AUTODESK UNIVERSITY 2013

60 Autodesk® Inventor® Tips in 60 Minutes

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Autodesk Inventor 3D CAD software has been the Autodesk flagship product for Digital Prototyping. Industries across the globe have been using Autodesk Inventor to create their designs and produce new digital prototypes. Over the recent years, productivity has been the key factor for any industry to get their designs or products to market faster. The emphasis on productivity often requires engineers and designers to work under great pressure. Come to this class to learn Autodesk Inventor tips and tricks to enhance your productivity and work with ease.

MA2822

Learning Objectives

At the end of this class, you will be able to:

Learning Objective 1: Be more productive while using Inventor
Learning Objective 2: Employ productivity-enhancing techniques while using Inventor
Learning Objective 3: Customize Inventor
Learning Objective 4: Use hidden capabilities in Inventor

About the Speaker

Sunith, after completing his Master's Degree in CAD/CAM started teaching AutoCAD and Autodesk Inventor for more than 10 years both online and at training locations across India. He's popularly known as CADProfessor & blogs at www.cadprofessor.in one of the highest rated blogs. He has taught more than 10,000 students, faculty and Industry Professionals till date on various CAD topics. His articles have reached AUGI Hot News, AUGI World Magazine, AUGI ATP, Autodesk Community India, AUGI India, number of reseller newsletters across the world. He has been teaching since 2003 on how to increase productivity in using CAD Tools. He is a faculty member at M.S. Ramaiah Institute of Technology, Bangalore and currently teaching various products of Autodesk to both undergraduate and post graduate students. He is one of the first Autodesk Certified Instructor & Autodesk Certified Evaluator from India and has conducted ACI Programs at India & Middle East.

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Autodesk Inventor Blog - http://goo.gl/R0BrRp

AutoCAD Blog - http://goo.gl/pSRs3k

AutoCAD & Autodesk Inventor Tips Blog - http://goo.gl/seJ3Uu

Twitter: @CADProfessor

Introduction

Autodesk Inventor is the foundation tool for Digital Prototyping. Every year Autodesk releases new version of Inventor which provides newer ways to solve a project sometimes, companies don't move ahead with subscription which can result in decreased productivity of the designers. As a result company loses the appropriate workflow needed for a particular project.

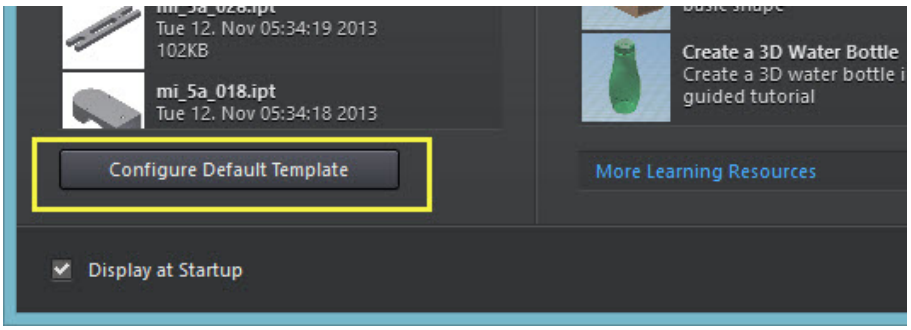
The class MA2822 showcases tips and techniques available in Autodesk Inventor and enables a user to learn different ways to be productive when working with Autodesk Inventor and also be more competitive, some tips exclusively available for Subscription Customers are showcased making a user / company understand the benefit of subscription.

Note: This hand out gives an exhaustive information on how to achieve productivity step by step.

Autodesk Inventor User Interface Tips

1) Units / Template type

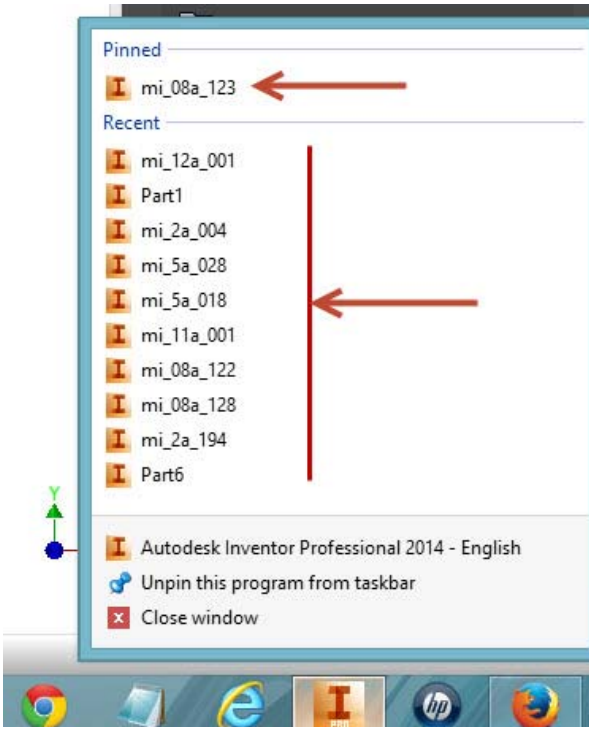
Units in the earlier version of Inventor could be changed only after entering into sketch / part / assembly environment. However this stands into effect only after restarting inventor. However this is now overcome by introducing Units in the Welcome Screen. A nice and quick way to change the units.



Access Default Template from Welcome Screen

2) Windows taskbar

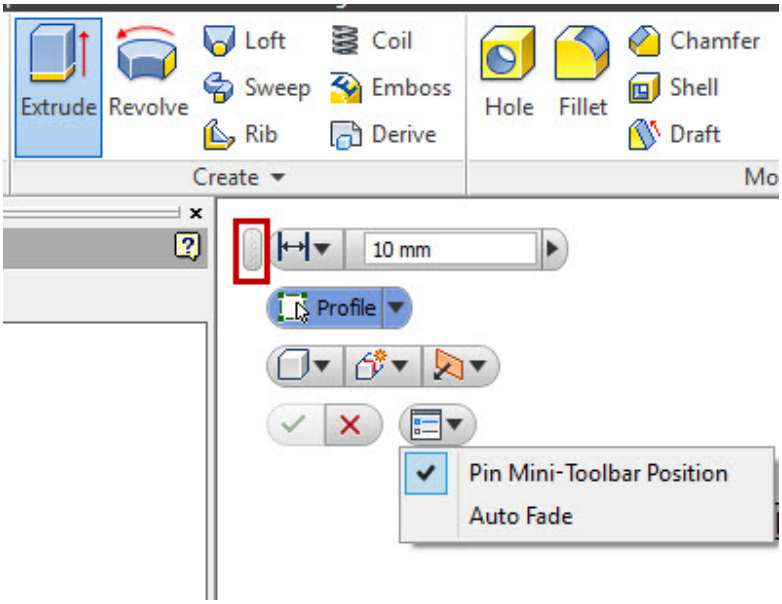
A user can open the recently created / opened files from Autodesk Inventor right from the Taskbar by Clicking and moving in the upward direction. There by helping a user save time in opening the model



3) Mini Toolbar customization

Mini Toolbar helps users to quickly make changes / inputs for a particular command. This has two options Pin Mini-toolbar position / Autofade

Note: This can be placed at Top Left Corner of the Graphics User Interface and also use the auto fade option if needed.



Mini Toolbar Option in Autodesk Inventor

4) Quick Access Toolbar

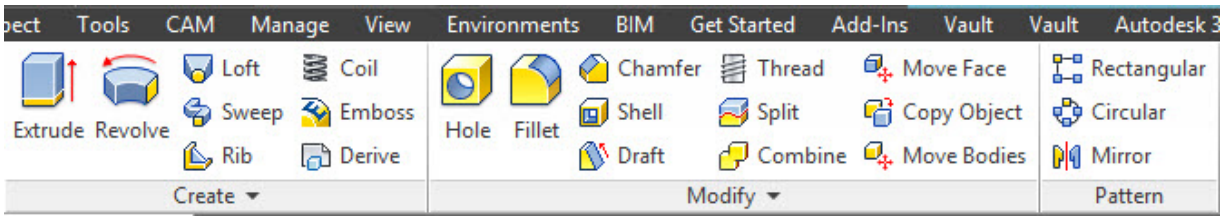
The Quick Access Toolbar (QAT) at the top of the screen displays commands across all environments. It includes frequently used commands, such as Undo and Redo.

You can add any number of commands to the QAT. For this class MA2822, since I use frequently Application Options. I can place the command in the QAT.

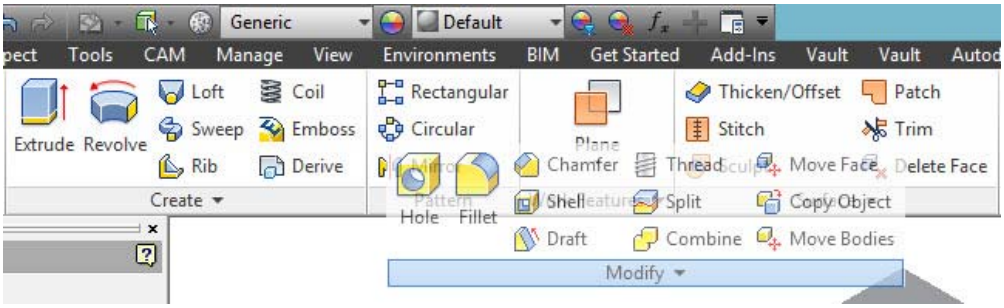
Quick Access Toolbar with Application Options

5) Working with Sticky Panels in Inventor

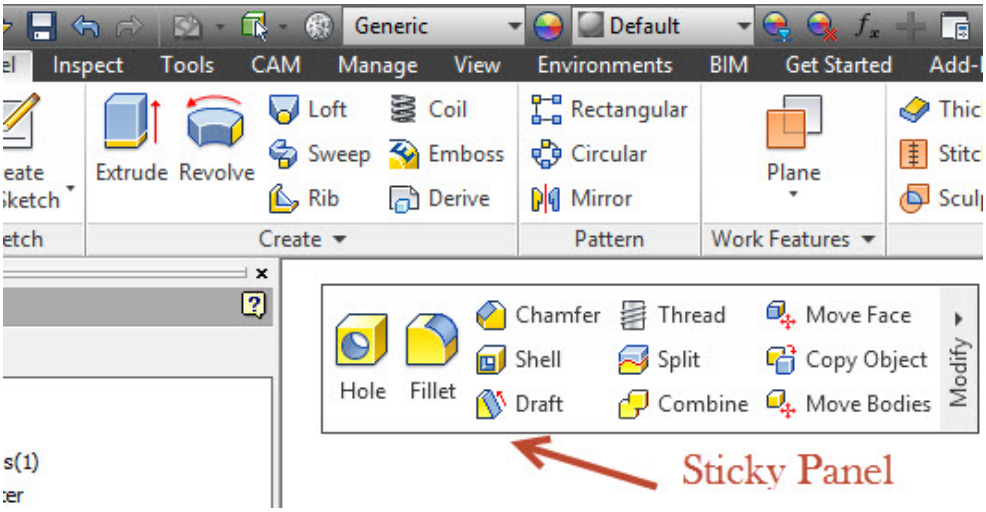
Sticky Panels are subset of panels found in the ribbon. Any panel within the Inventor tabs can be made as a sticky panel by dragging them away from the Ribbon and placing them in the GUI. The main advantage is that the commands within the panel can be accessed quickly rather than moving the cursor to the panel location. These Sticky panels appear to be present on the screen when other tabs are selected in Inventor. Making them accessible always.



Sticky Panels



Sticky Panel for Modify is being created



Sticky Panel

6) Function Hotkeys in Inventor 2014

F1 - Help	F6 - Home View
F2 - Pan	F7 - Slice Graphics in sketch mode
F3 - Zoom	F8 - Show All Constraints
F4 - Orbit	Alt + F8 - Open Macros Dialogue Box
F5 - Previous View	F9 - Hide All Constraints
Shift F5 - Next View	F10 - Alt + Tab

More shortcuts - **Click Here**

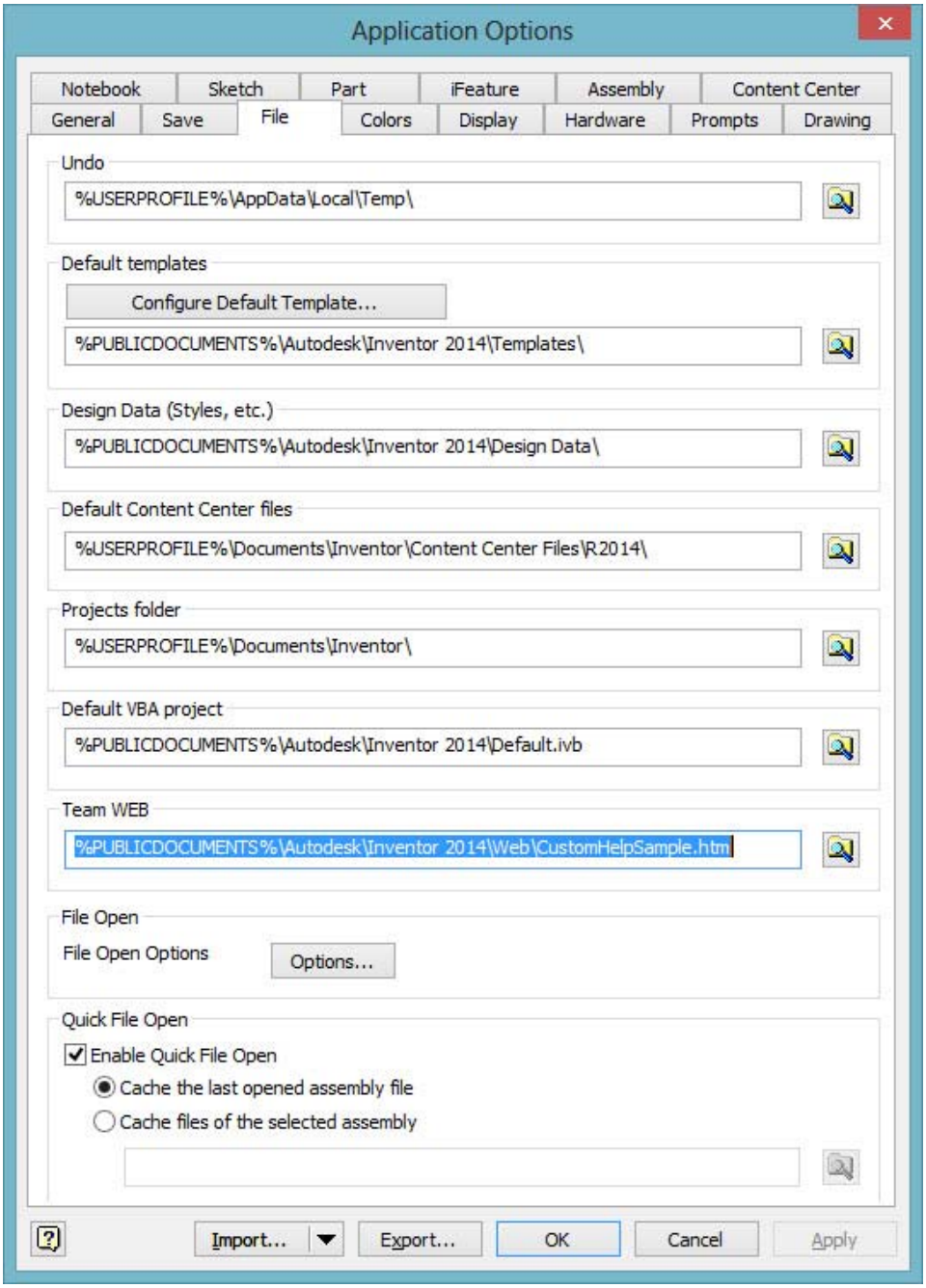
7) Inventor TEAM WEB

Specifies the location and name of the default URL or file launched by Team Web. Configure Team Web to launch the default learning resource of your choice. Access Team Web from Getting Started ➤ Launch panel ➤ Team Web.

