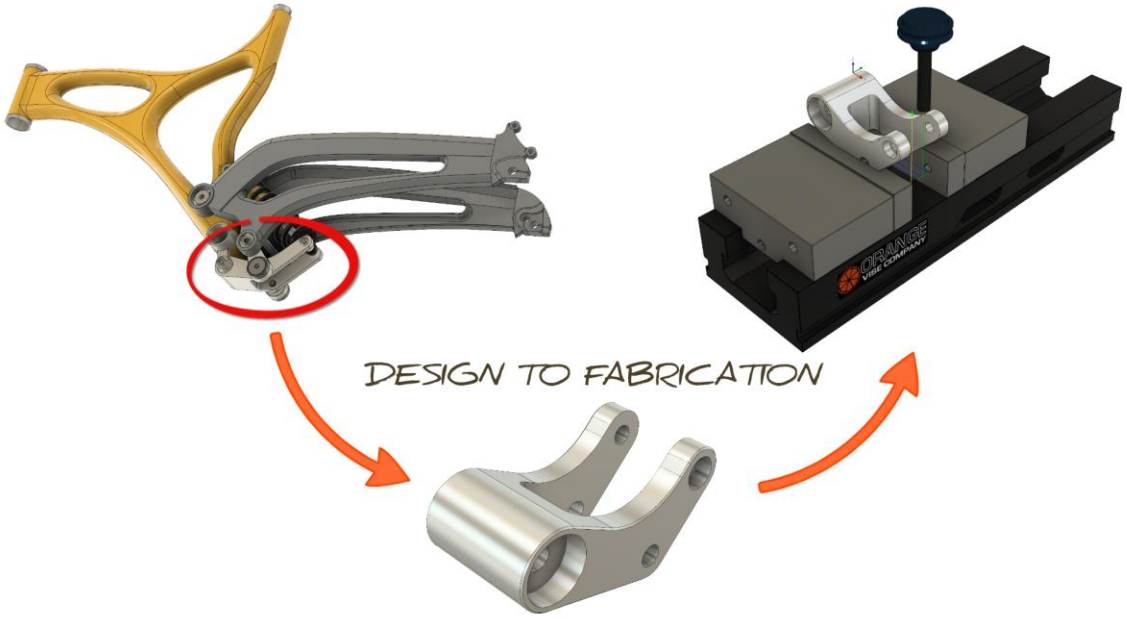
## Overview

CNC milling toolpaths are broadly classified as either 2D, 3D, 4-axis, and 5-axis, depending on the number of axes involved and how they move. The term, 2D, is a bit of a misnomer because all modern CNC machines control at least three axis and all three axes move at one time or another for every 2D machining operation. A more accurate term, 2-1/2D, is commonly used in CNC manufacturing.




## Learning Objectives

In this section you will learn how to:

- Create setups
- Apply 2D operations
  - Face
  - 2D adaptive clearing
  - 2D contour
  - Chamfer milling
  - Bore
- Simulate toolpaths and stock material removal
- Produce setup sheets
- Product NC code via post processing

## 2D vs. 3D Defined

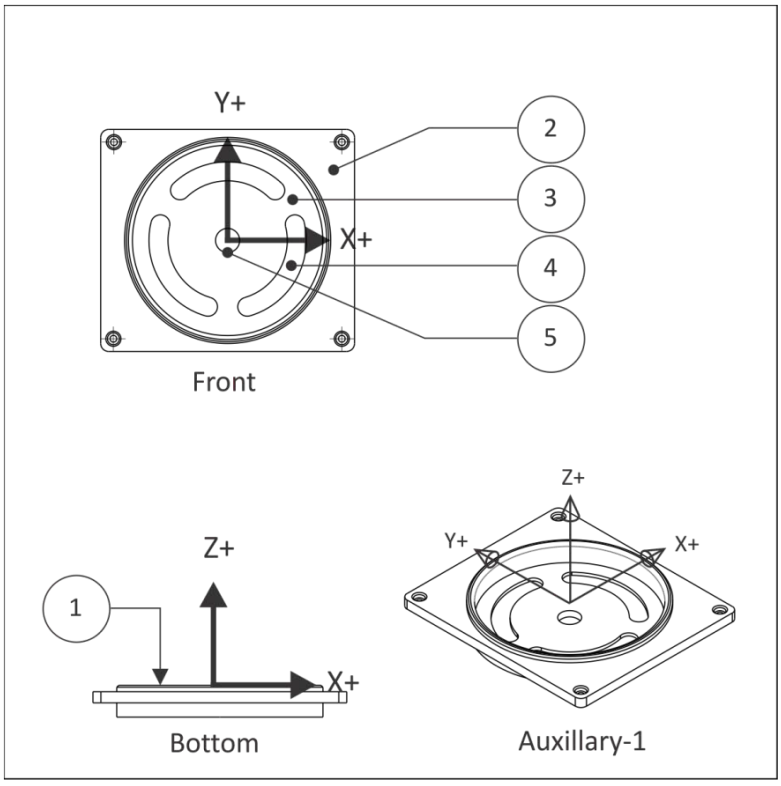
### 2D (Prismatic) Parts

2-1/2D milling toolpaths machine only in the XY plane. The Z axis is used only to position the tool at depth. The move to the cutting plane is a straight down feed, rapid, ramp or helical feed move.

What do you mean by 2-1/2 axis programming? All cutting happens on the XY plane. Z is simply used to position the tool to depth. We actually over-deliver, in that we support things like tracing a 3D path and thread milling.

The term prismatic is a term commonly used in engineering to describe 2-1/2D parts. There are, however, prismatic parts that require 4 or 5 axis machining, so the term is used in machining only to describe parts where all machined faces lie normal to the machine tool spindle. The XY axes are normal to the machine spindle and Z is used only to position the tool to depth (either in a feed or rapid motion).

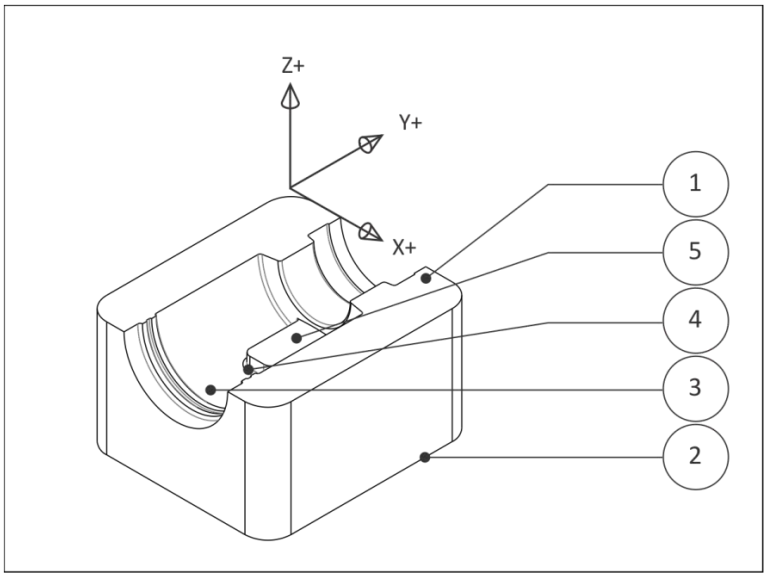
The image below shows a prismatic part. All machined features lie parallel to the XY plane. Each Z-level is machined by positioning the tool at a fixed Z-level and then moving the XY axes to remove material. Every feature can be reached with the tool approaching either from the Front or Bottom views. There are several cutting planes in this example, including the model top (1), top of the face where the holes start (2), the bottom of the pocket (3) where the slots begin, the bottom of the slots (4), and the bottom of the hole through the center (5).



**3D Parts**

3D refers to non-prismatic parts, including molds and complex organic shapes. Most consumer goods, for example, include 3D features. The image below shows half of a stamping die. This part is typical in that it includes both 3D and 2D features. The 2D features are the top face (1), and the outside contour (2).

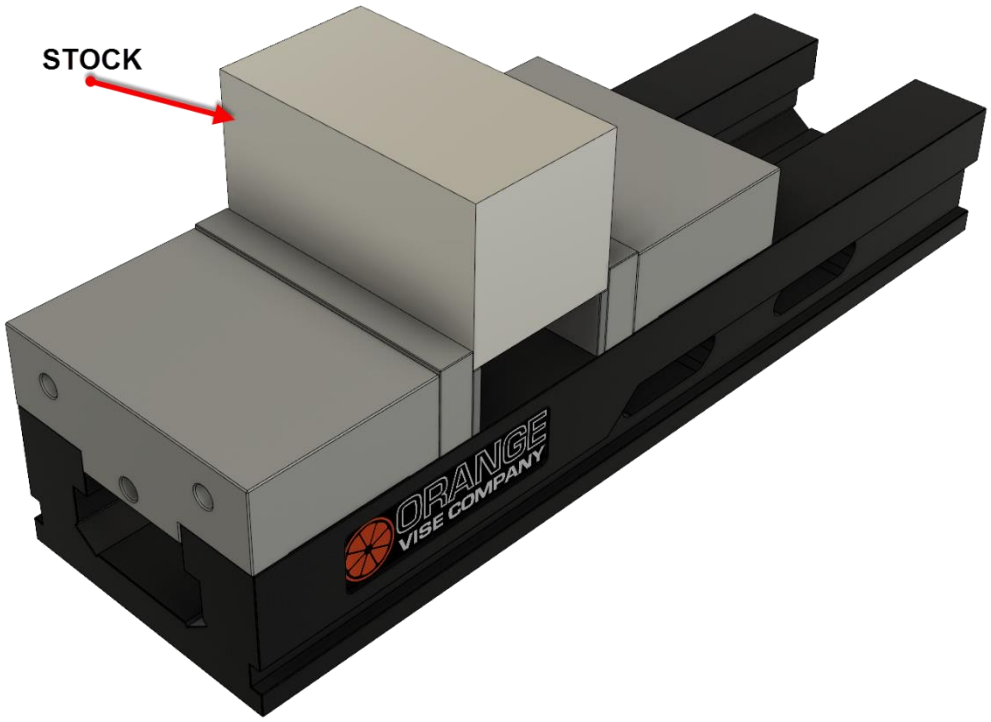
3D features, like the revolved surfaces (3) and fillet (4), require more complex machine motion. The revolved surfaces require XZ tool motion. The fillet requires XYZ tool motion. Even the flat (5) and cavity roughing (though technically planar) require 3D toolpaths because the adjacent revolved surfaces and fillet must be considered to prevent gouging the part. The calculations required to calculate these toolpaths are highly complex.



**Downhill mountain bike rocker arm**

**Objective:** Use the CAM workspace in Fusion 360 and 2-1/2D axis milling techniques to fully machine the provided part. This will require 3 job setups to machine the rocker arm.

Using 2.5 Axis Functionality, we will begin with a block of Aluminum Stock mounted in a Vise (see Figure 1), and will create three (3) 'job setups' that will be required to machine the Rocker Arm.

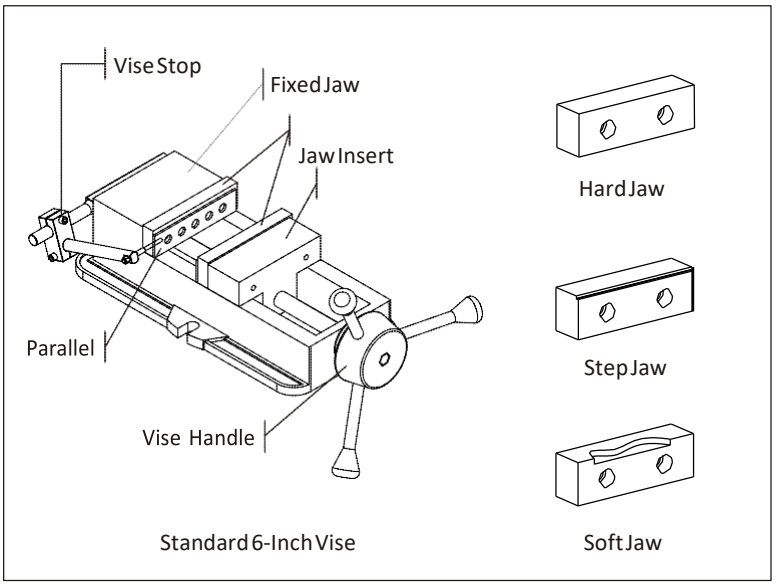


## 1.1.2 Fixture Component Terminology

### Vise and Accessories

The CNC vise is precision engineered and manufactured with components ground flat and perpendicular to within .0002 inches. The most common is referred to as a six inch (6”) vise, because the width of the jaws is six inches.

Once the vise is bolted to the table and aligned, parts are loaded into the vise and clamped by closing the jaws. The vise can exert tremendous force, so care is taken not to over-tighten the vise and deform fragile parts. Vise pressure must be appropriate to the part being held and expected cutting forces.

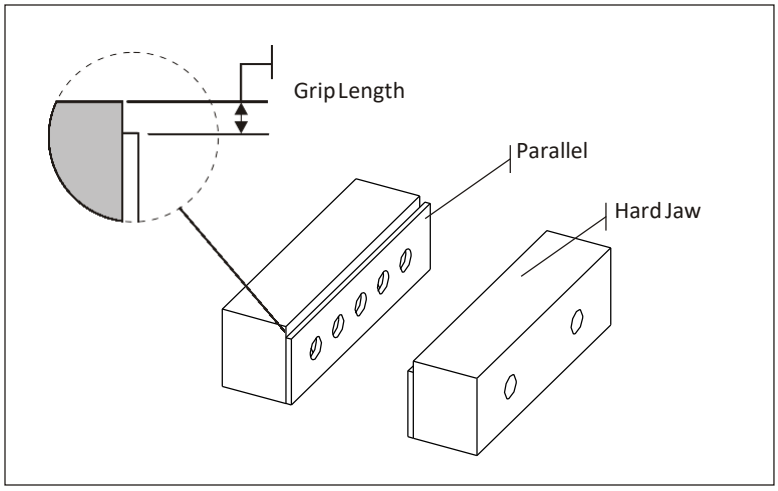



The **Fixed Jaw** remains stationary. The **Moving Jaw** opens when the **Vise Handle** is turned. It is a good practice to remove the vise handle after the jaws are closed and before running the program. This is done by sliding the handle off.

