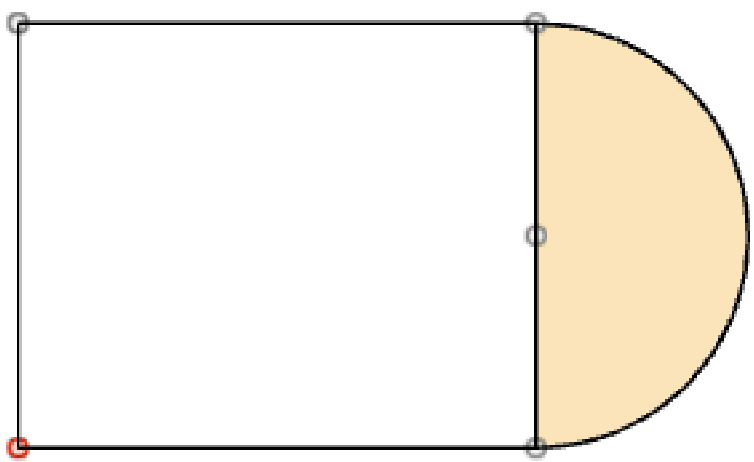
Profiles are closed loops that define cross sections for sketch-based features. A circle is always a profile, and a series of lines and curves form a profile when all of the end points connect. Some feature commands, like Extrude and Sweep, require a sketch profiles.

Sketch profiles are automatically detected and displayed as shaded regions. If two profiles overlap, the additional region is recognized as a separate profile.

In the following image, the sketch profiles are shown in yellow. The blue rectangular region under the cursor illustrates a selected profile.



If the end points of two sketch entities are close, but not connected, they display in red. This simplifies diagnosing the sketch to find out why a profile is not recognized as a region.



## Sketch Entity Commands

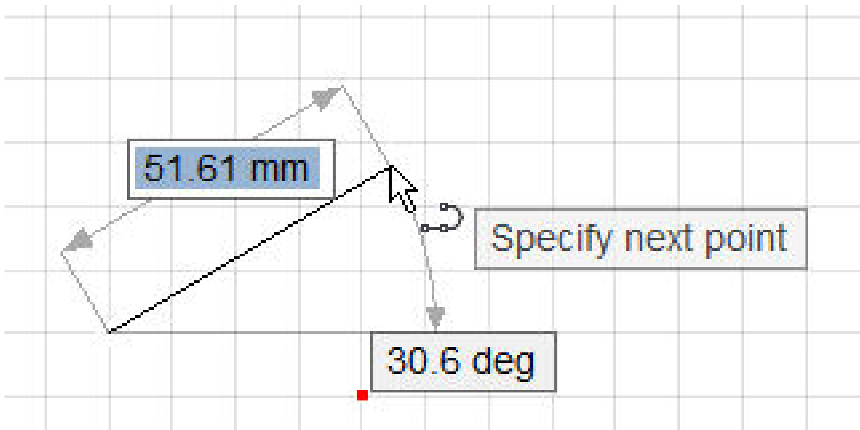
Sketch entity commands are used to create profiles for features, and to locate profiles.

A sketch entity is a single line or curve. Sketch geometry is multiple related sketch entities. Sketch geometry can be all of the entities in a sketch, or the entities in a particular profile. Sketch profiles are usually a closed region used to create features like extrusions. Sketch profiles that you use to create surfaces can be open.

### Draw Lines and Arcs

The Draw command creates a series of lines and arcs. A line segment is created each time you click. A tangent arc is created if you hold the left mouse button and drag.

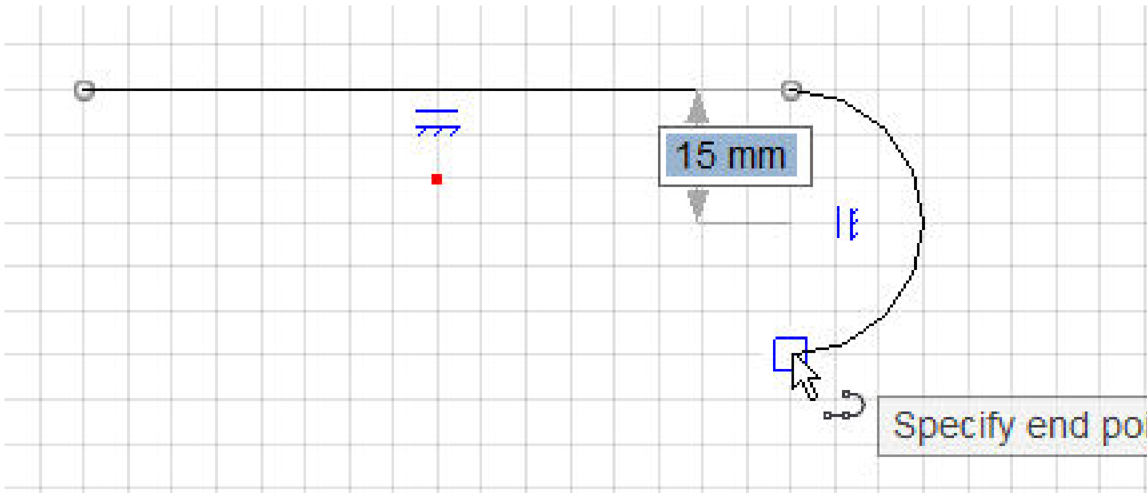
### Create lines




1. Click to place the starting point for the line, or enter the coordinates and click to create the point.
2. Click to place the second end point, or enter the length and angle and click to create the point. The angle field does not display when a constraint is inferred.
3. Continue to place points for additional line segments.

### Create tangent arcs

Create tangent arc segments while drawing lines, or add a tangent arc to an existing line.

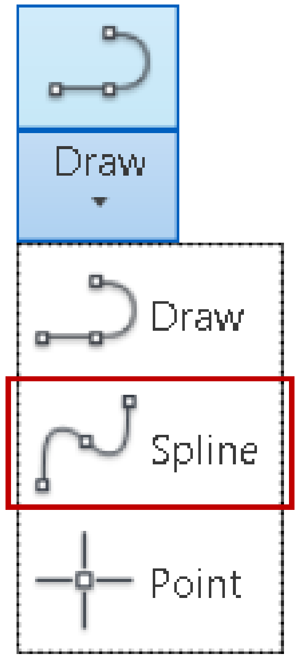



1 Pause over the last end point when drawing lines, or pause over an existing end point to add an arc.
2 Press the left mouse button and drag to preview the arc radius and included angle.
3 Move the cursor to other side of the line to change the direction of the arc.
4 Click to place the end point, or enter a value for the radius and then click to create the end point.

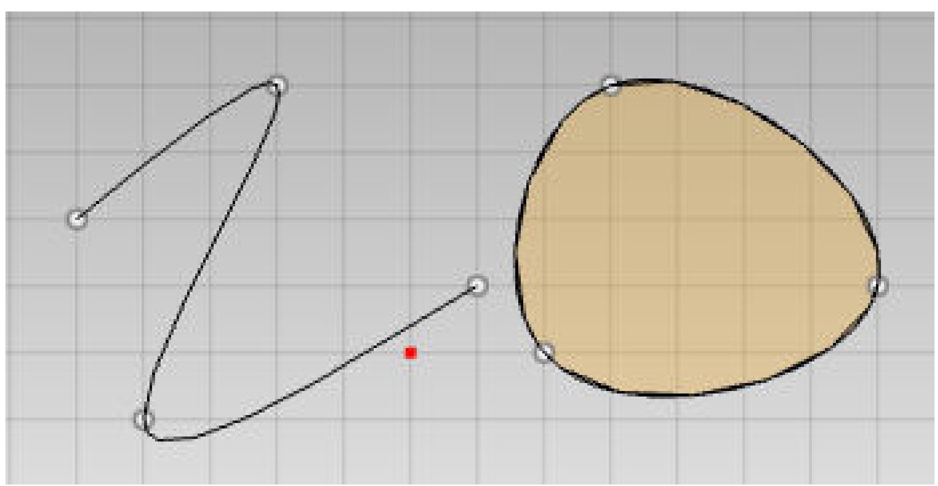
Reset the arc radius after entering a value by moving the cursor back to the starting point. If you press Tab after entering an arc radius value, the value is locked and will not reset.

## Spline Command

Splines are free form curves. You create a spline by placing points, and the spline solver creates a curve that goes through each point with smooth transitions.




You can create open or closed splines. Close a spline by clicking on the first point of the spline.



You can change the shape of the spline by moving the interior points. When you move a point, the spline solver changes only the curvature close to the point. It does not update the entire spline. You can make minor adjustments to the spline without affecting the overall shape.

Splines update locally instead of recalculating, so splines with identical points can have different curvature. If you create a spline and move interior points, the spline curvature changes only near those points. If you project those points into another sketch and create a spline, the new spline has different curvature. The first spline has local updates to the original curvature and the second spline calculates based on the final point locations.

---

**NOTE** It is possible to create self-intersecting splines (like a figure-8) using Spline. However, you cannot use self-intersecting splines to create features.

---

**Create Splines**

1 On the ribbon, Draw drop-down list, click Spline.
2 Click to place the starting point for the line, or enter the coordinates and click in the graphics window to create the point.
3 Continue to place points to define the spline.


To create a spline that is tangent to another line or spline, click an endpoint of the line or spline, and drag away from it. A tangent line indicator displays when the constraint is inferred.

4 Double-click to create an open spline, or click the start point to create a closed spline.

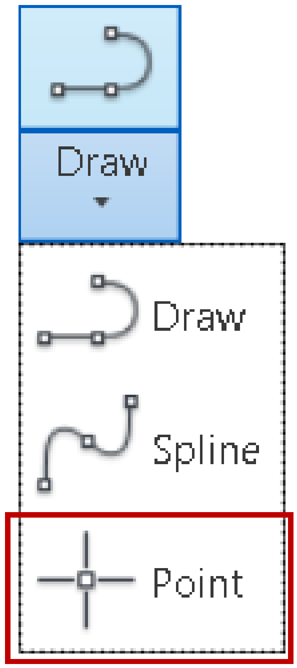
### Modify Splines

- Move interior points to modify the spline shape. Use persistent dimensions, constraints, and lock geometry to fix the location of points.

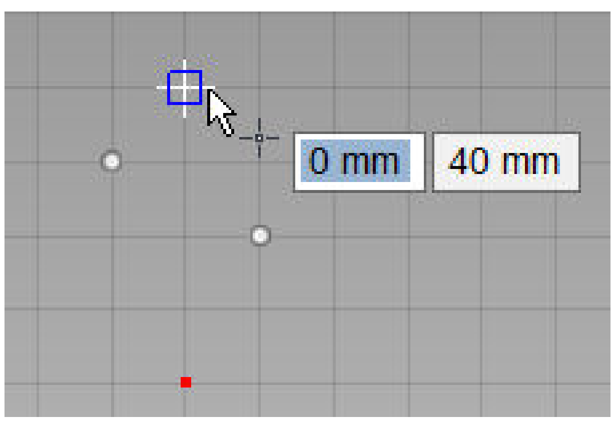
  The spline solver slightly adjusts the position of interior points to improve the spline curvature. The spline solver does not move the endpoints, or any interior points that are constrained. Using Lock Geometry on a point can change the curvature even though the point is in the same location.
- Use persistent dimensions, constraints, and lock geometry to fix the location of points.

Splines update locally instead of recalculating, so splines with identical points can have different curvature. Spline curvature changes only near the points you move. If you project those points into another sketch and create a spline, the new spline has different curvature. The first spline has local updates to the original curvature, and the second spline calculates based on the final point locations.

## Point Command

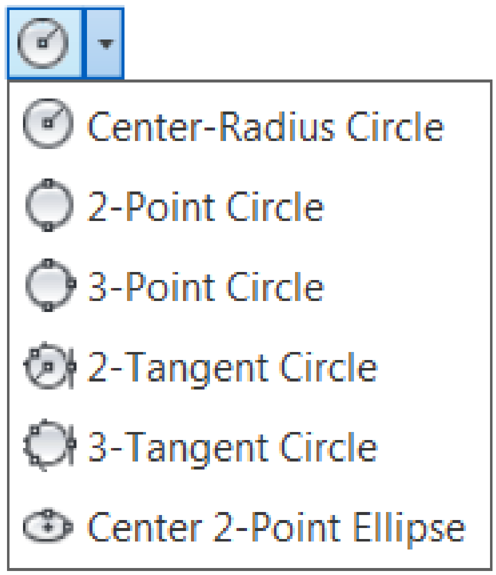



Place points by clicking in the graphics window or entering the coordinates in the value entry fields. You can quickly place a series of points at precise locations, and then create sketch entities by connecting the dots.



Coincident and midpoint constraints are automatically created when you place a point on an existing entity.

## Circles and Ellipses




Fusion has five circle commands. You can quickly create many circles without creating construction geometry. You can create ellipses by selecting the center point and the major and minor axes.

**Create a center-radius circle**

Place the center point, and then a point on the circumference.

**Create a 2-point circle**

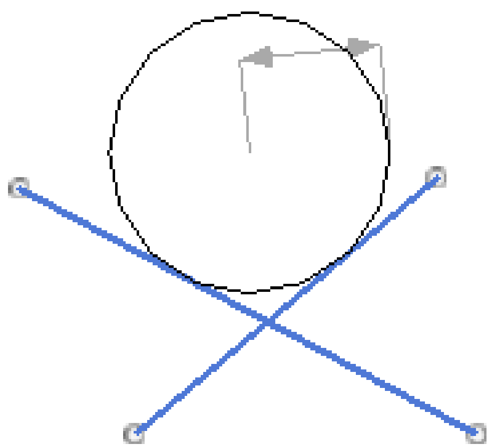
Place two points to define the diameter, or enter the coordinate values.

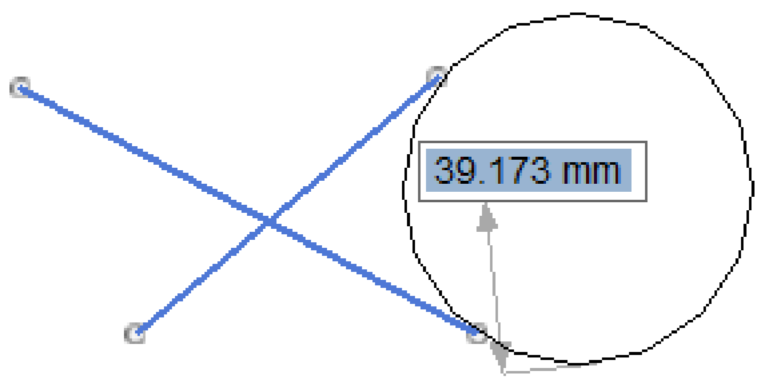
**Create a 3-point circle**

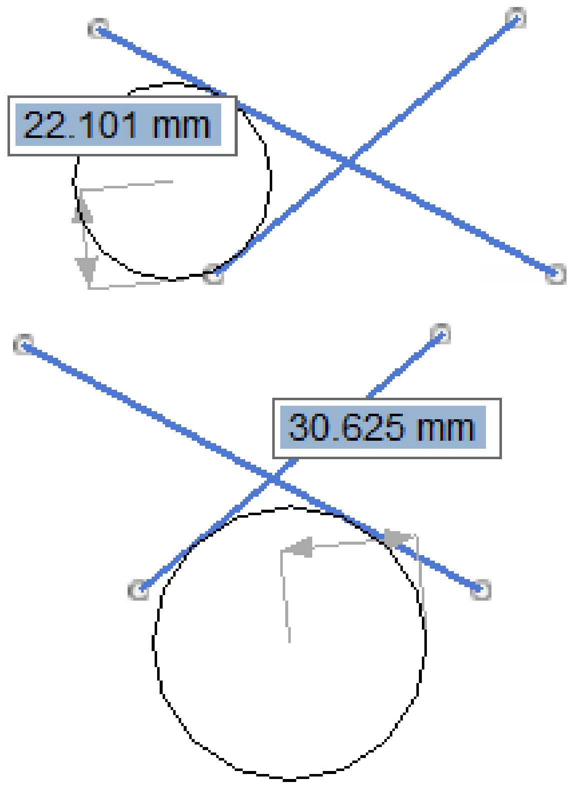
Place three points on the circumference, or enter the coordinates. The third point defines the diameter and the center of the circle.

**Create a 2-tangent circle**

Select two existing entities to define tangent locations on the circle, and then select the location of the center. There are four solutions for the center location:







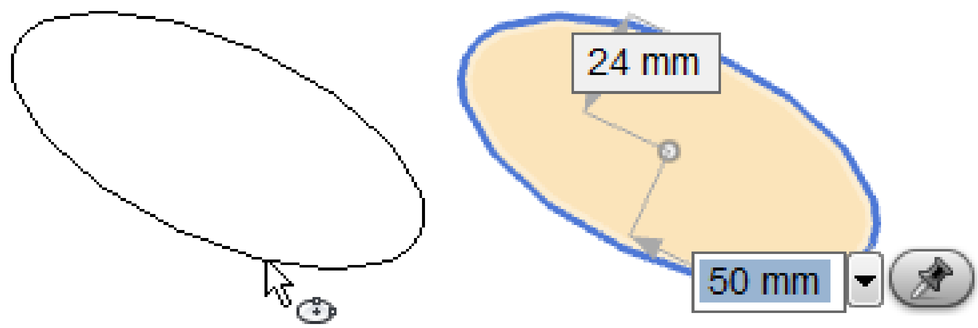

**Create a 3-tangent circle**

Select three existing sketch entities to define the tangent locations.

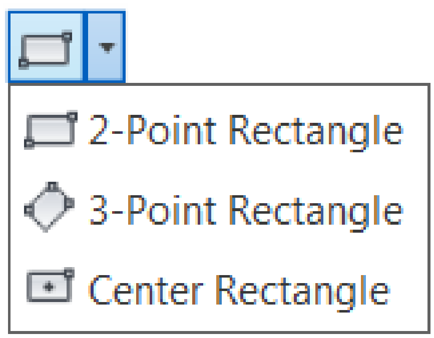
**Create a center 2-point ellipse**

Place the center point, and then the endpoint of one axis. The endpoint defines the orientation and the length of that axis. Place a point anywhere on the circumference of the ellipse.

Even though you can place the second point anywhere on the circumference, the ellipse has dimensions on the major and minor axes.




## Rectangle Command




Use the Rectangle command on the Sketch tab to create rectangles two ways: specifying diagonal corners, or specifying length and width. Each rectangle side is a line segment.

**Create a 2-point rectangle**

Place a point for the location of a corner, then place a point to define the length and height. The first side has a horizontal constraint.

**Create a 3-point rectangle**

Place a point for the location for a corner, then a point to define the length of one side. Place the third point to define the height of the rectangle. The sides have parallel constraints.

---

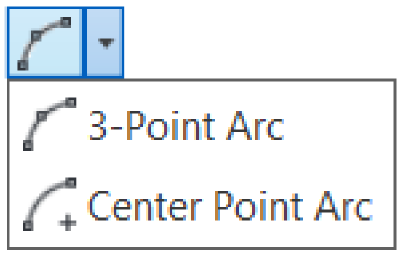
**NOTE** You cannot infer a parallel or perpendicular constraint to existing sketch entities, but you can infer a horizontal or vertical constraint for the first side.

---

**Create a Center Point rectangle**

Place the center point of the rectangle, and then a point to define length and height. The rectangle is aligned with the origin, but the sides only have parallel constraints.

## Arc Command




Use Arc on the Sketch tab to create arcs two ways: three points on the arc, a center point, and two endpoints.

**Create a 3-Point arc**

Place two points to define the endpoints of the arc. Place the third point to define the side for the arc and the radius.

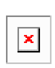
**Create a Center Point arc**

Place a point to define the center, a point to define the one endpoint and a radius, and a third point to define the included angle of the arc.

## Polygon Command

The Polygon command creates polygon shapes. Polygons can have up to 128 sides.

Fusion has three types of polygons: inscribed, circumscribed, and edge. You create inscribed and circumscribed polygons based on a theoretical circle. You create edge polygons by specifying the endpoints of an edge and the location of the center.

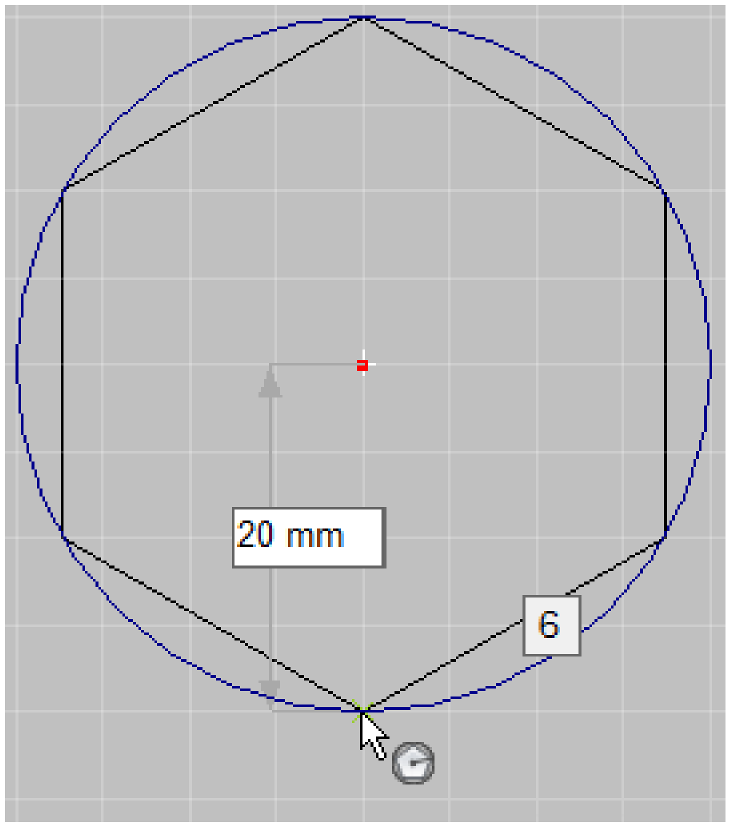


The vertex defines an Inscribed polygon.

The midpoint of the edge defines a Circumscribed Polygon.

**Create an Inscribed polygon**

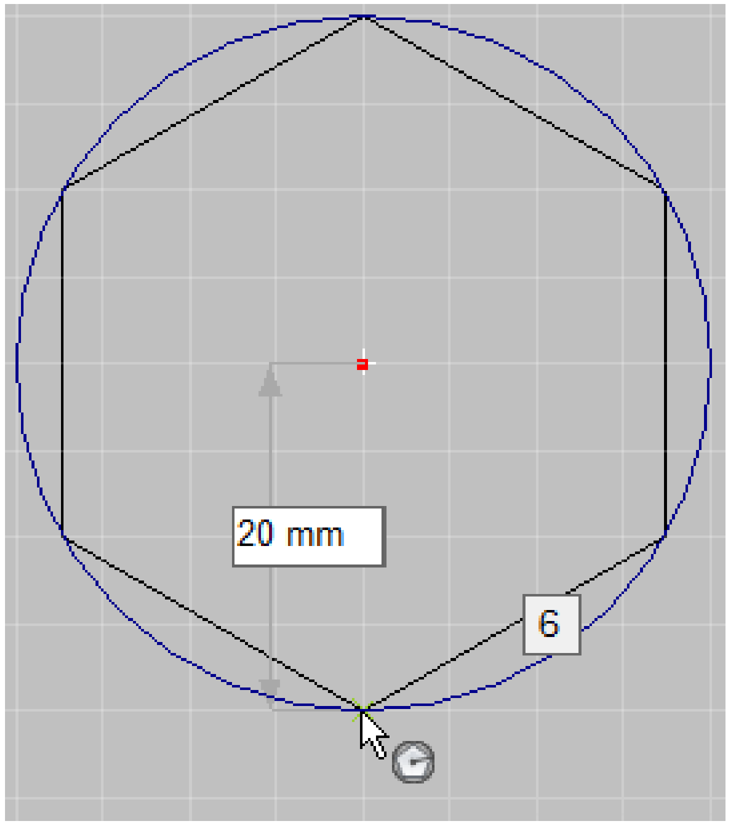
1




**Create a Circumscribed polygon**

1 Place a point to define the center. A temporary circle displays to preview the polygon.

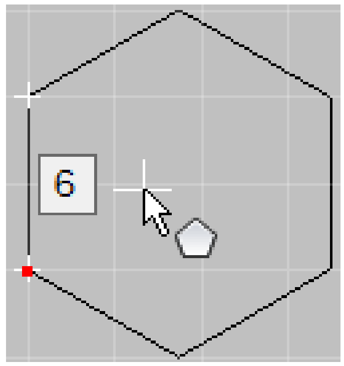
2 Place a second point to define the location of a vertex.




**NOTE** Set the number of sides before placing the points to improve the preview.

### Create an Edge polygon

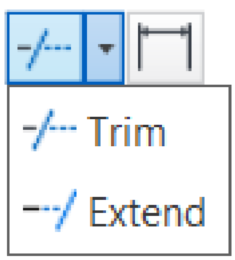
1 Place a point to define the location of an endpoint of an edge.
2 Place a second point to define the length and orientation of the edge.
3 Move the cursor to either side of the line and click to set the location of the center.




## Sketch Modification Commands

The sketch modification commands change existing sketch entities.

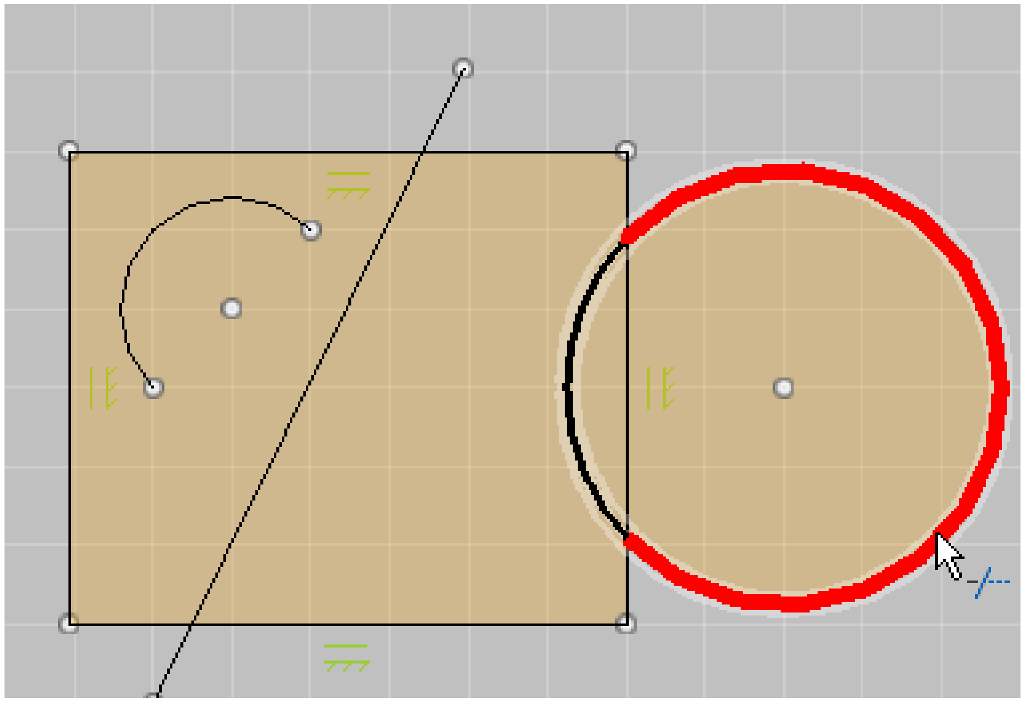
### Trim - Extend Command



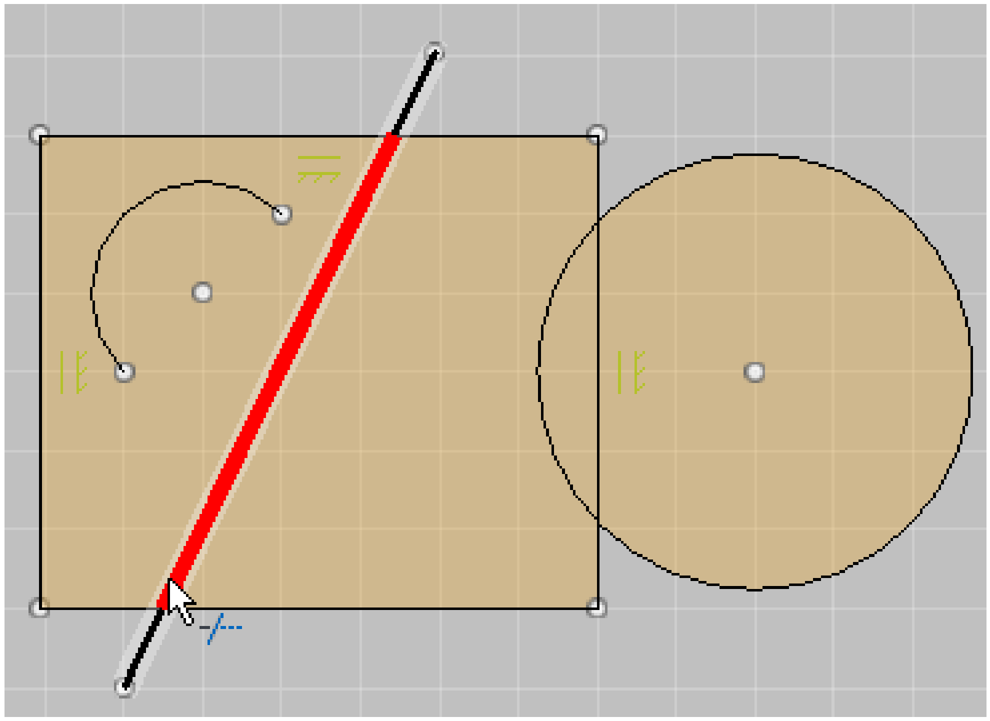

Use Trim and Extend to trim and extend sketch entities to other entities, remove sections of entities, or delete entities.

#### Trim and Entity

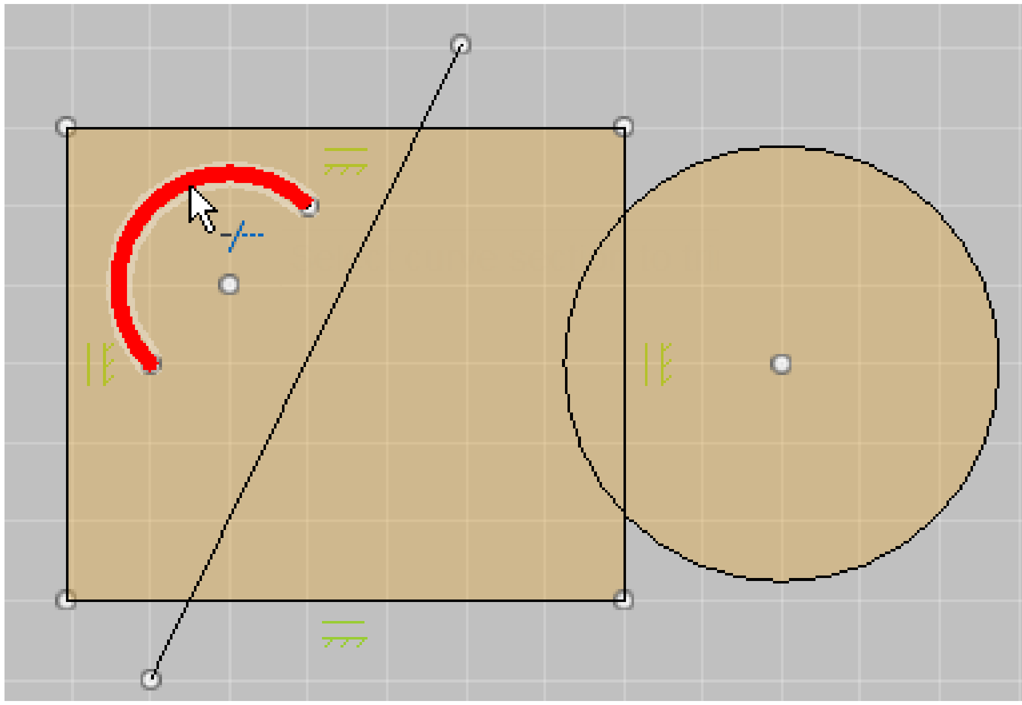
Pause the cursor over any sketch entity. The preview shows the section of the entity to delete. Click to delete that section. If the entity does not intersect another entity, the entire entity highlights.



The section is deleted if it intersects another entity.



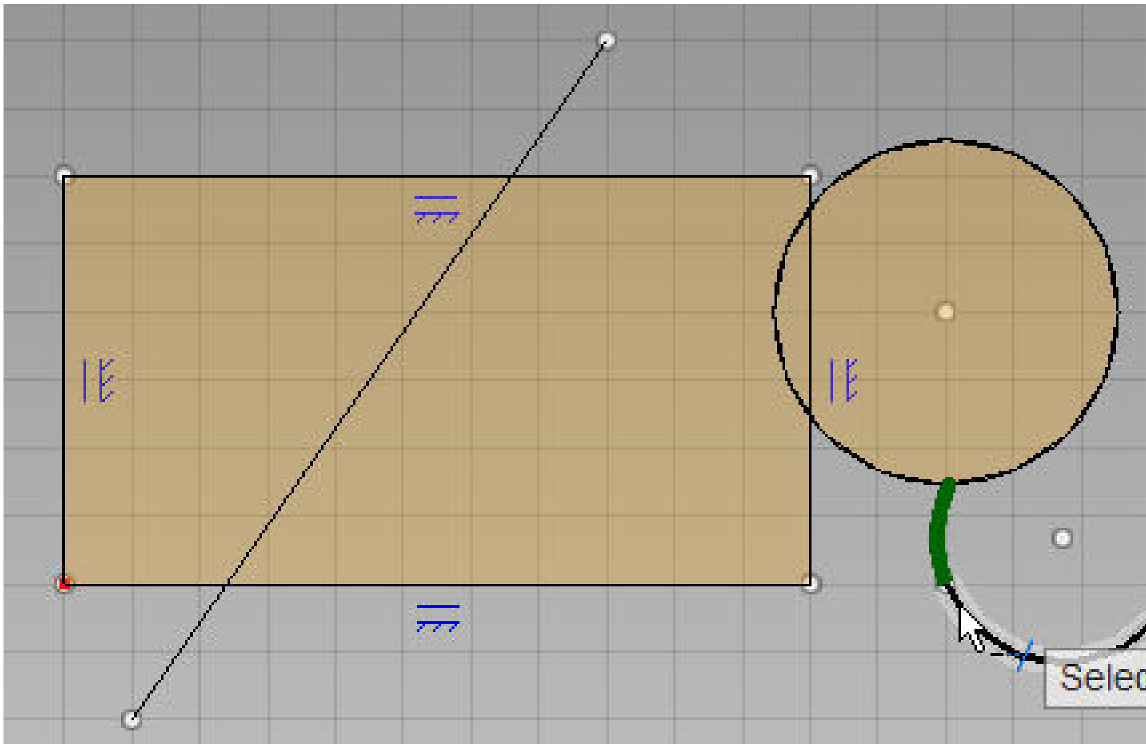
The section is deleted if it intersects two other entities, leaving two sketch entities.



The entity is deleted if it does not intersect any entities.

#### Extend an entity

Pause the cursor over any sketch entity. The added section highlights if the entity intersects another entity. If the entity can be extended at both ends (for example, an arc), the end that is closest to the cursor extends. If an entity cannot be extended against another entity, there is no preview, and a click has no effect.




## Sketch Fillet

The Fillet command places an arc of a specified radius at a corner or intersection of two lines.

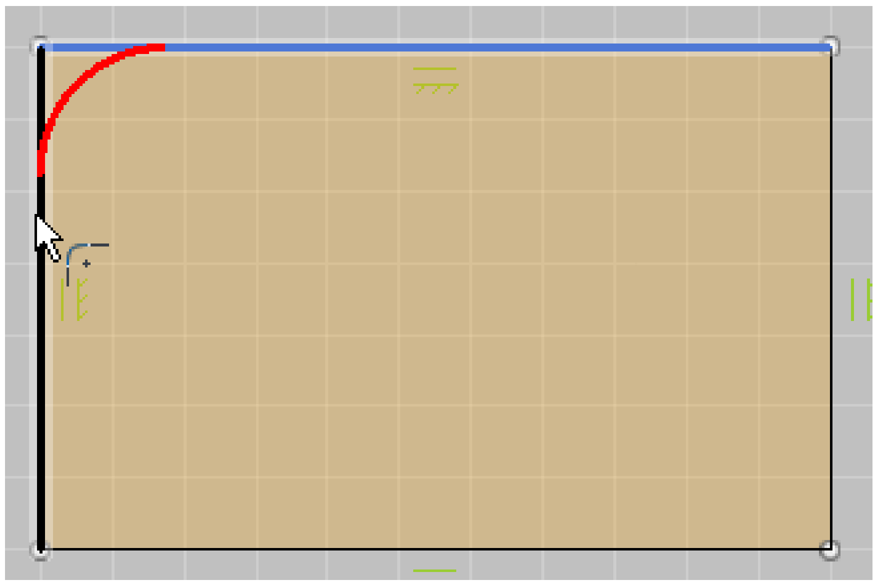
You can create fillets between:

- Two intersecting lines
- Two parallel lines
- A line and a circular arc that intersect
- Two circular arcs that intersect

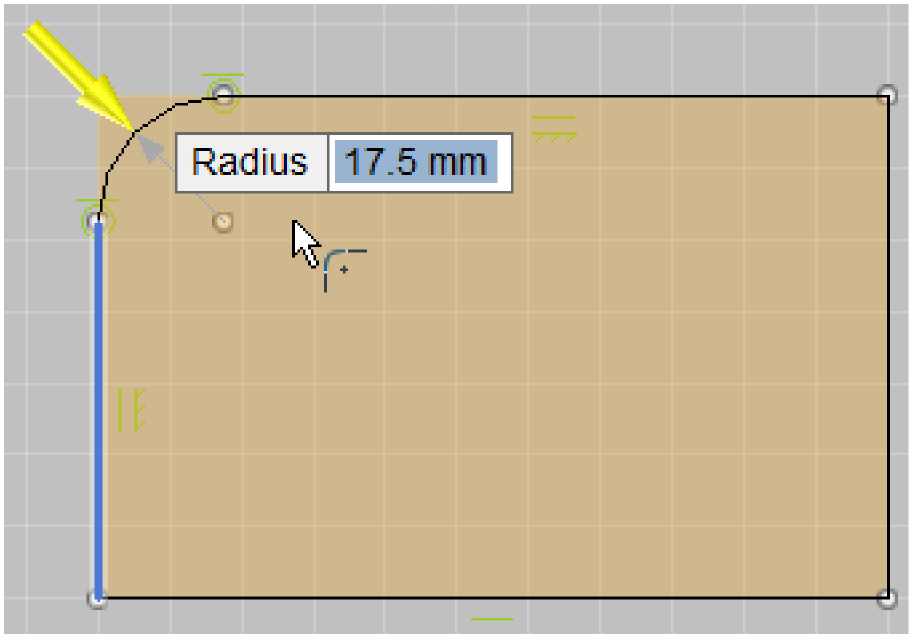
You cannot apply fillets to circles, ellipses, elliptical arcs, splines, and non-intersecting entities, except for parallel lines.

**Create a fillet**

1 Pause the cursor over the first entity for a fillet. The entity highlights if it is a valid a selection. Click to select the entity, then pause the mouse over the second entity. The fillet preview highlights for a valid selection.



2 Click to select the second entity.




The fillet previews with a value that is proportional to the smaller entity.

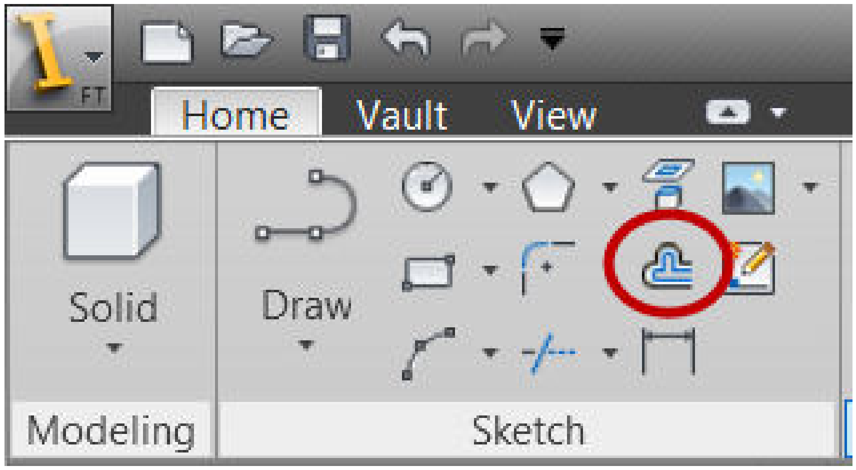
3 Click to select the second entity. The fillet previews with a value that is proportional to the smaller entity.

4 Enter a value in the box, or drag the arrow to change the size.

5 Click and drag the yellow arrow to define the fillet radius, or enter a value in the Radius edit box.

## Sketch Offset Command

The Offset command duplicates sketch geometry at a distance from the original.



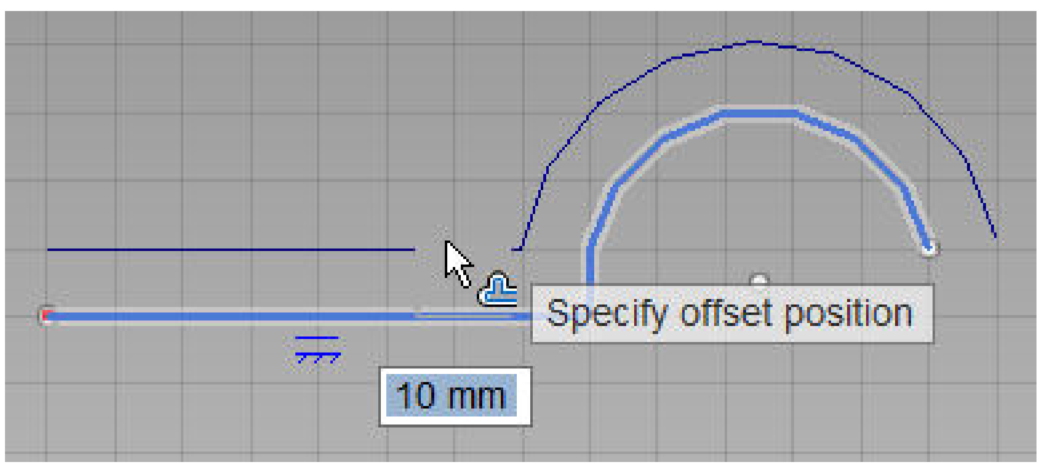

You can offset any sketch entities on the sketch plane. Offset supports individual lines and curves, a chain of connected entities, and profiles. You can only make one selection.

**Create an Offset**

1 In the graphics area, select an individual entity, a chain of connected entities, or a profile.

A valid selection highlights when you pause the cursor over it

2 Move the cursor, and click to create the offset, or enter a value.

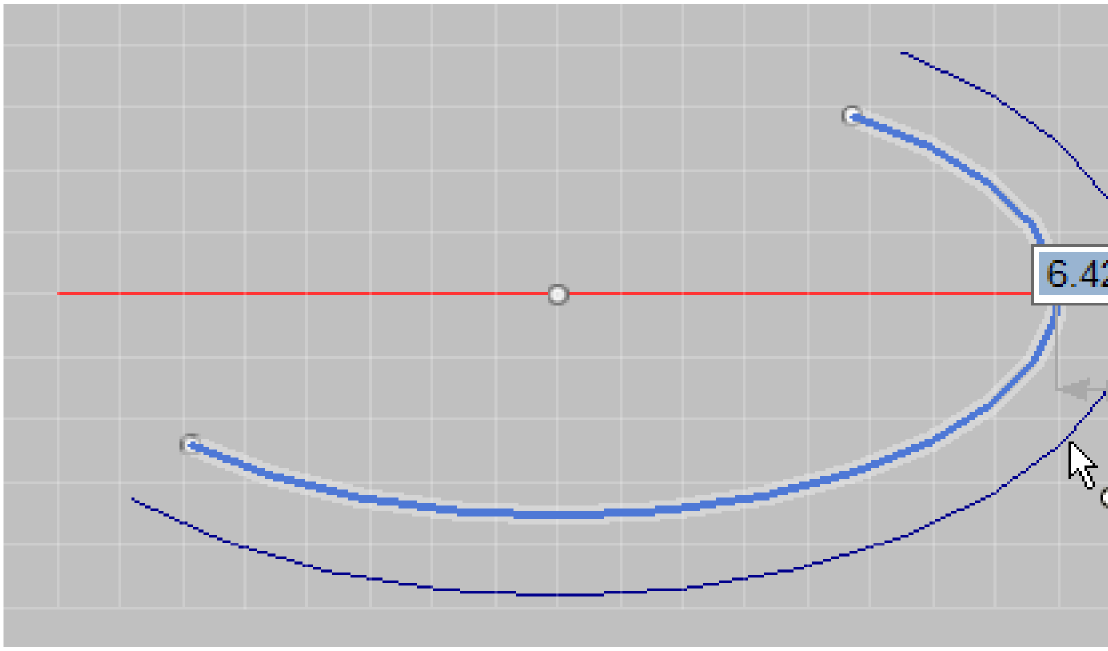


**NOTE** Offset entities do not keep a reference to the original geometry. You can change the length of any entity independently.

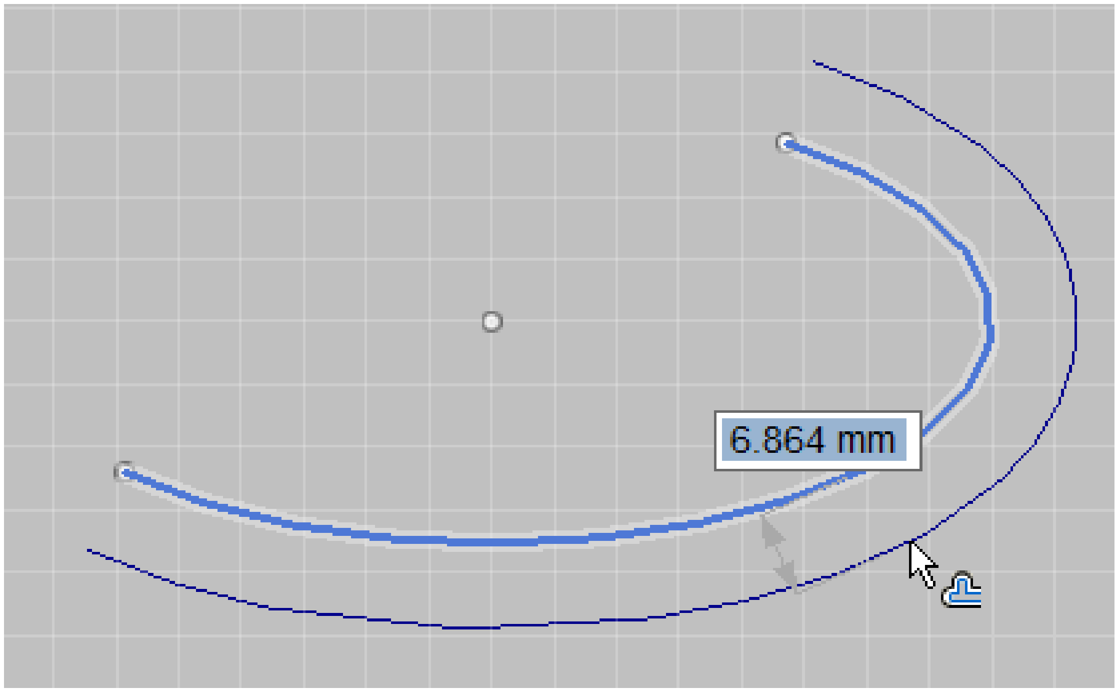
**Offset an ellipse**

- Select and ellipse or elliptical arc.

  If you click the ellipse near the major or minor axis, the offset result is an ellipse. The axis displays in the preview, and the offset distance is the same at the major and minor axes. Since an ellipse is created, the offset distance varies around the rest of the ellipse.

- If you click the ellipse away from the major or minor axis, the offset result is an oval. The axis does not display in the preview, and the offset distance is the same around the entire ellipse.

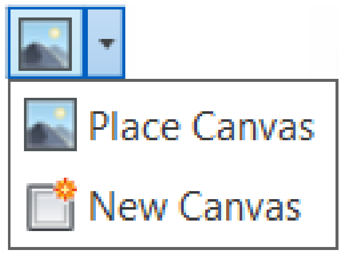



**NOTE** An oval offset is locked. You cannot change the size or create an offset from it.

**NOTE** The center of an offset ellipse is constrained to the center of the original ellipse. You can select the combined center point and press delete to create individual center points.

## Sketch Canvas

A sketch canvas places an image in the model. You can use it as a background, or trace over it to create features.




Fusion has two methods to add a canvas to a model.

- Place a sketch canvas on a work plane or planar face to use an existing image.
- Create a canvas on a work plane or planar face. Sketchbook Designer starts so you can create the image.

**Place a sketch canvas**

1. Click Place Canvas.
2. Browse to an image.
3. Select a planar face or work plane.
4. Use the preview controls to adjust the image size and rotation, and to mirror it horizontally and vertically.
5. Click OK to display the image.

**NOTE** Some image formats support a transparent background. Fusion keeps the transparency of images.

**Create a sketch canvas**

1 In the Place Canvas drop-down list, click New Canvas.
2 Select a planar face or work plane.
3 Sketchbook Designer starts with a new image.
4 Create the image and click Return to Autodesk Inventor Fusion to place the image on the canvas.

### Work with a Canvas

There are several commands for modifying the canvas on the context menu in the browser.

### Edit Canvas

The image manipulator controls the size and orientation.

### Calibrate

Displays a temporary dimension between two locations on the image. Change the dimension value to resize the image. Calibrate maintains the aspect ratio when resizing the image.

1. Right-click the canvas in the browser, and select Calibrate.
2. Click the image to set the anchor location. The anchor location remains fixed as the image changes size.
3. Click the image to specify the second location. A dimension displays with the distance between the points.
4. Double-click the dimension to change the value.
5. Press **Enter** to resize the image.

### Transparence

Changes the transparency of the image. Model geometry is visible behind a transparent image.

Right-click an image in the browser, and select Transparence. The image becomes transparent. Select Transparence again to make the image opaque.

### Show/Hide Background

Provides a two-step process to make a white background transparent and add anti-aliasing to the image.

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**NOTE** Some image formats support a transparent background. Fusion keeps the transparency of images.

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1. Right-click the image in the browser, and select Show/Hide Background. All white areas of the image become transparent.
2. Right-click the image in the browser, and select Show Hide Background again. The edges of the image become anti-aliased.

### Change colors in the image

1. In the browser, right-click the image, and select Edit in Sketchbook Designer.

   Fusion converts pure white colors (RGB = 255, 255, 255) to transparent. Make the background pure white if it does not become transparent. Use a slightly off-white color (RGB= 254, 254, 254) to prevent interior areas from becoming transparent.

#### Remove the transparency

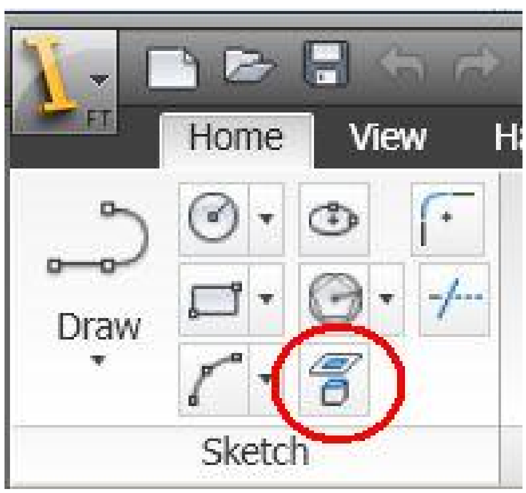
- Use**Update Image**.

#### Edit in Sketchbook Designer

Opens the image in Sketchbook Designer. The Fusion image is updated when you click Return to Inventor Fusion.

## Project Geometry

Use project geometry to create copies of existing model geometry and sketch entities in the active sketch.



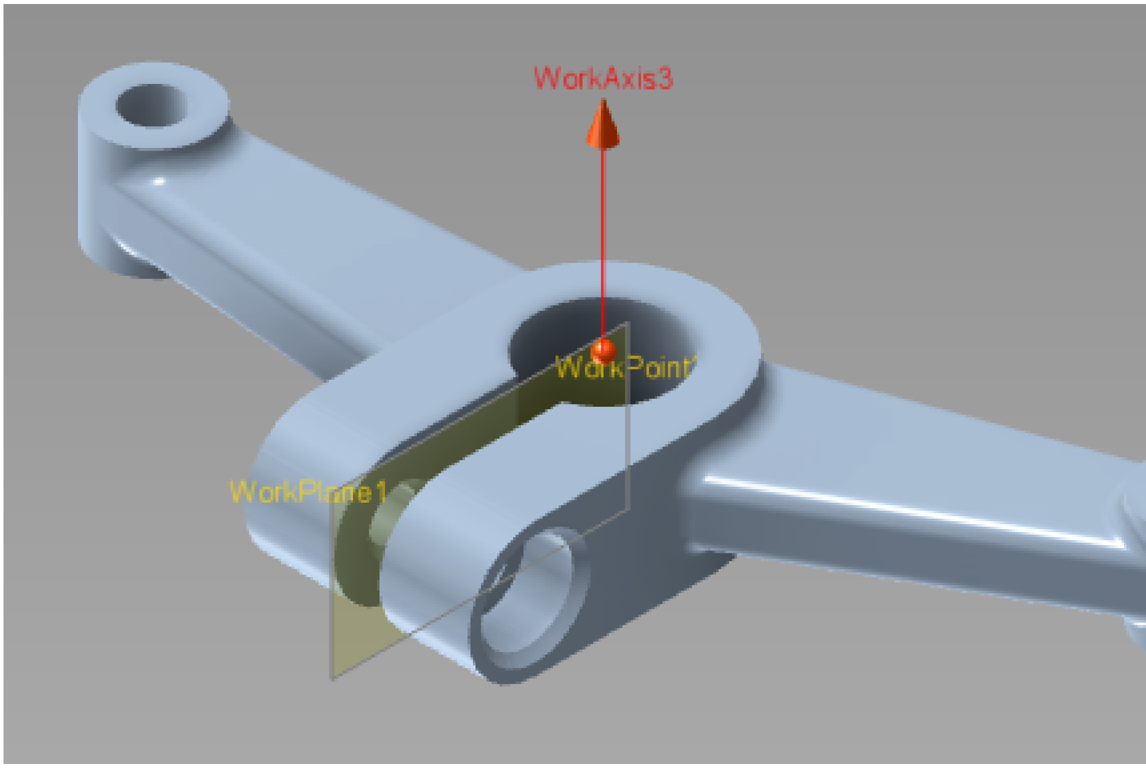

You can project model edges, work geometry, and sketch entities into the active sketch. Projected geometry is locked by default. For more information, see Locking Sketch Geometry (page 124).

If you create a sketch on an existing planar face, the edges of that face automatically project into the sketch.

**NOTE** Projected geometry is not linked to original geometry or entities. You can independently change the original or projected geometry

## Work Features and Construction Geometry

Work Features are planes, axes, and points created from model geometry, sketch entities, and existing work features. For example, you can create a work plane that is offset from a model face.




Work geometry is also called construction geometry, and it is used to aid the modeling process. For example, work planes are frequently used for creating sketches. Work features can be used for positioning model features during creation, or projecting a reference into a sketch.

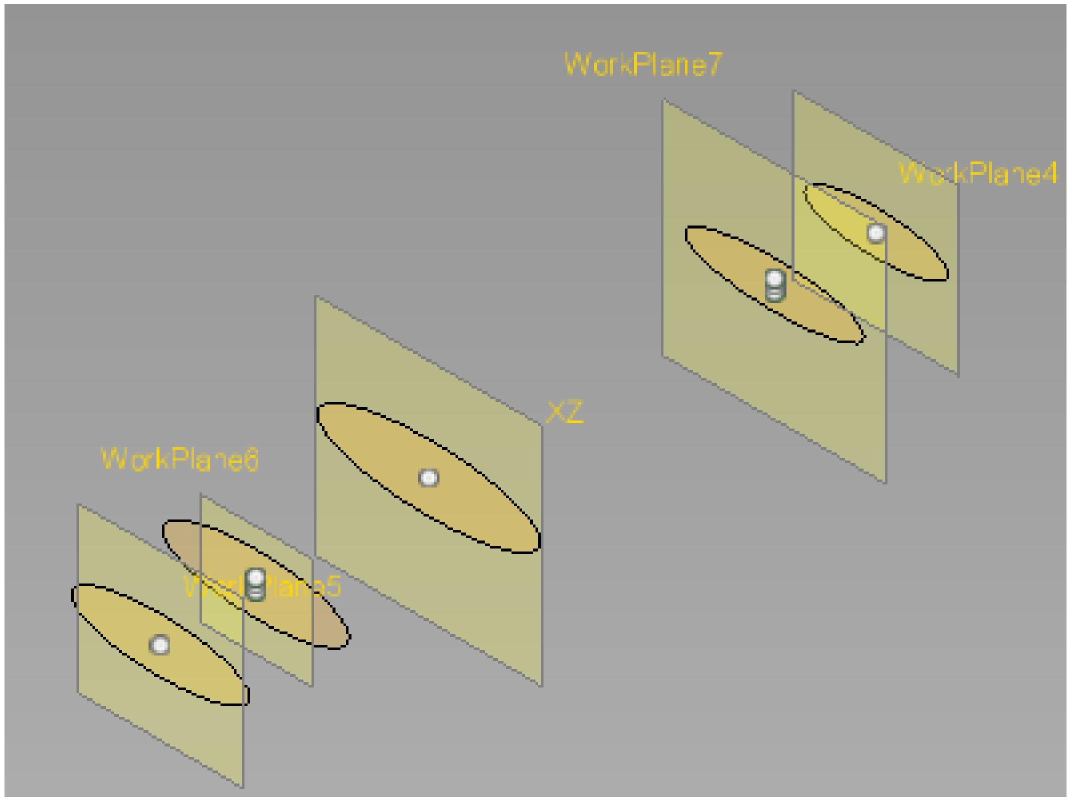
NOTE Since Inventor Fusion uses direct modeling, there is not a connection between the work feature and the geometry used to create it. For example, if you place a work axis on a hole, you can move the hole or the work axis independently.

### Create Work Geometry

Use work geometry to aid in modeling. Work planes, axes, and points are created using model geometry, sketches, and existing work features.

The work feature commands are located on the Construction panel. There are several options for each type of work feature.


In the image below, four work planes were created using the By Offset command on the XZ plane.



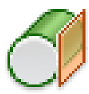
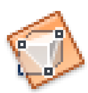
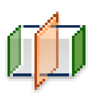
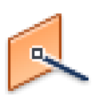
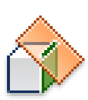

Each work feature command uses a specific set of selections. Some commands also require a value input. All work feature commands share a basic creation workflow. This example is for an offset work plane.

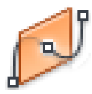
1 Start the By Offset work plane command from the Construction panel.
2 Select a model face, work plane, or sketch entity. When you select a sketch entity, the new work plane uses the sketch plane as the reference.
3 Drag the arrow manipulator or enter a value to offset the work plane.
4 Optionally, pause the cursor over a valid face and click when the To Face prompt displays.
5 Right-click and select OK from the marking menu.
6 Right-click and select Repeat from the marking menu to create another work plane.

### Work Feature Types

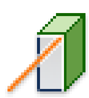
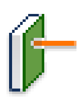
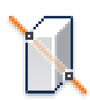
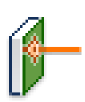
Inventor Fusion creates the following types of work features. Most work features are automatically created based on the geometry selections. Some work features, such as an offset work plane, also require a value to define the location.

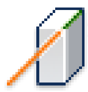
**Work Planes**

| Icon | Work Plane Type | Inputs |
|---|---|---|
|  | By Offset | Select a work plane or planar model face and enter the distance. |
|  | At Tangent | Select a cylindrical face and enter the angle. The plane is created where you clicked on the cylinder. |
|  | By Three Points | Select three model, sketch, or work points. |
|  | Between Two Planes | Select two parallel planes or planar model faces. The work plane is created halfway between them. |
|  | By Point and Axis | Select a model or work point, and a model edge, sketch line, or work axis. The plane is created perpendicular to the work axis through the point. |
|  | At Angle | Select a model edge, sketch line, or work axis, and enter the angle. |

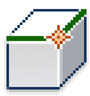
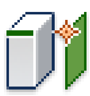
| Icon | Work Plane Type | Inputs |
|---|---|---|
|  | At Point on Path | Select a model edge, sketch line, or work axis. The work plane is created perpendicular to the path where you clicked. |
|  | Plane By Two Axes | Select two model edges, sketch lines, or work axes. |

## Work Axes

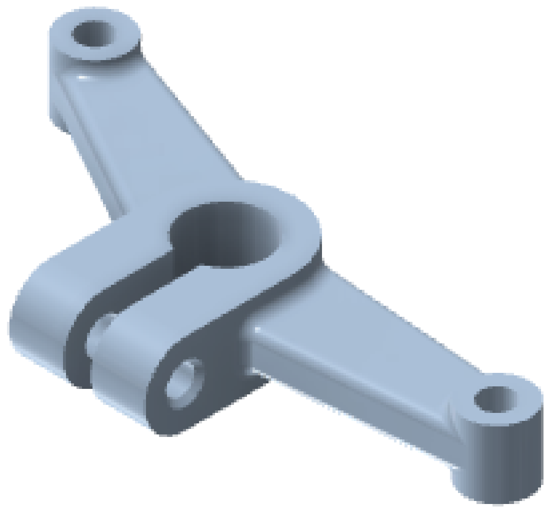
| Icon | Work Axis Type | Inputs |
|---|---|---|
|  | By Cylinder or Cone | Select a cylindrical face or a cone. |
|  | By Two Planes | Select two model faces or work planes. |
|  | At Pick Point | Select a model face or sketch profile. Work axis is created normal to it. |
|  | By Two Points | Select two vertices, sketch points or work points. |
|  | Normal at Plane and Point | Select a model or work plane, and a vertex or a work point. |

| Icon | Work Axis Type | Inputs |
|---|---|---|
|  | By Edge | Select a linear or circular edge on a model or a sketch profile. |

### Work Points

| Icon | Work Axis Type | Inputs |
|---|---|---|
|  | By Two Edges | Select two edges on a model or two models or sketch profiles that intersect. |
|  | By Three Planes | Select three model faces or work planes. |
|  | By Face/Plane/Profile & Axis | Select a work plane or model face, or a sketch profile and a work axis or sketch line. |
|  | By Circle/Spherical Surface Center | Select a circular edge or spherical surface. |

# Solid Modeling



Solid models in Fusion do not have a history. A model is a collection of features and geometry. With direct modeling, you can change geometry without editing the feature.

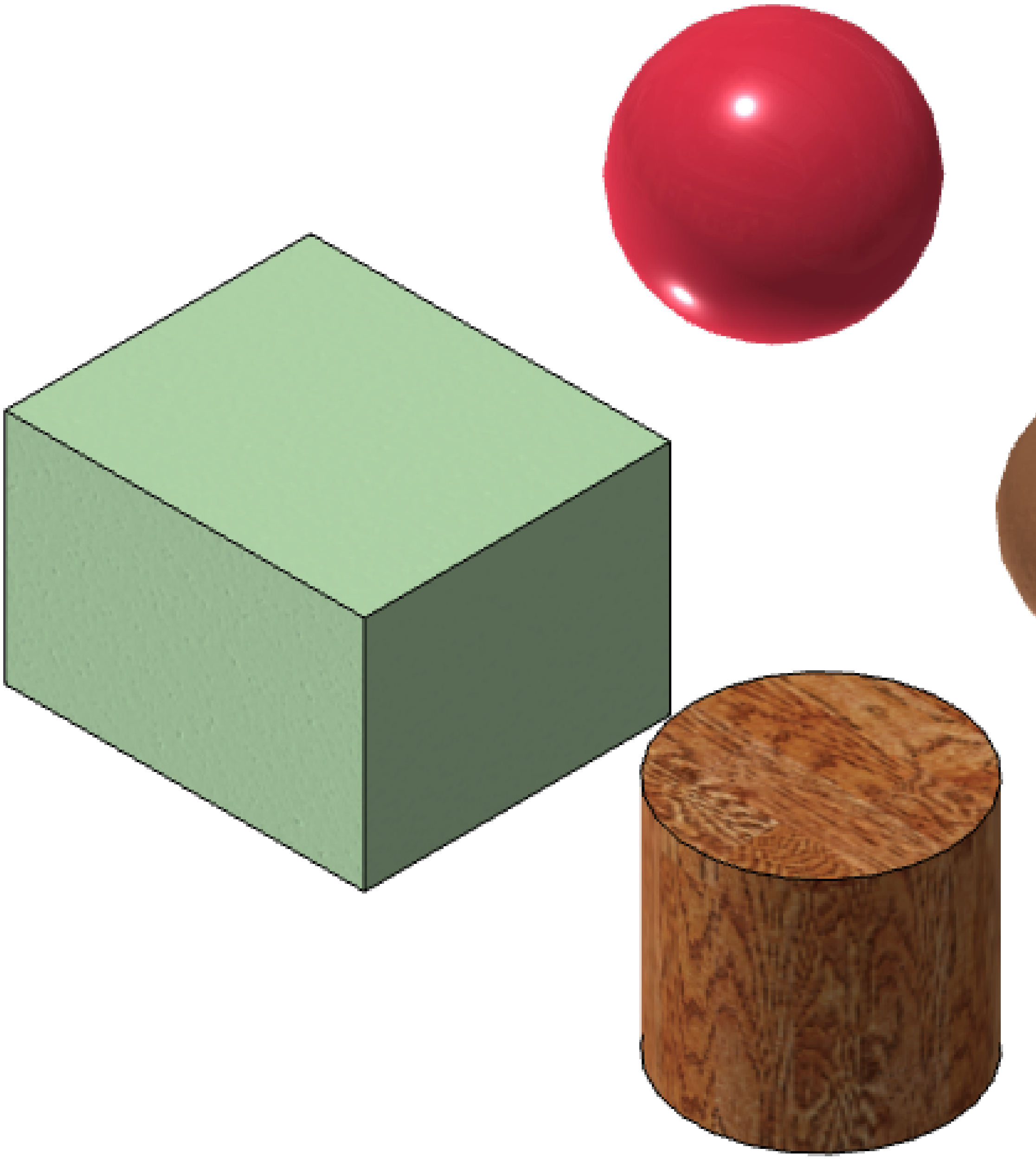
You can create standard modeling features like extrudes, fillets, and ribs. Standard Boolean options (Join, Cut, Intersect, New Component) are available. Most features become independent geometry after they are created. You can edit only a few features, like holes and patterns.

Each component has one solid body, but it is not necessary for the features to be connected. You can create multiple components in a model, including child components, to build an assembly.

## Primitive Features

The primitive feature commands combine sketching and feature creation to simplify creating basic solid geometry. Use the primitive commands to create boxes, cylinders, spheres, and torii (donuts). Like other modeling features, you can use primitives to join, cut, intersect, or create new components.

A feature node is added to the browser. The type of feature depends on the type of primitive created.



### Box

Create a rectangular or cubical box by specifying three linear dimensions. Options:

- Distance
- Taper angle
- Limit
- Boolean options

#### Show me how to create a cube

This video demonstrates using the box command to create a cube.

### Cylinder

Create a cylinder by specifying the radius and height. Options:

- Distance
- Taper angle
- Limit
- Boolean options

#### Show me how to create a cylinder

This video demonstrates using the cylinder command

### Sphere

Create a sphere by specifying the radius. Options:

- Boolean options

#### Show me how to create a sphere

This video demonstrates using the sphere command

### Torus

Create a torus by specifying the radii of two circles. The first radius is the distance from the center of the torus to the center of the tube. The second radius specified is the radius of the tube. Options:

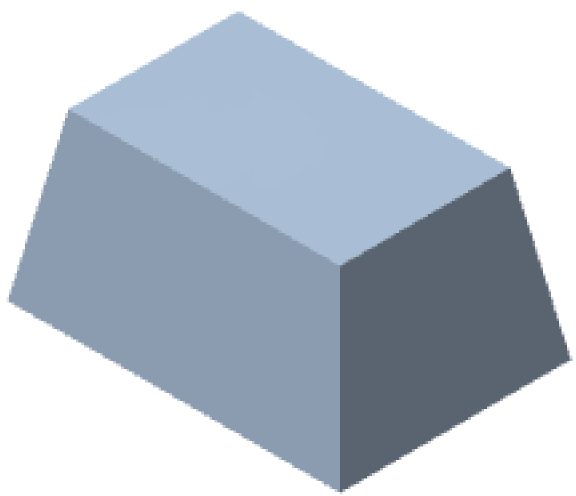
- Position
- Boolean operations

#### Show me how to create a torus

This video demonstrates using the torus command

## Box

Box creates an Extrude feature by adding depth to a rectangular region. You can add taper to the extrusion.



### Create a box

Create a rectangular Extrude feature on a work plane or face.

1. Click Box on the solids panel.

2 Select a face or work plane. The first corner is located where you clicked.

3 Click to place the opposite corner or enter values in the fields. The Extrude command ribbon displays.

4 Use the manipulators to set the height and taper, or enter the values in the command ribbon.

   **NOTE** If you selected a face for the extrusion, the feature automatically adds or removes material based on the direction. See step 6 for instructions on manually setting the Join or Cut Boolean option.

5 Select the Limit type

   a Distance is a numeric value for the extrusion length.

   b To determines the extrusion length when select a face or work plane.

   c All determines the extrusion length by the furthest face in the model.

6 Select the Direction

   a One Direction creates the extrusion in one direction.

   b Two Directions creates the extrusion in both directions. Each direction can have a different extrusion length.

   c Symmetric creates the extrusion in both directions. Each direction has the same extrusion length.

7 Select the Boolean Option

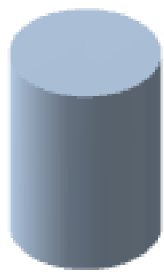
   a Join adds material.

   b Cut removes material.

   c Intersect removes all material from the solid that does not overlap the new feature.

   d New Component creates a child component in the active component.

## Cylinder



Cylinder creates an Extrude feature by adding depth to a circular region. You can add taper to the extrusion.

Cylinder is on the drop-down menu under Box .

**Create a cylindrical Extrude feature on a work plane or face**

1. Click on the drop-down arrow next to Box and select Cylinder.
2. Select a face or work plane. The center is located where you clicked.
3. Click to define the radius or enter the value in the field. The Extrude command ribbon displays.
4. Use the manipulators to set the height and taper, or enter the values in the command ribbon.

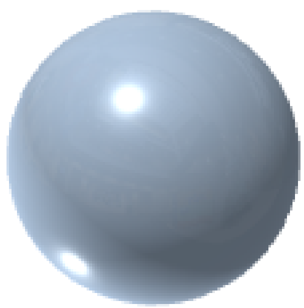
   **NOTE** If you selected a face for the extrusion, the feature automatically adds or removes material based on the direction. See step 6 for instructions on manually setting the Join or Cut Boolean option.

5. Select the Limit type.
   - Distance is a numeric value for the extrusion length.
   - To determines the extrusion length when select a face or work plane.
   - All determines the extrusion length by the furthest face in the model.
6. Select the Direction.
   - One Direction creates the extrusion in one direction.
   - Two Directions creates the extrusion in both directions. Each direction can have a different extrusion length.

- Symmetric creates the extrusion in both directions. Each direction has the same extrusion length.

7 Select the Boolean Option.
   - Join adds material.
   - Cut removes material.
   - Intersect removes all material from the solid that does not overlap the new feature.
   - New Component creates a child component in the active component.

## Sphere

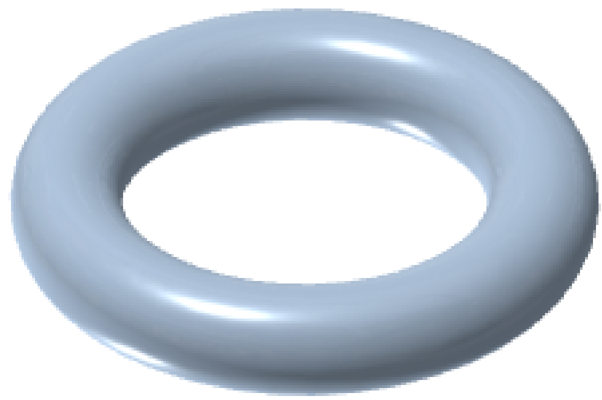


Sphere creates a spherical Revolve feature.

**Create a sphere**

1. On the Box drop-down list, click Sphere.
2. Select a face or work plane. The center locates at the point you click.
3. Click to define the radius, or enter the value in the field.
4. Select the Boolean Option.
   - **Join** Adds material.
   - **Cut** Removes material.
   - **Intersect** Removes all material from the solid that does not overlap the new feature.
   - **New Component** Creates a child component in the active component.

## Torus



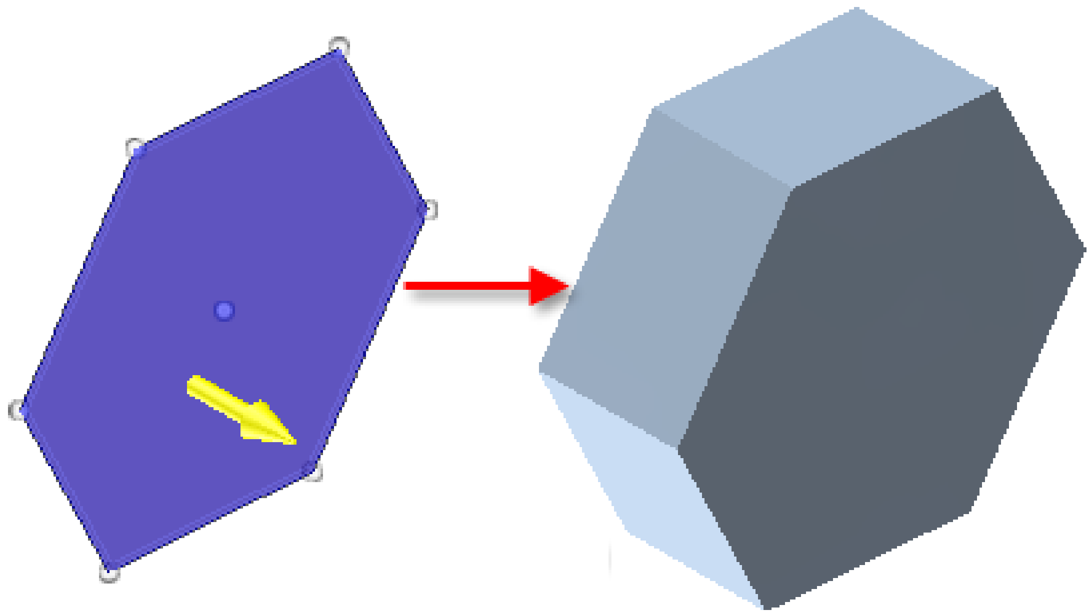
Torus creates a donut shaped Revolve feature by rotating a circle around an axis.

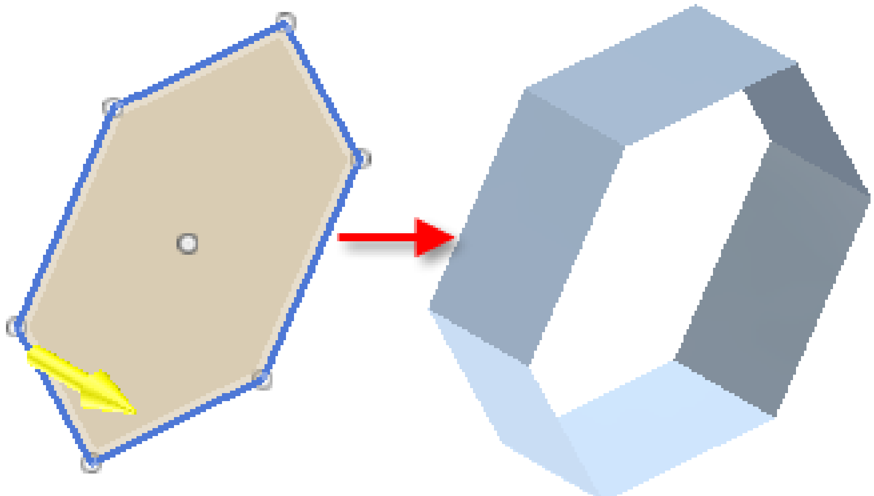
### Create a torus

1 On the Box drop-down list, click Torus.
2 Select a face or work plane. The revolution axis locates at the point where you click.
3 Click to define the radius of the revolution, or enter the value.
4 Click to define the cross-section radius, or enter the value.
5 Select the Boolean Option:
   - **Join** Adds material.
   - **Cut** Removes material.
   - **Intersect** Removes all material from the solid that does not overlap the new feature.
   - **New Component** Creates a child component in the active component.

## Sketch Based Features

### Extrude



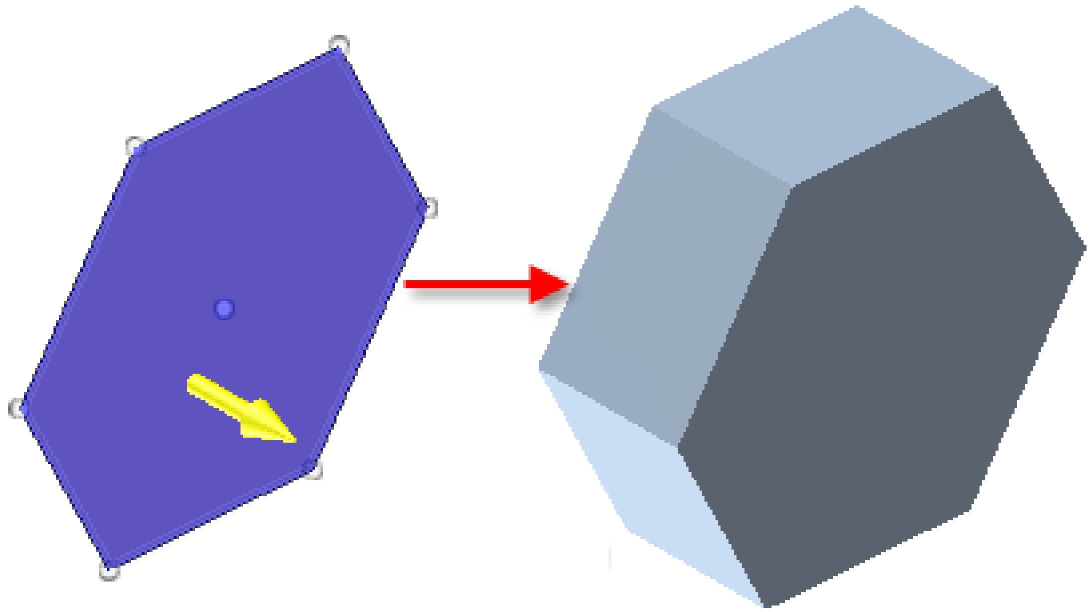


Extruded features are building blocks for creating and modifying models. You can create both solid and surface extrusions.

Extrude creates a feature by adding depth to sketch and model geometry. You specify the direction, depth, taper angle, and the type of feature for the extrusion.

An extruded solid requires a region. An extruded surface can use an open or closed profile.

#### Create a solid extrusion



Use the Extrude command to create a solid feature by adding depth to a region.

1 Switch the ribbon to Solid, if necessary. In the Solid panel, click Extrude.

2 In the graphics area, select one or more sketch regions or model faces. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.

3 Use the manipulators to set the height and taper, or enter the values.

   **NOTE** If you selected a face for the extrusion, the feature automatically adds or removes material based on the direction. See step 5 for instructions on manually setting the Join or Cut Boolean option.

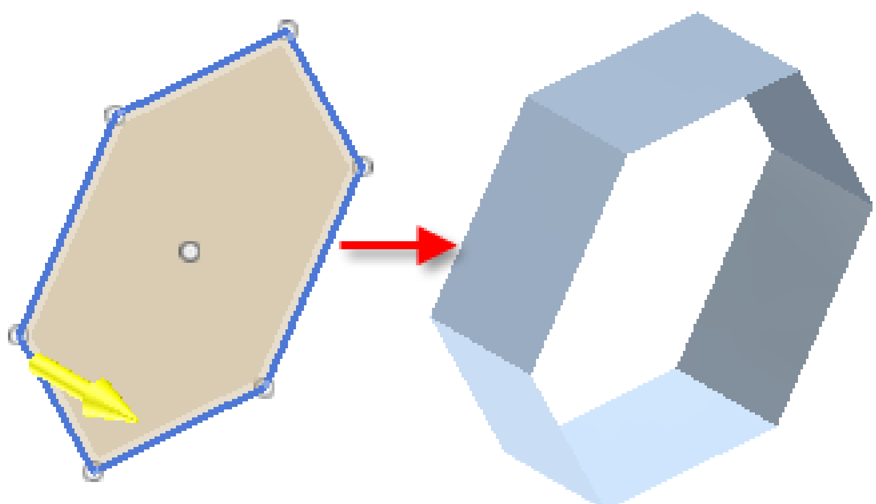
4 Select the Limit type
   - **Distance** A numeric value for the extrusion length.
   - **To** Determines the extrusion length when you select a face or work plane.
   - **All**Determines the extrusion length by the furthest face in the model.

5 Select the Direction
   - **One Direction** Creates the extrusion in one direction.

- **Two Directions** Creates the extrusion in both directions. Each direction can have a different extrusion length.
- **Symmetric** Creates the extrusion in both directions. Each direction has the same extrusion length.

6 Select the Boolean Option
- **Join** Adds material.
- **Cut** Removes material.
- **Intersect** Removes all material from the solid that does not overlap the new feature.
- **New Component** Creates a child component in the active component.

### Create a surface extrusion



Use the Extrude command to create a surface or component by adding depth to an open or closed profile.

1. Switch the ribbon to Surface, if necessary, in the Surface panel, click Extrude.
2. In the graphics area, select one or more sketch profiles, sketch entities, model faces, or model edges. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.

3 Use the manipulators to set the height and taper, or enter the values.

4 Select the Limit type:
   - **Distance**A numeric value for the extrusion length.
   - **To** Determines the extrusion length when you select a face or work plane.
   - **All** Determines the extrusion length by the furthest face in the model.

5 Select the Direction
   - **One Direction** Creates the extrusion in one direction.
   - **Two Directions** Creates the extrusion in both directions. Each direction can have a different extrusion length.
   - **Symmetric** Creates the extrusion in both directions. Each direction has the same extrusion length.

6 Select the New Surface or new Component for the Creation Option.

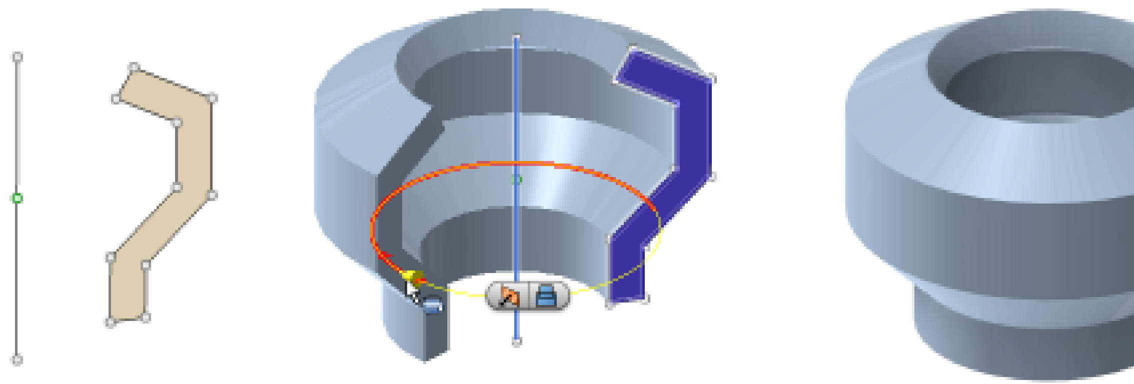
## Revolve

The Revolve command creates solid or surface features having radial symmetry such as stepped shafts and enclosures.

Revolve creates features by sweeping sketch and model geometry around an axis. The selection can be revolved through any angle measuring between zero and 360 degrees.

The axis can be a line in the profile, or a separate entity. An offset axis results in an axial hole in the feature. The axis cannot cross the region.

### Create a revolved solid



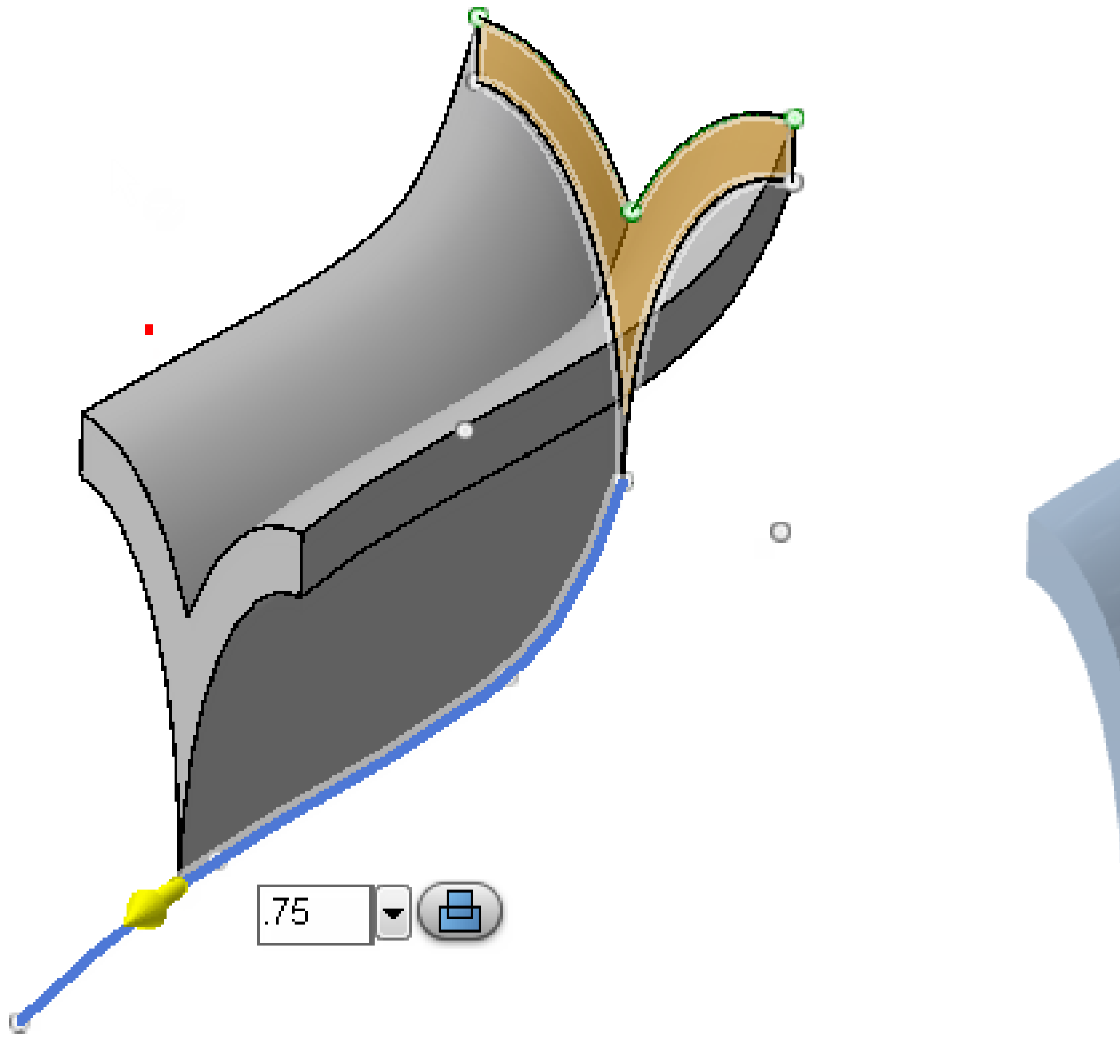
1. Click the drop-down arrow under Extrude, and then click Revolve.
2. Select one or more sketch profiles or model faces. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.
3. Select the axis for the revolution. If the selected axis is not on the sketch plane, the axis is temporarily projected to the plane.
4. Select the Limit type. Use the manipulator to set the rotation angle, or enter the value.
   - **Distance** A numeric value for the angle.
   - **To** Determines the angle when select a face or work plane.
   - **Full** Determines the angle by the furthest face in the model.
5. Select the Direction
   - **One Direction**Creates the revolve feature in one direction.
   - **Two Directions** Creates the revolve feature in both directions. Each direction can have a different angle.
   - **Symmetric** Creates the revolve feature in both directions. Each direction has the same angle.
6. Select the Boolean Option
   - **Join** Adds material.
   - **Cut** Removes material.
   - **Intersect** Removes all material from the solid that does not overlap the new feature.

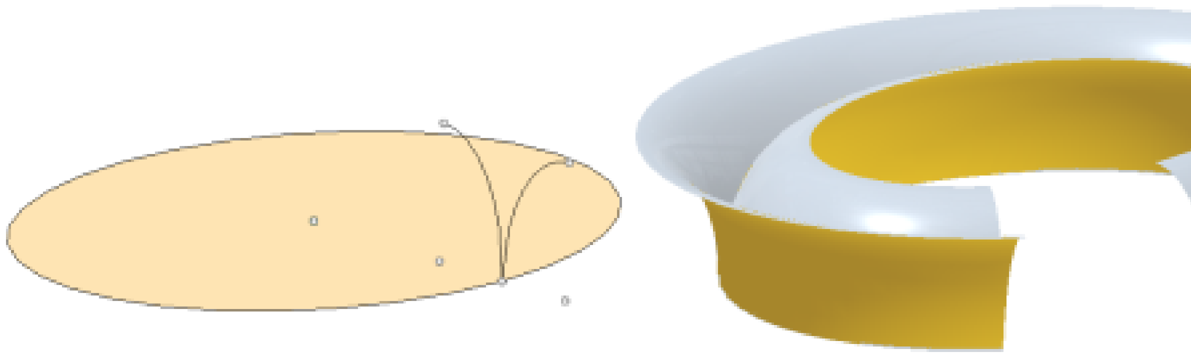
- **New Component** Creates a child component in the active component.

### Create a revolved surface

1. Click the drop-down arrow under Extrude, and then click Revolve.
2. Select one or more sketch profiles, sketch entities, model faces, or model edges. Multiple selections must be coplanar and in the same component. Sketch profiles and entities must be in the same sketch.
3. Select the axis for the revolution. If the selected axis is not on the sketch plane, the axis is temporarily projected to the plane.
4. Select the Limit type. Use the manipulator to set the rotation angle, or enter the value.
   - **Distance** A numeric value for the angle.
   - **To** Determines the angle when select a face or work plane.
   - **Full** Determines the angle by the furthest face in the model.
5. Select the Direction
   - **One Direction**Creates the revolve feature in one direction.
   - **Two Directions** Creates the revolve feature in both directions. Each direction can have a different angle.
   - **Symmetric** Creates the revolve feature in both directions. Each direction has the same angle.
6. Set the Creation Option to New Surface or New Component.

## Sweep





Sweep creates a feature by moving sketch or model geometry along a path. You can create both solid and surface sweep features.

**NOTE** A swept profile cannot intersect itself. Any curves in the path must have a radius greater than the width of the profile.

Paths for sweep features can be:

- A sketch or a model edge.
- Straight or curved.

### Create a swept solid

1. On the ribbon Solid tab, Extrude drop-down list, click Sweep.
2. Select one or more sketch profiles or model edges. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.
3. Select one or more sketch entities or model edges for the path. You can only select tangentially connected entities for the path.

   **NOTE** You can select a path from a different component than the profile.

4. Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path on the command ribbon.
5. Select the Orientation type:
   - **Perpendicular** keeps the region perpendicular to the path.
   - **Parallel** keeps the region parallel to the region sketch.
6. Select the Boolean Option:
   - **Join** Adds material.

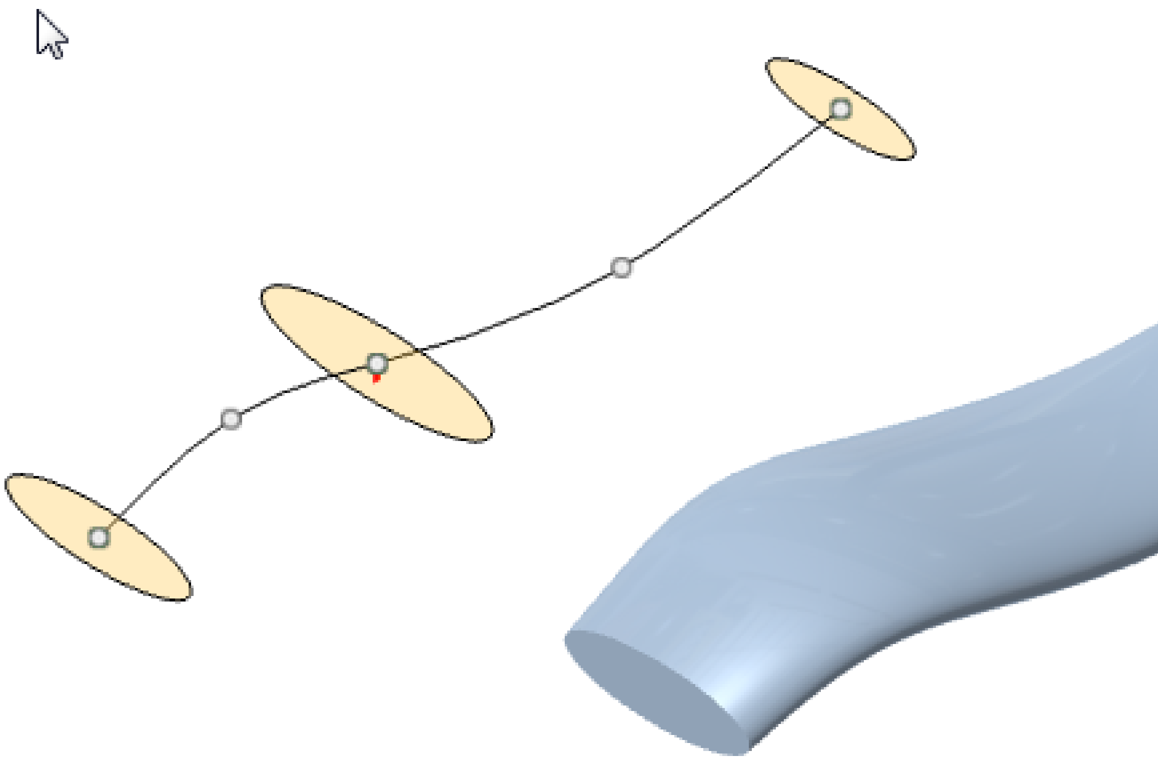
- **Cut** Removes material.
- **Intersect** Removes all material from the solid that does not overlap the new feature.
- **New Component** Creates a child component under the active component.

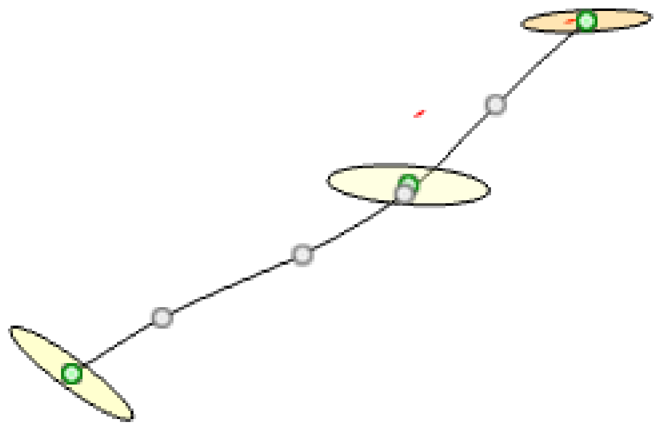
### Create a swept surface

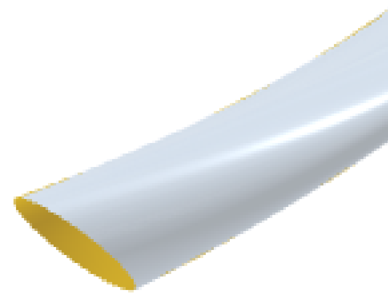
1. On the ribbon Surface tab, Extrude drop-down list, click Sweep.
2. Select one or more sketch profiles, sketch entities, model faces, or model edges. Multiple selections must be coplanar and in the same component. You can only select tangentially connected entities for the curve.
3. Select one or more sketch entities or model edges for the path. Multiple entities must be tangentially connected.
4. Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path in the command ribbon.
5. Select the Orientation type:
   - **Perpendicular** keeps the region perpendicular to the path.
   - **Parallel** keeps the region parallel to the region sketch.
6. Set the Creation option to New Surface or New Component.

Show me how to create swept features

## Loft





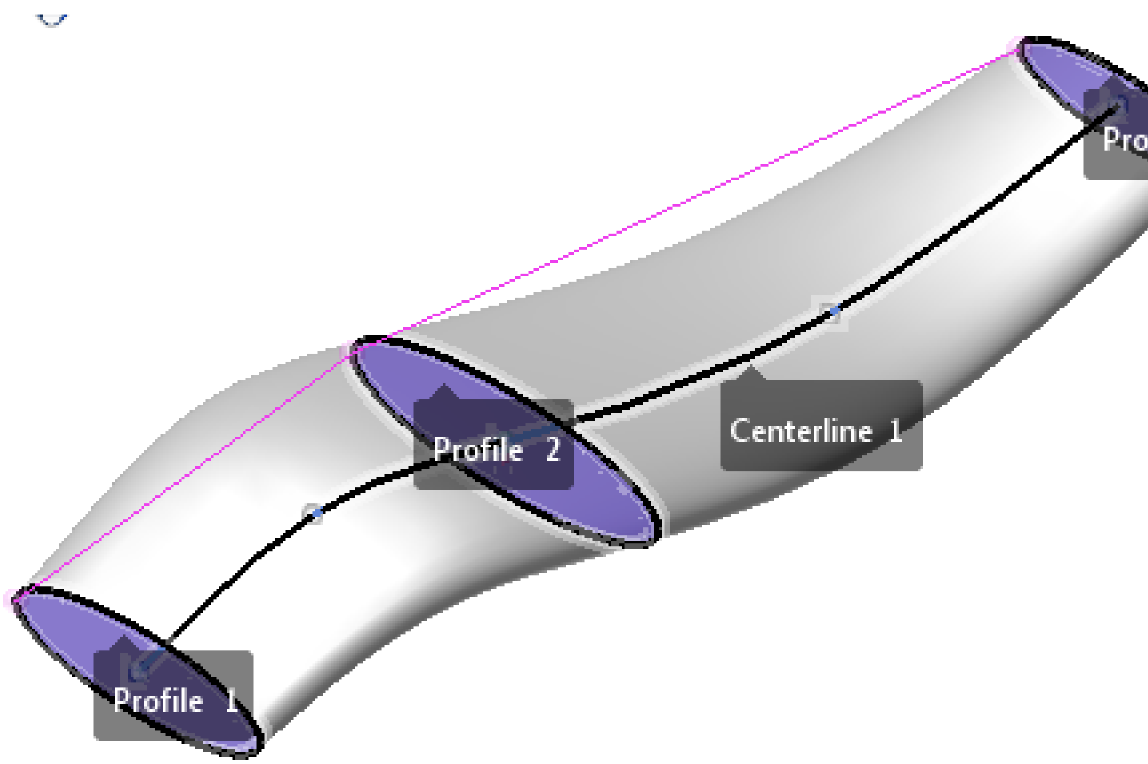


Loft creates complex, organic shapes. These shapes are common in the automotive, marine, and consumer products industries.

Loft creates a smooth shape that transitions between two or more sketch profiles or faces. You can select sketch profiles and model faces for a solid. You can select open or closed profiles for a surface. A surface loft cannot have both open and closed profiles.

You can also add a centerline to define a path for the loft, or you can change the curvature at each profile.

### Create a lofted solid




1. Switch the ribbon to Solid, if necessary, and in the Extrude drop-down list, click Loft.
2. In the graphics area, select the sketch regions or model faces. Select multiple regions in their order. You cannot select the first and last region, and then a middle region.
3. Optionally, select the centerline path that passes through each profile. The path can have multiple, tangent entities. Select each path entity.

   **NOTE** You can select the centerline path entities at any time.

4. In the command ribbon, click Closed to return to the first region in the loft. To close the loft, it is required that it extends more than 180 degrees.
5. Select the Boolean Option:

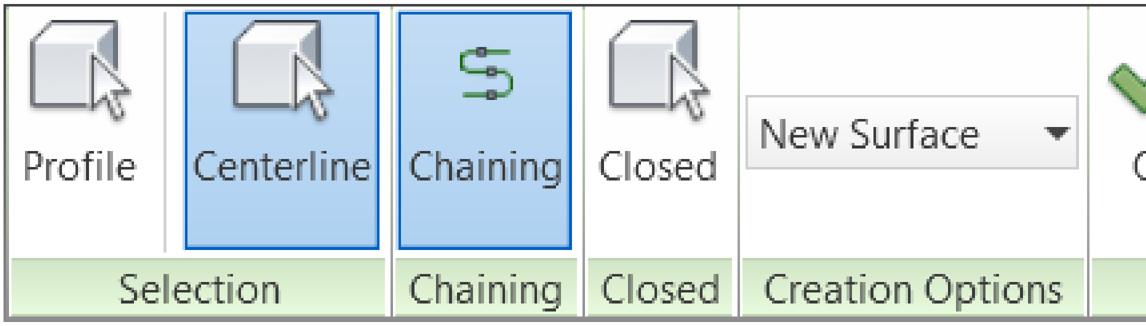
   a. **Join** Adds material.

b **Cut** Removes material.

c **Intersect** Removes all material from the solid that does not overlap the new feature.

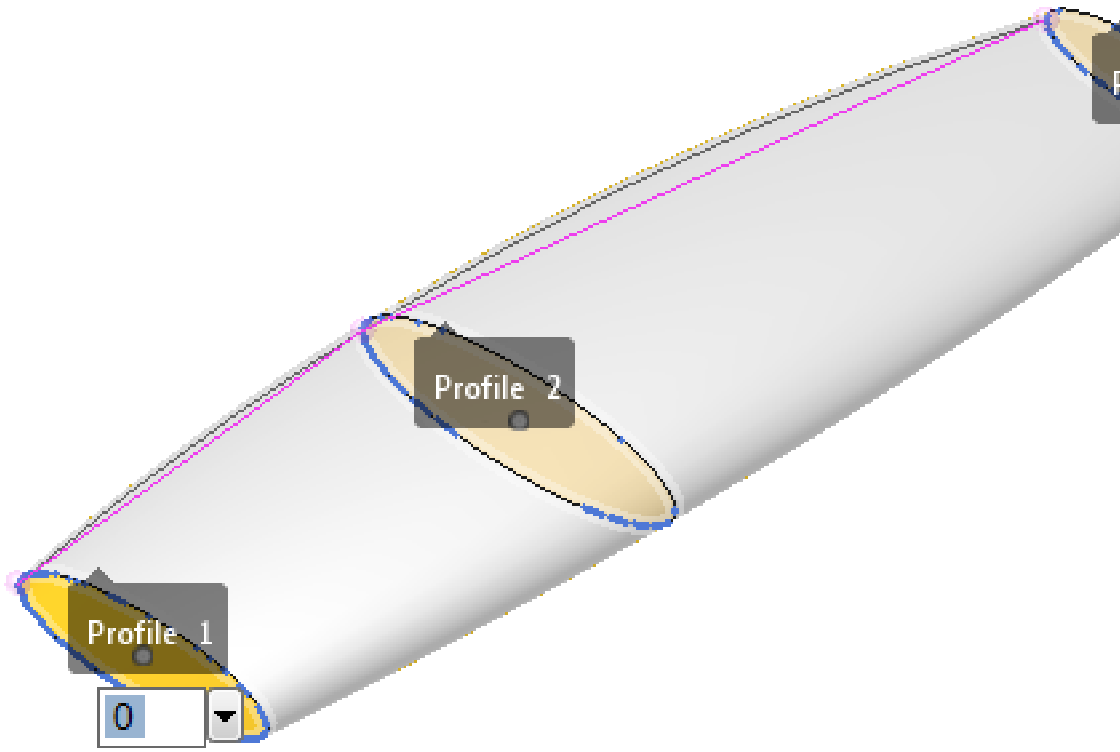
d **New Component** Creates a child component in the active component.

### Create a lofted surface

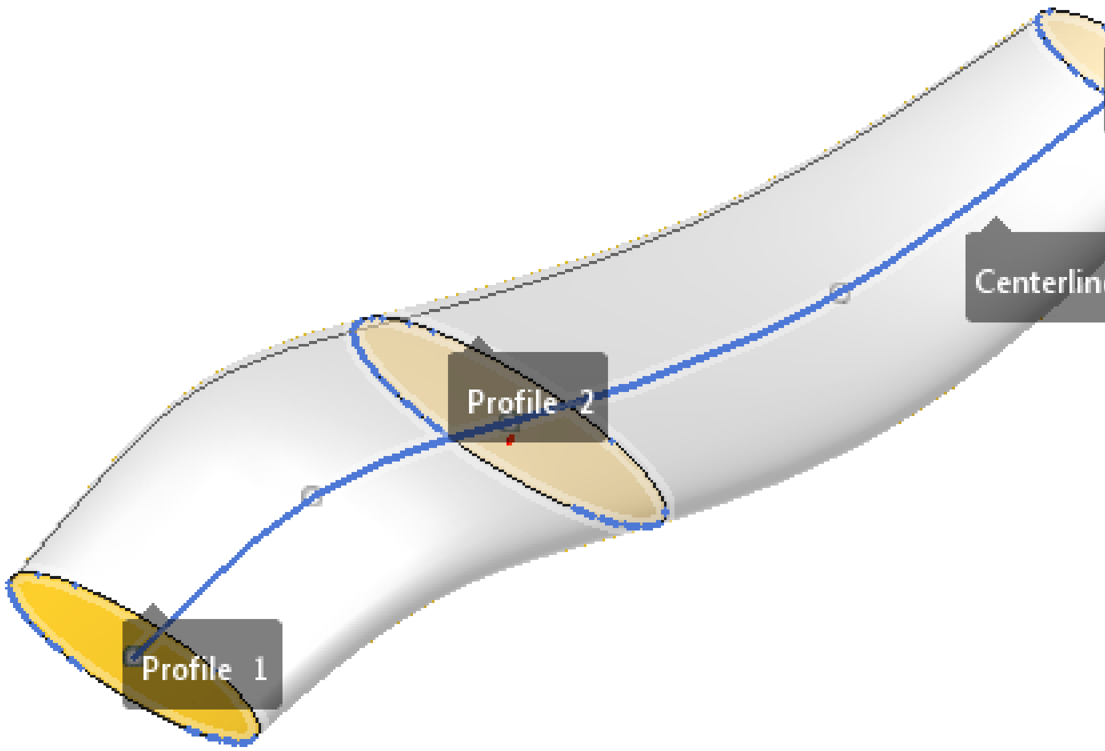



Use profiles that are open or closed. You can use a centerline path to control the curvature, or you can change the curvature at a profile. It is not required that the centerline path connects the profile centers, but it is required that it intersects the profiles.

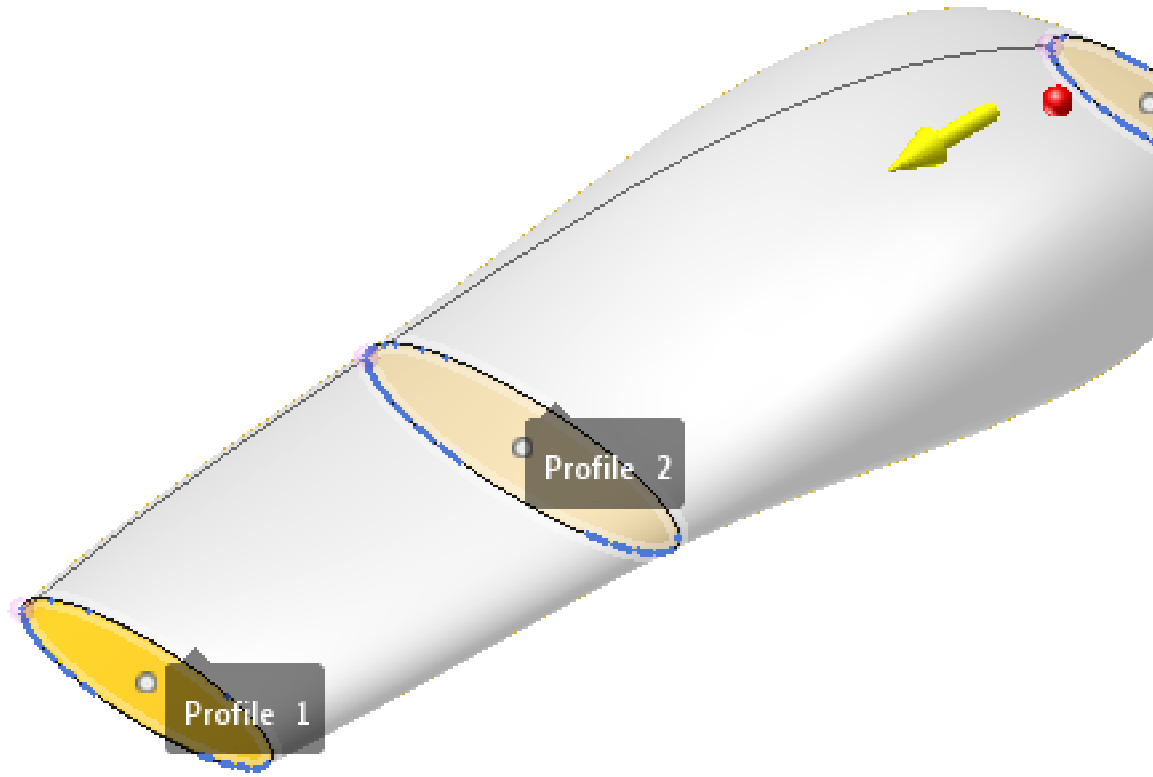
You can create the loft with natural curvature, with a centerline path, or with transition curvature changes at each profile.




This loft uses the natural curvature.




This loft uses a centerline path.