%PUBLICDOCUMENTS%\Autodesk\Inventor 2014\Web\CustomHelpSample.htm

The above file location can be replaced by web URL of your choice or intranet location within a company

Link can be changed from Application Options ➤ File ➤ Team Web

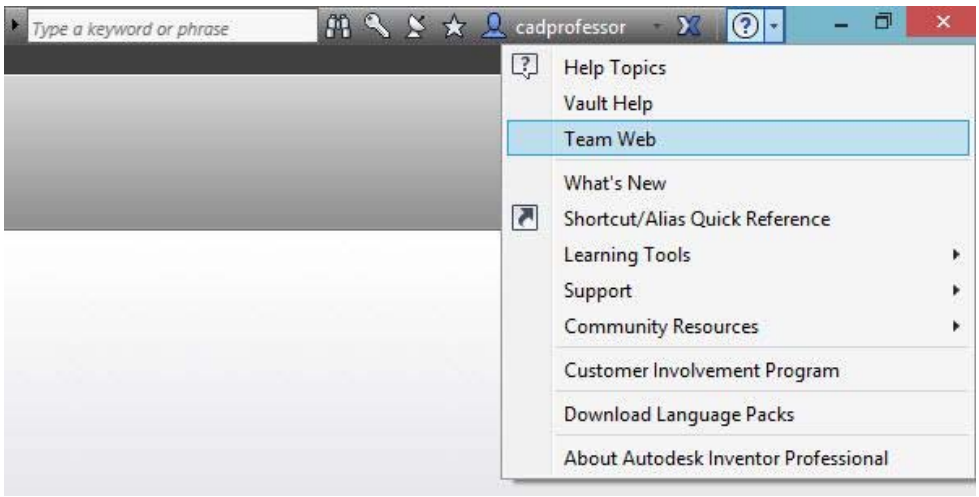


Team Web Customization

Different ways by which you can access TEAM WEB

1) Getting Started Tab >

2) Access Help from Top Right Corner of Inventor Screen >



3) Access from Welcome Screen

8) Access Help within Autodesk Inventor (User Interface)

If your organization does not allow access to internet or you work with Inventor on the go and want to access Inventor 2014 help then it becomes really difficult. Here is a simple tip to access Inventor 2014 help within your hard drive (393 Mb file)

Link –

English Autodesk Inventor 2014

Help Installer (exe - 402147Kb)

Help Install Instructions (html - 22Kb)

English Autodesk Inventor 2014

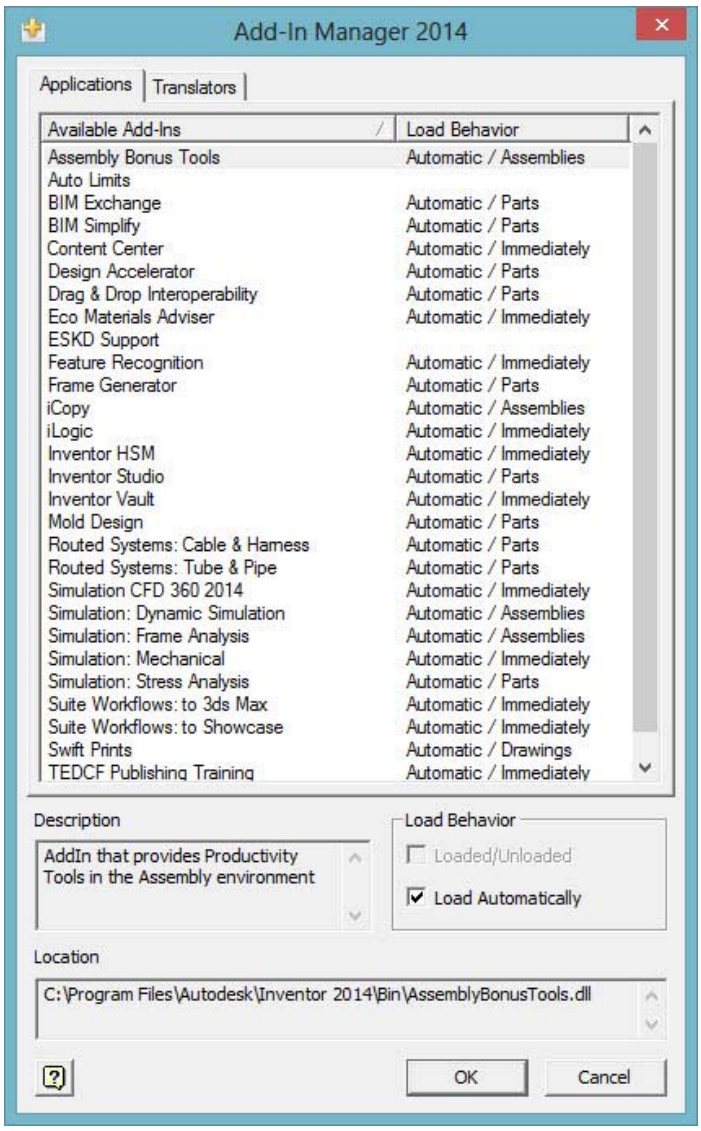
1. Access C:\Program Files\Autodesk\Inventor 2014\Bin\helpids.xml and ensure the UseOnline value is set to '1' as follows: UseOnline="1". If the value is set to '0', change the value to '1'.
2. Run regedit.exe
3. Access HKEY_LOCAL_MACHINE\SOFTWARE\Autodesk\Inventor\RegistryVersion18.0
4. Double-click **EnableOnlineHelp**
5. Change the Value Data field to 0.

9) Add-In Manager in Autodesk Inventor

The Add-Ins button opens an Add-In Manager dialog box that lets you select the add-ins you want to load or unload when Inventor starts up. You can access this tool by choosing Start ➢ All

Programs ➢ Autodesk ➢ Autodesk Inventor 2014 ➢ Tools or, when in Inventor, by selecting the Tools tab and clicking the Add-Ins button found on the Options panel. Once the Add-In Manager dialog box is open, click the add-in in the Available Add-Ins area, and at the bottom of the dialog box under Load Behavior, deselect the Loaded/Unloaded option to unload it.

Note: If there are add-ins you know you do not use, you can use the Add-In Manager to prevent them from loading when Inventor loads. This will slightly speed up the load time (and reduce the amount of RAM Inventor uses). Be sure you understand what specific add-ins do, because some are required for the proper operation of the software.



Add-In Manager in Inventor

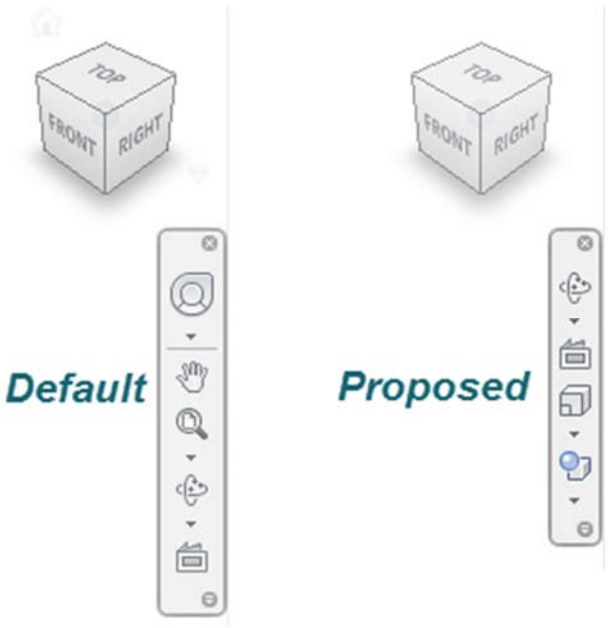
10) Using a Wheel Mouse and 3D-Input Device (Tip published at Cadalyst)

Wheel mouse in Inventor is recommended and this helps in controlling the PAN and ZOOM Scrolling the wheel will perform a Zoom In/Out, while pressing the wheel will perform the Pan function. In Inventor, the wheel zoom is reversed from AutoCAD. You can change this setting by clicking Application Options on the Tools tab, selecting the Display tab, and selecting Reverse Direction in the 3D Navigation group.

Customize the Navigation Bar

The navigation bar sits below the View cube in the upper right portion of the graphical canvas. By default, it has quite a few commands that are not really needed if you are using your mouse every day. Commands like the Steering Wheel, Zoom All, Pan, and Orbit are not terribly useful unless you travel with a laptop and forget the mouse.

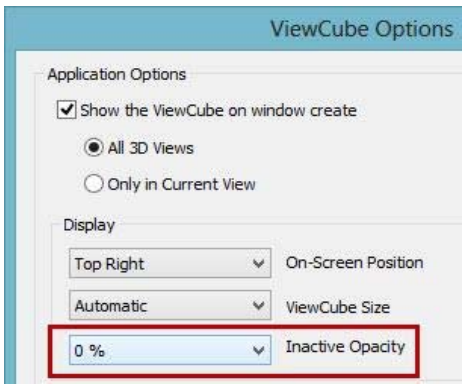
Typically, users can use the mouse center wheel for Zoom (scroll) and Zoom All (double-click it), and for zoom either use a 3Dconnexion mouse or pull out a keyboard trick and hold Shift while holding the middle mouse button. This leaves the Navigation bar pretty useless. The little arrow at the bottom of the bar allows customization of the commands. Here a user can have Projection and Visual Styles provided a user works in technical publications as well as PowerPoint.



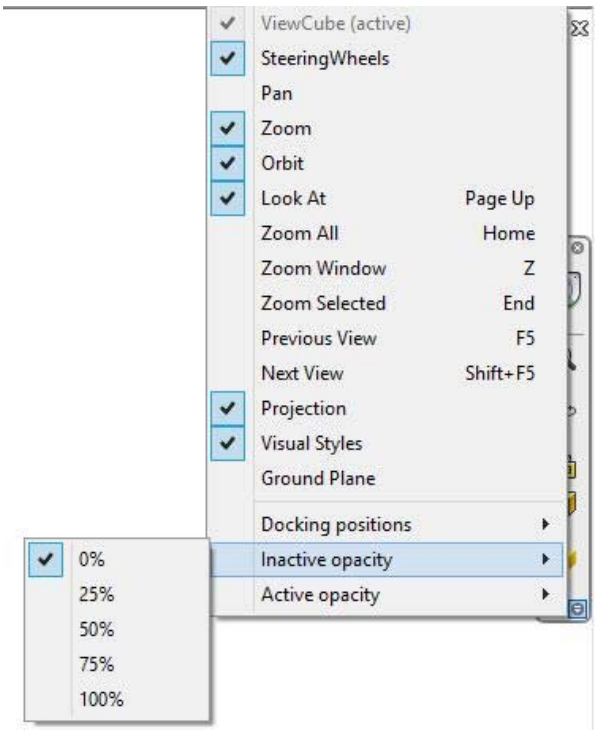
Note: Adding commands to Navigation bar below view cube is presently not supported. If available can further increase productivity.

11) Opacity Control of View Cube:

Opacity Control of View Cube provides clear work space with the CAD model in Part / Assembly / Drawing environment and a user can avoid panning. With Opacity set to 0% additional **workspace of 10%** is obtained when viewing a model within Autodesk Inventor.



View Cube Opacity Control



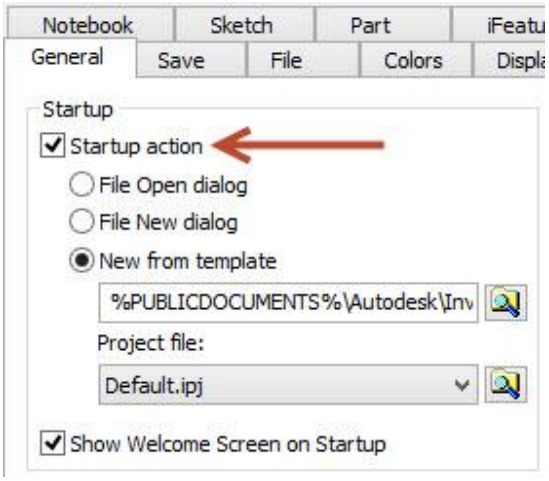
Inactive Opacity - 0%

Sketching:

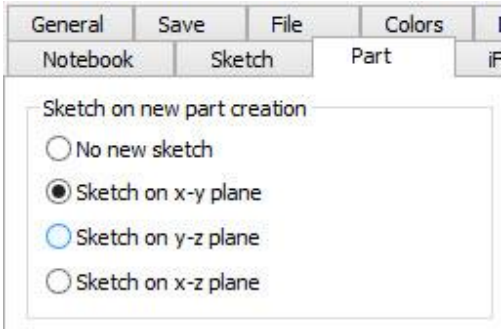
12) Load Sketching in Autodesk Inventor 2014

You can open Inventor Sketching Environment when you load Autodesk Inventor there by reducing time and increasing productivity when working with Autodesk Inventor. Open Application Options > General > Startup > Startup action > New from Template "%PUBLICDOCUMENTS%\Autodesk\Inventor 2014\Templates\Standard.ipt"

Note: The *.ipt file units depends on the type of template we select.



Startup Selection of template (Sketching)



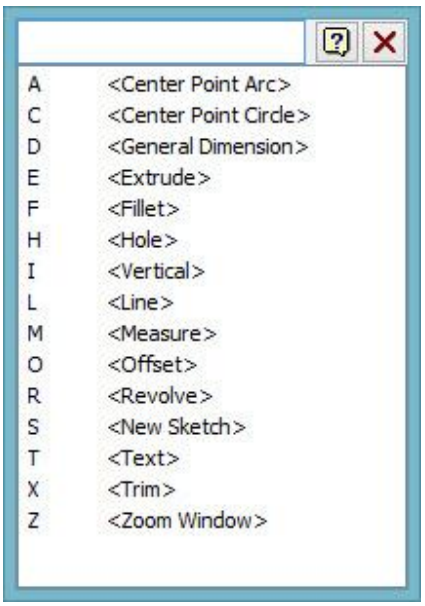
Select Sketching on X-Y Plane

13) Command Prompting in Inventor 2014

Similar to AutoCAD, Inventor also has a Prompting Interaction which helps a user to

- Show Command Prompting (Dynamic Prompts)
- Show Command Alias input dialogue
- Show autocomplete for command alias input

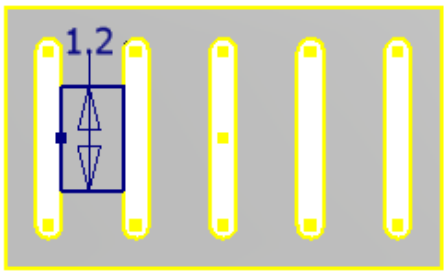
Command Prompting in Application Options



Command Prompting in Sketching Environment

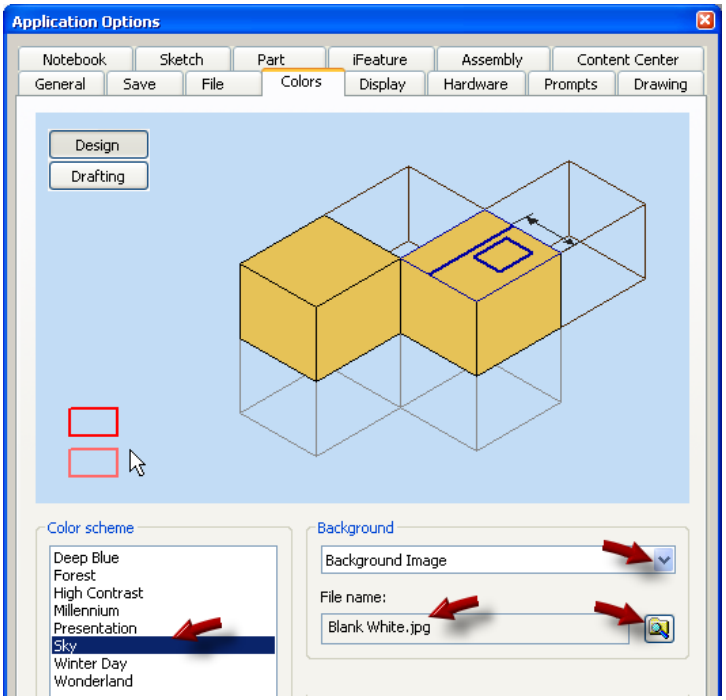
14) Autodesk Inventor White Background

Solid white back ground in Inventor can always be used because it allows a user to take quick screen captures for use in illustrations for a variety of publications (examples: how to videos, company standards documentation, build procedures, catalogs, and so on). However this poses a problem where the projected geometry appear as yellow and working with yellow lines on a white background makes these projected sketch lines difficult to see.



Projected Sketch

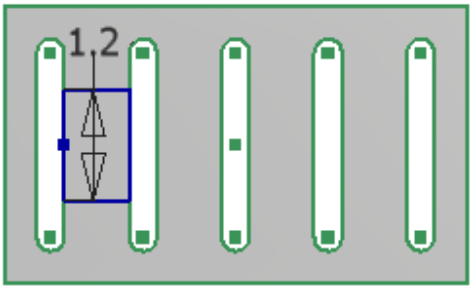
Additionally, if you use Inventor during a presentation or training, the white background used in the Presentation color scheme often shows up best, but again the yellow projected lines makes it difficult to work with, particularly with some projectors.



Changing background images

Rather than changing the projected line color used in the Presentation color scheme, you can use one of the other color schemes, and use a solid white background image. To do so, go to the **Tools** tab > **Application Options** button > **Colors** tab, and then change the **Background** setting to **Background Image**. Then use the browse button for the **File Name** setting to browse out and select a previously saved image file.

A user can select Sky color scheme (it uses green lines for the projected geometry color), with a file called Blank White.jpg. I created the file called Blank White.jpg using Microsoft Paint and saved it to the Inventor Backgrounds directory.



Better appearance

Yellow Line that comes because of Projected Geometry - to change this, use presentation with White Background (Save as Image - png) and use Sky as option and then change it to background option.

