Intro to 2D Strategies | CAM for Fusion 360

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. For more information, please refer to the Autodesk CNC Handbook.

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.

The term, Prismatic, is a term commonly used in engineering to describe 2-1/2D parts. There are, however, prismatic parts that require 4th 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).

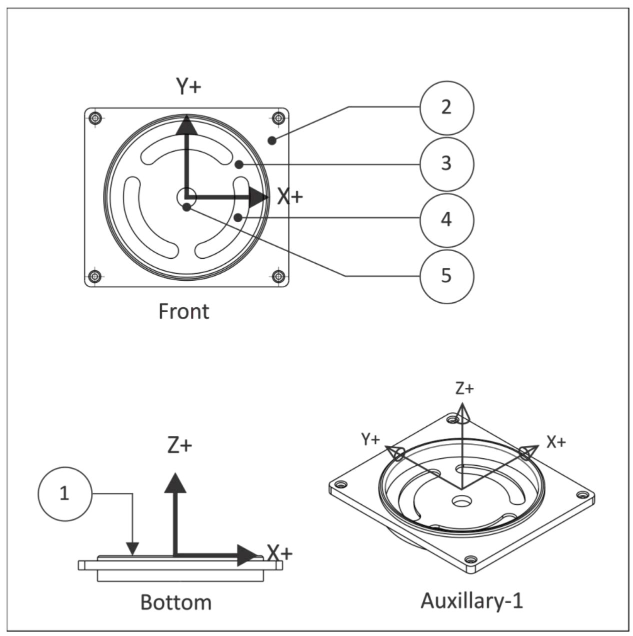


Figure 1: Prismatic Part (Orientation in CAD)

Figure 1 shows a prismatic part. All machined features lie parallel to the XY plane. Each Z-level can be 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).

Learning Objectives

Upon successful completion of this lesson, you will be able to:

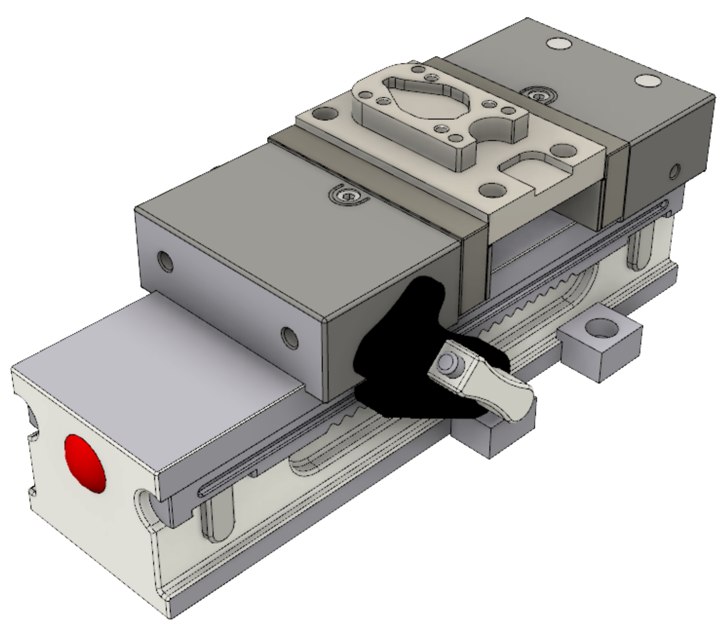
- Explain the difference between 2-1/2D and 3D machined parts.
- Explain the difference between common CAD and CAM graphics views
- Identify 2D machining features based on part geometry and your knowledge of tools and 2D toolpaths.
- Identify commonly used machining parameters for 2D tool path operations.
- Apply a Job Setup to a 2D Milled Part
- Apply a multitude of 2D Operations to a Milled Part
    - Facing Toolpaths
    - 2D Adaptive Toolpaths
    - 2D Contour Toolpaths
    - Chamfer Milling Toolpaths
    - Bore Toolpaths
- Produce Setup Sheets
- Simulate Toolpaths and Stock Material Removal
- Produce NC Code via Post Processing

Datasets Required

In Samples section of your Data Panel, browse to:

Fusion 101 Training > 09 – CAM > **09_2D_Strategies**

Open the design and follow the step-by-step guide below to get started with the lesson.



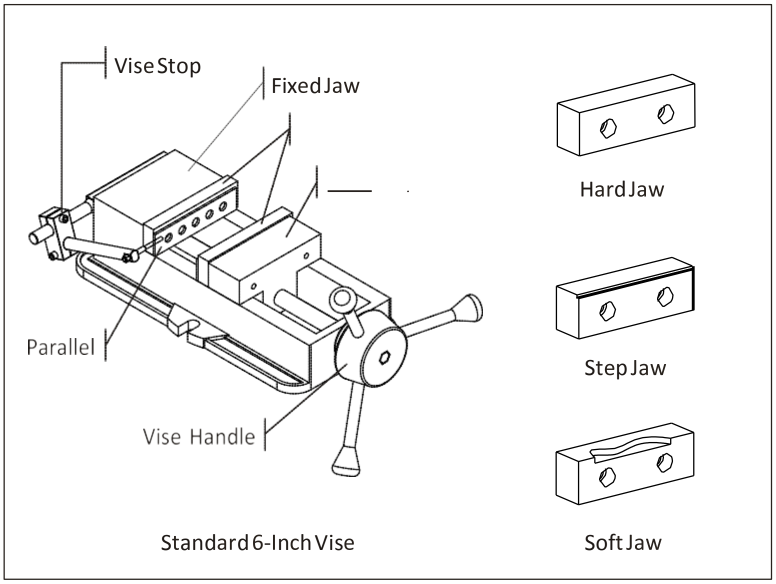
Lesson 1: Workholding & Job Setup

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.

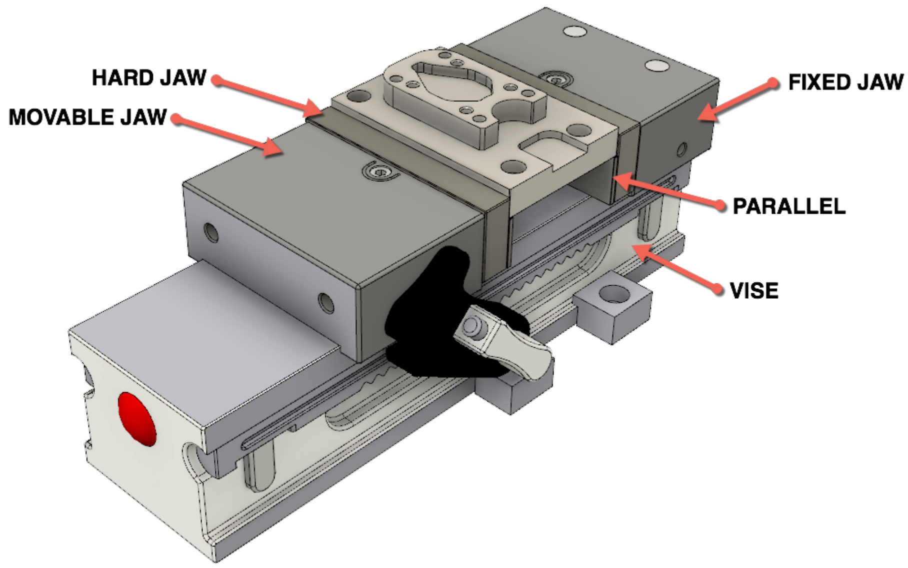


The **Fixed Jaw** remains stationary. The **Moving Jaw** opens when the **Vise Handle** is turned. It is a good practice to remove the vise handle after the jaws are closed and before running the program. This is done by simply sliding the handle off.

