

Contents

Chapter 1	Modeling with Inventor Fusion Technology	1
	Create Models with Inventor Fusion Technology	1
	Sketch Environment	4
	Plan and create sketches	5
	Work with Sketches	7
	Sketch Grids	10
	Sketch Constraints	11
	Sketch Dimensions	13
	Sketch Profiles	14
	Solid Modeling	15
	Surface Modeling	16
	Work with Multiple Components	16
	Create Components	18
	Position and Constrain Components	22
	Joints	23
Chapter 2	Inventor Fusion Technology User Interface	31
	Toolbar	31
	Browser	33
	Tools in the Graphics Window	36
	Heads-Up Display	36
	Marking Menu	38
	Manipulators	42

Selection Tool	45
Triad	46
Navigation commands	48
View Cube	48
ViewCube Overview	48
Reorient Views of Models with the View Cube	49
ViewCube Menu	49
Navigation Bar	50
Error Handling	51
Chapter 3 Inventor Fusion Technology Commands	55
Solid workspace	55
Sketch	55
Create Sketch	56
Line	57
Point	59
Circle Center Radius	60
Circle Two Point	61
Arc Three Point	62
Rectangle Two Point	63
Rectangle Three Point	64
Rectangle Center Corner	65
Ellipse	66
Fillet	66
Trim	70
Extend	72
Offset	74
Project	76
Sketch Dimension	77
Constraints	78
Stop Sketch	80
Grid Off	80
Grid On	81
Solid	81
Extrude	81
Revolve	84
Sweep	88
Fillet	91
Chamfer	93
Hole	95
Shell	97
Draft	98
Split Face	100
Split Body	101
Combine	102
Thicken	104

Find Features	105
Modify	108
Press Pull	109
Tweak Faces	110
Scale Command	111
Move	113
Move Components	114
Delete	115
Pattern	117
Rectangular Pattern	118
Circular Pattern	119
Mirror	121
Construction	122
Work Planes	122
Work Axes	123
Work Points	124
Select	125
Select	125
Selection Filters	126
Surface workspace	126
Sketch	126
Create Sketch	127
Line	128
Arc Three Point	130
Circle Center Radius	131
Ellipse	132
Circle Two Point	133
Sketch Dimension	134
Extend	135
Trim	137
Fillet	139
Grid Off	143
Grid On	143
Offset	143
Project	145
Rectangle Center Corner	146
Rectangle Three Point	147
Rectangle Two Point	148
Constraints	149
Point	151
Stop Sketch	152
Surface	152
Extrude	152
Revolve	156
Sweep	160
Patch	163

Merge	164
Fillet	166
Chamfer	167
Offset	169
Trim Surface	170
Extend	171
Reverse Normal	173
Split Face	174
Split Body	175
Thicken	176
Modify	178
Press Pull	178
Tweak Faces	179
Scale Command	180
Move	182
Move Components	183
Delete Faces	184
Pattern	184
Rectangular Pattern	185
Circular Pattern	186
Mirror	188
Construction	189
Work Planes	189
Work Axes	190
Work Points	191
Select	192
Select	192
Selection Filters	193
Assemble Workspace	193
Sketch	193
Arc Three Point	194
Circle Center Radius	195
Ellipse	196
Circle Two Point	197
Sketch Dimension	198
Create Sketch	199
Line	200
Extend	203
Trim	204
Fillet	206
Grid Off	210
Grid On	210
Offset	210
Project	212
Rectangle Center Corner	213
Rectangle Three Point	214

Rectangle Two Point	215
Constraints	216
Point	218
Stop Sketch	219
Joints	219
Joint Origin	219
Joint	220
Move Joints	222
Modify	223
Press Pull	223
Tweak Faces	224
Scale Command	225
Move	227
Move Components	228
Move Joints	228
Pattern	229
Rectangular Pattern	229
Circular Pattern	231
Mirror	233
Construction	234
Work Planes	234
Work Axes	235
Work Points	236
Place	237
Place	237
Place from Autodesk 360	238
Select	238
Select	238
Selection Filters	238
Options	239
Material	239
Physical Material	240
Visual Material	241
Display	241
Visual Style	242
Camera	246
Ground Reflection	247
Ground Shadow	250
Ambient Occlusion	251
Object Shadow	253
Screen	255
Enter Full Screen	255
Snap	255
Enable or Disable Adaptive Snap	256
Browser commands	256
Components	256

	Bodies	257
	Joints	257
	Features	258
Chapter 4	Import Data	259
Chapter 5	Export Data	261
Chapter 6	Legal Notices	263
	Index	265

Modeling with Inventor Fusion Technology

1

Inventor Fusion Technology is 3D modeling software that showcases intuitive direct manipulation capabilities for unrivaled ease of use. Designers can freely explore complex shapes and forms. Inventor Fusion makes it easy to open and edit 3D models from many sources and incorporate them into your design, enabling rapid design changes without limitations.

Create Models with Inventor Fusion Technology

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.

Overview of modeling in Fusion

A feature can be a solid or a surface. A solid has all of the faces for a shape and defines an enclosed 3D volume. 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.

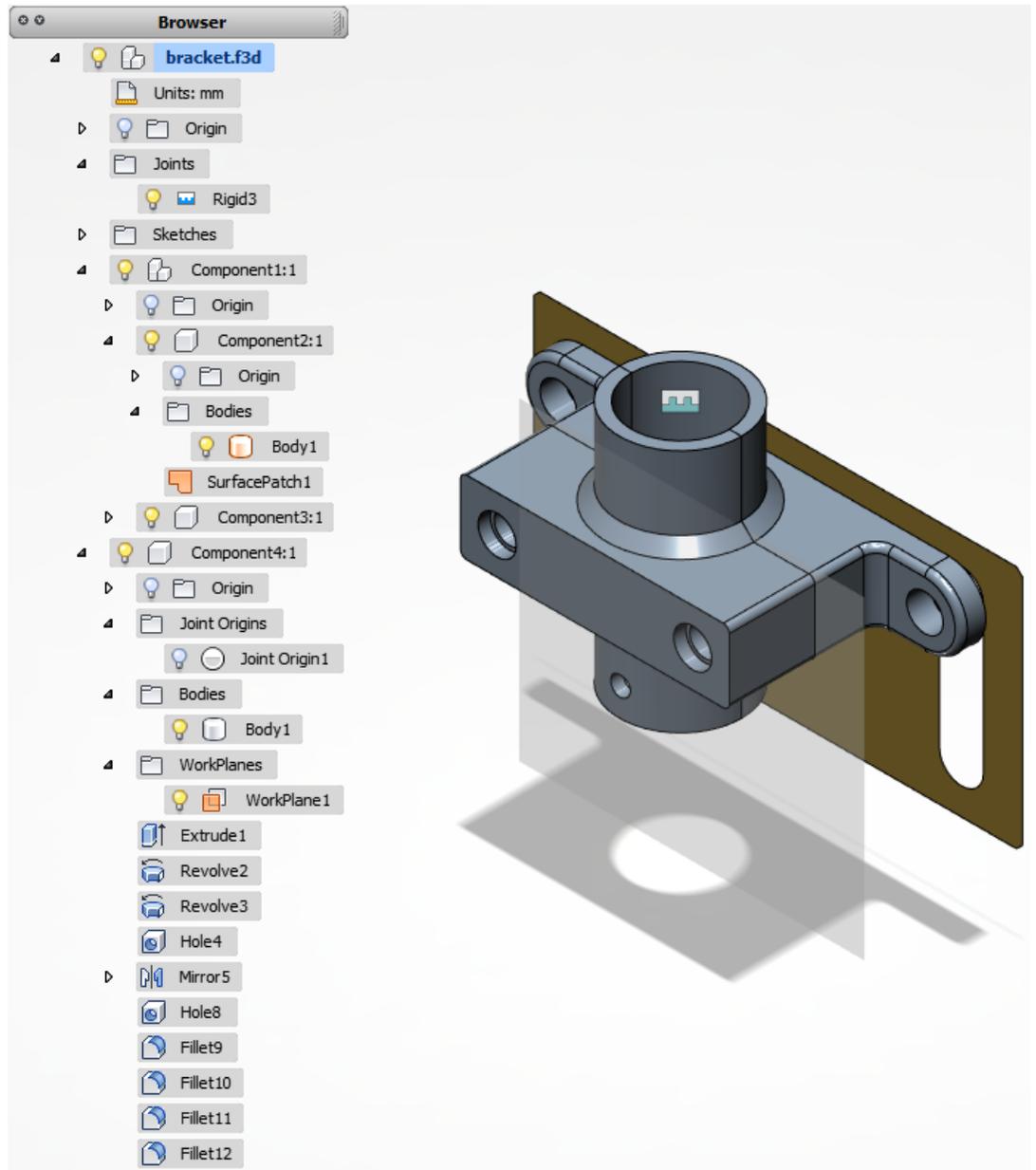
Fusion uses direct modeling instead of the parametric modeling used in Inventor. As a result, the concept of a feature differs in Fusion:

- 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.
- Inventor Fusion sees a feature as a collection of faces.

- 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 Tweak Face or Press 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.

Use features to create solid bodies in Fusion. Some features, such as extrude and revolve, require 2D sketches to create the feature.

Sample design in Fusion



Key elements of the design:

- Component1 is an assembly that contains two parts: Component2 and Component3.
- Component2 contains one surface body.
- The Sketches folders in the browser contains all the sketches for a component.
- Component4 contains a solid body and features. The Feature nodes in the browser represent geometry which was created using a feature command in Fusion.
- Component4 contains a work plane.
- 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.
- The root component contains the Joint node and the Joint Origins are contained in the relevant components.

Feature Command UI

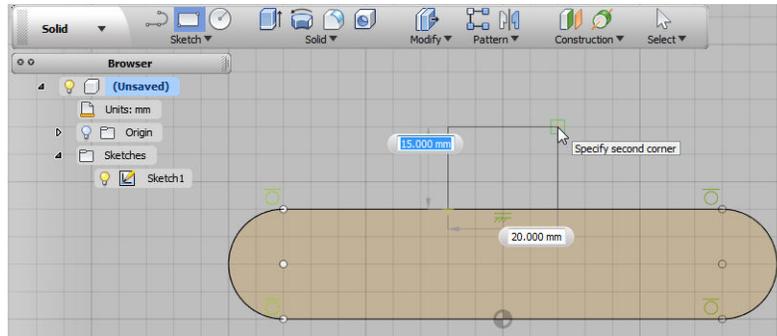
The heads-up display is the primary command interface. The heads-up display for a feature includes value entry fields, manipulators to change the size dynamically, and mini-toolbars for selecting options. Most feature commands have a dialog box in addition to the Heads-Up Display.

Related topics:

- [Sketch Environment](#) (page 4)
- [Solid Modeling](#) (page 15)
- [Surface Modeling](#) (page 16)
- [Work with Multiple Components](#) (page 16)

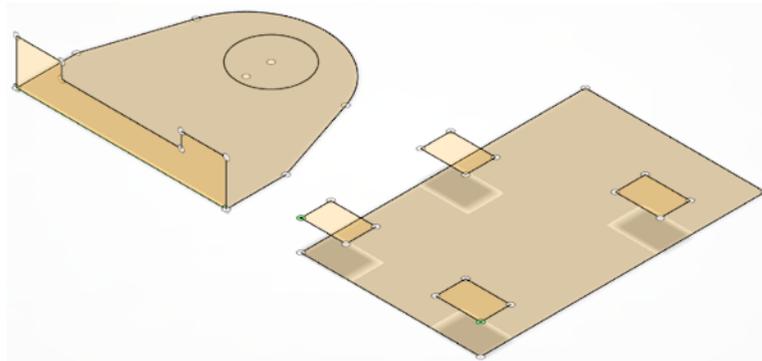
Sketch Environment

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



Plan and create sketches

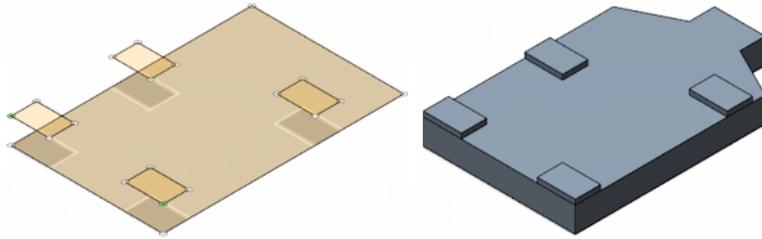
Define your design and create features with sketches. A layout sketch contains the references between important features. Detail sketches have the exact shape and size of features. Inventor Fusion provides several methods for creating sketches.



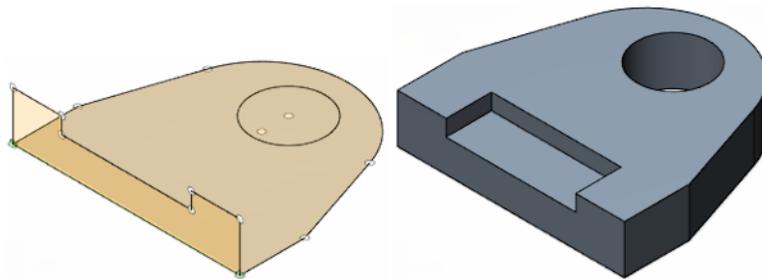
Plan sketches

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.

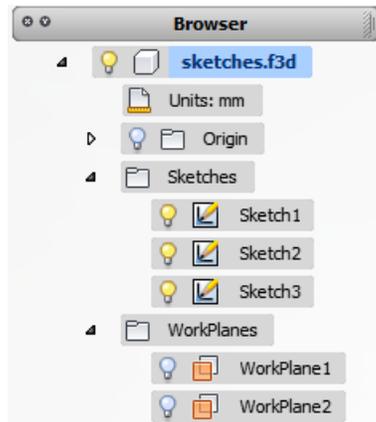
In many cases, a detail sketch is disposable. After you create the feature, you can delete the sketch. Keep a detail sketch if you want to reuse it, or use it for future reference.

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. The active component is indicated with a blue highlight in the browser. To set a component as the active component, move the mouse over the browser node then click the circle that is displayed. In the browser, the new 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 Create 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

Use the Create Sketch command or start any sketch command to create a new sketch. If you start Create Sketch or start any sketch command then select existing sketch geometry or a plane that already has a sketch, that sketch 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.
- Start a sketch command then select a plane that contains the sketch or select sketch geometry.

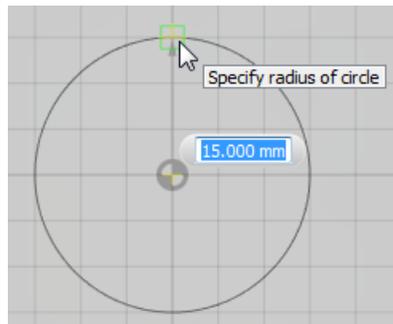
Exit a sketch

To exit a sketch:

- Right-click and select Sketch then select Stop Sketch.
- Select Stop Sketch from the Sketch pull down.
- Start any feature command.

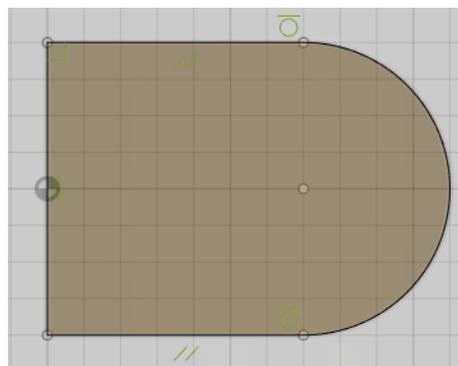
Sketch Grid and Snap Bar

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.

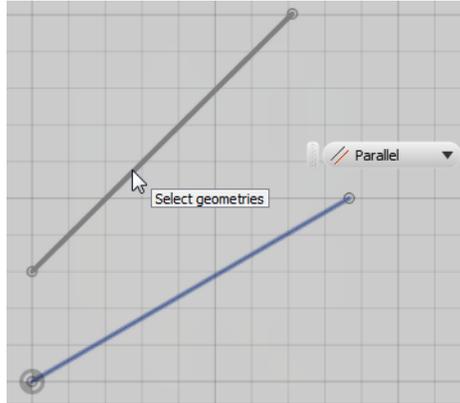


Constraints

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



You can add constraints to existing sketch entities using the Constraints command. Select the constraint type then select the geometry to constrain.



Dimensions

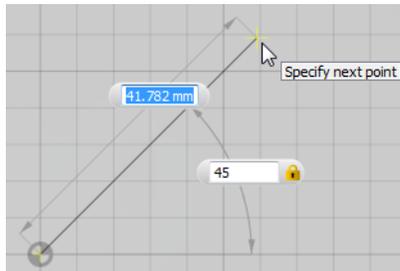
Use the Sketch Dimension command to add dimensions to sketch geometry. Sketch dimensions control the size of the geometry. Double-click a dimension to edit the value.

To display temporary dimensions on an object, select a sketch object when no command is active. These dimensions are for display only and cannot be edited.

Dimensions only display when a sketch is active.

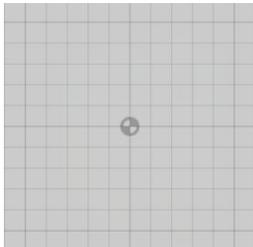
Precise Input

Precise Input provides value entry boxes so you can specify the entity size during creation. 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.

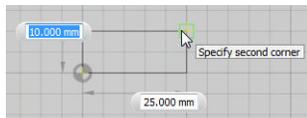


Sketch Grids

When a sketch plane is activated, a sketch grid is drawn that encompasses all the sketch entities on the plane. If there are no entities on the plane, the grid assumes a default size.



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



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

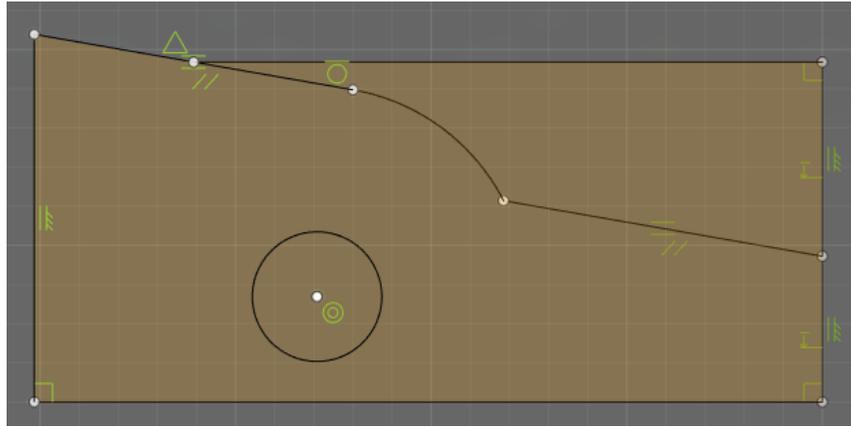


Control the display of the Grid using the Grid On and Grid Off commands in the Sketch drop down.

The grid spacing updates automatically as you zoom in and zoom out.

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.

Add constraints to existing geometry using the Constraints command in the Sketch pull down.

Constraint types

Con- straint	Glyph	Description
Coincident		Point to point or snap Point to entity
Collinear		Manually added only

Constraint	Glyph	Description
Concentric		Circle, arc, or ellipse have the same center point.
Midpoint		Point to midpoint of line
Fix/Unfix	Green geometry	Locks the position of the geometry
Parallel		Second line parallel to first line
Perpendicular		Second line perpendicular to first line
Horizontal		Line parallel to horizontal sketch axis
Vertical		Line parallel to vertical sketch axis
Tangent		Inferred to circles, arcs Manually added to circles, arcs, lines
Equal		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:

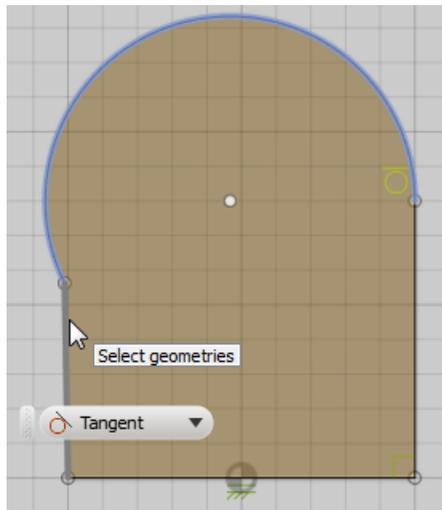
- 1 Coincident

- 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

Use the Constraints command in the Sketch pull down to manually constrain geometry. Select the desired constraint type from the mini-toolbar then select the geometry.



Sketch Dimensions

Sketch dimensions define the size of sketch entities. You can apply dimensions to entities using the Sketch Dimension command in the Sketch pull-down.

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.

Double-click a dimension to edit its value.

You can select entities to display the dimensions to verify the size of the entities. The dimensions disappear as soon as you click on a command or another entity.

Sketch dimensions:

- Display when the sketch is active.
- Prevent accidental changes to entity size.
- 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.

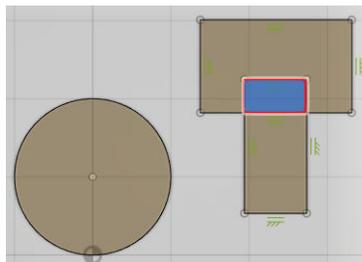
NOTE For some entities, such as arcs, you cannot place a dimension between the curve and another entity. If you select the entity, a radius dimension is created. You can create a dimension between the center point and other entities.

Sketch Profiles

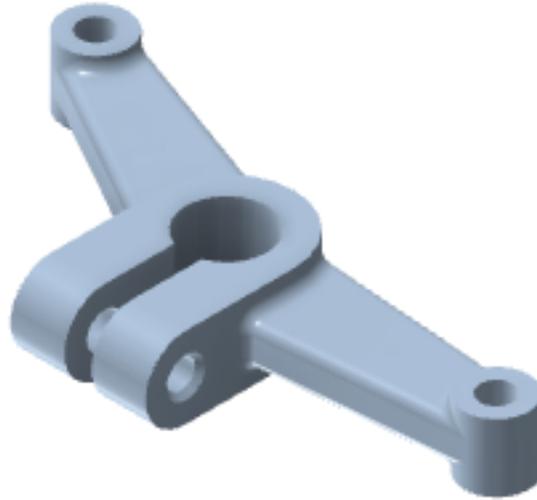
Profiles are coincident sketch geometry used to create a feature. Solid features require closed profiles. Surfaces features can use open or closed profiles.

Closed 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 closed profiles are shown in yellow. The blue rectangular region under the cursor illustrates a selected profile.



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 Body, New Component) are available. Most features become independent geometry after they are created.

Only fillets can be edited using an edit command from the browser. Use Press Pull, Tweak Faces, or other modification commands to edit features.

A component can have multiple solid bodies and surface bodies. You can create multiple components in a model, including child components, to build an assembly.

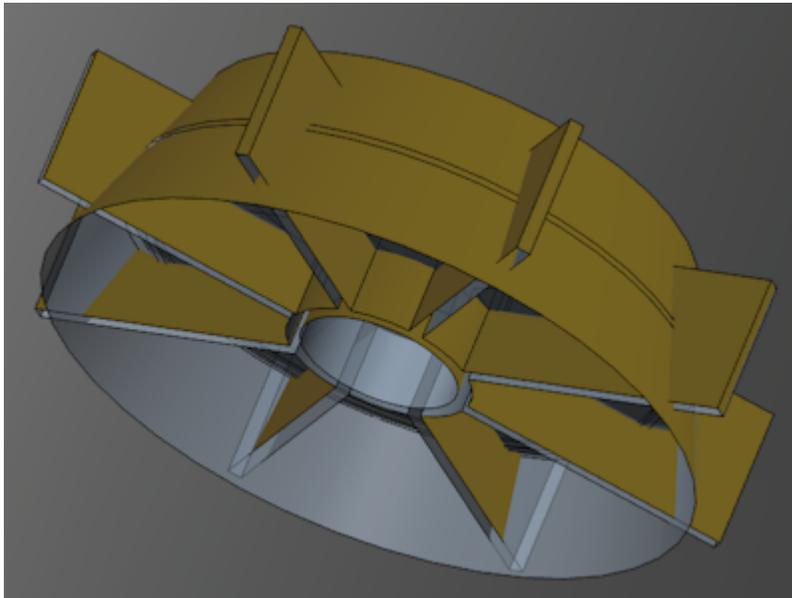
Related topics:

- [Solid workspace](#) (page 55)
- [Browser commands](#) (page 256)

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, and circular pattern using surfaces.

The Thicken command adds thickness to selected surfaces to make a solid. It can add or remove material from a solid, changing its mass properties. New Body and New Component are available for surface feature creation.

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 file. Unlike some other CAD products, Fusion does not require the creation of separate files on disk to use components.

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.

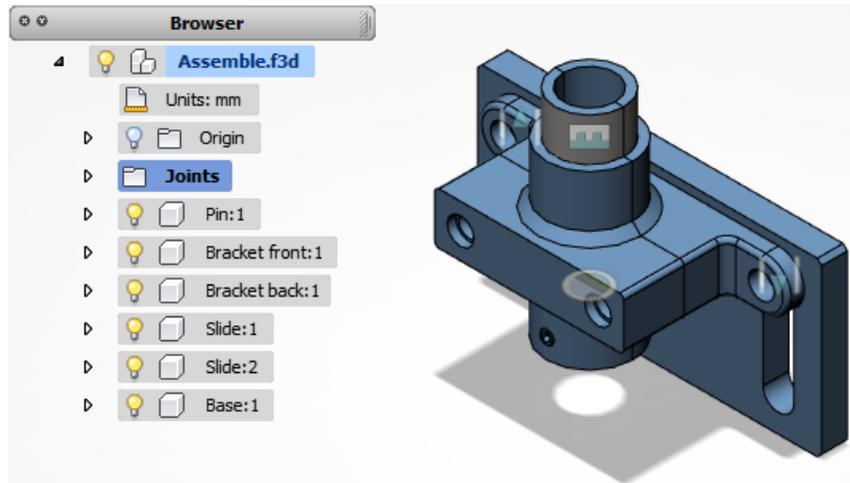
Every Fusion design has a single root component. The root component has the same name as the document itself, and is the top node in the browser.

A Fusion component can also contain:

- **Solid bodies** Zero or more solid bodies.
- **Surface Bodies** Zero or more surface bodies.
- **Sketches** Zero or more 2D sketches.
- **Work Geometry** Work planes, work axes, work points.
- **Features** Features are used to create geometry and group faces.
- **Joints** Joints position component occurrences relative to other occurrences in a model.

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. In the example below, the Slide component is referenced twice.



All instances of the same component share the same geometry, so a change to one applies to all instances of that component.

Related topics:

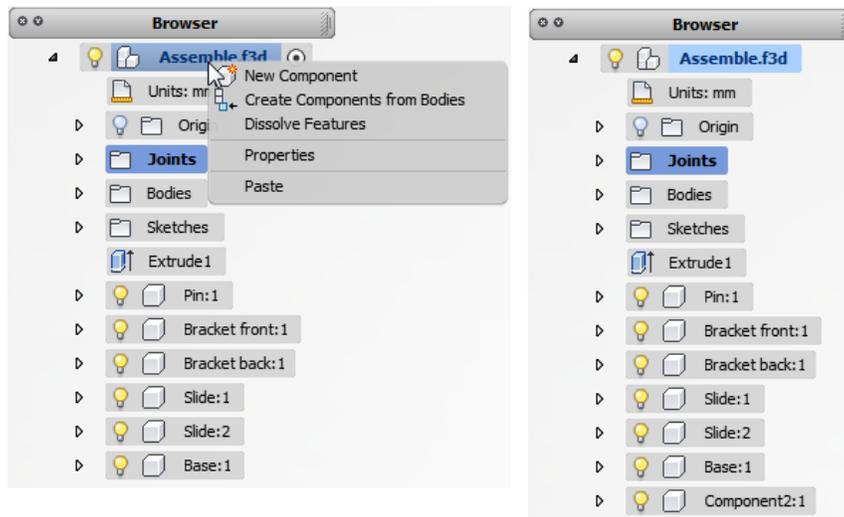
- [Assemble workspace](#) (page 193)
- [Browser commands](#) (page 256)

Create Components

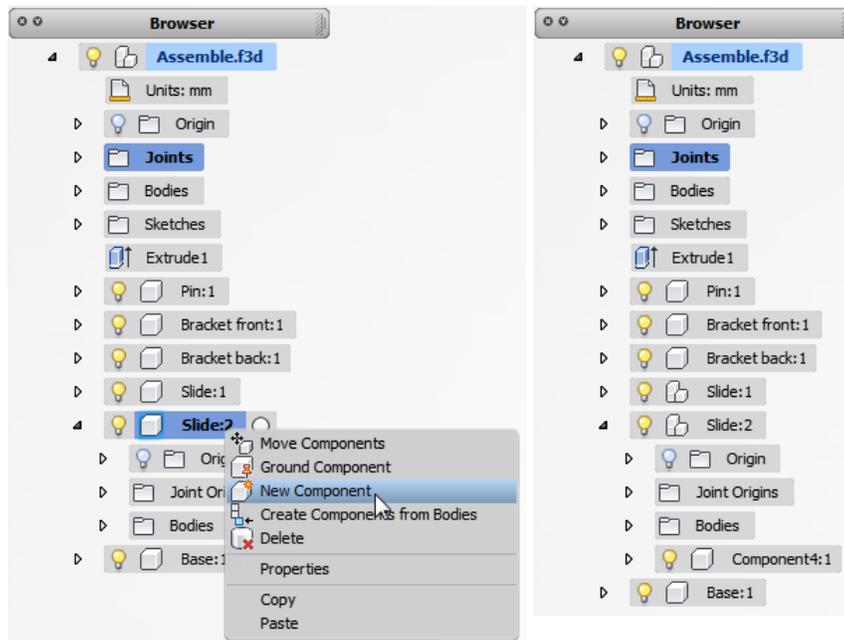
In Inventor Fusion, there are several methods to create components and populate them with geometry.

New Component Command

The New Component command is accessed 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). In the image, Component2:1 is added to the end of the browser.



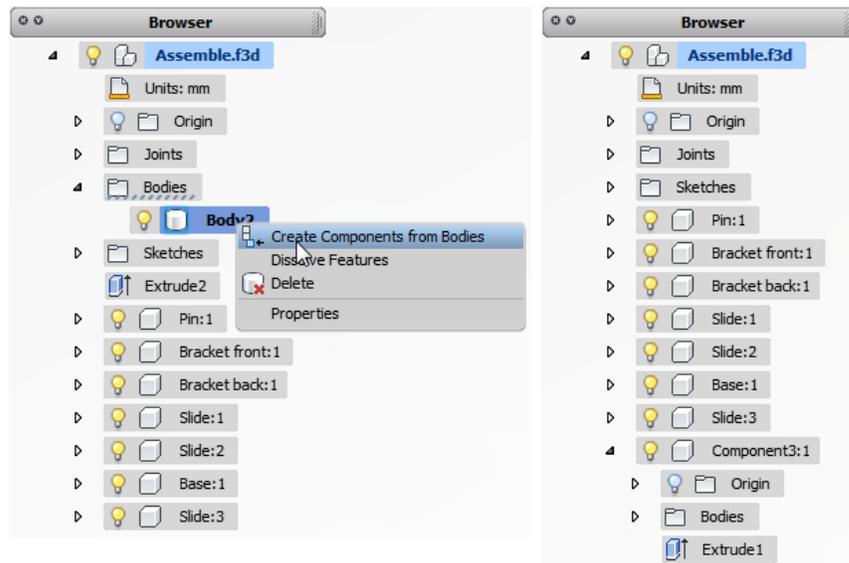
When you create a new component under a part component, the icon is changed to an assembly icon to show it has children. In the image, Component4:1 is added to the Slide component and Slide becomes an assembly.



Create Components from Bodies

The Create Components from Bodies command is accessed from the context menu when you right-click on a component or a body in the browser. The Create Components from Bodies command creates new components and moves the bodies into the component.

In the image, Component3:1 is created from Body2 and is added to the end of the browser.

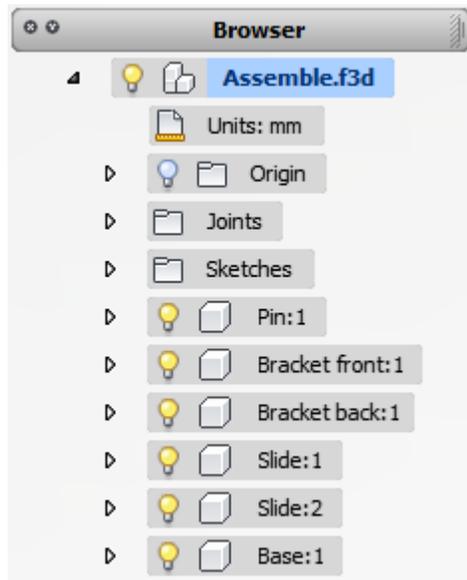


When Create Components from Bodies is used on a single body, a single component is created. When the command is used on a component, a new component is created for each body in the original component.

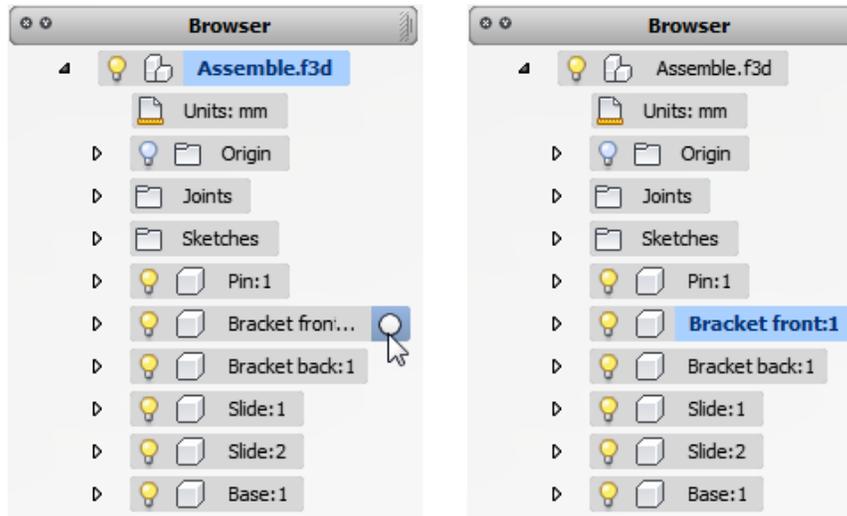
Activate Components

Inventor Fusion always has an active component, where newly created objects are placed. 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:

In the image, the root component (Assemble.f3d) is active.



In the image, Bracket front:1 is set as the active component.



To activate a component, move the mouse over the browser node for that component then select the circle that is displayed.

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 positioned on top of the original component in 3D space. Click and drag the component or use Move Component to change the position.

Position and Constrain Components

You can position component instances in 3D space, 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 Components

The [Move Components command](#) (page 114) uses the triad to change the location and orientation of components. Move Components cannot permanently position components.

Move Components makes local changes, so it can change the position of child components, such as the rod of a pneumatic cylinder.

Joints

[Joints](#) (page 219) position component occurrences relative to other occurrences in a model.

Different occurrences of the same subassembly can have different positions and orientations using joints. For example, one occurrence of a pneumatic cylinder can be shown extended and another occurrence can be shown contracted.

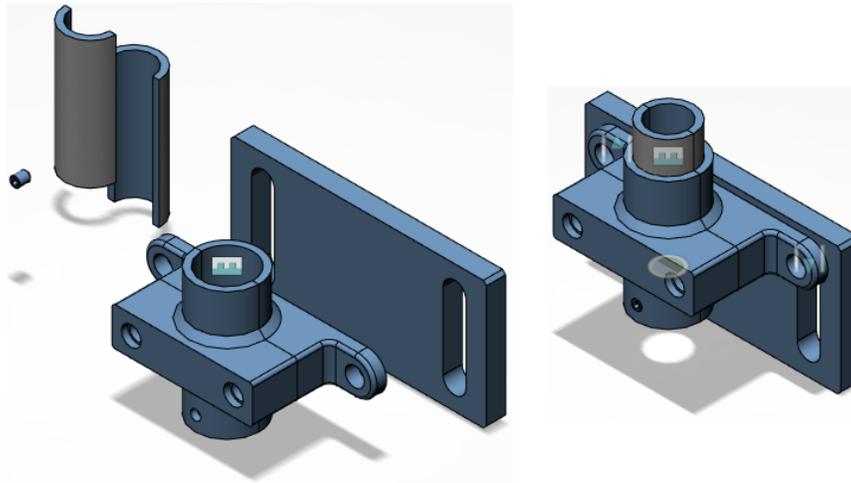
Ground Component

The [Ground Component](#) (page ?) command locks a component in its current location and orientation.

You can turn Ground Component on or off by right-clicking on the component in the browser and selecting Ground Component. A grounded component

has a pushpin on its browser icon. 

Joints



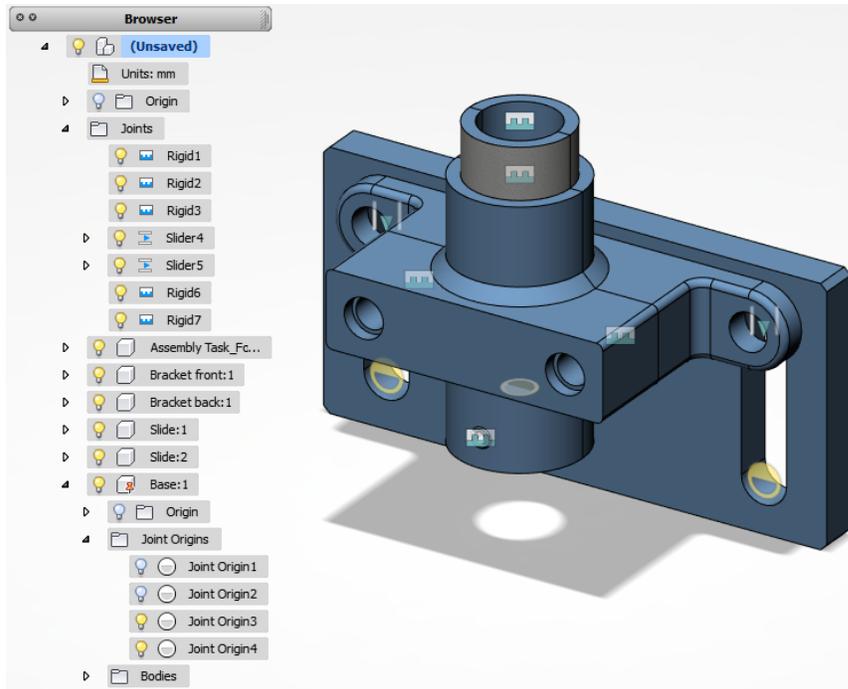
Joints are used to position components relative to one another in a model and to define their relative motion. A joint fully defines all degrees of freedom between two components.

Use the Joint command to define the position of the components by selecting the components and the joint type (rigid, revolute, etc). The allowable motion is determined by the joint type.

The Joint Origin command allows you to place a joint origin at a selected position on a component. Joint origins represent one half of a joint and are used to create a joint when connected to another joint origin on a different component. To use joint origins:

- Select a joint origin with the Joint command active.
- Drag and drop one joint origin on top of another .

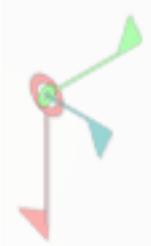
Joints and joint origins are listed in the browser and displayed in the graphics window. Joints are listed in the browser with the parent assembly. The movable degrees of freedom defined by the joint type are listed under each joint. Joint origins are listed in the browser with the associated component.



Joint types

	Joint Type	Description	Motion Allowed
	Rigid	Locks components together, removing all degrees of freedom.	None
	Revolute	Allows the component to rotate around joint origin.	1 rotation

	Joint Type	Description	Motion Allowed
	Slider	Allows the component to translate along a single axis.	1 translation
	Cylindrical	Allows the component to rotate and translate along the same axis.	1 translation 1 rotation
	Pin-slot	The component can rotate about an axis and translate about a different axis.	1 translation 1 rotation
	Planar	Allows the component to translate along two axes and rotate about a single axis.	2 translation 1 rotation

	Joint Type	Description	Motion Allowed
	Ball	Allows the component to rotate about all three axes using a gimbal system (three nested rotations).	3 rotation

Joint commands

Create joints

Use the Joint command

- 1 Switch the toolbar workspace to Assemble.
- 2 Click Joints in the Joints pull-down.
- 3 Select a joint origin or geometry to define an origin. The first component selected cannot be grounded or the joint origin cannot be on a body in the main assembly.

NOTE When selecting geometry, hold the <Command> key to lock the selection to the highlighted geometry then select the desired point for the origin location.

- 4 Select a second joint origin or geometry to define a second origin. The first joint origin is aligned with the second to show the component relationship when the joint values are zero. This is a temporary positioning used while the joint is defined.
- 5 A joint type is selected based on the geometry selected for the joint origins. Change the joint type in the Type pull-down if necessary. The components will animate to display the allowed motion for the joint.
- 6 Depending on the type of joint selected, you can change the Rotation Axis, Slide Axis, or Slide Plane.
- 7 Adjust the joint origin alignment by setting the alignment offset and angle. You can enter values in the fields or drag the manipulators in the graphics window. These adjustments allow you to tweak the relative position where the joint's values are zero.

- 8 Click Flip to reverse the orientation, if necessary.
- 9 Right-click then select OK to create the joint.

Drag joint origins

- 1 Click and drag a joint origin.
- 2 Drag the joint origin over another existing joint origin. The Joint command is activated.
- 3 A joint type is selected based on the geometry selected for the joint origins. Change the joint type in the Type pull-down if necessary. The components will animate to display the allowed motion for the joint.
- 4 Depending on the type of joint selected, you can change the Rotation Axis, Slide Axis, or Slide Plane.
- 5 Adjust the joint origin alignment by setting the alignment offset and angle. You can enter values in the fields or drag the manipulators in the graphics window. These adjustments allow you to tweak the relative position where the joint's values are zero.
- 6 Click Flip to reverse the orientation, if necessary.
- 7 Right-click then select OK to create the joint.

A single joint origin can be used to create multiple joints. The joint origin must be displayed to select it when creating joints. Use the light bulb icon in the browser to control the visibility of joint origins.

Edit joints

After joints are created, you can edit them to change the joint type, offset or angle values, or modify other joint settings.

- 1 Select a joint in the browser or the graphics window.
- 2 Right-click and select Edit Joint.
- 3 The joint manipulators and fields are displayed.
- 4 Make desired changes and select OK to apply the changes.

Lock joint values

After joints are created, you can manually lock their values (open degrees of freedom) to further control component movement. The lock applies to one

occurrence of the joint as determined by the nested components leading to the joint in the browser.

- 1 Expand the nested components and the Joints folder in the browser.
- 2 Expand the desired joint node in the browser.
- 3 Select the value (degree of freedom) node in the browser.
- 4 Right-click then select Lock.

Limit joint values

After joints are created, you can manually limit their values to ranges and optionally define a preferred resting point. You can also reverse the direction of positive values.

- 1 Expand the nested components and the Joints folder in the browser.
- 2 Expand the desired joint node in the browser.
- 3 Select the value (degree of freedom) in the browser.
- 4 Right-click then select Limit Value.
- 5 In the Limit Value dialog box, click Minimum then enter a value for the minimum position or drag the limit handle in the graphics window.
- 6 Click Maximum then enter a value for the maximum position or drag the limit handle in the graphics window.
- 7 Click reverse to switch the minimum and maximum value.
- 8 Click animate to show the motion of the joint. The component will animate until the box is unchecked.
- 9 Right-click then select OK.

Define joint origins

Joint origins can be defined outside the joint command. Joint origins define the geometry used to relate a joint's components. For example, a revolute joint binds a point and an axis. The points and axes are specified by joint origins.

- 1 Switch the toolbar workspace to Assemble.
- 2 Click Joint Origins in the Joints pull-down.
- 3 Select the face or edge to specify the position of the joint origin.

NOTE When selecting geometry, hold the <Ctrl> key to lock the selection to the highlighted geometry then select the desired point for the origin location.