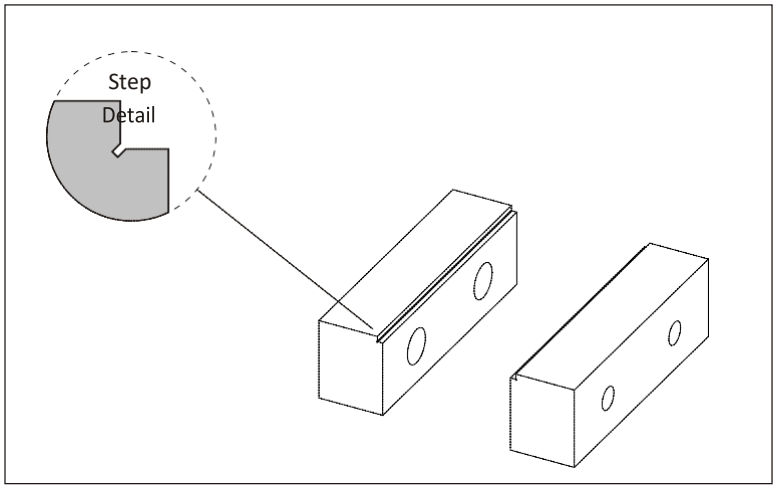
A **Vise Stop** is a device that allows the parts to be loaded into the vise precisely. This image shows a style of vise stop that is particularly useful because it is adjustable up-down and left-right.

**Hard Jaws** are made of hardened steel and precision ground on all sides. They are usually used along with parallels.

**Parallels** are thin steel plates, available in various widths, used to set the grip length of the vise jaws.

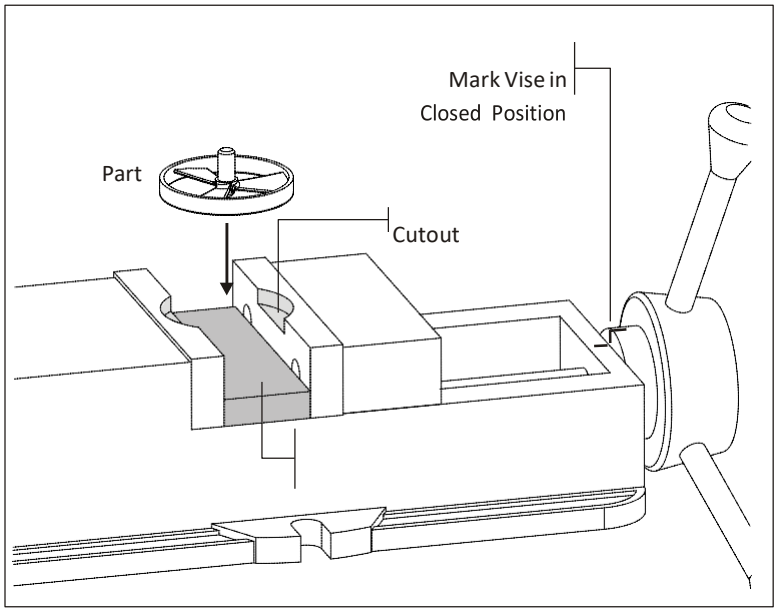



**Step jaws** are similar to hard jaws but include a step feature that eliminates the need for parallels.



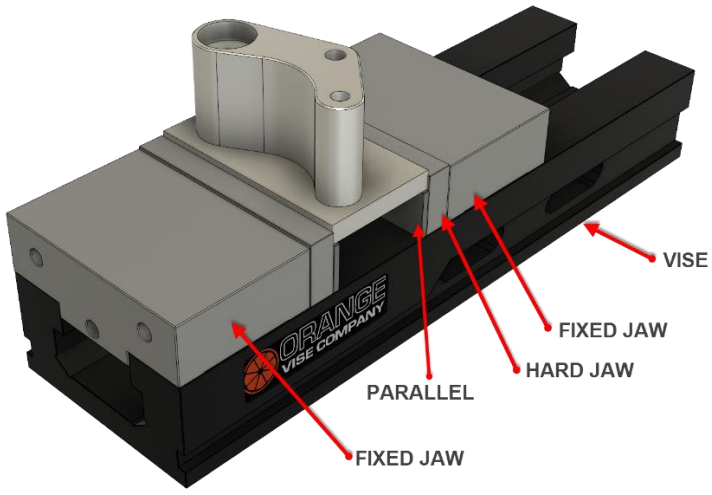
**Soft jaws** are blanks of aluminum used to grip parts that cannot be held using hard jaws. A cutout the same shape as the part is machined into the soft jaws to grip irregular shapes.

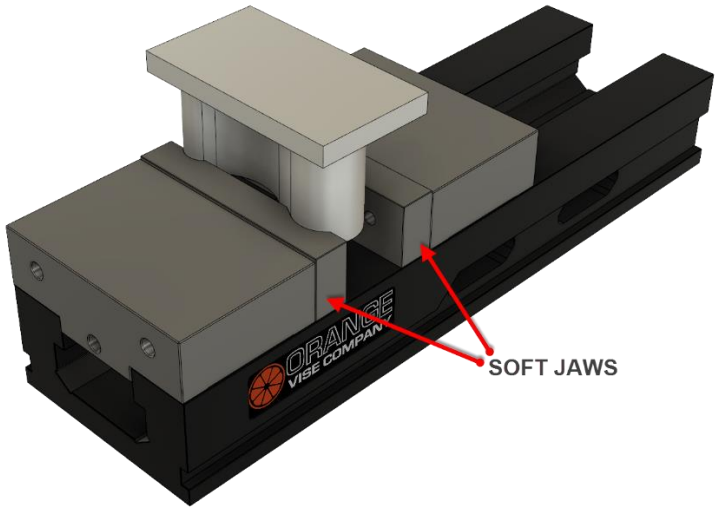
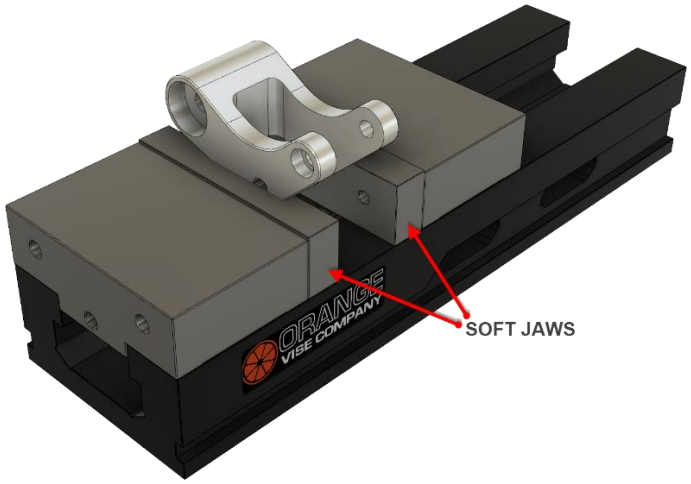
When machining the cutout, place a bar between the jaws to set the correct spacing. Use a torque wrench or mark the vise so it can be closed with the exact same pressure each time a new part is loaded. Remove the spacer before clamping the part.



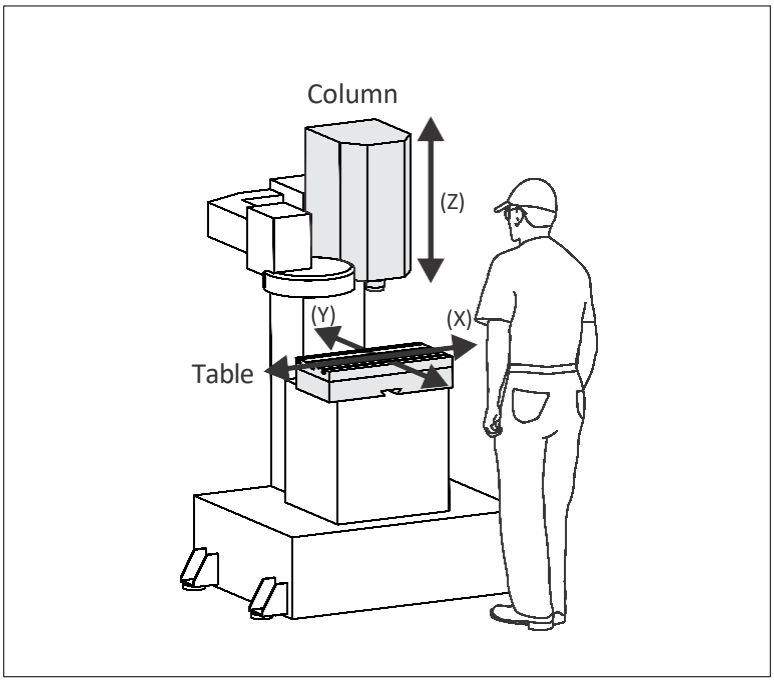

In Setup 1, we will:

1. Setup up a Job
2. Apply a multiple operations
3. Visit the Tool Library
4. Show Stock Simulation



| | |
|---|---|
| In Setup 2, we will:<br><br>1. Setup up a new Job<br>2. Apply a multiple operations<br>3. Visit the Tool Library<br>4. Show Stock Simulation |  |
| In Setup 3, we will:<br><br>1. Setup up another Job<br>2. Apply a 2D Pocket Operation<br>3. Visit the Tool Library<br>4. Show Stock Simulation<br><br>After all setups are created, we will create a Setup Sheet and output to a HAAS Post Processor. |  |

# Vertical Milling Center (VMC) Machine Terminology




CNC machines use a 3D Cartesian coordinate system. Figure 10 shows a typical.

Material to be machined is fastened to the machine table. This table moves in the XY-Plane. As the operator faces the machine, the X-Axis moves the table left-right. The Y-Axis moves the table forward-backward.

The machine column grips and spins the tool. The column controls the Z-axis and moves up-down.

# Work Coordinate System Terminology

To make programming and setting up the CNC easier, a Work Coordinate System (WCS) is established for each CNC program.

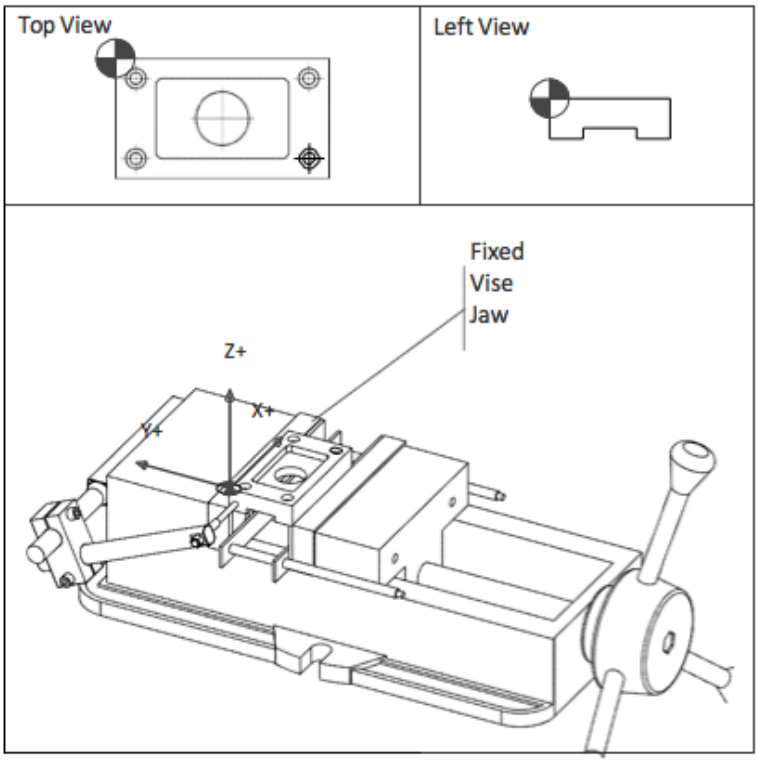
The WCS is a point selected by the CNC programmer on the part, stock or fixture. While the WCS can be the same as the part origin in CAD, it does not have to be. While it can be located anywhere in the machine envelope, its selection requires careful consideration.

The WCS location must be able to be found by mechanical means such as an edge finder, coaxial indicator or part probe.

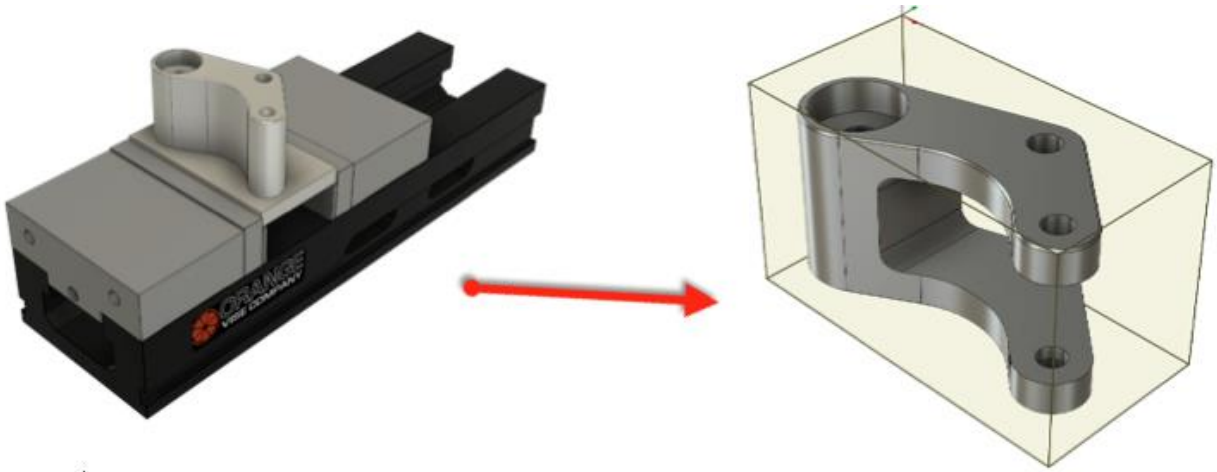
- It must be located with high precision: typically plus or minus .001 inches or less.
- It must be repeatable: parts must be placed in exactly the same position every time.
- It should take into account how the part will be rotated and moved as different sides of the part are machined.

The image below shows a part gripped in a vise. The outside dimensions of the part have already been milled to size on a manual machine before being set on the CNC machine.

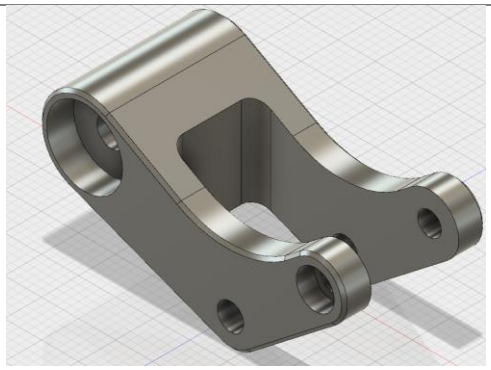
The CNC is used to make the holes, pockets, and slot in this part. The WCS is located in the upper-left corner of the block. This corner is easily found using an Edge Finder or Probe.