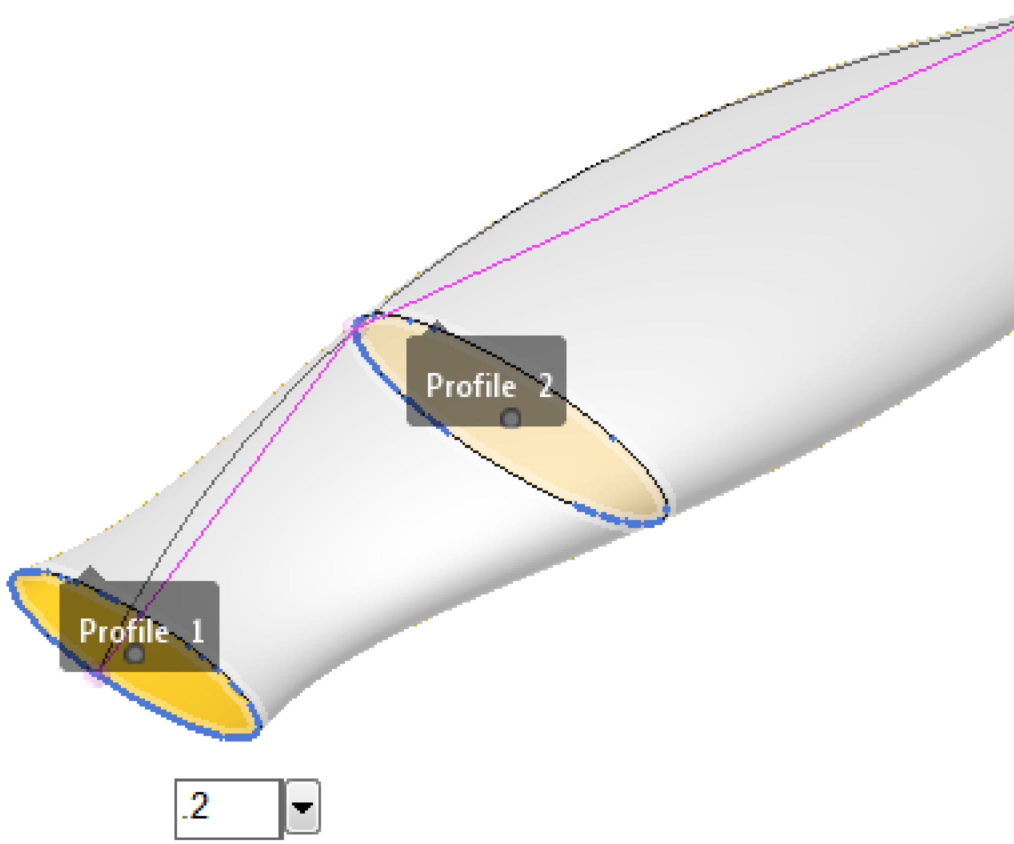

This loft uses directional control at the last profile.

1 Switch the ribbon to Surface, if necessary. In the Extrude drop-down list, select Loft.

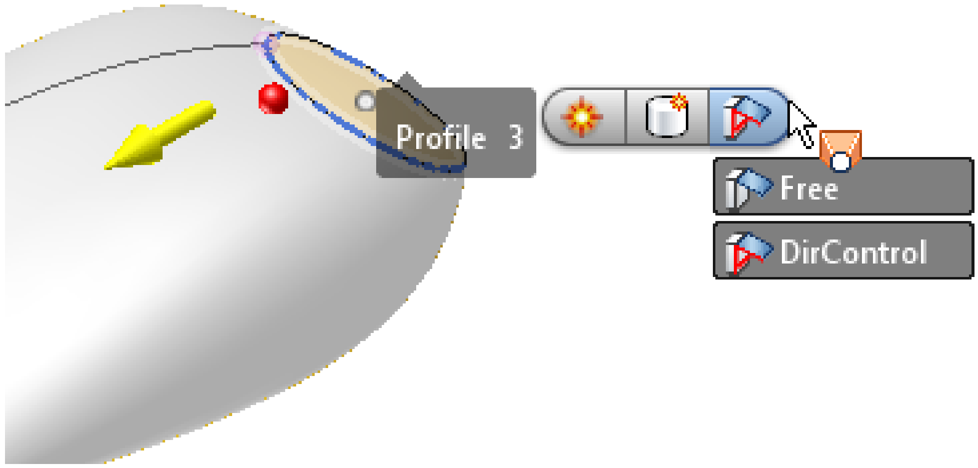
2 In the graphics area, select the sketch profiles and faces. You cannot select both open and closed profiles. Select the regions in order. You cannot select the first and last region, and then a middle region.

3 Optionally, change the transition curvature for the entire loft, set the transition curvature as you select each region, or select the centerline path.

   1 To change the transition curvature for the entire loft, in the edit box, enter a value between 0.0 and 1.0. In this image, the transition curvature is set to 0.2




   2 To change the transition curvature at a profile, on the command glyph, select DirControl (Directional Control). Use the manipulators to change the curvature.




**NOTE** If you select DirControl for a profile, the loft transition curvature is reset to 0.

3 Click the Centerline button, and select the path, which can have multiple, tangent entities. Chaining for the path is turned on by default. Turn chaining off to select the path entities manually.

4 In the command ribbon, click Closed to return to the first region in the loft. To close the loft, it is required that it extends more than 180 degrees.

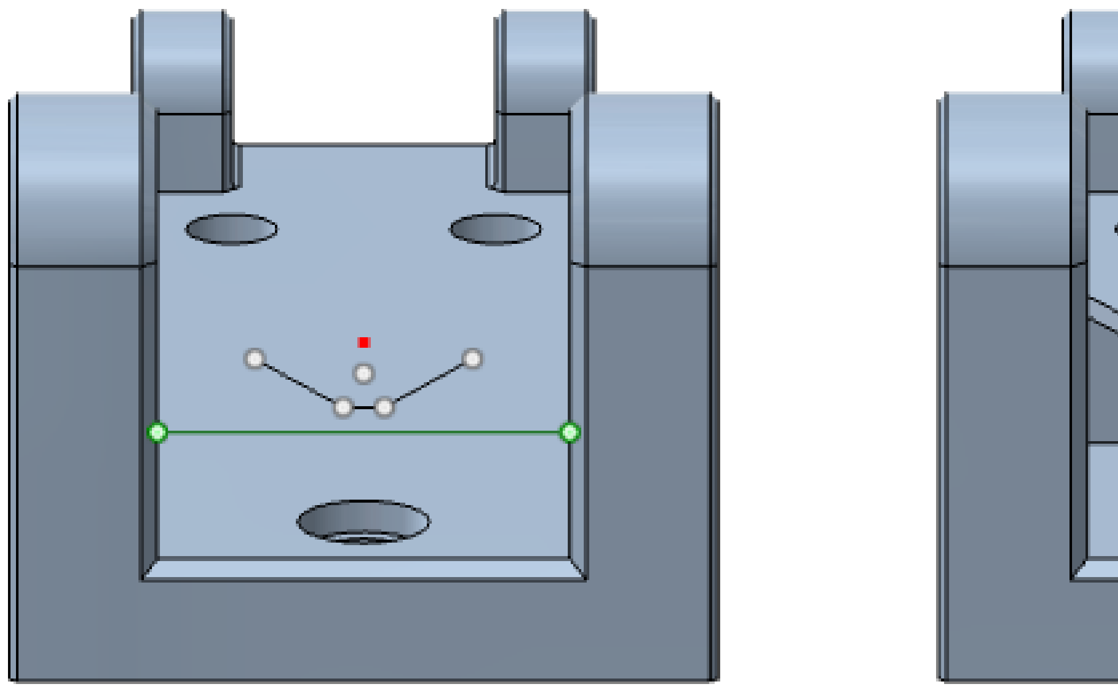
5 Select the Creation Option:

   1 **New Surface** Creates the surface in the active component.

   2 **New Component** Creates a child component for the surface under the active component.

**Show me how to create a surface loft**

## Rib

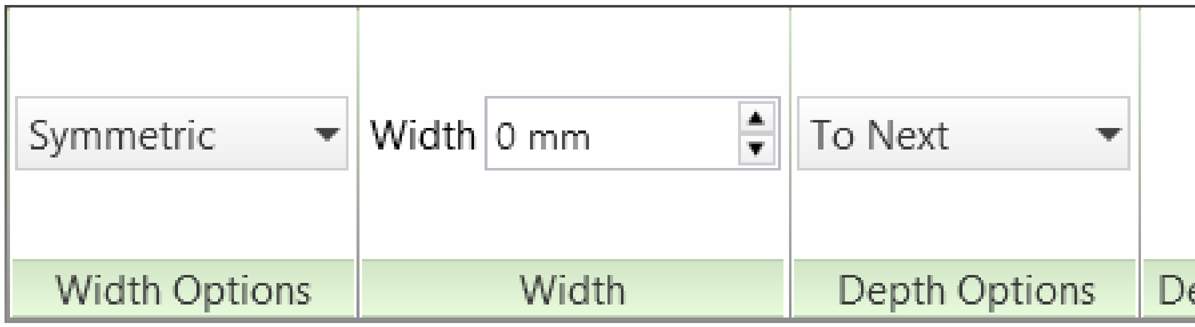


Ribs and webs are features that extend to adjacent faces. The sketch curve can be a series of lines and curves. The ends of the curve are automatically trimmed or extended to the adjacent faces.

- Ribs can have a thickness normal to the sketch plane and extend material planar to the sketch.
- Ribs can have a thickness planar to the sketch and extend material normal to the sketch plane.

Ribs and webs are often used in molds and castings. In plastic parts, they create rigidity and prevent warping.

**Create a rib**

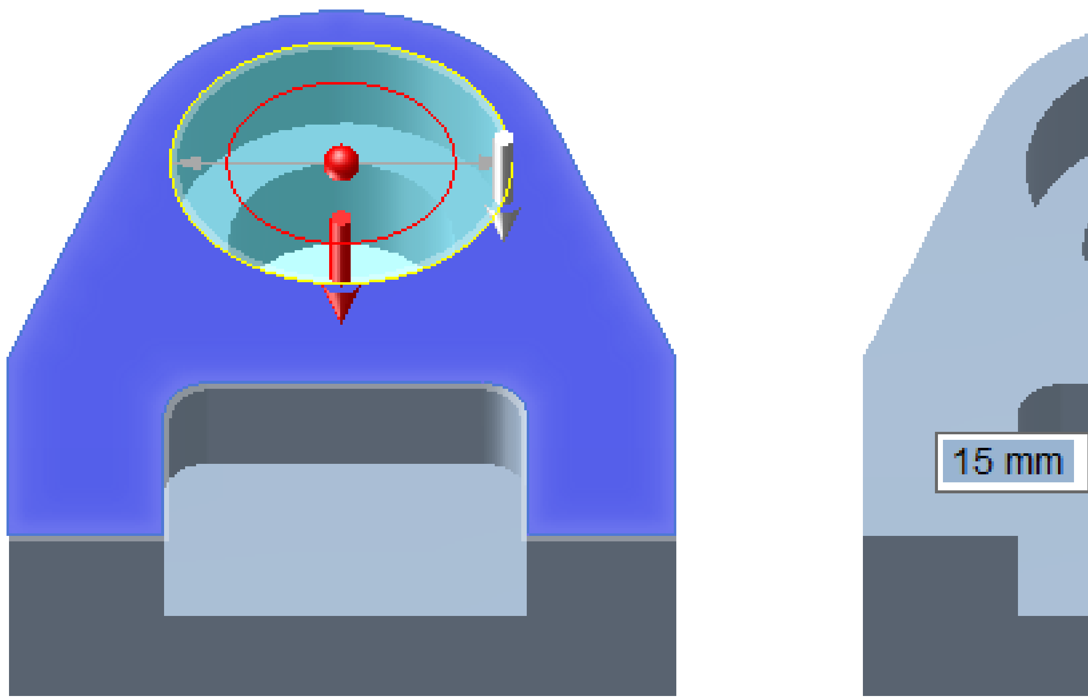



1 On the Extrude drop-down list, click Rib.
2 Select the sketch curve.
3 Set the width option.
4 Use the manipulator to set the width, or enter the value.
5 Set the depth option:
   - **To Next** Extends the curve to the adjacent faces to form a region.
   - **Depth** Sets a value for offsetting the curve. The end of the rib cannot extend past the faces. If the depth value is too large, the rib is trimmed back so the end still touches the face.
6 In Depth Direction, click Flip to change the side of the profile that extends.

**Show me how to create, and copy and paste, a rib**

## Placed Features

### Hole




The Hole feature creates three types of drilled holes: simple, counter bored, and countersunk.

You can reference other model geometry to set the location, diameter, and depth of the hole. When the ball manipulator is active, you can select points, lines, and edges for location references. When the arrow manipulator is active, you can click a face to set the hole depth.

You can edit a hole feature by right-clicking on it in the browser and selecting Edit Hole.

**Create a hole**

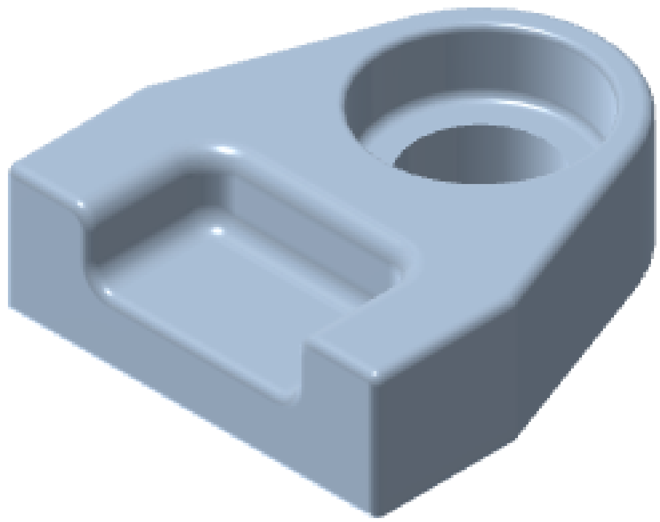
1. Start the Hole command.
2. Click a face. The manipulator displays where you click.
3. Use the ball to move the hole location on the face.

    a. Click a sketch point or an end point to snap to that location.
    b. Click a circle, arc, or ellipse to snap to the center.
    c. Click a line or edge to add a transient dimension.
    d. Click the arrow and drag it to change the hole depth. Pause the cursor over a hidden face to highlight it for the depth selection.

4. Click the ring and drag it to change the hole diameter. Click a cylindrical face to set the hole diameter.
5. Change the tip angle in the command ribbon.
6. For a Counter Bore hole, use the second arrow and ring to set the depth and diameter.
7. For a countersunk hole, use the second ring to set the diameter and change the angle in the command ribbon.

Show me how to create holes

## Fillet



Fillets round over exterior and interior edges. A concave fillet feature is called a fillet and a convex fillet feature is called a round.

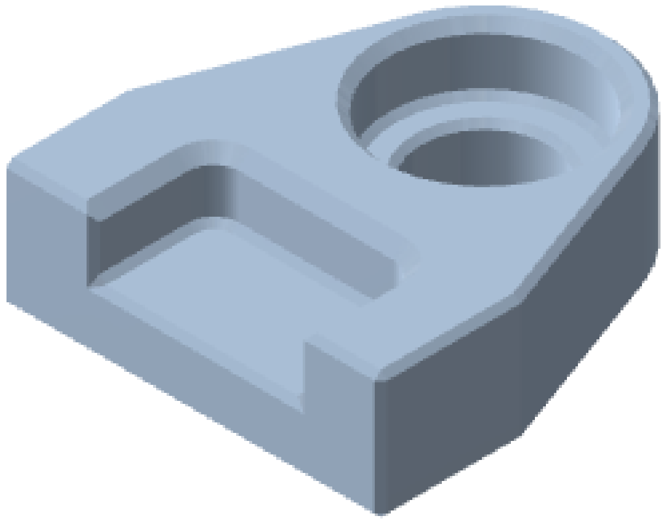
You can create a fillet on a solid or on a quilted surface.

You can edit a fillet by right-clicking on it in the browser and selecting Edit Fillet, or by using Press/Pull.

**Create a fillet**

1. Start Fillet.
2. Select an edge- the manipulator displays.
3. Deselect the Tangent Propagation option if you do not want tangentially connected edges.
4. Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
5. Deselect the Rolling Ball Corners option if you want a smooth transition between edges. Rolling ball fillets produce corners like a milling machine or a router.

## Chamfer



Chamfers create a bevel and removes material from an outside edge and adds material to an inside edge. Chamfers can be equal distance from the edge, a specified distance and angle from an edge, or a different distance from the edge for each face.

You can create a chamfer on a solid or on a quilted surface.

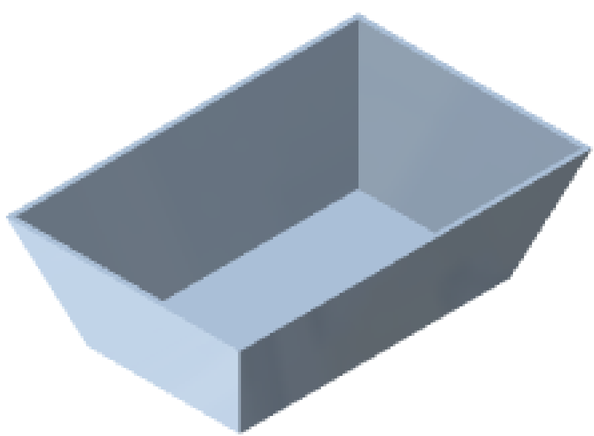
**Create a chamfer**

1. Start Chamfer.
2. Select an edge- the manipulator displays.
3. Deselect the Tangent Propagation option if you do not want tangentially connected edges.
4. Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
5. Select another Chamfer Option to change how the chamfer is defined.

## Modifying Features

Modifying features make changes to existing objects.

### Shell



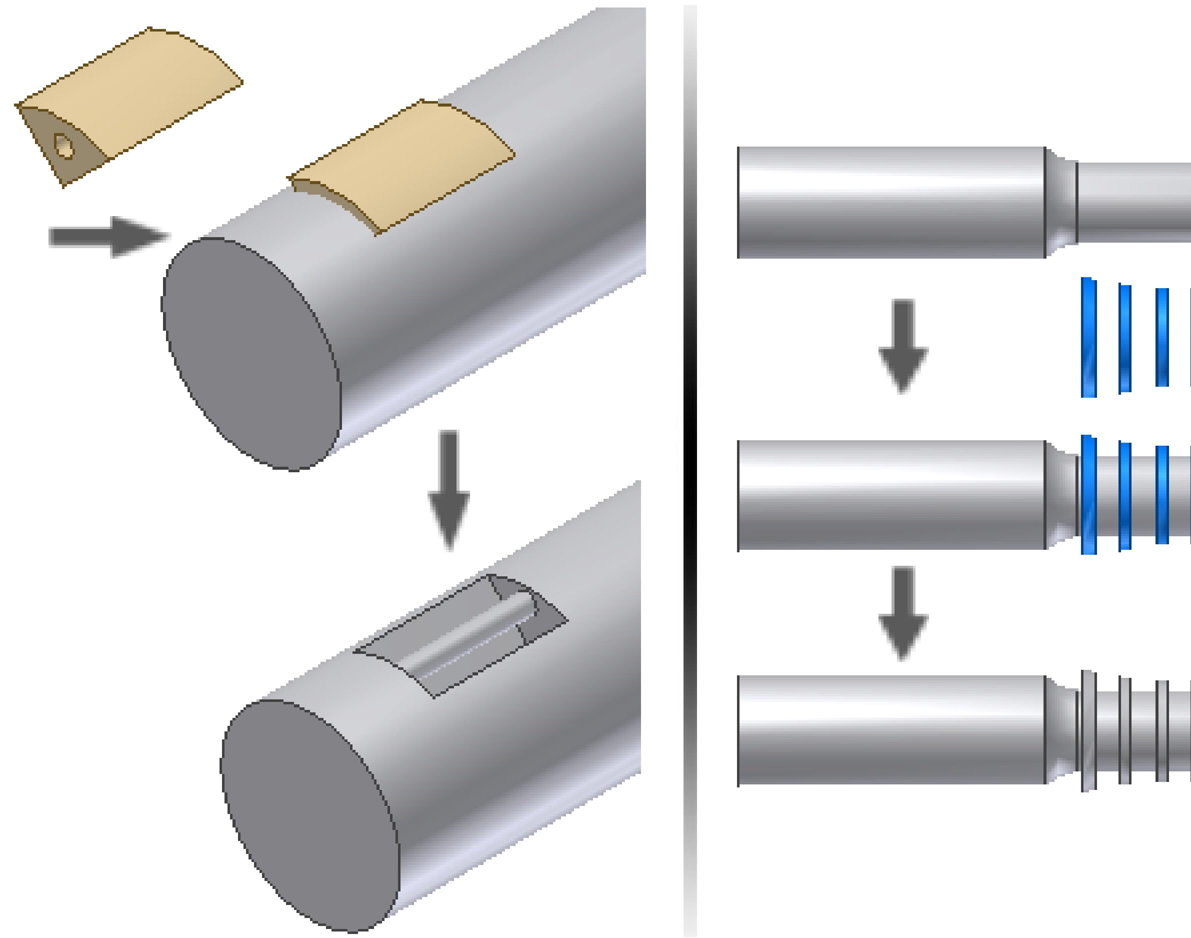
Use Shell to create a thin wall solid. The walls can offset towards the inside or outside of the solid.

Shell removes material from a part interior, creating a hollow cavity with walls of a specified thickness. Shell simplifies the modeling process. You create solid features to describe the model shape, and then remove unnecessary material. You can create a hollow solid, or remove faces to form an opening.

#### Shell a solid

1. Start Shell from the Solids panel
2. Select one or more faces to remove from a solid. You can also select the solid without selecting faces.
    1. Optionally, clear the check box for Tangent Propagation to prevent automatic selection of tangent faces.
3. Drag the manipulator to set the wall thickness, or enter a value.
4. Change the direction of the shell:
    - **Inside** Offsets the faces toward the interior of the part.
    - **Outside** Offsets the faces towards the exterior of the part.
    - **Both** Offsets the faces towards the interior and the exterior of the part. You can have different values for the offsets.

## Boolean



The Boolean command joins, cuts, or intersects the selected components. When you use the Boolean command, the Target is the component being acted upon, and the Tools are the components that change the model. You can select more than one tool to use in a Boolean operation.

The default behavior is to delete the tools after modifying the target. The component is kept in the browser, but the solid body is deleted. Choose Keep Tools if you do not want to delete the solids.

### Combine multiple components

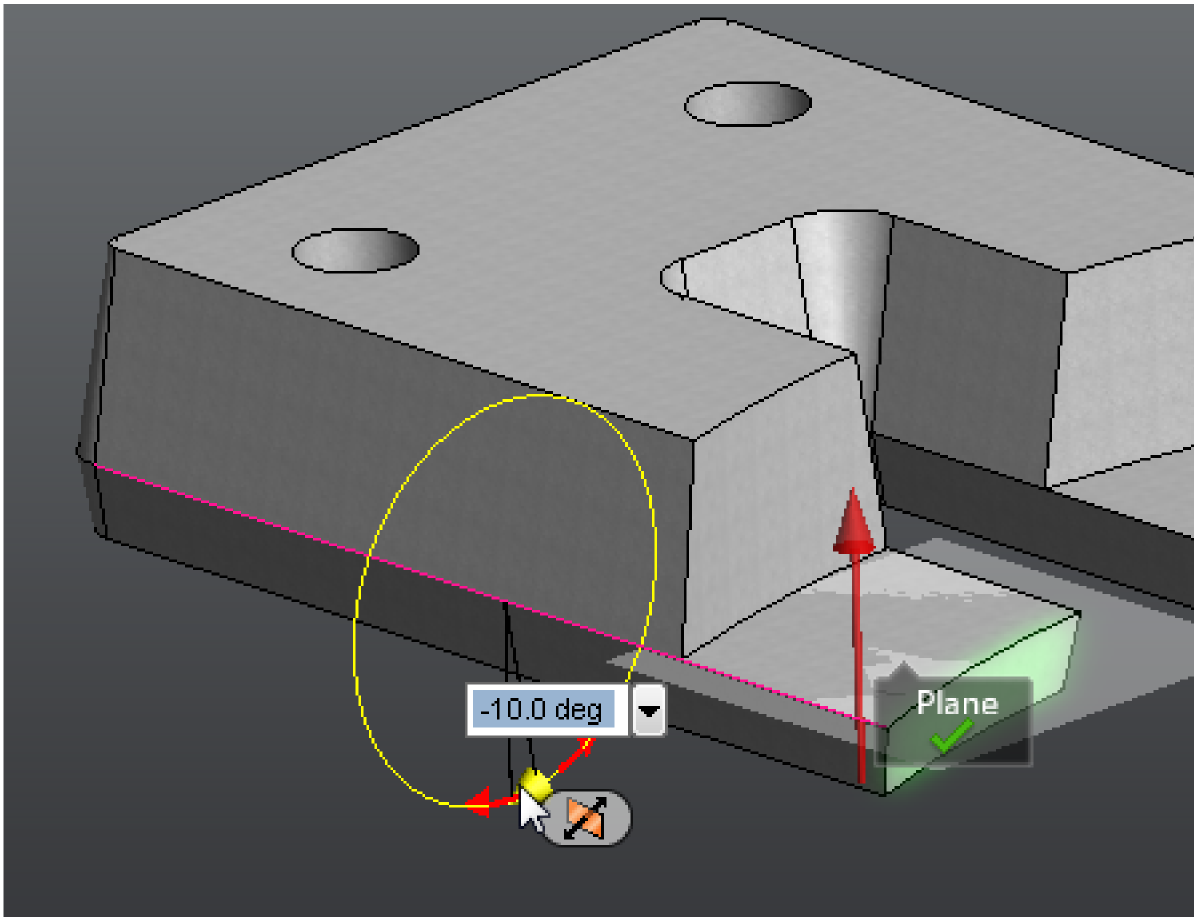
Use the Boolean command to combine solids in a model. Since a component only has one solid, the model must have multiple components with solid bodies.

1. Start Boolean
2. Select the Target solid.
3. Click on Tools, and select the solids that to modify the target.
4. Select the Boolean option.
    - a **Join** combines the target and tools into one solid.
    - b **Cut** removes the overlapping material from the target.
    - c **Intersect** keeps only the overlapping material.
5. Click New Component to create a component with the results.
6. Click Keep Tools to retain the tool solids.
7. Click OK to create the combined solid.

**Show me how to use Boolean operations**

## Draft Command

The Draft command in Inventor Fusion can be used to modify one or more component bodies by creating angled faces, with respect to a neutral plane.




This command is useful for creating parts that are manufactured using an injection molding or metal casting process. Some of the faces on such parts are angled so that the part can be removed from the mold easily. In these parts, Draft is applied to all of the side faces of the design.

You can also use Draft as a general modeling command for creating individual angled faces.

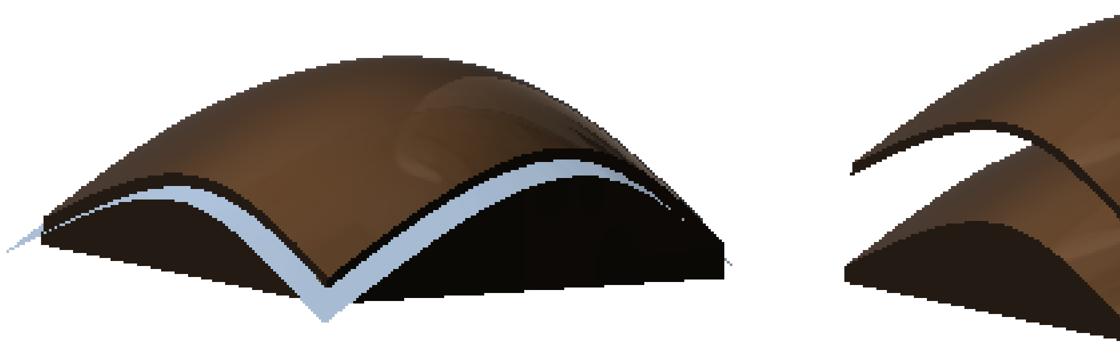
**Show me how to add draft to a solid**

This video demonstrates different methods to use the draft command.

### Apply draft to a solid

1. Invoke the Draft command.
2. Select a neutral plane around which faces are drafted, and the pull direction. The pull direction is the direction in which the mold is removed from the part after the molding process is complete.
3. Select the faces to be drafted.
4. Select the draft type:
   - One Way: specify a single draft angle.
   - Symmetric: specify a single draft angle that is applied above and below the neutral plane.
   - Two Way: specify two draft angles; one above the neutral plane and one below the plane.
5. Drag or enter a precise value into the draft angle entry box.
6. Click OK to complete the command.

## Split



Use Split to divide faces on a surface or solid, or divide a solid into two components. To start, create a work plane, surface, or sketch geometry to use as the splitting tool. Sketch geometry is projected onto the target faces.

You can split faces or a solid. Faces are typically split to add draft, delete an area, or to create new features. A solid is divided into two solids- one solid remains in the component and a child component is created for the second solid.

You can split a solid, surface, or quilt.

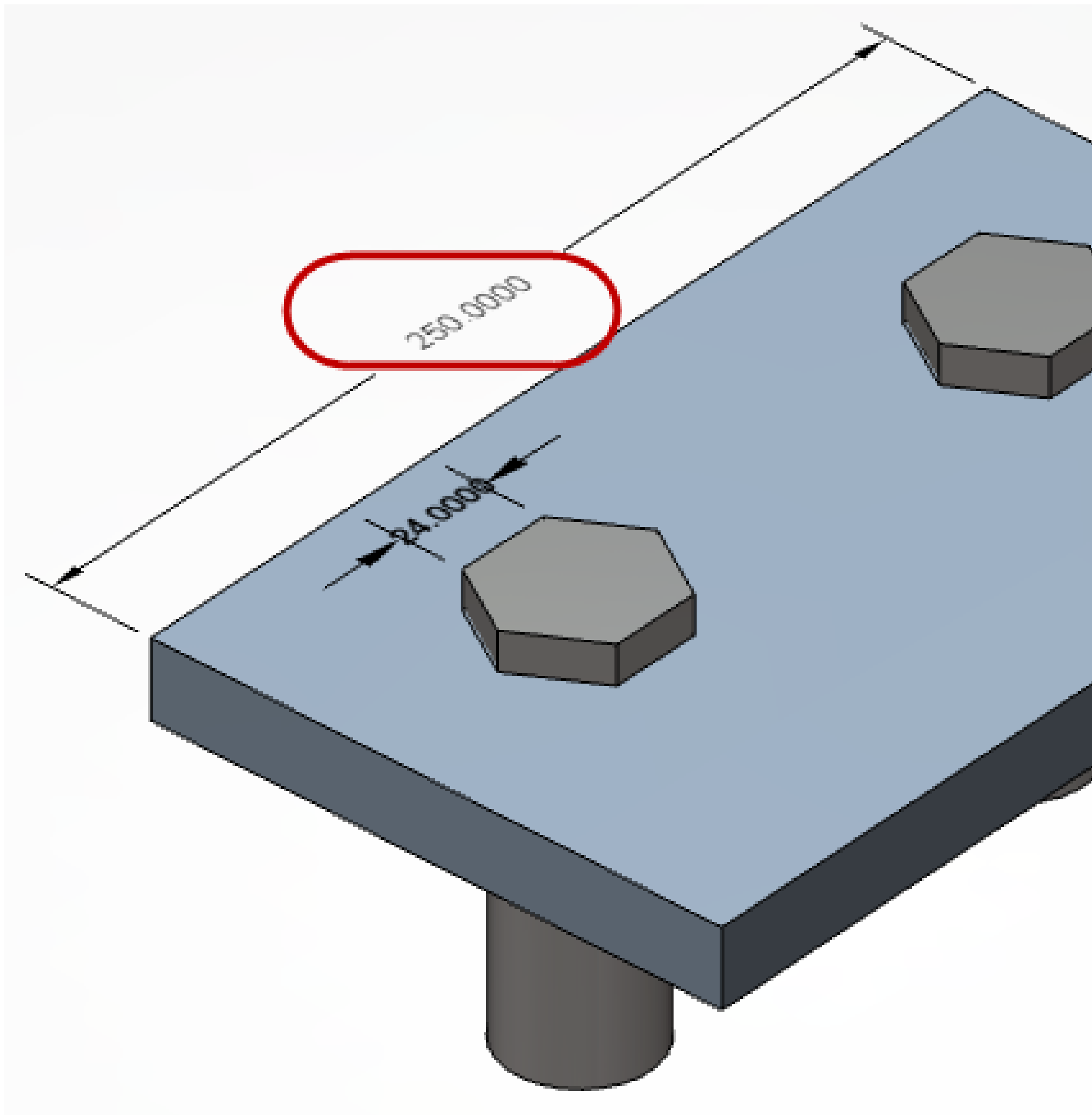
**Split a face, surface, or solid**

1. To start, create a work plane, surface, or sketch geometry to use as the splitting tool. Sketch geometry is projected onto the target faces.
2. Start the Split command.
3. Select the target faces or solid.
4. Click Split Tool, and select the intersecting work plane, surface, or sketch geometry. The Tool Chaining option automatically selects tangentially connected faces.
5. Click OK.

**Show me how to create a split**

## Scale Command

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling. Valid objects are components, solids, surfaces, and sketches.

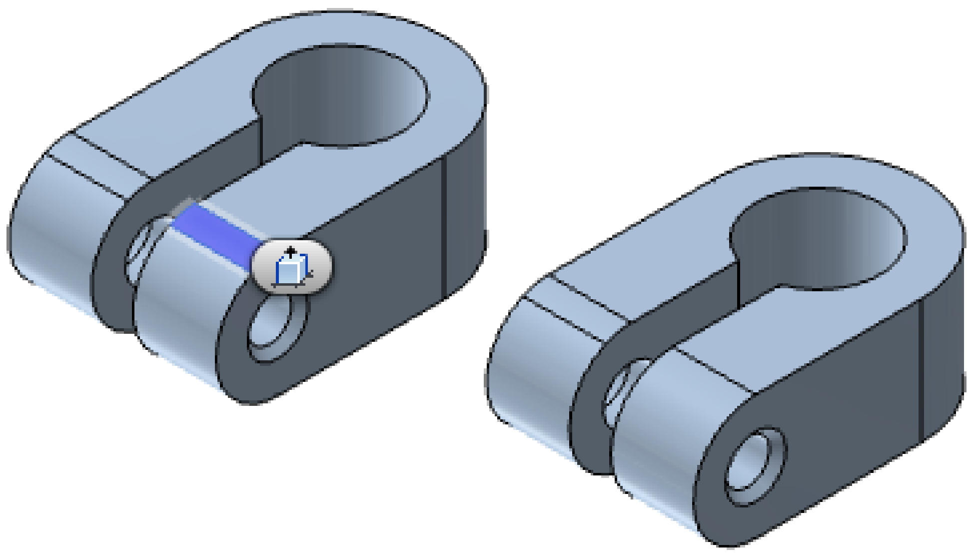



To scale an object:

1. Start the scale command.
2. Select the objects that you wish to scale.

3 Select a reference point to scale around.
4 Enter a scale factor or click and drag.
5 Click OK to complete the command.

## Delete

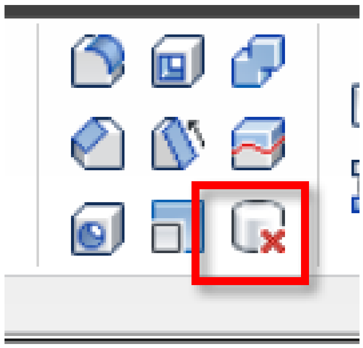


Delete removes a Face, Solid, Sketch Entity, or Construction Entity.

Delete cannot convert a solid into a quilted surface. If a face on a solid is deleted, adjacent faces are extended or modified to heal the model. An error displays if you select a face that cannot be replaced.

If you want to remove a face, use the Delete Face (page 252) command on the Surface ribbon.

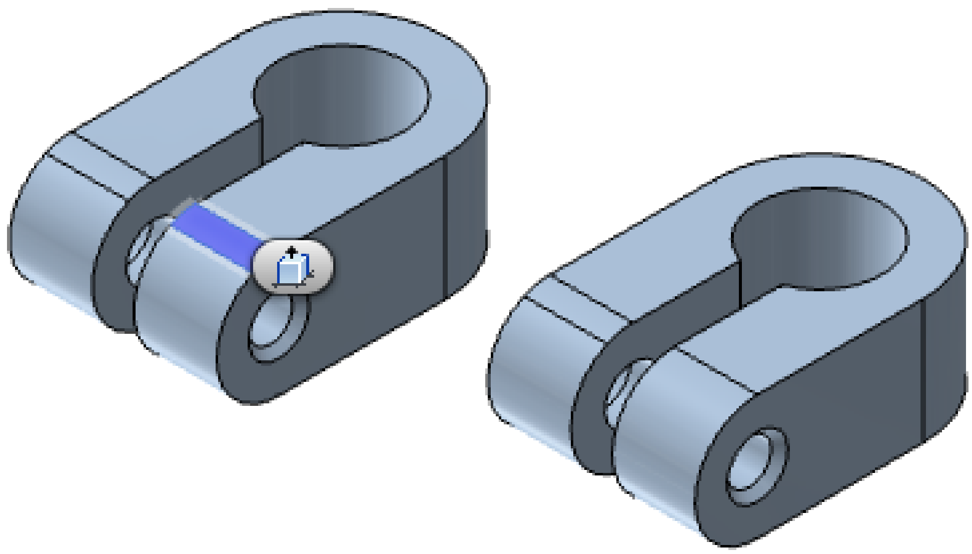
#### Delete an object from a model



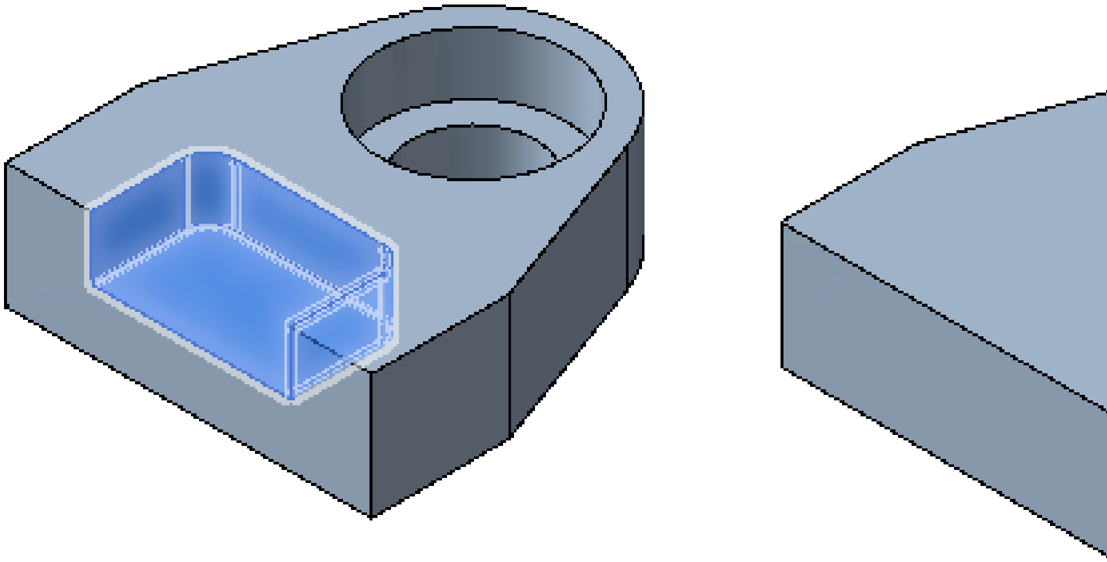
1 Switch the ribbon to Solid, if necessary.
2 Select a face, solid, sketch, feature, sketch entity, or construction entity.
3 Click Delete on the Solid panel. The selection is deleted, and the model is healed if necessary. An error displays if the model cannot be healed.

### Delete face examples

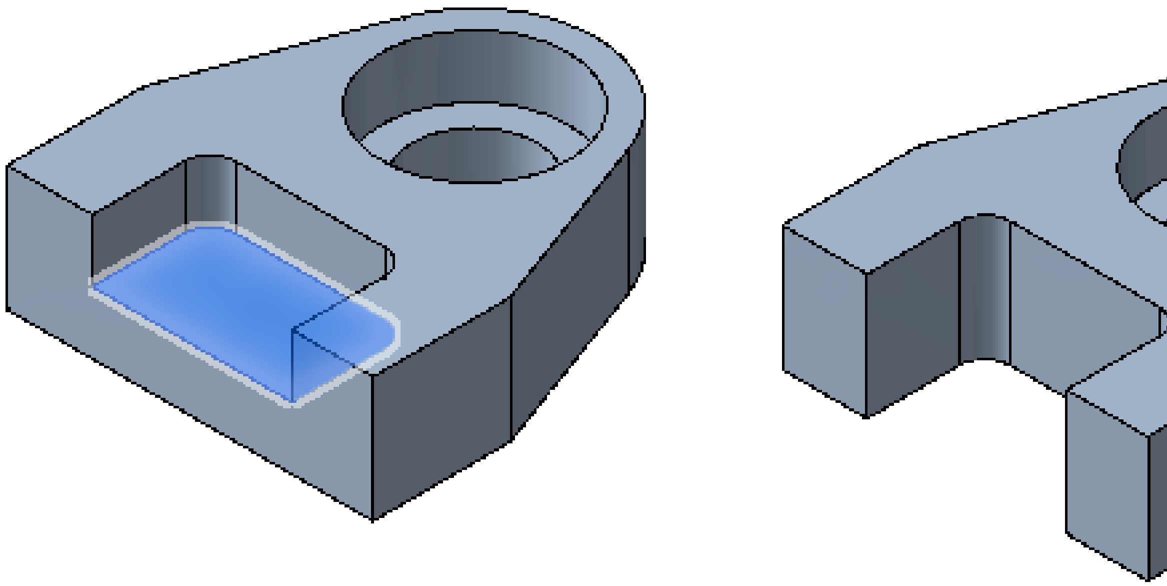
The delete command modifies the solid to heal the opening left by the deleted the face. The change depends on model geometry and the selected faces. This table has several examples of delete face operations.



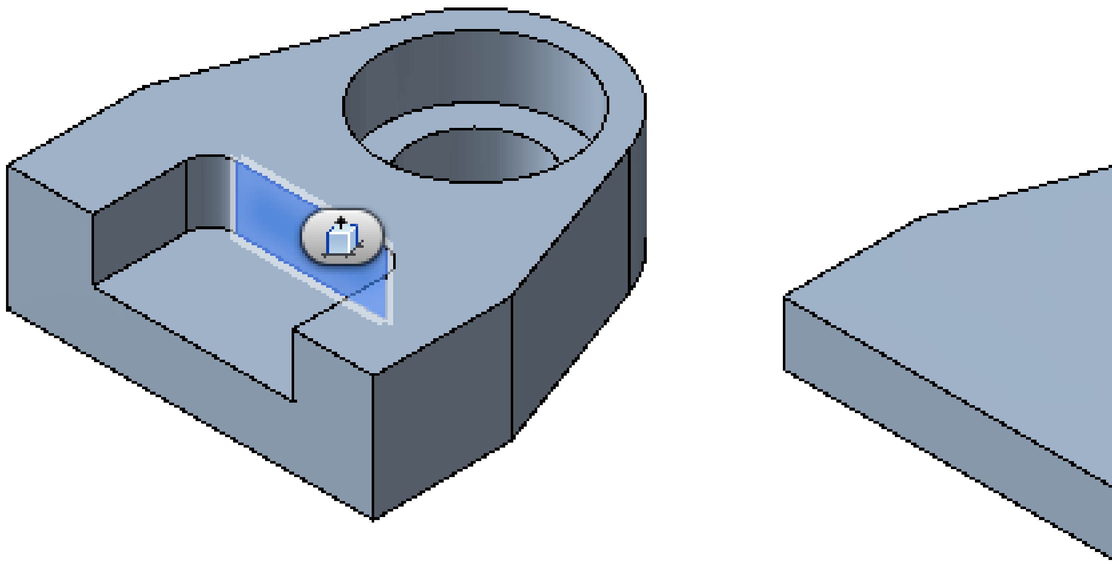
The selected face is tangent to the radial face. The radial face is extended to the planar face.



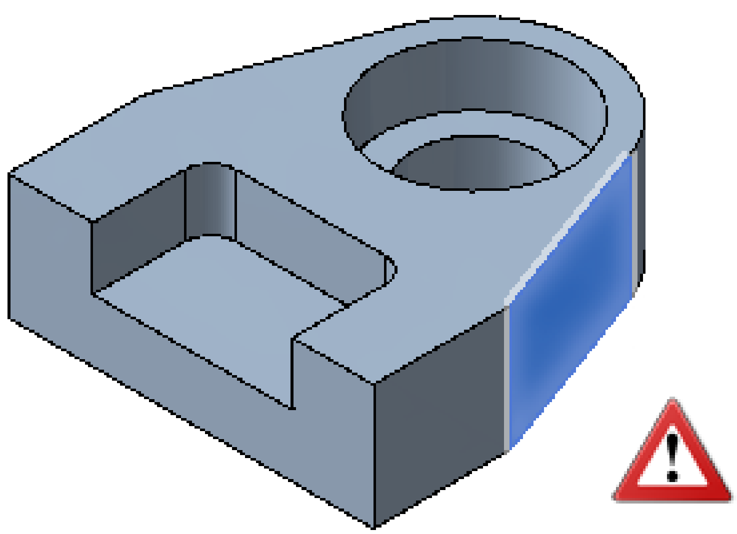
All faces in the pocket are deleted. The top and end faces are extended.



The model is cut, and the side faces of the pocket are extended

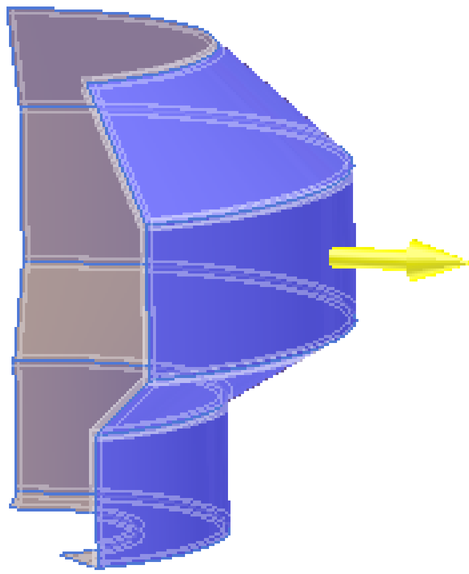


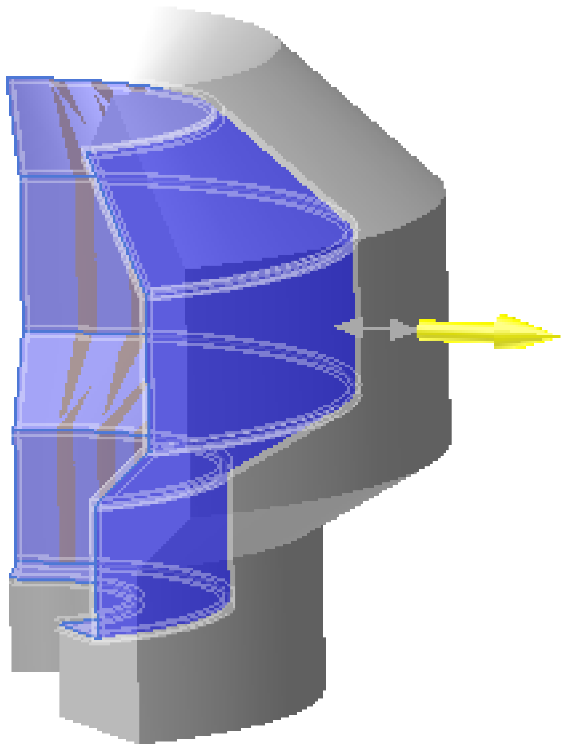
The model is cut, and the bottom face of the pocket is extended.



None of the adjacent can be extended to heal the model, so an error displays.

### Thicken





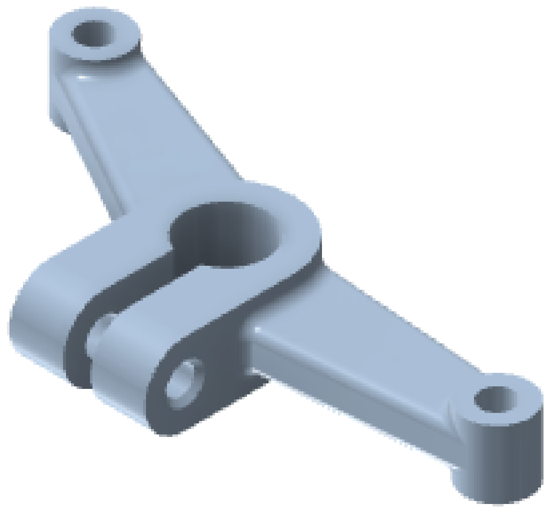
Thicken offsets surfaces and quilts, and adds sides to create a solid. All surfaces are offset an equal distance from the originals. You can select multiple surfaces and quilts.

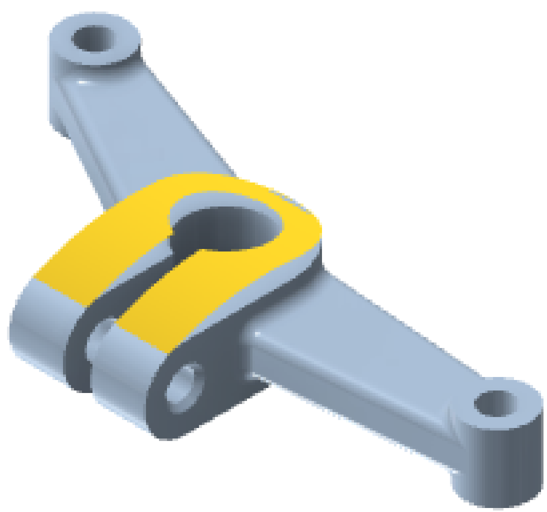
Thicken cannot remove a face, so the maximum offset value is set at the distance where a face disappears.

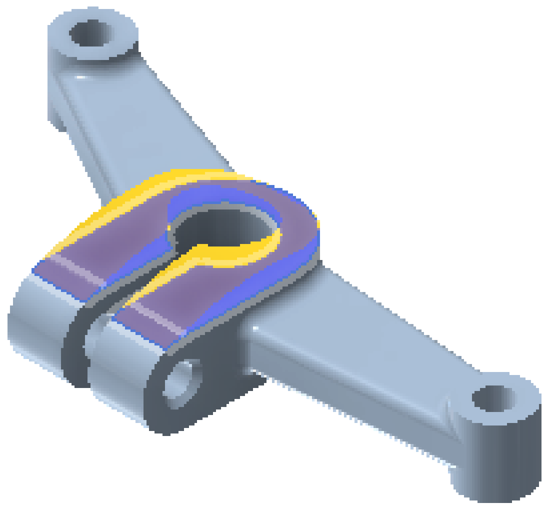
**Create a solid using Thicken**

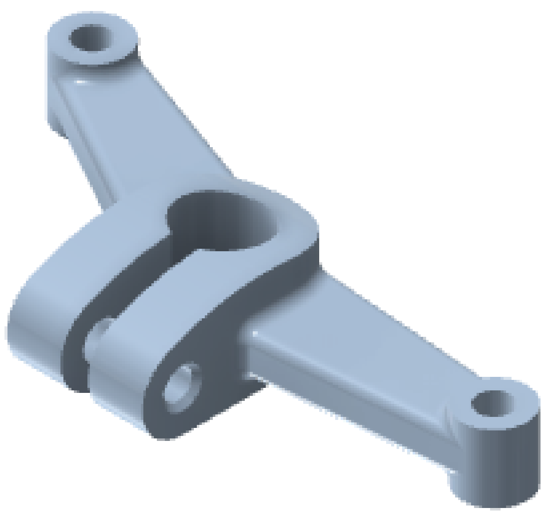
1. Start the Thicken command.
2. Select one or more surfaces or quilts.
3. Drag the arrow to set the offset, or enter a value.
4. Click OK to create a solid.

## Replace Face









Use Replace Face to replace an existing face with a surface or work plane. The replace face command extends adjacent faces to trim the new surface and stitch the face to the solid.

### Replace a model face

Use the Replace Face command to replace one or more faces with a new surface. Replace performs three actions:

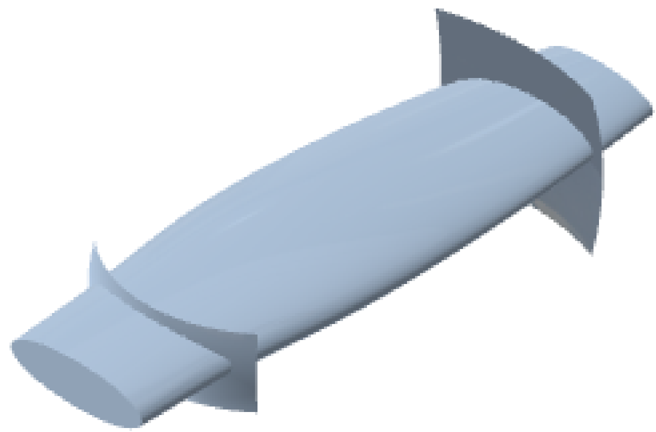
- Extends the adjacent faces to the new surface.
- Trims the new surface.
- Stitches the new surface to the source to create a solid.

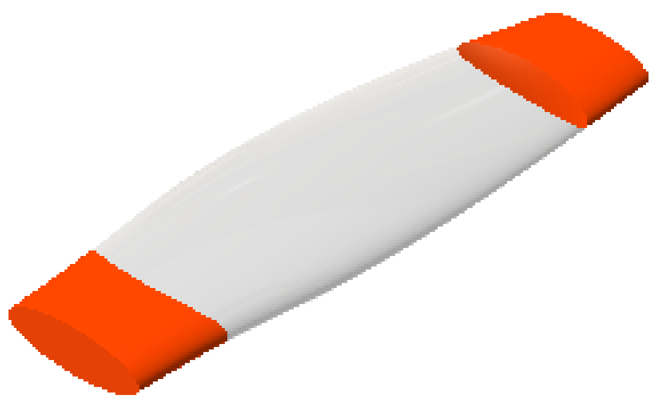
The target face must intersect the source faces, so it cannot be perpendicular.

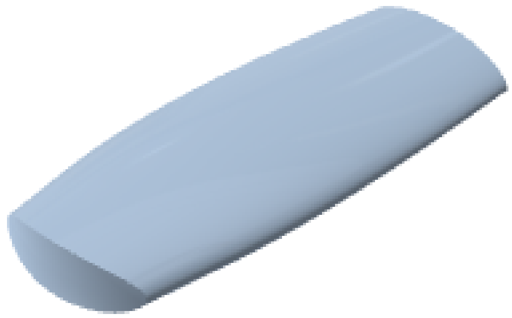
1. Expand the Solid panel and start Replace Face.
2. Select one or more Source faces that you want to replace.
3. Select the Target surface or work plane and click OK.
   An Extrude feature is created in the browser.

Show me how to create and use lofted surface

## Sculpt







Use the Sculpt command to select volumes defined by the intersection of solids, work planes, and surfaces.

Sculpt combines solids, work planes, and surfaces to create cells. Tools are the selected objects. Cells are the enclosed volumes where the objects intersect. You can use a cell to add or subtract material, or to create a component.

### Create a solid from multiple solids and surfaces

1. Expand the Solid panel, and click Sculpt.
2. Select the solids, surfaces, and work planes that enclose one or more volumes.

   On the command ribbon, click Hide Tools to turn off objects after they are selected.

   or

   Click Cells, and select the highlighted shapes.
3. Select the option for the output:
   - **Join** Adds the cells to the existing solid
   - **Cut** Removes the cells from the existing solid
   - **New component** Creates a component from the select cells
4. Click OK. The sculpt feature is added to the browser.

---

**NOTE** It is easier to see the enclosed volumes when the Hide Tools option is selected, especially in complex models.

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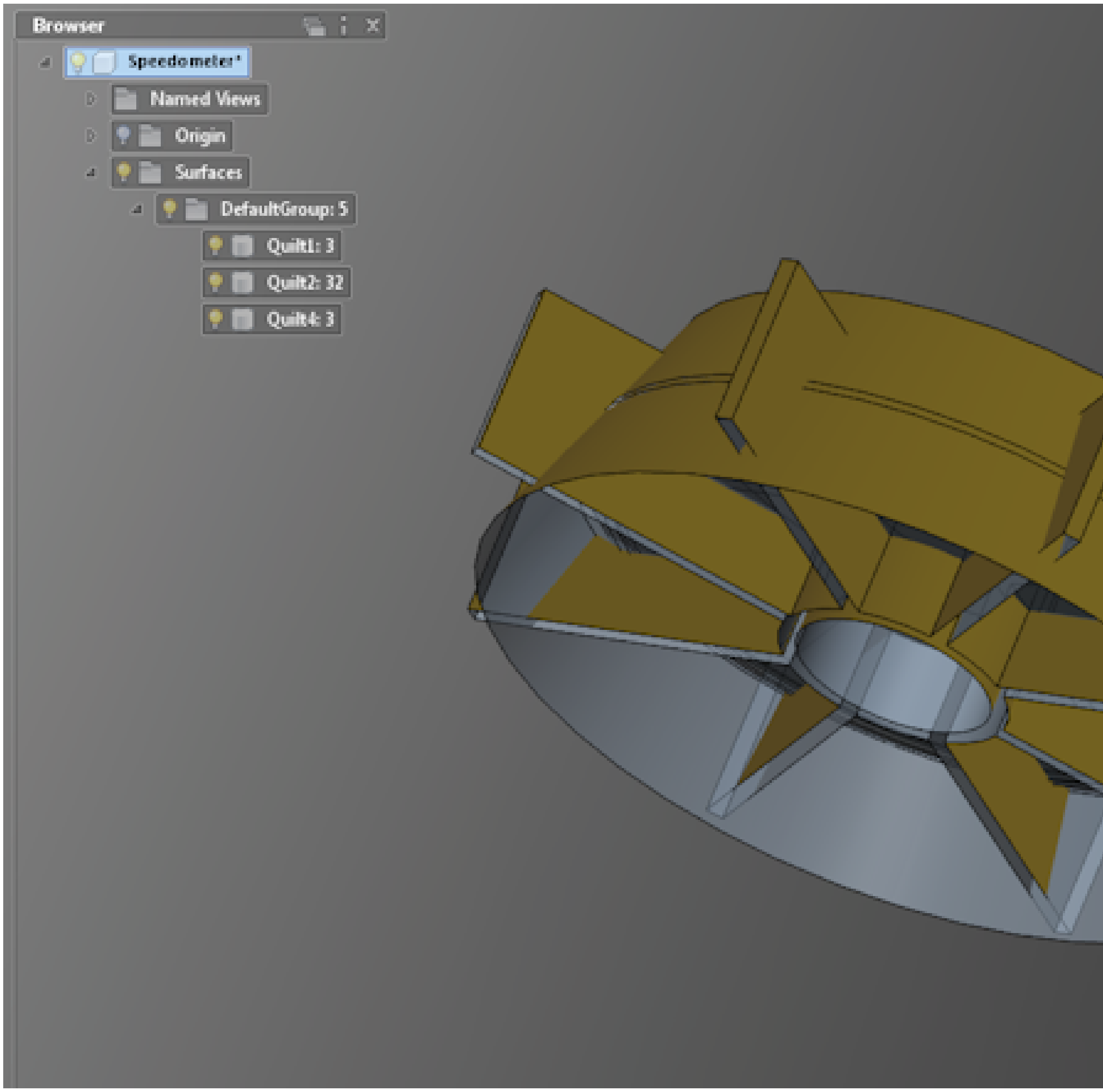
# Surface Modeling

A wide range of tools are available to create, edit, and arrange surfaces in an Inventor Fusion design. Surfaces can form an open or closed volume, but contain no mass.

The following image is an example of a Fusion design with surfaces.



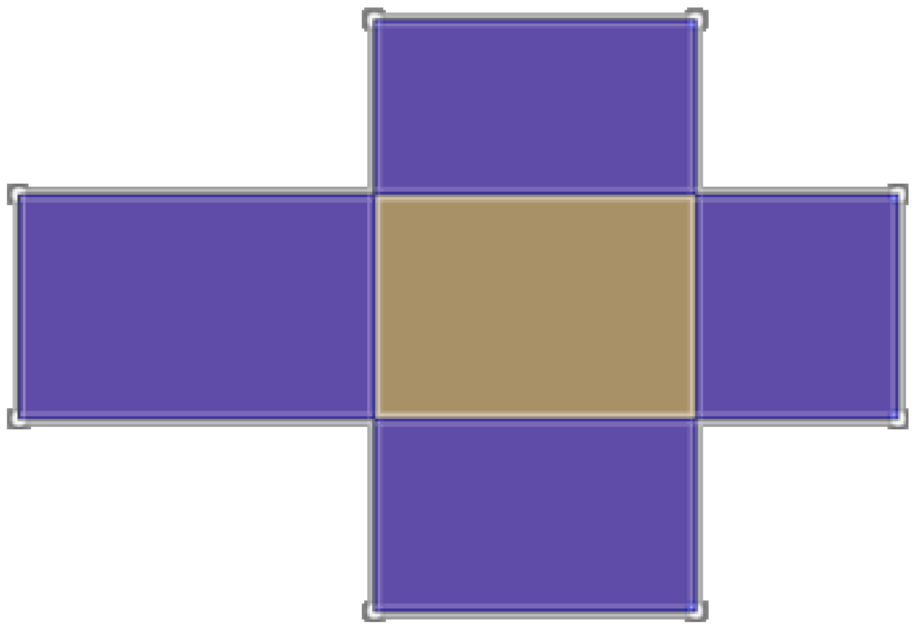

You can create mirror, rectangular pattern, circular pattern, and path pattern features using surfaces.

Thicken command adds thickness to selected surfaces to make a solid. It can add or remove material from a solid, changing its mass properties. Regular

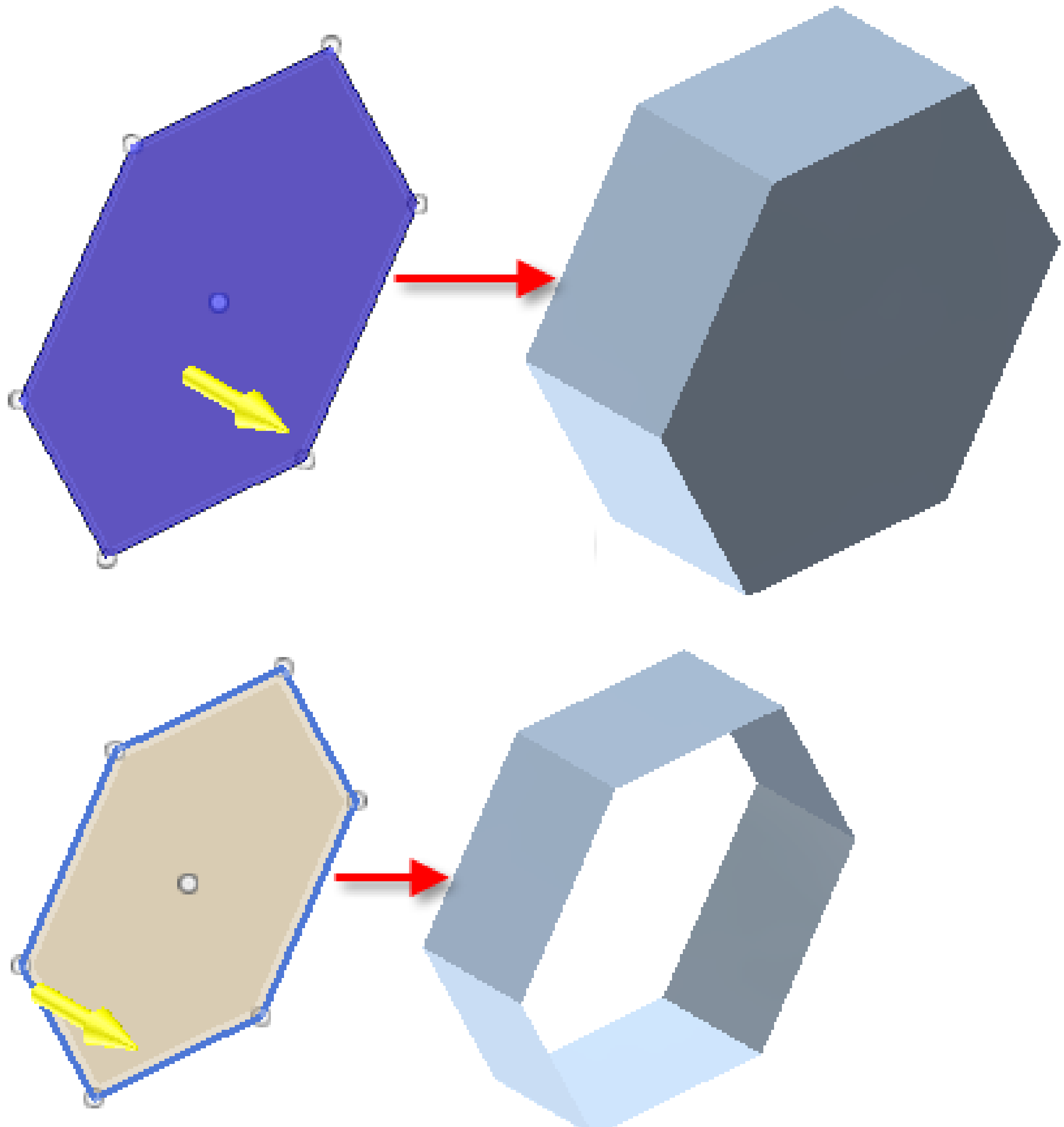
Boolean options (join, cut, intersect, new component) are available. By default, Inventor Fusion provides a precise solution. When a precise solution does not exist, an approximation is attempted

## Sketch Based

A Sketch-based feature defines the shape with sketch region. A region is the area created by a closed profile. In the image following, the overlapping rectangles form five regions.



## Extrude

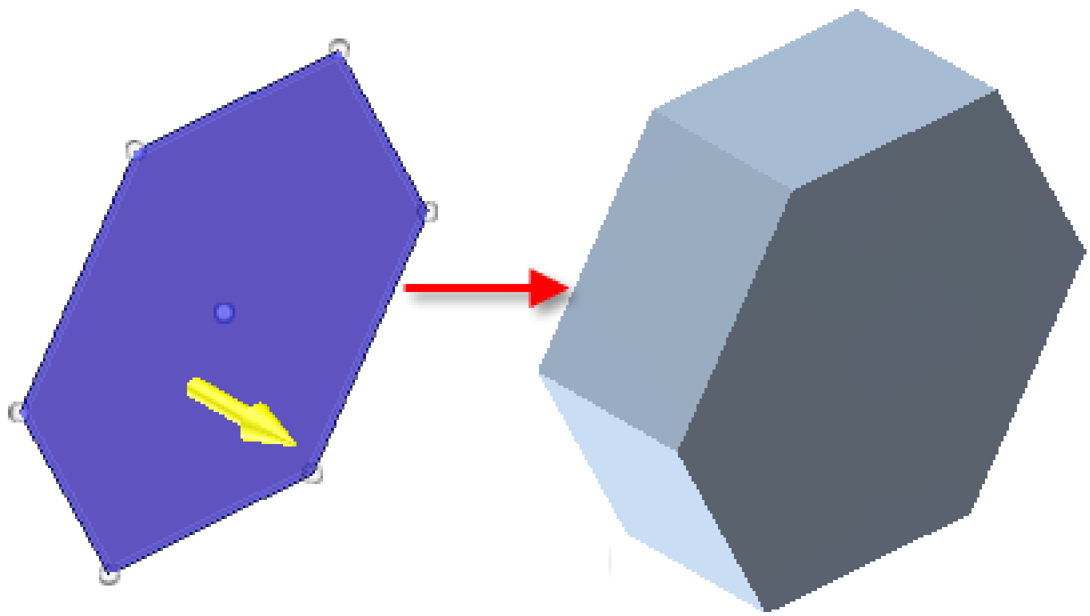


Extruded features are building blocks for creating and modifying models. You can create both solid and surface extrusions.

Extrude creates a feature by adding depth to sketch and model geometry. You specify the direction, depth, taper angle, and the type of feature for the extrusion.

An extruded solid requires a region. An extruded surface can use an open or closed profile.

### Create a solid extrusion



Use the Extrude command to create a solid feature by adding depth to a region.

1. Switch the ribbon to Solid, if necessary. In the Solid panel, click Extrude.
2. In the graphics area, select one or more sketch regions or model faces. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.
3. Use the manipulators to set the height and taper, or enter the values.

   **NOTE** If you selected a face for the extrusion, the feature automatically adds or removes material based on the direction. See step 5 for instructions on manually setting the Join or Cut Boolean option.

4 Select the Limit type

- **Distance** A numeric value for the extrusion length.
- **To** Determines the extrusion length when you select a face or work plane.
- **All**Determines the extrusion length by the furthest face in the model.

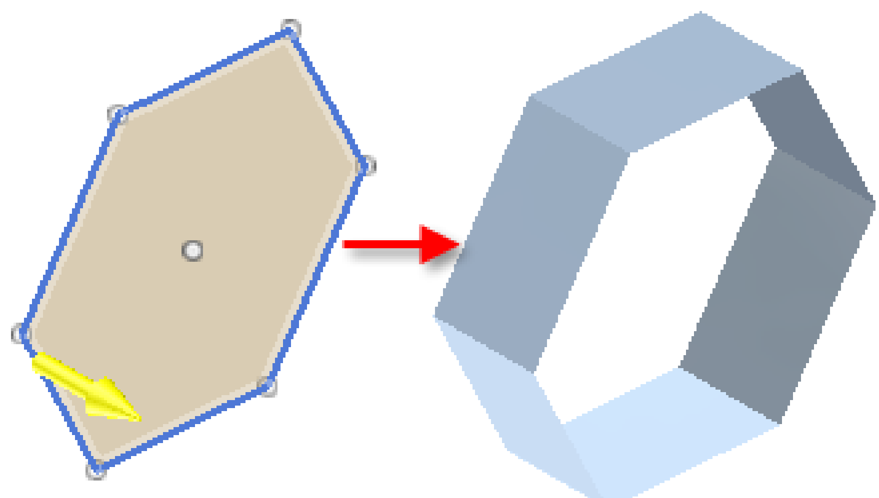
5 Select the Direction

- **One Direction** Creates the extrusion in one direction.
- **Two Directions** Creates the extrusion in both directions. Each direction can have a different extrusion length.
- **Symmetric** Creates the extrusion in both directions. Each direction has the same extrusion length.

6 Select the Boolean Option

- **Join** Adds material.
- **Cut** Removes material.
- **Intersect** Removes all material from the solid that does not overlap the new feature.
- **New Component** Creates a child component in the active component.

### Create a surface extrusion



Use the Extrude command to create a surface or component by adding depth to an open or closed profile.

1 Switch the ribbon to Surface, if necessary, in the Surface panel, click Extrude.
2 In the graphics area, select one or more sketch profiles, sketch entities, model faces, or model edges. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.
3 Use the manipulators to set the height and taper, or enter the values.
4 Select the Limit type:
   - **Distance**A numeric value for the extrusion length.
   - **To** Determines the extrusion length when you select a face or work plane.
   - **All** Determines the extrusion length by the furthest face in the model.
5 Select the Direction
   - **One Direction** Creates the extrusion in one direction.
   - **Two Directions** Creates the extrusion in both directions. Each direction can have a different extrusion length.
   - **Symmetric** Creates the extrusion in both directions. Each direction has the same extrusion length.
6 Select the New Surface or new Component for the Creation Option.

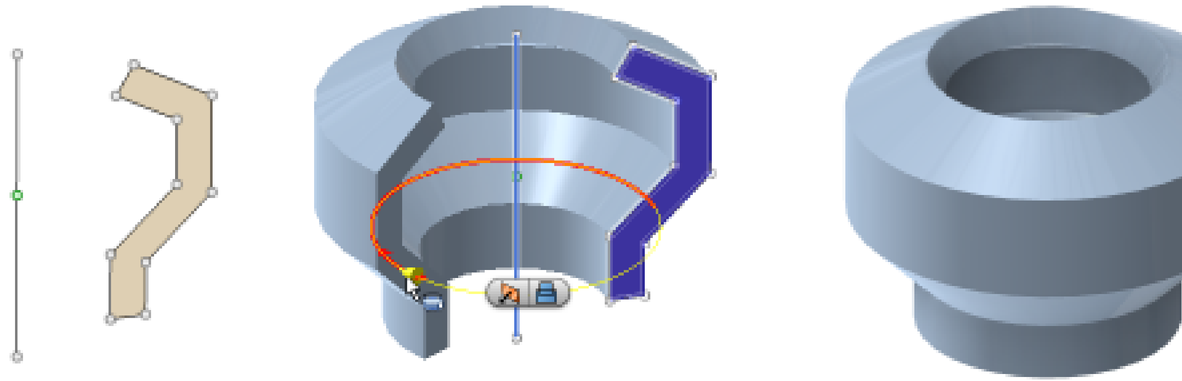
## Revolve

The Revolve command creates solid or surface features having radial symmetry such as stepped shafts and enclosures.

Revolve creates features by sweeping sketch and model geometry around an axis. The selection can be revolved through any angle measuring between zero and 360 degrees.

The axis can be a line in the profile, or a separate entity. An offset axis results in an axial hole in the feature. The axis cannot cross the region.

### Create a revolved solid



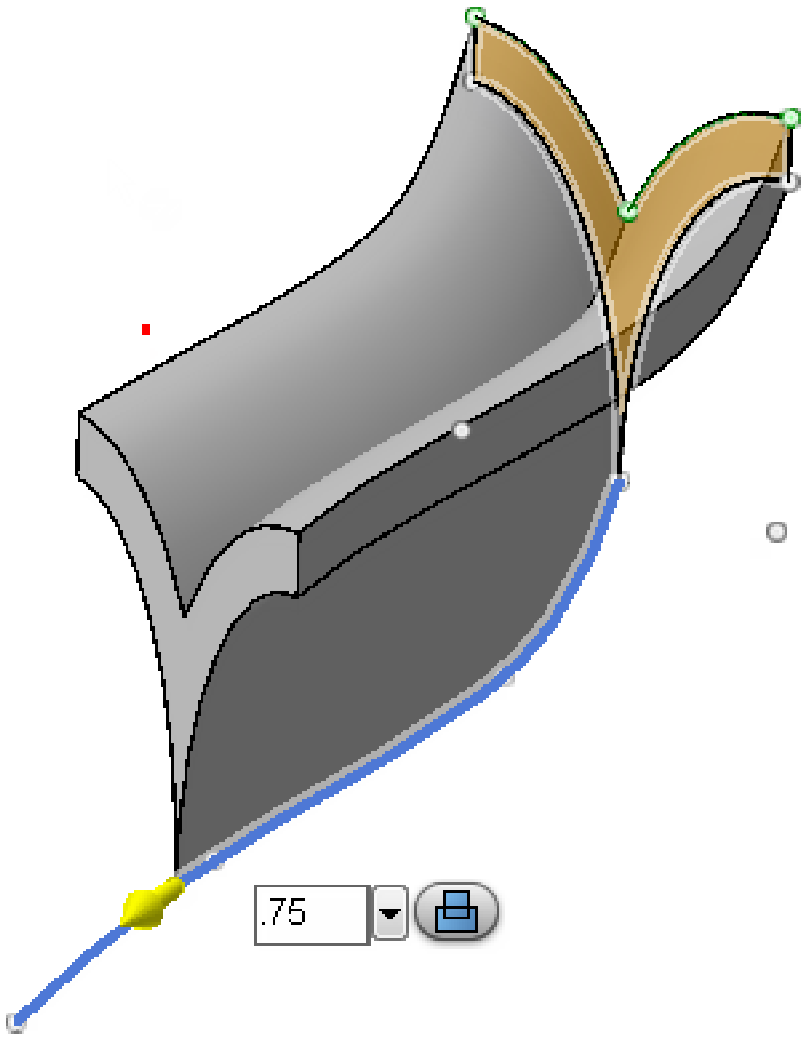
1. Click the drop-down arrow under Extrude, and then click Revolve.
2. Select one or more sketch profiles or model faces. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.
3. Select the axis for the revolution. If the selected axis is not on the sketch plane, the axis is temporarily projected to the plane.
4. Select the Limit type. Use the manipulator to set the rotation angle, or enter the value.
   - **Distance** A numeric value for the angle.
   - **To** Determines the angle when select a face or work plane.
   - **Full** Determines the angle by the furthest face in the model.
5. Select the Direction
   - **One Direction**Creates the revolve feature in one direction.
   - **Two Directions** Creates the revolve feature in both directions. Each direction can have a different angle.
   - **Symmetric** Creates the revolve feature in both directions. Each direction has the same angle.
6. Select the Boolean Option
   - **Join** Adds material.
   - **Cut** Removes material.
   - **Intersect** Removes all material from the solid that does not overlap the new feature.

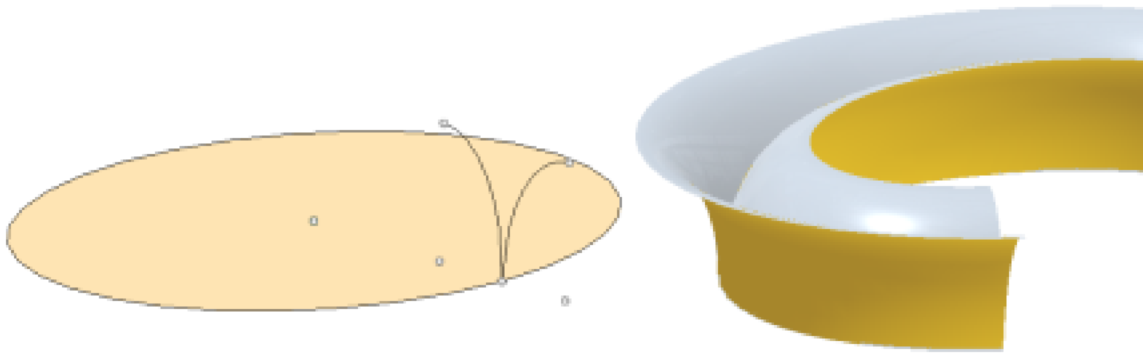
- **New Component** Creates a child component in the active component.

### Create a revolved surface

1 Click the drop-down arrow under Extrude, and then click Revolve.
2 Select one or more sketch profiles, sketch entities, model faces, or model edges. Multiple selections must be coplanar and in the same component. Sketch profiles and entities must be in the same sketch.
3 Select the axis for the revolution. If the selected axis is not on the sketch plane, the axis is temporarily projected to the plane.
4 Select the Limit type. Use the manipulator to set the rotation angle, or enter the value.
   - **Distance** A numeric value for the angle.
   - **To** Determines the angle when select a face or work plane.
   - **Full** Determines the angle by the furthest face in the model.
5 Select the Direction
   - **One Direction**Creates the revolve feature in one direction.
   - **Two Directions** Creates the revolve feature in both directions. Each direction can have a different angle.
   - **Symmetric** Creates the revolve feature in both directions. Each direction has the same angle.
6 Set the Creation Option to New Surface or New Component.

## Sweep



Sweep creates a feature by moving sketch or model geometry along a path. You can create both solid and surface sweep features.

**NOTE** A swept profile cannot intersect itself. Any curves in the path must have a radius greater than the width of the profile.

Paths for sweep features can be:

- A sketch or a model edge.
- Straight or curved.

### Create a swept solid

1. On the ribbon Solid tab, Extrude drop-down list, click Sweep.
2. Select one or more sketch profiles or model edges. Multiple selections must be coplanar and in the same component. Sketch profiles must be in the same sketch.
3. Select one or more sketch entities or model edges for the path. You can only select tangentially connected entities for the path.

   **NOTE** You can select a path from a different component than the profile.

4. Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path on the command ribbon.
5. Select the Orientation type:
   - **Perpendicular** keeps the region perpendicular to the path.
   - **Parallel** keeps the region parallel to the region sketch.
6. Select the Boolean Option:
   - **Join** Adds material.

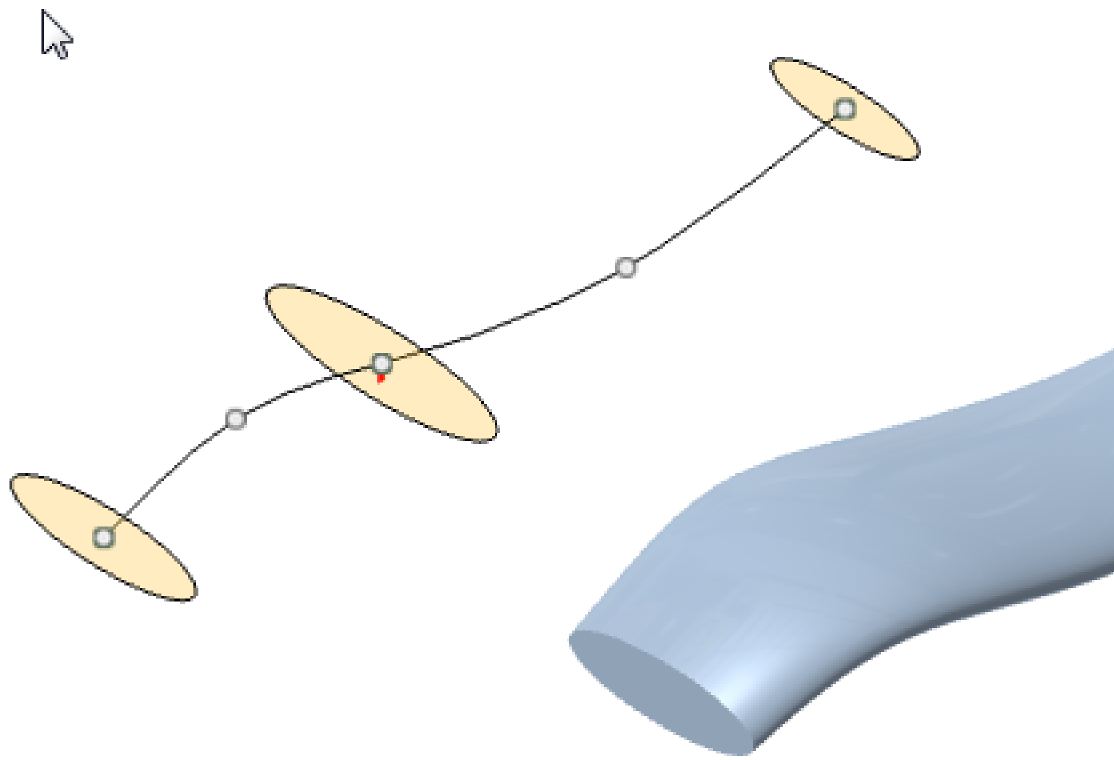
- **Cut** Removes material.
- **Intersect** Removes all material from the solid that does not overlap the new feature.
- **New Component** Creates a child component under the active component.

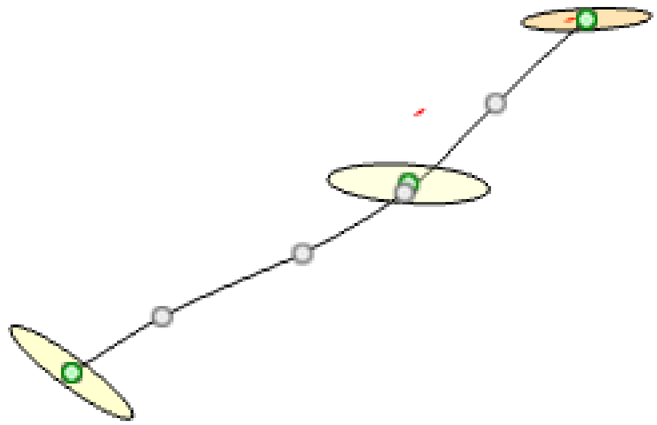
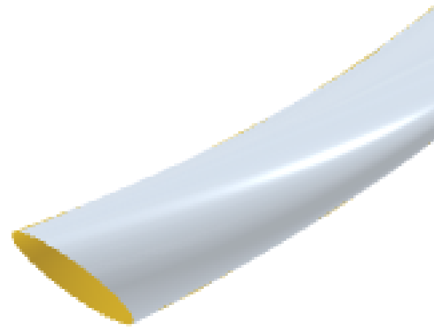
### Create a swept surface

1. On the ribbon Surface tab, Extrude drop-down list, click Sweep.
2. Select one or more sketch profiles, sketch entities, model faces, or model edges. Multiple selections must be coplanar and in the same component. You can only select tangentially connected entities for the curve.
3. Select one or more sketch entities or model edges for the path. Multiple entities must be tangentially connected.
4. Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path in the command ribbon.
5. Select the Orientation type:
   - **Perpendicular** keeps the region perpendicular to the path.
   - **Parallel** keeps the region parallel to the region sketch.
6. Set the Creation option to New Surface or New Component.

**Show me how to create swept features**

## Loft



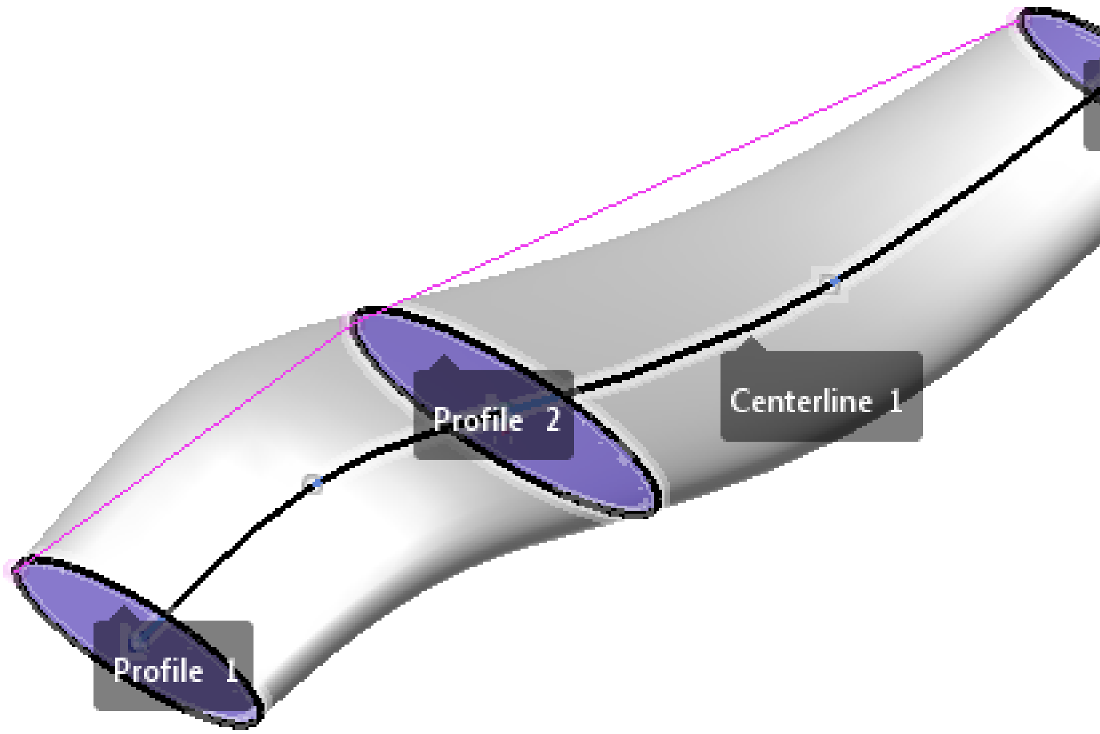



Loft creates complex, organic shapes. These shapes are common in the automotive, marine, and consumer products industries.

Loft creates a smooth shape that transitions between two or more sketch profiles or faces. You can select sketch profiles and model faces for a solid. You can select open or closed profiles for a surface. A surface loft cannot have both open and closed profiles.

You can also add a centerline to define a path for the loft, or you can change the curvature at each profile.

### Create a lofted solid




1 Switch the ribbon to Solid, if necessary, and in the Extrude drop-down list, click Loft.

2 In the graphics area, select the sketch regions or model faces. Select multiple regions in their order. You cannot select the first and last region, and then a middle region.

3 Optionally, select the centerline path that passes through each profile. The path can have multiple, tangent entities. Select each path entity.

   **NOTE** You can select the centerline path entities at any time.

4 In the command ribbon, click Closed to return to the first region in the loft. To close the loft, it is required that it extends more than 180 degrees.

5 Select the Boolean Option:

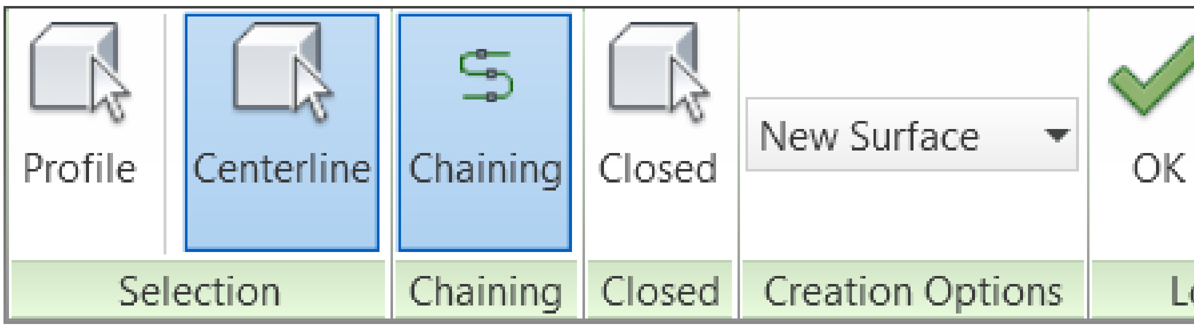
   a **Join** Adds material.

b **Cut** Removes material.

c **Intersect** Removes all material from the solid that does not overlap the new feature.

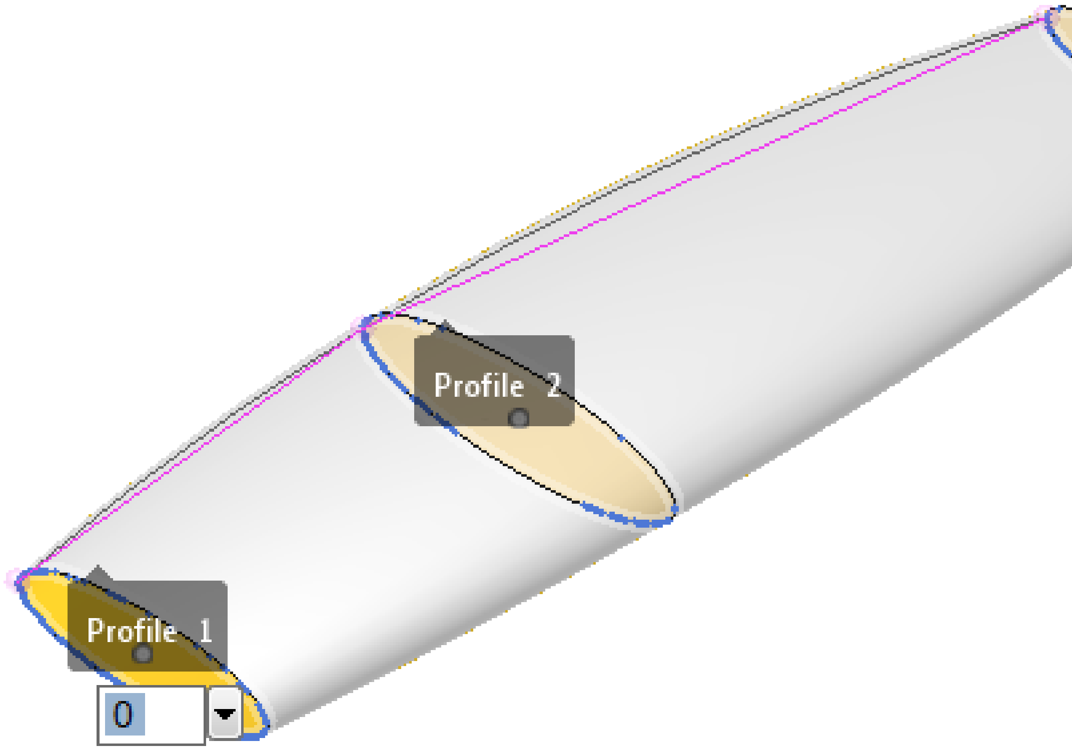
d **New Component** Creates a child component in the active component.

### Create a lofted surface

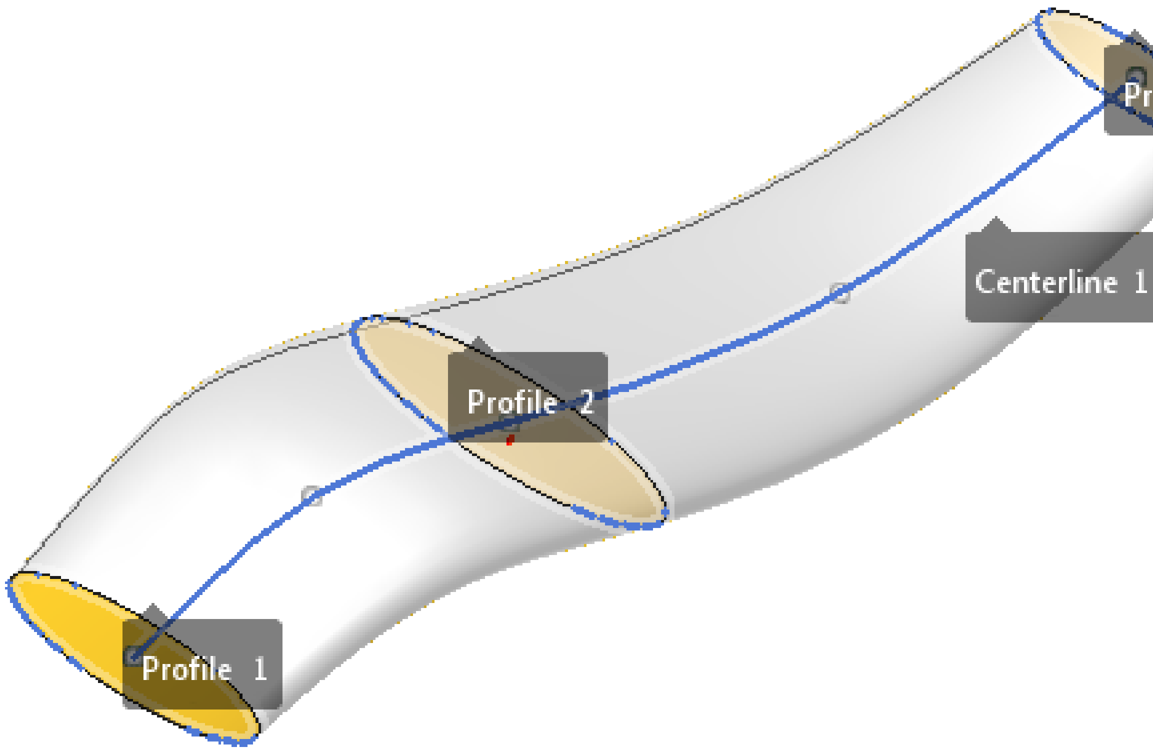



Use profiles that are open or closed. You can use a centerline path to control the curvature, or you can change the curvature at a profile. It is not required that the centerline path connects the profile centers, but it is required that it intersects the profiles.

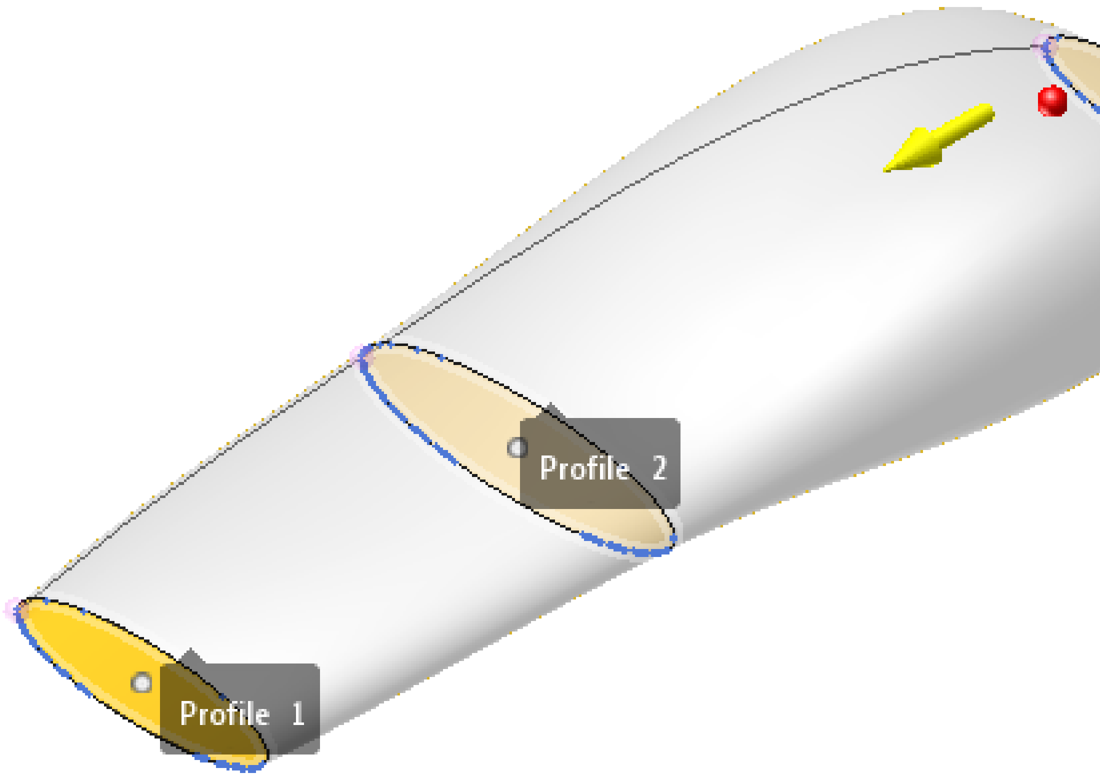
You can create the loft with natural curvature, with a centerline path, or with transition curvature changes at each profile.




This loft uses the natural curvature.




This loft uses a centerline path.



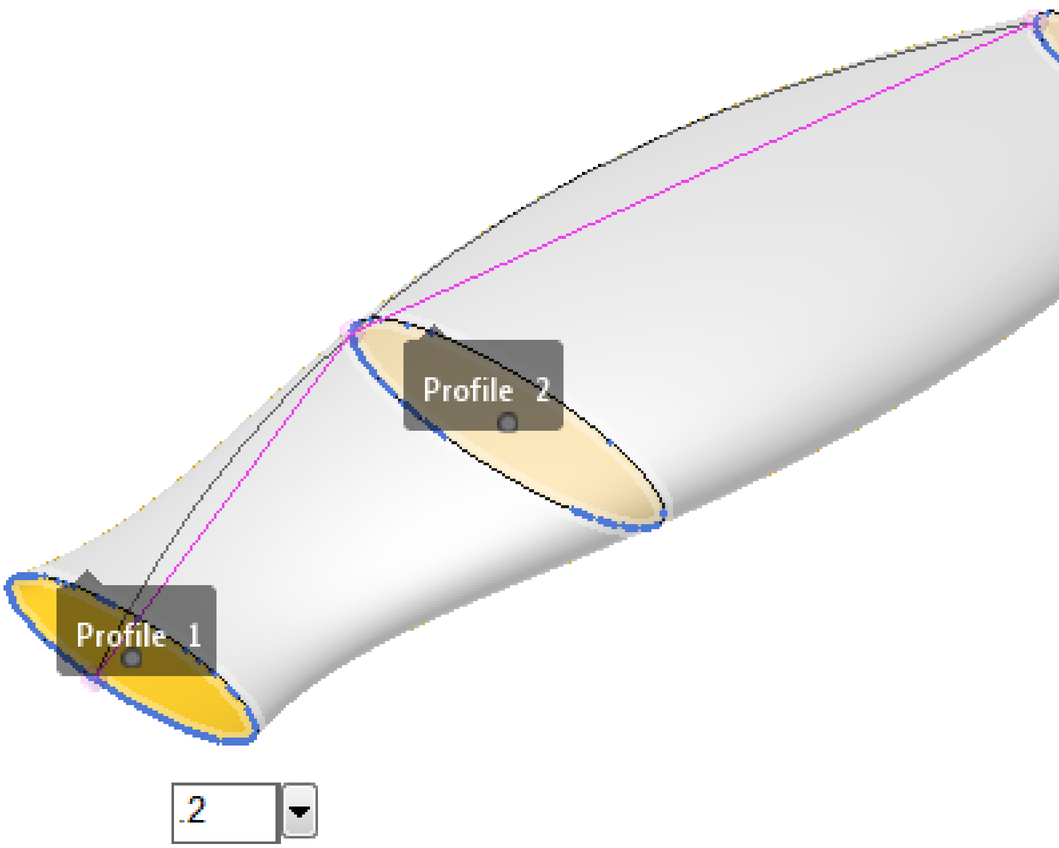

This loft uses directional control at the last profile.

1 Switch the ribbon to Surface, if necessary. In the Extrude drop-down list, select Loft.

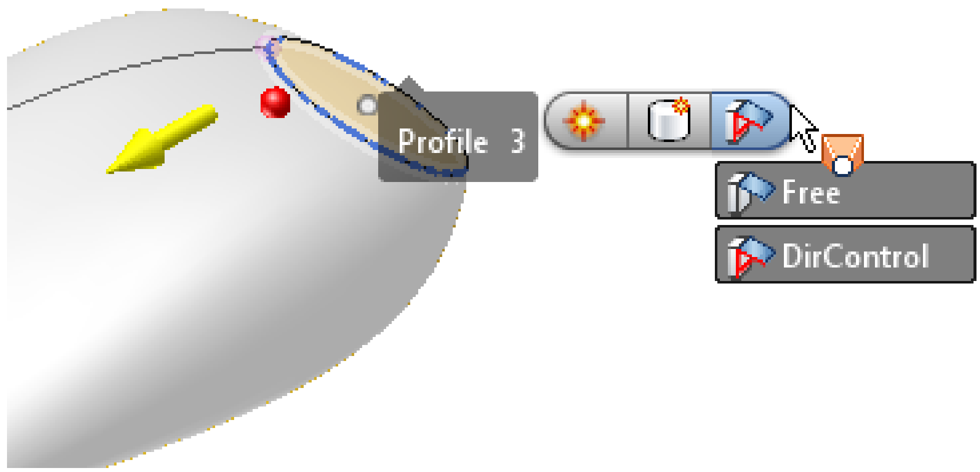
2 In the graphics area, select the sketch profiles and faces. You cannot select both open and closed profiles. Select the regions in order. You cannot select the first and last region, and then a middle region.

3 Optionally, change the transition curvature for the entire loft, set the transition curvature as you select each region, or select the centerline path.

   1 To change the transition curvature for the entire loft, in the edit box, enter a value between 0.0 and 1.0. In this image, the transition curvature is set to 0.2




   2 To change the transition curvature at a profile, on the command glyph, select DirControl (Directional Control). Use the manipulators to change the curvature.




**NOTE** If you select DirControl for a profile, the loft transition curvature is reset to 0.

3 Click the Centerline button, and select the path, which can have multiple, tangent entities. Chaining for the path is turned on by default. Turn chaining off to select the path entities manually.

4 In the command ribbon, click Closed to return to the first region in the loft. To close the loft, it is required that it extends more than 180 degrees.

5 Select the Creation Option:

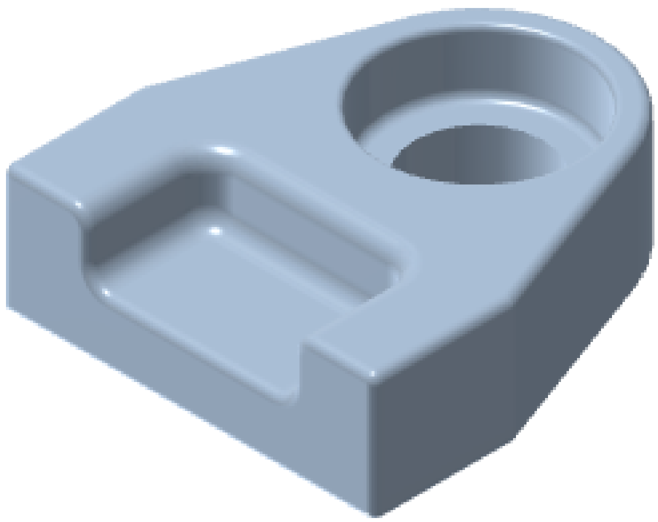
1 **New Surface** Creates the surface in the active component.

2 **New Component** Creates a child component for the surface under the active component.

**Show me how to create a surface loft**

## Placed Features

### Fillet



Fillets round over exterior and interior edges. A concave fillet feature is called a fillet and a convex fillet feature is called a round.

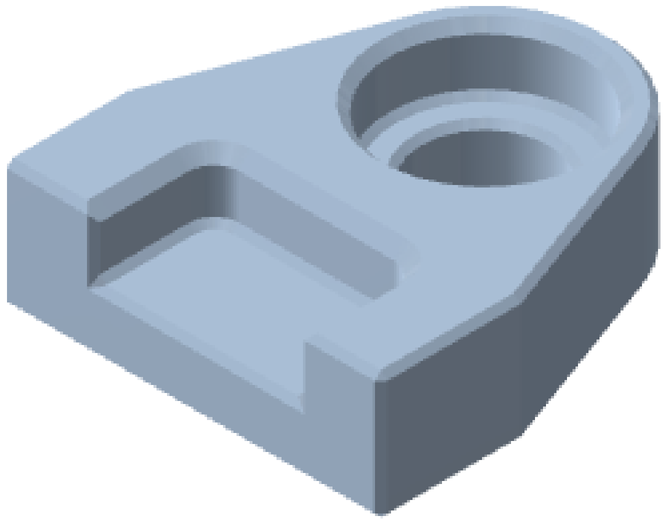
You can create a fillet on a solid or on a quilted surface.

You can edit a fillet by right-clicking on it in the browser and selecting Edit Fillet, or by using Press/Pull.

**Create a fillet**

1. Start Fillet.
2. Select an edge- the manipulator displays.
3. Deselect the Tangent Propagation option if you do not want tangentially connected edges.
4. Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
5. Deselect the Rolling Ball Corners option if you want a smooth transition between edges. Rolling ball fillets produce corners like a milling machine or a router.

## Chamfer



Chamfers create a bevel and removes material from an outside edge and adds material to an inside edge. Chamfers can be equal distance from the edge, a specified distance and angle from an edge, or a different distance from the edge for each face.

You can create a chamfer on a solid or on a quilted surface.

**Create a chamfer**

1. Start Chamfer.
2. Select an edge- the manipulator displays.
3. Deselect the Tangent Propagation option if you do not want tangentially connected edges.
4. Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
5. Select another Chamfer Option to change how the chamfer is defined.

# Modify Surfaces

Surface modification commands make changes to existing surfaces and quilts. Boundary Patch and Offset create surfaces based on existing geometry.

## Extend

Extend a face or surface by selecting one or more edges and moving them away from the face. You cannot extend a face on a solid.

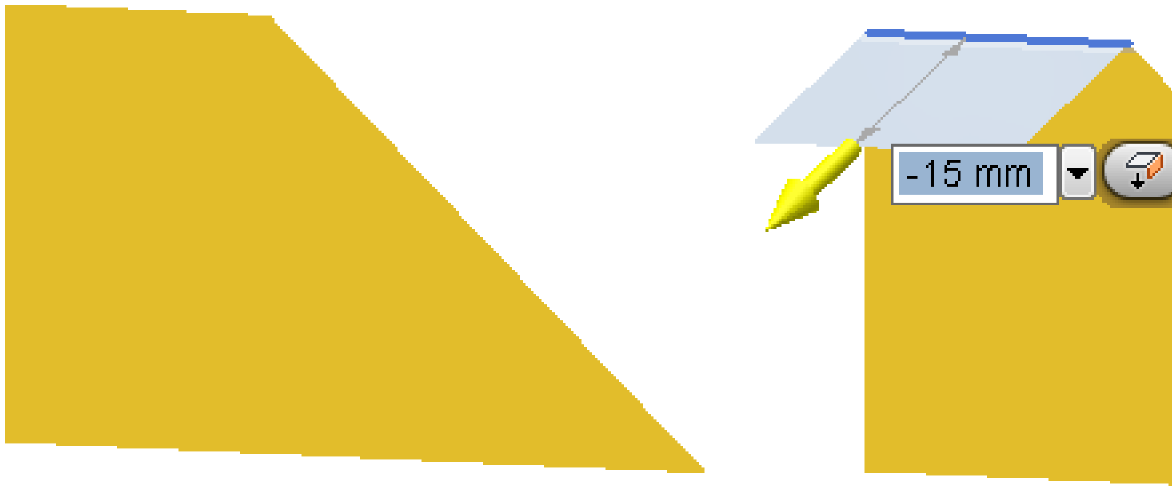
When you move the edge, you can extend the surface, or create a new, perpendicular surface.

You can extend planar and non-planer surfaces. There can be limitations when extending non-planer surfaces due to the curvature.

**Extend a surface**

Extension Type: Natural
Extension Direction: Aligned

Extension Type: Natural
Extension Direction: Non-Aligned




Extension Type: Perpendicular

1 On the ribbon, click the arrow under Solid and select Surface, and then. On the Surfaces panel, select Extend.

2 Select a surface edge.
   - You can select multiple edges. They can be unconnected, or coplanar.
   - The Tangent Propagation option automatically selects tangentially connected edges.
   - You cannot select Solid edges.

3 Select the Extension Type
   - Natural extends the current faces
   - Perpendicular creates new faces perpendicular to the current faces.

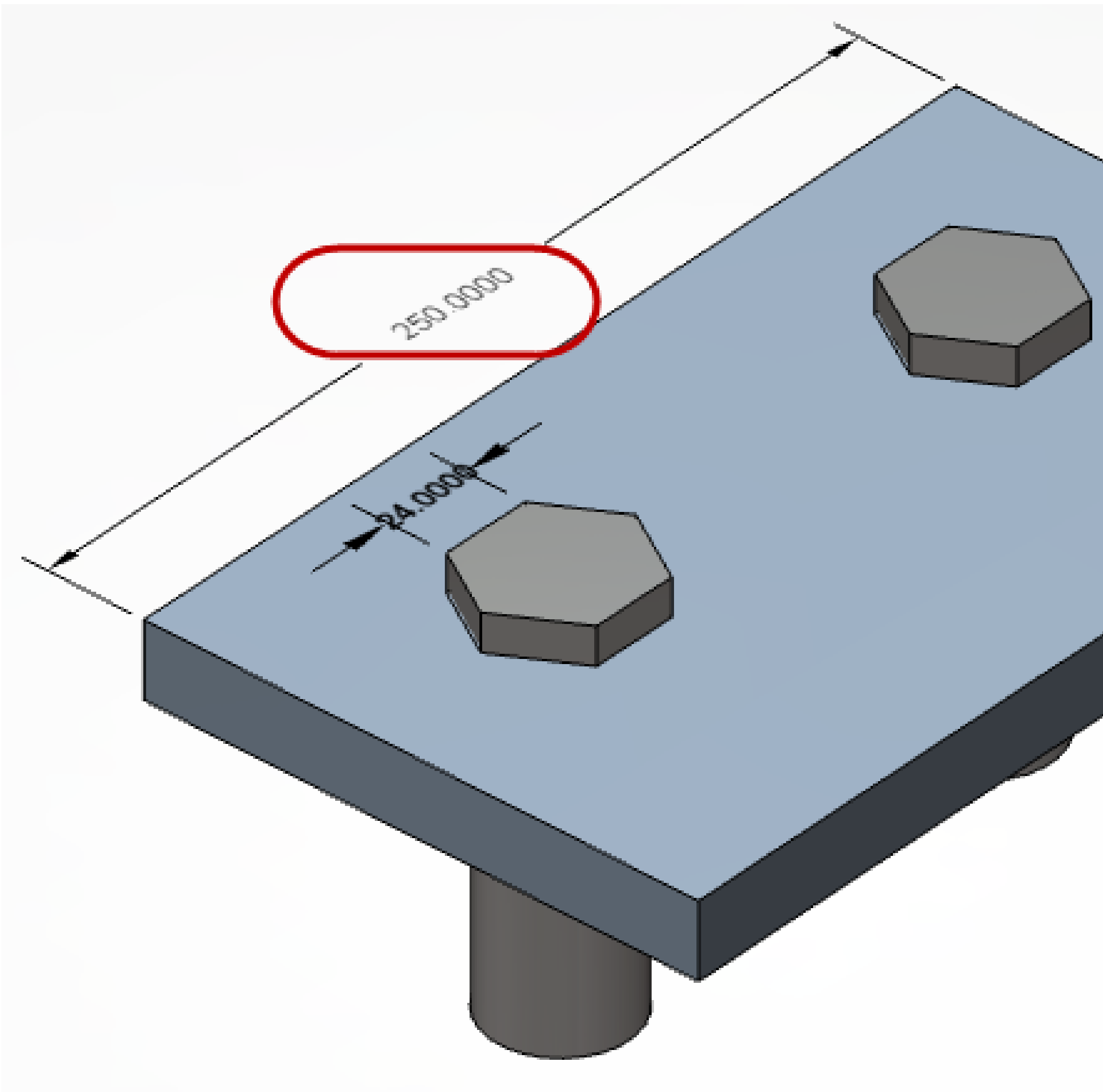
4 For the Natural extension type, select the Extension Direction.
   - Aligned extends the edges of the current faces.
   - Non-aligned creates new side edges perpendicular to the selected edge.

