What's New in Autodesk Inventor 2014

The focus for this Inventor release is to increase productivity for both new and long-time users in the area of assembly design. We also included productivity improvements for parts, sketching, drawing, and other environments based on long-time customer requests. Finally, we added tools that help you share Inventor models more broadly with your partners. This addition streamlines the sharing of models with building engineers and architects making it even easier to collaborate through Autodesk 360 services.

Customer Productivity	Workflow Interoperability	Connected Experience
• Ease of Assembly • Large Assembly Performance • Customer Satisfaction	• BIM Ready Models • Model Based Definition • Consistent Materials	• Integrated A360 Experience • Application Virtualization

The following is a brief outline of these tools and capabilities.

Greater Productivity

A new command called Connect transforms the assembly design process. Connect provides a one-step method of positioning components and defining motion.

If you are new to the product, it is easy to understand and master assembly behavior. If you already know the product, the streamlined assembly creation and management tools provide an engaging and productive experience.

Other productivity enhancements that we hope will delight you are:

- A new method of working with very large assemblies called Express mode loads assemblies five to ten times faster, greatly improving the time to work.
- New assembly diagnostic tools allow you to quickly understand and correct relationship errors for both constraints and connections.
- Assembly copy/paste functionality produces an identical copy with orientations and relationships intact.

- Improvements to the modeling core allow greater success in complex blends and sweeps.
- A new slot sketching tool dramatically reduces the time required to create obrounds and curved slots.

Enhanced Workflow Interoperability

Multiple Consistent Material workflows have been streamlined to make it easier to create custom libraries. Custom libraries can be shared between Inventor, Revit and AutoCAD files.

We also added tools that help you share Inventor models more broadly with your partners. With the new BIM interoperability features, you can transform detailed Inventor assemblies easily into simple native Revit Families for use as BIM content.

This addition streamlines the sharing of models with building engineers and architects making it even easier to collaborate through Autodesk 360 services.

Connected Experience

Finally, stay tuned for these enhancements: R2014 provides a simple, intuitive process to save files to, and open from Autodesk 360. You benefit by having access to your data, anytime, anywhere, and the ability to share anytime and with anyone. Get confirmation on designs, troubleshoot field maintenance, and resolve design sharing issues with Autodesk 360.

Keep reading for the details on these and all the other great enhancements

Assembly

Connect

The Connect command is a simple way to position components and describe motion. Creating a connection fully defines the component location and motion in one step. End, mid, and center points are used to associatively position components. The geometry you select determines the default connection type, or you can specify a connection type.

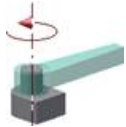The following describes the connection types and how they operate while positioning a component:

Rigid

Removes **all** degrees of freedom (DOF). Welded and bolted connections are examples of a rigid connection.

Rotational

Specify **one** rotational DOF. Hinges and rotating levers are examples of a rotational connection.

Slider

Specify **one** translational DOF. A slide block moving along a track is an example of a slider connection.
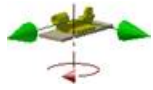
Cylindrical

Specify **one** translational and **one** rotational DOF. A shaft in a tube is an example of a cylindrical connection.

Planar

Specify **two** translational and **one** rotational DOF perpendicular to the linear. To place a component on a planar face, use this connection type. The component can slide or rotate on the plane.

Ball

Specify **three** rotational DOF. A ball and socket are an example of a ball connection.

Note:
- With the addition of the connect command, both constraints and connections are now described as relationships.
- In the browser, the Constraints folder is renamed to Relationships, and displays in both the Assembly View and the Modeling View.
- You can use connections with constraints to position components and describe motion.

For more information, check out the new **Place and connect parts** interactive tutorial or

Created connections are automatically converted to joints when you enter the Dynamic Simulation environment. The converted connections are listed as standard joints in the simulation environment browser. The following are the automatic conversions:

Inventor connection	Joint in Dynamic Simulation
Rigid	Welding
Rotational	Revolution
Slider	Prismatic
Cylindrical	Cylindrical
Planar	Planar
Ball	Spherical

You can edit the joints that are created automatically in Dynamic Simulation using the context menu in the browser.

Connections locked in the Assembly environment are also locked in Dynamic Simulation. To unlock a joint, right-click and clear the selection for **Lock dof**.

Lock and Protect

For connections that have one or more degrees of freedom, two options are available on the browser context menu:

- **Lock** maintains the current position, but is different from grounding a component. Lock eliminates all motion, but allows the component to change position when related components move. Grounding eliminates all degrees of freedom, and fixes the component position in space.
- **Protect** alerts you if an added relationship violates the required degrees of freedom.

Lock and Protect are not available for constraints.

In the following image, the Ball connection is locked.
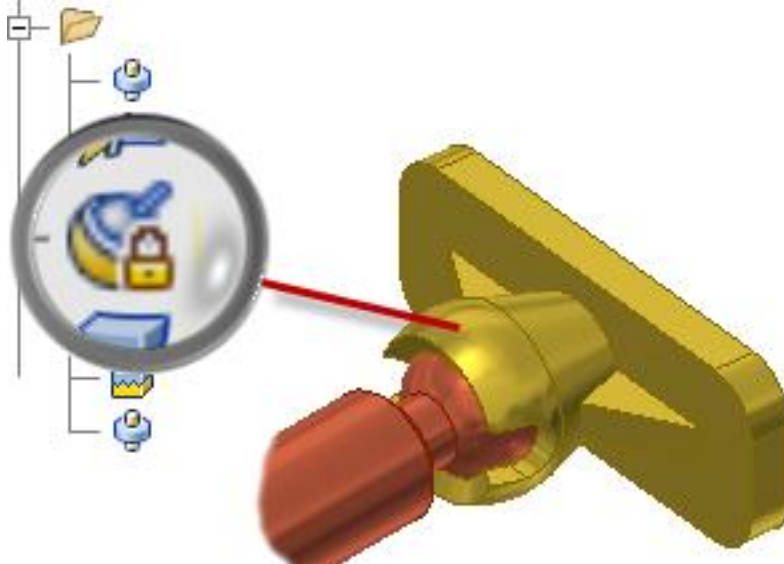
Relationship display commands

Three commands are added to help you manage relationships visually:

Show

To display a glyph that shows the relationship type, select a component. To suppress, unsuppress, or delete a relationship, use the glyph context menu.

Hide All

Remove all relationship glyphs from the display. Hide All does not modify relationships.

Show Sick

Display all relationships that are marked sick. To suppress or delete a relationship, use the glyph context menu.

Grounded component display

You can easily identify grounded components in the display. The **grounded icon** appears when you move your mouse over a grounded component.

