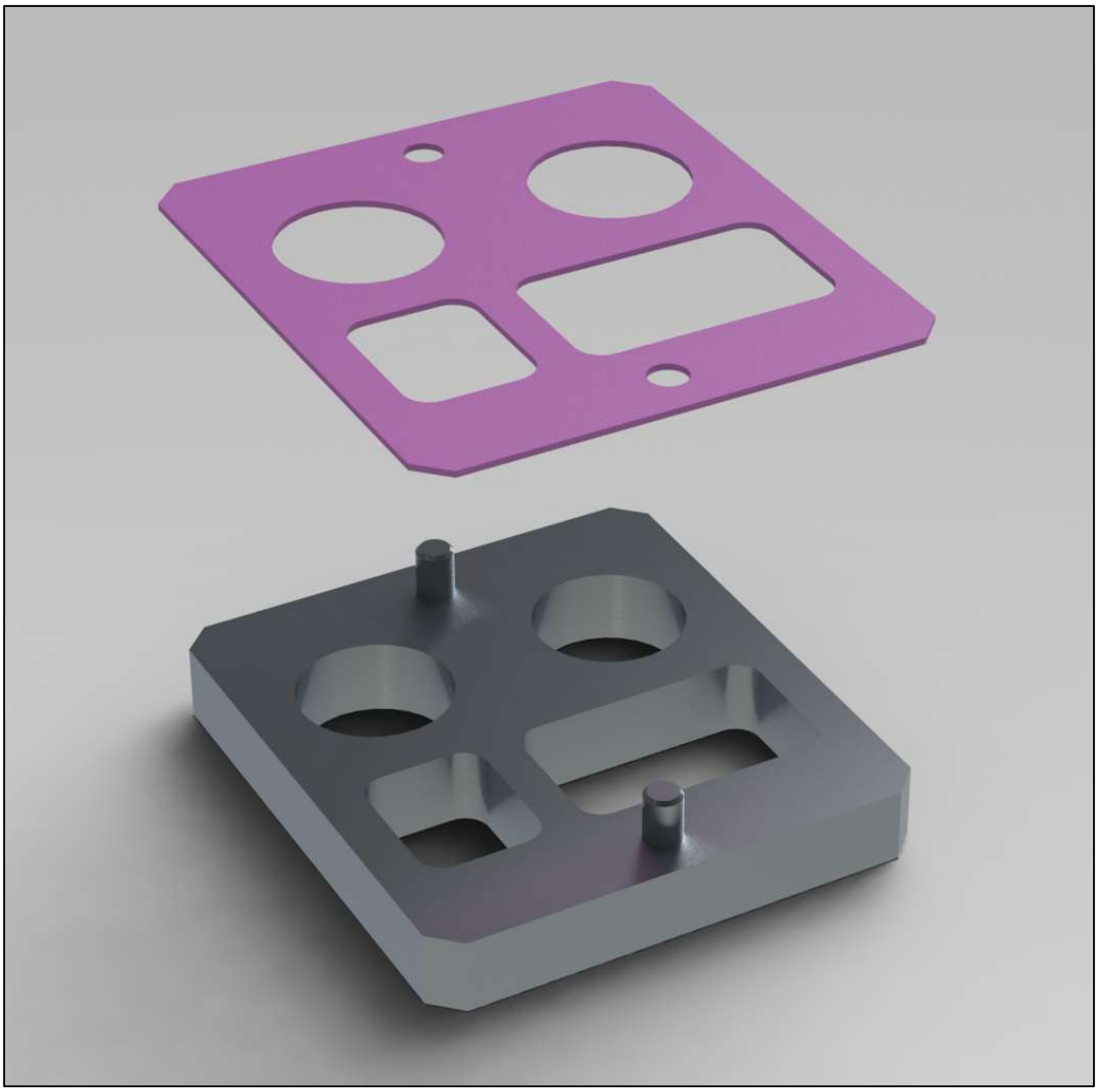
GASKET

THE GASKET PROJECT



Boston University College of Engineering
Engineering Product Innovation Center

The Gasket Project provides a brief introduction to HSMWorks, a Solidworks add-in used to create CNC programs for a variety of manufacturing machines. For a more comprehensive guide, see the **Intro to HSMWorks for 3-Axis Milling** guide.

TABLE OF CONTENTS

Rev 2024.09.15

PROJECT OVERVIEW

WHAT IS A GASKET?

A gasket is a shaped piece or ring of rubber or other material sealing the junction between two surfaces in an engine or other device. Gaskets help to seal mating surfaces by compensating for geometric variations resulting from surface irregularities and thermal expansion.

This project was chosen to reflect a common engineering scenario in which components are designed to fit existing parts or assemblies. When doing so, it is essential to consider which feature(s) to use as a datum.

WHAT IS A DATUM?

A datum is a plane, a straight line, or a point that is used as a reference when processing a material or measuring the dimensions of a target. It is usually an important functional feature that needs to be controlled during measurement.

In this project, the two dowel pins are used to establish a datum. By controlling the relationship between the dowel pins and all of the features used to define the boundaries of the gasket, we ensure that the that the elements of the assembly align properly. To make sure these parts fit together, it is essential to add a clearance fit.

WHAT IS A CLEARANCE FIT?

