# What's New in Autodesk Inventor 2016

Inventor 2016 enables you to break through boundaries typically associated with 3D modeling, delivering an invaluable connected design experience between you, the data, people, and processes involved in getting your job done. From providing seamless parametric, direct editing, and freeform design tools, to associatively connecting your Inventor data to non-native CAD formats, to truly integrating your electrical and mechanical data into a single design - Inventor 2016 helps you connect with your design like never before.
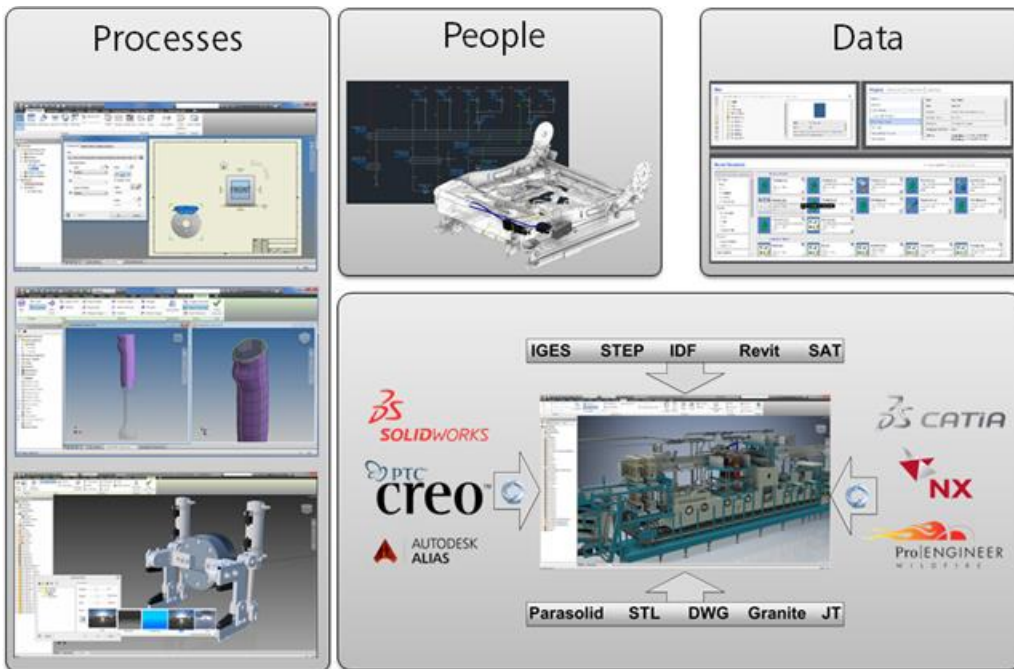
Work More Efficiently in a Multi-CAD Environment

New Technology for Importing Files from other CAD Systems.

- An associative import of CAD data from CATIA, SolidWorks, NX, Pro-E/Creo, and Alias files allows you to maintain a link to the selected file. The imported geometry in Inventor updates as the model changes.
- Selective import support allows you to only read in the geometry that is important to you, speeding the import process: The new Select tab provides you the option to specify which objects to import when importing a CATIA, SolidWorks, Pro-E/Creo, NX, Alias, STEP, IGES, or Rhino file.
- Multi-thread support has been enabled. This allows Inventor to more efficiently use the available hardware on which Inventor is running. As a result, you will experience improved performance when working on files from other CAD systems.
- The options for importing CATIA, SolidWorks, Pro-E/Creo, NX, Alias, STEP, IGES, Rhino, SAT, Parasolid Binary files are simplified and provide clear choices for import.

Associative DWG

- Quickly insert an AutoCAD DWG file in an Inventor part file as a DWG underlay using the Import command in the 3D Model tab, Create panel.
- You can add assembly relationships to underlay geometry.
- Use the Project DWG Geometry command to project DWG geometry, polylines, open or closed loops, and DWG blocks. Then you can use the projected sketch elements to create modeling features.
- Your 3D Inventor models based on the DWG geometry update when the 2D geometry changes in AutoCAD.

AutoCAD Electrical and Inventor Interoperability

- The new Electromechanical link between Inventor and AutoCAD Electrical provides a smooth data exchange between your 2D and 3D electrical designs.
- When you create a link between an AutoCAD Electrical and an Inventor assembly (not available in Inventor LT), the project files become associative: Design data changes made in one product are updated in the other via Sync.
- The Location View command on the new Electromechanical tab on the Assembly (not available in Inventor LT) ribbon in Inventor displays the devices and wiring contained in both the AutoCAD Electrical Drawings and Inventor Assemblies.

Shape Creation
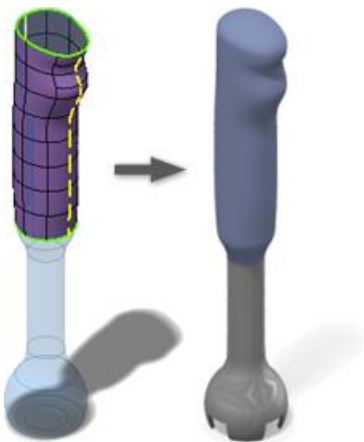
Use powerful new commands and workflows in the Freeform modeling environment. Some of the highlights are:
- Work with open surfaces or closed shapes.
- Convert existing model faces to freeform geometry for shape refinement.
- The new Freeform Thicken command creates solids, offset surfaces, or shell walls.
- Unweld edges to split and move a freeform body segment.
- Delete faces.

3D Printing Environment

A new environment is added that lets you position and orient your design within the print space of the selected 3D printer. You can also update the part in the print environment that does not impact the source document. When you are finished, you can send the results to Print Studio or other printing software to begin printing the part.
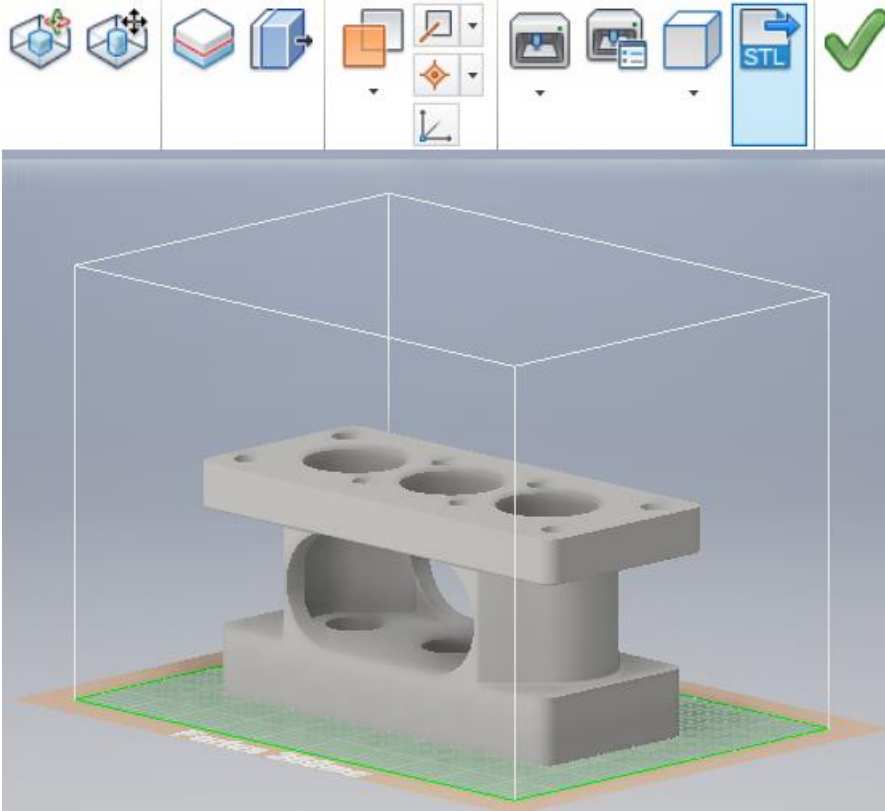
Drawing Environment

- Drawing view creation is simplified, and uses in-canvas tools.
- Text formatting is expanded with new options: Bullets and numbering, strike-through text, enhanced formatting (all caps, title case, lower case).
- Surface texture and feature control frame symbols are updated to the latest standards.
- Many new graphical symbols are available, and can be inserted in various types of drawing annotations.
- Balloon styles can use custom balloon shapes.
- A new Single Segment Leader option allows you to create drawing annotations with a single leader segment.
- You can align drawing annotations vertically, horizontally, or to an edge.
- Create a view sketch on a model with included work features and select those work features with the Project Geometry command.
- Quickly access and share your collection of sketch symbols through a new external Sketch Symbols Library. This library is an Inventor drawing file that, by default, is located in a subfolder of the Design Data location of your project. This new feature offers the following new functions:
  - A search and filter feature in the Sketch Symbols Library dialog box.
  - Preview the sketch in a preview pane within the dialog box before placing the sketch.
  - Browser expansion state remains throughout session when placing sketch symbols.
  - A Sketch Symbol Library can be created in IDW and DWG formats, allowing insertion of symbols contained within the library file from either format.
  - Save your sketch symbols to a customized library.

Improved Start Up and Learning Experience

The new start-up and learning experience combine to make learning and using Inventor faster and easier.
- Movable and resizable Inventor Home panels.
- Home provides greater access to common file tasks.
- Files can be removed from the recent documents list.
- Vault status is displayed in the recent documents list.
- Create custom shortcuts to provide fast access to frequently used files and locations.
- Help content was edited to provide better search results.
- Topic content was combined to reduce the number of clicks.

Graphics/Visualization/Studio Enhancements
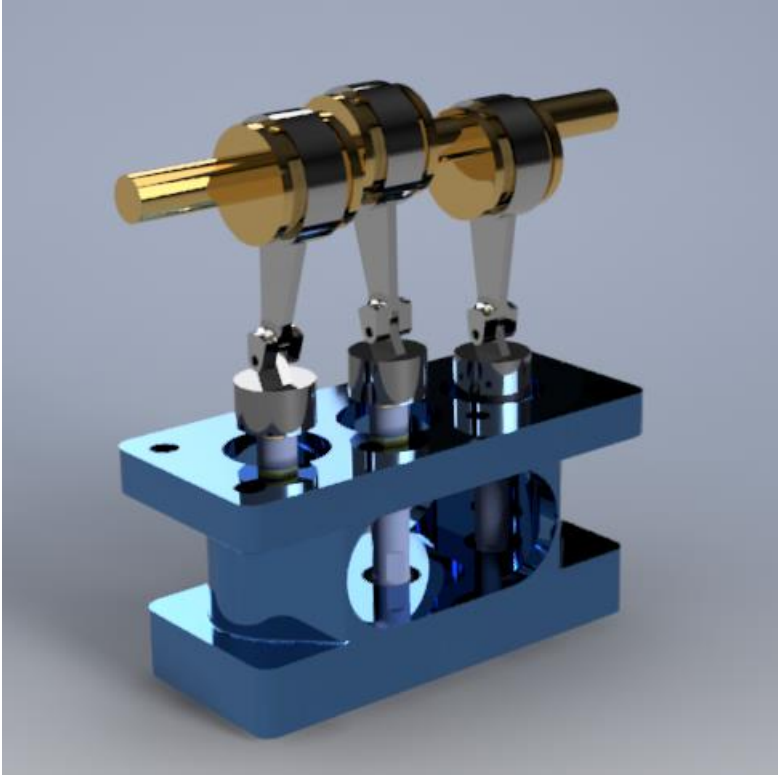
Some highlights are:

- ALL lighting styles in Inventor Studio are now associated with an IBL. A lighting style can have zero local lights but must have one IBL.
- IBL (Image Based Lighting) based lighting styles provides better lighting sources in Inventor Studio with enriched IBL collection.
- When you enter the Studio Environment, all legacy local lights are now disabled by default. It is recommended you use IBL for better rendering results. You can turn on specific local lights manually when necessary.
- All newly created lighting styles are associated with the default IBL automatically. You can change the associated IBL to another IBL as desired.
- The enhanced visual effect for shaded visual style makes it more consistent with the realistic visual style.
- The rendering engine in Inventor Studio was changed to RapidRT with advanced configurations for better quality rendering.
- The Studio Render Illustration settings have been moved to the View tab Appearance Panel Visual Styles drop-down menu. The new and enhanced Technical Illustration command creates a realistic illustration effect in the graphics window.

Workflow Enhancements

Among the many enhancements, are:

General

- Use Escape (Esc) key to cancel an operation in select processes.
- Multiple productivity enhancements made to dialog boxes.
- Add all window tile styles to the task bar.
- Dock an Inventor browser on any application's window edge.
- Hide all sketch dimensions now available in the Object Visibility menu.
- Import/Export external rules configuration for iLogic (not available in Inventor LT).

Sheet metal parts

- Multi-body support is added to sheet metal.
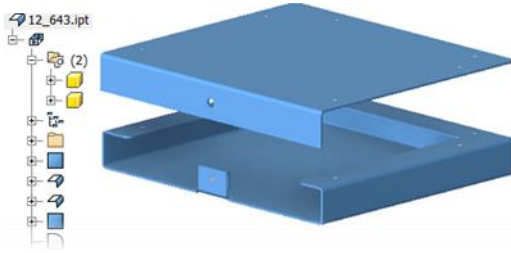
- Support for zero bend radius is added to many commands.
- Material thickness is detected when you convert a part to a sheet metal part.
- Punch tool shows a count of the center selections.

Part appearance and body, feature, or face color overrides can be pushed from derived part to new part

A new Use color override from source component option has been added to the Derived Assembly (not available in Inventor LT), Derived Part, Make Part, and Make Component dialog boxes. Selecting this option pushes the Part appearance and body, feature, or face color overrides to be pushed from the derived part to the new part.

Tube and Pipe setting enhancements (not available in Inventor LT)

You can now customize file names for fittings and populate part numbers into your parts list within the drawing environment. Previously, you could only change the name of your conduit items, but now you can update fittings as well.

Parts

- Face draft contains powerful new options that let you Fix or Move the parting line.
- Ruled Surface is added to the surfacing commands.
- The Mirror and Pattern commands support multiple solid body selection.
- Previously, nonlinear patterns of a solid body in a multi body part file have not been possible. You can now create nonlinear patterns for solid bodies.
- Drag a sketch above the parent feature in the browser to share it.
- The Measure command now allows you to measure an angle to the midpoint of any segment. This option is achieved by hovering your mouse over the midpoint of a segment until a yellow dot appears.

Sketch

- Identify which workplane or face a sketch was created on.
- The selected Show All or Hide All Constraints display setting remains active as you sketch and throughout your editing session.
- You can now create tangent dimensions between circular or arc geometry within a 2D sketch.
- The Initial View Scale property of the first drawing review placed on the sheet is added in the Sheet Properties group within the Format Text dialog box.
- The Sketch Dimensions option is added to the Object Visibility list. Select this option to display 2D or 3D sketches and hide all related sketch dimensions.
- Sharing a sketch is made visible by dragging above the feature in browser.
- New snap points added to the context menu: Endpoint, Apparent Intersection, Quadrant, and Mid of 2 points.
- You can now modify Start Sketch and select a view or sheet before sketching.

Assembly (not available in Inventor LT)

- New safety factor calculation warning displays in Stress Analysis.
- The Midplane option has been added to the assembly Pattern command. Select the Midplane option to create a pattern distributed on both sides of the original component.
- Replace All feature available for highlighted components within an assembly.
- Select multiple sick constraints within Design Doctor to delete.

Drawings

- You can start a drawing from any open model, and automatically apply the current model camera and representations in the base view.
- In-canvas tools in the Base View command simplify creation of a base view and projected views.
  For example, you can use the ViewCube to orient the model, set the view scale by dragging a view corner, or adjust projected views while creating the base view.

Presentations (not available in Inventor LT)

- Tweak command is redesigned to use direct manipulation tools.
- Creating and editing of trails is easier.
- Component selection is simplified.
- Auto Explode command is added to the ribbon and improved.

Keep reading for details on all of these great enhancements and more.

General

## Visualization, Graphics, and Studio Enhancements

Studio Enhancements

New support for Image Based Lighting (IBL)

All lighting styles in Inventor Studio are now associated with an IBL. IBL provides a better lighting source.

Image-based lighting from the modeling environment is now supported by Inventor Studio. To render from the modeling lighting environment, use the commands available on the View tab, then use the Current Lighting in the Studio Render Image dialog box, or change to other lighting styles from Render Image dialog box.
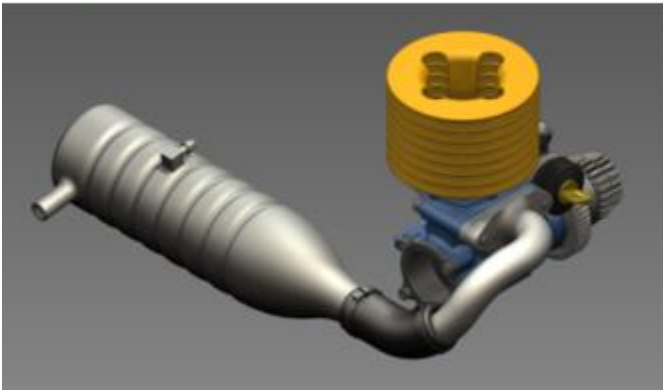
New 2016 IBL Lighting Style features

- IBL provides association
- Background turn on/off
- Support additive lights
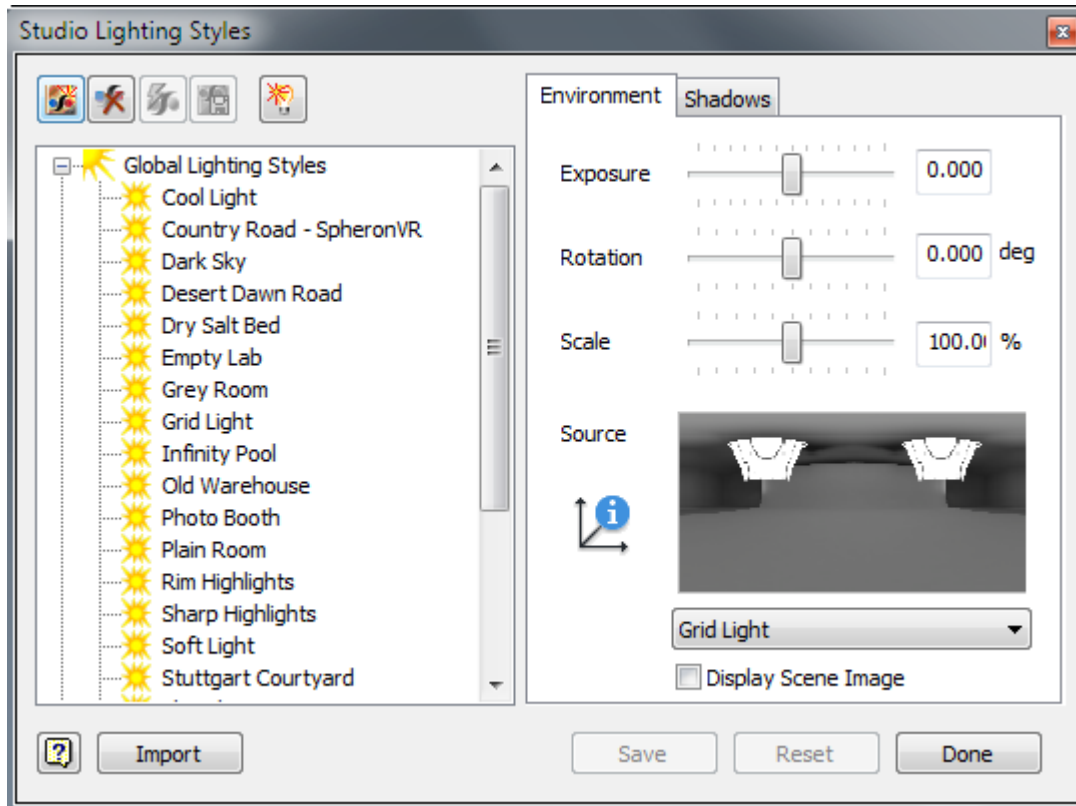
Advanced rendering engine RapidRT provides new configurations for rendering

- Time duration/iterations/unlimited mode
- Lighting and Material Accuracy modes

Obsolete options removed:

- Scene Style: The Ground plane and shadow effect settings specified in the View tab now display in the rendered model in Inventor Studio.
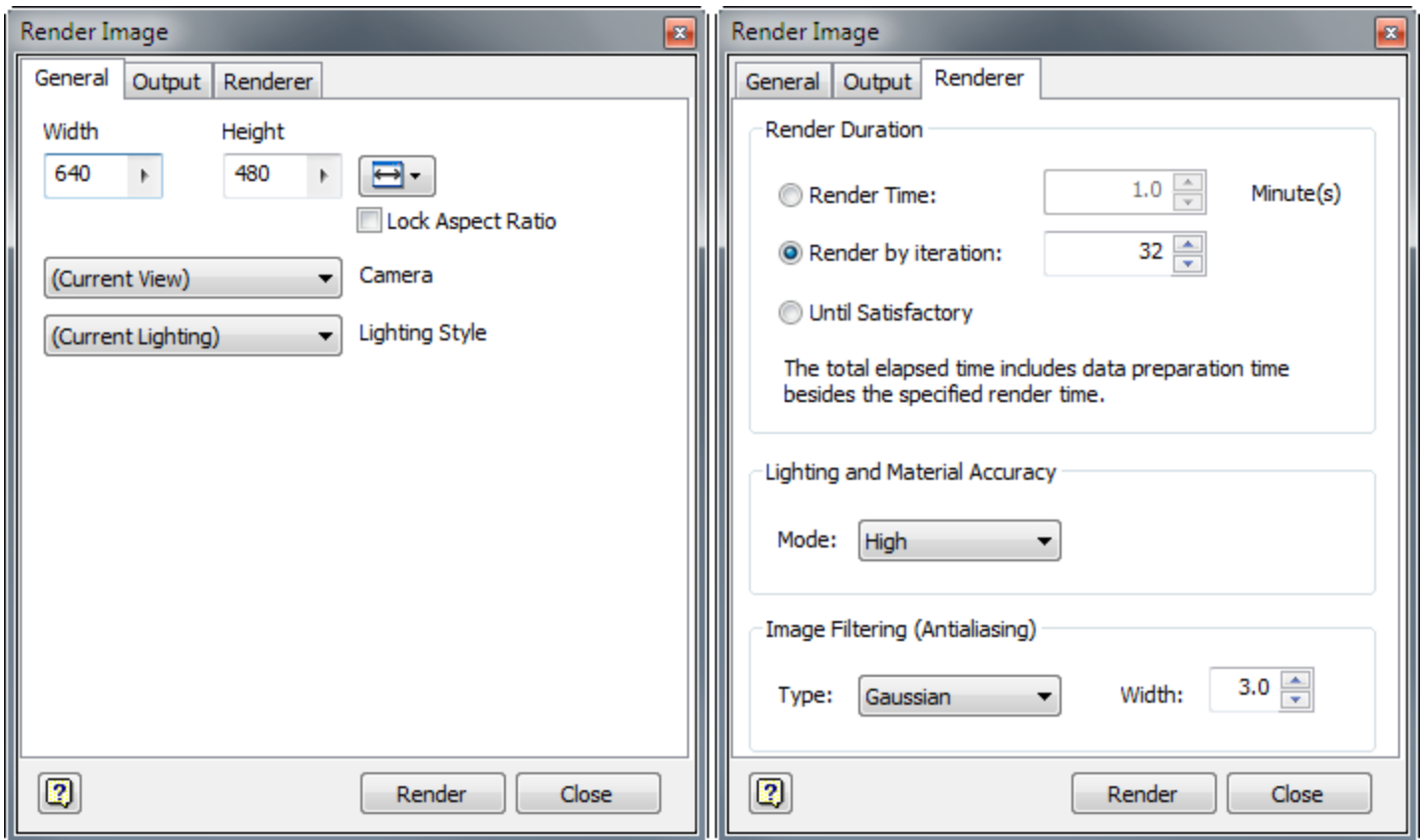
- The Studio Render Illustration settings have been moved to the View tab Appearance Panel Visual Styles drop-down menu. The new Technical Illustration command creates a realistic illustration effect in the graphics window.

