AUTODESK UNIVERSITY 2013

Beyond Design and into Manufacturing with Autodesk® Design Suites – Class Supplement

Scott Moyse - CADPRO Systems
Gavin Bath – CADPRO Systems

MA2274

Preparing your designs for manufacture isn't just about knocking out a 2D drawing and hitting print. There's a ton of metadata which can be created and capitalized on to improve the flow of information and the components themselves through the manufacturing process. These days, automated manufacturing with computer numerical control (CNC) is commonplace; therefore, it is necessary to have an efficient process for creating CNC cut files or models. In this class, we cover several ways to prepare and export data to CAM software ready for CNC machining. Which one is best for your company or process? Exporting that manufacturing data from Autodesk® Inventor® software is only part of the process. Learn how you can deliver that information to those within manufacturing and procurement. The class also features iPropWiz V6, an amazing Inventor add-in that not only supercharges iProperties, but also makes it easier to create CNC cut files.

LEARNING OBJECTIVES

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

- Use iPropWiz to supercharge your component properties and manufacturing data
- List the options for creating 2D CNC cut files
- Deliver manufacturing information and CAM information to those who need it
- List the options for creating 3D CNC cut files

ABOUT THE SPEAKER

Scott Moyse is a proud Cornish man & a MFG Applications Engineer for CADPRO Systems in New Zealand. He previously worked at SMI for over 9 years after moving from the UK while studying Motorsport Engineering. He started out as design support & quickly moved into programming their new CNC machine. Over the next 4 years he worked closely with both manufacturing & design to create & implement automated processes. This provided him with an invaluable insight into both departments operations. 4 years ago he moved back into design full-time, resulting in him taking the Design Manager position 3 years ago. Over the last 36 months he's implemented & managed Autodesk Vault Professional, improving communication, work allocation, organization & increased control over the design review process. Although he has no prior 'PLM' experience he has been deeply involved in process formation, implementation & development in an ever changing environment. He also regularly contributes on designandmotion.net

scott.m@cadpro.co.nz

INTRODUCTION

This hand-out has been written as a supplement to the class itself, rather than a script of the class. Some of the information may be covered in more or less detail to suit the hour timeframe. To start off with we will discuss iPropWiz, a 3rd party tool you can use to prepare manufacturing information associated with your designs. This will provide an understanding of how you can configure your Inventor design environment to benefit you in the long run. From there we will start to cover the various ways of producing CNC cut files or tool paths within Autodesk Inventor. Along the way we will slowly introduce the concept of using Autodesk Vault to manage your design data, then ultimately your CAM and manufacturing data, as well as some ways of presenting that to CAM programmers and staff on the shop floor.

IPROPWIZ – LEVERAGING YOUR MANUFACTURING INFORMATION

Most of the seasoned Autodesk Inventor users out there will have heard of iPropWiz. For those who haven't, or for those new to the game, iPropWiz is an Inventor Add-in created to simplify and ease the management of component properties. iPropWiz presents all of the properties you are interested in, within a single view. Over the years it's matured, moving way beyond just presenting properties to the user.

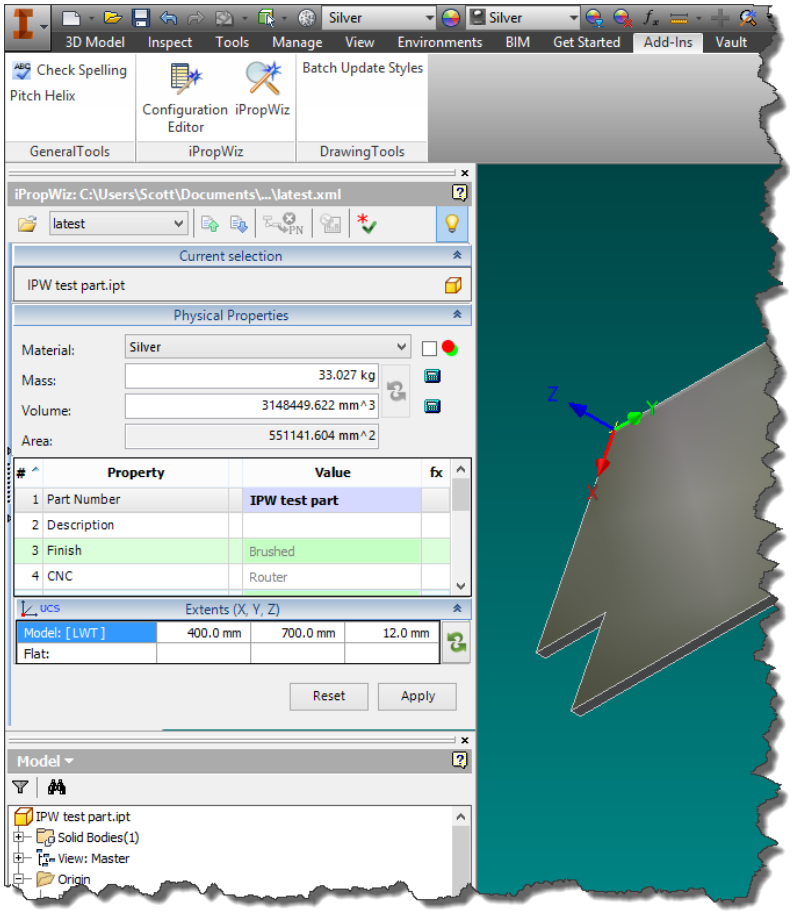


Figure 1: iPropWiz's non-modal dockable window

The iPropWiz property editor is a non-modal & dockable window, which means you can continue to work while it's open. Being able to keep it on a second monitor means you are constantly aware of, and working with the properties fundamental to the design & ultimately the manufacturing processes.

CONFIGURATION

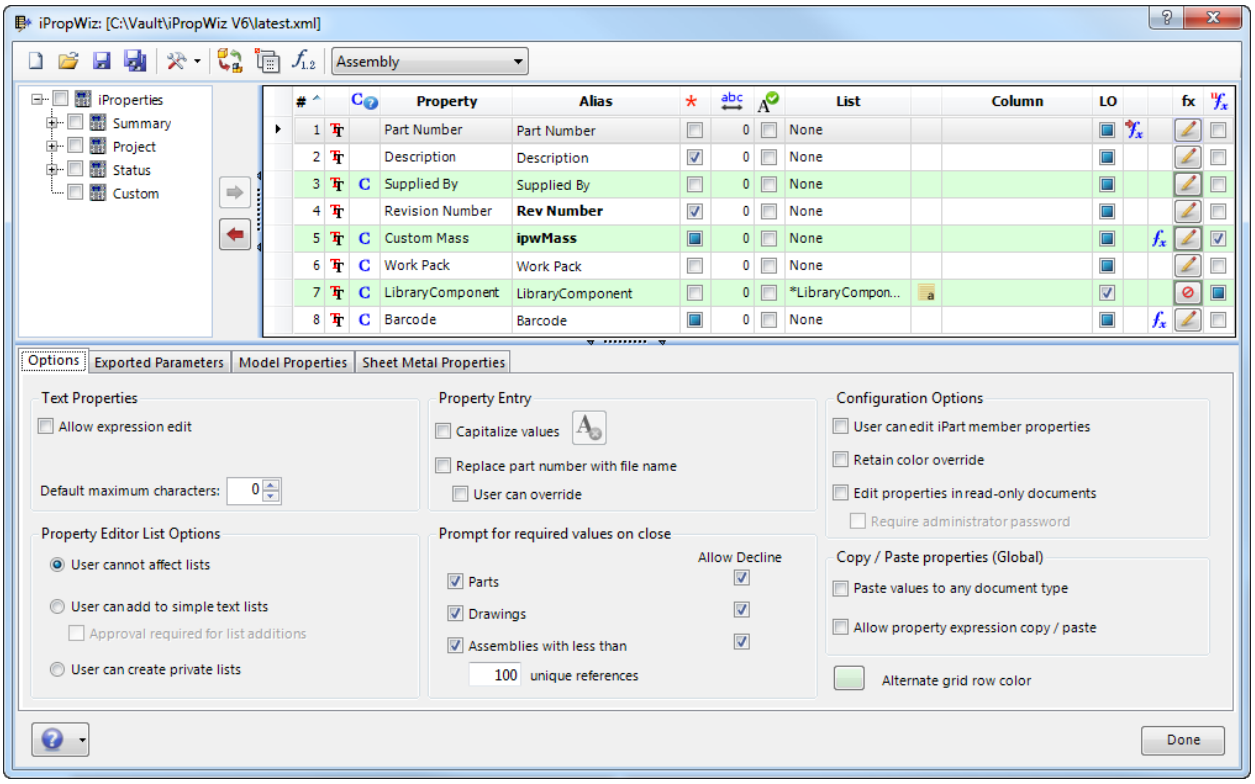


Figure 2: iPropWiz V6's Configuration Editor

Within the password protected administration dialogue box, you can set up formatting rules for physical properties, specify required properties, and manage lists & expressions. In addition you can configure iPropwiz to check property compliance in real time, or upon closing the files. All this will allow you to create a more stable environment for iLogic rules, Addins or Autodesk Vault Professional to guide the users work in the correct direction for the benefit of CAM & manufacturing.

LISTS & EXPRESSIONS

The new list types are some of the most powerful new features in iPropWiz V6 (Figure 3). The Multi-Property, Material & Parameter based multi-property lists allow the administrator to monitor the value of a targeted property. When that property matches the criteria set by the list rule, it takes control of other preset properties. This can either be in the form of enforcing a value, or chopping & changing lists assigned to these controlled properties. Using material as a good example, steel has a different set of finishing requirements than plastic. These tools allow you to present those options to the user. Very cool stuff! For the purpose of this class we will

use lists to create fixed values which will drive the categorization of parts, as they are checked into Vault.

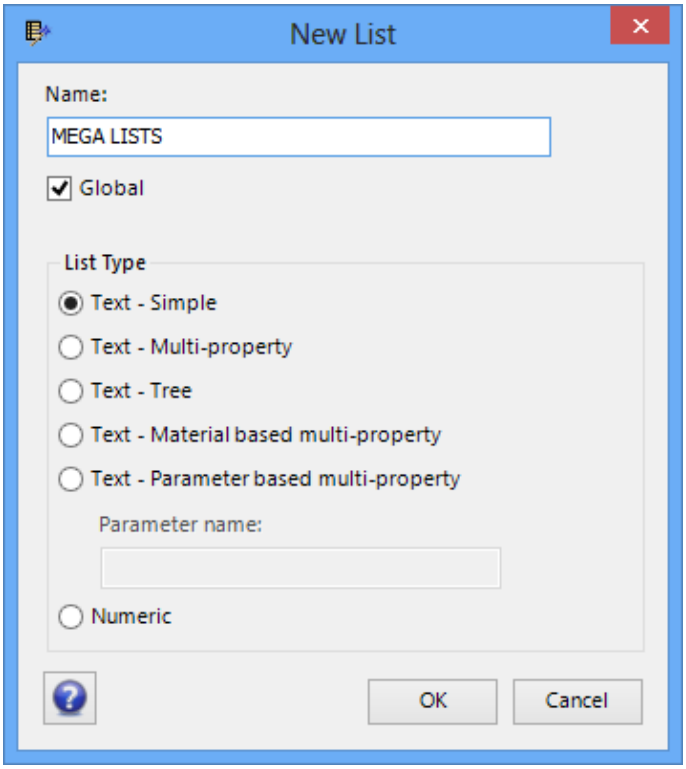


Figure 3: iPropWiz V6 sports clever new list types

MODEL PROPERTIES TAB

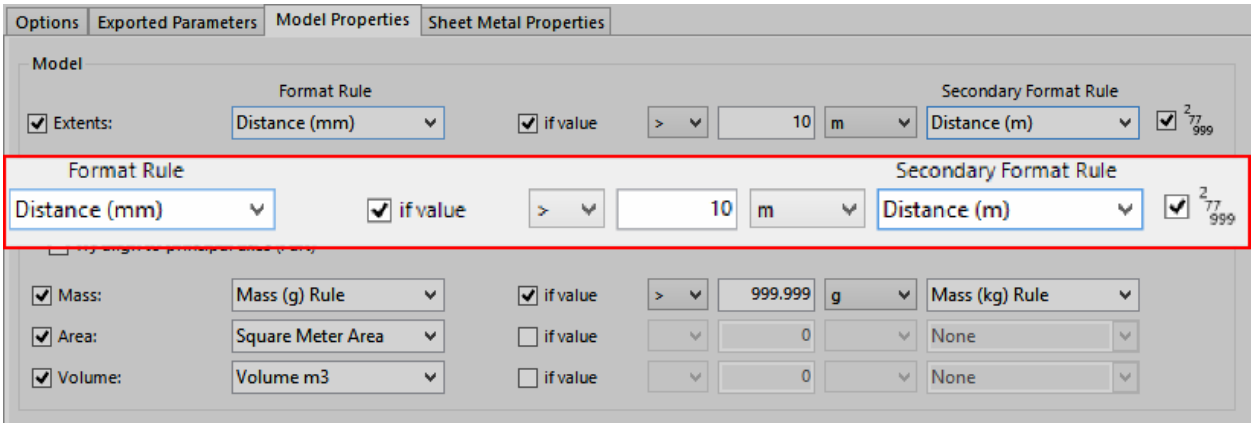


Figure 4: The Model Properties tab provides conditional formatting of physical properties

Notice the 2,77,999 tick box just to the right of the Secondary Format Rule dropdown list in Figure 4. That little sucker enables sorting of the 3 Extents properties from the smallest to the largest. This means you can use property expressions to map Length to the largest value & Thickness to the smallest… or whatever your standard may be. For instance, if you don't select

that option, you can permanently tie the Length property to the Extent in the X direction. In the woodworking industry some companies would want to allow the Inventor user to align the Length property with the grain direction of the part.