A clearance fit describes a geometric relationship in which clearance is designed between mating parts in an assembly. When designing assemblies, it is important to consider how much clearance to allow between features.

In this project, it is recommended to design .005" clearance around the dowel pins, and .015" around both internal and external features of the fixture body. This will ensure that the gasket fits over the dowel pins, and that enough clearance exists between the gasket and fixture that neither internal or external feature planes will be violated by the gasket.

Why might this matter? In the case of engine components, a poorly sized gasket can interfere with internal flow characteristics and/or external mounting relationships.

PROJECT WORKFLOW

Your mission is to use the components provided in your measurement kit (calipers, pins, threaded posts and straight edge) to measure all of the features of a specific fixture. Note: each fixture is custom, so be sure to compare your gasket to the fixture you begin with.

1. Create a hand drawn illustration of the fixture and transcribe all relevant dimensions necessary to create a CAD drawing. Note: you will be using the two dowel pins as a datum.
2. Create a 3D CAD model of a gasket based on your dimensions. Note: be sure to include clearance as described above.
3. Use HSMWorks to program a CNC routine to cut the gasket on our HAAS CM-1 3-axis milling machine.
4. CNC machine the gasket, and compare your results to the fixture used for measurement.

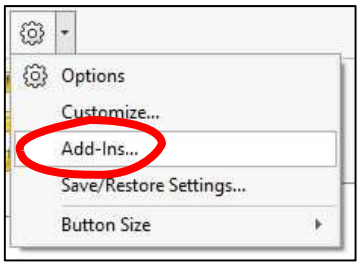
BEFORE YOU BEGIN

CONFIRM HSMWORKS IS ACTIVE

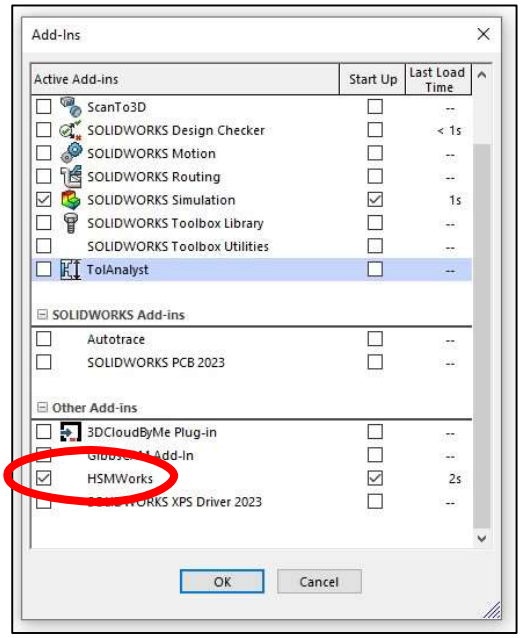
Confirm that the HSMWorks add-in is active in your Solidworks version with the following steps. If the add-in is not present, consult your course administrator.

Alternately, you may install the add-in on your personal computer which requires creating an Autodesk account with your Boston University email, and downloading the HSMWorks installer. Find more info here: https://www.autodesk.com/support/account/education/onboarding/students-guide

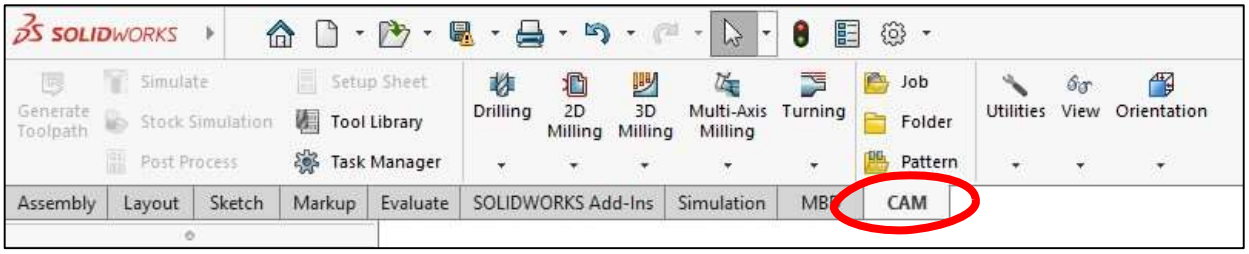
1. Located at the topmost toolbar of the Solidworks application, open the **Options** submenu and select **Add-Ins.**



2. In the **Add-Ins** popup window, scroll down to **Other Add-ins** and ensure that **HSMWorks** is present in the list and its checkbox is active.



3. Access the HSMWorks add-in by selecting the **CAM** tab located in the CommandManager. The main toolbar interface should appear as in the image below.