5 Drag the arrow manipulator, or enter a value in the HUD.

6 Click OK to create the extended face.

## Scale Command

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling. Valid objects are components, solids, surfaces, and sketches.

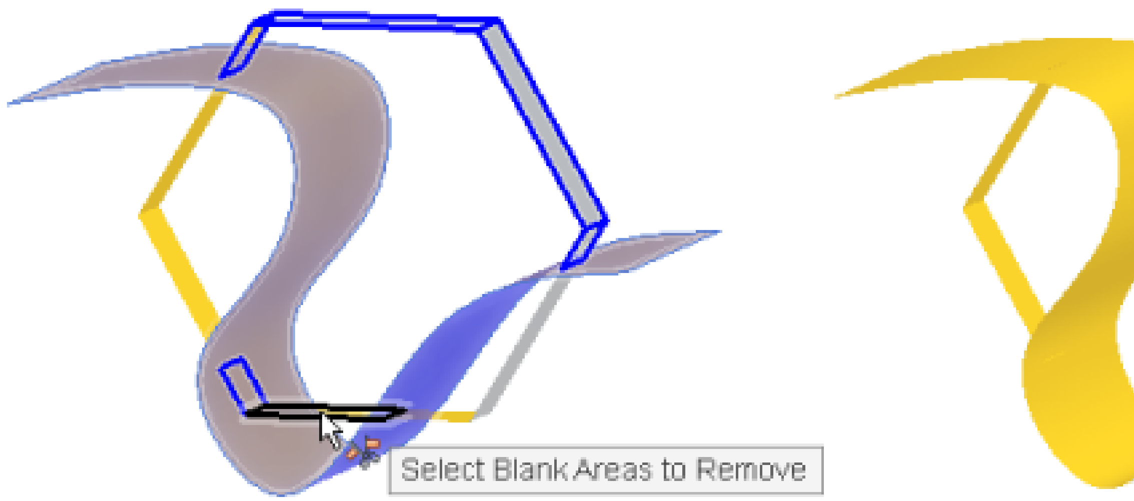



To scale an object:

1. Start the scale command.
2. Select the objects that you wish to scale.

3 Select a reference point to scale around.
4 Enter a scale factor or click and drag.
5 Click OK to complete the command.

## Trim Surface




Trim splits intersecting surfaces, and removes one or sections of a surface or quilt. You can use a surface, quilt, solid face, or a work plane as a split tool. You can use a connected series of sketch entities as a split tool, if they do not intersect the target surface.

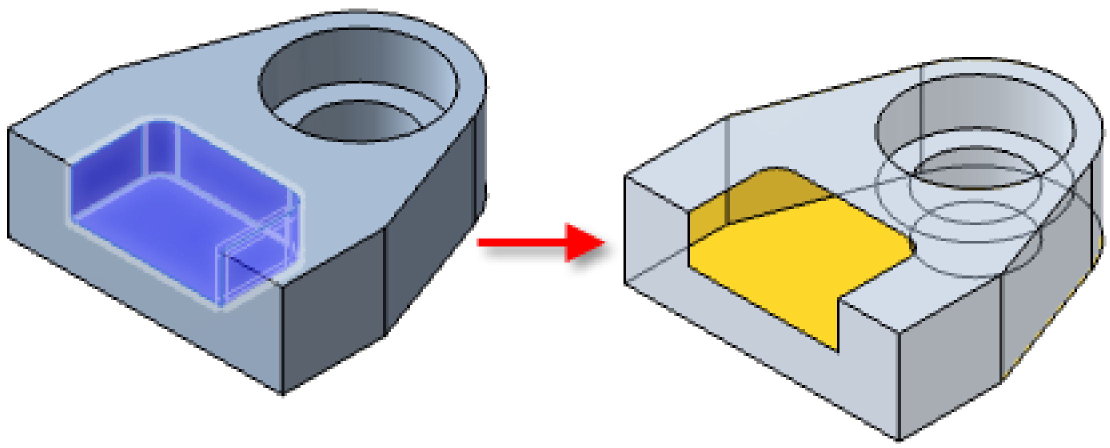
### Trim a surface

1 On the Surfaces panel, click Trim.
2 Select a Cutting tool. Valid feature selections include a surface, a quilt, a solid face, or a work plane. You can also use a connected series of sketch entities that do not intersect the target surface.
3 The intersecting surfaces highlight. Select the sections to remove.

   **NOTE** To use a selection window, hold down **Shift** as you drag the cursor.

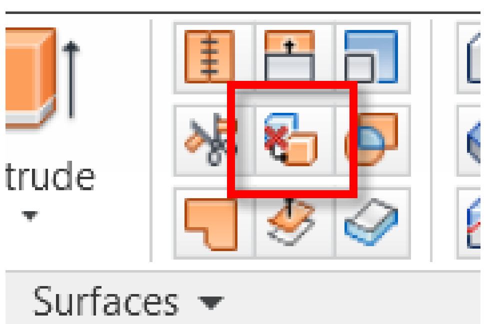
4 Click OK to delete the selected surfaces.

## Delete Face



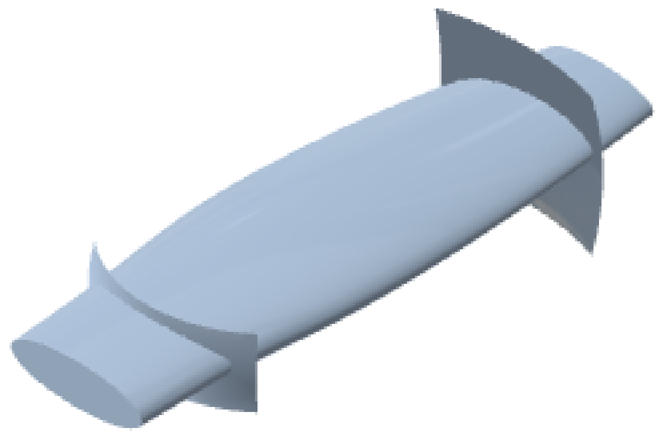
The Delete Face command removes one or more faces from a model. If the model is a solid, the remaining faces become a quilted surface.

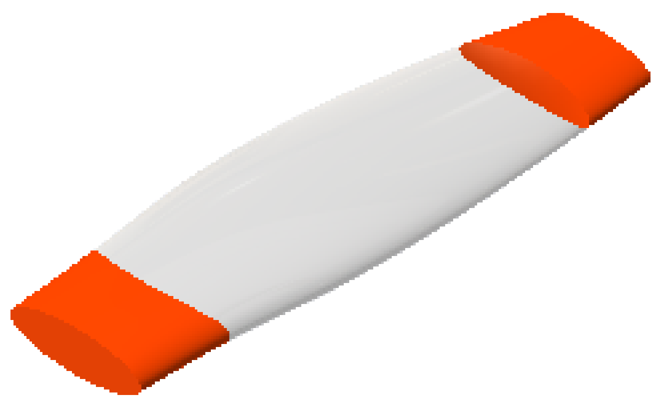
### Delete a face

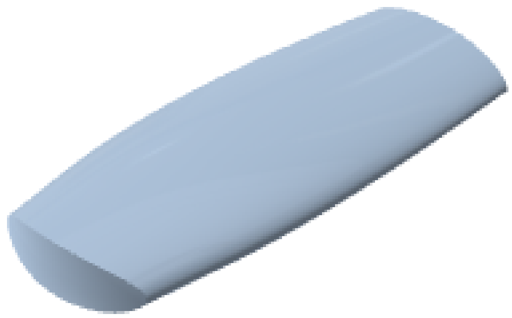



1 Switch the ribbon to Surface, if necessary.
2 Select one or more faces in a model. You can select faces on solids, surface quilts, and individual surfaces. You can select faces in separate components.
3 Click Delete Face on the Surfaces panel. If a face is on a solid model, the solid or solid feature is converted to a quilted surface.

## Sculpt







Use the Sculpt command to select volumes defined by the intersection of solids, work planes, and surfaces.

Sculpt combines solids, work planes, and surfaces to create cells. Tools are the selected objects. Cells are the enclosed volumes where the objects intersect. You can use a cell to add or subtract material, or to create a component.

### Create a solid from multiple solids and surfaces

1. Expand the Solid panel, and click Sculpt.
2. Select the solids, surfaces, and work planes that enclose one or more volumes.

   On the command ribbon, click Hide Tools to turn off objects after they are selected.

   or

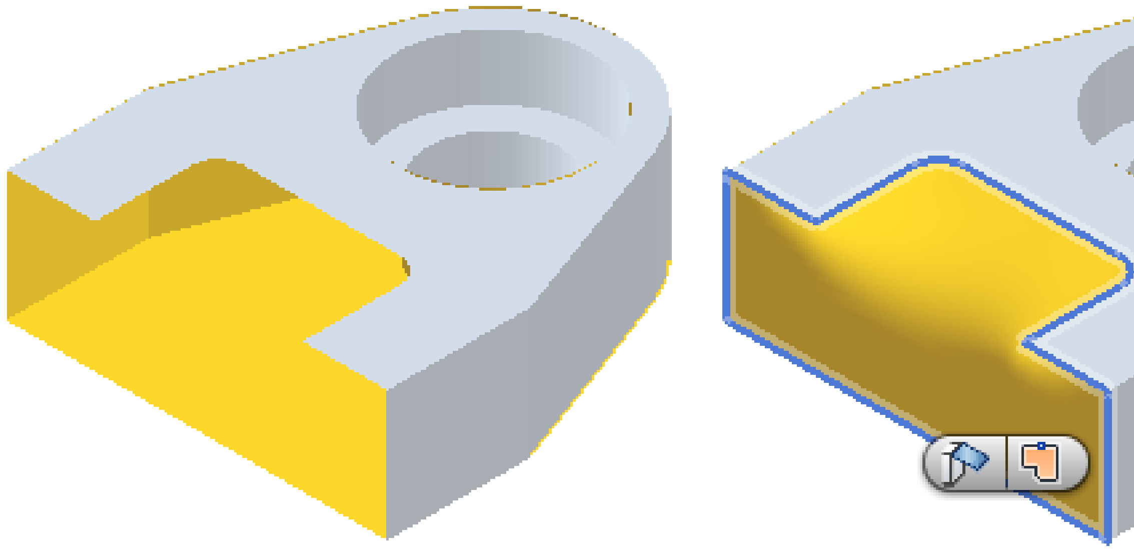
   Click Cells, and select the highlighted shapes.
3. Select the option for the output:
   - **Join** Adds the cells to the existing solid
   - **Cut** Removes the cells from the existing solid
   - **New component** Creates a component from the select cells
4. Click OK. The sculpt feature is added to the browser.

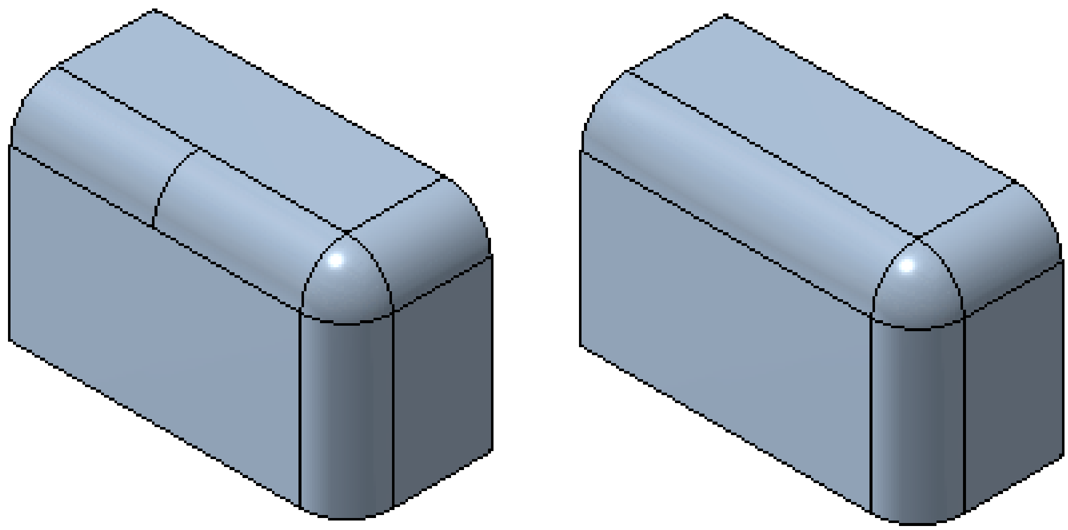
---

**NOTE** It is easier to see the enclosed volumes when the Hide Tools option is selected, especially in complex models.

---

## Patch/Merge





Patch/Merge creates a surface, also called a boundary patch, that replaces a missing face, or merges multiple faces into one surface.

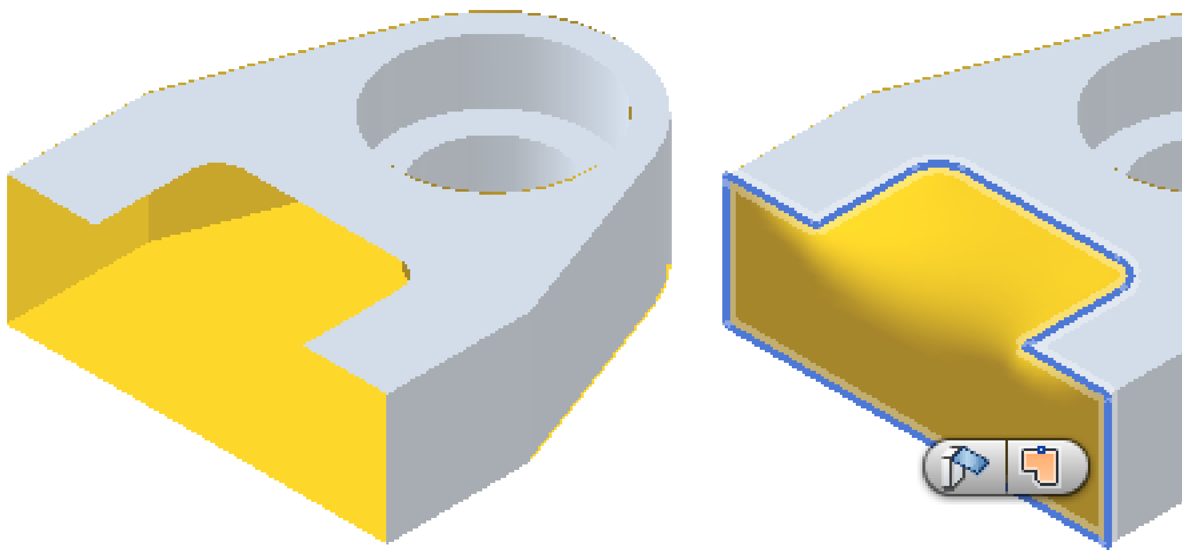
Use Patch to replace a missing face on a quilt, or create a face on an opening in a solid. You can create boundary patches to form a continuous quilt or create a solid. The missing face must have a connected series of edges that forms the boundary of the patch.

If the edges are not coplanar, the patch surface adds curvature to create a blend between the edges. You can control the curvature at each edge to produce a smoother surface.

Use Merge to replace multiple faces or surfaces with one surface. Merge is typically used to repair imported models that have extra faces, or to simplify a model for analysis.

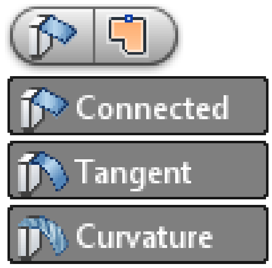
### Create a surface

The edges must form a closed profile. If the edges are not coplanar, surface curvature is added to blend between the edges. You can control the curvature at each edge to create a smoother surface.



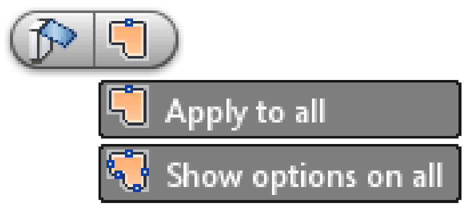
1. On the Surfaces panel, click Patch/Merge.
2. In the graphics area, select the edge of a missing face or an opening on a solid. All of the connected edges are automatically selected to create a closed profile.
3. On the command ribbon, in Creation Options, select New Surface or New Component.

4 On the command glyph in the graphics area, select the transition between the patch and the edges.




- **Connected** Creates a surface with G0 edges (the new surface edges are connected at an angle).
- **Tangent** Creates a surface with G1 edges (the new surface edges are tangential).
- **Curvature** Creates a surface with G2 edges (the new surface edges are blended with continuous curvature).

5 On the command glyph in the graphics area, select Apply to All or Show Options on All.

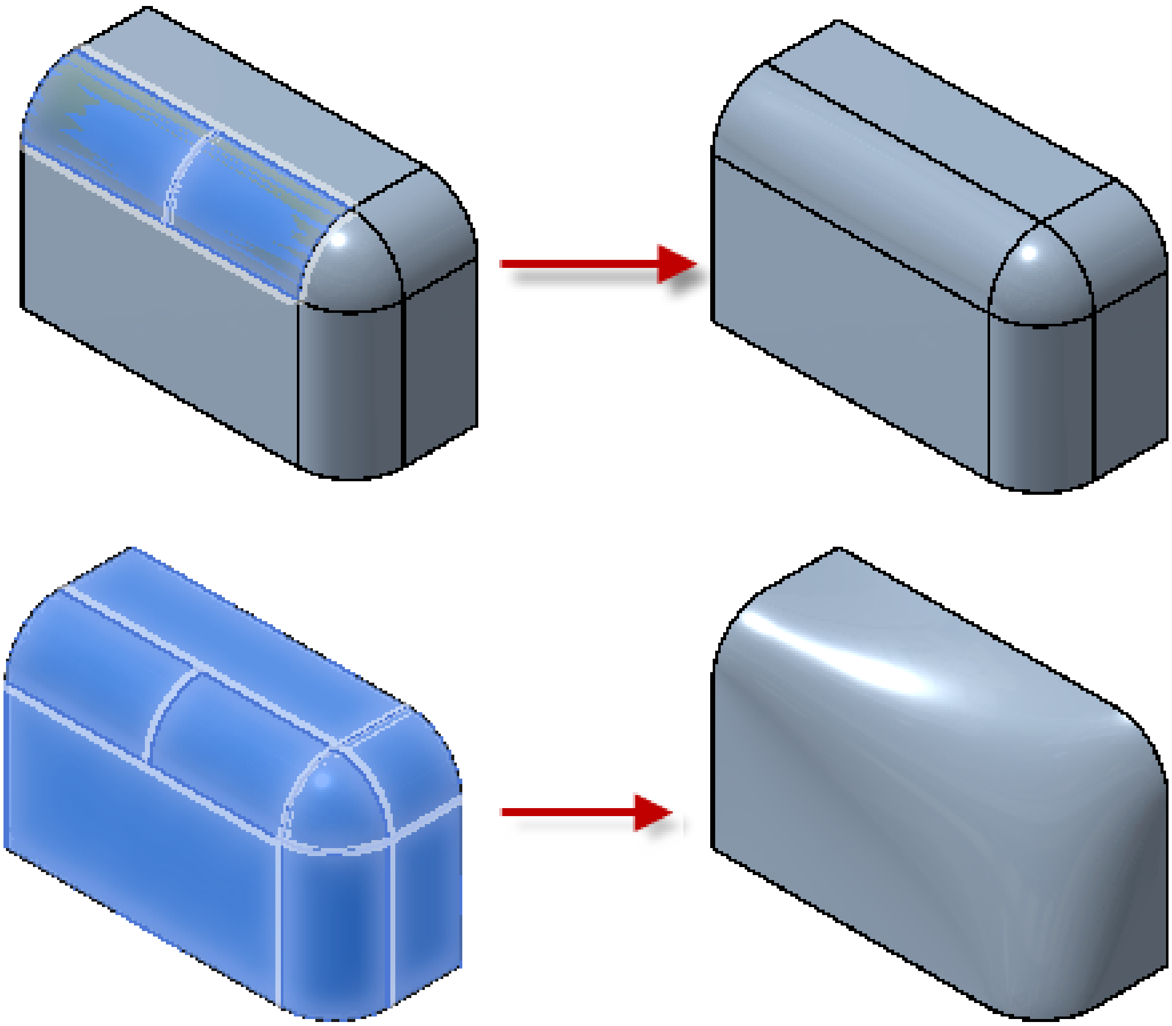



- **Apply to All** Applies the same setting for all edges.
- **Show Options on All** Displays the control glyph on each edge. Set the curvature on each edge individually to change the shape of the surface.

  On the glyph, **Hide Options on Others** turns off the individual edge controls.

6 Click OK to create the surface.

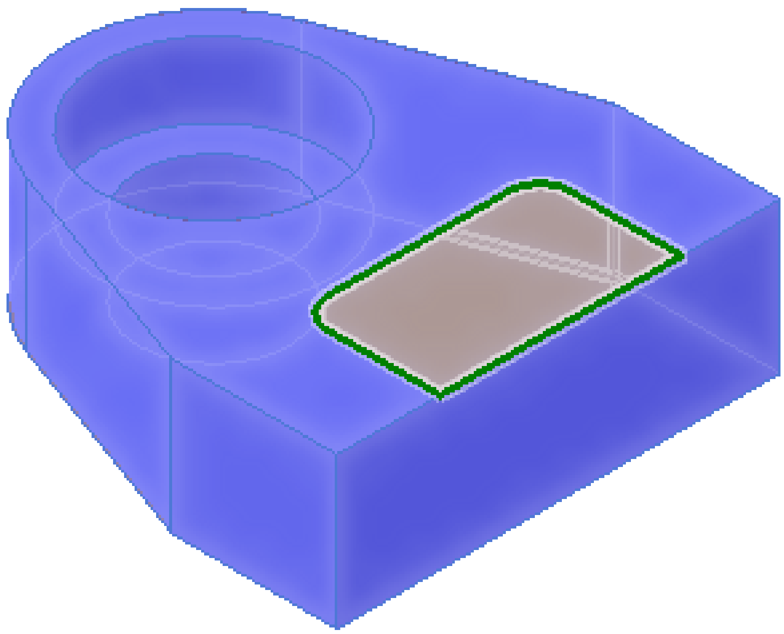
**Merge multiple faces**



1 Start Patch/Merge from the Surfaces panel.
2 Select two or more faces.
  On the command ribbon, clear the selection for Chaining to prevent automatic selection of all tangent faces.
3 Click OK to create the surface.

**NOTE** The cylindrical faces are replaced with a blended surface that approximates a cylinder.

## Stitch/Unstitch

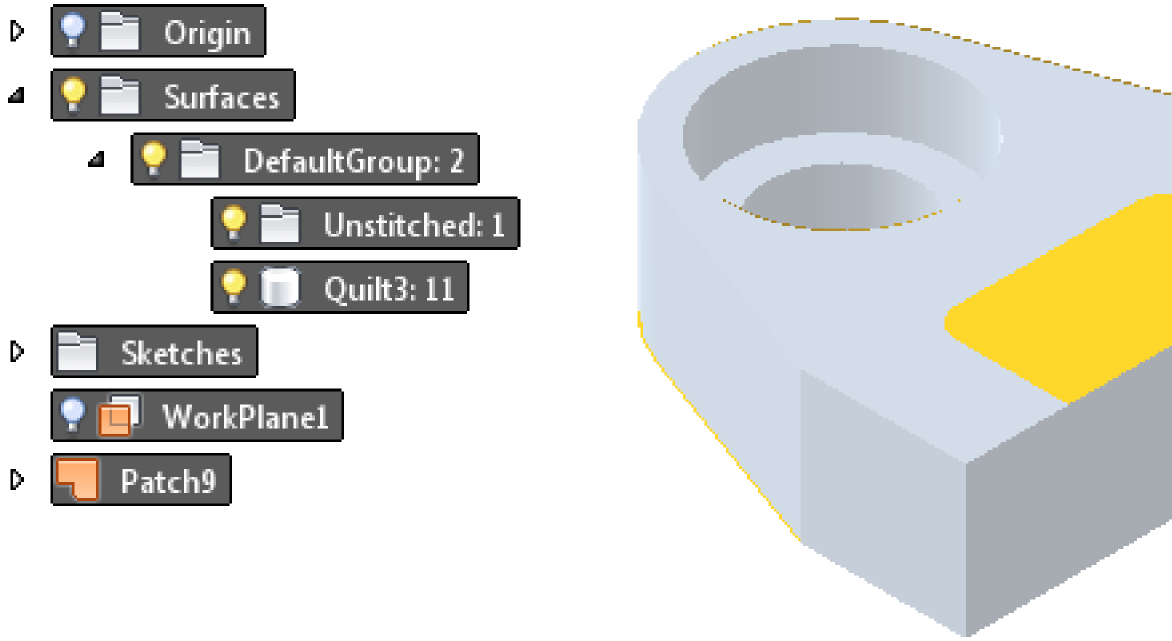


Stitch connects the edges of individual surfaces and quilts. The edges of the surfaces must be the same size and shape, and touching. Two or more stitched surfaces are called a quilt. It is difficult to create identical edges for complex surfaces, so Stitch can bridge small gaps.

After stitching, all of the surfaces are moved to a Quilt folder in the browser. Once a face is combined into a quilt, the browser feature is removed. If the quilt has no missing faces, it converts to a solid.

### Stitch

The edges of the surfaces must be the same size and shape, and touching. It is difficult to create identical edges for complex surfaces, so Stitch can bridge small gaps.




This model has a quilt and a surface.


After stitching, the quilt and the surface convert to a solid.

### Stitch surfaces into a quilt

1. Select Surface from the Modeling panel, then start Stitch from the Surfaces panel.
2. Select the surfaces and quilts. Edges that can be stitched are highlighted in green, and edges around missing faces are highlighted in red.

   **NOTE** You can use a selection window by pressing the Shift key while dragging the cursor.

3. Change the Tolerance setting if there is a gap between surfaces.
4. Click OK to stitch the surfaces and create the quilt.

### Unstitch

Unstitch divides a quilt into individual surfaces. Unstich is typically used to create several components from one model. For example, you can have an imported assembly model that comes in as a single solid. Unstitch separates the surfaces so you can create quilts for the individual components.

#### Unstitch a quilt

1. On the expanded Surfaces panel, click Unstitch.
2. Select one or more quilts to unstitch.

   **NOTE** To use a selection window, by hold down **Shift** while you drag the cursor.

3. Click OK. The surfaces are placed in an Unstitched folder in the browser.

## Offset Surface

The Offset command creates a surface or quilt with all surfaces at an equal distance from the originals. You can select multiple surfaces and quilts.

Offset cannot remove a face, so the maximum offset value is set at the distance where a face disappears.

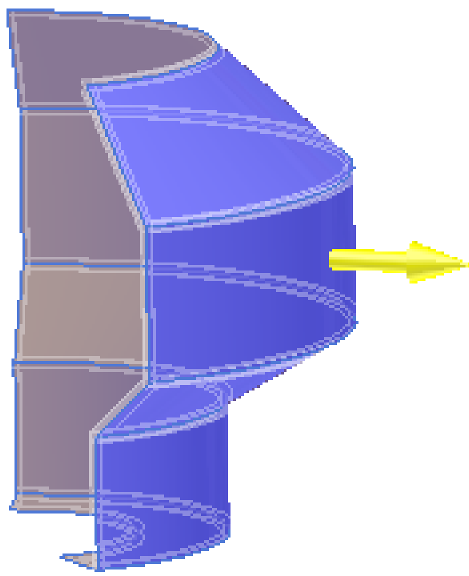
#### Offset a surface

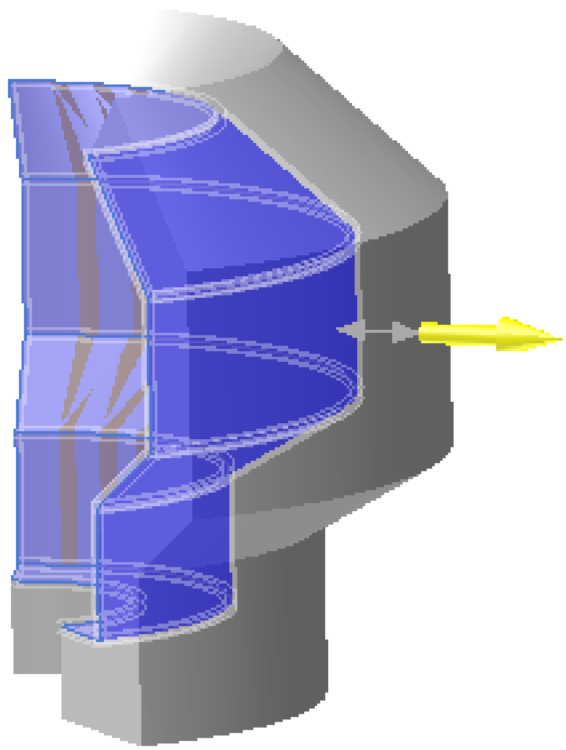
1. Start the Offset command.
2. In the graphics area, select one or more faces or quilts.

   **NOTE** To use a selection window, hold down `Shift` while you drag the cursor.

3. Drag the arrow to set the offset, or enter a value.
4. Click OK to create a solid.

### Thicken





Thicken offsets surfaces and quilts, and adds sides to create a solid. All surfaces are offset an equal distance from the originals. You can select multiple surfaces and quilts.

Thicken cannot remove a face, so the maximum offset value is set at the distance where a face disappears.

**Create a solid using Thicken**

1. Start the Thicken command.
2. Select one or more surfaces or quilts.
3. Drag the arrow to set the offset, or enter a value.
4. Click OK to create a solid.

## Split

Use Split to divide faces on a surface or solid, or divide a solid into two components. To start, create a work plane, surface, or sketch geometry to use as the splitting tool. Sketch geometry is projected onto the target faces.

You can split faces or a solid. Faces are typically split to add draft, delete an area, or to create new features. A solid is divided into two solids- one solid remains in the component and a child component is created for the second solid.

You can split a solid, surface, or quilt.

#### Split a face, surface, or solid

1 To start, create a work plane, surface, or sketch geometry to use as the splitting tool. Sketch geometry is projected onto the target faces.
2 Start the Split command.
3 Select the target faces or solid.
4 Click Split Tool, and select the intersecting work plane, surface, or sketch geometry. The Tool Chaining option automatically selects tangentially connected faces.
5 Click OK.

**Show me how to create a split**

### Reverse Normal

A surface has two faces. One side of the surface is positive, and one side is negative. The X-and Y-axes of the triad are always on the surface, and the Z-axis defines positive and negative. When a Shaded visual style is active, the positive side of a surface is gray, and the negative side is yellow.

You can use Reverse Normal to change the positive direction of the Z-axis. This action is typically used for repairing imported models. During the translation process, especially to and from neutral formats, some faces can be assigned the wrong orientation.

Reverse Normal is located on the expanded section of the Surfaces panel.

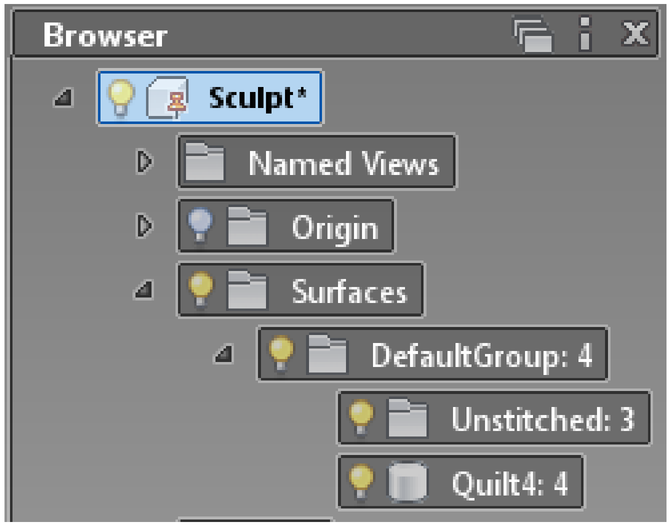
## Surfaces in the Browser

Fusion surface objects consist of quilts and unstitched single surfaces.

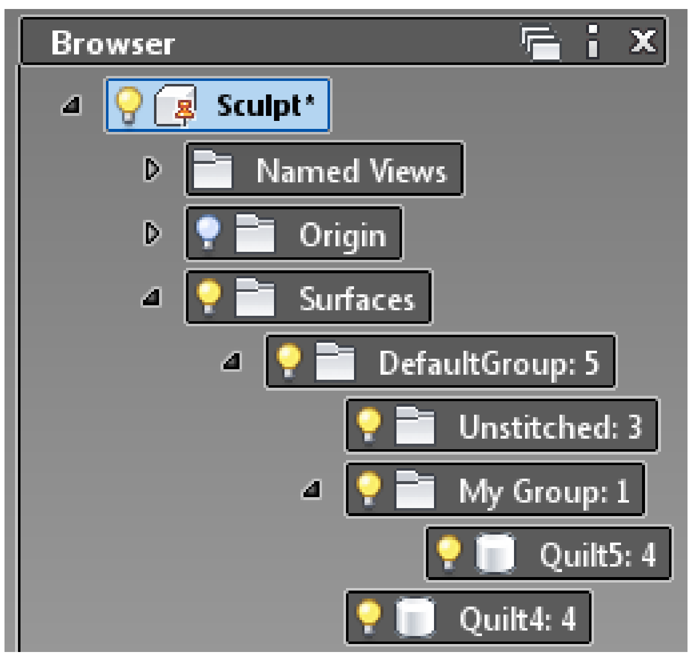
A quilt is a set of stitched surfaces. Extrusion of an open sketch with multiple curves is a typical example of a quilt. Fusion surfaces are visually represented (except in realistic view mode) as having two sides, a positive side and a negative side. The positive side is shown in a translucent faded color and the negative side is shown in dark yellow.

All surface objects are grouped under browser items called Surface Groups. Every component has a top-level browser item called Surfaces which contains all the surface groups. Surfaces that result from some operation are automatically added to a special surface group called DefaultGroup, which is typically the first surface group that gets created. You can create other surface groups using the New Group command. A surface group can be nested below another surface group. Quilts and unstitched surfaces also have corresponding browser items

The following image shows a model with one quilt and unstitched surfaces. The quilt (Quilt5) has four surfaces, and the unstitched group contains three surfaces. It also indicates that the default group has four items: three unstitched surfaces and one quilt.




This image indicates five items within the default group; two quilts and three unstitched surfaces. It also shows how nested groups can be created. You can organize surfaces using drag and drop.



## Validate Command

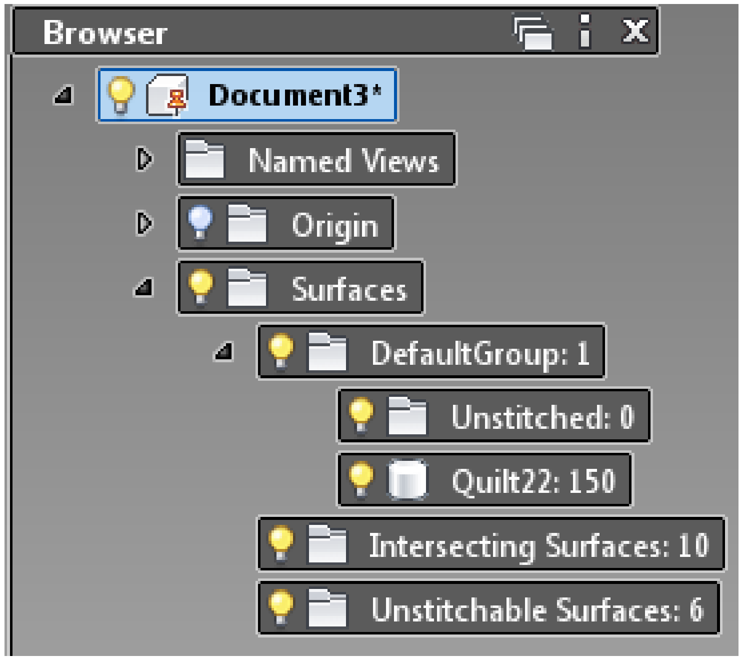
Validate verifies the quality of an imported model. It checks the selected bodies, attempts to repair bad surfaces, and then stitches together any surfaces that can create quilts or solids.

Validate performs the following for each selected solid body, quilt, or individual surface:

1. Unstitch all quilts and solid bodies.
2. Run a quality check on all individual surfaces.
3. Attempt to heal surfaces that did not pass the quality check.
4. Stitch the good surfaces.

The results of the Validate command can include:

- A solid.
- Quilted surfaces.
- Healthy surfaces that cannot be quilted.
- Intersecting surfaces.
- Unstitchable surfaces.
- Invalid surfaces.




### Use Validate on models

- On the ribbon, Modeling panel, expand the Surface panel and click Validate.

  or

  In the browser, right-click a surface group, and select Validate.

### Options

The "Tolerance" value affects the quality check and stitch processes. The default value is 0.001 cm.

The Checking Level has two possible values:

- **Basic** Covers a set of topological, geometric, and uncertainty errors.
- **Standard** Covers the Basic errors plus intersecting and overlapping faces.

## Geometry Error Tests

The Standard checking level includes all tests in the Basic level, plus tests for intersecting and overlapping faces.

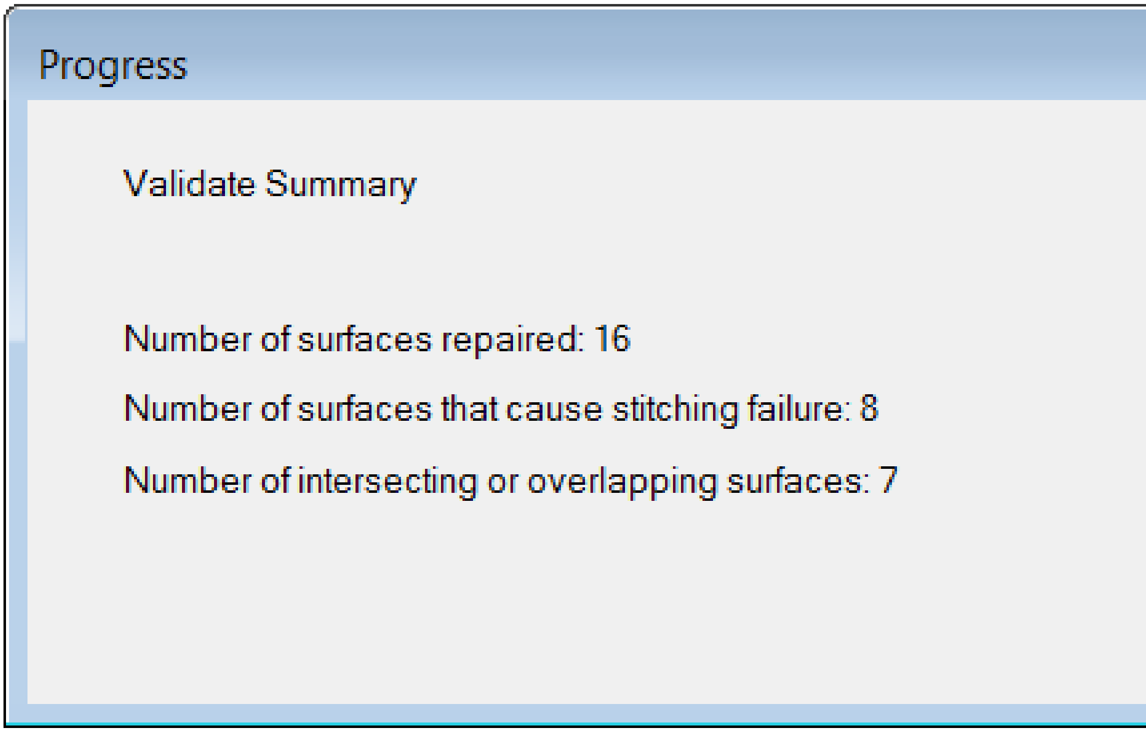
| Basic Option Tests | |
|---|---|
| **Self-intersecting surfaces** | A surface that folds onto itself is a self-intersecting surface. Surfaces must be continuous and smooth without changing direction. |
| **Self-intersecting curves** | Curve data comprises lines, arcs, or splines. Curves must be smooth without changing direction. Curves cannot reverse, twist, or intersect. |
| **Modeling uncertainty** | The bodies or surfaces have low-level errors. |
| **Irregular surfaces** | The surface that was generated during translation did not fit within system tolerances of the original surface in the imported file. This error can also be caused if the normal vector for a point on the surface is not pointing in the same direction as the rest of the surface.<br>These errors can occur if surfaces are twisted or collapse into a small area. |
| **Face orientation issues** | The loop direction is inconsistent with the normal direction of the face or inner loops do not agree with each other.<br>The top side of a face is referred to as the surface normal. Adjacent faces within a solid must all have a consistent normal direction. For example, the normal direction of the faces of a box must all point either inward or outward to be a valid solid. |
| **Loop orientation issues** | The outer loop is going in a wrong direction compared to the face normal direction. If the face includes islands (loops enclosed within the outer loop), the normal direction of the islands must point opposite to the outer loop. The start and endpoint and direction indicator define the loop direction.<br>In addition to the loop and islands having the same direction, a surface has a normal direction that must agree with the loop direction.<br>If any of the directions are opposite the others, an error is found. |

| Basic Option Tests | |
|---|---|
| **Duplicate vertices** | Start and endpoints of an edge are vertices. During translation, attempts are made to merge vertices that fall within the system tolerances. Duplicate vertices can occur when small edges make up a complex object. |
| **Irregular curve** | Math data is inconsistent in the curve definition or a vector is zero. This error can occur when the approximating surface does not fit within the system tolerance of the defining surface in the neutral file. |
| **Singularity surface** | A point on the surface vector is poorly defined. The surface normal cannot be determined. |
| **Degenerate surface** | The points that comprise the surface are in an area that is too small. |
| **Surface discontinuities** | The normal direction or curvature of the surface changed abruptly. Disconnected geometry can cause the error. Surfaces must be smooth (G1, G2) and cannot have an abrupt change in direction (G0). |
| **Curve discontinuities** | Curve data comprises lines, arcs, or splines. Curves must be smooth and cannot have an abrupt change in direction or an abrupt transition between curves (G0). |

| Standard Option- | |
|---|---|
| **Intersecting faces** | Two or more faces converge or pass through one another. |
| **Overlapping faces** | Two or more faces are coplanar or overlay each other. |

### Progress dialog box

A dialog box shows the progress of the check/heal/stitch process, and reports the results when the command is complete.




Click the Stop button to terminate the command at the earliest possibility. Any validation that is completed is preserved. To resume the command later, select the same set of bodies and start the command again. To cancel the command, stop the process and then undo.

**Body repair process**

The validate command attempts to automate the repair process. Some bodies cannot be automatically repaired. The following commands are useful for manually repairing bodies:

- Surface patch
- Reverse normal
- Surface delete
- Surface merge

# Direct Modeling

In Fusion, you can use direct modeling to make changes to model geometry without editing sketches or features. You can make simple changes, such as making an extrusion longer or shorter, or complex changes, such as modifying the curvature of an edge.

Related Topics:

- Dimensions and Body Constraints (page 357)

## Press - Pull Command

The Press/Pull command is one way to modify body geometry. In general, use Press/Pull as an offset style of modification. That is, the modified geometry is replaced with an offset of itself.

### Show me how to press/pull faces

Use press/pull to modify faces, fillets, and other geometry.

### Show me how to press/pull complex faces

You can press/pull complex faces and use split faces to press/pull a portion of a face.

**Command Interaction**

On the Ribbon, Marking Menu, or context menu, invoke Press/Pull. Click one or more faces (on one or more bodies), and drag over the arrow drag manipulator, or an area of the design with no geometry. You can drag, then release the mouse button, drag further, and repeat. When finished, start a new command, or click OK in the ribbon.

**Additional Functions of Press/Pull**

Press/Pull can invoke two additional commands: Fillet and Extrude. If, while in the Press/Pull command, you select a model edge, a fillet starts on that edge. If you select a sketch closed profile, an extrude starts on that profile.

## Move Command

The Move command in Inventor Fusion moves objects, including:

- Component instances

- Work Geometry
- Model body faces

You can use Move to make a wide range of geometric changes to a design. For instance, move one or more faces to modify the geometry of the body in a design. For information about using Move with components, see Position and Constrain Components (page 383).

**Show me how to move geometry**

This video demonstrates moving faces, aligning one face to another, and rotating a face

**Show me how to move geometry that contains dimensions**

This video demonstrates moving geometry that has dimensions. Both transient and persistent dimensions are used.

**Show me how to reorient the triad**

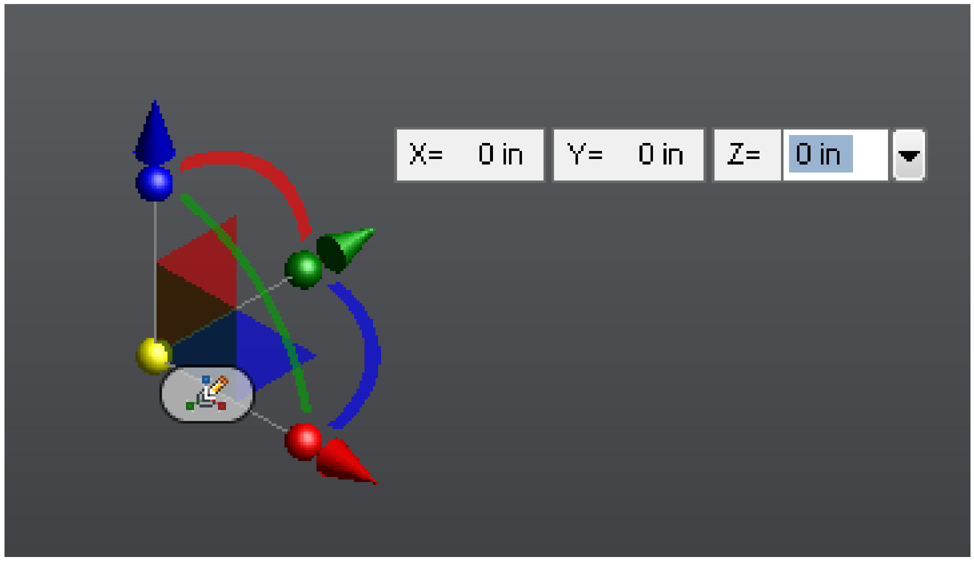
This video shows how to reorient the triad and the affect of different triad positions.

**Show me how to move edges**

This video shows how to move model edges and the effect on geometry.

### About the Move command

The Move command uses the Fusion triad manipulator:

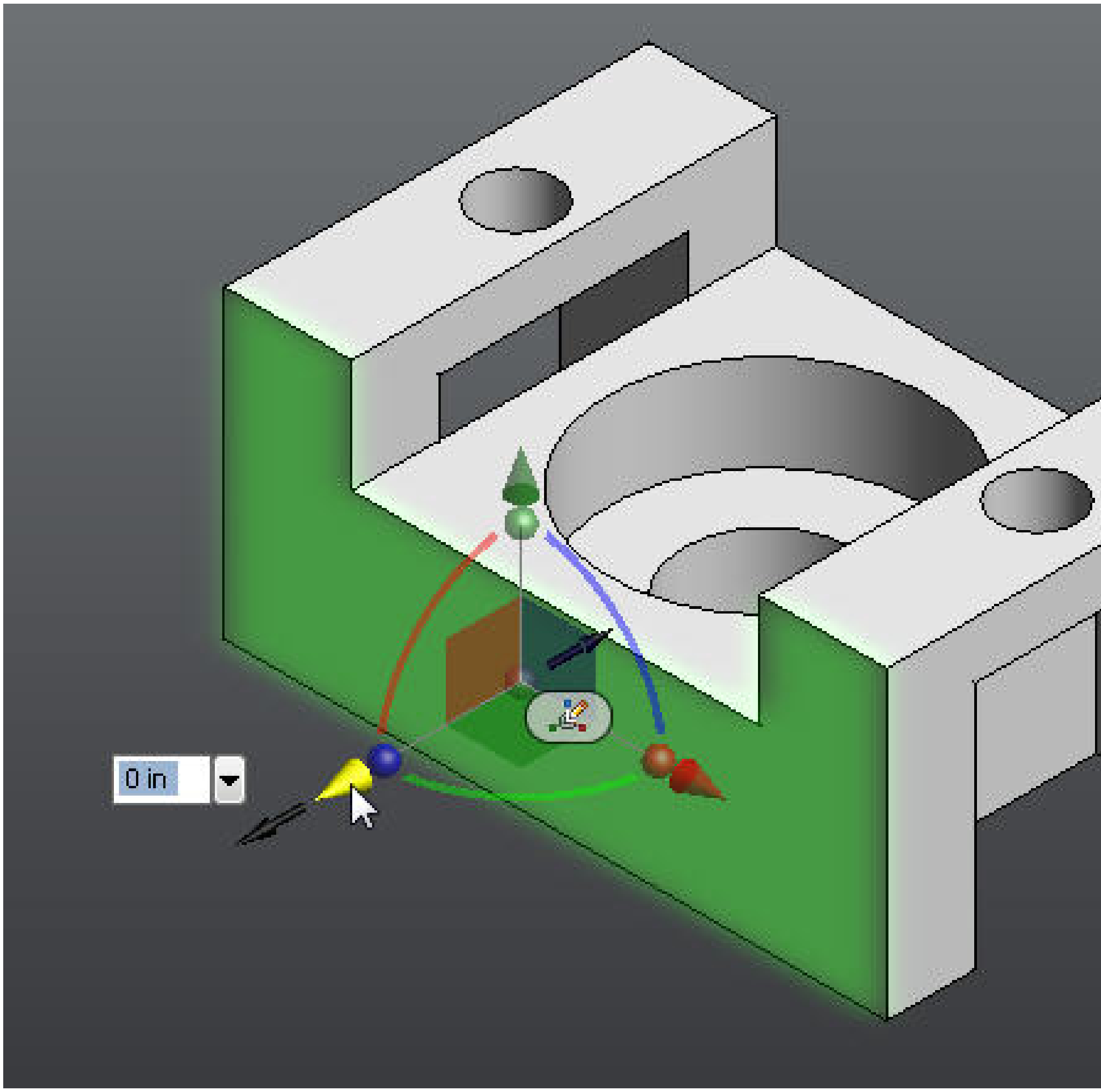


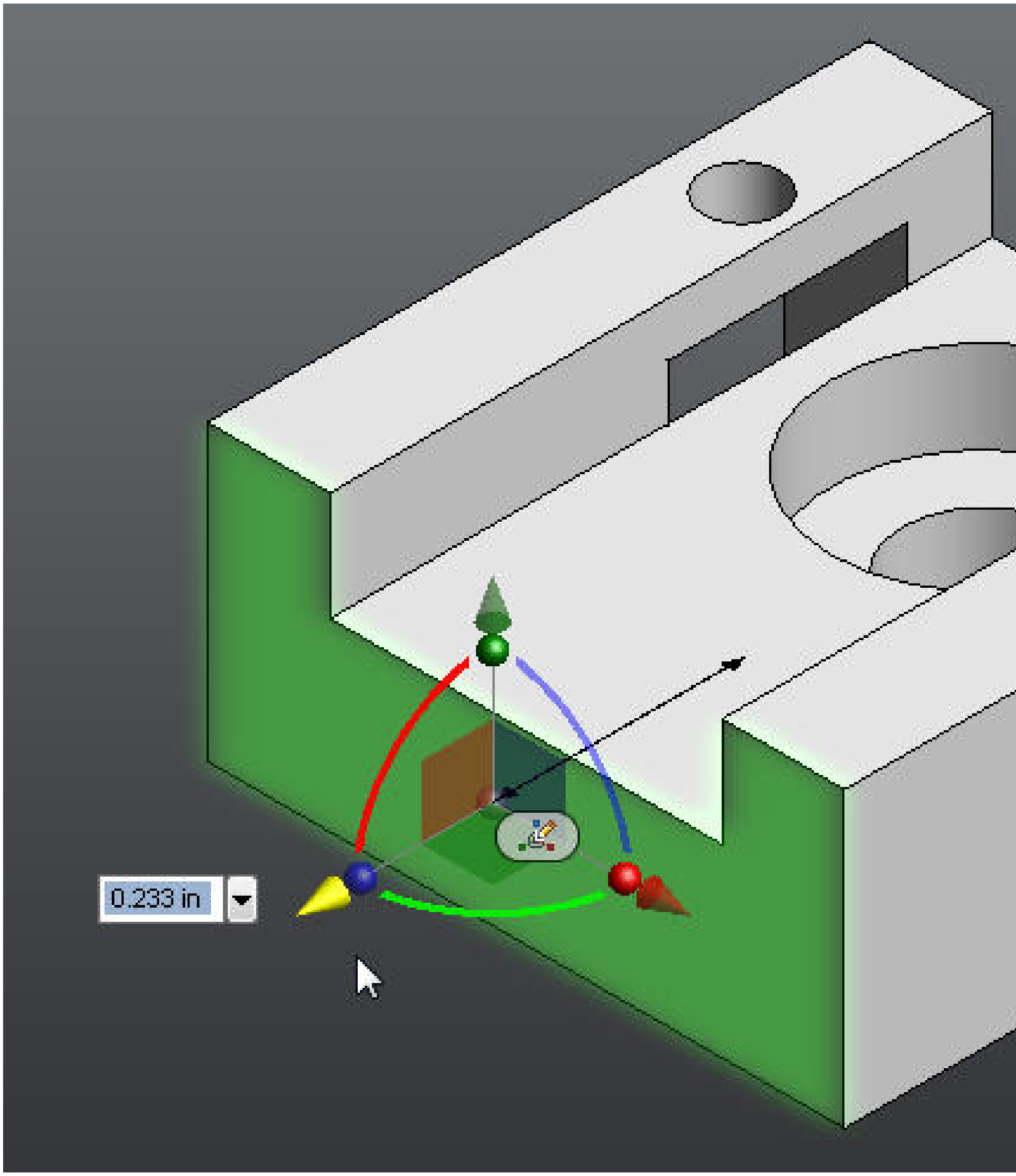

For more information about this manipulator, see triad (page 47).

**Use linear manipulators in Move to translate faces**

You can use the arrow-shaped manipulators on the Move triad to translate the selected model faces in the direction of the arrow. Drag either over the manipulator itself, or over an area of the screen where there are no model graphics or user interface elements.

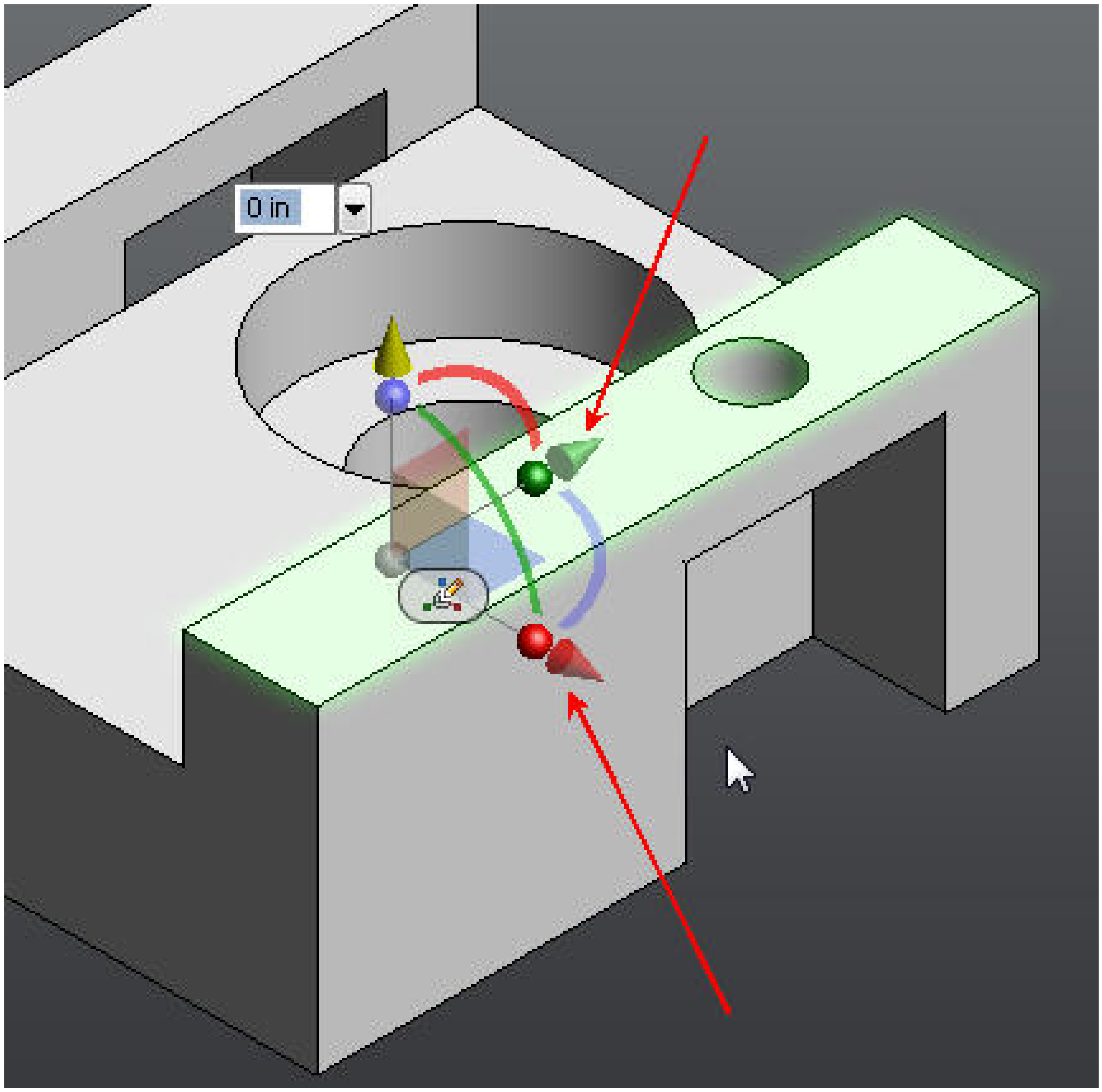
For instance, in this case, a single face is selected and moved along the yellow (yellow indicates selection) direction:


0 in


0.233 in

**Apply Move to geometry of any kind.**

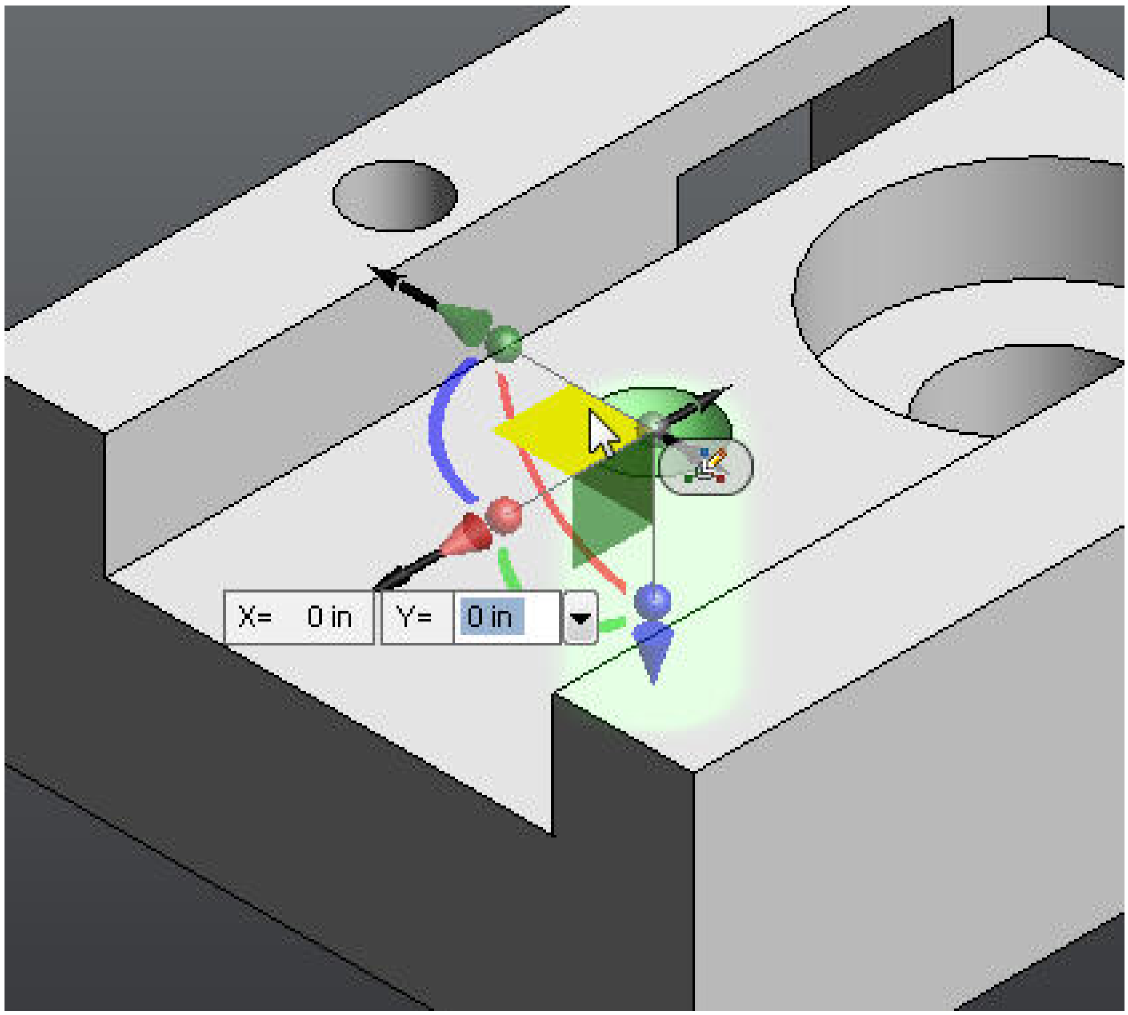
Some faces are limited in their ability to move, by the geometry of the model. In this case, the red and green translation manipulators are disabled.



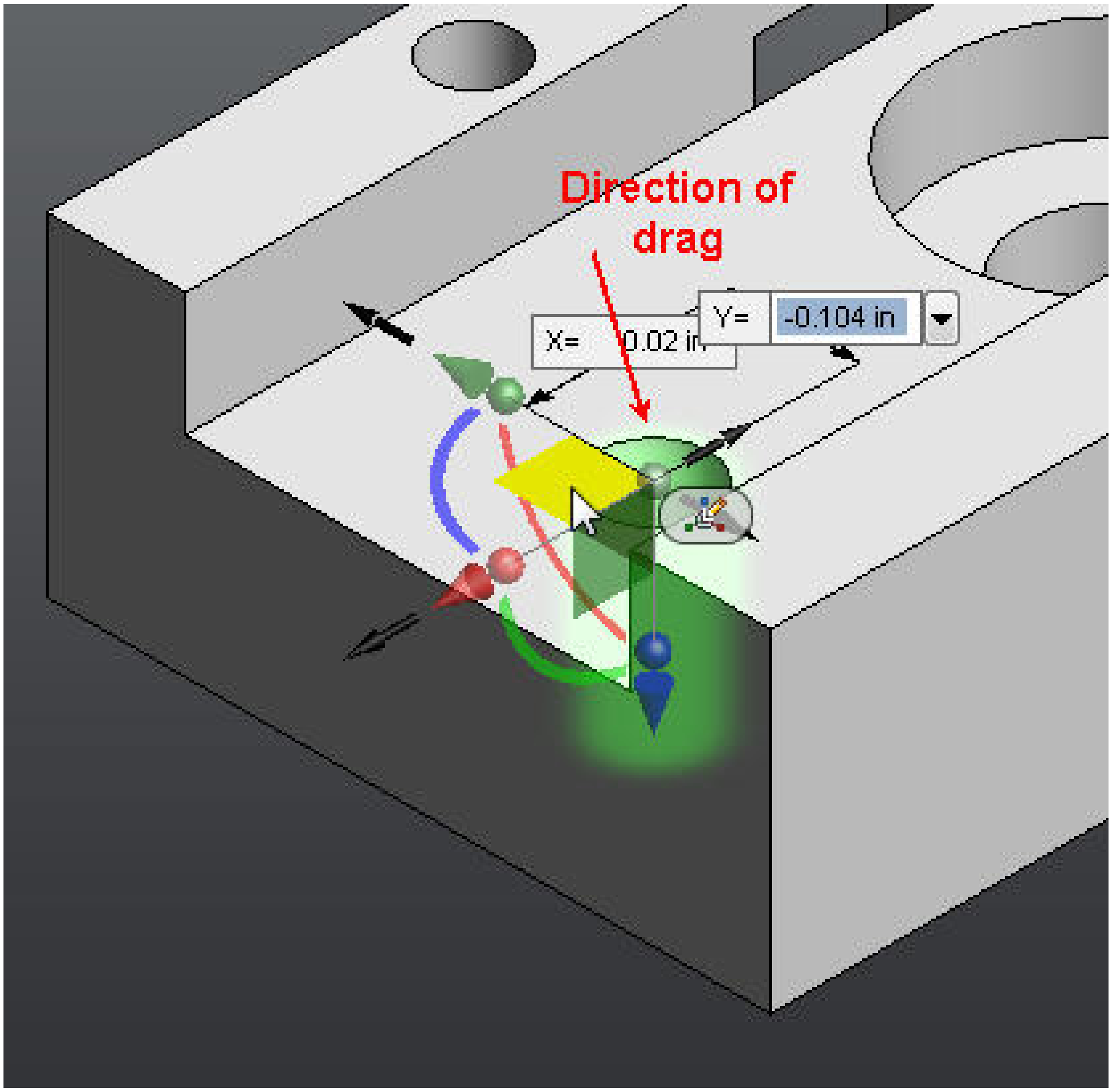
This planar face is not free to move in those directions. In some cases, the move command cannot detect these limitations. Manipulators are enabled, even though dragging in that direction does not result in a change to the model.

**Use plane manipulators in Move to translate faces**

The planar manipulators in the Move triad can be used to translate the selected faces in two directions at once.

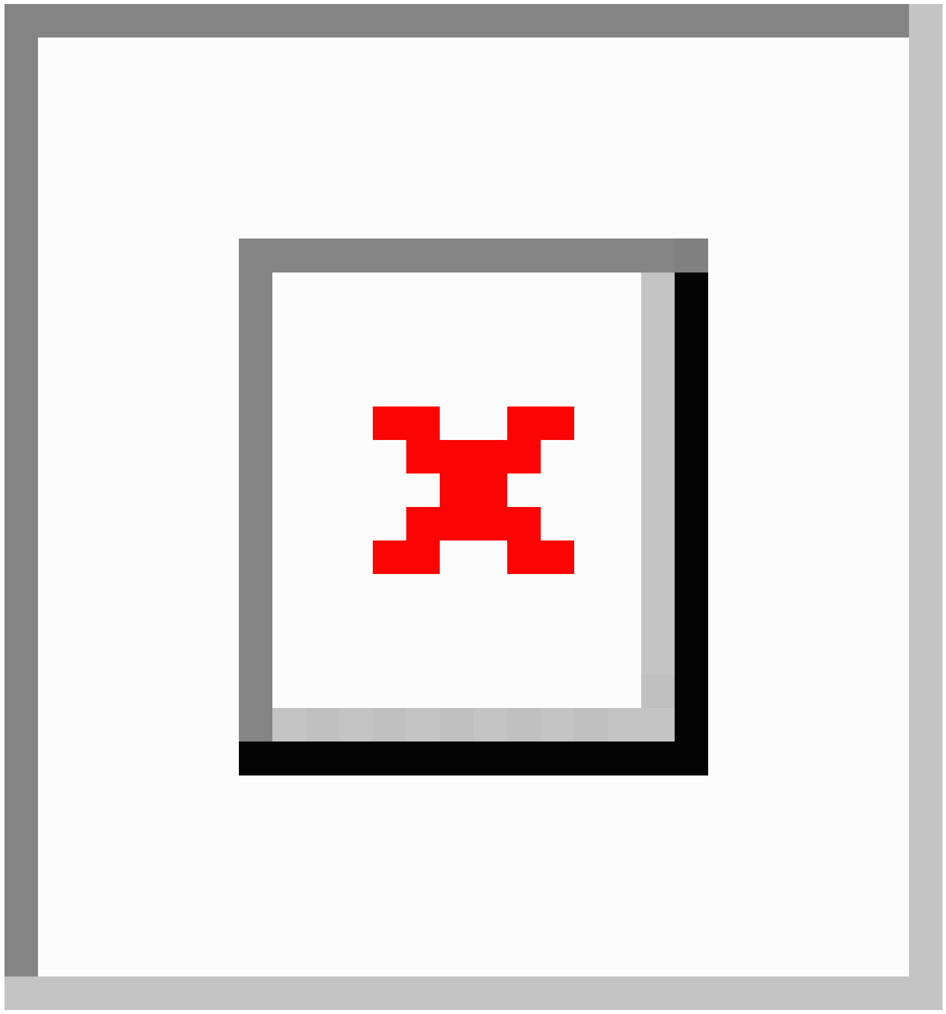


We use the highlighted plane manipulator to move the selected cylindrical face in the direction of either the green or red arrow at the same time.

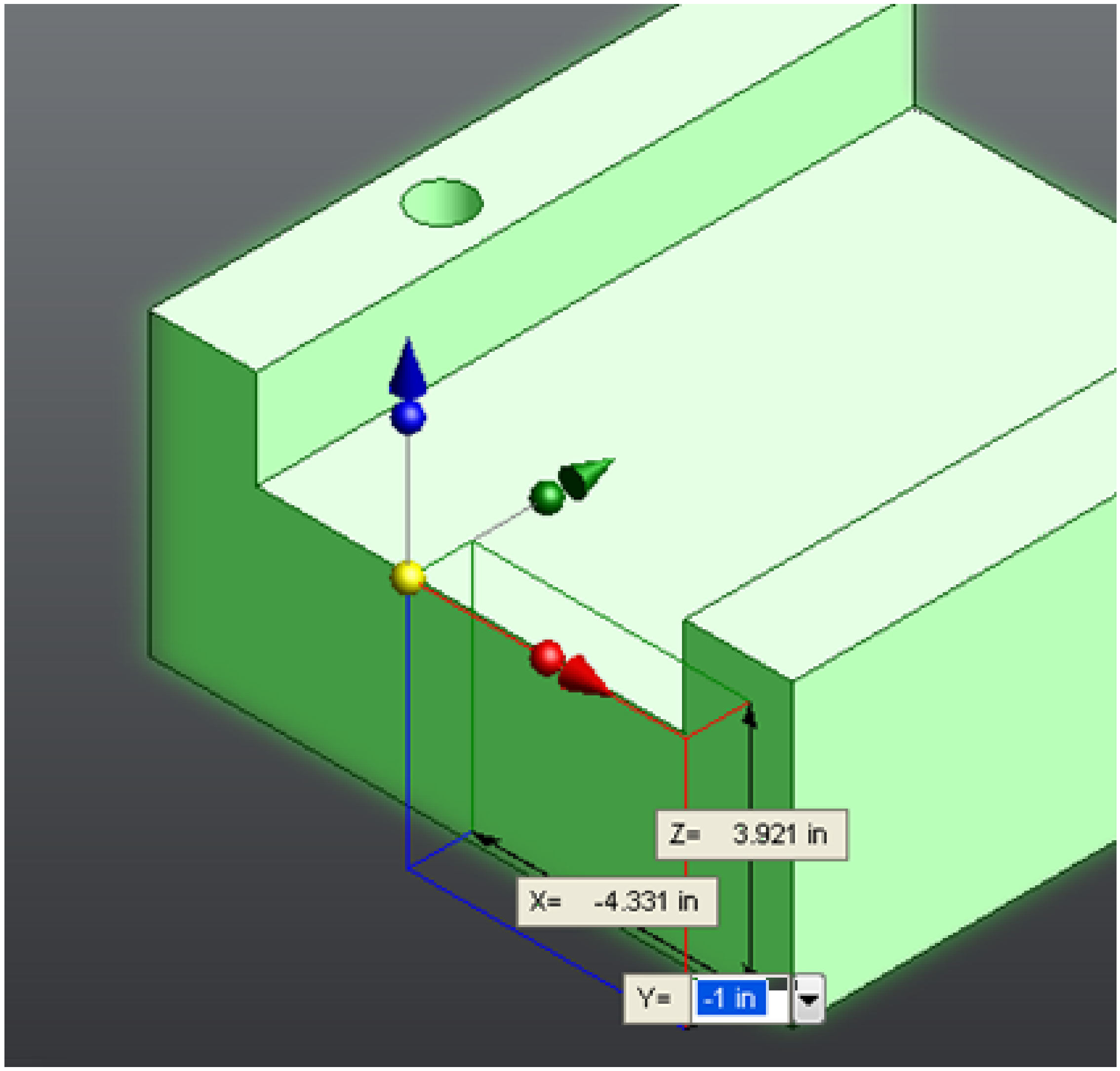



**Use the origin manipulator in Move to translate faces/features/components**

You can use the origin manipulator in the Move triad to translate the selected faces/features/components in all three directions at once.



The highlighted origin manipulator moves the selected component in the three directions of the green, red, and blue arrows at the same time:

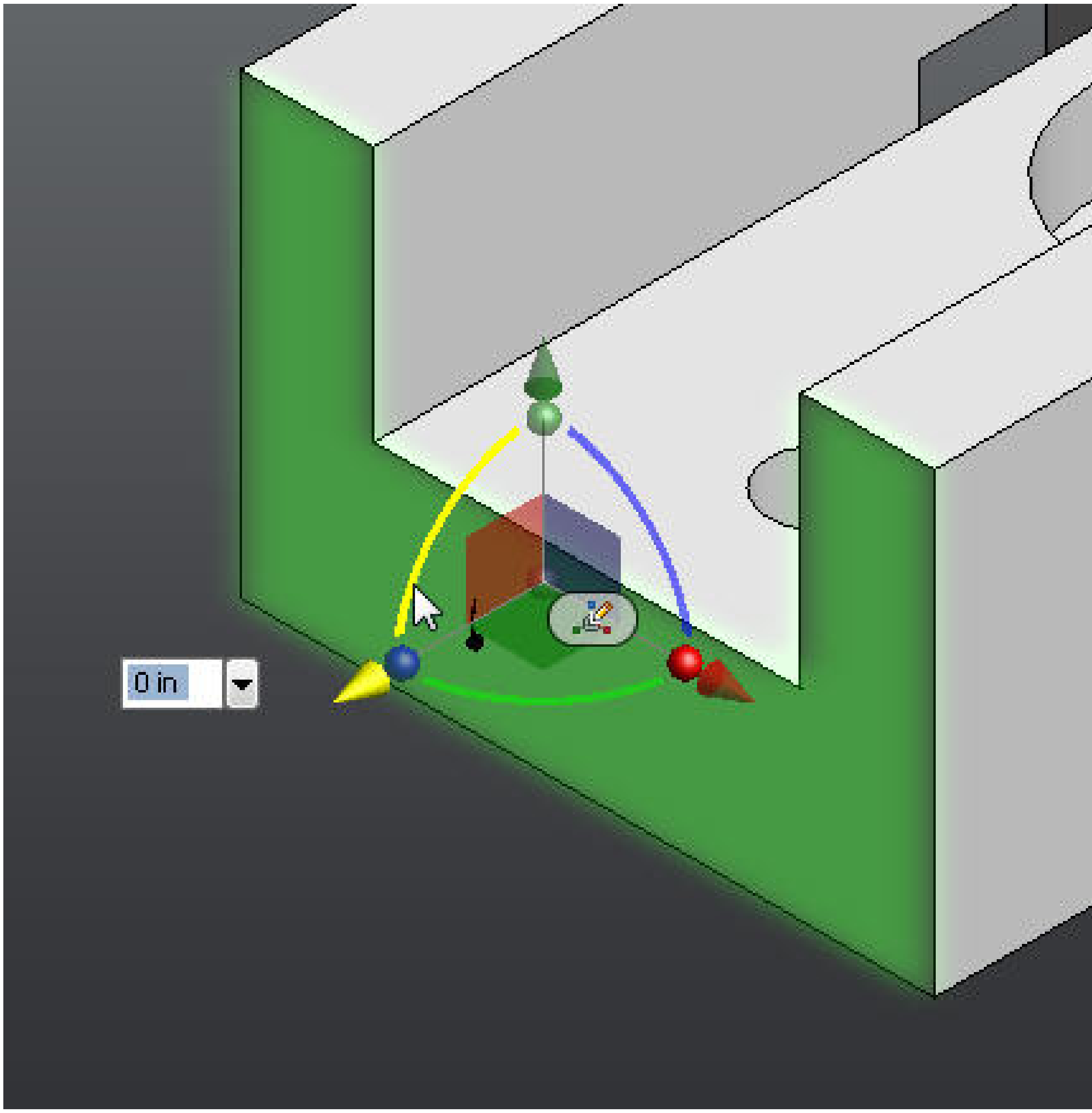



**Use Move to translate multiple faces at the same time**

You can modify more than a single face at a time. In the following image, we use Move to translate the two selected horizontal faces in the same operation:

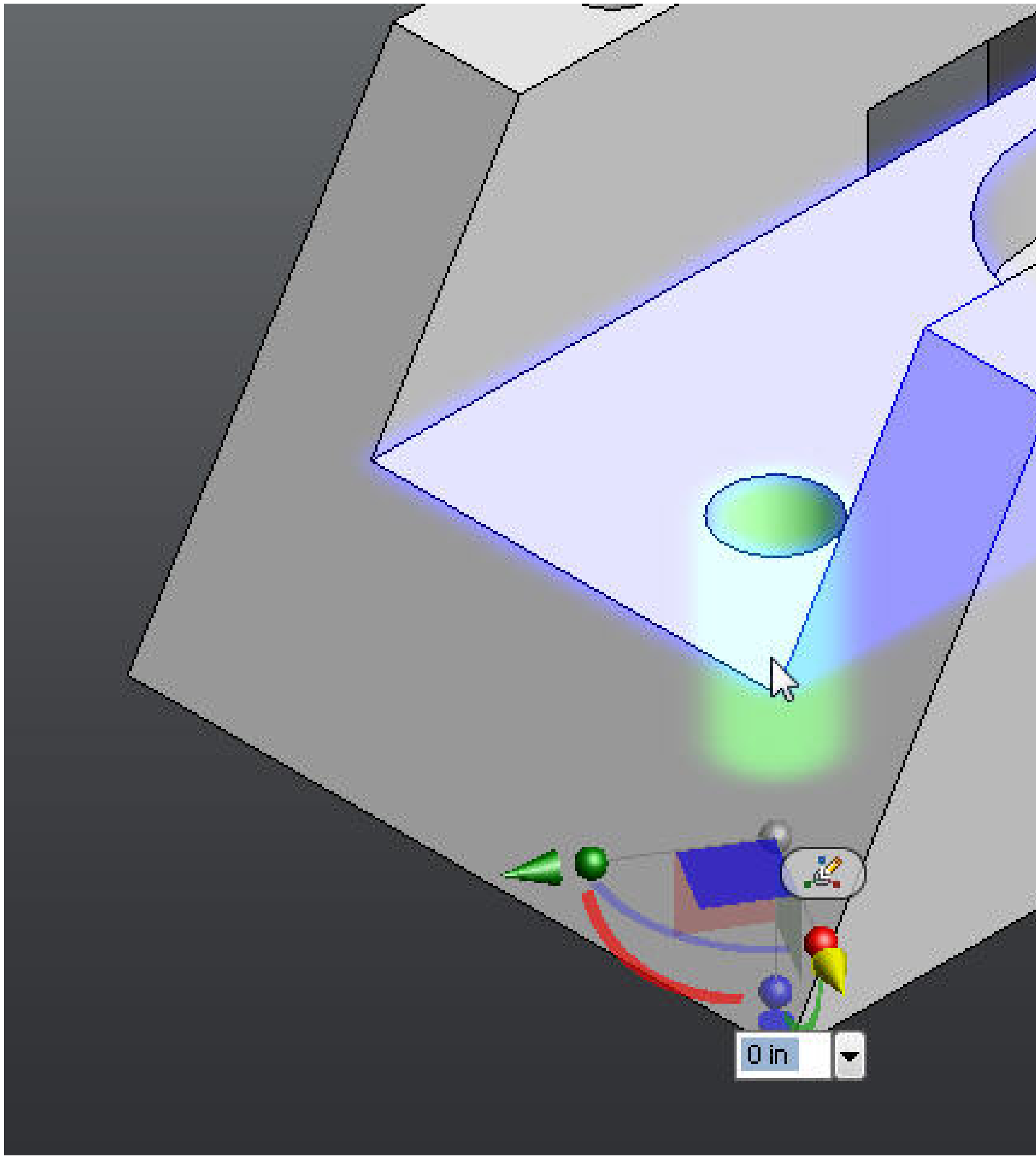
**Use Move to rotate faces**

Besides translation, you can use the Move triad to rotate faces. The rotate manipulators are the arcs on the triad. In this example, a rotate manipulator is used to tilt a model face:

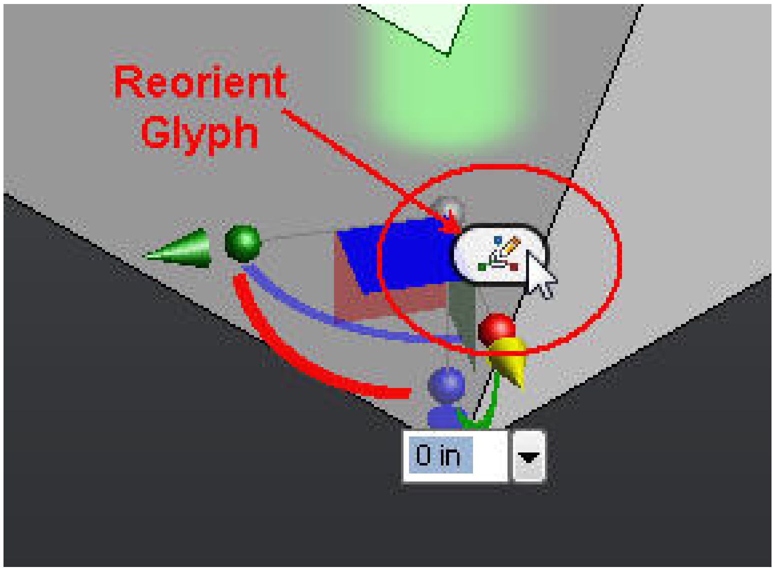


### Reorient the triad

When a face is selected in the Move command, Inventor Fusion places the Move triad at a default location and orientation.  At times, this orientation and position does not match the appropriate transformation.  For instance, in this case, a cylindrical face is selected.

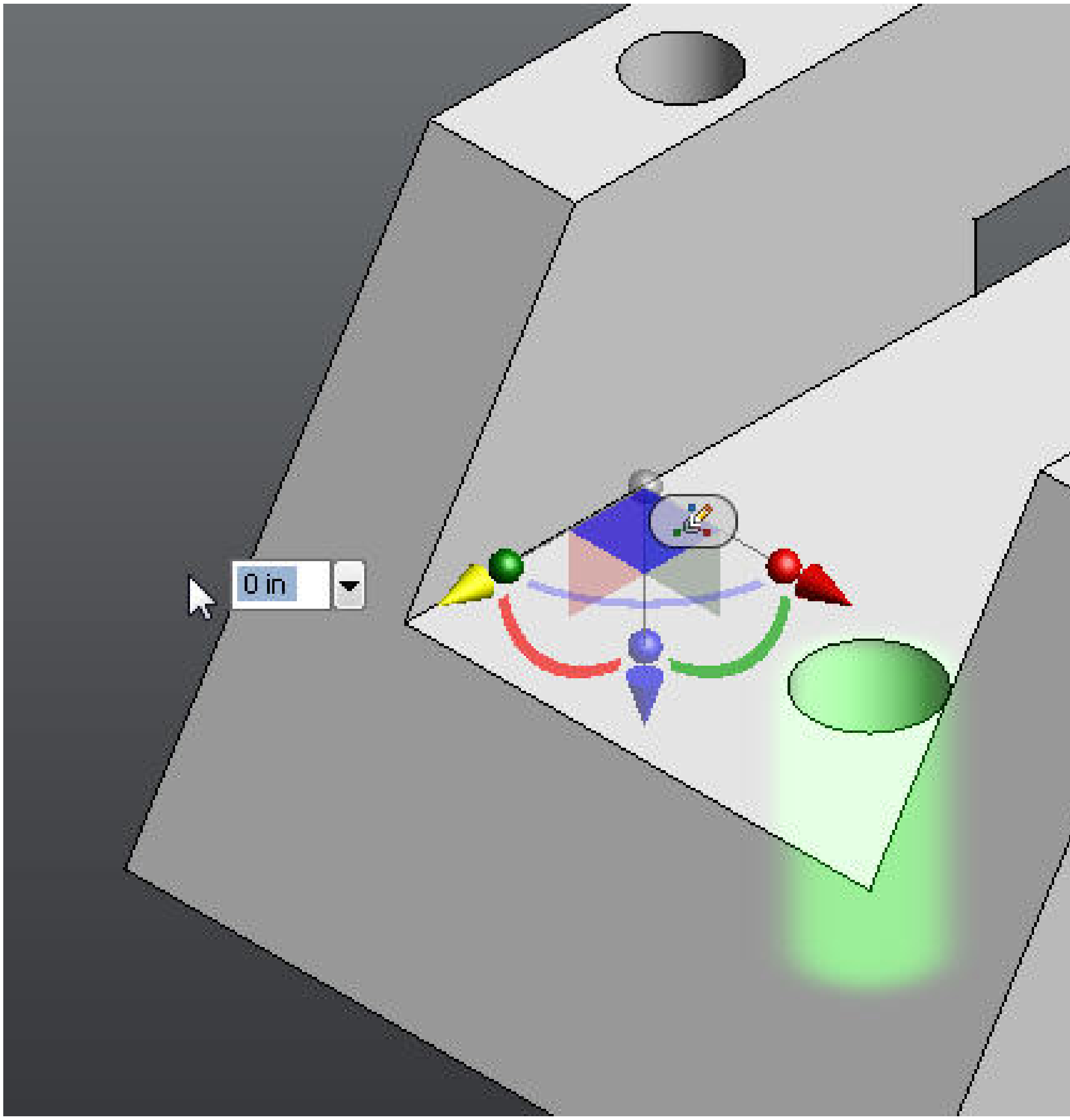

0 in

However, the default orientation is not ideal for moving the face along the rectilinear face that the hole is on.  So, use the Reorient glyph to change the default orientation to fit your intended design modification more closely.




If you click this glyph, the Move command enters reorient mode.  In this mode, you select geometry to reorient the triad, aligning the active triad manipulator with the selected geometry.  If a translate manipulator is active, you can select a linear edge. The triad moves so that the active translate manipulator aligns with the edge.  For instance, if the highlighted edge is selected while in reorient mode:

The triad reorients itself along the selected edge:




You can reorient manipulators along linear edges, work axes, cylinders, and planar faces. You can reorient the triad about an active rotate manipulator.

Reorienting about a rotate manipulator is identical to reorienting about the corresponding translate manipulator.  For instance, the blue rotate manipulator corresponds to the blue translate manipulator.

### Snap in the Move command

You can move one or more faces so they align precisely with other geometry in the design. For example, you want to make the slots in the side of the part the same size.  Select the face as shown in the following image. Then move the selected green face so that it aligns with the corresponding blue face on the opposite side of the design.

Use Move with its snap to geometry feature. If you select an applicable face that can be snapped to, Move transforms the active faces so that they are aligned. The program indicates that snapping is possible with the prompt Select to snap when the cursor is over an available snap face. In the previous case, the selected green face is aligned with the indicated blue face.

### Move and body constraints

During a move operation, Move obeys any body constraints or locked dimensions. For instance, a dimension has been created and locked between model geometry. As one face moves, the other follows along.

In this case, we created a dimension and locked its value:

If the top face of the design is selected in the Move command, the dimension is honored. The bottom face moves along with the top face:

For more information, see Dimensions and Body Constraints (page 357).

## Freeform Modeling

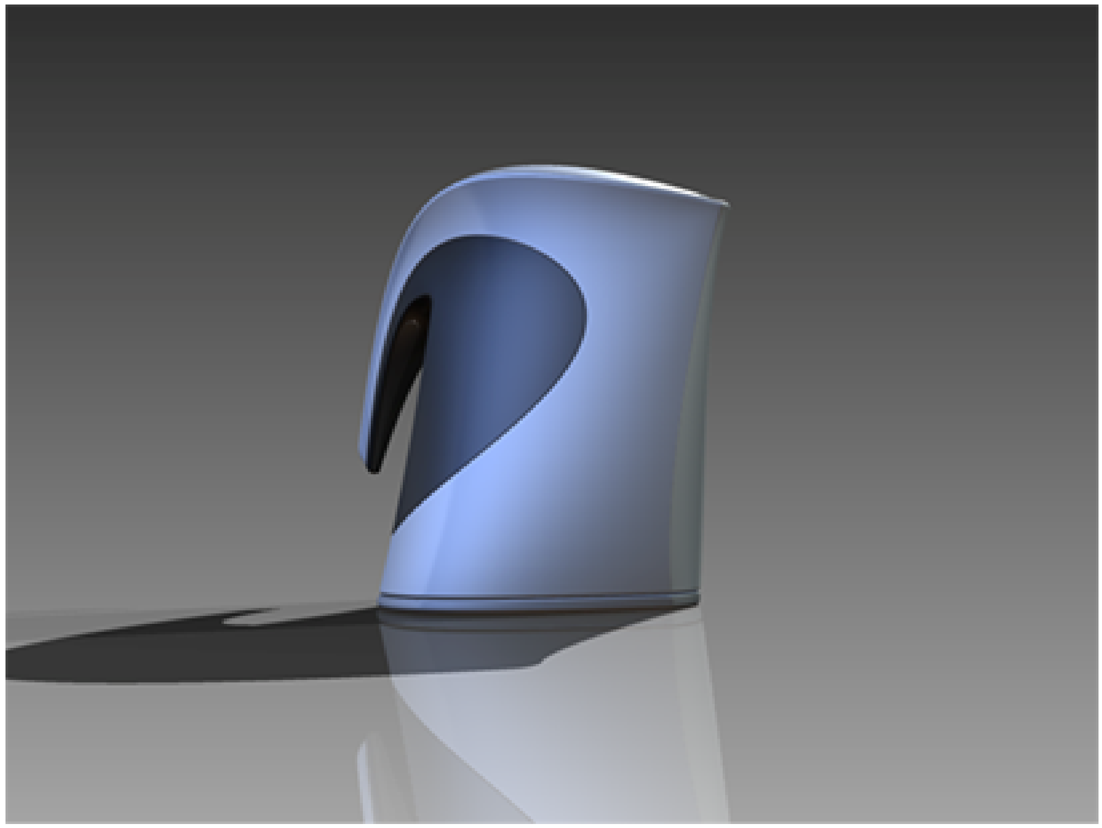
Inventor Fusion contains a set of freeform modeling commands that significantly enhance design possibilities. Use the tools to reshape solid body edges and surface edges, transforming geometric forms to more organic ones. The tools work within the Fusion modeling methodology, providing ultimate flexibility to create, and explore interesting shapes.

#### Key Freeform Editing Workflows

Using the freeform modeling commands in Fusion, you can:

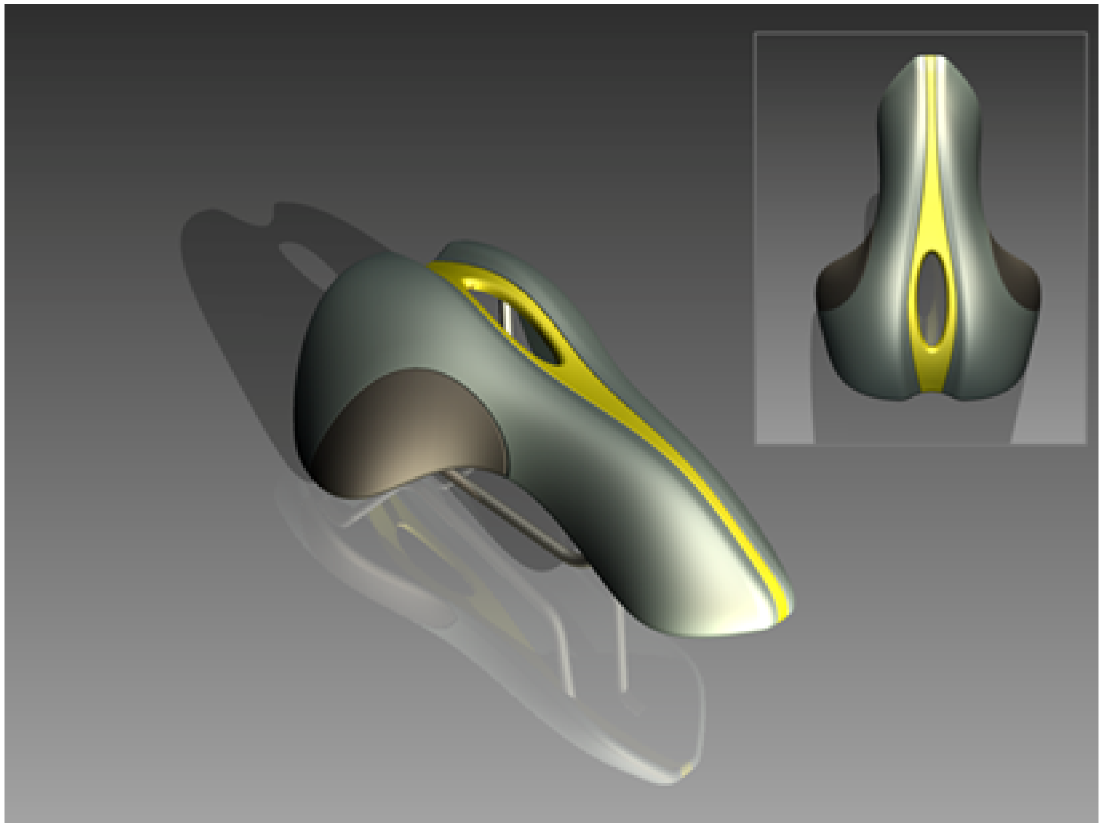
**Easily edit freeform shapes**

Edit freeform shapes by directly editing the intersection edges. Easily modify the shapes to explore freeform designs.



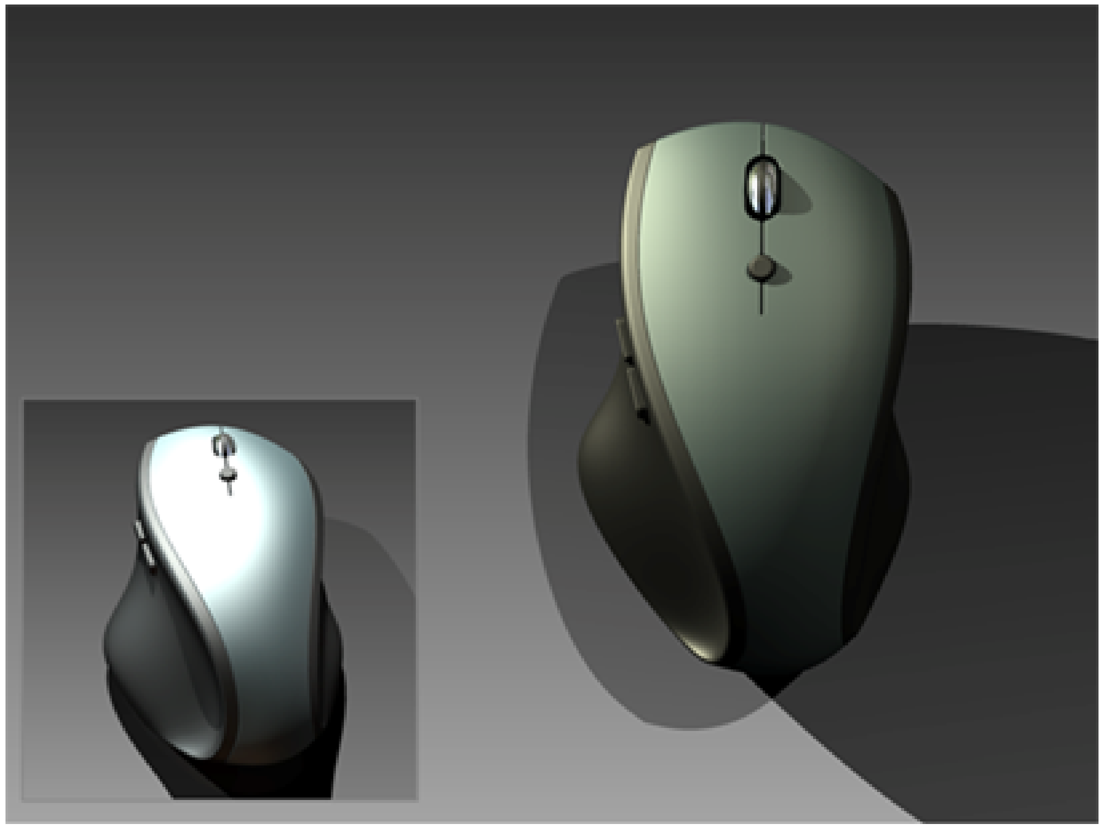
**Edit complex surface development**

Edit surface development that is sculpted from different directions. Modify these surfaces by directly editing the edges.



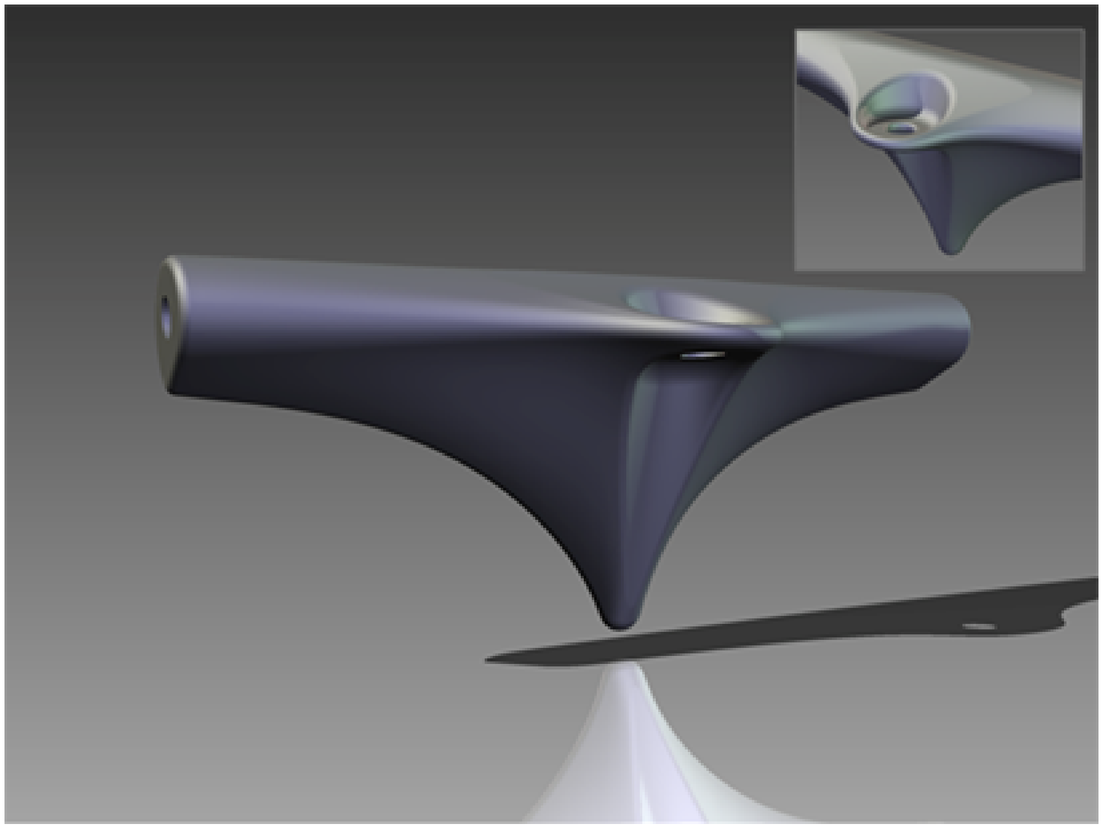
**Edit complex freeform shapes**

Edit freeform shapes that are difficult to build through surface modeling and re-edit these shapes easily.



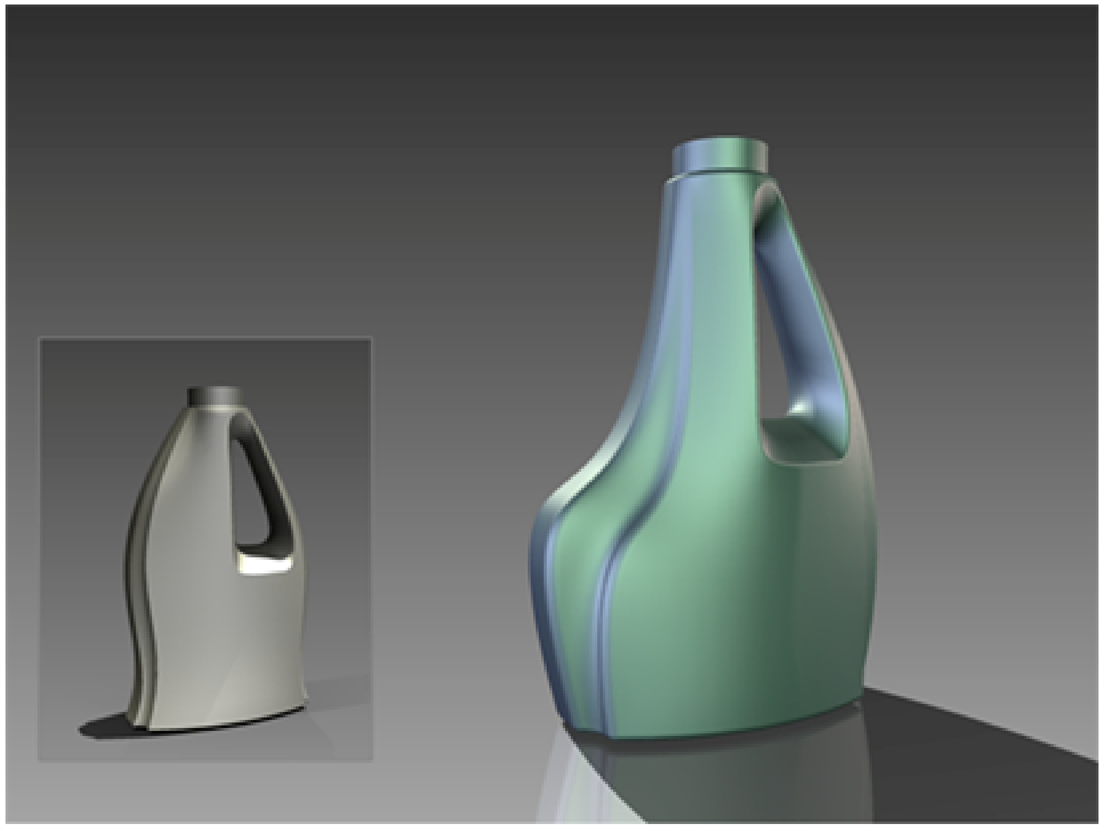
**Quickly edit complex engineering parts**

Quickly edit engineering parts that involve complex surfaces. Easily re-edit these complex parts.

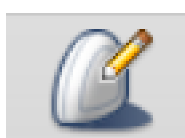


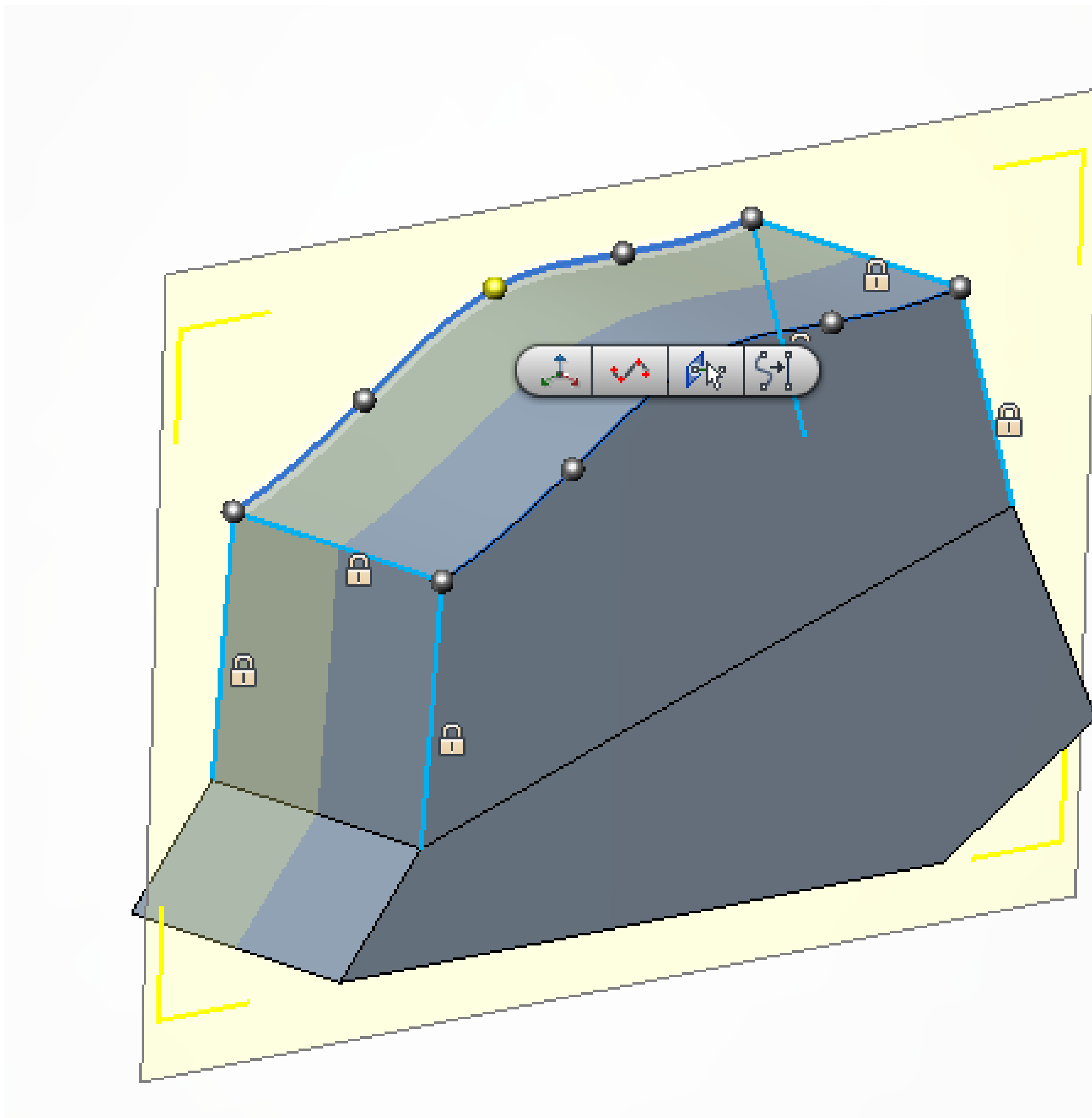
**Edit imported geometries**

Modify and improve shapes of any imported geometry. Easily re-edit the shape to make design changes efficiently.



## Edit Edge

Use the Edit Edge  command to reshape edges on a Fusion model. You can select edges of solids or surfaces.



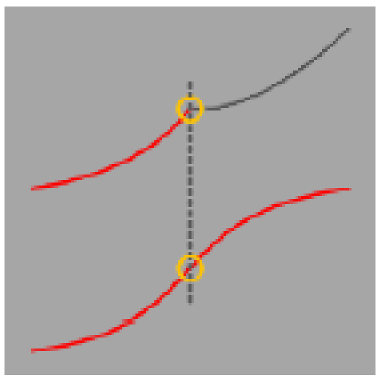
### Selections

- Edge

Use Edge option to reshape edges by dragging points on the edge.

- Merge Edges

Using the Merge Edges selection, you can merge the currently selected edge with an edge connected to its start or end points, making the edges G2 continuous.



- Plane

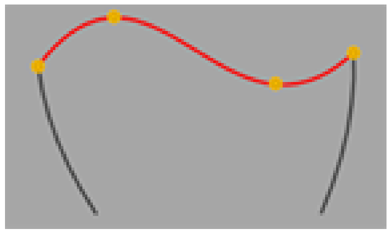
The Plane option pairs with the **Parallel to selected plane** on the Point Mode panel. Point moves only on the selected plane, you can't drag it off the selected plane. This is used when you want to make sure the edited points will remain parallel to the selected plane.

**Curve Mode**

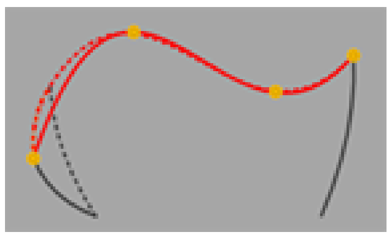
Curve mode controls how intermediate points on the selected edge react to changes. Select between Interpolation or Shape.

- Interpolation Curve Mode

  The start and end points and all intermediate points on the edge are Interpolation Points  .

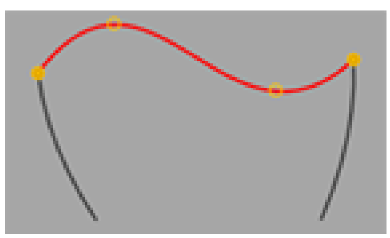


When the edge is edited, the Interpolation Points stay fixed and do not float with the edge.

