

MESHCAM PRO

CNC SOFTWARE FOR NON-MACHINISTS

Contents

Software Introduction	2
General Navigation and Loading your file	3
Load	5
Setup.....	5
Help	5
Welcome to MeshCAM.....	5
File	6
Edit.....	6
Menu	6
View.....	7
Geometry.....	7
Geometry Properties	7
Relief.....	7
Job Setup	8
Set Job Type	12
Define Stock	13
Set Program Zero.....	14
Set Retract Height.....	14
Set Max Depth.....	15
Set Machine Region.....	15
Creating Region Shapes	16
Center to Corner.....	16
Set Geometry Supports	17
Understanding Toolpaths.....	17
Roughing	17
Unified Finishing.....	18
Waterline Finishing.....	19
Parallel Finishing	19
Pencil Finishing.....	20
Cutout	20
Contour (2D).....	21
Pocket (2D).....	22
Simulating your CNC job	23
Saving your G-code.....	25

Software Introduction

You have a simple goal, to take a CAD file and get your mill to cut it. You don't want to become an expert on CAM software, you want parts today.

MeshCAM is made for people who just want to make parts now.

Here's the MeshCAM process:

1. Load a file from almost any CAD program
2. Build an efficient toolpath with minimal input
3. Save gcode that works on your CNC machine

Don't let the simplicity fool you, MeshCAM has lots of flexibility and options for advanced machining.

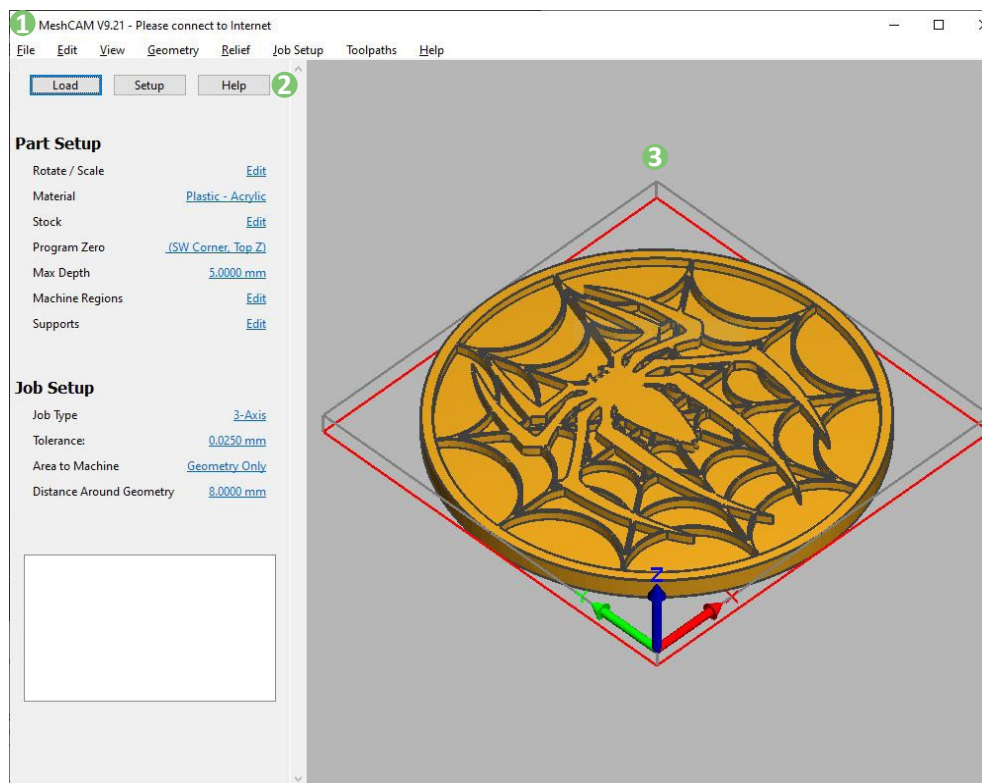
This manual will be broken up into four sections:

1. General navigation and loading your design file
2. Understanding toolpaths
3. Simulating your CNC job
4. Saving your G-code

General Navigation and Loading your file

MeshCAM's interface is segmented into three main areas:

1. Menu Bar
2. Design Tab
3. Workspace



1. Menu Bar

The Menu Bar is where you will find all your standard application menu options. Keep in mind that this is also where you will find your tool library.