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.

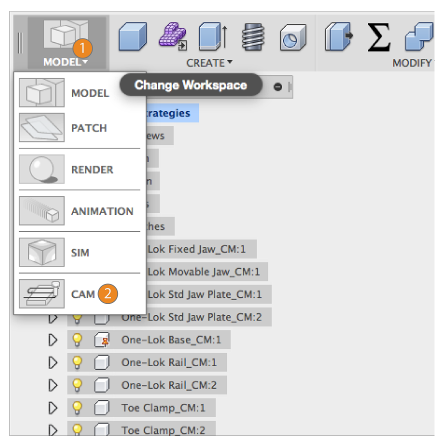
Hard Jaws are made of hardened steel and precision ground on all sides. They are usually used along with parallels.

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



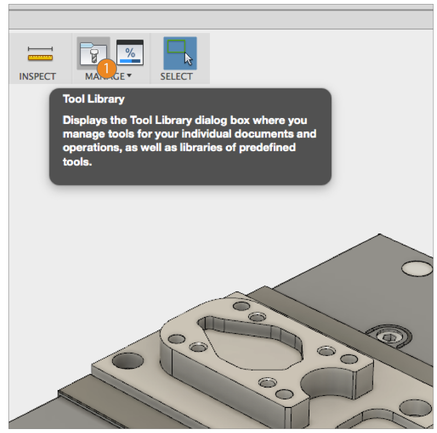
Step-by-step Guides:

Step 1: Activate the CAM Workspace.



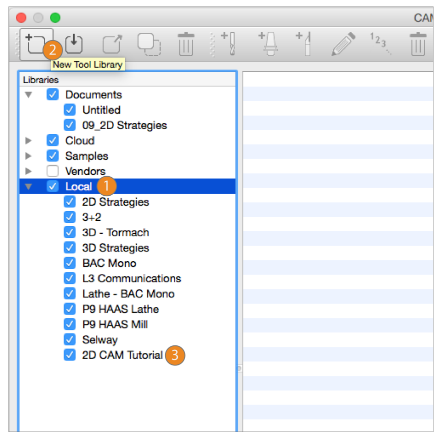
Step 2: – Start the **TOOL LIBRARY** command

1. Click **TOOL LIBRARY**



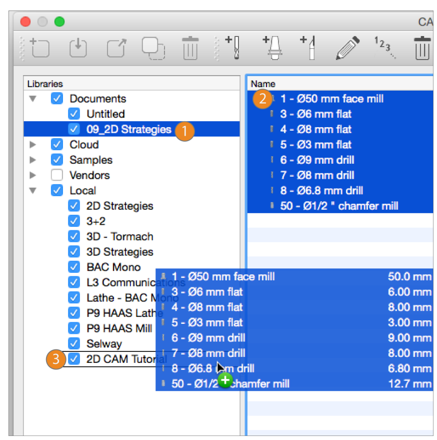
Step 3: – Create a **NEW TOOL LIBRARY**

1. Click on **Local**
2. Select the **NEW TOOL LIBRARY** Icon
3. Double Click on the **NEW TOOL LIBRARY** name and rename to **2D CAM Tutorial**.



Step 4: – Copy and Paste **TOOLS** into **NEW LIBRARY**

1. Click on **Documents** and select the Library **09_2D_Strategies**
2. Select **ALL** tools in the library, and drag and drop into your new library created.

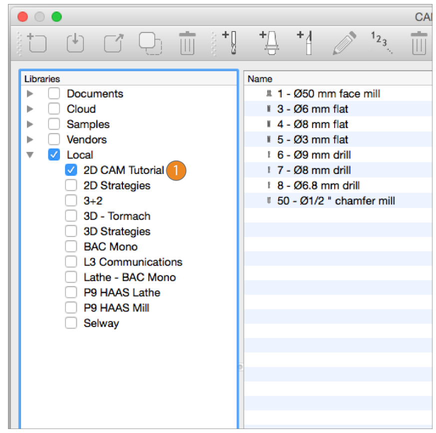


Step 5: – Turn off all other Libraries

1. Click off all other libraries and only show **2D CAM Tutorial.**

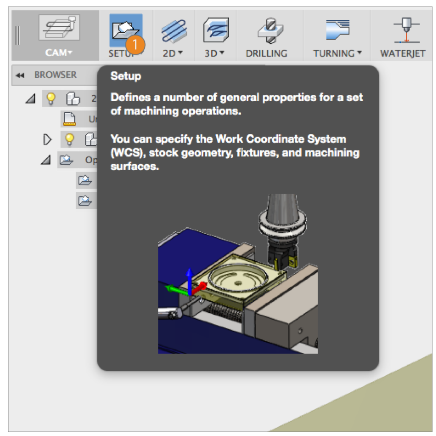
Then…

EXIT OUT OF TOOL LIBRARY



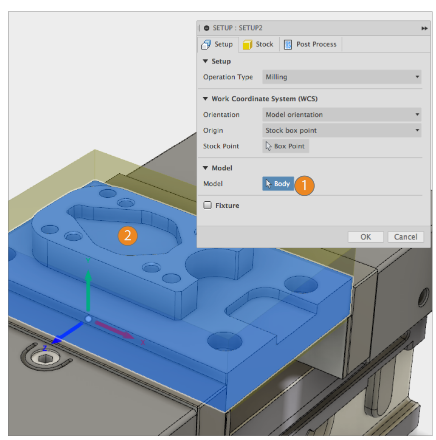
Step 6: – JOB SETUP

1. Click **SETUP**



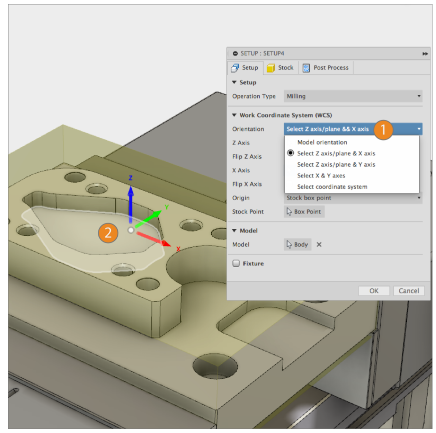
Step 7: – Select the **Part** you want to Machine

1. Under **MODEL**, active the **NOTHING** Icon
2. Select the **2D Strategies Part** in the Screen.



Step 8: – **Orientate** and **Locate** the Work Coordinate System (WCS) correctly.

