

Need for Speed: 102 Productivity Hacks for Inventor

Clint Brown

Senior Technical Services Manager

[@ClintBrown3D](https://twitter.com/ClintBrown3D)

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About the speaker



Clint Brown

A background in Automotive Engineering, Plastics, Machining, and Rotating Machinery. Clint has worked as a mechanical design engineer for the last 19 years. He's an Autodesk Inventor Certified Professional, having used most traditional CAD and analysis tools including AutoCAD (since R13), SolidWorks, SolidEdge, and Ansys. He has designed many products, machines, and systems including plastic fuel tanks (Nissan, Ford); several mobile vending units (Coca-Cola, SAB Miller); and the design and stress analysis of castings (Daikin, McQuay). He is also a patent holder.

About the Speaker



Some Stats

5

AU CLASSES
PRESENTED

10

AU'S ATTENDED

21

YEARS USING
AUTODESK PRODUCTS

10

YEARS OF
INVENTOR & VAULT

Presented at:

AU Las Vegas 2013 x2

AU London 2017 x1

AU London 2018 x2

First AU in 2010

First used AutoCAD
R10 in 1997

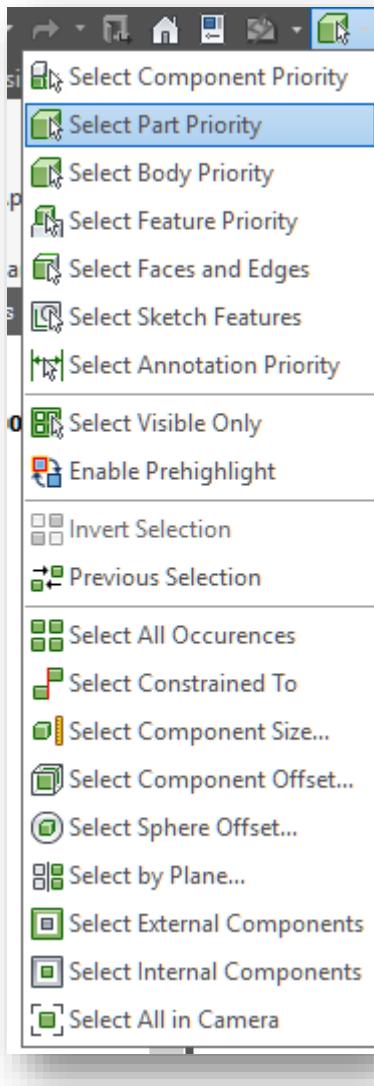
First used Inventor
R12 in 2008



Need for Speed: 102 Productivity Hacks for Inventor

@ClintBrown3D

Selection Filters



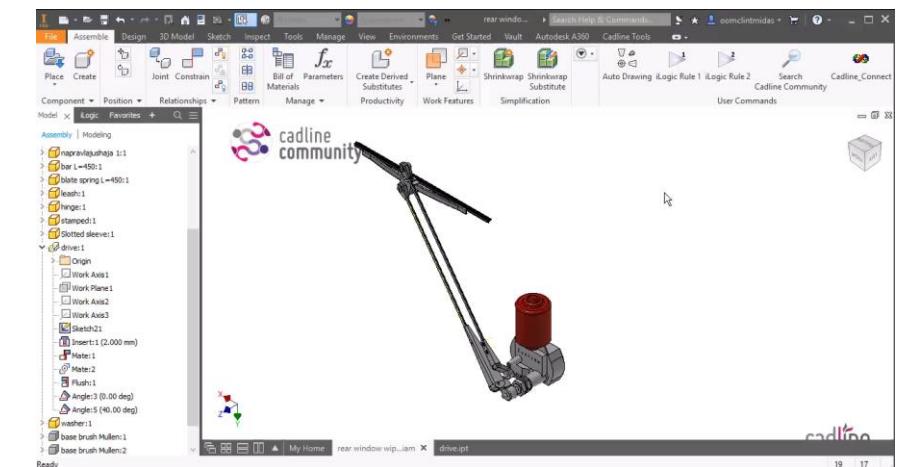
Each Inventor environment has a default Selection priority

Part: Faces and Edges

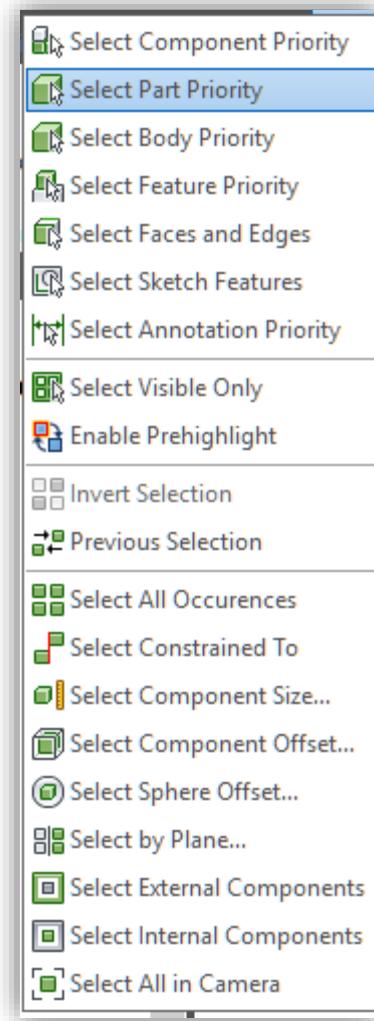
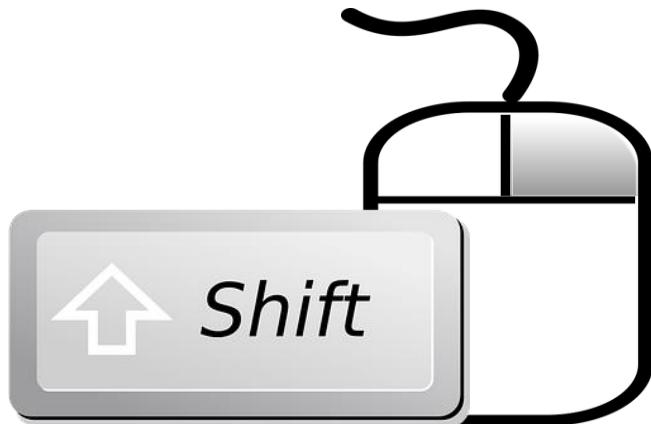
Assemblies: Components

Drawings: Edge

Changing the selection priority will give users faster access to specific geometry



Selection Filters

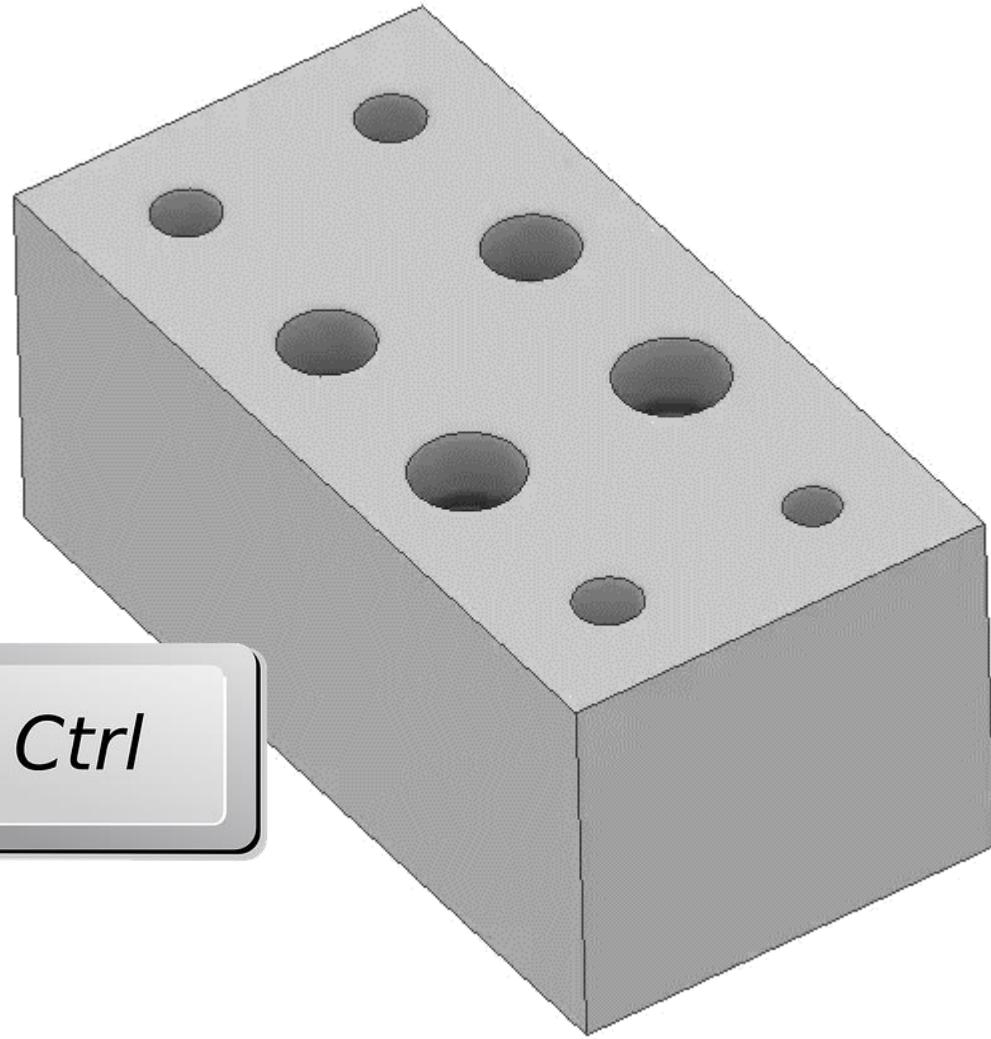
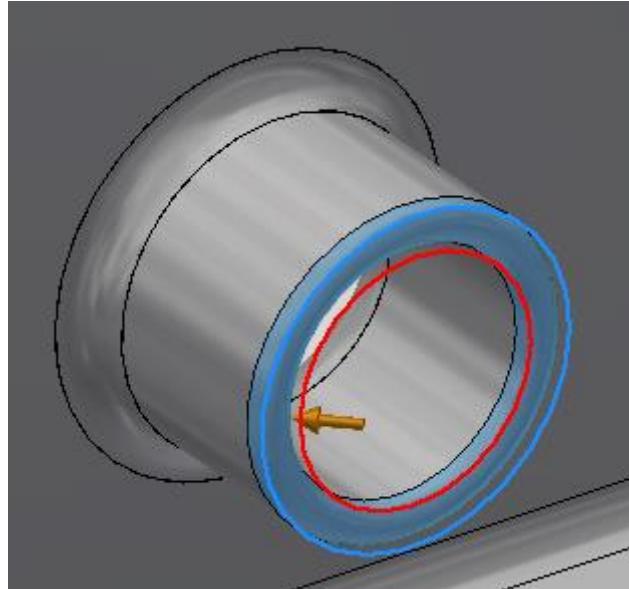


Access “Selection Filters”:

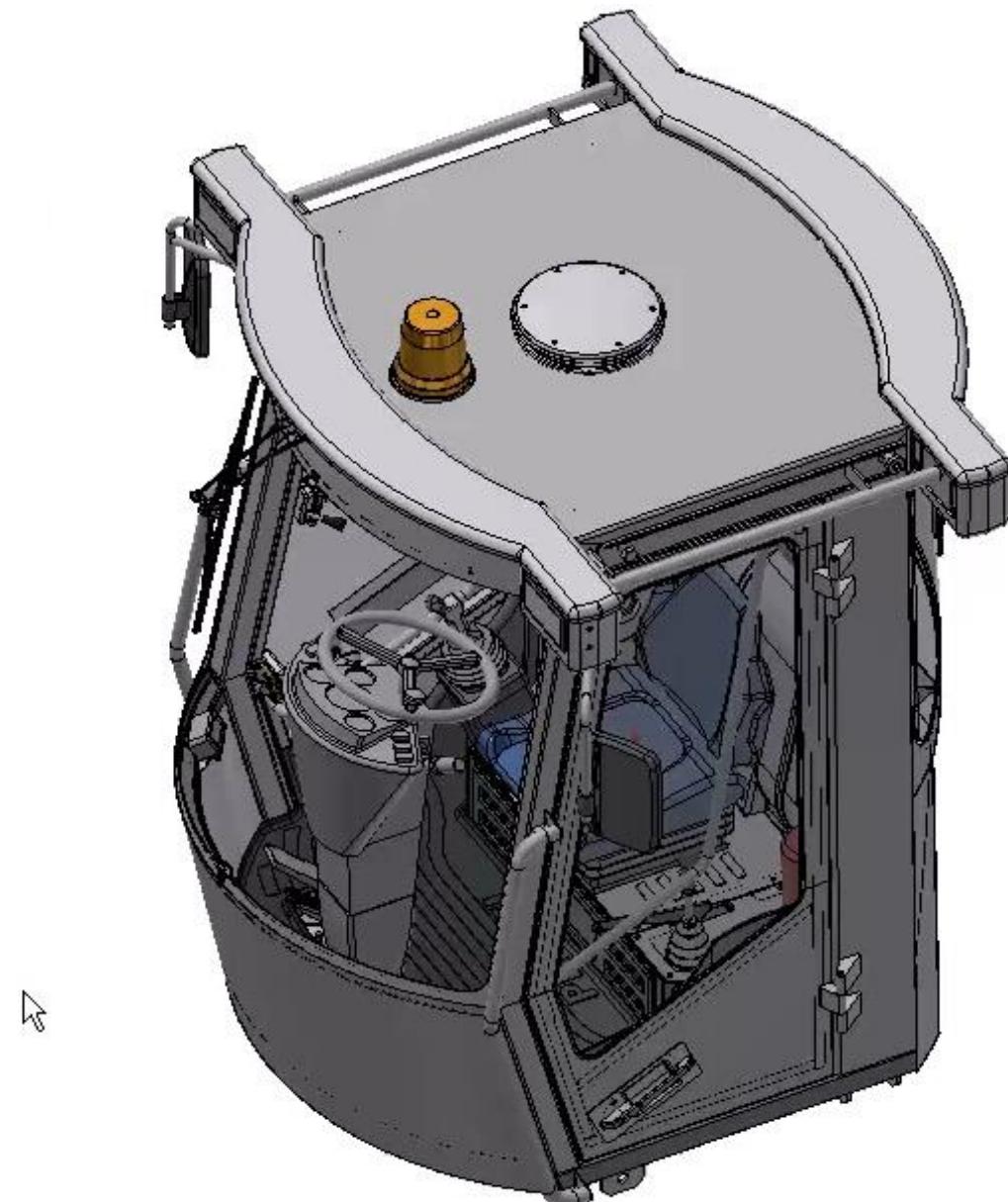
Shift + RMB

Ctrl to De-select objects

Holding Ctrl when selecting practically anything in Inventor, will allow you to de-select objects that have been selected



Assemblies: Isolate



Isolate is often overlooked. In assemblies, use it to show and hide components, this is especially useful when working on large assemblies, or when you need to hide a specific part, or if you just want to look at a specific set of parts

Box Select

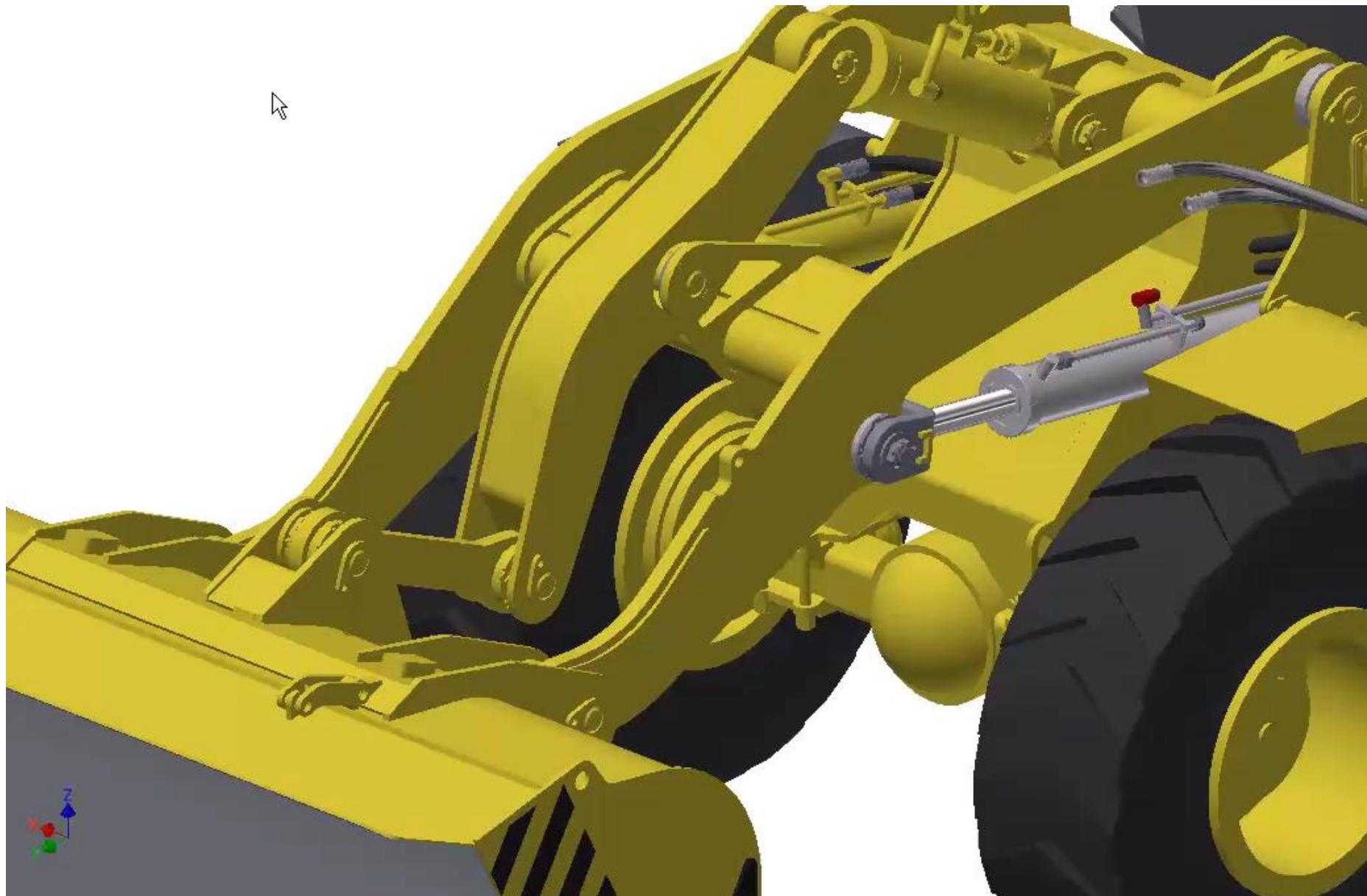
Left to Right

Everything fully inside the box



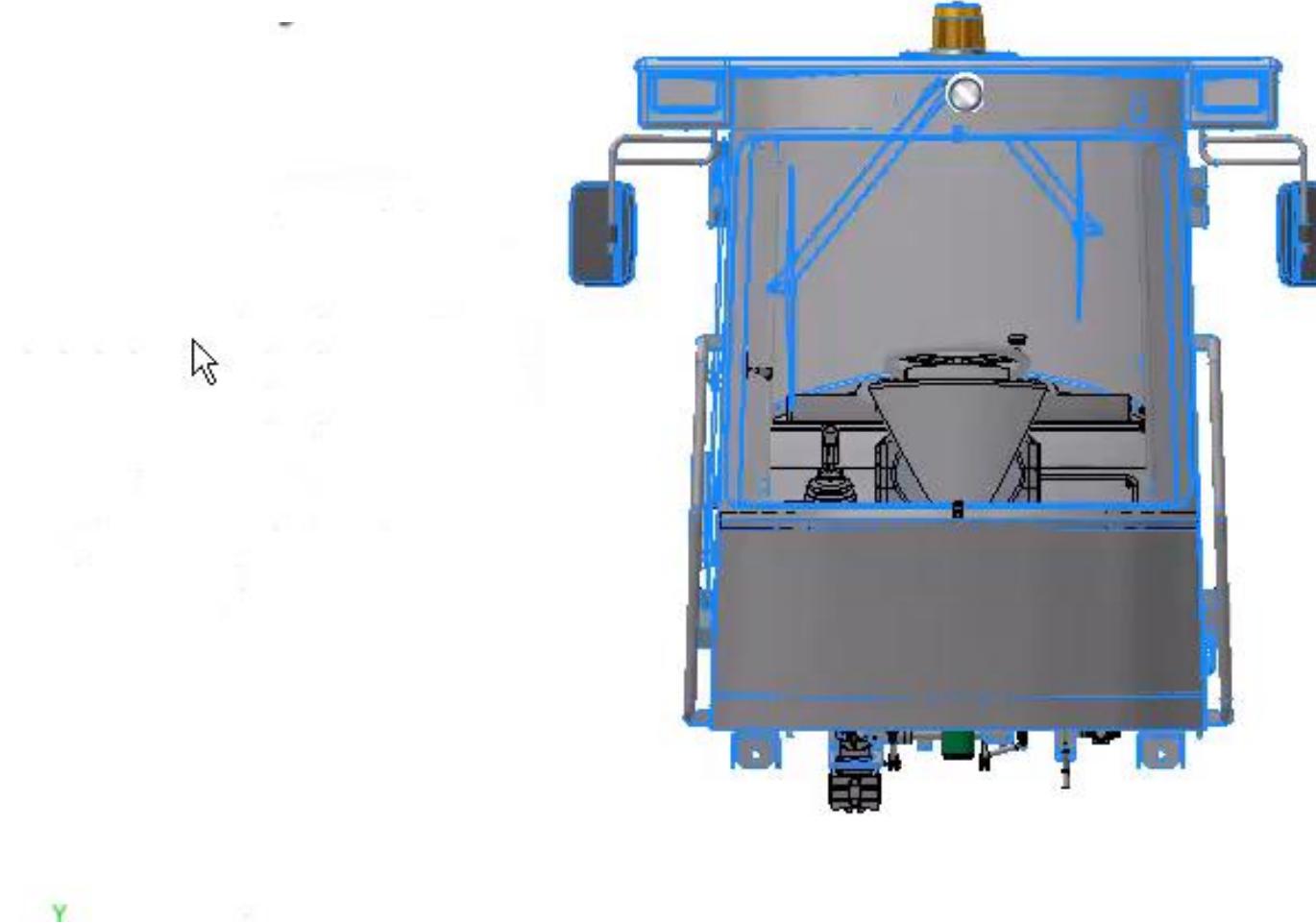
Right to Left

Everything touching the box



Assemblies: Invert Selection

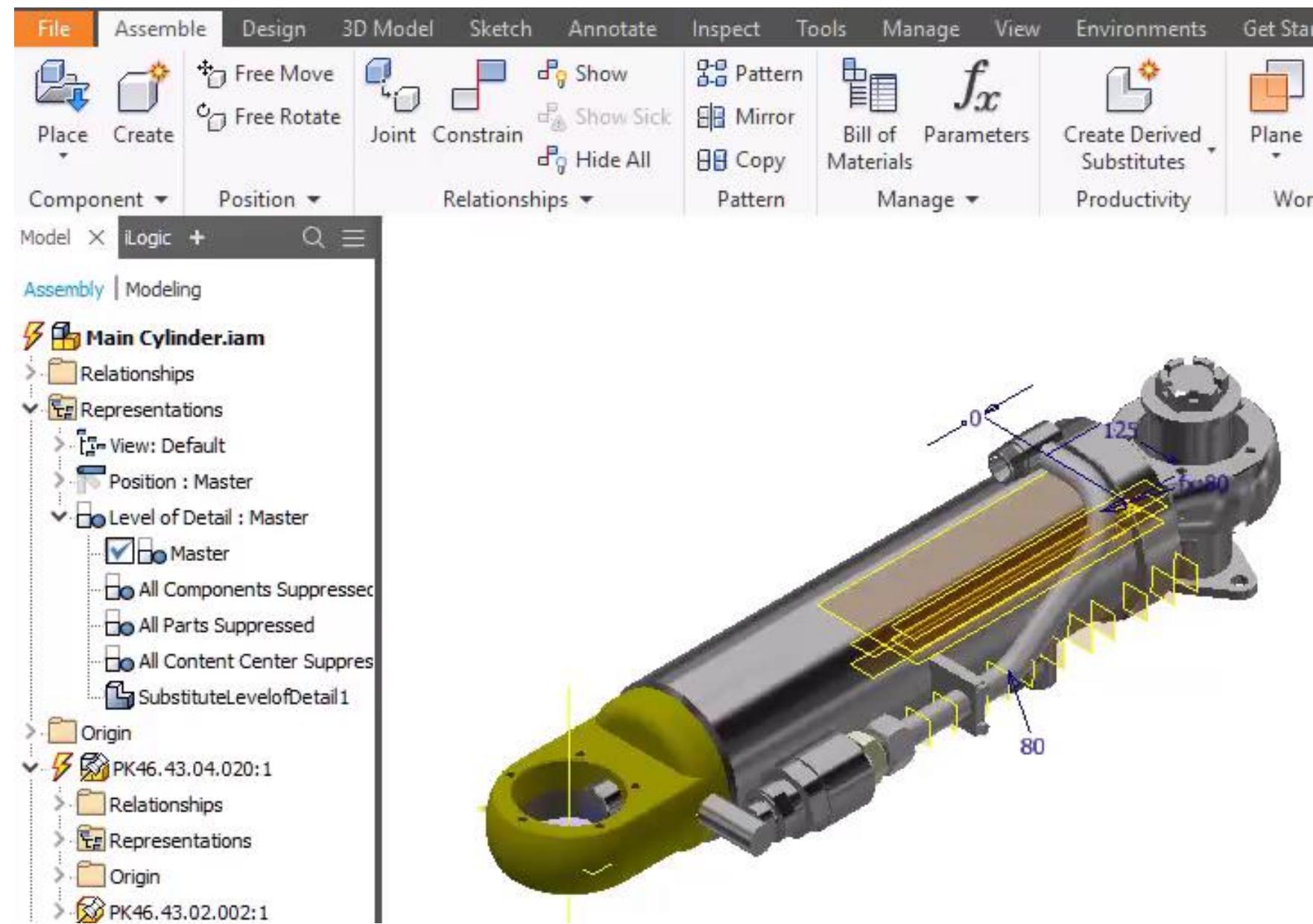
Use “**Invert Selection**” to select everything other than your current selection



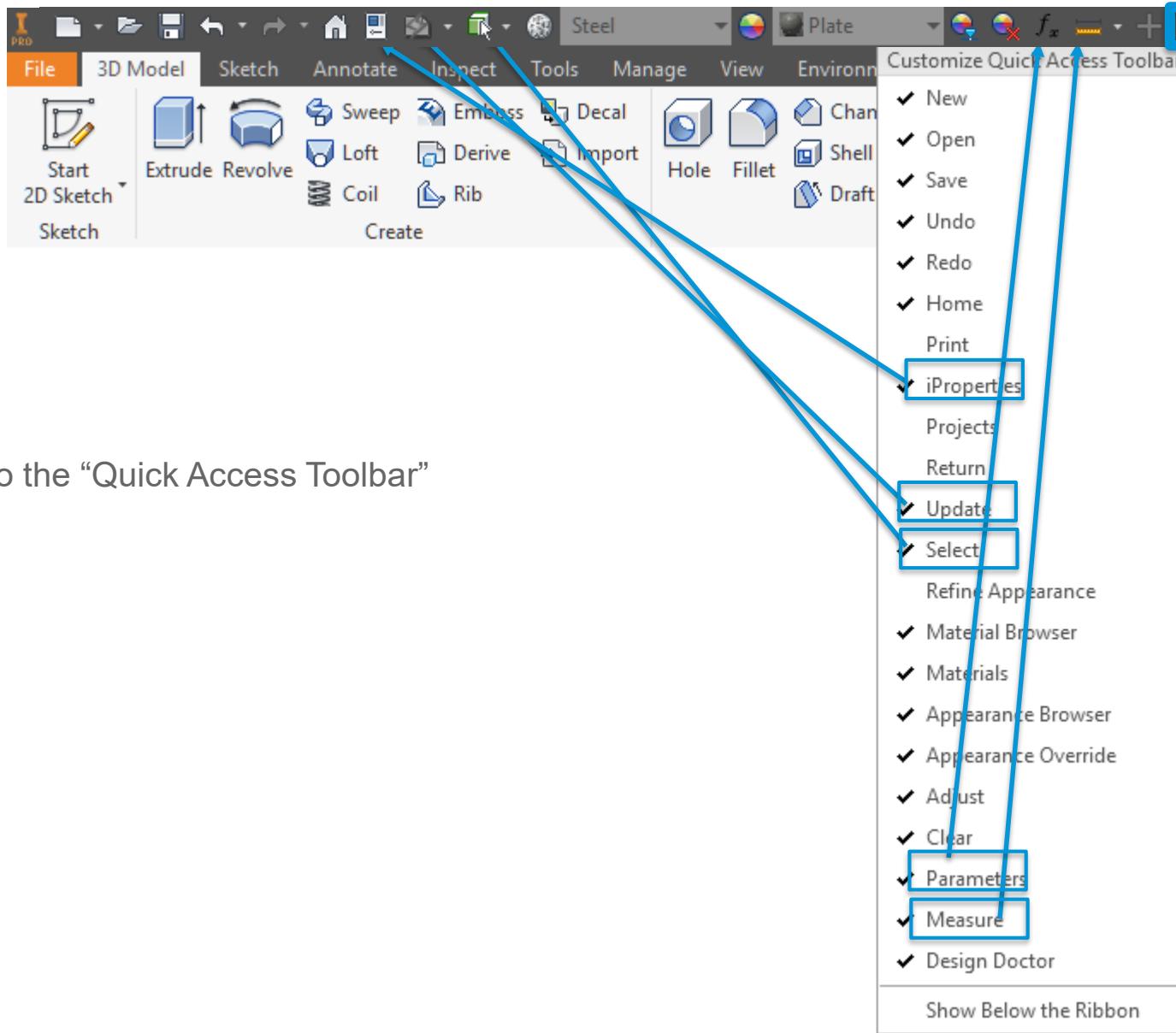
Object Visibility Filters

Toggle visibility of Workfeatures, sketches etc.

<input checked="" type="checkbox"/> All Workfeatures
<input checked="" type="checkbox"/> Origin Planes
<input checked="" type="checkbox"/> Origin Axes
<input checked="" type="checkbox"/> Origin Points
<input checked="" type="checkbox"/> User Work Planes
<input checked="" type="checkbox"/> User Work Axes
<input checked="" type="checkbox"/> User Work Points
<input checked="" type="checkbox"/> Construction Surfaces
<input checked="" type="checkbox"/> 2D Sketches
<input checked="" type="checkbox"/> 3D Sketches
<input checked="" type="checkbox"/> Sketch Dimensions
<input checked="" type="checkbox"/> 3D Annotations
<input checked="" type="checkbox"/> Component 3D Annotations
<input checked="" type="checkbox"/> Welds
<input checked="" type="checkbox"/> Weldment Symbols
<input checked="" type="checkbox"/> UCS Triad
<input checked="" type="checkbox"/> UCS Planes
<input checked="" type="checkbox"/> UCS Axes
<input checked="" type="checkbox"/> UCS Points



Speed: Use the quick access toolbar



Add commonly used tools to the “Quick Access Toolbar”

Templates



Part



Assembly



Drawing



Presentation

Overwriting the **Standard.upt/dwg/iam/iphn** templates with your customised templates will make new file creation quicker

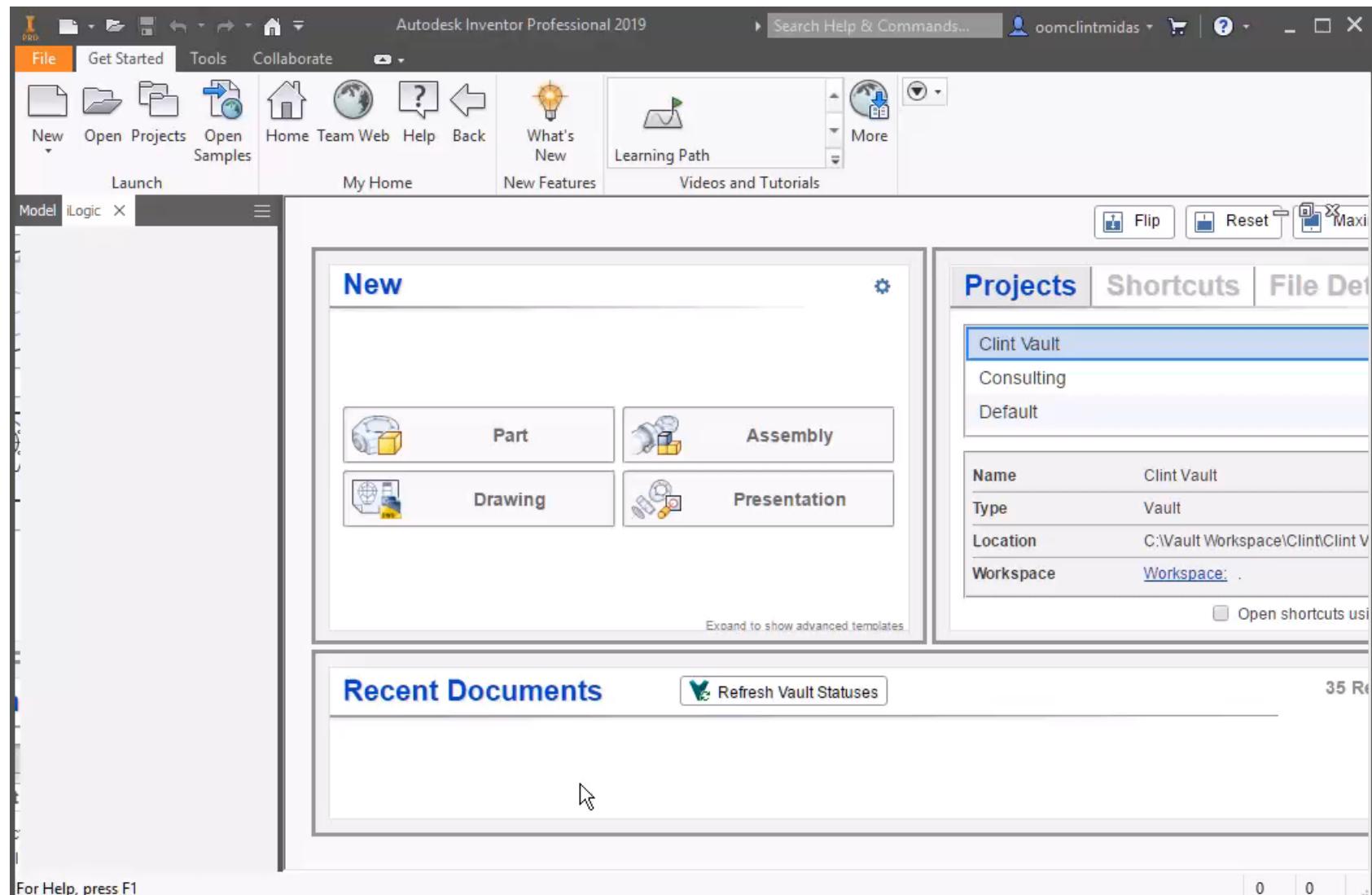
The screenshot shows the software's main interface with a focus on the 'New' template selection screen. The 'New' section displays four template categories: Part, Assembly, Drawing, and Presentation. The 'Recent Documents' section shows a list of recent files including 'STL-Mesh.iam', 'Plastic cover.upt', and 'FERRARAFORNI F...'. The 'Projects' and 'File Details' sections are also visible on the right.

Speed: Smart Templates

- Save pre-selected Material parts (i.e. Steel, PP etc.)
- Pre-populate iProperties + Parameters
- Re-use common shapes/start points
- Create sheet Metal templates too (Material and Thickness)

Assemblies

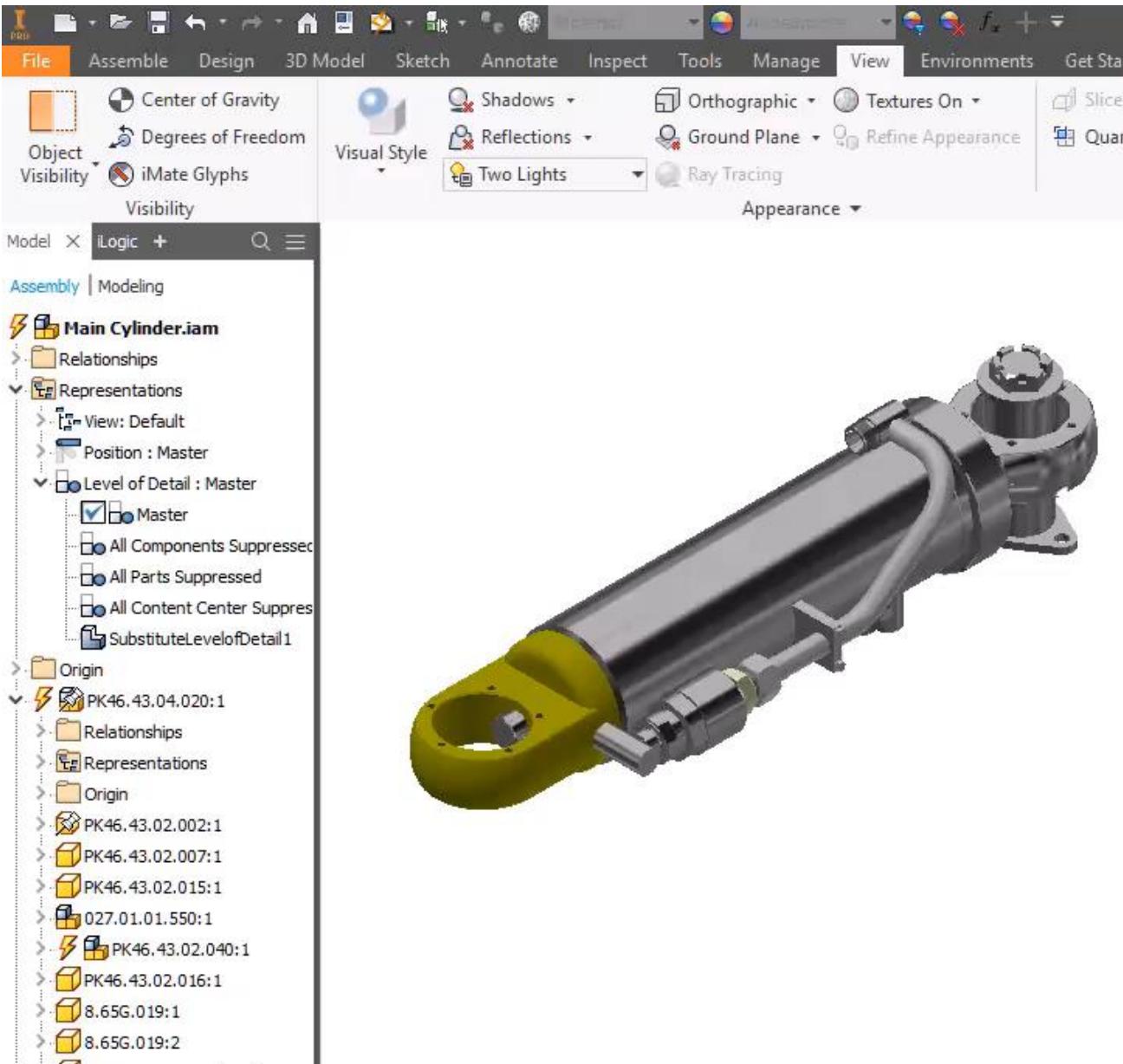
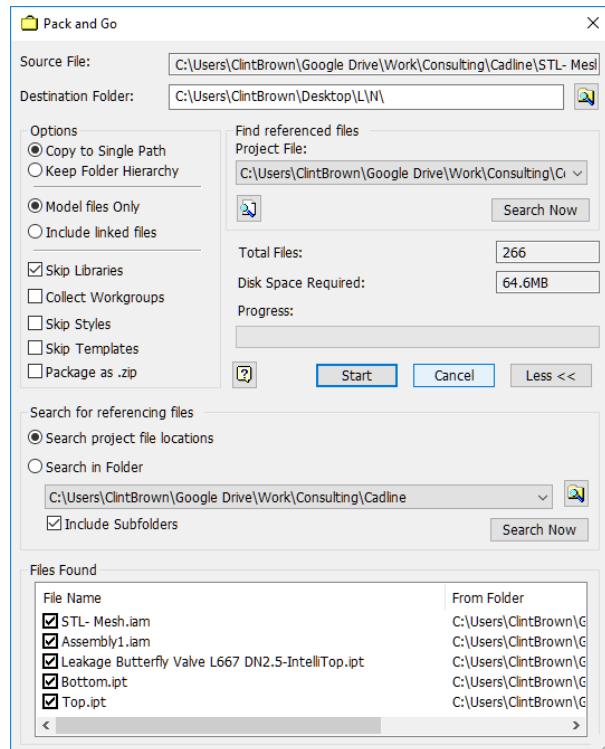
- Place virtual components in your template file
- Setup your BoM rows in the template file



Use Pack 'n go to Share files

Pack and Go packages an Autodesk Inventor file and all of its referenced files in a single location. All files that reference the selected Autodesk Inventor file from a selected project or folder can also be included in the package.

Use Pack and Go to archive a file structure, copy a complete set of files while retaining links to referenced files, or isolate a group of files for design experimentation.



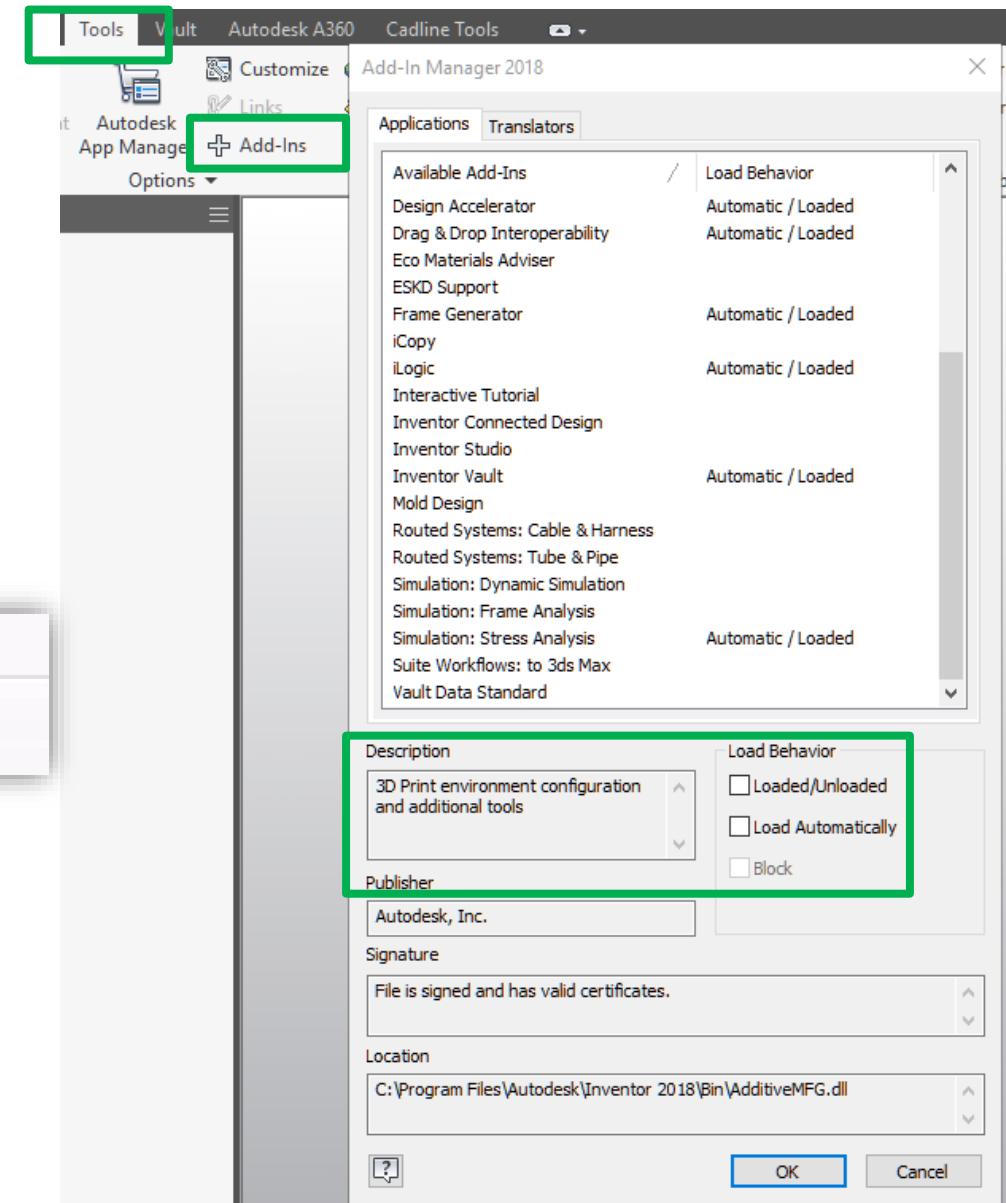
Speed: Manage Add-ins

Modern versions of Inventor no longer load all of the add-ins at start-up.

