BUMES Optimization and Vision

CAD/CAM Supporting Documentation

ME500 Advanced Manufacturing Final Project
Spring 2025
Wednesday AM Lab Group

Azamat Abdikarimov, Shrijit Banerjee, Aniruddha Dalal, Gaella Hawi, Kaleigh Hilal, Anirudh Sundaresan

Document Author: Anirudh Sundaresan

Table of Contents

Overview:

This CAM Support Document provides a comprehensive guide for using SolidWorks with HSMWorks to create CNC machining toolpaths.It walks through setting up parts and jobs, importing the correct tool libraries, and executing a wide range of CAM processes including 2D milling, drilling, and 3D adaptive clearing. The document also includes reference tables for tool parameters and outlines final steps such as simulation and post-processing for generating G-code.

1. SetUp:

1.1 Part SetUp:

In the Assembly Tab, click the drop down menu below Insert Components(Fig 1):

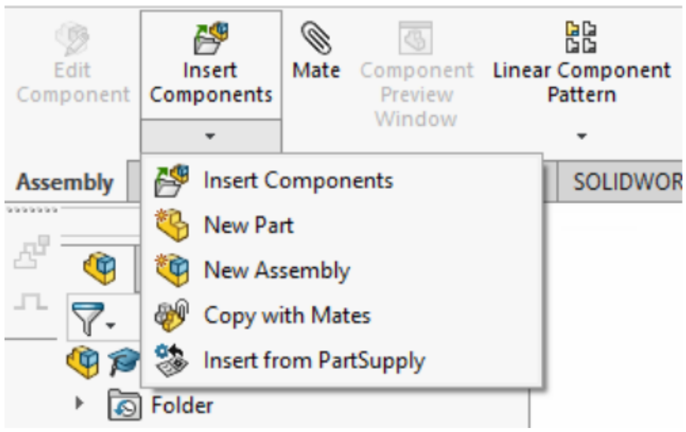


Figure 1: Insert Component

❖ The "Insert Components" option lets you add a part file to the assembly using the traditional bottom-up design approach.
  - Using the Mate option, you can align and assemble two or more components by applying appropriate mating constraints. The assembled bed along with the body is shown in figure 2.

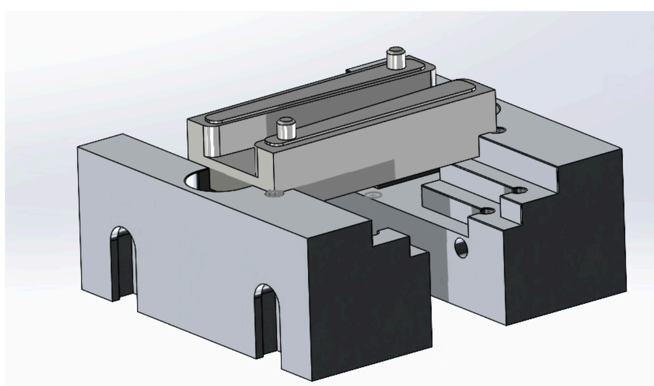


Figure 2: Component Assembly Using Mates

1.2 Job Setup:

❖ Click Job on the CAM tab as shown in fig 3:

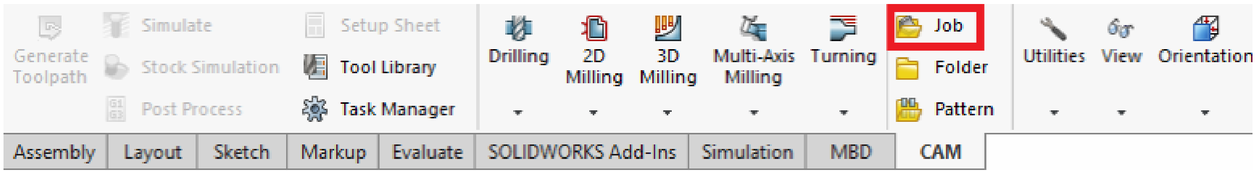


Figure 3: Job Option

- ❖ Adjust the settings in the PropertyManager for the job as follows:

  Type: Milling

  Model: Select the part that is to be machined.

