Autodesk® Inventor® Fusion Technology Preview

Autodesk Inventor Fusion: Getting Started

Autodesk®

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Autodesk Inventor Fusion TP2

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This is the Help for the second technology preview release of Autodesk Inventor Fusion released in October 2009. This content may not apply to prior or future releases.

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Working with Inventor Fusion User Interface

This section presents general topics related to the Inventor Fusion User Interface.

The Ribbon

Display and Organize the Ribbon

The ribbon is displayed automatically when you create or open a file, providing a compact palette of all commands necessary to create your model.

The horizontal ribbon is displayed across the top of the application window.

The ribbon minimize button minimizes the ribbon. The minimize button is located to the right of the ribbon tabs.

A temporary ribbon pane is displayed across the top of the application window when a command is active. Use the temporary pane to input command options, range limits, and other settings.

Ribbon Tabs and Panels

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

Some ribbon panels display a drop-down arrow. The arrow indicates there are additional commands related to that panel. Click the drop-down arrow to access the additional commands on the access table.

To display a hidden panel, right-click anywhere inside the ribbon, and click the name of the panel. To display or hide a panel, right-click anywhere inside the ribbon, and click or clear the name of a panel.

Floating Panels

If you pull a panel off a ribbon tab and into the drawing area or onto another monitor, that panel floats where you placed it. The floating panel remains open until you return it to the ribbon, even if you switch ribbon tabs.

Expanded Panels

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

Procedure

To minimize the ribbon using the minimize button

1. Click the ribbon minimize button to the right of the ribbon tabs.
2. The minimize behavior cycles through the following minimize options:
   - **Minimize to Tabs:**
     Minimizes the ribbon so that only tab titles are displayed.
   - **Minimize to Panels:**
     Minimizes the ribbon so that only tab and panel titles are displayed.
   - **Show Full Ribbon:**
     Displays tabs and full panels, including controls.

Other methods to minimize the ribbon

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

Double-click the name of the active ribbon tab.

To turn off the display of a panel

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

To switch the display of panel titles

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

Customize the Ribbon

You can customize the ribbon depending on your needs.

You can customize the ribbon in the following ways:

- You can change the order of ribbon tabs. Click the tab you want to move, drag it to the appropriate position, and release.

- On each tab, you can change the order of ribbon panels. Click the panel to move, drag it to the appropriate position, and release.
- You can hide panels. Right-click a tab and chose which panels to display.

Glyphs and Manipulators

As you use Inventor Fusion, you specify modes of operation, range limits, and other options. Since the Inventor Fusion user interface does not employ dialog boxes, there are other access modes to set these options:

- The ribbon
- Glyphs
- Manipulators

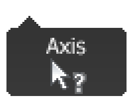
The Ribbon on page 4 is discussed in another section.

Glyphs

As you use commands in Inventor Fusion, symbols or glyphs, often appear next to the cursor. Glyphs indicate that you can select a mode of operation, or that certain commands are available for use.

Click and hold a glyph to display the options for the active command. Drag the cursor to the appropriate option and release to select it. If you pause the cursor over a glyph, a commandtip displays more information about the glyph.

The most frequently used glyphs are in the following table:

Glyph Icon	Glyph Title	Description
	Select Option	Select a command option.
	Go to Next	Go to the next input selection.
	Profile commands	This glyph is displayed when you select (highlight) a closed sketch profile. Click this glyph to select from a list of modeling operations to perform on the profile.

Manipulators

A Manipulator is a 3D command. It is usually an arrow, sphere, or ring that appears while a command is active. It sets the distance, angle, or direction of an operation, or for setting the location or size of a feature.

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

An active, selected manipulator appears yellow. When two or more manipulators are displayed, the active manipulator is yellow. The inactive manipulator appears red or gray.

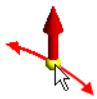
When there are two or more manipulators associated with a command, press the Tab key to cycle from the current manipulator to the next one. This also activates the heads-up display (HUD) field associated with the manipulator (including those with multiple HUDs). You can measure on page 34 geometry using the dropdown next to any HUD. You can enter simple arithmetic expressions and mix units when you enter values into the HUD.

A manipulator that is used to set the location of a feature (such as the Hole Center Manipulator) displays blue when the location is constrained (concentric with another cylindrical feature or aligned with the midpoints of two edges, and so on).

If you pause the cursor over a manipulator, a commandtip displays more information about the manipulator.

There are four basic manipulator types. The purpose of each varies by command. Some common examples are listed in the following table:

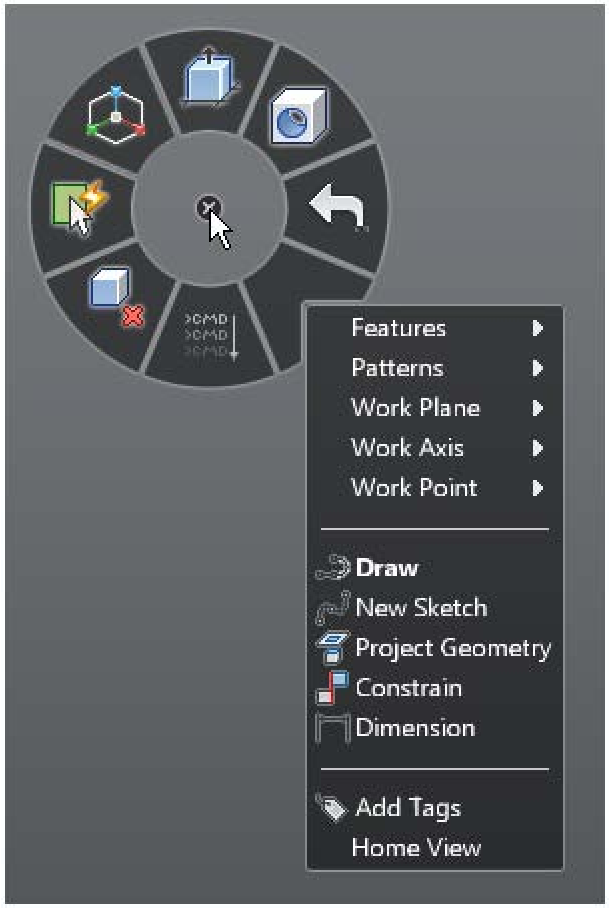
Manipulator	Manipulator Type	Description
	Linear Arrow	Used in the Extrude command to set length and direction, and in the Hole command to set the depth.
	Radial Arrow	Used in the Revolve command to set the rotation angle.

	Ring	Used in the Hole command to set the hole diameter as well as counterbore and countersink diameters.
	Sphere	Used in the Hole command to set the hole center location, in the Extrude command to set the taper angle, and in the Sweep command to sweep the selected profile along the path.

When using manipulators, you do not need to keep your cursor exactly on the 3D arrow. You can drag anywhere over empty graphics space. Many manipulators can also snap to other geometry on your model. While a manipulator is active, you can pause your cursor over the design and prompts appear over geometry that the manipulator can snap to.

Marking Menu

The Marking Menu is a spatially arranged, in-canvas menu used for executing and completing (or canceling) commands. The contents of the marking menu change based on the context in which it is invoked.



Marking Menu with Context Menu

The marking menu consists of eight wedges. Each wedge represents a command/operation. These commands are the seven frequently used commands and the eighth is a context menu which contains additional commands.

The default commands in the marking menu (context menu not shown) are:

1. Press Pull
2. Hole
3. Undo
4. Context Menu
5. Repeat Last Command
6. Delete
7. Select
8. Move

You can invoke and hide the marking menu through the following steps,

- To start the marking menu, right-click or right mouse down
- To close the marking menu, use the escape key, release the right mouse button when no marking menu item is preselected, or click the left mouse button when no item is selected.

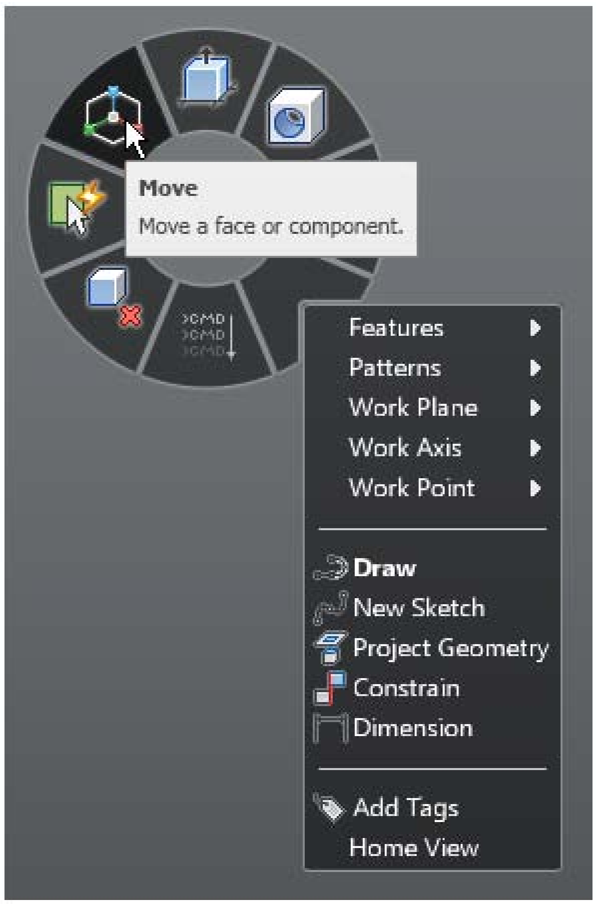
You can hover (pause) the cursor over an item to see the commandtip for that command.

You can select an item from the marking menu through the following steps,

1 Invoke the marking menu (using either the right-click or holding the right mouse down).

2 Drag the cursor to the appropriate item.

3 Once the cursor is over the appropriate item, release the cursor and click the item.



The marking menu item for Move is highlighted and the commandtip for the highlighted item is shown.

Gesture Behavior

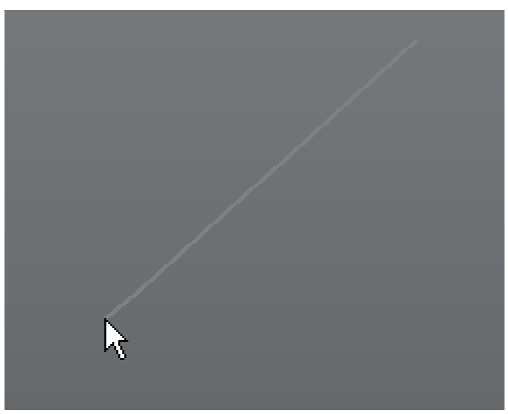
An alternative technique to execute a command in the marking menu involves gesture behavior. This is useful when you are well conversant with the marking menu layout and need a faster way to execute commands. Hence before using gesture behavior, a little practice with the marking menu to develop some muscle memory (familiarity) around the layout of the marking menu is helpful.

A gesture consists of starting the marking menu (right mouse down), immediately dragging the cursor to the location of the intended marking menu wedge and releasing the right mouse button before the entire marking menu is displayed. If these operations are completed within 250 milliseconds, only the selected wedge is briefly displayed to confirm that the operation was performed.

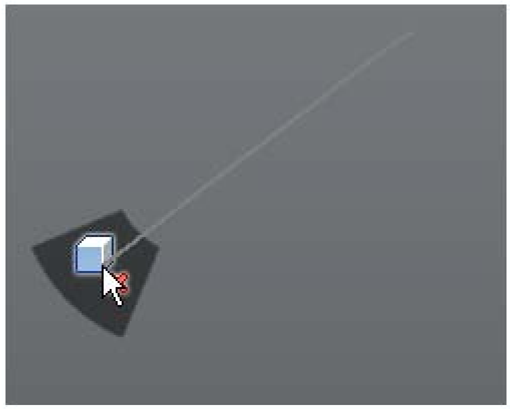
Here are the steps for executing a gesture,

1. Start the marking menu (right mouse down).
2. Within 250 ms, drag the cursor in the direction of the wedge for the appropriate operation.
3. Release the right mouse button.

During the drag gesture, a trail is visible in the canvas, showing the cursor path. When you release the cursor, the selected wedge is displayed for a brief time span. The command corresponding to this wedge then gets executed.



Visible train while dragging cursor in a gesture movement



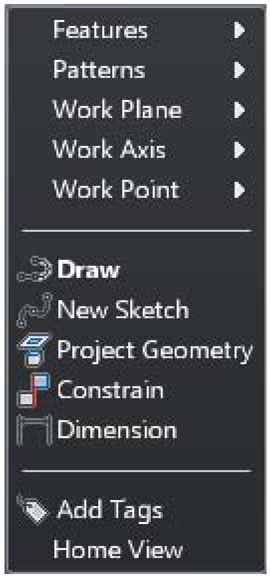
Marking menu wedge appears when cursor is released

Context Menu

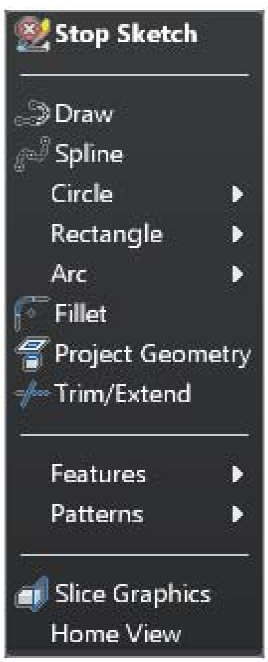
The marking menu displays a context menu in its 4th wedge. After you invoke the marking menu, drag the cursor to the appropriate operation on the context menu. When you release the cursor, the operation is selected.

The contents of the context menu change based on the current context depending on,

- Whether modeling mode/ sketch mode is on
- Whether any entity is selected



Context menu in modeling mode



Context menu in sketching mode

Context menu for Face selection

Context menu for edge selection

The context menu has a default menu item, which varies depending on the context and selection. The default menu item appears in bold text. When you select the 4th wedge of the marking menu in a gesture movement, the default context menu item is invoked.

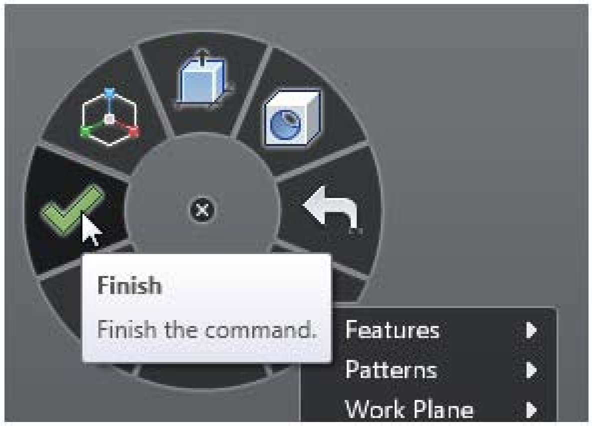
Primary Marking Menu Behavior when a Command is Active

When a command is active, certain items in the marking menu display a different behavior depending on the context. The 3rd wedge item which represents ‘Undo when executed outside a command, now assumes Cancel behavior within a command.

Item no. 3 represents 'Cancel' when in mid-command

Item no 3. represents Undo outside a command

When a command is executing and you select a different command, the current command is implicitly accepted if the input is valid. After this happens the new command is started. This technique can be utilized for quickly approving a command and ending it. The seventh wedge of the marking menu, which represents a Finish action when a command is active, is an easy way to do this. When you have provided the correct input for a command and wish to OK it, bring up marking menu and click the Finish item to implicitly OK the command and end it.

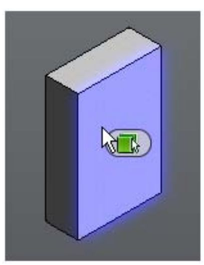
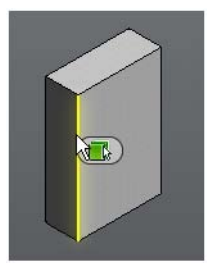


Item no 7. represents 'Finish' which can be used to commit an active command

Selection commands

Mechanical designs often have many objects in the canvas which can make selecting the appropriate object difficult.

The Select Other navigation commands in Inventor Fusion help you to select obscured or difficult-to-select geometry. The different options for the Select Other are accessible through a glyph which can be seen when you hover the cursor over a face/edge. This face/ edge is termed as the root face/ edge.

	
Select Other while hovering over a face	Select Other while hovering over an edge

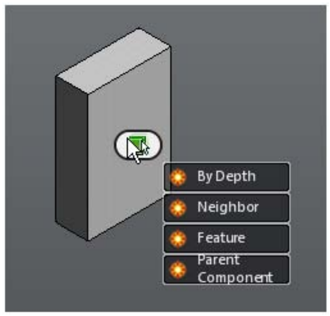
Click the glyph to see the fly-out menu containing the following options:

1 By Depth

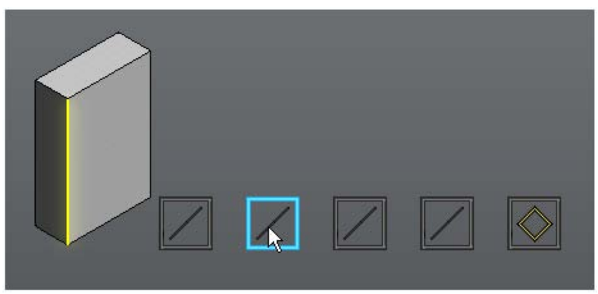
2 Neighbor

3 Feature

4 Parent Component



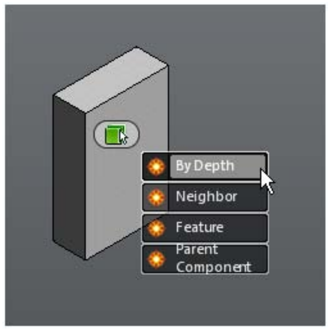
The Parent Component option directly selects the component which is the parent for the face edge. When you select By Depth, Neighbor, or Feature, a selection strip is displayed. The strip contains several frames each representing a possible selection.



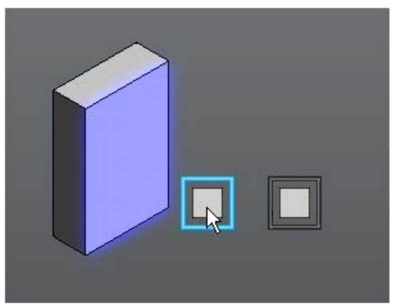
When you hover over a frame in the selection strip, the corresponding element is highlighted in the model. When you click a frame in the selection strip, the corresponding element is selected.

By Depth Selection

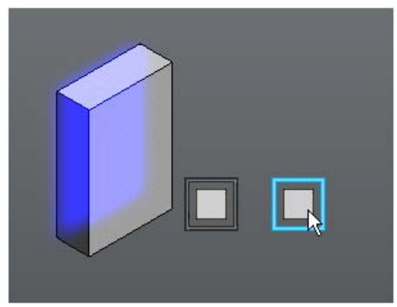
When you choose the ‘By Depth selection, eligible faces/ edges/ work planes/ work axes/ work points/ profiles that are partially or wholly obscured by the selected element are listed on the selection strip.



In the following example, eligible faces with different depth order are listed in the selection strip.



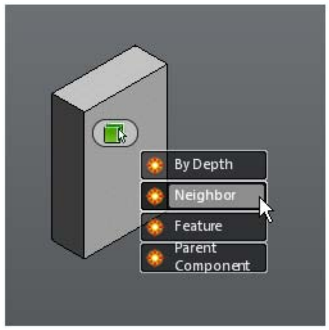
When you highlight the first frame in the Z Depth selection strip, the front most overlapping face element is also highlighted.



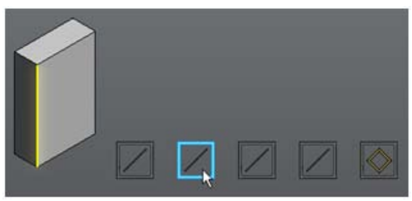
The next frame selected in the strip causes an eligible element with a different Z Depth to be highlighted.

Neighbor Selection

When you choose the Topological Neighbor selection, eligible elements that are topologically connected to the root face/ edge are listed on the selection strip. For the root face, list of connected edges and edge loops is displayed while for a root edge a list of connected faces is displayed.



In the following example, the topological neighbors of a root face are listed in the selection strip. The last item in the strip represents all the edge loops connected with the root face.

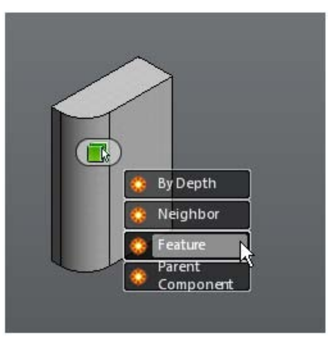


When you highlight a frame in the selection strip, the corresponding edge element is also highlighted.

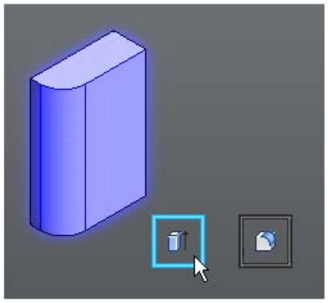
The last frame causes all edge loops to be highlighted.

Feature Selection

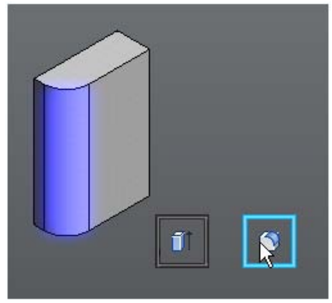
When you choose the Feature selection, eligible feature objects that are partially or wholly obscured by the selected element are listed on the selection strip.



In the following example, both the extrusion and the fillet are eligible features. The icons on the selection strip make it easy to identify the features by type.



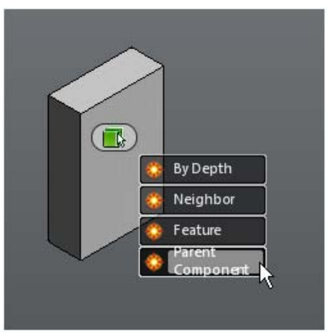
The icon in the selected frame in the selection strip identifies the extrude feature.



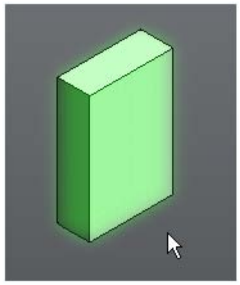
The icon in the selected frame in the selection strip represents the fillet feature.

Parent Component Selection

When you choose the Parent Component Selection, the parent component for the root face/edge gets selected. It is a short cut way to select the parent component of an entity.



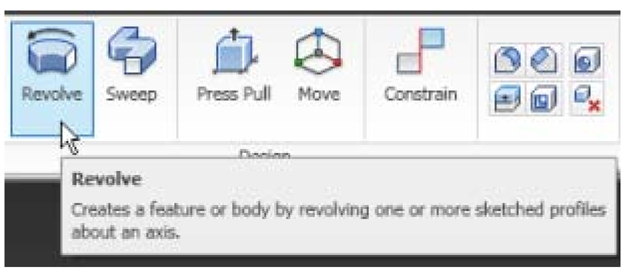
In the following example, the parent component of the root face gets selected.

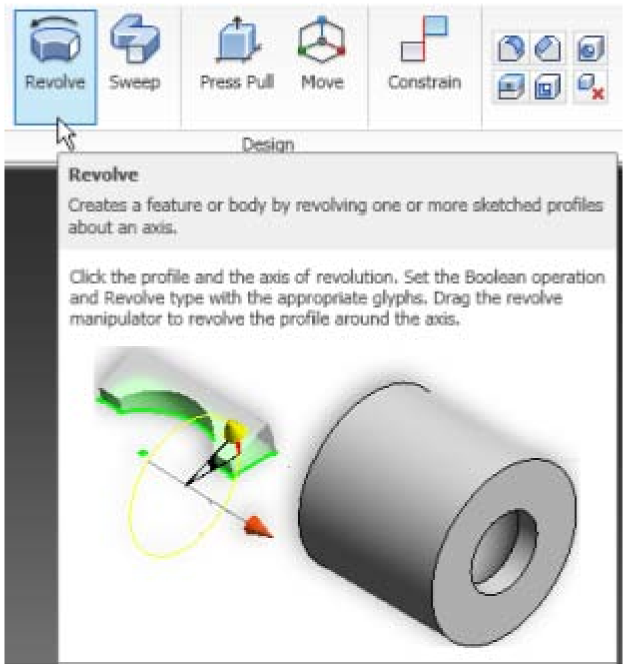


The parent component option from the fly-out was selected for the root face. Consequently the parent component of the root face is selected.

Enhanced tooltip

Many of the ribbon commands have enhanced (also referred to as progressive) tooltip which display information for interaction with commands. Initially, the name of the command and a short description of the command is displayed. If you continue to pause the cursor, the commandtip expands to display additional information.

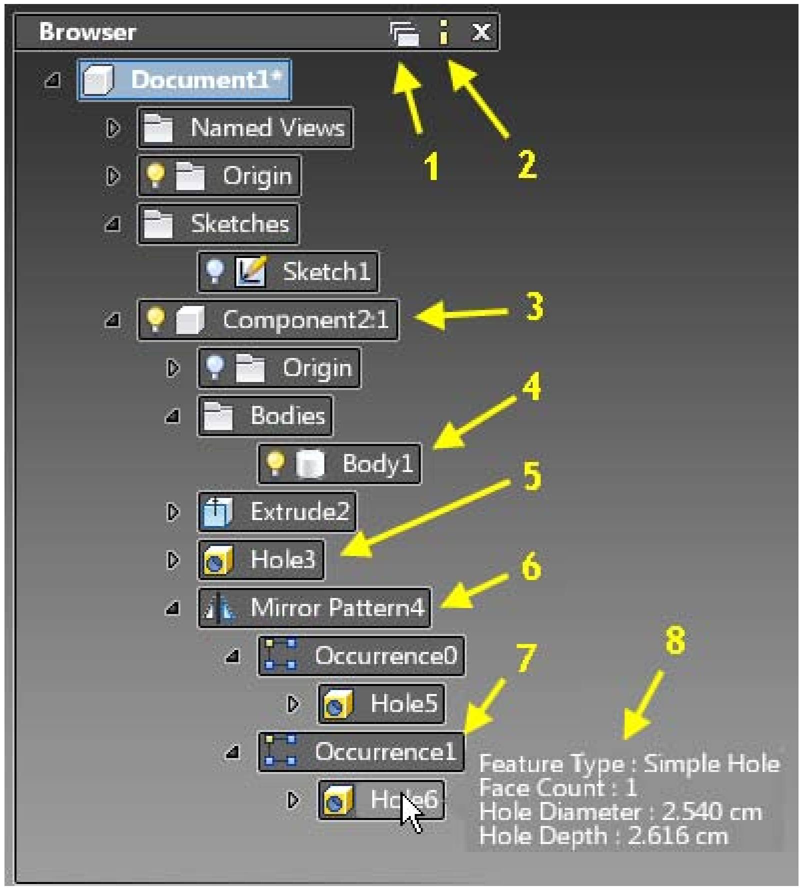




Browser and Copy/Paste

In Inventor Fusion, the browser presents an organized view of the data in your design. Objects selected in the browser are selected in the graphics and vice-versa. You can create new component instances in the browser. Bodies can be dragged or copied and pasted from one component to another.

1. Toggle Favorites Folder
2. Toggle Information Panel
3. Child Component Node
4. Body Node
5. Feature Node
6. Pattern Node
7. Occurrence Node
8. Information Panel



The blue node in the browser denotes the active component in your design. By double-clicking the component icon you can change the active component. This is important when creating new sketches, work features and features. All new objects that you create belong to the active component.

Information Panel

In Inventor Fusion, the model information is readily available. The browser includes an information panel for each node, which you can switch on or off. Pause the cursor over a component, body, feature, pattern, or occurrence to view information about it.

Create New Component Instances

Right-click the top-level Document node and select New Component from the context menu to create a child component under the document. Similarly, right-click any component node and select New Component from the context menu to create a child component under the selected component.

Additional Browser Functionality

Select Isolate Component to hide all but the selected component (UnIsolate Component redisplays the hidden components).

Use the Favorites folder to group frequently referenced components, bodies, features, and patterns.

Cut/Copy and Paste bodies from one component to another, or across documents.

Delete components, bodies, and features.

Dissolve patterns.

Cut/Copy and Paste

Cut / Copy and Paste offer user-interface paradigms for transferring objects from a source to a destination. Cut removes the object after pasting, whereas copy keeps the source object intact after pasting. Inventor Fusion currently supports the following objects for cut/copy and paste:

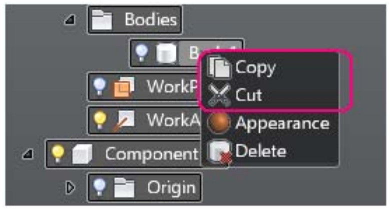
- One or more pieces of sketch geometries
- One or more body objects
- One or more components (component instances)
- Face sets

There is no undo/redo support for the cut or copy command. Paste and paste new are supported for undo/redo.

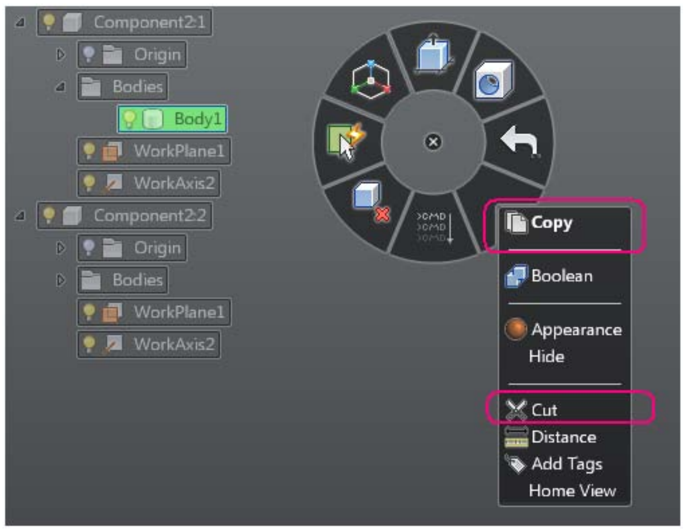
Cut/Copy Interace

Objects can be selected from either the browser or using a graphical selection. The cut and copy commands are available from various places as follows:

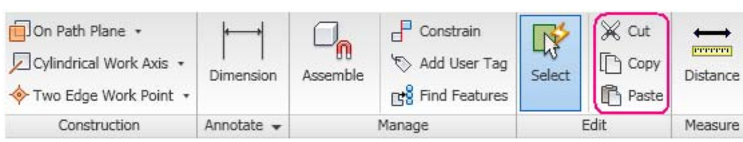
1. Browser: right-click a browser node



2 Marking Menu: right-click open space



3 Ribbon



4 Key strokes

This functionality has also been mapped to the key sequences Ctrl+X and Ctrl+C.

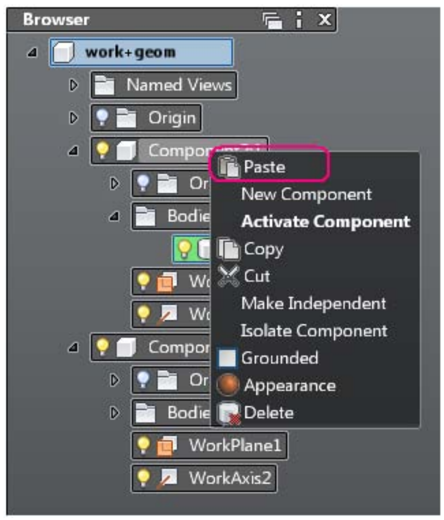
Note: If the user uses cut, the actual cut operation does not happen until the paste command is invoked. Every time a cut/copy command is used, the previously cut/copied objects are cleared from the clipboard.

Paste Interface

The paste command pastes previously cut/copied objects from the clipboard. At the time of pasting a valid paste container is either deduced or needs to be specified. Rules are as follows:

1 Browser Node

On a browser node that is a valid paste container. For example, a valid paste container for a set of sketch geometries is a sketch node in the browser. A component node is a valid paste container for a body or a component instance as long as those objects are not already included inside the component instance.

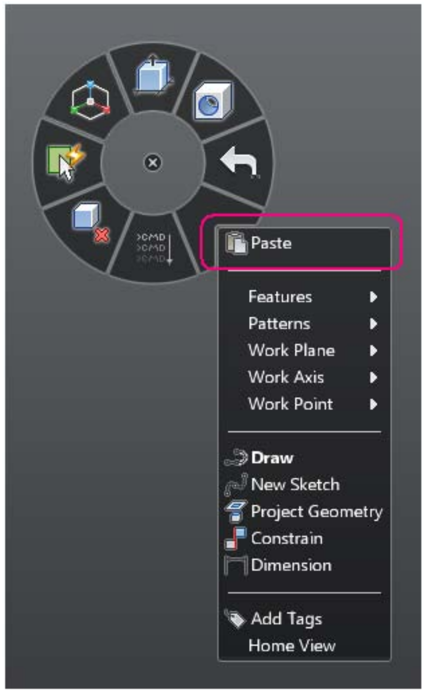


2 Graphical Selection

On a graphical selection that is a valid paste container. For example, a face or a work plane is valid paste containers for sketch geometries.

3 Empty Space

When the paste command is invoked in empty space, the active sketch or the active component is used as the paste container if that is valid.



4 Ribbon

5 Key Strokes

This functionality has been mapped to the key sequence Ctrl+V.

Note: Objects can be pasted in a different document from the one they were copied.

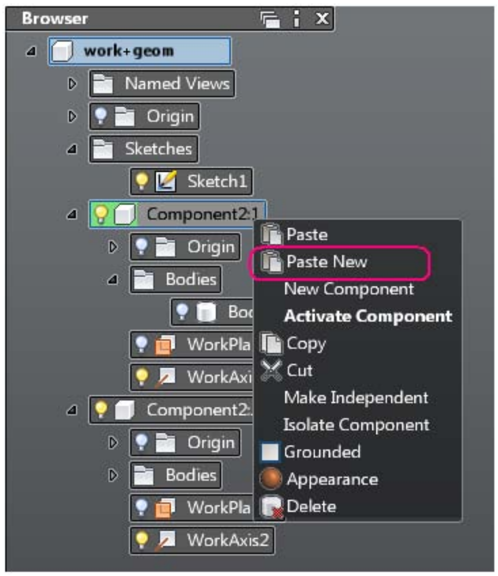
Implicit Paste Using Browser Drag and Drop

In addition, within a document, the user can drag and drop body objects and component objects. These drag-and drop operations result in an implicit cut and paste.

Paste New

When a component instance is cut/copied, it can be pasted as a "shallow copy" or as a "deep copy". In a shallow copy, a copy of the cut/copied component instance is added to the new owner as a new component instance; the structure under it is shared with other component instances. In a deep copy the entire subassembly under the component instance is copied.

To distinguish between the two, a command called "Paste New" is available when the cut/copied object is a component. Paste creates a shallow copy and paste new creates a deep copy.



Note: When a component is copied and pasted across documents, only a deep copy is possible, so only paste new is available.

Paste Behavior

When an object is pasted, there are two kinds of behavior depending on whether the paste is explicit (invoked using the paste or paste new command) or implicit (invoked by drag and drop in the browser).

Explicit Paste Behavior

When paste is done explicitly, the paste object is placed at the cursor and the user is allowed to move it around on the screen and place it by clicking with the left mouse button.

Explicit paste does not clear the clipboard, so it is possible to repeat the paste (or paste new) command to repeatedly paste.

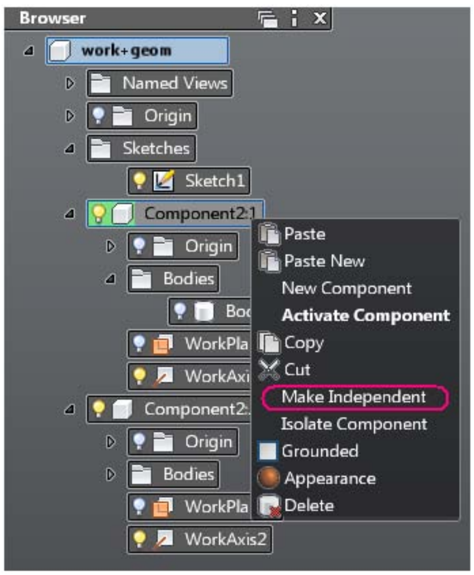
Implicit Paste Behavior

When paste is done implicitly, this is considered a restructure operation and the pasted object changes its place in the hierarchy. Its location and orientation in the world coordinate system remains unchanged. There is no user interaction needed after the drag-drop.

Implicit paste clears the clipboard.

Make Independent

The "Make Independent" command operates on a component instance selection either in the browser or a graphical selection.



This command makes the subassembly represented by the component instance "independent", for example, it is equivalent to a cut followed by a deep copy of the component instance in the same parent container.

There is no placement interaction when using this command, as this is also a "restructuring" operation.

Function Key Behavior

Some keys are reserved for specific purposes in Inventor Fusion. For example, F1 accesses Help in all Microsoft Windows applications. In Inventor Fusion, many of the F-keys (function keys) are reserved for global operations.

The following table contains all the reserved keyboard and mouse shortcuts in Inventor Fusion:

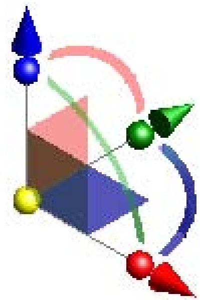
Function Key	Behavior
F2	Pan
F3	Zoom
Shift+F3	Zoom window
F4	Orbit
F6	Zoom all
F7	Slice graphics (see note)
F10	Toggle shortcut keytips in Application menu and Quick Access commandbar
Middle mouse button	Pan
Mouse wheel	Zoom
Shift+Middle Mouse button	Orbit

Note: Slice Graphics operation requires the selection of a cutting plane. First select a work plane or sketch profile, and then press the F7 key.

Triad

A Triad is a 3D in-canvas command for the movement and rotation of edges, faces, features, bodies, and components. You can interactively position a face or feature by dragging the triad in a planar move, axial move, or free movement. The selected area of the triad controls the movement.

When the triad is displayed, select or drag a triad segment to indicate the appropriate type of transform. In the heads-up display (HUD), you can enter coordinates to move a face or feature precisely. When you drag the triad, the X, Y, and Z coordinates dynamically update in the HUD.



- Red is the X axis
- Green is the Y axis
- Blue is the Z axis

When you first activate the triad, its origin sphere is coincident with the geometry to transform. Click a triad section or drag to indicate the appropriate type of transform. As you select other parts of the triad, you can drag or enter precise coordinates corresponding to your selection.

Triad Part	Description
Arrowheads	Move the triad along the axis.
Arcs	Rotate the triad around the axis.
Planes	Move the triad in the selected plane.
Sphere	Allows unrestricted movement in the view plane.

To reorient the triad relative to a different location without affecting the geometry selection, press the F5 key and then click the new location for the triad.

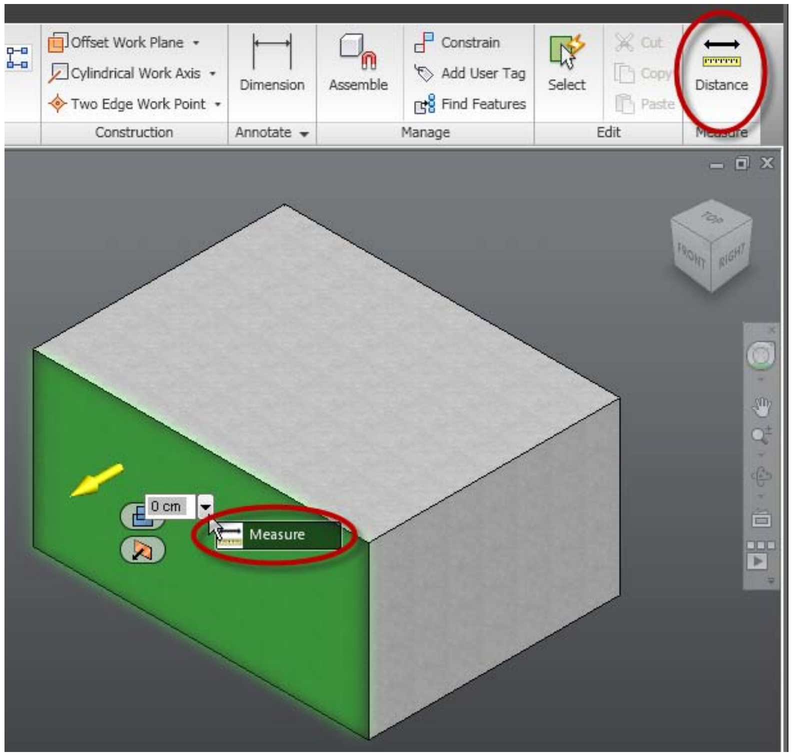
Measure

The Measure command provides two important functions:

- Provide geometry information (distance, angle, area, and so on).

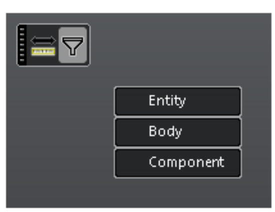
- Populate input boxes with measurement.

Access the measure command in the ribbon or in the fly out of any input box.



Selection Support

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



Measure Dialog Box

The measure dialog box is displayed when the command is started and persists until the command is terminated. The dialog box displays geometry information dependent on the selected entities.

click a row in the measure dialog box to copy that value to the clipboard. Use Ctrl + V to paste the value.

Click and drag the dialog box to reposition it.

Sample Workflow

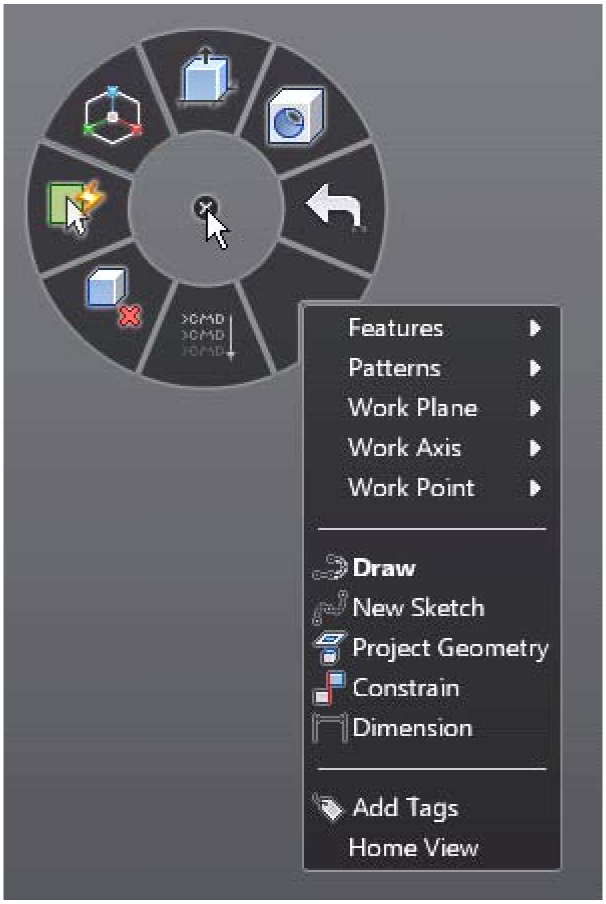
1. Select a face to extrude.
2. Start the measure command from the value input fly out and select the objects to measure.
3. To set the length of the extrusion equal to the length of the selected edge (2), click the Curve 2 Length row in the measure dialog box. The value is copied to the input box and as well as the clipboard.
4. After clicking the value, the measure dialog box is dismissed and the model is updated accordingly.

Menu and Command Access

Execute commands in Inventor Fusion in the following ways:

- Click the command button on the ribbon.
- Select the command from the marking menu.
- Select the command name in the context menu.

Most commands are available in both the ribbon and the context menu. A subset of frequently used commands appears in the marking menu.



For more information see Marking Menu on page 8.

Note: Right-click to display the marking menu and context menu.

Other commands in the Application Window

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

Application Menu

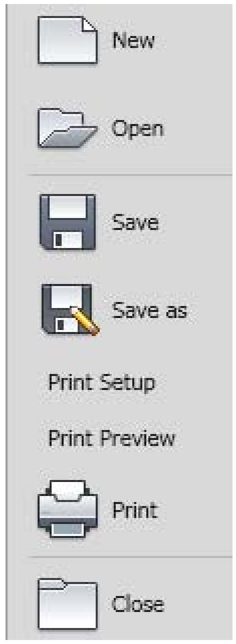
Click the application button to access commands to create, open, and export a file.

Access Common commands

Access common commands to start or export a file from the Application Menu.

Click the application button to:

- Create a file.
- Open an existing file.
- Save a file.
- Save a file as another name.
- Print a file.
- Close the application.



Quick Reference

This section contains descriptions of file access and print dialog boxes.

File Open

File Open

Access:	Opens when you perform operations requiring selection of a file.

Look in	Shows path of the active directory.
File list	The main window shows a list of the subfolders and files in the selected path. Double-click a subfolder to show the files it contains.

File name	Specifies the file to open, enter a file name, or select a file from the listed files.
Files of type	Filters file list to include only files of a specific type. Click the arrow to show list, and then highlight to select a file type.
Open	Open the selected file.
Cancel	Cancels the file open operation and closes the dialog box.

New File

New File

Creates a file.

Print

Print

Prints or plots all or any portion of a model.

Access:	Click  Print.

Sets the options for printing or plotting all or a portion of the active model.

Name	Specifies the printer or plotter. To change the printer or plotter, click the arrow and select from the list.
Properties	Opens the Print Setup dialog box used to set the paper size and orientation.
Print Range	Sets the page range to print. For a model, only the portion of the model that is displayed in the graphics window is printed (All is the only option available).

Number of Copies	Sets the number of copies to print. Enter the number of copies, or use the up or down arrow to select the number of copies.

You can print or plot all or any portion of a model.

To print a model

You can print or plot all or any portion of the active model.

1 Set up the view of the model. Only the portion of the model that is displayed in the graphics window prints.

2 Click  Print.

3 In the Print dialog box, enter the number of copies. If necessary, click Properties to open the Print Setup dialog box, and then change the paper size and orientation.

4 Click OK to print.

Save, Save As

Save, Save As

Saves a file with a specified name and file type.

Access:	In the Application Menu, select: **Save** **Save** The Save command saves the active document contents to the file specified in the window title, and the file remains open. **Save As** **Save As** Save As saves the active document contents to the file specified in the Save As dialog box. The original document is closed and the newly saved file is opened.

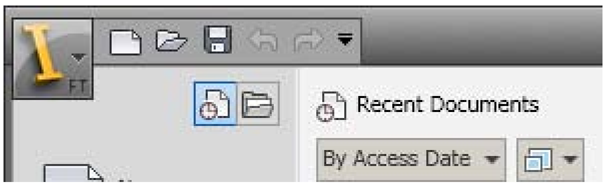
	The contents of the original file are unchanged.

Save in	Shows path of the active directory and specifies destination of the saved file.
Locations	
File name	Specifies the name of the file to save. If the file was saved, the file name is shown.
File of type	Filters file list to include only files of a specific type. Click the arrow to show list, and then highlight to select a file type. The extension is added to the file name.
Save	Saves highlighted file (with file name and type specified by one of the methods explained previously).
Cancel	Cancels the save operation and closes the dialog box.
Available file types:	
DWG	
SAT	Geometry objects stored in ASCII files. (versions 4.0 - 7.0).
STEP	An international format developed to overcome some of the limitations of current data conversion standards. Files created in other CAD systems can be converted to STEP format and imported into Inventor Fusion. (versions AP214 and AP203E2)

Recent Documents

View the most recently used files with the Recent Documents list.

Files display in the Recent Documents list with the most recently used file at the top by default.



Pinned Files

You can keep a file listed, regardless of files that you save later, using the push pin button to the right. The file is displayed at the bottom of the list until you turn off the push pin button.

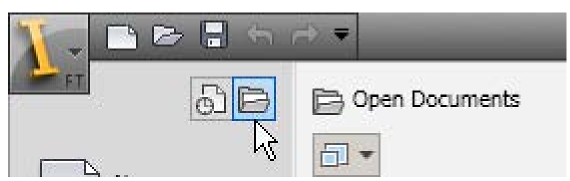
Sort and Group Options

Use the By Ordered list drop-down menu at the top of the Recent Documents list to sort or group files by:

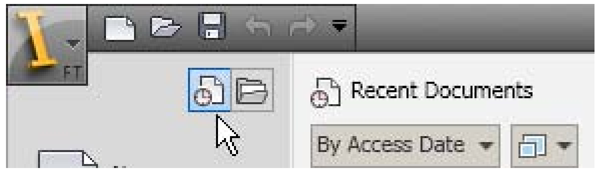
- File name
- File size
- File type
- Date the files were last modified.

Procedure

Click Open Documents to view open documents.



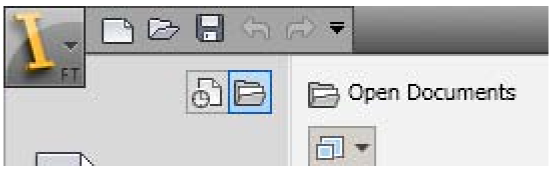
Click Recent Documents to view recent documents.



Currently Open Documents

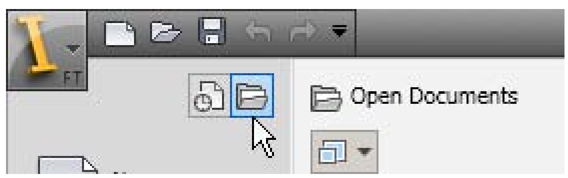
View only files that are currently open with the Open Documents list.

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

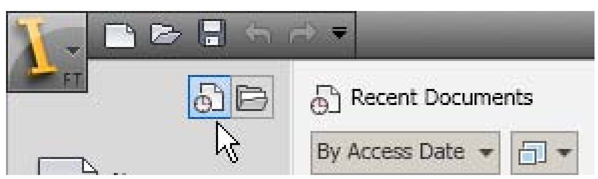


Procedure

Click Open Documents to view open documents.



Click Recent Documents to view recent documents.



Preview Documents

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

When you pause the cursor over a file in either of the lists, the following information is displayed:

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

Quick Access commandbar

Display frequently used commands with the Quick Access commandbar.

View Undo and Redo History

The Quick Access commandbar displays options to undo and redo changes to your file.

Add Commands and Controls

Add unlimited commands to the Quick Access commandbar. commands that extend past the maximum length of the commandbar are displayed in a drop-down menu.

To add a ribbon command to the Quick Access commandbar, right-click the command on the ribbon, and click Add to Quick Access commandbar. Commands are added to the right of the default commands on the Quick Access commandbar.

Move the Quick Access commandbar

Place the Quick Access commandbar either above or below the ribbon using the Customization button.

Procedure

To add and remove commands to the Quick Access commandbar

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

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

To add commands to the Customize Quick Access commandbar menu

➤ On the ribbon, right-click the command add, and select Add to the Quick Access commandbar.

To move the Quick Access commandbar menu above or below the ribbon

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

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

Status Bar

The Status bar displays across the bottom of active window. The status bar provides the following information:

- When you are in a command that requires you to perform an action to continue, a message displays on the far bottom left. It indicates the next step to proceed with the active command.

Keytips

Use the keyboard to access the Application Menu, Quick Access commandbar, and ribbon.

Press Alt or F10 to display shortcut keys for common commands in the application window. Use keytips to perform tasks without using your mouse.

When you select a keytip, more keytips are displayed for that command. Keytips appear as underlined characters to indicate which key or combination of keys on the keyboard to press to activate a command.

Use keytips to navigate in the Application Menu and in the ribbon using only the keyboard. Use the keyboard arrows to navigate to commands on the ribbon and Application Menu

Navigation commands

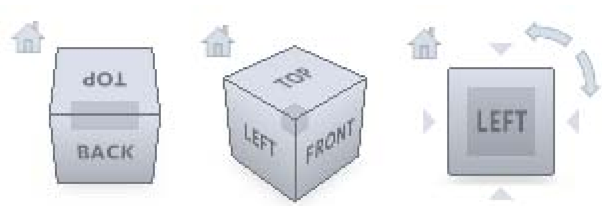
Navigation commands change the orientation and view of your model.

The display of a model can be adjusted by increasing or decreasing the magnification at which objects are displayed or rotating the view of the model. Use the View Cube, SteeringWheels, and commands in the Navigation panel to change the view of your model. You can create a view that defines an area of a model as the Home view and use preset views to restore known viewpoints of a model with the View Cube.

View Cube

Overview of the ViewCube

The ViewCube command is a persistent interface that you use to switch between standard and isometric views of your model. When you display the ViewCube command, it is shown 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 the cursor is positioned 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.



Control the Appearance of ViewCube

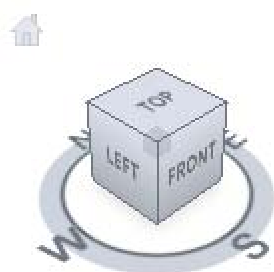
The ViewCube command is displayed in one of two states: inactive and active. When the ViewCube command is inactive, it appears partially transparent by default so that it does not obscure the view of the model. When active, it is opaque and may obscure the view of the objects in the current view of the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

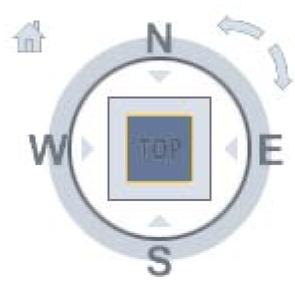
In addition to controlling the inactive opacity level of the ViewCube command, you can also control the following properties for the ViewCube command:

- Size
- Position
- Default orientation
- Compass display

Using the Compass

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





Reorient the View of a Model with the ViewCube

ViewCube is used to reorient the current view of a model. You can reorient the view of a model with the ViewCube by clicking 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

Use the ViewCube menu to restore and define the Home view of a model, switch between view projection modes, and change the interactive behavior and appearance of the ViewCube.