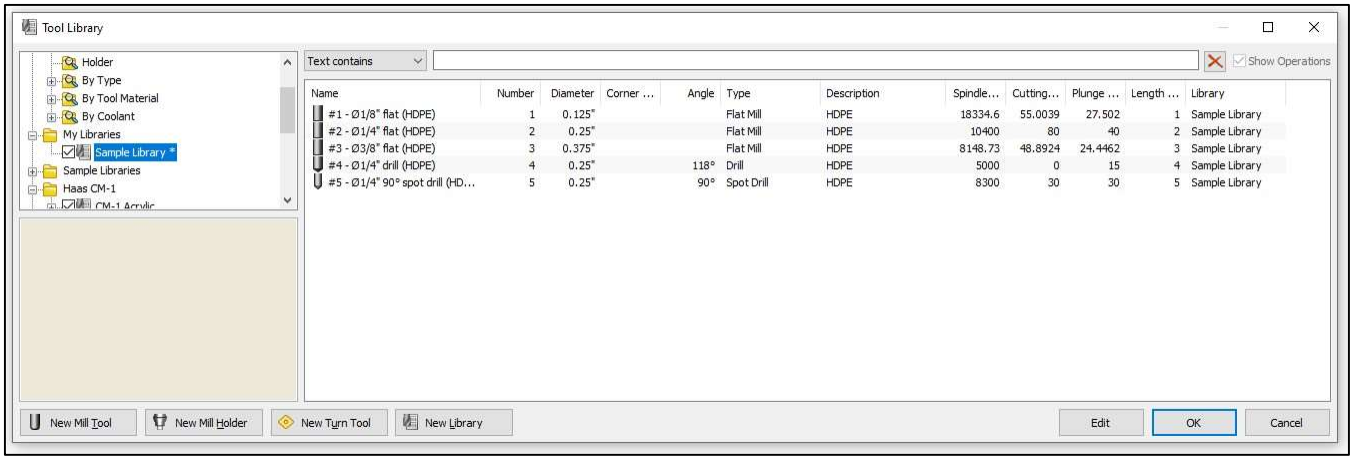


IMPORT TOOL LIBRARY

1. Select **Tool Library** in the CommandManager to open the Tool Library pop-up window.
2. Right click on **My Libraries** in the file tree, select **New Library,** and rename it **Sample Library** .
3. Right click on **Sample Library** , select **Import Tools from Library,** then open the file provided by your instructor titled **CM-1 HDPE.hsmlib.** The tool library can also be found at the following network location on an EPIC computer:

   EPIC > Equipment > Tool Libraries > HSMWorks > Haas CM-1

4. If done correctly, the Sample Library browser should be populated with 5 tools, similar to the image below.



IMPORT GASKET TO FIXTURE PLATE ASSEMBLY

1. Open the **cm1_fixture_plate** Solidworks assembly file provided by your instructor. If you are working at a computer station in EPIC, the file can be found at the following network location:

   EPIC > Equipment > Training > HSMWorks

2. Insert your gasket part file into the assembly, add a **Mate** feature, and center align the bottom face of your gasket with the top face of the fixture plate (Figure 1).

3. After confirming the Mate feature, select the **CAMManager** tab to access the HSMWorks module.

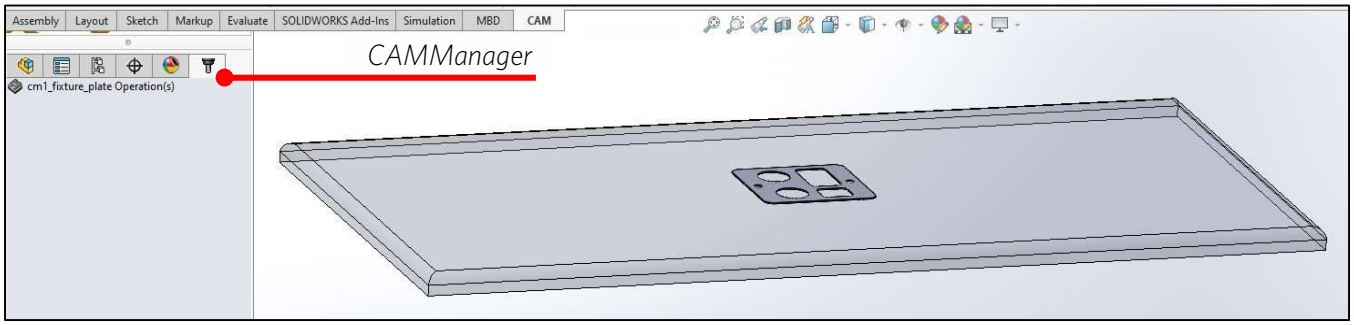


Figure 1

PROGRAMMING PROCESS STEPS

There are four main components to creating a CNC program using HSMWorks:

JOB SETUP involves defining the model to be machined, the stock dimensions, and the Work Coordinate System (WCS) and fixturing device as they are relative to the machine.

TOOLPATH CREATION involves selecting geometry and parameters that will drive the path a tool takes to produce a desired shape.

SIMULATION renders a graphical animation of the machining process, and can provide valuable feedback regarding safety and efficiency.

POST PROCESSING outputs setup and toolpath data as a single G-Code file to be used at the CNC machine.

JOB SETUP

4. Select the **Job** icon located in the CommandManager. This will open a **Job Properties** panel at the left side of the user interface (Figure 2).
5. Define the **Job Properties** parameters:
    a. Beneath **Model,** select the empty text field then select the gasket model in the Graphical Display.
    b. Open the submenu under **Stock**, select **Fixed Size Box**, and enter the following values:

Width (X):	3in
Depth (Y):	2.75in
Height (Z):	0.06in
Offset from bottom (-Z):	0in

        Note how the object that represents the stock in the graphical display updates accordingly, but is not aligned correctly to the gasket model.
    c. Beneath **Fixture**, select the text field then select the fixture plate model to assign it as a fixture component.
    d. In the **Work Coordinate System (WCS)** submenu, select **Use Z-Axis and X-Axis.**
    e. Highlight the first WCS field and select any vertical edge on the gasket model to align the Z axis.
    f. Highlight the second WCS field and select any horizontal edge on the gasket model to align the X axis.
    g. In the second **Origin** submenu, select **Bottom Center** to align the XYZ origin point accordingly.
    h. Select the green checkmark at the top of Job Properties to confirm the setup.