- 4 Optional, click Primary Axis in the dialog box and select a face or edge to define the orientation of the origin.
- 5 Optional, click Secondary Axis in the dialog box and select a face or edge to further define the orientation of the joint origin.
- 6 Right-click then select OK.

A single joint origin can be used to create multiple joints. The joint origin must be displayed to select it when creating joints. Use the light bulb icon in the browser to control the display of joint origins.

Modify joint values

You can change the offset and angle values of a joint occurrence. There are multiple ways to modify values:

- Click and drag a component. This causes the entire mechanism to move according to the degrees of freedom.
- Double-click the joint glyph in the graphics window.
- Use the Move Joints command. This movement is limited to the movement related to the selected joint.

Delete joints and joint origins

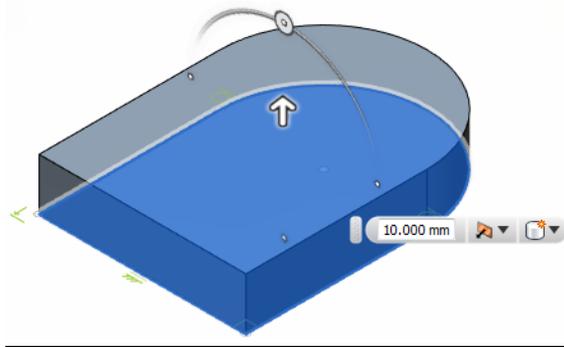
Joints and joint origins can be deleted or removed from an assembly. Deleting a joint will remove all occurrences of that joint.

- Right-click a Joint or Joint Origin in the browser then select Delete.
- Select a Joint or Joint Origin then press the <Delete> key.

Inventor Fusion Technology User Interface

2

The user interface in Inventor Fusion comprises standard interface features, such as the toolbar, 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.



Toolbar

The Inventor Fusion toolbar provides access to commonly used commands. Commands are grouped in pull-downs with the most commonly used commands displayed in the toolbar and other commands listed in the pull-down.

The toolbar is organized into workspaces based on the function of the commands available in the workspace.

- The [Solid workspace](#) (page 55) contains commands for creating and editing solid bodies.



- The [Surface workspace](#) (page 126) contains commands for creating and editing surface bodies.



- The [Assemble workspace](#) (page 193) contains commands for positioning and modifying components in an assembly.



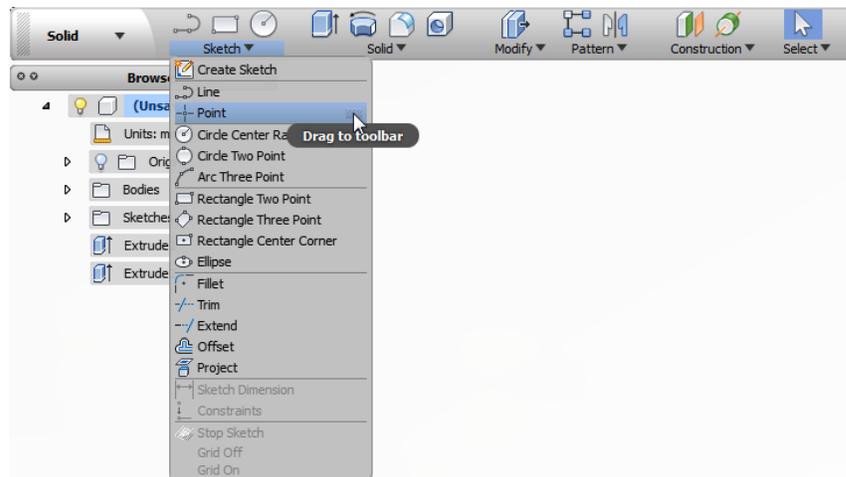
- The [Options workspace](#) (page 239) contains command for controlling the Inventor Fusion display and other options.



Customize the toolbar

You can organize the commands available in the toolbar by dragging the icons.

Drag the hatch at the end of the command name to add it to the top level toolbar.



Drag a command within the toolbar to reorder the commands. Commands cannot be dragged out of the pull-down that contains the 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.

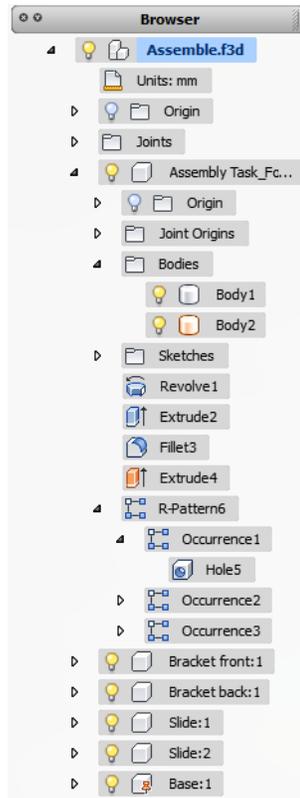
Objects are cross-highlighted in the browser and graphics window. Selecting an object in the browser also highlights it in the graphics window, and selecting an object in the graphics window highlights it in the browser. Cross-highlighting provides visual feedback and simplifies selecting a particular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

- Close the browser and expand/collapse the browser.
- Click the light bulb to turn visibility of an object on or off.
- The Units nodes displays the default units for the asset. The Units node provides access to change the units.
- The Origin folder has the X-Y-Z coordinate system. Components and created features are positioned relative to the parent coordinate system.
- A blue background indicates the active component.
- Joints position components relative to each other.
- Child components can contain features and other components. This hierarchy organizes the model by grouping related features and components together.
- Joint origins are used to create and position joints.
- A component can have multiple bodies. A body can be a solid or a surface.
- Sketches contain 2D geometry for creating features.
- 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.

- Objects can be mirrored and patterned 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.



Related topics:

- [Browser commands](#) (page 256)

Tools in the Graphics Window

Heads-Up Display

Inventor Fusion commands use a heads-up display. The heads-up display provides controls in the graphics window instead of on dialog boxes. There are three types of controls:

- Value Entry Fields
- Mini-toolbars
- Manipulators

Value Entry Fields

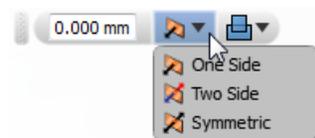
The value entry fields provide several methods for setting the value for a command or entity.



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

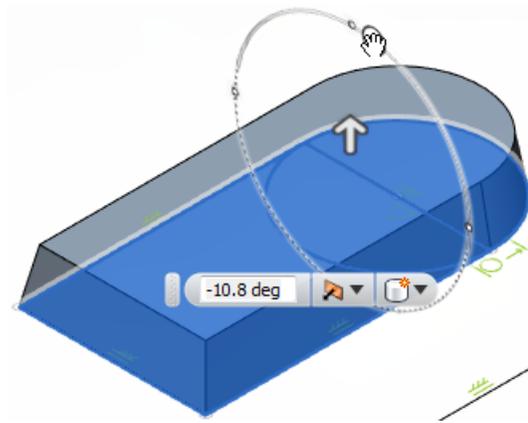
Mini-toolbars

Mini-toolbars can have value entry fields, glyphs, and buttons. Most mini-toolbars display near the cursor when a command is active.

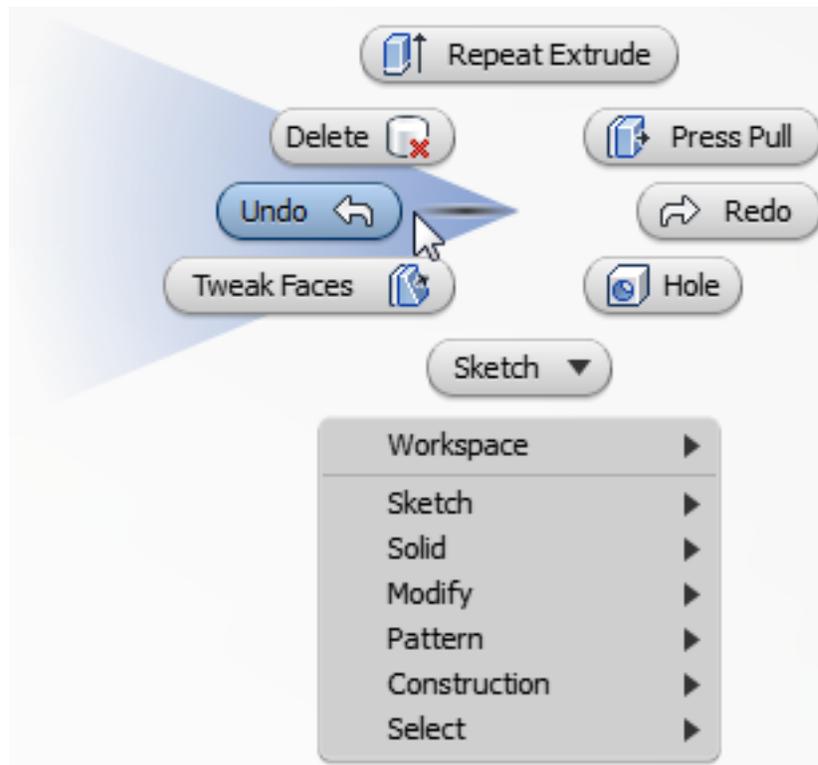


Manipulators

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. Sketch and Feature commands use arrow, angle, 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.

Each workspace (Solid, Surface, and Assemble) has its own marking menu.

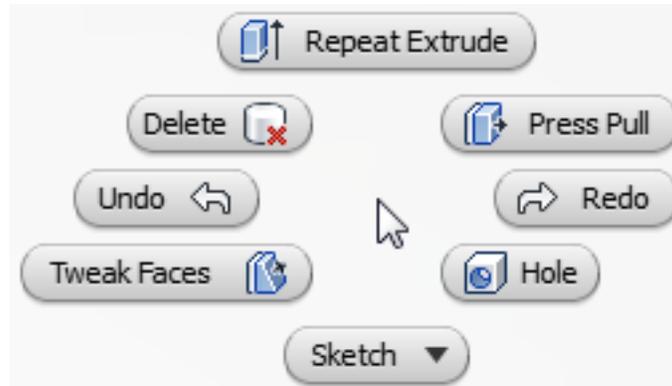
As you move the cursor from the center of the marking menu towards a node, the wedge displays on the screen. You can click anywhere in the wedge to select the item.

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 most 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:

	Sketch	Secondary Sketch	Solid	Surface	Assemble
North	Repeat last command	Stop Sketch	Repeat last command	Repeat last command	Repeat last command
North-east	Press Pull	Rectangle Two Point	Press Pull	Press Pull	Press Pull
East	Redo	Constraints	Redo	Redo	Redo
South-east	Hole	Project	Hole	Patch	Joint
South	Sketch	Line	Sketch	Sketch	Sketch
South-west	Tweak Faces	Offset	Tweak Faces	Reverse Normal	Joint Origin

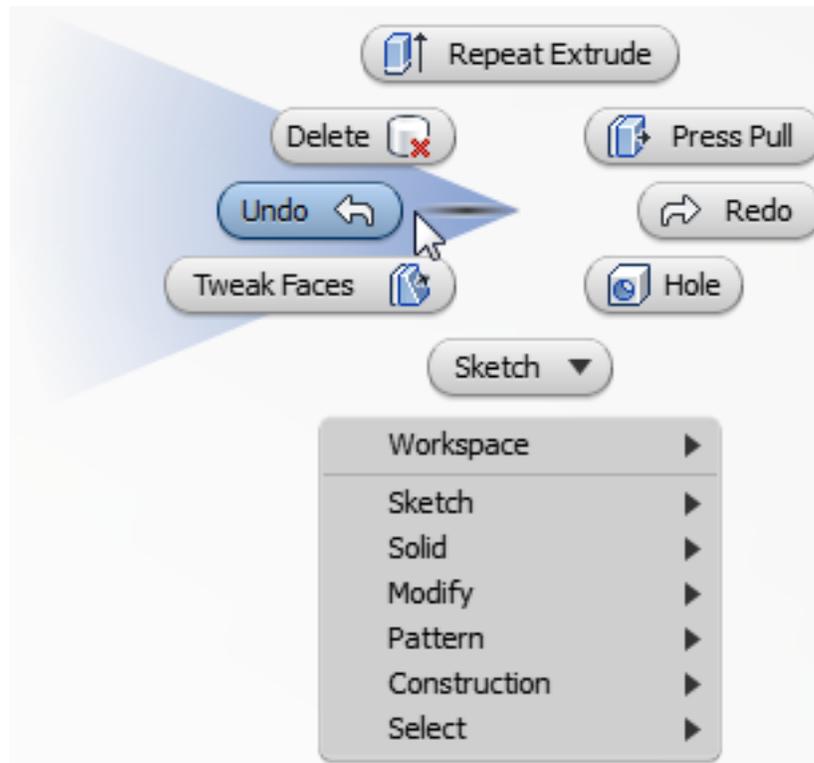
	Sketch	Secondary Sketch	Solid	Surface	Assemble
West	Undo	Sketch Dimension	Undo	Undo	Undo
Northwest	Delete	Circle Center Radius	Delete	Delete	Delete

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.

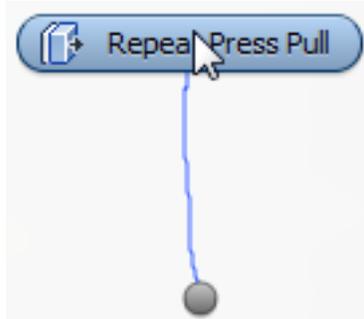


- 1 Right-click in the graphics window. A radial menu displays, centered on the current cursor position.
- 2 Move the cursor towards a menu item. A wedge displays showing which item is active.
- 3 Click anywhere in the wedge to select the item.

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.

Manipulators

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

The active manipulator lights up and inactive manipulators are dimmed. Click on a manipulator to activate it.

Sketch and Feature commands use arrow and ring manipulators. Move, Move Components, and Tweak Faces use a triad manipulator.

Many manipulators can 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 lights up and show a dashed line while inactive

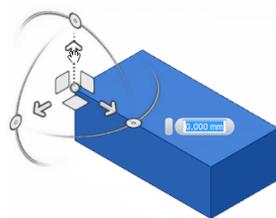
manipulators are dimmed. Click on a manipulator to activate it or press the <Tab> key to cycle between manipulators.

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

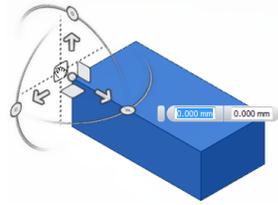
Manipulator Image	Type	Description
	Linear	Sets the distance value.
	Arc	Sets the rotation angle value.
	Ring	Sets the diameter and position for hole features.

Move Manipulator

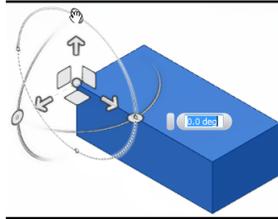
The triad provides constrained translation or rotation for an object. The triad has arrow, plane, and radial manipulators. You can translate using the arrows or planes; and rotate using the radial arrow.



Drag an arrow to move the object along the axis.



Drag a plane to move the object in two dimensions.



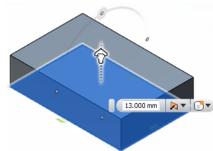
Drag an arc to rotate the object.

Use Manipulators

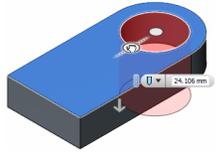
Drag the manipulator to complete (or preview) the operation. The value that is set by dragging the manipulator can also be set in mini-toolbar or the dialog box.

The active manipulator lights up and inactive manipulators are dimmed. Click on a manipulator to activate it.

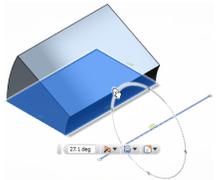
Many manipulators can also snap to other geometry on your model. Move your cursor over other geometry to display a prompt for snapping. When you snap a manipulator to other geometry, the manipulator becomes blue to indicate it is constrained.



Linear Sets the direction and distance value.



Ring Sets the diameter of a feature.



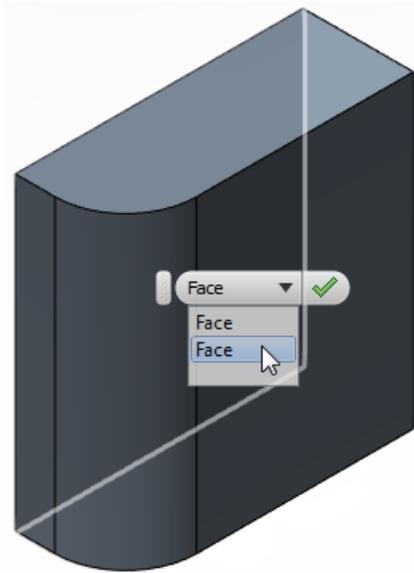
Arc Sets the angle value.

Selection Tool

Use the selection drop-down for geometry that is hidden or difficult to select. The selection drop down provides a list of objects that are available near the cursor.

To active select other, click and hold the left mouse button. Release the left mouse button then click the desired object from the list.

Objects highlight in the graphics window when you move the cursor over the name in the list.



Triad

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

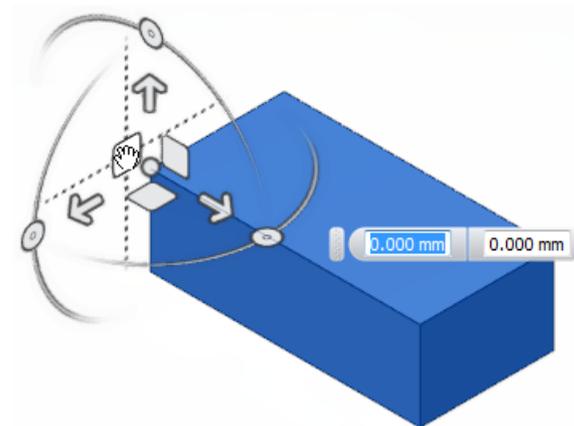
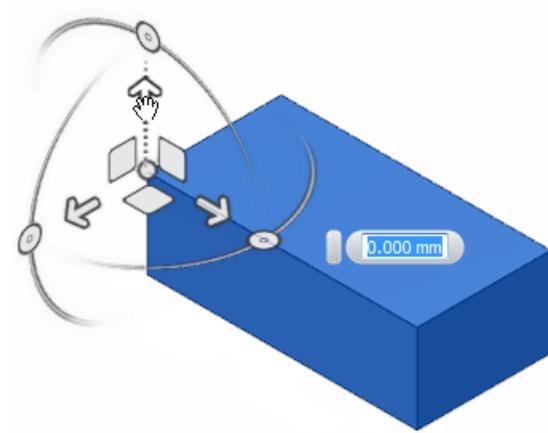
The active manipulator lights up. Click on a manipulator to activate it, then click and drag in the graphics window to move the object.

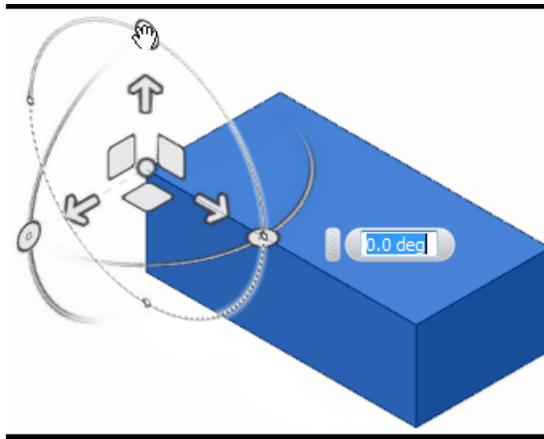


Manipulator	Description
Arrows	Move along the axis.
Planes	Move on the plane.
Arcs	Rotate around the axis.

Appearance of the Triad

The manipulator color highlighting indicates its status. The active manipulator lights up. The inactive manipulators are dimmed.





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.

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 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.
- **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** defines the Front view or Top view of the model based on the current view.
- **Reset Front** resets the Front view of the model to its default orientation.

Display the ViewCube menu

- Right-click the ViewCube or the Home icon to display the context menu.

Navigation Bar



The navigation at the bottom the window of the current model. It provides access to navigation commands.

To start a navigation command, click a button on the navigation bar.

Navigation commands

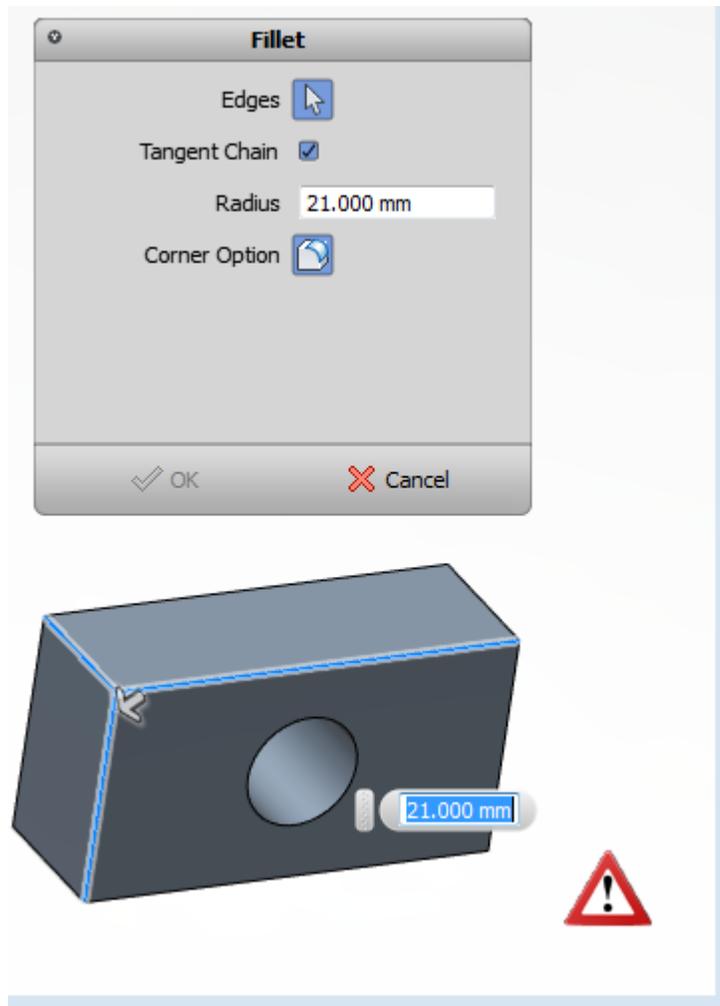
- **Orbit** A set of commands that rotate the current view.
- **Look At** Views faces of a model from a selected plane.
- **Pan** Moves the view parallel to the screen.
- **Zoom** Increases or decreases the magnification of the current view.
- **Fit** Positions the entire model on the screen.

Reposition the Navigation Bar

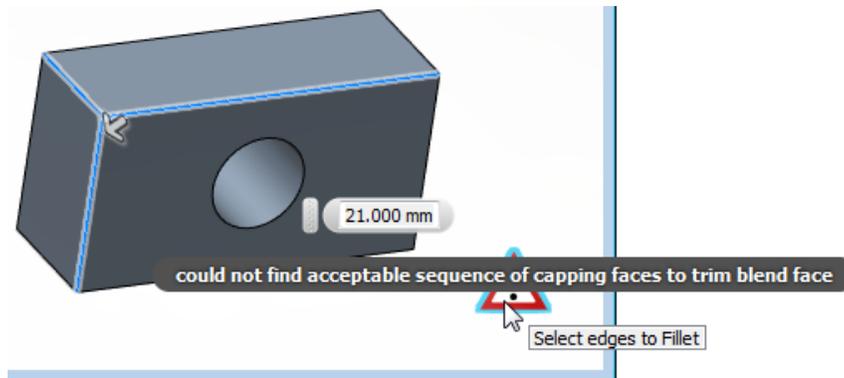
Click and drag the area at the left of the navigation bar to reposition it.

Error Handling

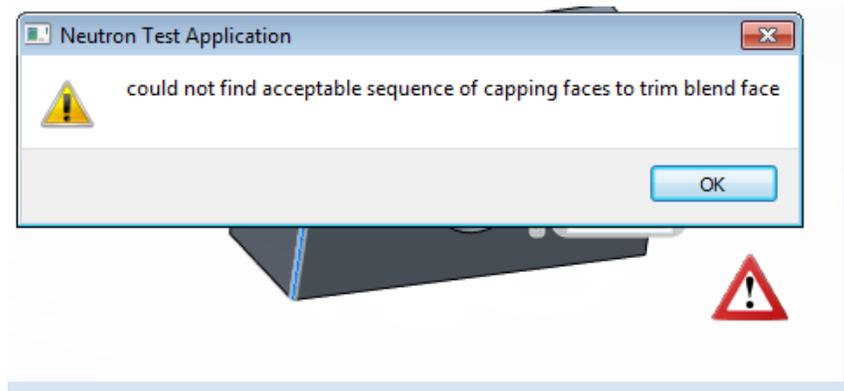
When an error condition occurs, the Error Glyph displays in the lower right-hand corner of the application window:



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.

Inventor Fusion Technology Commands

3

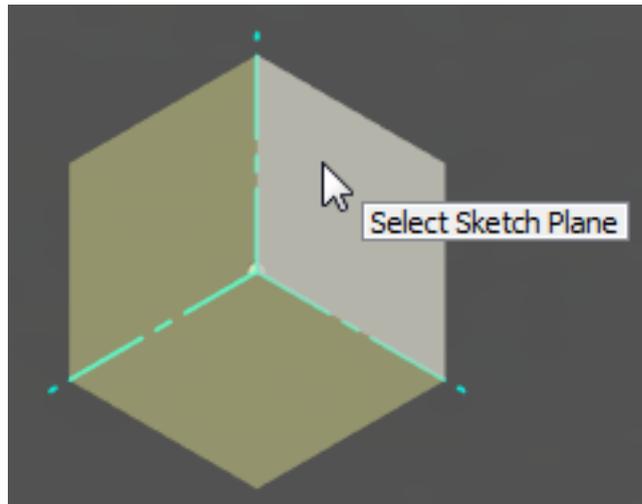
Solid workspace

Use the Solid workspace to create solid bodies in Inventor Fusion.

Sketch

Use the Sketch commands to create and modify 2D sketch geometry in Inventor Fusion.

Create Sketch



Creates a new sketch on the selected plane or face.

Create a sketch

- 1 Expand the Sketch pull-down then click Create Sketch.
- 2 Select the plane or face to create the sketch on. Valid selections are:
 - Origin planes
 - Work planes
 - Flat faces on geometry.

NOTE If you select an existing sketch entity or sketch node in the browser, that sketch is edited and a new sketch is not created.

Edit a sketch

There are two methods to edit a sketch.

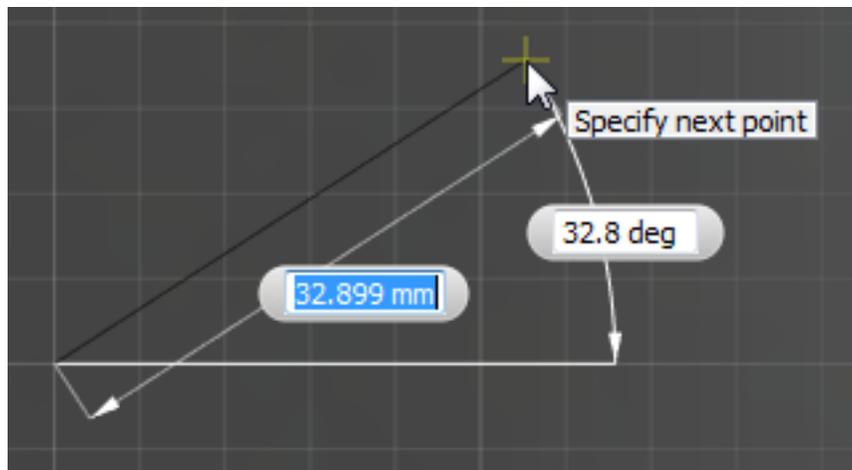
- 1 Right-click the sketch node in the browser then select Edit Sketch.
- 2 Start the Create Sketch command then select a sketch entity or the sketch node in the browser.

Line



The Line command creates a series of lines. A line segment is created each time you click.

Create lines



- 1 Click the Line command in the Sketch pull-down.
- 2 Click to place the starting point for the line.

- 3 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.
- 4 Continue to place points for additional line segments.
- 5 Right-click then select Cancel to end the command.

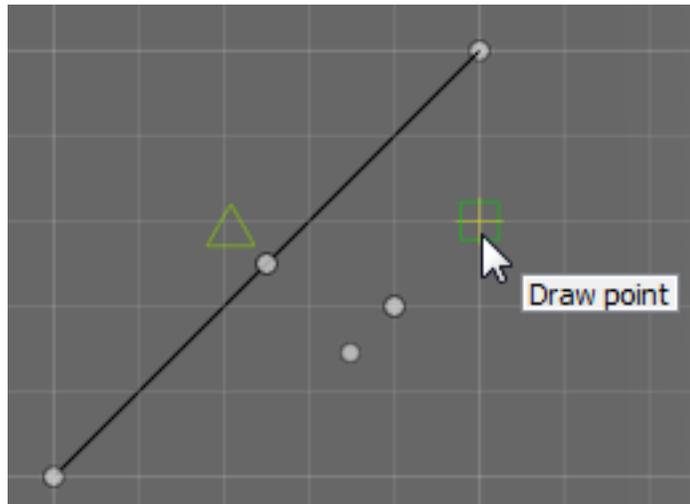
Create arcs



A tangent arc is created if you hold the left mouse button and drag while the line command is active.

- 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.
- 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.

Point

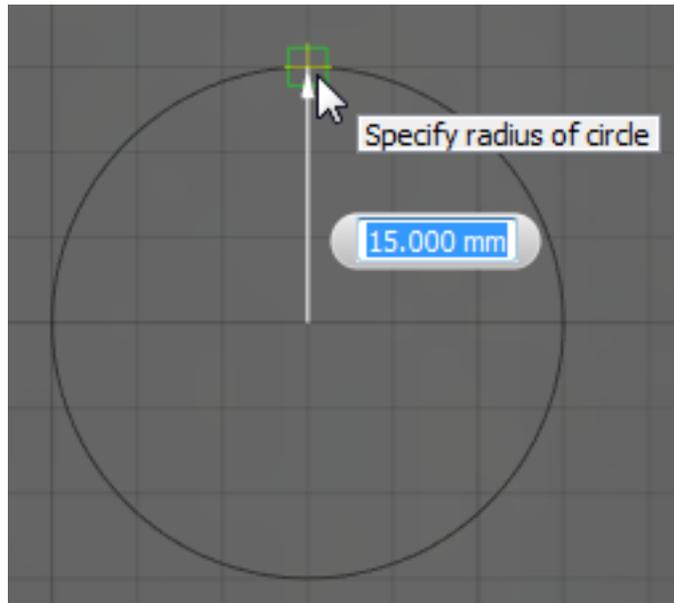


Place sketch points by clicking in the graphics window. You can quickly place a series of points at precise locations, and then create sketch entities by connecting the dots. Sketch points can also be used to position hole features.

Create sketch points

- 1 Expand the Sketch pull-down then click Point.
- 2 Click in the graphics window to create a point. You can snap to the grid or to existing geometry.
- 3 Click again to create additional points.
- 4 Right-click then select Cancel to end the command.

Circle Center Radius

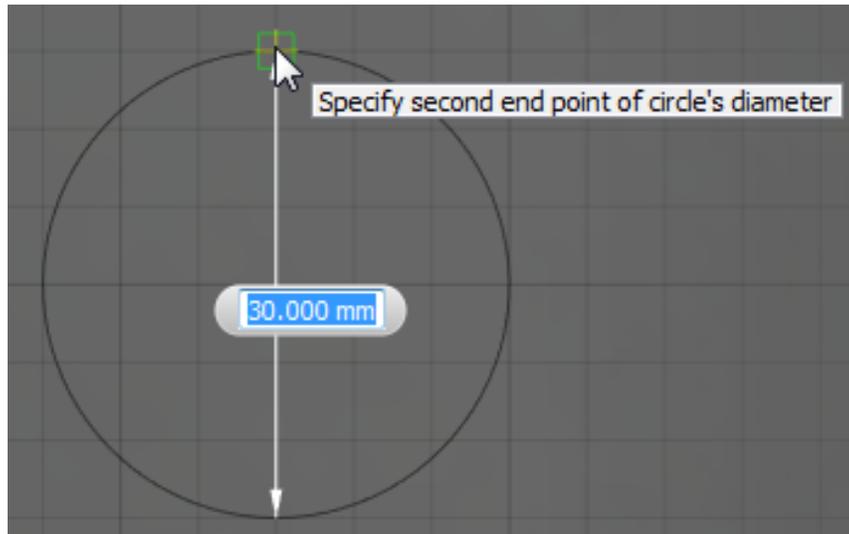


Creates a circle using two user-specified points. The points define the position of the center of the circle and the size of the circle.

Create a circle

- 1 Click Circle Center Radius on the Sketch pull-down.
- 2 Click in the graphics window to specify the center point of the circle.
- 3 To specify the radius of the circle:
 - Click a point in the graphics window.
 - Enter a value in the radius field then press <Enter>. Click a point to create the circle.
- 4 Click again to begin creating a new circle or right-click then select Cancel to end the command.

Circle Two Point



Creates a circle using two user-specified points. The points define the position and the size of the circle.

Create a circle

- 1 Expand the Sketch pull-down then click Circle Two Point.
- 2 Click in the graphics window to specify the first point of the circle's diameter.
- 3 To specify the diameter of the circle:
 - Click a point in the graphics window.
 - Enter a value in the diameter field then press <Enter>. Click a point to create the circle.
- 4 Click again to begin creating a new circle or right-click then select Cancel to end the command.

Arc Three Point

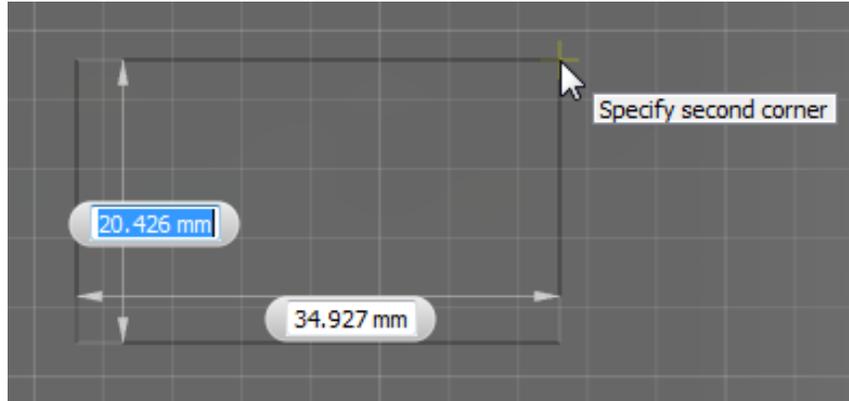


Creates an arc using three user-specified points. The three points define the position and size of the arc.

Create an arc

- 1 Expand the Sketch pull-down then click Arc Three Point.
- 2 Click in the graphics window to specify the first end point of the arc.
- 3 Click again to specify the second end point of the arc.
- 4 Click a third point to specify a point that lies along the arc.
- 5 Click again to begin creating a new arc or right-click then select Cancel to end the command.

Rectangle Two Point



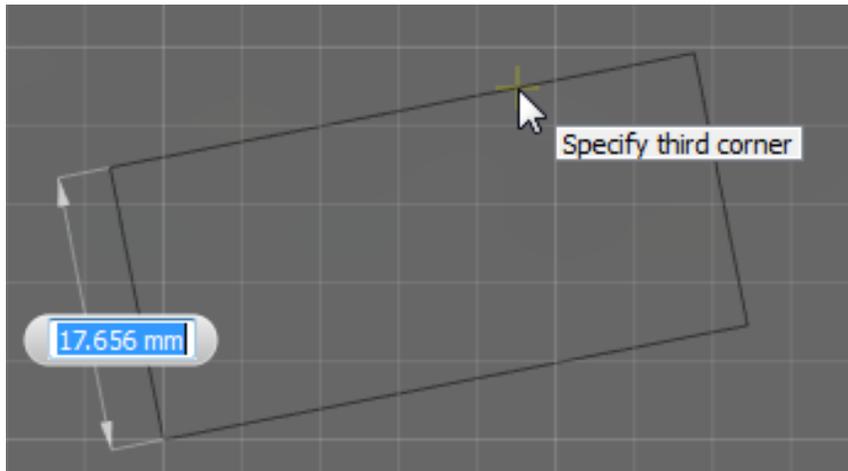
Create a rectangle by selecting two points to define the opposite corners of the rectangle.

Horizontal and vertical constraints are applied to the lines.

Create a rectangle using two points

- 1 Click Rectangle Two Point on the Sketch pull-down.
- 2 Click in the graphics window to specify the first corner.
- 3 To specify the opposite corner:
 - Click a point in the graphics window.
 - Enter values in the length and width fields then press <Enter>. Click a point to define the position of the rectangle.
 - Enter a value in one field then press <Enter> to lock the value. Click a point to define the other value.
- 4 Click a point to begin another rectangle or right-click then select Cancel to end the command.

Rectangle Three Point



Create a 3-point rectangle

Create a rectangle using three user-specified points. The selected points define the position and size of the four lines.

Parallel and perpendicular constraints are applied to the lines.

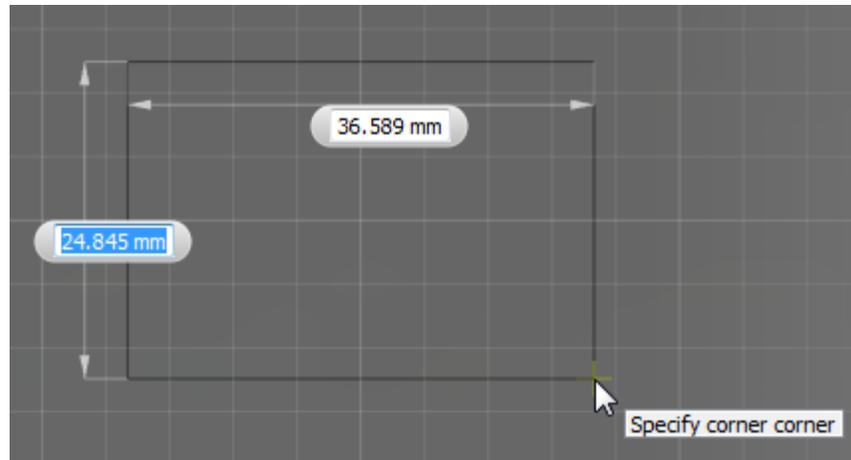
NOTE You cannot infer a parallel or perpendicular constraint to existing sketch entities, but you can infer a horizontal or vertical constraint for the first side.

Create a rectangle using three points

- 1 Expand the Sketch pull-down then select Rectangle Three Point.
- 2 Click in the graphics window to specify the first corner.
- 3 To specify the second point:
 - Click a point in the graphics window.
 - Enter a value in the field then press <Enter> to lock the value. Click a point to specify the angle of the line.
- 4 To specify the third point:
 - Click a point in the graphics window.
 - Enter a value in the field then press <Enter> to lock the value. Click a point to specify the position of the rectangle.

- 5 Click a point to begin another rectangle or right-click then select Cancel to end the command.

Rectangle Center Corner



Create a rectangle by selecting the center point then a corner. The first point defines the position of the rectangle. The second point defines the length and width.

Parallel and perpendicular constraints are applied to the lines.

Create a rectangle using the center point and a corner

- 1 Expand the Sketch pull-down then select Rectangle Corner Center.
- 2 Click in the graphics window to specify the first center of the rectangle.
- 3 To specify the corner:
 - Click a point in the graphics window.
 - Enter values in the length and width fields then press <Enter>. Click a point to define the position of the rectangle.
 - Enter a value in one field then press <Enter> to lock the value. Click a point to define the other value.
- 4 Click a point to begin another rectangle or right-click then select Cancel to end the command.

Ellipse



Creates an ellipse using three user-specified points. The points define the position and size of the ellipse.

Create an ellipse

- 1 Expand the Sketch pull-down then click Ellipse.
- 2 Click in the graphics window to specify the center point of the ellipse.
- 3 Click to specify the end point of the first axis of the ellipse.
- 4 Click a point to specify a point along the ellipse.
- 5 Click again to begin creating a new ellipse or right-click then select Cancel to end the command.

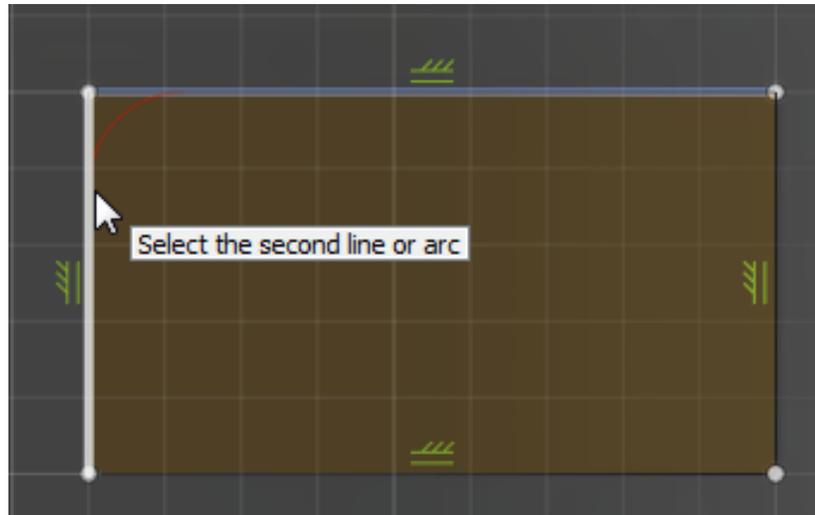
Fillet



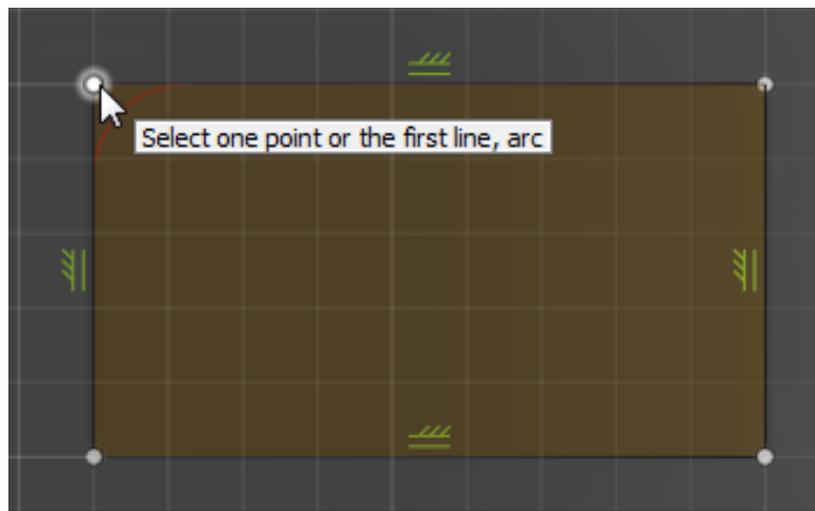
The Fillet command places an arc of a specified radius at a corner or intersection of two lines.