They are loaded when the first document is opened or created

Unload everything you do not need, including Add-ins you only use occasionally, they can always be added back in.

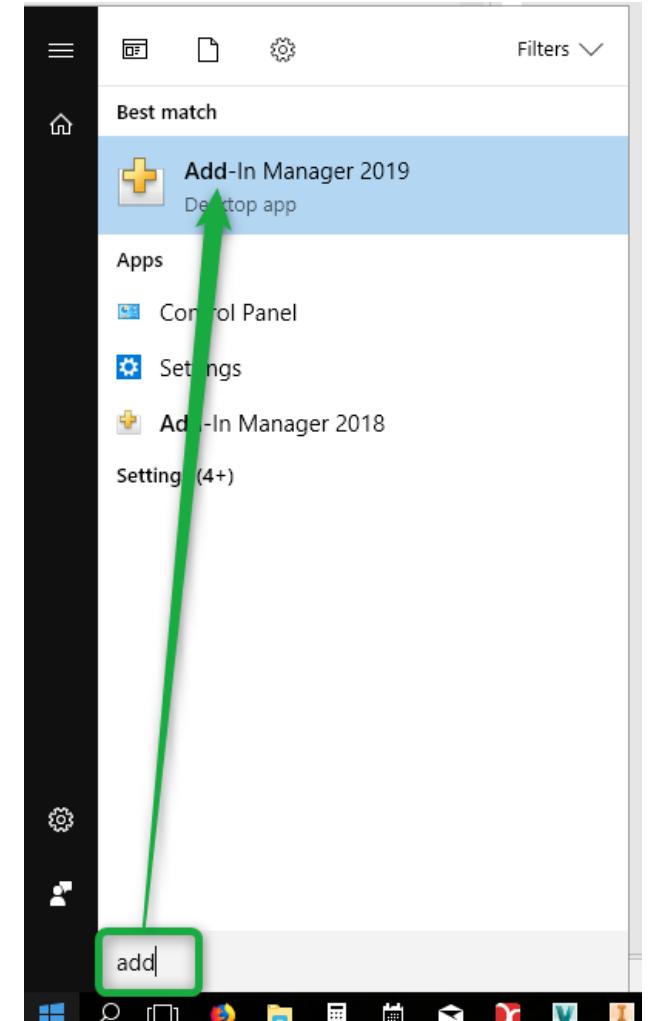
Note that Translators are loaded on demand



Speed: Manage Add-ins

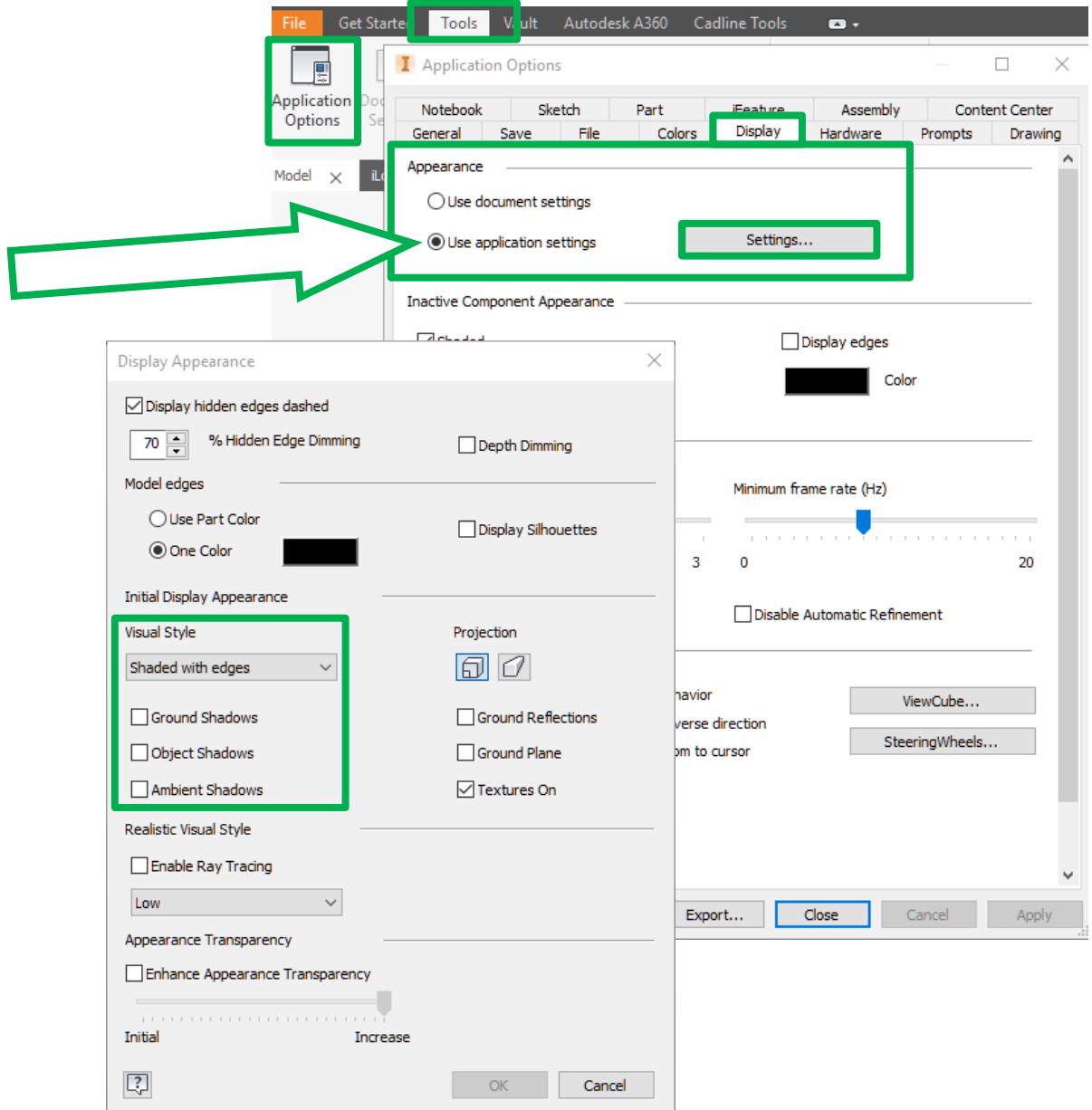
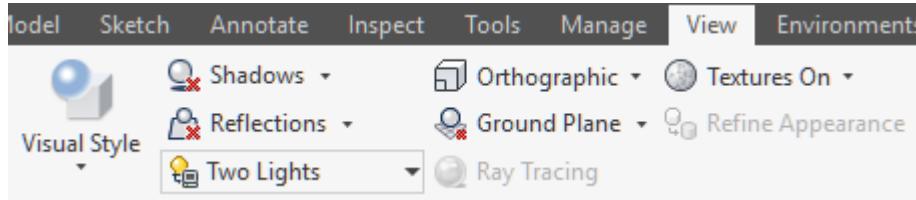
The Add-in Manager can also be loaded as a “Standalone” application.

This is useful if a specific add-in is causing stability issues.



Speed: View – Shadows Reflections & Visual Style

By default visual styles are set in the Inventor File



You can override this in your application options.

Benefits are that models appear as you prefer, and speed

Shadows and reflections slow you down

Speed: F-Key Shortcuts

The Function, or "F" keys on your keyboard provide shortcuts for model navigation inside of Autodesk Inventor, below is a list of these shortcuts;

- F2 – Pan
- F3 – Zoom
- F4 – Orbit
- F5 – Zoom Previous
- F6 – Home View

Bonus UI navigation tools:

The middle mouse button. Holding down the middle mouse button activates the "Pan" command (as per F2). Scrolling the middle mouse button will zoom the model in and out. Holding Shift and the middle mouse button will activate the "Orbit" command (as per F4).



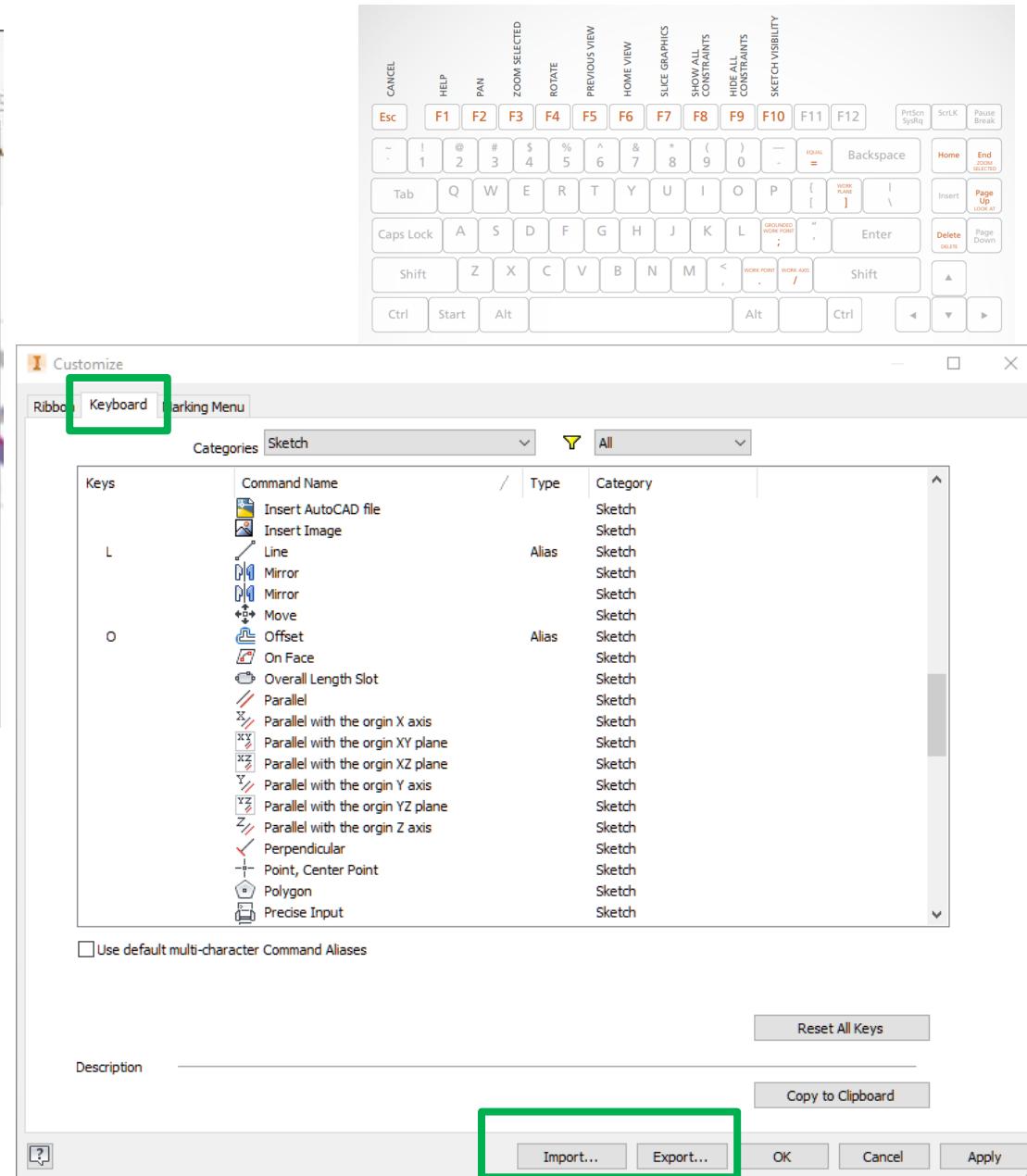
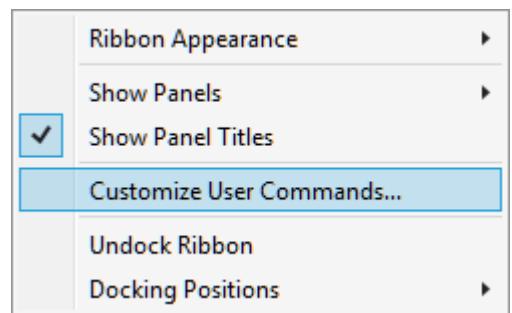
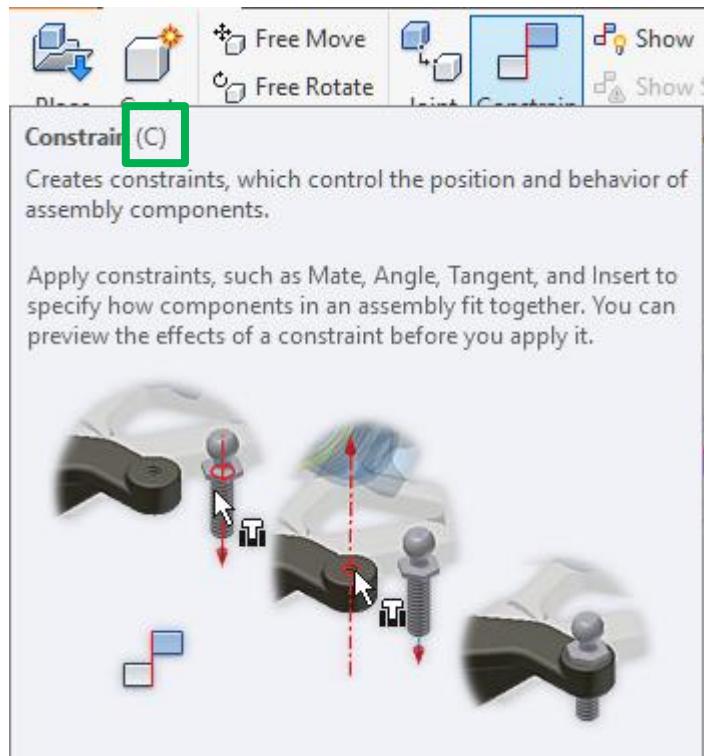
Speed: Keyboard Shortcuts

Hover your mouse over any command in Inventor, you will see the shortcut shown in brackets after the description. Example - Fillet (F).

The link below is to a PDF containing all Inventor shortcuts

Right Clicking on your Ribbon, and selecting “Customise User Commands” will allow you to modify shortcuts and set up shortcuts for commands that are not set.

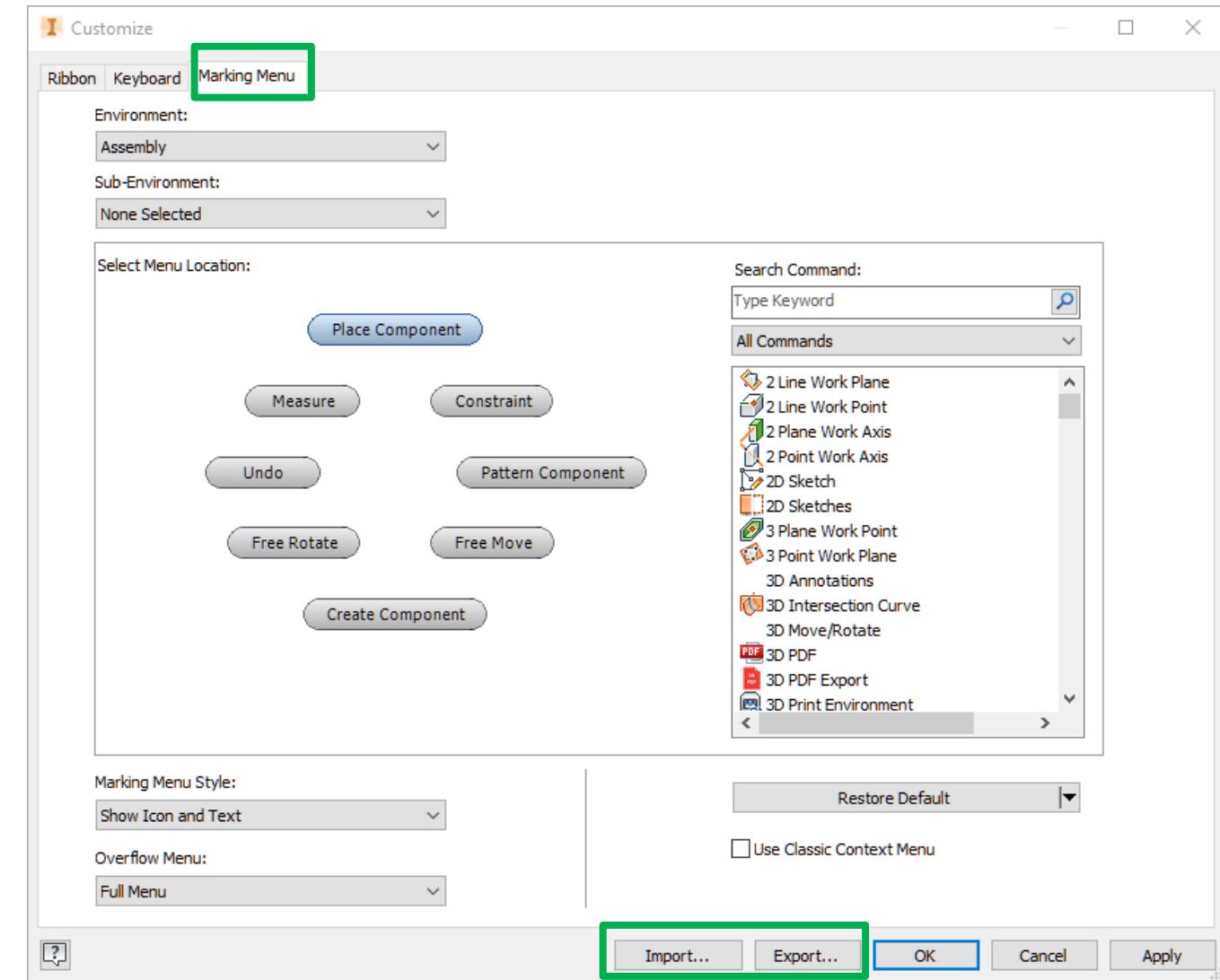
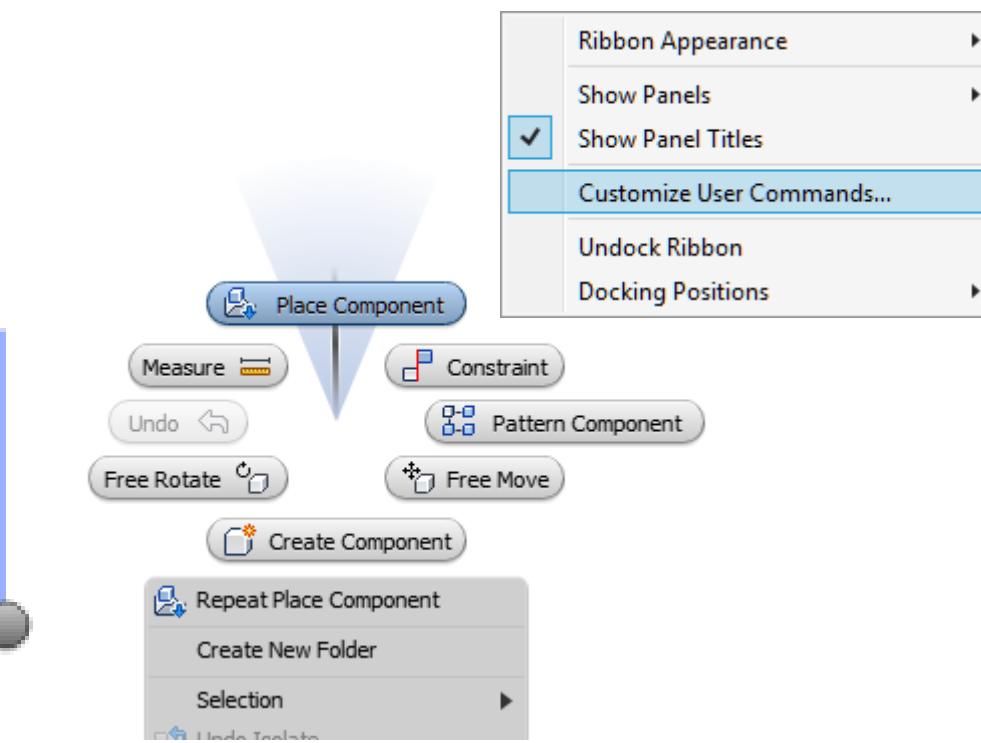
Share your settings using Export an import options



Speed: Customise the Marking (use Gestures)

Right Clicking on your Ribbon, and selecting “Customise User Commands” will allow you to modify your marking menu settings.

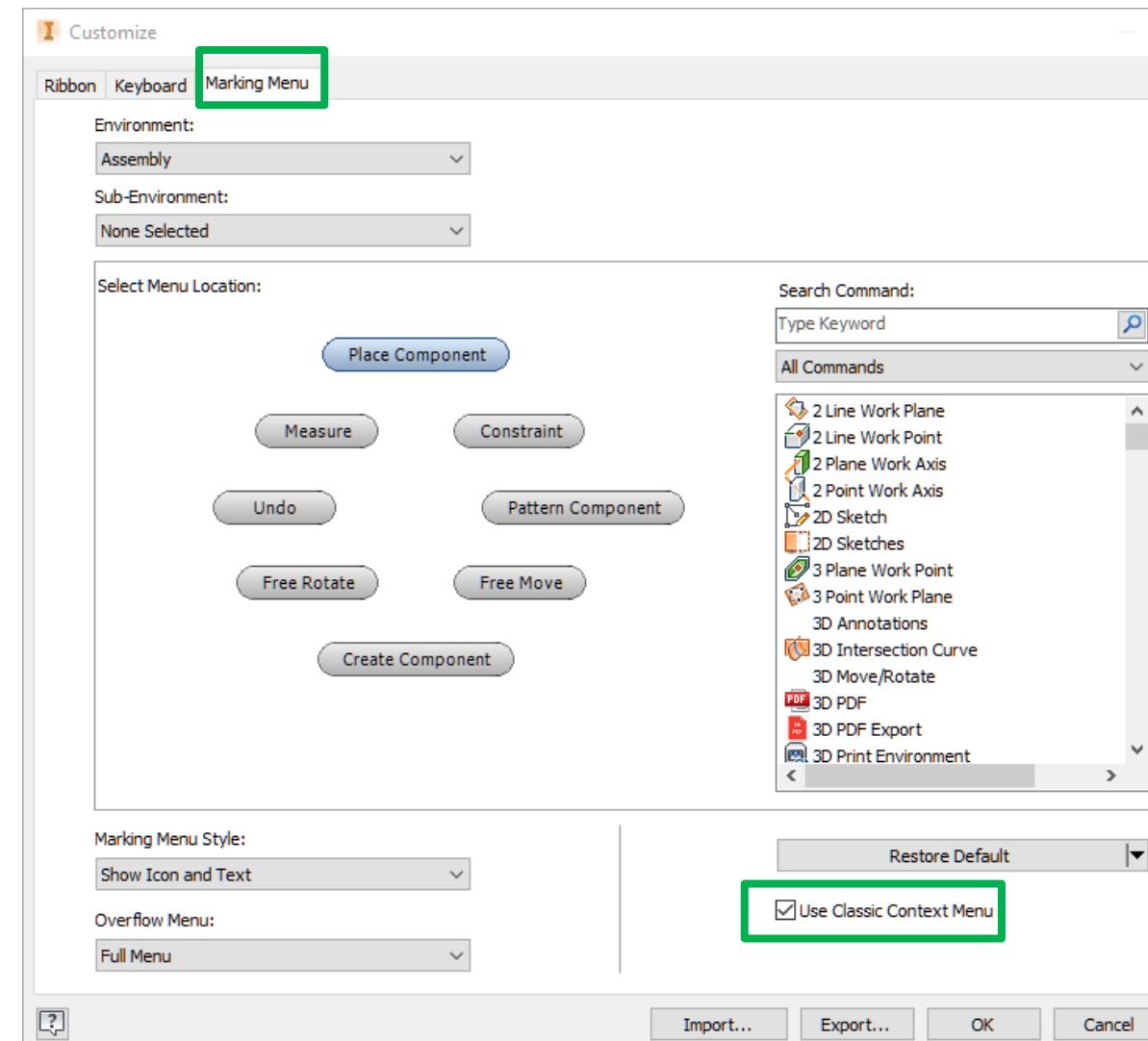
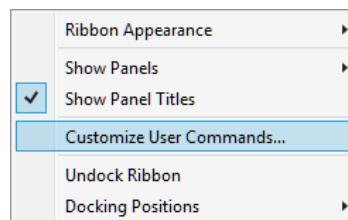
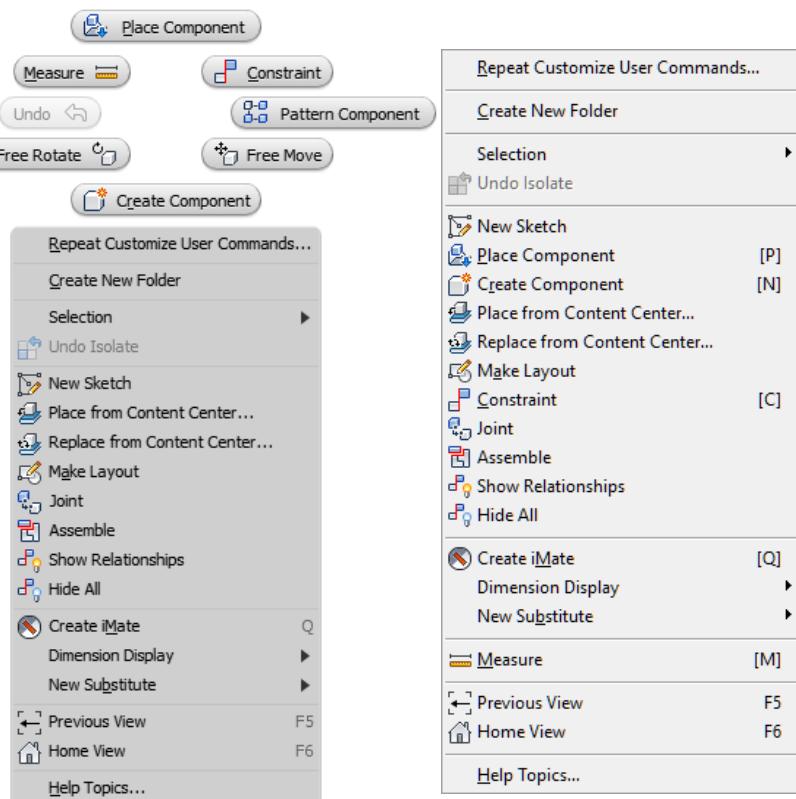
Share your settings using Import and Export options



Speed: Disable the Marking Menu (if it's not your thing!)

Right Clicking on your Ribbon, and selecting “Customise User Commands” will allow you to modify your marking menu settings.

Share your settings using Export an import options



Next 3 Tips need your IT department



HAVE YOU TRIED
TURNING IT OFF
AND ON AGAIN?



Speed: Anti-Virus Exceptions

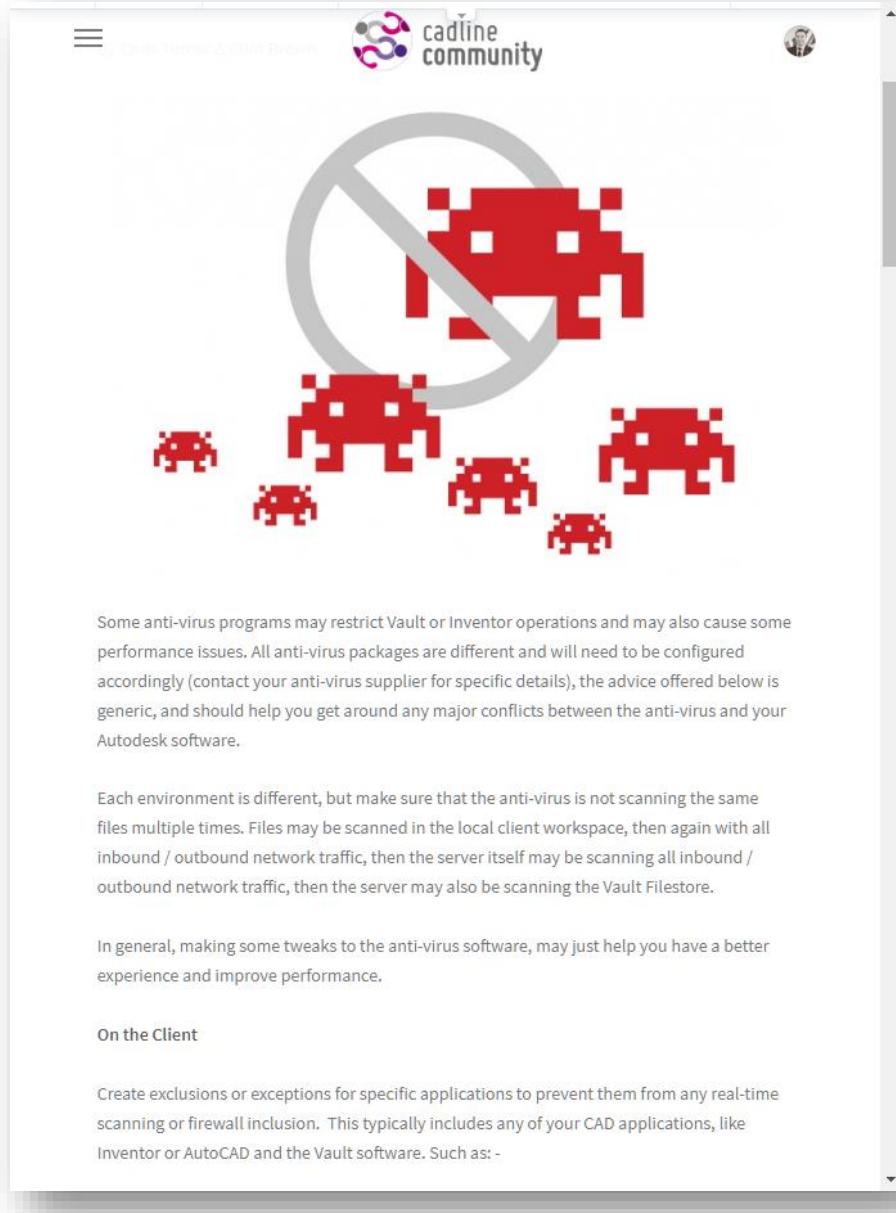
Your Company Anti-virus software is probably slowing you down.

Real time scanning, slows access to files

DLL & EXE blocking make parts of Inventor inoperable

Note

It is strongly recommended that you do **not lower the security on any computer without good reason**. Changes to the default configuration of your antivirus software are made completely at the risk of the administrator and should only be undertaken if the implications of lowering the security are completely understood



Some anti-virus programs may restrict Vault or Inventor operations and may also cause some performance issues. All anti-virus packages are different and will need to be configured accordingly (contact your anti-virus supplier for specific details), the advice offered below is generic, and should help you get around any major conflicts between the anti-virus and your Autodesk software.

Each environment is different, but make sure that the anti-virus is not scanning the same files multiple times. Files may be scanned in the local client workspace, then again with all inbound / outbound network traffic, then the server itself may be scanning all inbound / outbound network traffic, then the server may also be scanning the Vault Filestore.

In general, making some tweaks to the anti-virus software, may just help you have a better experience and improve performance.

On the Client

Create exclusions or exceptions for specific applications to prevent them from any real-time scanning or firewall inclusion. This typically includes any of your CAD applications, like Inventor or AutoCAD and the Vault software. Such as:-



Learn & Explore

Working with Inventor and antivirus software

By:  Autodesk.Support

Jun 30 2017

Issue:

When you work with Inventor and there is antivirus software active on your machine, you may experience slow performance. For example, you may have slow response times while using Inventor and refresh of Inventor UI. This is caused by antivirus software and you would like to optimize your performance.

Solution:

Inventor files normally should be excluded from real-time antivirus scanning. Users should also disable virus scanning during installation of the product.

Excluding Inventor files (.iam, .idw, .ipt and ipn at the very least) from real-time scanning is important since Inventor opens a lot of files (especially for a large assembly drawing, for example) and the virus/malware threat from them is extremely low.

Users can also exclude the workspace of their Inventor projects so that they are not scanned during work, which will give a better overall performance.

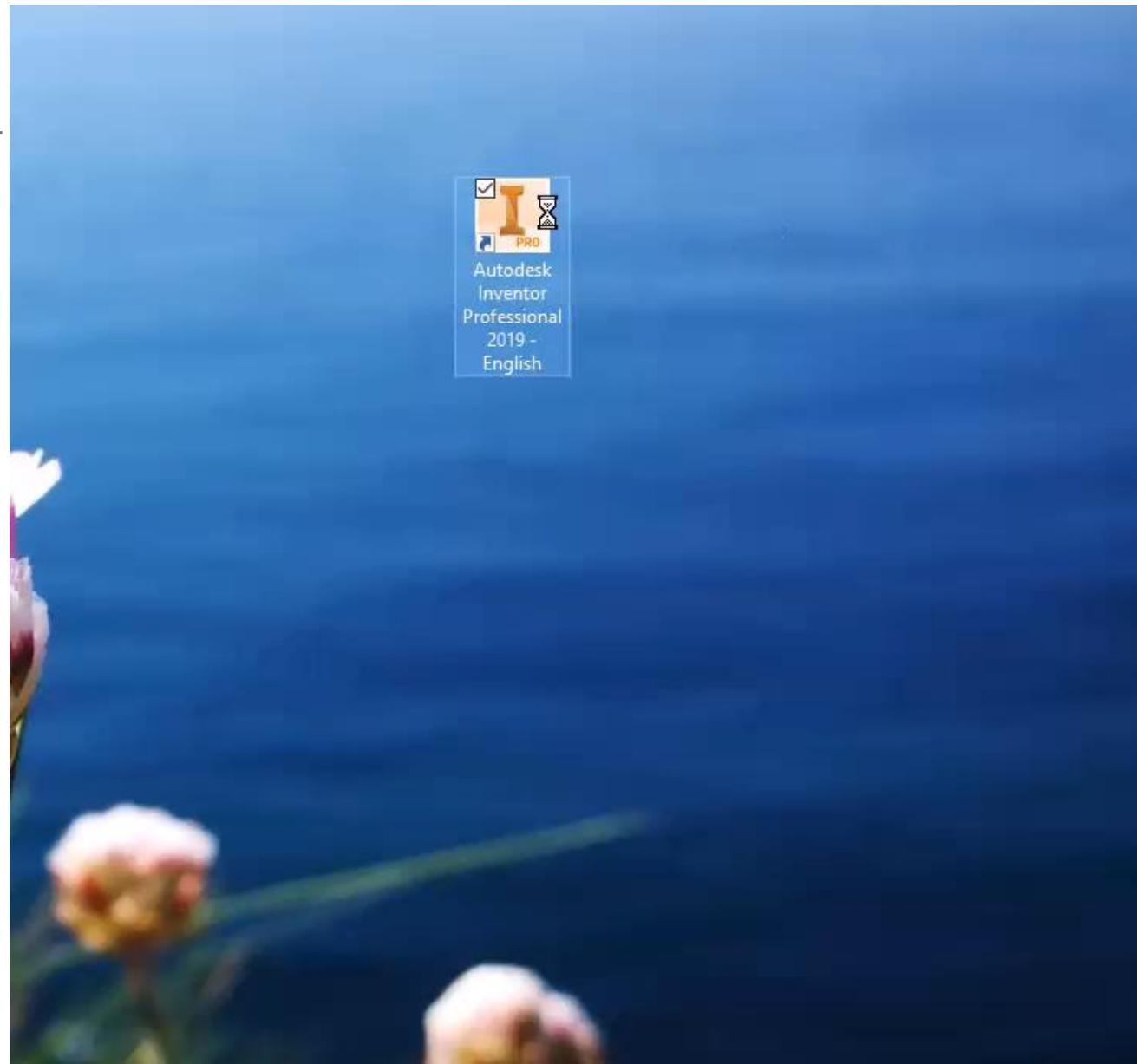
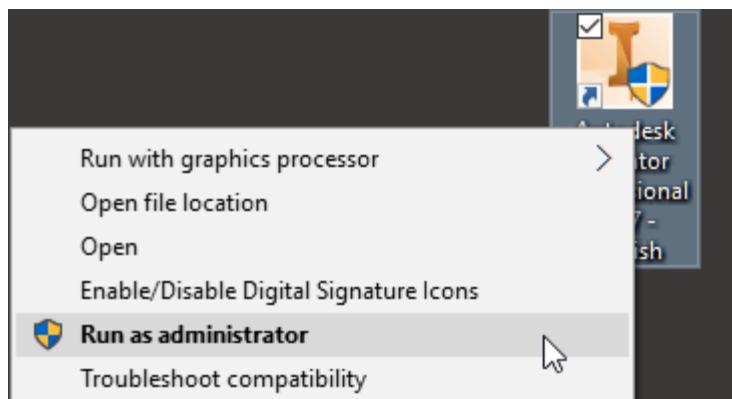
Additional files which could be excluded from virus scanning are .CHM file types (in

Speed: Run as Administrator

You may come across instances where Inventor will not load an add-in or a DLL, this is sometimes manifested in an error message, or a function just not working.

When installing Inventor, it should always be done with the UAC turned right down, with the user logged on as an Administrator, and with the anti-virus disabled. While logged on as Administrator, it is advised that Inventor is fired up, as this registers all of the relevant DLL's with the Windows operating system.

As a user, If you are still having issues, a nice workaround, is to run Inventor as Administrator, this can be achieved by right clicking on your Inventor icon and choosing "Run as Administrator"



Speed: Drivers: Graphics & 3D Mouse

AUTODESK. Clint Brown English

AUTODESK KNOWLEDGE NETWORK Search Vault Products

Autodesk Inventor 2019 Windows 10 64-bit

Recommended	Certified	Card	Generation	Memory (MB)	Type	Manufacturer	Driver Name	Driver
✓	✓	AMD FirePro W2100 (FireGL V)	2048	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W4100 (FireGL V)	2048	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W4300 (FireGL V)	4096	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W5000 (FireGL V)	2048	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W5100 (FireGL V)	4096	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W7000 (FireGL V)	4096	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W7100 (FireGL V)	4096	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W8000 (FireGL V)	4096	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W8100 (FireGL V)	8192	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	AMD FirePro W9000 (FireGL V)	6144	Workstation	AMD	AMD Radeon Pro 17.Q4.1		
✓	✓	NVIDIA Quadro GP100	Pascal	16384	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro GV100	Volta	32768	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 391.03	
✓	✓	NVIDIA Quadro K1200		4096	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K2000	Kepler	2048	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K2200		4096	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K2200M		2048	Mobile	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K4000	Kepler	3072	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K420		1024	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K4200		4096	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K5000	Kepler	4096	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K5200		8192	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K600	Kepler	1024	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K6000	Kepler	12288	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro K620		2048	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro M2000		4096	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro M4000	Maxwell	8192	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro M5000	Maxwell	8192	Workstation	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	
✓	✓	NVIDIA Quadro M6000	Maxwell	8192	Mobile	NVIDIA	Quadro Desktop/Quadro Notebook 385.90 64bit	

1. Ensure that your system **EXCEEDS** the minimum specs for running Inventor
2. Use the latest Graphics driver **FROM AUTODESK**, ask your IT department to [adjust your windows updates](#)
3. Ensure that your 3D Mouse driver is up to date, if you start having unexplained performance issues, unplug and uninstall it, see if this remedies the problem.