1. Under Work Coordinate System (WCS), pick the Orientation drop down and select '**SELECT Z axis/plan & X axis**.'
2. Pick the **Highlighted top face**, and the WCS will orientate in the top/center of the part with **'Z' facing north**.

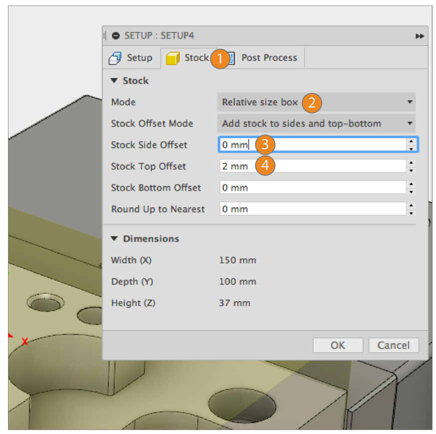


Step 9: – Change **STOCK** Options

1. Click on the **'STOCK'** Tab
2. Under **MODE**, select **Relative Size Box**
3. Under **STOCK SIDE OFFSET**, change to 0 mm
4. Under **TOP SIDE OFFSET**, change to 2 mm

THEN…

CLICK **OK** TO ACCEPT

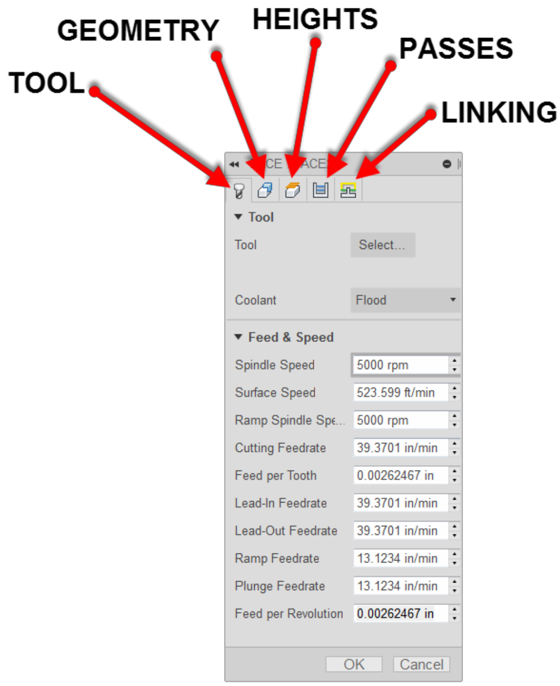


Lesson 2: Toolpath Operations

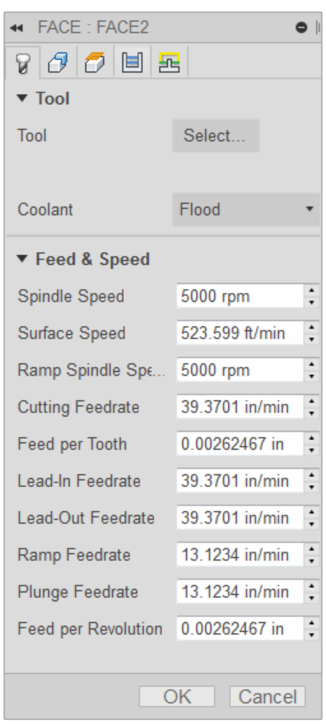
Understanding Toolpaths by Type and Use

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.

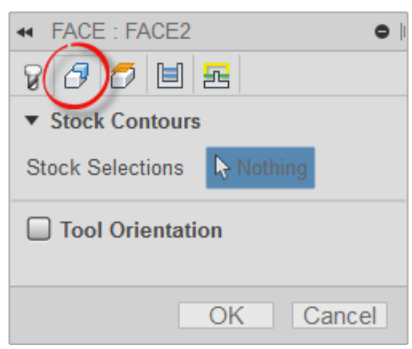
Type	Toolpath	Common Uses
Face	Face	• Finish face of part.
	Island Facing	• Finish face with open sides and bosses.
2D Contour	Contour	• Loops. • Partial loops. • Single edges. • Stick (single point) fonts. • Create dovetail, keyset, or saw cut.
	Chamfer	• Create chamfer using tapered mill or center drill. • De-burring.
	Fillet	• Creating fillet using Corner Round tool.
Pocket	Pocket	• Remove excess material. • Machining TrueType (outlined) fonts and logos.
	Slot Mill	• Straight slot. • 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. • Create milled OD threads of any size.



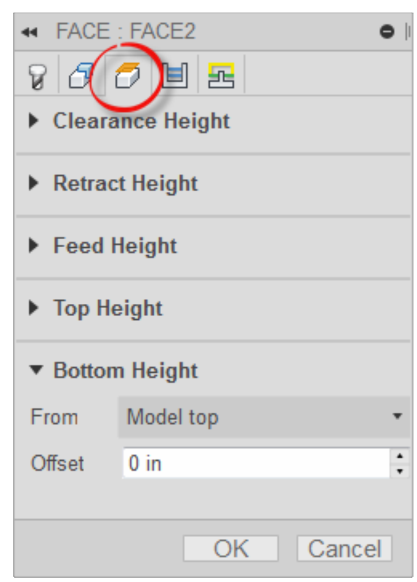
TOOL TAB– Defines the tool being used; as well as the feeds and speeds



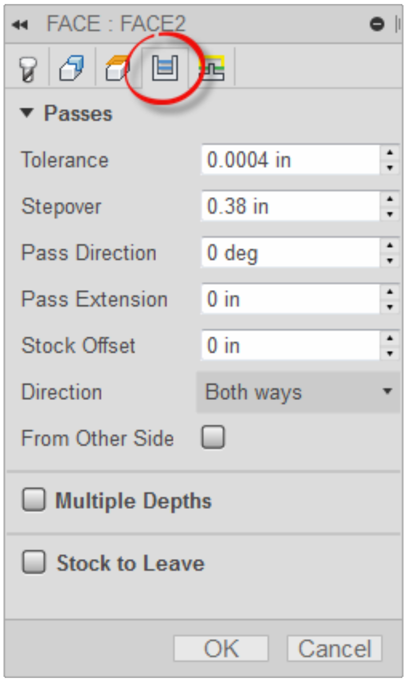
GEOMETRY TAB– Defines geometry being machined.