The ViewCube menu has the following options:

- **Go Home**

  restores the Home view saved with the model.

- **Orthographic**

  switches the current view to orthographic projection.

- **Perspective**

  switches the current view to perspective projection.

- **Perspective with Ortho Faces**

  switches the current view to perspective projection unless the current view aligns with a face view defined on the ViewCube.

- **Lock to Selection**

  uses the selected objects to define the center of the view when a view orientation change occurs with the ViewCube.**Note:** If you click Home on the ViewCube, the view returns to the Home view even if Lock to Current Selection is selected.

- **Set Current View as Home**

  defines the Home view of the model based on the current view.

- **Restore Default Home**

  resets the Home view of the model to its default orientation.

- **Set Current View as Front**

  defines the Front view of the model based on the current view.

- **Reset Front**

  resets the Front view of the model to its default orientation.

- **Properties**

  displays the dialog box so you can adjust the appearance and behavior of the ViewCube.

- **Help Topics**

  Opens the Help system and displays the topic for the ViewCube.

Procedure

To display the ViewCube menu, do the following:

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

SteeringWheels

SteeringWheels are tracking menus (that follow your cursor) from which you can access different 2D and 3D navigation commands from a single command.

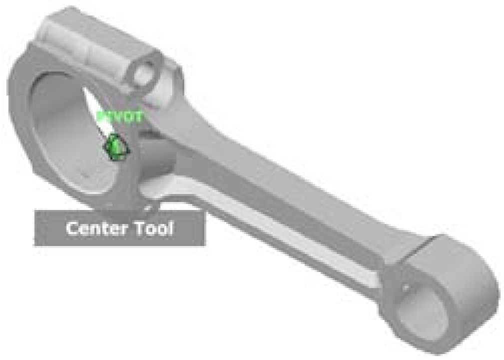
Navigation commands

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

Center command

With the Center command, you can define the center of the current view of a model. To define the center, drag the cursor over the model. A sphere is displayed in addition to the cursor. The sphere indicates that the point below the cursor in the model is the center of the current view. When you release the mouse button, the model is centered on the sphere.

Note: If a center point on a model cannot be identified, then an icon indicating that the operation cannot be performed. A circle with a diagonal line is displayed instead of the sphere.



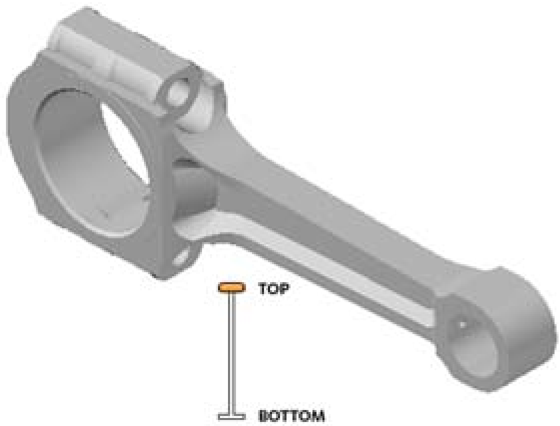
The point defined by the Center command provides a focal point for the Zoom command and a pivot point for the Orbit command.

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

Up/Down command

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

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



Procedure

1 Display one of the Full Navigation wheels or the Tour Building wheels.

2 Click and hold down the Up/Down wedge.The Vertical Distance indicator is displayed.

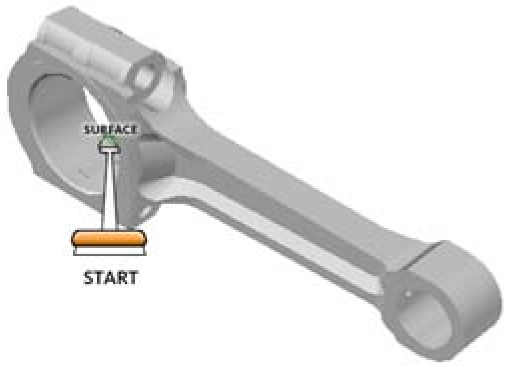
3 Drag up or down to change the elevation of the view.

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

Forward command

Use the Forward command to change the magnification of the model by increasing or decreasing the distance between the current point of view and the pivot point. The distance that you can move forward or backward is limited by the position of the pivot point.

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



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

Procedure

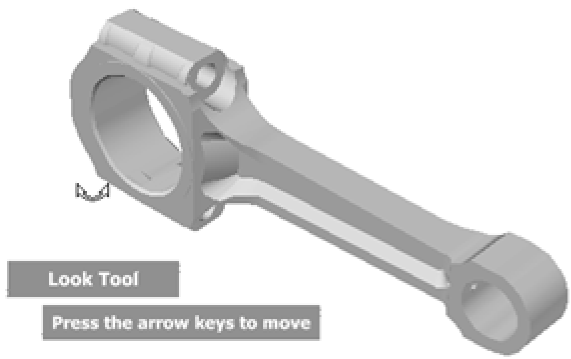
1. Display the big Tour Building wheel.
2. Click and hold down the Forward wedge within the scope of the model. The Drag Distance indicator is displayed.

   Note: If you click the Forward wedge once, the model moves forward 50% of the distance between the current location and the pivot point.
3. Drag the cursor up or down to change the distance from which you view the model.
4. Release the button on your pointing device to return to the wheel.

Look command

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

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



Walking through a Model

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

Invert Vertical Axis

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

Orbit command

You use the Orbit command to change the orientation of a model. The cursor changes to the Orbit cursor. As you drag the cursor, the model rotates around a pivot point while the view remains fixed.

Specify the Pivot Point

The pivot point is the base point used when rotating the model with the Orbit command. You can specify the pivot point in the following ways:

- **Default pivot point**

  . When you first open a model, the target point of the current view is used as the pivot point for orbiting the model.

- **Select objects**

  . You can select objects before the Orbit command is used to calculate the pivot point. The pivot point is calculated based on the center of the extents of the selected objects.

- **Center command**

  . You can specify a point on the model to use as the pivot point for orbiting with the NavSWheelCentercommand.htmCenter command .

- **Ctrl+Click and drag**

  . Press and hold down the
  Ctrl

  key before clicking the Orbit wedge or while the Orbit command is active. Then drag to the point on the model to use as the pivot point. This option is only available when using the big and mini Full Navigation wheels or the mini View Object wheel. **Note:** While the Orbit command is active, you can be press and hold the **CTRL** key at anytime to move the pivot point used by the Orbit command.

Maintain Up Direction

You can control how the model orbits around the pivot point by choosing to maintain the up direction of the model. When the up direction is maintained, orbiting is constrained along the *XY* axis and in the *Z* direction. If you drag horizontally, the camera moves parallel to the *XY* plane. If you drag vertically, the camera moves along the *Z* axis.

If the up direction is not maintained, you can roll the model using the roll ring which is centered around the pivot point. Use the properties dialog box for the SteeringWheels to control whether the up direction is maintained or not for the Orbit command.

Pan command

When the Pan command is active, the Pan cursor (a four-sided arrow) is displayed. Dragging the pointing device moves the model in the same direction. For example, dragging upward moves the model up, while dragging downward moves the model down.

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

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

Rewind command

As you use the navigation commands to reorient the view of a model, the previous view is saved to the navigation history. The navigation history holds a representation of the previous views of the model along with a thumbnail. A separate navigation history is maintained for each window; it is not maintained after the window is closed. Rewind navigation history is view-specific.

With the Rewind command, you can retrieve previous views from the navigation history. From the navigation history, you can restore a previous view or scroll through all the saved views.

When you hold down the button on the pointing device over the Rewind command on the wheel, the Rewind History panel is displayed. You can scroll through the navigation history. To restore one of the previous views in the navigation history, drag the bracket to the left in the Rewind History panel.

Note: Rewind history is not saved between sessions.

Walk command

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

Constrain the Walk Angle

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

Movement Speed

As you walk or fly through a model, you can control the movement speed. Movement speed is controlled by the distance in which the Cursor is moved from the Center Circle icon and the current movement speed setting. You can adjust the movement speed setting permanently or temporarily as you use the Walk command. To adjust the movement speed permanently, use the Properties dialog box for the SteeringWheels or the and keys when the Walk command is active. To increase movement speed temporarily, press and hold the + (plus) key while using the Walk command.

Change the Elevation

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

Zoom command

You use the Zoom command to change the zoom magnification of a model. The following mouse click and key combinations are available to control how the Zoom command behaves:

- **Click.**

  If you click the Zoom command on a wheel, the current view is zoomed in by a factor of 25 percent. If you are using the Full Navigation wheel, incremental zoom must be enabled in the Properties dialog box for the SteeringWheels.

- **SHIFT+click.**

  If you hold down the
  SHIFT

  key before you click the Zoom command on a wheel, the current view is zoomed out by a factor of 25 percent. Zooming is performed from the current location of the cursor, and not the current pivot point. **Note:** When you start the Zoom command from the Full Navigation wheel, incremental zooming must be enabled in the Properties dialog box for the SteeringWheels to use **CTRL**+click and **SHIFT**+click.

- **CTRL+click.**

  If you hold down the
  CTRL

  key before you click the Zoom command on a wheel, the current view is zoomed in by a factor of 25 percent. Zooming is performed from the current pivot point, and not the location of the cursor.

- **Click and drag.**

  If you click the Zoom command and hold down the button on your pointing device, you can adjust the magnification of the model by dragging up and down.

- **CTRL+click and drag.**

  When using the Full Navigation wheels or the mini View Object wheel, you can control the target point used by the Zoom command. By holding down the
  CTRL

  key, the Zoom command uses the location of the previous pivot point defined by the Zoom, Orbit, or Center command.

- **SHIFT+click and drag.**

  When using the Full Navigation wheels or the mini View Object wheel, zoom in to an area of the model by dragging a rectangular window around the area to fit in the window. Hold down the
  SHIFT

  key and then click and drag a window around the area in which to zoom. **Note:** If you hold down the **CTRL** key along with the **SHIFT** key, you can zoom in to an area of a model using a center-based window, instead of one defined by opposite corners.

- **Mouse wheel.**

  When a wheel is displayed, scroll the mouse wheel up or down to zoom the view of the model in or out.

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

Zoom Constraints

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

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

Navigation Wheels

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

2D Navigation Wheel

With this wheel, you can access basic 2D navigation commands; it is useful when you do not have a pointing device with a scroll wheel. The wheel includes the Pan and Zoom commands.

The 2D Navigation wheel wedges have the following options:

- **Pan:**

  Repositions the current view by panning.

- **Zoom:**

  Adjusts the magnification of the current view.
  Note: Pan and Zoom in a 2D SteeringWheel are used to pan or zoom the page space. In all other wheels, Pan and Zoom moves the camera.

- **Rewind:**

  Restores the most recent view orientation. You can move backward or forward by clicking and dragging left or right.

Full Navigation Wheels

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

Note: When one of the Full Navigation wheels is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the SHIFT key while pressing and holding the middle mouse button to orbit the model.

Big Full Navigation Wheel

The big Full Navigation wheel wedges include the following options:

- **Zoom:**

  Adjusts the magnification of the current view.

- **Rewind:**

  Restores the most recent view. Move backward or forward by clicking and dragging left or right.

- **Pan:**

  Repositions the current view by panning.

- **Orbit:**

  Rotates the current view around a fixed pivot point.

- **Center:**

  Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation commands.

- **Walk:**

  Simulates walking through a model.

- **Look:**

  Swivels the current view.

- **Up/Down:**

  Slides the current view of a model along the Z axis of the model.

Mini Full Navigation Wheel

The mini Full Navigation wheel wedges include the following options:

- **Zoom (Top wedge):**

  Adjusts the magnification of the current view.

- **Walk (Upper right wedge):**

  Simulates walking through a model.

- **Rewind (Right wedge):**

  Restores the most recent view. Move backward or forward by clicking and dragging left or right.

■ **Up/Down (Lower right wedge):**

Slides the current view of a model along the Z axis of the model.

■ **Pan (Bottom wedge):**

Repositions the current view by panning.

■ **Look (Lower left wedge):**

Swivels the current view.

■ **Orbit (Left wedge):**

Rotates the current view around a fixed pivot point.

■ **Center (Upper left wedge):**

Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation commands.

Tour Building Wheels

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

Big Tour Building Wheel

The big Tour Building wheel wedges have the following options:

■ **Forward:**

Adjusts the distance between the current point of view and the defined pivot point of the model. Clicking once moves forward half the distance as far as the object you clicked.

■ **Look:**

Swivels the current view.

- **Rewind:**

  Restores the most recent view. You can move backward or forward by clicking and dragging left or right.

- **Up/Downcommand:**

  Slides the current view of a model along the *Z* axis of the model.

Mini Tour Building Wheel

The mini Tour Building wheel wedges have the following options:

- **Walk (Top wedge):**

  Simulates walking through a model.

- **Rewind (Right wedge):**

  Restores the most recent view. You can move backward or forward by clicking and dragging left or right.

- **Up/Down (Bottom wedge):**

  Slides the current view of a model along the *Z* axis of the model.

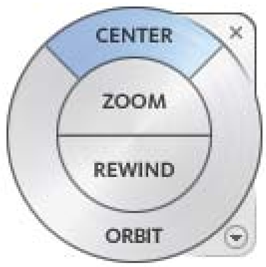
- **Look (Left wedge):**

  Swivels the current view.

Note: When the mini wheel is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the Shift key while pressing and holding the middle mouse button to orbit the model.

View Object Wheels

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



Big View Object Wheel

The big View Object wheel wedges have the following options:

- **Center:**

  Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation commands.

- **Zoom:**

  Adjusts the magnification of the current view.

- **Rewind:**

  Restores the most recent view orientation. You can move backward or forward by clicking and dragging left or right.

- **Orbit:**

  Rotates the current view around a fixed pivot point.

Mini View Object Wheel

The mini View Object wheel wedges have the following options:

- **Zoom (Top wedge):**

  Adjusts the magnification of the current view.

- **Rewind (Right wedge):**

  Restores the most recent view. You can move backward or forward by clicking and dragging left or right.

- **Pan (Bottom wedge):**

  Repositions the current view by panning.

- **Orbit (Left wedge):**

  Rotates the current view around a fixed pivot point.

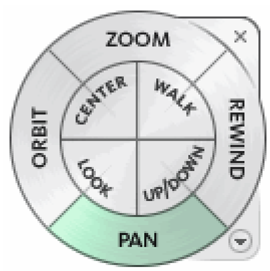
Note: When the mini wheel is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the Shift key while pressing and holding the middle mouse button to orbit the model.

Overview of SteeringWheels

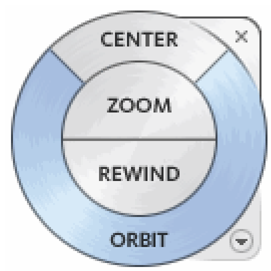
SteeringWheels, also known as wheels, can save you time by combining many of the common navigation commands into a single interface. Wheels are specific to the context from which a model is being viewed.

The following illustrations show the different wheels available:

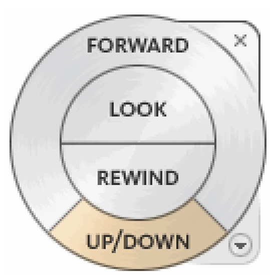
Full Wheels



Full Navigation Wheel



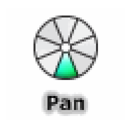
View Object Wheel



Tour Building Wheel

2D Wheel

Mini Wheels



Mini Full Navigation Wheel

Mini View Object Wheel

Mini Tour Building Wheel

Display and Use Wheels

Pressing and dragging on a wedge of a wheel is the primary mode of interaction. After a wheel is displayed, click one of the wedges and hold down the button on the pointing device to activate the navigation command. Drag to reorient the current view. Releasing the button returns you to the wheel.

Appearance of the Wheels

You can control the appearance of the wheels by switching between the different styles of wheels that are available, or by adjusting the size and opacity. Wheels (except the 2D Navigation wheel) are available in two different styles: big and mini.

The size of a wheel controls how large or small the wedges and labels appear on the wheel; the opacity level controls the visibility of the objects in the model behind the wheel.

Wheel tooltip, command Messages, and command Cursor Text

tooltips are displayed for each button on a wheel as the cursor is moved over them. The tooltips appear below the wheel and identify what action to perform if the wedge or button is clicked.

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

Wheel Menu

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

The Wheel menu has the following options:

- **Mini View Object Wheel.**
  Displays the mini View Object wheel.
- **Mini Tour Building Wheel.**
  Displays the mini Tour Building wheel.
- **Mini Full Navigation Wheel.**
  Displays the mini Full Navigation wheel.
- **Full Navigation Wheel.**
  Displays the big Full Navigation wheel.
- **Basic Wheels.**
  Displays the big View Object or Tour Building wheel.
- **Go Home.**
  Goes to the Home view saved with the model.
- **Fit to Window.**
  Resizes and centers the current view to display all objects.
- **Restore Original Center.**
  Restores the center point of the view to the extents of the model.
- **Level Camera.**
  Rotates the current view so it is relative to the *XY* ground plane.
- **Increase Walk Speed.**

Increases the walk speed used for the Walk command by two times.

- **Decrease Walk Speed.**

  Decreases the walk speed used for the Walk command by one half.

- **Orient to View.**

  Orients the camera to match the view angle of the selected view (a plan, elevation, section, or 3D view).

- **Orient to a Plane.**

  Adapts the view according to a specific plane.

- **Save View.**

  Saves the current view orientation with a unique name.**Note:** Use Save View only to save a 3D view with a unique name when you are viewing the default 3D view. If you are viewing a previously saved orthographic 3D view or a perspective (camera) 3D view, the view is saved with the new orientation and you are not prompted to supply a unique name.

- **Increase/Decrease Focal Length.**

  Acts as a zoom lens on the model, because it changes the focal length of the camera in a perspective view.

- **Move Crop Boundary.**

  Moves the position of the crop boundary around in a perspective view.

- **Re-center Crop Boundary.**

  Repositions the crop boundary to the center of the perspective view.

- **Help.**

  Displays the Help system and displays the topic about the wheels.

- **Properties.**

  Displays the dialog box where you can adjust the preferences for the wheels.

- **Close Wheel.**

  Closes the wheel.

Procedure

- Click the down arrow in the lower-right corner of the wheel or right-click the wheel.

Navigation Bar

The navigation bar is a user interface element from which you can access both unified and product-specific navigation commands. Unified navigation commands (such as Autodesk ViewCube and SteeringWheels) can be found across many Autodesk products. Product-specific navigation commands are unique to a product. The navigation bar floats over and along one of the sides of the window of the current model.

The unified and product-specific navigation commands are organized into separate areas of the navigation bar. The unified navigation commands are located on the two ends of the navigation bar, while the product-specific navigation commands are located in the center. Start navigation commands by clicking one of the buttons on the navigation bar. Or, select one of the commands from a list that is displayed when you click the smaller portion of a split button.

Available Navigation commands

The following unified navigation commands are available from the navigation bar:

- **ViewCube.**

  Indicates the current orientation of a model and is used to reorient the current view of a model.

- **SteeringWheels.**

Collection of wheels that offer rapid switching between specialized navigation commands.

The following product-specific navigation commands are available from the navigation bar:

- **Pan.**

  Moves the view parallel to the screen.

- **Zoom commands.**

  Set of navigation commands for increasing or decreasing the magnification of the current view of a model.

- **Orbit commands.**

  Set of navigation commands for rotating the current view of a model.

- **Look At.**

  Views faces of a model from a selected plane.

Reposition and Reorient the Navigation Bar

Adjust the position and orientation of the navigation bar by linking it to the ViewCube, docking it when the ViewCube is not displayed, or freely positioning it along one of the edges of the current window. When linked to the ViewCube, the navigation bar is positioned above or below the ViewCube and in a vertical orientation. When not linked or docked, the navigation bar can be freely aligned along one of the edges of the window of the current model.

You specify how the navigation bar can be repositioned from the Customize menu. When not linked to the ViewCube or docked, a grip handle is displayed. Drag the grip handle on the navigation bar to reposition it along one of the sides of the window of the current window.

If the side of the window that the navigation bar is aligned to is not long enough to show the entire navigation bar, it is truncated to fit. When truncated, a More Controls button is displayed and replaces the Customize button. When you click the More Controls button, a menu is displayed that contains the navigation commands that are not currently being displayed.

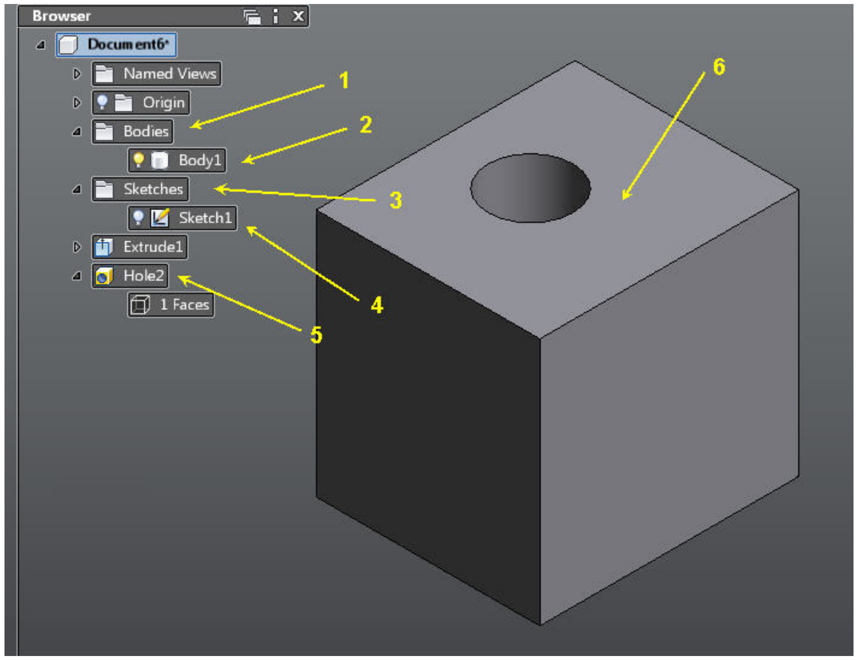
Control the Display of Navigation commands on the Navigation Bar

You can control which unified and product-specific navigation commands are displayed on the navigation bar with the Customize menu by clicking the drop-down arrow located on the bottom right of the navigation bar. The Customize menu is displayed by clicking the Customize button. From the

Customize menu, you click the navigation commands to display on the navigation bar. The position of the navigation commands on the navigation bar is predefined and cannot be changed.

Create 3D Models

This section describes some common techniques that are used in Inventor Fusion to create designs. It covers a wide range of modeling techniques that can be used to create and edit model geometry and components in an Inventor Fusion design. First, a few notes about the nature of an Inventor Fusion design. The following image is an example of a simple Fusion design:



This simple design has some elements that are worth pointing out:

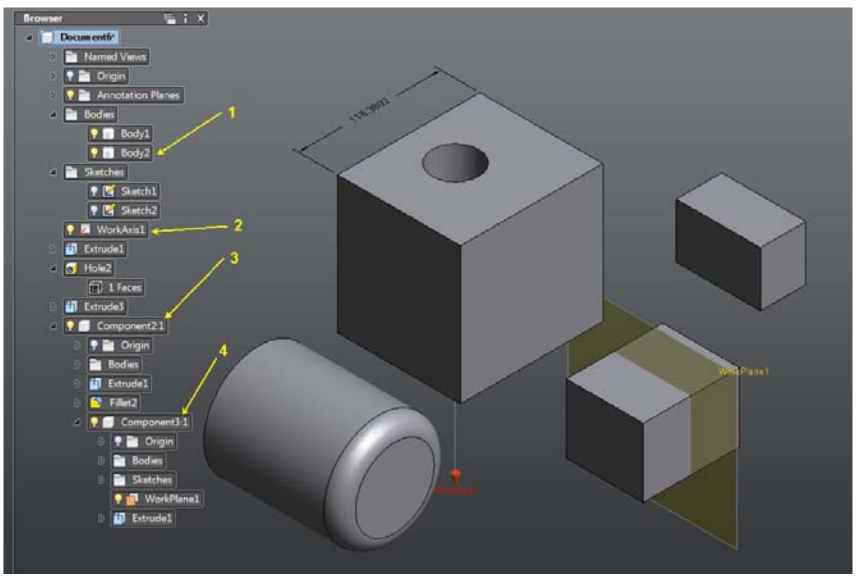
1. The Bodies folder in the browser. This folder contains all of the bodies for a component. This example has a single body Body1.
2. The Body node in the browser. Each body owned by the component has an entry in the Bodies folder.
3. The Sketches folder in the browser. This folder contains all of the sketches for a component. This example has a single sketch, which is invisible (the light bulb icon shows the visibility state for many items in a Fusion design).
4. The Sketch node in the browser. Each sketch owned by the component has an entry in the Sketches folder.
5. Feature nodes in the browser. These items represent geometry which has been create using a feature command in Fusion. In this example, the design has two features: Extrude1, and Hole2. The nodes underneath Feature nodes show individual body faces that were created by that feature.

6 The component body. The graphics area shows the geometry of the component. Most operations are performed by the user on this representation of the component.

An Inventor Fusion design may also contain:

- Multiple bodies
- Work geometry
- Annotation planes and annotations
- Named views
- Child components

An example of a more complex design is shown here:



Points of interest:

- The design contains a second body.
- The design contains a work axis.
- The design contains a child component (Component2).
- The design contains a second-level of component (Component3).

Create a Single Body

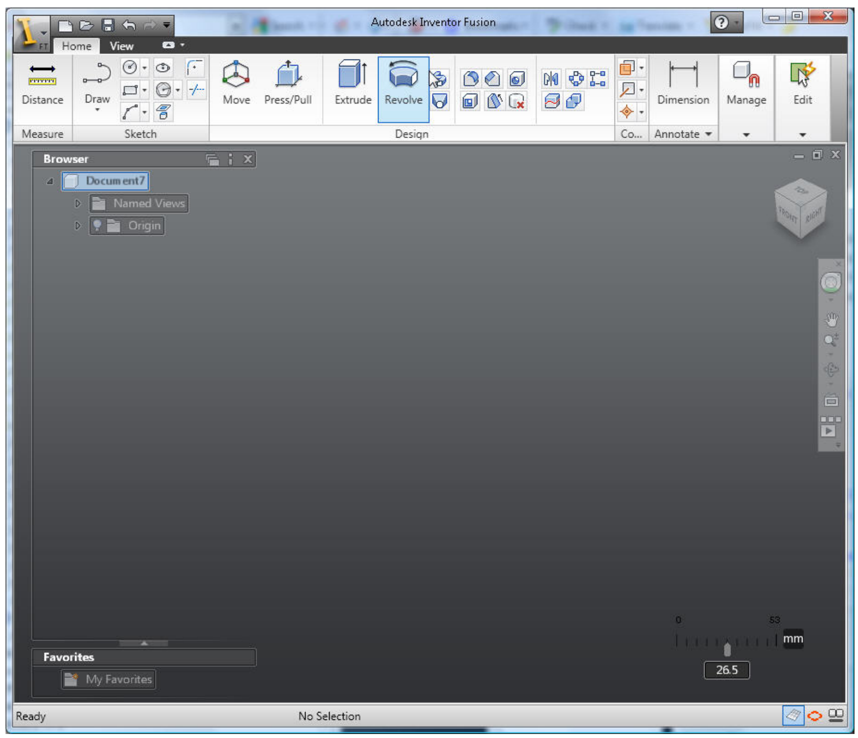
These steps illustrate how to start with an empty Fusion document, and create a single body in a simple design.

This example uses these Inventor Fusion capabilities:

- Sketching: Sketch on page 135
- Feature creation: Features on page 177

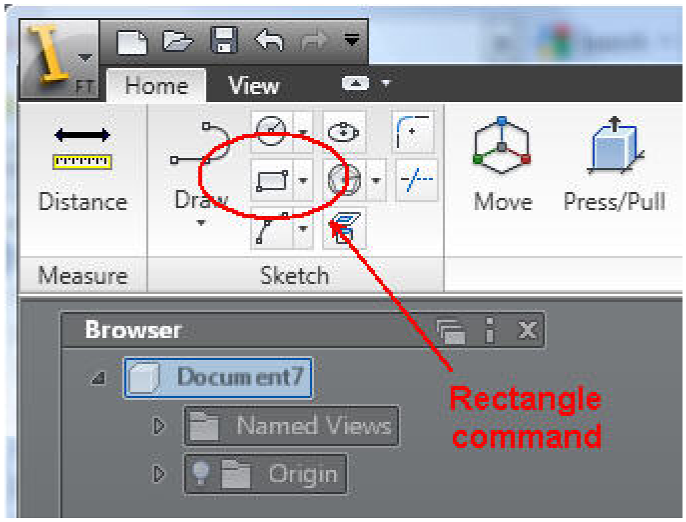
Starting point: An empty design

This example begins with an empty design. This is the state when Fusion is first invoked, or when the New icon is clicked. It looks like:



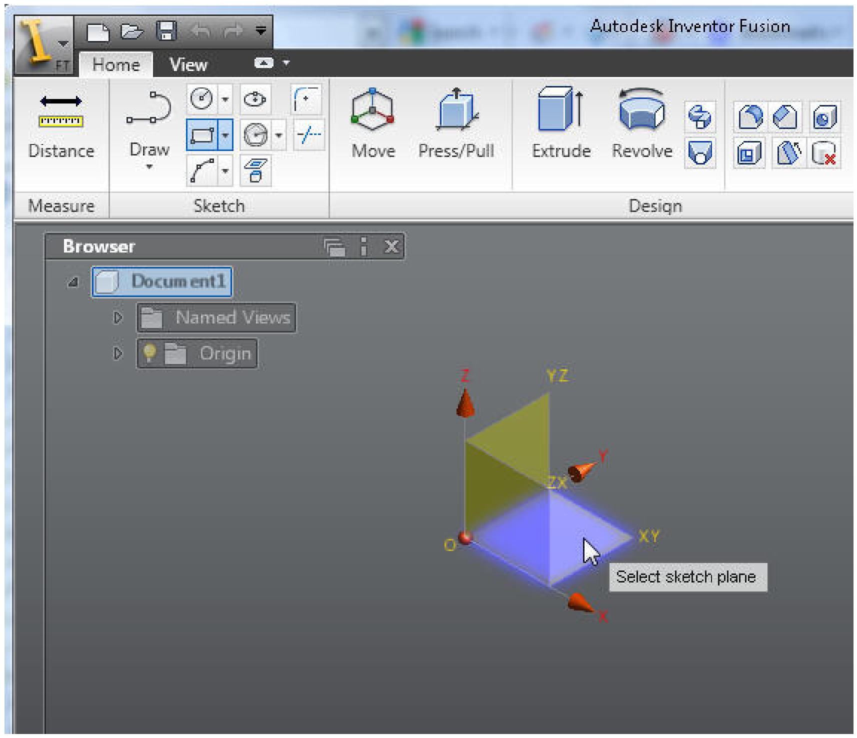
Step 1: Create a sketch rectangle

click the Rectangle command in the Ribbon

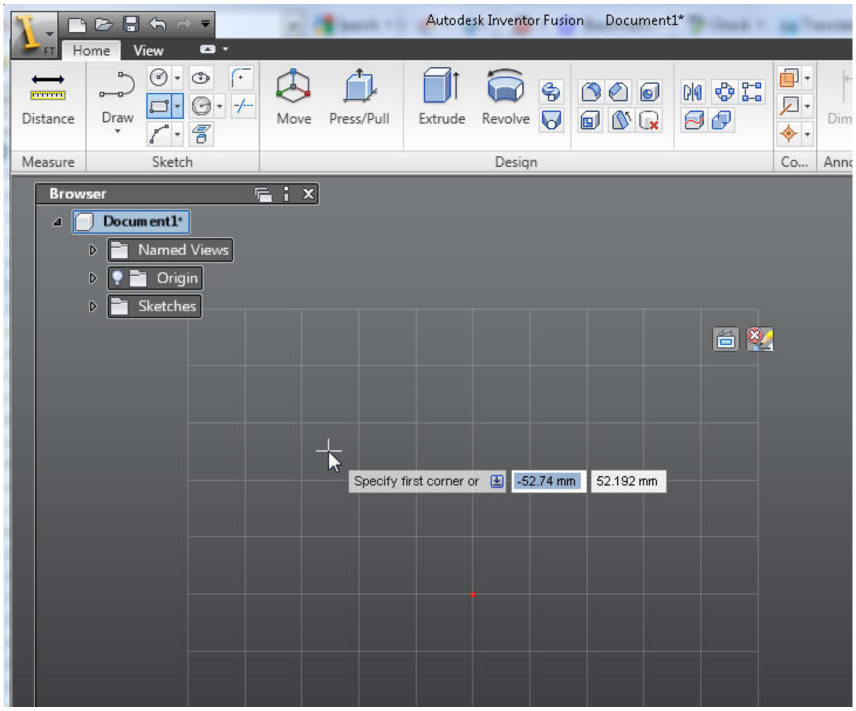


Because no sketch is currently active, the first step is to create a new sketch.

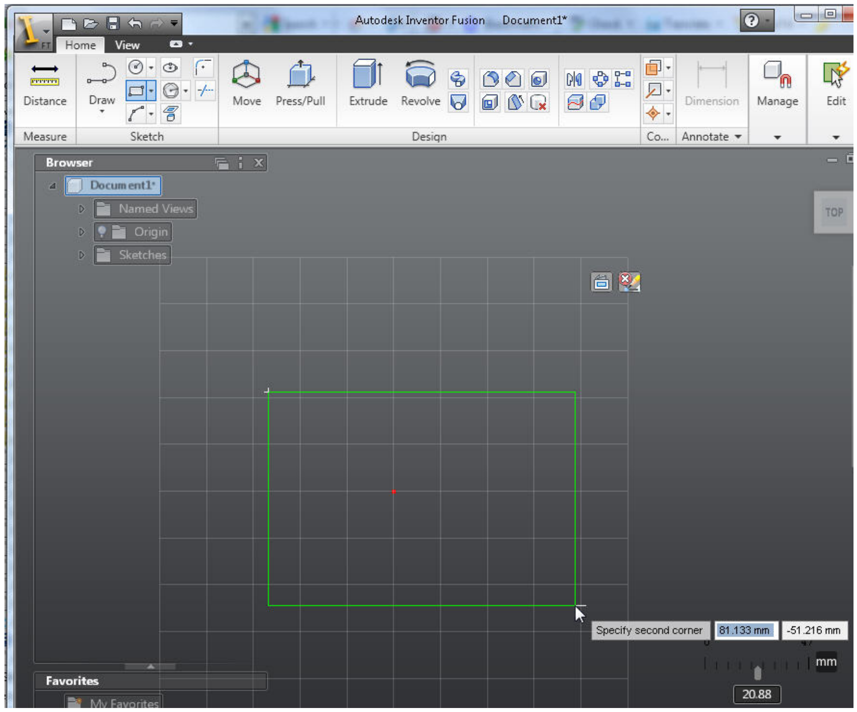
Select one of the origin work planes. In this example, we select the XY plane:



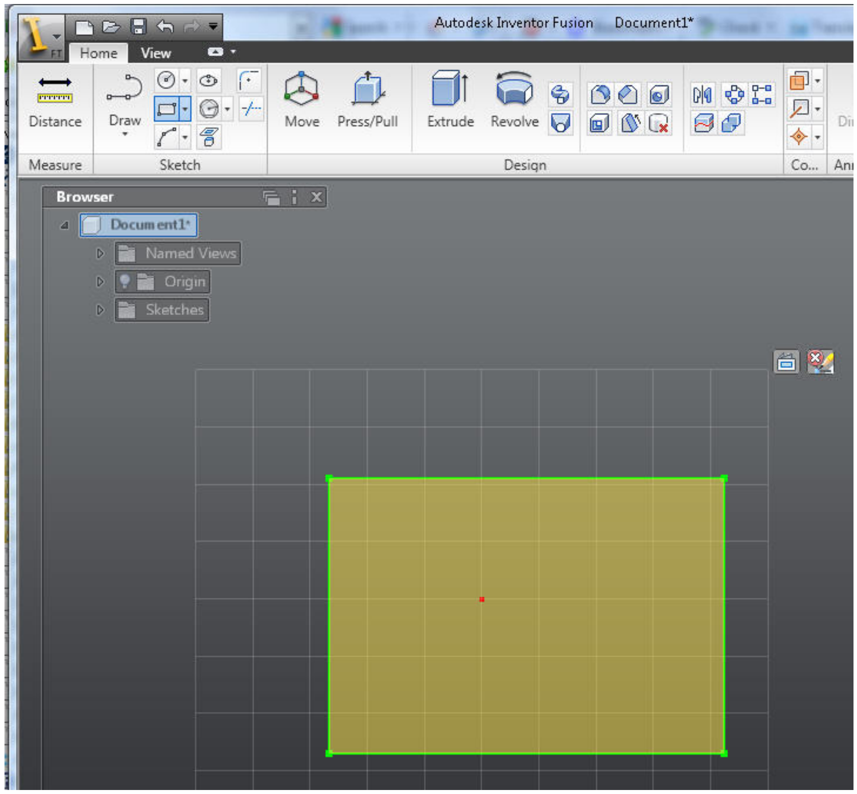
This creates and activates a new sketch, changes the view to look at the new sketch, and invokes the Rectangle command:



Click a first and second point to describe the rectangle:

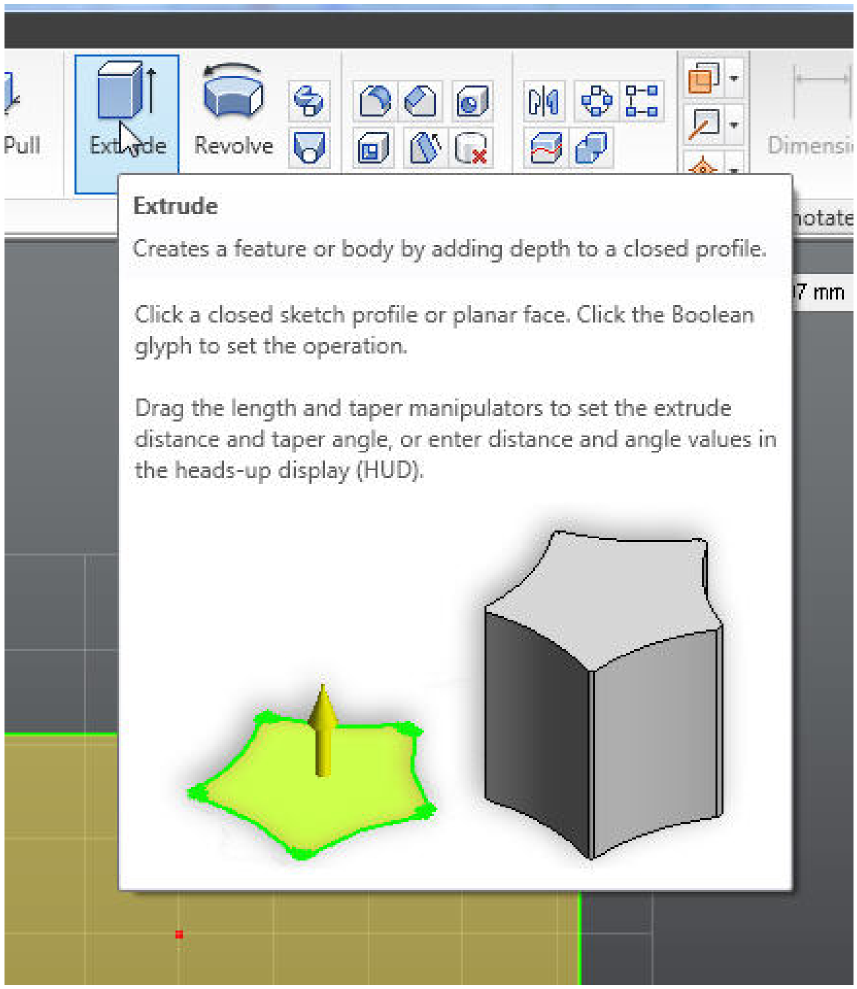


As soon as the rectangle is finished, Inventor Fusion recognizes the closed region, and shades it in a yellow color:

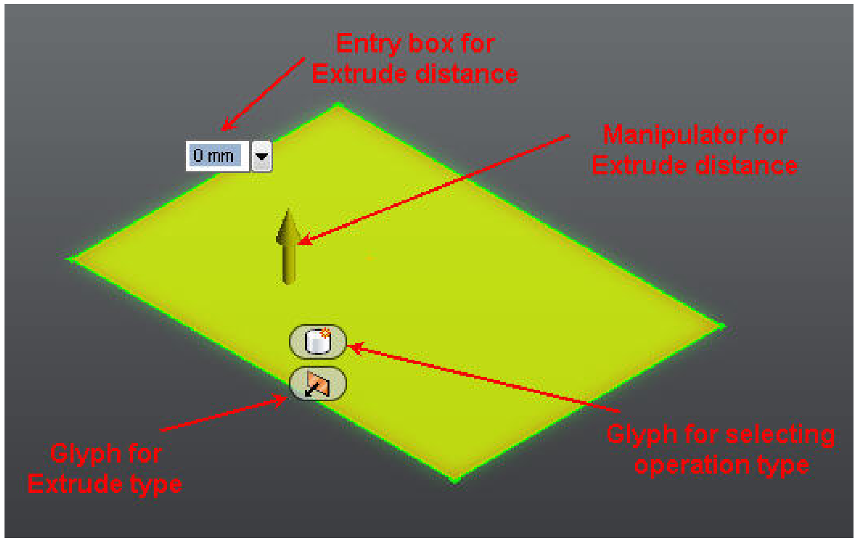


Step 2: Extrude the rectangle

Invoke the Extrude command. There are several ways to invoke Extrude. For this simple example, choose the Extrude command in the ribbon:

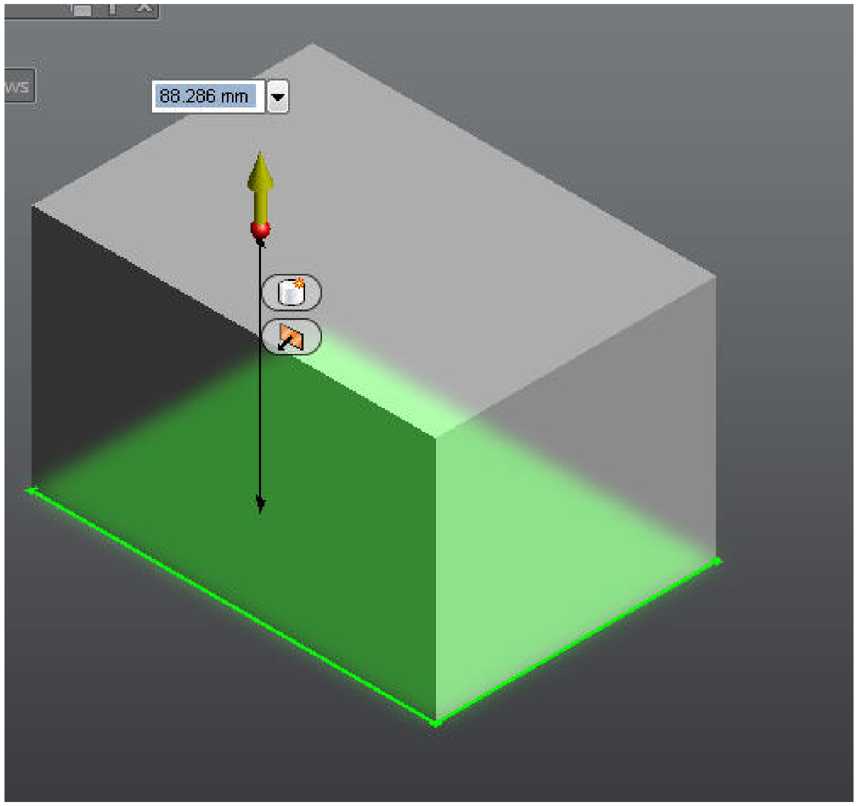


Because there is only a single closed region, Fusion automatically selects it. If more than one closed region were available, you would be required to explicitly select which region to use. Extrude shows the following elements:

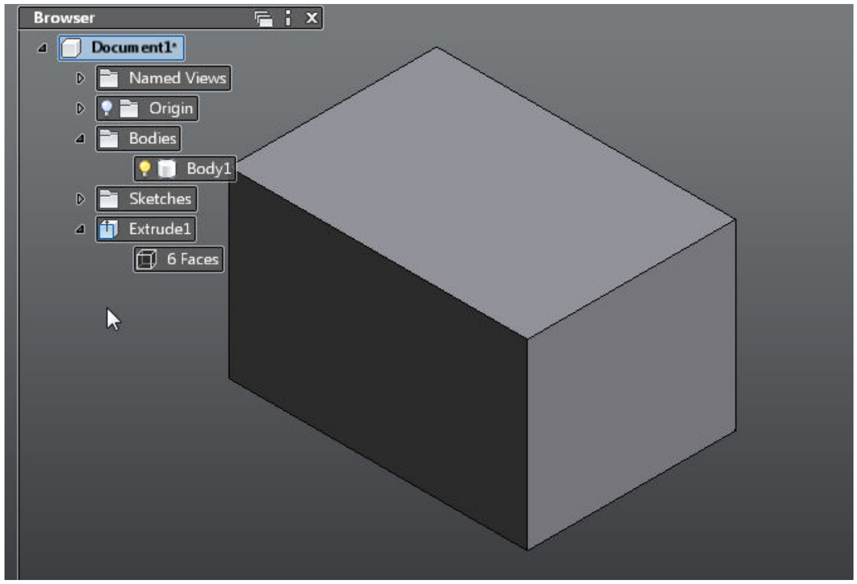


Note that these elements are also present in the ribbon, but are much more convenient to interact with in the graphics area

Drag to change the depth of the extrusion. You may drag over empty space, or directly over the distance manipulator. Dragging produces a preview of the extrusion:



Finish the command. There are a couple of ways to complete the command - starting another command, or clicking OK in the ribbon. In this case, click OK in the ribbon. This creates the body, as well as an Extrude feature in the browser:



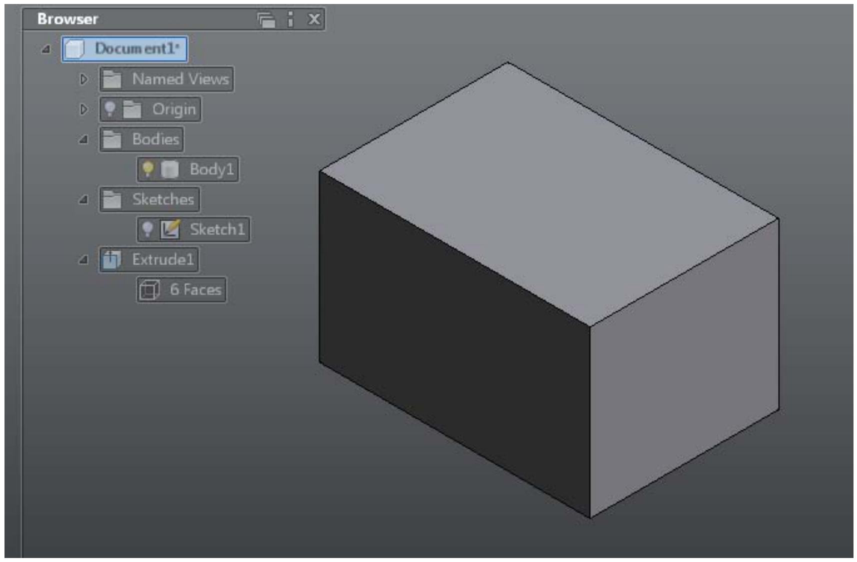
Create Multiple Bodies

The steps in this topic illustrate how to create a second body, starting with a Fusion document containing a single body, and create a second body. This example uses these Inventor Fusion capabilities:

- Sketching Sketch on page 135
- Feature creation Features on page 177

Starting point: a design containing a single body, in the root component

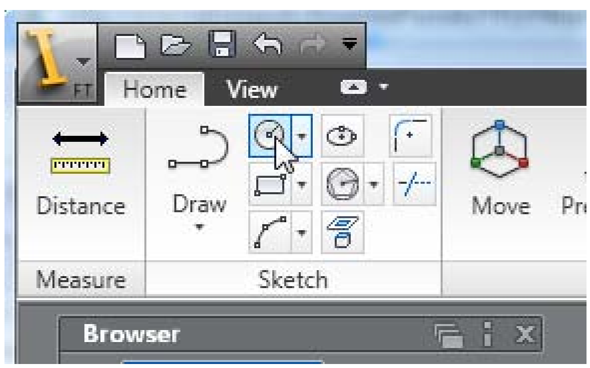
This is what the starting point for this example looks like:



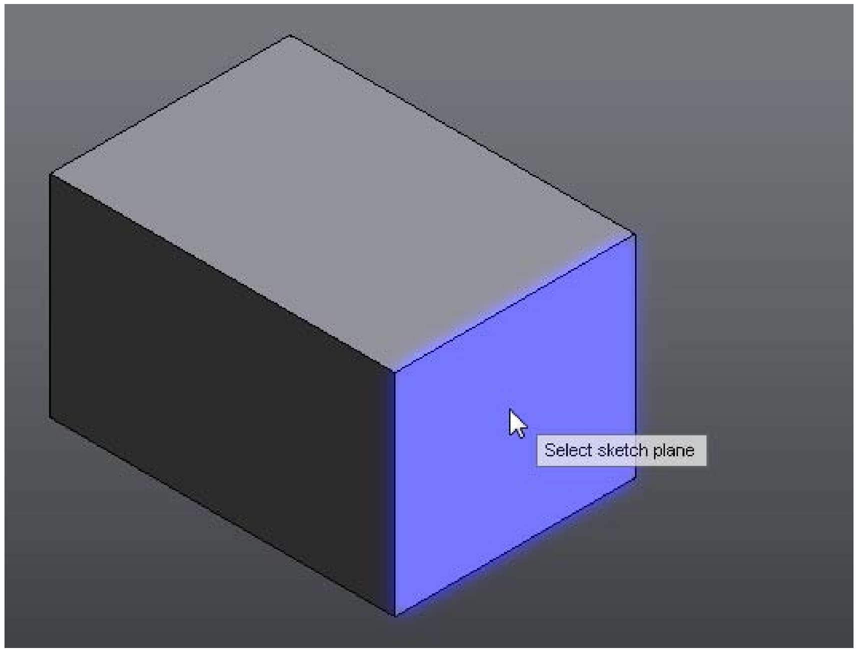
To learn how to create a design similar to this see: Create a Single Body on page 75

Step 1: Create a sketch and a sketch circle

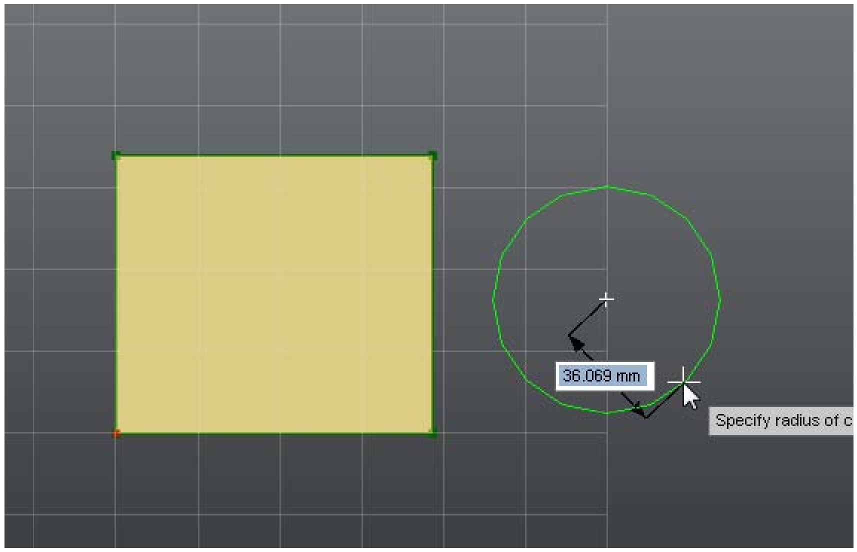
click the Circle command in the Ribbon



Because no sketch is currently active, the first step is to create a new sketch. Select a face of the existing body.