Place Component

The first component placed in an assembly is no longer grounded. To ground any component at the origin, use the context menu. The menu contains new options to rotate components in 90-degree increments about the X, Y, or Z axis, before you place them in the assembly.
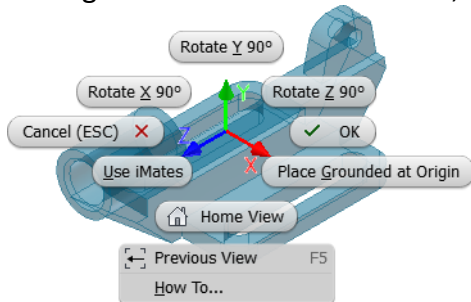
Free Move

With the Free Move command, display relationships as an elastic band, and visualize and manage relationships in a new way. Click a relationship icon, and then, to suppress, unsuppress, or delete relationships, use the context menu.
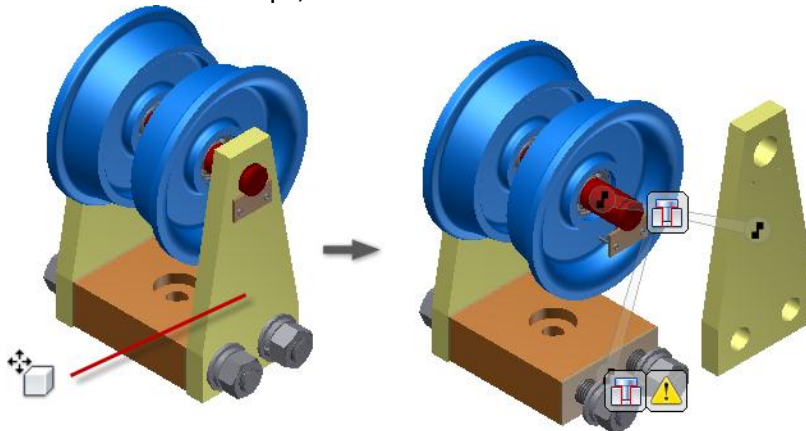
Assembly copy paste

In previous releases of Inventor, a copy and paste operation did not retain component connections or current orientation.
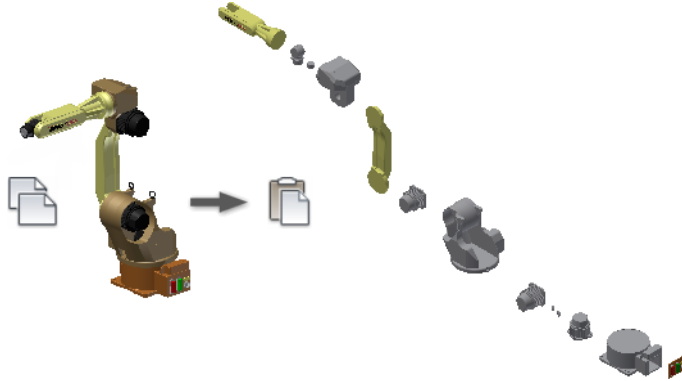
The enhanced copy and paste operation produces a duplicate of the copied components with connections and orientation intact.
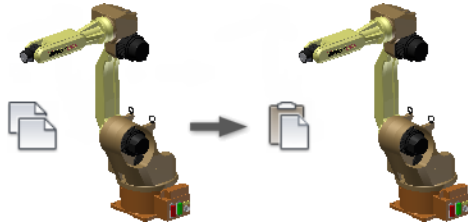
Symmetry constraints

The Symmetry constraint positions two objects symmetrically according to a plane or planar face. The Symmetry constraint is available in the Place Constraint dialog box.
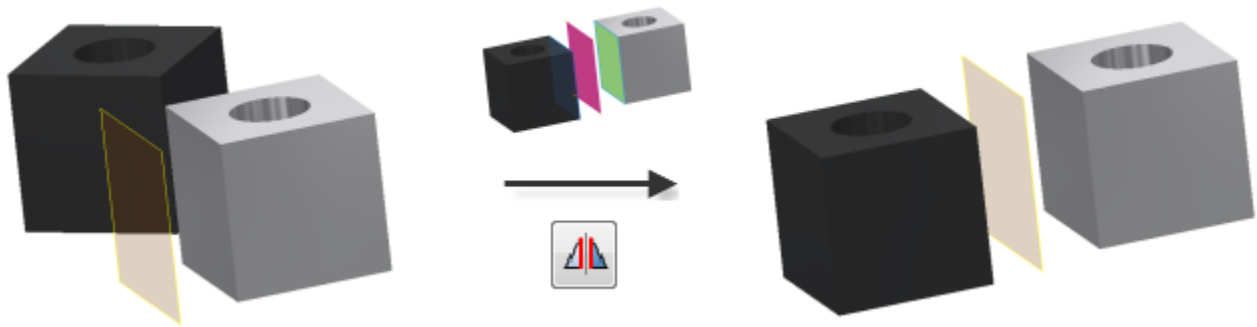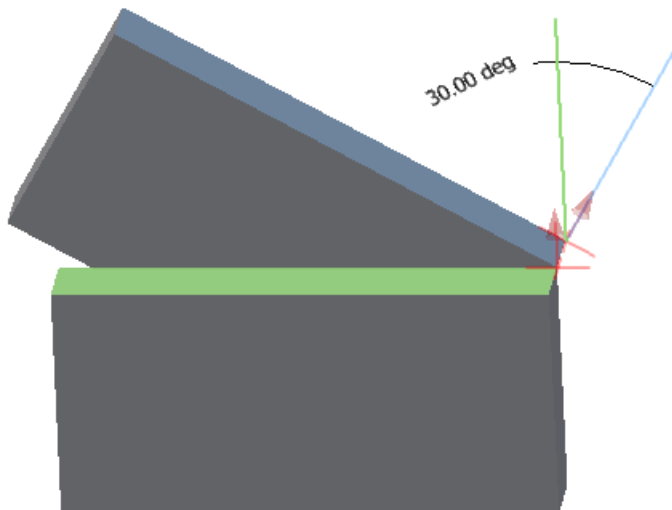

Improvements in Angular constraints

When you create an angular constraint, the status bar displays clear instructions to finish the Constrain command.

Angular constraints display direction vectors and the angle in the graphics window.

The model preview updates when the Angle value changes.

Express mode for large assemblies

Express mode introduces a new way of working with large assemblies. Large assemblies typically open 4-6x faster, dramatically improving your time to work.

Enable the Express option and set the unique files threshold in the Application Options, Assembly tab. Large assemblies that exceed the threshold open automatically in Express mode.

Load Full to exit Express mode and enable all commands.

Improved graphics performance

The following graphics enhancements improve the performance of pan, zoom, and rotate of large models in both Full and Express mode.

- The maximum frame rate is increased from 10 Hz to 20 Hz.
- The largest objects are drawn first.
- The graphics system now uses multiple cores to provide the best possible performance.
- A new background operation called consolidation is added. Consolidation draws objects that have the same appearance as a single group, improving the GPU usage.

In the following images, the 2013 model on the left displays the results of rotating the model using the maximum of ten frames per second. Increasing the frame rate increases the speed, but the choice of components to draw is not intelligent. The 2014 model on the right shows the results of rotating at the faster maximum frame rate of 20 Hz. The model remains recognizable, and all view operations are faster.
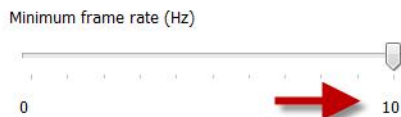
Enhancements in Insert Frame command

In previous versions of Inventor, construction lines and center lines are automatically included, and frame components are created from them.

In the current version, you can exclude construction lines or center lines from the geometry selection used to create frame components.

On the ribbon, click **Design tabFrame panelInsert Frame** . Then, in the Insert dialog box, select or clear the Select Centerline, or Select Construction options to specify your settings.
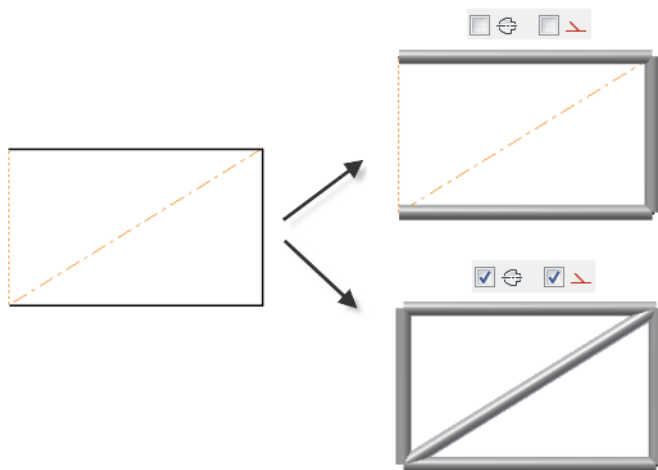
Camera setting in view representations

In previous versions of Inventor, the camera position is always automatically saved on closing the design view representation. When you rotate or zoom the model in a design view, tools to restore the previous camera settings are limited.