Ground Plane and Shadow Effect display in final rendering result

The Ground plane and shadow effect settings specified in the View tab will now display in the rendered model.

Graphics Enhancements

The enhanced Shaded visual style display effect is now more consistent with Realistic visual style. Enhanced visual effect for shaded view. The following two images compare the results of using Shaded View in 2015 and 2016. High rendering performance is maintained.
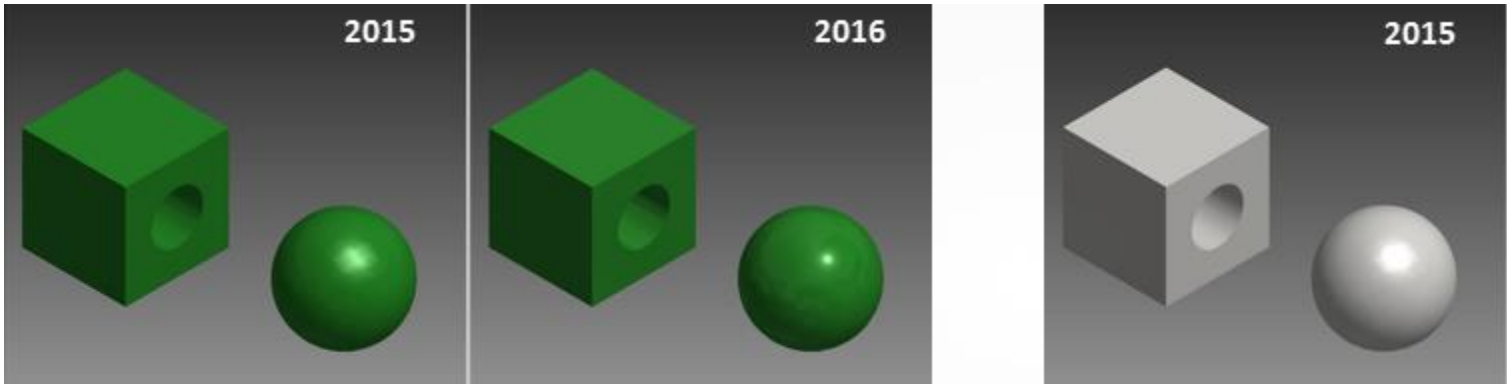
Technical Illustration Visual Style

The Studio Render Illustration settings have been moved to the View tab > Appearance Panel > Visual Styles drop-down menu. The new Technical Illustration command creates a realistic illustration effect in the graphics window.

New IBL styles

11 new lighting styles are added to the View tab > Appearance Panel > Lighting Styles menu. These include Dark Sky, Gray Room, Infinity Pool, Photo Booth, Tranquility Blue, Sharp Highlights, Rim Highlights, Cool Light, Warm Light, Soft Light, and Grid Light.

Changes to Ray Tracing

- The Ray Tracing modes are changed to Low, Draft, and High to indicate the lighting and material accuracy.
- The progress status bar is changed to display quality measurement instead of percentage progress.
- You can now save an image any time during Ray Tracing.
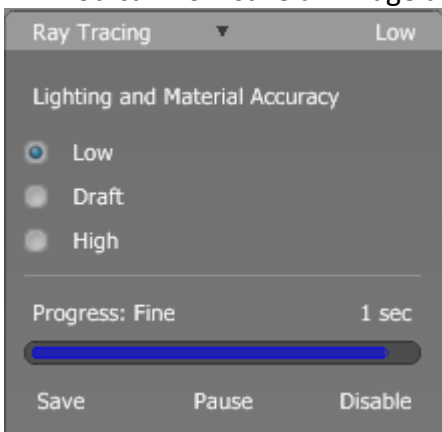
Visual enhancement in drawings

The new Reflection Environment settings in the drawing Document Settings dialog box allow you to apply specular effect and specify a Reflection map in a Drawing (.idw) file.
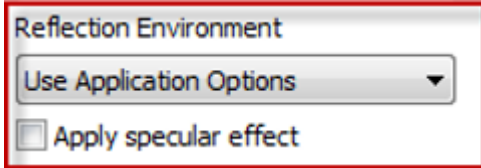
Text and Leader Text

Redesigned Format Text Dialog Box

The Format Text dialog box is simplified and refined to be clear and easy to use. Size of the dialog box is reduced to provide more space for editing in the graphic window.
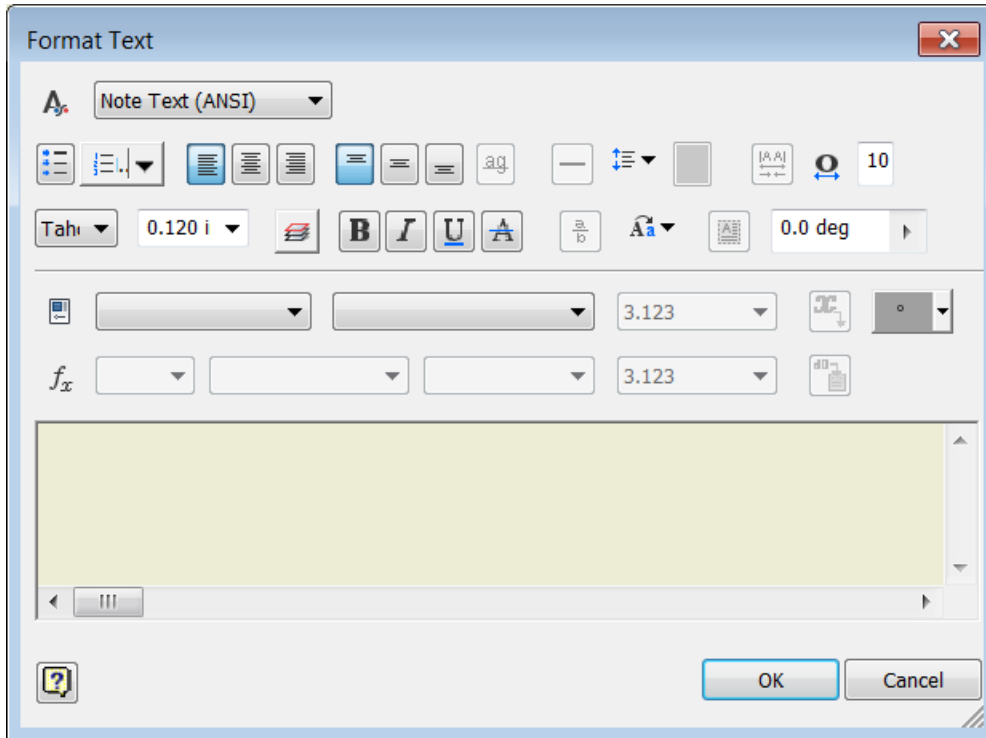
Bullets and Numbering in Text
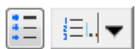
You can create bulleted or numbered lists in the Format Text dialog box. One bulleting style and several numbering styles are available.

Text Case Options

In the Format Text dialog box, select a string, and select a Text Case option to convert the string to UPPERCASE, lowercase, or Title Case.

Strikethrough Text

In the Format Text dialog box, click the Strikethrough button to apply strikethrough formatting for a string.

Enhanced Symbols List

New graphical symbols are added in the Symbols list in the Format Text dialog box.

Graphical symbols are organized in the following groups: Most Common, Geometric Characteristic, Material Removal Modifier, Additional Symbols, General, and Character Map.
Note: The active Feature Control Frame style controls the list of symbols available in the Format Text dialog box.

Edit Dimension Dialog Box

The symbol drop-down list has been removed and replaced by the graphic symbols as first-level options.

Text Preview in the Graphic Window

Text in a drawing file is dynamically previewed in the graphic window when you create or edit it. You can refine the appearance of text before you close the Format Text dialog box.
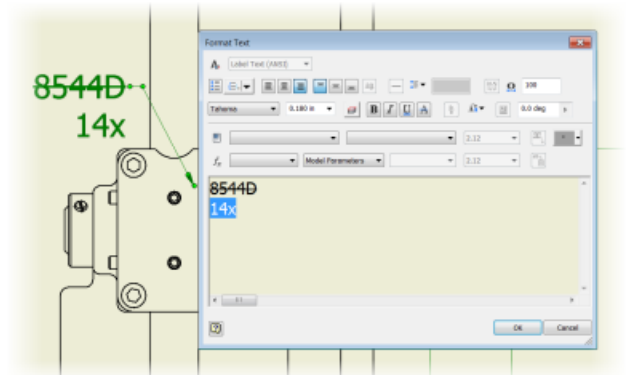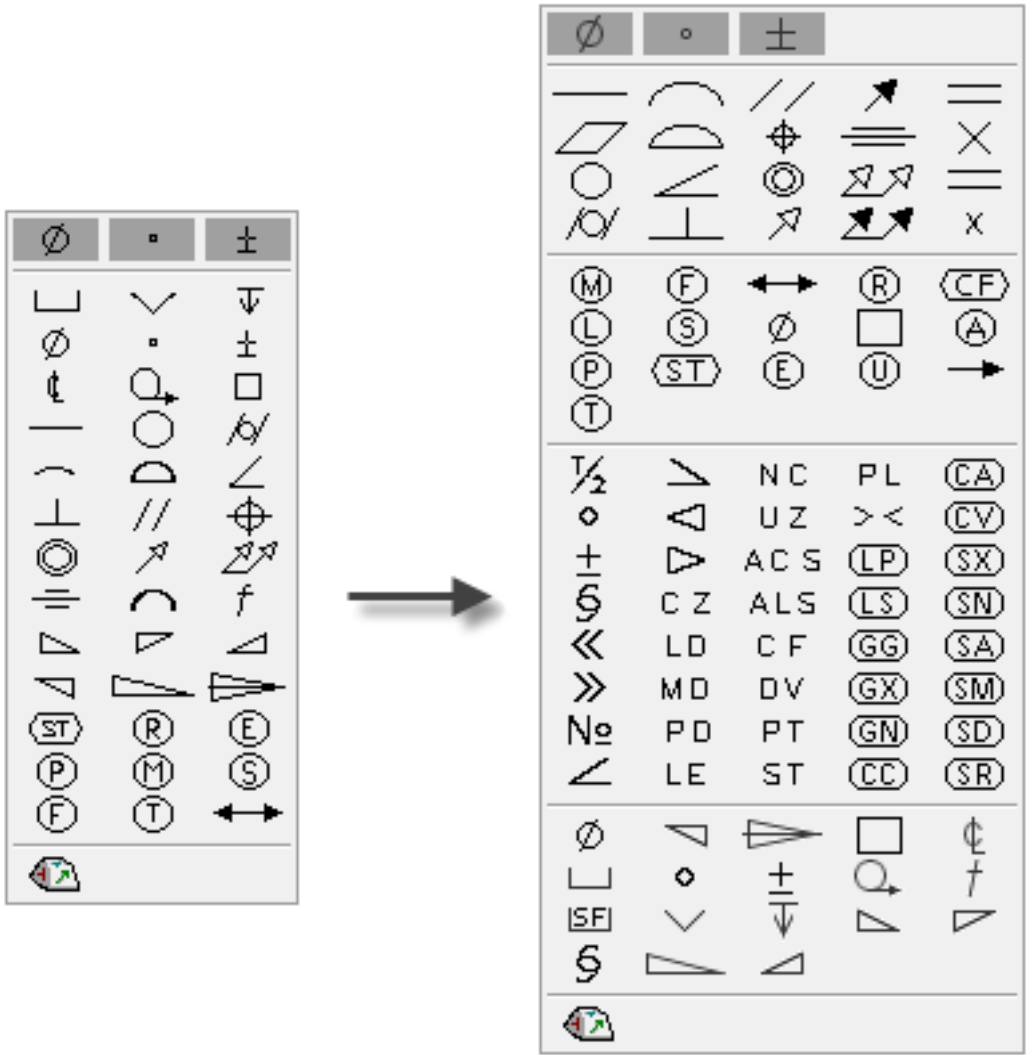
Zoom Edited Text In and Out

The Zoom In and Zoom Out buttons are removed from the Format Text dialog box. To zoom the edited text, place the cursor in the editing field, hold the CTRL key on your keyboard, and scroll the mouse wheel.

Rotation for Leader Text

In the Format Text dialog box, use the Rotation Angle option to rotate leader text in a specified angle.
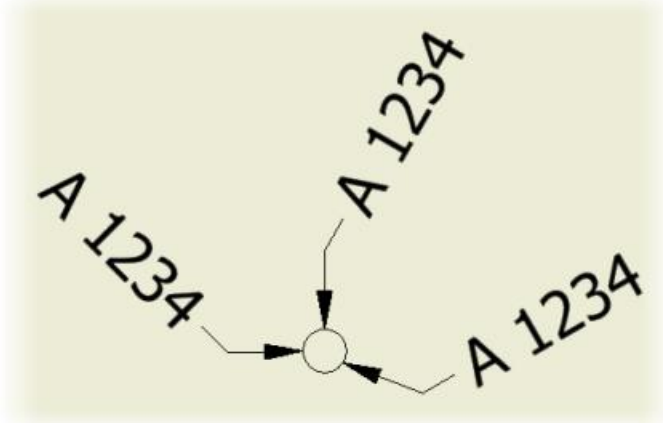
When you drag the end point of the landing line, text remains aligned to the landing line and keeps the original Rotation Angle.
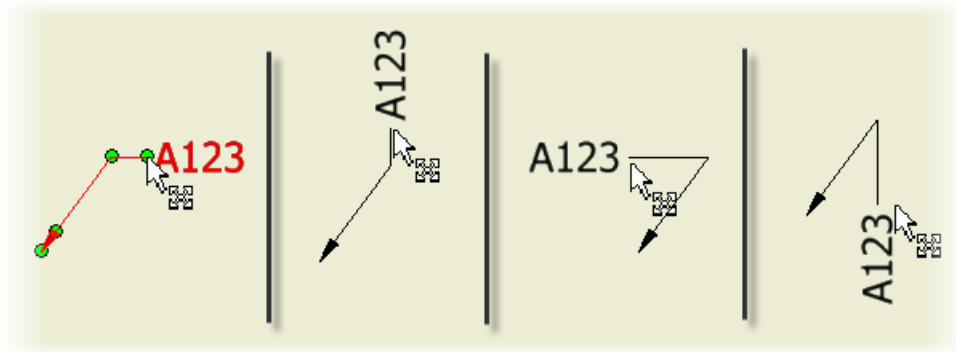
Retained Formatting for Copied Text

Numbered or bulleted lists and text formatting are retained when you copy text from a Microsoft Office document, or from AutoCAD.

Retained Formatting for Imported or Exported DWG Files

When you open an AutoCAD DWG file in Inventor, imported text is read-only, but keeps the original formatting. When you open an Inventor DWG file in AutoCAD, text keeps its formatting.

In-Canvas Text Rotation

Select a text object, and drag the blue rotation grip point to rotate text to the desired position.
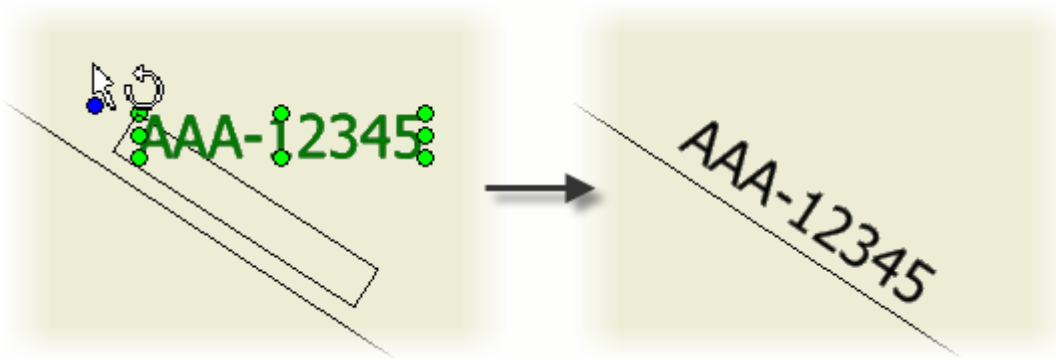
Other General Enhancements

Inventor Home

Inventor Home includes many new tools to make file management fast and simple.
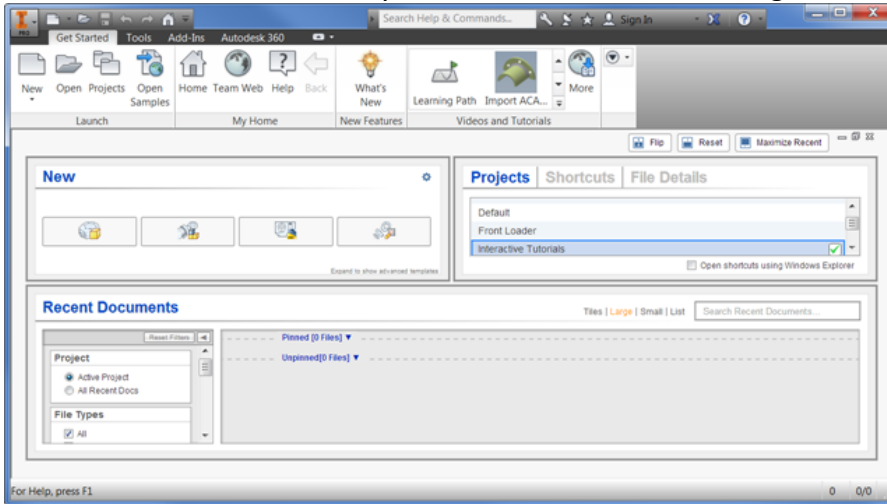
- The panel workspace is customizable.
  - Drag the handles to resize the panels.
  - Use Flip to reconfigure the panels.
  - Use Reset to restore the default layout.
  - Use Maximize Recent to fill the space with the Recent Documents panel.
- Presentation files are added to the New file list.
- Click Advanced in the New file panel to access all file templates.
- Vault status is displayed for recent documents.
- Projects, Shortcuts, and File Details tabs are added to Home.
  - Click Projects to set the active project file.
  - Click Shortcuts to create fast access to project locations, files, and folders.
  - Click File Details to view rich file information.
- New filter option allows you to view the information you want in the Recent Documents display.
  - Set the Project filter to show only files in the active project, or all recent documents.
  - Set the File Types filter to add or remove file types from the display.
  - The Sort By filter provides multiple sorting options.
  - The Date Modified filters allow you to fine tune the display with a wide range of dates.

Add or remove file types from the display.

- Move your mouse over a file in the Recent Documents list to view the available operations. For example:
  - Open the file
  - Remove from the list
  - Open with options
  - Open containing folder
  - Explore containing folder
  - File details
  - Clear unpinned documents

    Tip: The action to Clear unpinned documents from the recent files list is permanent and cannot be undone. You must open the file again to display in the recent documents list.

- Click Tiles, Large, Small, or List to change the display of the Recent Documents list.

Access Team Web and Online Help

Some proxy servers require authentication before granting access to the internet. If authentication is required, a new dialog box appears that lets you provide a username and password.

Work Plane

The Tangent to Surface and Parallel to Plane option supports a NURBS surface as the tangent selection.
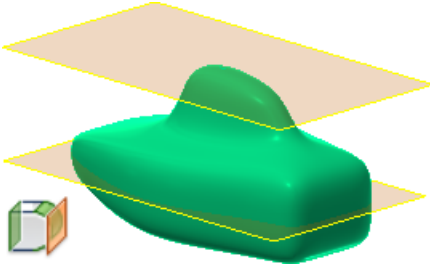
Changes to Content Center Libraries (not available in Inventor LT)

The Content Center libraries are updated to the latest industry standards.

You can now set the maximum length for Structural Steel to 50 m for all libraries. This setting is no longer limited to ANSI.

For complete details on the changes to the Content Center libraries, click here.

Enhancement Updates to Dialog Boxes

The following dialog boxes contain new features:

- **Parameters:** A Copy and Paste function is now available within the right-click context menu. The Paste option is available only when there is content copied to the clipboard.
- **Link Parameters:** This dialog box is now resizable to accommodate extra rows. When you resize this dialog box, the size persists from session to session.
- **Thicken/Offset dialog box:** The Apply option is added so you can execute multiple operations without exiting the command box.
- **Parameters dialog box:** The numerical sort order is now based on value rather than the leading digit.
- **Hole dialog box:** The Centers option now has a selection counter, displaying in parentheses the number of selections made.
- **Punch dialog box:** The Centers option now has a selection counter, displaying in parentheses the number of selections made.
- **Application Options/General tab:** A link is provided in Help options to download the local Help.
- **Resizable dialog boxes:** The following dialog boxes are now resizable and persist throughout your session.
  - Pack & Go
  - Symbol Library
  - Boundary Patch
  - Edit Balloon (drawings)
  - Project Editor
  - Link Parameters (parts and assemblies) dialog box

Update to Export Formats for Global Bill of Materials (BOM), Parts Lists, and Revision Tables (not available in Inventor LT)

A new list of acceptable formats to export your Global BOM, parts lists, and revision tables is now available in the drop-down menu.
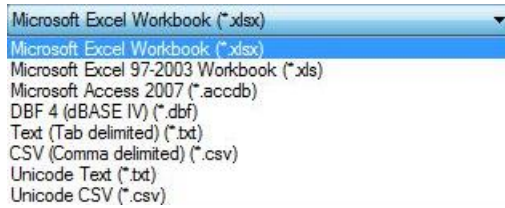
The Access, DBF, Text, CSV, and Unicode exports require Microsoft Access Database Engine to be installed on your computer, which is part of Microsoft Office installation. For 64-bit Inventor, install 64-bit Microsoft Office in order to export these files. If 64-bit Microsoft Office is not installed, the following error message displays:

Changes to Operating System Support (not available in Inventor LT)

Inventor 2016 now supports 64-bit Windows 7, Windows 8, and Windows 8.1 operating systems. Inventor LT also supports 32-bit Windows 7.

All inventor products, except for Inventor LT, no longer offer support for operating systems with 32-bit processors.