# SETUP 1

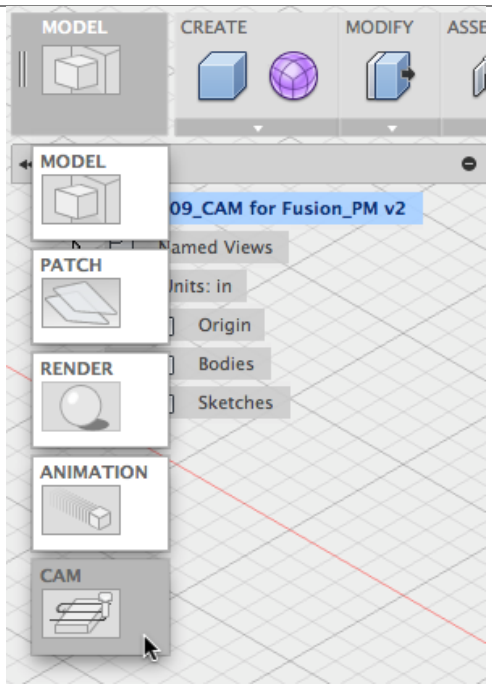


**Create the first setup**: In this section, you go to the CAM workspace, create a setup, then set your stock.

| | |
|---|---|
| Step 1 – Open the design<br><br>1. Open the Data Panel by clicking on the icon located at the top left of the menu bar. The Data Panel will slide open.<br>2. Double-click **09_CAM for Fusion** to open the design. |  <br><br>LAUNCH VIDEO |

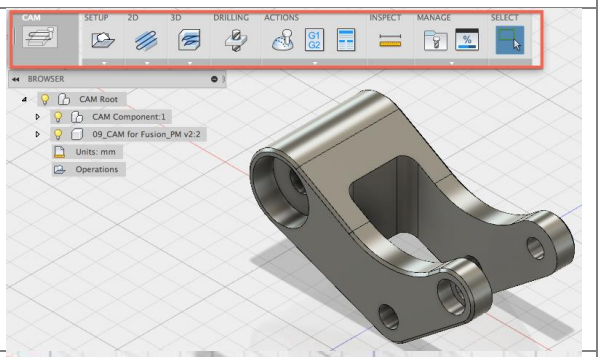
Step 2 – Go to the CAM workspace

1. Hover the mouse over the workspace switcher where it shows **MODEL**.
2. Click **CAM** to switch to the CAM workspace.



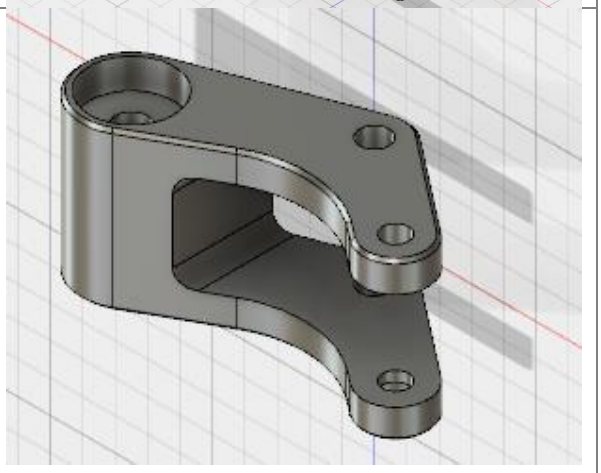

Step 3 – Verify the workspace

1. You can tell which workspace is active by looking at the toolbar. The toolbar should have CAM commands listed.



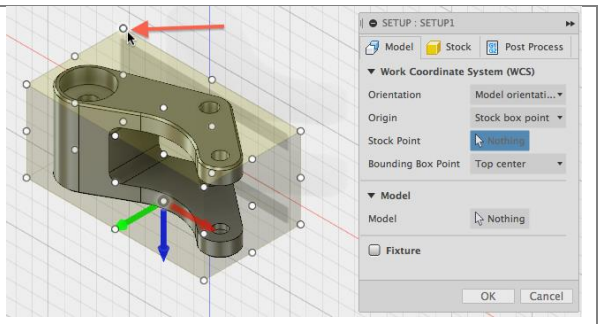

Step 4 – Orient the model

1. Use the ViewCube or the navigation commands to position the model as shown.



Step 5 – Create a setup and set the stock point

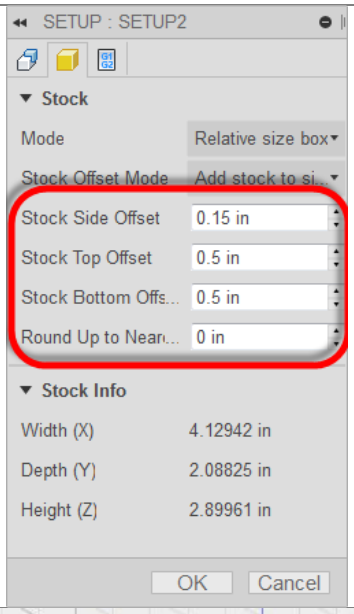
1. Click **Setup > Setup**.
2. Select **Stock Point**.
3. Click the **upper stock point** shown to move the triad.
4. Click the **head of blue arrow** to flip the orientation. We want the positive Z axis facing up.
5. Verify that the blue arrow (Z axis) is facing up, the green arrow (Y axis) is facing right, and the red arrow (X axis) is facing towards you.




LAUNCH VIDEO

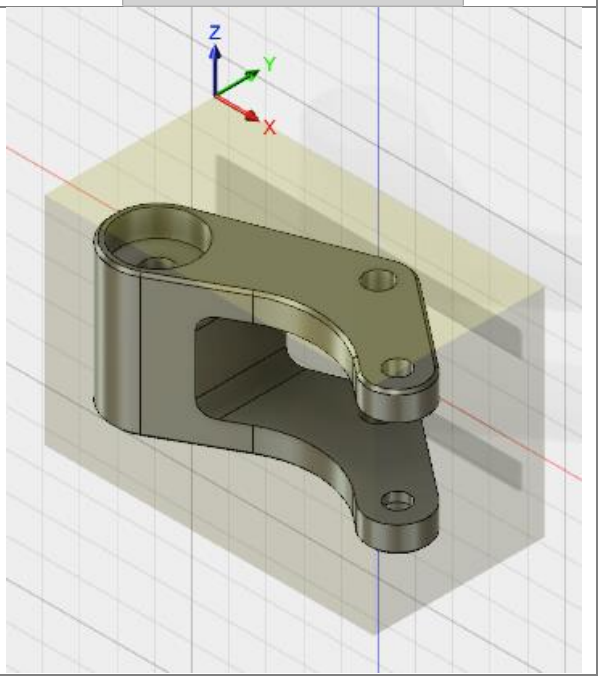
Step 6 – Set the stock size

1. Click the Stock tab.
2. Set the Mode to **Relative Size Box**.
3. Input these values:
   Stock Side Offset: **0.15 in**
   Stock Top Offset: **0.5 in**
   Stock Bottom Offset: **0.5 in**
4. Click **OK.**




Step 7 – Finished setup

You have defined the size of the stock we will be machining.



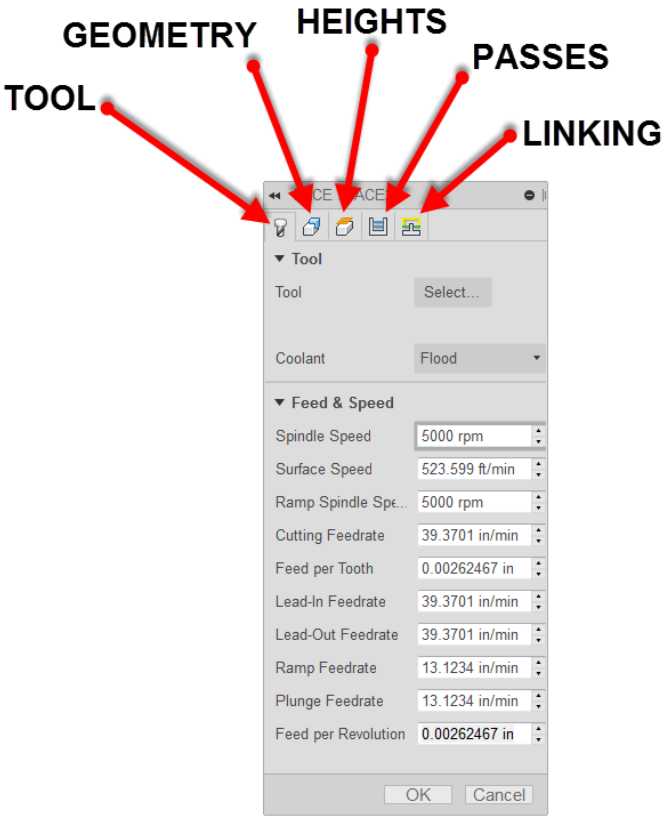
# Toolpath Operations

## Understanding Toolpaths by Type and Use

Before going further, it is helpful to understand how 2D toolpaths are classified in most CAM software. Please refer to the Autodesk CNC Handbook for more elaborate detail.

| Type | Toolpath | Common Uses |
|---|---|---|
| **Face** | Face | • Finish face of part. |
| | Island Facing | • Finish face with open sides and bosses. |
| **2D Contour** | Contour | • Loops.<br>• Partial loops.<br>• Single edges.<br>• Stick (single point) fonts.<br>• Create dovetail, keyset, or saw cut. |
| | Chamfer | • Create chamfer using tapered mill or center drill.<br>• De-burring. |
| | Fillet | • Creating fillet using Corner Round tool. |
| **Pocket** | Pocket | • Remove excess material.<br>• Machining TrueType (outlined) fonts and logos. |
| | Slot Mill | • Straight slot.<br>• Arc slot. |
| **Drill** | Drill | • Create spot drill, drill, tap, bore or reamed hole. |
| | Circular Pocket Milling | • Making holes greater than .75*in* diameter. |
| | Thread Mill | • Create ID threads over .75*in* diameter.<br>• Create milled OD threads of any size. |

# 2D Machining Tab Terminology




**Tool:** Defines the tool being used as well as the feeds and speeds.

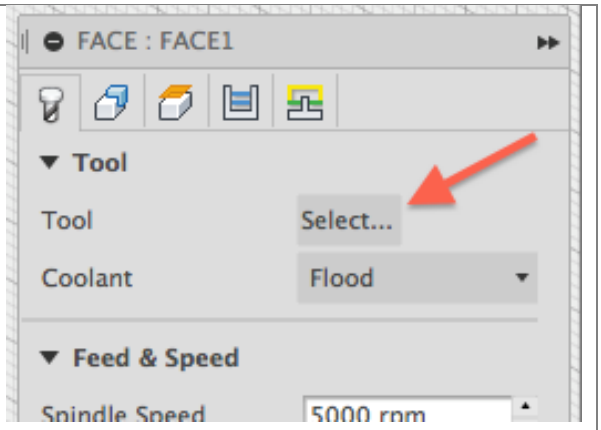
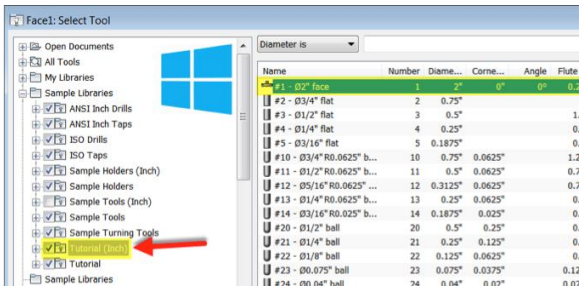
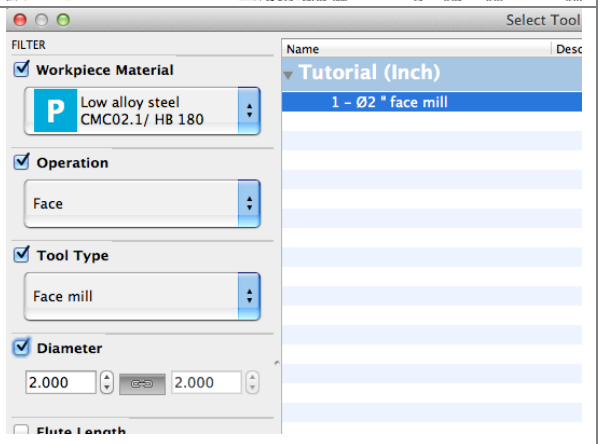
**Geometry:** Defines the geometry being machined.

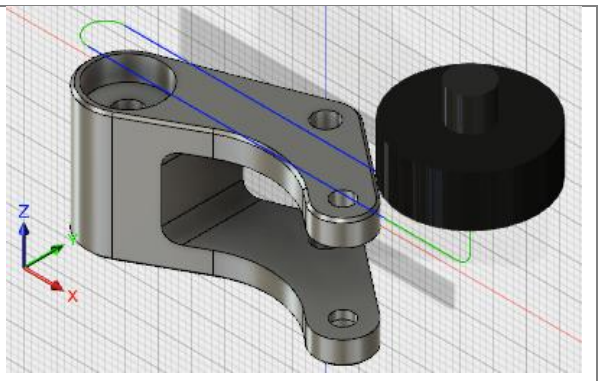
**Heights:** Controls heights the toolpath goes to such as cut depth and retract heights.

