

# Need for Speed: 102 Productivity Hacks for Inventor

Clint Brown

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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



## About the Speaker



# Some Stats

5

AU CLASSES  
PRESENTED

Presented at:  
AU Las Vegas 2013 x2  
AU London 2017 x1  
AU London 2018 x2

10

AU'S ATTENDED

First AU in 2010

21

YEARS USING  
AUTODESK PRODUCTS


First used AutoCAD  
R10 in 1997

10

YEARS OF  
INVENTOR & VAULT

First used Inventor  
R12 in 2008

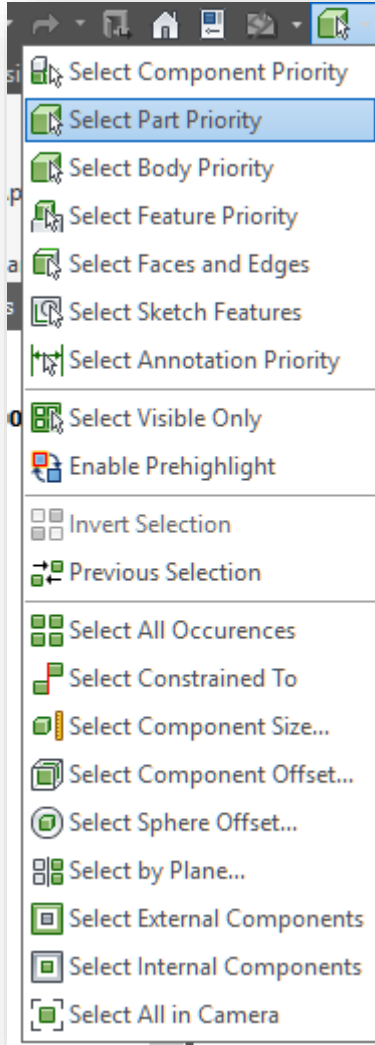




# Need for Speed: 102 Productivity Hacks for Inventor

@ClintBrown3D

# Selection Filters



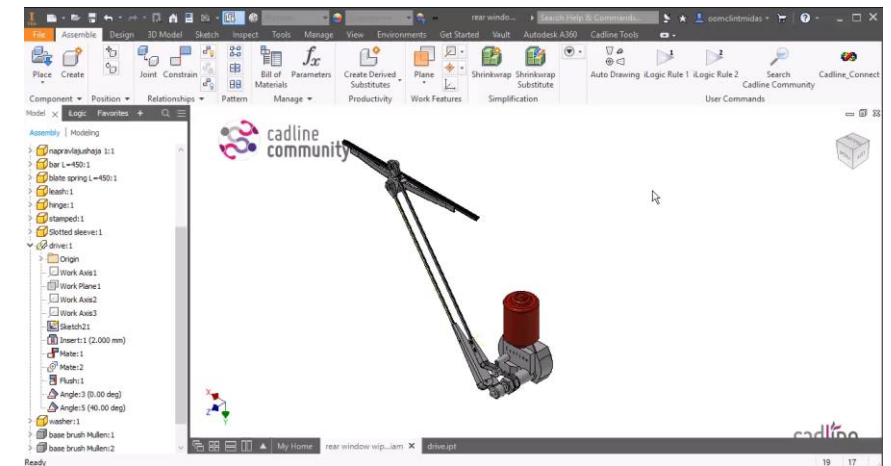
Each Inventor environment has a default Selection priority

**Part:** Faces and Edges

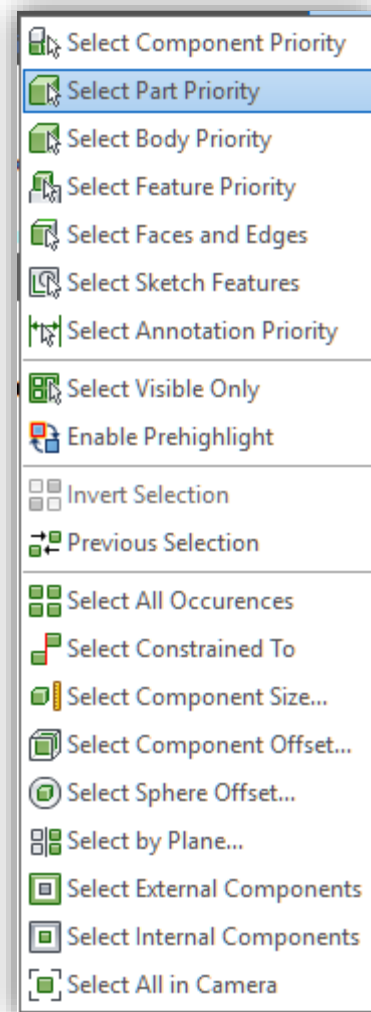
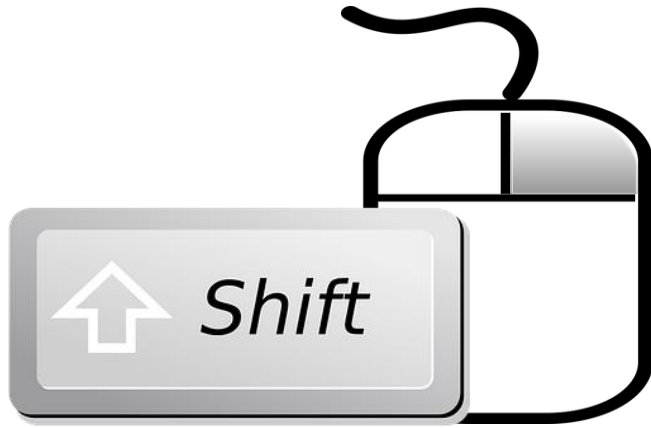
**Assemblies:** Components

**Drawings:** Edge

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



# Selection Filters

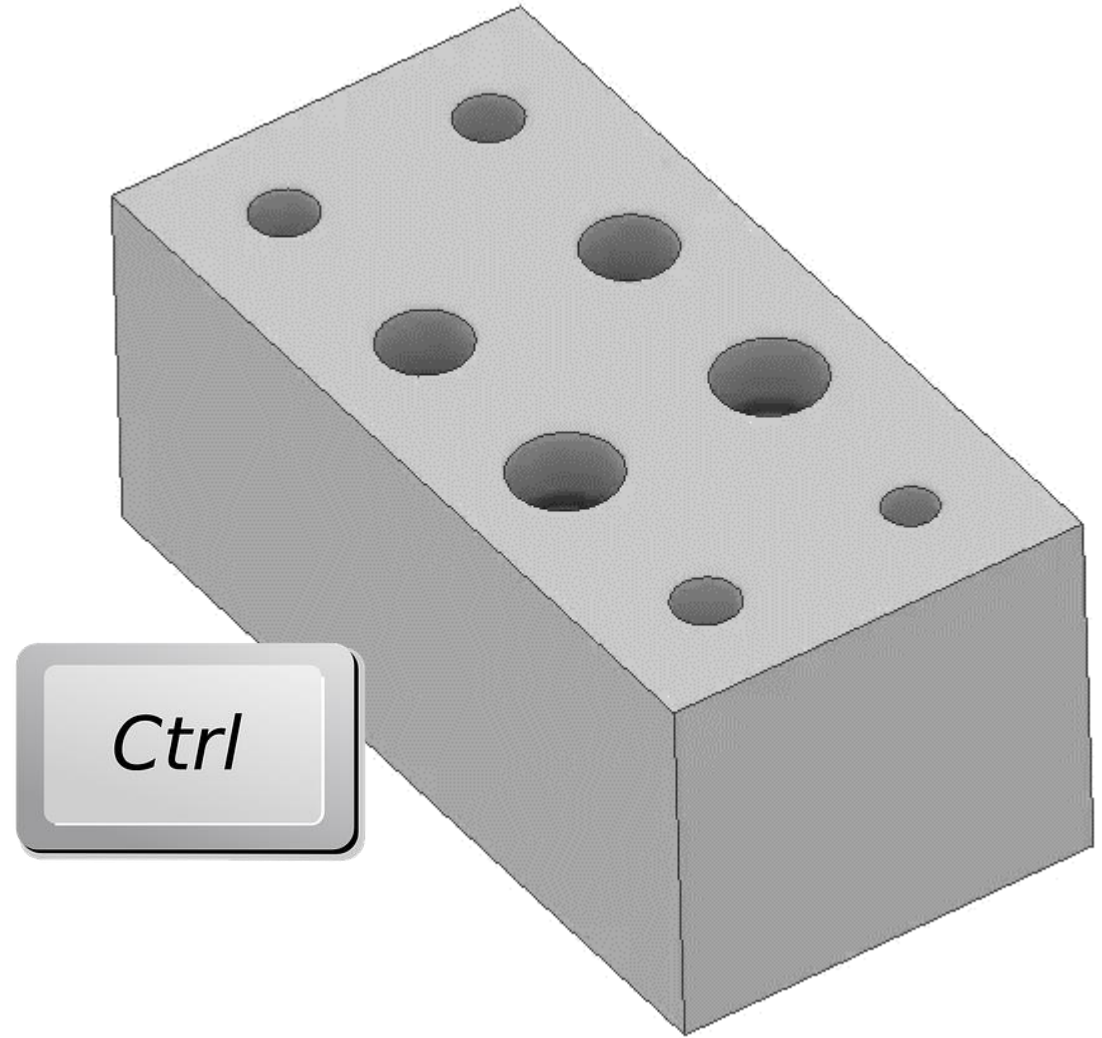
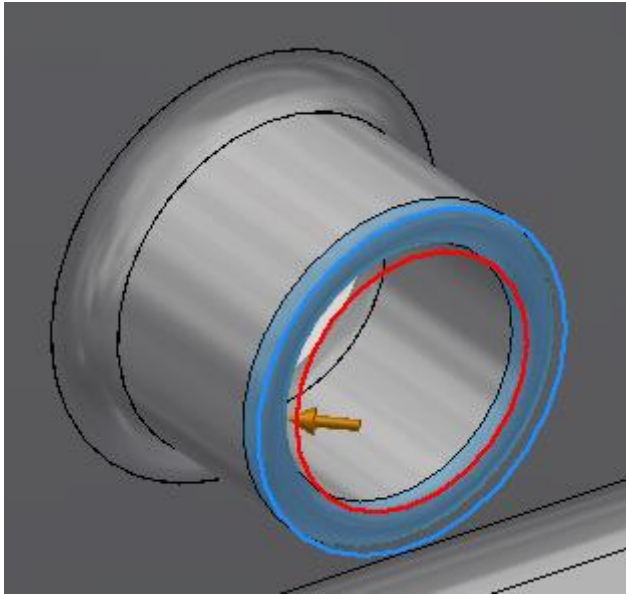


Access “Selection Filters”:

Shift + RMB

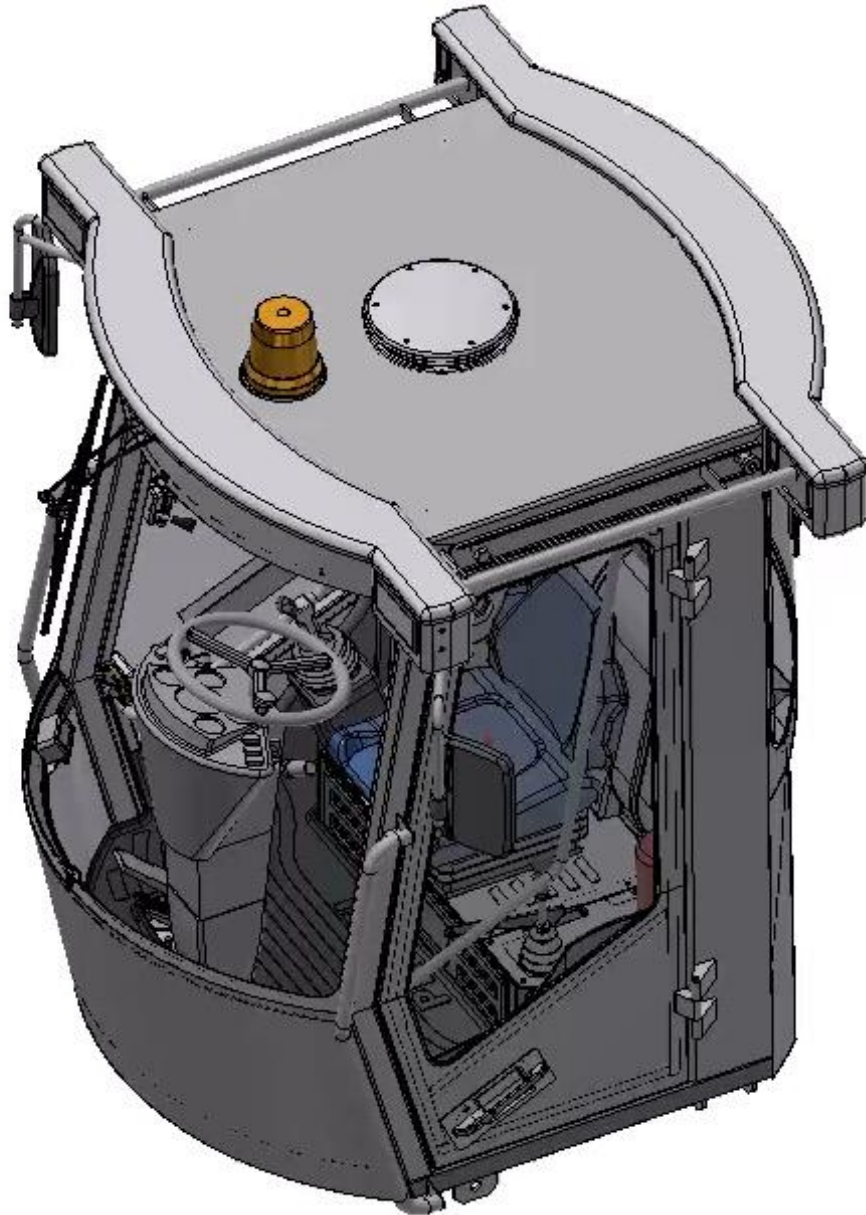
# Ctrl to De-select objects

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





# Assemblies: Isolate



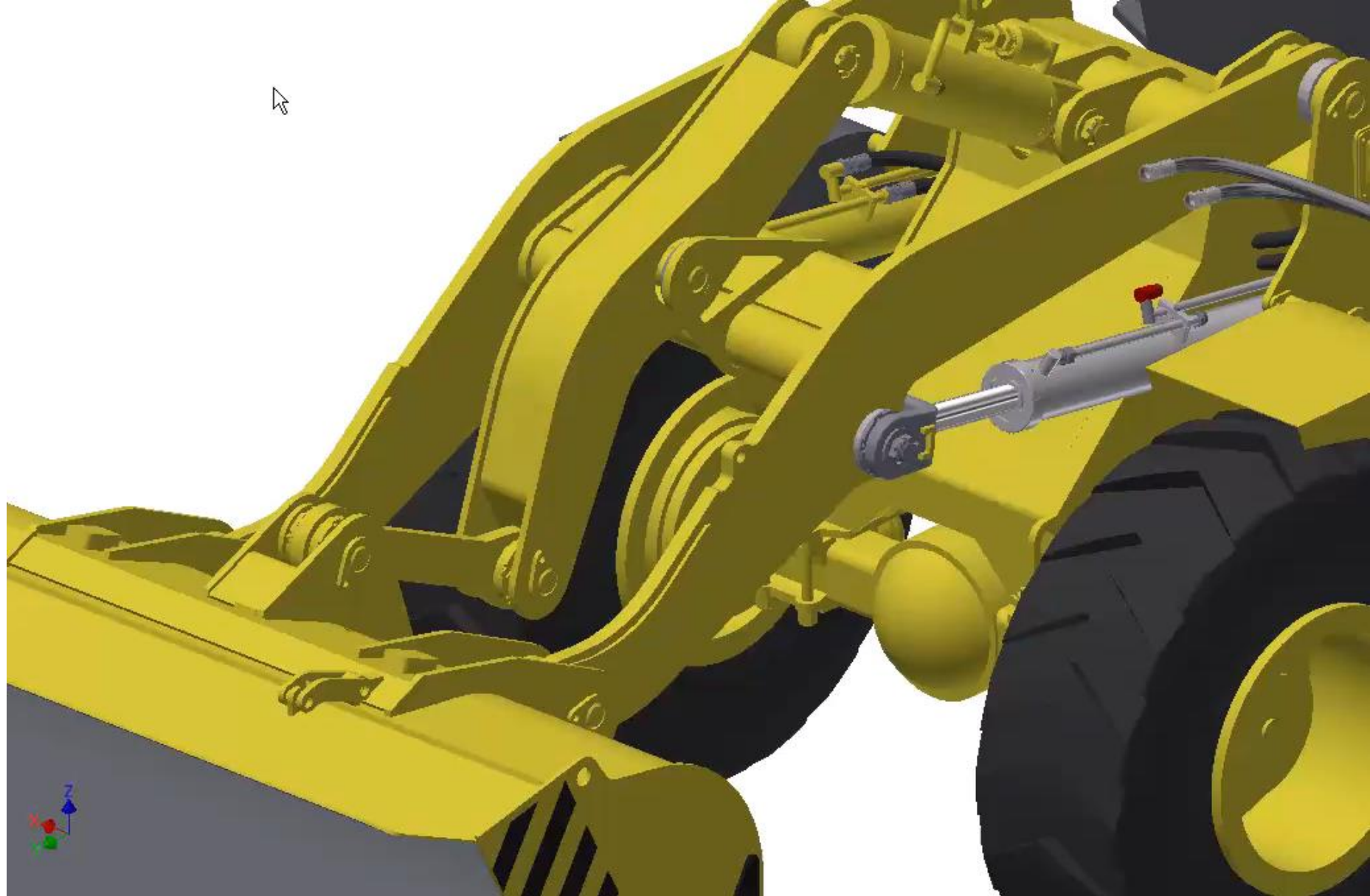
**Isolate** is often overlooked. In assemblies, use it to show and hide components, this is especially useful when working on large assemblies, or when you need to hide a specific part, or if you just want to look at a specific set of parts

# Box Select

**Left to Right**  
Everything fully inside the box

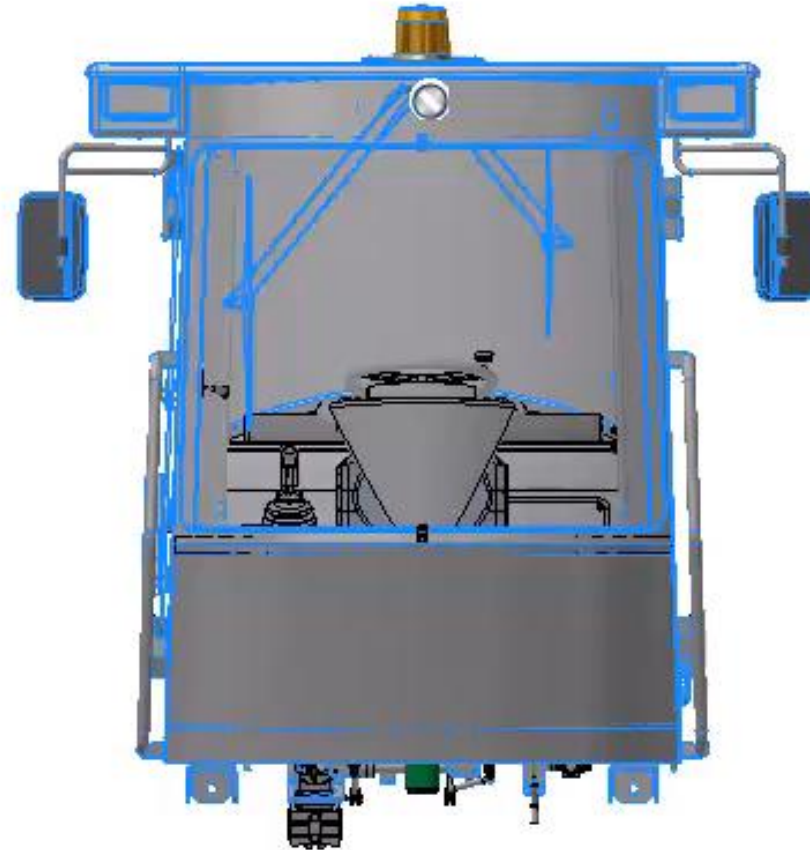


**Right to Left**  
Everything touching the box



# Assemblies: Invert Selection

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

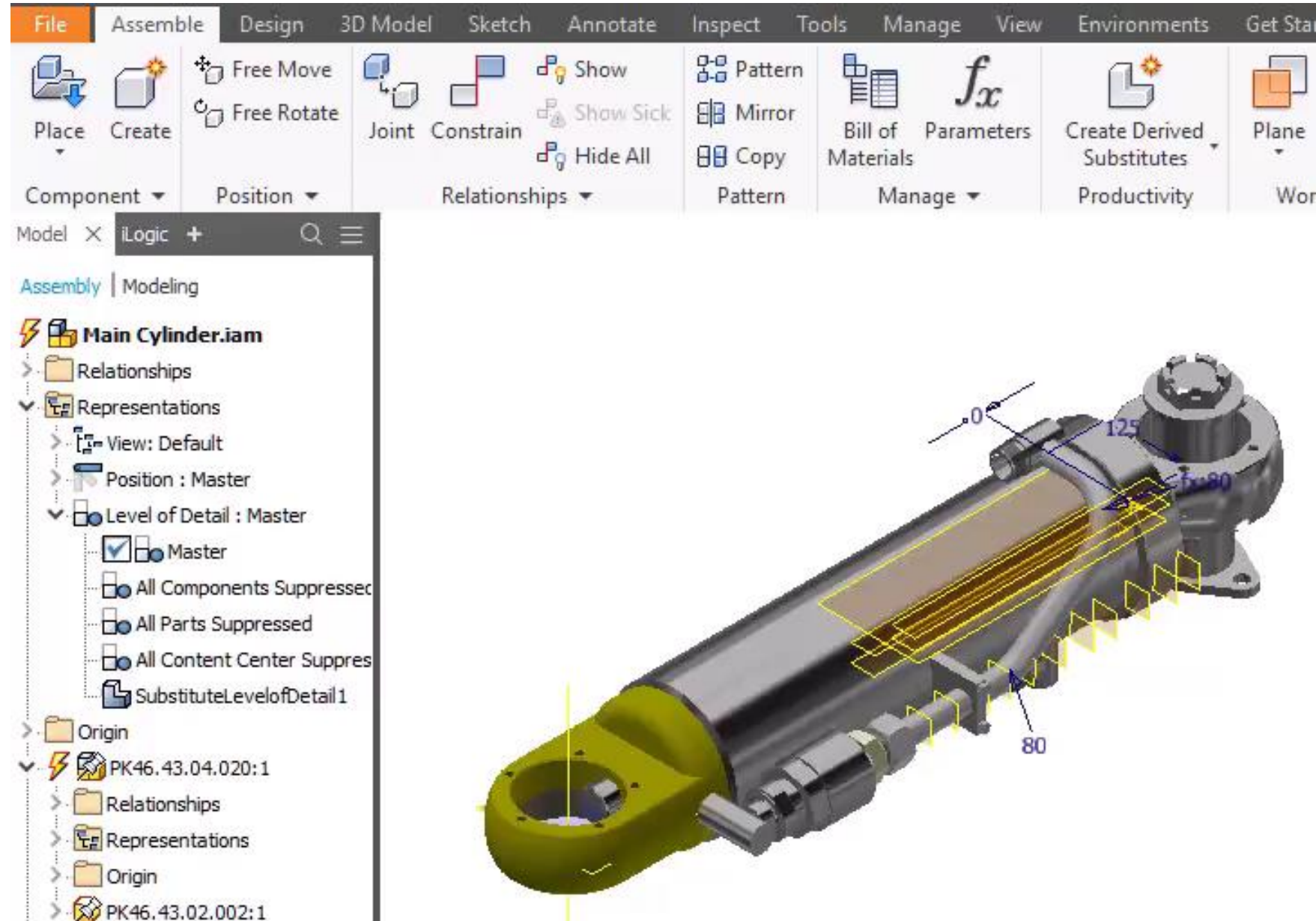




# Object Visibility Filters

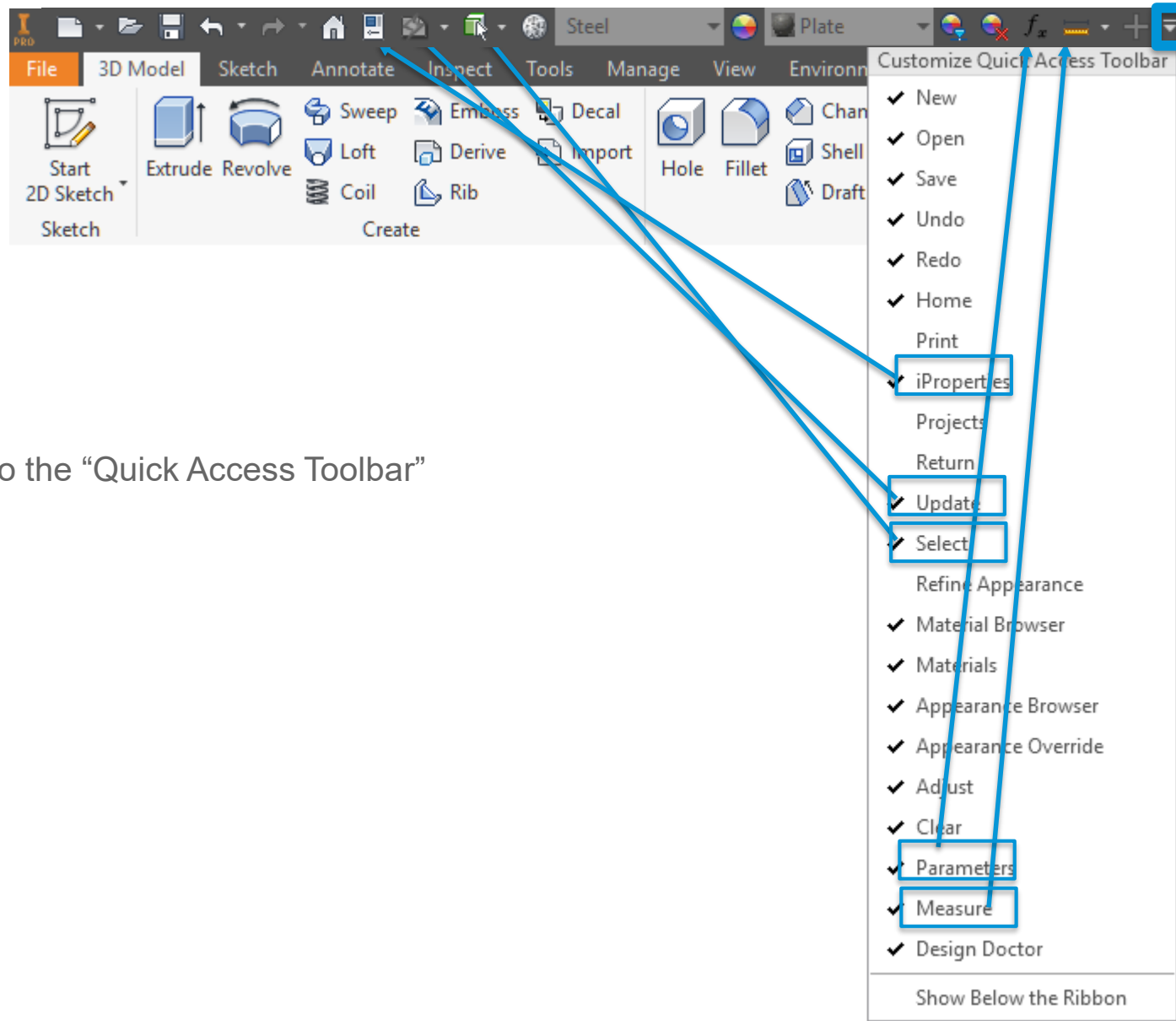
Toggle visibility of Workfeatures, sketches etc.

- All Workfeatures
- Origin Planes
- Origin Axes
- Origin Points
- User Work Planes
- User Work Axes
- User Work Points
- Construction Surfaces
- 2D Sketches
- 3D Sketches
- Sketch Dimensions
- 3D Annotations
- Component 3D Annotations
- Welds
- Weldment Symbols
- UCS Triad
- UCS Planes
- UCS Axes
- UCS Points



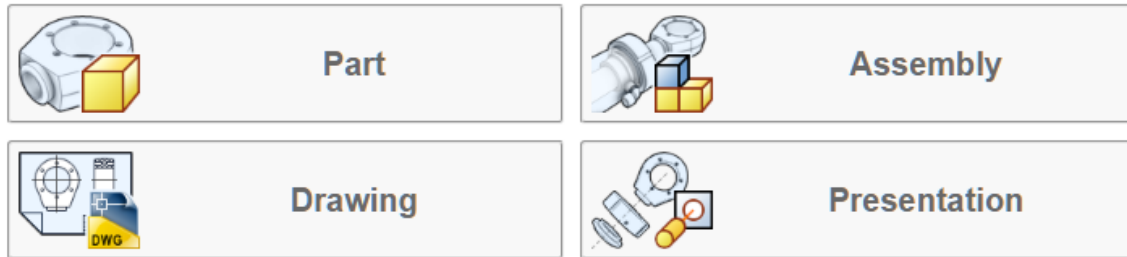


# Speed: Use the quick access toolbar

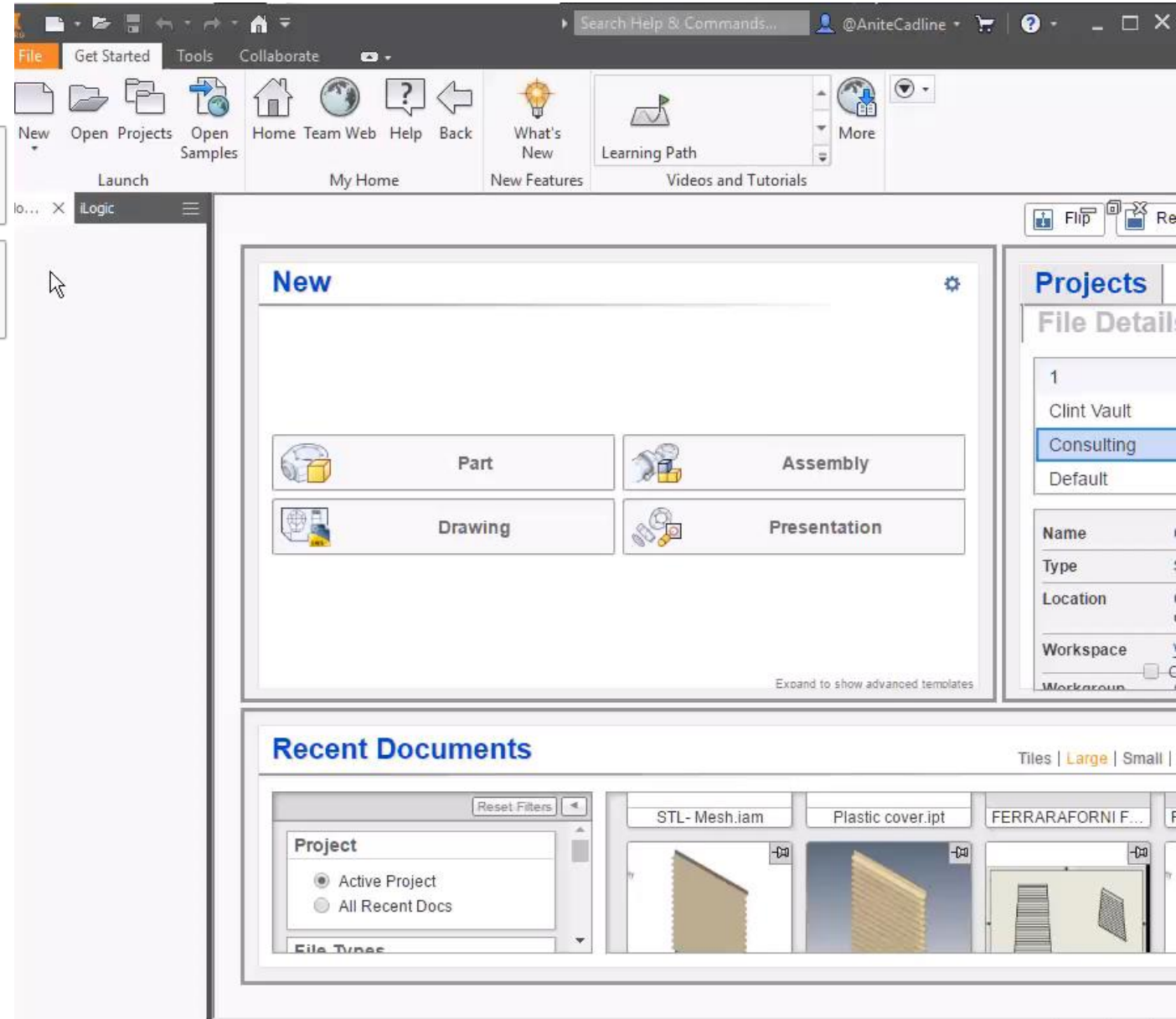


Add commonly used tools to the “Quick Access Toolbar”

# Templates



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

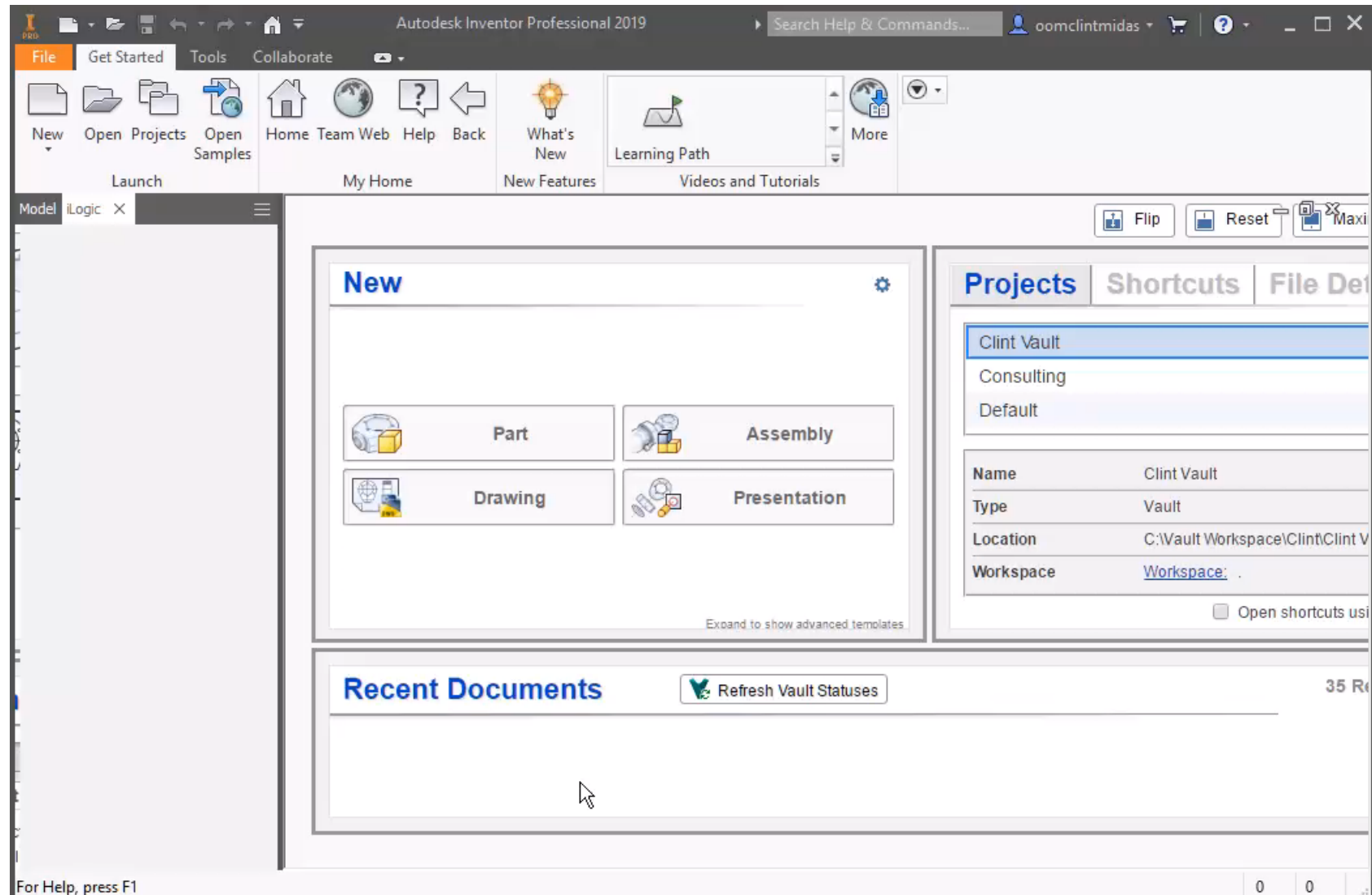


# Speed: Smart Templates

- 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

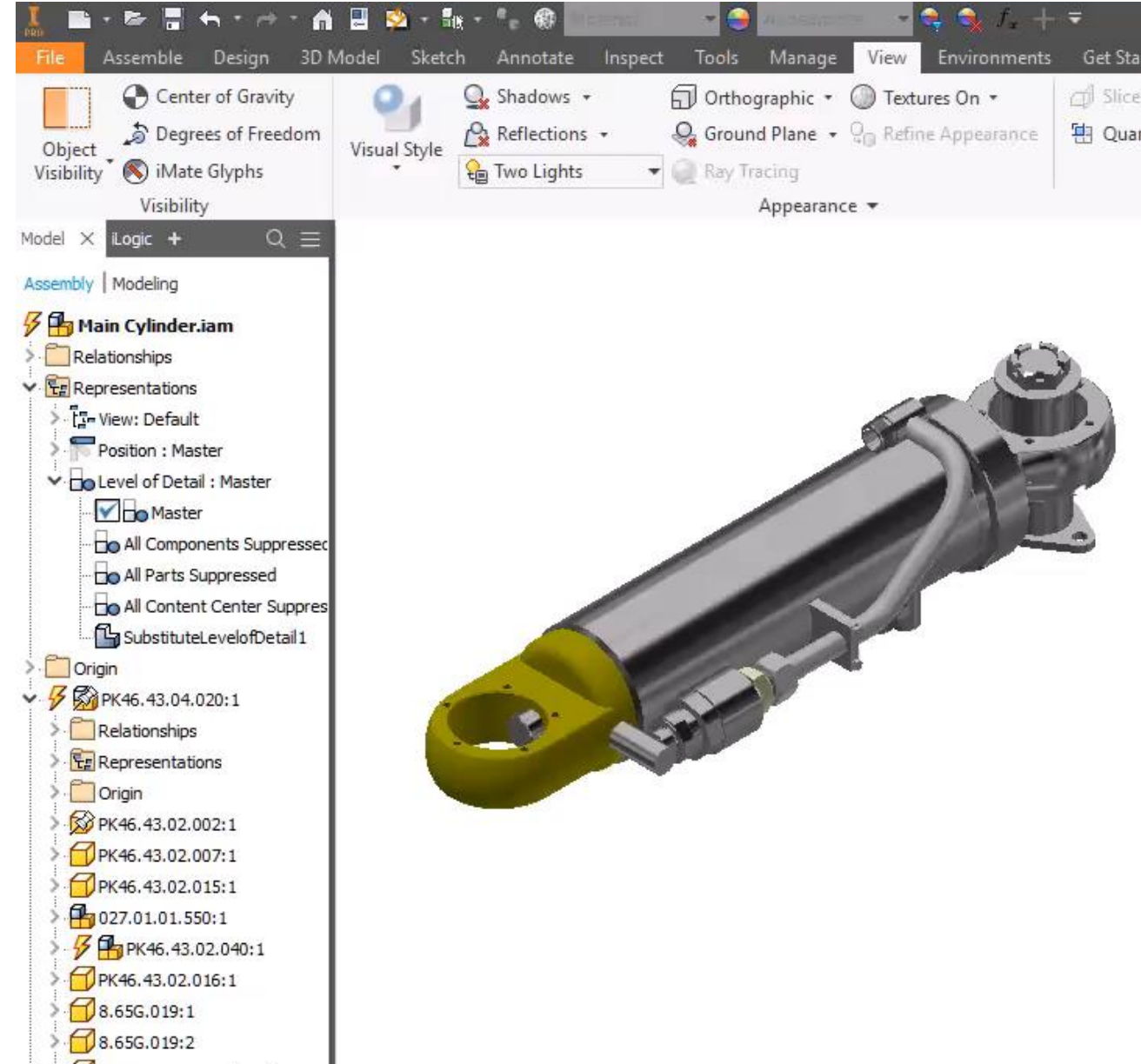
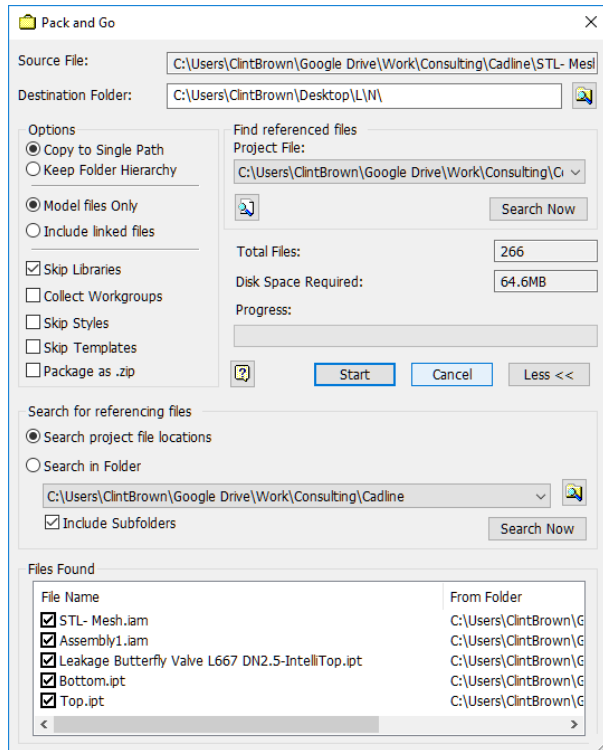
- Place virtual components in your template file
- Setup your BoM rows in the template file



# 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

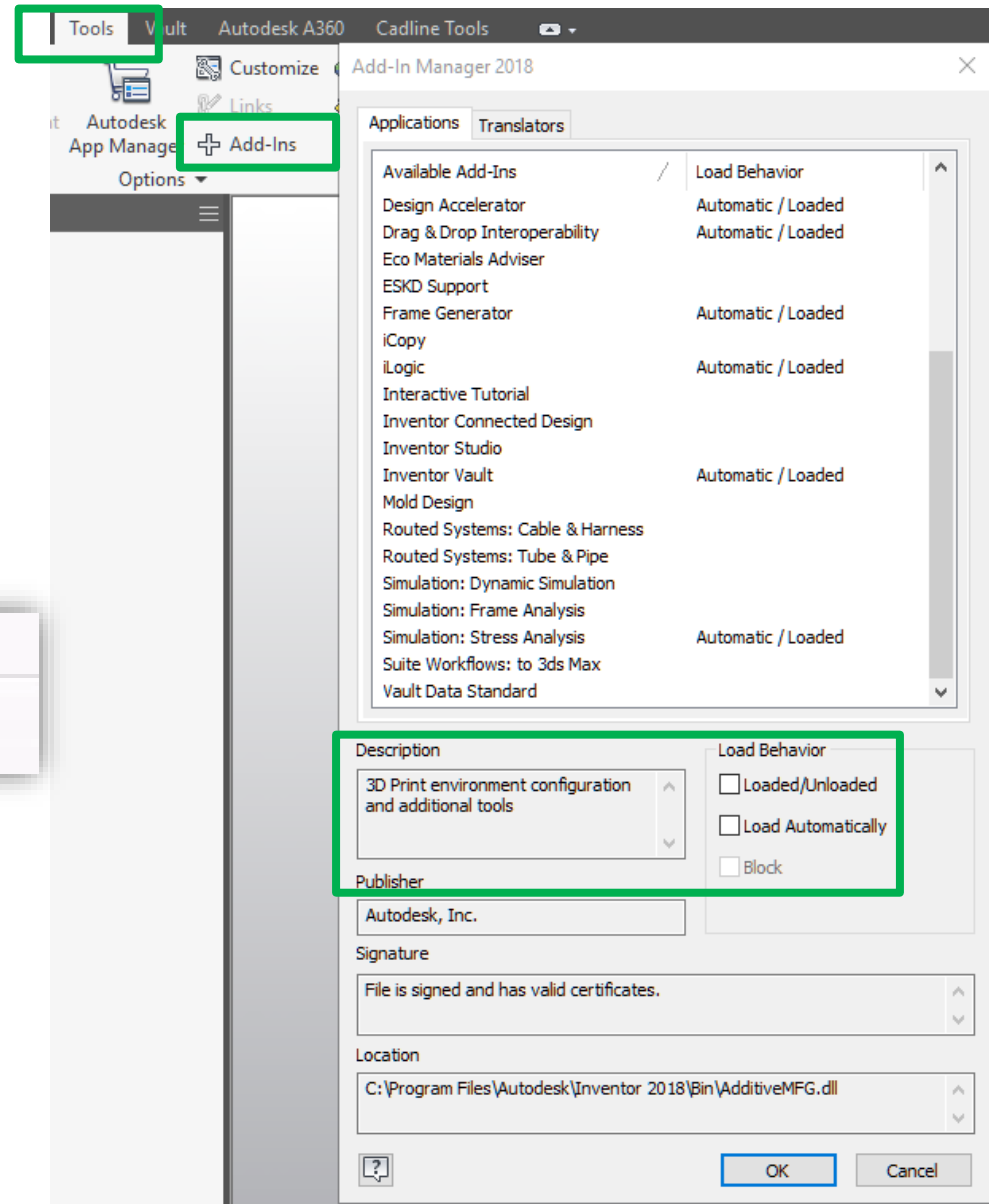
Modern versions of Inventor no longer load all of the add-ins at start-up.