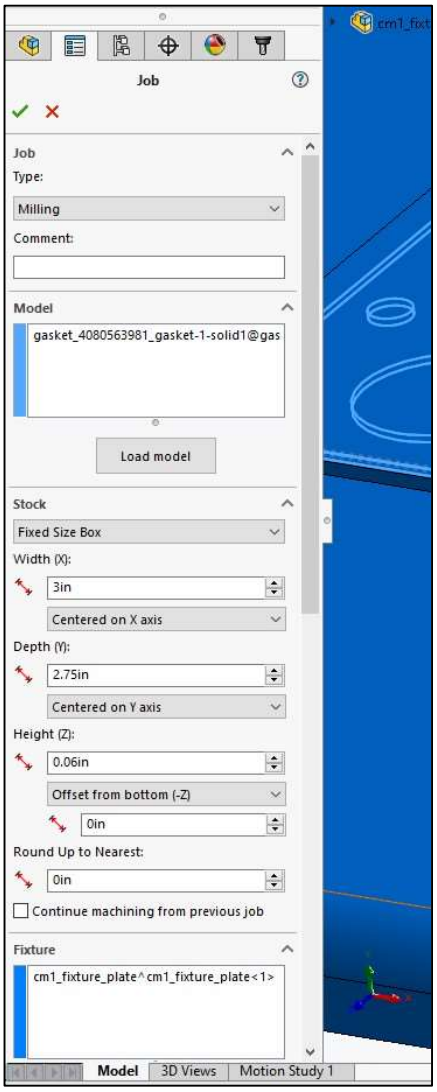


Figure 2

The WCS origin should now be aligned at the bottom center of both the stock and model and oriented as seen in Figure 3. This XYZ axes orientation is crucial as it directly relates to the machine that will make the part.

You can further confirm this orientation by inspecting the Front View using the View Orientation tool.

If the Job Setup needs to be edited, right click on Job in the CAMManager tree at the left side of the application and select Edit.

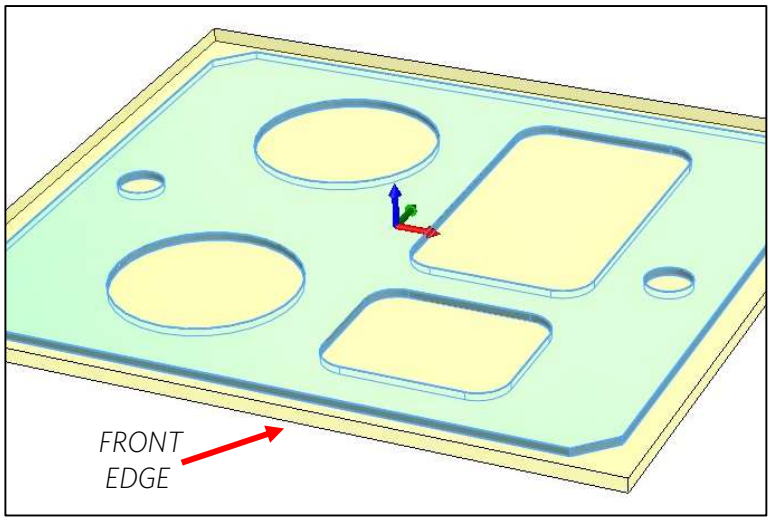
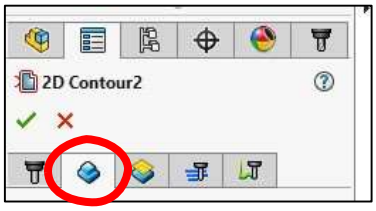


Figure 3

TOOLPATH CREATION

1. In the CommandManager, open the **2D Milling** submenu and select **2D Contour** (Figure 4) to open the **Toolpath Properties** panel.
2. Under **Tool**, select **Library > My Libraries > Sample Library > #1 Ø1/8" flat (HDPE)** and click the **Select** button to assign the tool to the operation.
3. In the Toolpath Properties panel, select the **Geometry** tab.



4. In the Graphical Display, select the bottom edge of each contour (Figure 6) you wish to machine, in the most practical and efficient order. It is common practice to machine the internal geometry first.
5. The black arrows alongside each contour denotes how the assigned tool will be offset relative to the contour path. Switch the orientation by highlighting a selection in the **Model** field, then selecting **Reverse** just beneath the field (Figure 6). Orient the arrow positions so they will produce the correct geometry for your gasket.
6. In the **Passes** tab, activate the **Preserve Order** checkbox (Figure 5). This step directly relates to the order of selection done in step 4.

Note: Hover the cursor over most icons, parameters, and value fields for contextual pop-ups!

