

Fundamentals of the CAM Workspace Using Fusion360: Learn 2D and 3D Toolpaths

Kevin Lee
LeeVerge Integration Inc.

Learning Objectives

- Discover a CAM setup by defining stock size and establish a work coordinate system
- Learn how to apply 2D toolpath operations by learning how to select geometry
- Understand geometry-driven toolpath operations by applying 3D strategies
- Learn how to verify toolpaths with stock simulation, output setup sheets, and postprocess G-code

Description

This class will focus on the fundamentals of Fusion 360 CAM software for both new and current users, by understanding workflows in the CAM workspace through job setups creation, stock definition, and work coordinate systems alignment. We will demonstrate how simple and easy it is to use Fusion 360 CAM by applying 2D toolpath operations with geometry selections that drive them and learning how geometry-based toolpaths can be applied using 3D operations. Class attendees will learn how to verify toolpaths with full stock simulation, create setup sheets, and postprocess G-code for their machines. These foundational concepts will help new users understand the benefits of truly integrated CAD/CAM using Fusion 360 to seamlessly update all toolpaths when CAD revision or design changes are required. This class will help you learn how to make anything with Fusion 360.

Speaker

Kevin Lee Managing Director
LeeVerge Integration Inc.

Kevin is an innovative entrepreneur specializing in advanced manufacturing solutions using integrated CAM technologies. He provides leadership and experience as managing director of his own company LeeVerge Integration Inc. located in southwestern Ontario, Canada.

LeeVerge Integration Inc. is a specialized business dedicated to providing CAD/CAM/CNC manufacturing solutions and professional services for its diverse clients.

Kevin's focus on integrated cad/cam solutions provides guidance to the firm and its clients looking for innovative technologies that deliver competitive advantage with long term sustainability. Kevin extends his knowledge and experience to his local college where he is a part-time professor teaching students cnc machining with gcode programming and cad/cam applications.

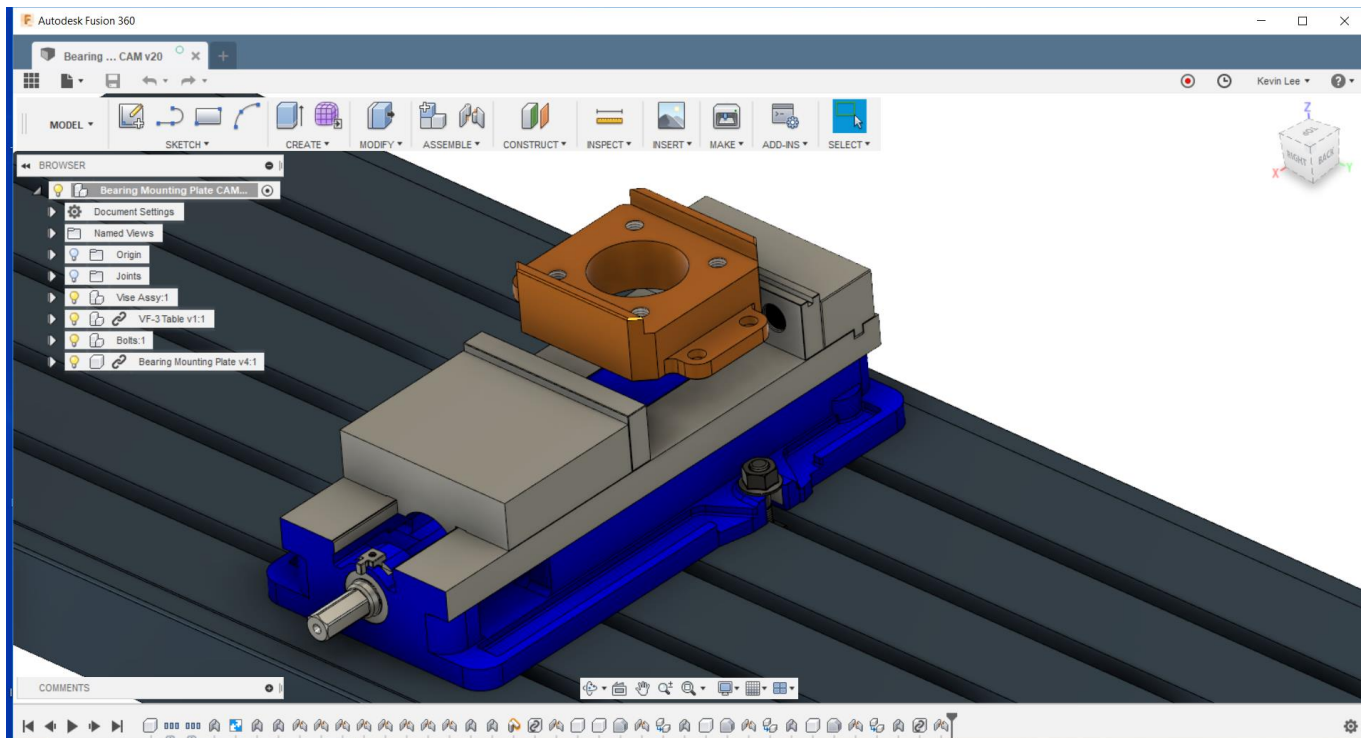
Integrated CAM with Fusion 360

12 Steps

There are many advantages to integrated CAM. It allows for seamless integration of parametric CAD designs with toolpath CAM regeneration that will improve your workflow.

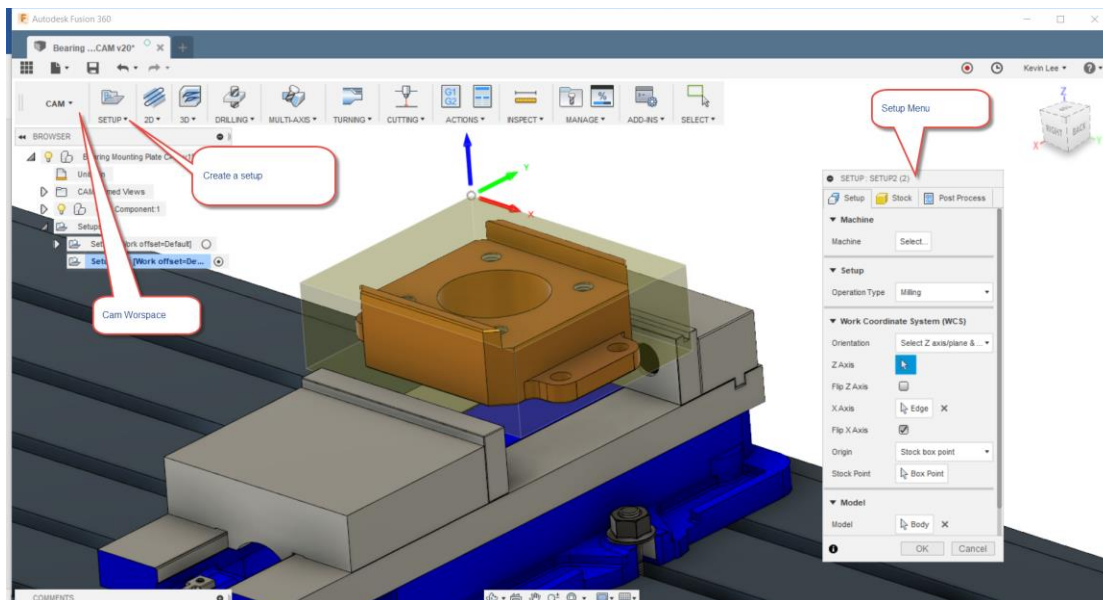
With leveraging either native CAD files imported into Fusion or parametric Fusion designs with in the modeling work space it allows for easier and faster CAM revision updates that increase efficiencies and reduces multiple extra steps during this manufacturing process.

Below are 12 basic steps that you can follow to help you get started with integrated CAM using Fusion 360.



Step 1: Create a Setup in the CAM Workspace:

To start, enter the CAM workspace and begin creating a setup by selecting the model that will be machined and define the rough stock size and work coordinate system, known as the WCS.



Select the part model body that you would like to create toolpaths on. Define the WCS orientation (blue, green, red arrow) by selecting the base of the blue arrow and then select top surface of the part. This sets the Z axis orientation of the NC program. Pick the blue aero head to flip the direction of Z positive if necessary.

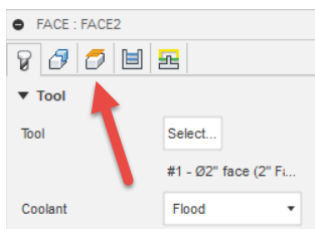
Under Origin drop down box, select stock box point and mouse click on a white dot to place the X & Y origin of the NC program. Next, select the "Stock" Tab to select fixed size box having X width (7.25") Y depth (6") Z height (2.25").

As an option, you can add a 0.25" for the "Round up to nearest" to round the nominal stock size to match something that would be easily available from material suppliers.

Approve the Setup by selecting OK.

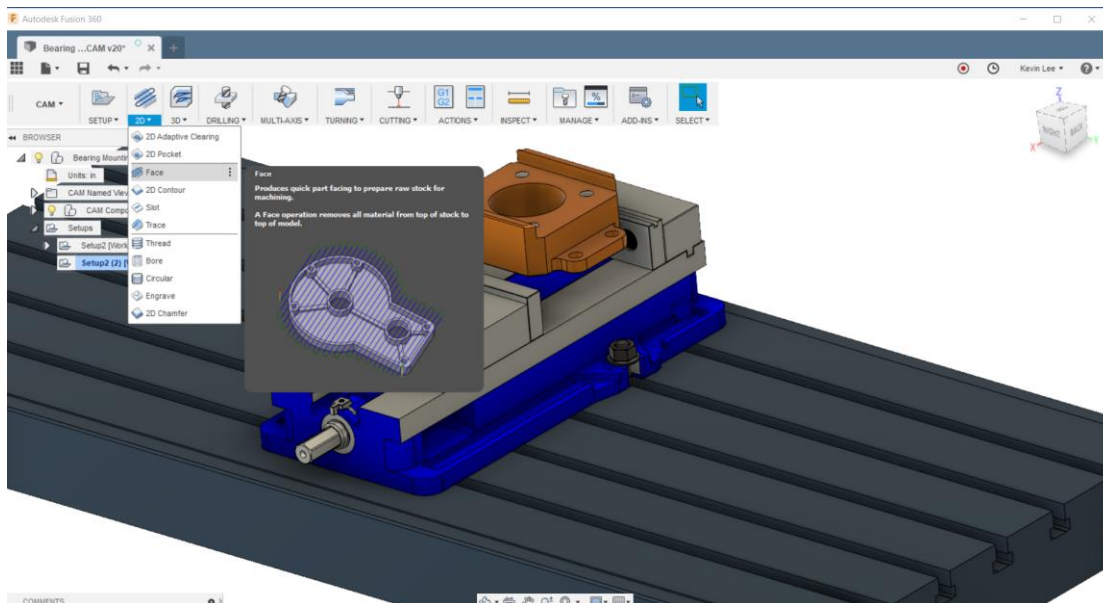
To create toolpaths, it needs to be understood that every toolpath is simple and easy to learn and navigate with the 5 tabs of information that control the parameters of toolpaths.