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

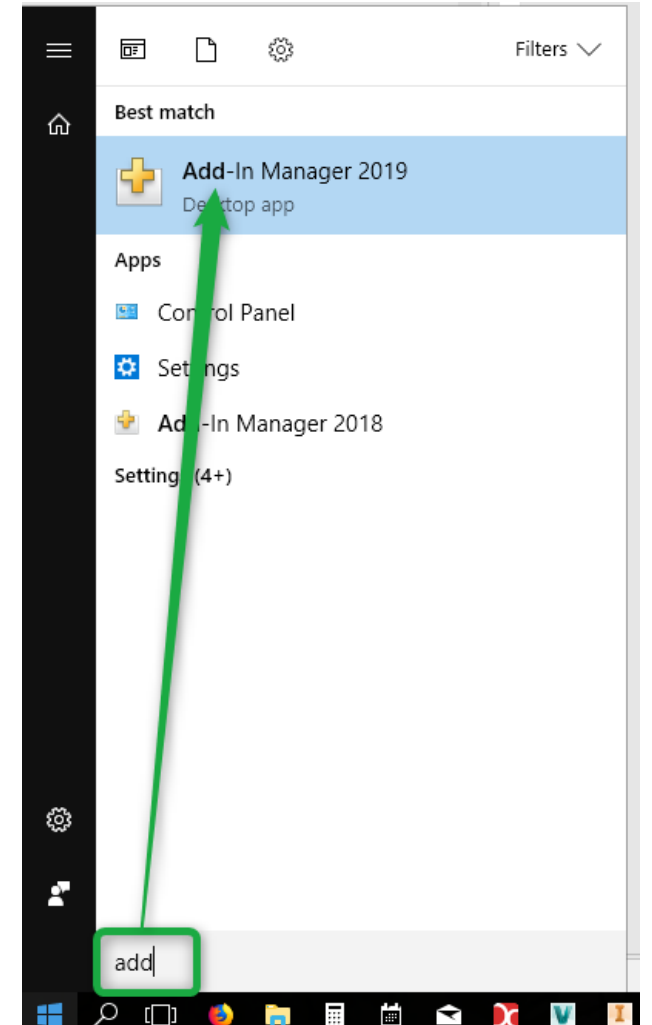
Loading FeatureCatalog ...



# Speed: Manage Add-ins

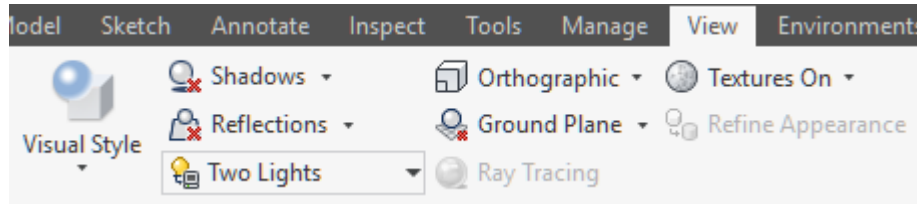
The Add-in Manager can also be loaded as a “Standalone” application.

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



# Speed: View – Shadows Reflections & Visual Style

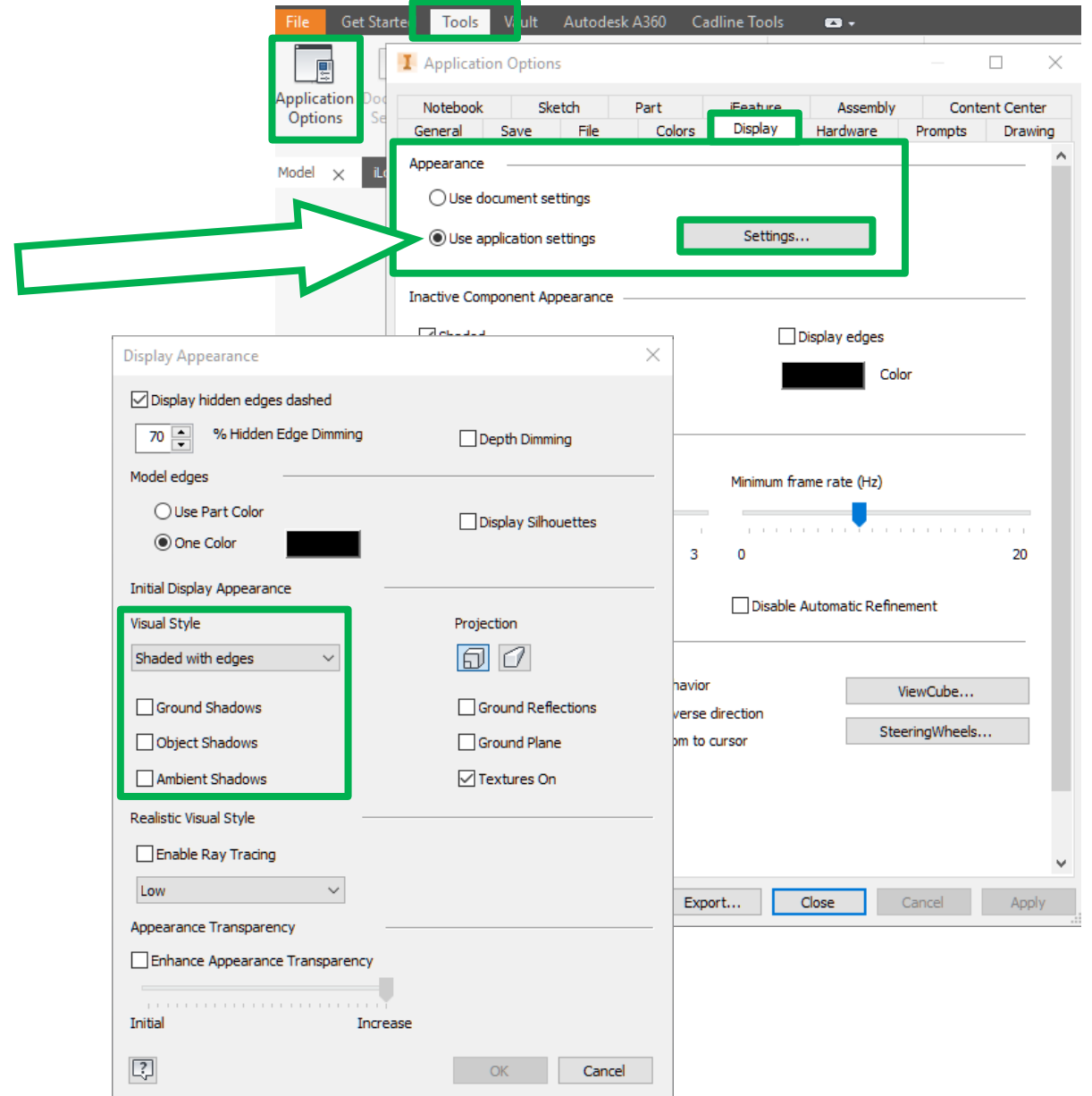
By default visual styles are set in the Inventor File



You can override this in your application options.

Benefits are that models appear as you prefer, and speed

Shadows and reflections slow you down



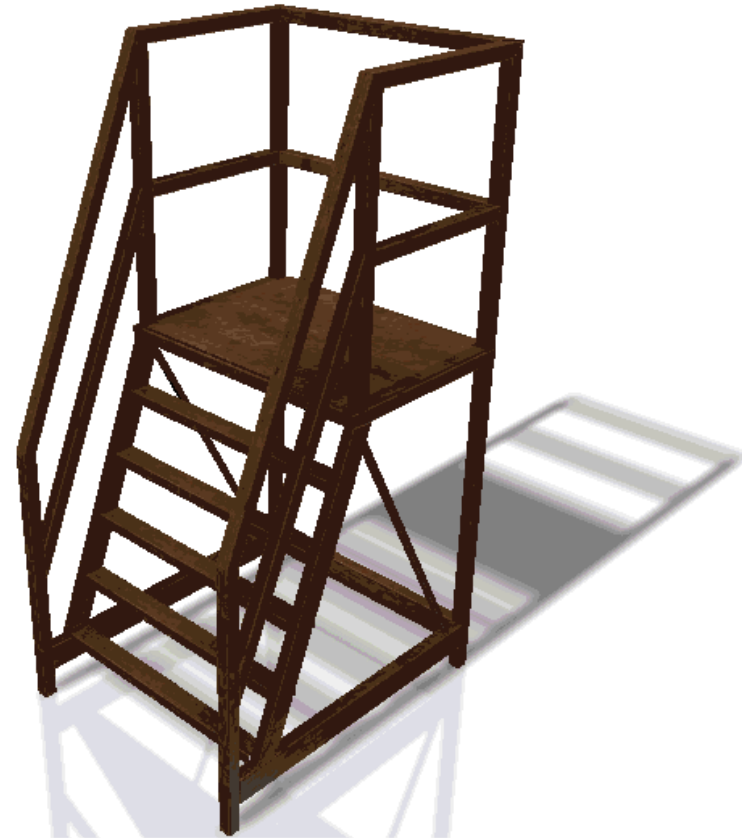
# Speed: F-Key Shortcuts

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).





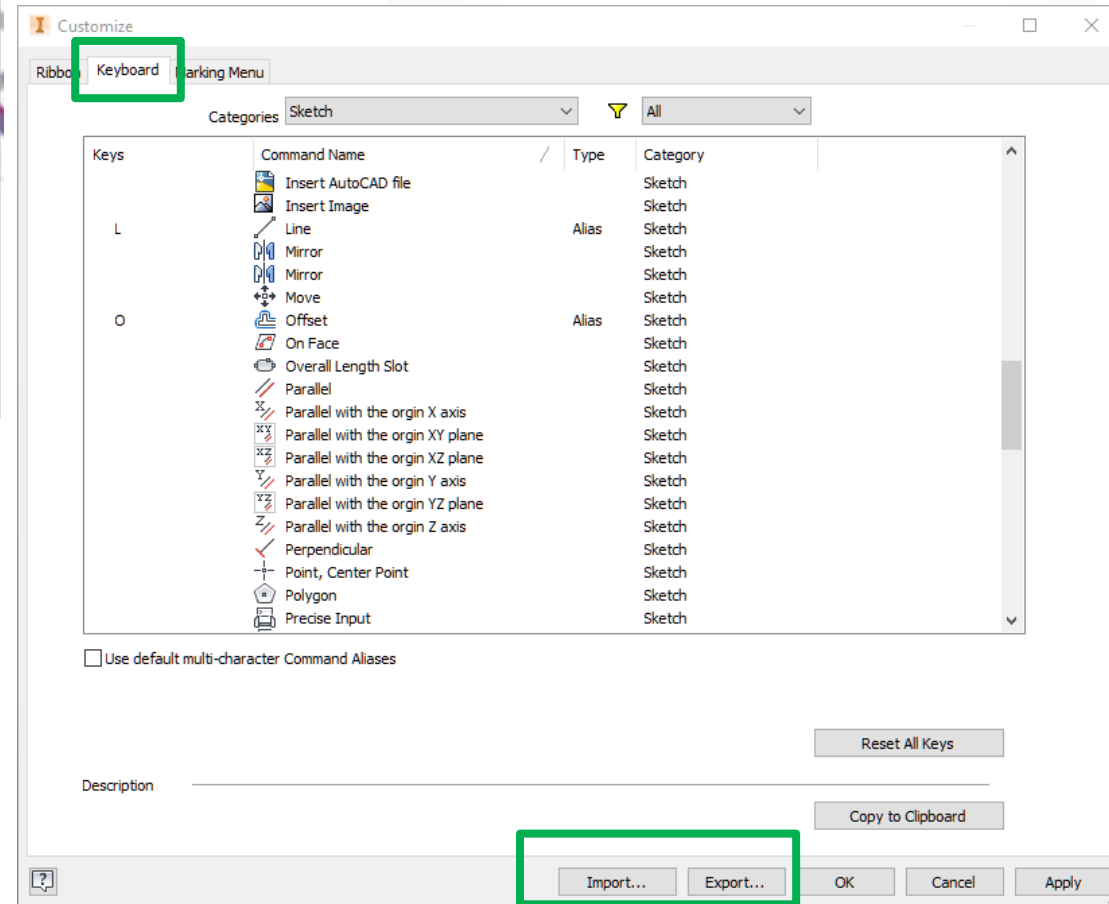
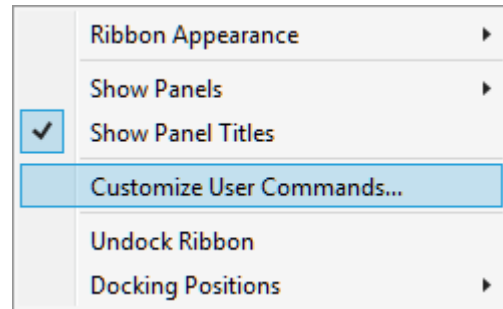
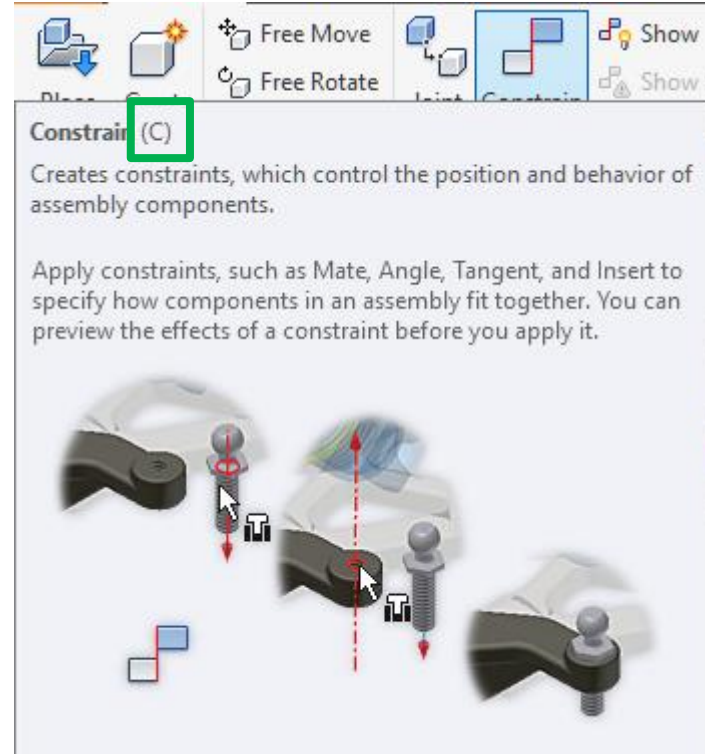
# Speed: Keyboard Shortcuts

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.

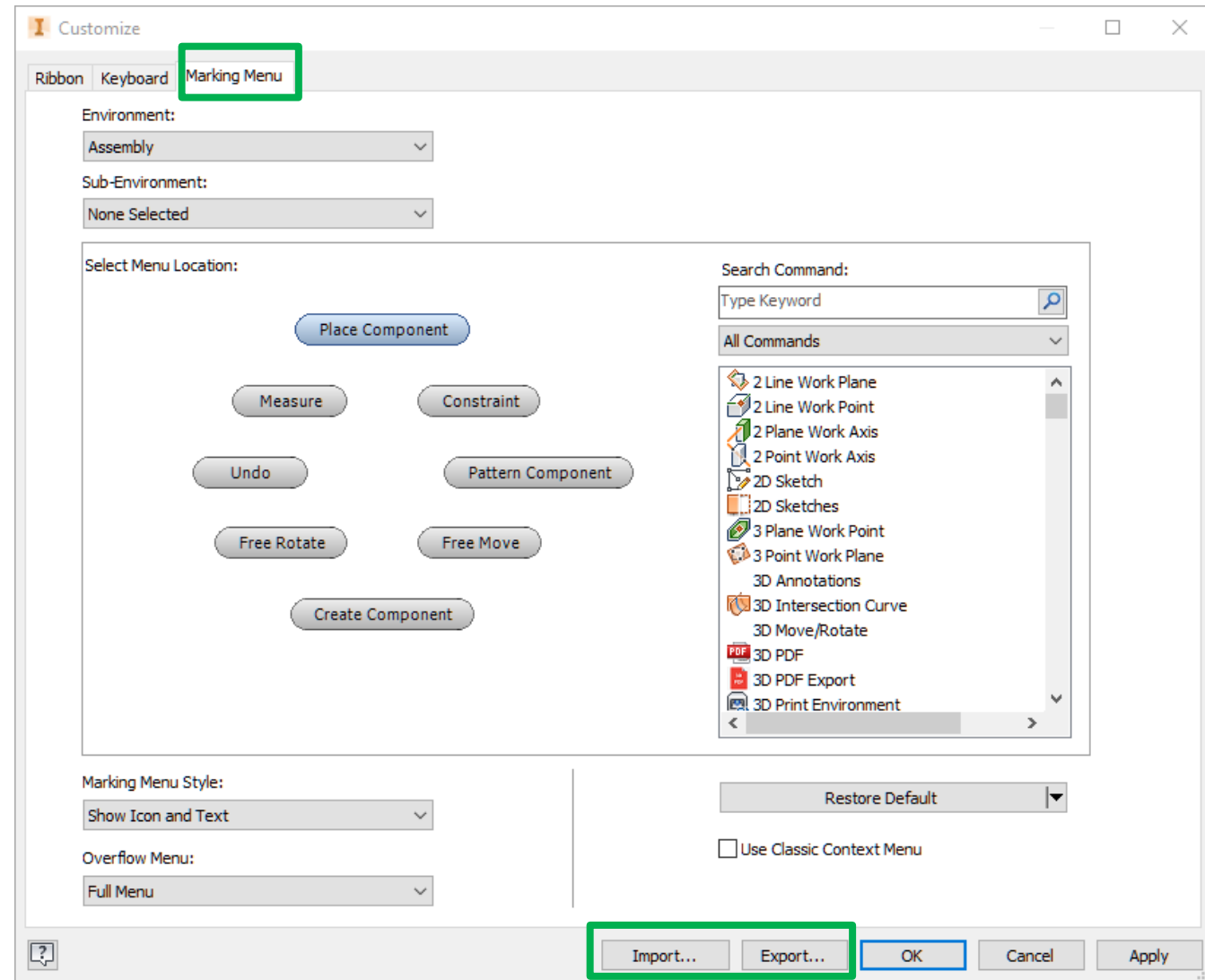
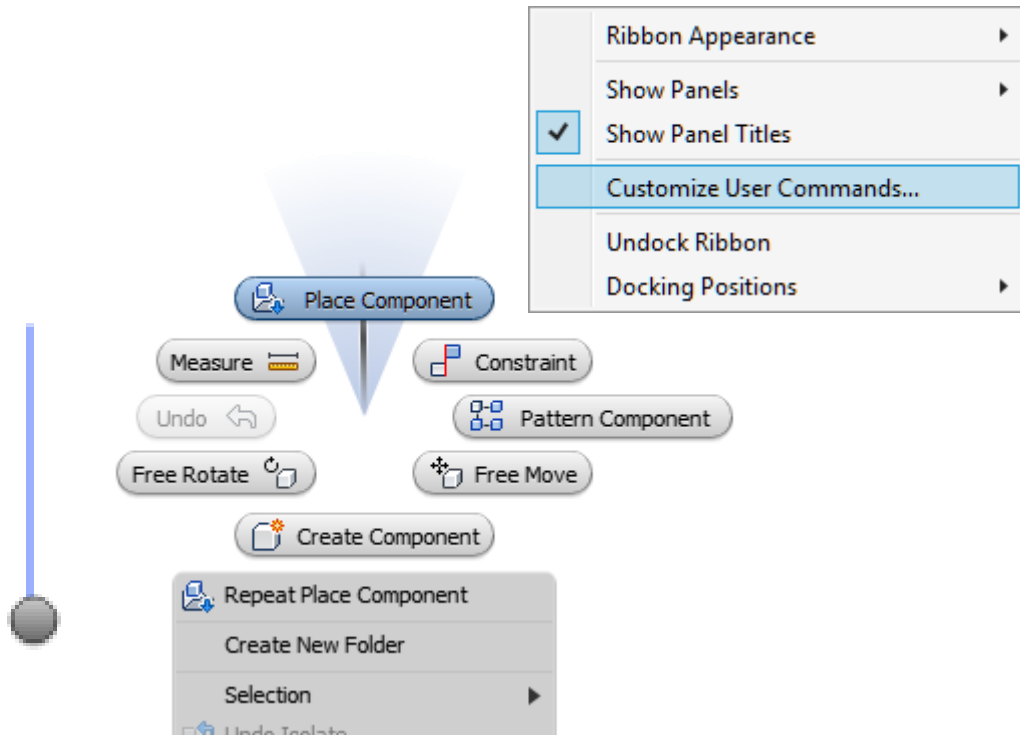
Share your settings using Export an import options



# 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.

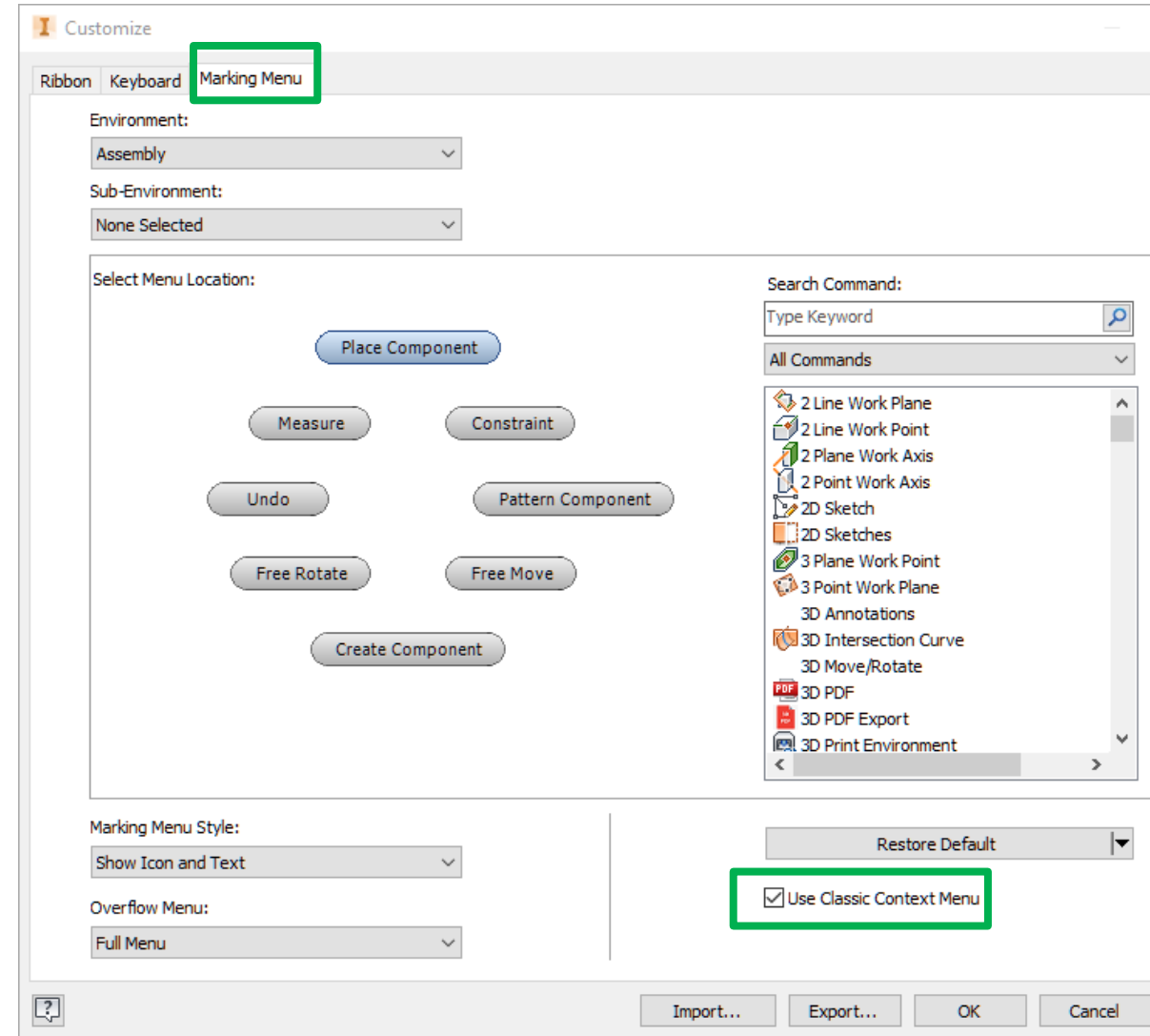
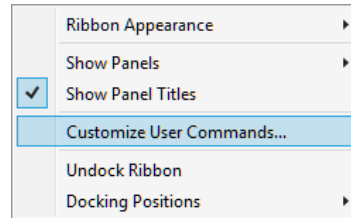
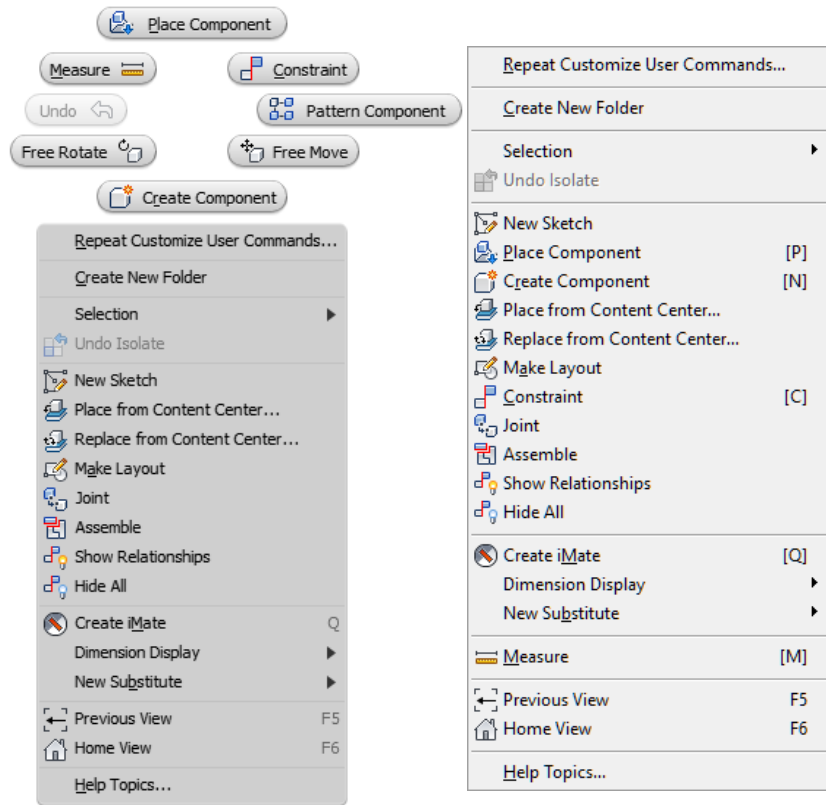
Share your settings using Import and Export options



# 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

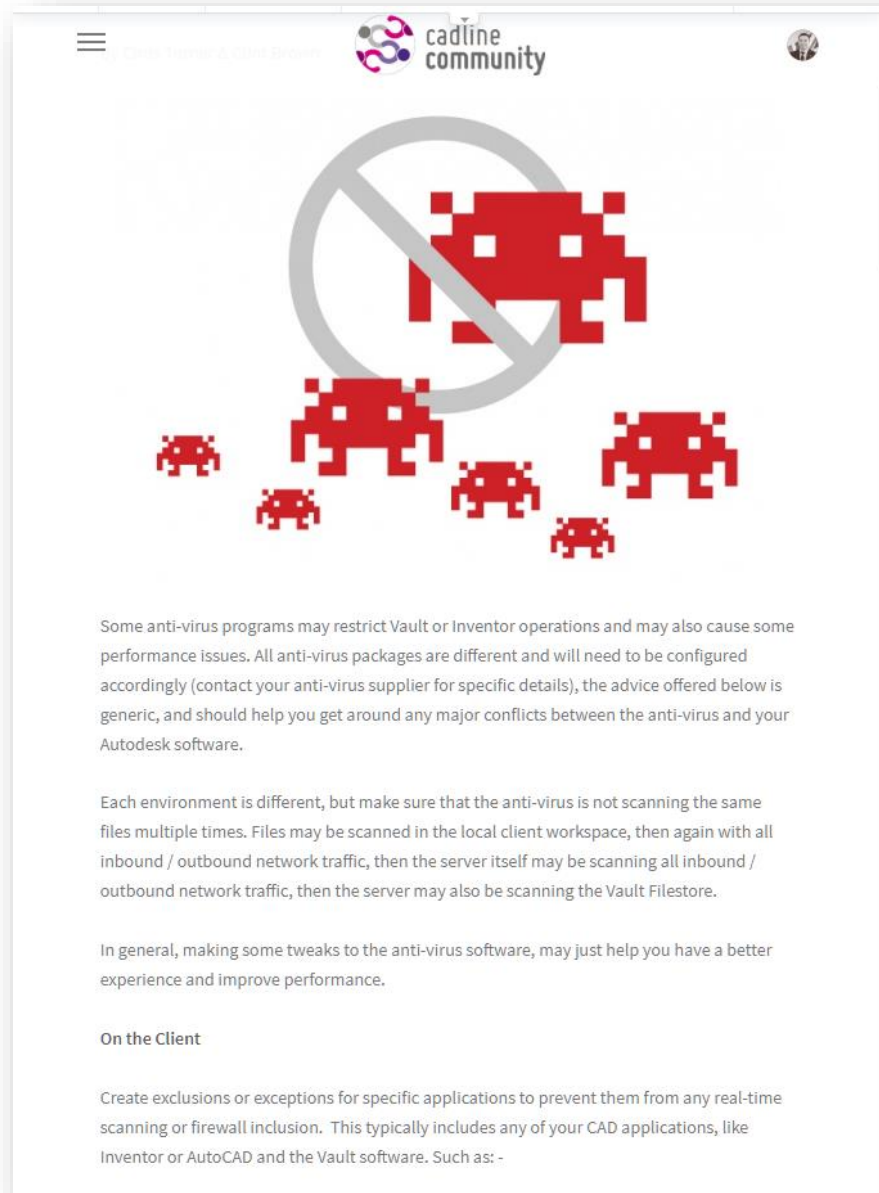
Your Company Anti-virus software is probably slowing you down.

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



The screenshot shows a web browser window displaying a community article. At the top, there is a navigation bar with a hamburger menu, the text 'Cadline Twitter & LinkedIn', the 'cadline community' logo, and a user profile picture. The main content area features a large graphic of a red alien spaceship with a grey prohibition sign over it, surrounded by several smaller red alien ships. Below the graphic, the article text reads: '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.' It continues with '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.' and 'In general, making some tweaks to the anti-virus software, may just help you have a better experience and improve performance.' The article is categorized under 'On the Client' and provides instructions: '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: -'.

## I INVENTOR

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.

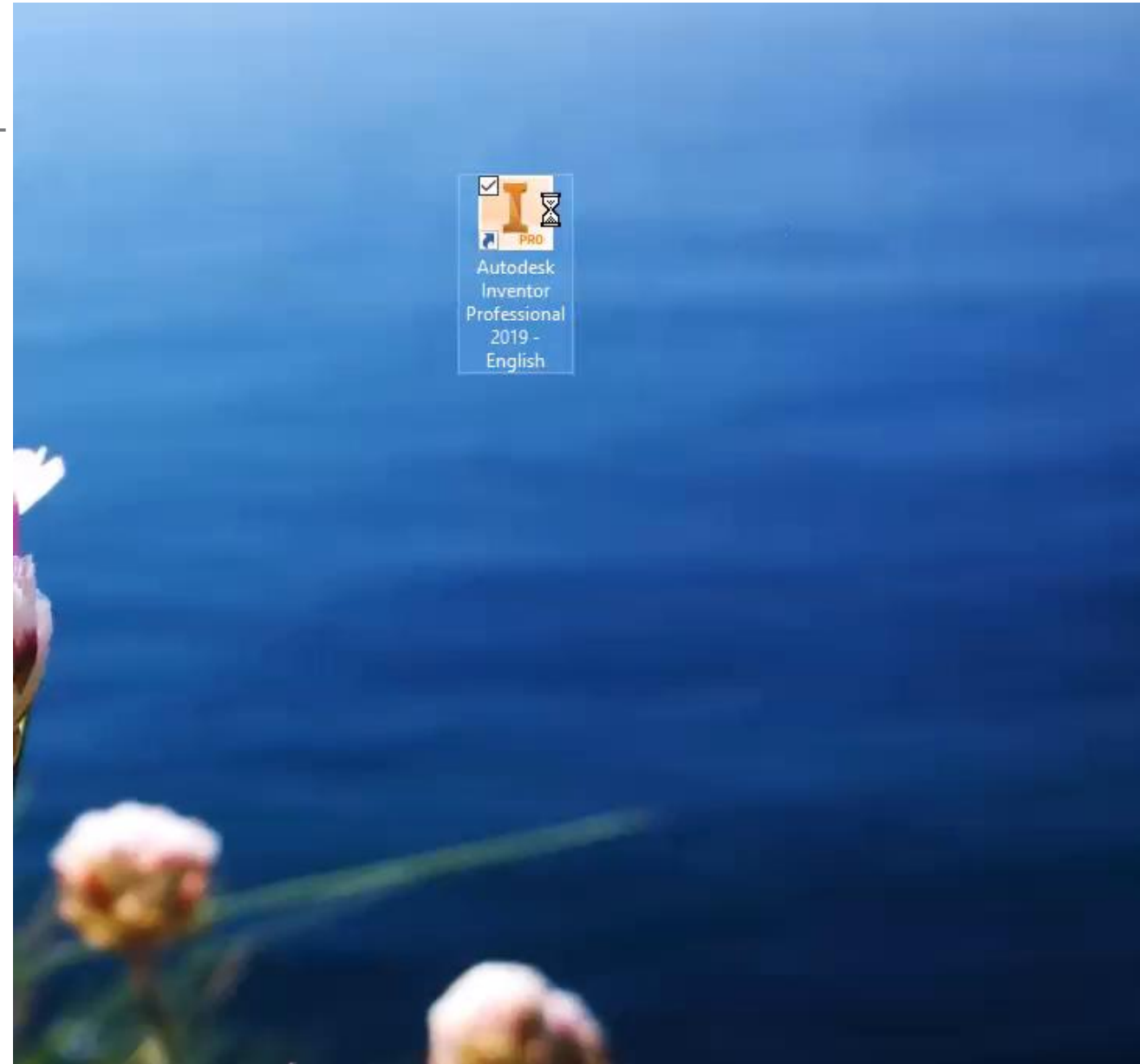
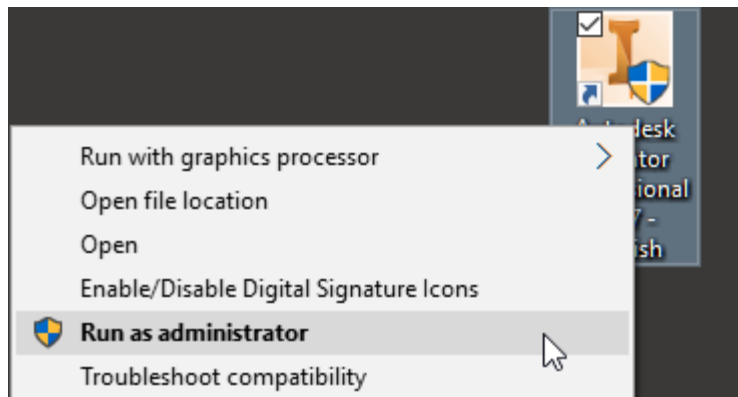
Additional files which could be excluded from virus scanning are .CHM file types (in

# 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  
 AUTODESK KNOWLEDGE NETWORK  
 Search Vault Products  
 Clint Brown English

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	⬇
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✓	✓	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 M5000M	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.



