

Inventor 2013 - What's New

It is that time of year again to release the details about the latest enhancements that are added to the newest release of Inventor. Here is the list of what's New for Inventor 2013.

User Interface

Help Experience

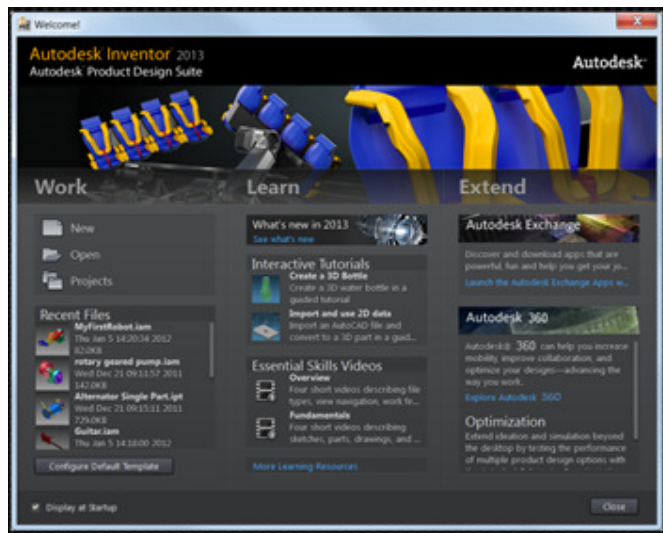
The default Autodesk Inventor Help is online, and is updated continuously.

- **Wiki Help**
Online Wiki Help is the default help in Autodesk Inventor. If you are connected to the internet when you access Help in Autodesk Inventor, the WikiHelp displays. The Wiki provides robust, relevant, and up to date information. In addition, the Wiki enables you to partner with Autodesk to share your knowledge with others.
- **Local Help Download** When you are not online, you can still have access to the Inventor Help locally. Access the internet, and then download from: <http://www.autodesk.com/inventor-2013-help-download>. When you are offline and you access the help, the local help displays.
- **Tutorial Data Sets** Download the tutorial data files referenced in the Inventor 2013 Tutorials, and the instructions to install them from: <http://www.autodesk.com/inventor-2013-help-download>.

Welcome Screen

When you start Inventor, a welcome screen displays with a clear starting point for new and returning users. The screen is divided into three sections:

- **Work Section** - Start new files, work on existing files, or configure the default templates.
- **Learn Section** - Open interactive tutorials, watch short introductory videos, or view the what's New topics.
- **Extend Section** - Start Autodesk Exchange, or explore the cloud with Autodesk 360.



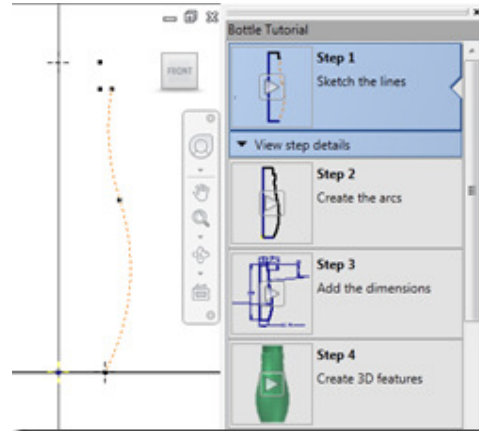
Essential Videos

A series of short videos designed to help you learn key tasks and workflows is included in the product. You can open the videos from the Welcome screen or the Get Started tab.

Interactive Tutorials

Inventor has a great new learning environment to help you get started. This environment leads you through tutorials with step by step video, supporting text, and in-canvas call-outs linked to the text.

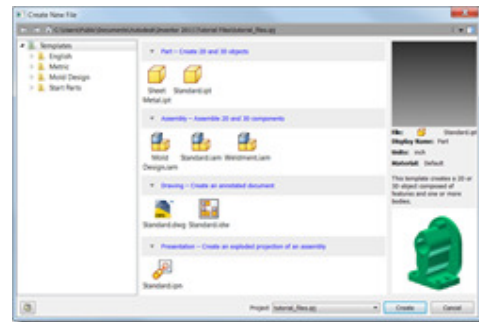
You can start the tutorials from the Welcome screen or the Get Started tab.



Create New File Dialog Box

The Create New File dialog box layout is redesigned to provide file information and better organization.

- Templates display in groups by type.
- Information about the template, such as units and materials, is displayed.
- A thumbnail displays for each template.
- An example image displays for each template type.



In-Command Marking Menu

In-command marking menus are aligned to have consistent placement of the OK, Done, Cancel, and Apply options. In-command marking menus display when you right-click while a command is active.



Context Sensitive Marking Menu

Context-sensitive marking menus are available for sketch objects in Inventor. Select objects, then right-click to display the marking menu. The marking menus make commonly used commands available, depending on the types of objects selected.

Context-sensitive marking menus are available when you right-click, or right-click and drag (gesture behaviour).

User-Defined Marking Menu

A user-defined marking menu is available in each Inventor environment. You must define the commands available in the marking menu before it can be used. Use the Marking Menu tab of the Customize dialog box to define the Ctrl+right-click menu.

Hold down Ctrl and, right-click to access the customizable marking menu. The Ctrl+right-click menu contains constraint commands in the 2D sketch and 3D sketch environments.