The 5 tabs are known as Tooling, Geometry, Heights, Passes, and Linking tabs that allow for specific parameters to be set within them.



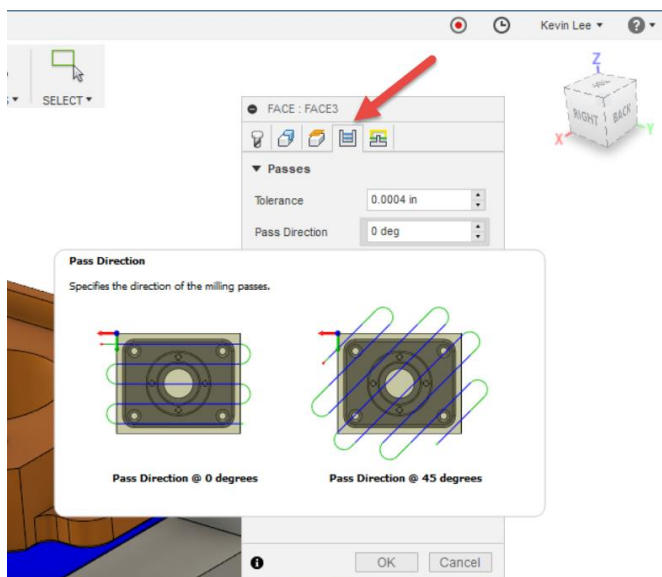
Step 2: Create a Facing Operation

Under the 2D CAM menu select Face toolpath. In the Tooling tab, select a 2" facing tool from the sample library and click OK.



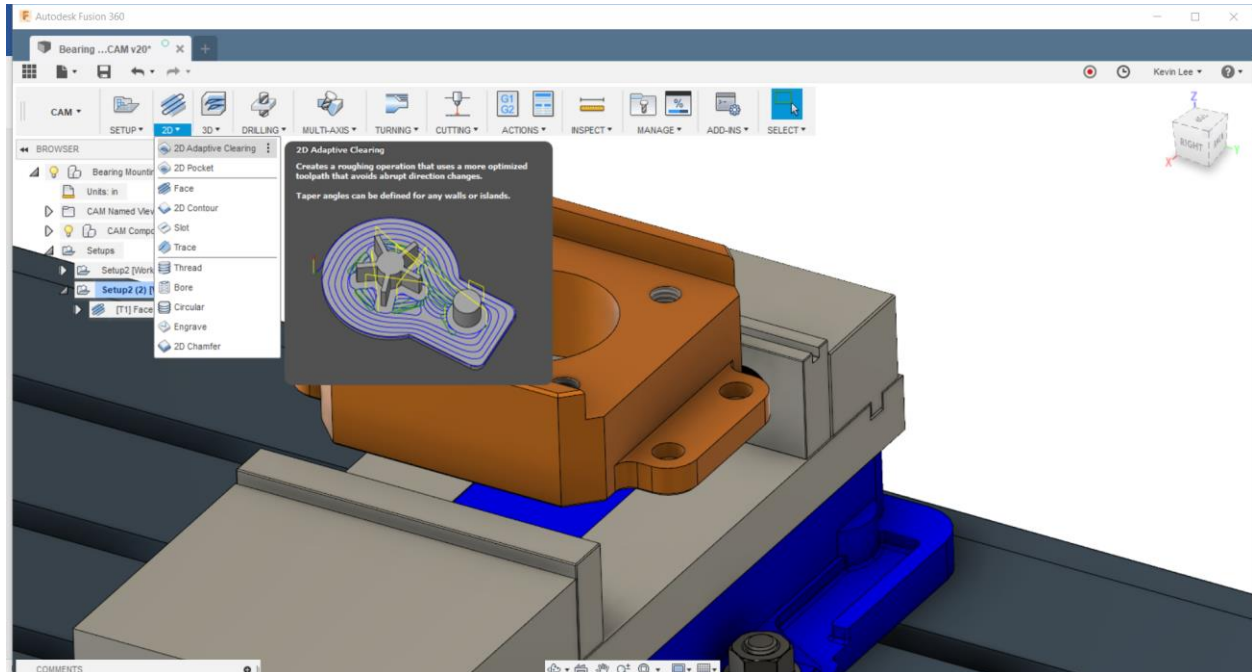
Fusion automatically creates the facing toolpath and understands the stock limitations as defined in the setup.

You can right click on the facing operation to modify other specific parameters like the Pass Direction. Note, when you pause the mouse over a parameter box a tool tip fly out will appear explaining what the parameter setting controls in the toolpath.

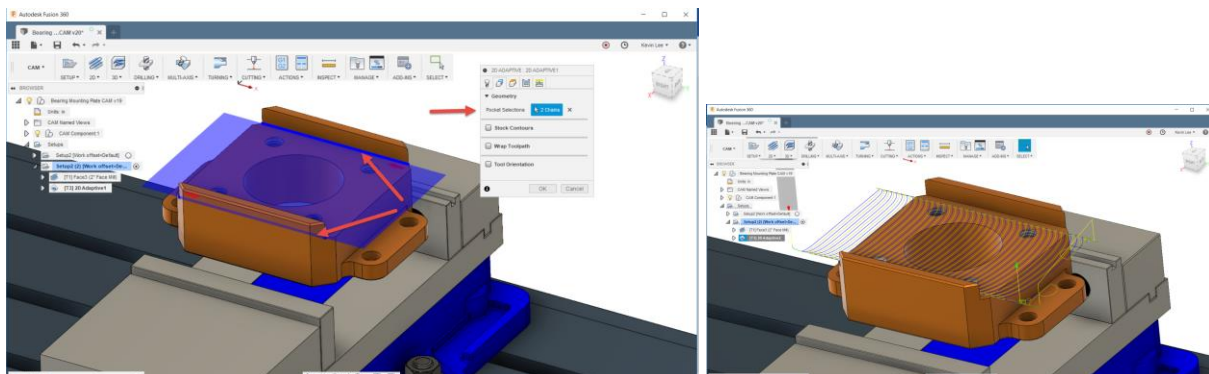


Step 3: Create a 2D Adaptive Roughing Operation

Under 2D CAM menu, select a 2D Adaptive Clearing toolpath strategy. In the tooling tab, select a 1/2" flat endmill from the sample library.



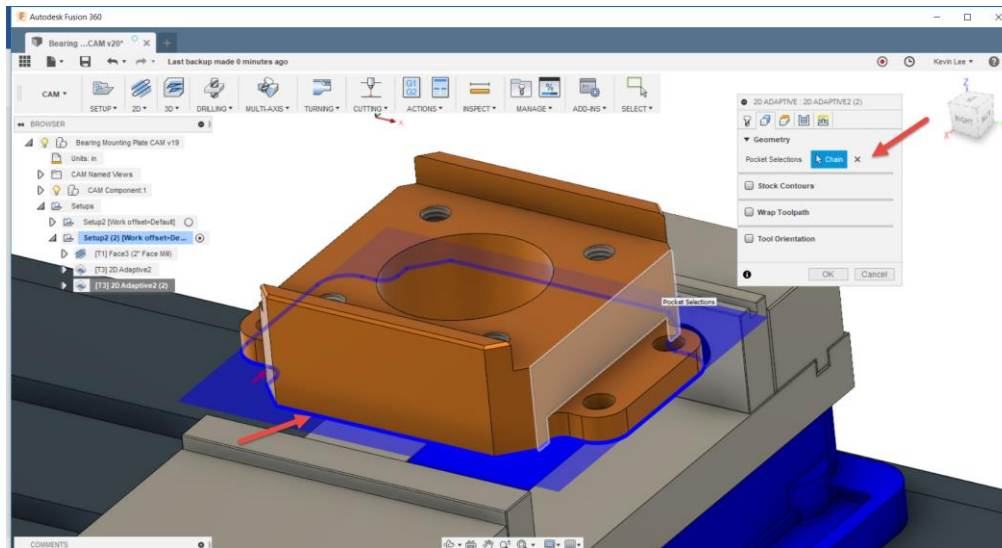
In the geometry tab, select the 2 inside edges on the upper open pocket. These edge selections automatically define the toolpath depth as selected in the Heights tab. In the 4th tab "Passes"; turn on "Stock to Leave" with 0.010" radially and axially stock to leave. We can set our "Optimal Load" and enable both ways adaptive strategy.



Step 4: Duplicating Operations

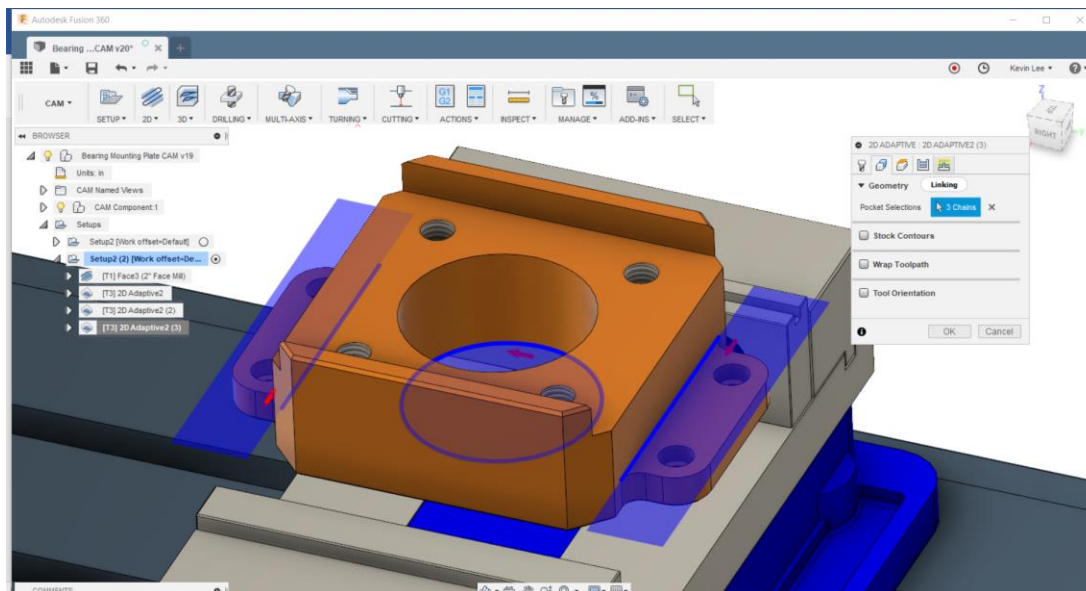
Now to build on our work we can save programming time by leveraging this roughing operation to rough away material on other part model features.