Security Enhancements to Add-In Manager (not available in Inventor LT)

When a non-Autodesk product add-in is loaded, a new dialog box displays. You can now choose to Allow or Block an add-in from loading. This setting will be retained for future sessions. You can examine and change the Allow/Block settings using the Add-In Manager.
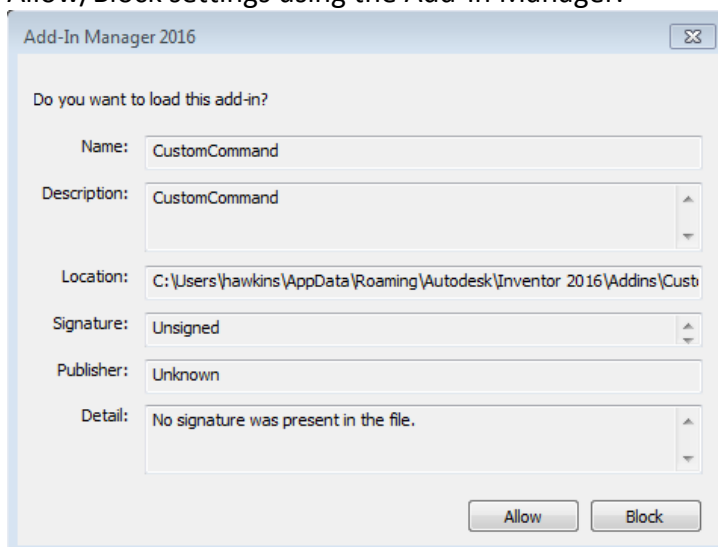
The following three security controls are now available in the Add-In Manager 2016 dialog box:
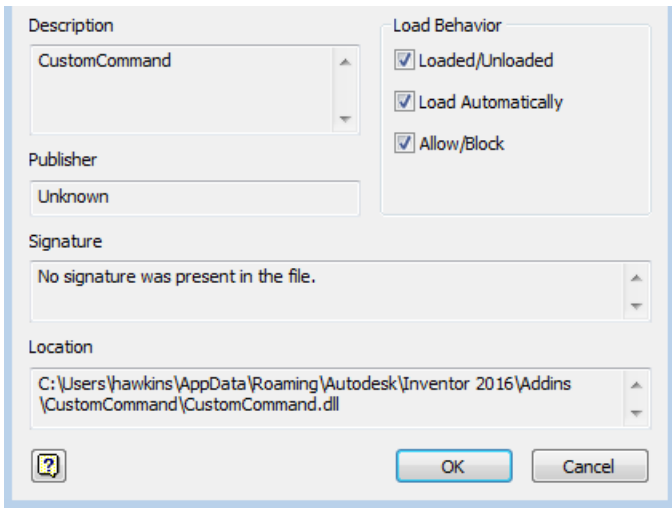
- Allow/Block check box in the Load Behavior area. This box is checked and disabled when Autodesk, Inc. approves the add-in.
- Signature text box shows the add-in is signed and has valid certificates.
- Publisher text box shows the add-in approved by Autodesk, Inc.

Enhanced Representation Support in Presentations (not available in Inventor LT)

Use the new Representations dialog box to change the current design view and associativity. To access this new dialog box, right-click the Explosion node within your model browser in a Presentation and select Representations.
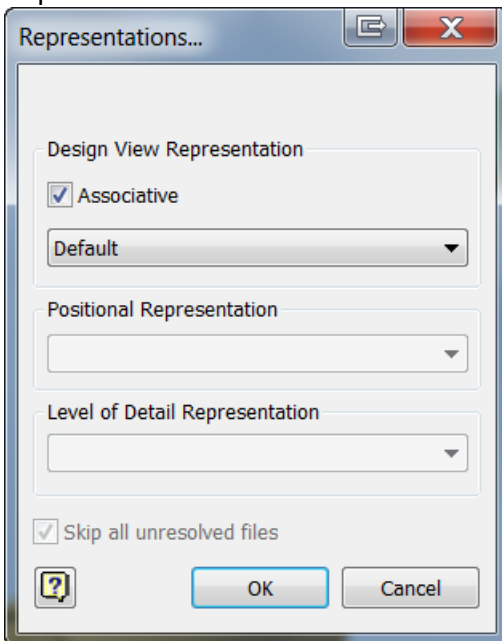
New Taskbar Menu Options Available for Tiling

Tile Horizontally and Tile Vertically menu options were previously only found in the top ribbon. Now there is convenient access to these options through the taskbar menu found at the bottom left corner of your workspace.

Added Window Select for Faces When Using Delete Face and Thicken Offset

In these commands, you can use a selection window to pick connecting faces of one set of body selections. If a surface body and a solid body are defined, you cannot select faces from both bodies at the same time. After you make the initial selection on a body, you can select more surfaces from the same body.

Standard window selection methods are:
- Cross selection: Select from right to left. Anything that is 100 percent contained within the selection frame is selected.
- Window selection: Select from left to right. Anything that is partially contained within the selection frame is selected.

Use Keyboard Shortcuts for Select Other

In model views, when you select a component or a subassembly, you can use the shortcut, Ctrl+A for Select Other. This option is listed in the shortcut and command alias reference in the Customize dialog box, Keyboard tab, and View category, where you can customize them.

Dock Inventor Browsers on Any Application or Window Edge

While working with one or more monitors, you can undock and drag any of the Inventor browsers. This can be done with modeling, iLogic (not available in inventor LT), Vault browser, or the Tutorial (Learn) browser. The browsers attach to one another and to any Inventor application edge. They attach either side by side or stacked.

An Always on top context option is available for all detached browsers. Browser positions and settings persist from session to session.

Use Escape (Esc) key to Cancel an Operation

Press the Escape key to abort or cancel an operation in process. The time it takes to cancel an operation may vary depending on the feature or data set that is in use. For operations with realistic preview, such as shell or fillet you can interrupt the preview.

Currently, not all operations are interruptible and some operations do not respond immediately when pressing the Escape key. A message displays when the cancellation is complete.

Copy Parameters to User Parameters

While working with Parameters, you can now copy an existing parameter, including all its settings and multivalues, as a User Parameter. Previously, recreation and manual entry was required for each value.

Import and Export External Rules Configuration for iLogic (not available in Inventor LT)

 You can now import/export iLogic configuration settings (as an XML file) to be shared between different users and machines. Once configured, externally mapped folders appear beneath a Standards Directory tree that will update/refresh immediately when content changes (sub folders and rules). In the past, external paths had to be manually added.

External rules configuration settings can also be leveraged and mapped during creation of deployments.

Global forms can now be refreshed through a right-click context option to reflect the most up-to-date form.

New Import and Export SMT File Extension Available for Autodesk Shape Manager (ASM) Files

Autodesk has introduced a new import and export file type associated with ASM files. You can now import and export ASM files using the SMT file extension.

Mockup 360 Now Available from the Inventor App Store

You can now download Mockup 360 from the [Inventor App Store](#). Mockup 360 is no longer available for install from the Inventor installer.

Update to Allow Part Appearance and Body, Feature, or Face Color to be Pushed from Derived Part to New Part

A new Use color override from source component option is available in Derived Assembly, Derived Part, Make Part, and Make Component dialog boxes. This checkbox, when checked, pushes the Part appearance and body, feature, or face color overrides applied to faces from the derived part into the new part. If unchecked, the appearance is set to default.

Example dialog boxes 1:
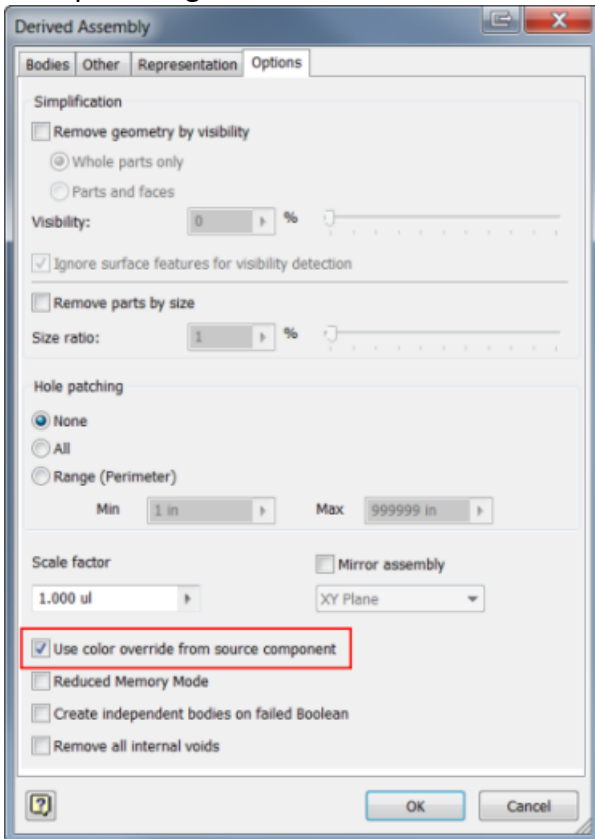
Example dialog boxes 2:
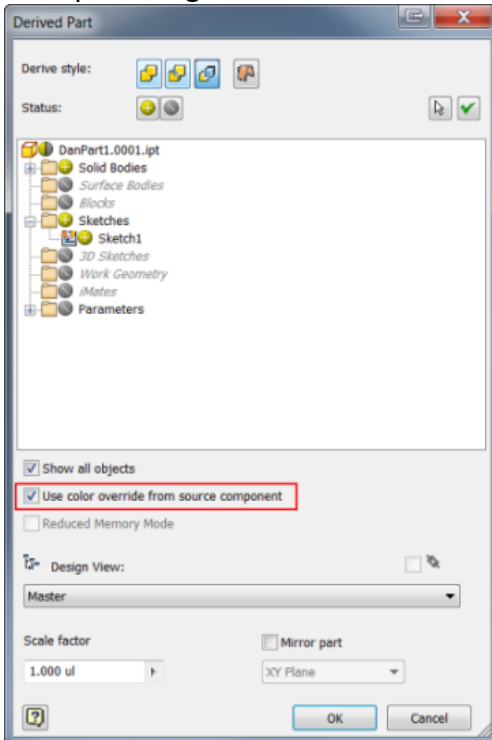
A global setting can be set to default the Use color override from source component check box to remain checked or unchecked throughout your session. This check box can be accessed through the Application Options dialog box under Make/Derive default, under the Tools tab in the ribbon.
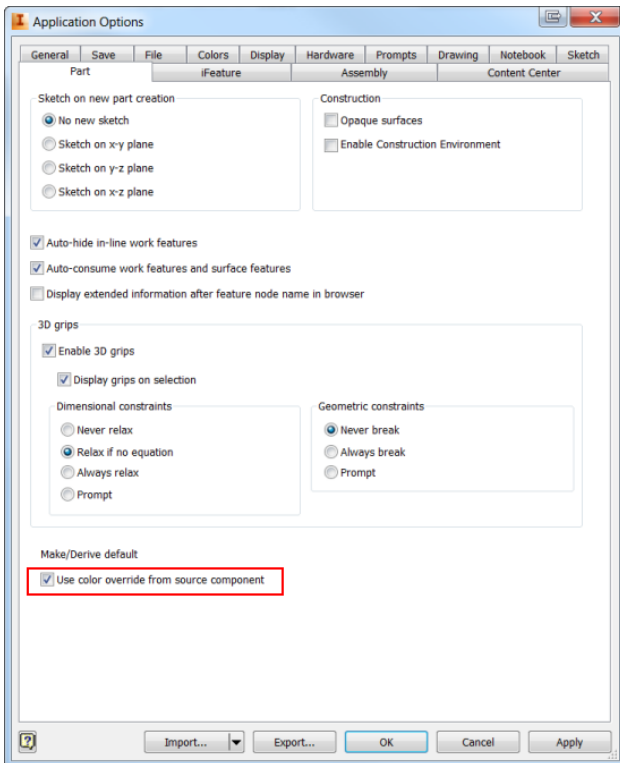
Tube and Pipe Setting Enhancements (not available in Inventor LT)

You can now customize file names for fittings and populate part numbers into your parts list within the drawing environment. Previously, you could only change the name of your conduit items, but now you can update fittings as well.

Updates to dialog box options within the Tube and Pipe environment:

Prompt for fitting file names (insert content as custom): By selecting this option, you are able to customize the name of your fittings within the Conduit File Names dialog box.

Enable Part Number Settings: Allows you to populate a part number into the parts list within the drawing environment. The part number is updated within the part file and then populates the parts list. This option also makes available the Part Number Settings options in the Conduit File Names dialog box.

Part Number Settings: The Part Number Settings field is made available by selecting the Enable Part Number Settings check box in the Tube and Pipe Settings dialog box. New options within this dialog box are:

- Inherit from Content Center: Uses the name provided by the Content Center Library as your file name.
- Use File Name: Allows you to customize file names, later allowing you the option to populate the information to the parts list within the drawing environment.
- Empty: Available in Inventor 2014. If selected, the File name remains empty.
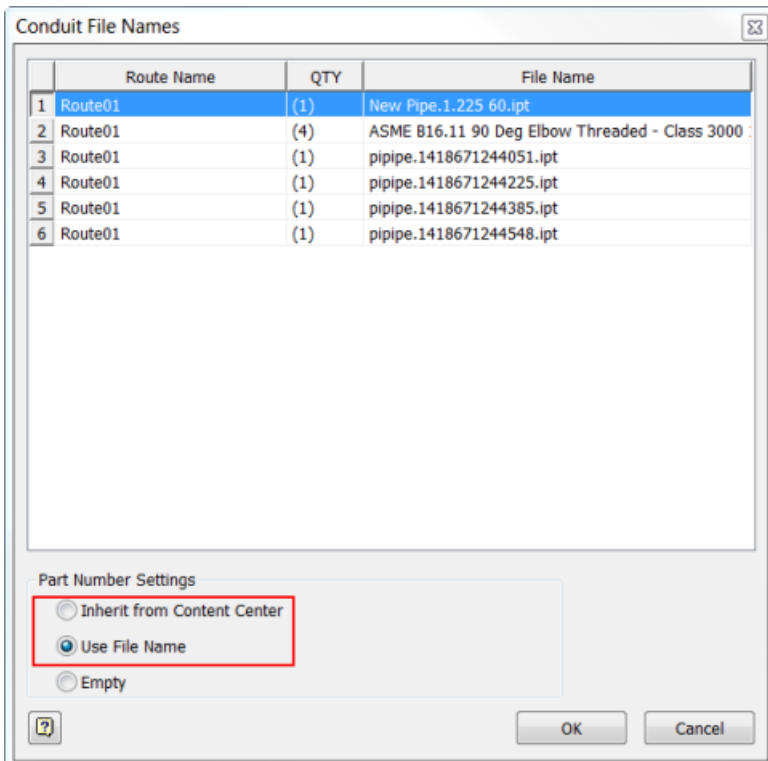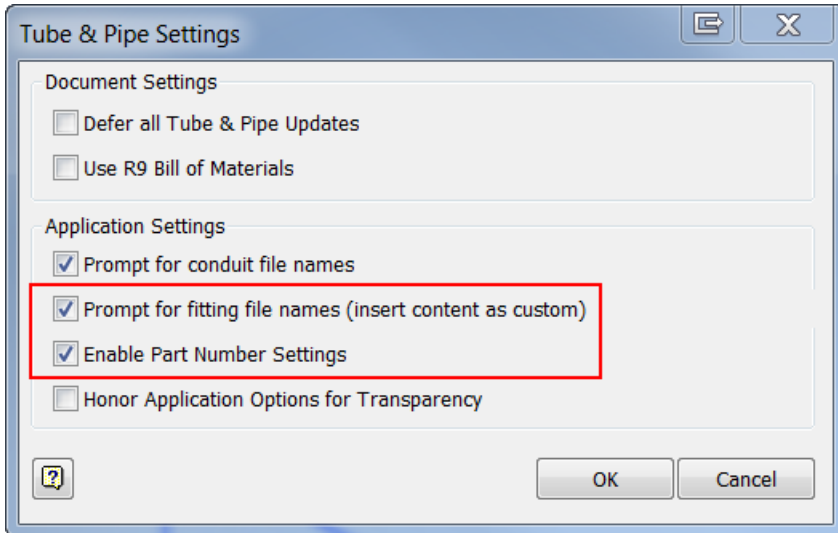
Additive Manufacturing
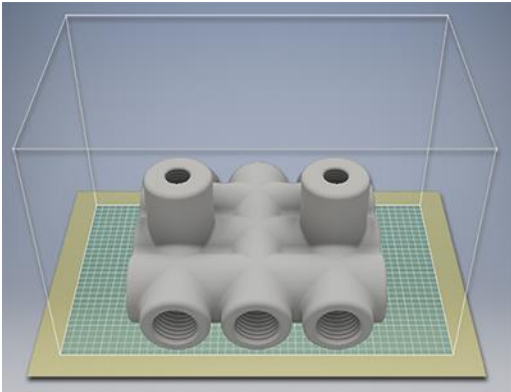
The 3D Print Environment provides tools for additive manufacturing or 3D printing. Models can be printed whole or in-part based on the 3D printer in use. The 3D Print environment is available only in the Part document. To print an assembly, first derive it into a part model using the option to keep each solid body as a separate body. In the derived part use the 3D Print Environment.

The 3D Print output is based on component or feature visibility. If you have features you do not want to print, change the visibility state or suppress the feature if doing so does not affect other features you want to print.

A list of 3D Printers is included. You can modify the list and specify a default 3D printer.

In the 3D Print environment you can:

Select the printer and print options you want to use.

Rotate the model into position relative to the build space.
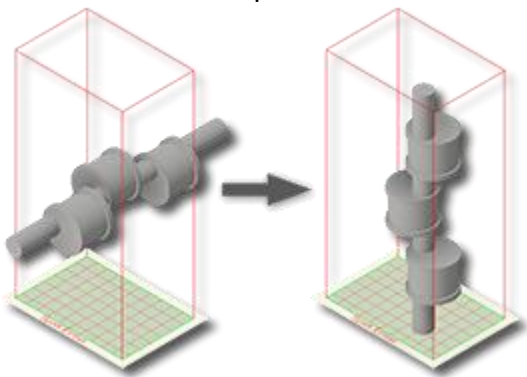
Position the model within the build space.
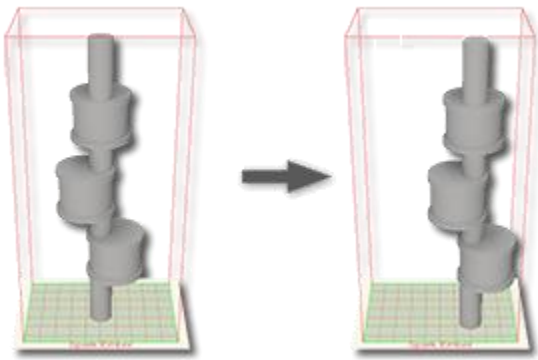
Partition components too large for the build space and place alignment features so the parts are easily assembled after printing.
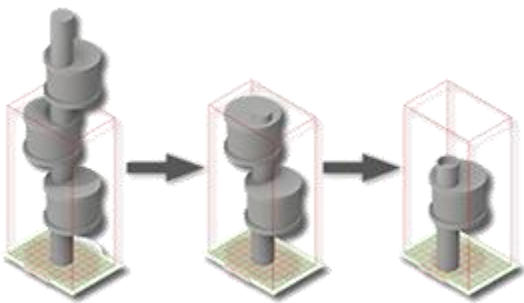
Use modeling commands, such as direct edit, to adjust features according to the needs of the printer or process. You can control quantity and location of alignment features and their size parameters.

Explicitly edit an alignment feature by exploding the partition and editing the sub-features. The explode action is persistent.
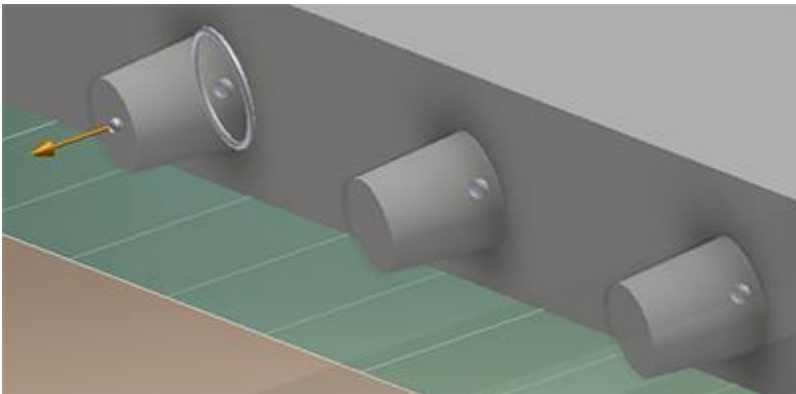
Parts
Sheet Metal Parts

Sheet Metal Multi-body

The multi-body part environment is extended to include sheet metal parts. You can now use a top-down workflow to create multi-body sheet metal parts. Use the Make Components or Make Part command and then open the derived components to create flat patterns for each part in a multi-body part file.
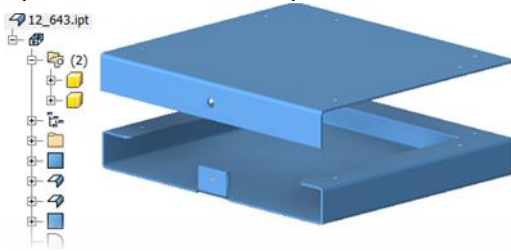
The New solid body option is available in the following commands:
- Face
- Contour Flange
- Contour Roll
- Lofted Flange
- Split

The following features support multi-body workflows:
- Flange, Corner Hem, Bend
- Cut, Fold, Rip, Punch

Zero Bend Radius
Support for zero bend radius is added to the following commands:

- Face
- Bend
- Flange
- Contour Roll
- Lofted Flange
- Hem
- Fold
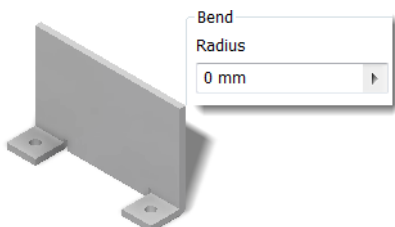
Punch Tool

The quantity of punch tool instances is added to the PunchTool dialog box, Geometry tab.
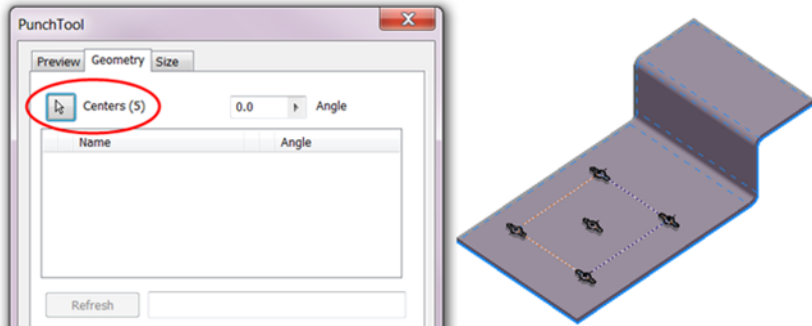
Thickness Detection