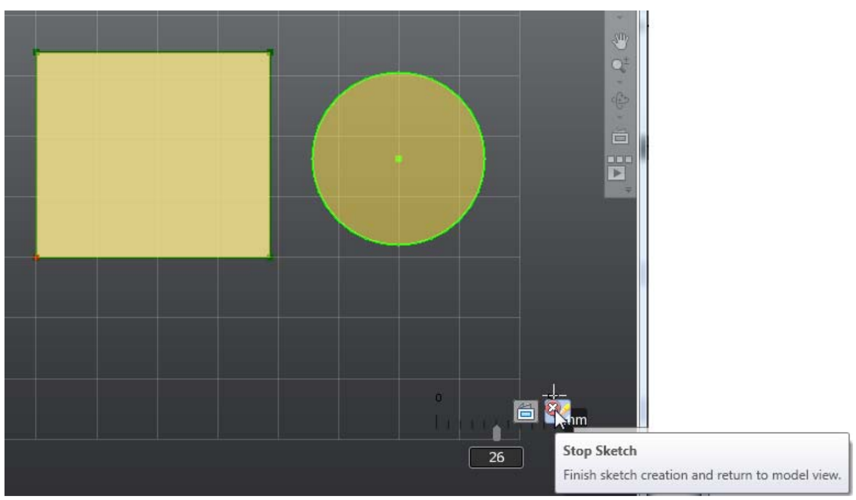


This creates and activates a new sketch, changes the view to look at the new sketch, and invokes the Circle command. Select a circle center and radius point, away from the existing body. Note that because we selected a face of the existing body, Fusion has generated sketch geometry that coincides with the edges of that face. This sketch geometry is fixed, and so can't be edited.

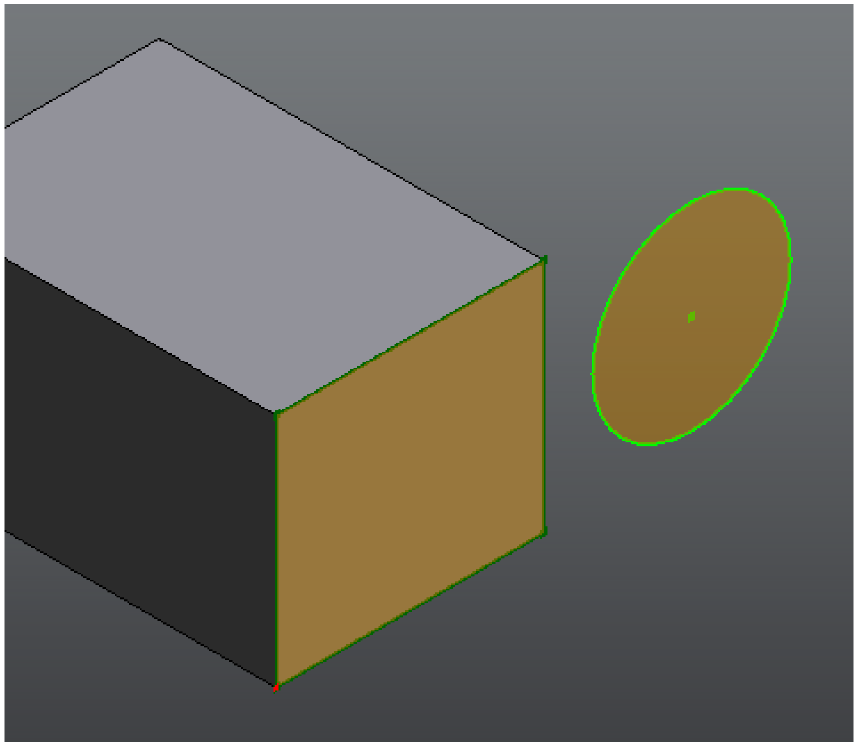


Step 2: Exit the sketch

In this example we illustrate how to exit sketch edit mode. click the Stop Sketch icon in the lower left corner of the screen.



This terminates the active sketch, and restores the view to where it was before sketching:

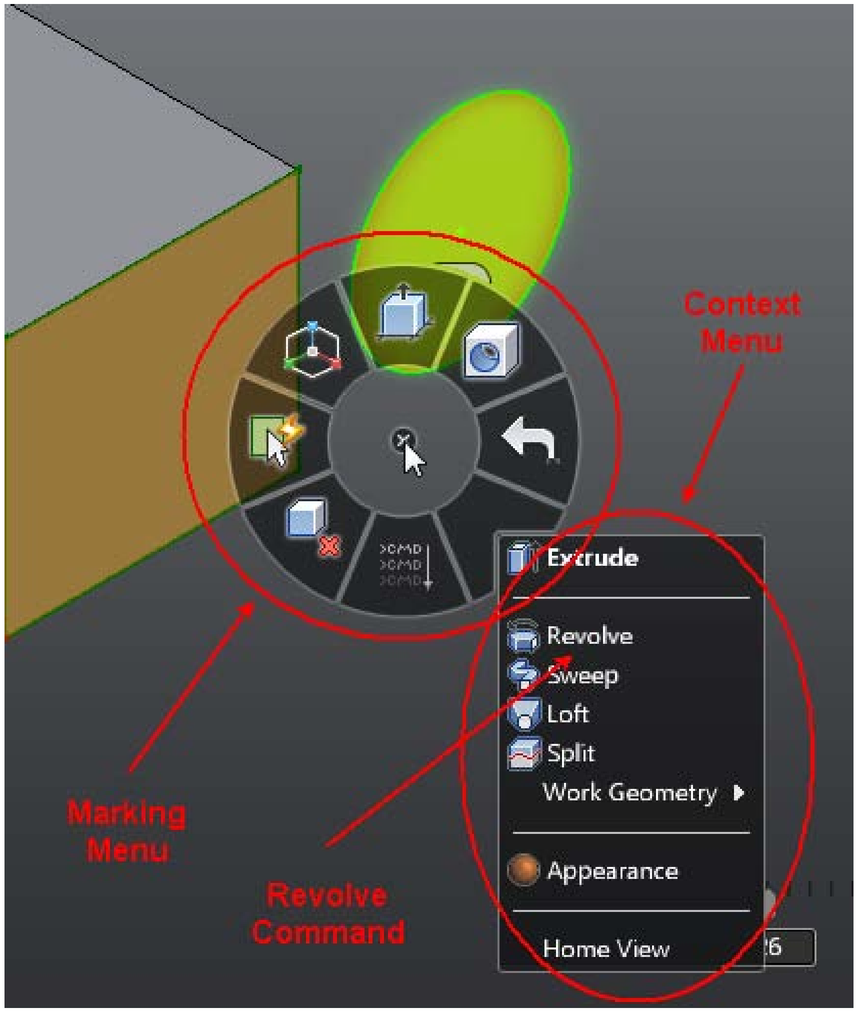


Note that Fusion recognizes the two closed regions, and shades them in a yellow color.

Step 3: Create a second body using the Revolve feature

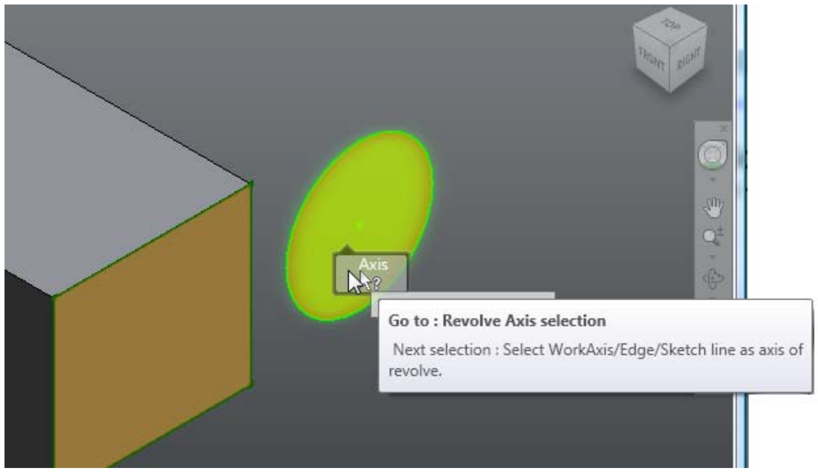
Invoke the Revolve command. There are several ways to invoke Revolve. For this simple example, we use the context menu.

Select the circular closed region, using the left mouse button. Click or hold down the right mouse button to bring up the marking menu and the context menu. Choose the Revolve command from the context menu:

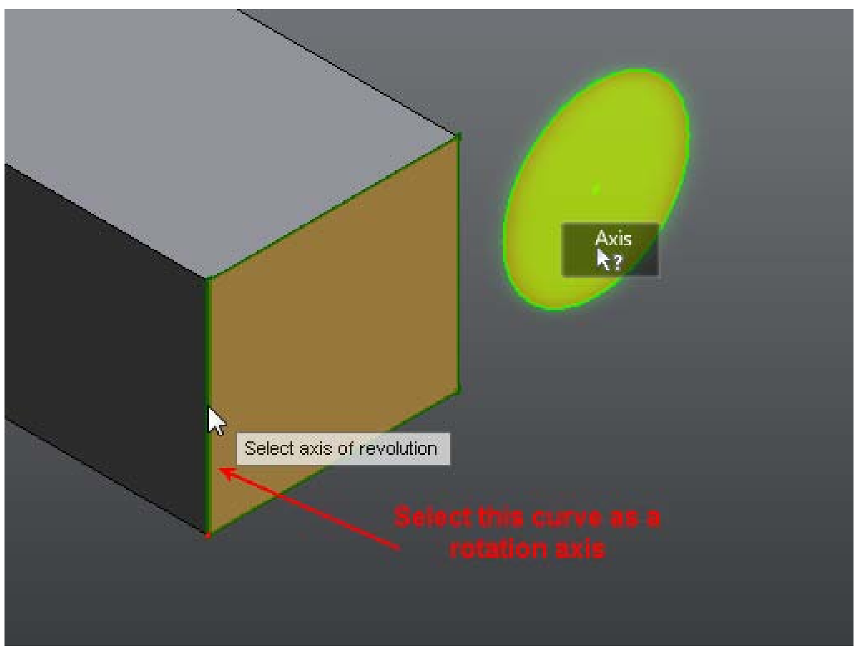


This brings up the Revolve command, with the circular region as the profile to Revolve.

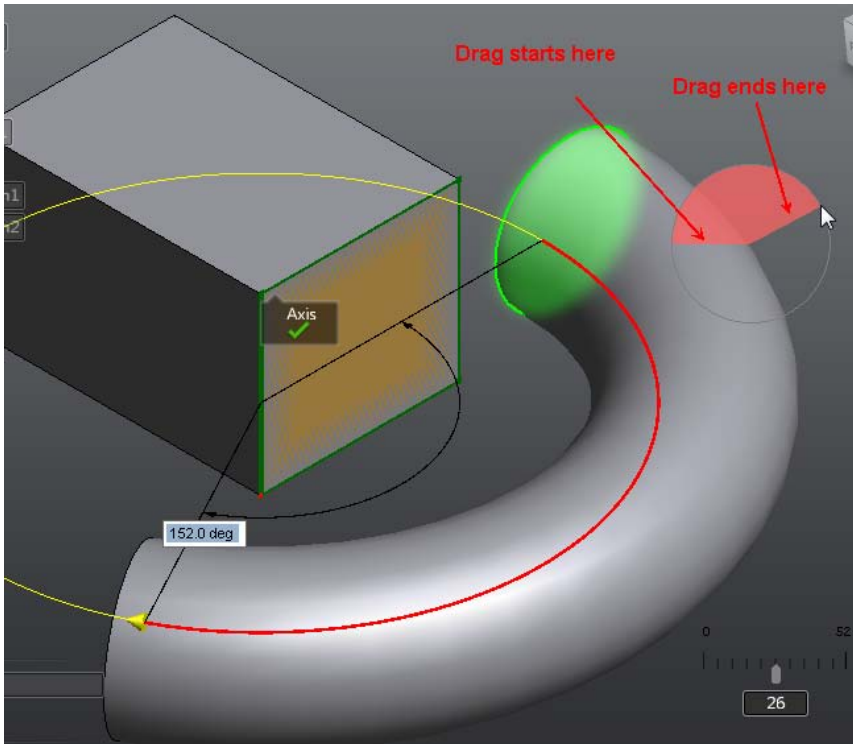
Revolve requires a second selection - an axis to revolve about. click the Axis glyph:



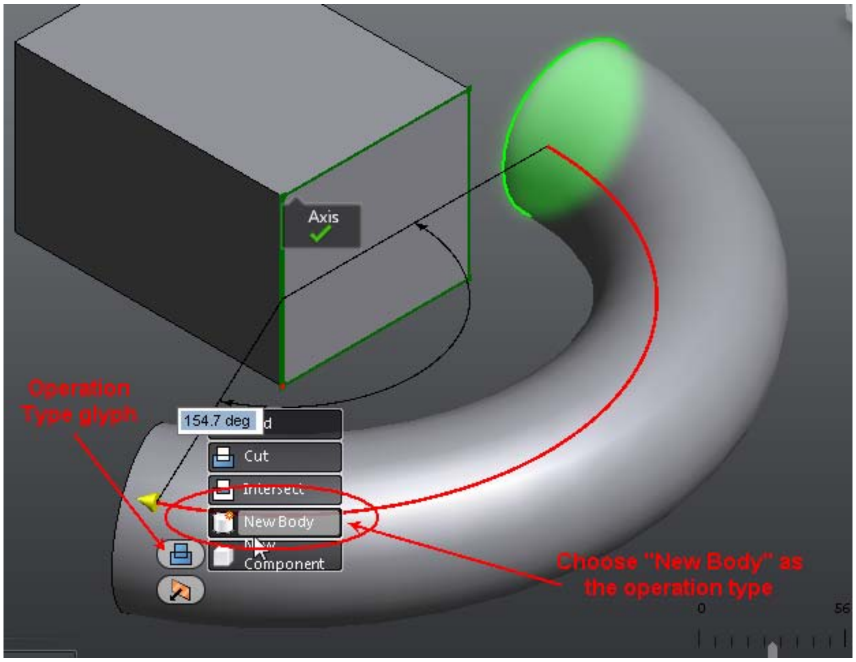
This prompts you to select an axis. You can select sketch lines or linear edges. In this case we select a sketch line from the face we sketched on:



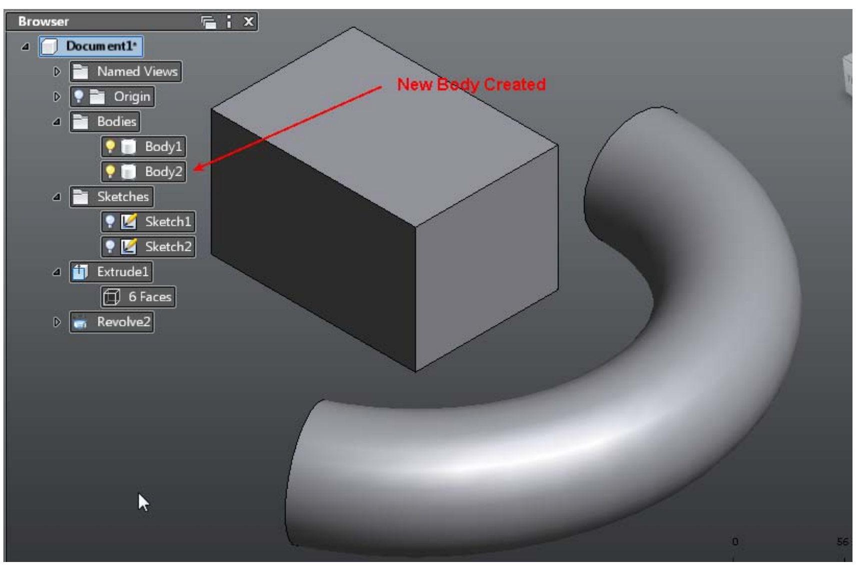
Once the axis is selected, begin dragging. In this example, we will drag in an area that is away from the manipulator. Click and drag the left mouse button in an area of the screen that is not over any graphics. The example drags to the upper right area of the screen. Note that the drag for Revolve,when dragging in open space is a 2-dimensional drag in the plane of the screen.



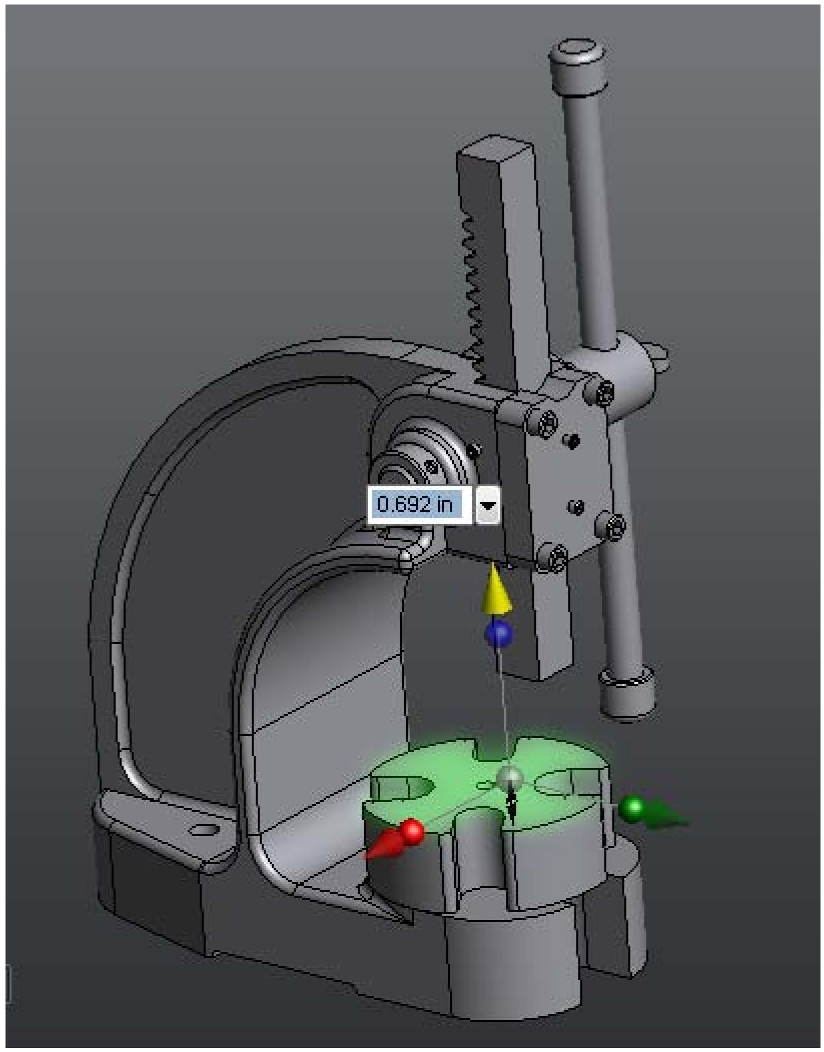
Next, change the Operation Type of the Revolve feature to create a new body. Click and hold down the left mouse button over the Operation Type glyph, and choose New Body.



Finally, click OK in the ribbon to create the Revolve and the new body. The results look like:



Modify a Body



This section discusses how to modify an already-existing geometric body in a Fusion design. This is one of the strengths of Inventor Fusion, and one of the main workflows for new users to understand to be productive when using Fusion.

Inventor Fusion provides a highly interactive environment for geometric editing. You can drag faces of the body into a variety of new positions using the commands described in the following section.

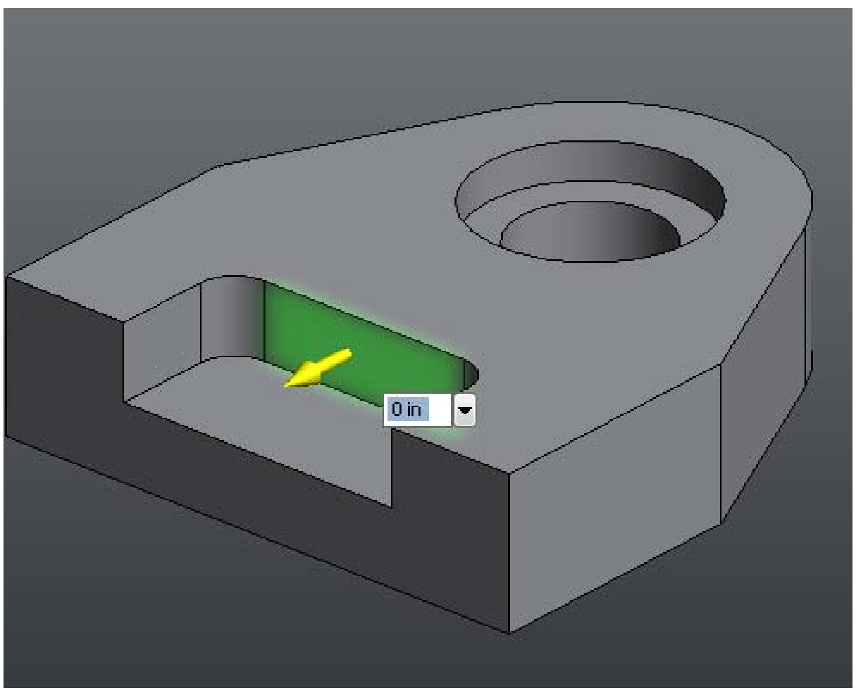
Related Topics:

- Dimensions and Body Constraints on page 184

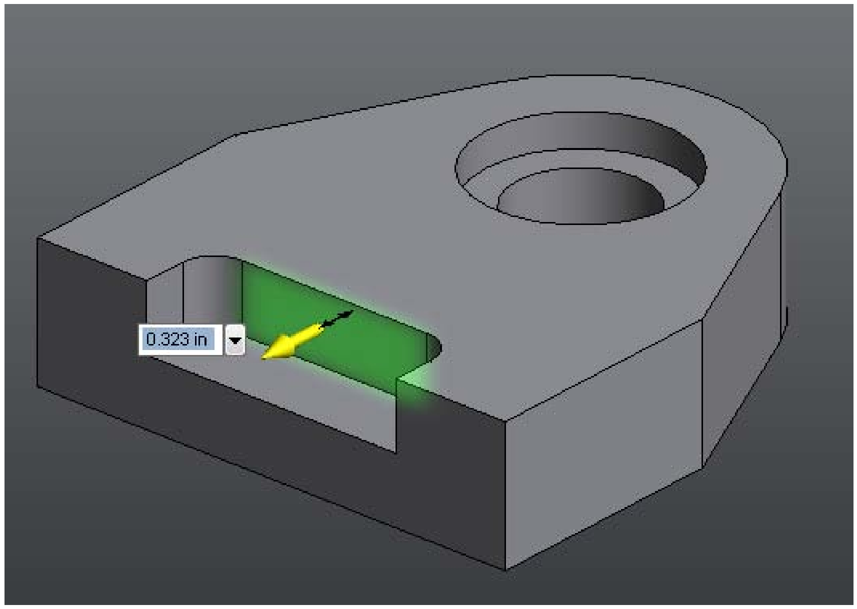
Press/Pull Command

The Press/Pull command is one way that a user can modify body geometry. In general, Press/Pull provides users with an Offset style of modification. That is, the modified geometry is replaced with an offset of itself.

For instance, in this simple design, the user can select a face such as this one:

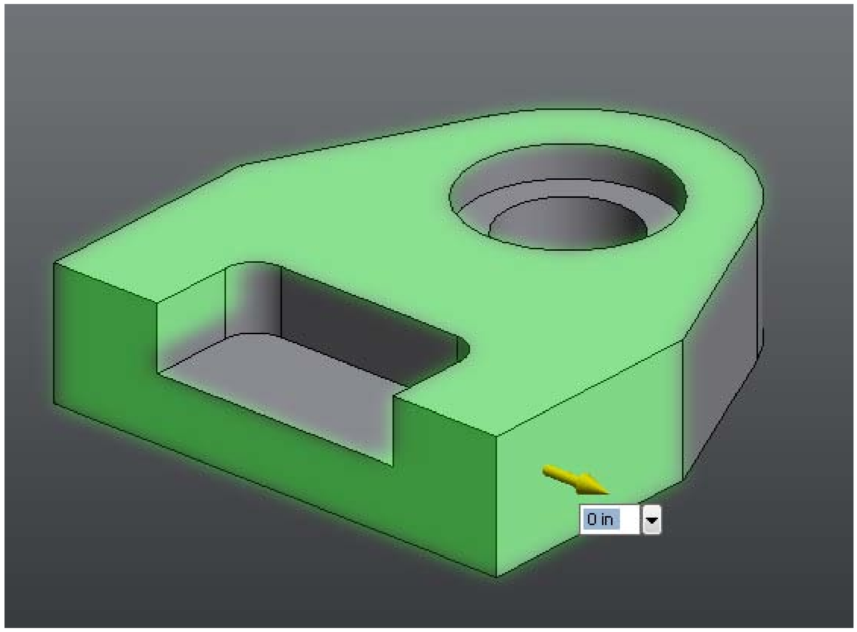


Resulting in this geometry:

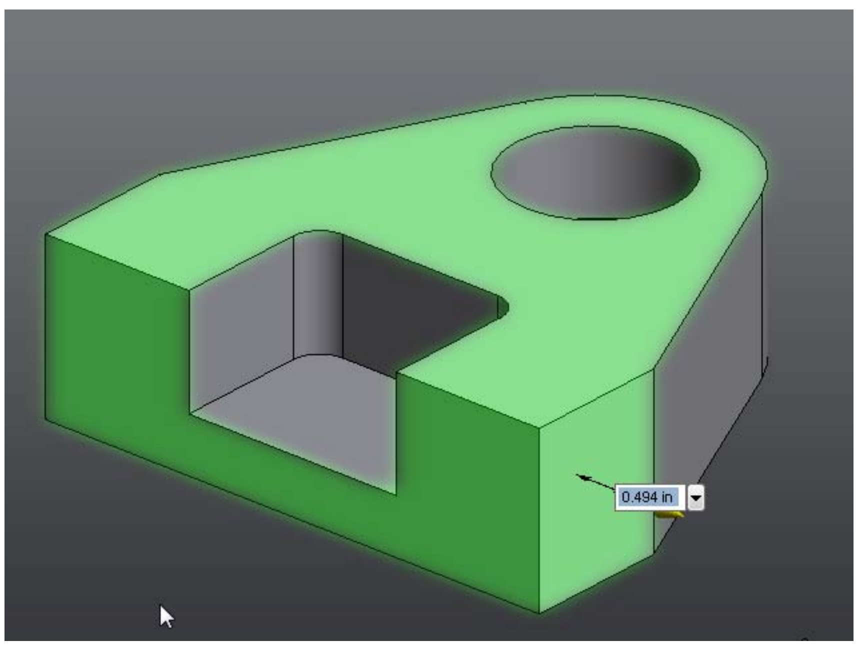


Multiple Faces

Press/Pull can be applied to multiple faces at once:

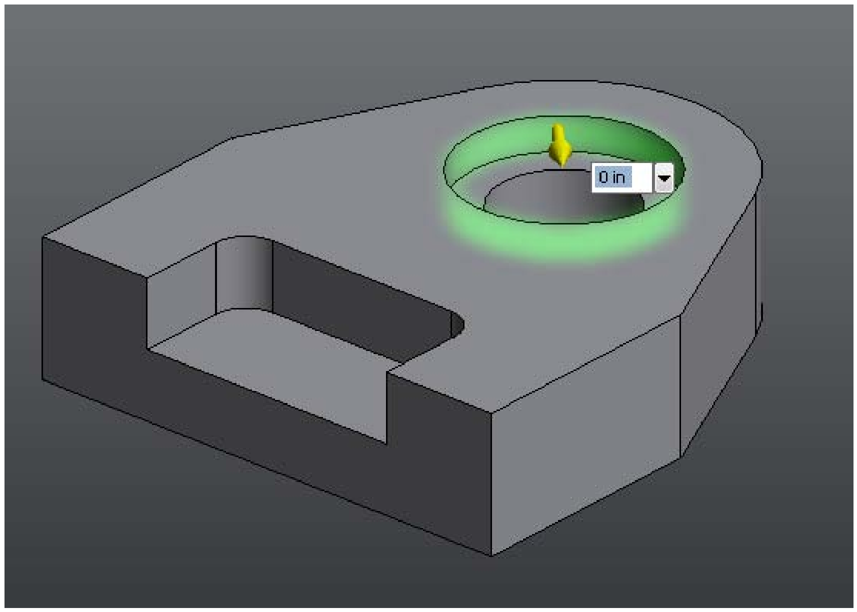


Resulting in:

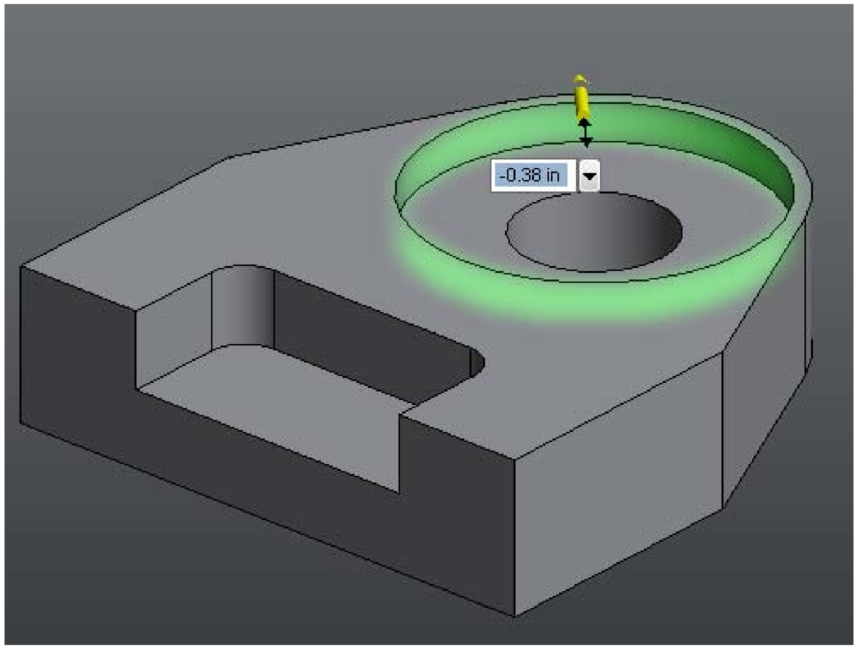


Cylindrical Faces

Press/Pull can be used to change the radius of cylindrical faces:

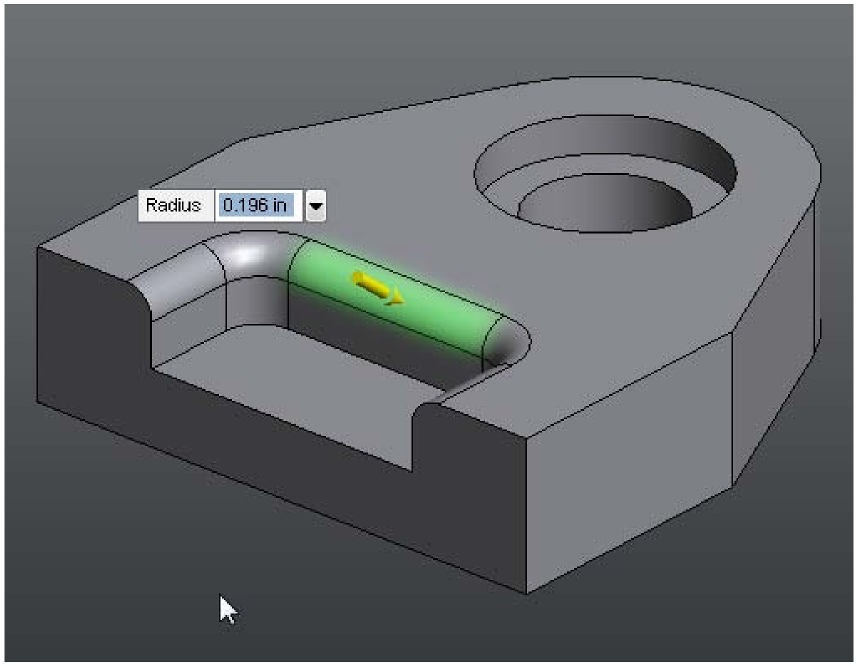


Resulting in:

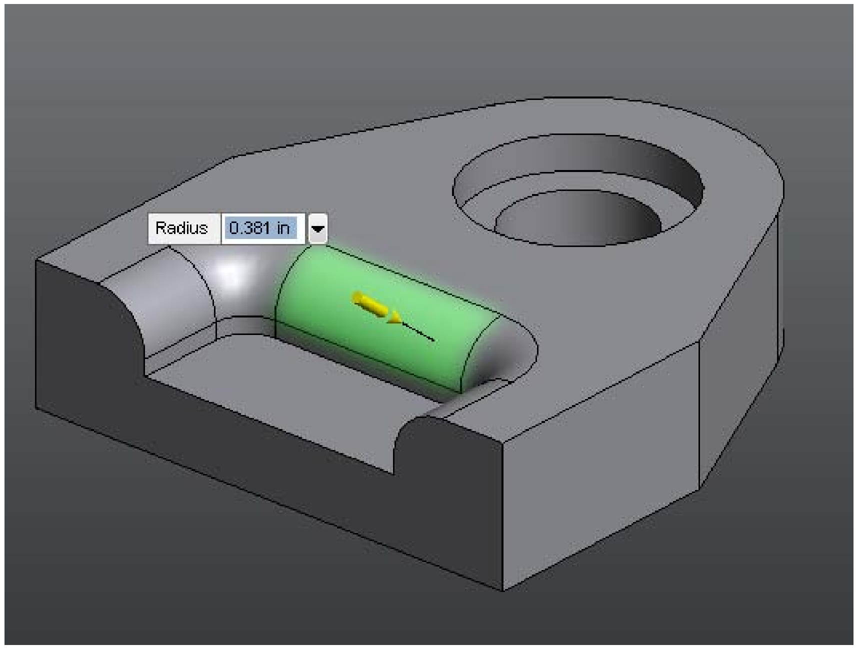


Fillet Faces

Press/Pull can be used to change the radius of Fillet geometry, even if that geometry was not created in Inventor Fusion, and without the user having to select all the faces involved:

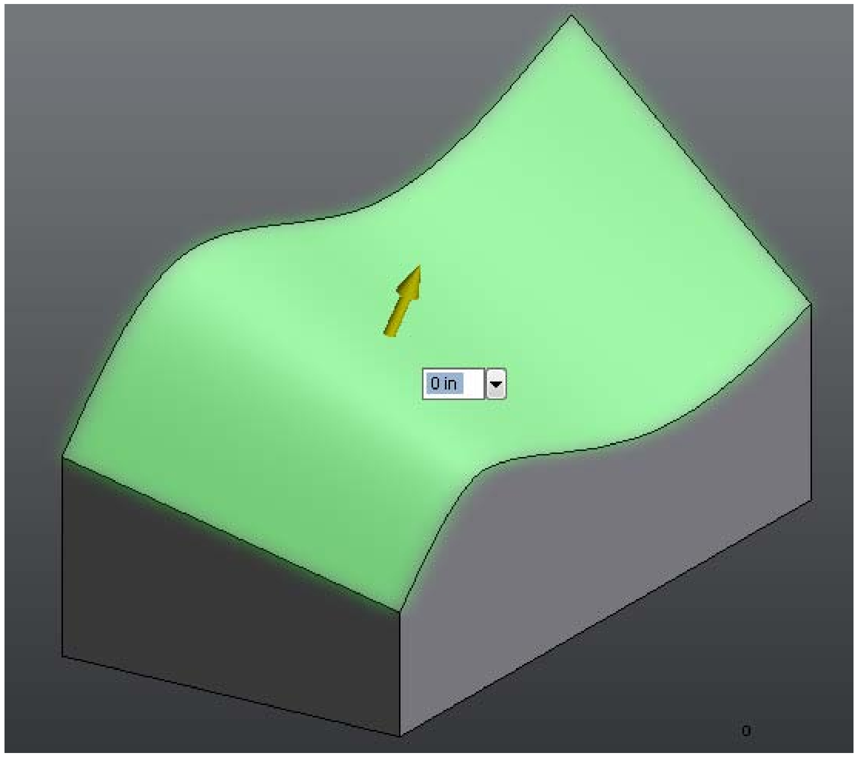


Resulting in:

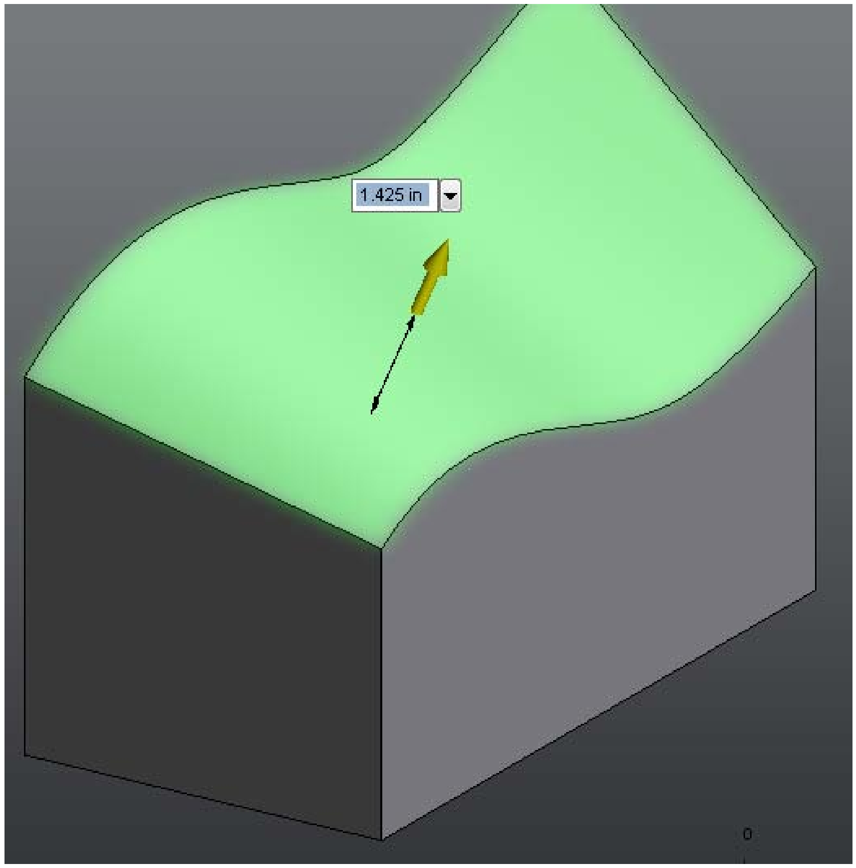


Complex Face Geometry

Press/Pull can be used to modify complex geometry as well as simple geometry:

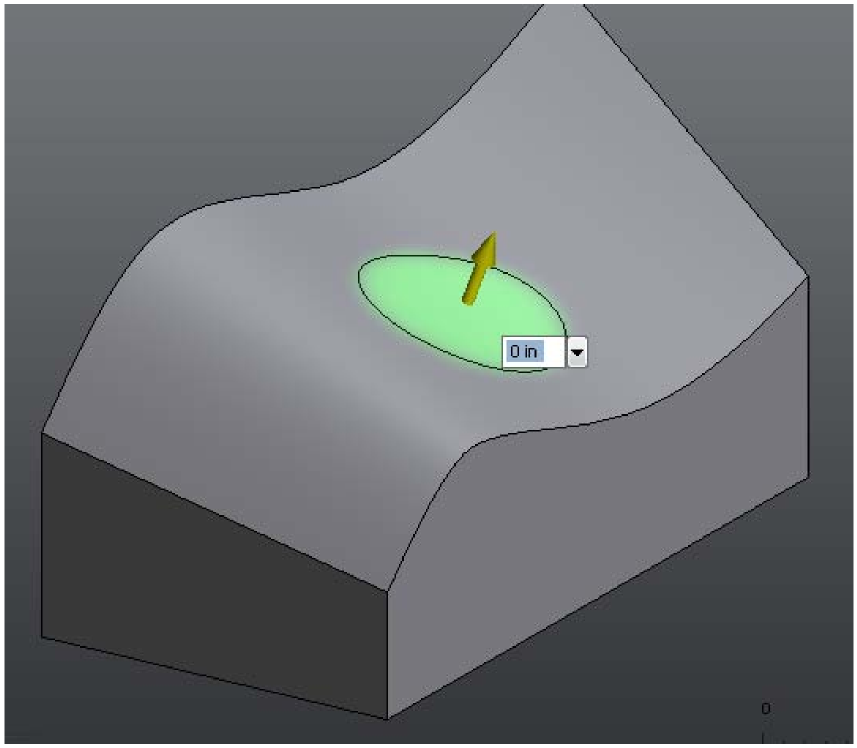


Resulting in:

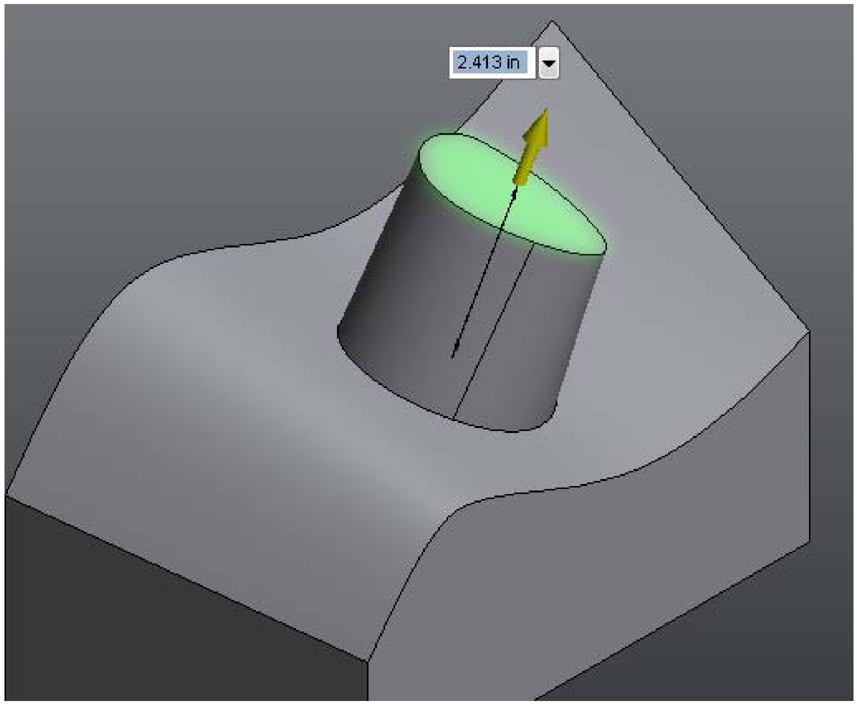


Used with Split Faces

Press/Pull can be used in conjunction with Split Faces to even add geometry to a design:



Resulting in:



Command Interaction

Press/Pull is extremely simple to use. invoke the command (using the Ribbon, Marking Menu, or Context Menu), select one or more faces (on one or more bodies), and drag using the left mouse button (either over the arrow drag manipulator, or over an area of the design with no geometry). You can drag, then release the mouse button, drag further, and repeat. When finished, start a new command (using any method), or click OK in the ribbon.

Other Things That Press/Pull Can Do

In addition to its use to modify bodies, Press/Pull can invoke two other commands: Fillet and Extrude. If, while in the Press/Pull command, you select a model edge, Press/Pull starts Fillet on that edge. If a sketch closed profile is selected, Press/Pull starts Extrude with that profile selected.

Move Command

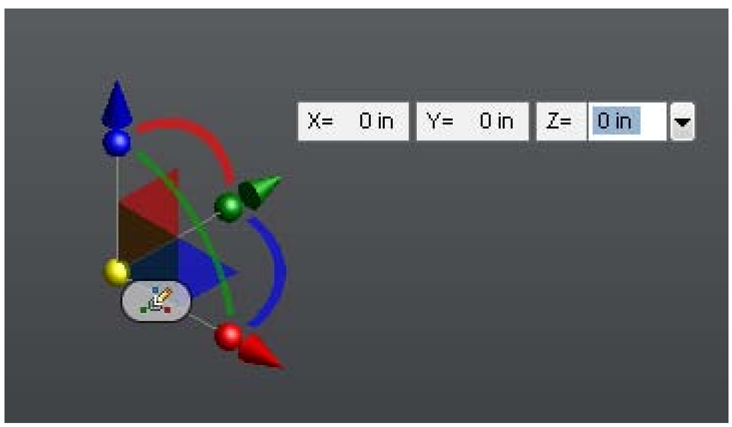
The Move command can be used to move a variety of objects in Inventor Fusion. These include:

- Component instances
- Work Geometry
- Model body faces

Use Move to modify the geometry of the design's body by moving one or more faces. See Position and Constrain Components on page 200 for information on using Move with components.

The Move command, when applied to model faces, is a very powerful editing command. You can make a wide range of geometric changes to a design, just using this one command.

The Move command uses the Fusion Triad manipulator:

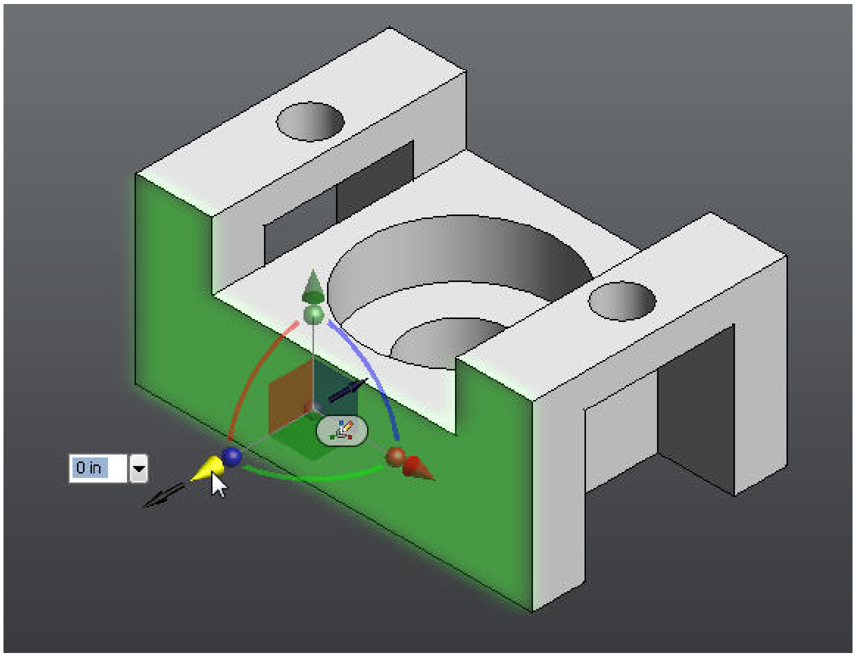


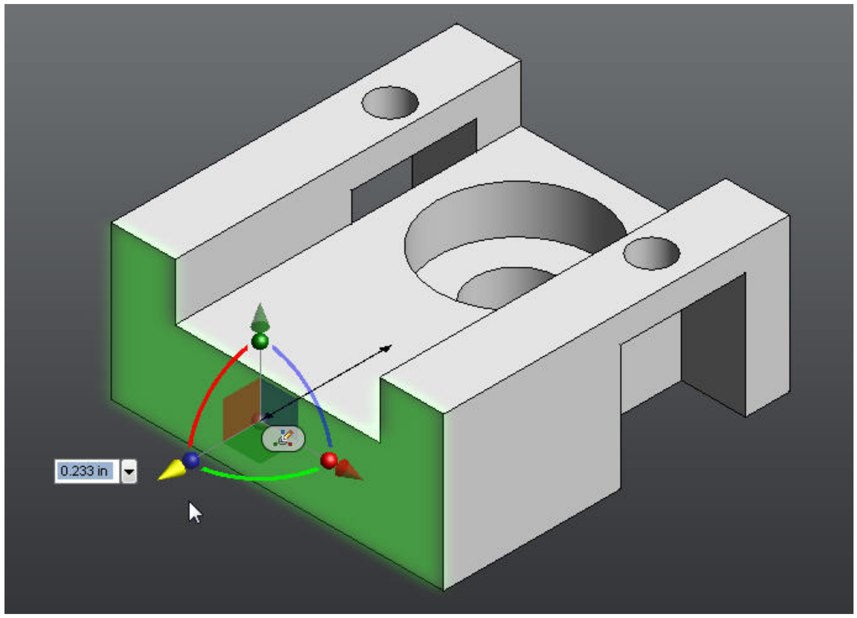
See this topic for more information on this manipulator: Triad on page 33.

Using linear manipulators in Move to translate faces

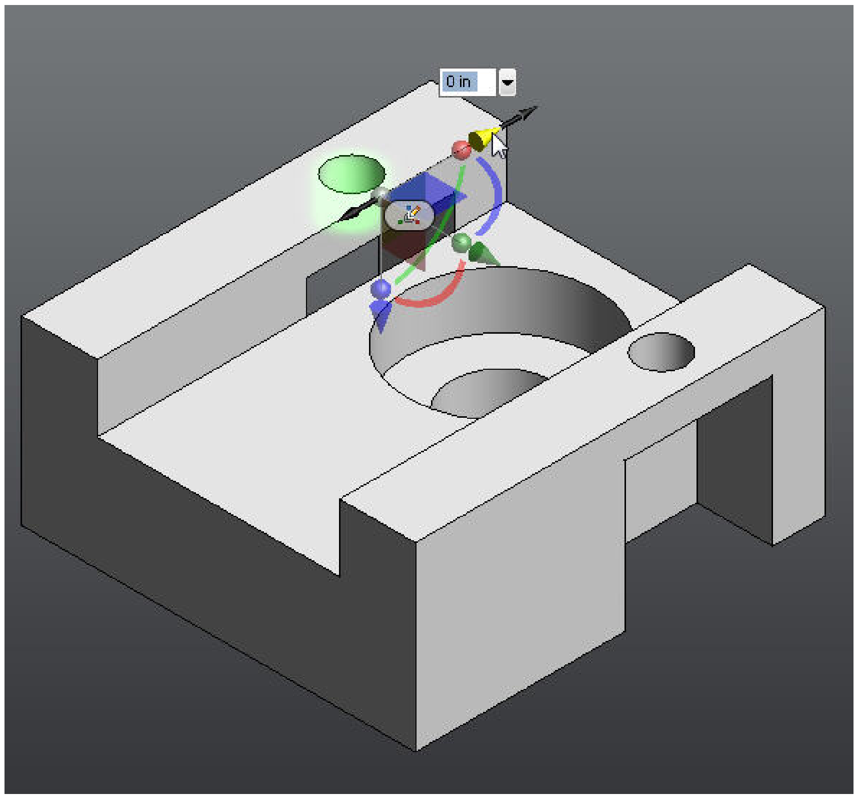
The arrow-shaped manipulators on the Move triad can be used to translate the selected model faces in the direction of the arrow. As with other Inventor Fusion manipulators, you can either drag over the manipulator itself, or over an area of the screen where there are no model graphics or user interface elements.