As soon as the 'Extents' tick box is enabled in iPropWiz Model Properties tab, it will create the following 6 custom iProperties:

Ordered Extents

- ipwModelExtents1
- ipwModelExtents2
- ipwModelExtents3

Aligned Extents

- ipwModelExtentsX
- ipwModelExtentsY
- ipwModelExtentsZ

SHEET METAL PROPERTIES TAB

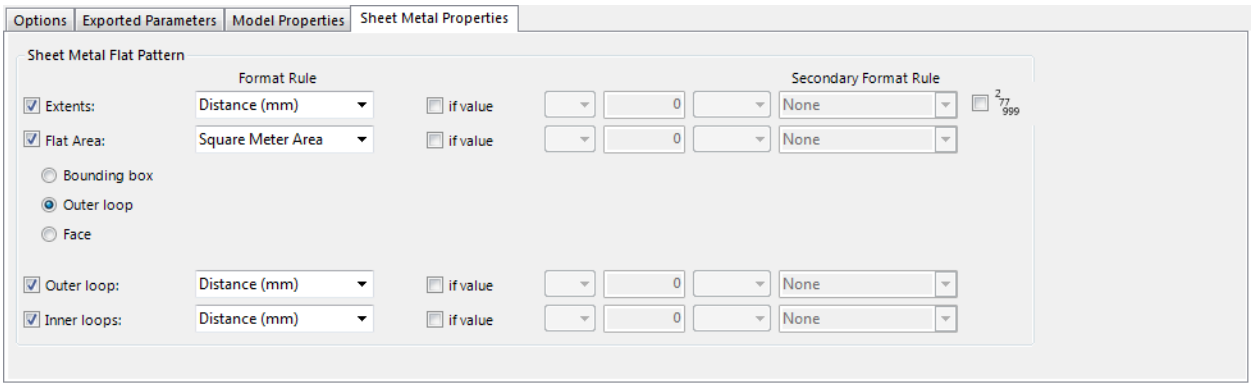


Figure 5: There are a number of tools for managing Sheet Metal properties

iPropwiz provides 3 options for exported flat pattern area, by default Inventor only provides one of these:

- Bounding box
- Outer loop area
- Face area.

The ability to automatically extract inner & outer loop distances can be utilized for pricing & planning CNC machine run times & consumable usage.

USER DEFINED UCS & COMPONENT EXTENTS

iPropWiz allows the user to place a User Defined UCS onto a part, then use it to calculate the extents of the entire component relative to that coordinate system.

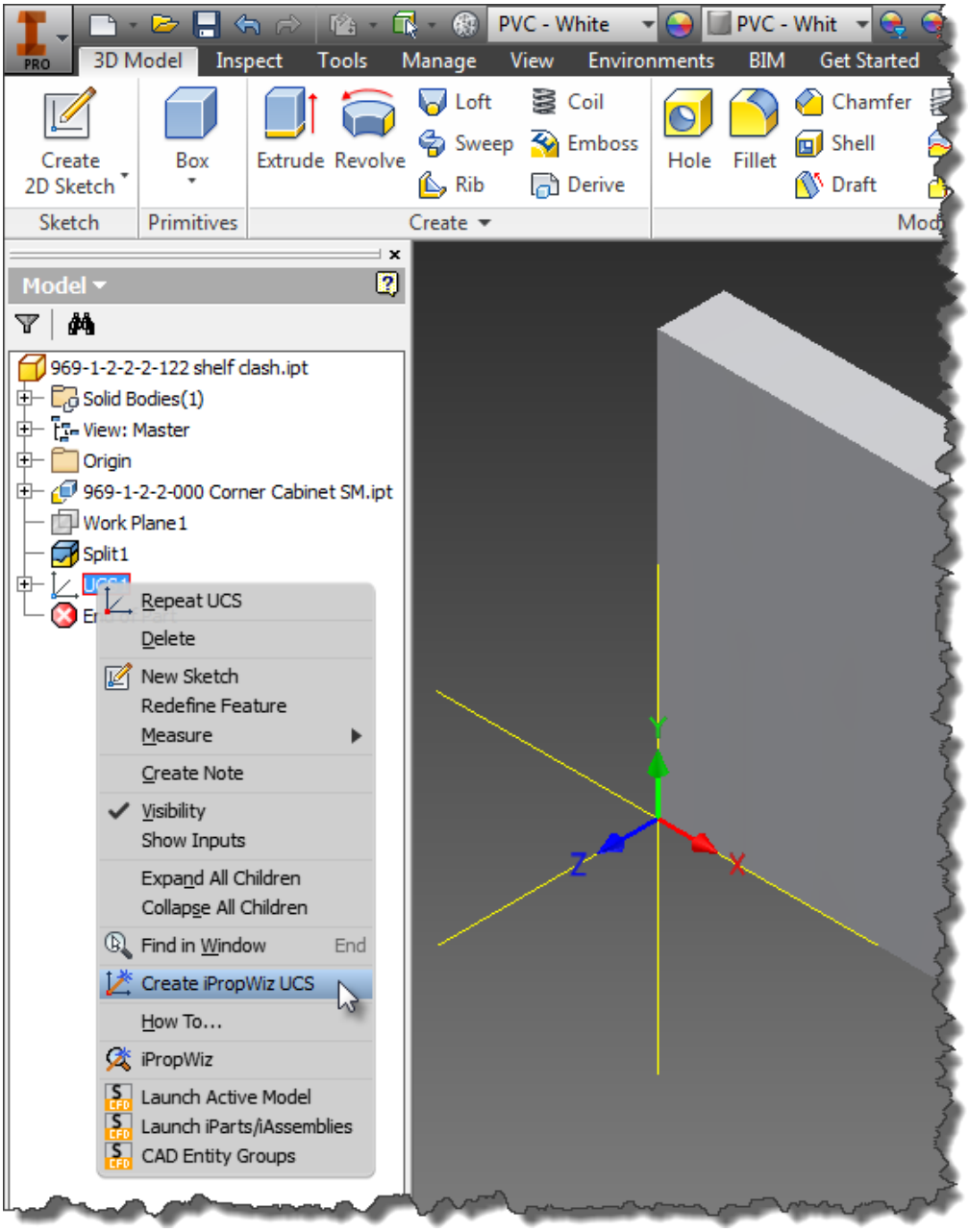


Figure 6: Create an iPropWiz UCS to calculate the extents of a component which isn't aligned to the WCS

With this in mind, you can now go back and take advantage of the Model Extents features I mentioned earlier. The result can be seen in Figure 7 below:

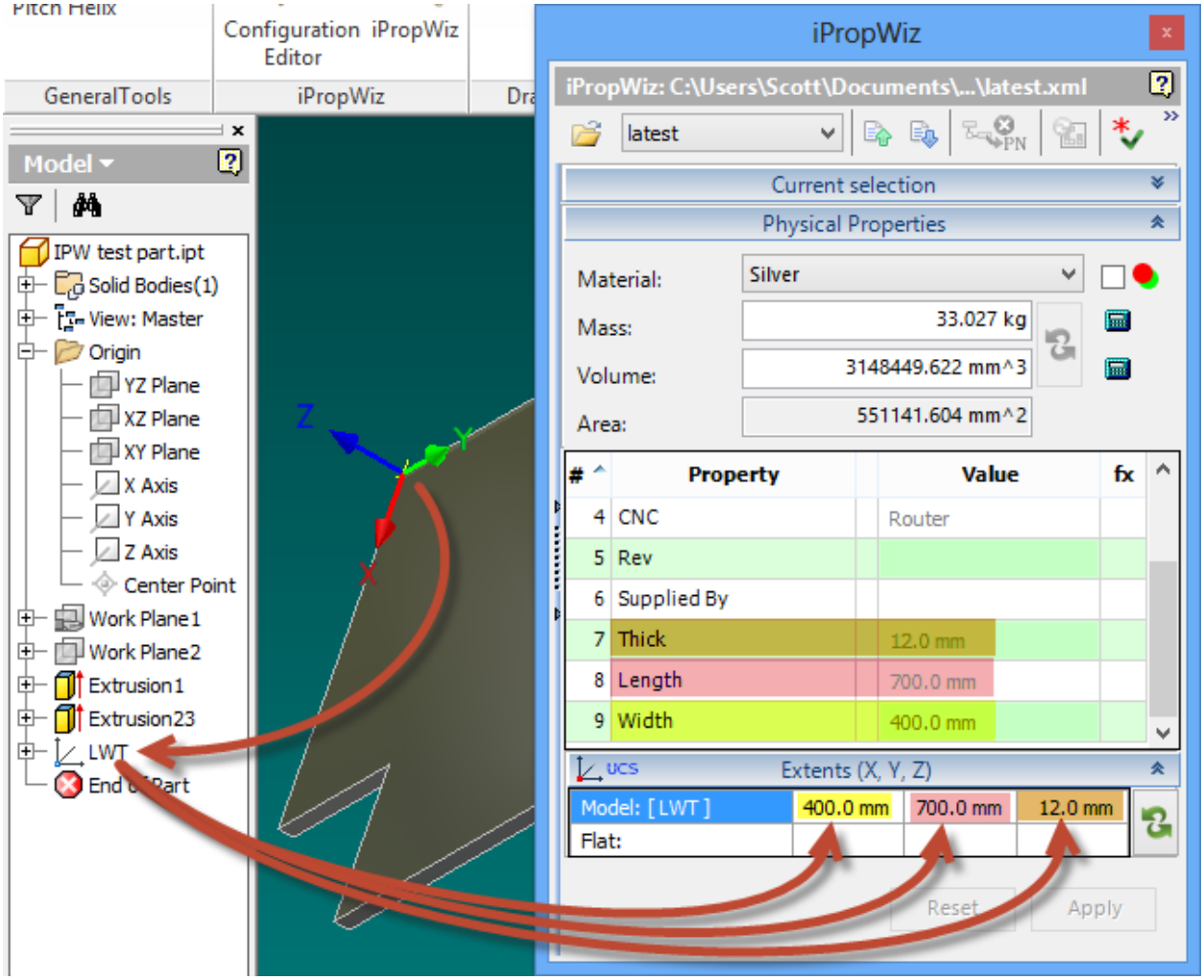


Figure 7: Using an IPW User Defined UCS enables you to pre-populate essential BOM information automatically.

The next advantage of placing a User Defined UCS is that it suddenly gives you an index for placing and orienting parts for CNC toolpathing & machining. For instance; the X axis can be aligned with any grain/pattern direction you may need to consider & the XY plane could be the bed of your CNC Machine or Clamp. Or maybe you have a saw blade add-on for your machine, but its axis is locked. With this you could ensure the appropriate axis is aligned with the modelled groove which represents the saw cut.

Another way to take advantage of this information, would be to create a little iLogic rule in your template document. That could constantly keep an eye on the values, then inform the user when the part is too large to manufacture on any machinery at your disposal or maybe it exceeds the available stock size?

2D CNC CUT FILES

I know of 5 distinct ways to get what CNC programmers need out of Inventor. The best method for you depends entirely on what you are cutting and the software you are using to program it. Depending on your situation you may be able to produce the perfect output for your needs. In the past I've had to resort to some fairly lengthy procedures to get what was needed. With building or Yacht interiors, the difficulty comes in dealing with the grain direction of the veneers or laminates on parts and subsequent grain matching. As a result I have investigated lots of options and discovered a few tricks along the way.

FLAT PATTERN EXPORT

As you may well know, in an Inventor Sheet-metal part you can create flat patterns. Once a flat pattern has been created, you can save out a dwg/dxf to AutoCAD which will allow you to develop the cut file further if need be.