Overflow Menu

The overflow menu displays below the marking menu. Shorter versions of the overflow menu are defined for Inventor to streamline the interface. The original, longer overflow menus are on by default. Use the Marking Menu tab of the Customize dialog box to access the short overflow menus.

The options available in the Customize dialog box to control the display of the overflow are:

- Full Menu
- Radial Menu Only
- Short Menu



Enhanced Error Messages

When you click Extrude or Revolve before you create a sketch, an error message displays, and you can start a new 2D sketch.

Default Application Options

The default setting for some Application Options is changed. The following options are affected:

- Sketch tab: Grid lines are off.
- Sketch tab: Edit dimension when created is on.

User Preferences

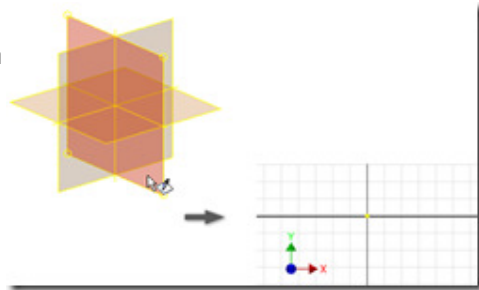
The following user preferences are removed from the installer, and are now available from within Inventor post-Installation.

- User Profile - The User Profile options are now available at the bottom of the Application Options dialog box: Click the down-arrow next to Import and select Use AutoCAD Related Settings or Use Inventor Settings.
- Measurement Units/Drawing Standard default settings - The Measurement Units/Drawing Standard default settings have been moved to the Configure Default Template dialog box. Access the Configure Default Template dialog box from:
 - The Welcome Screen
 - The Application Options dialog box: Select the File tab, and then click the Configure Default Template button.
- Part Modification - The Part Modification settings have been moved to The Application Options dialog box: Select the Drawing tab, and then select or deselect the Enable part modification from within drawings checkbox to enable or disable.

Sketch

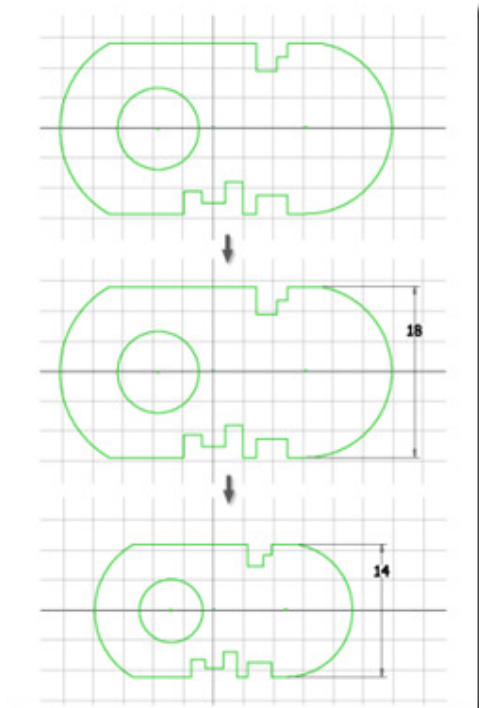
Origin Plane Display

When you start the Create 2D Sketch command, the origin planes display. Pick the edge or face of a plane to begin a new 2D sketch.



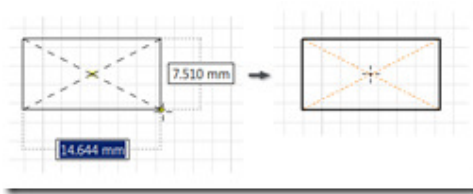
Set Sketch Scale by the First Dimension

The first dimension in the first part sketch determines the scale of the sketch.



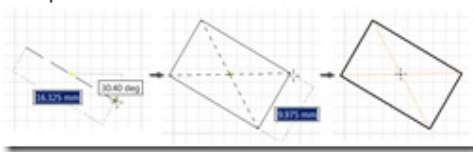
Create a Two-Point-Centre Rectangle

Use the Two Point Centre Rectangle command to create a rectangle defined by a centre point and a vertex.



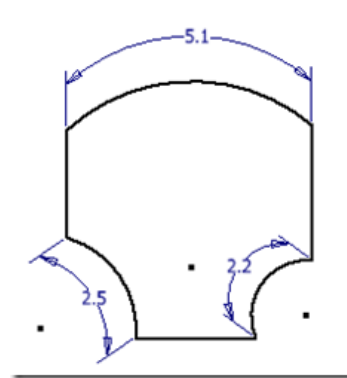
Create a Three-Point-Centre Rectangle

Use the Three Point Centre Rectangle command to create a rectangle defined by a centre point and two points that define the size and direction of the rectangle sides.



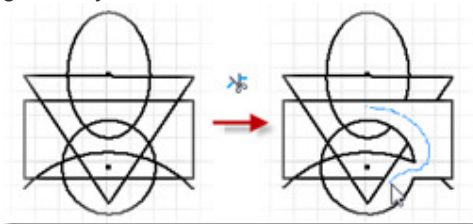
Dimension Arc Length

In the sketch environment, use the Dimension command, Arc Length option to add arc length dimensions.



Dynamically Trim Sketch Geometry

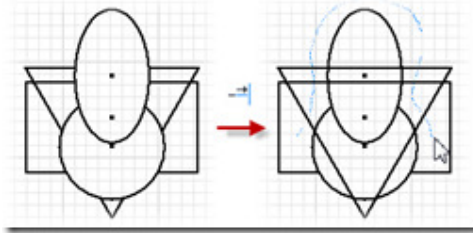
Use the Trim tool to dynamically trim lines and curves. Create or edit a sketch. Then on the ribbon, click Sketch tab – Modify panel - Trim. In the graphics window, click and drag the cursor to trim sketch geometry.



