

323169

Supplementing Your Design with Inventor CAM and Inventor Nesting

Stan Wile
IMAGINiT Technologies

Learning Objectives

- Learn how to integrate Inventor CAM and Inventor Nesting into existing workflows
- Learn how to create G-code from existing Inventor models
- Learn how to create nesting from existing Inventor assemblies
- Compare Autodesk's products for CAM and nesting with existing third-party software

Description

From cataloging tooling to final post processing, this class will walk through the world of Inventor CAM software. We'll review several real-world examples to see the different machining operations and their options. For anyone from hobbyist to professional, Inventor CAM really is the one-size-fits-all software for your machining needs. We'll then turn to Inventor Nesting and look at how Inventor quickly creates nested sheets from assemblies, as well as from other 2D profiles. Examples will include sheet metal and woodworking projects. In both workflows, we'll modify the source files to show that the machining paths and nestings automatically update.

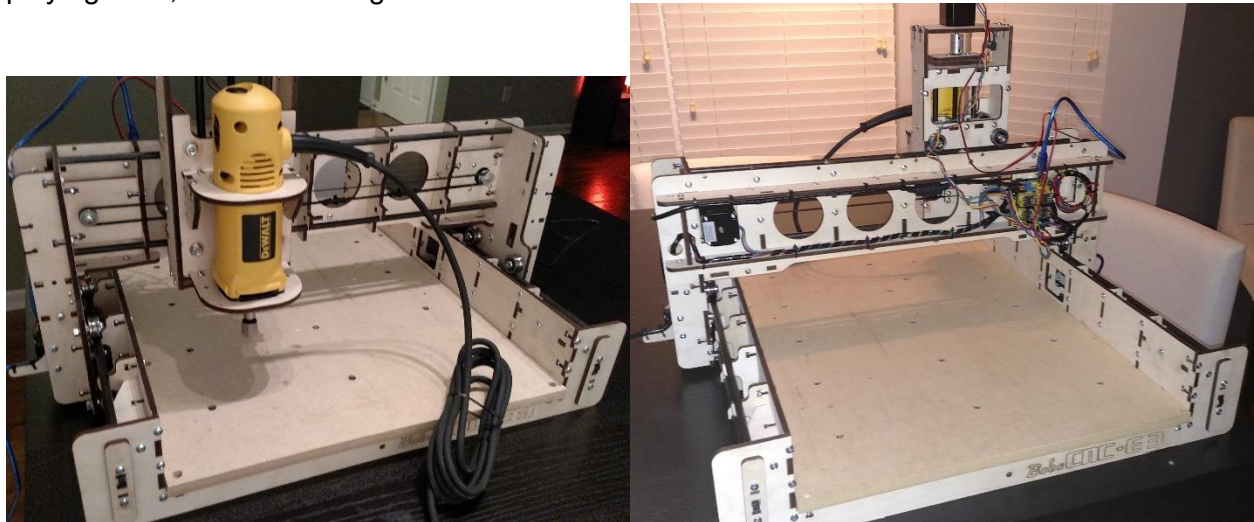
Speaker(s)

Diversified and experienced, Stan is a valuable team member of IMAGINiT Technologies. As an accomplished and proven engineer, he has significant experience in all aspects of 3D conceptual design. Stan has been instrumental in implementing the switch from AutoCAD to Inventor for many firms. He has also implemented Vault to solve data management issues involving multiple engineers working on the same projects. He has written user manuals for all drafting procedures including data management with Vault and an entire plant manual for ISO 9000 certification.

Stan has quickly become one of IMAGINiT's "go to AEs" for the most complex of implementations, including Atlas Copco, Parker Hannifin, and Duke Energy. He is also one of our AutoCAD Electrical experts. Stan is an adept and experienced instructor who teaches all Inventor course offerings, as well as AutoCAD and AutoCAD Electrical.

Introduction to Inventor CAM

Having been out of the manufacturing industry for over 11 years, I decided it was time to brush up on my CAM and nesting skills. After looking over quite a number of small CNC machines, I settled on a wooden kit. You read right, a wooden CNC router! Who knew? It was the BobsCNC E3. The E4 wasn't out at the time or I would have more than likely purchased that one because of the slightly larger footprint. It took me about three evenings to assemble, another evening playing with the included code sender software, and I was ready to start playing... uh, I mean working.



For 700 bucks you have EVERYTHING you need in this kit to start. Even the router is included. I don't know of a cheaper and easier way to get started in CNC milling and open whole new possibilities for your home shop. After assembling it, I really expected quite a bit of play but when the servos turn on, it actually becomes quite rigid. My first few cuts were a series of circles. I stacked them up and looked for deviation between them. They were all exactly the same. Perfect circles! Impressive for a \$700 all-inclusive kit!

Ok enough about the BobsCNC. I have since sold it to a co-worker and I'm looking to go bigger. I just wanted to start with pointing out that these machines don't have to cost thousands. And with the CAM capability built into Fusion and Inventor, anyone can move into the world of CNC.

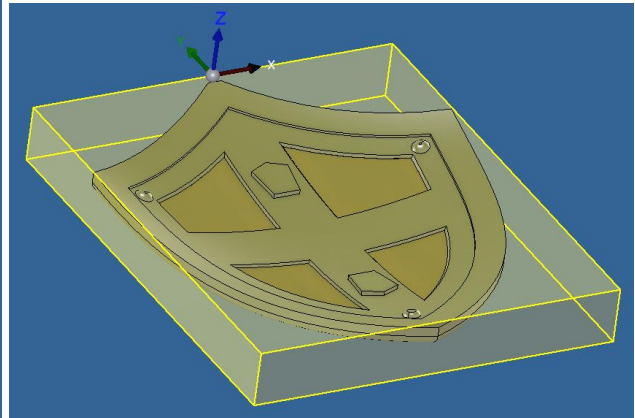
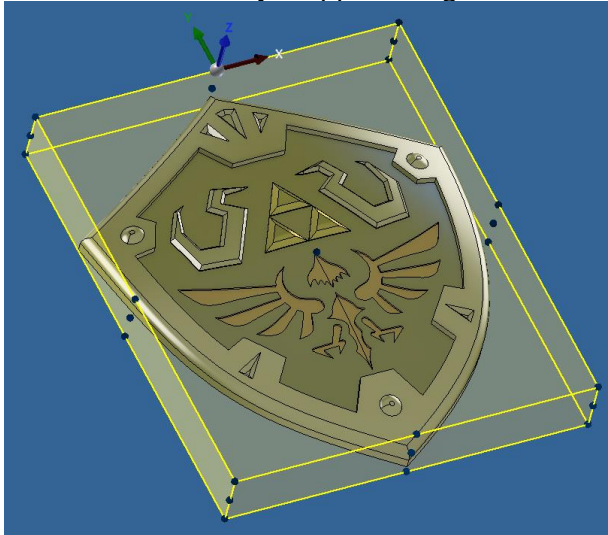
The CNC Process

After modeling the 3D part you are looking to machine, simply switch to the CAM environment and start defining the toolpaths. If you need to modify the part, switch back to the modeling environment and make the changes. When returning to the CAM side of Inventor or Fusion, the toolpaths update to reflect the changes made to the model.

The Setup

After switching to the CAM environment, the first step is creating a Setup. Depending on the complexity of the part, you may end up with multiple setups. Each one defines the type of operation (Milling, Turning, or Cutting), the XYZ directions, the zero position, as well as the stock size and shape.

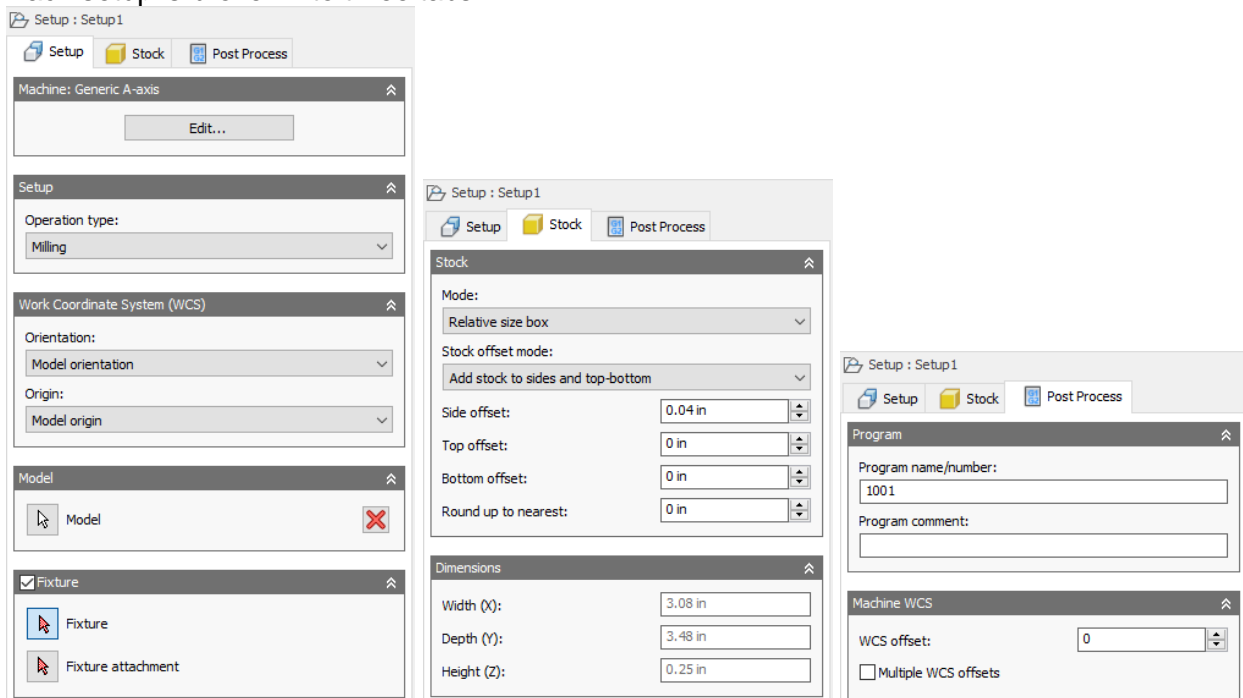
Any part that needs to flip will need multiple setups. The first part that I made that required multiple setups was a shield I pulled off GrabCAD. I made some modifications because some of the details were only mapped images.



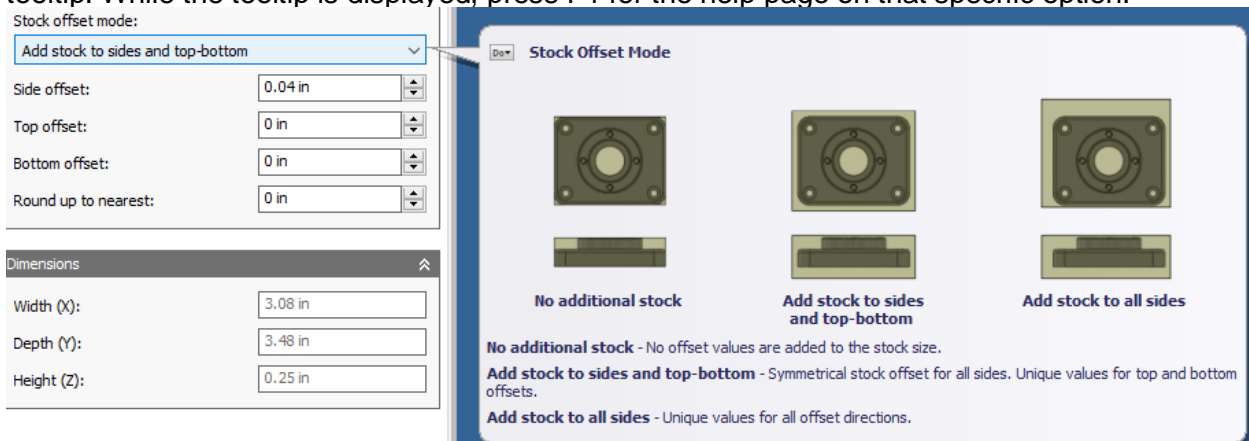
This was machined at only three inches high as a test on some scrap. I was still trying to see what the machine was capable of. But it does show multiple setups. Looking back, I should have finished each side with a Pencil operation to clean up the edges. But I was still learning Inventor's CAM operations at the time and only had one router bit.