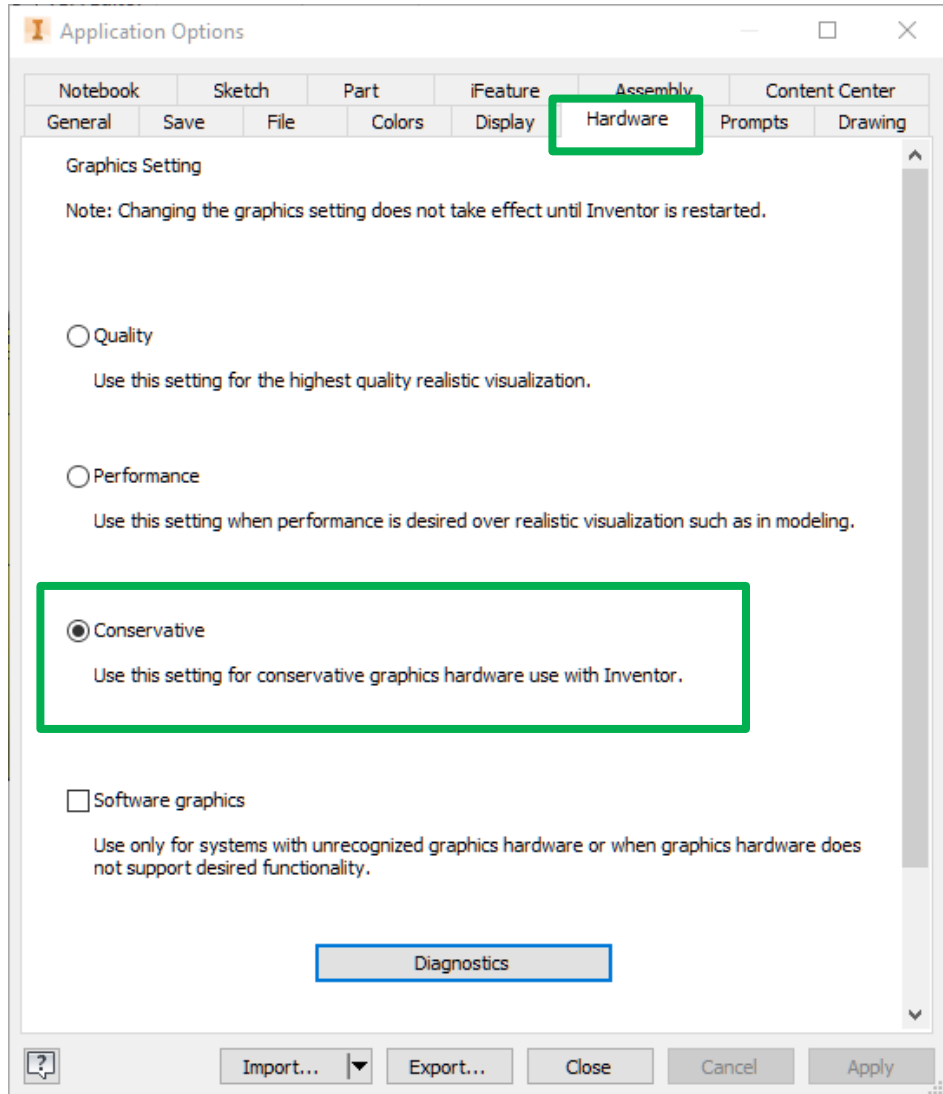
<https://www.3dconnexion.co.uk/index.php?id=76&redirect2=www.3dconnexion.co.uk>

Back Over to you

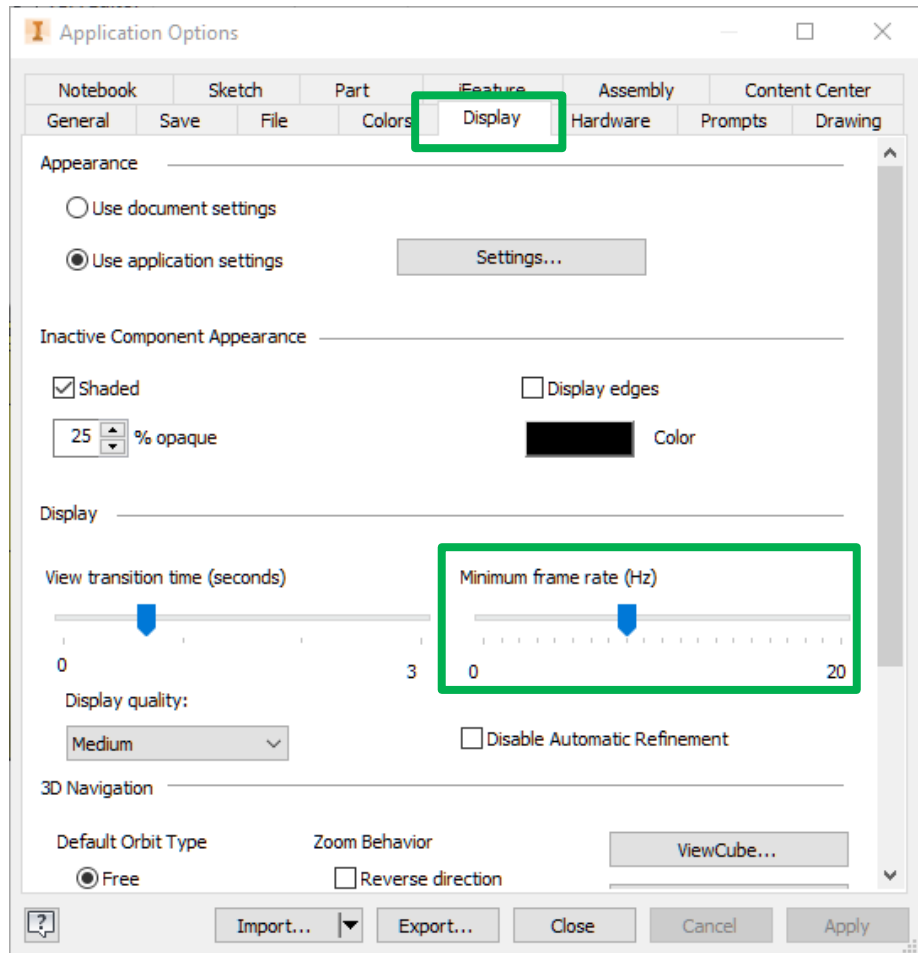


# Speed: Graphics

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

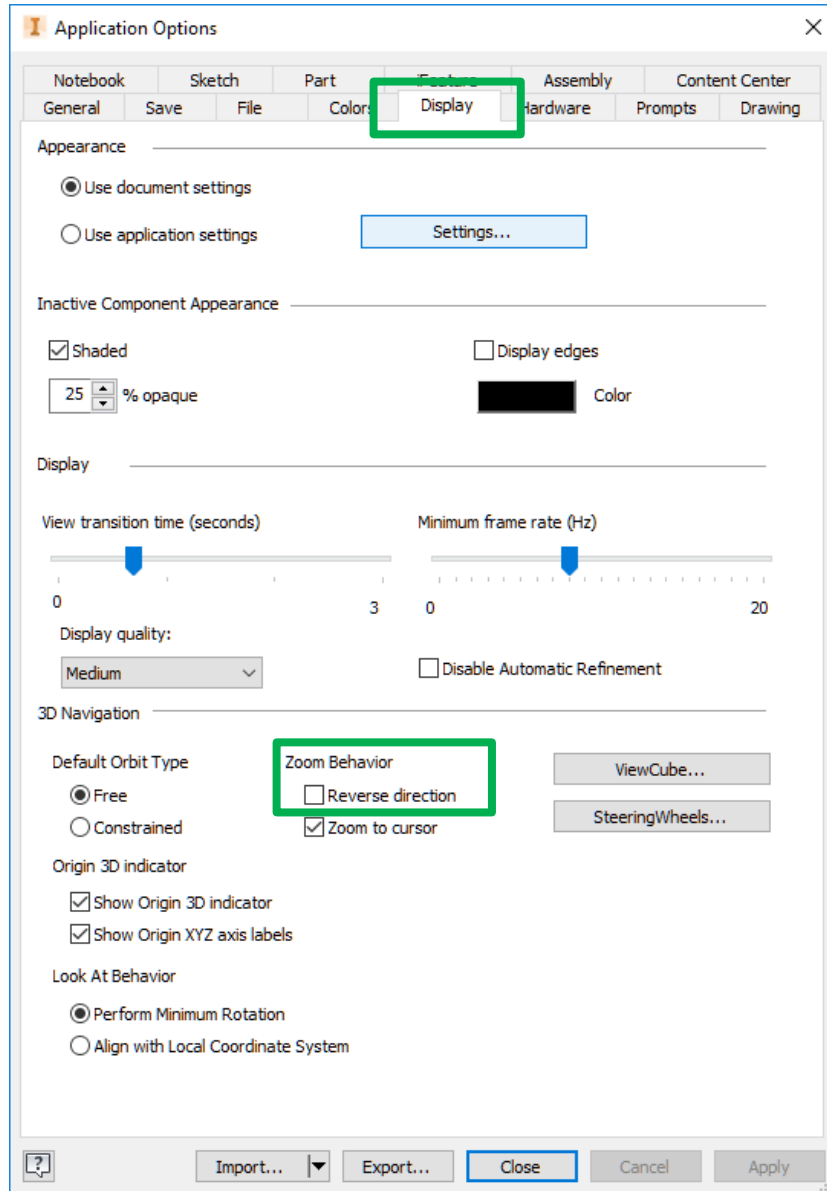


# Speed: Graphics

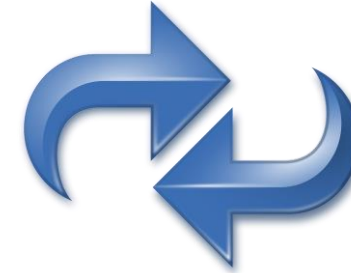


If you are still experiencing issues, try a lower frame rate, it's worth experimenting with a setting that will suit your system (In Application Options)

# Speed: Zoom Direction



Tools > Application Options > Display > Zoom Behavior

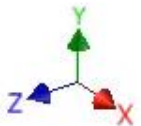
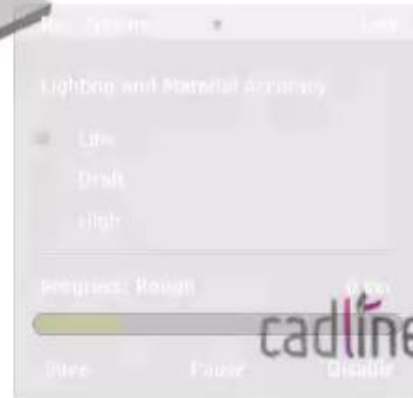


# Rendering – Ray Trace still images



Save images of your ray trace while it's rendering

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

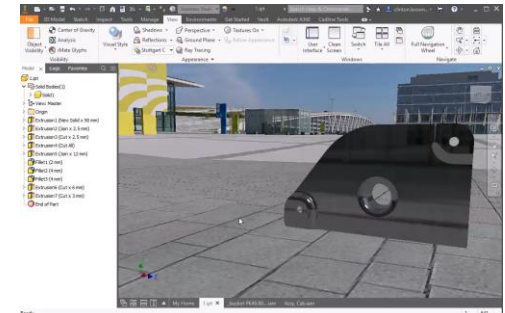




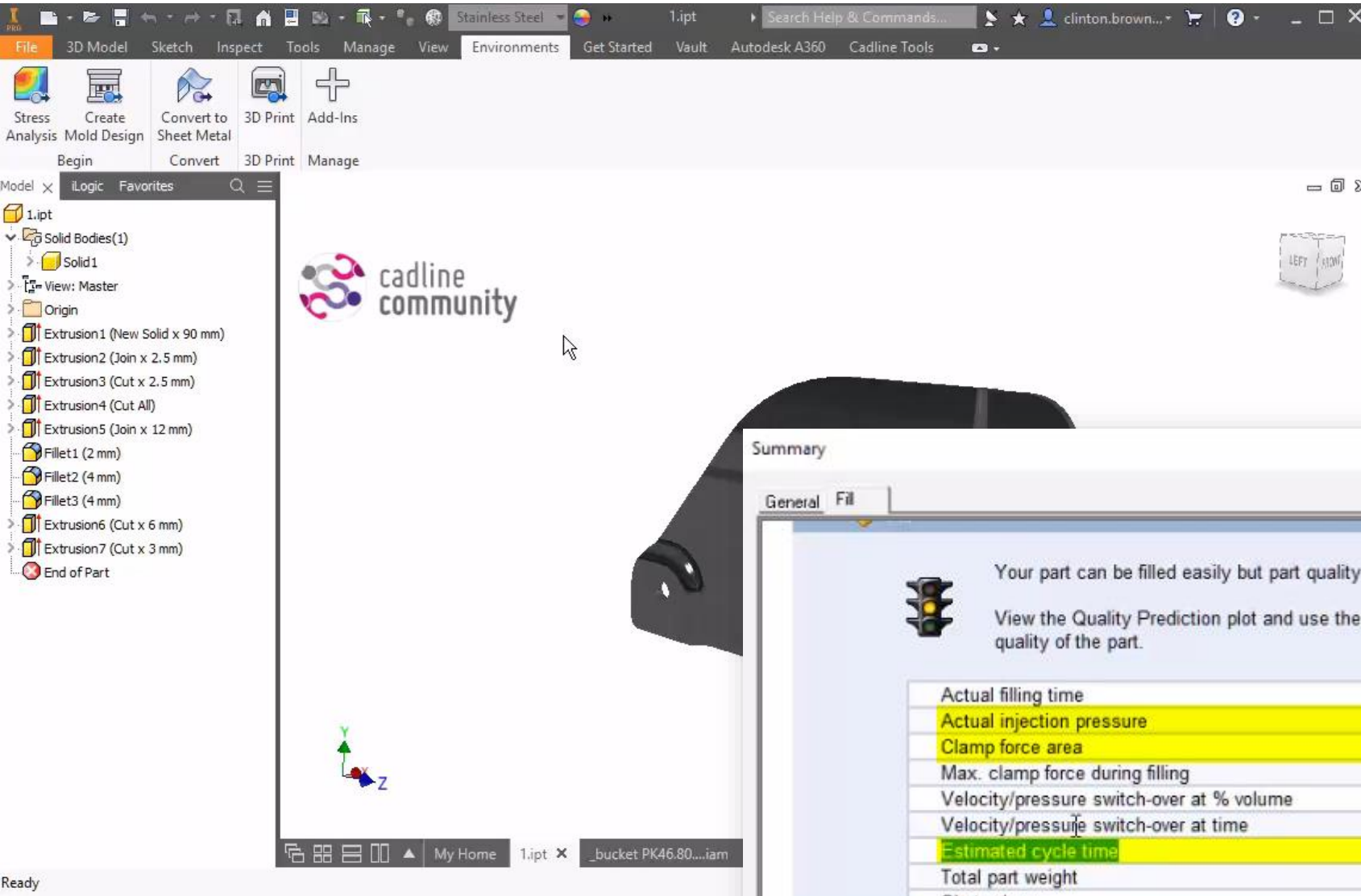
# Environments



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

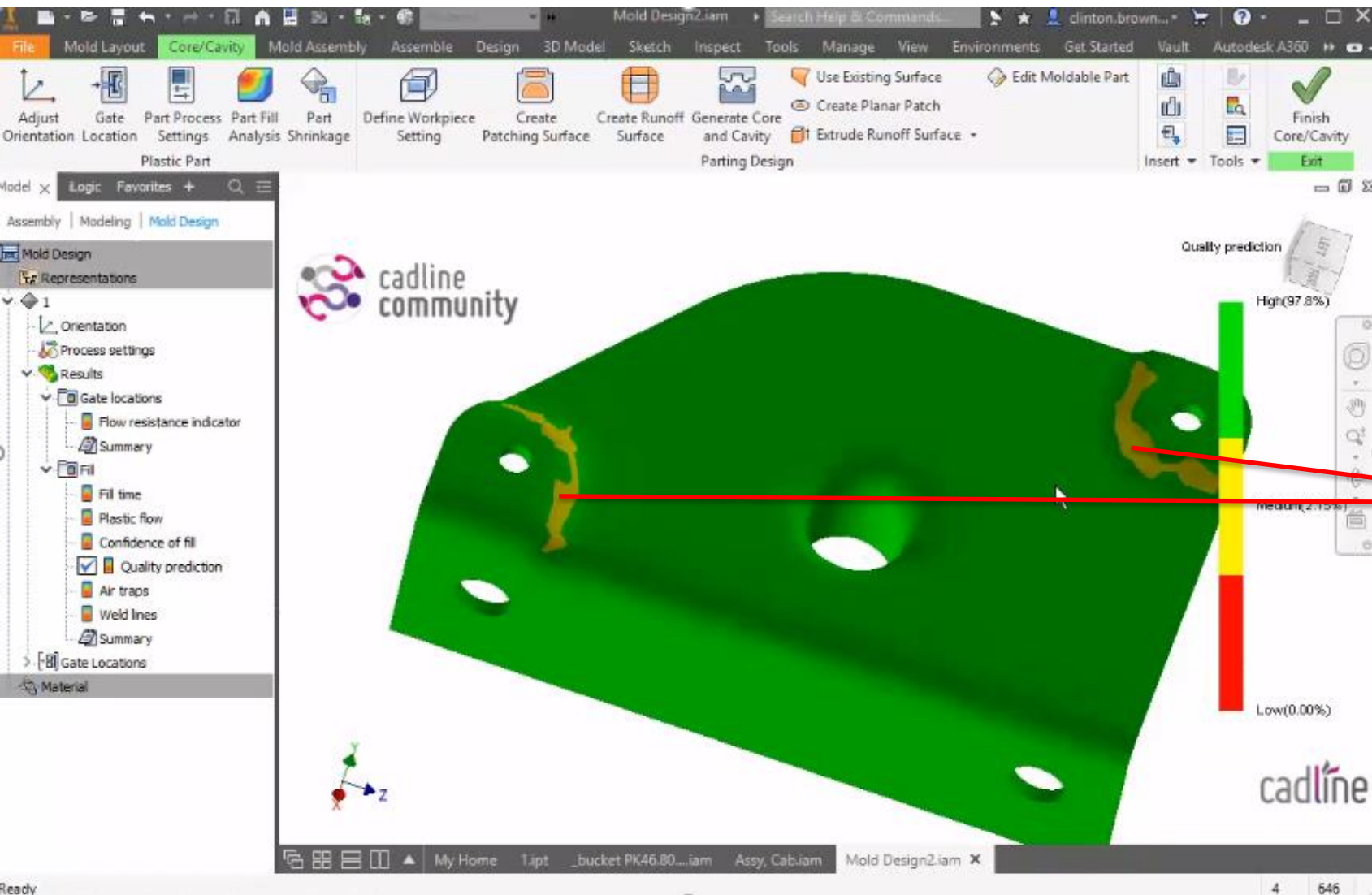
Setting material

Selecting Default processing parameters

Estimated Gate location

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

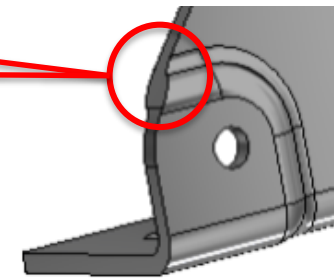
# Do you design Plastic parts? Use the Tooling environment



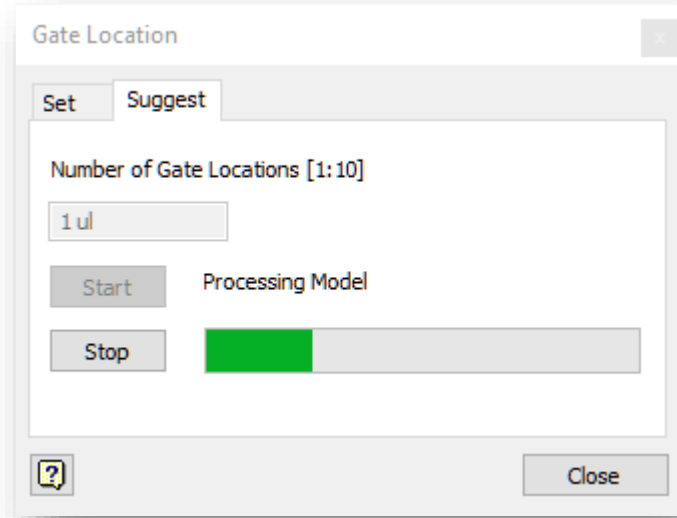
Use Part Fill analysis to look for potential moulding issues like weld lines

Quality Prediction is useful for sink mark areas

Confidence of fill is also very useful

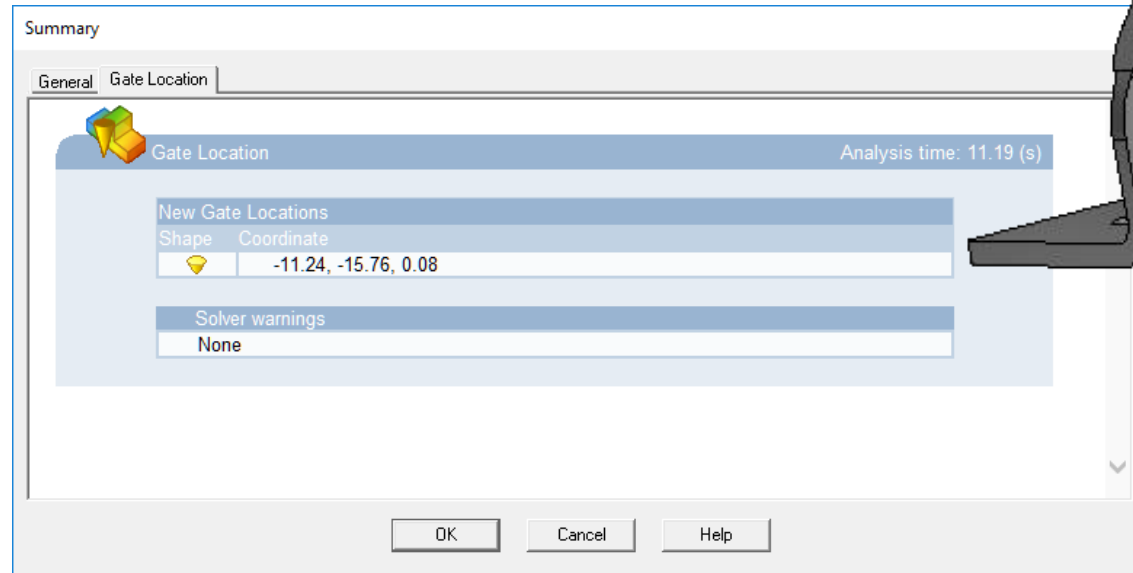
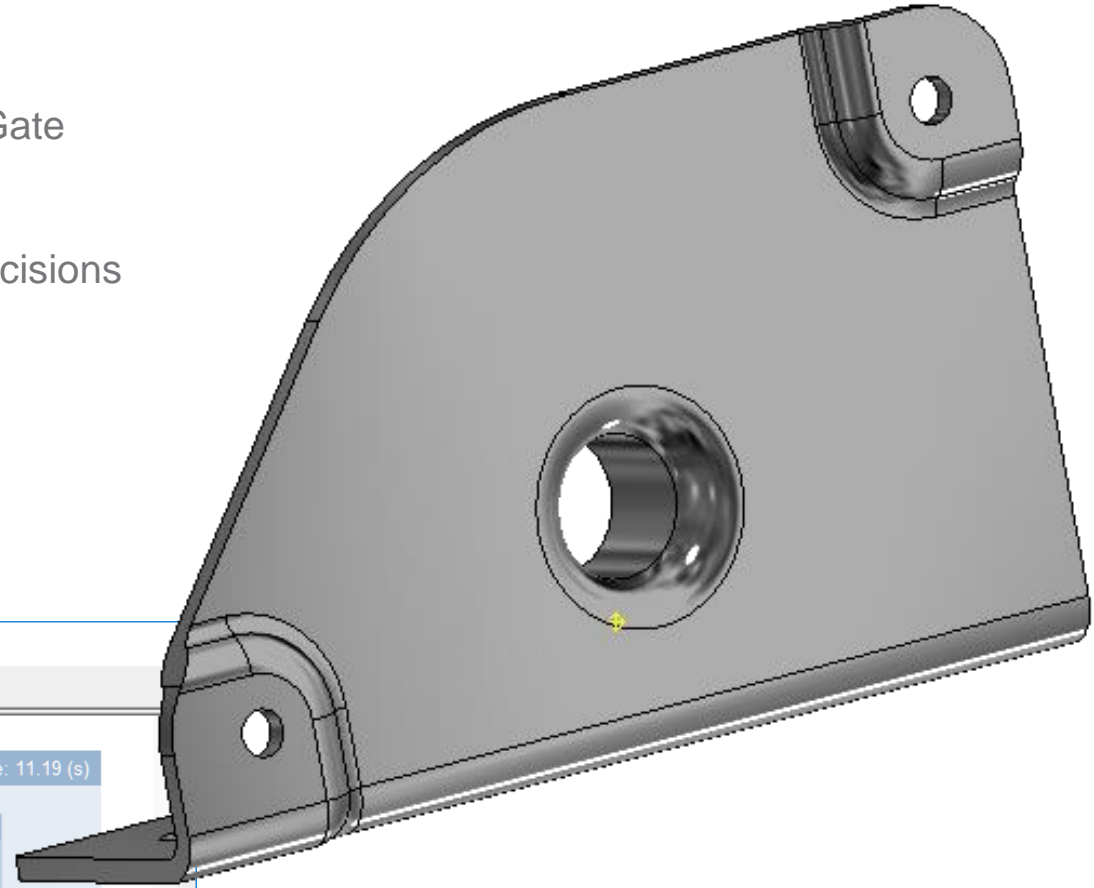


# Do you design Plastic parts? Use the Tooling environment



Inventor can suggest the best Gate location for the moulding

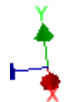
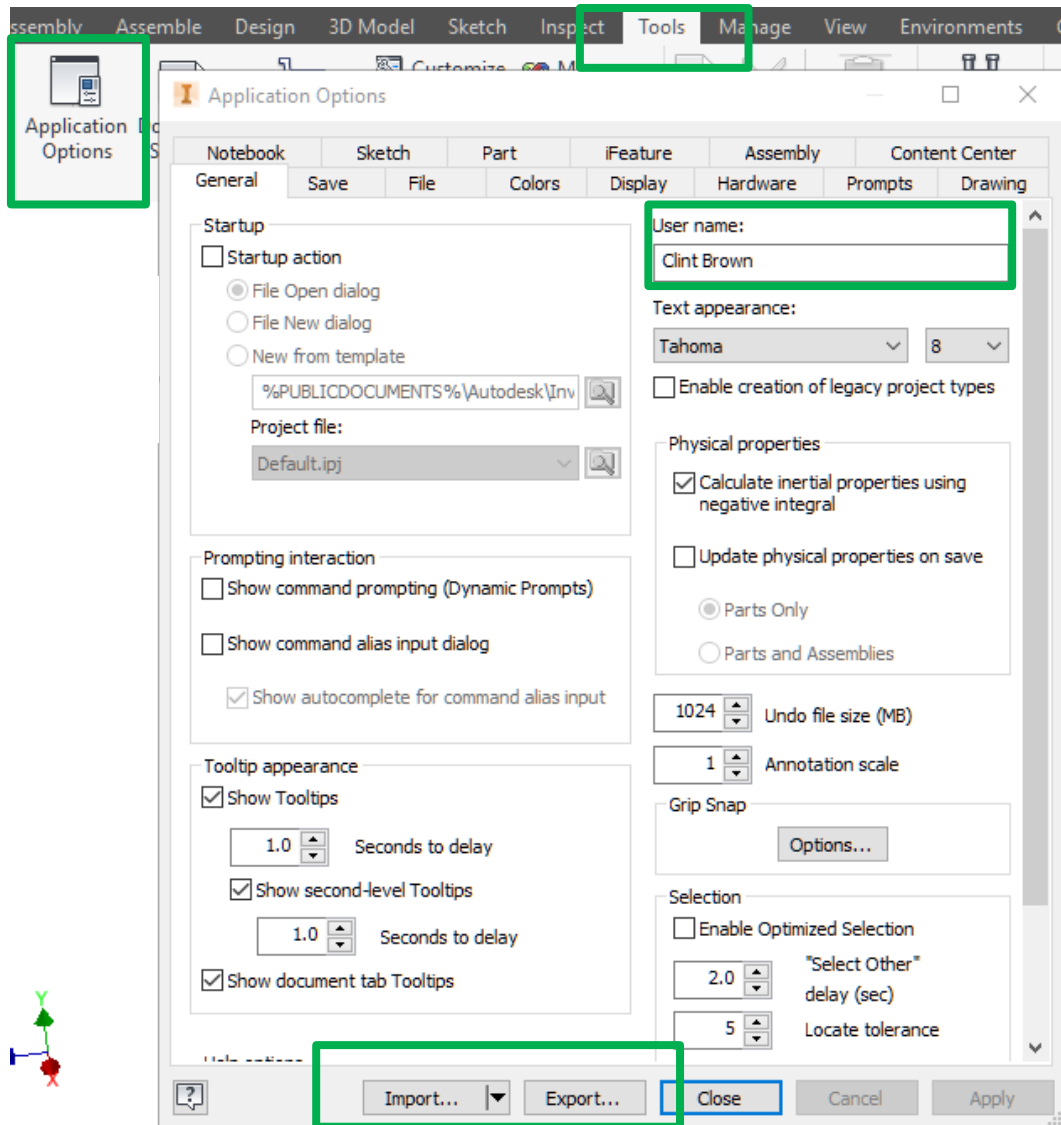
This may help inform design decisions





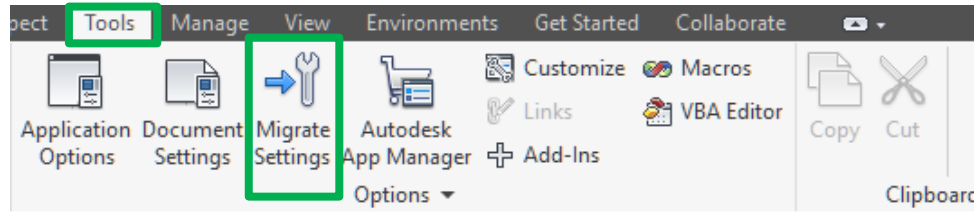
# Application Options

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

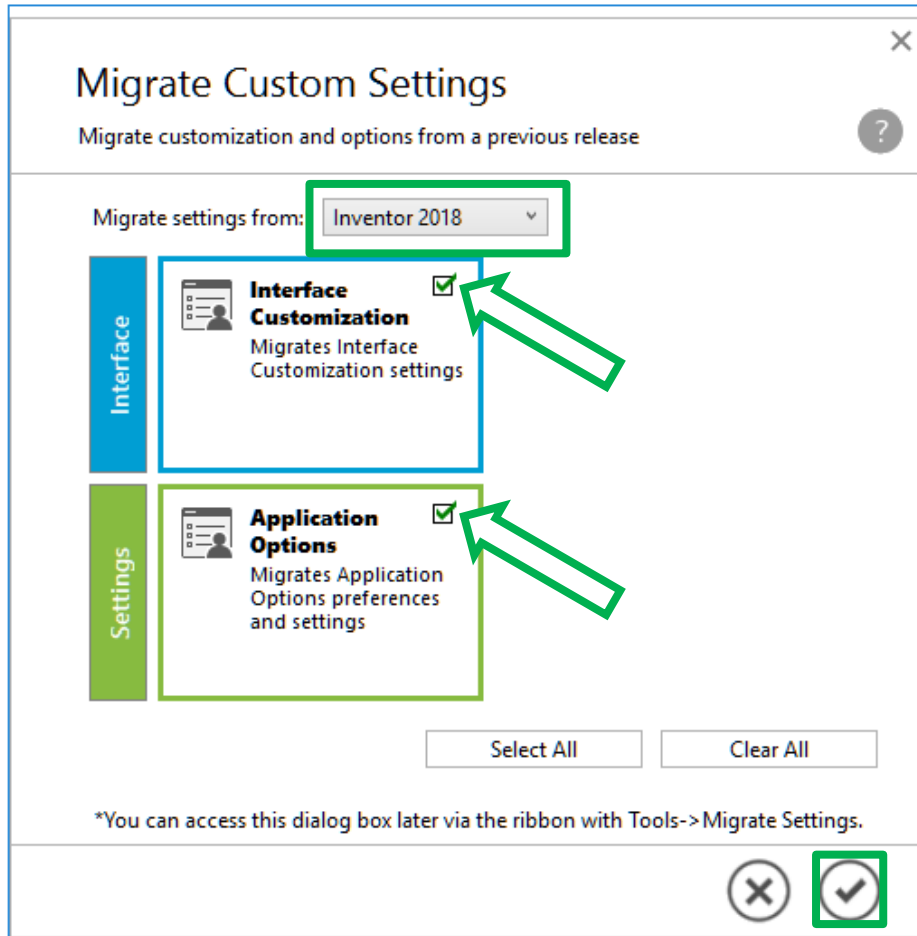




# Migrate Application Options + UI (2019)



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



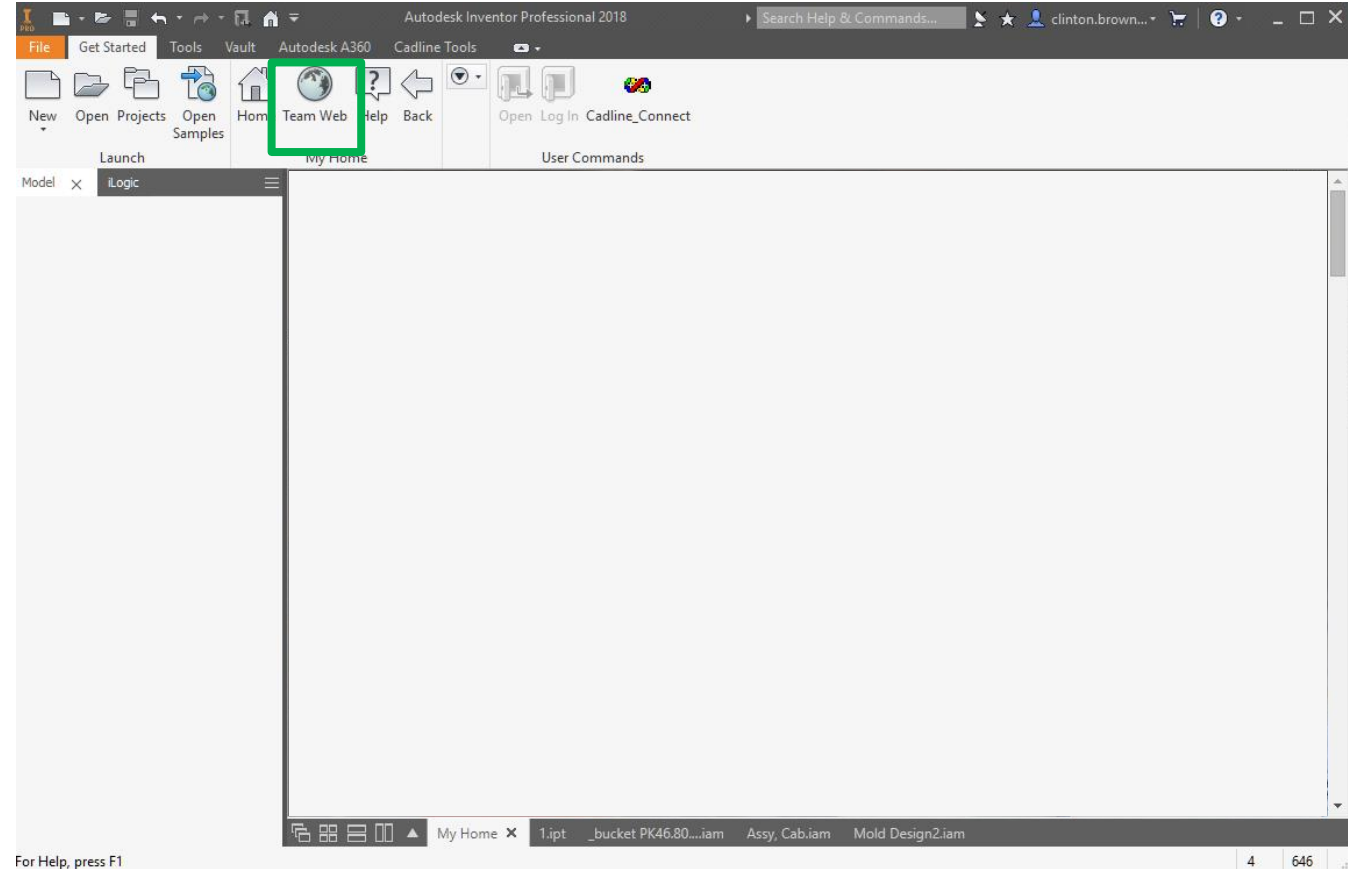
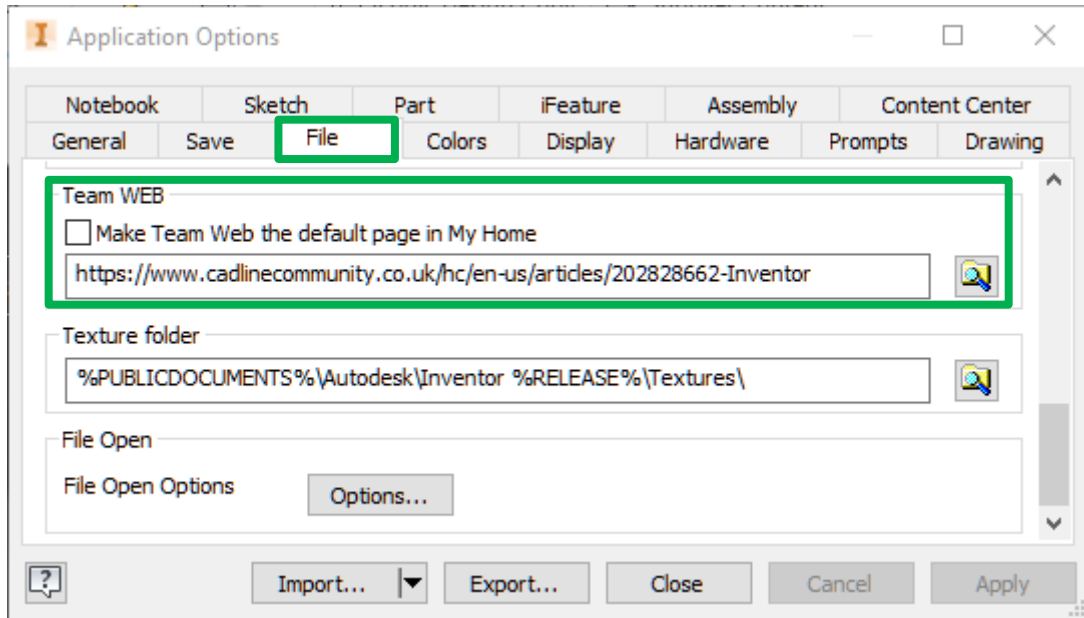
# Team Web

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`



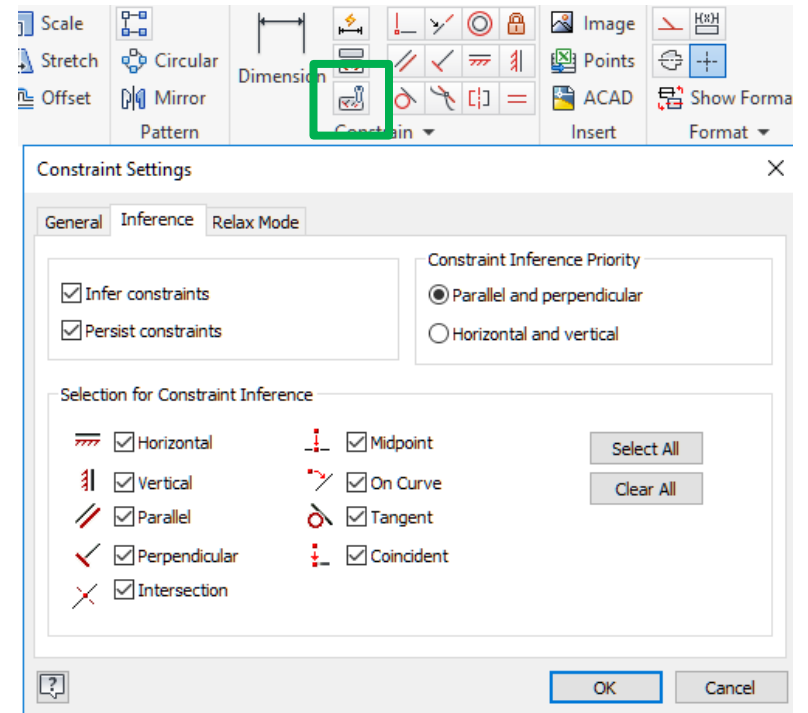
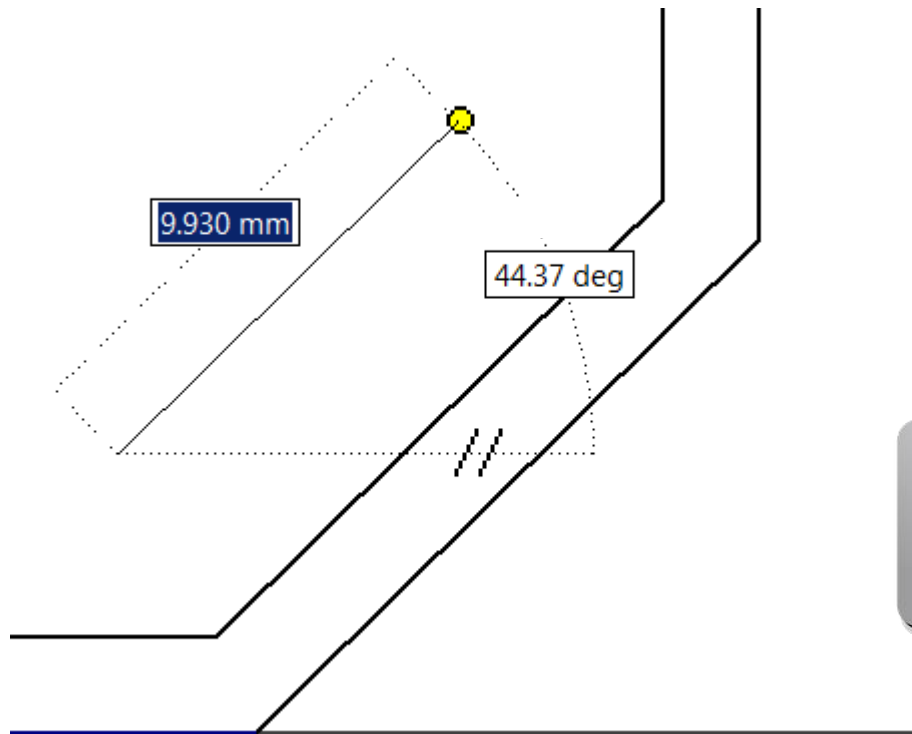
# Sketches: Arcs from the Line Command

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.

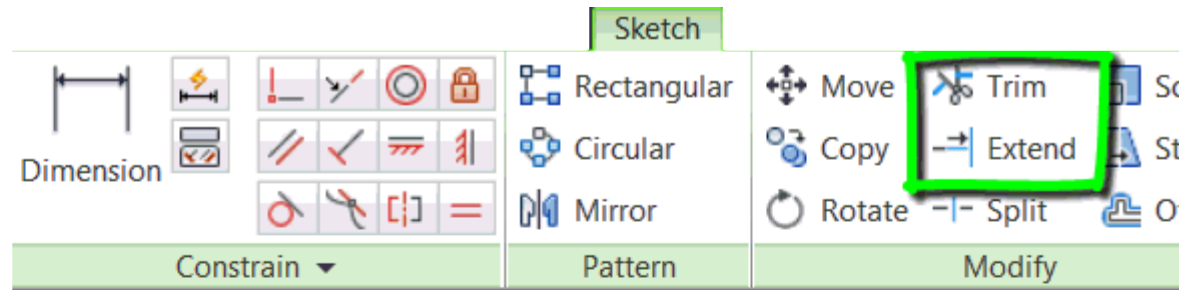


# Sketches: Inferred Constraints

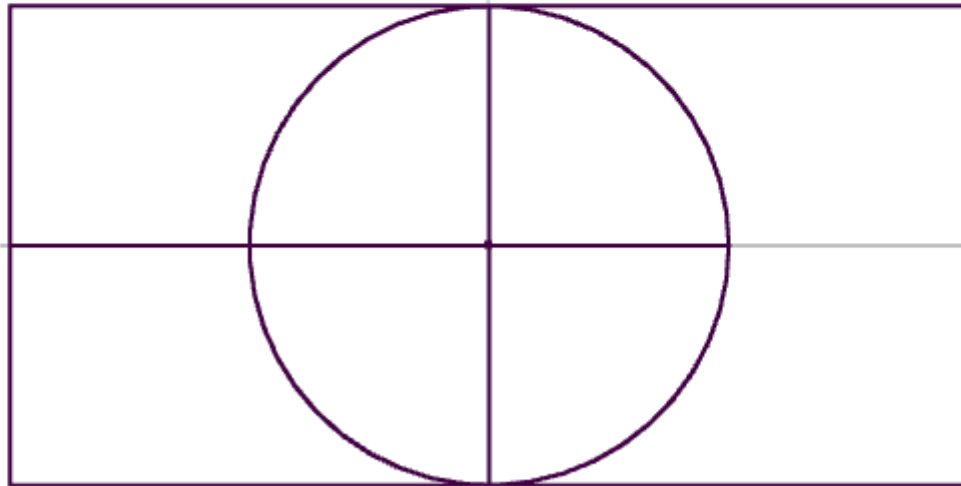
Hold the **CTRL** key when drawing in a model sketch to prevent Inventor creating inferred constraints. This ignores your constraint settings



# Sketches: Trim and Extend

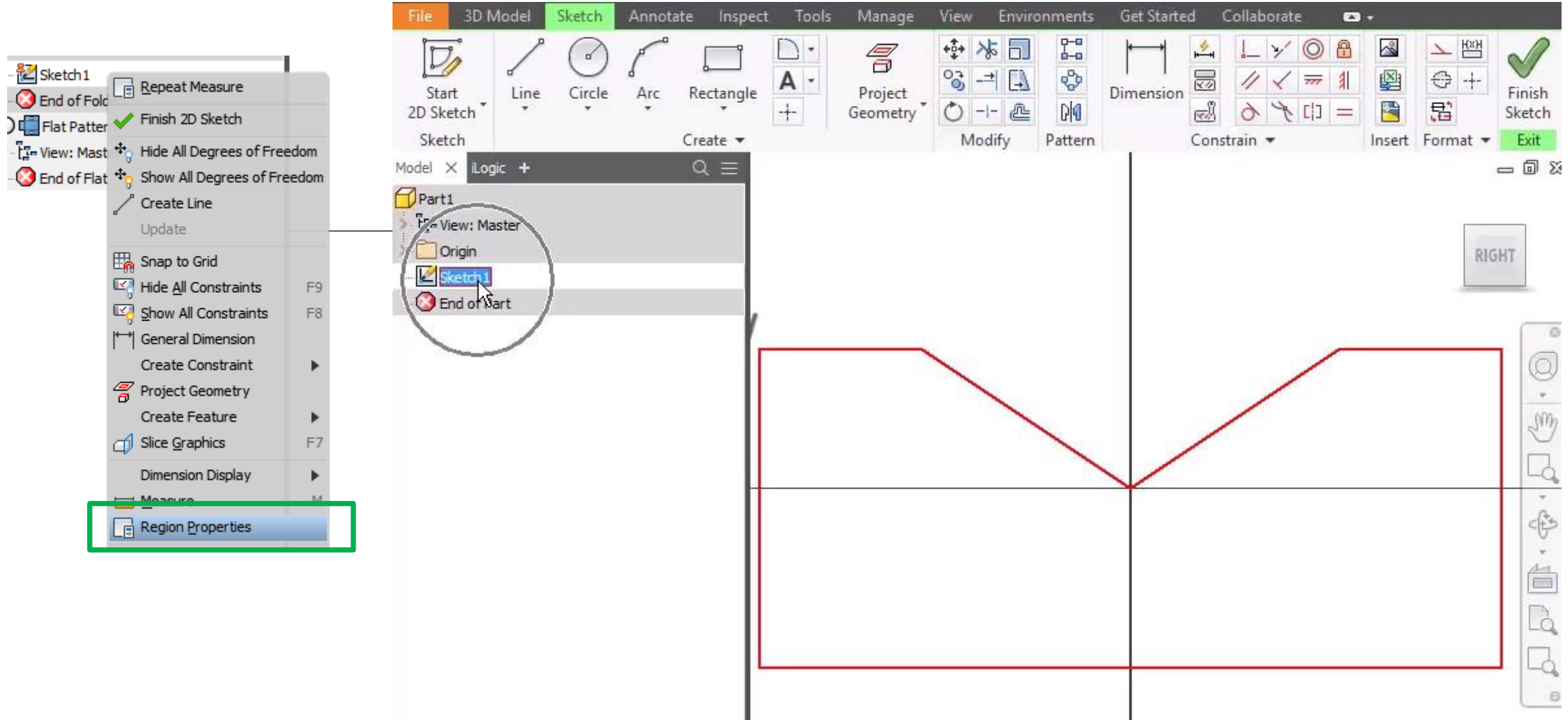


When in Sketch mode, you can switch between the trim and extend command, by simply holding down the **Shift** key on your keyboard.