  Stock: From Solid
- ❖ Choose the stock from the flyout FeatureManager Design Tree.
- ❖ Fixture: Select the soft jaws.
- ❖ Work coordinate System (WCS): Use Coordinate System

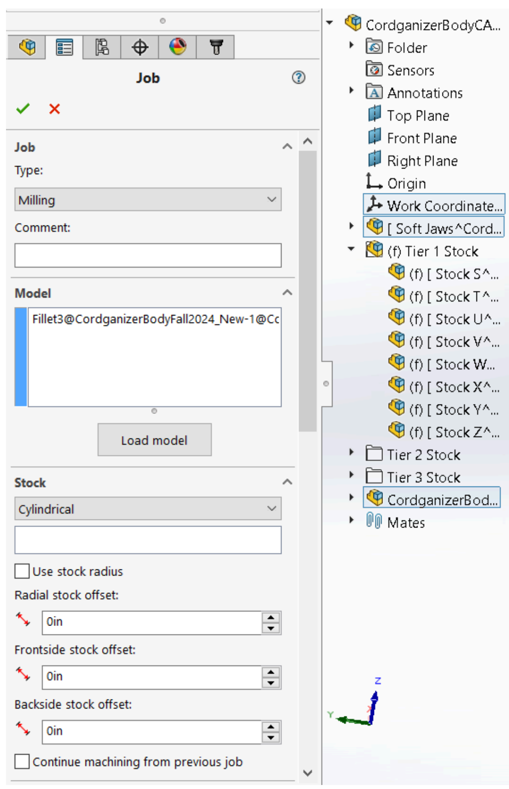


Figure 4: Job Setting

1.3 Tool Library:

- ❖ Download the Tool library file from blackboard and import into Solidworks using the options “Tool Library” as shown(Fig 5):

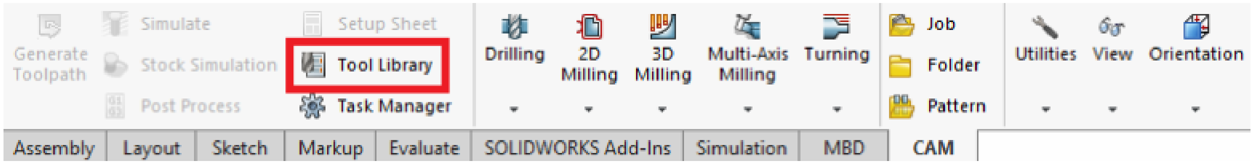


Figure 5: Tool Library Option

- ❖ Right Click on My Libraries.
- ❖ Select Import library to import the tool library.

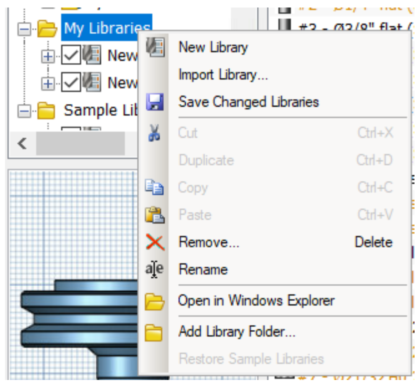


Figure 6: Importing Tool library

- ❖ When you open the tool library, it should appear as shown below.