Right click on your adaptive operation and select duplicate or Ctrl D on your keyboard. In the duplicate operation under the Geometry Tab reselect the outer edge profile of the part to rough away the material on the outside of the part.

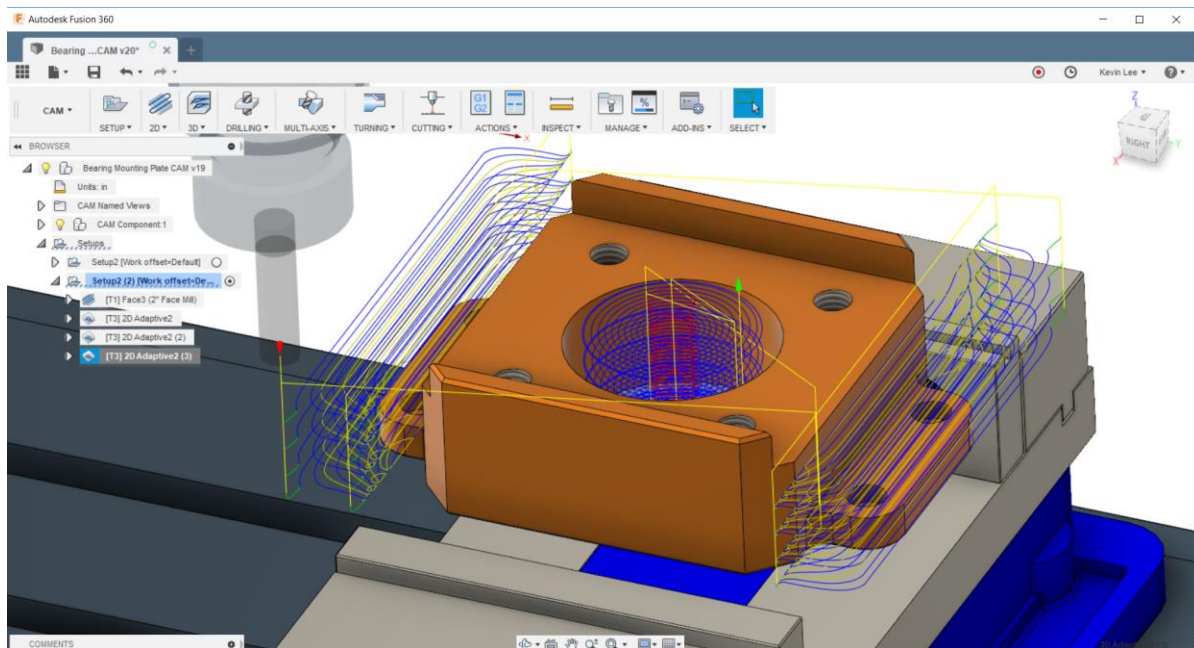


Once complete, we can duplicate the 2D adaptive operation once more so we can rough material away on the internal bore and exterior tab features on the model all with one toolpath.

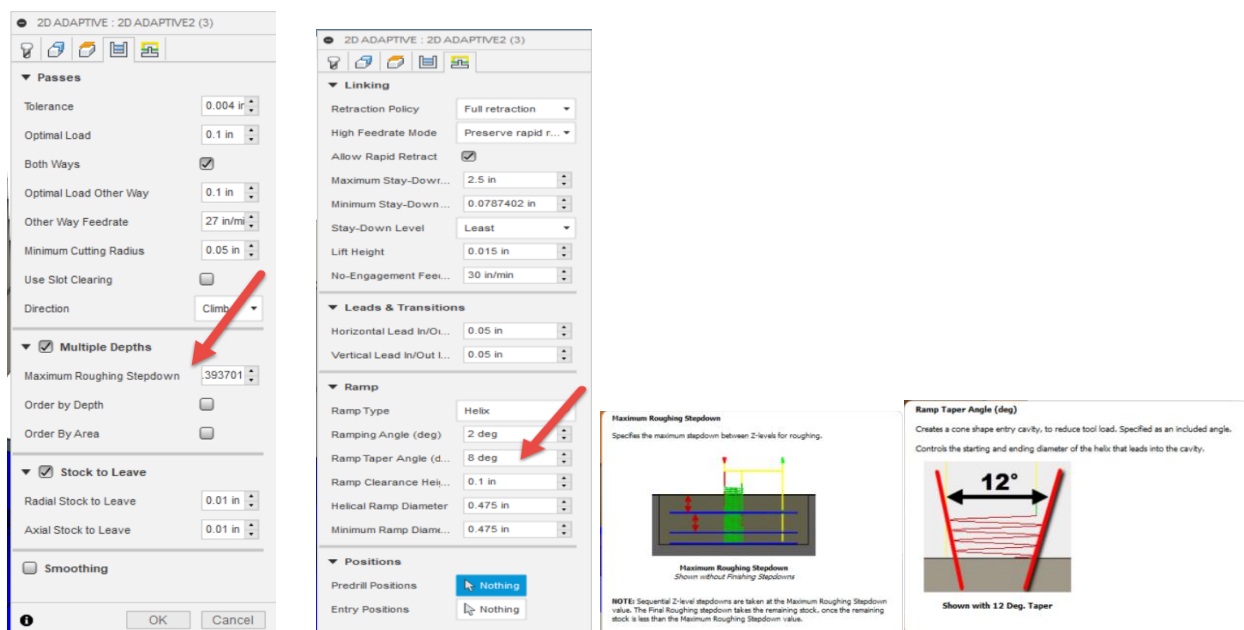
Select these 3 edges on the model.



If we enable depth cuts and ramp taper angle parameters, toolpath should look like below.



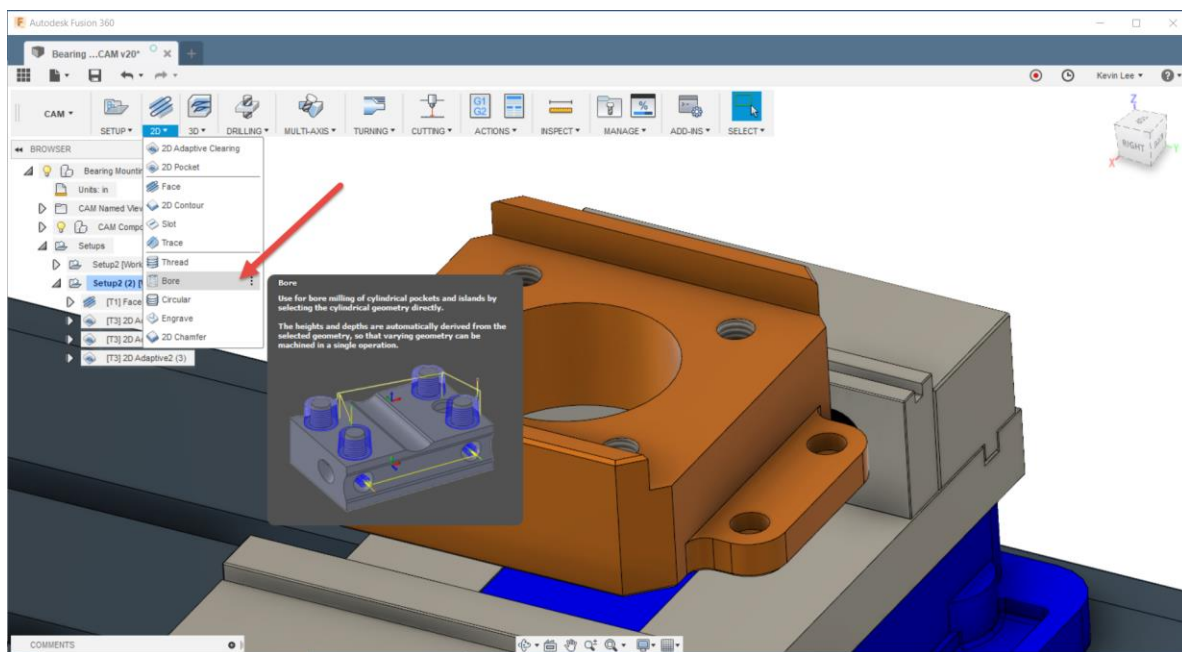
Some parameters worth noting are the Multiple Depths in the “Passes” Tab and Ramp Taper Angle found in the “Linking” Tab.



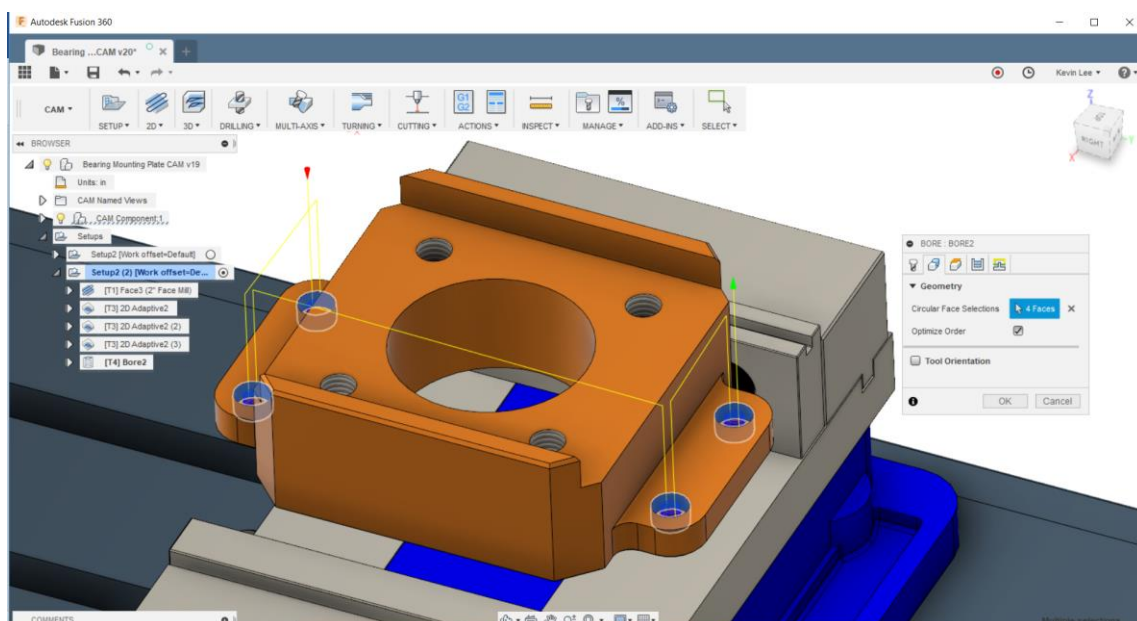
Step 5: Create a Finish Bore Operation

We can use a 2d Bore operation to machine the counterbore features on this part. Use this bore operation for milling of cylindrical pockets and islands by selecting the cylindrical geometry directly. The heights and depths are automatically derived from the selected geometry, so that varying geometry can be machined in a single operation.