**Passes:** Controls how the tool will go about removing material.

**Linking:** Controls how the tool enters/exits and transitions between cutting movements.

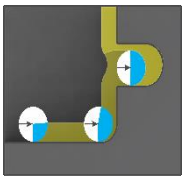
**Face Operation:** In this exercise, you create a face operation.

| | |
|---|---|
| Step 1 – Start the Face command<br><br>1. Click **2D > Face**.<br>2. In the Face dialog box, click **Select** next to **Tool**. |  <br>LAUNCH VIDEO |
| Step 2 – Windows library<br><br>1. Select **2" Face Mill** from the **Tutorials (Inch)** library.<br>2. Click **OK**.<br><br>The face operation automatically recognizes the top of the stock and machines down to the top of the model. |  |
| Step 2 – Mac library<br><br>1. Check the **Diameter** box.<br>2. Enter **2.0** in the field and click the **link** box.<br>3. Select the **2" Face Mill.**<br>4. Click **OK**.<br>5. Click **OK** to accept the tool post processor information.<br><br>The face operation automatically recognizes the top of the stock and machines down to the top of the model. |  |

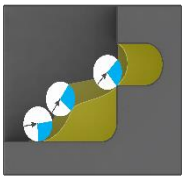
| | |
|---|---|
| Step 3 – Finish the operation<br><br>1. Click **OK** to create the face operation. |  |

**Adaptive Clearing:** Create an adaptive clearing operation to remove material around the outside of your design.

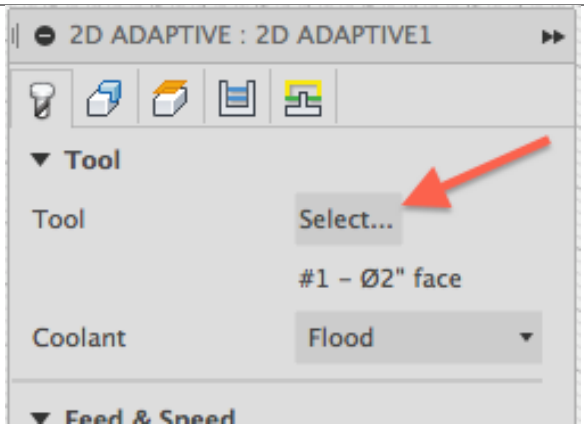
Adaptive clearing calculates paths based on a sophisticated algorithm that constantly considers the remaining material and maintains optimal tool engagement throughout the cut. Since we keep cut load constant, you can push the tool a lot faster and deeper through the material."



LEGACY CLEARING

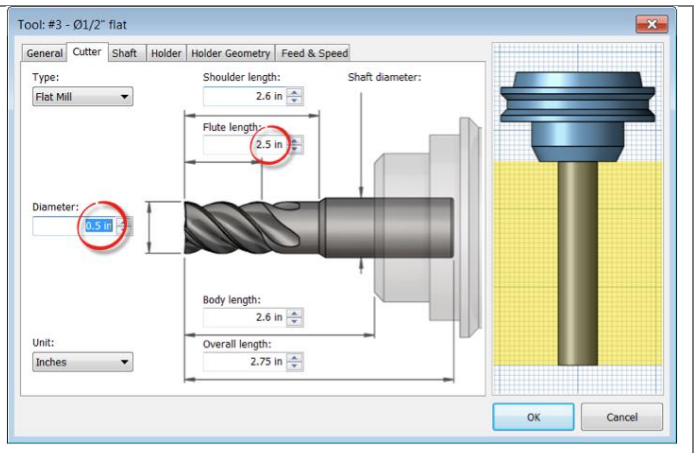


ADAPTIVE CLEARING

| | |
|---|---|
| Step 1 – Start the 2D Adaptive Clearing command<br><br>1. Click **2D > 2D Adaptive Clearing**.<br>2. In the 2D Adaptive dialog box, click **Select** next to **Tool**. |  <br>LAUNCH VIDEO |

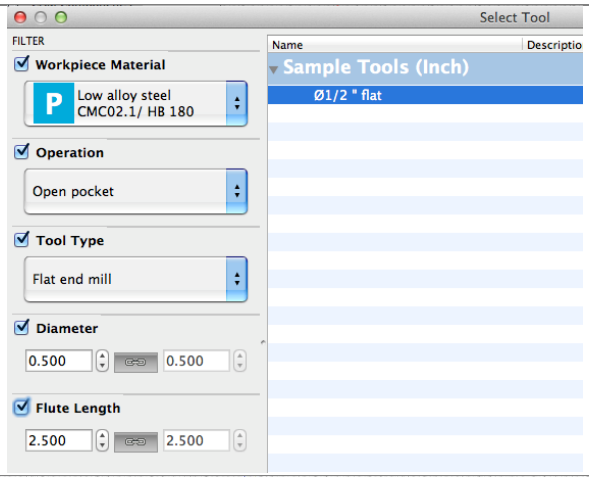
Step 2 – Windows library

1. Select a Flat End Mill with:
   Diameter equal to **0.5 in**
   Flute Length greater than **2 in**
2. Click **OK.**



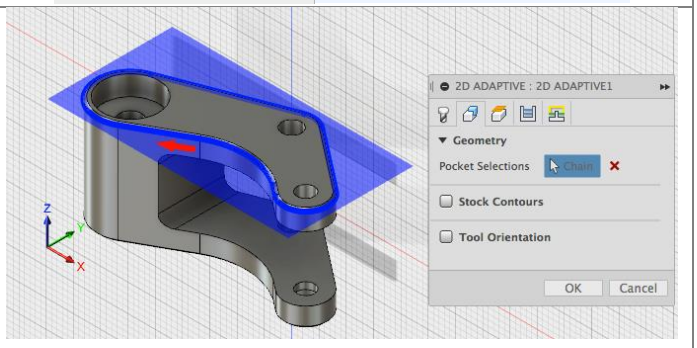

Step 2 – Mac library

1. Check the box for **Diameter**.
2. Enter **0.5** in the field and click the link button.
3. Check the box for **Flute Length**.
4. Enter **2.5** in the field and click the link box.
5. Select the **½" Flat** tool.
6. Click **OK**.
7. Click **OK** to accept the tool post processor information.




Step 3 – Select the geometry

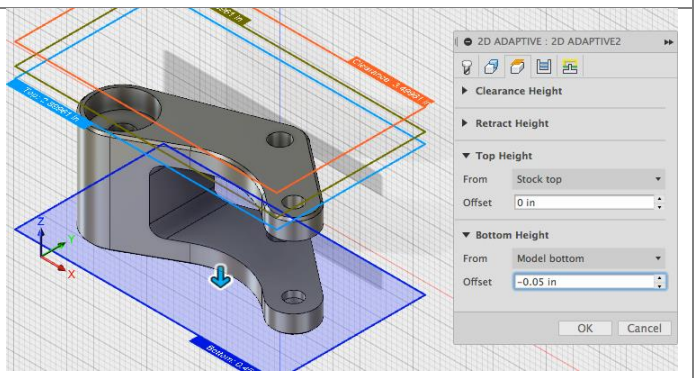
1. Select the **Geometry** tab.
2. Select the **top outside edge** of the part.




Step 4 – Set the heights

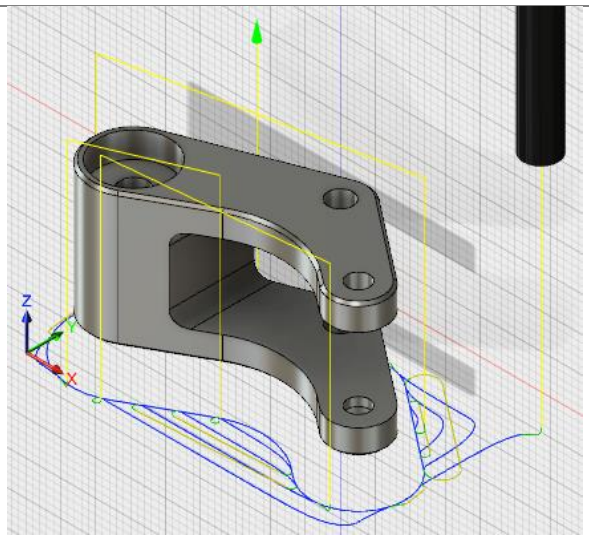
1. Select the **Heights** tab.
2. Set the **Bottom Height** to **Model bottom**.
3. Under Bottom Height, set the Offset to **– 0.05 in**.

Machining past the part during roughing gives you a nice finish/transition when you face the backside.



Step 5 – Finish the operation

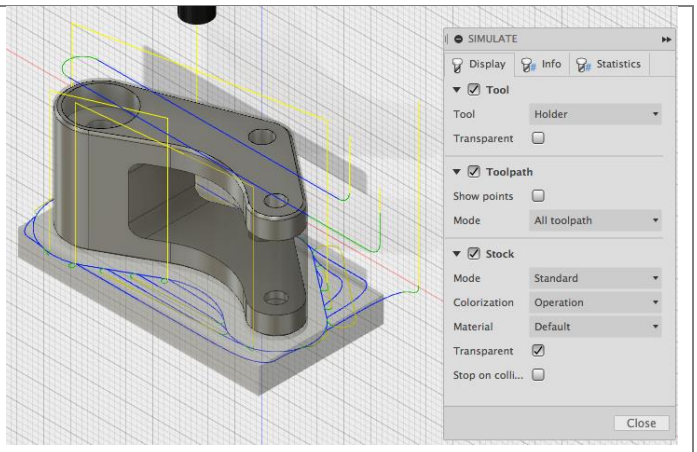
1. Click **OK** to create the 2D Adaptive Clearing operation.




**Simulate the Toolpaths:** Let's simulate the two toolpaths that we have created so far. This gives you an opportunity to verify that the operations are set up the way you expect.

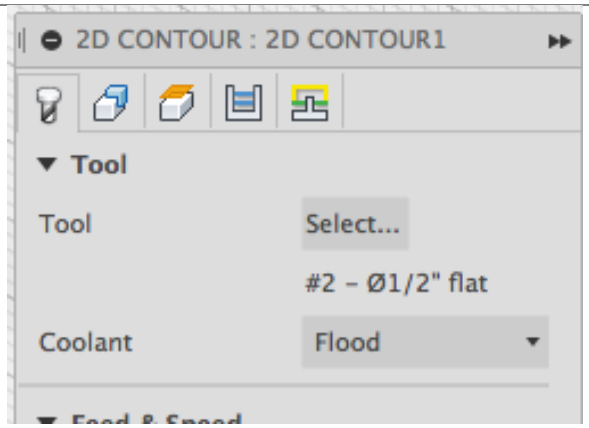
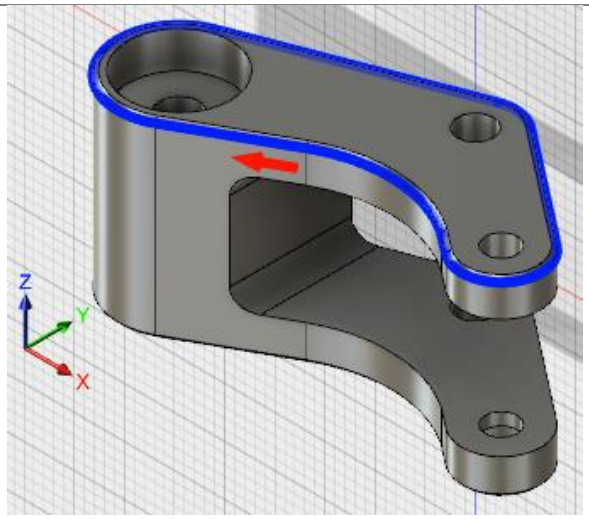
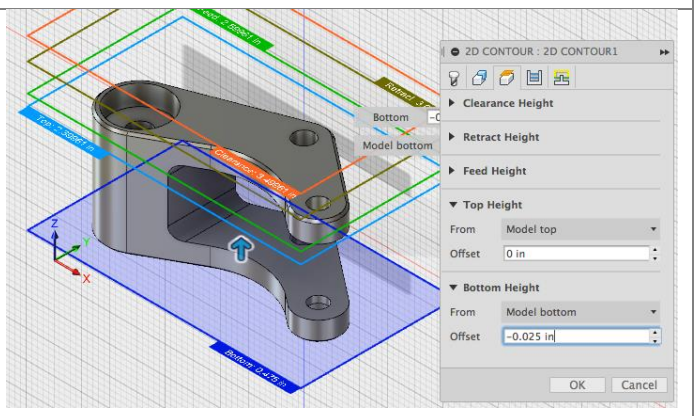
Step 1 – Simulate the toolpaths

1. In the browser, right-click on the Setup and select **Simulate**.
2. In the Simulate dialog box, click the boxes for **Toolpath**, **Stock**, and **Transparent**.
3. Click Play to see the simulation.
4. After verifying stock simulation, click **Close**.

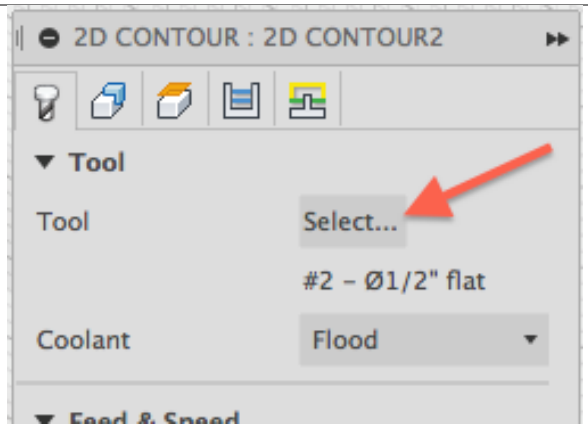
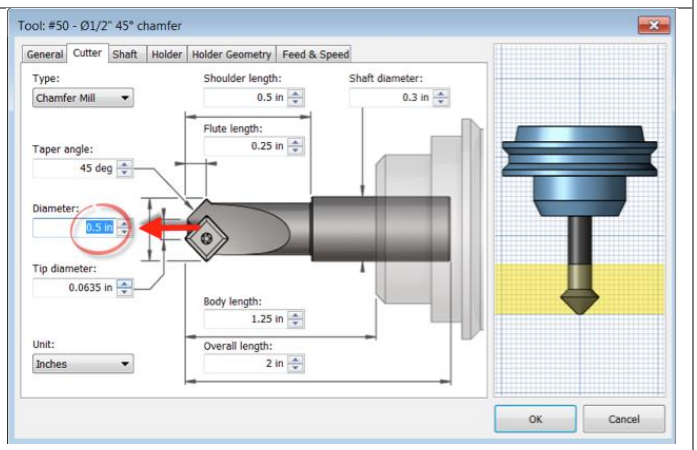
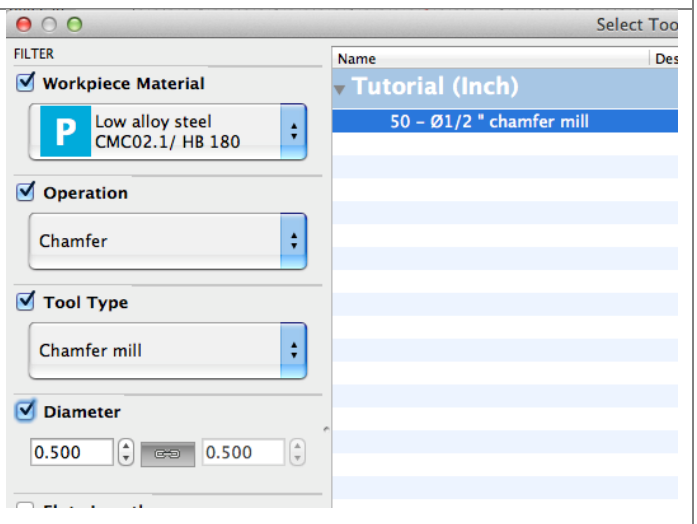