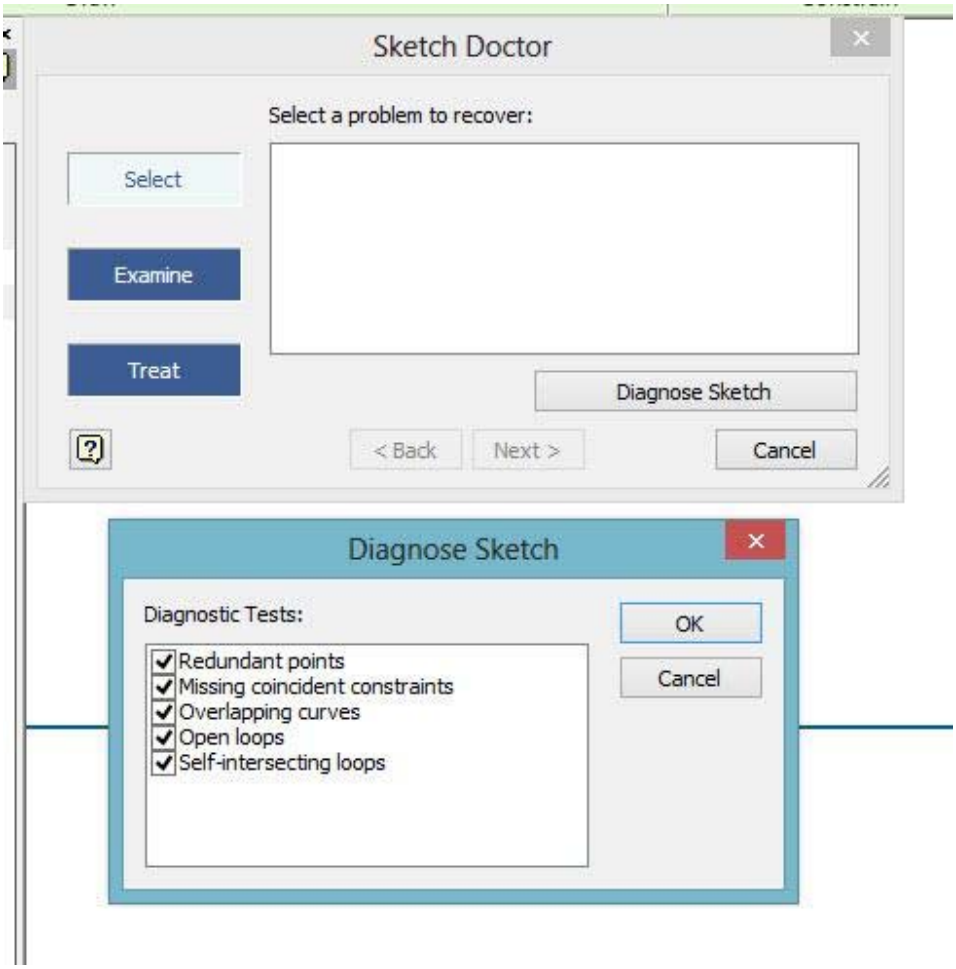
15) Sketch Doctor: (Sketching)

Sketch Doctor in Autodesk Inventor helps users rectify sketch related issues. When a user selects sketch doctor by Right Clicking in sketching interface, the sketch doctor dialogue box opens. It is divided into three segments Select > Examine > Treat

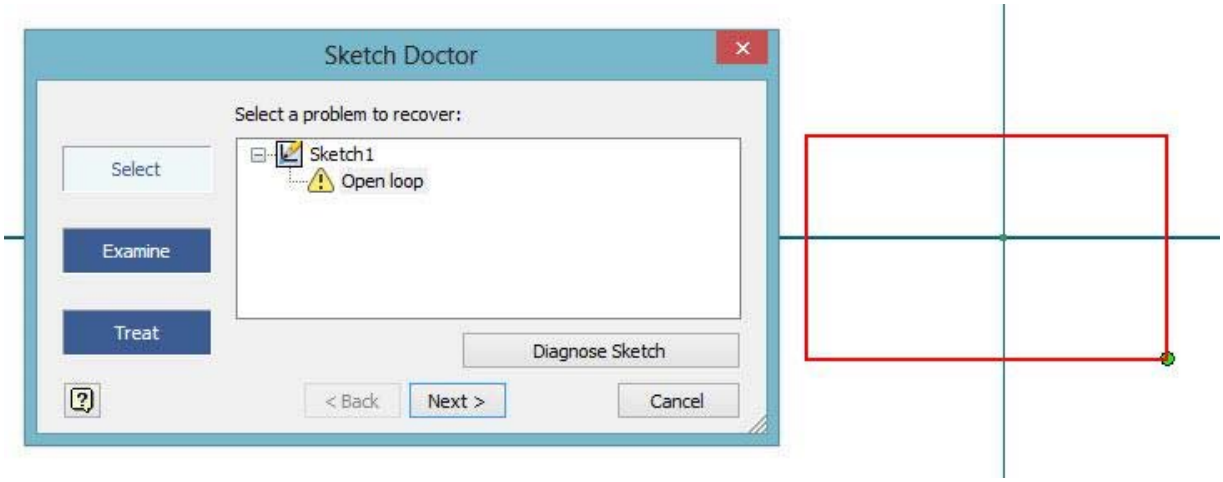
Select: This helps a user to select the sketch and diagnose a sketch to check for redundant points, missing coincident constraints, overlapping curves, open loops and self-intersecting curves.

Examine: This helps a user to identify the type of problem that exists in sketching and provides a possible solution to rectify the same.

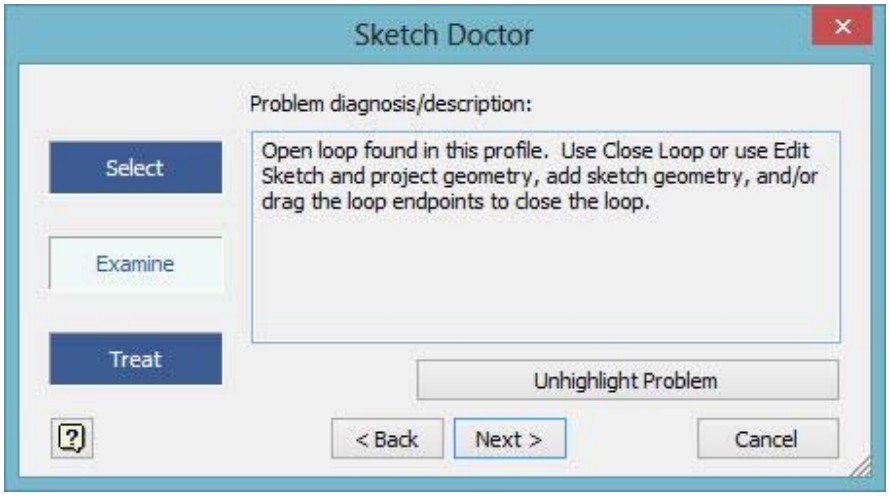
Treat: This helps a user to close the open loop sketch or solving the error within the sketching environment.



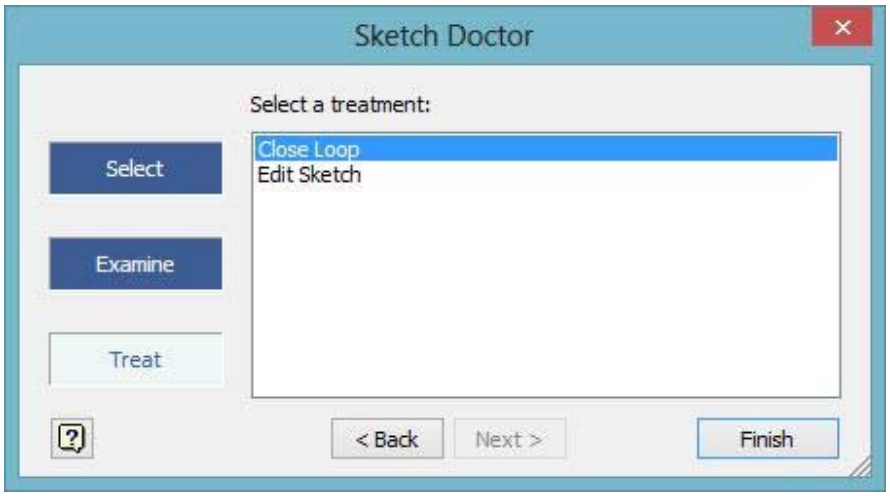
Sketch Doctor Selection Process



Identification of Problem



Examination of Problem

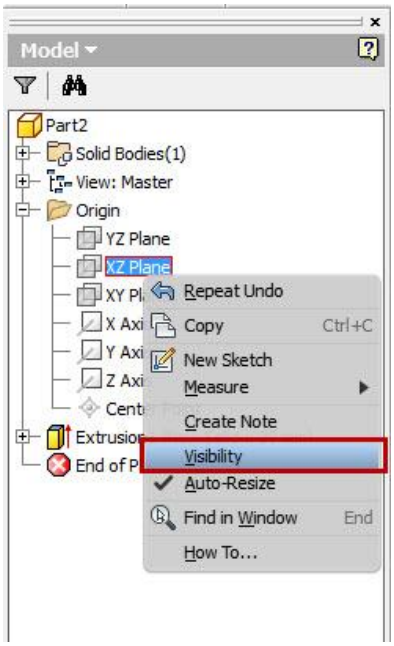


Treating of problem identified

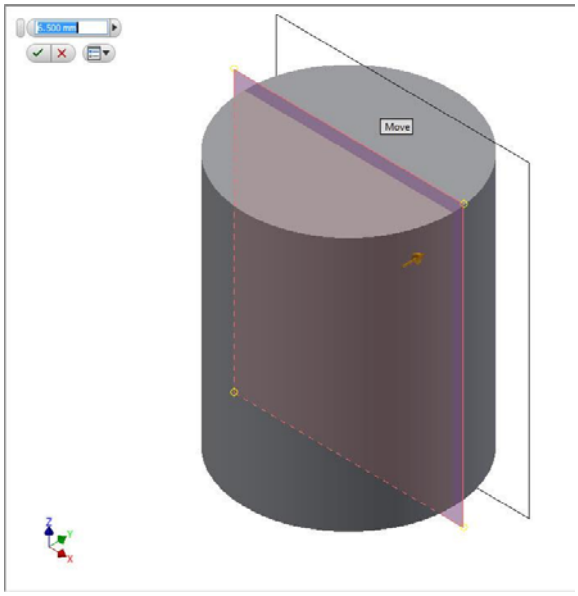
16) Creating an Offset Workplane and Sketch

The ability to create an offset work plane and sketch at the same time is something a quick process. Conventionally we create a plane later offset the plane and then use sketching tool to sketch on that plane.

The above tip can be executed by selecting one of the Origin Planes from the browser bar (active - visible) followed by pulling the plane after selecting sketch command to required distance.



Sketch Plane Selection



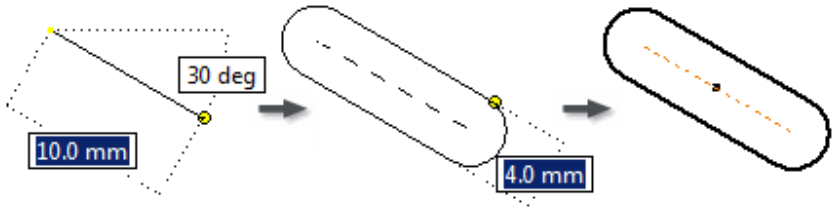
Moving the Sketch Plane from the Origin Plane (X-Y)

17) Slots

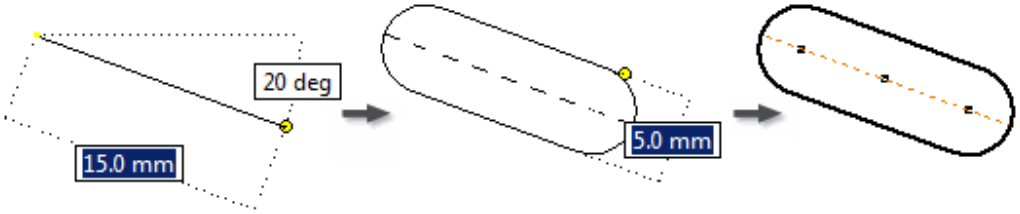
You can sketch linear or arched slots with a single command, similarly as other basic shapes like rectangles or circles.

Five new Slot commands are available:

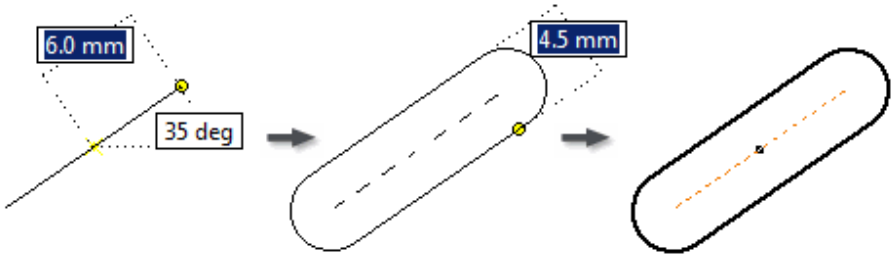
- **Center to Center Slot** command creates a linear slot defined by orientation and length of the slot centerline, and by slot width.



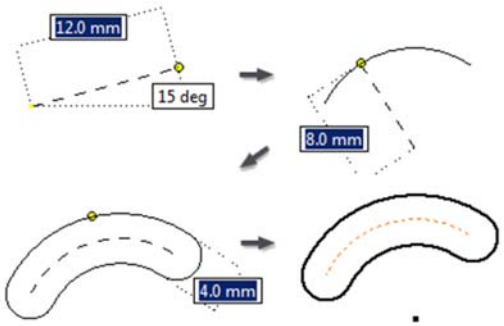
- **Overall Slot** command creates a linear slot defined by slot orientation, length, and width.



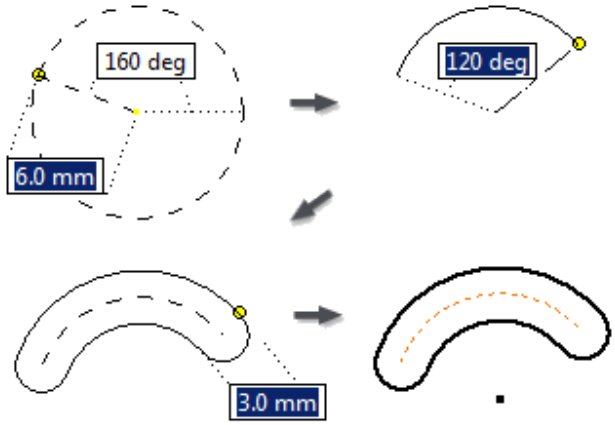
- **Center Point Slot** command creates a linear slot defined by the slot center, orientation and length of the slot centerline, and by slot width.



- **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.



18) Trim and Extend Control

Trim and Extend Command can be used in different way within Autodesk Inventor sketching environment. Some are listed below

Ctrl Key: Selects Termination Lines for Trim or Extend Command

Shift Key: Holding the Shift Key toggles between Trim and Extend Command

Hold Left Mouse Button: When a user continuously holds the left mouse button the **Dynamic Trim or Extend** is enabled until the mouse button is released.

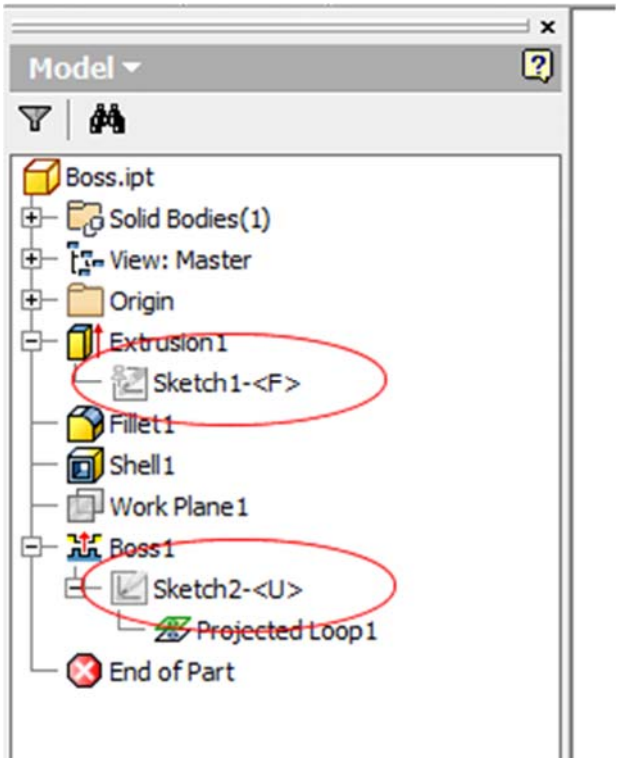
Right Click: Command option to switch between Trim, Extend or Split

19) Fully Constrained Sketch

As a professional user, you know that all your sketches should be fully constrained, but how many times have you closed a sketch only to realize that you didn't check the status bar. Now all you need to do is look in the browser.

Sketch Status from [TEDCF Publishing](http://www.trainingtutorial.com/Sketch_Status_2014.htm) is a simple but powerful add-in that gives you more control over your sketches.

Open a part or sheet metal part and all the sketches in the file are automatically renamed to show their constraint status.



Sketch Status

In the example above Sketch1 is fully constrained, and Sketch2 is under-constrained.