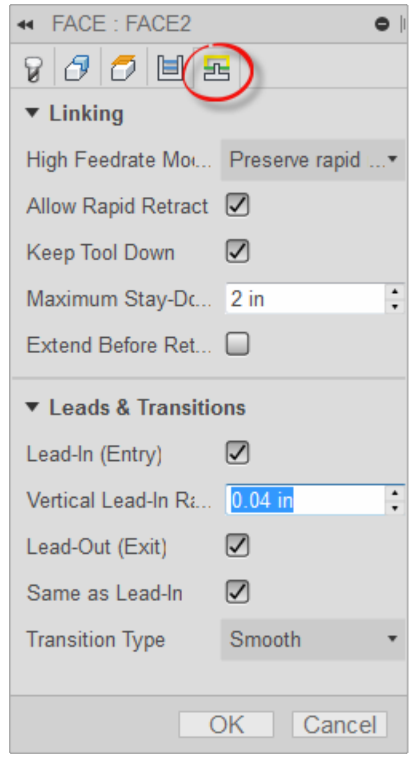
HEIGHTS TAB– Controls heights the toolpath goes to such as cut depth and retract heights



PASSES TAB– Controls how the tool will go about removing material.

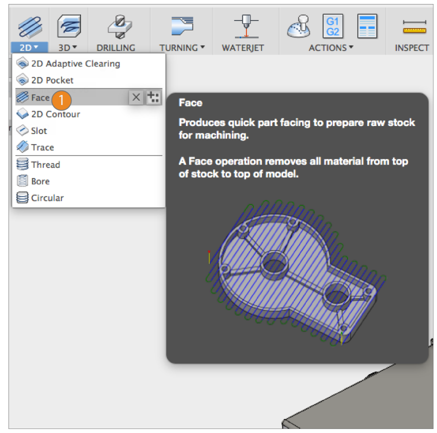


LINKING TAB– Controls how the tool enters/exits and transitions between cutting movements



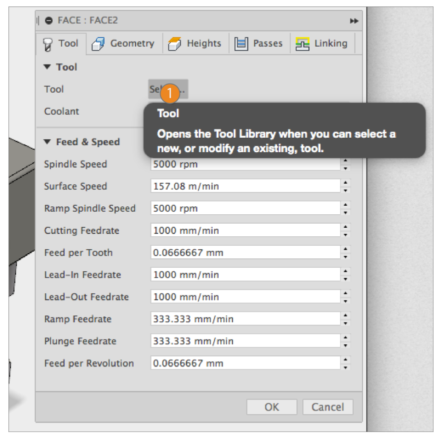
Step 10: – The **FACE** Operation

1. Under **2D Operation**, click on the **FACE Operation**



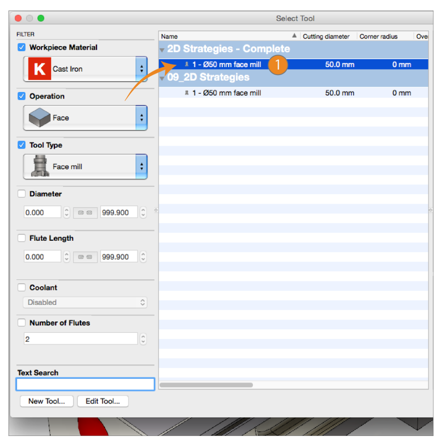
Step 11: – Access **TOOL LIBRARY**

1. Click on **SELECT** under **TOOL**



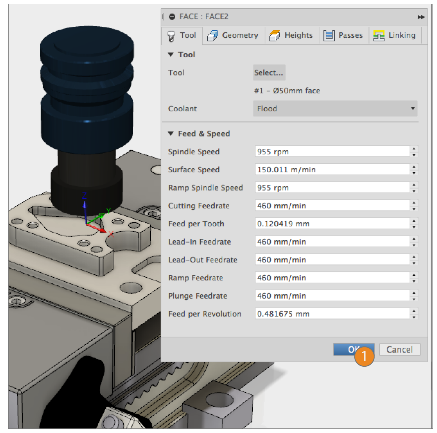
Step 12: – Select a Face Mill

1. Select the **#1 50 mm Face Mill**
2. **Click OK**

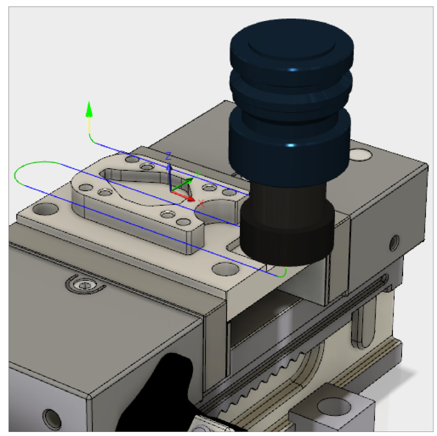


Step 13: – The **FACE** Operation

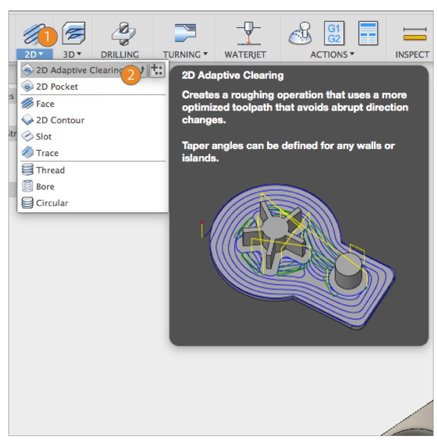
1. Click **OK**



Step 14: – **FACE Operation COMPLETE**

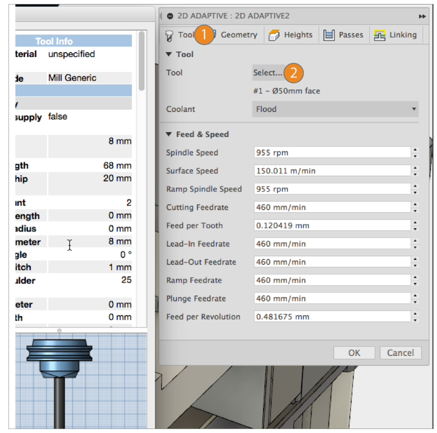


Step 15: – Apply a **2D Adaptive Clearing** Operation



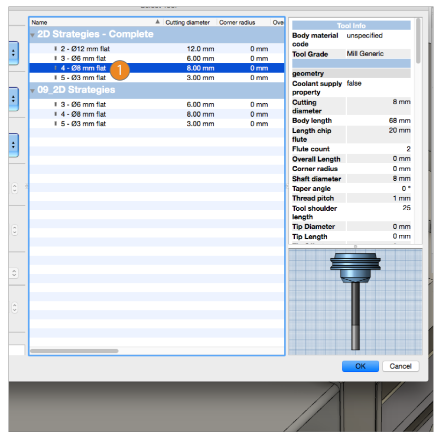
Step 16: – Select a NEW TOOL

1. Click **Tool**



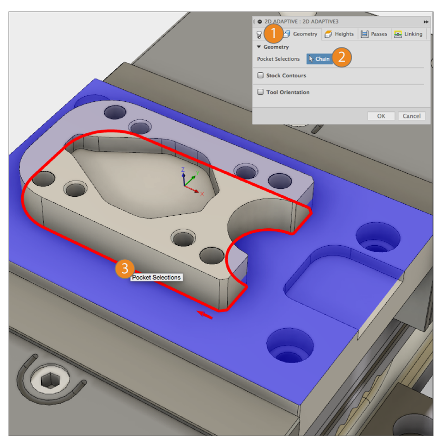
Step 17: – **Select a #4 8 mm Flat End Mill**