In the 2D cam menu select Bore operation.



Select a 1/4" Endmill from the sample library, select the four cylindrical counter bore faces and adjust the helical pitch to 0.025" found in the passes tab.

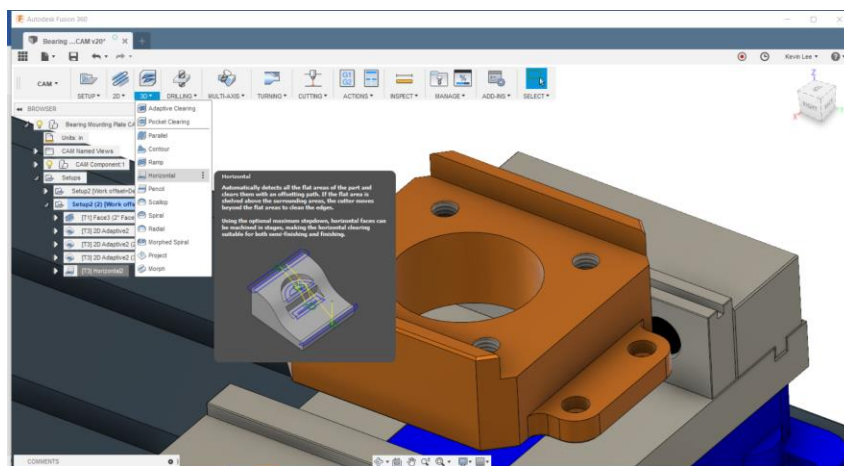


Step 6: Create a Geometry-driven 3D “Horizontal” toolpath Strategy

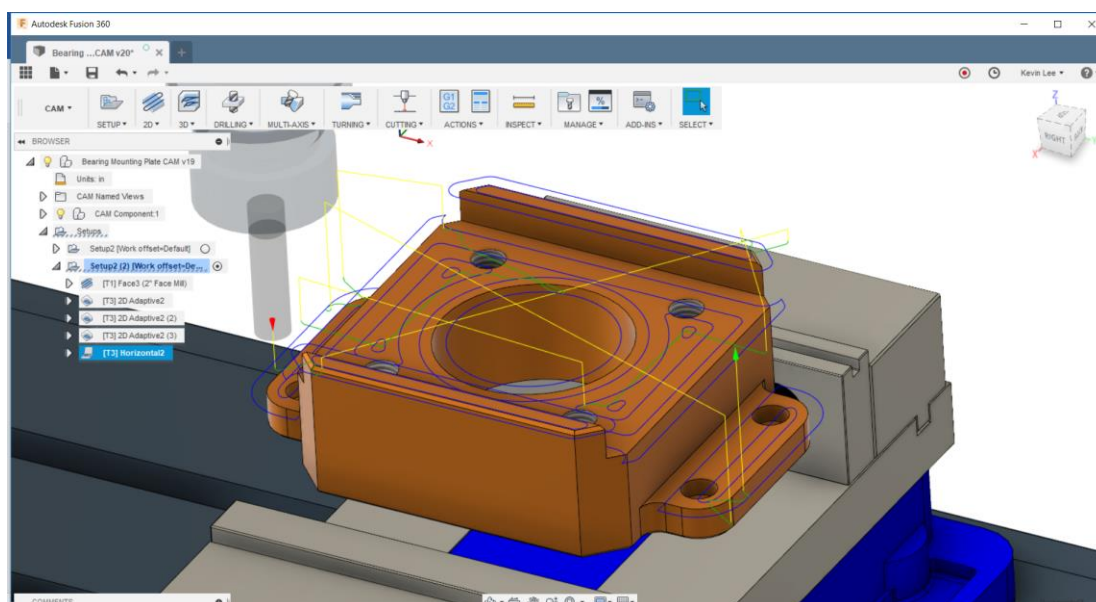
Fusion 360 cam toolpaths are geometry driven from the model. As we have created some 2D toolpath operations we had to select edge geometries to drive them. The advantage of using 3D toolpath operations is that they are geometry driven from the model and allow us to control them in specific areas by defining machining boundaries.

Under the 3D CAM menu, select the Horizontal toolpath strategy. In tooling tab, select the 1/2” flat endmill.

The Horizontal toolpath strategy automatically detects all the flat areas of the part and clears them with an offsetting toolpath. If the flat area is shelved above the surrounding areas, the cutter moves beyond the flat areas to clean the edges.

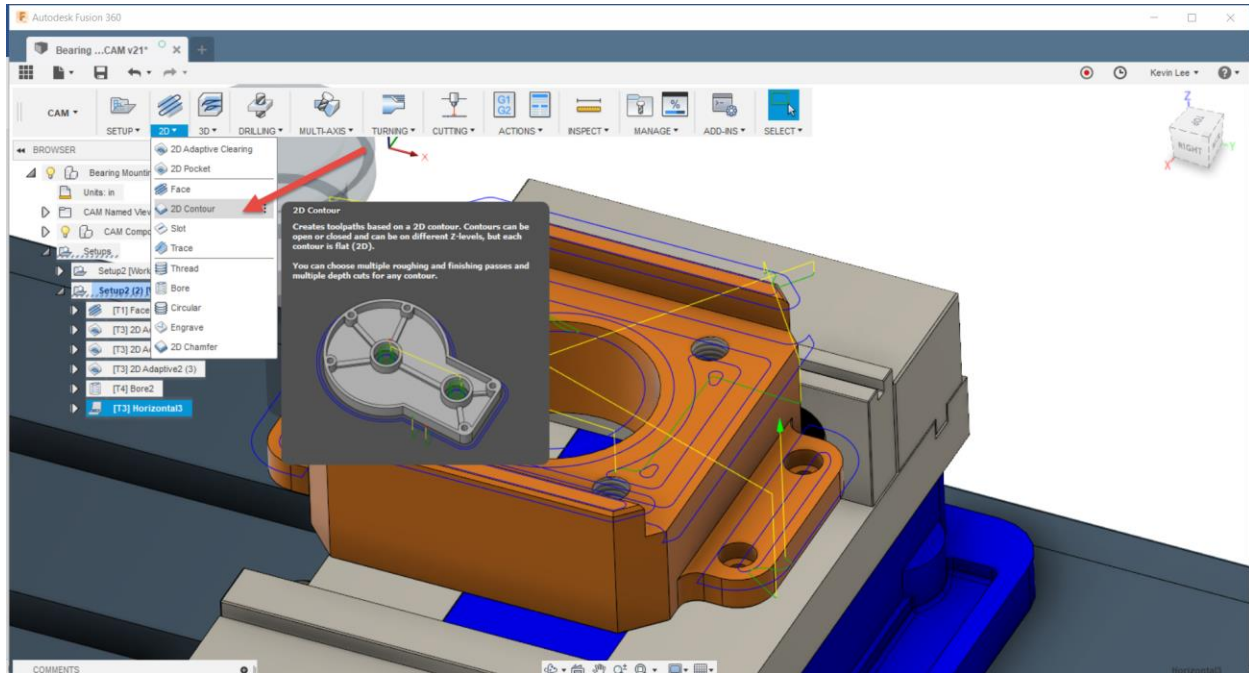


Select a 1/2” Endmill from your tool library and OK to confirm for toolpath generation.