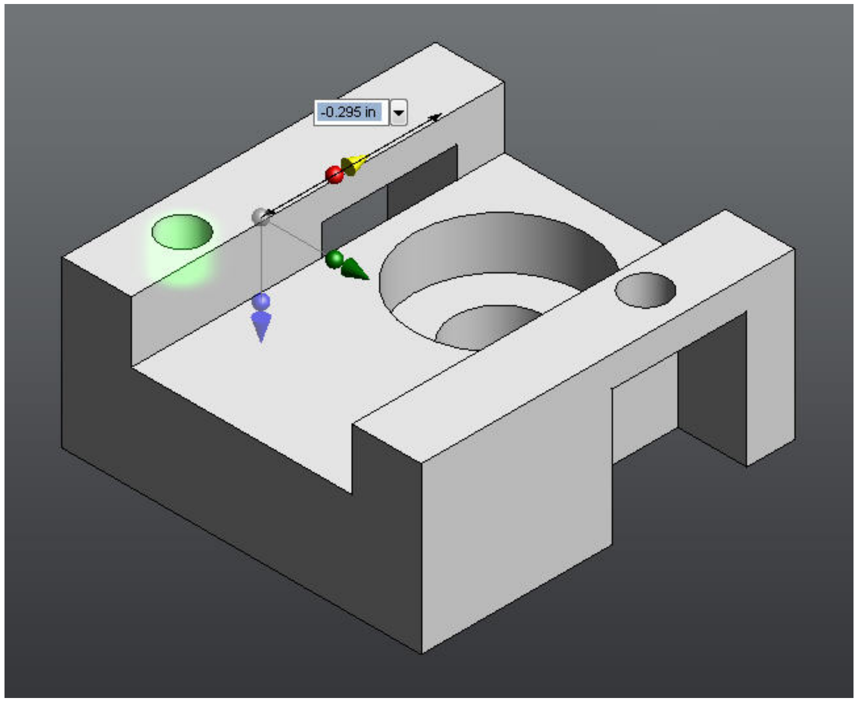
For instance, in this case, a single face is selected and moved along the yellow (yellow indicates selection) direction:



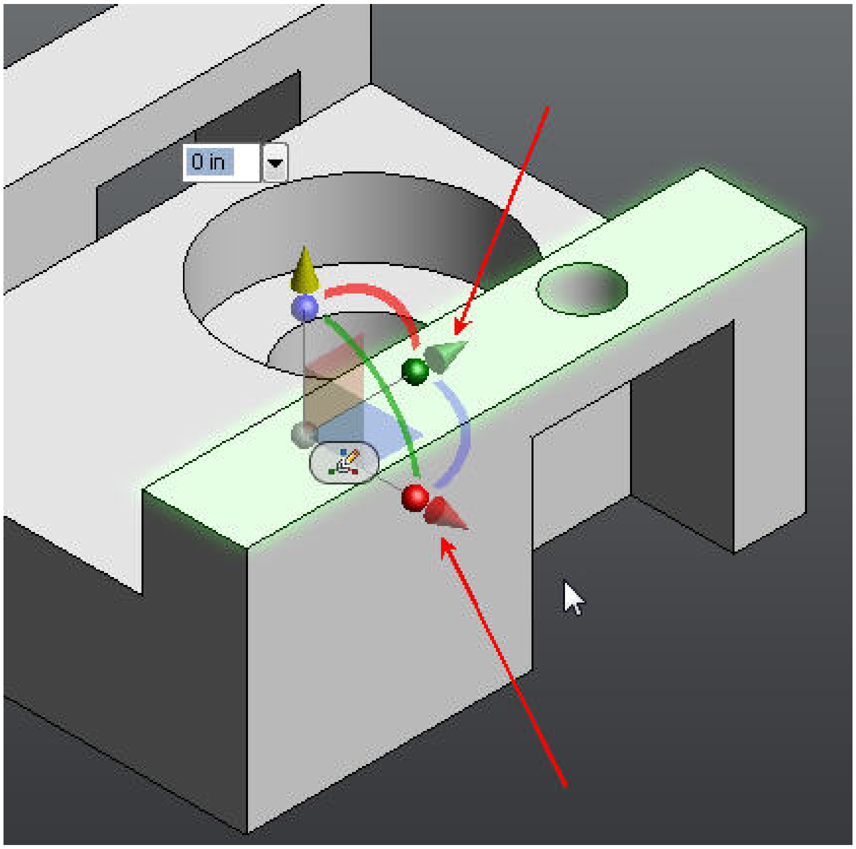

0.233 in

Move can be applied to geometry of any kind. In this example, one of the cylindrical holes is translated along one side of the model:





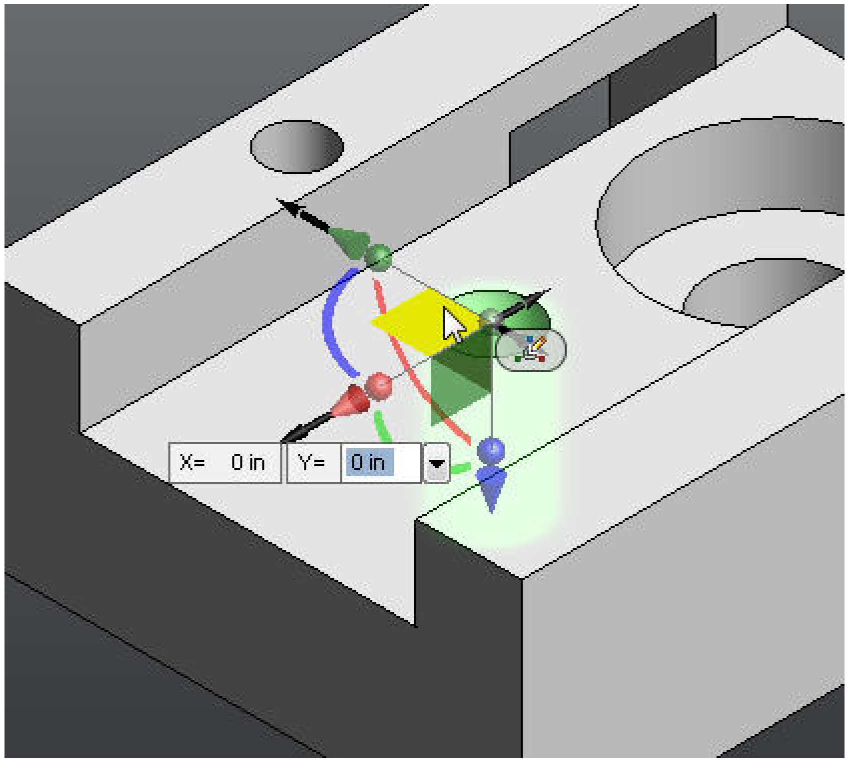
Note: Some faces are limited in their ability to move, by the geometry of the model. Note that in this case the red and green translation manipulators are disabled:



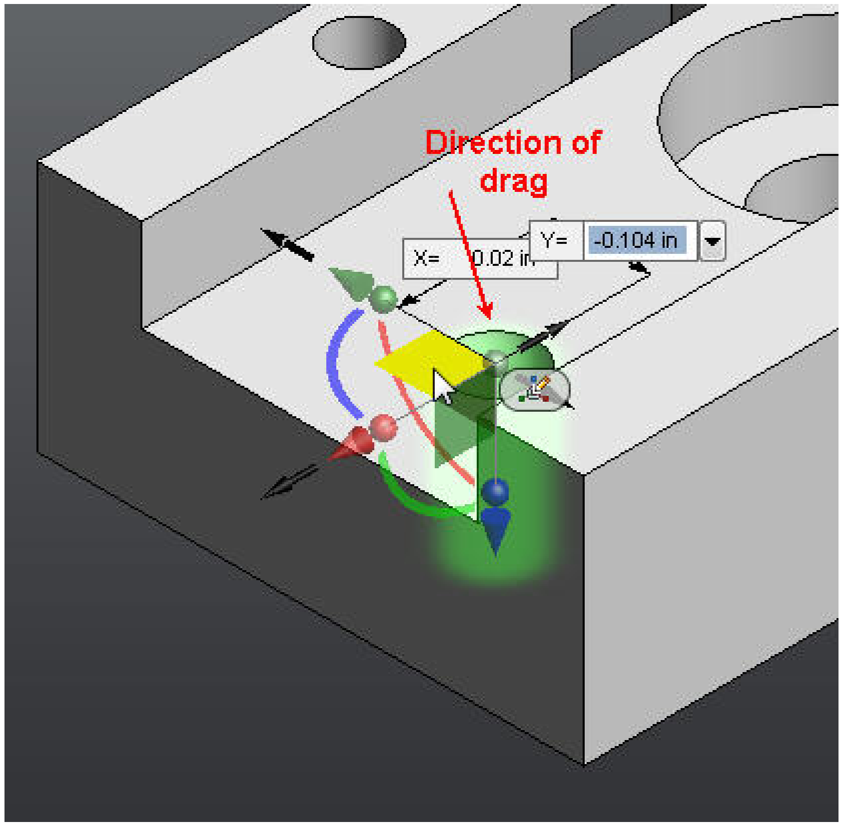
This planar face is not free to move in those directions. In some cases, the move command may not be able to detect these limitations, and manipulators are enabled, even though dragging in that direction does not result in a change to the model.

Using plane manipulators in Move to translate faces

The planar manipulators in the Move triad can be used to translate the selected faces in two directions at once.

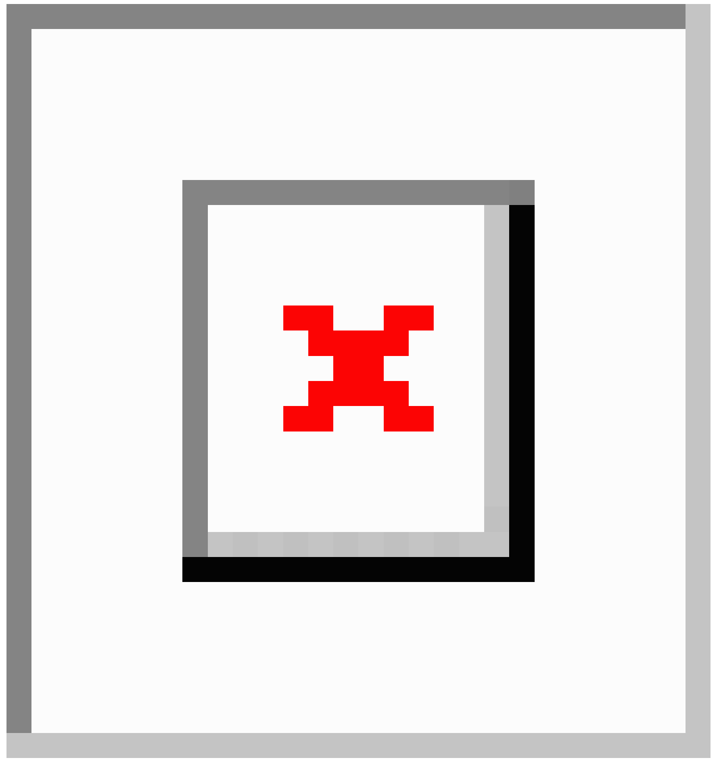


In this case, the highlighted plane manipulator can be used to move the selected cylindrical face in either the direction of the green arrow or the red arrow at the same time.

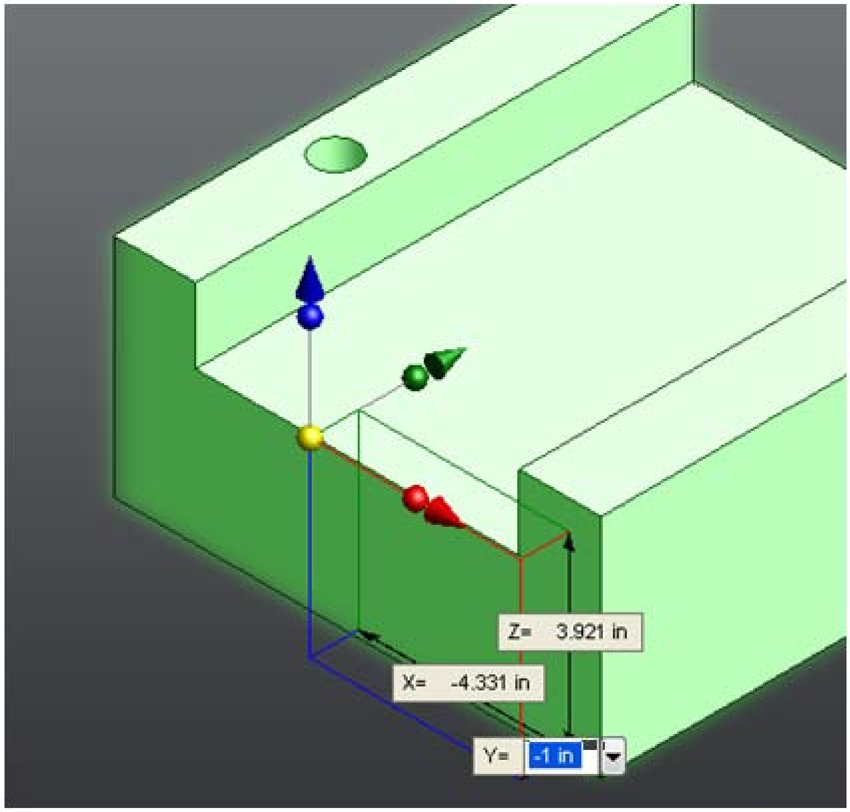


Using the origin manipulator in Move to translate faces/features/components

The origin manipulator in the Move triad can be used to translate the selected faces/features/components in all three directions at once.

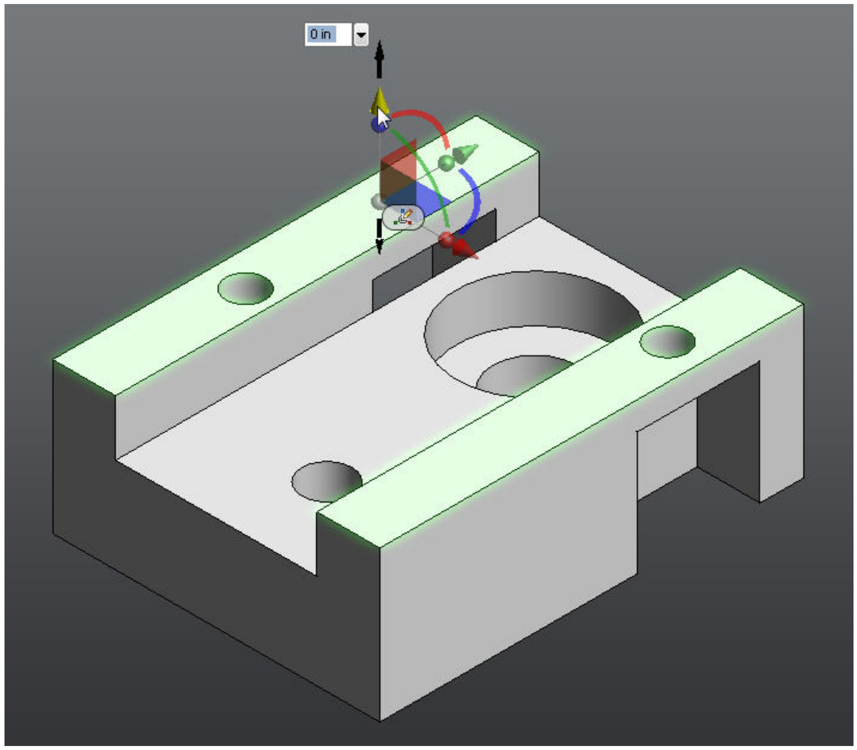


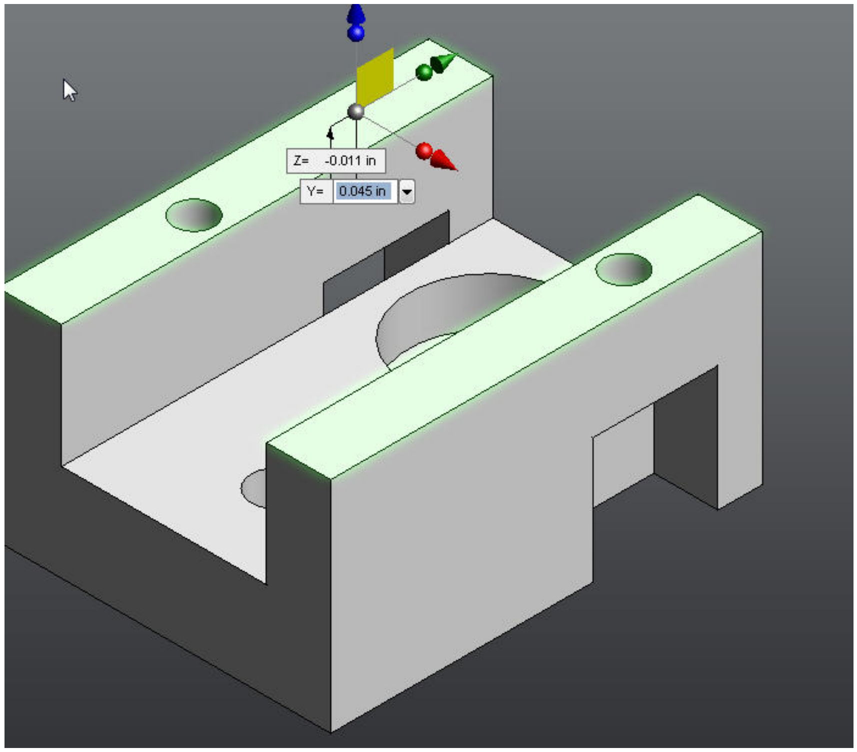
In this case, the highlighted origin manipulator can be used to move the selected component in all the three directions of the green arrow, red arrow and blue arrow at the same time:



Using Move to translate multiple faces at the same time

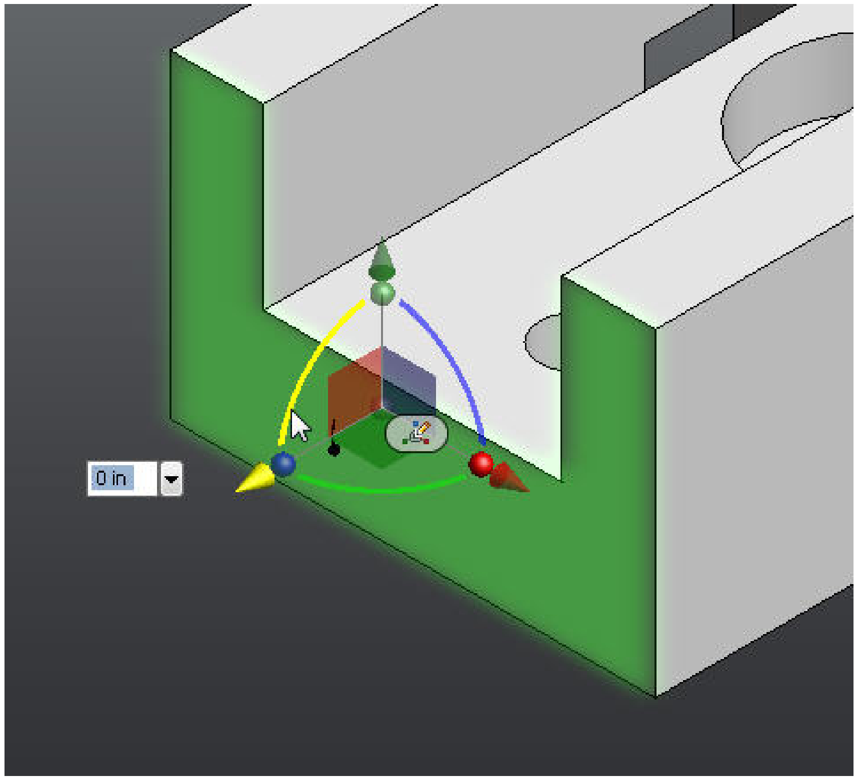
Move can be used to modify more than a single face at a time. In the following image, Move can be used to translate the two selected horizontal faces in the same operation:

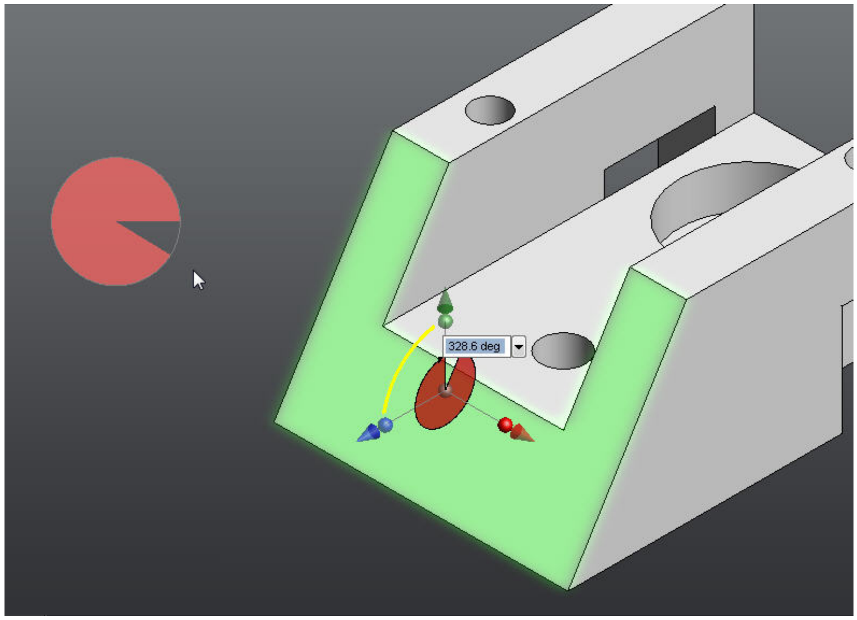




Using Move to rotate faces

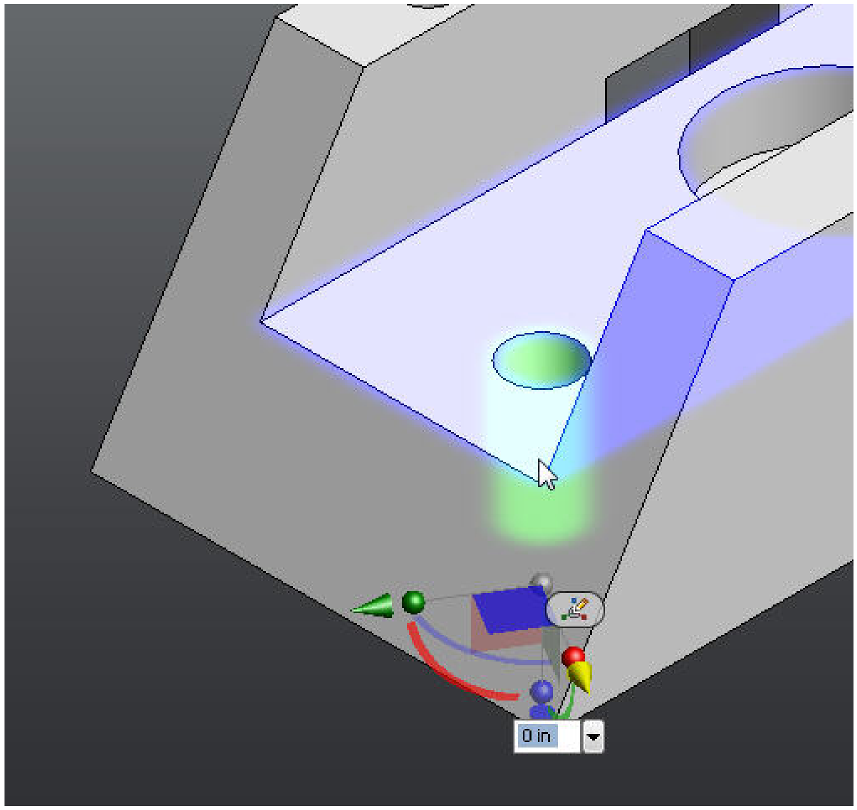
Besides translation the Move triad can be used to rotate faces as well. The rotate manipulators are the arcs on the triad. In this example, a rotate manipulator is used to tilt a model face:





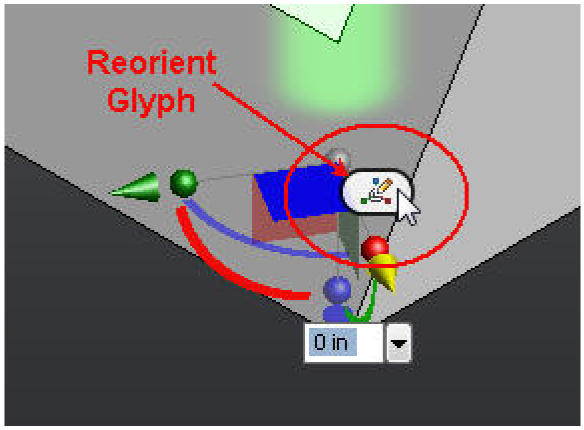
Reorienting the triad

When a face is selected in the move command, Inventor Fusion will place the Move triad at a default location and orientation. At times, this orientation and position does not match the appropriate transformation. For instance, in this case, a cylindrical face is selected.

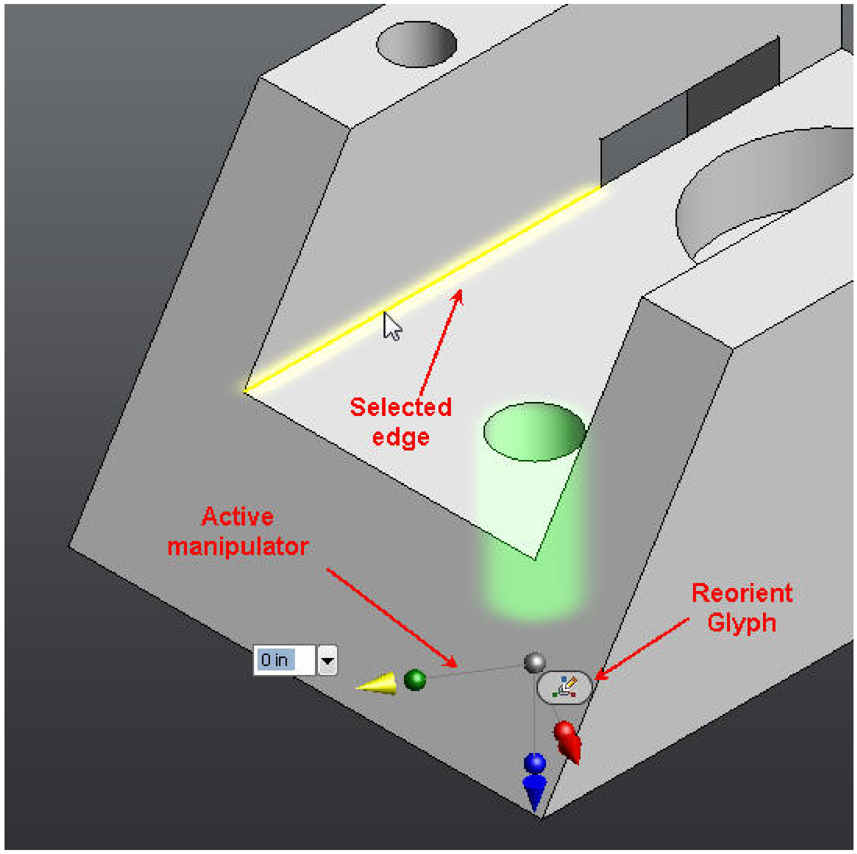


However, the default orientation is not ideal for moving the face along the rectilinear face that the hole is on. So, to change the default orientation to more fit your intended design modification.

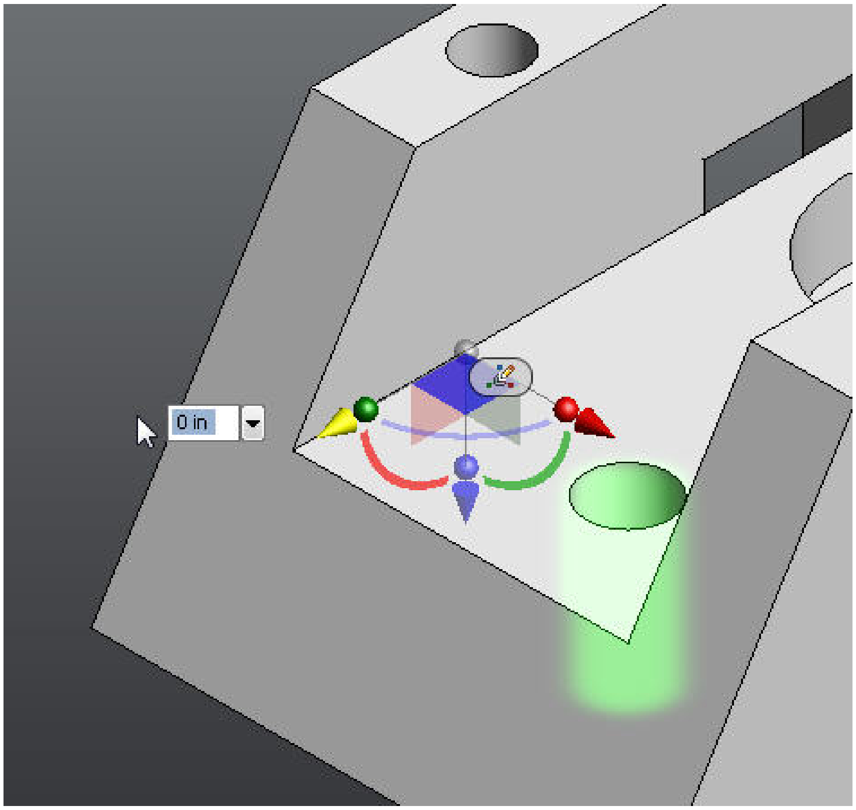
The Reorient glyph can be used to accomplish this:



If you click this glyph, the move command enters reorient mode. In this mode, geometry selections are used to reorient the triad, by aligning the active triad manipulator with the selected geometry. If a translate manipulator is active, a linear edge can be selected, and the triad is moved so that the active translate manipulator is aligned with the edge. For instance, if the highlighted edge is selected while in reorient mode:



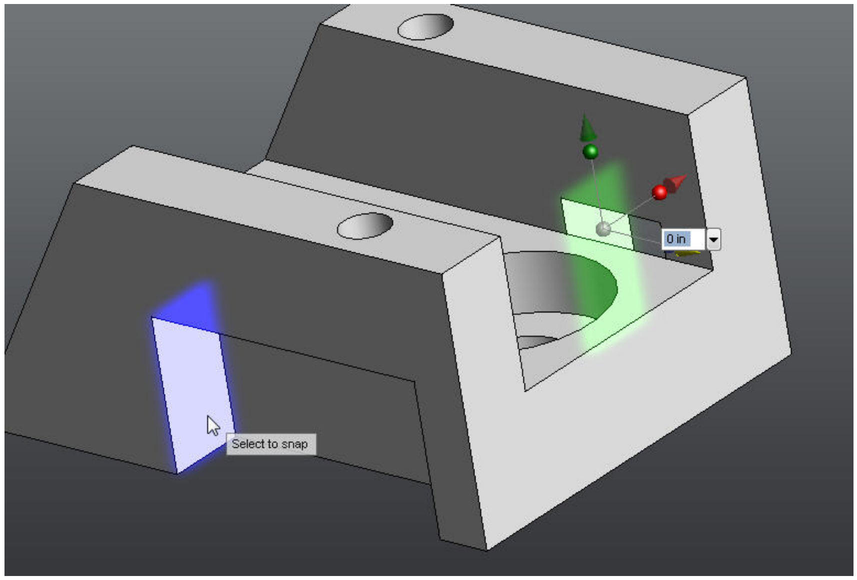
The triad is reoriented along the selected edge:



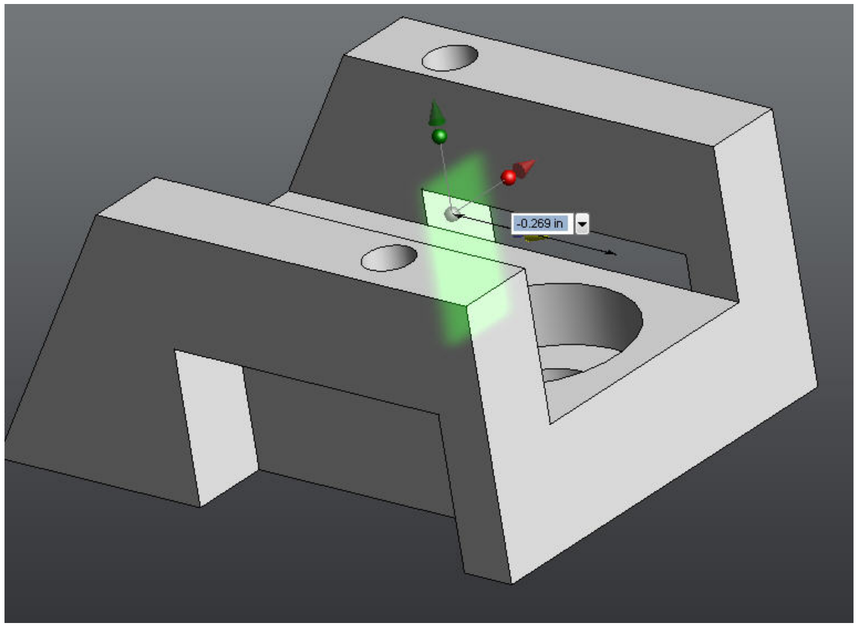
Translate manipulators can be reoriented along linear edges, work axes, cylinders, and planar faces. The triad can be reoriented about an active rotate manipulator similarly. Reorienting about a rotate manipulator is identical to reorienting about the corresponding translate manipulator. For instance, the blue rotate manipulator corresponds to the blue translate manipulator.

The use of snapping in the Move command

You can move one or more faces so they align precisely with other geometry in the design. For example, you want make the slots in the side of the part the same size. Select the face as shown in the following image. You want to move the selected green face so that it aligns with the corresponding blue face on the opposite side of the design.



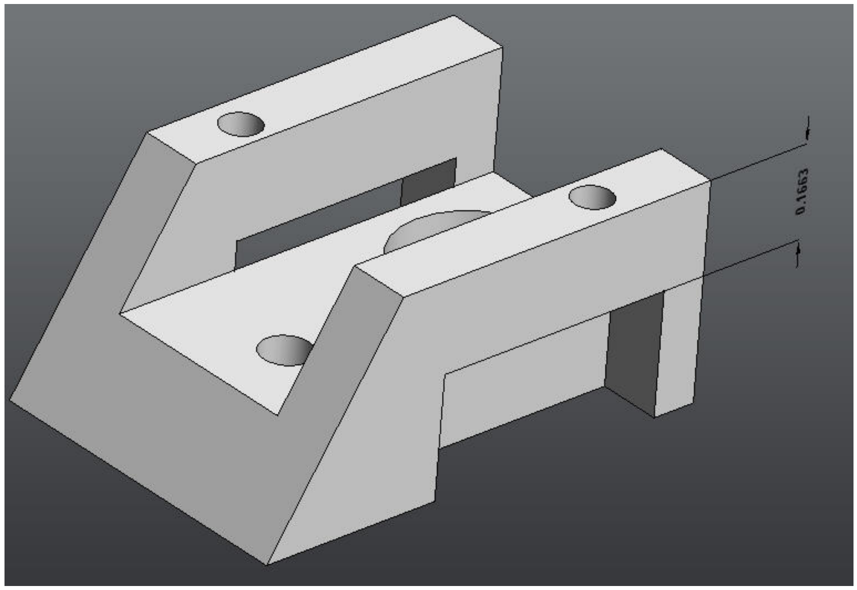
Use the Move command to achieve this with its snap to geometry feature. When a face is selected, if an applicable face is clicked that can be snapped to, move transforms the active faces so that they are aligned. Inventor Fusion indicates that snapping is possible with the prompt Select to snap when the cursor is over an available snap face. In the previous case, the selected green face is aligned with the indicated blue face.



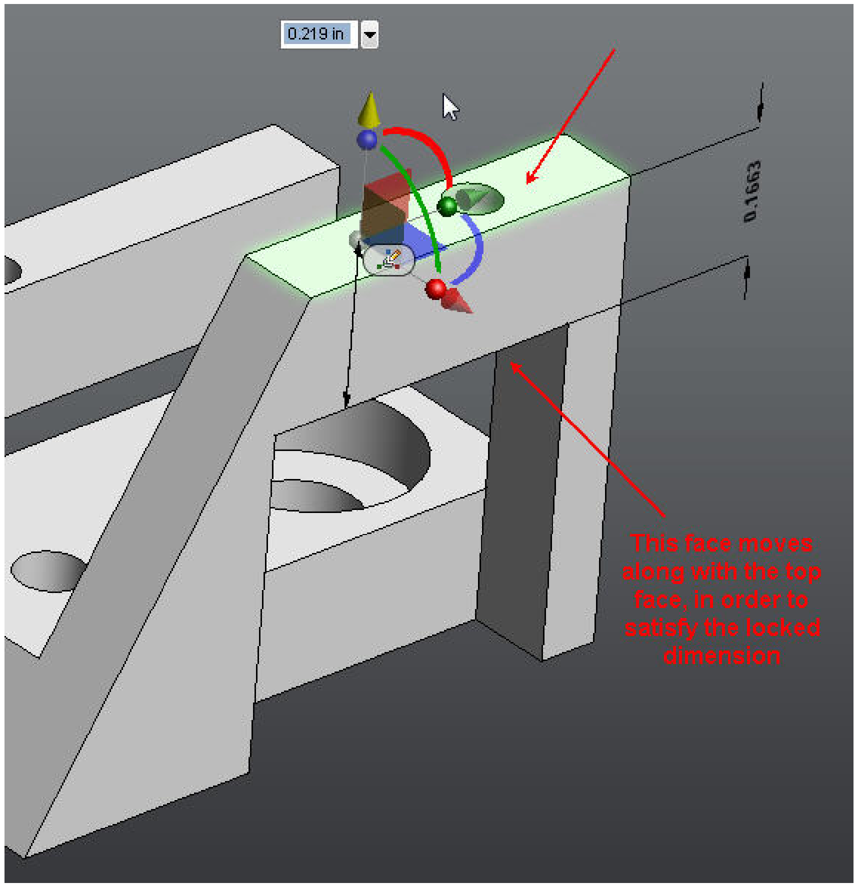
Move and body constraints

The Move command obeys any body constraints or locked dimensions during a move operation. For instance, a dimension has been created and locked between model geometry. Note that as one face moves, the other follows along.

In this case, the user has created a dimension and locked its value:



If the top face of the design is selected in the Move command, the dimension is honored, and so the bottom face moves along with the top face:



See Dimensions and Body Constraints on page 184 for more details.

Draft Command

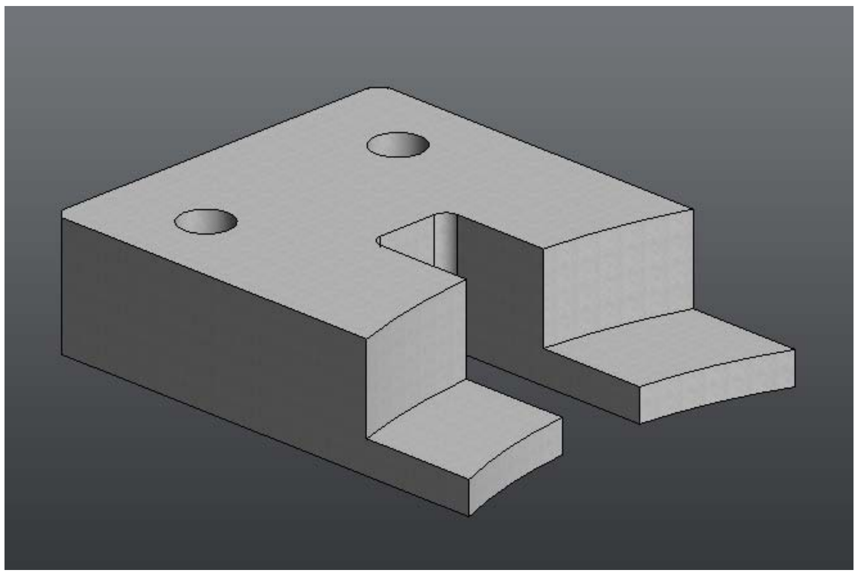
The Draft command in Inventor Fusion can be used to modify one or more component bodies by creating angled faces, with respect to a neutral plane.

A primary goal of this command is to be used for creating molded parts - parts that are manufactured using an injection molding or metal casting process. Such parts usually have faces that are slightly angled, to make removal of the part from the mold easier. In these cases, the draft is usually applied to a selection of several faces, most often all of the side faces of the design.

However, this command can also be used as a general modeling command for creating individual angled faces.

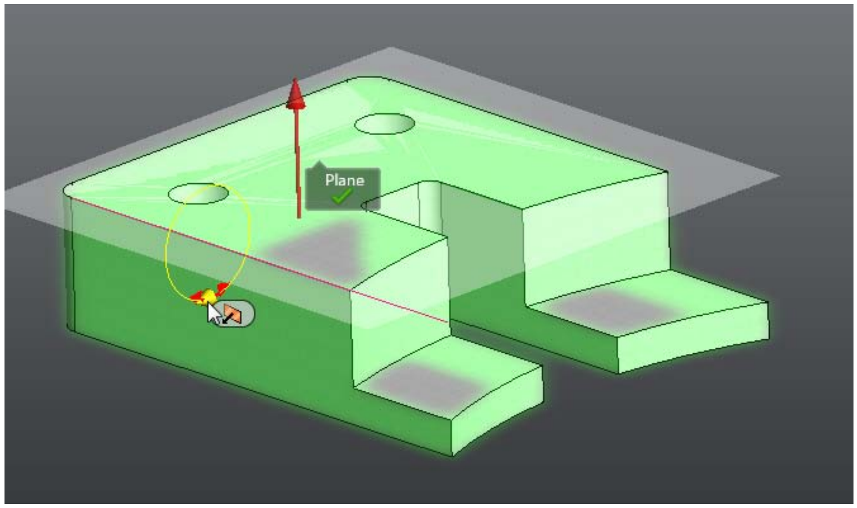
Using Draft to model a simple molded part

In this example, draft can be used to add draft to a part so that it can be easily removed from a mold. The following image is the original part:

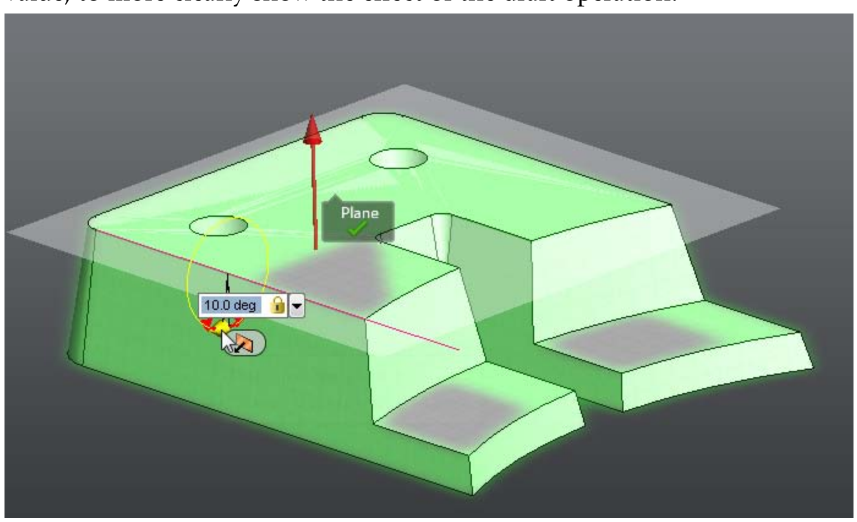


Invoke the Draft command. The first input this command requests is a neutral plane. This selection serves two purposes: It specifies the plane around which faces are drafted, and it also specifies the pull direction. That is, the direction in which the mold is removed from the part, after the molding process is complete. The second input is the faces to be drafted.

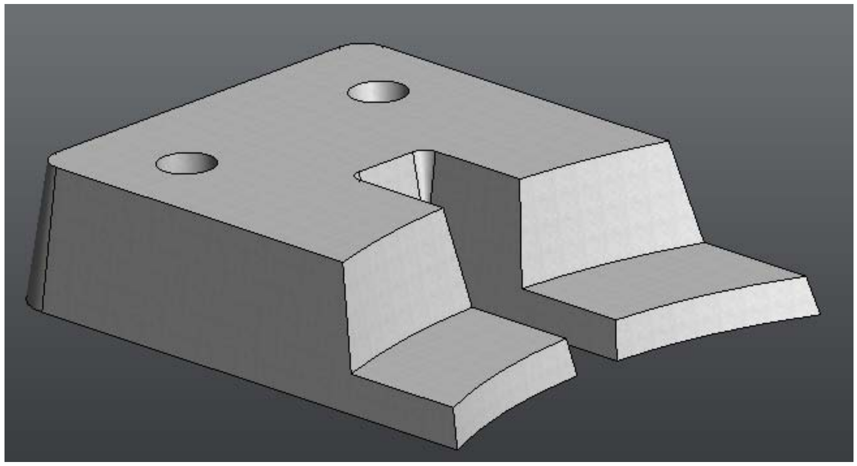
In this example, the user has selected all of the vertical faces as draft faces:



Next, you would drag, or enter a precise value into the draft angle entry box. In the following image, the draft angle has been set to an unusually high value, to more clearly show the effect of the draft operation.

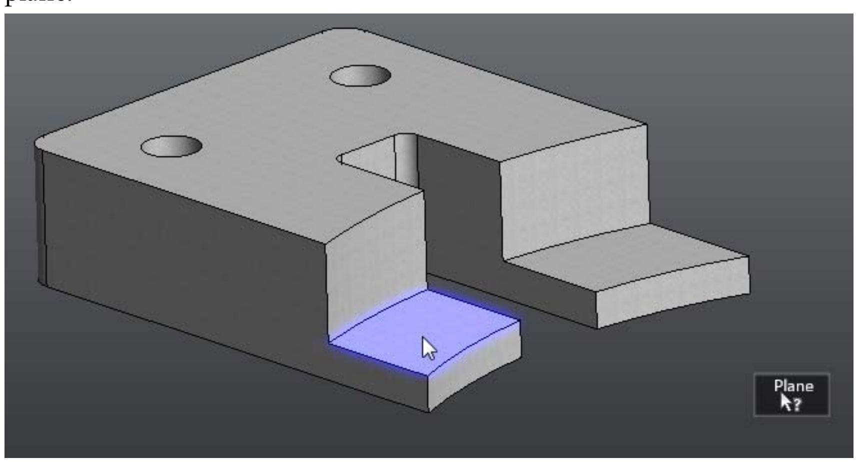


Finally, you OK the command to apply the draft to the model:

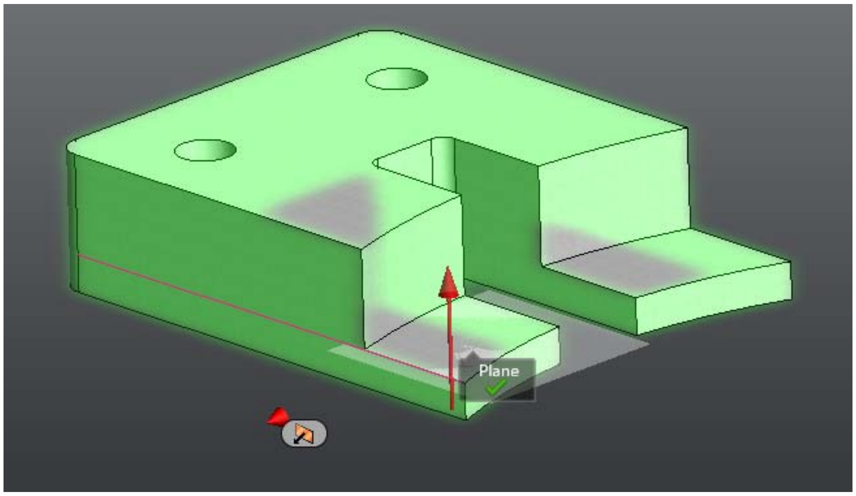


Using Draft to model a part that is to be manufactured with a split mold

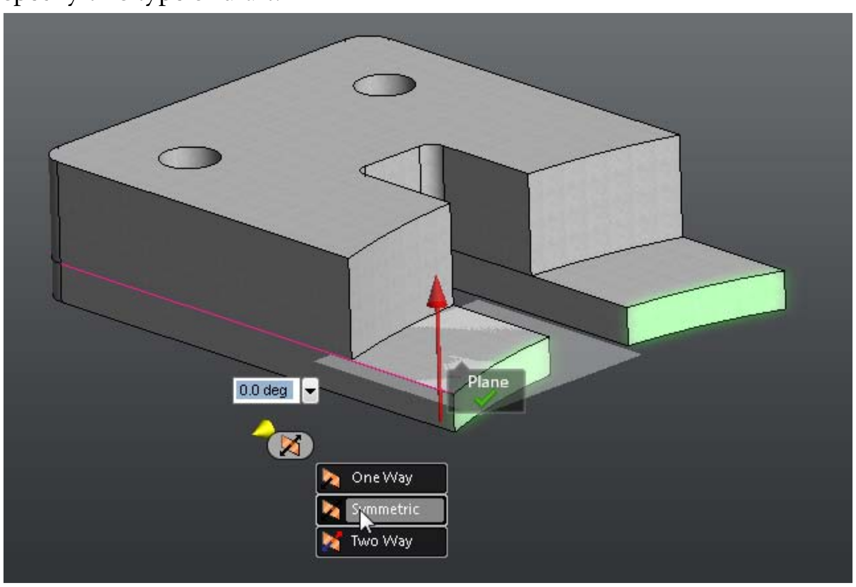
Sometimes, mold designers use a two-piece mold, which splits in the middle, to remove the part after cooling. Using the same example as previously, this time the user picks the plane shown in the following image as the neutral plane:



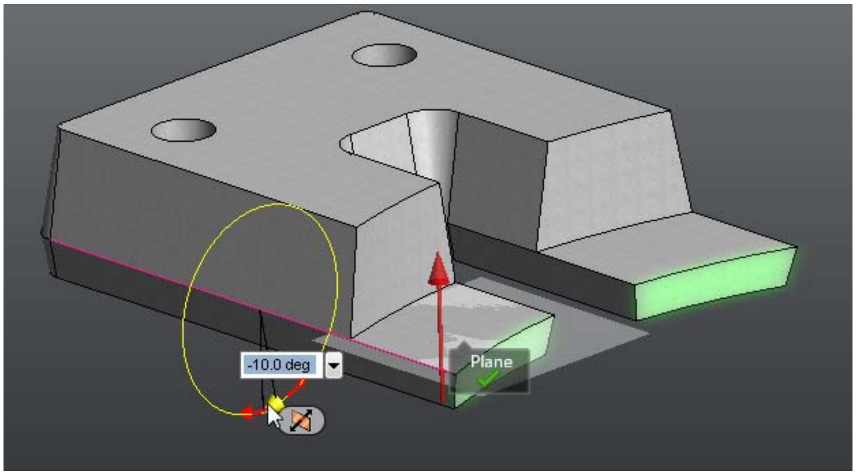
This selection indicates that the mold will be split along this plane. The vertical planes are selected as faces to draft:



Next, you specify that this is a symmetric draft. Use the Draft Type glyph to specify this type of draft:



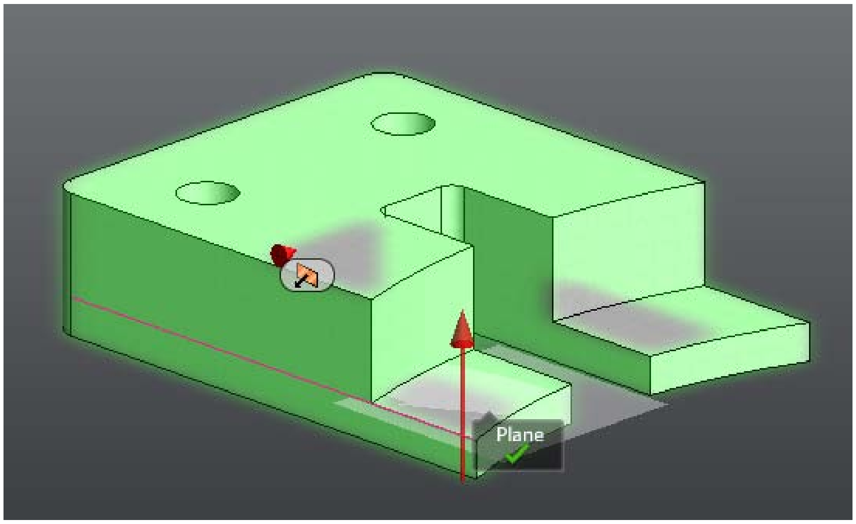
Drag or enter a draft angle. The result shows that the model has been split along the neutral plane, and the draft applied symmetrically to both sets of faces:



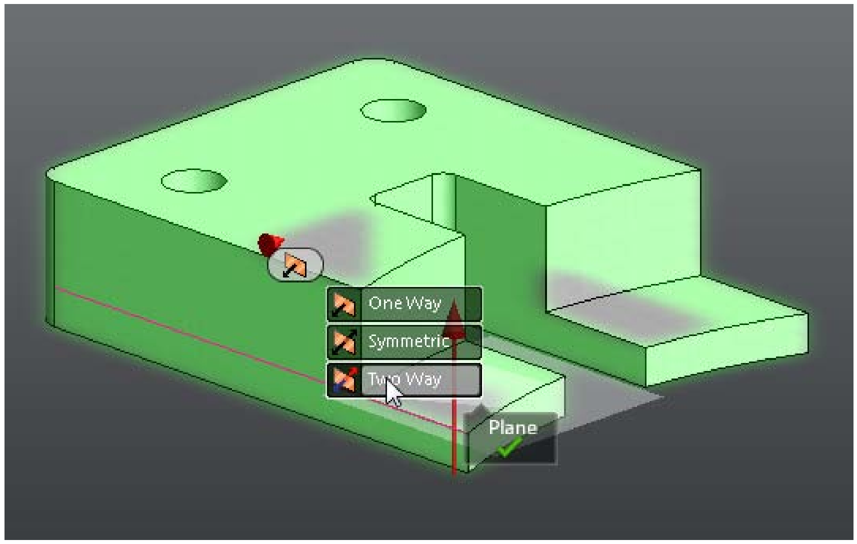
Using multiple draft angles

Sometimes you may want to perform a draft operation similar to the previous topic, but apply a different angle to faces above the neutral plane than to those below. You would use the two-way draft type to achieve this.

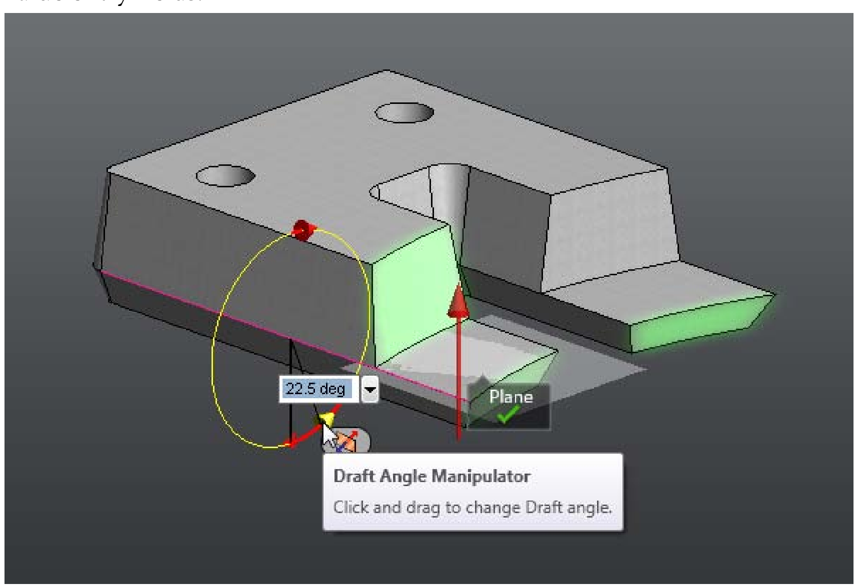
The following example is identical to the previous example. Assuming that you have already picked the neutral plane and faces to draft, the command looks like:



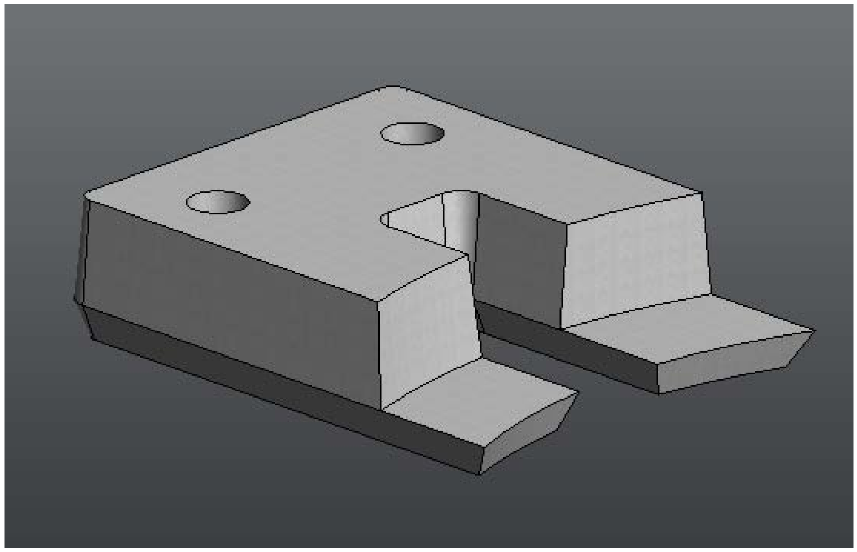
This time, in the Draft Type glyph menu, select two way.