With new commands in the view representation context menu, you can easily save and restore the camera settings.

- **Save Current Camera** command saves the current position of the camera, and switches off the AutoSave command.
- **Restore Saved Camera** command restores the last saved position of the camera.
- **AutoSave** command saves the last camera position automatically on closing the view representation. This option corresponds to the legacy behavior.
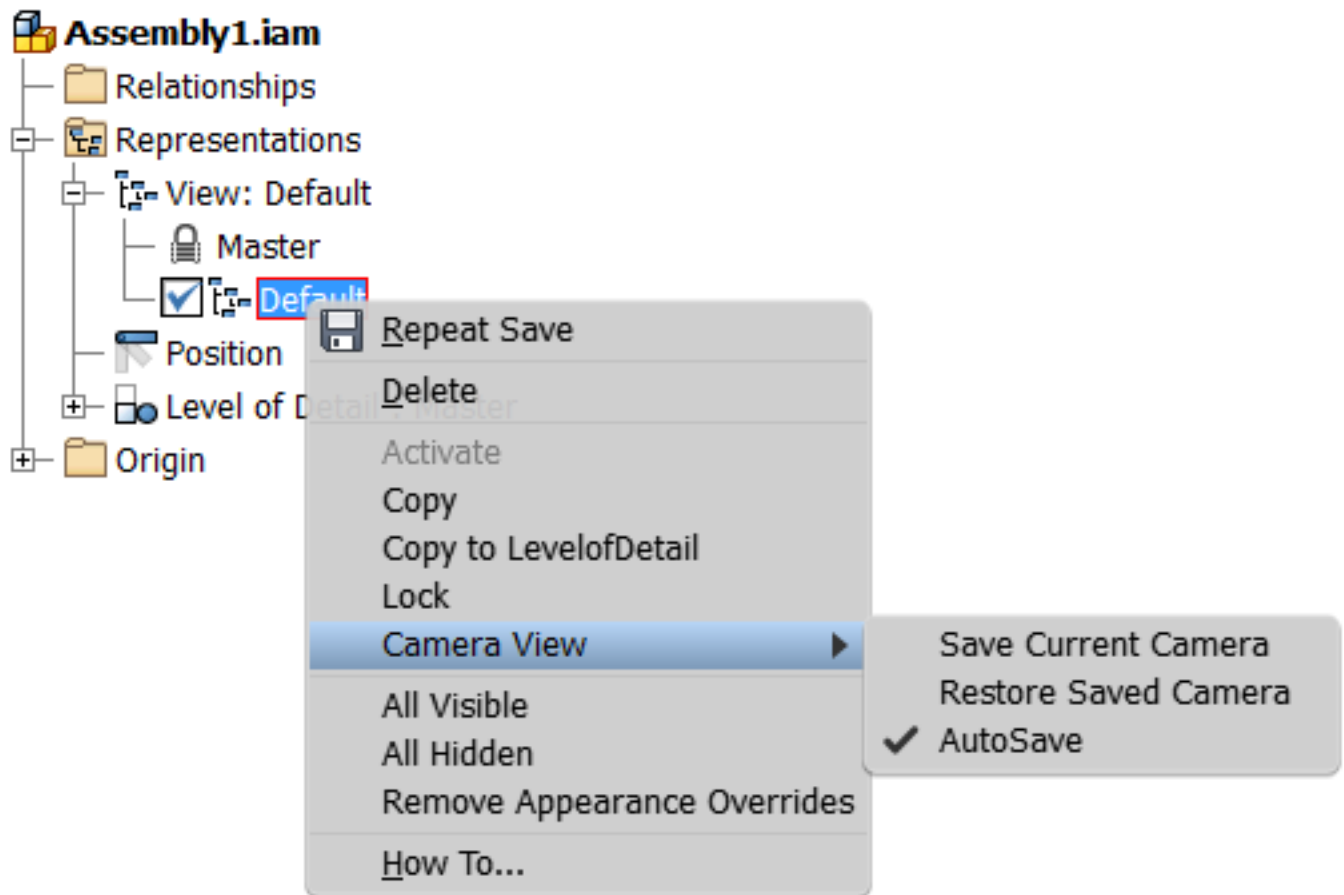
Improved export of recorded driven constraint sequences

The default settings used to export recorded driven constraint sequences are optimized. As a result, the WMV or AVI export provides optimal output quality by default.

When you customize settings in the WMV or AVI export dialog boxes, your customization is persistent for the entire session.

Appearance property

The model Appearance value is added as a selectable property. You can add the Appearance property to a bill of materials or a parts list.
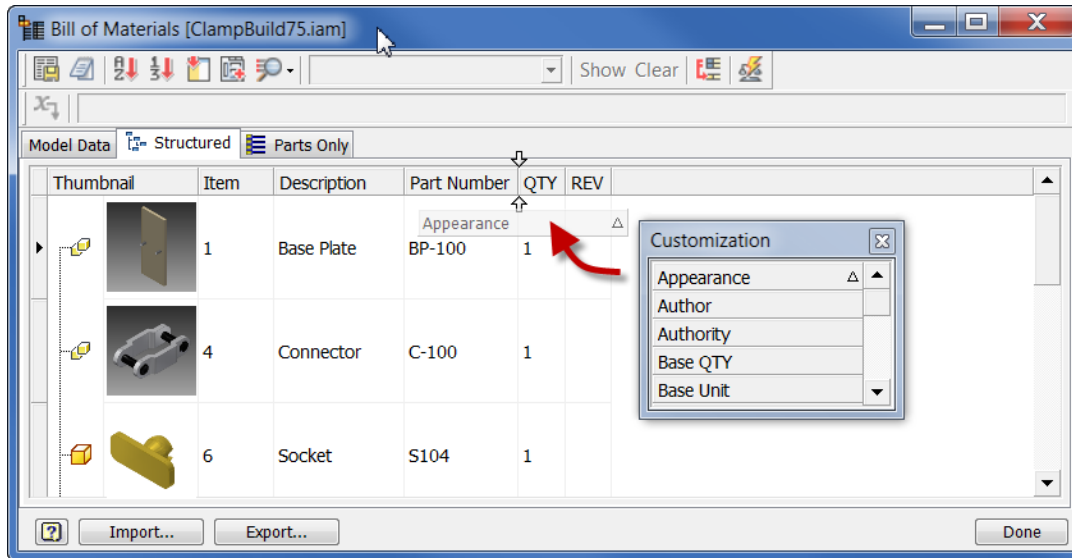
Mold design, Extend Runoff Surface

  The Core/Cavity environment contains a new command for creating runoff surfaces. Previously, all tooling runoff surfaces were created along the XY plane or a user specified direction.