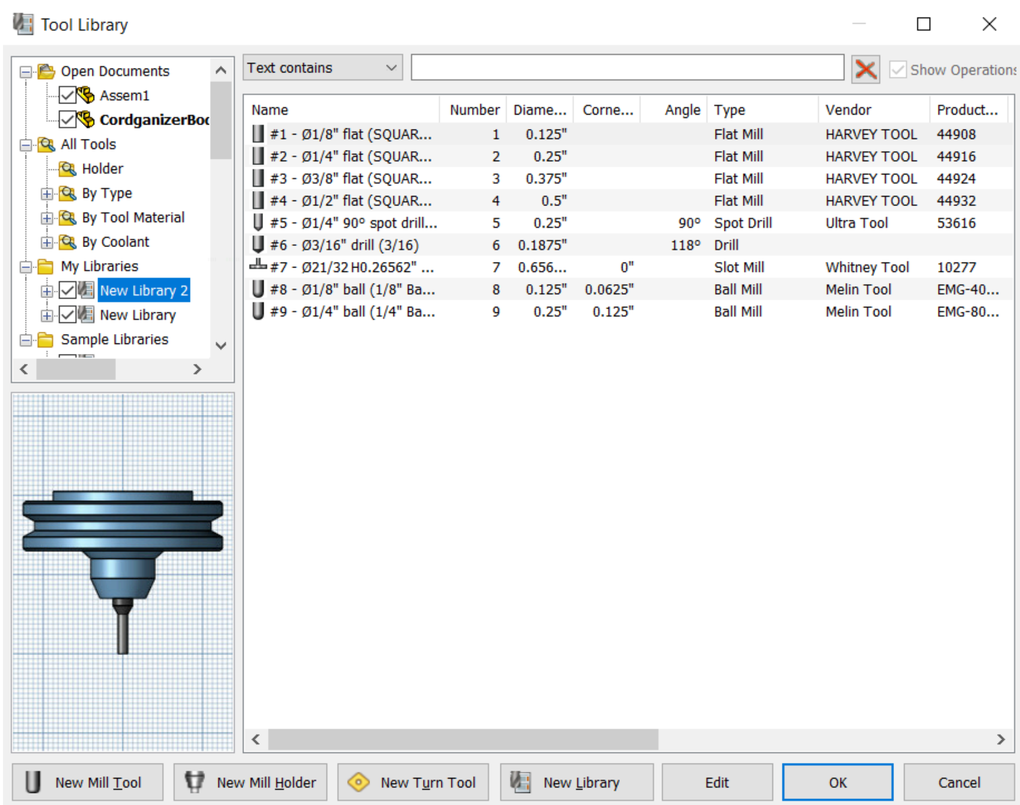


Figure 7: Tool Library

2. Cam Process for Body:

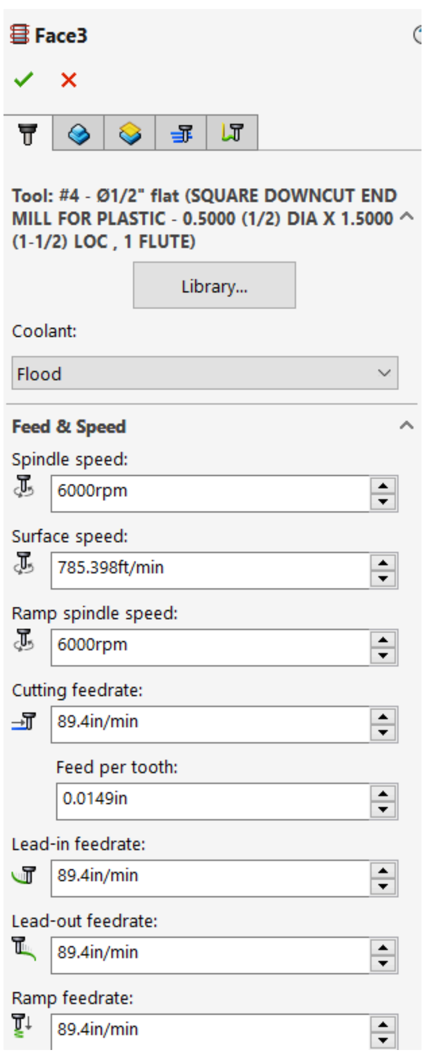
2.1 2D Milling

2.1.1 Facing

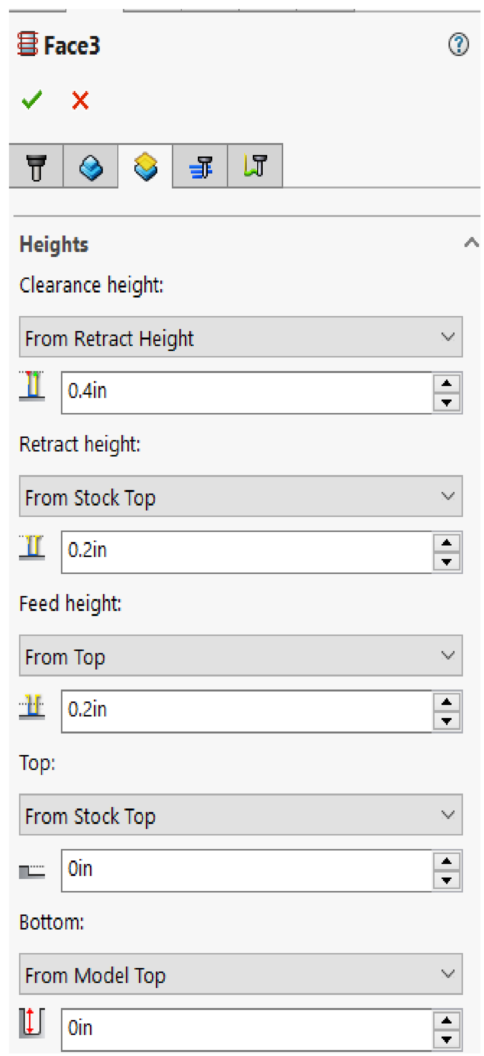
Facing is important because it flattens the top surface of the stock, ensuring a consistent and accurate starting height. This helps improve the precision of all subsequent machining operations and removes any surface irregularities.

The first step should always be to face the stock to make it the correct height.