Dynamically Extend Sketch Geometry

Use Extend to dynamically extend lines and curves. Create or edit a sketch. Then on the ribbon, click Sketch tab – Modify panel - Extend. In the graphics window, hold down the left mouse button, and

move the cursor to extend sketch geometry.

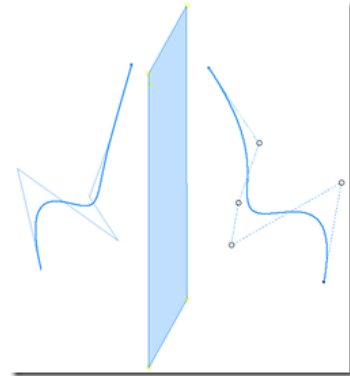


3D Move/Rotate Direct Manipulation

The 3D Move/Rotate command uses a mini-toolbar to manipulate the X, Y, Z values. The mini-toolbars provide a clear, consistent interaction with commands in Inventor. As you drag the triad, the new geometry previews. You can visualize your changes before you commit.

3D Sketch Mirror Command

The Mirror command is available in the 3D Sketch environment. Use origin planes, work planes, or flat faces to mirror 3D sketch geometry.



3D Sketch Properties

The Sketch Properties toolbar and Geometry Properties dialog box are available for 3D sketch geometry. With these controls, you can customize the line colour, line type, and line weight of selected geometry. These controls function the same for 3D sketch geometry as they do for 2D sketch geometry.

3D Sketch Line Align to Plane

In the 3D Sketch Line command, you can snap points to a plane. The selected points are not constrained to the plane. With the line command active, right-click and select Align to Plane then select the desired plane.

Precise Input in 3D Sketch

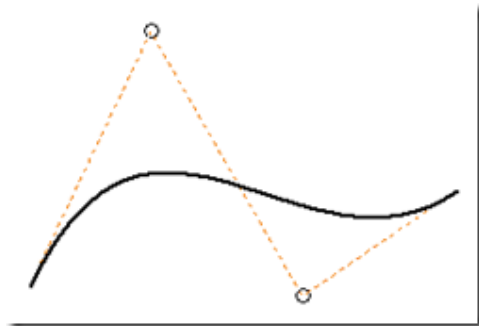
The Precise Input toolbar in the 3D Sketch environment uses a mini-toolbar. The mini-toolbars provide a clear, consistent interaction with commands in Inventor.

The Precise Input mini-toolbar is off by default. To display the mini-toolbar, expand the Draw panel in the 3D Sketch ribbon tab, and click the Precise Input command.

Control Vertex Splines

Inventor supports two types of splines: interpolation, and control vertex. Interpolation splines and control vertex splines have separate commands in the ribbon for forward creation.

After you create a spline, you can change the spline type using the right-click menu. A change in the spline type sometimes causes the loss of constraints and dimensions. Interpolation splines fit the spline through selected points. This type of spline is available in previous releases of Inventor.



In a control vertex spline, the selected points define the vertices of the control frame. The control

frame defines the shape of the spline. You can constrain and dimension the control points and frames. Control vertex splines provide a different method to create and modify splines. Control vertex splines are available in both 2D sketches and 3D sketches.

You can apply tangent, perpendicular, and smooth (G2) constraints anywhere along control vertex splines. For example, draw a line that is coincident to the control vertex spline but not at the end point of the spline. You can now apply a tangent constraint between the spline and the line, and to other geometry types besides lines.

Drag 3D Sketch Geometry

Modifying geometry in a 3D sketch works like modifying geometry in a 2D sketch. You can click and drag points to reposition them, and roughly position geometry without entering coordinates.

You can modify geometry when a 3D sketch is not active. Drag points on a 3D sketch at any time like you can on 2D sketch geometry or right-click a point and invoke the 3D Move/Rotate command.

The changes to a 3D sketch display in a preview while you enter changes by dragging points and using the 3D Move/Rotate command. You can view and understand the effects of your changes before you commit.

Equation Curves

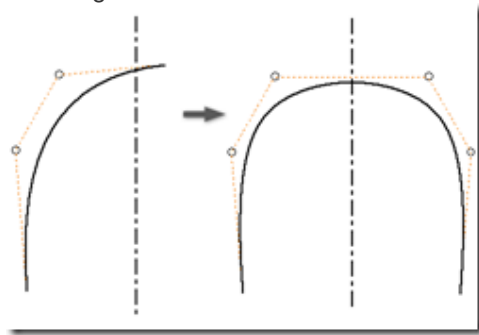
Equation curves are useful to model complex geometry, such as gear tooth profiles or sweep paths for hydraulic pumps. Equation curves are available for 2D and 3D sketches.

You can enter explicit equations into 2D sketches ($y(x) = x^3 / (1\text{mm} * 1\text{mm}) + 3$).

Parametric equations are supported in 2D and 3D sketches. When working with implicit equations, provide the equations for X, Y, and Z (for 3D sketches) and the range of values to evaluate.

2D Mirror Self Symmetric

The mirror command in the 2D sketch environment has a Self-Symmetric option. This option is available when the geometry selected to mirror is an open spline and crosses the mirror line. When this option is selected, the mirror command creates a single spline that is symmetric about the mirror line.