2. Design Tab

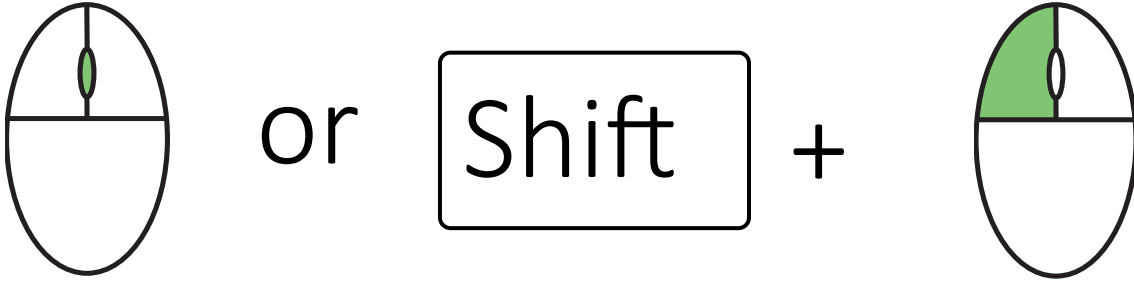
The Design Tab is where you will edit your setup and workspace and create your design.

3. Workspace

The Workspace is where you will be able to see your project and the different actions you can take on your project.

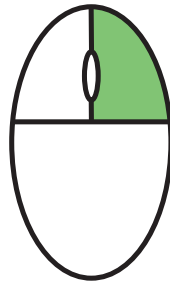
Zoom

To Zoom in and out of the workspace use your mouse scroll wheel. This will let you zoom in to get a clearer view of the actions you take on your project or zoom out to get a look at the entire workspace. Another way to zoom in or out can be done by selecting shift and left-clicking while moving your mouse up and down.



Pan

When zoomed into your workspace it may be necessary to move left, right, up, or down. This is known as panning. Right click and hold while moving your mouse to pan around the workspace.



Welcome to MeshCAM

Welcome to MeshCAM is a pop up that is prompted to you when launch MeshCAM. This is a great place to start for first project or loading a model easily. There will also be a small section with files that have recently been imported into MeshCAM.

Load

Load will allow you to import a file type into MeshCAM to work on your project. Selecting load will open your file explorer.

Select a File

Select a File allows you to open file types to be imported into MeshCAM. File types include STEP Files, MeshCAM Files, 2D files, 3D files, and Image Files.

Setup

The Setup menu will allow you to input your machine, tool types, and registration information.

List Tools

List Tools will display all of the tool types currently saved into your MeshCam.

Select Tool

The Select Tool settings will allow you to choose between the various tool types that are currently saved into MeshCAM. Selecting the select tool button will prompt you a Select Tool pop up. This pop up will allow you to select what tool type you wish to use for a toolpath from a list of tool types saved into MeshCAM. This pop up will also allow you to add, delete, copy, or edit any tool types.

Set Machine

Selecting Set Machine will prompt a Select Machine pop up. This pop up will list various machine types to choose from. You will be able to add machines to the list, edit machines from the list, and select machines from the list.

Select Machine

Selecting a Machine will save the machine into the software and give and allow MeshCAM to operate within the physical parameters that the selected machine supports.

Register

Register will allow you to register for MeshCAM Standard or MeshCAM Pro. Selecting the register will prompt you with a Registration Information pop up.

Registration Information

The text box next to Name is where you will type in your name corresponding to your registration code. The text box next to the Registration Code allows you to type in or paste your registration code.

Help

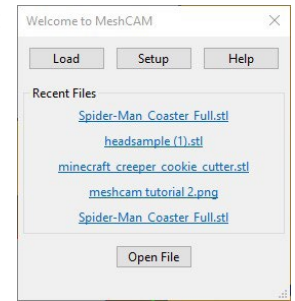
The Help menu has items related to application assistance and registration.