If you know the importance of fully constraining all your sketches, you know how important this add-in is. It's free, so there's no reason you shouldn't try it.

http://www.trainingtutorial.com/Sketch_Status_2014.htm

20) I CHECK IT ADD-IN

There are several add-ins available that check to ensure that users have not forgotten to fill out required iProperties. One such add-in is i CHECK IT from Tata Technologies. i CHECK IT goes far beyond just checking iProperties, however; it will also check to ensure that the file naming convention has been followed, that the first sketch of a model is constrained to the origin, that all sketches are fully constrained, that only approved dimension styles are used, and much more. There are over 100 checks that can be defined. To know more download from Apps Store

This app checks Sketching / Drawing Environment Files

http://apps.exchange.autodesk.com/INVNTOR/Home/Index

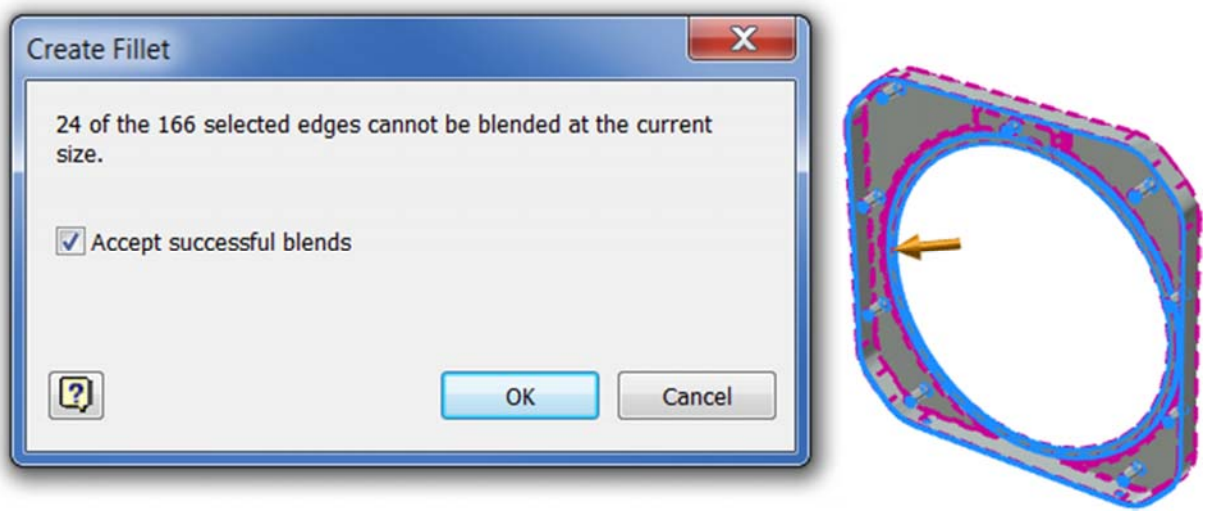
You get a > 30-day free trial version.

Part Modeling:

21) 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.



If you attempt to place multiple fillets at once, you might be alerted to the fact that some of the edges could not be blended successfully using the current radius size, in which case you can choose to place the successful fillets and skip the others.

One of the most common issues with edge fillets arises when you select multiple edges that converge

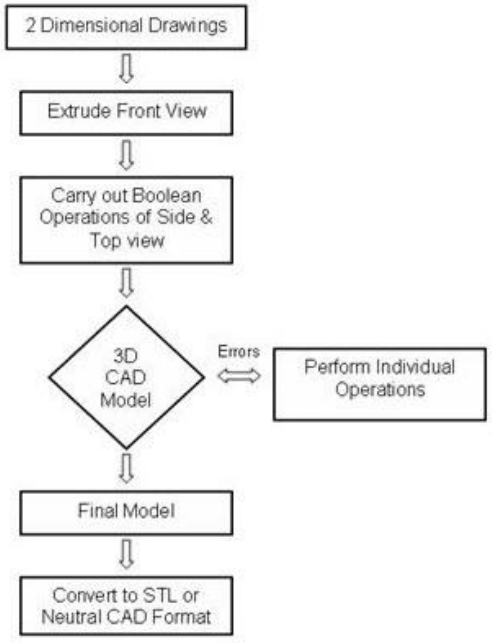
On a single corner vertex. Unfortunately, when this happens, the error message might erroneously indicate an issue with the fillet size. However, you can generally resolve this issue by removing one or more of the competing edges from the selection, applying the fillet feature, and then applying another fillet feature using the previously removed edge(s). Keep this in mind as you place edge fillets, and remember that just because you can place multiple sizes of edge fillets all in one feature doesn't mean it's always the best solution.

22) Changing the Units of a Part File

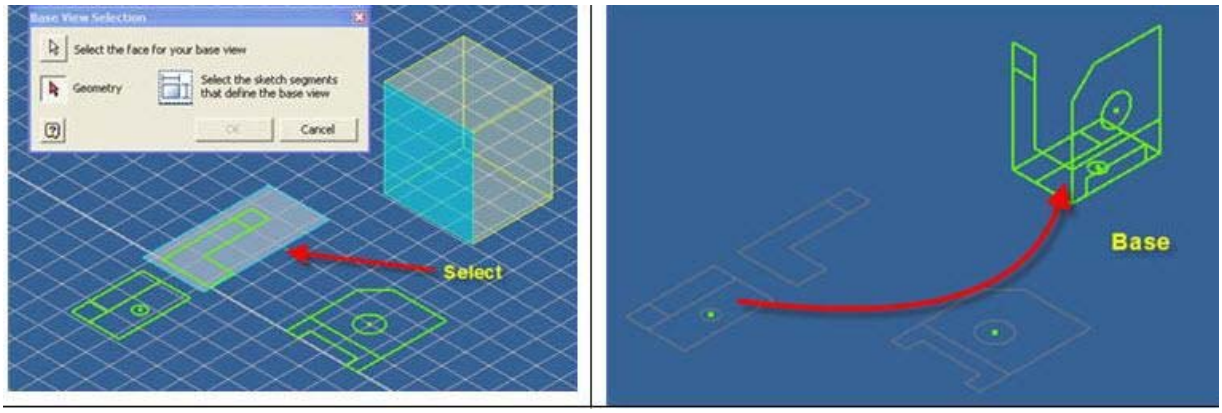
If you start a part file using the wrong template (inches instead of millimeters or millimeters instead of inches), you can change the base units of the file by clicking the Document Settings icon on the Tools tab of the Ribbon menu and selecting the Units tab. Changing the base units will automatically convert parameters but will not override parameter inputs. For instance, if you enter a value of 3 inches for a dimension and then change the units of the file to millimeters, the dimension will show 76.2 mm; however, when you edit the dimension, you will see the original value of 3 inches.

23) 2D to 3D App in Autodesk Inventor

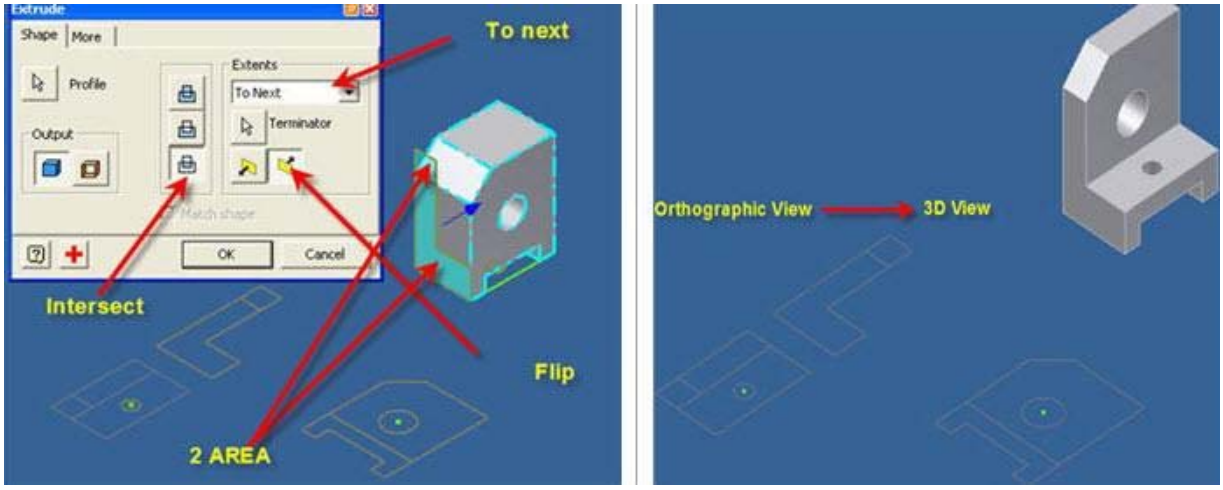
2D to 3D App was first made available from Autodesk Labs and provides a great help in converting legacy 2D AutoCAD files into native Inventor 3D models.



Flow chart for 2D to 3D conversion



Conversion from 2D to 3D Stage 1

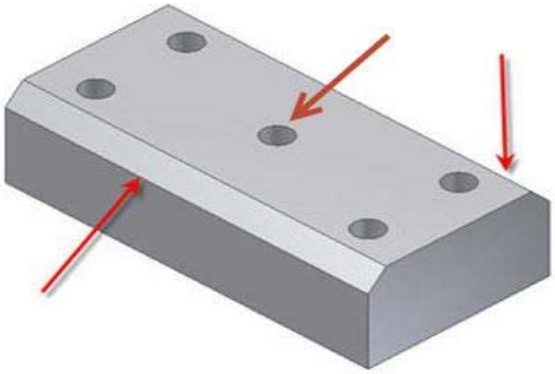


Conversion from 2D to 3D Stage 2

24) Feature Recognition

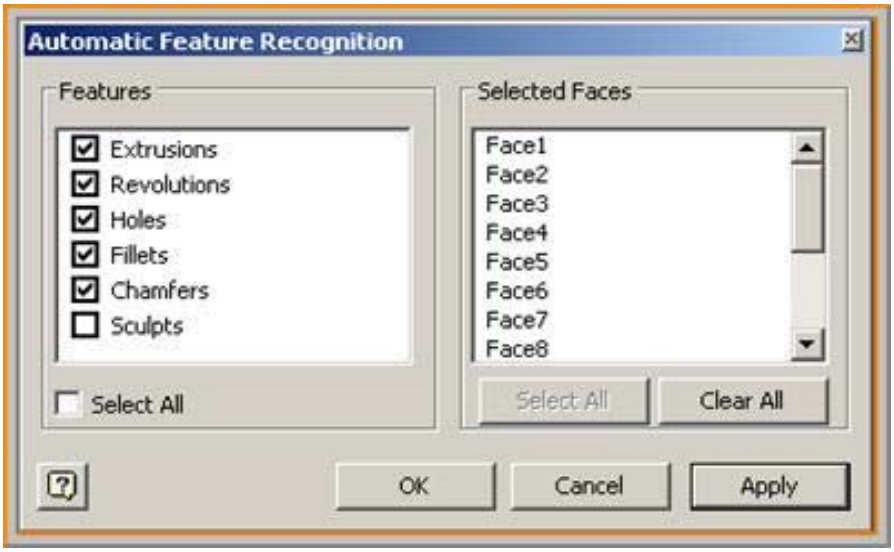
Feature Recognition is a tool that converts neutral 3D CAD models, such as STEP, SAT, or IGES solids, into full-featured Autodesk® Inventor® models. Feature mapping could be executed automatically or interactively as needed to maintain design intent.

For more details visit the link below to download the Feature Recognition App – **Click Here**

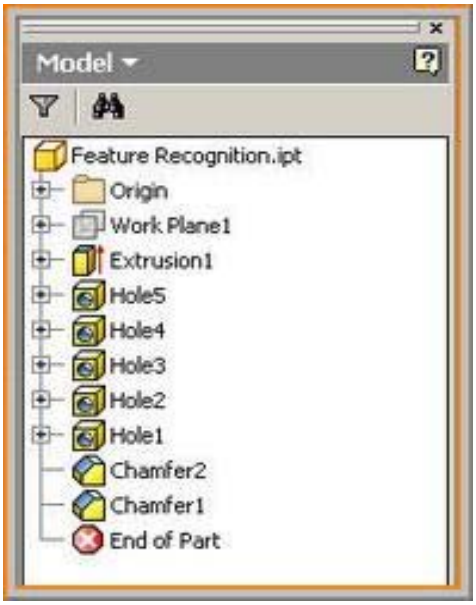


SAT Model with Chamfer and 5 Holes

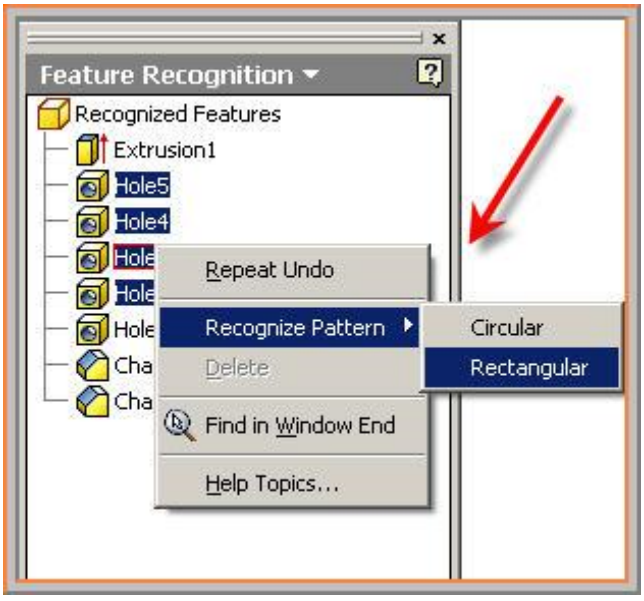
Recognize Features



Types of Features to Recognize



Recognized Features

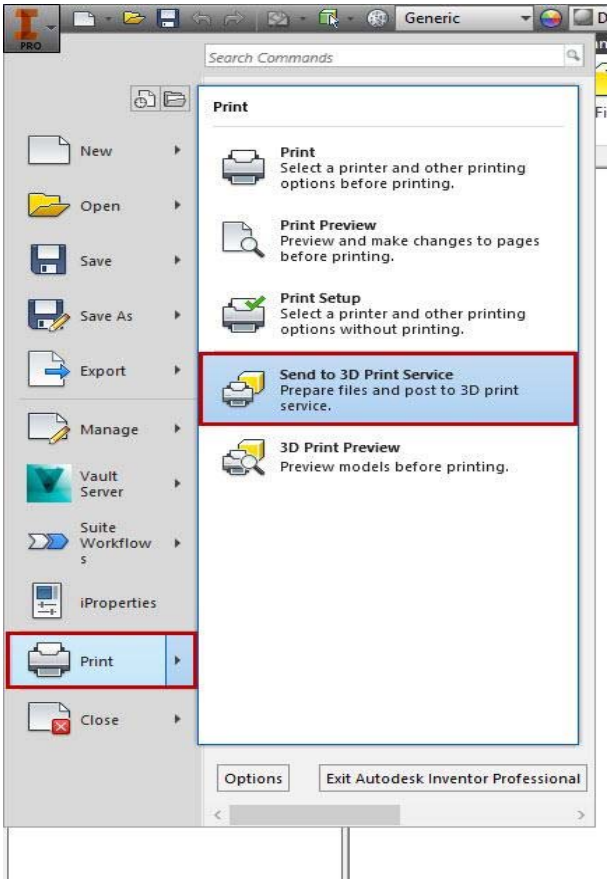


Combine holes to make rectangular pattern

25) Printing your 3D Models from Autodesk Inventor 2014.

Autodesk Inventor now has option to print your 3D CAD models by converting it into STL Files and later you can upload them into Autodesk Recommended 3D printing companies.

- Set up an account
- Upload your STL file
- Request a cost estimate
- Place your order
- Pay for your order



The orders vary from $15 - $25 per cubic inch

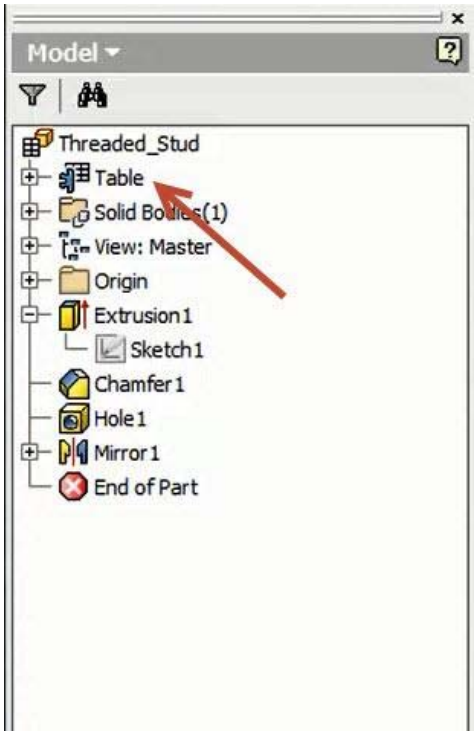
Note: Additional tips related to 3D printing are provided here -

http://images.autodesk.com/adsk/files/tips_for_optimizing_your_model.pdf

http://usa.autodesk.com/adsk/servlet/pc/item?id=22025960&siteID=123112

26) How to convert iPart to a Standard Part

You can convert an iPart to a standard part file by right-clicking the table in the Model browser and choosing Delete. The part will assume the active members' feature values and states.