When you convert a part to sheet metal, you are now prompted to select a base face. This pick allows Inventor to measure the material thickness for you. After you select a face, the Sheet Metal Defaults dialog box appears. The measured thickness is displayed in the Thickness value box. Click OK to accept the measured thickness value.
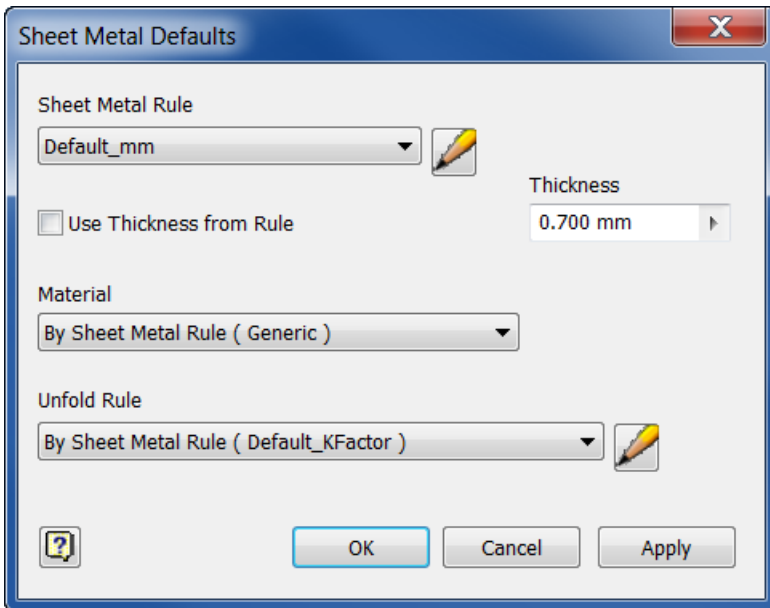
Dismiss Warning

You can permanently dismiss the warning message when you make edits to the flat pattern.
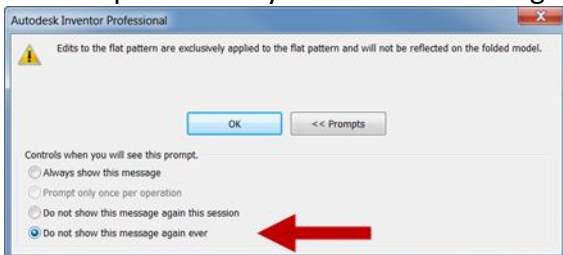
Export DWG and DXF

Some shop tools were unable to use an exported DWG or DXF flat pattern file if a bend centerline crossed a hole in the bend. A new option is added to the Geometry tab in the Flat Pattern DXF or DWG Export Options dialog box. Check the new Trim Centerlines at Contour option in the Geometry tab to trim bend centerlines to the edge of the cut.
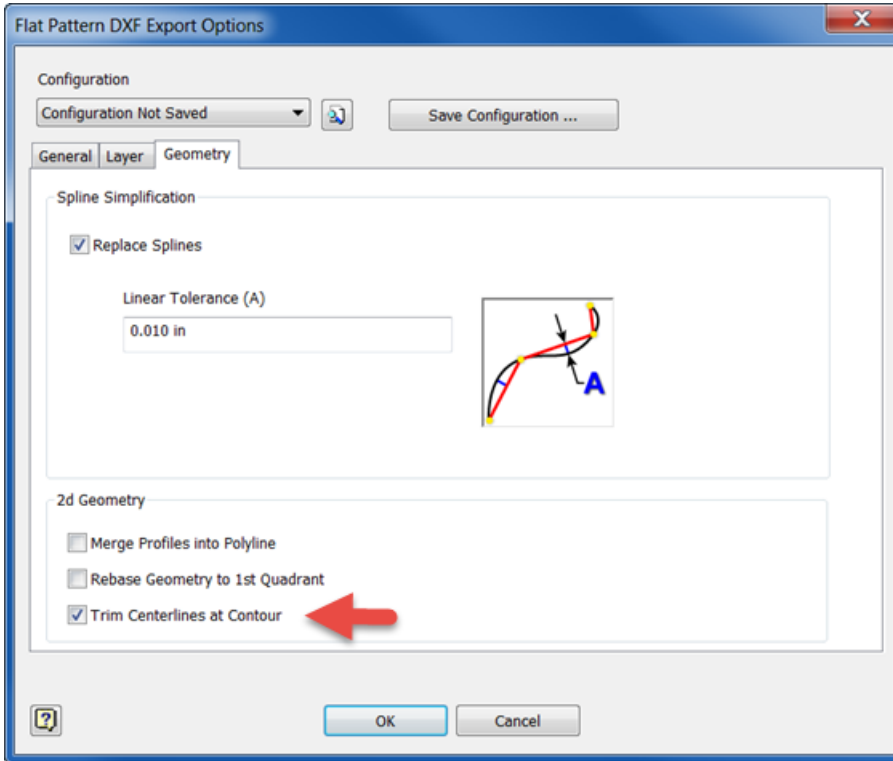
Freeform Parts

Freeform Environment

When you start a Freeform command a Freeform ribbon tab is activated providing full access to the environment.

Work Features are added to the Freeform ribbon and can be created in the Freeform environment. For example, create a work plane to use as a flatten reference or create a work plane to use as an alignment plane.

When you exit the Freeform environment the topology is simplified. Simplification means that there are fewer faces and edges in the model, but the freeform body shape is preserved.
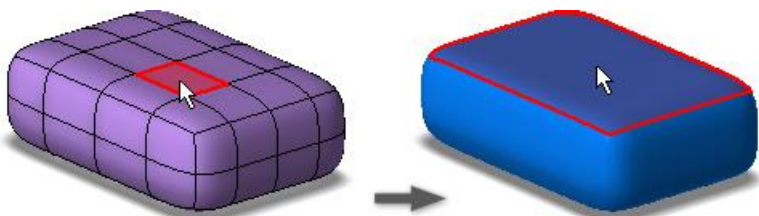
Edit Form

The following enhancements are added to the Edit Form dialog box:

- The dialog box is reorganized for better access to options.
- The Extrude option is added to the dialog box. Both previous extrude workflows; holding the ALT key and using the context menu when editing, are preserved.
- A Reset option is added next to Undo and Redo.
- The **Soft Modification** option is added. Set the type and the falloff gradient to control the influence your changes have over the surrounding area. In the following image, soft modification is off in the upper right image, and on in the lower right image.
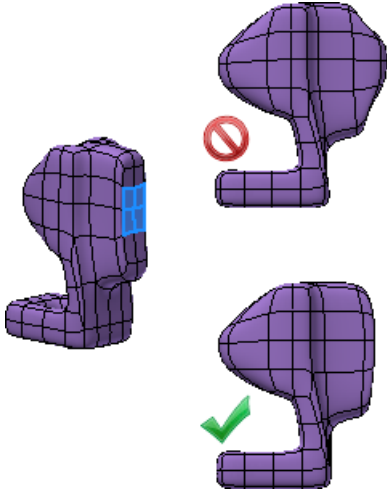
Freeform Plane

Use the Freeform Plane command to create a planar sheet body and then manipulate the faces.
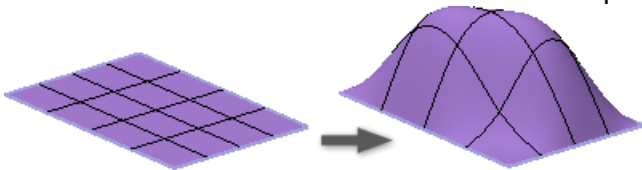
Freeform Cylinder

The Cylinder command supports creation of closed or open cylinder bodies.
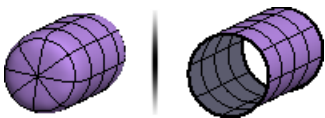
Edit Form

The Edit Form command supports extruding open sheet edges and faces.
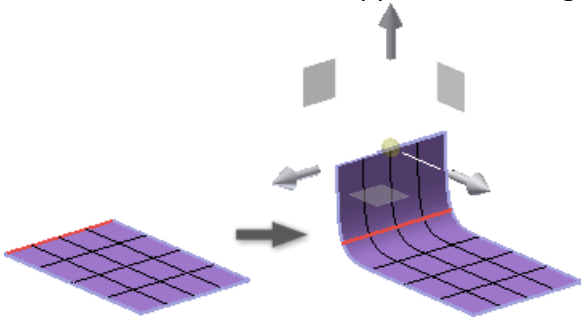
Freeform Face

Use the Freeform Face command to create a three-dimensional sheet body and then use Edit Form or Thicken to extrude the face or selected edges.
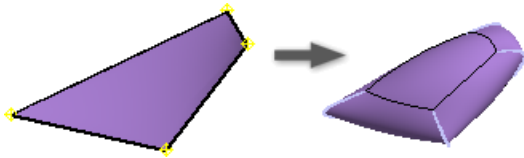
Delete Command

The Delete command now supports the Delete key. You can also delete faces to create an open sheet body. Delete one or more faces from a freeform shape, and then use the Freeform Face command to close the opening.
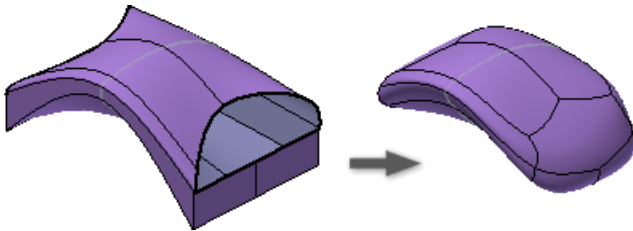
Convert to Freeform

Use the Convert command to copy solid body or surface faces to the freeform environment. Use the freeform edit tools to manipulate the shape.
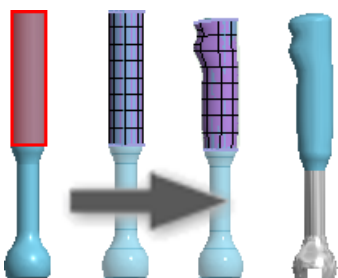
Body Management

Intelligent body management. Freeform bodies and surfaces are converted to their respective solids or surfaces when you exit the environment.
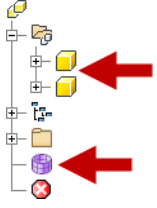
Align Form

Use Align Form to precisely move a freeform body to a target plane or planar face.
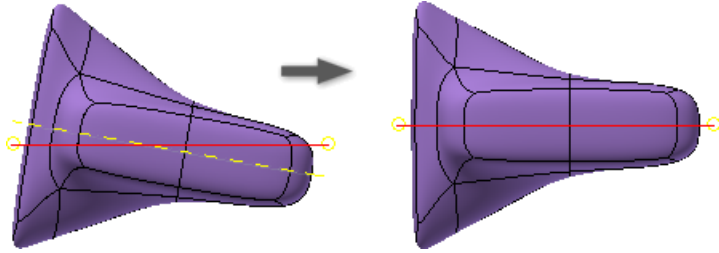
Flatten

Use Flatten to move the selected vertices to single plane passing through the vertices. Select at least four vertices to use Flatten.
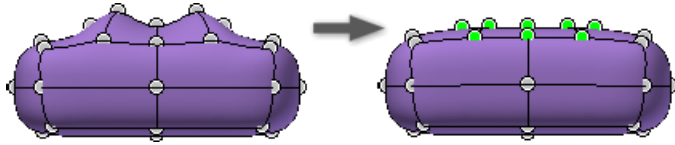
Weld Vertices

Use Weld Vertices to join vertices or to close a hole in a freeform body. Pick points individually to move the first vertex to the second vertex.
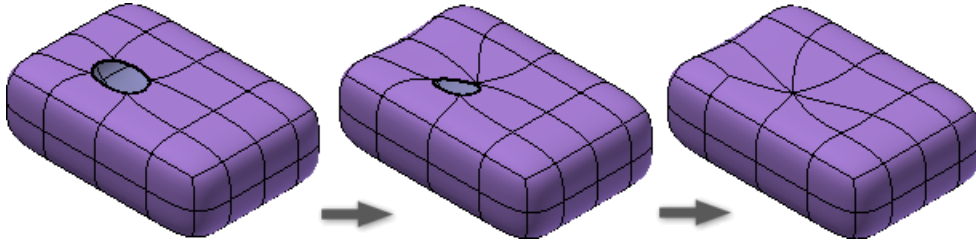
Unweld Edges

Use Unweld Edges to disconnect an edge or a loop to enable separating the edges.
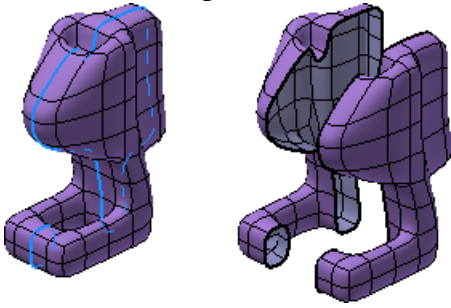
Insert Point

Use **Insert Point** to add control points to your model.
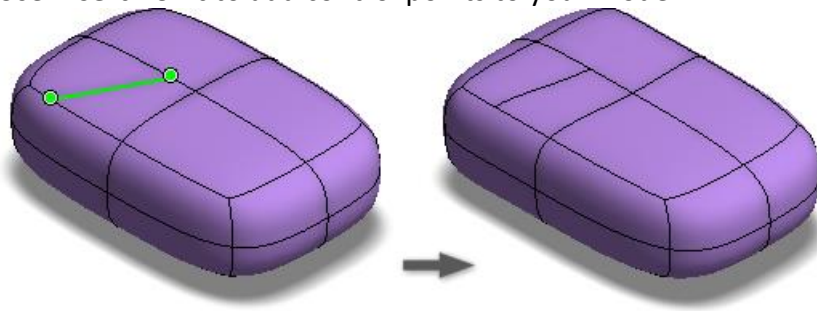
Merge Edges

Use Merge Edges to close gaps and join edges.
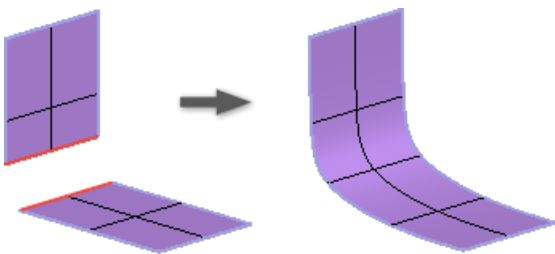
Thicken

The Thicken command is added to the freeform environment. You can choose Sharp, Soft, and No Edge solutions to thicken a body. The following image shows an open sheet body thickened with the No Edge option.
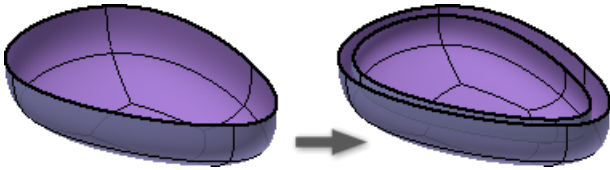
Mirror

The Mirror command is added to the freeform environment. You can use a work plane or planar face to mirror a body.
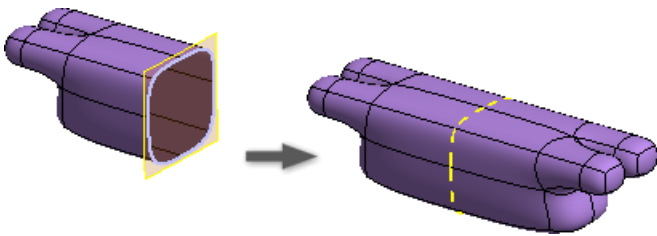
Crease and Uncrease

The Crease and Uncrease commands are available in the ribbon. Previously they were available only in a context menu. Creased edges are colored.
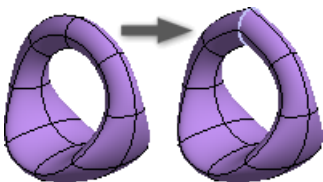
Bridge

The Bridge command has added the ability to select edges.
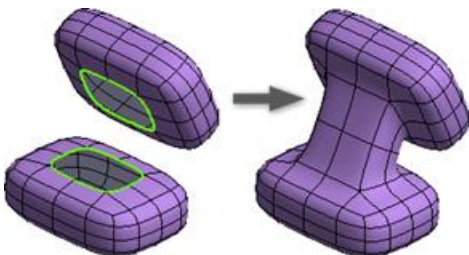
Add Distance

Use the Add Distance command to place a reference dimension between points, planes, edges, and faces.
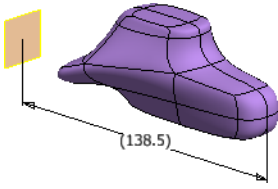
Make Uniform

Use the **Make Uniform** command to relax the freeform body and make the intervals between star points uniform. You can also use this command as a repair operation if you encounter issues with the body.
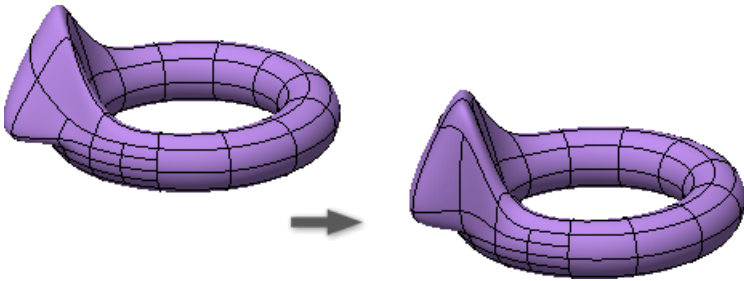
Copy

The Copy command supports copying bodies and faces. The Edit Form, Transform option is active when you use Paste so you can easily move the duplicated geometry.
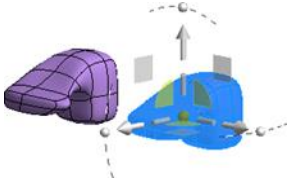
Toggle Translucent

Enable the Toggle Translucent command to see though the body and view existing geometry or sketches.
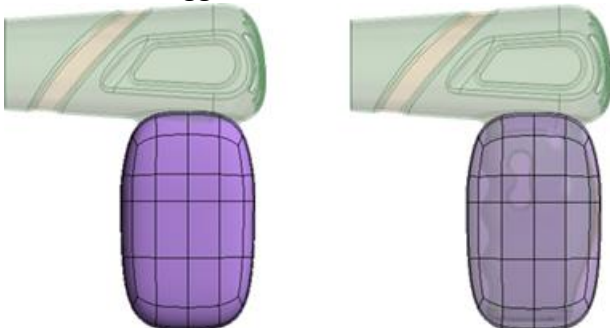
Toggle Select Through

Use Toggle Select Through to filter crossing window geometry selection. When enabled, geometry selection is normal. When toggled off, only geometry that is visible in the view is selected.
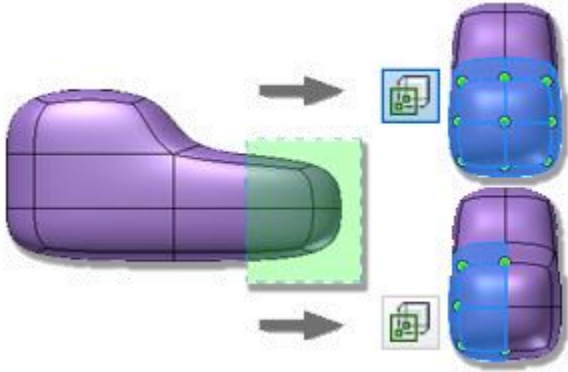
Match Edge

The **Match Edge** command is more robust and works with open sheet bodies. In addition, the following new functionality is added:

- Expand the Matches folder in the browser, right-click the matched edge, and select Rematch if the source geometry changes.
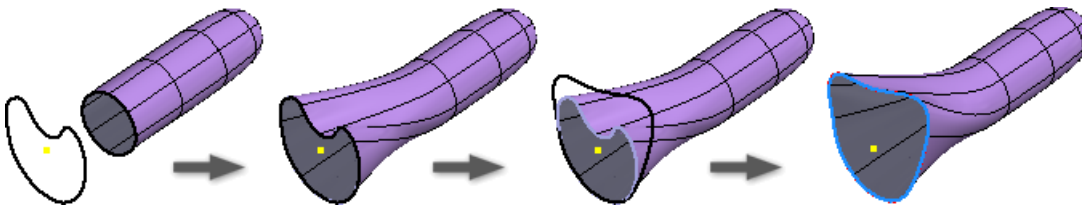
- G0, G1, and G2 continuity control is provided between a NURB edge and a freeform body edge.
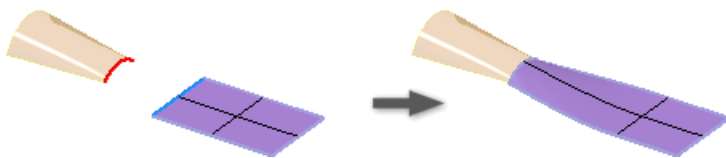
Selection

Freeform selection is enhanced to include the following:
- You can now window select vertices.
- Vertex display is enabled on bodies when you start a command that requires vertex selection.
- Select an edge, press Shift and double-click another edge in the loop range to add all edges between the two selections.
- Select a face, press Shift and double click another face in the loop range to add all faces between the two selections.
- Select a vertex, press Shift and double-click another vertex in the loop range to add all vertices between the two selections.

Symmetry

Symmetry is displayed with a different line type and color.
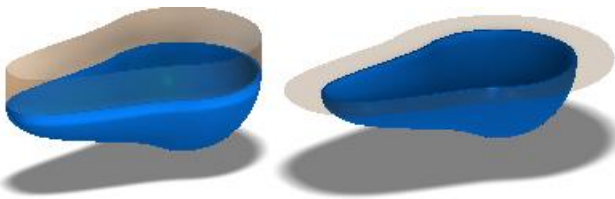
Part Design

Face Draft

Face draft enhancements allow you to control which part of the model thickness is preserved.
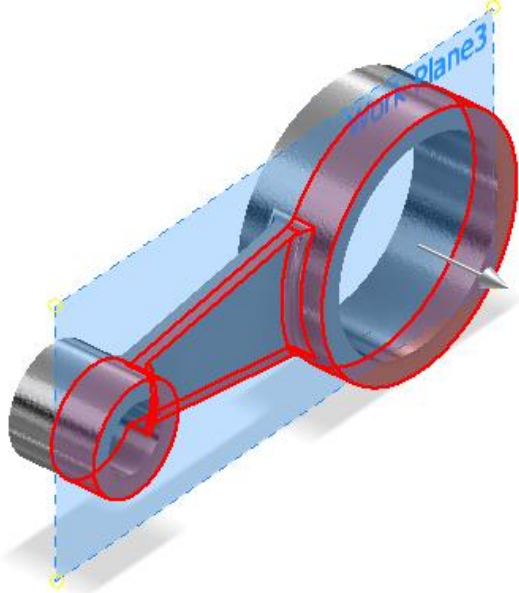
Ruled Surface

The Ruled Surface command is added to create Normal, Tangent, or Sweep surfaces. Ruled surfaces are used to create pockets, split a complex body after shelling, or to create a parting surface for mold design.
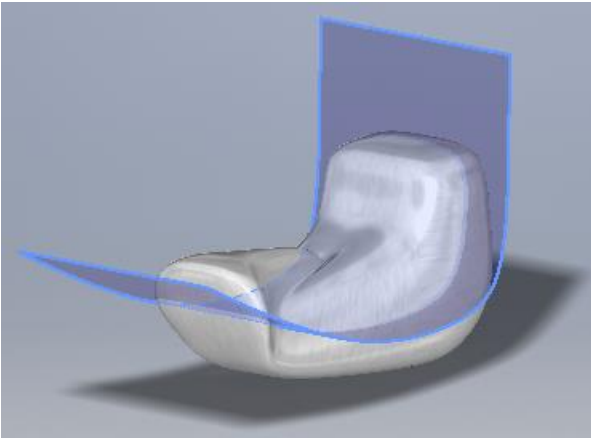

Silhouette Curve

New exclude options are added to the Silhouette Curve command to provide greater control of curve generation.
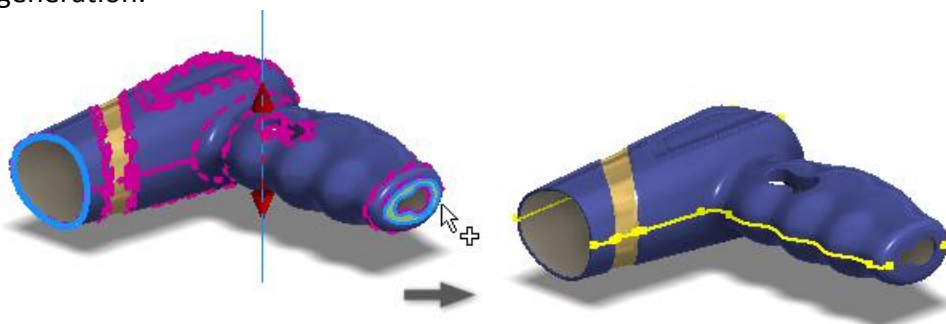
Parting Line

Surfaces can be used to generate a parting line split.