Getting Started

Selecting Getting Started will open a web browser and navigate to the MeshCAM web page. This page will have a list of various tutorial to choose from and videos to watch that will have interactive step by step demonstrations of how to operate MeshCAM.

Register

Register will allow you to register for MeshCAM Standard or MeshCAM Pro. Selecting Register will prompt you with a Registration Information pop up.



Menu

Now that the Welcome to Meshcam prompt is no longer in the way, there will be open access to the menu toolbar. The menu contains important features and it's good to know where they are located.

File

The File menu allows you to find actions related to control over projects and files within MeshCAM.

Open

Open allows you to open file types to be imported into MeshCAM. File types include STEP Files, MeshCAM Files, 2D files, 3D files, and Image Files.

Open 2D DXF

Open 2D DXF will allow you to browse for 2D files (*.dxf, *.svg) to import into MeshCAM to start on your project.

Save Relief as Bitmap

Save Relief as Bitmap will save the current relief in MeshCAM as a bitmap.

Save Relief as STL

Save Relief as STL will save the current relief in MeshCAM as an STL.

Save Project

Save Project allows the user to save any current progress on a project. If the project has been previously saved, you will be prompted to browse for a location to save and name your project file. This action will save the project as a MeshCAM Job (*.mcf) file type.

Close

Close will force quit MeshCAM. Therefore, if progress has not been saved it is important to do so before closing out the application.

Edit

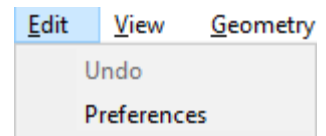
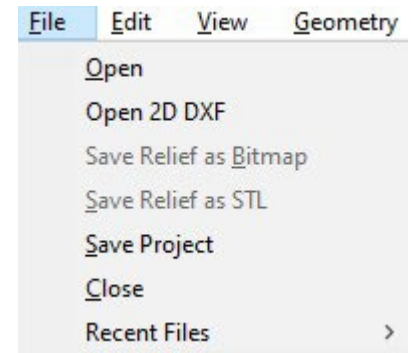
The action in the edit menu will directly affect the design space within MeshCam.

Undo

The undo button will reverse any action done in MeshCAM.

Preferences

Preferences will prompt a small toolbox. This toolbox will allow you to have control over Graphic Preferences, Geometry Preferences, and Global Preferences. Both Graphic and Geometry Preferences have check boxes to toggle the actions on and off. Graphic Preferences has a check box to show full program zero. Global Preferences allows the user to change the user interface units of measurement and to reset "don't ask again" settings.

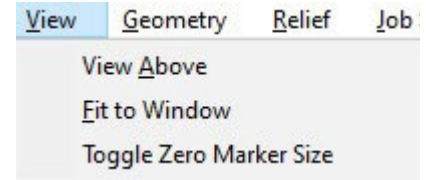
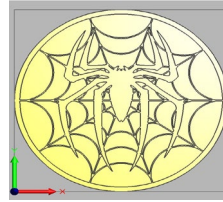


View

The view menu actions will allow you to change how the MeshCAM workspace is viewed.

View Above

View above will change the workspace view to a top down view, fitting the workspace on the screen as pictures to the right.



Fit to Window

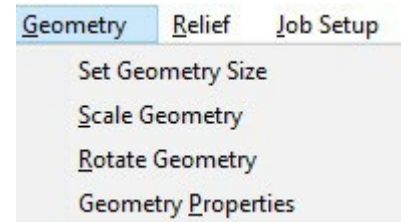
Fit to Window will change the workspace view to fit as much of the project onto the screen as possible. Fit to window will zoom out if needed as well as zoom in if needed.

Geometry

Geometry will allow you to manipulate your project's scaling, sizing, and orientation. Also, showing you the dimensions of your project.

Set Geometry Size

Set Geometry Size will prompt you to select one of the three X, Y, and Z-axis. Upon selecting an axis, you will have the action to set a new size for the selected axis with the appropriate unit of measurement given.