2D Sketch Constraints

Some 2D sketch constraints are enhanced:

- Perpendicular constraints accept arcs.
- You can apply tangent, perpendicular, and smooth (G2) constraints:
 - Anywhere along control vertex splines, projected geometry, or equation curves.
 - To the endpoints of interpolation splines.

Add User Parameters to Text

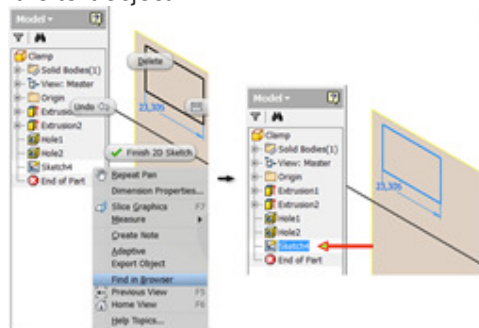
You can add user-defined parameters of True/False or Text type to text in model or drawing sketches, and to drawing notes.

Double-Click Sketch Text to Edit

To edit sketch text, in the Sketch environment, double-click the text object.

Find Sketch in Browser Using a Sketch Dimension

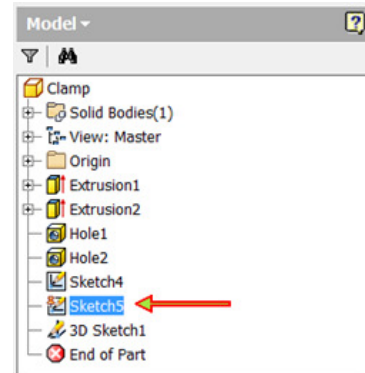
Use the Find in Browser command to locate a sketch. In the graphics window, select a 2D or 3D sketch dimension, right-click, and select Find in Browser. The related sketch highlights in the browser.



Display of Fully Constrained Sketches

A new icon is added to the Part browser to indicate fully constrained 2D sketches.

In the Assembly environment, the indicator icon updates its appearance when you exit the Sketch environment. In the Part environment, it updates interactively as you add or remove constraints.



Measure to the Midpoint of a Segment

In 2D Sketches, when using the Measure Distance command, you can select a midpoint of a linear segment, and a circular arc.

Sketch Environment Status Bar

Commands that were previously located on the right-click menu are now available on the Status Bar in both the 2D and 3D sketch environments. Commands on the 2D Status Bar:

- Snap to Grid
- Show all Constraints
- Hide all Constraints
- Dimension Display
- Slice Graphics
- Show All Degrees of Freedom
- Hide All Degrees of Freedom

Commands on the 3D Status Bar:

- Dimension Display

Associative Sketch Geometry Projection

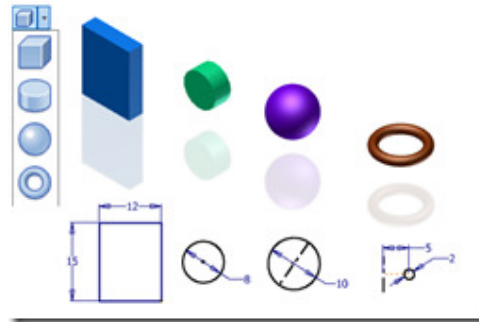
When you edit a part in the context of an assembly, you can project sketch geometry from another part into the active part. This geometry is now associative by default. The sketch and the part are set as Adaptive to keep the geometry associative.

On the Assembly tab, an Application Option controls whether projected sketch geometry is associative. The option is called Enable associative sketch geometry projection during in-place modelling.

Part

Primitive Shape Creation

The function to create primitive shapes is added to a new panel in the 3D Model tab. Primitive shapes automate the sketch, extrude, and revolve process to create a box, cylinder, sphere, or torus. Primitives can add or remove material, or create new bodies.



Offset Base Solid

The Offset command is added to the Edit Base Solid environment. Offset provides the means to modify radial and planar faces of imported solids without opening Inventor Fusion. You can enter a precise value, press to remove thickness, or pull to add thickness. Offset does not add a new feature to the model.

Change Appearance for Multiple Features in One Step

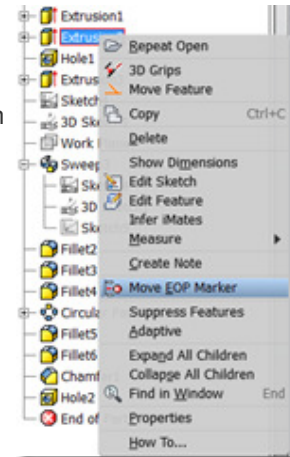
Select multiple part features, right-click, and click Properties. The Feature Properties dialog box with the Feature Appearance option displays. Select an appearance from the list, and click OK to apply the appearance to all selected features.



End of Part Marker

In the browser in the part environment, you can move the End of Part marker directly under a selected feature. Right-click the feature and then click Move EOP Marker. Move EOP Marker command is not available when you select multiple features in the browser, or when it is already located under the selected feature.

You can also move End of Part marker below the Origin folder.



Move End of Part Marker in the Feature List

In the part environment, you can move the End of Part marker directly to the top, or directly to the end of features in the browser. When the End of Part marker is located within your part features, right-click the End of Part browser entry. When nothing is selected, right-click in the browser, and click Move EOP to Top, or Move EOP to End.