Use **Extend Runoff Surface** to create runoff surfaces that extend along the component tangent or rule. For example, the following image displays a tangent runoff surface on the left, and a rule runoff surface on the right.
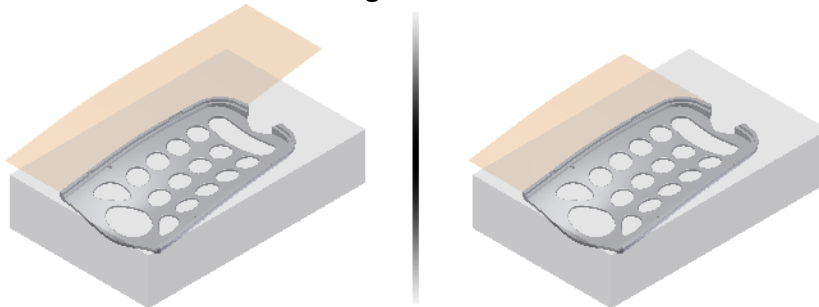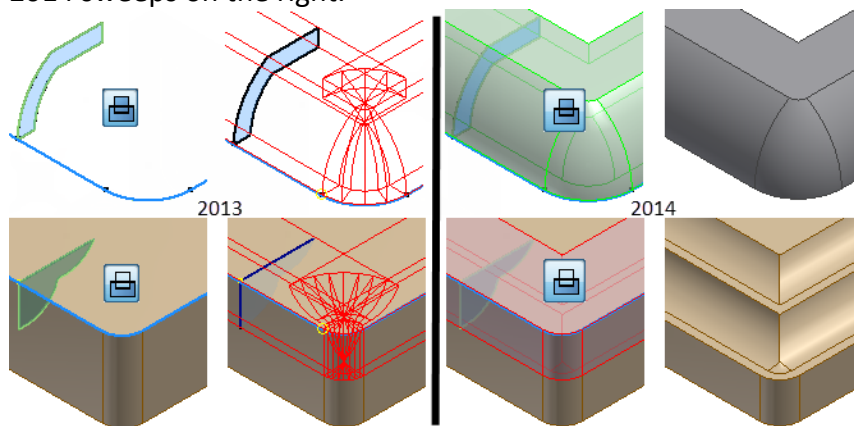
Part

Self-intersecting sweeps

With the enhanced sweep command, you can create sweeps that were previously considered too complex to solve. You can sweep large profiles along small fillets and bends.

The following image displays examples of previous sweep failures on the left, and successful 2014 sweeps on the right.


Non tangent surface path sweeps

 Previously, creating a swept surface using a profile that was perpendicular to a closed G0 (not smooth or tangent) path failed. New sweep functionality lets you create a surface using a perpendicular profile along a closed G0 path.
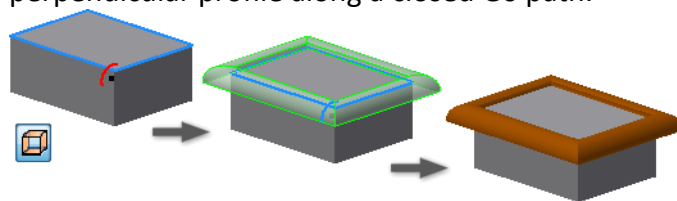
New Boundary Patch options

Two options are added to Boundary Patch.

- **Automatic Edge Chain** Provides a streamlined workflow when you select multiple edges. You can include edges that are not tangent.
- **Boundary Patch** Create complex shapes using the added weight factors for tangent and smooth boundary patch types.
- The following image shows the effect of using different weight factors. The image on the right uses the maximum weight factor of 1.0
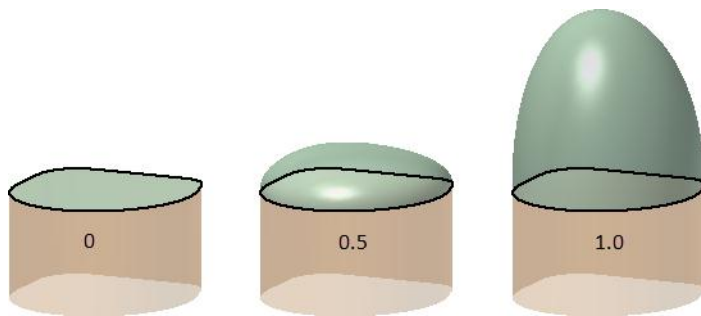

Fillet success

In previous releases, if one edge in a fillet selection set failed, the entire fillet operation failed. In this release, any blend failures open a dialog box with the option to fillet the successful edges.

After you fillet the initial edge selection, you can repeat the operation using the same size radius. This action usually blends more edges.

In the model used in the following image, an additional 17 edges are filleted after the first 142 edges are successfully filleted.
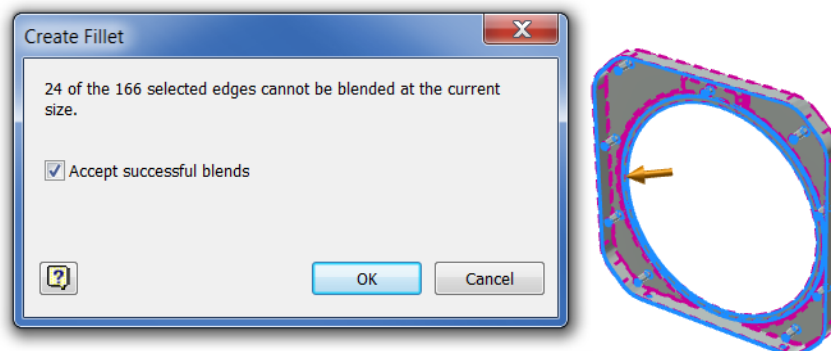
Work Point on a sphere

In previous releases, a sphere was not a valid selection to locate a work point. You can now define a work point at the center of a sphere.
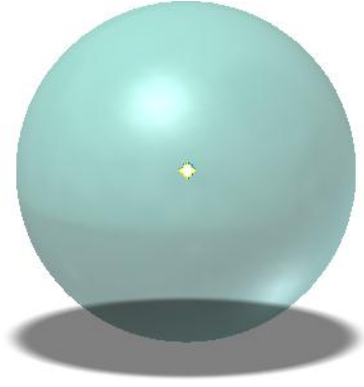
Derived part color override

In previous releases, you were not able to change the color override setting when you placed a derived part. An edit after placing the part was required to remove the color override.

In this release, you can keep or discard the source component color override when you place a derived part.
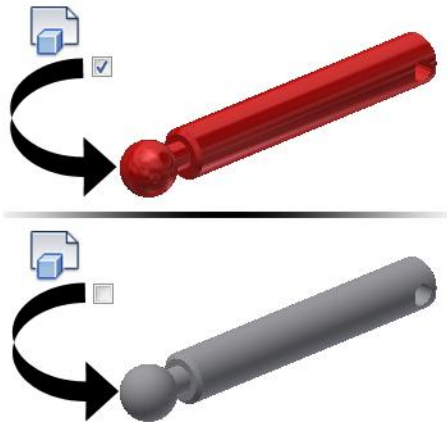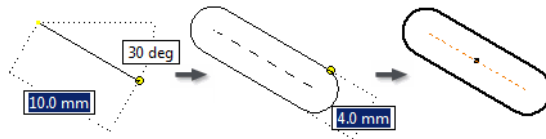
Sketch Design

Linear and arched slots

You can sketch linear or arched slots with a single command, similarly as other basic shapes like rectangles or circles.

Five new Slot commands are available:

Center to Center Slot command creates a linear slot defined by orientation and length of

the slot centerline, and by slot width.