Replace Face

Replace Face supports both surface and solid faces as replacement faces.
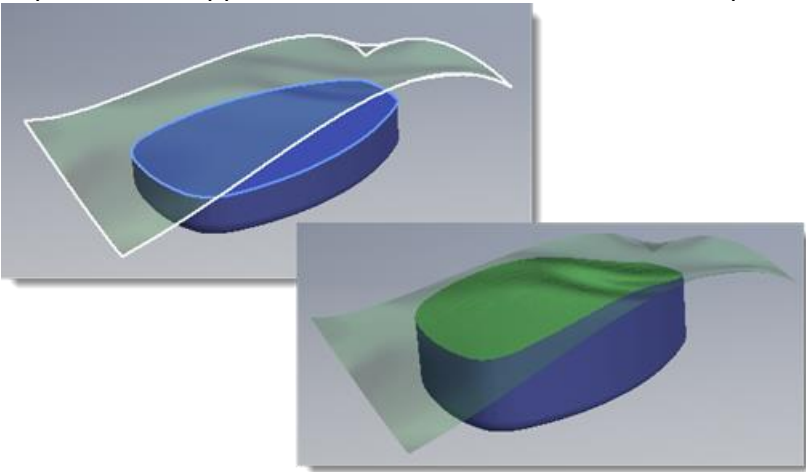
General Design

- The **Mirror** and **Rectangular Pattern** commands now support selecting multiple solid bodies. Use these new options to create new bodies or join the bodies.
- The **Measure** command now allows you to measure an angle to the midpoint of any segment. This option is achieved by hovering your mouse over the midpoint of a segment until a yellow dot appears.
- You can now use the **Rectangular Pattern** command to create nonlinear solid body patterns.
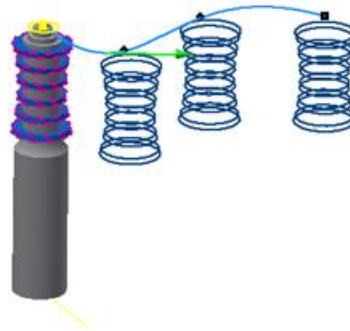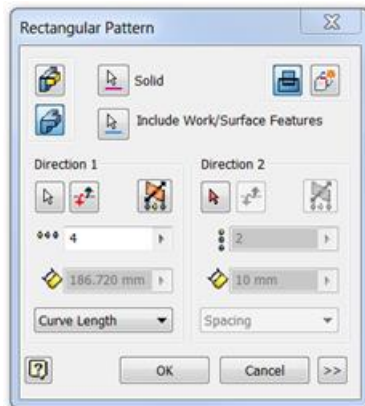

Enhancements to Boundary Patch

The Boundary Patch command no longer requires the input edges to form a closed loop(s) to preview and create a patch.

The Boundary Patch command now tries to create a patch whenever it is possible. If the input edges do not form a closed loop, it creates a patch that is large enough to encompass all input edges. It is no longer necessary for the boundary of the open loop patch to correspond to input edges.

The user interface is unchanged. You can still specify continuity constraints (G1/G2) and tangent scale values for each input edge. The OK and Apply buttons are enabled when it is possible to create a patch from input edges. You do not need to specify whether an open or closed loop patch is created. The Boundary Patch command automatically determines what to create.

If feature preview is enabled, the preview gets updated every time when you select or deselect an edge as shown in the following images going from left to right: 1) One edge is selected, 2) A second edge is added, 3) A third edge is added and a closed loop is formed.
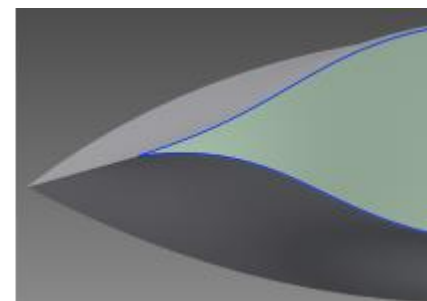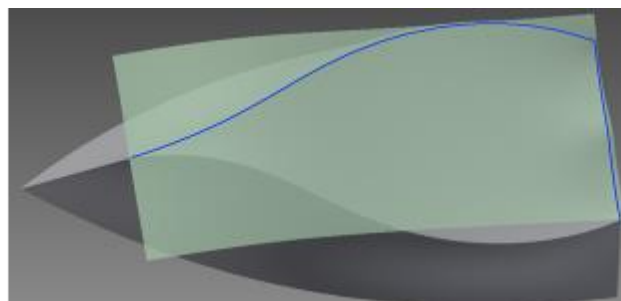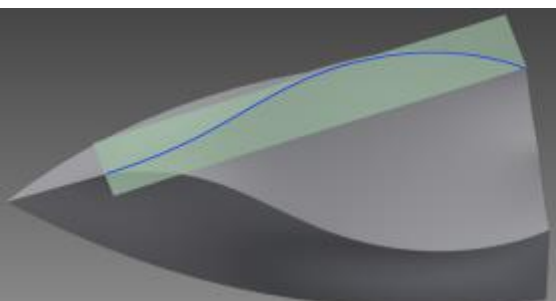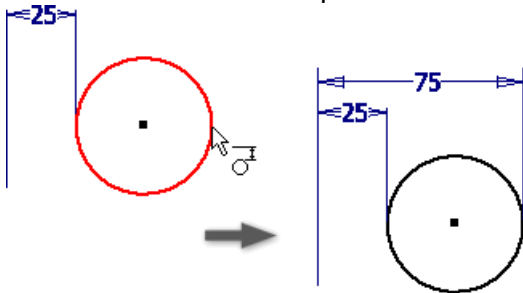
Sketch

Sketch

Identify the Sketch Source Plane

You can now identify which face or plane a sketch was initially created on. Select a sketch in the browser and select Show Input in the context menu to display the source plane or face.

Create Tangent Dimensions Between Circular or Arc Geometry Within a 2D Sketch

You can now create tangent dimensions between circular or arc geometry within a 2D sketch. This workflow will allow you to fully constrain a circle using tangent dimensions instead of a traditional diameter dimension, or the use of the center point of the circle.


The Show/Hide All Constraints Display Setting Persists

The selected Show All or Hide All Constraints display setting remains active as you sketch and throughout your editing session. Previously the show or hide setting did not persist when you created new geometry or when you started a new sketch.
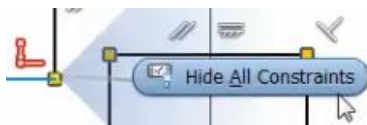
Hide Sketch Dimensions

The Sketch Dimensions option is added to the Object Visibility list. Select this option to display 2D or 3D sketches and hide all related sketch dimensions. Previously, you could not hide sketch dimensions separately. This option is available in both parts and assemblies.
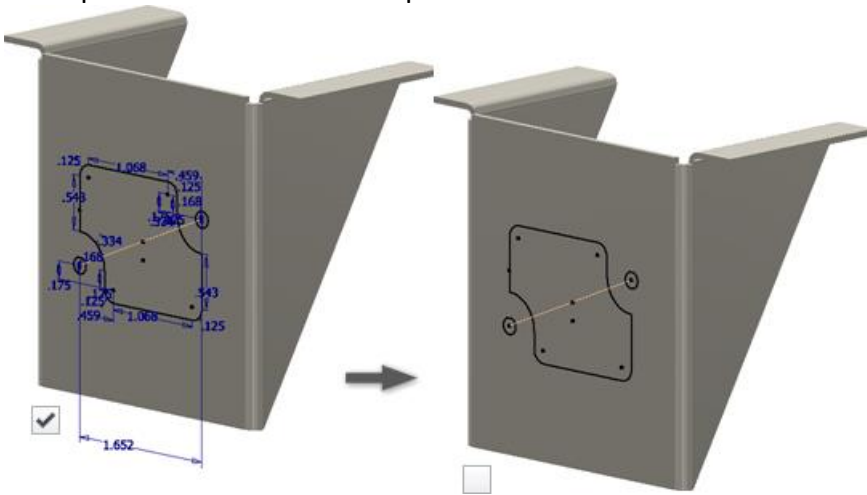

Consistency in Sketch Sharing

When sharing a sketch, by dragging it above its feature within the browser, it is now automatically made visible. The visibility state of the shared sketch is now consistent with the context menu method of choosing Share Sketch.

3D Sketch Wrap to Surface

The Wrap to Surface option of the Project to Surface command is enhanced. Wrap to Surface is more predictable and can now wrap to a wider variety of surfaces. The following image shows the results of wrapping a straight line onto two planes and a curve in both 2015 and 2016.
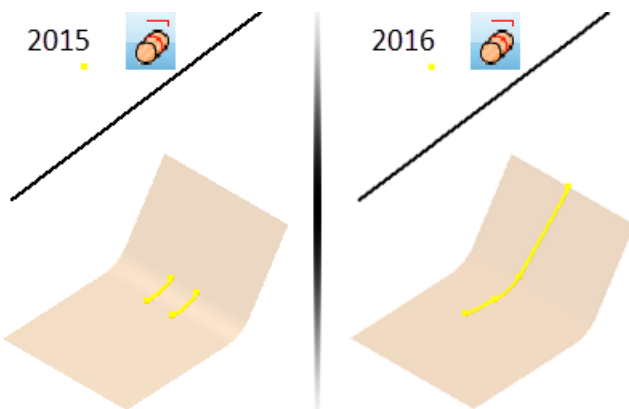
New Snap Points in the Context Menu

Previously, the only snap points that were available through the right-click > Point Snaps option were Midpoint, Center, and Intersection. The following snap points are added:

- Endpoint
- Apparent Intersection
- Quadrant
- Mid of 2 points

Endpoint
Midpoint
Intersection
Apparent Intersection

Center
Quadrant
Tangent

Mid of 2 points
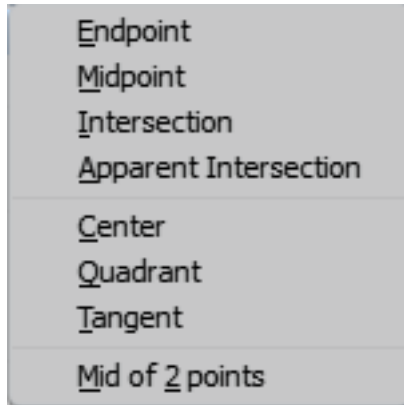
3D Sketch Silhouette Curve

The create silhouette curve command includes new options. You can now choose to exclude selected faces, straight faces, and internal faces from curve generation.
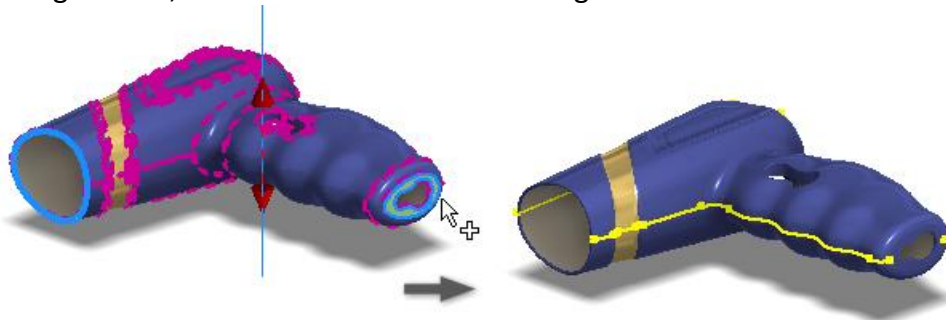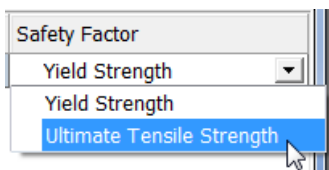
Assembly

Assembly

Stress Analysis

Changing the Safety Factor for materials triggers a new warning message. While assigning materials In Stress Analysis, if you change Safety Factor from Yield Strength to Ultimate Tensile Strength (UTS), a warning message displays. The warning advises that the safety factor calculation of one or more materials is based on UTS. UTS uses Principal Stress and is appropriate for brittle materials.

Large Assembly Performance

The Disable Automatic Refinement option is added to the Application Options, Display tab. Disabling automatic refinement provides a coarser model display and improves graphics performance in large assemblies such as when changing a level of detail. Coarse faceting is used until you select the Refine Appearance command in the View tab.
Tip: Disabling refinement can also improve graphics performance in complex models.
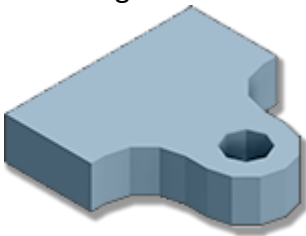

Replace Selected Components

The Replace command now supports more than a single component selection. Multiple similar or dissimilar component selections can be made before or after the Replace command is invoked.
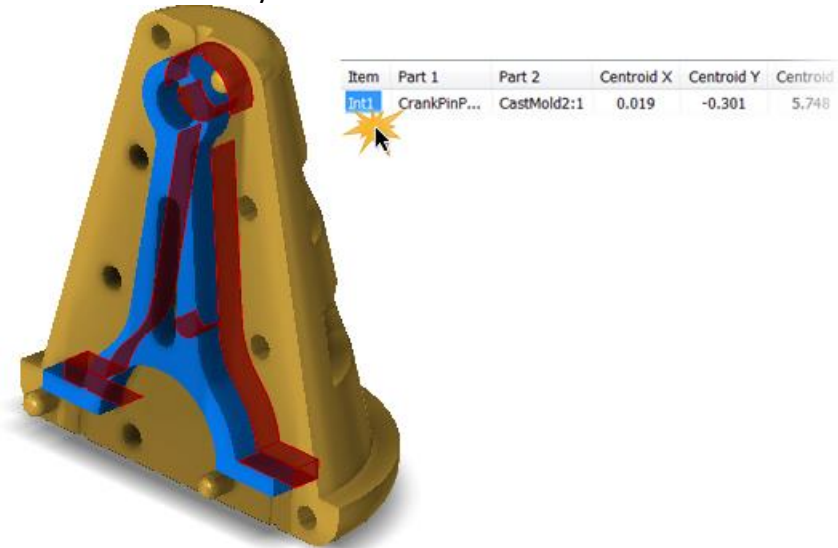
Analyze Interference

The following new functionality is added to the Analyze Interference workflow:

- You can use a crossing window to add components to the selection set. Previously you selected components one at a time.
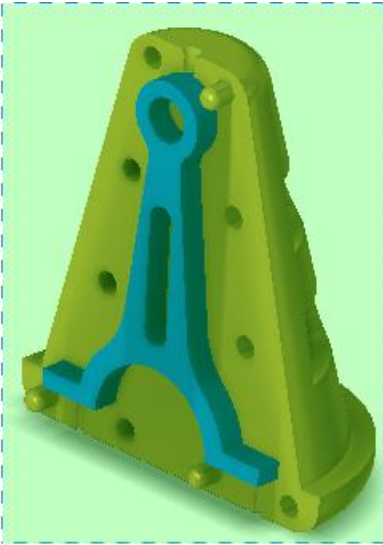

- You can now zoom to an interference. If interferences are detected, expand the dialog box and double-click an entry to zoom to the interference.

| Item	Part 1	Part 2	Centroid X	Centroid Y	Centroid
Int1	CrankPinP...	CastMold2:1	0.019	-0.301	5.748

Assembly Pattern

The Midplane option is added to the assembly Pattern command. Select the Midplane option to create a pattern distributed on both sides of the original component.
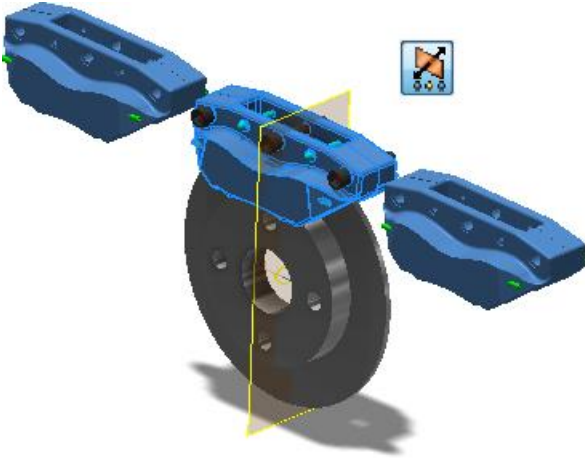
Act on Multiple Sick Constraints

You now can select multiple sick constraints within the Design Doctor and delete or suppress them.
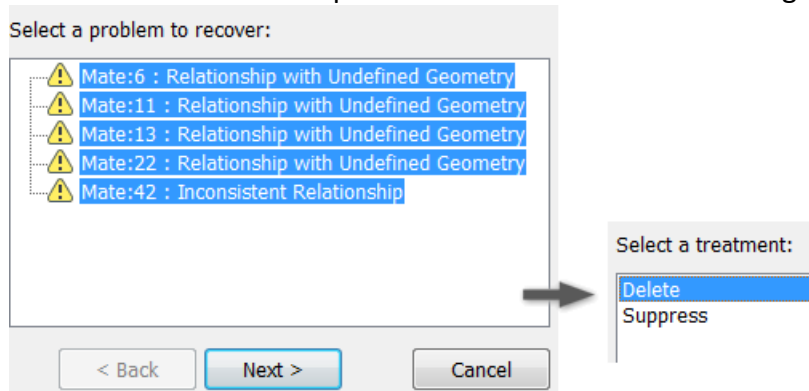
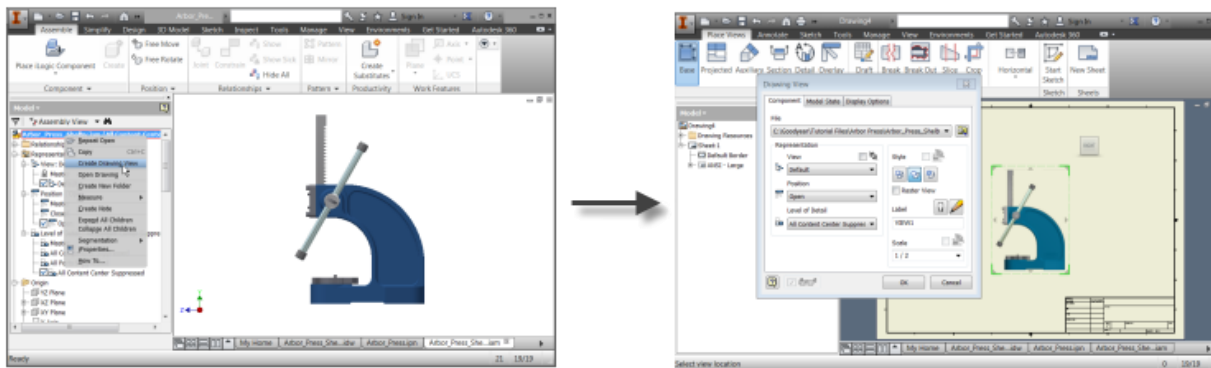# Drawings

General Enhancements in Drawings

Fast Drawing Creation

A new Create Drawing View command merges several steps to one, so you can quickly create drawing views from a model file. In the model file, arrange the model as desired. Then right-click the root component in the browser, and click Create Drawing View.

The following model properties are automatically propagated from the model to the drawing:
- All active representations (design view, position, and level of detail representation)
- Current camera including Orthogonal/Perspective setting
- Active iPart/iAssembly member

Fast drawing creation is supported in the following environments: Part, Weldment Assembly, Assembly, Presentation, Sheet Metal, and Flat Pattern. It also supports iPart, and iAssembly factories and member files.

Centralized External Sketch Symbol Library Features

You can now easily access and share your collection of sketch symbols through a new external Sketch Symbol Library. This library is an Inventor drawing file that, by default, is located in a subfolder of the Design Data location of your project.

Sketch Symbols dialog box
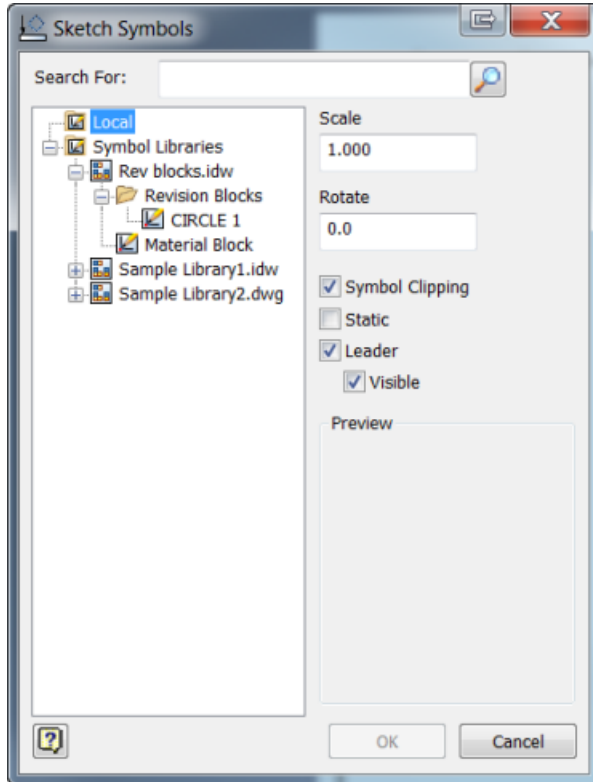
Search and filter function

You can now search and filter your libraries with more accurate results when inserting sketch symbols from a Symbol Library.

Preview sketch symbol

You can now see a preview of your selected sketch symbol before placing it.