Each setup is broken into three tabs.

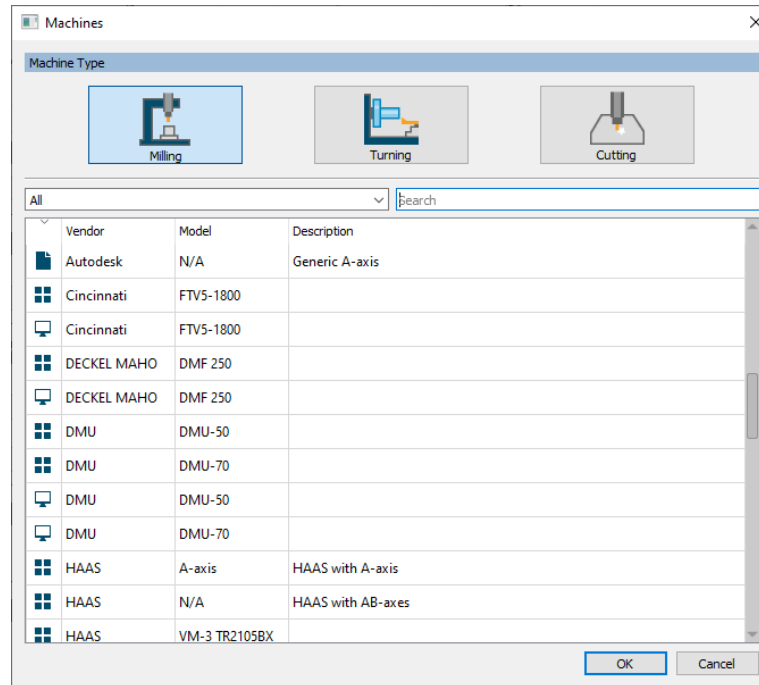


Each area will be briefly explained in this document. Having a good understanding of all the options available, will create better tool paths with fewer errors and frustrations later. If you see an option you are unsure of, hover over the option and a summary of information will appear a tooltip. While the tooltip is displayed, press F1 for the help page on that specific option.

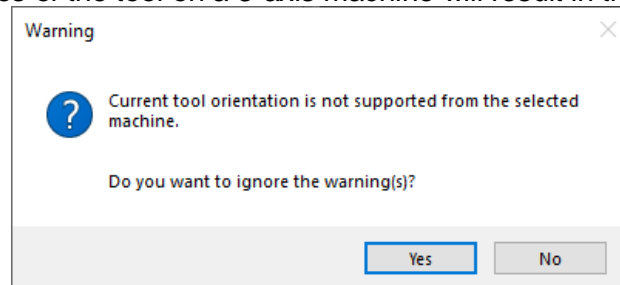


Machine - Setup

Allows selection of the machine to be used for this setup.



After selecting this, Inventor will warn when an operation is attempted that is outside of the selected machines capability. For example, attempting to change the axes of the tool on a 3-axis machine will result in this following warning.



Setup Type – Setup

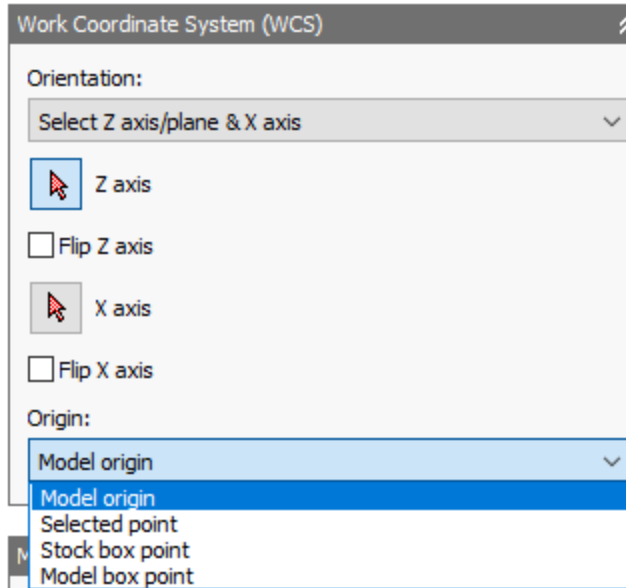
The type of setup is selected automatically when a machine is selected. If no machine is selected, the type of setup can be selected here. Other options in the setup panel are determined by the type of setup selected.

Work Coordinate System (WCS) – Setup

The default is Model Orientation. Typically, the model's orientation doesn't line up with where it makes sense to zero the setup stock before machining. The other orientation options in the drop-down changes what selections can be made on the model to manually define the Z axis and XY plane.

I have found that Select Z axis/plane & X axis works pretty much every time. Once the X and Z axis have been defined, select the origin point for the newly defined WCS. This can reference

the model or the corners of the stock box. In extreme cases, you can even place an Inventor work point to be used as the origin of the setups WCS.



Safe Z – Setup (Turning Only)

Sets the safe Z plane that the tool is to start, and return to, between operations.

Model – Setup

If only a single body or part is in use, no selection is needed. If an assembly or multi-body part is in use, use this to select the part to machine. All other parts will be ignored.

Fixture/Chuck – Setup

If the fixture is included in the assembly, select those parts. The toolpaths will watch for collisions when simulating the toolpaths.

Stock – Stock

This is where the stock dimensions and shape are defined. The default is relative to the model with offsets. There is also an option for selecting another part/solid if the starting stock has been modeled beforehand.

Dimensions – Stock

This is a read only section that displays the stock dimensions.

Program – Post Process

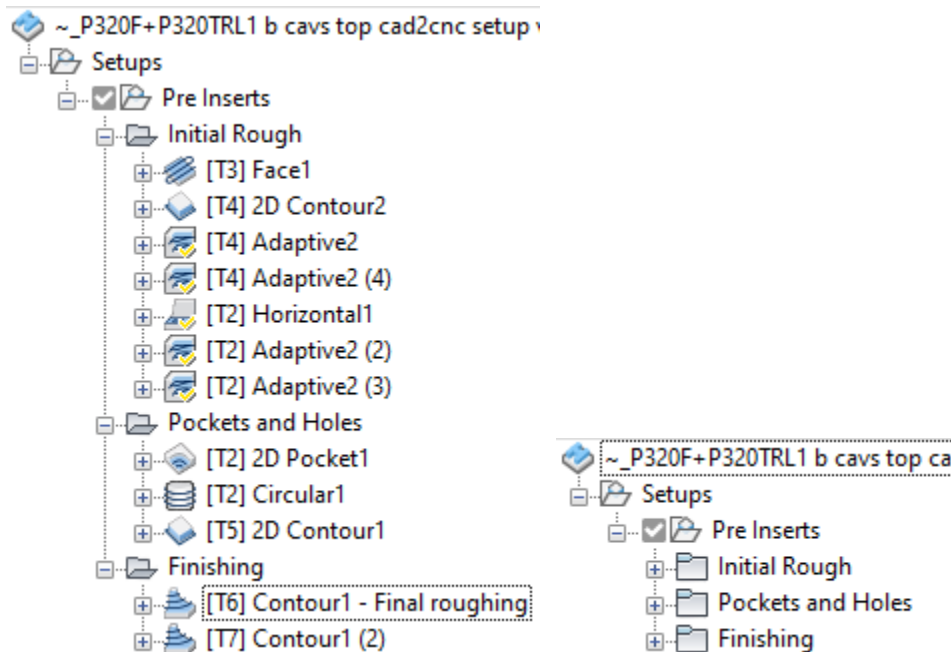
Entering a number/name and comment here will carry into the post setup sheet when they are generated.

Machine WCS – Post Process

If the CNC controller has several available fixture offsets, this identifies the workpiece WCS for the setup. 1 being the first available fixture offset.

If multiple fixtures are in place, check the Multiple WCS offsets box to create duplicates for each fixture.

Inside of each setup, folders can be created to better organize complex machining operations. Once the folders are created, simply drag any operations into the folders. If a folder is selected when creating a new operation, that operation will start in that folder.



Creating Operations

All the operations use the same five tabs. However, the contents of these tabs can drastically change between the various types of operations.

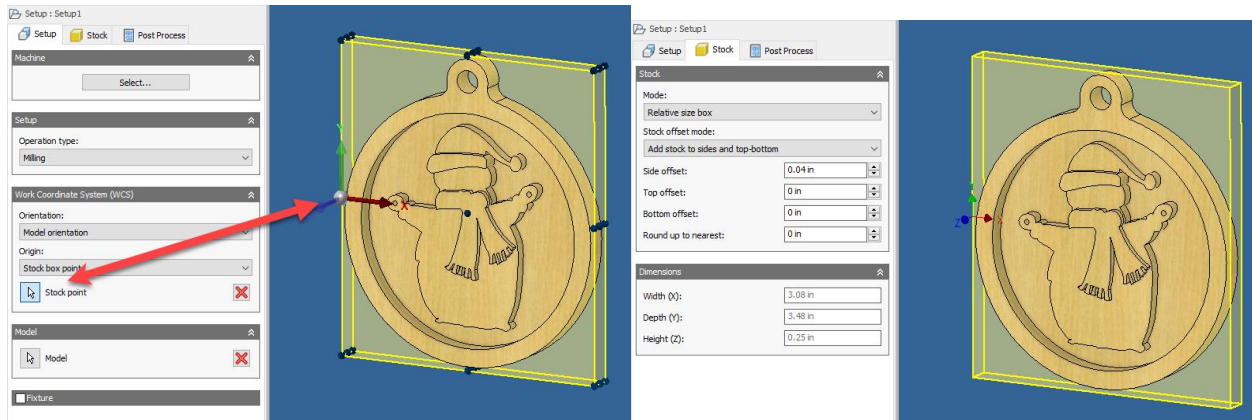


Rather than try to explain each of the options it might be better to walk through the process from Start to finish. I'll start with a few simple wooden parts and then move into some more complex metal ones.

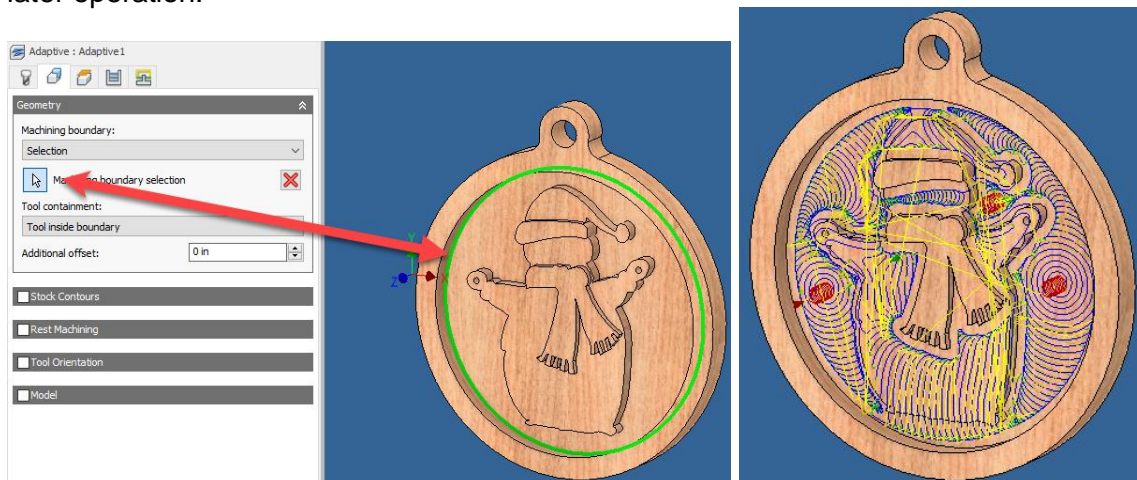
For the first part, I'll start with one of many Christmas ornaments I made last year.

First create a setup and select the WCS. In this case Z is in the proper direction but I did change the origin to the side instead of the middle. I modeled this to the thickness of some scrap 1/4" plywood. So, the Top and Bottom offset was changed to 0.