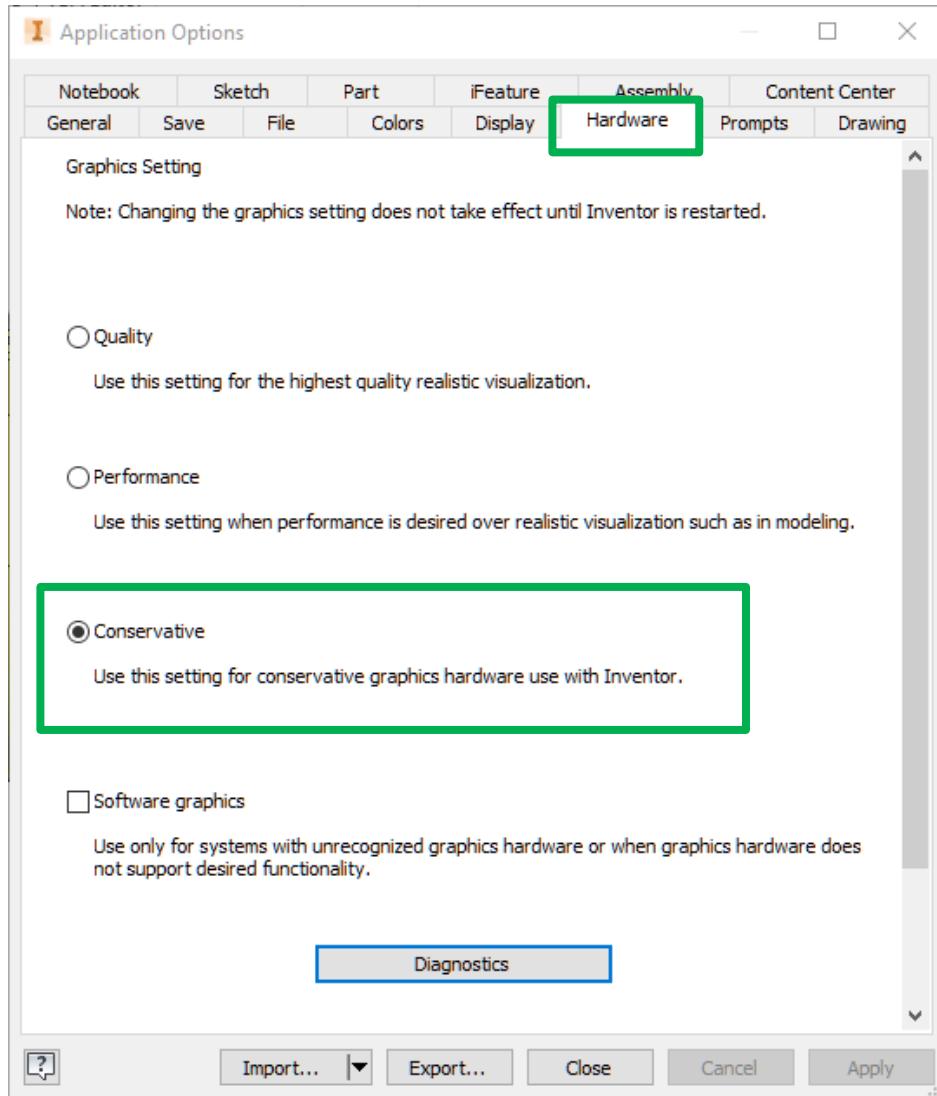
<https://www.3dconnexion.co.uk/index.php?id=76&redirect2=www.3dconnexion.co.uk>

[Link to Inventor Certified Graphics Cards](#)

Back Over to you

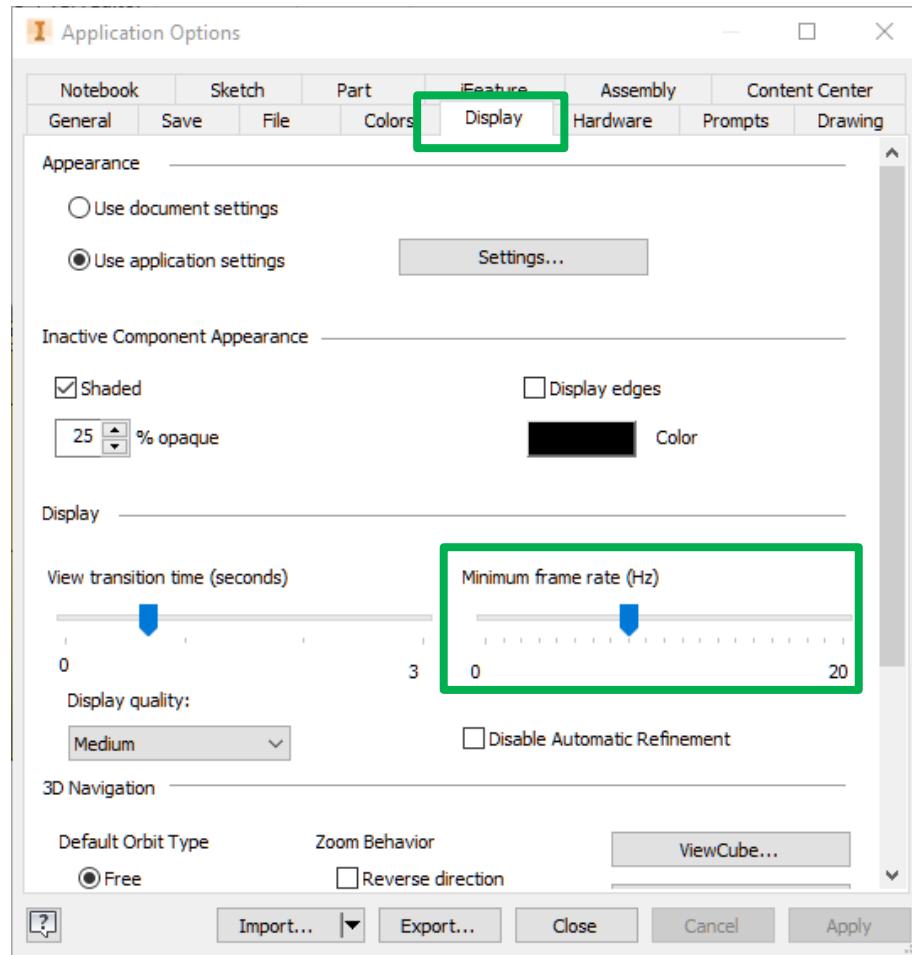


Speed: Graphics



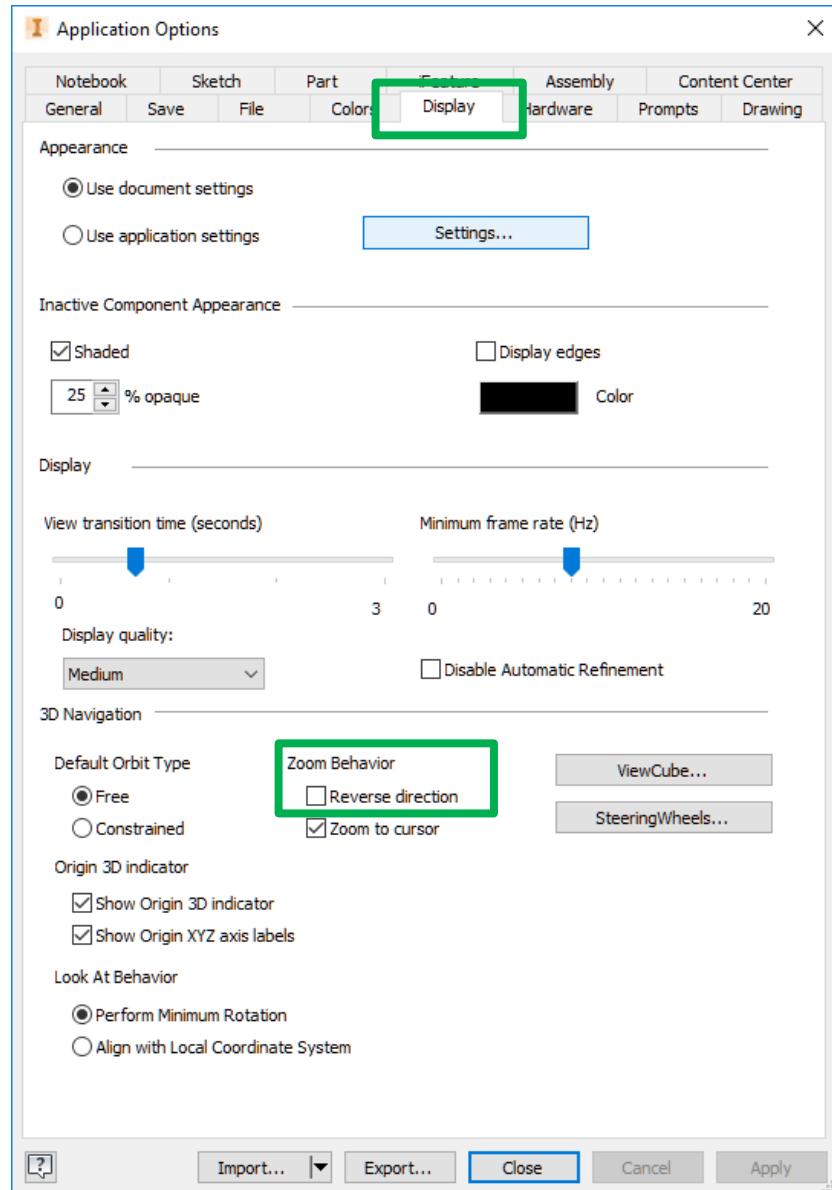
1. If you are experiencing issues, try a lower hardware setting (In Application Options)

Speed: Graphics

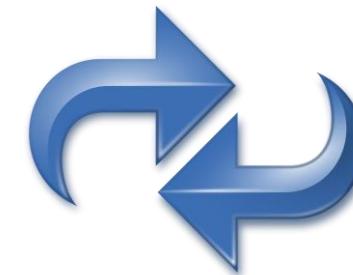


If you are still experiencing issues, try a lower frame rate, it's worth experimenting with a setting that will suit your system (In Application Options)

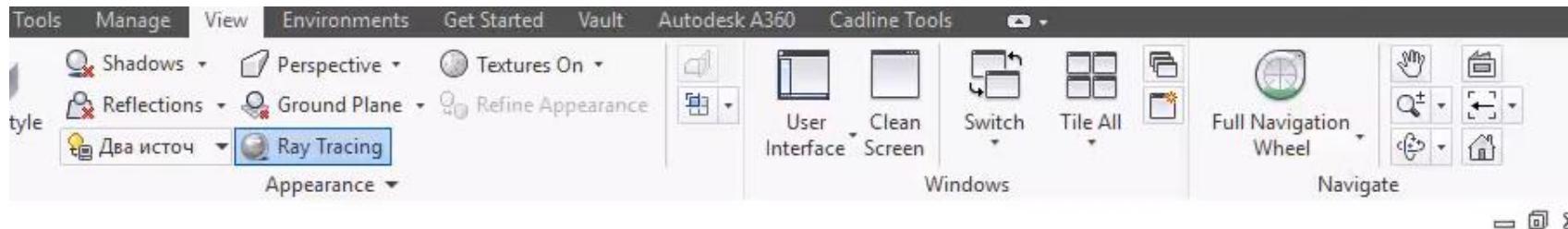
Speed: Zoom Direction



Tools > Application Options > Display > Zoom Behavior



Rendering – Ray Trace still images



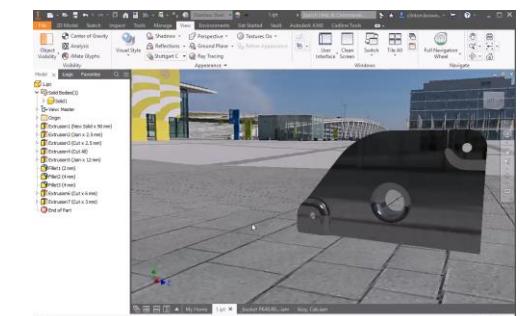
Save images of your ray trace while it's rendering

Can help with initial views while a large / time consuming render completes

Environments



You can turn off an environment, but still use it's lighting and reflections



Do you design Plastic parts? Use the Tooling environment

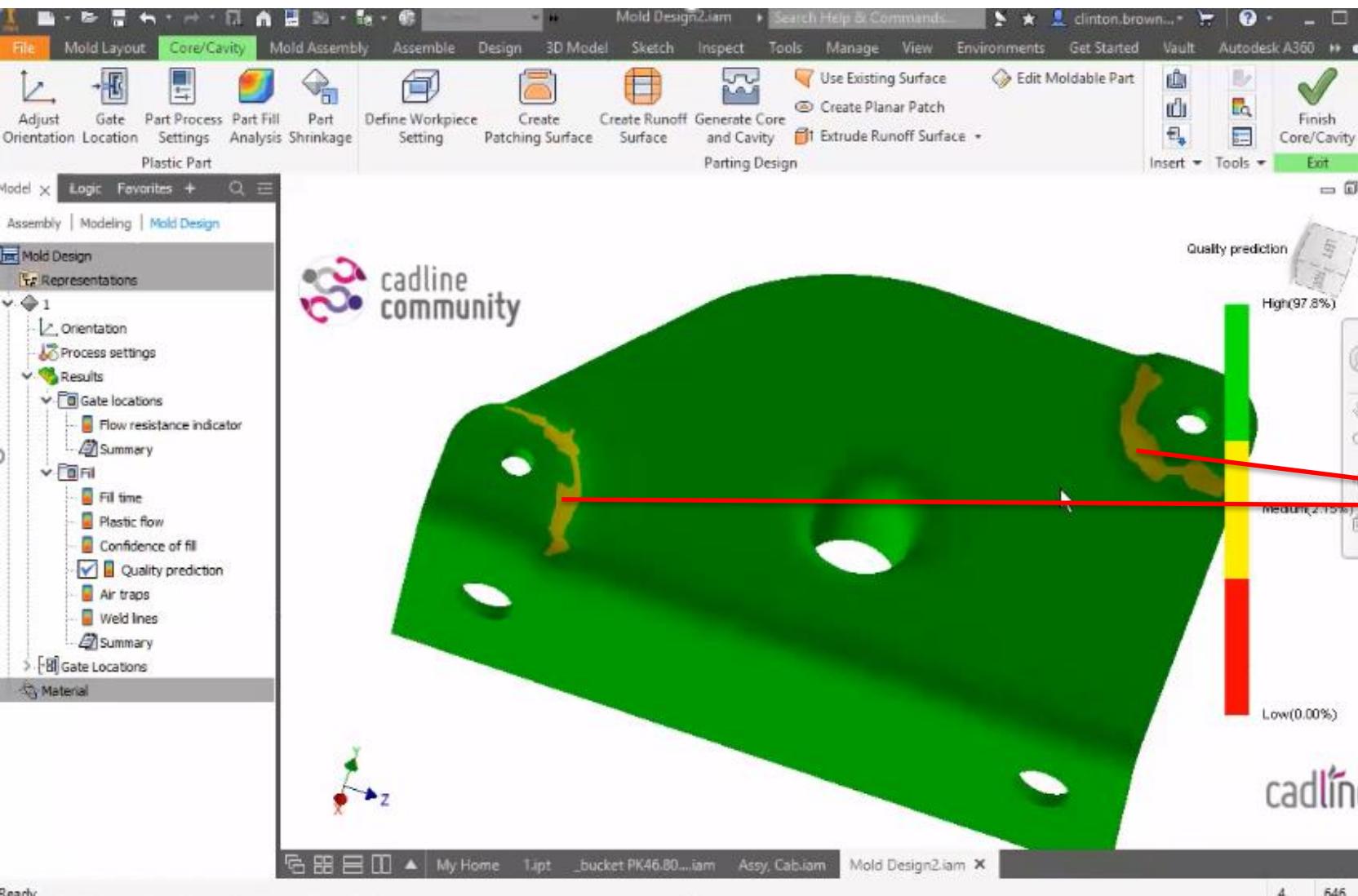
After some basic setup

Setting material
Selecting Default processing parameters
Estimated Gate location

You can report on Cycle time and Clamping pressure, which should help with costings

Actual filling time	0.90 (s)
Actual injection pressure	8.438 (MPa)
Clamp force area	6.6755 (cm ²)
Max. clamp force during filling	0.383 (tonne)
Velocity/pressure switch-over at % volume	99.06 (%)
Velocity/pressure switch-over at time	0.88 (s)
Estimated cycle time	14.52 (s)
Total part weight	10.090 (g)
Shot volume	13.1498 (cm ³)

Do you design Plastic parts? Use the Tooling environment

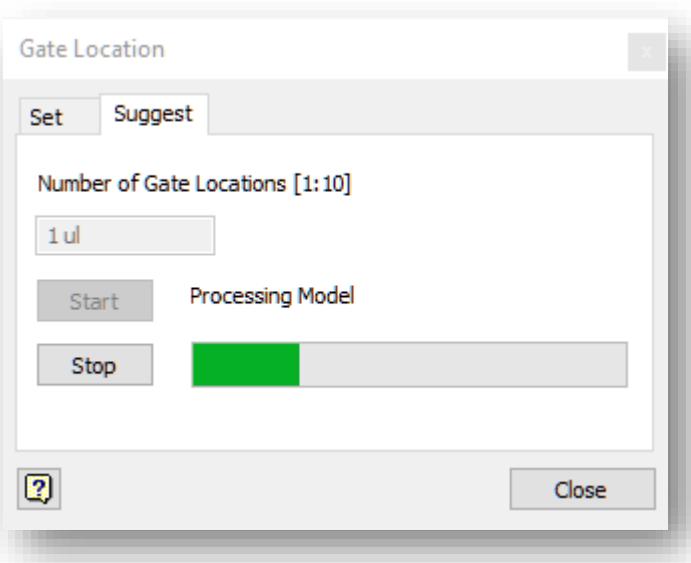


Use Part Fill analysis to look for potential moulding issues like weld lines

Quality Prediction is useful for sink mark areas

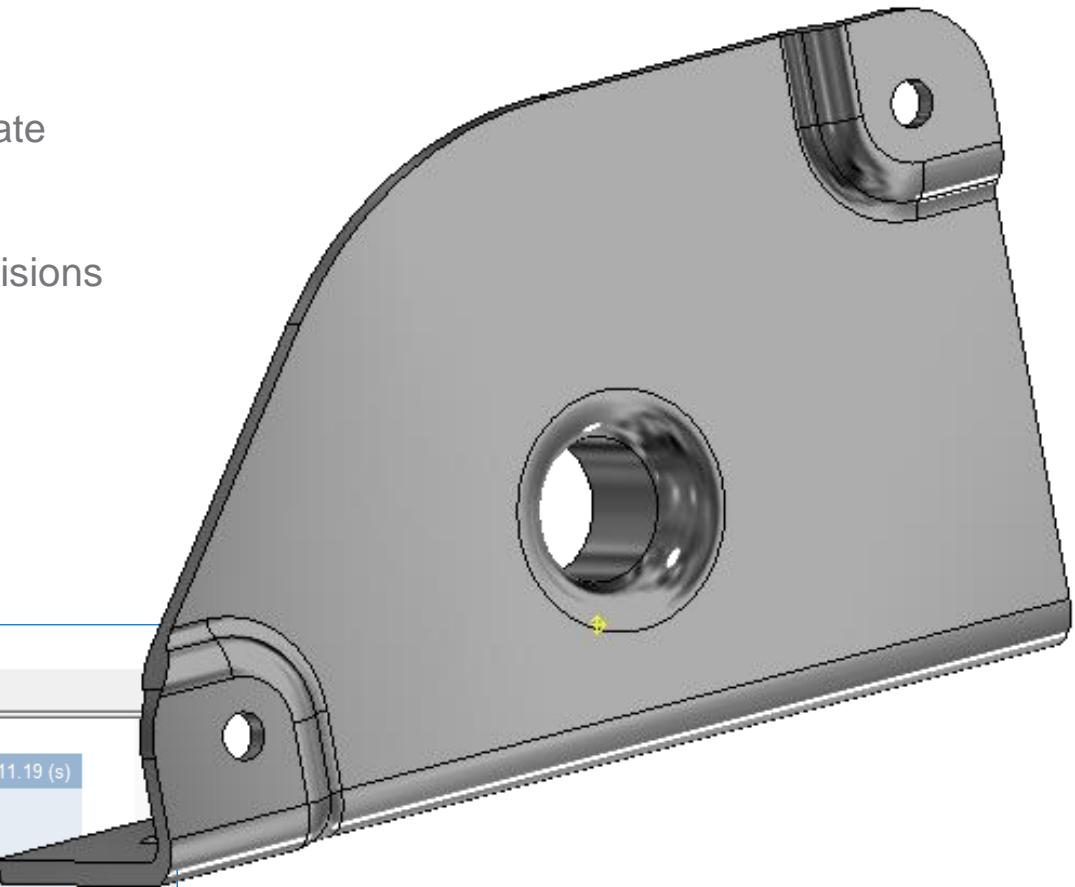
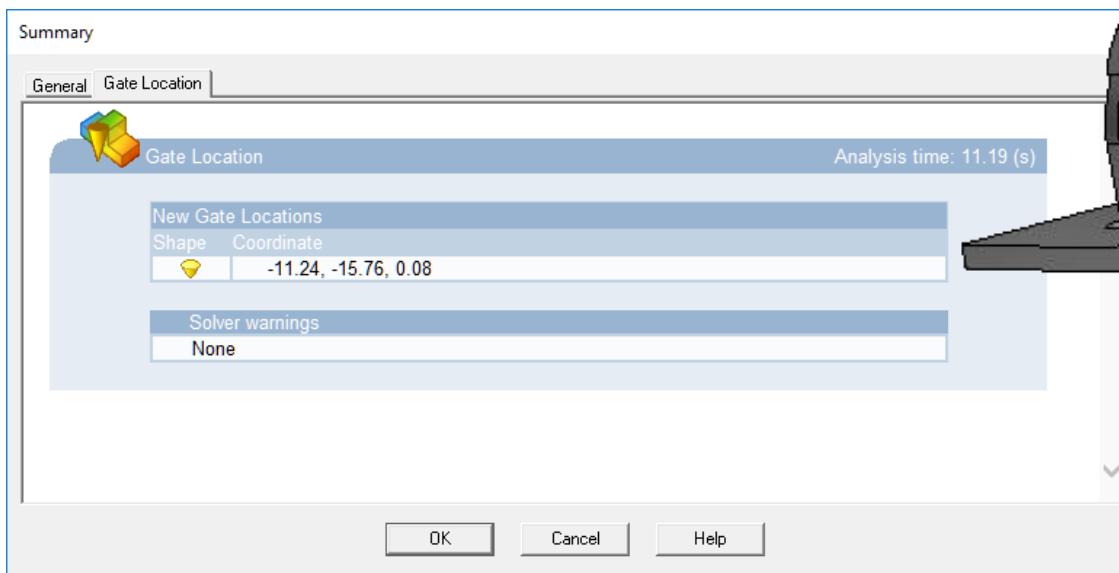
Confidence of fill is also very useful

Do you design Plastic parts? Use the Tooling environment

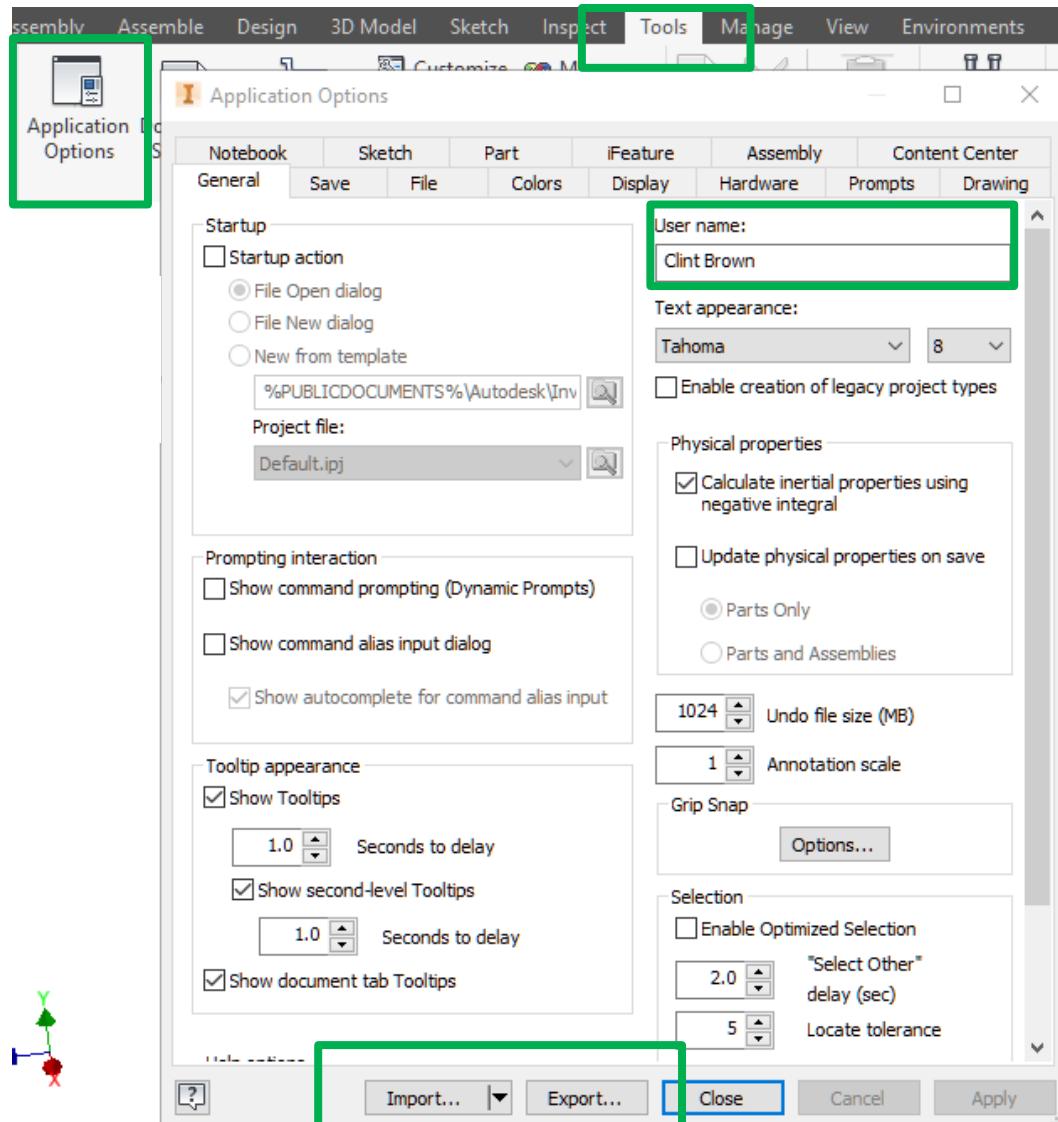


Inventor can suggest the best Gate location for the moulding

This may help inform design decisions

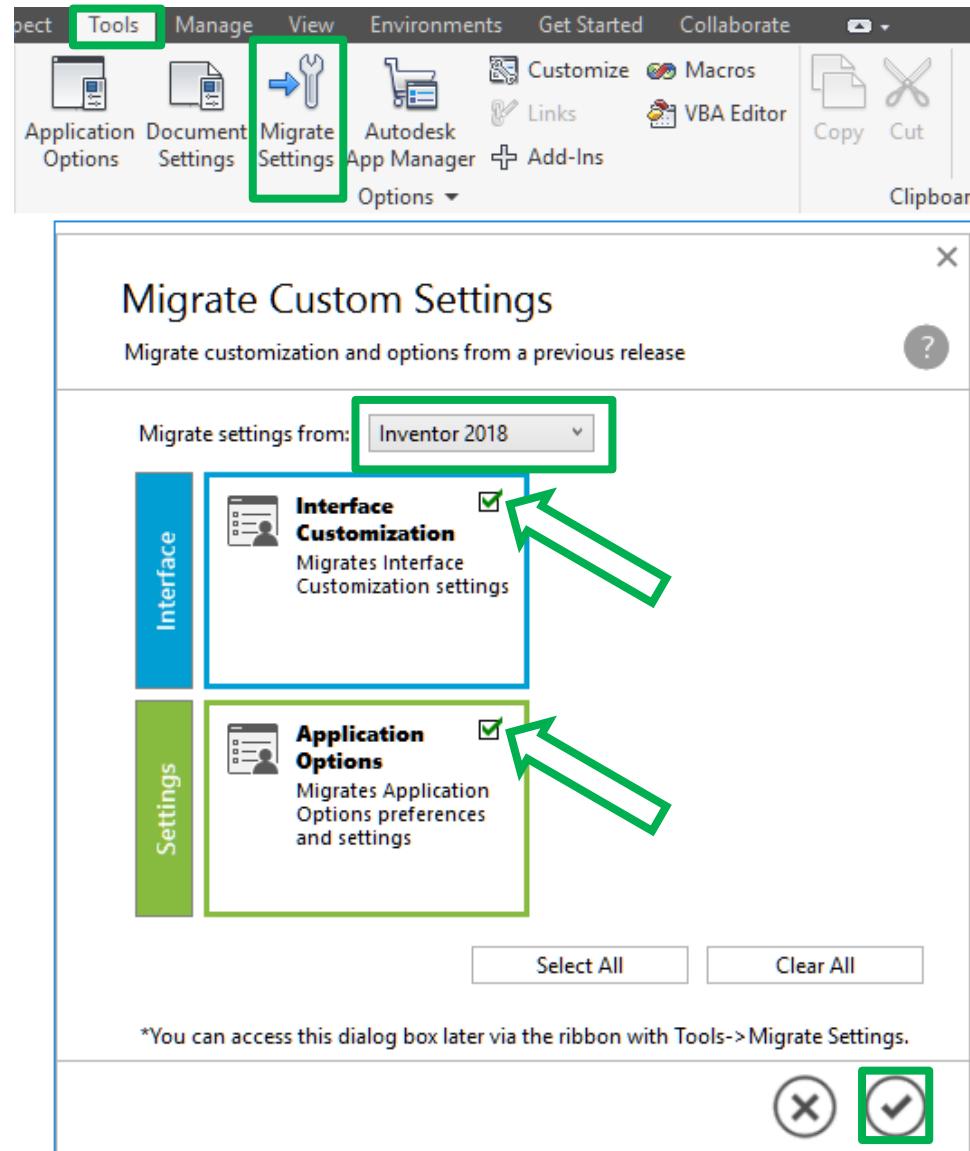


Application Options



Share your Application Options (remember to change your user name!)

Migrate Application Options + UI (2019)



Migrate your “Application Options” and UI customisations from previous version of Inventor

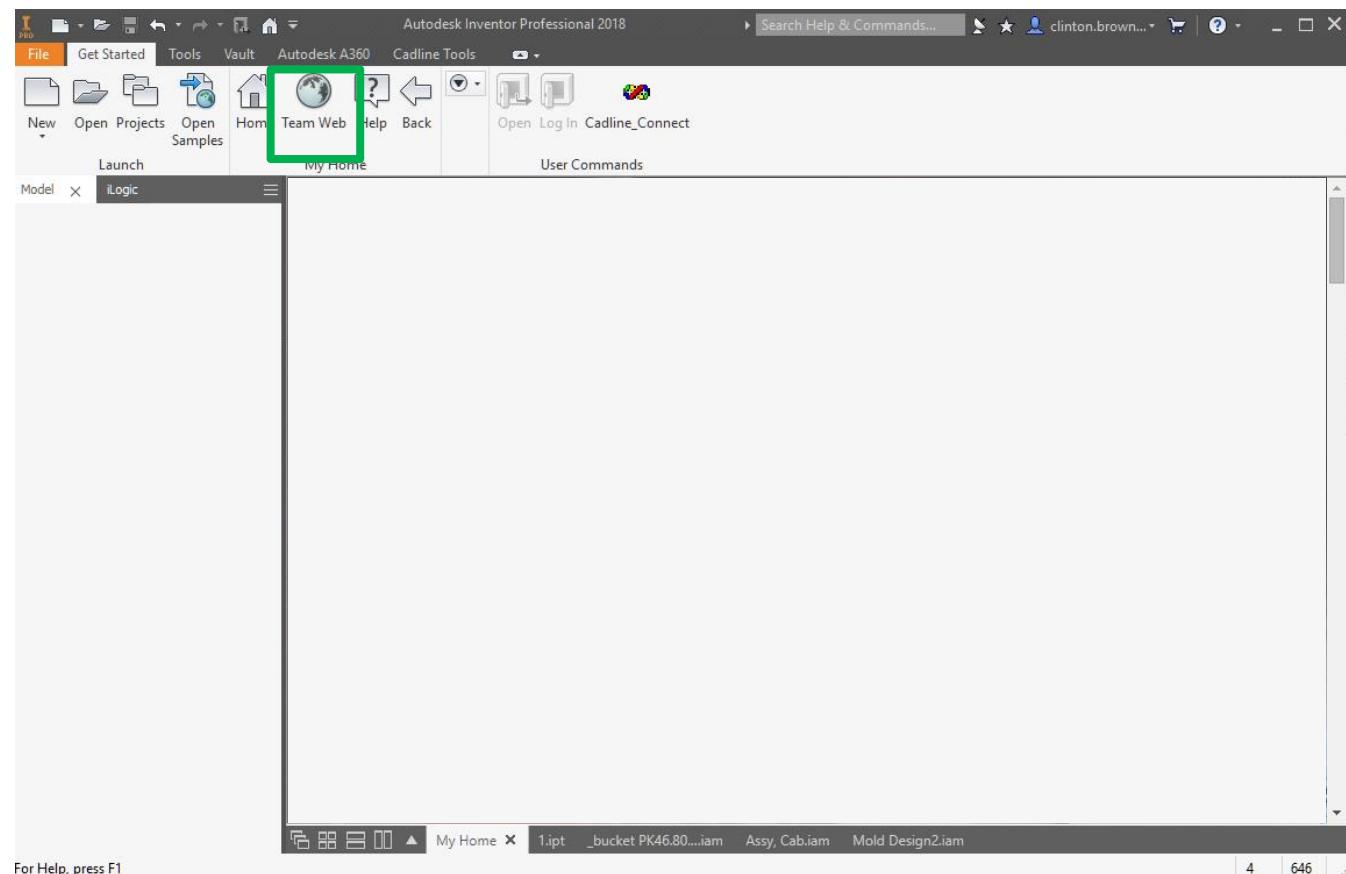
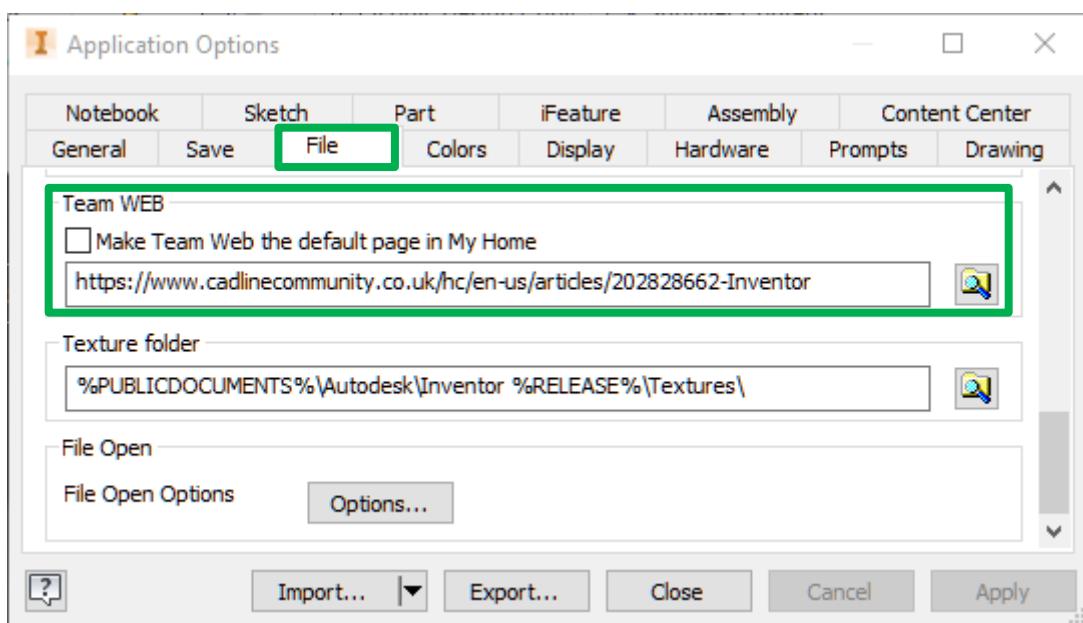
Team Web

In Application Options:

Set team web to something useful, like Cadline Community, or your intranet

Default URL is:

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Web\en-US\CustomHelpSample.htm



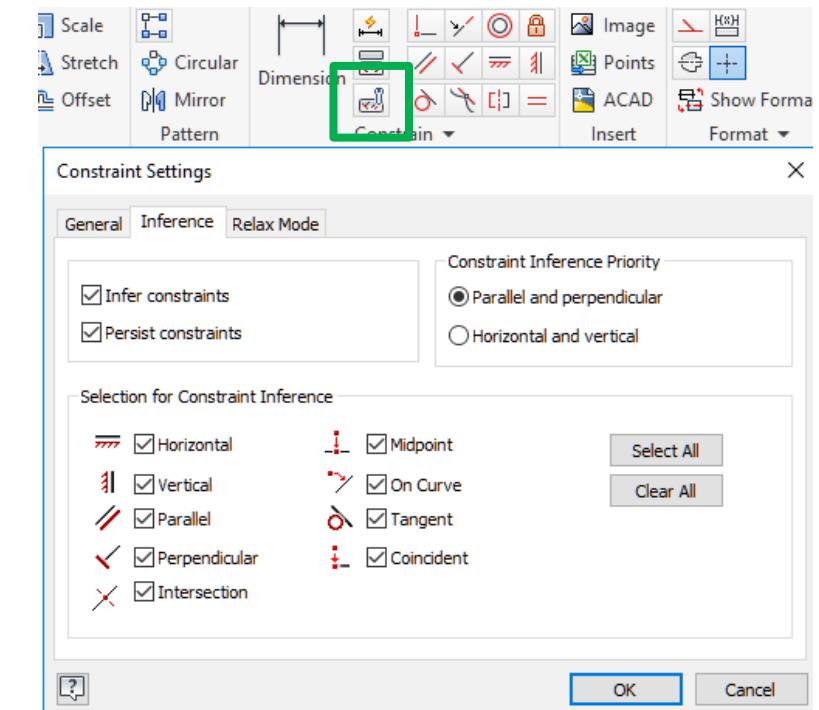
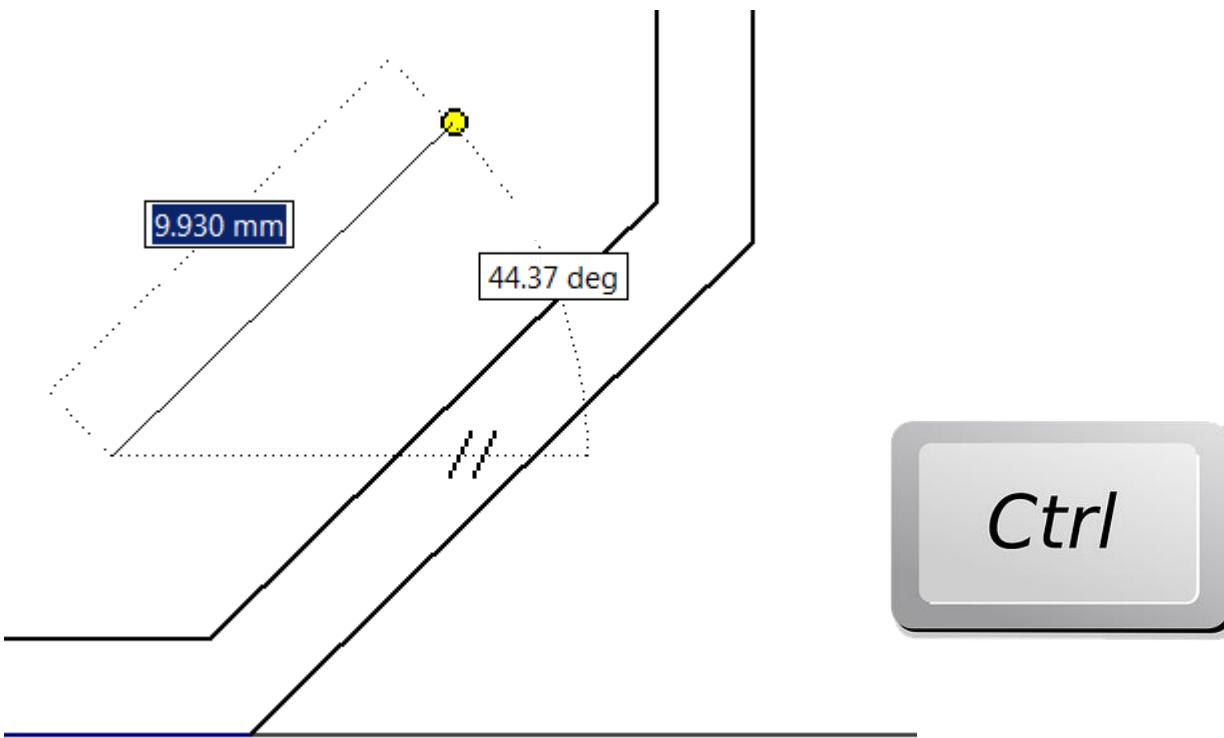
Sketches: Arcs from the Line Command

After drawing your line, move your mouse pointer to the start point of your new line (the end point of your last line). You can then **click and drag** away from that point and create a number of different arcs. Perpendicular or Tangent ones in multiple directions depending on the direction of your drag.

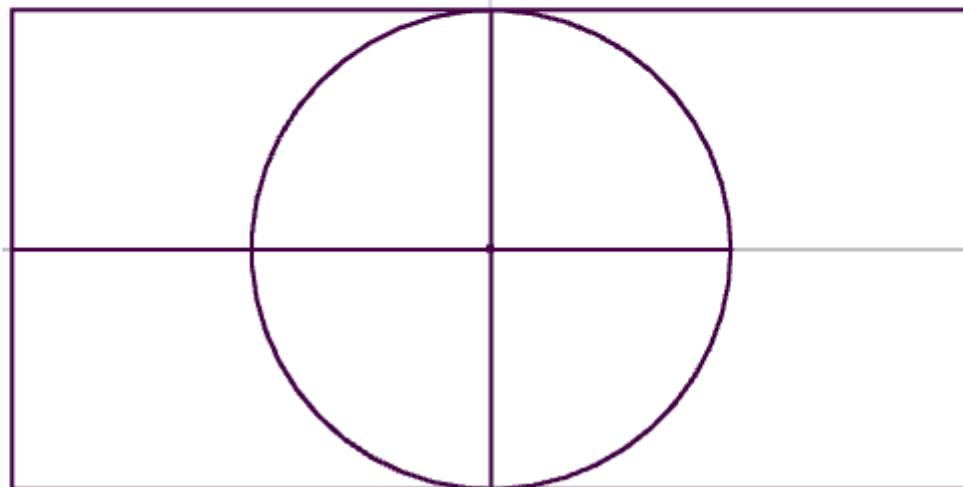
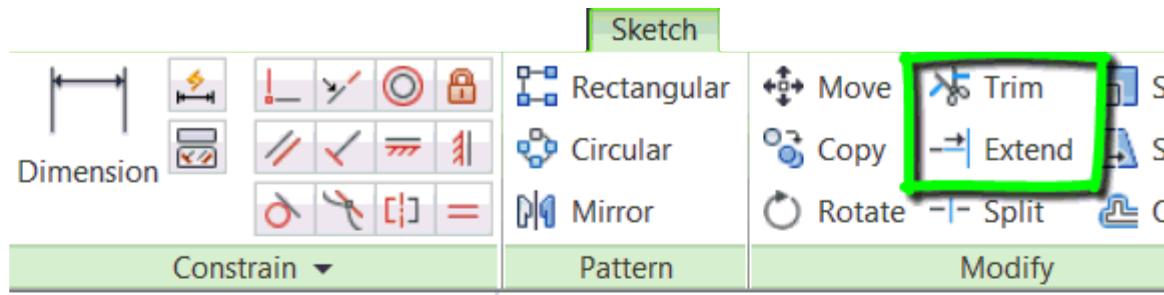


Sketches: Inferred Constraints

Hold the **CTRL** key when drawing in a model sketch to prevent Inventor creating inferred constraints. This ignores your constraint settings



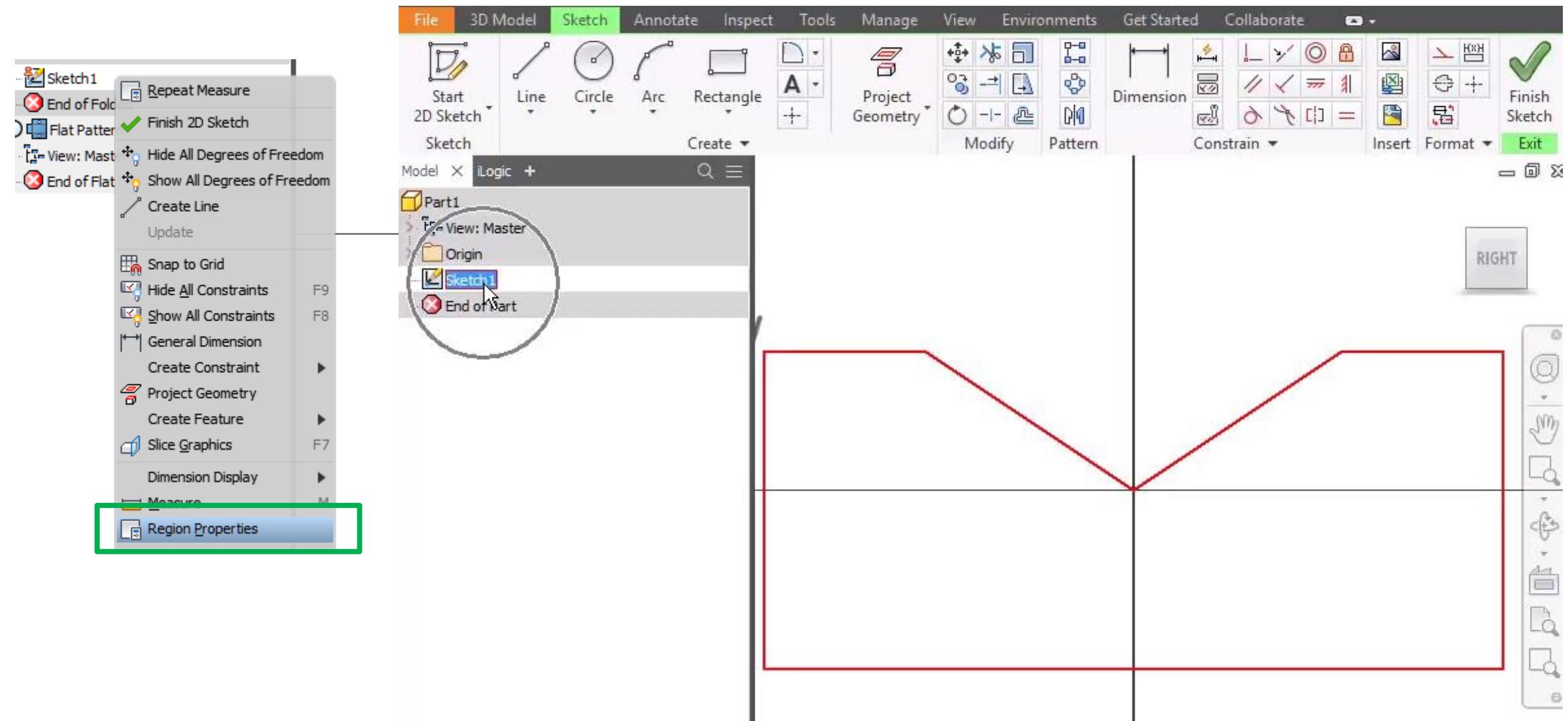
Sketches: Trim and Extend



When in Sketch mode, you can switch between the trim and extend command, by simply holding down the **Shift** key on your keyboard.