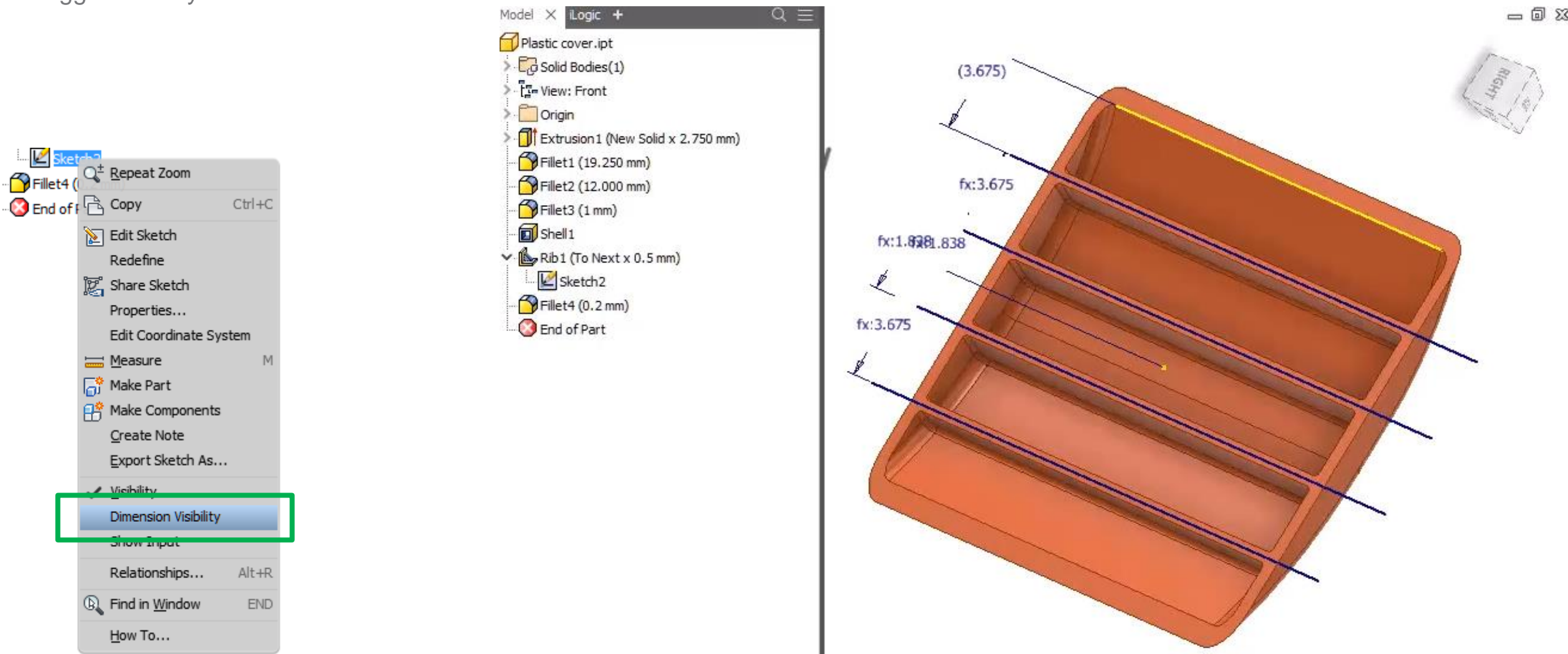


# Sketches: Region Properties



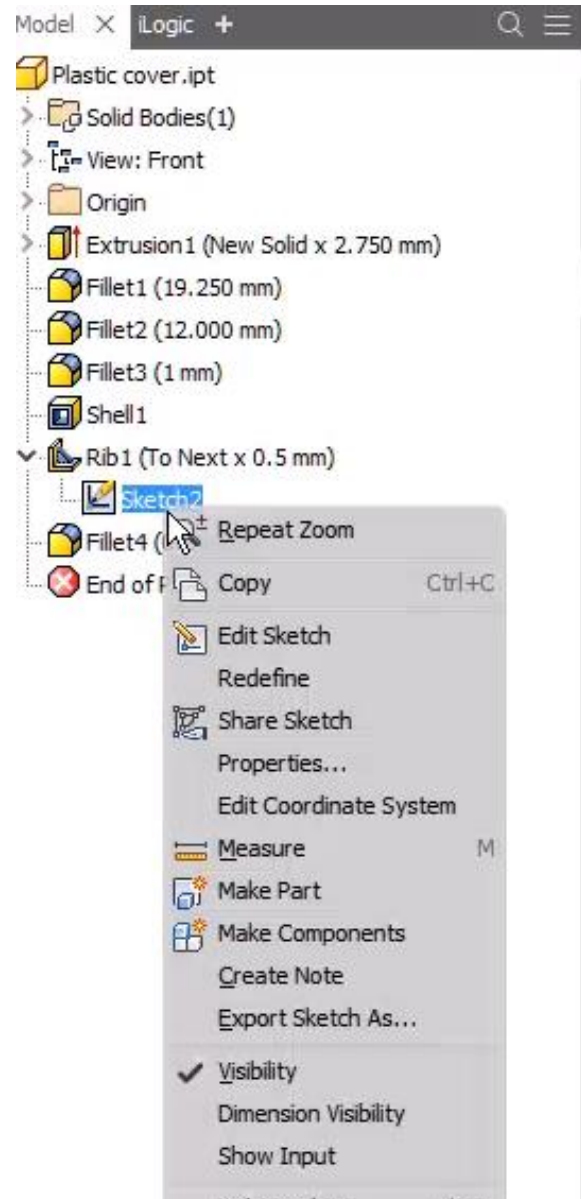
# Sketches: Dimension Visibility

Toggle visibility of dimensions on a sketch



# Sketches: Export Sketch to DXF

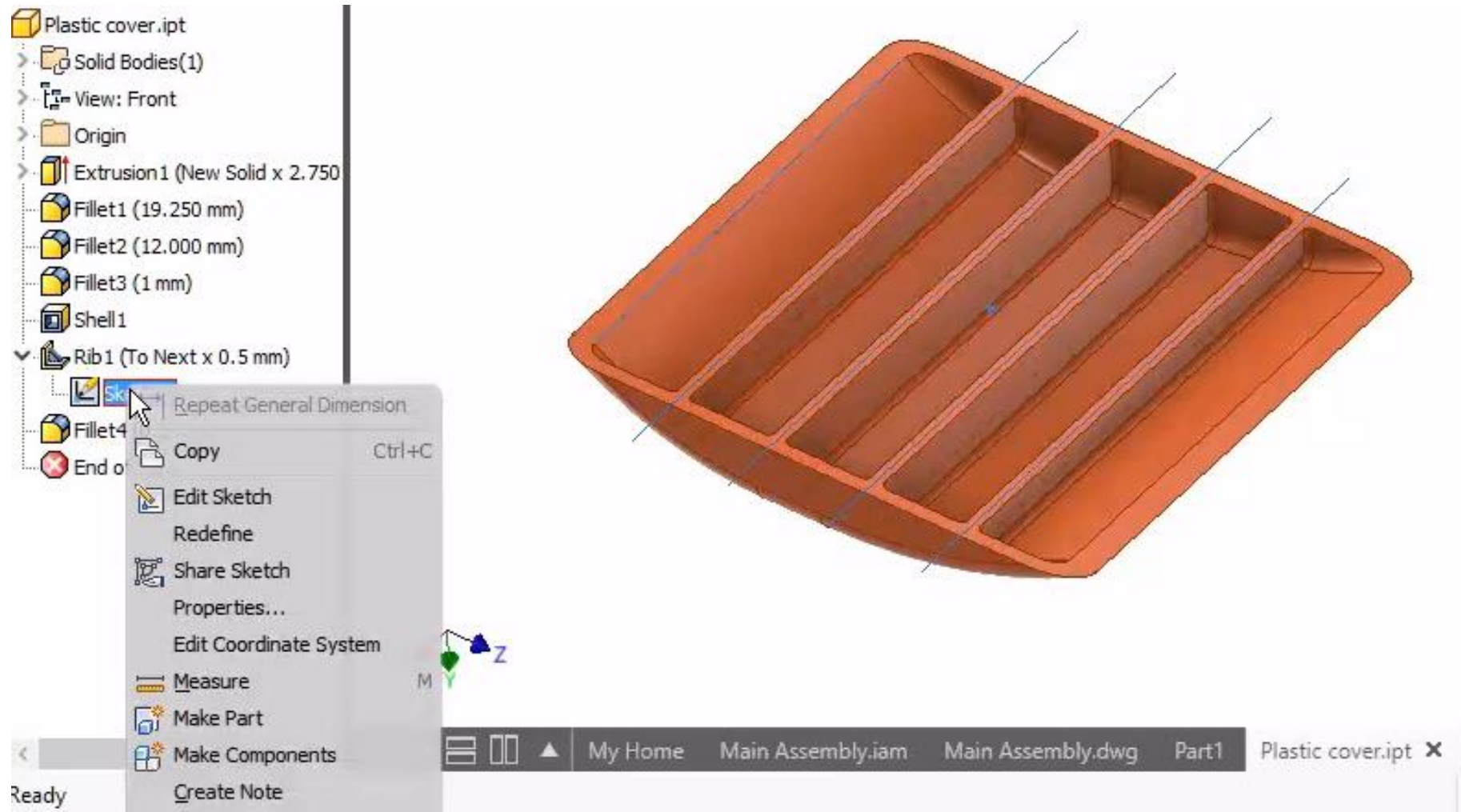
Any Sketch in Inventor can be exported out to DXF





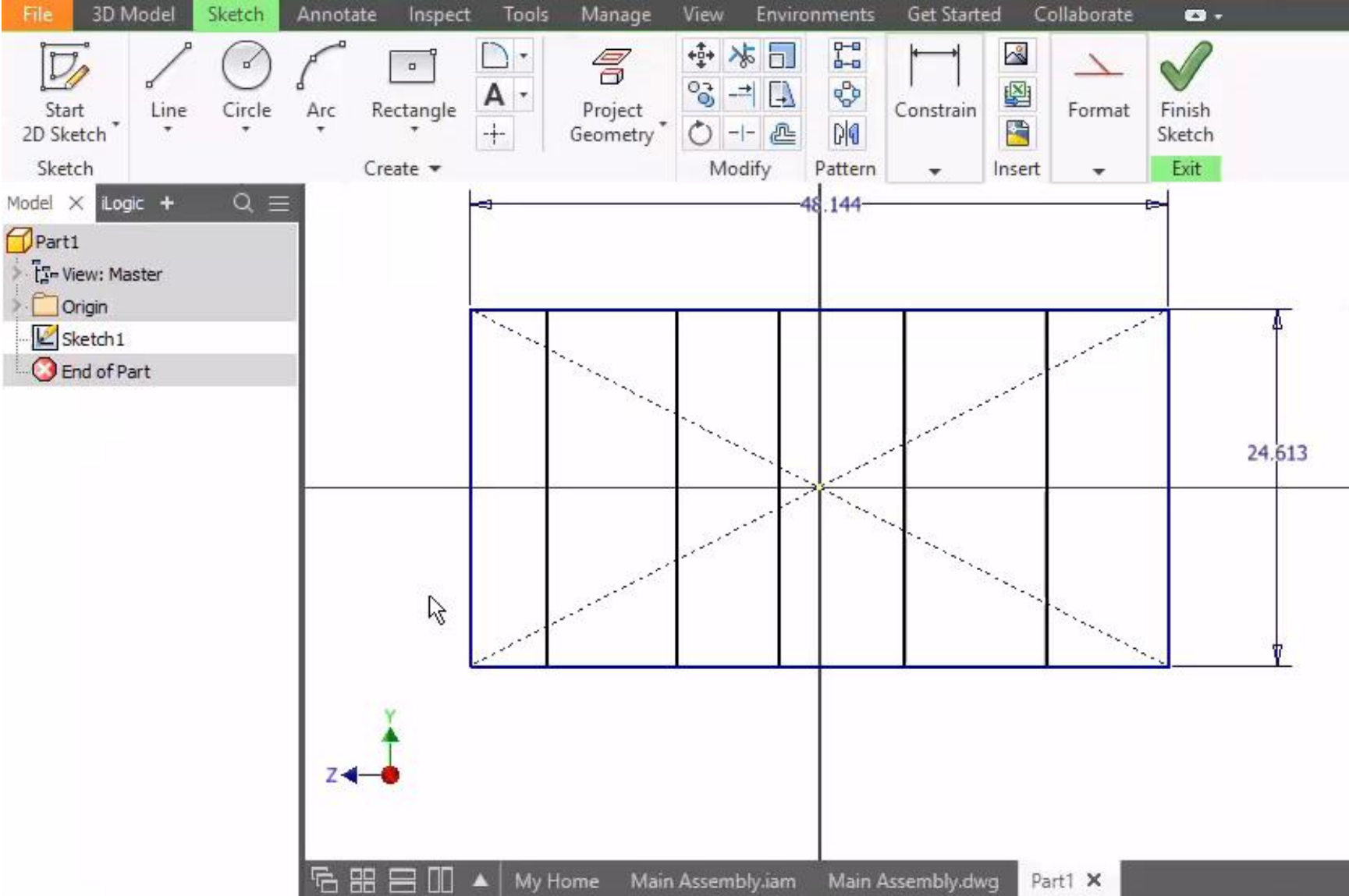
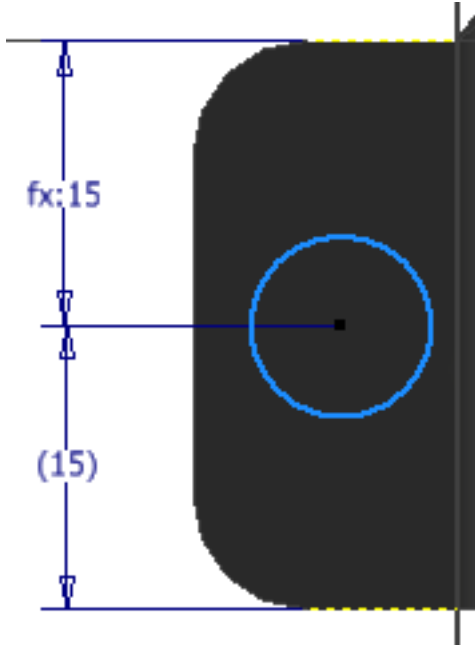
# Sketches: Change Display Colour + Linetype

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

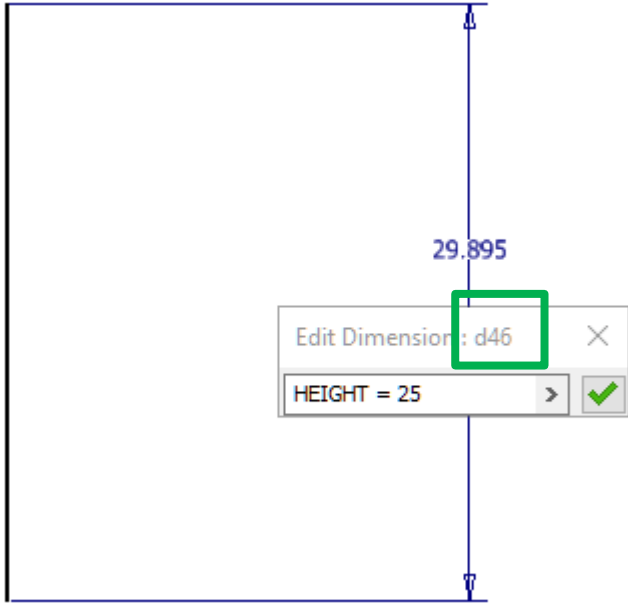


# Sketches: Equal Spacing

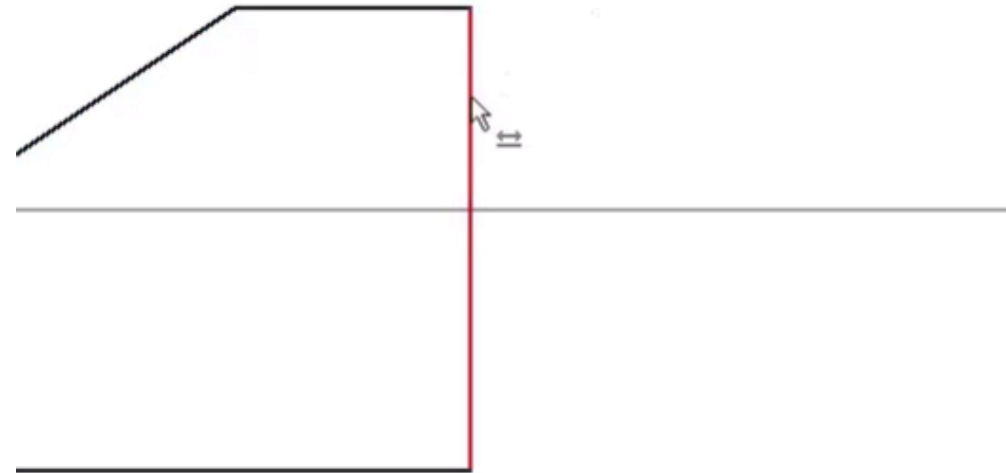
Use a driven dimension as a reference to space geometry evenly



# Parts/Sketches: Parameter naming on the fly

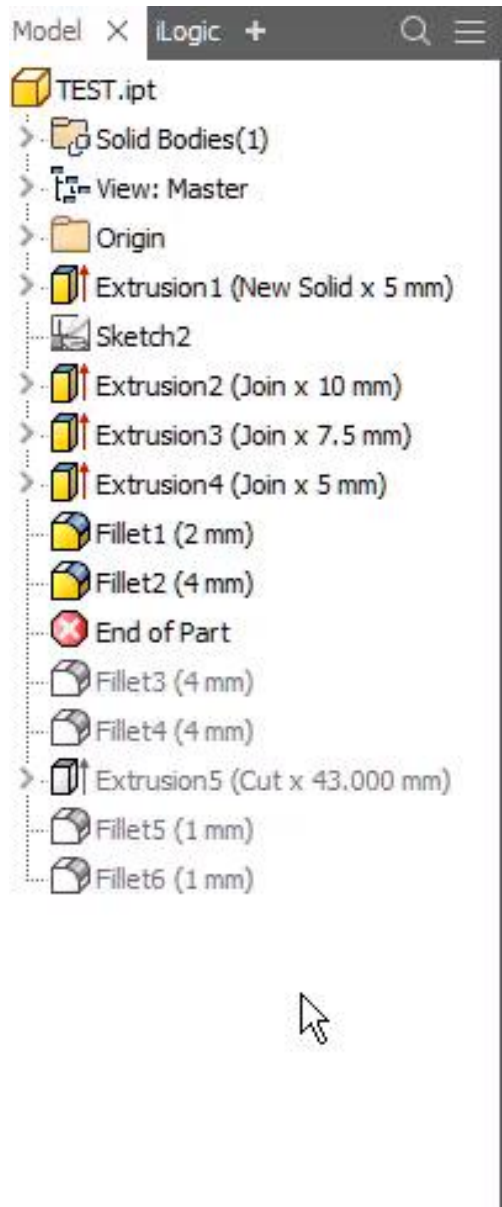


Save time by naming your parameters as you create them

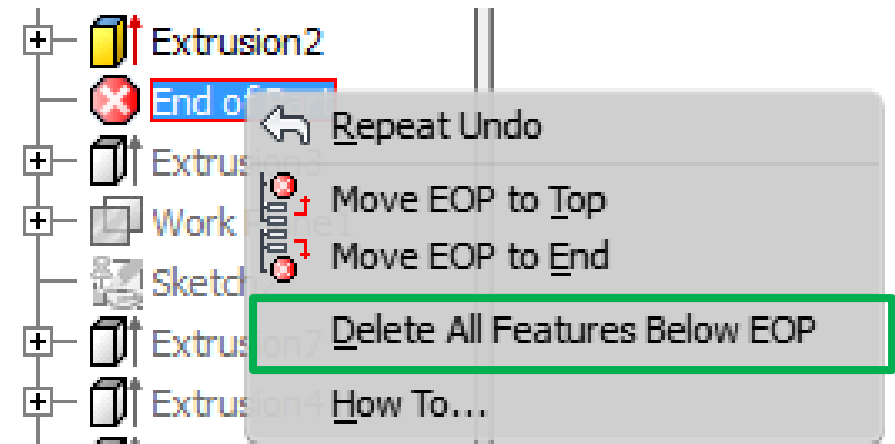




# Parts: EOP – Delete Features below the EOP



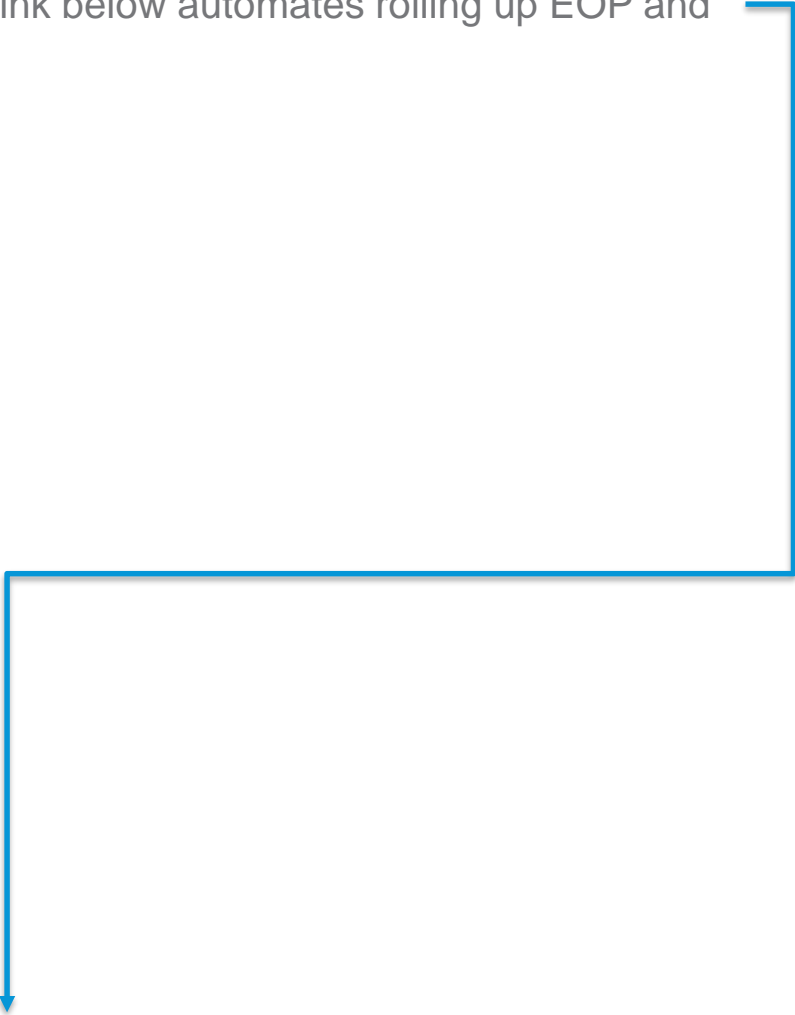
All features below the EOP can be deleted by simply right clicking on the EOP and selecting “Delete All Features Below EOP”



# 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)
- Extrusion2 (Join x 2.5 mm)
- 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
- End of Part

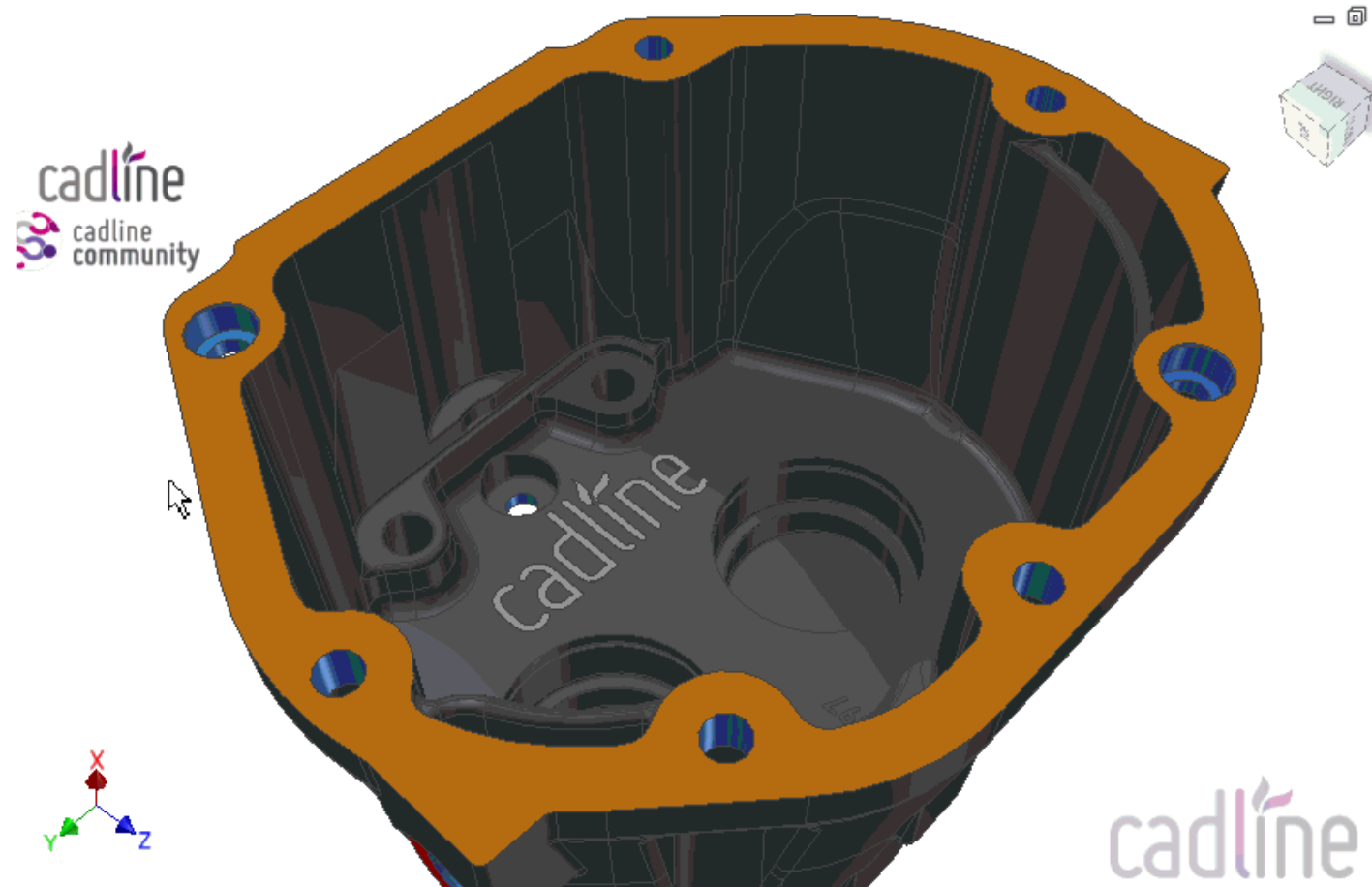
Location:	C:\Users\ClintBrown\Google Drive\Work
Size:	384.50 KB
Size on disk:	384.50 KB

- Origin
- End of Part
- Extrusion1 (New Solid x 90 mm)
- Extrusion2 (Join x 2.5 mm)
- 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

Location:	C:\Users\ClintBrown\Google Drive\Work
Size:	218.00 KB
Size on disk:	218.00 KB

# Parts: Flat Faces to DXF

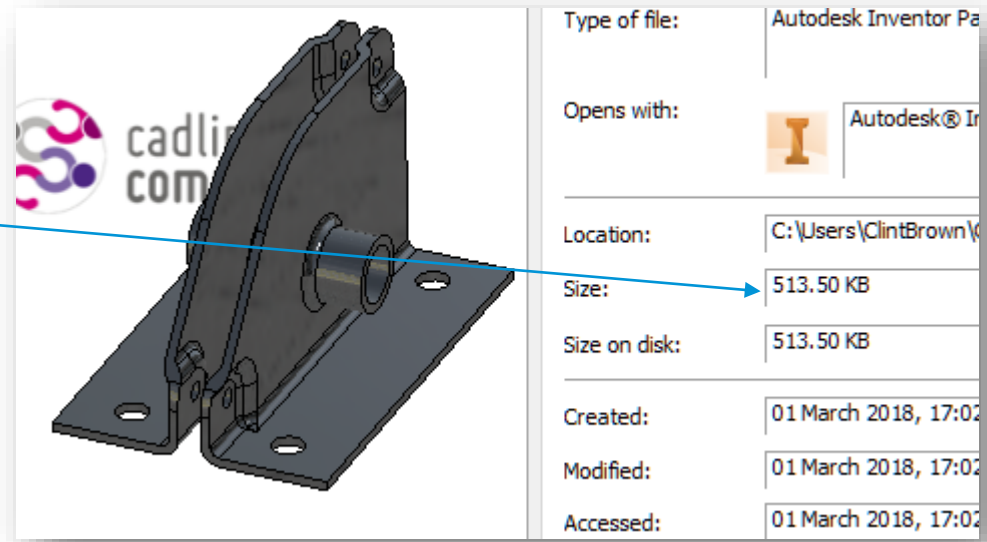
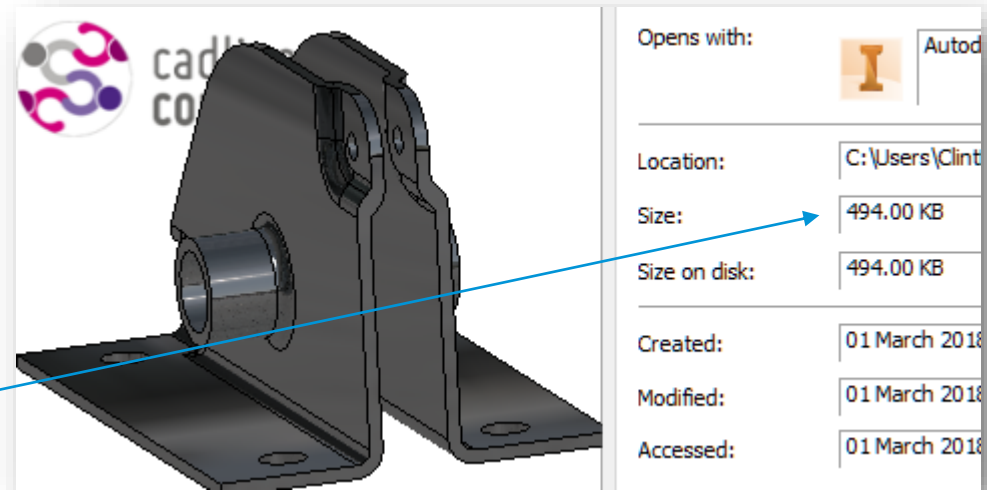
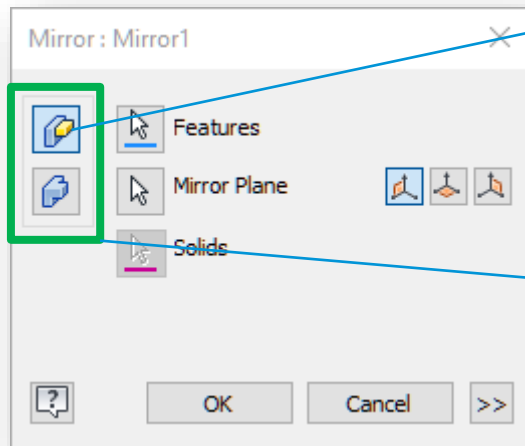
Any Flat face in an Inventor part file can be exported to DXF with a simple right click



# Parts: Mirror Solids

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

Sometimes with bigger file size



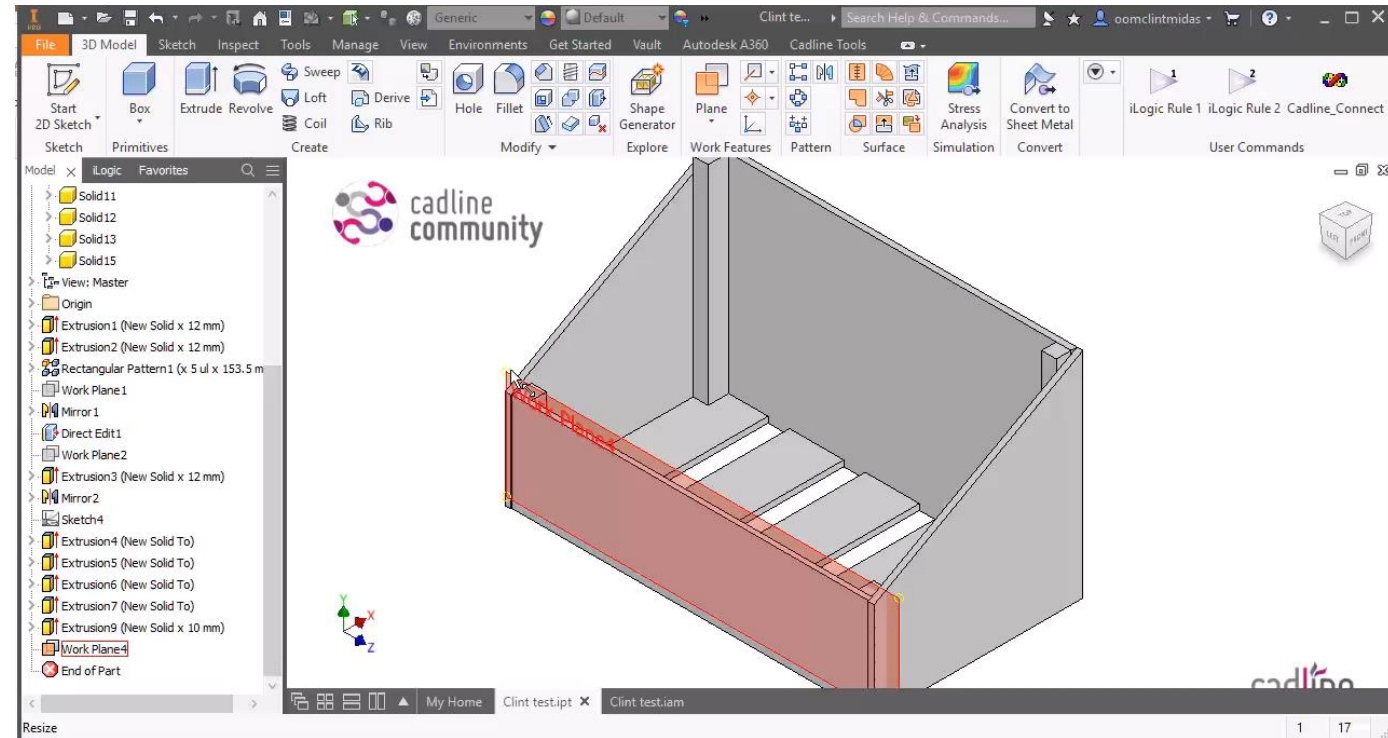
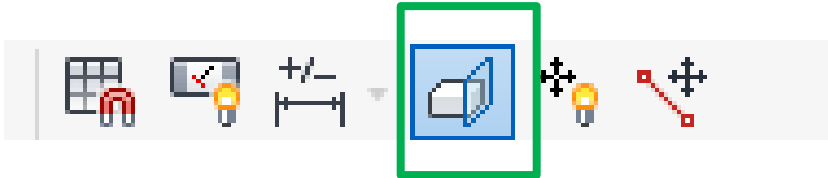
# Parts: Slice Graphics - Sketches

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

Keyboard shortcut is **F7**

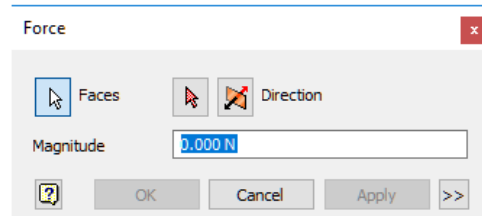
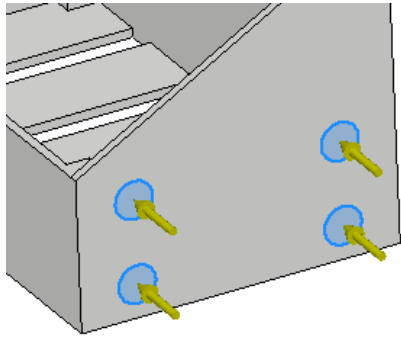
Inventor now has a neat toggle in the sketch environment

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

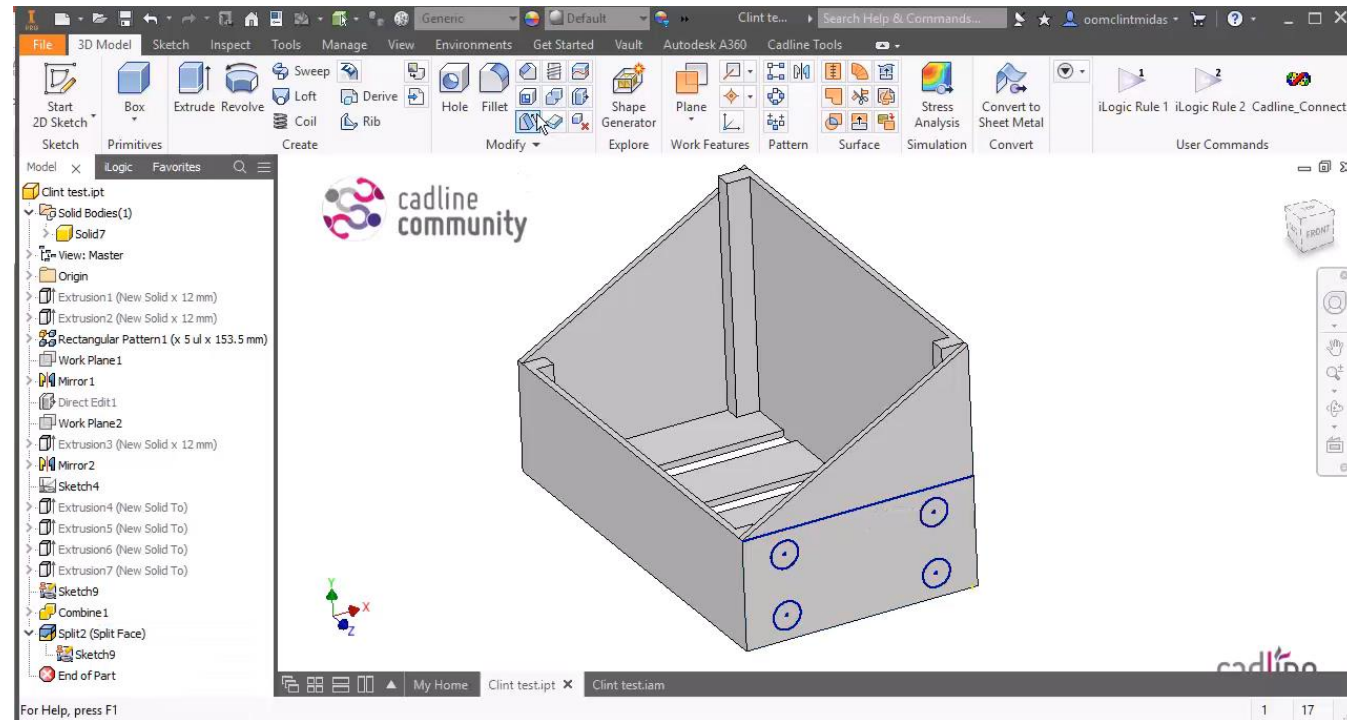
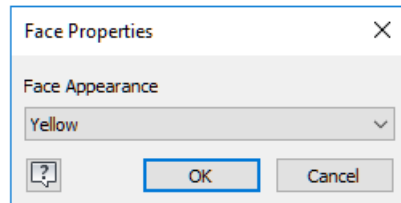
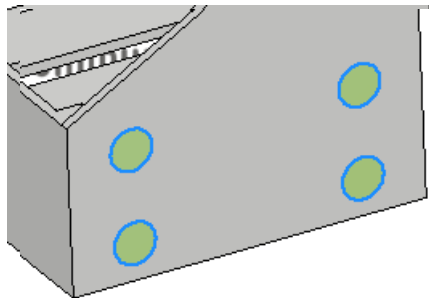


# Parts: Split Command – Split faces

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.