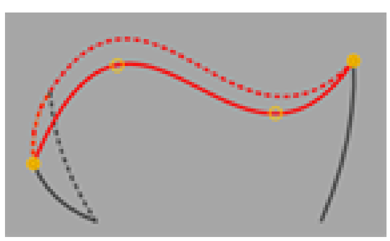


- Shape Curve Mode

  All intermediate points on the edge are Shape Points  ( the start and end points are Interpolation Points).



When the edge is edited, the Shape Points float with the edge.



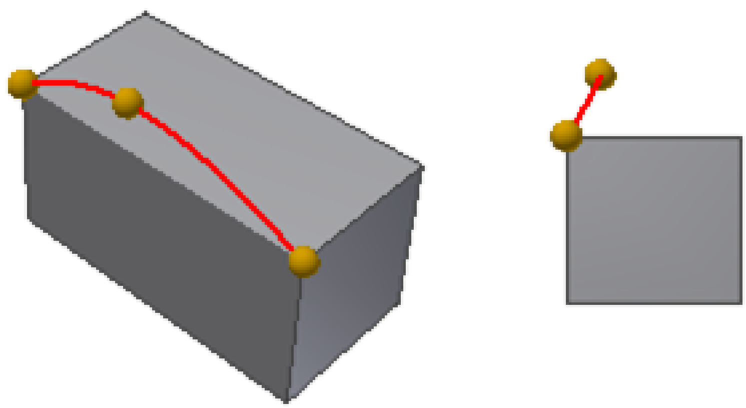
When you edit a specific edge, you can edit it in Interpolation Point mode or Shape Point mode.

**Point Mode**

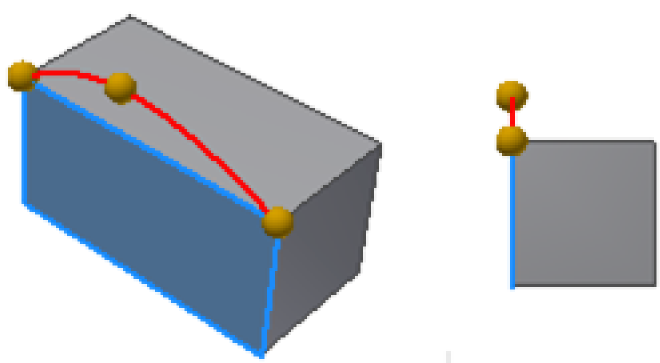
You can constrain your edits so they are parallel to the coordinate plane, parallel to a selected plane, or parallel to the screen.

### Constrain Edits Parallel to Plane

By default, edits you make are relative to the screen. In the following example, the edge edit appears to be planar, but when you look at the side view of the form, you see that it is not.



You can constrain your edits so they are parallel to a selected plane.



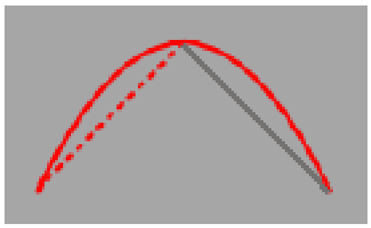
**Merge Option**

As you select the combine or blend merge option, Merge Edges is automatically selected. You can also select Merge Edges, and then select the merge option.

The two types of continuity on the Merge Option panel are:

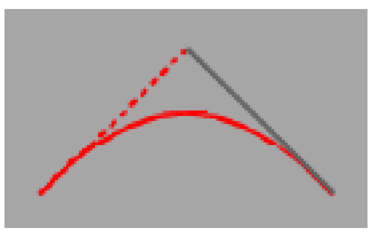
- Combine

For Combine option, the selected edges combine to become one continuous edge, with the point where the two original edges join remaining fixed.



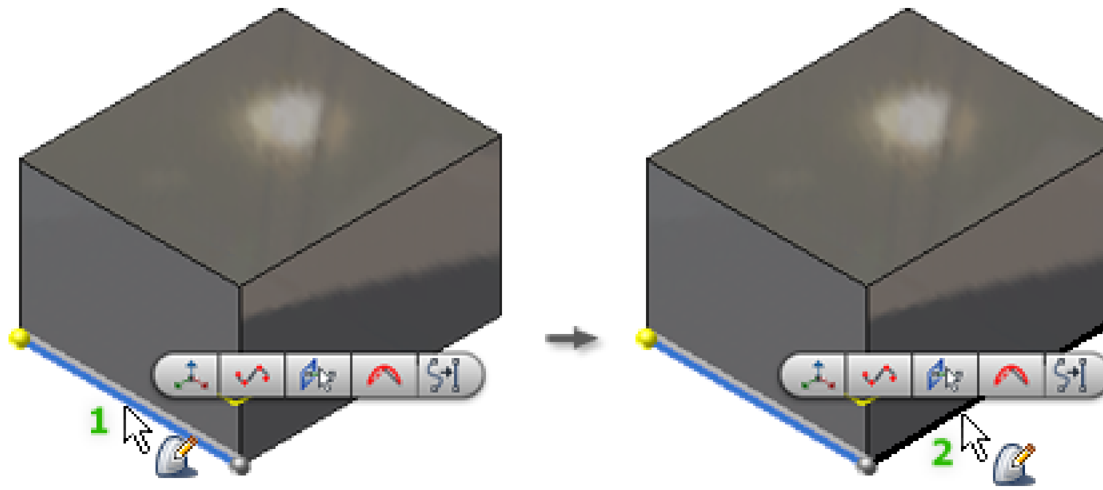
- Blend

For Blend option, the selected edges blend to become one continuous edge, with the point where the two original edges join not remaining fixed.

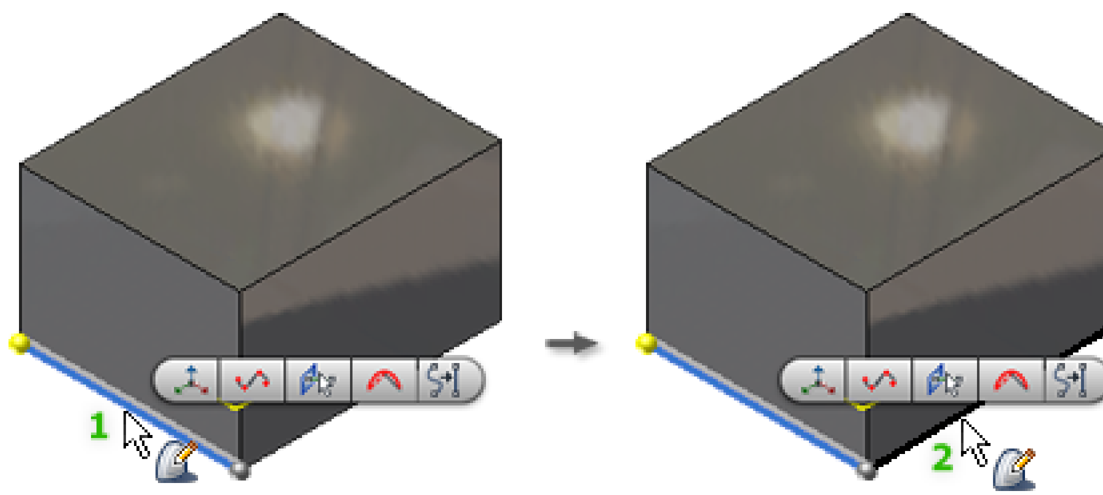


**To merge edge**

- Combine Option




■ Blend Option




Use one of the three workflow below to merge edges.

■ Workflow (1)

1 Select Edit Edge.
2 Click Merge Edges.

3 Select the Merge Option - Combine or Blend.
4 Select an edge.
5 Select another edge.

- Workflow (2)

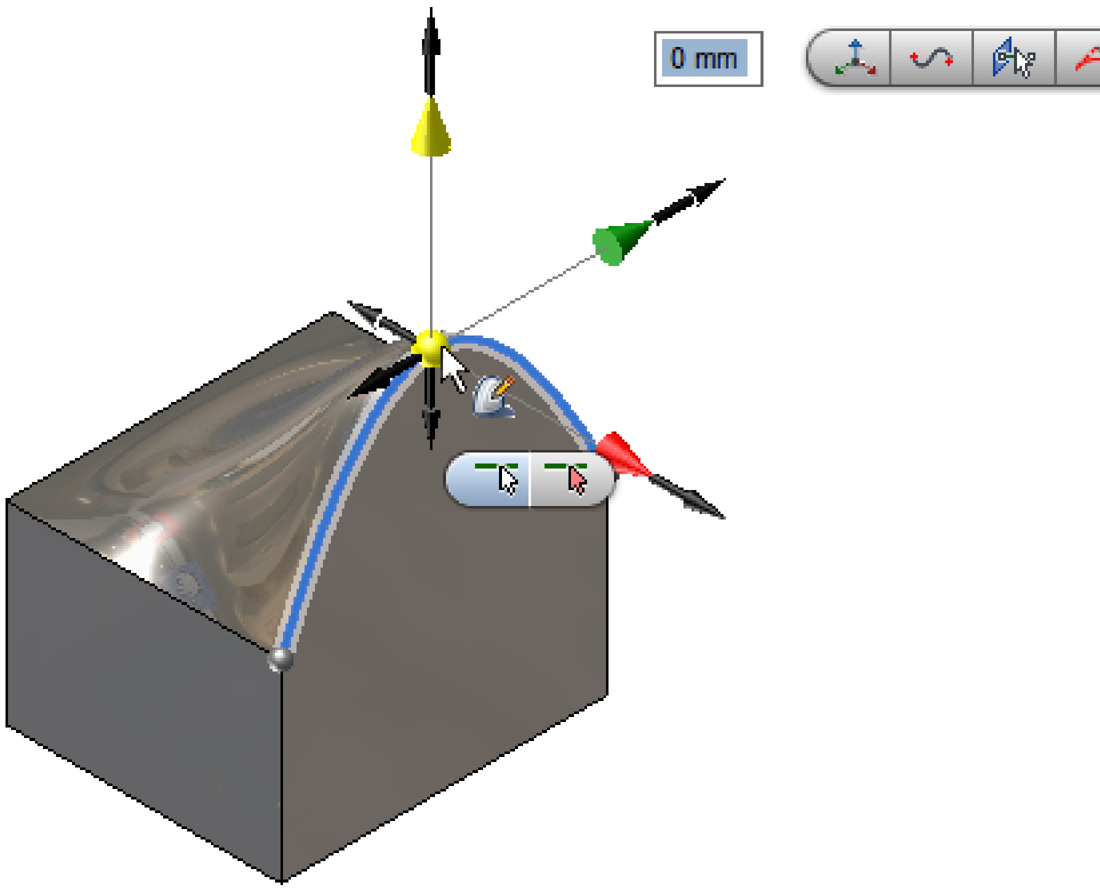
1 Select Edit Edge.
2 Select an edge.
3 Click Merge Edges.
4 Select the Merge Option - Combine or Blend.
5 Select another edge.

- Workflow (3)

1 Select Edit Edge.
2 Select the Merge Option - Combine or Blend. Notice the Merge Edges command is automatically activated.
3 Select an edge.
4 Select another edge.

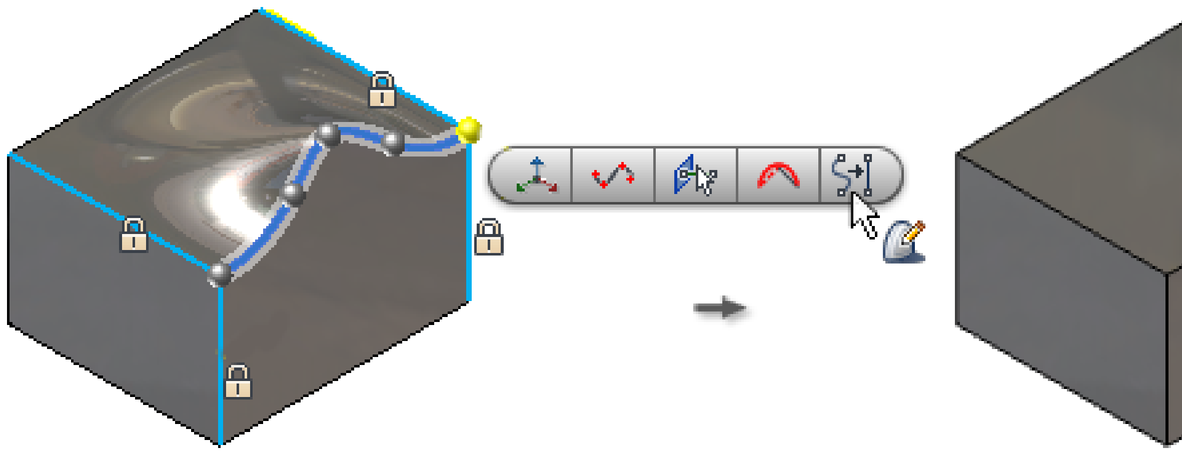
### Triad Switch

Instead of dragging on points, turn on the Triad Switch and use the triad to reshape the edge. By selecting the appropriate element of the triad, you can drag the point along the X, Y or Z axis with a defined distance.




**Straighten**

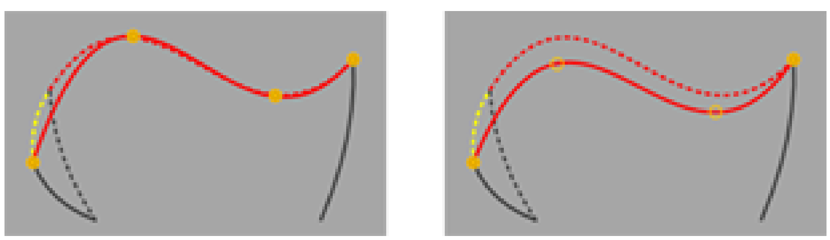
Use Straighten command to select the edge you want to straighten. The selected edge becomes a straight line connecting the start and end points of the edge.



### Extend or Shorten Edges

Extend or shorten edges by dragging the end points. When you extend an edge in the direction of the edge tangent, a yellow guideline appears.

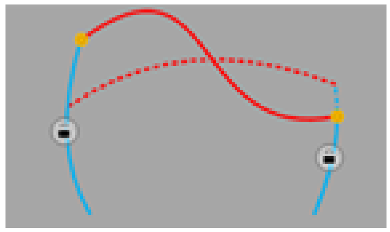
- If the edge is a line, and you drag the point along the guideline, the edge extends and shortens following the line.
- If the edge is an arc, and you drag the point along the guideline, the edge extends and shortens following the arc.
- If the edge is a freeform curve, and you drag the point along the guideline, the edge extends following an arc radius of the point and shortens following the freeform curve.



### Set Editing Freedom

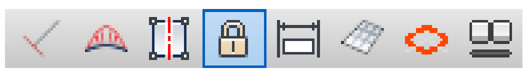
You can fix (or hold) the start and end points of edges in space so they do not move while you freeform edit. For greater editing control, you can fix individual edges.

By default, all edges are fixed except the currently edited edge.

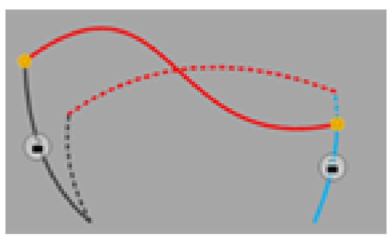


When you edit the endpoints of an edge, connected edges are extended or shortened, but maintain their shape.

You can explicitly toggle the edge status between Hold  and Free  by clicking the edge lock icon at the status bar.



When the lock is open, edits to the selected edge can affect the edge.



### Edit with Symmetry Attributes

While editing with the Edit Edge command, you can edit with Symmetry On or with Symmetry Off. Use the Assign Symmetry command to control symmetry properties. While in the Edit Edge command, use the Symmetry Mode toggle to turn symmetry on/off.

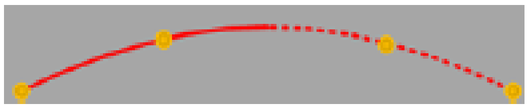
When you edit with Symmetry On, edits you make to the selected edge are also made to the symmetrical edge partner based on the symmetry attributes set for the body.

**Self-symmetric edge**

A self symmetric edge is an edge that passes through the defined symmetry plane.

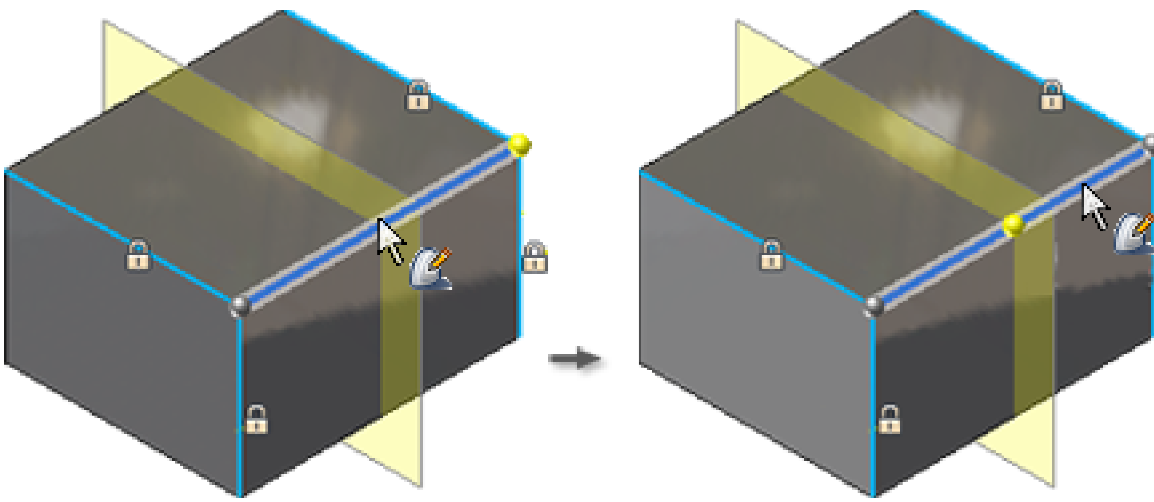
When you select an edge with the **Self Symmetric Edge** condition set for the body, points display on both halves of the edge.

When you edit one half of the selected edge, the other half edits automatically.



When you edit a point on the symmetry plane, you can move it only on the symmetry plane. You cannot pull it off the symmetry plane.

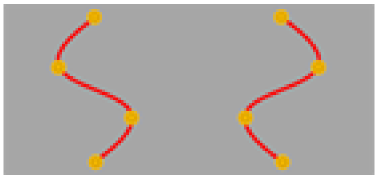
If you add a point to the selected edge, a new point is also added to the self symmetrical partner edge.



**Twin-symmetric edge**

Symmetric twin edges are edges that are "mirrored" through the defined symmetry plane.

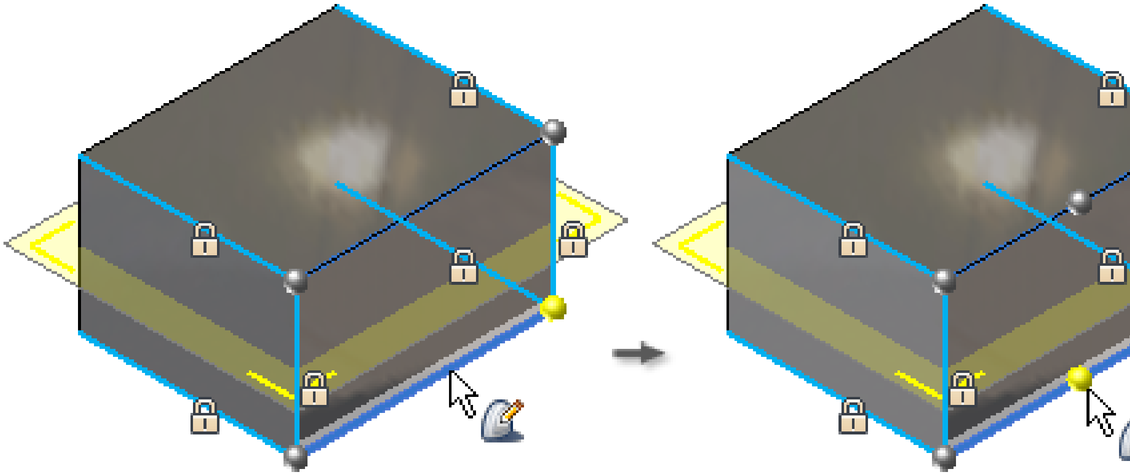
When you select an edge with the **Symmetric Twin Edge** condition set, the symmetrical partner edge is selected automatically. Points display on both the selected edge and the partner edge.



When you edit the selected edge, the partner edge edits automatically.

If the point you are editing is on the symmetry plane, the partner edges share the point. When you edit the point, you can move it only on the symmetry plane. You cannot pull it off the symmetry plane.

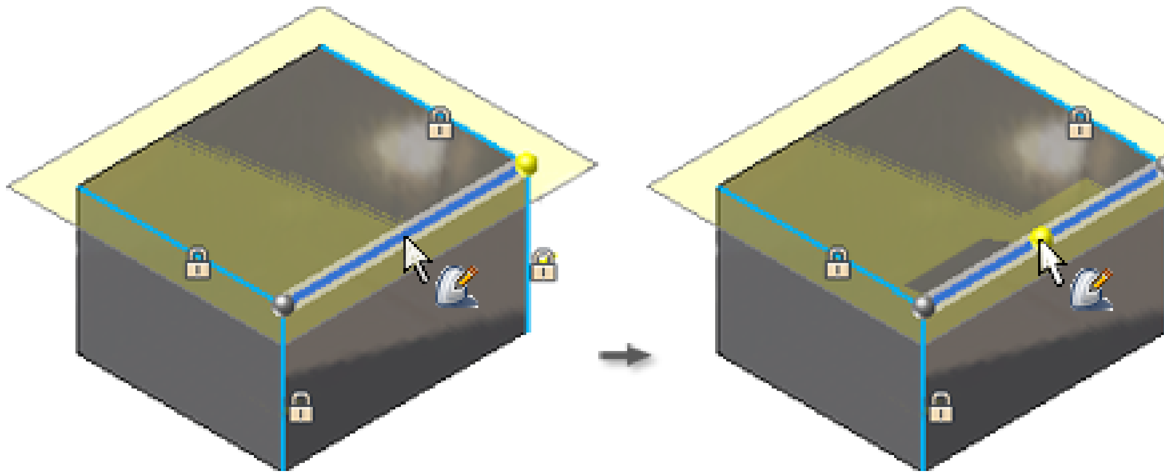
If you add a point to the selected edge, a new point is also added to the twin symmetrical partner edge.



### Edge on symmetry plane

An edge on symmetry plane is an edge that lies on the symmetry plane.

During editing, all points (start, end, and intermediate) can move only on the symmetry plane.



**Show me how to edit edges on a component with symmetry attributes**

## Assign Symmetry

The **Assign Symmetry**  command sets symmetry attributes for both solids and surfaces.

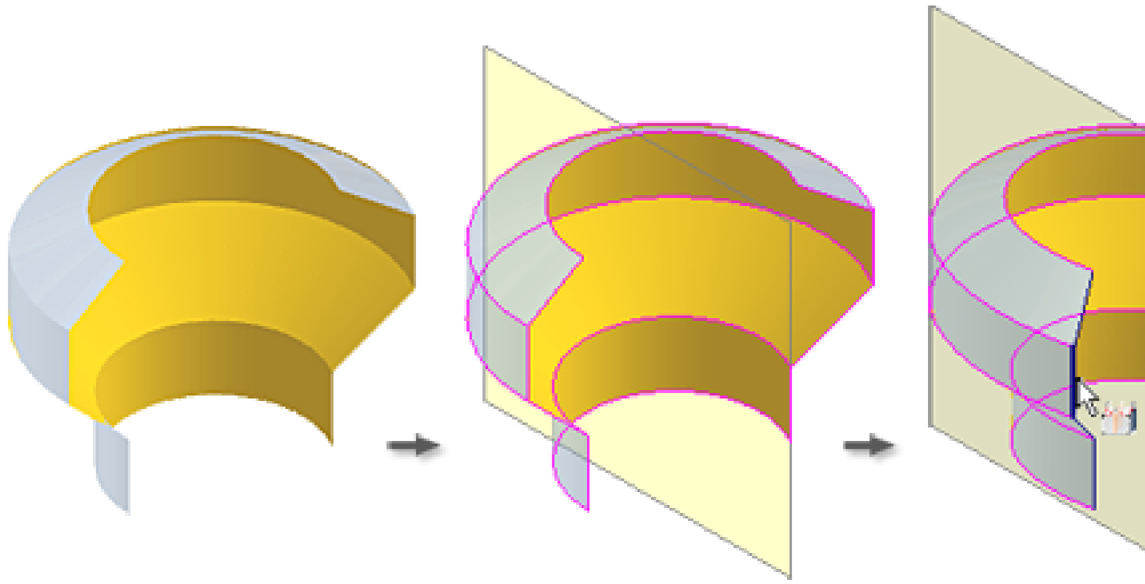
### To establish symmetrical conditions

1. Select Assign Symmetry. If there is a solid or surface at the top level of the document, temporary planes are displayed at the middle of the solid or surface.
2. Select a symmetry plane. Inventor Fusion detects symmetry attributes automatically.
3. Select a symmetry option.
4. Select the edges.

When you edit an edge of the body with Symmetry On, the symmetrical counterpart edits automatically.

### Form Symmetry

All edges of the form automatically recognize their symmetrical counterparts. Detected symmetry conditions are indicated with color. When you freeform edit any of the edges, the symmetrical counterpart updates automatically.



You can use Form Symmetry to start with a basic geometric form and edit edges to sculpt the form to a complex freeform.

### Symmetry Plane

Define the symmetry plane used to detect or set symmetrical partner edges.

The plane can be a coordinate system plane, a temporary plane, a flat face on the body, or you can define it to be a plane that passes through the center of a selected edge and perpendicular to it.

You can select the plane in the modeling window or in the browser.

---

**NOTE** To select another plane, click Plane on the Plane Selection panel.

---

### Target

If there are multiple components in the document or the model has multiple bodies, use Target to select the other component.

---

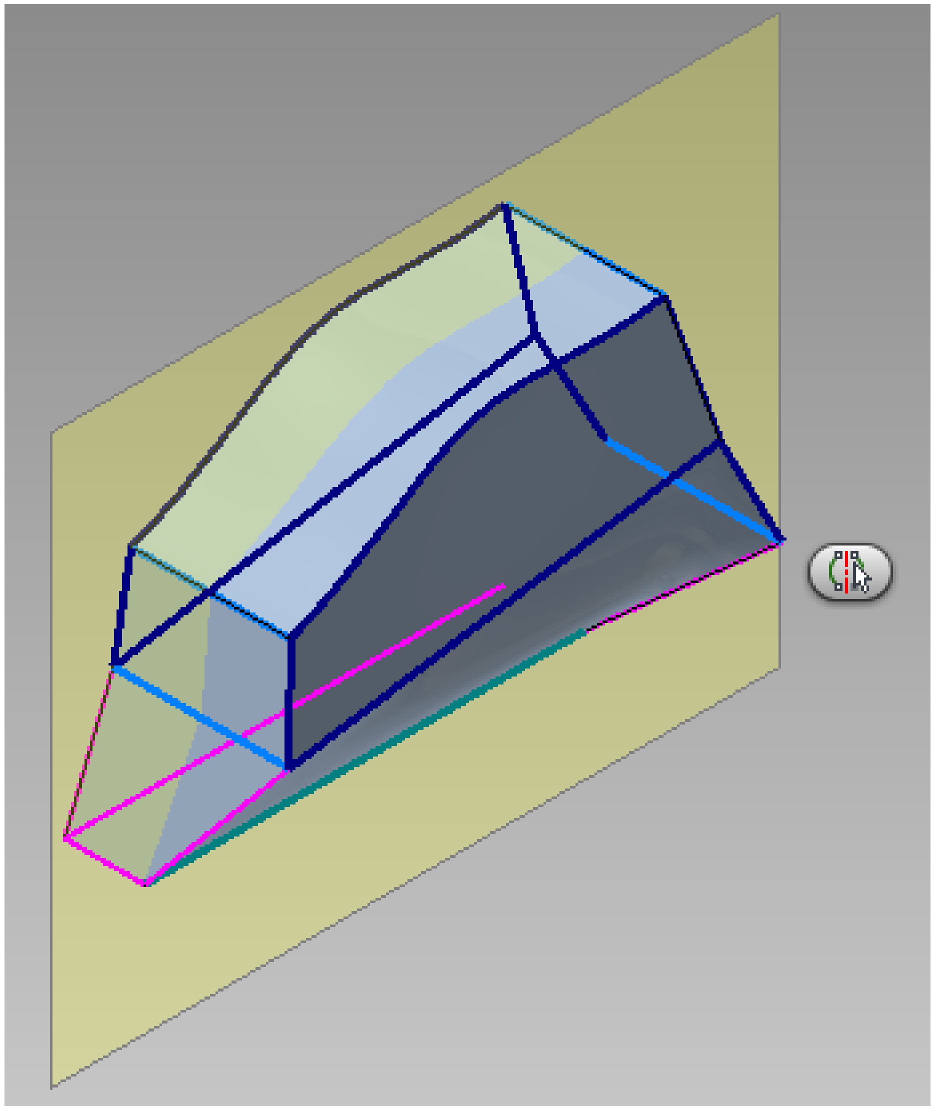
**NOTE**

Inventor Fusion remembers and displays the previous selected plane on the selected component. Select Plane to display the temporary planes of the target.

---

### Symmetry Conditions

Symmetry conditions are automatically detected with Form Symmetry after you select a component. Detected symmetry conditions are indicated with color.



**Self-symmetric Edge**

A self symmetric edge is an edge that passes through the defined symmetry plane.

Edges that have this relationship display with the light blue color.

**Twin-symmetric Edge**

Symmetric twin edges are edges that are "mirrored" through the defined symmetry plane.

Edges that have this relationship display with the dark blue color.

---

**NOTE** When you select edges, the first selection remains stationary, and the second selected edge adjusts to be symmetrical.

---

### Edge on Plane

An edge on plane is an edge that lies on the symmetry plane.

Edges that have this relationship display with the light green/blue color.

### No Identified Symmetry Condition

Edges that have no identified symmetry conditions display with the magenta color.

**Show me how to assign symmetry to a component**

## Edge Evaluation

Use the Edge Evaluation command to display the curvature comb for selected edges.

### Options

Display mode controls how the curvature is represented in the graphics window.

Display min-max radius controls the display of the minimum and maximum radius of the selected edge.

Density sets the number of combs displayed.

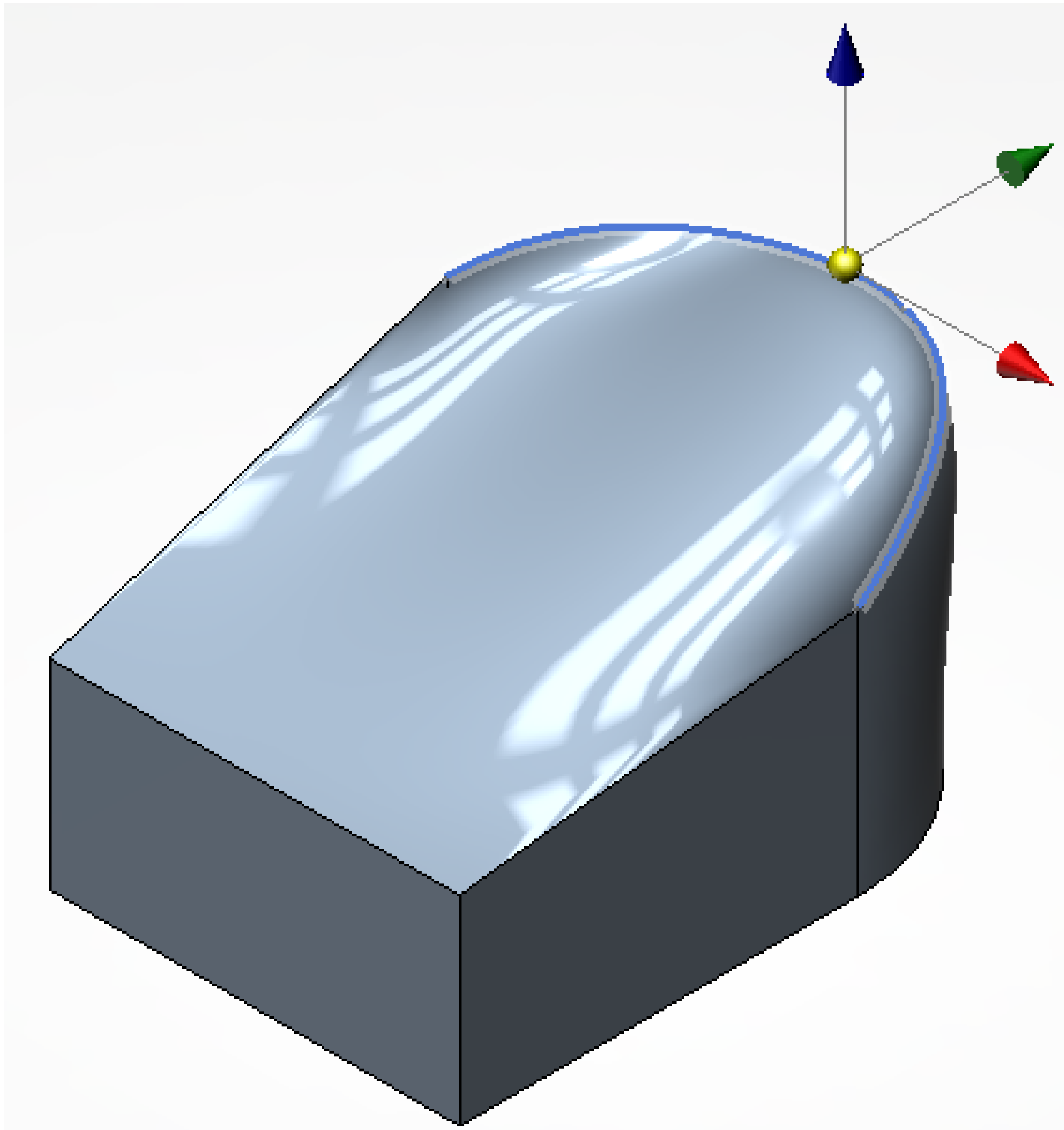
Scale sets the scale of the curvature comb.

**Display the curvature plot of edges for evaluation**

**Show me how to display the curvature plot of edges for evaluation**

## Move Edge

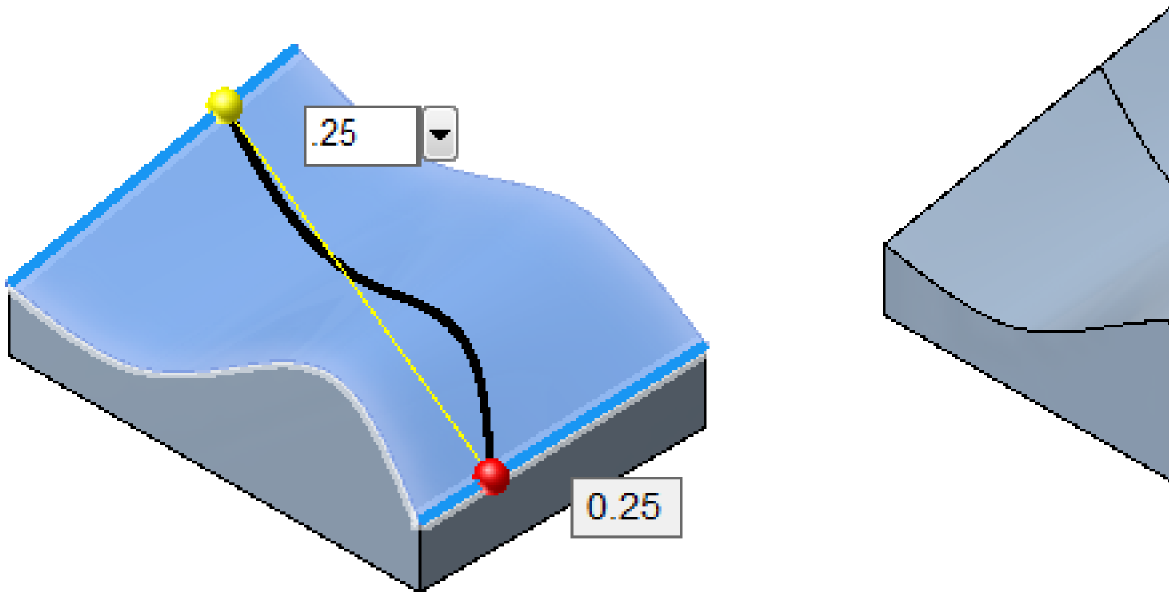
Use the Move command to move edges. When you move an edge, the adjacent faces are refit to accommodate the move.



**Show me how to move an edge**

This video shows how to move model edges and the affect on geometry.

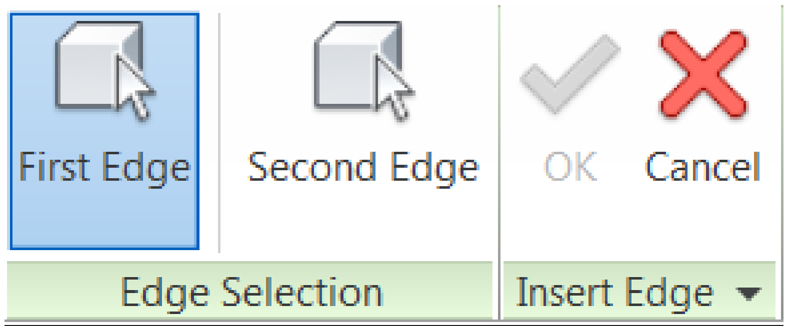
## Insert Edge




Insert Edge adds an edge to a face or surface by splitting the face between two points. You select points on two edges of a face, and a line between those points is projected to split the face and create the edge.

You can adjust the location of a point by dragging the manipulator or entering a value between 0.0 and 1.0.

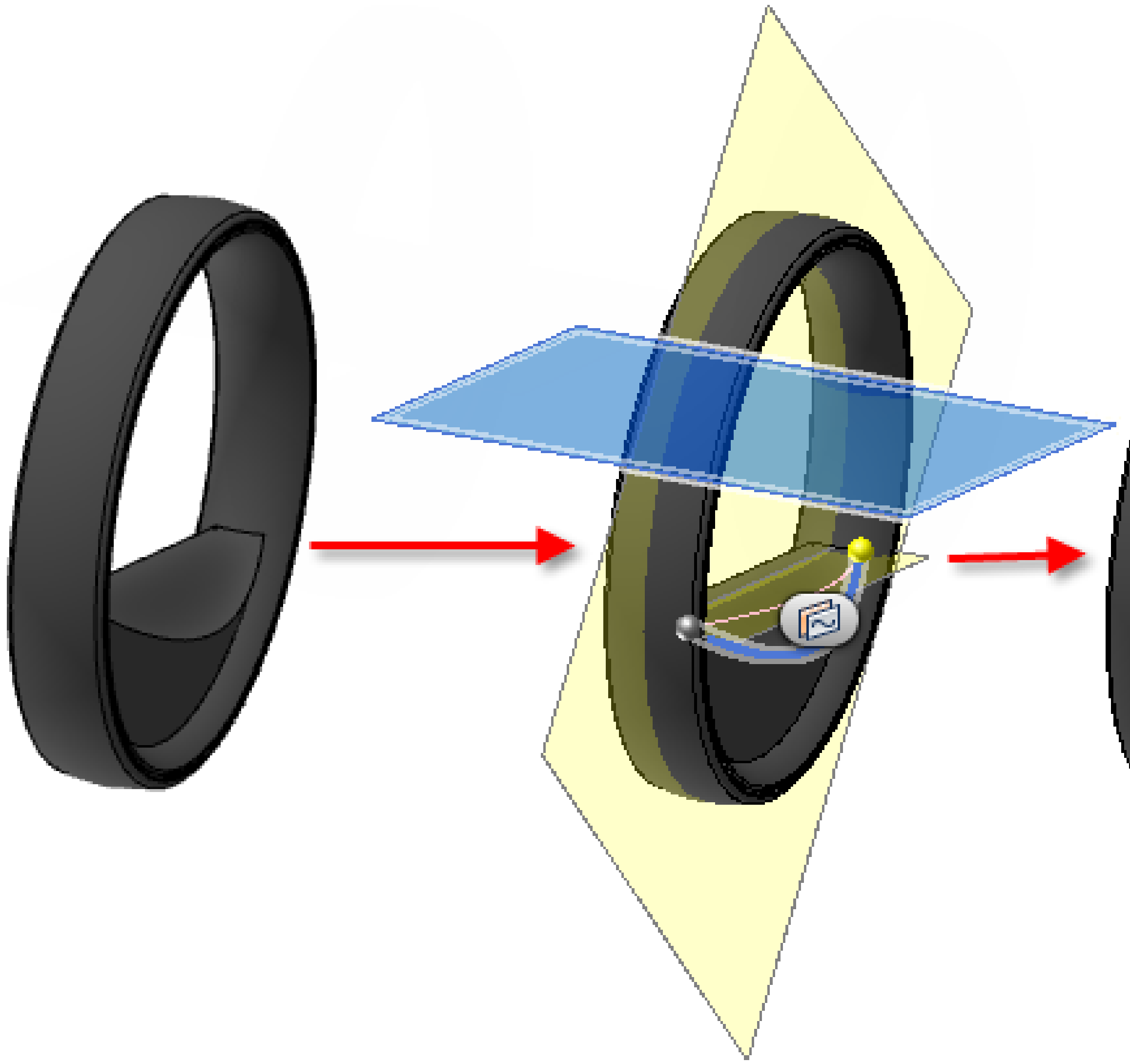
**Insert an edge**




1. On the Edit Form panel, start the Insert Edge command.
2. Click on two edges of a face to define the endpoints of the new edge. A preview displays of the line connecting the points and the projected edge on the face.
3. Drag the manipulators, or enter values between 0.0 and 1.0 to set the locations of the end points.
4. Click OK to create the edge and split the face.

**Show me how to insert an edge onto the curved face**

## Planarize Edge

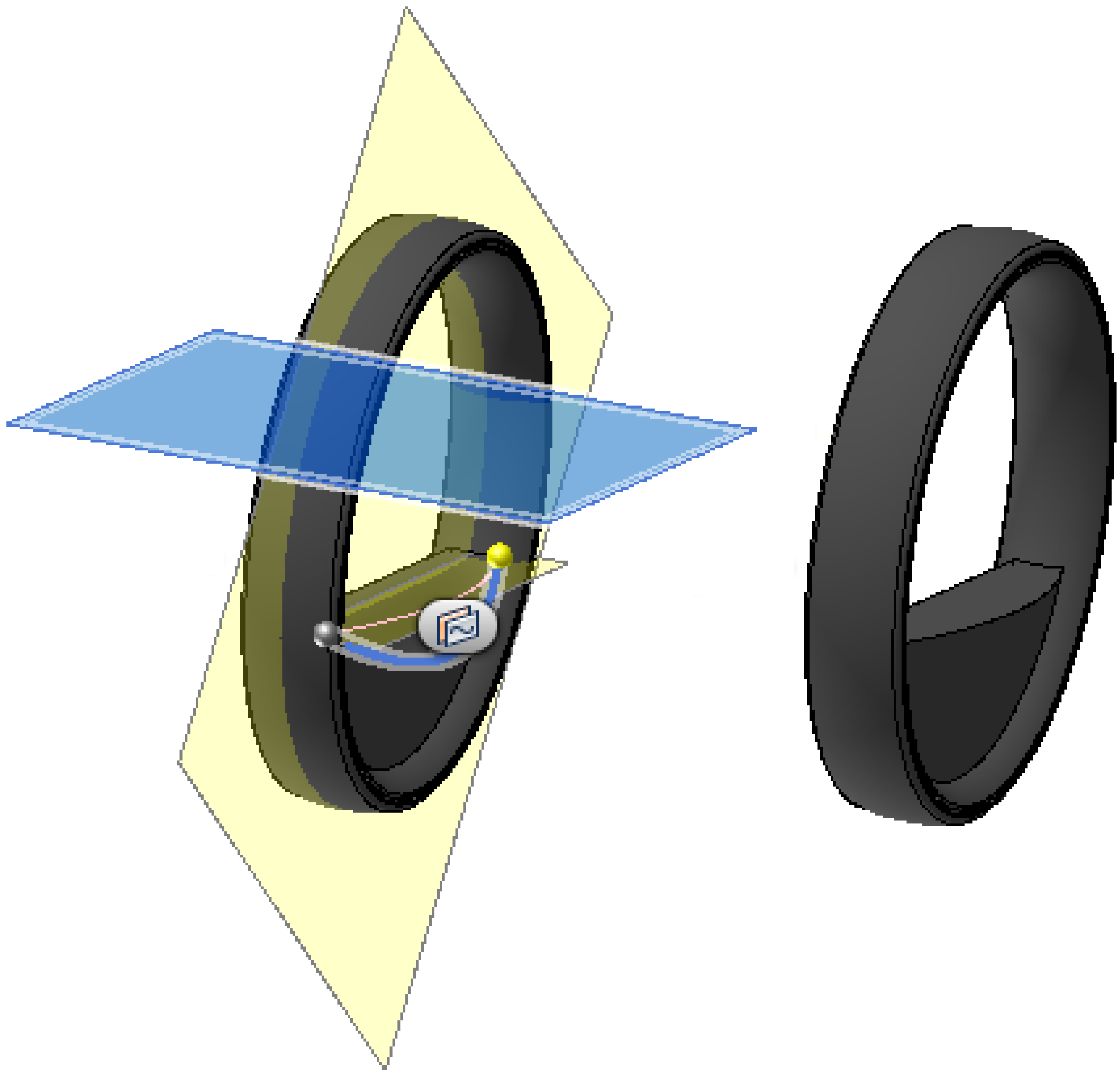


The Planarize Edge command modifies an edge to make it parallel to a plane.

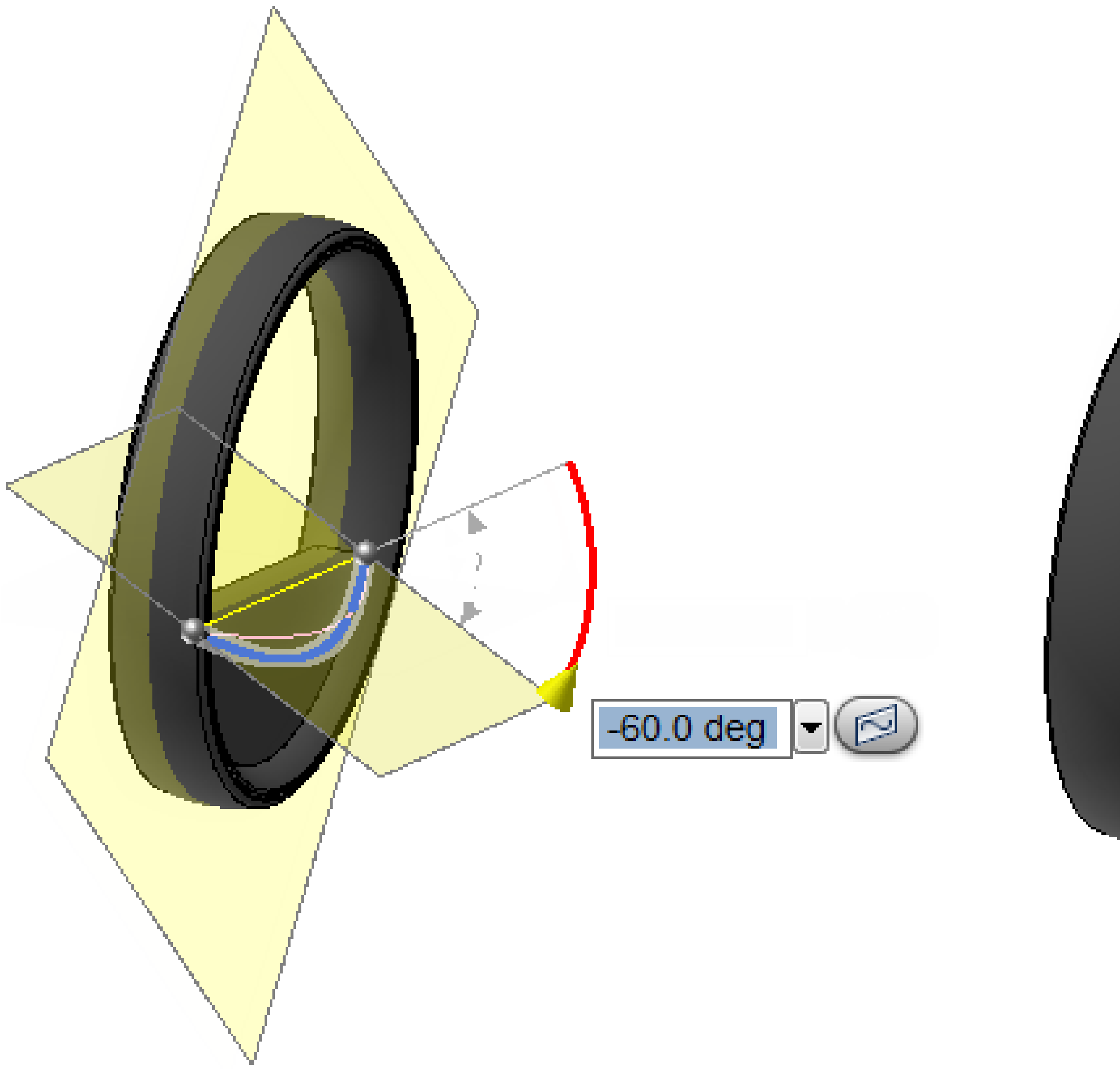
You can select an existing work plane, a planar surface, or a planar model face. You can also define a plane during the command by selecting the end points of the edge and specifying the angle.

### Planarize an edge

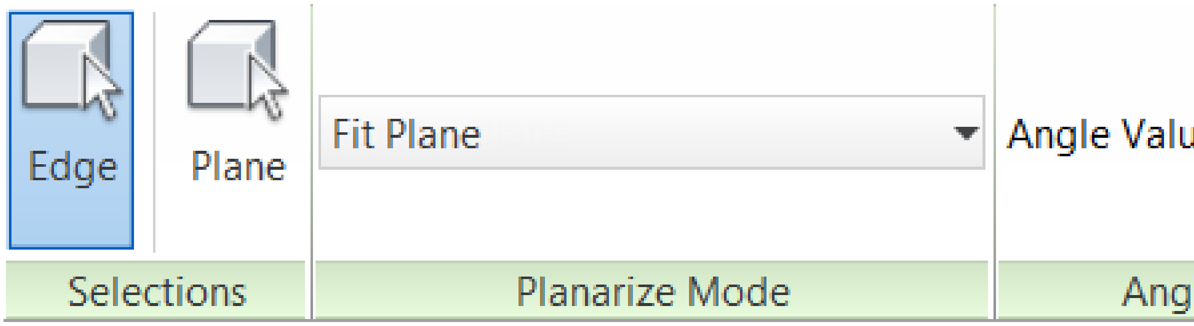
The edge is projected to the plane, and the curvature of the adjacent faces is updated.



The selected plane must be parallel to a line connecting the end points of the edge. If the parallel plane is offset from the edge, a temporary work plane is constructed.




The fit plane is coincident with the endpoints of the edge, and the angle of rotation defines the position.




1. On the Form Edit panel, click Planarize Edge.
2. In the graphics area, select the model edge.
3. Set the Planarize Mode to Parallel Plane or Fit Plane.
   - **Parallel plane** Select a work plane, a planar face, or a planar surface that is parallel to the end points of the edge. If the plane is offset from the edge, a temporary work plane is constructed.
   - **Fit Plane** A temporary work plane displays that is coincident to the end points of the edge. Enter an angle to define the work plane.
4. Click OK. The edge projects to the plane, and the curvature of the adjacent surfaces updates.

## Mirror and Pattern

Mirrors and Patterns create copies of faces, features, solids, and components. You can make copies that are symmetrical about a plane, arranged in a rectangular or circular pattern, or patterned along a path.

Mirrored and patterned objects are linked, so changes to one instance update the other instances. You can break the link between objects by right-clicking on the feature in the browser and selecting Dissolve. This action deletes the feature, but the objects remain.

### Mirror

Mirror creates and maintains symmetrical features, solids, and components. The objects can be mirrored about a work plane or planar face.

### Mirror objects

1. Start the Mirror command.
2. In the graphics area, select the objects to mirror.
3. On the command ribbon, click Plane, or click the Select Mirror Plane glyph.
4. Click OK to mirror the object.

### Patterns

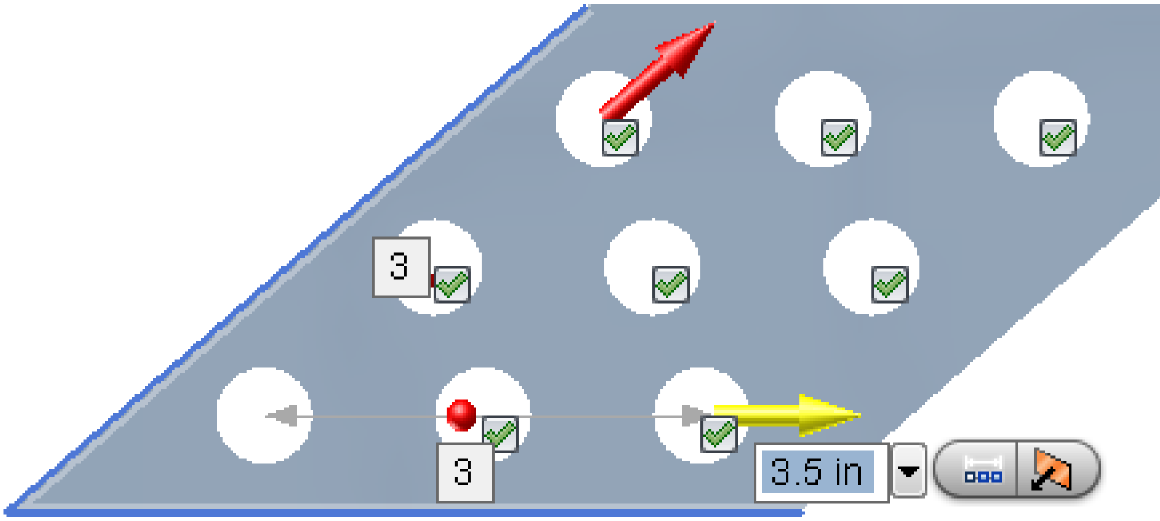
You can create rectangular or circular patterns, or a pattern along a path.

Patterns can be edited. In the browser, right click the pattern, and click Edit. The command ribbon and HUD display.

### Create rectangular patterns

Rectangular patterns create copies of objects in one or two directions. When you select the direction, two perpendicular manipulators display. You can select a second edge to change the angle of the second direction. To create the pattern in one direction, set the number of occurrences to 1.

1. Start the rectangular pattern command.
2. In the graphics area, select the objects to pattern.
3. Click Direction, and select a linear edge or work axis. Perpendicular manipulators display.
4. For a non-perpendicular pattern, select a second linear edge or work axis.
5. Use the ball manipulator to set the number of occurrences.
6. Click a check mark to suppress an occurrence.
7. Set the pattern type to Extent or Spacing.
   - For Extent, drag the arrow manipulator to set the total distance. The occurrences are evenly spaced.
   - For Spacing, drag the arrow manipulator to set the distance from one occurrence to the next.




Selecting two edges of this part creates a non-perpendicular pattern.

**Show me how to create a rectangular pattern**

### Create circular patterns

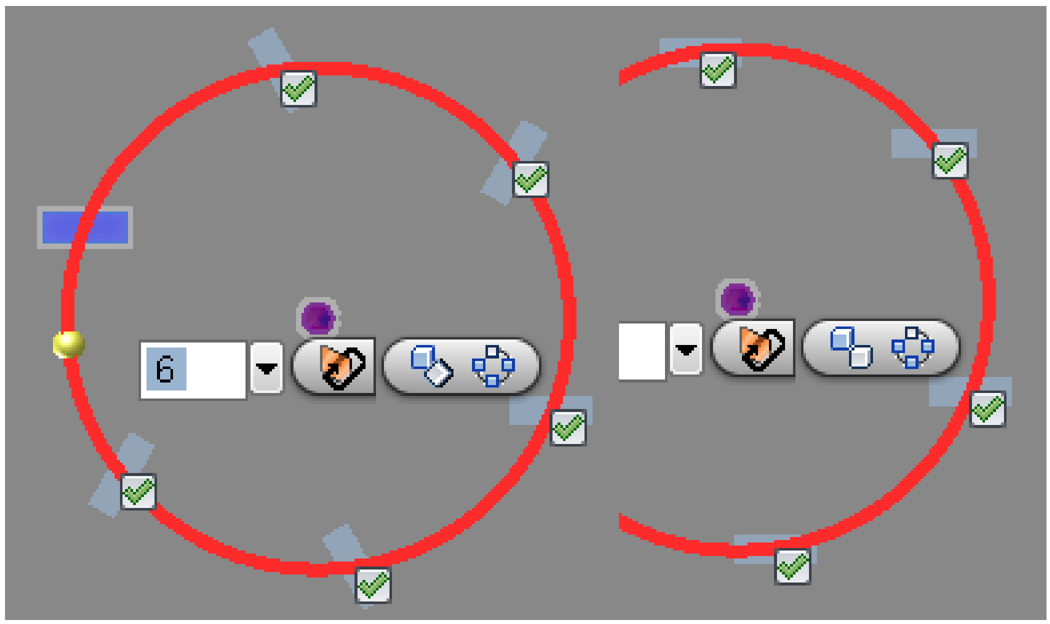
The axis for a circular pattern can be a sketch line, a linear edge, a work axis, a cylindrical face, or a circular edge.

1. Start the rectangular pattern command.
2. In the graphics area, select the objects.
3. Click Center and select a sketch line, linear edge, a work axis, a cylindrical face, or a circular edge.
4. Use the ball manipulator to set the number of occurrences. The ball manipulator displays only up to eight instances, but you can enter a larger value.
5. Click a check mark to suppress an occurrence.
6. Set the pattern type to Full, Angle, or Symmetric.
   - **Full** The copies are evenly spaced.
   - **Angle** Drag the arrow manipulator to set the total angle.

- **Symmetrical**Drag the arrow manipulator to set the total angle. You cannot have an even number of copies for a symmetrical pattern.

7 Set the pattern type to Rotate or Identical.

- **Rotate** Changes the orientation of each copy to match its position.
- **Identical** Keeps the orientation of the original object.

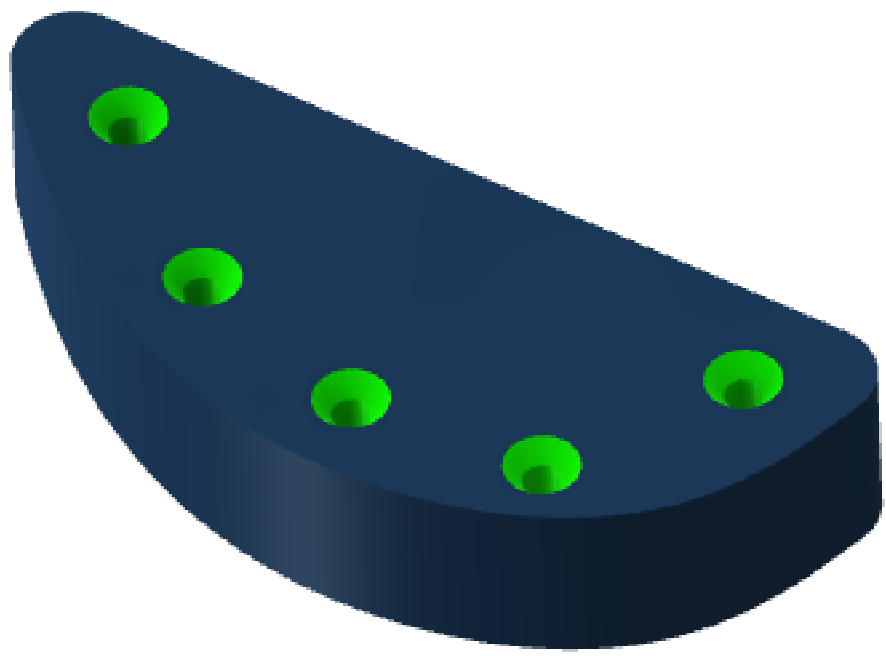



Rotate orientation

Identical orientation

Show me how to create a circular pattern

**Create a pattern along a path**



Path patterns create copies of objects by following the selected edge or sketch line.

1 Start the Path Pattern command.
2 In the graphics area, select the objects to pattern.
3 Click Direction, and select sketch entities or edges. Tangent curves and edges are automatically selected.
4 Set the pattern type to Extent or Spacing.
   - **Extent** Drag the arrow manipulator to set the total distance. The occurrences are evenly spaced.

- **Spacing** Drag the arrow manipulator to set the distance from one occurrence to the next.

5 Drag the first ball manipulator to set the starting point for the pattern.

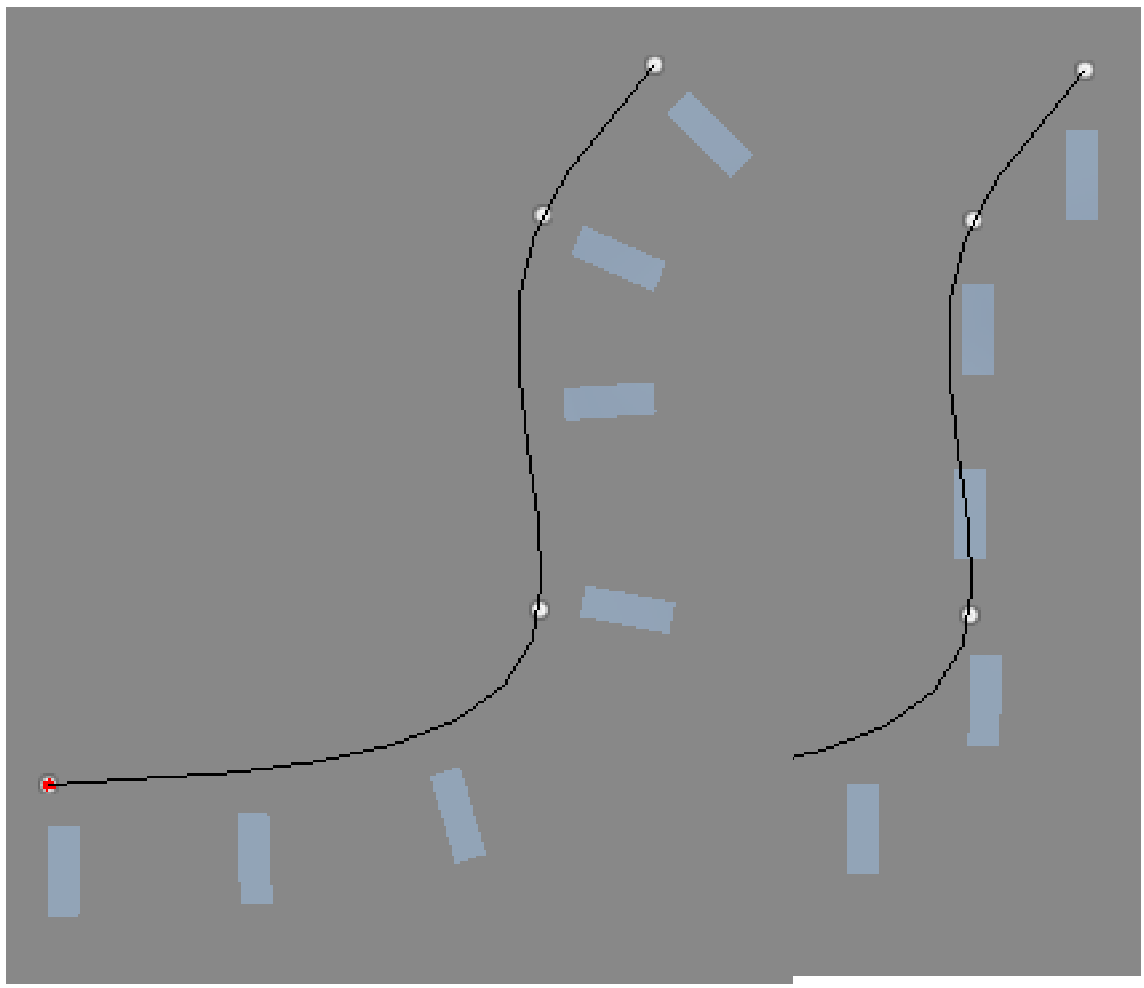
6 Drag the second ball manipulator to set the number of occurrences.

7 Click a check mark to suppress an occurrence.

8 Set the pattern Type to Path Direction or Identical

   a **Path Direction** Changes the orientation of each copy to match its position along the path.

**b** **Identical** Keeps the orientation of the original object.



Path Direction orientation Identical orientation

**Show me how to create a pattern along a path**

# Cut, Copy, and Paste

You can copy components, solids, surfaces, and features. You can paste surfaces and solid features into the graphics window, and solids or components into the browser or graphics window. You can also paste objects into other documents.

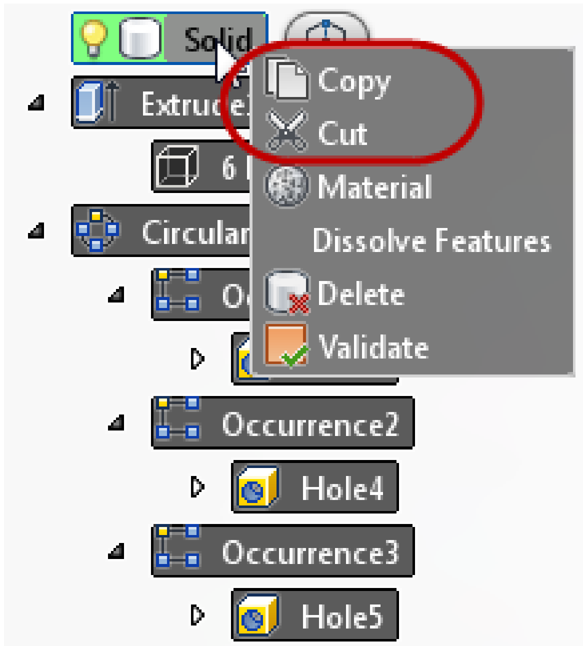
You can cut, copy, and paste:

- One or more pieces of sketch geometries.
- One or more body objects.
- One or more components (component instances).
- Face sets.

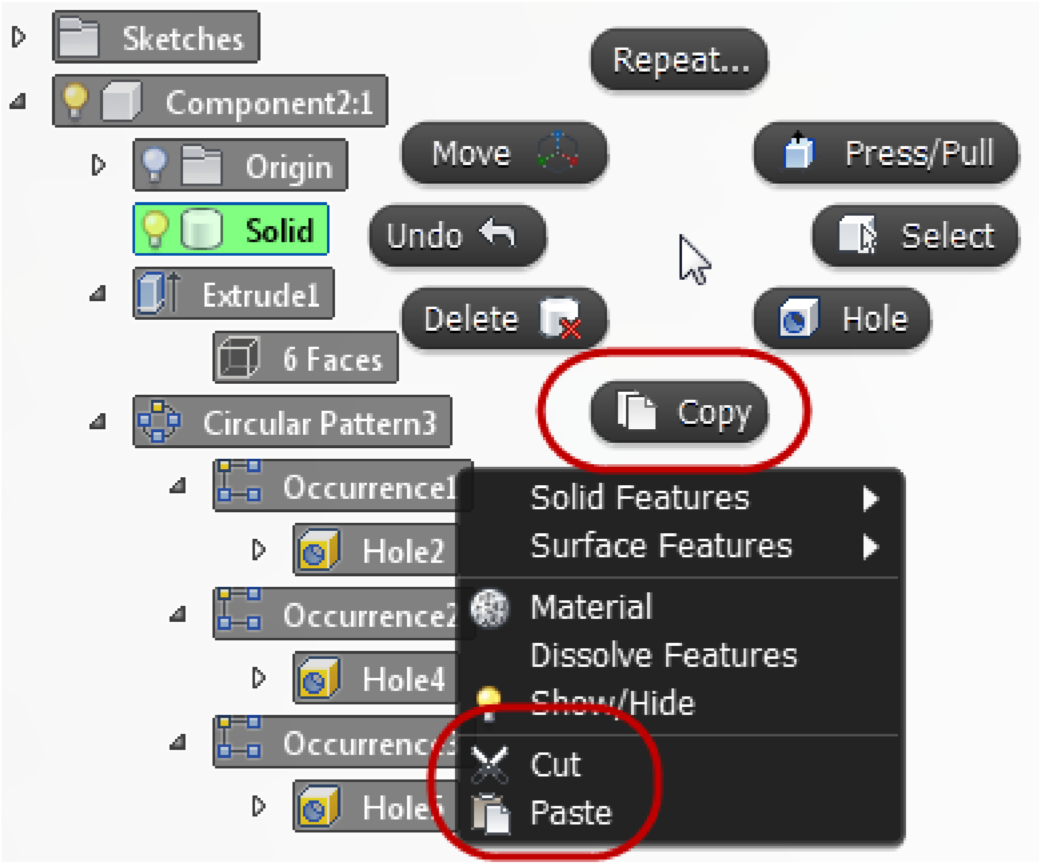
### Cut and copy interface

You can select objects from the browser, the graphics window, or the ribbon. The cut and copy commands are available from:

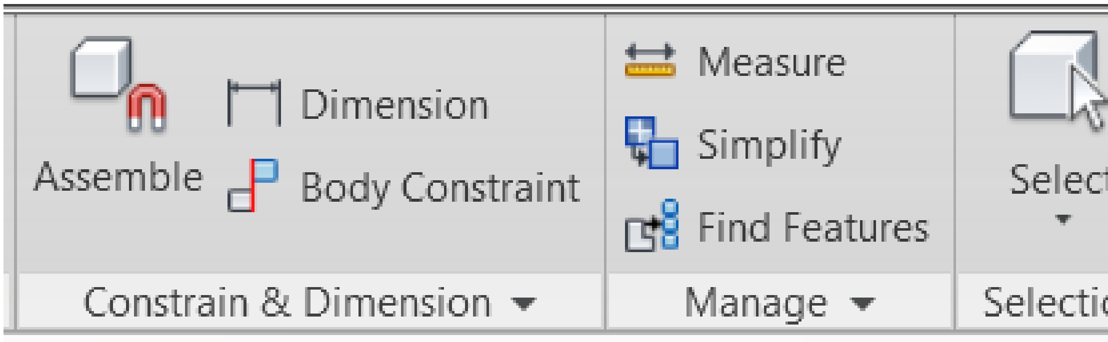
- Browser: right-click a browser node.




- Marking Menu: right-click open space.




- Ribbon



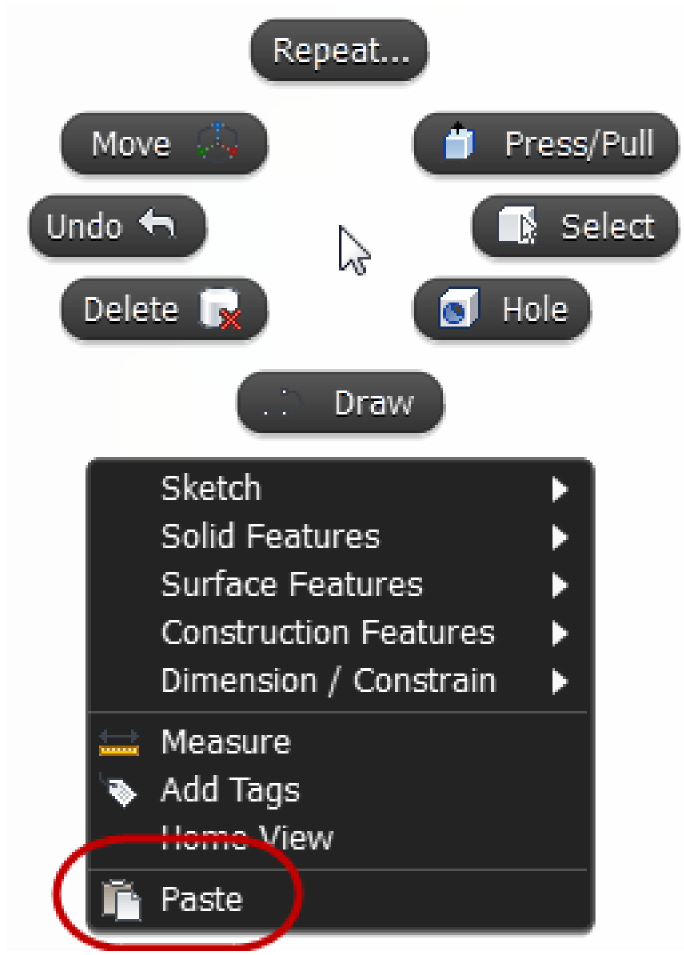

- Key strokes

This functionality is mapped to the key sequences `CTRL+X` and `CTRL+C`.

**Note:** If you use cut, the actual cut operation does not happen until the paste command is invoked. Every time a cut and copy command is used, the previously cut and copied objects are cleared from the clipboard.

### Paste interface

The paste command pastes previously cut and copied objects from the clipboard. Paste only displays on the context menu for valid targets. For example, Paste only displays for a solid if the target component does not contain a solid.




### Solid Features

Solid features can be copied from the browser or graphics window, and pasted into the graphics window.

**Copy solid features**

- Select a solid in the browser, or in the graphics window by clicking on a face and choosing the Parent from the Selection Commands_TOPIC_12 menu.
- Copy from the browser context menu, the graphics window context menu, or the ribbon.

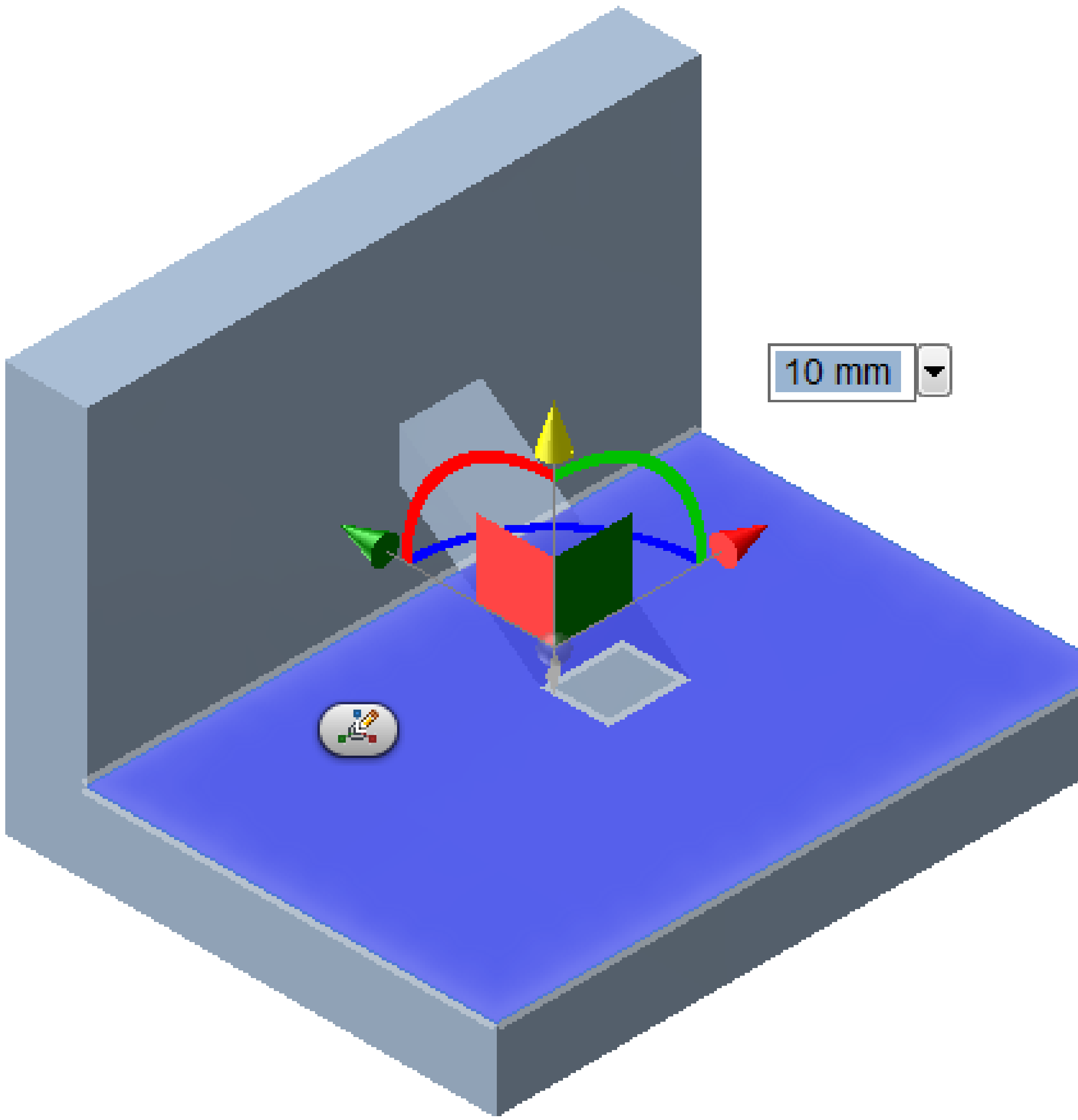
- Copy from active or inactive components.
- Hole is the only placed feature that can be copied. See the Placed Features-Solid (page 193) topic for more information.
- Multiple features can be copied. You can select the features from different components.

**Paste solid features**

- Select Paste from the graphics window context menu and ribbon.
- Select a face on a solid for pasting a dependent feature.
- A feature is pasted into the same component as the selected target face. The component does not have to be active.
- You can select a planar or curved face for the paste target, but it must be larger than the pasted feature.
- You cannot select a work plane or surface for pasting a solid feature.
- Hole is the only placed feature that can be pasted.

**Paste ribs**

Ribs are a hybrid sketch based and placed feature. The rib profile updates based on the selected model faces. This hybrid behavior makes paste more complicated. See the rib video for more information.




1 Select Paste from the graphics window context menu or the ribbon.

2 Click on a face to select the location of the rib. The rib is pasted where you click the face.

3 The rib profile must intersect the adjacent face. If necessary, use the arrow manipulator to move the rib closer to the adjacent face until the preview displays.

4 The X-Y plane is aligned with the selected face. If necessary, use the arc manipulator to rotate the triad so the Y-axis (green arrow) points towards the adjacent face.

5 Use the X-axis (red) arrow manipulator to move the rib along the face.

   **NOTE** Pause the cursor over the value field and select Measure to get the exact distance between the rib preview and a face or edge. Click on the measurement to enter the value, and use an equation to calculate the distance. See the Measure Command_TOPIC_13 (page 54) topic for more information.

---

**NOTE** The rib has the same width and orientation as the original. Use the arrow and arc manipulators to change the width of the rib or the angle of the profile.

---

**Show me how to create, and copy and paste, a rib**

### Surfaces

Individual surfaces, surface features, and quilts can be copied and pasted.

#### Copy surfaces and Quilts

- Select surfaces in the graphics window by clicking on a face and choosing the parent from the Selection Tool (page 38)menu.
- Select a quilt in the graphics window or the quilt folder in the browser.
- Select surface features in the browser or in the graphics window or the ribbon.
- Select individual surfaces in the graphics window.
- Copy a quilt from the browser context menu, the graphics window context menu, or the ribbon.
- Copy a surface feature or an individual surface from the graphics window context menu or the ribbon.

#### Paste surfaces and quilts

- Pasting from the browser context menu places surfaces in the selected component.

- Paste from the graphics window context menu or the ribbon places surfaces in the active component.

### Solids

A component can only have one solid. You can only copy a single solid, and paste into a component that does not have a solid.

#### Copy solids

- Select a solid in the browser, or click a face in the graphics window and choose the parent from the Selection Tool (page 38)menu.
- Copy from the browser context menu, graphics window context menu, or the ribbon.

#### Paste solids

- Paste the solid only in a component that does not have a solid.
- Paste from the browser context menu places the solid in the selected component. The component does not have to be active.
- Paste from the graphics window context menu or the ribbon places the solid in the active component.
- The pasted component is placed in the same location as the original, and the Move triad is active. Use the manipulators to change the component position and orientation.

---

**NOTE** Create an empty component for the copied solid by right-clicking in the browser and selecting New Component.

---

#### Drag and Drop solids

Use Drag and Drop in the browser to move a solid to an empty component.

Select the solid in the browser and drag it to a component. The solid is only moved if the target component does not have a solid.

### Components

Components can be copied from the browser, graphics window, or ribbon. You can paste a component into the browser or graphics window.

---

**NOTE** You can select a component in the graphics window by double-clicking a face.

---

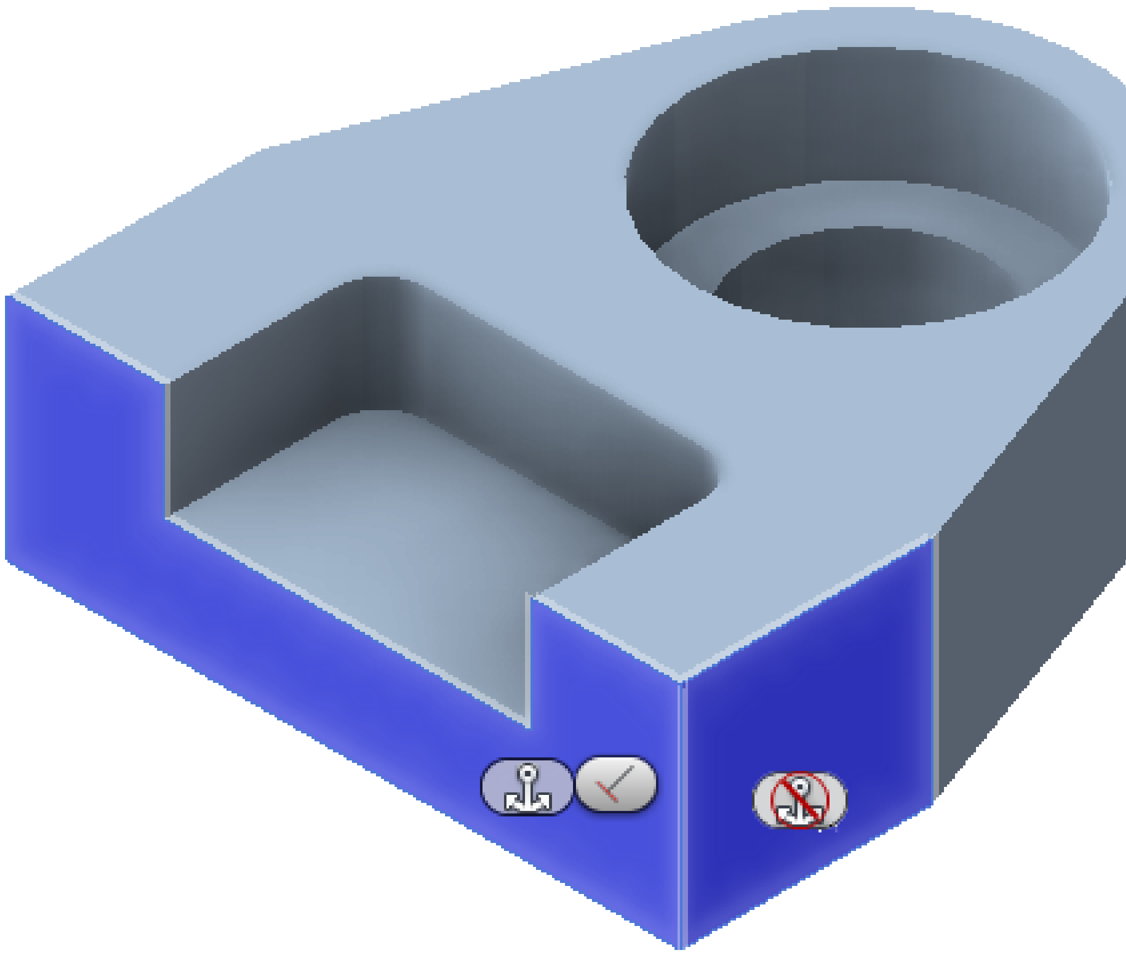
### Paste components

- Paste creates another instance of the original. Both instances are linked, so edits to one updates the other.
- Paste New creates a copy of the original. The original and copy are not linked, so edits to one do not update the other.
- Use Paste and Paste New from the browser to place the component in the selected component.
- Use Paste and Paste New from the graphics window context menu or the ribbon to place the component in the active component.
- The pasted component is placed in the same location as the original, and the Move triad is active. Use the manipulators to change the component position and orientation.

### Make Independent

Use Make Independent from to break the link between instances of a component. Make Independent is available on the browser and graphics window context menus.

# Body Constraints



Inventor Fusion uses direct modeling, which allows you to change model geometry without impacting other features. This behavior makes it easy to create and modify models, but you can also make unintended changes to the model. Define the geometry relationships as the model matures so that you can lock down the shape of the model.

### Overview of body constraints

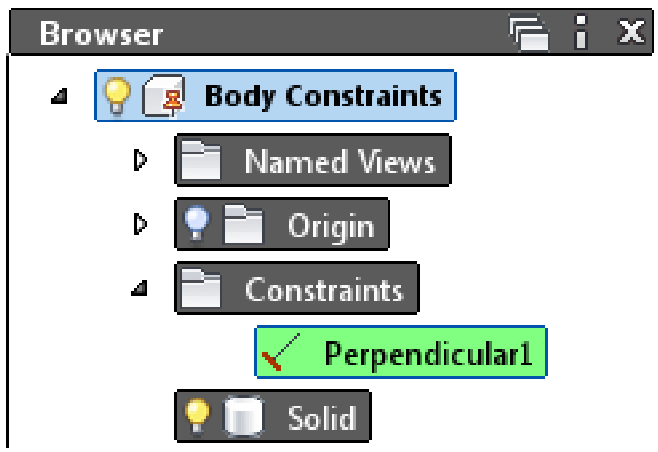
Use body constraints to make planar faces coplanar, parallel, or perpendicular, and to make cylinders concentric. Body constraints create relationships between the faces, but do not control the size or location in the model.

---

**NOTE** Defining the relationships between geometry is also called capturing design intent. For example, a motor has mounting feet that are bolted onto a flat surface. Adding coplanar constraints to the mounting feet captures the design intent for mounting the motor.

---

Body constraints are added to a Constraints folder in the browser. The faces are highlighted when you click on a constraint. You can delete a constraint, but you cannot edit it to change the type or the face selections.




Body constraints are used with model dimensions to define the model and capture relationships. Use body constraints to create relationships between geometry, such as making two cylinders concentric. Use dimensions to locate and define the size of geometry and features, such as the diameter and location of a hole. See the Working With Model Dimensions and Body Constraints (page 348) section for more information.

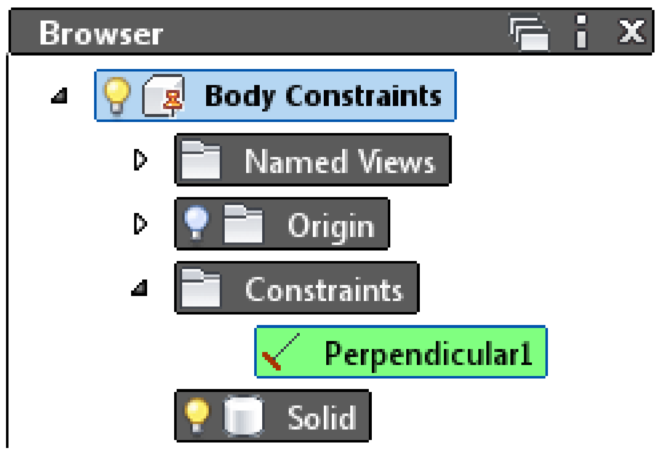
Body constraints create relationships between geometry in a single component. Use assembly constraints to position components and create constraints between them. See the Position and Constrain Components (page 383) topic for more information.

### Create body constraints

Use body constraints to constrain faces and edges within a single component. Body constraints cause the body to maintain it's shape as changes are made. For example, if you make two faces parallel, both faces update if one of them is moved to a different angle.

Body constraints define relationships between model geometry in a single component. You cannot create body constraints between components. Use Assemble to create geometric constraints between components.

Body constraints do not display in the graphics window, but they are added to a Constraints folder in the browser. Click on a constraint to highlight it in the graphics window.




**The following types of Body constraints can be created:**

- Coplanar: Two planar faces are in the same plane
- Center: Two cylindrical faces are aligned along the same axis.
- Parallel: Two planar faces are parallel
- Perpendicular: Two planar faces are perpendicular

1. Start Body Constraint from the Constrain & Dimension panel.
2. Select the Constraint Type. The face selections are filtered based on the constraint type.
    1. Only planar faces can be selected for Coplanar, Parallel, and Perpendicular constraints.

2 Only cylindrical faces can be selected for Center constraints.

3 If desired, change the constraint type before you apply the constraint. The faces are deselected if they are not valid for the new constraint type.

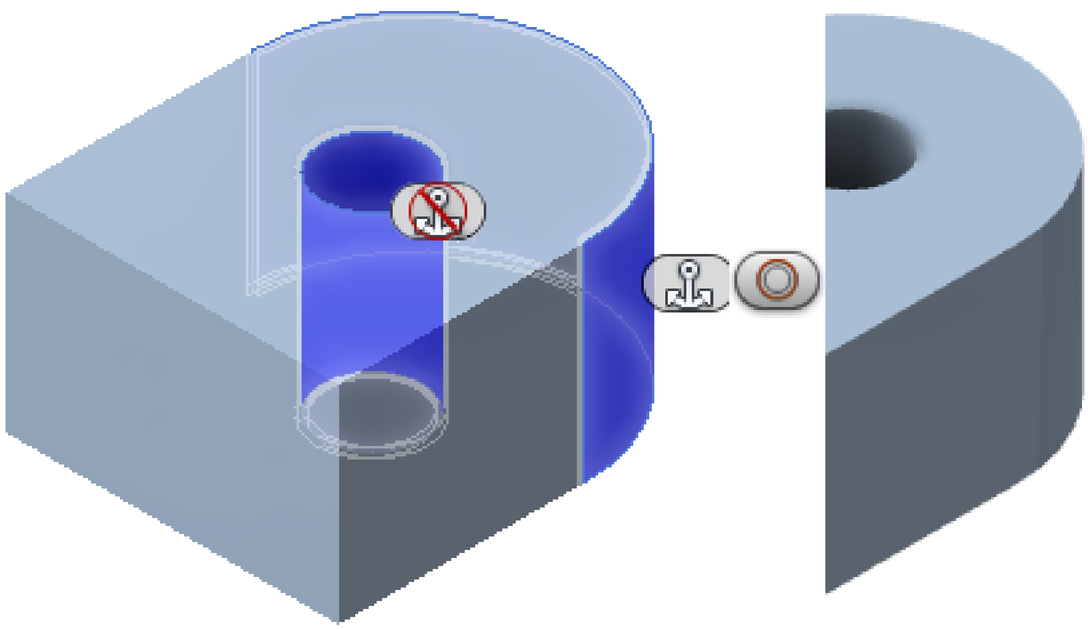
4 The first face that you select is marked with the Anchor glyph, and the second face is marked with the Non-anchor glyph. When you apply a constraint, the anchored face remains where it is and the other face moves.

5 You can switch the anchored and non-anchored faces by pressing the Tab key, or clicking one of the anchor glyphs.

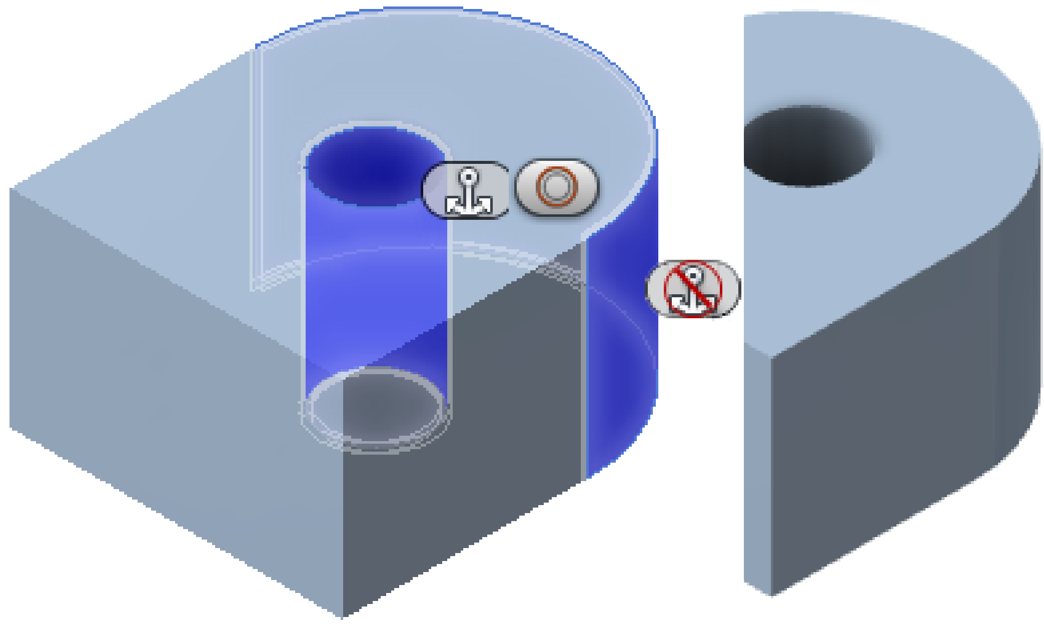
---

**NOTE** The anchored state is temporary. After you create the constraint, either face can move to satisfy the constraints.

---



The filleted end was selected first, so the hole moves to become concentric.



The hole was selected first, so the model is shortened to make the filleted end concentric.

### Constraints in the browser

Body constraints are added to a Constraints folder in the browser. The faces highlight when you pause the cursor over a constraint. You can rename constraints to make it easier to identify them. You cannot edit a constraint; you can only delete it from the browser and create a new one.

### Special behavior and limitations

- If a modeling change divides a face, only one of the new faces keeps the constraints. It is not possible to predict or specify which face will keep the constraint.
- Only the DWG format supports body constraints.
- The Draft command overrides body constraints. Any conflicting constraints are deleted.
- Using a face to create some features, like Revolve, will delete the original face. Any constraints to that face are also deleted.

### Work with body constraints and model dimensions

Use a combination of body constraints and model dimensions to control the size and shape of a model. Body constraints define geometric relationships between faces and edges within a single component. Model Dimensions define the size and location of model edges, both within a component and between components.

**Body constraints:**

- Make model geometry coplanar, concentric, parallel, or perpendicular.
- Do not define the size or location of faces or edges.
- Cannot be created between components.

**Model Dimensions:**

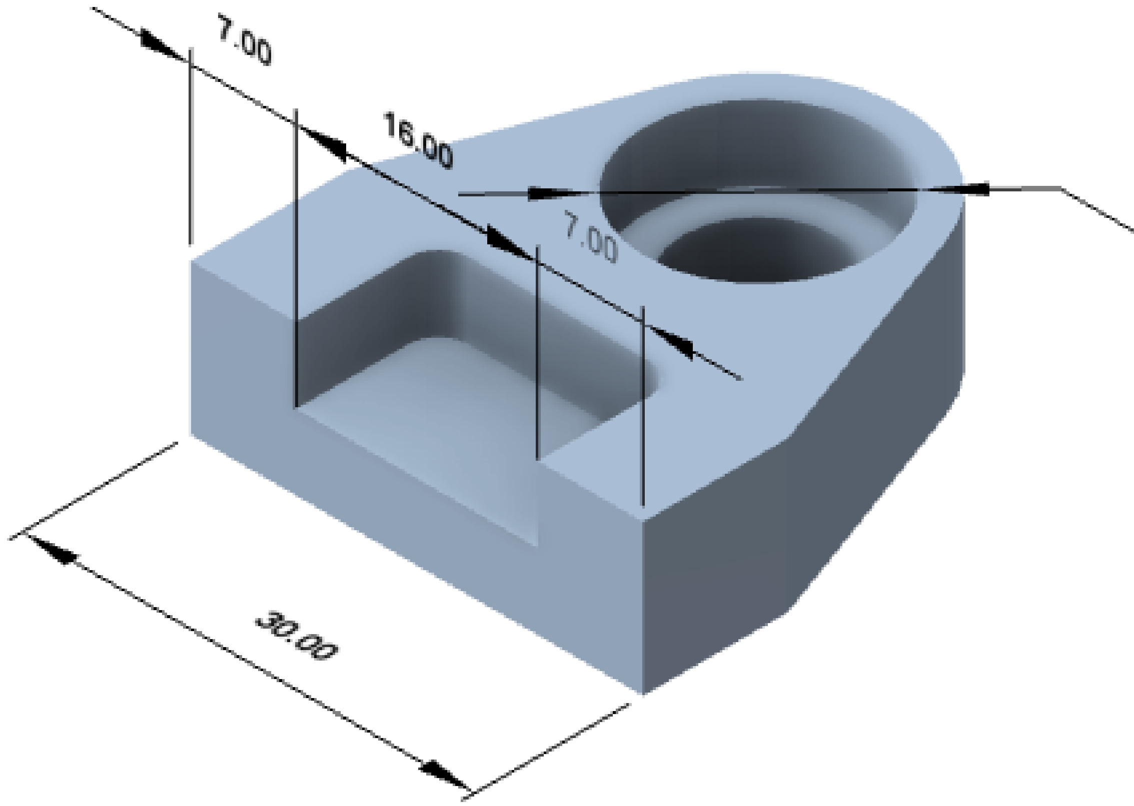
- Define the size of or distance between edges.
- Do not control geometric relationships between faces.
- Can be created between components.

When the model is changed, Inventor Fusion analyzes the body constraints and model dimensions to maintain the shape of the model and the size of features. The body constraints are solved first, and then the model dimensions. An error displays if there is a conflict.

For more information on using dimensions to define the size and location of model edges, see the Model Dimensions and Annotations (page 349) topic.

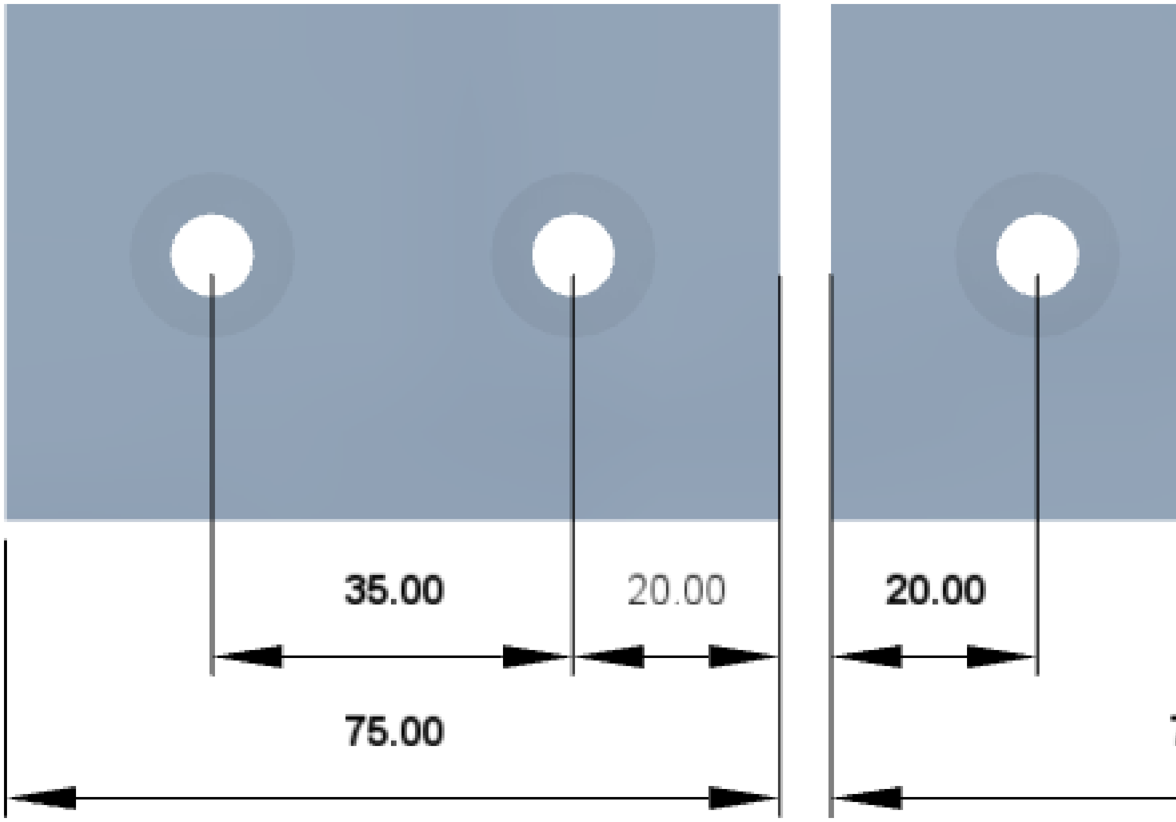
For information using assembly constraints to position components relative to each other, see the Position and Constrain Components (page 383).

# Model Dimensions and Annotations



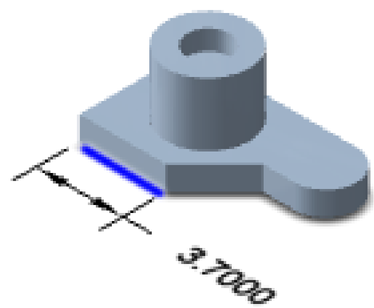
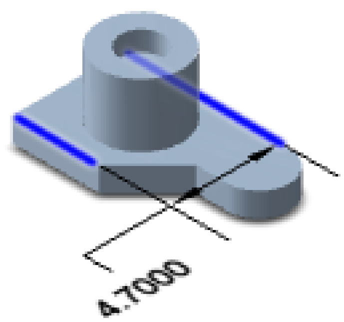
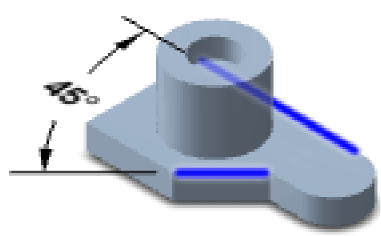

Using direct modeling in Inventor Fusion, you can change model geometry without impacting other features. This behavior makes it easy to create and modify models, but you can also accidentally change them. Define and monitor important dimensions as the model matures so that you can lock down the location and size of features.

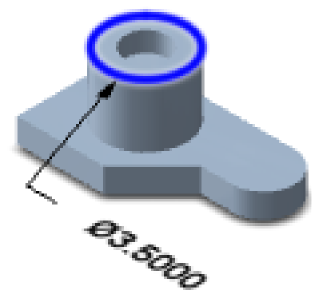
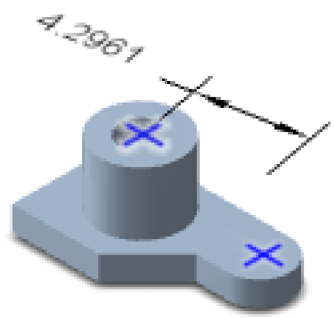
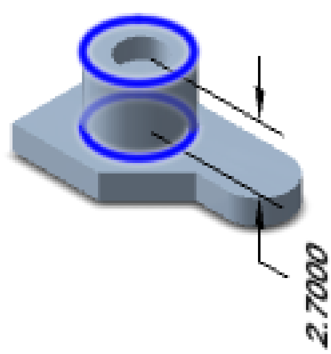
**NOTE** Defining the size and location of geometry is also called capturing design intent. In the image below, the block has two holes. The design intent for the dimensions on the left is to maintain the distance between the holes. The design intent for the dimensions on the right is to maintain the distance between the holes and the edges.

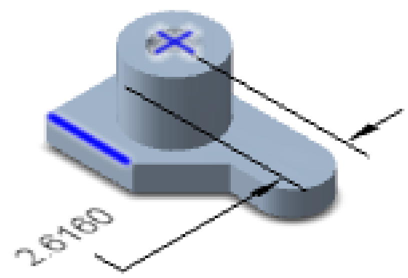
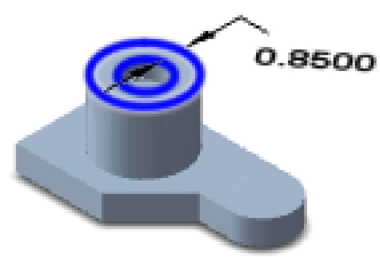

35.00
20.00
20.00
75.00

### Types of Dimensions

You can create model dimensions on linear and circular edges. You can create the body dimensions shown in the following table:

| Selection | Example | Dimension type | Notes |
|---|---|---|---|
| One linear edge |  | | |
| Two parallel linear edges |  | | reated, no distinction from one-edge dimension. |
| Two linear edges, not parallel, but in the same plane or parallel planes |  | | b key to swap gth dimension when edges are on different faces. |

| Selection | Example | Dimension type | Notes |
|---|---|---|---|
| One circular edge |  | | |
| Two circular edges, non-concentric |  | | |
| Two circular edges, concentric and not coplanar |  | | |

| Selection | Example | Dimension type | Notes |
|---|---|---|---|
| One linear edge and one circular edge |  | | |
| Two circular edges, concentric and co-planar |  | | |

### Create dimensions

Body dimensions can either define (locked dimensions) or annotate (unlocked dimensions) a model. This behavior is like sketch dimensions (page 121).

You can create a dimension for the size of an entity or for the distance between entities. For more information, see the previous section, Types of Dimensions.

Dimensions use ANSI or ISO styles. New dimensions are created using the active style, but existing dimensions do not update if the style changes. You can manually change the style of existing dimensions.

The precision for linear and angular dimensions is defined in the style. You can change the precision for a dimension type (linear or angular), but you cannot change the precision for an individual dimension.

1. Expand the Constraint & Dimension panel, and set the dimension style to ANSI or ISO.
2. On the Constrain & Dimension panel, click Dimension.
3. In the graphics area, select a linear edge, an arc, or a circle.

   To dimension the length of a linear edge, press tab to select the annotation plane.
4. To dimension the distance between two entities, select a second linear edge, arc, or circle.
5. Click to place the dimension.
6. Select the entities for the next dimension.
7. Press `Enter` or select OK from the marking menu to end the command.

### Define planes for dimensions

Dimensions are created on annotation planes. Annotation planes are automatically created on model faces when you place a dimension. If all dimensions on that plane are removed, the plane is deleted. In the browser, the planes are added to the Annotation Planes folder. If you delete an annotation plane in the browser, you also delete all dimensions on that plane.

To define the annotation plane for a linear edge, press `Tab` to select one of the adjacent faces. If you select an arc, circle, or two entities, there is only one valid annotation plane.

### Dimension Precision

The linear and angular dimension types have different precisions, and the ANSI and ISO styles have different precisions for linear dimensions.

To change the precision for a dimension type, right-click a dimension, and on the Dimension/Constrain>Dimension Precision list, select a new precision.

---

**NOTE** The precision setting changes all of the dimensions in the model. You cannot change the precision of a single dimension.

---

### Edit Dimensions

Edit a dimension by double-clicking it. The value field and the anchor glyph display. Enter a new value for the dimension, or change the anchor location by moving the cursor to the other end of the dimension.

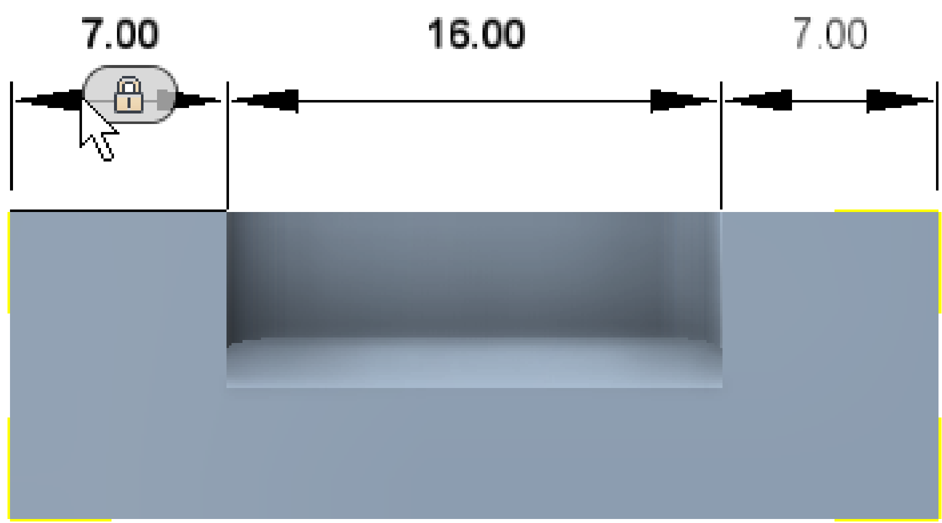
**NOTE** The anchor location defines which end of the dimension is fixed.

**NOTE** When you press `Enter` or select **OK** from the marking menu, the dimension automatically switches to the locked state. This switch occurs even if you do not change the dimension value.

### Locked and Unlocked Dimensions

You can lock or unlock body dimensions. This behavior is like sketch dimensions (page 121). Model dimensions (locked) define the size and location of geometry. Annotation dimensions (unlocked) display the current value of a dimension.

Dimensions are unlocked by default. Locked dimensions display in bold. A padlock glyph displays when you pause the cursor over a dimension.



### Change a dimension between locked and unlocked

- Pause the cursor over the dimension, and click the padlock glyph.
- Double-click the dimension to edit it. The dimension is automatically locked, even if you do not change the value.

### Dimension Anchors

Most dimensions, except the radius of an arc or the diameter of a circle, have one end anchored. The anchored end stays in position when the dimension updates, and the other end moves. You can change which end is anchored when you edit the dimension.

To set the anchor for a dimension, double-click it and move the cursor to one end. The anchor glyph displays and automatically attaches to that end of the dimension.

### Special Behavior and Limitations

Some changes can make a dimension invalid. During the preview, the dimension color changes to light gray, and the dimension is deleted when you complete the change.

Only the DWG format supports dimensions. If you save the model in another file type, the dimensions are lost.

### Work with model dimensions and body constraints

Use a combination of body constraints and model dimensions to control the size and shape of a model. Body constraints define geometric relationships between faces and edges within a single component. Model Dimensions define the size and location of model edges, both within a component and between components.

**Body constraints:**

- Make model geometry coplanar, concentric, parallel, or perpendicular.
- Do not define the size or location of faces or edges.
- Cannot be created between components

  **NOTE** Use the Assemble command to create constraints between components

**Model Dimensions:**

- Define the size of or distance between edges.
- Do not control geometric relationships between faces.
- Can be created between components.

When the model changes, Inventor Fusion analyzes the body constraints and model dimensions to maintain the shape of the model and the size of features. The body constraints are solved first, and then the model dimensions. If a conflict occurs, an error displays.

# Dimensions and Body Constraints

Dimensions and body constraints can constrain the faces and edges in a model in certain ways, while allowing free-form Direct Manipulation of the model in other ways.

**Body constraints**

You can use body constraints to constrain faces and edges within a single component, but not between different components.

Body constraints cause the body to change shape to meet the constraints. In contrast, Component (assembly) constraints treat each component rigidly. Inventor Fusion solves body constraints first, and then component constraints. For information about how to constrain or assemble components relative to each other, see Position and Constrain Components. (page 383)

Inventor Fusion respects the design intent of the constraints when you use the Move and Press/Pull commands, and when you change dimension values. If you move a face that is constrained to another face, then that face moves exactly as you specify, and the other face also moves.