Overall Slot command creates a linear slot defined by slot orientation, length, and width.
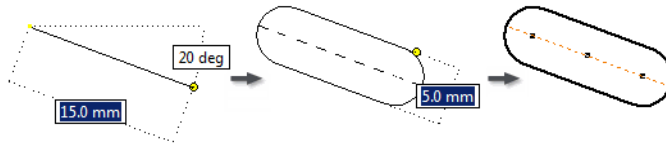
Center Point Slot command creates a linear slot defined by the slot center, orientation and length of the slot centerline, and by slot width.
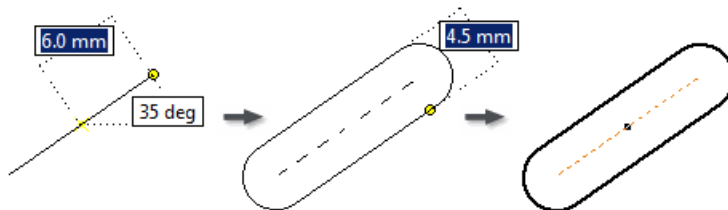
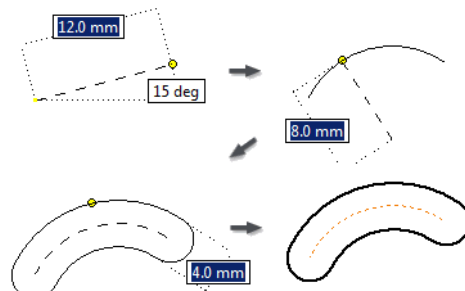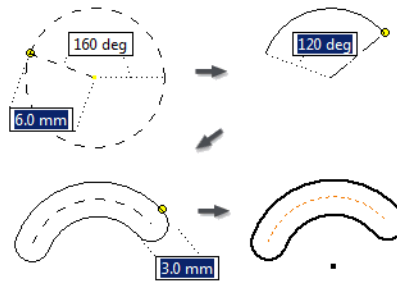**Three Point Arc Slot** command creates an arc slot defined by a three-point center arc, and

by slot width.

Center Point Arc Slot command creates an arc slot defined by a center point, two-point

center arc, and by slot width.

Drawings

Split tables

You can split a parts list, hole, revision, or general table, and then relocate the table segments to the desired place on the drawing sheet.

To split a table, right-click text in a table row, and click Table   Split Table. The table splits into two segments. In the browser, the new table segment displays as a child of the parent table. To cancel the table split, right-click a table segment, and click Un-split Table.
Edits to the parent segment of the table are promoted to child segments. Changes that you make to a child segment do not affect other table segments.
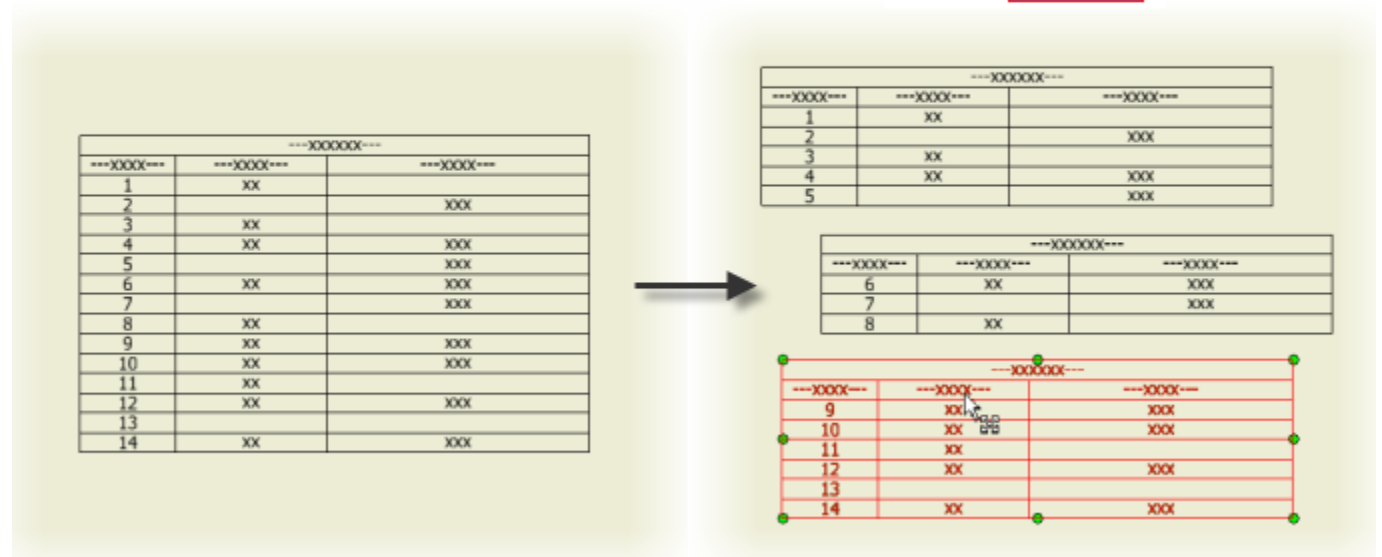
Raster views

A new option in the drawing view dialog box creates raster views that do not become precise over time. You control when to make the views precise.

Raster views can help you defer the cost of precise drawing view recompute when you edit a large model. For example, open the drawing document and convert all views to raster. Open the assembly file and edit the model. The drawing document displays the changes without waiting for a full recompute.

A raster view displays in the browser with a diagonal red line through it. Green brackets frame a raster view in the display.

  Use the browser context menu to convert an individual view to a precise or raster view. Use the context menu in the display and select Make All Views Precise, or Make All Views Raster to convert multiple views.
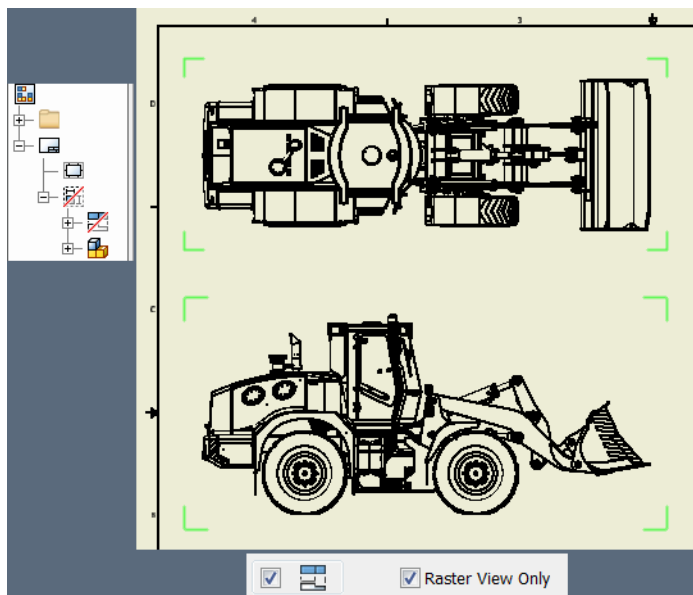
Appearance property

The model Appearance value is added as a selectable property. You can add the Appearance property to a bill of materials, or a parts list.

PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	APPEARANCE
1	1	BP-100	Base Plate	Metal
2	1	SB-101	Slide Base	Sky Blue Dark
3	1	SBE-103	Shaft Ball End	Metallic Paint
4	1	C-100	Connector	Metal
5	1	L3-100	Lever	Smooth - Off-White
6	1	S104	Socket	Nickel
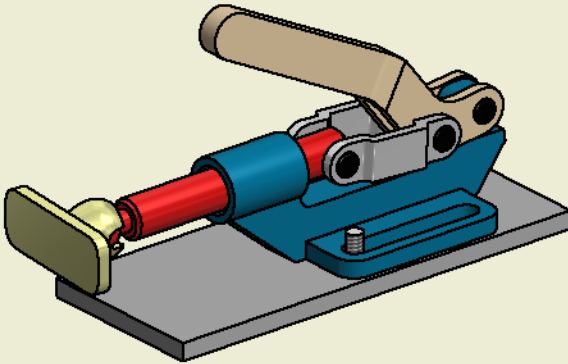
Communication and Interoperability

BIM Simplify

If complete product definition is not required, you can transform your models to simpler representations with Simplify tools in Inventor. You can remove intellectual property from your catalog models before making them accessible to customers. Alternatively, use this workflow to prepare Revit family files in Inventor, which contain native Revit form features.

