

CUAUV Guide to CAM and CNC

Version 1

Nikki Hart (njh84)
Spring 2023

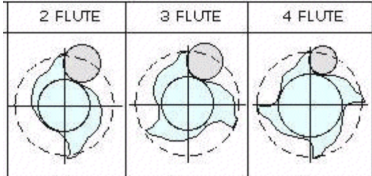
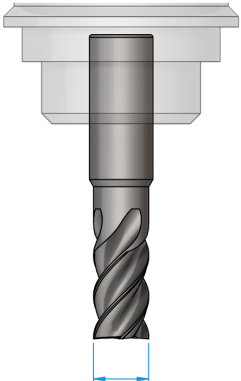
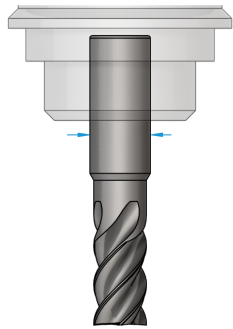

Table of Contents

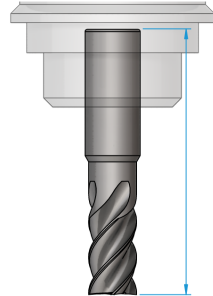
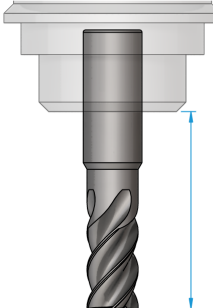
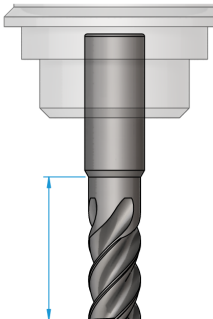
1. Key Terms
2. Design Considerations
 - a. Common Features
 - i. Deep Pockets
 - ii. Pocketing Patterns
 - iii. Internal Fillets
 - iv. External Fillets
 - v. Hole Patterns
 - vi. Large Holes
 - vii. O-ring Grooves
 - b. Non-machinable Features
 - i. Sharp Enclosed Internal Corners
 - ii. Small Feature at Large Depths
 - c. DFM Considerations
 - i. Feature Depth Best Practices
 - ii. Minimizing Material to be Cleared
 - iii. Reducing Number of Setups
 - iv. Reducing Tool Switching
 - v. Optimizing Tool Path
3. Plan to Manufacture
 - a. Setup Design
 - b. Tool Selection
 - i. Common Tools and Tool Inventory
 - ii. Constraints on Tool Selection
 - iii. Objectives for Tool Selection
 - c. Fixturing
4. Detailed Instructions on CAM
 - a. Tool Setup
 - i. Cutter (Tool Dimensions)
 - ii. Cutting Data (Feeds and Speeds)
 - b. Stock
 - c. General Guidelines for Operation Parameters
 - i. Stepover and Stepdown for Peripheral Milling with End Mills
 - ii. Stepover and Stepdown for Facing Tools
 - iii. Boring and Ramping Angles
 - iv. Heights
 - d. Specific Operations
 - i. Facing
 - ii. Adaptive
 - iii. Contour
 - iv. Drilling
 - v. Boring

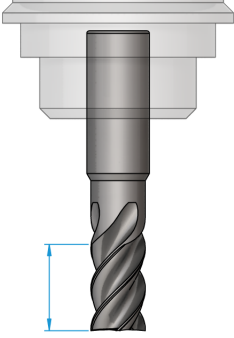

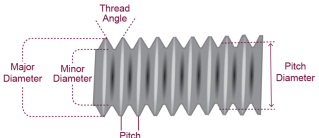
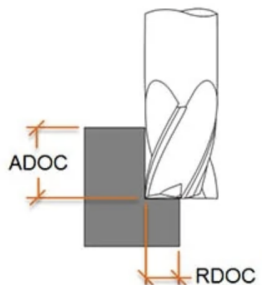
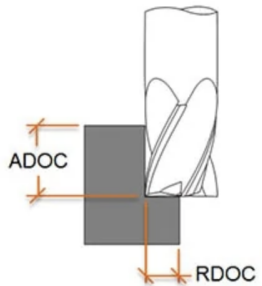
- e. Simulation, Post-Processing, and Setup Sheet
- 5. Mistakes to Avoid
- 6. References

1 Key Terms

The following are key terms related to tooling and manufacturing.

Term	Description and Notes	Image
Flute	The flutes of an end mill, drill, or tap are the spiraling cutting edges. The number of flutes, often used interchangeably with number of teeth, is the number of cutting edges. Thus, “number of flutes” for a face (shell) mill, which does not have flutes, refers to the number of teeth.	 <p>[1]</p>
HSS, Carbide, and Ti Coated	These are all names of tool materials, which can be found on the tool packaging. Tools with a silver shaft and gold flutes are generally Ti coated.	N/A
Tool diameter	The diameter of the cutting portion of a tool.	
Shaft diameter	The diameter of the shaft of a tool. This will usually be	
Reduced neck style	A reduced neck style tool, often used interchangeably with “reach tool,” is a tool where a portion of the tool below the shank/shaft but above the flutes	 <p>Reduced Neck Style</p>

	has a reduced diameter. The reduced diameter prevents the shaft from rubbing against the part, generating heat and worsening finish, during deep-pocket milling.	[2]
Overall length	The overall length of a tool is the length from end to end.	
Length below holder	The length below holder for a tool is the exposed length and varies by setup. The length below holder must be larger than the shoulder length to ensure chips can clear out through the tops of the flutes.	
Shoulder length	The shoulder length is the length of the fluted portion of the tool plus the length of the tapered channels above the flutes that allow for chip clearing.	