Constraints are propagated through patterns. When you move a face that is part of a pattern, then all corresponding faces in that pattern move in the same way. If any of those faces are constrained to other faces, the other faces also move. If Inventor Fusion cannot find a suitable solution, then the preview stops and the error glyph displays.

You can access the following types of Body constraints from the Constrain command:

- Coplanar: Two planar faces are made to lie in the same plane
- Center: Two cylindrical faces are made to lie along the same axis

- Parallel: Two planar faces are made to be parallel
- Perpendicular: Two planar faces are made to be perpendicular

When you use the Constrain command to create body constraints:

- Only the appropriate types of faces are available for selection (cylindrical for Center constraints, planar for the other types).
- The first face that you select is marked with the Anchor glyph (grounded). The second face that you select is marked with the Non-anchor glyph (not grounded). When you apply a constraint, the grounded face remains where it is, and the other face (or faces, as appropriate) moves.
- You can switch to the grounded face with the Tab key, or click the Anchor or Non-anchor glyph, before you apply the constraint.
- You can change the constraint type before you apply the constraint. Faces that are not valid for the new constrain type are filtered out.
- The grounded state is temporary. After you create the constraint, either or both faces can move to satisfy the constraints. There is no way (nor no need) to make a face permanently grounded. The solver never moves unconstrained faces. (In contrast, assembly components can be grounded.)

**NOTE** Instead of using body constraints to move a face, you can use the move triad, possibly aligning it to an edge. By selecting the appropriate element of the triad, you can force the face to move in only certain limited directions.

### Locked dimensions

Dimensions can drive the model, or function as pure annotations (non-driving). For more information about dimensions as annotations, see Dimensions.

Dimensions and explicit body constraints are solved together. No precedence is given to one or the other.

You can lock a dimension to force the model to maintain that dimension at the current value, regardless of other changes. Locked dimensions are shown in bold.

Locked dimensions constrain edges. The adjacent faces move to adjust the edges to the correct size and the correct position.

When you pause the cursor over a dimension, a lockable dimension shows a Lock glyph which means the dimension is locked. If it shows an Unlock glyph, the dimension is unlocked.

To lock a dimension, do one of the following:

- Double-click the dimension and change its value (click `Enter` when done, or `Escape` to cancel).
- On the context-menu, click the check box for Lock to lock the dimension at its current value. Clear the check box to unlock.
- Click the Lock glyph.

**NOTE** If a double-click does nothing, the Lock check box is not available on the context menu, and no glyph displays when you pause the cursor over a dimension, then you cannot lock the dimension.

When editing the value of a dimension:

- A preview of the new value displays. A delay in the preview minimizes unnecessary previews of intermediate values as you enter them.
- Most dimensions show an anchor glyph to indicate the grounded side. The other side moves in response to the new value of the dimension. To swap the anchor, move the cursor to the other side. Indicator lines and circles represent the original position of the editing edges. Indicator lines and circles display temporarily when the dimension is in preview mode.

### Details and limitations

- Sometimes invisible constraints remain to maintain obvious perpendicularity and parallelism. If the invisible constraint is not appropriate, move the faces slightly before you apply the constraint, or lock the dimension.
- If a Move or Press/Pull operation causes an edge or face to split, and the edge or face has a dimension or constraint attached to it, then the dimension or constraint is automatically applied to one of the resulting entities. The other edge/face does not inherit any constraining behavior from the original. It is not possible to predict or specify which edge or face acquires the constraint.
- You can create a situation where a dimension becomes invalid and changes color during the preview. If you finish the command with dimensions in this state (sick), then the sick dimensions are deleted.
- If you create a set of constraints that cannot be solved, the error glyph appears. Undo or delete constraints to get back to a properly solved model.
- You can save dimensions and body constraints in only the DWG format. They are not saved in any other format.
- Constraints are not respected in the Draft command.

- Under some circumstances, invisible intermediate states can fail for no visible reason. To avoid these situations, by move the elements through a different route, or move a little bit at a time, and accept each move.

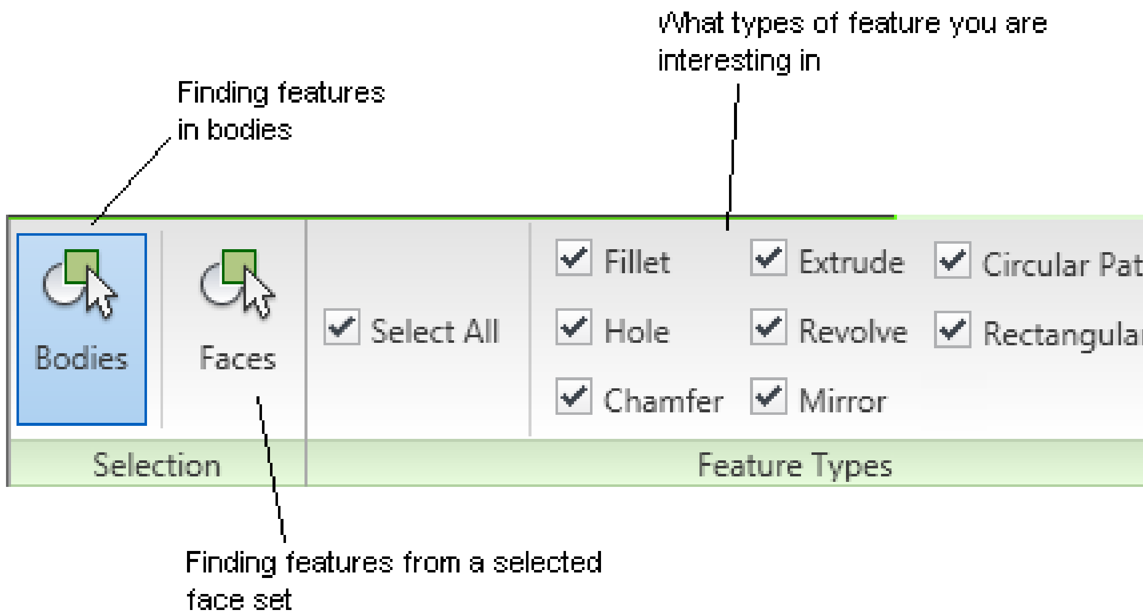
# Find Features

Find Features, also called feature recognition, is a process to extract design feature information from a solid model. Currently, Find Features supports:

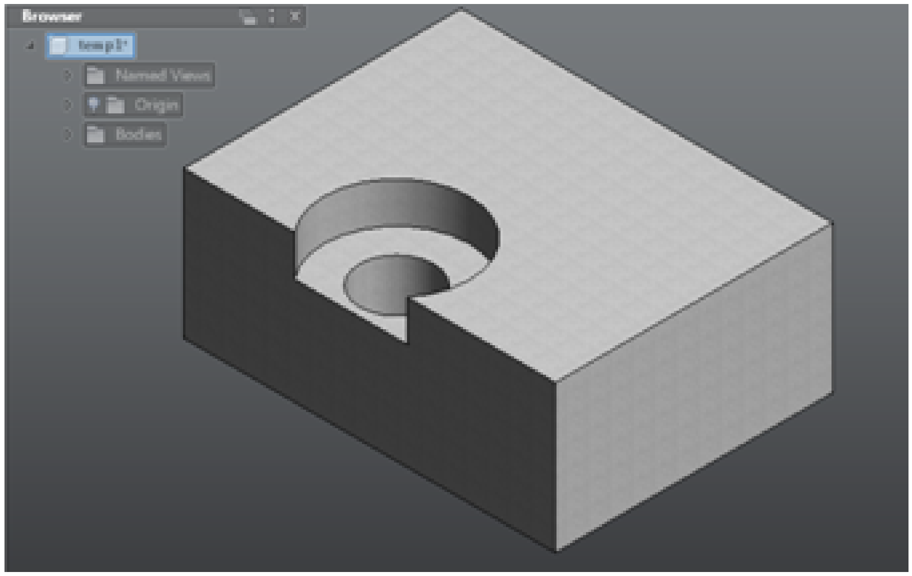
- Fillet
- Hole
- Chamfer
- Extrude
- Revolve
- Patterns
- Mirror

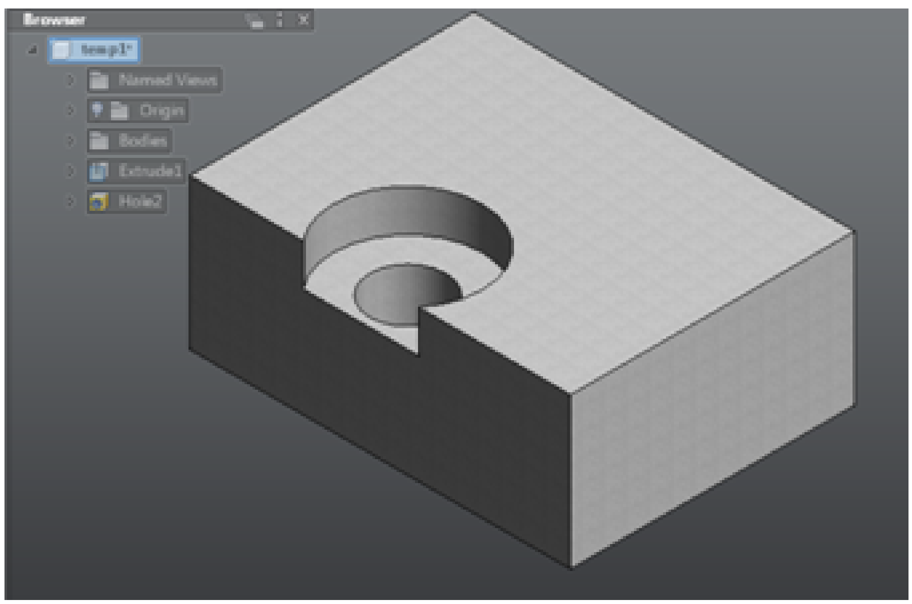
The Find Features command adds features to the browser. The command does not modify the geometry of the body. Features aid in selecting geometry to modify the body. You can perform Find Features on bodies or faces. You can specify the feature types to recognize.

When an Inventor file (IPT, IAM) is opened in Inventor Fusion, Find Feature creates a Fusion feature for each Inventor feature in the model.

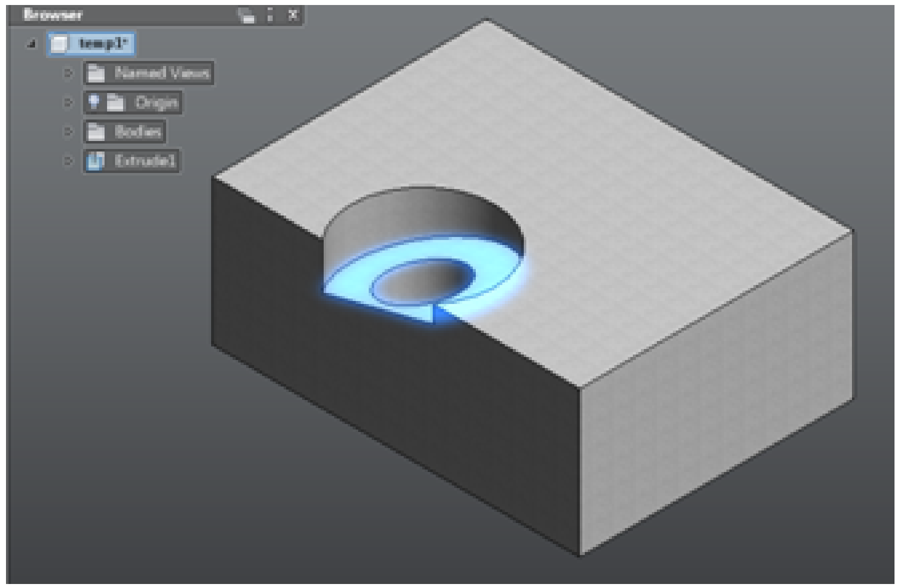



In the following image, Find Features adds an extrude (the rectangular body) and a counter bore hole (the cylindrical cut) to the browser. After a feature is found, it is managed the same as features created with traditional methods. Features can be edited, deleted, or dissolved.




If a downstream operation modifies a feature face, the owning feature is re-evaluated. If the face set does not satisfy the feature definition, the feature is removed from the browser. In the previous counter bore hole example, if you move the highlighted face, the feature is not a valid counter bore hole. It is removed from the browser at the end of the move operation.




If Find Features recognizes a feature in a way that does not fit your intent, you can change it. In the browser, right-click and change the feature type. For example, to recognize a hole as a revolve, you can dissolve the feature, which removes the feature from the browser, but leaves the geometry on the model.

# Model Simplification

The Simplify command removes constant radius fillets, equal distance chamfers, and holes. It provides tools for selecting the features based on type and size.

### Show me how to simplify a model

Model simplification is typically used to prepare models for simulation analysis. Many CAD models are too complicated for CAE applications like FEA and CFD. Small features can cause a significant increase in analysis time. CAE analysts typically remove these details before meshing the model. While most small features can be removed, some features are important for the analysis. The Simplify command checks the model for features and provides tools for removing them based on type and size.

---

**NOTE** Simplify does not analyze the model, so it does not recognize geometry that does not have a feature in the browser. For example, simplify finds a hole feature, but not a cylindrical cut. Use the Find Features (page 360) command to convert model geometry to features.

---

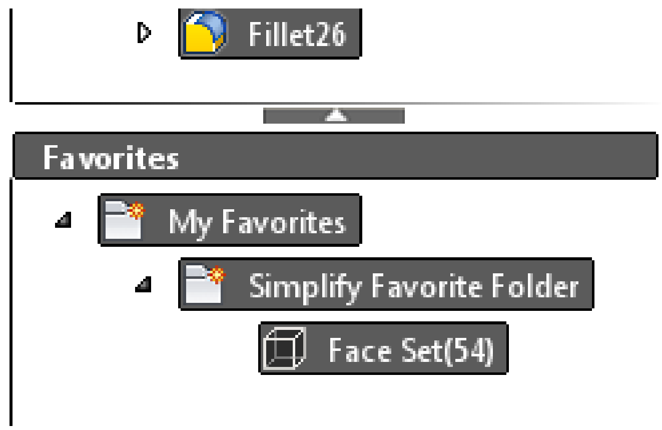
### Model simplification results

The wizard can take three actions on the selected features:

**Delete** Removes the selected features and extends the adjacent faces to close the gaps. If the faces cannot be extended for a particular feature, then that feature is not removed.

**Select** Retains the selection set after you click OK. You can quickly select a group of features to edit with modeling commands.

**Create Favorites** Creates a Simplify Favorites Folder in the browser with the feature definitions. You can click the folder in the browser to select all model features that meet the criteria.




**Feature types in the Simplify dialog box**

The Simplify dialog box contains a check box for each feature type, and sliders to set the range of sizes. You can deselect feature types, and filter them by size.

---

**NOTE** Simplify does not analyze the model. It retrieves only features that are identified in the browser. Run the Find Features command to identify features in imported models.

---

In addition to feature type, you can filter for fillet geometry, and counterbores on holes.

- Fillets:
- Convex
- Concave
- Radius

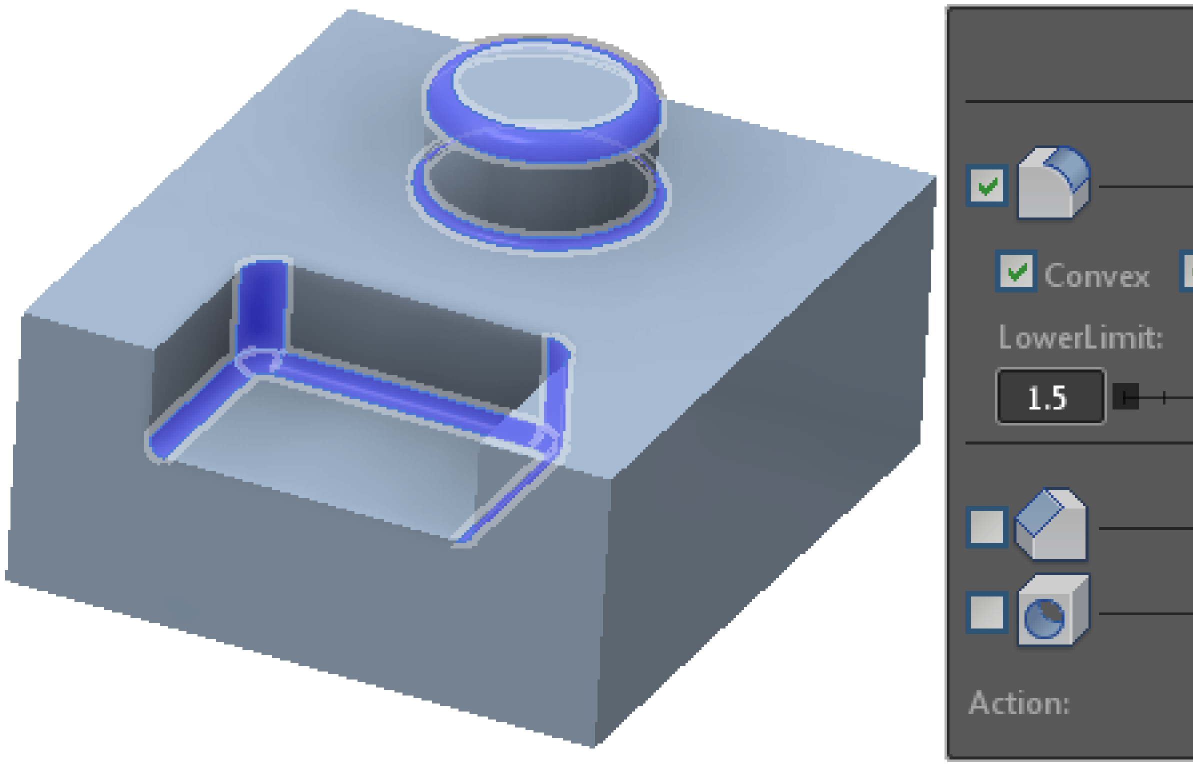
- Chamfers:
- Leg Length

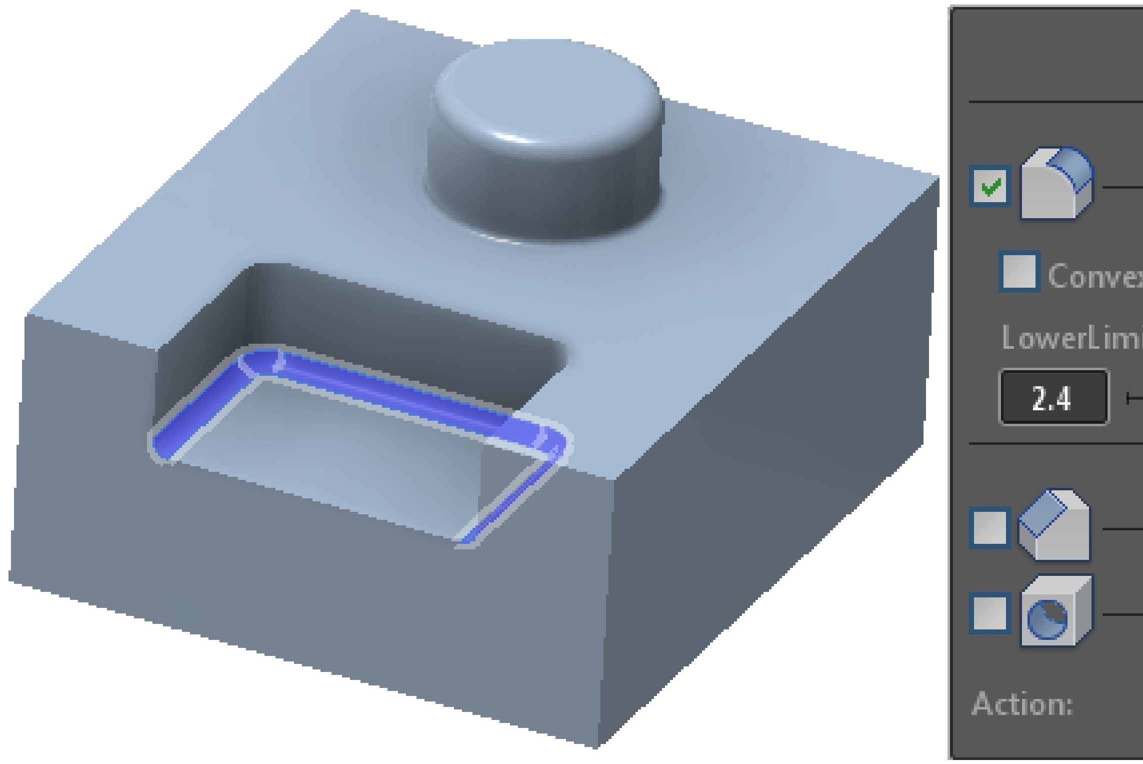
- Holes:
- Depth of Holes
- Diameter of Holes
- Depth of Counterbores
- Diameter of Counterbores

**Feature Sizes**

The black slider button sets the lower limit, and the red button sets the upper limit. You can drag the slider buttons to change the limits, or click the number and enter a new value. You cannot make the lower limit greater than the upper. As you change the limits, features outside of that range are deselected.

In the images below, the settings filter for concave fillets between 2.4 mm and 5.1 mm radius.




Convex
LowerLimi
2.4
Action:

**Remove features from a model**

### Simplify a model

1 On the Simulation panel, click Simplify. All fillet, chamfer, and hole features are automatically selected.

   **NOTE** Simplify does not analyze the model. Use the Find Features (page 360) command to convert geometry into features.

2 Deselect feature types to remove them from the selection set.

3 Change the size limits to reduce the selection set. You can drag the black and red slider buttons, or click a number to enter a value.

4 Select an Action method

   - **Delete** Removes the selected features and extends the adjacent faces to close the gaps. If the faces cannot be extended for a particular feature, then that feature is not removed.
   - **Select** Retains the selection set after you click OK. You can select a group of features quickly to edit with modeling commands.
   - **Create Favorites** Creates a Simplify Favorites folder in the browser with the feature definitions. You can click the folder in the browser to select all model features that meet the criteria.

5 Right-click in the graphics window, and select OK.

---

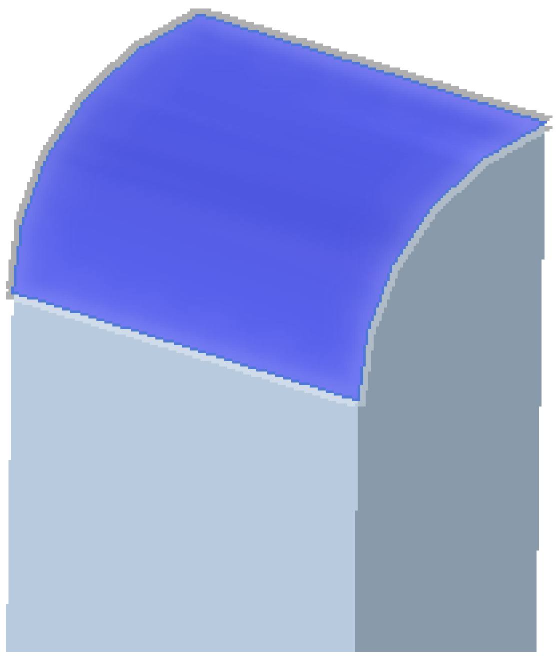
**NOTE** You can click anywhere on the background of the Simplify dialog box to drag it to a new location.

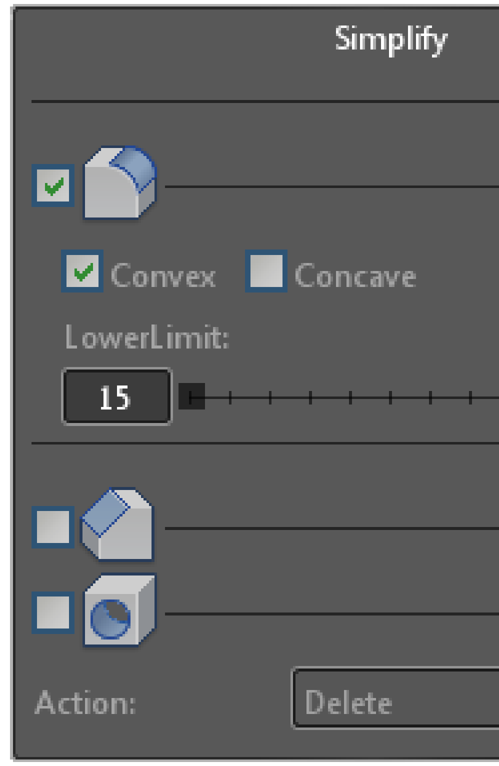
---

### Large feature limitations

When a feature is deleted, the adjacent faces are extended to close the gap. If the feature is large enough to remove a model face, the Simplify command cannot repair the body. In these cases, the features are not deleted.

In the example below, the Simplify command found the fillet feature, but does not remove it because the model requires a new top face.



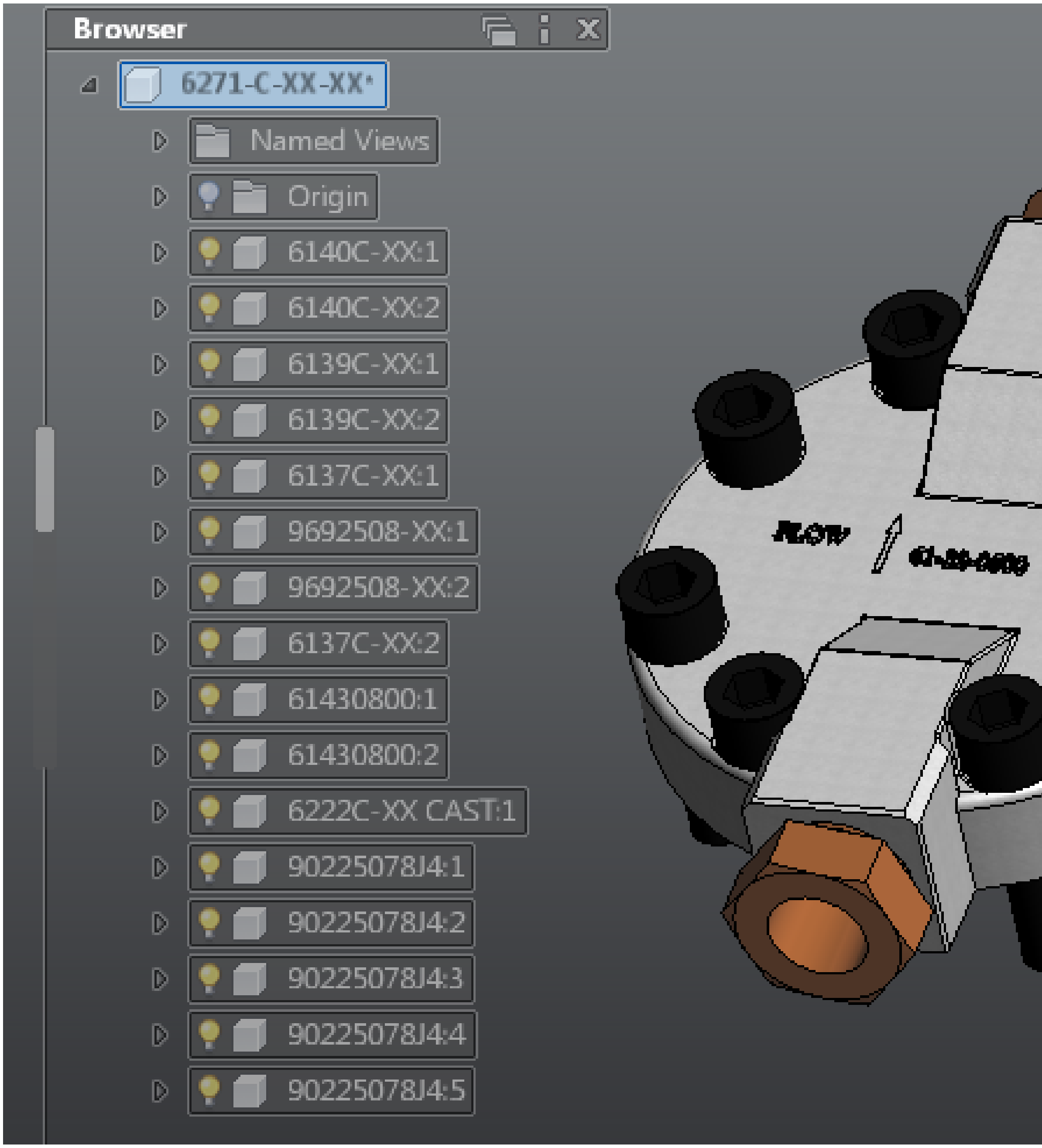



## Work with Multiple Components

Sometimes a design requires that you create and manage several different components. For instance, if your design contains 25 bolt/washer/nut combinations, do not manually create each fastener as a separate set of features. Instead, create just a single version of the bolt, nut, and washer, and then replicate that set of components across your design. Inventor Fusion has commands to support the placement and management of components and component instances.

**Note:** In Inventor Fusion, you can manage all the components in your design in a single Fusion DWG file. Unlike some other CAD products, Fusion does not require the creation of separate files on disk to use components.

Using components, you can easily and accurately create designs such as the following:


Browser
6271-C-XX-XX*
Named Views
Origin
6140C-XX:1
6140C-XX:2
6139C-XX:1
6139C-XX:2
6137C-XX:1
9692508-XX:1
9692508-XX:2
6137C-XX:2
61430800:1
61430800:2
6222C-XX CAST:1
90225078J4:1
90225078J4:2
90225078J4:3
90225078J4:4
90225078J4:5
FLOW

Components in Inventor Fusion are organized hierarchically.  That is, each component can have zero or more child components, and those child components can have child components, and so on.

Every Fusion design has a single root component. The root component has the same name as the document itself, and is the top node in the browser.

A Fusion component can also contain:

- **Solid** A single solid body.
- **Surface Bodies** Zero or more surface bodies, quilts, or user-defined groups containing surface bodies or other groups.
- **Sketches** Zero or more 2D sketches.
- **Work Geometry** Work planes, work axes, work points.
- **Features** For a description of a feature in Fusion, see  NO LABEL .

### Components and Component Instances

Inventor Fusion supports the reuse of components in your designs. You can use any component but the root component multiple times in your design. Each version of the component refers to the same component geometry, and is called a *component instance*. In the browser, each instance is numbered:

All instances of the same component share the same geometry, so a change to one applies to all instances of that component.

Other topics related to components include:

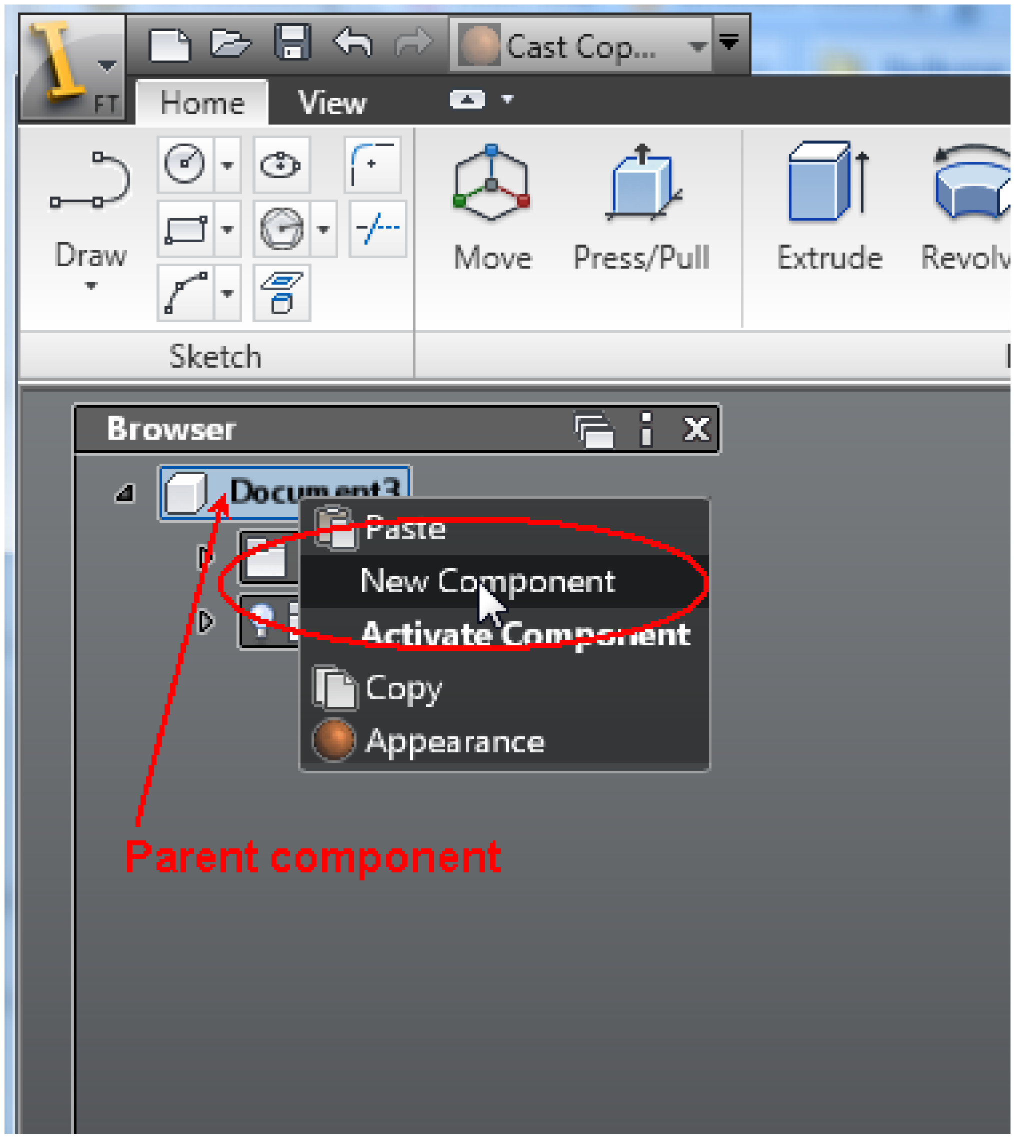
- Create Components (page 377)
- Position and Constrain Components (page 383)

## Create Components

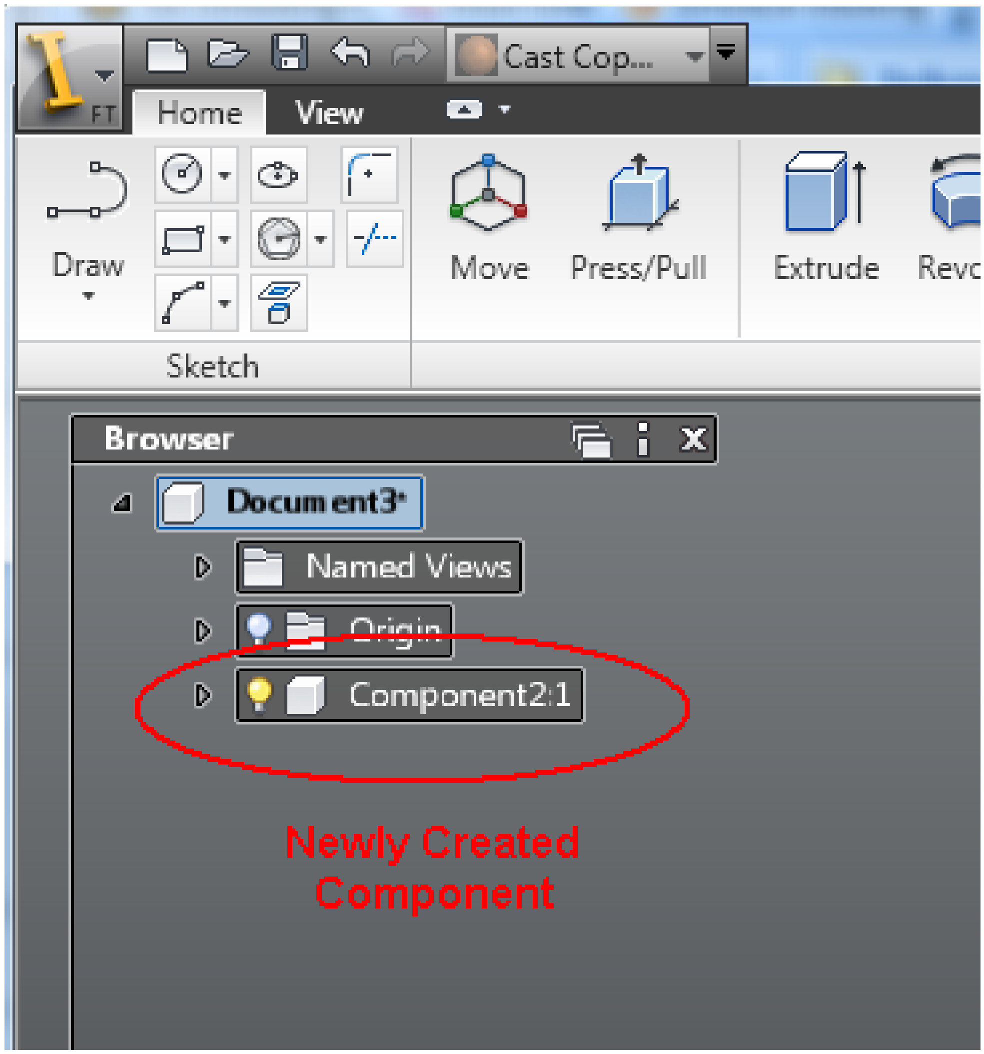
In Inventor Fusion, there are several methods to create components and populate them with geometry.

**New Component Command**

This command is located on the context menu when you right-click a component node in the browser.

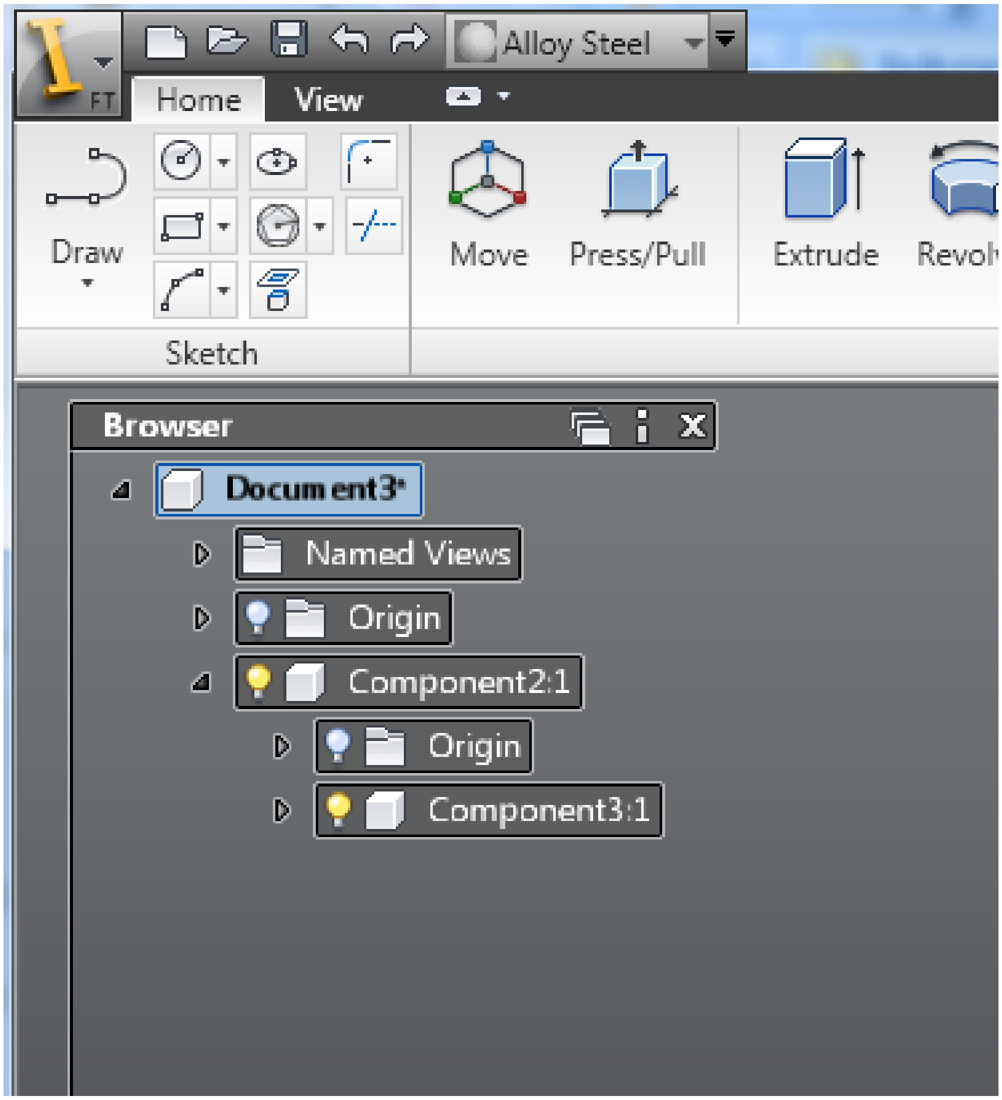



The New Component command creates an empty component that is a child of the selected component (which can be the root component).



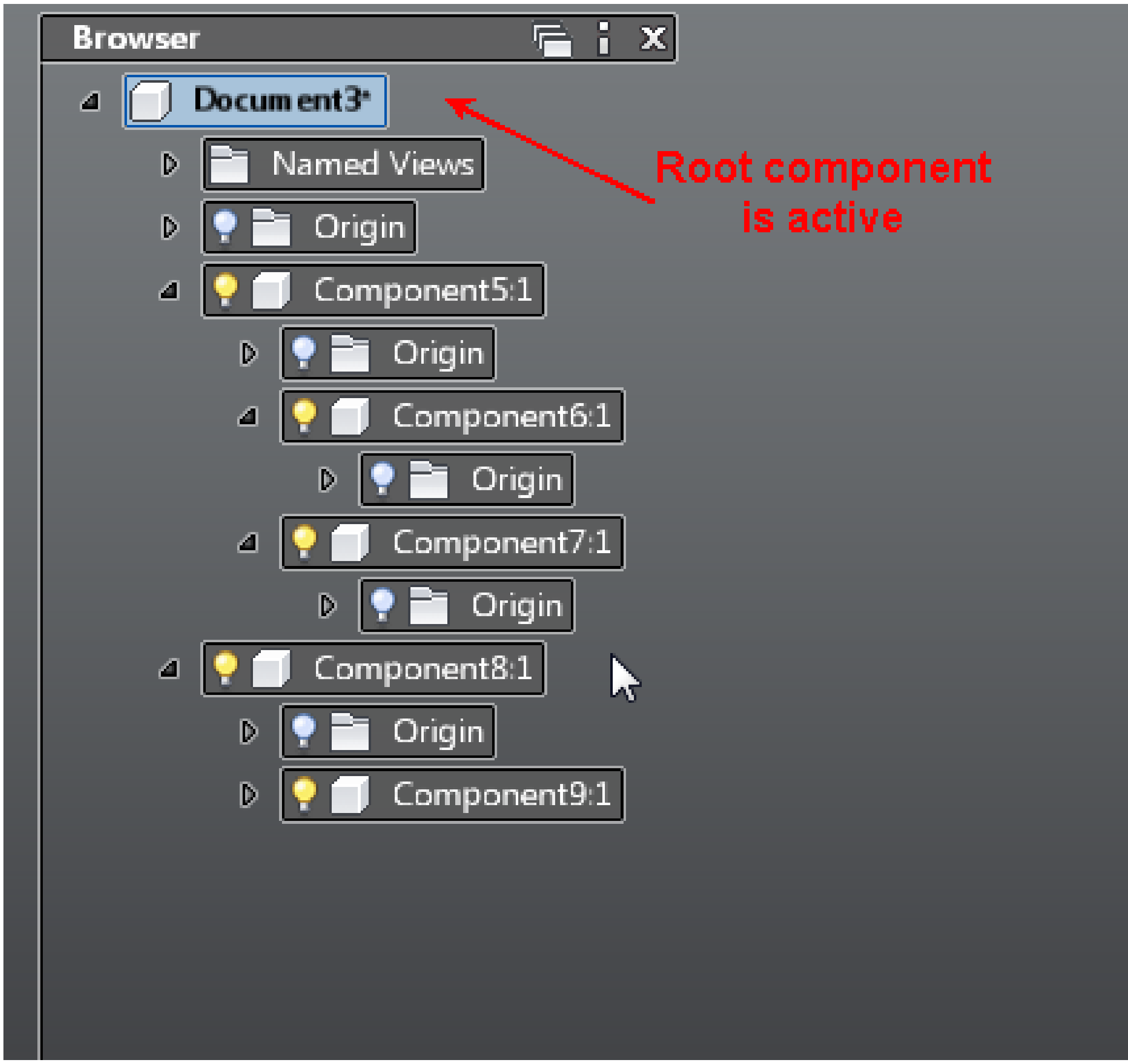

If you right-click Component2 and select New Component, a child component of that component is created.

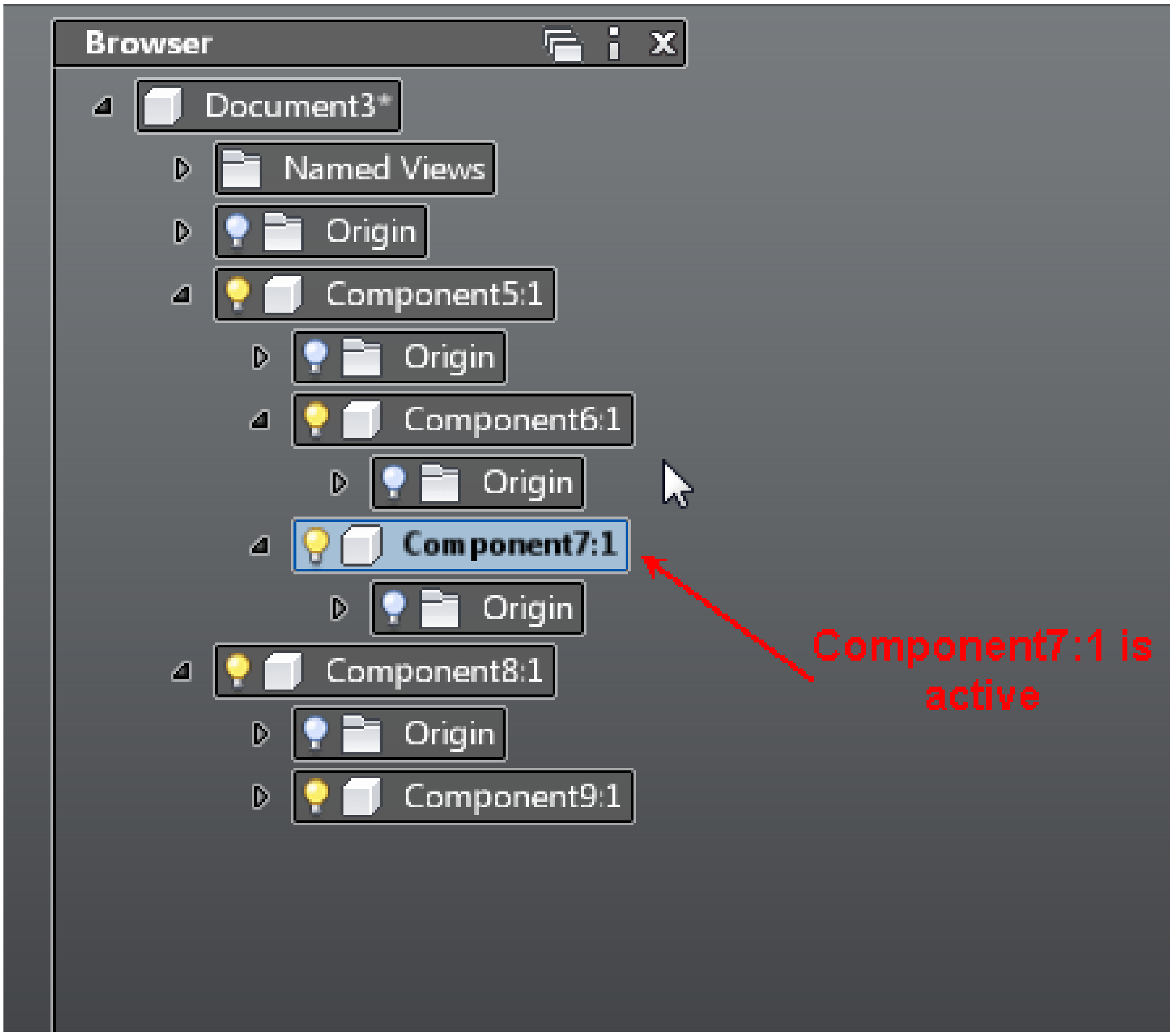
:




### Activate Components

Inventor Fusion always has an active component, where newly created objects go. These new objects can include body geometry (for example, if new features are created), work geometry, and sketches.  A blue highlight shows the active component:






**Activate a component**

- Double-click the browser node for the component.

or

- Use the Activate Component command.

### Add solid geometry to the active component

- Create new features with commands such as Extrude, Revolve, Loft, and so on. For more information, see Create a Single Body (page 98).

or

- Drag or copy and paste bodies from other components.

### Create new instances of components

- Using the Copy and Paste commands, select a component to copy.
- select the intended new parent component (which could be the root component).

The resulting new instance of the component that you copied and pasted is attached to the cursor. You can drag it and click a point to place the new instance.

## Position and Constrain Components

You can position component instances in 3D, and create constraints to keep instances precisely positioned relative to other geometry in your design.

Inventor has three methods to position and constrain components:

**Move**

Move (page 275) uses the triad to change the location and orientation of components. Move cannot permanently position components.

Move can change the position or modify the geometry of a component. Move makes local changes, so it can change the position of child components, such as the rod of a pneumatic cylinder. Move cannot permanently position a component.

**Assemble**

Assemble uses geometric constraints to determine how components in the assembly fit together. Assemble can temporarily position components or create permanent constraints.

Assemble treats components as rigid bodies, so it can only change the position of components. It cannot change the size or shape of model geometry or the position of child components. For example, Assemble can change the position of a pneumatic cylinder assembly but cannot extend or retract the rod.

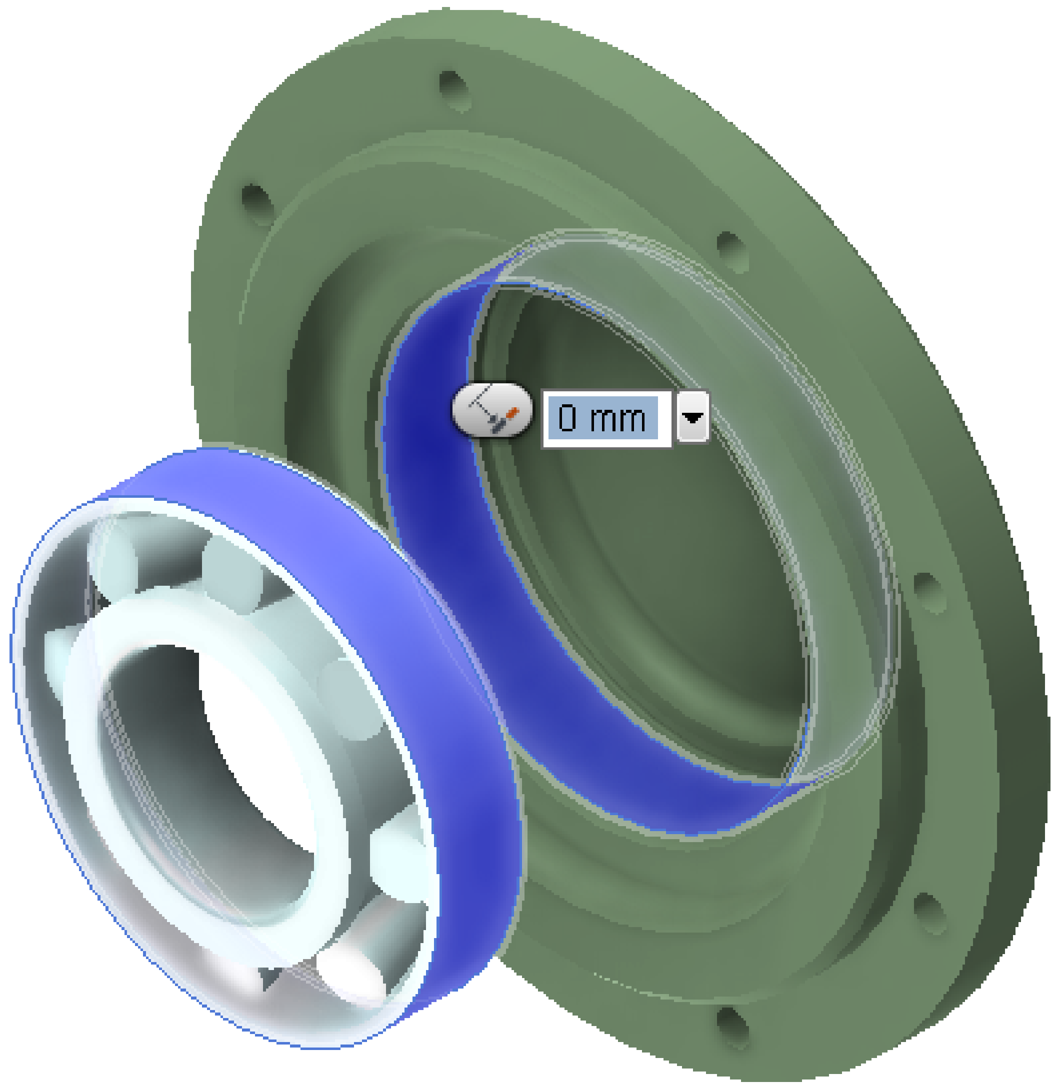
### Grounded

Grounded is a setting that locks a component in its current location and orientation.

You can turn Grounded on or off by right-clicking on the component in the browser and selecting Grounded. A Grounded component has a pushpin on its browser icon. 


**NOTE** Grounded does not lock the model geometry. For example, you can use Move to change the position of the solid in a grounded component.

## Assemble Components




In the Assemble command, geometric relationships determine how components in the assembly fit together. Assemble can change the location and orientation of components. It can also create constraints so that components maintain their positions.

### Overview of Assemble

---

**NOTE** Assemble moves the first selected component to the second one. If the first component is grounded or fully constrained, the second component moves to the first one.

---

Components are initially free to move and rotate in the model. As you apply constraints, you remove degrees of freedom, restricting the ways components can move. When your model is complete, most components will be fully constrained.

Assemble treats components as rigid bodies, so it cannot change the size or shape of model geometry or the position of child components. For example, Assemble can change the position of a pneumatic cylinder assembly, but cannot extend or retract the rod.

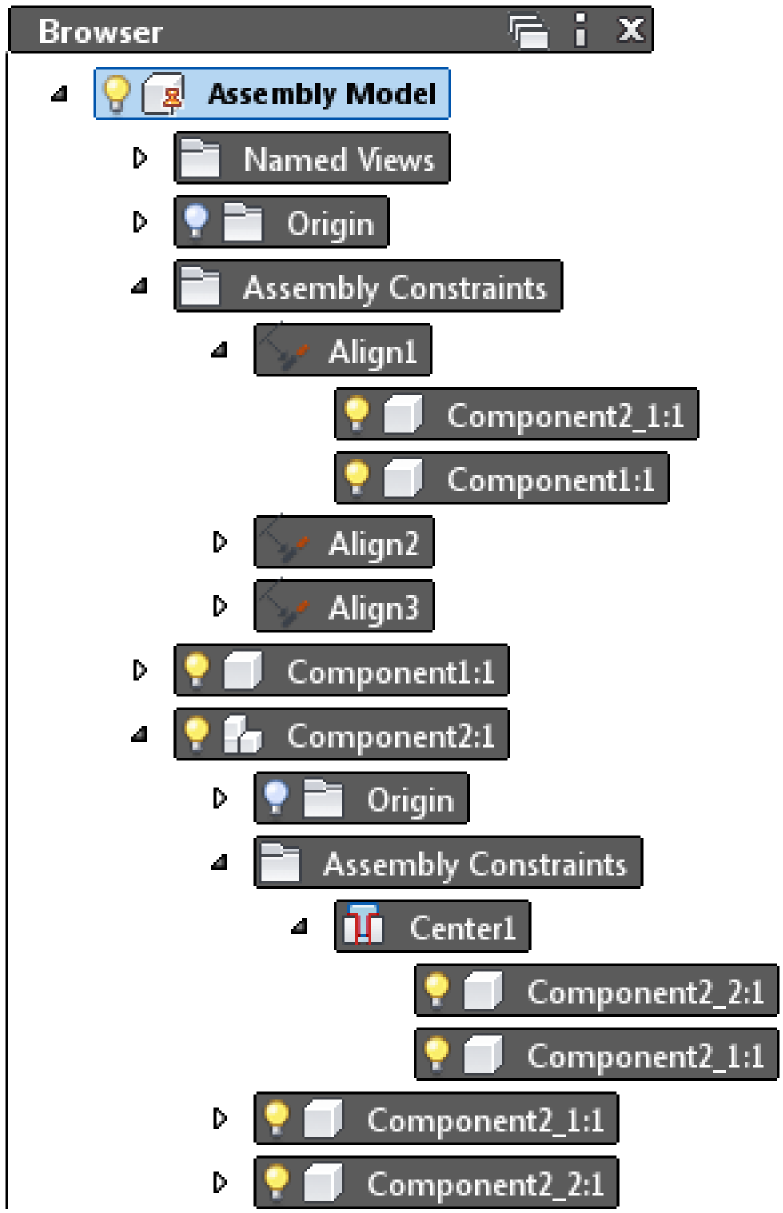
---

**NOTE** Treating components as rigid bodies is important for performance and model stability. Each constraint is analyzed when the model is changed, so checking child components and model geometry would take much longer and could cause unexpected updates.

---

Constraints are added to an Assembly Constraints folder in the browser. A folder is created at the level above the components.

In the image below, Copmonent2:1 was active when the constraints where created. The Align1 constraint between between Component1 and Component2_1 are placed in a folder at the top level. The Constraints between Component2_1 and Component2_2 are placed in a folder under Component2.




You can edit, suppress, delete and rename assembly constraints from the browser.

### Geometry selections and constraint types

Valid geometry selections:

- Planar Face
- Cylindrical face
- Linear Edge
- Circular Edge
- Work Plane
- Work Axis
- Work Point

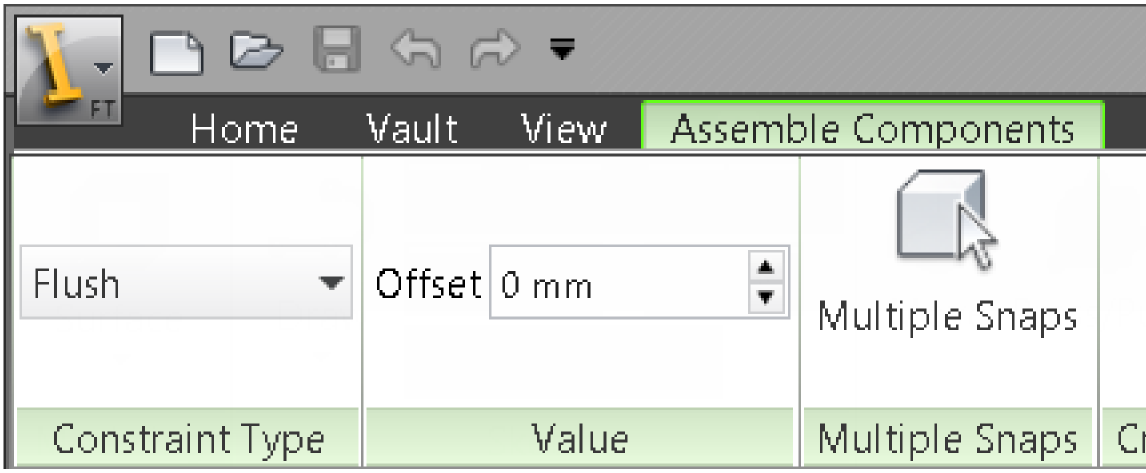
Available constraint types

- An **Align** constraint positions edges, axes, and planar faces in alignment. For example, cylinder axes become collinear and edges are aligned with planar faces.
  - The **Mate** condition positions planar faces parallel and opposite each other. The faces can be offset from one another.
  - The **Flush** condition positions planar faces parallel and in the same direction. The faces can be offset from one another.
- An **Angle** constraint positions planar faces and linear edges at a specified angle.
- A **Tangent** constraint positions cylinders, planes, edges so they contact at the point of tangency. Tangency can be inside or outside a curve.
- A **Center** constraint aligns both the axes and the ends of two cylinders or circular edges. For example, a bolt can be placed in a hole with one constraint.

### Position or constrain a component

**Show me how to assemble components**

Use Assemble to define the location and orientation of components by creating geometric relationships between components using model geometry or work features. You can change the position of components, or create constraints so the components maintain their positions as the model changes.




**Use Assemble to position or constrain a component**

1. Start Assemble from the Constrain & Dimension panel.
2. Select a face or edge on the first component.
3. Select a face or edge on the second component.
4. Select the Constraint Type from the ribbon or from the constraint glyph in the graphics window. The list displays the valid constraints for the selected faces.

   Delete clears the selected geometry.
5. Enter an Offset Value.
6. Select Multiple Snaps to automatically select geometry for related constraints.
7. Deselect Create Constraints to position the component without constraining it.
8. Continue selecting sets of faces for constraints.
9. Click OK to position or constrain the components.

---

**NOTE** You can drag the component instead of selecting the second face. Drag allows you to select a hidden face without rotating the model or using the Select tool.

---

**Valid Geometry Selections:**

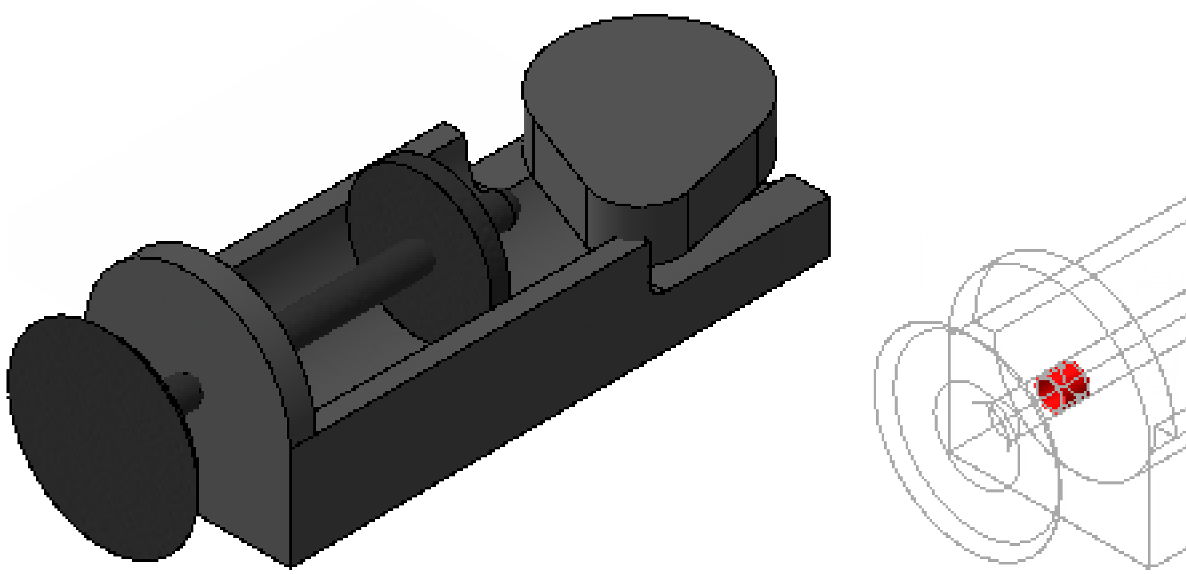
- Planar face

- Cylindrical face
- Linear Edge
- Circular Edge
- Work Plane
- Work Axis
- Work Point

**Constraint Types:**

- Align
  An Align constraint has either a Mate or Flush condition.
- Angle
- Tangent
- Center

## Find Interferences



The Interference command analyzes a set of components for interferences. It extracts each interference volume and creates solid components.

You can select individual components or a component assembly for analysis. Select at least two components. If a component assembly is selected, the interference command checks for interferences between the child components.

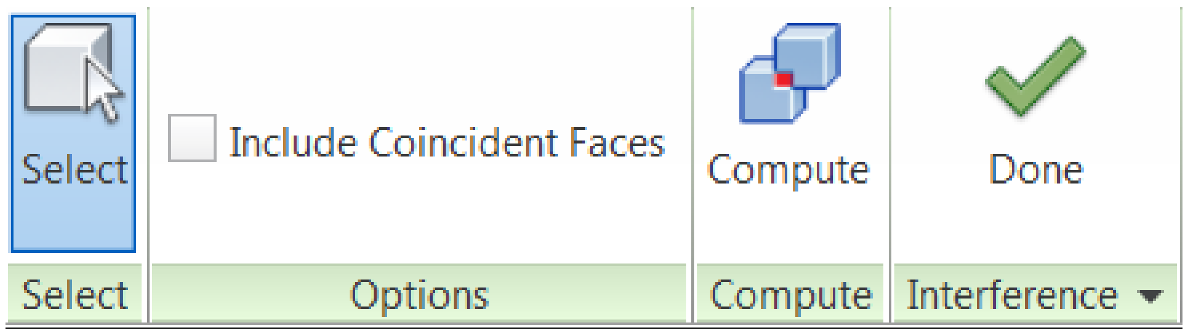
The interferences display in the model, and you can create a separate component for each interference. You can use an interference component as a reference to update the selected components. You can use the Boolean (page 200) command to remove the interference volume from components.

### Interference Results

The Interference Results dialog box displays information for each interference. Each interference can be saved as a solid component. You can select which interferences to save, and specify whether to display interference previews in the graphics window.

In this example, there are two interferences and one coincident face. The coincident face has 0 volume. You cannot save a coincident face as a component.

**Check for interference**




The Interference command analyzes a set of components for interferences. You can select the entire model, multiple components, or a component assembly. Interference does not check for interferences between features within a component.
Select only the components necessary for interference checking. The interference command checks the volume of each component against the other components in the selection set.

The Include Coincident Faces option checks if individual faces are coincident, such as two faces that are coplanar, or a pin that has the same diameter as a hole.

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**NOTE** Limit the number of components in the selection set to improve performance and improve the results. This option can add a significant amount of time, especially if the geometry is complicated. Also, many components have co-planar faces. For example, the faces of a part mounted on a plate coplanar.

---

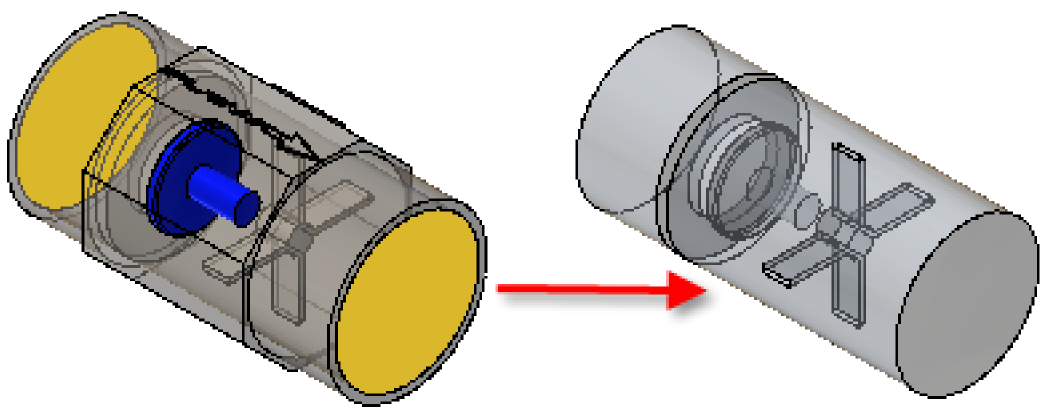
1. On the Manage panel, start the Interference command.
2. Select the components from the browser or graphics window. You can select individual components within a component assembly. An individual component is the default selection in the graphics window.
3. Optionally, select Include Coincident Faces. Any faces that touch are included as an interference.
4. Click Compute to analyze the selection. The Interference Results dialog box displays.
5. Select which interferences to keep. A component is created for each selected interference.
6. Optionally, deselect Show All Interferences to hide the interference previews.

7 Click OK to create a component for each interference. The components use the name from the Interference dialog box.

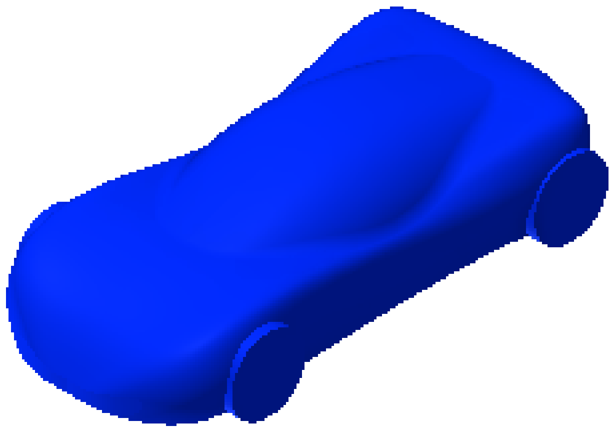
## Fluid Volume

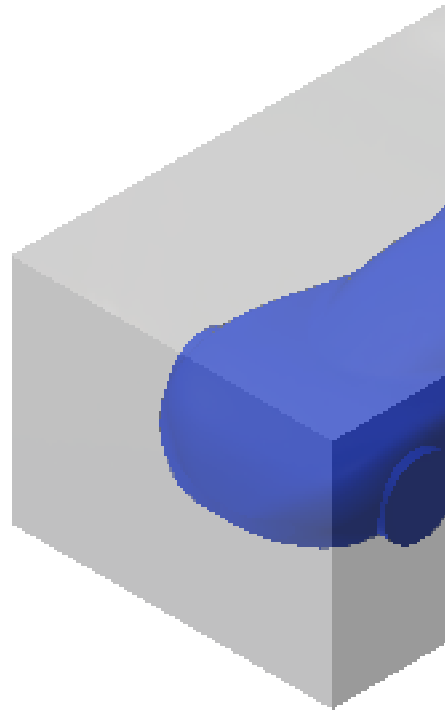

Use Fluid Volume to create a solid model based on the interior or exterior shape of the model. These models are typically used for CFD analysis.

The internal volume is a solid that matches the interior shape of the model. The model faces are copied to create the solid. Therefore, any openings result in a surface with missing faces. Cover (cap) the openings with a surface before you create the internal volume.



The external volume is a box, cylinder, or hemisphere with a void created by subtracting the model. If the fluid flow is only outside the model, cover any openings. Openings in the model result in the void with both the interior and exterior shape.





**Show me how to create fluid volume**

**Create surfaces to cover openings in models**

The internal fluid volume command requires surfaces that cover all openings in a model. The surfaces can be:

- Boundary patches that exactly match the opening.
- A work plane or surface that extends past the opening.
- A developed surface that conforms to a curved shape.

For more information about the modeling commands, see the links to topics, and view the Create Lofted Features video.

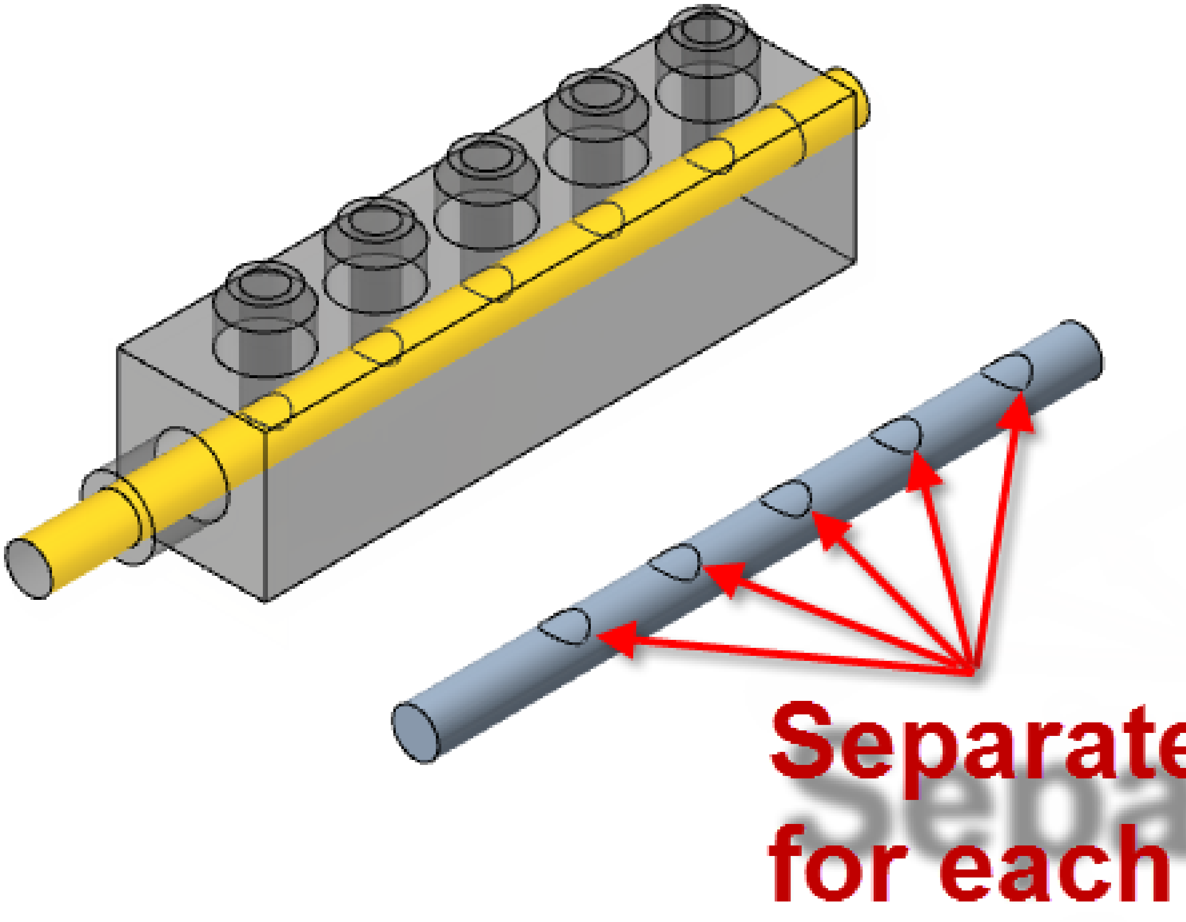
- **For a planar surface**, create an Offset Work Plane (page 157). If the work plane is inside the model, anything past it is not included. To remove sections of the model that are not required for simulation, create an internal bounding surface.

- **For an opening that has a connected set of edges**, use Patch/Merge (page 255). Create a surface that matches the opening. A Boundary Patch can also blend across a curved shape.
- **For more complicated openings**, use surfacing commands, such as Loft, Sweep, and Revolve (page 221).

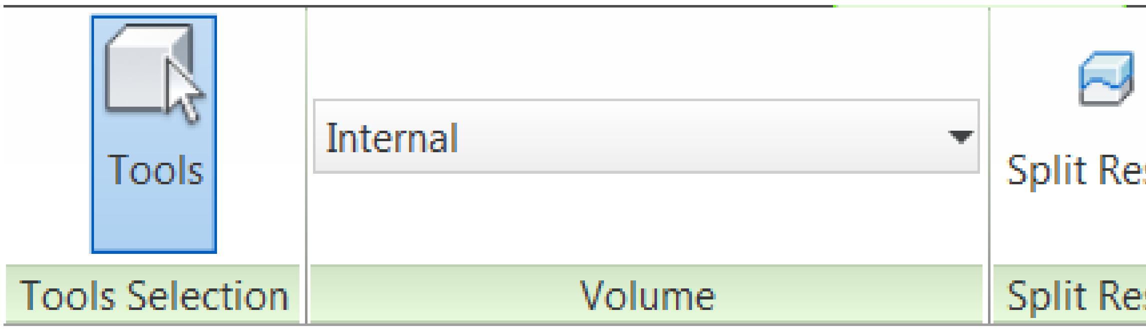
### Create separate faces with Split Result

The fluid volume command automatically merges tangent or smooth faces, so the surface cap for a hole can become part of the surrounding face. The Split Result option creates separate faces in the volume for each surface in the model. A typical use for split faces is to simplify geometry in a hydraulic manifold.

In the image below, a cylindrical extrusion covers the holes where each port intersects the main passage. A work plane at the end of manifold removes the fitting from the interior volume. By default, the interior volume is a simple cylinder. Split Result creates a separate face for each intersection of a port and the main passage.

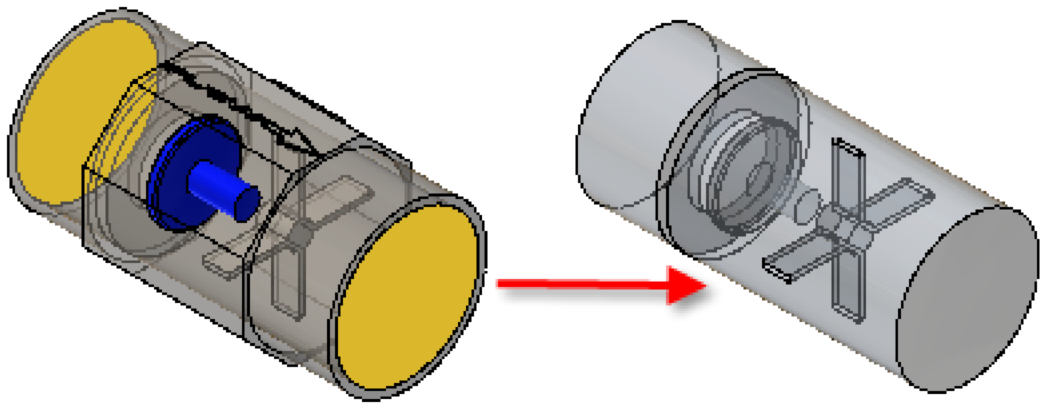

Separate
for each

### Create an interior volume




Use the Interior Volume option to create a solid body from the enclosed interior shape of the model. Multiple components can be used to create the solid.

Create surfaces to cover (cap) any openings in the model. In this image, the yellow faces are surface caps on the inlet and outlet ports, and the blue check valve is a separate component.



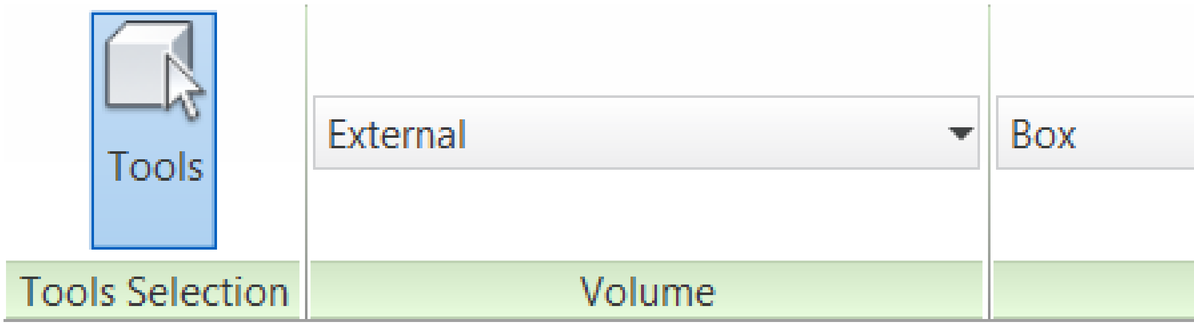
1. Create surfaces (caps) to cover any openings, if necessary.
2. From the Home tab, Simulation panel, start the Fluid Volume command.
3. Select the solid components and any surfaces. The error symbol displays until the internal volume is enclosed.
4. Set the Volume to Internal, if necessary.

5 If necessary, split the interior volume by clicking the Split Result button and selecting surfaces, work planes, or solid bodies that intersect the model.

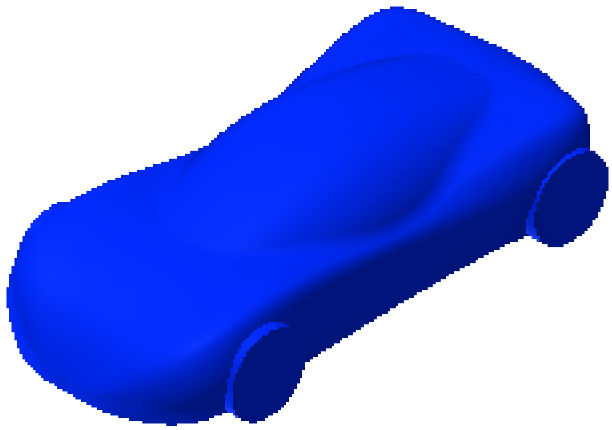
6 Click OK.

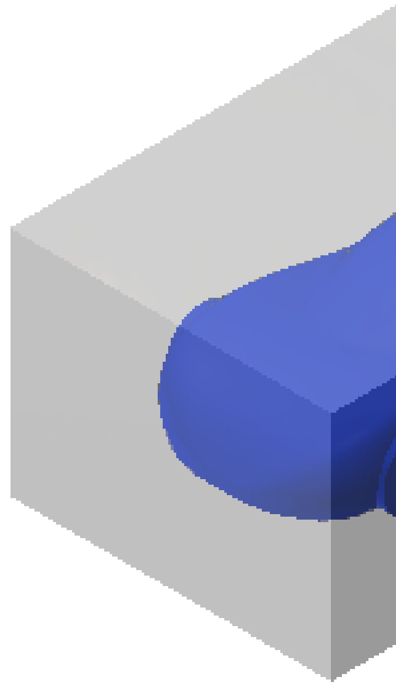
   A new component is created that matches the interior shape of the model. If you used Split Result, the model faces are split.

### Create an exterior volume




Use the Exterior Volume option to create a solid body that envelopes the model. The model is subtracted from the volume, leaving a void. Multiple components can be used to create the solid.



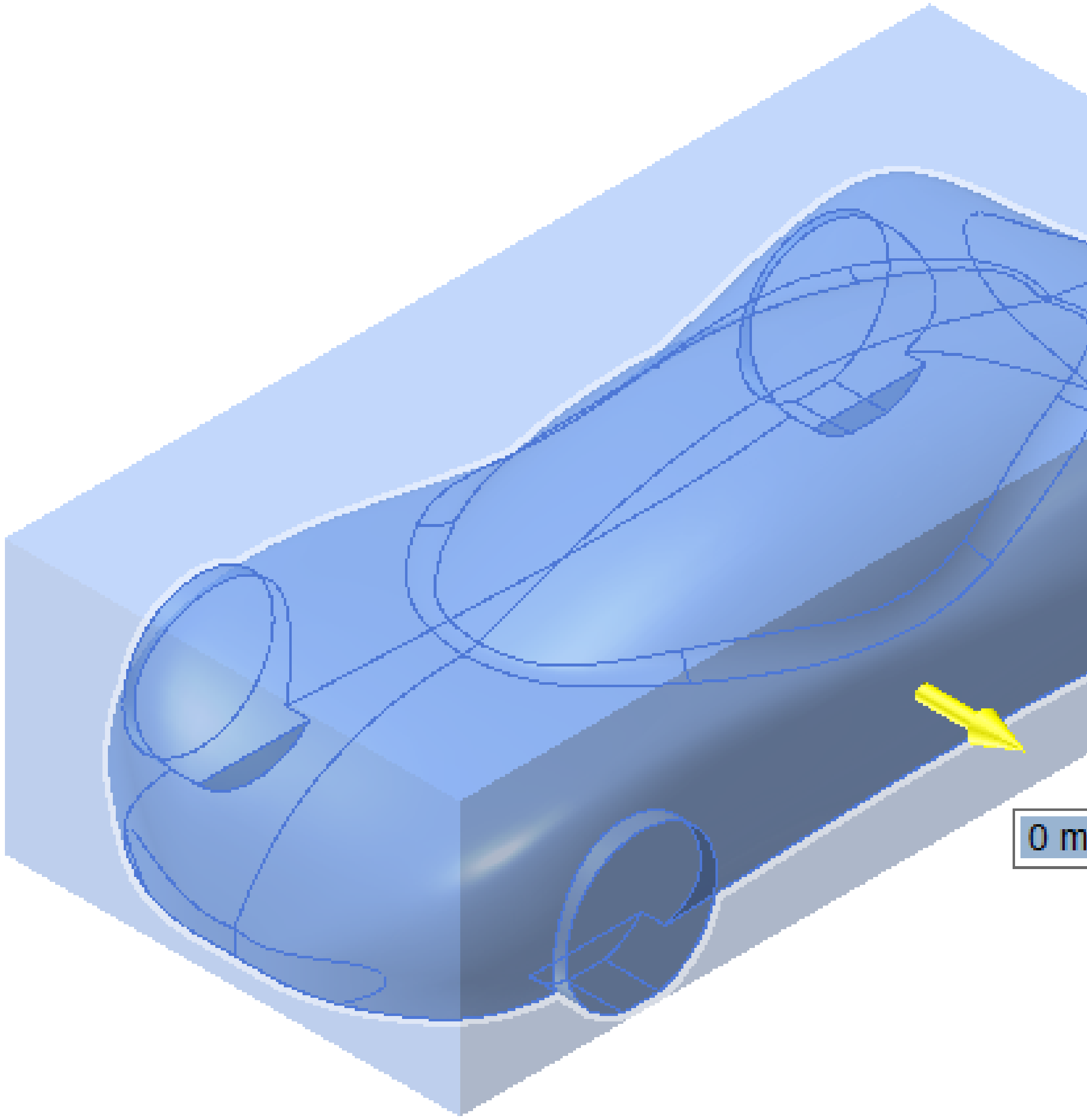


1 From the Home tab, Simulation panel, start the Fluid Volume command.
2 Select the solid components and any surfaces. If the analysis includes flow through the model, use the Selection Tool (page 38) to select interior components.

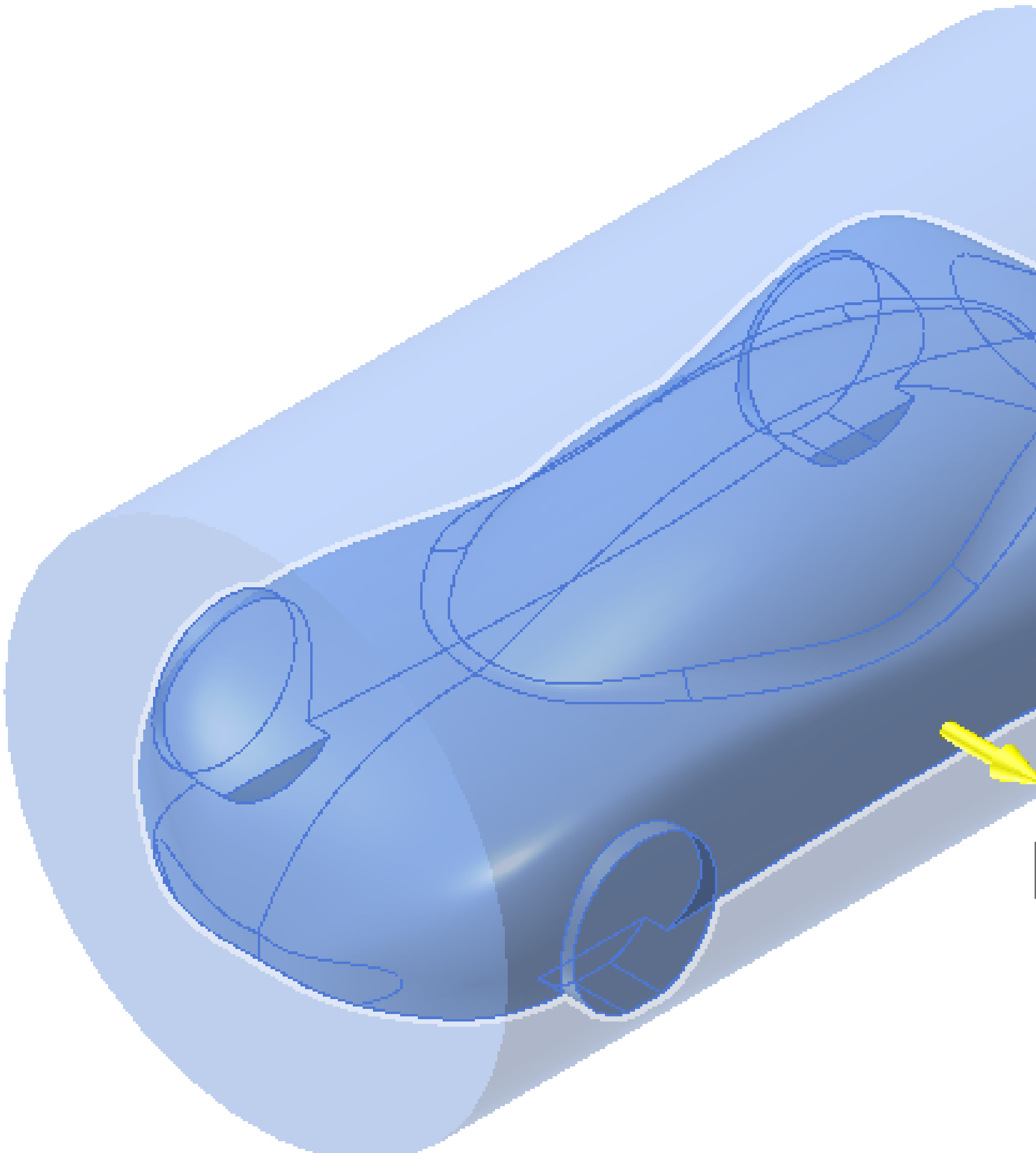
**NOTE** The solids and surfaces are also called toolbodies.

3 Set the Volume to External, if necessary.

4 Select the shape:



The **Box** faces are parallel to the origin planes.



The **Cylinder** axis is aligned with the X-axis by default. Change the alignment by selecting the Y-axis or Z-axis from the glyph in the mini toolbar.

The bottom of the **Hemisphere** is perpendicular to the X-axis. Change the alignment by selecting the Y-axis or Z-axis from the glyph in the mini toolbar. Click the Flip button to change the direction of the hemisphere.

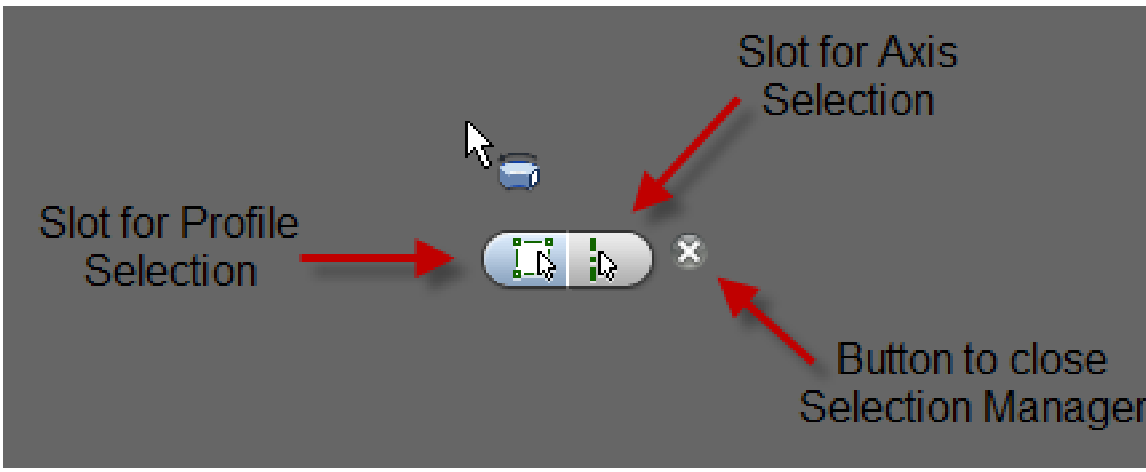
5 Drag the manipulator or enter a value for the Offset. The size of the box and the cylinder increases equally in all direction. The bottom of the hemisphere is fixed and the radius increases.
6 If desired, keep all model faces by clicking the Split Result button.
7 Click OK.

A new component is created with a void that matches the exterior shape of the model.

## Selection Manager

The Selection Manager helps you understand the possible single or multiple selection inputs required by a command to perform an operation in Inventor Fusion.

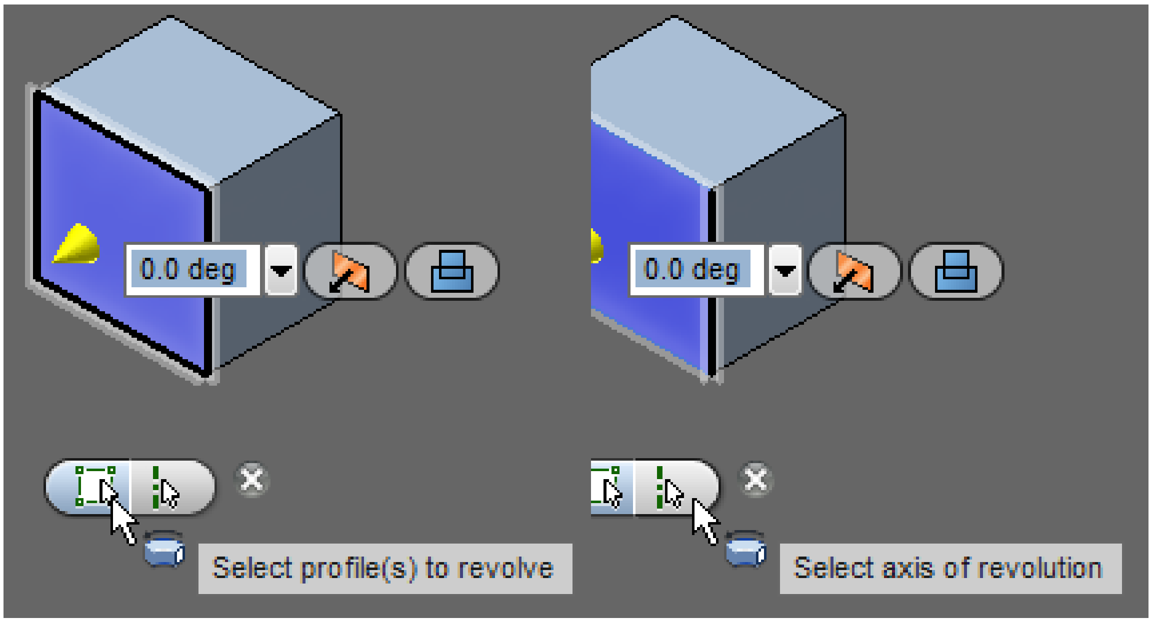
The Selection Manager contains a slot for each type of selection, and provides information about the selection inputs.




### Work with the Selection Manager

Pause the cursor over a selection slot, and the entities selected for it highlight.

Click a particular selection slot to activate it. The selections performed from this point onwards are considered for this active selection type.

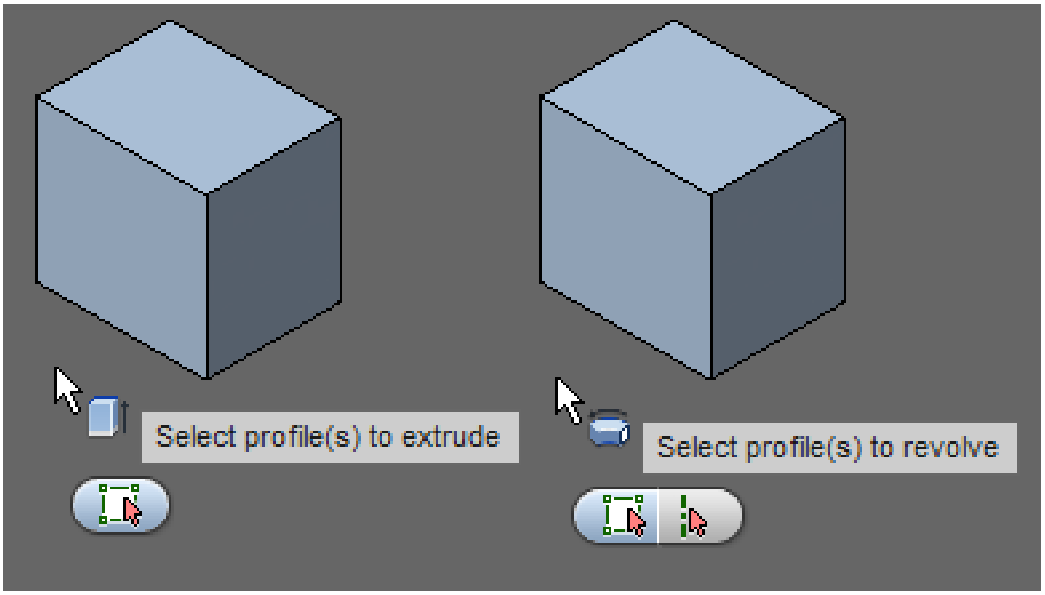



Entity selected for Profile highlights.

Entity selected for Axis highlights.

#### Feedback about selections

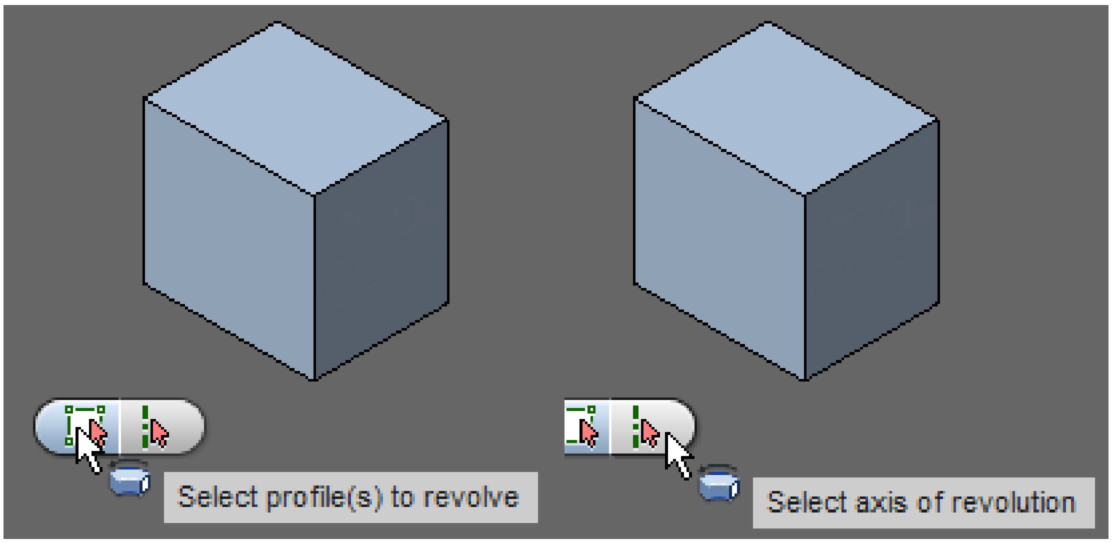
- Number of selections required by the command.



Extrude command requires one selection. Selection Manager displays a single slot.

Revolve command requires two selections. Selection Manager displays two slots

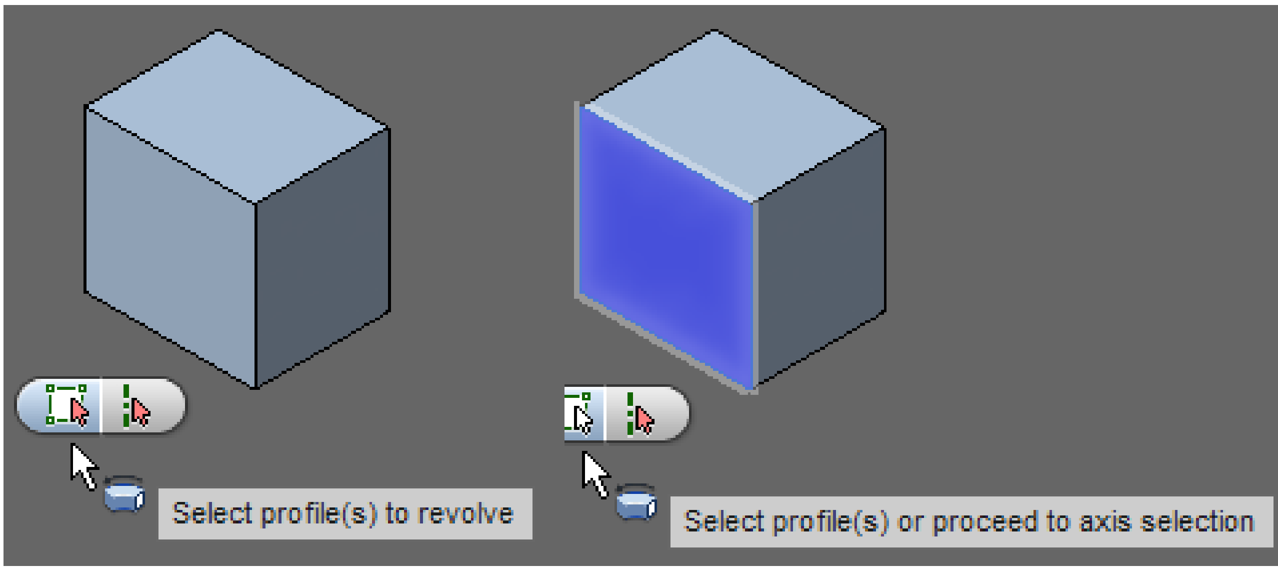
- The type of each selection.



First slot in Revolve represents selection for Revolve Profile

Second slot in Revolve represents selection for Revolve Axis

- Status of the selection - whether selection is satisfied.



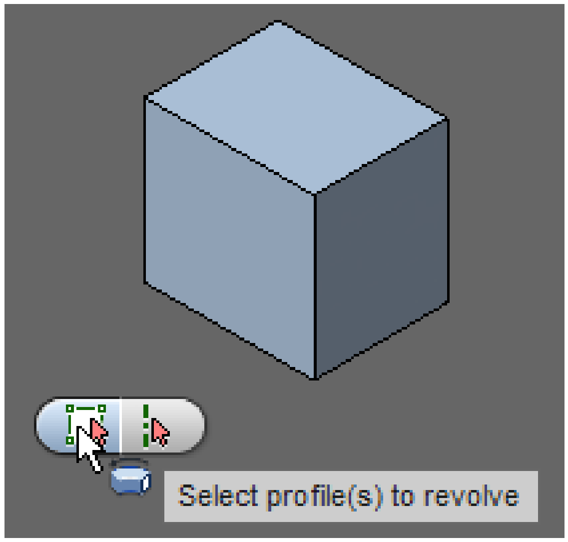

The Profile for this revolve is not selected. The indicator for the selection displays in red.

The Profile for this revolve is selected. The indicator for the selection displays in white.

- The active selection

When a command requires multiple input selections, at any given time, one of the selections behaves as the active selection. This means that the selection made is used for the active selection type.

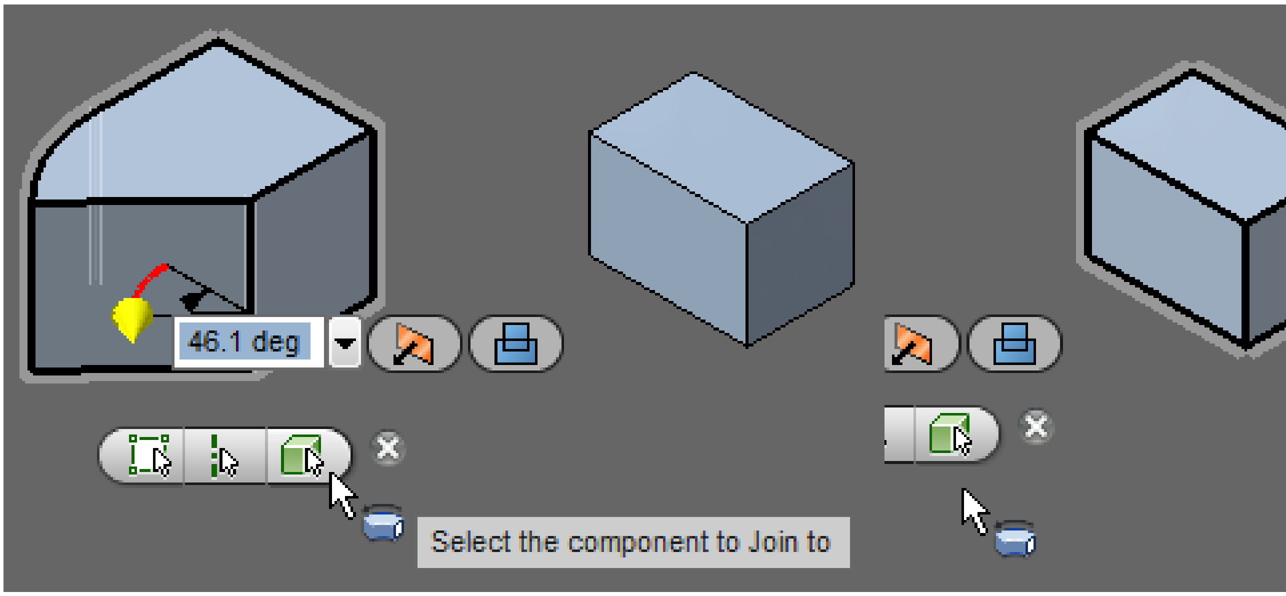
The Selection Manager automatically sets the active selection based on the user input in most of the cases. The active selection displays as a blue slot in the Selection Manager. If the Selection Manager cannot automatically decide on the active selection, you can click the selection slot to do so.




**Additional selections**

- Join to target component

For certain commands, such as Extrude, Revolve, Sweep, and Loft, a facility is provided to define the component to which the output of the command belongs to. This selection is called Join to target component.

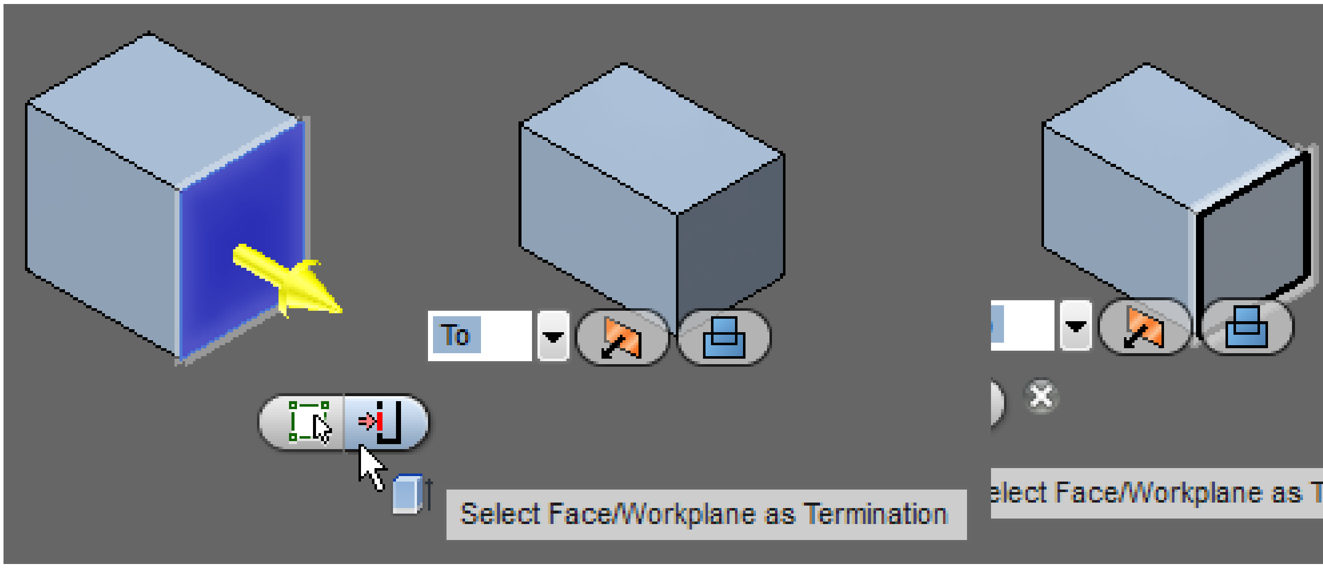



The Join to target component selection slot displays in the Selection Manager. It shows as selection satisfied, since the parent component is taken by default.

A different component is selected for the Join to target component selection, which is now highlighted.

- Termination

  For certain commands, you can define an entity as a limit. The modeling operation executes up to this limit. This selection is the Termination selection. When you choose the To option for a command, this selection type shows up in the Selection Manager.




The To option is chosen for Extrude command. The termination slot displays in Selection Manager.

The selection for Termination is successfully completed. The indicator changes color from red to white.

# Materials and Model Appearance

# 5

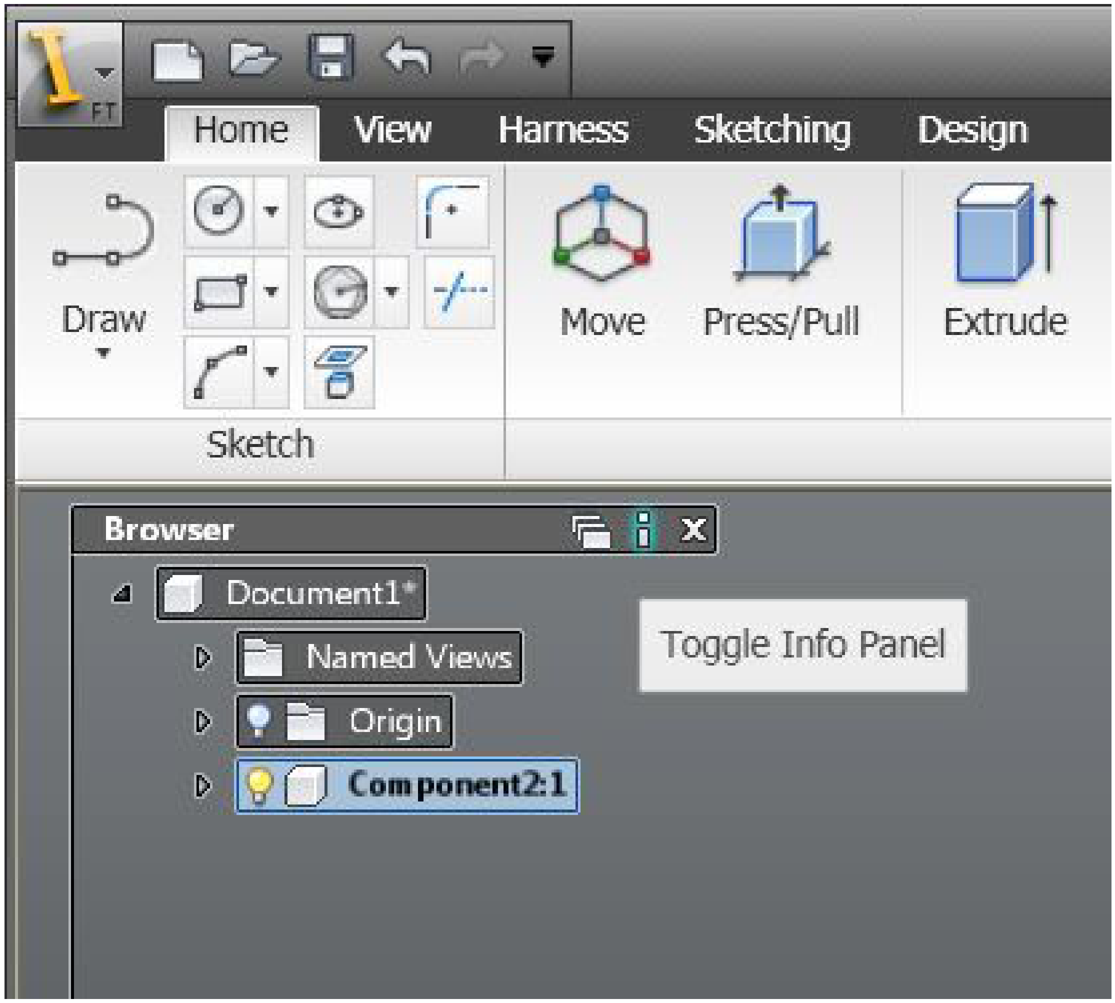
## Physical Materials

When you create a body or component in Inventor Fusion, a default physical material (material) is assigned to it.