You can create fillets between:

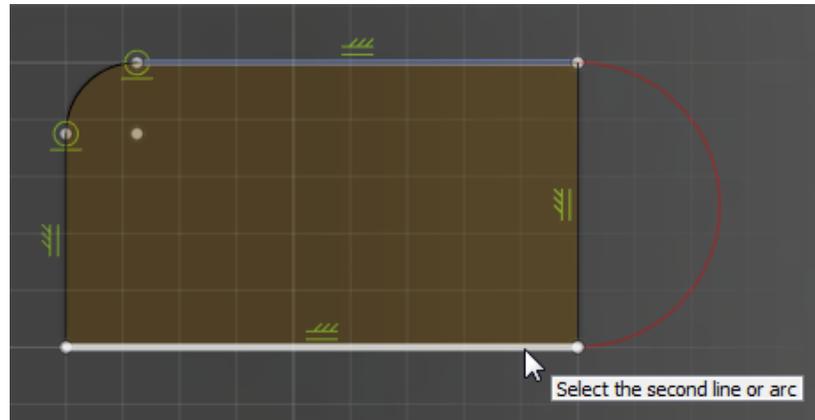
- Two intersecting lines



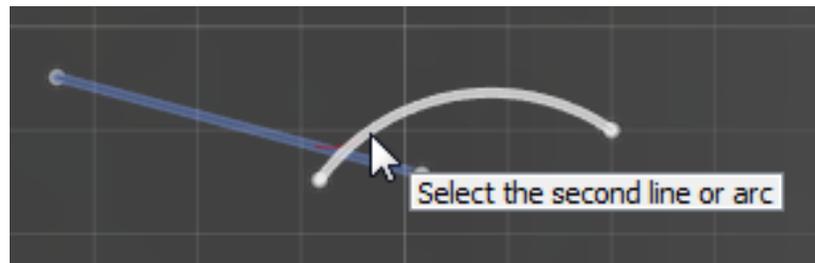
- Point at two intersecting lines



- Two parallel lines



- A line and an arc that intersect



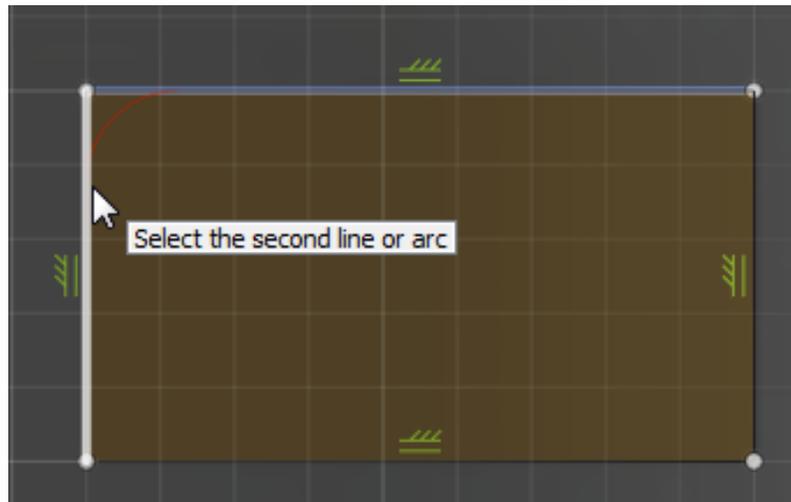
- Two circular arcs that intersect



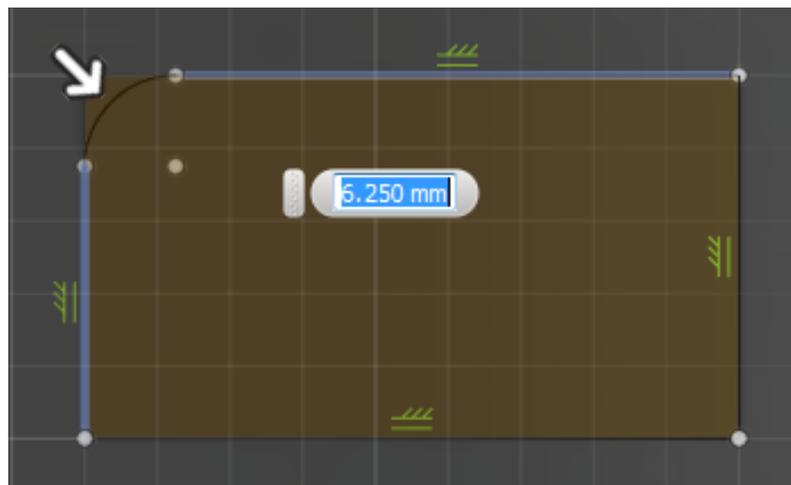
Fillet entities

- 1 Expand the Sketch pull-down then select Fillet.

- 2 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.
- 3 Pause the mouse over the second entity. The fillet preview highlights for a valid selection.

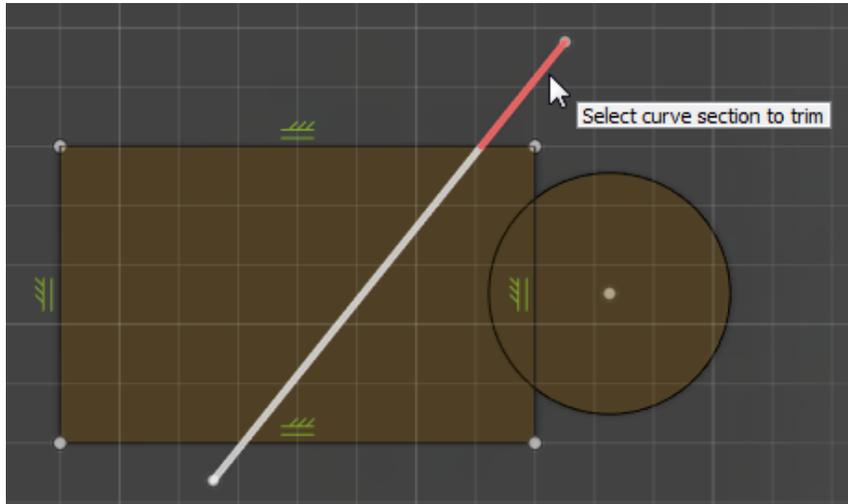


- 4 Click to select the second entity.
- 5 The fillet previews with a value that is proportional to the smaller entity. Enter a value in the box, or drag the arrow to change the size.



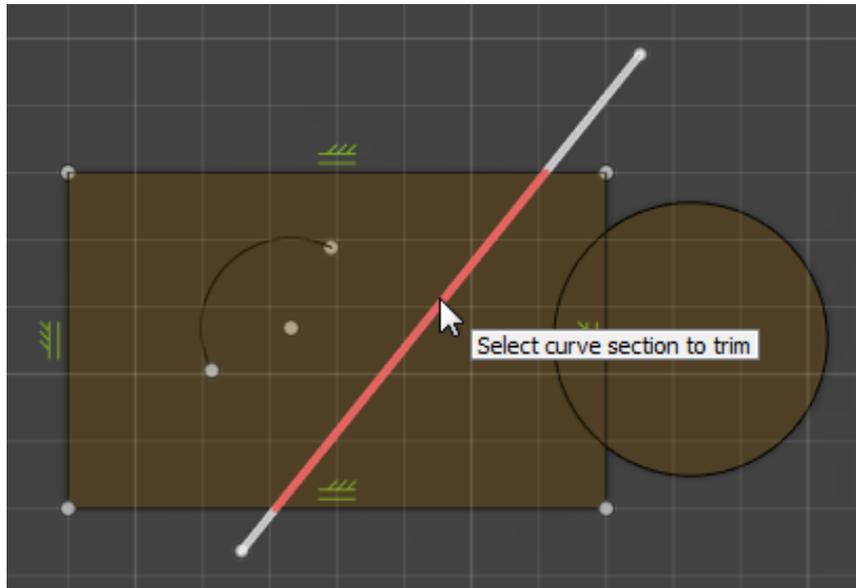
- 6 Right-click then select OK or press <Enter> to complete the command.

Trim

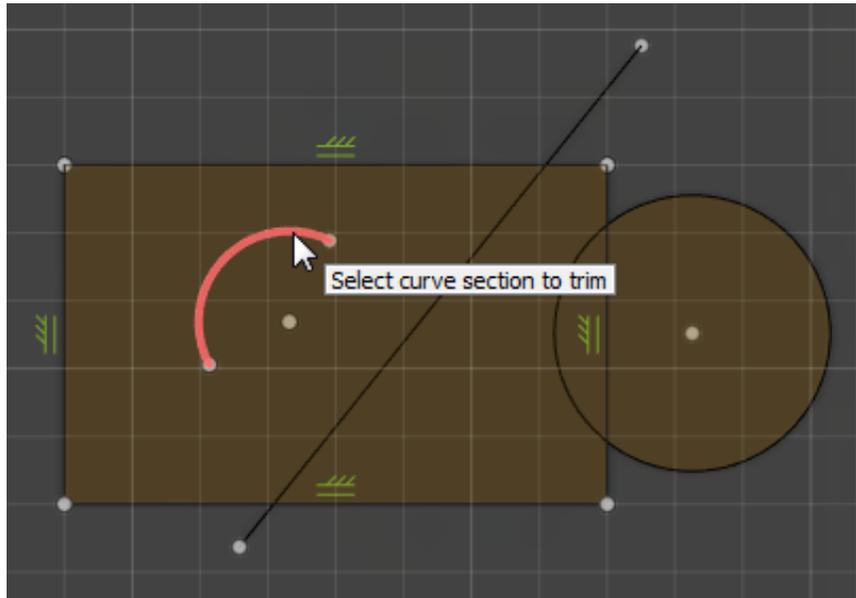


Removes a portion of the selected geometry up to the next intersection with existing geometry. If no intersection exists, the selected entity is deleted.

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.



Trim an entity

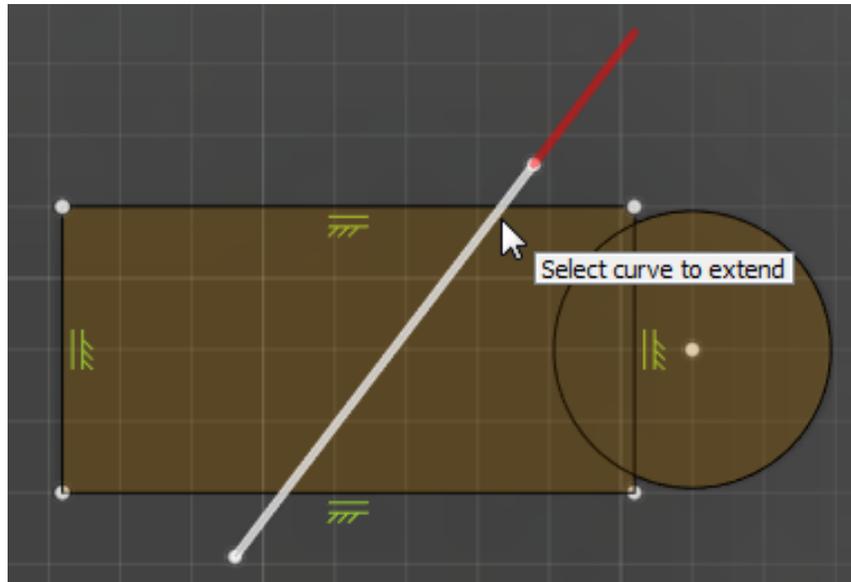
- 1 Expand the Sketch pull-down then select Trim.
- 2 Hover over sketch geometry to see a preview of the trim.
- 3 Click geometry to trim to the next intersection.
- 4 Continue clicking geometry to trim.
- 5 Right-click then select Cancel to end the command.

Extend



Extends the selected geometry to the next intersection of geometry. If no intersection exists, the geometry cannot be extended.

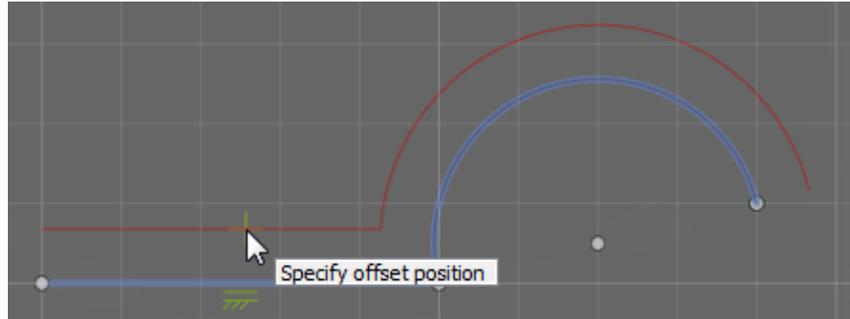
The Extend command will project existing geometry to extend to the theoretical intersection.



Extend an entity

- 1 Expand the Sketch pull-down then select Extend.
- 2 Hover over sketch geometry to see a preview of the extension.
- 3 Click geometry to extend to the next intersection.
- 4 Continue clicking geometry to extend.
- 5 Right-click then select Cancel to end the command.

Offset



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.

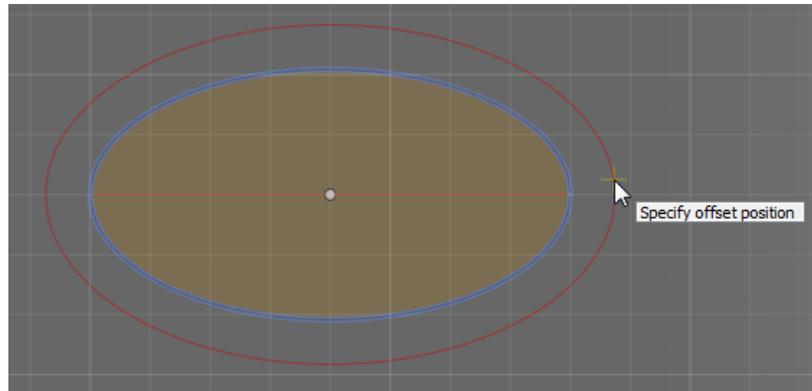
NOTE Offset entities do not keep a reference to the original geometry. You can change the length of any entity independently.

Offset sketch entities

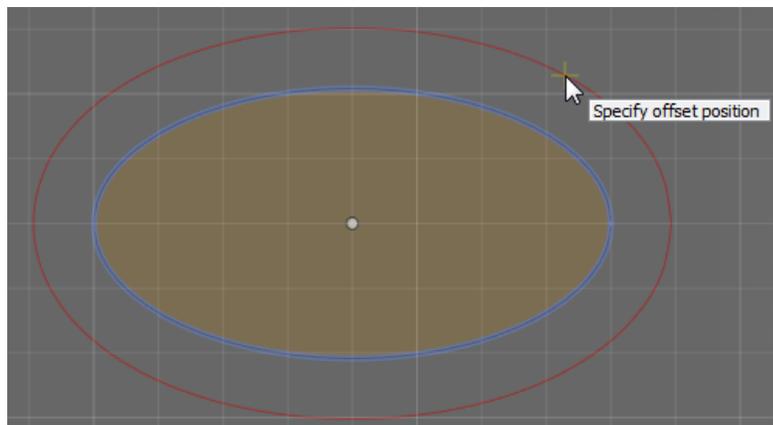
- 1 Expand the Sketch pull-down then select Offset.
- 2 In the graphics area, select an individual entity, a chain of connected entities, or a profile.
A valid selection highlights when you pause the cursor over it
- 3 Move the cursor then click to create the offset.
- 4 Select another entity to offset or right-click then select Cancel end the command.

Offset an ellipse

- Select an 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.

Project

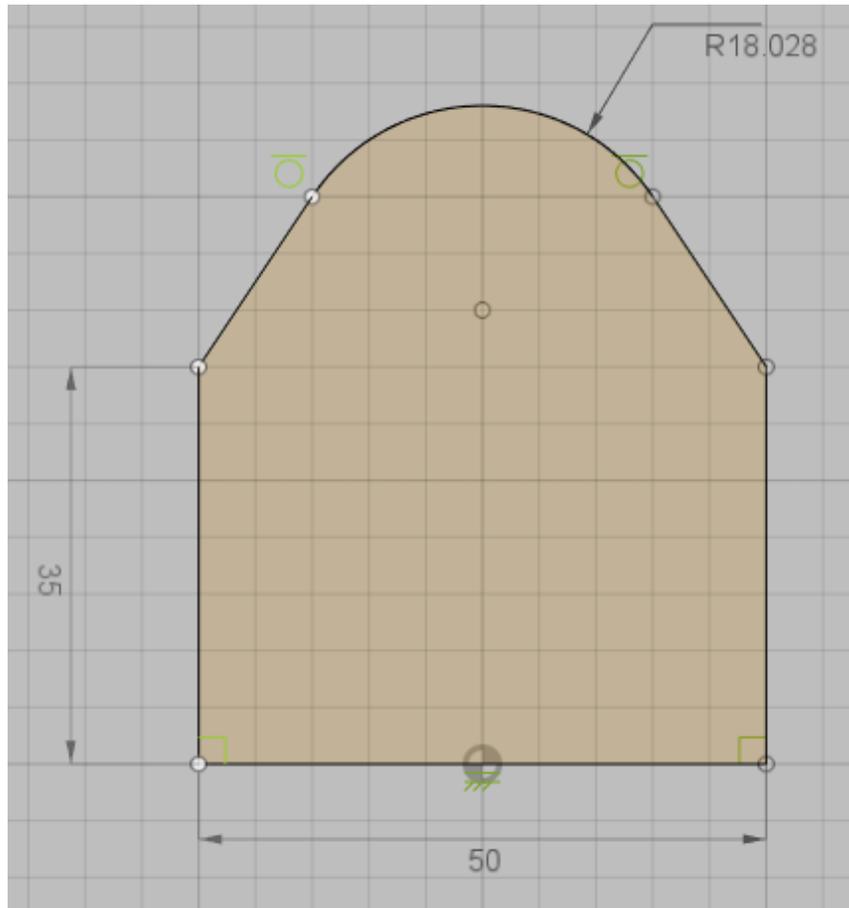
Use project geometry to create copies of existing model geometry and sketch entities in the active sketch.

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.

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

Sketch Dimension



Creates a new sketch on the selected plane or face.

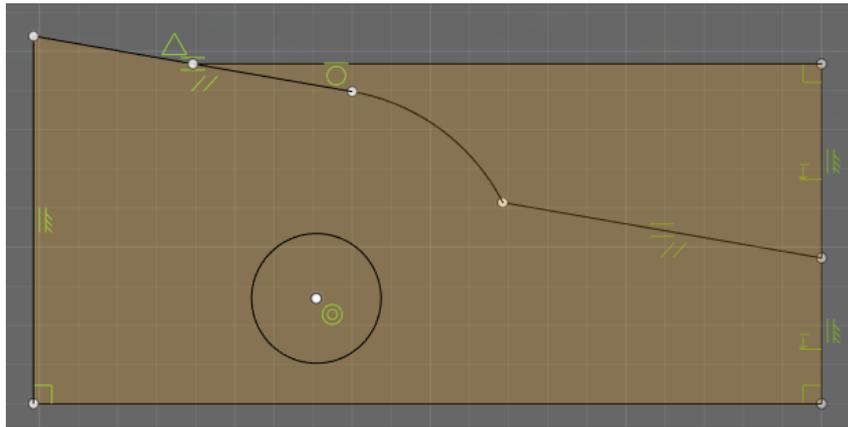
Dimension sketch geometry

- 1 Expand the Sketch pull-down then click Sketch Dimension.
- 2 Select the geometry to dimension. Multiple geometries can be selected. For example, you can select two lines to create an angle dimension or a line and a point to get a distance.
- 3 Pick a point to place the dimension.

- 4 Select other geometry to continue dimensioning or right-click then select Cancel to end the command.

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.

Add constraints to existing geometry using the Constraints command in the Sketch pull down.

Constraint types

Con- straint	Glyph	Description
Coincident		Point to point or snap Point to entity

Constraint	Glyph	Description
Collinear		Manually added only
Concentric		Circle, arc, or ellipse have the same center point.
Midpoint		Point to midpoint of line
Fix/Unfix	Green geometry	Locks the position of the geometry
Parallel		Second line parallel to first line
Perpendicular		Second line perpendicular to first line
Horizontal		Line parallel to horizontal sketch axis
Vertical		Line parallel to vertical sketch axis
Tangent		Inferred to circles, arcs Manually added to circles, arcs, lines
Equal		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 Parallel
- 6 Horizontal
- 7 Vertical

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

Add Constraints Manually

Use the Constraints command in the Sketch pull down to manually constrain geometry.

- 1 Expand the Sketch pull-down then click Constraints.
- 2 Select the constraint type from the mini-toolbar.
- 3 Select the geometry to apply the constraint.
- 4 Continue applying constraints as needed.
- 5 Right click then select OK to end the command.

Stop Sketch

Exits the active sketch.

- On the Sketch pull-down, click Stop Sketch.
- On the marking menu, select Sketch then Stop Sketch.

Grid Off

Turns off the display of the sketch grid when a sketch is active.

Expand the Sketch pull-down then click Grid Off.

Grid On

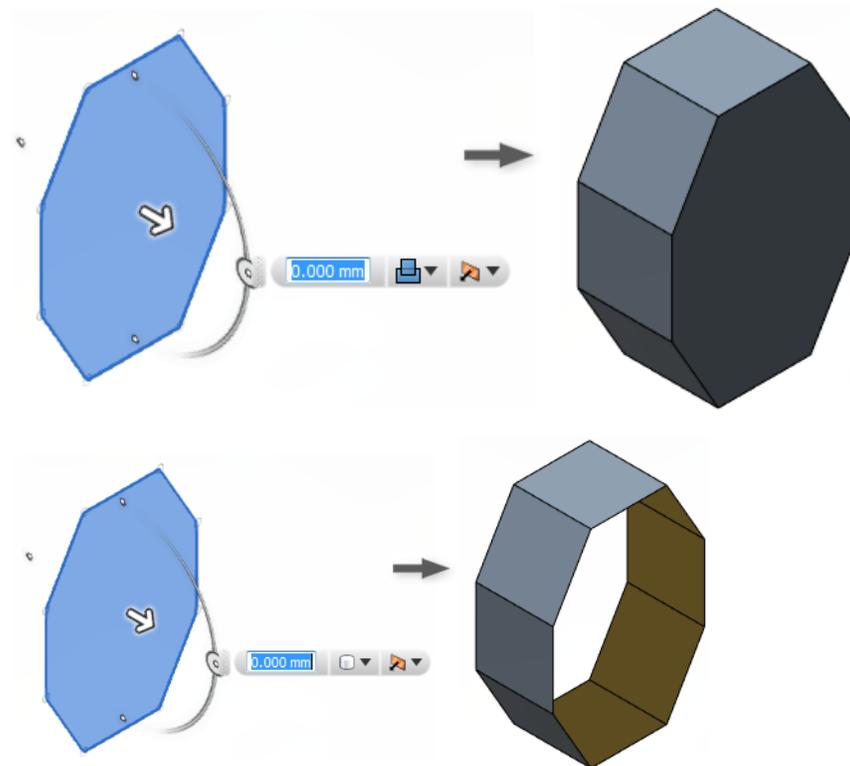
Turns on the display of the sketch grid when a sketch is active.

Expand the Sketch pull-down then select Grid On.

Solid

Use the Solid commands to create and modify solid bodies in Inventor Fusion.

Extrude

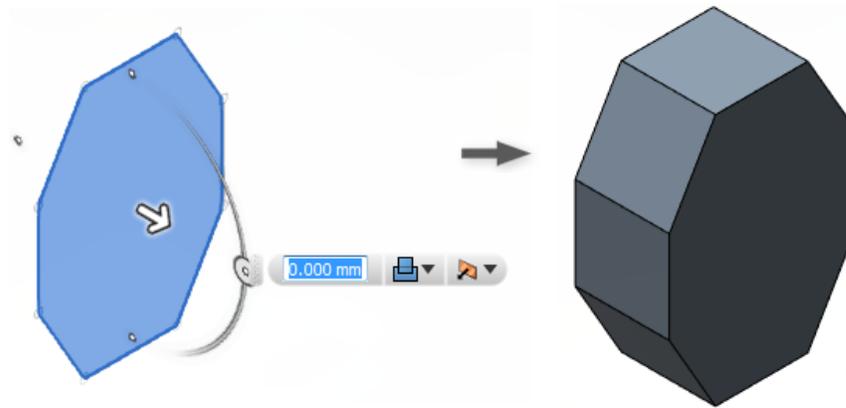


Extruded features are building blocks for creating and modifying models. You can create both solid and surface extrusions.

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.

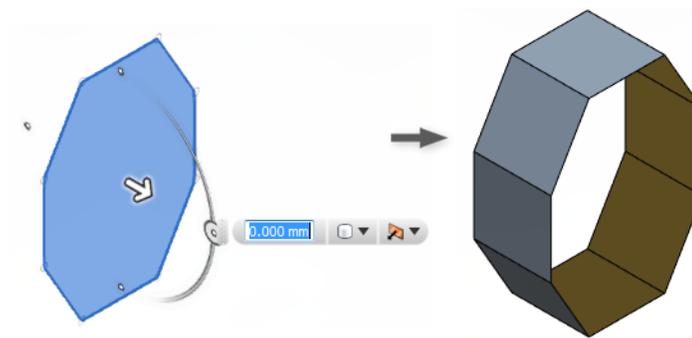
Create a solid extrusion



- 1 Switch the toolbar workspace to Solid.
- 2 Click Extrude in the Solid pull-down.
- 3 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.
- 4 Select the Direction Type
 - **One Side** Creates the extrusion in one direction.
 - **Two Side** 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.
- 5 Use the manipulators to set the height and taper, or enter values in the dialog box.
- 6 Select the Boolean Type
 - **Join** Adds material.

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

Create a surface extrusion



- 1 Switch the toolbar workspace to Surface.
- 2 Click Extrude in the Surface pull-down.
- 3 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.
- 4 Select the Direction Type
 - **One Side** Creates the extrusion in one direction.
 - **Two Side** 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.
- 5 Use the manipulators to set the height and taper, or enter the values.
- 6 Select the Boolean Type
 - **New Body** Creates the extrusion as a new surface body.
 - **New Component** Creates a child component in the active component.

Extrude dialog box

Profile

Enables the selection of sketch profiles.

Along Distance

Specifies the distance to extrude.

Against Distance

Available for Two Side extrusions. Specifies the second extrusion direction.

Taper Angle

Specifies the angle to taper the extrusion.

Direction Type

Specifies the method to control the size of the extrusion

- **One Side** Creates the extrusion in one direction.
- **Two Side** 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.

Boolean Type

Specifies the affect the extrusion has on the model.

- **Join** (solid only) Adds material.
- **Cut** (solid only) Removes material.
- **Intersect** (solid only) Removes all material from the solid that does not overlap the new feature.
- **New Body** Creates the extrusion as a new body.
- **New Component** Creates a child component in the active component.

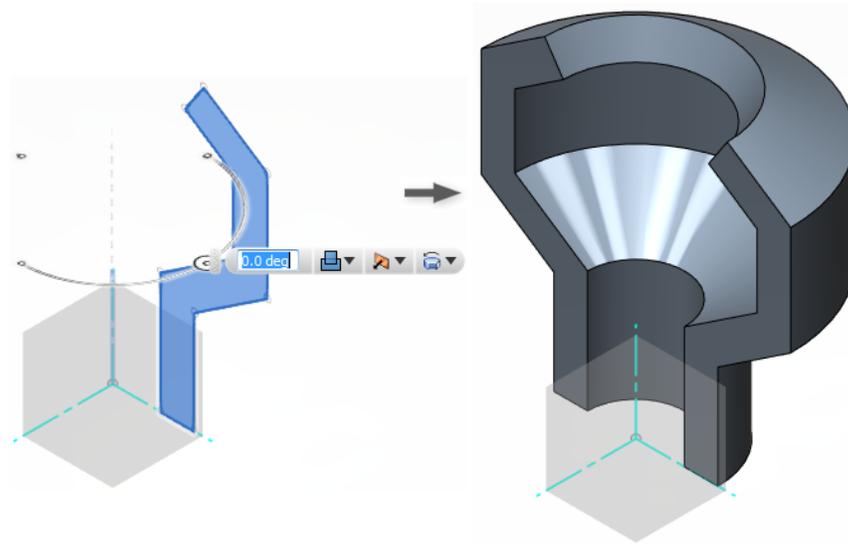
Revolve

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.

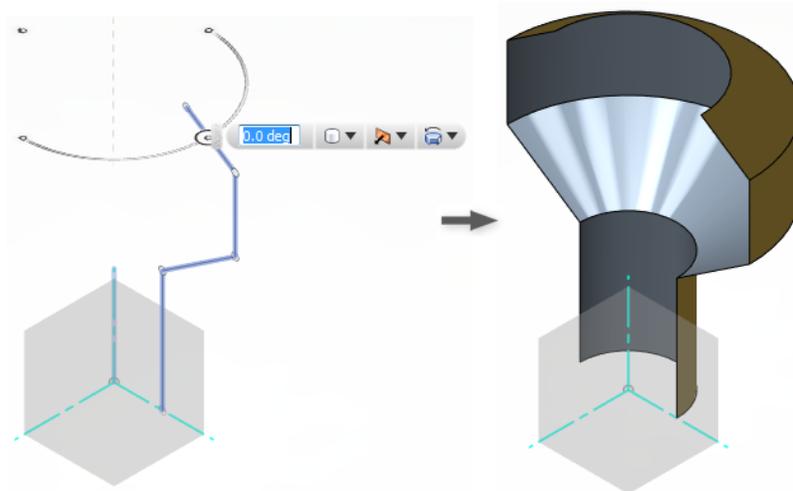
Create a revolved solid



- 1 Switch the toolbar workspace to Solid.
- 2 Click Revolve in the Solid pull-down.
- 3 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.
- 4 Set the selection to Axis in the dialog box then select the axis for the revolution.
- 5 Select the Direction Type
 - **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 Type
 - **Angle**
 - **Full**
- 7 Specify the Along Angle and Against Angle by dragging the manipulator or entering a distance in the dialog box.
- 8 Select the Boolean Type
 - **Join** Adds material.
 - **Cut** Removes material.
 - **Intersect** Removes all material from the solid that does not overlap the new feature.
 - **New Body** Creates the revolution as a new solid body.
 - **New Component** Creates a child component in the active component.
- 9 Right-click then select OK to create the revolve.

Create a revolved surface



- 1 Switch the toolbar workspace to Surface.
- 2 Click Revolve in the Surface pull-down.

- 3 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.
- 4 Set the selection to Axis in the dialog box then select the axis for the revolution.
- 5 Select the Direction Type
 - **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 Type
 - **Angle**
 - **Full**
- 7 Specify the Along Angle and Against Angle by dragging the manipulator or entering a distance in the dialog box.
- 8 Select the Boolean Type
 - **New Body** Creates the revolution as a new surface body.
 - **New Component** Creates a child component in the active component.
- 9 Right-click then select OK to create the revolve.

Revolve dialog box

Profile

Enables the selection of sketch profiles.

Axis

Enables the selection of the axis of revolution.

Along Angle

Specifies the angle to revolve.

Against Angle

Specifies the opposite angle for Two Side revolutions.

Direction Type

Specifies the method to control the size of the revolution

- **One Side** Creates the revolution in one direction.
- **Two Side** Creates the revolution in both directions. Each direction can have a different angle.
- **Symmetric** Creates the revolution in both directions. Each direction has the same angle.

Type

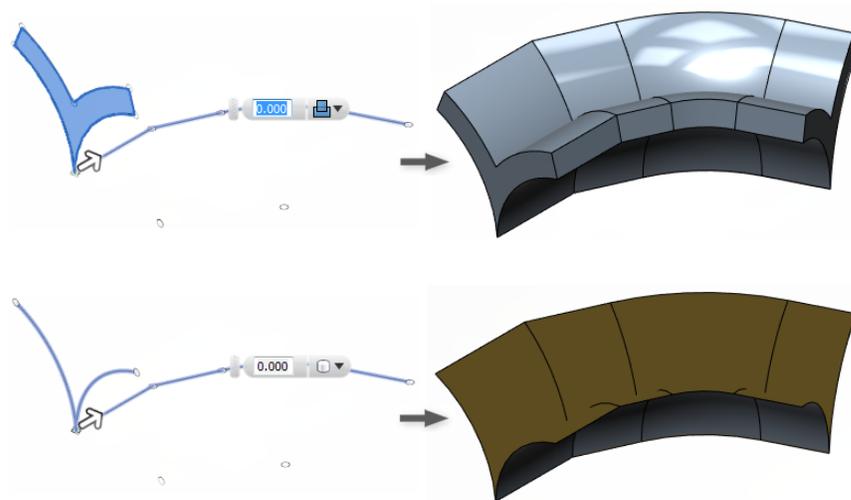
Select whether to specify an angle to revolve or revolve a full 360 degrees.

Boolean Type

Specifies the affect the extrusion has on the model.

- **Join** (solid only) Adds material.
- **Cut** (solid only) Removes material.
- **Intersect** (solid only) Removes all material from the solid that does not overlap the new feature.
- **New Body** Creates the revolution as a new body.
- **New Component** Creates a child component in the active 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 Switch the toolbar workspace to Solid.
- 2 Expand the Solid pull-down then click Sweep.
- 3 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.
- 4 Set the selection to Path then select one or more sketch entities or model edges for the path.

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

- 5 Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path in the dialog box.
- 6 Select the Orientation type:
 - **Perpendicular** keeps the region perpendicular to the path.
 - **Parallel** keeps the region parallel to the region sketch.
- 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 Body** Creates the sweep as a new solid body.
 - **New Component** Creates a child component under the active component.
- 8 Right-click then select OK to create the sweep.

Create a swept surface

- 1 Switch the toolbar workspace to Surface.
- 2 Expand the Surface pull-down then click Sweep.
- 3 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.
- 4 Set the selection to Path then select one or more sketch entities or model edges for the path.
- 5 Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path in the dialog box.
- 6 Select the Orientation type:
 - **Perpendicular** keeps the region perpendicular to the path.
 - **Parallel** keeps the region parallel to the region sketch.
- 7 Select the Boolean Option:
 - **New Body** Creates the sweep as a new solid body.
 - **New Component** Creates a child component under the active component.
- 8 Right-click then select OK to create the sweep.

Sweep dialog box

Profile

Enables the selection of sketch profiles.

Path

Enables the selection of sweep path.

Along Value

Specifies the percentage value of the sweep.

Full Path

Enable the Full Path option to sweep the profile along the entire path.

Orientation

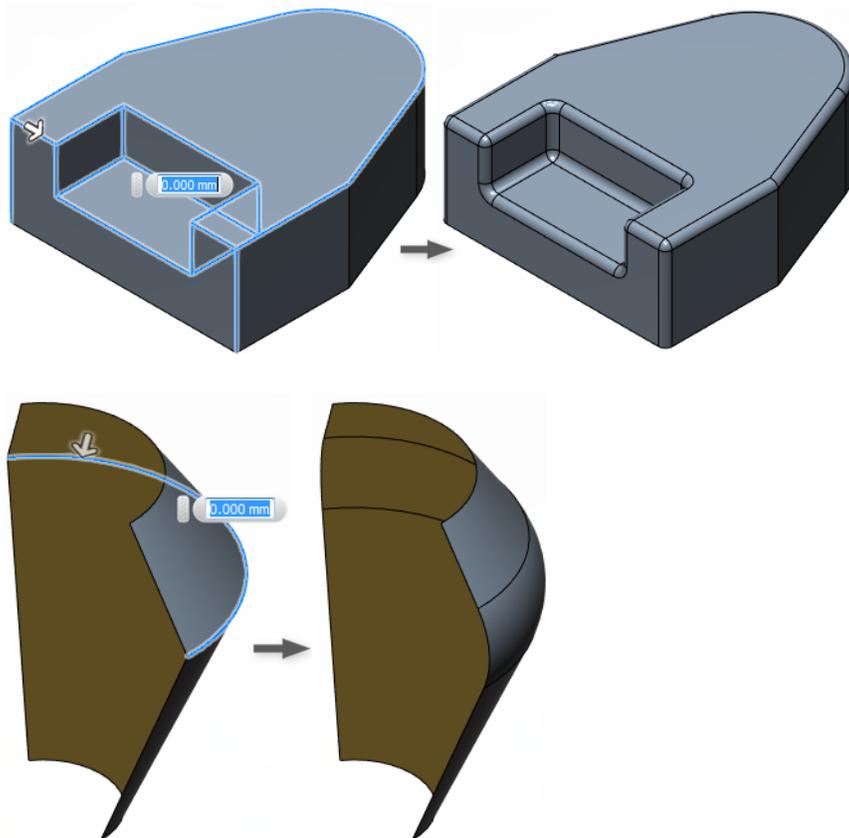
- **Perpendicular** keeps the region perpendicular to the path.
- **Parallel** keeps the region parallel to the region sketch.

Boolean Type

Specifies the affect the sweep has on the model.

- **Join** (solid only) Adds material.
- **Cut** (solid only) Removes material.
- **Intersect** (solid only) Removes all material from the solid that does not overlap the new feature.
- **New Body** Creates the sweep as a new body.
- **New Component** Creates a child component in the active component.

Fillet



Fillet features 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 then selecting Edit Fillet, or by using Press/Pull.

Create a fillet

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid pull-down or Surface pull-down then select Fillet.
- 3 Deselect the Tangent Chain if you do not want tangentially connected edges.
- 4 Select the edges to fillet.
- 5 Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
- 6 Deselect the Corner Option if you want a smooth transition between edges. The Corner Option produces corners like a milling machine or a router.
- 7 Right-click then select OK to create the fillet.

Fillet dialog box

Edges

Enables the selection of edges to fillet.

Radius

Specifies the radius for the fillet.

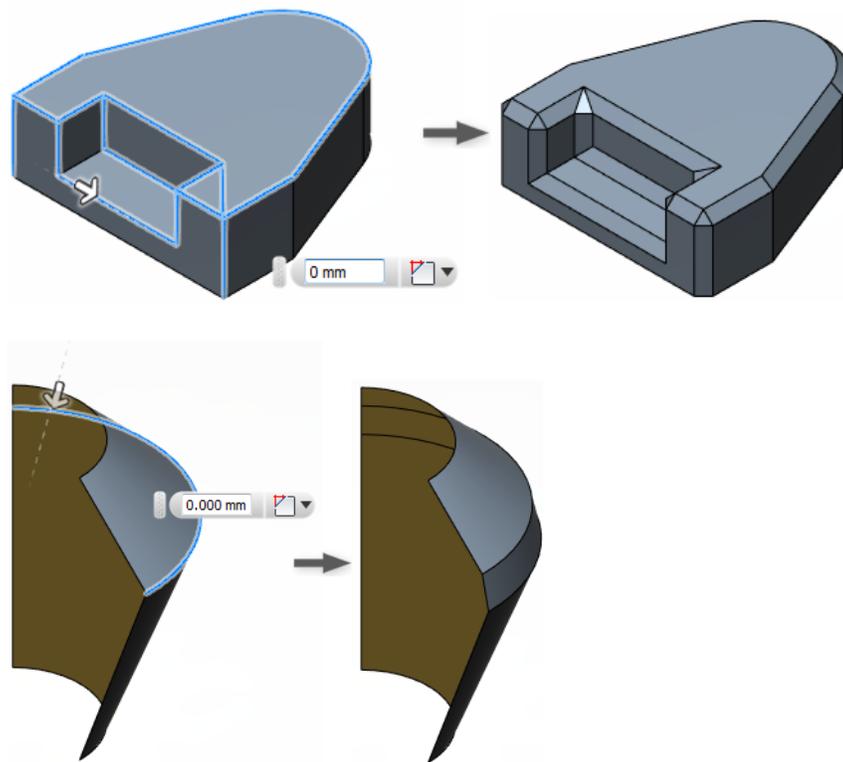
Corner Option

Creates rolling ball corners when enabled.

Tangent Chain

Specifies whether to include tangentially connected edges or select individual edges.

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 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid pull-down or Surface pull-down then click Chamfer.
- 3 Deselect Tangent Chain if you do not want tangentially connected edges.
- 4 Select the edges to chamfer

- 5 Select the type of chamfer to apply.
 - **Equal distance** Specify a single distance for both sides of the chamfer.
 - **Two distances** Specify a distance for each face of the chamfer.
 - **Distance and angle** Specify a distance and angle to create the chamfer.
- 6 Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
- 7 Right-click then select OK to create the chamfer.

Chamfer dialog box

Controls the settings of the Chamfer command.

Edges

Enables the selection of edges to chamfer.

First Distance

Specifies the first distance for the chamfer.

Second Distance

Available for two distances chamfers. Specifies the second distance for the chamfer.

Angle

Available for distance and angle chamfers. Specifies the angle for the chamfer.

Tangent Chain

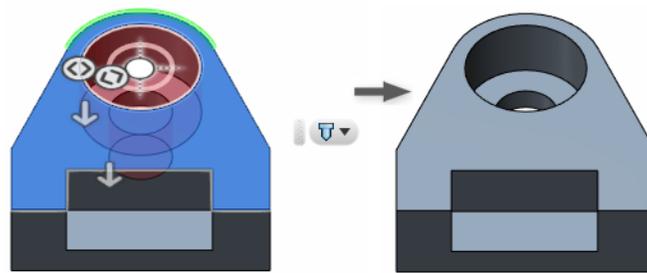
Specifies whether to include tangentially connected edges or select individual edges.

Chamfer Type

The chamfer type specifies the method to control the size of the chamfer.

- **Equal distance** Specify a single distance for both sides of the chamfer.
- **Two distances** Specify a distance for each face of the chamfer.
- **Distance and angle** Specify a distance and angle to create the chamfer.

Hole



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

You can reference other model geometry to set the location, diameter, and depth of the hole. When the center mark 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 Switch the toolbar workspace to Solid.
- 2 Click the Hole command in the Solid pull-down.
- 3 Select the face to position the hole.
- 4 Use the center mark to move the hole location on the face.
 - Click a sketch point or an end point to snap to that location.
 - Click a circle, arc, or ellipse to snap to the center.
 - Click a line or edge to add a transient dimension.
- 5 Click the manipulator on the edge of the hole and drag it to change the hole diameter or enter a value in the dialog box.

NOTE You can select the manipulator then select a cylindrical edge to set the diameter.

- 6 Change the tip angle in the dialog box.
- 7 Select the Hole Type from the drop-down.
 - Simple

- Counterbore
 - Countersunk
- 8 Drag the arrow manipulator to set the depth or enter a value in the dialog box.
 - 9 For a Counterbore hole, use the second arrow and ring to set the depth and diameter or enter values in the dialog box.
 - 10 For a Countersunk hole, use the second ring to set the diameter or enter a value in the dialog box. Change the angle in the dialog box.
 - 11 Right-click the select OK to create the hole.

Hole dialog box

Hole Type

Select Simple, Counterbore, or Countersunk.

Face

Enables the selection of the face to start the hole.

Depth

Specifies the depth of the hole.

Diameter

Specifies the diameter of the hole.

Tip Angle

Specifies the angle of the drill point for the hole.

Counterbore Depth

Specifies the depth of the counterbore portion of the hole.

Counterbore Diameter

Specifies the diameter of the counterbore portion of the hole.

Countersunk Diameter

Specifies the diameter of the countersunk portion of the hole.

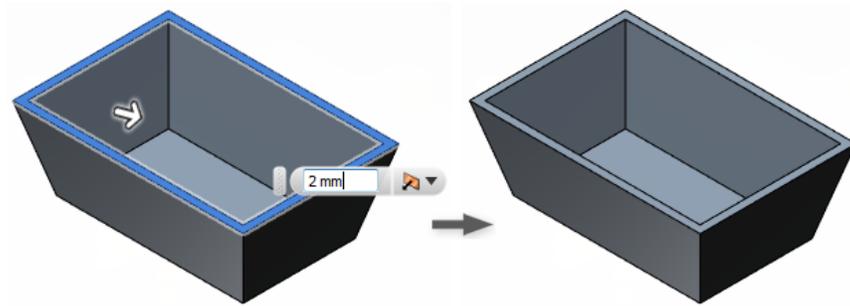
Countersunk Angle

Specifies the angle of the countersink.

Flip Direction

Reverses the direction of the hole.

Shell



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

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 Switch the toolbar workspace to Solid.
- 2 Expand the Solid pull-down then click Shell.
- 3 Select one or more faces to remove from a solid. You can also select the solid without selecting faces.
 - Optionally, clear the check box for Tangent Chain to prevent automatic selection of tangent faces.
- 4 Select 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.
- 5 Drag the manipulator to set the Inside Thickness and Outside Thickness, or enter values in the dialog box.
- 6 Right-click then select OK to create the shell.

Shell dialog box

Faces

Enables the selection of faces to remove from the solid.

Inside Thickness

Specifies the wall thickness inside the solid.

Outside Thickness

Specifies the wall thickness outside the solid.

Direction

Select the direction for the wall thickness.