This results in two separate angle manipulators and two corresponding angle value entry fields:



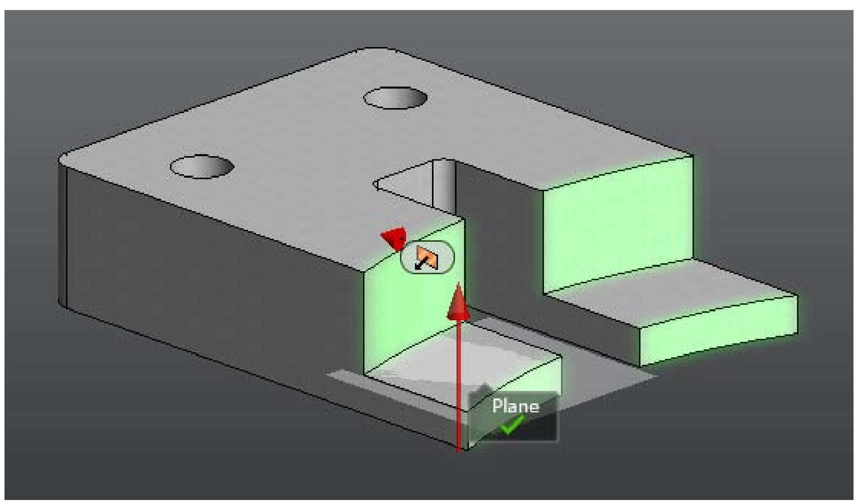
The final result shows two different angles applied to each set of faces:



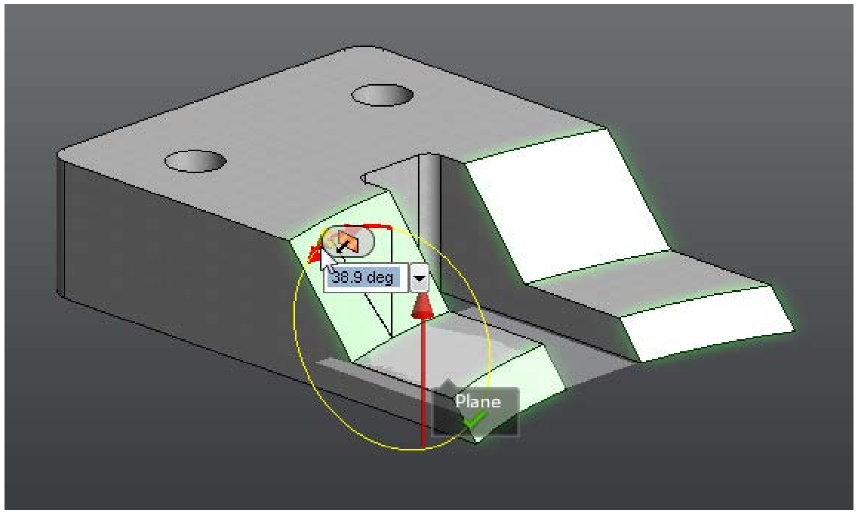
Using Draft as a general body modification command

In addition to using Draft to add an angle to all vertical faces, for the purposes of easier mold extraction, Draft can be used as a general command to edit a design.

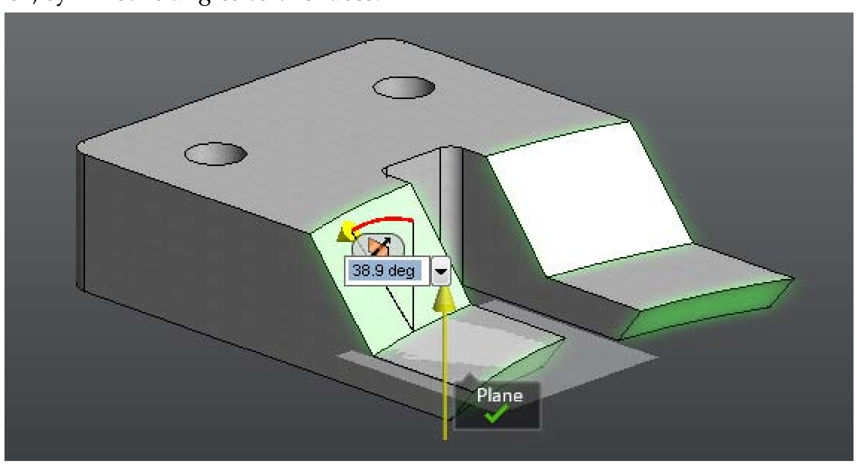
Using the same model as in previous examples, select a neutral plane and four faces to draft:



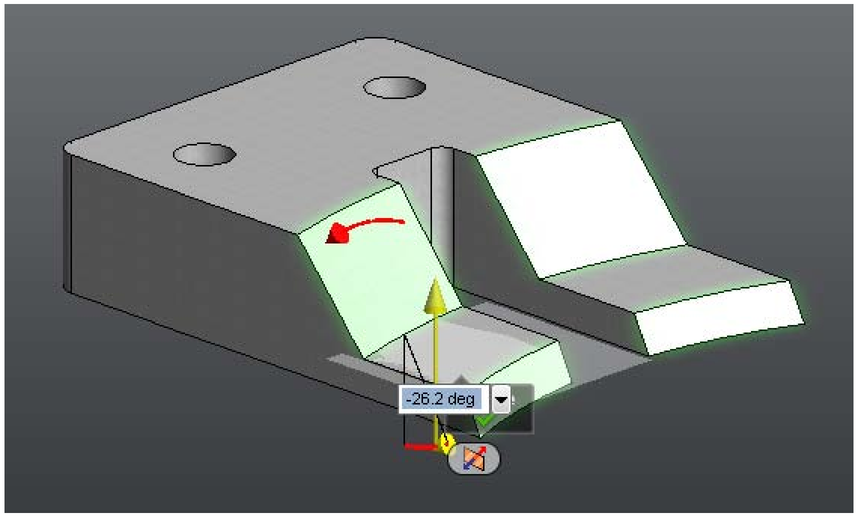
You can use Draft to apply a single angle to the faces:



Or, symmetric angles to the faces:



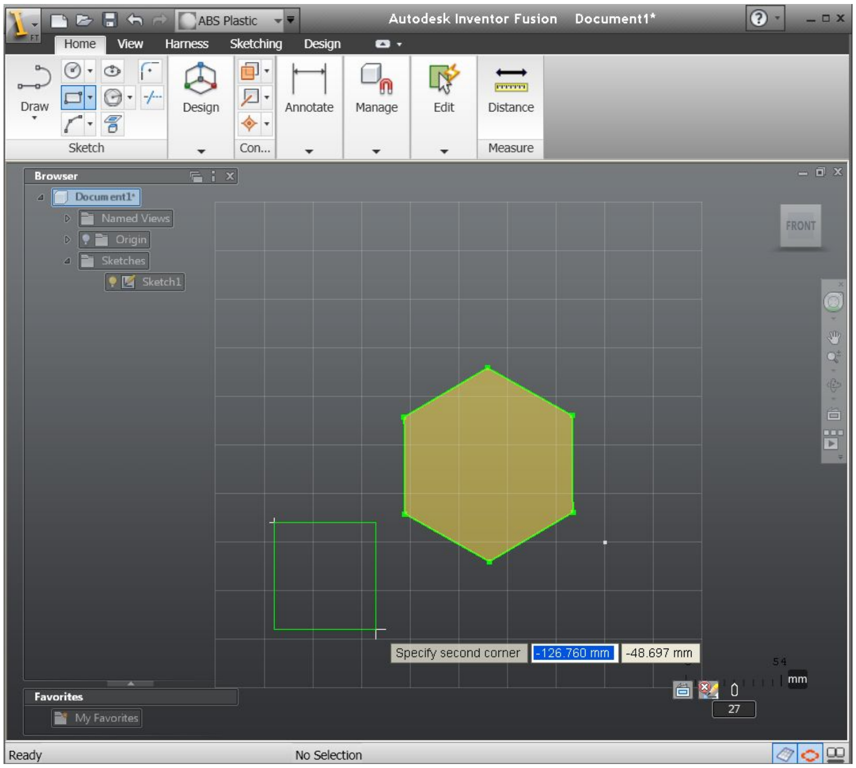
Or, two different angles to the faces:



Sketch

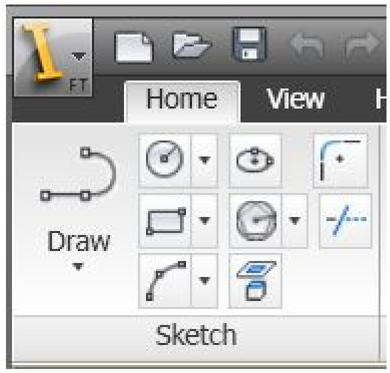
Introduction

If you are creating a new part in Inventor Fusion, the first operation you perform is Sketching. This section explains Sketch creation, Sketch editing and other Sketch features in detail to help you learn the Sketching environment.



Starting a Sketch

To start a Sketch, you can pick any Sketch command. All the Sketch commands are available in the Home Tab:



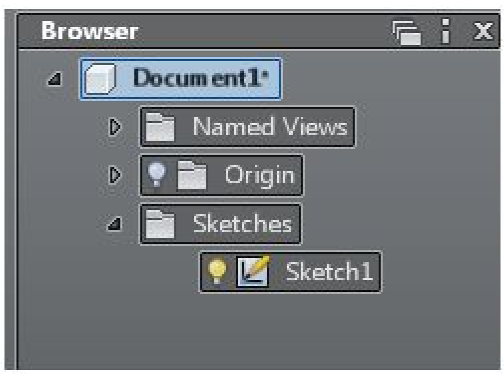
Sketch commands are also available from the context menu if there is no command currently active:

Once a Sketch command is selected, you are prompted to define a Sketch Plane on page 138. At this point, you can pick any work plane or planar face to sketch on. Once the Sketch Plane is defined, you see the Sketch Grid on page 138 by default.

The Sketch Plane

The Sketch Plane is a plane on which you can draw Sketch entities. A Sketch Plane is always created as a child of the currently active component. A component can have many Sketch Planes, but only one Sketch Plane in a document can be active at a time.

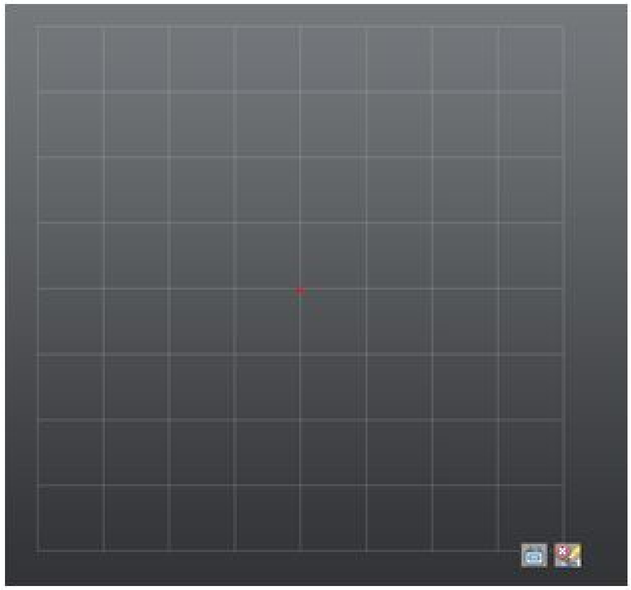
A Sketch plane can be defined using a planar face, a Workplane, or an existing Sketch Profile on page 171. When a Sketch Plane is created, a node is added to the browser:



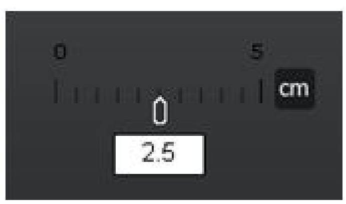
You can click the light bulb icon to make the Sketch entities on the plane invisible/visible. You can double-click the pencil icon to activate that Sketch Plane, so you can add/remove its Sketch entities. If you activate a Sketch, any Sketch that is already active is deactivated.

The Sketch Grid

When a Sketch Plane on page 138is activated, a Sketch Grid is drawn that encompasses all the Sketch entities on the plane. The camera is updated automatically to look at the Sketch Plane. If there are no entities on the plane, the Grid assumes a default size.



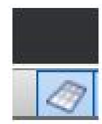
The Sketch Grid has two icons associated with it. The first is a Look-At icon that you can use to change the camera to look at the Sketch Plane if you perform any camera operations that would change the planes orientation. The second icon is a Stop Sketch icon that deactivates the Sketch and returns you to the select mode.

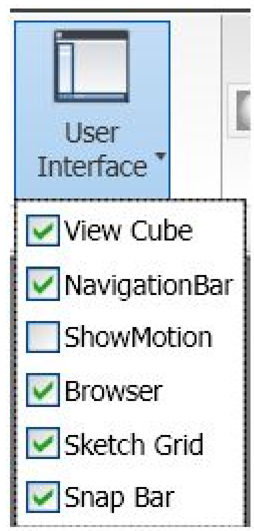


The Grid aids in creating accurate Sketches by precisely snapping to points defined by its spacing. A snap point is indicated by a red rectangle.

Snap points are available throughout the Sketch Plane – they are not restricted to the visible Grid.

You can hide the Grid in two ways:

1 Click the Grid icon at the bottom right of your commandbar. 

2 Go to the View Tab, and under the “User Interface” drop-down box, 

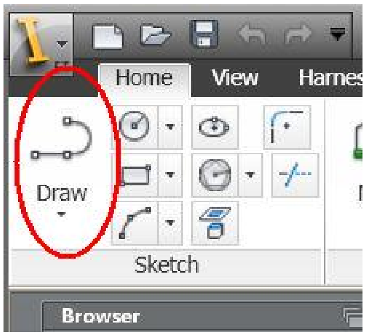
uncheck the Sketch grid check box.

The Sketch Grids spacing is controlled by the snap bar which appears to the bottom-right of the canvas. If the snap bar is set to 0, the Grid assumes a default spacing value.

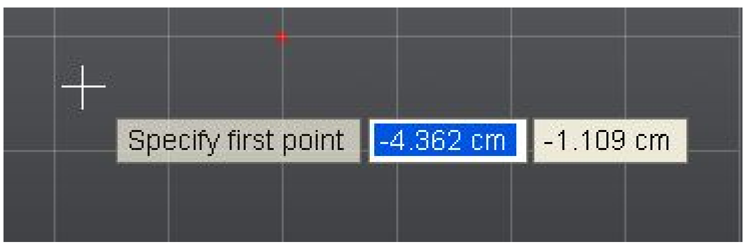
When a Sketch Plane on page 138 is activated, a Sketch Grid will be drawn that encompasses all the Sketch entities on the plane. The camera is updated automatically to look at the Sketch Plane. If there are no entities on the plane, the Grid assumes a default size.

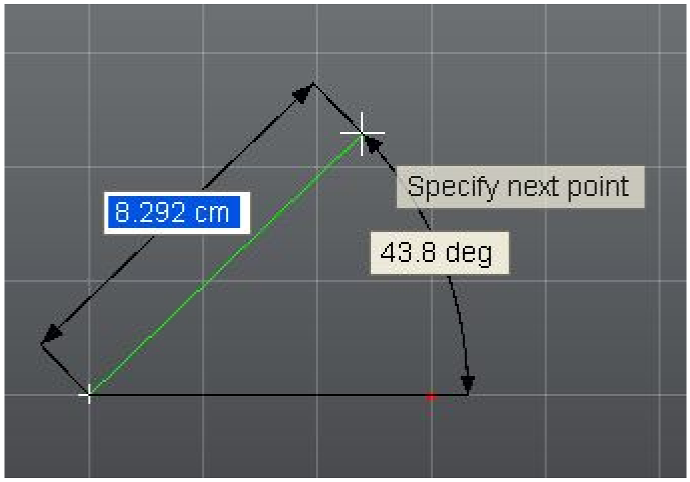
Line/Arc Segment Creation

Click the Line/Arc command button on the ribbon to activate the Sketch Line/Arc command. If there is no Sketch Plane on page 138 currently active, you are prompted to select one.



When you activate the Line/Arc command, you can define points by clicking the mouse, or by typing the values into the Heads-Up Display (HUD) text boxes. When relevant, you get HUD text boxes where you input line lengths and angles from the vertical/horizontal. This helps you to create more precise geometry. You can press the TAB key to navigate from one HUD text box to another.

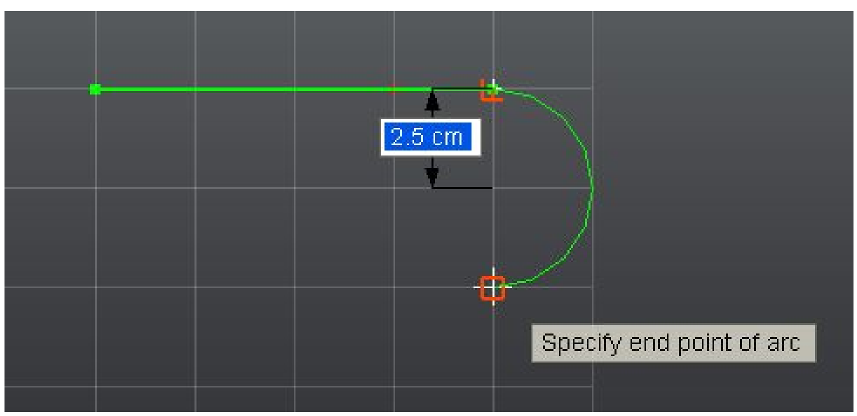




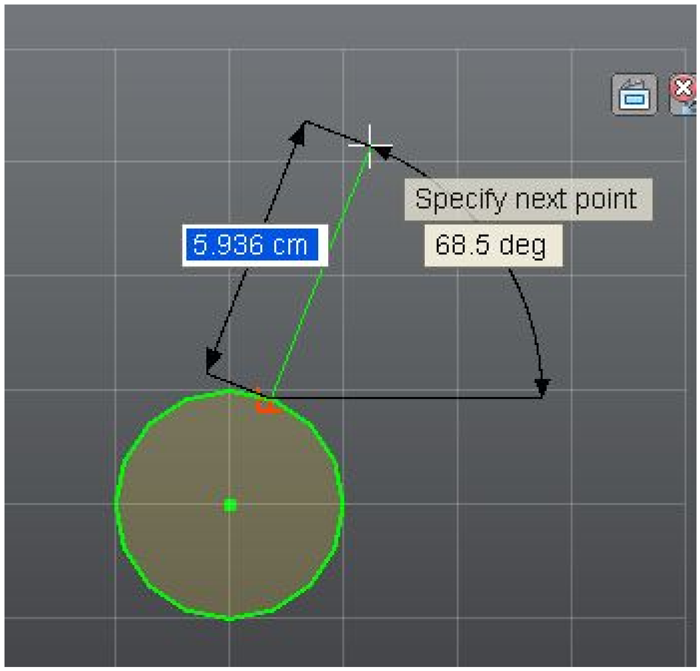
If there are existing sketch geometries on your Sketch Plane, you also receive feedback on points that lie on other sketch entities, midpoints, and Grid snap points. See Sketch Inferencing on page 162 to learn more.

Although you cannot define explicit constraints while creating Sketch entities, some constrains are implicitly detected when creating/editing Sketch entities. See Sketch Constraints on page 169 to learn more.

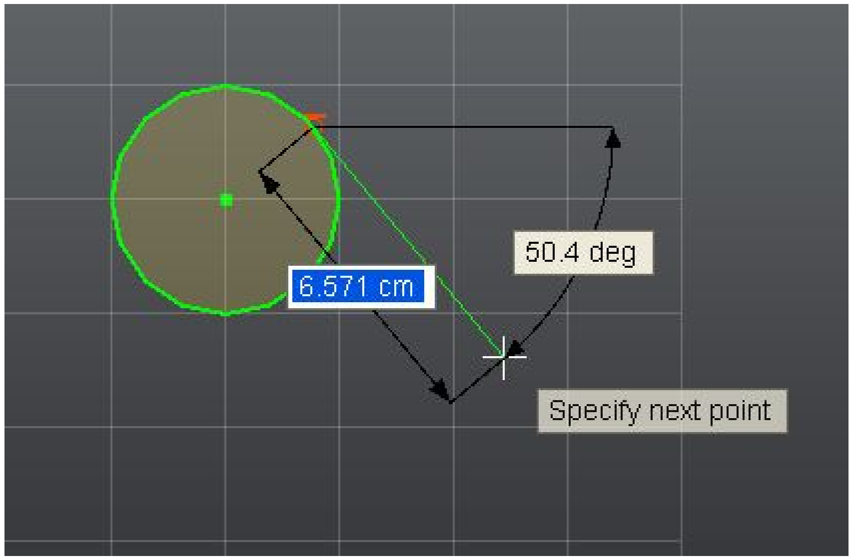
To create an arc segment while creating a line, click and hold the existing start or endpoint, and drag to create the arc. Release the cursor to end the arc.



To create a line perpendicular or tangent to a circle or arc, click and hold the start point. Then drag the cursor to end the line segment. Drag the cursor perpendicular from the start point to create a perpendicular line. See the Sketch Inferencing on page 162 page to learn more.



Line drawn perpendicular to a circle



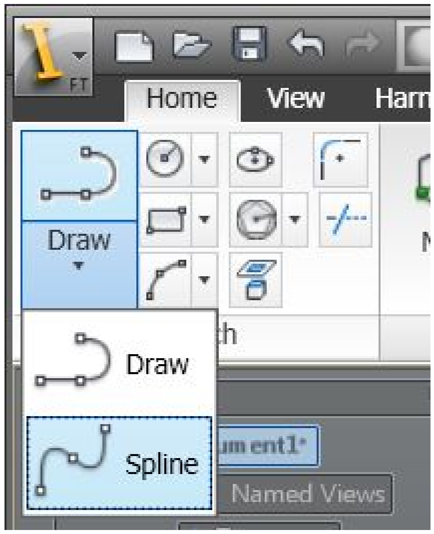
Line drawn tangent to a circle

When one line segment is created, the creation of the second segment begins where the first ended, in effect producing a polyline. To end the creation of a line or set of lines, double-click the last point you want. Then you can start creating a new line independently of any previously created lines.

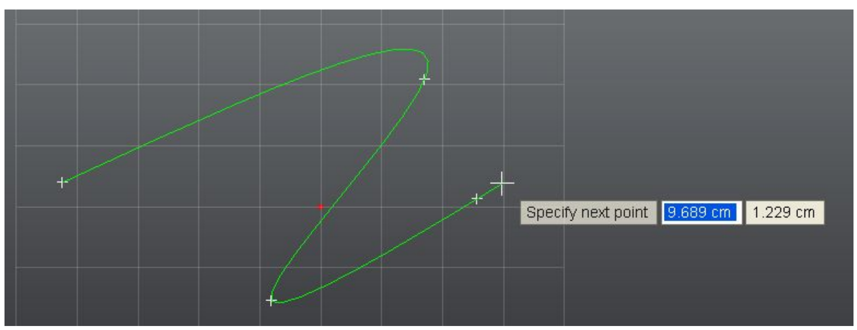
To exit the Line command, press the Esc key or pick any other command. Any line segment that is currently in a preview state will be discarded.

Spline Creation

Click the Spline command button on the ribbon to activate the Spline command. If there is no Sketch Plane on page 138 currently active, you are prompted to select one.



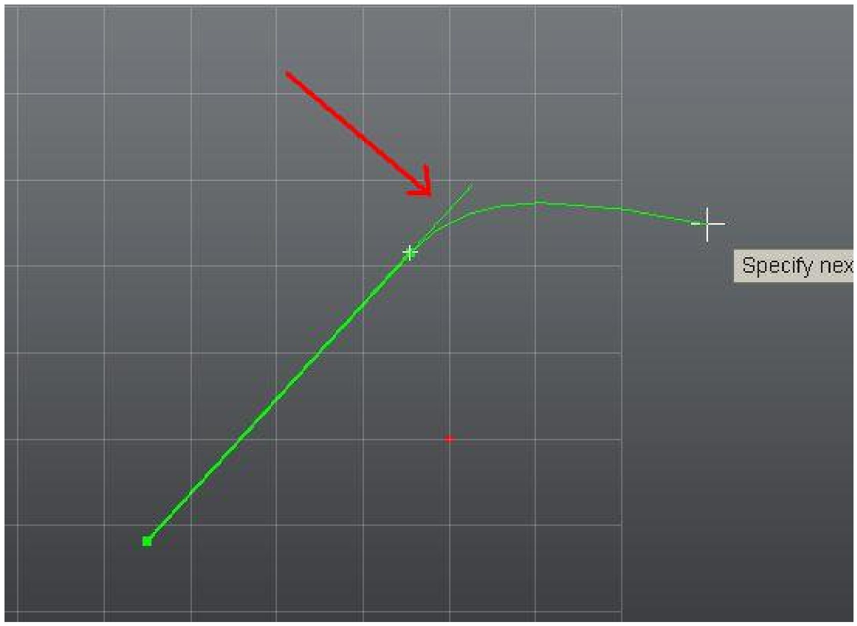
When you activate the Spline command, you can define fit points by clicking the mouse, or by typing the values into the Heads-Up Display (HUD) text boxes. This helps you to create more precise geometry. You can press the Tab key to navigate from one HUD text box to another.



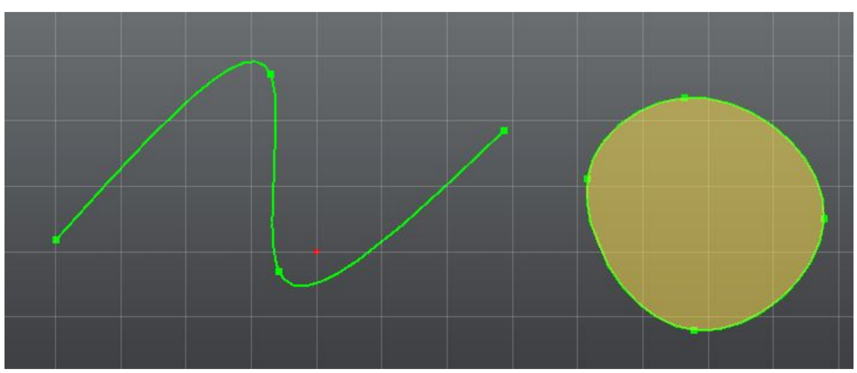
If there are existing sketch geometries on your Sketch Plane, you also receive feedback on points that lie on other sketch entities, midpoints, and Grid snap points. See Sketch Inferencing on page 162 to learn more.

Although you cannot define explicit constraints while creating Sketch entities, some constrains are implicitly detected when creating/editing Sketch entities. See Sketch Constraints on page 169 to learn more.

You can create a spline that is tangent to another line or spline by clicking the start or endpoint of the line or spline and dragging away from it. You see a tangent line indicator when the constraint is inferred.



You can create open splines as well as closed splines. You can complete the creation of an open Spline entity by double-clicking at the last point. To create a closed Spline, click the start point of the current spline entity – it also is defined at the endpoint. The current entity creation is completed and you can start creating a new entity.

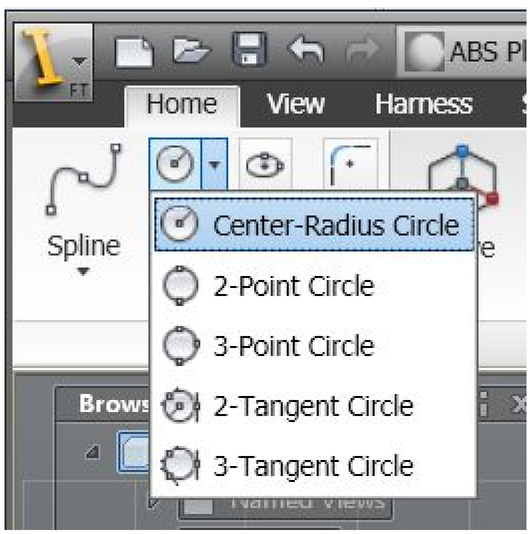


Note: It is possible to create self-intersecting splines (like a figure-8) using the spline command, but such non-manifold entities are not supported by the Sketch Profile on page 171 recognition system, and may cause undesirable results.

To exit the Spline command, press the Esc key or pick any other command. Any Spline that is currently in a preview state is discarded.

Circle Creation

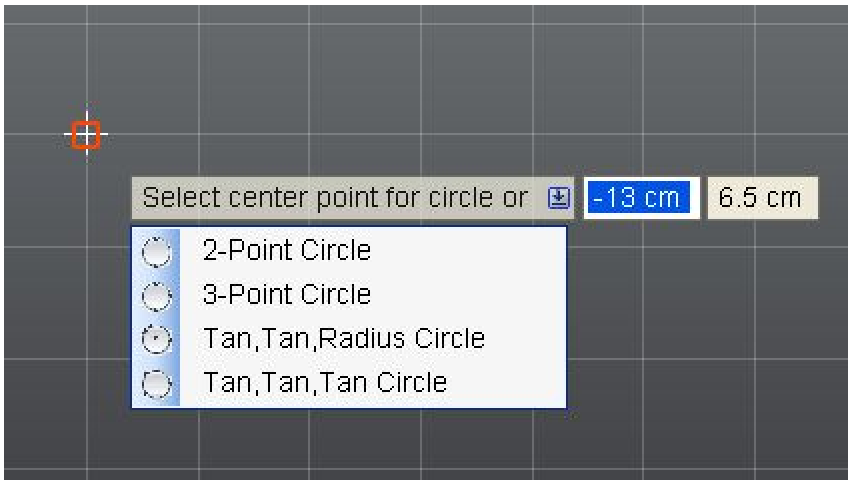
Click the Circle command button on the ribbon to activate the Circle command in the Center-Radius mode. If there is no Sketch Plane on page 138 currently active, you are prompted to select one. You can also define a Circle in other ways by picking any one of the drop-down options on the command button.



- Center-Radius Circle: Define a circle by clicking once to pick a center point, and then clicking again to define the radius.
- 2-Point Circle: Define a circle by picking the endpoints of its diameter
- 3-Point Circle: Define a circle by picking three unique points on its circumference.
- 2-Tangent Circle: Define a circle by picking two tangent lines and then a point to place its center.

- 3-Tangent Circle: Define a circle by picking three tangent lines.

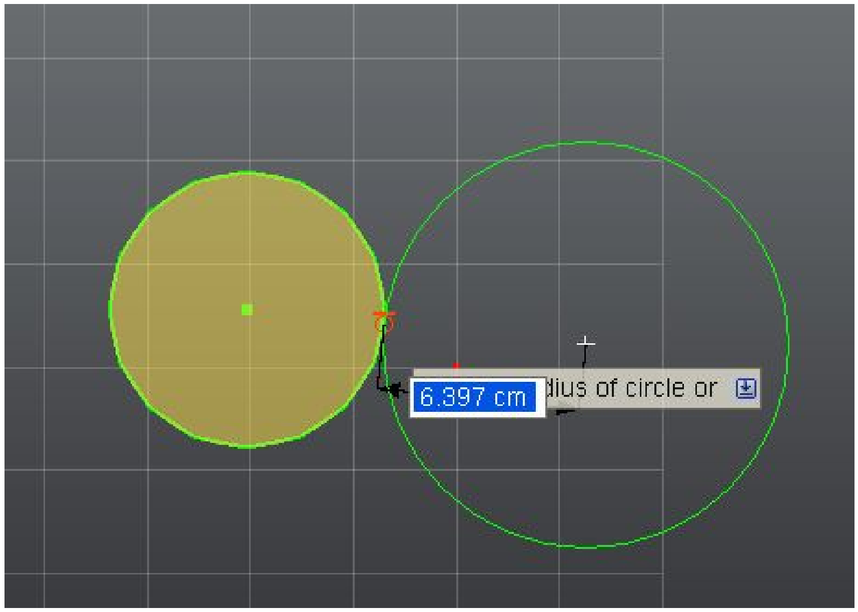
You can also switch between the different Circle creation options from the drop-down menu that is available when you are not in the middle of a creation operation. Press the down arrow key to get this option.



If there are existing sketch geometries on your Sketch Plane, you receive feedback on points that lie on other sketch entities, midpoints, and Grid snap points. See Sketch Inferencing on page 162 to learn more.

Although you cannot define explicit constraints while creating Sketch entities, some constrains are implicitly detected when creating/editing Sketch entities. See Sketch Constraints on page 169 to learn more.

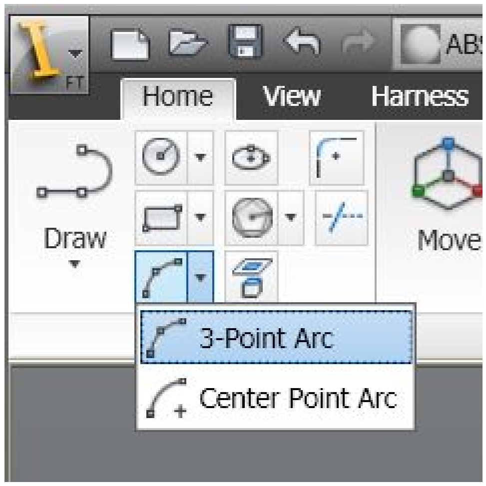
If you are creating a Circle such that it is tangent to another Line or Circle, you see a red tangent symbol (a circle with a line on top) when the constraint is inferred.



To exit the Circle command, press the Esc key or pick any other command. Any Circle that is currently in a preview state is discarded.

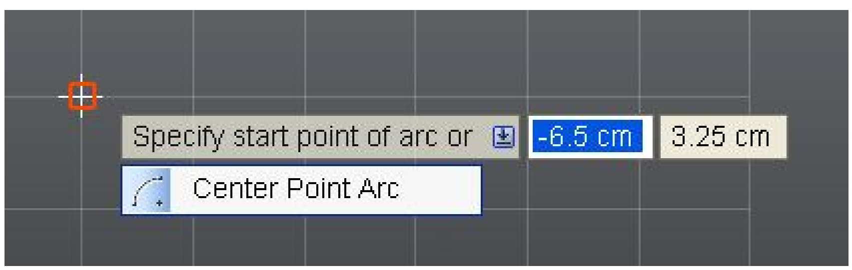
Circular Arc Creation

Click the Circular Arc command button on the ribbon to activate the Circular Arc command in the 3-Point Arc mode. If there is no Sketch Plane on page 138 currently active, you are prompted to select one. You can also define a Circular Arc in the Center Point Arc mode by picking the drop-down option on the command button.



- 3 Point Arc: Define a Circular Arc by picking the start point, endpoint and a third point on the Arc which is not collinear with the other two points.
- Center Point Arc: Define a Circular Arc by picking the center point, start point and endpoint of the Arc.

You can also switch between the two Arc creation options from the drop-down menu that is available when you are not in the middle of a creation operation. Press the down arrow key to get this option.



If there are existing sketch geometries on your Sketch Plane, you receive feedback on points that lie on other sketch entities, midpoints, and Grid snap points. See Sketch Inferencing on page 162 to learn more.

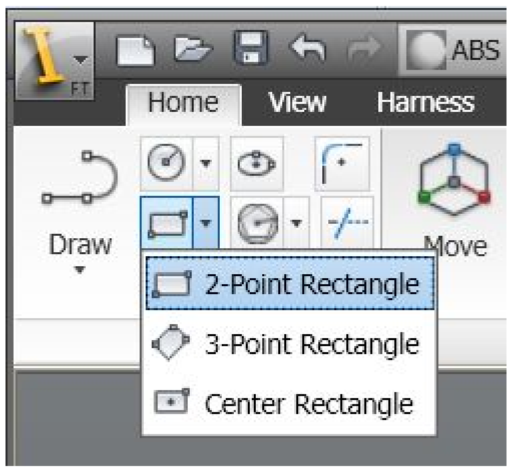
Although you cannot define explicit constraints while creating Sketch entities, some constrains are implicitly detected when creating/editing Sketch entities. See Sketch Constraints on page 169 to learn more.

You can create an arc such that it is tangent to an existing Line or Circle/Circular Arc.

To exit the Circle command, press the Esc key or pick any other command. Any Circle that is currently in a preview state is discarded.

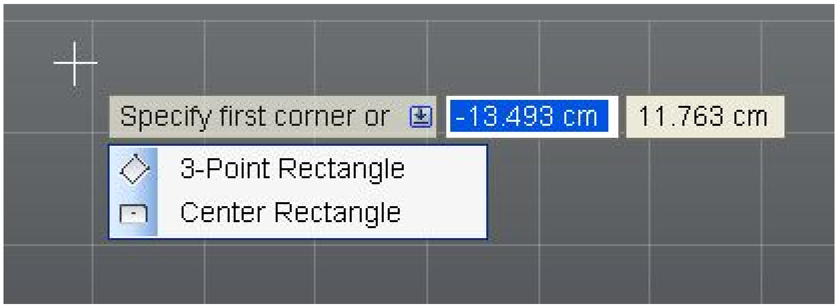
Rectangle Creation

Click the Rectangle command button on the ribbon to activate the Rectangle command in the 2-Point mode. If there is no Sketch Plane on page 138 currently active, you are prompted to select one. You can also define a Rectangle in other ways by picking any one of the drop-down options on the command button.



- 2-Point Rectangle: Define a Rectangle by picking the top-left and bottom right point.
- 3-Point Rectangle: Define a Rectangle by picking two endpoints of a side, and then picking a point to define the length of the other side. This is the only type of rectangle that can be created such that its sides are not parallel to the X and Y axes of the Sketch Plane.
- Center Rectangle: Define a Rectangle by picking its center point and any vertex.

You can also switch between the different Rectangle creation options from the drop-down menu that is available when you are not in the middle of a creation operation. Press the down arrow key to get this option.



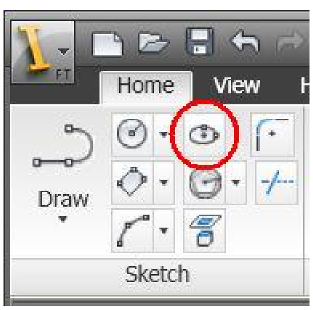
If there are existing sketch geometries on your Sketch Plane, you receive feedback on points that lie on other sketch entities, midpoints, and Grid snap points. See Sketch Inferencing on page 162 to learn more.

Although you cannot define explicit constraints while creating Sketch entities, some constrains are implicitly detected when creating/editing Sketch entities. See Sketch Constraints on page 169 to learn more.

To exit the Rectangle command, press the Esc key or pick any other command. Any Rectangle that is currently in a preview state are discarded.

Ellipse Creation

Click the Ellipse command button on the ribbon to activate the Ellipse command. If there is no Sketch Plane on page 138 currently active, you are prompted to select one.



You can define an ellipse by picking the center point, followed by another point to define the major axis and finally a third point to place the minor axis.

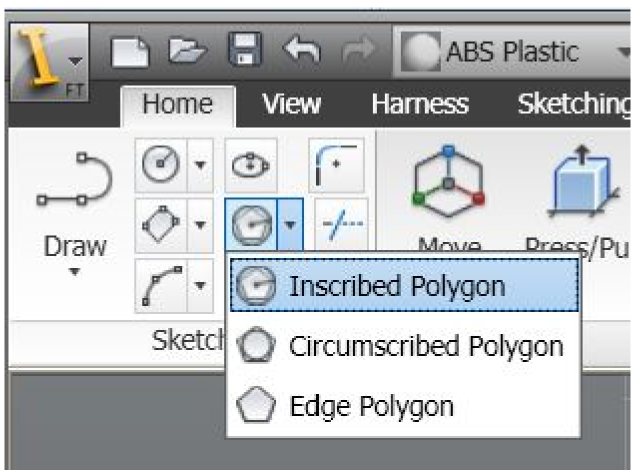
If there are existing sketch geometries on your Sketch Plane, you receive feedback on points that lie on other sketch entities, midpoints, and Grid snap points. See Sketch Inferencing on page 162 to learn more.

Although you cannot define explicit constraints while creating Sketch entities, some constrains are implicitly detected when creating/editing Sketch entities. See Sketch Constraints on page 169 to learn more.

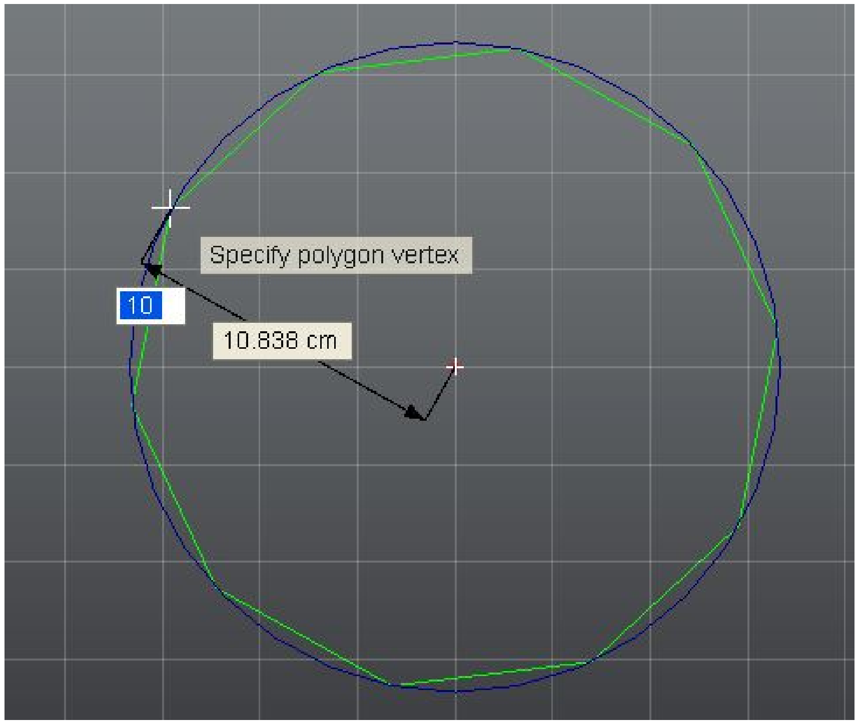
To exit the Ellipse command, press the Esc key or pick any other command. Any Ellipse that is currently in a preview state are discarded.

Polygon Creation

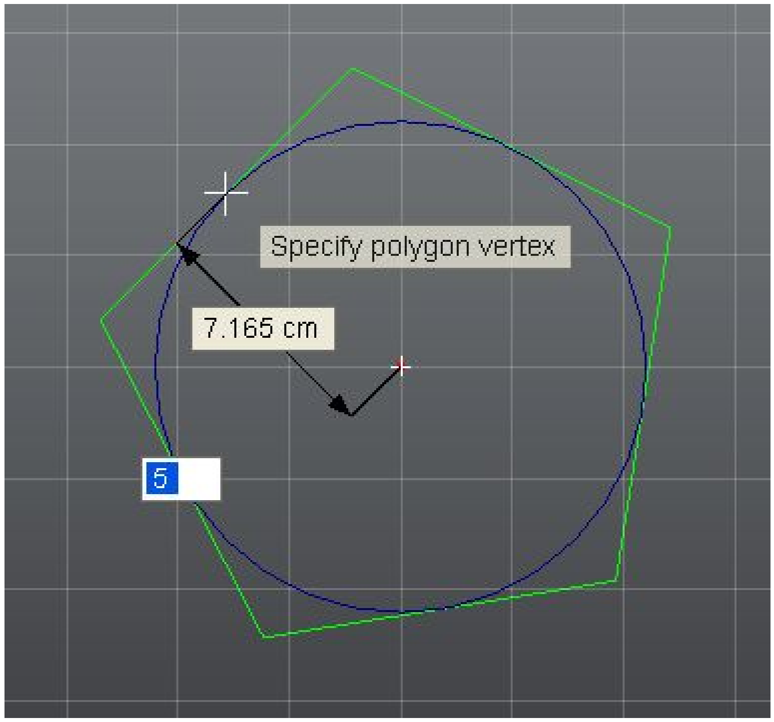
Click the Polygon command button on the ribbon to activate the Polygon command in the Inscribed mode. If there is no Sketch Plane on page 138 currently active, you are prompted to select one. You can also define a Polygon in other ways by picking any one of the drop-down options on the command button.



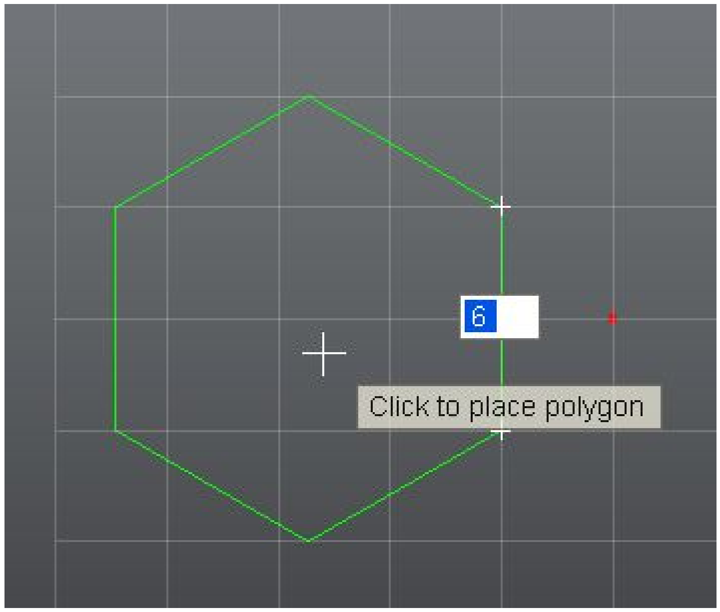
- Inscribed Polygon: Define the center and radius of the Polygons circumscribed circle. Define the number of sides by typing a value into the Heads-Up-Display (HUD) text box. You see a preview of the circumscribed circle in blue.



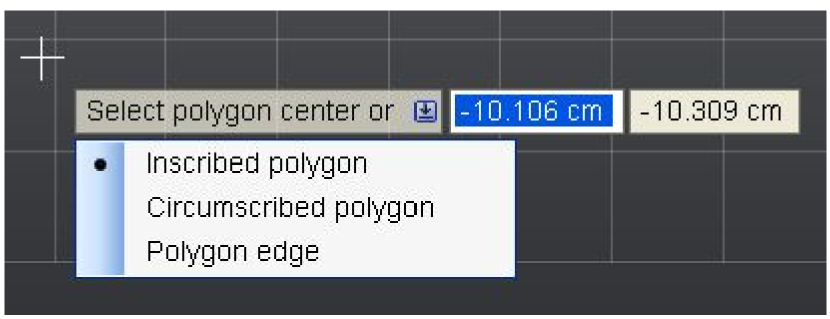
- Circumscribed Polygon: Define the center and radius of the Polygons Inscribed circle. Define the number of sides by typing a value into the HUD text box. You see a preview of the inscribed circle in blue.



- Edge Polygon: Define a Polygon edge, and then place the polygon by clicking on either side of the edge. Define the number of sides by typing a value into the HUD text box. You see a Polygon preview as you move your cursor from one side of the edge to another.



You can also switch between the different Polygon creation options from the drop-down menu that is available when you are not in the middle of a creation operation. Press the down arrow key to get this option.



If there are existing sketch geometries on your Sketch Plane, you receive feedback on points that lie on other sketch entities, midpoints, and Grid snap points. See Sketch Inferencing on page 162 to learn more.

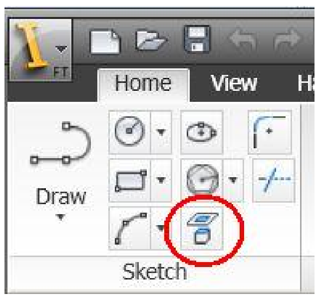
Although you cannot define explicit constraints while creating Sketch entities, some constrains are implicitly detected when creating/editing Sketch entities. See Sketch Constraints on page 169 to learn more.

To exit the Polygon command, press the Esc key or pick any other command. Any Polygon that is currently in a preview state are discarded.

Once a Polygon is created, editing one edge length does not cause all edges to update to that length. The edges and vertices are independent of each other apart from the inferred Constraints.

Project Geometry

Click the Project Geometry command button on the ribbon to activate the Project Geometry command. If there is no Sketch Plane on page 138 currently active, you are prompted to select one.



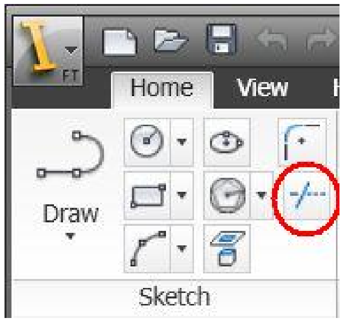
With this command, you can project model edges, work geometry and even Sketch geometry from other Sketch Planes onto your current Sketch Plane. Projected geometry is locked by default. See Locking Sketch Geometry on page 176 for further information on this topic.

If you sketch on an existing planar face, that faces edges are automatically projected onto the new Sketch Plane to assist you in creating your sketches.

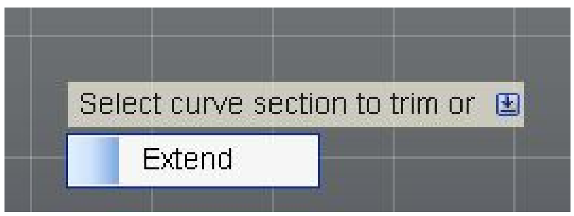
To exit the Project Geometry command, press the Esc key or pick any other command.

Trim/Extend

Click the Trim/Extend command button on the ribbon to activate the Trim/Extend command. If there is no Sketch Plane on page 138 currently active, you are prompted to select one.

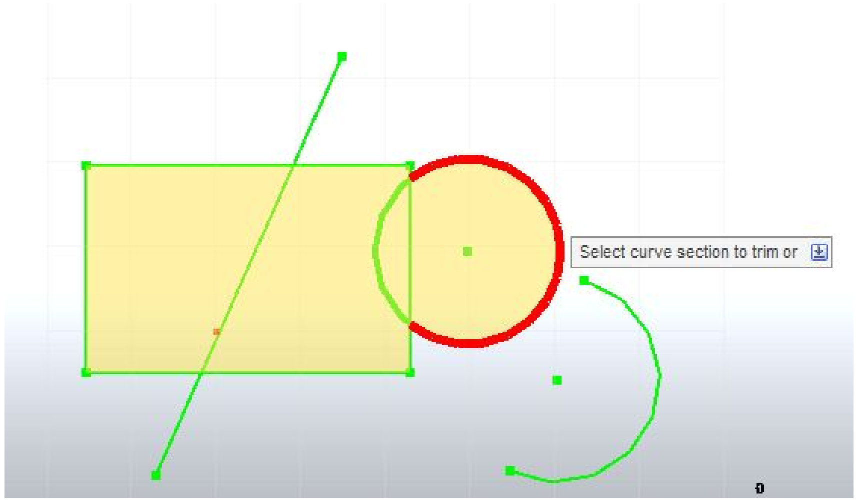


With this command, you can trim Sketch entities against other entities on the Sketch plane, or extend them to other entities. The command is in the Trim mode by default. You can switch to the Extend mode by using the drop-down menu (press the down arrow key). Invoking the drop-down menu again switches back to the Trim option.



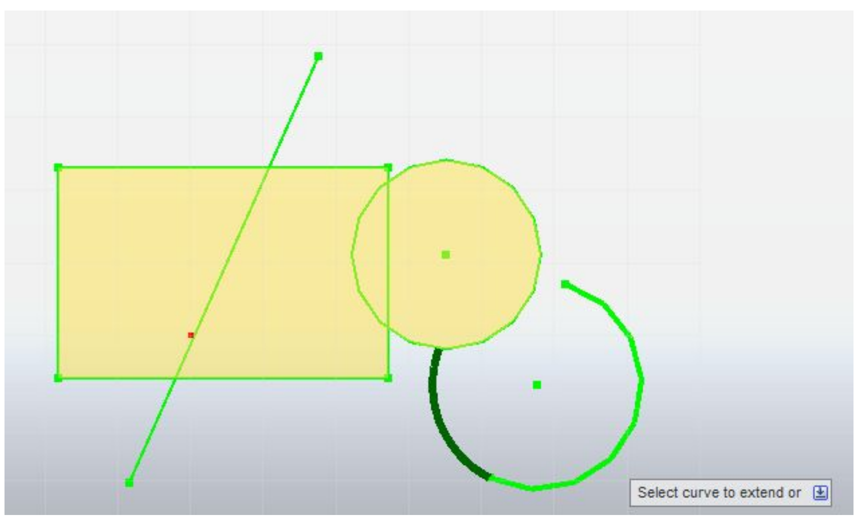
Trim

When you are in the Trim mode, you can hover over any sketch entity. If it is possible to trim that entity against another entity, you see the section to be trimmed highlighted in red. The section to be trimmed always lies under your mouse cursor. A Trim operation can delete an entity if it cannot be trimmed against any other entity. In such a case the entire entity is highlighted in red. Clicking the mouse when you see a preview commits the Trim operation.