- ❖ Tool - Tool 4 is selected from the tool library with the parameters shown below (Fig 9).
- ❖ Geometry - Do not select any geometry.
- ❖ Heights - Fill in the heights as follows(Fig 8):



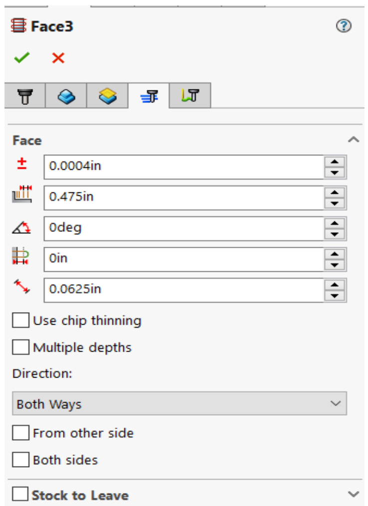
Face Tool



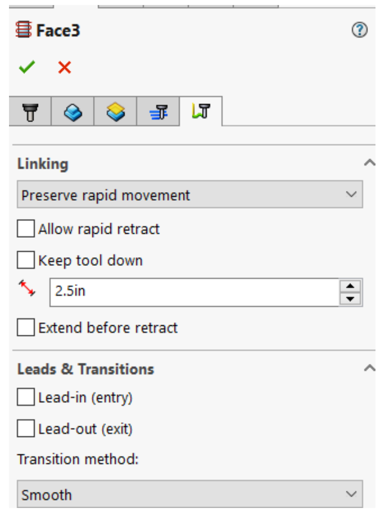
Face Height

Figure 8:Facing Parameters

- Passes : The important settings to change are below(Fig 9):
  - – Stock Offset: 0.0625in
  - – Direction: Both Ways
- Linking: The important settings to change are below(Fig 9):
  - – Linking: Preserve rapid movement