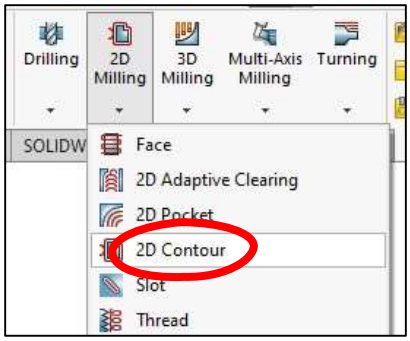


Figure 4

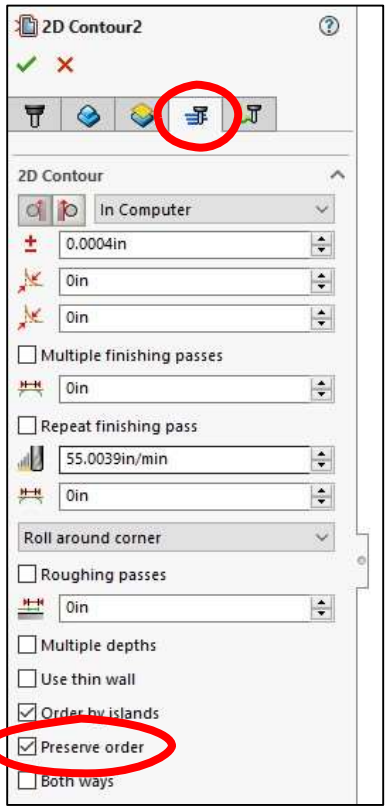


Figure 5

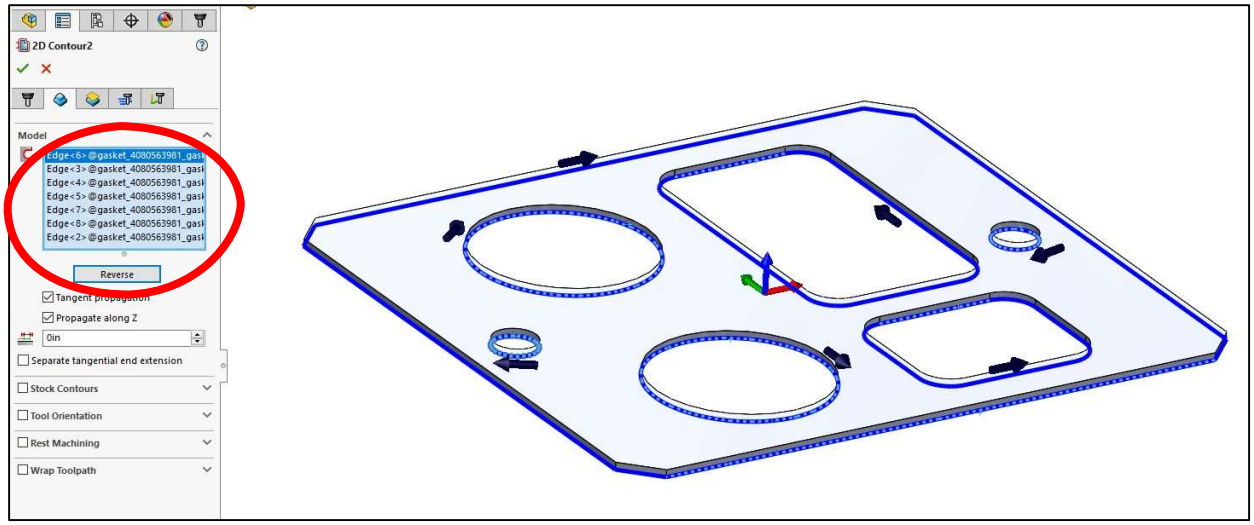


Figure 6

7. Select the green checkmark at the top of the Properties panel to confirm the parameters and generate the toolpath. The graphical display should resemble Figure 7.