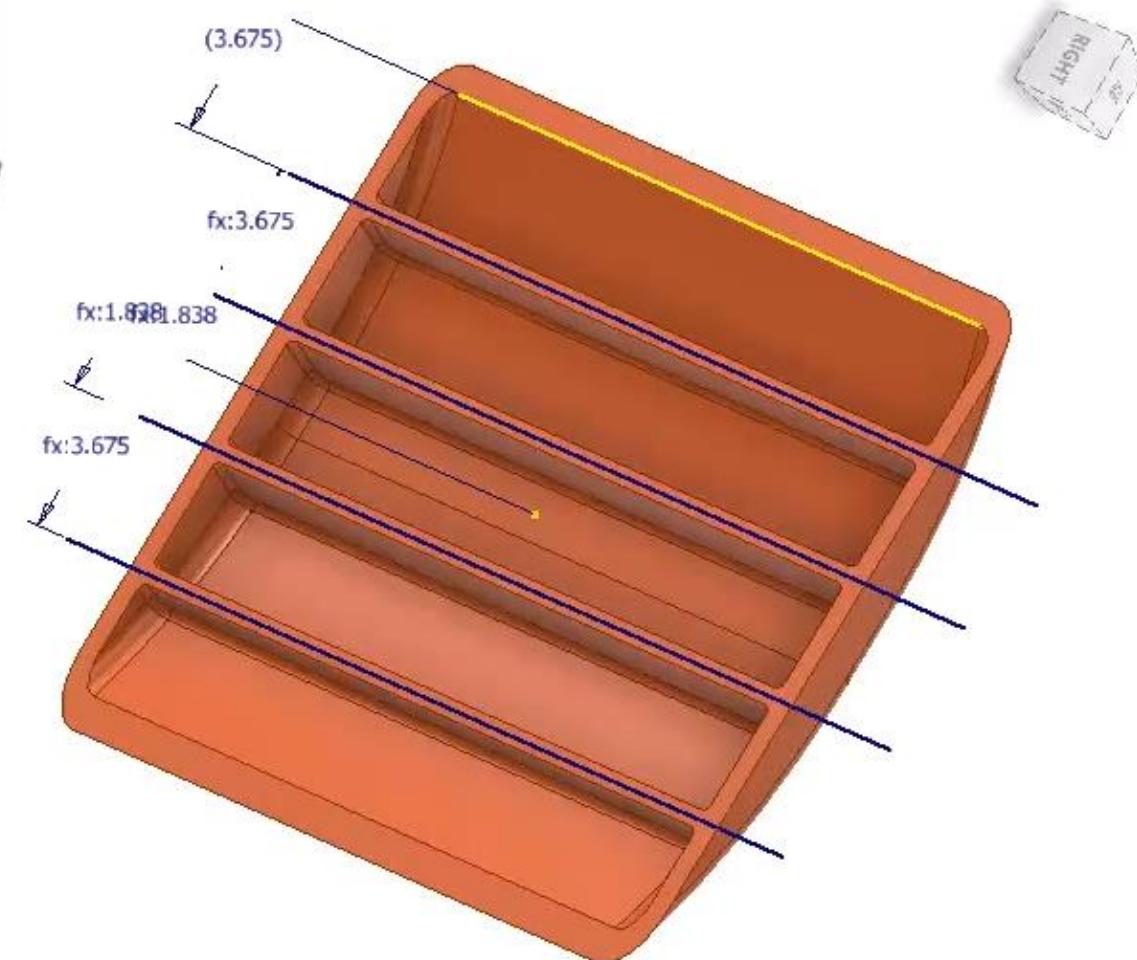
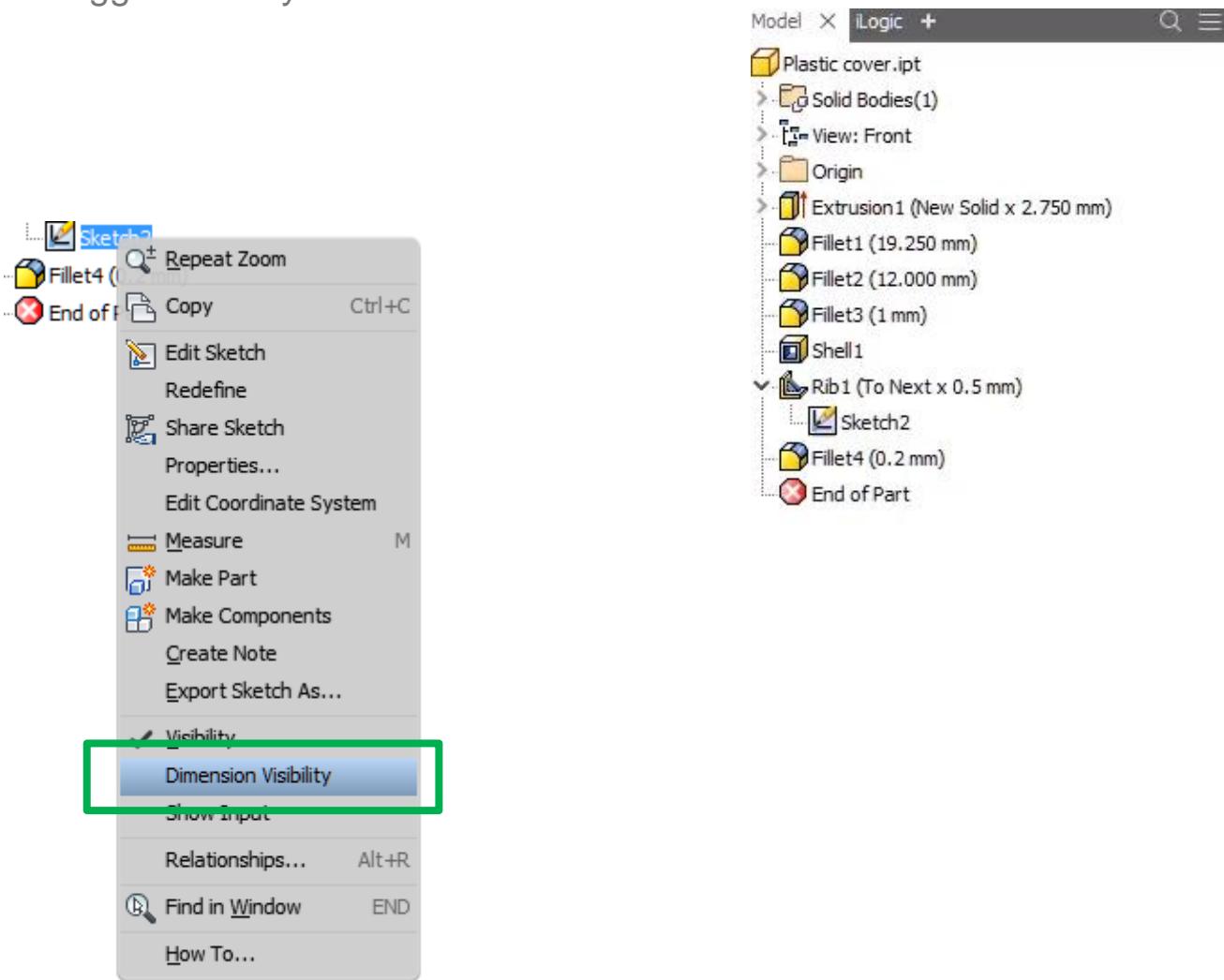


Sketches: Region Properties



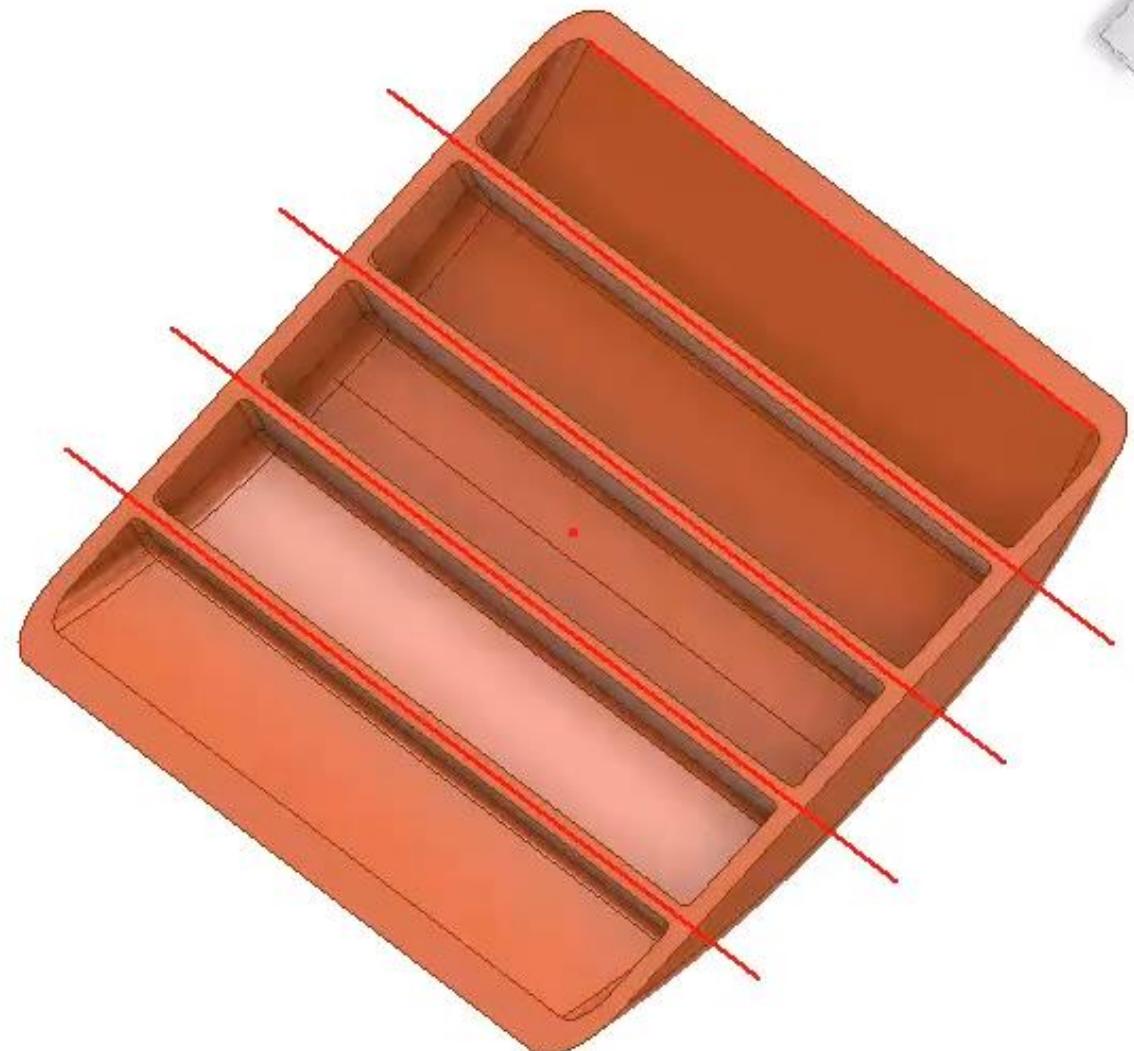
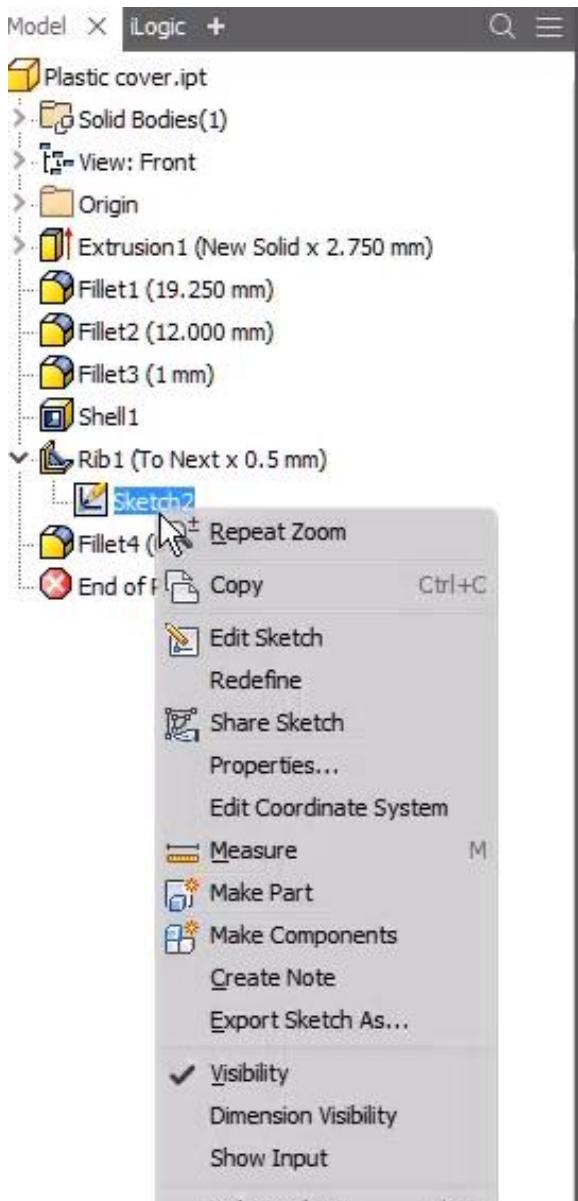
Sketches: Dimension Visibility

Toggle visibility of dimensions on a sketch



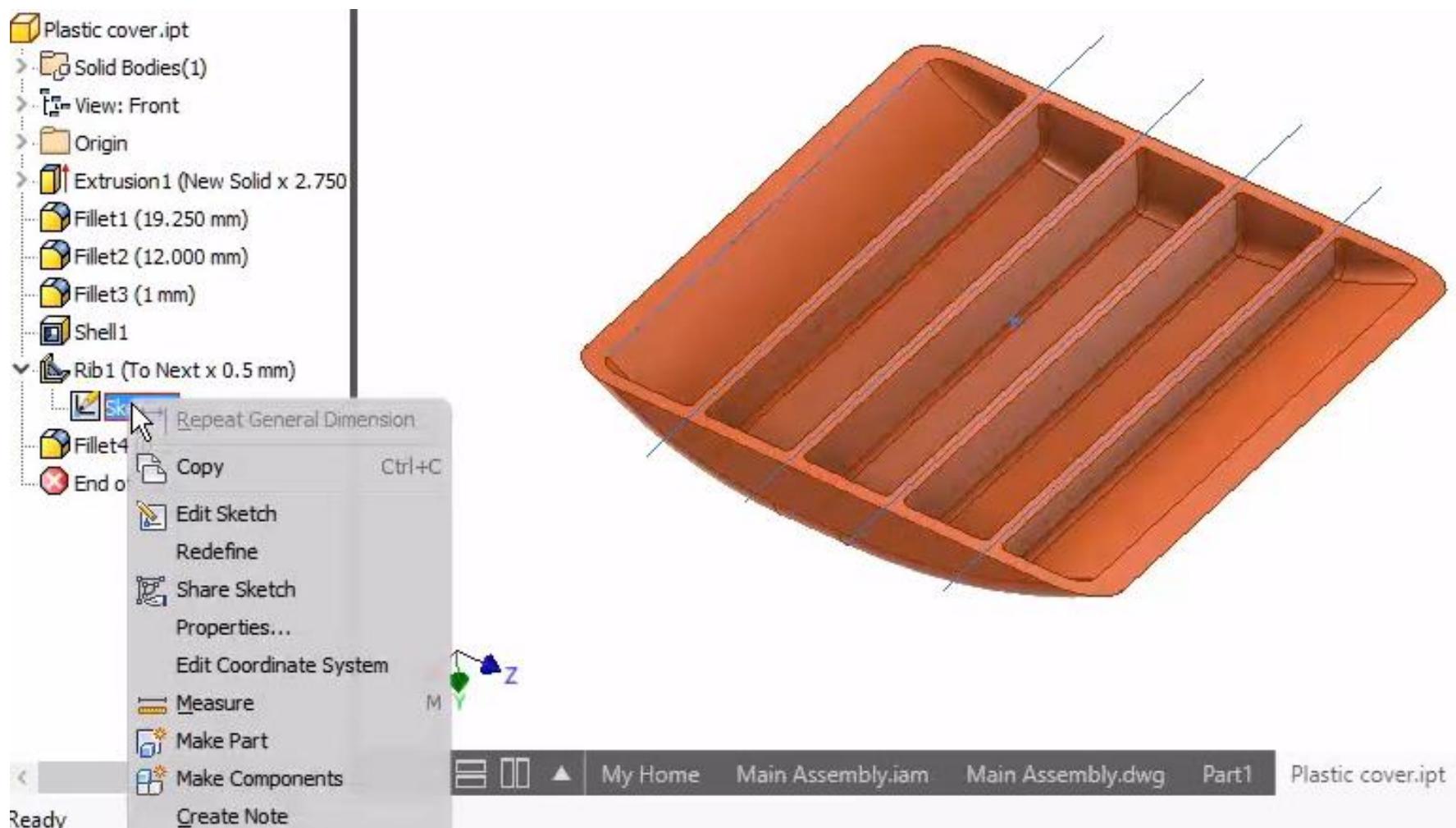
Sketches: Export Sketch to DXF

Any Sketch in Inventor can be exported out to DXF



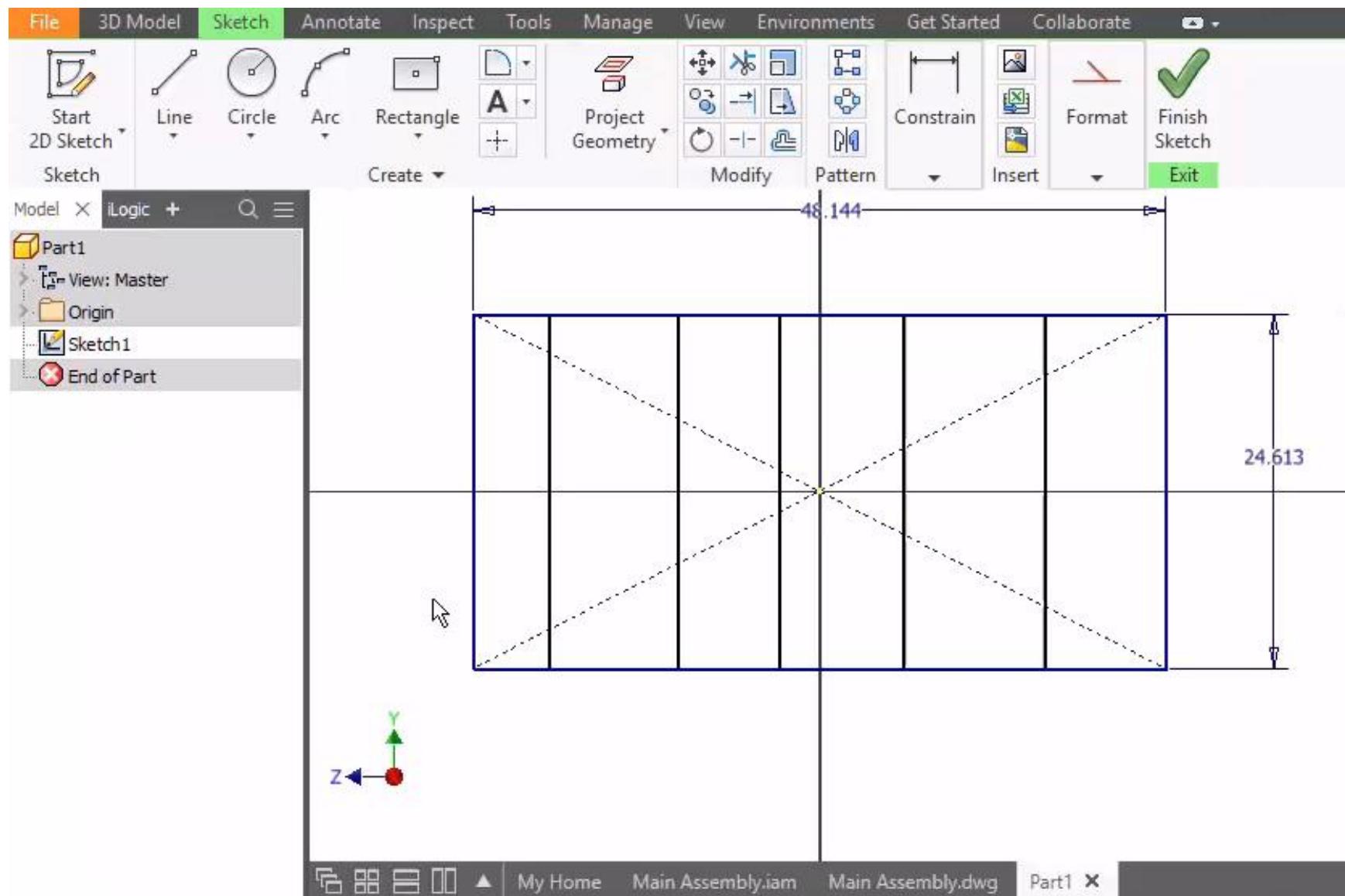
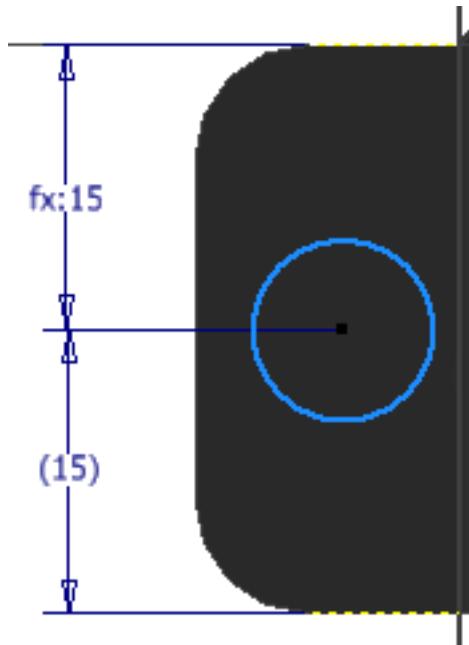
Sketches: Change Display Colour + Linetype

Adjust the appearance of sketches from "Properties" on the right-click menu

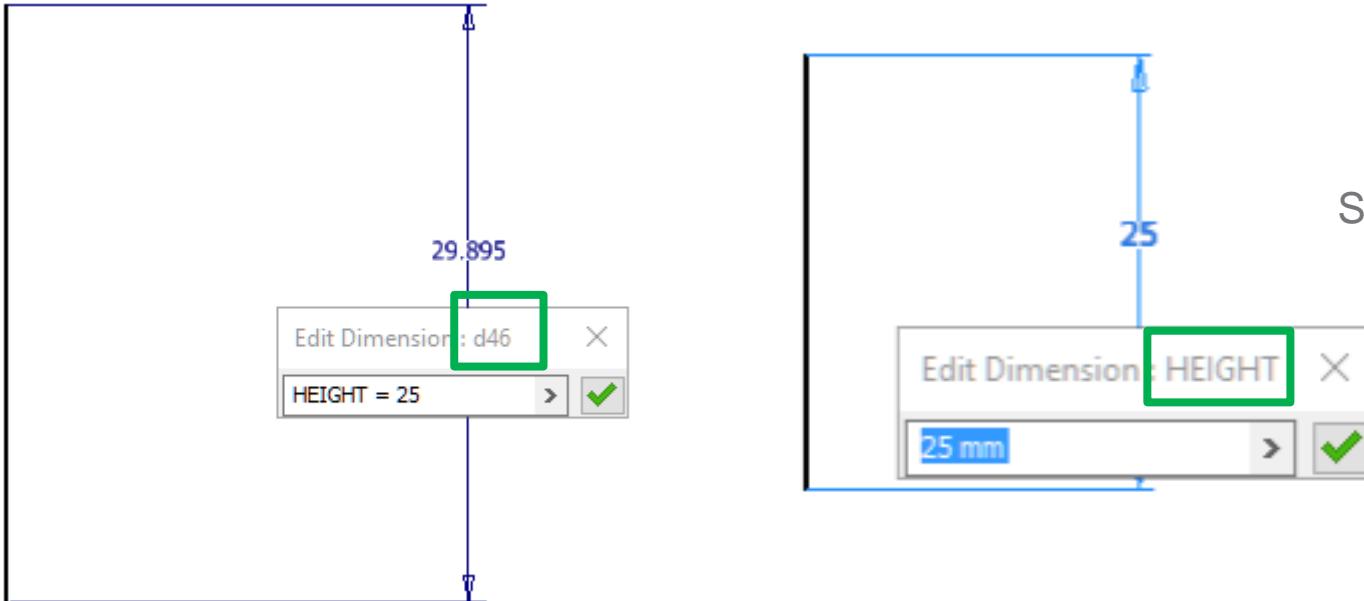


Sketches: Equal Spacing

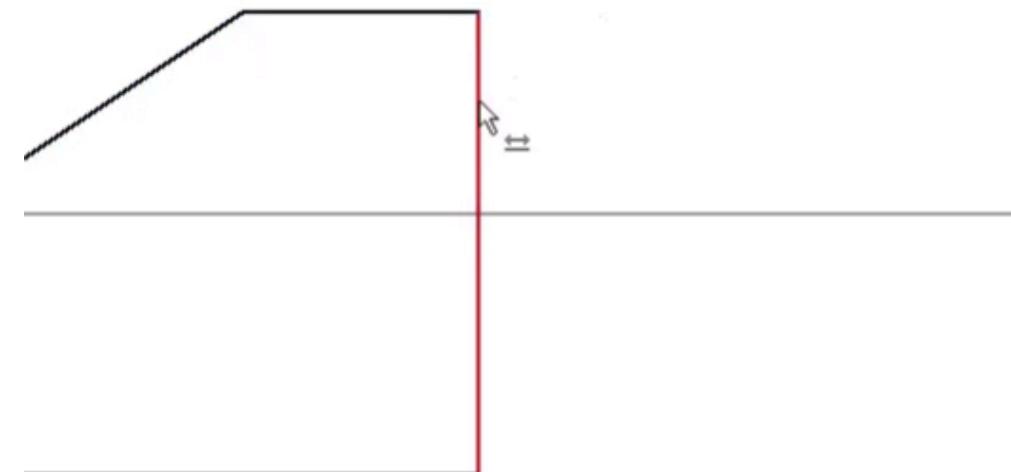
Use a driven dimension as a reference to space geometry evenly



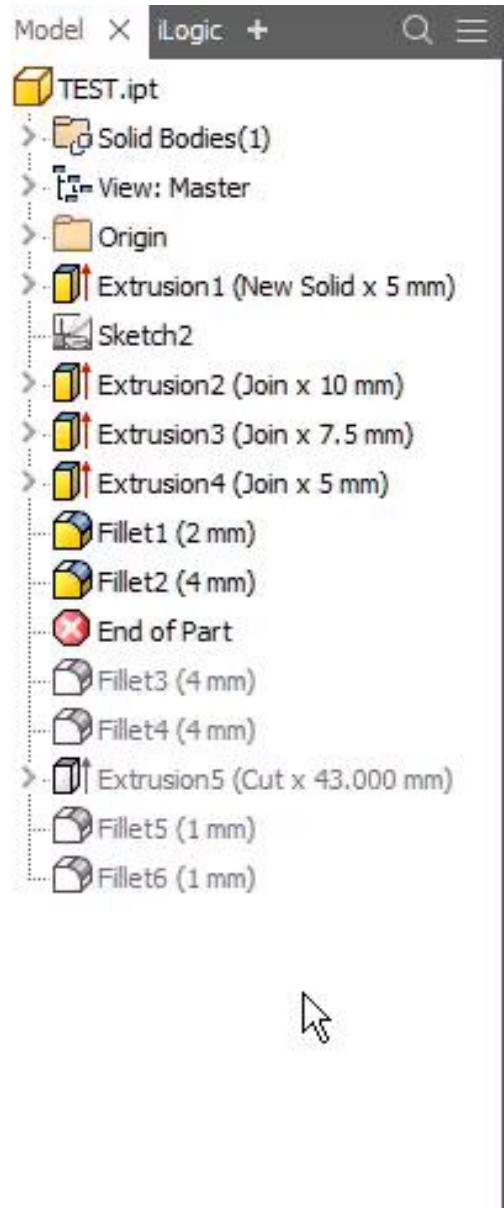
Parts/Sketches: Parameter naming on the fly



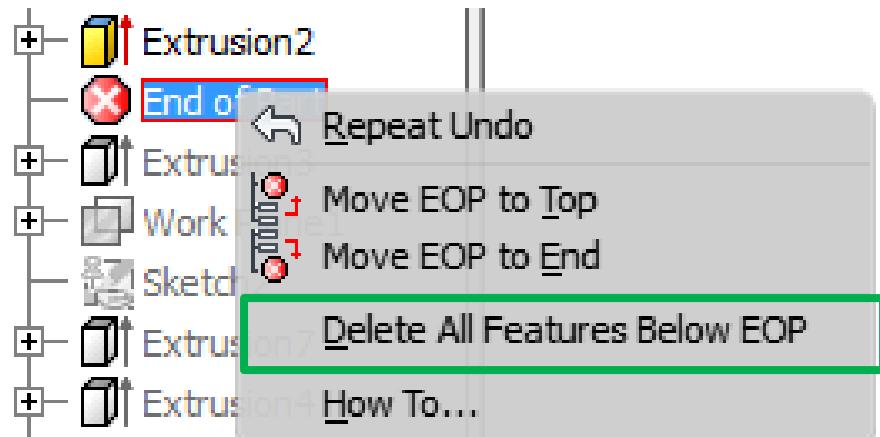
Save time by naming your parameters as you create them



Parts: EOP – Delete Features below the EOP



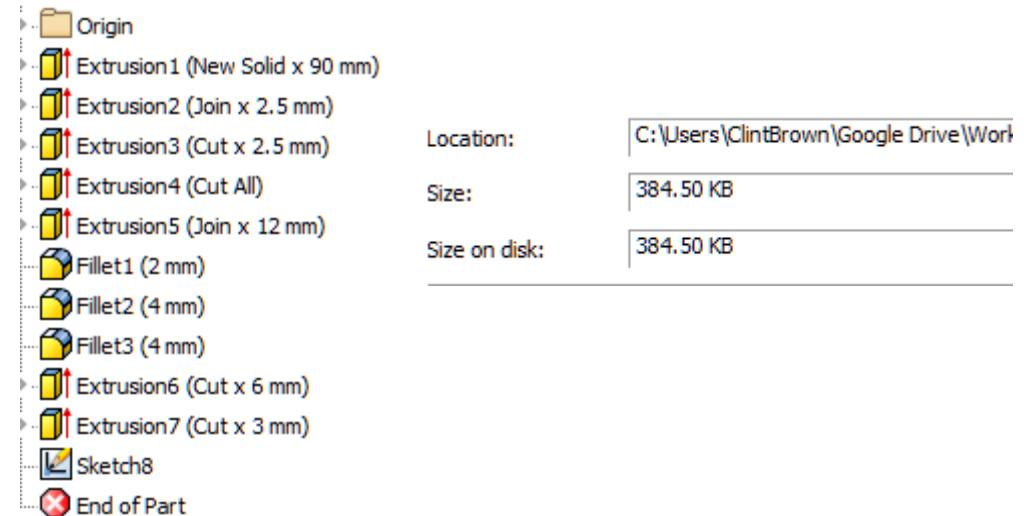
All features below the EOP can be deleted by simply right clicking on the EOP and selecting “Delete All Features Below EOP”



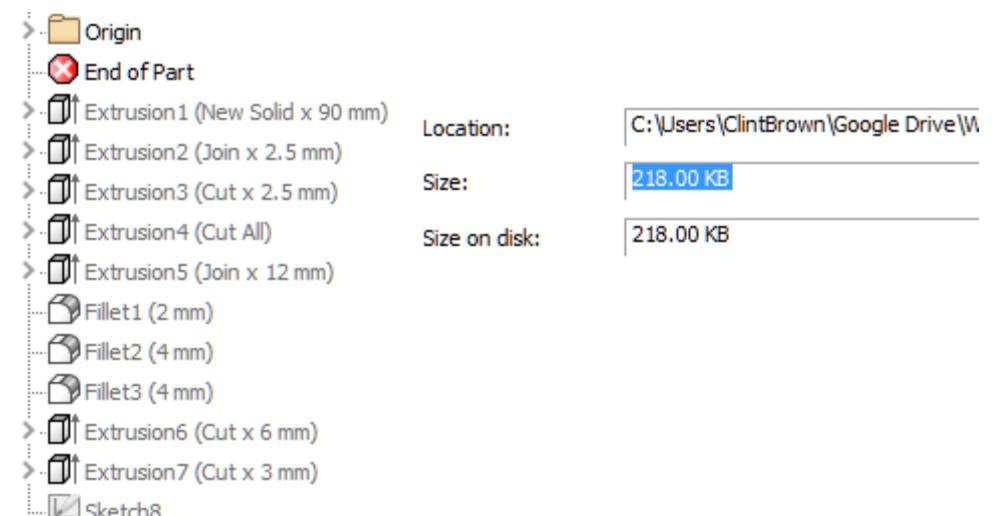
Parts: EOP – File Size & Emails

Rolling up your end of part makes Inventor files smaller

iLogic rule in the link below automates rolling up EOP and attaching to email



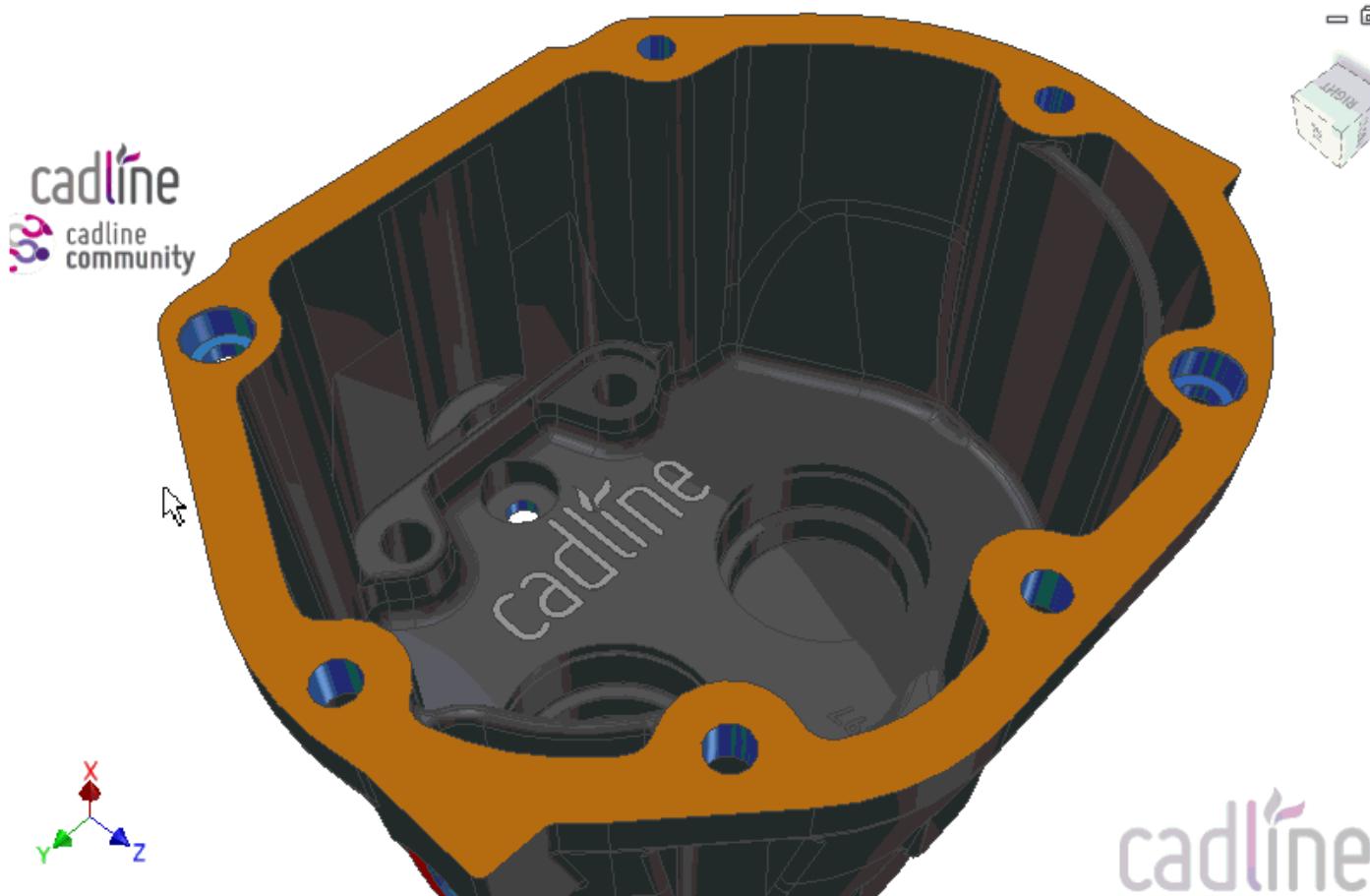
Origin		
Extrusion1 (New Solid x 90 mm)	Location:	C:\Users\ClintBrown\Google Drive\Work\
Extrusion2 (Join x 2.5 mm)	Size:	384.50 KB
Extrusion3 (Cut x 2.5 mm)	Size on disk:	384.50 KB
Extrusion4 (Cut All)		
Extrusion5 (Join x 12 mm)		
Fillet1 (2 mm)		
Fillet2 (4 mm)		
Fillet3 (4 mm)		
Extrusion6 (Cut x 6 mm)		
Extrusion7 (Cut x 3 mm)		
Sketch8		
End of Part		



Origin		
End of Part	Location:	C:\Users\ClintBrown\Google Drive\Work\
Extrusion1 (New Solid x 90 mm)	Size:	218.00 KB
Extrusion2 (Join x 2.5 mm)	Size on disk:	218.00 KB
Extrusion3 (Cut x 2.5 mm)		
Extrusion4 (Cut All)		
Extrusion5 (Join x 12 mm)		
Fillet1 (2 mm)		
Fillet2 (4 mm)		
Fillet3 (4 mm)		
Extrusion6 (Cut x 6 mm)		
Extrusion7 (Cut x 3 mm)		
Sketch8		

Parts: Flat Faces to DXF

Any Flat face in an Inventor part file
can be exported to DXF with a
simple right click

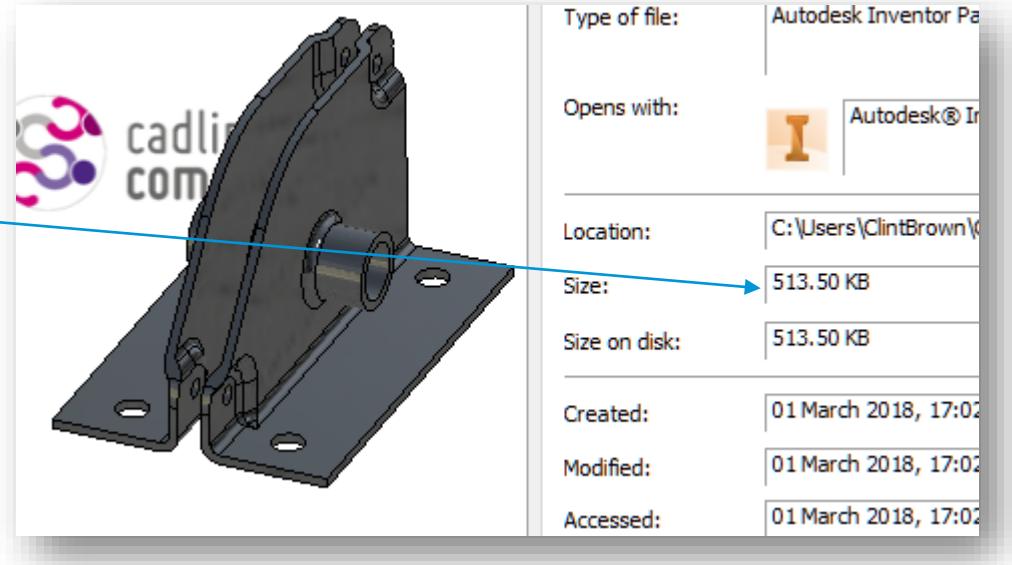
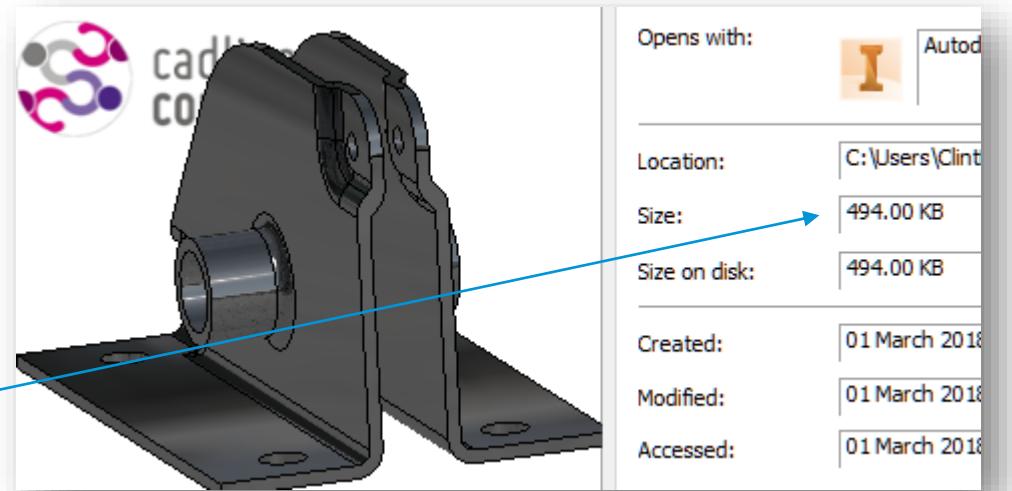
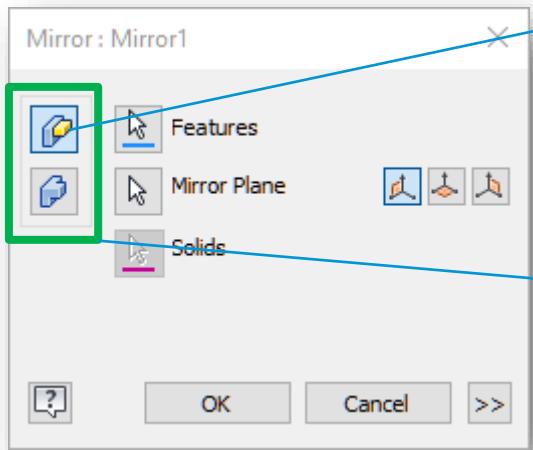


cadline

Parts: Mirror Solids

Mirror Solids instead of features generally compute faster, also have less errors

Sometimes with bigger file size



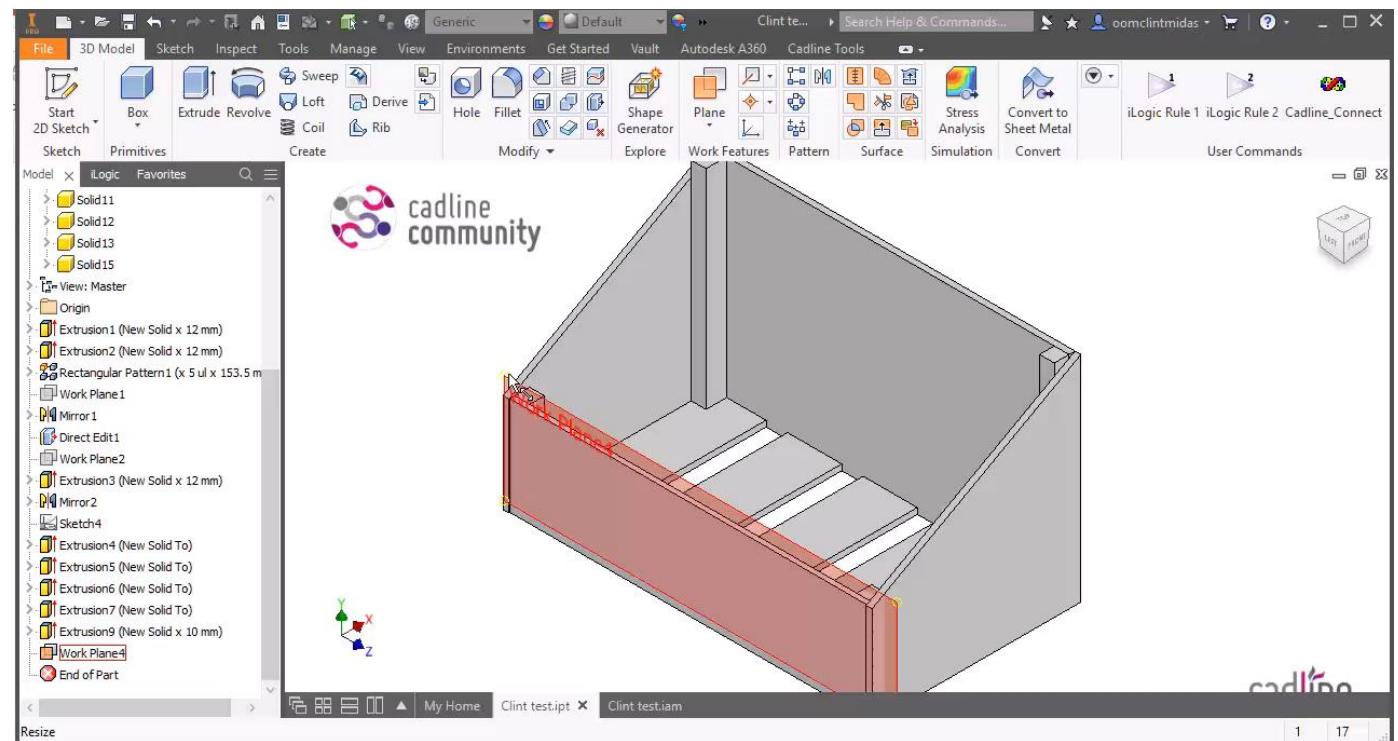
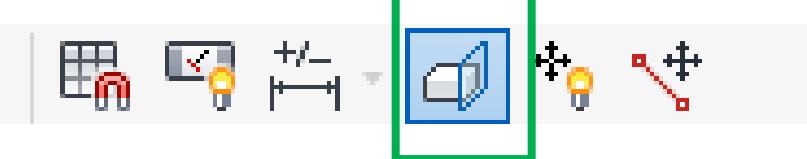
Parts: Slice Graphics - Sketches

Use Slice Graphics to see a temporary section through your model whilst sketching

Keyboard shortcut is **F7**

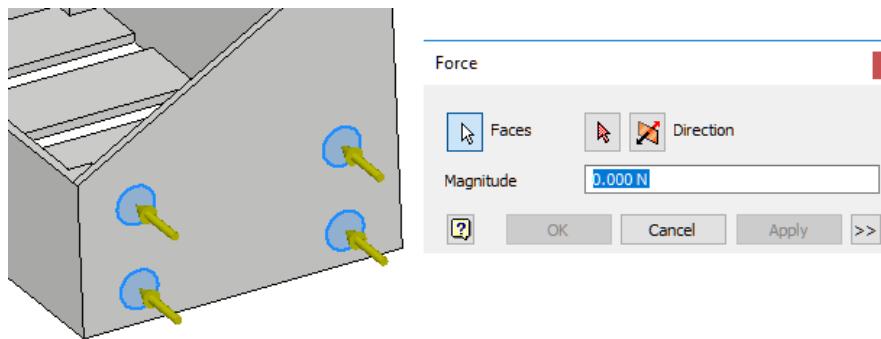
Inventor now has a neat toggle in the sketch environment

