



Fusion – tips and tricks Volume 2
**How to improve your workflow in
the CAM workspace (for machining).**

Welcome to Volume 2 of the Fusion tips and tricks guide, created for the CAM workspace (with a focus on machining). This guide is open to all experience levels and by will help improve your overall Fusion skills and become a better CAM programmer. The tips come recommended by the Autodesk team. We have broken them down into the three categories that matter most to programmers; how to work smarter, how to work faster and how to work easier.

Table of Contents

Housekeeping Tips – Know where to look

- Each button will have a tooltip appear when you hover over it
- Use the ‘i’ button for further information on each command
- Find the latest post processors on our online library

How to work smarter – Key functions within Fusion

- Know the difference between 2D, 3D and multi-axis toolpaths
- Use Machining Time to estimate and quote jobs
- Use manufacturing models to modify parts
- Use Manual NC to add-in custom code
- Use the in-process stock option to instantly view the stock after each toolpath

How to work faster – Use these time saving functions

- Pattern toolpaths
- Derive toolpaths
- Drag tools between operations to copy over tools
- Hole Recognition to speed up programming
- Hole Recognition to speed up drilling and boring programming
- ‘Align to View’ to quickly set up different toolpath orientations

How to work easier – Use automation as much as possible

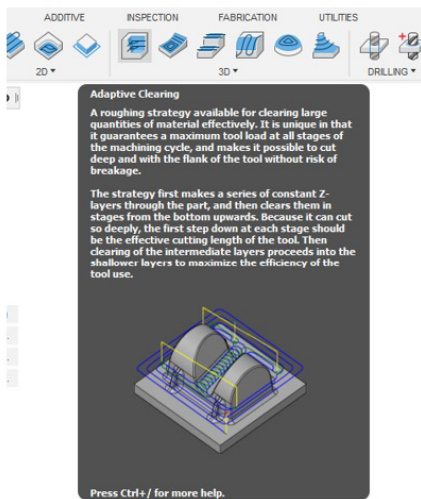
- Use toolpath templates to quick start programming
- Use formulas in the toolpath value fields to automate calculations
- Use ‘from preceding setup’ as the stock option when doing subsequent setups
- Part alignment will save you hours setting up your machine

Housekeeping Tips – Knowing where to look:

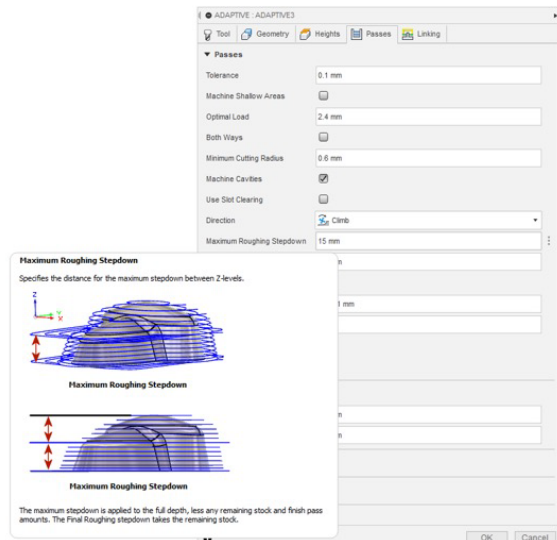
Let’s start with some basic housekeeping tips that focus on knowing where to look for information when you’re stuck.

A - Each button will have a tooltip appear when you hover over them.

Fusion makes it easy to learn with tooltip pop-ups. Every command and button has a tooltip associated to it, that will appear when you hover your mouse cursor over it. Use these to learn and explore new features that you have not used before.



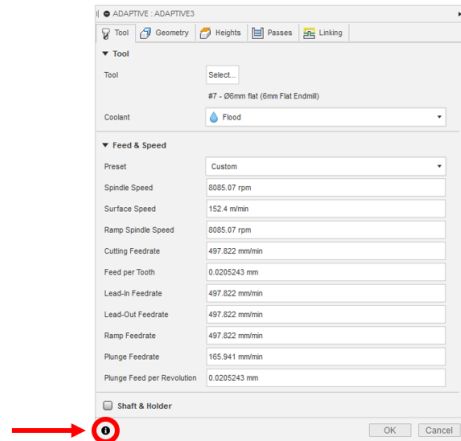
Tooltips appear over commands in the ribbon



Tooltips appear over commands within functions

B - Use the ‘i’ button for further information on each command.

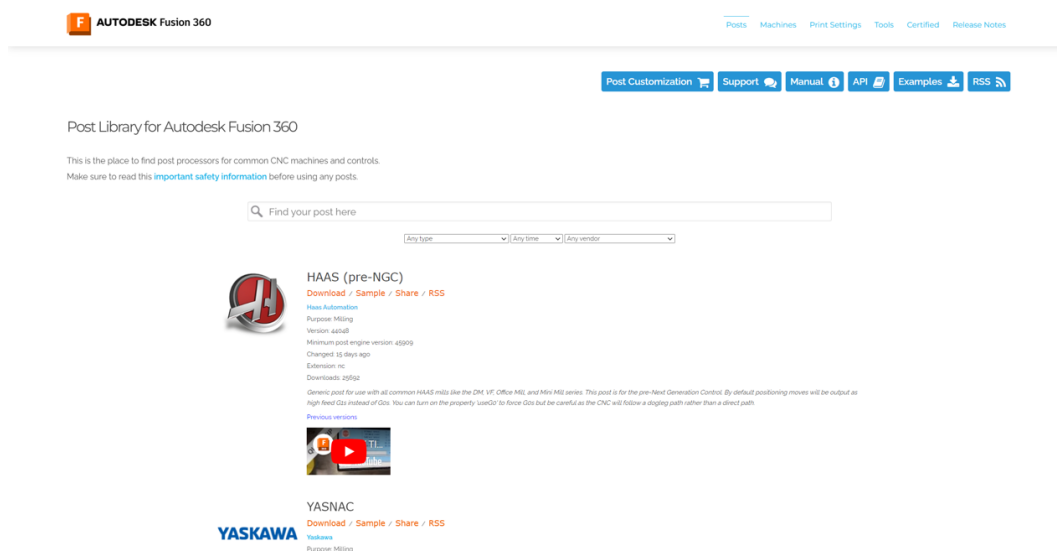
If the tooltips are not enough, you can always access our full product documentation page for a further breakdown on each function and command. Clicking the ‘i’ button found towards the bottom left of each command box will take you directly to the function’s page in our online documentation.



‘i’ symbol will take you to the product documentation page

C - Find the latest post processors on our online library.

The last step to get usable G-code out of Fusion and into your machine is to ‘post’ out your toolpath operations with the correct post processor. These posts are freely available to all users and are regularly updated, with new machines being added every few months. Find the latest versions of these post processors in our online library: <https://cam.autodesk.com/hsmposts>.