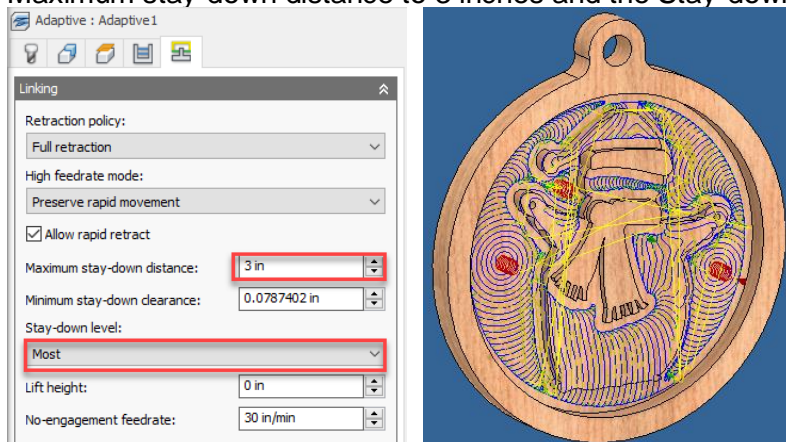




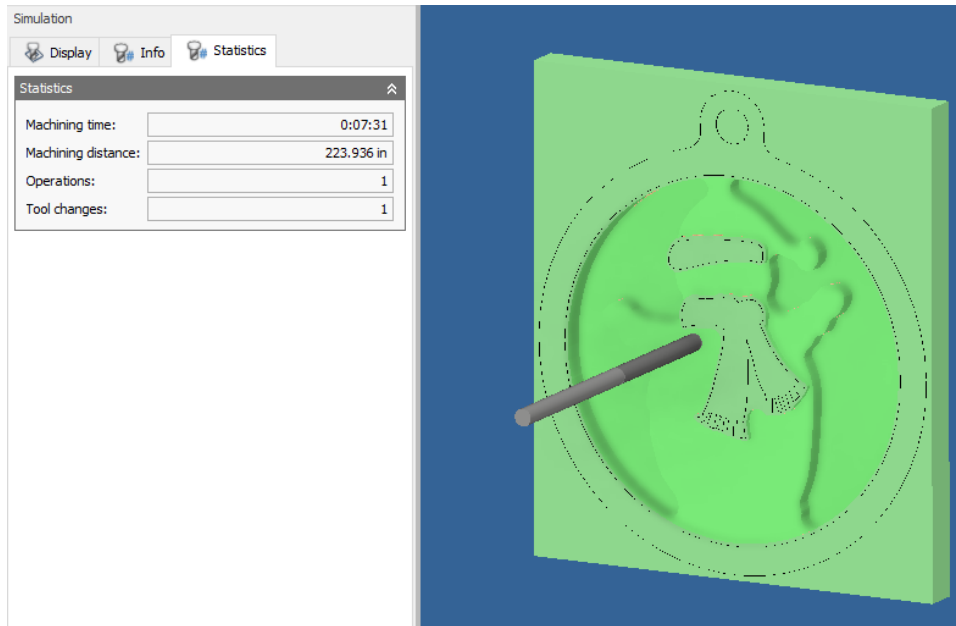
For the roughing pass, I started with the 3D Adaptive operation. After selecting a 1/8" flat end mill on the Tool tab, we move to the Geometry tab. Here I'm defining the machining boundary by selecting the inside circle. This will keep the tool inside the part and the I'll cut the profile in a later operation.



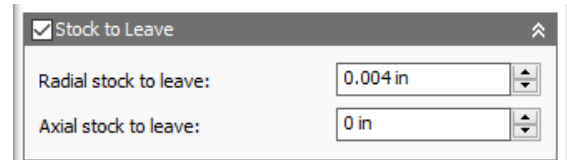
There is a lot of Z axis travel and that is the slowest axis on my machine. So, I changed the Maximum stay-down distance to 3 inches and the Stay-down level from Least to Most.



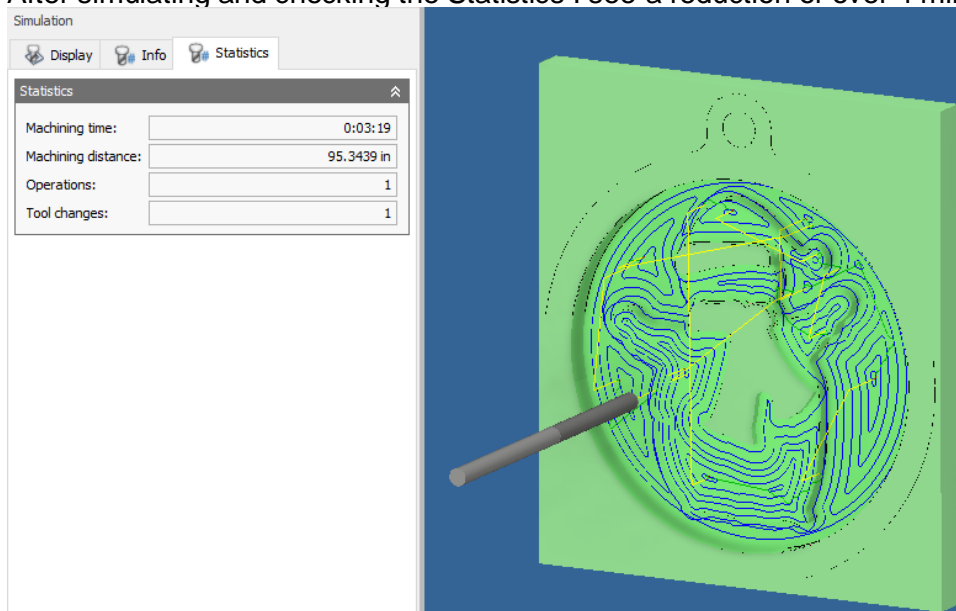
That made for a much less Z axis travel. When simulating the path, the Statistics tab shows a 7.5 minute run time.



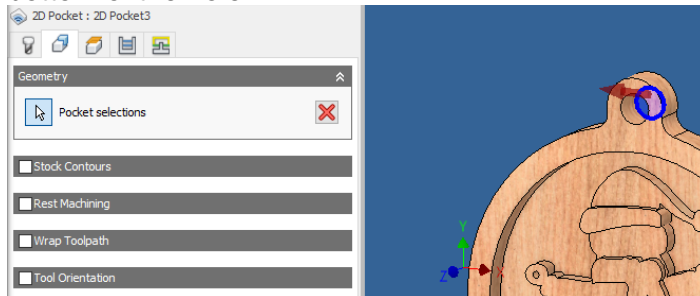
The Horizontal operation could do this as well. So, I tried the same tool and boundary with that operation. I should note that on stock to leave I left a bit on the radial to clean off later in a finishing operation. However, Axial was set to zero as I don't plan on going over the horizontal surfaces again. This same option is available on the Adaptive operation as well.



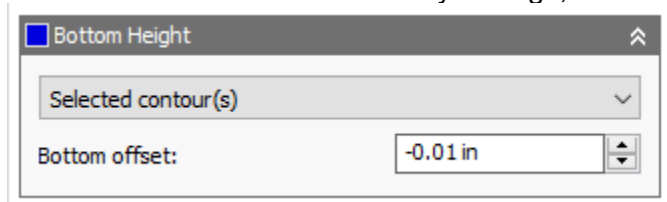
After simulating and checking the Statistics I see a reduction of over 4 minutes in run time.



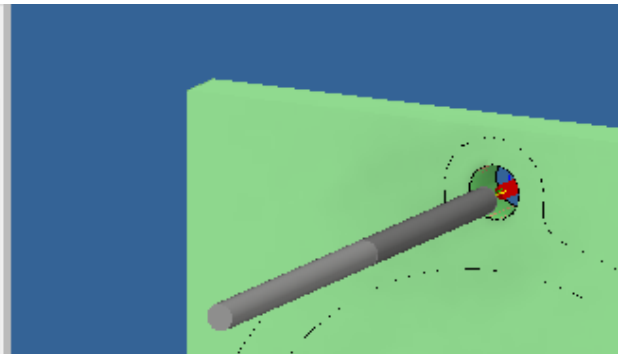
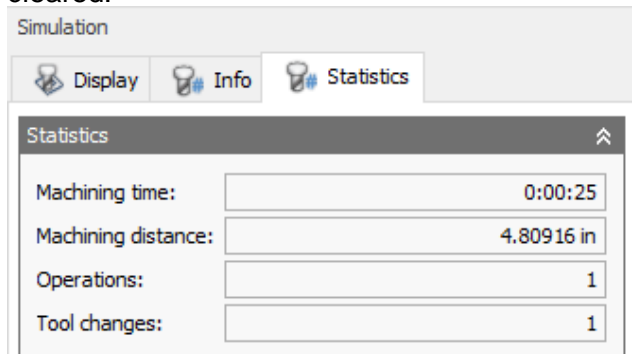
For the roughing, I'll settle for the Horizontal operation and move on. To mill the hole on the top, I selected a 2D Pocket operation using the same tool. For the Geometry selection, I selected the bottom of the hole.



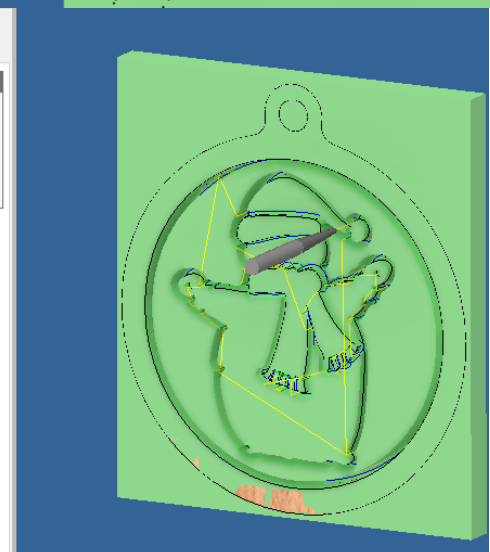
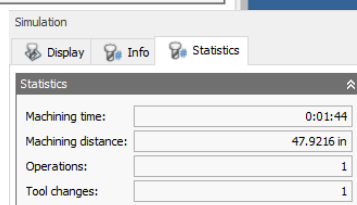
To ensure it will cut the whole way through, the Bottom Height was set to a negative value.



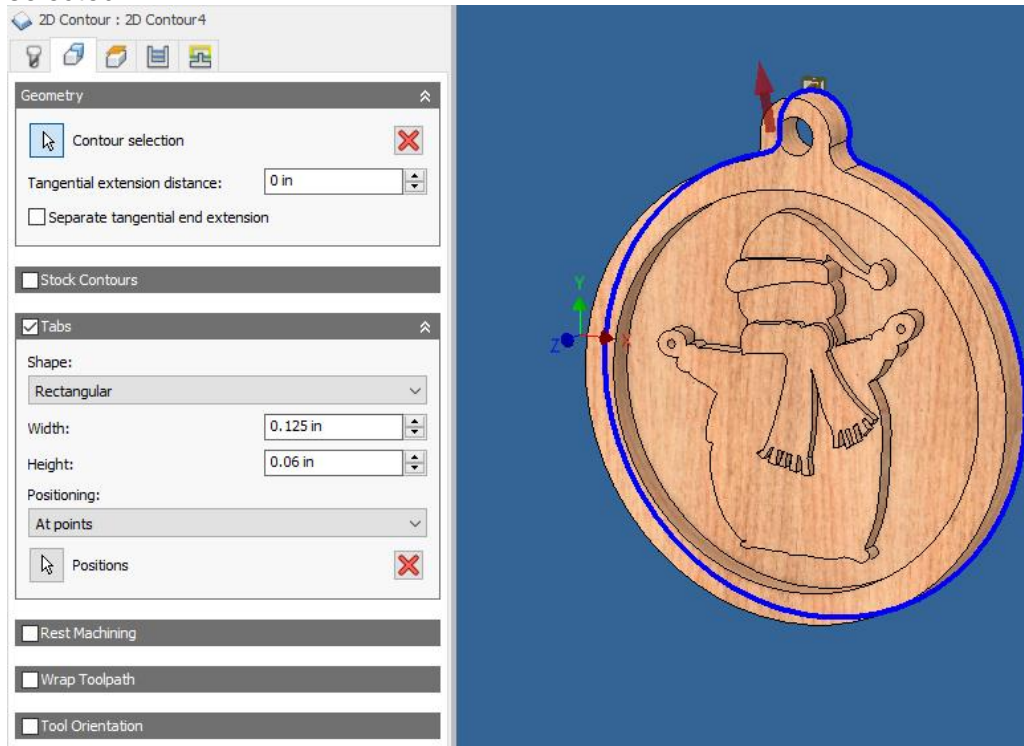
Stock to Leave is checked by default. There will be no finishing operation here so that was cleared.