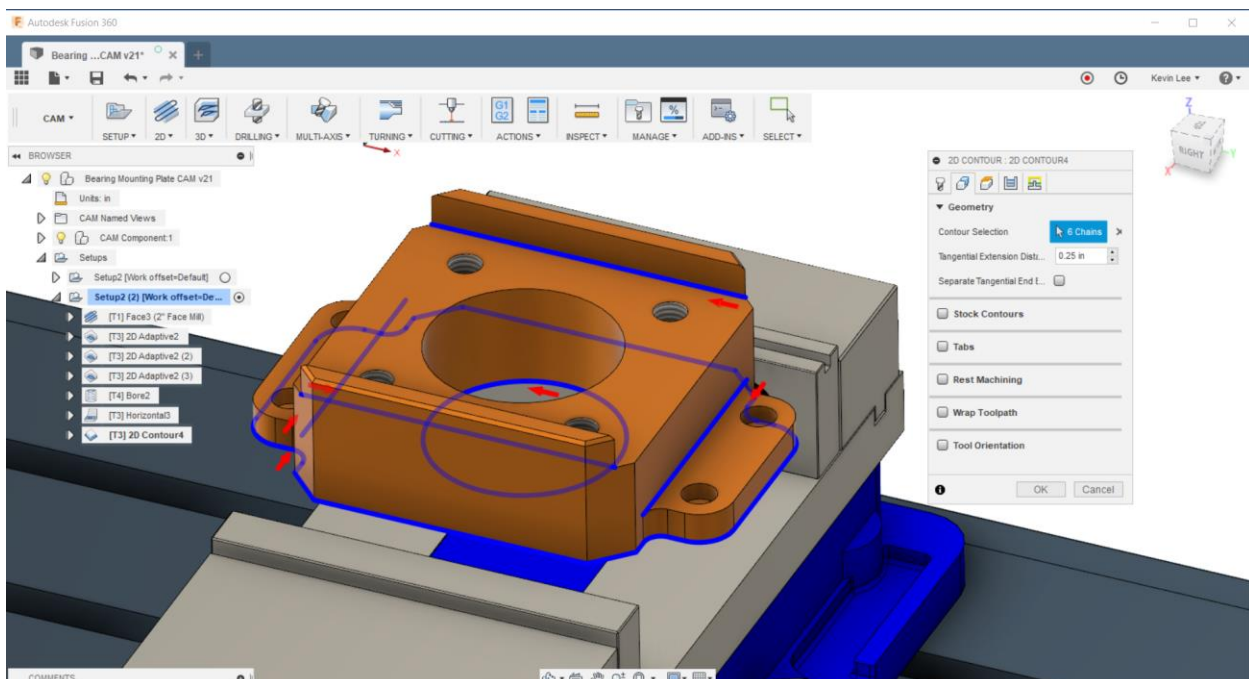


Step 7: Create a Finish Contour Operation

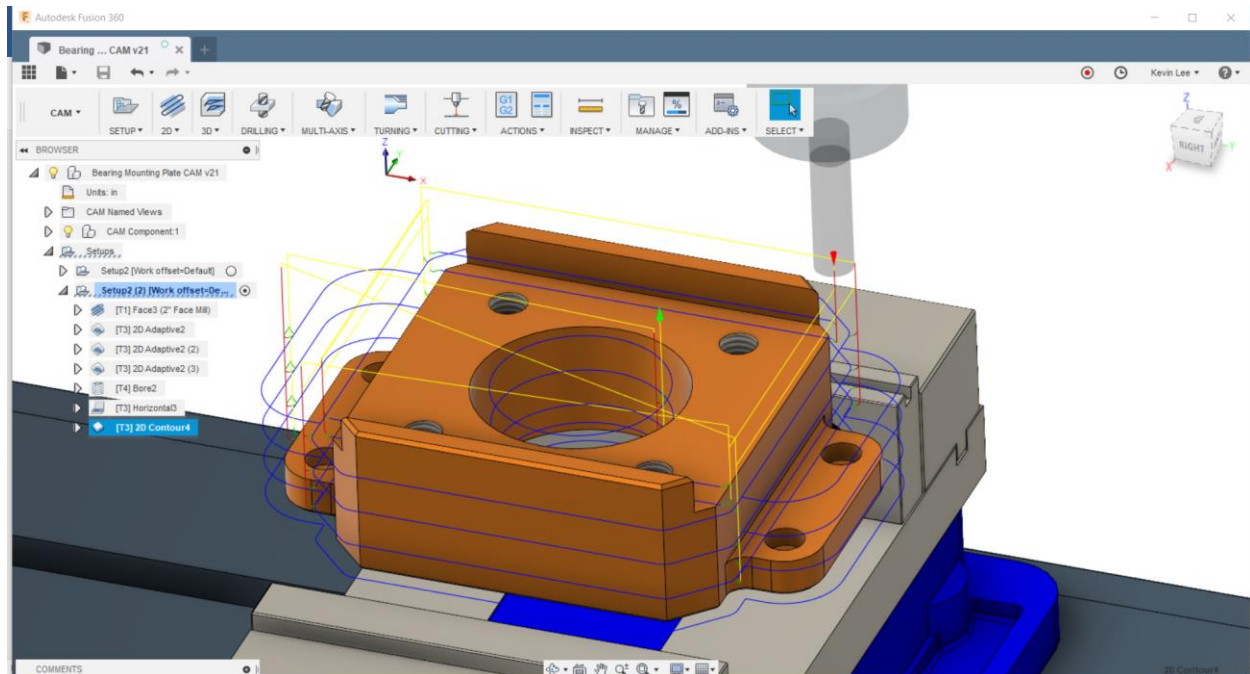
This 2D toolpath operation creates toolpaths based on a 2d contour. Contours can be open or closed and can be on different Z levels, but each contour is flat (2D). You can choose multiple roughing and finishing passes and multiple depth cuts for any contour.



Select all six 2d contour edges that require finishing as shown below. Turn on multiple depths set to 0.5" and enable radius wear compensation under compensation type drop down menu.

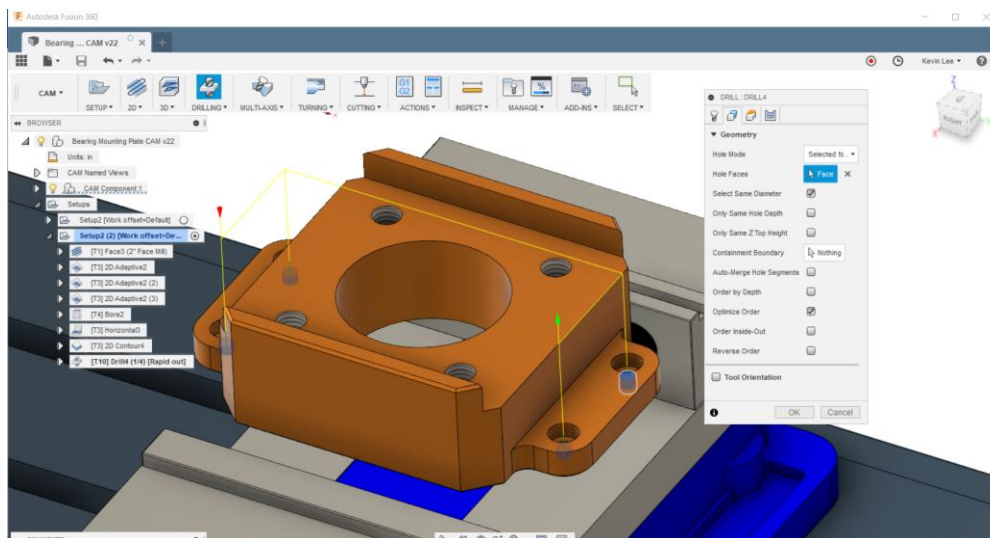


The finish contour operation should look similar to the image below.



Step 8: Create a Drilling Operation

Create a new drilling operation under the drilling drop down and select a 1/4" drill from the sample library.



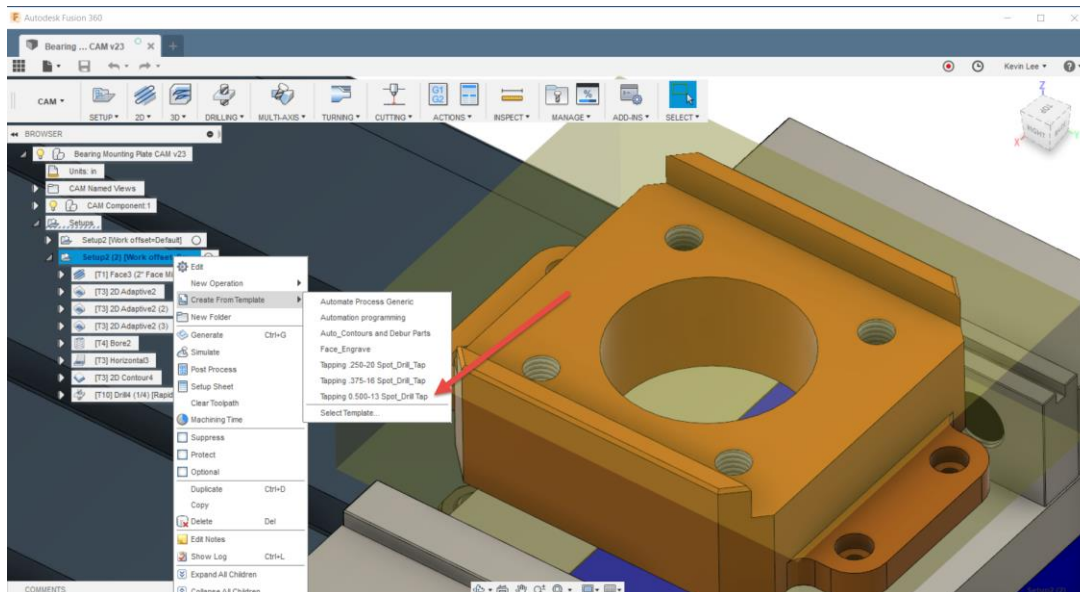
In the geometry tab select a cylindrical face of one hole and enable the “Select Same Diameter” parameter in the geometry tab. With this enabled, a single selection can detect all matching holes. In the heights tab, enable “Drill Tip Through bottom” and select Ok to confirm toolpath.

Step 9: Deploy a Strategic Spot, Drill, Tapping “Template”

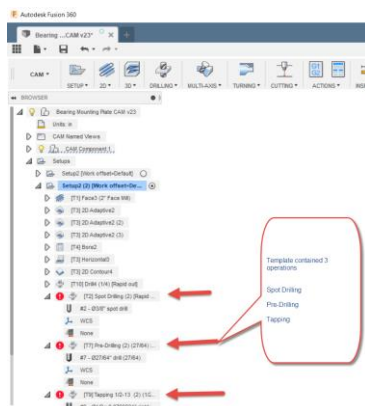
Leverage Fusion 360 templates to quickly deploy strategic toolpath operations. Templates are pre-recorded toolpath operations that can be saved with a unique name and be re-deployed on new parts that need similar toolpath operations programmed.

Using templates in the Fusin 360 Cam workspace can be the “Game changer” that will help you automate some of your programming operations.

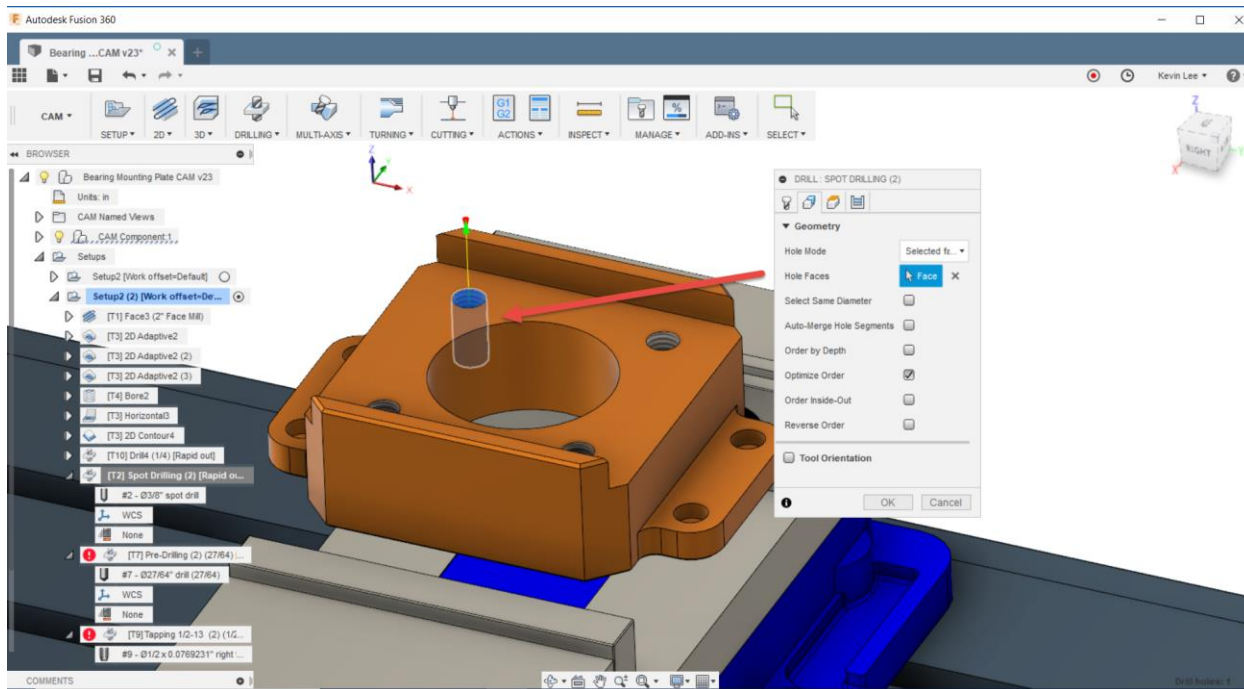
To deploy a prerecorded template, right click on your active setup and select create from template. In the drop down menu select the template named “Tapping 0.500-13 Spot_Drill Tap”.