How to work smarter – Know these key functions within Fusion:

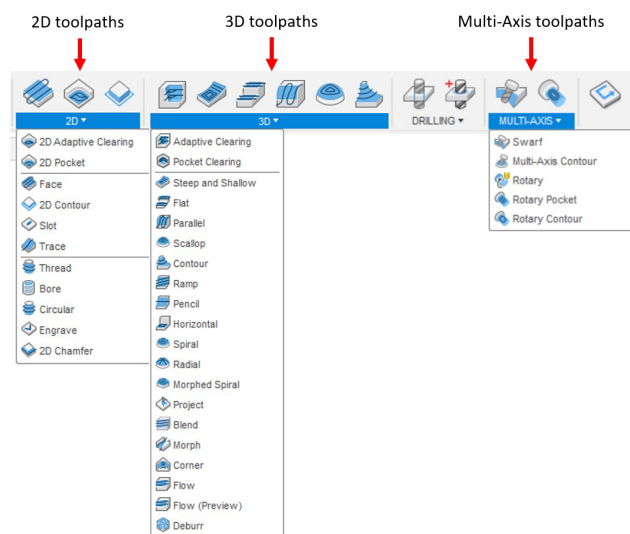
These tips ensure that you know how to use key functions in the CAM workspace.

- **Know the difference between 2D, 3D and multi-axis toolpaths**

In the milling tab you'll notice options for 2D, 3D and multi-axis toolpaths. It's important to know the differences between each and when to use them.

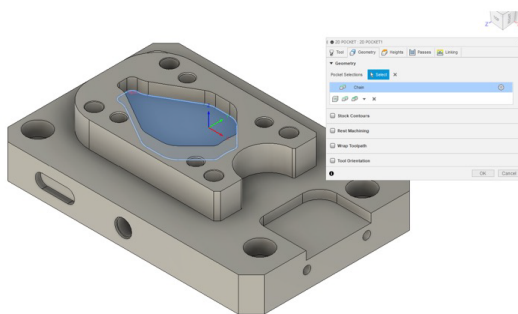
It's a common misconception that 2D toolpaths are only for 2.5 axis machines, 3D toolpaths are only for 3-axis machines and multi-axis toolpaths are only for 4-and 5-axis machines. This is indeed not the case and is instead based on the geometry of the part and the type of cutting motion you need.

Multi-axis toolpaths generate motions that only a 4-or 5-axis machine can use, but 2D and 3D toolpaths can be used across the entire range of milling axes. Let's take a look at each:

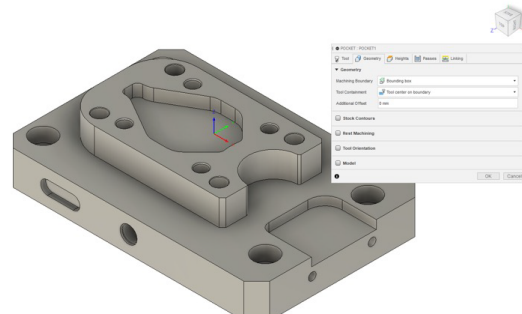


2D toolpaths are commonly used to machine parts that are prismatic, that is, all the surfaces of the part are horizontal or vertical. In a 2D toolpath, you must select the geometric boundary and depth of cut to be machined from the CAD model. These are not good candidates for parts with varying heights and freeform surfaces. These 2D toolpaths have a 'tool orientation' function and can be used for positional multi-axis motions (otherwise known as indexing). They also have a 'wrap' function which allows them to wrap around cylindrical surfaces, thus supplying 4-axis simultaneous cutting capabilities too.

3D toolpaths are used to machine parts that are more freeform, that is, where the surfaces are not simply vertical or horizontal but vary across the part. Unlike 2D toolpaths, 3D toolpaths are considered ‘model and stock aware’ and so a geometric boundary does not need to be selected as they will automatically recognise the part and calculate the toolpath for you. All 3D toolpaths have the ‘tool orientation’ capability and can be used for positional multi-axis motions. However, some 3D toolpaths go further and have a full ‘multi-axis’ capability that enables them to do simultaneous multi-axis motions too.



2D toolpaths work at one depth and may require you to select the geometry.



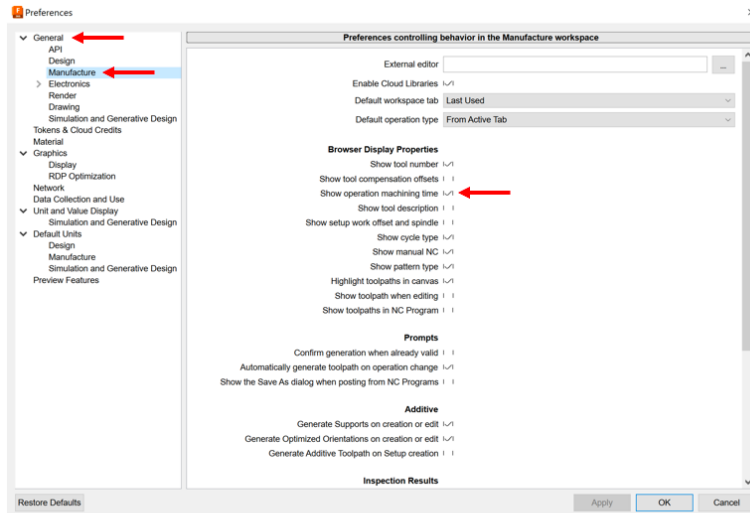
3D toolpaths are model and stock aware.

Multi-axis toolpaths are a more specific category of toolpaths and can only be used for certain multi-axis simultaneous motions. They perform the milling motions that are not covered by 2D and 3D toolpaths. Like 3D toolpaths they are ‘model and stock aware’, however due to the uniqueness of each, they do have their own geometry settings that need selecting.

Each toolpath across the 2D, 3D and multi-axis range serves different purposes. Learn how to use each of them so that you can maximise the toolpath options available to you.

▪ **Use Machining Time to estimate and quote jobs**

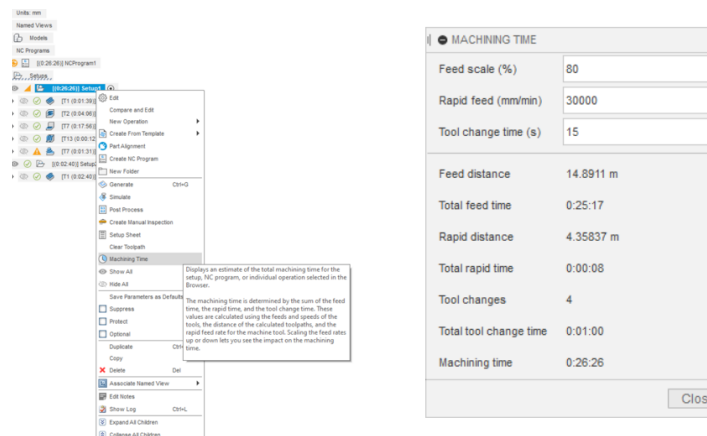
There are two methods of viewing machining time within Fusion. The first is the ‘Show operation machining time’ option found in the ‘Preferences’ settings under ‘General’ > ‘Manufacture’ option. Turning this on will give you an immediate time estimation for each toolpath. This is very handy to verify and find problems as you programme through your part.



In your Preferences you can turn on the ‘Show Operation Machining Time’ setting

The second method is to right click on the setup itself and select the option ‘Machining Time.’ This will bring up the estimation for the time taken to perform the machining operations.