Face passes



Face Linking

Figure 9:Facing Parameters

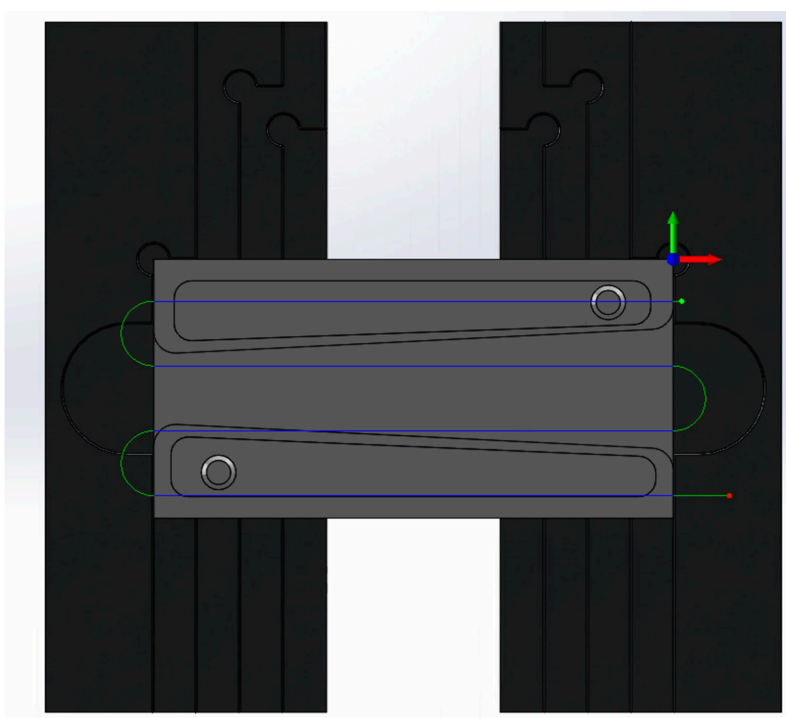


Figure 10: Face Tool Path

2.1.1 2D Contour

2D Contour is used to machine the outer edges or profiles of a part. It follows the selected boundary to cut along walls or outlines, helping shape the final dimensions of the part.

Basic

- ❖ Tools:
  - **Slot (Locking):** Tool 2 was used to create the slot required for the locking feature.
  - **Chamfer:** Tool 5 was used to create the chamfered edge.
- ❖ Geometry:
  - Edges and Faces can be selected.
  - Reverse will change which side of the contour is being cut as shown below in figure 11.

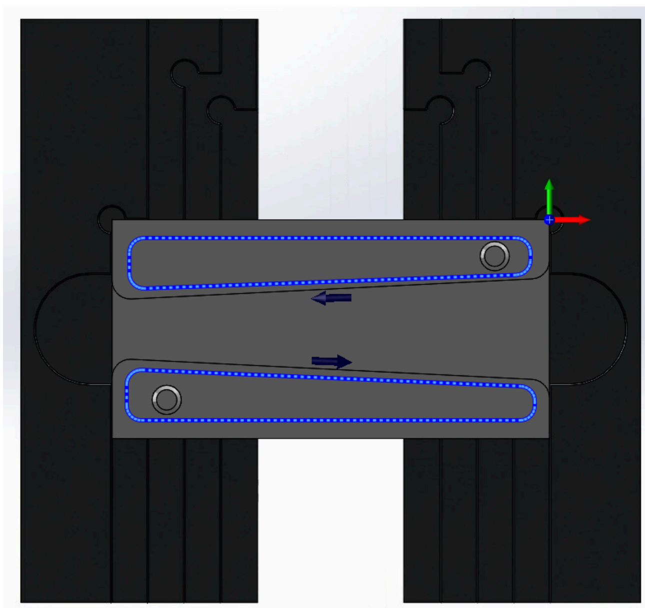


Figure 11: Toolpath Showing Cutting Direction for 2D Contour Operation

- ❖ Heights:
  The same height parameters were used for both the Chamfer and the Slot operations, as shown below.
  - Clearance height: From Retract height: 0.4in
  - Retract height: From Stock Top: 0.2in
  - Feed height: From Top: 0.2in
  - Top: From Stock Top: 0in
  - Bottom: From Contour: 0in

- ❖ Passes:
  - Check Repeat finishing pass for a better finish.

- Check Multiple depths.
- Check Finish only at final depth.
- Uncheck Stock to Leave in general, unless using the 2D contour as a roughing operation.

2.2 3D Milling

2.2.1 Adaptive Clearing

Adaptive clearing is important because it improves machining efficiency and tool life by maintaining a constant tool load throughout the operation. It allows for faster material removal, reduces tool wear, and prevents sudden load spikes that could damage the tool or part.