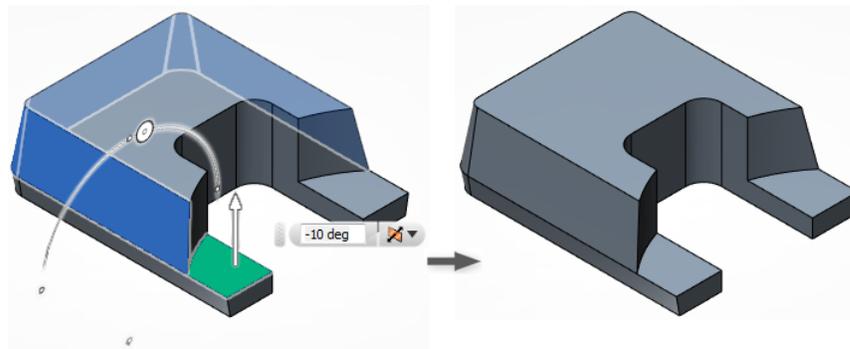
- **Inside** Offsets the faces toward the interior of the part.
- **Outside** Offsets the faces towards the exterior of the part.
- **Both** Offsets the faces towards the interior and the exterior of the part.
You can have different values for the offsets.

Tangent Chain

Specifies whether to include tangentially connected faces or select individual faces to remove.

Draft

The Draft command is used to modify one or more component bodies by creating angled faces, with respect to a neutral plane.



The Draft 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.

Apply draft to a solid

- 1 Switch the toolbar workspace to Solid.
- 2 Expand the Solid pull-down then click Draft.
- 3 Select a neutral plane around which faces are drafted.
- 4 Select the faces to be drafted.
- 5 Select the draft type:
 - **One Side** Specify a single draft angle.
 - **Two Side** Specify two draft angles; one above the neutral plane and one below the plane.
 - **Symmetric** Specify a single draft angle that is applied above and below the neutral plane.
- 6 Drag or enter a precise value into the angle entry box.
- 7 Click OK to complete the command.

Draft dialog box

Plane

Enables the selection of the neutral plane.

Faces

Enables the selection of the faces to draft.

Angle1

Specifies the draft angle.

Angle2

Available for Two Side direction type. Specifies the second draft angle.

Tangent Chain

Specifies whether to include tangentially connected edges or select individual edges.

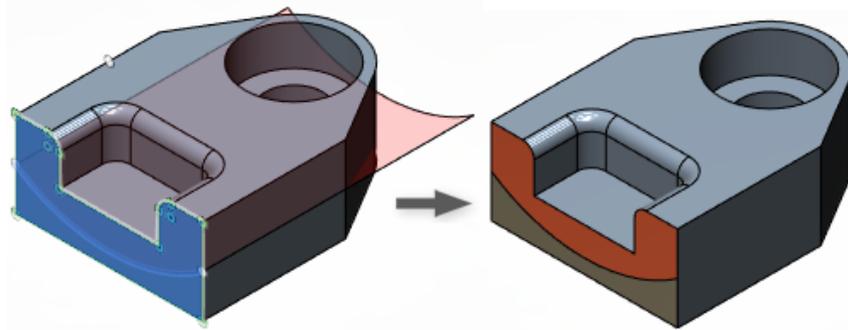
Flip Direction

Click to flip the direction of the draft.

Direction Type

- **One Side** Specify a single draft angle.
- **Two Side** Specify two draft angles; one above the neutral plane and one below the plane.
- **Symmetric** Specify a single draft angle that is applied above and below the neutral plane.

Split Face



Use Split Face to divide faces on a surface or solid. The faces can be any face on a solid body or surface body. The Splitting Tool can be a work plane, surface, or sketch geometry. Sketch geometry is projected onto the target faces.

Faces are typically split to add draft, delete an area, or to create new features.

Split a face

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid or Surface pull-down then select Split Face.
- 3 Select the faces to split.
- 4 Click Splitting Tool in the dialog box then select the intersecting work plane, surface, or sketch geometry.
- 5 Right-click then select OK.

Split Face dialog box

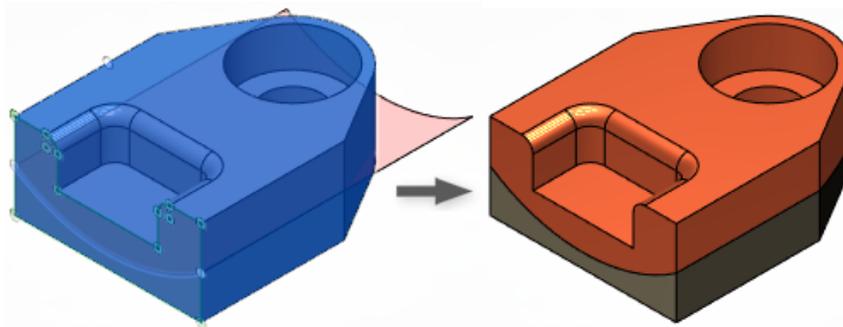
Faces to Split

Enables the selection of faces to split.

Splitting Tool

Enables the selection of a surface, work plane, or sketch geometry to divide the bodies.

Split Body



Use Split Body to divide a single body into two bodies. The bodies can be solid bodies or surface bodies. The Splitting Tool can be a work plane, surface, or sketch geometry. Sketch geometry is projected onto the target faces.

Split bodies

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid or Surface pull-down then select Split Body.
- 3 Select the bodies to split.
- 4 Click Splitting Tool in the dialog box then select the intersecting work plane, surface, or sketch geometry.
- 5 Right-click then select OK.

Split Body dialog box

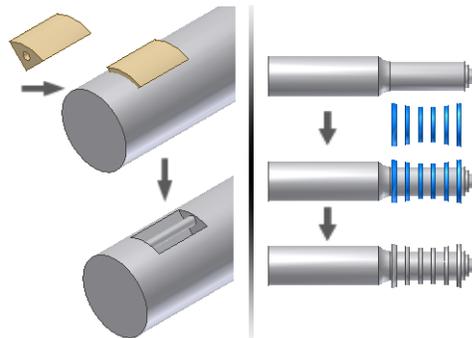
Body to Split

Enables the selection of bodies to split.

Splitting Tool

Enables the selection of a surface, work plane, or sketch geometry to divide the bodies.

Combine



The Combine command joins, cuts, or intersects the selected bodies. When you use the Combine command, the Target Body is the body or component being acted upon, and the Tool Bodies are the bodies that change the model. You can select more than one tool body to use in a Combine operation.

The default behavior is to delete the tool bodies 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

- 1 Switch the toolbar workspace to Solid.
- 2 Expand the solid pull-down the select Combine.
- 3 Select the Target Body.
- 4 Select the Tool Bodies that modify the target.
- 5 Select the Boolean Option.
 - a **Join** combines the target and tools into one solid.
 - b **Subtract** removes the overlapping material from the target.
 - c **Intersect** keeps only the overlapping material.

- 6 Click New Component to create a component with the results.
- 7 Click Keep Tools to retain the tool solids.
- 8 Right-click then select OK to create the combined solid.

Combine dialog box

Target Body

Enables the selection of the body being acted upon.

Tool Bodies

Enables the selection of the bodies that change the model.

Boolean Operation

Select the affect the tool bodies have on the target body.

- a **Join** combines the target and tools into one solid.
- b **Subtract** removes the overlapping material from the target.
- c **Intersect** keeps only the overlapping material.

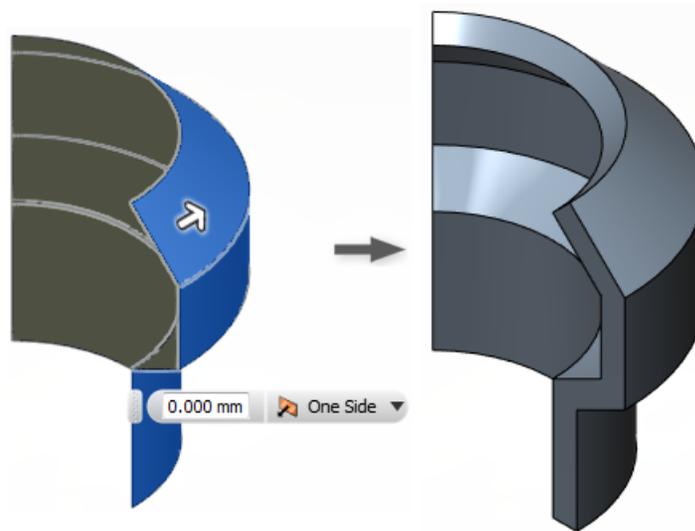
New Component

Creates a new component with the results.

Keep Tools

Retains the tool bodies after the combine results.

Thicken



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

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

Thicken a face, surface, or quilt

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid pull-down or Surface pull-down then click Thicken.
- 3 Select one or more faces, surfaces, or quilts.
- 4 Select the Direction Type
 - **One Side** Thickens the faces in one direction.
 - **Symmetric** Thickens the faces in both directions. Each direction has the same distance.
- 5 Drag the arrow to set the offset, or enter a value in the dialog box.
- 6 Select the Boolean Type
 - **Join** Adds material.
 - **Cut** Removes material.

- **Intersect** Removes all material from the solid that does not overlap the new feature.
- **New Body** Creates the extrusion as a new body.
- **New Component** Creates a child component in the active component.

7 Right-click then select OK to thicken the faces.

Thicken dialog box

Faces

Enables the selection of faces, surfaces, or quilts.

Thicken

Specifies the distance to thicken the selected faces.

Direction Type

- **One Side** Thickens the faces in one direction.
- **Symmetric** Thickens the faces in both directions. Each direction has the same distance.

Boolean Type

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

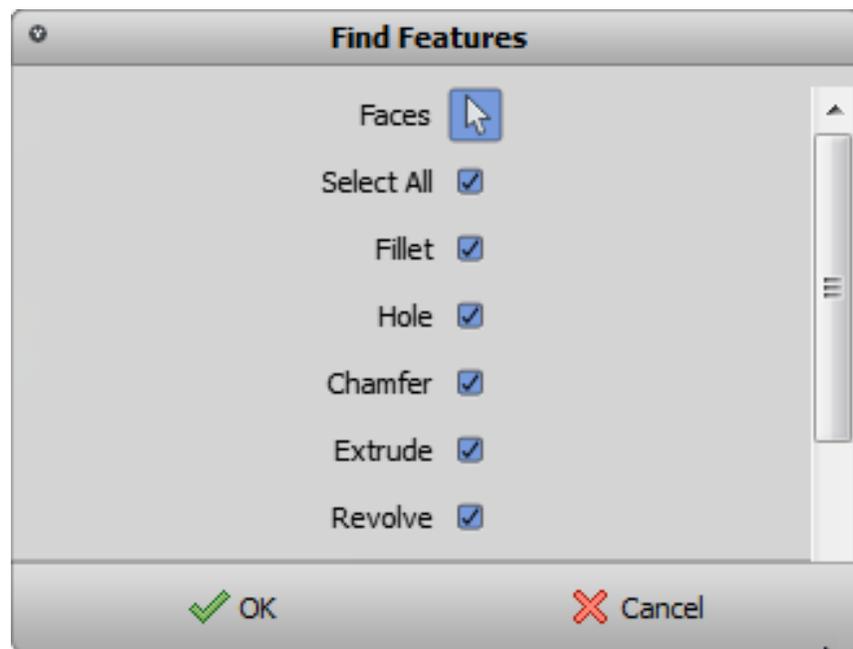
Find Features

Find Features, also called feature recognition, is a process to extract design feature information from a solid model. Find Features supports:

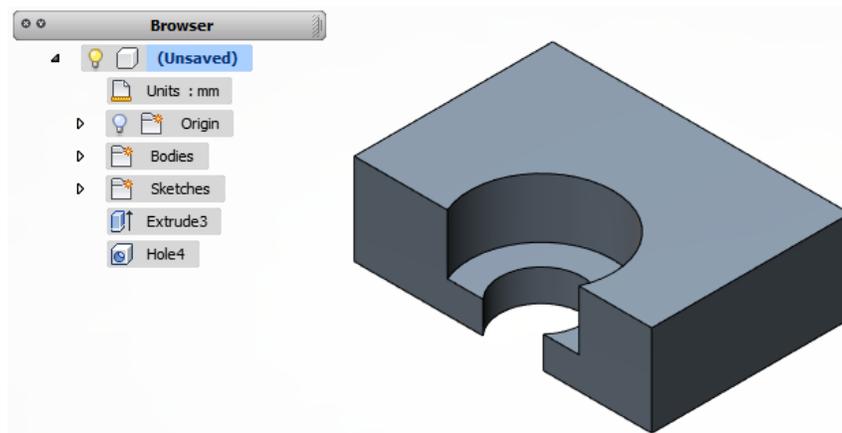
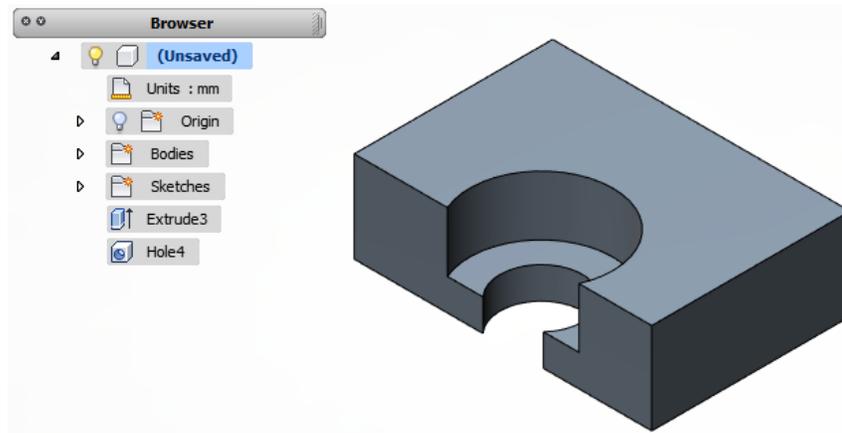
- Fillet
- Hole
- Chamfer
- Extrude

- Revolve
- Mirror
- Rectangular Pattern
- Circular Pattern

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.



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.



To find features in a model

- 1 Switch the toolbar workspace to Solid.
- 2 Expand the Solid pull-down then click Find Features.
- 3 Select the faces or bodies to analyze.
- 4 Select the features to create:
 - Fillet
 - Hole
 - Chamfer
 - Extrude

- Revolve
 - Mirror
 - Rectangular Pattern
 - Circular Pattern
- 5 Right-click then select OK.

Find Features dialog box

Faces

Enables the selection of faces or bodies to analyze.

Select All

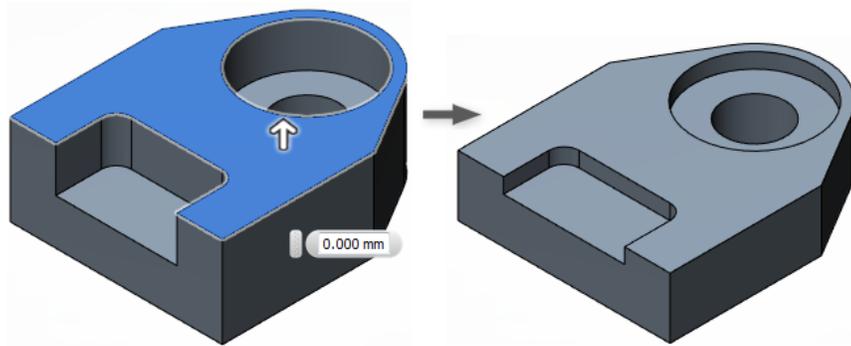
Selects all the available features.

- **Fillet**
- **Hole**
- **Chamfer**
- **Extrude**
- **Revolve**
- **Mirror**
- **Rectangular Pattern**
- **Circular Pattern**

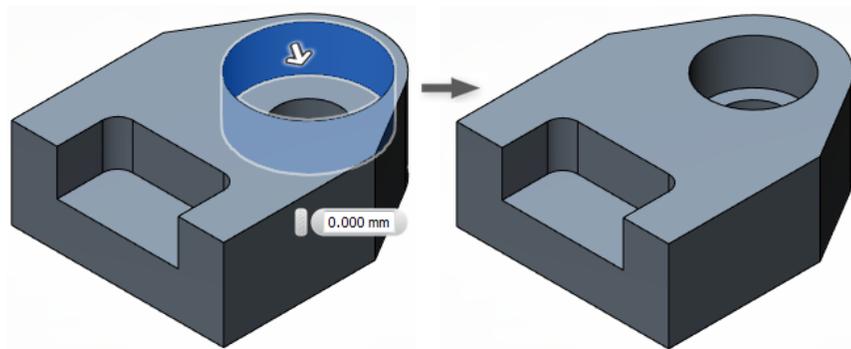
Modify

Use the Modify commands to make changes to solids, surfaces, and components in Inventor Fusion.

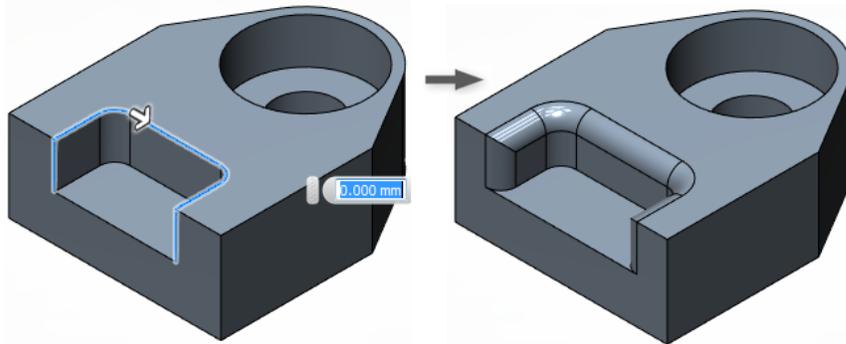
Press Pull



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.



Use Press Pull to edit fillet, holes, and chamfers.



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.

How to modify geometry using Press Pull

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Press Pull in the Modify pull-down.
- 3 Select the faces to modify.
- 4 Drag the arrow manipulator or enter a value in the field in the dialog box.
- 5 Right-click then select OK to accept the change.

Press Pull dialog box

Selection

Enables the selection of faces or edges.

Distance/Radius

Specifies the distance to move the face or the radius of the arc face.

Tweak Faces

Use Tweak Face to modify the 3D model. Tweak face moves the selected faces and trims or extends adjacent faces.

Edit geometry using Tweak

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Tweak Faces.
- 3 Select the faces to tweak.
- 4 Drag a manipulator or enter values in the dialog box.
- 5 Right-click and select OK.

NOTE You can snap to existing geometry when tweaking faces. Drag a manipulator to activate it then select the geometry to snap to.

Tweak dialog box

Faces

Enables the selection of faces to tweak.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Scale Command

Enlarges or reduces selected bodies based on a specified scale factor.

Scale an object

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Scale.
- 3 Select the body that you wish to scale.
- 4 Click Point in the dialog box then select a reference point to scale around.
- 5 Select Uniform or Non Uniform for the Scale Type
- 6 Set the Scale Factor:
 - For Uniform scale, drag the arrow manipulator or enter a value for Scale Factor.
 - For Non Uniform scale, drag the arrow manipulators or enter values for the X Distance, Y Distance, and Z Distance.
- 7 Right-click then select OK to accept the changes.

Scale dialog box

Scale Type

- **Uniform** Uses a single scale factor to scale the object.
- **Non Uniform** Use a different scale factor to scale the object in the X, Y, and Z direction.

Bodies

Enables the selection of bodies to scale.

Point

Enables the selection of a reference point to scale around.

Scale Factor

For Uniform scale, specifies the value for the scale.

X Distance

For Non Uniform, specifies the value to scale in the X direction.

Y Distance

For Non Uniform, specifies the value to scale in the Y direction.

Z Distance

For Non Uniform, specifies the value to scale in the Z direction.

Move

Use the Move command to move solid bodies, surface bodies, and work geometry.

Move bodies

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Move.
- 3 Select the bodies, or work geometry to move.
- 4 Drag a manipulator or enter values in the dialog box.
- 5 Right-click and select OK.

Move dialog box

Selection

Enables the selection of bodies and work geometry to move.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Move Components

Use the Move Components command to change the position of component instances.

Move components

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Move Components.
- 3 Select the components to move.
- 4 Drag the manipulators or enter values in the dialog box.
- 5 Right-click then select OK.

Move Components dialog box

Components

Enables the selection of components to move.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

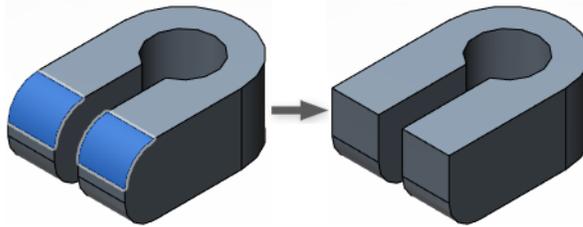
Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

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.

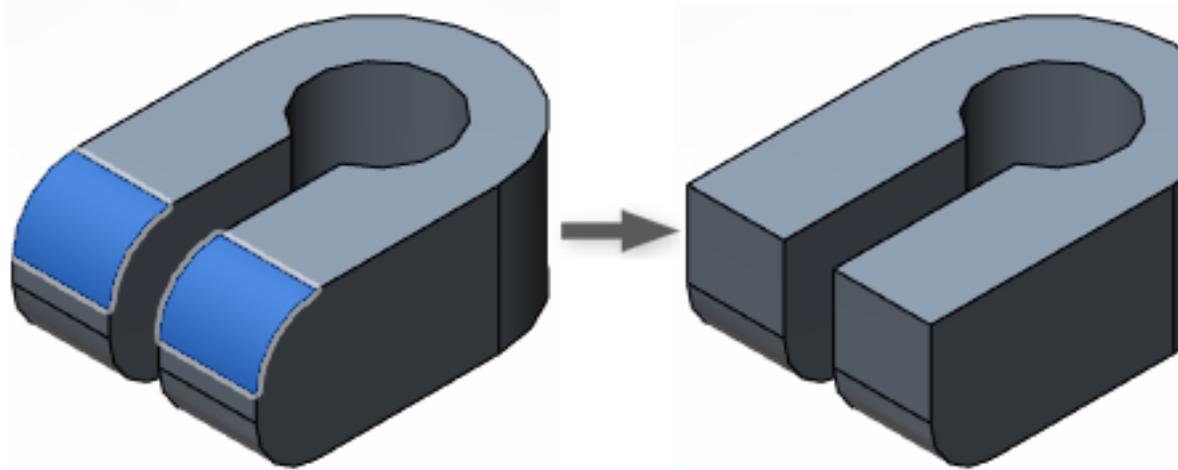
If you want to remove a face, use the Delete Face command on the Surface pull-down of the Surface toolbar workspace.

Delete an object from a model

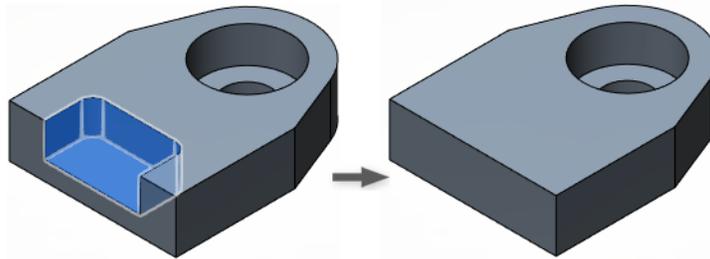
- 1 Switch the toolbar workspace to Solid.
- 2 Select a face, solid, sketch, feature, sketch entity, or construction entity.
- 3 Expand the Solid pull-down then click Delete. The selection is deleted, and the model is healed if necessary. An error displays if the model cannot be healed.

Delete face examples

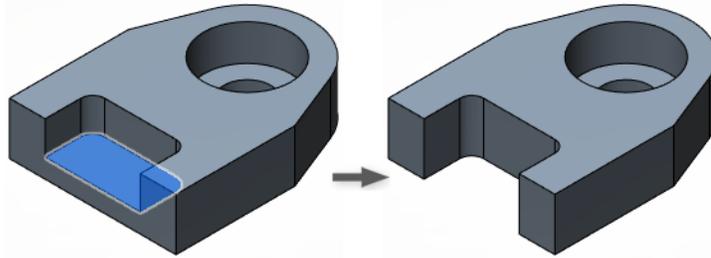
The delete command modifies the solid to heal the opening left by the deleted face. The change depends on model geometry and the selected faces. This table has several examples of delete face operations.



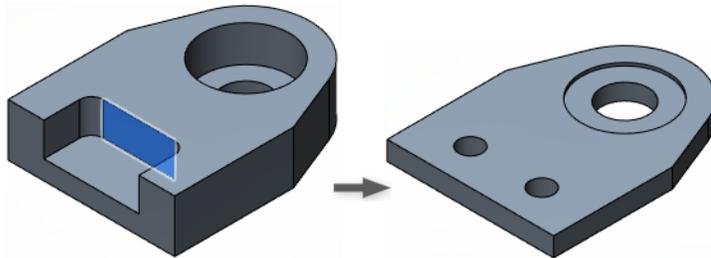
The selected face is tangent to the planar faces. The planar faces are extended to intersect.



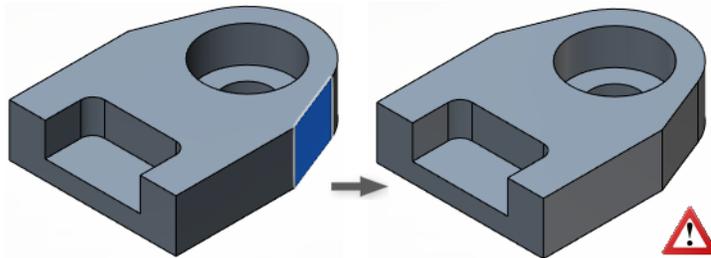
All faces in the pocket are deleted. The top and end faces are extended.



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.

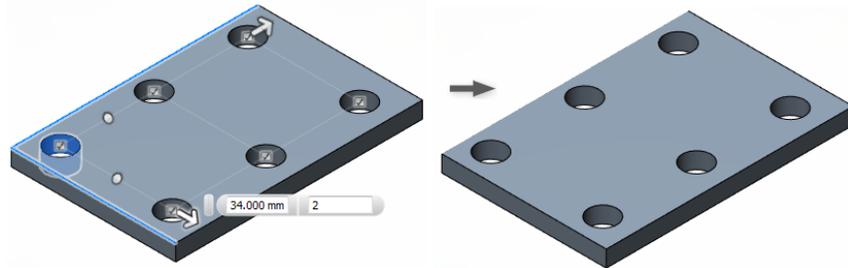


None of the adjacent can be extended to heal the model, so an error displays.

Pattern

Use the Pattern commands to pattern or mirror geometry in Inventor Fusion.

Rectangular Pattern



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.

You can pattern faces, features, or bodies.

Create rectangular patterns

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Rectangular Pattern in the Pattern pull-down.
- 3 In the graphics area or the browser, select the faces, features, or bodies to pattern.
- 4 Click Directions, and select a linear edge or work axis. Perpendicular manipulators display.
- 5 For a non-perpendicular pattern, select a second linear edge or work axis.
- 6 Use the arrows to set the U Distance and V Distance or enter values in the fields.
- 7 Use the dot manipulator to set the U Grid Entries and V Grid Entries or enter values in the fields.
- 8 Click a check mark to suppress an occurrence.
- 9 Right-click the select OK to create the pattern.

Rectangular Pattern dialog box

Objects

Enables the selection of objects to pattern. Valid objects are faces, features, or bodies.

Directions

Enables the selection of edges to control the U and V directions.

Suppress

Displays check boxes in the graphics window at each occurrence. Uncheck a box at an occurrence to suppress that occurrence from the pattern.

U Grid Entries

Specifies the number of occurrences along the first direction.

U Distance

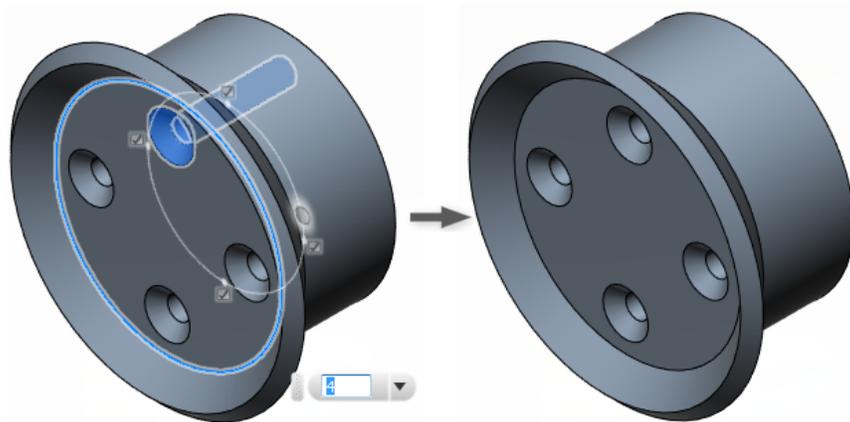
Specifies the overall distance in the first direction. Occurrences are spaced evenly along the distance.

V Grid Entries

Specifies the number of occurrences along the second direction.

V Distance

Specifies the overall distance in the second direction. Occurrences are spaced evenly along the distance.

Circular Pattern

The circular pattern command copies selected objects around a selected axis. You can pattern faces, features, or bodies.

The axis for a circular pattern can be a sketch line, a linear edge, a work axis, a cylindrical face, or a circular edge.

Create a circular pattern

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Pattern pull-down then click Circular Pattern.
- 3 In the graphics area or the browser, select the faces, features, or bodies to pattern.
- 4 Click Circular Axis in the dialog box then select a sketch line, linear edge, a work axis, a cylindrical face, or a circular edge.
- 5 Use the ball manipulator to set the number of occurrences or enter a value in the field.
- 6 Click a check mark to suppress an occurrence.
- 7 Set the Circular Type to Full or Angle.
 - **Full** The copies are evenly spaced around a complete circle.
 - **Angle** The copies are evenly spaced around the specified angle.
- 8 If Circular Type is set to angle, enter the value in the Total Angle field.
- 9 Right-click the select OK to create the pattern.

Circular Pattern dialog box

Objects

Enables the selection of objects to pattern. Valid objects are faces, features, or bodies.

Circular Axis

Enables the selection of the axis to pattern around. Valid objects are a sketch line, linear edge, a work axis, a cylindrical face, or a circular edge.

Suppress

Displays check boxes in the graphics window at each occurrence. Uncheck a box at an occurrence to suppress that occurrence from the pattern.

Total Angle

Available when the Circular Type is set to Angle. Specifies the total angle to space the occurrences around.

Count

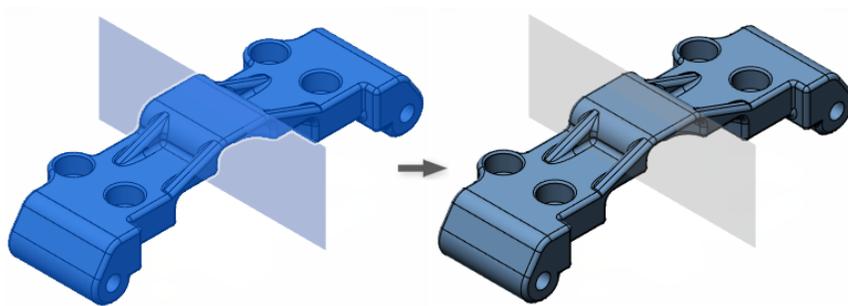
Specifies the total number of copies.

Circular Type

Controls how the copies are positioned around the pattern.

- **Full** The copies are evenly spaced around a complete circle.
- **Angle** The copies are evenly spaced around the specified angle.

Mirror



Mirrors the selected objects to the opposite side of a selected face or plane.

Valid object types are faces, features, or bodies.

Mirror objects

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Mirror in the Pattern pull-down.
- 3 In the graphics area or the browser, select the faces, features, or bodies to mirror.
- 4 Click Mirror Plane in the dialog box then select the plane or face to mirror around.
- 5 Right-click then click OK to create the mirror.

Mirror dialog box

Objects

Enables the selection of objects to mirror. Valid objects are faces, features, or bodies.

Mirror Plane

Enables the selection of the plane or face to mirror around.

Construction

Work features or construction 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.

Work Planes

A work plane is a construction feature that defines the parametric location of a sketch plane in 3D space. A work plane is useful when no planar face exists to use as a sketch plane, for example, when sketching on curved faces.

Create work planes

Inventor Fusion creates the following types of work planes. Most work planes 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.

Icon	Work Plane Type	Inputs
	Offset Plane	Select a work plane or planar model face and enter the distance.

Icon	Work Plane Type	Inputs
	Plane At Angle	Select a model edge, sketch line, or work axis, and enter the angle.
	Tangent Plane	Select a cylindrical face and enter the angle. The plane is created where you clicked on the cylinder.
	Midplane	Select two parallel planes or planar model faces. The work plane is created halfway between them.
	Plane Through Two Edges	Select two model edges, sketch lines, or work axes.
	Plane Through Three Points	Select three model, sketch, or work points.

Work Axes

A work axis is a construction feature that defines an axis in 3D space. A work axis is useful when no geometry exists to use as an axis.

Create work axes

Inventor Fusion creates the following types of work axes. Work axes are automatically created based on the geometry selections.

Icon	Work Axis Type	Inputs
	Axis Through Cylinder	Select a cylindrical face or a cone.
	Axis Perpendicular At Point	Select a model or work plane.
	Axis Through Two Planes	Select two model faces or work planes.
	Axis Through Two Points	Select two vertices, sketch points or work points.
	Axis Through Edge	Select a linear or circular edge on a model or a sketch profile.

Work Points

A work point is a construction feature that defines a point in 3D space. A work point can be projected onto a sketch and incorporated into dimension and constraint schemes.

Create work points

Inventor Fusion creates the following types of work points. Work points are automatically created based on the geometry selections.

Icon	Work Axis Type	Inputs
	Point at Vertex	Select a vertex, work point, or sketch point.
	Point Through Two Edges	Select two edges on a model or two models or sketch profiles that intersect.
	Point Through Three Planes	Select three model faces or work planes.
	Point at Center of Circle/Sphere	Select a circular edge or spherical surface.
	Point at Edge And Plane	Select a work plane or model face, or a sketch profile and a work axis or sketch line.

Select

Use the select commands to control selection sets in Inventor Fusion.

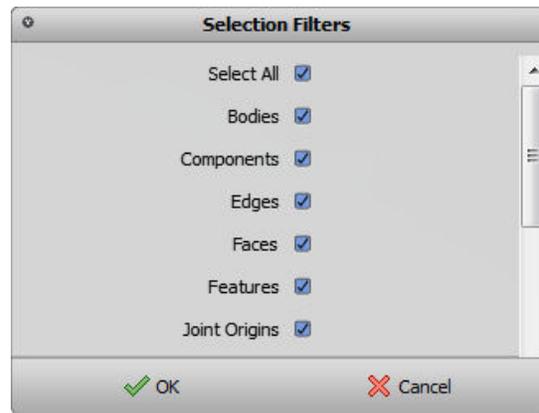
Select

Enables the selection of objects in the graphics window.

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.

Access the Selection Filters command from the Select pull-down in the Solid, Surface, or Assemble workspaces.



The selection filters control only selection in the graphics window. You can still select objects in the browser

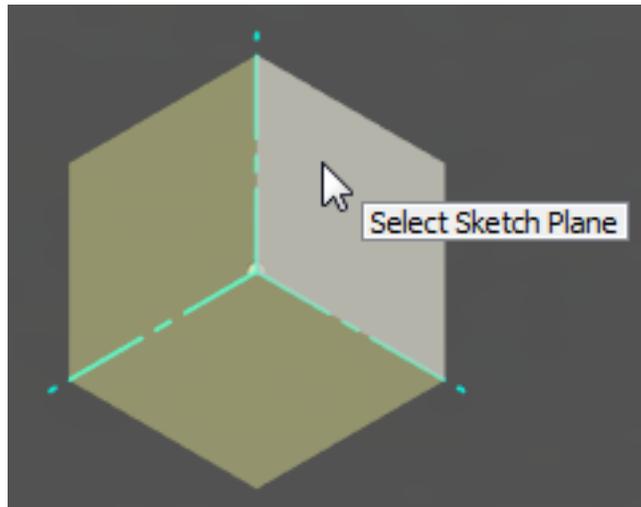
Surface workspace

Use the Surface workspace to create surface bodies in Inventor Fusion.

Sketch

Use the Sketch commands to create and modify 2D sketch geometry in Inventor Fusion.

Create Sketch



Creates a new sketch on the selected plane or face.

Create a sketch

- 1 Expand the Sketch pull-down then click Create Sketch.
- 2 Select the plane or face to create the sketch on. Valid selections are:
 - Origin planes
 - Work planes
 - Flat faces on geometry.

NOTE If you select an existing sketch entity or sketch node in the browser, that sketch is edited and a new sketch is not created.

Edit a sketch

There are two methods to edit a sketch.

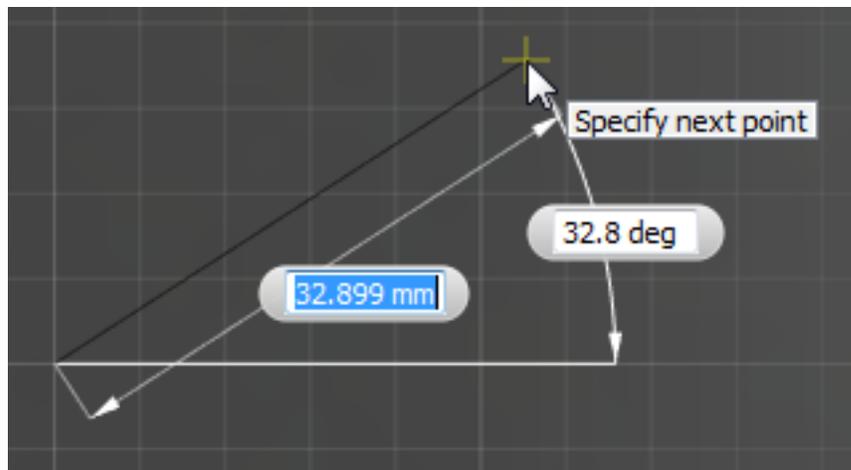
- 1 Right-click the sketch node in the browser then select Edit Sketch.
- 2 Start the Create Sketch command then select a sketch entity or the sketch node in the browser.

Line



The Line command creates a series of lines. A line segment is created each time you click.

Create lines



- 1 Click the Line command in the Sketch pull-down.
- 2 Click to place the starting point for the line.

- 3 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.
- 4 Continue to place points for additional line segments.
- 5 Right-click then select Cancel to end the command.

Create arcs



A tangent arc is created if you hold the left mouse button and drag while the line command is active.

- 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.
- 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.

Arc Three Point

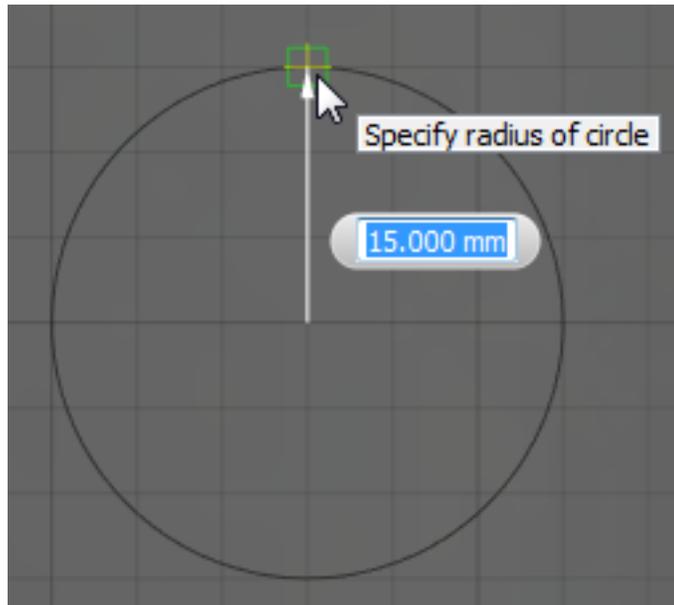


Creates an arc using three user-specified points. The three points define the position and size of the arc.

Create an arc

- 1 Expand the Sketch pull-down then click Arc Three Point.
- 2 Click in the graphics window to specify the first end point of the arc.
- 3 Click again to specify the second end point of the arc.
- 4 Click a third point to specify a point that lies along the arc.
- 5 Click again to begin creating a new arc or right-click then select Cancel to end the command.

Circle Center Radius



Creates a circle using two user-specified points. The points define the position of the center of the circle and the size of the circle.

Create a circle

- 1 Click Circle Center Radius on the Sketch pull-down.
- 2 Click in the graphics window to specify the center point of the circle.
- 3 To specify the radius of the circle:
 - Click a point in the graphics window.
 - Enter a value in the radius field then press <Enter>. Click a point to create the circle.
- 4 Click again to begin creating a new circle or right-click then select Cancel to end the command.

Ellipse

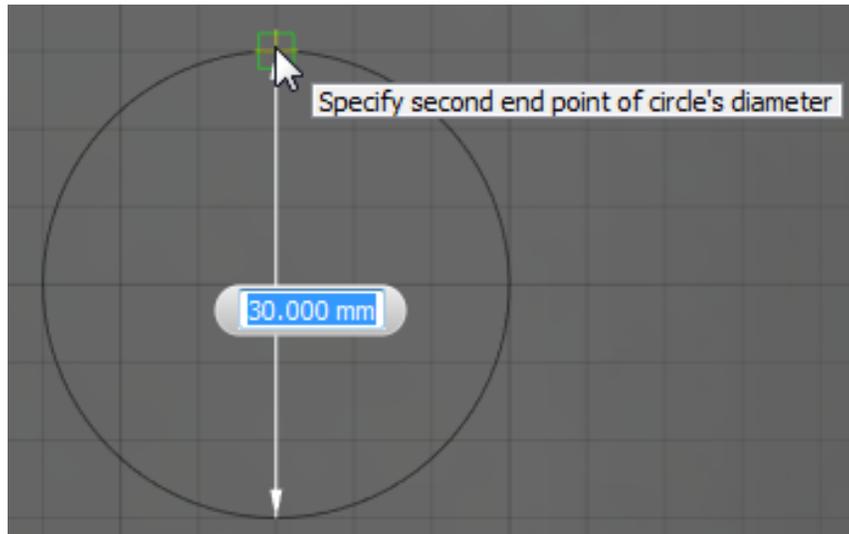


Creates an ellipse using three user-specified points. The points define the position and size of the ellipse.

Create an ellipse

- 1 Expand the Sketch pull-down then click Ellipse.
- 2 Click in the graphics window to specify the center point of the ellipse.
- 3 Click to specify the end point of the first axis of the ellipse.
- 4 Click a point to specify a point along the ellipse.
- 5 Click again to begin creating a new ellipse or right-click then select Cancel to end the command.

Circle Two Point

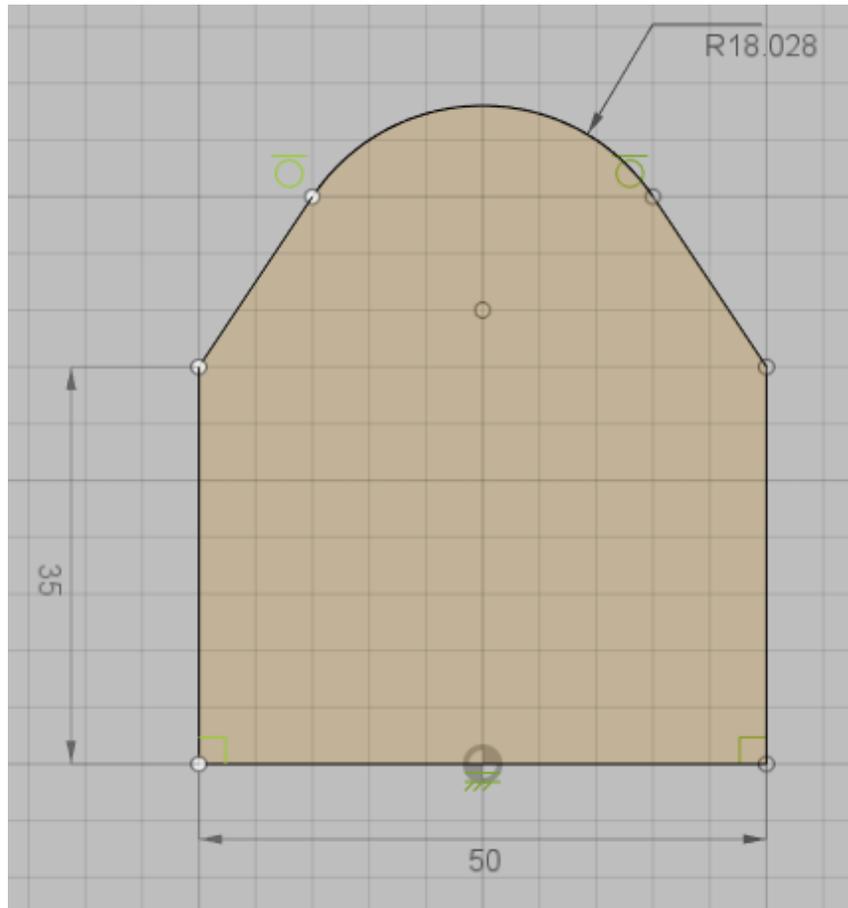


Creates a circle using two user-specified points. The points define the position and the size of the circle.