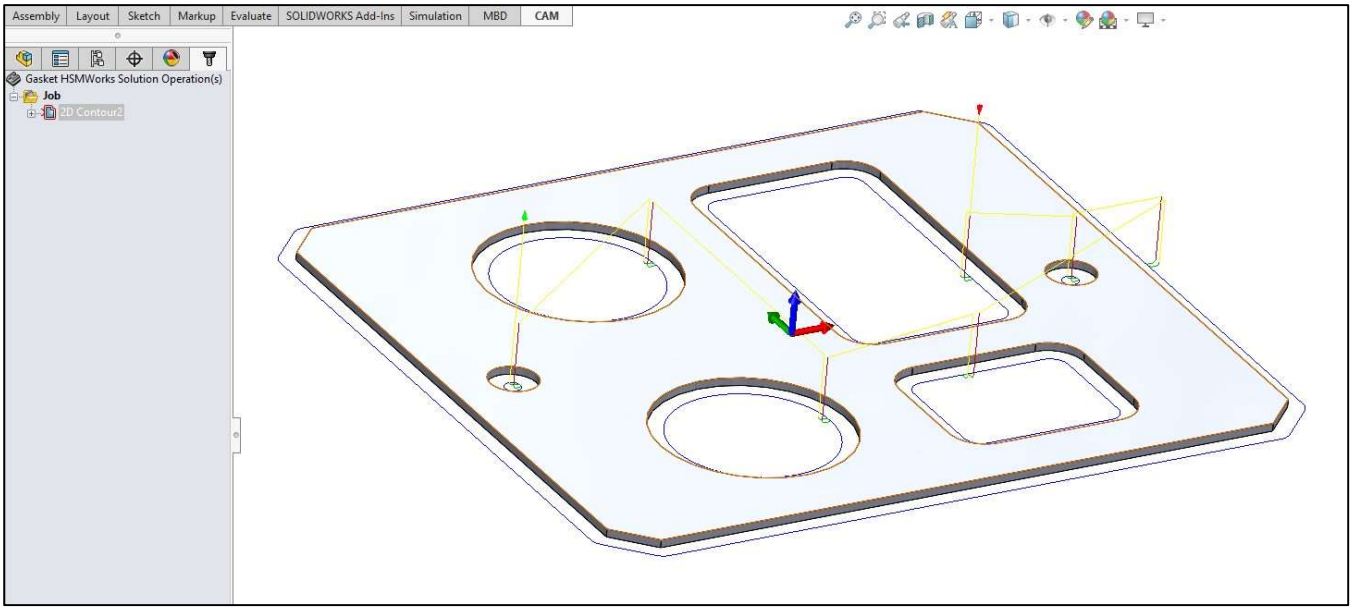


Figure 7

SIMULATION

1. With the **2D Contour** operation highlighted in the **CAMManager** panel, select **Stock Simulation** in the CommandManager to open its properties panel.
2. In the Stock Simulation Properties panel, ensure that the **Tool Visibility** buttons **tool, shaft,** and **holder** are active (Figure 8). This will provide a graphical rendering of the tool which should be dimensionally accurate to the actual tool setup at the machine.
3. Activate the **Check against fixture** and **Stop on clash when animating** buttons. These functions will signal the user if the tool comes into contact with the fixture plate, which is a hazard.
4. Move the animation **Speed** slider to the 2nd mark.
5. Select the **Play** icon to begin the animation.

6. A pop-up window will detail if a clash has occurred, otherwise it will list information on the operation.

If a clash has occurred, you must review and edit the 2D Contour operation parameters, then re-run the simulation.

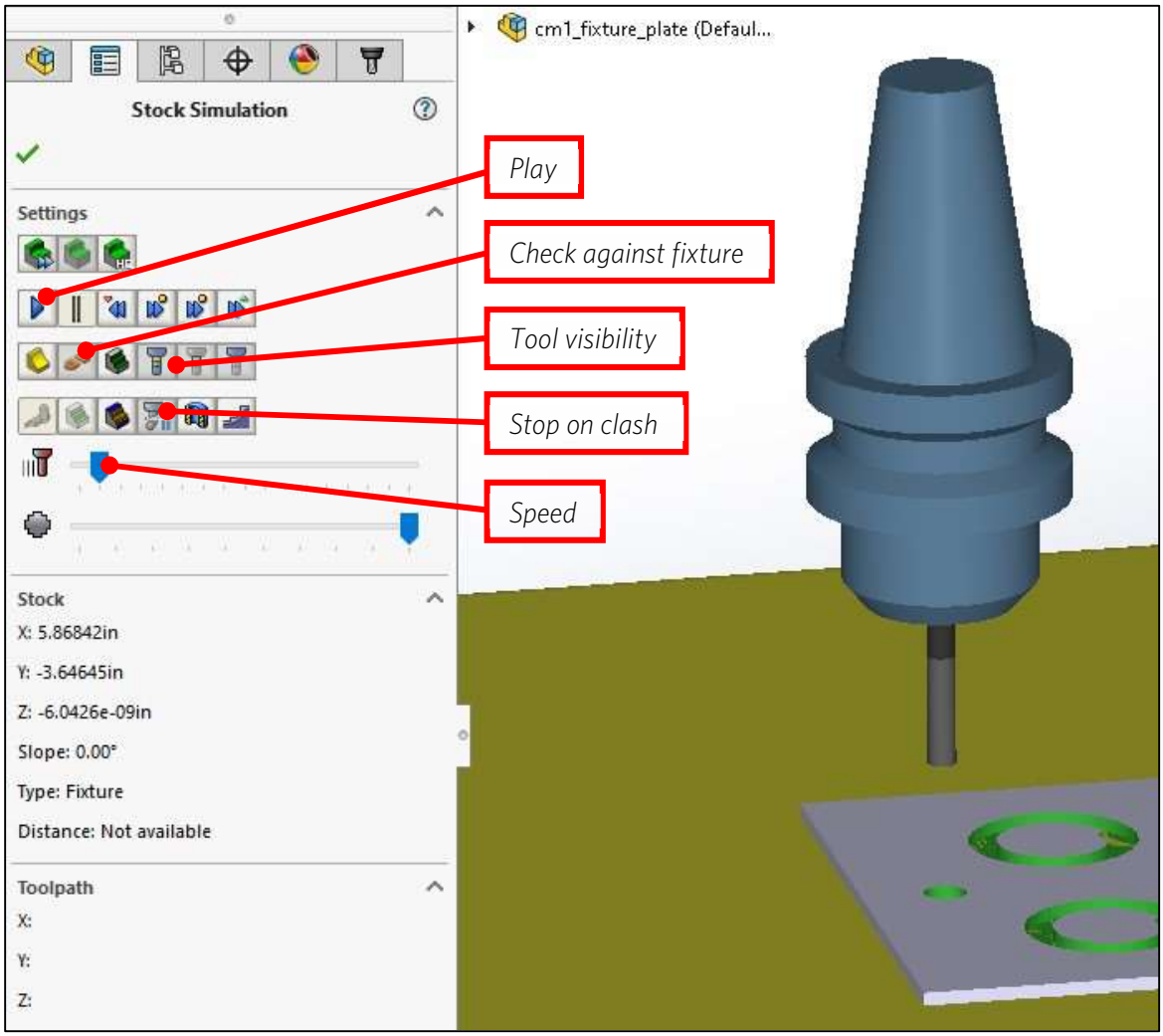


Figure 8

7. Select the green checkmark to exit the simulation. If you need to edit the operation, right click on it in the CAMManager tree and select **Edit.**