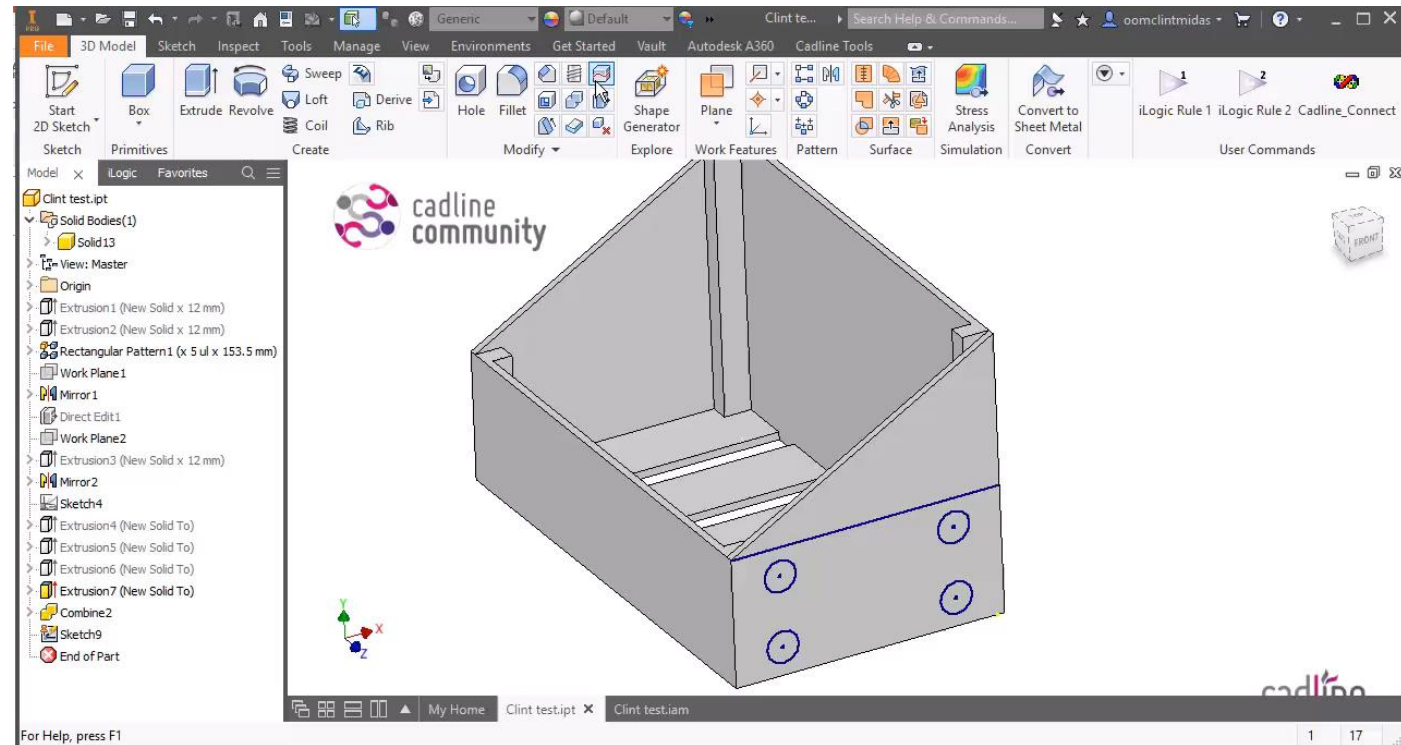
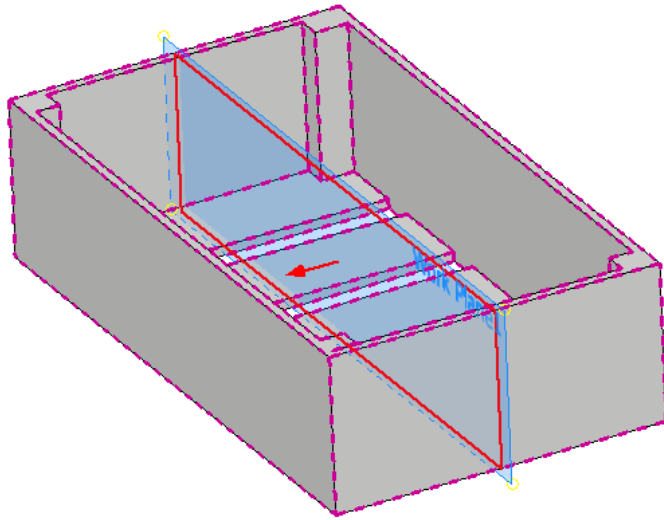
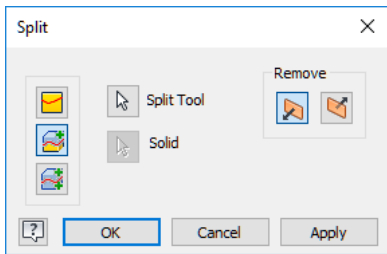




# Parts: Split Command – Trim Solid

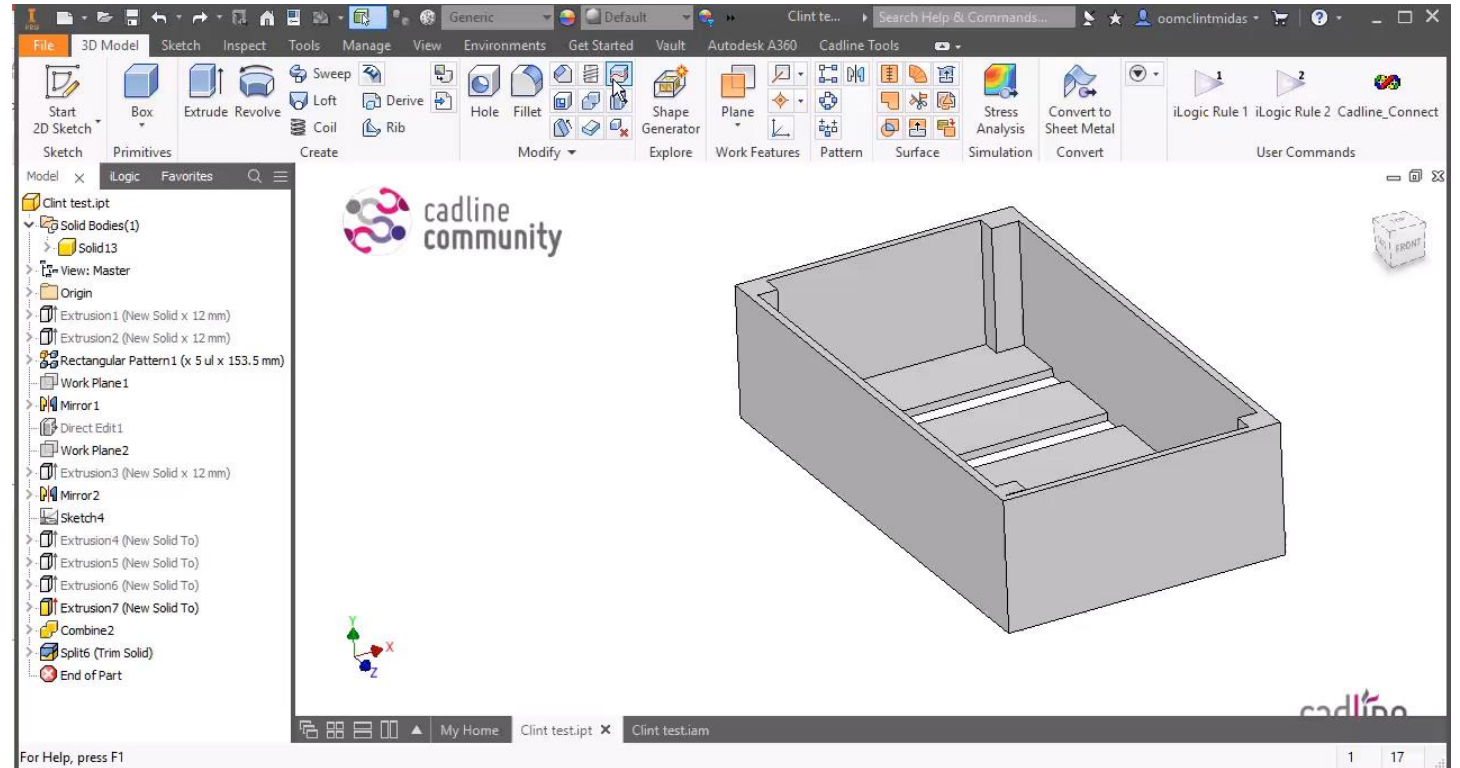
Use “Trim Solid” to cut parts to size

Bonus Tip: Use work planes as the Split Tool



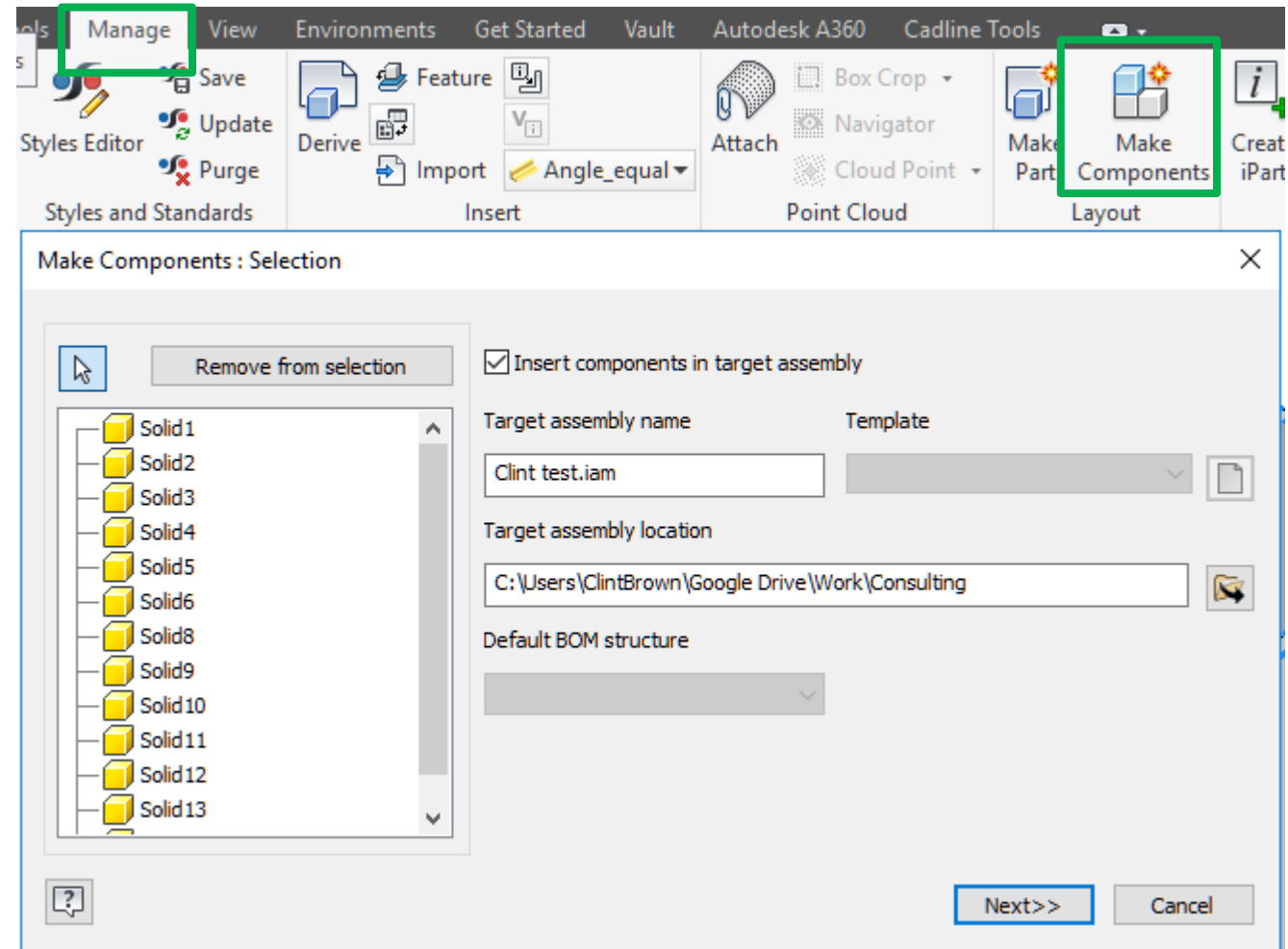
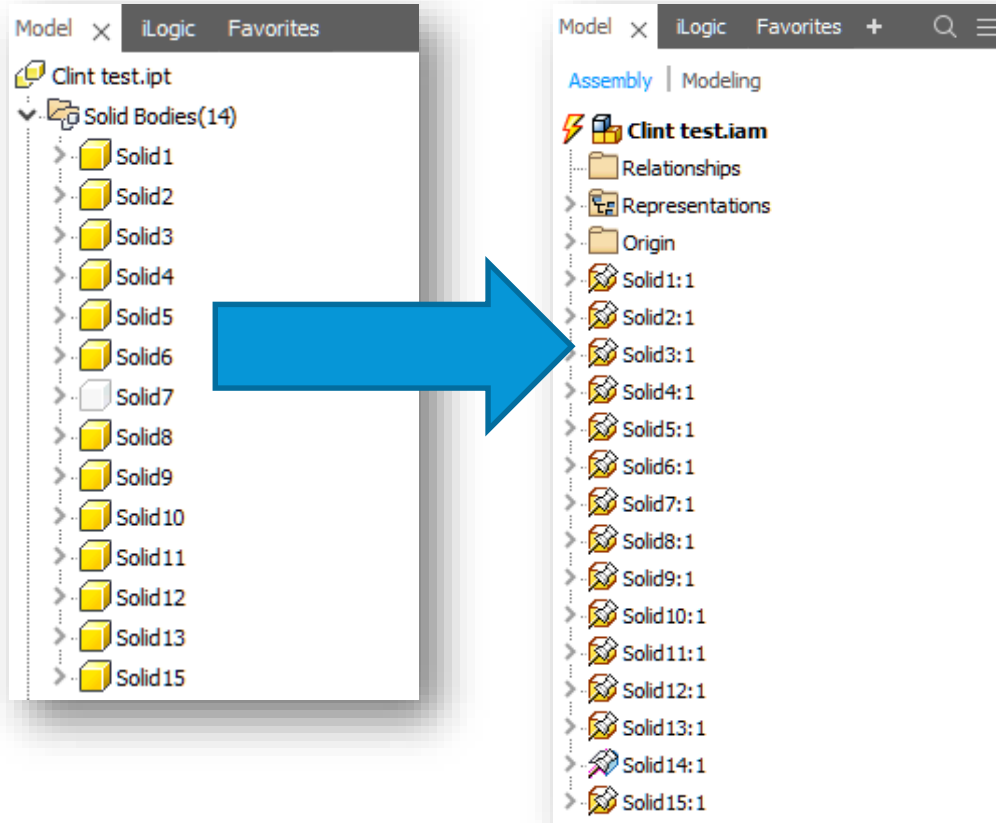
# Parts: Split Command – Split Solid

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



# Parts: Multi-body solids & Make Components

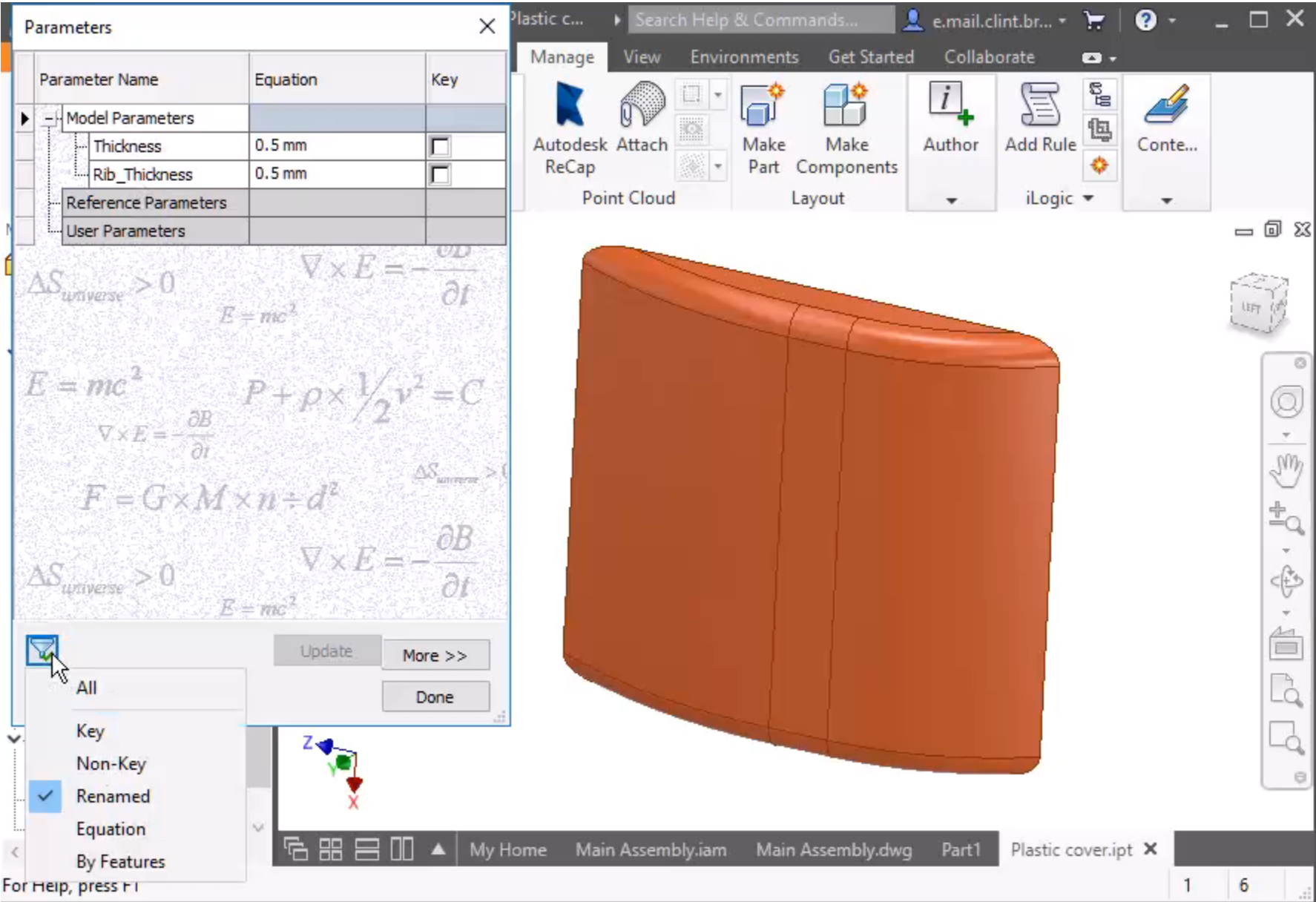
Convert Multi-Body Solids to Assemblies using the Make Components command



# 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

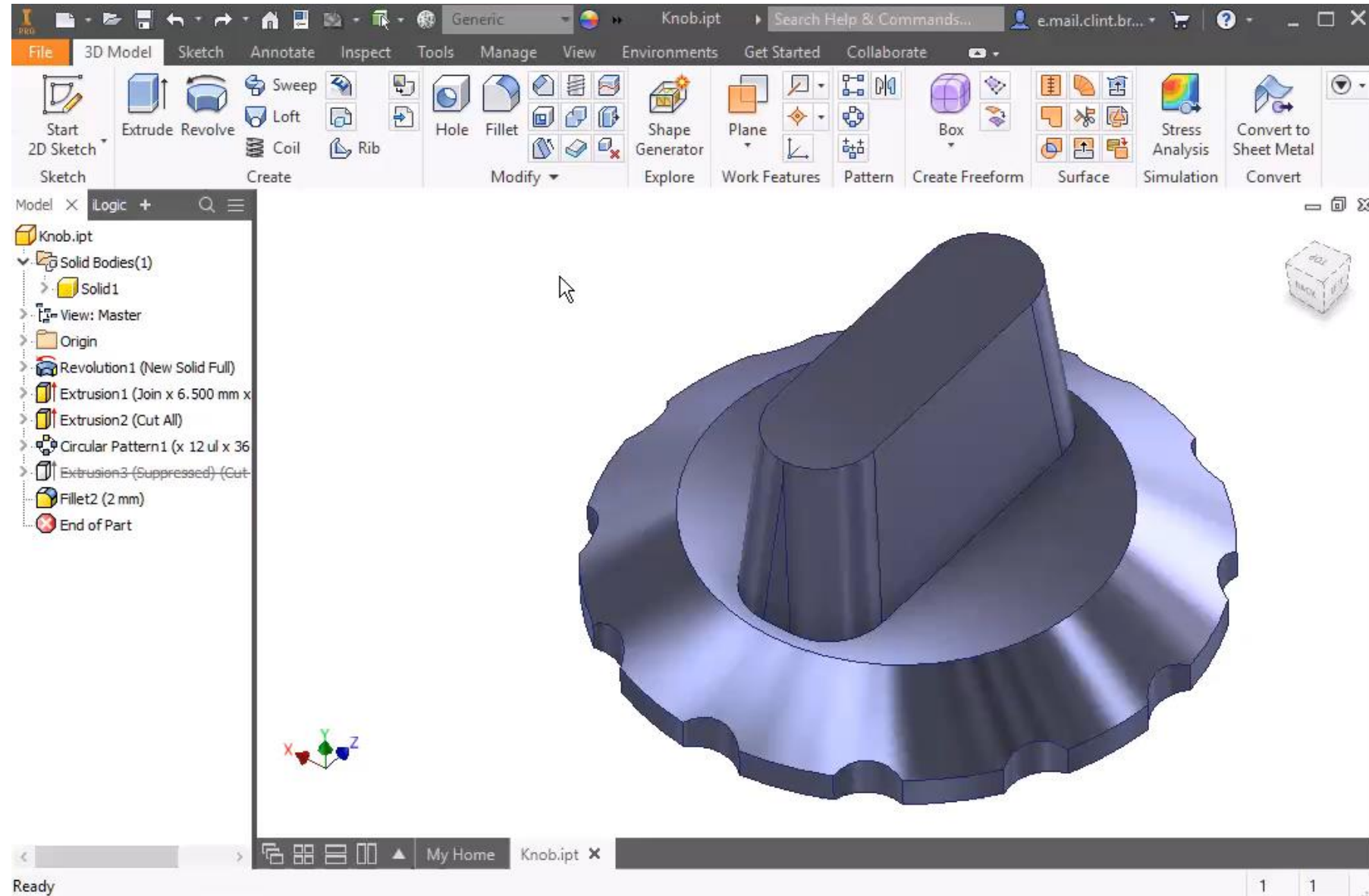




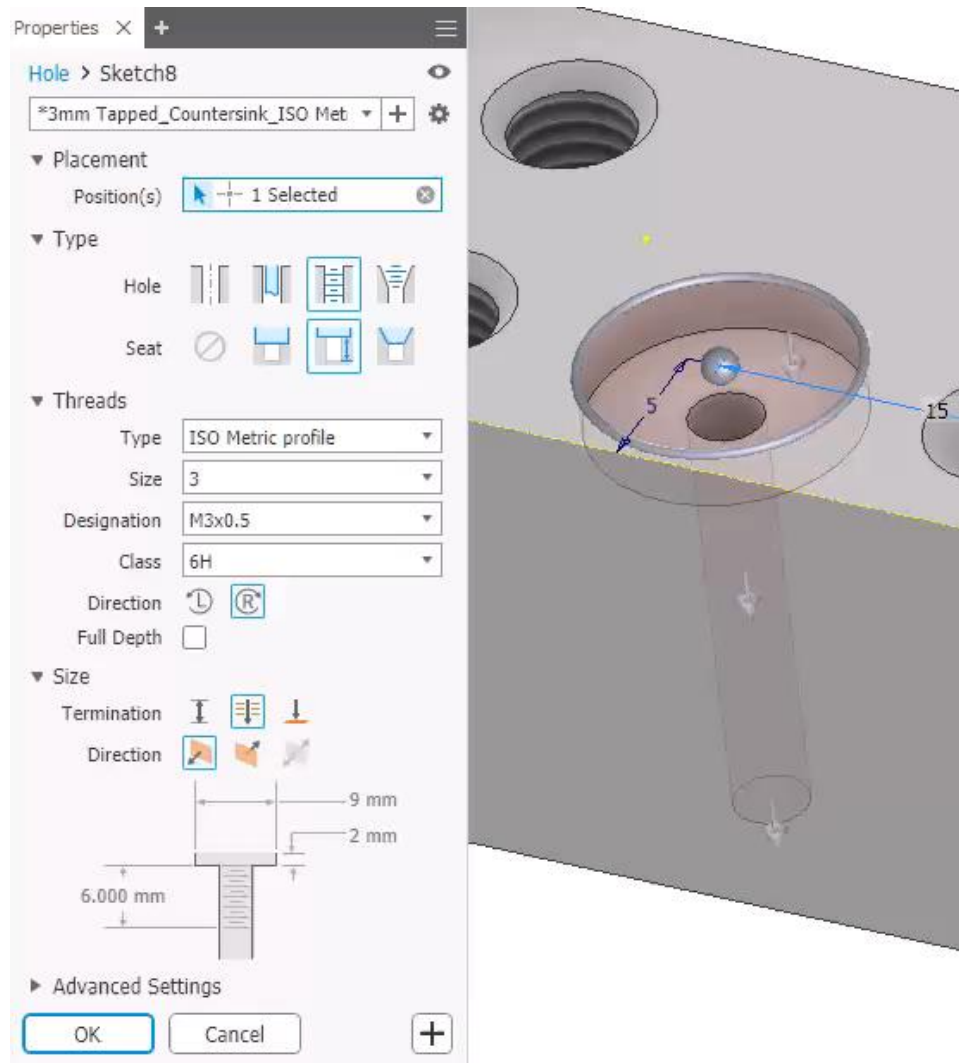
# Parts: All Fillets + All Rounds

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

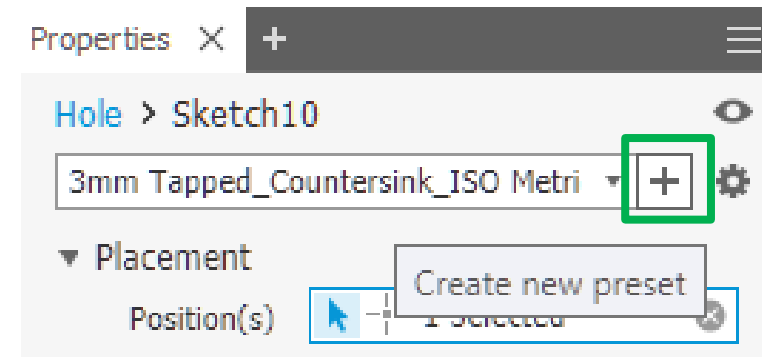
The feature will continue to calculate the fillets, even after the geometry changes



# 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:

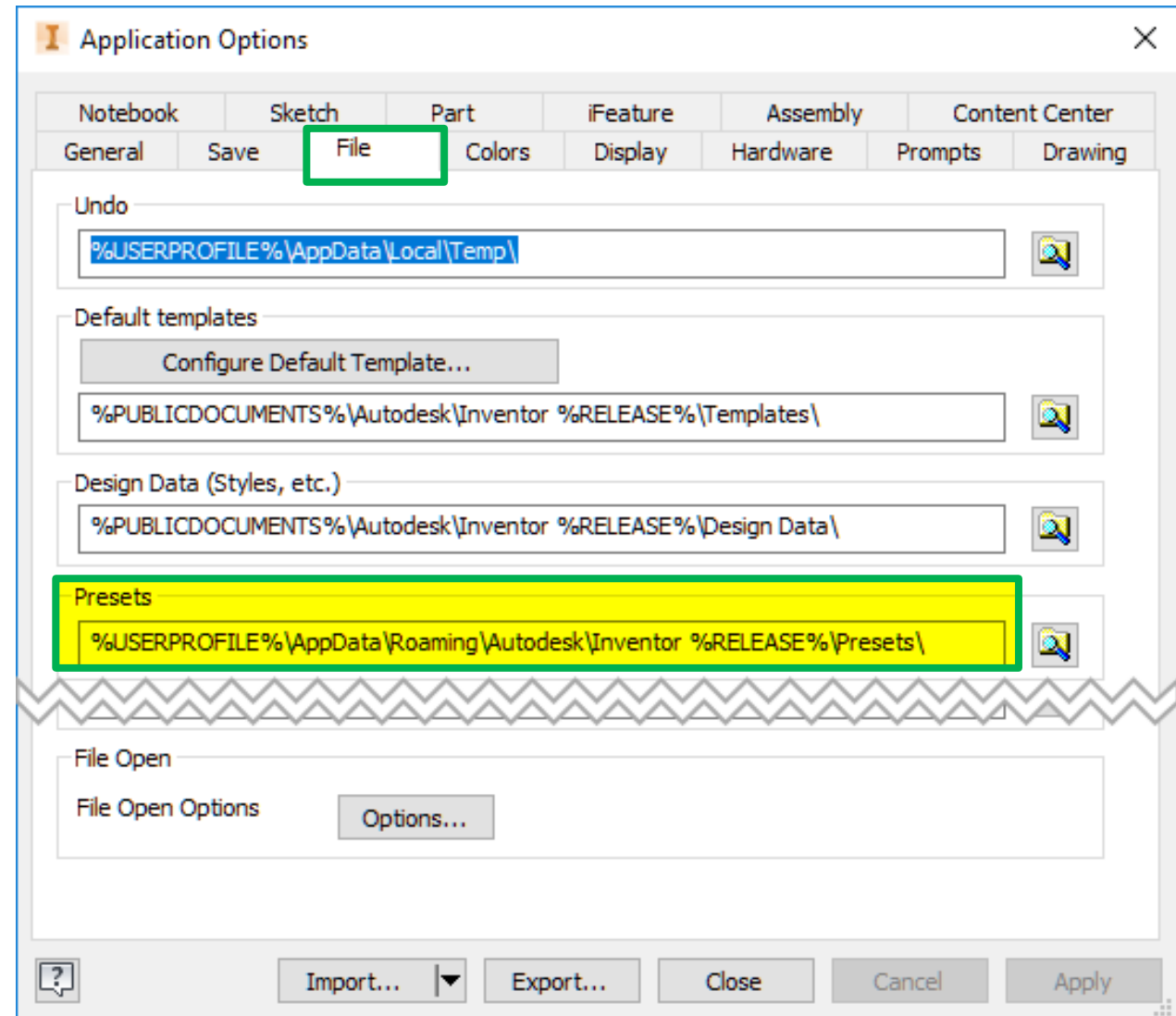
- Choose it from the drop down
- Modify the settings to suite
- Save the current setting



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

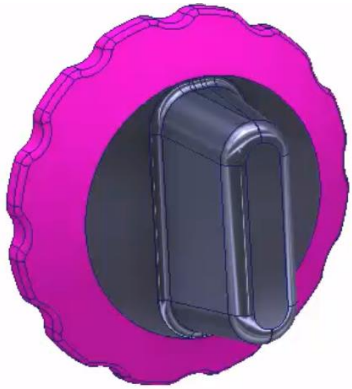
# Parts: Sharing Hole Pre-sets (2019)

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



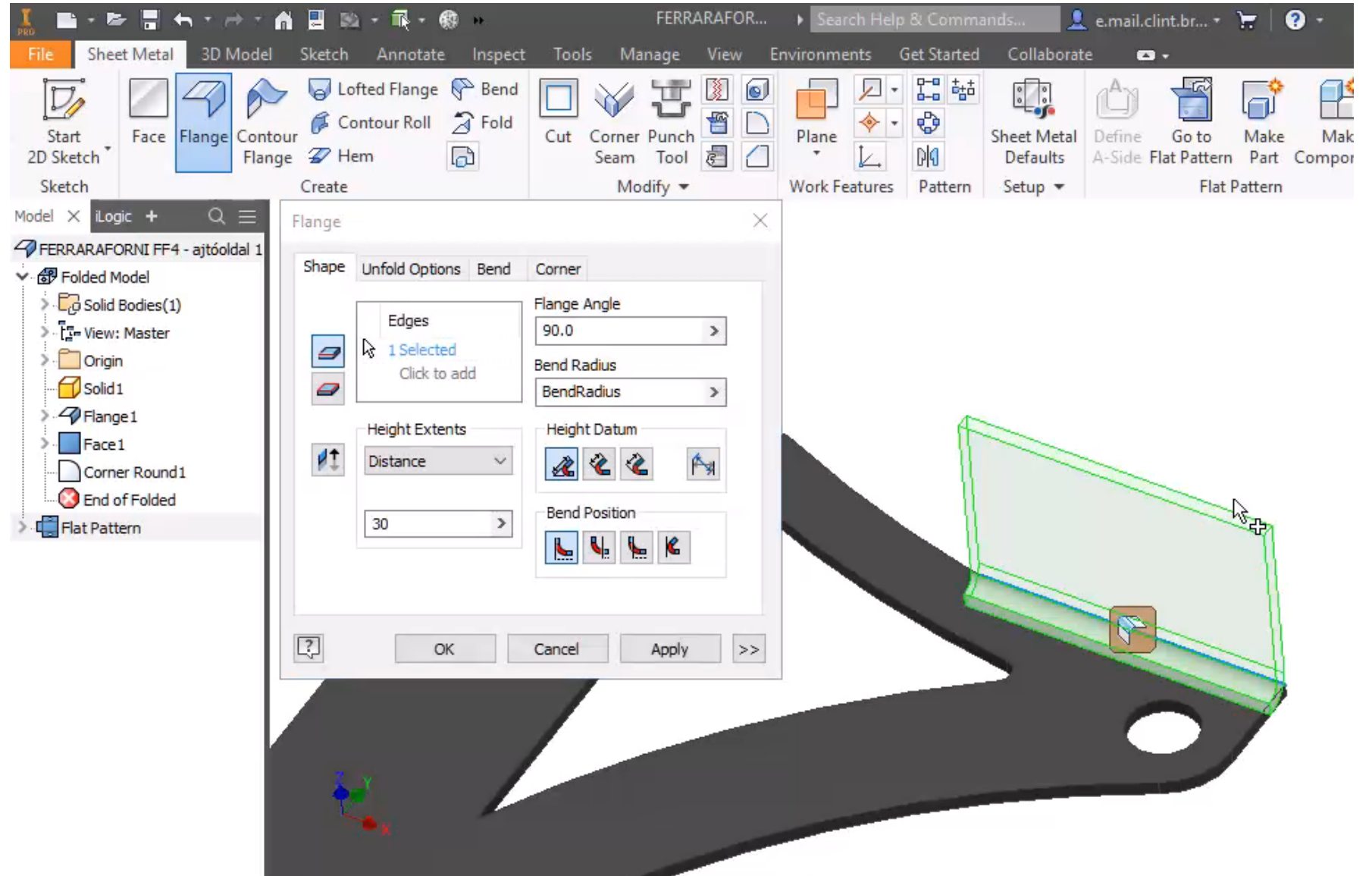
# Parts: Select Tangencies

“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

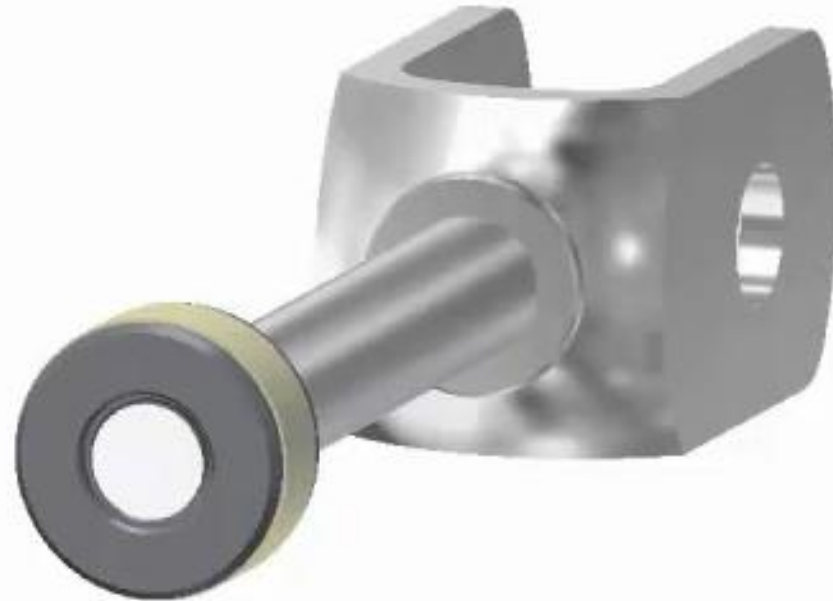
Use “Perspective” views for renders and images

Tip: it’s faster to work in Orthographic mode





# Get some Perspective

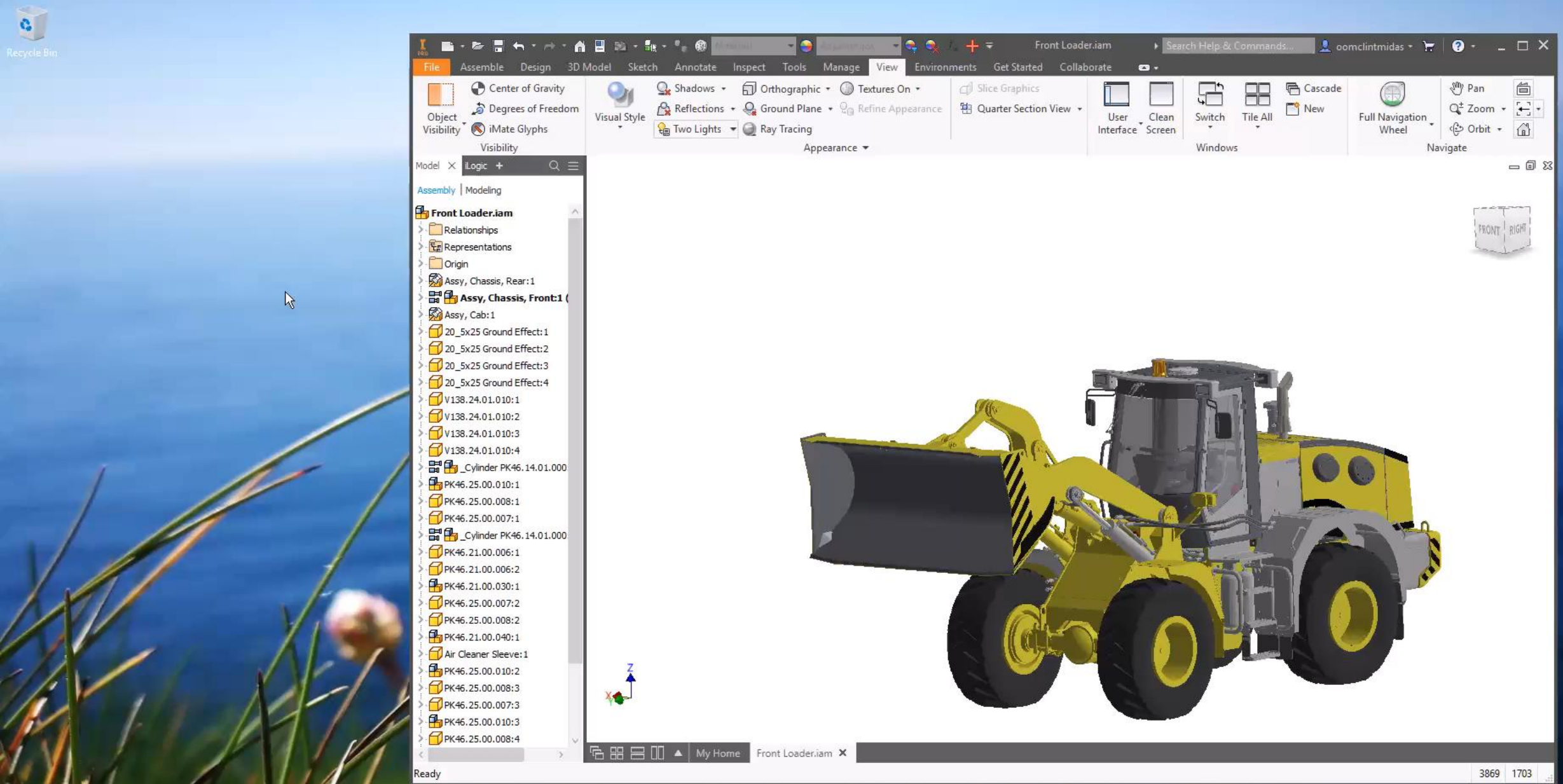


## **Perspective (Ctrl + Shift + F3)**

To adjust the level of perspective

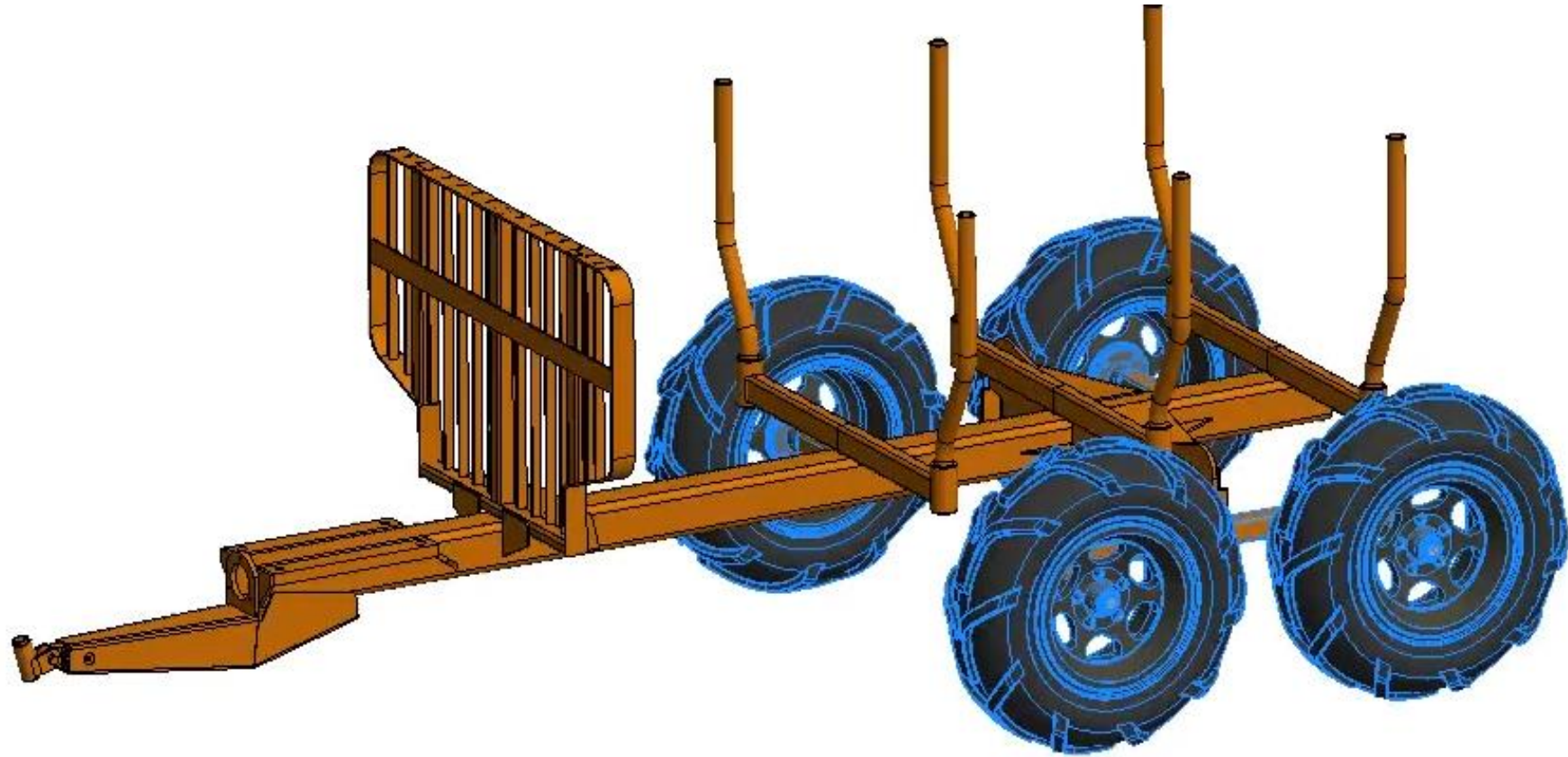
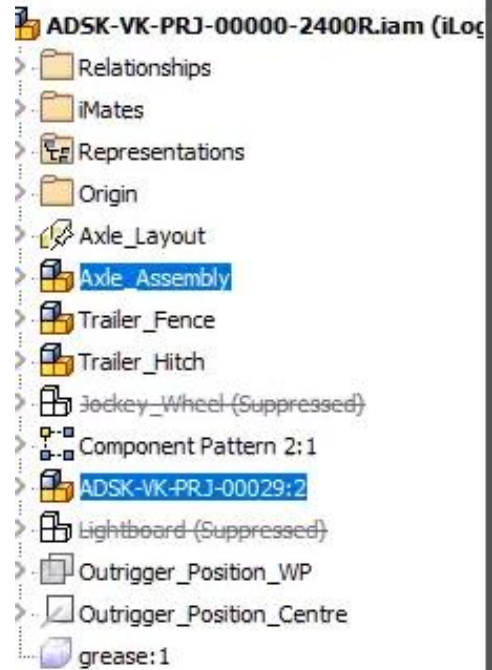
Ctrl + Shift + F3 and scroll up and down.