Create a circle

- 1 Expand the Sketch pull-down then click Circle Two Point.
- 2 Click in the graphics window to specify the first point of the circle's diameter.
- 3 To specify the diameter of the circle:
 - Click a point in the graphics window.
 - Enter a value in the diameter field then press <Enter>. Click a point to create the circle.
- 4 Click again to begin creating a new circle or right-click then select Cancel to end the command.

Sketch Dimension



Creates a new sketch on the selected plane or face.

Dimension sketch geometry

- 1 Expand the Sketch pull-down then click Sketch Dimension.
- 2 Select the geometry to dimension. Multiple geometries can be selected. For example, you can select two lines to create an angle dimension or a line and a point to get a distance.
- 3 Pick a point to place the dimension.

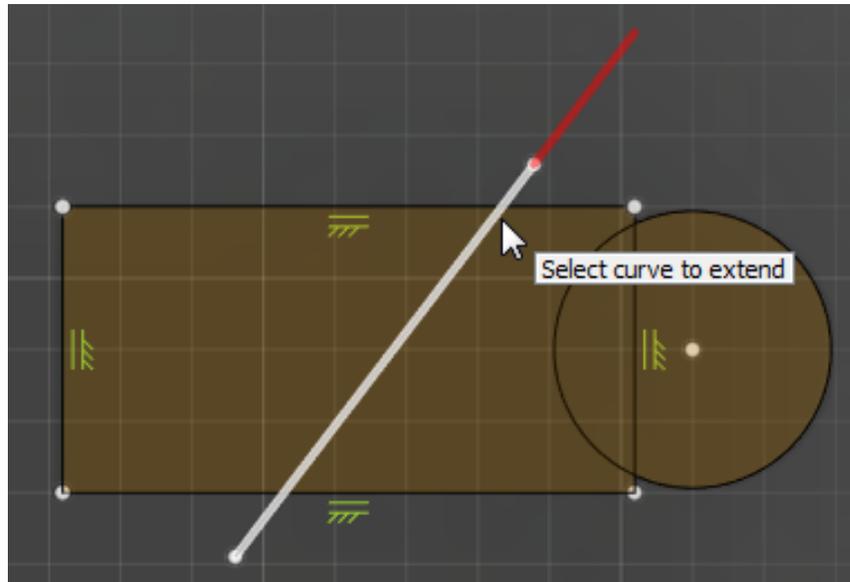
- 4 Select other geometry to continue dimensioning or right-click then select Cancel to end the command.

Extend



Extends the selected geometry to the next intersection of geometry. If no intersection exists, the geometry cannot be extended.

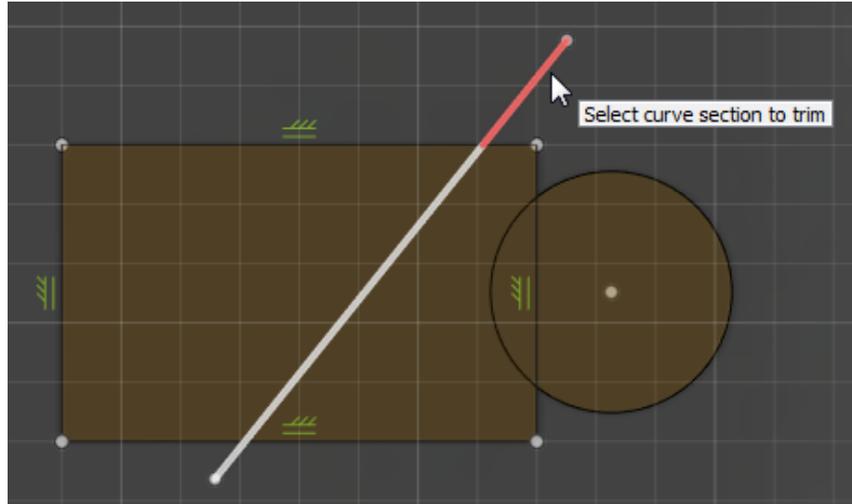
The Extend command will project existing geometry to extend to the theoretical intersection.



Extend an entity

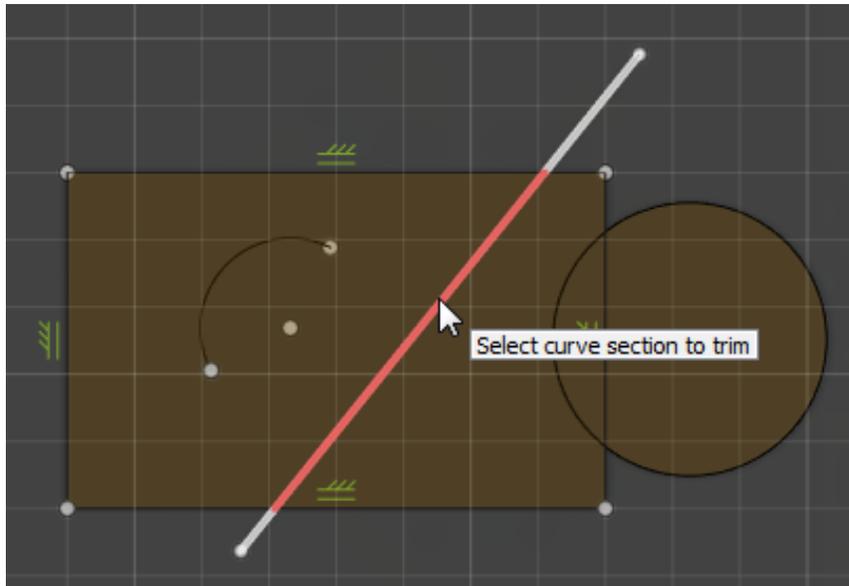
- 1 Expand the Sketch pull-down then select Extend.
- 2 Hover over sketch geometry to see a preview of the extension.
- 3 Click geometry to extend to the next intersection.
- 4 Continue clicking geometry to extend.
- 5 Right-click then select Cancel to end the command.

Trim

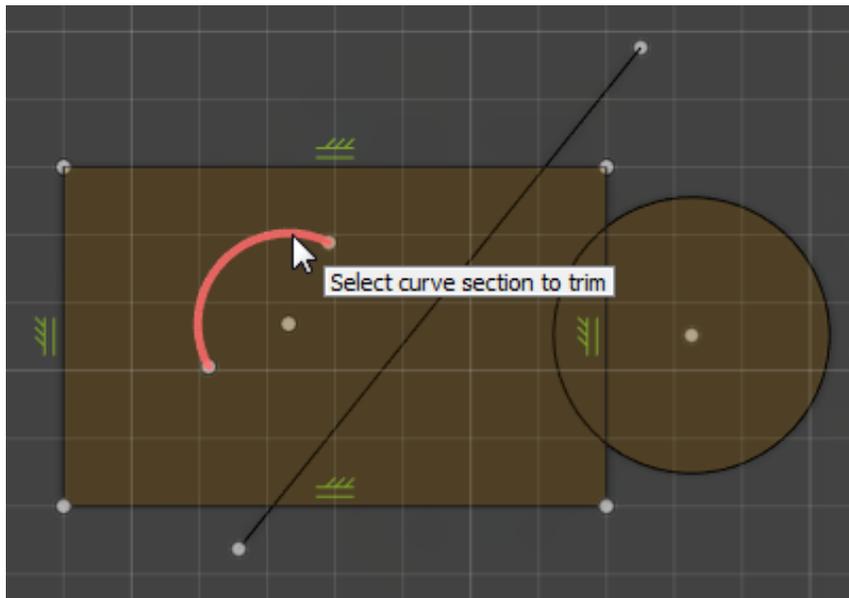


Removes a portion of the selected geometry up to the next intersection with existing geometry. If no intersection exists, the selected entity is deleted.

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.



Trim an entity

- 1 Expand the Sketch pull-down then select Trim.
- 2 Hover over sketch geometry to see a preview of the trim.
- 3 Click geometry to trim to the next intersection.
- 4 Continue clicking geometry to trim.
- 5 Right-click then select Cancel to end the command.

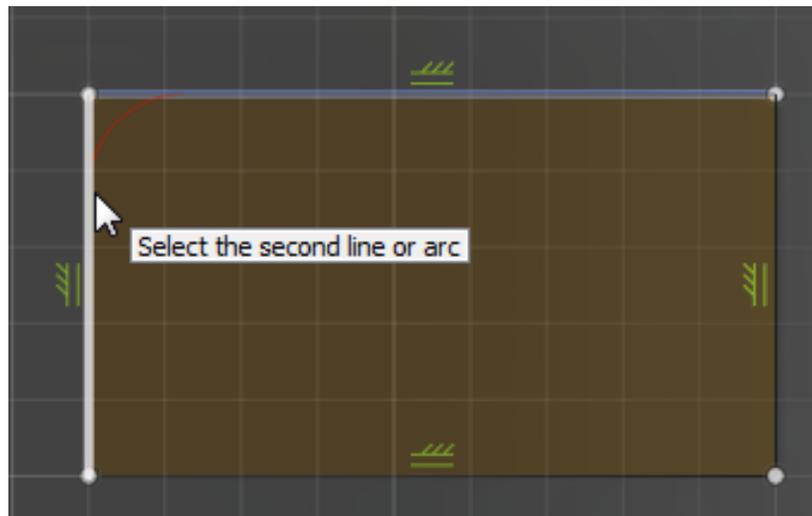
Fillet



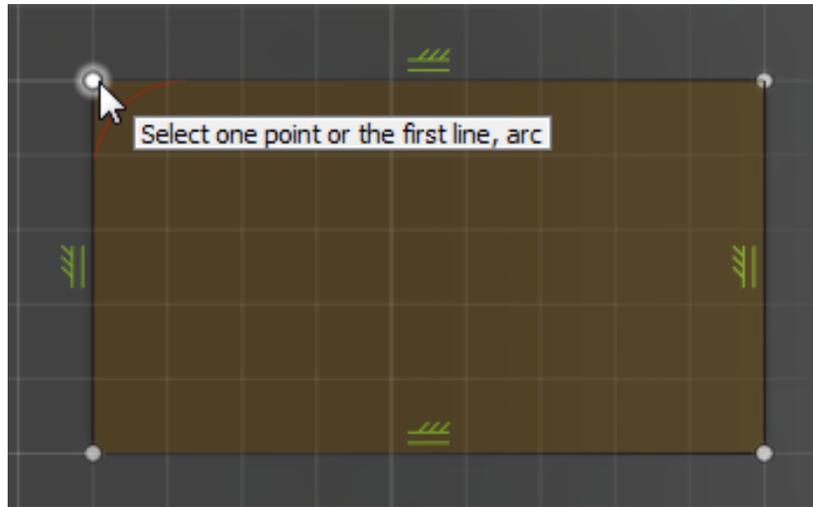
The Fillet command places an arc of a specified radius at a corner or intersection of two lines.

You can create fillets between:

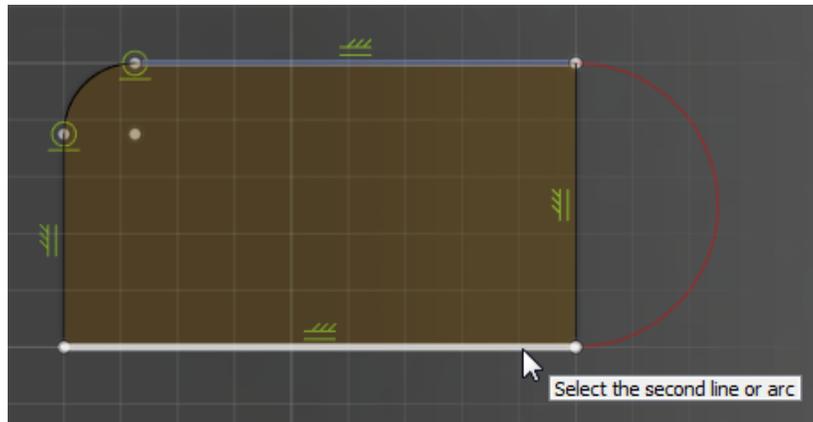
- Two intersecting lines



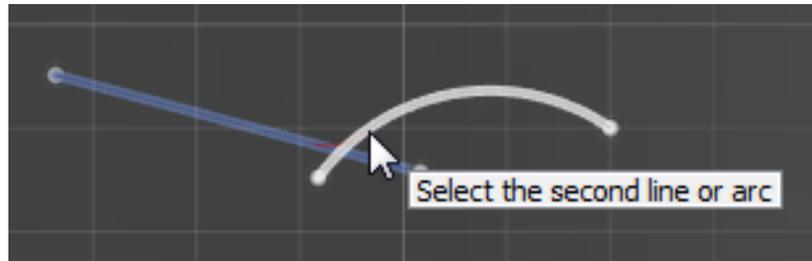
- Point at two intersecting lines



- Two parallel lines



- A line and an arc that intersect

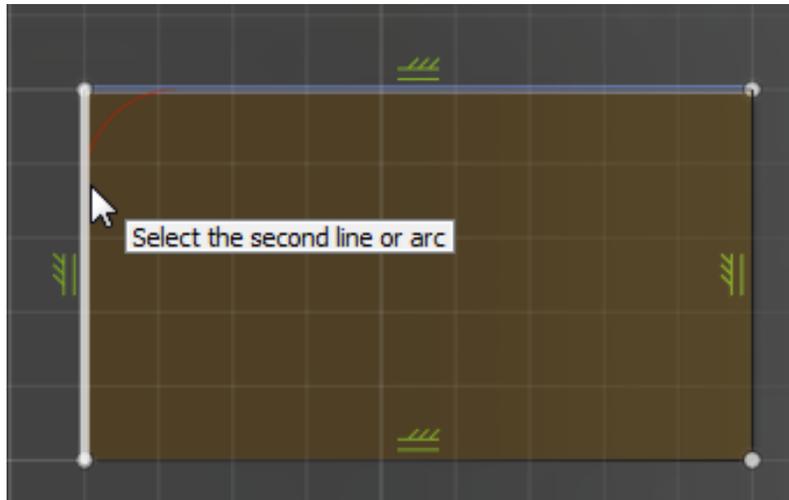


- Two circular arcs that intersect

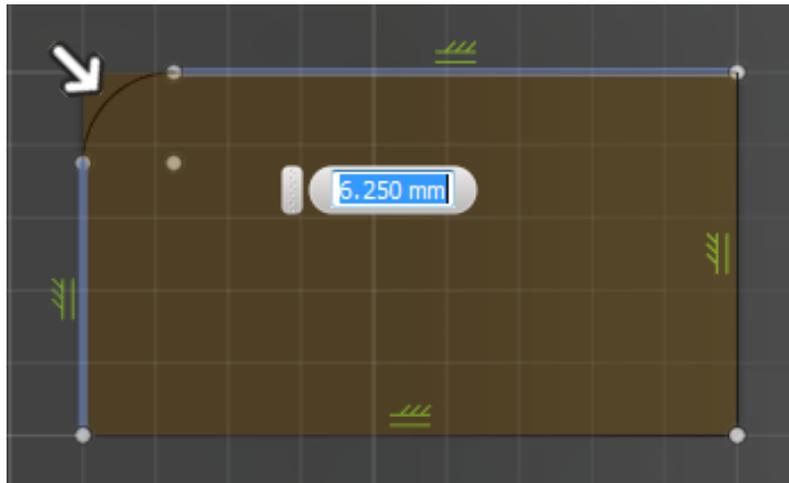


Fillet entities

- 1 Expand the Sketch pull-down then select Fillet.
- 2 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.
- 3 Pause the mouse over the second entity. The fillet preview highlights for a valid selection.



- 4 Click to select the second entity.
- 5 The fillet previews with a value that is proportional to the smaller entity. Enter a value in the box, or drag the arrow to change the size.



- 6 Right-click then select OK or press <Enter> to complete the command.

Grid Off

Turns off the display of the sketch grid when a sketch is active.

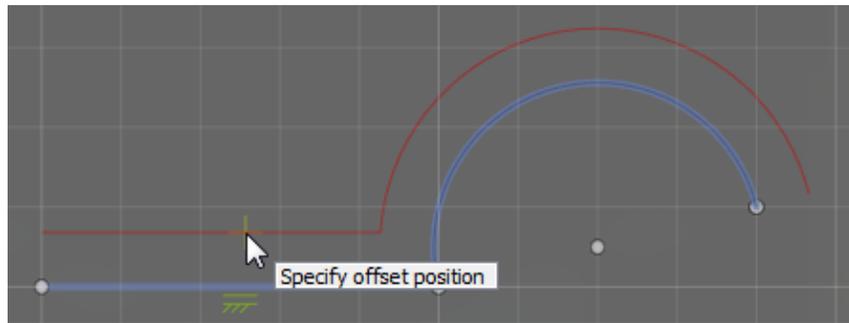
Expand the Sketch pull-down then click Grid Off.

Grid On

Turns on the display of the sketch grid when a sketch is active.

Expand the Sketch pull-down then select Grid On.

Offset



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.

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

Offset sketch entities

- 1 Expand the Sketch pull-down then select Offset.
- 2 In the graphics area, select an individual entity, a chain of connected entities, or a profile.

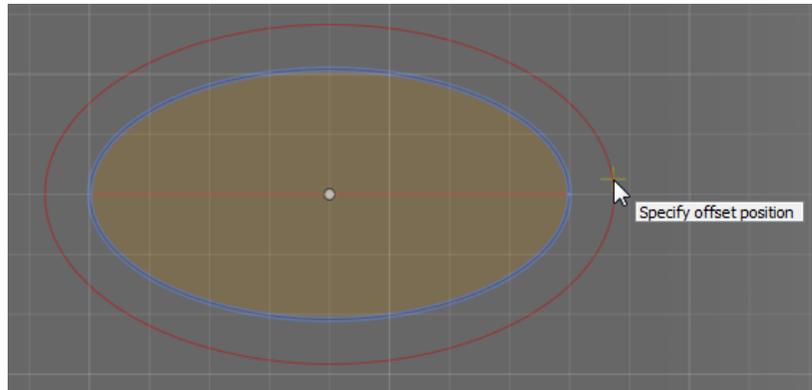
A valid selection highlights when you pause the cursor over it

- 3 Move the cursor then click to create the offset.
- 4 Select another entity to offset or right-click then select Cancel end the command.

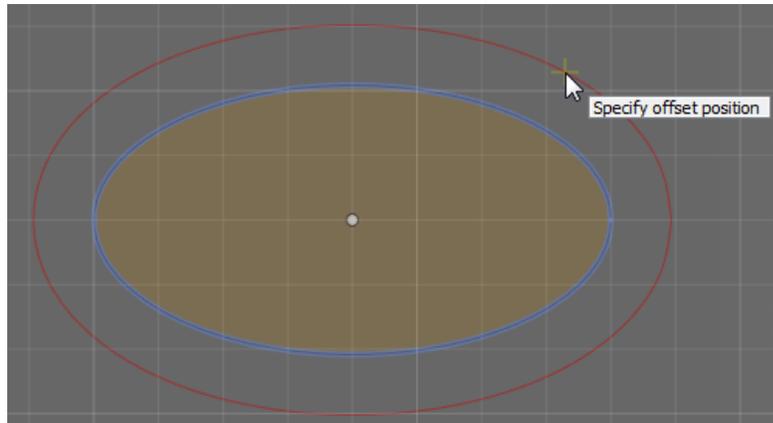
Offset an ellipse

- Select an 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.

Project

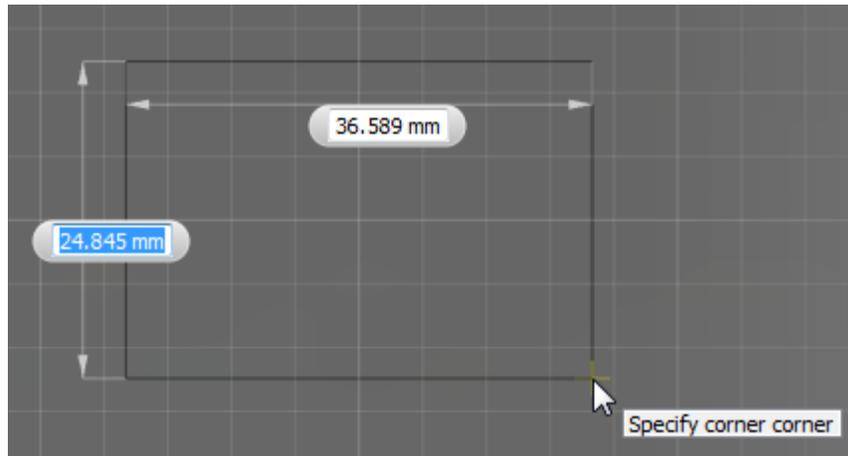
Use project geometry to create copies of existing model geometry and sketch entities in the active sketch.

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.

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

Rectangle Center Corner



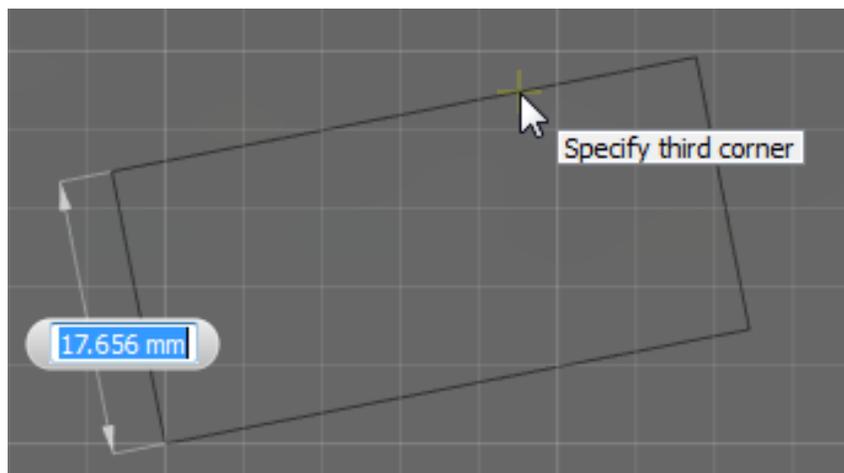
Create a rectangle by selecting the center point then a corner. The first point defines the position of the rectangle. The second point defines the length and width.

Parallel and perpendicular constraints are applied to the lines.

Create a rectangle using the center point and a corner

- 1 Expand the Sketch pull-down then select Rectangle Corner Center.
- 2 Click in the graphics window to specify the first center of the rectangle.
- 3 To specify the corner:
 - Click a point in the graphics window.
 - Enter values in the length and width fields then press <Enter>. Click a point to define the position of the rectangle.
 - Enter a value in one field then press <Enter> to lock the value. Click a point to define the other value.
- 4 Click a point to begin another rectangle or right-click then select Cancel to end the command.

Rectangle Three Point



Create a 3-point rectangle

Create a rectangle using three user-specified points. The selected points define the position and size of the four lines.

Parallel and perpendicular constraints are applied to the lines.

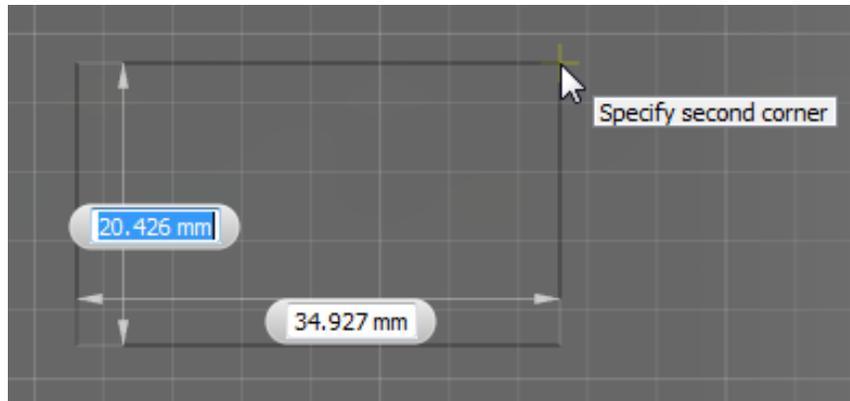
NOTE You cannot infer a parallel or perpendicular constraint to existing sketch entities, but you can infer a horizontal or vertical constraint for the first side.

Create a rectangle using three points

- 1 Expand the Sketch pull-down then select Rectangle Three Point.
- 2 Click in the graphics window to specify the first corner.
- 3 To specify the second point:
 - Click a point in the graphics window.
 - Enter a value in the field then press <Enter> to lock the value. Click a point to specify the angle of the line.
- 4 To specify the third point:
 - Click a point in the graphics window.
 - Enter a value in the field then press <Enter> to lock the value. Click a point to specify the position of the rectangle.

- 5 Click a point to begin another rectangle or right-click then select Cancel to end the command.

Rectangle Two Point



Create a rectangle by selecting two points to define the opposite corners of the rectangle.

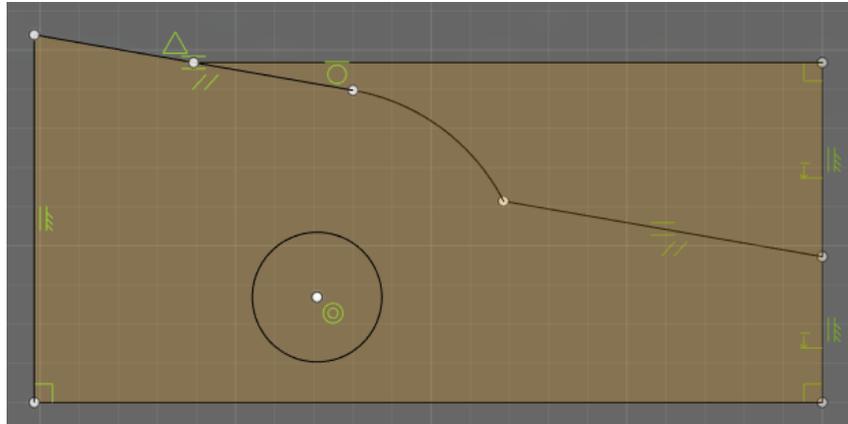
Horizontal and vertical constraints are applied to the lines.

Create a rectangle using two points

- 1 Click Rectangle Two Point on the Sketch pull-down.
- 2 Click in the graphics window to specify the first corner.
- 3 To specify the opposite corner:
 - Click a point in the graphics window.
 - Enter values in the length and width fields then press <Enter>. Click a point to define the position of the rectangle.
 - Enter a value in one field then press <Enter> to lock the value. Click a point to define the other value.
- 4 Click a point to begin another rectangle or right-click then select Cancel to end the command.

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.

Add constraints to existing geometry using the Constraints command in the Sketch pull down.

Constraint types

Con- straint	Glyph	Description
Coincident		Point to point or snap Point to entity
Collinear		Manually added only
Concentric		Circle, arc, or ellipse have the same center point.

Constraint	Glyph	Description
Midpoint		Point to midpoint of line
Fix/Unfix	Green geometry	Locks the position of the geometry
Parallel		Second line parallel to first line
Perpendicular		Second line perpendicular to first line
Horizontal		Line parallel to horizontal sketch axis
Vertical		Line parallel to vertical sketch axis
Tangent		Inferred to circles, arcs Manually added to circles, arcs, lines
Equal		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 Parallel
- 6 Horizontal
- 7 Vertical

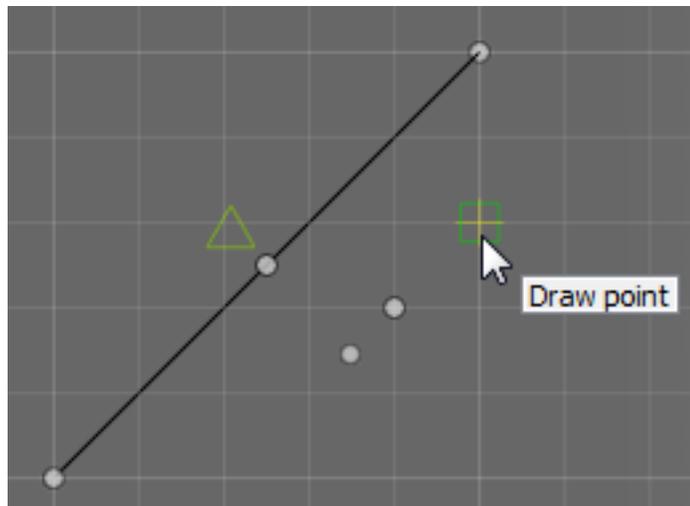
The equal and collinear constraints cannot be inferred. You can manually add these constraints after creating the entities.

Add Constraints Manually

Use the Constraints command in the Sketch pull down to manually constrain geometry.

- 1 Expand the Sketch pull-down then click Constraints.
- 2 Select the constraint type from the mini-toolbar.
- 3 Select the geometry to apply the constraint.
- 4 Continue applying constraints as needed.
- 5 Right click then select OK to end the command.

Point



Place sketch points by clicking in the graphics window. You can quickly place a series of points at precise locations, and then create sketch entities by connecting the dots. Sketch points can also be used to position hole features.

Create sketch points

- 1 Expand the Sketch pull-down then click Point.
- 2 Click in the graphics window to create a point. You can snap to the grid or to existing geometry.
- 3 Click again to create additional points.
- 4 Right-click then select Cancel to end the command.

Stop Sketch

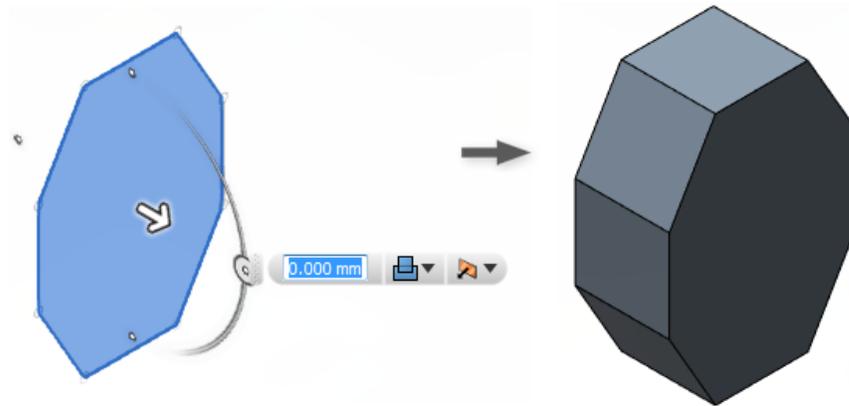
Exits the active sketch.

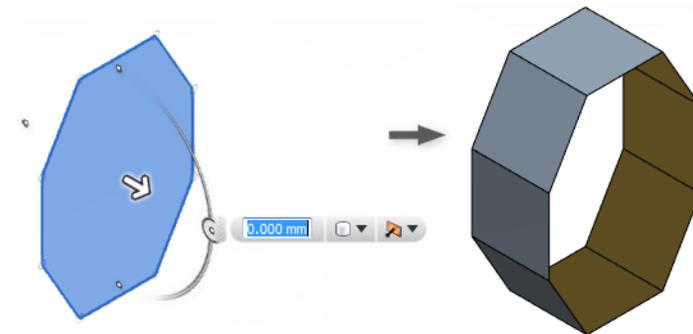
- On the Sketch pull-down, click Stop Sketch.
- On the marking menu, select Sketch then Stop Sketch.

Surface

Use the Surface commands to create and modify Surface bodies in Inventor Fusion.

Extrude



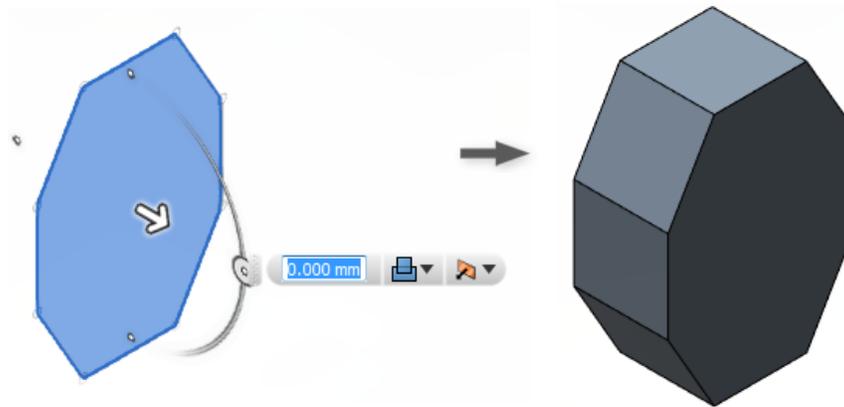


Extruded features are building blocks for creating and modifying models. You can create both solid and surface extrusions.

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.

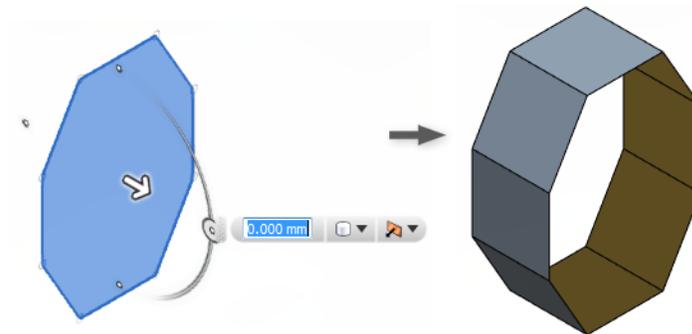
Create a solid extrusion



- 1 Switch the toolbar workspace to Solid.
- 2 Click Extrude in the Solid pull-down.
- 3 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.

- 4 Select the Direction Type
 - **One Side** Creates the extrusion in one direction.
 - **Two Side** 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.
- 5 Use the manipulators to set the height and taper, or enter values in the dialog box.
- 6 Select the Boolean Type
 - **Join** Adds material.
 - **Cut** Removes material.
 - **Intersect** Removes all material from the solid that does not overlap the new feature.
 - **New Body** Creates the extrusion as a new solid body.
 - **New Component** Creates a child component in the active component.

Create a surface extrusion



- 1 Switch the toolbar workspace to Surface.
- 2 Click Extrude in the Surface pull-down.
- 3 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.
- 4 Select the Direction Type
 - **One Side** Creates the extrusion in one direction.

- **Two Side** 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.
- 5 Use the manipulators to set the height and taper, or enter the values.
 - 6 Select the Boolean Type
 - **New Body** Creates the extrusion as a new surface body.
 - **New Component** Creates a child component in the active component.

Extrude dialog box

Profile

Enables the selection of sketch profiles.

Along Distance

Specifies the distance to extrude.

Against Distance

Available for Two Side extrusions. Specifies the second extrusion direction.

Taper Angle

Specifies the angle to taper the extrusion.

Direction Type

Specifies the method to control the size of the extrusion

- **One Side** Creates the extrusion in one direction.
- **Two Side** 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.

Boolean Type

Specifies the affect the extrusion has on the model.

- **Join** (solid only) Adds material.
- **Cut** (solid only) Removes material.
- **Intersect** (solid only) Removes all material from the solid that does not overlap the new feature.

- **New Body** Creates the extrusion as a new body.
- **New Component** Creates a child component in the active component.

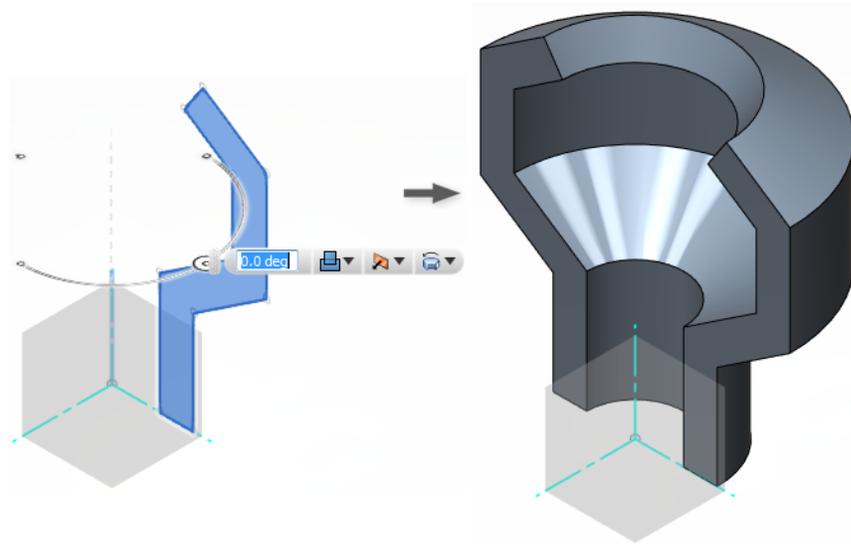
Revolve

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.

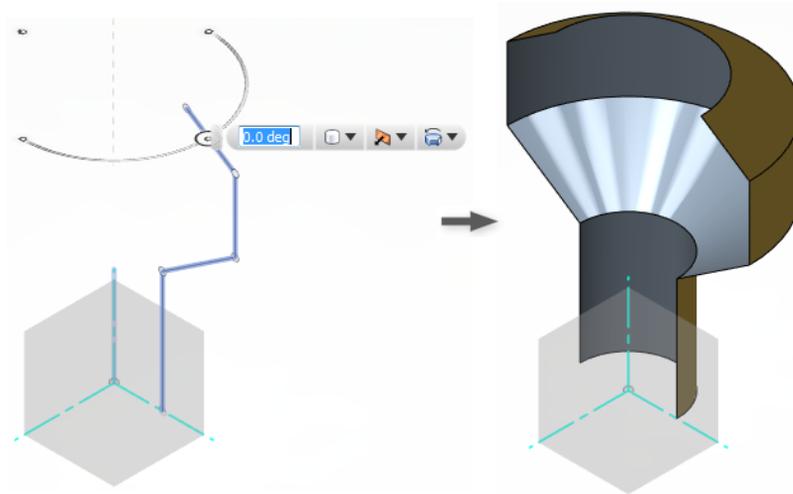
Create a revolved solid



- 1 Switch the toolbar workspace to Solid.
- 2 Click Revolve in the Solid pull-down.
- 3 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.

- 4 Set the selection to Axis in the dialog box then select the axis for the revolution.
- 5 Select the Direction Type
 - **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 Type
 - **Angle**
 - **Full**
- 7 Specify the Along Angle and Against Angle by dragging the manipulator or entering a distance in the dialog box.
- 8 Select the Boolean Type
 - **Join** Adds material.
 - **Cut** Removes material.
 - **Intersect** Removes all material from the solid that does not overlap the new feature.
 - **New Body** Creates the revolution as a new solid body.
 - **New Component** Creates a child component in the active component.
- 9 Right-click then select OK to create the revolve.

Create a revolved surface



- 1 Switch the toolbar workspace to Surface.
- 2 Click Revolve in the Surface pull-down.
- 3 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.
- 4 Set the selection to Axis in the dialog box then select the axis for the revolution.
- 5 Select the Direction Type
 - **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 Type
 - **Angle**
 - **Full**
- 7 Specify the Along Angle and Against Angle by dragging the manipulator or entering a distance in the dialog box.

- 8 Select the Boolean Type
 - **New Body** Creates the revolution as a new surface body.
 - **New Component** Creates a child component in the active component.
- 9 Right-click then select OK to create the revolve.

Revolve dialog box

Profile

Enables the selection of sketch profiles.

Axis

Enables the selection of the axis of revolution.

Along Angle

Specifies the angle to revolve.

Against Angle

Specifies the opposite angle for Two Side revolutions.

Direction Type

Specifies the method to control the size of the revolution

- **One Side** Creates the revolution in one direction.
- **Two Side** Creates the revolution in both directions. Each direction can have a different angle.
- **Symmetric** Creates the revolution in both directions. Each direction has the same angle.

Type

Select whether to specify an angle to revolve or revolve a full 360 degrees.

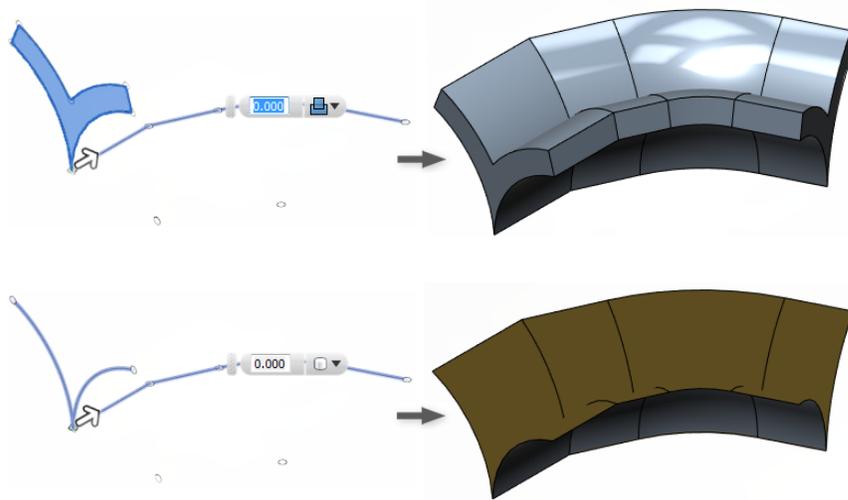
Boolean Type

Specifies the affect the extrusion has on the model.

- **Join** (solid only) Adds material.
- **Cut** (solid only) Removes material.
- **Intersect** (solid only) Removes all material from the solid that does not overlap the new feature.
- **New Body** Creates the revolution as a new body.

- **New Component** Creates a child component in the active 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 Switch the toolbar workspace to Solid.
- 2 Expand the Solid pull-down then click Sweep.
- 3 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.

- 4 Set the selection to Path then select one or more sketch entities or model edges for the path.

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

- 5 Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path in the dialog box.
- 6 Select the Orientation type:
 - **Perpendicular** keeps the region perpendicular to the path.
 - **Parallel** keeps the region parallel to the region sketch.
- 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 Body** Creates the sweep as a new solid body.
 - **New Component** Creates a child component under the active component.
- 8 Right-click then select OK to create the sweep.

Create a swept surface

- 1 Switch the toolbar workspace to Surface.
- 2 Expand the Surface pull-down then click Sweep.
- 3 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.
- 4 Set the selection to Path then select one or more sketch entities or model edges for the path.
- 5 Use the manipulator to set the distance along the path, or enter the percentage value. You can select Full Path in the dialog box.
- 6 Select the Orientation type:
 - **Perpendicular** keeps the region perpendicular to the path.
 - **Parallel** keeps the region parallel to the region sketch.
- 7 Select the Boolean Option:
 - **New Body** Creates the sweep as a new solid body.

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

8 Right-click then select OK to create the sweep.

Sweep dialog box

Profile

Enables the selection of sketch profiles.

Path

Enables the selection of sweep path.

Along Value

Specifies the percentage value of the sweep.

Full Path

Enable the Full Path option to sweep the profile along the entire path.

Orientation

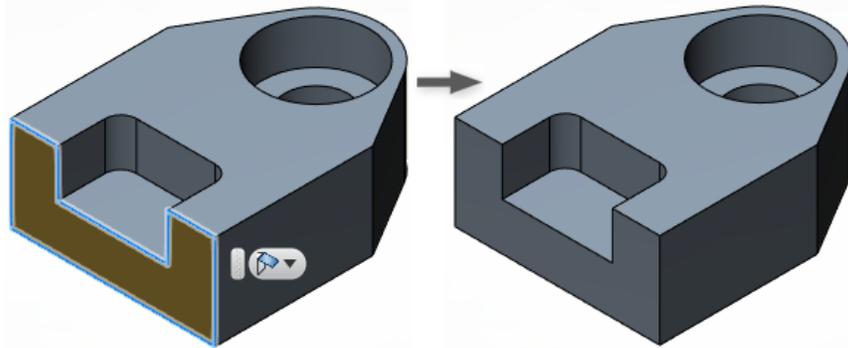
- **Perpendicular** keeps the region perpendicular to the path.
- **Parallel** keeps the region parallel to the region sketch.