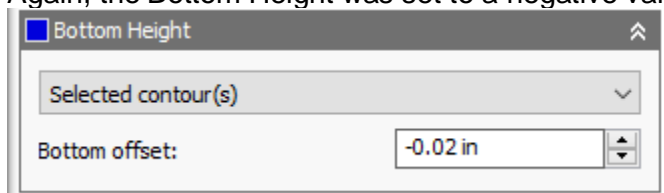
The only thing left is to clean up the details on the inside and then cut out the profile. For the clean-up, that brings us to the Pencil operation. This operation traces all tight inside edges and fillets that other tools can't reach. For the tool I'm using a 1mm bull nose. I selected the inside circle again as the Machining boundary. The same boundary used in the first operation. Everything else was left as default.



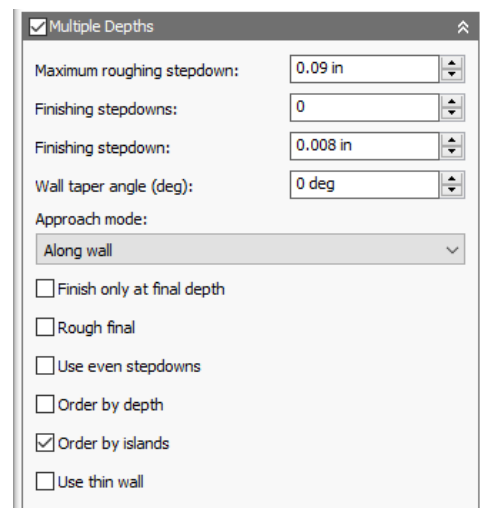
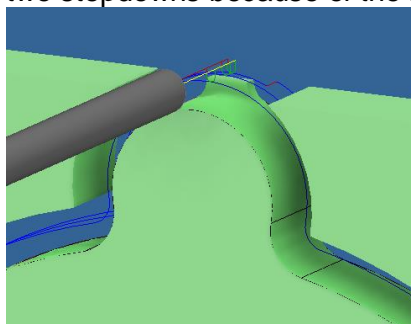
Finally, to cut out the profile. This is done with the 2D Contour operation. On the Tool tab the $\frac{1}{4}$ flat end mill was selected again. On the Geometry tab, two things were done. First the Contour selection. The outside profile was selected. And then a Tab was added to the top. This was sanded off by hand later. For the positioning of the tab, a point at the top of the model was selected.



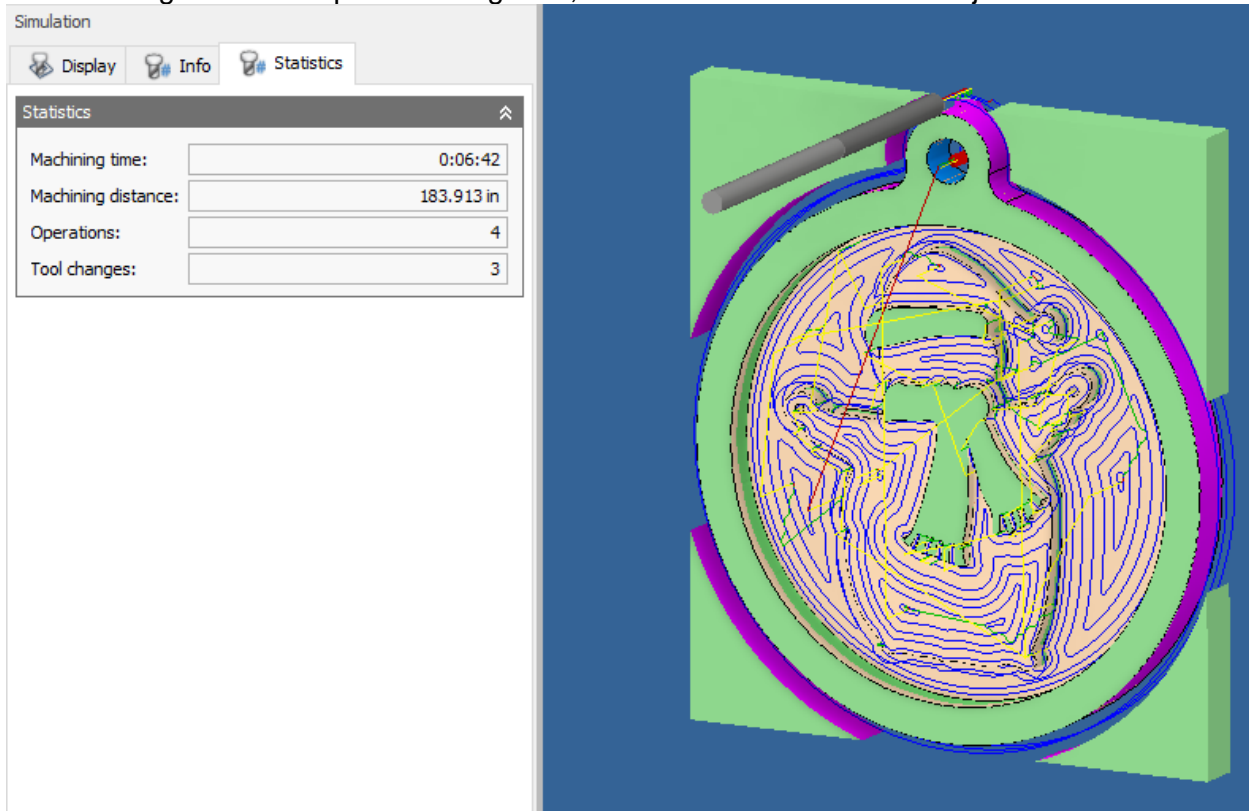
Again, the Bottom Height was set to a negative value.



On the Passes tab, Multiple Depths were used. I wanted two stepdowns because of the feed speeds used.

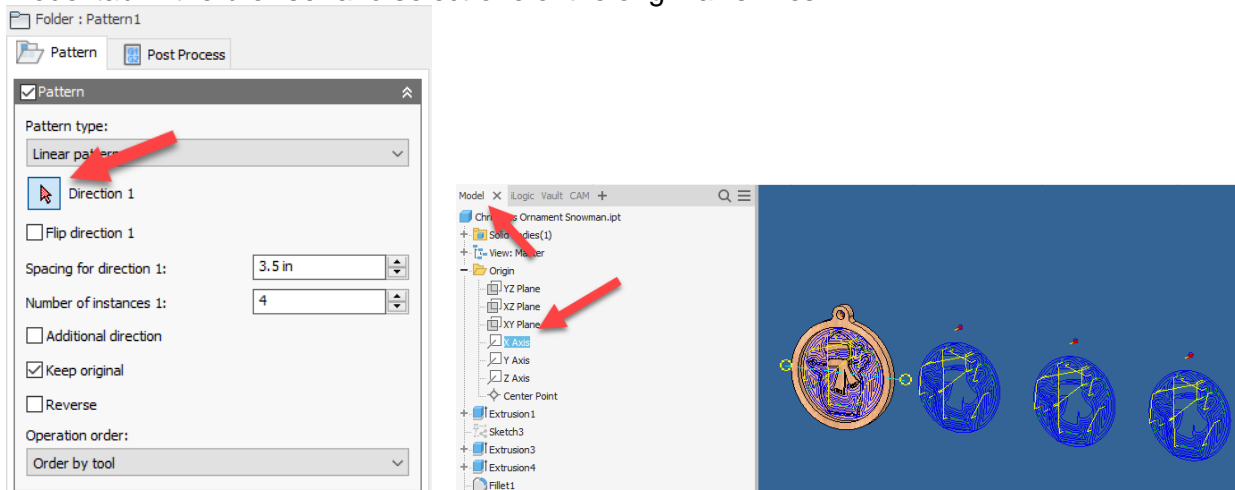


After running the all the operations together, I can see I have a run time of just under 7 minutes.

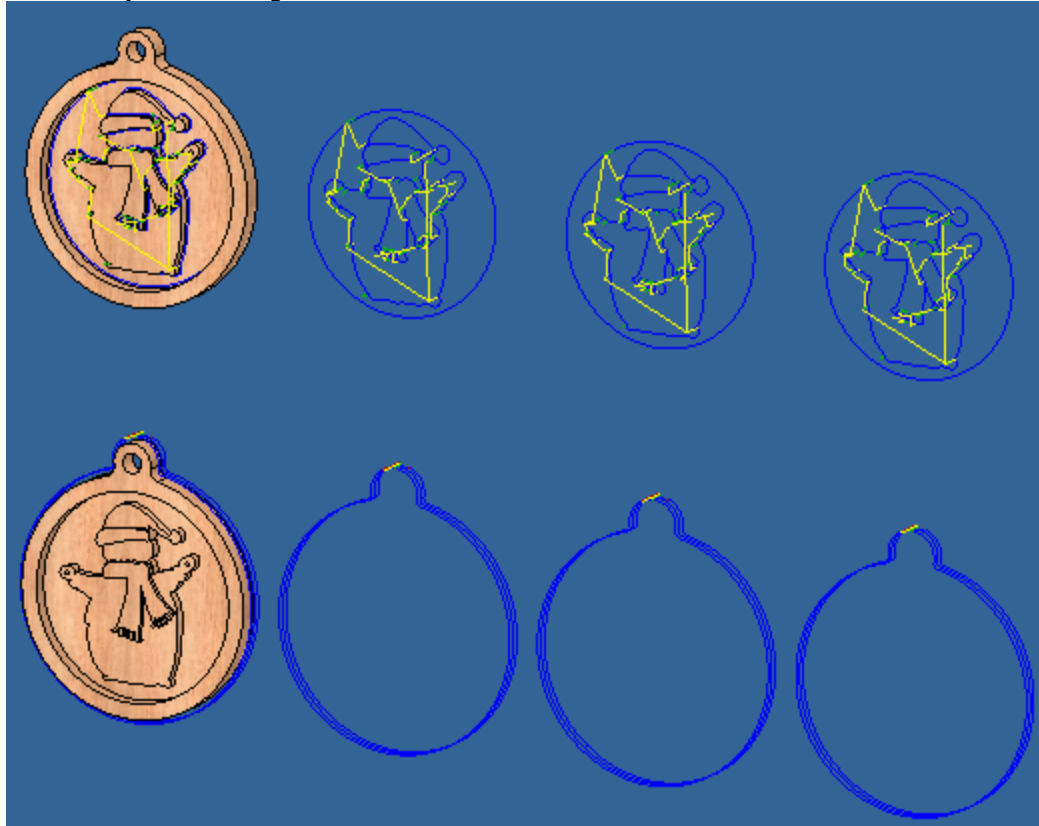


I really want four of these. Running these separate would require several tool changes for each part. I would rather run all four at once. That can be done with a Pattern.

Select the operations that need to be patterned and then start the Pattern command. Typically, the Direction is selected from model geometry. But in this case, I have nothing pointing in the direction needed on the model. So, after selecting the Direction 1 arrow, switch over to the Model tab in the browser and select one of the origin axis lines.



This was repeated for the Pencil operation and finally the 2D Contour operation due to the necessary tool changes.



Finally, the paths are ready for posting. Because my machine requires a manual tool change, I need to generate each tool as a separate file. Select the first pattern in the browser and start the Post Process command. Here the post, final file location, and program name are defined.

Once the file is built, the code is opened in the NC editor for review and final tweaks.

Post Process

Configuration Folder

C:\Users\Public\Documents\Autodesk\Inventor CAM\Posts

Setup

Post Configuration

Enter search text

All

All vendors

Acramatic / acramatic

Open config

Output folder

C:\Users\Stan\AppData\Local\Inventor CAM\nc

Open folder

NC extension

.nc

Program Settings

Program name or number

101

Program comment

Snowman - Roughing

Unit

Document unit

☐ Reorder to minimize tool changes

☒ Open NC file in editor

Property

Value

(Built-in) Allow helical moves

Yes

(Built-in) High feedrate mapping

Preserve rapi...