It is not possible to create a body or component that has no material. If you import a body or component that does not have a material associated with it into Fusion, it is assigned a default material. Faces and other entities like Workplanes, Sketch profiles, and so on, do not have materials. Inventor Fusion supports all of the same materials as Autodesk Inventor. The default material in Fusion is Alloy Steel.

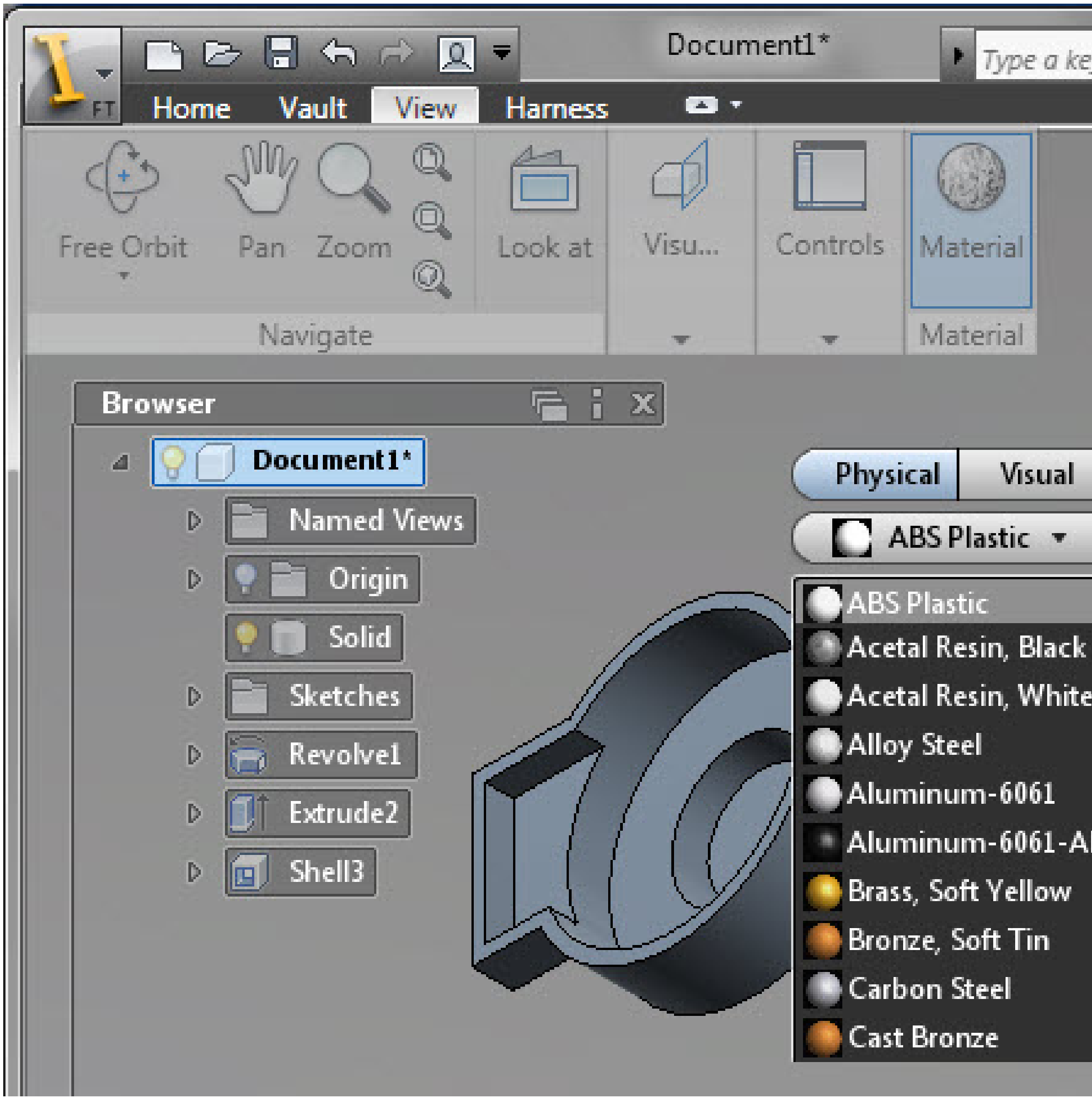
**Note** When importing a file from inventor, the part is assigned the material present in its iProperties. If it does not have an iProperty type of material, it is assigned the default Alloy Steel Material.

A body with a material has certain physical properties like density, volume, and mass, depending on the material it is made of. To see this information, in the Browser panel, click Toggle Info Panel.




With the Info Panel turned on, move the cursor to a browser node for a body or component to obtain information about physical properties. The mass and volume of a component is the sum of the mass and volumes of its child bodies and components.

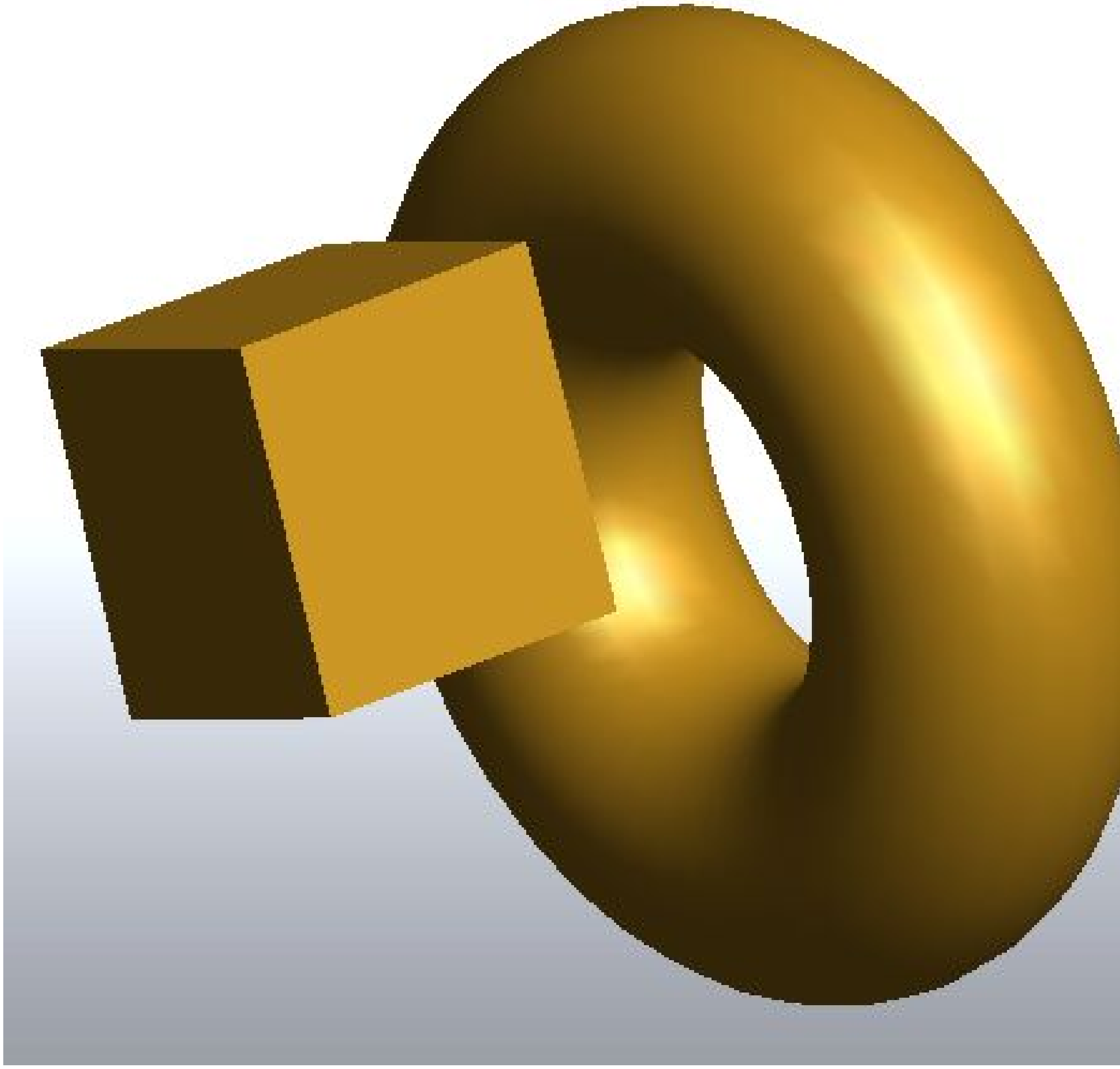
A body or component typically inherits the material of its immediate parent. You can set the entire document to have a particular material, and any object created under it has that Material. To override the material of child objects, select them, and on the context menu or the ribbon, View tab, use the Material command.



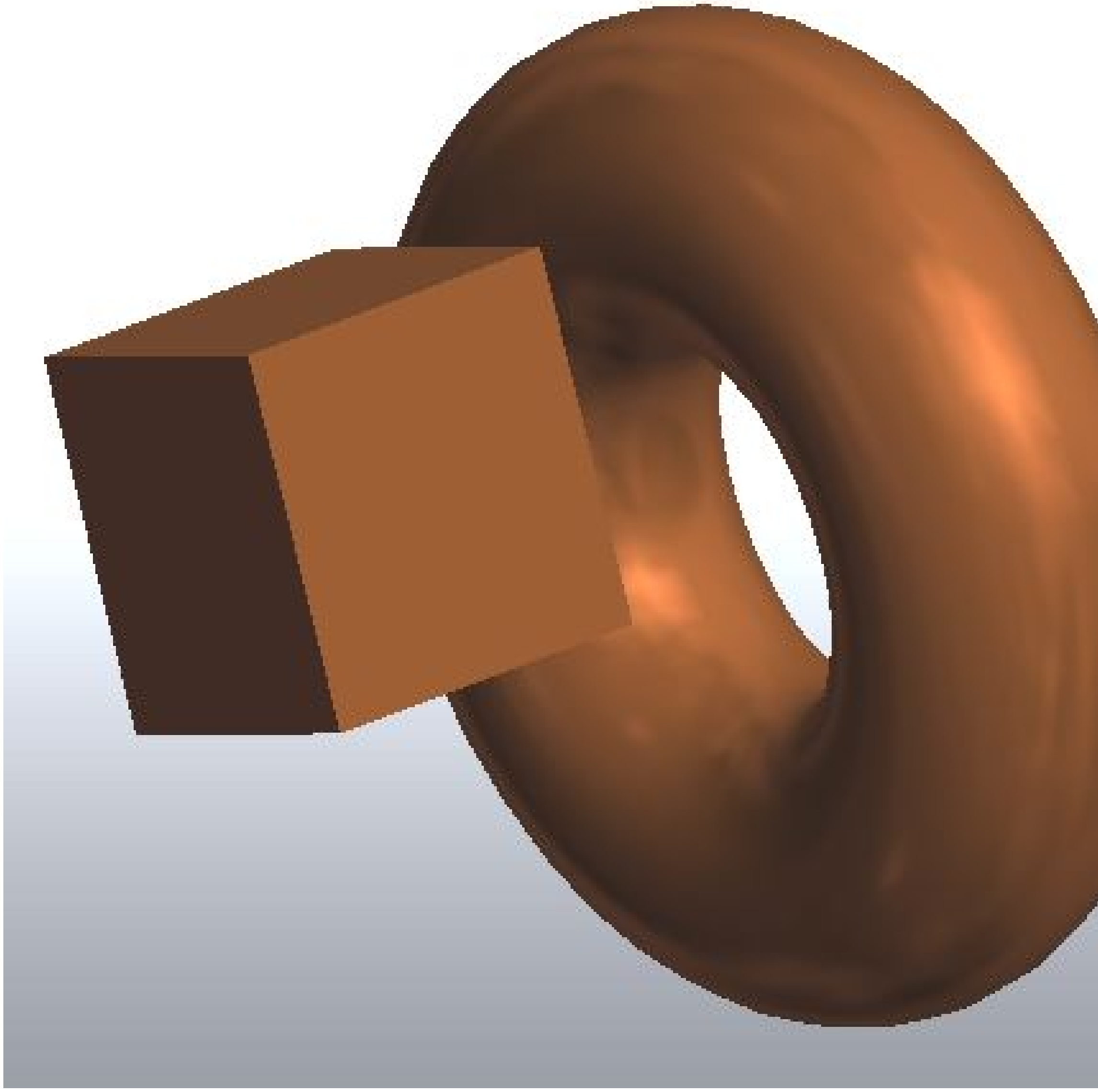

On the Physical material drop-down list, select a material.

Every Material has a default appearance associated with it. When a component or body is assigned a Material, it also assumes the appearance of the Material.

This drop-down menu is enabled only if one of the selected items supports Materials. You can pick any item from the drop-down, and it is applied to all valid selections.



**Brass**


**Copper**

You can change the appearance of a component, body, or face using Appearance Overrides. For more information, see Appearance Overrides (page 420) on the Appearance page.

# Appearance

The Autodesk Material Library 2011 provides the appearances for Inventor Fusion.

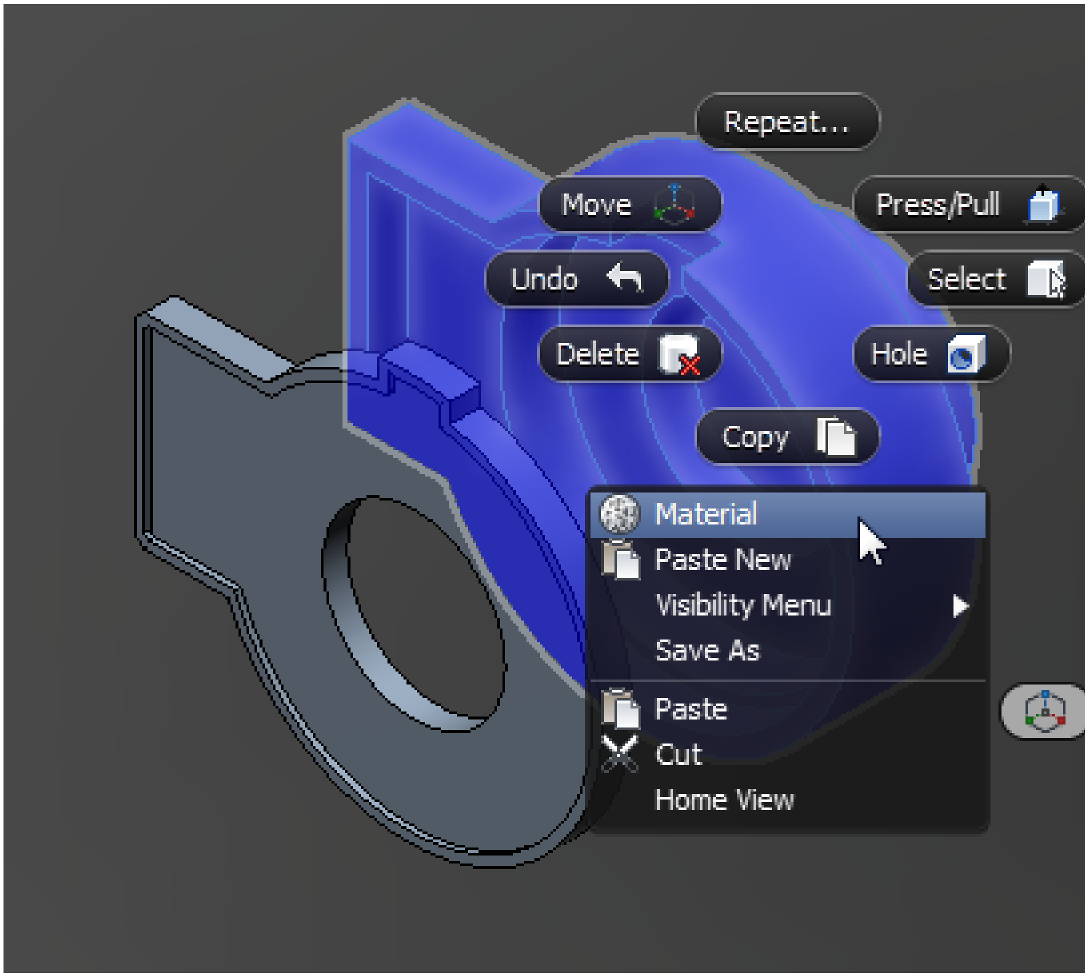
A graphics card that supports Autodesk Feature level 3_0 or higher is required to render realistic materials. Cards that support Feature level 2_0 (the minimum required to run Inventor Fusion) show low-quality materials that look different from the Realistic Materials. For more information, see the System Requirements (page 567).

Alloy steel is the default material for the appearance of objects in Inventor Fusion. You can override the appearance at the component, body, or face level. Changes in appearance of an object, such as from copper to aluminum, do not affect physical properties, such as mass.

**NOTE** When importing a file from Inventor, the appearance overrides on faces are imported. The appearance overrides on the body are not imported, and remain the same as the active appearance that you saved in Inventor.

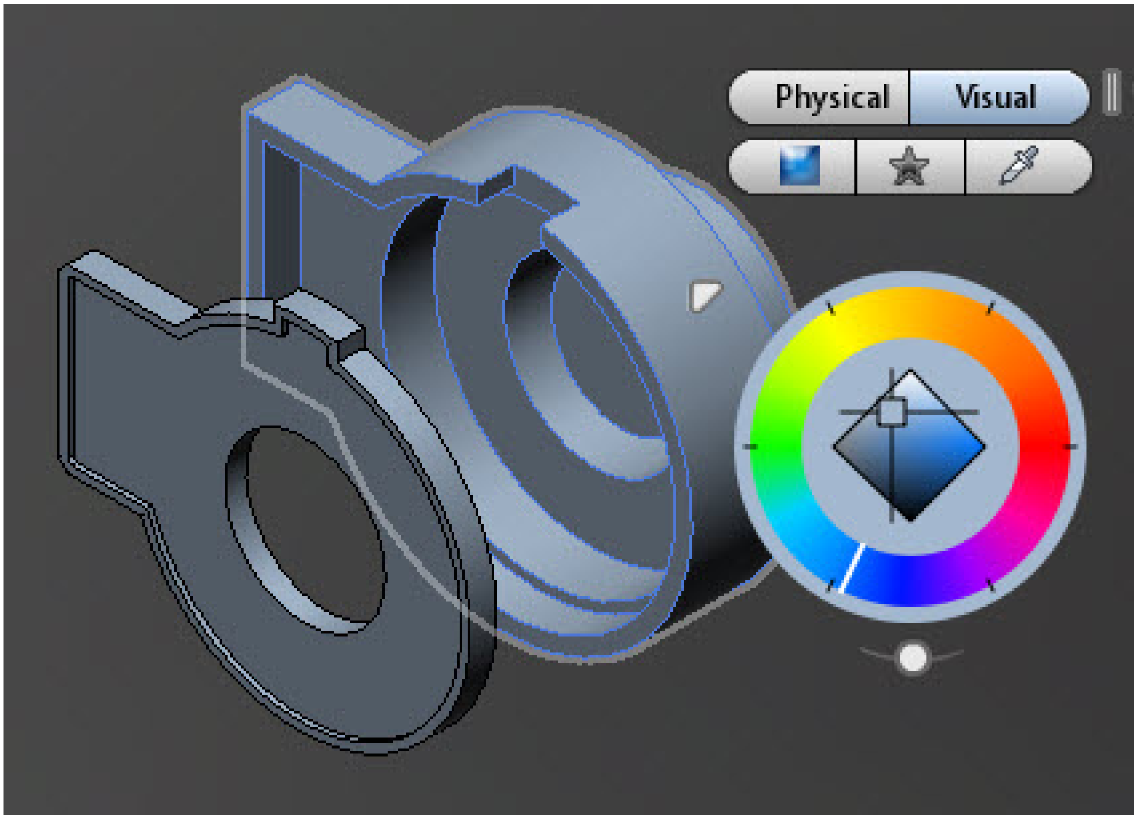
**Change the appearance of an object**

1 Select the object in the canvas or in the browser.
2 On the ribbon, View tab, or context (right-click) menu, select Material.

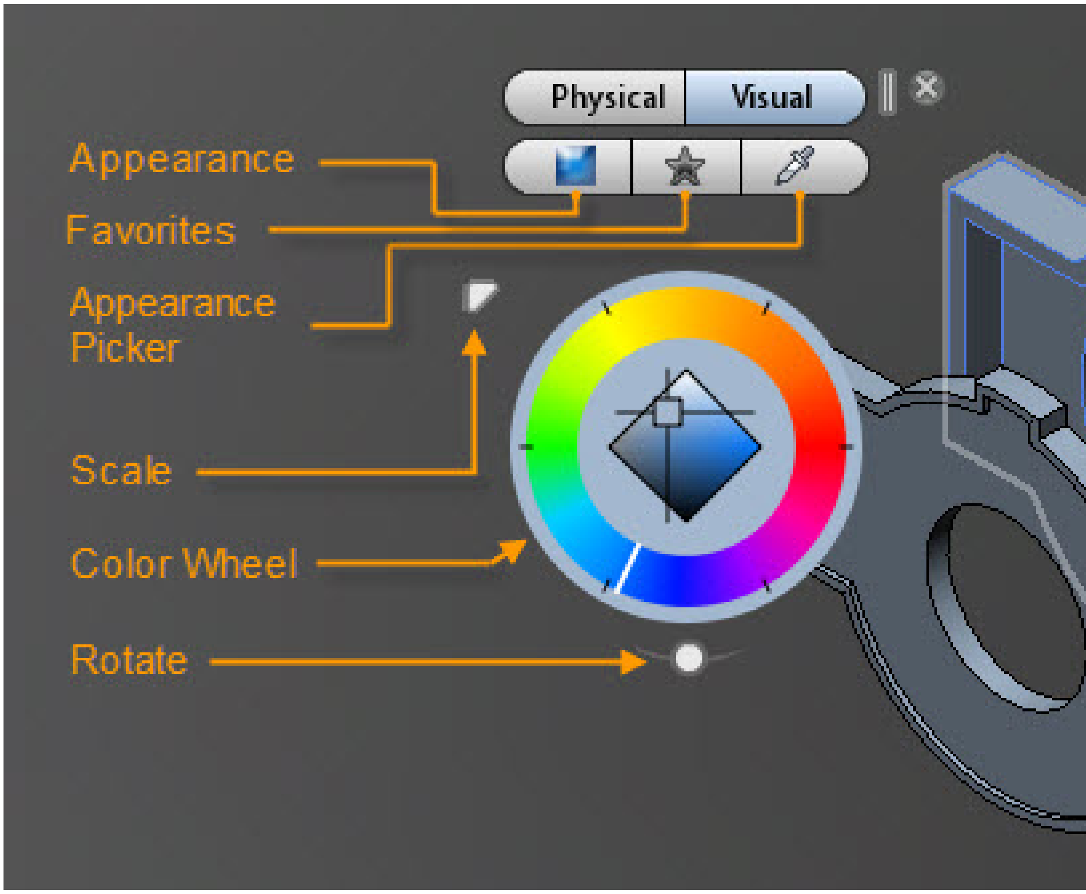



With the Material command, you control physical material and appearance.

3 To change the appearance of the selected objects, in Material, select Visual.




The Visual portion of the Material command provides a number of controls to change the appearance of the selected objects.




**Descriptions of commands**

### Appearance

Adjusts the appearance of selected objects. Select an appearance type from the list. Overrides the appearance of the physical material, but does not change physical properties. To return to the default physical material appearance, click Reset. To select an appearance from the Autodesk Material Library, click Autodesk Library.

### Favorites

Provides access to saved appearance favorites. To save an appearance for reuse, click Add to favorites. Automatically stores recently used appearances.

### Appearance Picker

Copies the appearance of an existing object. Click the Appearance Picker button, then select a face which has the desired appearance.

### Scale

Scales any applied texture. To adjust the scale factor of the applied texture, click and drag the manipulator.

### Color Wheel

Adjusts the color of selected objects. The outer portion of the wheel adjusts the color. The inner portion of the wheel adjusts brightness and saturation. Adjustments override the appearance of the physical material, but do not change physical properties.

### Rotate

Rotates the applied texture. To adjust the angle of the applied texture, click and drag the manipulator.

## Customize appearances

You can customize appearance with the tools outlined previously, or you can select a standard Autodesk appearance. In the Appearance menu, click Autodesk Library. The Material Library dialog box displays all of the appearances by category.

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**NOTE** The same appearances are available in Autodesk Inventor. You cannot edit a standard appearance.

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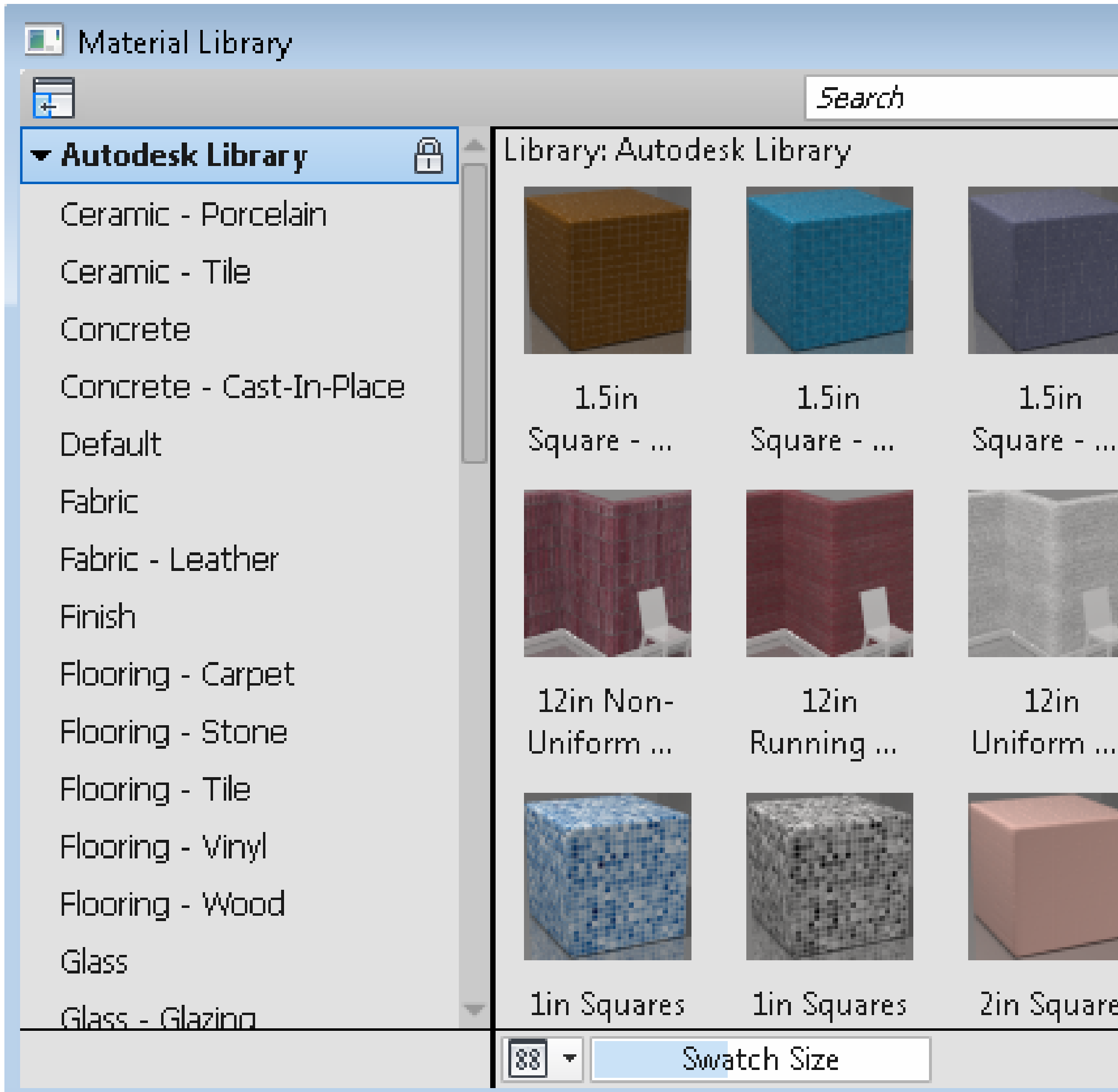
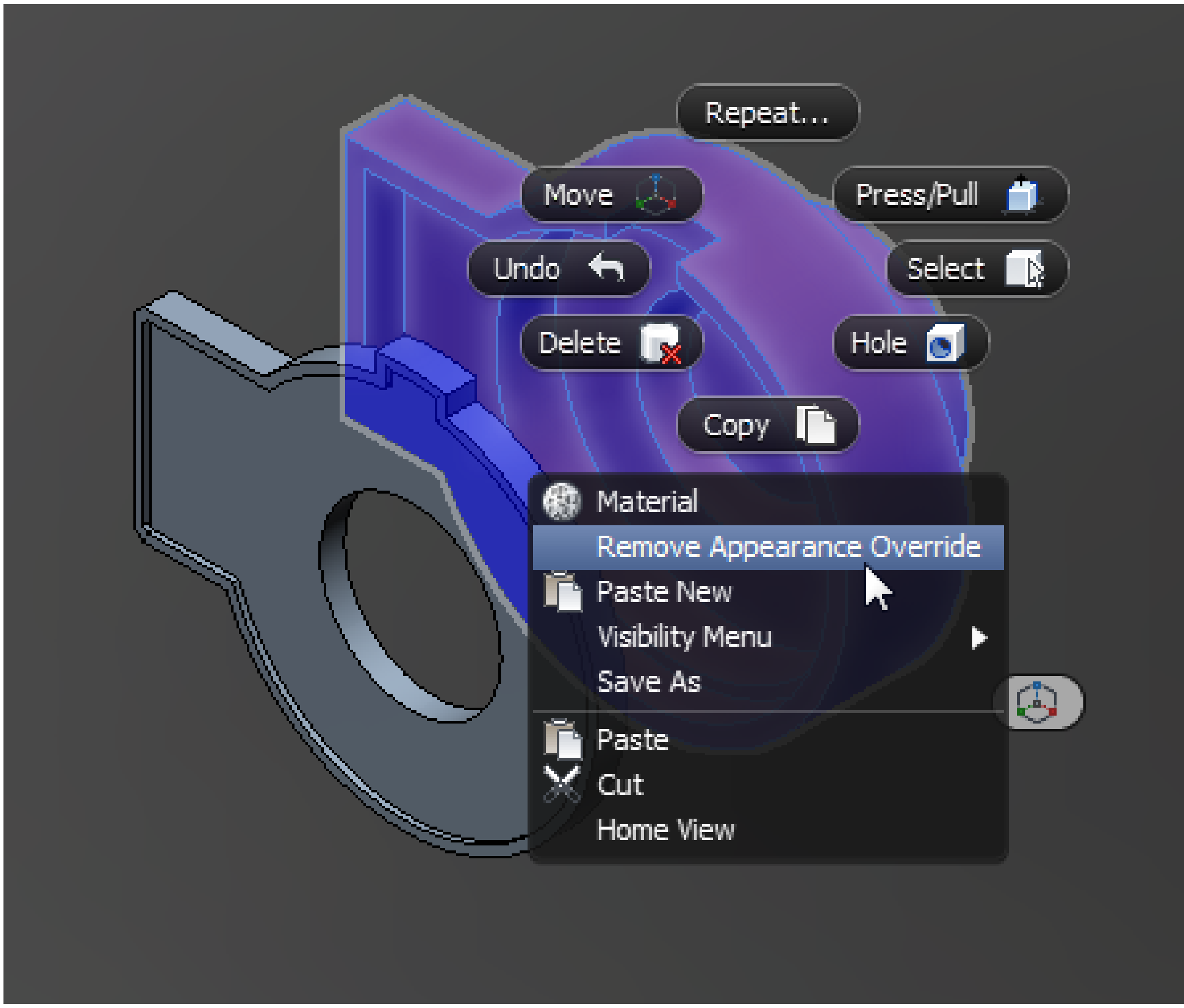



Move your cursor over any color square to apply an appearance preview on your selected objects. To apply the appearance to your selection, click, double-click, or right-click any square. and select Assign to Selection.

To remove an appearance override of an object, select it in the canvas or in the browser, and right-click. On the context menu, click Remove Appearance Override.

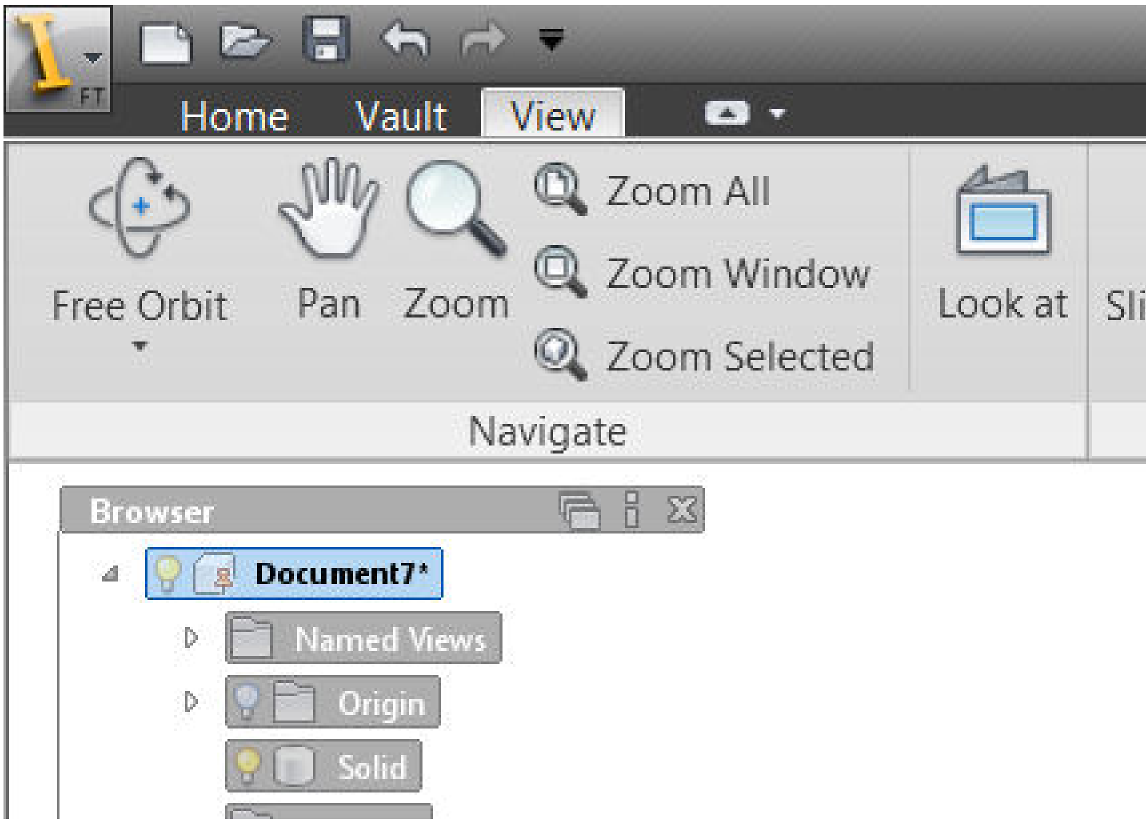



**Appearance Override priorities**

Appearance overrides applied at a component level have the highest priority, and affect the appearance of their children. The second priority is faces, and the last priority is bodies. If a face has an appearance override, changing the appearance of the body does not remove the override. A change to the appearance of the component removes the override.

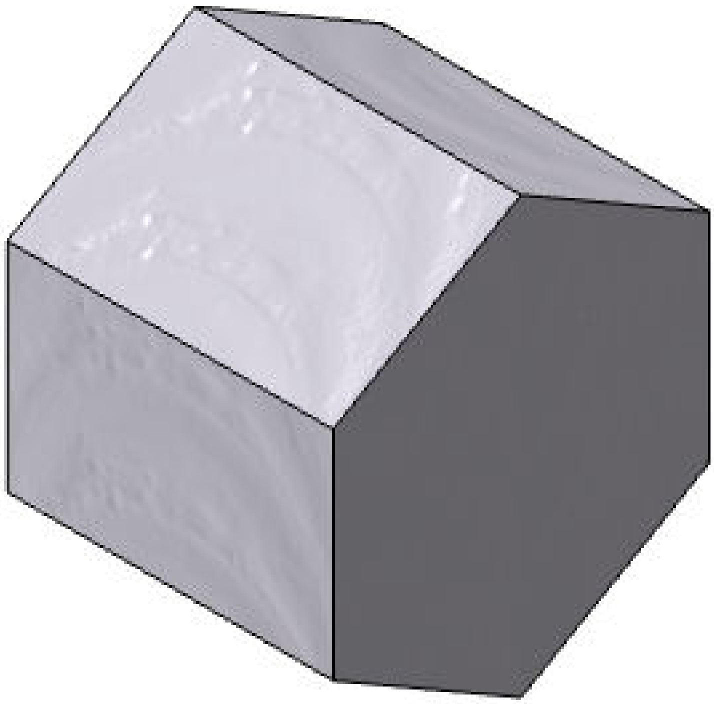
## Visibility of Edges

You can turn visibility of edges on or off in your model. On the ribbon, View tab, Visual panel, use the drop-down menu. Edge display is an application setting. Your current setting is restored the next time you start the application.

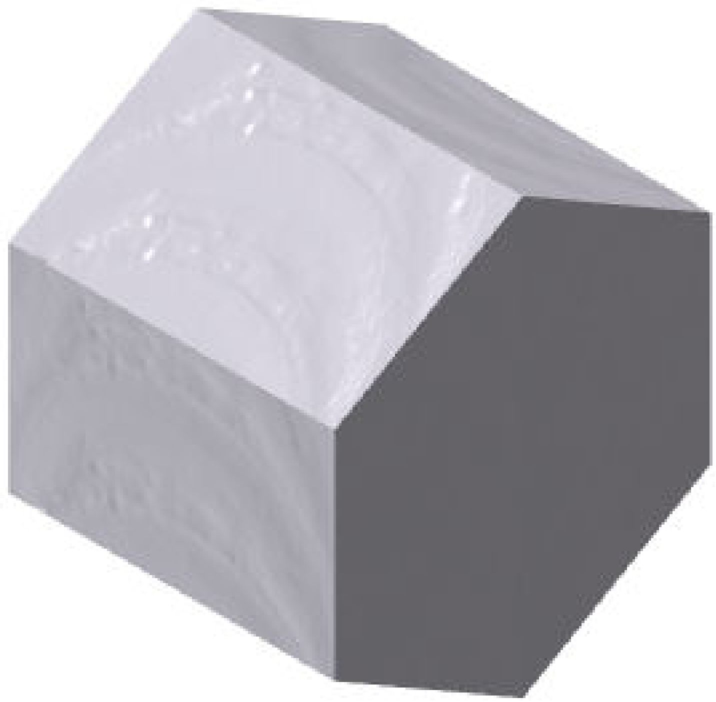



The Shaded With Edges option draws both faces and edges. The Shaded option draws only faces. Inventor Fusion does not support a Wireframe (Edges-only) style.

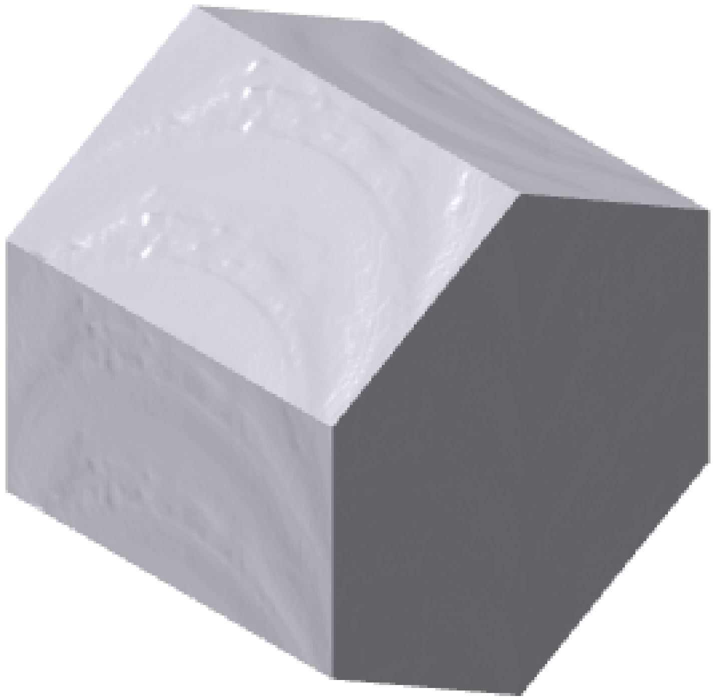
The **Shaded with Edges** option draws both faces and edges.



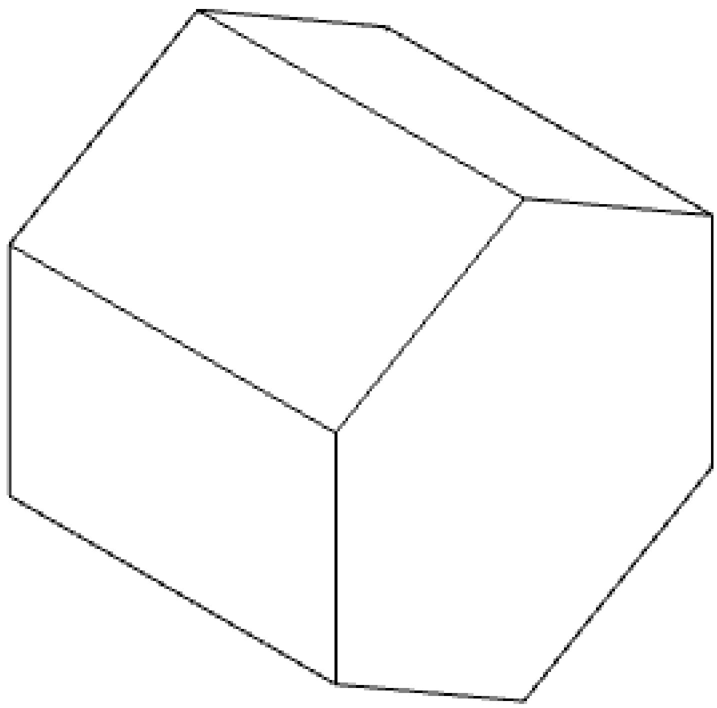
The **Shaded** option draws only faces.



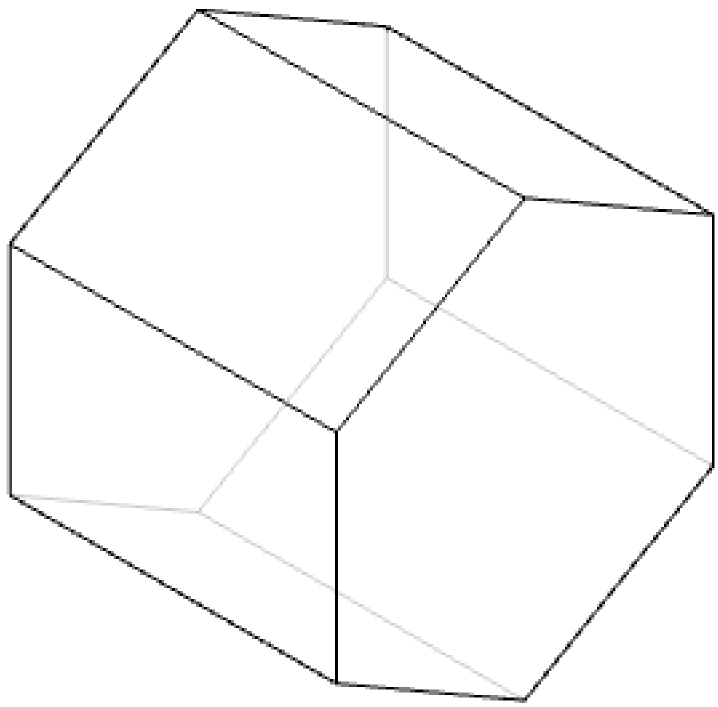
The **Realistic** option displays a realistically textured model with high quality shading.



The **Wireframe** option displays model edges only.

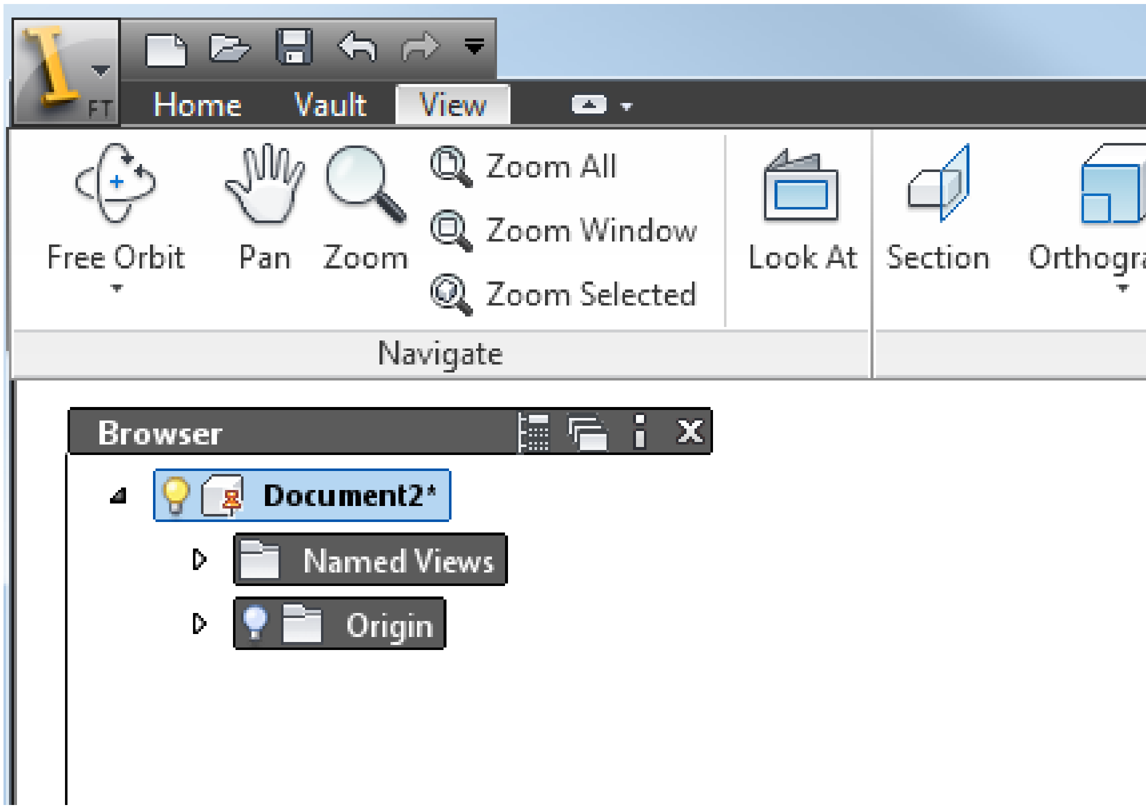


The **Hidden Line** option displays model edges with hidden edges shown.



## Effects

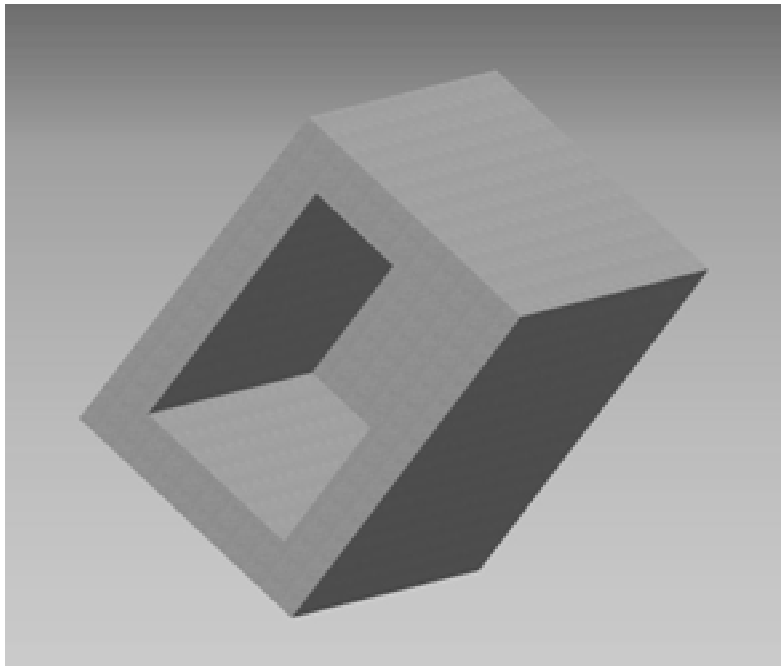
Visual effects are available on the ribbon View tab, Visual Styles panel.



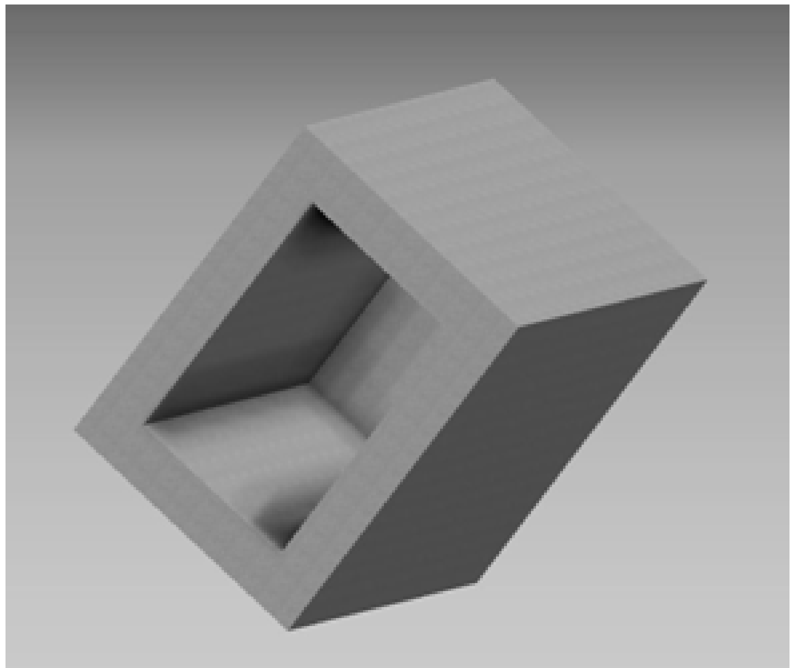

All effects are turned off by default. You can turn any effect on or off by clicking the checkbox. Effect settings are application-specific and are saved and restored the next time you launch the application. Active visual effects can slow down performance.

**Ambient Occlusion**

Ambient occlusion takes into account the attenuation of light due to occlusion. In the simplest terms, when an object obscures any object area, that object area receives less light and appears darker. When active, Ambient Occlusion degrades performance noticeably.



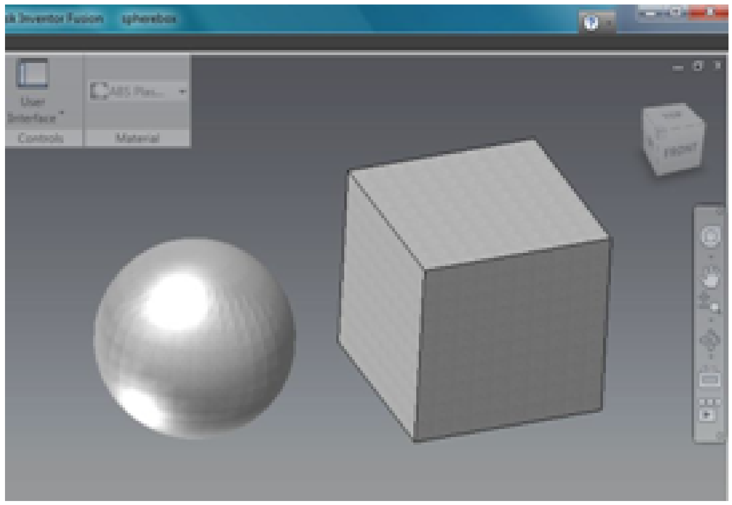
**Ambient occlusion off**



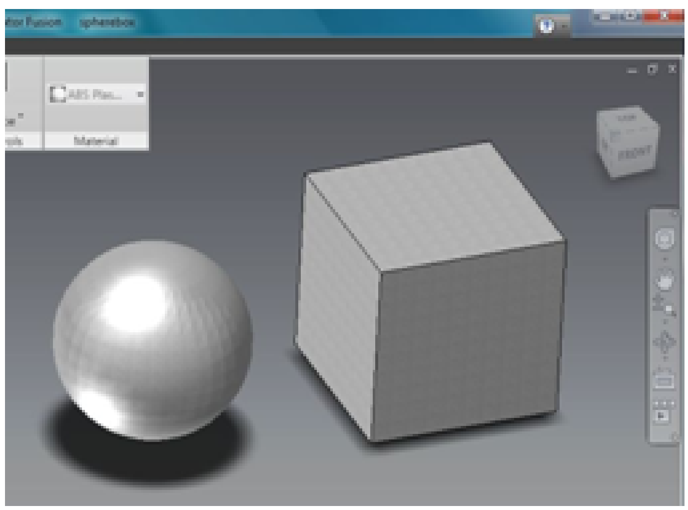
**Ambient occlusion on**

### Shadow

Shadow casts a shadow of the object on the ground as though a light is directly overhead. The orientation of the ground plane is fixed and is the bottom plane as defined by the View Cube. As geometry in the scene changes (components are added, deleted or removed) the ground plane adjusts to keep its position below all the objects in the scene. Its orientation does not change. However, if visibility of any geometry such as a feature or surface is turned off, no shadow is cast. Work features, annotations, and sketch geometry are not visible in shadows.



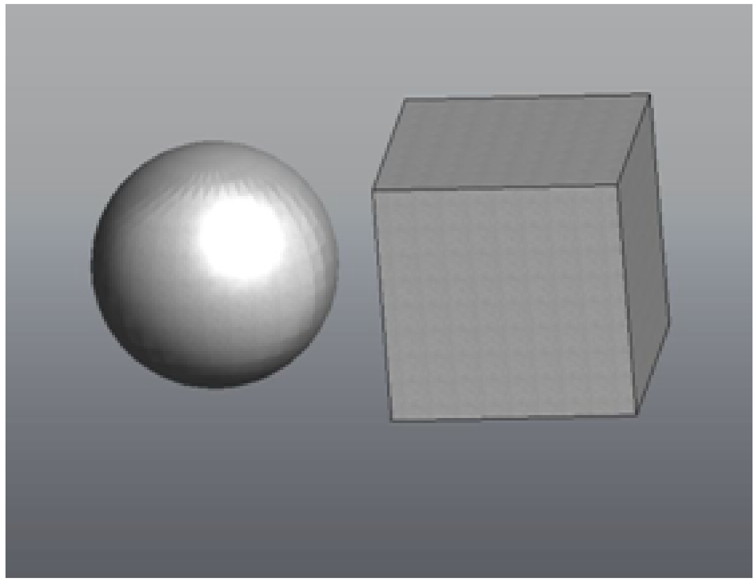

**Shadow off**



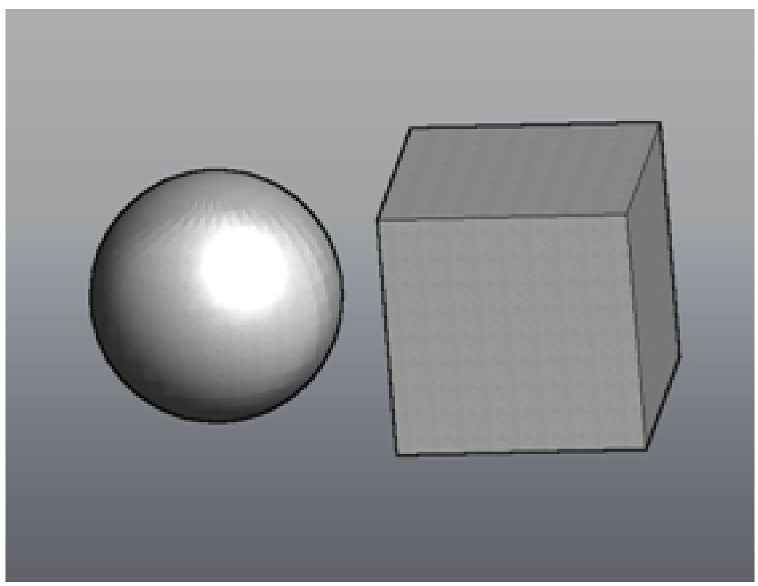

**Shadows on**

### Silhouette

Silhouette draws an outline on the objects in the scene. This effect is applicable only if your rendering setting is Shaded with edges. If your setting is shaded, and edges are not visible, this effect does nothing. Although it has no edges, a sphere has a circular silhouette,.



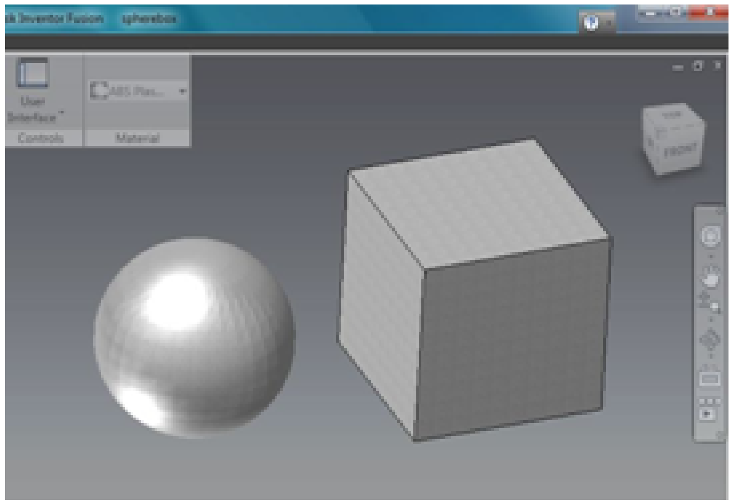
Silhouette off



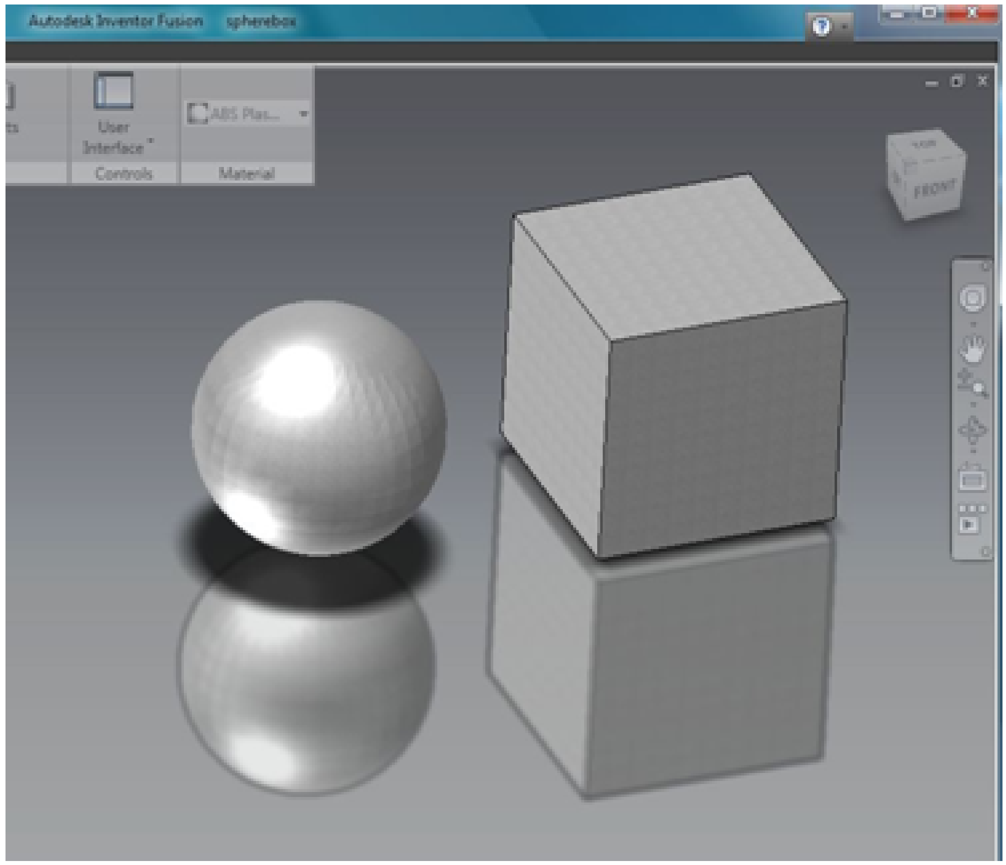
Silhouette on

### Floor Reflections

This effect casts a reflection on the floor.



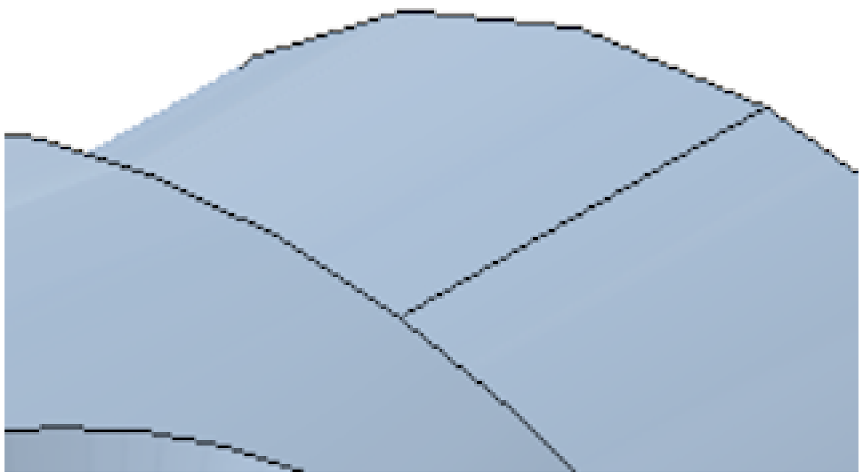
**Floor Reflections off**



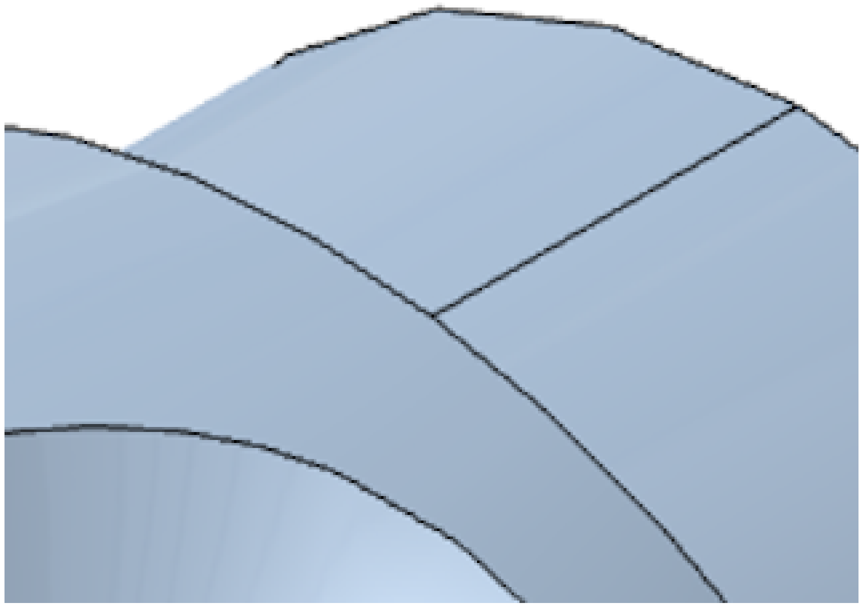

Floor Reflections on

## Anti-Aliasing

This effect reduces the pixelation in the graphics window.



**Anti-Alias off**

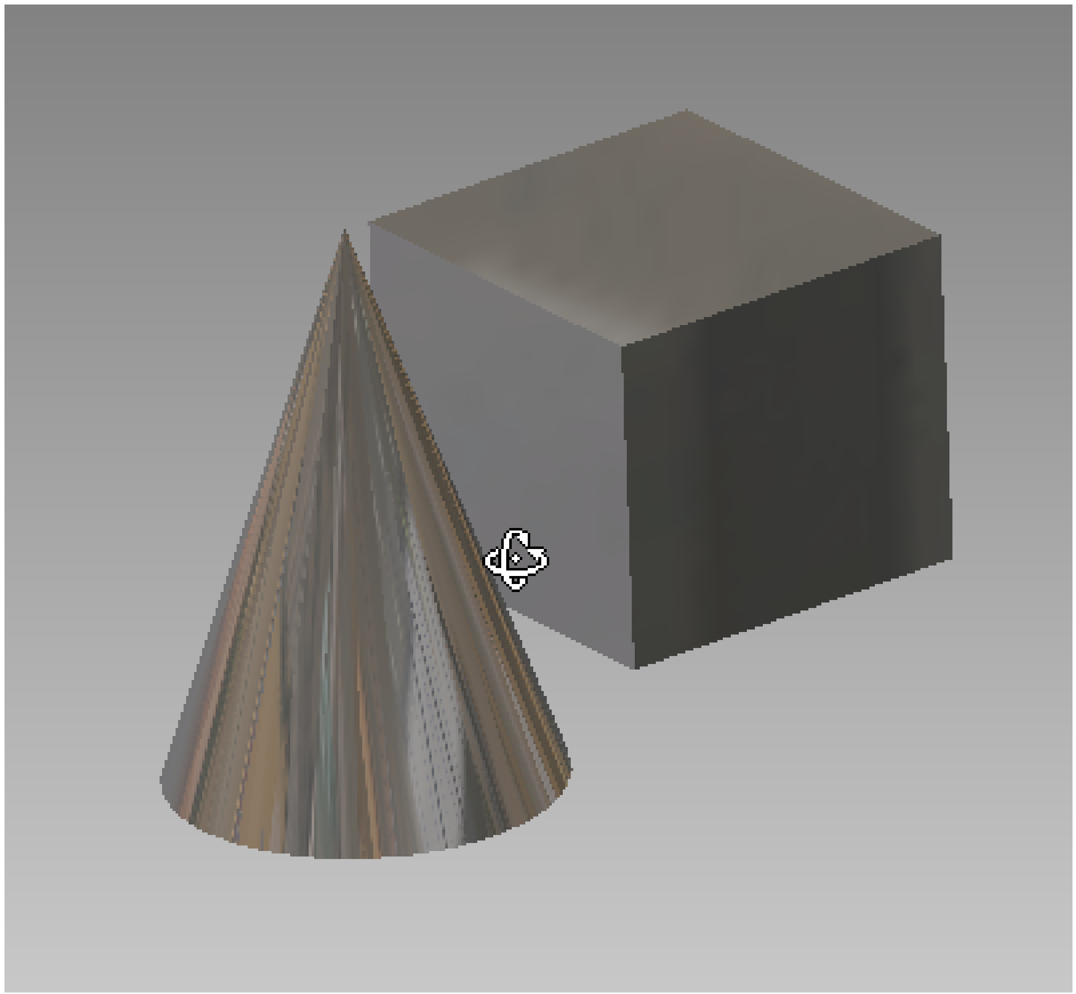


**Anti-Alias on**

### Adaptive Performance

Turn On Adaptive Performance to optimize the graphics for performance. For example, Effects are disabled during the view operations.

Adaptive Performance off



Adaptive Performance on

You can produce realistic looking images using Effects in combination with realistic appearances.

# Views of Models

## Orthographic Views

You can set the view to Orthographic using a drop-down list in the View Tab, Visual Styles panel:


Use orthographic camera mode to confirm visually or compare the relative dimensions of entities.

In orthographic camera mode, a model displays so that all of its points project along lines parallel to their positions on the screen. All same-length parallel edges display as the same length, even when you orient them with one edge closer to you than the other. In orthographic camera mode, a 3D model appears flat and unlike objects observed in the real world.

**Note:** The term camera mode indicates only the particular view method used for models in the graphics window. You cannot record actions that take place in the graphics window by choosing either orthographic camera mode or perspective camera mode.

## Perspective Views

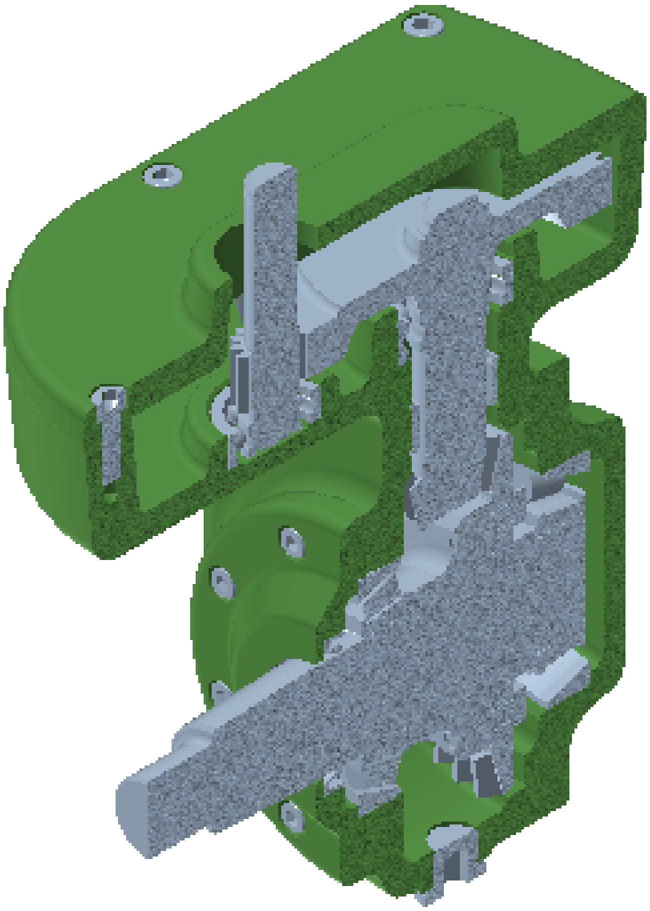
You can set the view to Perspective using a drop-down list in the View Tab, Visual Styles panel:

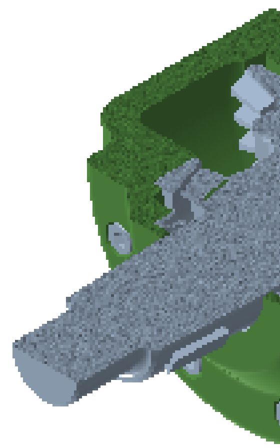

In perspective camera mode, the model displays as a human eye sees it. Objects further away from the camera appear smaller than objects closer to the camera. If two lines are parallel and are along the line of sight, they appear to converge as they move further away from the camera.

Use Perspective Camera mode to do a realistic walkthrough of a scene.

**Note:** The term camera mode indicates only the particular view method used for models in the graphics window. You cannot record actions that take place in the graphics window by choosing either Orthographic Camera mode or Perspective Camera mode.

## Section Command

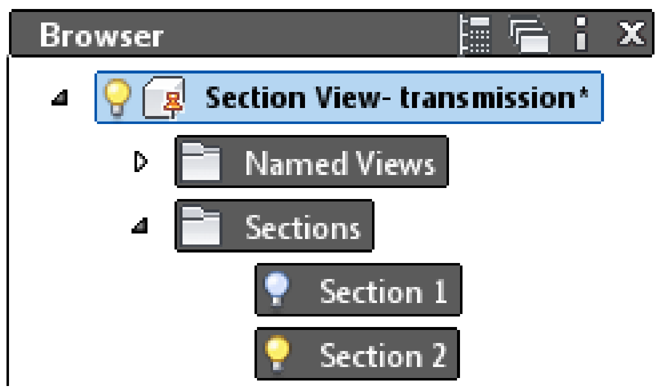


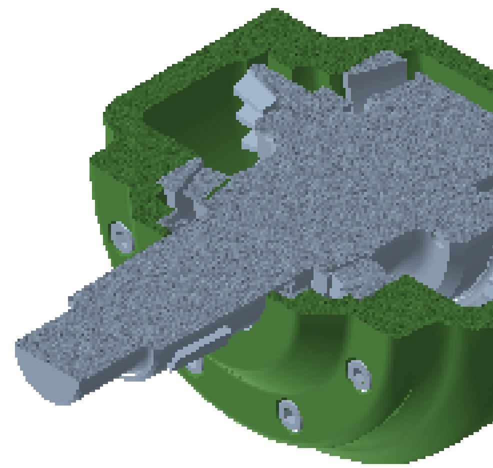


The Section command creates a view of the interior of a component by hiding the model graphics on one side of a selected plane.

You can use the manipulator to change the offset and angle of the section plane.

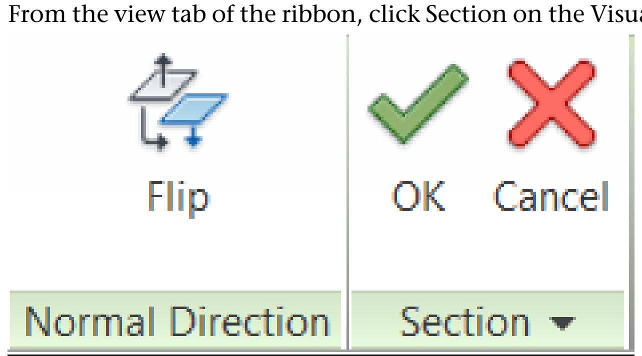
Section views are saved in a Sections folder in the browser. Click the light bulb to activate a section view. Only one section view can be active at a time.



### Create a section view

1 From the view tab of the ribbon, click Section on the Visual Styles panel.

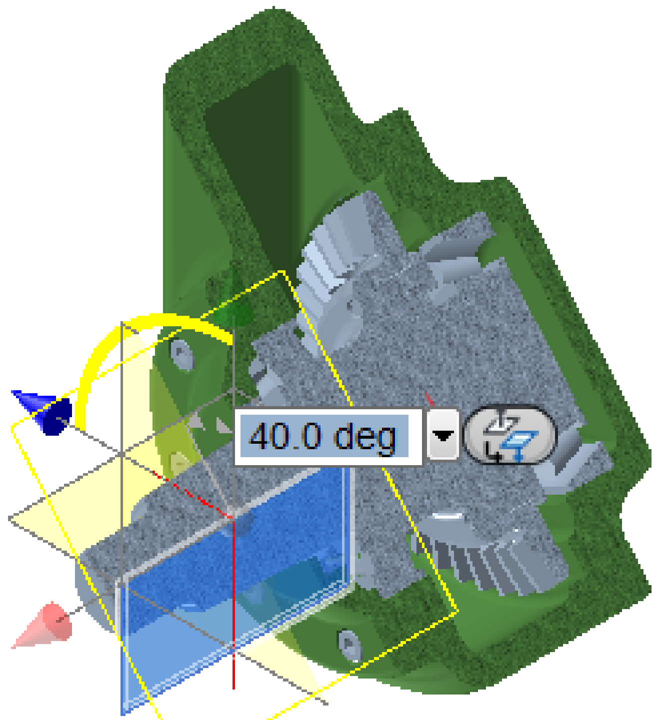



2 Select a work plane, planar face, or planar surface.

3 Select Flip to choose which side of the graphics is hidden.

or

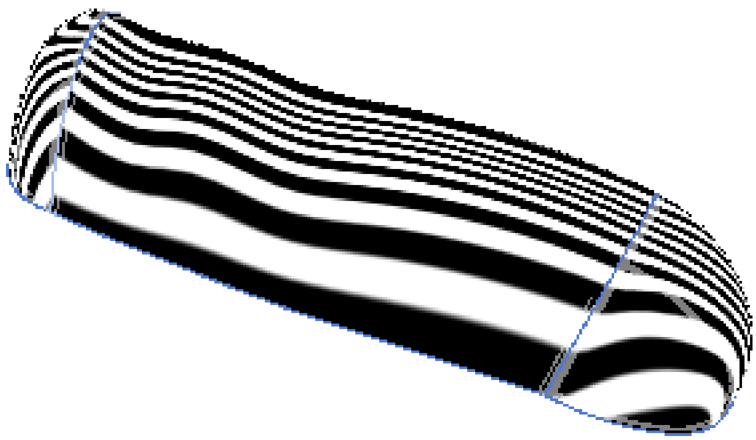
Use the manipulator to change the offset and angle of the section plane.




4 Click OK to create the section view.

5 Expand the Sections folder in the browser and click the light bulb to turn off the section view. If you turn on the visibility of another section view, the current one is turned off.

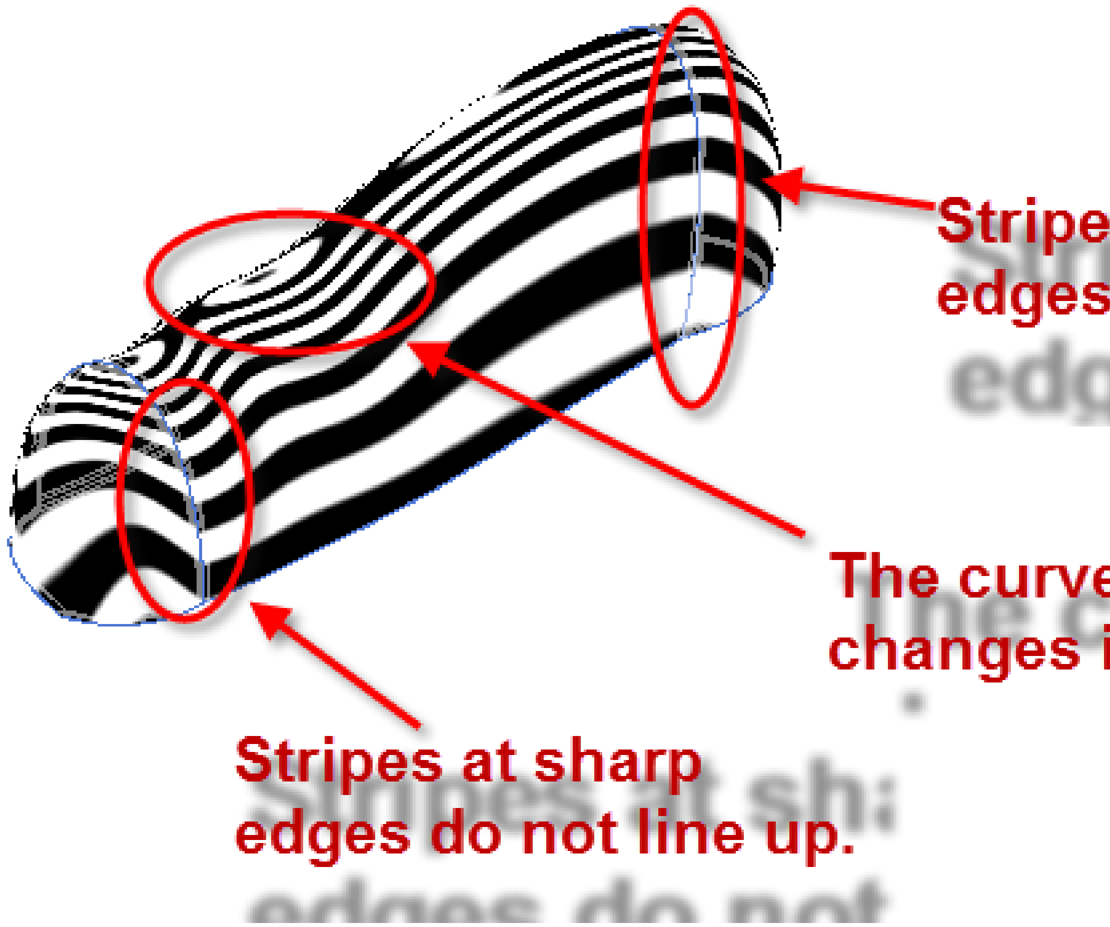
## Zebra Analysis



Use the Zebra Analysis command to view the curvature of each face and the transitions between faces. The zebra stripes are the reflection of lines that are projected onto the model.

Since the zebra stripes are a reflection, the display is different on each face and the appearance changes as you rotate and zoom the model. You can change the direction and spacing of the lines to improve the contrast of the surface curvature and transitions.

The zebra stripes provide visual feedback for surface curvature and the transition between surfaces.

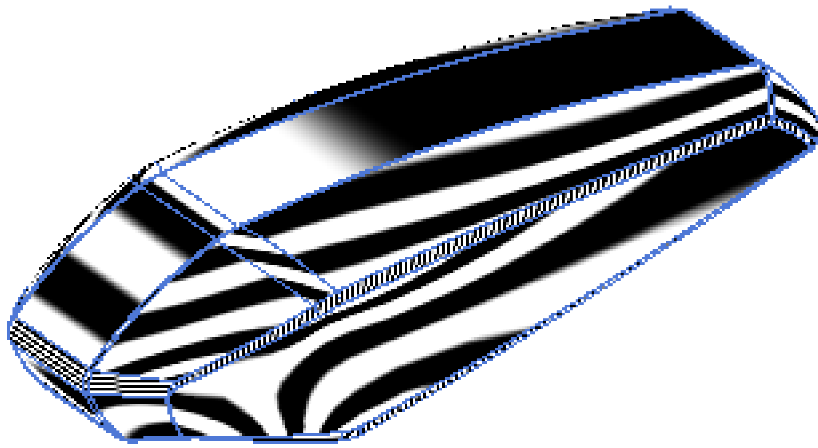



You can select one or more surfaces, quilts, solid features, solid bodies, and components. The stripes only display while the command is active.

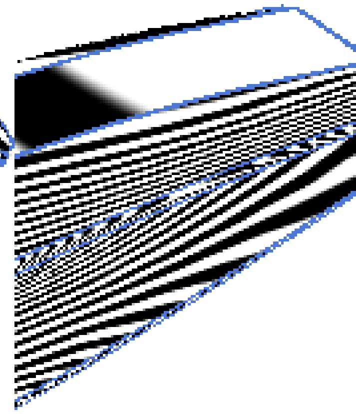
### Zebra stripe examples

The zebra stripe reflection changes as you rotate and zoom the model. Zebra stripes are typically used in industrial design to emphasize the appearance of reflections. For most models, it is helpful to change the stripe orientation and density after you change the view.

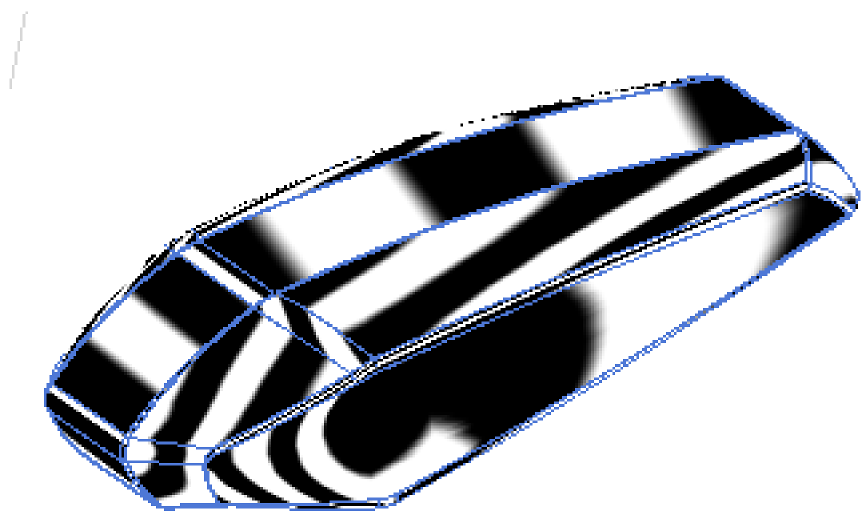
The images show the effects of different stripe directions and densities. The stripe appearance is dependent on the model orientation and size of the faces you are inspecting. Setting the direction and density is usually an interactive process as you change the model orientation and zoom factor.



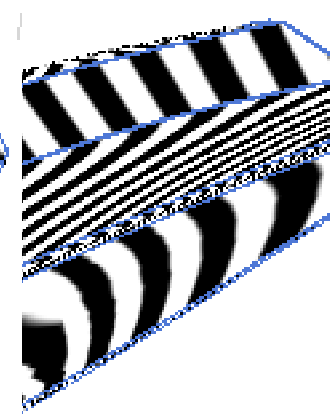
Horizontal stripes, low density



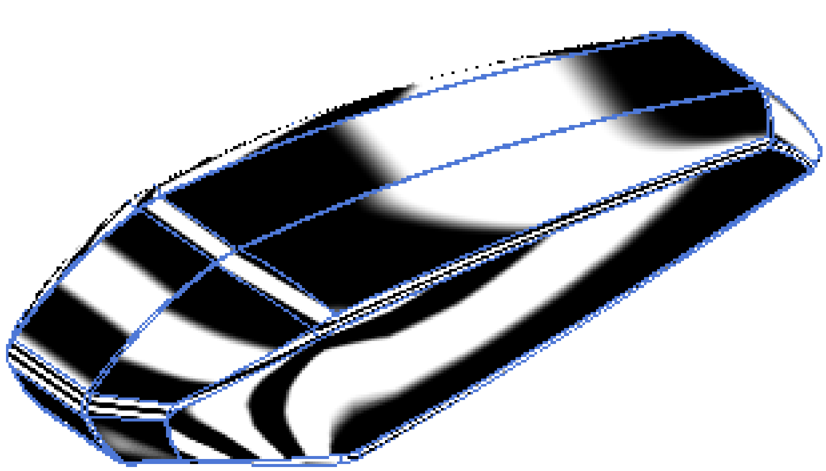
Horizontal stripes, higher density



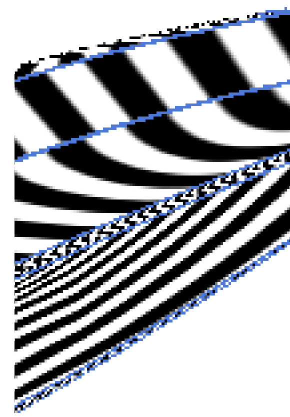
Vertical stripes, low density



Vertical stripes, higher density



Radial stripes, low density



Radial stripes, higher density

**Display zebra stripes on the model**

The zebra analysis command includes controls for the visibility, direction, and density of the stripes. You can adjust the direction of the density of the stripes to enhance the appearance for the current view orientation and zoom factor. The zebra stripes do not display after the command ends.

1 On the View tab, Visual Styles panel, click Zebra analysis.
2 Select one or more surfaces, quilts, solid features, solid bodies, and components.
3 Set the stripe direction to Horizontal, Radial, or Vertical.
4 Drag the slider to change the density.

---

**NOTE** You can select and deselect objects while the command is active.

---

# User Tags

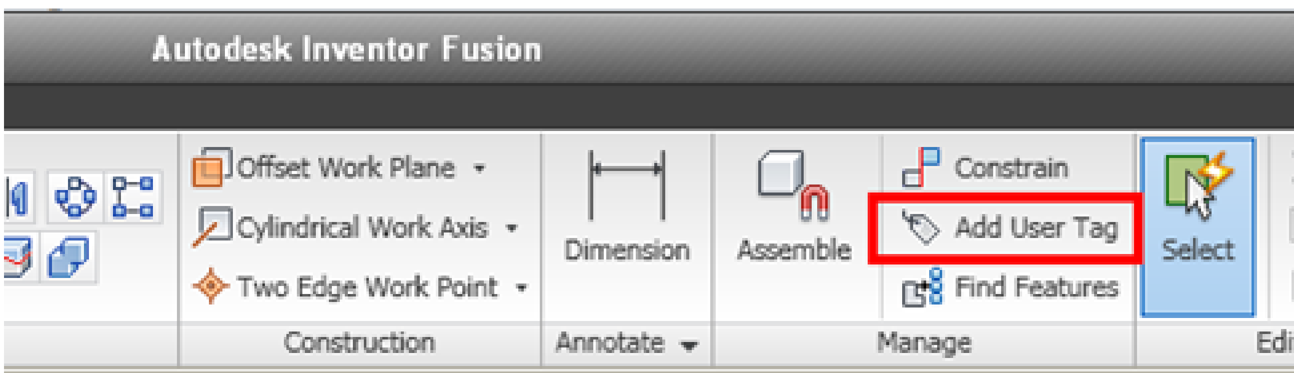
# 6

You can add tags to any face or body, and then use the tags to search and locate particular entities.

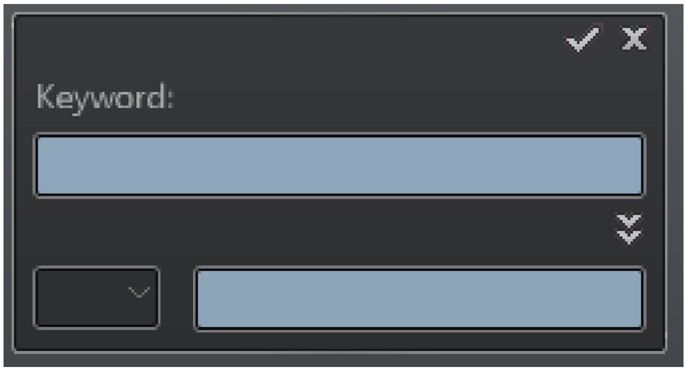
These User Tags consist of a tag name and an optional tag value. Inventor Fusion also supports Feature Tags. Feature Tags are predefined properties of features such as Fillet Radius or Hole Dia.

### Add User Tag

The Add User Tag command sets the tag name and value for the selected entities. The command is located on the Ribbon.




To assign a user tag, click Add User Tag, and use the dialog box to enter the tag. To expand the dialog box, click  .




There are two ways to set the tag name and value.

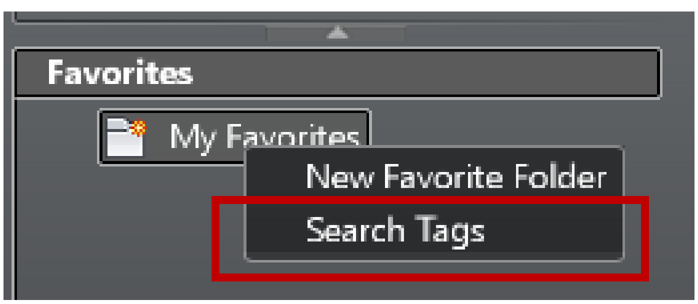
- In Keyword, enter the tag name and value separated by a comma.
- Expand the dialog box, select the tag name in the drop-down menu, and enter the tag value in the field. To enter a new tag name, select New.

Then select the objects or multiple objects to assign the tag to them.

Click OK  to apply the tag. Click Cancel  to terminate the command without applying the tag.

**Search User Tags**

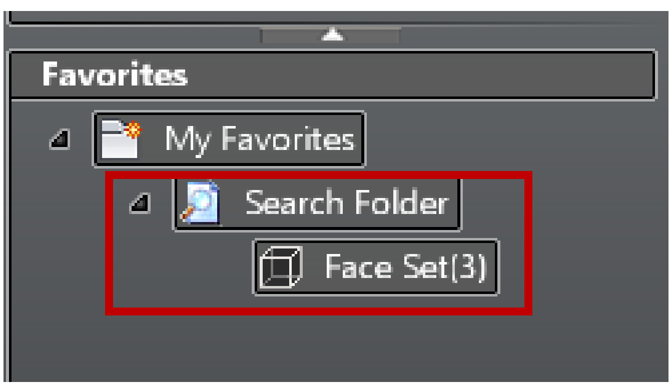
You can search user tags and predefined feature tags. In the browser, in Favorites, create, and access searches. To create a search, in the browser, in Favorites, My Favorites, use the context menu and select Search Tags.




When you create a search, the search dialog box displays. Enter the tag or feature name and value and then click OK to create the search.


### User Tag Search

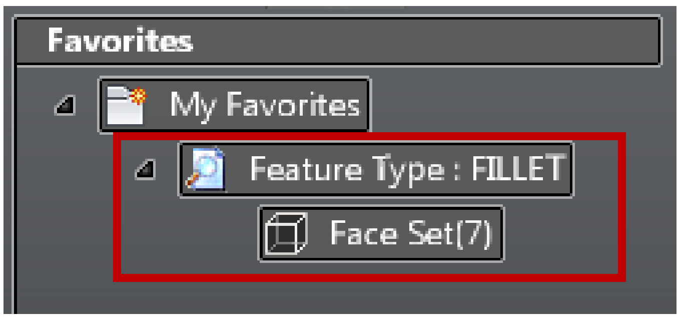
- This command searches tags added using the Add User Tag command.
- You can query any keyword (tag name, value). The search result is placed under a new search folder browser node.


### Feature Tag Search

- This command searches predefined feature tags.

- You can enter any feature type to search for all faces of that feature type.

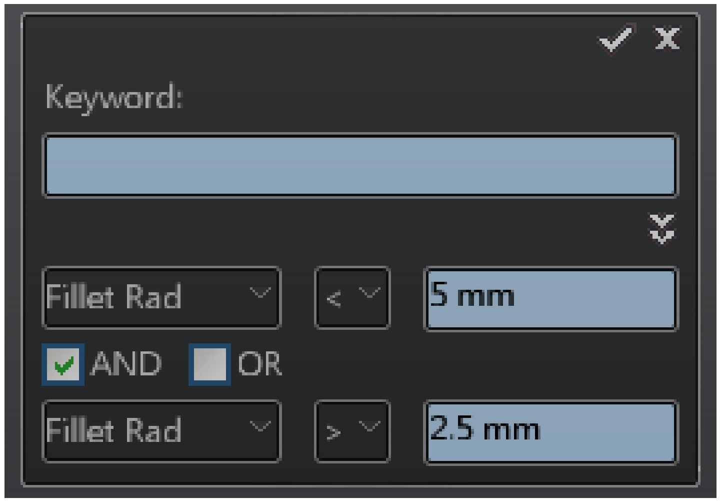




### Advanced Tag Search

- Click the expand button  to display advanced search options.
- Set search criteria by defining up to two conditions.
- There is a logical operator 

between the two options.
- The tag name drop-down list contains recently used tag names and a predefined list of feature tags names. (Fillet Rad, Hole Dia, Hole Type, Hole Depth, CB Dia, CB Depth, CS Dia, CS Angle, Extrude Dist)

■ You can set a comparison for tag values (, , =, or !=).




■ In the browser, the search results node is added to the My Favorites folder. This folder updates when you click or expand the folder node. You cannot create a search folder that is empty, but a search folder where the query once returned values, but now no longer does, is allowed.

# Import Data

# 7

Part and assembly files from other CAD systems can be imported for use in Autodesk Inventor Fusion. The import operation does not maintain associativity with the original file. Changes to the original file after the import operation do not affect the imported part or assembly. Likewise, changes to the imported part or assembly do not affect the original file.

When assemblies are imported, Autodesk Inventor Fusion attempts to preserve the assembly and subassembly structure of the original file. However, for SAT files, the subassembly structure is decomposed so that all subassembly parts become components of the top-level assembly.

The import process creates base features in Autodesk Inventor Fusion representative of the geometry and topology in the source file. You can use Autodesk Inventor Fusion commands to adjust the base features and add new features to the Autodesk Inventor Fusion feature tree.

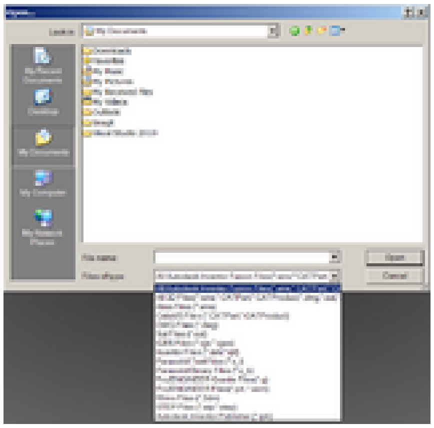
## Translate files into Autodesk Inventor Fusion data

You can open or import part and assembly files from other CAD systems.

#### To translate a file In Autodesk Inventor Fusion

1 Click the Open button.

2 In the Open dialog box, in Files of type, select the file type.



3 Browse to and select the file to import.

4 Click Open to import the file.

### Import CATIA V5 files

Open and change models created in CATIA V5 (versions R6 - R19). Autodesk Inventor Fusion translates assembly and part files, solids, multi-solids, surfaces, and more. After the import operation is complete, you have a base feature or features that match the geometry and topology of the original file. Use commands in Autodesk Inventor Fusion to adjust the base features and add new features to the feature tree.

Types of CATIA V5 files you can import:

- *.CATPart (part)
- *.CATProduct (assembly)

After changing the file, you can continue to open it in Autodesk Inventor Fusion.

### Import Pro/ENGINEER files

Open and change models created in Pro/ENGINEER. Autodesk Inventor Fusion translates assembly and part files, solids, multi-solids. After the import operation is complete, you have a base feature or features which match the geometry and topology of the original file. Use Autodesk Inventor Fusion commands to adjust the base features and add new features to the feature tree.

Types of Pro/ENGINEER files you can import:

- *.prt* (part) (up to version 4.0)
- *.asm* (assembly) (up to version 4.0)

- *.g (Granite) (up to version 5.0)

After changing the file, you can continue to open it in Autodesk Inventor Fusion.

### Import SolidWorks files

Open and change models created in SolidWorks. Autodesk Inventor Fusion translates assembly and part files, solids, multi-solids. After the import operation is complete, you have a base feature or features which match the geometry and topology of the original file. Use Autodesk Inventor Fusion commands to adjust the base features and add new features to the feature tree.

Types of SolidWorks files you can import:

- *.sldprt (part)
- *.sldasm (assembly)

### Import STEP files

You can import a STEP file (versions AP214 and AP203E2). The solid body is saved in an Autodesk Inventor Fusion file, and no links are maintained to the original file.

If an imported STEP file contains one part, it produces an Autodesk Inventor Fusion file with a single body. If it contains an assembly, it produces an Autodesk Inventor Fusion file with multiple components.

### Import SAT files

You can import a SAT file (versions 4.0 - 7.0). The solid body is saved in an Autodesk Inventor Fusion file, and no links are maintained to the original file.

If an imported SAT file contains a single body, it produces an Autodesk Inventor Fusion part file with a single body. If it contains multiple bodies, it produces an Autodesk Inventor Fusion File with multiple components.

# Export Data

# 8

You can export Autodesk Inventor Fusion parts and assemblies to other CAD system formats. The export operation for part and assembly files creates files as if they were created in the native CAD system format. The use of the exported files in the native CAD system is typically seamless.

The export operation does not maintain associativity with the Autodesk Inventor Fusion file. Changes to the Autodesk Inventor Fusion file after the export operation do not affect the exported part or assembly. Likewise, changes to the exported part or assembly do not affect the Autodesk Inventor Fusion file.

When you save Inventor Fusion data, it is saved to a 2010 format DWG file. You can choose to save as an AutoCAD 2009 format DWG file. Inventor Fusion model intelligence is not preserved in the AutoCAD 2009 DWG file. The model is exported as AutoCAD solids.

### Export Data to Other Formats

You can export solid data in the following file formats:

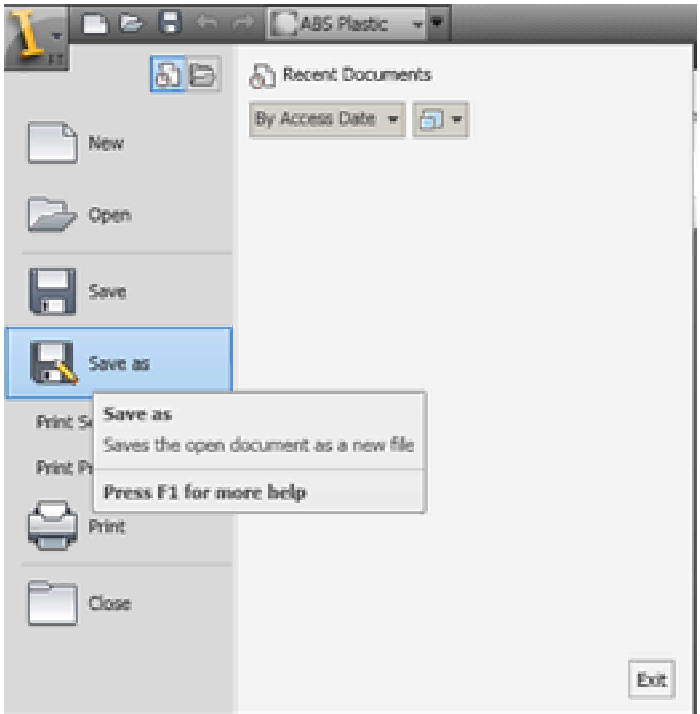
- Catia V5, parts and assemblies
- Pro/ENGINEER, parts and assemblies
- STEP
- Autodesk Shape Manager SAT

You can export viewing and faceted data in the following file formats:

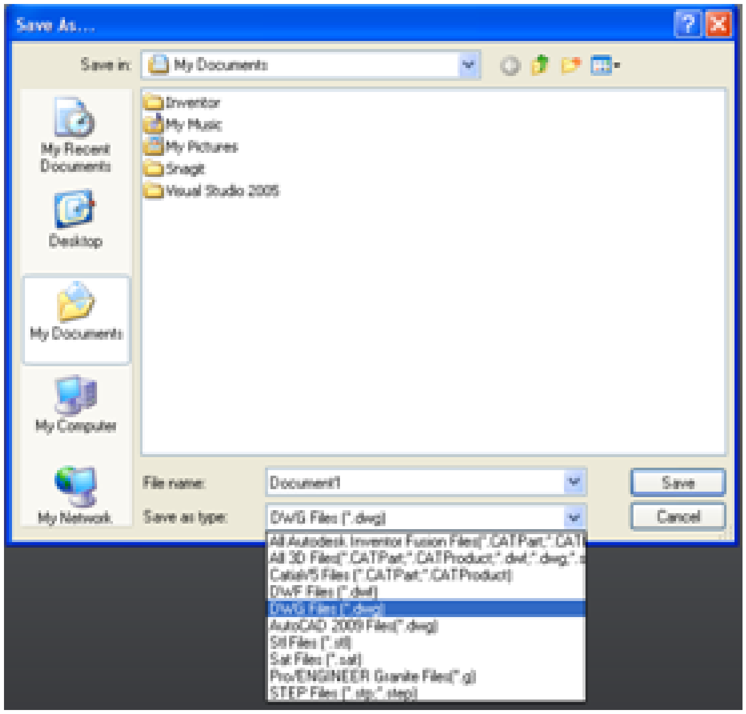
- STL
- DWF
- IGES

To save Autodesk Inventor Fusion files to other file formats:

1 Select  ▸ Save As.




2 In Save as type, select the appropriate file type from the list.




3 Enter the file name. If you do not enter a file name extension, the file is saved with the extension of the selected file type.

4 Click Save to export the Autodesk Inventor Fusion data to the file.

# Interoperability

# 9

Autodesk Inventor Fusion facilitates data exchange between many Autodesk software products. This exchange can be import/export of files. Some products are more integrated, such as Inventor Change Manager.


- Autodesk Inventor (page 469)
- AutoCAD (page 475)
- Autodesk Simulation Mechanical (page 476)


## Import Inventor Data

You can import an Autodesk Inventor file. The solid body is saved in an Autodesk Inventor Fusion file. Excluded, suppressed, and invisible objects become visible when importing IPT and IAM files.

You can import these types of Inventor files:

- *.ipt (part)
- *.iam (assembly)

Changes made to a part file within Inventor Fusion can be integrated back to the original part using Change Manager in Inventor.

You can edit an Inventor model within Inventor Fusion, free from the rigid structure of history-based modeling. Then open the part within Inventor, and accept or reject Inventor Fusion model edits through a rich interface provided by the add-in.

This workflow helps you to make rapid and unrestrained model edits in Inventor Fusion. Then those model edits can be realized within the Inventor model.

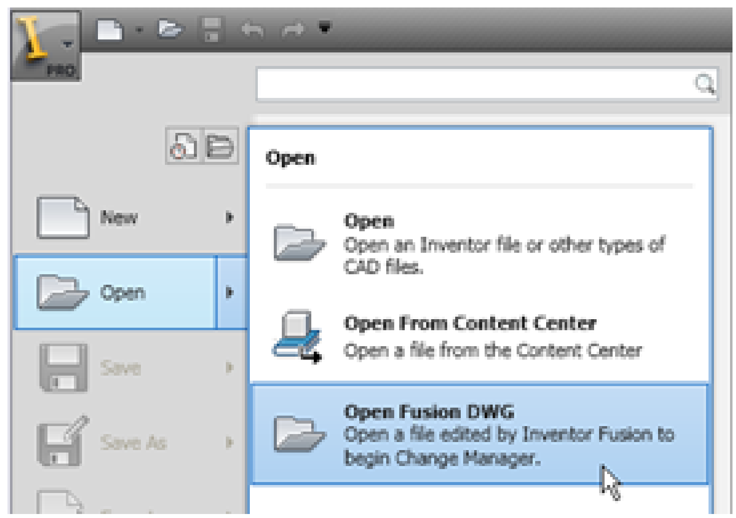
For more information see Inventor Fusion Change Manager (page 470).

## Change Manager in Inventor Fusion

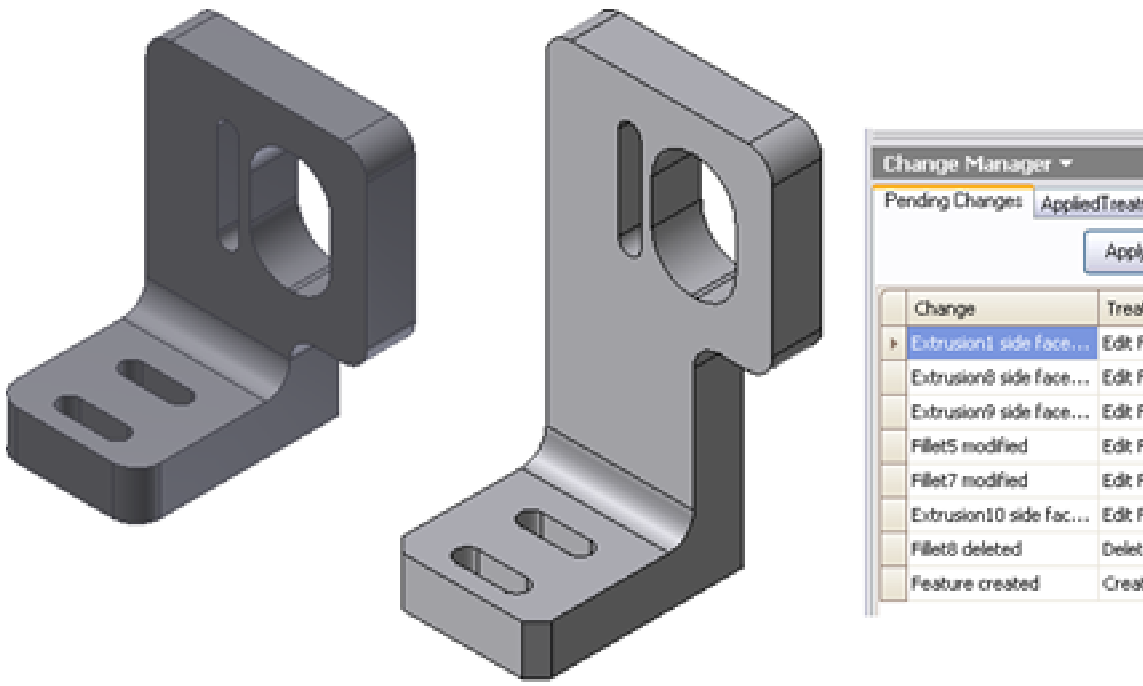
Change Manager is an Inventor add-in that is included when you install Autodesk Inventor Fusion Technology. With this add-in, you can identify and manage changes made to a part with Fusion. Use Change Manager to map non-parametric Fusion edits to editable, parametric Inventor features.

### Overview of Change Manager

Change Manager provides the means to edit an Inventor model (.ipt) within Inventor Fusion Technology, free from the rigid structure of history-based modeling. The file is saved in a Fusion-flavored DWG format. Use the Inventor Open Fusion DWG command to open the edited model in Inventor. You can then accept or reject the Fusion edits using the new Change Manager browser interface provided by the add-in.




Consider the example of a slotted bracket. Using Fusion, we replace the two bottom fillets with chamfers, and increase the overall height of the bracket.



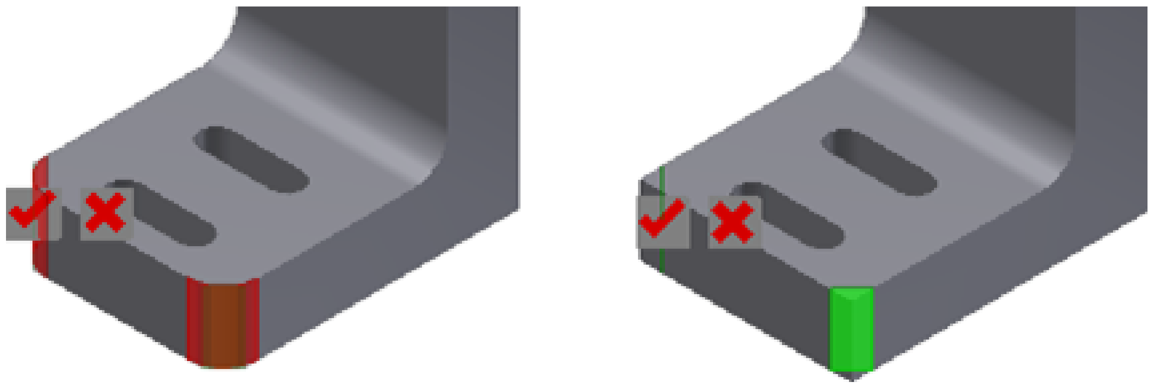

Original Inventor Part    Part Edited in Fusion    Change Manager Browser L

When the Fusion DWG file is opened in Inventor, the Change Manager Environment is activated. The Change Manager browser lists eight differences between the original part and the new part. The graphics show the color:

- Red: Deleted fillets
- Green: Newly created chamfers
- Blue: Old position of modified faces
- Yellow: New position

For each change listed in the browser, a set of Treatments is offered. The preferred treatment is a parametric update to the Inventor model. In our example, we can Apply Treatment to the Edit Feature for both Extrusion1 and Extrusion8.