LAUNCH VIDEO

**2D Contour:** Create another 2D Contour. This is the finishing pass for this setup.

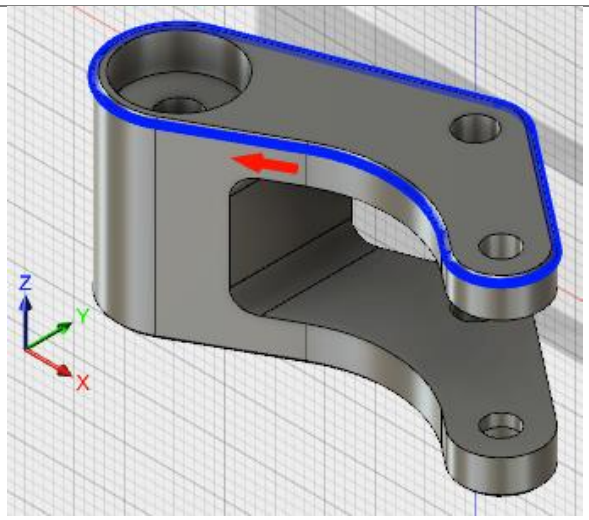
| | |
|---|---|
| Step 1 – Start the 2D Contour command<br><br>1. Click **2D > 2D Contour**.<br>2. CAM for Fusion 360 remembers the last tool you used, so we will use the **½" Flat End Mill** for this operation. |  |
| Step 2 – Select the geometry<br><br>1. Select the **Geometry** tab.<br>2. Select the same outside edge you selected for the 2D Adaptive Clearing operation.<br><br>The red arrow indicates the direction the tool will follow either outside or inside the profile. In this example, the tool will follow on the outside of the profile. Click the arrow to flip the direction. |  |
| Step 3 – Set the heights<br><br>1. Select the **Heights** tab.<br>2. Under Top Height, set **From** to **Model top**.<br>3. Under Bottom Height, set **From** to **Model bottom** and enter **– 0.025 in** for the Offset.<br>4. Click **OK**. |  |

**Chamfer Mill Operation:** Use a 2D Contour operation to chamfer the top edge of the part.

| | |
|---|---|
| Part 1 – Start the 2D Contour command<br><br>1. Click **2D > 2D Contour**.<br>2. In the 2D Contour dialog box, click **Select** next to **Tool**. |  <br>**LAUNCH VIDEO** |
| Step 2 – Windows library<br><br>1. Use a **45 degree Chamfer Mill** with a **0.5 in** Diameter.<br>2. Click **OK**. |  |
| Step 2 – Mac library<br><br>1. Set the Operation to **Chamfer**.<br>2. Check the Diameter box.<br>3. Enter **0.5** in the field and click the link button.<br>4. Select the **1/2 " chamfer mill**.<br>5. Click **OK**. |  |

Step 3 – Select the geometry

1. Select the **Geometry** tab.
2. Select the same edge you selected in the last two operations.

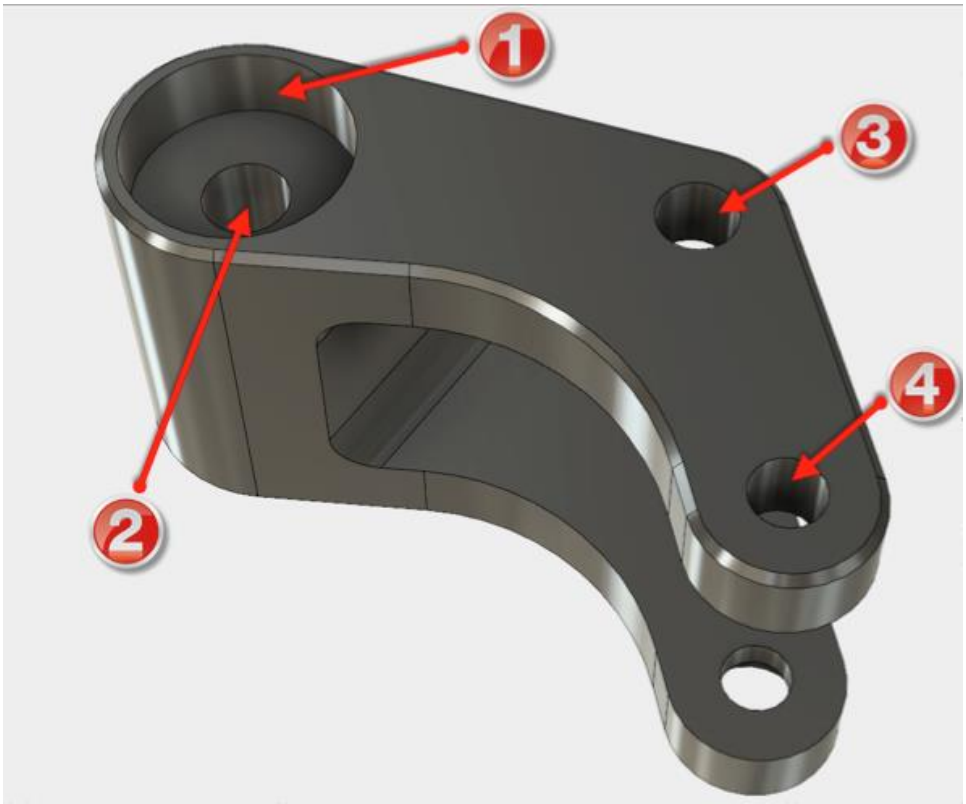


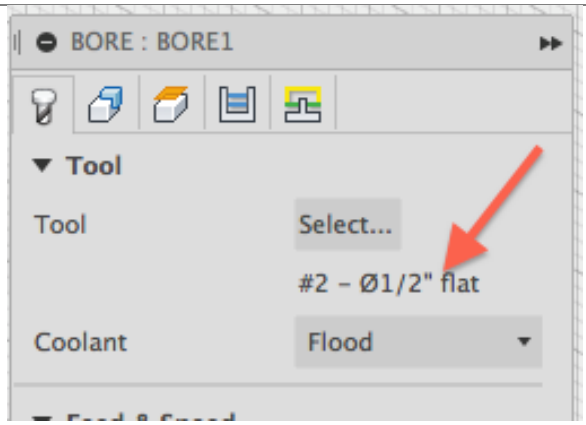

Step 4 – Set the chamfer tip offset

1. Select the **Passes** tab.
2. The Chamfer option is present because the tool is a chamfer mill.
3. Set the Chamfer Tip Offset to **0.05 in**.
4. Click **OK**.

The center of a spinning tool is not rotating as it is at the center of rotation. Instead, it is just pushing material. By using a Tip Offset the tool slides down the chamfer and is cutting instead of pushing material resulting in a far better surface finish and avoiding the possiblity of a step at the bottom of the chamfer.

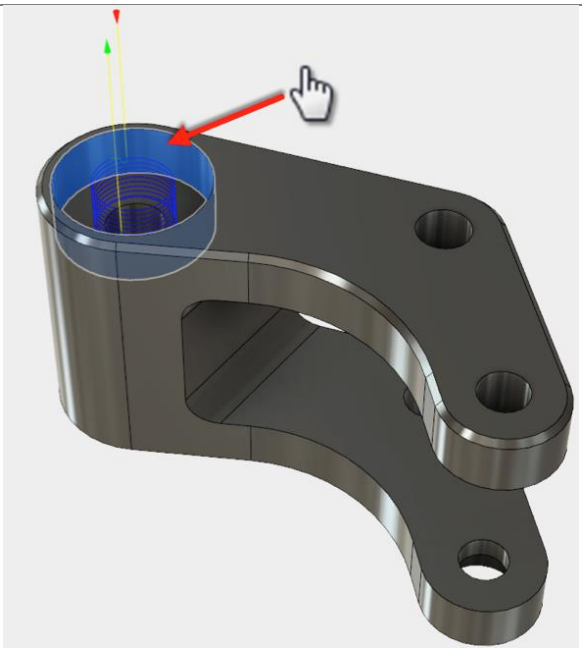
**Bore operation:** In this section, we will bore out four holes using two operations and two flat end mills with different diameters.




| | |
|---|---|
| Step 1 – Start the Bore command<br><br>1. Click **2D > Bore**.<br>2. In the Bore dialog, box click **Select** next to **Tool**.<br>3. Select the same **1/2" end mill** used in the 2D adaptive operation. |  <br><br>LAUNCH VIDEO |

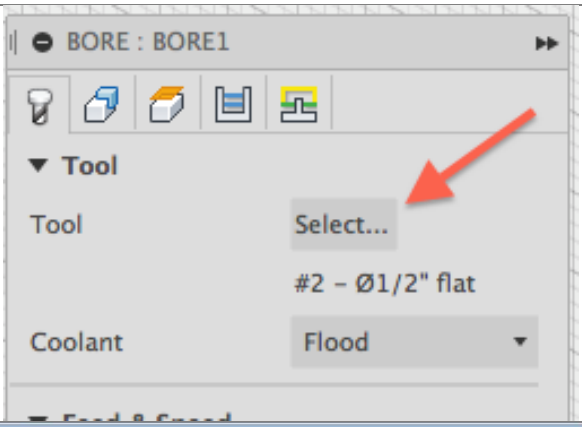
Step 2 – Select the geometry

1. Select the **Geometry** tab.
2. Select the inside circular face shown.
3. Click **OK**.



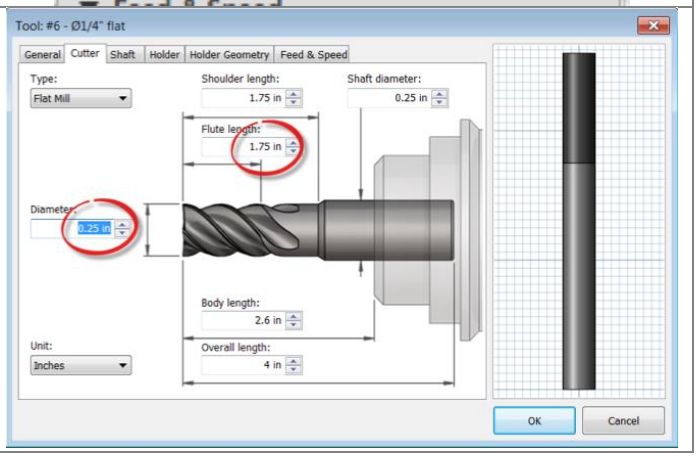
Step 3 – Bore the other holes

1. Right-click and select **Repeat Bore**.
2. In the Bore dialog box, click **Select** next to **Tool**.



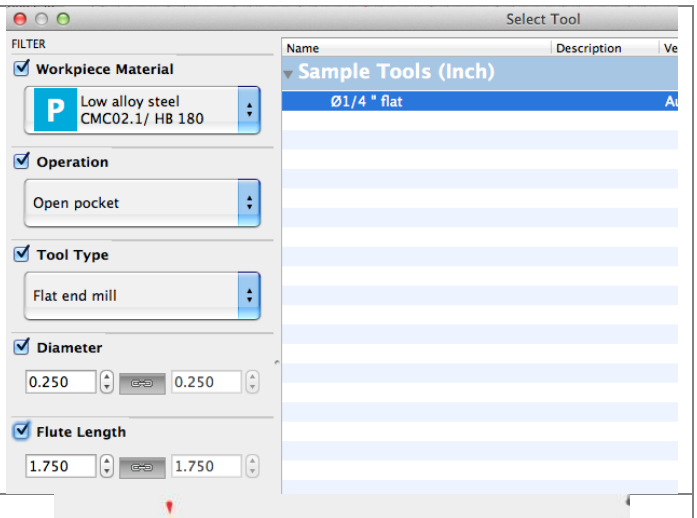

Step 4 – Windows library

1. Select a **flat end mill** with:
   Diameter equal to **0.25 in**.
   Flute Length greater than **1.25 in**.
2. Click **OK**.



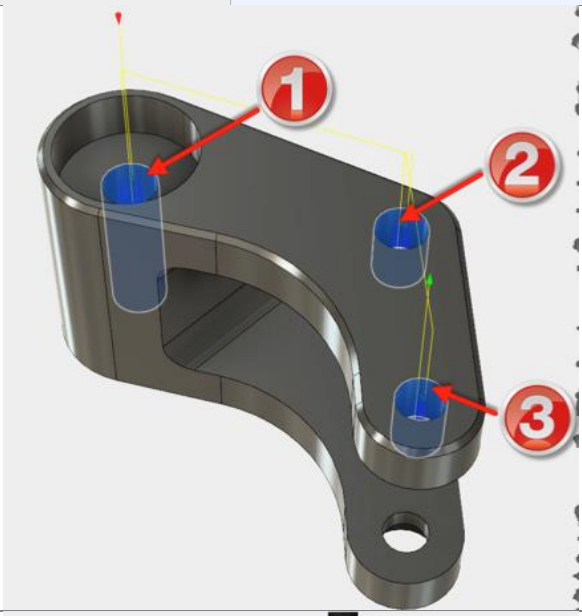
Step 4 – Mac library

1. Check the box for **Diameter**.
2. Enter **0.25** in the field and click the link button.
3. Check the box for **Flute Length**.
4. Enter **1.75** in the field and click the link button.
5. Select the **1/4" flat** tool.
6. Click **OK**.
7. Click **OK** to accept the tool post processor information.