Boolean Type

Specifies the affect the sweep has on the model.

- **Join** (solid only) Adds material.
- **Cut** (solid only) Removes material.
- **Intersect** (solid only) Removes all material from the solid that does not overlap the new feature.
- **New Body** Creates the sweep as a new body.
- **New Component** Creates a child component in the active component.

Patch



The Patch command creates a surface, also called a boundary patch, that replaces a missing face.

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.

Create a surface

- 1 Switch the toolbar workspace to Surface.
- 2 Click Patch on the Surface pull-down.
- 3 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.
- 4 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 Right-click then click OK to create the surface.

Patch dialog box

Selection

Enables the selection of edges.

Enable Chaining

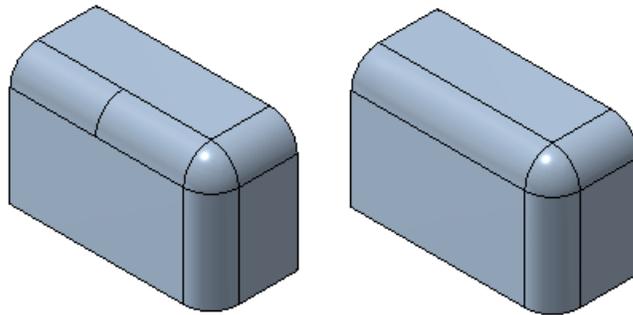
When enabled, clicking a single edge selects all adjacent edges.

Connectivity

Controls the transition between the patch and the selected 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).

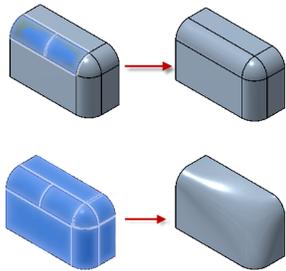
Merge



Use the Merge command to combine multiple faces into one 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.

Merge multiple faces



- 1 Switch the toolbar workspace to Surface.
- 2 Expand the Surface pull-down then select Merge.
- 3 Select two or more faces.
Clear the selection for Enable Chaining to prevent automatic selection of all tangent or adjacent faces.
- 4 Right-click then click OK.

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

Merge dialog box

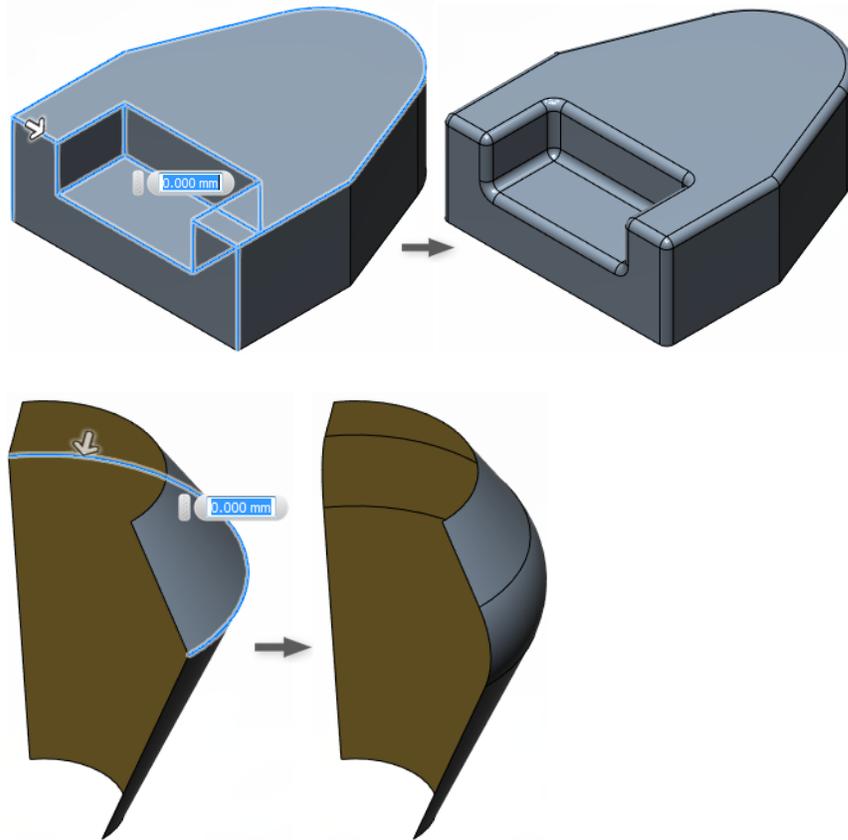
Selection

Enables the selection of faces to merge.

Enable Chaining

When enabled, clicking a single face selects all tangent or adjacent faces.

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 then selecting Edit Fillet, or by using Press/Pull.

Create a fillet

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid pull-down or Surface pull-down then select Fillet.

- 3 Deselect the Tangent Chain if you do not want tangentially connected edges.
- 4 Select the edges to fillet.
- 5 Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
- 6 Deselect the Corner Option if you want a smooth transition between edges. The Corner Option produces corners like a milling machine or a router.
- 7 Right-click then select OK to create the fillet.

Fillet dialog box

Edges

Enables the selection of edges to fillet.

Radius

Specifies the radius for the fillet.

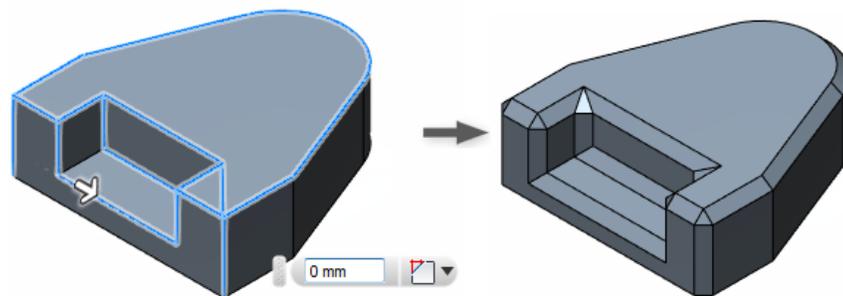
Corner Option

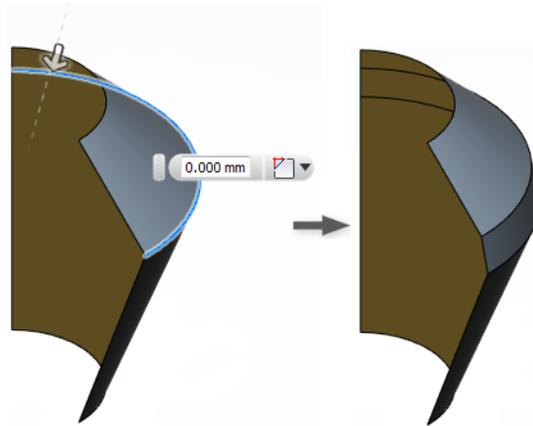
Creates rolling ball corners when enabled.

Tangent Chain

Specifies whether to include tangentially connected edges or select individual edges.

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 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid pull-down or Surface pull-down then click Chamfer.
- 3 Deselect Tangent Chain if you do not want tangentially connected edges.
- 4 Select the edges to chamfer
- 5 Select the type of chamfer to apply.
 - **Equal distance** Specify a single distance for both sides of the chamfer.
 - **Two distances** Specify a distance for each face of the chamfer.
 - **Distance and angle** Specify a distance and angle to create the chamfer.
- 6 Drag the manipulator or enter a value. You can only drag the manipulator in the positive direction.
- 7 Right-click then select OK to create the chamfer.

Chamfer dialog box

Controls the settings of the Chamfer command.

Edges

Enables the selection of edges to chamfer.

First Distance

Specifies the first distance for the chamfer.

Second Distance

Available for two distances chamfers. Specifies the second distance for the chamfer.

Angle

Available for distance and angle chamfers. Specifies the angle for the chamfer.

Tangent Chain

Specifies whether to include tangentially connected edges or select individual edges.

Chamfer Type

The chamfer type specifies the method to control the size of the chamfer.

- **Equal distance** Specify a single distance for both sides of the chamfer.
- **Two distances** Specify a distance for each face of the chamfer.
- **Distance and angle** Specify a distance and angle to create the chamfer.

Offset

The Offset command creates a surface, face, 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 Switch the toolbar workspace to Surface.
- 2 Expand the Surface pull-down then select Offset.
- 3 In the graphics area, select one or more faces or surfaces.

- 4 Select the Boolean Type:
 - **New Body** Creates a new body to contain the surface.
 - **New Component** Creates a new component to contain the surface.
- 5 Drag the arrow to set the offset, or enter a value.
- 6 Right-click then select OK.

Offset dialog box

Faces

Enables the selection of faces or surfaces to offset.

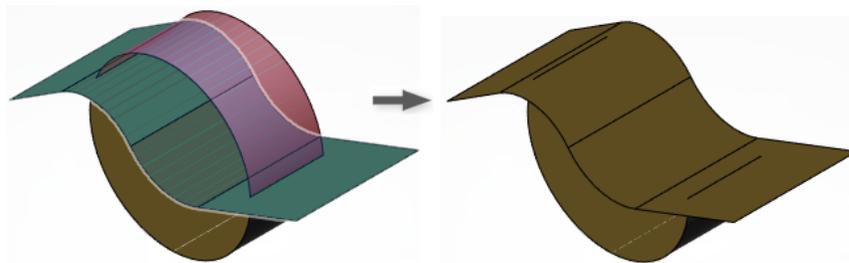
Distance

Specifies the distance to offset.

Boolean Type

Select to create a new body or new component that contains the surface.

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 surfaces

- 1 Switch the toolbar workspace to Surface.
- 2 Expand the Surface pull-down then select Trim.

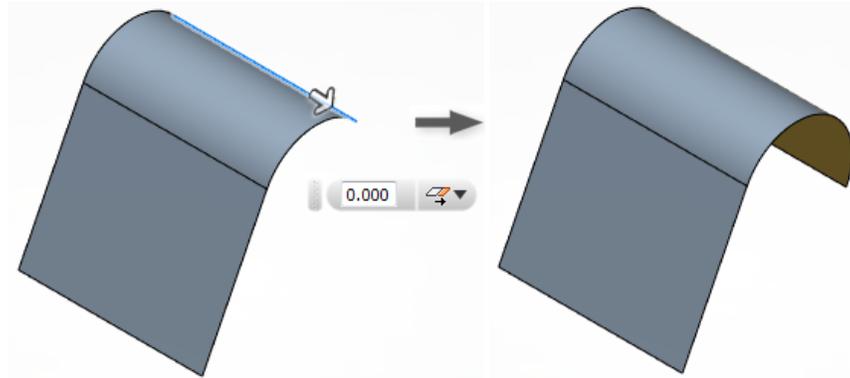
- 3 Select a Trim tool. Valid feature selections include a surface, a quilt, a solid face, or a work plane. You can also use a connected series of sketch entities that do not intersect the target surface.
- 4 The intersecting surfaces highlight. Select the sections to remove.
- 5 Right-click then click OK to delete the selected surfaces.

Trim dialog box

Trim Tool

Enables the selection of the surface to use as a trim tool.

Extend



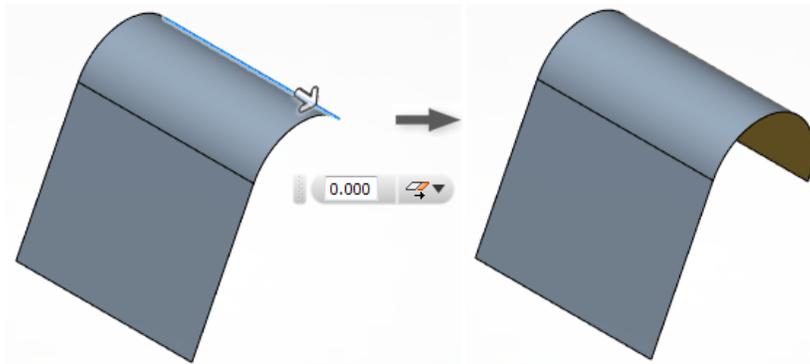
Extend a face or surface by selecting one or more edges and moving them away from the face. You cannot extend a face on a solid.

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

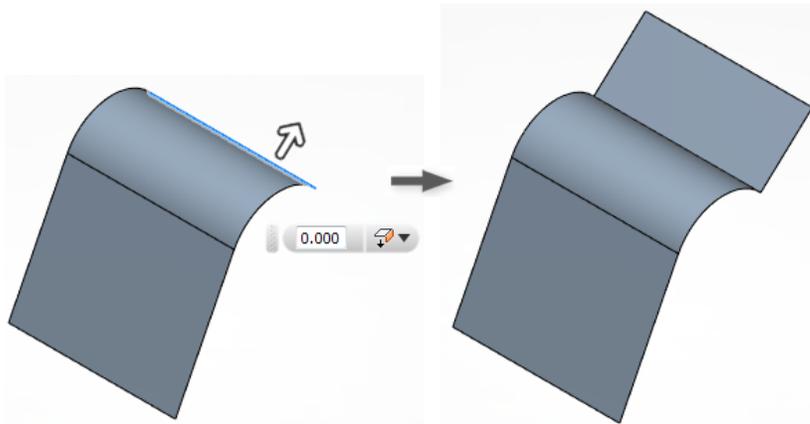
You can extend planar and non-planar surfaces. There can be limitations when extending non-planar surfaces due to the curvature.

Extend type

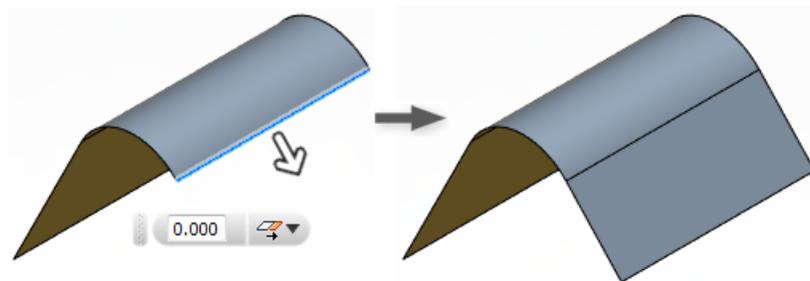
- **Natural** Extends the current faces



- **Perpendicular** Creates new faces perpendicular to the current faces.



- **Tangent** Creates new flat faces tangent to the current face.



Extend a surface

- 1 Switch the toolbar workspace to Surface.
- 2 Expand the Surface pull-down then click Extend.
- 3 Select a surface edge.
 - You can select multiple edges. They can be unconnected, or coplanar.
 - You cannot select Solid edges.
- 4 Select the Extension Type
 - **Natural** Extends the current faces
 - **Perpendicular** Creates new faces perpendicular to the current faces.
 - **Tangent** Creates new flat faces tangent to the current face.
- 5 Drag the arrow manipulator, or enter a value.
- 6 Right-click then select OK to create the extended face.

Extend dialog box

Edges

Enables the selection of edges to extend.

Value

Specifies the distance to extend the surfaces.

Extend Type

Determines the direction of the new surface.

- **Natural** Extends the current faces
- **Perpendicular** Creates new faces perpendicular to the current faces.
- **Tangent** Creates new flat faces tangent to the current face.

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 the normal direction of a surface

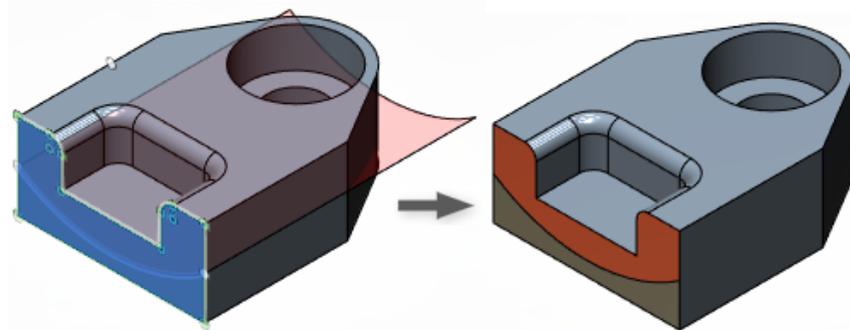
- 1 Switch the toolbar workspace to Surface.
- 2 Expand the surface pull-down then select Reverse Normal.
- 3 Select the faces to change.
- 4 Right-click then select OK.

Reverse Normal dialog box

Faces

Enables the selection of faces to change.

Split Face



Use Split Face to divide faces on a surface or solid. The faces can be any face on a solid body or surface body. The Splitting Tool can be a work plane, surface, or sketch geometry. Sketch geometry is projected onto the target faces.

Faces are typically split to add draft, delete an area, or to create new features.

Split a face

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid or Surface pull-down then select Split Face.
- 3 Select the faces to split.

- 4 Click Splitting Tool in the dialog box then select the intersecting work plane, surface, or sketch geometry.
- 5 Right-click then select OK.

Split Face dialog box

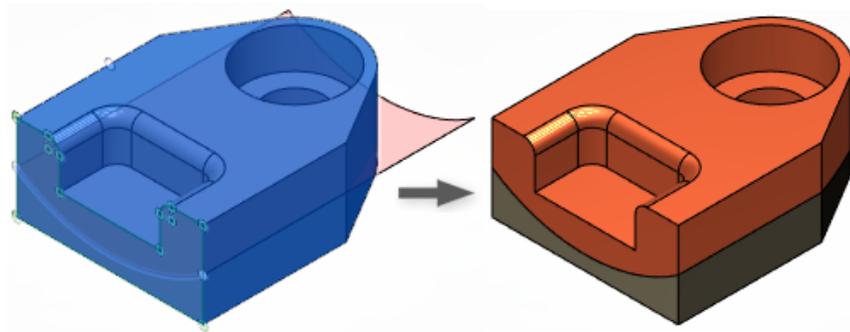
Faces to Split

Enables the selection of faces to split.

Splitting Tool

Enables the selection of a surface, work plane, or sketch geometry to divide the bodies.

Split Body



Use Split Body to divide a single body into two bodies. The bodies can be solid bodies or surface bodies. The Splitting Tool can be a work plane, surface, or sketch geometry. Sketch geometry is projected onto the target faces.

Split bodies

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid or Surface pull-down then select Split Body.
- 3 Select the bodies to split.
- 4 Click Splitting Tool in the dialog box then select the intersecting work plane, surface, or sketch geometry.
- 5 Right-click then select OK.

Split Body dialog box

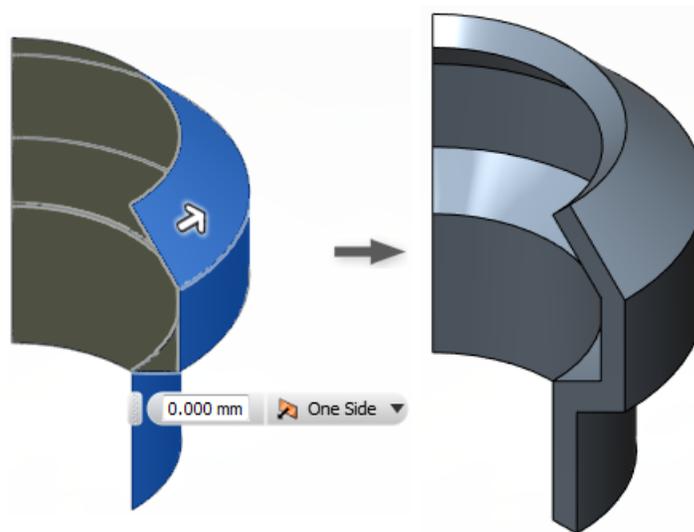
Body to Split

Enables the selection of bodies to split.

Splitting Tool

Enables the selection of a surface, work plane, or sketch geometry to divide the bodies.

Thicken



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

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

Thicken a face, surface, or quilt

- 1 Switch the toolbar workspace to Solid or Surface.
- 2 Expand the Solid pull-down or Surface pull-down then click Thicken.
- 3 Select one or more faces, surfaces, or quilts.

- 4 Select the Direction Type
 - **One Side** Thickens the faces in one direction.
 - **Symmetric** Thickens the faces in both directions. Each direction has the same distance.
- 5 Drag the arrow to set the offset, or enter a value in the dialog box.
- 6 Select the Boolean Type
 - **Join** Adds material.
 - **Cut** Removes material.
 - **Intersect** Removes all material from the solid that does not overlap the new feature.
 - **New Body** Creates the extrusion as a new body.
 - **New Component** Creates a child component in the active component.
- 7 Right-click then select OK to thicken the faces.

Thicken dialog box

Faces

Enables the selection of faces, surfaces, or quilts.

Thicken

Specifies the distance to thicken the selected faces.

Direction Type

- **One Side** Thickens the faces in one direction.
- **Symmetric** Thickens the faces in both directions. Each direction has the same distance.

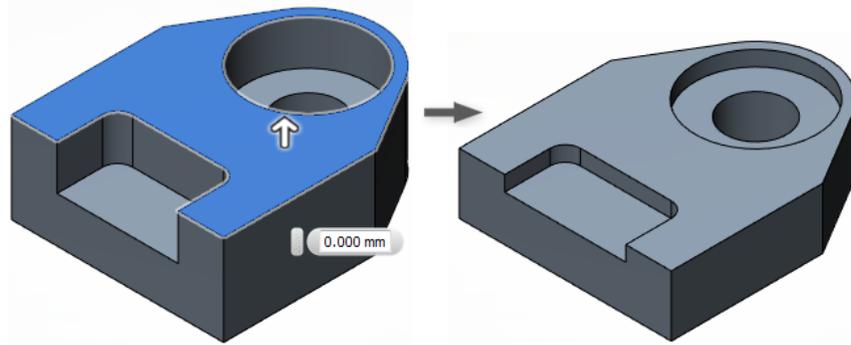
Boolean Type

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

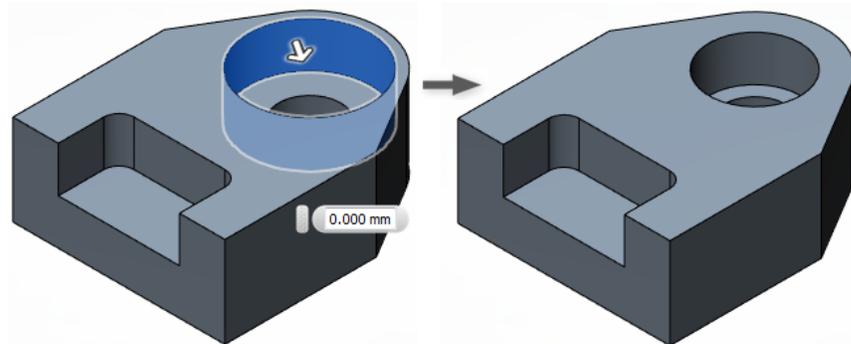
Modify

Use the Modify commands to make changes to solids, surfaces, and components in Inventor Fusion.

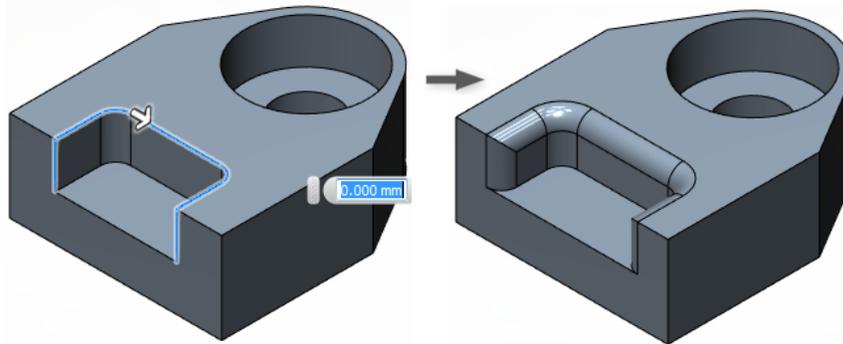
Press Pull



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.



Use Press Pull to edit fillet, holes, and chamfers.



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.

How to modify geometry using Press Pull

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Press Pull in the Modify pull-down.
- 3 Select the faces to modify.
- 4 Drag the arrow manipulator or enter a value in the field in the dialog box.
- 5 Right-click then select OK to accept the change.

Press Pull dialog box

Selection

Enables the selection of faces or edges.

Distance/Radius

Specifies the distance to move the face or the radius of the arc face.

Tweak Faces

Use Tweak Face to modify the 3D model. Tweak face moves the selected faces and trims or extends adjacent faces.

Edit geometry using Tweak

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Tweak Faces.
- 3 Select the faces to tweak.
- 4 Drag a manipulator or enter values in the dialog box.
- 5 Right-click and select OK.

NOTE You can snap to existing geometry when tweaking faces. Drag a manipulator to activate it then select the geometry to snap to.

Tweak dialog box

Faces

Enables the selection of faces to tweak.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Scale Command

Enlarges or reduces selected bodies based on a specified scale factor.

Scale an object

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Scale.
- 3 Select the body that you wish to scale.
- 4 Click Point in the dialog box then select a reference point to scale around.
- 5 Select Uniform or Non Uniform for the Scale Type
- 6 Set the Scale Factor:
 - For Uniform scale, drag the arrow manipulator or enter a value for Scale Factor.
 - For Non Uniform scale, drag the arrow manipulators or enter values for the X Distance, Y Distance, and Z Distance.
- 7 Right-click then select OK to accept the changes.

Scale dialog box

Scale Type

- **Uniform** Uses a single scale factor to scale the object.
- **Non Uniform** Use a different scale factor to scale the object in the X, Y, and Z direction.

Bodies

Enables the selection of bodies to scale.

Point

Enables the selection of a reference point to scale around.

Scale Factor

For Uniform scale, specifies the value for the scale.

X Distance

For Non Uniform, specifies the value to scale in the X direction.

Y Distance

For Non Uniform, specifies the value to scale in the Y direction.

Z Distance

For Non Uniform, specifies the value to scale in the Z direction.

Move

Use the Move command to move solid bodies, surface bodies, and work geometry.

Move bodies

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Move.
- 3 Select the bodies, or work geometry to move.
- 4 Drag a manipulator or enter values in the dialog box.
- 5 Right-click and select OK.

Move dialog box

Selection

Enables the selection of bodies and work geometry to move.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Move Components

Use the Move Components command to change the position of component instances.

Move components

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Move Components.
- 3 Select the components to move.
- 4 Drag the manipulators or enter values in the dialog box.
- 5 Right-click then select OK.

Move Components dialog box

Components

Enables the selection of components to move.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

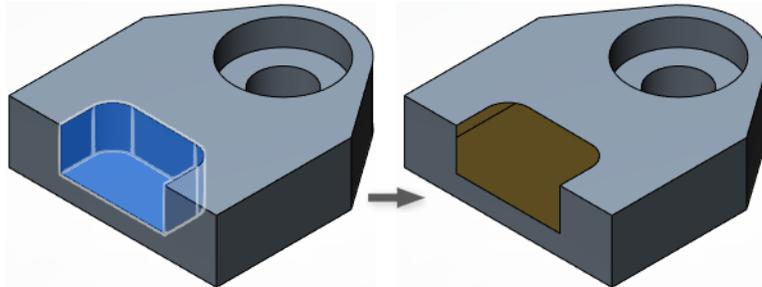
Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Delete Faces



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.

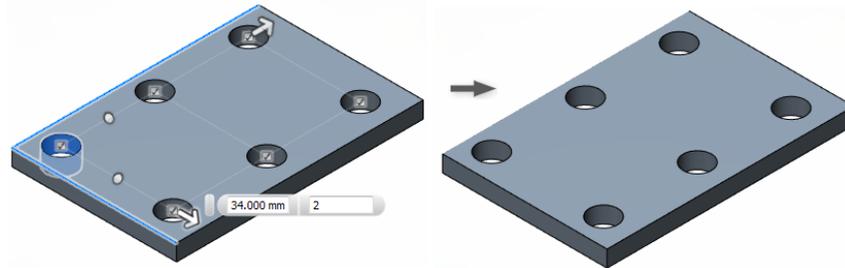
Delete a face

- 1 Switch the toolbar workspace to Surface.
- 2 Expand the Modify pull-down then select Delete Faces.
- 3 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.
- 4 Right-click then select OK.

Pattern

Use the Pattern commands to pattern or mirror geometry in Inventor Fusion.

Rectangular Pattern



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.

You can pattern faces, features, or bodies.

Create rectangular patterns

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Rectangular Pattern in the Pattern pull-down.
- 3 In the graphics area or the browser, select the faces, features, or bodies to pattern.
- 4 Click Directions, and select a linear edge or work axis. Perpendicular manipulators display.
- 5 For a non-perpendicular pattern, select a second linear edge or work axis.
- 6 Use the arrows to set the U Distance and V Distance or enter values in the fields.
- 7 Use the dot manipulator to set the U Grid Entries and V Grid Entries or enter values in the fields.
- 8 Click a check mark to suppress an occurrence.
- 9 Right-click the select OK to create the pattern.

Rectangular Pattern dialog box

Objects

Enables the selection of objects to pattern. Valid objects are faces, features, or bodies.

Directions

Enables the selection of edges to control the U and V directions.

Suppress

Displays check boxes in the graphics window at each occurrence. Uncheck a box at an occurrence to suppress that occurrence from the pattern.

U Grid Entries

Specifies the number of occurrences along the first direction.

U Distance

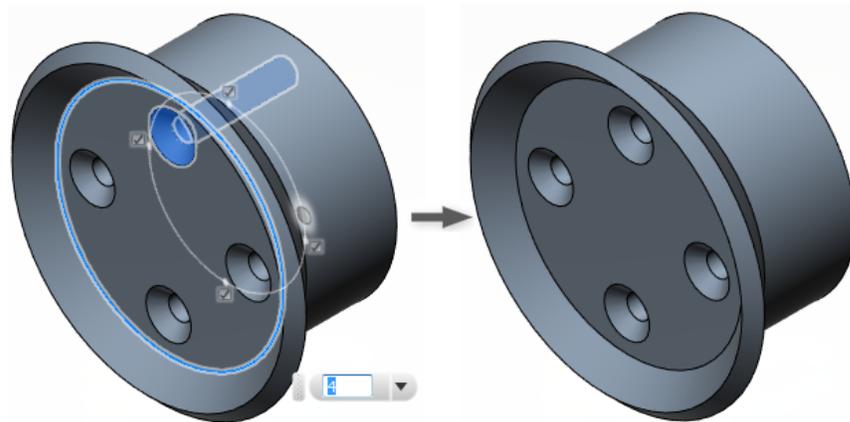
Specifies the overall distance in the first direction. Occurrences are spaced evenly along the distance.

V Grid Entries

Specifies the number of occurrences along the second direction.

V Distance

Specifies the overall distance in the second direction. Occurrences are spaced evenly along the distance.

Circular Pattern

The circular pattern command copies selected objects around a selected axis. You can pattern faces, features, or bodies.

The axis for a circular pattern can be a sketch line, a linear edge, a work axis, a cylindrical face, or a circular edge.

Create a circular pattern

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Pattern pull-down then click Circular Pattern.
- 3 In the graphics area or the browser, select the faces, features, or bodies to pattern.
- 4 Click Circular Axis in the dialog box then select a sketch line, linear edge, a work axis, a cylindrical face, or a circular edge.
- 5 Use the ball manipulator to set the number of occurrences or enter a value in the field.
- 6 Click a check mark to suppress an occurrence.
- 7 Set the Circular Type to Full or Angle.
 - **Full** The copies are evenly spaced around a complete circle.
 - **Angle** The copies are evenly spaced around the specified angle.
- 8 If Circular Type is set to angle, enter the value in the Total Angle field.
- 9 Right-click the select OK to create the pattern.

Circular Pattern dialog box

Objects

Enables the selection of objects to pattern. Valid objects are faces, features, or bodies.

Circular Axis

Enables the selection of the axis to pattern around. Valid objects are a sketch line, linear edge, a work axis, a cylindrical face, or a circular edge.

Suppress

Displays check boxes in the graphics window at each occurrence. Uncheck a box at an occurrence to suppress that occurrence from the pattern.

Total Angle

Available when the Circular Type is set to Angle. Specifies the total angle to space the occurrences around.

Count

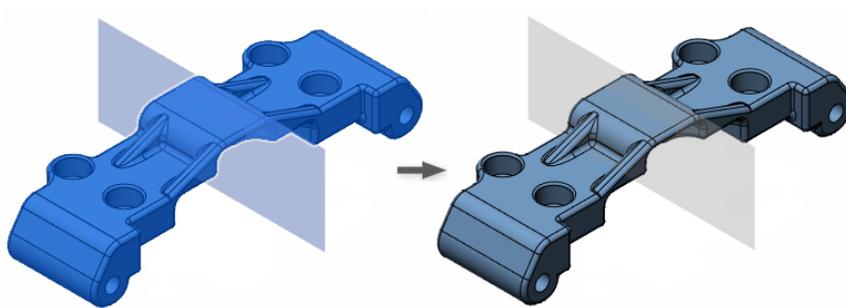
Specifies the total number of copies.

Circular Type

Controls how the copies are positioned around the pattern.

- **Full** The copies are evenly spaced around a complete circle.
- **Angle** The copies are evenly spaced around the specified angle.

Mirror



Mirrors the selected objects to the opposite side of a selected face or plane.

Valid object types are faces, features, or bodies.

Mirror objects

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Mirror in the Pattern pull-down.
- 3 In the graphics area or the browser, select the faces, features, or bodies to mirror.
- 4 Click Mirror Plane in the dialog box then select the plane or face to mirror around.
- 5 Right-click then click OK to create the mirror.

Mirror dialog box

Objects

Enables the selection of objects to mirror. Valid objects are faces, features, or bodies.

Mirror Plane

Enables the selection of the plane or face to mirror around.

Construction

Work features or construction 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.

Work Planes

A work plane is a construction feature that defines the parametric location of a sketch plane in 3D space. A work plane is useful when no planar face exists to use as a sketch plane, for example, when sketching on curved faces.

Create work planes

Inventor Fusion creates the following types of work planes. Most work planes 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.

Icon	Work Plane Type	Inputs
	Offset Plane	Select a work plane or planar model face and enter the distance.

Icon	Work Plane Type	Inputs
	Plane At Angle	Select a model edge, sketch line, or work axis, and enter the angle.
	Tangent Plane	Select a cylindrical face and enter the angle. The plane is created where you clicked on the cylinder.
	Midplane	Select two parallel planes or planar model faces. The work plane is created halfway between them.
	Plane Through Two Edges	Select two model edges, sketch lines, or work axes.
	Plane Through Three Points	Select three model, sketch, or work points.

Work Axes

A work axis is a construction feature that defines an axis in 3D space. A work axis is useful when no geometry exists to use as an axis.

Create work axes

Inventor Fusion creates the following types of work axes. Work axes are automatically created based on the geometry selections.

Icon	Work Axis Type	Inputs
	Axis Through Cylinder	Select a cylindrical face or a cone.
	Axis Perpendicular At Point	Select a model or work plane.
	Axis Through Two Planes	Select two model faces or work planes.
	Axis Through Two Points	Select two vertices, sketch points or work points.
	Axis Through Edge	Select a linear or circular edge on a model or a sketch profile.

Work Points

A work point is a construction feature that defines a point in 3D space. A work point can be projected onto a sketch and incorporated into dimension and constraint schemes.

Create work points

Inventor Fusion creates the following types of work points. Work points are automatically created based on the geometry selections.

Icon	Work Axis Type	Inputs
	Point at Vertex	Select a vertex, work point, or sketch point.
	Point Through Two Edges	Select two edges on a model or two models or sketch profiles that intersect.
	Point Through Three Planes	Select three model faces or work planes.
	Point at Center of Circle/Sphere	Select a circular edge or spherical surface.
	Point at Edge And Plane	Select a work plane or model face, or a sketch profile and a work axis or sketch line.

Select

Use the select commands to control selection sets in Inventor Fusion.

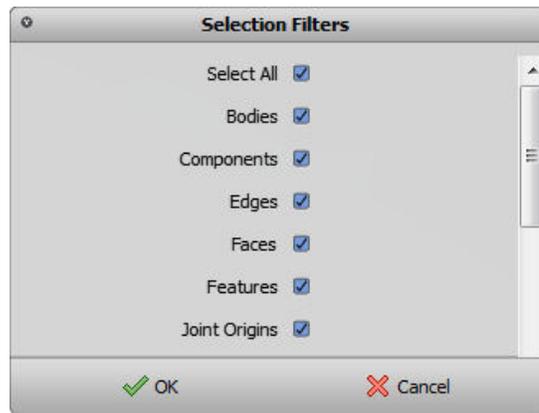
Select

Enables the selection of objects in the graphics window.

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.

Access the Selection Filters command from the Select pull-down in the Solid, Surface, or Assemble workspaces.



The selection filters control only selection in the graphics window. You can still select objects in the browser

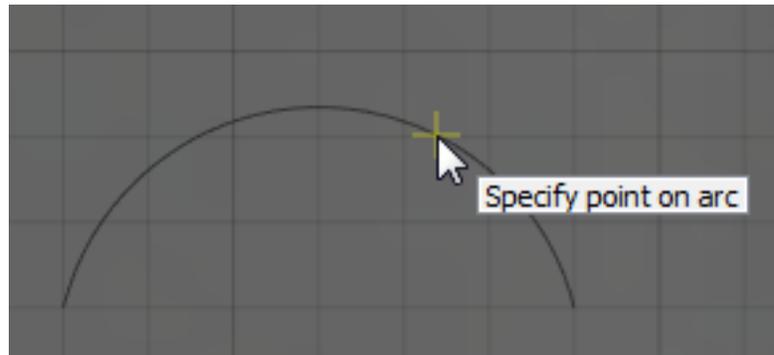
Assemble Workspace

Use the Assemble workspace to position and manage components in Inventor Fusion.

Sketch

Use the Sketch commands to create and modify 2D sketch geometry in Inventor Fusion.

Arc Three Point

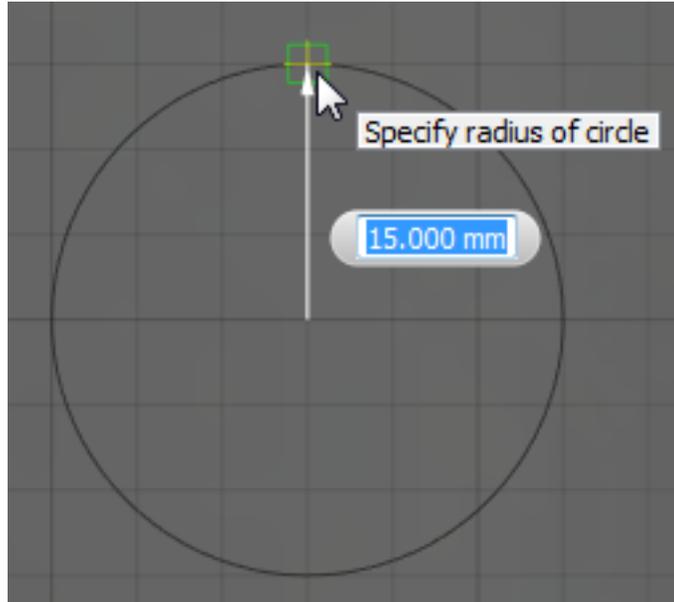


Creates an arc using three user-specified points. The three points define the position and size of the arc.

Create an arc

- 1 Expand the Sketch pull-down then click Arc Three Point.
- 2 Click in the graphics window to specify the first end point of the arc.
- 3 Click again to specify the second end point of the arc.
- 4 Click a third point to specify a point that lies along the arc.
- 5 Click again to begin creating a new arc or right-click then select Cancel to end the command.

Circle Center Radius



Creates a circle using two user-specified points. The points define the position of the center of the circle and the size of the circle.

Create a circle

- 1 Click Circle Center Radius on the Sketch pull-down.
- 2 Click in the graphics window to specify the center point of the circle.
- 3 To specify the radius of the circle:
 - Click a point in the graphics window.
 - Enter a value in the radius field then press <Enter>. Click a point to create the circle.
- 4 Click again to begin creating a new circle or right-click then select Cancel to end the command.

Ellipse

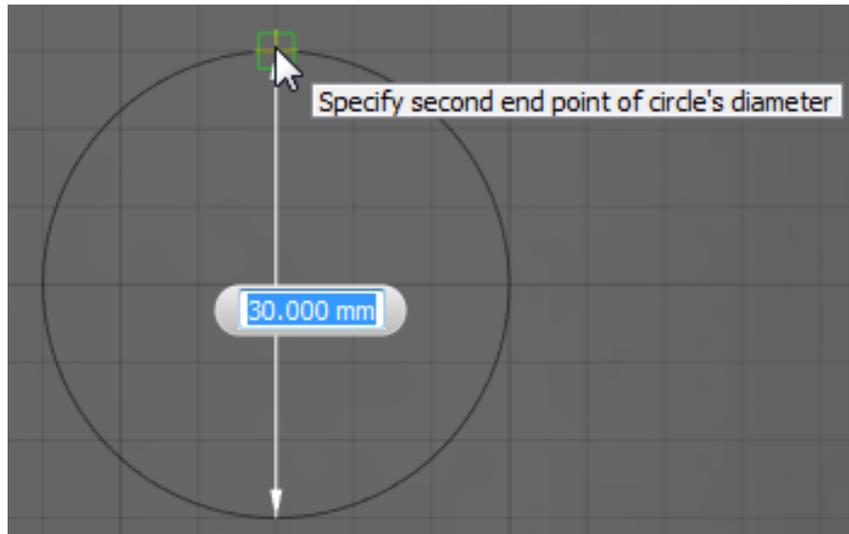


Creates an ellipse using three user-specified points. The points define the position and size of the ellipse.

Create an ellipse

- 1 Expand the Sketch pull-down then click Ellipse.
- 2 Click in the graphics window to specify the center point of the ellipse.
- 3 Click to specify the end point of the first axis of the ellipse.
- 4 Click a point to specify a point along the ellipse.
- 5 Click again to begin creating a new ellipse or right-click then select Cancel to end the command.

Circle Two Point

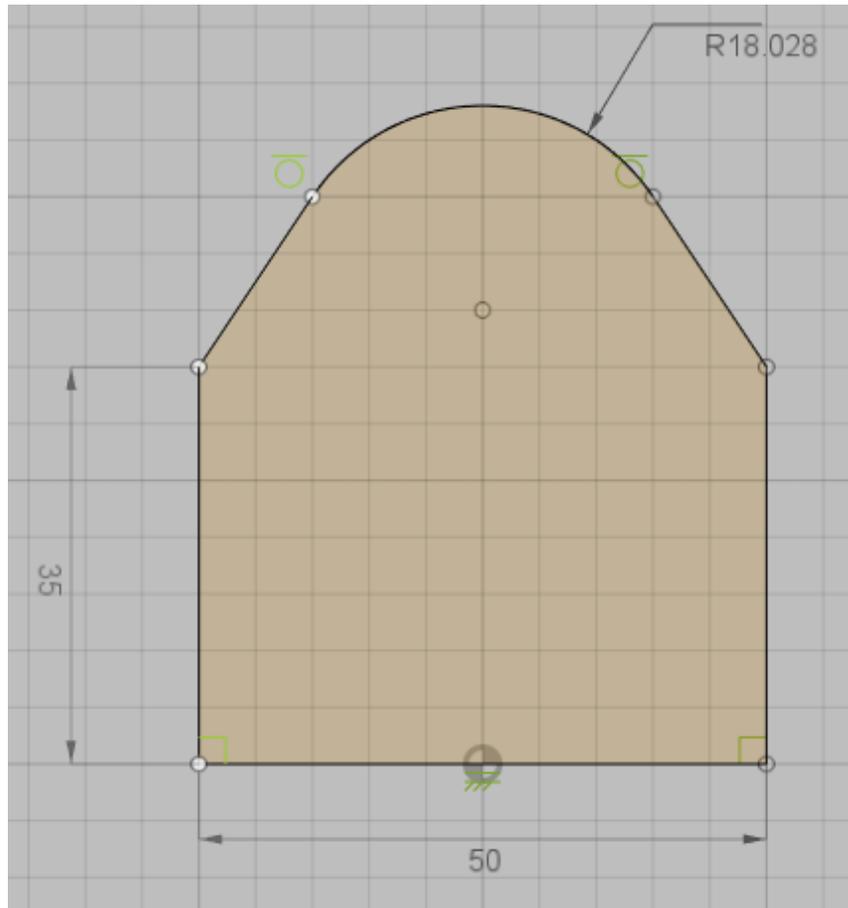


Creates a circle using two user-specified points. The points define the position and the size of the circle.