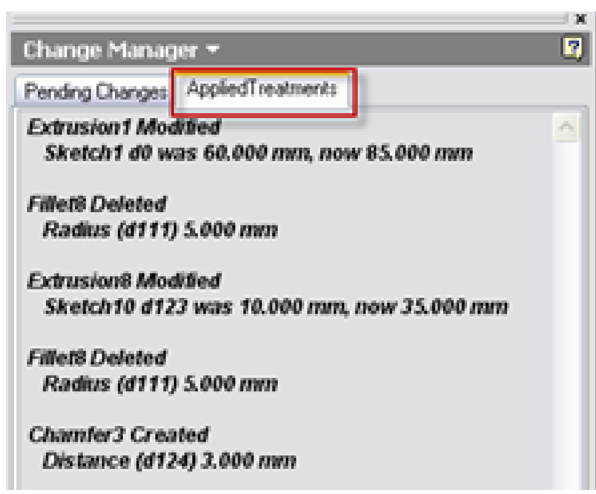
After these two treatments are applied, only two remaining changes are detected. After every change to the Inventor model, either through Apply Treatment or direct user edits, Change Manager reevaluates the differences between the Inventor model and the Fusion model. In this case, the updates to Extrusion1 and Extrusion8 repositioned the slots with both the wide and narrow radii, eliminating these differences. All that remains to be done is to Apply Treatment to the deleted Fillet8 (shown in red), Then Apply Treatment to accept the chamfer replacements (shown in green).



**Note** You can also accept or reject changes from within the graphics window. Click the red check mark to apply a treatment. Click the red X to ignore the change.

Apply All applies the preferred treatment to each change in order.

After you finish accepting or rejecting changes, in the Change Manager browser, click the Applied Treatments tab to confirm the precise nature of the changes.

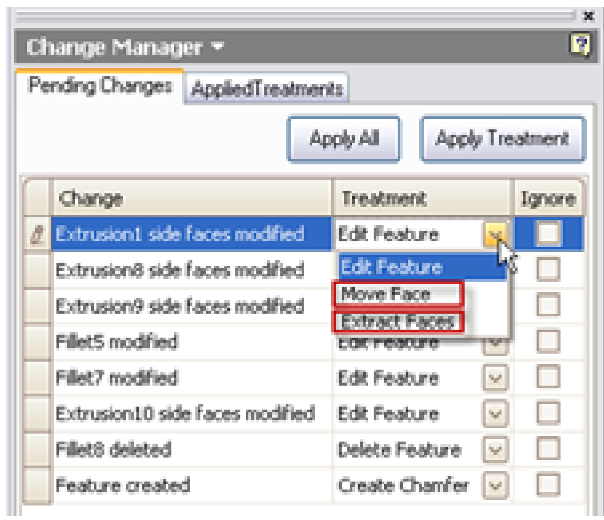


### Alternative treatments

Not all features are supported for parametric edits. Now, Change Manager can edit only the parameters for fillets, chamfers, extrusions, revolutions, patterns, and holes. Hole types that can be created include: From Sketch, Linear, Concentric, and On Point.

Features that are currently unsupported include Draft, Loft, Shell, and Sweep.

In those cases where Change Manager cannot find a parametric edit for the Inventor model, two alternative treatments called Move Face and Extract Faces are available. To access the alternative treatments, in the Change Manager browser, Treatment column, click the down arrow to the right of Edit Feature.

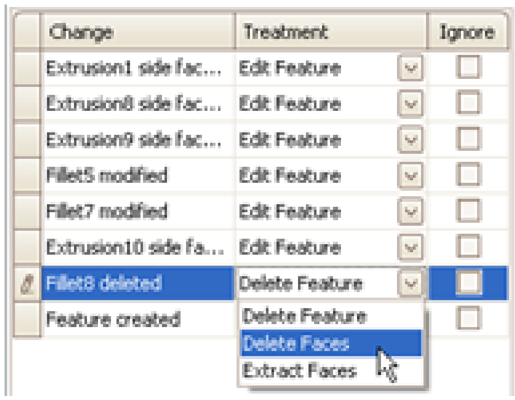



The Move Face treatment copies the faces from the Fusion model and convert them to parametric, associative faces in Inventor. The new faces are fully editable using Direct Manipulation techniques.

**Note** Refer to the Inventor Help topic: Move Face.

The Extract Faces treatment copies the faces from the Fusion model and then attempts to sculpt the Inventor model to create a solid body. If the sculpt fails for some reason, the copied surfaces remain in the Inventor model. You can then use these faces as references for trying some manual edits to the Inventor model. Extract Faces is available for all features.

Whenever Change Manager lists Delete Feature as a suggested treatment, it also offers the option to Delete Faces. With Delete Feature, the feature is permanently deleted from the Fusion-edited model. As an alternative, Delete Faces deletes the faces and places a Delete Face feature node in the Model browser. By deleting the Delete Face node in the browser, you can restore the deleted faces later , if necessary.




At any time, you can edit the Inventor model directly. By selecting the Model browser view button, you can switch to the usual model browser and perform Inventor model edits. You can also use Finish Change Manager and return later by selecting the Change Manager button on the Environments toolbar.

### Edit parts within an assembly

#### Open an Inventor assembly file (.iam) in Fusion, and modify individual parts

1. Start Fusion, and open the .iam file using the Files of type drop-down list.

   The assembly file opens in Fusion and each of the parts appears in the Fusion browser.

2. In the browser, right-click the part to edit, and select Activate Component.

   The selected part is activate, and is ready for editing.

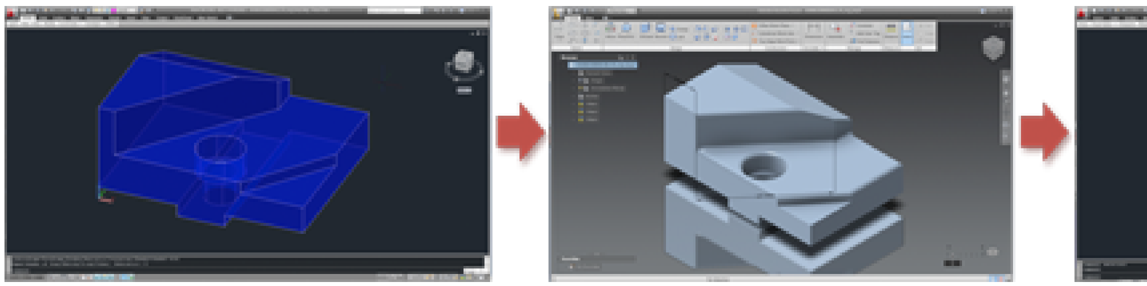
3. When you finish editing the part, in the browser, right-click the part name, and select Save As.

   The part is saved with a DWG file extension.

4. In Inventor, use the Open Fusion DWG command to open the edited part and start Change Manager.

5. Apply or ignore the suggested treatments in the Pending Changes list.

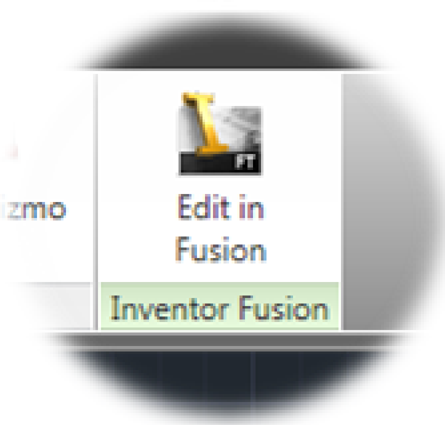
# Integration of AutoCAD and Fusion



Autodesk Inventor Fusion and AutoCAD both use DWG files. You can use the Open command in Fusion, to open and modify AutoCAD 3D files, and in AutoCAD, to open and modify Fusion files.

In an AutoCAD session, you can invoke Fusion and modify 3D AutoCAD data without saving and closing the current file. This workflow is available if both Fusion 2012 and AutoCAD 2012 are installed.

**To edit 3D AutoCAD geometry using Fusion**




1 Create 3D bodies in AutoCAD, or open a file that contains 3D bodies.
2 Select one or more 3D bodies.
3 On the ribbon, click Edit in Fusion. Fusion launches, and imports the selected bodies.

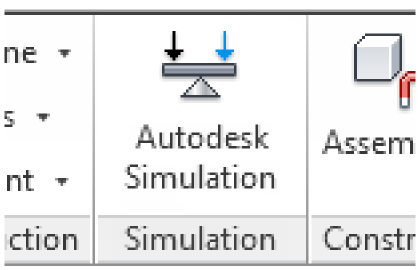
4 Make the desired changes to the geometry.

5 When you finish your changes, click Return to AutoCAD. The modified bodies are saved and replaced in the AutoCAD file.

# Integration of Autodesk Simulation Mechanical and Fusion

Seamless integration exists between Inventor Fusion and Autodesk Simulation Mechanical. You can open a model in Inventor Fusion and send it to Autodesk Simulation Mechanical for simulation analysis. You can open the model for simulation analysis in Autodesk Simulation Mechanical, send it for editing in Inventor Fusion. After modifications are finished, you can automatically reload the updated model for analysis.

**Send Inventor Fusion models to Autodesk Simulation Mechanical for analysis**




The Autodesk Simulation command is available when you open any DWG file in Inventor Fusion if:

- The model is not empty.
- Autodesk Simulation Mechanical is installed on your machine.

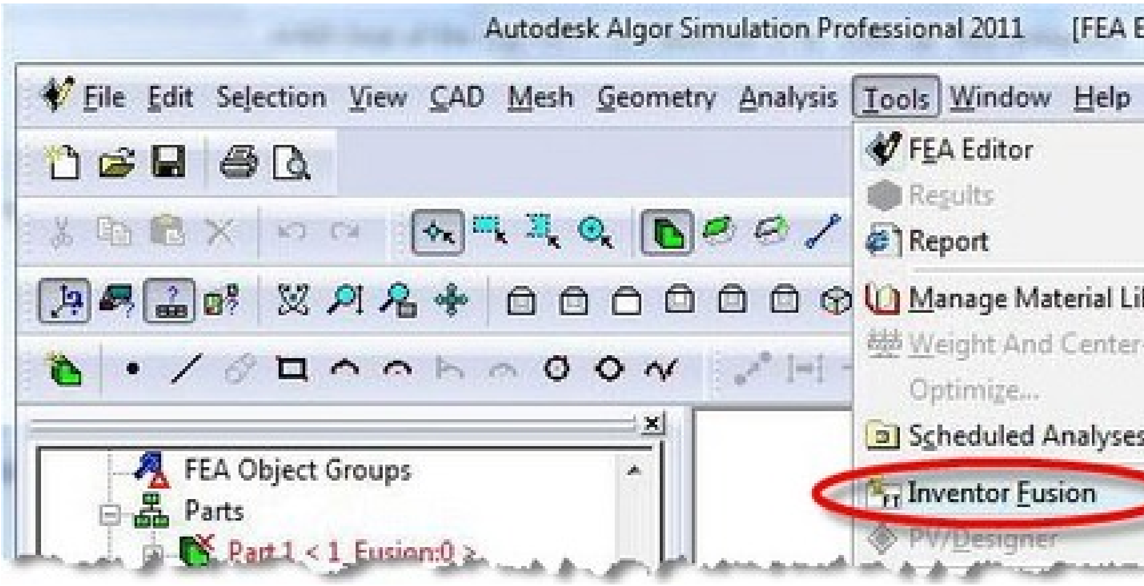
If a file format is other than DWG, you are prompted to save the file to DWG. With the model in DWG file format, Inventor Fusion launches Autodesk Simulation Mechanical and loads the model for processing inside it. Then you can assign loads and boundary conditions to geometry and perform an analysis.

**Edit models opened in Autodesk Simulation Mechanical inside Inventor Fusion**

Autodesk Simulation Mechanical provides a way to send the model opened for simulation to Inventor Fusion for editing purposes.

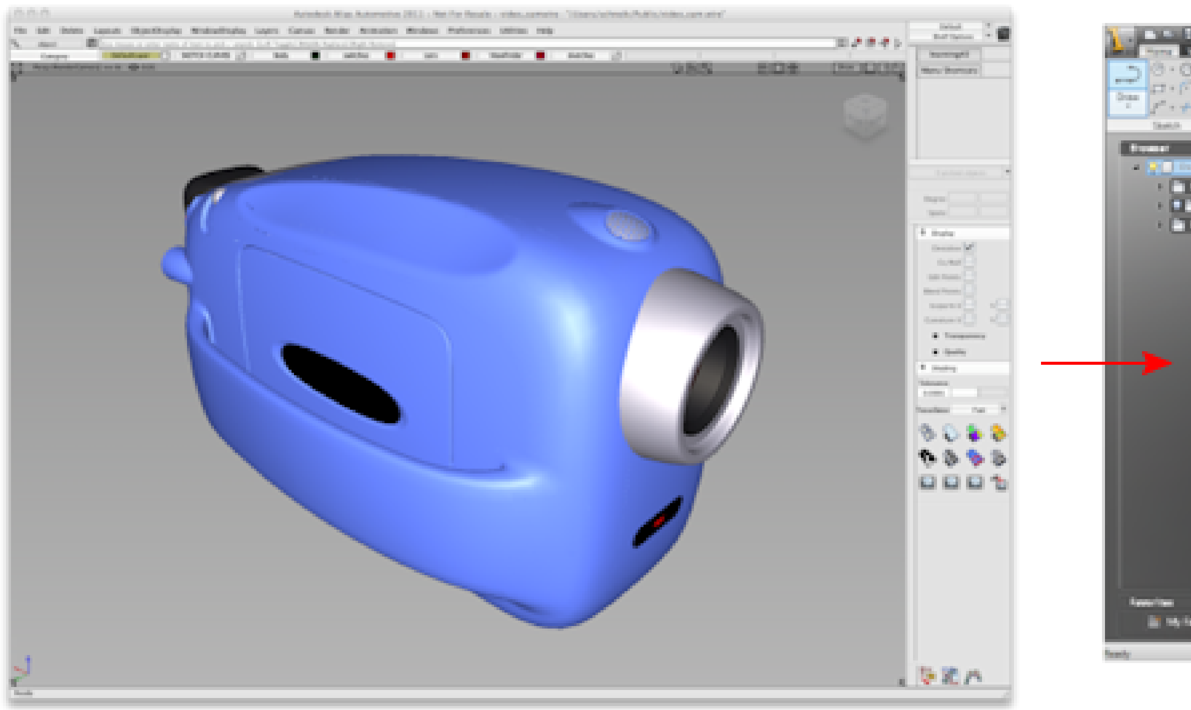
When you use the File Open dialog box to open an Inventor Fusion DWG file in Autodesk Simulation Mechanical, the model automatically loads in Inventor Fusion. If Inventor Fusion is not running in the background, then Autodesk Simulation Mechanical invokes a new instance of the application and loads the model into it.

If a file format is other than DWG, then save it to DWG in Inventor Fusion. To modify the model, in Autodesk Simulation Mechanical, on the Tools menu, click Inventor Fusion, or activate the Inventor Fusion session that loaded the model.



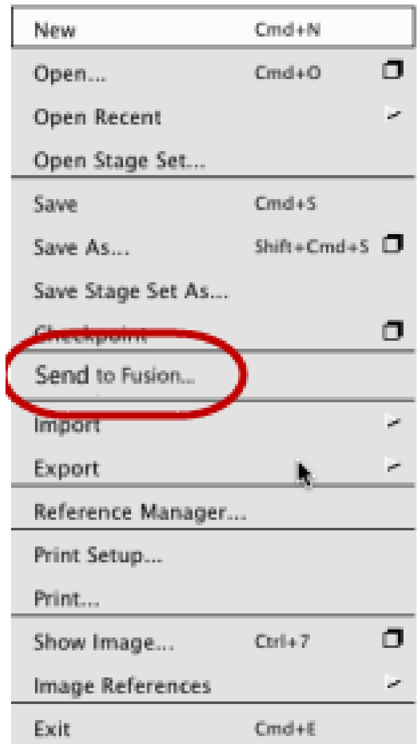

You can edit the model using Inventor Fusion commands. Then, on the Inventor Fusion Ribbon, Home tab, invoke the Autodesk Simulation command to send the updated model back to Autodesk Simulation Mechanical.

## Integration of Alias Design and Fusion



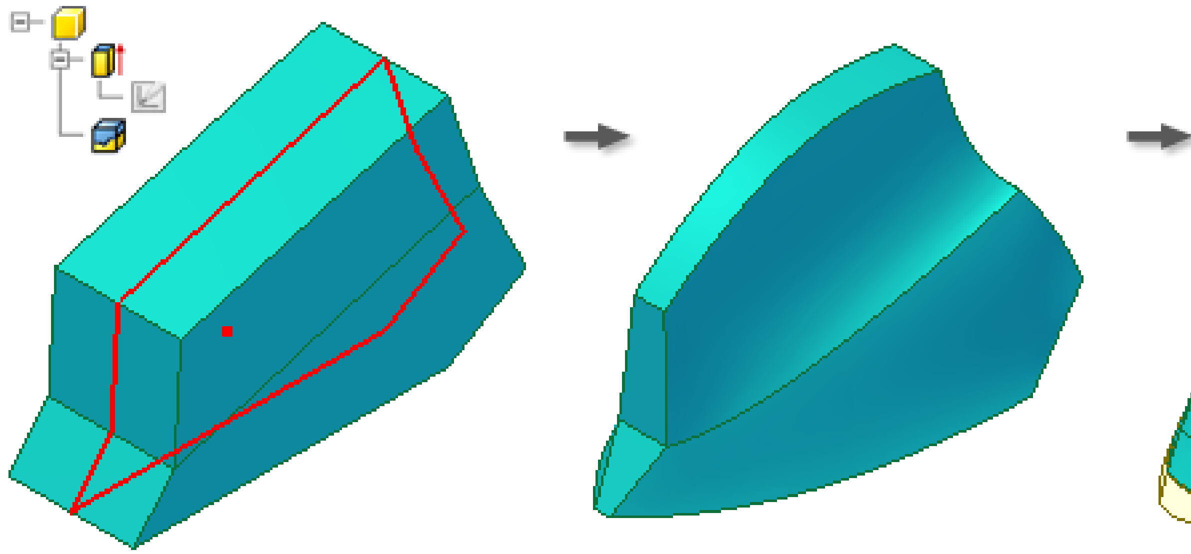
In an AliasDesign session, you can send the surface data to Fusion. Use Fusion to repair and stitch surfaces into solids. If necessary, use Fusion commands to add geometry such as holes, ribs, or fillets. Export the solid data to the desired CAD format once the design is ready.

To use Fusion to prepare AliasDesign data:




1. Create surface data in AliasDesign.
2. Select Send to Fusion.
3. Use Validate or other surface commands to repair and stitch the surfaces to create a solid.
4. Add geometry such as holes, ribs, or fillets to refine the model.
5. Save the model to the desired CAD format.

# Integration of Inventor and Fusion



In Inventor, use Edit Form or Edit Copy of Form to launch Fusion with the selected geometry. Make you changes in Fusion then select Return to Inventor to push the changes back to Inventor. An Alias Freeform feature is added to the Inventor Browser to contain the changes.

To edit Inventor geometry using Fusion:

1. Create the desired geometry in Inventor.
2. Start Edit Form or Edit Copy of Form.
3. Select the solid body or features to edit in Fusion. Fusion launches and imports the selected geometry.
4. Make changes in Fusion. All Fusion editing commands are available including Press/Pull, Move, and Edit Edge.
5. Click Return to Inventor. The modified body is pushed back to Inventor as an Alias Freeform feature.

---

**NOTE** Only one body can be exchanged between Inventor and Fusion. Changes made to the body will be pushed back to Inventor but any additional bodies or surfaces created in Fusion will not.

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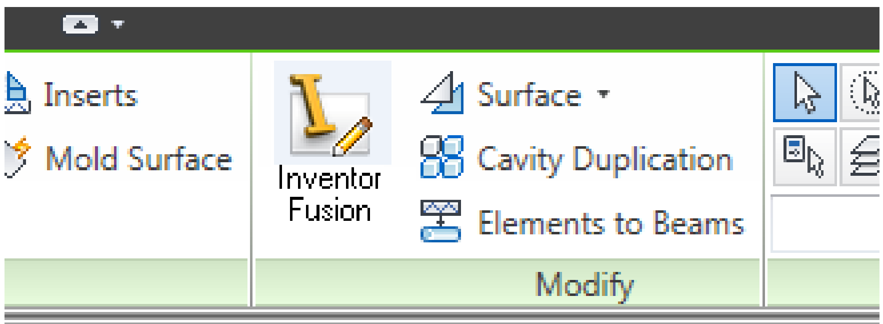
# Integration of Moldflow and Fusion

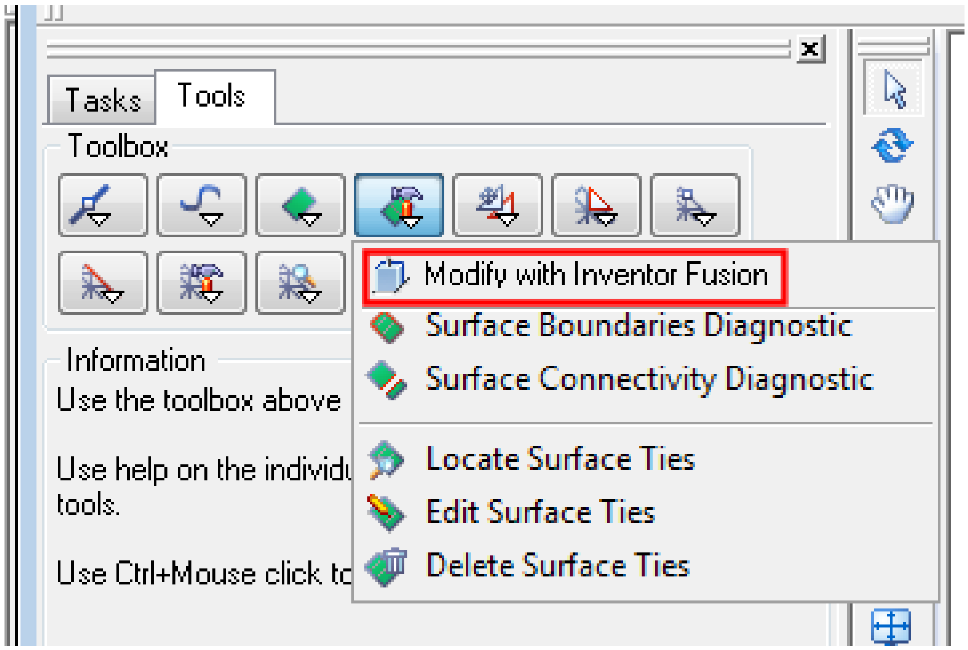
Many times during a flow analysis project using Autodesk Moldflow, users want to change the part design. Inventor Fusion can be used to make many of the changes necessary. The model can be exchanged between Moldflow and Fusion.

With a model open in Moldflow, click the Inventor Fusion button to send the model to Fusion. You can modify the model using any of the Fusion commands and return the updated model to Moldflow.




**Ribbon interface**

Classic interface



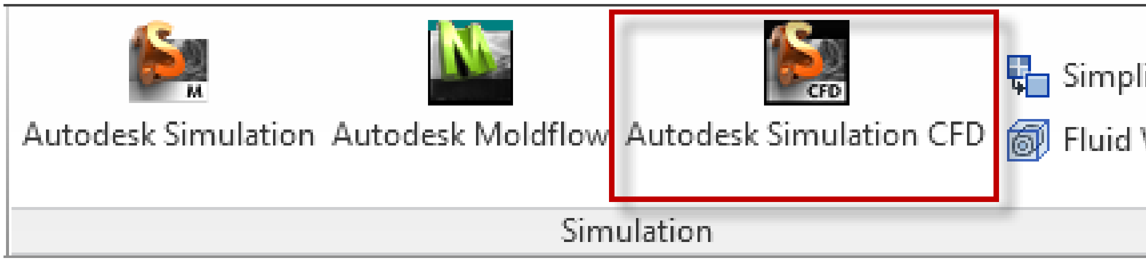

Tools Pane

# Integration of Simulation CFD and Fusion

Autodesk Simulation CFD and Inventor Fusion are seamlessly integrated. After opening a model in Inventor Fusion, you can transfer it to Autodesk Simulation CFD to simulate the fluid flow and heat transfer.

**Transfer a model from Inventor Fusion to Simulation CFD**

On the Home tab, Simulation panel, click Autodesk Simulation CFD.




The Simulation CFD command is available in Inventor Fusion if:

1. The model is not empty.
2. Autodesk Simulation CFD is installed on your computer.

**NOTE** The file must be in the DWG format. Simulation CFD prompts to save the file as a DWG, if necessary.

After clicking the Autodesk Simulation CFD button, the Design Study Manager opens. The Design Study Manager is an interactive tool for coordinating your Inventor Fusion models and Design Studies. It manages Design Studies, Designs, and Scenarios, and allows you to modify the models in Fusion without exiting Simulation CFD. The Design Study Manager opens automatically when you launch Autodesk Simulation CFD from Inventor Fusion.

# Vault Add-In

# 10

A vault is a repository where documents are stored and managed.

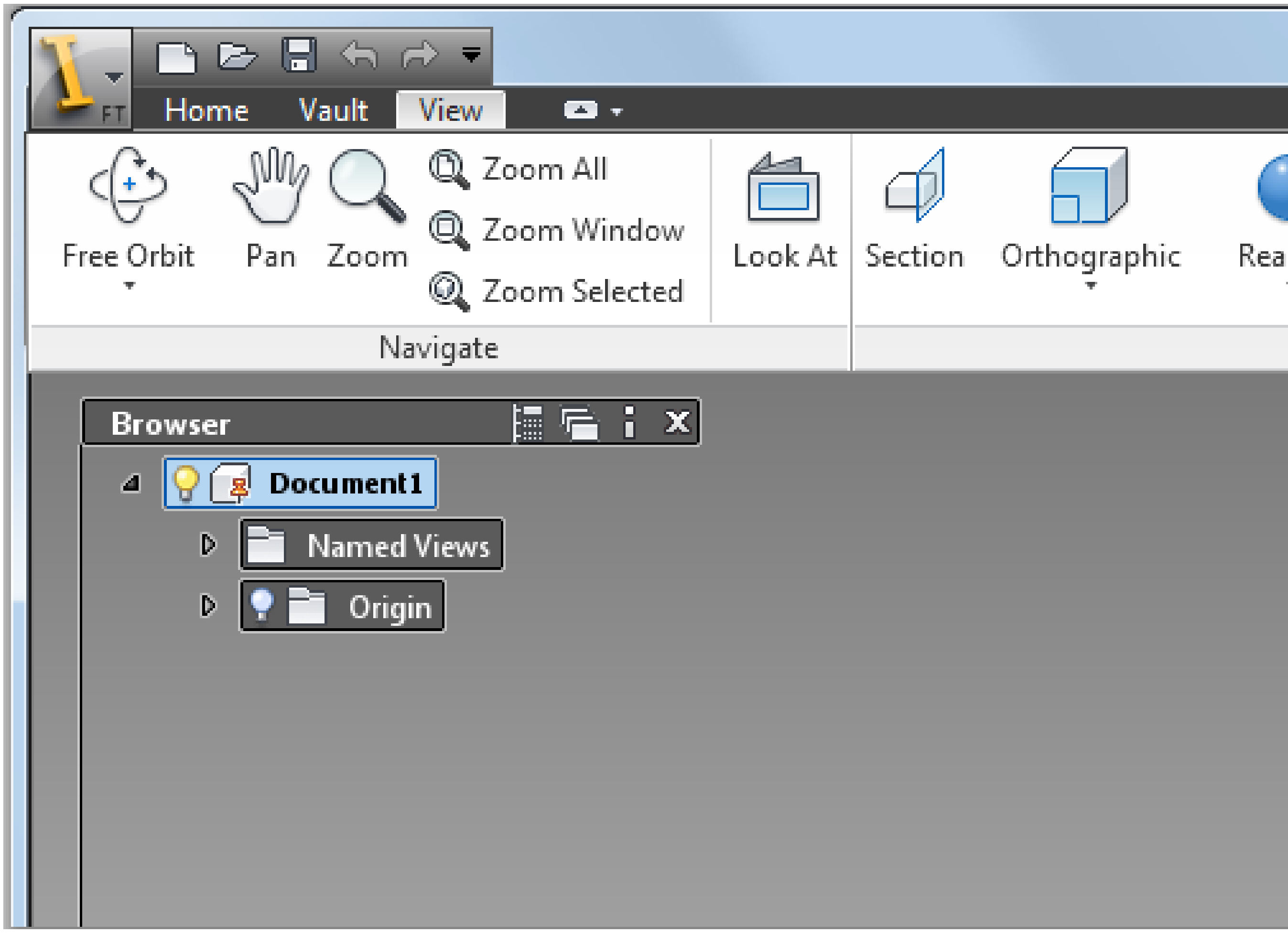
Autodesk Vault consists of two components: a server and a client. The server stores all the versions of your documents. If you store all of your data in a common, centralized location, you can share and manage all of your information with your design team. With the client, you can access the data that is stored in the Vault from the working folder. The working folder is a folder on your computer/network to which files are copied from the vault.

There are two forms of clients:

- **Vault Explorer** A stand-alone client that provides full access to the Vault.
- **Vault Add-in** A tool that performs many of the same vault tasks as Autodesk Vault Explorer within Inventor Fusion. You can manage Fusion files directly within the Inventor Fusion Environment.

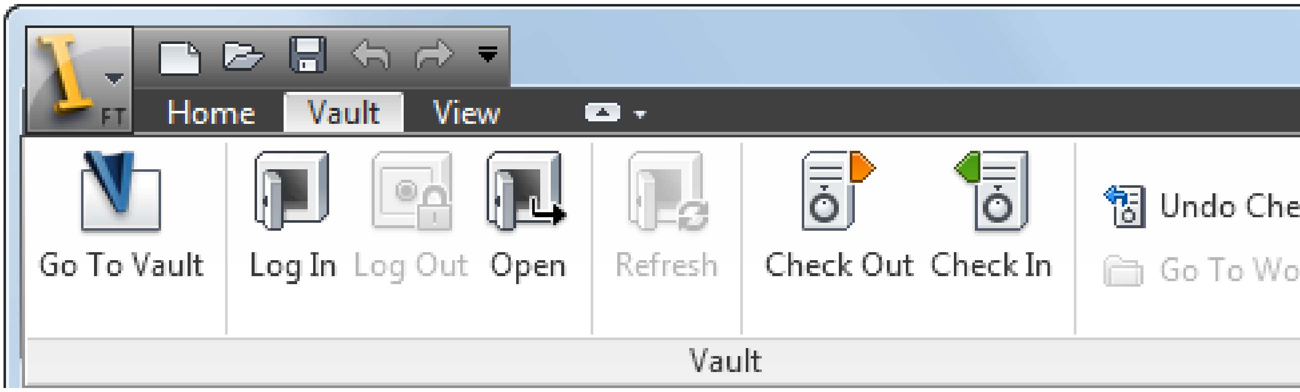
## Invoke Vault Functionality

You can turn on and off the functionality of the Vault Add-In using the Vault switch, as shown in the following image.



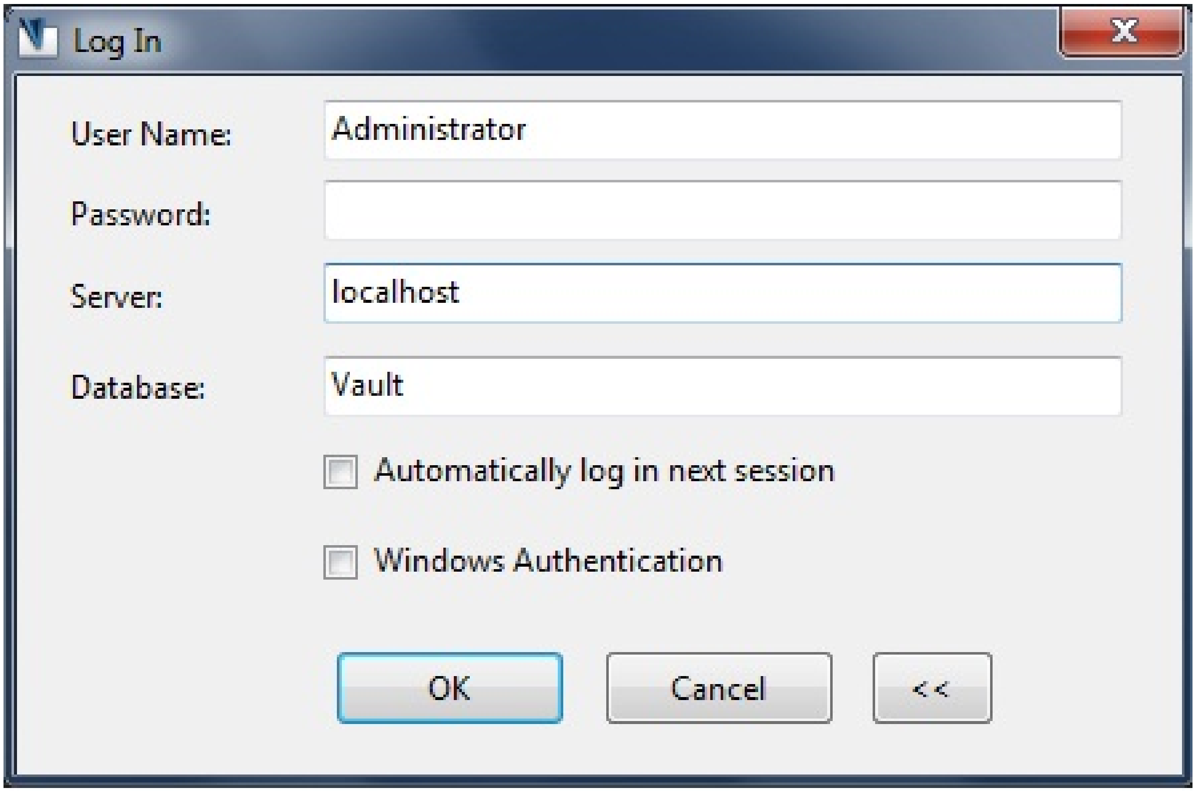

### Vault Tab

Vault Tab consists of Vault commands on the Ribbon. Commands are enabled/disabled based on availability of particular commands.




**Vault Commands**

- **Log In** Before you can log into a Vault in Autodesk Fusion, you must establish an account. Use a unique user name and password assigned by the server administrator. Contact the administrator of your server for your account information and server location.




You can save your login settings so that the login dialog box is automatically filled in the next time you start Inventor Fusion Vault.

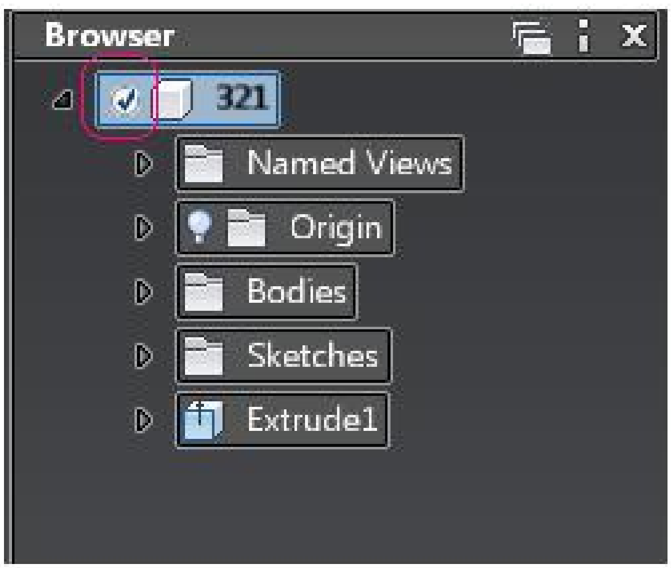
- **Log Out** Closes your connection to a vault. You can log back in to the same vault using the same account or a different account, or log in to a different vault.
- **Go To Vault** You can start the Autodesk Vault Explorer application from within Autodesk Fusion for vault administration and file maintenance.
- **Go To Workspace** Launches file selection dialog box showing the local Workspace Folder.
- **Open** Opens any Autodesk Fusion file stored in a vault into the current workspace. The file opens in the Fusion Environment as read-only file. If you wish to modify the file, you can check out the file.
- **Refresh** Retrieves a copy of the most recent design data that is available in vault.
- **Check Out** Changes the read-only attribute of the local copy to read/write. After a file is checked out, you can edit it. Only one member of a team can check out a file at one

time. No one else can modify a checked out file until it is checked back into the vault. Use Vault Explorer, User Name column, to see who has an item checked out.

- **Undo Check Out** Returns the selected checked out file to a checked in state in the vault without versioning it forward. Any changes made to the file are not stored in the vault.
- **Check In** Checks files into the vault for the first time, and when you are done working on an Autodesk Fusion file that is checked out of the vault. Check the file back into the vault to store the changes with the new version history. When you check a file in, include a comment with the file describing the changes made to this version of the data. The latest version of a file in the vault is the last version that was checked in. When a file is checked in, the version in the vault is incremental and the latest changes are now available for others to check out.

### Vault File Status

After you log in to Vault, the browser in Fusion shows file status at the top node of the browser.




The following are the icons used with Autodesk Vault in Inventor Fusion.

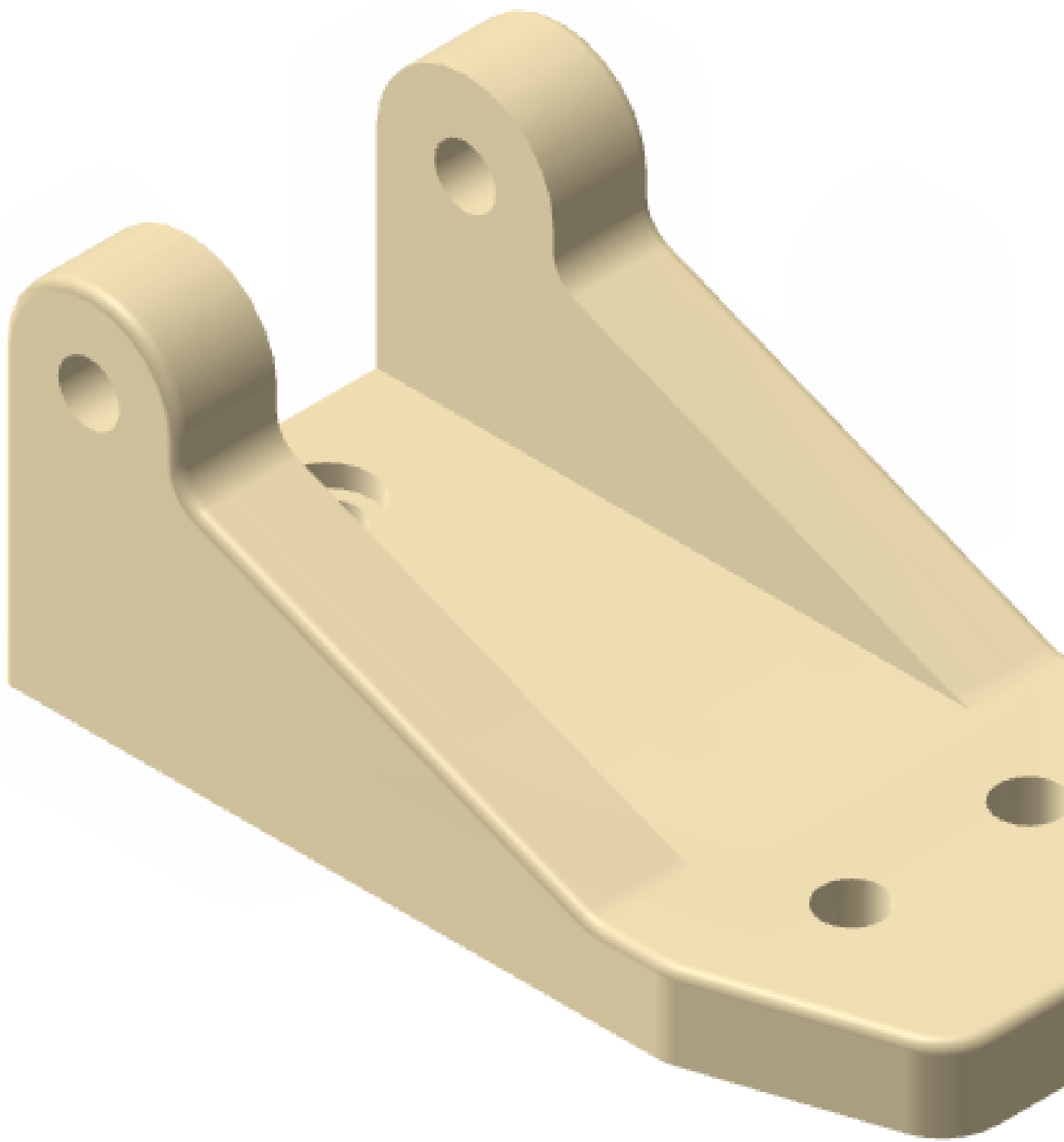
| Icon | Description and Required Action |
| --- | --- |

| | |
|---|---|
|  | You are logged in, but the active document is not in the vault. You can check in the file to add it to the vault. |
|  | File is in the vault and available to check out. The version in your working folder is the same as in the vault. Also referred to as the Latest Version. |
|  | File is in the vault and available to check out, but the local version is newer than the latest version in the vault. This status means that your local file was changed without checking it out. To save these changes, check out the file with the Don't Get Local Copy option. |
|  | File is in the vault and available to check out, but the local copy is out of date. Get the latest version from the vault. |
|  | File is checked out to you and the local version is the same as in the vault. Also referred to as the Latest Version. |
|  | File is checked out to you and the local copy is newer than the latest version in the vault. This status means that you changed the file since it was checked out, but did not check it back in. |
|  | File is checked out to you and the local copy is older than the latest version in the vault. This status means that you started with a version for the vault that was older than the latest version. Then you checked it out to promote it to the latest. |
|  | File is checked out by someone else, and the local copy is the same as in the vault. Also referred to as the Latest Version. This state happens if the other person did not check changes back into the vault, or is still editing the file. |
|  | File is checked out to someone else, but the local copy is newer than the latest version in the vault. This status happens if the other person checked changes into the vault, but kept the file checked out. |
|  | File is checked out to someone else, but the local copy is older than the latest version in the vault. Refresh to update to the latest available version. This action changes the status of the document to Latest Version. |

# 11 Tutorials

# Advanced Modeling Techniques

## About this tutorial



Learn how to use model dimensions and body constraints to define geometrical relationships, and how to reference geometry during modeling operations.

| | |
|---|---|
| **Category** | **Advanced Users** |
| **Time Required** | 60 Minutes |
| **Tutorial Files Used** | Body Constraints and Dimensions.dwg |

Inventor Fusion uses direct modeling to simplify creating and modifying geometry. Since there are not any relationships, some changes can affect other parts of the model. Body constraints create relationships between faces, and model dimensions can control size and how the model updates. There are also modeling techniques that can help control how geometry changes.

### Objectives

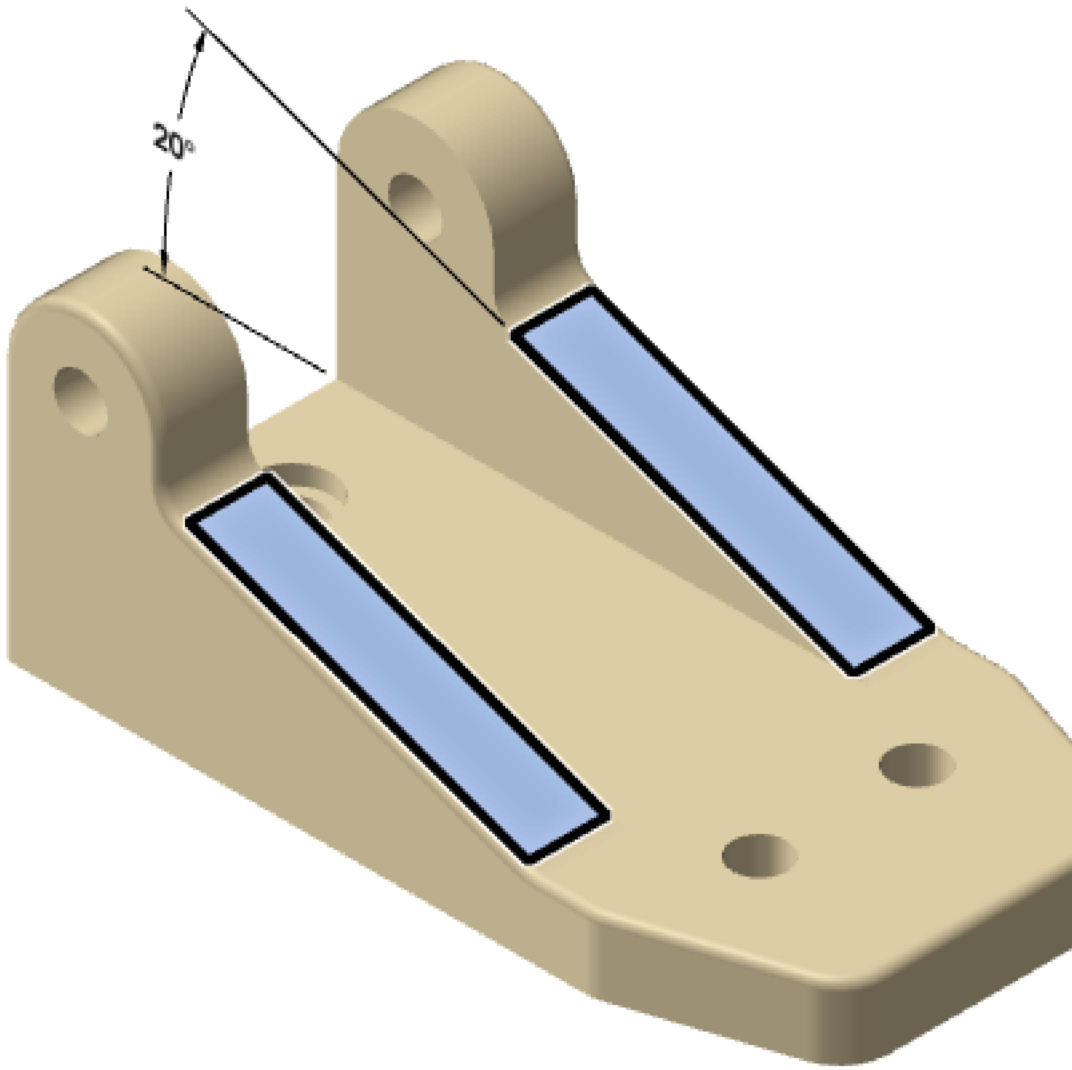
- Learn how to create relationships between model faces with body constraints.
- Learn how to control model size and updates with model dimensions.
- Learn techniques for adding draft to a model.

### Navigation Tips

- Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

## Use body constraints and model dimensions

In this section of the tutorial, we use body constraints and model dimensions to control how the model updates.




Body constraints create relationships between model faces. For example, the vertical extrusions on this model are symmetrical, and the large radii are concentric with the holes.

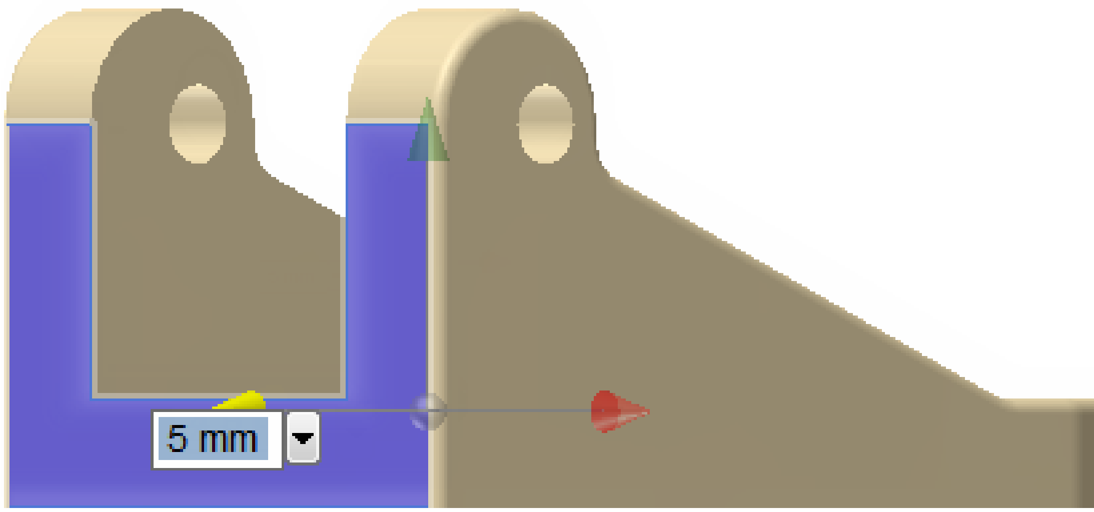
Model dimensions are used to annotate the model, and to control the size and location of edges. You can lock dimensions to hold the size during updates.

Usually, only critical dimensions remain locked, while other dimensions are locked during modeling to maintain size or proportions and then unlocked or deleted.

One of the advantages of direct modeling is that you can easily change the model geometry. Since body constraints and locked dimensions restrict this functionality, they are usually limited to cases where there is an important relationship.

1 Open the tutorial file Body Constraints and Dimensions.dwg.

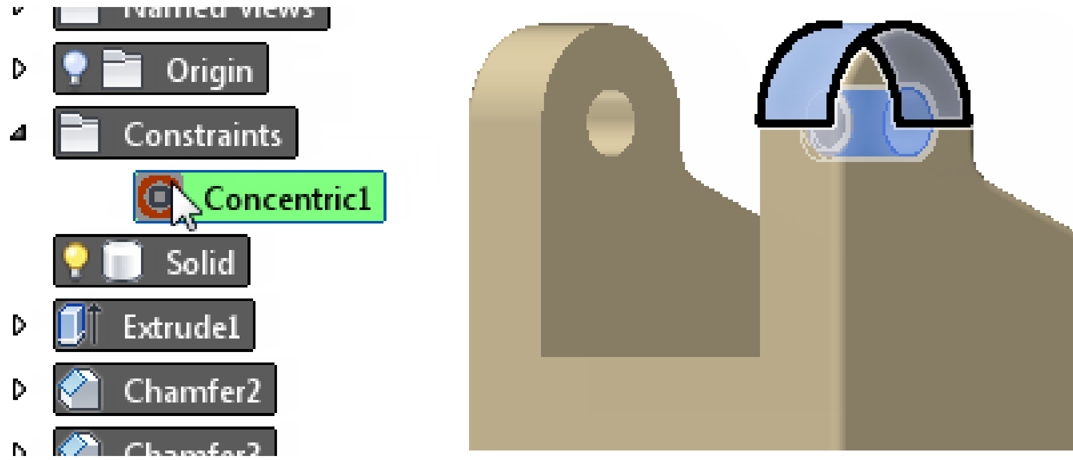
2 Expand the Named Views folder in the browser and click Corner View, then press `Home` to zoom into the model. Select the Left face and start Move. As you drag the manipulator, the top radius moves but the hole does not. We can add a constraint to keep the radius and the hole concentric.



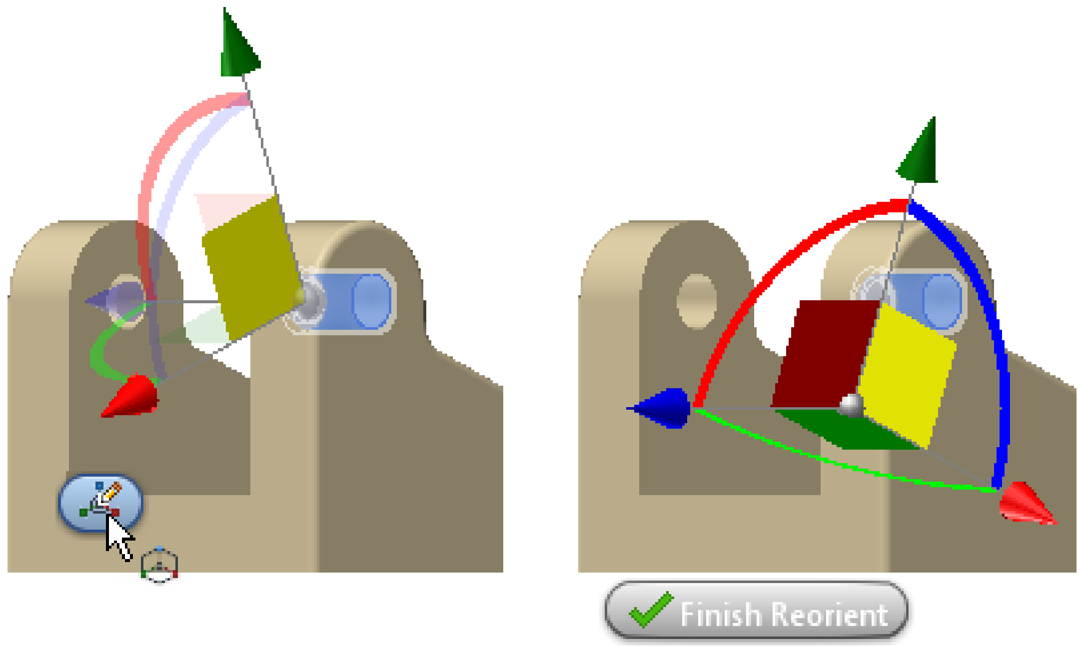

3 Press Escape to cancel the change.

4 Start the Body Constraint command from the Constrain and Dimension panel. Select the Center constraint type on the command ribbon, and then select the hole and the radius.

5 Press `Enter` to create the constraint. A Constraints folder is added to the browser, and the faces highlight when you pause the cursor over the constraint. You cannot edit constraints, but you can delete them to make the faces independent again.

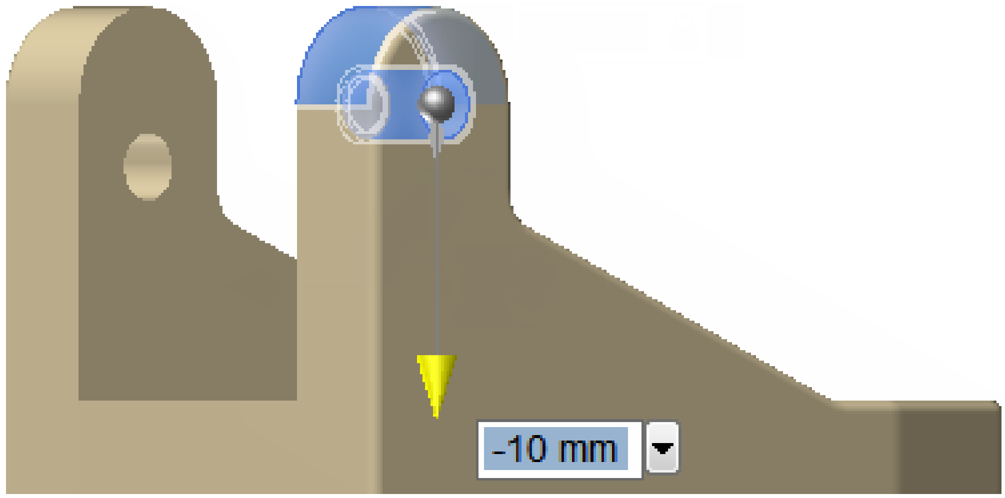



6 Start move and select the hole. The initial triad orientation depends on where you clicked on the hole. Click the Reorient Triad button. Select one of the plane manipulators, and then the back face. Click the axis manipulator that is parallel to the hole, and then select the hole to align it.




**NOTE** The triad can have a different orientation, depending on the initial triad orientation and which manipulator plane you selected. Make sure one of the axis manipulators is vertical .

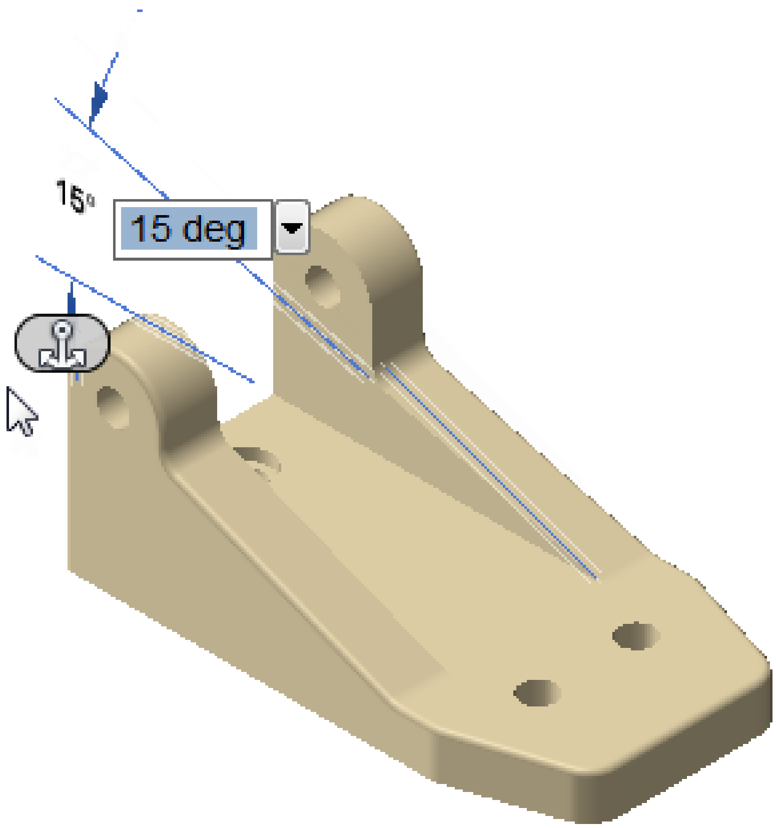
7 Select the vertical manipulator and drag it upwards. The hole and the radius stay concentric, but the hole on the other side does not move. In this model, the extrusions on both sides move together because the back face connects them. The holes do not have a relationship because they were created separately.

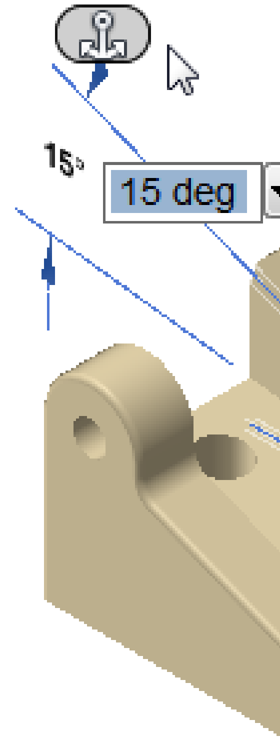



8 Press Esc to cancel the command. Start Body Constraints and select the holes.

**NOTE** The back face connects the extrusions, so they update together. You could also select either radius for the constraint and the model would have the same update behavior. However, if a cut is made in the back face of the part, the extrusions would not stay connected. A shaft goes through the holes, so they must be aligned even if the geometry changes.

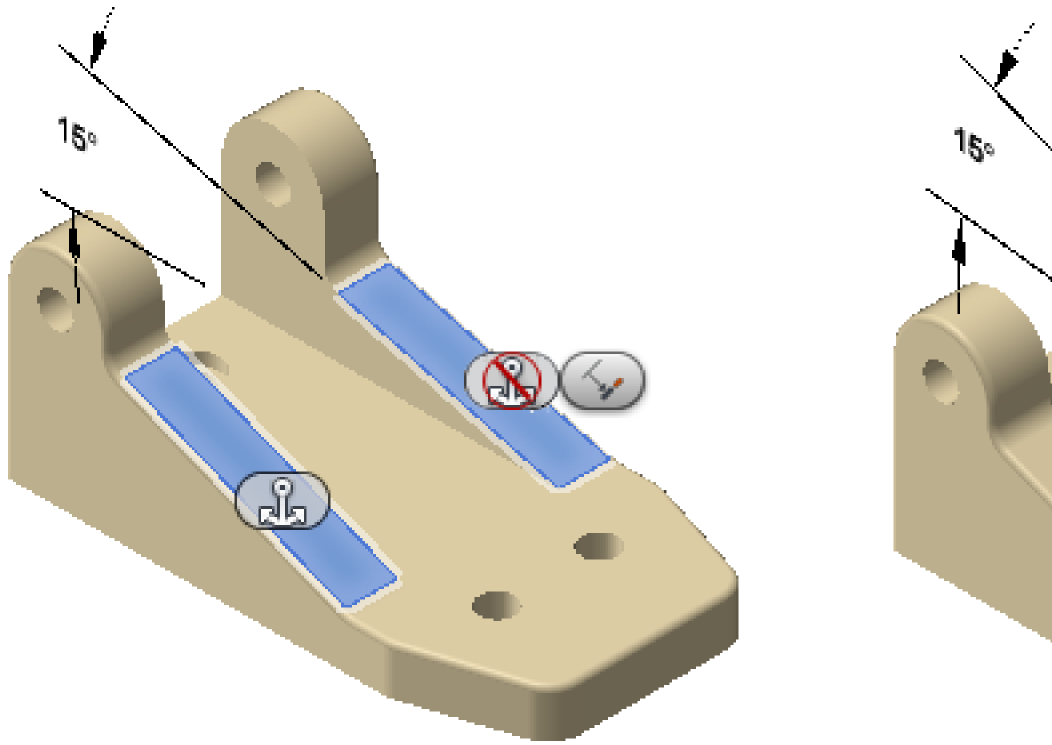
9 Return the model to the Isometric view and start the Dimension command. Select the edge of the angled face and the bottom edge of the cut. Click to place the dimension behind the model and press `Esc` to end the command. Double-click the dimension and enter 15. Move the cursor above and below the dimension- the anchor glyph indicates which edge is fixed. Position the cursor so the top face of the bracket remains fixed and press `Enter`.



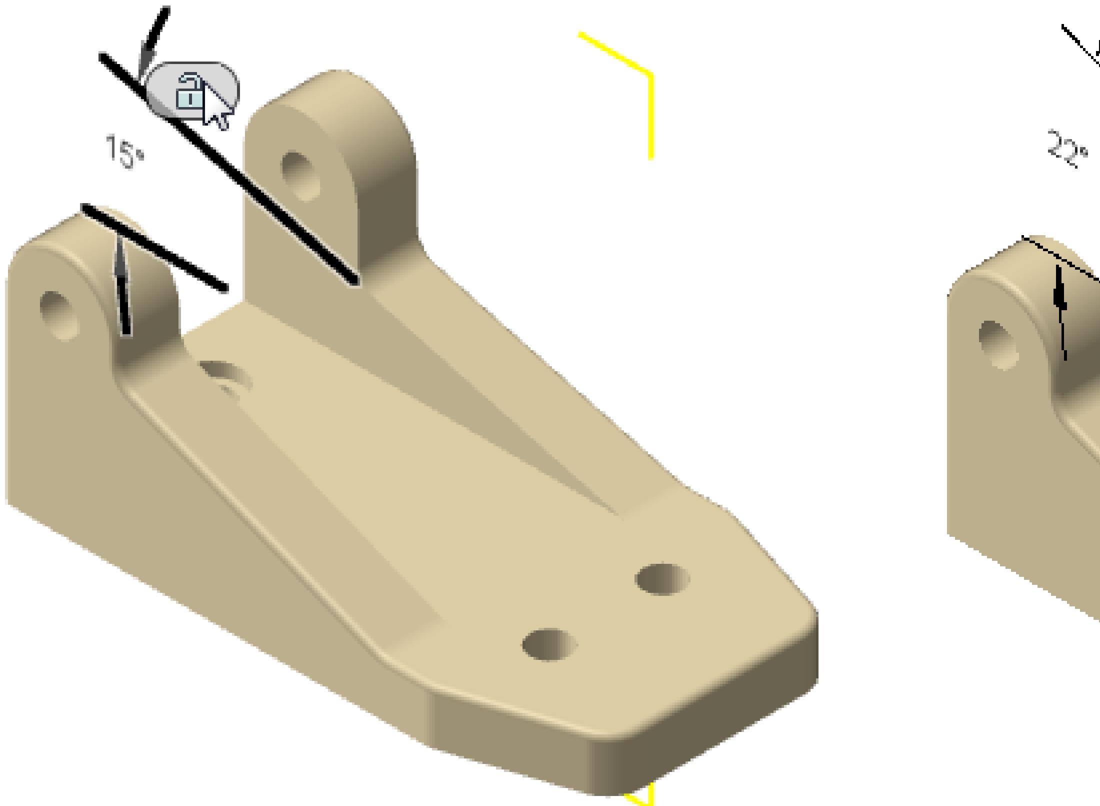

**10** Start the Body Constraint command, select the Coplanar constraint type, and then select the angled faces.

**11** Start the Body Constraint command, select the Coplanar constraint type, and then select the angled faces. The anchor glyph indicates which face is fixed and which face moves. Activate the anchor for the face without the dimension and press `Enter`.

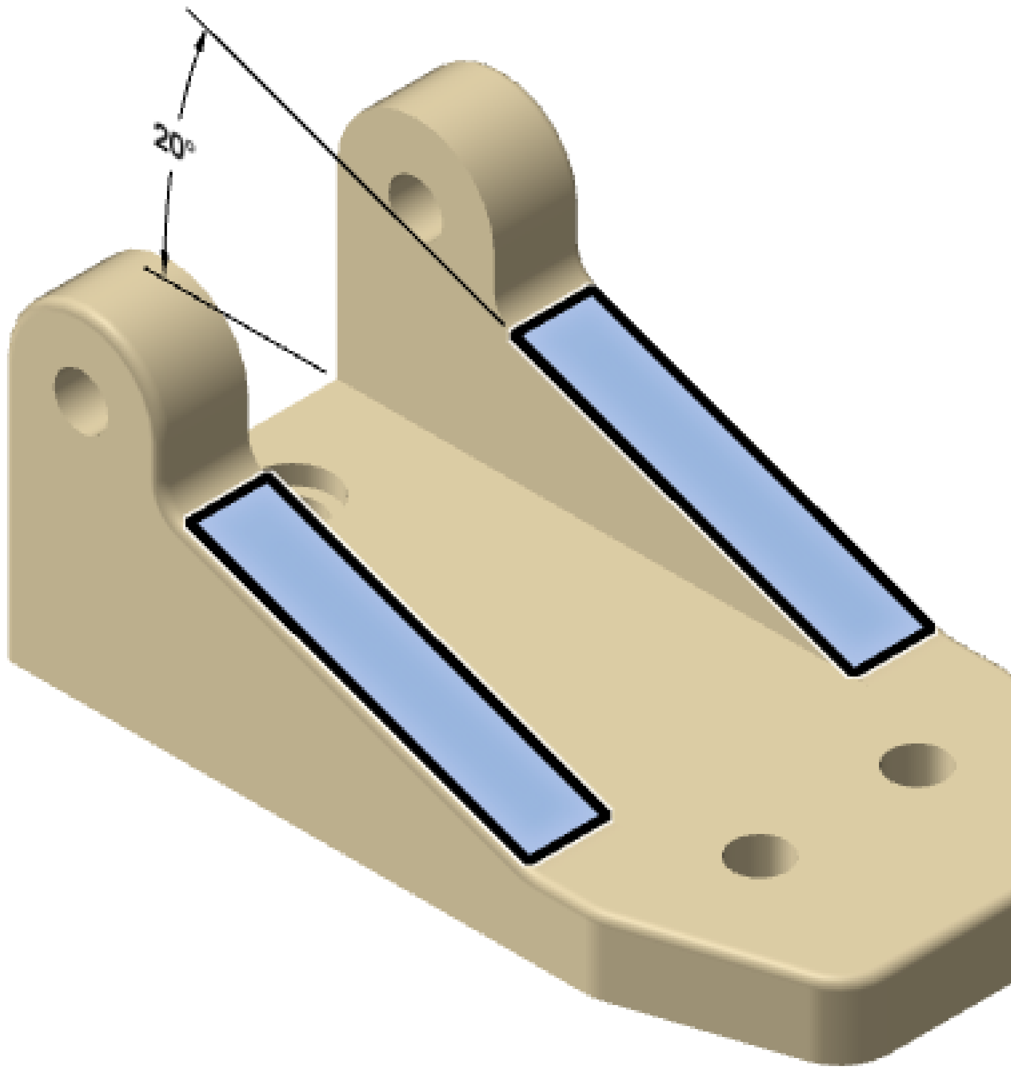



Adding the constraint causes the top face of bracket to tilt. The face with the dimension was not anchored, so it moved to align with the other face. Since the dimension was automatically locked when you edited it, the top face of the bracket moved to maintain that dimension.

**12** Click Undo to restore the model geometry. Click on the dimension, and then click the padlock button to unlock it. Apply the coplanar body constraint again, and make sure that the non-dimensioned face is anchored. Press `Enter` to apply the constraint. This time, only the face with the dimension moves, and the dimension updates with the new value.




**13** Double-click the dimension and enter `20`. Position the cursor so the top face of the bracket remains fixed and press `Enter`- both faces update to the 20 degree angle. The anchor only effects the update behavior when the constraint is created. After that, either face can be changed and the other one updates to match.

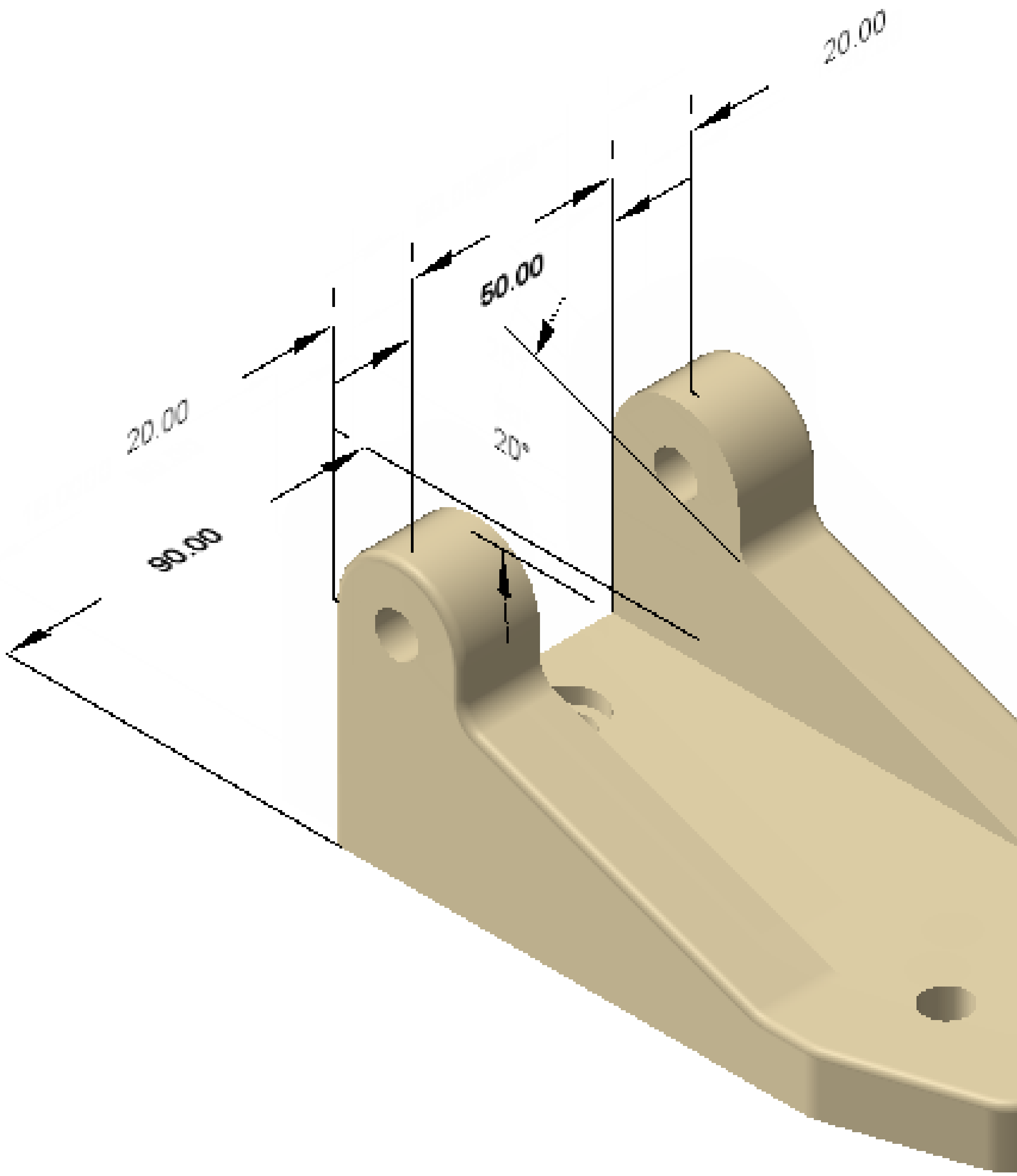



In the next section of the tutorial, we look at more methods for using model dimensions to define geometry and create annotations.

The same model is used for the next section. We work with other model geometry, so you can keep the model open, or close it and open the original model again.

## Use model dimensions

In this section, we use model dimensions to change the size of the model, lock a critical dimension, and annotate the model.

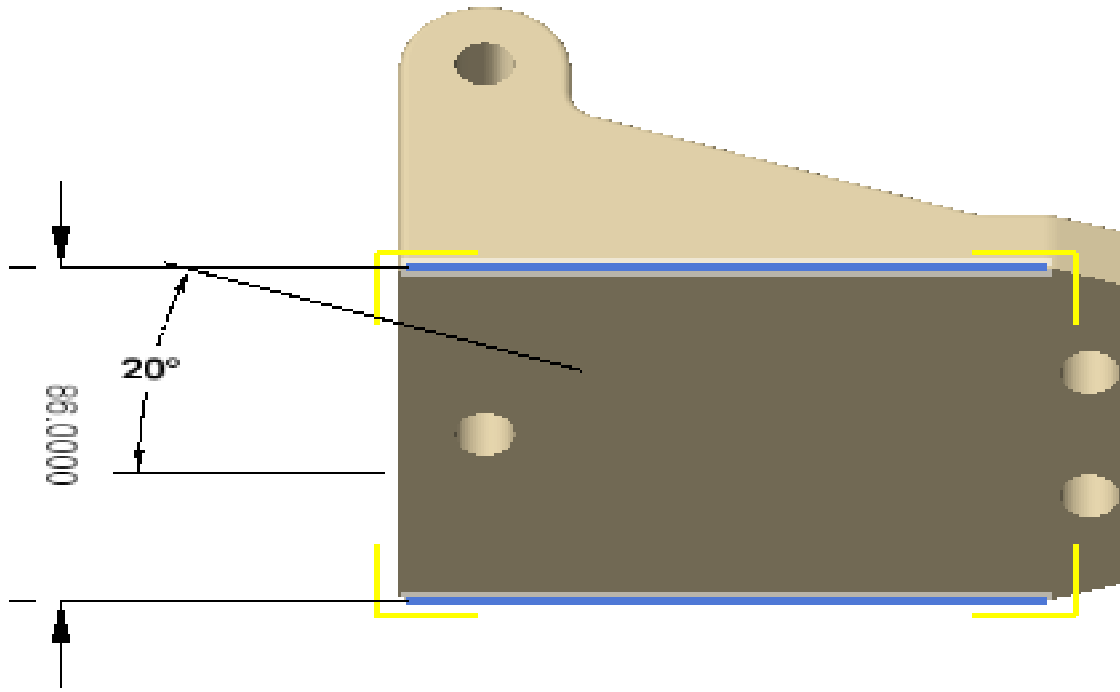

20.00
50.00
20.00
20°
90.00

Model dimensions display on annotation planes. The annotation planes are created automatically, and they are reused when possible. For some dimensions, such as the length of an edge, you can choose an alternate annotation plane.

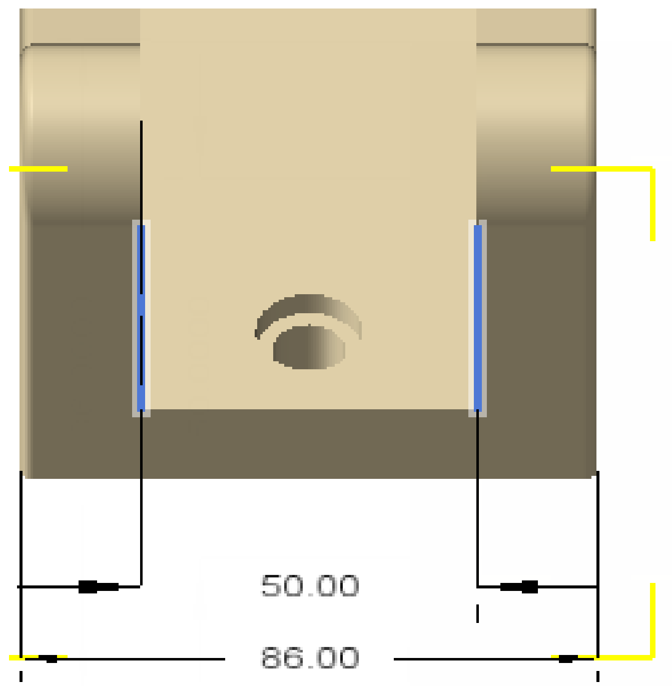
Model dimensions are created on edges, so you cannot create a dimension between faces. By default, a dimension displays the current value and updates as the model changes. You can lock a dimension to keep the current value. Also, dimensions are locked automatically after editing.

1 Open the tutorial file Body Constraints and Dimensions.dwg, if necessary. Rotate the model and click the Front- Bottom edge of the ViewCube.

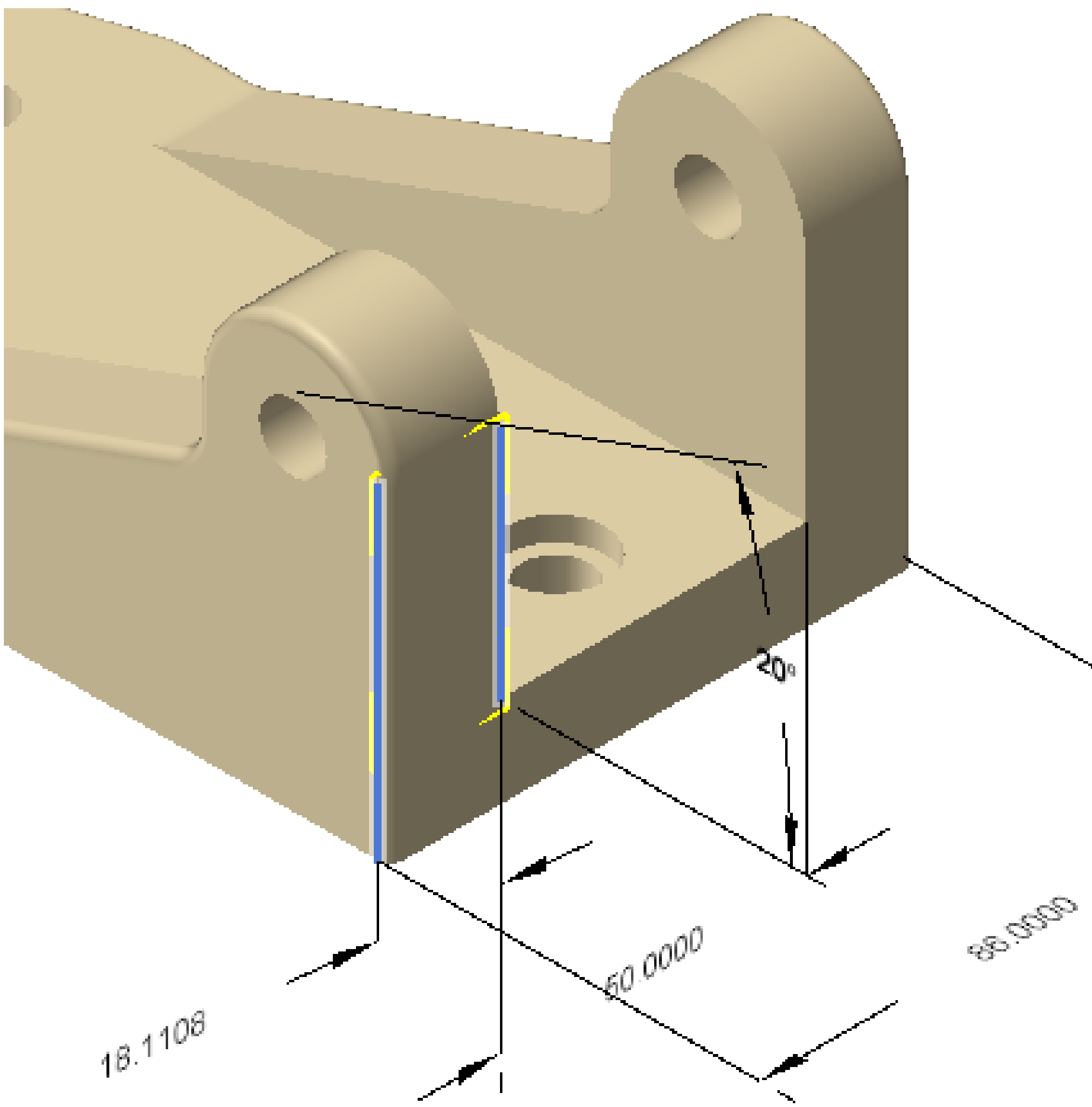
2 Start Dimension from the Constrain and Dimension panel. Select the side edges and place the dimension.



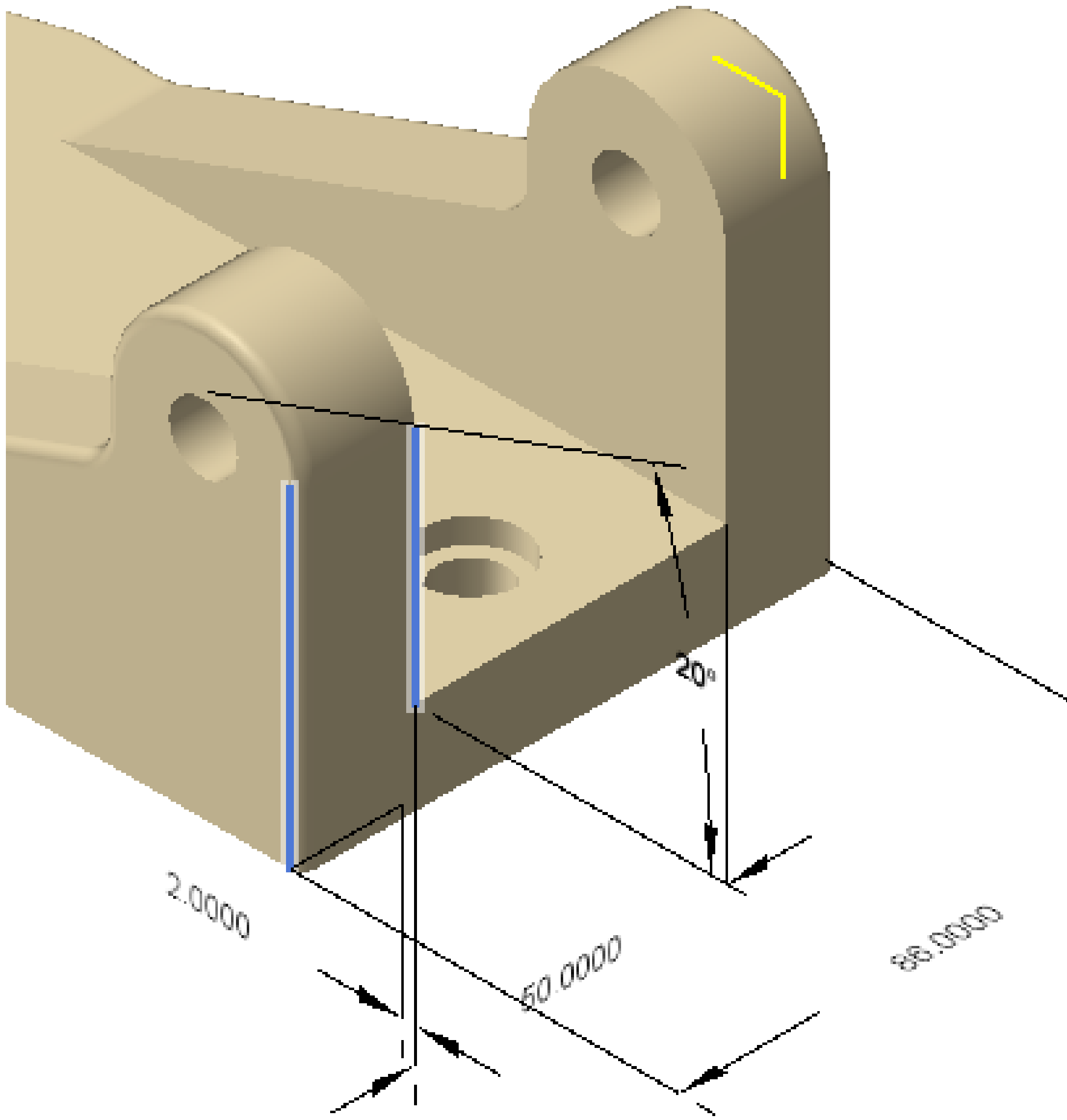

3 Click the Left-Top edge of the ViewCube. Select the right inside edge, and then the left inside edge.



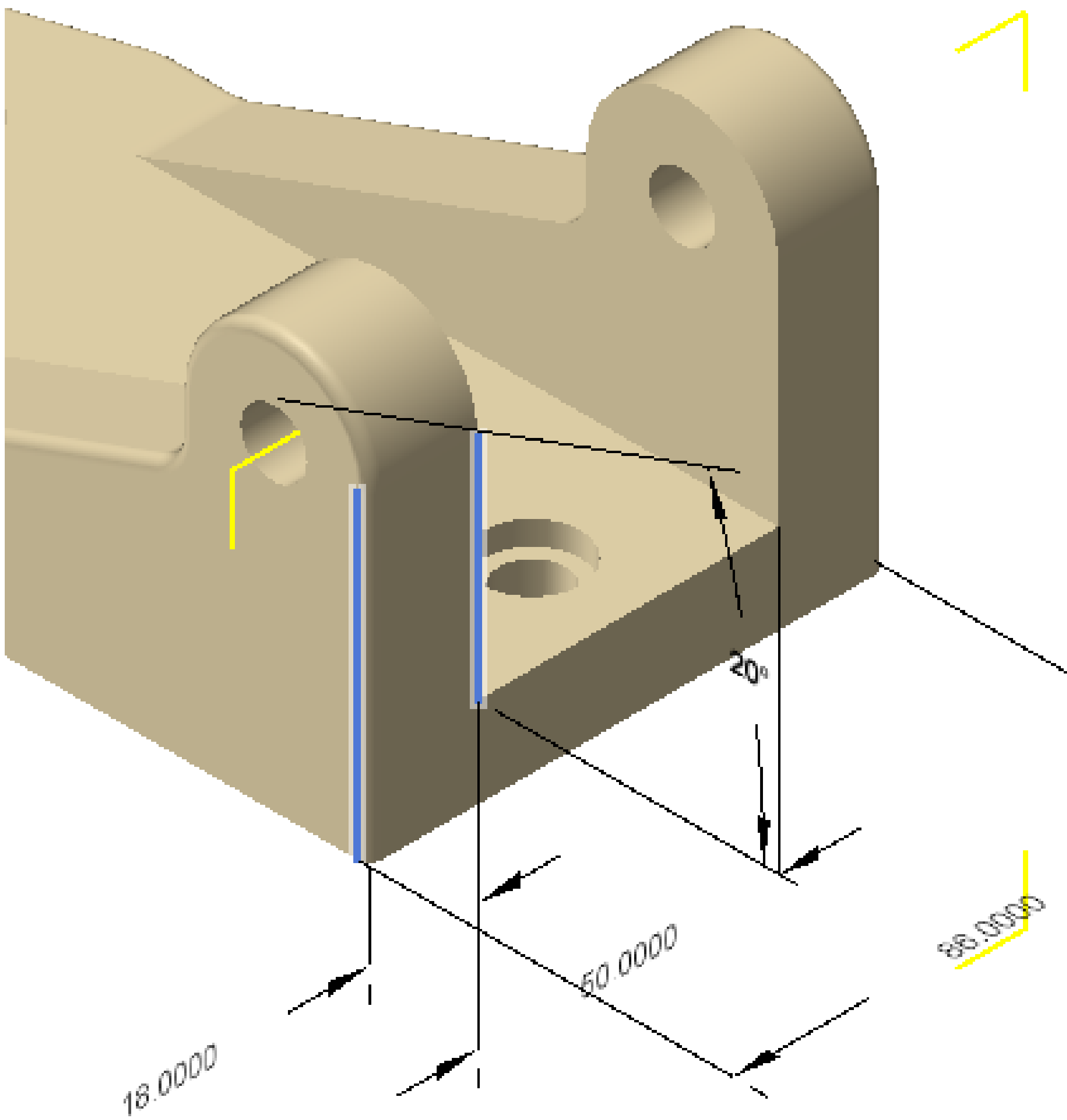

4 Click the Left-Top_-Back corner of the ViewCube. Select the edge between the fillet and the back face, and the inside edge of the left face. Drag the dimension away from the model- the default annotation plane is coplanar with the edges.




Press `Tab` to switch the annotation plane to the Back face- the dimension value is 2 mm.



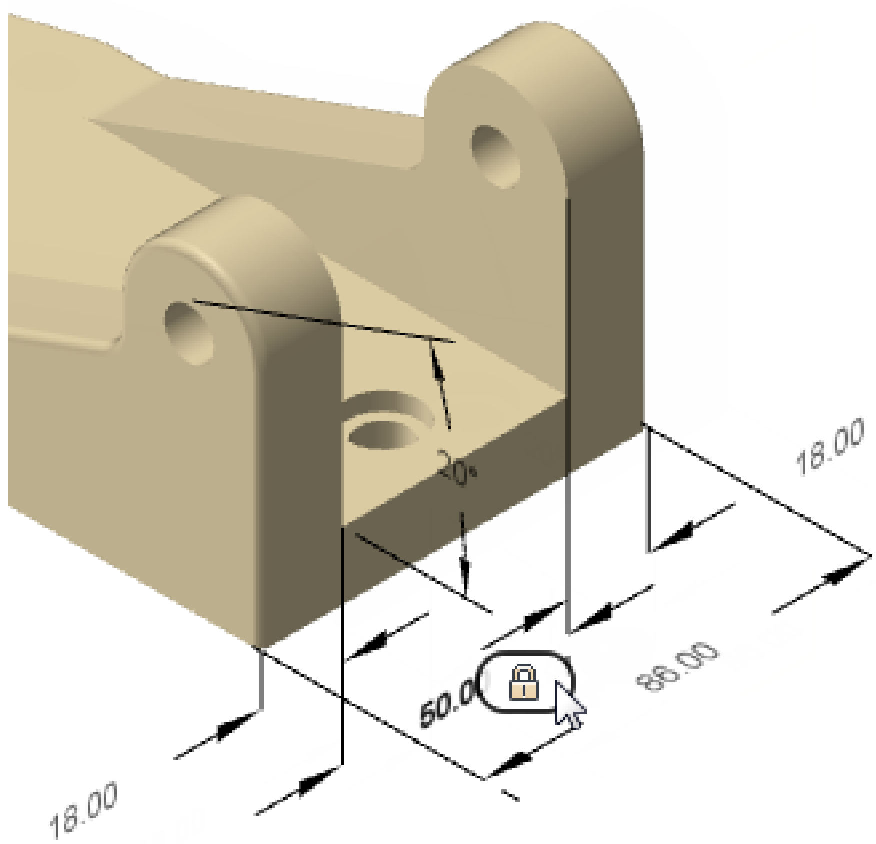

Press `Tab` again to switch the annotation plane to the Left face- the dimension value is 18.0000 mm. Place the dimension, and then dimension the other side.




**NOTE** Since these dimensions are attached to a fillet edge, you cannot edit or lock them.

5 Right-click on a dimension, and select Dimension/Constrain> Dimension Precision>2.12 from the context menu.

6 Pause the cursor over the 50.00 dimension and click the padlock glyph.



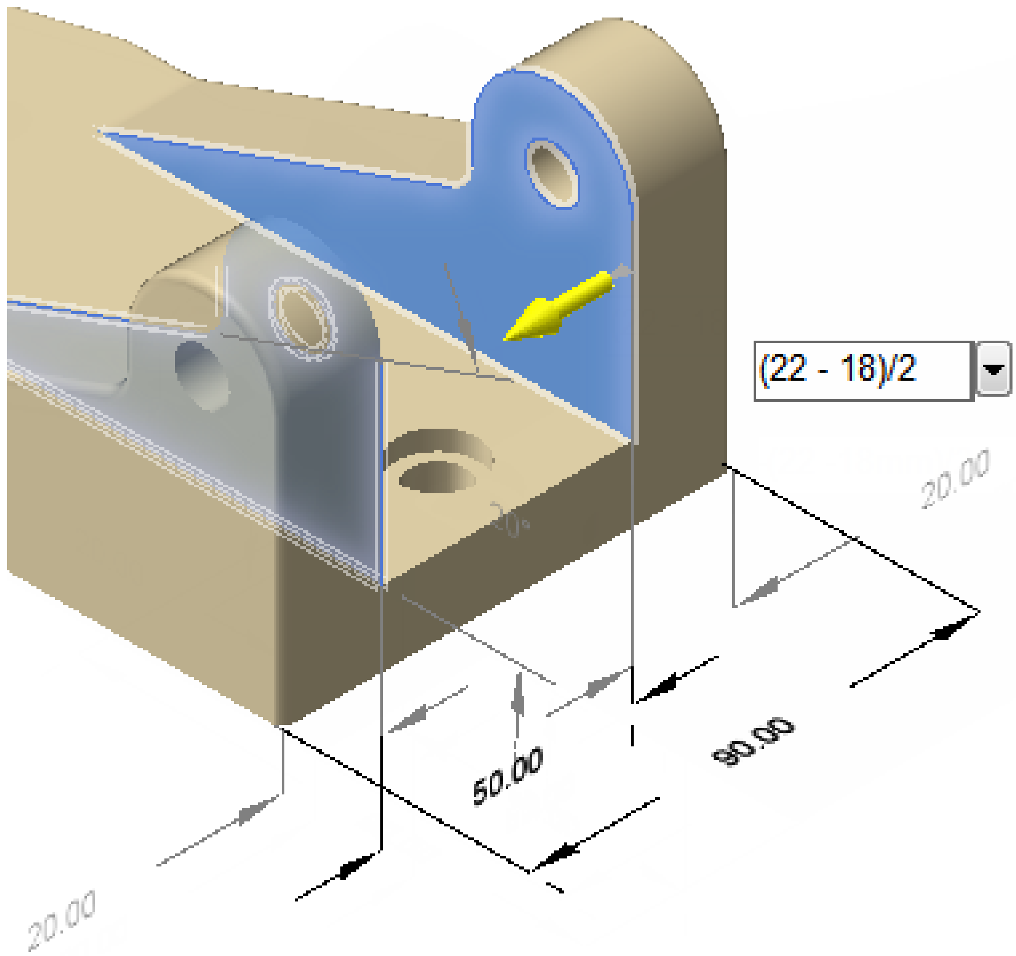

7 Double-click the 86.00 dimension and enter `90.00`. Since the 50.00 mm dimension is locked, one of the side dimensions updates to 22.00 mm.

8 We need to center the 50 mm dimension, but we cannot edit the dimension on either side. We can use an equation in Press/Pull to move an inside face and shift the cut.

    1 Start Press/Pull and select the inside face. Since we know both dimensions, we can enter them in the value field.

2 Press the `Delete` key to clear the field, and then enter `(22 - 18)/2` . Both dimensions update to 20.00 mm, so we know the cut is centered.

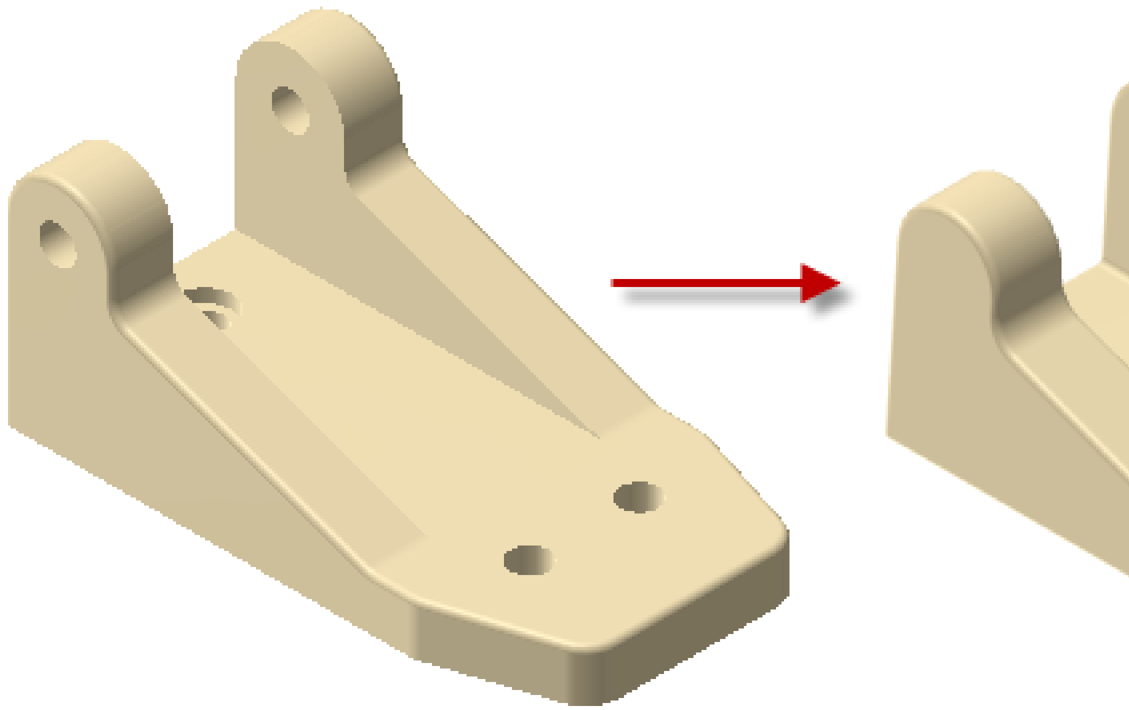
3 Press `Enter` to move the face.




In the next section of the tutorial, we simplify the model and add draft to create a casting pattern. We use the original file for the next section, so close this file.

# Convert the model to a casting pattern

In this section of the tutorial, we modify the model to create a casting pattern. We simplify the model, add draft, and then add fillets and rounds.



Before we can start modifying the model, we need the parting line, pull direction, minimum section thicknesses. We also need to know which surfaces are machined. For this part:

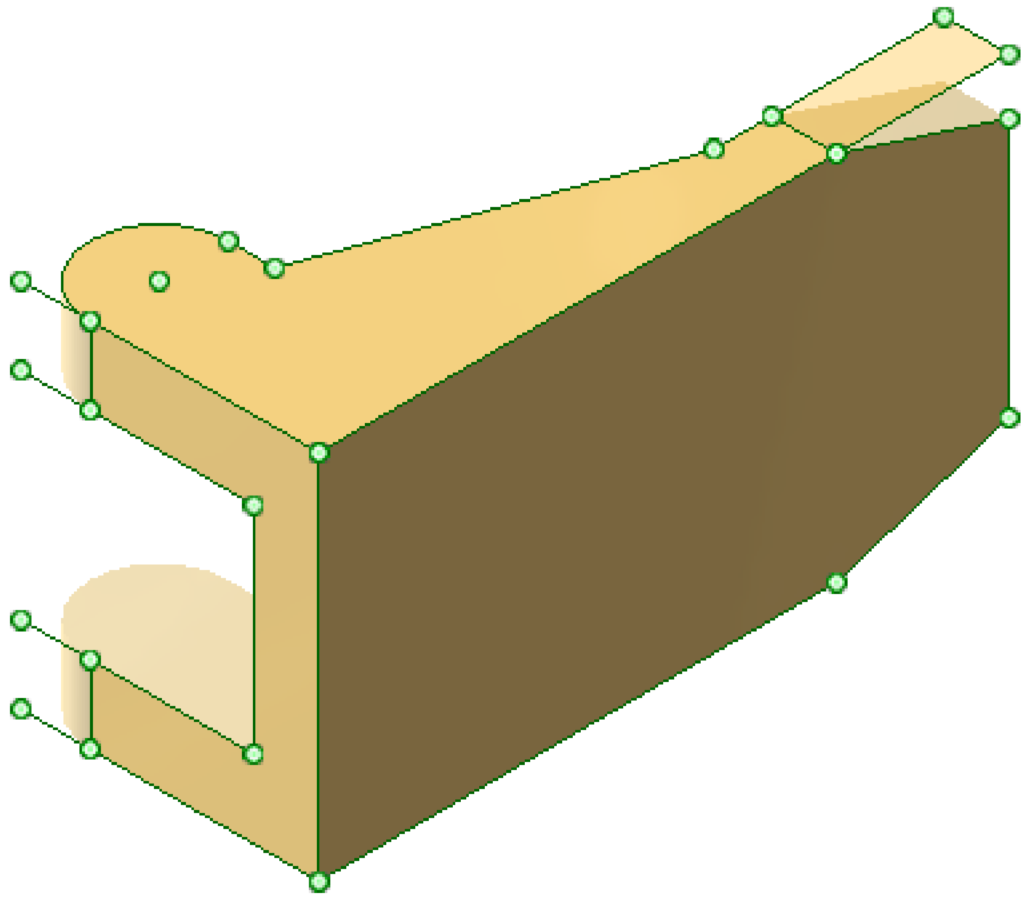
- The bottom of the part is the parting line, so the pull direction is vertical.
- The bottom of the part is machined.
- The width at the top of the extrusions cannot be reduced.
- The inside faces of the extrusions are machined.

1 Open the tutorial file Body Constraints and Dimensions.dwg, and save it as Advanced Modeling Tutorial- casting pattern.

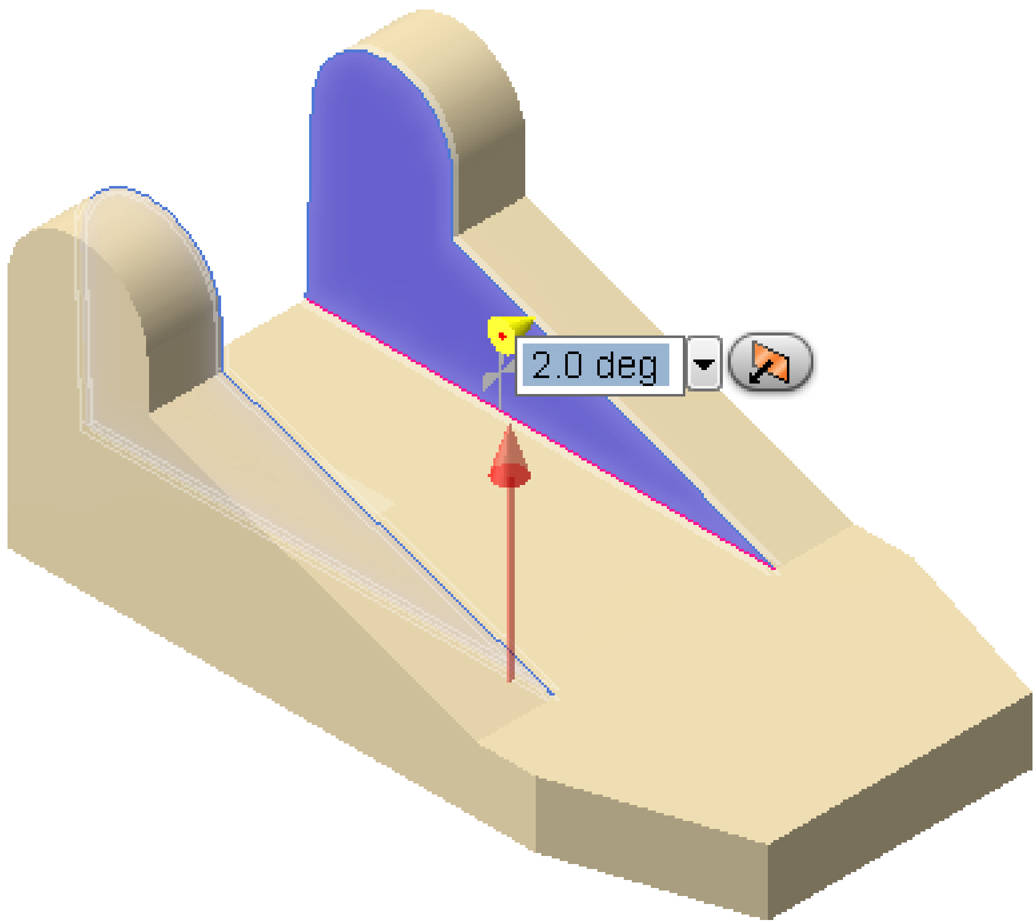
2 Start the Simplify command from the Simulation Panel. It automatically selects all fillet, chamfer, and hole features.

We need the chamfered faces, so deselect the Chamfer filter. Confirm the Action is set to Delete, and click the check mark.

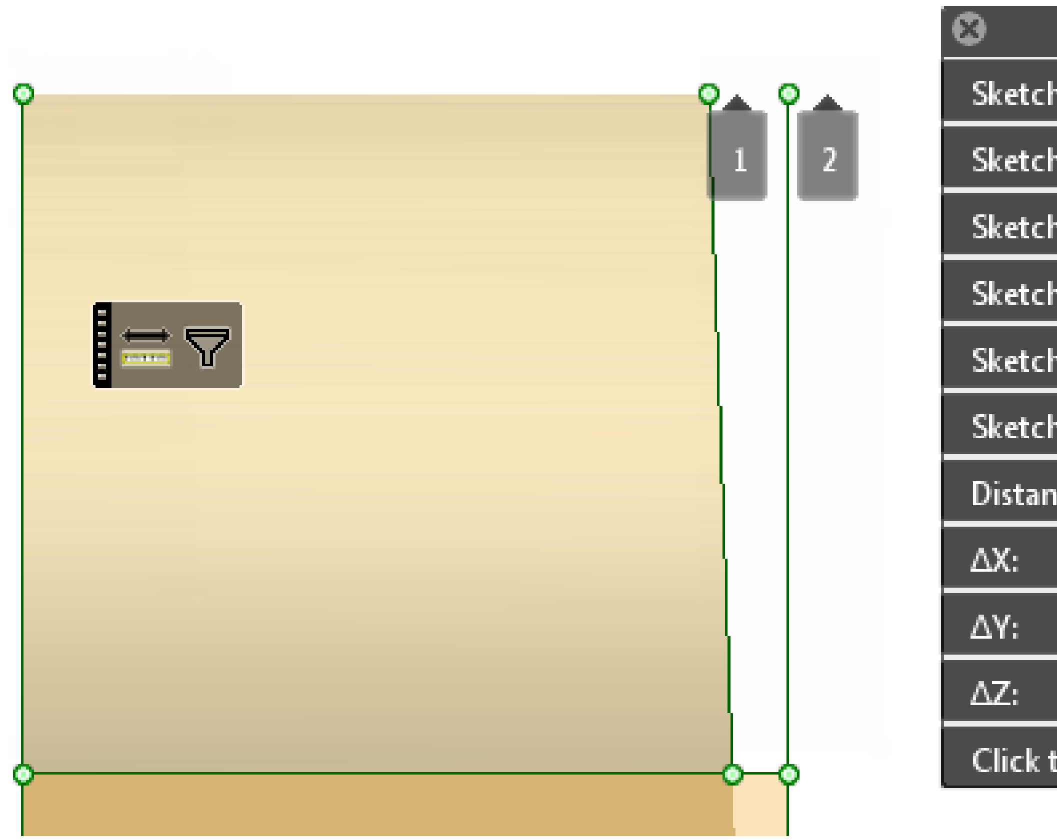
3 Before we add draft, we should capture the current size and shape of the model. Since Fusion does not create a link between sketches and the model, projected geometry in sketches does not update. Create sketches on the Left, Front, and Bottom faces, and project geometry as necessary. Rename the sketches to match the views, and turn off the visibility.



4 Start the Draft command and select the top face of the base for the neutral plane. Select the inside faces, enter `2` for the draft angle, and then press Enter. The taper makes the extrusions narrower, so we need to increase the width.



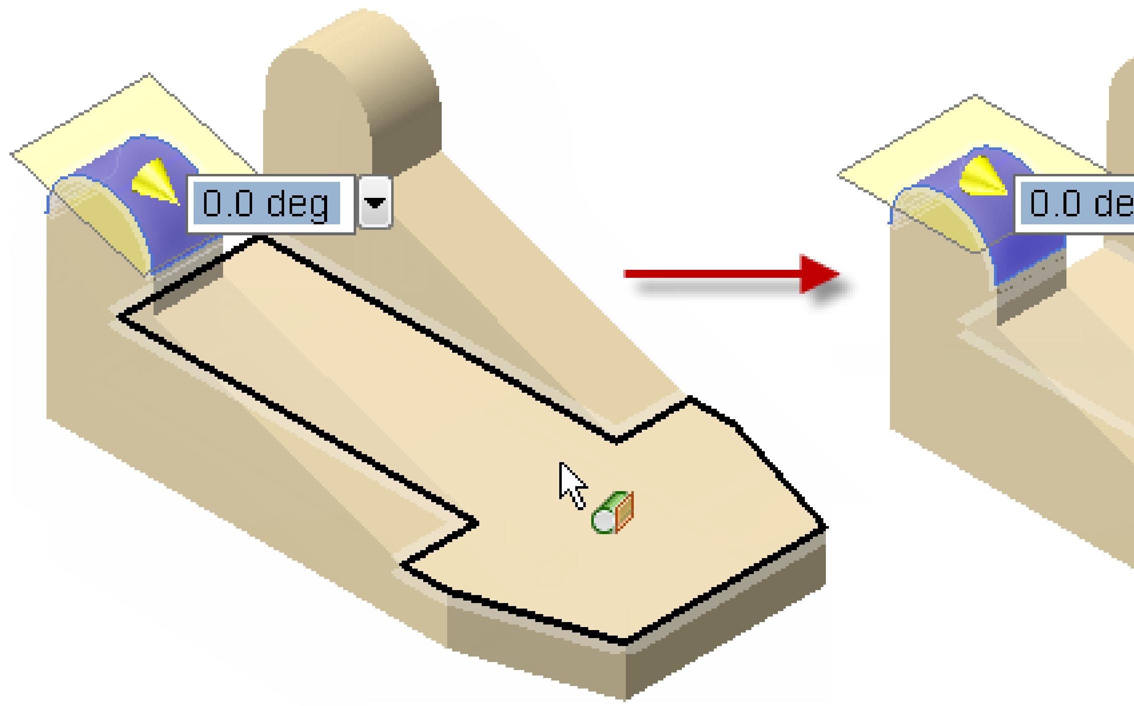

5 Use the ViewCube to rotate the model to the Left face. Expand the Sketches folder in the browser and edit the sketch for the left face. Project the inside edge of an extrusion so we can measure how much material was removed. Start Measure and select the endpoints- the gap is 1.9 mm. We also need a 2 mm machining allowance, so we will increase the width by 4 mm.




Start Press/Pull, and 4 mm to each inside face.