<p>Flute length</p>	<p>The flute length of a tool is the length of the tool that has cutting edges, or flutes. This can also be referred to as “length of cut,” as only this portion of the tool can clear material.</p>	
<p>Tip angle</p>	<p>The tip angle of a drill is the angle between the bottom edges of the flutes. Common angles for drills are 118 degrees and 135 degrees, or 120 degrees and 140 degrees for spot drills.</p>	
<p>Thread pitch and threads per inch (TPI)</p>	<p>The pitch of a screw or tap is the distance between threads. This is a common measurement of how fine or coarse threads are for metric screws. For imperial/standard tools, threads per inch (TPI) is more common; this is the reciprocal of the screw’s pitch in inches. TPI is the second number in common thread sizes (#4-40, #6-32, #8-32, etc.).</p>	 <p>[3]</p>
<p>Axial depth of cut (ADOC)</p>	<p>The axial depth of cut is the depth of a single milling pass in the direction of the central axis of the tool. It is normally given either in inches or in multiples of the tool diameter.</p>	 <p>[4]</p>
<p>Radial depth of cut (RDOC)</p>	<p>The radial depth of cut is the width of a single milling pass in the direction of the tool radius or diameter. It is normally given either in inches or in multiples of the tool diameter. The maximum RDOC for any tool is 100% of its diameter.</p>	

2 Design Considerations

A. Common Features

I. Deep Pockets

A deep pocket, or a large amount of material removed from the center of a piece of stock to create an enclosure, is one of the most common features of a CNC part on our team. The large amount of material to be removed makes an enclosure a prime candidate for use of the CNC due to the repetitive and high-scale nature of the passes necessary to clear the material, which would be time-consuming and difficult for a manual machinist.

II. Pocketing Patterns

Complex pocketing patterns are generally present as weight savings for planar parts such as frame plates or racks trusses. Because a large number of pockets tend to be present on these parts, and because these features may use shapes or angles that are hard to produce manually, these features are milled using a CNC.

III. Internal Fillets

Internal fillets are any fillets that round an indented corner. There are two key types of internal fillets: those that appear at the inside corners of an enclosed shape and those that are not bounded by an enclosed shape.

An example of the first case is the fillets on the inside four corners of a rectangular enclosure or the three corners of a triangular pocket feature. Internal fillets are present for any and all enclosed internal corners on CNC or manually milled parts because sharp corners cannot be produced with a cylindrical end mill.

For internal corners that are not bounded by an enclosed shape, a sharp corner may be achieved by reorienting the part setup, but placing a fillet at the corner will reduce manufacturing time if the sharp corner is an unnecessary feature.

IV. External Fillets

External fillets are those that round a protruding corner. Beyond reducing stress concentrations, these tend to be used both to improve aesthetics and to make holding and handling the part safer.

V. Hole Patterns

Both clearance and tapped hole patterns can be produced rapidly on the CNC. Patterns with more than 8 holes or non-circular hole patterns will often be produced on the CNC

both because they appear on the sealing surfaces of large CNC enclosures and because they would be too time-consuming for a manual machinist.

VI. Large Holes

Large holes can be produced easily on a manual mill by using multiple drill bits in succession, but they are often present on CNC parts and can be produced rapidly using a boring operation and a single tool.

VII. O-ring Grooves

Non-circular O-ring grooves should only be produced on a CNC. A small end mill can precisely clear out the groove in steps.

B. Non-machinable Features

I. Sharp Enclosed Internal Corners

As mentioned in [Section 2.A.III](#), sharp internal corners cannot be produced in the case that they are enclosed¹, and sharp internal corners are generally not necessary even when the corners are not enclosed.

II. Small Features at Large Depths

While we do own reach tools, or tools designed to produce features at large depths, these tools are strengthened either by a relatively large diameter or by the fact that they are only used axially (for example, a long drill bit). While “small” and “large” here are subjective, in general, features at a depth of five times the tool diameter will create poor finish, and features at a depth of ten times the tool diameter are not safe to produce with the tools and machines we have available. This does not apply to pure drilling operations with a drill bit. Anecdotal data from our team on safe depths is enumerated in [Section 2.C.I](#) below.

C. DFM Considerations

I. Feature Depth Best Practices

As mentioned in 2.B.II above, a good rule of thumb for determining safe depths for features of a certain size is that a feature produced using radial passes of a tool of diameter D may cause poor finish if the tool’s length below the tool holder is greater than $5D$ and should not be produced in Emerson if the tool’s length below the holder is at or above $10D$. The following table lists examples² of depths at which we have been able to produce good finish for common tool sizes and associated internal fillet radii.

¹ There exist ways to approximate, achieve, or replicate sharp internal corners, but they are not necessary for the parts we make on CUAUV. If you would like to learn about specialized machining techniques relevant to internal corners, you can read more from Reference [5].

² This table was produced based both on experience (as documented in the CNC Diary) and on the lengths of tools we have for each tool diameter. This table approximately aligns with the $5D$ depth rule.

Max Depth of Feature	Min End Mill Size	Min Feature Internal Radius
Up to .3"	.0625"	.03125"
.3"-.875"	.125"	.0625"
.875"-1.5"	.25"	.125"
1.5"-2.5"	.5"	.25"
2.5"-3.4"	.75"	.375"

As an example, based on the table above, the internal corners of an enclosure that has an internal depth of 3" should not be rounded to a fillet radius lower than .375".

II. Minimizing Material to be Cleared

Our enclosures tend to be produced from rectangular stock and require a large amount of material to be cleared from the center. While this is unavoidable, the material to be cleared can be reduced when designing the enclosure by ensuring that the internal dimensions of the enclosure are as small as possible to fit any necessary components. Additionally, because CNC milling is a subtractive manufacturing process, any protrusions from an enclosure (for example, a flange or the protruding tube present on the 2023 ZED Camera enclosure) increase the stock box dimensions and therefore the material to be cleared. When possible, avoid adding thin or narrow but long extrusions to enclosures.