POST PROCESSING

Post processing is the final step in preparing your program to run on a CNC machine. This function computes all of your operations into a single G-Code file specific to the brand of controller used at the machine.

1. Right click on **Job** in the CAMManager tree and select **Post Process (All)** .
2. In the Post Process pop-up window (Figure 9) enter "haas" in the Post Configuration search bar, then select "**HAAS (pre-NGC) / haas"** from the dropdown list.

3. In the **Program name or number** field, enter the numbers **01**.
4. In the **Program comment** field enter: [FIRST NAME] [LAST INITIAL] COASTER
5. Ensure the **Open NC file in editor** checkbox is active.
6. Select the **Post** button at the bottom of the window and choose the appropriate directory location to save the file. **Staff will advise whether the file should be saved to a USB drive or network location.**
7. Format the file name as [FIRST NAME] [LAST INITIAL] COASTER and select **Save.**
8. The **Autodesk HSM Edit** application window should automatically open to display your program as G-Code (Figure 10).
9. In the header of the G-Code, ensure that the **ZMIN** value is not less than zero.

If ZMIN is a negative value, the tool will machine into the fixture plate. You will need to revise your program.

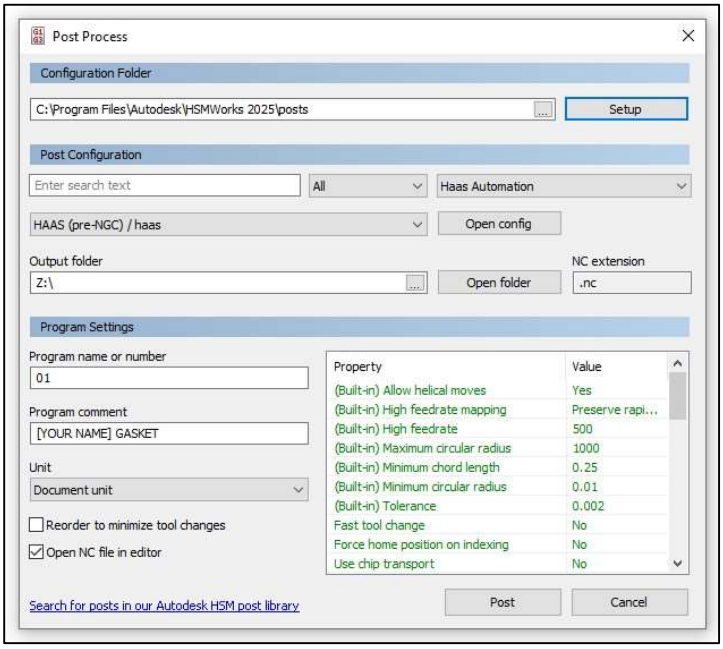


Figure 9

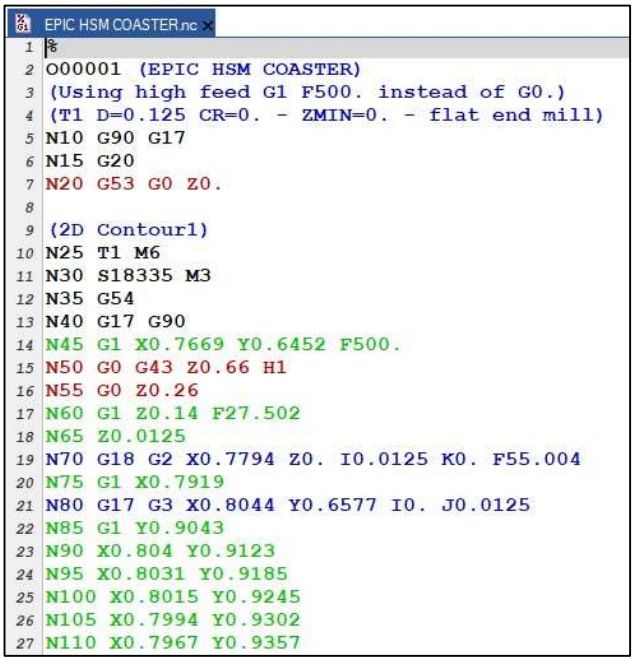


Figure 10

An EPIC staff member must approve your program prior to machining.

CALIPER USE – BEST PRACTICES

CLEAN AND INSPECT THE CALIPER JAWS AND WORKPIECE

Pinch and pull paper between the closed jaws of the calipers. This removes grease and debris from the jaws. Inspect both the calipers and the work being measured. Look for burrs, debris and imperfections.