Note that this machining time is considered ‘perfect’ and therefore does not always account for other factors that might slow down the machine movements. Our advice is to add 30% to the time shown when quoting job times to factor this in.

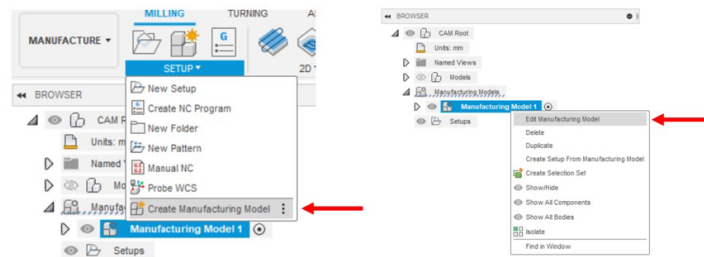


Right clicking on any setup and selecting ‘Machining Time’ will give you an estimation of time to machine. Use this for information and to help with quoting.

- **Use manufacturing models to make one-off edits to your part**

There are times when you'll need to edit the CAD model of the part you're programming. Sometimes, you may wish to remove a feature that is not relevant, or you may want to edit geometry to create a better toolpath. Rather than editing the original CAD model which will affect all other setups and toolpaths, you can instead create a unique 'Manufacturing Model.' A Manufacturing Model is a separate design instance that lives in the Manufacturing workspace and allows you to freely edit the design without affecting the original model.

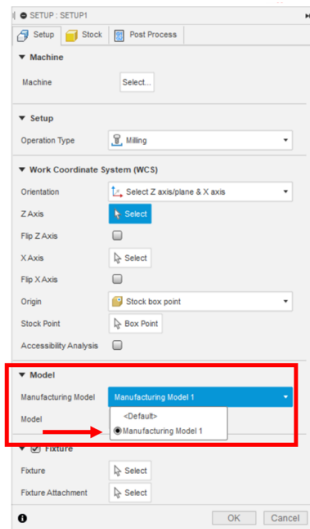
The command 'Create Manufacturing Model' is found under the 'Setup' tab in the ribbon at the top. By selecting this, you will create a new manufacturing model and enter a unique design environment that will give you the key commands needed to edit and change your model. You can create as many manufacturing models as you need.



Creating a manufacturing model allows you to make edits and modifications to your part without changing the original design.



Use it to edit your designs to allow for better toolpath creation. In the example above, we have removed the cutouts so that our spiral toolpath will better move around the part.



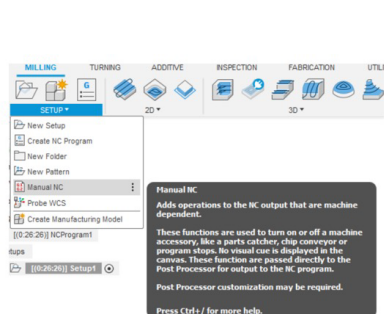
Then when you come to creating your manufacturing setup, you can choose which manufacturing model to work on.

You can choose the manufacturing model you want to work on when creating the manufacturing setup.

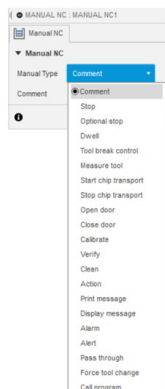
▪ Use Manual NC to add-in custom g-code

The 'Manual NC' function is a great way to add more commands to your g-code from within the Fusion environment, without having to modify your post processor. These will appear as extra lines in your g-code when you post out your operations.

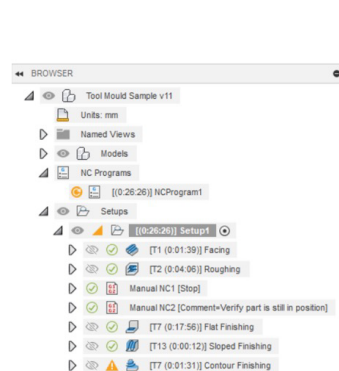
To do this, select the 'Manual NC' option found in the 'Setup' tab. From there you can choose from a range of g-code commands to add in. Options include comments, machine stops and the ability to pass through specific g-code commands relevant to your machine controller. Adding in extra commands in this space gives you the freedom and capability to have more control over how your machine will run.



The Manual NC function can be found under the Setup tab.



Here you can add a variety of different NC commands into your G-code.



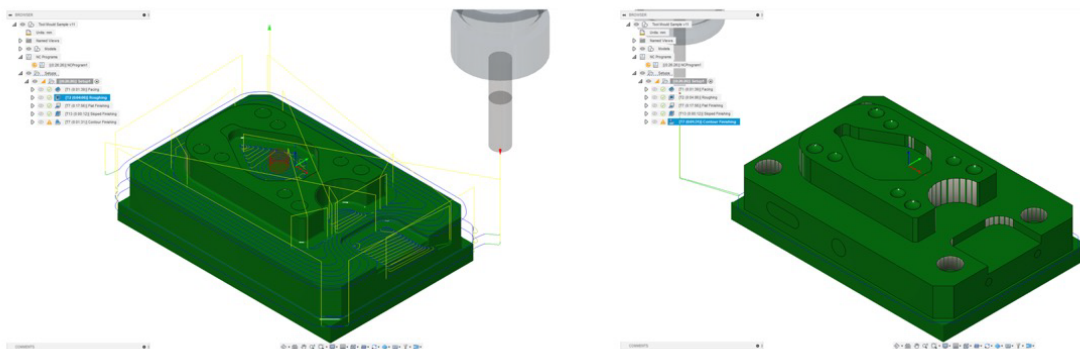
These appear in your toolpath tree and can be moved around as needed.

- Use in-process stock option to quick view the stock after each toolpath

Once you've created your toolpaths, it's helpful to visualise how the part will look after each stage. You do have the option to run a full CNC simulation using the 'simulate' command, however this can take a little time to load. A quicker option is to turn on a setting called 'in-process stock', found at the bottom of the screen. Now when you click on a toolpath, it will show you the remaining stock up to that moment. Use this quick snapshot view to dial in your toolpaths faster.



Turn on the 'Display in-process stock' setting found at the bottom of the screen.



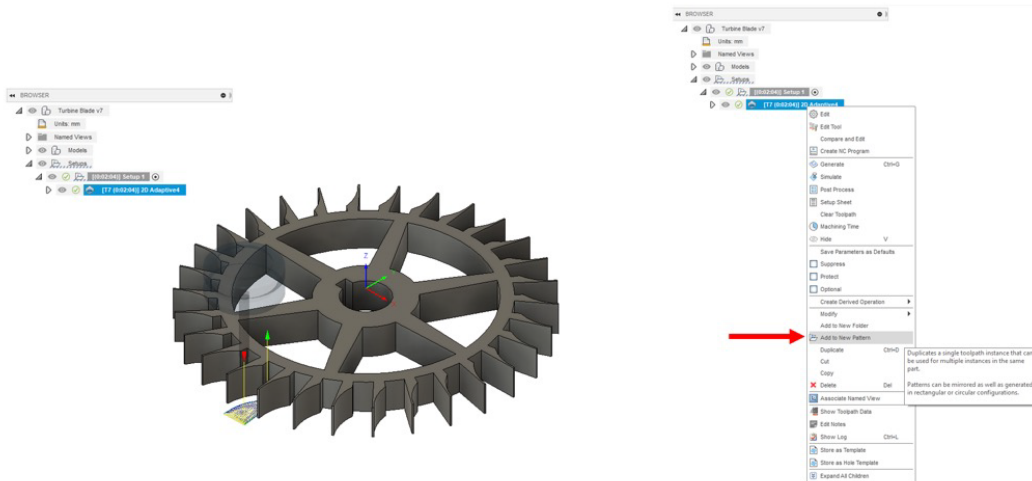
This will show the stock analysis whenever you click on a toolpath.