Bonus Tip: use “Project Cut Edges” to project edges into your sketch

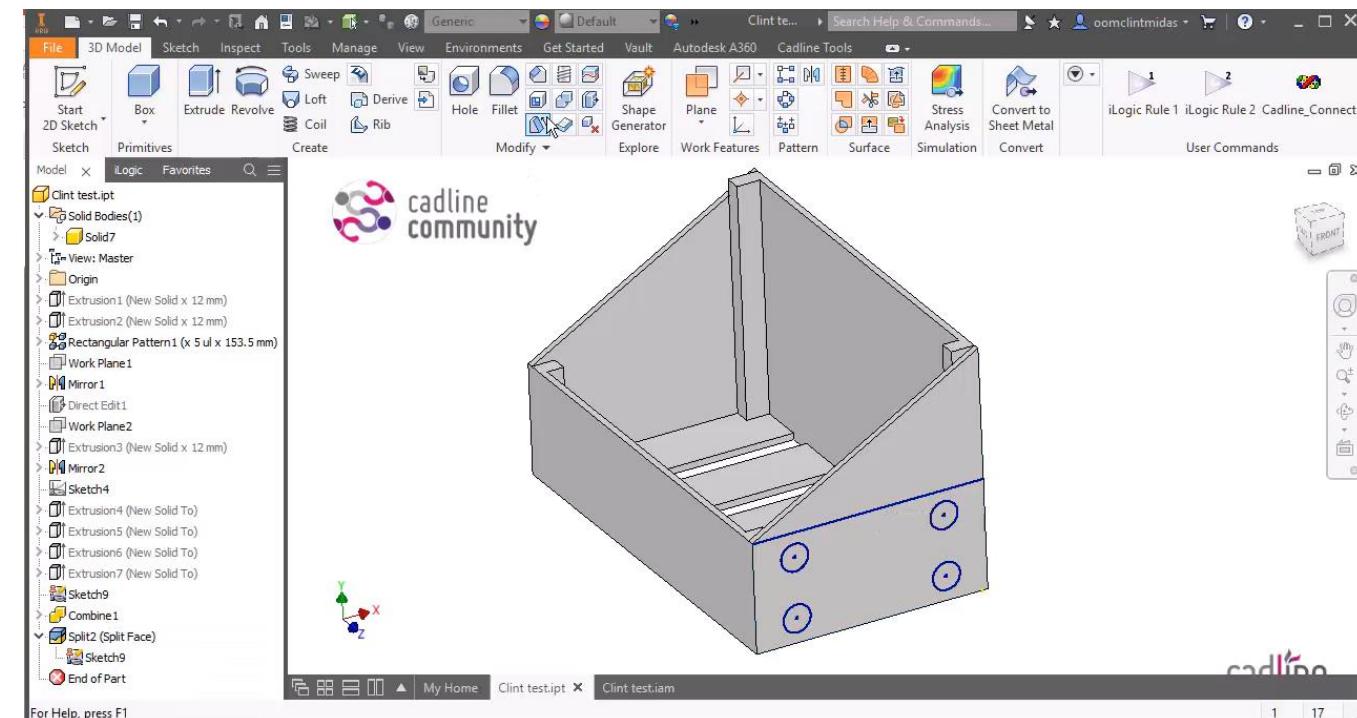
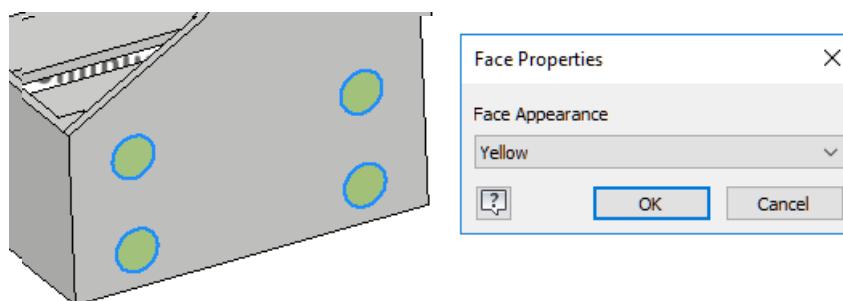


Parts: Split Command – Split faces

Use Split faces for FEA, define specific load/boundary conditions, use parametric sketches to adjust them



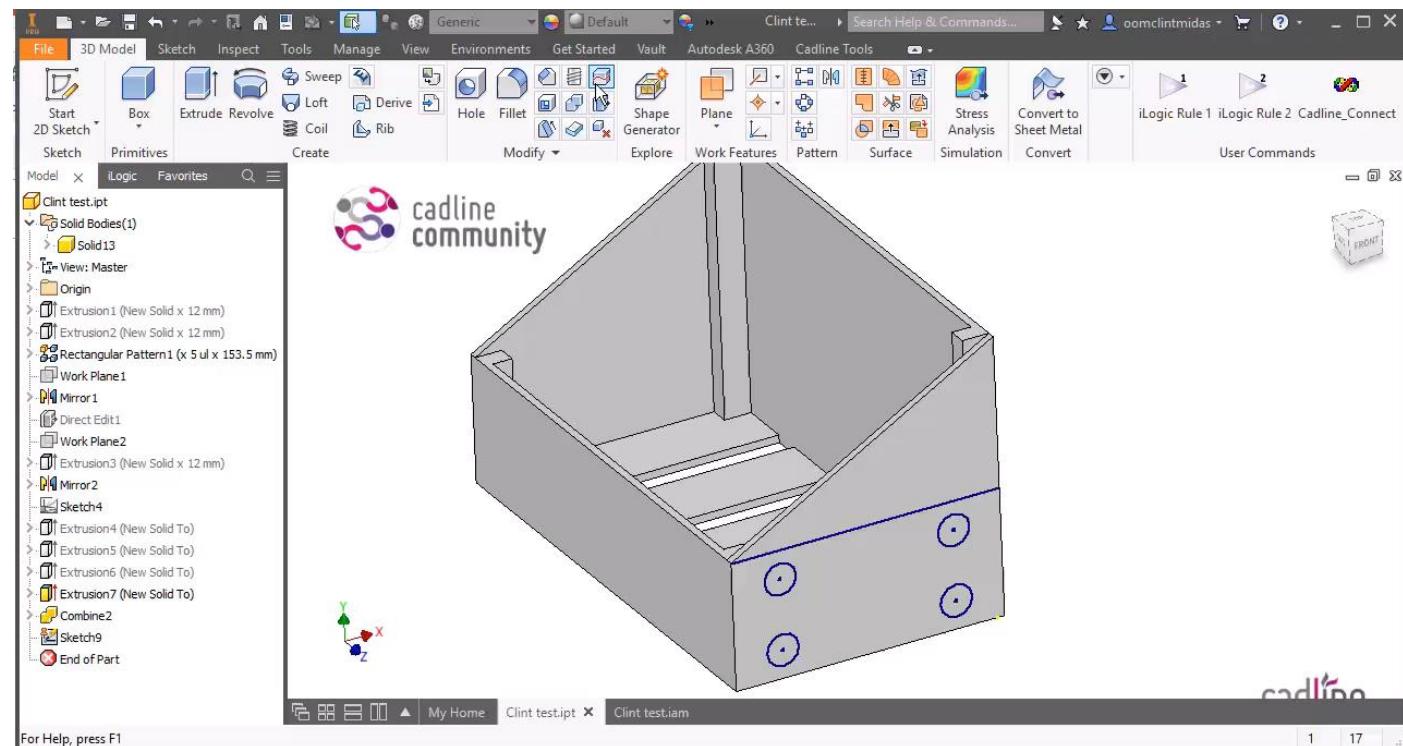
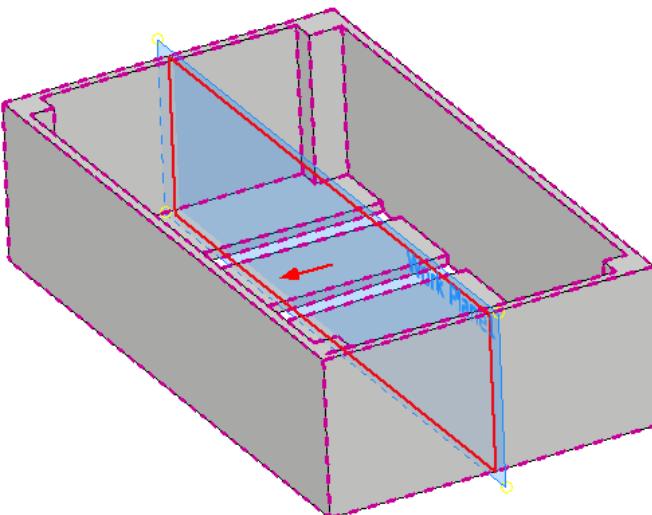
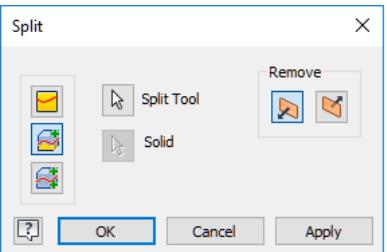
Use Split faces for demarcating different colours on a face.



Parts: Split Command – Trim Solid

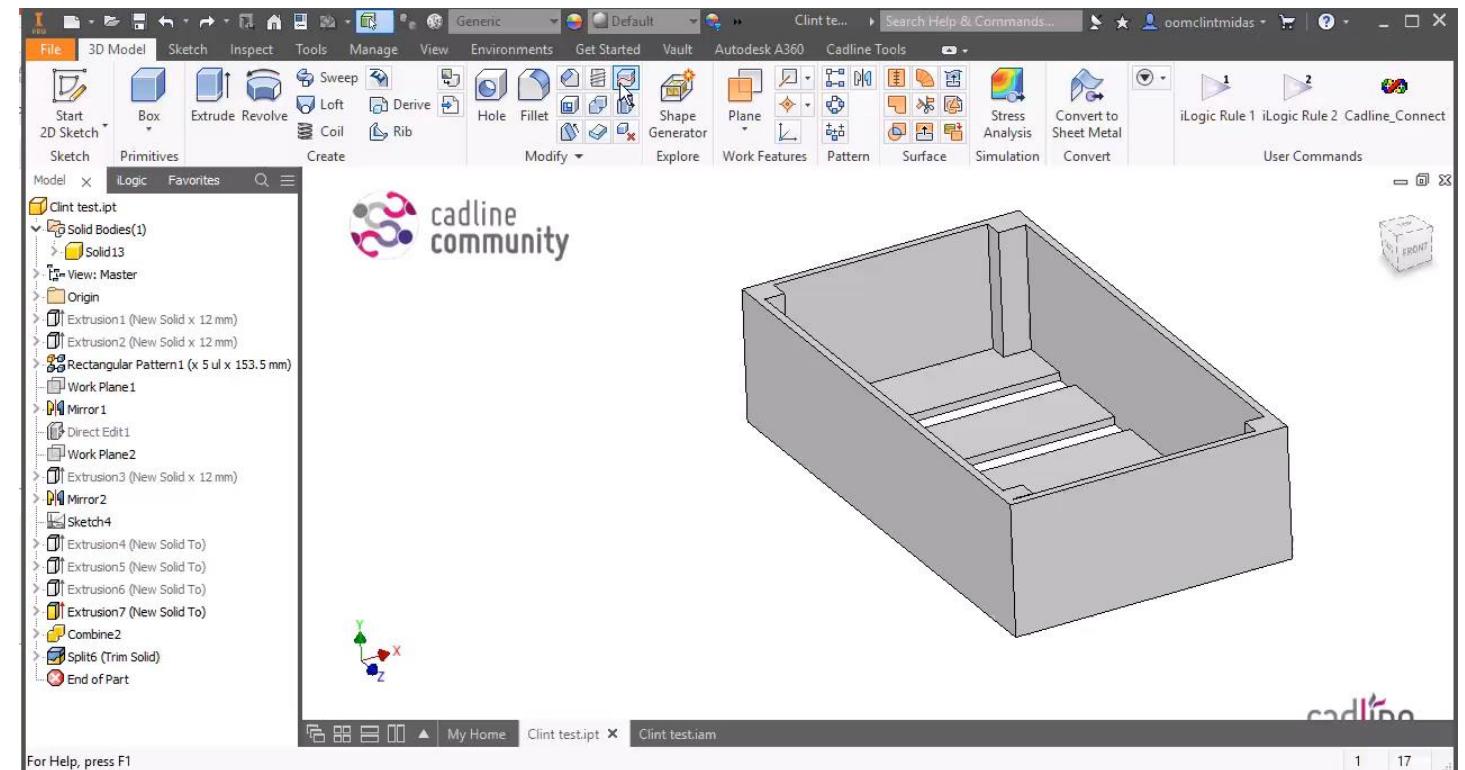
Use “Trim Solid” to cut parts to size

Bonus Tip: Use work planes as the Split Tool



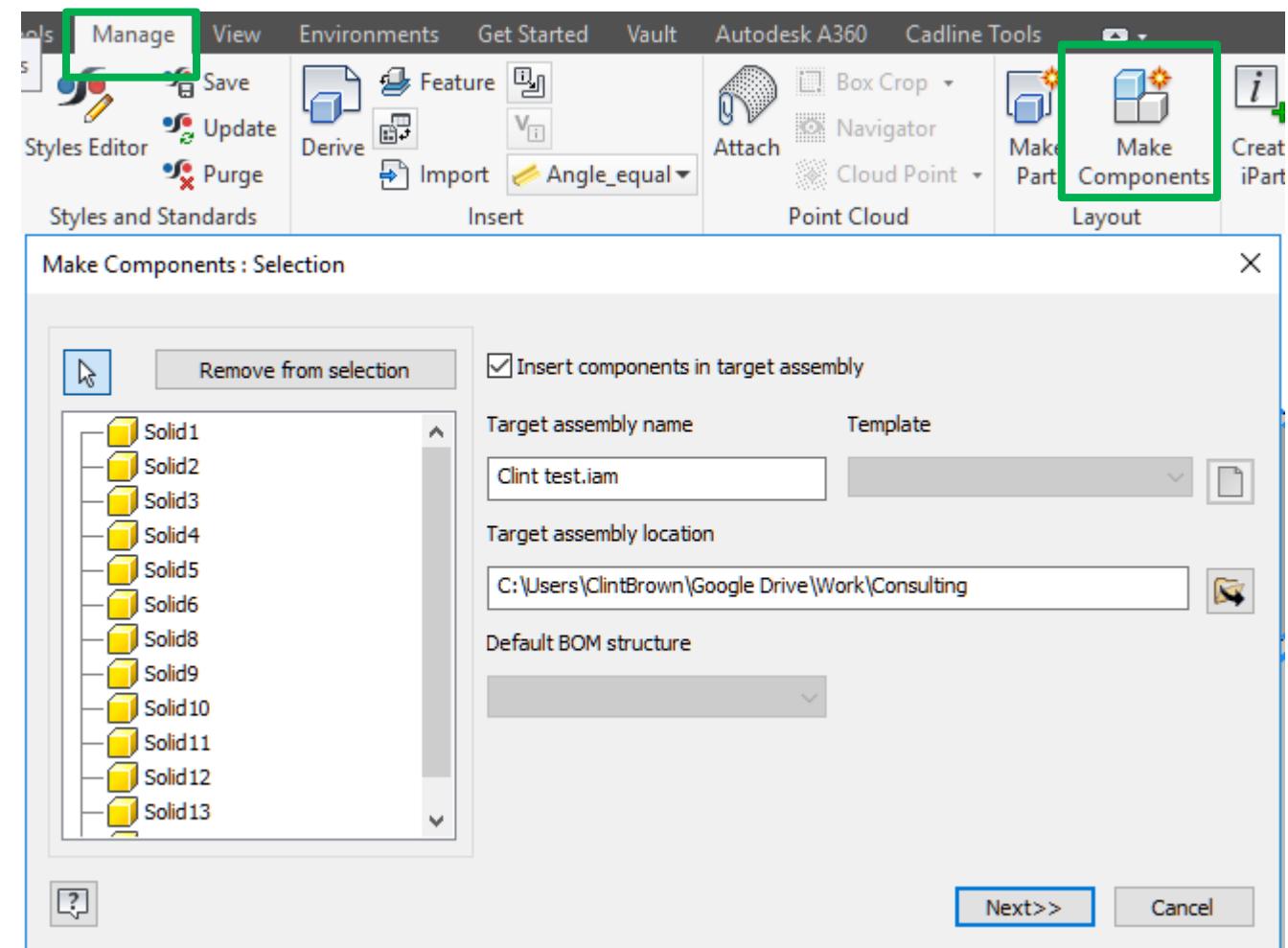
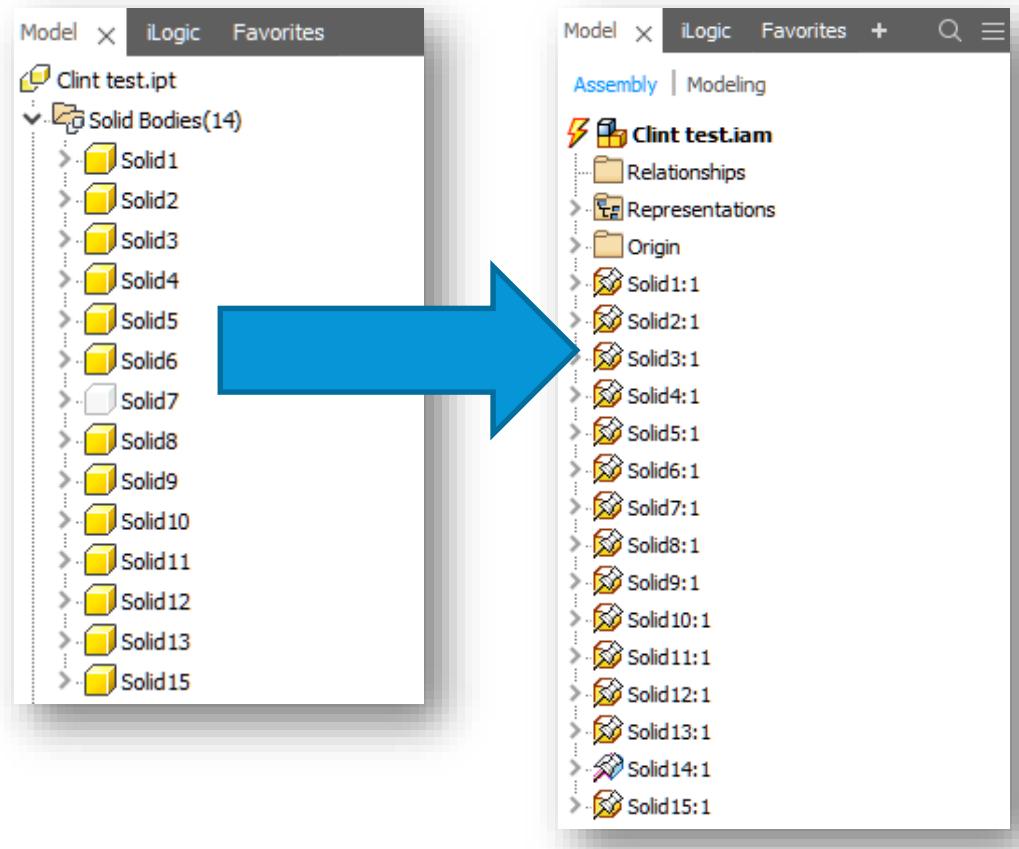
Parts: Split Command – Split Solid

Use “Split Solid” to split your model in separate solids.



Parts: Multi-body solids & Make Components

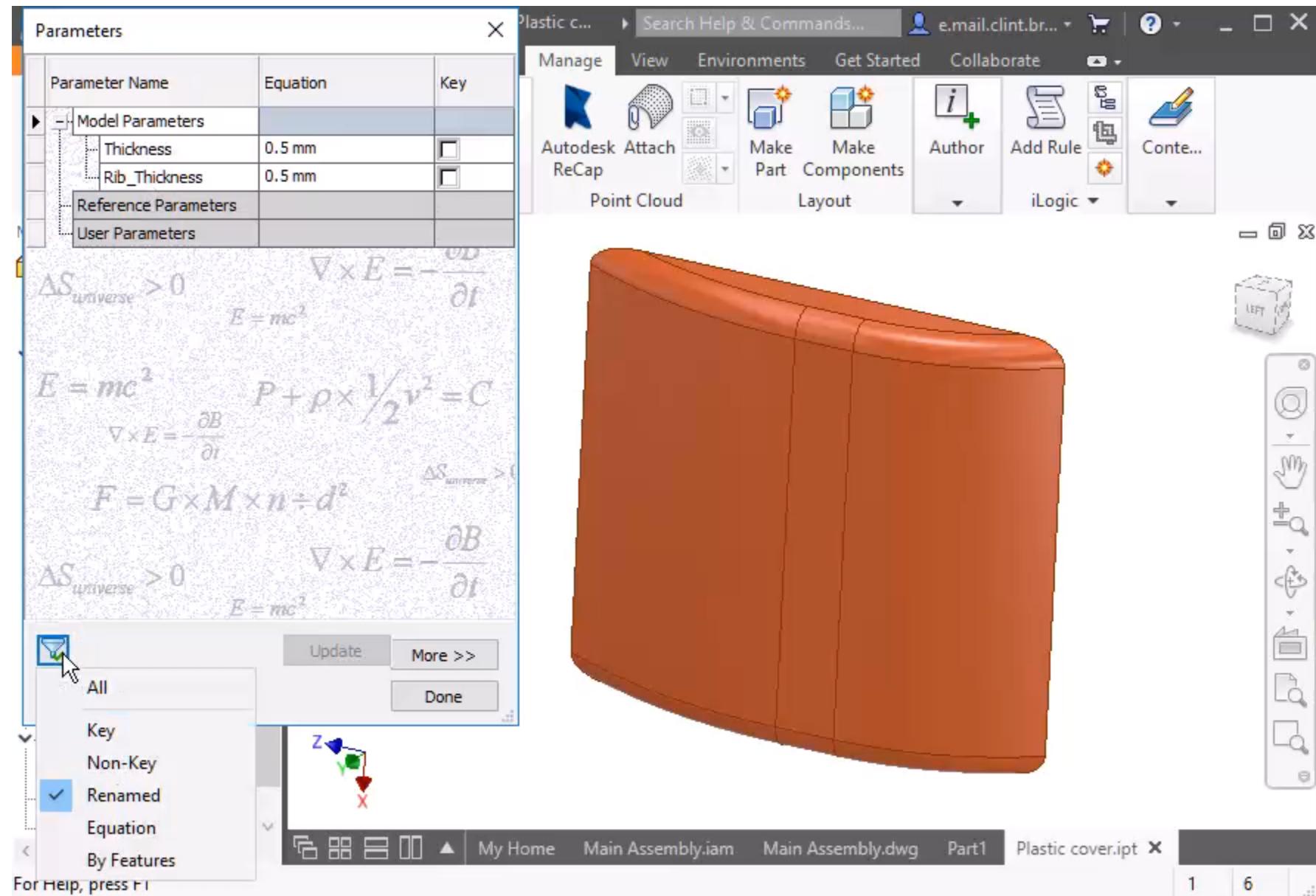
Convert Multi-Body Solids to Assemblies using the Make Components command



Parts: Parameter Filters

Use the Parameter filter to see all “key” and all renamed parameters

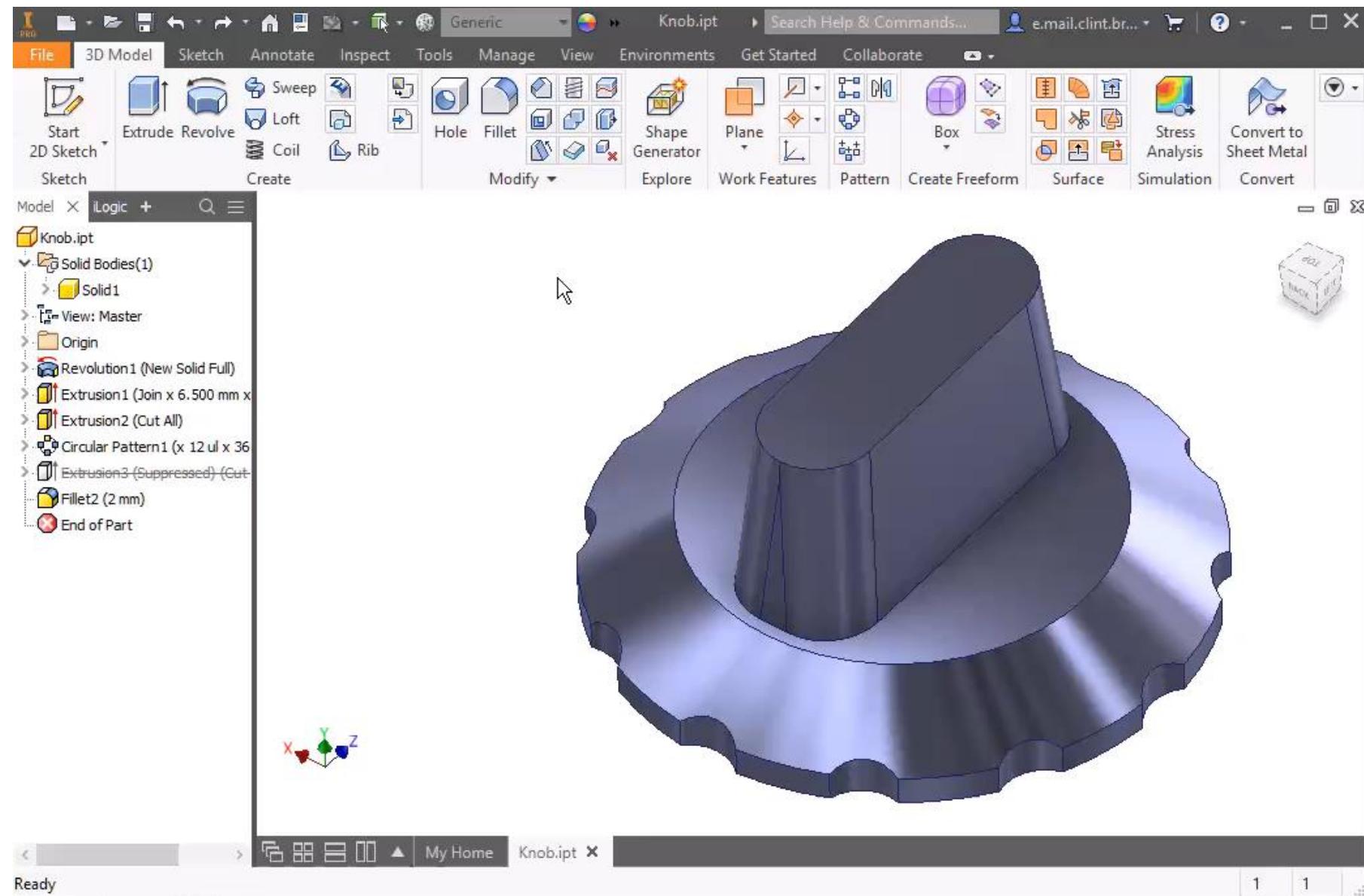
2018.1 bought in the “By Feature” filter which allows a user to see which parameters were used to create a feature



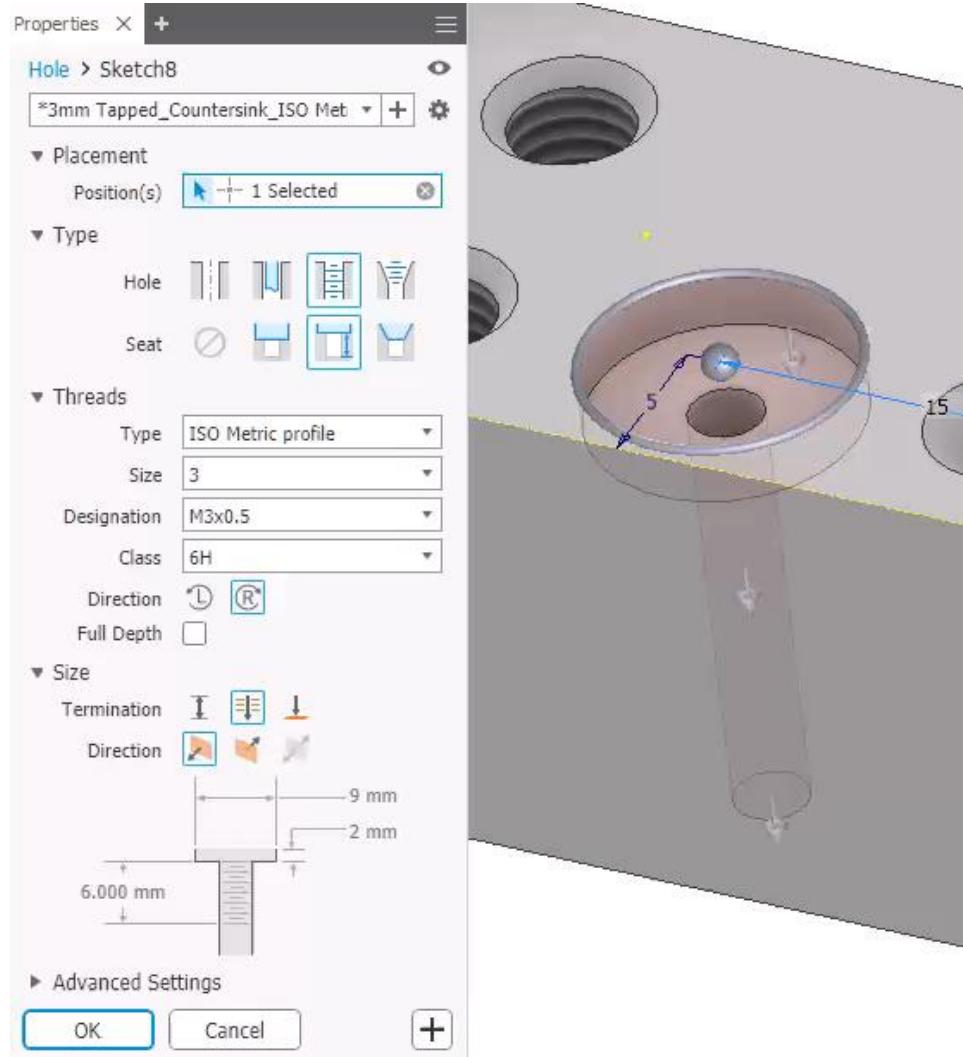
Parts: All Fillets + All Rounds

Useful for cast/plastic components that have a specified minimum radius.

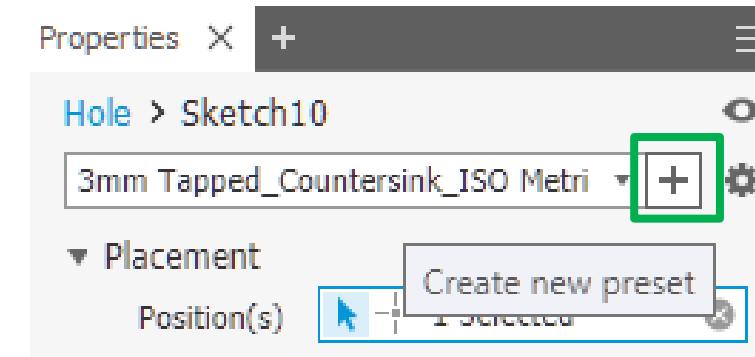
The feature will continue to calculate the fillets, even after the geometry changes



Parts: Creating a Hole Pre-set (2019)



- Specify the hole
- Click +
- Give the new pre-set a logical name



Parts: Edit Hole Pre-sets (2019)

Pre-sets can be saved, renamed or deleted via settings

There is no “Edit” for pre-sets

Edit a Pre-set:

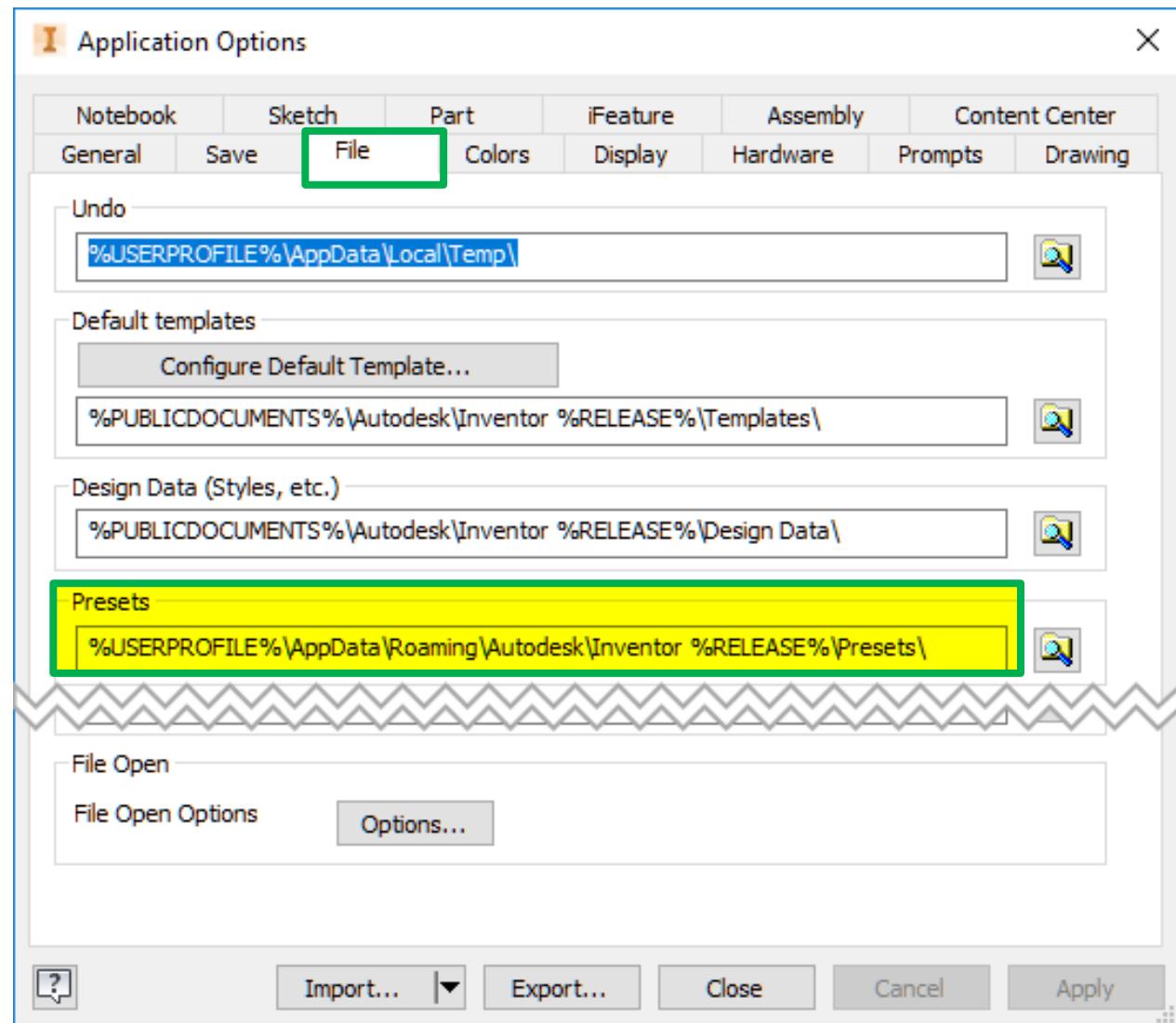
- Choose it from the drop down
- Modify the settings to suite
- Save the current setting



- Choose the old pre-set from the menu
- Delete the old pre-set

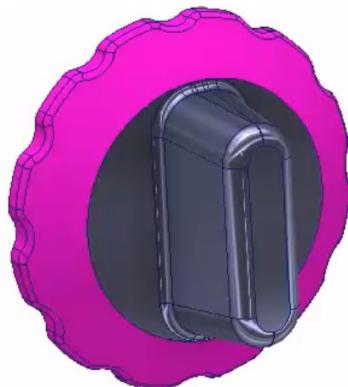
Parts: Sharing Hole Pre-sets (2019)

By default Hole Pre-sets are stored locally, this location could be set to a network location



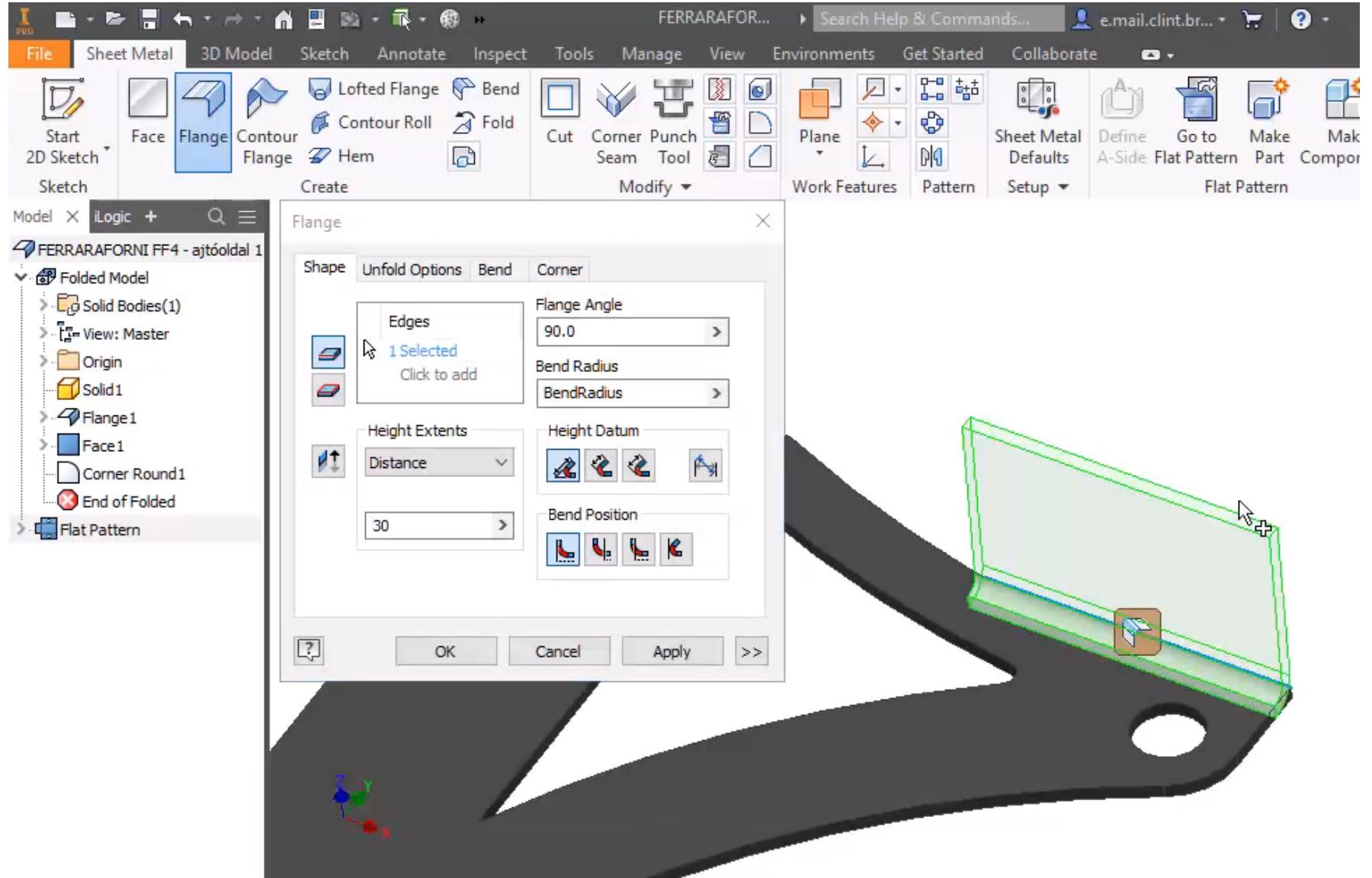
Parts: Select Tangencies

“Select Tangencies” selects all faces or edges tangent to the current selection



Parts: Custom Sheet Metal Flanges

Specify specific flange widths and offsets in the expanded menu.



Get some Perspective

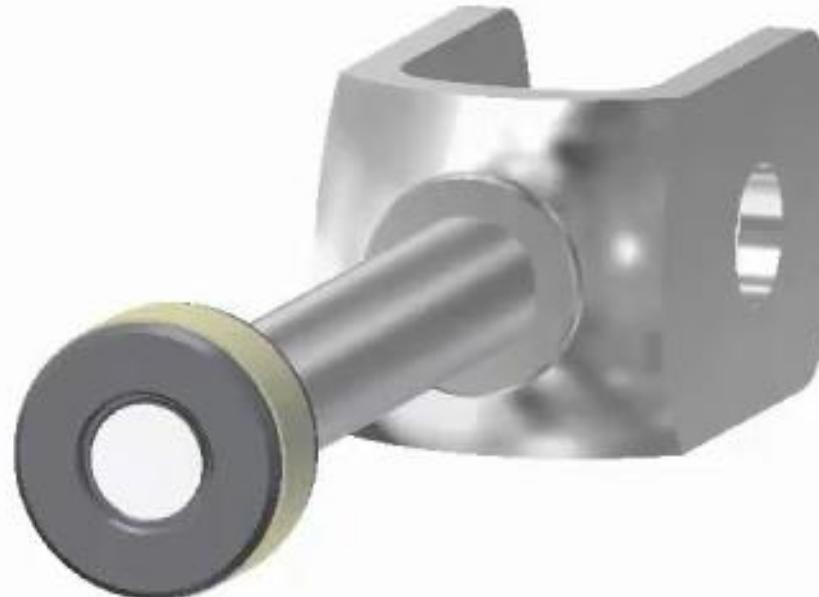
Right Click on the view cube > Click Perspective

Use "Perspective" views for renders and images

Tip: it's faster to work in Orthographic mode



Get some Perspective

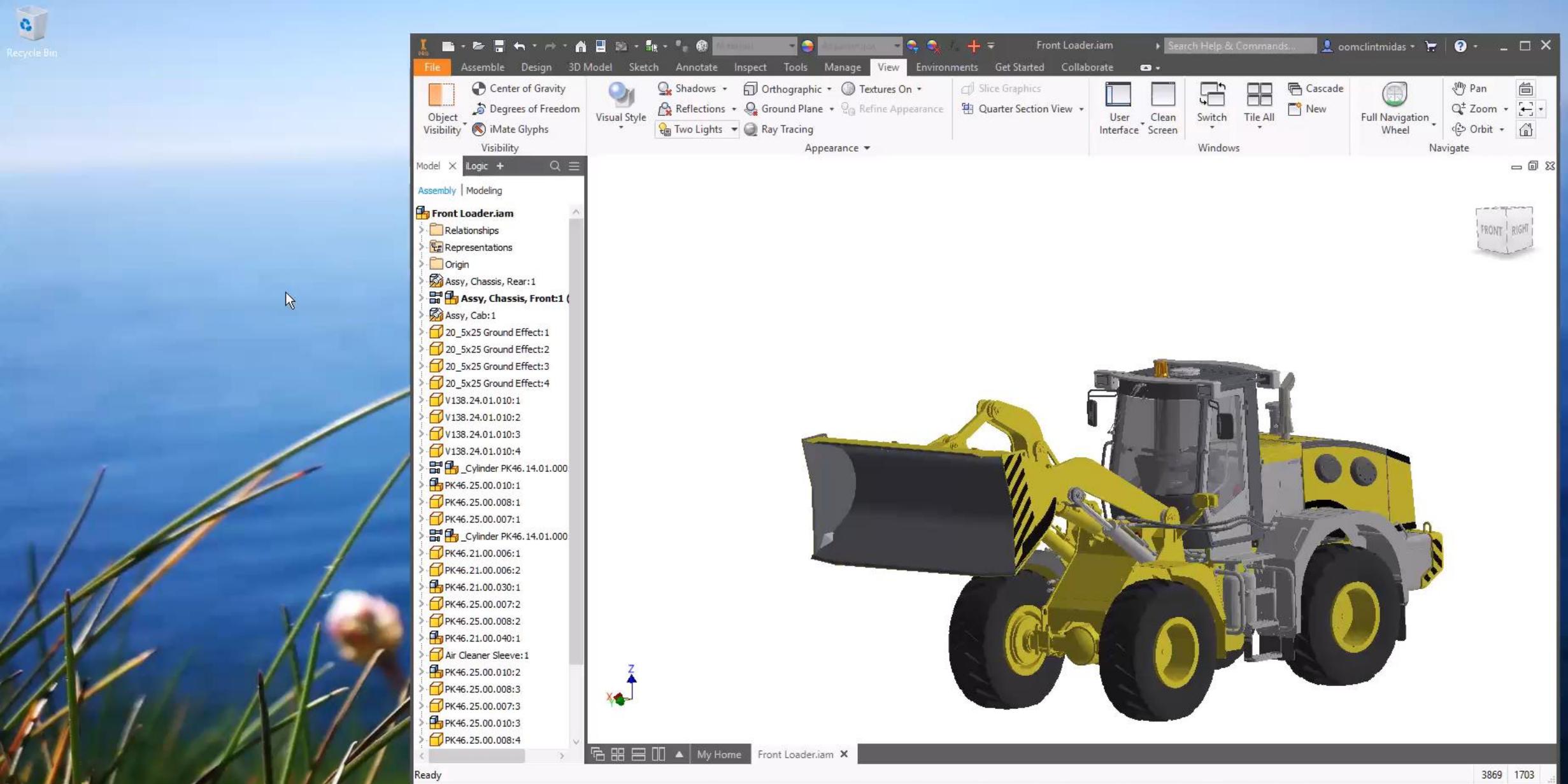


Perspective (Ctrl + Shift + F3)

To adjust the level of perspective

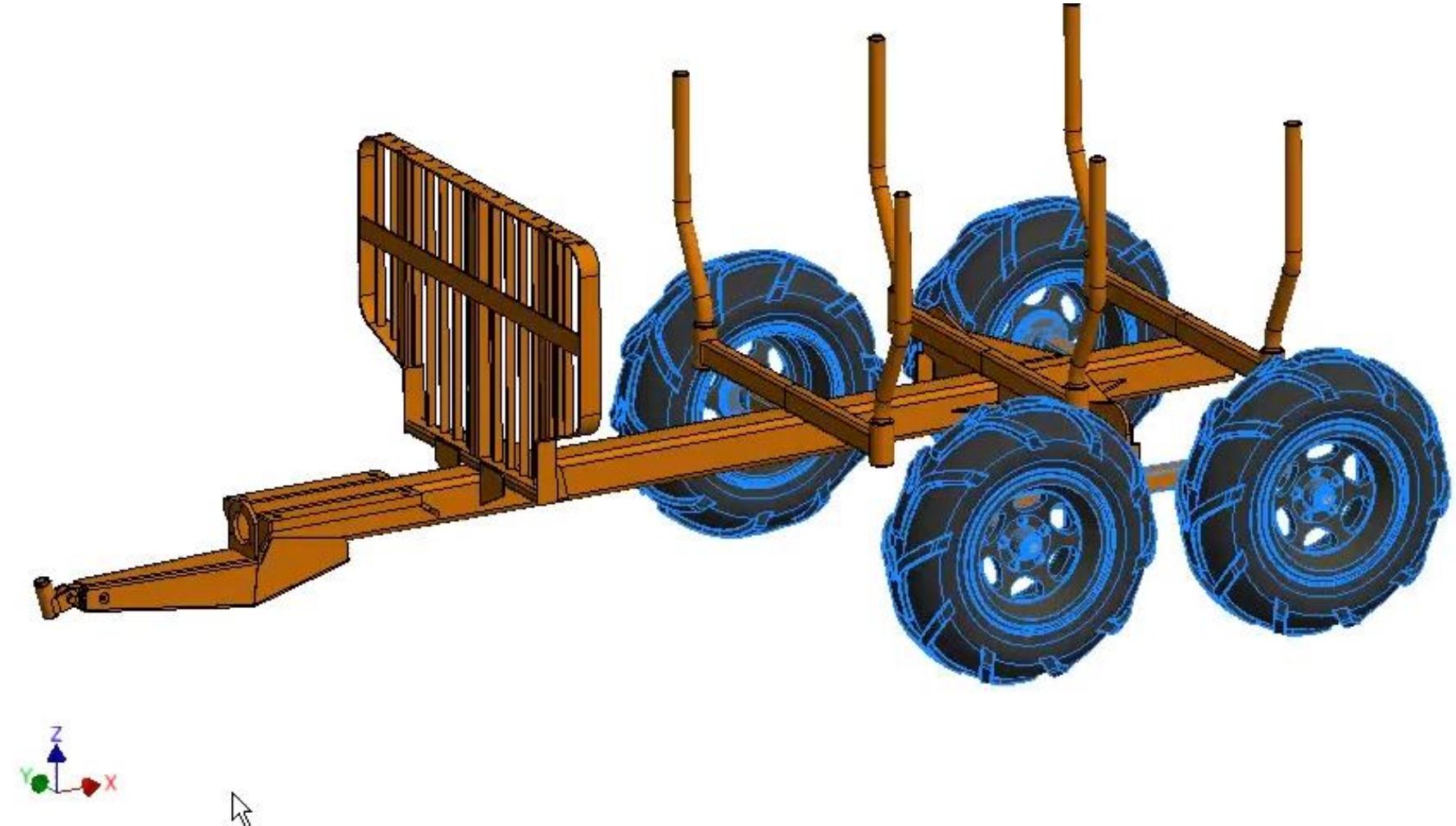
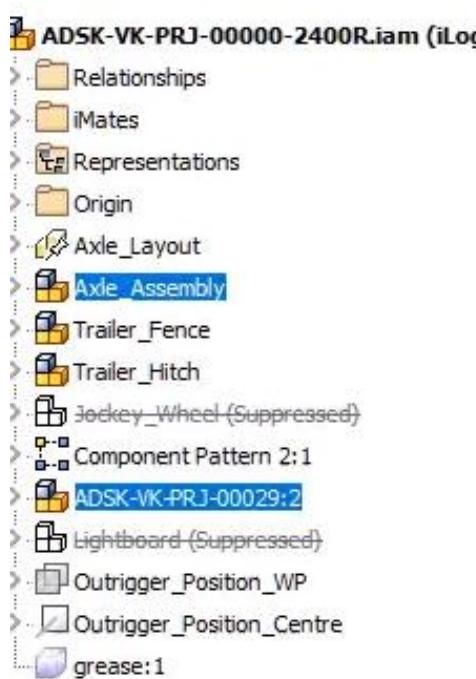
Ctrl + Shift + F3 and scroll up and down.