THEN CLICK **OK**

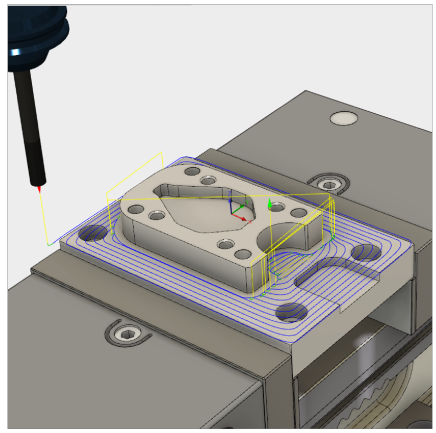


Step 18: – Select Geometry

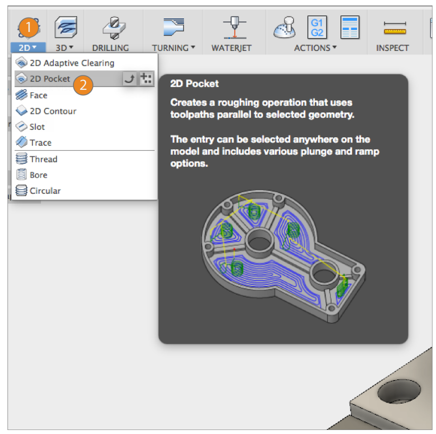
1. Click the **Geometry** Tab
2. Activate **Pocket Selection**
3. Click the outside of the Boss (IN RED)
4. CLICK **OK**



Step 19: – Toolpath Generated

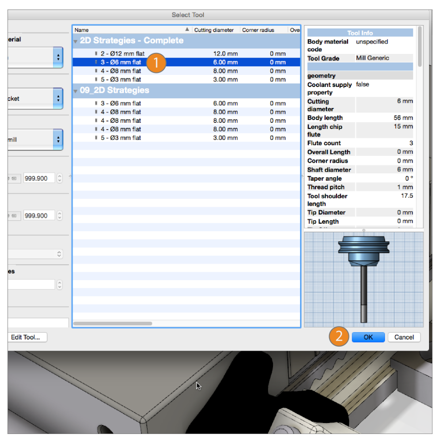


Step 20: – Apply a **2D Pocket** Operation



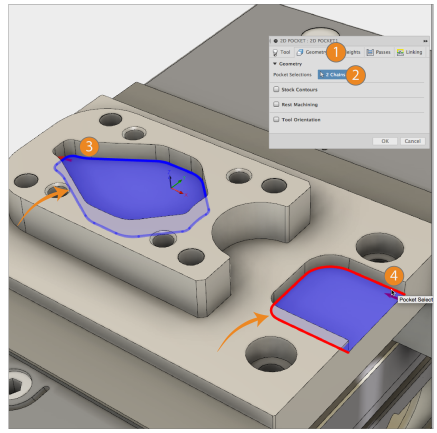
Step 21: – Pick the correct tool

1. #3 6mm Flat End Mill

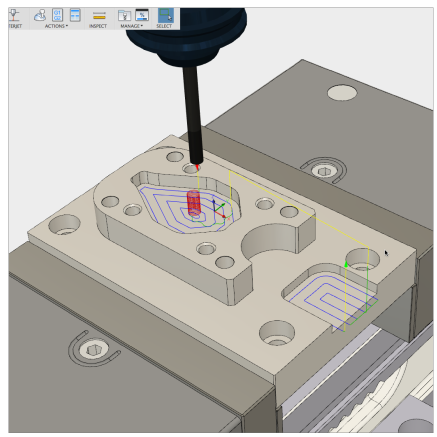


Step 22: – Select Geometry

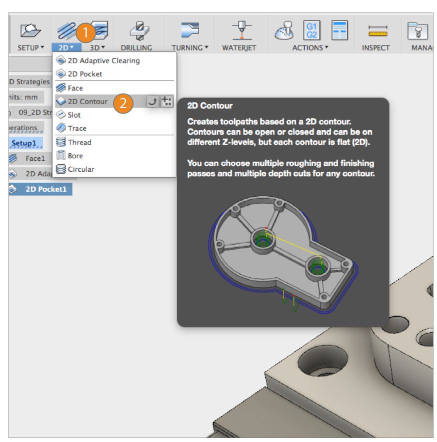
1. Click the Geometry Tab
2. Activate **Pocket Selection**
3. Click the inside embossed bottom **EDGE**
4. Click the bottom **EDGE** of the embossed open pocket (IN RED)
5. CLICK **OK**

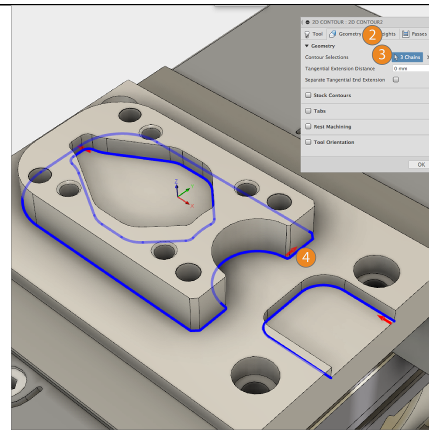
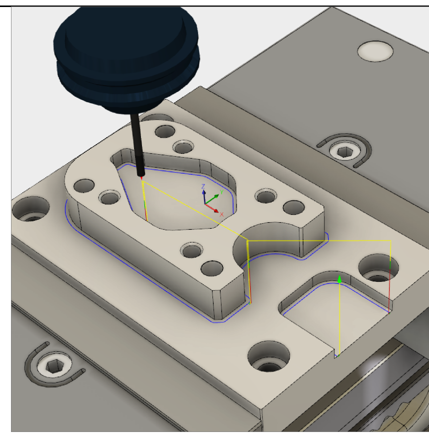
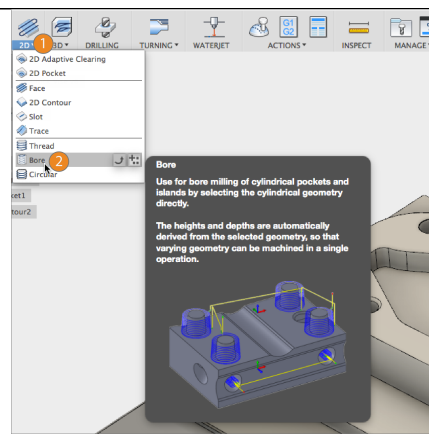


Step 23: – Toolpath Generated



Step 24: – Apply a **2D Contour** Operation for a finishing pass.