III. Reducing Number of Setups

Rotating or re-gripping a part in the vise can be necessary to allow access to certain faces, machine the entire piece of stock, or expose a certain datum for zeroing. While additional setups sometimes cannot be avoided to produce necessary features, the number and duration of setups can be reduced by using simple features that can be produced in pre-machining.

IV. Reducing Tool Switching

When small or specialized features are to be produced on the CNC, tool switching becomes necessary. An increased number of tools means increased setup time and increased machining time, especially when the additional tools rerun previous passes to achieve a certain feature size or shape.

One particularly time-consuming example is a small fillet radius on the four internal corners of a rectangular enclosure. While a larger version of this cosmetic feature would require only a single large end mill to bore out the enclosure, creating a small fillet requires switching to a smaller tool and re-milling the perimeter at a reduced feed rate and stepdown.

Another example is internal bottom edge fillets; not only are these features often unnecessary, but they require that material is left behind and then milled using a ball end mill, which is more inefficient at clearing material [6], [7].

Increasing the size of features when possible and removing cosmetic features can increase efficiency in CNC machining. The trade-offs associated with this aspect of DFM should be considered when making design decisions driven by aesthetics.

V. Optimizing Tool Path

When designing specialized features, it is best practice to prevent tools from having to change directions sharply. Doing this can cause very poor finish. This can easily be prevented by increasing fillet diameters to be larger than the tool used to cut them, allowing the tool to take a rounded path instead of taking a sharp 90 degree turn, as well as by never creating features that have acute angles. Use simplified and enlarged geometry where possible to create the best finish.

3 Plan to Manufacture

A. Setup Design

When making a plan to manufacture, it is important to consider how each setup affects the next. Certain operations can expose or remove important datums, while others can create surfaces by which a part should not be gripped. For standard enclosures on our team, the bottom of the enclosure will generally be machined first, leaving five flat faces by which the enclosure can be gripped or placed on parallels. Then, features such as scallops on the flange that would make gripping or zeroing the part more difficult, as well as features such as the sealing face and O-ring groove, which are never to be gripped in a vise, can be machined in the final setup.

It is essential that as many operations as possible are completed in a single setup. This reduces the risk of misaligned features inherent to the variability of multiple setups. In particular, because sealing surfaces have the most critical tolerances, facing and O-ring groove milling operations must be done in a single setup. Usually, two to three setups are sufficient for most parts, depending on whether any holes or other features appear on the sides of an enclosure.

B. Tool Selection

I. Common Tools and Tool Inventory

End mills are the most versatile tools; they can be used for facing, milling contours, clearing large amounts of material for both internal and external features, and boring large holes that would normally require switching between multiple drill sizes. Small end mills can also be used to create O-ring grooves.

Reach or reduced neck end mills are often a better option for deep pocketing or for milling tall external features. While a regular end mill can be used to clear the majority of the material due to the larger length of cut, reach end mills prevent rubbing already-machined faces with the shaft of a tool when milling the deepest features.

While end mills can be used for facing, facing tools are generally the best selection for creating faces on parts due to the amount of material they can clear quickly and the better finish afforded by a large tool.

Drills are used to create holes that are small in diameter (requiring only one drill tool) or deep (not reachable by an end mill). End mills should only be used to bore wide and short holes, as they are generally less efficient than drills but can reduce tool switching.

Spot drills and center drills are used to create a lead-in for a drill. Spot drills are generally a better choice, as the large tip angles allow the tip of the drill to engage, preventing deflection. However, center drills still guide drills into place.

Once an updated CUAUV tool library is created in Fusion, it will be linked here.

II. Constraints on Tool Selection

It is imperative that a sufficiently long tool is selected for each operation. If the flute length of a tool is not long enough or a tapered reach tool is not selected, the shank, or the cylindrical part of the tool above the flutes, can rub against the part. This creates poor finish and creates dangerously high frictional forces that can generate high heat, break a tool, or even dislodge a part from the vise. In the worst case, an insufficiently long tool length below the tool holder can result in the tool holder crashing into the part, which would cause irreparable damage to the part and likely the machine.

When deciding on a tool length, ensure not only that the flute length is greater than the depth or height of the part, but that the total length of the tool that can be exposed below the tool holder is greater than the height of the stock. Depending on the order of operations chosen, it is possible that the tool holder could crash into material that has not yet been cleared, not just features of the designed part.

The diameter of a tool being used to mill features must also be the same size or smaller than any fillets present. Attempting to mill small features with a large tool can cause a large amount of material to be cleared—often more than the ideal stepover for a tool—or can result in chatter and very poor finish when forcing the tool into small corners.

III. Objectives for Tool Selection

Additional considerations when selecting tools include operation efficiency and reducing vibrations and chatter. Larger tools will be able to clear material more quickly due to the larger allowable stepover and stepdown. The largest possible tool diameter and smallest

possible tool length should be selected for a given operation, as longer tools increase chatter, while wider tools are stronger and sturdier against vibrations.

Facing tools will create better finish than end mills and are the best choice for the bottom and top faces of a part.

C. Fixturing

On our team, fixtures for parts are generally only used for plate parts where the perimeter of the stock will be cleared away. To enable completing a plate in a single setup (assuming through or coplanar features), holes present on the part can be pre-machined on both the stock and a fixture plate. The stock can then be secured to the fixture plate using fasteners, and the fixture can be gripped in the vise to expose more surfaces of the stock to the machine. As many holes as possible should be captured to reduce vibrations.