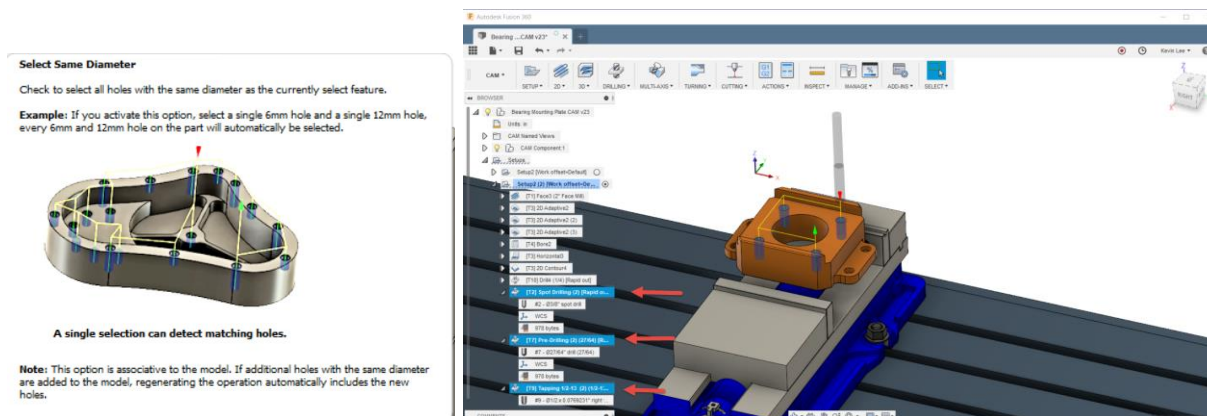
Once selected, the toolpath operations contained in the template will be added to your toolpath operation order found in your setup. These 3 operations are “red flagged” showing that they require geometry to be re-selected and to be regenerated.



On each operation, move into the Geometry Tab and select the cylindrical face of one hole.



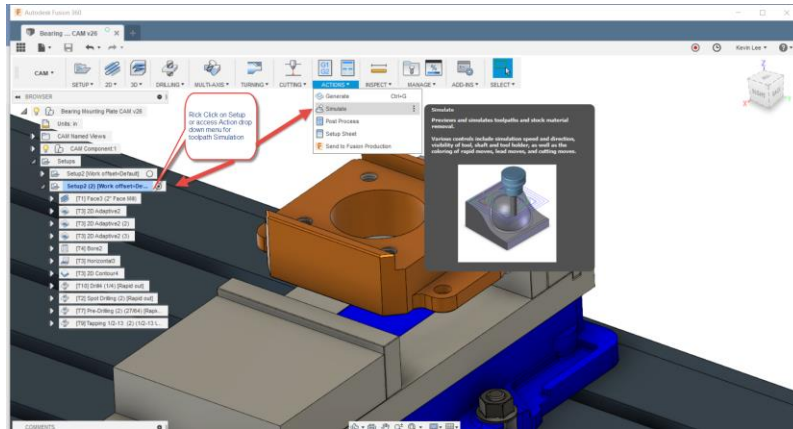
Note, by checking “Select Same Diameter” Fusion360 will select all the holes with the same diameter as the currently selected face. Select OK to complete operation for regeneration.



Repeat this face selection process in the geometry tab for both the pre-drill and tapping operations.

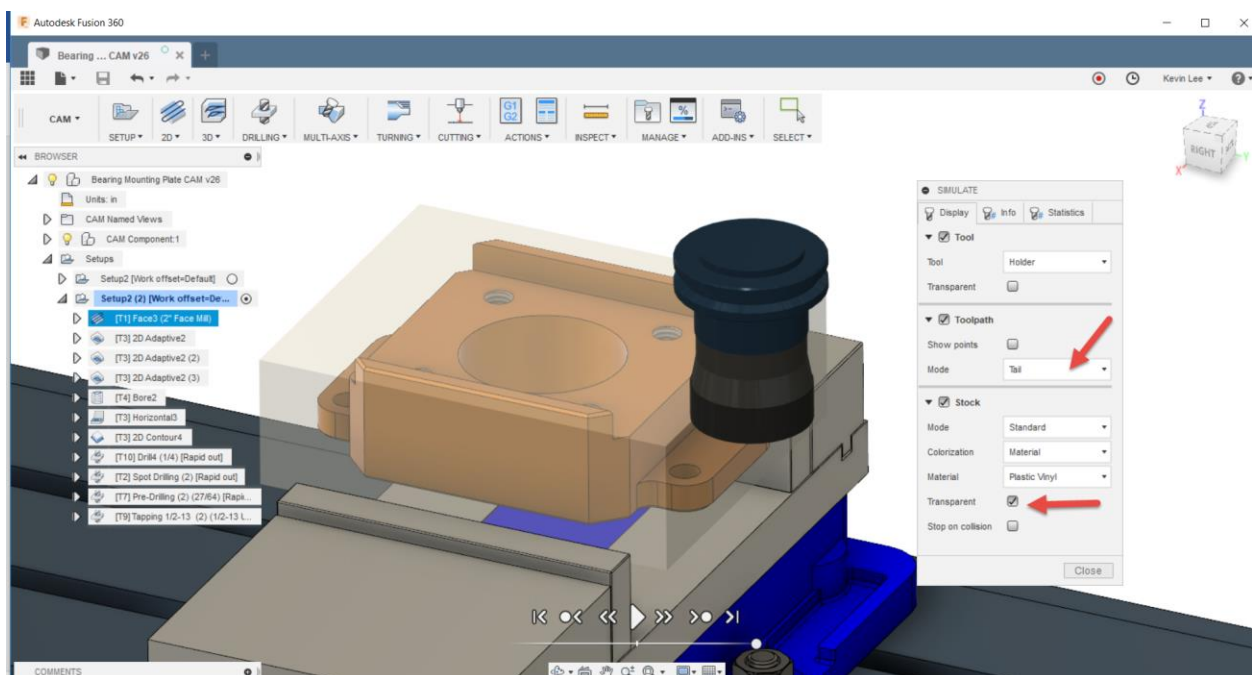
Step 10: Verify Toolpaths with Stock Simulation

With toolpath operations complete we can now Simulate them to verify there are no collisions. There are two ways to access toolpath Simulation. You can right click on the setup itself or select the Actions drop down menu to simulate toolpaths.

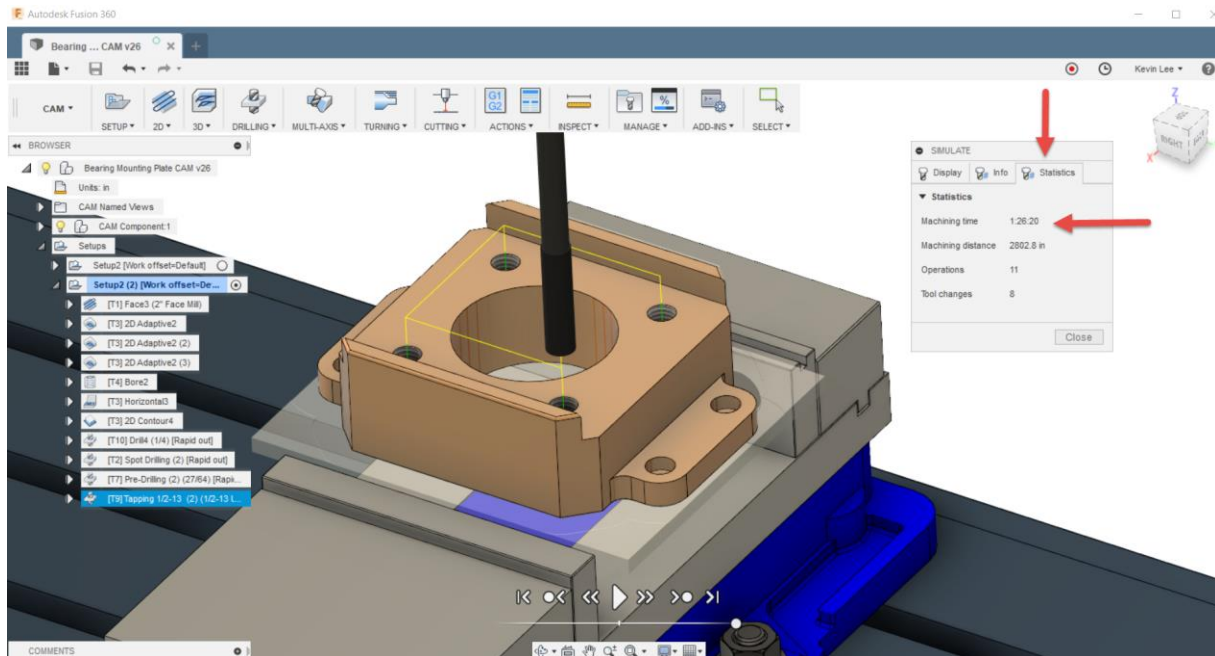


The Simulation menu has several parameters for controlling the visual simulation of the toolpaths. Selecting “transparent stock” and toolpath mode set to “Tail” allows for a clean simulation showing comparison between cad model and machined part.

Select the play button and simulate all toolpaths.