# 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

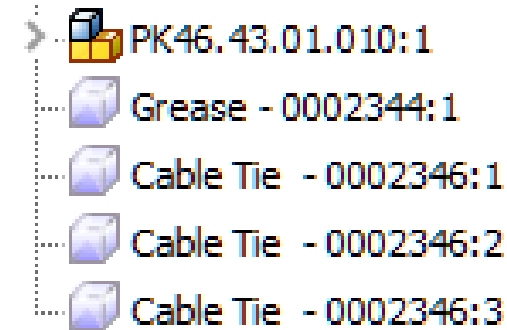
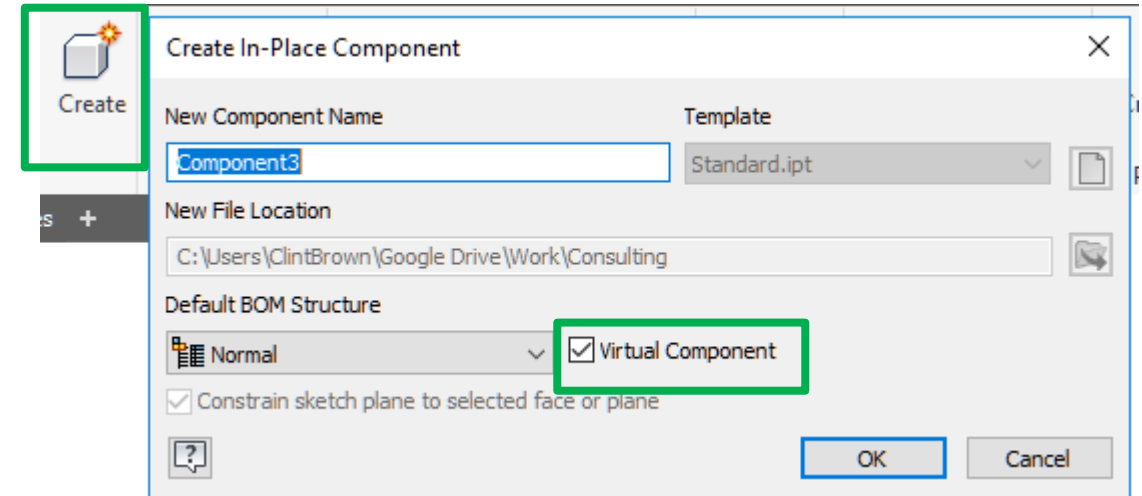
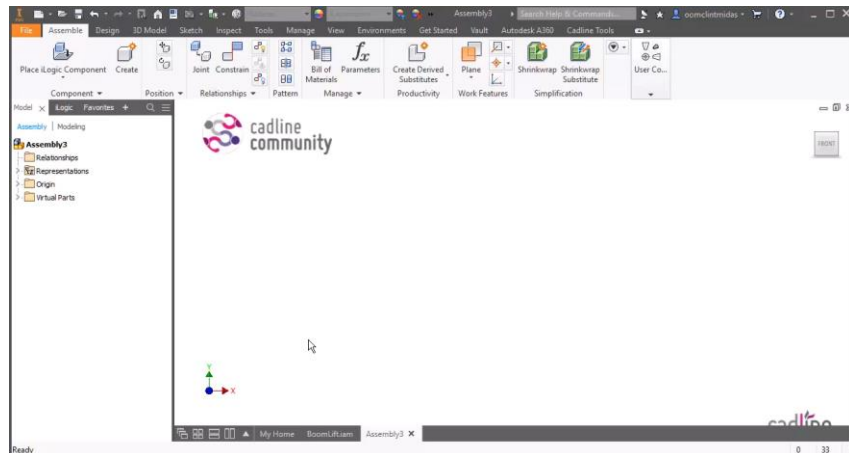


# Assemblies: Virtual Components

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

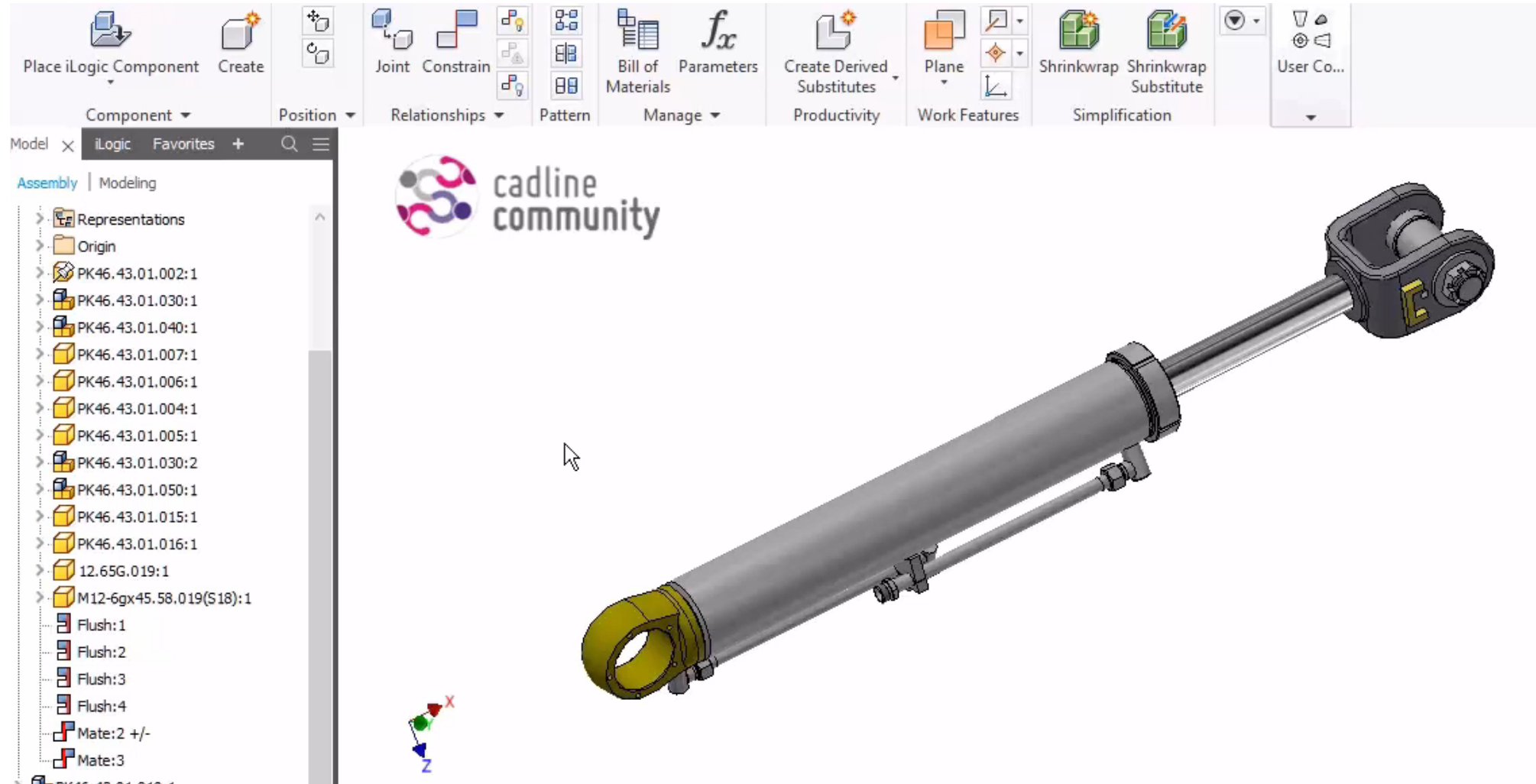
**Bonus Tip 2:** Save commonly used Virtual Components in your Assembly file





# 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

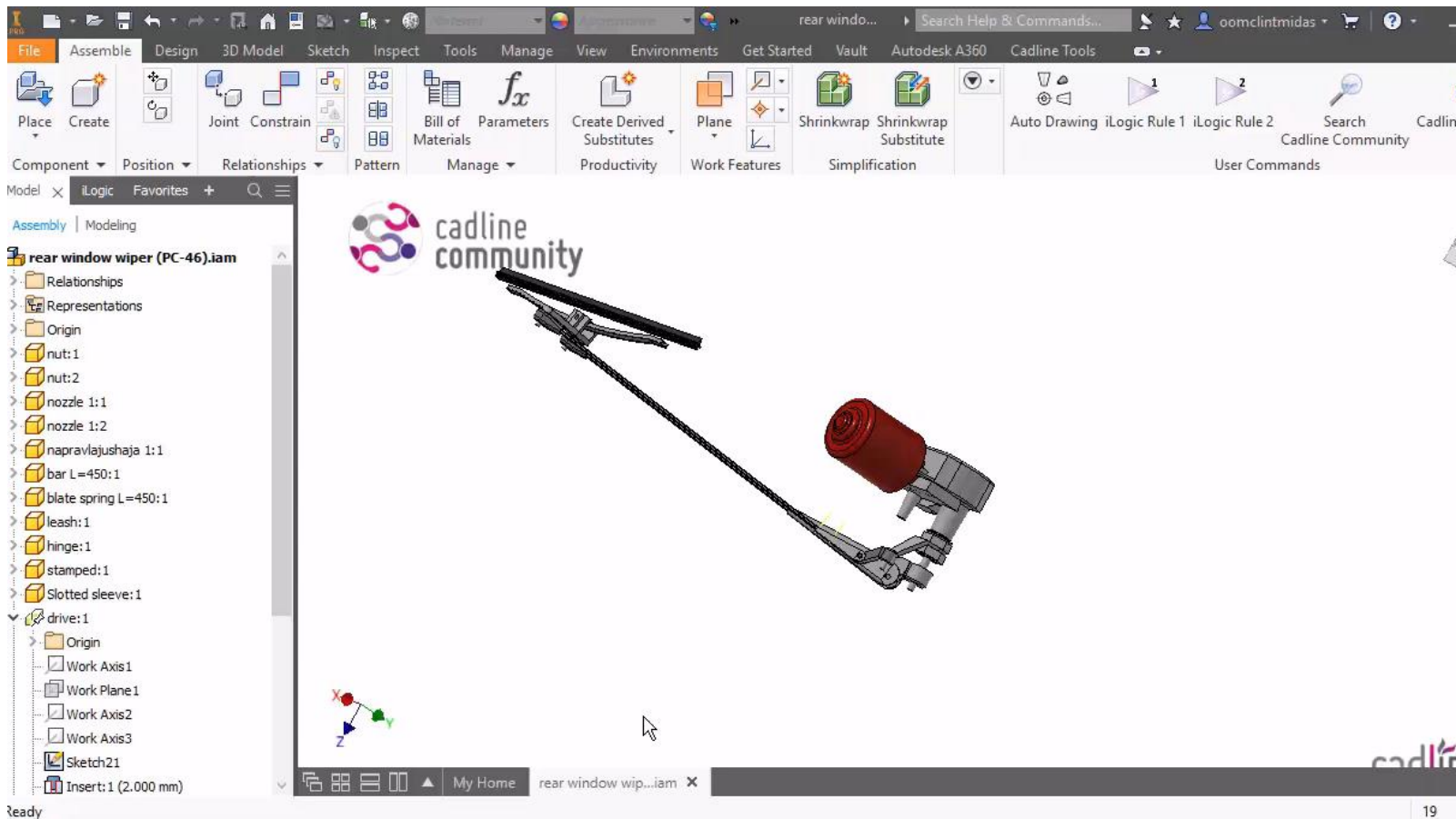
CAPITALISE

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

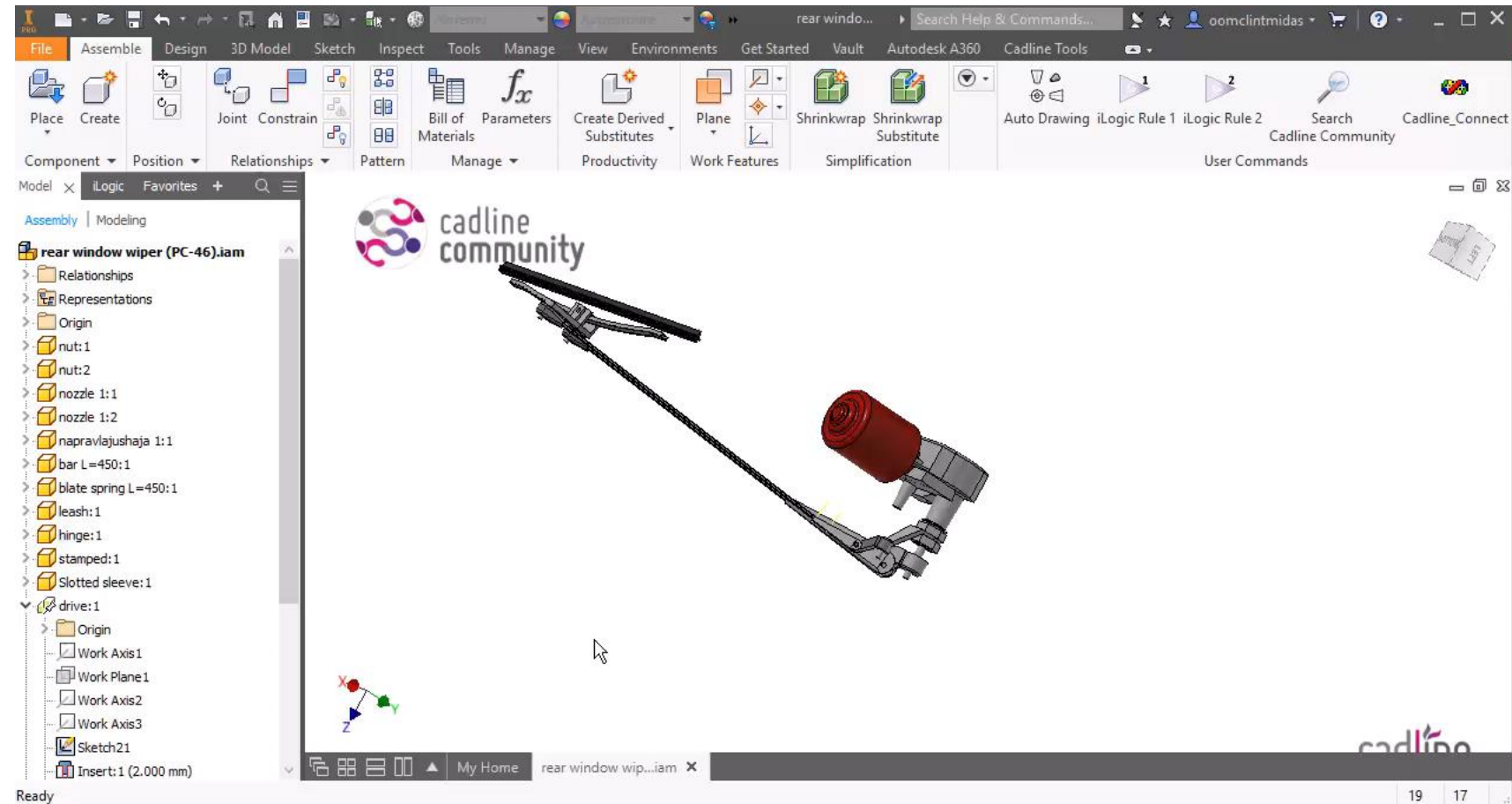
Set up your BoM template.

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!

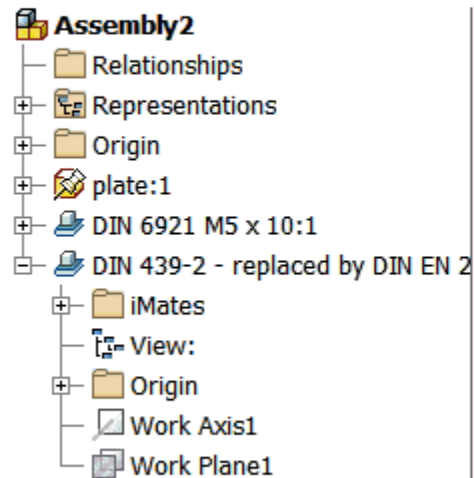


# Assemblies: Alt + Drag Constraints

## Alt + Drag

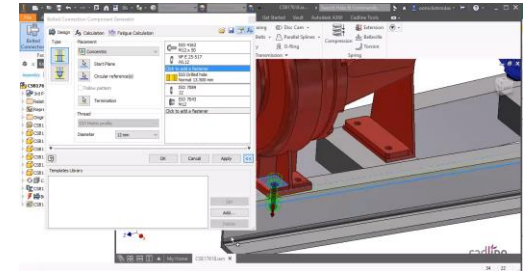
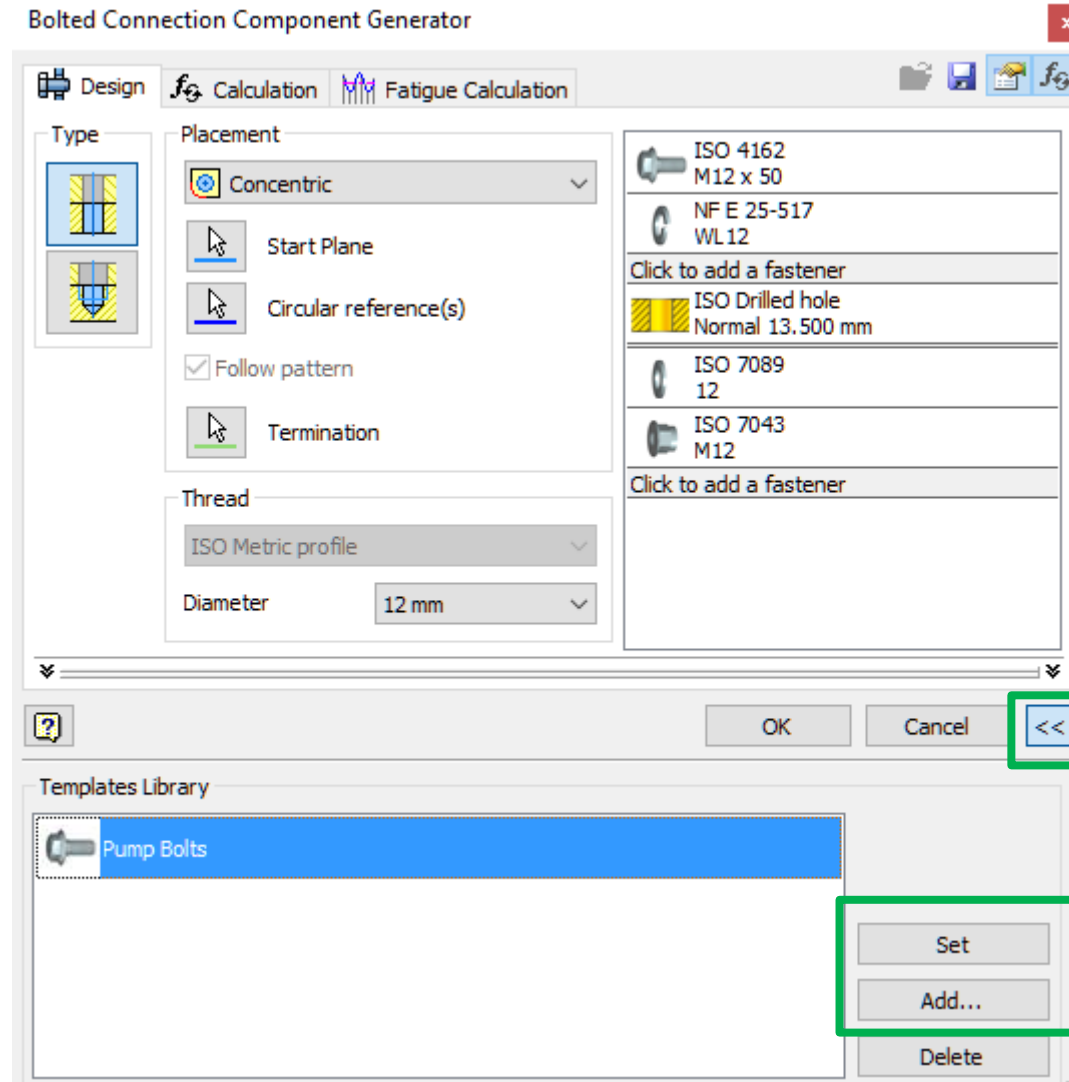
If you hold down ALT and then drag (hold down your left mouse button) geometry it will create a constraint, based on the geometry that you select, the animation below shows how I have placed an insert constraint.

**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:

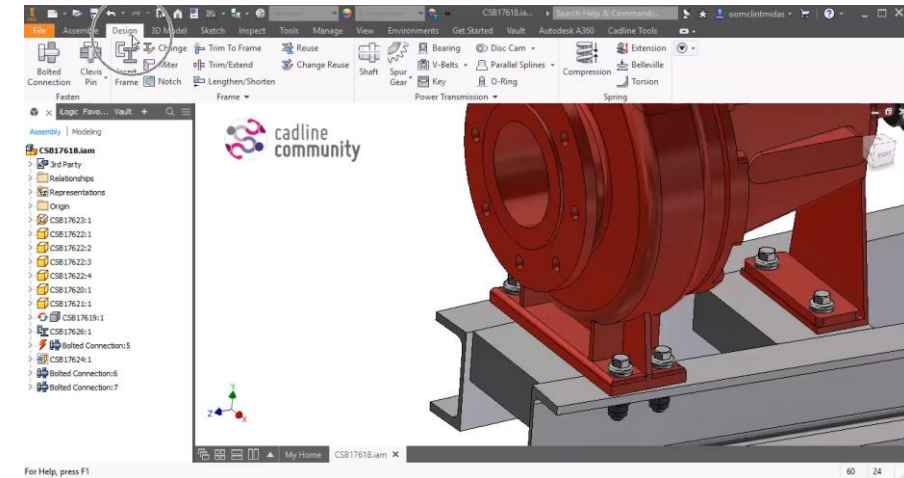
Type  
Insertion Position  
Mass

Family	BS 4 : Part 1 (Universal columns) - Stru
Size	203x203x71
Material	Steel, Mild
Appearance	As Material
Part Number	BS 4 - 203x203x71 - 5400
Length	5400.000 mm
Width	206.400 mm
Height	215.800 mm
Area	6604403.344 mm <sup>2</sup>
Volume	48830842.982 mm <sup>3</sup>
Mass	383.322 kg

Insert Position

Custom Point

Done

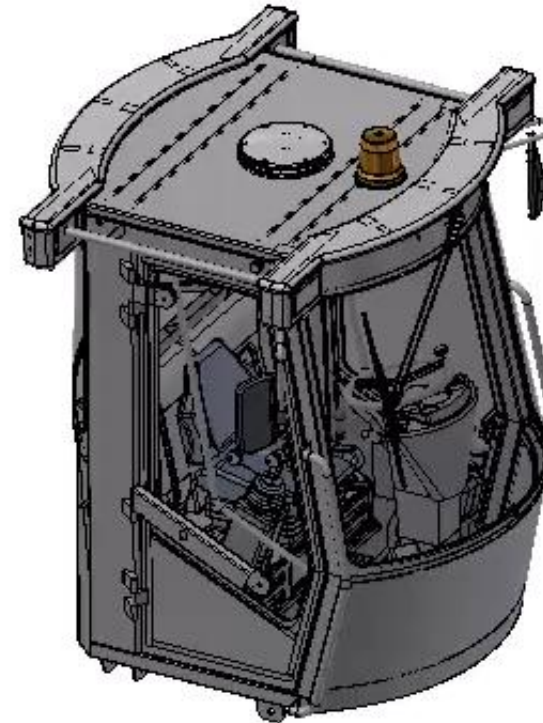
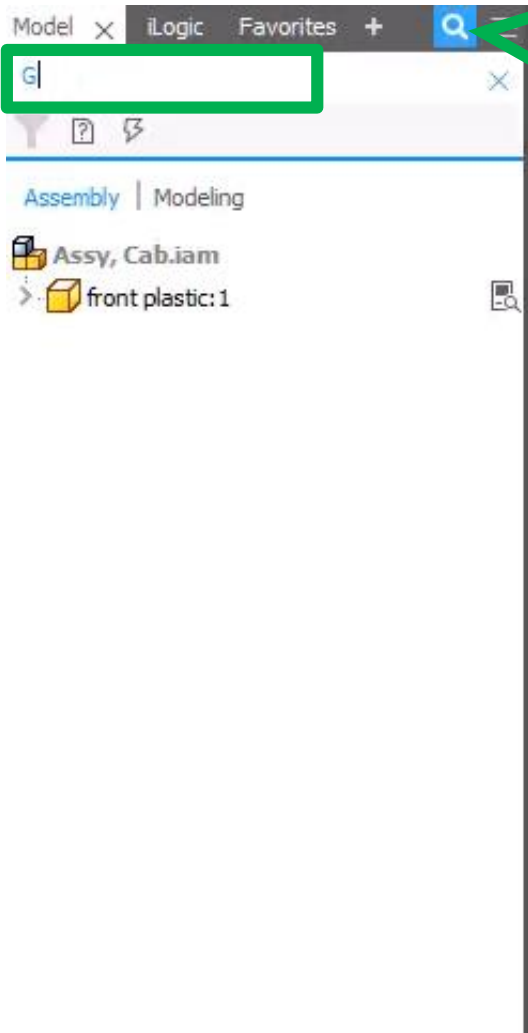




# Assemblies: Browser Search

Introduced with Inventor 2018.

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

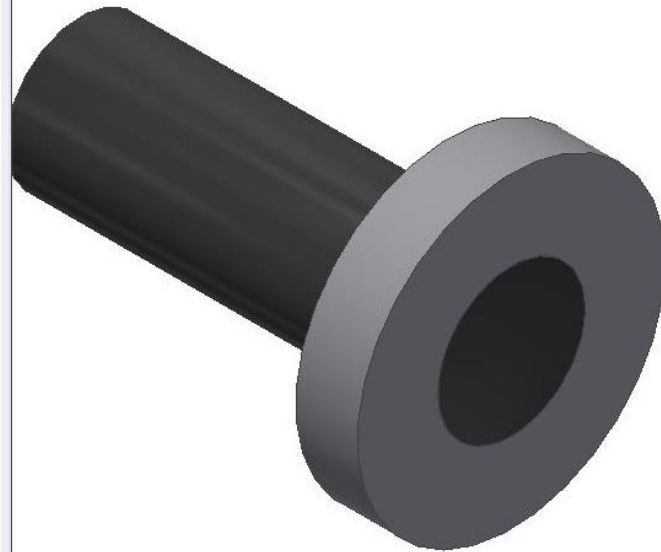
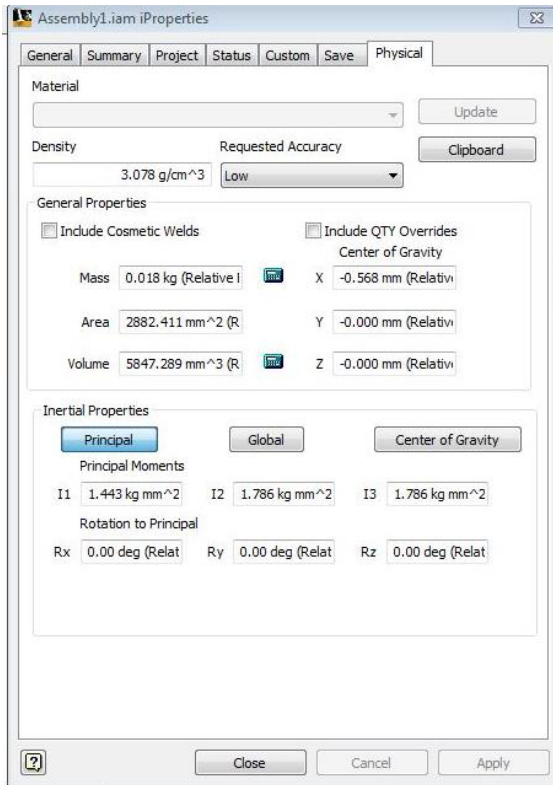


# Assemblies: Surface Area

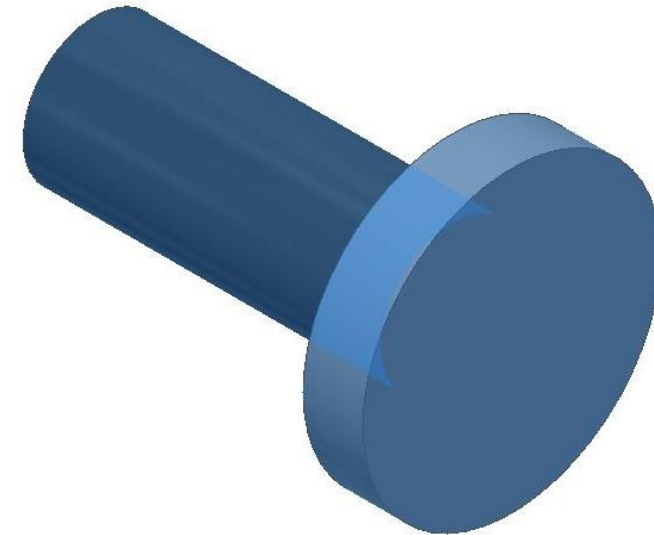
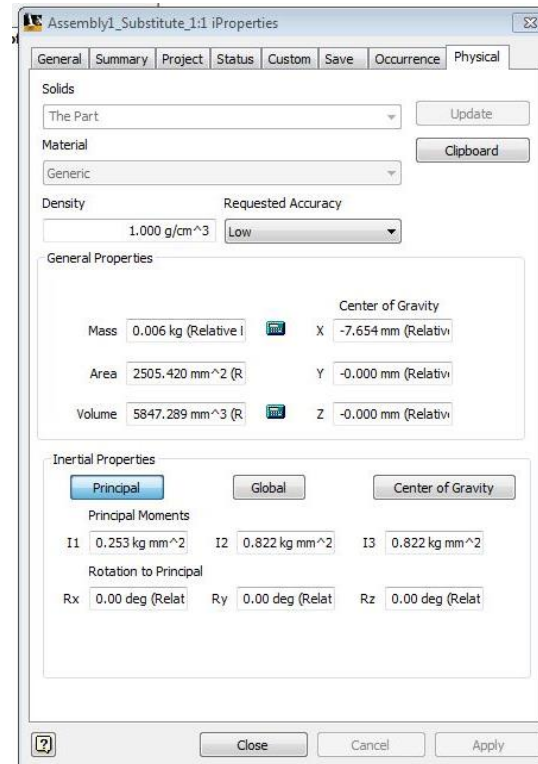
The Surface area shown in an assembly is the total Surface area of all parts in the assembly.

For Painting or Coating create shrinkwrap for a more accurate result

**Assembly = 2882.411mm<sup>2</sup>**



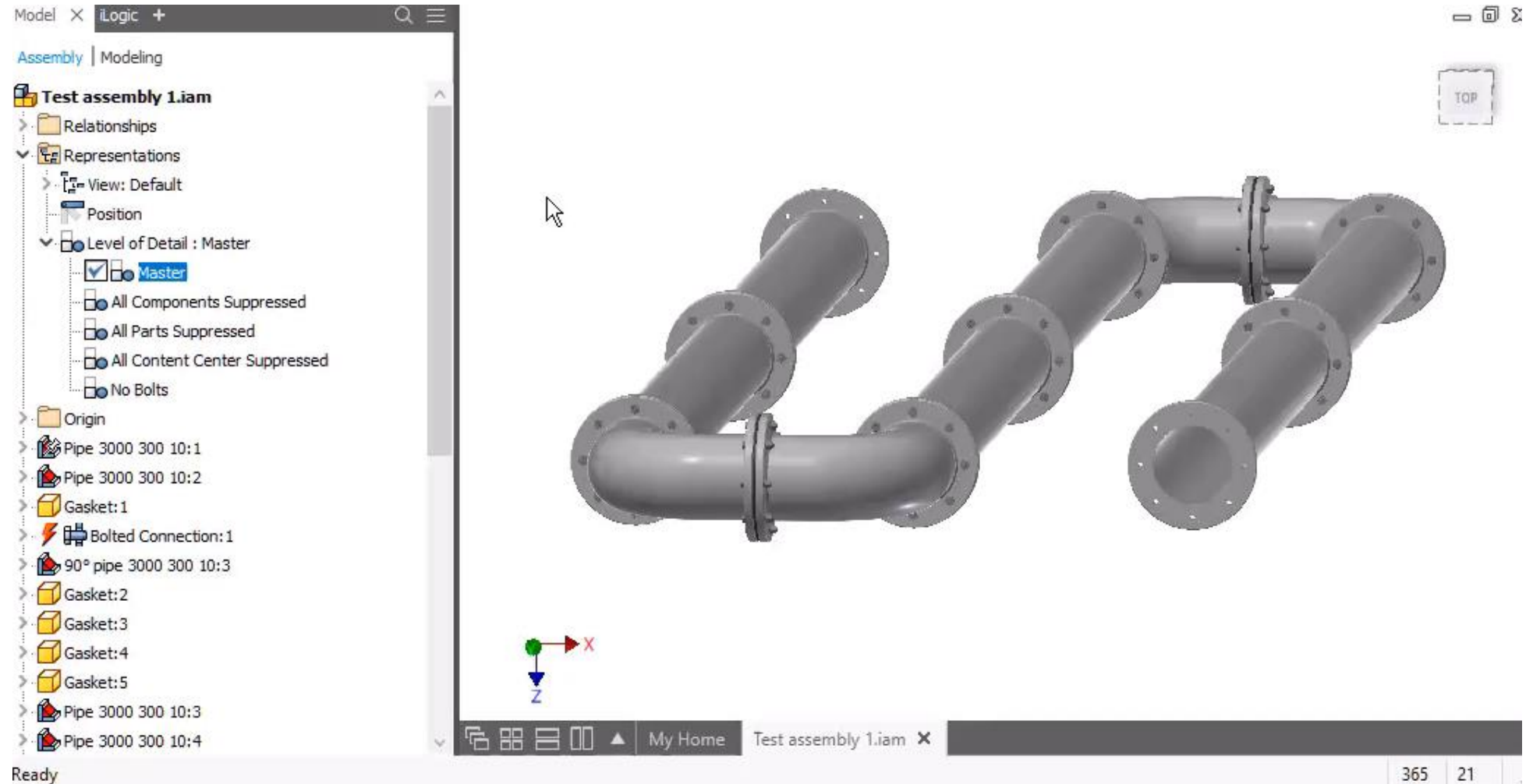
**Shrinkwrap = 2505.420mm<sup>2</sup>**



# Assemblies: Shrink Substitute LoD's via iLogic

Reduce model complexity with LoD's

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



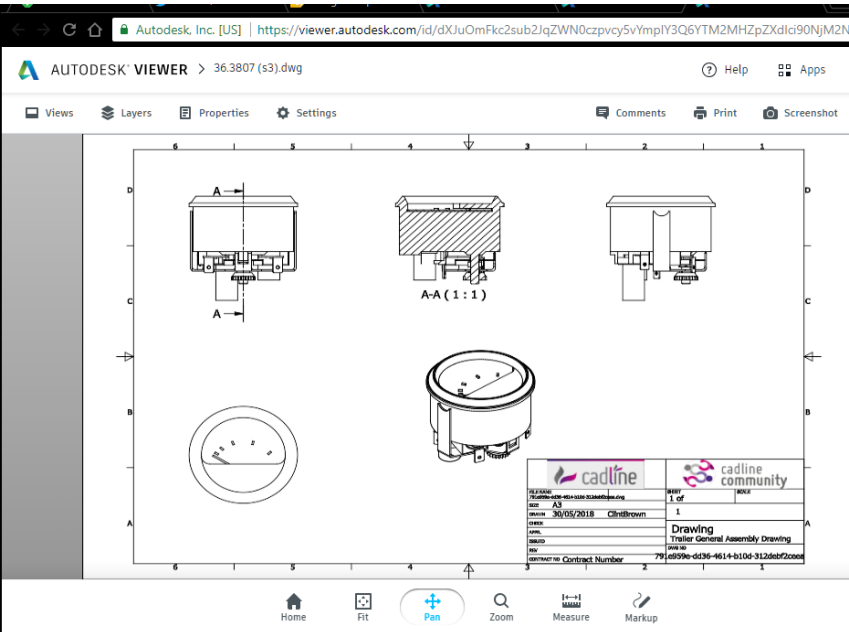
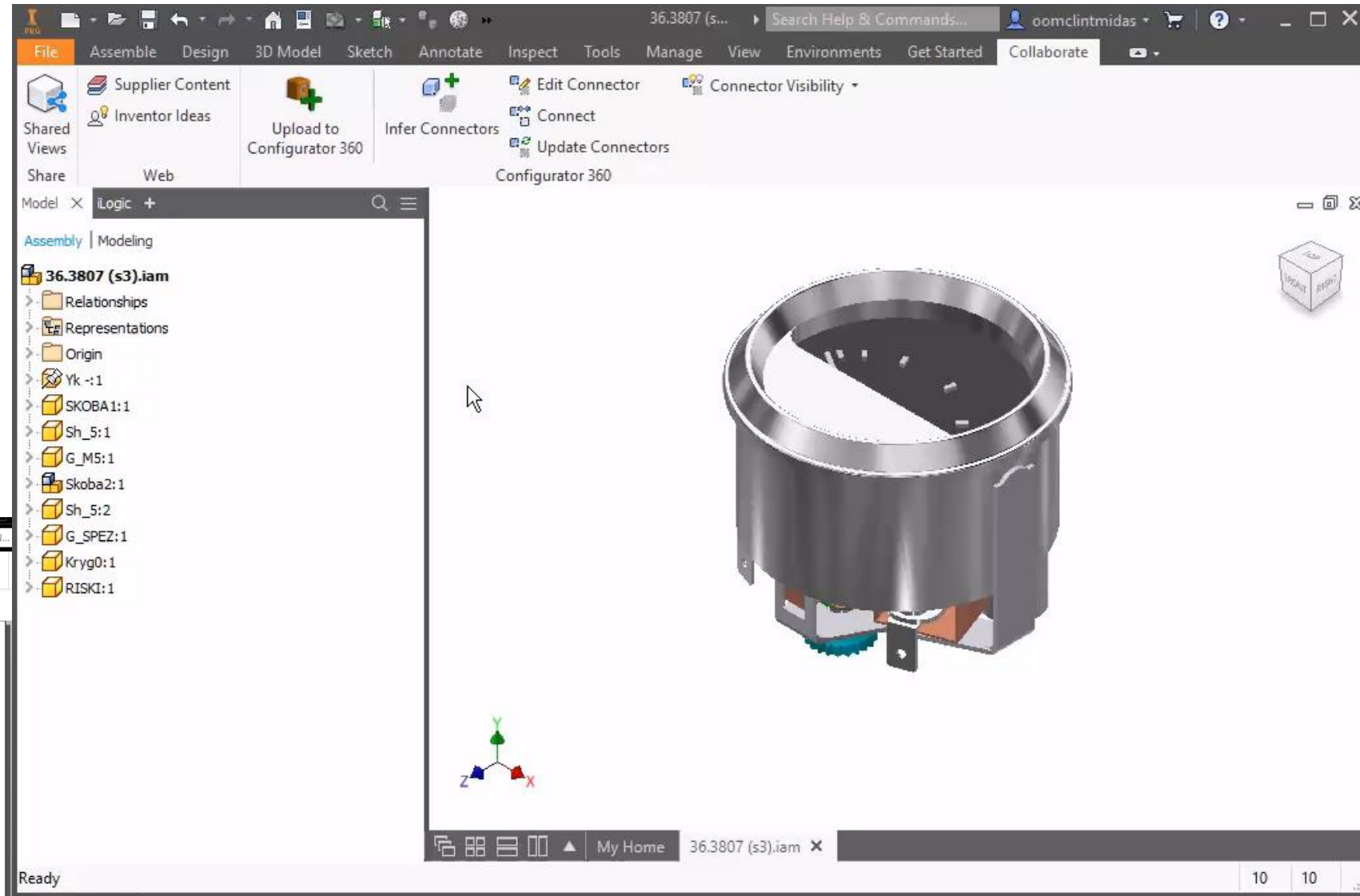
# Collaborate with Shared views (new in 2019)