Fast Client image



Assemblies: Assembly Folders

- Use Assembly Folders to organise complex assembly data
- Folders can be used to Isolate or Suppress sets of components

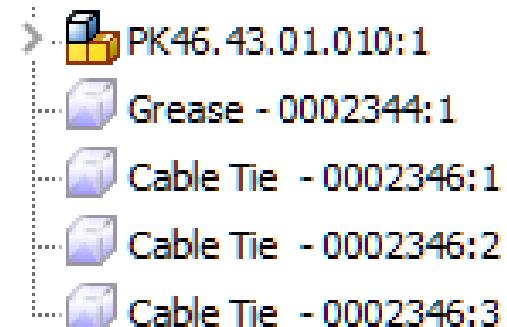
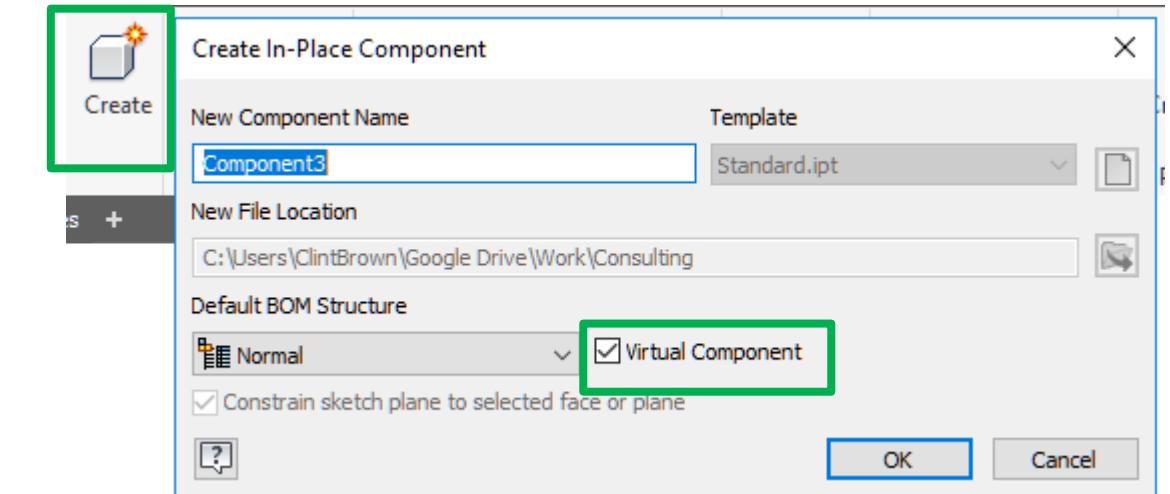
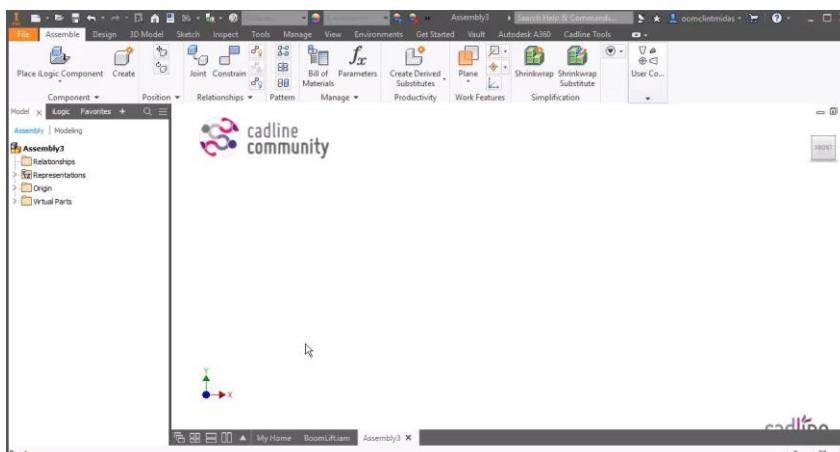


Assemblies: Virtual Components

Use Virtual Components to add parts to your Inventor BoM that you might not normally model

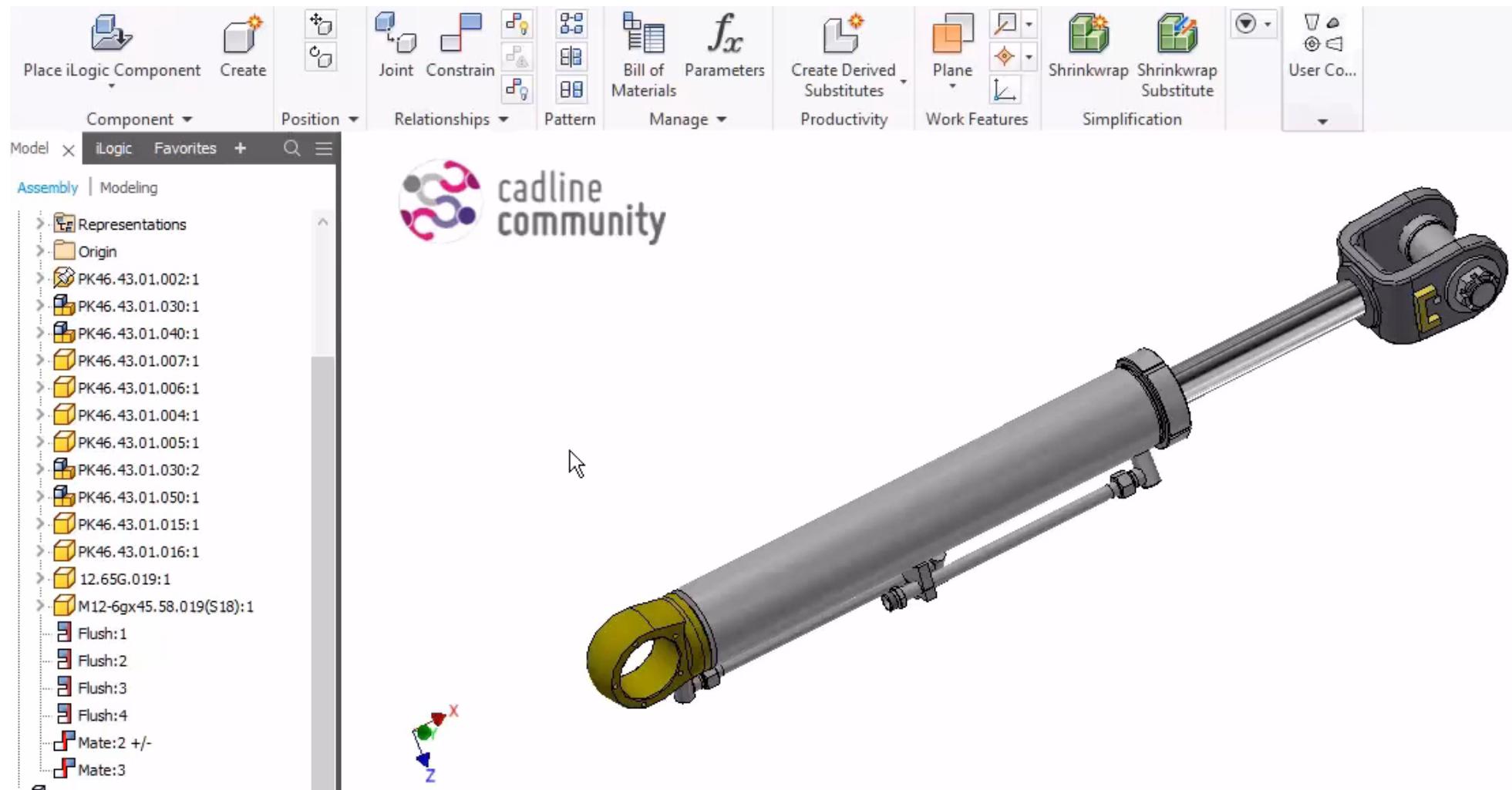
Bonus Tip: Use iLogic to add these parts on the fly, see link below [_____](https://www.cadlinecommunity.co.uk/hc/en-us/articles/212726349-Inventor-2017-iLogic-Create-Virtual-Parts-on-the-Fly)

Bonus Tip 2: Save commonly used Virtual Components in your Assembly file



Assemblies: Display Mass on a BoM

Use a Virtual Components to calculate the Mass of the Assembly, and display this on a BoM



`iProperties.Value("VIRTUAL PART", "Custom", "Assembly Mass") = iProperties.Mass`

Assemblies: BOM – Check properties & Mass Manipulation

The BoM environment is a good place to check materials and assign properties to many files at once.

Copy/Paste between columns

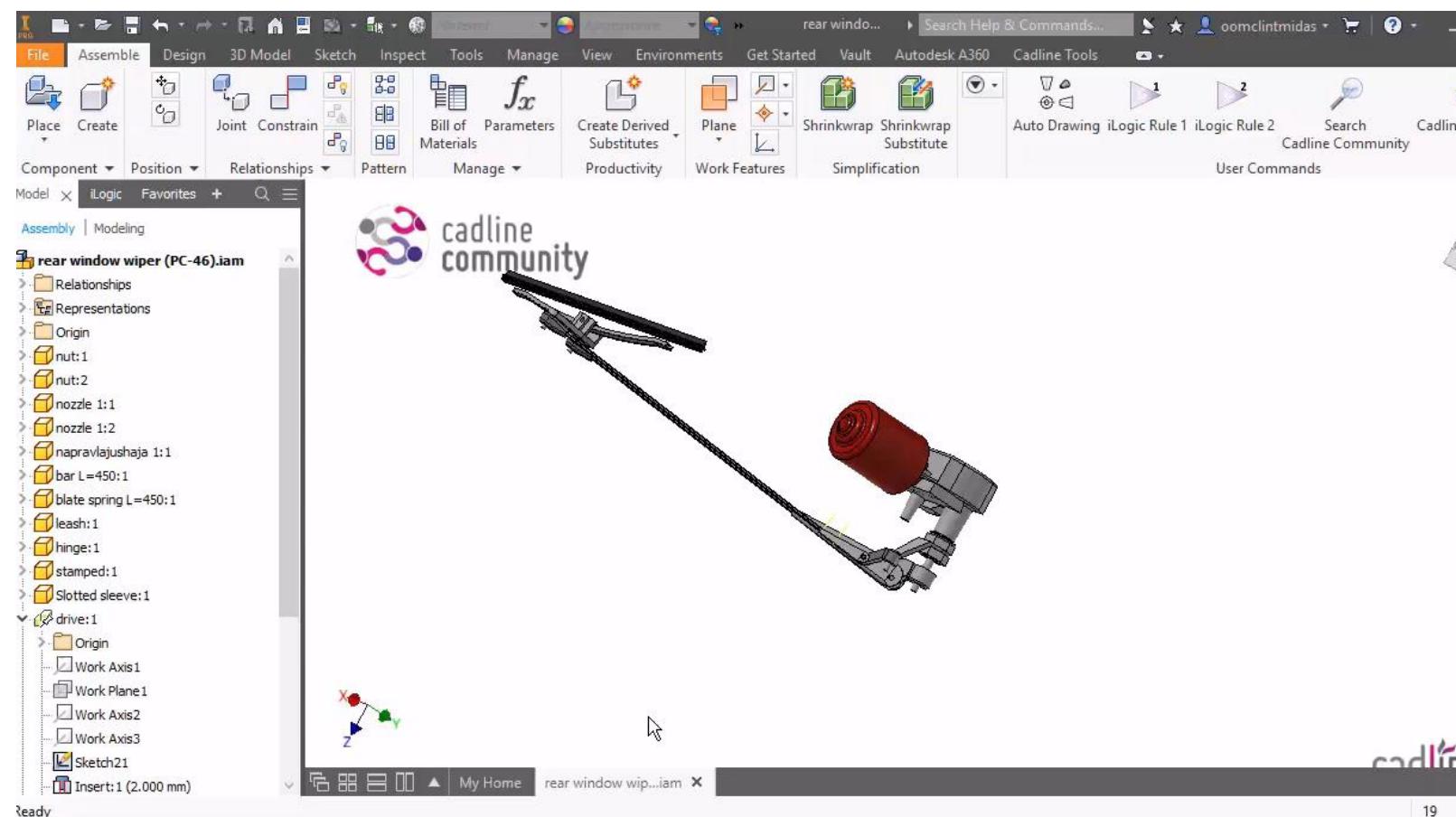
CAPITALISE

Create Expressions

Change Materials

Bonus tip: (Inventor 2018.1 onwards), right click on a BoM row entry to open it.

Note: For Vaulted Assemblies, the children (Sub assemblies & parts) will need to be checked out if their properties are to be edited.



Assemblies: BOM – Customise Default Template

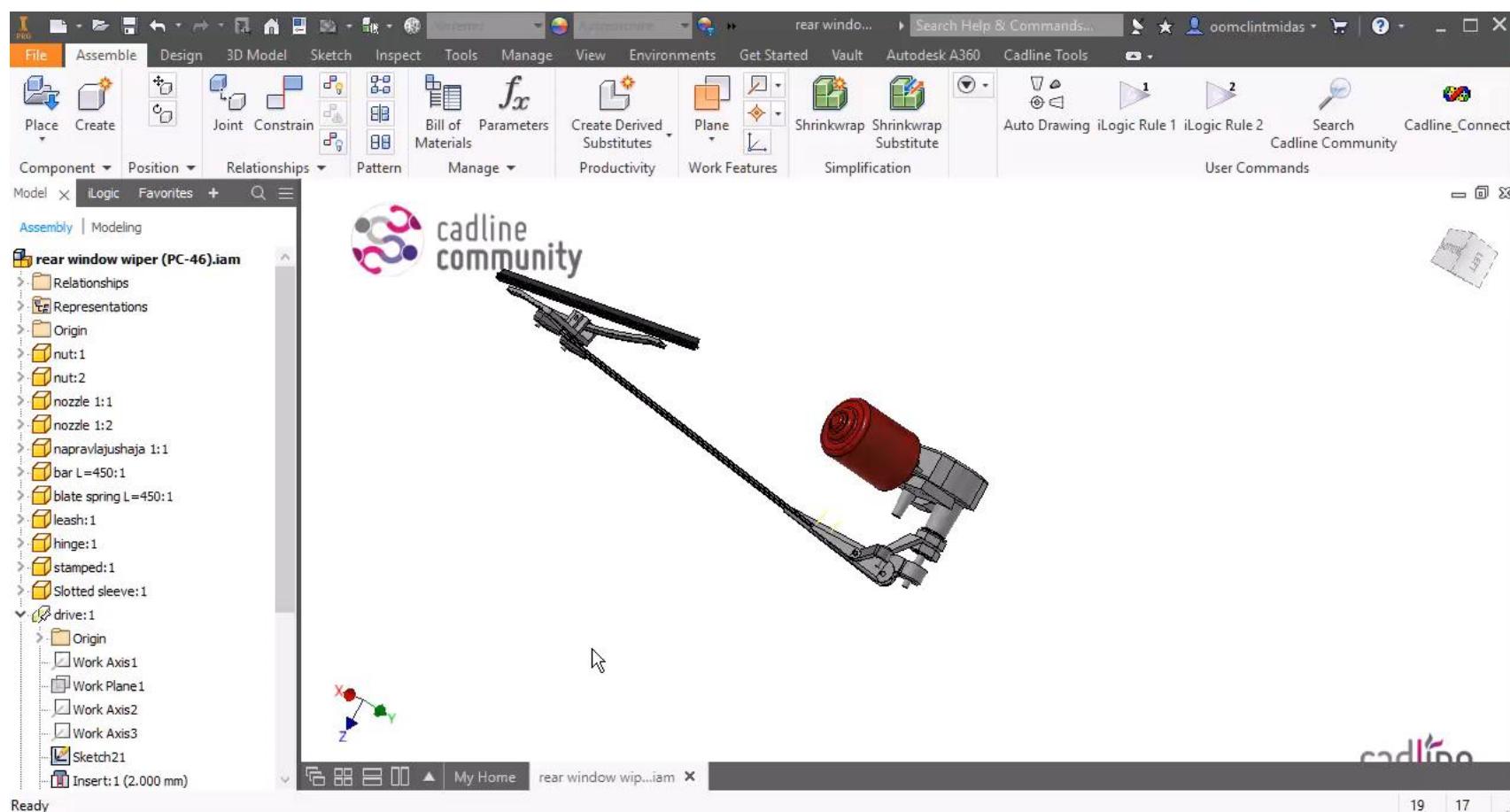
Set up your BoM template.

Choose the Rows to include, enable the BoM views that you use by default.

Export out an XML file

Open your default template, import the XML, “Save as Template”

Done!

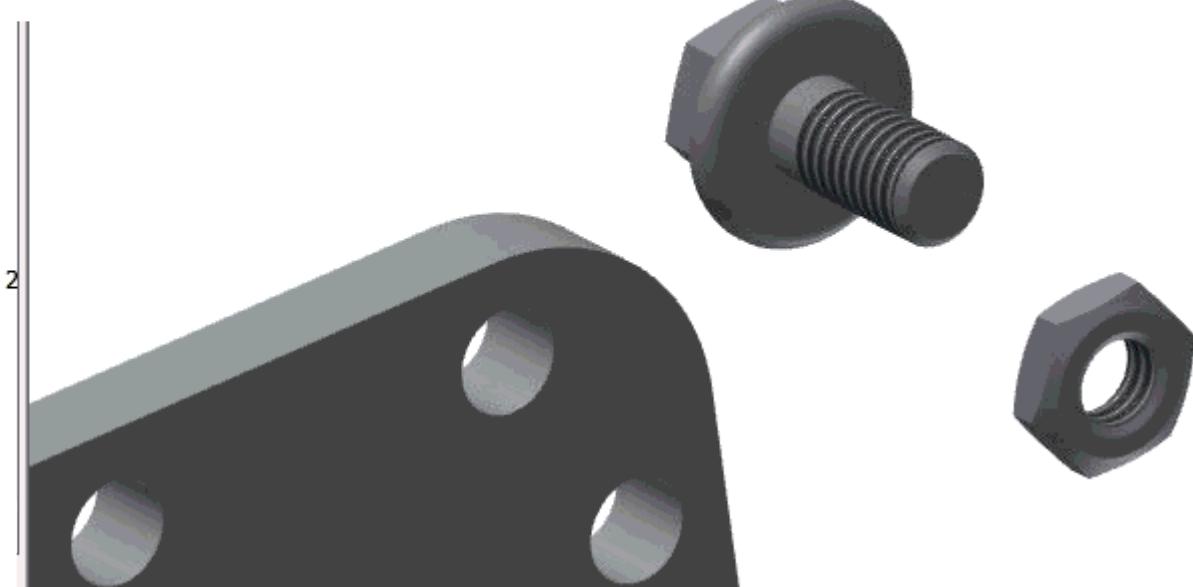
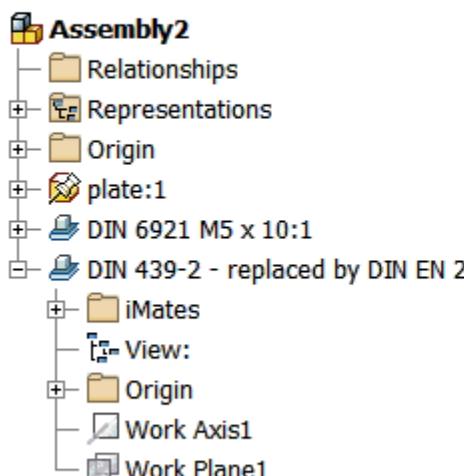


Assemblies: Alt + Drag Constraints

Alt + Drag

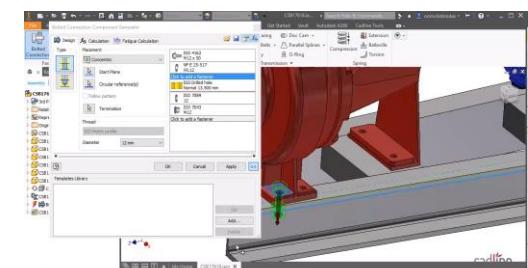
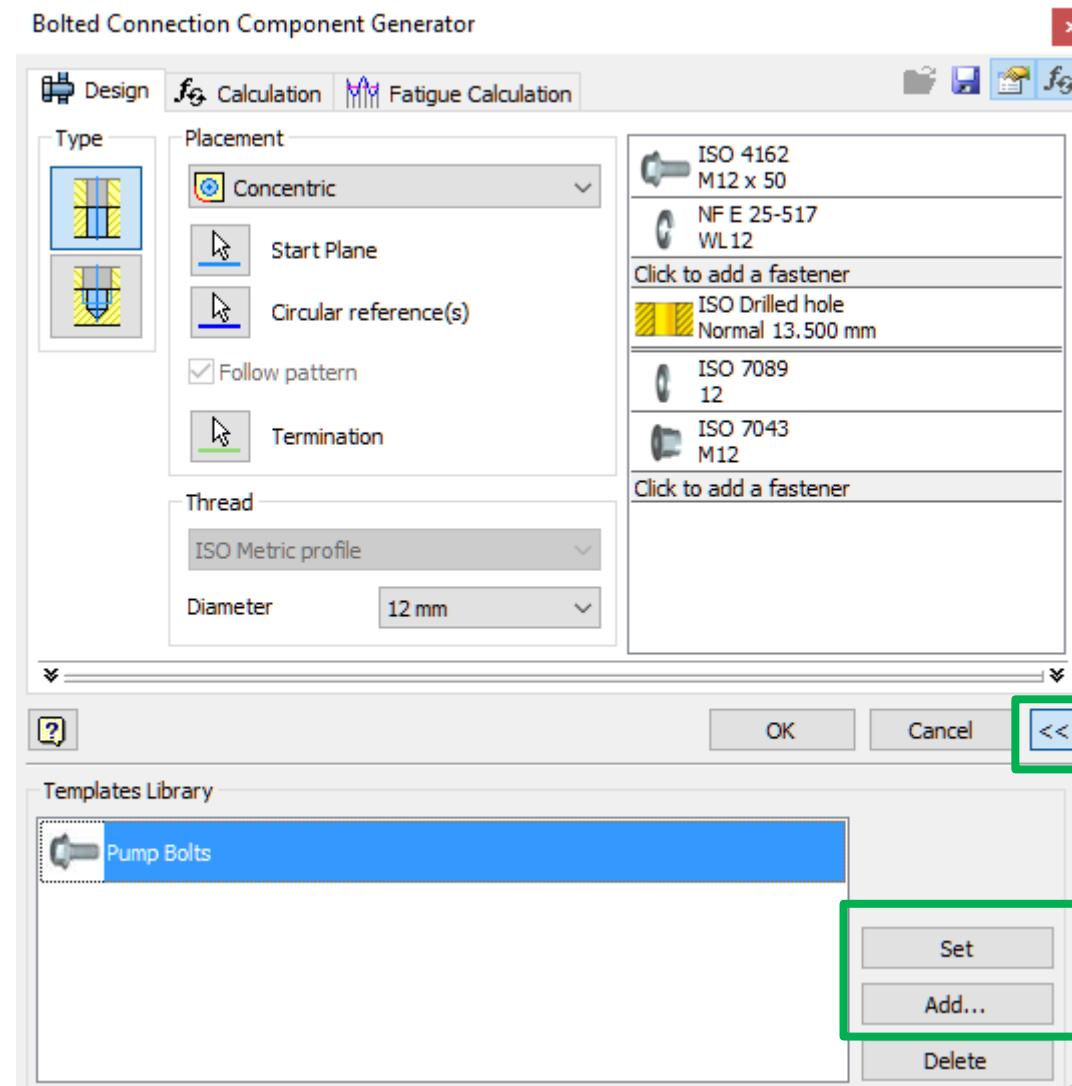
If you hold down ALT and then drag (hold down your left mouse button) geometry it will create a constraint, based on the geometry that you select, the animation below shows how I have placed an insert constraint.

Note: The Alt + Drag workflow will create the following Constraints, Mate, Tangent and Insert



Assemblies: Save Favourite Bolted Connections

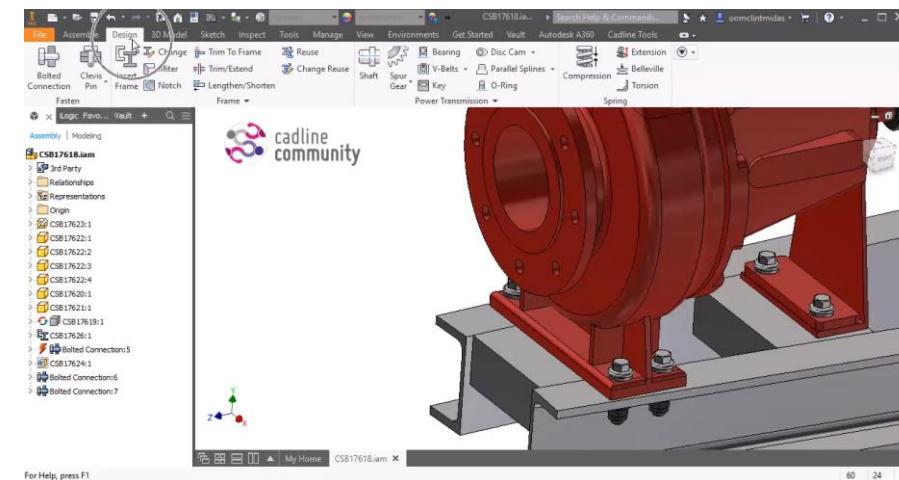
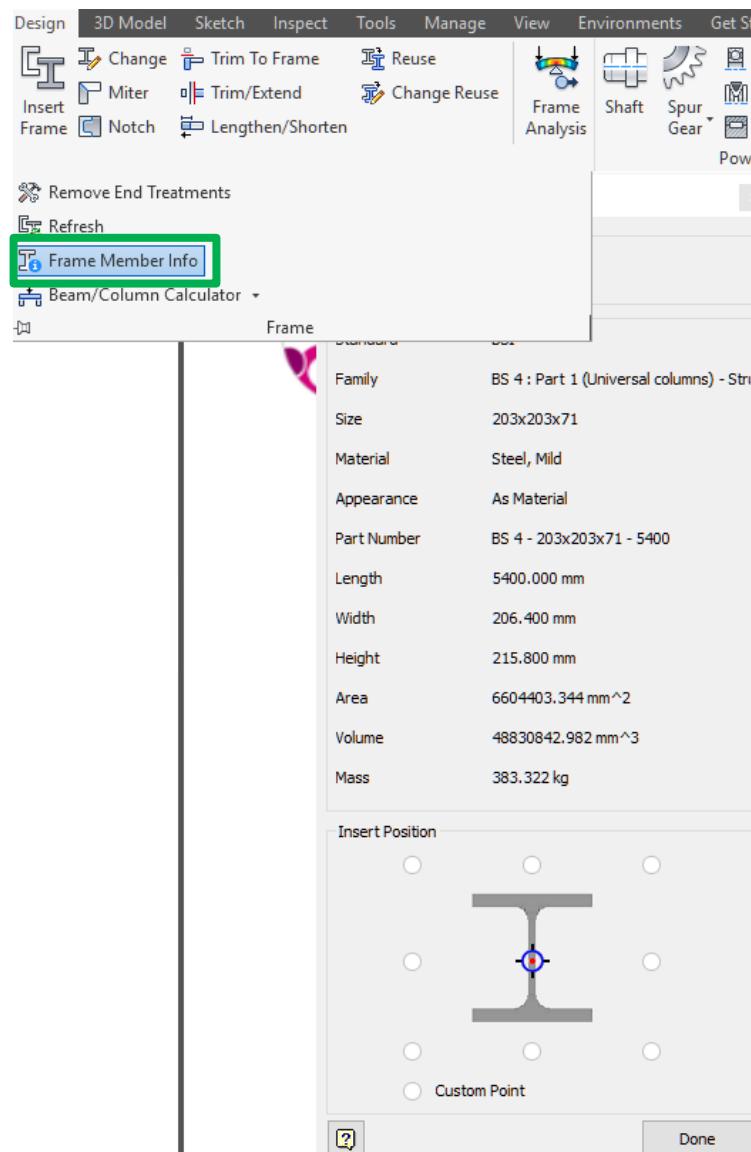
Save and re-use your favourite Bolted Connections



Assemblies: Frame Member Info

Use “Frame Member Info” to see detailed properties of individual frame members, including:

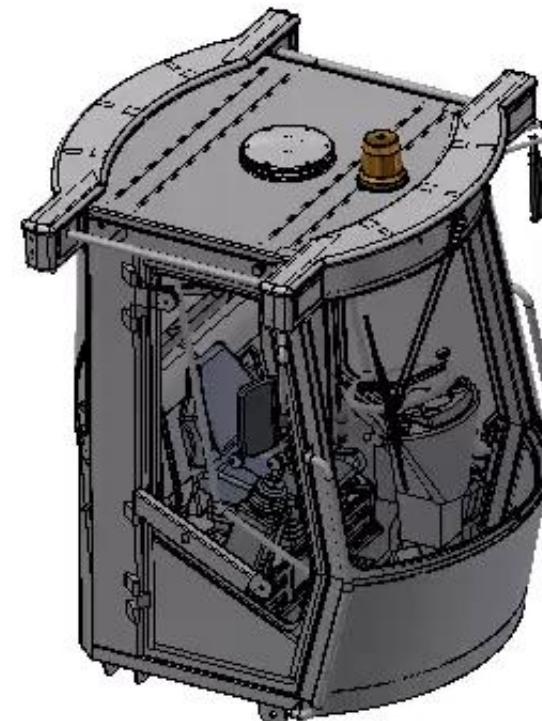
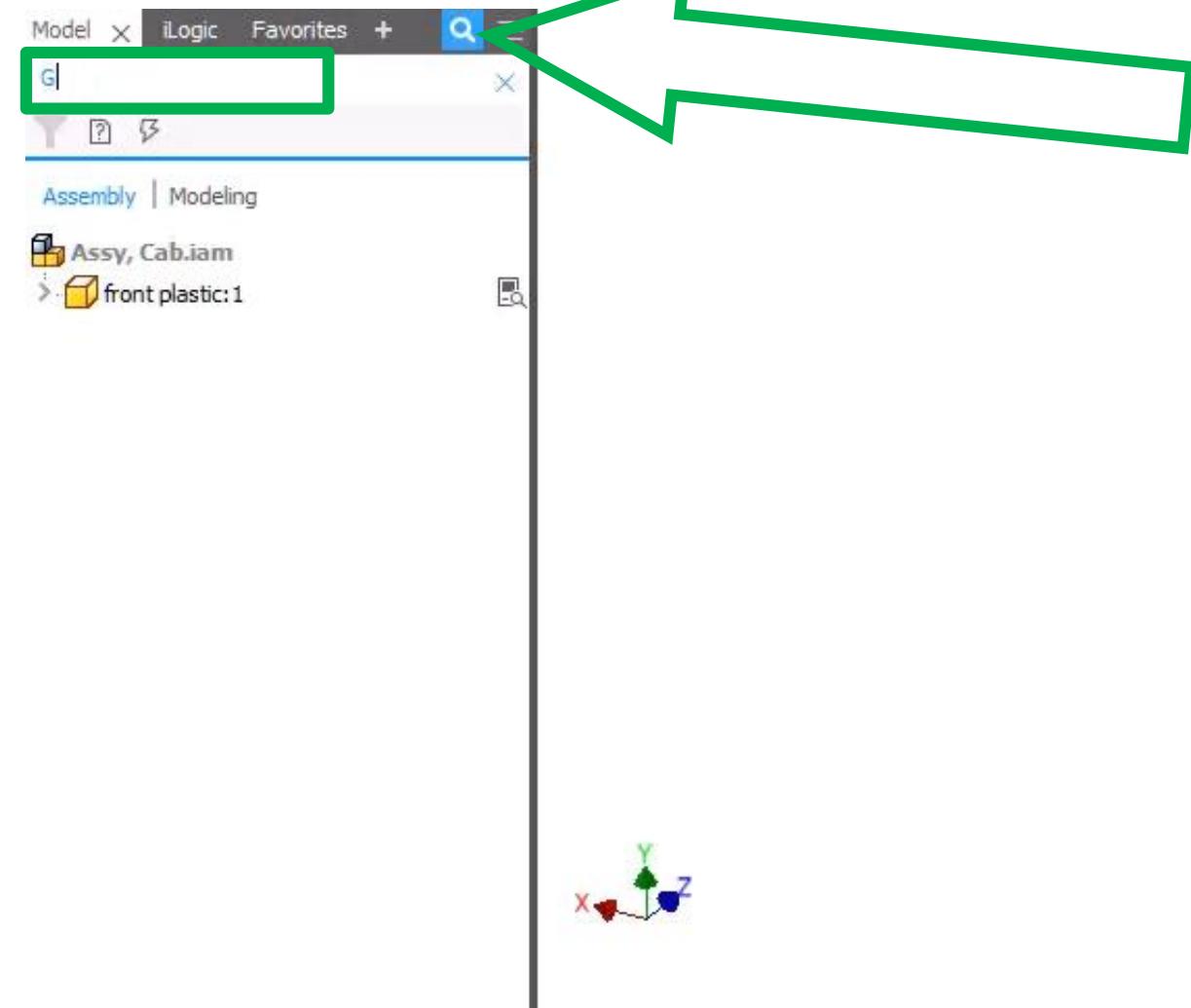
Type
Insertion Position
Mass



Assemblies: Browser Search

Introduced with Inventor 2018.

Search for parts in an assembly (at any level) based on name, part number, or iProperty (including custom iProperty values)

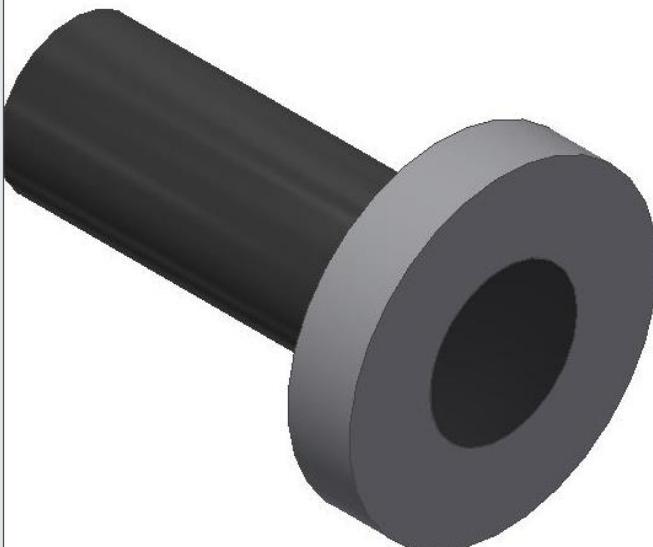
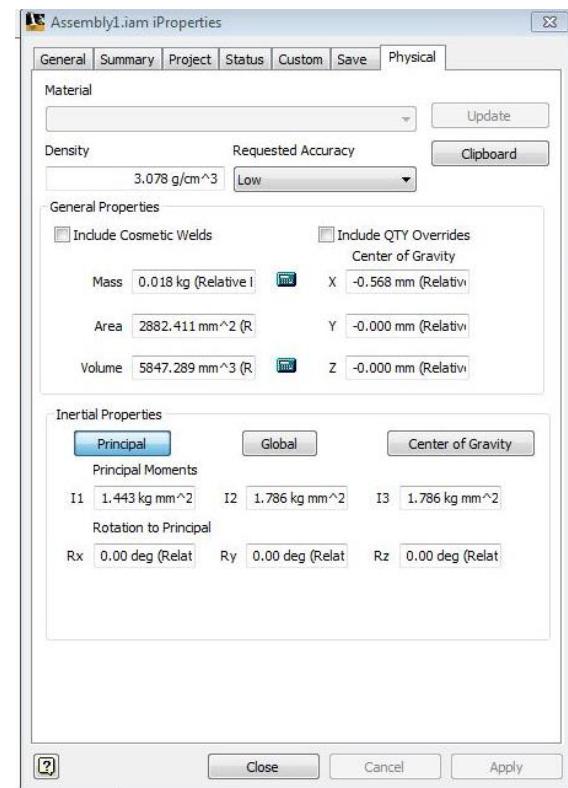


Assemblies: Surface Area

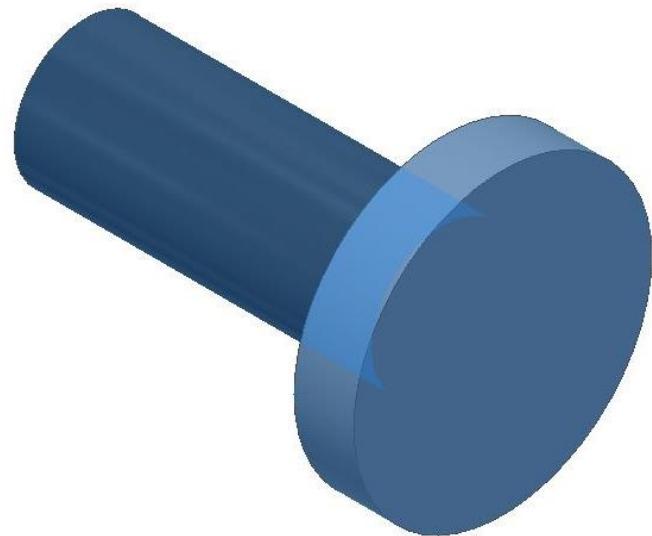
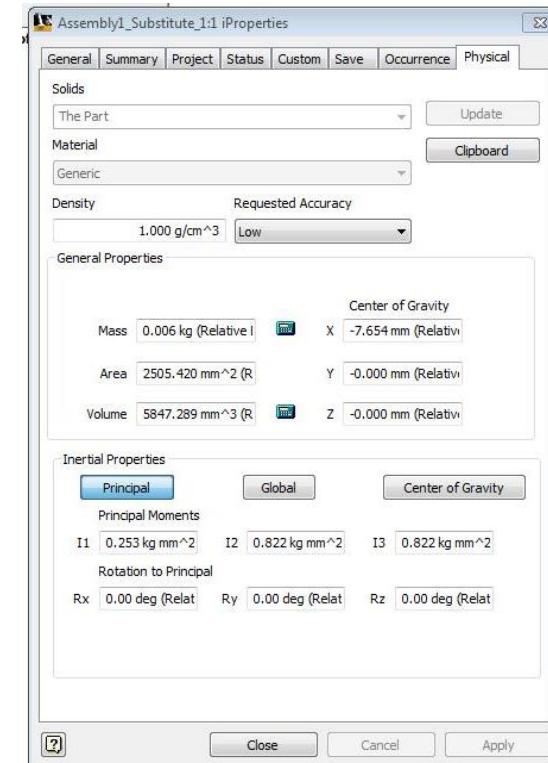
The Surface area shown in an assembly is the total Surface area of all parts in the assembly.