How to work faster – take advantage of the time saving functions

The next set of tips are aimed to help you work faster and highlight ways of cutting down on programming time.

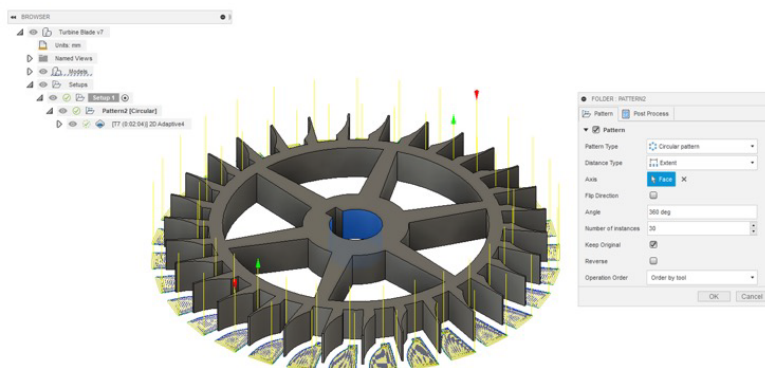
- Use Pattern toolpaths where possible

When the geometry allows, it's much easier and faster to pattern your toolpaths. To do this create a single instance of the toolpath you need, then right click on it, and select 'Add to new pattern.' From here, you can select the pattern type, the direction and the quantity you need. Fusion will then replicate this toolpath around your part. Do this whenever possible to save on time and complexity.



Create a single instance of the toolpath for one feature.

Right click on the toolpath in question and select the 'Add to New Pattern' option.



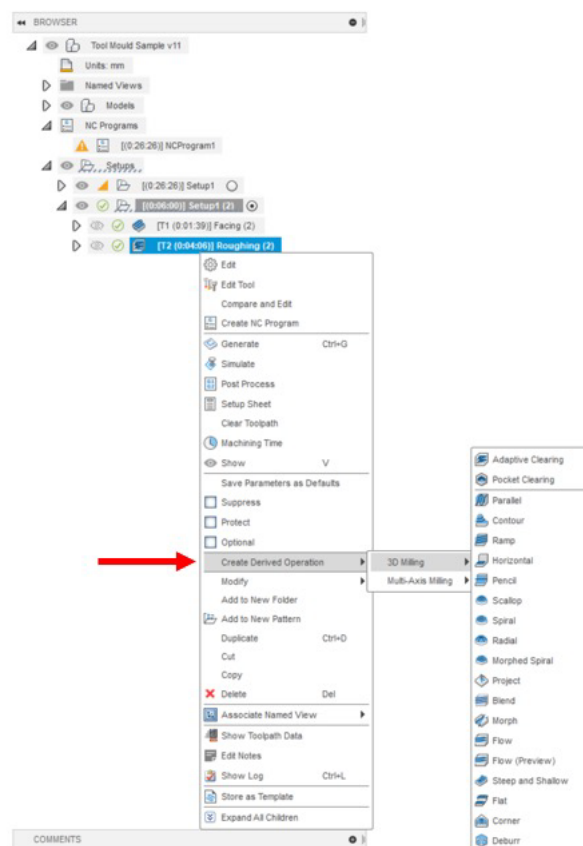
Choose the number of instances and type of pattern you need, and Fusion 360 will replicate this toolpath around your part.

- **Use Derive toolpaths where possible**

There are times when you'll want to carry forward the settings from one toolpath to another. For this you can take advantage of the 'Create Derived Operation' command.

To get here right click on the toolpath you want to copy the settings from and select 'Create Derived Operation,' then choose the next toolpath operation that you need. This creates that new toolpath with the exact same settings. Getting proficient at deriving toolpaths will speed up your programming time and ensure that you do not forget to set crucial settings.

Getting proficient at deriving toolpaths will speed up your programming time and ensure that you do not forget to set crucial settings.

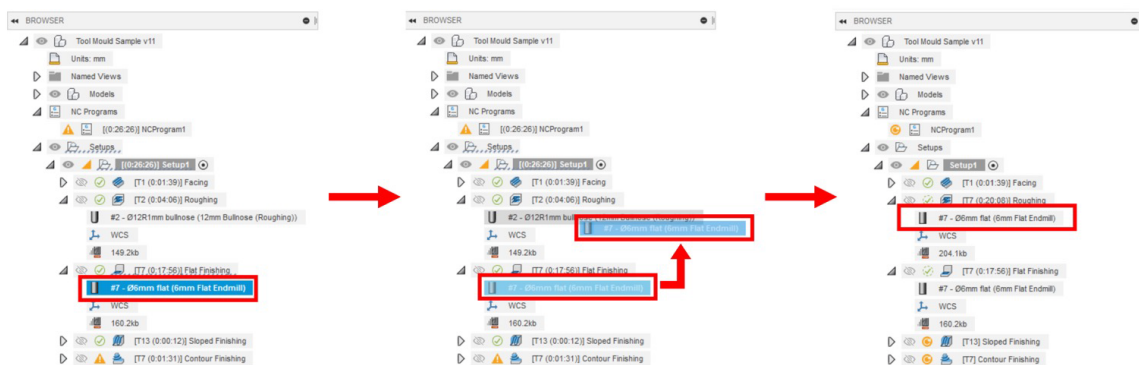


Right click on the toolpath in question and select the 'Create Derived Operation' option.

- **Drag the tools between toolpath operations.**

There are instances when you may want to edit certain toolpaths so that they all use the same tool. The time-consuming method would be to edit each toolpath individually (if you have many toolpaths, this time would really add up). Luckily there's a faster method of updating the tool used in a toolpath. You can simply drag and drop the tool used by one toolpath into another. What this does, is copy over the tool to the new toolpath and automatically recalculate it for you. No manual intervention or edits are required.

This method is much faster at updating the tool selection and can save you a considerable amount of time when programming especially if you need to update many toolpaths en masse.