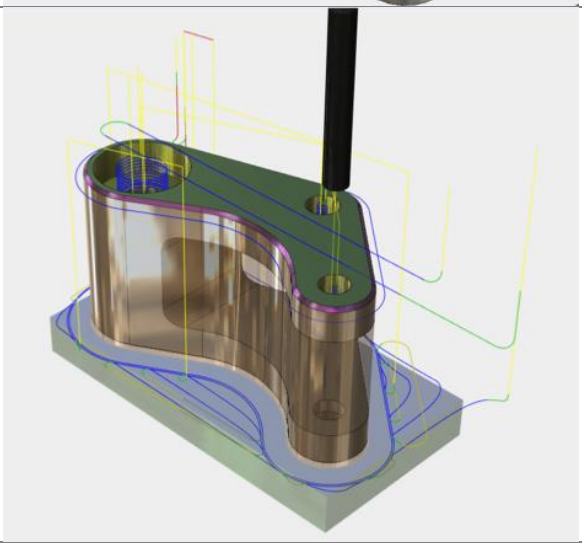

Step 5 – Select the holes

1. Select the **Geometry** tab.
2. Select the three holes
3. Click **OK**.



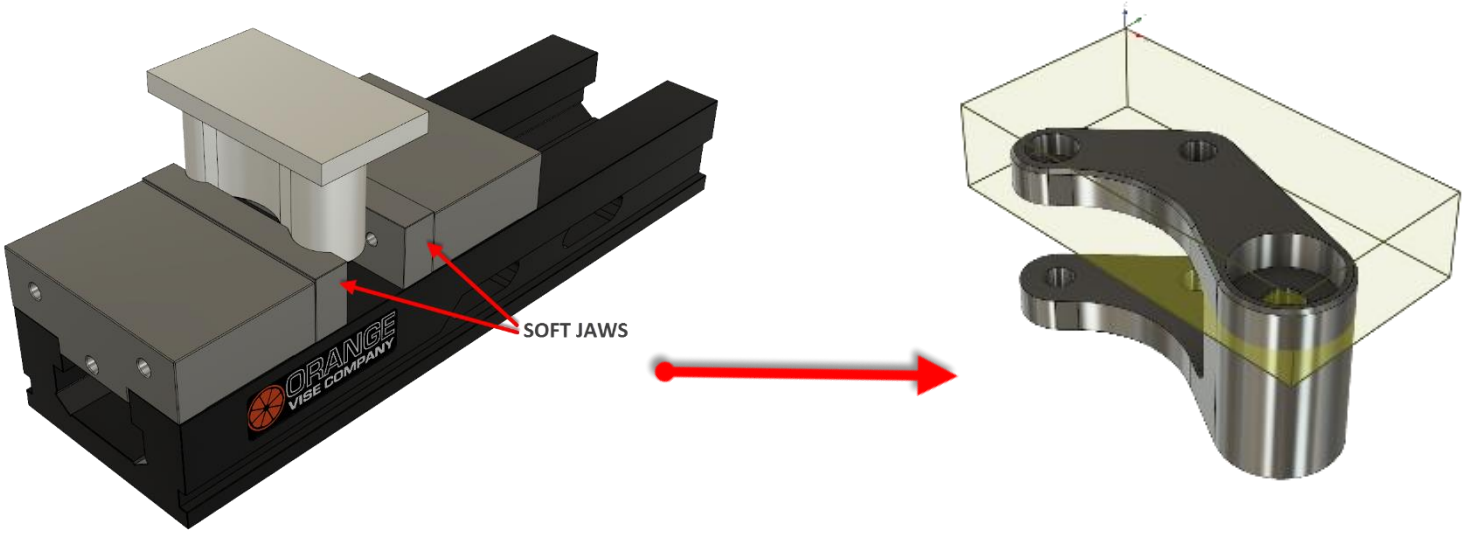

Step 6 – Simulate the setup

1. In the browser, right-click on the Setup and select **Simulate**.
2. In the Simulate dialog box, click the boxes for **Toolpath**, **Stock**, and **Transparent**.
3. Click Play to see the simulation.
4. After verifying stock simulation, click **Close**.

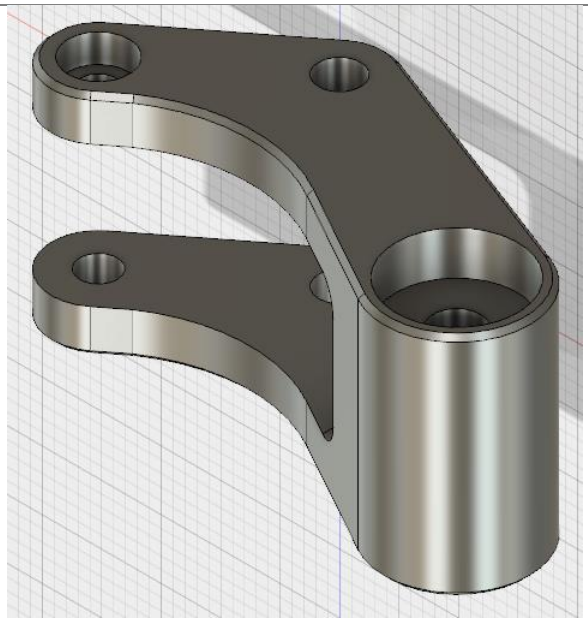


# SETUP 2

In this setup, we will machine the Rocker Arm based off the orientation/position seen in below, and use a HAAS Vertical Milling Center (VMC) Machine. The work coordinate system (WCS) for Setup 2 will be defined in upper right corner as well.



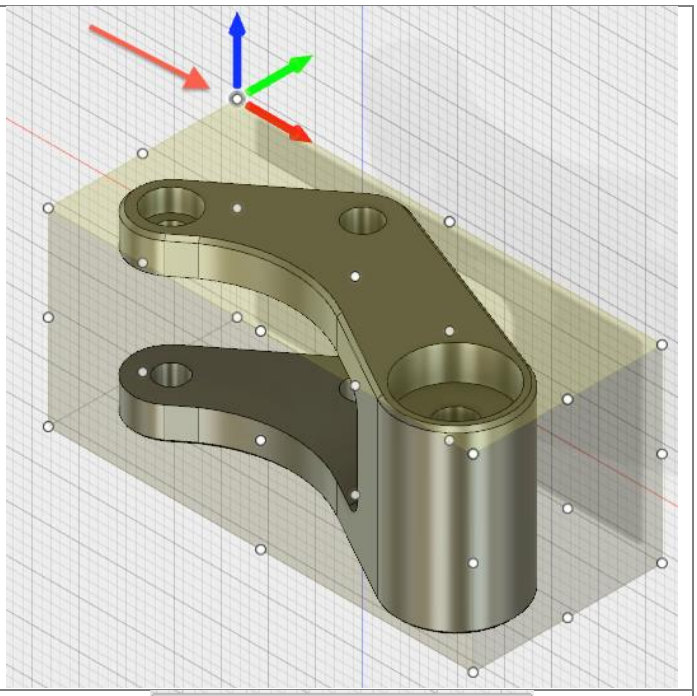

**Create Setup 2:** Create the second setup and stock to continue machining the part.

| | |
|---|---|
| Step 1 – Orient the model<br><br>1. Use the ViewCube or the navigation commands to position the model as shown. |  |

Step 2 – Create a setup and set the stock point

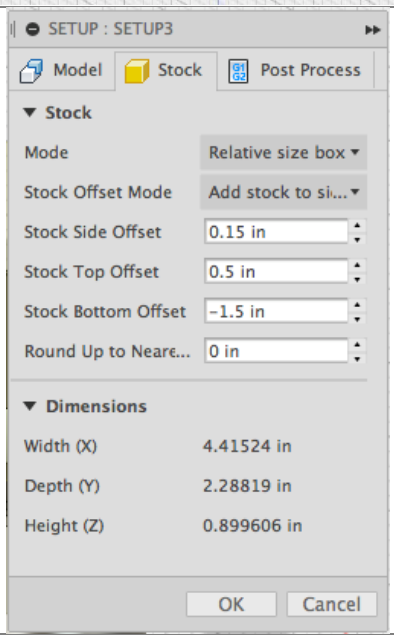
1. Click **Setup > Setup**.
2. Select **Stock Point**.
3. Click the **upper stock point** shown to move the triad.
4. Click the **head of red arrow** to flip the orientation. We want the positive X axis facing towards you.

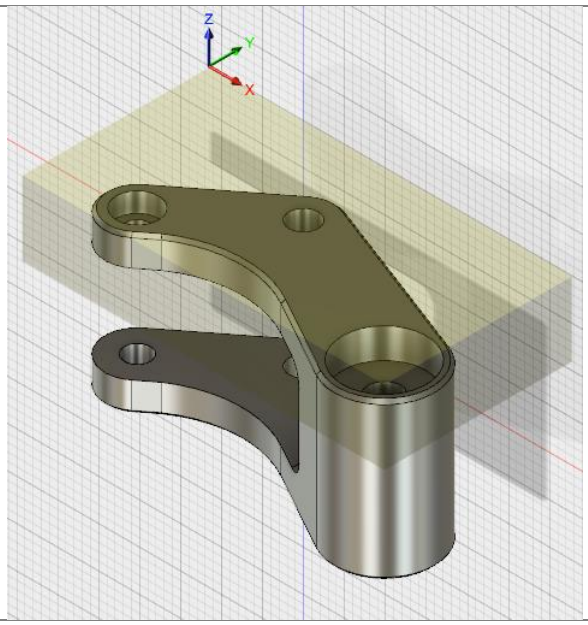
Verify that the blue arrow (Z axis) is facing up, the green arrow (Y axis) is facing right, and the red arrow (X axis) is facing towards you.



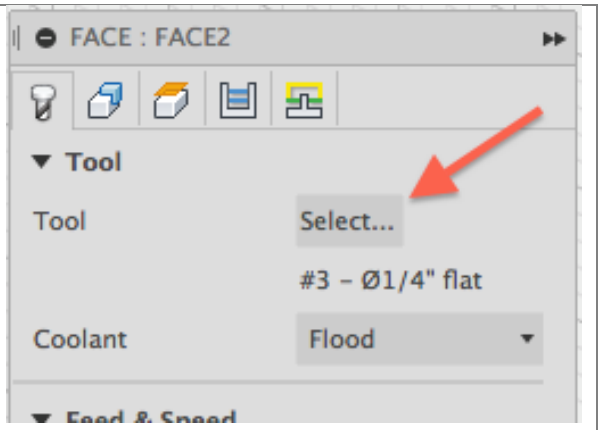
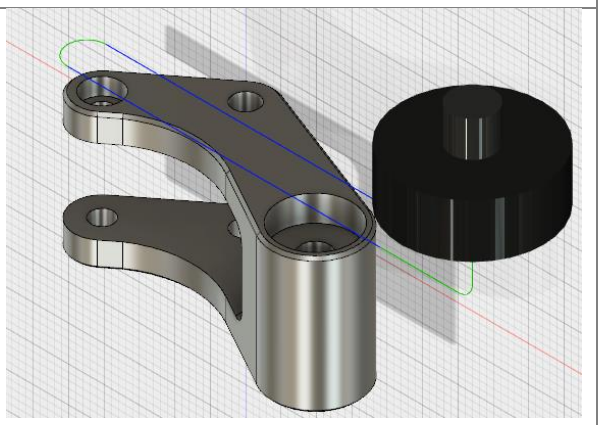
Step 3 – Set the stock size

1. Click the **Stock** tab.
2. Set the Mode to **Relative Size Box**.
3. Input these values:
   Stock Side Offset: **0.15 in**
   Stock Top Offset: **0.5 in**
   Stock Bottom Offset: **-1.5 in**
4. Click **OK.**

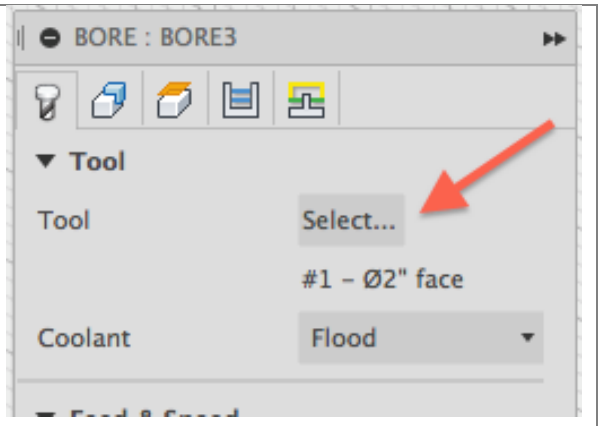
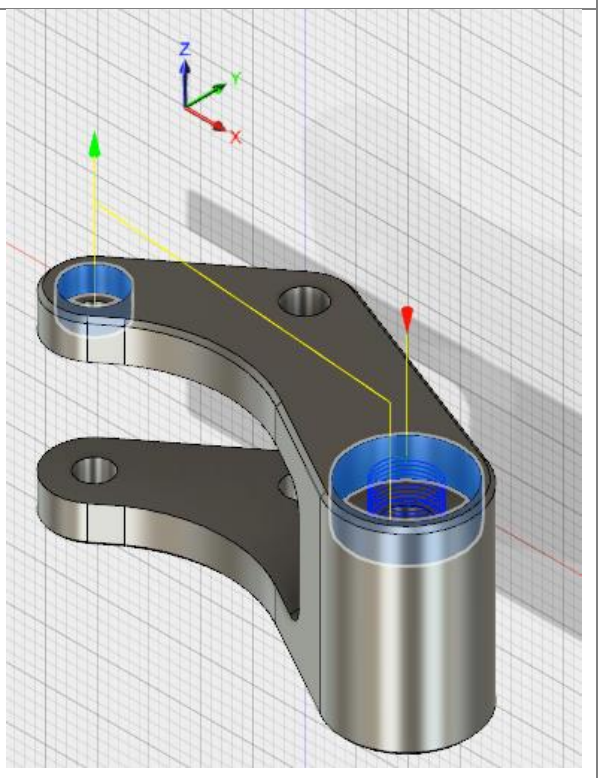


| | |
|---|---|
| Step 4 – Finished setup<br><br>You have defined the size of the stock we will be machining.<br><br>The stock for Setup2 is shorter than in Setup1 because what's left to machine is the remainder of the material at the 'bottom' of the part; so there is no need to fully model the stock. Also, we won't need to 2D Adaptive Clear the outside of the part, since we did that in SETUP 1. |  |