For Painting or Coating create shrinkwrap for a more accurate result

Assembly = 2882.411mm²



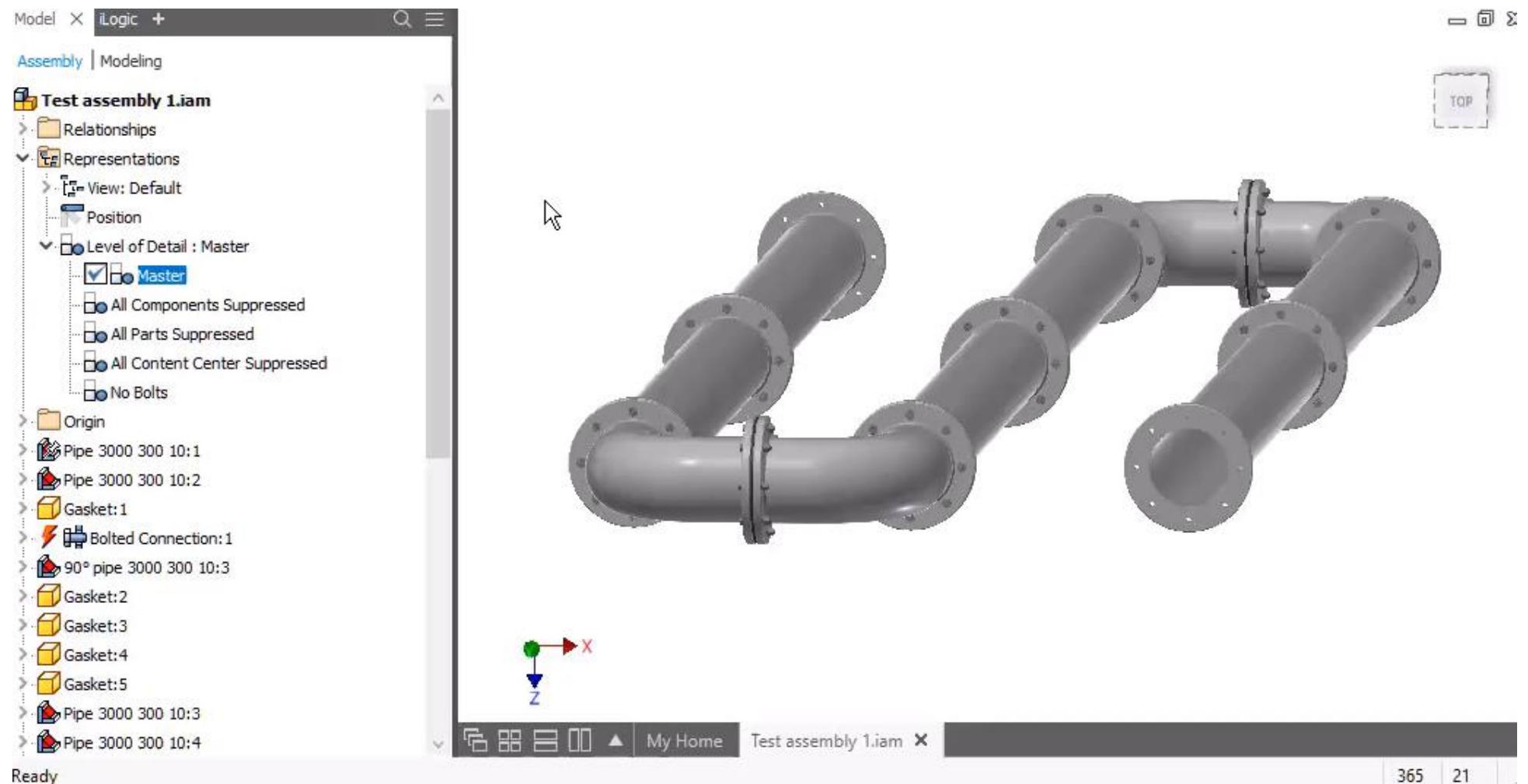
Shrinkwrap = 2505.420mm²



Assemblies: Shrink Substitute LoD's via iLogic

Reduce model complexity with LoD's

Use Shrinkwrap substitute LoD iLogic Rule to quickly create simplified LoD's



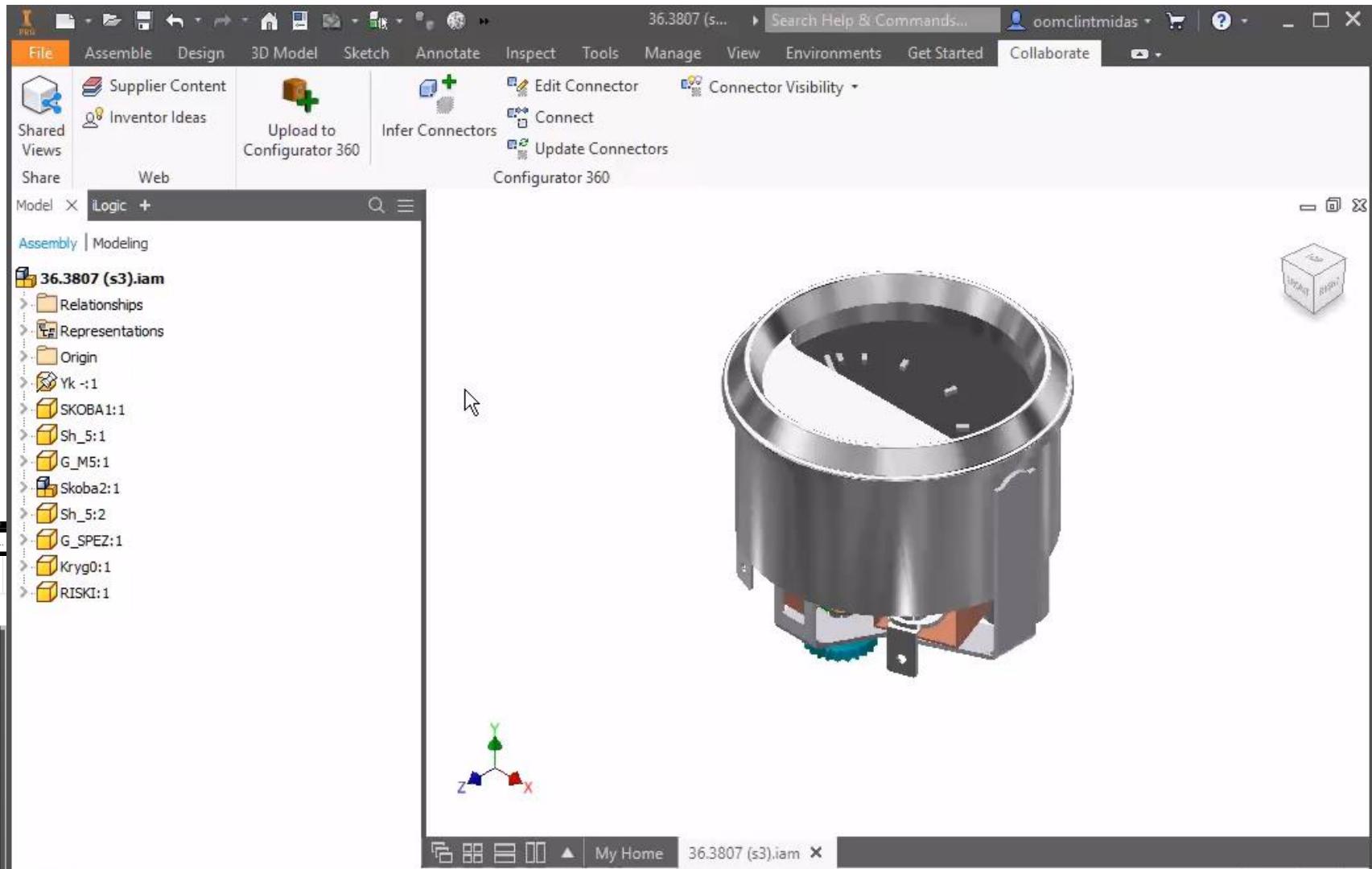
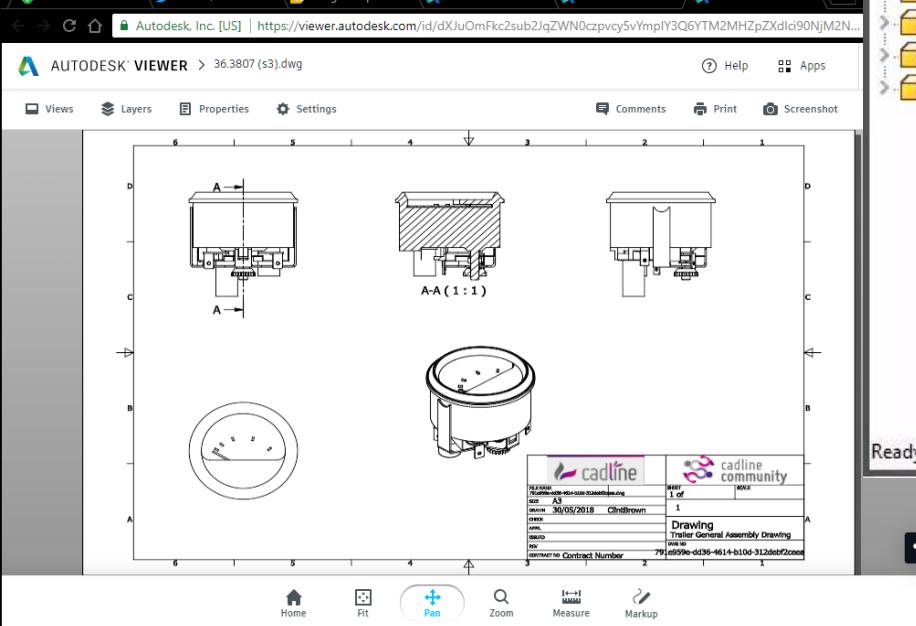
Collaborate with Shared views (new in 2019)

Shared Views last 30 days (unless renewed)

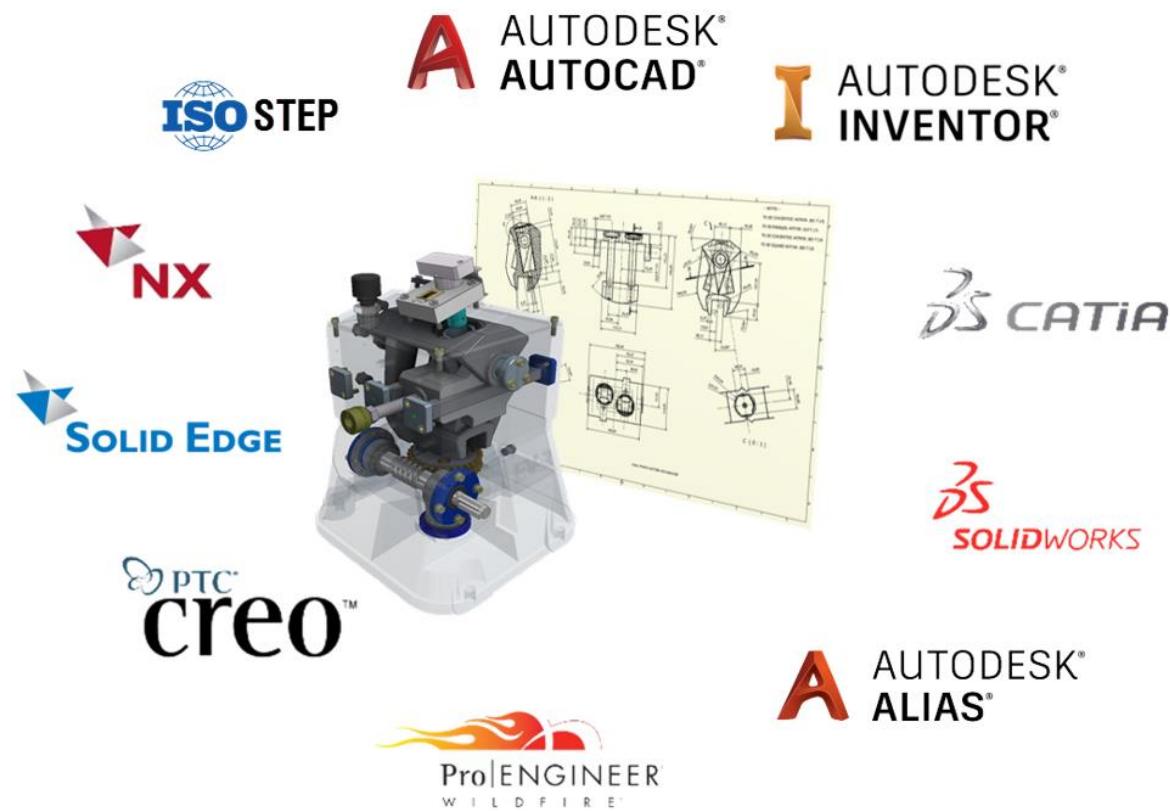
Allows collaboration over the web

Great for collaborators who do not have CAD tools!

Works with parts, assemblies and drawings



Lost in Translation: ANYCAD v STEP v IGES



Always try to get hold of the native CAD files, Inventor works with practically every mainstream CAD vendor, AnyCAD workflow is better than a file translation

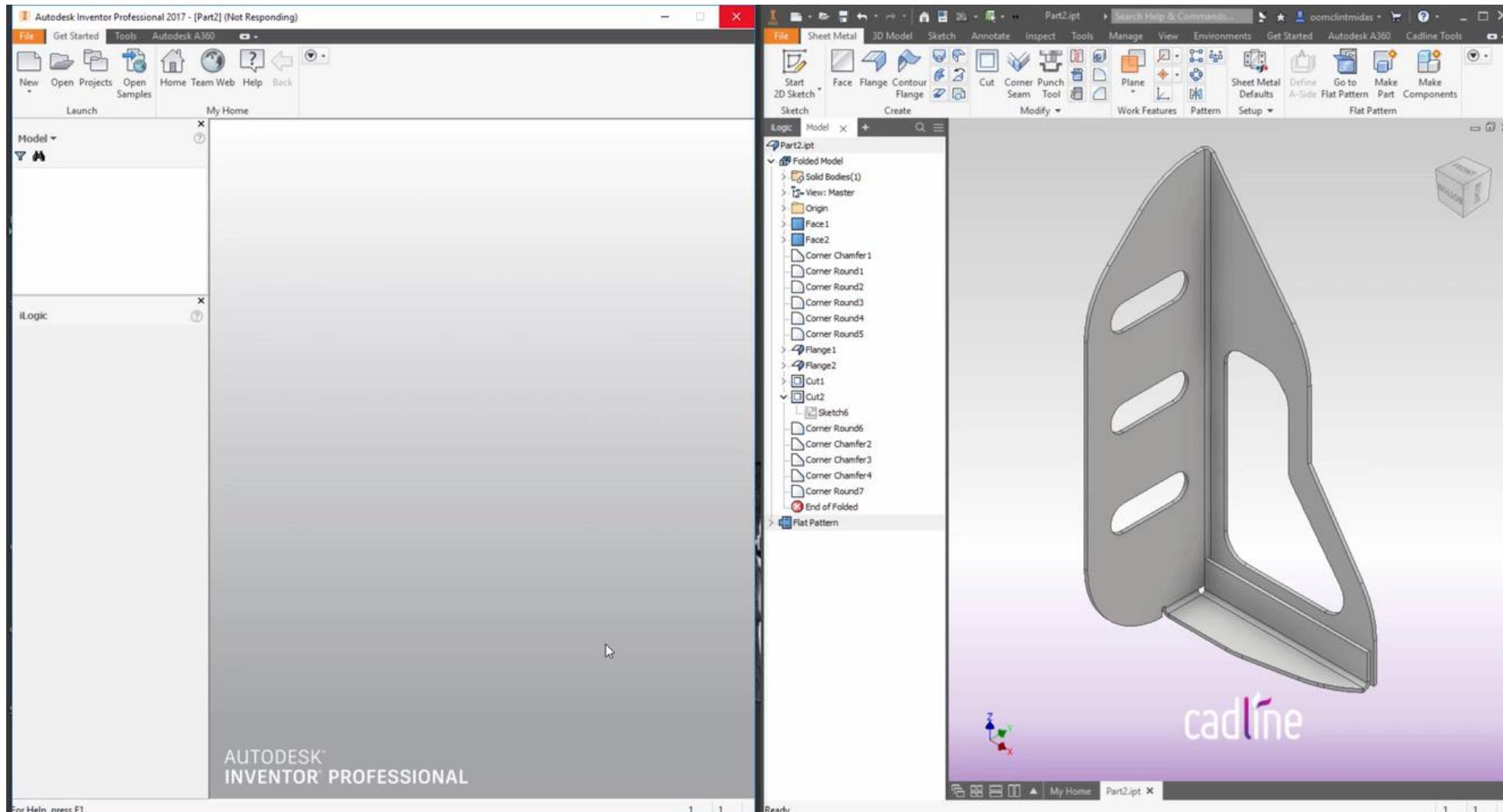
The **IGES** standard was last updated in **1996**

STEP has been available for over 20 years with continual development and wide adoption between CAD vendors

Backward Compatibility

Starting with Inventor 2017.4, you can **reference** future (newer) versions of Inventor part and assembly files.

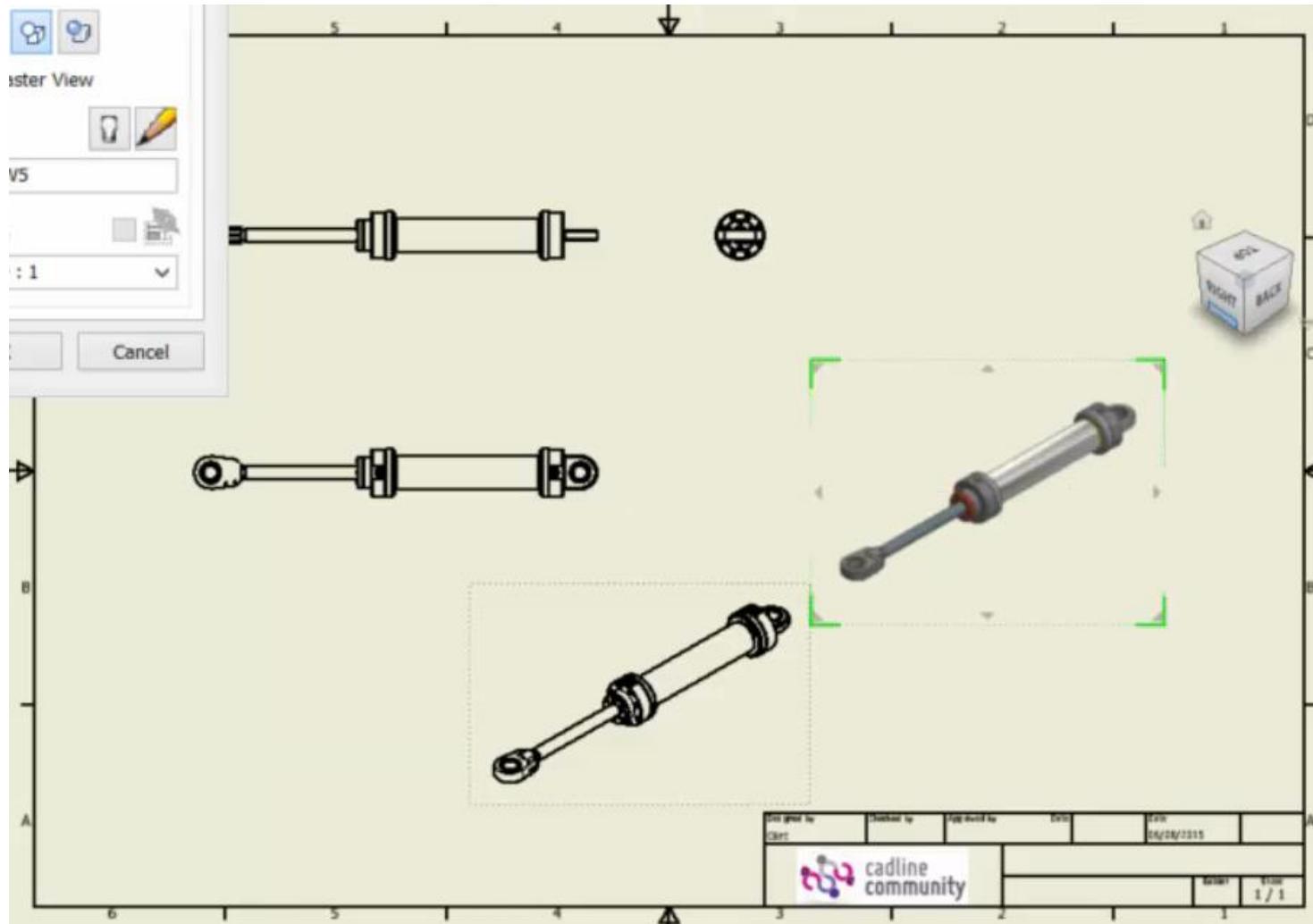
For example, you can now reference an Inventor 2018 part file in Inventor 2017



Inventor 2017.4

Inventor 2018

Get some Perspective - Drawings

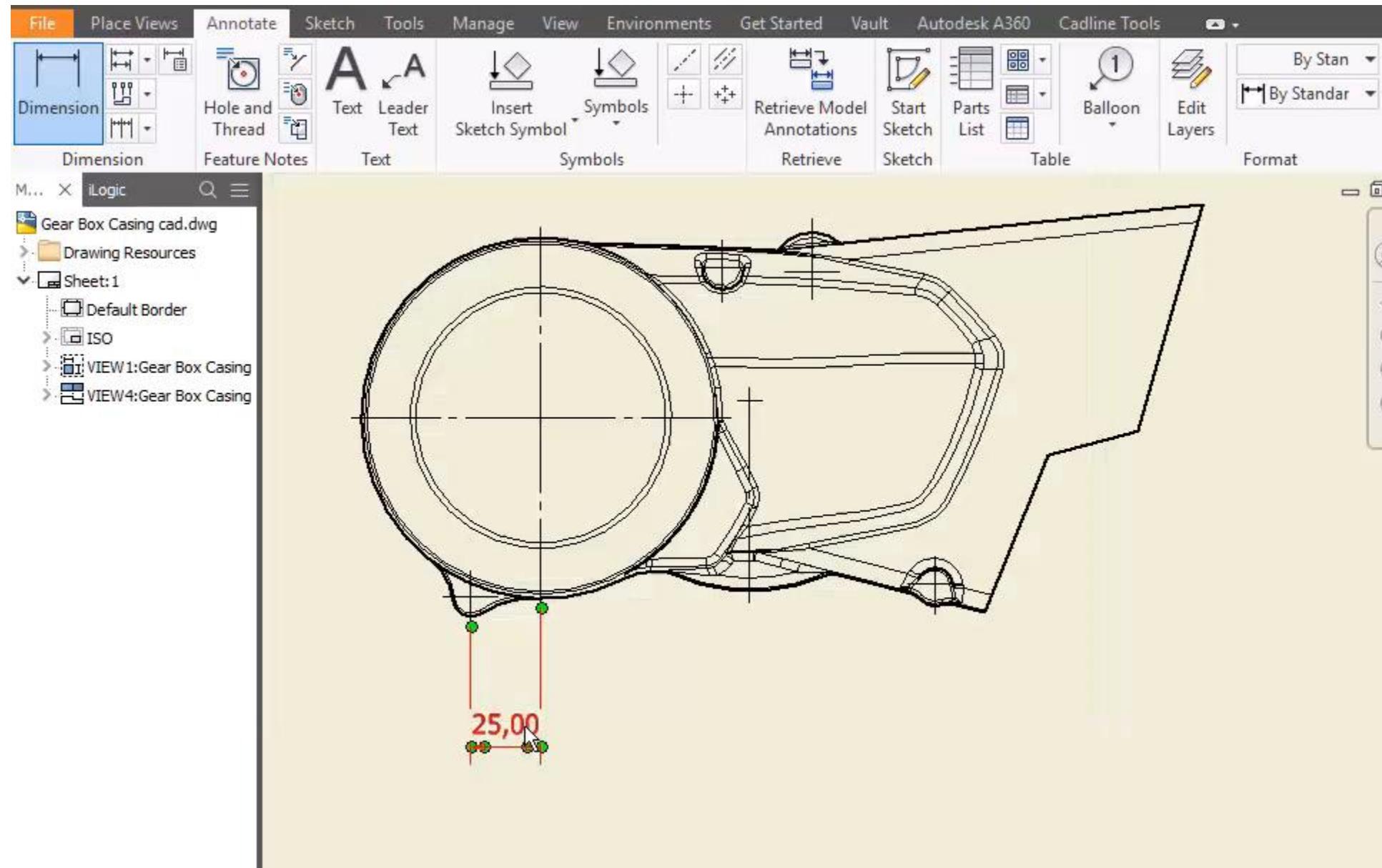


Since Inventor 2016, we can now use the "View Cube" to orientate our views, and as such, we can place Perspective views as opposed to Isometric views on our drawings, giving more realistic appearance to a "pictorial" view.

Drawings: use AutoCAD for dimensions

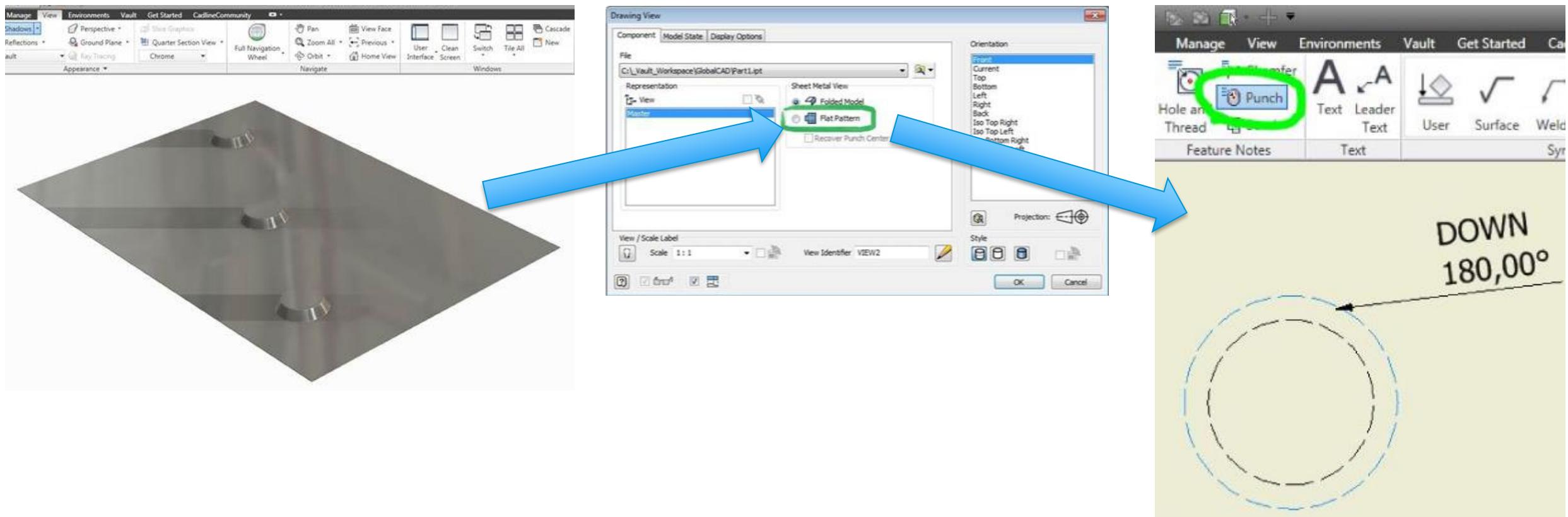
Did you know that you can dimension your Inventor DWG files in AutoCAD?

AutoCAD uses the same dimension style as your Inventor drawing

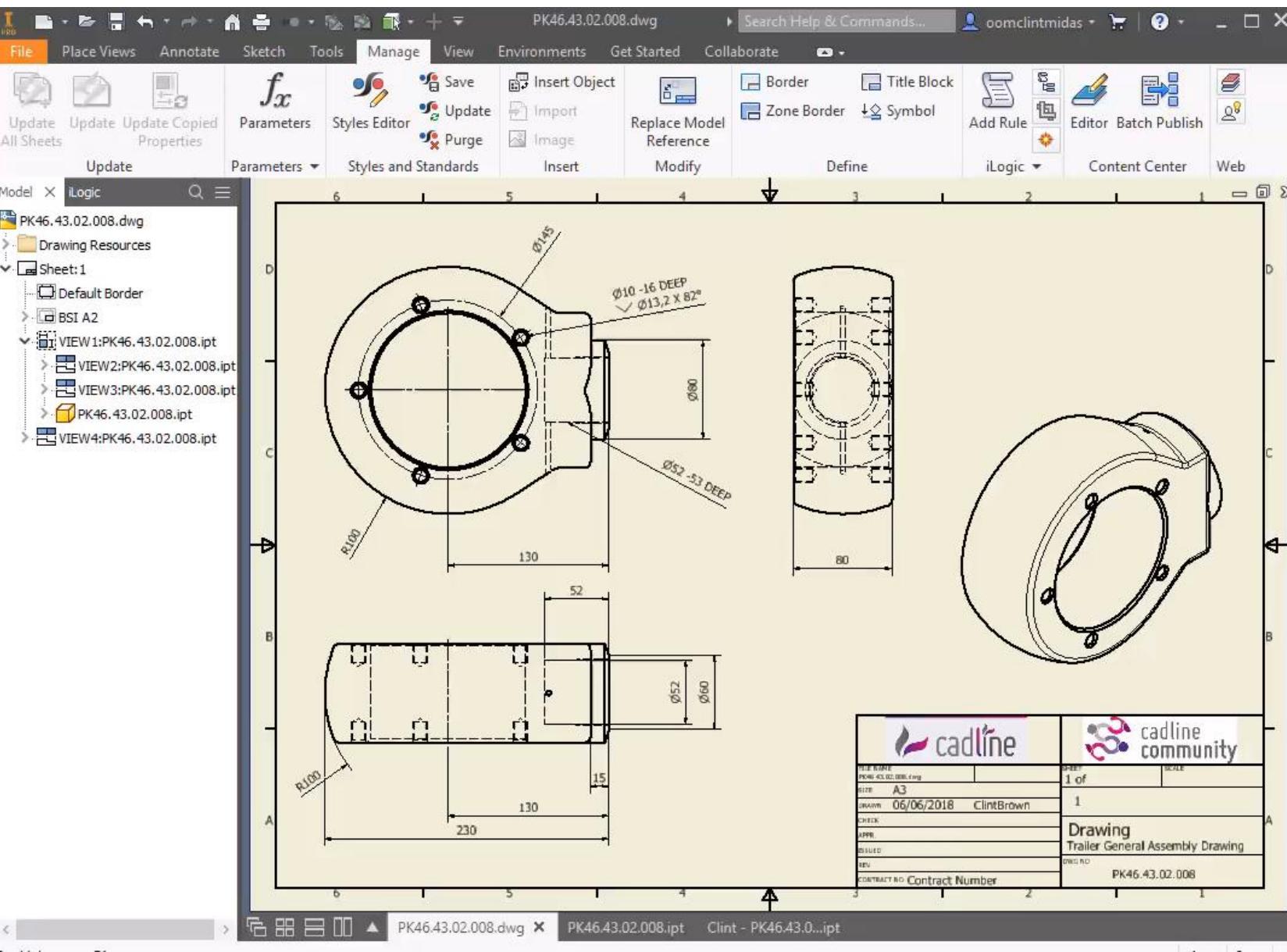


Drawings: Annotating Punch tools

On “Flat” sheet metal parts, you need to create a Flat Pattern, in order to use the annotation tools for punches



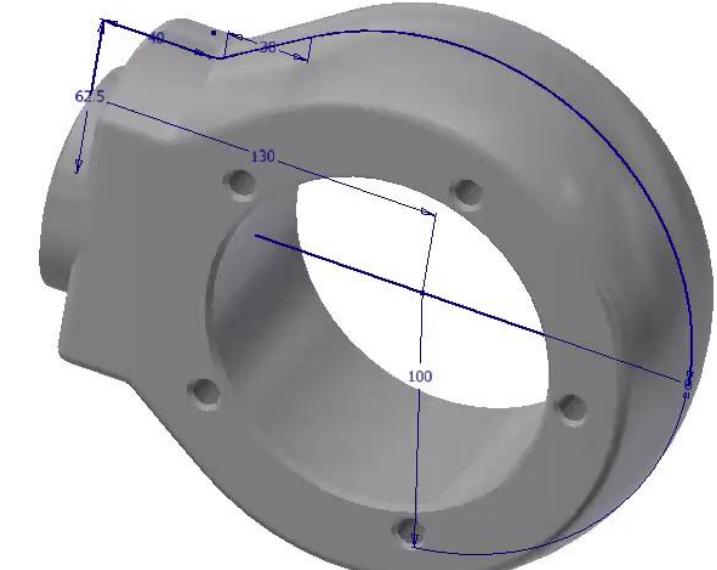
Drawings: Replace Reference



Introduced in Inventor 2011

Allows you to re-use your existing drawings

Works when the geometry is the same
i.e. "File" > "Save As"

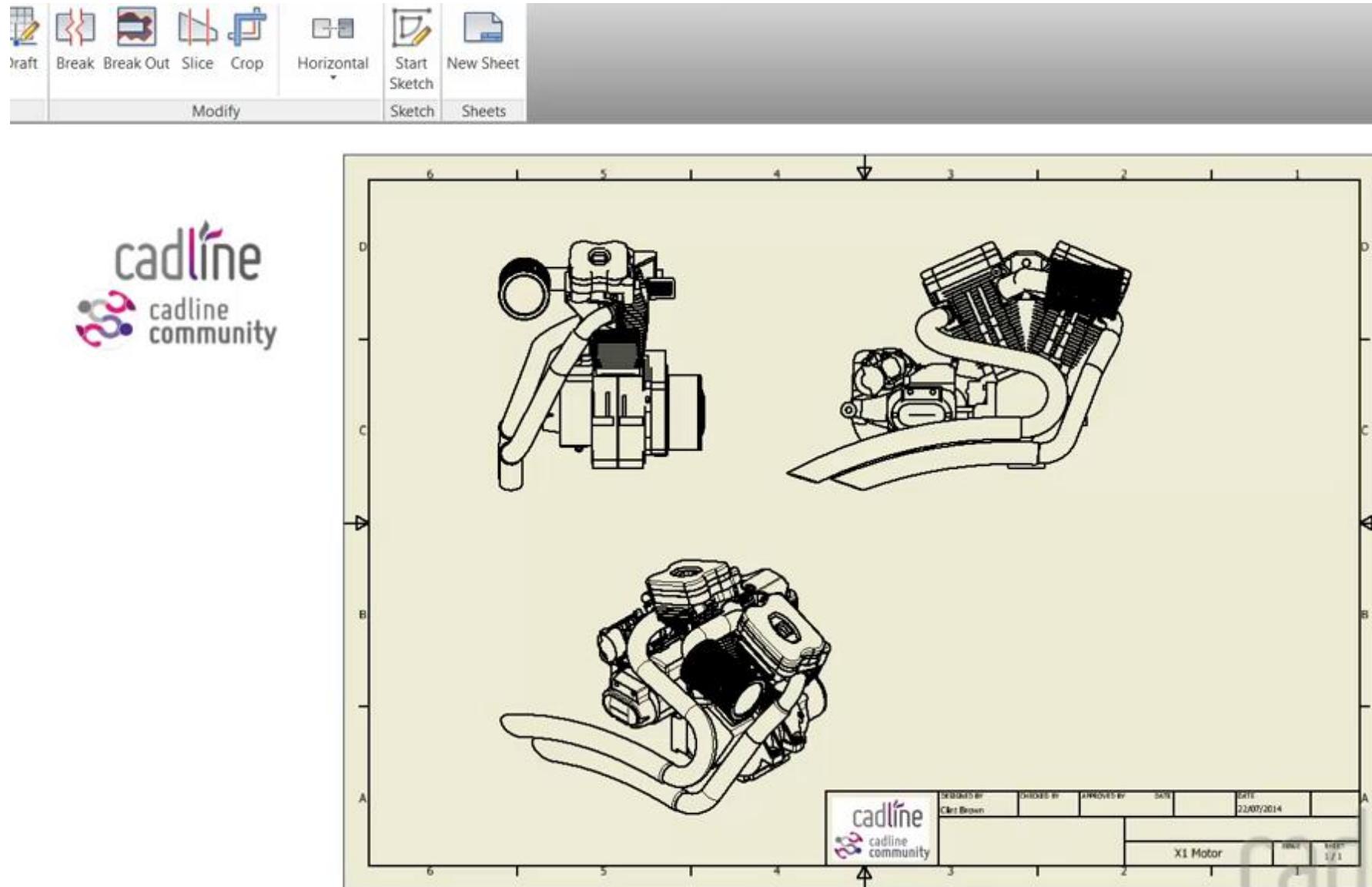


Drawings: Sketchy lines

Revit & AutoCAD 2015 introduced
“Sketchy Lines”

Open your Inventor DWG in
AutoCAD, change the view to
“Sketchy Lines”

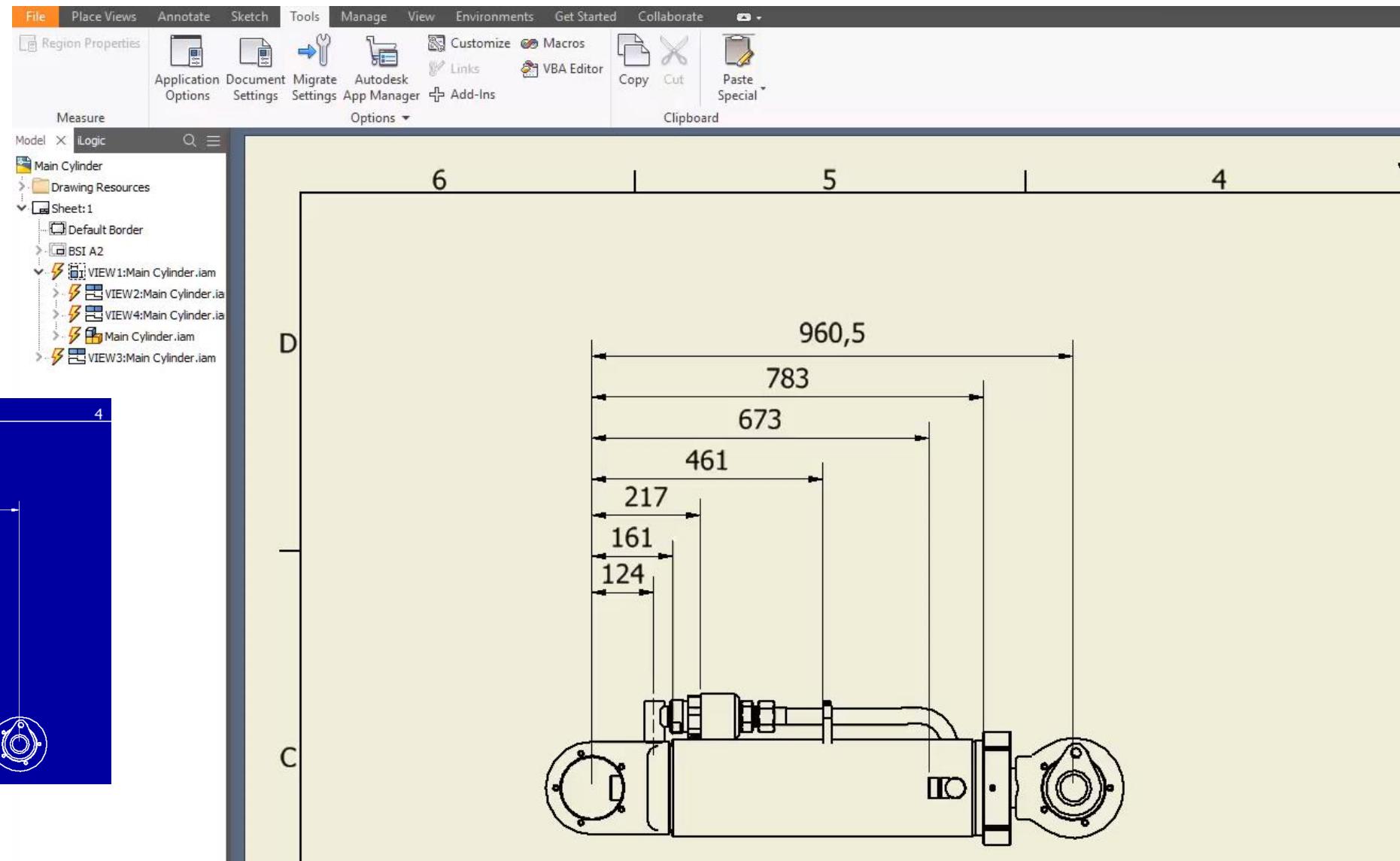
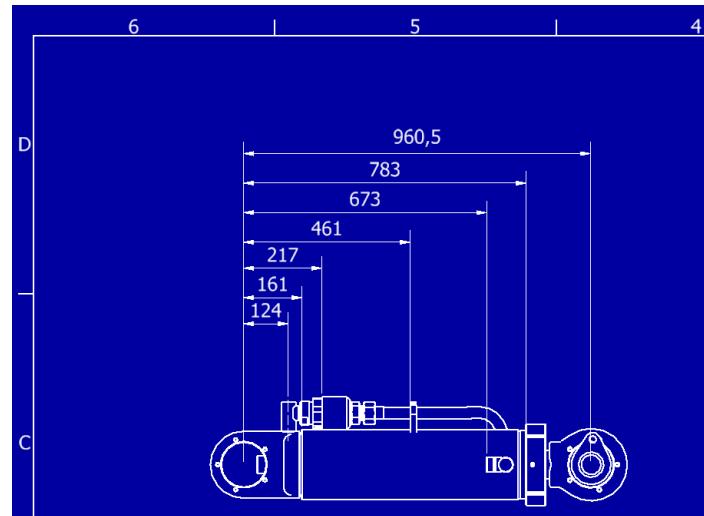
Export to PDF or Image – “Sketchy
Lines” will not show in Inventor



Drawings: Change your sheet colour

Tools > Document Settings

Note that this setting is file specific

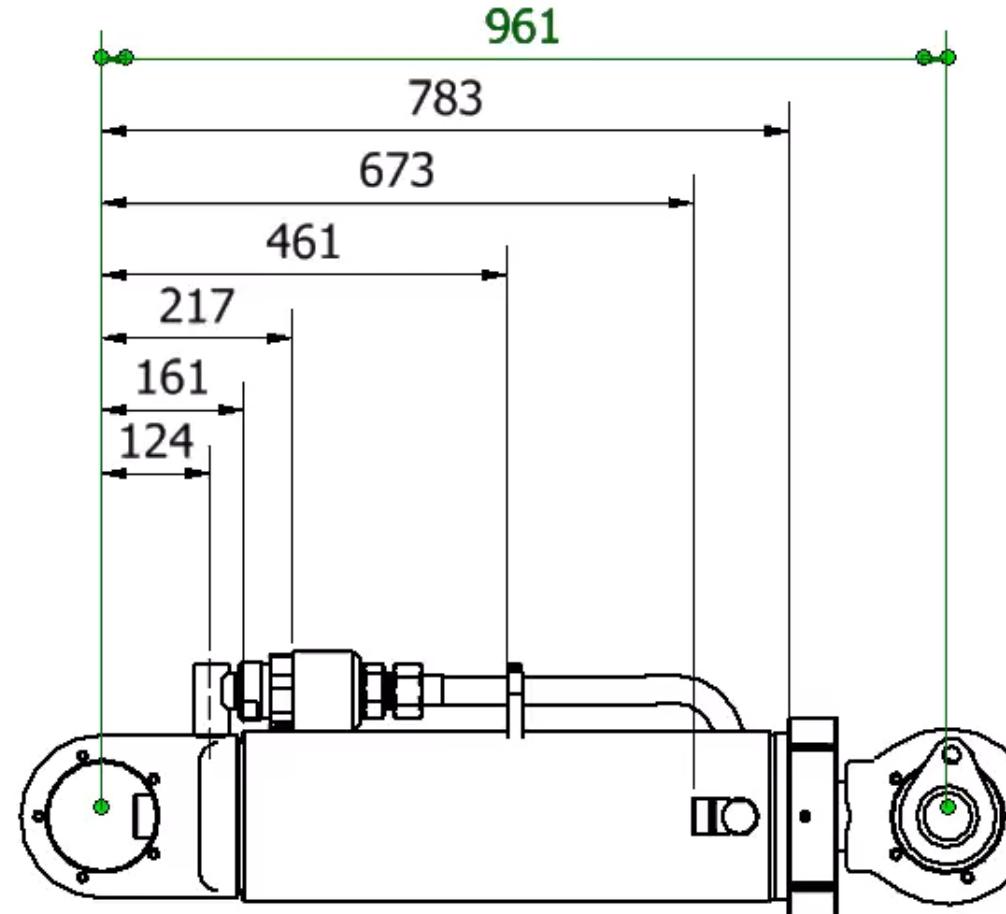
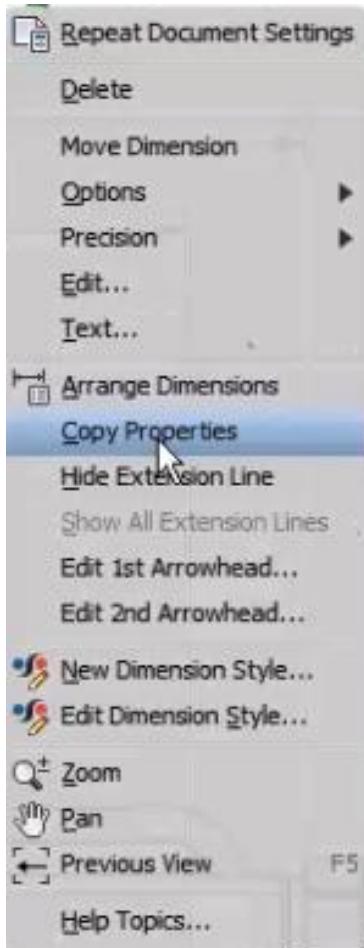