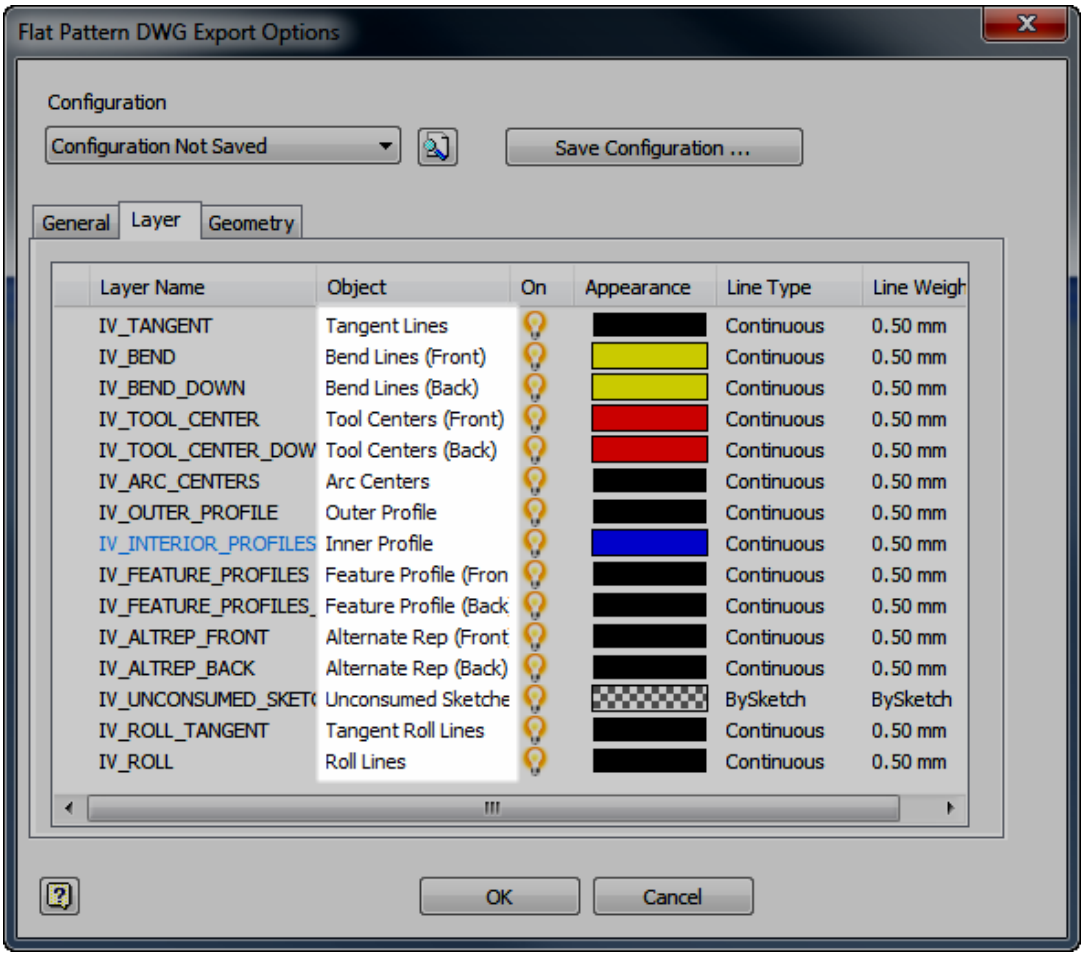


Figure 8: Layer Mapping options for Flat Pattern Export

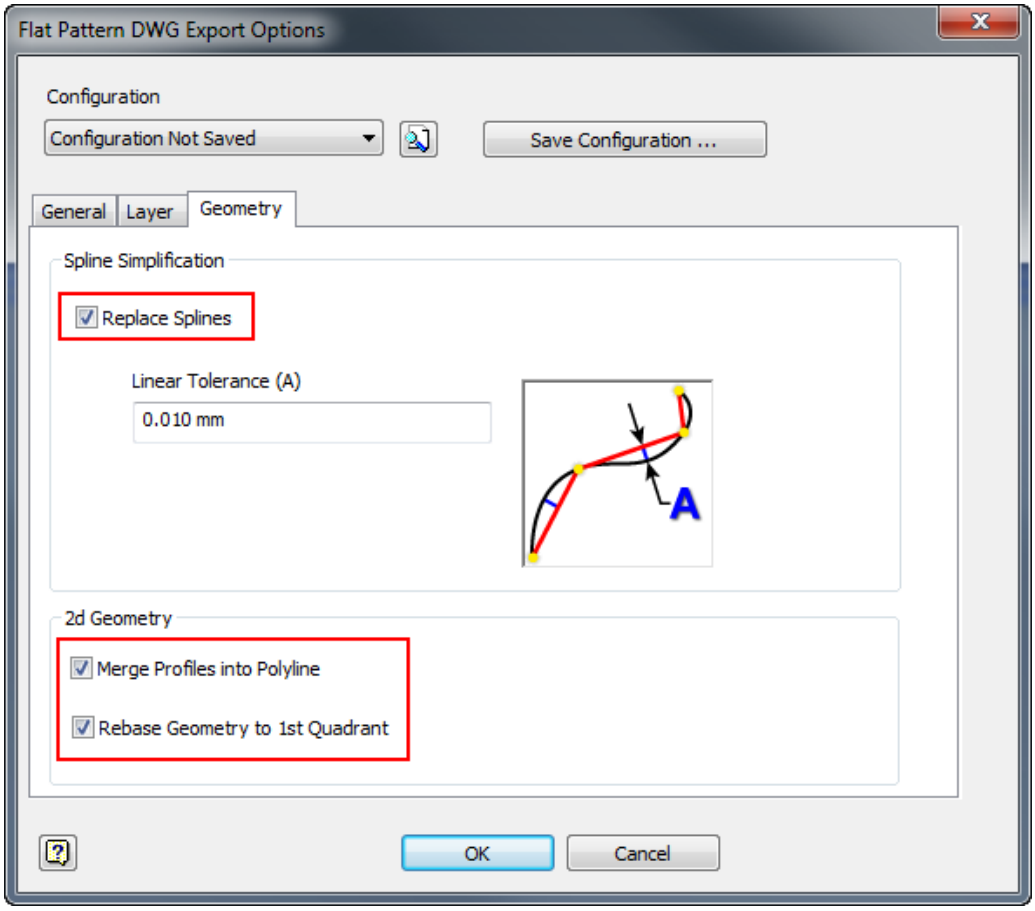


Figure 9: Geometry Clean Up settings for Flat Pattern Export

There is some scope to run a macro in AutoCAD to do some clever post processing on the flat patterns, to streamline the CAD to CAM process. For a lot of people this would mean getting the part number into the cut file.

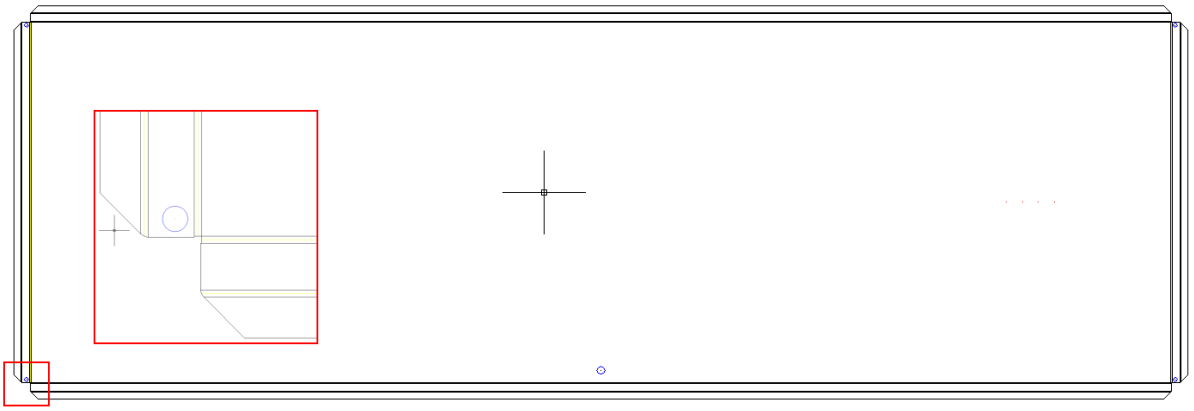


Figure 10: The result of a Flat Pattern Export automates the assignment of layers for some of the model features

Pros:

- Merge Profiles into Closed Polylines
- Spline Conversion to Polylines
- Object Recognition: Inside & Outside, and Features on the upper & lower faces.
- Rebase Geometry to 1st Quadrant
- To a certain extent ability to control the layers
- Semi-Automated Length, Width & Thickness values for BOM & Parts list.

Cons:

- No support for exporting text with the cut file/flat pattern!
- No feature recognition, so there's no method to include feature depths in the 2D data.
- The process has to be carried out manually on each part. So it's the same repetitive process over and over again.
- Sheet metal thickness won't link through to body extrusion value easily when using multi body modelling techniques.

FLAT PATTERN EXPORT ADD-IN EXAMPLE

Here is an example of how automating Inventor can make the process of creating sheet metal flat patterns a breeze.

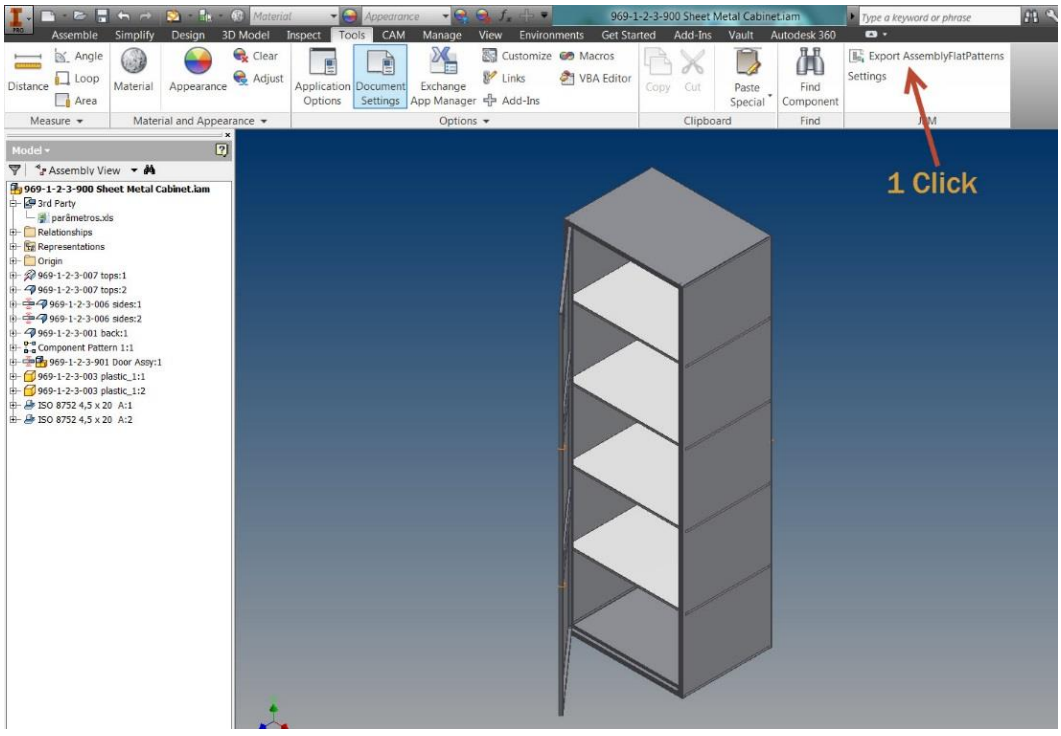


Figure 11: Export Assembly Flat Pattern

For many companies, the creation of 2D data for profile cutting can be a hugely repetitive, manual task which costs significant time. We tend to find that the requirements for exactly how the data is presented varies a lot. For this reason, it's difficult to come up with a "one size fit's all" approach for automating this task. This particular example is for a manufacturer who requires a table of information in a cut kit as well as 2D DWG files for each component. The tool runs as an Inventor addin and works in the assembly environment.

With an assembly file loaded, the command is executed and the routine runs through every sheet-metal component in the assembly, creates its flat pattern (if it doesn't already exist,) and then exports the flat pattern as a DWG.

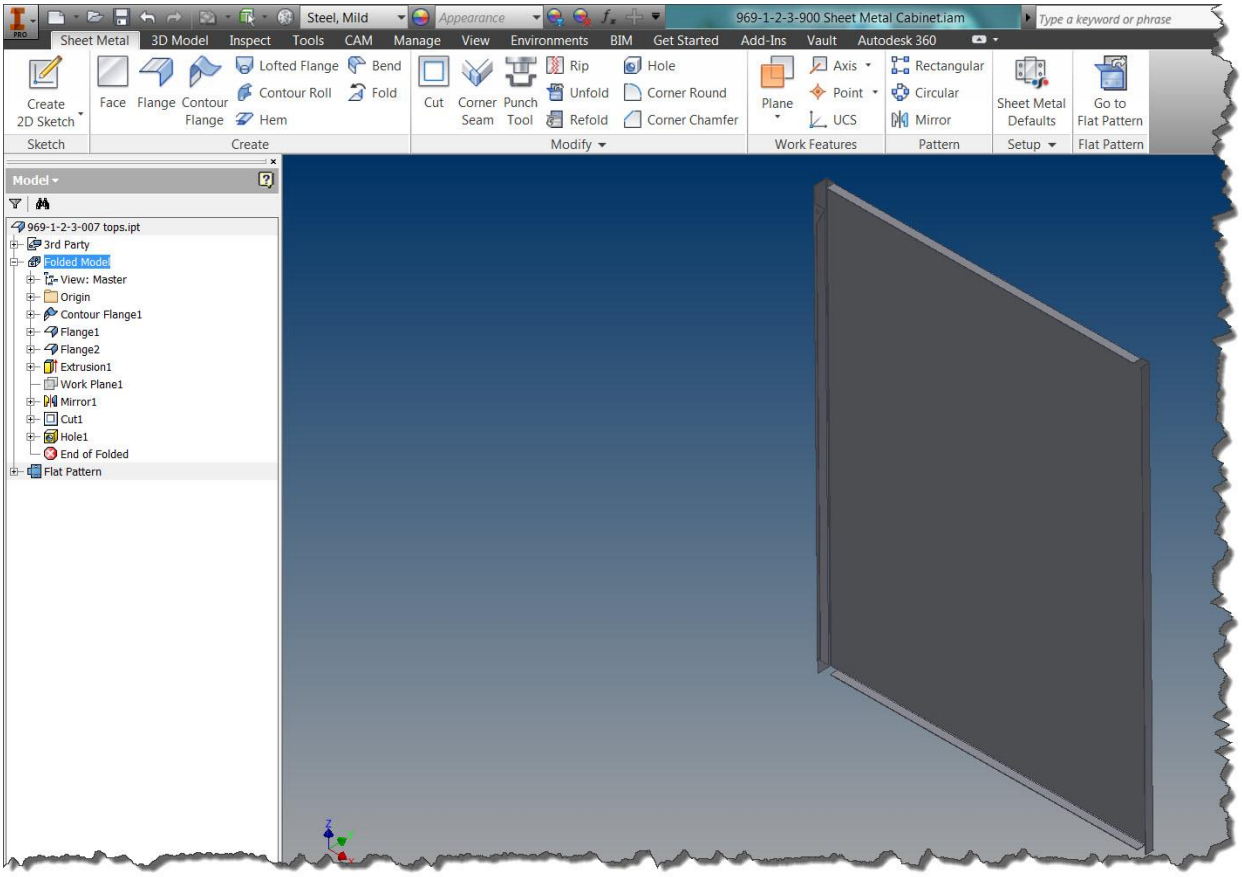
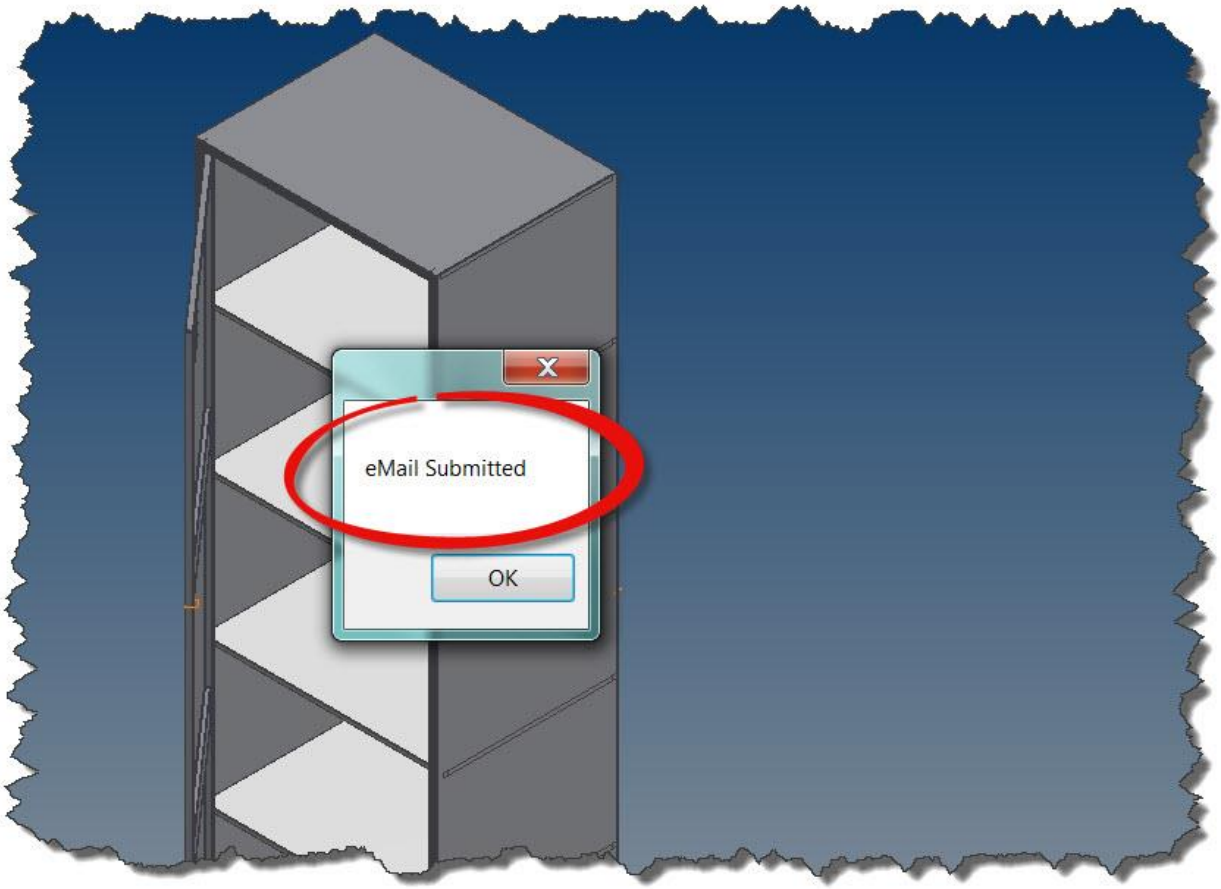


Figure 12: Flat patterns created automatically for all sheet-metal components.