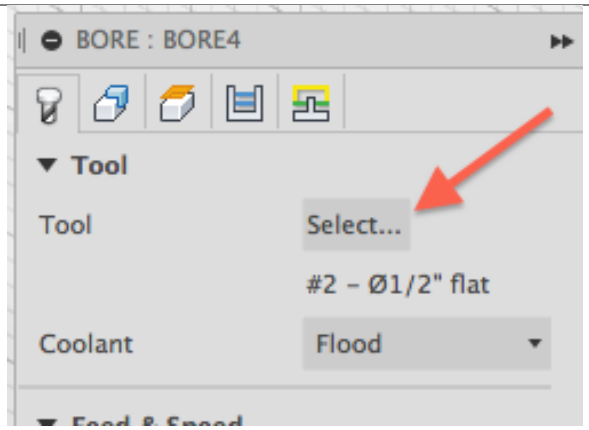
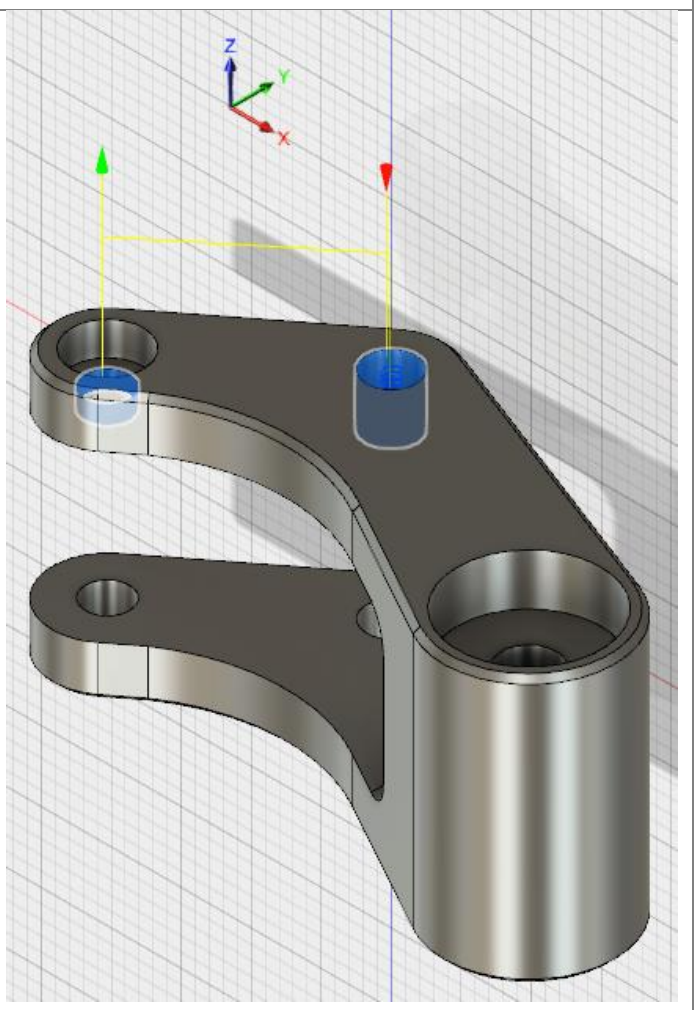
**Face operation:** Create a face operation and use the same face mill as in the previous face operation.

| | |
|---|---|
| Step 1 – Start the Face command<br><br>1. Click **2D > Face**.<br>2. In the Face dialog box, click **Select** next to **Tool**.<br>3. Select the same **2" Face Mill** that was used in the first face operation.<br>4. Click **OK** to accept the tool.<br>5. Click **OK** to create the operation. |  |
| Step 2 – Face operation created<br><br>The face operation is create for the second setup. |  |

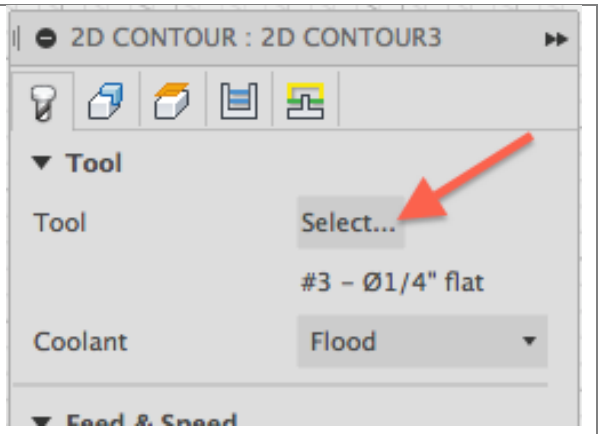
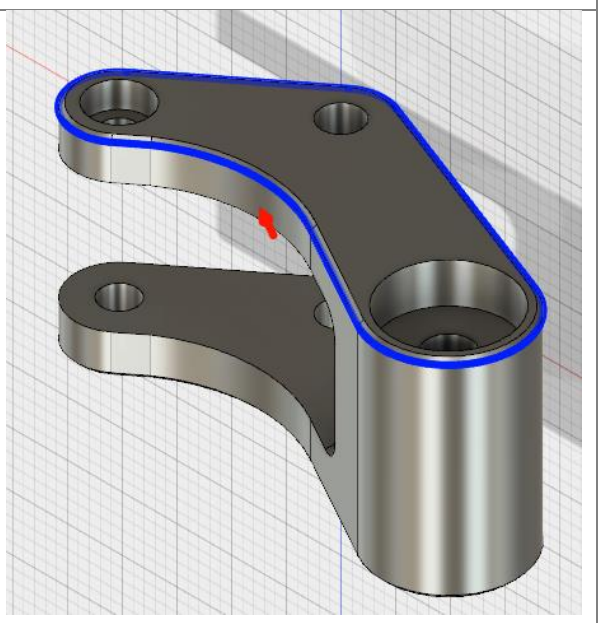
**Bore larger holes:** Create a bore operation to machine the counterbores on two holes.

| | |
|---|---|
| Step 1- Start the Bore command<br><br>1. Select **2D > Bore**.<br>2. In the Bore dialog box, click **Select** next to **Tool**.<br>3. Select the **1/2" Flat Mill** you used earlier.<br>4. Click **OK** to accept the tool. |  |
| Step 2 – Select the geometry<br><br>1. Select the **Geometry** tab.<br>2. Select the **two cylindrical faces** shown. These are the counterbores for the holes.<br>3. Click **OK** to create the bore operation. |  |

**Bore small holes:** Create a bore operation to machine the two small holes.

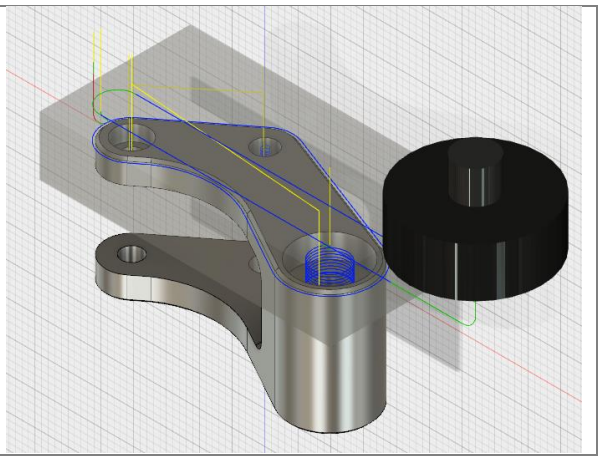
| | |
|---|---|
| Step 1 – Start the Bore command<br><br>1. Click **2D > Bore**.<br>2. In the Bore dialog box, click **Select** next to **Tool**.<br>3. Select the **1/4" Flat Mill** tool used previously.<br>4. Click **OK** to accept the tool |  |
| Step 2 – Select the geometry<br><br>1. Select the **Geometry** tab.<br>2. Select the **two holes** shown.<br>3. Click **OK** to create the operation. |  |

**Chamfer edge:** Finish this setup by chamfering the edge of the part.

| | |
|---|---|
| Step 1 – Start the 2D Contour command<br><br>1. Select **2D > 2D Contour**.<br>2. In the 2D Contour dialog box, click **Select** next to **Tool**.<br>3. Select the same **1/2" Chamfer Mill** used earlier.<br>4. Click **OK** to accept the tool. |  |
| Step 2 – Select the geometry<br><br>1. Select the **Geometry** tab.<br>2. Select the edge of the part. Be sure to select the lower outer edge of the chamfer feature.<br>3. |  |
| Step 3 – Set the chamfer tip offset<br><br>1. Select the **Passes** tab.<br>2. The Chamfer option is present because the tool is a chamfer mill.<br>3. Set the Chamfer Tip Offset to **0.05 in**.<br>4. Click **OK**. |  |

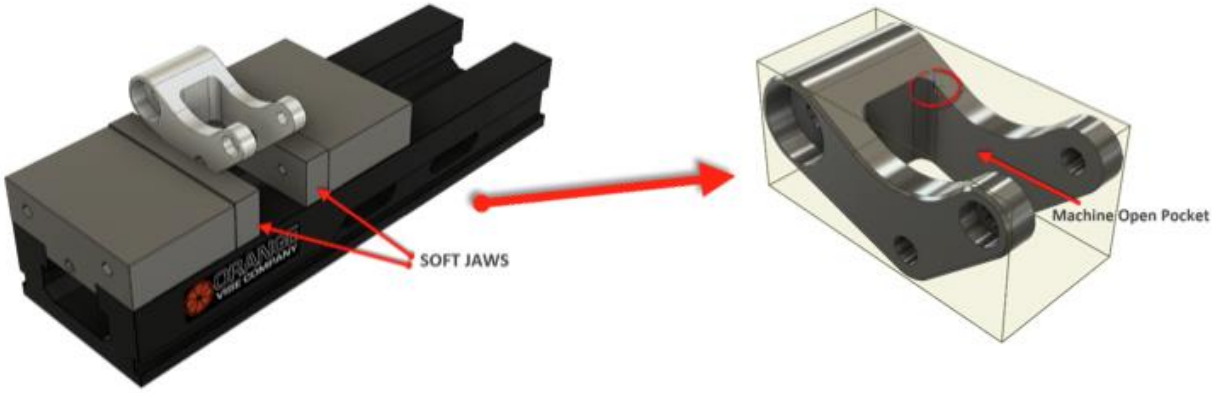
Step 4 – Simulate the setup.

1. In the browser, right-click on the Setup and select **Simulate**.
2. In the Simulate dialog box, click the boxes for **Toolpath**, **Stock**, and **Transparent**.
3. Click Play to see the simulation.
4. After verifying stock simulation, click **Close**.

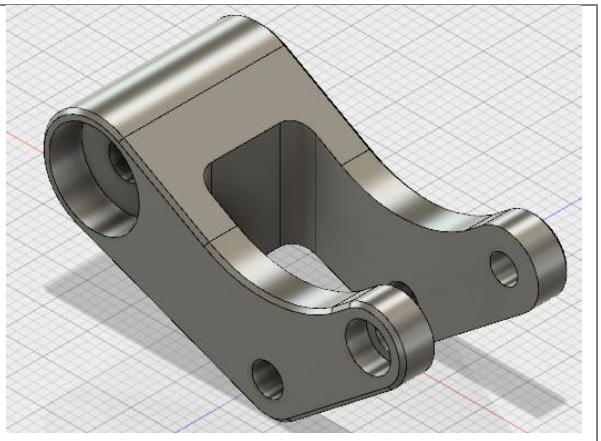
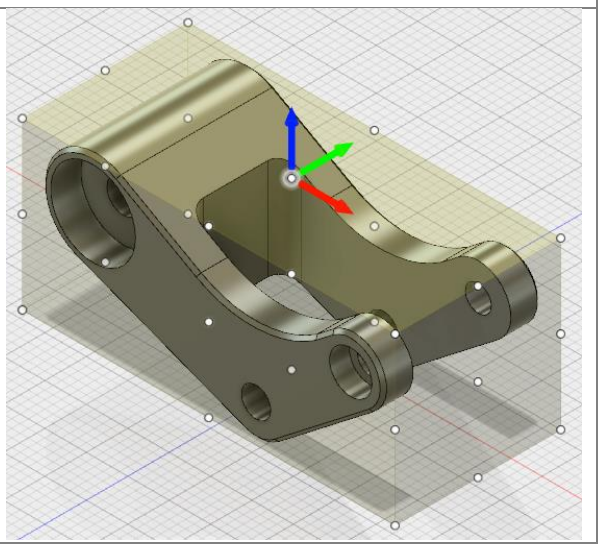


# SETUP 3

In this setup, we will machine the Rocker Arm based off the orientation/position seen in the image below, and use a HAAS Vertical Milling Center (VMC) Machine. The work coordinate system (WCS) for Setup 3 will be defined in the upper center.



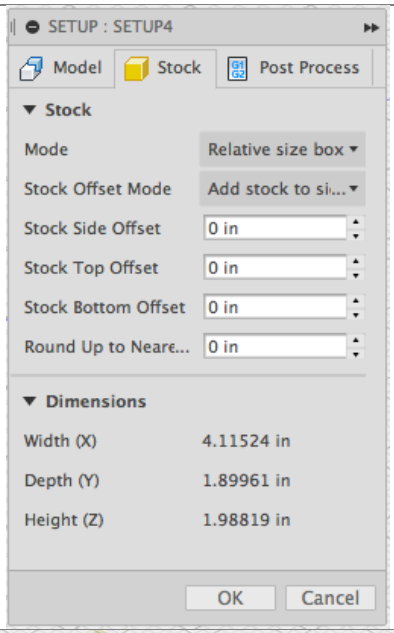
**Create Setup 3:** Create the second setup and stock to continue machining the part.

| | |
|---|---|
| Step 1 – Orient the model<br><br>1. Use the ViewCube or the navigation commands to position the model as shown. |  <br><br>**LAUNCH VIDEO** |
| Step 2 – Create a setup and set the stock point<br><br>5. Click **Setup > Setup**.<br>1. Select **Stock Point**.<br>2. Click the **center stock point** on the top face as shown to move the triad.<br>3. Click the **body of blue arrow** then select a vertical edge to set the orientation. We want the positive Z axis facing up.<br><br>Verify that the blue arrow (Z axis) is facing up, the green arrow (Y axis) is facing right, and the red arrow (X axis) is facing towards you. |  |

Step 3 – Set the stock size

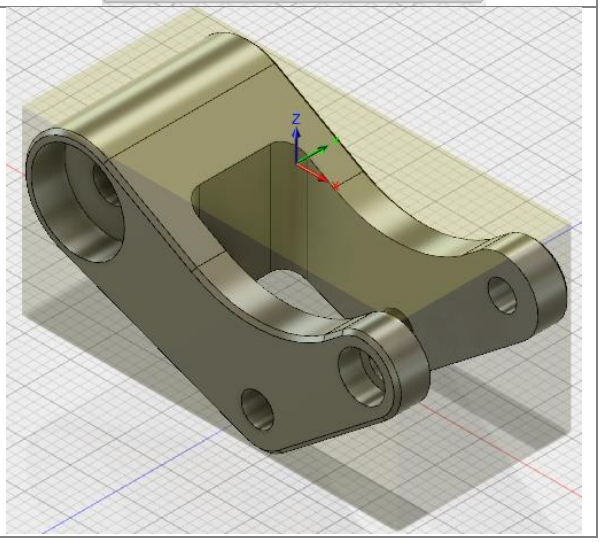
1. Click the **Stock** tab.
2. Set the Mode to **Relative Size Box**.
3. Input these values:
   Stock Side Offset: **0 in**
   Stock Top Offset: **0 in**
   Stock Bottom Offset: **0 in**
4. Click **OK.**

The stock offset values are set to zero. Since we only have the open pocket left to machine, there is no need to add material to the outside of the part.