(Built-in) High feedrate

0

(Built-in) Maximum circular radius

1000

(Built-in) Minimum chord length

0.25

(Built-in) Minimum circular radius

0.01

(Built-in) Tolerance

0.002

Optional stop

Yes

Preload tool

Yes

Separate words with space

Yes

Search for posts in our Autodesk HSM post library

Post

Cancel

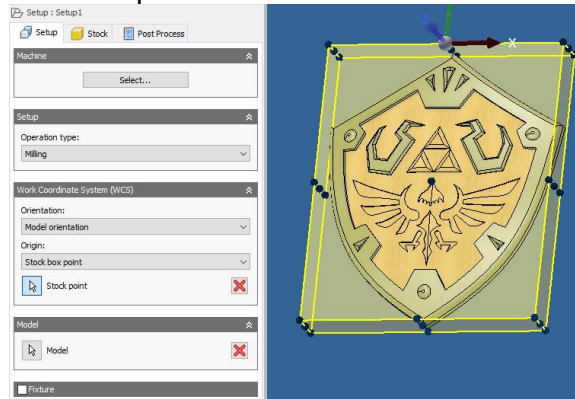

```

1 : (PGM, NAME="1001")
2 ; T4 D=0.125 CR=0 - ZMIN=-0.27 - FLAT END MILL
3 ; T11 D=0.0394 CR=0.0197 TAPER=3.7DEG - ZMIN=-0
4 : G90 G40 G94
5 G17
6 G70
7 M26
8 ; HORIZONTAL2
9 M9
10 M26
11 :T4 M6
12 M26
13 :T11
14 S30000 M3
15 H0
16 M8
17 G0 X1.0929 Y-0.2731
18 Z0.2
19 Z-0.0206
20 G1 Z-0.0465 F30
21 X1.0928 Y-0.2732 Z-0.0479 F25
22 Y-0.2734 Z-0.0492
23 X1.0926 Y-0.2737 Z-0.0505
24 X1.0925 Y-0.2742 Z-0.0517
25 X1.0923 Y-0.2748 Z-0.0529
26 X1.092 Y-0.2755 Z-0.054
27 X1.0917 Y-0.2763 Z-0.0551
28 X1.0914 Y-0.2772 Z-0.056
29 X1.0911 Y-0.2782 Z-0.0568
30 X1.0908 Y-0.2793 Z-0.0575
31 X1.0905 Y-0.2805 Z-0.0581
32 X1.0902 Y-0.2817 Z-0.0585
33 X1.09 Y-0.283 Z-0.0588
34 X1.0897 Y-0.2843 Z-0.059
35 G17 G3 X1.0889 Y-0.2985 Z-0.06 I1.1481 J-0.2948
36 G2 Y-0.3275 I0.86 J-0.3129 F30
37 G3 X1.1125 Y-0.332 I1.1009 J-0.3283
38 X1.1218 Y-0.3008 I0.015 J0.0134

```

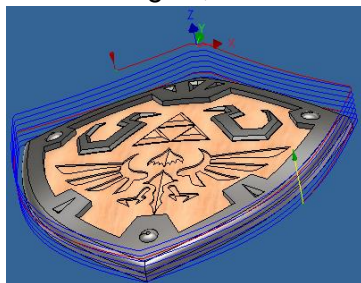
If necessary, setup sheets can be created for each setup or operation as well.

On to the next part. This part has machining on the front and back. That will require two setups. I drilled a small hole through the stock to help zero the second setup.



For roughing, a 3D adaptive operation was used. The Machining boundary was set to silhouette so that the rest of the stock was left alone. The Bottom Height was moved up to ensure the profile will not cut out completely. Both Radial and Axial stock was left for later finishing passes.

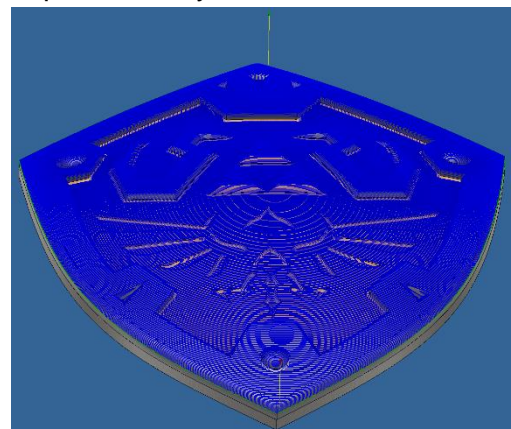
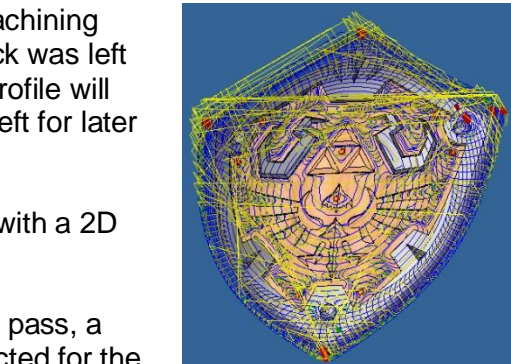
While the large tool was still in place, the profile was cut with a 2D Contour. Again, the Bottom Height was moved up.

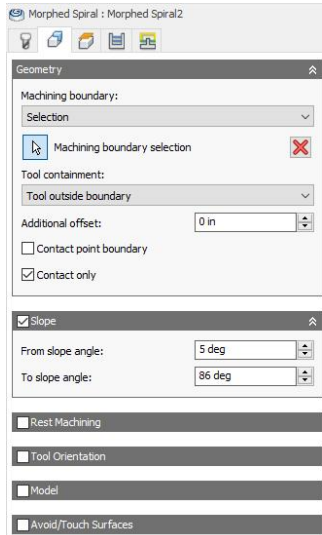


Finally, for the first finishing pass, a Parallel operation was selected for the gradual curves on the inside. The boundary was set to Selection and the outside edge was selected. With containment defaulting to Tool Center On Boundary, the path will stay inside and not roll around the outside. On the Passes tab, the stepover was changed to

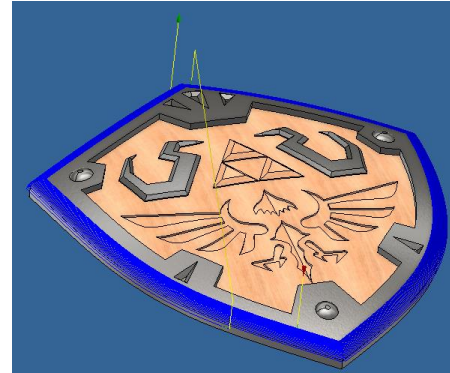
a much smaller amount.

To clean up the outside fillet, a Morphed Spiral was selected. The boundary selected was the outside, and inside, edge of the surrounding fillet. Slope was also enabled to limit the path from running down past the fillet. I played around with the To Slope Angle until the path stopped dropping over the side. 86 degrees ended up being the magic number.

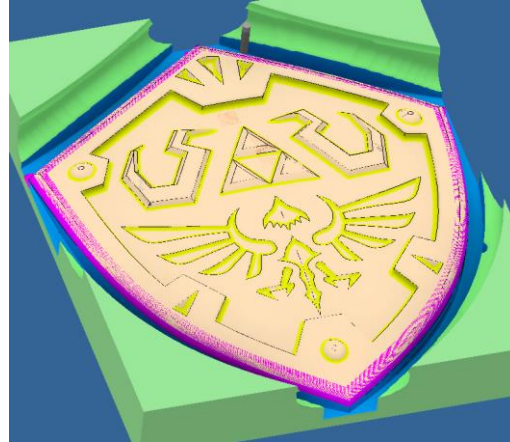
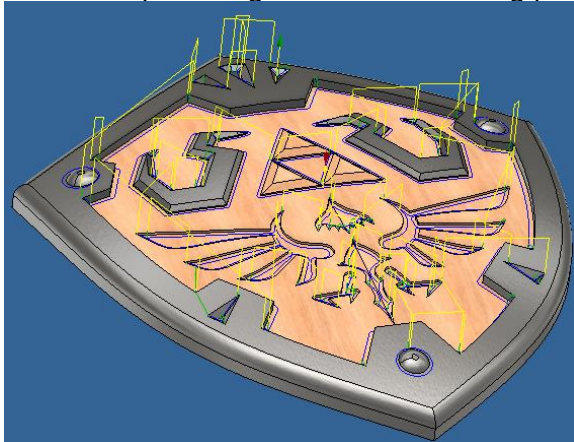




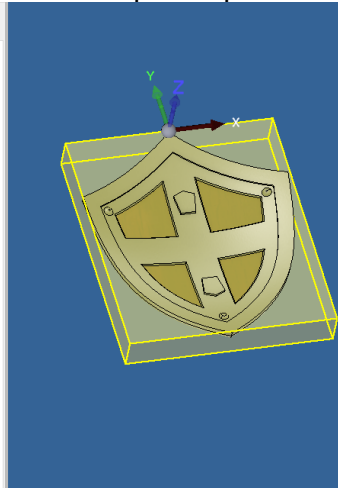
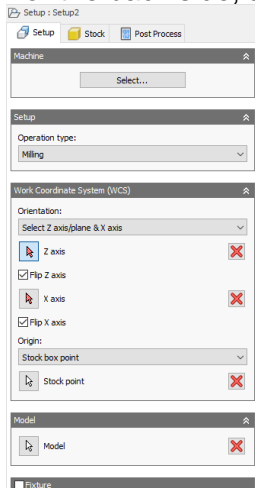
Finally, on the Passes tab, the Stepover was changed to the same value as the first finishing pass used in the Parallel operation above.



To clean up the edges after the finishing passes, the Pencil operation was used.



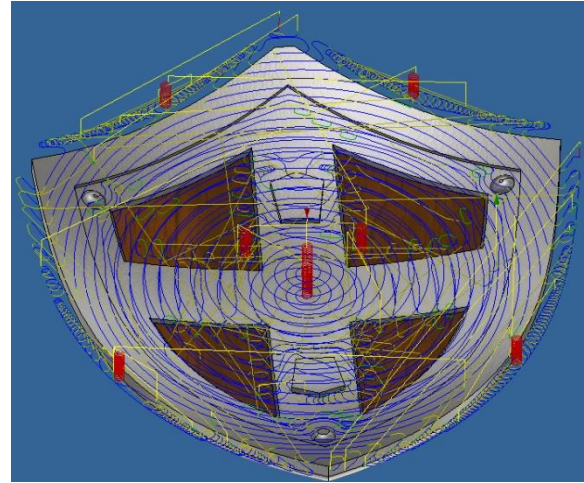
For the back side, a new setup is required.



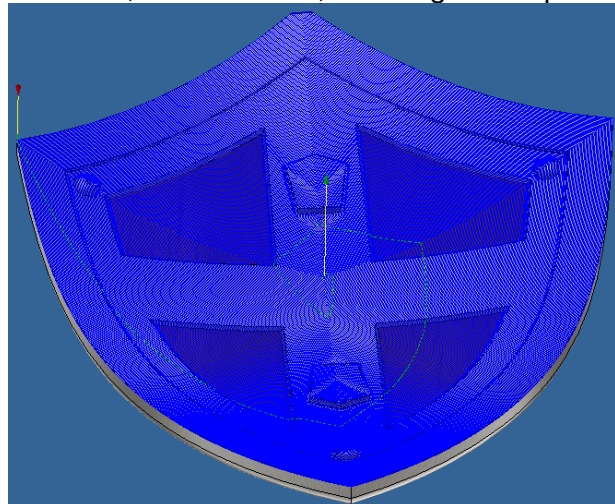
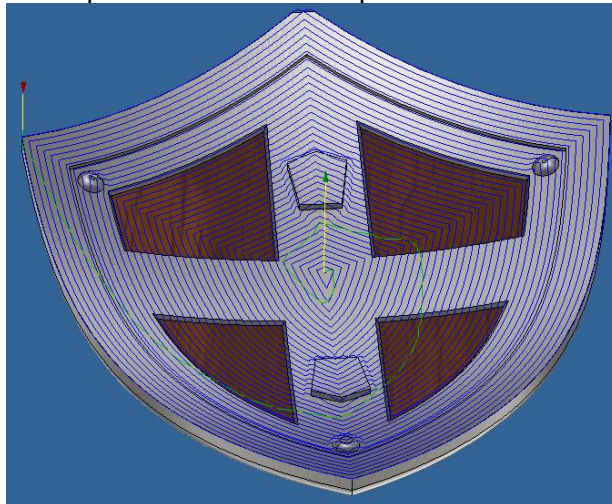
The WCS orientation needs to be completely flipped around from the models UCS. After moving the origin to one of the back side stock points, the Z and X axis was flipped.