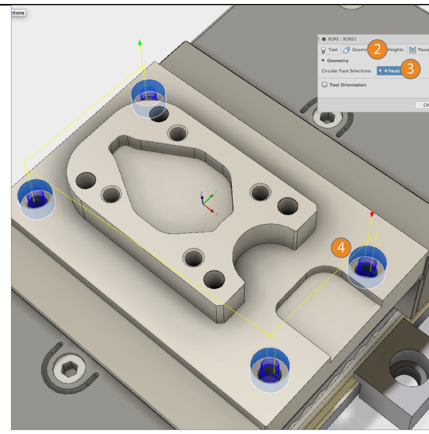


Step 25: – Select correct **TOOL** and **GEOMETRY**. 1. Under the Tool Tab, select a **#5 3mm Flat End Mill.** 2. Click the **Geometry** Tab. 3. Activate **Pocket Selection** 4. Click the **3 EDGES** shown. CLICK **OK** **TIP: CLICK ON THE RED ARROW TO HAVE THE TOOLPATH FOLLOW ON THE OUTSIDE/INSIDE OF THE BLUE LINE.**	
Step 26: – Toolpath Generated	
Step 27: – Create a **BORE** Operation	

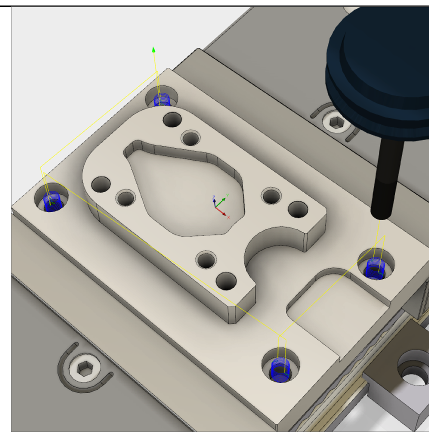
Step 28: – Select correct **TOOL** and **GEOMETRY**.

1. Under the Tool Tab, select a **#4 8mm Flat End Mill.**
2. Click the **Geometry** Tab.
3. Activate **Circular Face Selections**
4. Click the **4 Internal Faces of the Holes** shown.

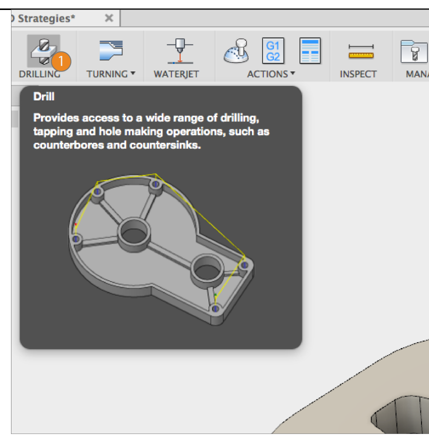
CLICK **OK**

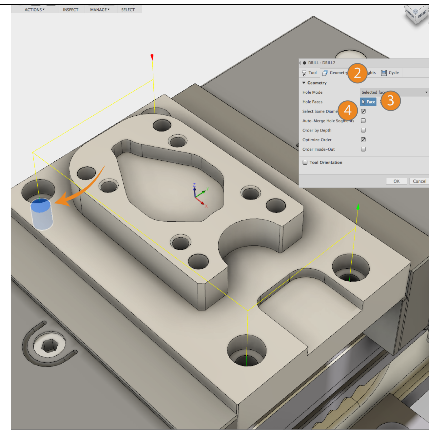
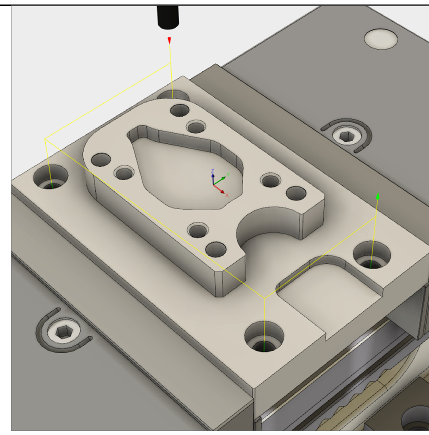


Step 29: – Toolpath Generated

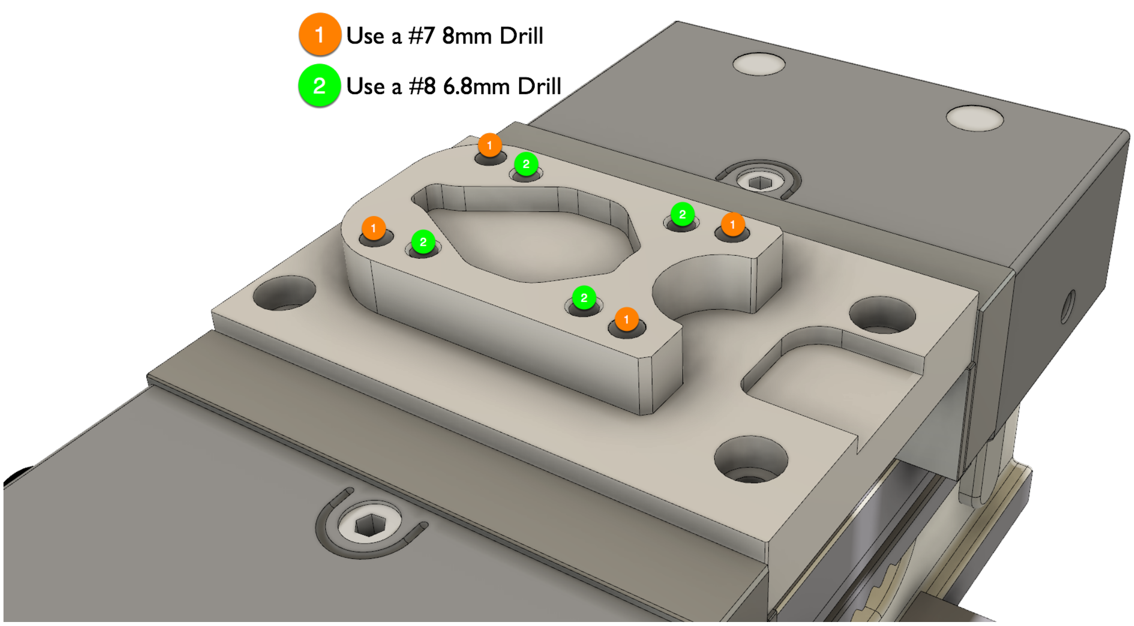


Step 30: – Create a **DRILL** Operation

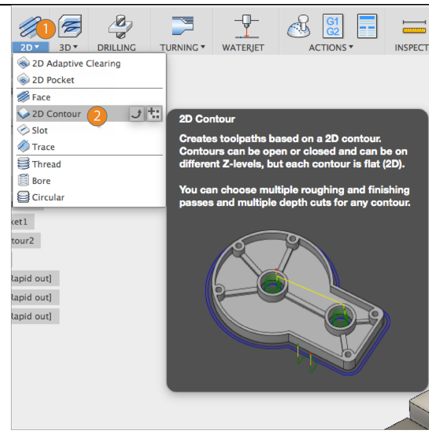


Step 31: – Select correct **TOOL** and **GEOMETRY**. 1. Under the Tool Tab, select a **#6 9 mm Drill**. 2. Click the **Geometry** Tab. 3. Activate **Hole Faces** 4. Click on **'Select Same Diameter'** 5. Select hole as shown CLICK **OK**	
Step 32: – Toolpath Generated	

Step 33: – Repeat **Step 31**, and use the designed Drill bits below for the holes designated.