2.2.1.1 Adaptive Clearing of Central Pocket Region

❖ Tool:

- Used Tool 4 for this process
- Check Shaft & Holder: Pull away(Fig 12)

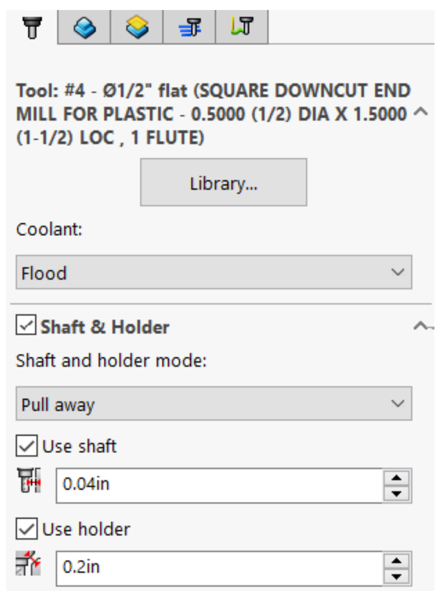


Figure 12 :Tool Setting

❖ Geometry:

- Machining Boundary: None
- Check Rest Machining: From Previous Operation(s)

❖ Heights:

- Clearance height: From Retract height: 0.4in
- Retract height: From Stock Top: 0.2in

- Top: From Stock Top: 0in
- Bottom: From Selection - choosing upper face of center body (fig 13): 0in

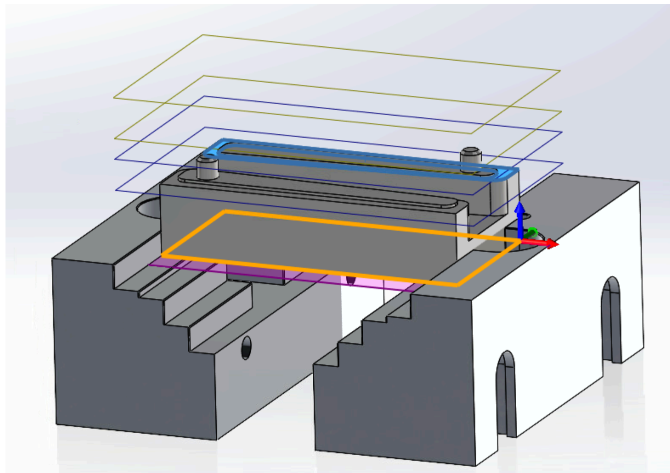


Figure 13 : Stock Setup and Toolpath Heights in Fixture Assembly

❖ Passes:
- Check Flat area detection
- Check Machine cavities.
- Check Use slot clearing.
- Direction: Climb
- Uncheck Stock to Leave

❖ Linking:
- Retraction policy: Minimum Retraction

2.2.1.2 Adaptive Clearing for Central Channel Region

❖ Tool:
- Used Tool 4 for this process
- Check Shaft & Holder: Pull away
- Feed Rate - 35.2in/min (Fig 14)

Feed Rate Calculation:

Feed rate(F_m) = $n * z * F_t$

where,

F_{int} = F_m = Feed rate (in/min) = 52.8 in/min

n = 6000RPM
z = number of teeth = 1
F_t = feed per tooth (in/tooth)

h = ratio of the height = 0.667in

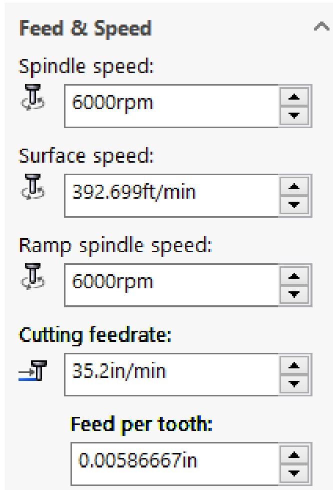


Figure 14: Feed Rate

Calculating the initial feed per tooth:

$F_{t\text{-original}} = F_{m\text{-int}} / n * z = 52.8 / 1 * 6000 = 0.0088$ in/tooth

Calculating the new feed per tooth:

$F_{t\text{-new}} = h * F_{t\text{-original}} = 0.667 * 0.0088 = 0.0058696$ in/tooth

Calculating the new feed rate:

$F_{m\text{-new}} = 6000 * 1 * 0.0058 = 35.2$ in/min

- Geometry:
  - Machining Boundary: None
  - Check Rest Machining: From Previous Operation(s)
- Heights:
  - Clearance height: From Retract height: 0.4in
  - Retract height: From Stock Top: 0.2in
  - Top: From Stock Top: 0in
  - Bottom: From Selection - choosing upper face of center body (fig 13): 0in
- Passes:
  - Check Flat area detection
  - Check Machine cavities.
  - Check Use slot clearing.
  - Direction: Climb
  - Uncheck Stock to Leave

❖ Linking:

- Retraction policy: Minimum Retraction.