For the roughing operation, Silhouette was selected as the boundary with containment set to Tool outside boundary. Stock to leave was set to leave some Radial and Axial material for a finishing pass later.

A second roughing pass with a small diameter tool as was used. Because of the gradual slope in the part, a Scallop operation was used. Silhouette was used as the boundary. Only a slight bit of material was left in the Stock to leave for a final finishing pass. The maximum stay down was increased to keep the tool down for the entire pass.

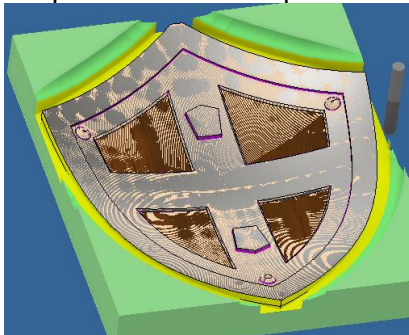


This operation was then duplicated with no stock to leave, a smaller tool, and a tighter stepover.



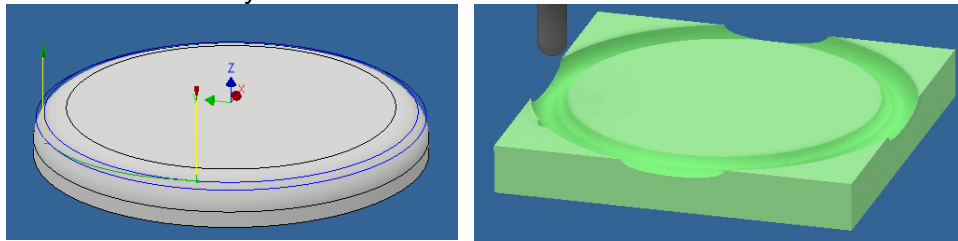
Pencil was used on the back side for the sharp corners.

Finally, the 2D Contour was used to finish cutting out the part from the stock. Again, the bottom of part was moved up.



Knowing what options exist, and what each one does, is key to successful and efficient paths.

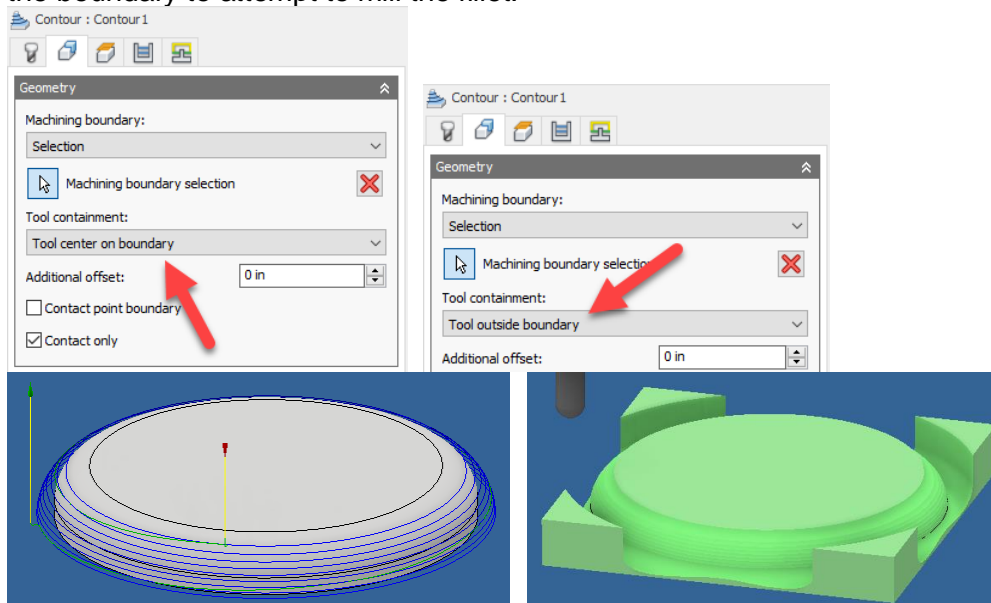
For a simple example of this options type trouble shooting, here is a simple profile that represents an actual question recently e-mailed to us. This is a simple $\frac{1}{4}$ " fillet that the user wanted to create with a $\frac{1}{4}$ " ball end. However, the path only seems to be going around twice and not the full way down the fillet.



This is quite simple when you understand the options but can be quite frustrating to a new user.

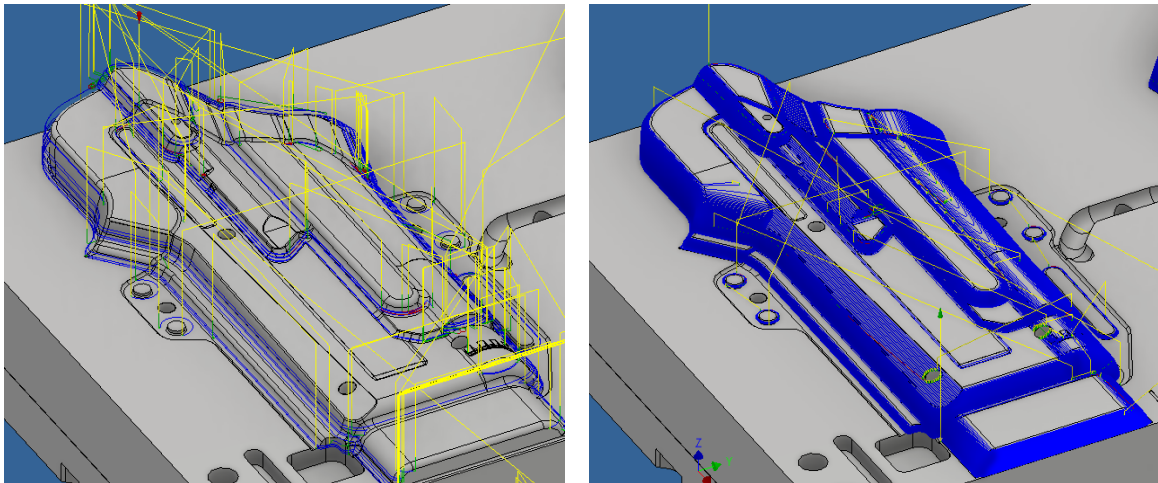
The problem this user is experiencing is not really a problem, the tool containment is set to "Tool center on boundary" by default and so the center of the tool cannot go outside the selected boundary. Which in this case was the inside and outside of the fillet.

The user is expecting the tool to cut the entire fillet but because of the containment option, the tool is unable to go outside the boundary far enough for the ball to mill the bottom of the fillet. The fix... Change the Tool Containment to Tool Outside Boundary. Now the entire tool can leave the boundary to attempt to mill the fillet.

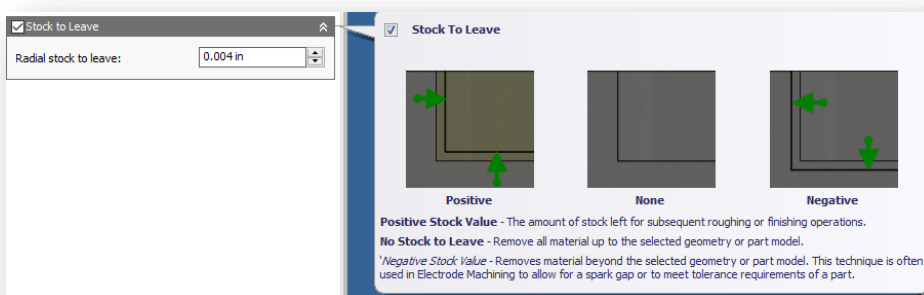
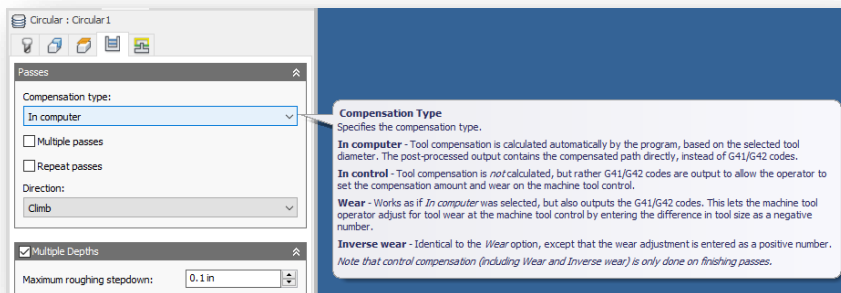


For one final example, here the Slope operation was used with excluded surfaces, bottom height, and boundaries were used to greatly modify the same Contour operation. First a final roughing pass only touching between 30-90 degrees with a fairly large step down as well as some Stock to Leave.

And then the final finishing pass was the same Contour operation but with different options set to slope, stepdown, and no stock to leave.



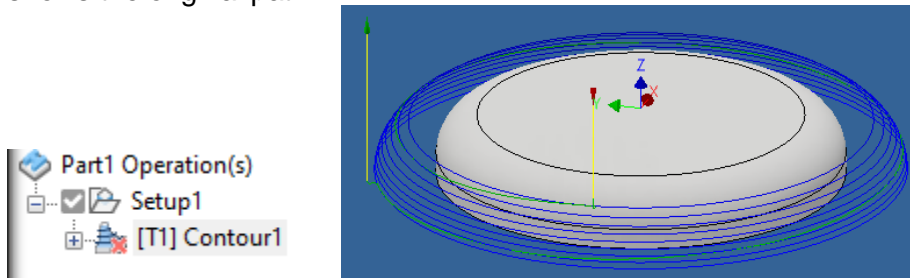
The help is great in helping to get familiar with the various options in each operation. Take the time to review them and play with each to see the differences that are made as each option is tweaked.



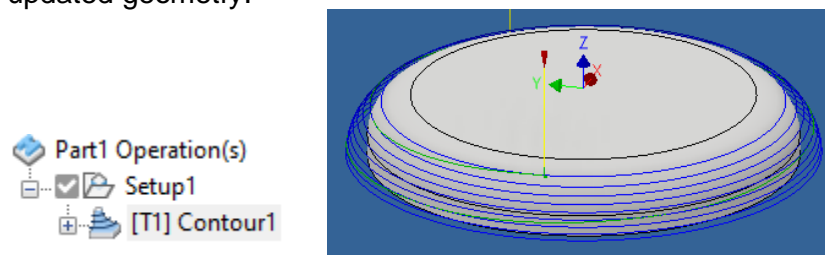
For even more help, press F1 while the tool tip is showing, and a webpage will open to the help of that operations settings and options.

Finally, I want to point out the since this is all happening inside of inventor, should the model ever change, the paths simply need to be regenerated. There is no need to start over recreate

all the operations. If you don't want the paths to be regenerated, each operation can be locked to keep it from ever updating. Going back to one of the previous examples, here the circle was changed from 3" to 2.5". The browser shows the toolpath is out of date and the preview still shows the original path.



Right clicking on a setup, or single operation, and selecting Generate Toolpath will cause the operations to build new toolpaths, using the same selections and options, based on the new updated geometry.



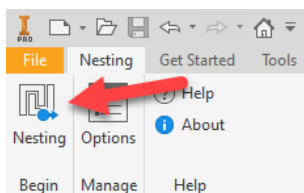
Introduction to Inventor Nesting

Inventor's Nesting environment quickly runs through an assembly, or collection of parts/2D profiles, and creates nests that can be saved out for a laser, waterjet, or turret punch type machine. I have even used this in some of my own woodworking projects to get the best use of plywood sheets that I then manually cut down.

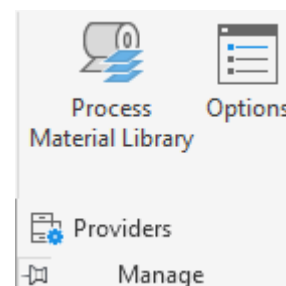
First you should know what files can be used.