For other parts, fixturing generally can and should be avoided. Select setups such that grippable surfaces are not machined away on the first setup and such that as many faces to be machined as possible are exposed. Fixtures are less stable than gripping a part in a vise and can be subject to far more vibrations, especially for larger parts where surfaces are machined at a great distance from the fixture and fasteners. Fasteners can even release themselves due to the vibrations the part undergoes during machining. Only use a fixture when it is not possible to access necessary surfaces without it.

4 Detailed Instructions on CAM

A. Tool Setup

When setting up tools, first check whether the tool you plan to use is available in the CUAUV Tool Library. If not, or if you would like to learn more about calculating tool parameters, detailed instructions on setting up tools are included below.

I. Cutter (Tool Dimensions)

To create a new tool, open the Tool Library and press the plus symbol, then select a tool type. Most tools we use will appear under “Milling” (ball end mills, flat end mills, and face mills) or “Hole Making” (drill, center drill, spot drill, tap right hand). Once a tool type has been selected, navigate to the “Cutter” tab.

Next, enter tool parameters into the boxes. Note that in Fusion 360, clicking on the geometry boxes will highlight the desired input graphically on the tool model, allowing you to see what each parameter looks like on the specific tool. Additionally, hovering over these boxes will bring up a brief description of the parameter. Descriptions of some common parameters can also be found in [Section 1](#) of this document.

Both milling and hole making tools will generally prompt you for the number of flutes, material, tool diameter, shaft diameter, overall length, length below holder, shoulder length, and flute length. Additional parameters may include tip angle for drills and thread pitch (recall that this is the reciprocal of threads per inch, which is more common for imperial/standard tools) for taps. We generally leave more specific parameters, such as a face mill's corner radius and taper angle, to the Fusion defaults.

Most of these parameters will be available on the tool packaging. If this is no longer available, many of these parameters can be measured with calipers. To measure the tool diameter of a tool with an odd number of flutes or otherwise complicated geometry, place the tool in calipers and spin it until the calipers expand to the full tool diameter. The most common drill tip angle is 118 degrees.

II. Cutting Data (Feeds and Speeds)

Feeds and speeds depend on the tool material and size, material to be cut, and even the type of operation and cutting conditions. Rules of thumb for the feeds and speeds for end mills and facing tools (peripheral milling and facing), drills, and taps cutting aluminum with coolant are described below.

a. End Mills and Facing Tools

The range of surface speeds for end mills and facing tools cutting aluminum is 600-1000 FPM. It is generally fine to use 1000 FPM; you can enter this into the "Surface speed" box and the spindle speed and ramp spindle speed will automatically be updated using the following equation:

$$RPM = FPM * \frac{12 \text{ in/ft}}{\text{Tool Diameter} * \pi}$$

Never exceed 10,000 RPM, even if the surface speed needs to be adjusted to below 600 FPM, as 10,000 RPM is the limit for the machines in Emerson. Smaller tools will therefore have smaller FPM. Additionally, you may want to manually reduce the ramp spindle speed by 30% (though we often forget to do this).

Feedrates should be found by entering the feed per tooth (in inches per tooth), as this parameter is tabulated for certain tool diameters, tool materials, and stock materials. In general, the following FPTs can be used, but FPTs should be reduced for reach tools [8]:

Tool Diameter	Feed per Tooth (Inches per Tooth)
1/16"	0.001
3/16"	0.002

1/4"	0.002
3/8"	0.003
1/2"	0.004
5/8"	0.005
3/4"	0.006

A feed per tooth of 0.006-0.007 inches can be used for face mills. Fusion will then calculate the cutting feedrate using the following formula:

$$\text{Feedrate} = \text{RPM} * \# \text{Flutes} * \text{FPT}$$

The lead-in feedrate and lead-out feedrate should be manually reduced to $\frac{2}{3}$ of the cutting feedrate, and the ramp feedrate and plunge feedrate (under "Vertical Feedrates") should both be manually set to 20-25% of the cutting feedrate.

b. Drills

The two most common drills we use are spot and center drills (1/8"-3/16") and #43 drills to create tapped #4-40 holes. For spot and center drills, use a speed of 5000 RPM and a cutting feedrate of 40 IPM. Set the ramp and vertical (plunge and retract) feedrates to 13 IPM. For #43 drills, set the speed to 10000 RPM and the vertical (plunge and retract) feedrates to 33 IPM.

For other drills, you can use a surface speed of 150 FPM (the range for aluminum is 150-275 FPM). Fusion will then calculate the spindle speed using the same speed calculation shown in the end mill section. Then, select a plunge/retract feed per revolution from the following table and have Fusion calculate the plunge and retract feedrates [9].

Tool Diameter	Feed (Inches per Revolution)
1/16"	0.001
1/8"	0.003
3/16"	0.006
1/4"	0.007
1/2"	0.012
3/4"	0.016
1"	0.019

>1"	0.025
-----	-------

c. Taps

For taps, the spindle speed should be between 150 RPM and 250 RPM (250 RPM is standard). The surface speed will be automatically calculated, and the feedrates are preset by the tapping operation in Fusion.

d. Engraving

B. Stock

When creating a new setup in Fusion, you will be prompted to set up the stock box as well as the Work Coordinate System (WCS), or origin.