Extend

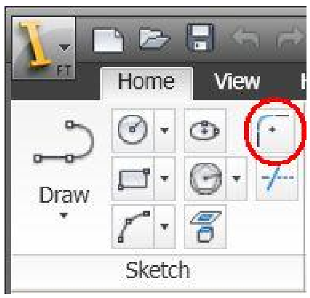
When you are in the Extend mode, you can hover over any sketch entity. If it is possible to extend that entity against another entity, you see the section to be extended highlighted in dark green. If the entity can be extended at two ends (for example, an arc), the end that extends is the one closest to your mouse position. If an entity cannot be extended against another entity, you do not see a preview and clicking it has no effect. Clicking the mouse when you see a preview commits the Extend operation.



To exit the Trim/Extend command, press the ESC key or pick any other command.

Sketch Fillet

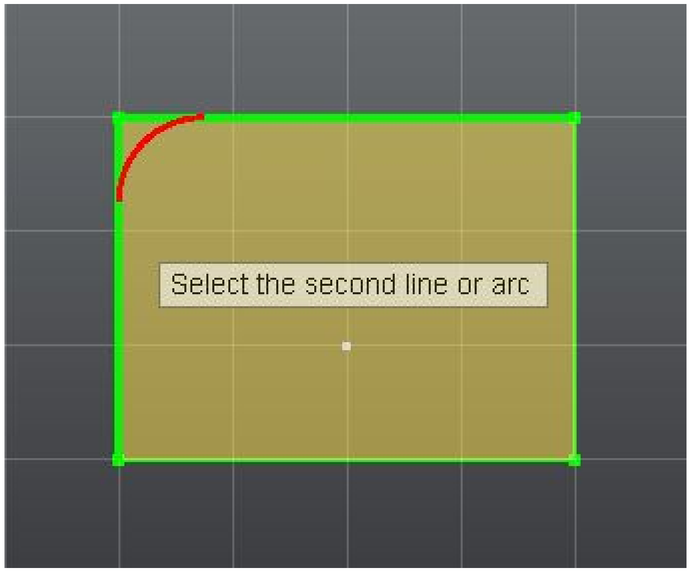
Click the Sketch Fillet command button on the ribbon to activate the Sketch Fillet command. If there is no Sketch Plane currently active, you are prompted to select one.



With this command, you can create fillets between two intersecting lines, two parallel lines, a line and a circular arc that intersect, and two circular arcs that intersect. Fillets cant be applied to circles, ellipses, elliptical arcs and splines. Fillets also cant be applied to non-intersecting entities, with parallel lines being an exception.

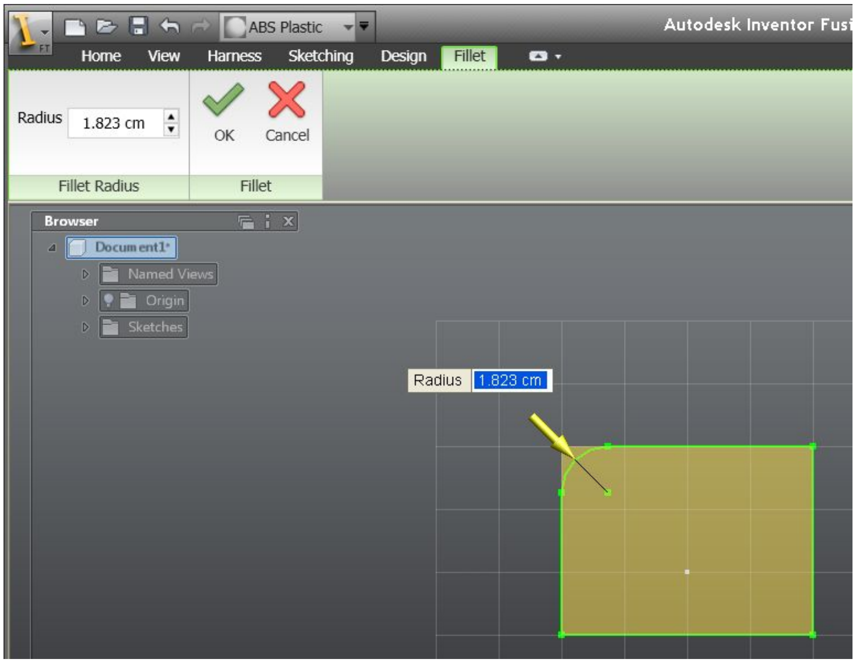
When you invoke the Sketch Fillet command, you see a Context Ribbon. The ribbon starts out with just a Cancel option in it. You are prompted to select the two entities to fillet. When you mouse over an entity, it highlights in light green if it is a valid selection. If not, you do not see any highlight. Clicking a highlighted entity selects it for the fillet operation.

Once you have selected one valid entity, moving your cursor over other entities shows you a preview of the fillet, if a fillet can be created between the selected entity and this entity. The preview is displayed as a red arc. To calculate the starting radius of the fillet arc, we consider the lengths of the lines and radii of the arcs involved in the operation. The fillet arc length is 25% of the smaller value.



Highlighting another valid entity removes the existing preview and show you a new preview. If two entities can be filleted in more than one way (e.g. two lines intersecting to form an X have four possible fillet options) then the selection pick points determines which solution is created. See FilletPreview.avi on page 160 for a demonstration.

Once you have selected two valid entities, you see a fillet preview and an updated Context Ribbon.



You can click and drag on the yellow arrow to define the fillet radius. You can also type into the Radius edit box to change the fillet radius.

A zero radius is not valid. If you drag the arrow such that the radius is zero, or you type in 0.0 in the box, you see an error message. Change the value to something valid

Click OK to commit the preview and exit the command. The Sketch Fillet differs from other Sketch commands in that invoking another command also commits the preview and exit the command. Click cancel or press Esc to discard the preview and exit the command.

Sketch Inferencing

Introduction

While you are in the process of creating a new entity, your active Sketch Creation command picks up certain inferences to aid you in creating your entity more accurately. For example, you can precisely connect one line to another. Inferences may also be detected when editing Sketch entities.

Note: Some inferences produce temporary constraints that are only available when creating a particular entity. They are not persisted and cannot be added, removed or changed. See Sketch Constraints on page 169 for more information.

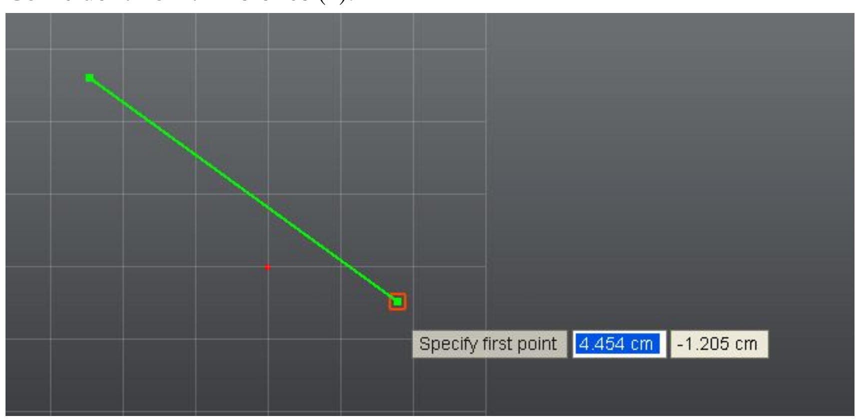
Some types of inferences are dependent on “touching” other Sketch entities with your mouse when in the middle of creating a new Sketch entity. For example, if you are moving your mouse when creating a line, and move it over a point or another line, the command remembers that you “touched” those entities and they now become reference entities for your currently active Sketch command. Touching a point or curve itself results in an inference.

Here is a list of inferences that are picked up and displayed to you through some graphical symbol. The number in parenthesis next to the inference name indicates the priority given to that inference, with 1 being the highest priority.

Geometry Inferences

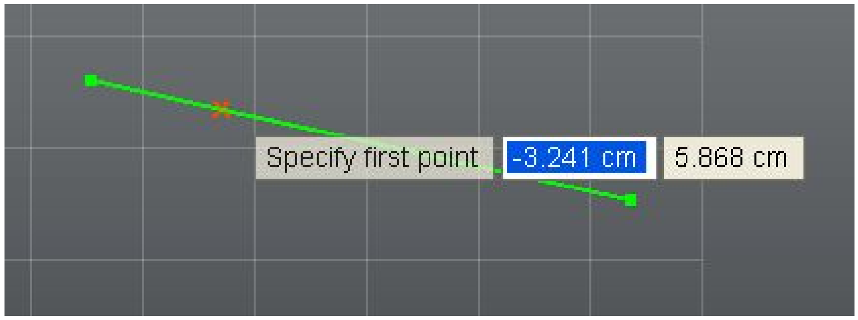
Geometry inferences are displayed when your mouse is over some key geometry points.

- Coincident Point Inference (1):



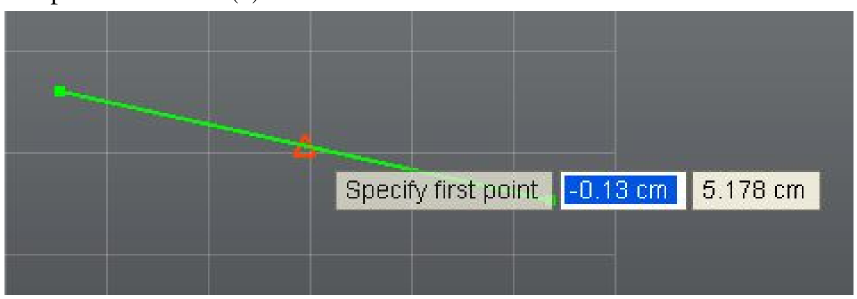
If your cursor touches an existing Sketch point, you see a red square symbol. If the point happens to be the center of a circle or circular arc, then you see a red circle instead of a red square. If you click your mouse when you see this inference, your new point is guaranteed to coincide with the inferred point.

- Point On Curve Inference(2):

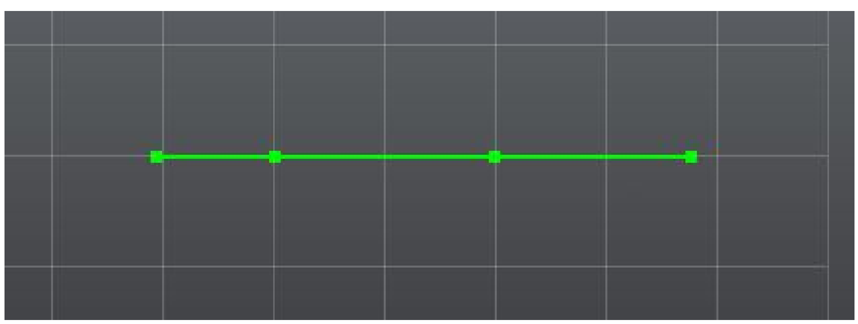


The red X symbol on any curve indicates that your cursor is touching a curve. If you click your mouse when you see this inference, then your new point is guaranteed to lie on the curve.

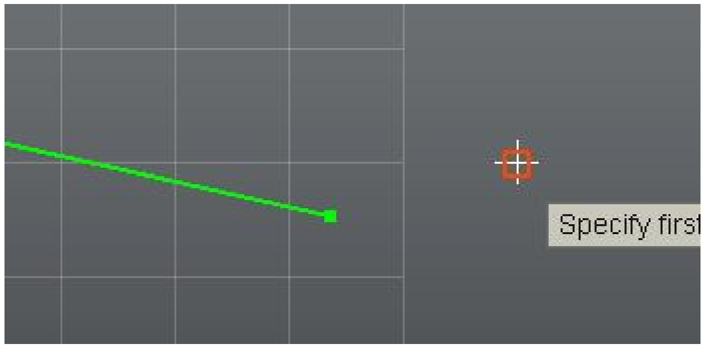
- Midpoint Inference (5):



If your cursor touches the midpoint of a straight line, you see a red triangle symbol indicating the midpoint. If the line has multiple segments, as in the following image, you do not see this inference.



- Sketch Grid Point Inference (6):

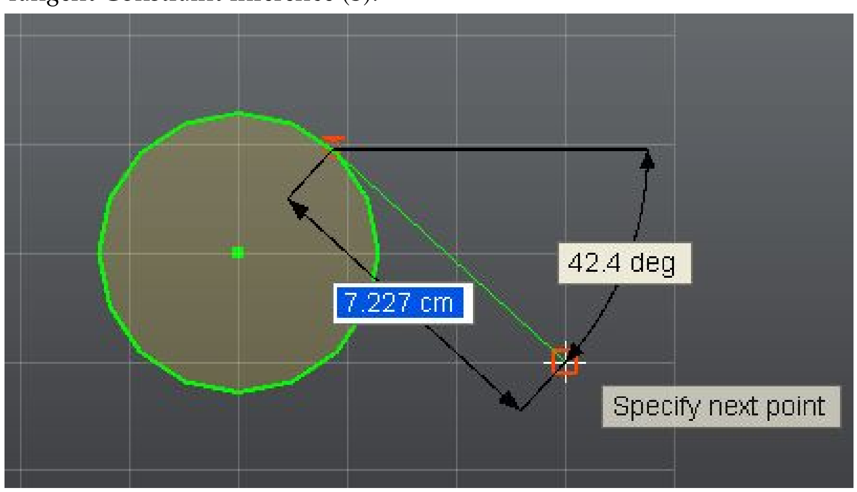


If your cursor touches Sketch Grid snap point (on the visible Grid or the infinite Grid) you see a red square symbol. To learn more about Grid snapping, see the Sketch Grid on page 138 page.

Constraint Inferences

Constraint inferences are displayed based on your creation action. These inferences produce temporary constraints that affect how your entity is created. Some of these constraints is also inferred when you are editing a Sketch entity. See Sketch Constraints on page 169 for more information:

- Tangent Constraint Inference (3):

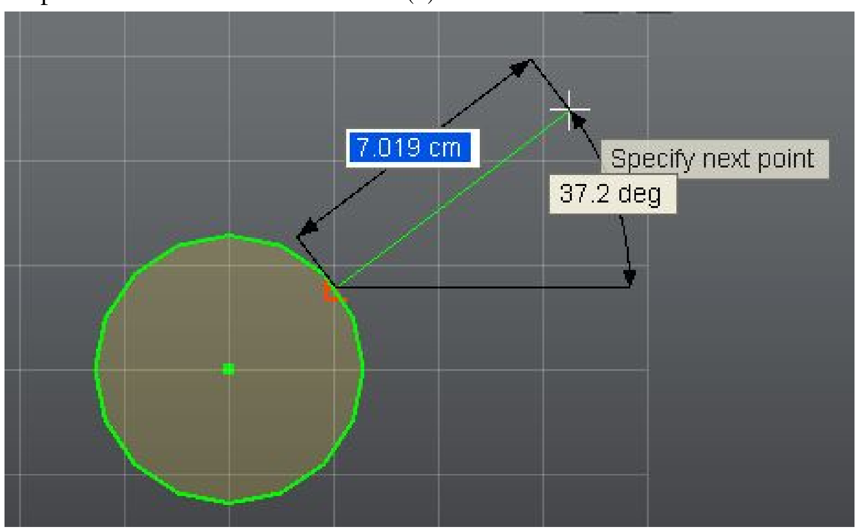


Consider a situation in which you have a circle (or circular arc) drawn on your Sketch Plane. You now start drawing a line. If the first point on your line lies on the circle (use the Point-On-Curve inference to achieve this) and you drag/move the mouse in a direction roughly tangential to the circle, a Tangent constraint is inferred. You see a red circle symbol with a red line drawn tangent to it.

If you actually clicked the point on the circle before moving your mouse, your first point is fixed and you only infer this constraint when you move your mouse in a direction roughly tangential to the circle at your picked point.

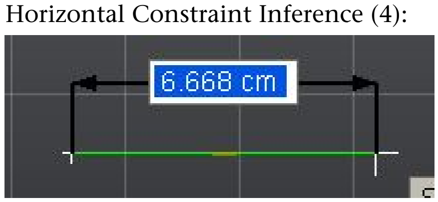
However, if you pressed your mouse left button on the circle (without releasing it) and dragged the mouse in a direction tangential to the circle, your first point is not fixed. You can drag the mouse in any direction and your line adjusts to remain tangential to the circle. Releasing the mouse will define both points of your line simultaneously. This behavior is not limited by the bounds of an arc – it will exist for the entire logical circle that the arc is a part of.

- Perpendicular Constraint Inference (3):

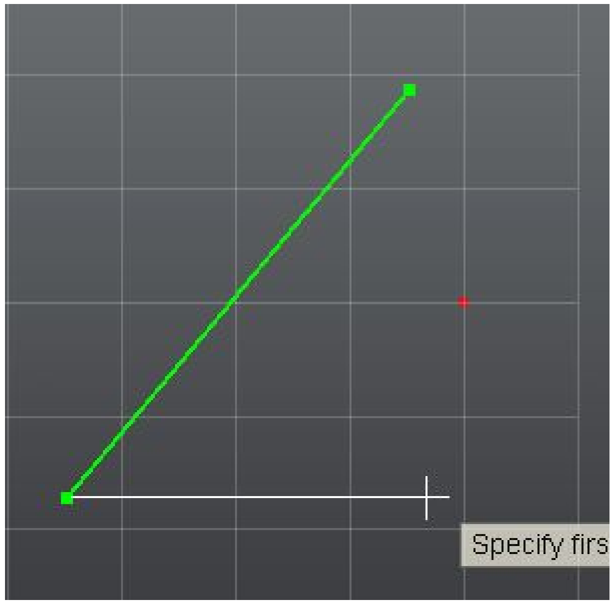


This is very similar to the Tangent Constraint, except that it is inferred when you move/drag in a direction perpendicular to the circle or circular arc.

- Horizontal Constraint Inference (4):



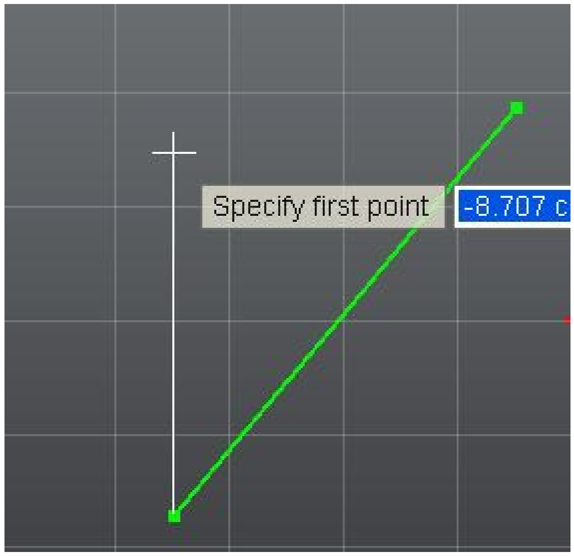
A horizontal constraint is inferred if you are drawing a line and drag your mouse such that the line is parallel to the X-axis of the Sketch Plane. It is indicated by a horizontal red line symbol.



Another kind of horizontal constraint can be inferred by “touching” a point and moving the mouse in a horizontal direction. You will see a white line stretching from the “touched” (referenced) point to your current mouse position. This means that the point that your command is in the process of defining is constrained to have the same X-axis value as your “touched” point.

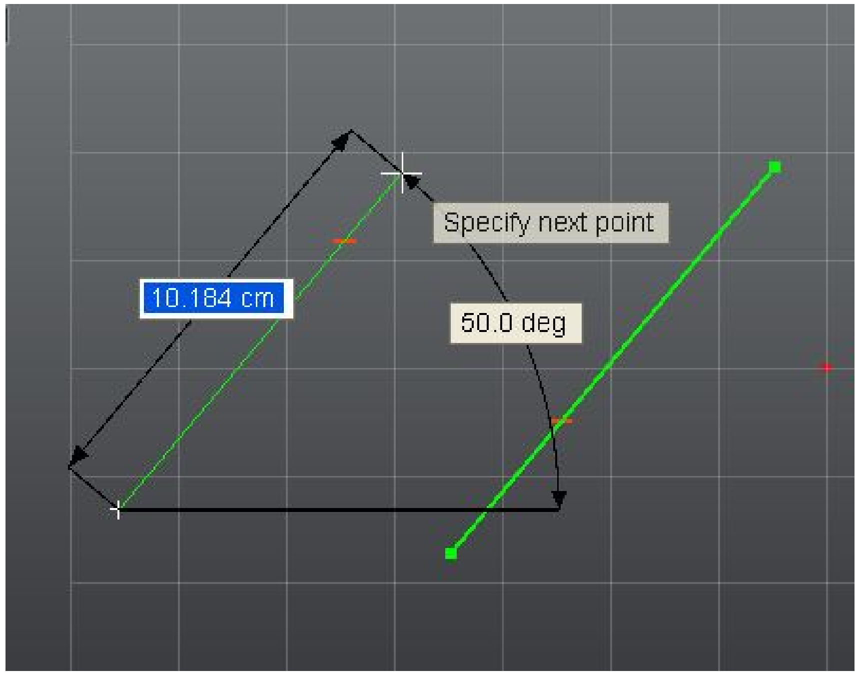
- Vertical Constraint Inference (4):

A vertical constraint is inferred if you are drawing a line and drag your mouse such that the line is parallel to the Y-axis of the Sketch Plane. It is indicated by a vertical red line symbol.



Another kind of vertical constraint can be inferred by "touching" a point and moving the mouse in a vertical direction. You will see a white line stretching from the "touched" (referenced) point to your current mouse position. This means that the point that your command is in the process of defining is constrained to have the same Y-axis value as your "touched" point.

- Parallel Constraint Inference (4)



A parallel constrained is inferred when you "touch" a line that does not share endpoints with the line you are currently trying to create, and then moving your mouse in a direction roughly parallel to that line. Your line will be adjusted to be parallel to the "touched" (referenced) line and you will see a red horizontal line symbol on both the line being created and the "touched" line.

Sketch Constraints

When you are editing a Sketch entity on page 172, some constraints inferred at the time you created the entity are recomputed to aid in editing the entity. See Sketch Inferencing on page 162 to learn more about the types of inferences. These temporary constraints work together, so that editing one entity can affect many other entities. The following list considers each inference described in the Sketch Inferencing section and discusses whether the inference will produce a constraint at edit time or not.

- Coincident Point Inference

This inference results in a temporary constraint at edit time. If two entities share a point and that point is moved, it will affect both entities.

- Point-On-Curve Inference
  This inference results in a temporary constraint at edit time. If one entity has a point that lies on another entity, moving the second entity will affect the first entity. In case the entities are lines, editing the reference line will adjust the constrained line such that it maintains its direction, but still has an endpoint touching the reference line. In some cases, you can move the reference line in such a manner that the constrained line segment no longer touches the reference line segment, but it still touches the infinite line that the reference segment is part of.

- Midpoint Inference
  This constraint does not result in a temporary constraint at edit time. If you had a line whose endpoint was the midpoint of a second line, changing the length of the second line will have no effect on the first line, although the midpoint is now changed.

- Sketch Grid Inference
  This inference does not result in any kind of constraint at edit time. If you change the Sketch Grid on page 138s snap values, it has no impact on any existing entity.

- Tangent Inference
  This inference results in a temporary constraint at edit time. If a line was created such that it was tangent to a circle, then moving either endpoint of the line will adjust the line and the circle such that the tangency is maintained.

- Perpendicular Inference
  This constraint does not result in a temporary constraint at edit time. If you created a line such that it was perpendicular to a circle, the perpendicular relationship wont be maintained on editing either entity.

- Horizontal Inference
  This inference does not result in any kind of constraint at edit time.

- Vertical Inference
  This inference does not result in any kind of constraint at edit time.

- Parallel Inference
  This inference does not result in any kind of constraint at edit time.

Removing a Constraint

An inferred Sketch constraint cannot be removed through any user action. The only way to remove it is to delete one or more of the Sketch entities that participate in the constraint.

Stopping a Sketch

To exit a sketch command, press the Esc key, or click the specific sketch command button on the ribbon. To exit the sketch environment do one of the following:

- Click the Stop Sketch glyph in the sketch environment:

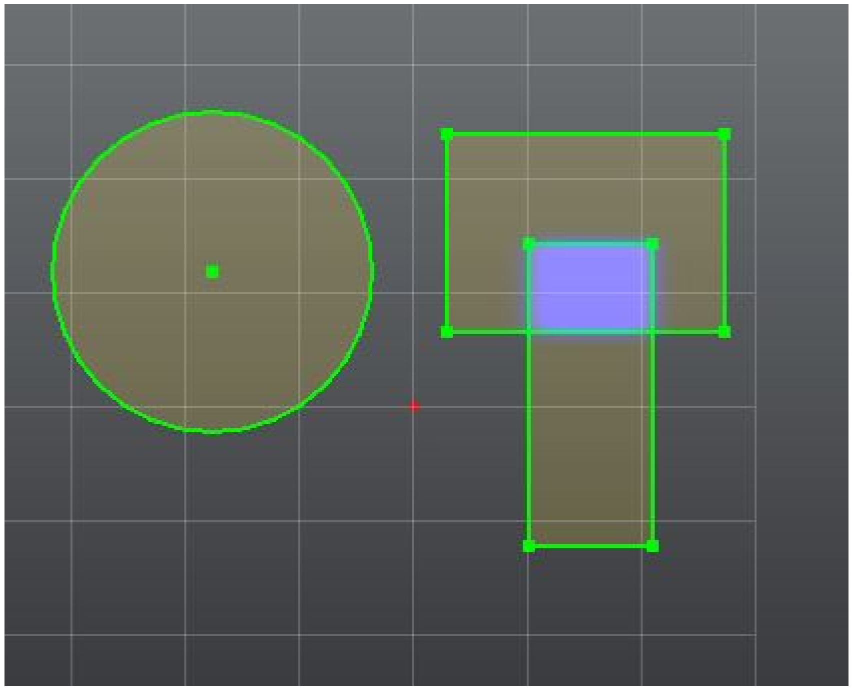
  

- Select Stop Sketch from the context menu. You can invoke the context menu by right-clicking:

Sketch Profiles

When you create Sketch entities such that they form closed regions, Sketch Profiles are automatically detected and displayed as filled regions. Each closed region corresponds to one Sketch Profile. You may then select one or more Sketch Profiles to perform further operations on, like Extrude, for example.



In the image above, the Sketch Profiles are shown in Yellow. The blue region shows a highlighted Profile.

Editing a Sketch Entity

After creating a Sketch entity, you may wish to reposition it or edit its dimensions, or delete it.

Selecting a Sketch Entity

click a Sketch entity to select it. When an entity is selected you will see its dimensions. Note that selecting one edge of a polygon does not select the entire polygon. It will only select the edge. To select the entire polygon, select the area that the polygon lies in.

Select Area: During the Select command, click and drag the left mouse button to draw a rectangle and select objects which are inside of the rectangle. Drag the cursor from Left to Right to only select objects completely inside the selection area. Drag the cursor from Right to Left to select anything inside or crossing the selection area.

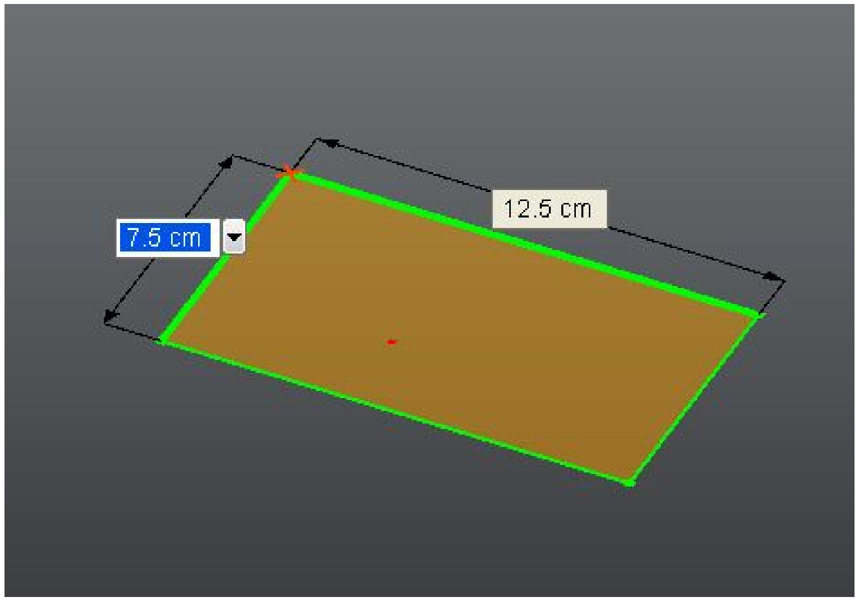
Note: When in sketch mode (for example, you have an active Sketch Plane), select area will only select Sketch curves. It will not select Sketch points or Sketch profiles on page 171. Outside of Sketch, select area will select normal topology, but will not select Sketch curves. It will select Sketch profiles.

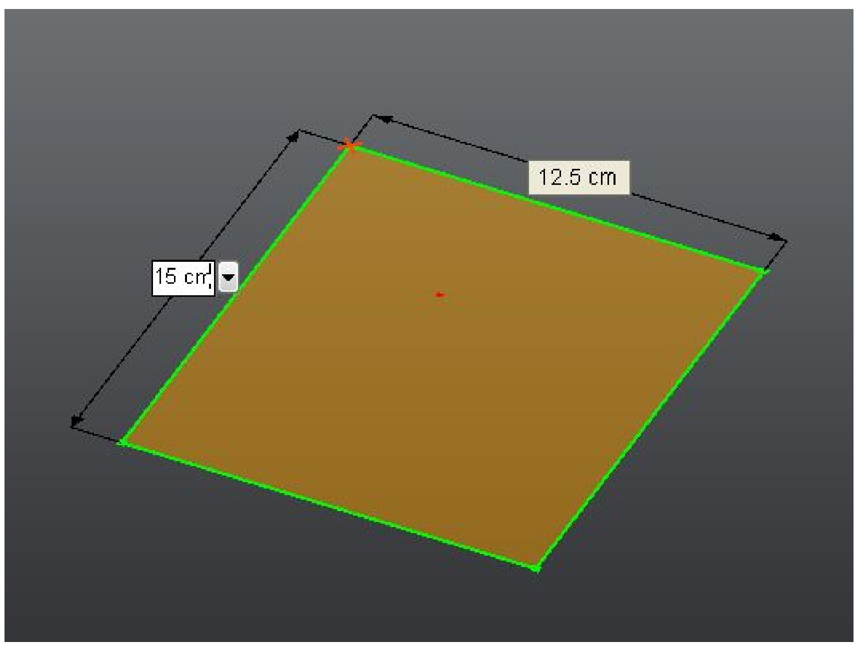
Repositioning a Sketch Entity

You can click and drag a selected entity to a new position and release the cursor to "drop" it in its new location. Note that Sketch Constraints on page 169 may come into play if the entity you are trying to move is connected to other entities, and the connected entities may also move/resize to maintain those constraints.

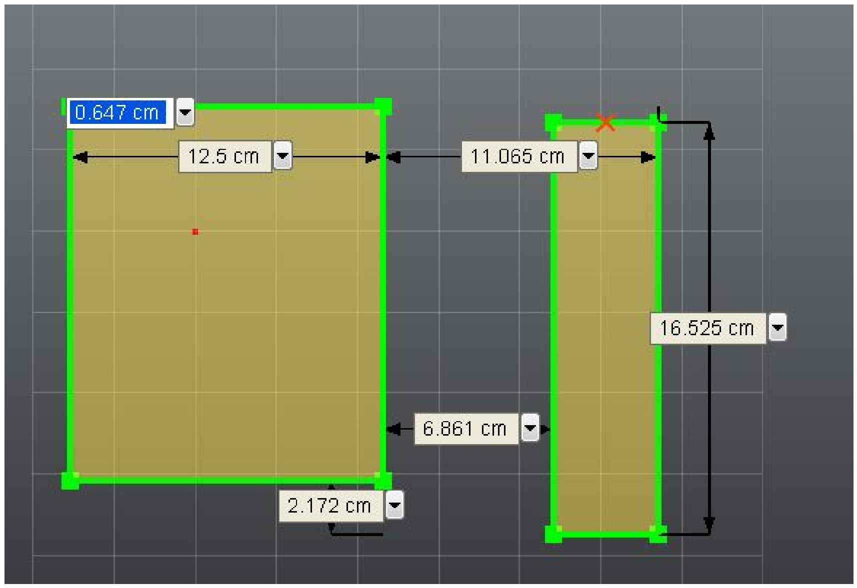
Editing Sketch Dimensions

When you select a Sketch entity, you will see its dimensions. Change the dimension value by entering a new value in the heads-up display (HUD).


7.5 cm
12.5 cm


15 cm
12.5 cm

Selecting multiple sketch entities will display additional dimensions. As you edit a dimension, other visible dimensions are locked to their current their value as the sketch changes.



A red X appears on dimensions that are being edited. The X indicates which endpoint is fixed while the dimension is being edited. To lock or unlock a sketch entity, click the entity, then invoke the marking menu (RMB) and select Lock/Unlock Geometry from the context menu. See Locking Sketch Geometry on page 176 for more information.

Selecting multiple entities will also create dimensions between the selected entities, where appropriate.

You can also edit certain entities by selecting them and dragging your mouse. For example, you can pick the endpoint of a line and drag it to extend the line.

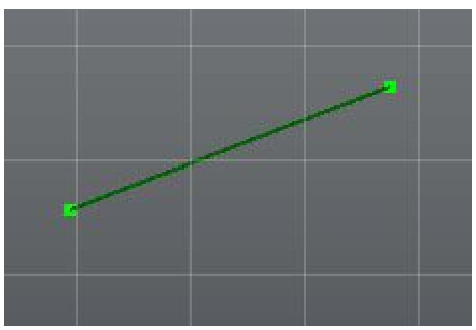
Deleting a Sketch entity

Select the Sketch entity that you wish to delete and press the delete key. The entity will be deleted. You can use area select to select multiple entities and delete them at once. You can also delete an entire Sketch Plane by right-clicking the Sketch Planes browser node and picking the “Delete” option. All Sketch entities on the plane will be deleted.

Locking Sketch Geometry

A Sketch entity can be locked by selecting it, right-clicking and picking the Lock/Unlock Geometry Option. It can be unlocked by selecting the menu item again.

When an entity is locked, it is drawn in dark green. User-created edges are unlocked by default. Projected Geometry on page 157 is locked by default.



Locking a curve does not automatically lock its endpoints or vertices. When a curve is locked, it cannot be moved. In the previous image, the line is locked; however its endpoints are not. This reduces the edit operations you can perform on the curve and even on curves or points connected to it. In the previous example, you can extend the line in either direction, but you cannot move the position of the line or change its direction.

Locking geometry will not eliminate any inferred Sketch Constraints on page 169. You will not be able to edit entities in a way that would break those constraints.

Features

This page describes the concept of “Features” in Inventor Fusion.

A Feature in Inventor Fusion is essentially a collection of faces that represents a shape corresponding to a mechanical feature.

Inventor Fusion uses the Direct Modeling paradigm as opposed to the Parametric Modeling paradigm used by Inventor. As a result, the concept of a Feature in Inventor Fusion is a little different. The main differences are:

1. Features in Inventor Fusion are not “ordered”, in that the order in which they are created and represented in the browser does not affect how the model behaves under further operations.
2. A feature created in Inventor Fusion exists only as long as it is “valid”, for example, the shape defined by the feature conforms to the definition of the feature. If you create an Extrude feature and then tweak one of the faces created by the Extrusion using the Move command such that the Extrude is no longer valid, the Extrude feature ceases to exist.
3. Features in Inventor Fusion that offer the Edit function are edited “directly”, for example, editing the feature does not cause a cascading effect on other features in the model as it would in a history-based modeler.

Inventor Fusion provides the following Features:

- Extrusion
- Revolution
- Sweep
- Loft
- Hole
- Fillet
- Chamfer
- Shell

- Rectangular Pattern
- Circular Pattern
- Mirror

Of these, Extrusion, Revolution, Sweep and Loft are Sketched Features. They are created from one or more 2D sketches on page 135.

Each Feature is created by a command that allows the user to provide the inputs necessary to create the Feature using the direct manipulation user interface.

Features can also be created on an imported model by using the Find Features on page 181 command.

Once a Feature is created, it is represented in the browser under the Component.

A Feature can be selected by clicking on the browser or by selecting one of its faces and using the Select Other on page 18 command.

Once a Feature is selected, it can be deleted by using the Delete command.

Features can be patterned on page 178 using the Rectangular Pattern or Circular Pattern commands.

Some Features provide an Edit command, which allows the Feature to be edited in place. The following Features provide an Edit command:

- Hole
- Fillet
- Chamfer
- Rectangular Pattern
- Circular Pattern

When a Feature is edited, the same direct manipulation interface as the one that is used to create the Feature is made available to the user to modify the input parameters of the Feature.

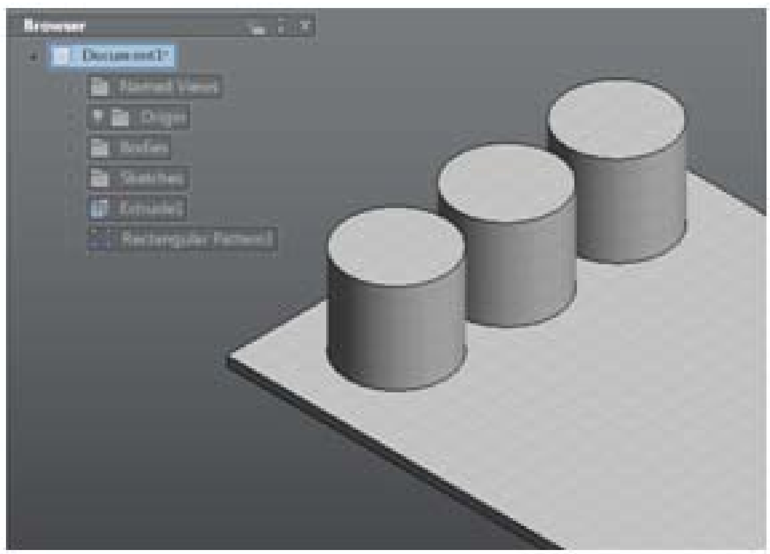
Pattern

Pattern is a specialized feature which can create copies of geometry. The copied geometry can be arranged in a circular pattern on page 180 or a rectangular pattern on page 181. Each copy of the geometry is called an occurrence. Changes to any one occurrence will effect all other occurrences in the pattern, for example the modification done by operations such as move, fillet, push/pull and draft in one occurrence will be propagated to all occurrences.

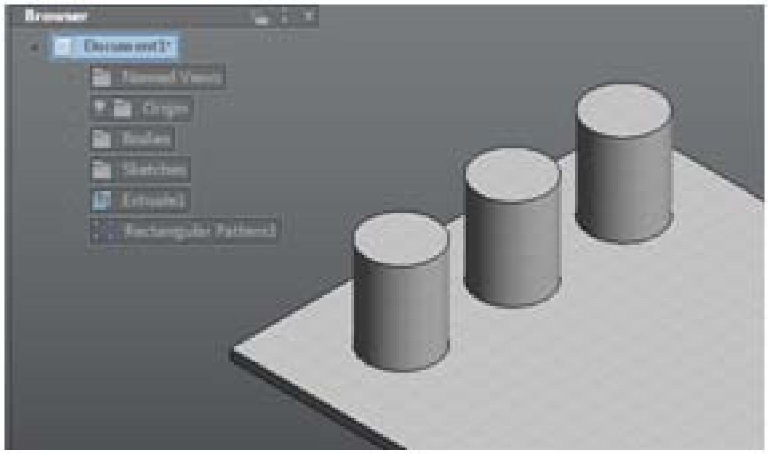
Inventor Fusion supports two types of patterns: rectangular and circular. Rectangular pattern positions occurrences in rows and columns. Circular pattern positions occurrences in an arc or circular pattern.

Example:

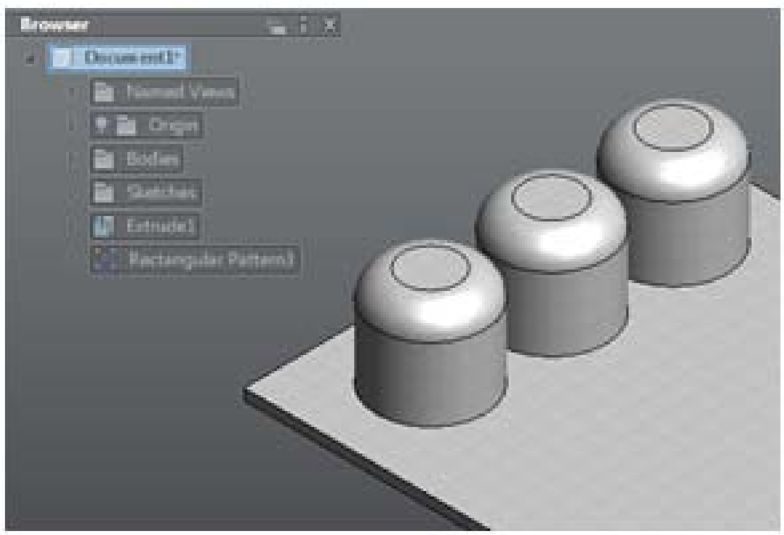
A rectangular pattern is created from a cylindrical extrusion.



A change in the size of any of the occurrences affects all occurrences.



Adding a fillet to any occurrence affects all occurrences.



Circular pattern

Circular patterns allow you to select a center around which to pattern geometry. The center must be a circular face like a cylinder, cone, or torus.

Rectangular Pattern

Rectangular patterns allow you to select a direction you can then drag the manipulators to define a pattern that follows the given direction. The direction selection must be an edge.

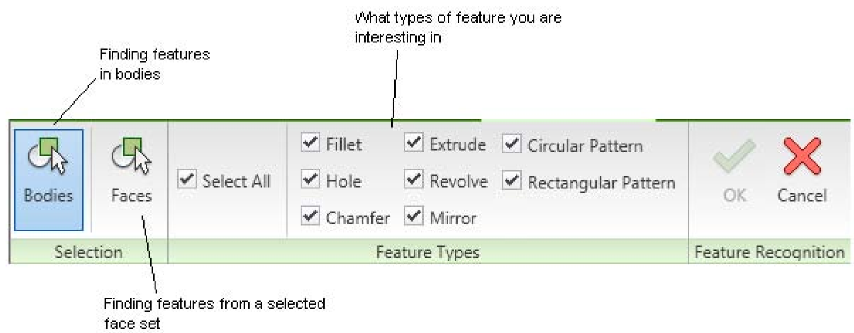
You can also drag away from the direction, in a perpendicular manner to create a rectangular pattern in either direction from one simple selection.

Find Features

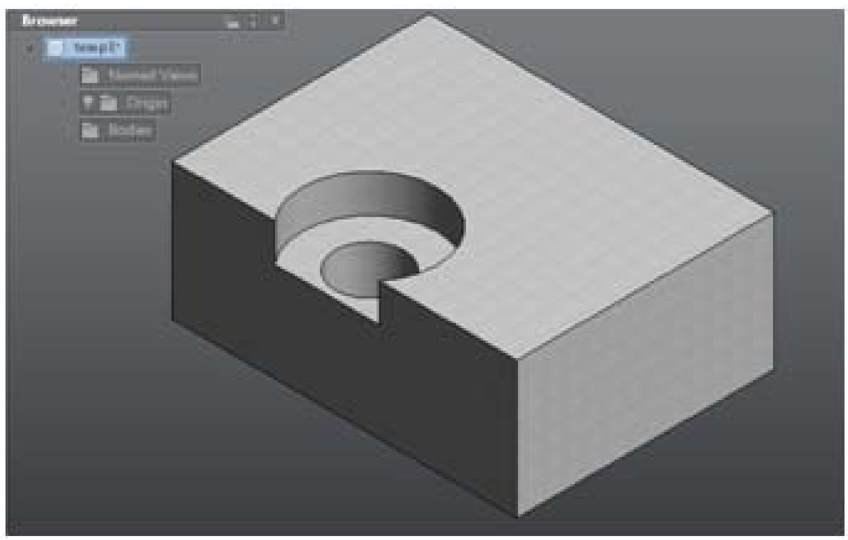
Find Features is also referred to as feature recognition. It is a process to extract design feature information from a solid model. Currently, Find Features supports:

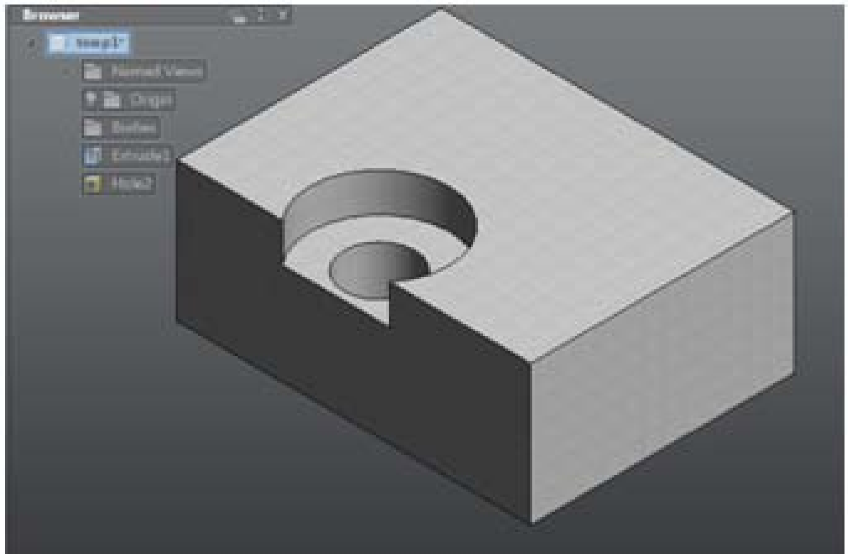
- Fillet
- Hole
- Chamfer
- Extrude
- Revolve
- Patterns
- Mirror

The Find Features command adds features to the browser. The command does not modify the geometry of the body. Features aid in selecting geometry to modify the body. Find Features can be performed on bodies or faces. The user can also specify the feature types to recognize.

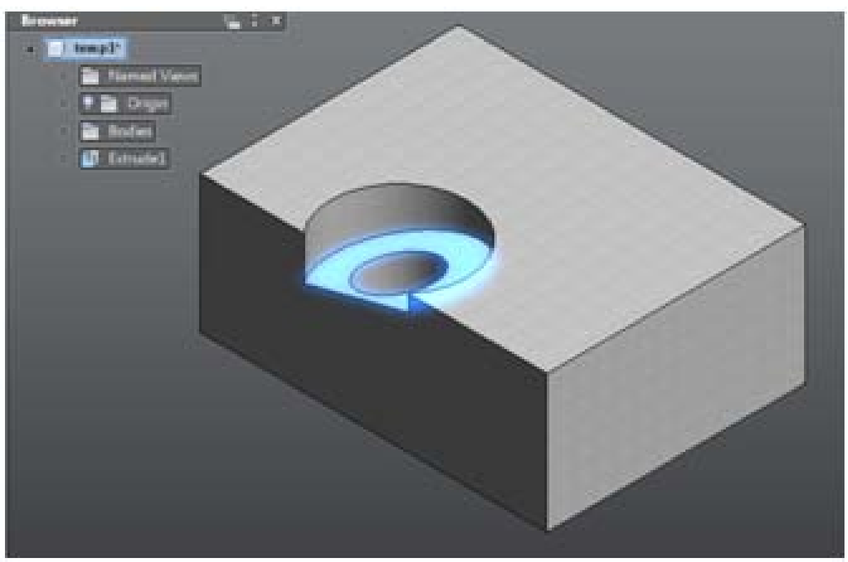


In the following image, Find Features adds an extrude (the rectangular body) and a counter bore hole (the cylindrical cut) to the browser. Once a feature has been found, it is managed the same as features created with traditional methods. Features can be edited, deleted or dissolved.





If a downstream operation modifies a feature face, the owning feature will be re-evaluated. If the face set does not satisfy the feature definition, the feature will be removed from the browser. In the previous counter bore hole example, if the user moves the highlighted face, the feature will not be a valid counter bore hole and is removed from the browser at the end of move operation.



If a find features recognizes a feature in a way that does not fit your intent you can change it. In the browser you can right-click and change the feature type. For example recognize a hole as a revolve. You can also dissolve the feature which removes the feature from the browser but leaves the geometry on the model.

Dimensions and Body Constraints

Dimensions and Body Constraints are used to constrain the faces and edges of the model in certain ways while still allowing free-form Direct Manipulation of the model in other ways.

Dimensions can be used to drive the model, or can be used as pure annotations (non-driving). For more information about dimensions as annotations, see Dimensions.

Body constraints are used to constrain faces and edges within a single component. For information on how to constrain or assemble components relative to each other, see Position and Constrain Components on page 200.

Overview

Inventor Fusion will respect the design intent of the constraints while using the Move and Press/Pull commands and when changing dimension values. If you are moving a face that is constrained to another face, then the other face is also moved in order to respect the constraint.

Constraints are propagated through patterns. If you are moving a face that is part of a pattern, then all corresponding faces in that pattern move in the same way. If any of those faces are constrained to any other faces, they are also moved as appropriate.

The face or faces that you are explicitly moving are always moved exactly as you specify. The other faces are moved to respect constraints.

If Inventor Fusion cannot find a suitable solution, then the preview stops and the Error Glyph appears.

Note that you don't always need to use body constraints to merely move a face in a constrained way. You can use the move triad, possibly aligning it to an edge, and by selecting the appropriate element of the triad, you can force the face to only move in certain limited directions.

Body constraints

Body constraints include the following types:

- Coplanar: Two planar faces are made to lie in the same plane
- Center: Two cylindrical faces are made to lie along the same axis
- Parallel: Two planar faces are made to be parallel
- Perpendicular: Two planar faces are made to be perpendicular

All of these body constraint types are accessible from the Constrain command.

Note: In addition to body constraints, the Constrain command offers the ability to create inter-component (assembly) constraints. Component constraints are discussed here: Position and Constrain Components on page 200

Body constraints are used to constrain faces and edges within a single component. You can constrain faces or edges between different bodies within the same component, but you cannot use body constraints between faces or edges of different components.

Body constraints cause the body to change shape to meet the constraints. In contrast, Component (assembly) constraints treat each component rigidly. Inventor Fusion solves body constraints first, and then component constraints.

When you use the Constrain command to create body constraints:

- Only the appropriate type of faces are available to be selected (cylindrical faces for Center constraints, planar faces for the other types).
- The first face that you select is marked as grounded, using the Anchor glyph. This means that when the constraint is first applied, the grounded face will remain where it is, and the other face will move (as well as any other faces that may need to move).
- You can switch the grounded face with the Tab key, before applying the constraint.
- The groundedness is temporary; after the constraint is created, either or both faces may move to satisfy the entire set of constraints. There is no way (nor no need) to make a face permanently grounded. Unconstrained faces are never moved by the solver. (In contrast, assembly components can be grounded.)

Locked Dimensions

In addition to explicit body constraints, you can lock a dimension to force the model to maintain that dimension at the current value regardless of other changes. The following video shows the basic operation:

Locked dimensions are shown in bold.

You can lock a dimension in one of two ways:

- Double-click the dimension and change its value (pressing Enter when done, or Escape to cancel)
- Use the context-menu Lock command (check box) to lock the dimension at its current value

You can uncheck the Lock check box to make a dimension unlocked again.

Only certain dimensions can be locked. If double-clicking does nothing, and the Lock check box is not available on the context menu, then the dimension cannot be locked.

Dimensions and explicit body constraints are solved together. No precedence is given to one or the other.

Locked dimensions constrain edges. The adjacent faces are moved to make the edges be the correct size and in the correct position.

When editing the value of a dimension:

- Inventor Fusion shows you a preview of the new value. There is a built-in delay to minimize unnecessary previews of intermediate values as you are typing.
- Most dimensions show an anchor glyph. This indicates which side is grounded, and the other side will move in response to the new value of the dimension. Move the mouse to the other side to swap the anchor.

Details

Inventor Fusion sometimes includes invisible provisional constraints to maintain obvious perpendicularity and parallelism. If this is not appropriate, you can move the faces slightly, before applying the constraint or locking the dimension.

If a Move or Press/Pull operation causes an edge or face to be split, and the edge/face has a dimension or constraint attached to it, then Inventor Fusion will automatically apply the dimension/constraint to one of the resulting entities. The other edge/face will not inherit any constraining behavior from the original. It is not possible to predict or specify which edge/face will acquire the constraint.

It is possible to create situations where a dimension becomes invalid. In this situation, the dimension changes color during the preview. If you finish the command with dimensions in this state (sick), then the sick dimensions are deleted.

It is possible to create a set of constraints that cannot be solved. In such a case, the error glyph will appear. You will have to Undo or delete constraints to get back to a properly solved model.