Shared Views last 30 days (unless renewed)

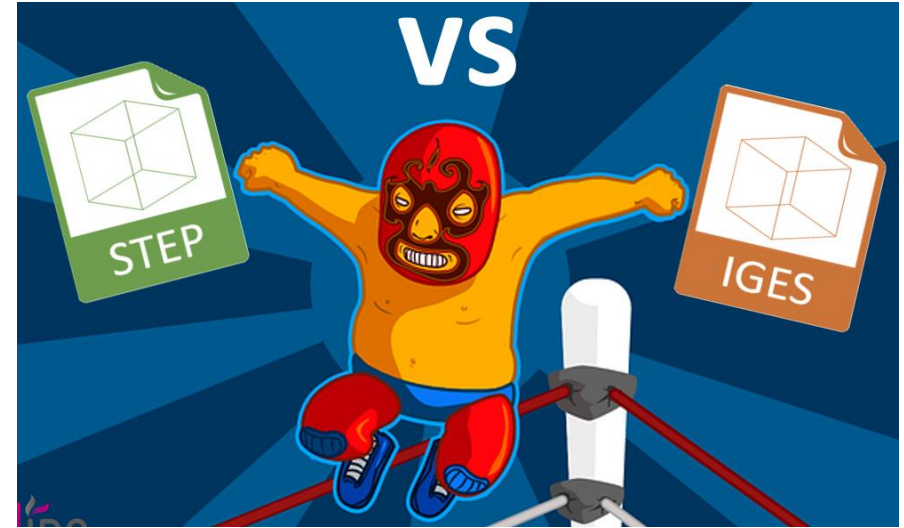
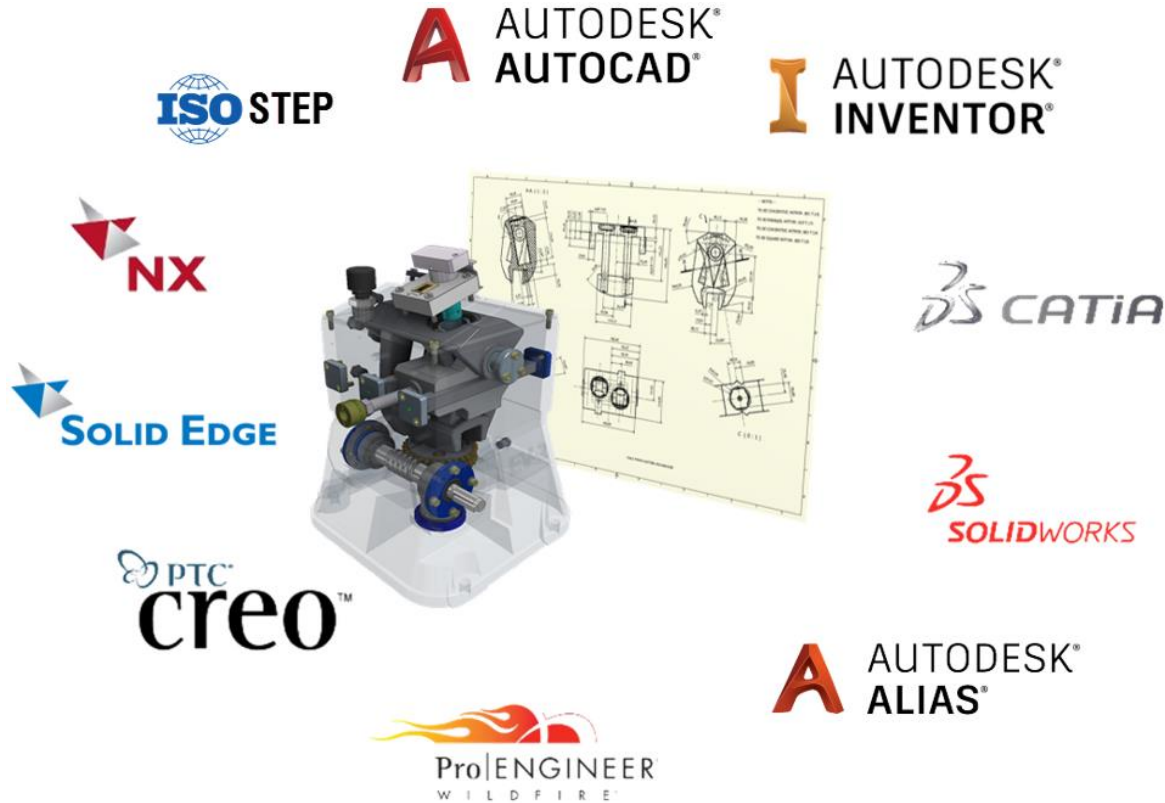
Allows collaboration over the web

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 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**

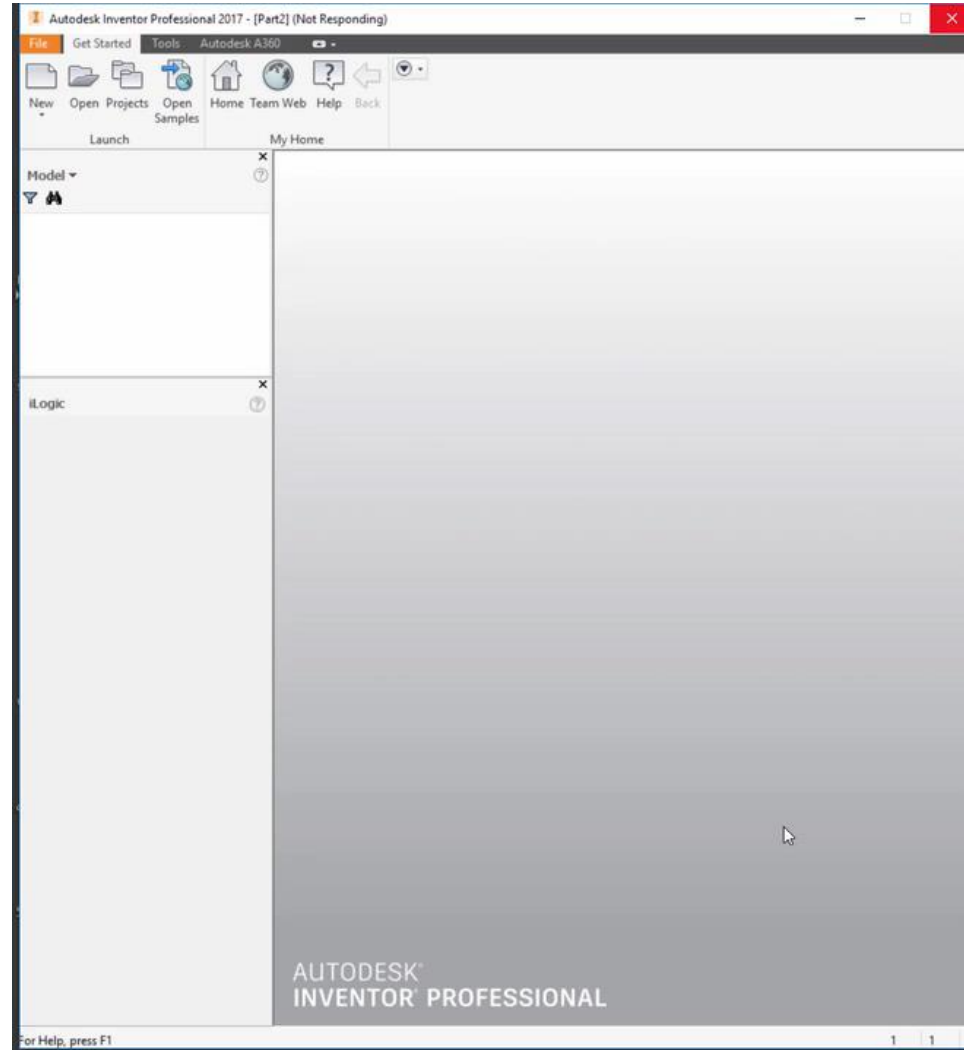
**STEP** has been available for over 20 years with continual development and wide adoption between CAD vendors



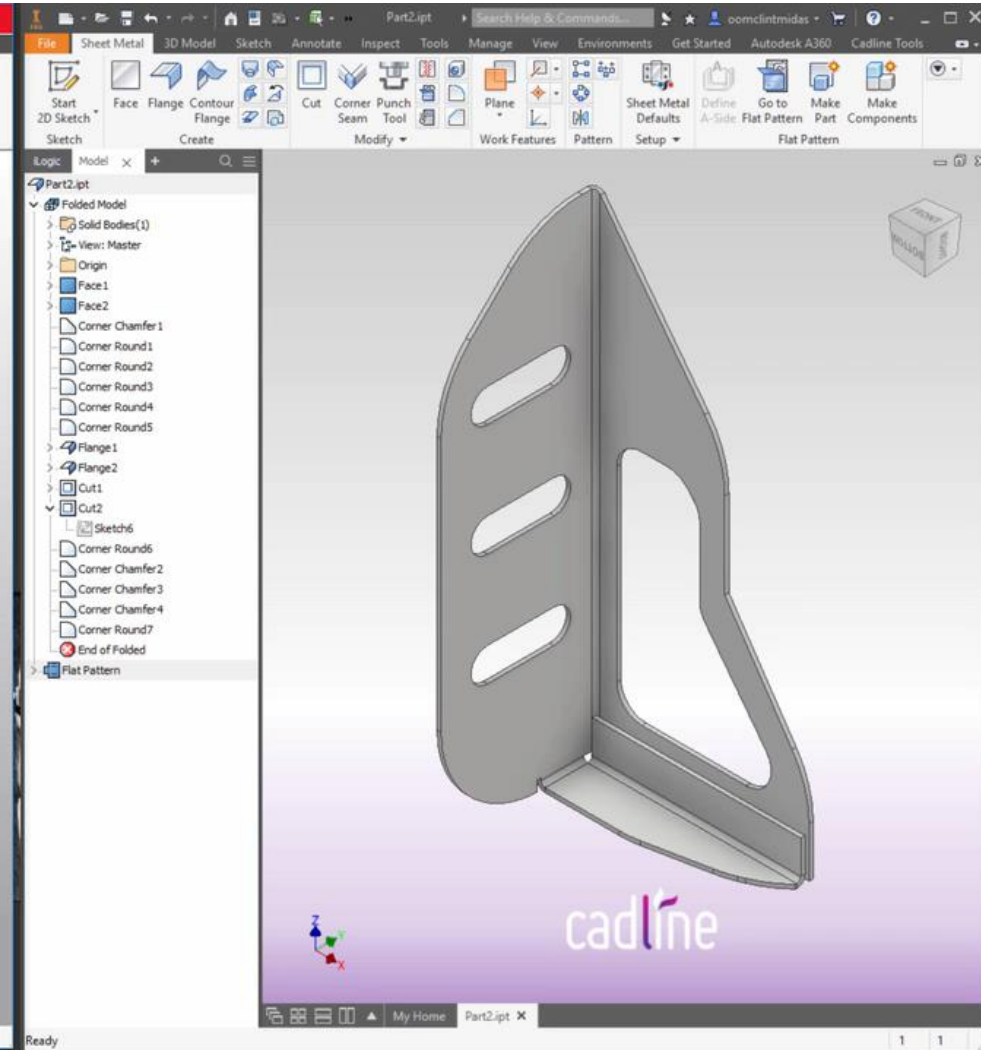
# Backward Compatibility

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



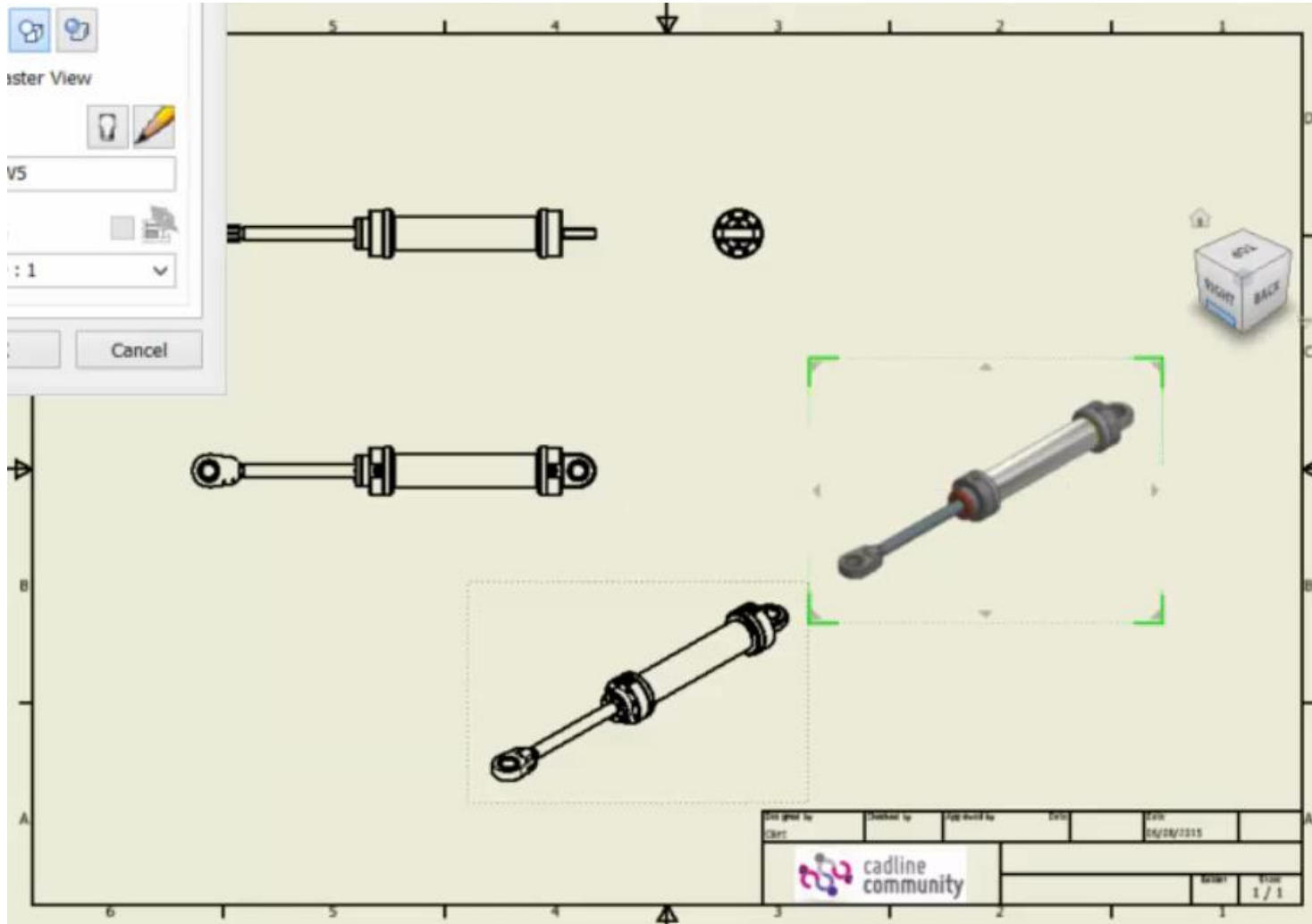
Inventor 2017.4



Inventor 2018



# Get some Perspective - Drawings

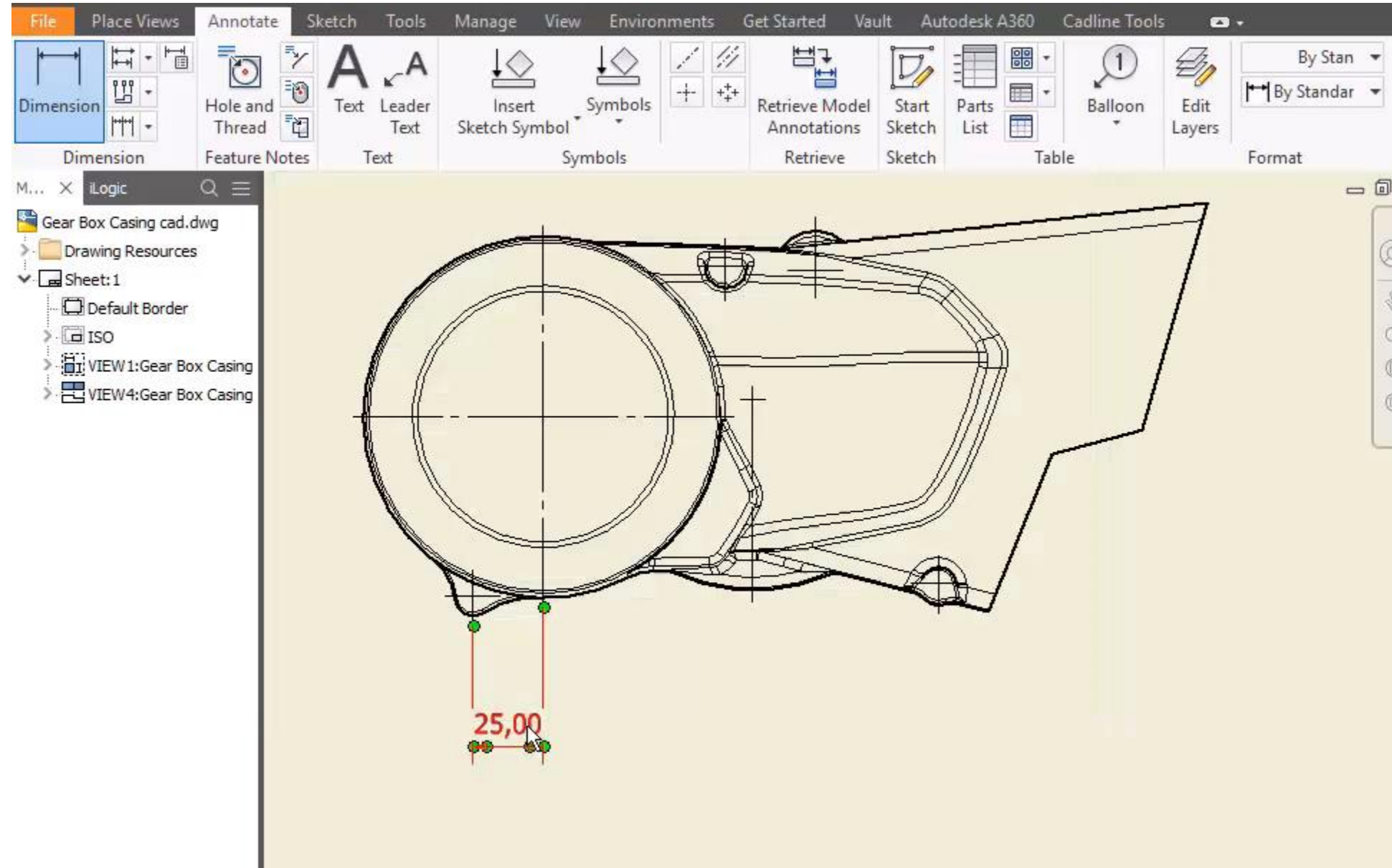


Since Inventor 2016, we can now use the "View Cube" to orientate our views, and as such, we can place Perspective views as opposed to Isometric views on our drawings, giving more realistic appearance to a "pictorial" view.

# Drawings: use AutoCAD for dimensions

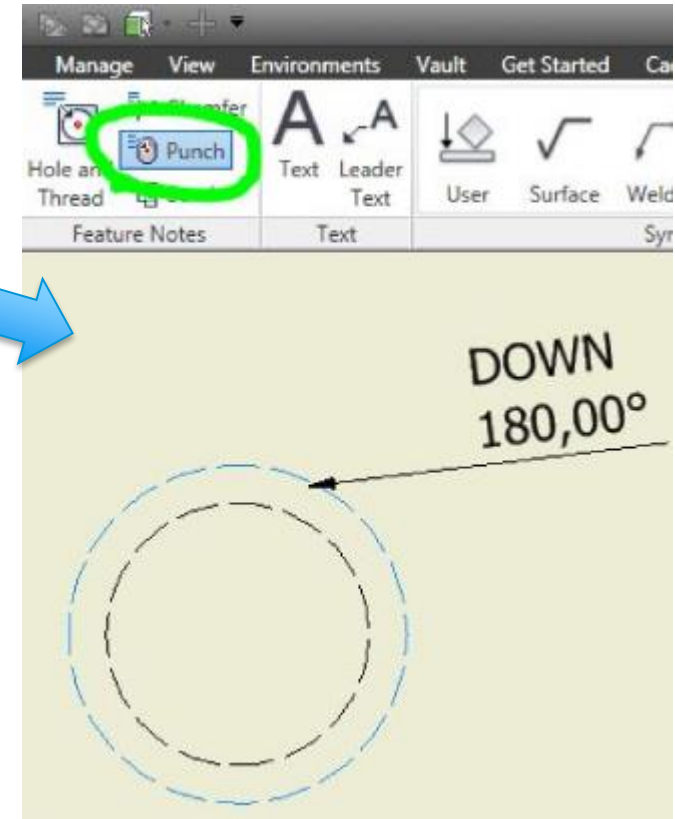
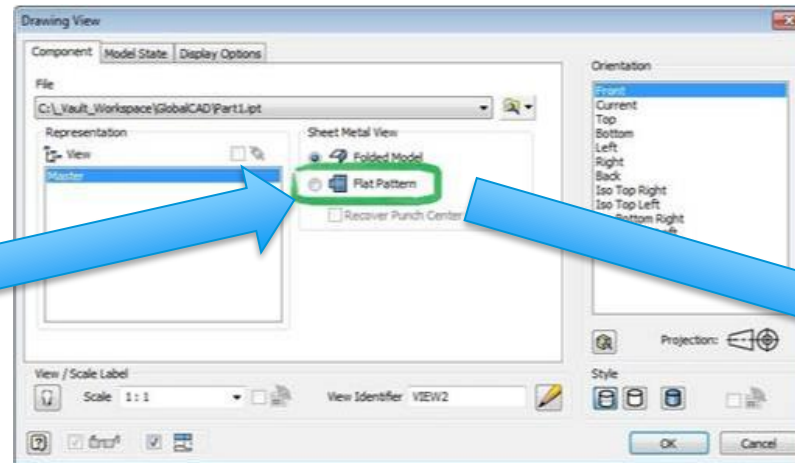
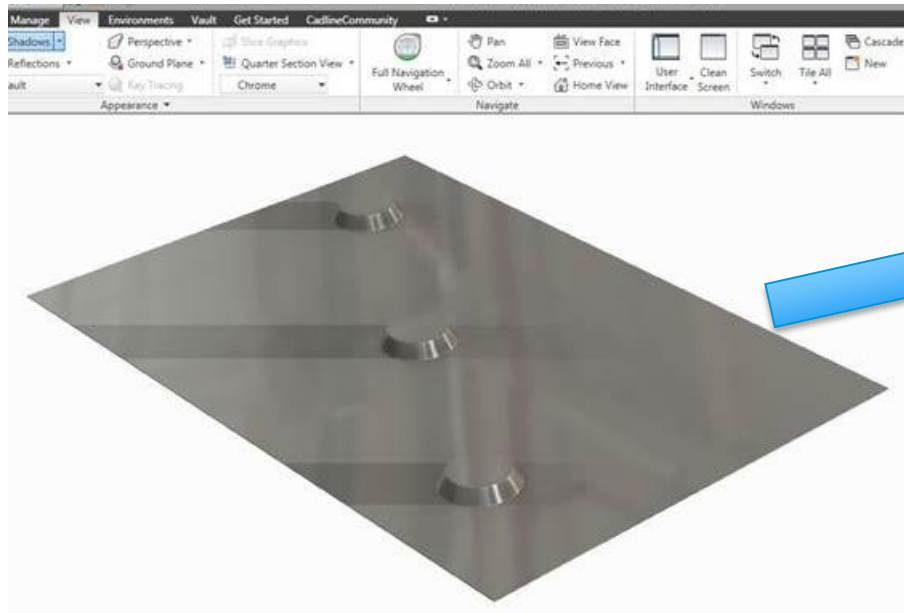
Did you know that you can dimension your Inventor DWG files in AutoCAD?

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



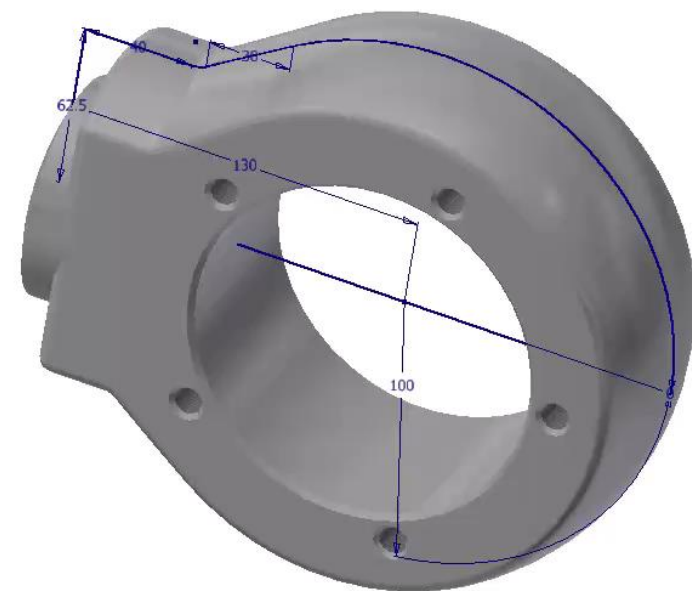
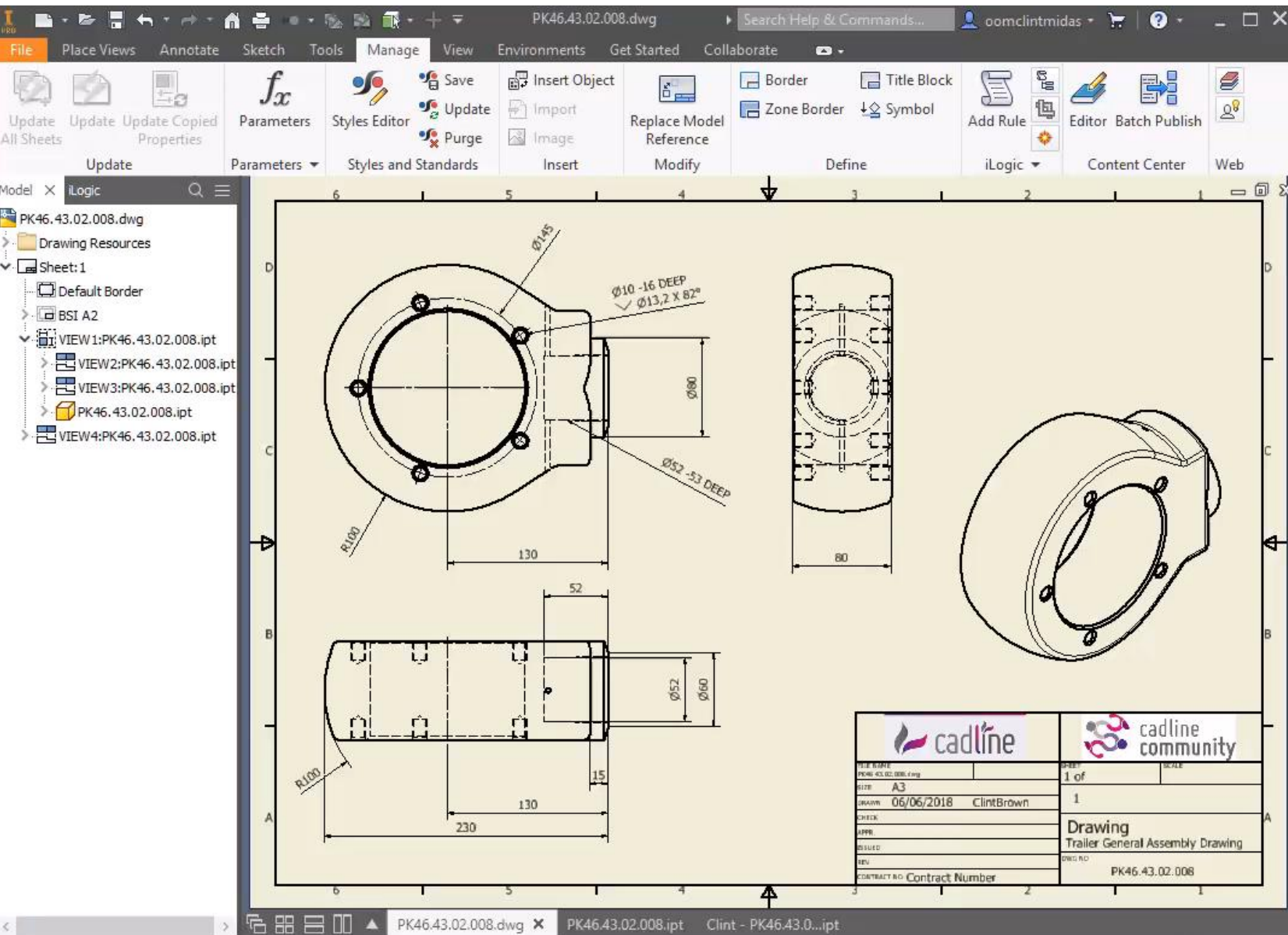


# Drawings: Replace Reference

Introduced in Inventor 2011

Allows you to re-use your existing drawings

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



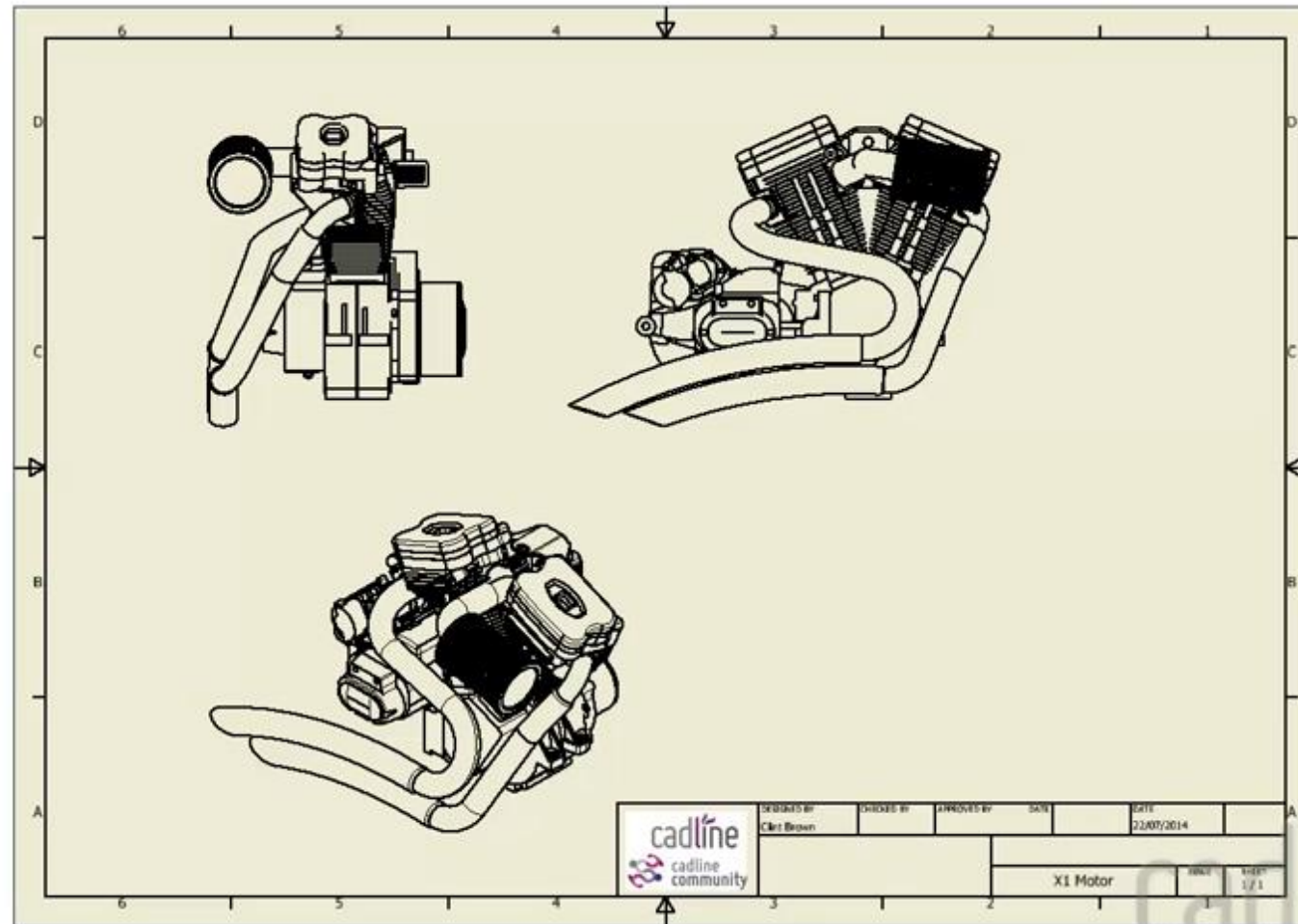


# Drawings: Sketchy lines

Revit & AutoCAD 2015 introduced “Sketchy Lines”

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

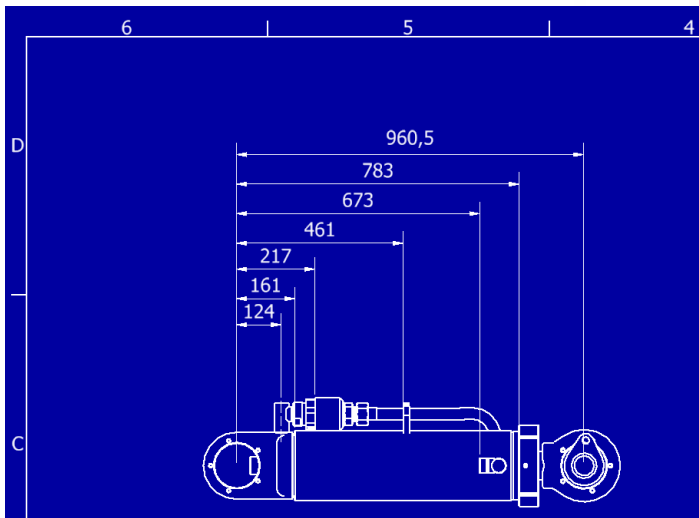
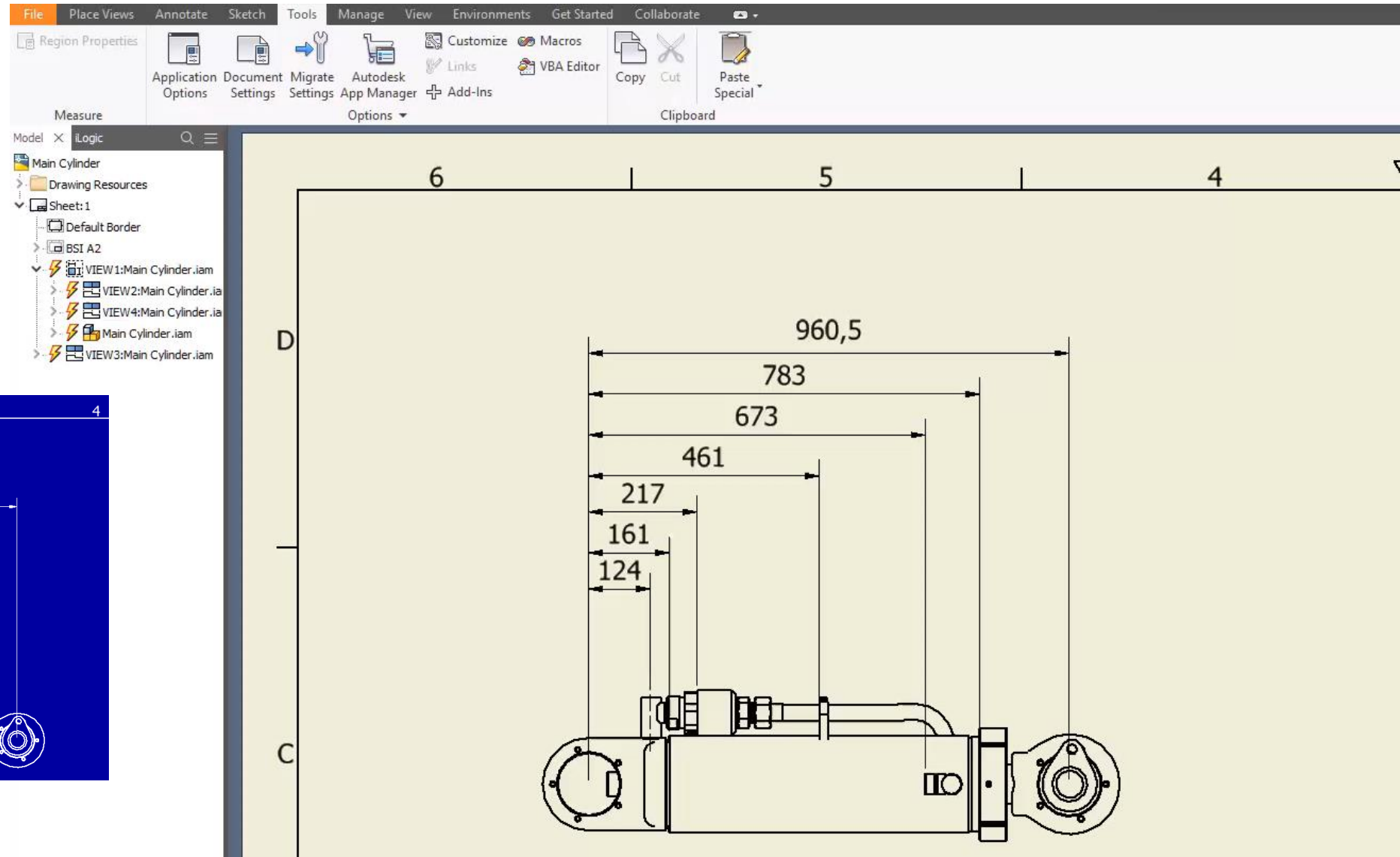
Export to PDF or Image – “Sketchy Lines” will not show in Inventor



# Drawings: Change your sheet colour

Tools > Document Settings

Note that this setting is file specific

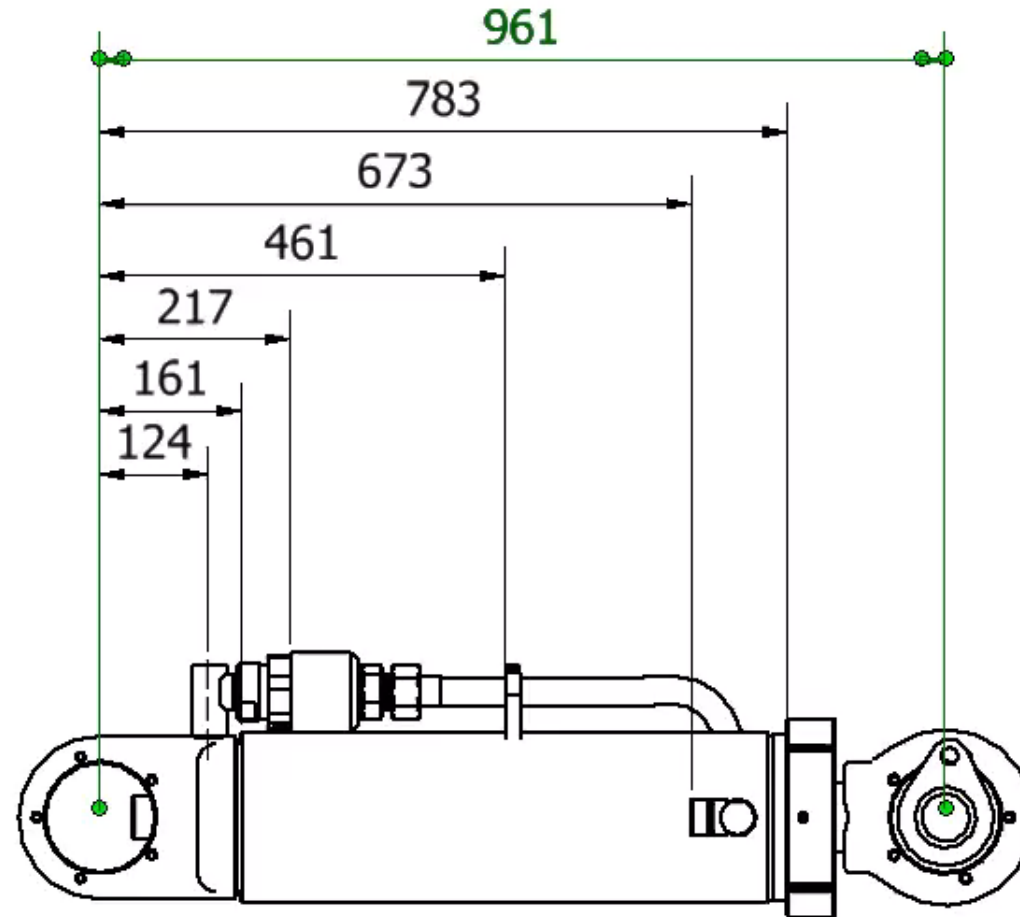
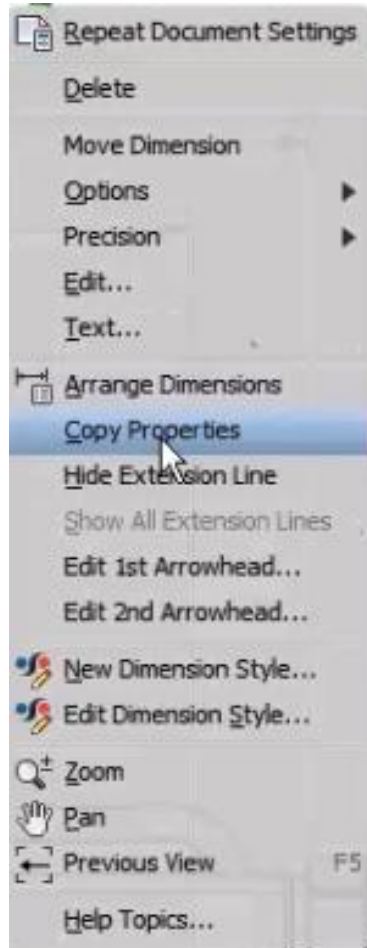