CALIBRATE THE BEZEL

When fully closed, the indicator should point to the "0" mark. If not, loosen the bezel clamp screw and rotate the bezel until the indicator aligns with the "0." Re-tighten the clamp screw. This is referred to as "zeroing" out the caliper.

ALIGN CALIPER WITH FEATURE OF MEASUREMENT

Pay careful attention that the calipers are parallel with the feature of measurement, and that the jaws are perpendicular to flat surfaces when measuring. Misalignment will give you an imprecise measurement.

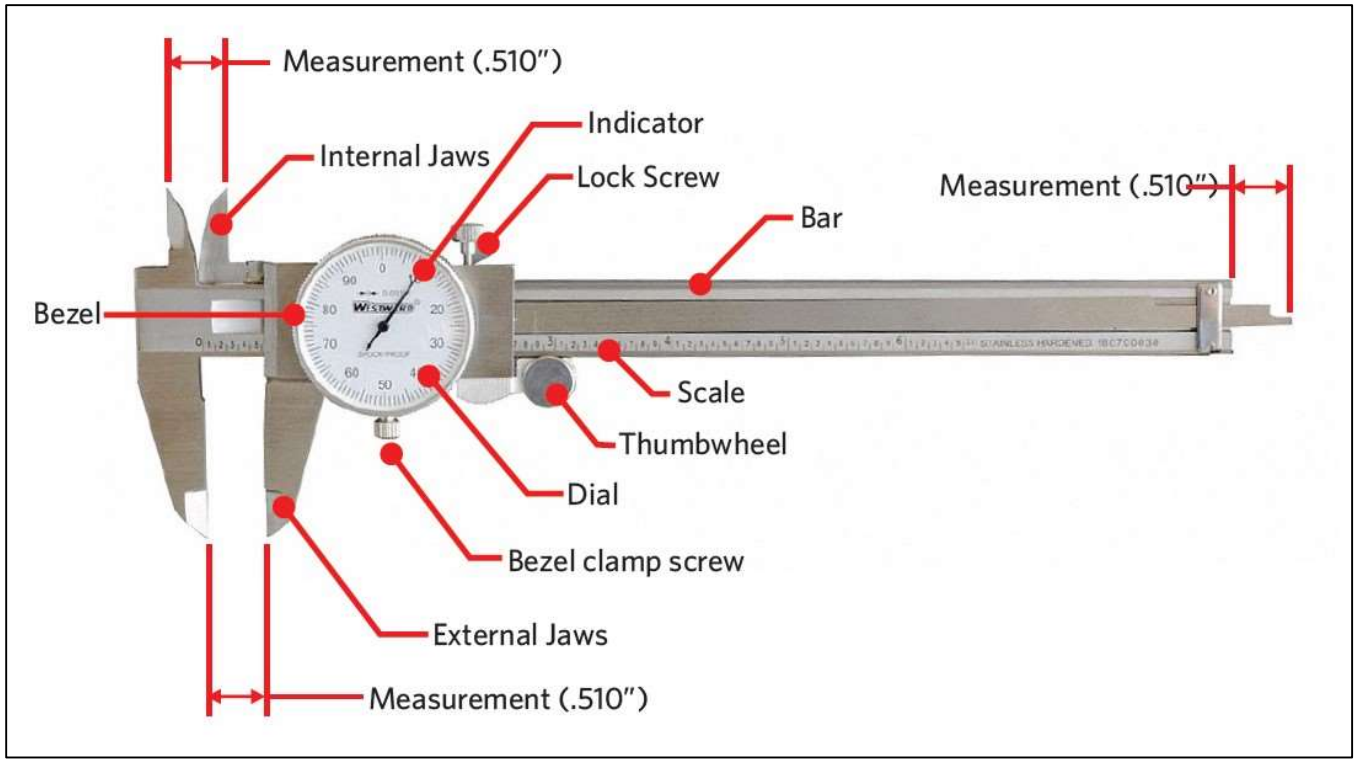
DUPLICATE THE MEASUREMENT

If new to the use of calipers, it is good practice to measure the same feature multiple times and see if you are able to duplicate your measurements. Seek guidance if you struggle to achieve consistent results.

RETURN TO ZERO

When done measuring, return the jaws to a closed condition and confirm that the indicator still reads zero. Misalignment suggests that your measurement may have resulted in a faulty reading.

CALIPER NOMENCLATURE



EPIC CNC MILL SPECIFICATIONS

Since 2022, EPIC has acquired four Haas CNC mills to support projects associated with the College of Engineering and the greater Boston University community. Capable of machining a variety of metals and plastics, Haas machine tools provide a high level of speed and precision.

HAAS CM-1

- 30,000 RPM spindle
- 5.0 HP
- Max Torque 8.0 ft-lbf@3000RPM
- Max Cutting Feed Rate of 500IPM
- 20-Pocket Automatic Tool Changer
- Work Envelope of 12″x10″x14″
- Taper ISO 20 Toolholder

HAAS VF-2

- 8,000 RPM spindle
- 30 HP
- Max Torque 90 ft-lbf@2000RPM
- Max Cutting Feed Rate of 650IPM
- 20-Pocket Automatic Tool Changer
- Work Envelope of 30″x16″x20″
- Taper ISO 40 Toolholder