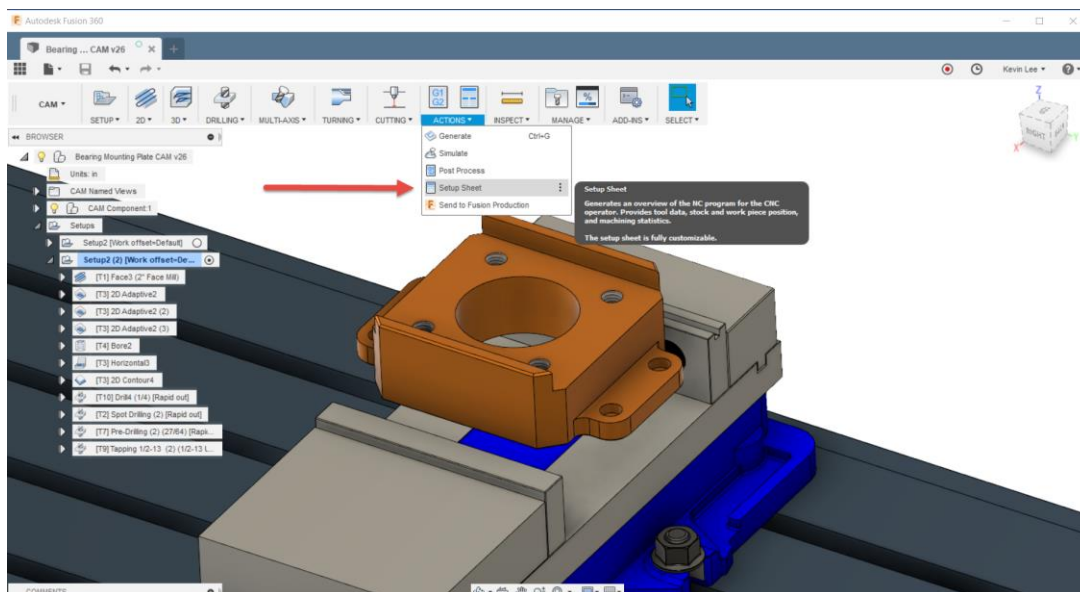


With Simulation completed we can move to the Statistics tab in the Simulate menu to review machining run time.



Step 11: Create a Setup Sheet

Under the Actions drop down menu, select Create a Setup Sheet. Setup sheets generate an overview of the NC program for the operator. It provides a screen capture of your setup, tool data, stock and work piece position and machining statistics.



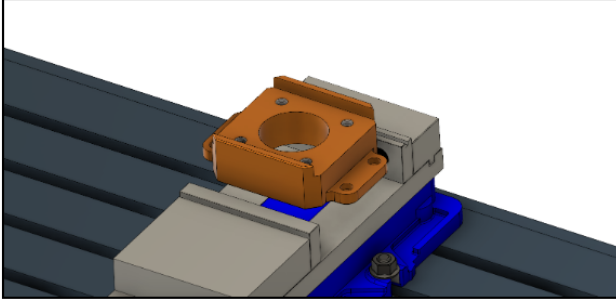
Setup sheets are extremely helpful in capturing the important tooling data used in toolpath operations.

Setup Sheet for Program 001

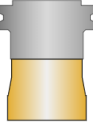
PROGRAM COMMENT: Test Fusion

JOB DESCRIPTION: Setup2

DOCUMENT PATH: Bearing Mounting Plate CAM v26

Setup	
<p>WCS: #0</p> <p>STOCK:</p> <p>DX: 7in</p> <p>DY: 6in</p> <p>DZ: 2.25in</p> <p>PART:</p> <p>DX: 7in</p> <p>DY: 5in</p> <p>DZ: 2in</p> <p>STOCK LOWER IN WCS #0:</p> <p>X: 0in</p> <p>Y: -6in</p> <p>Z: -2.25in</p> <p>STOCK UPPER IN WCS #0:</p> <p>X: 7in</p> <p>Y: 0in</p> <p>Z: 0in</p>	

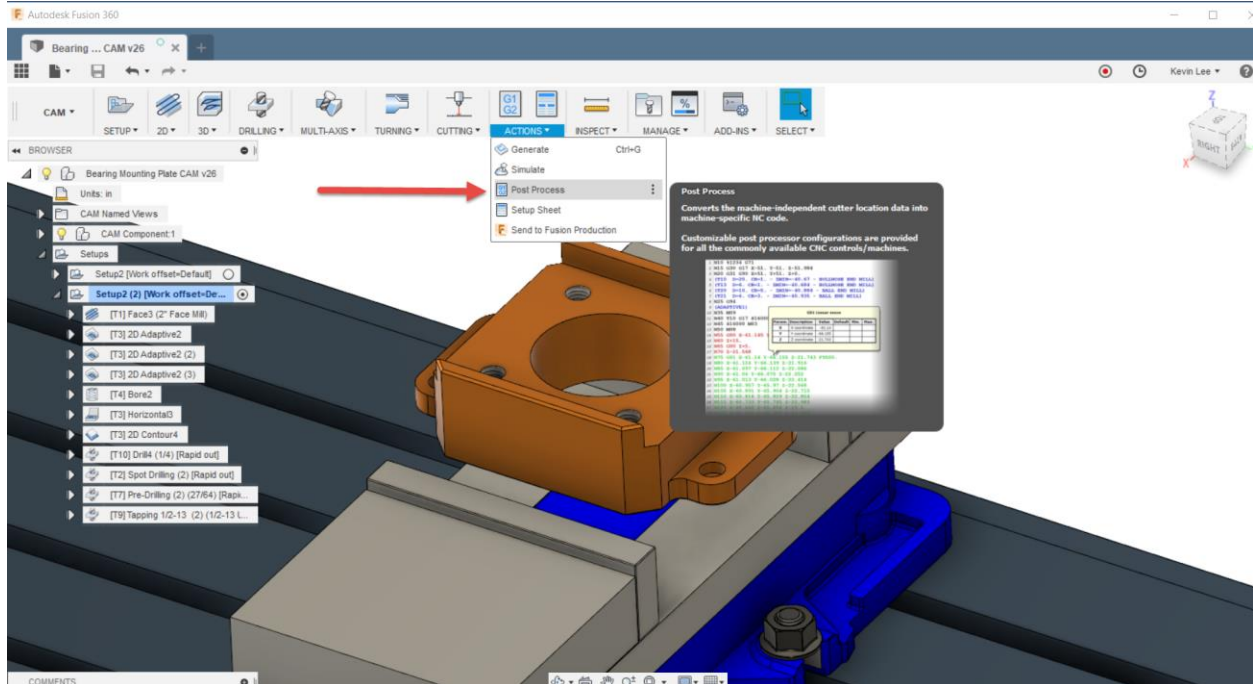
Total
<p>NUMBER OF OPERATIONS: 21</p> <p>NUMBER OF TOOLS: 8</p> <p>TOOLS: T1 T2 T3 T4 T7 T9 T10 T19</p> <p>MAXIMUM Z: 0.969in</p> <p>MINIMUM Z: -2.105in</p> <p>MAXIMUM FEEDRATE: 65in/min</p> <p>MAXIMUM SPINDLE SPEED: 6500rpm</p> <p>CUTTING DISTANCE: 3563.745in</p> <p>RAPID DISTANCE: 807.967in</p> <p>ESTIMATED CYCLE TIME: 2h:10m:53s</p>

Tools			
T1 D1 L1 TYPE: face mill DIAMETER: 2in LENGTH: 1.563in FLUTES: 5 DESCRIPTION: 2" Face Mill VENDOR: Maritool PRODUCT: MSAP16-D050A05R-25.4	MINIMUM Z: -0.03in MAXIMUM FEED: 65in/min MAXIMUM SPINDLE SPEED: 6500rpm CUTTING DISTANCE: 74.696in RAPID DISTANCE: 1.66in ESTIMATED CYCLE TIME: 1m:9s (0.9%)	HOLDER: Maritool CAT40-FMA1.0-1.5M VENDOR: Maritool PRODUCT: CAT40-FMA1.0-1.5M	
T2 D2 L2 TYPE: spot drill DIAMETER: 0.375in TIP ANGLE: 118° LENGTH: 1.181in FLUTES: 3	MINIMUM Z: -0.605in MAXIMUM FEED: 19.685in/min MAXIMUM SPINDLE SPEED: 6500rpm CUTTING DISTANCE: 2.2in RAPID DISTANCE: 30.311in ESTIMATED CYCLE TIME: 7s (0.1%)	HOLDER: BT40 - B4C4-0016	
T3 D3 L3 TYPE: flat end mill DIAMETER: 0.5in LENGTH: 2.25in FLUTES: 3	MINIMUM Z: -2.03in MAXIMUM FEED: 30in/min MAXIMUM SPINDLE SPEED: 6500rpm CUTTING DISTANCE: 3424.295in RAPID DISTANCE: 555.083in ESTIMATED CYCLE TIME: 2h:3m:54s (94.7%)	HOLDER: Maritool CAT40-ER32-2.35 VENDOR: Maritool PRODUCT: CAT40-ER32-2.35	
T4 D4 L4 TYPE: flat end mill DIAMETER: 0.25in LENGTH: 2in FLUTES: 3	MINIMUM Z: -1.78in MAXIMUM FEED: 40in/min MAXIMUM SPINDLE SPEED: 6500rpm CUTTING DISTANCE: 5.458in RAPID DISTANCE: 50.333in ESTIMATED CYCLE TIME: 13s (0.2%)	HOLDER: Maritool CAT40-ER32-2.35 VENDOR: Maritool PRODUCT: CAT40-ER32-2.35	
T7 D7 L7 TYPE: drill DIAMETER: 0.422in TIP ANGLE: 118° LENGTH: 4.319in FLUTES: 1 DESCRIPTION: 27/64	MINIMUM Z: -1.657in MAXIMUM FEED: 29in/min MAXIMUM SPINDLE SPEED: 6500rpm CUTTING DISTANCE: 10.811in RAPID DISTANCE: 100.935in ESTIMATED CYCLE TIME: 24s (0.3%)	HOLDER: Maritool CAT40-APU13 Drill Chuck VENDOR: Maritool PRODUCT: CAT40-APU13	

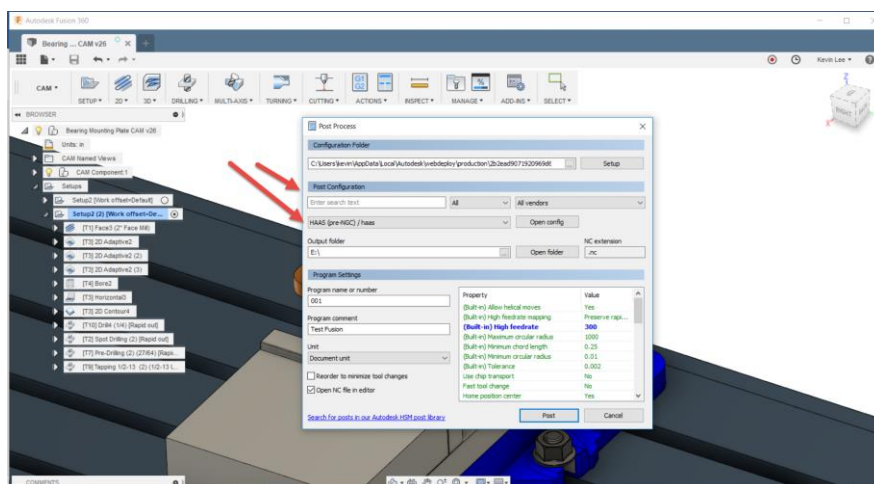
Step 12: Post Process G-Code

Post processing essentially captures all of the toolpath data/parameters and converts them into machine specific NC code. This NC code is commonly known as G&M code language that most machines controllers use to machine parts.