- IPT: These can be:
 - Sheet metal files containing flat patterns, a simple sketch, or an extrusion
 - Inventor solids that have a planar (flat) face
 - Inventor Composites that have a planar (flat) face
 - Non-native files (such as STL, STEP, and so on) saved as IPT; these must also have a planar (flat) face
 - Non-native files are supported as of the 2019.2 Update.
- IAM: These must reference at least one supported IPT.
- DXF: Most DXF files will work, as long as they define 2D closed shapes on standard layers.
- DWG: These must also define 2D closed shapes on standard layers. Avoid using DWG-specific features, such as Paperspace.

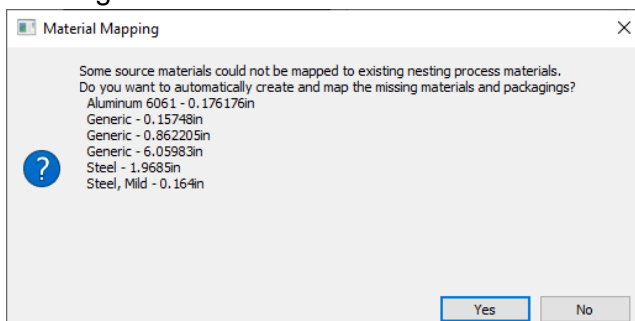
To setup a template and predetermine what types of files are nested, start the Nesting environment. From the Home screen switch to the Nesting tab and start a nest.



Here setup the materials to be processed for this future template using the Process Material Library. The Materials and thicknesses need to match the material names and thicknesses used in the parts to be nested.



If the materials are not setup a head of time, you will see this when creating a nest.



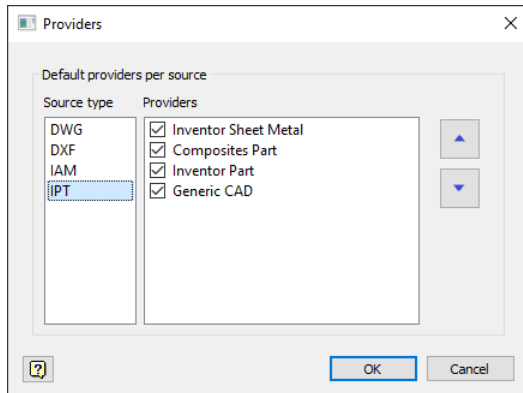
When this happens, generic materials and packages are created for each material and thickness found.

Each will need to be reviewed and the proper packages (overall nesting sheet sizes and grain direction) will need to be defined.

Review the Options and make any changes desired. The settings under the Options consist of display colors, extract\export

layers. There are several other options there to review as well.

Finally, from the drop down of the Manage panel, select the providers this template should look for when creating a nest.



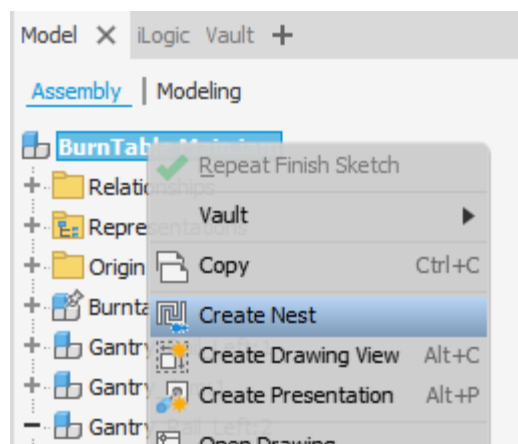
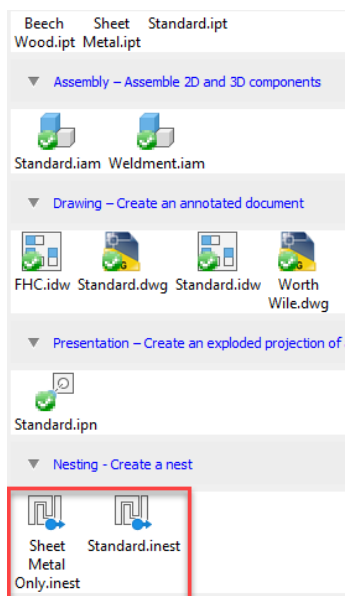
Then from the File drop down Save as Template.

Note that this template needs to be in the default Public location as well as your default template location. Some of the commands only reference the default template location and not the location the project file calls out as the template folder.

I would imagine this will be fixed in a future release, but for now place a duplicate copy in the public folder...

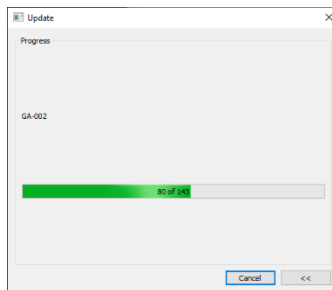
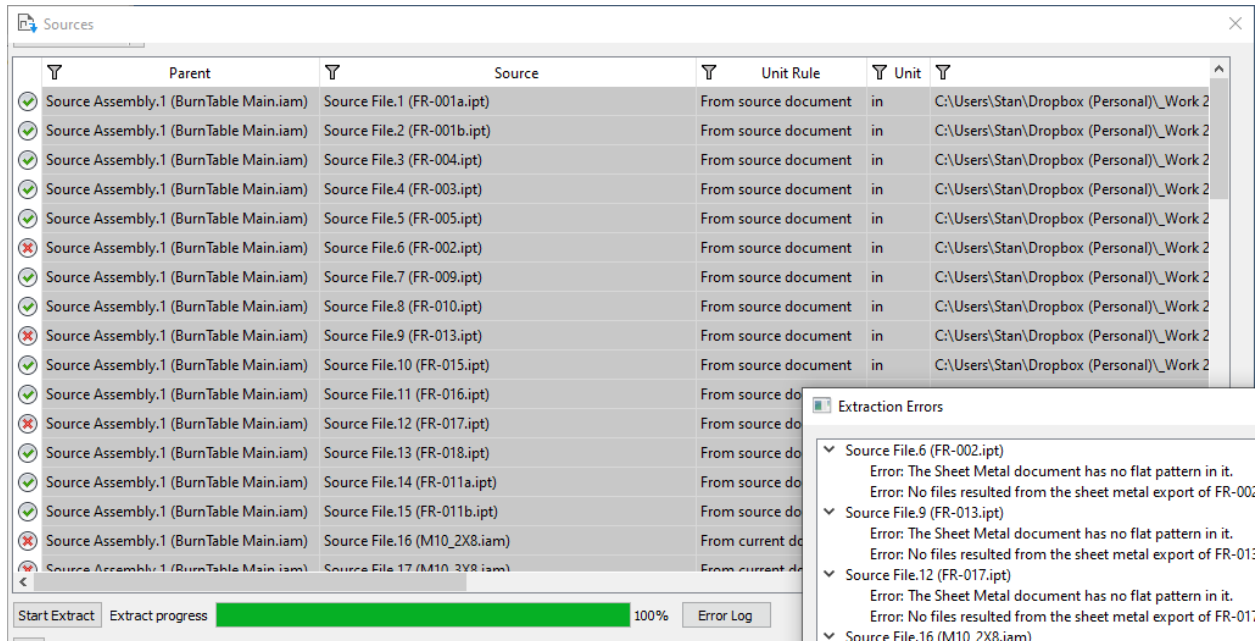
C:\Users\Public\Documents\Autodesk\Inventor 2020\Templates\en-US

There are multiple ways to start a new nest. From the New File dialog select one of the templates that ends in .inest. Or from an existing assembly, right click on the top browser node and Create Nest. *This second option is the one that pulls templates from the Public location mentioned above.*



When creating from an assembly, it will immediately start pulling in the files allowed by the selected templates providers list. If you start a brand-new nest from a template, the second step will be to select the source files using the Sources button. Once the sources have been all selected, press Start Extract. When starting from an assembly, you are taken to this point automatically.

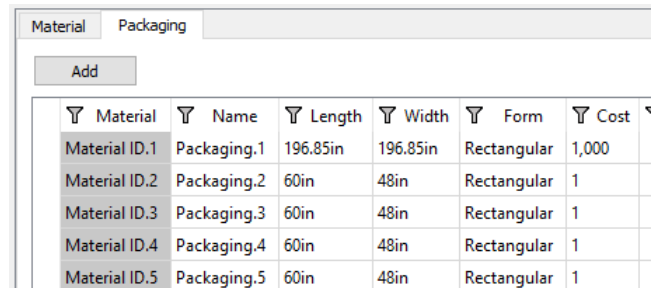
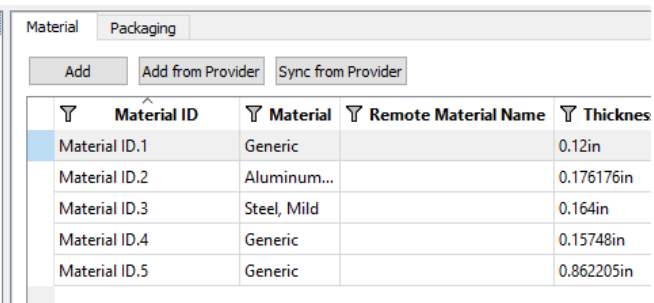
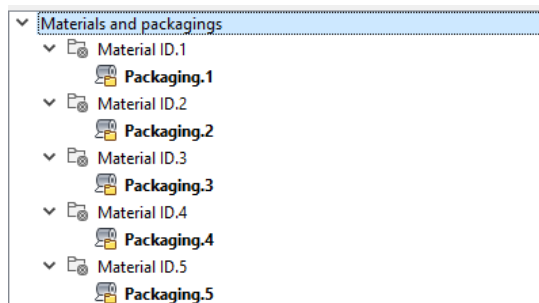
Some files may show red X's and did not import. Press the Error Log button to see what problems were encountered and caused the reason for each exclusion.



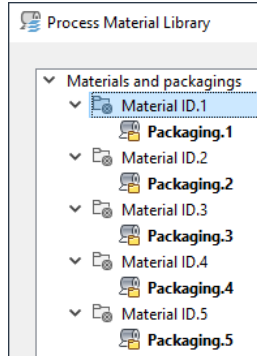
Pressing OK starts the process of creating all of the 3D solids the nests will be built from.

These shapes are all referenced under the Geometry Sources in the browser.

From the Process Material Library, review the materials and packaging. Each of these can be renamed and modified as needed.



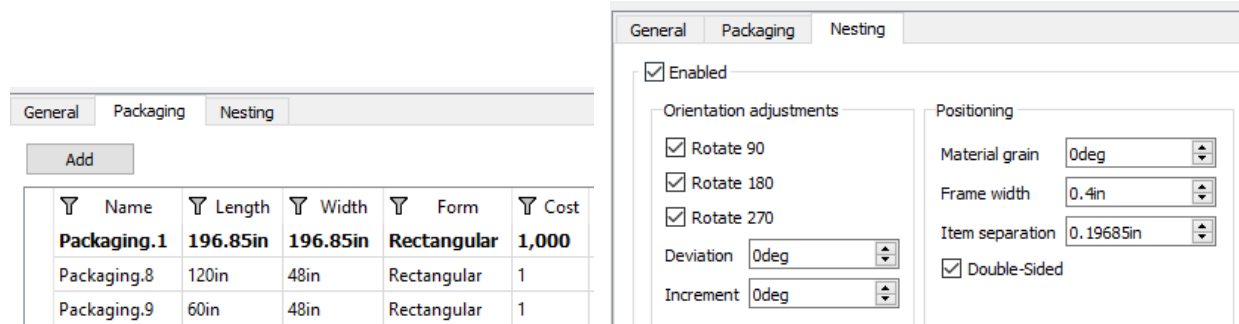
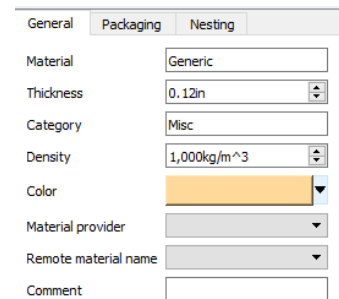
Selecting each material gives even more options.