Create a circle

- 1 Expand the Sketch pull-down then click Circle Two Point.
- 2 Click in the graphics window to specify the first point of the circle's diameter.
- 3 To specify the diameter of the circle:
 - Click a point in the graphics window.
 - Enter a value in the diameter field then press <Enter>. Click a point to create the circle.
- 4 Click again to begin creating a new circle or right-click then select Cancel to end the command.

Sketch Dimension



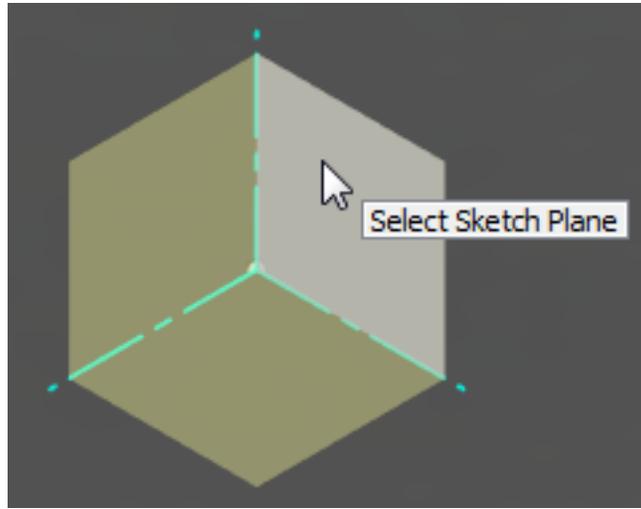
Creates a new sketch on the selected plane or face.

Dimension sketch geometry

- 1 Expand the Sketch pull-down then click Sketch Dimension.
- 2 Select the geometry to dimension. Multiple geometries can be selected. For example, you can select two lines to create an angle dimension or a line and a point to get a distance.
- 3 Pick a point to place the dimension.

- 4 Select other geometry to continue dimensioning or right-click then select Cancel to end the command.

Create Sketch



Creates a new sketch on the selected plane or face.

Create a sketch

- 1 Expand the Sketch pull-down then click Create Sketch.
- 2 Select the plane or face to create the sketch on. Valid selections are:
 - Origin planes
 - Work planes
 - Flat faces on geometry.

NOTE If you select an existing sketch entity or sketch node in the browser, that sketch is edited and a new sketch is not created.

Edit a sketch

There are two methods to edit a sketch.

- 1 Right-click the sketch node in the browser then select Edit Sketch.

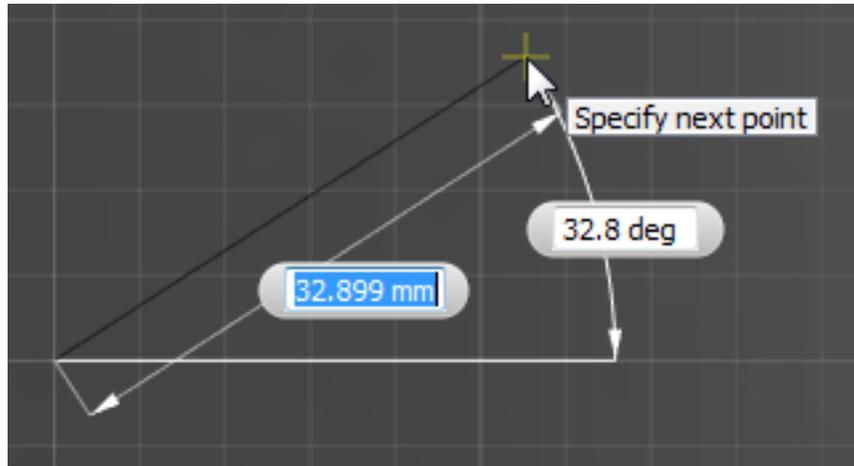
- 2 Start the Create Sketch command then select a sketch entity or the sketch node in the browser.

Line



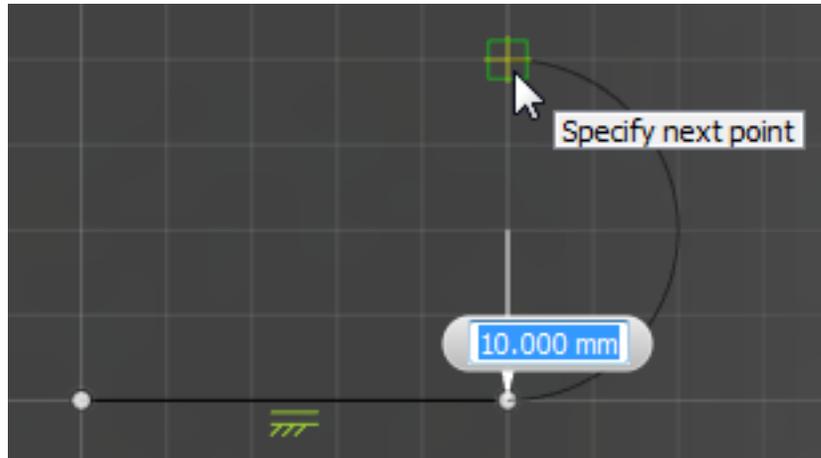
The Line command creates a series of lines. A line segment is created each time you click.

Create lines



- 1 Click the Line command in the Sketch pull-down.
- 2 Click to place the starting point for the line.
- 3 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.
- 4 Continue to place points for additional line segments.
- 5 Right-click then select Cancel to end the command.

Create arcs



A tangent arc is created if you hold the left mouse button and drag while the line command is active.

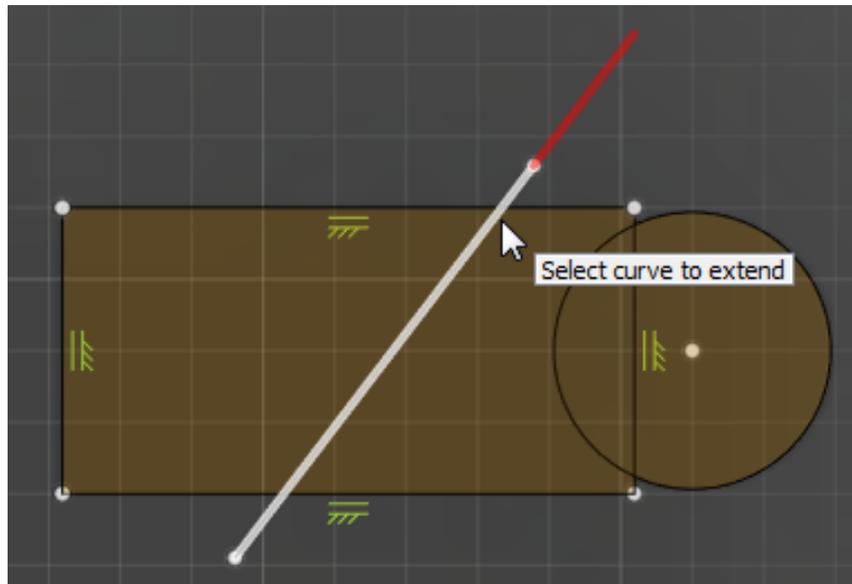
- 1 Pause over the last end point when drawing lines, or pause over an existing end point to add an arc.
- 2 Press the left mouse button and drag to preview the arc radius.
- 3 Move the cursor to other side of the line to change the direction of the arc.
- 4 Click to place the end point, or enter a value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

Extend



Extends the selected geometry to the next intersection of geometry. If no intersection exists, the geometry cannot be extended.

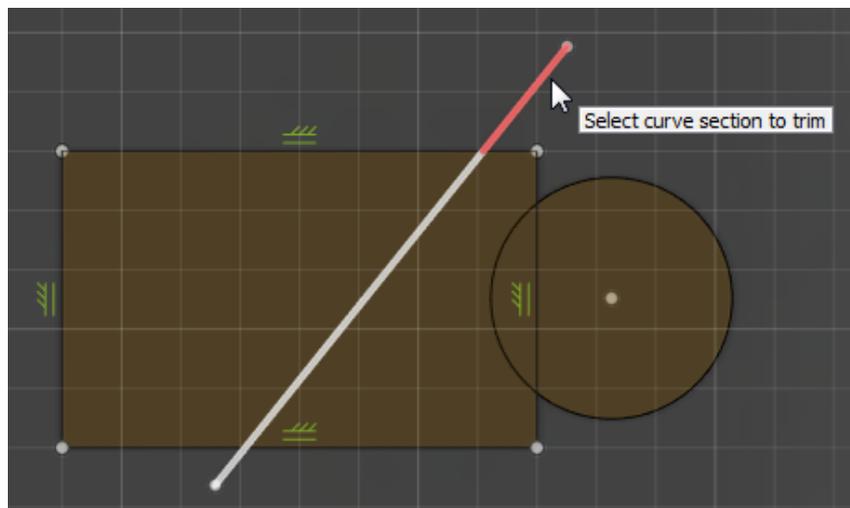
The Extend command will project existing geometry to extend to the theoretical intersection.



Extend an entity

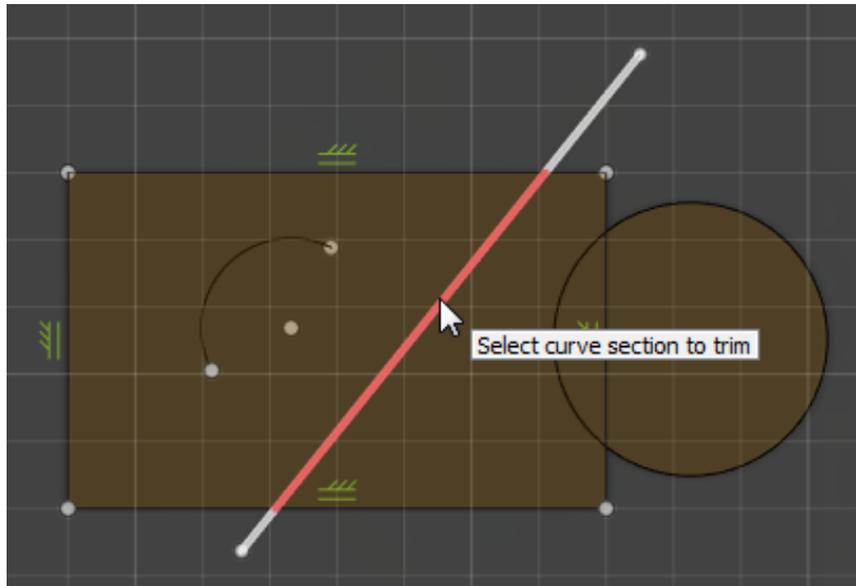
- 1 Expand the Sketch pull-down then select Extend.
- 2 Hover over sketch geometry to see a preview of the extension.
- 3 Click geometry to extend to the next intersection.
- 4 Continue clicking geometry to extend.
- 5 Right-click then select Cancel to end the command.

Trim

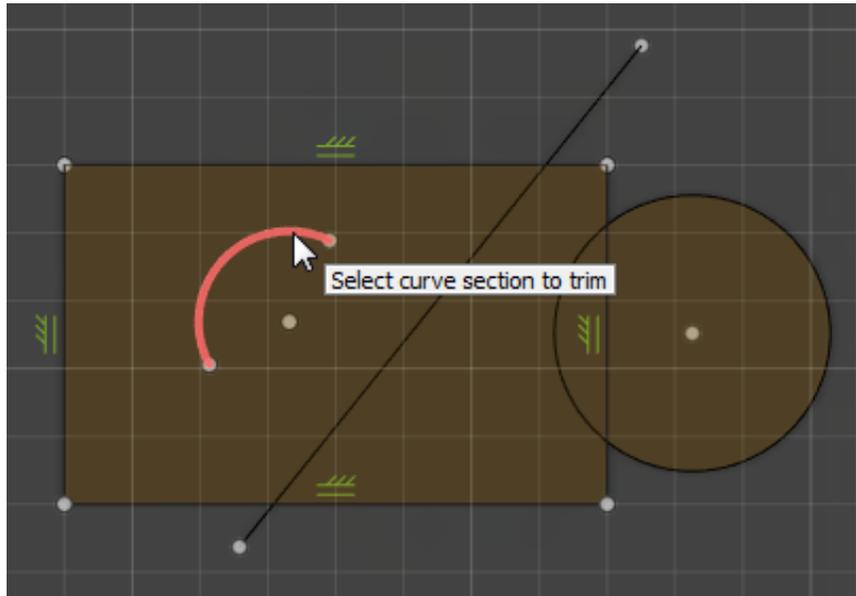


Removes a portion of the selected geometry up to the next intersection with existing geometry. If no intersection exists, the selected entity is deleted.

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.



Trim an entity

- 1 Expand the Sketch pull-down then select Trim.
- 2 Hover over sketch geometry to see a preview of the trim.
- 3 Click geometry to trim to the next intersection.
- 4 Continue clicking geometry to trim.
- 5 Right-click then select Cancel to end the command.

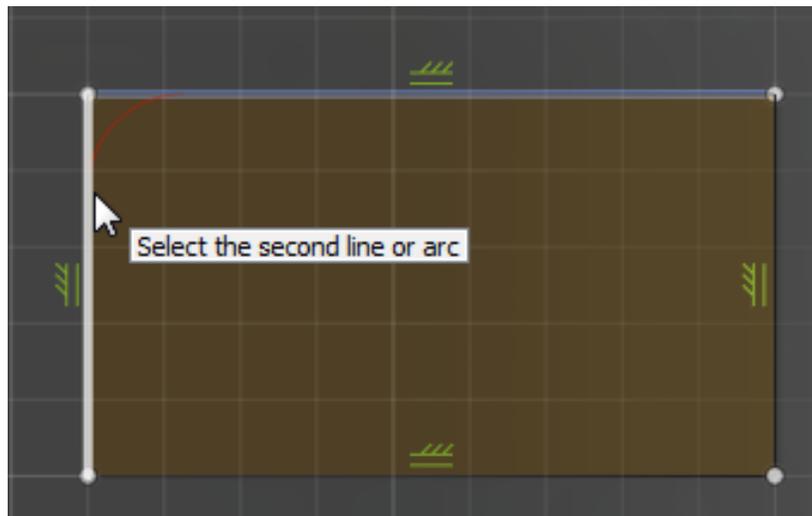
Fillet



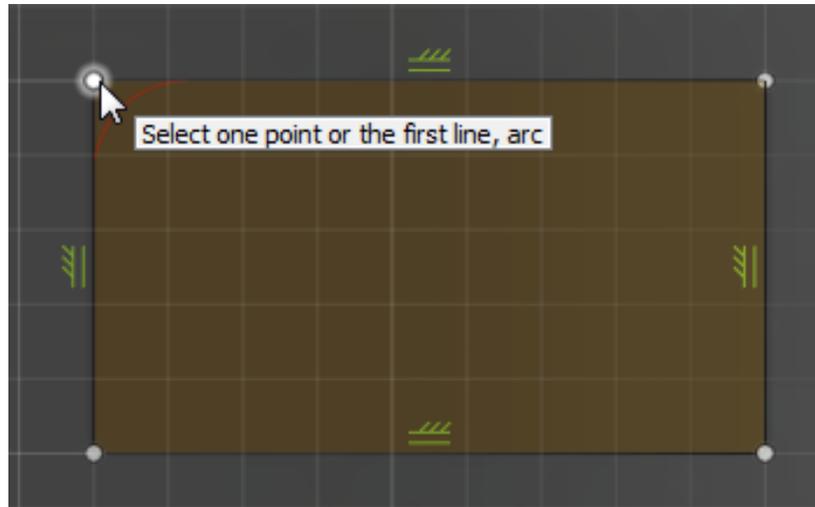
The Fillet command places an arc of a specified radius at a corner or intersection of two lines.

You can create fillets between:

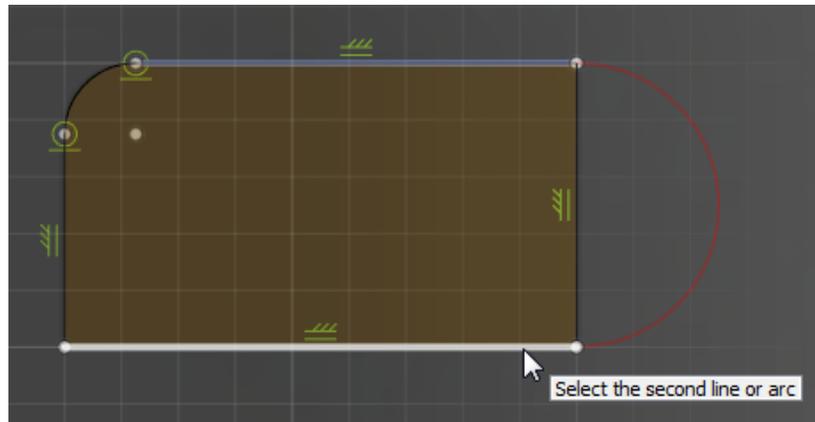
- Two intersecting lines



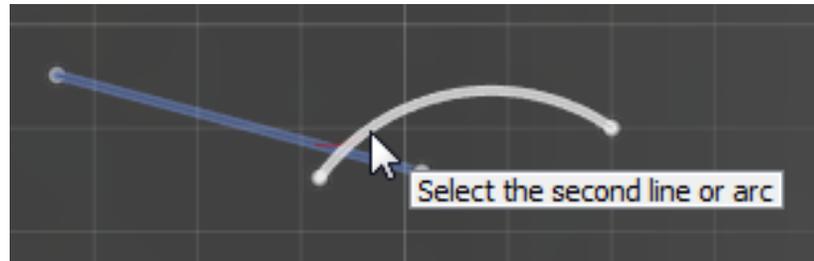
- Point at two intersecting lines



- Two parallel lines



- A line and an arc that intersect

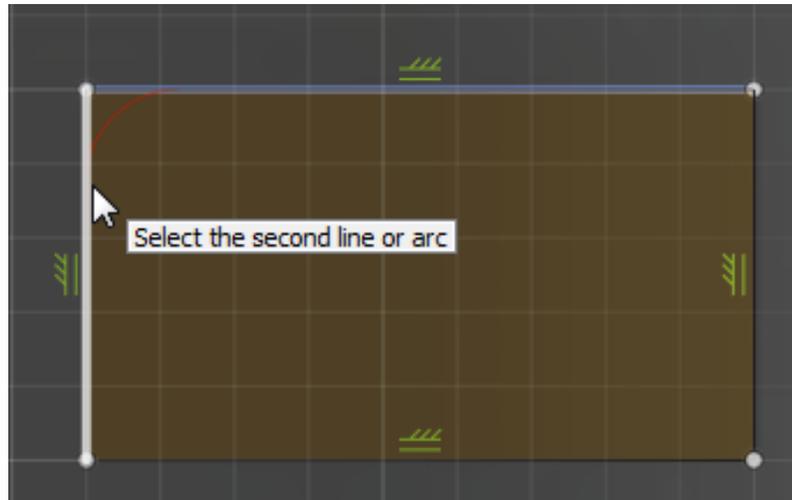


- Two circular arcs that intersect

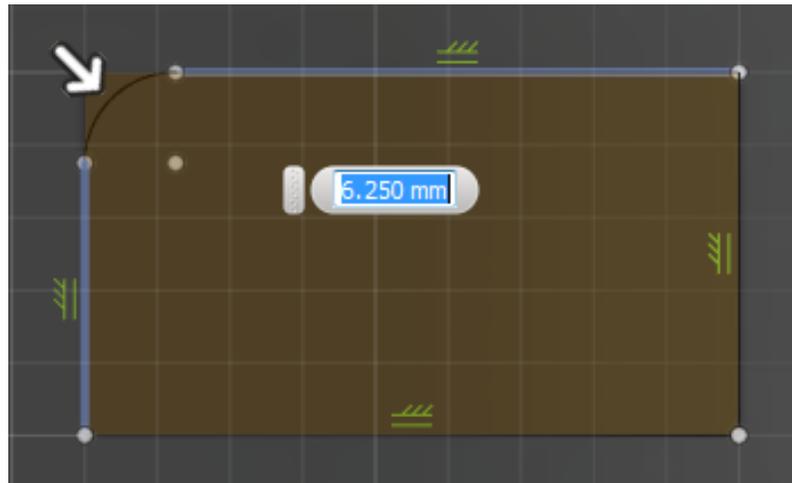


Fillet entities

- 1 Expand the Sketch pull-down then select Fillet.
- 2 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.
- 3 Pause the mouse over the second entity. The fillet preview highlights for a valid selection.



- 4 Click to select the second entity.
- 5 The fillet previews with a value that is proportional to the smaller entity. Enter a value in the box, or drag the arrow to change the size.



- 6 Right-click then select OK or press <Enter> to complete the command.

Grid Off

Turns off the display of the sketch grid when a sketch is active.

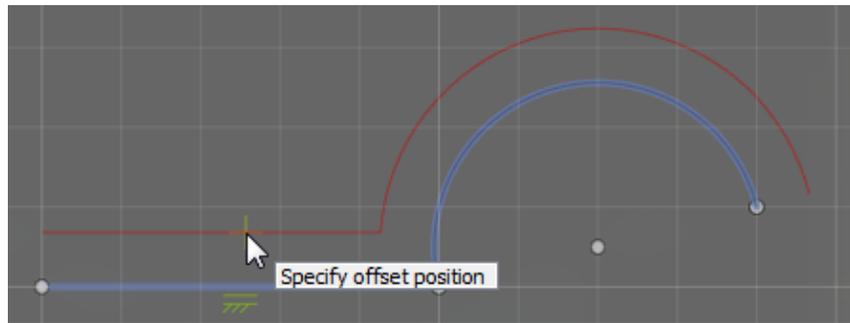
Expand the Sketch pull-down then click Grid Off.

Grid On

Turns on the display of the sketch grid when a sketch is active.

Expand the Sketch pull-down then select Grid On.

Offset



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.

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

Offset sketch entities

- 1 Expand the Sketch pull-down then select Offset.
- 2 In the graphics area, select an individual entity, a chain of connected entities, or a profile.

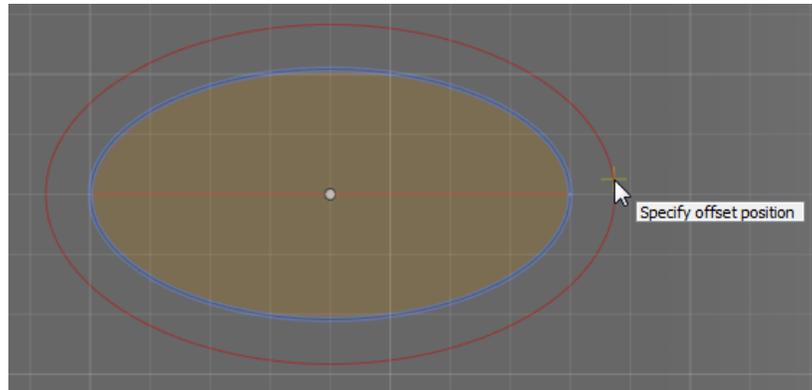
A valid selection highlights when you pause the cursor over it

- 3 Move the cursor then click to create the offset.
- 4 Select another entity to offset or right-click then select Cancel end the command.

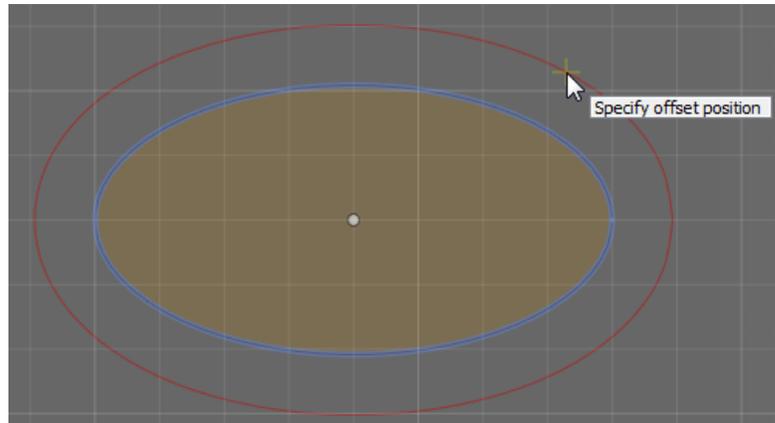
Offset an ellipse

- Select an 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.

Project

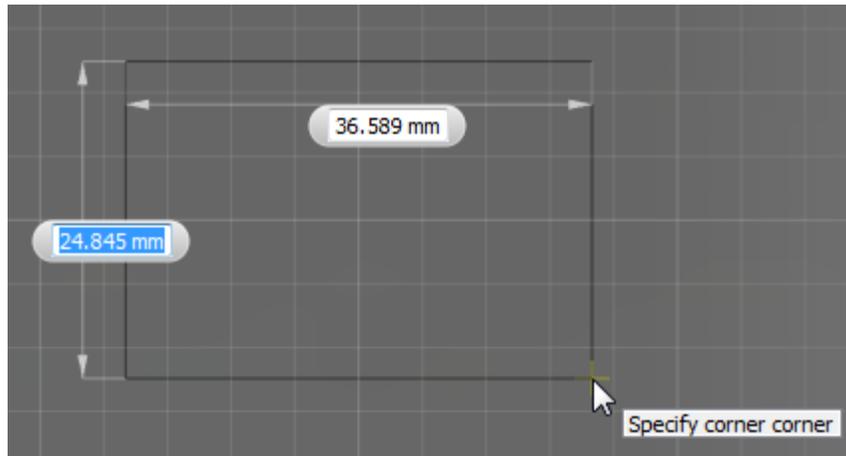
Use project geometry to create copies of existing model geometry and sketch entities in the active sketch.

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.

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

Rectangle Center Corner



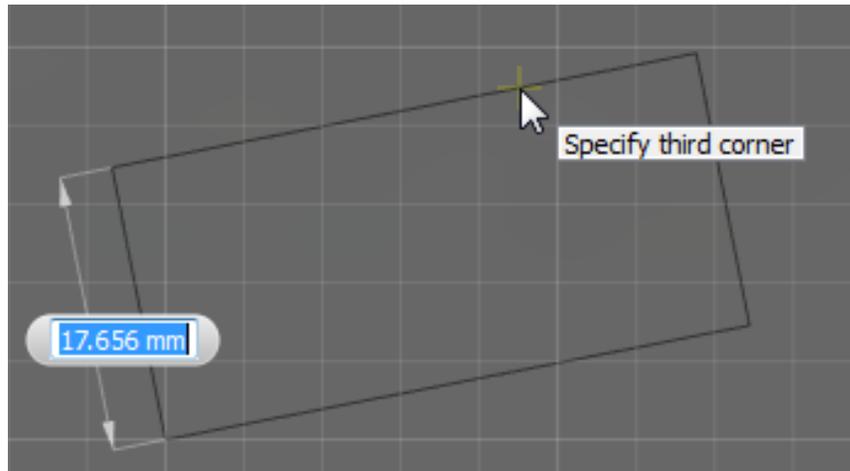
Create a rectangle by selecting the center point then a corner. The first point defines the position of the rectangle. The second point defines the length and width.

Parallel and perpendicular constraints are applied to the lines.

Create a rectangle using the center point and a corner

- 1 Expand the Sketch pull-down then select Rectangle Corner Center.
- 2 Click in the graphics window to specify the first center of the rectangle.
- 3 To specify the corner:
 - Click a point in the graphics window.
 - Enter values in the length and width fields then press <Enter>. Click a point to define the position of the rectangle.
 - Enter a value in one field then press <Enter> to lock the value. Click a point to define the other value.
- 4 Click a point to begin another rectangle or right-click then select Cancel to end the command.

Rectangle Three Point



Create a 3-point rectangle

Create a rectangle using three user-specified points. The selected points define the position and size of the four lines.

Parallel and perpendicular constraints are applied to the lines.

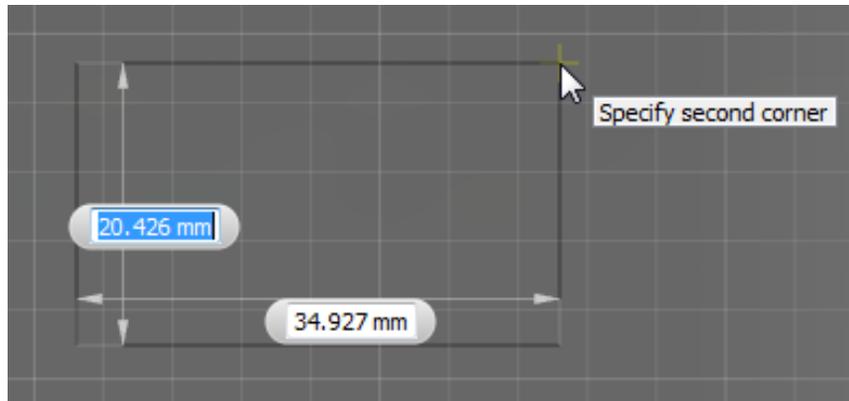
NOTE You cannot infer a parallel or perpendicular constraint to existing sketch entities, but you can infer a horizontal or vertical constraint for the first side.

Create a rectangle using three points

- 1 Expand the Sketch pull-down then select Rectangle Three Point.
- 2 Click in the graphics window to specify the first corner.
- 3 To specify the second point:
 - Click a point in the graphics window.
 - Enter a value in the field then press <Enter> to lock the value. Click a point to specify the angle of the line.
- 4 To specify the third point:
 - Click a point in the graphics window.
 - Enter a value in the field then press <Enter> to lock the value. Click a point to specify the position of the rectangle.

- 5 Click a point to begin another rectangle or right-click then select Cancel to end the command.

Rectangle Two Point



Create a rectangle by selecting two points to define the opposite corners of the rectangle.

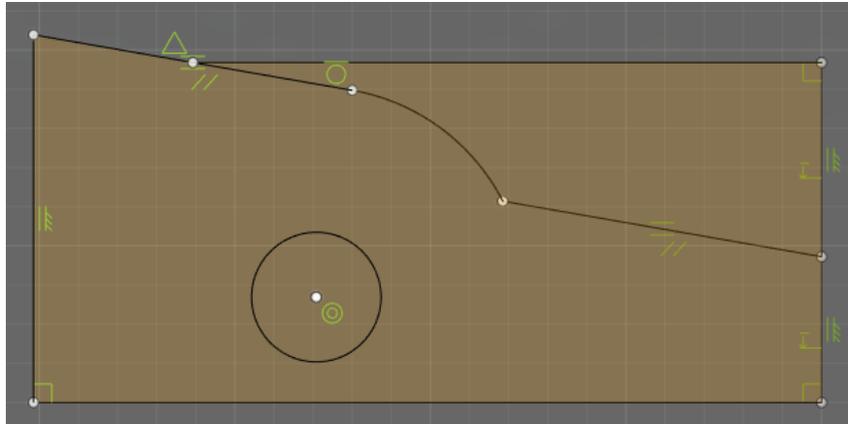
Horizontal and vertical constraints are applied to the lines.

Create a rectangle using two points

- 1 Click Rectangle Two Point on the Sketch pull-down.
- 2 Click in the graphics window to specify the first corner.
- 3 To specify the opposite corner:
 - Click a point in the graphics window.
 - Enter values in the length and width fields then press <Enter>. Click a point to define the position of the rectangle.
 - Enter a value in one field then press <Enter> to lock the value. Click a point to define the other value.
- 4 Click a point to begin another rectangle or right-click then select Cancel to end the command.

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.

Add constraints to existing geometry using the Constraints command in the Sketch pull down.

Constraint types

Con- straint	Glyph	Description
Coincident		Point to point or snap Point to entity
Collinear		Manually added only
Concentric		Circle, arc, or ellipse have the same center point.

Constraint	Glyph	Description
Midpoint		Point to midpoint of line
Fix/Unfix	Green geometry	Locks the position of the geometry
Parallel		Second line parallel to first line
Perpendicular		Second line perpendicular to first line
Horizontal		Line parallel to horizontal sketch axis
Vertical		Line parallel to vertical sketch axis
Tangent		Inferred to circles, arcs Manually added to circles, arcs, lines
Equal		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 Parallel
- 6 Horizontal
- 7 Vertical

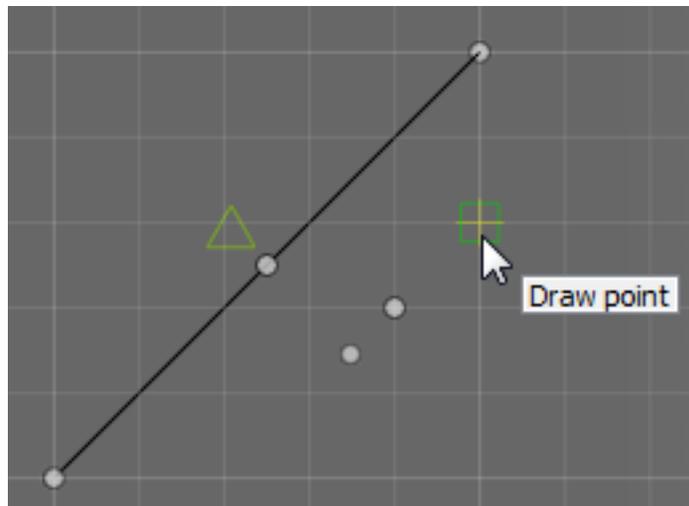
The equal and collinear constraints cannot be inferred. You can manually add these constraints after creating the entities.

Add Constraints Manually

Use the Constraints command in the Sketch pull down to manually constrain geometry.

- 1 Expand the Sketch pull-down then click Constraints.
- 2 Select the constraint type from the mini-toolbar.
- 3 Select the geometry to apply the constraint.
- 4 Continue applying constraints as needed.
- 5 Right click then select OK to end the command.

Point



Place sketch points by clicking in the graphics window. You can quickly place a series of points at precise locations, and then create sketch entities by connecting the dots. Sketch points can also be used to position hole features.

Create sketch points

- 1 Expand the Sketch pull-down then click Point.
- 2 Click in the graphics window to create a point. You can snap to the grid or to existing geometry.
- 3 Click again to create additional points.
- 4 Right-click then select Cancel to end the command.

Stop Sketch

Exits the active sketch.

- On the Sketch pull-down, click Stop Sketch.
- On the marking menu, select Sketch then Stop Sketch.

Joints

Use the Joints commands to create and manage joints in Inventor Fusion.

Joint Origin

Use the Joint Origin command to define a joint origin on a component. Joint origins define the geometry used to relate a joint's components. For example, a revolute joint binds a point and an axis. The points and axes are specified by joint origins.

Joint origins can also be created while in the Joint command. The Joint Origin command provides more positioning options for the origins that are available while in the Joint command.

Place joint origins

- 1 Switch the toolbar workspace to Assemble.
- 2 Click Joint Origins in the Joints pull-down.
- 3 Select the face or edge to specify the position of the joint origin.
- 4 Optional, click Primary Axis in the dialog box and select a face or edge to define the orientation of the origin.

- 5 Optional, click Secondary Axis in the dialog box and select a face or edge to further define the orientation of the joint origin.
- 6 Right-click then select OK.

Joint Origin dialog box

Origin

Select the face or edge to define the position of the joint origin.

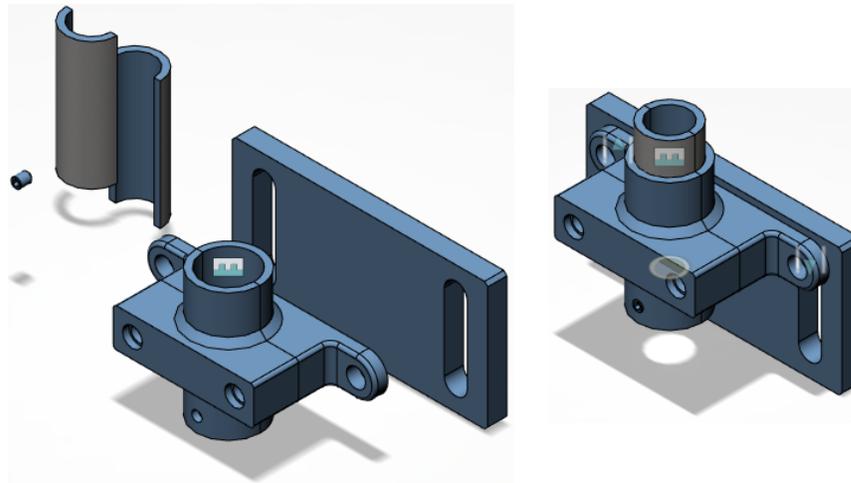
Primary Axis

Select a face or edge to define the primary axis orientation of the joint origin.

Secondary Axis

Select a face or edge to define the secondary axis orientation of the joint origin.

Joint



Use the Joint command to define joints between components. Joints are used to position component occurrences relative to other occurrences in a model.

Position components using joints

- 1 Switch the toolbar workspace to Assemble.
- 2 Click Joints in the Joints pull-down.

- 3 Select a joint origin or geometry to define an origin. The component occurrence selected for the first joint origin cannot be grounded or the joint origin cannot be on a body in the main assembly.

NOTE When selecting geometry, hold the <Command> key to lock the selection to the highlighted geometry then select the desired point for the origin location.

- 4 Select a second joint origin or geometry to define a second origin. The first joint origin is aligned with the second to show the component relationship for a joint value of zero. This is a temporary positioning used while the joint is defined.
- 5 A joint type is selected based on the geometry selected for the joint origins. Change the joint type in the Type pull-down if necessary. The components will animate to display the allowed motion for the joint.
- 6 Depending on the type of joint selected, you can change the Rotation Axis, Slide Axis, or Slide Plane.
- 7 Adjust the joint origin alignment by setting the alignment offset and angle. You can enter values in the fields or drag the manipulators in the graphics window. These adjustments allow you to tweak the relative position where the joint's values are zero.
- 8 Click Flip to reverse the orientation, if necessary.
- 9 Right-click then select OK to create the joint.

A single joint origin can be used to create multiple joints. The joint origin must be displayed to select it when creating joints. Use the light bulb icon in the browser to control the visibility of joint origins.

Joint dialog box

Joint Origins

Select the joint origins or select geometry to define the joint origins.

Angle

Specify an angle to rotate the joint origins.

Offset

Specify a value to offset the joint origins.

Flip

Reverses the orientation of the joint.

Type

Select the type of joint to create. Depending on the type selected, you can change the orientation of the allowed movement.

- Rotation Axis
- Slide Axis
- Slide Plane

On Change

Shows the motion allowed by the joint when a change is made to the joint type.

Animate

Turns on the animation of the joint. Uncheck the box to end the animation.

Move Joints

Offsets the joint a specified distance.

Move joints

- 1 Switch the toolbar workspace to Assemble.
- 2 Expand the Joints pull-down then select Move Joints.
- 3 Select the joint glyph in the graphics window or select the joint in the browser.
- 4 Drag the manipulator or enter a value to specify the distance.
- 5 Right-click then select OK.

Move Joints dialog box

Joint Input

Select the joint to modify.

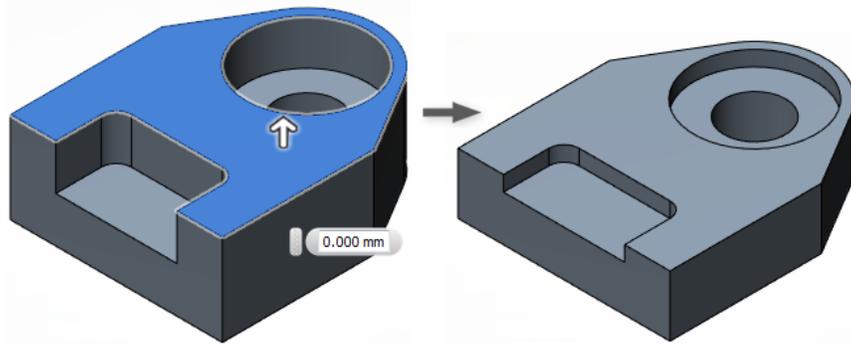
Distance

Specify the distance to offset.

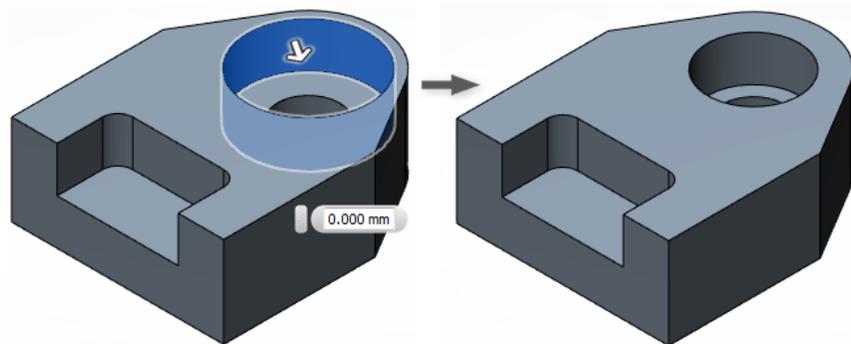
Modify

Use the Modify commands to make changes to solids, surfaces, and components in Inventor Fusion.

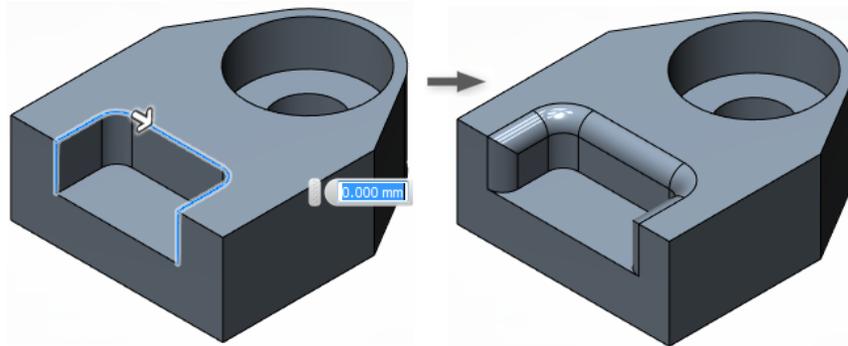
Press Pull



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.



Use Press Pull to edit fillet, holes, and chamfers.



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.

How to modify geometry using Press Pull

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Press Pull in the Modify pull-down.
- 3 Select the faces to modify.
- 4 Drag the arrow manipulator or enter a value in the field in the dialog box.
- 5 Right-click then select OK to accept the change.

Press Pull dialog box

Selection

Enables the selection of faces or edges.

Distance/Radius

Specifies the distance to move the face or the radius of the arc face.

Tweak Faces

Use Tweak Face to modify the 3D model. Tweak face moves the selected faces and trims or extends adjacent faces.

Edit geometry using Tweak

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Tweak Faces.
- 3 Select the faces to tweak.
- 4 Drag a manipulator or enter values in the dialog box.
- 5 Right-click and select OK.

NOTE You can snap to existing geometry when tweaking faces. Drag a manipulator to activate it then select the geometry to snap to.

Tweak dialog box

Faces

Enables the selection of faces to tweak.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Scale Command

Enlarges or reduces selected bodies based on a specified scale factor.

Scale an object

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Scale.
- 3 Select the body that you wish to scale.
- 4 Click Point in the dialog box then select a reference point to scale around.
- 5 Select Uniform or Non Uniform for the Scale Type
- 6 Set the Scale Factor:
 - For Uniform scale, drag the arrow manipulator or enter a value for Scale Factor.
 - For Non Uniform scale, drag the arrow manipulators or enter values for the X Distance, Y Distance, and Z Distance.
- 7 Right-click then select OK to accept the changes.