3. Cam Process for the Lid:

3.1 Bore Milling

Bore milling is important for accurately enlarging or finishing internal circular cavities, such as holes for bearings or shafts. It ensures precise diameters and smooth surface finishes, which are critical for proper mechanical fit and alignment in assemblies.

❖ Tool: Tool 1 is used for the bore milling.
❖ Geometry: Select the face of the hole.

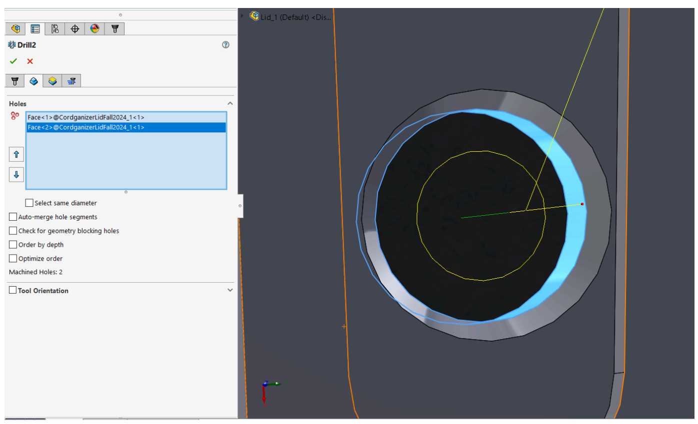


Figure 15: Hole Geometry Selection

❖ Passes:

- Cycle: Bore milling
- Pitch: Should match maximum step down.
- Diameter: 0.205in

3.2 Drilling-rapid out

Drilling - Rapid Out is used to quickly retract the drill after reaching the desired depth, minimizing idle time during tool withdrawal. This improves machining efficiency and reduces overall cycle time without affecting hole quality.

❖ Tool: Tool 5 (1/4” Spot Drill) is used to create the chamfer.

- Geometry:
  - Select the face of the chamfer at the top of the hole.
- Heights:
  - Clearance height: From Retract height: 0.4in
  - Retract height: From Stock Top: 0.2in
  - Feed height: From Top: 0.1in
  - Top: From Hole Top: 0in
  - Bottom: From Hole Bottom: 0in
- Passes: Drilling- rapid out.

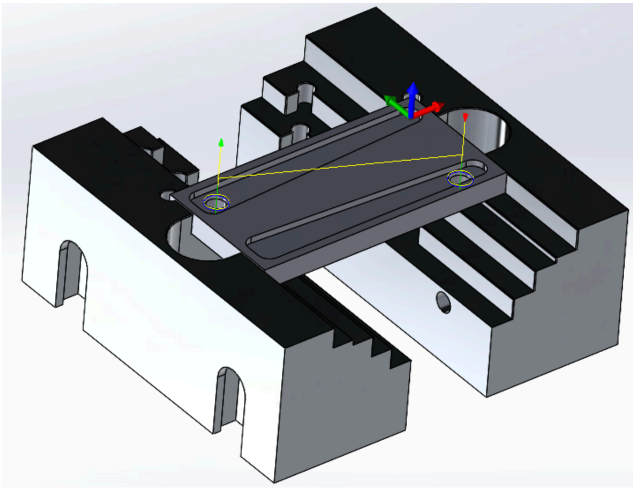


Figure 16 : Drilling Rapid-out

3.3 2D Pocket

2D Pocketing is important for removing material from enclosed areas within a part, such as slots, cavities, or recesses. It allows for efficient roughing and finishing of flat-bottomed features, which are common in mechanical components and assemblies.

Tool:Tool 1 (⅛" flat drill) is used to machine the slot shown in Figure 17.