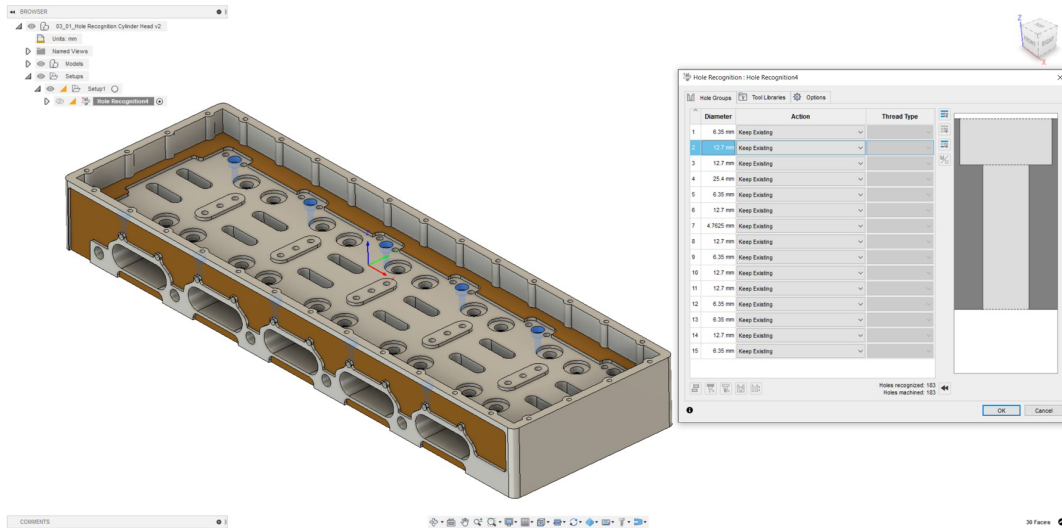


Dragging and dropping the tool settings between different toolpaths negates the need to open up the toolpath settings.

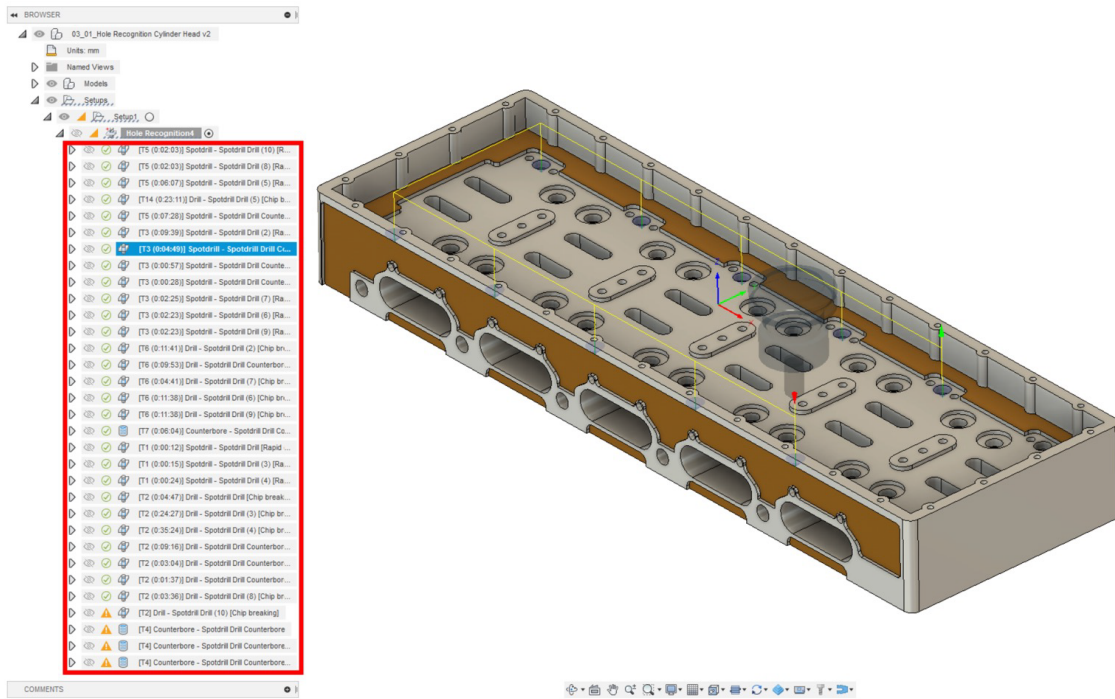
- **Use Hole Recognition to speed up drilling and boring programming**

Whether you only have to make a few holes or 100s, Hole Recognition is a valuable tool used to generate drilling and boring toolpaths in an instant.

By selecting the 'Hole Recognition' command, Fusion will find all the holes on your model and suggest a variety of hole making operations for each. It refers to your active tool library and will suggest drilling or boring options so that you will only get the hole making operations that can actually be made with the tools available. So there's no need to spend time checking if you have the right tool. It can even be applied to multi-axis operations too. Once you hit ok, Fusion will generate all of your hole making toolpaths in an instant, saving you a lot of time and energy (and no doubt your sanity too).



Hole Recognition will identify all the holes on the part, refer to your active tool library and suggest hole making operations.

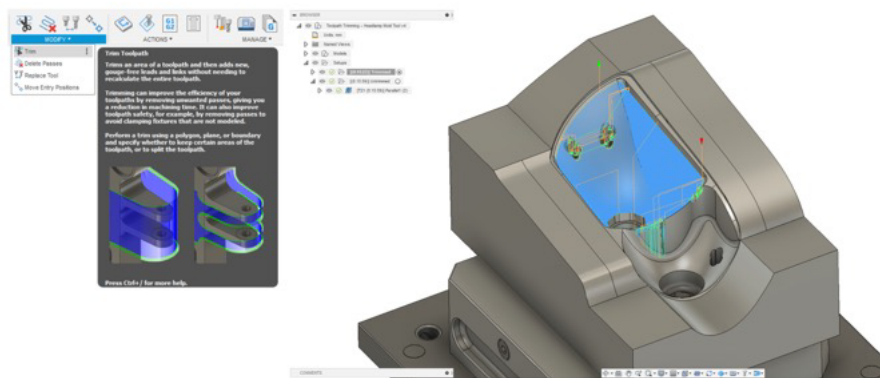


Hole Recognition will identify all the holes on the part, refer to your active tool library and suggest hole making operations.

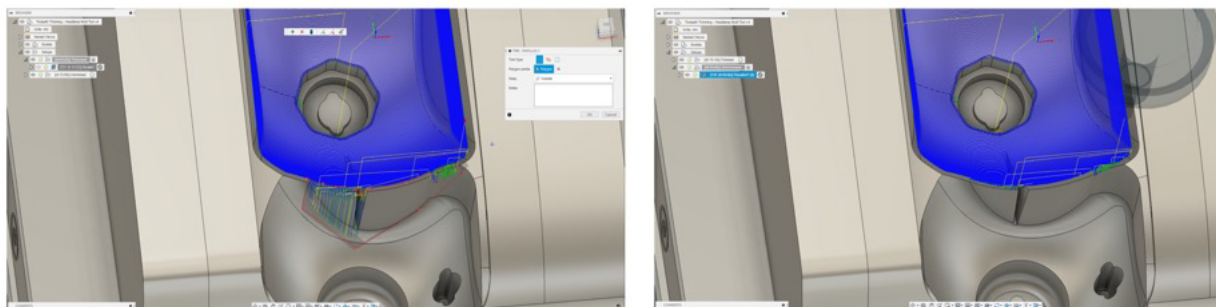
- **Use Toolpath Trimming to speed up programming**

Often, we see users struggling to control a toolpath that is either cutting too much air or cutting areas it should not be, and hours can be wasted. Fortunately, there is an easier way to contain toolpaths, and that is by using the ‘toolpath trimming’ command.