Once the flat patterns have been created, an email is automatically generated with a HTML table that contains thumbnail images of each component as well as dimensional information, material, and assembly quantities. Additionally, the DWGs are attached.

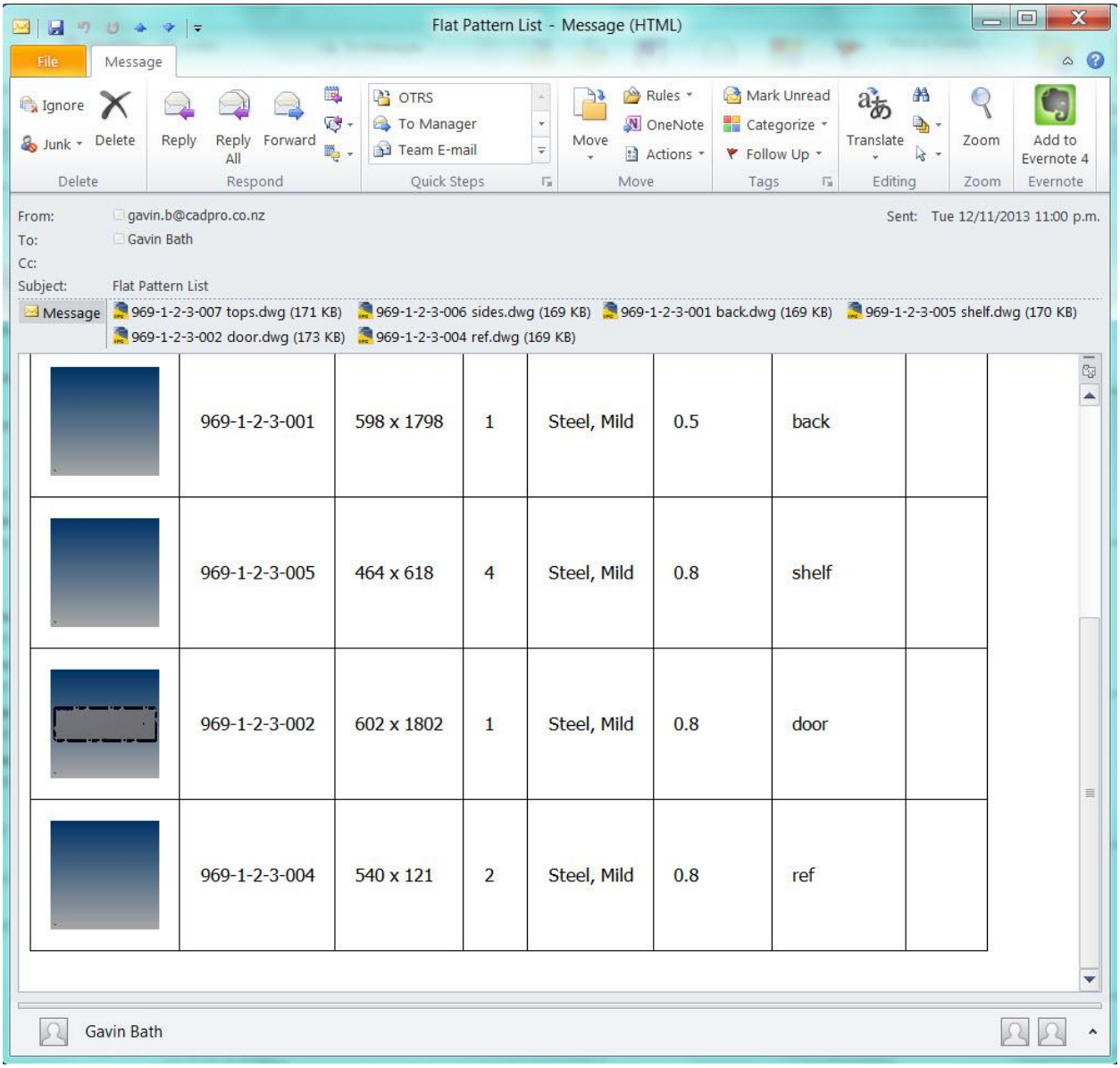


Figure 13: Example email generated by tool

This customization can be very easily edited to manipulate the data that is output, for example; changing from DWG to DXF file format.

Pros:

- Provides confidence in accuracy of quantities and dimensional information
- Really Fast / very efficient
- Email trail provides job ordering history
- Not easy to fudge quantities, add spares etc. without modifying assembly

Cons:

- Not easy to override quantities, add spares etc. without modifying assembly
- Reliance on programmer to edit customization if requirements change
- Doesn't handle non sheet-metal parts (unless identified somehow)
- Doesn’t handle grain direction on grain-sensitive materials

EXPORT FACE

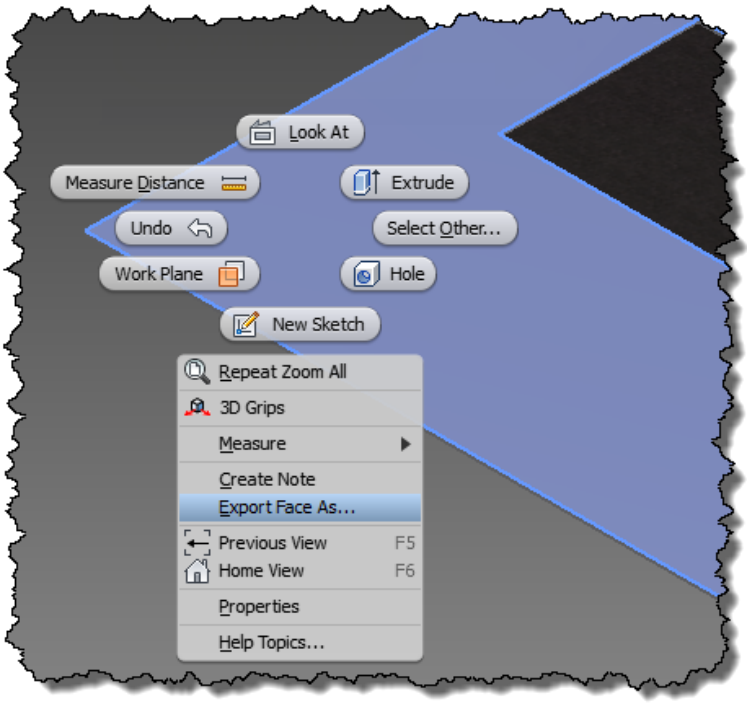


Figure 14: Export Face Radial Menu Command

I have a couple of videos showing you how to export a face of your part and a sketch in your part as a dxf/dwg file. People often use it for its simplicity, since you can quickly knock up a 2D cut file and fire it off to your CAM programmer.

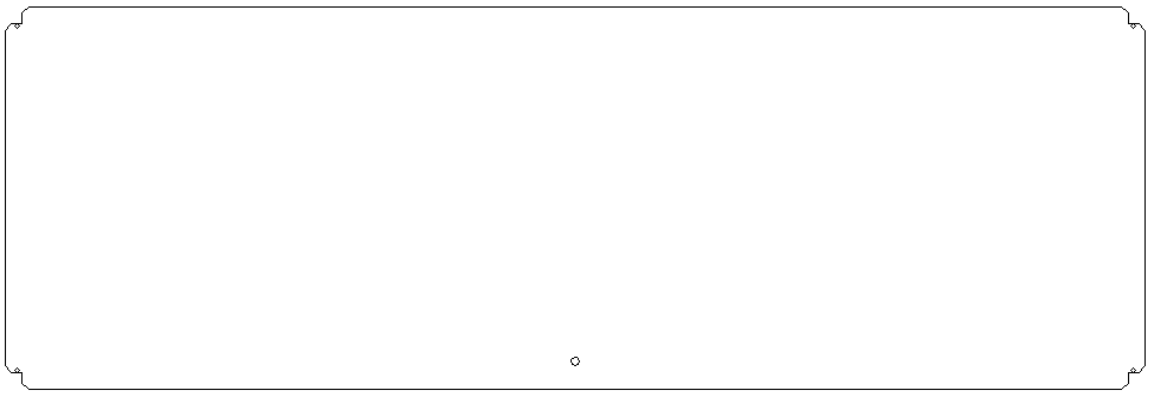


Figure 15: Export Face captures the face outline nicely. But it’s missing a lot of 'manufacturing' information

Export Face Pros:

- It's Quick and Simple
- There are some AutoCAD export options for objects & file type
- The result is closed polylines

Export Face Cons:

- Zero layer mapping options like there are in the Flat Pattern tool
- It doesn't capture geometry of the entire part, such as rebates, bevels and hidden features etc.
- No ability to include text in the export for engraving
- No ability to color the features by type for export

EXPORT SKETCH AS

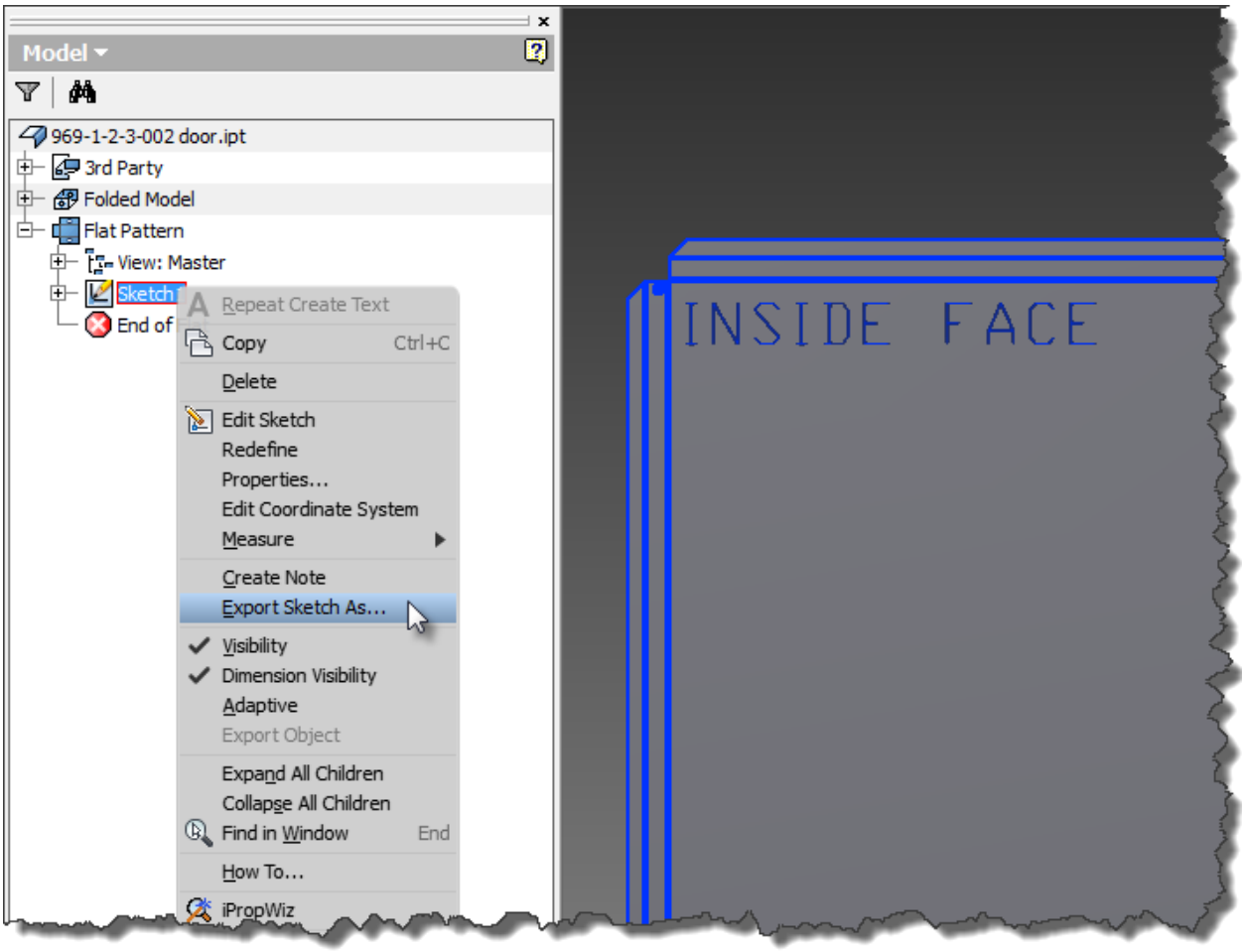


Figure 16: Export Sketch As allows you to capture additional information which can be added to the sketch

Export Sketch Pros:

- Quick and Simple process
- Some AutoCAD export options for objects & file type
- The sketch tools allow you to project geometry not included in the Face export
- Resulting geometry made up of closed polylines

Export Sketch Cons:

- Zero layer mapping options unlike the Flat Pattern export tool
- No ability to export text for engraving
- No ability to color the features by type for export

To clarify the engraving text issue. In both situations above you could emboss the text and have it exported with the face or sketch. However, this results in a text outline instead of a text center line. This is very inefficient for CNC machines to cut when the number is only there for practical rather than aesthetic reasons.