Persistent Display Settings in Parameters

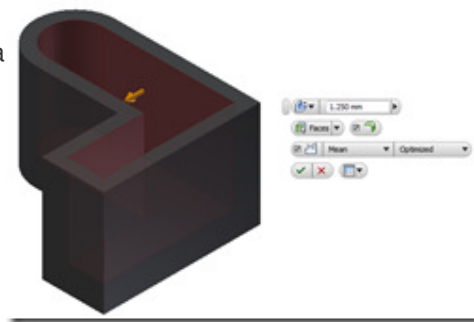
In the Parameters dialog box, the state of the display of folders now persists. For example, if you collapse the Model Parameters folder, it remains collapsed the next time you open the dialog box.

Window Select Area when Selecting a Hole Centre

In parts and assemblies environments, sketch points within a selected area can serve as centres of holes. Click the Hole command. In the Placement area drop-down menu, ensure that From Sketch is selected, and click Centres. Click and drag to select an area. Sketch points within the selected area are used for the centres of the holes.

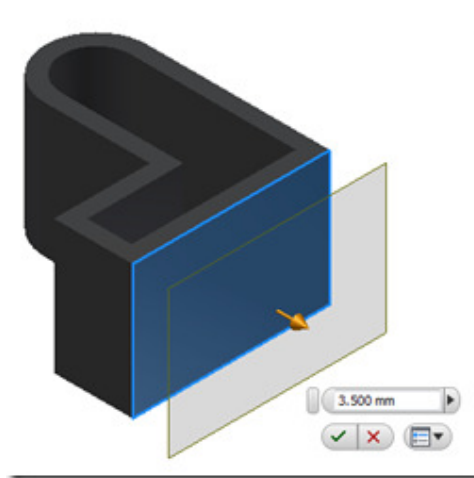
Shell Command Direct Manipulation

The Shell command has direct manipulation controls and a mini-toolbar. The Shell dialog box is collapsed by default.



Work Plane Direct Manipulation

Offset and angle work planes use direct manipulation and mini-toolbars for creation and edit. This workflow is consistent with other Inventor features.



Error Glyphs

An error glyph is displayed next to the mini-toolbar when a feature cannot be calculated. The error glyph is displayed while the command is active so you can address the error before committing the change.

Click the error glyph to display a description of the error. Error glyphs are available for:

- Fillet
- Chamfer
- Shell
- Equation Curves

Disable Construction Environment

The construction environment is disabled by default in Inventor. Therefore, commands related to the construction environment are not available. We recommend that you use the repair environment instead.

The construction environment is available in Inventor when you open legacy parts that contain construction data. Enable the construction environment using the option in the Application Options, Parts tab.

Part View Representation

You can now create view representations in a multi-body part file. Part view representations are similar to assembly view representations and capture the following display characteristics:

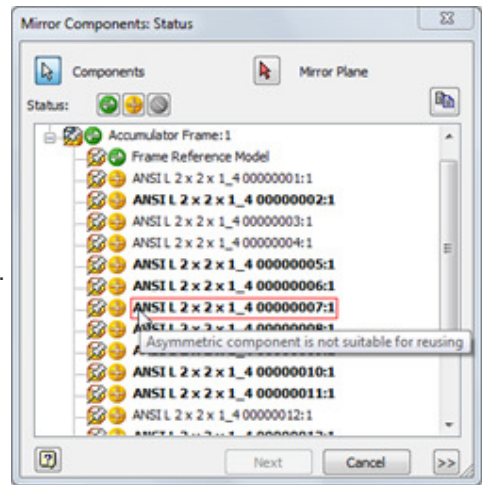
- Part Appearance
- Sketch Visibility
- Sketch Block Visibility
- Work Feature Visibility
- UCS Visibility
- Browser Expansion State
- Zoom Magnification
- Viewing Angle

Assembly

Reuse Components in Assembly Mirroring

Components not suitable for reusing are highlighted in the Mirror Components dialog box. If reused components cause a design error, a message informs you about corresponding fixes.

In the Mirror Components dialog box, use Copy to insert a list of components not suitable for reusing to the clipboard. Then paste the list to a document.



Remove Internal Voids for Derived Components

When you create derived or shrinkwrap solid body parts, select the Remove All Internal Voids option to fill in all internal void shells.

Alignment of a Frame Member through Geometry

In Frame Generator, you can select geometry to align frames instead of defining a rotation angle. On the Design panel, click Insert Frame. Specify the frame member and its position, and check the Align box. Select desired geometry (planar face, linear edge, work axis, work plane, or sketch line) and the inserted frame is aligned by the X axis.

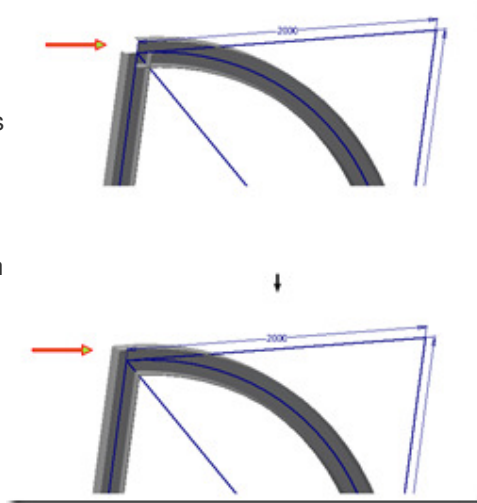
Sometimes geometries are not selected due to a potential conflict. For example, the frame cannot be aligned to the geometries that belong to a:

- Frame that already aligns to the selected frame.
- End treatment generated by Mitre, Notch, Trim To Frame, Trim/Extend, and Shorten (curved or merged beam).
- Part or assembly that has an assembly constraint with the selected frame directly or indirectly.