Simplify Tools for Assembly
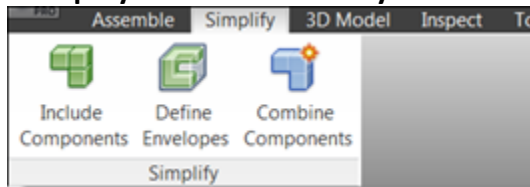
Include Components Create a View representation using selected components.

Define Envelopes Simplify a component or a part by replacing it with a set of box or cylindrical envelopes.

Create Simplified Part Create a simplified part using the current view representation. Hidden components are not included.
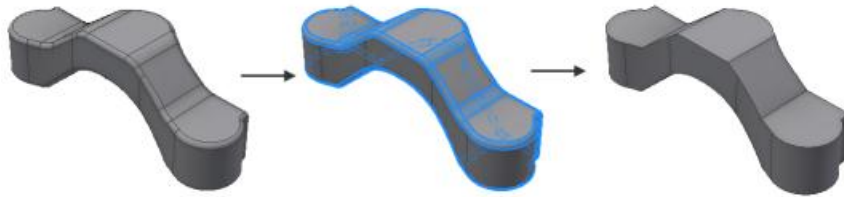
BIM Feature Recognition environment for Part

In the **BIM Feature Recognition** environment, use geometry simplification tools to reduce model complexity. To represent your model with features recognized by Revit, use feature recognition.
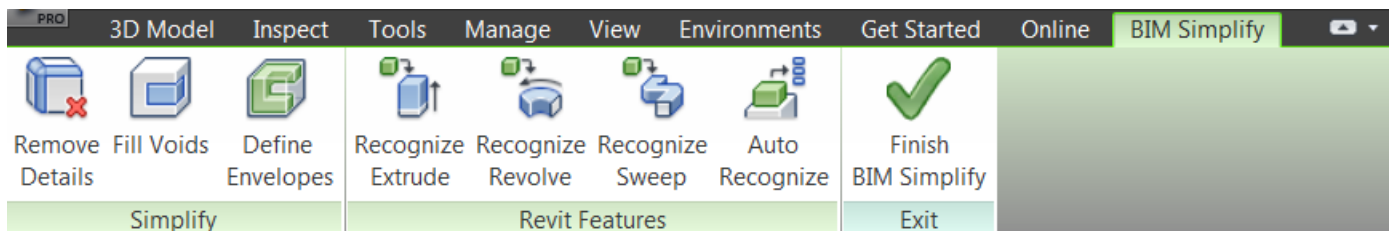
Remove Details Simplify a model by removing recognized fillets, chamfers, or custom-selected faces.

Fill Voids: Cap holes and pockets with surface patches that fill model voids.
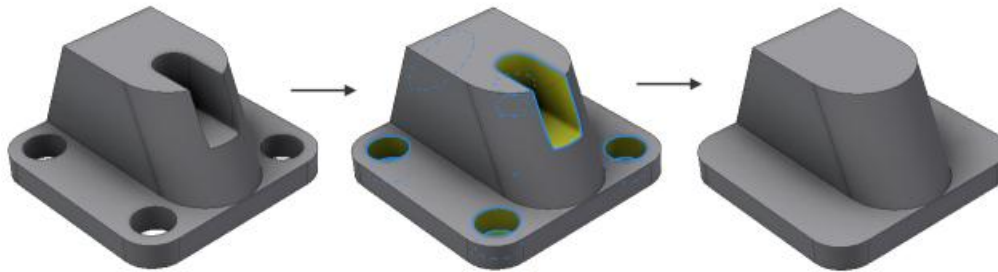
Define Envelopes Simplify a model by replacing selected components with box/cylindrical envelopes or creating simple extrude features that represent their shape.
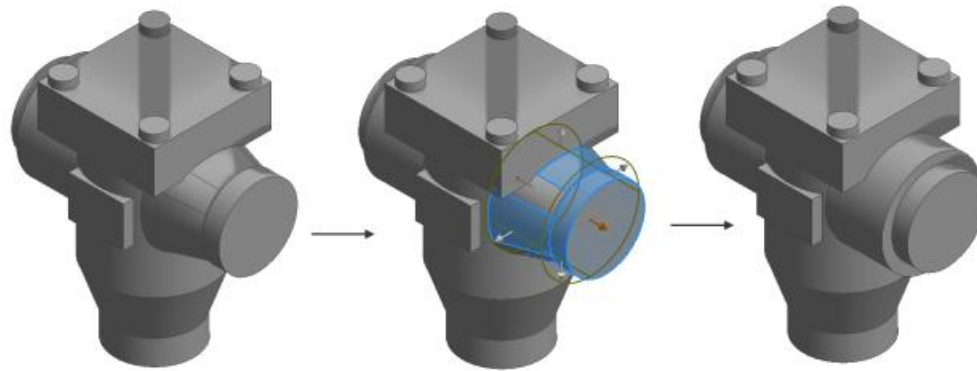
Recognize extrude Recognize extrude features, and convert them to extrusions recognized by Revit.

Recognize revolve Recognize revolve features, and convert them to revolutions recognized by Revit.

Recognize sweep Recognize sweep features, and convert them to sweeps recognized by Revit.

Auto Recognize Automatically recognize extrude and revolve features, and remove fillets and chamfers.

BIM Exchange

A native Revit Family file can be exported using **Export Building Components** in the **BIM Exchange** environment for **Part** documents. In addition, Connectors, Properties, Parameters, Orientation, Category and Insertion Point information can be added to the Revit Family file format.

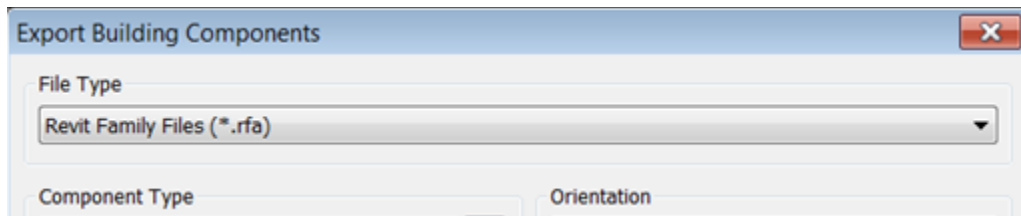Before exporting a Revit Family file, Inventor users must prepare the model so that model features can be translated into native Revit form features. In Inventor 2014, extrude, revolve and sweep features can be translated. Inventor users can convert an existing model using the BIM Simplify environment, or they can create a compatible model by using only these three features to construct the model geometry.

Check Revit Features

Displays an HTML report with information on features the model contains as defined in the Revit Family file feature translation rules.

This command is used to determine how successfully the features will be translated when exported as a Revit Family file. The report contains the status for each feature translation and the reason(s) for any unsuccessful translations.

Translators

Support for exporting the following versions is added. You can now export:
- Parasolid version 9.0 through 25.0
- Pro/ENGINEER Granite version 1 through 8.0
- CATIA V5 version R10 through V5-6R2012.