Under the Actions drop Down menu, select Post Process.



Under the post configuration drop down menu, select an appropriate processor that will match your machine controller.

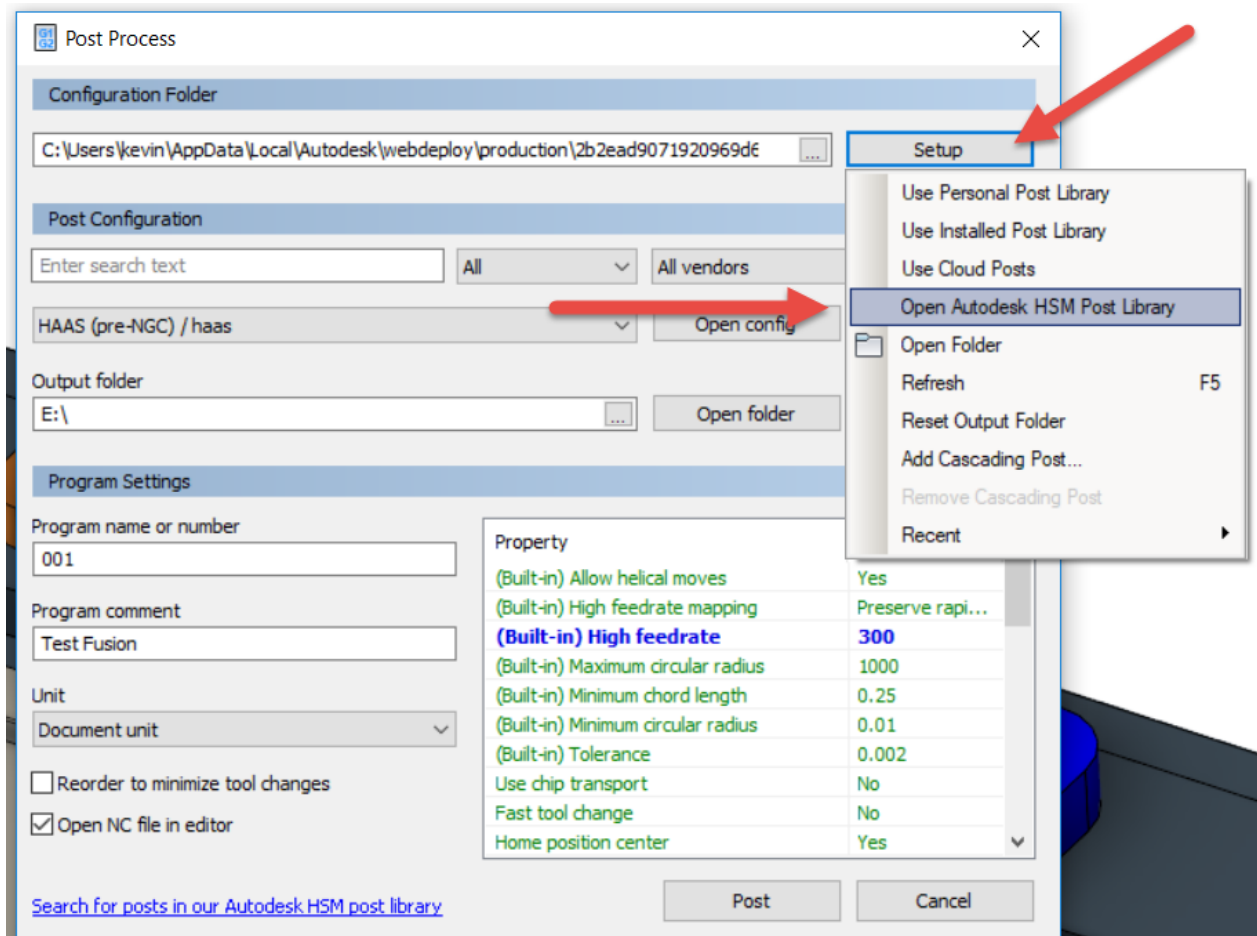


Fusion 360 comes standard with many post processors available for a wide variety of machine specific controllers. To ensure your using the most up to date post processor; select Setup.

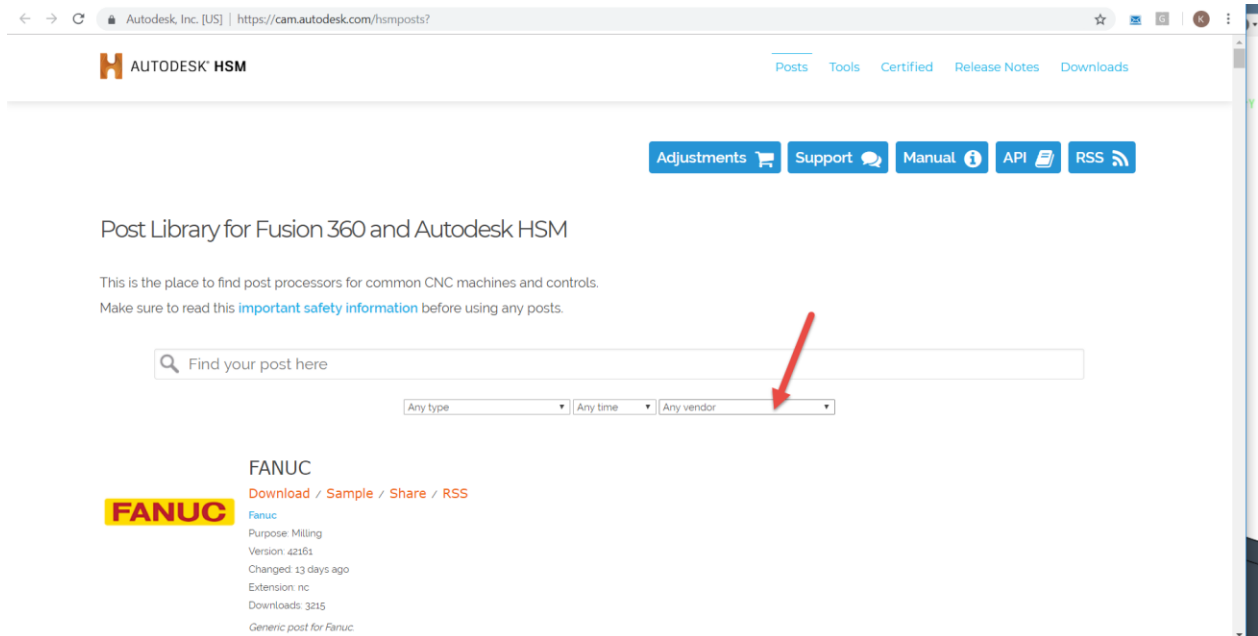
This will open a drop-down list where you can Open Autodesk HSM Post Library.

This web link can be used to access the Autodesk HSM Post Library

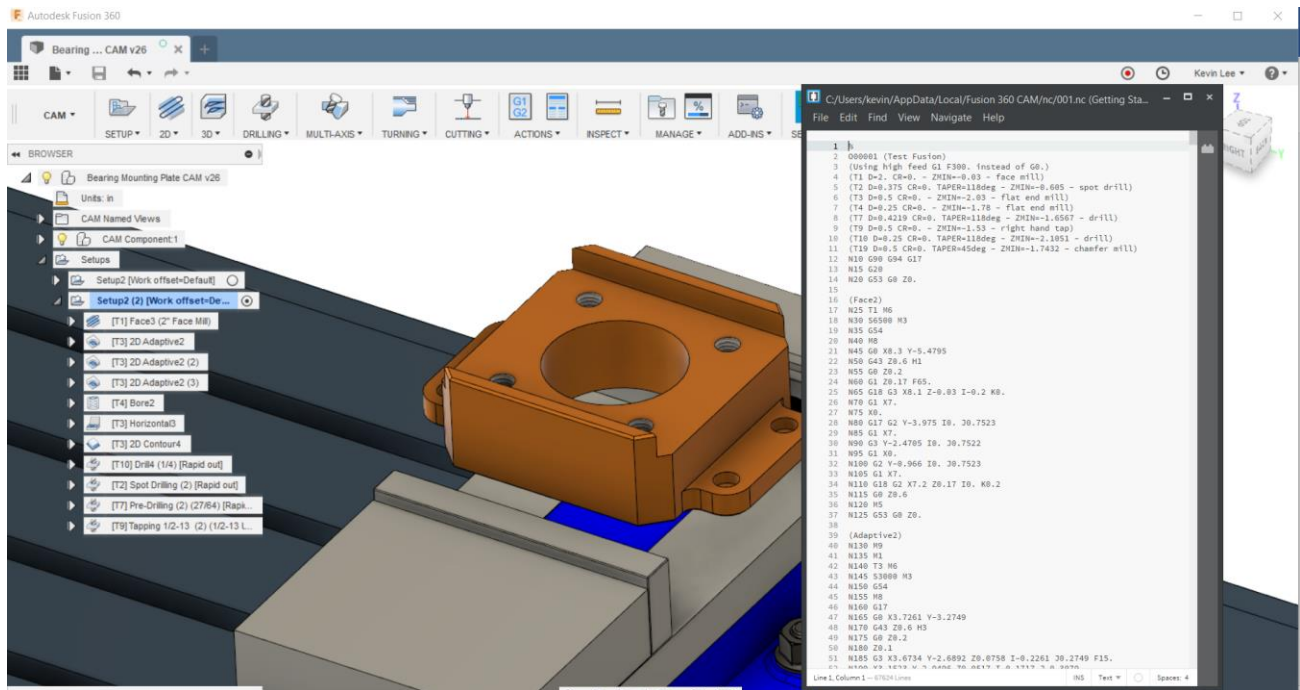
<https://cam.autodesk.com/hsmposts>



Here you will find a drop-down list where you can select a machine manufacturing vendor to view all available HSM post processors that can be machine model specific.



Once you have selected a Post Processor a NC Editor will launch to show you the NC program that you can upload to your machine controller.



THANK YOU

Thank you to everyone who have registered for my AU2018 class focused on the:

Fundamentals of the CAM Workspace Using Fusion360: Learn 2D and 3D Toolpaths.

Hopefully you found these 12 basic steps helpful in understanding the fundamentals of integrated cam with Fusion 360 and the workflow inside the CAM workspace.

For further assistance and Fusion 360 Cam support your welcome to reach out directly through our website at www.leeverage.ca or by email at info@leeverage.ca.



Follow us on
Instagram

@leeverageintegration



Like us
on
Facebook

@LeeVergeIntegration