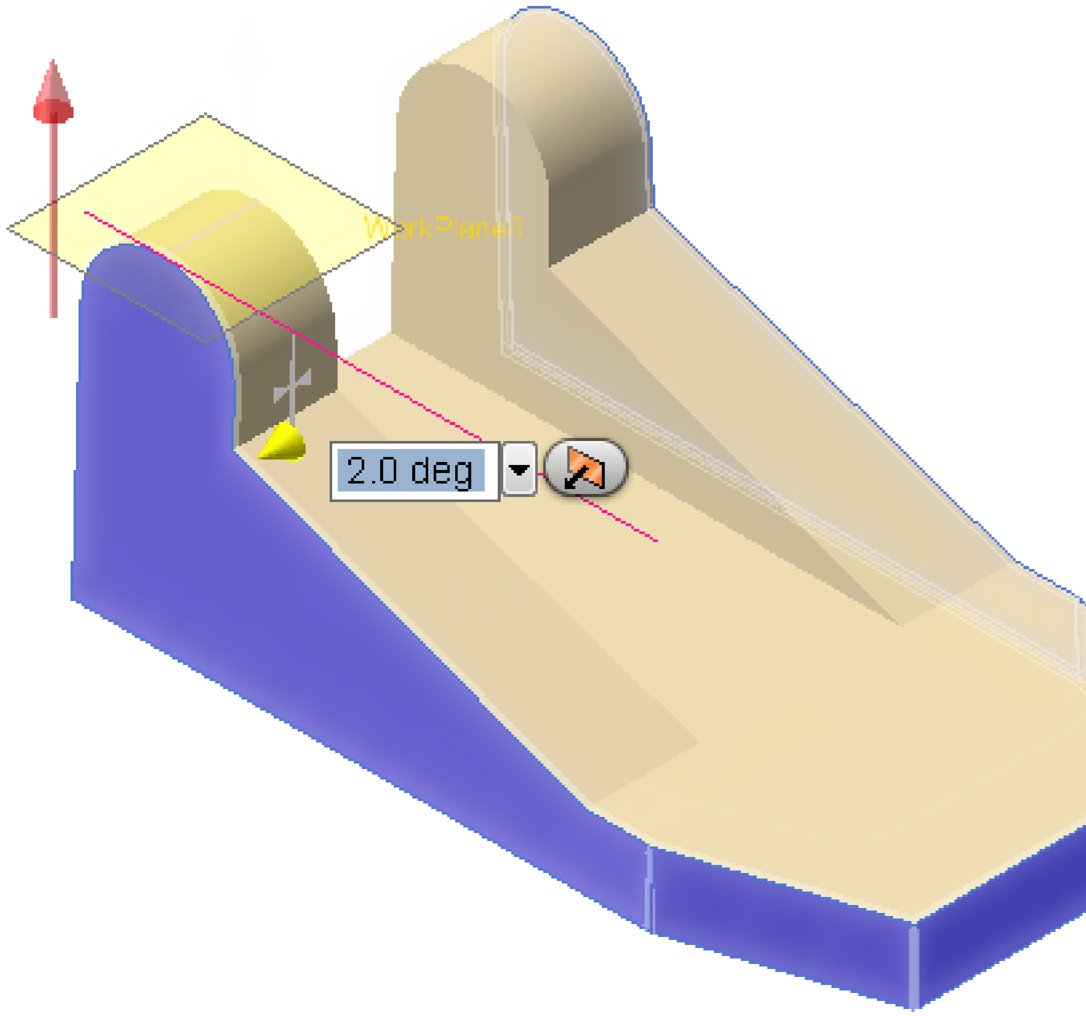
6 The draft on the outside cannot remove material. If we use the bottom of the model as the neutral plane, we have to add material. A simpler method is to create a work plane tangent to the top of the extrusions, and use it as the neutral plane.

Click the drop-down arrow next to Plane and select Plane At Tangent. Select one of the rounds, and then select the top face of the base to provide an angular reference.

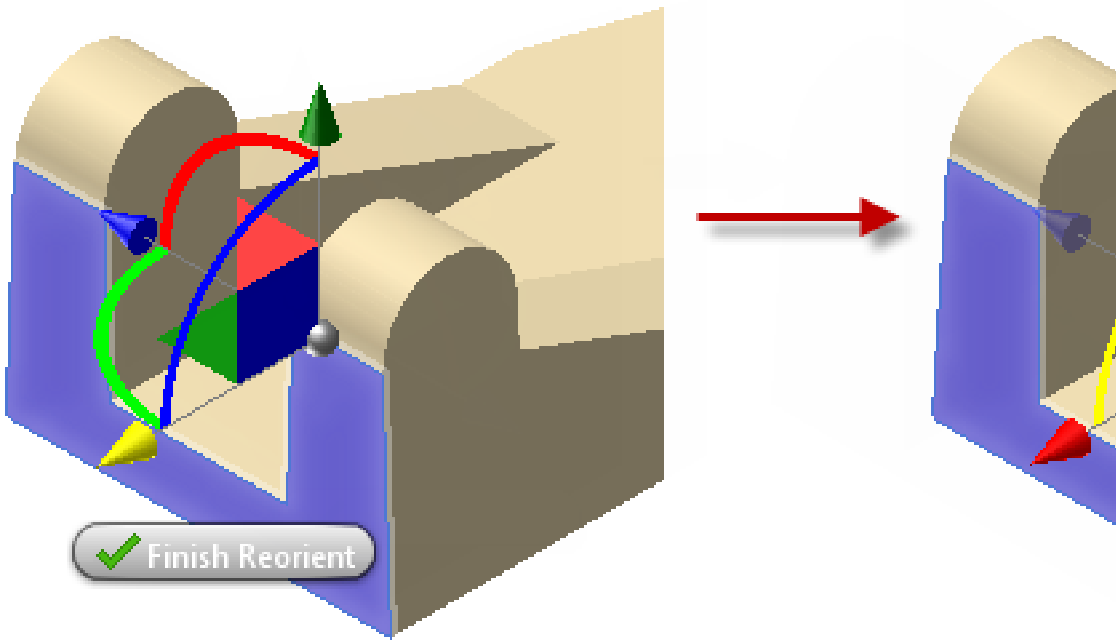



7 Start Press/Pull and add 2 mm to the bottom face of the part for a machining allowance.

8 Start Draft and select the work plane, and then select the outside faces. When you enter a draft angle, the error symbol displays. Press the Ctrl key, click a face to remove it from the selection set, and then release the Ctrl key to update the preview. The error does not display when you remove the Left face. Since that face is tangent to the round, the taper cannot be added from the neutral plane. Add the other faces, if necessary, and create the 2.0 degree draft.




**9** Rotate the model and click the Top-Left-Front corner of the View Cube. Since the left face is tangent to the round, we cannot use the Draft command to add the taper. The solution is to use Move to rotate the face at the tangent. Start Move and select the Left face. Click the Reorient Triad button, select a tangent edge on the face, and then click Finish Reorient. Select the arc manipulator and rotate the face 2.0 degrees.




10 Now that all of the faces have draft, we can add fillets and rounds to finish the pattern. Create 8 mm fillets on the short edges, and 2 mm fillets on the rest of the edges.

# Introduction to Direct Modeling

Inventor Fusion uses Direct Modeling. Direct modeling does not use parameters, and the features do not have a history. Direct modeling simplifies model creation and editing because you can change model geometry without editing the original feature.

## About this tutorial

| | |
|---|---|
| **Category** | **New Users** |
| **Time Required** | 30 Minutes |
| **Tutorial File Used** | |

Inventor Fusion uses Direct Modeling. Fusion models do not use parameters, and the features do not have a history. You can create standard model features like extrusions and fillets. After you create a feature, you simply move and change the size of the geometry.

### Objectives

- Learn the Fusion user interface for creating and editing features.
- Create primitive features.
- Create an edge fillet.
- Use the manipulators to create and modify geometry.
- Use the triad to move geometry.

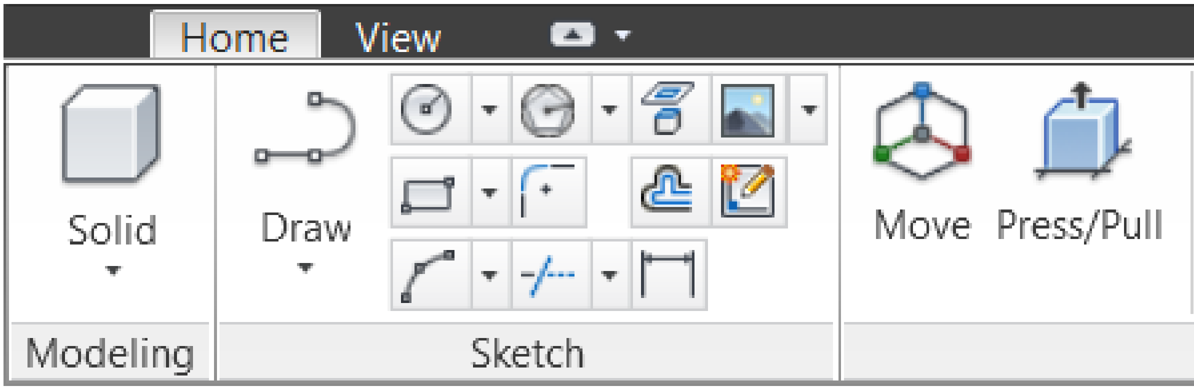
### Navigation Tips

- Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

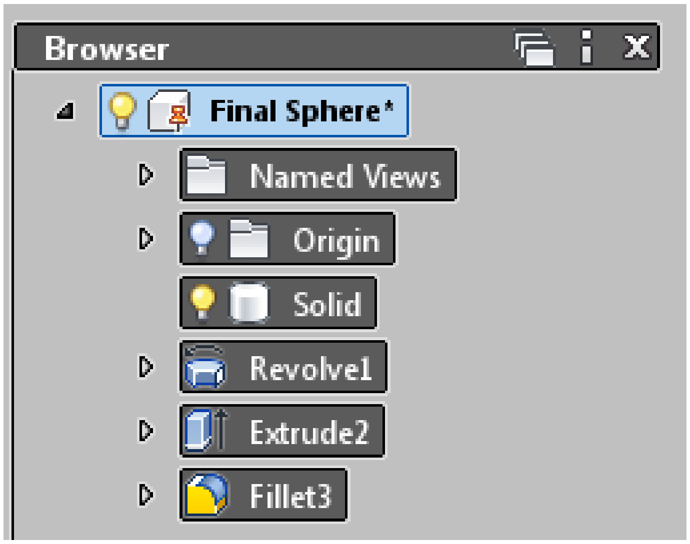
## The Fusion User Interface

The primary components of the Fusion user interface are the ribbon, the browser, and a Heads Up Display (HUD). The HUD provides access to most modeling commands in the graphics window. This provides a very efficient modeling workflow by reducing mouse motion.

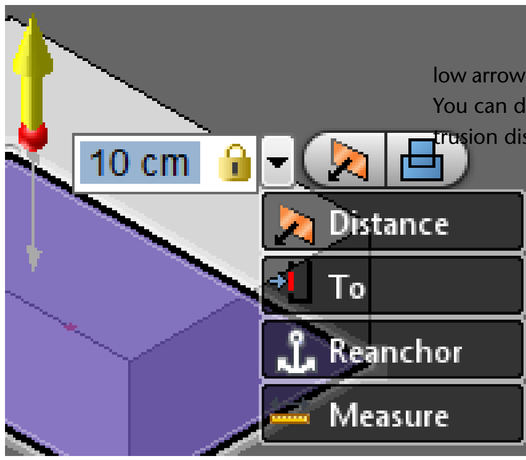
The ribbon has two tabs- Home contains the modeling commands, and View contains commands for navigating the model and the appearance and display controls.




The browser is an organized list of objects in the model. Each component can have a solid, surfaces, sketches, features, and other modeling objects. In the image below, the model has three features- a revolve, an extrude, and a fillet.



The HUD includes several types of controls including a mini-toolbar, buttons, glyphs, and manipulators. The HUD is context sensitive, which means that controls only display when they are active.




oolbar includes a value entry field ns for selecting options. The yellow arrow is a manipulator for an extrusion. You can drag the arrow to change the extrusion distance.


When you pause the cursor over the model, the selection glyph displays. A list of selection options displays when you move the cursor to the glyph.

## Create Primitive Features

Fusion has four basic types of feature commands.

- Primitive features are basic geometric shapes, like a box or a sphere.
- Sketch-based features use a profile to define the shape.

■ Placed features are created on existing geometry. Fillets and chamfers are created on model edges, and holes are created on faces.

■ Modifying features change existing geometry. For example, the taper command adds a draft angle to model faces.

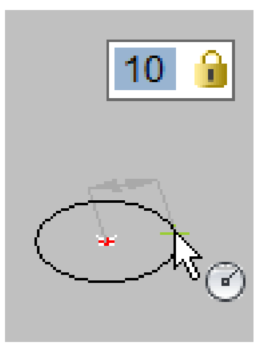
The primitive commands simplify modeling because they combine the sketch and feature creation for common shapes. Since direct modeling does not use parameters or history, you can rapidly create model geometry and adjust the size and position later.

In this section of the tutorial, we use the Cylinder and Sphere commands to create a solid, Push/Pull to change the diameter of the cylinder, and the Material command to change the appearance.

1 Create a new document.

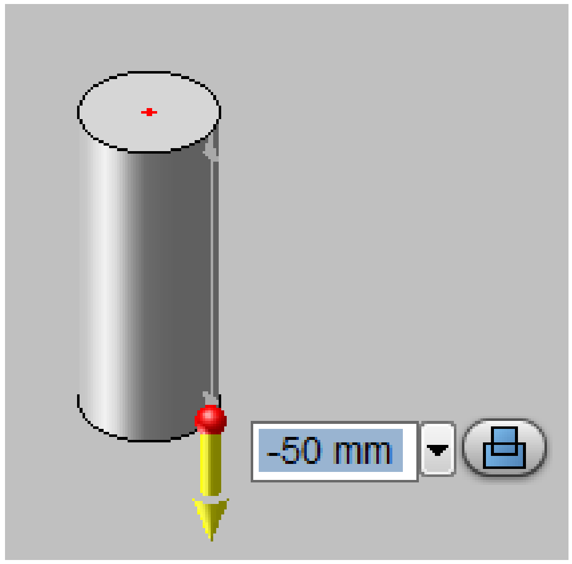
2 On the Home tab of the ribbon, click on the drop-down arrow under Box and select Cylinder. The XY plane displays and the tool tip prompts you to select the center point.

3 The red dot is location of the origin. Click on the red dot and move the cursor away. The cylinder previews and the radius dimension updates as you move the cursor. Type `10.0` and press `Enter`. A padlock displays in the edit field to indicate that you entered the value.

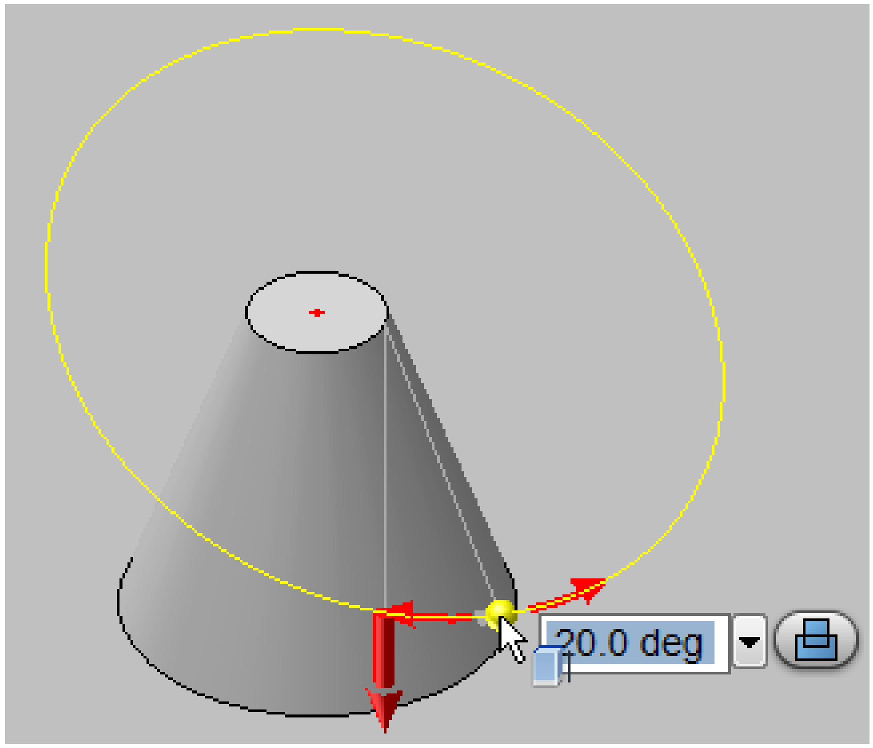



4 Move the cursor and note the size does not change. The value is set until you enter a new value.

5 Click in the graphics window, and note that a yellow arrow displays. The arrow is a manipulator, and the length of the cylinder changes as you drag it. Drag the manipulator until the value is -50 mm and click.




6 Click on the red sphere at the tail of the arrow. The sphere becomes yellow, and the arrow becomes red. The active manipulator is always yellow. A yellow circle displays and red arrows indicate that the sphere moves on the circle. Drag the sphere back and forth to change the taper of the cylinder. Enter `20.0` and click to accept the value.




7 On the ribbon, click on Push/Pull and select the cylinder. The face highlights and the manipulator arrow displays. Drag the manipulator until the end of the cylinder is a point. An error displays if the value is less than -10.0 mm. Click OK on the command ribbon to change the diameter, then click Done to exit the command.

8 On the ribbon, click on the drop-down arrow under Cylinder and select Sphere. Click on the red dot, set the sphere radius to 15 mm, and create the sphere.

9 Move the cursor to the view cube in the upper right-hand corner and click on the house. The house sets the view to isometric and zooms into the model. Click on a corner of the view cube and drag it to rotate the model.

## Modify Geometry

In this section of the tutorial, we change the size and location of features, edit a counterbored hole, and add fillets and chamfers.

With direct modeling, you can move faces to change the size and shape of a part. Initially, any object in the model can change. As the design progresses, you can define the size and location of geometry to limit the type of changes.

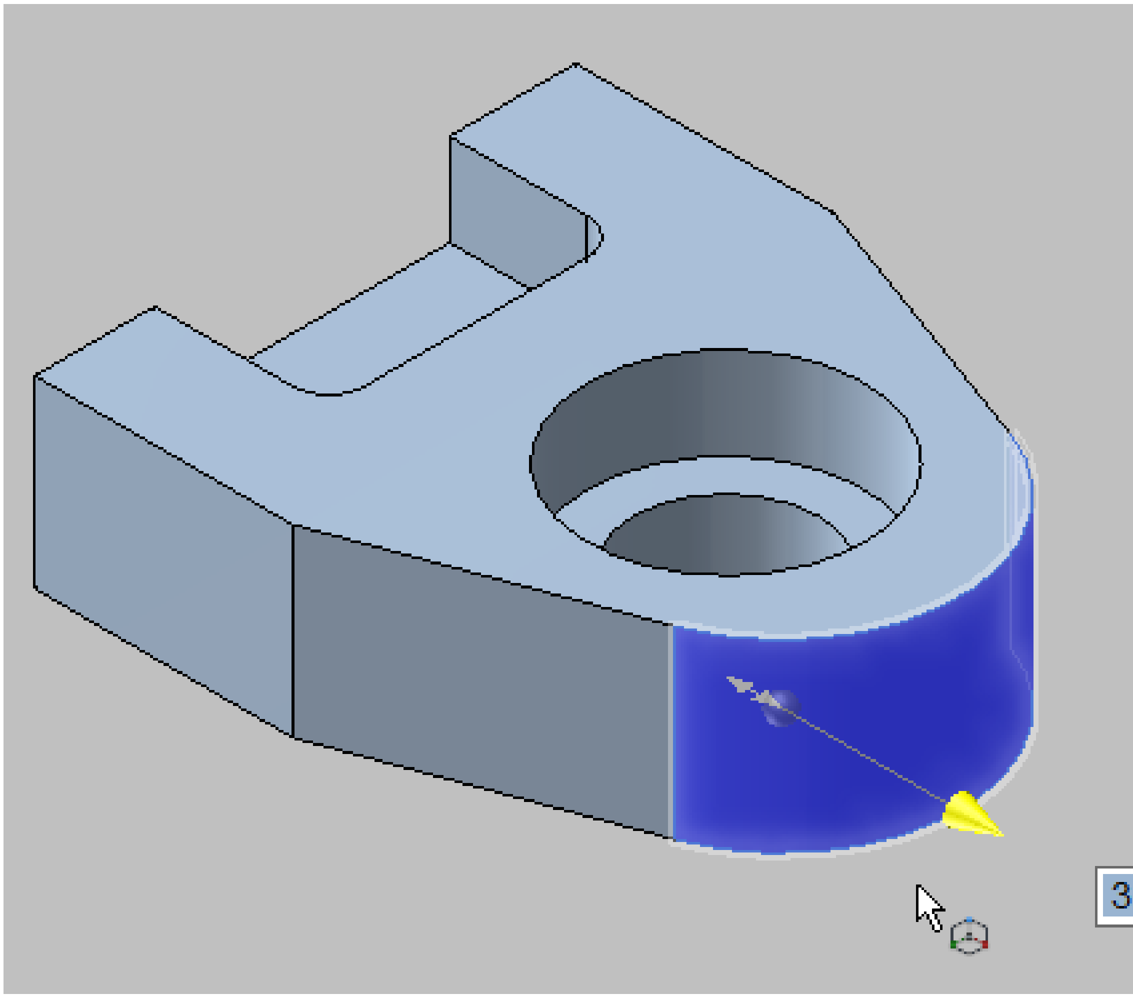
1 Open the Modify Geometry.dwg tutorial file.

2 Rotate the model so the large fillet is visible. Click on the fillet face to select it. On the Solids panel, click Move. The triad displays in the graphics window.

3 The triad has four types of manipulators. The sphere moves the object in the X, Y, and Z directions. The arrows move the object along the X-, Y-, or Z- axes. The planes move the object in the XY-, XZ-, or YZ- planes. The arcs rotate the object around the axis.

4 Some of the triad manipulators are disabled because the fillet cannot move in that direction. The manipulator still display, but they are dimmed and cannot be selected. Click on the green arrow to move the fillet along the Y-axis. The arrow becomes yellow to indicate it is active.

5 Click and drag in the graphics window to move the fillet. The manipulator has priority, so you do not have to click on the arrow to drag it. As you move the fillet, the model preview updates. Drag the fillet 3 mm from the model. Press `Enter` or click OK on the command ribbon to accept the change.

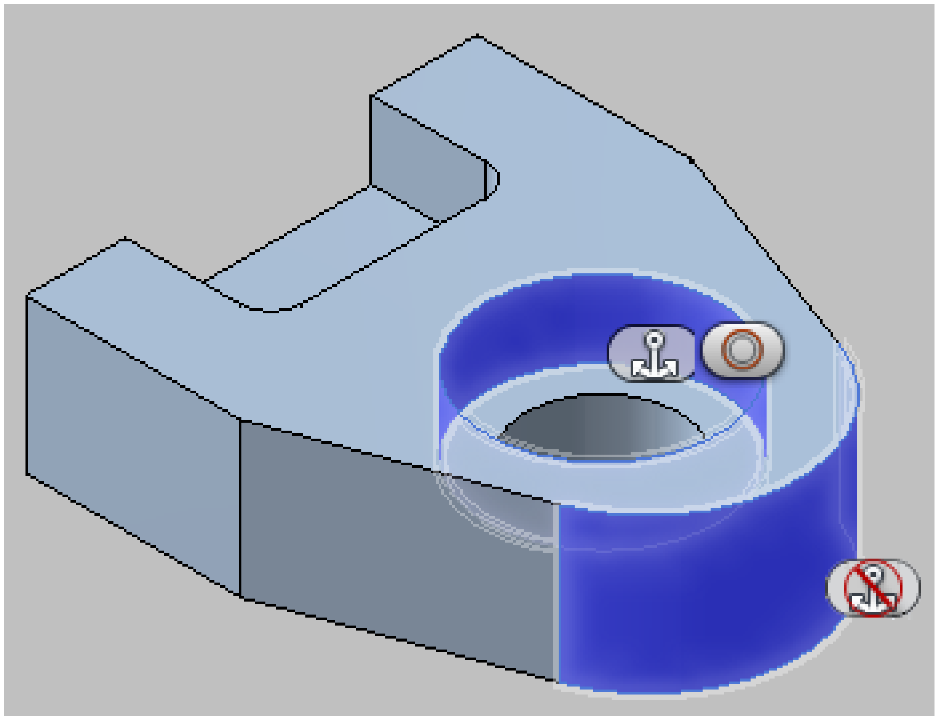



**NOTE** If you drag the fillet towards the model, it removes geometry. For example, it cuts into the hole or remove it entirely.

6 We can apply a body constraint to make the hole and the fillet concentric. Start Body Constraint from the Constrain and Dimension panel of the ribbon. Select the Center constraint type from the command ribbon, select a face on the hole and a face on the fillet and press Enter.

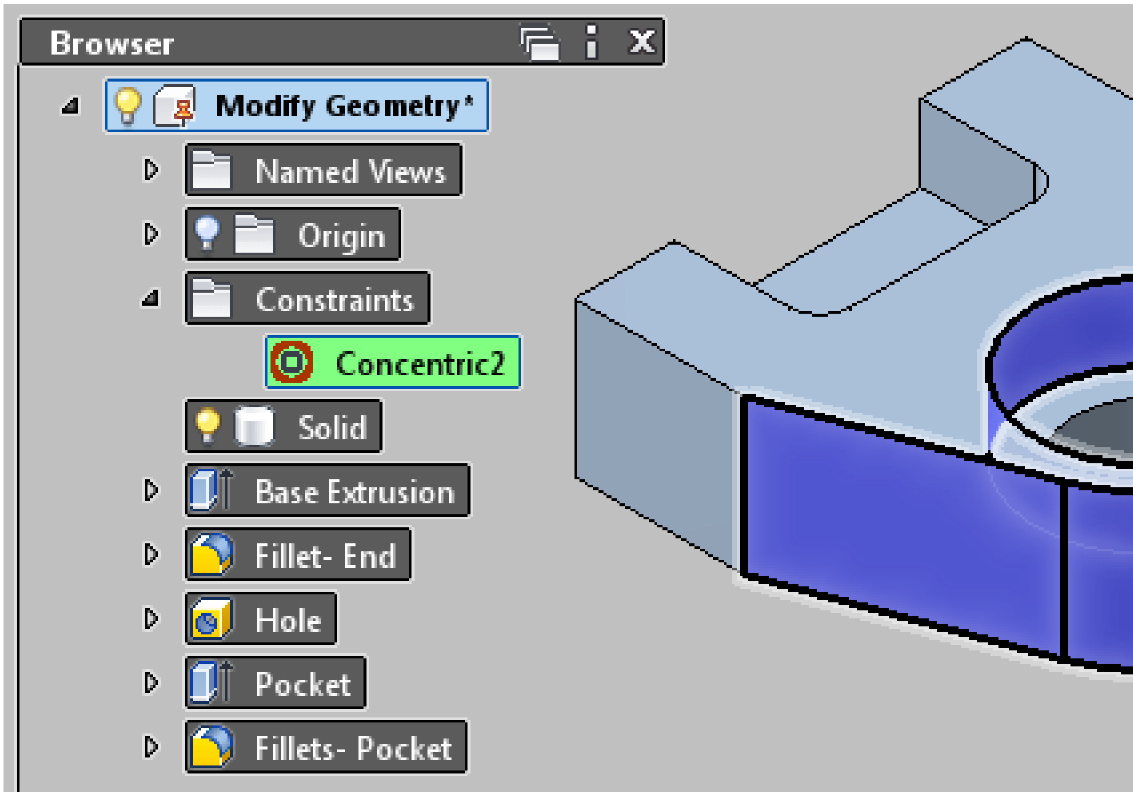
By default, a body constraint moves the second selection to the first one. In the next step, we use constraint options to control which object moves.

7 Click Undo to move the features back to their original positions. Start Body Constraint and select the hole and then the fillet. Anchor symbols display next to the highlighted faces. One anchor has a red slash to indicate it is not anchored.

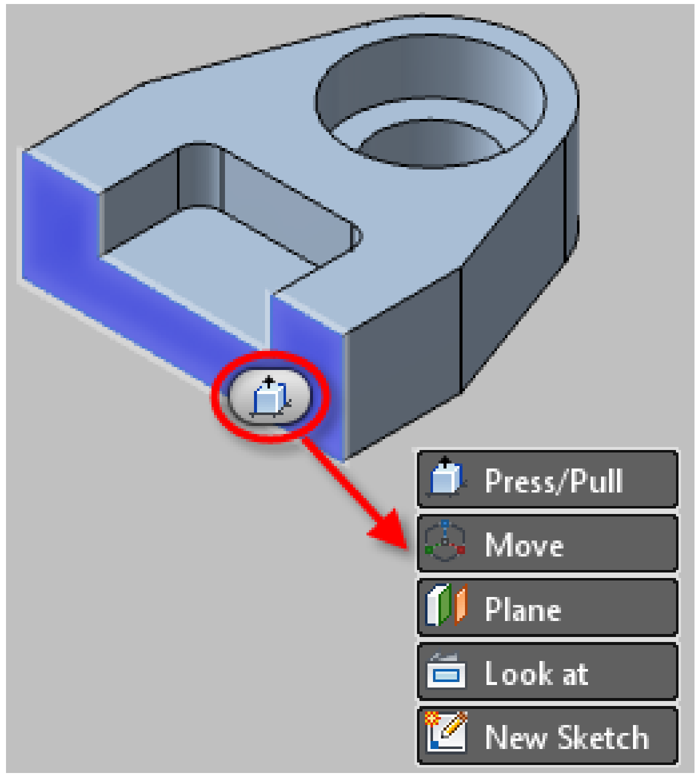


8 Click on the anchor for the fillet. The anchor for the hole now has the red anchor. Press `Enter` to move the hole to fillet.

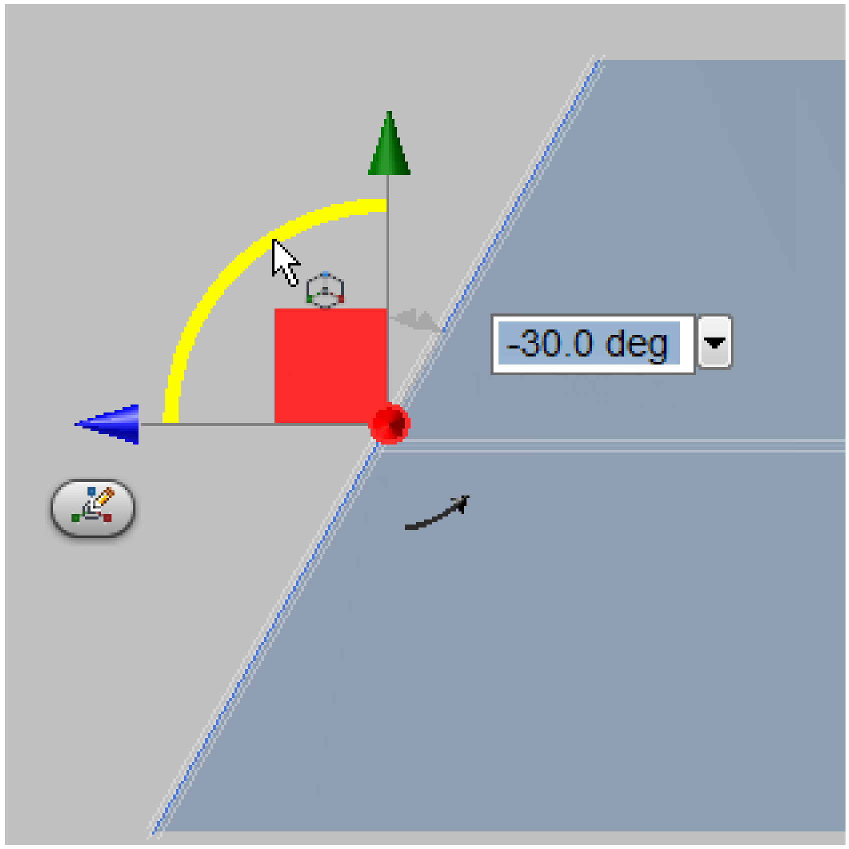
9 A constraints folder is added to the browser. Expand the browser, and click on the constraint to highlight the model faces. You can delete a body constraint, but you cannot edit it.




**10** Click on the house icon above the view cube to return to the Isometric view. Click on the end face. Pause the cursor over the command glyph to display the menu, and select Move.



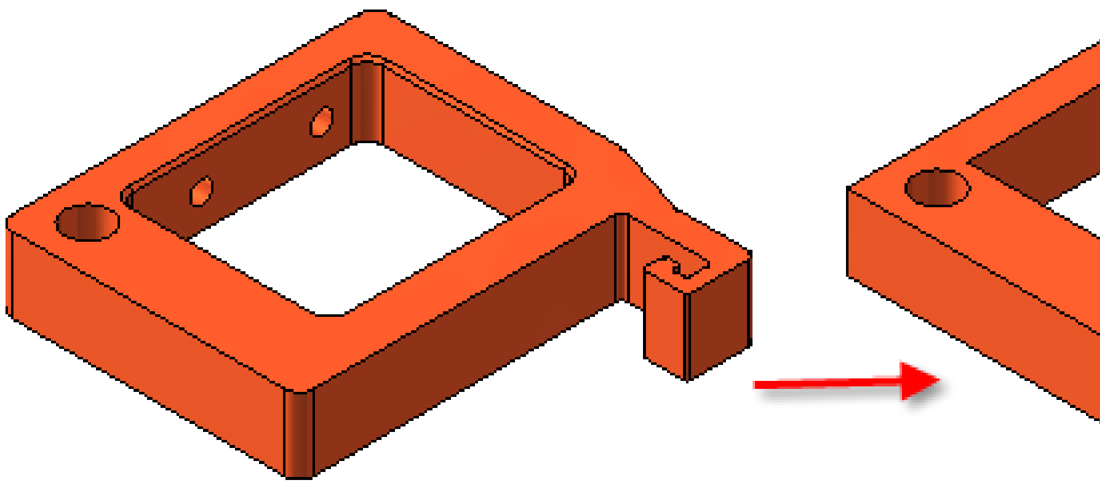

11 Click the Right side of the view cube to rotate the model. Click on the arc manipulator to activate it. Drag the manipulator to change the angle of the face. You can tilt the face towards or away from the model. Drag the manipulator to -30.0 degrees, or type the value and press `Enter`. Click in the graphics window to accept the change.




12 Return to the isometric view. Click on the top edge of the pocket and select Fillet from the command glyph. The fillet command automatically selects tangent edges. Drag the manipulator to change the size. Since this fillet removes material, you can only drag the manipulator towards the body. Drag the manipulator to 1 mm and click to create the fillet.

# Preparing models for CAE

## About this tutorial



Learn how to prepare a model for analysis programs by removing unnecessary features.

| | |
|---|---|
| **Category** | **CAE Analysts** |
| **Time Required** | 20 Minutes |
| **Tutorial File Used** | Find Features.sat, Simplify Model.dwg |

Inventor Fusion provides tools to simplify models for CAE applications. Most models have small features, like fillets, that complicate the analysis. These features do not affect the model behavior, but they can add a significant amount of time for meshing and calculations.

Fusion has two tools for removing features from models:

- The Find Features command analyzes the geometry and creates features.
- The Simplify command provides filters for selecting fillet, chamfer, and hole features.

Preparing models analysis is usually a two-step process. The Simplify command requires features, so you must first convert the geometry with the Find Features command.

### Objectives

- Learn how to find features in models.
- Learn how to filters to select features by type and size.
- Learn how to work with feature selections.

### Navigation Tips

- Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

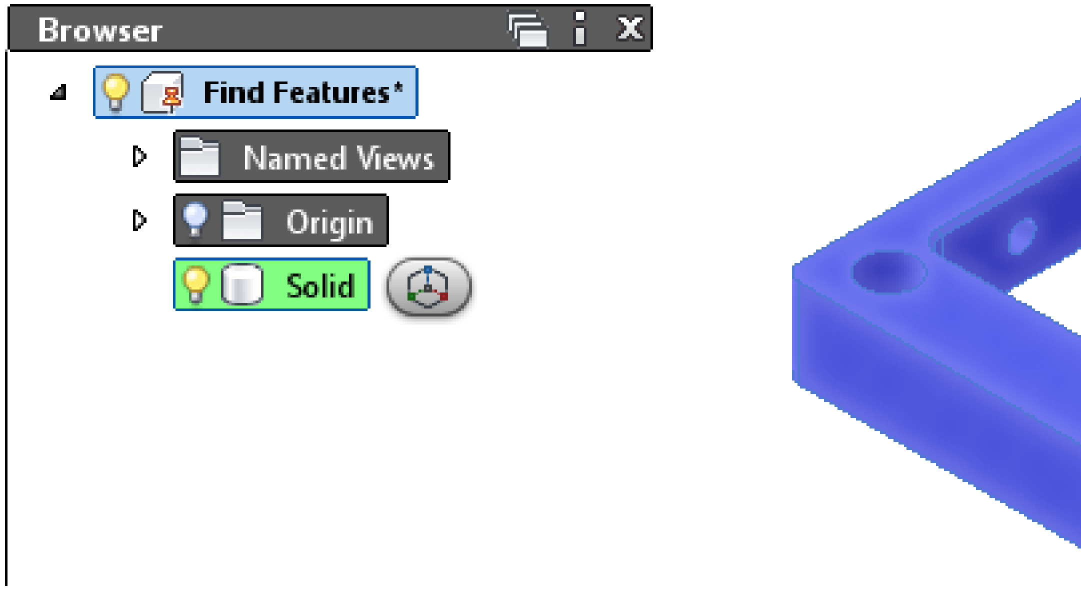
## Find Features in Models

In this section, we use the Find Features tool to analyze the geometry in an imported model and create features.

The Find Features command is usually the first step in the process of simplifying models for CAE applications. Many models only have geometry, which must be converted to features before using the Simplify command. For example, the Find Features command converts cylindrical cuts into hole features.

1 Open the Find Features.sat model.

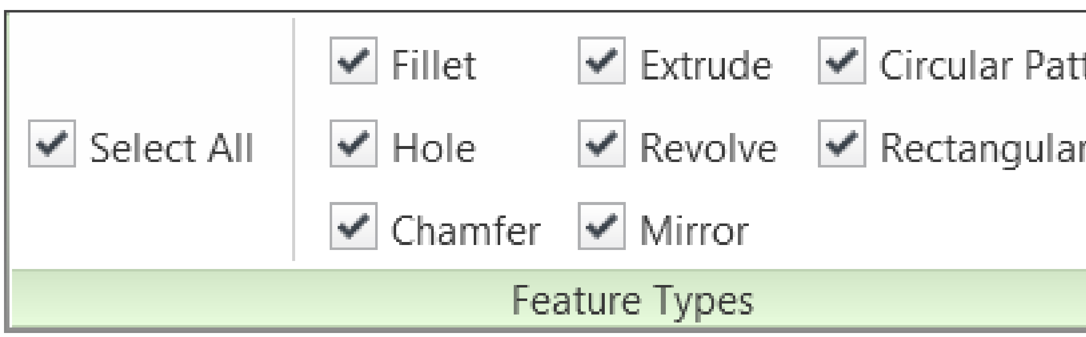
2 The browser contains a list of the objects in the model. Since the model was imported from a neutral file format, the browser only has an entry named Solid. Click on the icon next to Solid. The entire model is selected and highlighted in the graphics window.




3 The Find Features model opens in the isometric home view. Use the tools in the Navigation bar to rotate, pan, and zoom. The model has fillets on every corner and chamfers on the inside edges. There are two small counterbored holes on one side. Click the Home button above the View Cube to return to the Isometric View.

4 Locate the Manage panel on the Home Ribbon, and start the Find Features command.

5 The Find Features command ribbon replaces the standard ribbon.




6 Find Features can analyze the model for eight types of features. All feature types are selected by default.

7 Click Select All to clear the feature type selections, then select Fillet and Chamfer.

   The model is analyzed for geometry that matches the selected features. Limiting the types of features speeds up feature recognition process and produces more consistent results.

   **NOTE** The Find Features results can vary with the feature type selection. For example, a cylindrical cut can be recognized as a Hole or a Revolve feature, since they create the same geometry.

8 Select Fillet and Chamfer.

9 Click on Solid in the browser, which selects the entire model. The OK button is enabled because you have satisfied the requirements for feature types and geometry.

   **NOTE** You can also select a group of faces for analysis. Press Shift while clicking on faces to define the select set. Selecting specific geometry can reduce the analysis time for a complex model.

10 Click OK. The browser has a list of the individual model features. There are two chamfers and 18 fillets.

11 Click on a chamfer feature in the browser to highlight in the model. There is one chamfer feature for eight individual chamfers because they are tangentially connected.

12 Go down the browser list and pause the cursor over each fillet icon. The edges of the fillet feature highlights in the model but face does not. You can scan the browser for a feature without selecting each object.

13 Start the Find Features command, select Hole as the Feature type, click the solid, and click OK. The three hole features added to the browser.

14 Select one of the small holes in the browser. Right-click and select Edit Hole. The geometry was recognized as a counterbored hole.

The next section of the tutorial uses a new model for the Simplify command. You can close this model.

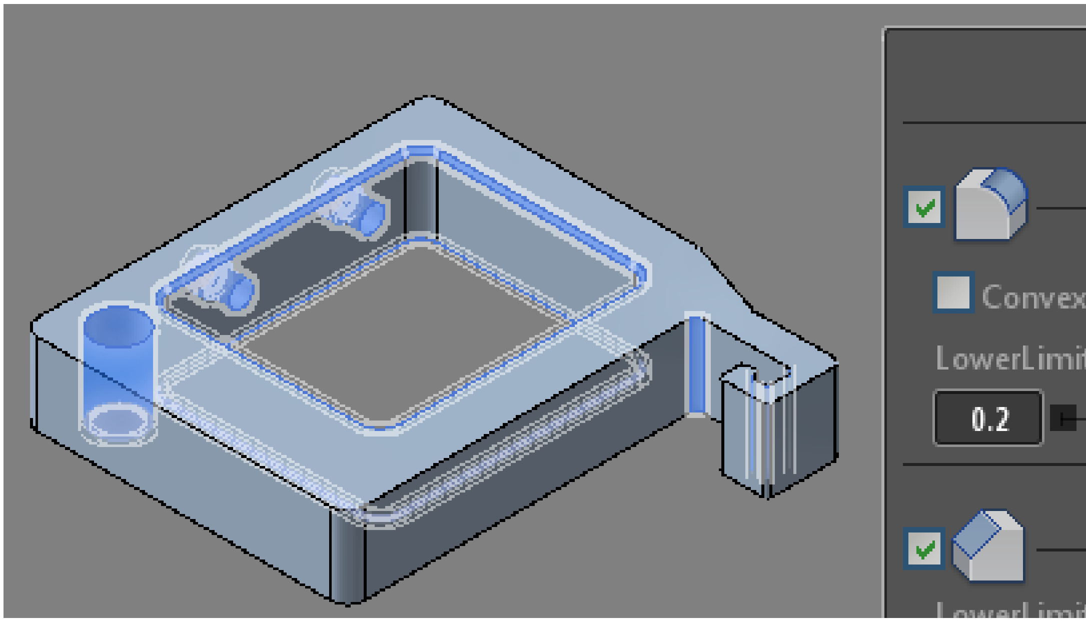
## Select Features with Simplify

The Simplify command has two sets of functionality. It provides filters for selecting features by type and size, and it has options for working with a selection set.

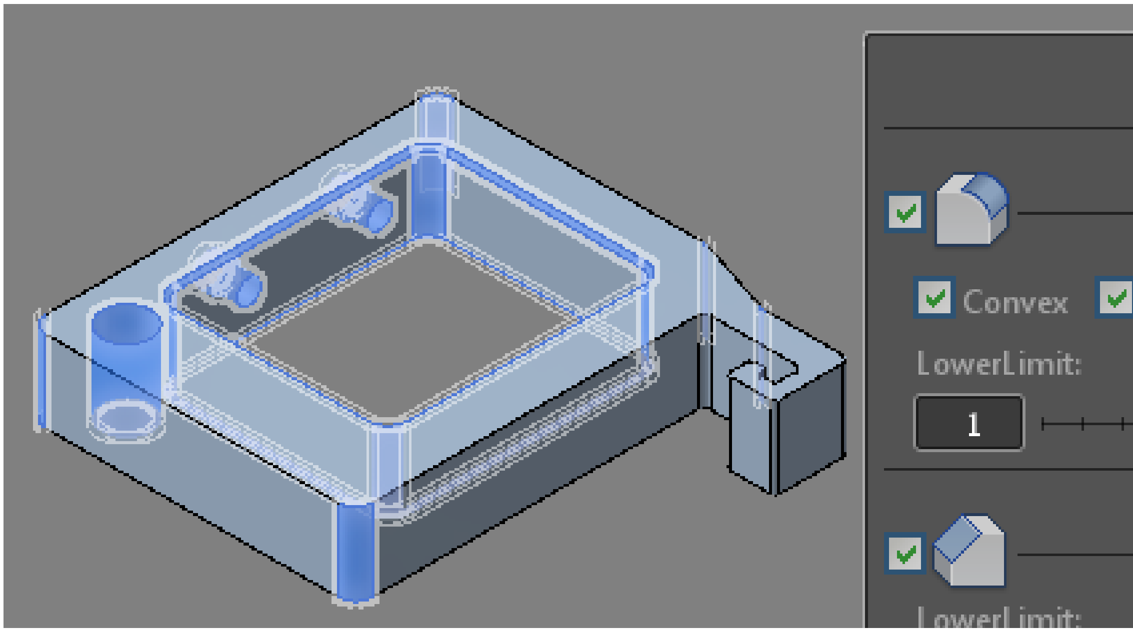
The dialog only displays the filter controls for feature types in the model. For example, if a model only has fillets and holes, the chamfer filter controls do not display. If you uncheck a feature type, the filter options are hidden.

To reposition the dialog, click and drag anywhere on the dialog.

1 Open the Simplify Model.dwg model.
2 From the Manage panel, click Simplify. The Simplify dialog displays the fillet, chamfer, and hole features in the model. The selected features are highlighted in the model.
3 Each feature type has a range of sizes. You can reduce the range by dragging the black or red square on the slider. The black square is the minimum size and the red square is the maximum size. You can also enter a value directly in the field. The highlighting updates when you change the range.
4 Fillet features can be concave or convex. Deselect the convex fillets. The outer fillets are removed from the select set.
5 Use the slider to remove large radius fillets from the select set. Drag the red square to the left. The large interior fillets are removed. Continue dragging the slider until the fillet between the hook and the main body is removed. Click in the Upper Limit field, enter `1.0`, and press `Enter` to restore that fillet.

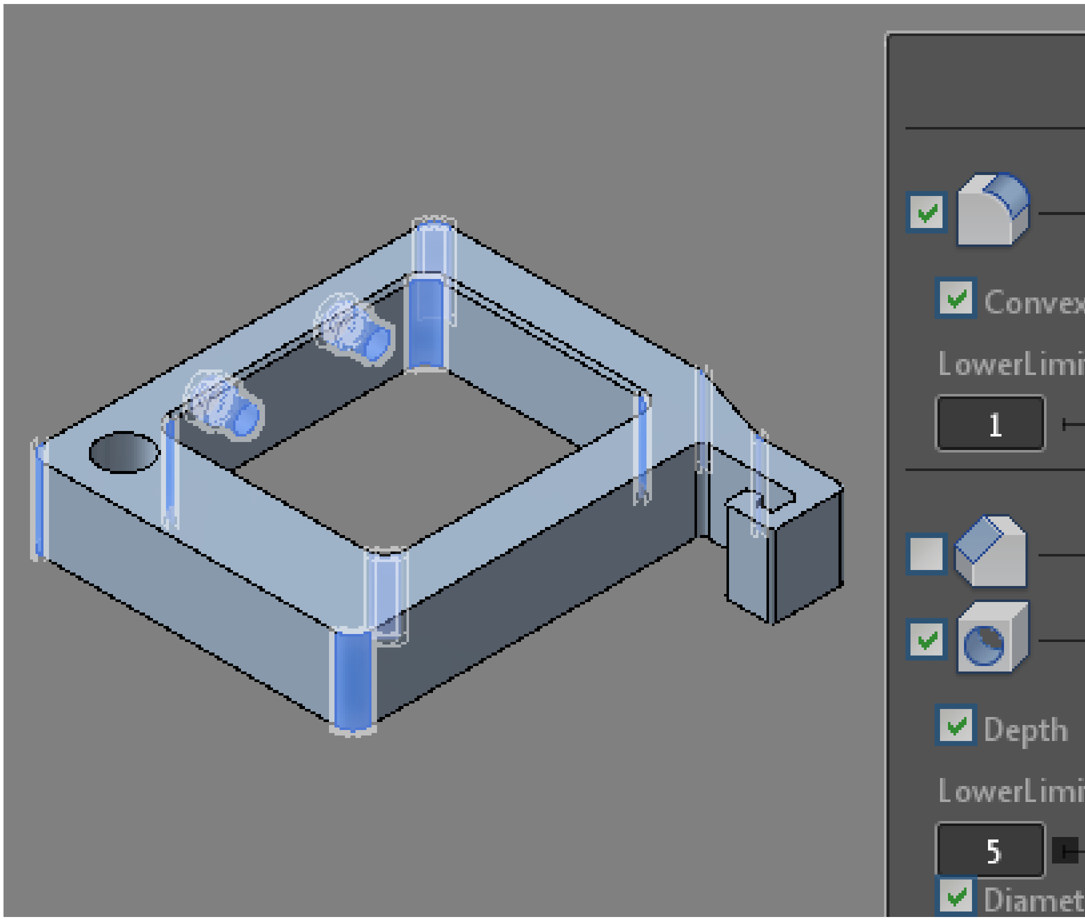


6 We have the parameters that define the small fillets around the hook. To keep those fillets, invert the settings to limit the select set to the fillets on the rest of the model. Click Convex to include all of the fillets. Changing the fillet type option also resets the size limits. Click in the Lower Limit field, type `1.0`, and press `Enter`. The small fillets are removed from the selection set.




7 Holes have four filters: hole depth, hole diameter, counterbore depth, and counterbore diameter. Drag the Upper Limit slider on the hole diameter to remove the large hole from the selection set.

8 Click the check mark at the upper right-hand corner of the dialog. The selected features are deleted and the model is healed.




The next section of the tutorial uses a new model to explore working with sets of features. You can close this model or save it with a different name.

## Work with Selections in Simplify

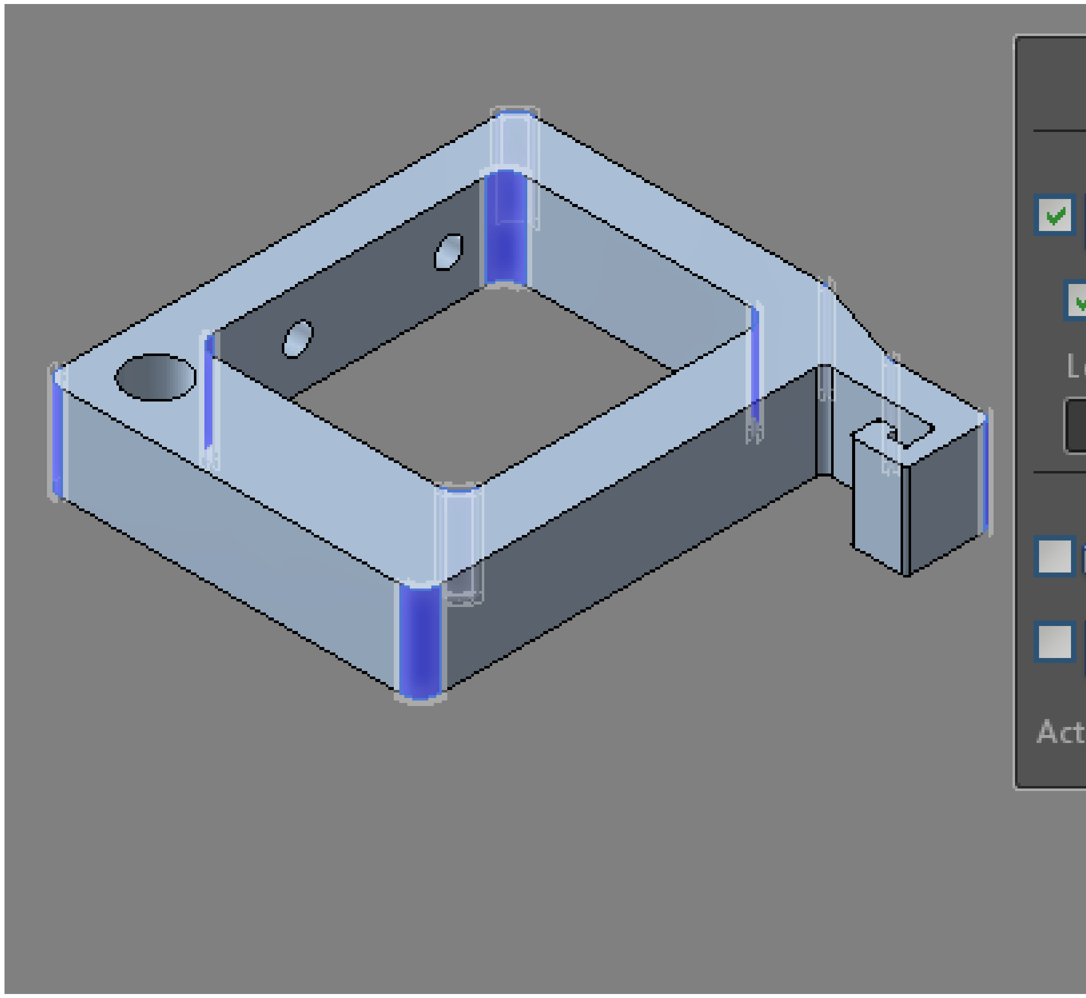
In the previous section, we explored how to use Simplify to filter feature by type and size. Analysis frequently requires different versions of the model. The Simplify command can create select sets with different collections of features. You can use the sets to modify or delete the features.

Simplify has three Actions for selection sets:

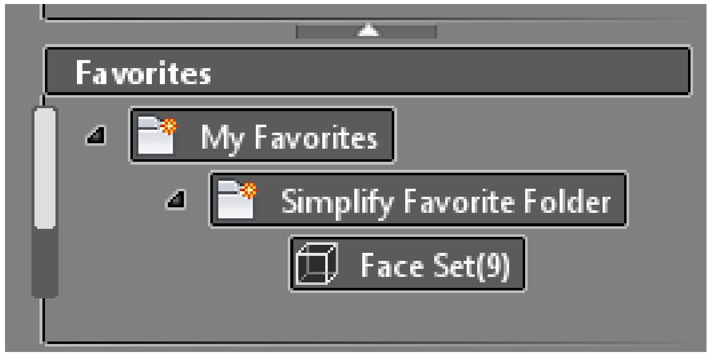
- **Delete** is the default setting. The geometry is removed and the model is healed.
- **Select** keeps the selection set when the dialog is closed. You can use Shift-select or Ctrl-select to remove individual features from the set.
- **Create Favorite** adds a Simplify Favorites Folder to the Favorites section of the browser for the selection sets. You can quickly create multiple sets of features.

1. Open the Simplify Model.dwg file. This file is also used for the previous section of the tutorial, so save the file as Simplify Model with feature sets.dwg.
2. Start Simplify and uncheck the fillet and hole filters. Press `Enter` to delete the chamfers.
3. Start Simplify and uncheck the Hole filter. Change the Lower Limit for the fillets to `1.0` to remove the large fillets from the select set.

4 Change the Action to Create Favorite and Press `Enter`.



5 The Favorites pane is at the bottom of the browser. By default, the height is set to the minimum. Drag the arrow to increase the height. Expand the My Favorites, and then expand the Simplify Favorites Folder. The fillet selection set is named Face Set(9). Change the name to `Small Fillets`.

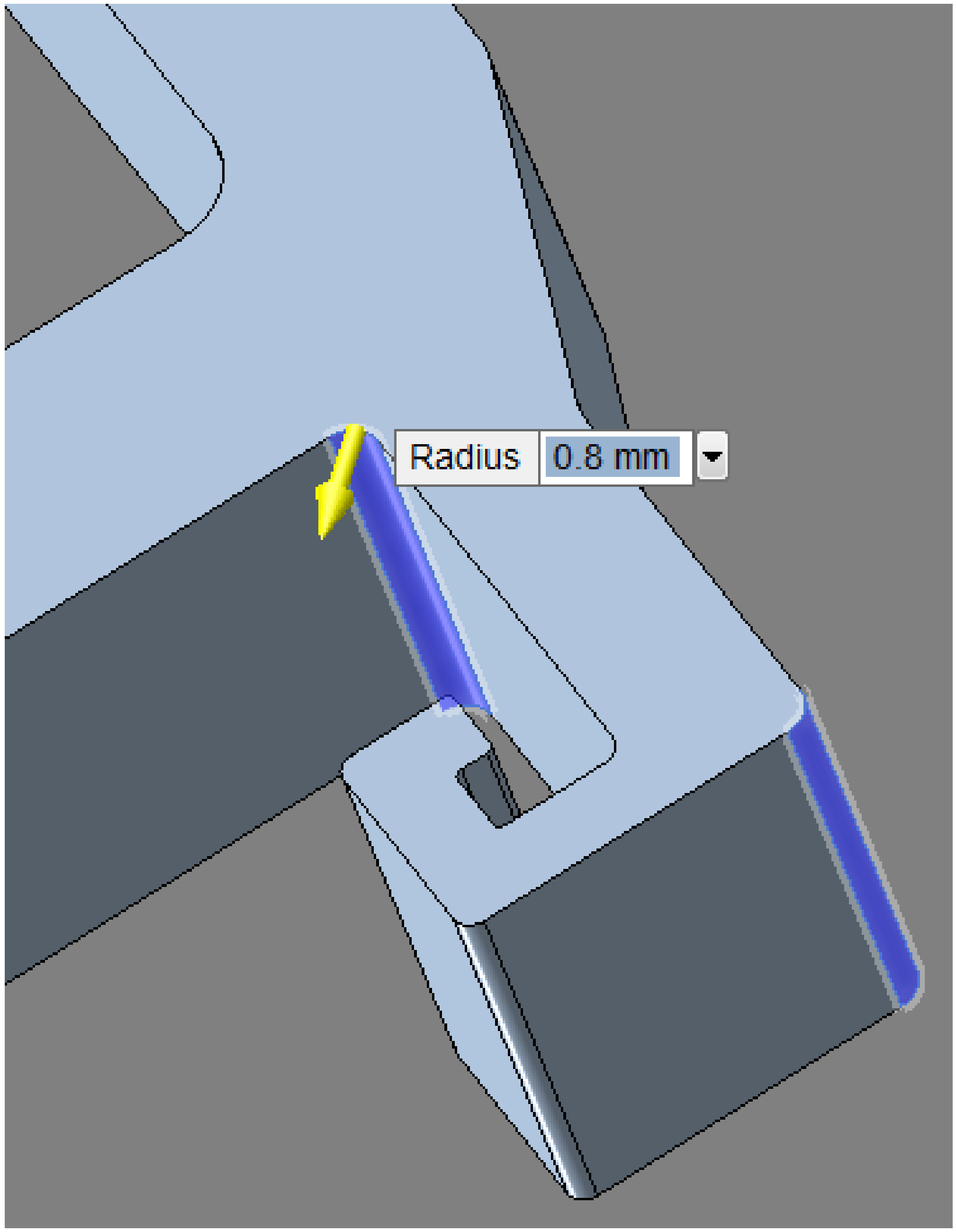



6 Start Simplify and uncheck the Hole filter. Change the Upper Limit for the fillets to `1.0` To remove the small fillets from the select set. Set the Action to Create Favorite, if necessary, and press `Enter`.

7 A second Simplify Favorite Folder is created in the Favorites browser. Change the name of the set to `Large Fillets`. Drag the Large Fillets set up to the first Simplify Favorites Folder. Right-click on the second Simplify Favorites Folder and select Delete.

8 Start Simplify and uncheck the Hole filter. Drag the Fillet sliders so the Lower Limit is greater than 0.5 and the Upper Limit is less than 2.0. Change the Action setting to Select and press `Enter`. Two fillets are selected in the graphics window.

9 Click Press/Pull on the Solid panel. A yellow arrow displays on one of the fillets. The yellow arrow is a manipulator. Dragging a manipulator changes the size of a feature.


Radius
0.8 mm

10 Click on the arrow and drag the manipulator. An error displays if the fillet size becomes too large or too small. Enter 1.2 in the value edit field. Click OK in the command ribbon, and then click Done.

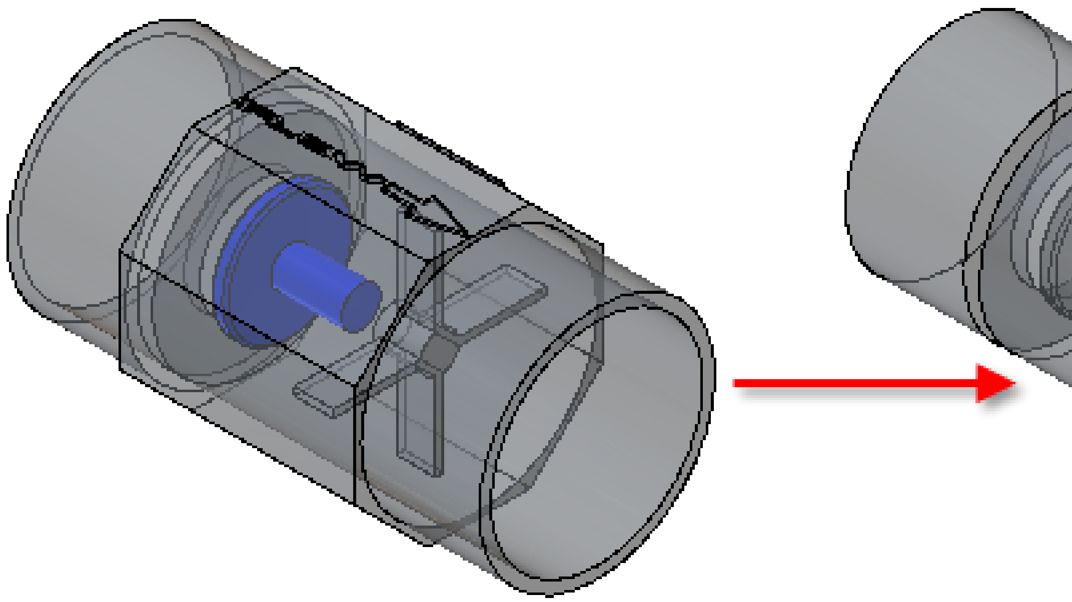
11 Create a new select set with just the fillets at the end of the hook. Click on the Small Fillets set in the Simplify Favorites Folder. Press Shift and click on the fillets away from the end of the hook. There are four fillets remaining in the select set.

12 Right-click in Favorites browser and select New Favorite Folder. A My Favorites is created for the select set is created for the four fillets. Change the name of the set to `Hook Fillets`. Drag the Hook Fillets set up to the Simplify Favorites Folder. Right-click on the second My Favorites folder and select Delete.

The Simplify dialog provides a quick way to delete unnecessary features and create groups of features. You can use the groups to create alternative versions of the model for analysis.

# Creating models for CFD Analysis

## About this tutorial



Learn how to prepare a model for CFD analysis by creating an internal or external volume.

| | |
|---|---|
| **Category** | **CFD Analysts** |
| **Time Required** | 30 minutes |
| **Tutorial Files Used** | Axial Check Valve.sat, Car.sat |

The Fluid Volume command creates internal and external solid volumes. The internal volume is a solid part created from the inside of a model. The external volume is a solid part that encloses the model. An external volume can be a box, cylinder, or hemisphere.

The Fluid Volume command creates a new component, and does not modify the original model. You can create multiple versions of the solid volume for design studies. You can save the solid volumes with the original model to simplify management, and you can also save each component as a separate file.

**Objectives**

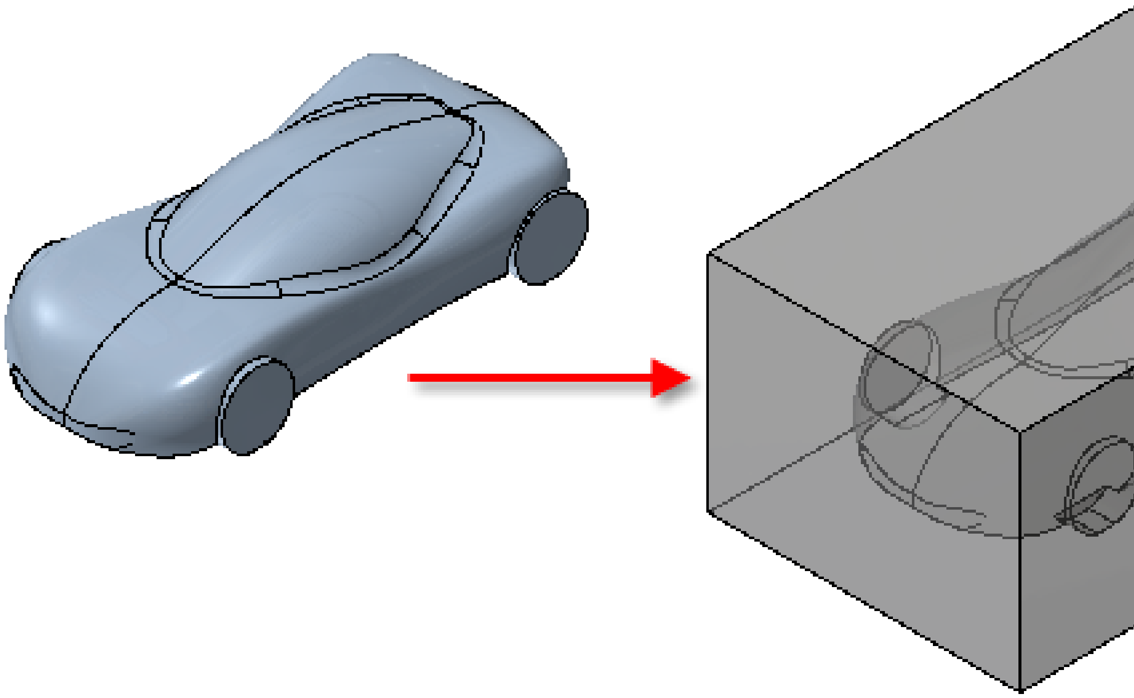
- Learn how to cover openings in models before creating an internal volume.
- Learn how to create box, cylindrical, and hemispherical external volumes.
- Learn how to change the size and orientation of an external volume.

### Navigation Tips

- Use Next or Previous at the bottom-left to advance to the next page or return to the previous one.

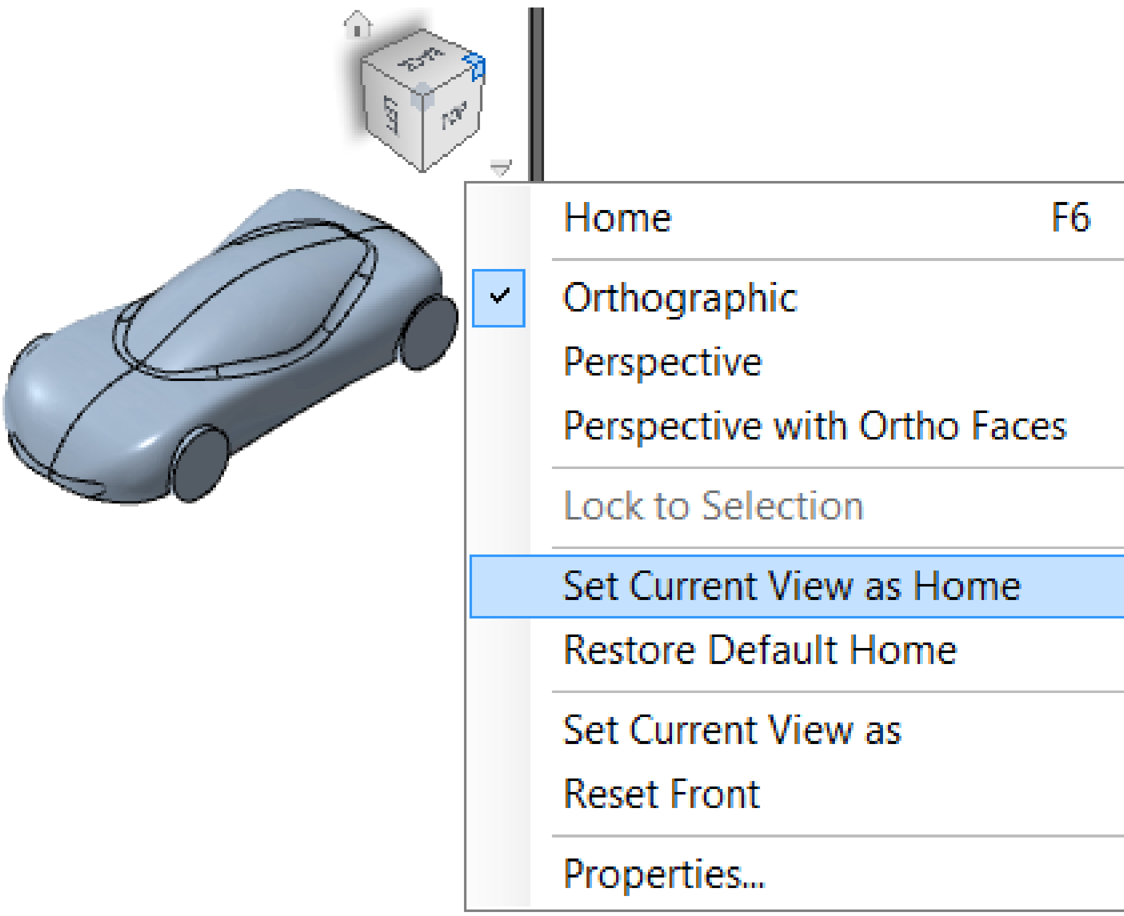
## Create External Fluid Volumes

In this section of the tutorial, we use the Fluid Volume command to create a volume that encloses the model of a car. The resulting solid is a new component in the document.



The fluid volume command can create three shapes- a box, a cylinder, or a hemisphere. The box is the most commonly used shape for CFD simulations as it most closely resembles a wind tunnel. The cylinder can also be used to simulate a wind tunnel. The hemisphere is useful for solar heating analyses.

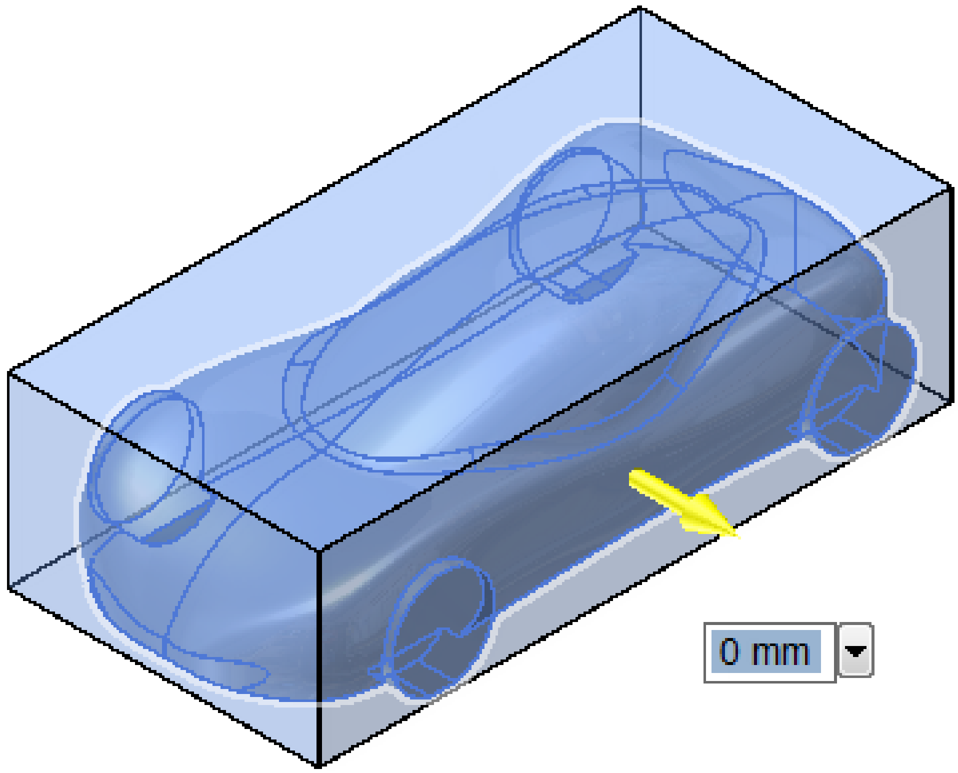
1 Open the Car.sat tutorial model.
2 The isometric view of the model shows the bottom of the car. Click Top on the View Cube, then click the upper left-hand corner of the View Cube. This isometric view shows the top, back, and left sides of the car. Click the drop-down arrow under the View Cube and select Set Current View as Home> Fit to View. The model now returns to this isometric view when you click the Home button.




3 On the Home tab>Simulation panel, start the Fluid Volume command.

4 Set the Volume to External and the Shape to Box.

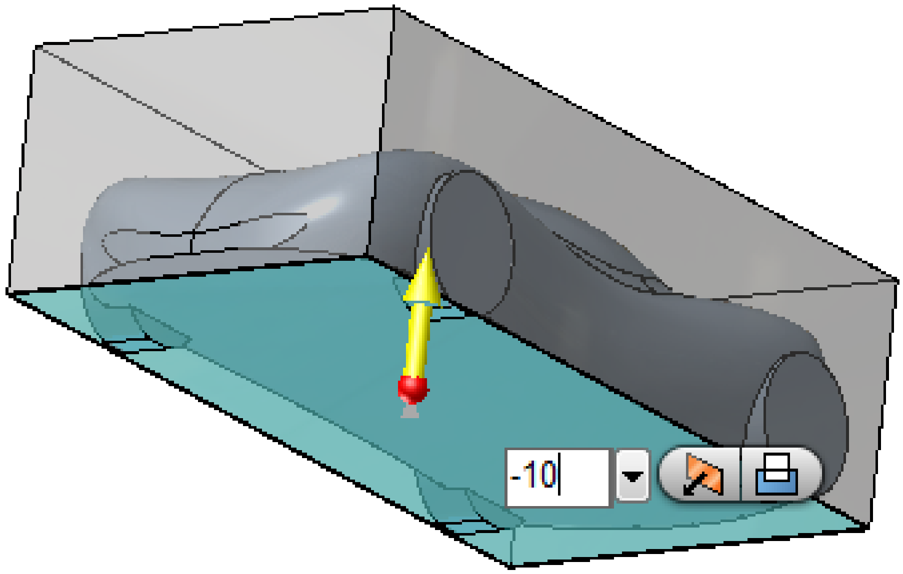
5 Click on the car in the graphics window. The box preview displays with an arrow manipulator for the offset. Drag the manipulator to change the offset- the size changes equally in all directions. We need different offsets in each direction for CFD analysis, so drag the manipulator back to 0.0 and click OK to create the box.



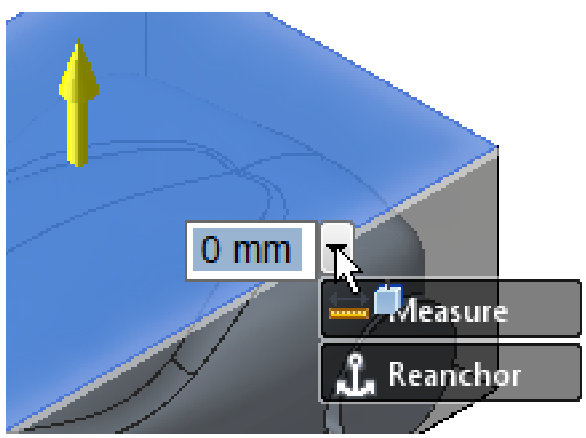

6 The external volume is created as a new solid component. To simplify viewing the cavity and to prevent edits to the original model, turn off the visibility for the car solid. In the browser, click the light bulb next to Solid.

7 Click Front on the View Cube. The bottom of the box touches the bottom of the tires. The box must be slightly above the tires to simplify meshing and analysis.

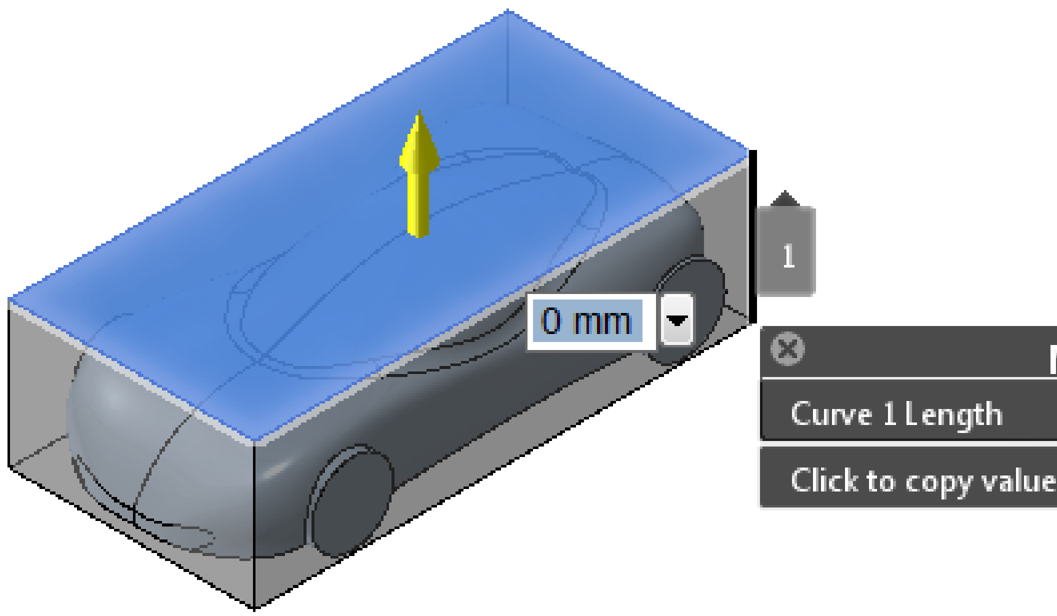
8 Drag a corner of the View Cube to rotate the model so the bottom of the box is visible. Start Press/Pull from the Solid panel of the ribbon, and click on the bottom of the box. Enter -10 mm and click OK to change the size.




**9** Click on the house above the View Cube to return to the Home view. For this tutorial, the top requires an offset at least 50% greater than the model height. The Press/Pull command stays active, so click on the top of the box. Pause the cursor over the drop-down arrow next to the value entry field and select Measure.




Click on a vertical edge of the box- the Measure dialog displays with the length.




Click Curve 1 Length of the Measure dialog to enter it into the field. The offset is now equal to the original height. Press the End key and enter /2 after the value in the edit field. Press enter to accept the value.

**10** Repeat this process to offset the front, back, and sides of the box. Click Done to exit the Push/Pull command.

**NOTE** The minimum offsets for an incompressible external flow simulation are typically five chord lengths upstream and 10 downstream. For this exercise, one chord length in each direction is acceptable.

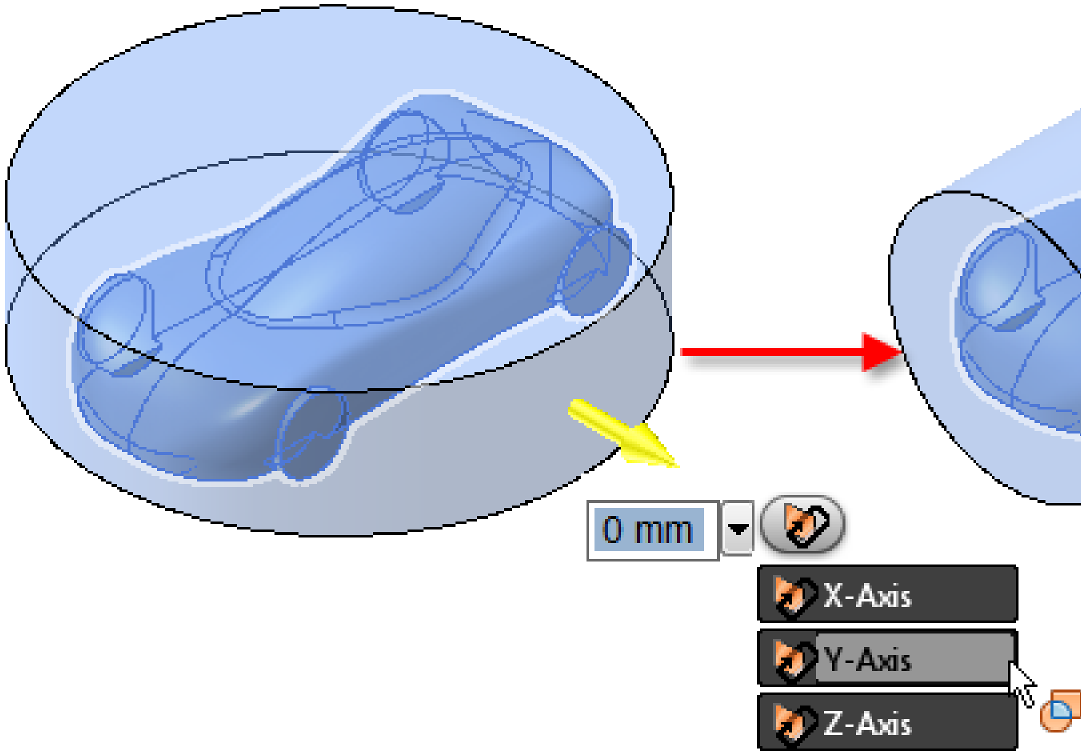
**11** Next, we create a cylindrical external volume. Click the light bulbs in the browser to turn on the visibility for the car solid and turn off the visibility for the box component.

**12** Start Fluid Volume from the Solid panel of the ribbon.

**13** Set the Volume to External if necessary, and set the shape to Cylinder.

**14** Select the car model in the graphics window. The cylinder previews around the car, and the cylinder axis is parallel to the Z-axis.

**15** Pause the cursor over the Axis Selection glyph next to the edit box and select the Y-axis. The cylinder is now aligned with the length of the car.

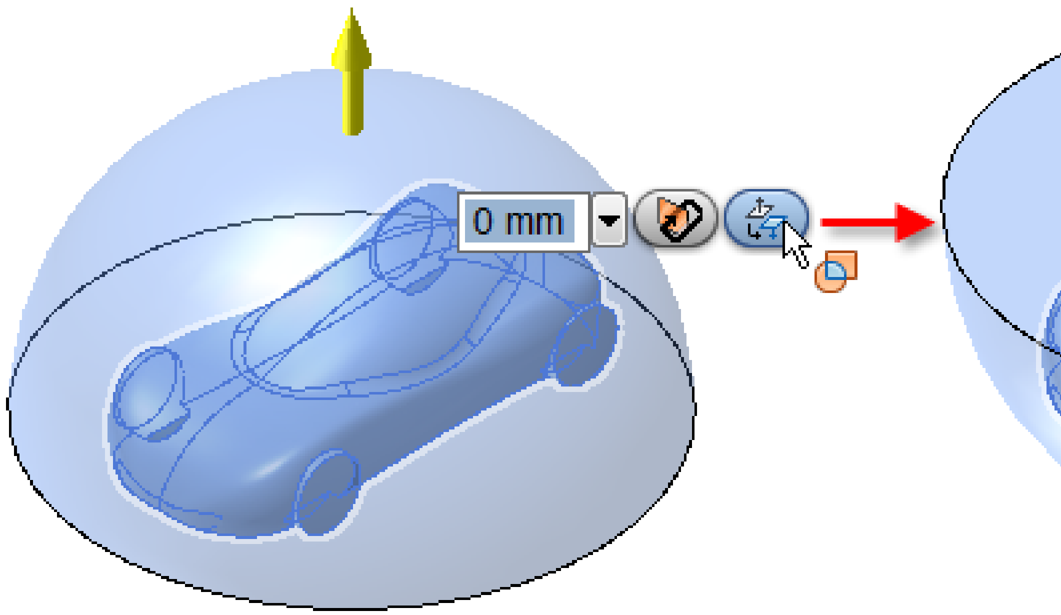



16 Drag the Manipulator to change the offset to 2000 mm.

17 Click OK, and click the light bulb next to the new component to turn off the visibility of the cylindrical volume.

18 Next, we create a hemispherical exterior volume. Right-click in the graphics window and select Repeat from the marking menu. The Fluid Volume command starts again. Set the Shape to Hemisphere and select the car.

19 Select Z-axis from the selection glyph and click the Flip glyph to reverse the direction of the hemisphere. Click Flip again so the hemisphere radius is in the positive Z-direction. Drag the manipulator to increase the radius by 1000 mm.



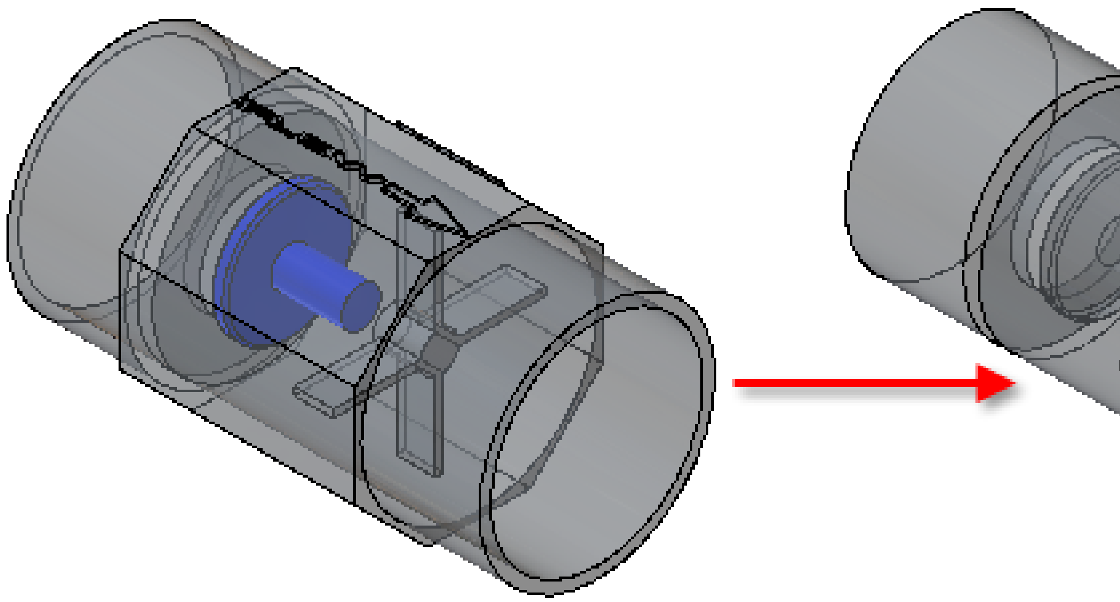

**20** Click OK to create the component.

**21** Right-click on each component and select Save As to create a file for that external volume. If you want to keep the external volume components in the car model, click Save from the Quick Access Toolbar.

## Create Internal Fluid Volumes

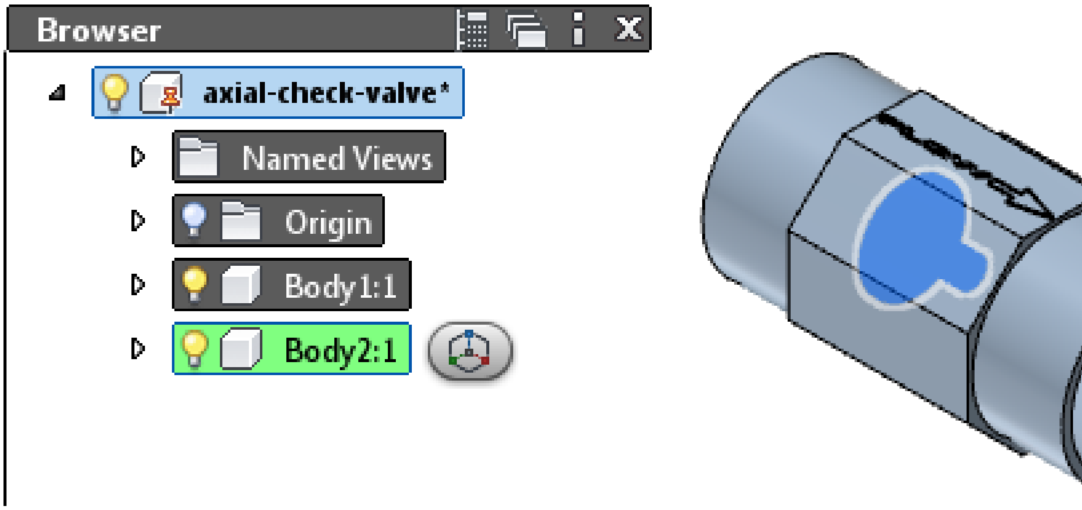
In this section of the tutorial, we use the Fluid Volume command to create an internal fluid volume within an axial check valve. The Fluid Volume command converts a fully enclosed, internal volume into a solid model for meshing and simulation. The resulting part is a new component in the document.

We create a section view to show the inside of the model, constrain the position of the components, create surfaces to close the inlet and outlet openings, and create the internal volume.



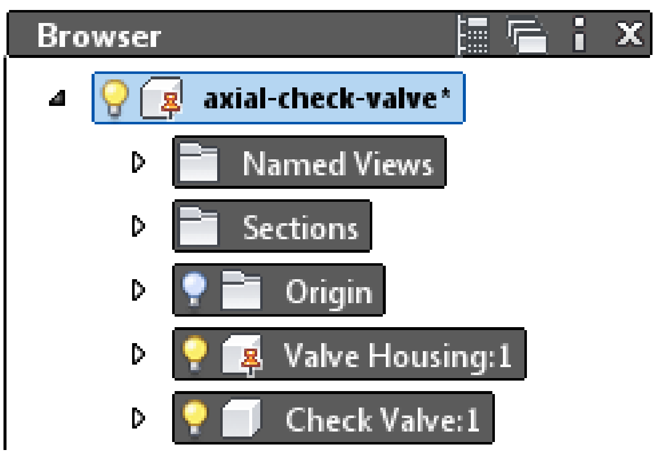
1 Open the tutorial file Axial Check Valve.sat.

2 The model browser shows two components. Click on the Body2, which is the check valve, and the highlight is visible through the outer body of the check valve.



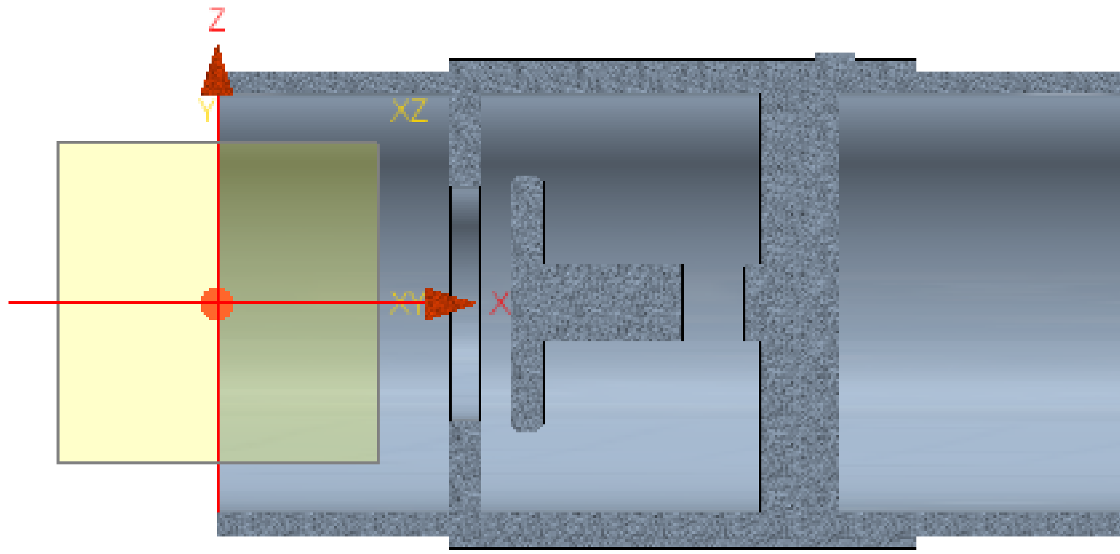

3 We can change the browser names to make them more descriptive. Rename Body1 to Valve Housing, and Body2 to Check Valve.

4 The parts of an imported model start in the correct position, but they can be moved. The housing never moves, but the check valve has open and closed positions. Once we fix the location of the valve housing, and we can constrain the check valve to the housing. Right-click on the Valve Housing in the browser and select Grounded. The icon has a pushpin to indicate that it is fixed in position.




5 We can create a section view to access the check valve. Click on the light bulb next to the origin folder to display the origin geometry. In the upper

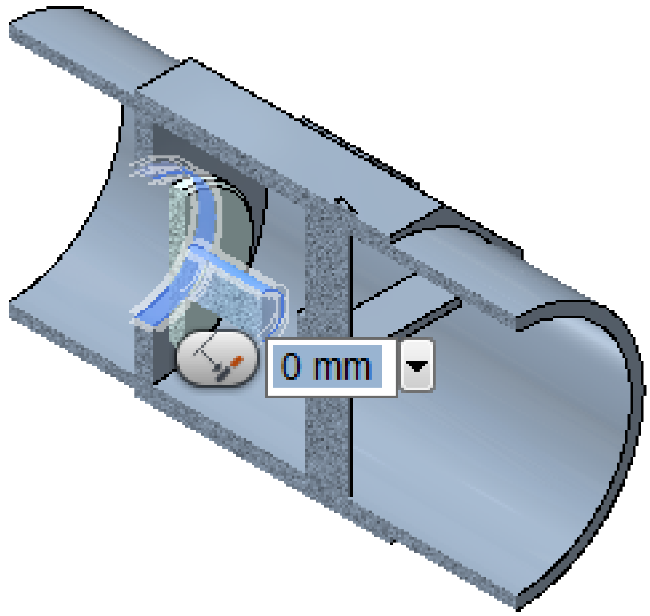
right corner of the graphics window, click on the Front face of the View Cube. Switch the ribbon to the View tab, and select Section from the Visual Styles panel. Click on the visible origin plane, and the front half of the model is removed from the display.




6 Press Enter to create the section view. A Sections folder is added to the browser. Expand the folder, and click the light bulb to turn off the section view. Turn the section view on again so we can create assembly constraints between the parts.

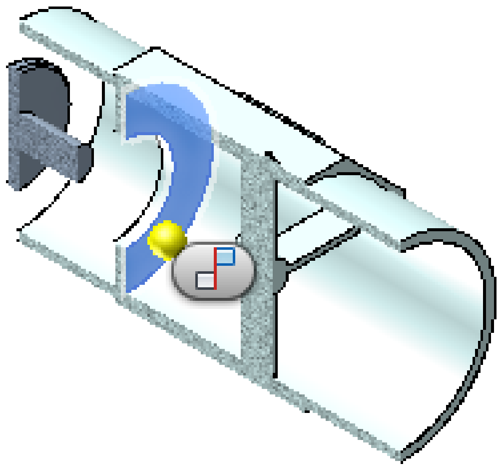
7 Click on the House icon above the view cube to return to the isometric view. Click on the light bulb next to the origin folder to turn off the origin geometry.

8 Switch the ribbon to the Home tab, and select Assemble from the Constrain & Dimension panel. Select the shaft of the check valve, and then select a cylindrical surface in the housing. A glyph with the axial constraint symbol and a value entry field display. Press enter to create the constraint.




9 Double-click on the check valve in the graphics window to select the component, and drag it past the hole in the housing. Since we created an axial constraint, the check valve can only move along the housing.

10 Start the Assemble command again. Click on the face of the housing web with the hole. Rotate the model and click on the mating face of the check valve. The check valve moves against the face. The value entry field is active, so type 5 mm and press Enter to accept the value. Press Enter again to create the constraint.

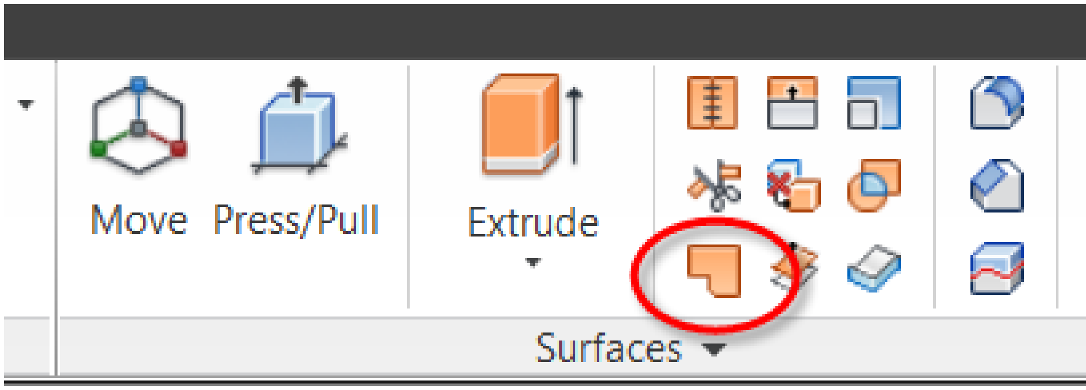



**11** Click Front on the View Cube, and note the gap between the parts. Expand the Assembly Constraints folder, and double-click the second constraint. The value entry field displays so you can change the distance between the parts. Press Esc to cancel the edit. You can quickly change the constraint value and create internal fluid volumes for design studies in CFD.

**12** Turn off the section in the browser and return to the isometric view.

**13** The Fluid Volume command requires a fully enclosed model, so we have to create surfaces to cover the holes in housing. The command automatically finds the intersections between faces, so the edges do not have to match. For this tutorial, we use different methods for the inlet and outlet ports. The boundary patch exactly matches the opening, while the work plane extends past the model.

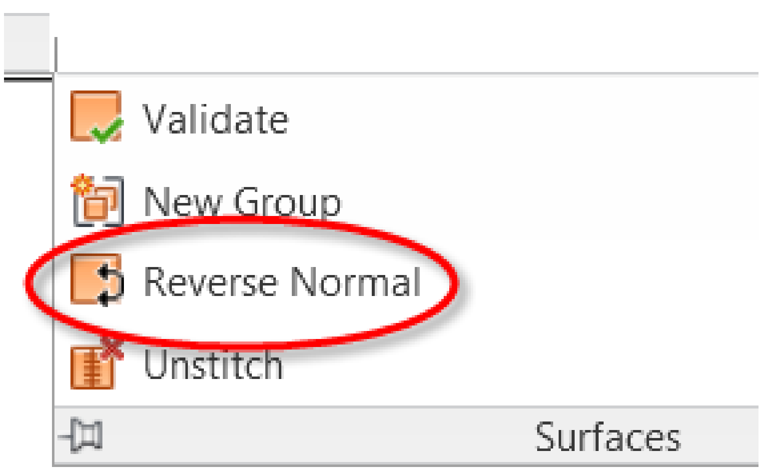
**14** Switch to the Home tab of the ribbon. In the Modeling panel, click the drop-down arrow under Solid and select Surface. The ribbon updates to display the surface commands. Click the Boundary Patch icon on the Surfaces panel.

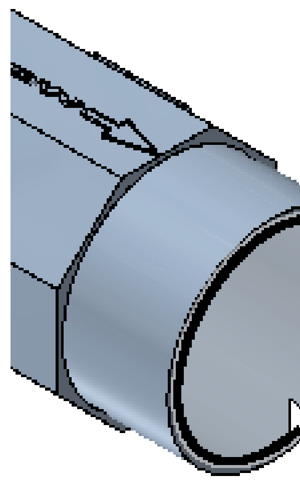



**15** Click on the inner circular edge of the port and press Enter. A circular surface is created that exactly matches the hole.

**NOTE** Surfaces have a positive and negative side. Fusion uses a translucent grey color for one side of a surface and yellow for the other.

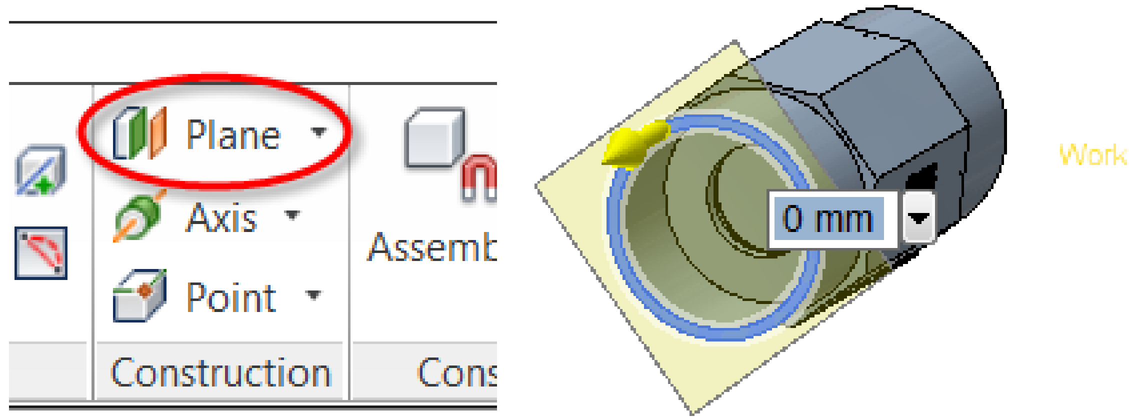
For this exercise, the yellow side of the surface faces outward to improve visibility. If the boundary patch is grey, expand the Surfaces panel and select Reverse Normal. Click on the surface and press Enter.



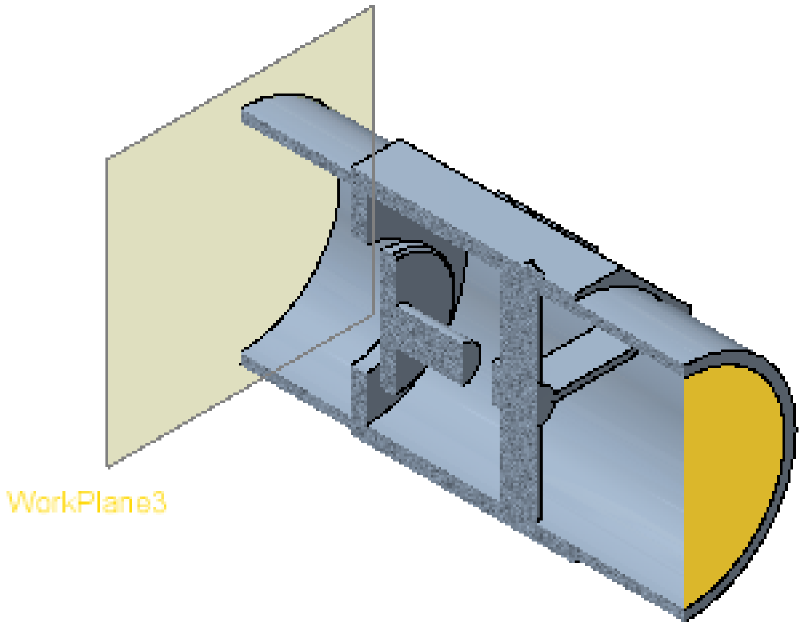



**16** Rotate the view so the other end of the model is visible.

**17** Click Plane By Offset in the construction panel, and select the end of the housing. The work plane previews with the offset set to 0.0 mm. Press Enter to create the work plane on the end of the housing.

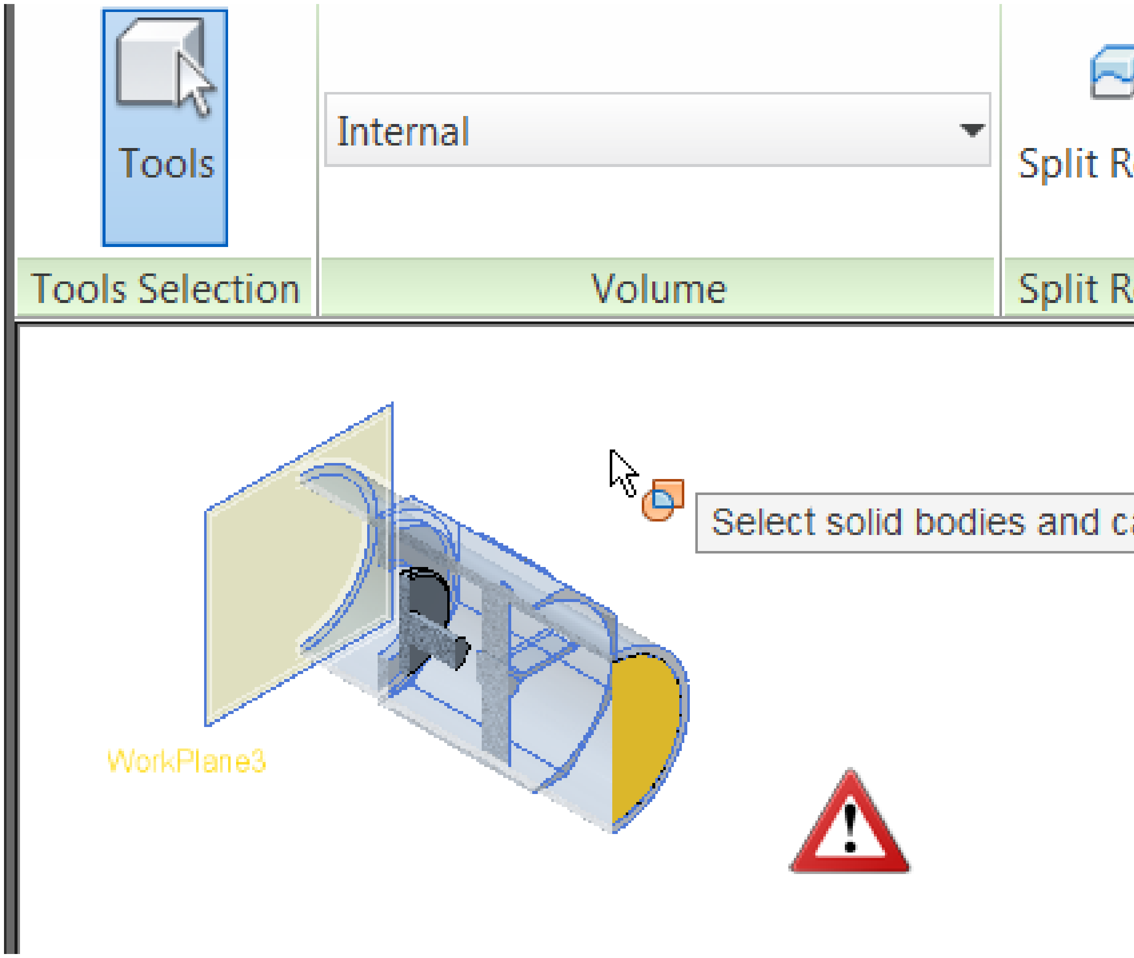



**18** Return the model to isometric view, and turn on the section from the browser. The model internals and the surfaces are now visible.

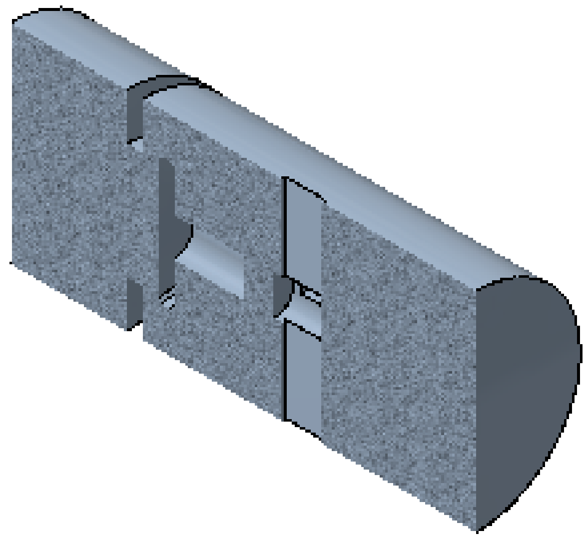



19 Save the model as Axial Check Valve- Ready for Fluid Volume.dwg.

20 On the Modeling panel of the ribbon, expand the Solids panel and select Fluid Volume. Set the Volume option to Internal. Select the Valve Housing, the Check Valve, and both surfaces. An error symbol displays until the selections form an enclosed volume. Click OK on the command ribbon to create the internal volume.




21 The internal volume is created as a separate component. In the browser, click the light bulb next to the Surfaces folder, the Valve Housing, and the Check Valve. Inspect the internal volume and confirm the check valve was subtracted from the solid.



22 Expand the internal volume component, right-click on the Solid node, and select Copy.

23 Create a new document. Right-click in the graphics window and select Paste, then click OK. Save this document as Axial Check Valve- Internal Volume- 5 mm.dwg, and close it.

24 Now that the model preparation is complete, you can change the check valve position and create new internal volume modules. The command creates a new component each time, so you can quickly create multiple versions.

# System Requirements

# 12

## Operating System

- Microsoft® Windows® 7 (32-bit or 64-bit)
- Microsoft® Windows® Vista (32-bit or 64-bit), SP2
- Microsoft®Windows® XP (32-bit), SP3
- Microsoft®Windows® XP (64-bit), SP2

**NOTE** Microsoft .Net Framework 4.0 is installed prior to the installation of Fusion if .Net is not already present on the system.

## Hardware

**Minimum requirements:**

- Intel Pentium 4, AMD Athlon 64 or AMD Opteron or later with 2.0GHz or faster, or comparable
- 1.0+ GB RAM
- 2.0+ GB of free disk space (for installation)
- Microsoft Direct 3D 9 or 10 graphics support with 64+ MB
- 1,024 x 768 Screen Resolution

**Recommended Specifications:**

These specifications are over and above the minimum requirements.

- 2 GB RAM
- 1,280 x 1,024 Screen Resolution

# Graphics Processing Unit (GPU) Requirements

**Minimum specifications required:**

- Direct 3D 9-Compatible Graphics Card with Pixel Shader 2.0 support. Supports most commonly used Inventor Fusion effects. Does not support the Ambient Occlusion Effect and does not support Realistic Materials.
  - Technical note: Supports Autodesk Graphics Feature level 2_0

**Recommended specifications:**

- Direct 3D 9-Compatible Graphics Card with Pixel Shader 3.0 support or Direct 3D 10-Capable graphics card. Supports all Inventor Fusion Effects, including Ambient Occlusion and Realistic Materials.
  - Technical note: Supports Autodesk Graphics Feature Level 3_0

### Test graphics card capabilities

The Inventor Fusion Install directory contains a Graphics Capabilities Application, AdskDiagOutput.exe. The location is *C:\Program Files\Autodesk\Inventor Fusion [version]*, or the equivalent on your machine. Run this program to produce a text file, *out.txt*, in the install directory.

---

**NOTE** For Windows Vista and Windows 7, right-click and select Run As Administrator.

---

Open the output file to view the system and graphics information.

The line called Feature Level tells you what level of Graphics support your card offers Inventor Fusion.

- If the value is less than 2_0, Inventor Fusion does not work well or at all on your machine. The Inventor Fusion development team cannot address some graphics issue.
- If the value is 2_0, Inventor Fusion works on your machine, but has only basic graphics capabilities. For example, Realistic Appearances are not available, and the default rendering is used.
- If the value is 3_0 or higher, your machine supports all effects used by Inventor Fusion.

**NOTE** For issues with a graphics card that meets the requirements, please email the *out.txt* diagnostics file and a description of your problem to *labs.iv.fusion@autodesk.com*. If we need more information, such as screen shots or sample files, a member of the Inventor Fusion Development Team contacts you.

# Legal Notices 13

Autodesk Inventor Fusion 2013



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