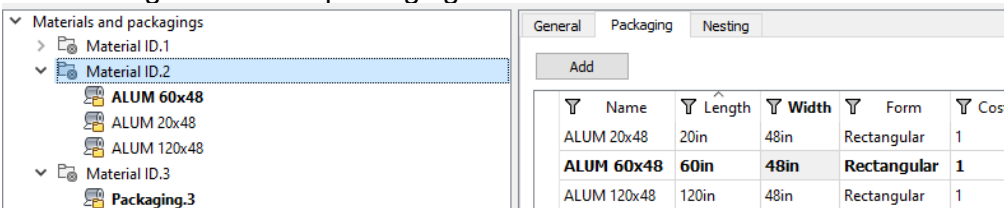
General is general information that will carry forward into the nests and reports.

Packaging allows the creation of additional sheet sizes to be used when calculating the nesting study. Select a row and then right click on and select Set as Default Packaging. This will automatically include it in future nesting studies.

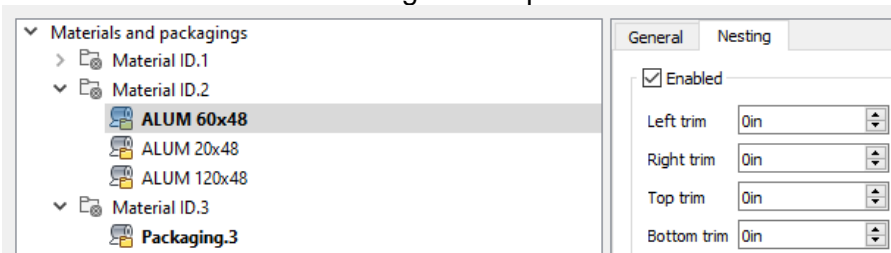
Finally, the Nesting tab allows control of the rotation and material grain direction for each material.



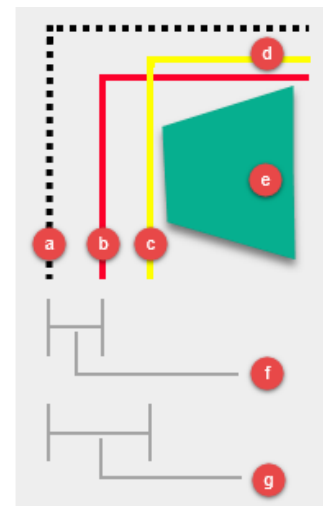
After adding the available packaging sizes...



select each one and set the edge Trim options.

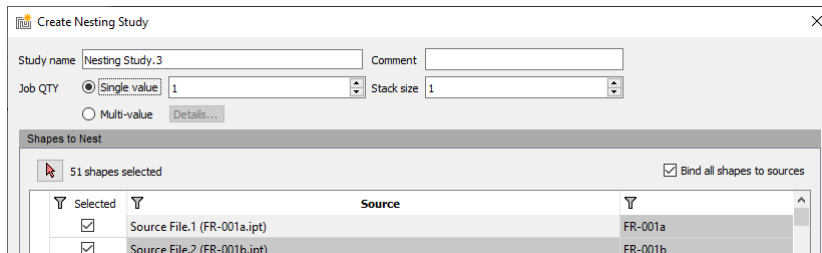


- Sheet perimeter
- Frame line
- Left trim line
- Top trim line
- Nested shape
- Frame value
- Left trim value

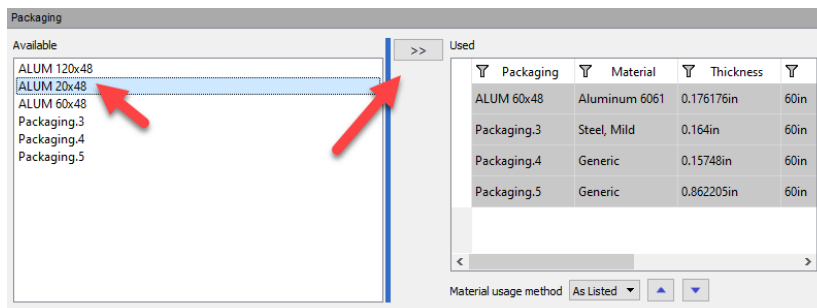


Most of the Materials and Packaging should be setup in a template at some point. You don't want to be creating materials and packages over and over.

From the Nesting tab press Create to create a nesting study. At the top of the dialog, the study can be named, and the quantity of parts needed can be changed. If the study is made up of multiple sources, use the Multi-value option to set different options for each source.



In the Packaging section of the dialog, the default packages are automatically pushed over to the Used section on the right side. Any other package sizes you wish to use can be manually pushed over at this time. In my example above, I created an ALUM 20x48 because I have three pieces of 20x48 that I wish to use up. The 20x48 package can be selected and pushed to the right. Then move it to the top so it is used first in the study.

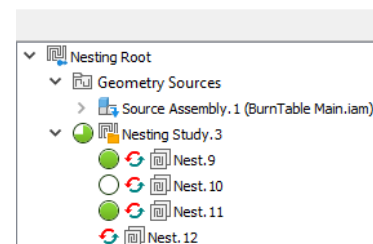


If the right-side section is scrolled all the way to the right, you will find the QTY of each package that can be used in the study. They are all set to infinity by default.

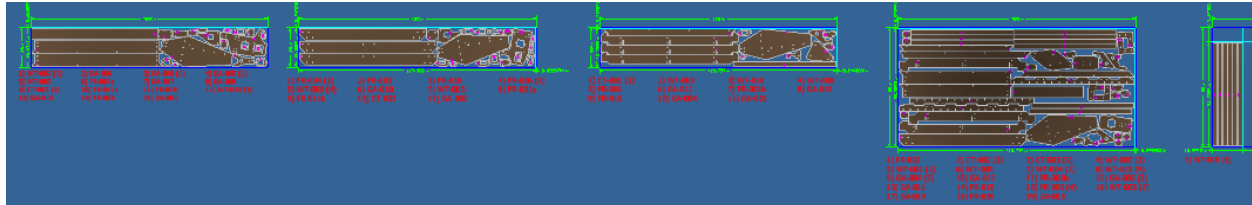
Used				Used			
Y	Packaging	Y	Material	Y	Thickness	Y	
	ALUM 20x48		Aluminum 6061		0.176176in		20in
	ALUM 60x48		Aluminum 6061		0.176176in		60in
	Packaging.3		Steel, Mild		0.164in		60in
	Packaging.4		Generic		0.15748in		60in
	Packaging.5		Generic		0.862205in		60in

ess	Y	Length	Y	Width	Y	Inventory QTY
		20in		48in		∞ 3
		60in		48in		∞
		60in		48in		∞
		60in		48in		∞
		60in		48in		∞

Press OK and the different materials will separate into their own nests. Be patient, this calculation can take some time.

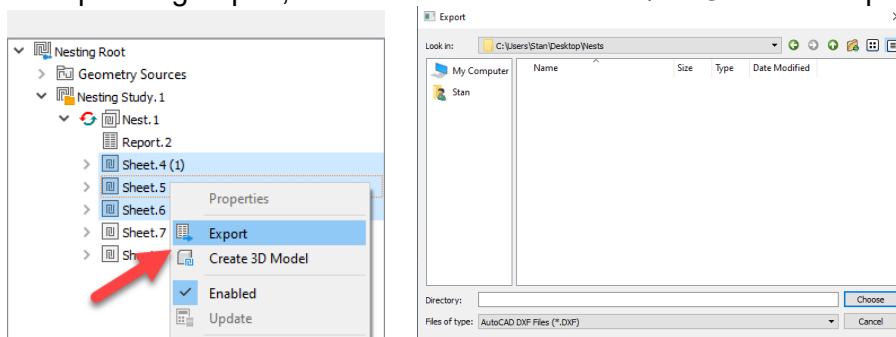


And now we see the results. The three 20x48 sheets were used first and then it went to the 48x60 sheets. If something wouldn't have fit on the 48x60 the 48x120 sheet would have been used.

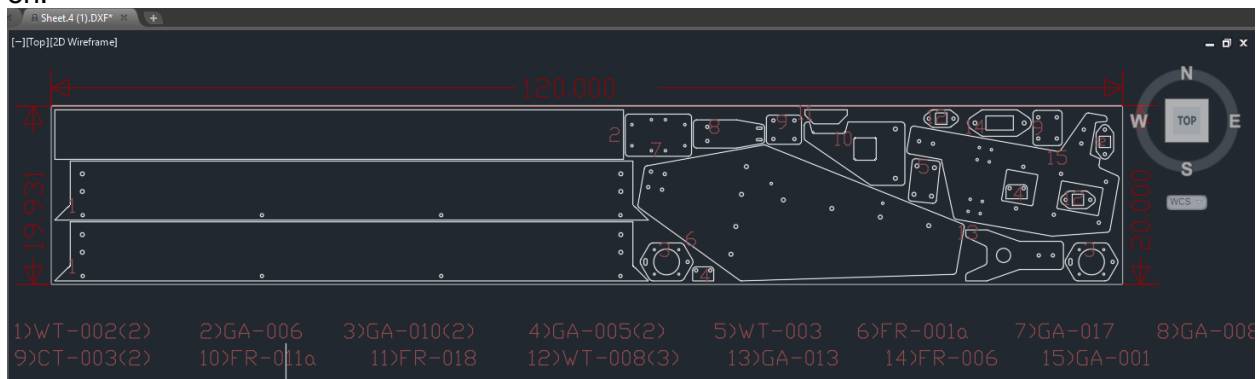


The generated sheets can be renamed in the browser. These sheet names will become the DXF/DWG names when they are exported.

The final sheets can be exported individually, or in batches. Right click on them and select Export. Or press the Export button on the ribbon after selecting the sheets to export. After pressing Export, select the folder the DXF's/DWG's are to export to.



You can then select one of the Export configurations available in the template. These settings control what is exported, how it is exported, and what AutoCAD layers the objects are placed on.



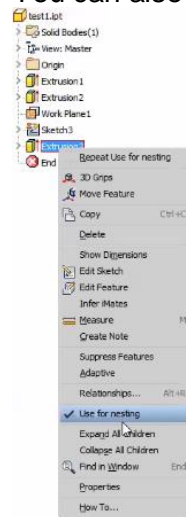
So that is the basic process for creating a nest. Let's finish this by discussing the various ways a source file is brought into this nesting environment.

1. DWG/DXF's will work if they have closed 2D profiles.
2. IAM's must have at least one supported IPT.
3. IPT's use these rules in the following order.

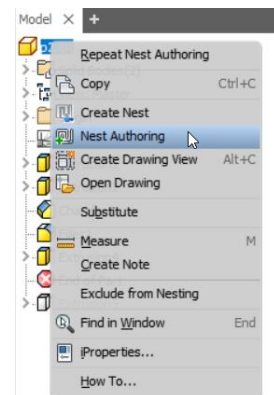
- The part is a sheet metal part with a flat pattern, simple sketch, or an extrusion. Found in that order.
- Inventor solids that have a planar face. Nesting will determine the projection direction normal to that flat face.
- By default, when you bring this IPT file into Inventor Nesting, its shape is interpreted for nesting following this order of rules in the model browser.
 - The last unconsumed (not used by an extrusion or revolve feature) sketch is used, provided it is the last node in the tree and is visible, or;
 - The first extrusion in the model is used.

Objects that follow End of Part in the model browser are ignored.

You can also right-click on any node in the browser and select Use for nesting.



When you nest an extrusion, Inventor Nesting detects its thickness from the extrusion referenced for the profile selection. That is if the extrusion was created with a parameter. If To Next or To was used, Inventor Nesting will not be able to detect the thickness of the extrusion. Similarly, when you nest a sketch, no thickness value is found. When no thickness value is available, a default value of 0.12 inches is assigned. Alternatively, you can assign a thickness value for Inventor Nesting by right clicking on the top browser node and selecting Nest Authoring. After entering a value in the Thickness dialog that opens, a parameter is added to named NestingThickness.



Parameter Name	Consumed by	Unit/Type	Equation
Model Parameters			
d0	Extrusion1	in	0.375 in
d1	Extrusion1	deg	0.0 deg
d2	Extrusion2	in	0.375 in
d3	Extrusion2	deg	0.0 deg
NestingThickness		in	0.375 in
User Parameters			

When importing DWG/DXF profiles, select the materials for each in the Sources dialog. This sets the material and the thickness for each.

And like the CAM environment, when the parts are updated, they can automatically update in the Nesting environment and all studies can simply be recalculated.