This command will allow you to quickly draw out a boundary, either by freehand or by using existing geometry and planes, so that you can then remove irrelevant toolpath motions. The toolpath will then recalculate accounting for the trimming, and you’ll be left with a more efficient toolpath. This can also be used to split toolpaths in half should you wish to control and manage it that way.



There are many instances when a toolpath might go outside the desired area.



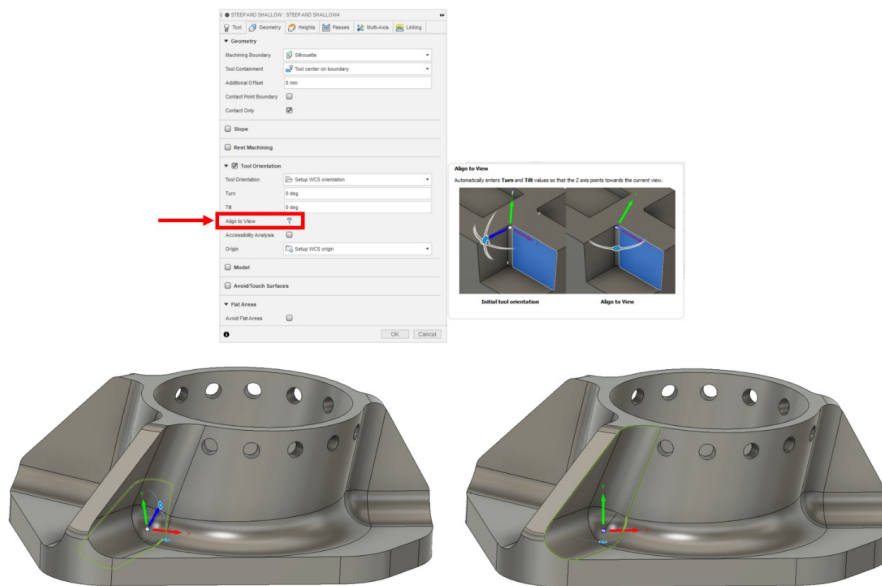
Before Toolpath Trimming

After Toolpath Trimming

ADDITIONAL TIP: use the ‘Delete Passes’ command to have even more granular editing control with your toolpaths.

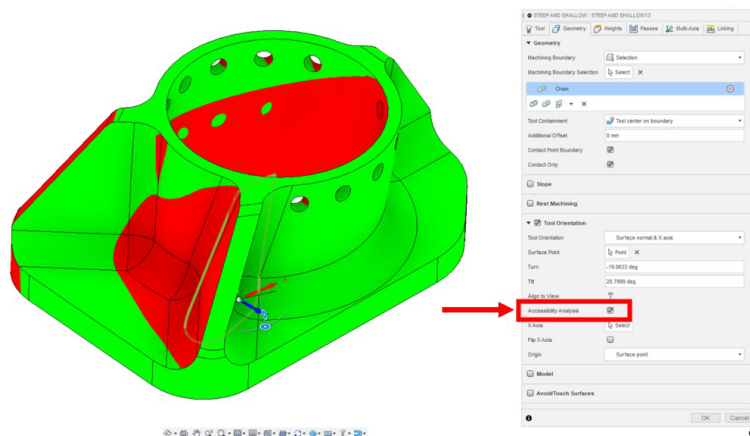
- Use ‘Align to View’ to quickly set up different toolpath orientations.

Another great and often overlooked function to speed up programming is the ‘Align to View’ command found in the ‘Tool Orientation’ section. This command instantly aligns the toolpath’s z-axis to be in line with the camera view. There’s no need to select faces or planes and it’s a fast and straightforward way to get the alignment you need. It’s particularly useful when there are no flat reference surfaces to choose or if the angle of approach is complicated.



Align to View will orientate the axis so that the Z-axis is in-line with the camera view.

Combine this with the ‘Accessibility Analysis’ function and you’ll get a clear picture whether your part can be accessed from its current angle of approach.



Follow it up with an Accessibility Analysis to see what is and is not reachable. Note that this will not consider multi-axis movements at this stage.

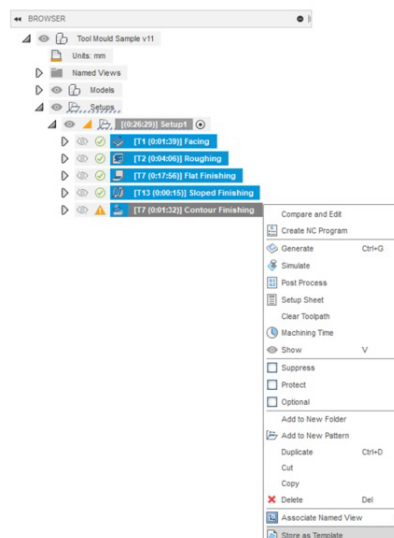
How to work easier – Know how to automate as much as possible

This final set of tips will build upon the earlier two and aims to help you unlock the full automating capabilities within the Fusion CAM workspace. Mastering these techniques will save you a considerable amount of time and make your CAM programming experience more enjoyable.

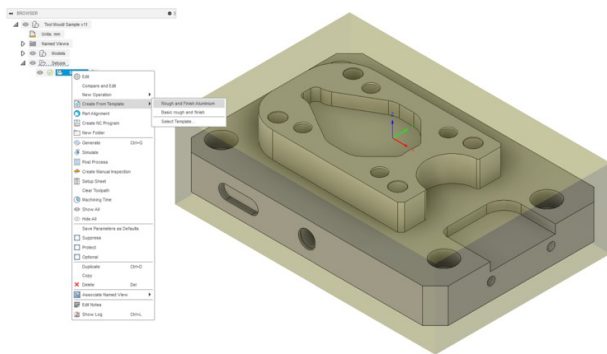
- Use toolpath templates to quick start your programming (and to also create a rapid quoting system).

Once you become familiar with programming in Fusion, you can take advantage of toolpath templates to save you time and effort. You most likely already have a favourite set of toolpaths with specific settings that you use each time. Rather than having to create them individually, you can instead save them as a template and call them up every time you start a new setup.

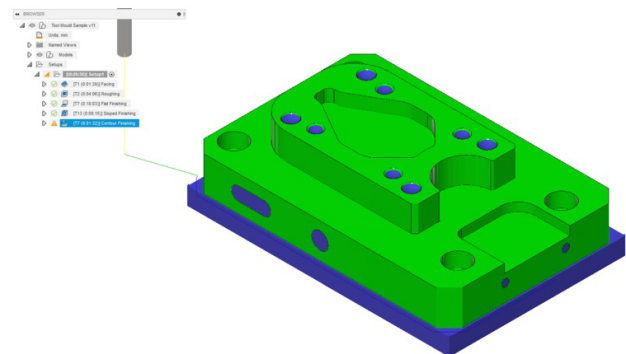
Take this a step further and create different templates for the different parts you might machine. A common example is to create a template of starting toolpaths for each material type that you work with.



To create a template, highlight all of the toolpaths that you wish to add to a template. Then right click and select 'Store as Template'. Then give your template a name.



To bring in this template, first create a setup, then right click on it and select 'Create From Template', then choose the template you wish to add it.