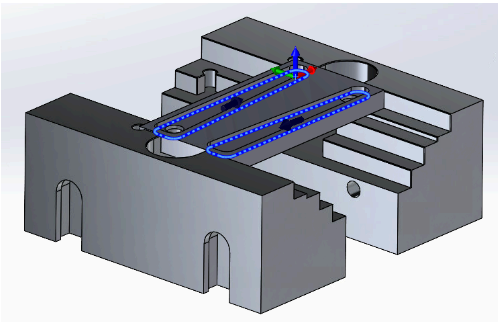


Figure 17 :Central Slot Pocketing Toolpath

Geometry

- Corner edges were manually selected to define the slot boundary.
- Tangent propagation was enabled for smooth transitions along connected edges.
- The reverse button can be used if the toolpath direction needs correction.

❖ Heights:

- Clearance height: From Retract height: 0.4in
- Retract height: From Stock Top: 0.2in
- Feed height: From Top: 0.2in
- Top: From stock Top: 0in
- Bottom: From Contour: 0in

❖ Passes

- Step Down is set to 0.1 in
- Stepover is set to 0.004 in

❖ Linking

- Choose Preserve Rapid Movement
- Ramping - Helix (refer fig 18)

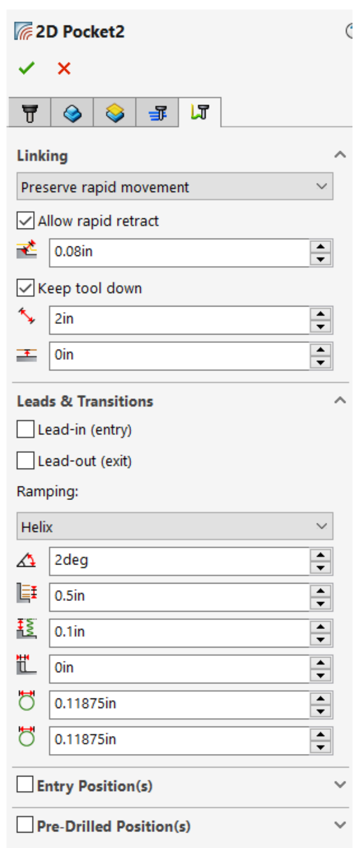


Figure 18: Linking and Ramping Settings

4. Final Step

4.1 Check the Order of Operations

- ❖ The operations are sent to the mill in the same sequence listed in the Job folder.
- ❖ You can change their order by dragging them up or down.

4.2 Run a Stock Simulation

- ❖ Stock Simulation helps visualize how the part will be machined.
- ❖ Ensure that "Check against fixture" is enabled for collision detection..

Figure 19: Check against fixture

- ❖ Ensure that "Stop on clash when animating" is enabled to pause the simulation on collisions.

Figure 20: Stop on clash when animating

- ❖ After the animation finishes, it's useful to click the button that compares the current stock with the final model.

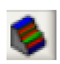


Figure 21: Compares the current stock and mode

5. Post Process

- ❖ Right-click Job in the CAM Manager
- ❖ Click Post Process (All).
- ❖ Under the Post Configuration section,
  - Choose the HAAS (pre-NGC) / haas
  - Change the output folder
- ❖ Under the Program Settings section,
  - Change the Program name or number
  - Write a Program comment of your choosing.
  - Check off Open NC file in editor
- ❖ Click Post.
- ❖ Click Save
- ❖ The G-Code Editor Window will appear.
- ❖ Now the G-code can be loaded onto a USB drive and transferred to the CNC machine.

6. Document Descriptions:

Reference Files for CAM and Design:

The following SolidWorks files were created and used throughout the project for part design and CAM operations:

- CordganizerBodyCAMFall2024_02_New.SLDASM - CAM /assembly file for the Body component
- CordganizerBodyFall2024_New.SLDPRT – Updated Body part design fil
- CordganizerLidFall2024_1.SLDPRT – Updated Lid part design file
- Soft Jaws_Body.SLDPRT – Custom soft jaw designed to securely fixture the body during machining
- Soft Jaws_Lid.SLDPRT – Custom soft jaw designed to hold the lid part during machining
- CordganizerLidFall2024_01_New.SLDASM - CAM /assembly file for the lid component
- 331.nc – G-code generated for the body machining
- O00332.nc – G-code generated for the lid machining