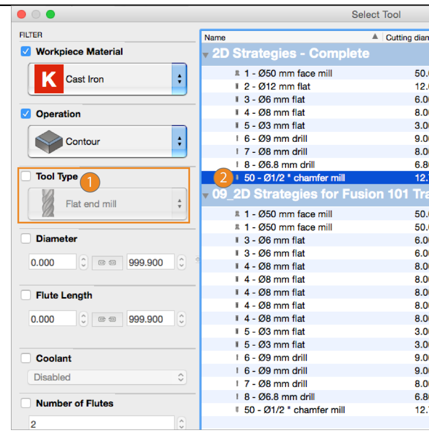


Step 34: – Create a **2D Contour** Operation



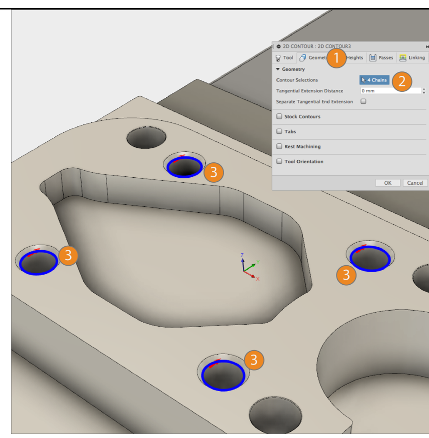
Step 35: – Pick the correct tool

1. Check **OFF** Tool Type
2. Select **#50 1/2" Chamfer Mill**



Step 36: – Select correct **GEOMETRY**.

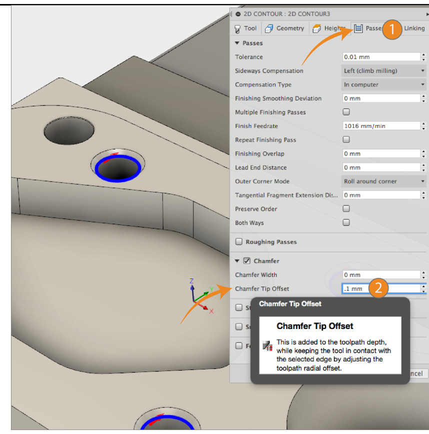
1. Click the **Geometry** Tab.
2. Activate **Contour Selections.**
3. Select edges as shown



Step 37: – Edit the **PASSES** Tab

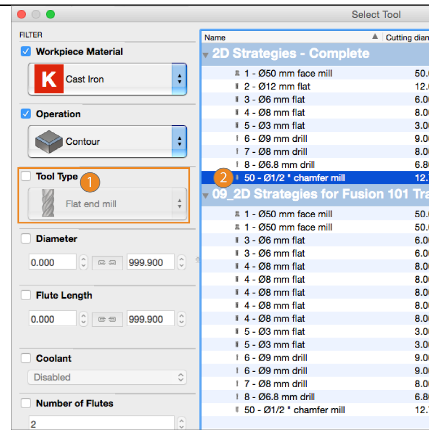
1. Click the **PASSES** Tab.
2. Under **Chamfer**, select **Chamfer Tip Offset**, and type in **0.1 mm**

CLICK OK



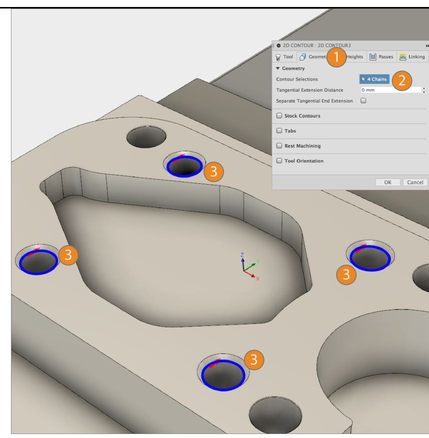
Step 38: – Pick the correct tool

3. Check **OFF** Tool Type
4. Select **#50 1/2" Chamfer Mill**



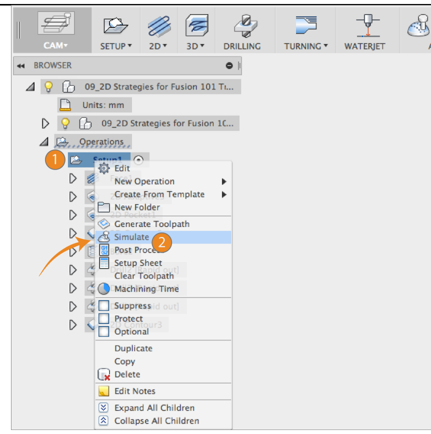
Step 39: – Select correct **GEOMETRY**.