CREATE A DRAWING

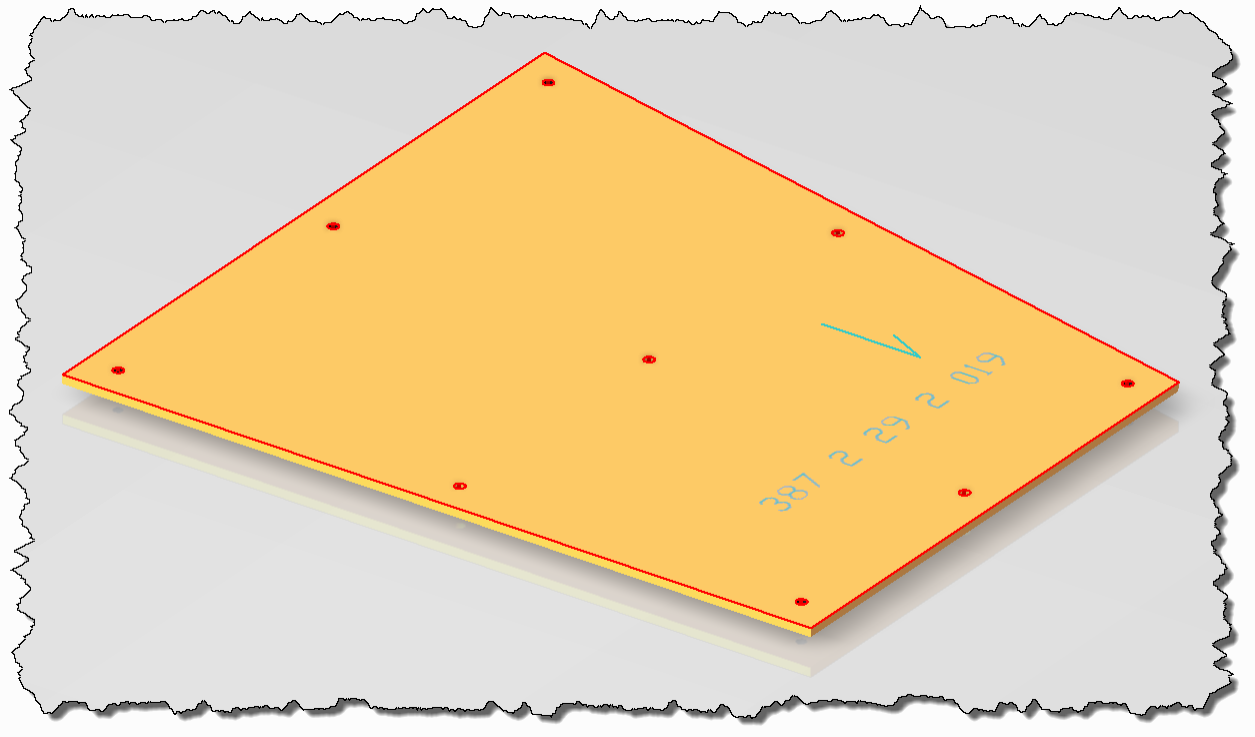


Figure 17: Inventor Part with Associative Sketch Cut File

This is my preferred method of generating cut files if you have a requirement for engraving or etching text onto your parts.

There are two reasons for wanting to create cut files using the drawing environment, they are:

1. 2D labelled cut files saved in a single file

In this case you may want to:

- Group your 2D cut files by material and thickness
- Then use leaders or balloons to access and display your parts iProperties
- Add instructions to the individual views via icons/symbols or notes for the CAM Programmer(s).

2. 2D cut files via model sketches

- Group your 2D cut files by material and thickness
- Display colored model sketches instead of visible edges
- Display positioned sketch text for CNC engraving

I'll focus on the 2nd option here:

1. Create your part
2. Add a sketch for any projected geometry (reference geometry)
3. Add a sketch for 'sketched' geometry and sketch text
4. Look at the top face (the one with the sketches on it) & set the view cube view as Front. This step is optional, but helps placement of the part views later
5. Rinse & Repeat for the other parts
6. Create a drawing
7. Start placing views making sure the views are perfectly normal to the sketch faces
8. Right click on each view in the browser & select 'Get Model Sketches'
9. Check that all sketches are visible (Sketch containing reference geometry won't be visible if they contain 'sketched' geometry as well)
10. Turn off the Visible Edges layer (to reduce/remove duplicate geometry)
11. Save as AutoCAD DWG
12. In AutoCAD use Quick Select to select geometries by color & add to the appropriate layer
13. Use the Overkill command to clean up the geometry
14. Close polylines using the polyline edit command & the multiple option.
15. Check all the cut files are 'water tight' and only open where intended.

Following that workflow alone would get you there. However, if you use Inventor DWG's as your company's standard drawing file format, then the following video demonstrating the use of the 'EXPORTLAYOUT' command in AutoCAD, may result in a few less files for you. Thanks to Paul Munford for letting me know about this one.

http://www.youtube.com/watch?feature=player_embedded&v=igPHBF9zbBE

So you may be thinking what a mission it is creating all this lot, however, you can automate the process via an Add-in or a macro. So I thought I would share a video of one of our cut file macros at work:

http://www.youtube.com/watch?feature=player_embedded&v=6zj33QGWPI8

What really gets me excited about this workflow is its associativity with the base files. You can do the hard work of creating your CNC cut files while the product is still being designed. Any subtle changes which occur prior to the design being released for construction are associatively updated. All with very little effort and at the click of a button are exported en masse to a dxf/dwg for some pre CAM prep.

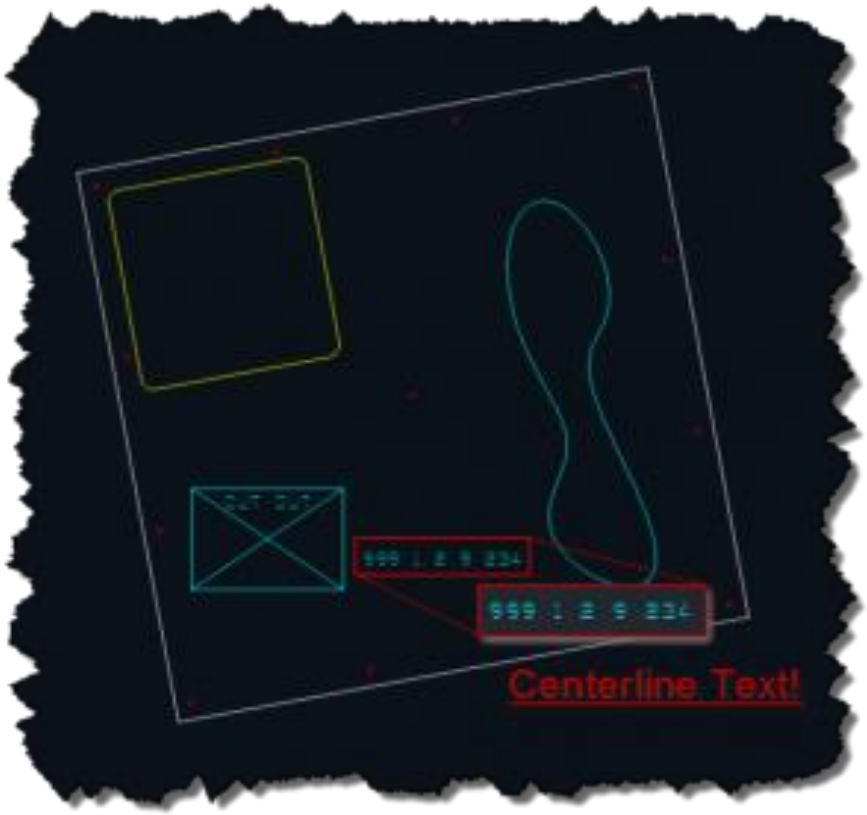


Figure 18: Once exported to DXF/DWG the txt font becomes centerline text

Pros:

- Associative Cut files
- Allows you to manage exactly what geometries you want in the cut file
- Export of the 'txt' font text results in center line text in AutoCAD which is perfect for CNC machines
- Geometry Layers & Color settings are maintained on Export which can then be leveraged efficiently in AutoCAD

- Batch processing of cut files is possible natively, with zero customization
- Result is a single file containing multiple cut files which allows for efficient preparation in AutoCAD prior to CAM import

Cons:

- Resulting geometries are individual & open
- Splines aren't converted to polylines
- Based on the listed workflow, all geometries are on the same layer
- Result is a single file containing multiple cut files which can mean a lot of work splitting the cut files up in the CAM software prior to nesting.

SETTING UP A SKETCH TEXT STYLE

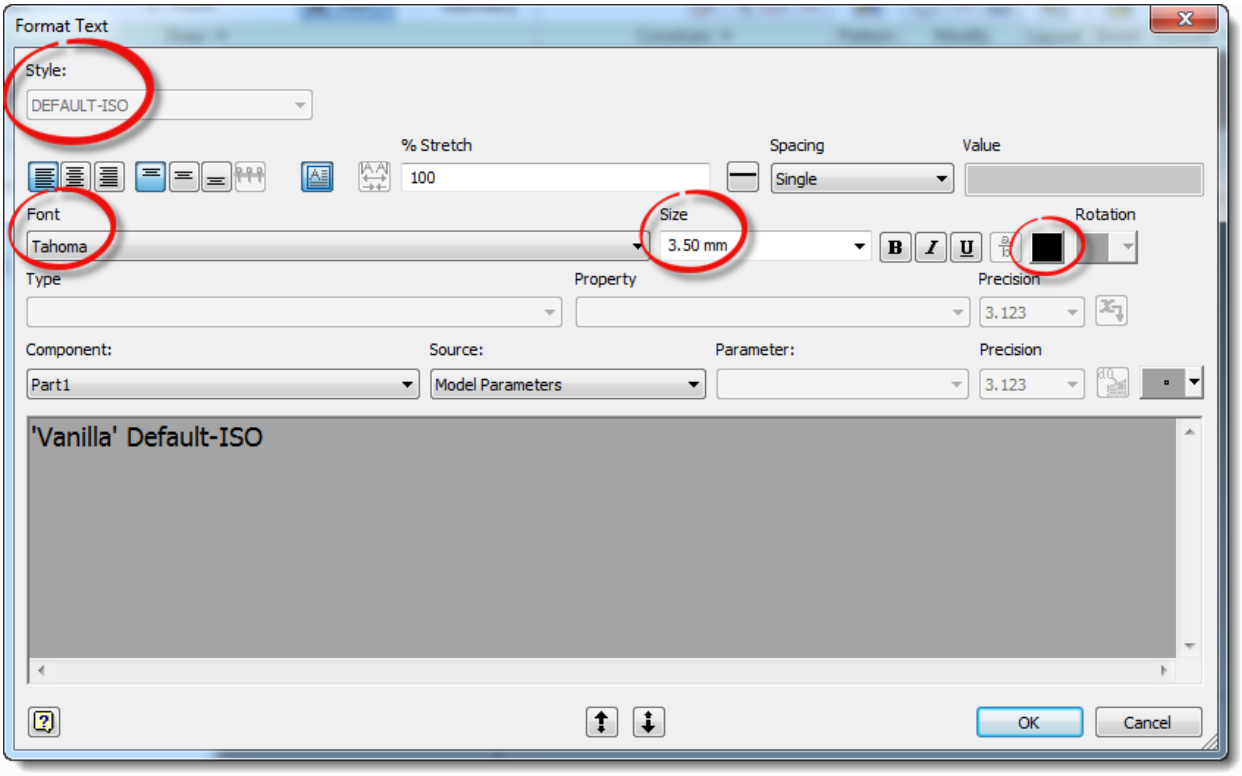


Figure 19: Sick of changing these settings all the time?

Do you find yourself swearing at Inventor when you have to repetitively set the font, size & color of your sketch text in the part environment? It used to really irritate me that the dialogue was identical to the one in the drawing environment, but the text style drop down is permanently greyed out and set to 'Default ANSI' or 'Default ISO' depending on your default standard. I just wanted to pick my own Style which had the presets I use most often and couldn't. But there is a sneaky undocumented work around which works a treat.

If you were to go and take a look in the styles & within all the Inventor folders on your local drive, for any trace of a text style named 'Default ANSI' or 'Default ISO' etc then you won't find

anything. So the trick is to create a text style matching the exact name of the style you can't find, then just configure it with the settings you are so fed up with selecting each time you create a new text box.