Dimensions and body constraints are saved when using the DWG format. They are not saved in any other format.

Limitations

Current limitations:

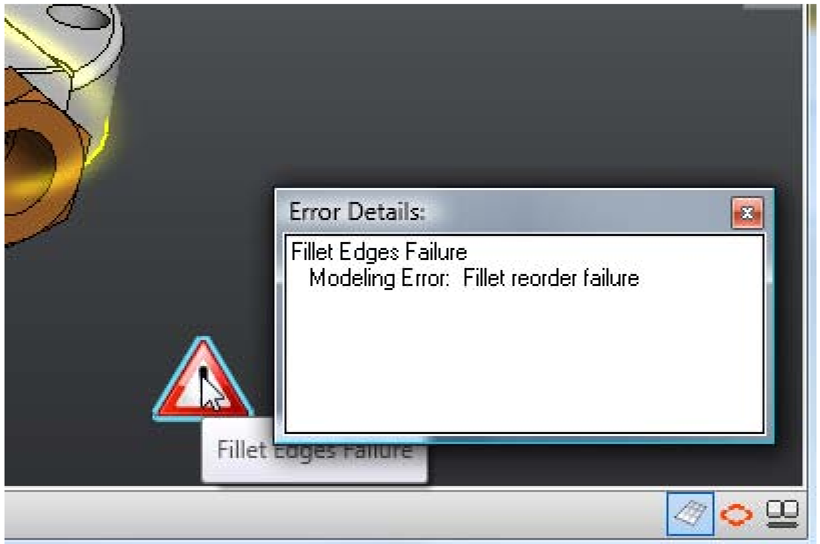
- Constraints are not respected when using the Draft command.
- Under some circumstances, invisible intermediate states may fail for no visible reason. Some of these situations may be worked around by moving the elements through a different route. Other situations may be worked around by moving a little bit at a time, and accepting each move.

Error Handling

At times, commands on certain geometry, or with certain inputs, will result in an error condition. When this happens, Inventor Fusion will display the Error Glyph, in the lower right hand corner of the application window:

If you hover over this glyph, a commandtip will be displayed which gives a brief description of the error:

If you click the error glyph, more information on the error is given, in a dialog box:



Use the upper-right x icon in this dialog to dismiss it.

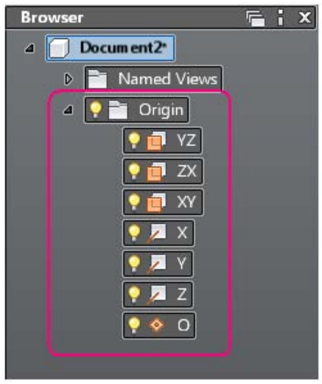
The error glyph will disappear when the condition that is causing it is removed. For instance, errors in Fillet are often caused when the fillet radius is too large. entering a smaller radius, or dragging the manipulator to a legal value will cause the error glyph to disappear.

When the error glyph is displayed, any command in progress cannot be committed. So, the OK check mark will be disabled whenever the command is in an error state. You can cancel the command, but not commit it.

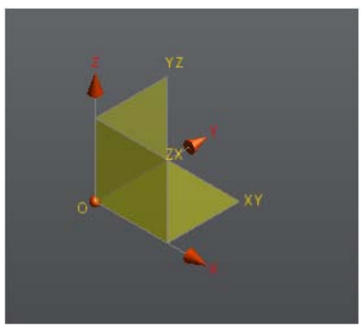
Work Geometry

Work geometries are construction objects used to create model geometry. Work geometries include work planes, work axes, and work points. They are useful to create features on page 177.

Components on page 193 contain default work planes, work axes, and a work point at the origin. Expand the Origin node in the browser to display these work geometries in the browser.



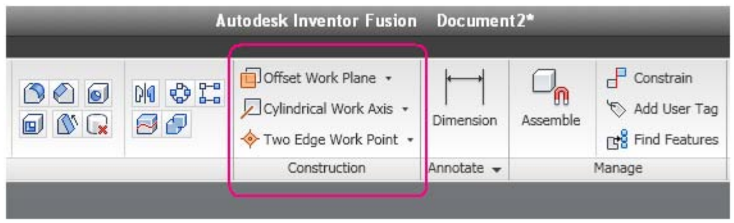
The three default work planes are the YZ, ZX, and XY planes. The three default work axes are the X, Y, and Z axes. The single default work point is the O point which represents the value 0, 0, 0 in the model. The default work planes and axes are based on the origin point. The following image shows the origin geometries in the graphics window.



Note: The origin geometries are not visible by default. When a new sketch on page 135 is started, the geometries are displayed automatically. Use the light bulb icon to manually control the visibility of planes, axes, and point.

Create Work Geometry

Work geometry creation commands are grouped as work plane, work axis, and work point creation commands. Each group is arranged in a drop down. Access the different creation methods by clicking the drop down arrow.



Create work geometry by selecting a work geometry command then selecting existing geometry. Except for Offset Work Plane and Angle Work Plane, no values are required to create work geometry. The geometry you select defines the position of the work geometry.

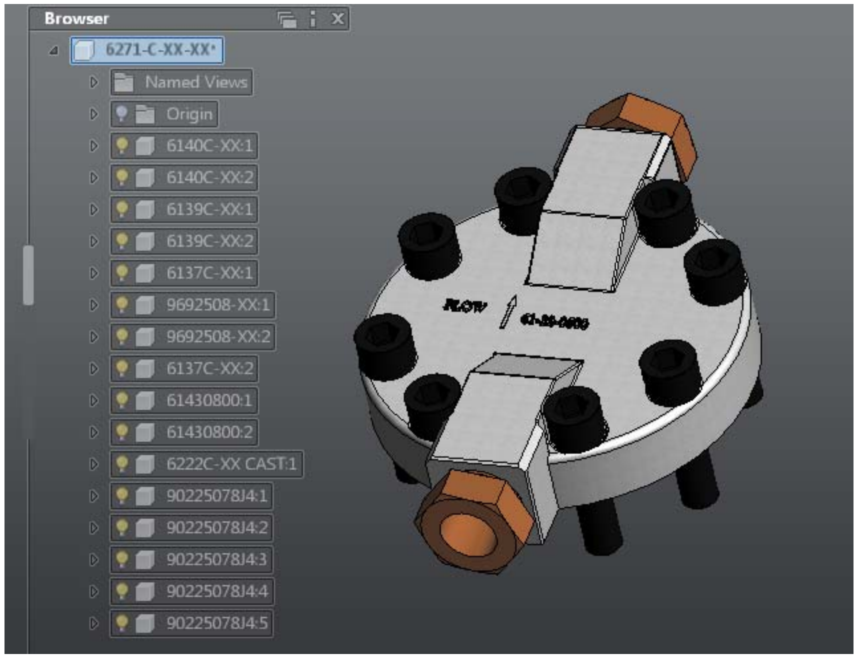
All work geometry commands work with preselected geometry or geometry can be selected after the command is activated. Once the inputs are satisfied, the geometry is created and the command is terminated. It is not necessary to terminate the command.

Working with Multiple Components

Some designs need to be able to 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 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, in the TP1 and TP2 releases, you manage all the components in your design in a single Fusion DWG file. Unlike some other CAD products, Fusion does not require the creation of separate files on disk to use components.

Using components, you can easily and accurately create designs such as this:



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.

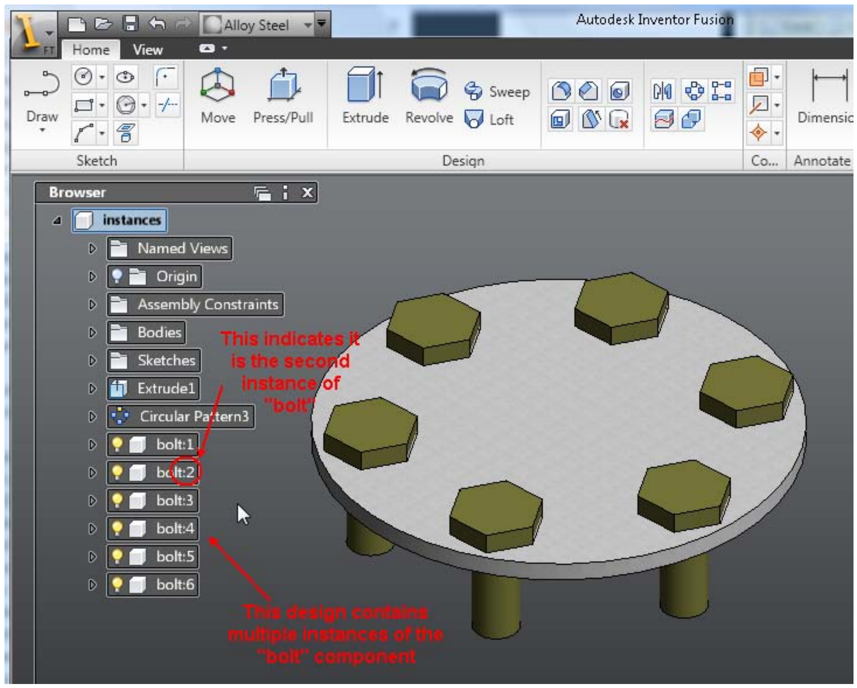
Every Fusion design has a single root component. The root component has the same name as the document itself, and is the top node in the browser

A Fusion component can also contain:

- Bodies: zero or more solid bodies
- Sketches: zero or more 2D sketches
- Work Geometry: Work planes, work axes, work points
- Features: see Features on page 177 for a description of what a Fusion feature is

Components and Component Instances

Inventor Fusion supports the reuse of components in your designs. Any component but the root component may be used multiple times in your design. Each version of the component refers to the same component geometry, and is called a *component instance*. Component instances are indicated by the instance number in the browser:



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

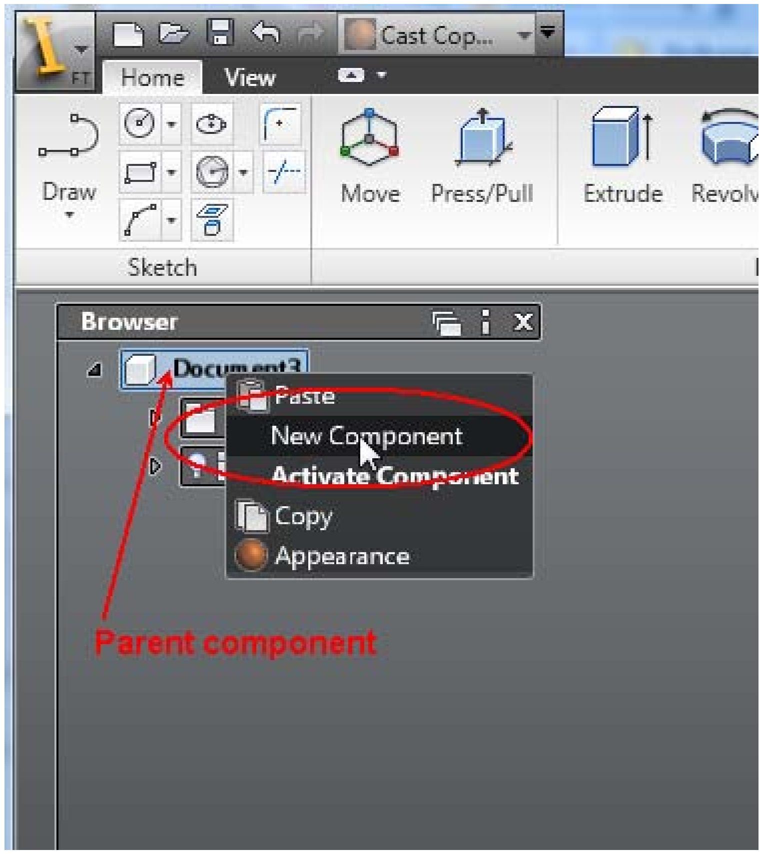
Creating Components

There are several methods in Inventor Fusion to create components, and populate them with geometry

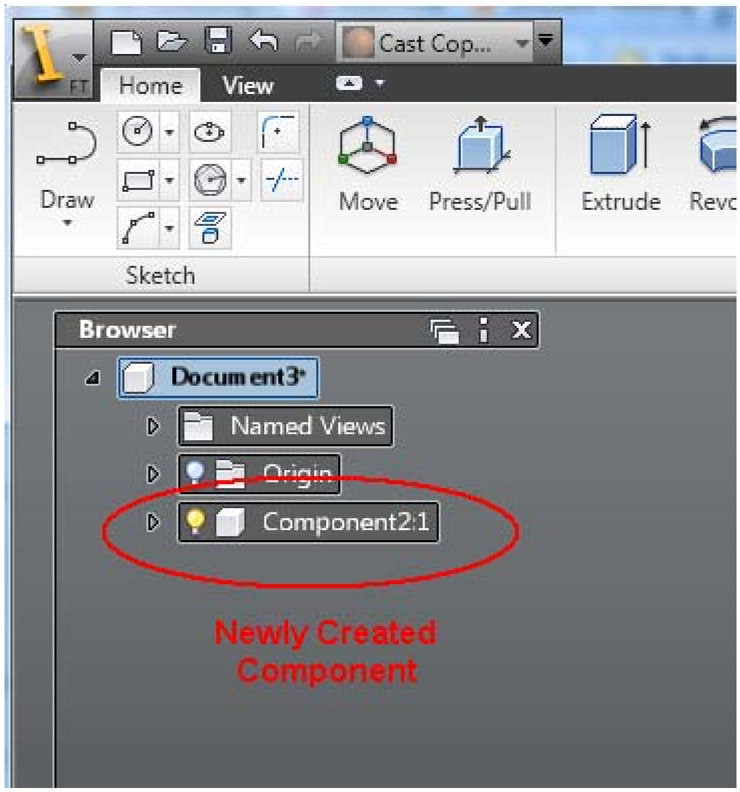
Creating an empty new component

The New Component command creates a new, empty component that is a child of the selected component (which may be the root component). This command is accessed by clicking the right mouse button when the cursor is over a component browser node, which will bring up the context menu.

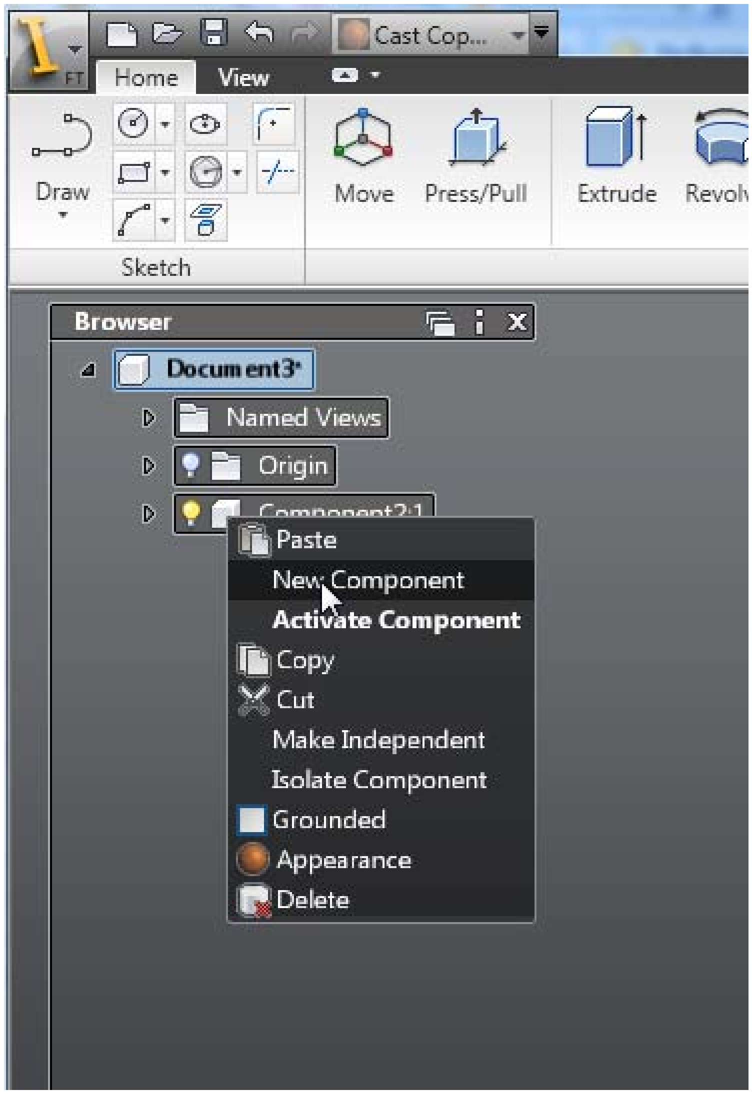
For instance, choosing New Component while the root component (the document) is selected



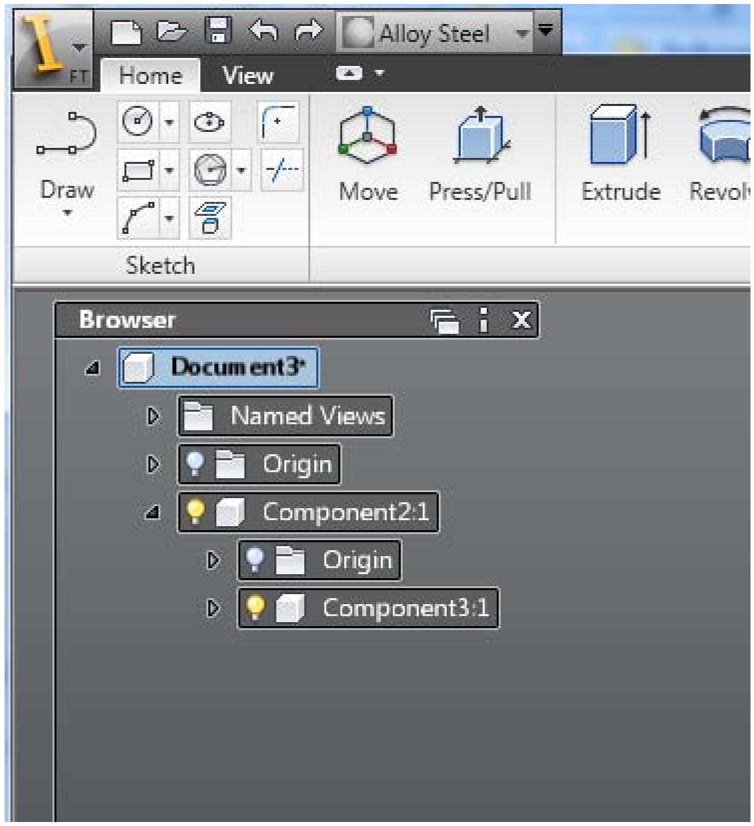
Will create a new component which is a child of the root component



Further, if Component2 is selected, and the New Component command is chosen:

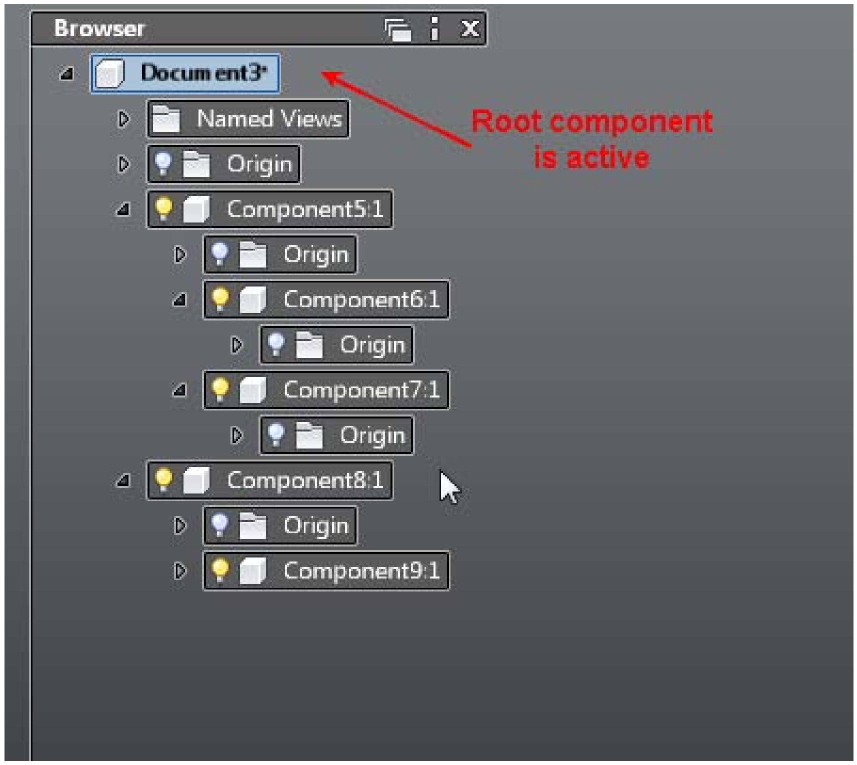


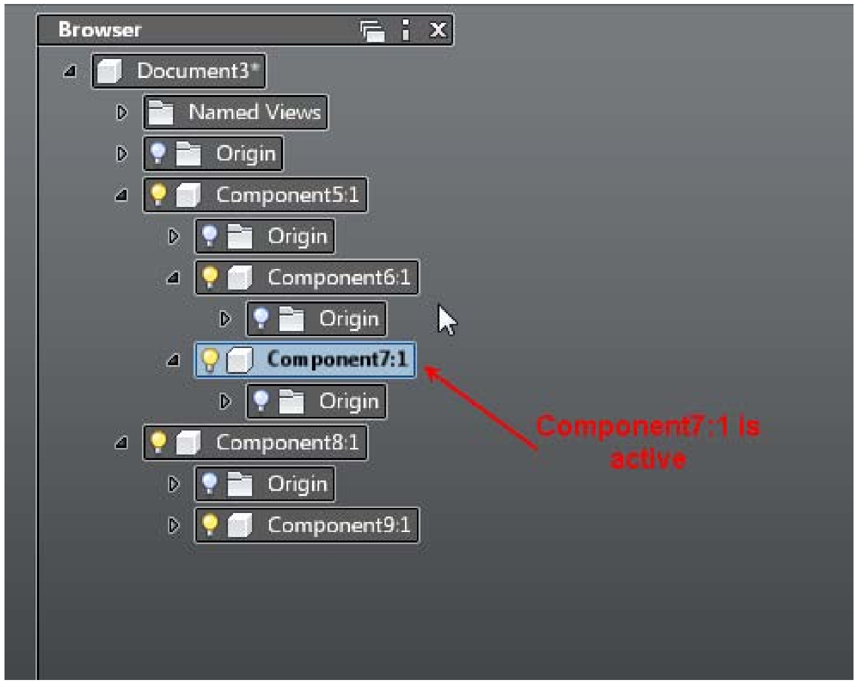
A child component of that component is created:



Activating a component

Inventor Fusion always has an active component. The active component is where newly-created objects go. This includes body geometry (for example, if new features are created), work geometry, and new sketches. The active component is indicated by a blue highlight:





There are two ways to activate a component: Double-click the component's browser node, and using the Activate Component command.

Adding solid geometry to the active component

There are two main ways to add geometry to the body of a component. The first is by creating new features, such as Extrude, Revolve, Loft, and so on. The second is by dragging or copy/pasting bodies from other components.

The primary method to create geometry is using feature commands. See Create a Single Body on page 75 for information about how to use features to create geometry in a component.

Next, you can create geometry in a body by re-using body geometry from other components. You can drag a body from one component to another (which has the effect of moving the body from one to another), or you can select a body in a component, use the Copy Command to make a copy, and then the Paste command to copy that body to a different owning component.

Creating new instances of a component

To create a new instance of a component in your design, use the Copy and Paste commands. select an existing component, choose Copy from the context menu, then select the intended new parent component (which could be the root component), and choose Paste. The result is a new instance of the copied component, which is attached to the cursor, and can be dragged to a new position. A click of the left mouse button will place the new instance.

Position and Constrain Components

This topic covers how to position component instances in 3D, and how to optionally create constraints to keep instances precisely positioned relative to other geometry in your design.

Each component instance, unless constrained, is free to move in 3D. Any child components owned by a component will move with the parent.

The three commands which can be used to move or constrain components are

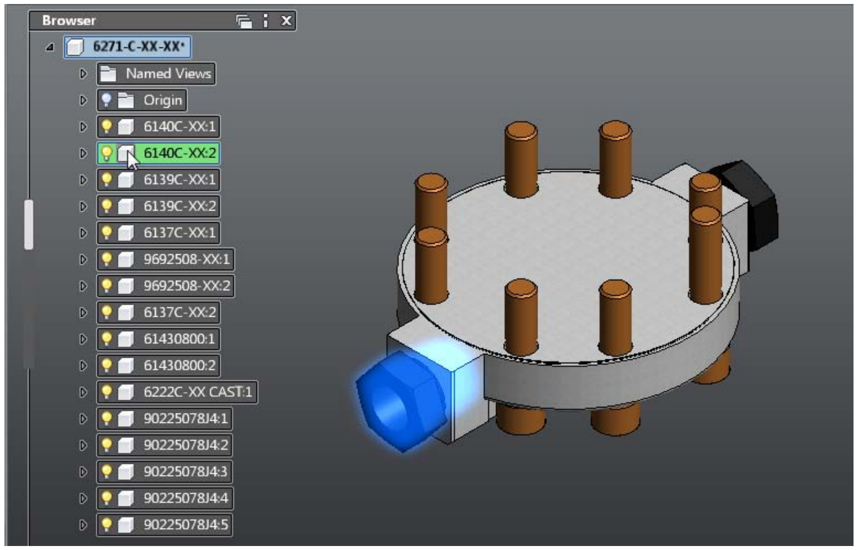
- Move
- Assemble
- Constrain

Subtopic: Moving components without constraining

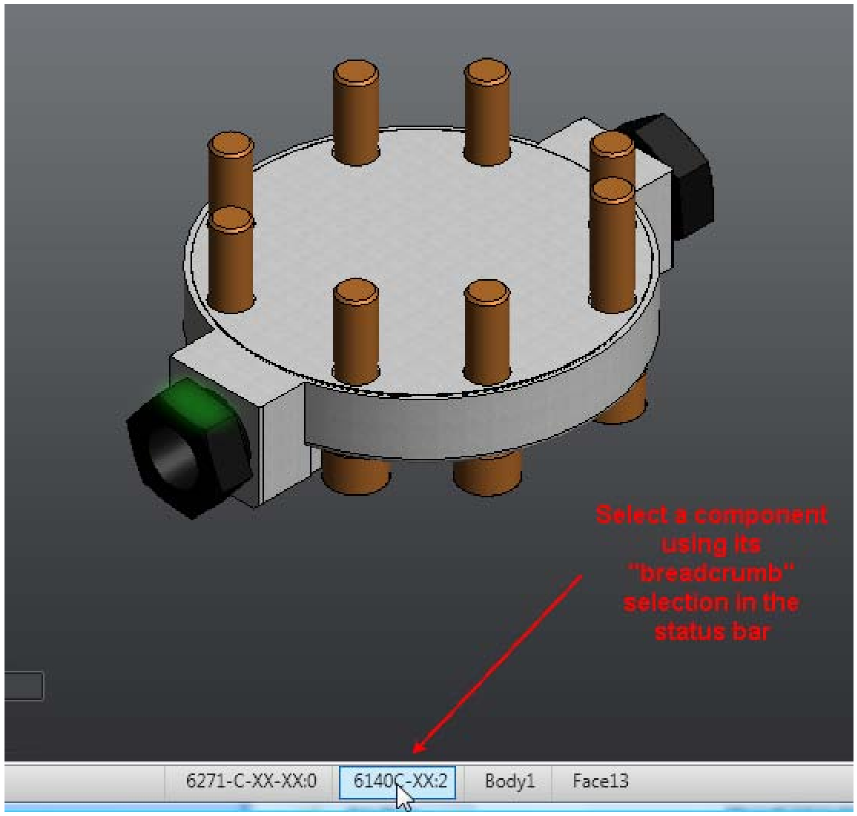
Sometimes, you want to just move a component, and don't particularly care that it stays in its new position if other components move. If so, you can use Move or Assemble to translate or rotate components.

To use Move to transform a component, start Move, and then select the entire component (remember that Move can also be used to edit body geometry by moving one or more faces). The easiest way to select a component is to use the browser or the *selection breadcrumbs* in the Fusion status bar.

Selecting a component in the browser:



Selecting a component using the status bar selection breadcrumbs:



Once the component is selected, use the Move command's manipulators to translate or rotate a component. Note that if other components have constraints to the component being moved, they will also move to satisfy the constraints.

If you want to position a component relative to other components, you can use the Assemble command. The Assemble command can be used with or without constraints. In this case, we don't want to create constraints, but Assemble allows us to easily snap together component geometries, even without constraints. Assemble is a highly interactive command, that allows dynamic dragging of the selected component, and infers other geometries to snap to, as the component is dragged. You can also to perform this same snapping in a more directed way, by explicitly picking the geometries to snap. The following video shows both styles of interaction;

Constraining components

In some design workflows, it is desirable to create persistent relationships between components, to assure that components stay together when one is moved. For instance, if your design contains a bolt/nut/washer assembly, you might want to make sure that these individual components stay together.

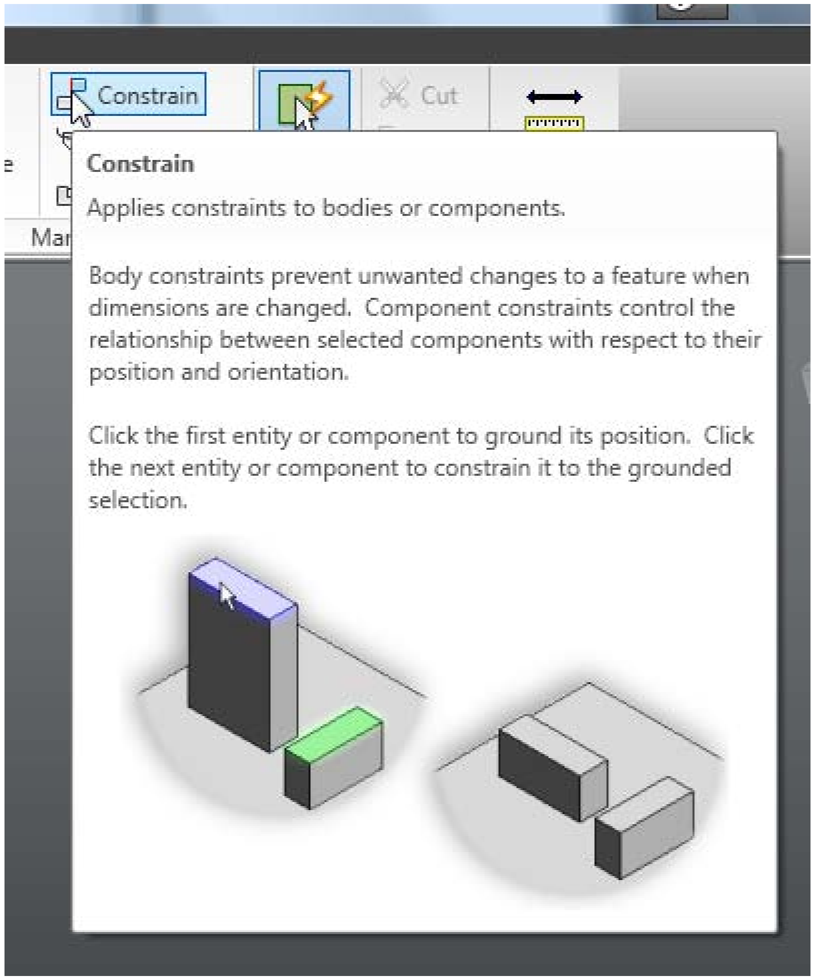
You can create *constraints* between geometries on components. A constraint is a relationship between geometry on two different component Some examples of constraints are:

- Align: aligns the two geometries. This could be making two plane coplanar, or aligning the axes of two cylinders
- Tangent: forces the two geometries to be tangent to each other - for instance a plane and a cylinder, or two cylinders can be made tangent
- Center: this is a special constraint that is used for operations like inserting a bolt into a hole. It is a combination of an Align between two cylinders, and an Align between two planes
- Angle: specify the angle between the geometries - for instance 2 plane can be at a specified angle

There are two main commands for creating constraints:

- The **Constrain** command
- The **Assemble** command

The Constrain command



This command serves two purposes:

- Creating constraints between components. This topic will discuss this usage of Constrain
- Creating body constraints. A body constraint is similar to a component constraint, except that it is used, not to position components, but to change the geometry of a single body in a component. See Dimensions and Body Constraints on page 184 for more information about this usage of the Constrain command

The two usages of Constrain are controlled by the Type input on the ribbon dialog:

In general, when creating component constraints, you select two geometries and specify the Constraint Type. The first geometry selected will be the component that will move as a result of the constraint, while the second will generally remain fixed. Some Constraint Types support the ability to flip the solution (for example, from Mate to Flush when creating an Align between planes). Other constraints (for example Align) allow you to optionally specify an offset value. Still others (for example Angle) usually require a value. Clicking OK or beginning a new command commits the command.

The Assemble command

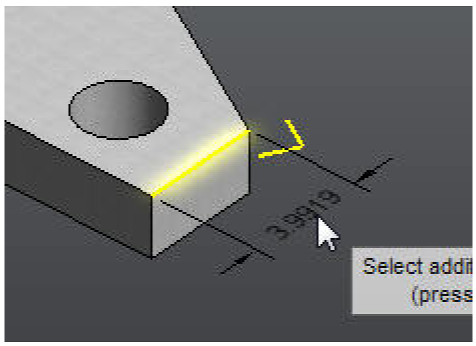
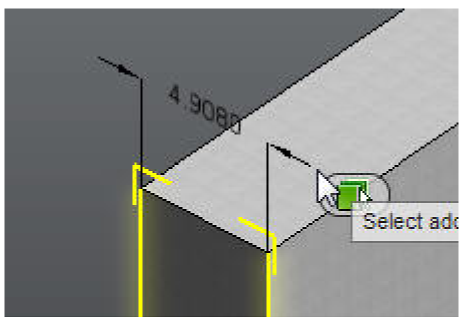
This command, as described earlier in this topic, can be used to position components. However, if the Create Constraints flag is enabled, the Assemble command will also create persistent component constraints. The interaction is identical with or without Create Constraints enabled.

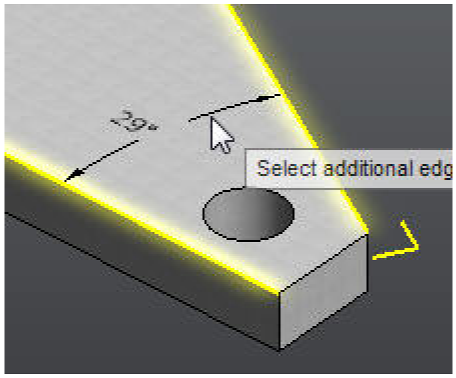
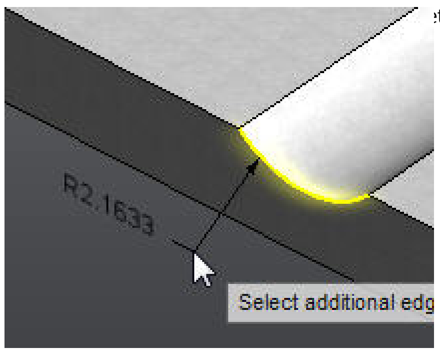
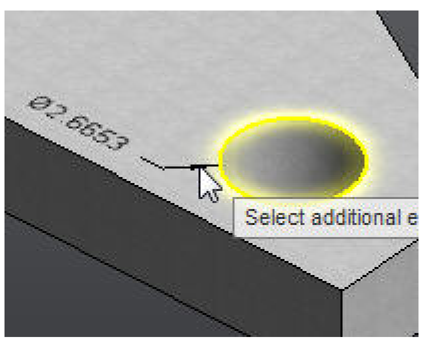
Dimensions as Annotations

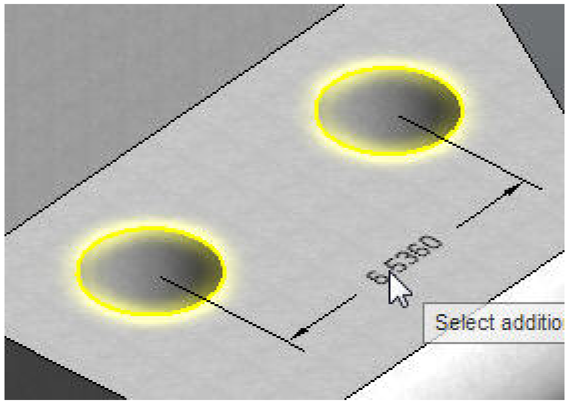
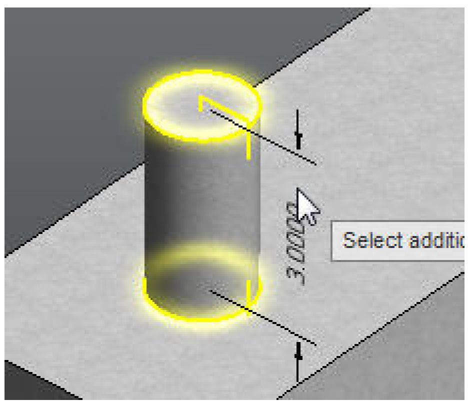
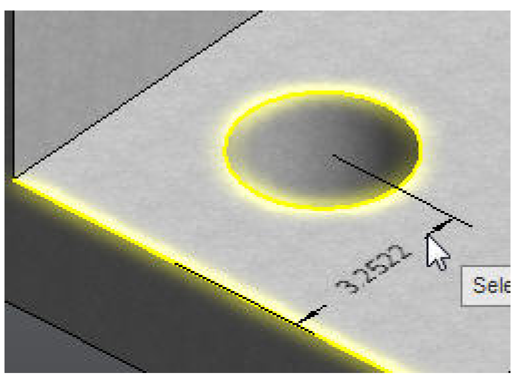
Dimensions are used for two purposes in Inventor Fusion: (1) as annotations, and (2) as drivers of the model. When dimensions are used as annotations, they are driven by the model and the value cannot be modified directly. This section discusses dimensions used as annotations. When dimensions are used as drivers, you can double-click the dimension and change its value. The model is then updated in some way to meet the new constraint imposed by the dimension's value. For more information about driving dimensions, see Dimensions and Body Constraints on page 184.

Types of Dimensions

You can create annotation dimensions using any of the following:

Selection	Example	Dimension type	Notes
One linear edge			
Two parallel linear edges			Once created, no visual distinction from one-edge dimension

Two linear edges, not parallel, but in the same plane or parallel planes		Use Tab key to swap to Length dimension when edges are on different faces
One circular edge	 	

Two circular edges, non-concentric	
Two circular edges, concentric and not coplanar	
One linear edge and one circular edge	

Other Details

Dimension precision can be controlled from the context menu. All dimensions use the same precision. (Angular vs. length dimensions have independent precision.)

Dimensions are always created on an *annotation plane*.

- Annotation planes are created automatically as necessary. Existing annotation planes are shared/reused if possible.
- When creating a dimension, you can use the Tab key to select the appropriate annotation plane from the available ones.
- Annotation planes are usually associative to a face or set of edges.
- Annotation planes are displayed/controlled from the Browser.
- Dimensions belong to their annotation plane, which implies:
  - Annotation planes can be turned on/off, which affects all of the owned dimensions.
  - Annotation planes can be deleted, which also deletes all the owned dimensions.
- Annotation planes will highlight by showing four corners.

Named views respect the visibility-state of annotation planes. This means you can turn off an annotation plane, and create/update a named view, and whenever you return to the named view, the saved state of the annotation planes is restored. This is convenient when working with larger sets of dimensions.

Dimensions and annotation planes are always created on the root component. Dimensions can refer to edges of any component in the assembly hierarchy. Inter-component dimensions can never be locked. If you drag a body from one component to another, Inventor Fusion deletes any dimensions referring to that body.

Limitations

Current limitations:

- There are no other types of annotations (no notes, no GD&T symbols, and so on).
- There are no provisions for creating 2D layouts (no sheets, no title blocks, and so on.

■ There are no formatting or style mechanisms, other than the precision.

Note: The context menu always uses the active selection in preference to the merely highlighted. So if you have a dimension selected, and then hover over a different non-selected dimension, and click right, it will tell you the lock-status of the *selected* dimension, and *not* the dimension under the mouse.

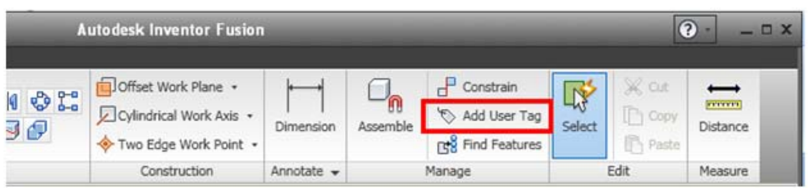
User Tags

Tags can be added to any face or body. These tags can then be used for searching/locating particular entities.

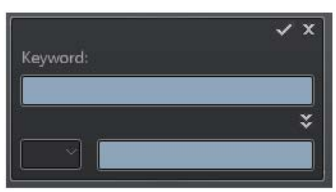
These user-defined tags are called User Tags. A user tag consists of a tag name and an optional tag value. Inventor Fusion also supports Feature Tags. Feature Tags are predefined properties of features such as Fillet Radius or Hole Dia.

Add User Tag

The Add User Tag command sets the tag name and value for the selected entities. The command is accessed in the Ribbon.



Use the user tag dialog box to assign user tags. The dialog box can be expanded using the  button.



To assign a user tag, start the command, select the objects, then enter the tag name and value. You can select multiple objects to assign the same tag to the objects.

There are two ways to set the tag name and value.

■ Enter the tag name and value separated by a comma in the Keyword field.

- Expand the dialog box, select the tag name in the drop-down menu, and enter the tag value in the field. To enter a new tag name, select New.

You must click OK to apply the tag. You can also click Cancel to terminate the command without applying the tag.

Search User Tags

You can search user tags and predefined feature tags. Searches are created and accessed in the Favorites area of the browser. Create a search using the context menu on the My Favorites node in the browser.

When a search is created, the search dialog is displayed. Enter the tag or feature name and value then click OK to create the search.

User Tag Search

- This will search tags added using the Add User Tag command.
- You can query any keyword (tag name, value) and the search result is placed under a new search folder browser node.

Feature Tag Search

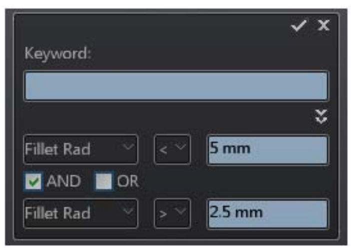
- This will search predefined feature tags.

- You can enter any feature type to search for all faces of that feature type.

Advanced Tag Search

- Click the expand button  to display advanced search options.
- You can set search criteria by defining up to two conditions.
- There is a logical operator 

 between the two options.
- The tag name drop-down list contains recently used tag names and a predefined list of feature tags names (Fillet Rad, Hole Dia, Hole Type, Hole Depth, CB Dia, CB Depth, CS Dia, CS Angle, Extrude Dist).
- You can set a comparison for tag values (, , =, or !=).



- The search results node is added to the My Favorites folder in the browser. This folder is updated when you click or expand the folder node. You cannot create a search folder that is empty but a search folder where the query once returned values but now no longer does is allowed.

Import Data

Import Data

Part and assembly files from other CAD systems can be imported for use in Autodesk Inventor Fusion. The import operation does not maintain associativity with the original file. As a result, changes to the original file after the import operation do not affect the imported part or assembly. Likewise, changes to the imported part or assembly do not affect the original file.

When assemblies are imported, Autodesk Inventor Fusion attempts to preserve the assembly and subassembly structure of the original file. However, for SAT files, the subassembly structure is decomposed so that all subassembly parts become components of the top-level assembly.

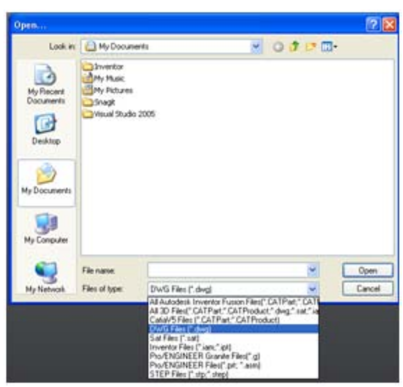
The import process creates base features in Autodesk Inventor Fusion representative of the geometry and topology in the source file. You can use Autodesk Inventor Fusion commands to adjust the base features and add new features to the Autodesk Inventor Fusion feature tree. You cannot modify the original definition of the base features.

Translate files into Autodesk Inventor Fusion data

You can open or import part and assembly files from other CAD systems.

In Autodesk Inventor Fusion, do the following to translate a file:

1. Click the Open button.
2. In the Open dialog box, select the Files of type drop down then select the file type.



3 Browse to and select the file to import.

4 Click Open to import the file.

Importing CATIA V5 files

Open and change models created in CATIA V5 (versions R6 - R19). Autodesk Inventor Fusion translates assembly and part files, solids, multi-solids, surfaces, and more. After the import operation is complete, you have a base feature or features which match the geometry and topology of the original file. Use Autodesk Inventor Fusion commands to adjust the base features and add new features to the feature tree.

These types of CATIA V5 files can be imported:

- *.CATPart (part)
- *.CATProduct (assembly)

After changing the file, you can continue to open it in Autodesk Inventor Fusion.

Importing Pro/ENGINEER files

Open and change models created in Pro/ENGINEER. Autodesk Inventor Fusion translates assembly and part files, solids, multi-solids. After the import operation is complete, you have a base feature or features which match the geometry and topology of the original file. Use Autodesk Inventor Fusion commands to adjust the base features and add new features to the feature tree.

These types of Pro/ENGINEER files can be imported:

- *.prt* (part) (up to version 4.0)
- *.asm* (assembly) (up to version 4.0)
- *.g (Granite) (up to version 5.0)

After changing the file, you can continue to open it in Autodesk Inventor Fusion.

Importing STEP files

You can import a STEP file (versions AP214 and AP203E2). The solid body is saved in an Autodesk Inventor Fusion file, and no links are maintained to the original file.

If an imported STEP file contains one part, it produces an Autodesk Inventor Fusion file with a single body. If it contains an assembly, it produces an Autodesk Inventor Fusion file with multiple components.

Importing SAT files

You can import a SAT file (versions 4.0 - 7.0). The solid body is saved in an Autodesk Inventor Fusion file, and no links are maintained to the original file.

If an imported SAT file contains a single body, it produces an Autodesk Inventor Fusion part file with a single body. If it contains multiple bodies, it produces an Autodesk Inventor Fusion File with multiple components.

Inventor Data

You can import an Autodesk Inventor file. The solid body is saved in an Autodesk Inventor Fusion file. Excluded, Suppressed, and Invisible objects will become visible when importing IPT and IAM files.

These types of Inventor files can be imported:

- *.ipt (part) (up to Inventor 2010)
- *.iam (assembly) (up to Inventor 2010)

Changes made to a part file within Inventor Fusion can be integrated back to the original part using Track Changes in Inventor. Track Changes is an add-in for Inventor which will allow users to identify and manage any changes made to a part in Inventor Fusion. Track Changes is intended to map non-parametric Inventor Fusion edits to editable, parametric Inventor features.

This functionality allows users to edit an Inventor model within Inventor Fusion (free from the rigid structure of history-based modeling), open the part within Inventor, and accept or reject Inventor Fusion model edits through a rich interface provided by the add-in.

This workflow is intended to assist users in making rapid & unrestrained model edits in Inventor Fusion, and subsequently allowing those model edits to be realized within the Inventor model.

For more information see the Track Changes Help in Inventor.

Export Data

You can export Autodesk Inventor Fusion parts and assemblies to other CAD system formats. The export operation does not maintain associativity with the Autodesk Inventor Fusion file. As a result, changes to the Autodesk Inventor Fusion file after the export operation do not affect the exported part or assembly. Likewise, changes to the exported part or assembly do not affect the Autodesk Inventor Fusion file.

The objective of the Autodesk Inventor Fusion export operation, for part and assembly files, is to create files as if they were created in the native CAD system format. As a result, use of the exported files in the native CAD system is typically seamless.

When you save Inventor Fusion data it is saved to a 2010 format DWG file. You can choose to save as an AutoCAD 2009 format DWG file. Inventor Fusion model intelligence will not be preserved in the AutoCAD 2009 DWG file. The model is exported as AutoCAD solids.

Export Data to Other Formats

You can also export solid data in the following formats:

- Catia V5 parts and assemblies
- Pro/Engineer parts and assemblies
- SETP files
- Autodesk ShapeManager SAT files

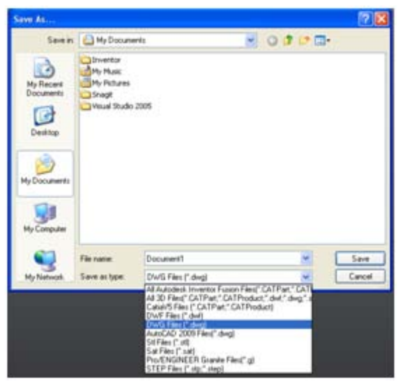
You can export viewing and faceted data in the following formats:

- STL files
- DWF Files

To save Autodesk Inventor Fusion files to other formats:

1 Select  Save As.

2 Click the arrow next to the Save as type box, and then select the appropriate file type from the list.



3 Enter the file name in the box. If you do not enter a file name extension, the file is saved with the extension shown for the selected file type.

4 Click Save to export the Autodesk Inventor Fusion data to the file.

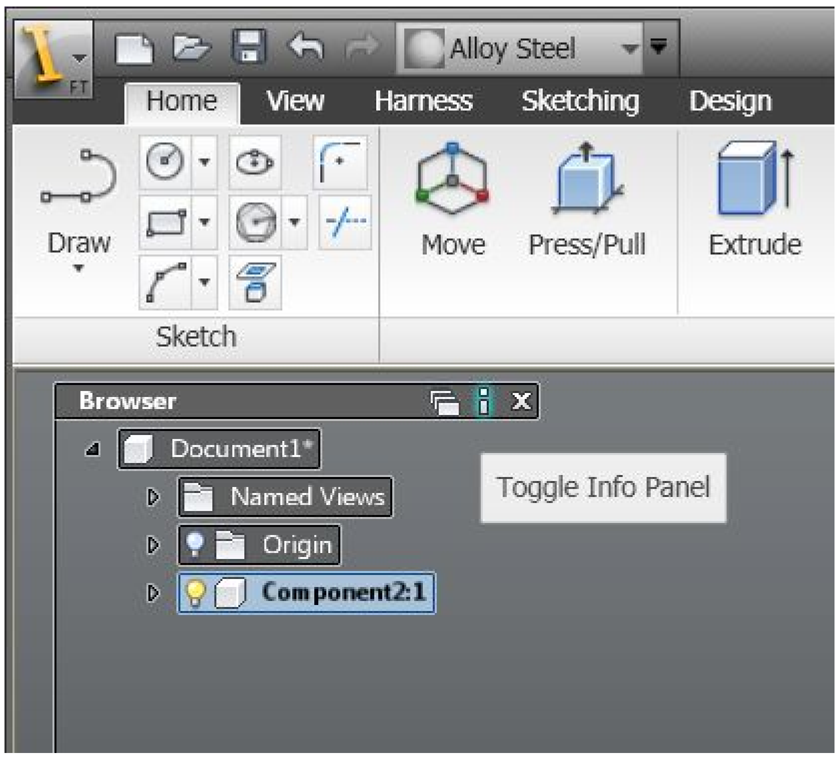
Materials and Model Appearance

Physical Materials

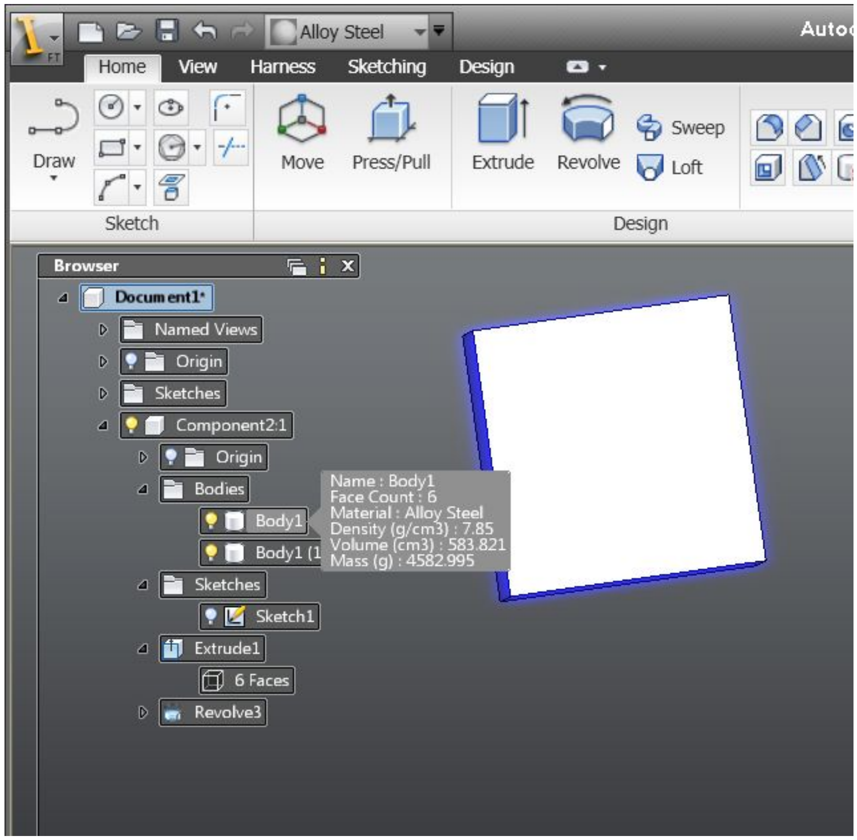
When a body or component is created in Inventor Fusion, it will be assigned a default Physical Material (henceforth called Material). It is not possible to create a body or component that has no Material. For the same reason, any body or component imported into Fusion that does not have a material associated with it is assigned a default material. Faces and other entities like Workplanes, Sketch profiles and so on do not have Materials. Inventor Fusion supports all the same materials as Autodesk Inventor. The default Material in Fusion is “Alloy Steel”.