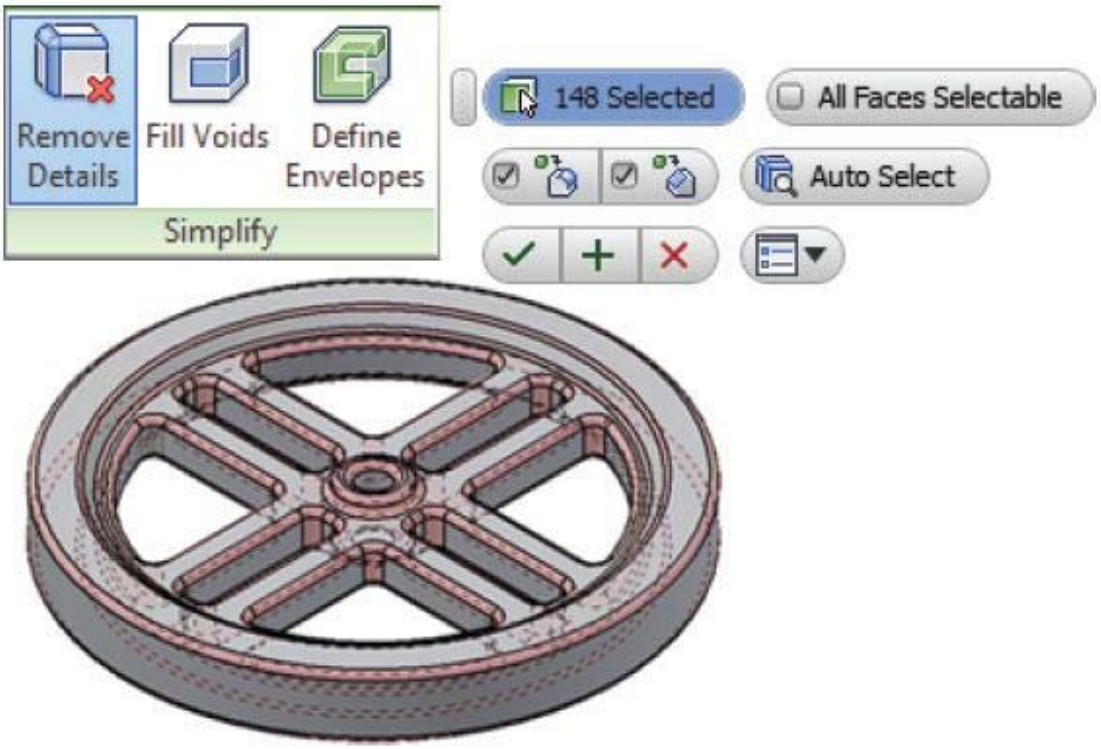


When Table is deleted will convert iPart into an Active Part

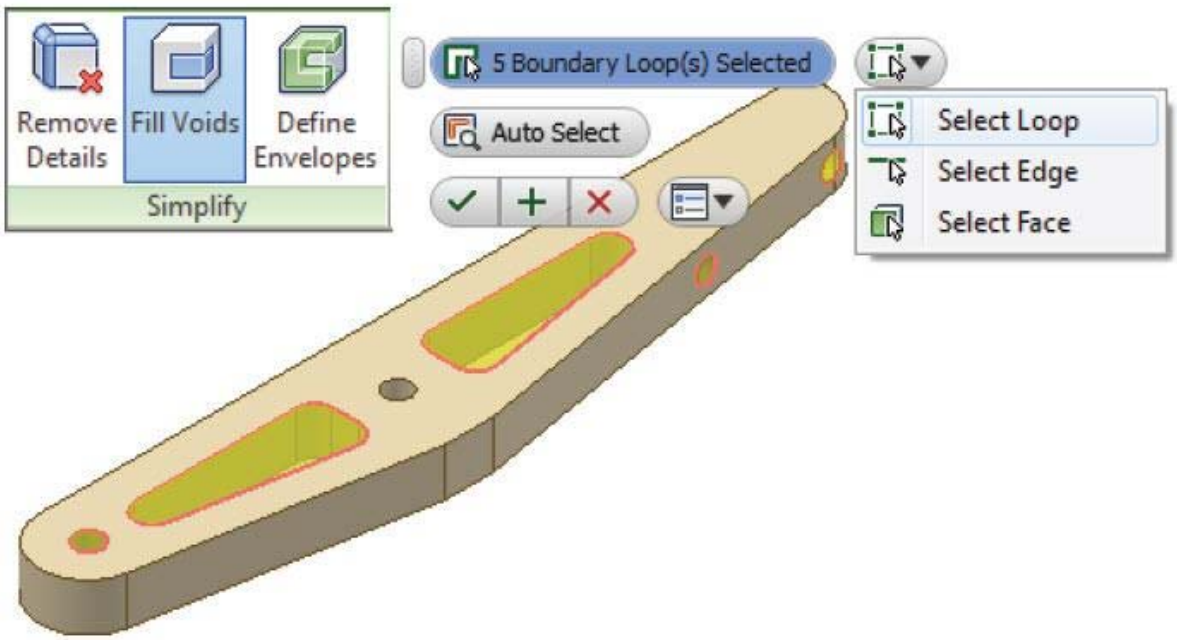
27) Part Model Simplification: (Building Information Modeling – Ready)

In addition to the tools available to simplify assembly models, you can also use the BIM Feature Recognition and Simplify tools in the parts environment to reduce details that are not needed for BIM models, and you can use the Recognize Revit Features tools to make your models friendlier to Autodesk Revit

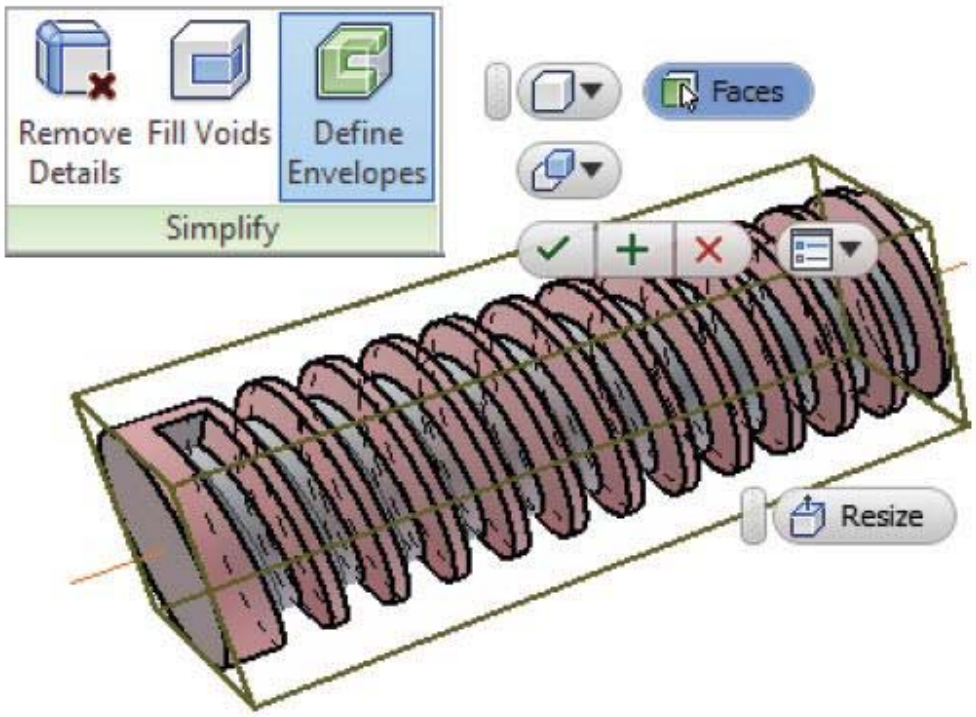
Note: One of the issues with handling part models created in Inventor to the BIM user is that the models often contain cosmetic or manufacturing features that are important in the mechanical design, but are not required for the BIM layout and planning application. For example, if you create a model of a machined part in Inventor, you will likely need to include and specify rounded edges, holes, and other features as part of the mechanical design. However, these features are not important to the BIM user and can slow down the BIM application. So to help with this, Inventor includes part simplification tools to remove detail and voids, as well as reduce the number of faces and edges in a part model.



Remove Details Options



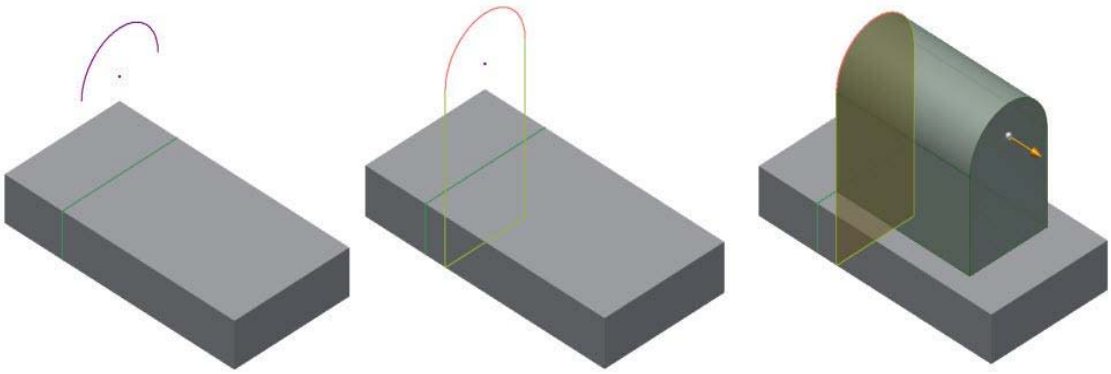
Fill Voids



Define Envelopes

28) <u>Extruding Solids from Open Profiles</u>

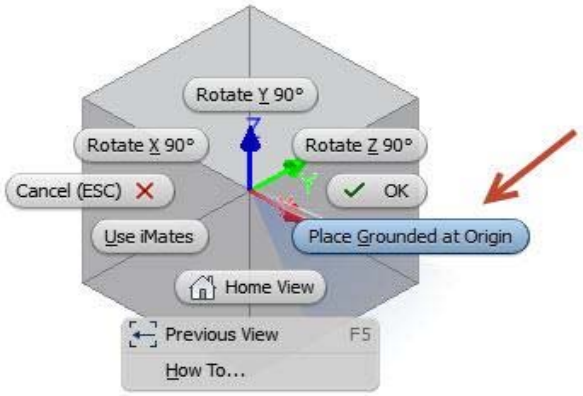
In addition to extruding surfaces from open profiles, you can extrude solids, provided sufficient geometry is present to allow the open profile to solve correctly. This technique employs the Match Shape option and is the default solution when an open profile is selected in the Extrude tool while the solid output is selected



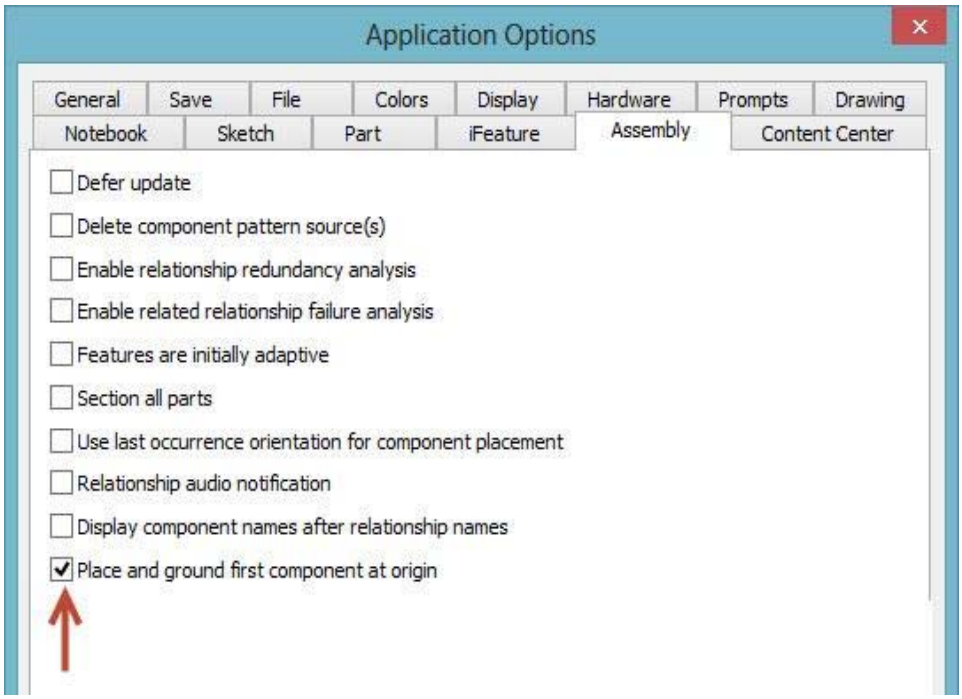
Assembly Modeling

29) Grounded Part

When we start Assembly Modeling and insert the first part they are not grounded in Inventor 2014 Version. So care must be taken to ensure you select grounded part from marking menu.

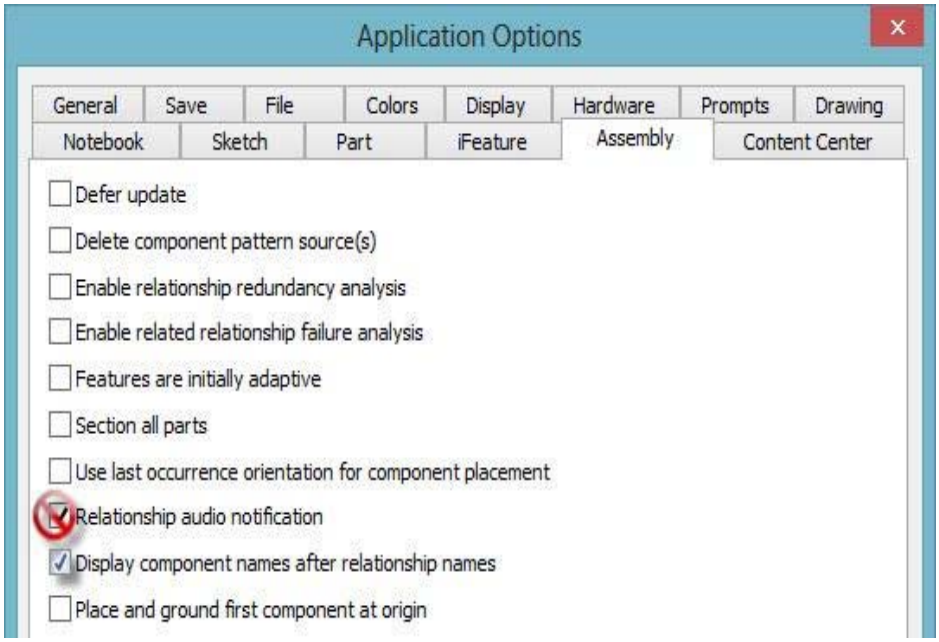


Another option is to input a permanent solutions by selecting from Application Options



30) Constraint Audio Notification

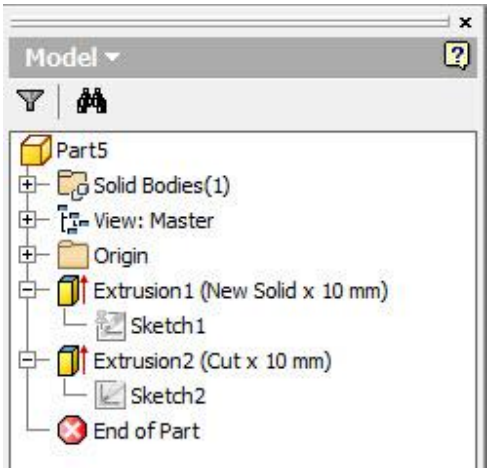
When two components / parts are assembled in the assembly environment with default settings, we hear a sound (Tock) and this at times is annoying to users or when you do a corporate training / presentation or trying to explain the sequence of assembly for a vendor. This can be overcome by switching off the Relation Audio notification and enabling Display Component names after relationship names



Note: By enabling the Display Component names after relationship names Autodesk Inventor displays the component instance names with the relationship.
When the check box is cleared, Autodesk Inventor does not display the component instance names with the relationship. For example, if constraint 12 is a Mate constraint between rod:1 and cylinder:1, the browser displays:

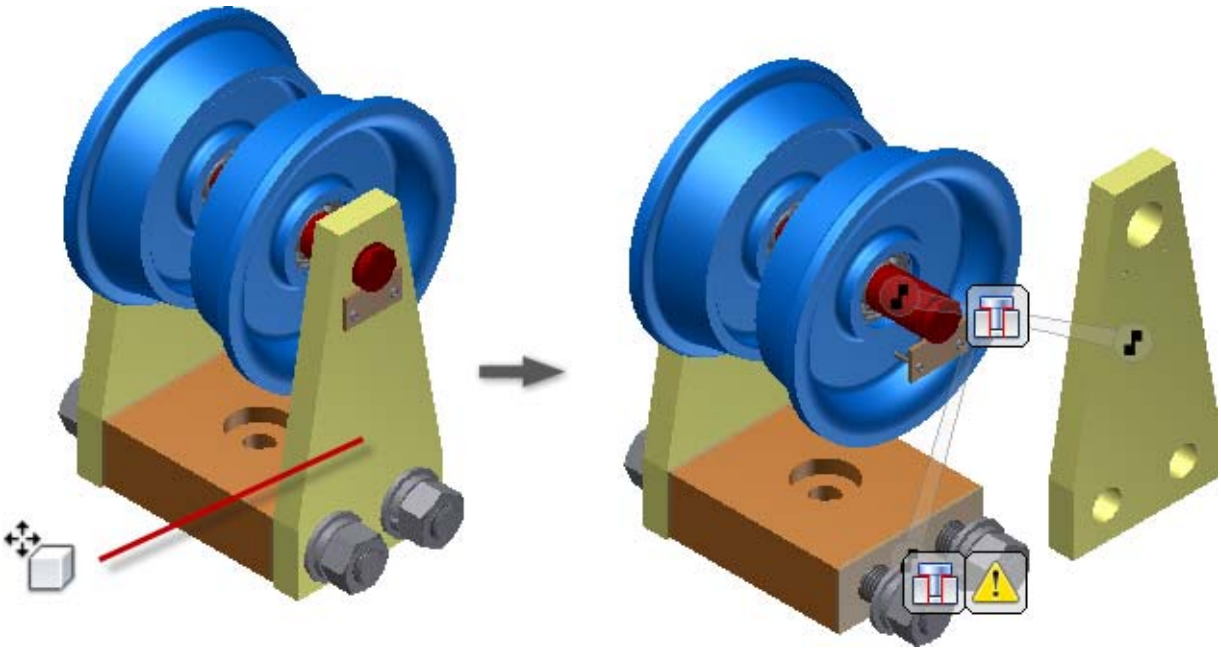
 Mate:12(rod:1, cylinder:1) When option enabled