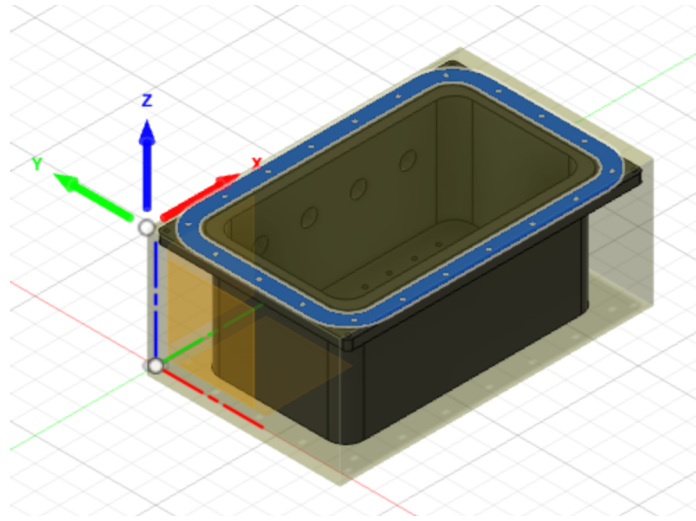
To set up stock in Fusion, make sure the stock has been pre-machined as desired, then measure the dimensions of the stock. The default stock mode, fixed size box, has a few options for the positioning of the model within the stock box. Leaving the model centered within the stock box can be useful when first starting a part, as it will leave stock on the bottom face (not exposed in the first setup) that can be faced on the CNC in the second setup, resulting in a nice finish. However, selecting this option does not expose out the distance between the top face to be milled and the stock box. This means that when the part is flipped over, the remaining stock to be machined will either have to be measured and the part model aligned with one face of the stock, or the remaining stock to be machined will have to be calculated based on the stock and part dimensions.

Once parts of the stock have been machined, the "From preceding setup" stock mode can be used to carry over geometry from the previous setup. This is useful if you have filleted external corners, causing a corner origin to no longer exist on the model.

You also have the option of using the "Relative size box" stock mode, then the "Add stock to all sides" stock offset mode. You can then specify the offset between the stock and every outer face of the model. Faces that have been machined can be set to zero, while faces that have not been machined can be aligned precisely with the existing part features. If the "Center" model position option was used in the first setup, all of the remaining stock in the axial direction can be added as an offset to the non-machined face, and one half of the non-machined length and width can be added to each of the lateral faces.

It is imperative that you correctly set your WCS for each setup. On the first setup, ensure that the "Origin" option is set to stock box point. It is most common to select the back left corner of the stock to be the origin, with the X axis pointing to the right, the Y axis pointing towards what will be the back of the machine, and the Z axis pointing up. This can be achieved by selecting the "Orientation" option to be "Select Z axis/plane & X axis," then selecting the top face of the

model for the “Z Axis” option and an edge along the width of the part for the “X Axis” option. The “Flip Axis” checkboxes can be adjusted as necessary to achieve the desired orientation.



It is very important to pay attention to the WCS position and orientation for subsequent setups. Consider which faces of the part will not only remain after the first setup but be reachable by the probe. If an extrusion or flange is now present at the top of the new setup, for example, the probe will not be able to reach past it to other lateral faces. This means that even if the flange still has rough, un-machined edges, the WCS must appear on one of those un-machined points in the new setup.

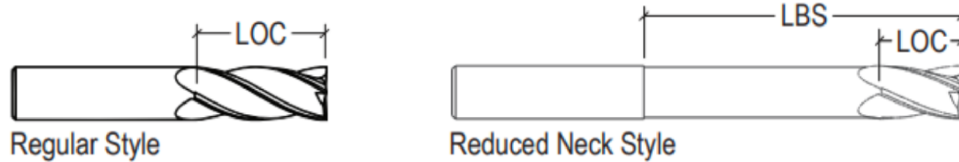
Depending on what faces are reachable by the probe and whether they were machined in the first setup, you will either select “Stock box point” (un-machined face) or “Selected point” (machined face) under “Origin.” Recall that if you are using a box stock option but have filleted external corners, the desired origin may no longer exist on either the model or the stock box. This would require changing the stock mode to something that better represents the existing geometry.

C. General Guidelines for Operation Parameters

I. **Stepover and Stepdown for Peripheral Milling with End Mills**

As with feeds and speeds, there are optimal stepovers and stepdowns for certain tools and material combinations. As seen in the table below [2], the maximum RDOC used for roughing passes is 40% of the tool diameter, while the optimal range for ADOC is 1D-2D for short/non-reach tools and up to the maximum length of cut (flute length) for reach tools, which is typically about 1.5D. Many of our tools are long enough to be considered reach tools but are not reduced-neck style and therefore have long lengths of cut; this does not mean it is safe to use the entire length of cut. Thus, consider the maximum ADOC to be 1.5D for all end mills.

Milling Process	Style	ADOC	RDOC
Slot (Full Slotting)	Non-Reached	75%-125% Diameter	100% Diameter
	Reached	Up to Max LOC	100% Diameter
Rgh (Traditional Roughing)	Non-Reached	125%-200% Diameter	30%-40% Diameter
	Reached	Up to Max LOC	30%-40% Diameter
Fin (Finishing)	N/A	Up to Max LOC	4%-6% Diameter



In practice, we do not use these maximums in conjunction with each other. When using the ADOC max, the RDOC should be reduced, and vice versa. In fact, our team tends to use very conservative values of an ADOC of .8D or less and an RDOC of .4D. In the future, increasing the ADOC (and reducing the RDOC, if necessary) should be considered to increase tool life.

The following table includes starting points for ADOC/RDOC combinations for roughing passes with end mills of various sizes, based on our team's experience. This table should be updated with less conservative values if they are used successfully in the future.

Tool Diameter	ADOC (Stepdown)	RDOC (Stepover/Optimal Load)
1/16"	0.01 in	0.025 in
1/8"	0.05 in	0.05 in
1/4"	0.2 in	0.1 in
3/8"	0.25 in	0.15 in
1/2"	0.3 in	0.2 in
3/4"	0.3 in	0.1 in

For finishing passes, the RDOC should be reduced to no more than 6% of the tool diameter, and the ADOC can be increased significantly.