Enhancement to browser expansion state when placing sketch symbols

The library's browser expansion is now remembered when you select a symbol from the specified library and select OK . It will remain expanded in the last viewed position, based on the state it was left in before closing the dialog box. This will be remembered throughout your session.

For example, when you select OK, the next time you insert a symbol from the library, the last sketch symbol that was inserted, will be selected automatically from within the last library accessed.

Both IDW and DWG file types are supported in your sketch symbol libraries

The ability to place IDW or DWG file formats into the sketch symbol library is now possible. A Sketch Symbol Library can be created in IDW and DWG formats allowing insertion of symbols contained within the library file from either format.
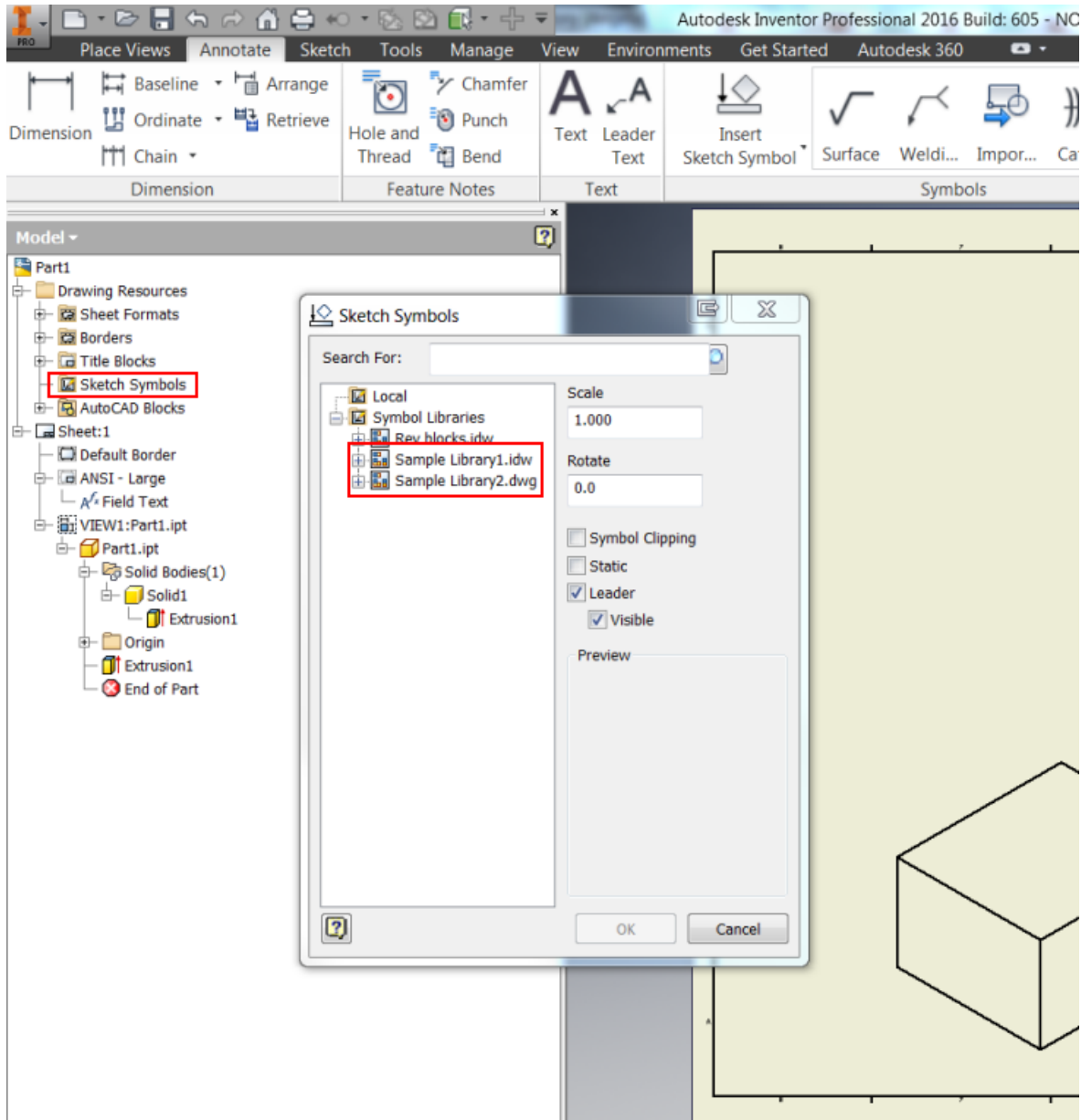

Save all to symbol libraries

To save all sketch symbols to a given library, right-click on the Sketch Symbol, Drawing Resources node in your browser and select Save All to Symbol Library.

Save to desired sketch symbol library

Save your sketch symbols to a library of your choice. Right-click on a symbol in the graphics from beneath a sheet or beneath the Sketch Symbols node under Drawing Resources to display the Save to Symbol Library dialog box. Here, you can choose which library to save your sketch symbols to. To create a library, select the Create a new library icon. To create a folder within a library, select the Create a new folder icon.

Retain your library's folder structure when saving your sketch symbols to a library. To accomplish this, click the Retain Resource Folder Structure check box.
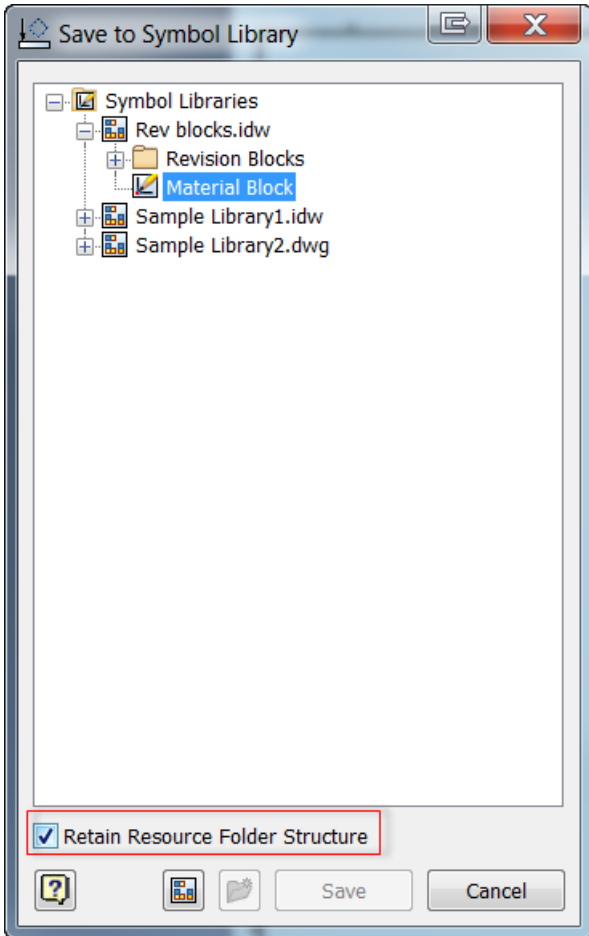
New Sketch Symbol Library folder entry available in Application Options dialog box

A new configuration entry is available in the Application Options dialog box under the file tab called Sketch Symbol Library folder. Here, you can specify a folder outside of the Design Data structure to store your sketch symbol library templates. Select the Browse icon to locate a file or Make New Folder to create a location to store your libraries.
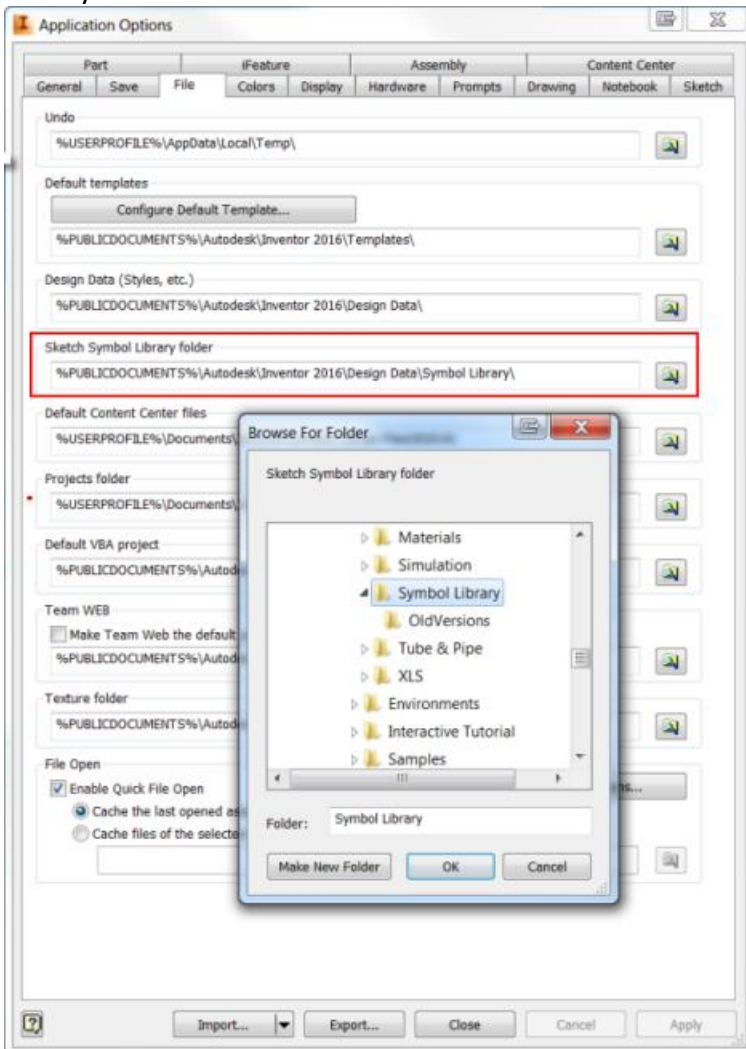

Initial View Scale Property in Title Block or Text

The Initial View Scale property scale of the first drawing review placed on the sheet is added in the Sheet Properties group within the Format Text dialog box. You can insert the Initial View Scale property to title blocks in your drawing templates or to Text objects in your drawing. Initial View Scale property value is not changed when you reorder views in the browser or delete the first drawing view.

Work Features Can Now Be Projected Into a View Sketch

You can now create a view sketch on a model with included work features and select those work features with the Project Geometry command. This capability in a drawing view sketch allows you to reference, constrain, or annotate other sketch geometry to projected work features that were included in the view.
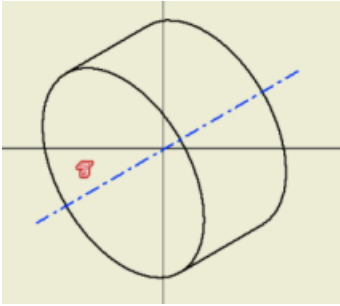
# Hidden Line Views for Standard Fasteners

A new option is available to exclude all standard fasteners from hidden line views.
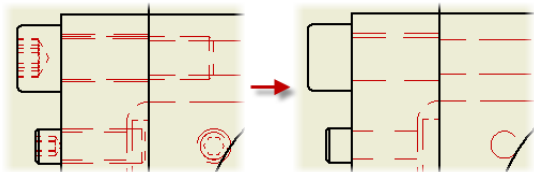
In the Drawing View dialog box, on the Display Options tab, choose a Hidden Lines option:
- Select Never to remove hidden lines for all standard fasteners.
- Select Obey Browser to respect the browser Hidden Lines setting for each standard fastener.

Note: The Hidden Lines option applies only on Content Center parts from the Fasteners folder and Sheet Metal folder.
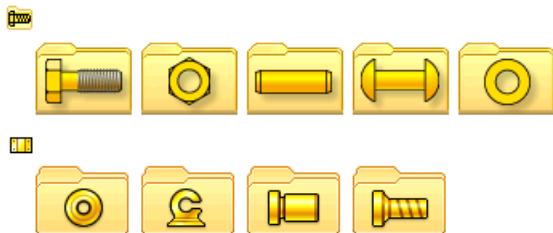
Transparent Components in Drawing Views

In the Drawing Browser, select the Transparent option in the component context menu to make a component transparent in a drawing view. Transparent components do not hide geometry of other components.

Note: Component transparency applies only to precise drawing views.
In the example below, the highlighted component is set as transparent in the drawing view.
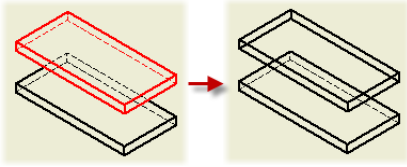
Workflow Modified to Start Sketch

The drawing Start Sketch command is modified to work similarly as the Start 2D Sketch command in the modeling environment. When you click Start Sketch, you have to select a sheet, drawing view, or existing sketch in the browser or graphic window. The new or edited sketch is highlighted in the browser and opened for editing.

If the current drawing sheet does not include any drawing views or sketches, a sheet sketch is started automatically.

Tip: Alternatively, you can select a sheet, drawing view, or sketch before you click Start Sketch.
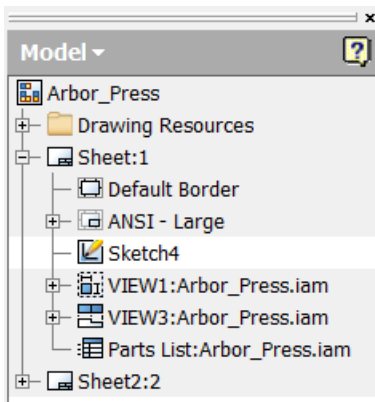
Base and Projected Views

Model Orientation in Base Views

When you create a base view, View Cube is displayed next to the upper right corner of the base view. Click a View Cube face, edge, or corner, or click the rotation manipulators to set the desired orientation of the model. The View Cube replaces the Orientation options from the Drawing View dialog box.
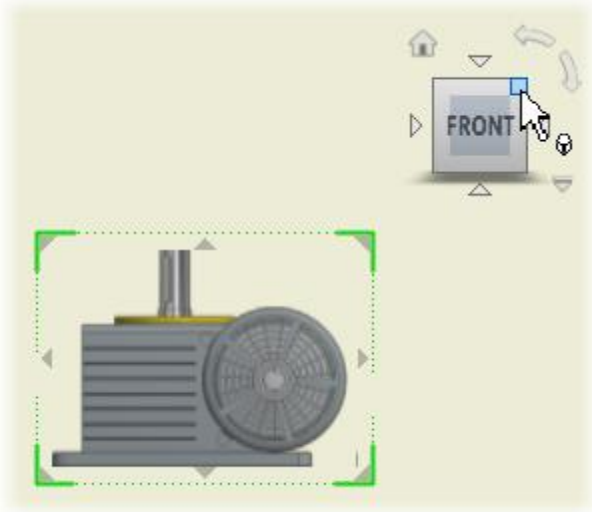

Temporary Base View Position

When you start the Base View command and specify the model, the base view is placed in a temporary position on the drawing sheet. Hover over the base view to display the Move manipulator. Then drag the base view to the desired place.
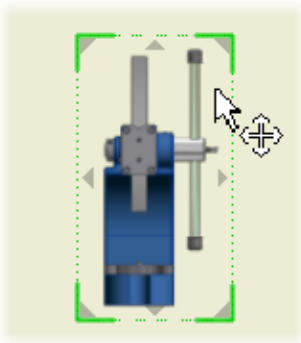
Custom View Orientation

Right-click the View Cube, and click Custom View Orientation to enter the Custom View environment. Then specify the orientation of the custom view.
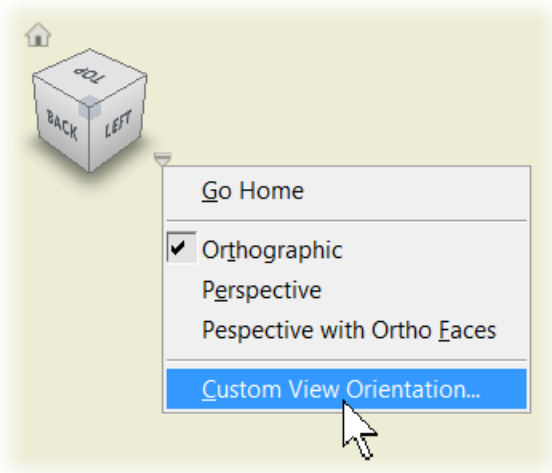
Orthographic and Perspective Projection

By default, Orthographic projection is used to create drawing views. To use Perspective projection, when you create a base view or edit a base view or independent projected view, right-click the view cube, and click Perspective.
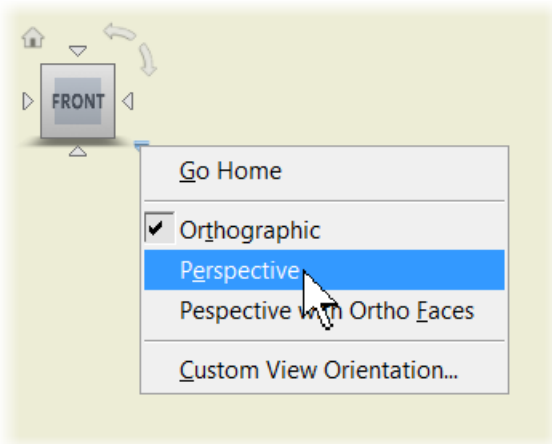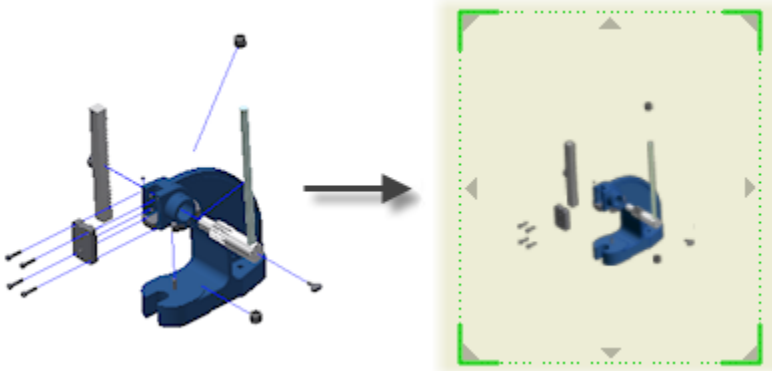
Saved Camera for Presentation Views

For drawing views based on a presentation view of the model, you can access and apply camera settings saved in the presentation file (.ipn). Right-click the View Cube, and click Saved Camera.
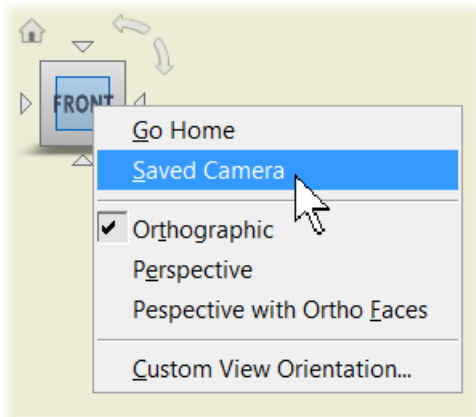
Model orientation in the drawing view changes to match orientation saved in the presentation view.
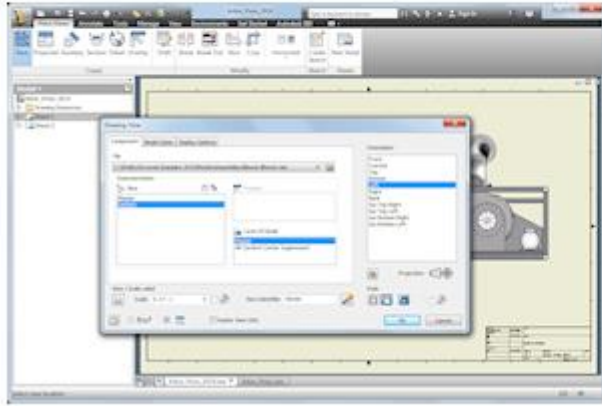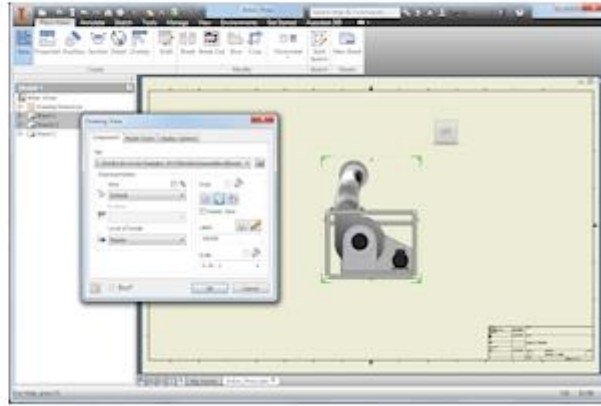
Revised Drawing View Dialog Box

The Drawing View dialog box is streamlined to support new workflows with direct interaction in the graphic window.