 Mate:12 When option not enabled



31) Free Move (Assembly)

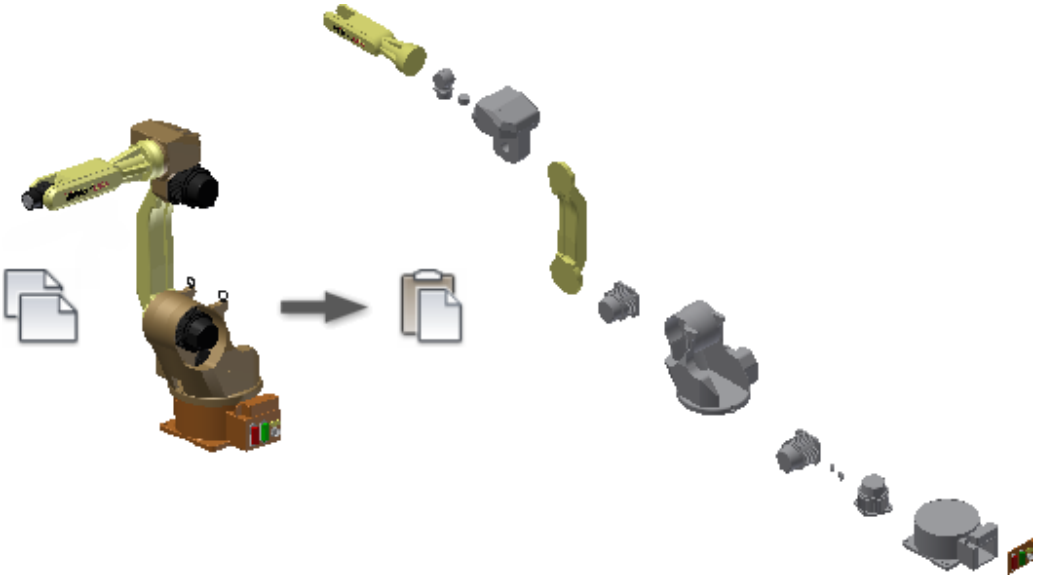
With the Free Move command, display relationships as an elastic band, and visualize and manage relationships in a new way. Click a relationship icon, and then, to suppress, unsuppress, or delete relationships, use the context menu.



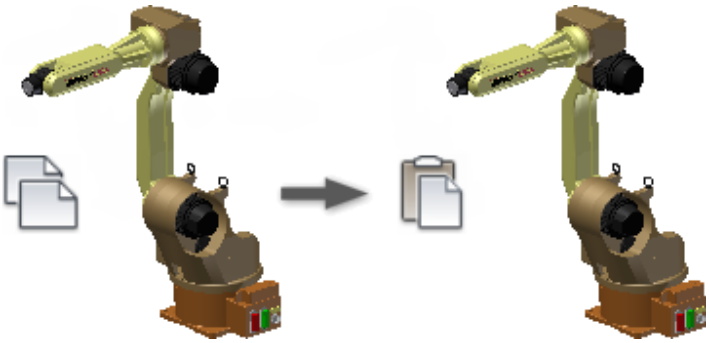
32) Assembly copy paste

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.



33) Joints in Autodesk Inventor 2014

The Joint command is a simple way to position components and describe motion. Creating a joint 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 joint type, or you can specify a joint type.

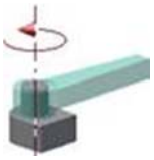
The following describes the joint types and how they operate while positioning a component:


Rigid

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


Rotational

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




Slider

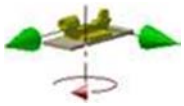
Specify one translational DOF. A slide block moving along a track is an example of a slider joint.


Cylindrical

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


Planar

Specify two translational and one rotational DOF perpendicular to the linear. To place a component on a planar face, use this joint 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 joint.

Note:

- With the addition of the joint command, both constraints and joints 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 joints with constraints to position components and describe motion.

34) 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.

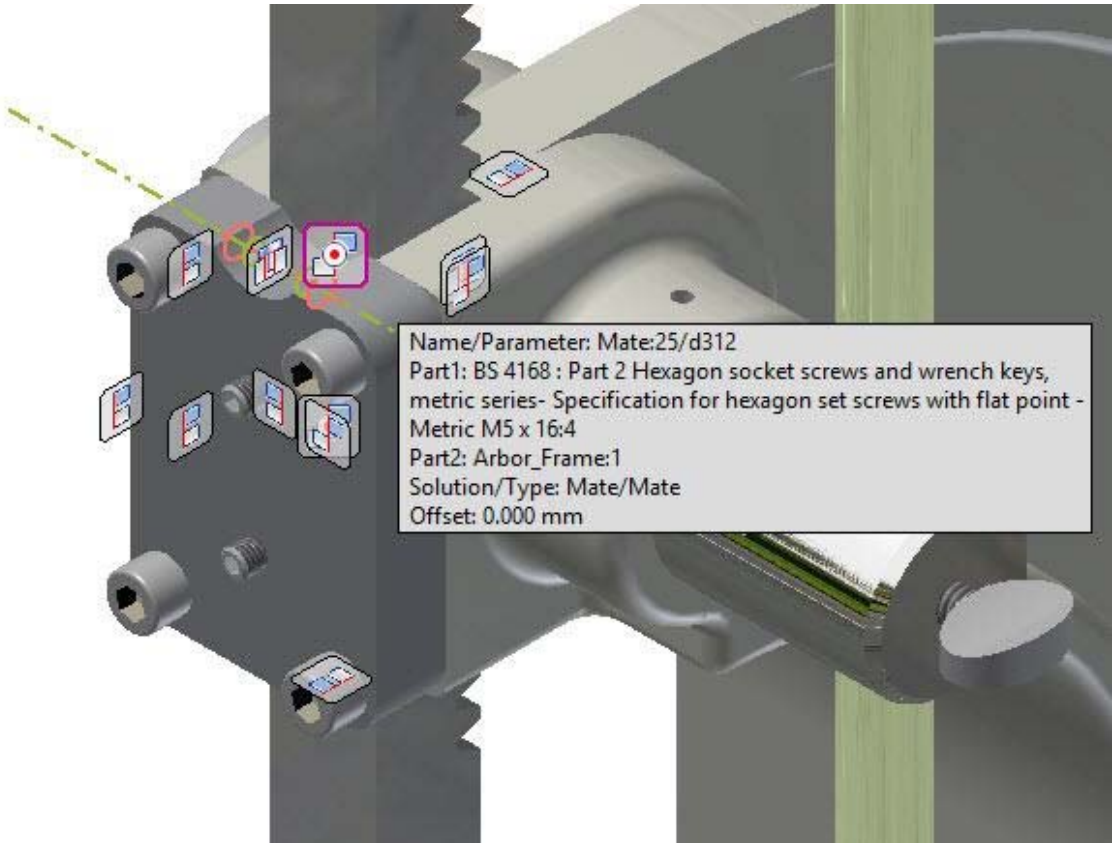
Express Mode OFF in the above case

Express Mode ON in the above case

35) Show and Hide Relationship Tools

In addition to locating constraints in the browser, you can use the Show Relationships tool to select a component and locate all of the assembly relationships that involve it. Once the relationship glyphs are shown, you can right-click any of them and choose from a number of options, such as Edit, Delete, Suppress, and so on. You can then use the Hide Relationships tool to turn off the display of the relationship glyphs in the graphics area.

If any assembly relationships contain errors, you can use the Show Sick Relationships tool to find them quickly.



Show and Hide Relationship Tools

36) Place Assembly Constraints Using Alt-Drag

Rather than using the Place Constraint dialog box, you can press and hold the Alt key and then drag a component into position. Constraints are inferred based on the type of geometry selected. The constraint is previewed in the graphics area as you drag over the components involved. To set a specific constraint, release the Alt key and enter one of the following shortcut keys. You can press the spacebar to flip the constraint solution, from Mate to Flush, for example.

- M or 1 changes to a Mate constraint.
- A or 2 changes to an Angle constraint.
- T or 3 changes to a Tangent constraint.
- I or 4 changes to an Insert constraint.
- R or 5 changes to a Rotation motion constraint.
- S or 6 changes to a Translation motion (slide) constraint.
- X or 8 changes to a Transitional constraint.

To see the Alt-drag method in action, select the Get Started tab, and click the Show Me Animations button. Click the Assemblies – Constraints link in the list and then choose Alt-drag Shortcut Animation from the next list.

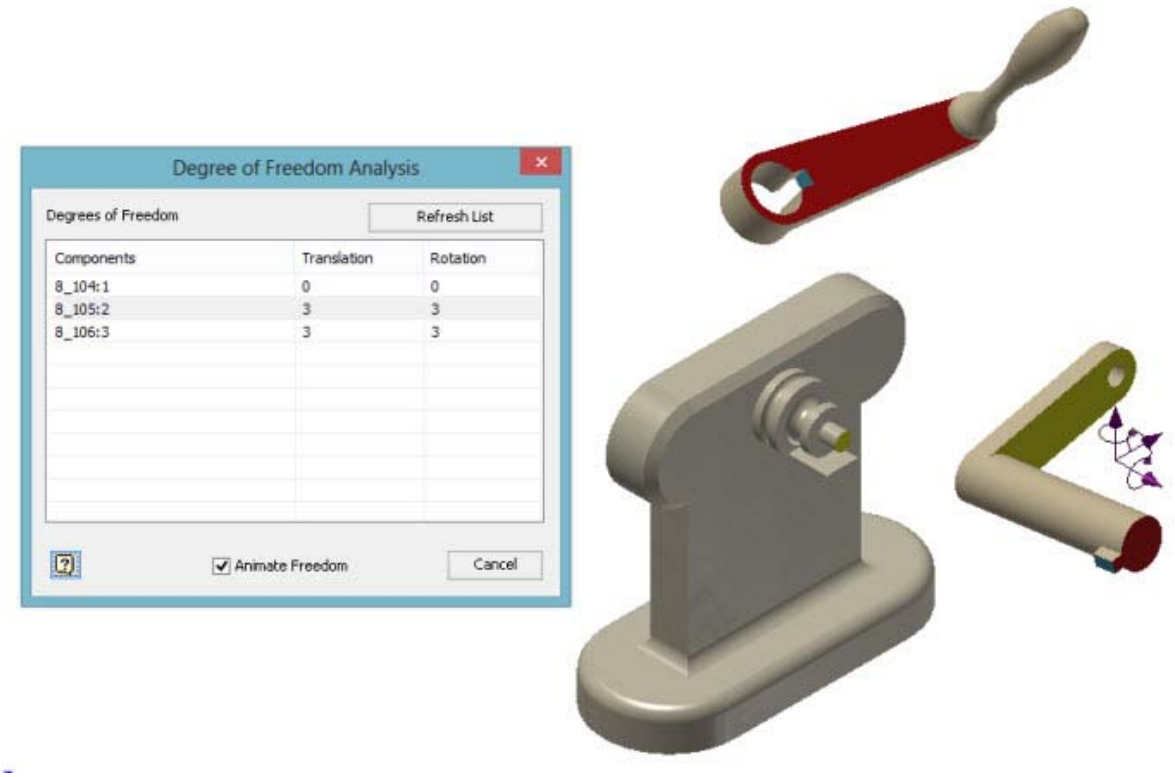
37) Degree of Freedom Analysis

Degree of Freedom Analysis can be accessed from

Ribbon: Assemble tab ➤ Productivity panel ➤ Degree of Freedom Analysis 

This command displays a dialog box that lists the number of translational and rotational degrees of freedom for all the occurrences in the active assembly.

A dialog box displays a list of all first-level components in the active assembly with a count of remaining degrees of translational and rotational freedom.



Degree of Freedom Analysis

Productivity

38) Creating Mixed-Unit Formulas

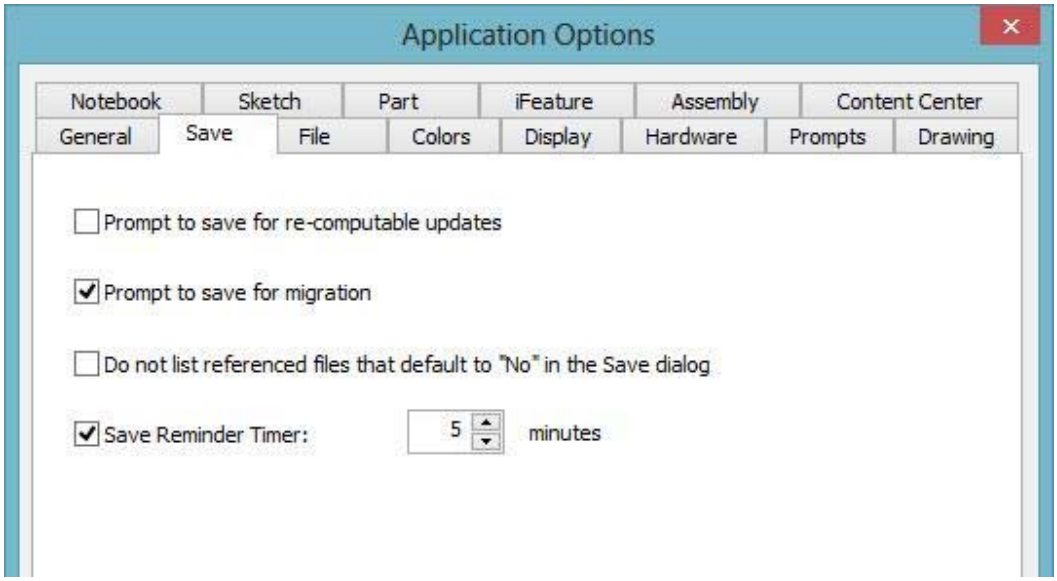
You can mix any acceptable units in the same parameter formula and allow Inventor to do the conversion. For instance, entering the following into a dimension is perfectly acceptable:

3.25 ft – 1 m + 3 cm + 0.125 in = 23.775 mm

This would return a value of 23.775 mm in a millimeter part.

39) Adjusting the Save Options

Inventor does not have an automatic save function, but instead has a save reminder utility that allows you to save by just clicking within the bubble to launch a standard save operation. To adjust the save timer settings, select the Application Options button and then select the Save tab.



Save Time Reminder

40) Migrating Templates to Autodesk Inventor 2014 Using Task Scheduler

Before migrating any templates, make sure you've backed everything up! I've never had a problems with migration, but you only have to be wrong once!

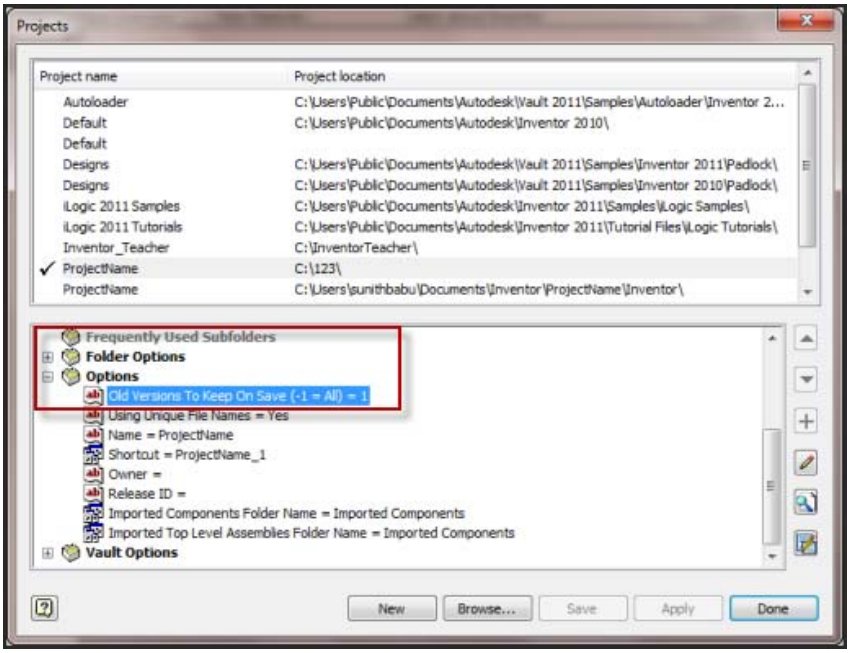
To access Task Scheduler, go to **Windows Start>All Programs>Autodesk Inventor 2014>Tools**. Task Scheduler will be in there.

41) Project File Version

Each time you save an Autodesk Inventor file, the previous version of the file is stored in an *OldVersions* folder under the folder containing the file.

You can specify how many versions of each file are stored in the *OldVersions* folder.