Up until this point, only peripheral milling operations, or operations that do not use the entire tool diameter, have been discussed. Slotting operations are not commonly used,

as Fusion uses boring operations with a reduced feedrate to allow deep pockets to be cut using only peripheral milling.

II. Stepmover and Stepdown for Facing Tools

For facing tools, a stepover of about 70%-90% of the tool diameter should be used with between a 0.01" (finishing pass) and 0.05" (MAE 5250 C-block) depth of cut. For manual machining, we use a 0.025" depth of cut. For large parts, it is safer to use a lower depth of cut, as cuts in the direction parallel to the vise may be subject to more vibration and chatter compared to small parts where the entire part is gripped by the vise. While the entire tool diameter could be utilized, using a reduced stepover ensures no material is left behind.

III. Boring and Ramping Angles

For boring operations, adaptive clearing operations that utilize boring, and ramping operations, a ramping angle of 2 degrees should be used.

IV. Heights

The default heights set in Fusion tend to work well, but check them as you set up your operations to make sure they are all safe heights.

Starting from the bottom of the list, the bottom height, which is the lowest your operation will reach, should be set to the final geometry of the feature you are machining. For facing operations, this is the face to machine; for contour and adaptive operations, this is the bottom of the feature, and for drilling or boring operations, this is the bottom of the hole (or a little past the bottom of the hole for through holes). In general, the offset (usually 0) can be specified from the "Selection" option in the dropdown and you can select the feature to be machined, though more specific options can be used for specific operations. (For example, facing operations might use "Model Top," and boring or drilling operations might use "Hole bottom." These will be specified in more detail in [Section 4.D.](#))

The top height is the height at which the operation will begin. This should be carefully set to the extremity of the material the tool may encounter during this operation. For example, facing operations should always use "Stock top," and subsequent operations should use "Selection" to select the last machined face. If there is any question of whether there will be extra material, add a positive offset to ensure the correct stepdowns are maintained.

Feed height, retract height, and clearance height should all be set to some positive offset from the stock top so that when linking passes or retracting the tool does not hit the part. Clearance height is generally the highest.

D. Specific Operations

I. Facing

To set up a facing operation, select “Face” from the 2D menu. Change the parameters as described below, organized by tab.

a. Tool

Select a tool from the Fusion file library or from the CUAUV tool library. This will import all the “Feed & Speed” parameters; you should not have to change these.

b. Geometry

The top stock face in the setup should automatically be selected. This will ensure the entire stock is machined, not just the model’s top face, which is important if the model top is an extrusion or the opposite face has a flange.

c. Heights

Follow the guidelines in Section [4.C.IV](#) to set these heights. Since facing operations normally only happen on the top and bottom surfaces, the bottom height can be set using “Model top,” or you can also use “Selection” and select the face. “Offset” should be set to 0. The top height should be set to “Stock top” with an offset of 0. If you anticipate the top surface of your stock being rough or uneven, add some positive offset to the top height to be safe.

d. Passes

This is where your stepdown and stepover will be set as described in Section [4.C.II](#). Put your calculated stepover value into “Stepover,” then select the “Multiple Depths” checkbox and add your roughing stepdown to “Maximum Stepdown.” Check “Both Sides” if you want to allow the tool to go in both directions across the part (to prevent it from retracting and starting again at its original position). Check “Finishing Step” and enter a finishing stepdown for the final pass. If you want, you can even slow the finishing feedrate under “Finish Feedrate” for an even nicer finish. Do not select “Stock to Leave” unless you plan to remove this stock with another operation later.

e. Linking

Fusion defaults should be fine here, with the exception of the first dropdown, which should be changed to “Always use high feedrate.” This will import the tool’s high feedrate. “Allow rapid retract,” “Keep tool down,” “Lead-In,” and “Lead-Out” should all be selected.

II. Adaptive

To set up an adaptive operation, select “2D Adaptive Clearing” from the 2D menu. Change the parameters as described below, organized by tab.

a. Tool

Select a tool from the Fusion file library or from the CUAUV tool library.

b. Geometry

Under “Geometry,” select the bottom face of the region to be cleared. If this is the first adaptive operation on a certain geometry, check the “Stock Contours” box to ensure the operation “sees” the extra stock material and preserves the correct stepover. If this is an adaptive operation that follows another and is being used to more accurately machine small geometry, you may want to select the “Rest Machining” option, which will only cut geometries a larger tool could not reach (see image below). If this option is selected, specify the “Tool Diameter” to the diameter of the last tool used and the “Corner Radius” to the “Minimum Cutting Radius” of the last operation. To finish an entire contour as well as refine small geometry (as in, radial but not axial stock remains), you might instead select a contour operation.

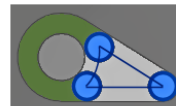
Rest Machining

Limits the operation to only remove material that a previous tool or operation could not remove.

Rest stands for **RE**maining **ST**ock.

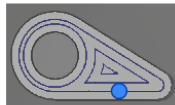


Area to machine.
Pocket shown in green.

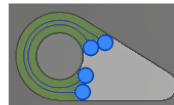


Previous Operation.
Not all stock is removed.

With a smaller tool you can machine the entire pocket, or use Rest Machining to clear the remaining stock.



Use Rest Machining OFF.
All areas are machined.



Use Rest Machining ON.
Previously uncut areas are machined.