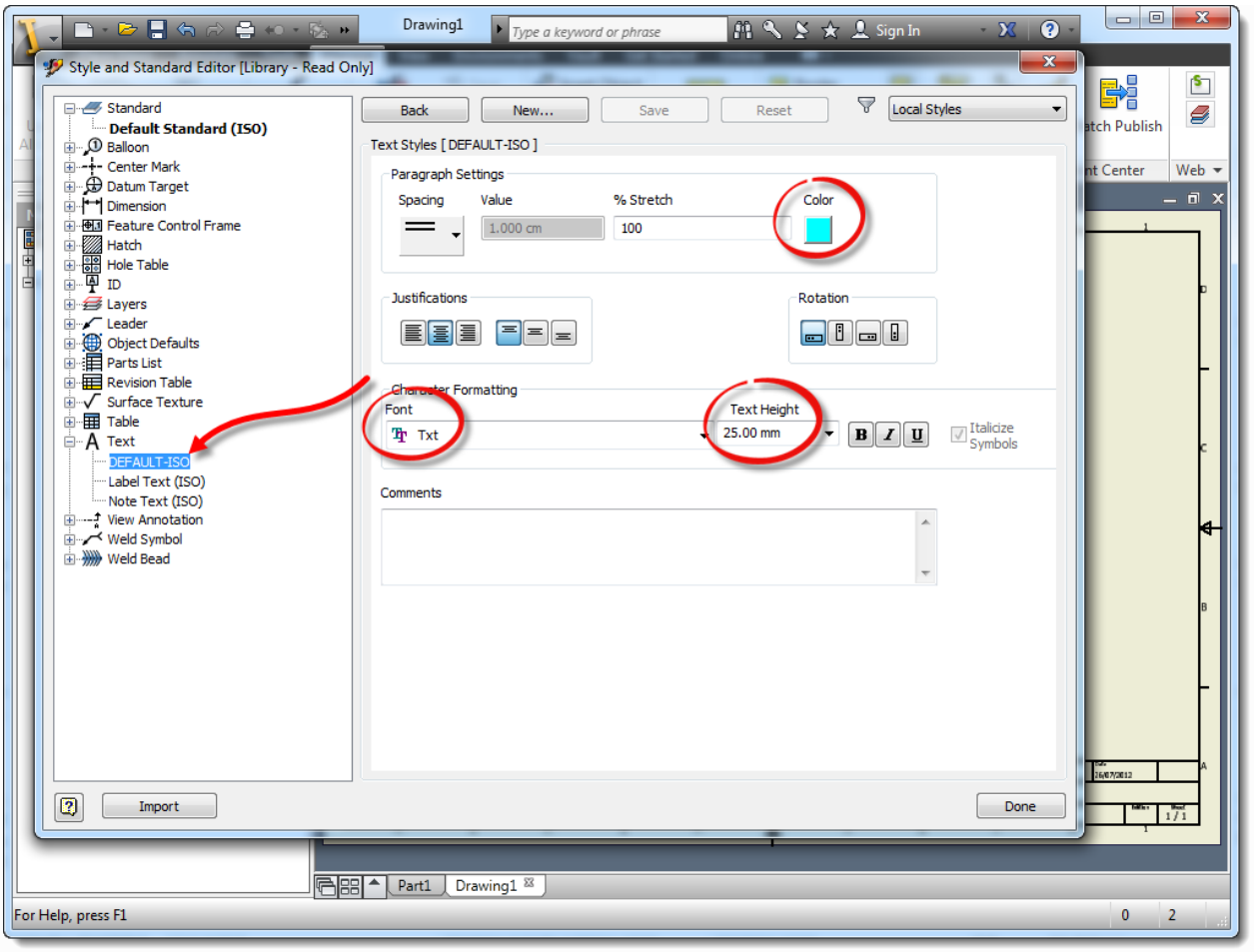


Figure 20: Create a new text style with an identical name as the 'greyed out' text style in the text editor

The next time you create a text box in a sketch all of your settings will be just so. If you've been creating a tonne of CNC cut files requiring text engraving with Inventor, it's at this point you may weep a little (I certainly did).

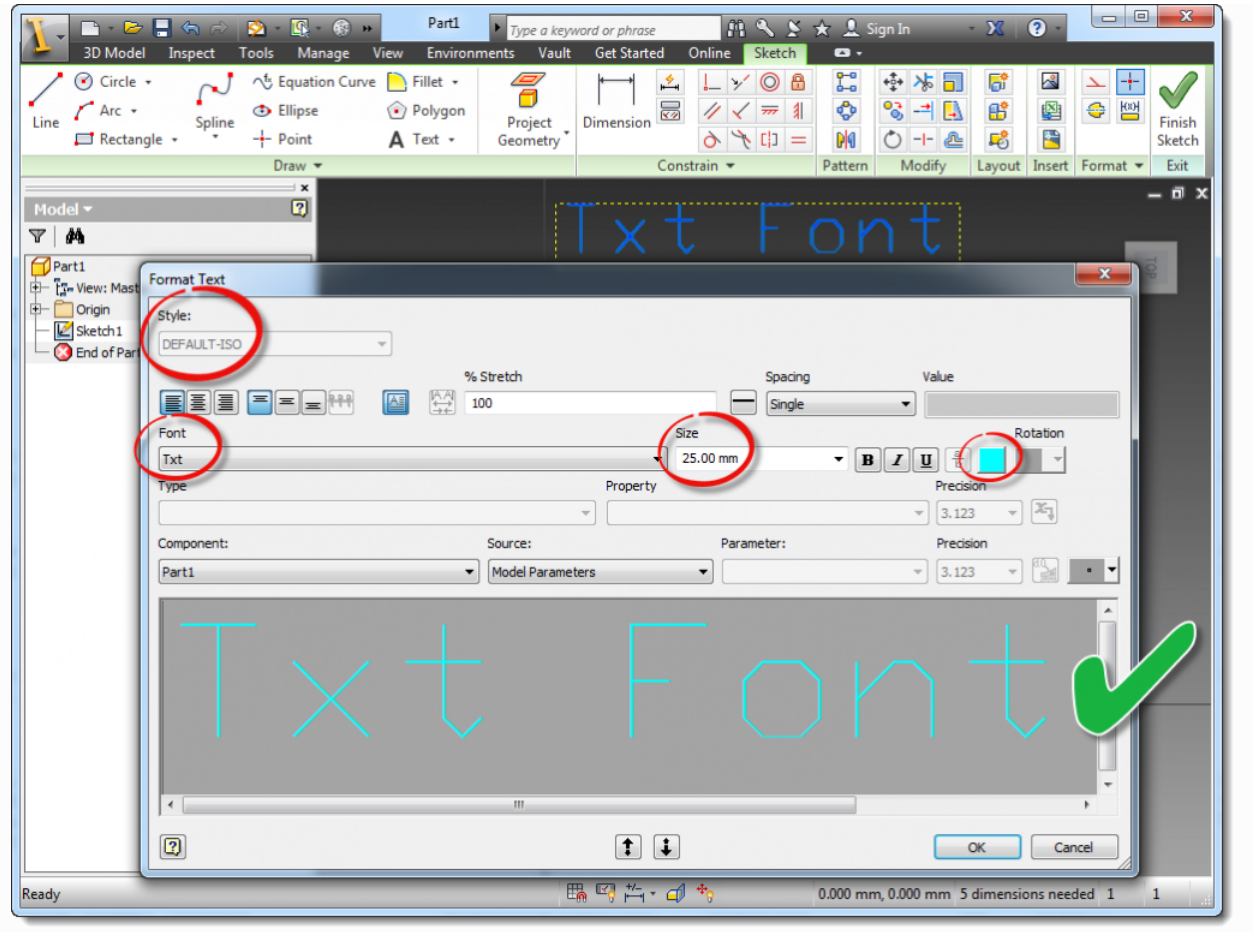


Figure 21: Lovely Jubbly check that out! Perfect settings every time

Thanks to the individual who shared this solution with me a number of years ago via the Autodesk Discussion Forums.

DELIVERING/PRESENTING MANUFACTURING INFORMATION

VAULT

Delivering the CAM & manufacturing related information to those who need it has been referred to throughout this hand-out & the presentation. However, there are a few additional things which can be done with Vault to focus the view of the end users within manufacturing. I'll focus on different CNC operators to illustrate the example.

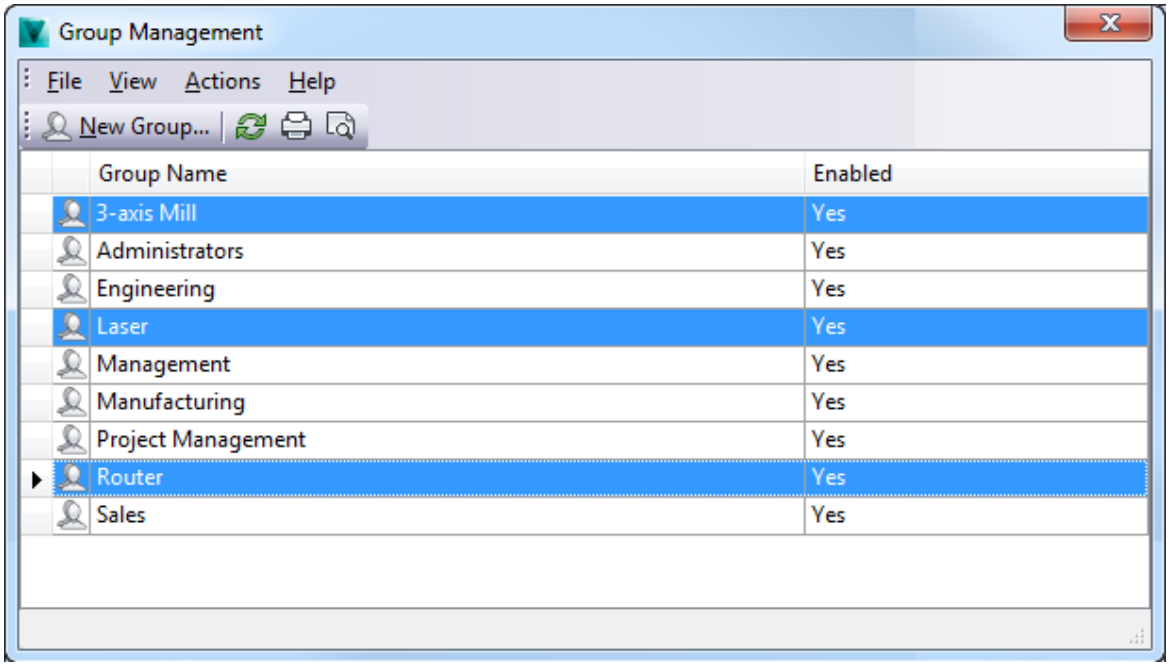


Figure 22: Create a group for each CNC Type

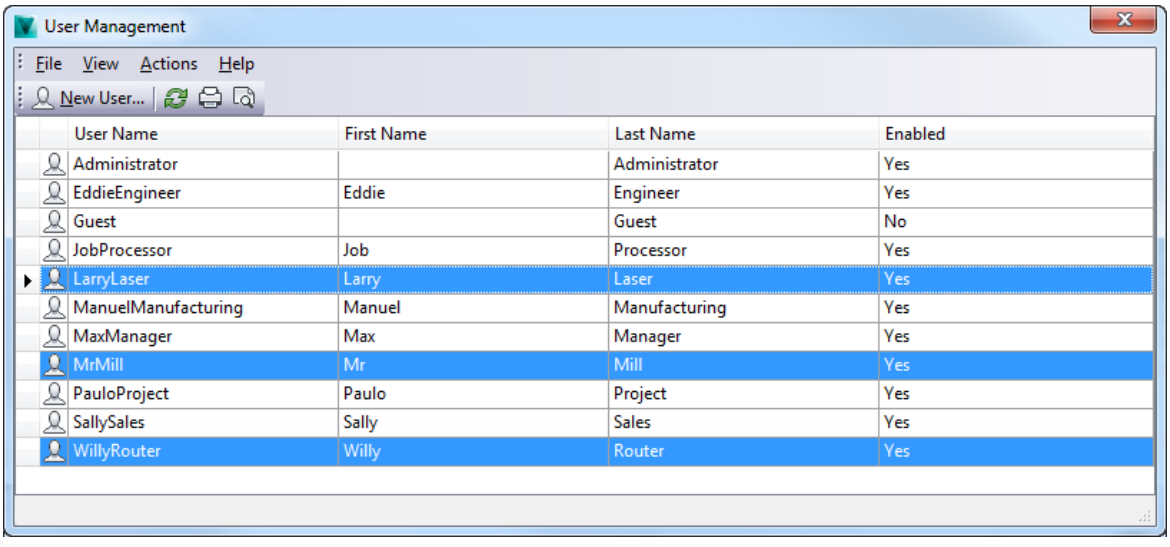


Figure 23: Create CNC users and add them to their respective CNC Groups

Create a lifecycle for each CNC type:

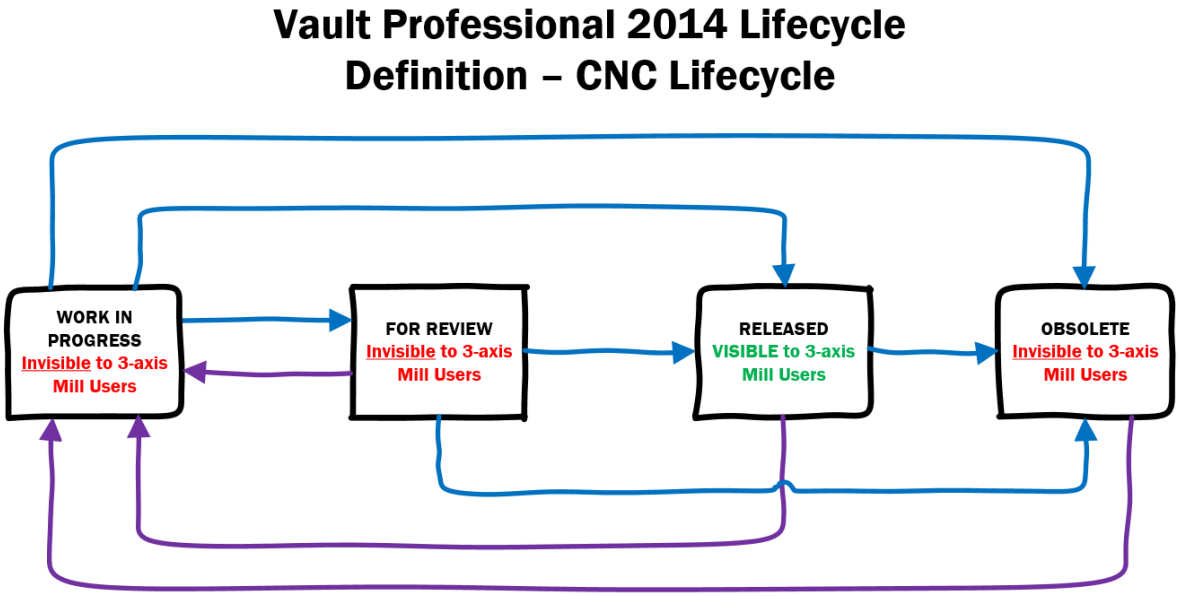


Figure 24: The 3-axis Mill group can only see files participating in the CNC for their CNC type when the files are released

Create a Category for each CNC type & assign a Lifecycle Definition, a Revision Scheme and all the properties you want enforced on files of that category.