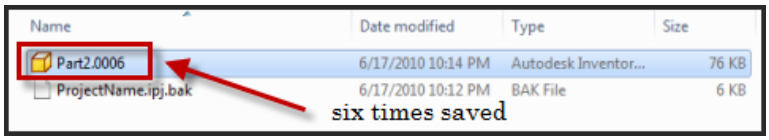
1. Expand *Options*.
2. Select *Old Versions To Keep On Save*.
3. Click the *Edit selected item* button (on the right-hand portion of the dialog box).
4. Enter the number of versions to keep.



Once the specified number of saved versions is reached, subsequent saves eliminate the oldest version.

Old versions are formatted as follows:

- First save of existing file = file name.*0001*.extension.
- Second save of existing file = file name.*0002*.extension.
- Subsequent saves are named in a similar manner.

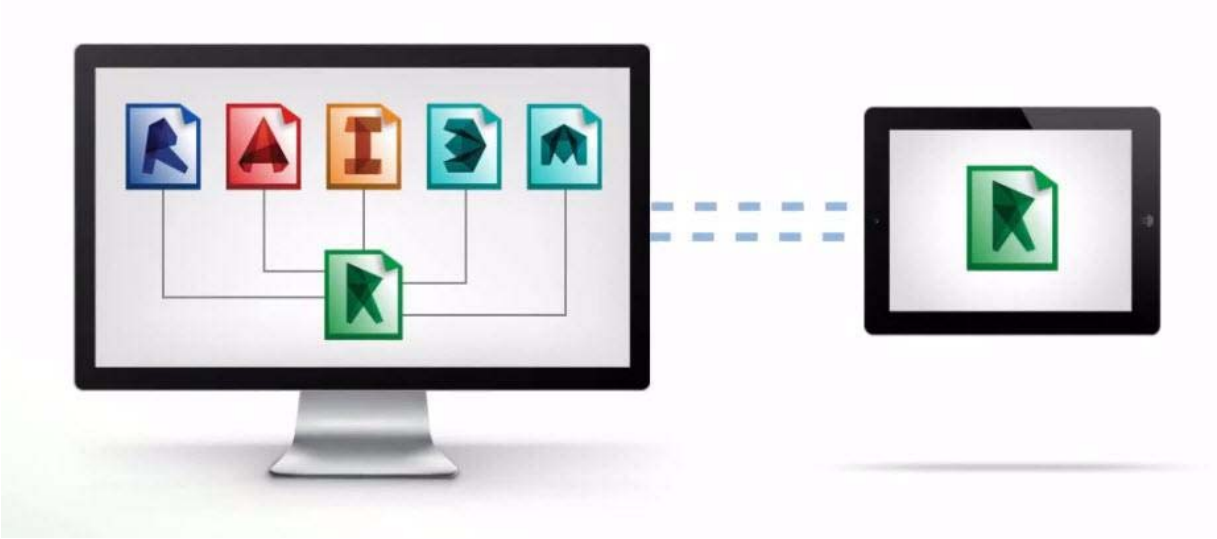


You can open an older version of a file as a read-only file. Alternatively, you can restore an old version as the current version of the file. Before the restoration of an old version, the file is saved as the most recent version in the *OldVersions* folder.

42) Autodesk Remote

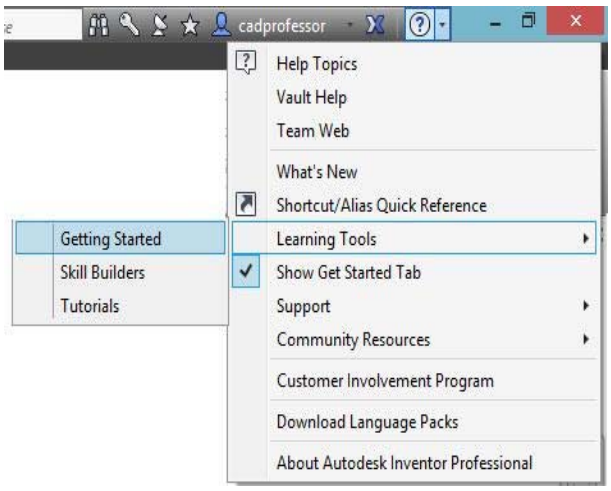
Autodesk Remote helps you connect your desktop / laptop inventor software - service to drive Autodesk software installed on your primary computer from a remote PC or iPad® to access native

design data. The iPad app, compatible only with Autodesk® Inventor® software, enables you to work anywhere you can access the Internet. This service is available with purchase of Autodesk® Subscription for select Autodesk products.



Watch the video to learn more about Autodesk Remote - [Click Here]

43) Autodesk Inventor - Getting Started / First Time learner.



Learning map helps you build and track skills

A First time inventor learner can get a complete learning experience by clicking on Access Help Icon > Learning Tools > Getting Started

This provides a new and interesting learning portal within Autodesk.com site, which is structured and easy to understand and learn.

Simplified Getting Started

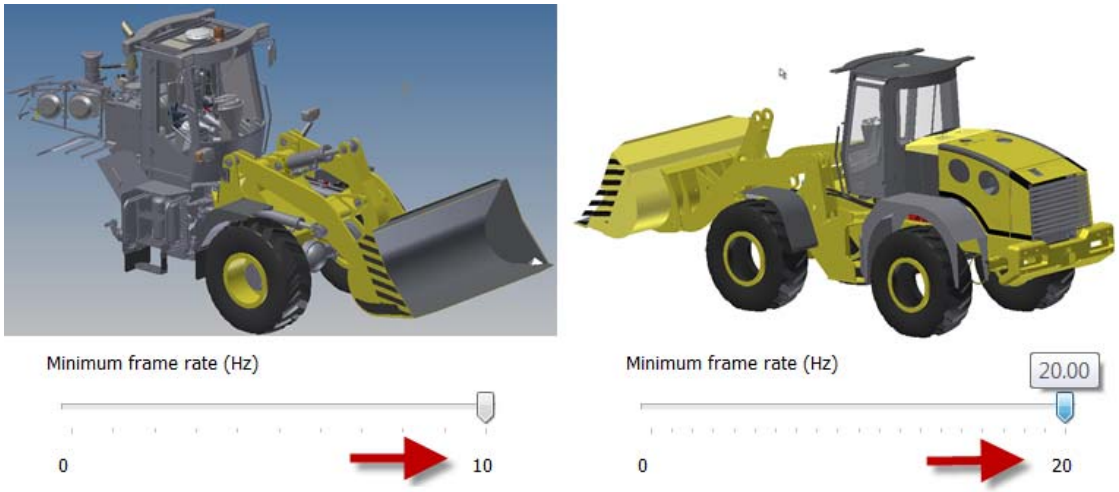
Link - http://akn.autodesk.com/maps/ENU/99994

44) 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.

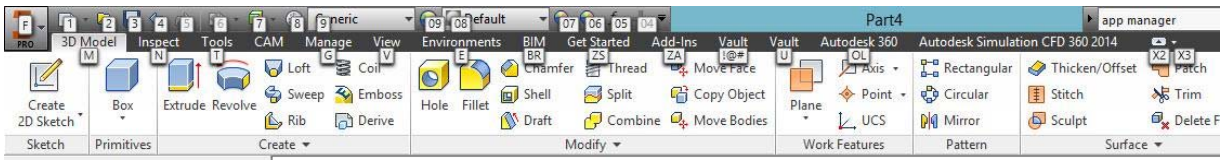


45) Key Tips using ALT key

Keytips are shortcut keys for performing tasks without using your mouse. Using Keytips, you can navigate in the Application Menu and the ribbon using only the keyboard. Use the keyboard arrows to navigate to commands on the ribbon and Application menu.

Press ALT or F10 to display shortcut keys for common tools in the application window. Keytips appear as underlined characters to indicate which key or combination of keys to press to activate a command.

When you select a keytip, more keytips display for that tool.

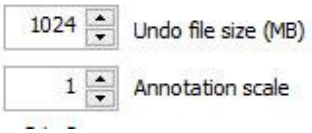


46) Undo File Size

The Undo File Size is very essential when a user works with large assembly files. However, it is essential to reduce the Undo File Size from 1024 MB to 256 MB so that the user gets the additional RAM to work with Inventor 2014.

The Undo File Size can be accessed from Tools > Application Options > General Tab

Autodesk recommends adjusting this in 4 MB increments.



Undo File Size

47) iPart Factories in Assemblies

You should be aware that Inventor does not allow an iPart factory file to be placed into an assembly.

If you attempt to do so, only a member of the factory will be placed instead. However, keep in mind that if you create a regular part file, place it into an assembly, and then turn it into an iPart factory, changing the factory table does not place a member file but simply updates the factory. Using factory files in assemblies in this manner is not the intended workflow for iParts.

48) iFeatures Are Powerful Tools

These tools allow you to quickly create standard features in your models. Examples include o-ring grooves, louvers, bosses, ribs, electrical connector punches, patterns of holes, and an infi nite number of other features. Another major advantage of iFeatures is that they enforce standards. Since the iFeature can be designed to allow the user to select predefined sizes only, the possibility of error is greatly reduced. Take a few moments to examine your designs, and you'll likely see many opportunities for iFeatures.

49) RAM Requirements for Autodesk Inventor 2014

When it comes to RAM, the more your system has, the better it will handle large and complex assemblies in Inventor. You can use the number of unique parts in your assemblies (unique parts as opposed to multiple instances of the same part) as a general rule of thumb when determining how much RAM to consider. Here is a list of recommendations based on unique part count:

◆ More than 5,000 unique parts: 6 GB of RAM

◆ More than 10,000 unique parts: 12 GB of RAM

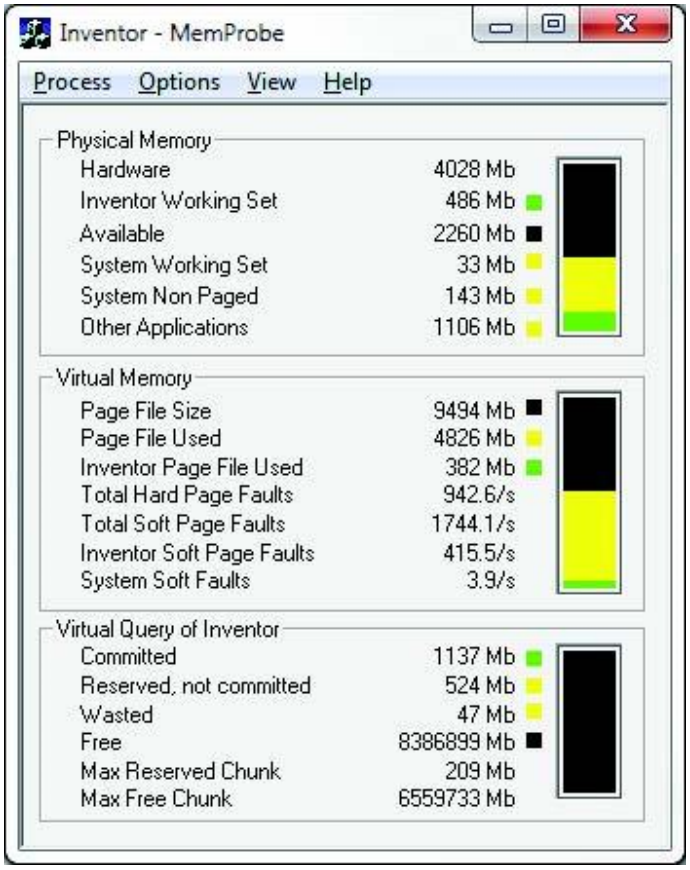
◆ More than 15,000 unique parts: 18 GB of RAM

◆ More than 20,000 unique parts: 24 GB of RAM

50) Memory Probe

Included in your install of Inventor is a utility designed to monitor memory use for your system. The MemProbe utility, shown in Figure, looks at the Inventor process and displays its use of physical and virtual memory. It can often be useful in troubleshooting issues of capacity and slow performance. You can find this tool at the following location. You might find it useful to create a desktop shortcut to it, if you find yourself using it often:

C:\Program Files\Autodesk\Inventor 2014\Bin\memprobe.exe



51) Participate in CIP and CER

The Customer Involvement Program (CIP) aids in collecting information about your use of the Inventor software. Customer error reporting (CER) aids in sending information to Autodesk when the software program closes unexpectedly.

CIP Details:

To guide the direction of the Autodesk design software in the future, your use of the Inventor software is forwarded to Autodesk if you participate in the CIP. You can access this feature by selecting Help ➢ Customer Involvement Program. In the Customer Involvement Program dialog box, you can select a level of participation and then click the OK button. The following information might be collected:

◆ Autodesk product version and name

◆ Inventor commands and time spent

◆ Error conditions (catastrophic and nonfatal)

◆ Information such as system configuration, IP address,

CIP Agreement

CER Details:

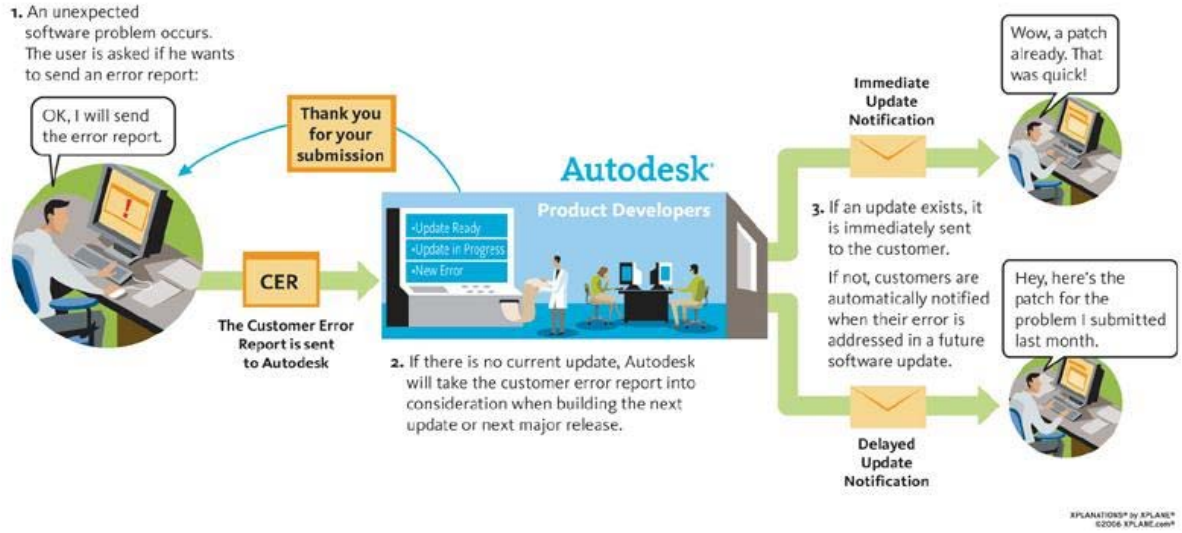
Customer error reporting is a process by which Inventor users can report crashes to Autodesk. A software crash happens when the software program closes unexpectedly. When you have an unexpected error, Inventor shows a dialog box, and you can choose to send the error to Autodesk. CER records the subset of the code that was in use before the crash. The CER report collects a variety of information, such as the following:

◆ Operating system and graphics driver name, version, and configuration

◆ Autodesk software name and version

◆ List of recently used Autodesk commands

◆ Lines in the code where the crash happened



CER Workflow

52) Autodesk 360

With your Autodesk® 360 account you can save files to the cloud. View and edit them using any web browser or mobile device.

Storage

Receive 5 GB of free storage instantly simply by creating an account. Additional storage is available with Autodesk® Subscription.

Viewing

View and edit 2D and 3D design files through a web browser using Autodesk 360 or via a mobile device using Autodesk® 360 or AutoCAD® WS mobile apps.

Collaboration and Sharing

Share files with your team. Invite them to view, markup, share, and download designs directly in a web browser or mobile device. Follow user comments with the ongoing activity stream.

Rendering

Autodesk® 360 provides powerful cloud rendering capabilities that reduce time and costs by enabling users to produce compelling, photorealistic visualizations without tying up the desktop or requiring specialized rendering hardware.

Pixlr

Pixlr™ products are free* online and mobile image editing tools and utilities for creating, editing, and sharing images.

Recap Photo Trial

Create 3D models from photos in the cloud with Autodesk® ReCap™ Photo Trial.

Fusion 360

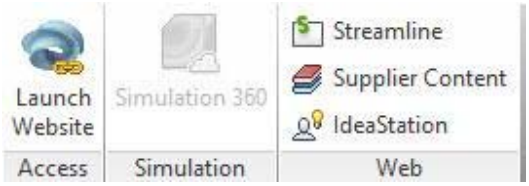
- Autodesk® Fusion 360™ is the next-generation 3D design tool that seamlessly combines industrial and mechanical design with collaboration in one easy-to-use and affordable solution.

Mockup 360

- Autodesk® Mockup 360™ is a real-time collaboration and digital design mockup tool that enables engineers and project managers in manufacturing environments to work with suppliers and customers in a simple, cloud-based environment.