Drawings: Precision & Copy Properties

Set Precision from a right click

Similar to “MATCHPROP” in AutoCAD

“Copy Properties” of one dimension to others



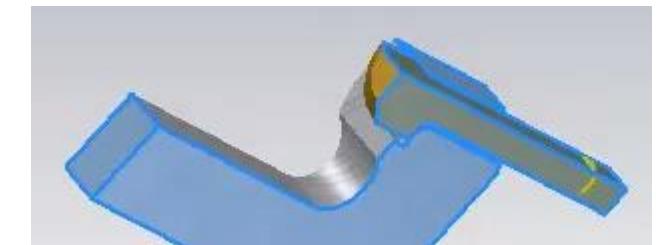
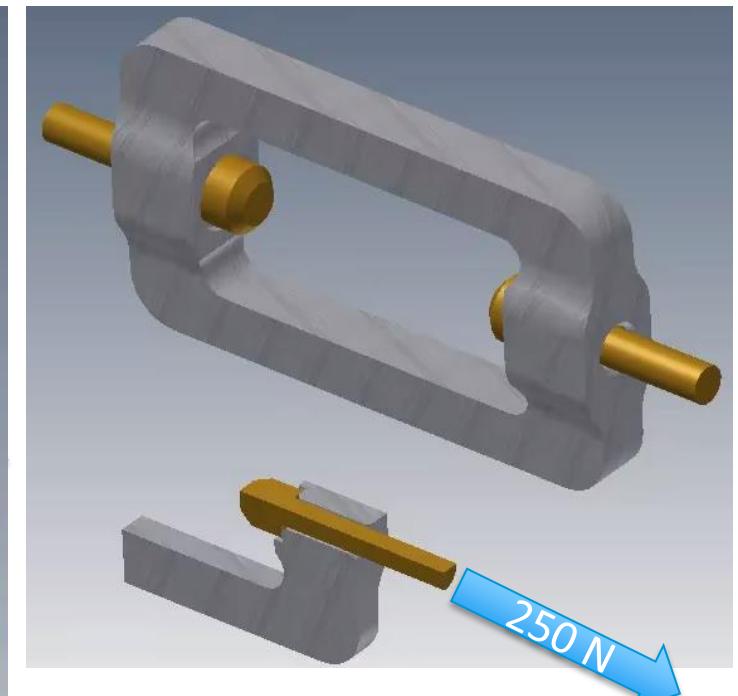
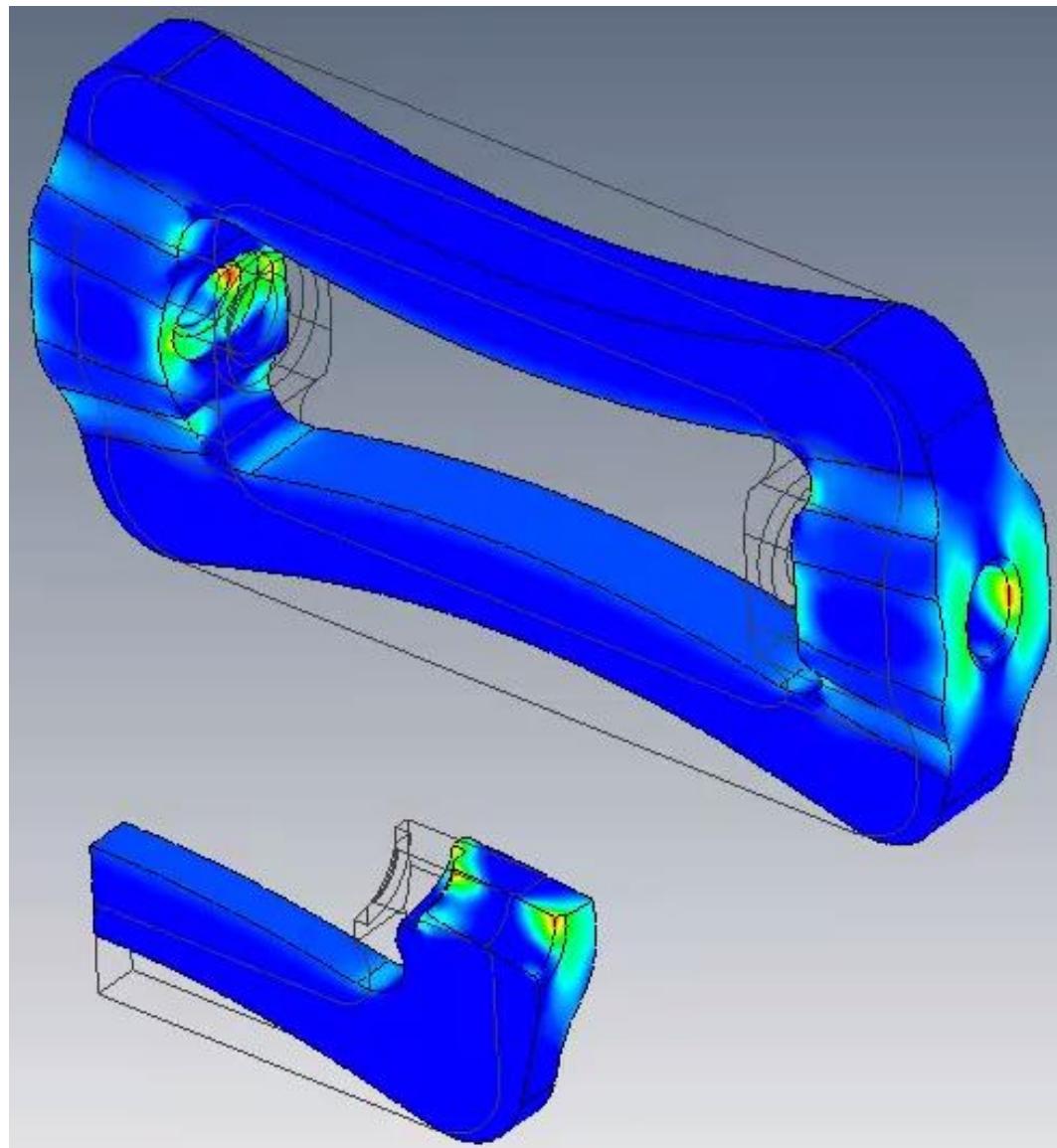
Simulation: Symmetry

Inventor does not have a Symmetry Command in the Simulation Environment

Instead use a Frictionless Constraint

Remember to check your loads! In this example, the load is divided by 4, as the model is split into 4

See the full video here:



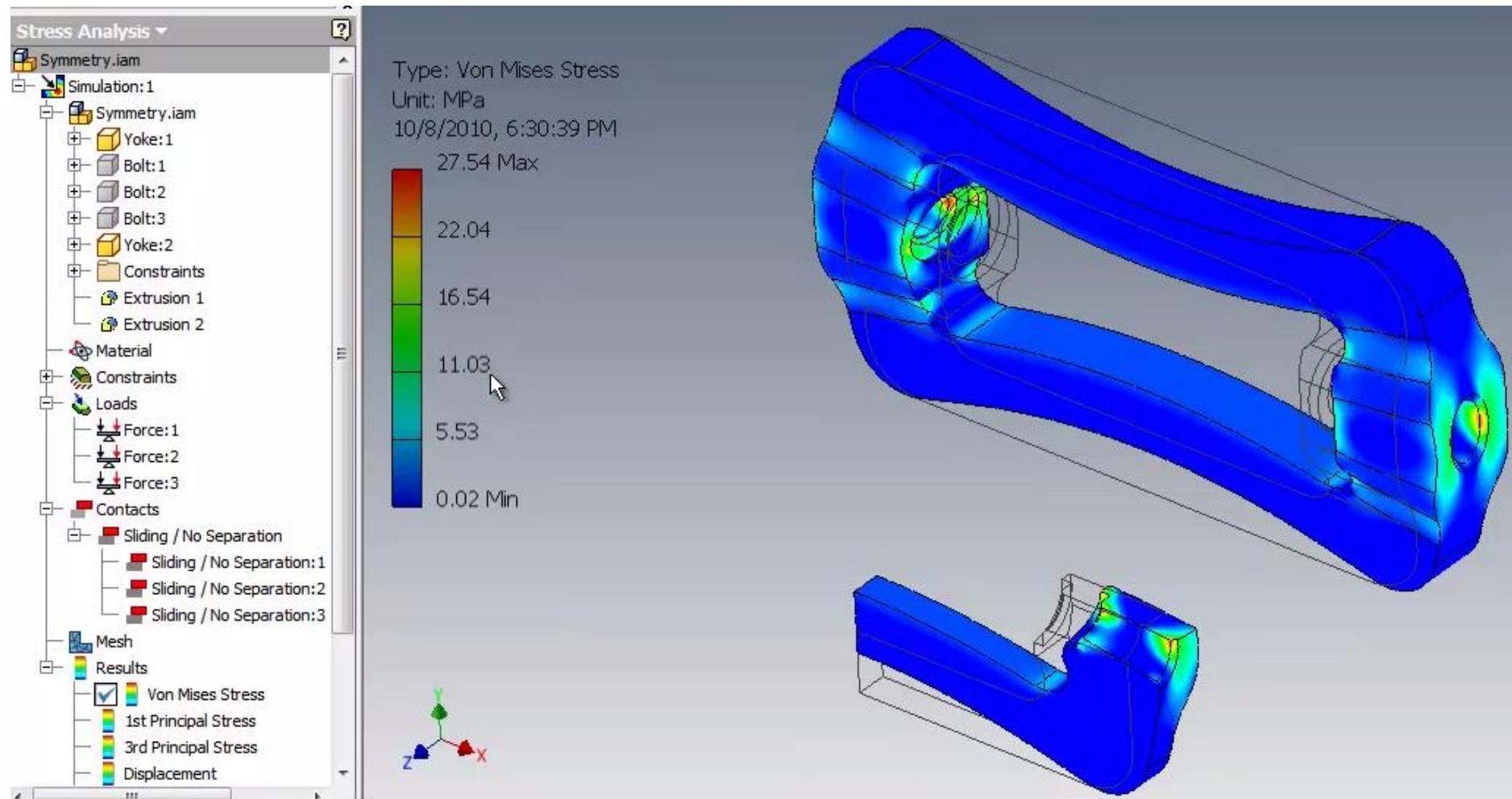
Frictionless Constraints on the faces highlighted in blue

Simulation: Component Visibility

Toggling visibility of Components:

Does not remove them from analysis

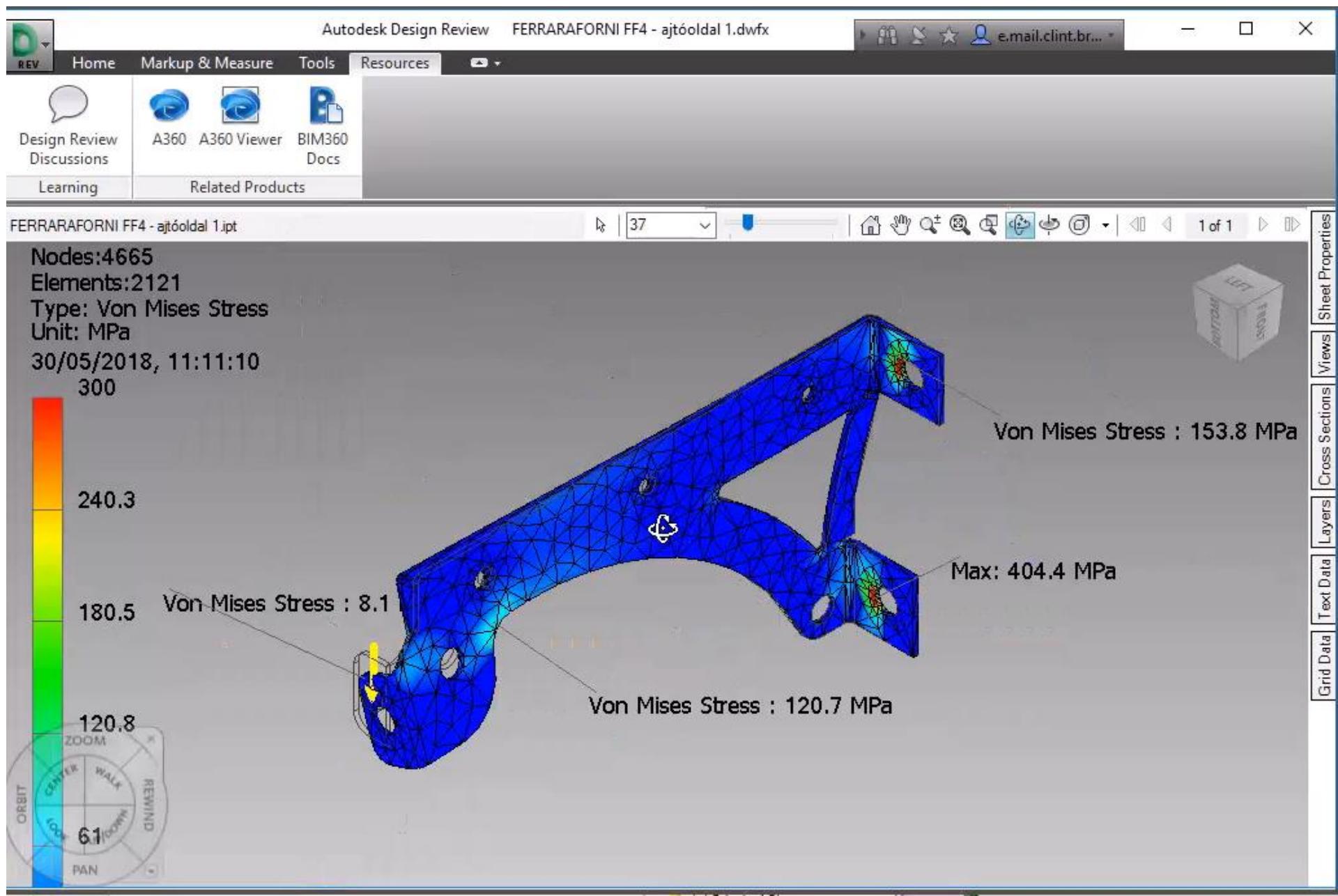
Hides them in the results, making it easier to interpret your results



Simulation: Share your results via DWF

Export out a DWF with your results showing

Note: this is not a supported workflow, and does not work with very large or complex models



Parts: Direct Edits

Use Direct Edits to modify “Dumb Geometry”, or even to make tweaks to existing designs

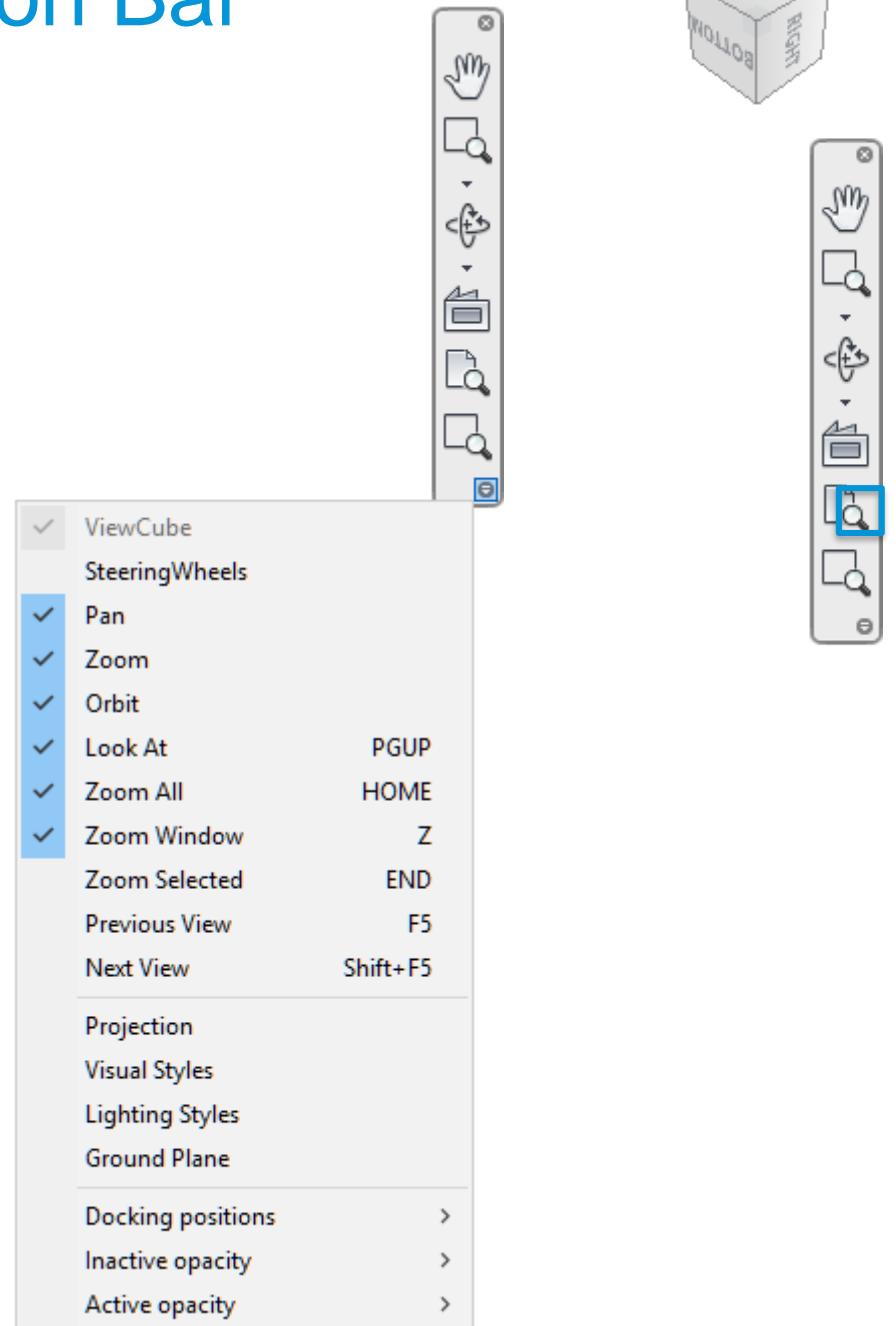
Remember all changes are parametric

The screenshot shows the Autodesk Inventor interface with the following elements:

- Top Bar:** Includes 'New', 'Open Projects', 'Open Samples', 'Home', 'Team Web', 'Help', 'Back', 'What's New', 'Learning Path', 'More', and 'Videos and Tutorials'.
- Left Panel:** 'Launch' tab with 'Model' and 'iLogic' buttons, and 'My Home' section with a 'New' button.
- Central Area:** A 'File Open' dialog box is open, showing a list of files in the 'Cadline' folder. The file 'etal.stp' is selected and highlighted in blue.
- Bottom Left:** 'Parameters' table with columns: Parameter Name, Consumed by, Unit/Type, Equation, Nominal Value, Tol., Model Value, Key, and Comment. It lists 'Model Parameters' (d2, d5, d6, d7, d8, d9) and 'User Parameters'.
- Bottom Right:** 'Reset Tolerance' buttons (blue plus, green triangle, yellow circle, red minus), 'Done' button, and 'Options...', 'Open', and 'Cancel' buttons.
- Bottom Bar:** Buttons for 'Add Numeric', 'Update', 'Purge Unused', 'Link', and 'Immediate Update'.

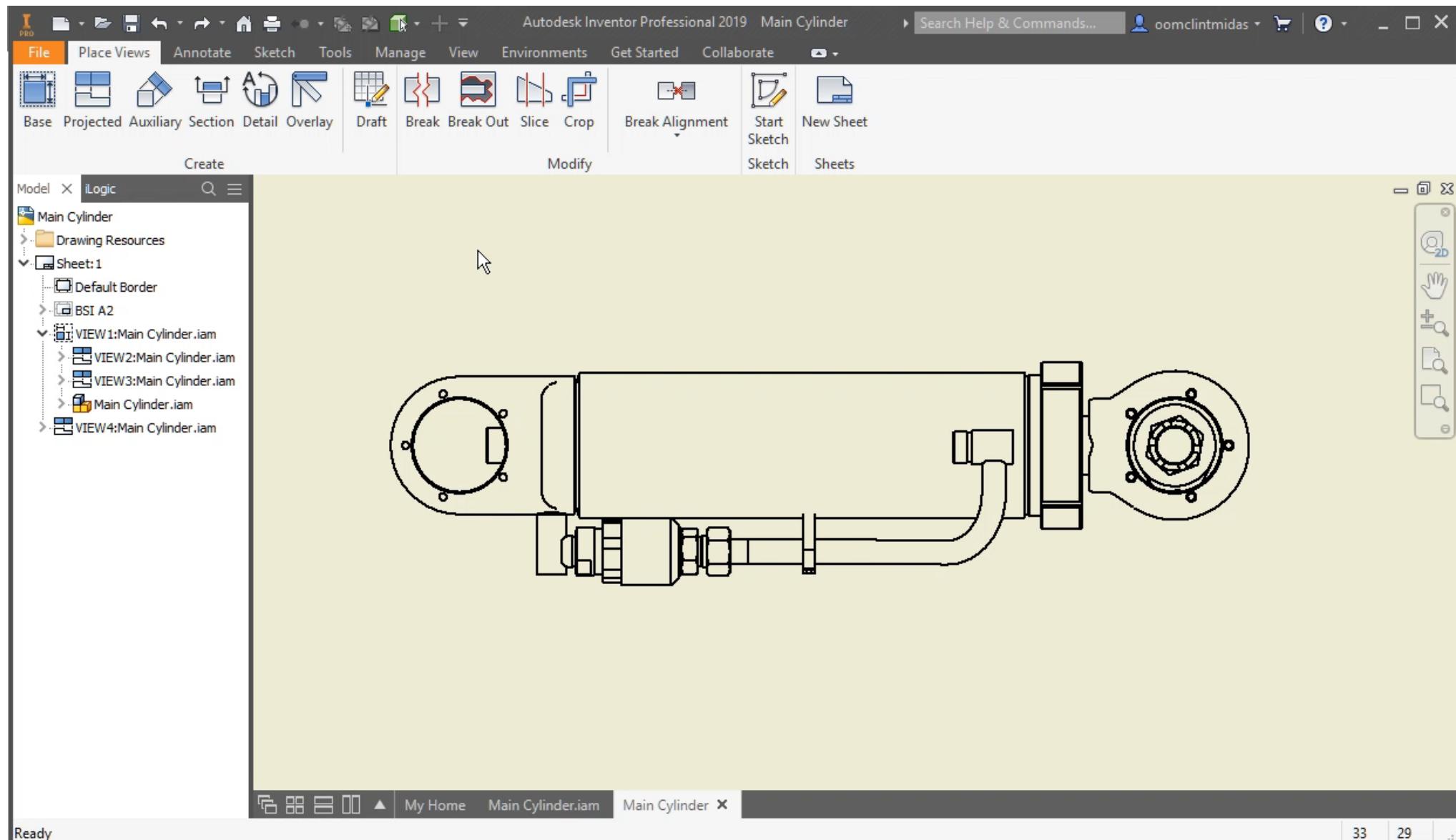
Customise the Navigation Bar

Add the navigation tools that you use most commonly to the Navigation toolbar



Drawings: Set Transparency

Inventor 2016 introduced
“Component
Transparency” in the
drawing environment.

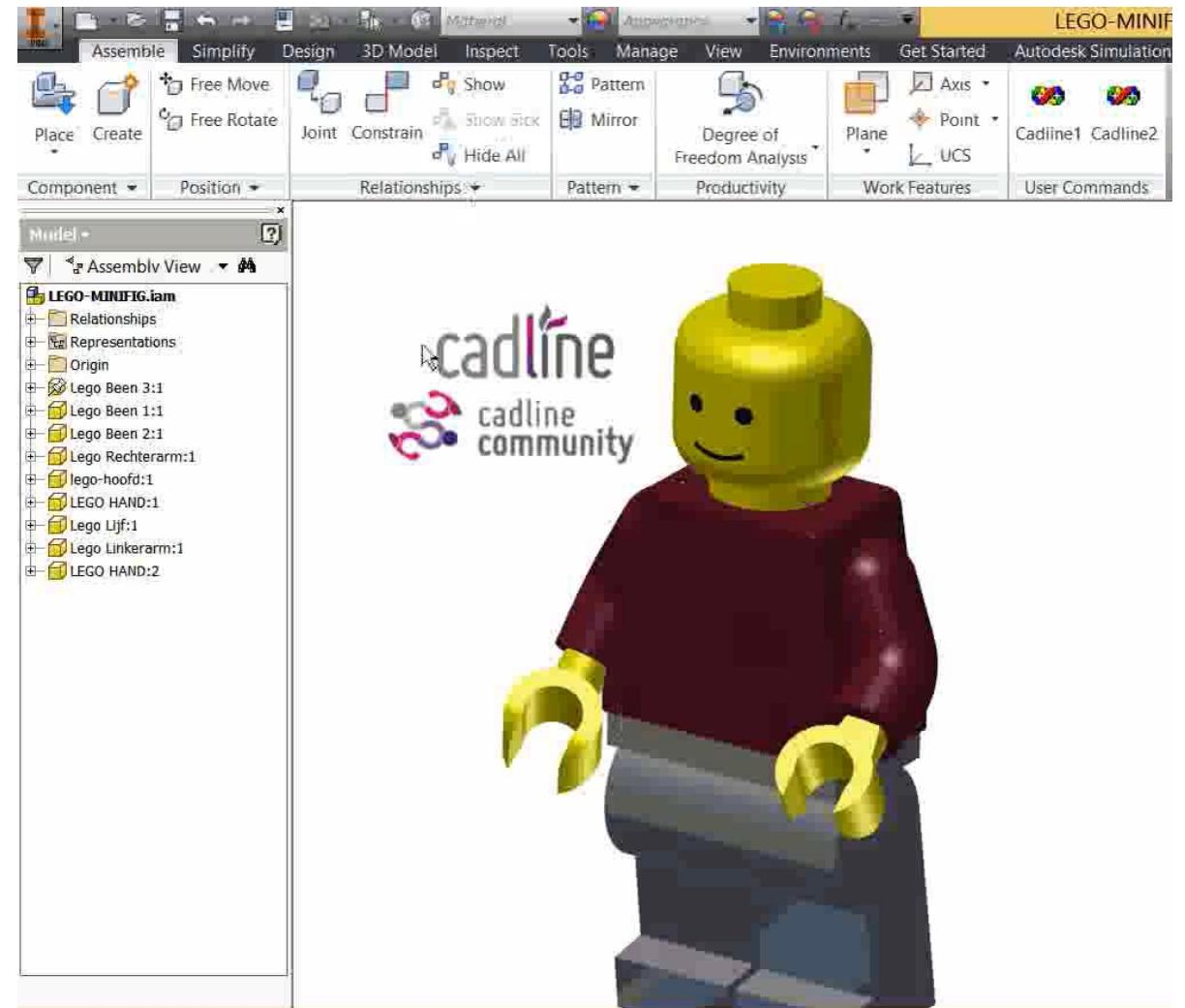


Assemblies: Degrees of Freedom Analysis

This tool is located under the Productivity Panel of the Assemble Tab.

When you click on ***"Degree of Freedom Analysis"*** a table will appear, showing all of the parts in the assembly.

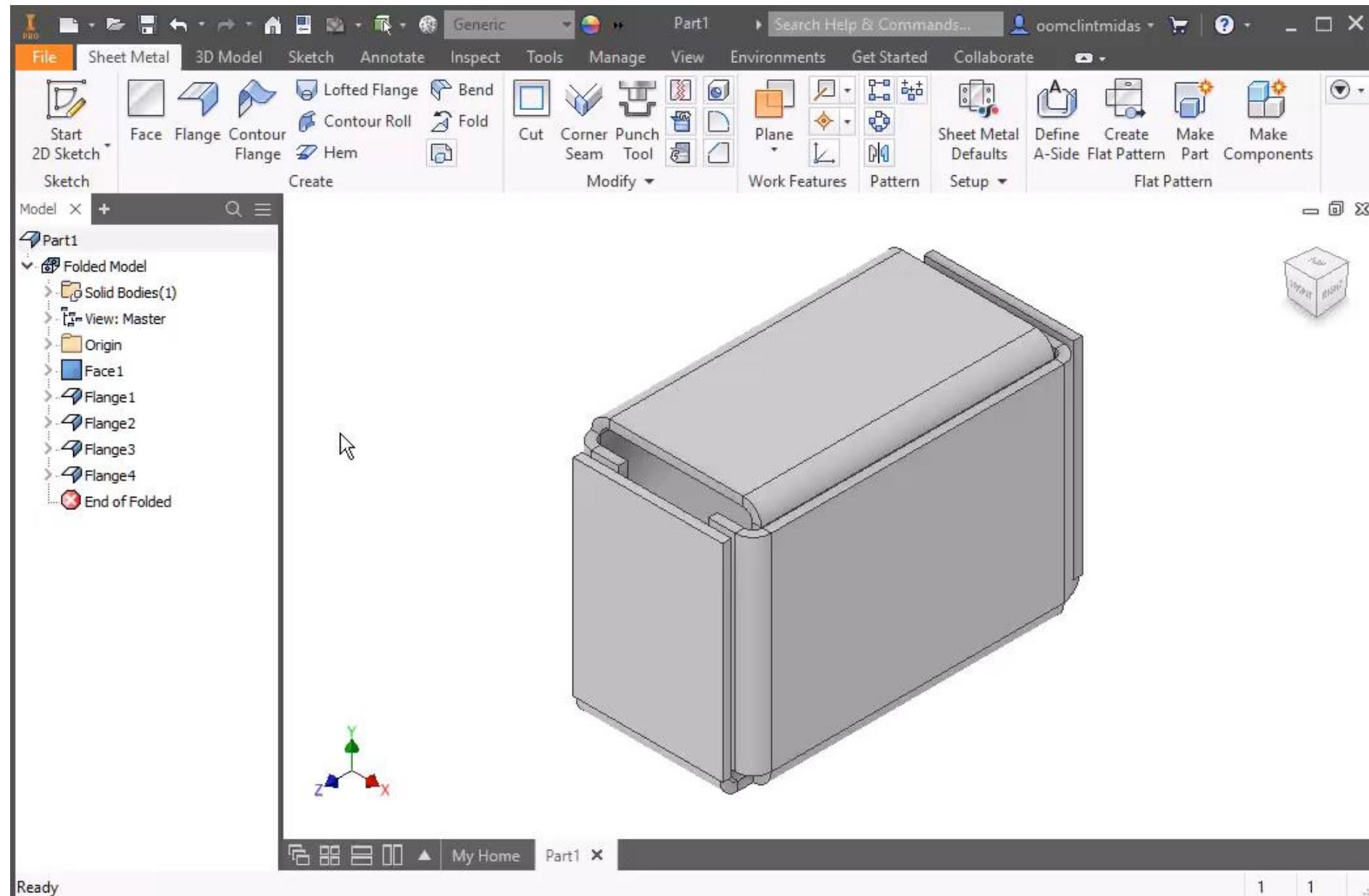
If you tick the Animate Freedom box, animations of the available movement of the part will be shown on screen



Code Free iLogic

Forms:

Perhaps one of the most overlooked areas of iLogic is the forms interface. Forms allow you to easily interact with iProperties and Parameters in your models and drawings. If you still use prompted entries to enter critical design information, you may want to consider changing to a more modern looking form based interface.



iLogic: Some Best Practice

Keep things simple:

Don't try to do everything in one rule.

Comment your code:

- ‘Adding comments to your code, will help your colleagues, or your memory when reviewing code later

Use External Rules where possible:

- Far easier to update if things change. Embedded rules are difficult to “Fix”

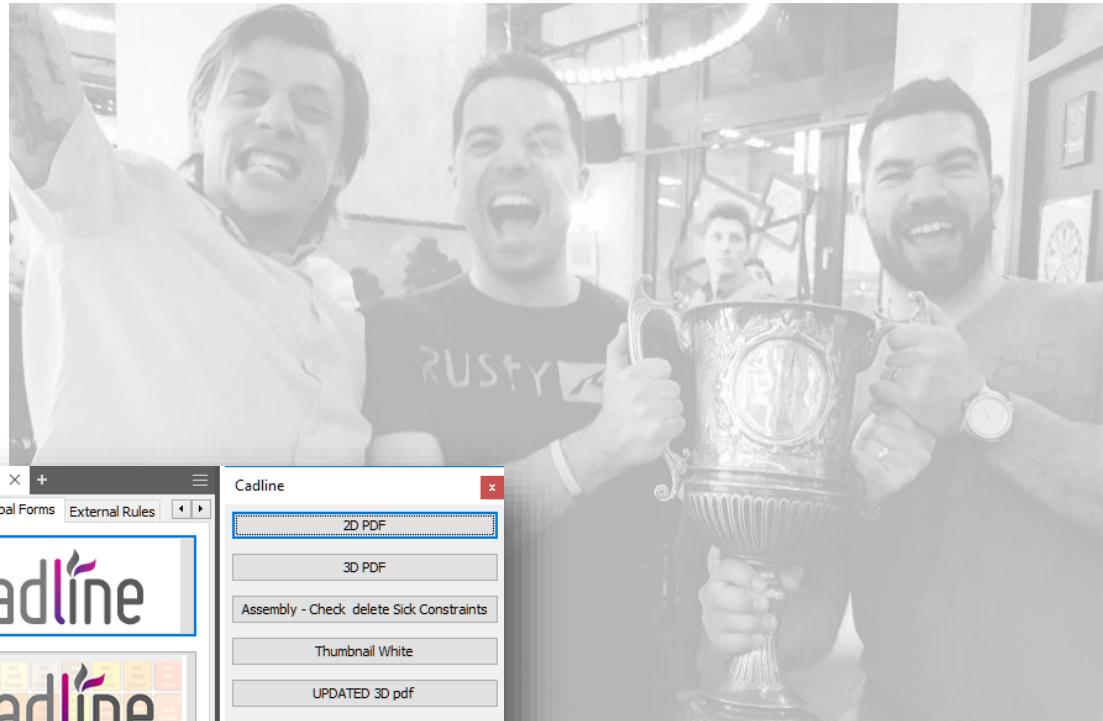
2 disadvantages – 1: The rules don’t run on parameter changes
2: The rules are not embedded in the files if sharing externally

Use forms to link external rules

- Add rules to your forms, these can be pushed like buttons

Event Triggers:

- Running rules “After Save” - ensure that these do not change the file (Ok for file exports (STEP etc.) not good for changing iProperties or parameters, as the file will need to be updated



Drawings: Fixing Overridden Dimensions

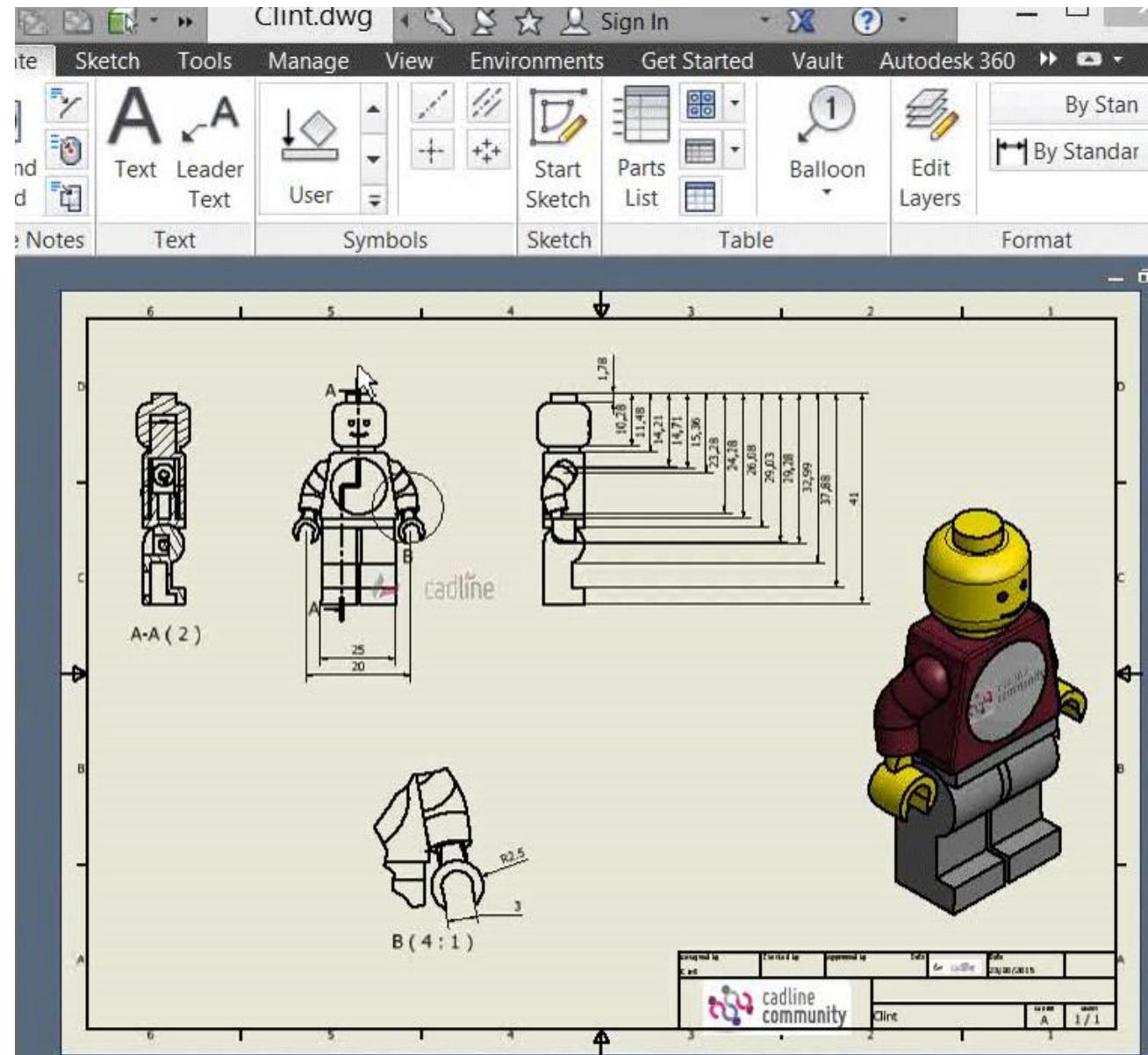
Choose “**Select All Overridden Dimensions**” tool from the selection set dropdown

Right Click and choose “**Edit**” this will bring up the Edit dimension dialogue.

Delete the overridden value and then **untick** the “**Hide Dimension Value**” box.

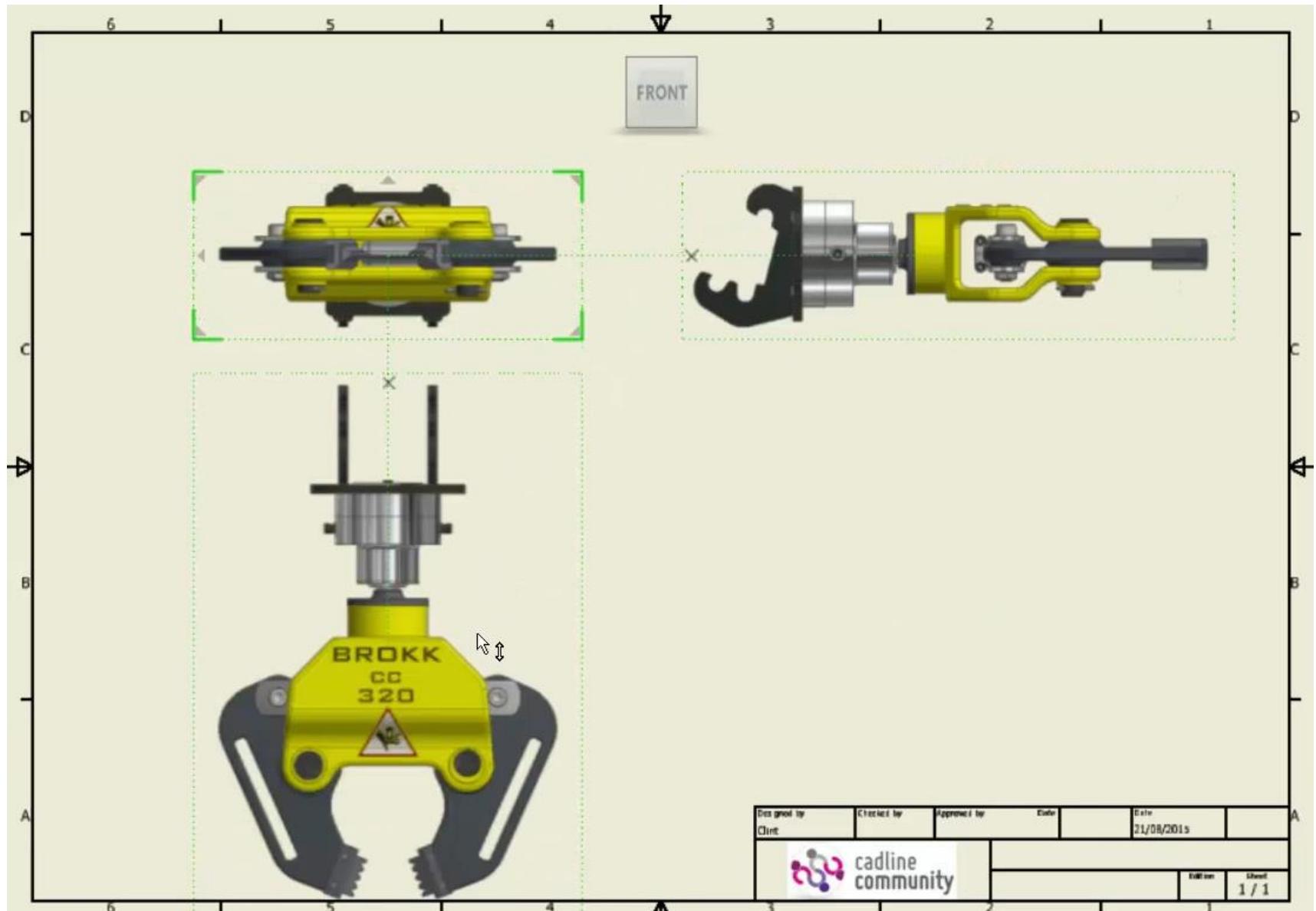
To cycle through all of the dimensions, Right Click and select “**Repeat Select All Overridden Dimensions**” then **Right Click** and choose “**Edit**”

Repeat the process until all of the offending dimensions have been corrected.



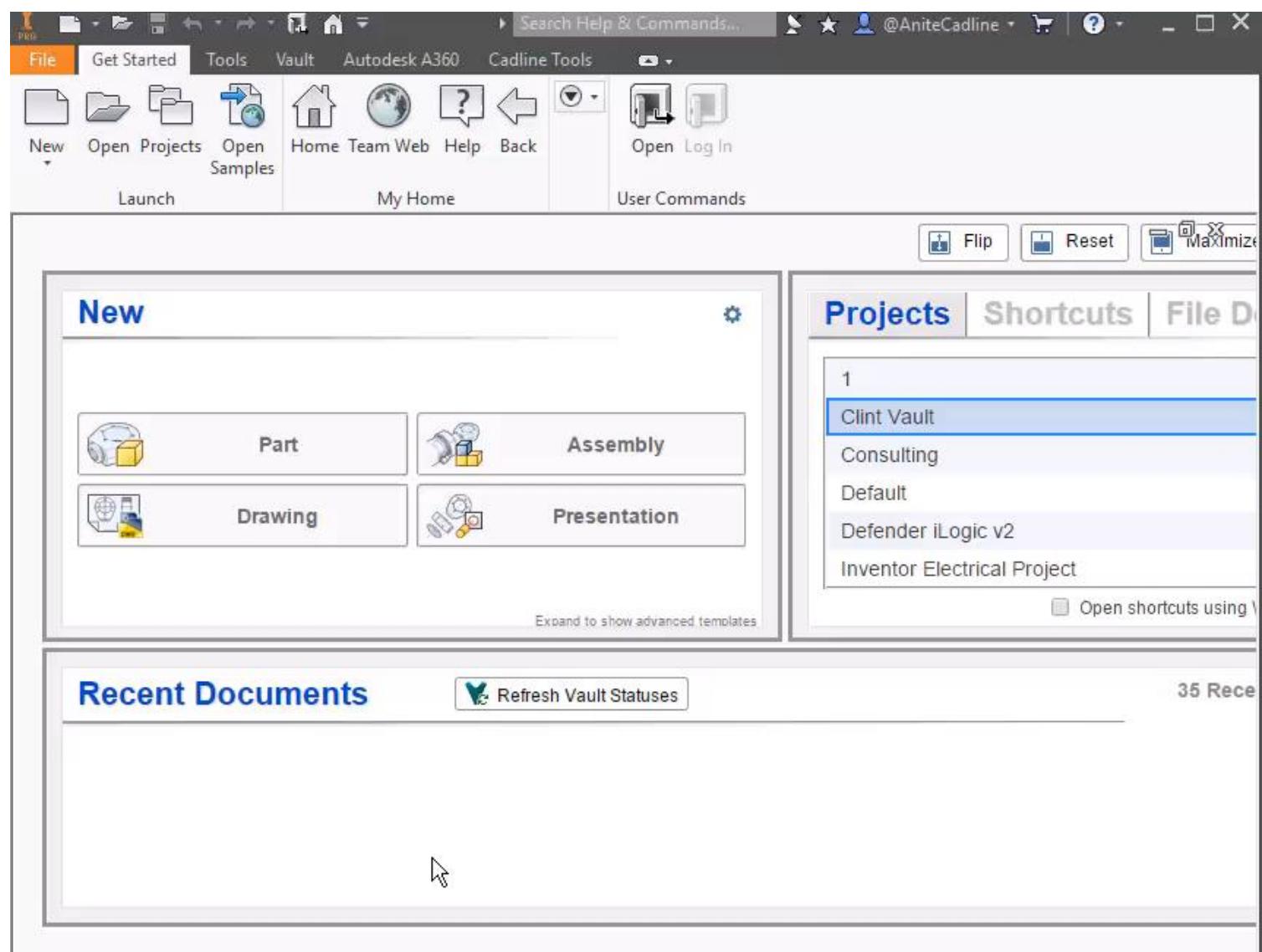
Drawings: View Scales

Set the scale of the views on the drawing by simply dragging the corner of the Base view.
(introduced in 2016)



Open from Vault Tips

- Use the “Browse Vault” Button on the “file> Open” dialogue
- Add “Open from Vault” to the quick Access toolbar
- Add “User Commands” to “Log In” and to “Open” from Vault





Questions ?

@ClintBrown3D



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