- Support for importing the following versions is added. You can now import:
- CATIA V5 version R6 through V5-6R2012
- NX version Unigraphics 13 through NX 8.0

- SolidWorks version 2001 plus through 2012
- Up to Wildfire 5.0, Creo Parametric version 1.0 through 2.0

Revit interoperability sub-component

The Revit interoperability sub-component allows you to:

- Import a .rvt (Revit Project) file in the Place Component dialog box.
- Export a .rfa (Revit Family) file in the Export Building Component dialog box. A Revit Family file, is content file in Revit, such as a chiller, pump, air conditioner, furnace, tank, furniture, etc.

You must select the checkbox to install Revit interoperability functionality to interact with Revit models to extract and record building information. If you do not install the Revit interoperability sub-component, you will not be able to import and export the Revit files.

Point Cloud enhancements

Autodesk ReCap indexes Point Cloud scan files, and creates .rcs and .rcp files that you can use in Inventor. You can collect multiple scans into a single Reality Capture Project (.rcp) file, and then attach them as a group within Inventor.

 The new Insert at Origin setting allows you to insert the Point Cloud at 0,0,0 of the layout when attaching a Point Cloud. By default, the insert point is an arbitrary point (the pick point on the graphic screen).

Autodesk 360

The new Launch Website command, located in the Autodesk 360 tab, launches the Autodesk 360 website, where you manage files stored in the cloud.

You can Open from and Save files to Autodesk 360 from the top of the locations panel in file dialog boxes such as the Save As, Save Copy As, Open, Place Component and Create Component dialog boxes. Autodesk 360 is available in all file dialog boxes that include a locations panel.

While adding one file that has dependencies to the Autodesk 360, you can use Autodesk 360 to add all file references. For example, adding one assembly file that comprises ten part files adds all ten part files, and the project file. This new feature tracks and uploads/downloads the referenced files automatically.

A new Autodesk 360 drive is included in Window Explorer. You can drag and drop documents to the Autodesk 360 drive from Windows Explorer. Files added to your Autodesk 360 drive from Windows Explorer will automatically upload and sync with your Autodesk 360 account. All referenced files are uploaded during this process.

An Autodesk 360 notification icon is automatically installed in the taskbar notification area. A message displays if any files you have added to the cloud are not in sync with the corresponding file on your local computer. Mouse over the notification icon anytime to check status.

Autodesk Inventor IdeaStation

It's our mission to make Autodesk Inventor the world's best design solution. But we can't do it without you.

The Autodesk Inventor IdeaStation is a forum dedicated to improving Autodesk Inventor software by sharing your ideas for improving the product directly with the developers.

We need you to help us with our mission to make Autodesk Inventor the best design solution in the world. Do you have a great idea for a new feature or enhancement you'd like to see in Autodesk Inventor? Share your wish list directly with the Autodesk Inventor Development Team, and let us turn your ideas into reality.

Post your ideas, see what your peers are thinking, and give kudos to the ideas you like the best. The more kudos your idea receives, the better the chance of it making it into the product. At the IdeaStation, you can even track the status of ideas as they grow from initial concept to ultimate release!

The Inventor Team is listening. We consider all great ideas for inclusion in our development plan.
  Share your ideas now at the [Autodesk Inventor Idea Exchange](#).

CAD Administrator Tools

Team Web provides access to your custom help

If you require specific workflows designed to meet your company standards, and if the content is subject to change, use Team Web. Launch a site on your company-wide intranet using the Team Web command, and the More Learning Resources link on the Welcome screen. Team members can quickly access customized company help.

Configure your site in the Application Options dialog box, Files Tab. Specify a Web site, page, or file you want the Team Web command to launch. This customized link is accessible on the Welcome Screen > More learning Resources, and on the Help menu > Team Web.
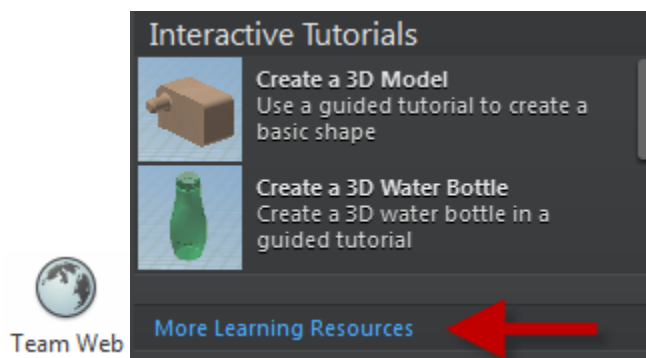

Include assembly graphics in Update Design and Migration tasks

In Task Scheduler, in both the Update Design and Migration tasks, you can control whether to include assembly graphics. Saving the graphics allows Express workflows. The option, Include assembly graphics for Express Mode, is added to the options dialog box for each task. It is turned off by default.

Enhancements for Materials

**Improved navigation in the Material Browser**
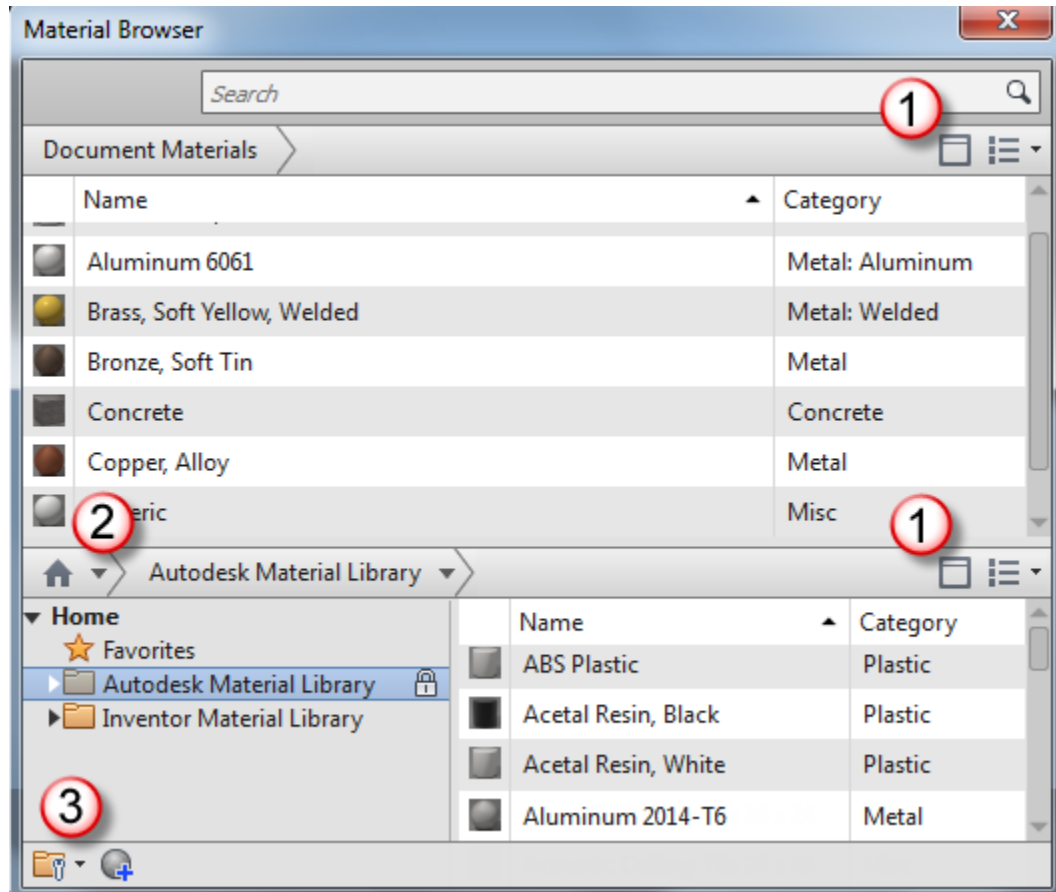
New to the browser:

1. Show/Hide library panel and tree icon give you greater control over your view.

2. Home button to control the display of your saved libraries.

3. ![icon] icon provides more access to Library operations.