Requires additional information about the tool previously used to cut the boundary.

c. Heights

Follow the guidelines in Section [4.C.IV](#) to set these heights. The bottom height should be automatically set to “Selected contour(s),” but if not, use “Selection” to select the face to be machined. “Offset” should be set to 0. If you completed a facing operation before this operation, you can set the top height to the model top with an offset of 0 (or an offset of the amount of stock left behind).

d. Passes

This is where your stepdown and stepover will be set as described in Section [4.C.I](#). Put your calculated stepover value into “Optimal Load.” If the tool being used to clear the material is larger than the features in the geometry, make sure to set the “Minimum Cutting Radius” to a non-zero value; it should default to 10% of the tool diameter. Select the “Multiple Depths” checkbox and add your

roughing stepdown to “Maximum Roughing Stepdown.” If you plan to use a different (usually smaller) tool for the final passes, check the “Stock to Leave” box and specify the radial (X and Y) and axial (Z) values. If the next operation will be a contour operation, you may want to specify the axial stock to leave as zero. Leaving axial stock will greatly increase the number of passes required to finish the radial surfaces.

e. Linking

Fusion defaults should be fine here, with the exception of the first dropdown, which should be changed to “Always use high feedrate.” This will import the tool’s high feedrate. If the “Ramp” settings are not already specified to a “Ramp Type” of “Helix,” a “Ramping Angle” of 2 degrees, and a “Ramp Taper Angle” of 1 degree, then update those settings.

III. Contour

To set up a contour operation, select “2D Contour” from the 2D menu. Change the parameters as described below, organized by tab. Contour operations will have many of the same settings as adaptive operations, but they are generally used to finish faces or achieve small geometry and will therefore specify a single pass by default. In general, they should not be used for clearing large amounts of material.

a. Tool

Select a tool from the Fusion file library or from the CUAUV tool library.

b. Geometry

Under “Geometry,” select the contour (curvature/chain, not the face) that the operation will finish. As mentioned earlier, this operation should generally be used on geometries that have already been partially machined. However, if this is the first operation on a certain geometry, then select the bottom face of the region to be machined and check the “Stock Contours” box to ensure the operation preserves the correct stepover. If this is being used only to more accurately machine small geometry (not to finish entire faces), instead of selecting “Stock Contours,” select the “Rest Machining” option, then specify the “Tool Diameter” to the diameter of the last tool used and the “Corner Radius” to the “Minimum Cutting Radius” of the last operation. This can save time by making the toolpath shorter, but it can also cause the path to miss material that was left behind. See Section [4.D.II](#) under “Geometry” for more information.

c. Heights

Follow the guidelines in Section [4.C.IV](#) to set these heights. The bottom height should be automatically set to “Selected contour(s),” but if not, use “Selection” to select the face to be machined. “Offset” should be set to 0. If you completed a facing operation before this operation, you can set the top height to the model top with an offset of 0 (or an offset of the amount of stock left behind).

d. Passes

Contour operations will be set to a single pass by default and, unlike adaptive operations, will allow you to specify finishing parameters directly in the operation. Follow the guidelines in Section [4.C.I](#) for roughing and finishing stepovers and stepdowns.

Ideally, contour operations will only be used for the final passes on a geometry; therefore, you should be using your final tool size (one that fits the model geometry) and specifying only finishing passes. If this is true, you can leave the “Minimum Cutting Radius” at the default of 0, then choose to select or not select “Multiple Finishing Passes.” If you do select “Multiple Finishing Passes,” specify the number of finishing passes and the calculated finishing stepover. If you want to debur the part or ensure no material is left behind due to possible tool deflection, you can also check the “Repeat Finishing Pass” box.

If you have more material left over than what can be cleared with just a couple finishing passes, you may want to use an adaptive operation instead. If you choose to go with the contour operation to achieve a certain geometry, enter your calculated roughing stepover into “Maximum Stepover” and specify the number of stepovers by calculating how many stepovers it would take to clear the amount of material left over using the specified stepover size. You might increase this number just to be safe, as this is the maximum, not the minimum, number of stepovers the operation will take.

Unless the height of the geometry to be finished is less than two times the diameter of the tool, you should check the box for “Multiple Depths.” If you are unsure, still check the box and enter the 2D value into “Maximum Roughing Stepdown”; if only one stepdown is necessary for the geometry, only one will be used. If you increase the number of finishing stepdowns from the default of 0, you can apply a finishing stepdown in a similar way that you would apply a finishing stepover to finish both the radial and the axial faces. “Finish Only at final depth” can be used to specify that the finishing pass on the radial faces should only occur at the final depth (ensuring that there are no stepdowns and therefore no lines between finishing passes) and should only be used when the finishing stepover is very small to prevent over-engaging the tool.

As with adaptive operations, you can also select the “Stock to Leave” option if you have additional operations following this one.

e. Linking

Fusion defaults should be fine here, with the exception of the first dropdown, which should be changed to “Always use high feedrate.” This will import the tool’s

high feedrate. You should not need to use the “Ramp” settings, as contour operations should not be used for clearing large amounts of material.

IV. Drilling and Tapping

To set up a drilling or tapping operation, select “Drill” from the Drilling dropdown.

- a. Tool
Select a tool from the Fusion file library or from the CUAUV tool library.
- b. Geometry
Under “Geometry,” select the internal face of each hole to drill or tap using the same tool.
- c. Heights
Follow the guidelines in Section [4.C.IV](#) to set these heights. The bottom height should be set to “Hole Bottom” with an offset of 0 (or a small negative number to deburr a through hole). The top height should be the hole top if the top of the hole is on a surface that has already been machined, or the stock top if the face has not yet been machined.
- d. Cycle
Select the type of drilling operation. The most common drilling modes for us will be “Drilling – rapid out,” which we use for both spot drilling and for relatively shallow holes (such as the #4-40 tapped holes that appear on most of our flanges), as well as “Tapping.” For deeper holes, use “Deep drilling – full retract” and specify pecking parameters to allow chips to be cleared. A good starting point for pecking depth is 0.05 in.

V. Boring

To set up a boring operation, select “Bore” from the 2D dropdown.