Size of the dialog box is reduced:

Some dialog box options are moved, changed, or removed:

Component tab

- Style controls, the Raster View option, and label and scale controls are moved from the base dialog box area to the Component tab.
- The Projected View option is removed and projected view creation is controlled directly in the graphics window.
- The View, Position, and Level of Details Representation controls are changed from list boxes to drop-down lists.
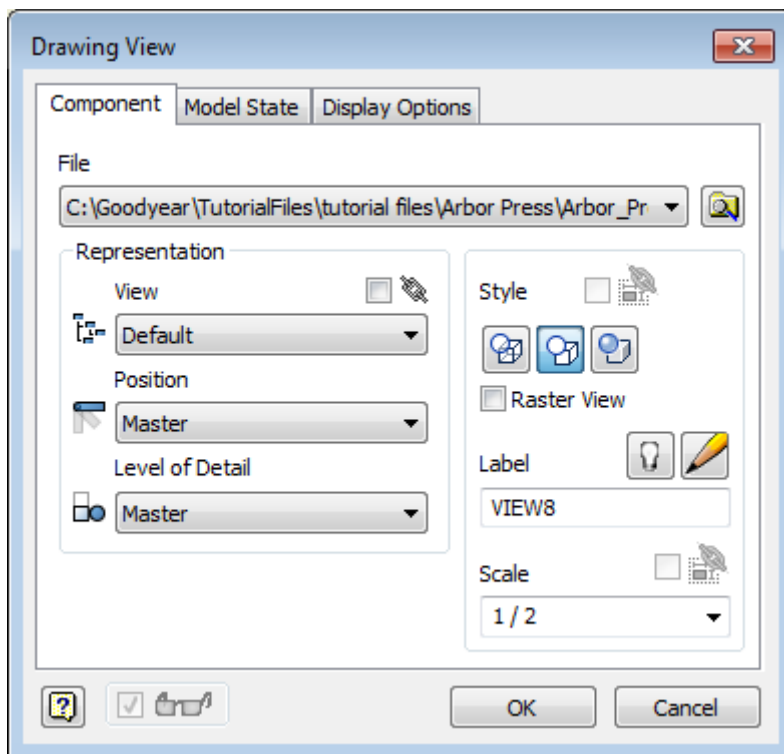
Display Options tab

- The most frequently used controls are on top.
- The All Model Dimensions option is visible only for Part models. The option is hidden and replaced with the Interference Edges option for other types of referenced files.
- Show Trails option is moved to the Components tab > Presentations frame.
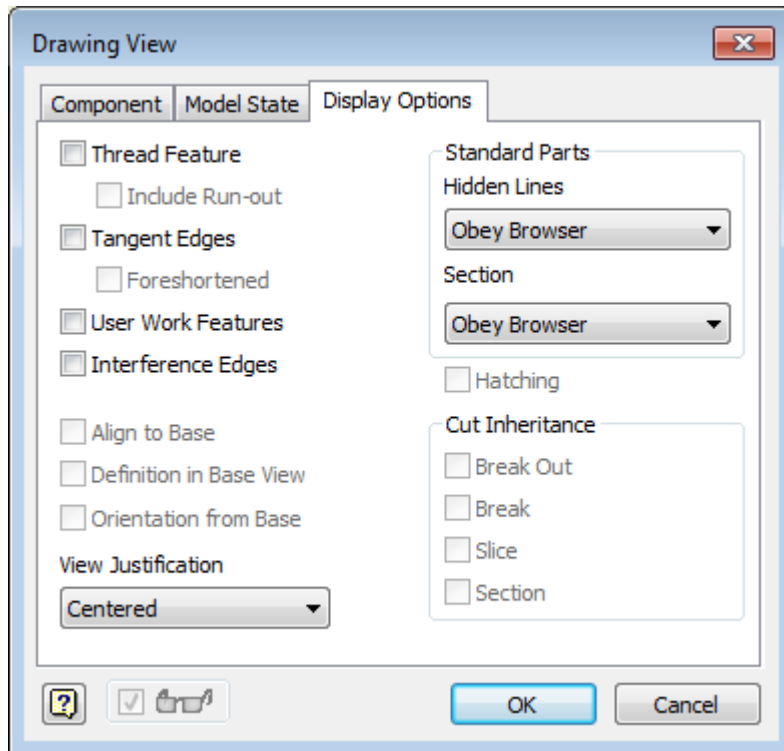
Model State tab

- Weldment controls are moved from the Display Options tab to the Model State tab to be colocated with the other weldment-specific options.
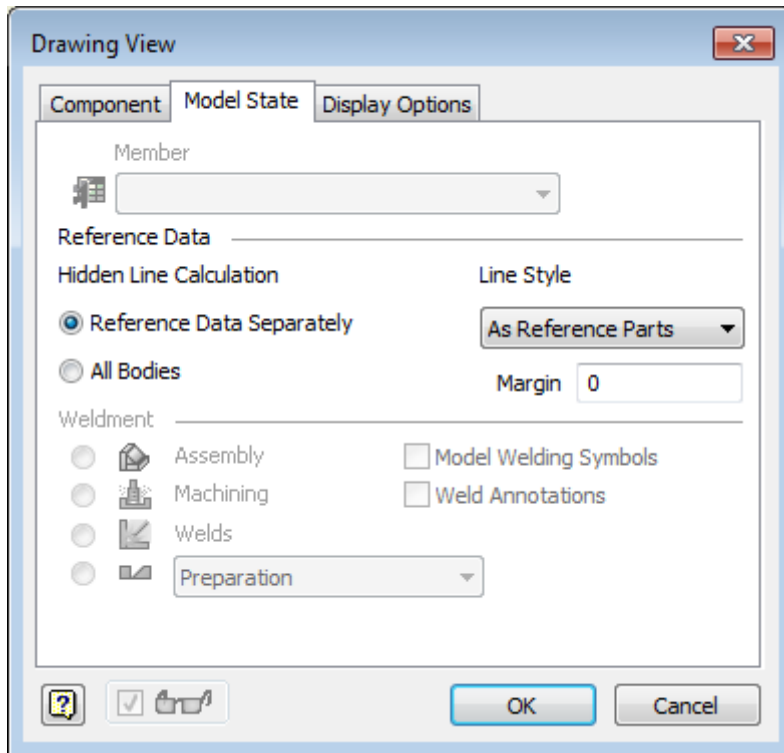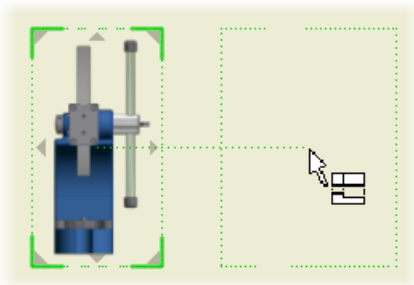
Projected Views

Use the direct interaction tools to create multiple projected views with the Base View command, and to place projected views in the drawing.
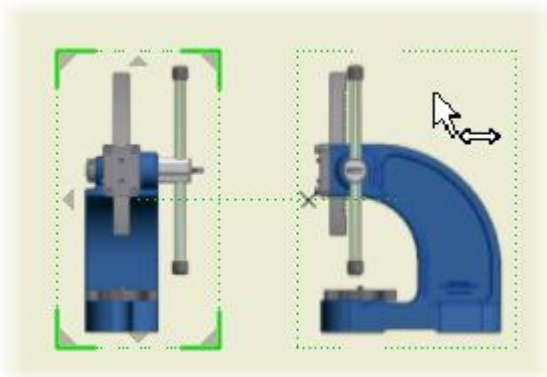
To create an orthographic or isometric projected view, click a triangular icon on the base view border. The projected view is placed in a default location.
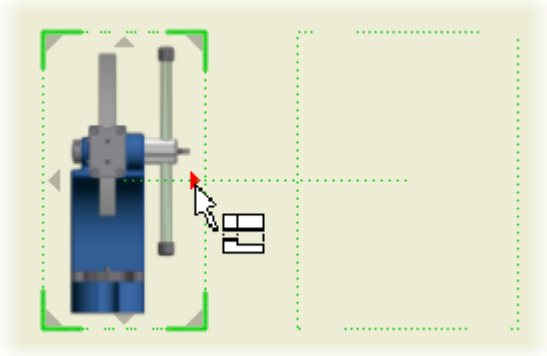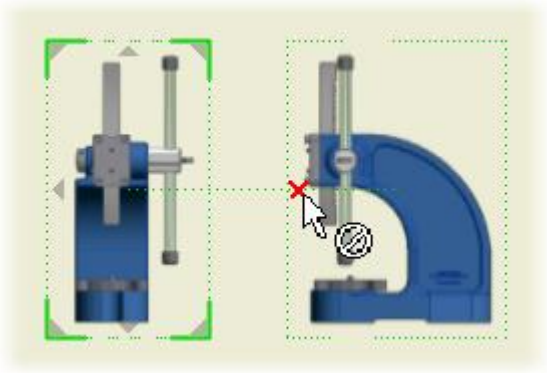

To add a projected view to a specific place on the sheet, drag the projected view bounding box to the desired location. Then click to create the view.

To move a projected view, drag the view to the desired position.

To remove a projected view, click the x mark.

Drawing View Scale

When you create a base view and projected views, a temporary scale value is used to fit the model view to the sheet. Drag a corner of the base view to change the view scale directly in the graphic window.

By default, within the range of scales predefined in the current Standard style, the view size snaps to the preset scale values. Hold the CTRL key to enable free dragging, and set arbitrary view scale.
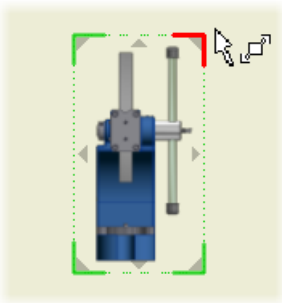
Child projected views are dynamically resized to respect the new base view scale.

# Editing of Base and Isometric Projected Views

When you edit a base view or an isometric projected view, direct-editing tools are available.

Use the View Cube to specify new orientation and type of projection for the view. Drag a view corner to change the scale.
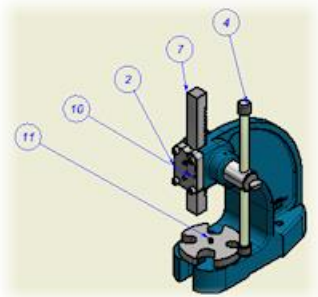
Balloons

Custom Balloons in Balloon Styles

You can use a sketch symbol as the default balloon shape in a balloon style. In the balloon style, switch on the Used-Defined Symbol button, and select a symbol from the list.
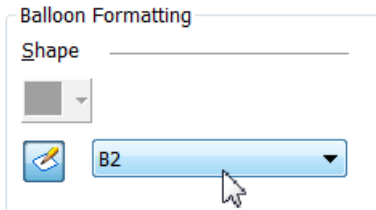
Tip: To make your custom balloon shapes available in all drawings within a project, add sketch symbols defining the custom balloons to the centralized sketch symbol library.

## Align To Edge Option

A new To Edge option is available to align balloons. You can select any linear edge on the drawing sheet to specify direction of the aligned balloons.

Corrected Split Balloons

When you insert a split balloon, the middle line is connected to the circle.

Migrated balloons are not automatically updated to the new type. Edit a legacy balloon to update its appearance.
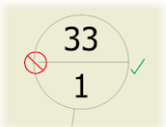
Window Selection for Aligned Balloons

Any window selection that includes balloons is valid, and displays Align commands in the right-click menu. After you start an alignment command, all objects that cannot be aligned are automatically removed from the selection set.
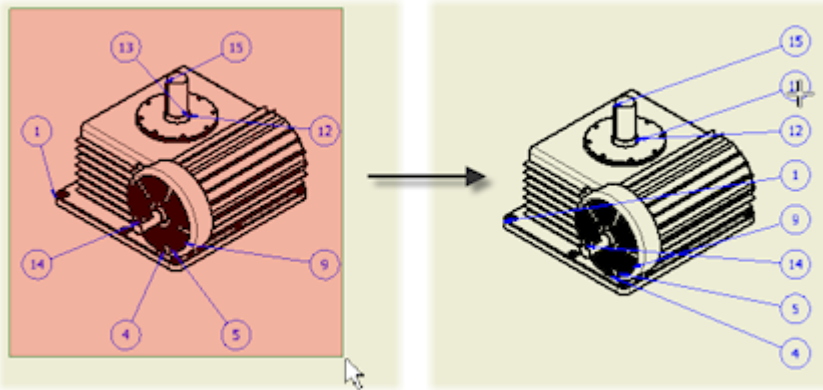
Improved Auto Balloon command

Placement of automatic balloons was improved to optimize the balloon start points, and position and length of balloon leaders.

Symbols and Symbol Styles

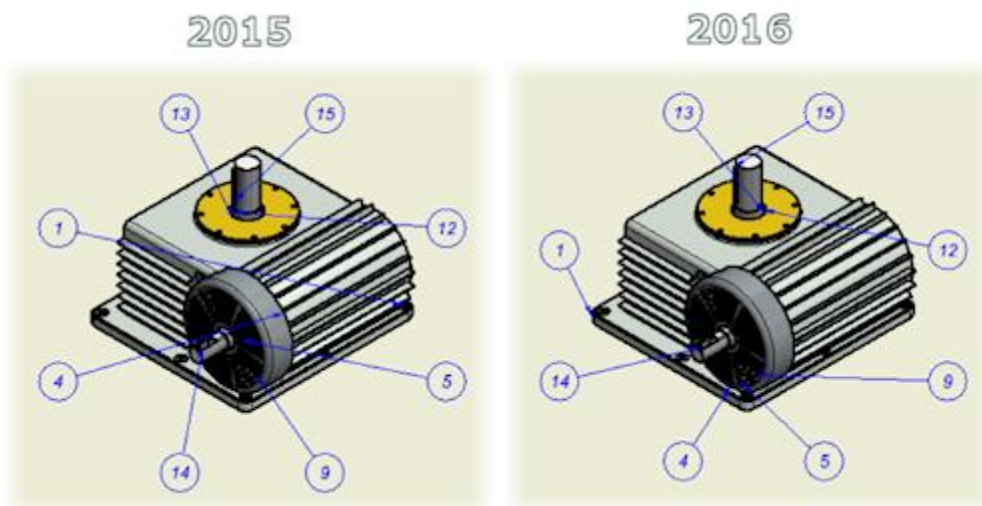## Enhanced Symbols List in Drawing Annotations

New graphical symbols are added to the list accessed by the Symbols button, and can be inserted in various types of drawing annotations.

The Symbols list is divided into six segments: Common Symbols, Geometric Characteristic, Material Removal Modifier, Additional Symbols, General Symbols, and Character Map.

Note: The active Feature Control Frame style controls the list of symbols available in drawing annotations.

Double-Click Editing for Symbols

Double-click editing is now available for more types of symbols:
- Surface Texture Symbols
- Feature Control Frames
- Weld Symbol

Surface Texture Symbol Enhancements

The following improvements are available in the Surface Texture dialog box:
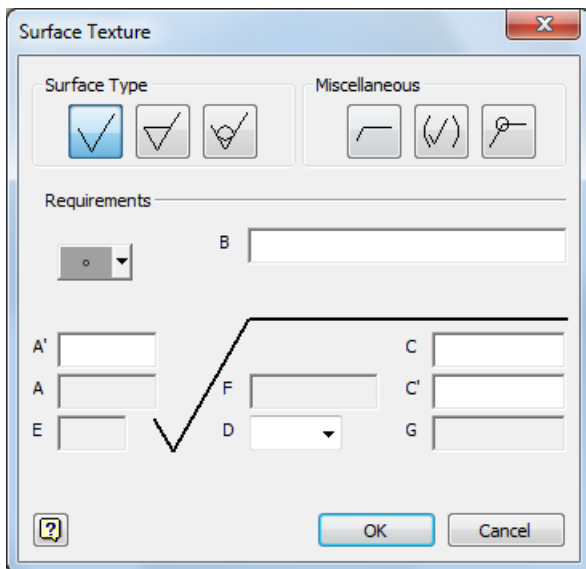
Symbol layout

Various layouts of Requirements fields are available. The layout in the Surface Texture dialog box corresponds to the current drafting standard.

Improved tooltips

A tooltip shows a detailed description for each Requirement field.
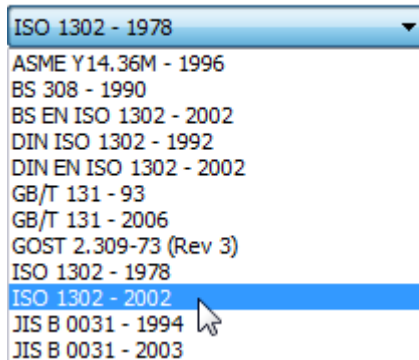
Inset Symbol

Use the Insert Symbol button to add graphical symbols to any of the Requirements fields.
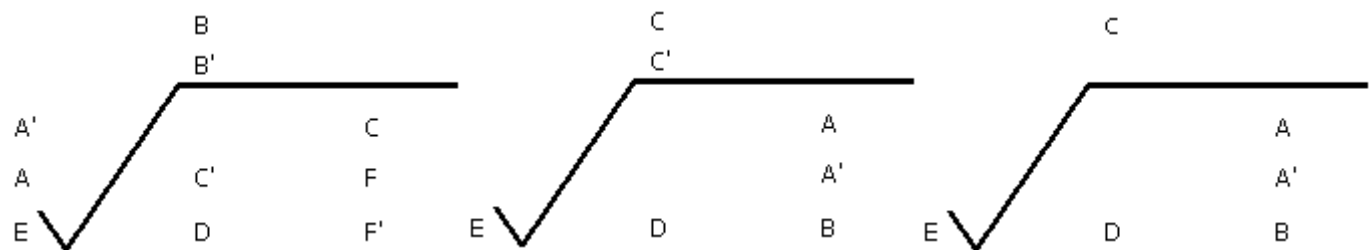
Surface Texture Symbol Styles

Surface texture symbol style is updated to provide symbol formats corresponding to a number of drafting standards. When you define a Surface Texture style, in Style and Standard Editor, select the desired standard from the Standard Reference list.

Standard Reference

ISO 1302 - 1978
ASME Y14.36M - 1996
BS 308 - 1990
BS EN ISO 1302 - 2002
DIN ISO 1302 - 1992
DIN EN ISO 1302 - 2002
GB/T 131 - 93
GB/T 131 - 2006
GOST 2.309-73 (Rev 3)
ISO 1302 - 1978
ISO 1302 - 2002
JIS B 0031 - 1994
JIS B 0031 - 2003

The Requirements field in the Surface Texture style panel shows a preview of the surface texture symbol that corresponds to the selected standard. Examples of symbol layouts (DIN ISO 1302 - 1992, DIN EN ISO 1302 - 2002, and JIS B 0031 - 2003) are on the following images:

Note: When you open a drawing created in previous versions of Autodesk Inventor, the appearance of surface texture symbols is retained. To apply a new layout to the surface texture symbols, change the Standard Reference in the Surface Texture style.

Feature Control Frame Enhancements

The following improvements are available in the Feature Control Frame dialog box:

Top note field

A new edit field is available to add notes in the top of the feature control frame symbols.

Insert Symbol

Use the Insert Symbol button to add graphical symbols to any of the editing fields.

Insert Letter

Use the Insert Letter button to add a letter to the editing fields. The Insert Letter list replaces the single letter buttons available in the legacy dialog box.
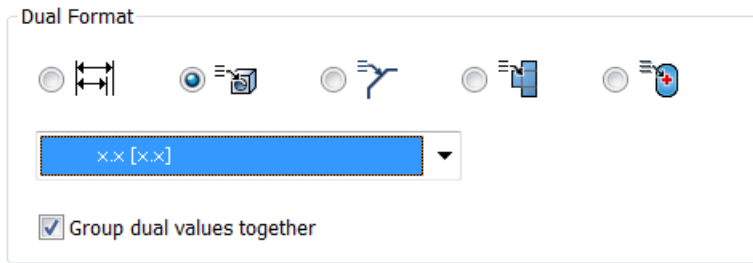
Merged Format for Dual Hole Notes

A new Group Dual Values Together option is available to support adjacent placement of dual dimensions within hole notes. The option is available in Styles and Standard Editor Dimension Style   Alternate Units tab, and it is selected by default.


The following image shows two formats currently available for hole notes with dual dimensions. The Group Dual Values Together option is selected for the hole note at the top, and it is unselected for the hole note at the bottom.
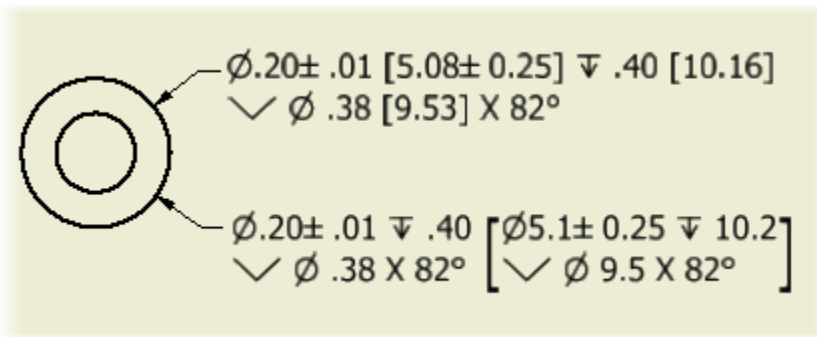
Pressed Joint Welding Symbol

The Pressed Joint symbol is added to the list of graphical welding symbols.

Tip: To make the symbol available in the drawing, in the current Weld Symbol style, in the Symbol Filter list, select the Pressed Joint symbol.
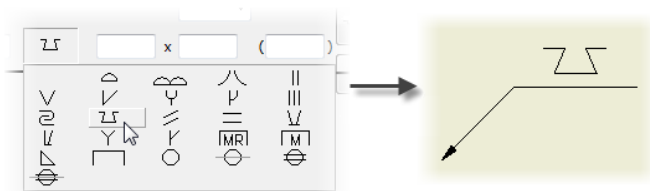
Annotation Leaders

Single Segment Leader Option

Select the Single Segment Leader option to change the default procedure and create drawing annotations with a single leader segment. The Single-Segment Leader option is available for balloons, leader text, surface texture symbols, welding symbols, feature control frames, datum targets, and sketch symbols. You set the option separately for each type of drawing annotations.
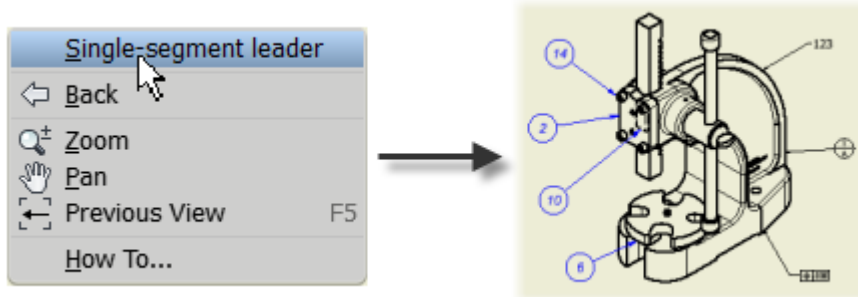

Aligned Leader Annotations

Drawing annotations with leaders can now be aligned in one step. In the drawing, select the annotations to align. Right-click, and on the menu, click Align > Vertical, Horizontal, or To Edge.
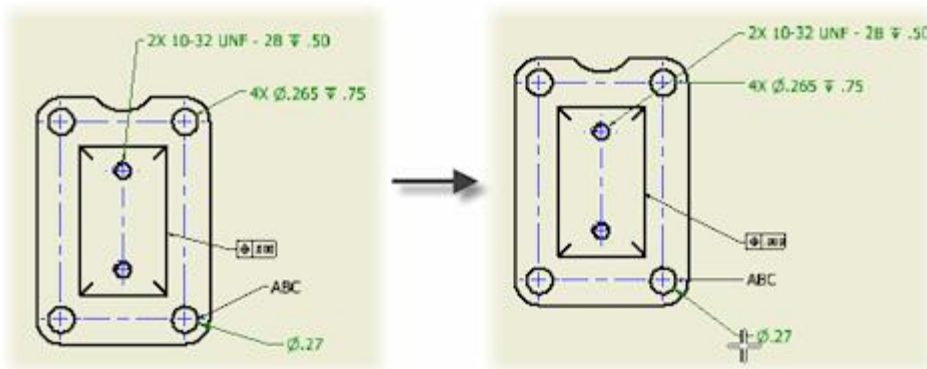
Window Selection for Aligned Annotations

Any window selection that includes annotations supported in the Align commands is valid, and displays Align commands in the right-click menu. After you start an alignment command, all objects that cannot be aligned are automatically removed from the selection set.
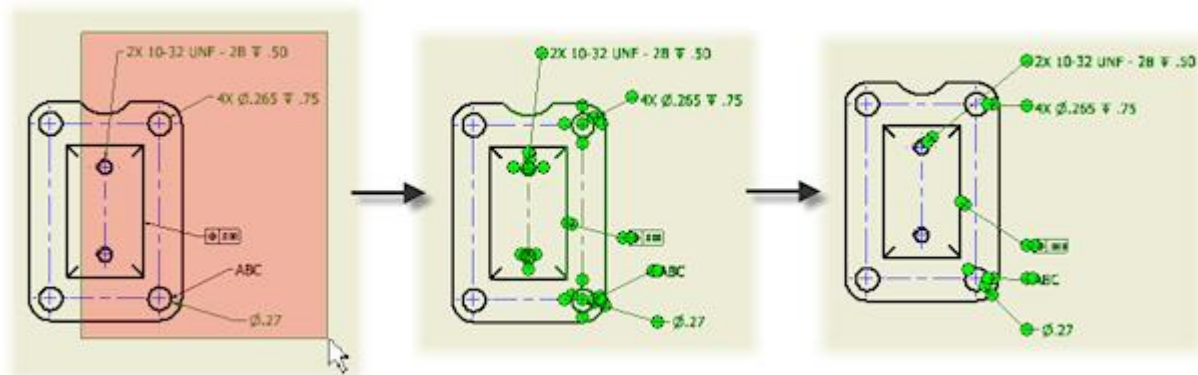
No Snap for Annotation Leaders

The default behavior and CTRL-key behavior for placing drawing annotations are switched. Hold CTRL key to enable 15° leader snapping.
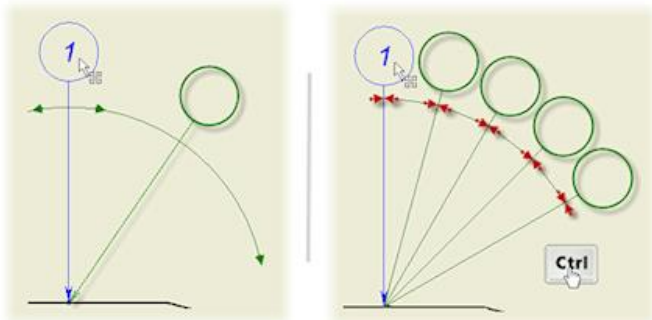
Presentations

General Enhancements in Presentations

Window Selection and Selection Filters
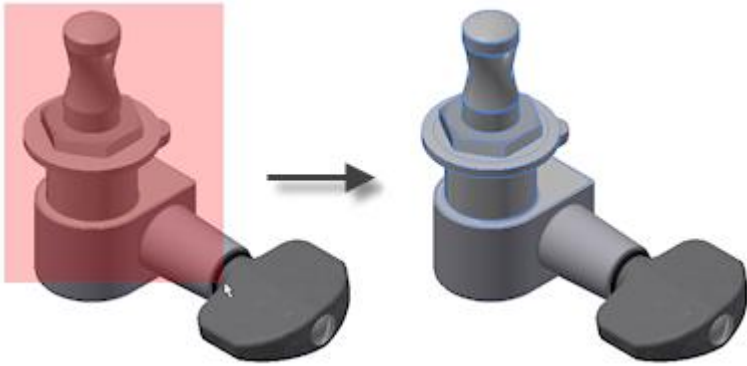
The Select command works similarly as in other environments, including color highlighting and window selection.