Templates can give you an 80%+ head start on machining jobs, especially if you use the same type of toolpaths and settings regularly.

Using templates is one of the best pieces of advice we can give any machinist. Not only can it give you a huge head start when you begin programming a new part, but you can also share them with your colleagues and teammates so that the programming expertise is shared amongst the shopfloor..

PRO TIP: Combine templates with 'Machining Time' estimation to create a rapid quoting system on new incoming parts.

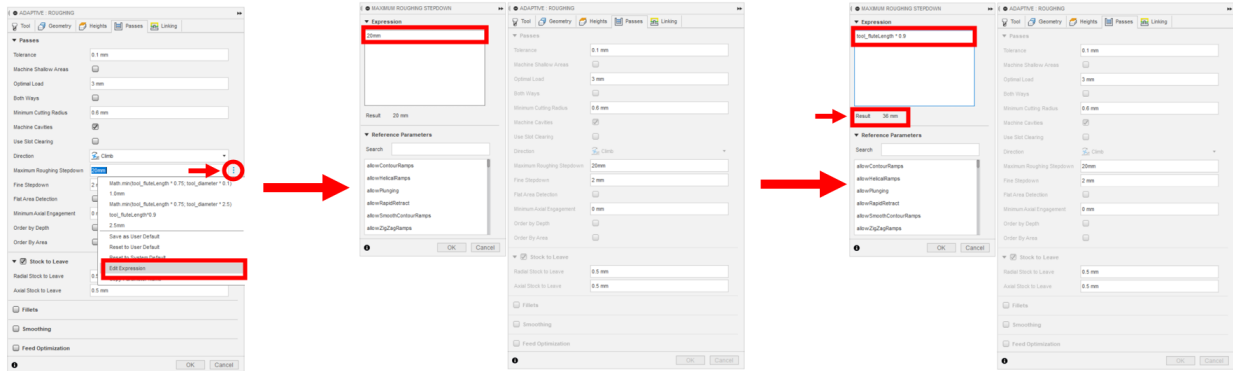
▪ Use formulas to define toolpath values

Within the toolpath settings, there are a number of value fields that allow you to define how the tool will cut. Examples of this include the tool stepover, stepdown, number of passes etc. Whilst they allow you to type values in manually, it's a much more effective method to use a formula to automatically calculate a value instead.

To do this, select the three vertical dots that appear to the right of any value field, then select the 'Edit Expression' option. Here you can select from pre-defined formulas or construct your own by calling up expressions from the in-built library of parameters. These include options like tool length, tool diameter, and flute length. The full list be found here:

<https://www.autodesk.com/support/technical/article/caas/tsarticles/ts/5D2gWS3yp6Q56lEN30ZFik.html>

A common example that we often see is to set the 'Maximum Roughing Stepdown' field to be equal to the value of 90% of the tool flute length. Incorporating formulas like this to automate your settings reduces the number of decisions you need to make and helps prevent errors.



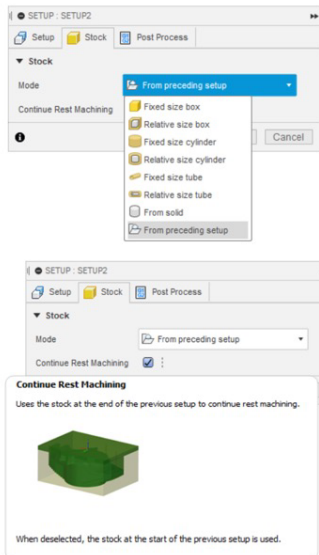
Select 'Edit Expression'.

Type in the desired formula in the Expression box.

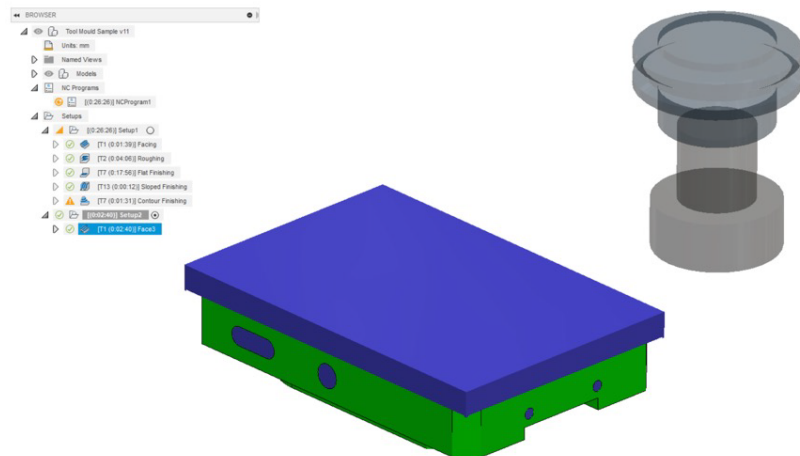
The value will automatically be calculated.

- Use 'from preceding setup' as the stock option when doing subsequent setups

If you want to create additional setups on the same part (to program the other side of the part for example), Fusion has an option to make choosing the remaining stock easy and fast. Select the option 'from preceding setup' within the 'Stock' tab when selecting your stock and turn on the option 'continue rest machining.' This will recognize what is left over from earlier operations and carry that over to your new setup.



When in the 'Stock' tab, select the options 'From Preceding Setup' & 'Continue Rest Machining'

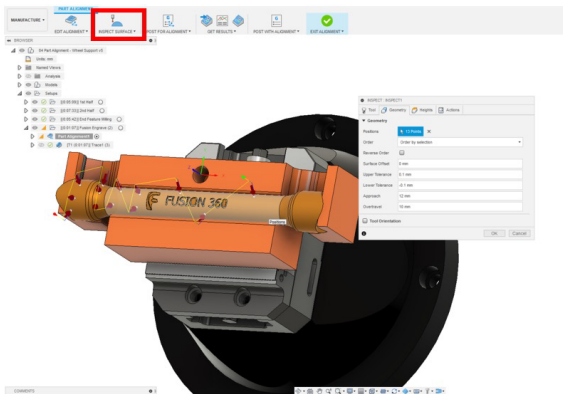


This will carry the remaining stock over from the previous operations to your current setup.

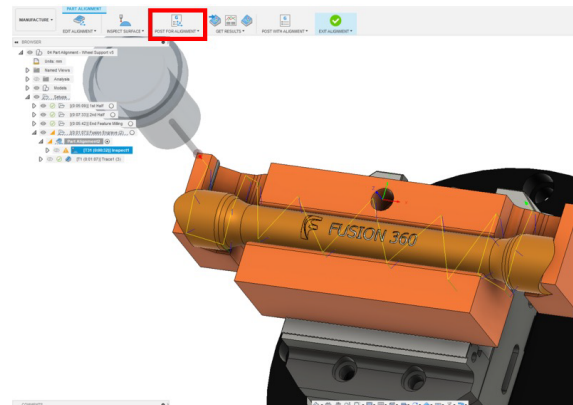
- **Part alignment will save you hours with setting up your machine**

If there is one capability that could save you hours with your programming, it's Part Alignment. There are instances where you'll need to position an already machined part back onto a machine accurately. This process, when done manually, can involve potentially hours of gently knocking and shimming the part into the correct place with the correct alignment. That is why Fusion's automatic part alignment functionality is a huge timesaver that can reduce a multi hour-long process down to minutes.