End Treatment Support on Curved Frame Generator Beams

In assemblies with frame members, you can now cut treatments on curved or merged members. End treatments of an arc curved frame are trimmed following the curve shape. End treatments of a spline curved frame are trimmed through tangent direction.

Use these Frame generator tools to edit end treatments on curved or merged beams: Mitre, Notch, Trim To Frame, Trim/Extend, Lengthen/Shorten.

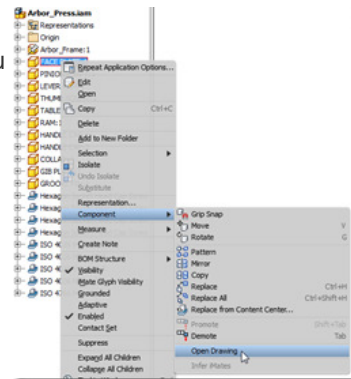


Drawing

Open Component Drawing File from Assembly

The Open Drawing command is moved from the first level of the menu to the second level for parts and sub-assemblies located in a parent assembly. The functionality is not changed.

To open a component drawing, in the Assembly browser, right-click a part or sub-assembly, and then click Component - Open Drawing.

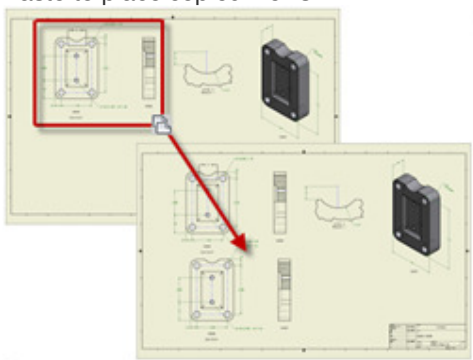


Delete Multiple Drawing Sheets in one step

In the browser, select multiple drawing sheets, right-click, and click Delete. Review or refine the selection in a dialog box, and then confirm the sheet deletion. All selected sheets are deleted from the drawing.

Copy Drawing Views to the Same Sheet

Select one or more views, right-click, and click Copy. Right-click in the graphics window, and click Paste to place copied views.

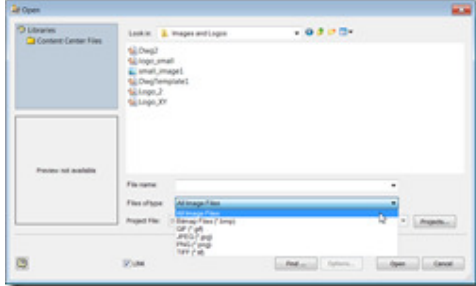


Disable Leader Snapping for Drawing Annotations

Hold down CTRL to disable leader line snapping in 15-degree steps. Then place or move the leader node at any point in the drawing.

Insert Images of Five Types in Title Blocks

Insert image types BMP, GIF, JPG, PNG, and TIF into a title block.



Show All Child Components in a Parts List

In the Parts List dialog box, click the + sign in front of a parent row in a structured parts list to display all child component rows. Alternatively, right-click a parent row, and click Expand All Children.

Default Source Models for Base Views

The Base View command selects by default the last active model document as the source for the placed view. If the last active model document is closed, no model is automatically selected.

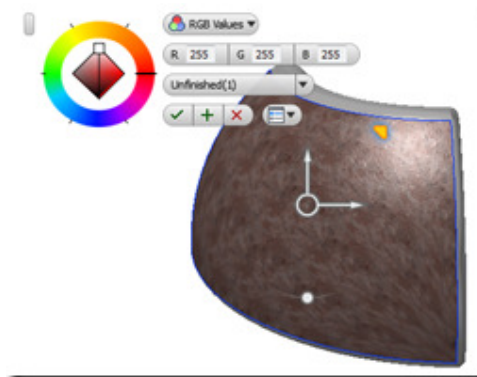
Materials

Appearance and Texture Tools In-Canvas

New in-canvas appearance editing tools make it easier to modify textures right in the scene, and provide immediate feedback. Appearance Mini Toolbar Provides access to the appearance properties in-canvas with real-time updates. In the mini toolbar you can modify the appearance colour, change the assigned appearance, and change the texture mapping.

Texture Manipulator Provides tools to scale, move, and rotate the appearance texture map in-canvas. Contextual feedback helps to determine the necessary modifications.

The texture manipulator works on the selection of a feature or face the automated adjust appearance process reduces the level of library interaction required.



Materials and Appearances

Autodesk shared libraries for Materials and Appearances replaces Inventor Material and Colour Styles, for a more consistent design and viewing experience. The installed libraries include:

- Autodesk Inventor Material Library - A consolidated subset of the Autodesk Material Library with a focus on manufacturing related materials.
- Autodesk Material Library - A compliment of materials and appearances accessed by multiple products and suites. Materials or Appearances selected from this library have the same appearance and material properties, whether viewed in Inventor or Revit.
- Autodesk Appearance Library - A compliment of appearances accessed by multiple products and suites.