Note: When importing a file from inventor, the part is assigned the Material present in its iProperties. If it does not have an iProperty of type Material, it will be assigned the default Alloy Steel Material.

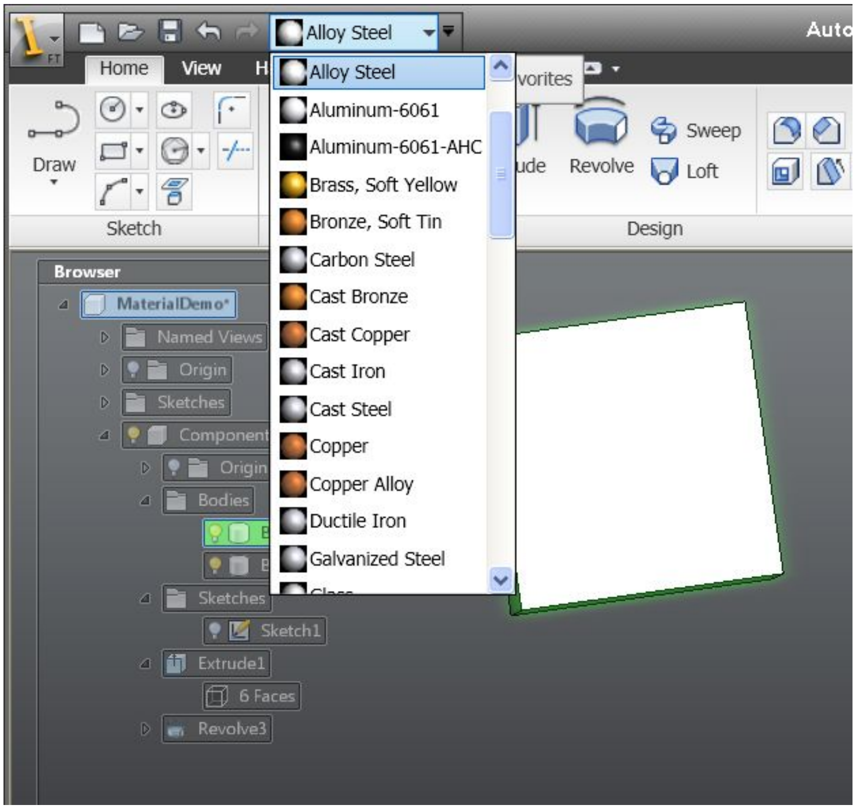
A body with a Material has certain physical properties like density, volume and mass depending on the material it is made of. To see this information, click the Toggle Info Panel button in the Browser Panel:



Once the Info Panel is turned on, you can obtain physical property information about a body or component by moving your cursor on its browser node. The mass and volume of a component is the sum of the mass and volumes of its child bodies and components.



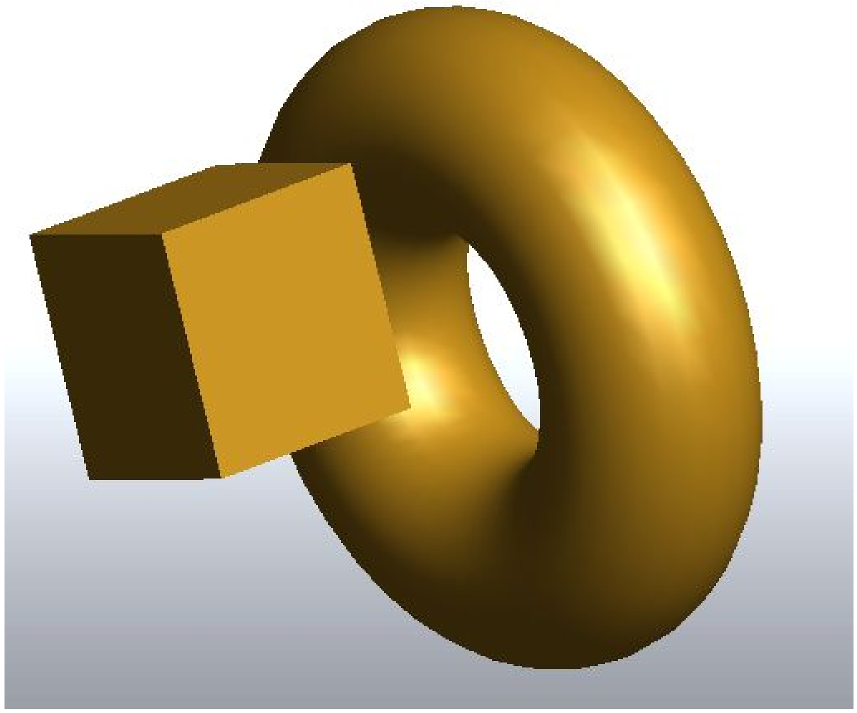
A body or component typically inherits the Material of its immediate parent. You can set the entire document to have a particular material, and any object created under it will have that Material. You can override the material of child objects by selecting them and using the drop-down menu available in the Quick Access commandbar.



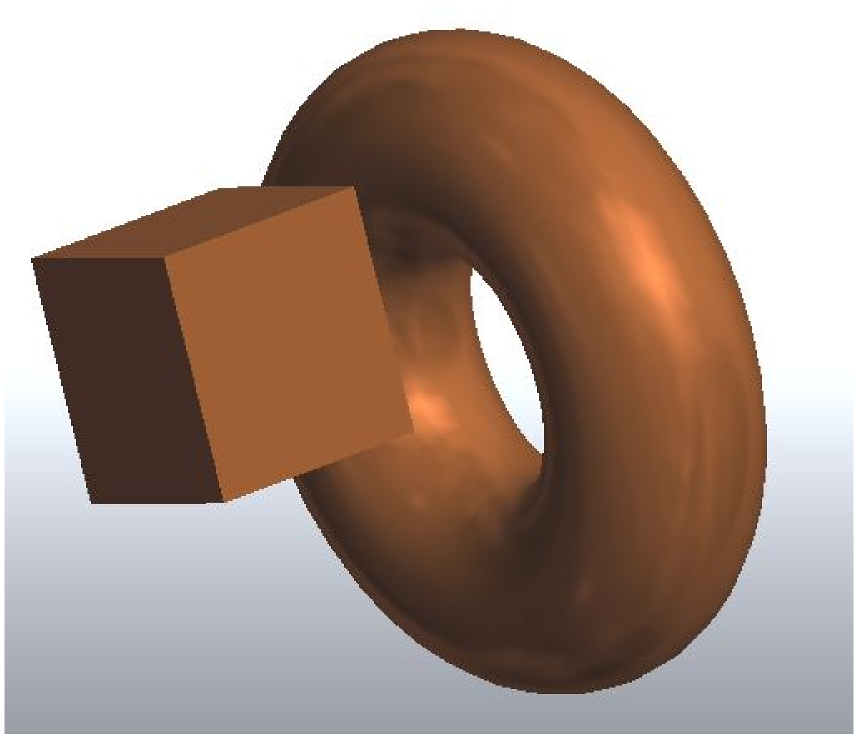
This drop-down menu is only enabled if one of your selected items supports Materials. You can pick any item from the drop-down and it will be applied to all valid selections.

Every Material has a default appearance associated with it. When a component or body gets assigned a Material, it also assumes the appearance of the Material.

Brass



Copper



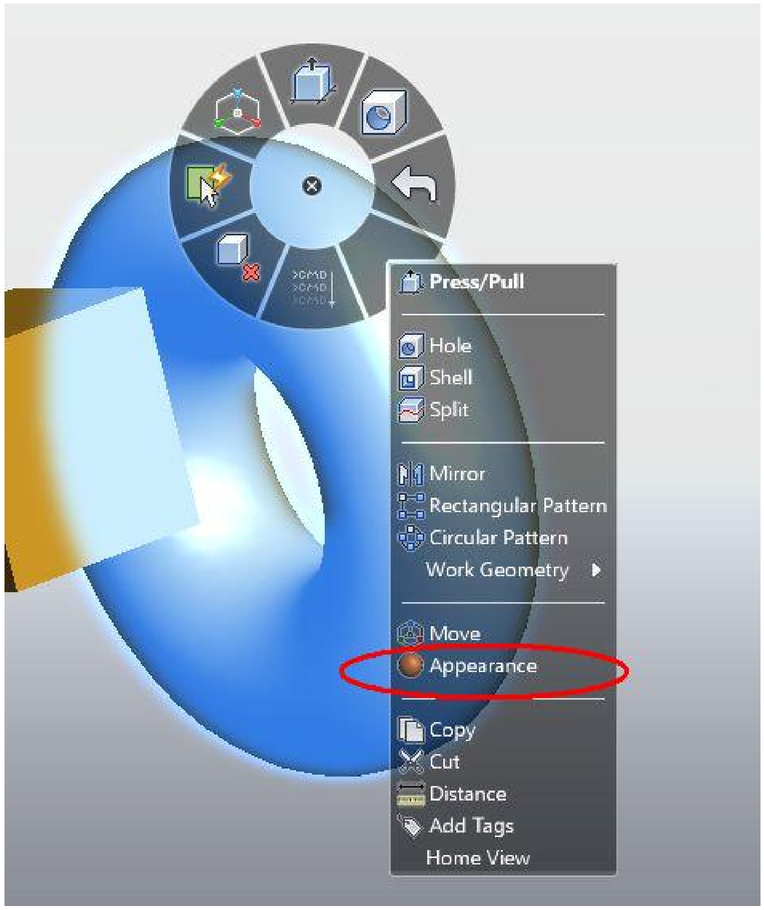
You can change the appearance of a component, body or face using Appearance Overrides on page 222. See the Appearance page for more information.

Appearance

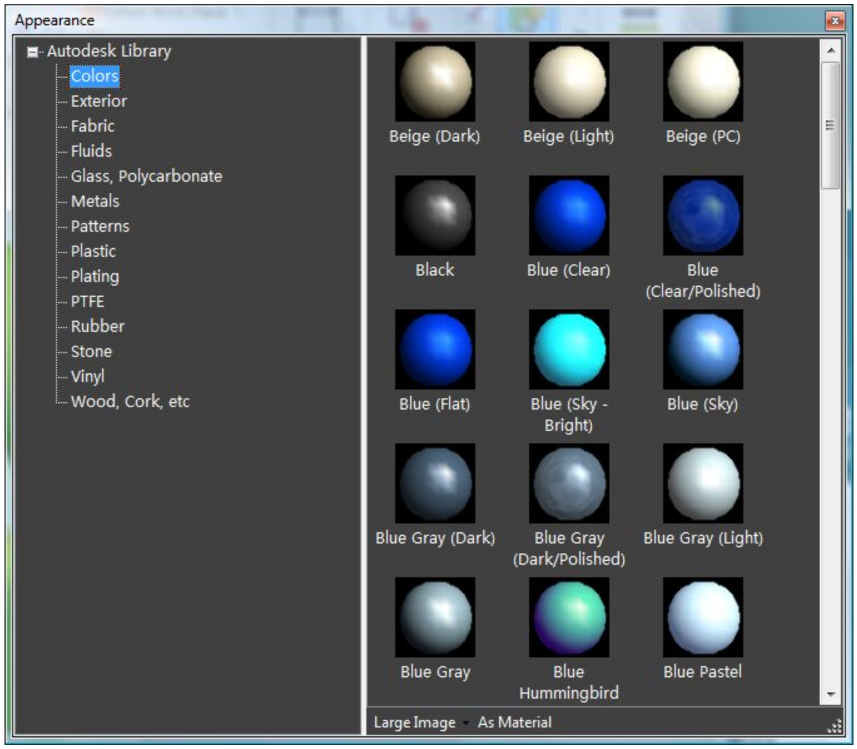
An object in Fusion gets a default appearance based on its Material. This appearance can be overridden at the component, body or face level. Changing the appearance of an object will not affect its physical properties like mass. If your object is made of Copper Material and you change its appearance to Machined Aluminum, it will not get the properties of Aluminum. It will only look like Aluminum.

Note: When importing a file from Inventor, appearance overrides applied in Inventor are not imported presently.

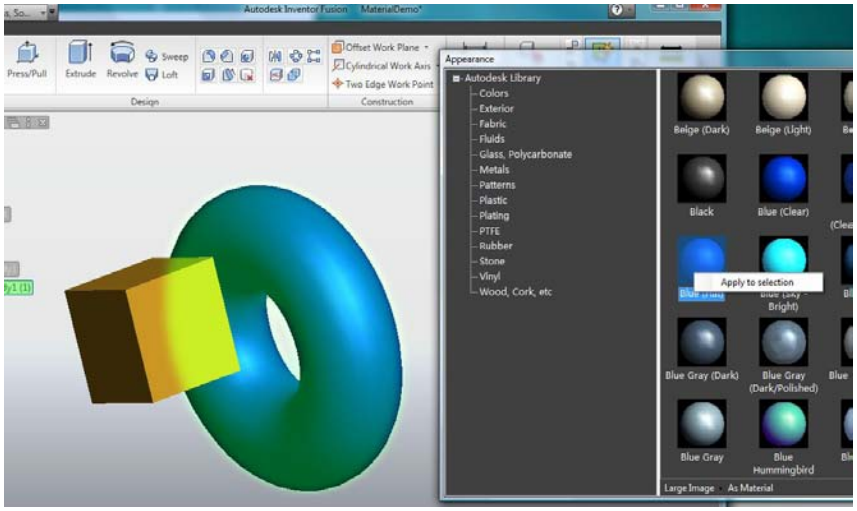
To change the appearance of an object, select it in the canvas or in the browser and right-click. The context menu will have an Appearance item:



Clicking the menu item will open the Appearance Dialog. This is a modeless dialog that shows you all the appearances available in Fusion, categorized in groups. These are the same appearances that would be available to you in Autodesk Inventor.

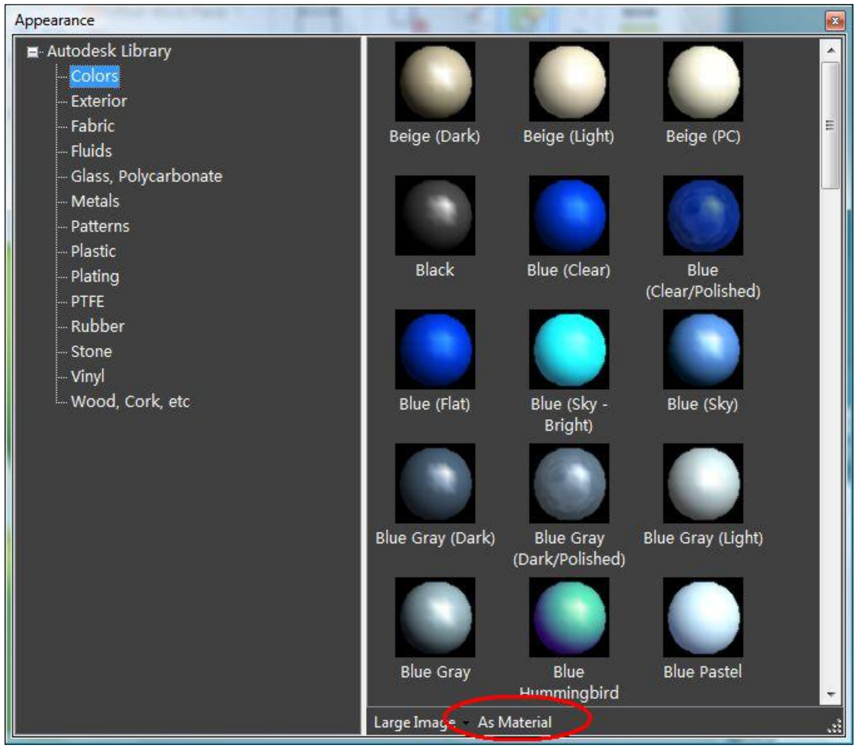


Moving your mouse over any color square will apply an appearance preview on your selected objects. You can double-click any square or right-click and select the "Apply to Selection" option to apply the appearance to your selection.



To remove an appearance override, open the Appearance dialog box and click the As Material button at the approximate center and bottom of the dialog box.

Setting the Material on a component or body will change the appearance of the object. Face appearance overrides are not presently removed when the Material of their parent body or component changes. You can use the “As Material” feature to restore the faces appearance to its parents appearance if you need to.

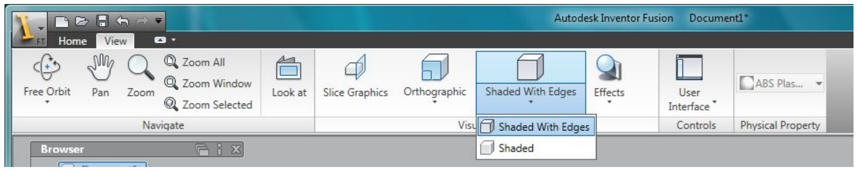


Appearance Override priorities

Appearance overrides applied at a component level have the highest priority and effect the appearance of all their children. The second priority is given to Faces, and the last priority is given to bodies. Hence, if a face has an appearance override, changing the bodys appearance will not remove it, but changing the components appearance will remove it.

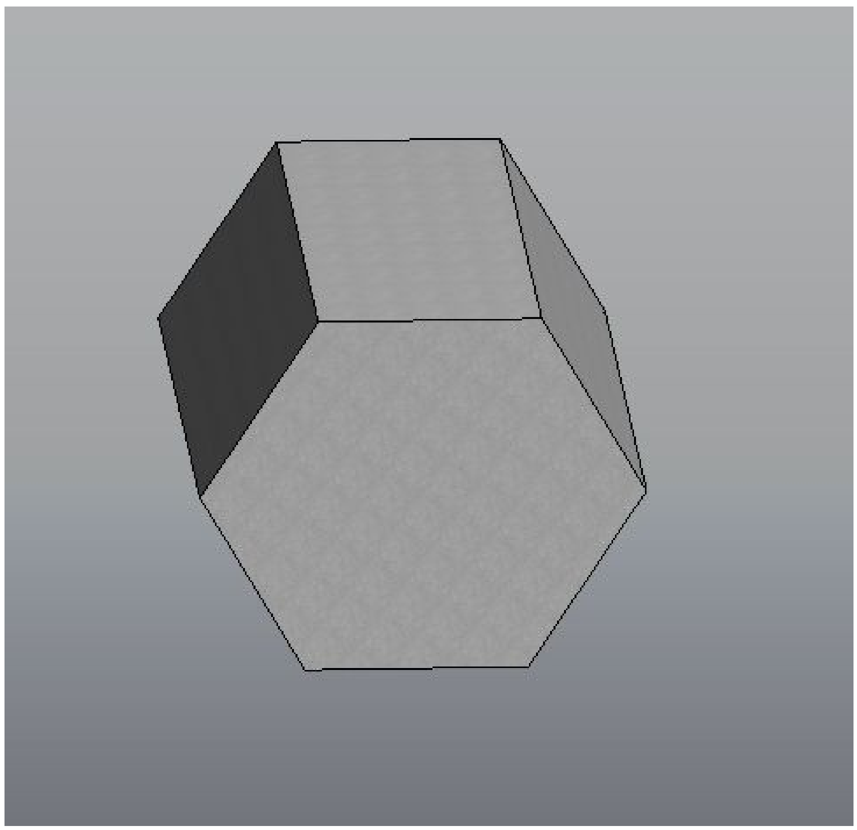
Edge Visibility

You can turn edges on or off in your model by using the drop-down menu in the view ribbon. Edge display is an application setting and your current setting will be restored the next time you start the application.

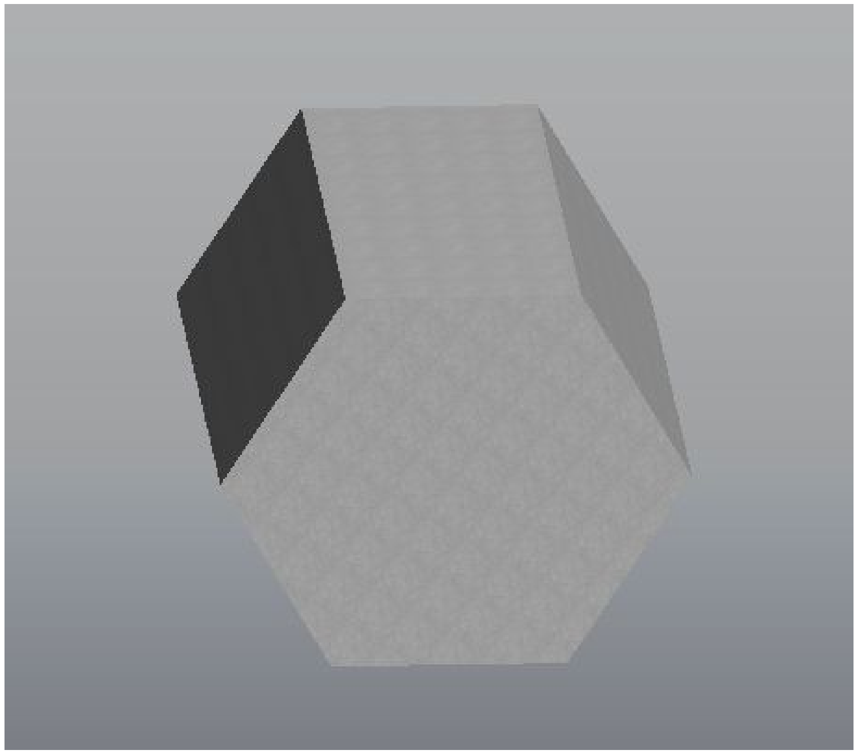


The Shaded With Edges option will draw both faces and edges. The Shaded option will only draw faces. Inventor Fusion does not support a Wireframe (Edges-only) style at present.

Shaded with Edges

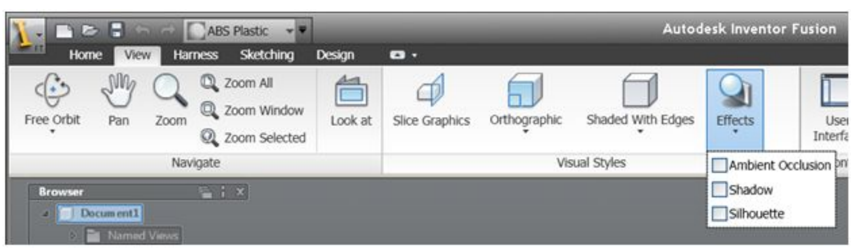


Shaded



Effects

Visual effects are available in the View tab of the Ribbon Control.

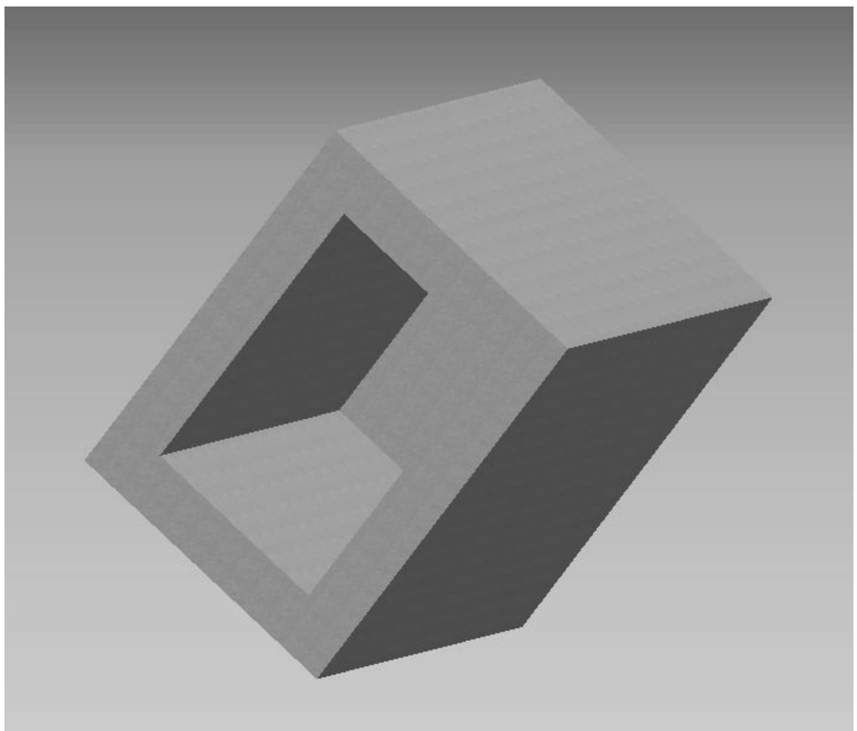


All effects are turned off by default. You can turn any effect on or off by clicking the check box. Effect settings are application-specific and will be saved and restored the next time you start the application. Note that turning any of these visual effects on may slow down performance.

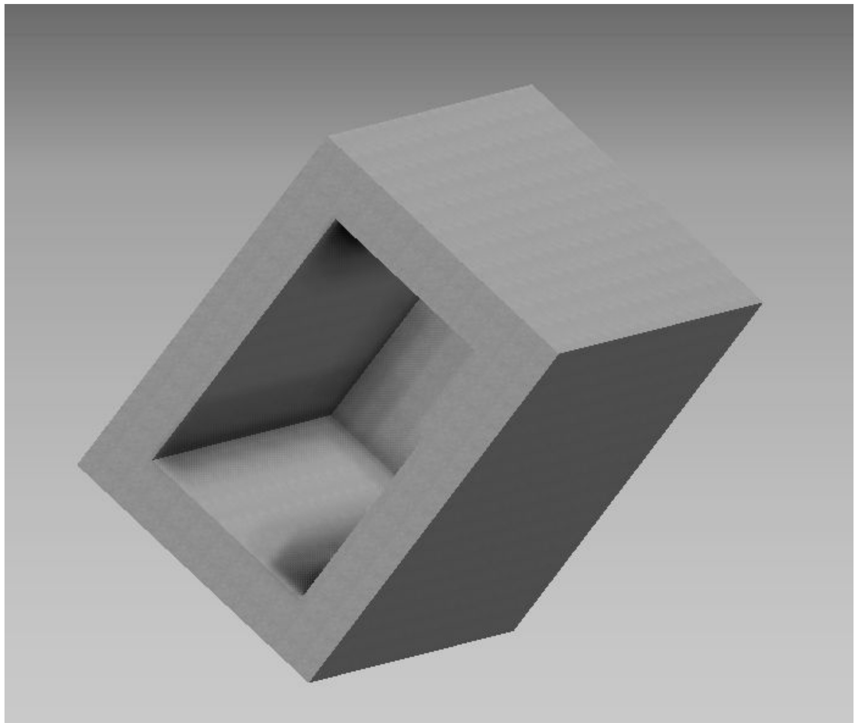
Ambient Occlusion

Ambient occlusion is an effect that takes into account attenuation of light due to occlusion. In the simplest terms, any object area that is obscured by another object would receive less light and would appear darker. Note that this effect is computationally expensive and will result in noticeable performance degradation.

Ambient occlusion off



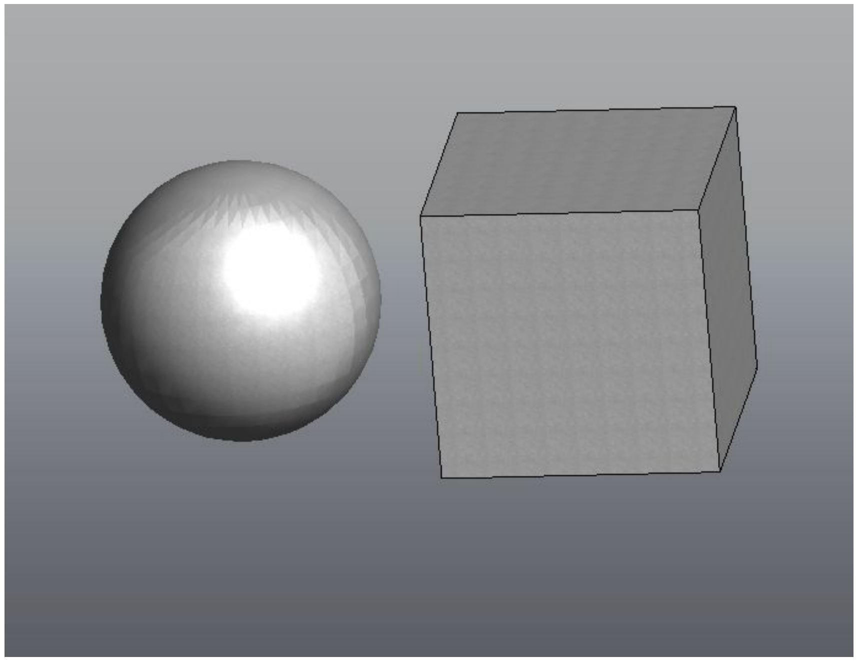
Ambient occlusion on



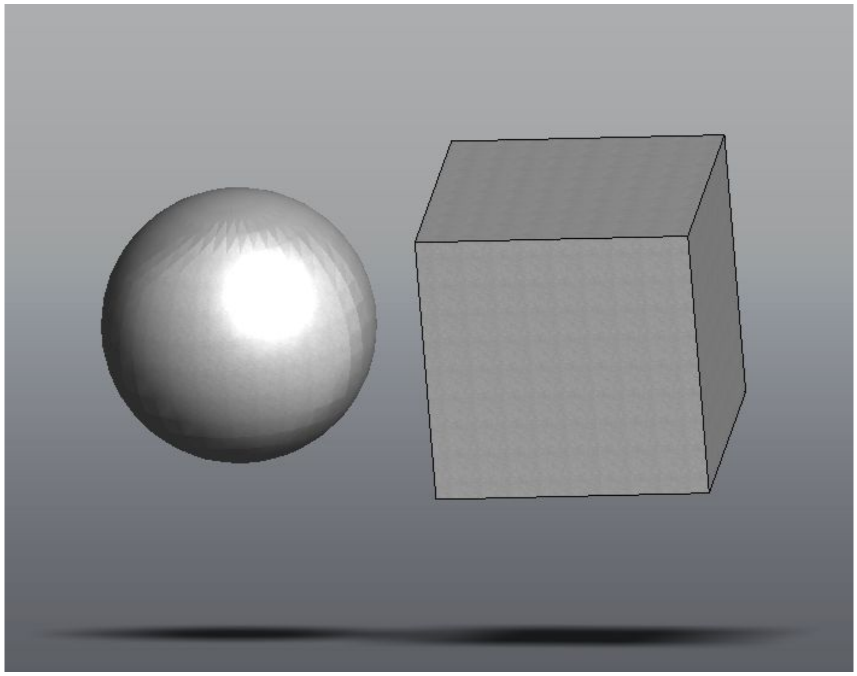
Shadow

This effect casts a shadow of the object on the ground assuming that a light is directly overhead. The ground is not fixed in world space. It does not rotate with the object. It is the floor plane in view space.

Shadow off



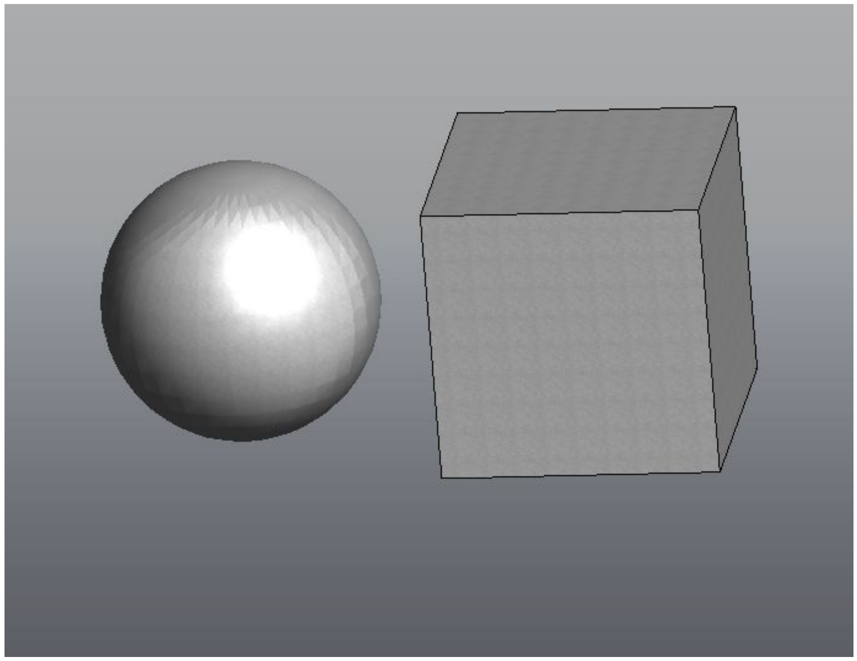
Shadows on



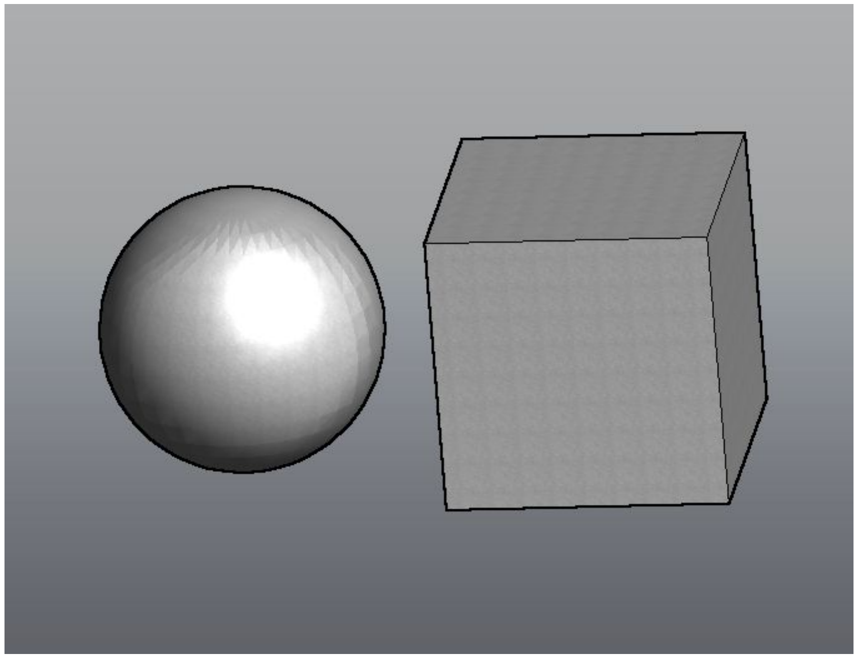
Silhouette

This effect draws a silhouette (outline) on the objects in the scene. This effect is only applicable if your Edge Visibility on page 226 is "Shaded With Edges". If your setting is "Shaded" and edges are not visible, this effect will do nothing. A sphere will also have a circular silhouette, although it has no edges.

Silhouette off



Silhouette on



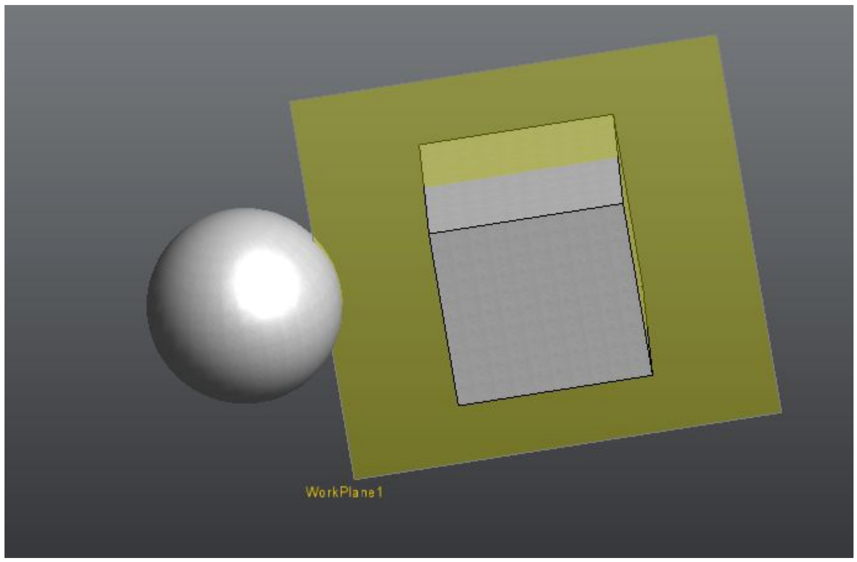
Slice Graphics

Use Slice Graphics to cut away part of the scene to better see other objects in the scene. It can slice the scene along any planar surface (a planar face or workplane). Select the surface to slice along before activating this feature.

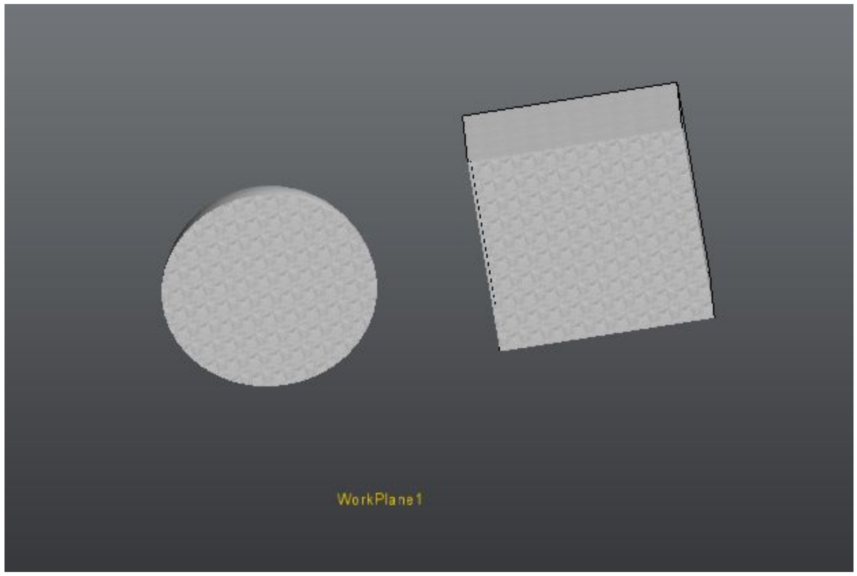
The Slice Graphics feature can be activated or deactivated by clicking a button in the View Tabs Visual styles panel:

It can also be activated or deactivated by pressing the F7 function key. When Slice Graphics is active, it will cut any bodies that intersect the plane. It will also create temporary "caps" to close any holes created by the slicing action, as you can see in the following images.

Before slicing along the workplane



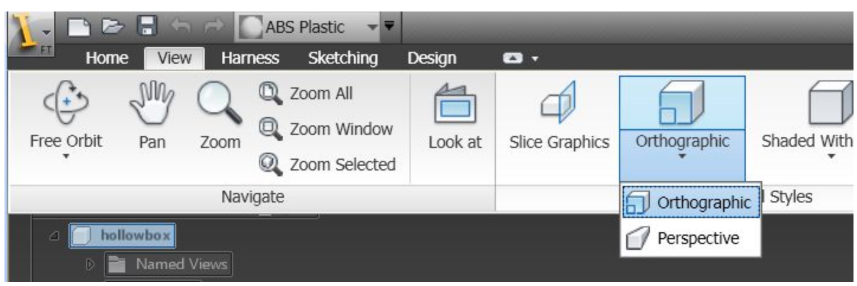
After slicing along the workplane



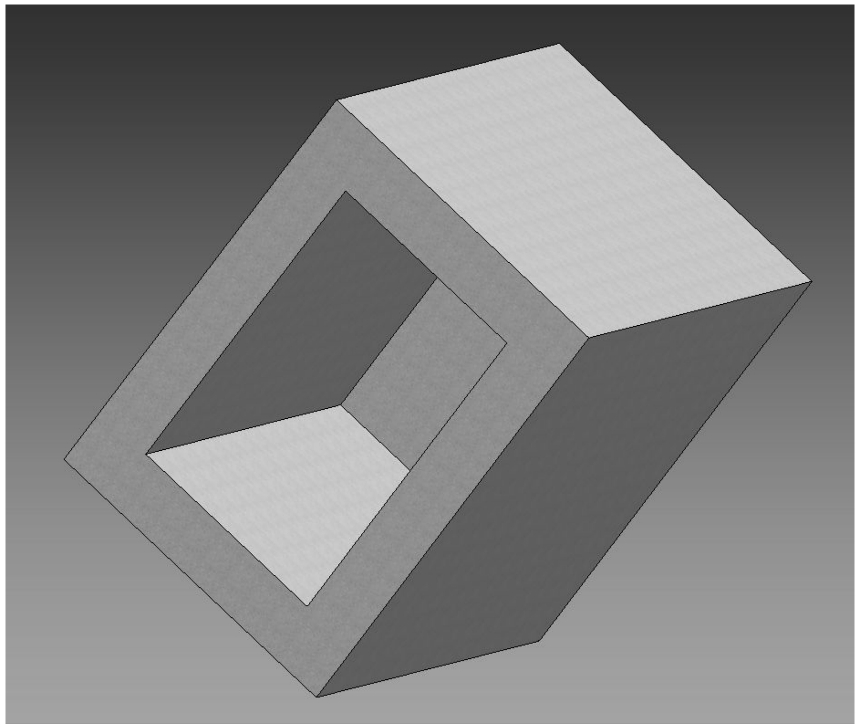
Views of models

Orthographic views

You can set the view to Orthographic using a drop-down list in the View Tabs Visual Styles panel:



In Orthographic Camera mode, a model is displayed so all its points project along parallel lines to their positions on the screen. All same-length parallel edges display as the same length, even when you orient them so one edge is closer to you than the other. In Orthographic Camera mode, a 3D model appears flat and unlike objects observed in the real world.

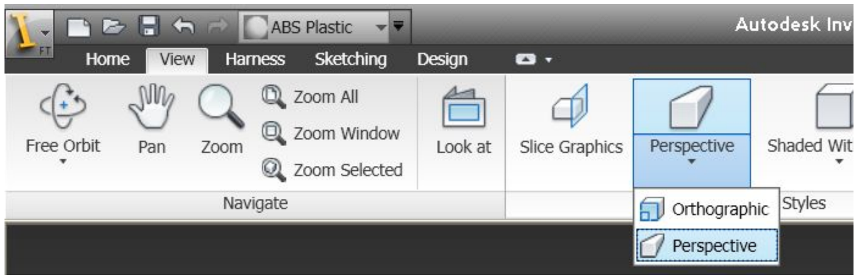


Use Orthographic Camera mode to confirm visually or compare the relative dimensions of entities.

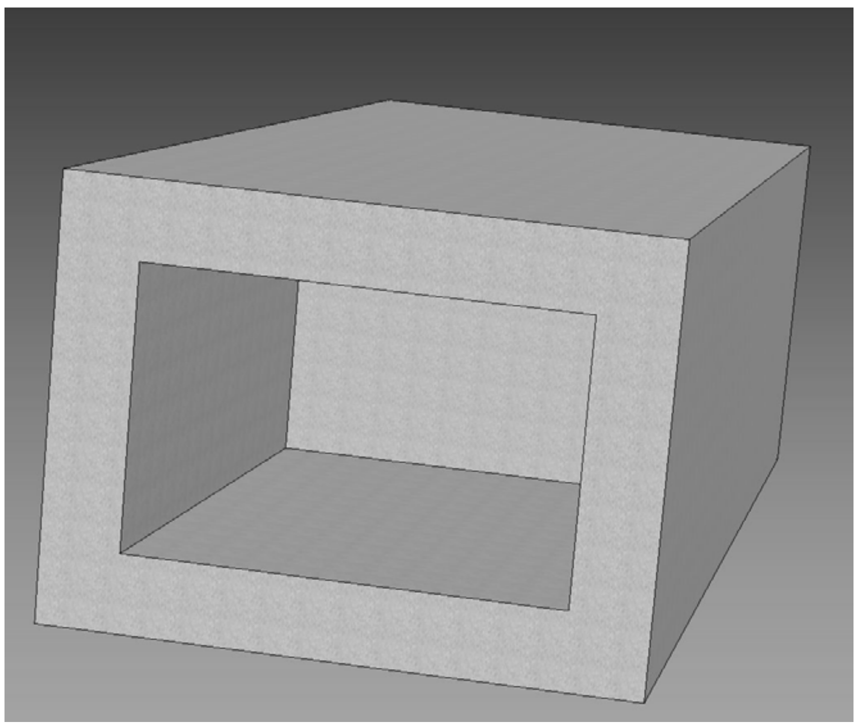
Note: The term camera mode indicates only the particular view method used for models in the graphics window. It is not meant to indicate that you can record actions that take place in the graphics window by choosing either Orthographic Camera mode or Perspective Camera mode.

Perspective views

You can set the view to Perspective using a drop-down list in the View Tabs Visual Styles panel:



In perspective camera mode, the model is displayed as it would be seen with a human eye. Objects further away from the camera appear smaller than objects closer to the camera. Also, if two lines are parallel to each other and are along the line of sight, they will 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. It is not meant to indicate that you can record actions that take place in the graphics window by choosing either Orthographic Camera mode or Perspective Camera mode.

Modeling Paradigms

There are many different types of modeling commands. In mechanical design, the need for manufacturing precision has led most commands to use some for of sold or surface BREP (Boundary Representation) modeling technology.

Within these commands there are several approaches and technologies that are combined to deliver modeling commands with various capabilities and strengths. For the sake of keeping this page short(ish) we will leave the discussion of high level modeling out and move onto those technologies that directly affect Inventor Fusion.

- Modeling with Features
- History compared to history-free modeling
- Explicit or Direct modeling
- Parametric modeling
- Dialog vs direct design manipulation

To date most successful modeling commands have fallen into two categories

- Parametric history based feature modeling
- Explicit history free modeling

It is not very important to understand everything about the two different technologies (and the new hybrid applications emerging) but it can be interesting. There is a lot of miss-information on these topics as different points of view try to make one modeling technology sound better than the other. Autodesk does not believe one is better than the other. We believe that you need both and that you should have the benefits of both as part of your solution.

Modeling with features

Mechanical designs are not non-descriptive blobs. They are precisely designed functional objects where features serve a purpose and have defined size. It is convenient to be able to assign names to these design features. It can be easier to work with these features because defined behaviors can help make the creation and editing predictable and efficient.

As an example: One common mechanical feature is a hole. It is drilled. Different drills bits come in different standard sizes and with different point angles. Holes can be counter sunk, counter bored and tapped. Rather that make a design create a cylinder and a cone to represent a hole. A hole feature can group the geometry together present the user with the list of available types and sizes making the definition of the geometry fast and easy. Holes are often positioned in specific ways. Away from two edges or concentric to some other geometry are common examples. Hole features can request specific positioning input, again simplifying the process over having to do this manually.

Feature based modeling command offer a comprehensive set of features that can be used to build up a design. Examples of features are:

- Extrude
- Revolve
- Hole
- Fillet
- Chamfer
- Sweep
- Loft
- Rib
- Pattern
- Mirror

One important point about features is that they not only aid creation but also aid editing. To change the hole diameter you can edit the feature and simple type in a new number rather than having to offset the cylinder and re-trim the cone.

Using a set of features you can describe and define most mechanical design and maintain some intelligent editing of those designs.

History compared to History free modeling.

When you design with features you implicitly create some order to them. This is because you make features serially one after another. Sometimes these features have relationships. You have to have a plate to place and create a hole. In this case, the hole has a relationship to the plate.

You can think of history like a recipe. You can replay a history of features and the same model will result every time. You can change dimensions in a feature and then replay the history to generate new geometry for a new model.

As an example: we create a 500mm x 300mm x 100mm Extrusion to create a plate. Next we create a hole with diameter of 20mm and a depth of 80mm. The hole is located 25mm from a corner of the plate.

If you change the size of the plate the hole updates. If the plate is made thinner, in this case less than 80mm the hole would punch through the plate.

History free modeling does not create a repeatable recipe. History free models are geometry. In a history free modeler, features are created independent of other features. When an edit is made, only the selected geometry is affected. You can make changes to geometry that may not be possible in a history based modeler because relationships prevent the specific change. To contrast, history based modelers are better at dealing with edits to a model where geometry changes (model faces become consumed are reappear depending on size).

Dialog box editing compared to Direct Manipulation

Direct manipulation refers to how a change is performed . With direct editing to make a change, you push, pull or otherwise modify geometry. With dialog editing changes are made by entering data into the dialog box and then either committing the change or a preview is updated.

Think of a cube as an example of this. Using direct edit modeling, you edit the cube by pushing and pulling faces. Using indirect modeling, you edit the cube by changing the value of parameters in a dialog box.

Inventor Fusion

Inventor Fusion is a direct manipulation, feature based history free modeler. Autodesk Inventor is a feature based history parametric modeler. In both applications, you create sketches, dimensions, and features to generate a 3D model. The difference is how the information is stored and modified. Inventor Fusion stores the information with the geometry. You modify geometry directly to change the model. Changes only affect the geometry being manipulated. Autodesk Inventor stores the information in parameters features and their history. To make a change, you modify parameters or relationships between features and the remaining features are recalculated based on that information.

System Requirements

Operating System

- Microsoft® Windows® Vista (32-bit or 64-bit), SP1
- Microsoft®Windows® XP (32-bit), SP2, SP3
- Microsoft®Windows® XP (64-bit), SP2

Hardware

Minimum requirements:

- Intel Pentium 4, AMD Athlon 64 or AMD Opteron or later with 2.0GHz or faster processor; or compatible
- 1.0+ GB RAM
- 2.0+ GB of free disk space (for installation)
- Microsoft Direct 3D 9 or 10 graphics support with 64+ MB
- 1,024 x 768 Screen Resolution

Recommended Specifications:

These specifications are over and above the minimum requirements.

- 2 GB RAM
- 1,280 x 1,024 Screen Resolution

Graphics Processing Unit (GPU) Requirements

Minimum requirements:

- Direct 3D 9-Compatible Graphics Card with Pixel Shader 2.0 support. Supports most commonly used Inventor Fusion effects; does not support the Ambient Occlusion Effect.
    - Technical note: Supports Autodesk Graphics Feature level 2_0

Recommended Specifications:

- Direct 3D 9-Compatible Graphics Card with Pixel Shader 3.0 support or Direct 3D 10-Compatible graphics card. Supports all Inventor Fusion Effects, including Ambient Occlusion.
  - Technical note: Supports Autodesk Graphics Feature Level 3_0

Testing your Graphics Card Capabilities:

In your Inventor Fusion Install directory ("C:\Program Files\Autodesk\Inventor Fusion Technology Preview 2" or the equivalent on your computer) you will find a Graphics Capabilities Application "AdskDiagOutput.exe".

Open a command prompt (Start Menu- Run cmd) and navigate to this folder. Type "AdskDiagOutput out.txt" and press Enter. The program runs and produces an output file (out.txt or any other name you choose).

Open the output file. The entry displays something like this:

Autodesk System and Graphics Device Information Reporting command

System Information:

Operating System: 32-bit Windows Vista Business (6.0, Build 6000) (6000.vista_rtm.061101-2205)

Language: English (Regional Setting: English)

System Manufacturer: Dell Inc.

System Model: Precision M70

BIOS: Default System BIOS

Processor: Intel(R) Pentium(R) M processor 1.86GHz (1 processor, 1 core)

System Memory: 1022MB RAM

Page File: 1022MB used, 1279MB available

DirectX Version: DirectX 10

DxDiag Version: 6.00.6000.16386

DirectX Version: 10.0

Graphics Device Information:

Manufacturer: NVIDIA

Chip Type: Quadro FX Go1400

DAC Type: Integrated RAMDAC

Graphics HW Memory: 256 MB

Display Mode: 1920 x 1200 (32 bit) (60Hz)

VendorID: 0x10DE

DeviceID: 0x00CC

SubSystemID: 0x019B1028

RevisionID: 0x00A2

Graphics driver file: nvd3dum.dll,nvapi.dll

Driver file version: 7.15.10.9746

Driver Date: 8 December 2006 12:25:00 UTC (Coordinated Universal Time)

DDI Version: 9Ex

WHQL Level: False

Autodesk Graphics Feature Level:

Feature Level: 3_1

The last line "Feature Level" will tell you what level of Graphics support your card offers Inventor Fusion.

- If the value is less than 2_0, then Inventor Fusion may not work well or at all on your computer. If you find any graphical defects, they may not be addressed by the Inventor Fusion Development Team.
- If the value is 2_0, Inventor Fusion works on your computer but the Ambient Occlusion effect will not work. If you turn the effect on, you will see no change to your image.
- If the value is 3_0 or higher, all effects used by inventor Fusion are supported on your computer.

In case you have any issues with your graphics card, and you believe the graphics card meets the requirements, e-mail the previous text file along with a description of your problem to labs.iv.fusion@autodesk.com. You may be contacted by a member of our team and asked to provide further information like screenshots, sample files and so on to help fix the issue.

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