Scale dialog box

Scale Type

- **Uniform** Uses a single scale factor to scale the object.
- **Non Uniform** Use a different scale factor to scale the object in the X, Y, and Z direction.

Bodies

Enables the selection of bodies to scale.

Point

Enables the selection of a reference point to scale around.

Scale Factor

For Uniform scale, specifies the value for the scale.

X Distance

For Non Uniform, specifies the value to scale in the X direction.

Y Distance

For Non Uniform, specifies the value to scale in the Y direction.

Z Distance

For Non Uniform, specifies the value to scale in the Z direction.

Move

Use the Move command to move solid bodies, surface bodies, and work geometry.

Move bodies

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Move.
- 3 Select the bodies, or work geometry to move.
- 4 Drag a manipulator or enter values in the dialog box.
- 5 Right-click and select OK.

Move dialog box

Selection

Enables the selection of bodies and work geometry to move.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Move Components

Use the Move Components command to change the position of component instances.

Move components

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Modify pull-down then select Move Components.
- 3 Select the components to move.
- 4 Drag the manipulators or enter values in the dialog box.
- 5 Right-click then select OK.

Move Components dialog box

Components

Enables the selection of components to move.

X Distance

Specifies the distance to move in the X direction.

Y Distance

Specifies the distance to move in the Y direction.

Z Distance

Specifies the distance to move in the Z direction.

X Angle

Specifies the angle to rotate around the X axis.

Y Angle

Specifies the angle to rotate around the Y axis.

Z Angle

Specifies the angle to rotate around the Z axis.

Move Joints

Offsets the joint a specified distance.

Move joints

- 1 Switch the toolbar workspace to Assemble.
- 2 Expand the Joints pull-down then select Move Joints.
- 3 Select the joint glyph in the graphics window or select the joint in the browser.
- 4 Drag the manipulator or enter a value to specify the distance.
- 5 Right-click then select OK.

Move Joints dialog box

Joint Input

Select the joint to modify.

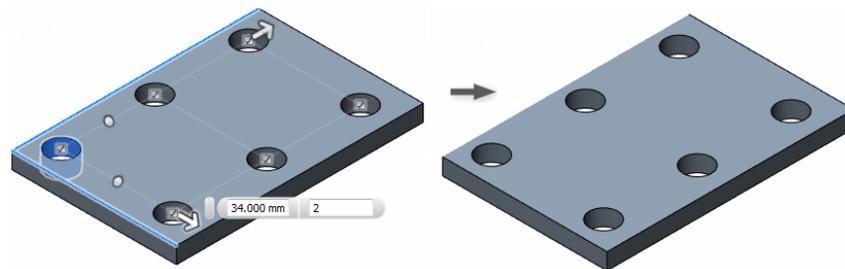
Distance

Specify the distance to offset.

Pattern

Use the Pattern commands to pattern or mirror geometry in Inventor Fusion.

Rectangular Pattern



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.

You can pattern faces, features, or bodies.

Create rectangular patterns

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Rectangular Pattern in the Pattern pull-down.
- 3 In the graphics area or the browser, select the faces, features, or bodies to pattern.
- 4 Click Directions, and select a linear edge or work axis. Perpendicular manipulators display.
- 5 For a non-perpendicular pattern, select a second linear edge or work axis.
- 6 Use the arrows to set the U Distance and V Distance or enter values in the fields.
- 7 Use the dot manipulator to set the U Grid Entries and V Grid Entries or enter values in the fields.
- 8 Click a check mark to suppress an occurrence.
- 9 Right-click the select OK to create the pattern.

Rectangular Pattern dialog box

Objects

Enables the selection of objects to pattern. Valid objects are faces, features, or bodies.

Directions

Enables the selection of edges to control the U and V directions.

Suppress

Displays check boxes in the graphics window at each occurrence. Uncheck a box at an occurrence to suppress that occurrence from the pattern.

U Grid Entries

Specifies the number of occurrences along the first direction.

U Distance

Specifies the overall distance in the first direction. Occurrences are spaces evenly along the distance.

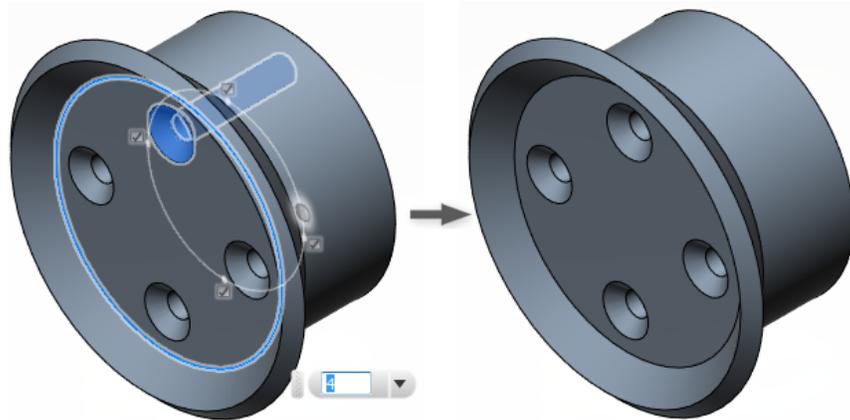
V Grid Entries

Specifies the number of occurrences along the second direction.

V Distance

Specifies the overall distance in the second direction. Occurrences are spaced evenly along the distance.

Circular Pattern



The circular pattern command copies selected objects around a selected axis. You can pattern faces, features, or bodies.

The axis for a circular pattern can be a sketch line, a linear edge, a work axis, a cylindrical face, or a circular edge.

Create a circular pattern

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Expand the Pattern pull-down then click Circular Pattern.
- 3 In the graphics area or the browser, select the faces, features, or bodies to pattern.
- 4 Click Circular Axis in the dialog box then select a sketch line, linear edge, a work axis, a cylindrical face, or a circular edge.
- 5 Use the ball manipulator to set the number of occurrences or enter a value in the field.
- 6 Click a check mark to suppress an occurrence.
- 7 Set the Circular Type to Full or Angle.
 - **Full** The copies are evenly spaced around a complete circle.
 - **Angle** The copies are evenly spaced around the specified angle.

- 8 If Circular Type is set to angle, enter the value in the Total Angle field.
- 9 Right-click the select OK to create the pattern.

Circular Pattern dialog box

Objects

Enables the selection of objects to pattern. Valid objects are faces, features, or bodies.

Circular Axis

Enables the selection of the axis to pattern around. Valid objects are a sketch line, linear edge, a work axis, a cylindrical face, or a circular edge.

Suppress

Displays check boxes in the graphics window at each occurrence. Uncheck a box at an occurrence to suppress that occurrence from the pattern.

Total Angle

Available when the Circular Type is set to Angle. Specifies the total angle to space the occurrences around.

Count

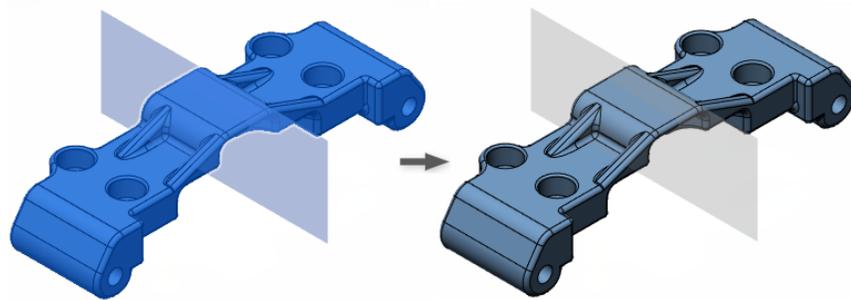
Specifies the total number of copies.

Circular Type

Controls how the copies are positioned around the pattern.

- **Full** The copies are evenly spaced around a complete circle.
- **Angle** The copies are evenly spaced around the specified angle.

Mirror



Mirrors the selected objects to the opposite side of a selected face or plane. Valid object types are faces, features, or bodies.

Mirror objects

- 1 Switch the toolbar workspace to Solid, Surface, or Assemble.
- 2 Click Mirror in the Pattern pull-down.
- 3 In the graphics area or the browser, select the faces, features, or bodies to mirror.
- 4 Click Mirror Plane in the dialog box then select the plane or face to mirror around.
- 5 Right-click then click OK to create the mirror.

Mirror dialog box

Objects

Enables the selection of objects to mirror. Valid objects are faces, features, or bodies.

Mirror Plane

Enables the selection of the plane or face to mirror around.

Construction

Work features or construction 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.

Work Planes

A work plane is a construction feature that defines the parametric location of a sketch plane in 3D space. A work plane is useful when no planar face exists to use as a sketch plane, for example, when sketching on curved faces.

Create work planes

Inventor Fusion creates the following types of work planes. Most work planes 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.

Icon	Work Plane Type	Inputs
	Offset Plane	Select a work plane or planar model face and enter the distance.
	Plane At Angle	Select a model edge, sketch line, or work axis, and enter the angle.

Icon	Work Plane Type	Inputs
	Tangent Plane	Select a cylindrical face and enter the angle. The plane is created where you clicked on the cylinder.
	Midplane	Select two parallel planes or planar model faces. The work plane is created halfway between them.
	Plane Through Two Edges	Select two model edges, sketch lines, or work axes.
	Plane Through Three Points	Select three model, sketch, or work points.

Work Axes

A work axis is a construction feature that defines an axis in 3D space. A work axis is useful when no geometry exists to use as an axis.

Create work axes

Inventor Fusion creates the following types of work axes. Work axes are automatically created based on the geometry selections.

Icon	Work Axis Type	Inputs
	Axis Through Cylinder	Select a cylindrical face or a cone.

Icon	Work Axis Type	Inputs
	Axis Perpendicular At Point	Select a model or work plane.
	Axis Through Two Planes	Select two model faces or work planes.
	Axis Through Two Points	Select two vertices, sketch points or work points.
	Axis Through Edge	Select a linear or circular edge on a model or a sketch profile.

Work Points

A work point is a construction feature that defines a point in 3D space. A work point can be projected onto a sketch and incorporated into dimension and constraint schemes.

Create work points

Inventor Fusion creates the following types of work points. Work points are automatically created based on the geometry selections.

Icon	Work Axis Type	Inputs
	Point at Vertex	Select a vertex, work point, or sketch point.

Icon	Work Axis Type	Inputs
	Point Through Two Edges	Select two edges on a model or two models or sketch profiles that intersect.
	Point Through Three Planes	Select three model faces or work planes.
	Point at Center of Circle/Sphere	Select a circular edge or spherical surface.
	Point at Edge And Plane	Select a work plane or model face, or a sketch profile and a work axis or sketch line.

Place

Use the Place commands to place components from an external file into the current Inventor Fusion model.

Place

Place a component in your model.

Place components in a model

- 1 Switch the toolbar workspace to Assemble
- 2 Expand the Place pull-down then click Place.
- 3 Select the file to place then click Open.

- 4 The component is placed at the origin and the Move command is activated. Drag the manipulators or enter values to reposition the component.
- 5 Right-click then select OK.

Place from Autodesk 360

Place a component in your model from Autodesk 360.

Place components from Autodesk 360

- 1 Switch the toolbar workspace to Assemble
- 2 Expand the Place pull-down then click Place from Autodesk 360.
- 3 Log in to your Autodesk 360 account.
- 4 Select the file to place then click Open.
- 5 The component is placed at the origin and the Move command is activated. Drag the manipulators or enter values to reposition the component.
- 6 Right-click then select OK.

Select

Use the select commands to control selection sets in Inventor Fusion.

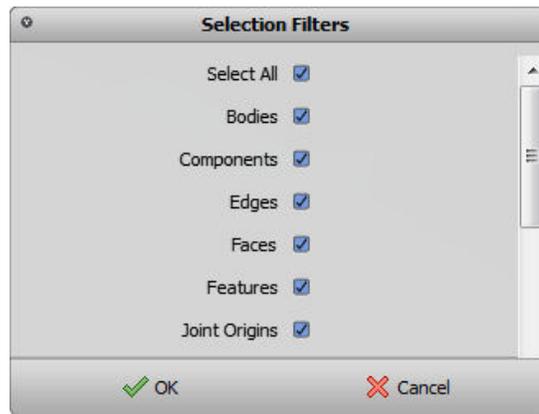
Select

Enables the selection of objects in the graphics window.

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.

Access the Selection Filters command from the Select pull-down in the Solid, Surface, or Assemble workspaces.



The selection filters control only selection in the graphics window. You can still select objects in the browser

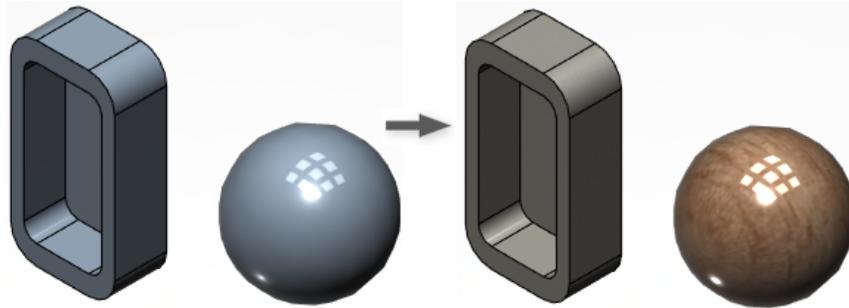
Options

The Options workspace contains commands to control the display and other behaviours of Inventor Fusion.

Material

Use the Material commands to control the physical materials and visual material of components, bodies, and faces in Inventor Fusion.

Physical Material



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 work planes, sketch profiles, and so on, do not have materials.

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, right-click the component in the browser then select Properties.

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.

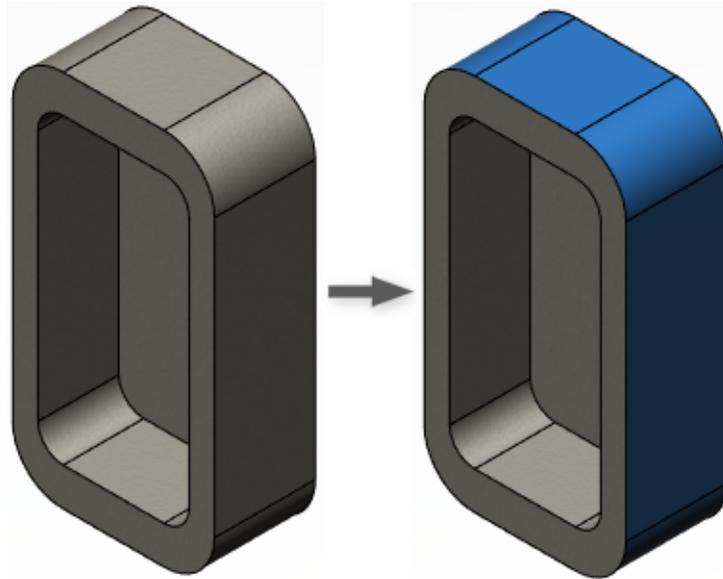
Physical materials can be assigned to bodies or components.

You can change the appearance of a component, body, or face using Visual Styles.

Assign a physical material to a component

- 1 Switch the toolbar workspace to Options.
- 2 Select the body or component in the browser.
- 3 Click the Physical Materials pull-down then select the desired material.

Visual Material



The color of a component is assigned based on the assigned physical material. The color can be overridden using the Visual Material pull-down. The override only applies to the selected occurrence.

Visual materials can be assigned to faces, bodies, and components.

The Visual Material is for appearance only and does not affect the physical properties such as density.

Assign a visual material to a component

- 1 Switch the toolbar workspace to Options.
- 2 Select the face, body, or component in the browser.
- 3 Click the Visual Materials pull-down then select the desired material.

Display

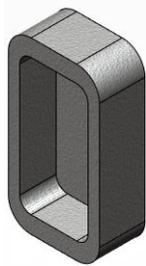
Use the Display commands to control the appearance of objects in the graphics window in Inventor Fusion.

Visual Style

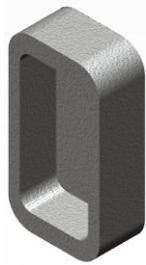
You can turn the visibility of edges on or off in your model. In the Options workspace, select the Visual Style pull-down. The Visual Style is an application setting. Your current setting is restored the next time you start the application.

The Shaded With Edges option draws both faces and edges. The Shaded option draws only faces.

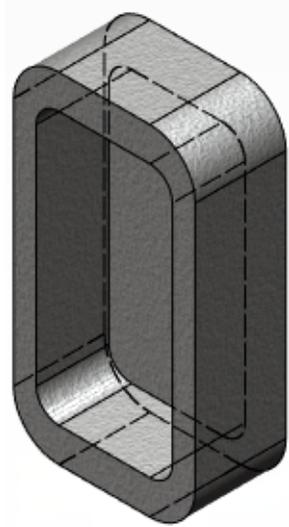
The **Shaded with Edges** option draws both faces and edges.



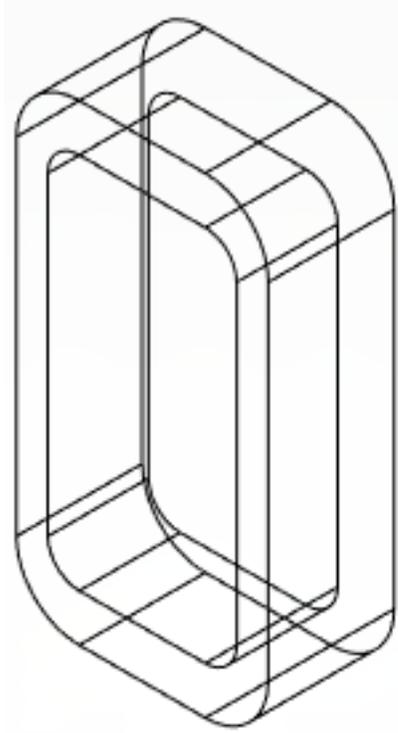
The **Shaded** option draws only faces.



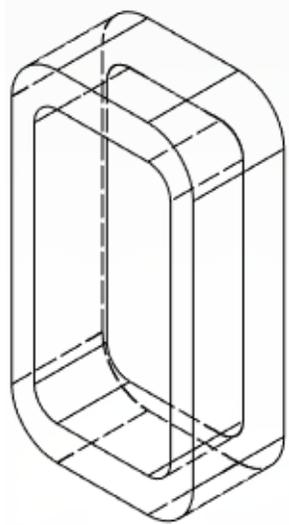
The **Shaded with Hidden Line** option displays the shaded model with hidden lines in a dashed line style.



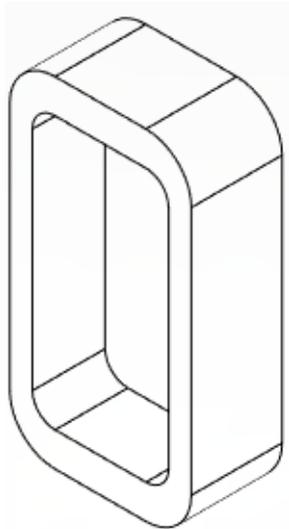
The **Wireframe** option displays model edges and hidden lines with the same line style.



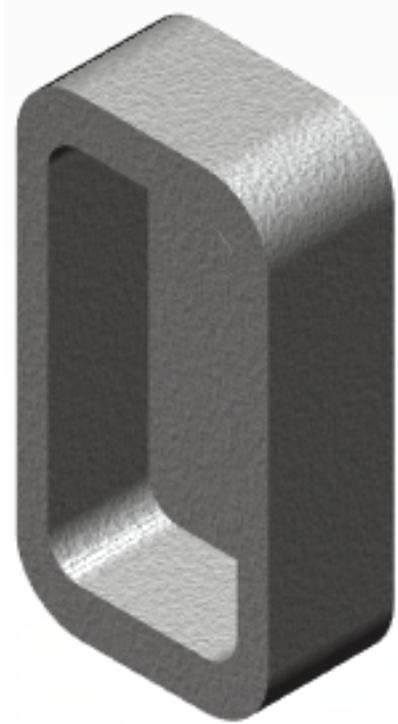
The **Wireframe (Hidden Line)** displays model edges and hidden lines. The hidden lines are displayed with a dashed line style.



The **Wireframe (Hidden Line Removed)** displays only model edges.



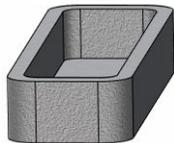
The **Realistic** option displays a realistically textured model with high quality shading.



Camera

Use the Camera options to display the model in orthographic mode or perspective mode.

Orthographic Views



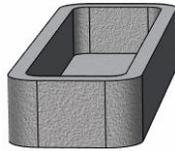
Set the view to Orthographic using a drop-down list in the Display pull-down in the Options toolbar workspace.

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



Set the view to Perspective using a drop-down list in the Display pull-down in the Options toolbar workspace.

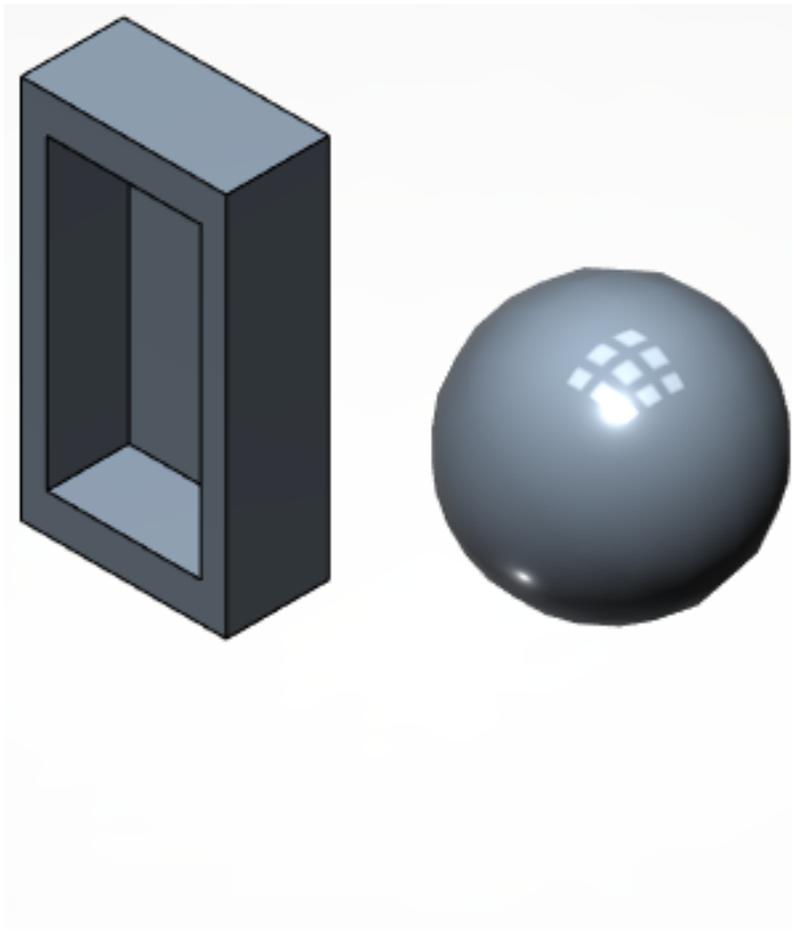
In perspective camera mode, the model displays as a human eye sees it. Objects further away from the camera appear smaller than objects closer to the camera. If two lines are parallel and are along the line of sight, they appear to converge as they move further away from the camera.

Use Perspective Camera mode to do a realistic walkthrough of a scene.

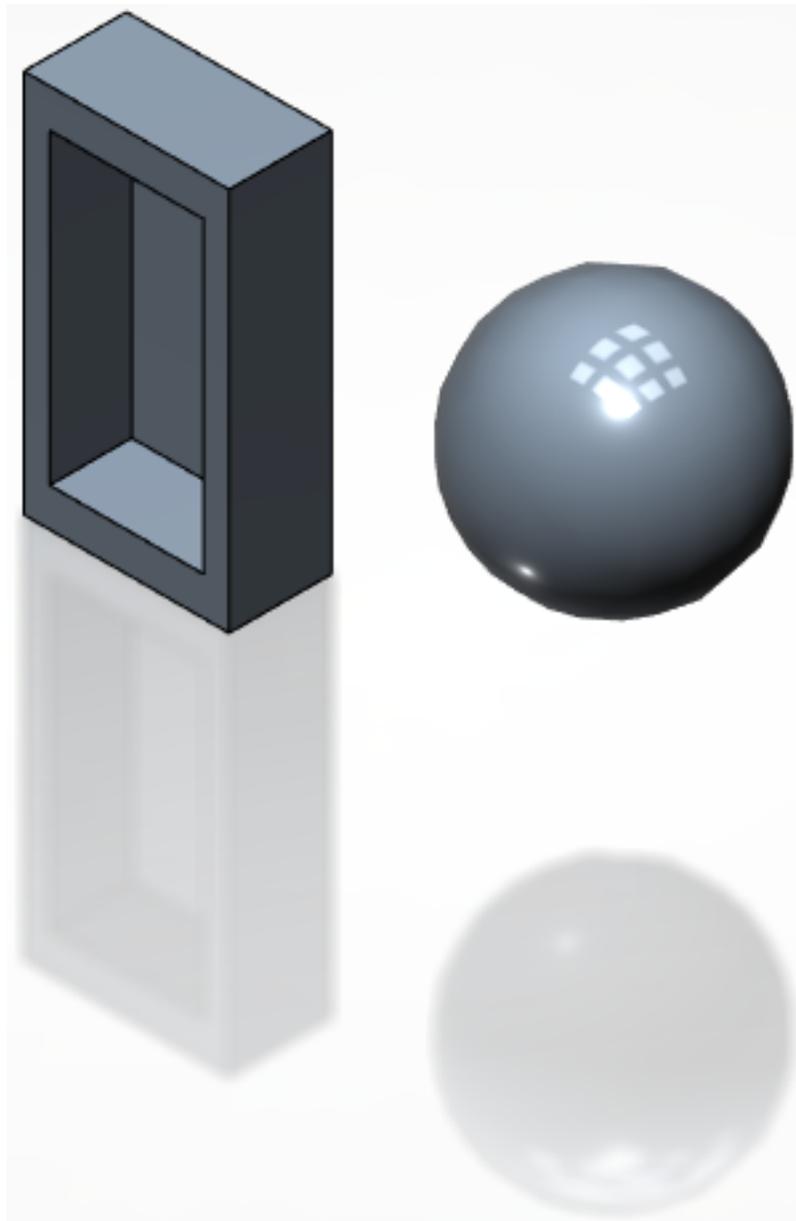
Note: The term camera mode indicates only the particular view method used for models in the graphics window. You cannot record actions that take place in the graphics window by choosing either orthographic camera mode or perspective camera mode.

Ground Reflection

Ground reflection casts a reflection on the floor.



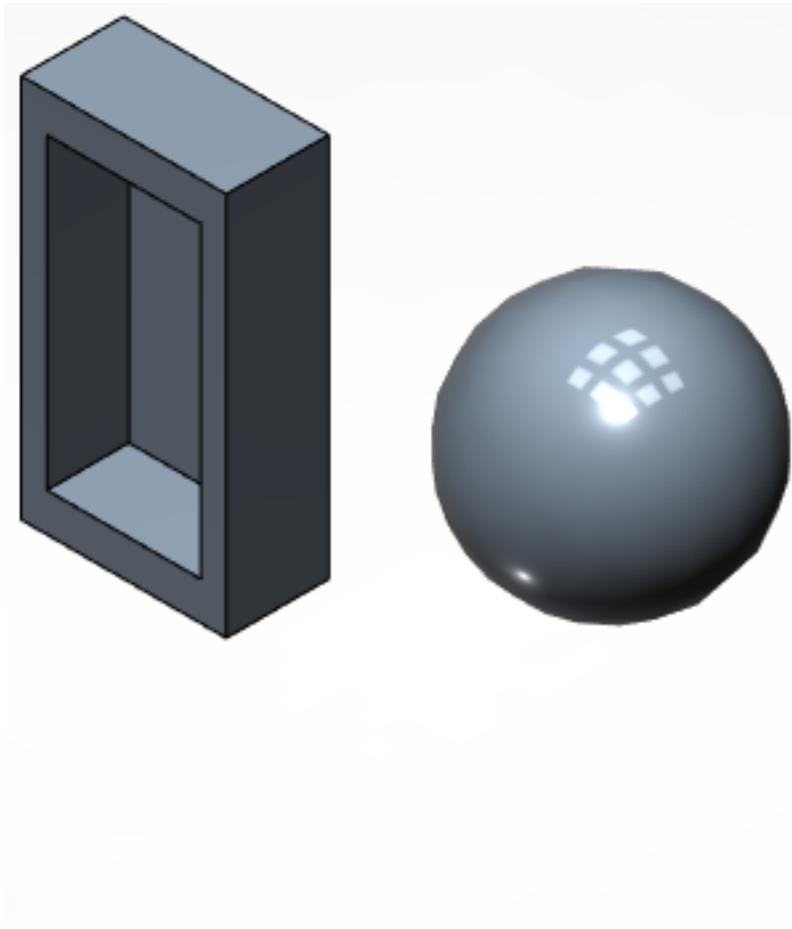
Floor Reflections off



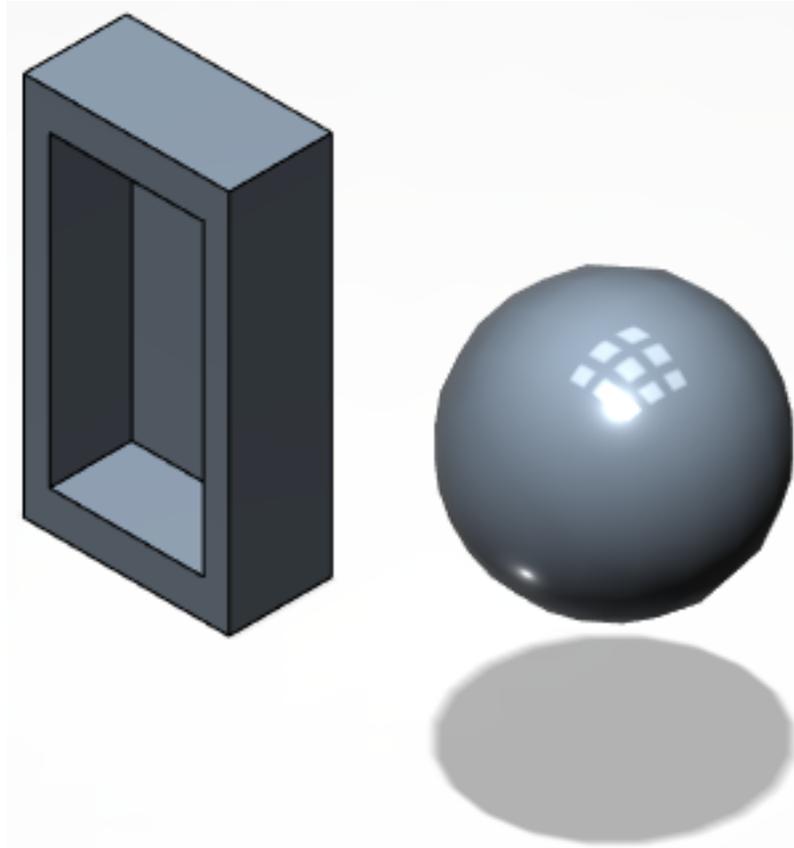
Floor Reflections on

Ground 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 and sketch geometry are not visible in shadows.



Shadow off

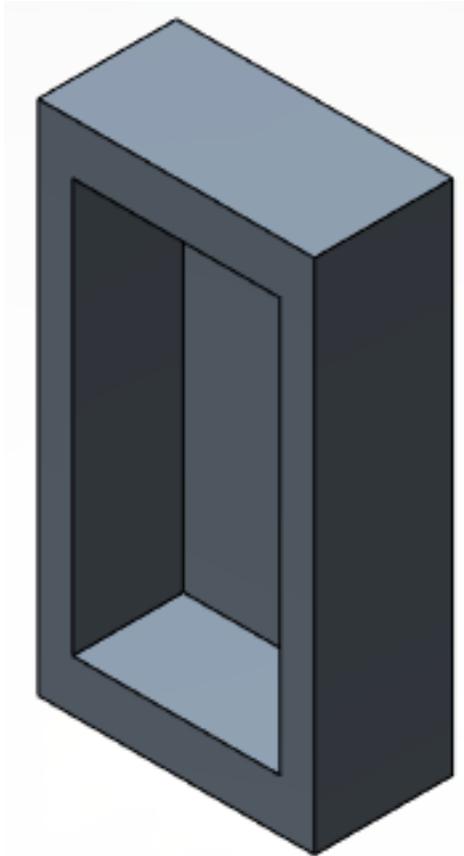


Shadows on

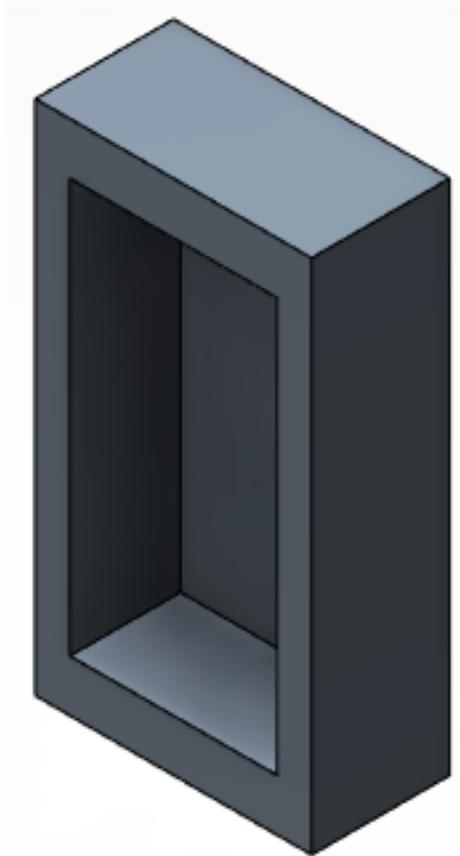
Ambient Occlusion

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.



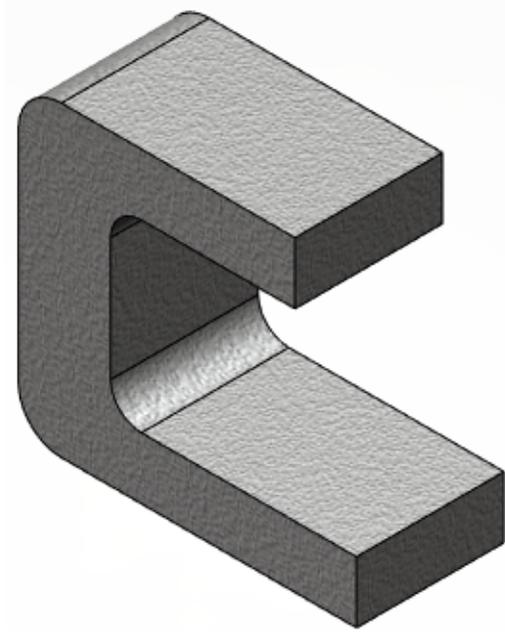
Ambient occlusion off



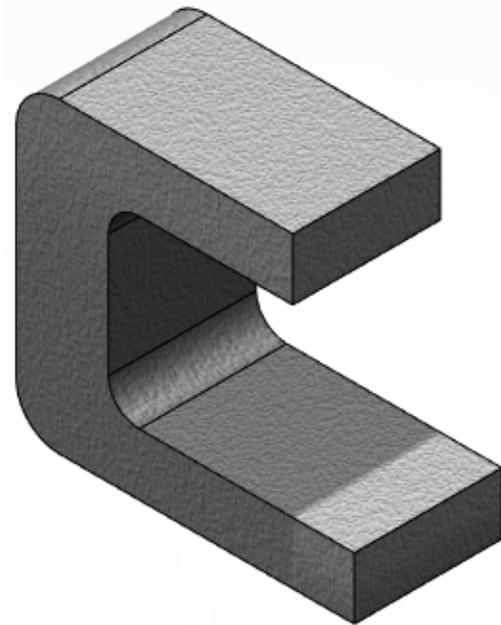
Ambient occlusion on

Object Shadow

Object Shadow casts shadows of the objects onto themselves and other objects in the model.



Object Shadow off



Object Shadow on

Screen

Use the Screen commands to control the size of the Inventor Fusion window.

Enter Full Screen

Switches the interface between Full Screen mode and Normal mode.

Snap

Use the Snap commands to control the display of the grid in Inventor Fusion.

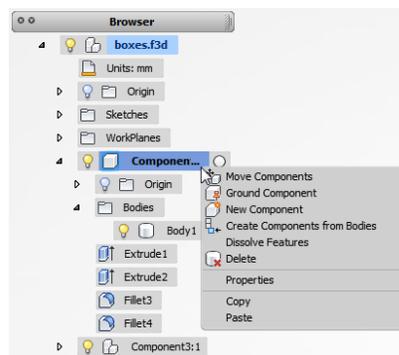
Enable or Disable Adaptive Snap

Controls the display of the grid when editing a sketch.

Browser commands

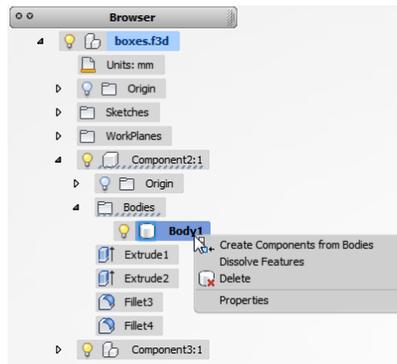
There are commands that can be accessed in the browser in Inventor Fusion. Access these commands using the icons in the browser or through the right-click context menus.

Components



- **Move Components** Starts the Move Components command with the selected component.
- **Ground Components** Locks the position of the component in 3D space.
- **New Component** Inserts a new component within the selected component.
- **Create Components from Bodies** Creates a new component within the selected component for each body in the component.
- **Dissolve Features** Removes features from the browser. This does not affect the geometry.
- **Delete** Removes the component occurrence from the model.
- **Properties** Displays the Properties dialog box for the selected component.
- **Copy** Places the selected component on the clipboard.
- **Paste** Inserts the component from the clipboard.

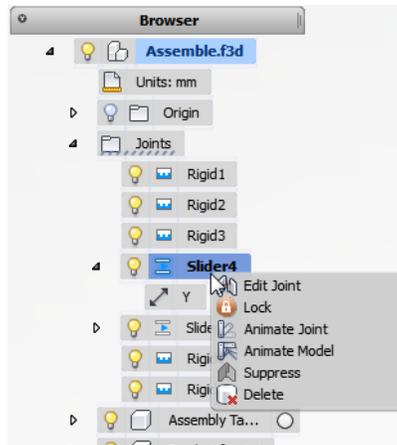
Bodies



- **Create Components from Bodies** Creates a new component and moves the selected body.
- **Dissolve Features** Removes features contained by the body from the browser. This does not affect geometry.
- **Delete** Deletes the body from the model.
- **Properties** Displays the Properties dialog box for the body.

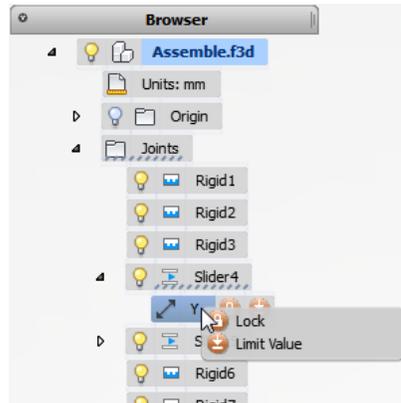
Joints

Joint node



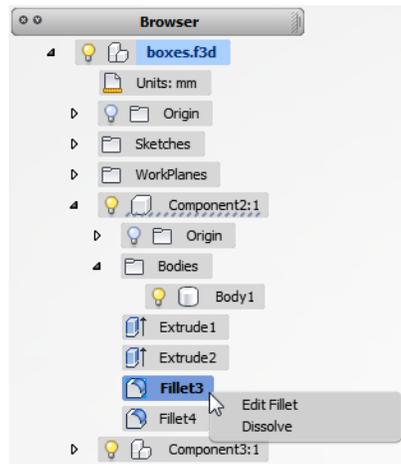
- **Edit Joint** Displays the Edit Joint dialog and manipulators.
- **Lock** Locks the motion of the joint.
- **Animate Joint** Shows the allowable motion of the joint.
- **Animate Model** Shows the allowable motion of the model.
- **Suppress** Removes the joint from participating in the position of the component.
- **Delete** Removes the joint.

Joint movement node



- **Lock** Locks the motion of the joint in the selected direction.
- **Limit Value** Specify a minimum and maximum value for the joint movement. These values affect the animation of the Animate Joint and Animate model commands.

Features



- **Edit feature** The availability of this command depends on the type of feature selected. Displays a dialog box and manipulators to modify the feature.
- **Dissolve** Removes the feature from the browser. This does not affect the geometry.

Import Data

4

Part and assembly files from other CAD systems can be imported for use in 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, 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 Inventor Fusion representative of the geometry and topology in the source file. You can use Inventor Fusion commands to adjust the base features and add new features to the Inventor Fusion feature tree.

Import Data from other CAD systems

You can import data in the following file formats:

- DWG
- IPT
- IAM
- SAT
- STP, STEP

To import files into Inventor Fusion

- 1 Expand the Application Menu then select Open.
- 2 In the Open dialog box, select the file type in the Files of Type pulldown.
- 3 Browse to and select the file to import.
- 4 Click Open to import the file.

Export Data

5

You can export 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 Inventor Fusion file. Changes to the 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 Inventor Fusion file.

Export Data to Other Formats

You can export solid data in the following file formats:

- STEP
- Autodesk Shape Manager SAT

You can export viewing and faceted data in the following file formats:

- STL

To export an Inventor Fusion file to another file format

- 1 Expand the Application Menu then select Export
- 2 In Export dialog box, 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 Inventor Fusion data to the file.

Legal Notices

6

Autodesk Inventor Fusion 2013

© **2012 Autodesk, Inc. All Rights Reserved.** Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

Third Party Software Credits and Attributions

Trademarks

The following are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and other countries: 123D, 3ds Max, Algor, Alias, Alias (swirl design/logo), AliasStudio, ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Homestyler, Autodesk Intent, Autodesk Inventor, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSketch, AutoSnap, AutoTrack, Backburner, Backdraft, Beast, Beast (design/logo) Built with ObjectARX (design/logo), Burn, Buzzsaw, CAiCE, CFdesign, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Creative Bridge, Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, Design Web Format, Discreet, DWF, DWG, DWG (design/logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DWFX, DXF, Ecotect, Evolver, Exposure, Extending the Design Team, Face Robot, FBX, Fempro, Fire, Flame, Flare, Flint, FMDesktop, Freewheel, GDX Driver, Green Building Studio, Heads-up Design, Heidi, Homestyler, HumanIK, IDEA Server, i-drop, Illuminate Labs AB (design/logo), ImageModeler, iMOUT, Incinerator, Inferno, Instructables, Instructables (stylized robot design/logo), Inventor, Inventor LT, Kynapse, Kynogon, LandXplorer, LiquidLight, LiquidLight (design/logo), Lustre, MatchMover, Maya, Mechanical Desktop, Moldflow, Moldflow Plastics Advisers, Moldflow Plastics Insight, Moldflow Plastics Xpert, Moondust, MotionBuilder, Movimento, MPA, MPA (design/logo), MPI, MPI (design/logo), MPX, MPX (design/logo), Mudbox, Multi-Master Editing, Navisworks, ObjectARX, ObjectDBX, Opticore, Pipeplus, Pixlr, Pixlr-o-matic, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProMaterials, RasterDWG, RealDWG, Real-time Roto, Recognize, Render Queue, Retimer, Reveal, Revit, RiverCAD, Robot, Scaleform, Scaleform GfX, Showcase,

Show Me, ShowMotion, SketchBook, Smoke, Softimage, Softimage|XSI (design/logo), Sparks, SteeringWheels, Stitcher, Stone, StormNET, Tinkerbox, ToolClip, Topobase, Toxik, TrustedDWG, U-Vis, ViewCube, Visual, Visual LISP, Voice Reality, Volo, Vtour, WaterNetworks, Wire, Wiretap, WiretapCentral, XSI.

All other brand names, product names or trademarks belong to their respective holders.

Disclaimer

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. "AS IS." AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

Index

B

browser 33