Step 4 – Finished setup

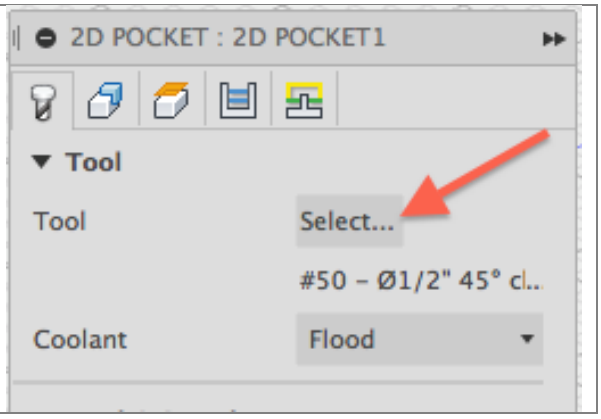
You have defined the size of the stock we will be machining.



**Create Setup 3:** Create the second setup and stock to continue machining the part.

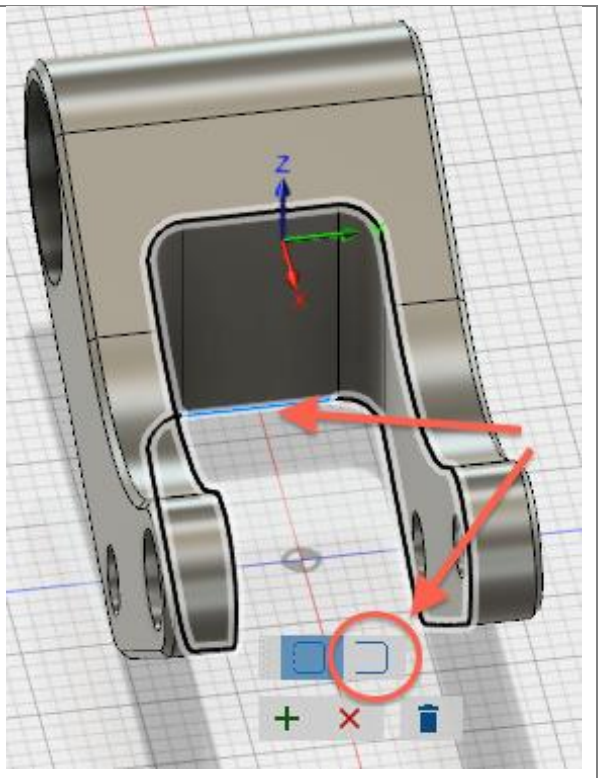
Step 1 – Start the Bore command

1. Click **2D > 2D Pocket**.
2. In the 2D Pocket dialog box, click **Select** next to **Tool**.
3. Select the **1/4" Flat Mill** tool used previously.
4. Click **OK** to accept the tool.



Step 2 – Select the geometry

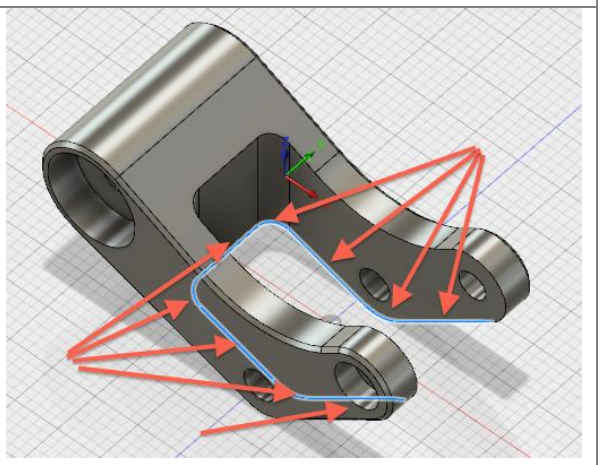
1. Select the **Geometry** tab.
2. Select the **edge** shown then click the **blue profile** preview that is displayed. This displays the mini toolbar.
3. Click the **Open contour** button in the mini toolbar.




**LAUNCH VIDEO**

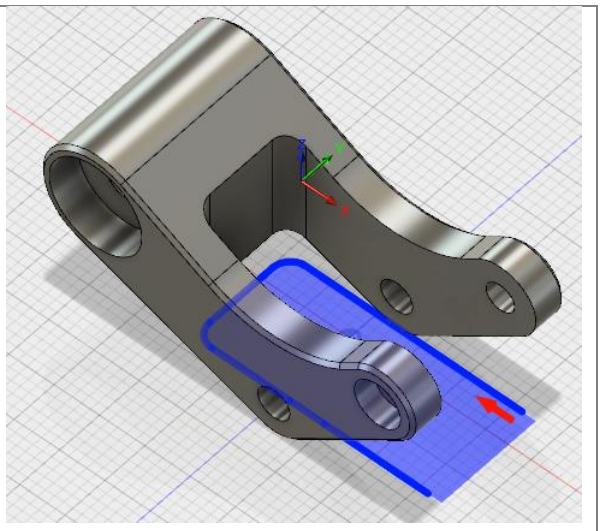
Step 3 – Select the profile.

1. Select the edges shown. There are nine total edges to select.
2. Click the **+ Accept current contour** button in the mini toolbar.



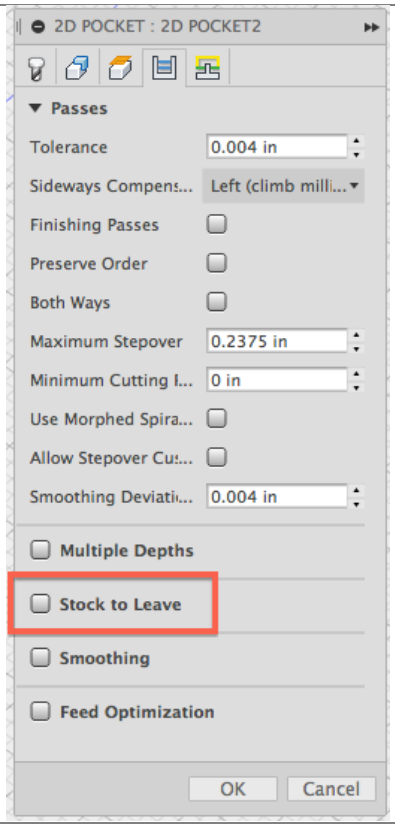
Step 4 – Verify the path

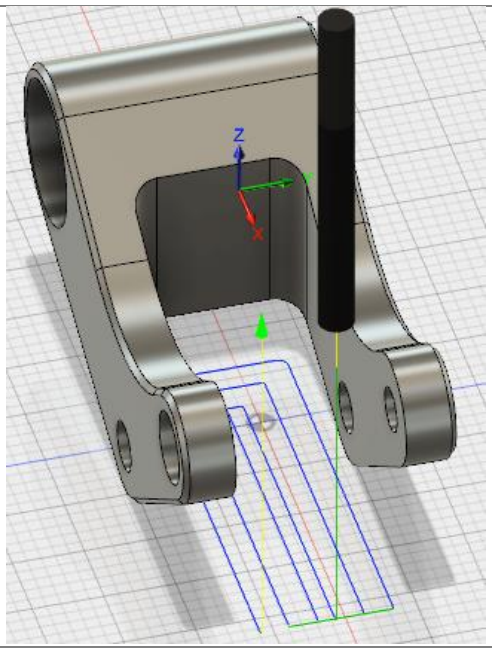
1. The toolpath should look like the image shown. If needed, click on the **red arrow** to flip the direction of the toolpath to machine inside the pocket.



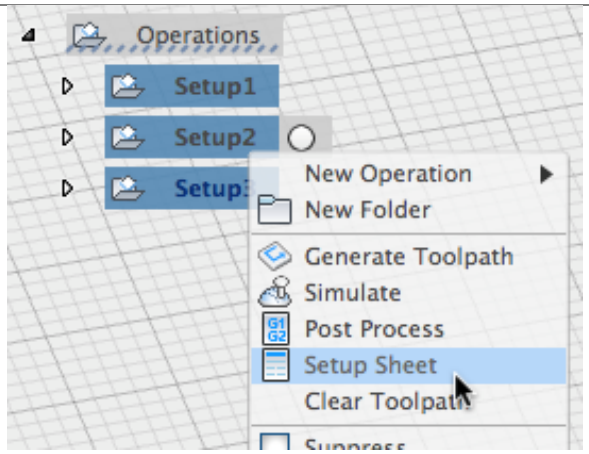
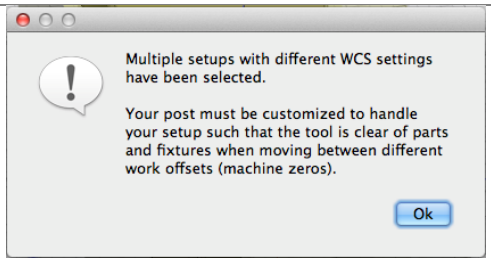
Step 5 – Set the passes

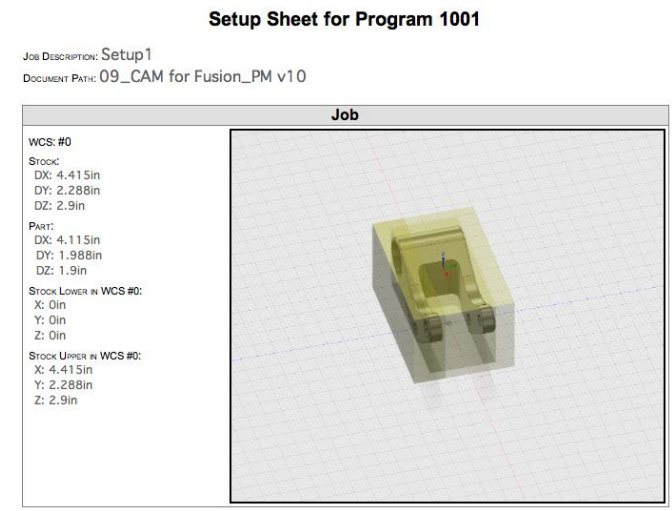
1. Select the **Passes** tab.
2. Unselect the **Stock to Leave** option.
3. Click **OK** to create the operation.



| | |
|---|---|
| The setup is complete. |  |

**Setup Sheet:** The Setup Sheet command generates an overview of the NC program for the CNC operator. It provides tool data, stock and work piece positioning; as well as machining statistics.

| | |
|---|---|
| Step 1 – Start the Setup Sheet command<br><br>1. Select **Setup1**, hold **Shift** then select **Setup3**. This selects all three setups.<br>2. Right-click then select **Setup Sheet**. |  |
| Step 2 – Specify the location<br><br>1. Navigate to a location to save the file then click **Save**.<br>2. Click **OK** in the warning dialog box. |  |

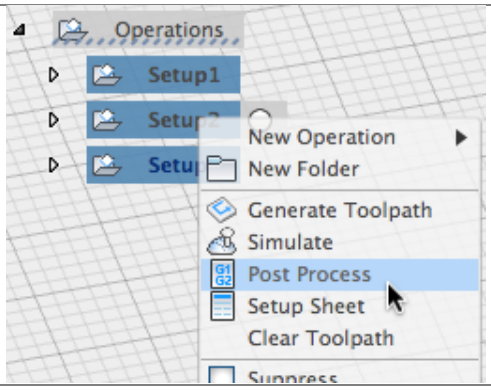
| | |
|---|---|
| Step 3 – Setup sheet<br><br>1. The Setup Sheet is displayed in your default internet browser.<br>2. Review the content then close your browser when done. |  |

# 5.0 Post Processor

A post processor is essentially a printer driver for CNC machines; a unique configuration file that allows our Post Processor System to turn your programmed toolpaths into CNC programs (G-Code) that your machine control executes to cut parts.

Fusion 360 comes with a standard library of "Posts". These library posts are included because they have been proven to make good parts using standard machine defaults. As the complexity of your setups increases, and you learn more about your CNC, you will probably want modifications made to one of these library posts that produce code in a particular way or with particular options enabled. This requires a post edit. Autodesk has a dedicated Post Development Team that while not working with machine tool vendors to produce more standard library posts helps our Autodesk CAM Resellers and end-users with post requests.

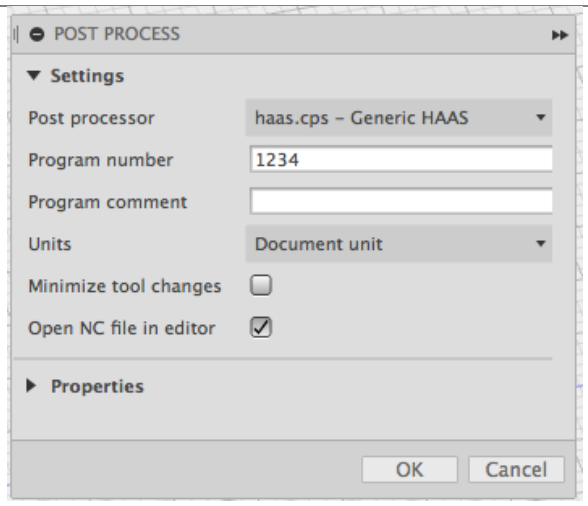
**Post Processor:** In this section, you post the CNC code from the three setups you created.

| | |
|---|---|
| Step 1 – Start the Post Process command<br><br>1. Select the three setups in the browser.<br>2. Right-click then select **Post Process**. |  |

Step 2 – Configure settings

1. Click **OK** in the warning dialog box about multiple WCS.
2. Select **haas.cps – Generic HAAS** as the Post Processor.
3. Enter **1234** in the Program Number field.
4. Click **OK**.
5. Enter a name and click **Save**.

Many machines, like the Haas, require programs to be a 4 digit number. So, the post forces the users to use a program name that the control will accept. If your control will accept another naming convention, like full alpha-numeric program names, the post can be easily modified.




Step 3 – Review the code

1. The G-Code is displayed in the editor.
2. Close the editor and return to Fusion.

```
%
O01234
(T1 D=2. CR=0. - ZMIN=-0.5 - face mill)
(T2 D=0.5 CR=0. - ZMIN=-0.8642 - flat end mill)
(T3 D=0.25 CR=0. - ZMIN=-1.9882 - flat end mill)
(T50 D=0.5 CR=0. TAPER=45deg - ZMIN=-0.5394 - chamfer mill)
N10 G90 G94 G17
N15 G20
N20 G53 G0 Z0.

(Face1)
N30 T1 M6
N35 S94 M3
N40 G54
N45 M8
```