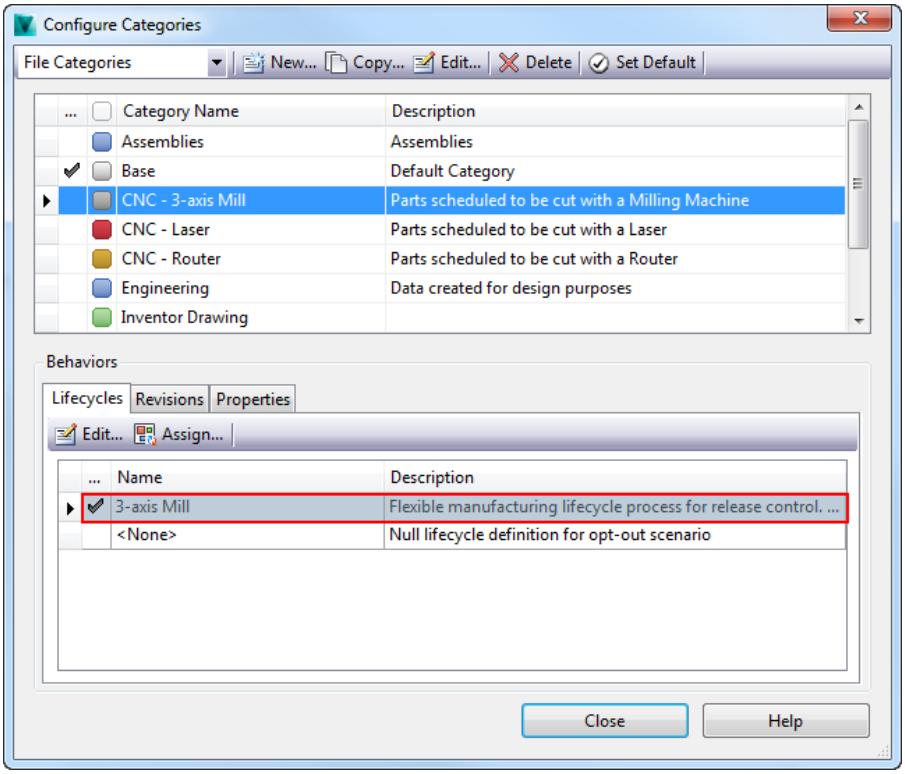


Figure 25: The 3-axis Mill Lifecycle has been assigned to the 3-axis Mill Category

Now you can create a rule for that category so any files meeting the Rule Criteria when checked in for the first time get categorized automatically.

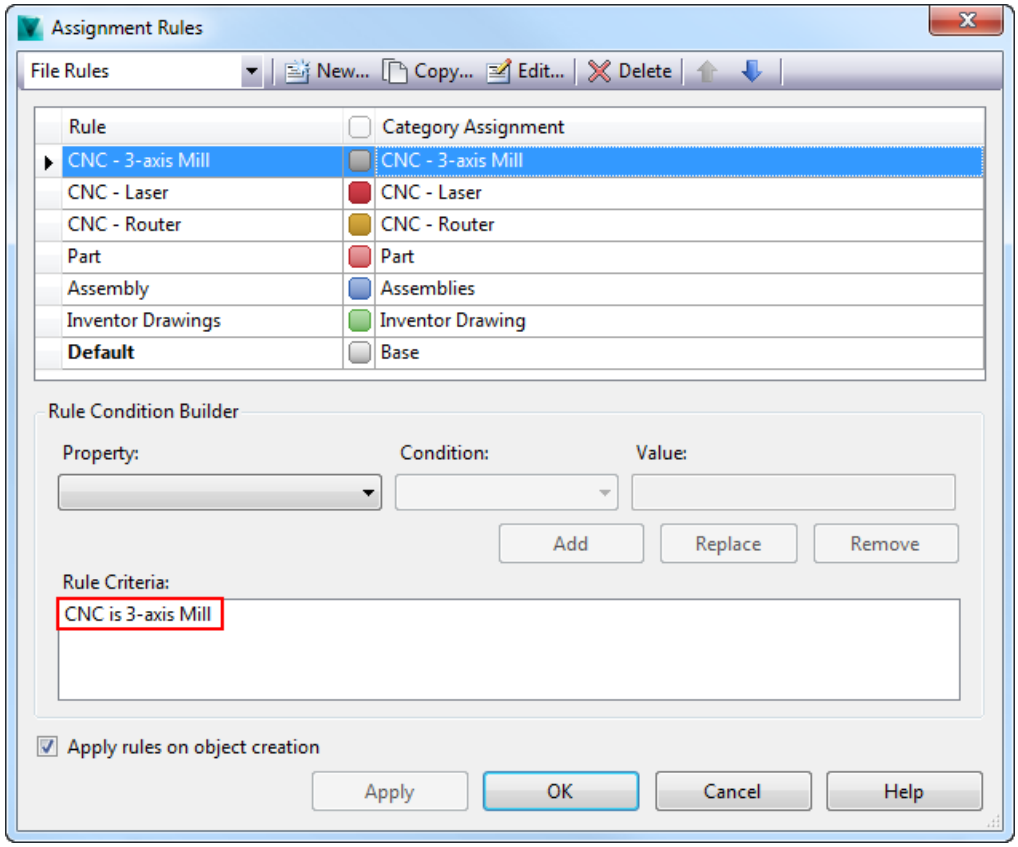


Figure 26: Using the Inventor CNC property you can automatically assign categories upon first Check In

The idea behind this approach means you can configure Vault in such a way that CAM programmers & CNC operators can only see files which are relevant to them. You could expand on these lifecycles allowing them to move the files off into another state once they have been programmed or machined etc.

You could then take advantage of Vault's excellent reporting tool to present status information to management and clients.

If you don't want users to have to rummage about in the depths of folder structures you can simply create some Search folders and combined them with some predefined Custom views in much the same way as I did in the section on Parts sorted by material & thickness into Assemblies.

3D CNC CUT FILES

This is where you start to see data management software come into its own. Vault isn't just a means to look after your CAD data, but you can use it to make the most of it & the information

contained within it. We will now show you how you can use all that metadata to speed up the production of CNC cut files & get your designs to CAM faster.

PARTS SORTED BY MATERIAL & THICKNESS INTO ASSEMBLIES

Using Vault's Custom View tools, sort and group parts by material & thickness:

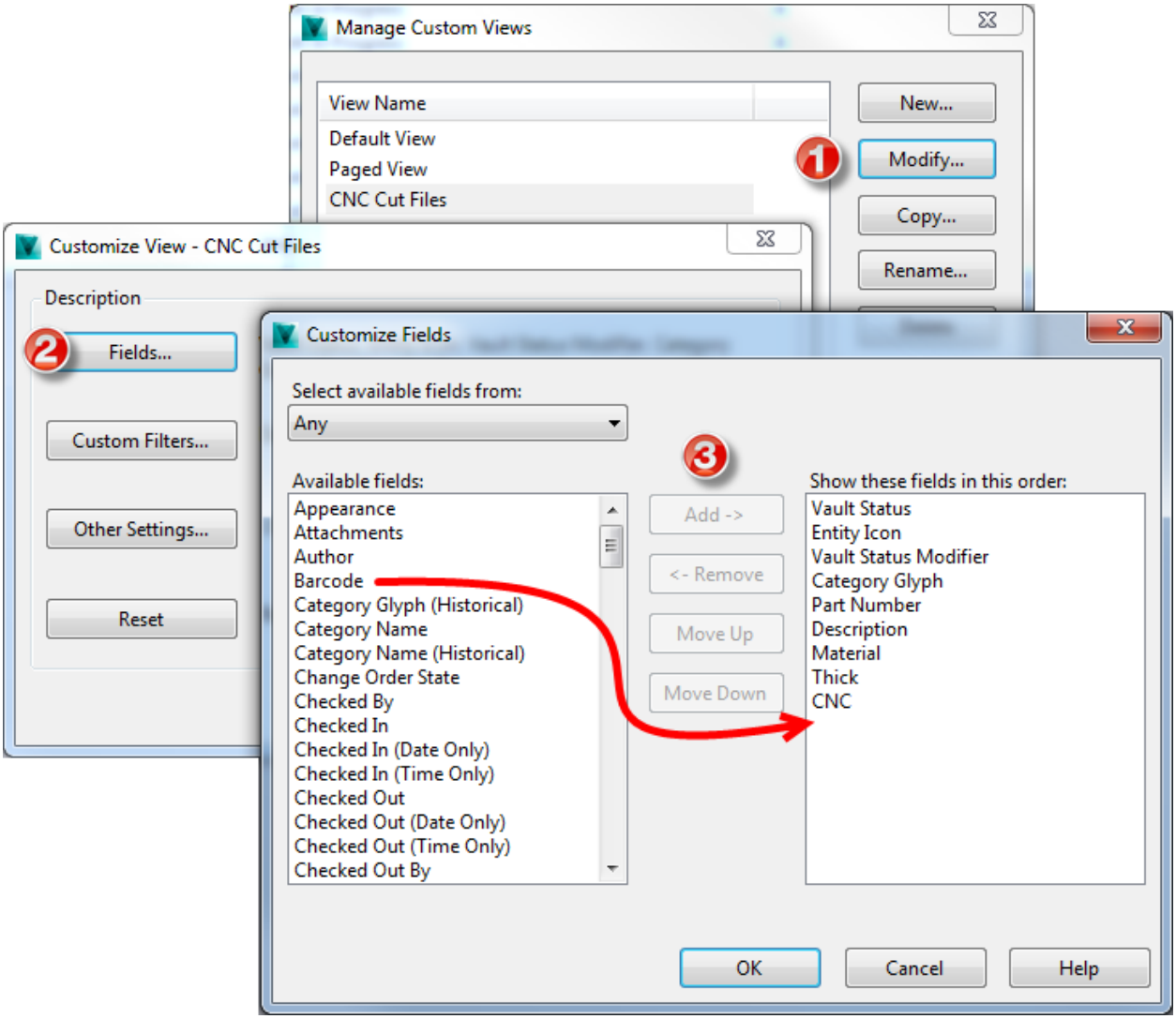


Figure 27: Create a custom view in Vault Explorer

You can also setup Custom Filters & Other settings for your custom view. These tools can be surprisingly powerful once you learn to use them:

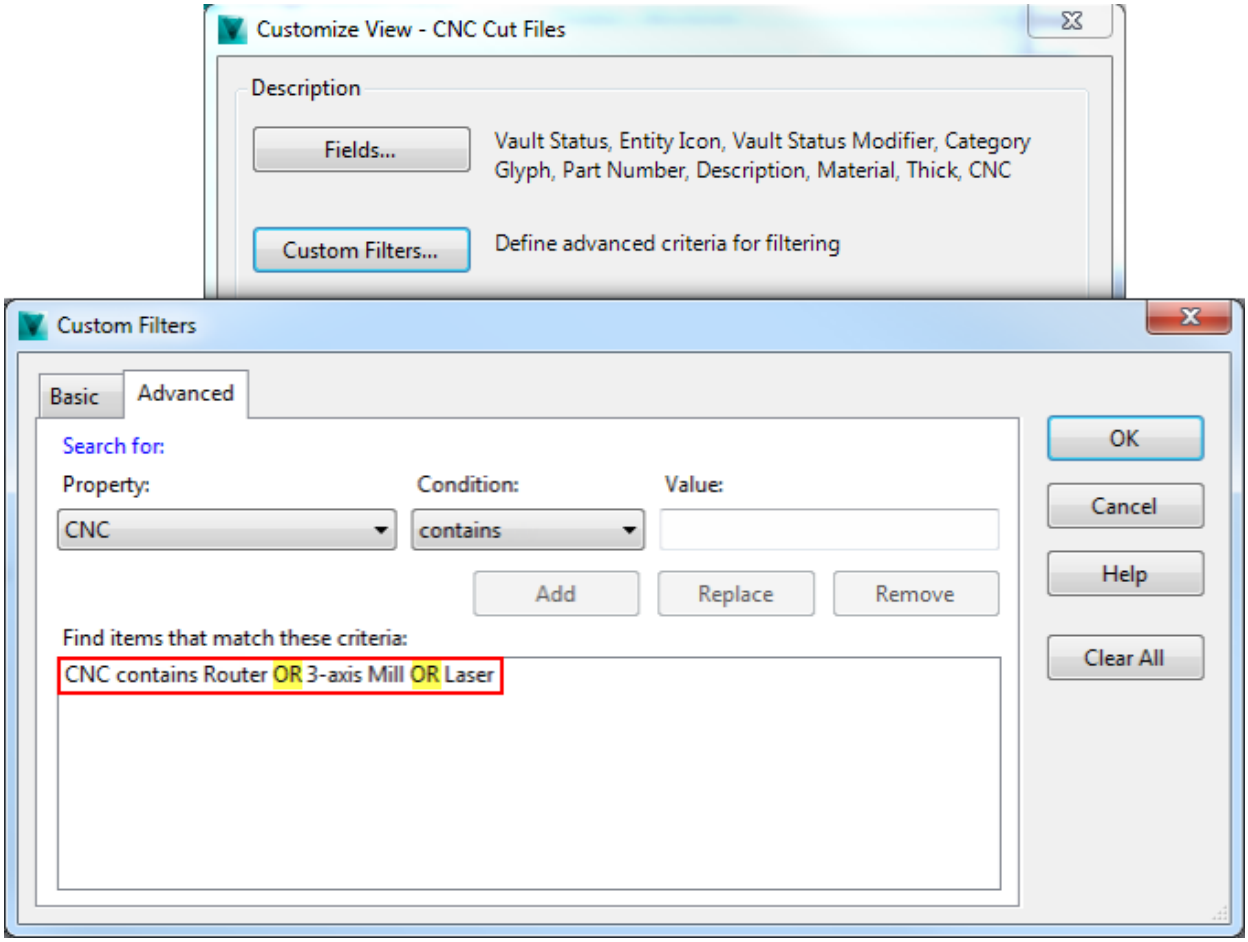


Figure 28: This is a sneaky Vault tip to get around the standard 'AND' nature of this dialogue