3. Click the **Geometry** Tab.
4. Activate **Contour Selections.**
5. Select edges as shown

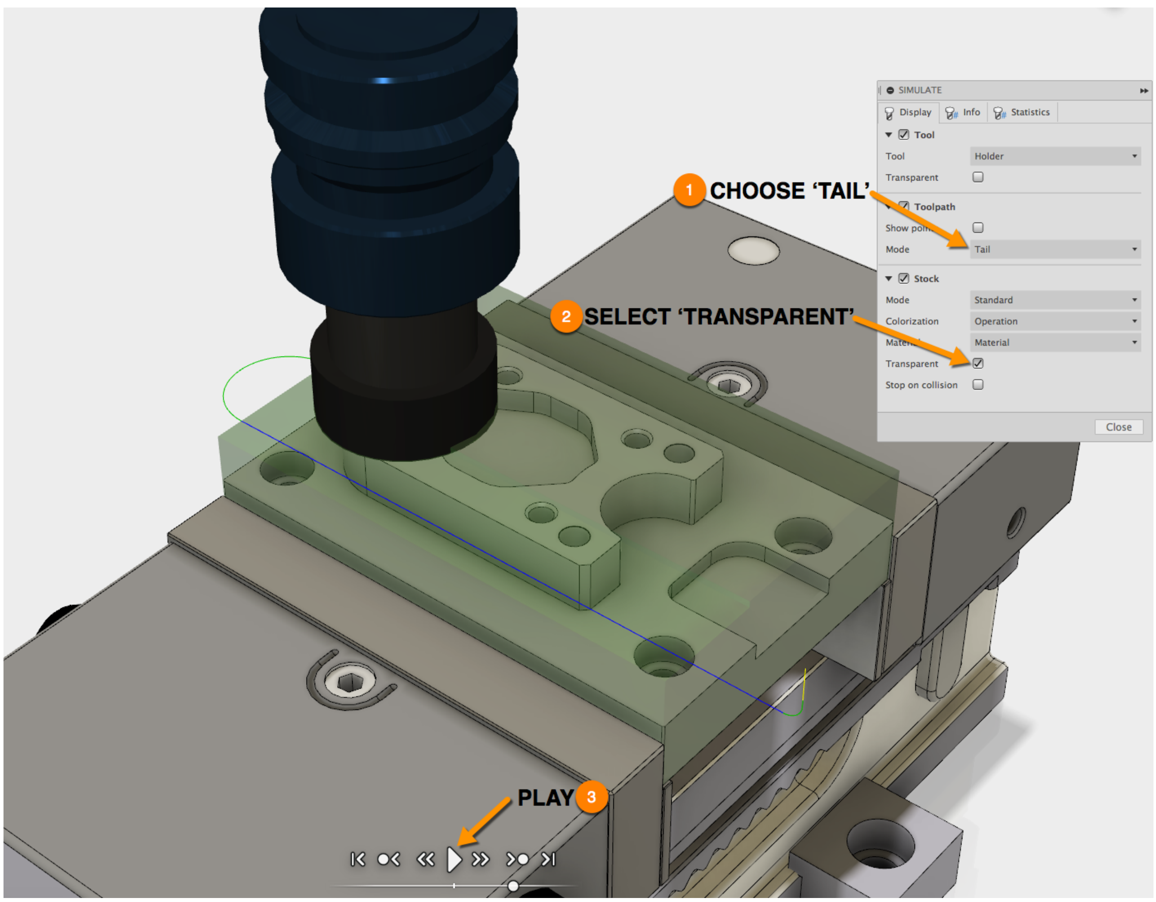


Step 40: – Simulate the Job

1. **RIGHT CLICK** on **SETUP** in the browser.
2. Select **SIMULATE**.



Step 41: – Choose the following **SIMULATE** presets below:

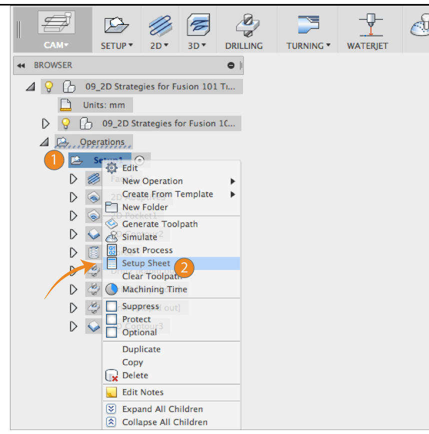


Setup Sheet

The Setup Sheet feature allows you to generate 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 42: – Create a Setup Sheet

1. **RIGHT CLICK** on **SETUP** in the browser.
2. Select **Setup Sheet**
3. Choose a location where to save it.
4. HTML will be generated of a Setup Sheet (see below).

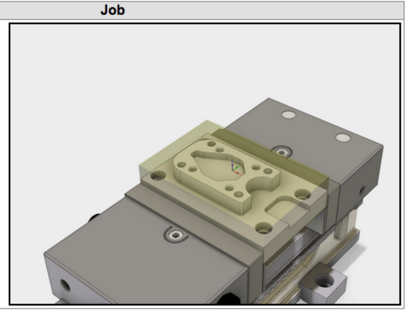


Setup Sheet for Program 1001

Job Description: Setup1
Document Path: 09_2D Strategies for Fusion 101 Training v4

Job

WCS: #0
Stock:
DX: 150mm
DY: 100mm
DZ: 37mm
Part:
DX: 150mm
DY: 100mm
DZ: 35mm
Stock Lower in WCS #0:
X: -75mm
Y: -50mm
Z: -37mm
Stock Upper in WCS #0:
X: 75mm
Y: 50mm
Z: 0mm



Total

Number Of Operations: 9
Number Of Tools: 8
Tools: T1 T3 T4 T5 T6 T7 T8 T50
Maximum Z: 15mm
Minimum Z: -36mm
Maximum Feedrate: 1256mm/min
Maximum Spindle Speed: 10500rpm
Cutting Distance: 6954.28mm
Rapid Distance: 2936mm
Estimated Cycle Time: 12m:25s

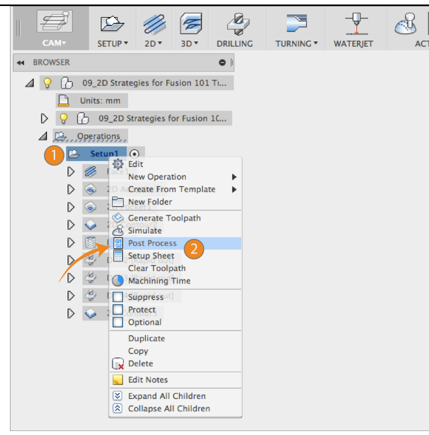
Operation 1/9 Description: Face1 Strategy: Facing WCS: #0 Tolerance: 0.01mm Maximum stepover: 47.5mm	Maximum Z: 15mm Minimum Z: -2mm Maximum Spindle Speed: 955rpm Maximum Feedrate: 460mm/min Cutting Distance: 600.24mm Rapid Distance: 22mm Estimated Cycle Time: 1m:19s (10.6%) Coolant: Flood	T1 D1 L1 Type: face mill Diameter: 50mm Length: 50mm Flutes: 4 Holder: BT40 - B4C3-0040
Operation 2/9 Description: 2D Adaptive3 Strategy: Adaptive 2D WCS: #0 Tolerance: 0.1mm Stock to Leave: 0.5mm Optimal load: 3.2mm Load deviation: 0.32mm	Maximum Z: 15mm Minimum Z: -16.5mm Maximum Spindle Speed: 3900rpm Maximum Feedrate: 1256mm/min Cutting Distance: 3514.11mm Rapid Distance: 775.83mm Estimated Cycle Time: 3m:57s (31.8%) Coolant: Flood	T4 D4 L4 Type: flat end mill Diameter: 8mm Length: 68mm Flutes: 2 Holder: Default Holder
Operation 3/9 Description: 2D Pocket1 Strategy: Pocket 2D WCS: #0 Tolerance: 0.1mm Stock to Leave: 0.5mm Maximum stepover: 5.7mm	Maximum Z: 15mm Minimum Z: -21.5mm Maximum Spindle Speed: 5233rpm Maximum Feedrate: 1256mm/min Cutting Distance: 960.78mm Rapid Distance: 159.27mm Estimated Cycle Time: 1m:47s (14.3%)	T3 D3 L3 Type: flat end mill Diameter: 6mm Length: 56mm Flutes: 3 Holder: Default Holder

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 producemore standard library posts, helps our Autodesk CAM Resellers and end-users with postrequests.

For more information on Post Processors, please review the [Autodesk Post Processor Manual.]

Step 43: – Outputting to a Post Processor 1. **RIGHT CLICK** on **SETUP** in the browser. 2. Select **Post Process**	
Step 44: – Outputting to a Post Processor 1. From Post Processor, select your Machine For Inquiries on specific posts, please email **cam.posts@autodesk.com**