Using the library browsers, you can:

- Search for specific materials or appearances by name.
- Add materials or appearances to the document.
- Access materials and appearances in other libraries.
- Create custom libraries to suit your manufacturing requirements.
- Access additional libraries, add categories, and manage custom libraries.
- Quickly add new materials or appearances.

The shared libraries comprise one or more assets, such as Physical properties, Appearance properties, Thermal properties, and so on. Each product uses assets according to the product requirements.

Material and Appearance Libraries and Projects

The project file specifies the material and appearance libraries that you want immediately accessible for that project. During any session you can open additional material and appearance libraries for that session.

Material and Appearance Access

There are multiple access points for tools to work with materials and appearances.

- Material Browser - Use for creating and modifying materials.
- Appearance Browser - Use for creating and modifying appearances.
- Adjust command - Use for modifying assigned appearances.
- Clear Overrides command - Use for removing appearance overrides.
- Assigns the specified material to the selection set.
- Assigns the specified appearance to the selection set.

Work with Materials and Appearances

Materials and appearances are assets that define the composition and look of your design.

Materials are grouped in categories, such as metal, wood, ceramic, and so on, with subtypes for each category. For example, the metal category might include subtypes of aluminium, steel, titanium, and so on.

Appearances are also grouped in categories such as colour, patterns, texture images, bump maps, and other properties. Each type has properties unique to the type. The combined properties provide a unique appearance.

A collection of dialog boxes, a browser, an editor, and an asset browser are available for managing materials and appearances.

Material and Appearance Browsers

The browsers provide access where you can manage libraries and their content. Create custom libraries and populate them with assets. A toolbar at the base of the dialog box provides library access and management tools, and includes the material and colour style migration tool.

Material and Appearance Editors

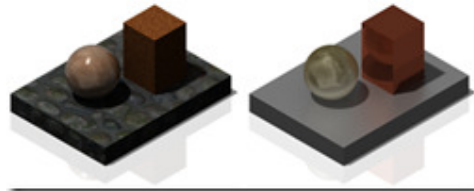
Editors provide the means to duplicate and modify assets. You can create custom assets and keep them in custom libraries. The editors present information specific to the material or appearance, and the types or categories from which they are made.

Asset browser

While you edit a material or appearance asset, the corresponding assets display in the browser. You can replace material and appearance assets for any document material.

Visualization

The common use of texture maps to define the appearance of components, and modelling with textures displayed, adds visual complexity. To reduce visual complexity and increase efficiency, a pair of controls are added to the View tab - Appearance panel.

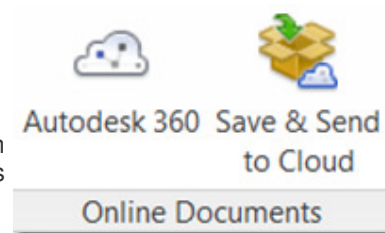


Using Textures On and Textures Off, you can quickly override the display of textures while modelling, and redisplay them for visualization. You can set these controls to persist with each model using Document settings, or with each session using Application Option settings. The display of all textures except, threads and decals, are turned off. Threads and decals display regardless of the texture setting.

Online Services

Online services

Autodesk 360 online services are available for all subscription customers. Integrated access to online services is added to Autodesk Inventor 2013, and all subscription customers have access to online document storage. You can use your secure login to access documents from any location at any time. All you need is a web browser to view your data. Collaboration just got easier.



Autodesk Exchange Apps

Autodesk Exchange Apps is an online resource where you can browse and purchase Apps (including product-specific content such as models, training materials, and eBooks) – and is available to several Autodesk Products. You can download some items for free, and some for a price. Purchasing is handled via PayPal at this time. Purchases and downloads can be tracked online, or by using the Autodesk Exchange App Manager that is included with the respective Autodesk product.

Simulation

Analysis of Thin Wall Bodies

In the stress analysis environment, new functionality performs the analysis of thin wall components. You find the bodies within your model that meet the thin component criteria. Then you simplify the geometry to generate mid surfaces or offsets, and perform the analysis.

On the Prepare panel, use Find Thin Bodies to find and highlight bodies that meet the shell features criteria.

Then, use Mid surface and Offset commands to simplify the model geometry. Mid surface inspects the selected bodies and automatically builds a median surface based on the geometry. Shell features are considered in the analysis for such surfaces. Offset creates a shell feature from selected faces. Specify the thickness, and distance is calculated as thickness/2. You can perform an automatic face chain selection.

API for Dynamic Simulation

An API for the Dynamic Simulation environment opens up some very exciting co-simulation workflows with other simulation tools. The API allows you to control the magnitude of any load at each time step and it also allows you to extract resultant information like position and acceleration, enabling you to interface with external control system simulations.

To learn more about the API, open Programming Help on the Autodesk Inventor Help menu, click Community Resources and then Programming Resources.

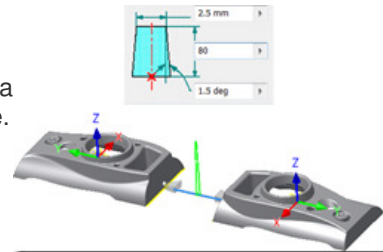
Enhancements in Mesh Creation and Solution Time

Multi-core machines see enhancements relative to mesh creation and analyses. All available machine cores are now used for both mesh generation and the solution phase of your simulation resulting in faster performance.