New tabbed interface in the Material Editor

Navigate easier within the Material Editor dialog box using its new tabbed interface. In this interface, you can distinguish the various material assets and their properties.
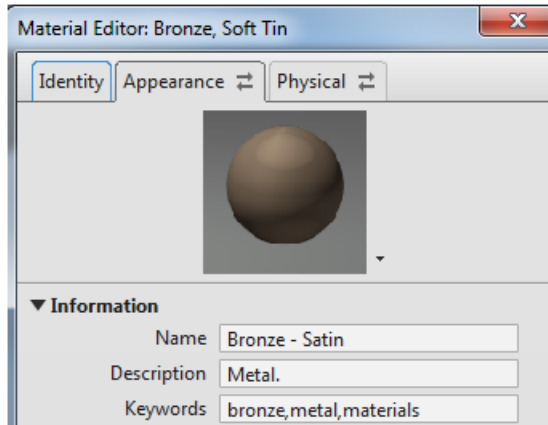
Multi-select You can now multi-select materials and appearances to add to a library

Lock & refresh icon When being modified, the shared library for both material browser and asset browser display a lock icon.
The shared library for both material browser and asset browser display a refresh icon to indicate that a document material and/or appearance is out of date. You can update by clicking the refresh icon.

Visual aids Italic and underline text (with tooltips) indicate differences between the document materials and appearances and the library. significant number of enhancements were made in Inventor 2013 Service Pack 1.1. Review the Autodesk Inventor 2013 Service Pack 1.1 for full information.

New notifications and behaviors for shared assets

You now have greater control over assets in your material and appearance libraries:

- You will be notified when you add or copy an asset (appearance) to a library if that asset name that already exists in the library. When you add an asset to a shared library that has the same name and different properties, you will be notified that an asset with that name already exists and have the option to keep both or replace. If you choose to replace, all materials referencing the asset are updated to use the new asset.
- You will be notified when you rename an appearance asset that is in use. For example, if you rename the color Green to Bright Green and Green is referenced by multiple materials, all materials referencing the asset Green are updated to use the new color name, Bright Green if you choose to rename the asset.
- The option to delete is now disabled on an asset that is shared with one or more materials in a library. You can now find out where the asset is being used by selecting the new **Where Used** from the context menu in the Appearance Browser.
- You will be notified when attempting to add a material to a library with a material name that already exists in the library and have the option to keep both or replace.

Deploy additional software

Include add-ins, components, and other applications when creating a deployment. On the Create Deployment product landing page, click anywhere in the **Include Additional Software** box to access the controls.

Control access to Autodesk Exchange

Block access to Autodesk Exchange when creating a deployment. On the Create Deployment product landing page, click anywhere in the **Autodesk Inventor 2014** box to access the Autodesk Exchange Apps Access control. Select to block access to Autodesk Exchange within Inventor.

Export errors from Task Scheduler log file to Excel

  Task Scheduler generates log files in .mdb format to report task errors. You can export the log file entries of one or more tasks from the Task Scheduler .mdb file format to an Excel spreadsheet.

The Excel file generates a separate sheet for each error type. The first column of each sheet lists the error files. The second column lists the corresponding task names.
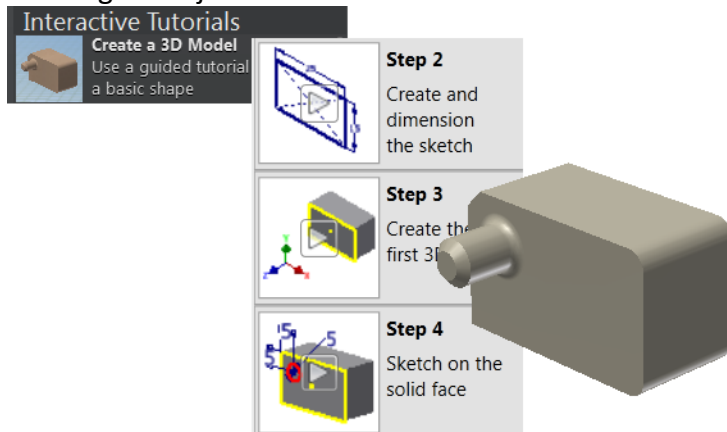
Export Errors to Excel is available on the Task Scheduler File menu, and on the context menu for a selected task.
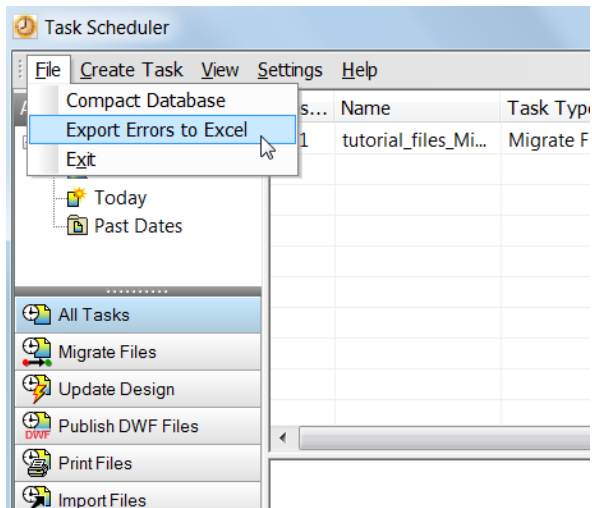
Learning Tools

Basic skills interactive tutorial

A new basic skills interactive tutorial steps you through the commands and workflows for creating 3D objects in Inventor.

Connect interactive tutorial

A new tutorial introduces the new assembly Connect command. The tutorial leads you through applying explicitly defined connections to locate components and define motion in one step.
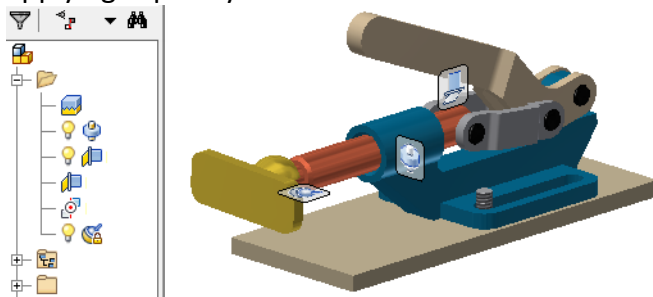
Create interactive tutorials

You can create your own interactive tutorials for internal use or to share with others.

Download the instructions and starter files for 64-bit systems here:

http://www.autodesk.com/interactive_tutorial-2014-64bit

Download the instructions and starter files for 32-bit systems here:

http://www.autodesk.com/interactive_tutorial-2014-32bit

Location for interactive tutorials no longer restricted

The interactive tutorials were previously installed in the Program Files area, which is a restricted access location. The new install location is in a public area. With no administrator rights required, anyone can add new interactive tutorials.

  The new location is: <drive>:\Users\Public\Documents\Autodesk\Inventor 2014\Interactive Tutorial\en-US\

Interactive tutorial visual enhancements

The new size, shape, and color of the ribbon command pointer, and the video play button provide greater visibility.