The following filters are available on the Quick Access toolbar:

- **Component** selects either sub-assemblies or parts.
- **Part** restricts selection to parts only.
- **Trail** selects trails.

Note: The Component and Part filters are available also on the Tweak Components minitoolbar.

Browser Views

The Filter icon is replaced with name of the current browser view.

Click the arrow to switch between the available browser views.
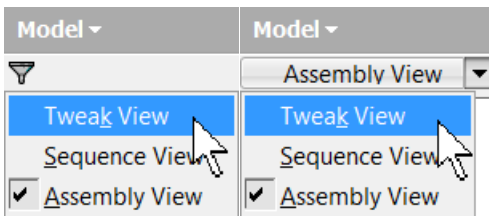
Consistent Tweak Names

Tweak names are now consistent in all browser views.

Design View Representations

You can change the design view representation for an existing presentation view.

You can control associativity between the presentation view and the definition of the design view representation in the model file.

In the Representations dialog box:

- Select a design view representation from the list.
- Select the Associative option to update the presentation view with any edits of the design view representation in the source assembly file.
- Cancel the Associative option to preserve the current design view representation in the presentation view.
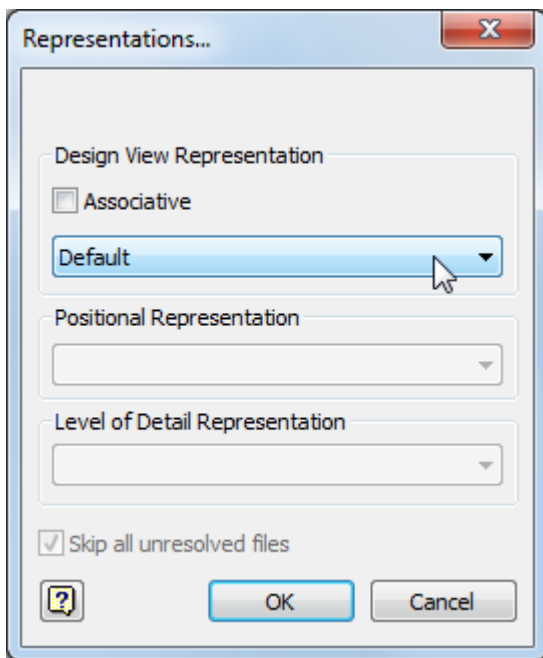
Easier Access to the Auto Explode Command

Auto Explode command is added to the ribbon for easier access.

The command is available when you select an assembly or subassembly.

Tweaks, Trails, and Auto Explode

Tweak Components Minitoolbar

Tweak command is redesigned to use direct manipulation tools arranged in the Tweak Components minitoolbar.

The Tweak Components minitoolbar includes many new commands.

Triad Manipulator
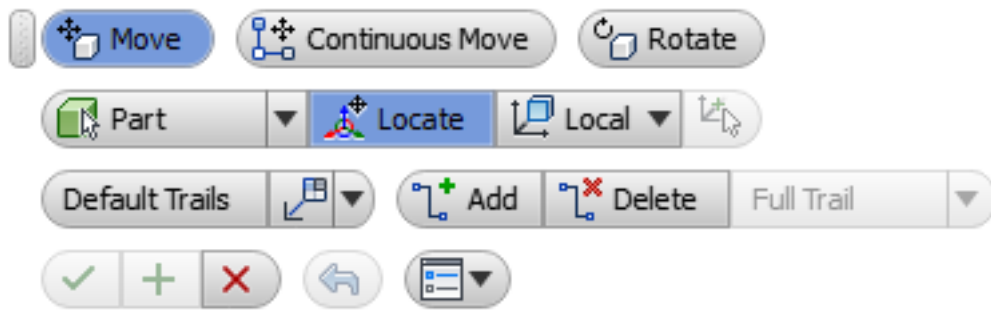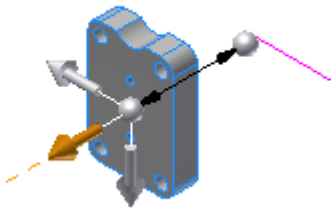
The triad manipulator is modernized and its appearance is similar as in Inventor direct edit commands.

The triad can use the world coordinate system or local coordinate system of the attached component. When local coordinate system is used, you can align the triad to the component geometry.

Move and Continuous Move

Two options are available to create transitional tweaks:
- **Move** clears the selection set of components when you click Apply. You can create a sequence of tweaks with a different selection sets.
- **Continuous Move** retains the selection set of components after you click Apply. You can add multiple tweaks for the same selection set, or slightly change the original selection as you create tweaks.

Editable Selection Set of Tweaked Components

When you create a tweak, you can dynamically add or remove components included in the selection set.

When you edit an existing tweak, you can redefine the set of components participating in the tweak.
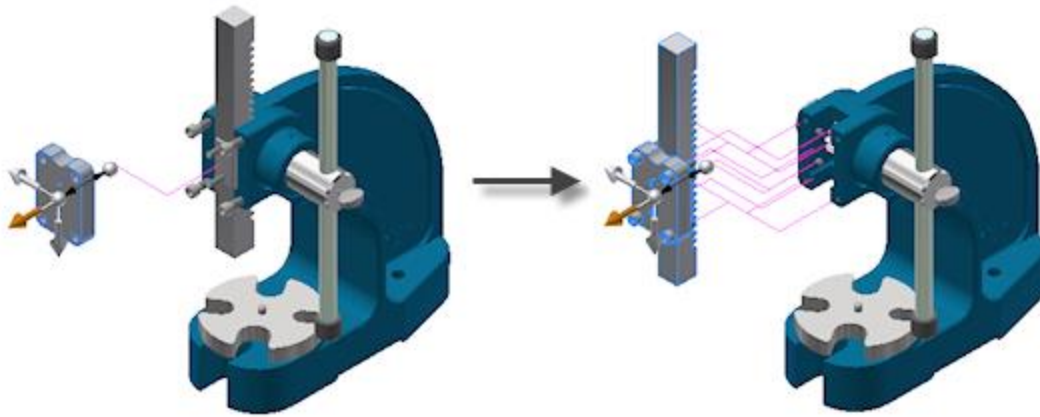
Dynamic Preview of Tweaked Components

When you create a tweak, component positions and created trails are dynamically previewed.

When you edit a tweak, components are rolled back so that you can preview the edited tweak.

In-Canvas Editing of Tweak Distance or Angle

When you create or edit a tweak, an edit box with the distance or angle value is attached to the triad manipulator. When you edit the value, the tweak is updated accordingly.
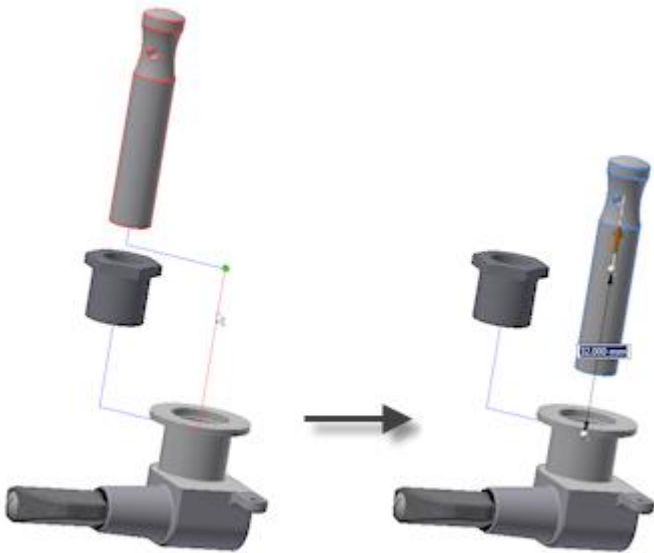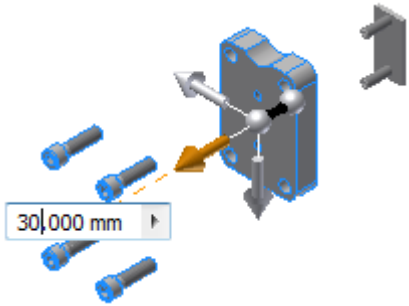

Undo Button in Tweak Components Command

When you create tweaks, you can go back one step and cancel the previous tweak by the Undo button. The tweak history and the Undo button are available until you click Apply in the Tweak Components minitoolbar.
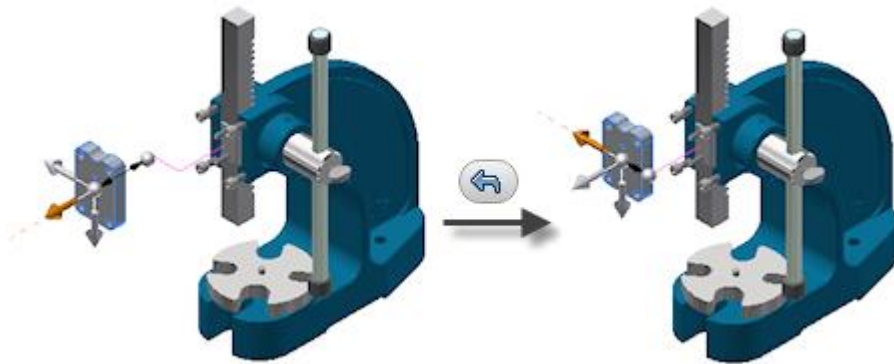
Default Trails

When you create a tweak, you select an option to create default trails:

- **None** cancels creating trails.
- **All Components** creates trails for all selected components.
- **All Parts** creates trails for all selected parts, excludes subassemblies from creating trails.
- **Single** creates a single trail for the entire tweak regardless of the number of moved components. The trail starts from the triad location.

Enhanced Trail Editing

Use the direct editing tools on the Tweak Components mini toolbar to manually add, edit, or delete trails or trail segments.

Manual Trails

When the default trails do not match your needs, you can add trails or trail segments manually. On the Tweak Component minitoolbar, click ⊥⁺ Add , and select Full Trail or Trail Segments. Then in the graphic window, click a point on the tweaked component to add a trail.

A full trail denotes a complete component path in a presentation view. A trail segment represents a specific translational or rotational tweak.
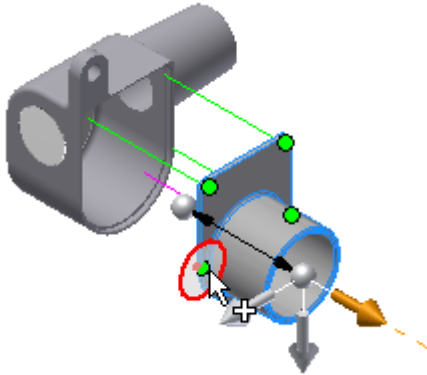
Dynamic Preview in Auto Explode

When you use the Auto Explode command, select the Preview box to dynamically preview the exploded view of the model. The preview updates whenever you change the settings in the Auto Explode dialog box.
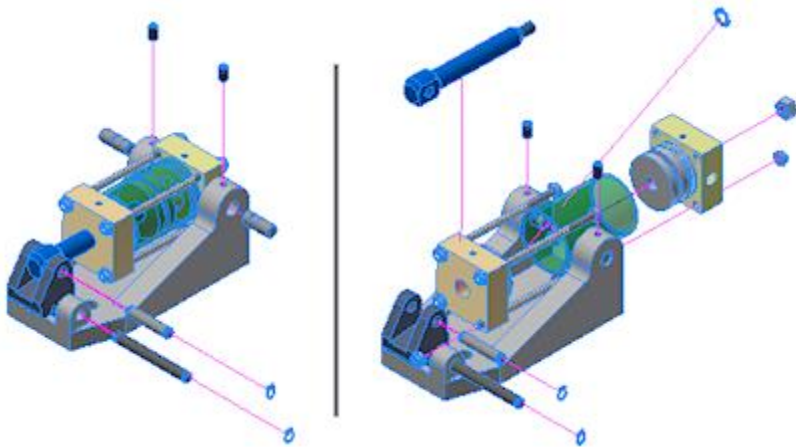
Levels of Auto Explosion

Two options are available to autoexplode the model:

- **One Level** separates all selected components down to one level in the hierarchy.
- **All Levels** separates all selected components down to all levels in component hierarchy.

Trails for Auto Exploded Components

When you autoexplode a model, several options are available to create trails:

- **None** cancels creating trails.
- **All Components** to create trails for all components included in auto explode.
- **All Parts** to create trails for exploded parts only.
- **Single** to create a single trail for each tweak created by the Auto Explode command.

Tweaks Created by Auto Explode

When you autoexplode a model, individual tweaks for the exploded components are added to the presentation view. The tweaks can be edited separately, so you can fine tune the position of the components in the exploded model view.
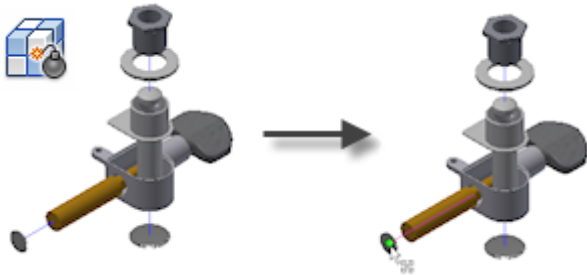
Communication and Interoperability

Importing Files from other CAD Systems Productivity Enhancements

Increased Performance When Opening Files from Other CAD Systems

Multithread support is enabled. Multithreading allows Inventor to use multiple processors if the hardware you are using contains a multi-core processor. As a result, performance is increased when working on files from other CAD systems.

New access to the Import command

- To import into a new file:
    1. **OpenImport CAD Formats**
    2. **Get Started tab Launch panel Import CAD Formats** When you import a file, Inventor now automatically detects whether the imported file is a part or assembly and creates the new document accordingly. For example, to import a 3rd party assembly file as a part, you must now first create or have a part file open, and then Import the 3rd party assembly file into the part file.
- To import into a part file:
    1. **3D Model tabCreate panel Import**
- **Manage tabInsert panel Import**
- To import into an assembly: **Assemble tabComponent panel Place Imported CAD**.

Importing a File from Other CAD Systems

Depending on the file type, click Options or Open in the Import dialog box:

- To access CATIA, Solidworks, Pro-E/Creo, NX, Alias, STEP, IGES, Rhino, SAT, Pro-E Granite, ProE Neutral, Parasolid Binary files: Click **Open** to access options for importing the selected file.
- To access DWG, DXF, STL, Revit, DWF, and IDF files: Click **Options** to access options for importing the selected file.
- To open an AutoCAD DWG file in view only mode in Inventor:
    1. **OpenOpen DWG**

Import as Reference Model: (CATIA, Solidworks, NX, Pro-E/Creo, Alias, and AutoCAD files)

The new Reference Model option maintains a link to the selected file which enables you to monitor and update as the model changes. Use this option if the design is evolving and you are not required to edit the referenced model. If changes are made to the file in its respective product after import, the Inventor file reflects the changes. To import an AutoCAD file as an associative reference file, use the Import command from within a part file.

Import as Convert Model (CATIA, Solidworks, NX, JT, Pro-E/Creo, Rhino, IGES, Alias, Parasolid, and SAT files)

The Convert Model option creates new Inventor files which are not linked to the original. Use this option if you plan to modify the model for a new design.

Simplified import options

The options for importing CATIA , Solidworks, Pro-E/Creo, NX, Alias, STEP, IGES, Rhino, SAT, Parasolid Binary files are simplified and provide clear choices for import.

Selective import

The new Select tab provides you the option to specify which files to import when importing a CATIA, Solidworks, Pro-E/Creo, NX, Alias, STEP, IGES, or Rhino file.

Customize Properties for Imported CATIA, Pro-E/Creo, Solidworks, and NX files

Add standard Inventor property values in the new XML data files to map properties from CATIA, Pro-E/Creo, Solidworks, and NX to standard Inventor properties.

The location of the new design data files is by default: Users > Public > Public Documents > Autodesk Inventor [version] > Design Data > Import Properties.

Linking to Mechanical Desktop (MDT) Components Option Removed (not available in Inventor LT)

The link in the proxy file belonging to legacy data sets are now broken upon migration. Once the link is broken, it is converted into a base solid.

New Associative DWG and Enhanced DWG Workflows

You can now create 3D Inventor parts associative to the source 2D geometry created in AutoCAD. The associative projected geometry can be used to locate Inventor models in an Assembly. The 3D part created in Inventor associatively updates when the 2D geometry changes in AutoCAD.

Use the Import command in the 3D Model tab, Create panel to insert an AutoCAD DWG file with no conversion as an associative DWG underlay.

The following workflows are now available:

- Use the Project DWG Geometry command to project single geometry elements, polylines, open or closed loops, and DWG blocks. Then you can use the projected sketch geometry to create associative model features.
- Right-click the AutoCAD file in the browser to open the new Layer Visibility dialog box. You can select the individual layers you want to display, or control the layer visibility for the whole DWG.
- Reposition the DWG data in the part file by using Triad either moving or rotating with a specified value.
- In an assembly file, you can use the Joint, Constrain, and Assemble commands to create relationships between DWG underlay geometry and a part.
- In an assembly file, you can project DWG geometry in another part that is also open for edit.
- When the original DWG file is updated in AutoCAD, the corresponding Inventor files are updated if the geometry or position changes.

Electromechanical - Linking AutoCAD Electrical and Inventor Projects

An Electromechanical project allows you to create a link between an AutoCAD Electrical project and an Inventor project. When you create a link between an AutoCAD Electrical and an Inventor project, the project files become associative: Design data changes made in one product are updated in the other through Sync.

The process to link a project file from one product to the other involves two steps:

- In either AutoCAD Electrical or Inventor, create the link file and point to the project for that product.
- In the other product, complete the link by browsing to the previously created link file and pointing to a project in that product.

You can now install the Electrical Catalog Browser from the Inventor Installer. The Electrical Catalog Browser for Inventor allows you to insert 3D parts by selecting a catalog value from the database. The utility includes the Catalog Browser interface, the link to the AutoCAD Electrical database, and a sample set of 3D electrical parts.

Learning Tools

Learning Tools

Videos and Tutorials panel
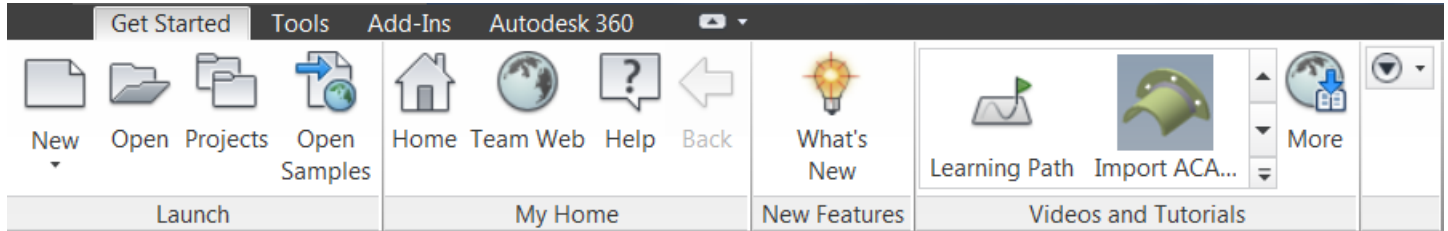
The following changes are made to the Videos and Tutorials panel:

- The panel is enabled by default.
- The Learning Path link for the get started tutorials is moved to the panel.
- The 12 Learning Path guided tutorials no longer display individually in the Videos and Tutorials panel. You can access these tutorials from the Learning Path.
- An Import AutoCAD geometry guided tutorial is included in addition to the Learning Path tutorials.
- A More tutorials icon is provided with a direct link to the guided tutorial download site.

Tip: Click this button to view the new tutorials we release frequently throughout the release cycle.

Help and Search

- Multiple help topics received an edit clean up and are tagged to provide better search results.
- Many dialog box reference topics are combined with the parent task topic to reduce mouse clicks.

Tutorials

The standard written tutorials that run in the Web browser are retired for 2016. You can still access the tutorials from the [Inventor 2014](#)and [Inventor 2015](#)online help.

Continue to increase your knowledge of Inventor by following the guided tutorials available from the Get Started tab - Videos and Tutorials panel in Inventor.

New guided tutorials are available for download throughout the year [from here](#). Guided tutorials provide a no fail learning experience that runs inside the Inventor application. These tutorials lead you through information in a side panel with step by step video, text, and model files. In-canvas visual clues linked to the text in the panel point to commands and other information.

Exclusive Online Training Offers for Autodesk Customers

Autodesk now maintains a site with links to free 30-day trials from online training publishers. Each of these online training publishers provide 30-day free access to their entire library of content. Registered Autodesk customers are eligible. Sign up here: http://knowledge.autodesk.com/free-learning-resources