Tooling

Virtual Sprue

An expand section is added to the Mould Process settings dialog box to create a virtual sprue. With a virtual sprue, you can perform a mould fill analysis without creating a mould base or an actual sprue. To create the virtual sprue, select a runner sketch, and specify the virtual sprue parameters.



Machine the Core/Cavity into Mould Plates

With an option that is added to the Workpiece Pocket command, you can machine the core/cavity directly into the A/B plates. To use the new option, in the dialog box click the Pocket Type drop-down menu, and select Merge Core and Cavity into Mould Base Plates.

Include Surfaces from Source Model

An option is added to include surface bodies from the source model when you derive the plastic part. This option provides greater flexibility for creating complex hole patches and runoff surfaces. Select the surfaces to include when you derive the part.

Locate Sprue Bushing

A new option in the Sprue Bushing dialog box provides greater flexibility in setting the location. You can locate the sprue on a runner sketch by selecting the sketch and specifying a ratio value. The ratio value adjusts the position of the sprue along the selected sketch geometry. If you select a sketch point or an end point, ratio is not available.

General Performance and Capacity

Two significant enhancements improve the performance and capacity of mould assemblies:

- Multi-body in the mouldable part creates an insert for improved performance, capacity, and robustness.
- The derived mould part uses Reduced Memory Mode when the plastic part is placed in the mould assembly.

Additional enhancements are:

- The Delete option is supported in the context menu of both Create Patching Surface and Create Runoff Surface.
- When you place a plastic part, the colour style of the original part is inherited.

Select Reference Centre for Part Shrinkage

The Specify UCS option is added to the Part Shrinkage dialog box. Create a UCS in the part file at the required location, and then select the UCS to use it as the centre for part shrinkage.

Update Core/Cavity

You can save time during the design process by independently updating the core/cavity features without updating the whole mould design. To update the core/cavity without updating the whole design, click the Core/Cavity update button in the core/cavity or mould environment.

Interoperability

Point Cloud

A point cloud is a set of vertices in a three-dimensional coordinate system, defined by X, Y, and Z coordinates. The vertices typically represent the external surface of an object. You can:

- Attach one or more point clouds to both part and assembly files.
- Move, rotate, scale, crop, and un-crop a point cloud.
- Decrease the point density.
- Attach work points to individual cloud points for the creation of work axes and work planes through the cloud.
- If you use the Autodesk Factory Design Suite, snap factory assets directly to points in the point cloud.



Multiple Solid Parts Import

The Import Multiple Solid Parts as Assembly option is no longer available. Files that contain multiple solid parts are imported as Inventor multi-body parts. Use the Make Components command to create part files from the individual bodies.

This change affects import options for JT, Parasolid, IGES, STEP, NX, and SolidWorks files.

For SAT files, the options are Import as a Single Part, and Import as an Assembly.

Updated Translators

The following translators are updated:

- CATIA V5
 - Import: R21
 - Export: R20
- JT
 - Import/Export: Version 9.5
- Granite
 - Import/Export: Version 7.0
- NX
 - Import: NX8
- Parasolid
 - Import/Export: Version 24
- Rhino
 - Import: 5.0
- Solidworks
 - Import: 2012
- SAT

- Export: Version 7.0 only (version 4-6 are no longer supported for export)

Task Scheduler

Purge

Run the new Purge task to purge styles and materials for multiple files in one operation. Use this task when you transition to using a style library with your design project.

A purge operation:

- Removes every unused style from all of the Autodesk Inventor files (parts, assemblies, drawings) that you specify for processing.
- Removes styles that you no longer need from legacy documents.
- Often reduces the memory requirements for working with large assemblies.

Any styles that you do not save in another document or a style library are permanently lost in purging.

Installation and Deployment

CAD Manager and IT Manager Resource

The CAD Manager and IT Manager Resource is dedicated to CAD Managers and IT Managers. Links are provided on this centralized location to get you to the information you need for installing, licensing and deploying Inventor, the Product Design Suites and Factory Design Suites.

Additionally you will find links to topics on:

- Data Migration
- Language Packs and Language Switching
- Product Support resources

Data Migration and You

There is a new Data Migration resource that details the factors to consider when deciding how to manage your data from one release to the next. Determining if and what to migrate may depend upon any combination of the following: the type of data you create, the quantity of files you manage, and whether you are using a data management system (such as Vault).

The new Data Migration resource includes detailed step-by-step procedures for migrating design and data files. To determine the best migration path for your files, the Data Migration resource breaks down the different types of content and guides you through the migration process for each type.

Language Packs and Language Switching

A language pack is a mechanism used by some Autodesk products to provide another language for the user interface and the documentation. You can now install additional standalone language packs, and switch between languages without having to uninstall and reinstall.

Language switching functionality is now supported for:

- Autodesk Inventor R2013
- Autodesk Inventor Professional R2013
- Autodesk Inventor LT R2013
- AutoCAD R2013
- Autodesk 3ds Max Design R2013

When you launch Autodesk Inventor or Autodesk Inventor LT by a language-specific Start menu shortcut, that language becomes the current language for Inventor.

Deploy Local Help

The online Wiki Help is now the default Help for Autodesk Inventor and LT. If you are online and access the Help, the online Wiki Help displays.

If you need to work offline and need access to the product help, you can download the Help locally to your computer. When you are offline, the local Help will display.

CAD Managers and IT Managers - If you want to provide the Local Help to your user base, you can deploy the Local Help through the deployment wizard.