- a. Tool
Select a tool from the Fusion file library or from the CUAUV tool library.
- b. Geometry
Under “Geometry,” select the internal face of the hole to bore.
- c. Heights
Follow the guidelines in Section [4.C.IV](#) to set these heights. For hole drilling and tapping operations, the bottom height should be set to “Hole Bottom” with an offset of 0 (or a small negative number to deburr a through hole). When using a spot or center drill, specify the bottom height either with a negative offset from the hole top (starting point for #4-40: 0.015 in) or with “To chamfer diameter” (starting point for #4-40: 0.04 in). The top height should be the hole top if the top of the

hole is on a surface that has already been machined, or the stock top if the face has not yet been machined.

d. Passes


Select “Use Ramp Angle” and enter a ramp angle of 2 degrees. If the bore size is near or above two times the diameter of the tool, a single pass may leave material in the center of the bore. To prevent this, select the “Multiple Passes” option. “Stepover” can be set to the same stepover value or smaller than what you would select for an adaptive clearing operation. Enter the number of stepovers. If the number of stepovers is greater than what can be achieved with as large of a stepover size as was specified, some of the passes will be discarded; decrease the number of stepovers or the stepover size until the warning disappears. As with other operations, you can specify whether to include a finishing pass and the finishing stepover as well as whether any radial stock should be left behind.

e. Linking

Change the first dropdown to “Always use high feedrate.”

E. Simulation, Post-Processing, and Setup Sheet

To post-process the CAM, or to generate the G-code, go to Actions→Post Process. Here, you should change three of the settings: “Post,” “File name,” and “Output folder.” For the file name and output folder, choose a meaningful name and location so you can find the file. For the “Post” setting, make sure to select the Emerson post-processor every time, or you will have issues when you go to run the CNC. Finally, click “Post” at the bottom right of the window.

If you have not already imported the Emerson post-processor, first download it from the CUAUV→Mechanical→CNC Google Drive folder ([or click here](#)). In the post-processor window you already have open in Fusion, click the folder icon next to the “Post” dropdown, click the import button (), and select the downloaded file.

A setup sheet will give you useful information for setting up and running your program, from stock and part sizes to tool numbers and lengths as well as a list of operations. (Here, “length” refers to the length below holder, which helps ensure you leave enough of the tool out when setting up your tools.) You can pull this up on your computer in Emerson as a guide. To generate a setup sheet, go to Actions→Setup Sheet and select the destination folder (the filename will be automatically generated based on the Fusion file name and version number).

5 Mistakes to Avoid

As mistakes are encountered, notes on how to identify and avoid these problems will be added here.

I. **Not tightening the tool holder enough**

The large loads and vibrations a tool experiences during a CNC operation can result in a tool slipping or vibrating out of the holder if it is not gripped properly. One way to identify this is if the tool or tool path appears to be ramping when it should be milling at a constant height. If you do not identify this visually, another way to identify this is hearing the passes progressively get louder. If you encounter this issue, immediately stop the machine and re-tighten all your tools. Do not forget to re-measure the tools using the CNC once they have been set up again.

II. **Manually adjusting the speed instead of the feed on the CNC**

It is common and even encouraged to use the handle on the machine to adjust the feedrate as necessary. However, it is easy to accidentally select “speed” instead of “feed” and cause the tool to stop spinning when you mean to stop its lateral motion. This can cause a non-rotating tool to crash into your part at full speed. Take care to select the correct handle jog option, and if you are adjusting the speed, never reduce the speed to at or near zero.

6 References

Data that does not have an explicitly-named reference is based either on personal experience and the CUAUV Spring 2023 CNC Diary or metrics used in existing CAMs (such as past team CAMs or the MAE 5250 C-block CAM project). Unlabeled images are from Fusion 360.

1. Tool flutes: <https://www.cs.cmu.edu/~rapidproto/students.03/zdb/project2/CNCflutes.htm>
2. Feeds and speeds: https://ecomm.productivity.com/customer/docs/skudocs/SF_H35AL-3.pdf
3. Pitch diagram: https://www.aatprod.com/hrf_faq/what-is-the-pitch-diameter/
4. RDOC and ADOC relationship: <https://www.harveyperformance.com/in-the-loupe/depth-of-cut/>
5. Ways to approximate or achieve sharp internal corners: <https://makeitfrommetal.com/machining-square-inside-corners-the-nightmare/>
6. When to use or not use fillets: <https://www.fictiv.com/articles/fillets-when-to-use-em-when-to-lose-em>
7. Flat and ball nose end mills: <https://www.autodesk.com/products/fusion-360/blog/machining-fundamentals-introduction-milling-tools/#:~:text=Because%20of%20this%2C%20Flat%20End.roughing%20passes%20have%20left%20behind>
8. Cornell Mars Rover DOC Charts

9. Machining Data Handbook, page 3-33

HARDNESS BHN	CONDITION	SPEED fpm	FEED - Inches Per Revolution								HSS TOOL MATERIAL except as noted
			NOMINAL HOLE DIAMETER - Inches								
			1/16	1/8	1/4	1/2	3/4	1	1-1/2	2	
30 to 80 500 kg	Cold Drawn	140 275	.001 -	.003	.007	.012	.016	.019	.025	.030	M10 M7 M1
75 to 150 500 kg	Solution Treated and Aged	140 275	.001 -	.003	.007	.012	.016	.019	.025	.030	M10 M7 M1

10. Fusion Heights Explained (note that this swaps “clearance height” and “retract height”!):

<https://www.nyccnc.com/fusion-360-cam-heights-explained/>

11. CUAUV CNC Diary:

https://docs.google.com/spreadsheets/d/14_Pq7-YefiAi9pMy_jhjYBsgcAatPTl3BqaQV5OGisk/edit?usp=sharing