Drag & drop the grouped parts from Vault Explorer into an Inventor Assembly

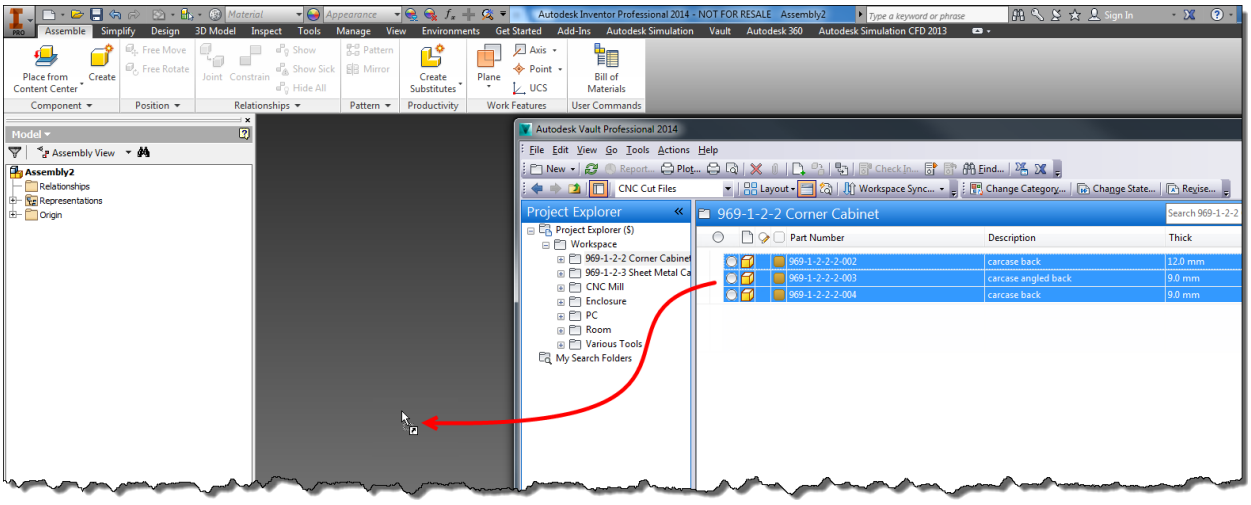


Figure 29: Drag & Drop CNC parts from Vault. Vault shows the Custom Filter created above activated

Constrain parts into place using the IPW User Defined UCS aligning axis if appropriate

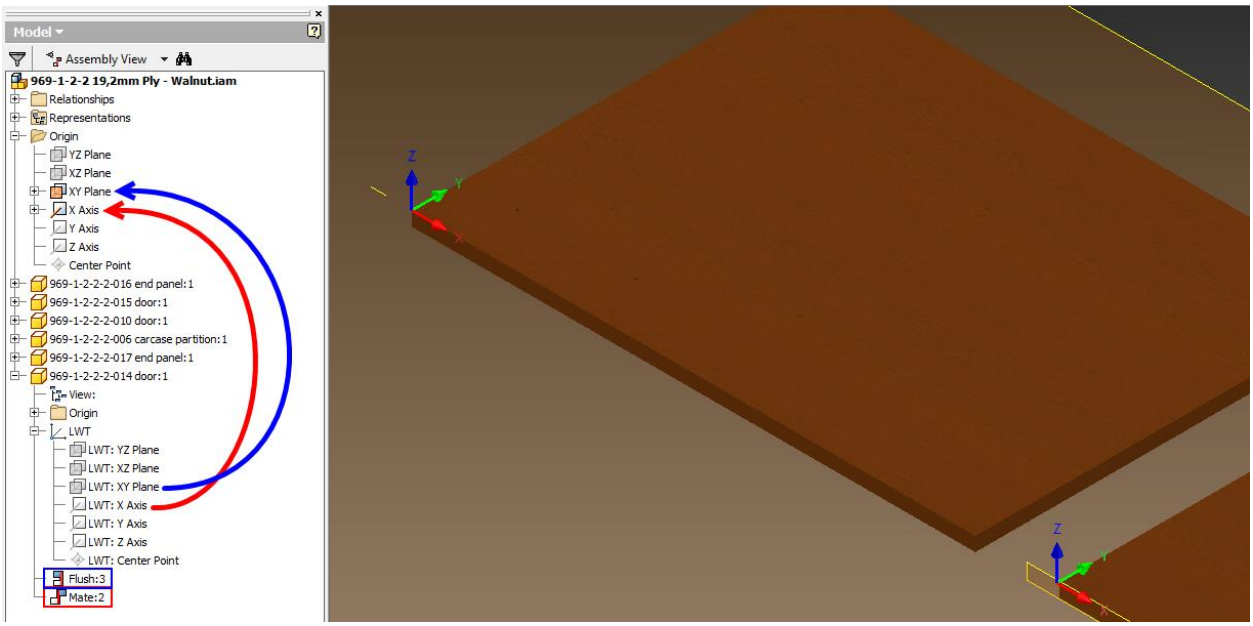


Figure 30: Constrain IPW UCS with the Assembly Origin

Save the assembly into a folder of the same name using the material & thickness as the filename for the assembly. Use a unique identifier as well, such as a contract/project number.

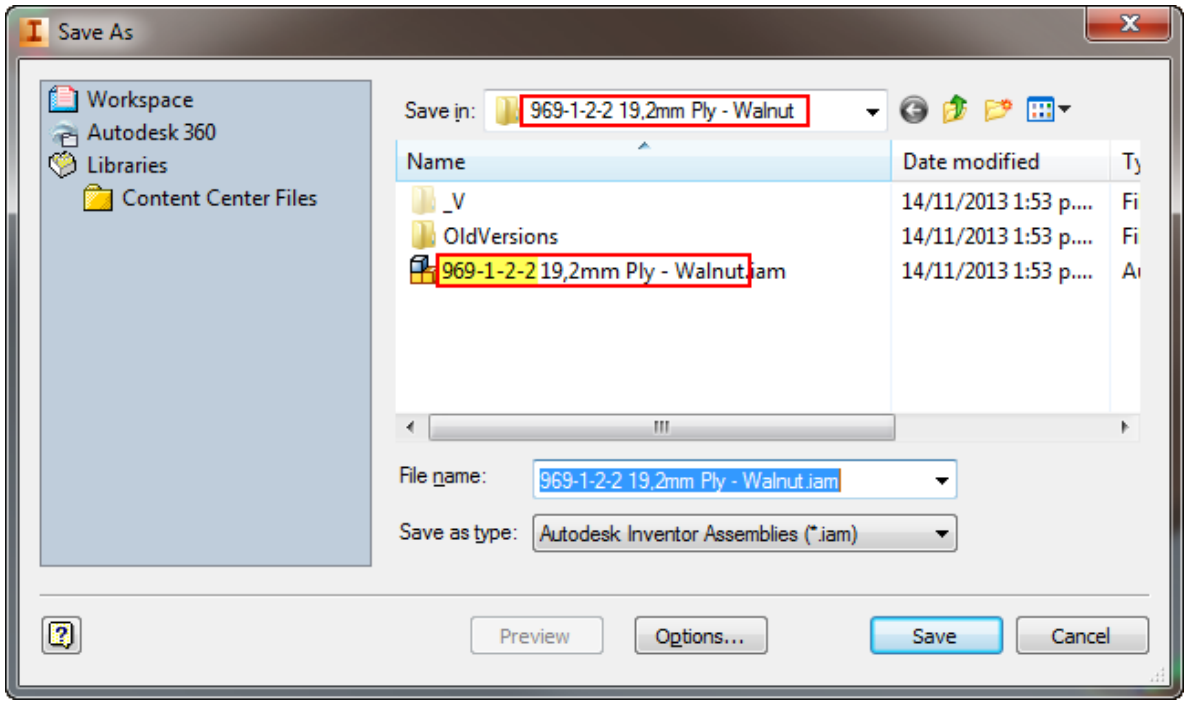


Figure 31: Yellow Highlight is the unique section of the filename

Check all the CNC Cut file assemblies into Vault, you can dump them all into a blank assembly to check in at once if you want. Without checking in that 'dummy' parent assembly.

INVENTOR HSM EXPRESS

Inventor HSM Express is a free 2.5D fully integrated CAM system which runs inside Inventor and Solidworks. For prismatic components, this software will allow you to very quickly generate quality CNC code to run your machine tools. For Inventor or Solidworks users, the HSM Express addin can be downloaded from cam.autodesk.com. When installed, the addin will add a CAM tab to your ribbon and also a CAM browser. Selecting either of these will take you into CAM mode shown here:

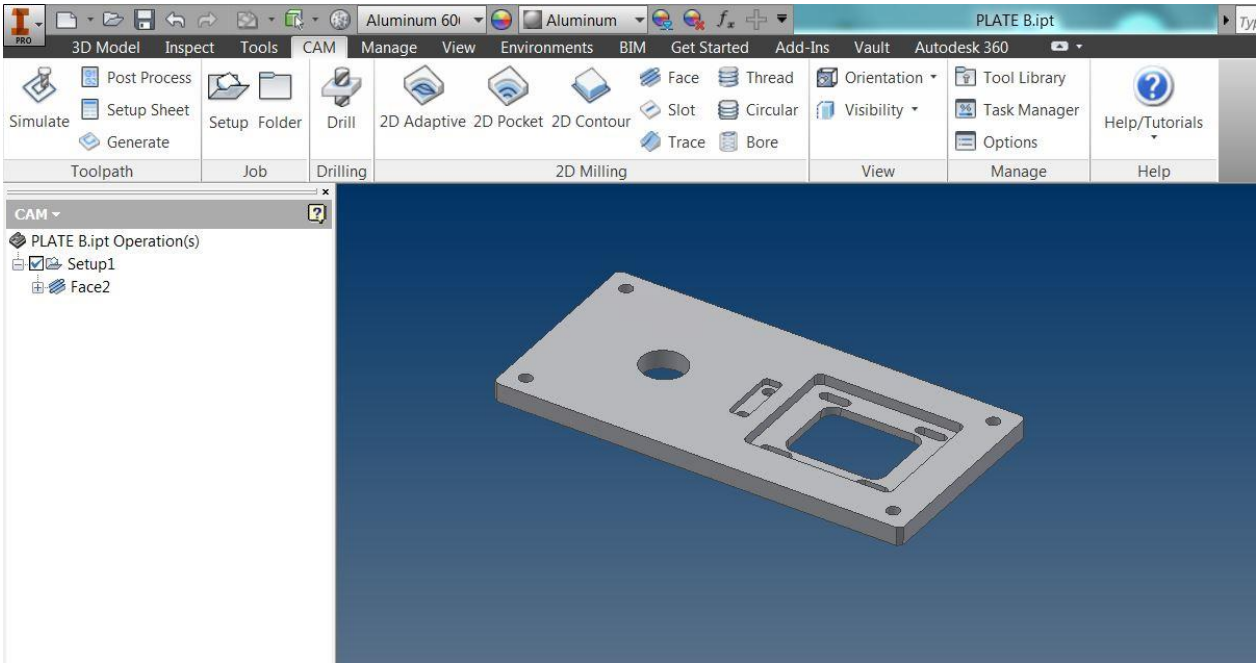


Figure 32: CAM Browser and Ribbon loaded in Inventor

Like many Inventor workflows, you generally work from left to right across the ribbon. Starting with creation of setups, where you define stock size and orientation, you then move through the toolpath creation which converts Inventor features to toolpaths automatically based on tooling, strategies and other parameters specified by you. Toolpaths can be visualized as lines, or a simulation process can be run which animates the tool movement. A still image from a simulation can be seen below:

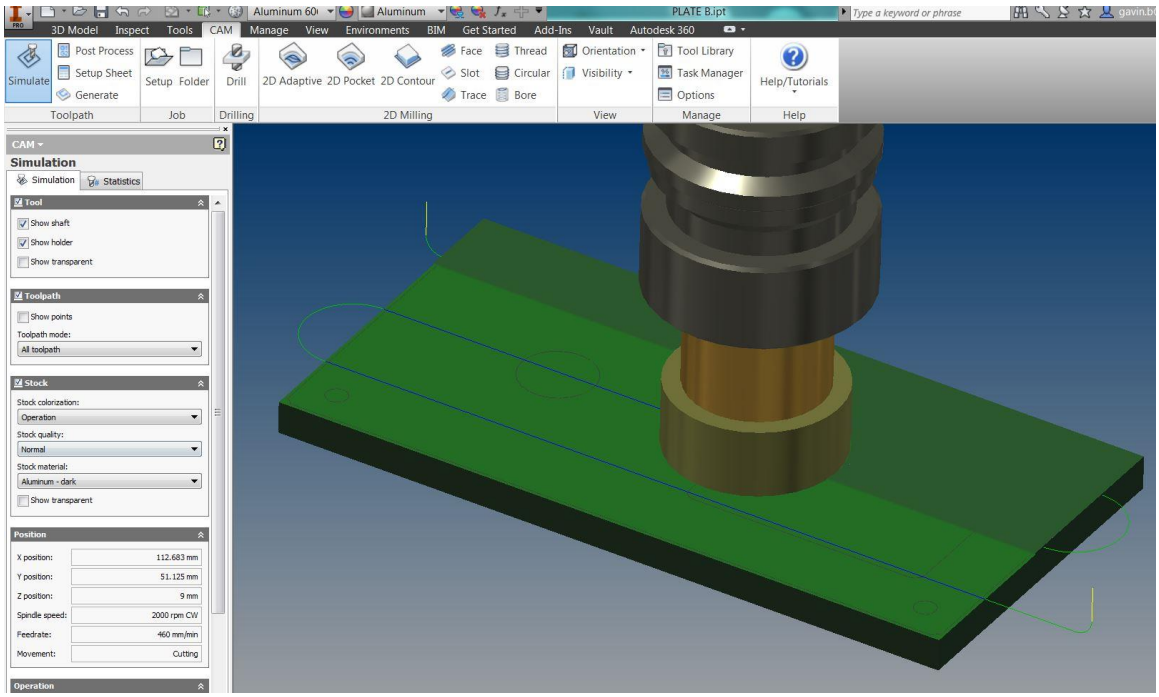


Figure 33: Toolpath simulation running

This simulation feature is invaluable for checking the logic the software has applied and ensuring the tool doesn't go anywhere untoward.

Once you are happy with your simulated toolpaths, the file can be post-processed to generate the G-code required to feed to your machine controller. This G-code can also be backplotted to ensure accuracy of the conversion and to visualize the instructions given by the code. If there are issues with the backplot, this can be an indication that modifications to your post-processor are required.

Pros:

- Fully integrated with CAD package
- Easy to use interface with good help
- Fantastic support for developing post processors
- Keeps the CAM data with the part file, reducing the number of files to manage & increasing the odds the correct version will be machined.

Cons:

- Limited to 2.5D (no 3D operations e.g. surfacing or helical interpolation. This will be in the paid version)
- Adds CAM data to part files which increases file size
- No nesting

HSM WORKS FINISH PASS – SEE THIS IN CLASS