Configurator 360

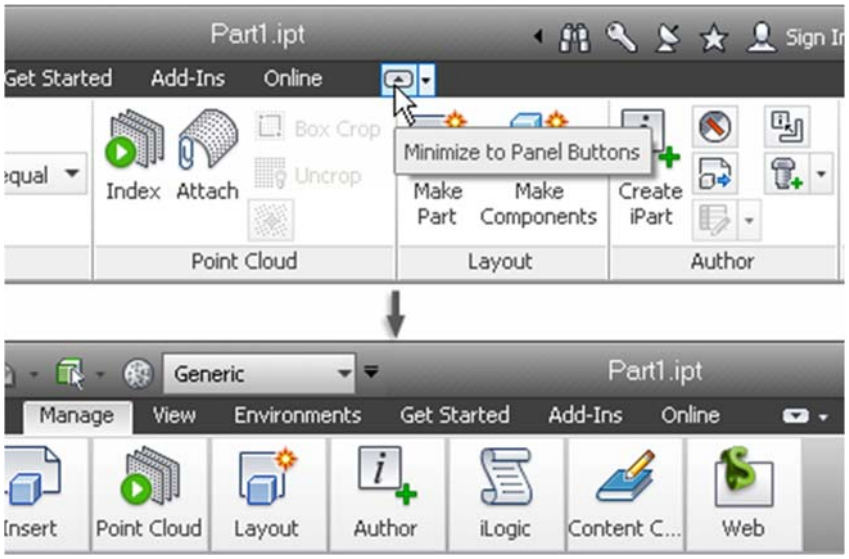
- Autodesk® Configurator 360™ is a cloud-based service that helps custom product manufacturers of all sizes deploy an online catalog of products with a built-in product configurator.



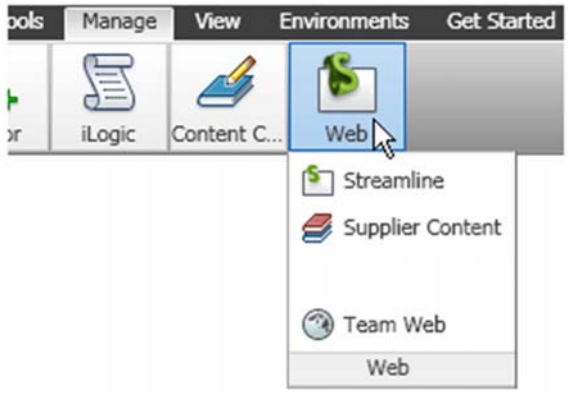
53) Increase Screen Space

The ribbon is designed to provide more available graphic space for your models.

Click the Minimize button once. The panels reduce to panel buttons.



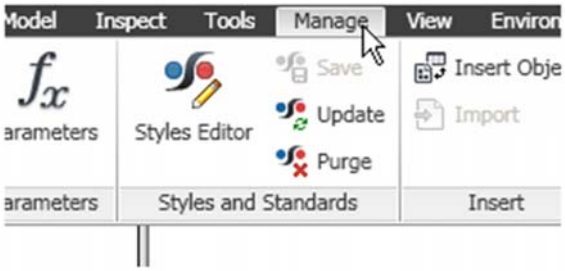
Pause the cursor over a panel to display the commands on that panel.



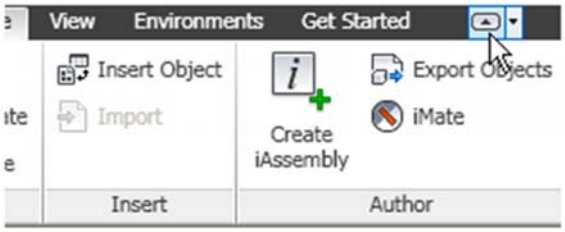
Click the Minimize button again to reduce to the tab and panel titles.

Click the Minimize button again to reduce to the tabs.

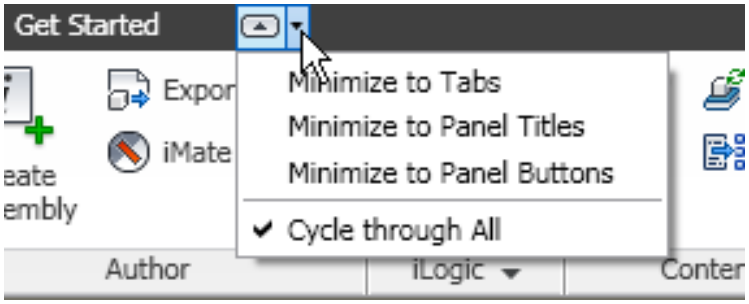
Click a tab title to reveal the tab.



Click the Minimize button again to restore the full ribbon.



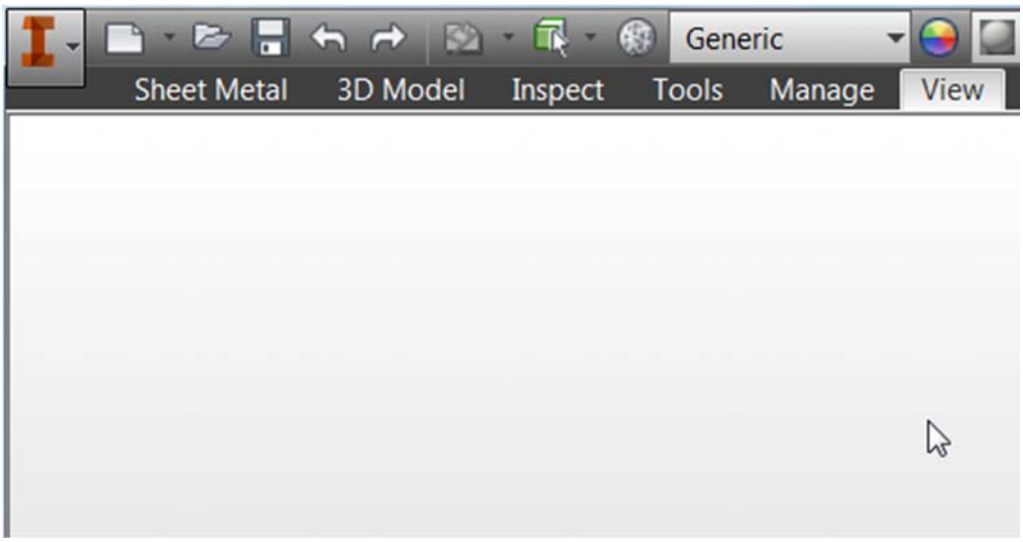
As an alternative to cycling through the ribbon states, select a state from the drop-down menu.



You can also use the Clean Screen command to maximize model space instantly.

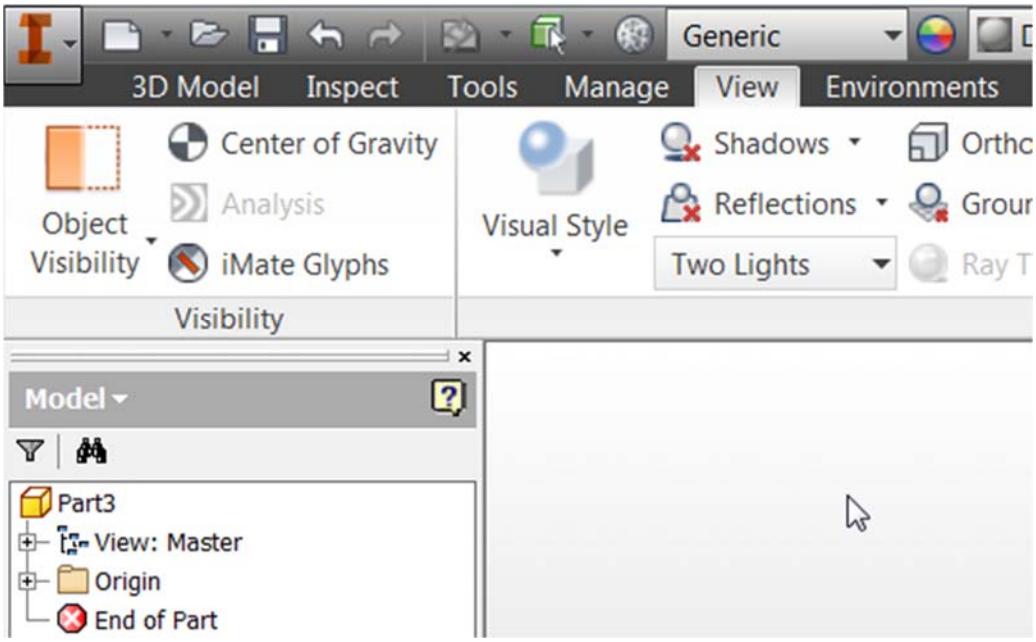
On the View tab, click Clean Screen.

This command hides the browser and maximizes the graphics area to the application window. Tab panels are minimized if they were in a maximized state. The command also maximizes the Autodesk Inventor window to your screen.



You can also use the keyboard shortcut: press Ctrl 0 (zero) to switch from Clean Screen display. The browser is restored and the tabs are also restored to whatever state they were in previously.

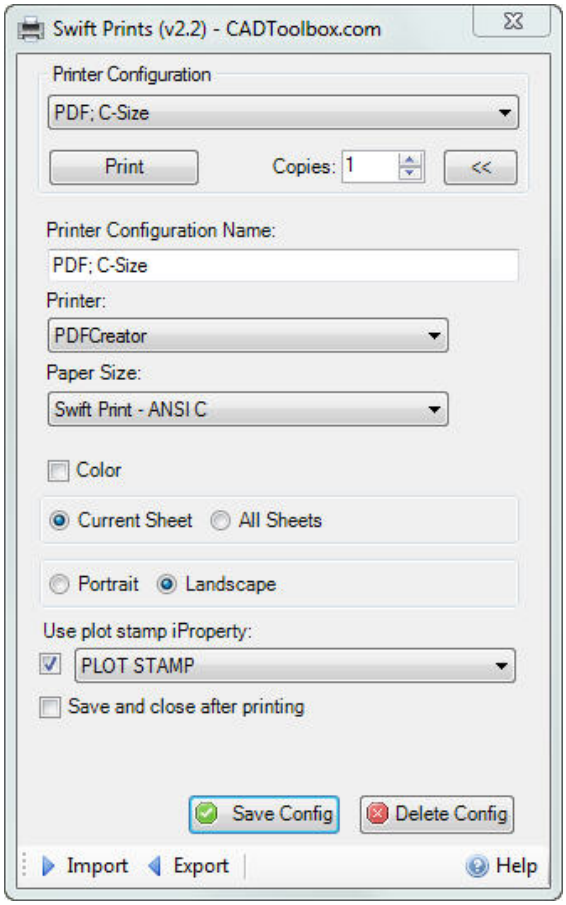
Click the Minimize button, as needed, to restore the tabs to their full display.



54) Swift Prints

This allows a user to create templates for common printer configurations that you can quickly access later for one click printing of Inventor drawings.

http://cadtoolbox.com/swift-prints/



55) End of Part

The End of Part Marker, or EOP sits at the end of Inventor's feature browser to let us know that there are no more features.

The End of Part Marker can be used as a productivity tool in its own right. Here are five great tips for making use of the EOP.

Five EOP Productivity Tips.

There are five ways to use the EOP:

- **You can step through the feature tree to see how the part was created.**
- **You can use the EOP to place features at the correct place in the tree.**
- **You can use the EOP to help repair broken features.**
- **You can delete all the features below the EOP.**
- **You can roll the EOP up to make the part file smaller.**

56) Autodesk Inventor IdeaStation

The Autodesk Inventor IdeaStation can be accessed within the Software from **Autodesk 360** tab ➤ **Web** panel ➤ **IdeaStation**.

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

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 Autodesk Inventor Idea Exchange, you can even track the status of ideas as they grow from initial concept to ultimate release!

Drawing

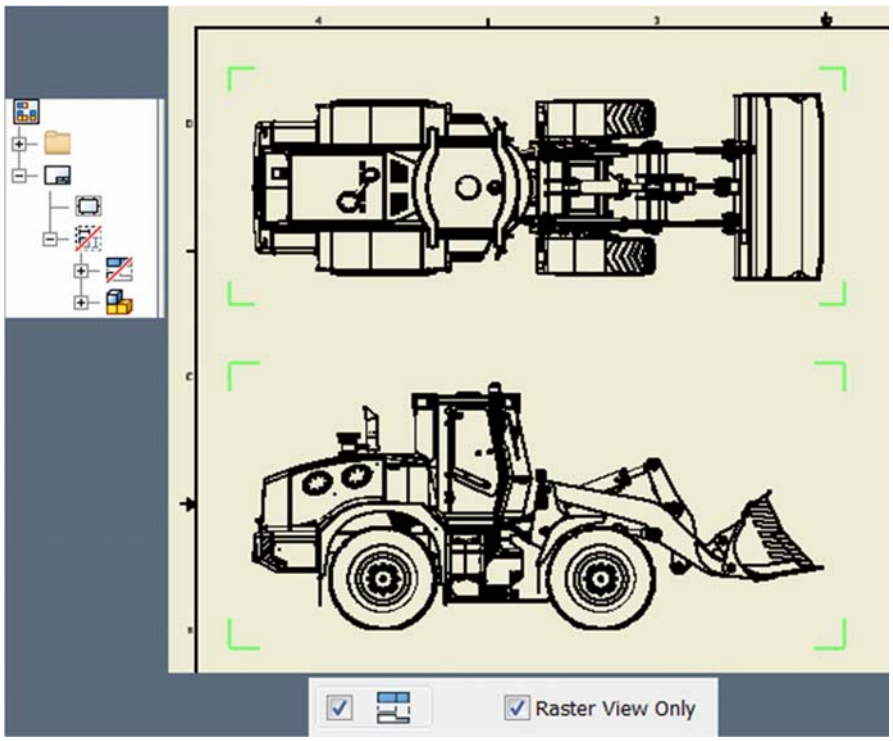
57) Raster views

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.



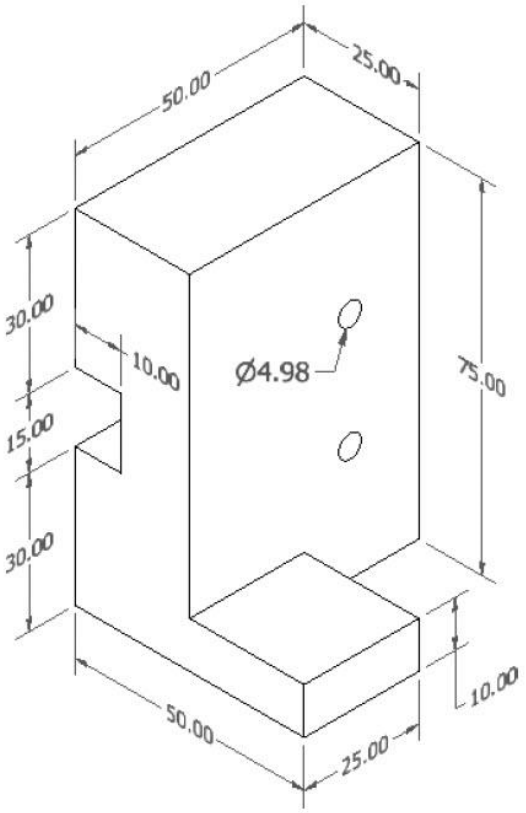
58) Import Hatch Patterns

You can load hatch patterns from an external PAT file to add custom hatch patterns to your drawings. To do this, edit a hatch and select other from the Pattern list, and then click Load. You can also do this in the Style and Standard Editor to add custom hatches to your company standard.

59) Dimensions in Isometric Views

Isometric dimensions are functionally identical to orthographic dimensions; however, they cannot be "moved." That is, they cannot be detached and reattached to different geometry, and they can't be moved using the dimension Move option on the dimension's context menu. All formatting options and behavior are otherwise identical. Although placing isometric dimensions may seem unconventional, you can, in fact, often eliminate the number of views required to concisely communicate the design by placing dimensions on an isometric view.

A quick way to place dimensions on an isometric view is to right-click the view and choose Retrieve Dimensions.



Isometric Dimensions

60) Do you need to use Content Center (Content Center)

Many people find the file structure of Content Center–generated files to be at odds with the way they store purchased and standard parts. If you find this is the case with your setup, you might want to use Content Center to generate parts but then save copies of them under your own file structure as needed. Often, commonly used components are generated from Content Center in this manner and then placed in a company's purchase part directory after the Part Number and Description iProperties are adjusted.

61) Adding a New Category to Content Center (Content Center)

If you need a unique category for your own specific parts, you can simply add a new family of parts instead of using an existing category. You can do this from the Content Center Editor (accessed from the Manage tab). Make sure the library view is set to view only the read/write library in which you intend to create the new category and that the tree view is enabled. Then right-click in the blank space in the Category View pane, and choose Create Category. You can also create a subcategory by right-clicking an existing category and selecting Create Category.

Adding to Content Center

Reference:

Video Learning & Book

1) Autodesk Inventor 2014 Video Learning Content – www.trainingtutorial.com
2) Mastering Autodesk Inventor 2014 (book) - http://goo.gl/VEAdvw

Blogs

1) Inventor Tales > http://www.inventortales.com
2) CADSetterOut > http://cadsetterout.com
3) John Evans Design > http://www. http://designandmotion.net
4) Imaginit (Mark Flayler) > http://blogs.rand.com/manufacturing/mark-flayler.html
5) Autodesk MFG Northern Europe > http://autodeskmfg.blogspot.com
6) Inventor Trenches > http://inventortrenches.blogspot.com
7) Autodesk Mfg Blog Rolls > http://autodeskmfg.typepad.com/blog/blog-roll.html
8) Inventor Topix > http://inventortopix.blogspot.com
9) Wiki help > http://help.autodesk.com