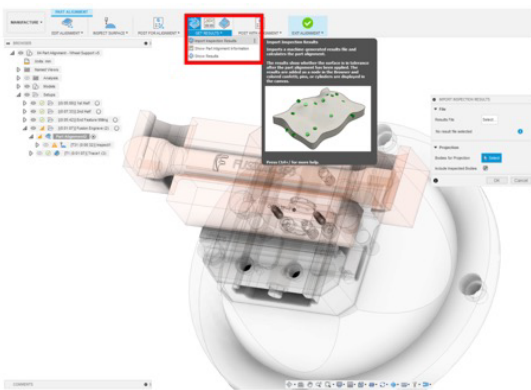
Part alignment can be found under the 'Inspection' tab and gives you the choice to align your part in 3-, 4- or 5-axes. It's a 2-step process to get the part aligned. First, you setup and run a probing cycle on your part to determine its current positioning. Then you feed these results back into your setup and Fusion will showcase any misalignment and apply a positional adjustment in the outputted g-code which will move it into the correct place before the machining operations start.



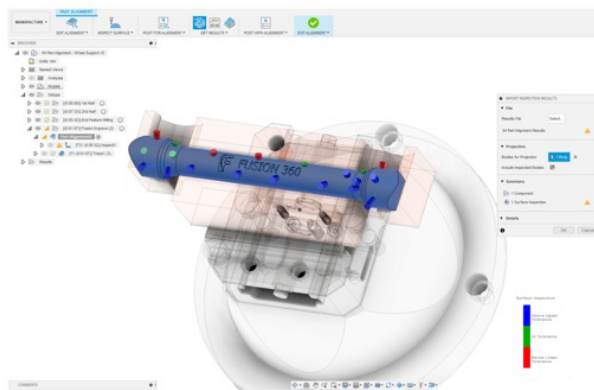
The first stage of doing a part alignment is to setup an inspection cycle from the 'Part Alignment' tab.



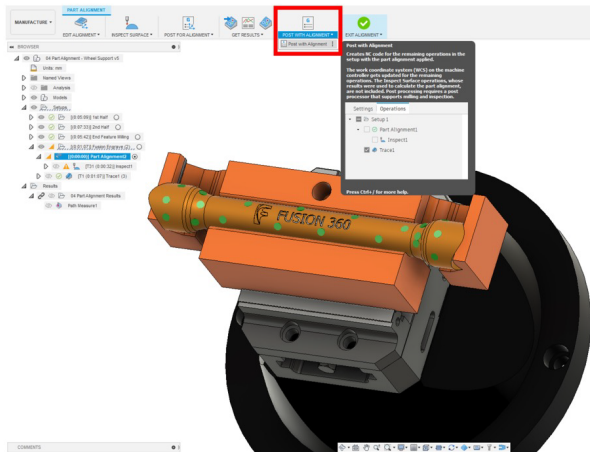
Then post this inspection cycle out and run it on your CNC machine. The CNC machine should output a text file of the results.



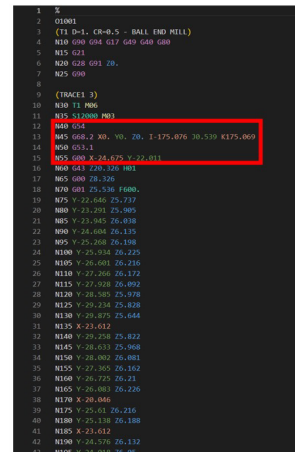
The second stage is to import the results from the CNC machine probing cycle back into Fusion 360.



This will display the tolerance of each point or 'how well aligned the part is'.



Once the results are brought in, Fusion 360 will add an auto adjustment to the setup with the 'Part Alignment' attached. The final step is to export out this setup.

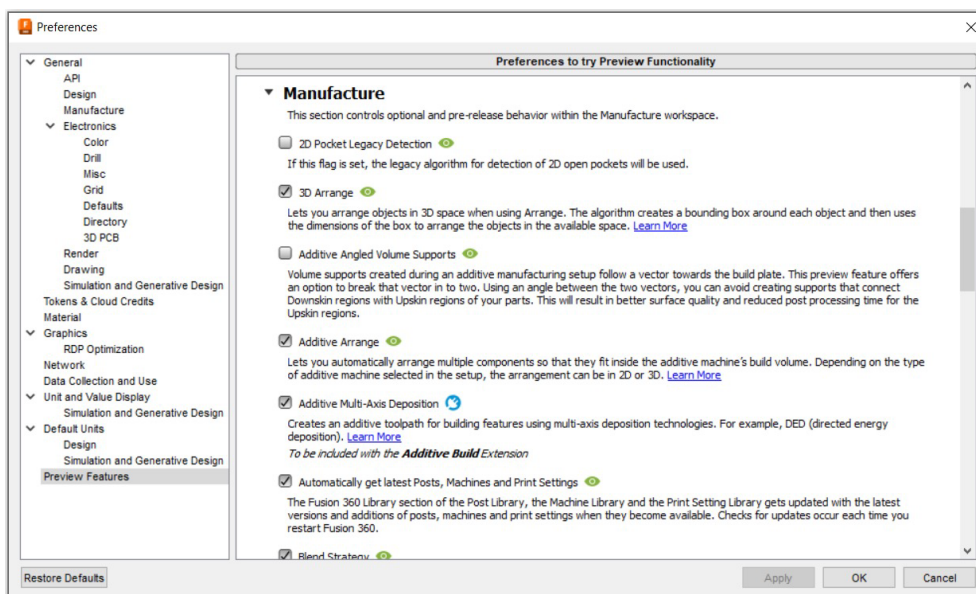


When reviewing the g-code, you'll notice that a position adjustment has been added in prior to the first toolpath cutting motion.

The steps to setup a Part Alignment are very well explained in this video: <https://www.youtube.com/watch?v=y2D1Ct8nMEM>

Bonus tip: Check out the Preview Features tab for the latest capabilities coming to Fusion

Fusion is constantly improving the software with updates and new capabilities being added approximately every 6 weeks. Prior to their release these new features are often available as a preview for you to explore and test before full release. You can turn these on in the 'Preview Features' tab which is found in your 'Preferences'. Feel free to explore and experiment with them as you please. They'll keep you at the forefront of the Fusion's CAM capabilities and you'll likely find some settings that will greatly improve your CAM workflows.



The information contained in this document represents the current view of Autodesk, Inc. as of the date of publication, and Autodesk assumes no responsibility for updating this information. Autodesk occasionally makes improvements and other changes to its products or services, so the information within applies only to the version of Autodesk Fusion offered as of the date of publication. This document is for informational purposes only. Autodesk makes no warranties, express or implied, in this document, and the information in this document does not create any binding obligation or commitment on the part of Autodesk. Without limiting or modifying the foregoing, Autodesk Fusion services are provided subject to the applicable terms of service located at <https://www.autodesk.com/company/terms-of-use/en/general-terms>. Autodesk, the Autodesk Logo and Fusion are registered trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2023 Autodesk, Inc. All rights reserved.