# Drawings: Precision & Copy Properties

Set Precision from a right click

Similar to “MATCHPROP” in AutoCAD

“Copy Properties” of one dimension to others



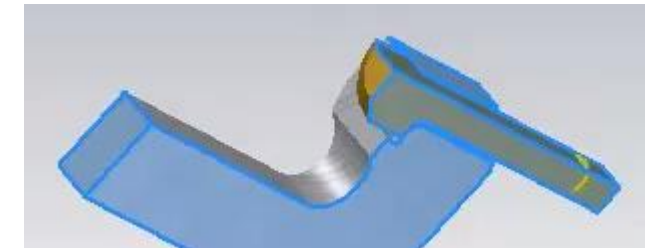
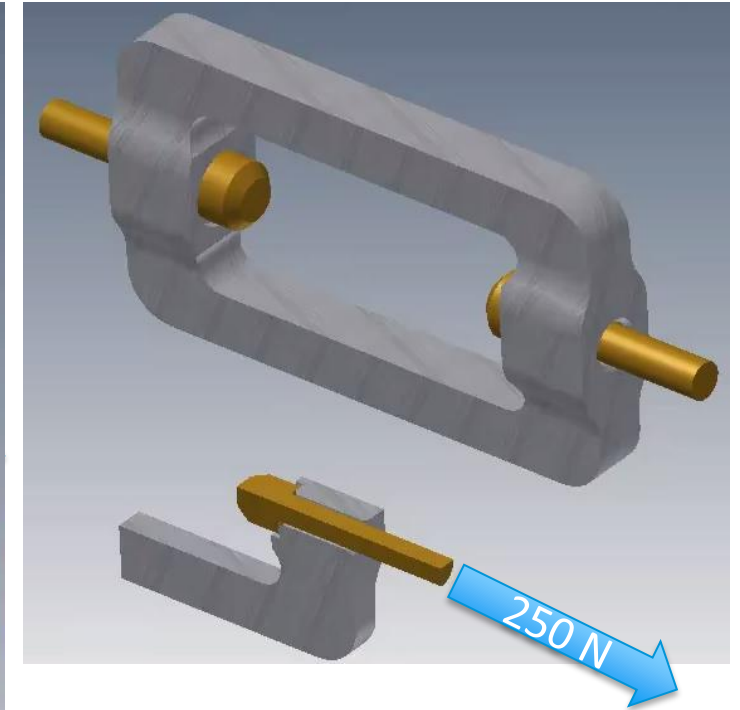
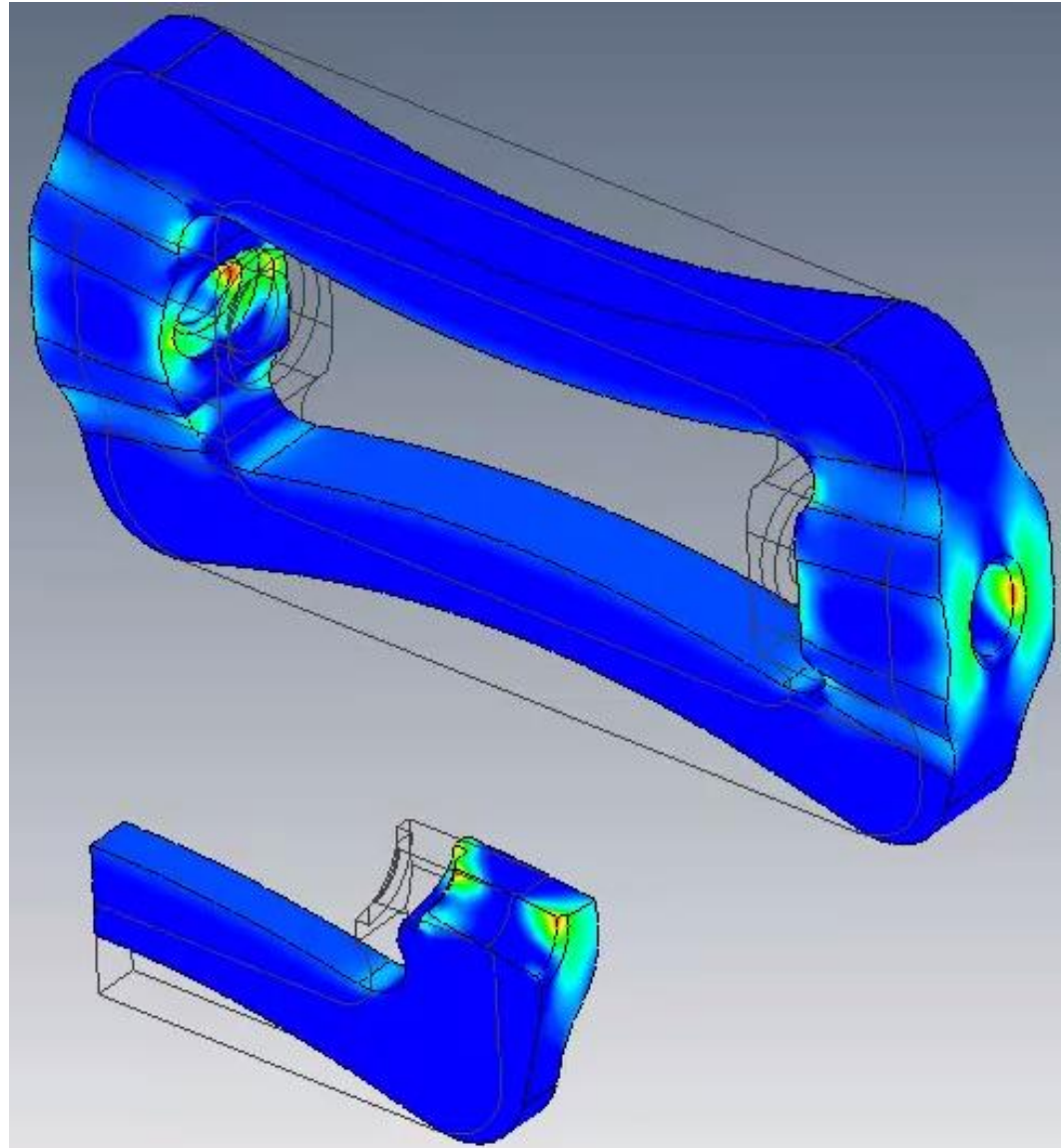
# Simulation: Symmetry

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

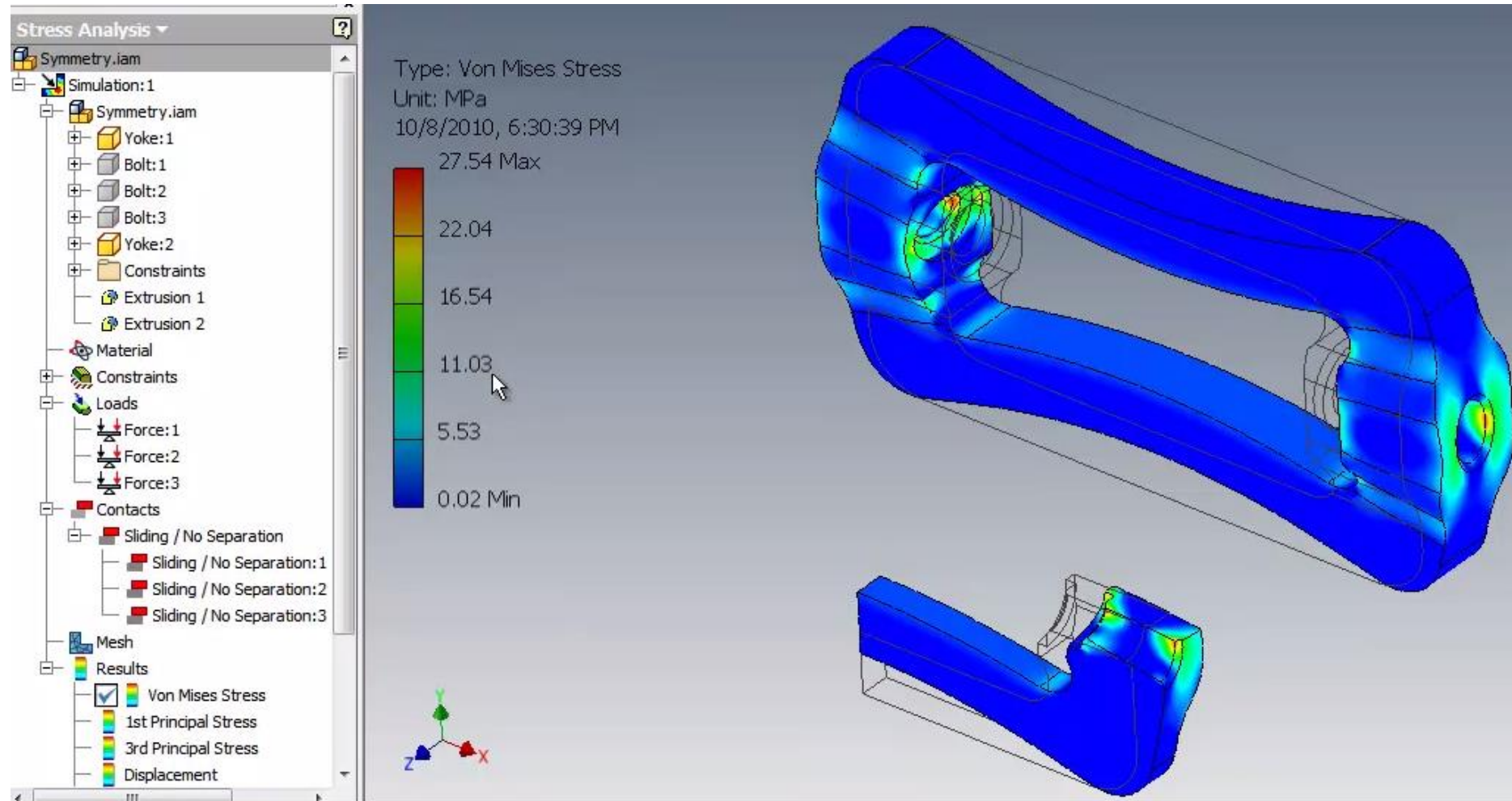


# Simulation: Component Visibility

Toggle visibility of Components:

Does not remove them from analysis

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

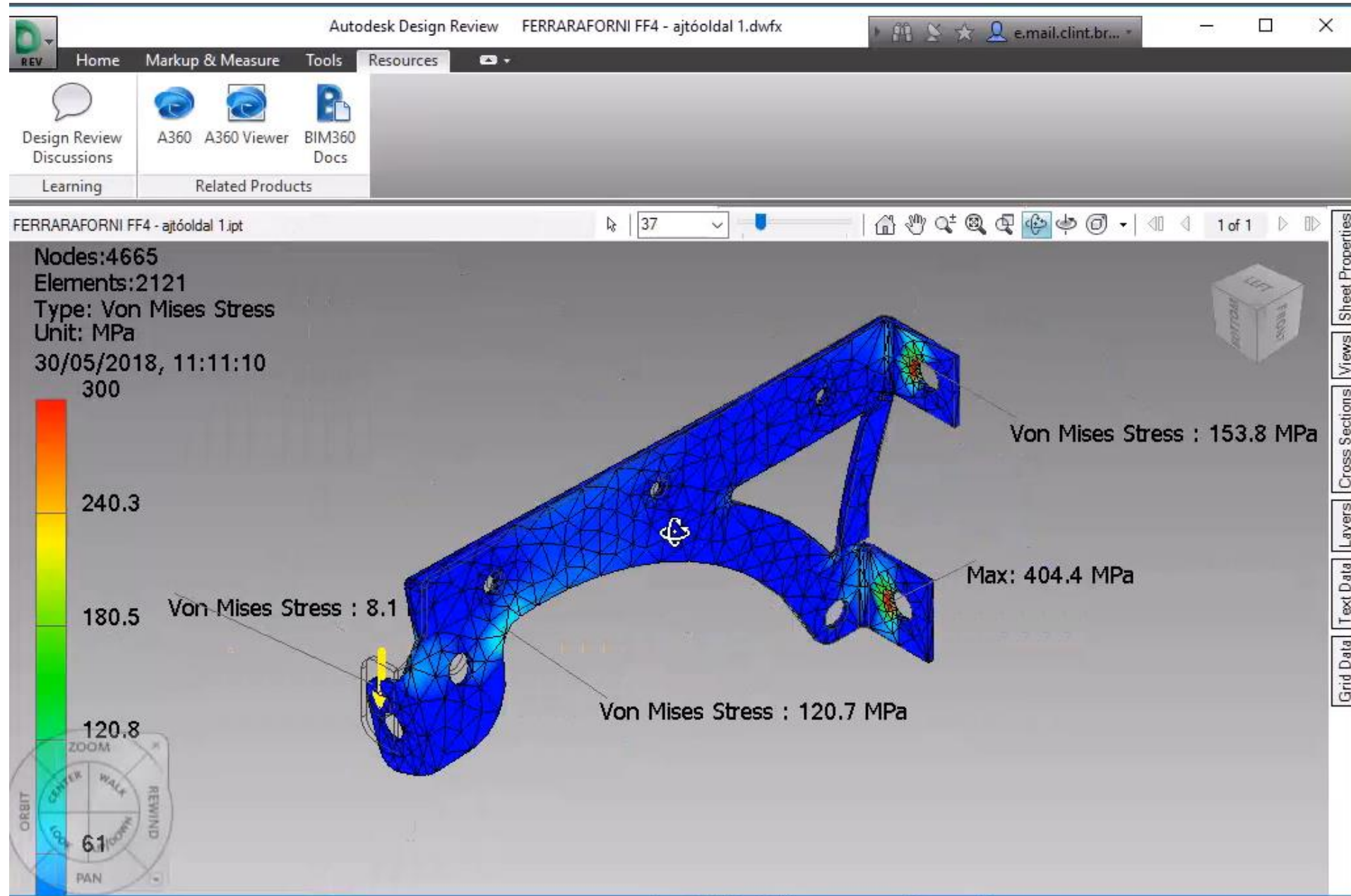




# Simulation: Share your results via DWF

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. The Parameters dialog box is open, displaying a table of model parameters. Below the table, there are mathematical equations:  $E = mc^2$ ,  $\nabla \times E = -\frac{\partial B}{\partial t}$ , and  $P + \rho \times \frac{1}{2} v^2 = C$ . The Open dialog box is also open, showing a file list with columns for Name, Date modified, and Type. The file 'etal.stp' is selected.

Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Ex	Comment
<b>Model Parameters</b>									
d2	Direct Edit1	mm	2 mm	2.000000	●	2.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d5	Direct Edit1	mm	2 mm	2.000000	●	2.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d6	Direct Edit1	mm	-6.750 mm	-6.750000	●	-6.750000	<input type="checkbox"/>	<input type="checkbox"/>	
d7	Direct Edit1	mm	-2.500 mm	-2.500000	●	-2.500000	<input type="checkbox"/>	<input type="checkbox"/>	
d8	Direct Edit1	mm	4.000 mm	4.000000	●	4.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d9	Direct Edit1	mm	-5.250 mm	-5.250000	●	-5.250000	<input type="checkbox"/>	<input type="checkbox"/>	
<b>User Parameters</b>									

# Customise the Navigation Bar

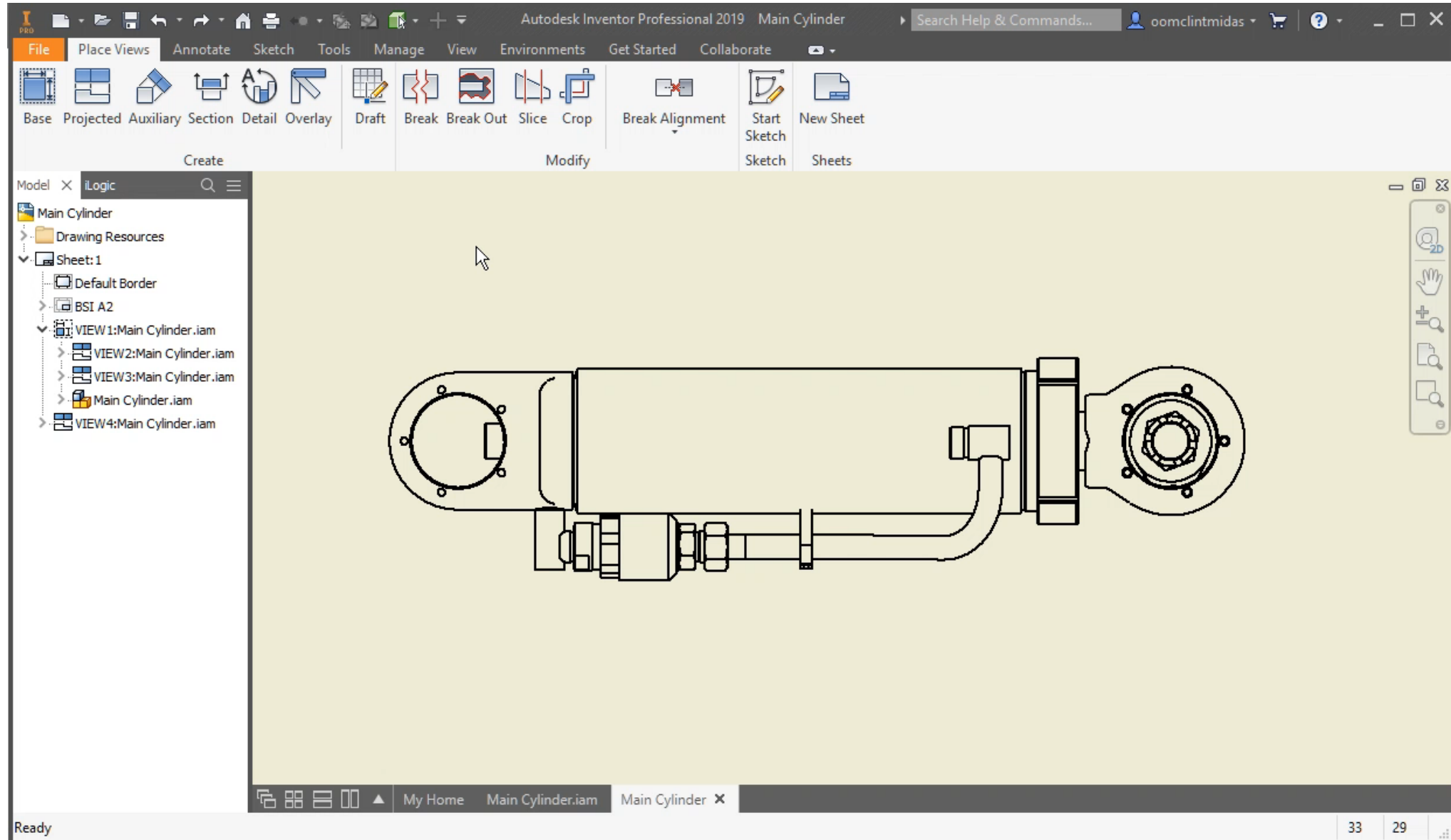
Add the navigation tools that you use most commonly to the Navigation toolbar

- ✓ ViewCube
- SteeringWheels
- ✓ Pan
- ✓ Zoom
- ✓ Orbit
- ✓ Look At PGUP
- ✓ Zoom All HOME
- ✓ Zoom Window Z
- Zoom Selected END
- Previous View F5
- Next View Shift+F5
- Projection
- Visual Styles
- Lighting Styles
- Ground Plane
- Docking positions >
- Inactive opacity >
- Active opacity >



# Drawings: Set Transparency

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

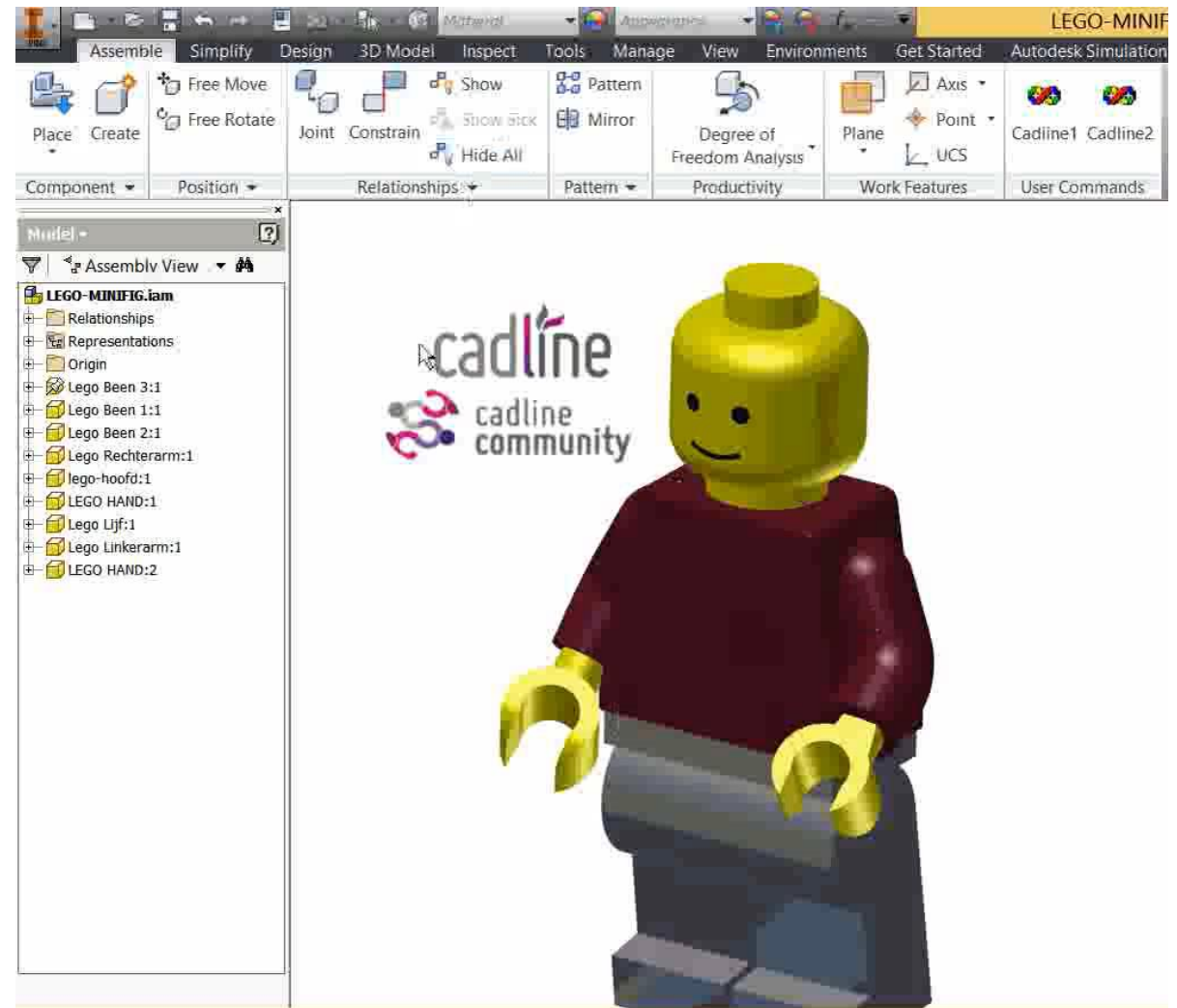


# Assemblies: Degrees of Freedom Analysis

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

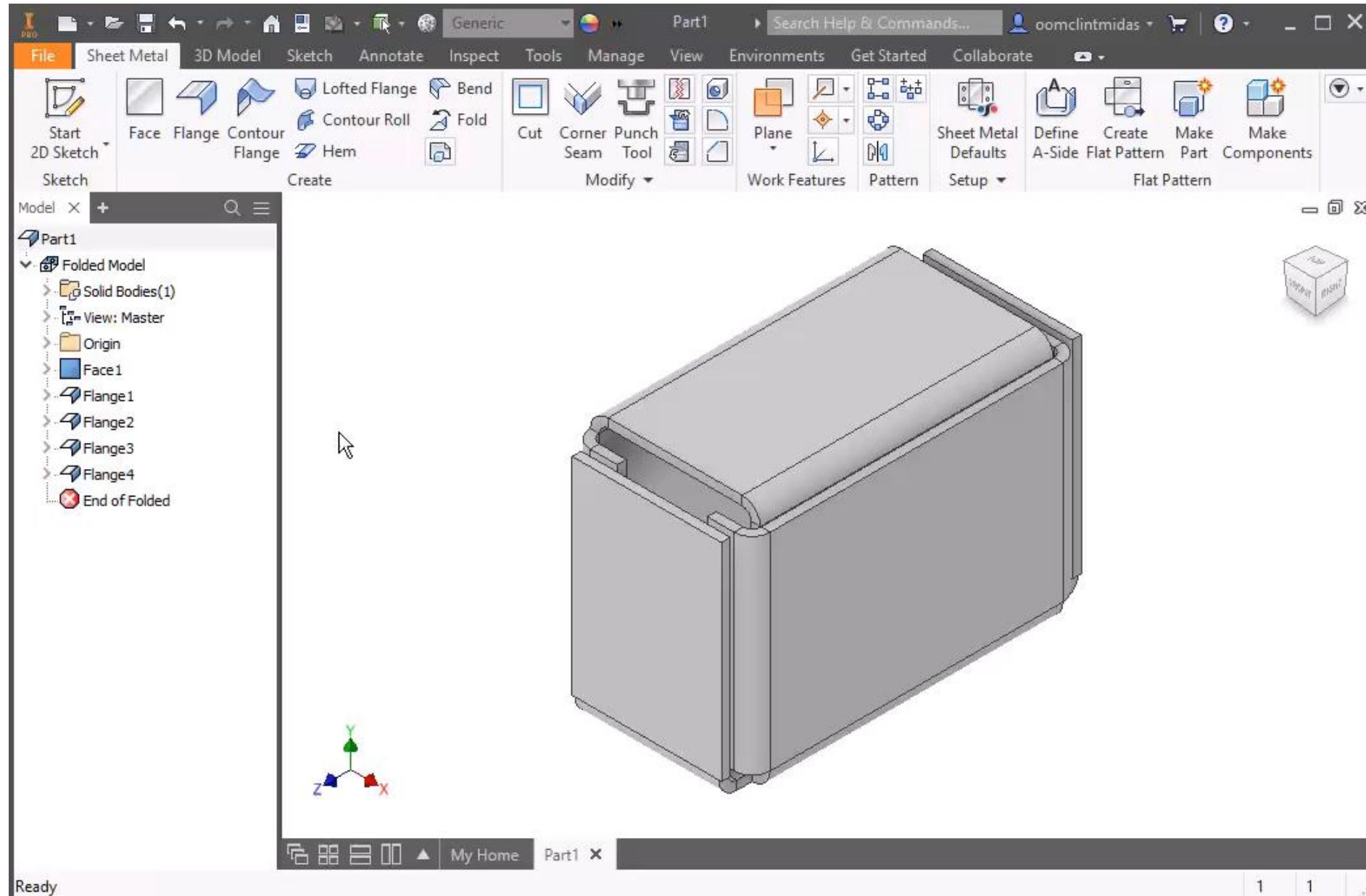




# Code Free iLogic

## 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 you 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"

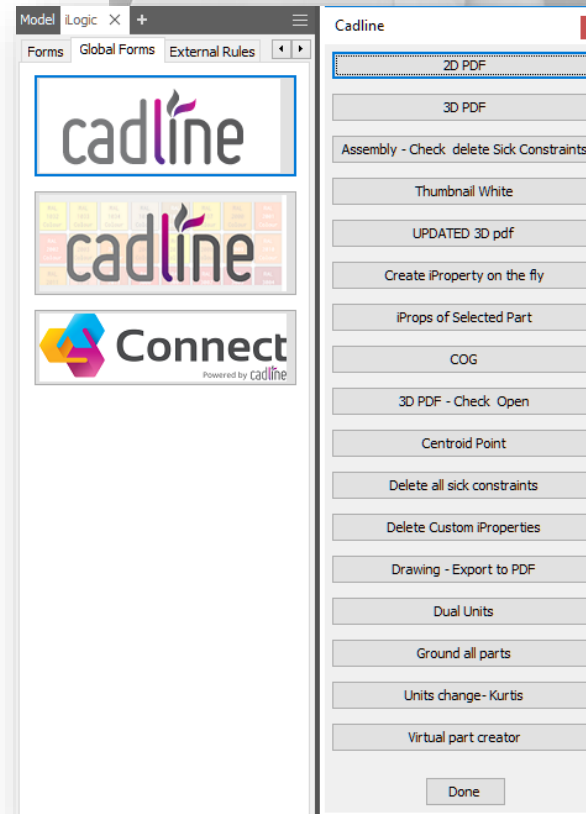
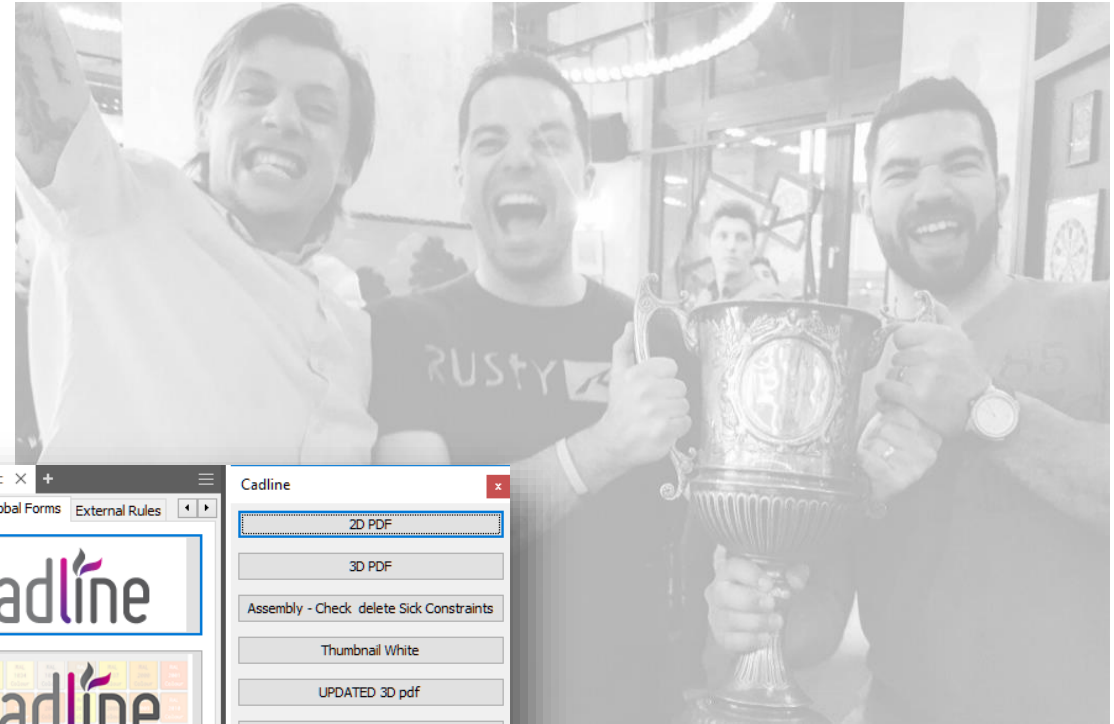
2 disadvantages – 1: The rules don't run on parameter changes  
2: The rules are not embedded in the files if sharing externally

## 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

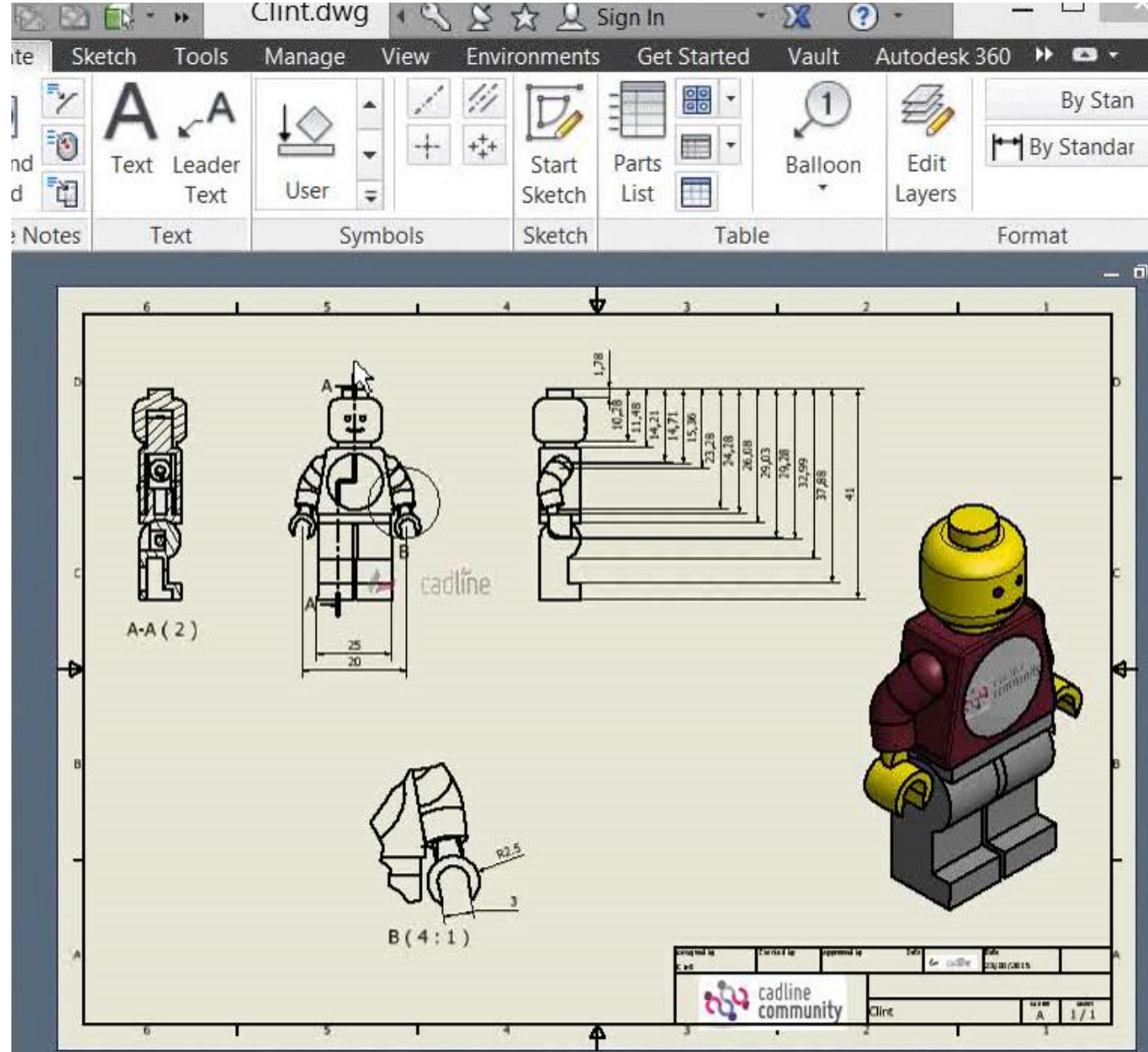
Choose **“Select All Overridden Dimensions”** tool from the selection set dropdown

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

Delete the overridden value and then **un-tick** the **“Hide Dimension Value”** box.

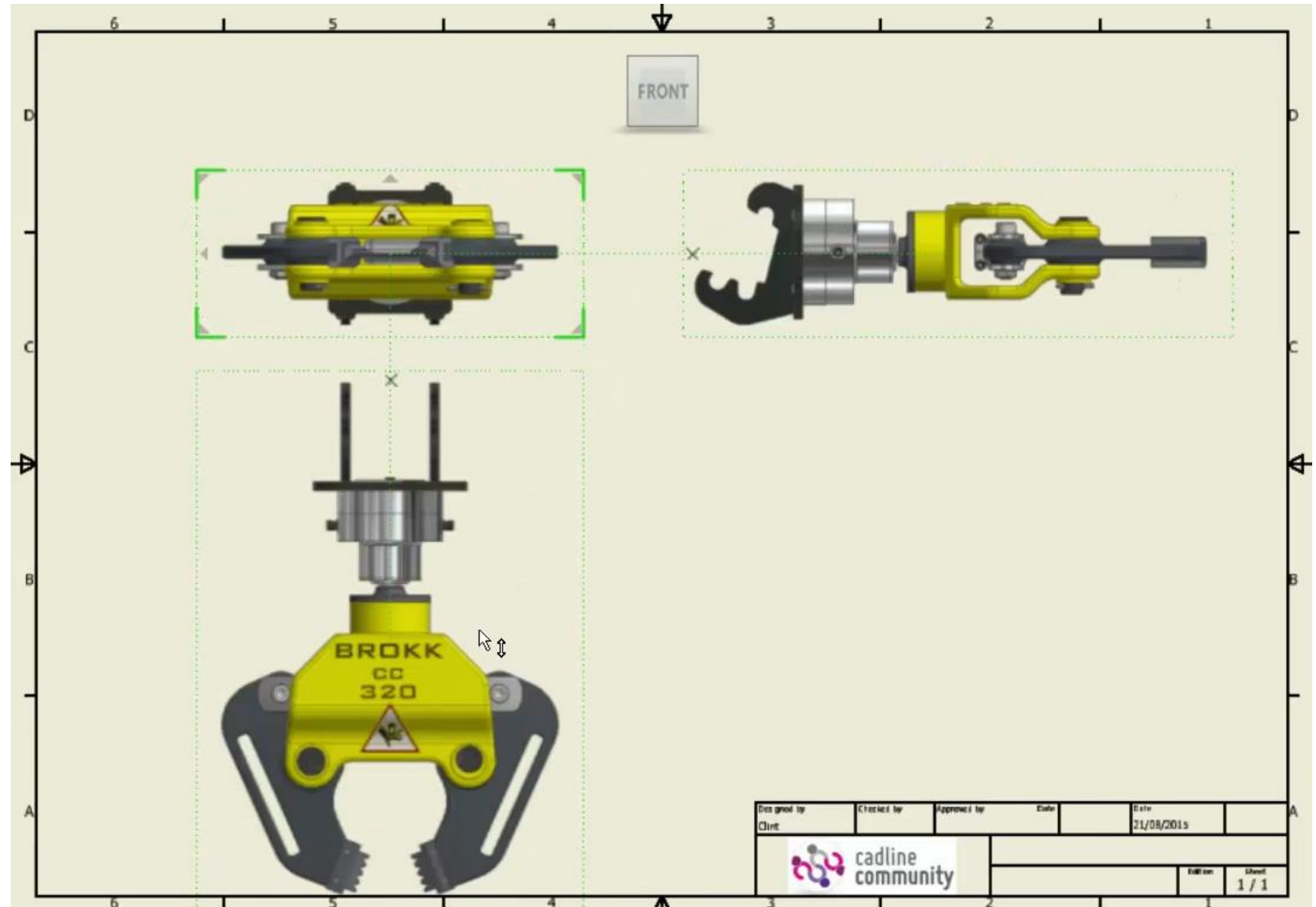
To cycle through all of the dimensions, Right Click and select **“Repeat Select All Overridden Dimensions”** then **Right Click** and choose **“Edit”**

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



# Drawings: View Scales

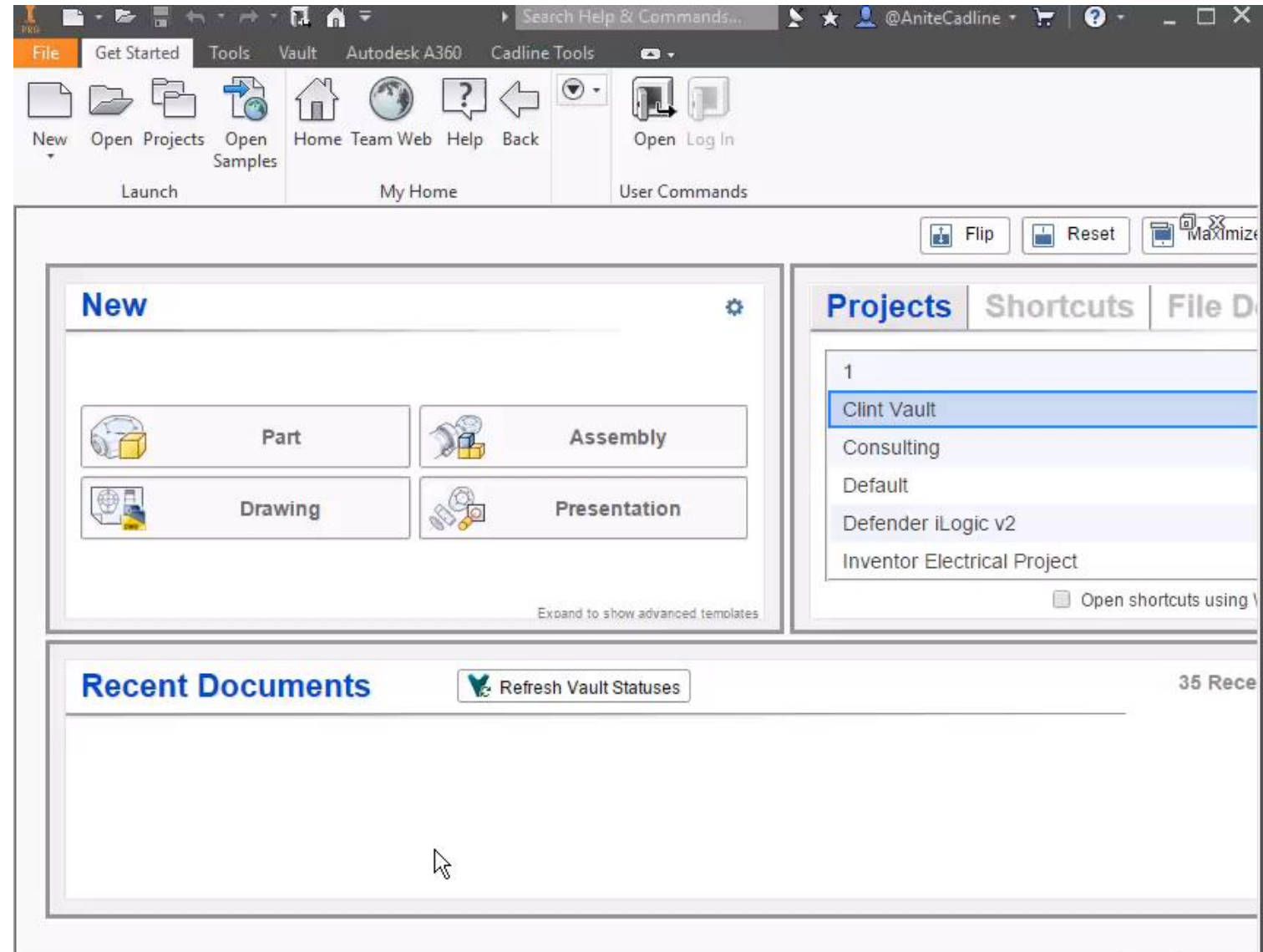
Set the scale of the views on the drawing by simply dragging the corner of the Base view. (introduced in 2016)





# 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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