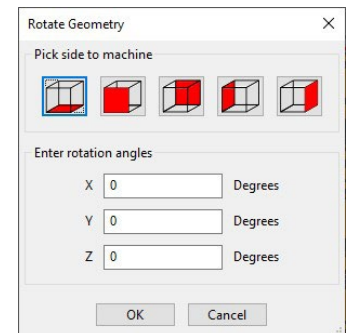


Scale Geometry

Scale Geometry will allow you to manipulate the proportions between all three axes. Scale geometry has a check box for the axis to be uniform. If uniform all the axes will have a direct correlation. When changing one axis it will impact the other axis. However, if the uniform box is not checked, you can change the scaling as you wish without a direct correlation between all the axes.

Rotate Geometry

Rotate Geometry will allow you to change the orientation of your project and where your desired location to machine is. Upon selecting rotate geometry, a prompt will appear. The prompt will allow you to decide what side of the project you wish to use your machine on. The prompt will also allow you to enter rotation angles to orientate your project freely. Here you will be able to change the degrees of all three axis.



Geometry Properties

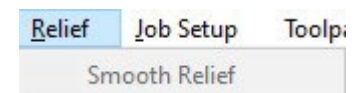
Upon selecting geometry properties, you will be prompted with a toolbox listing the geometry dimensions of your project. The dimensions include the min, max, and size for each axis of your project in the corresponding units of measurement.

Relief

Relief will allow you to smoothen your relief if your project is the correct format.

Smooth Relief

Selecting this button will smooth your relief.



Job Setup

Job setup allows you to choose what machine and tool types you will be using for your project and will allow you to input the specific actions for your machine to take on your project. In this job setup you will also be able to set the specifications of your stock of choice.

Select Machine

A select machine prompt will pop up allowing you to scroll through a list of various machine types. The prompt has three buttons being Add, Edit, and Select.

Edit

The Edit button of the Select Machine prompt will only allow you to edit any machine types that have been added by you. You can choose what machine you want to edit by left clicking on the name of the machine you wish to edit and ensuring that it is highlighted once clicking. By clicking the machine you wish to edit a Machine Configuration pop up will appear with the machine's name and measurements allowing you to go in and edit each feature. The machine configuration pop up will have an OK and Cancel button. OK button will close the prompt and save the newly edited machine as a copy of the original machine leaving both the originally added machine and new edited machine in your list of machine types. The Cancel button will close the pop up and will not save any inputted measurements to the list of machine types.

Add

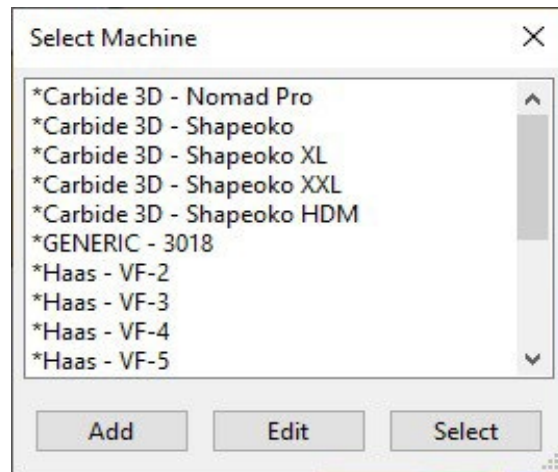
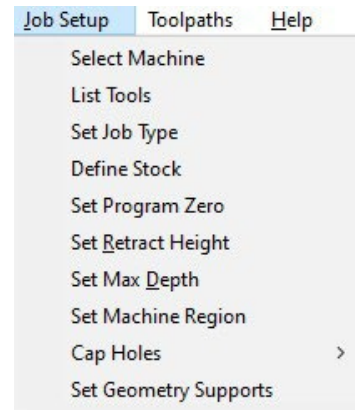
The Add button will prompt a Machine Configuration pop up. This pop up will allow you to input the name of your machine and the measurement for your spindle and travel. At the bottom of the machine configuration pop up there will be an OK and Cancel button. The OK button will close the prompt and save the current machine and its measurements to the list of other machine types. The Cancel button will close the prompt and will not save the machine to the listed machine types.

Spindle

You will be able to input the power for your spindle in watts, the minimum and maximum rpm (revolutions per minute), and RPM rounding. RPM rounding allows the machine to automatically round RPM changes to a specific range that may perform better on your CNC machine.

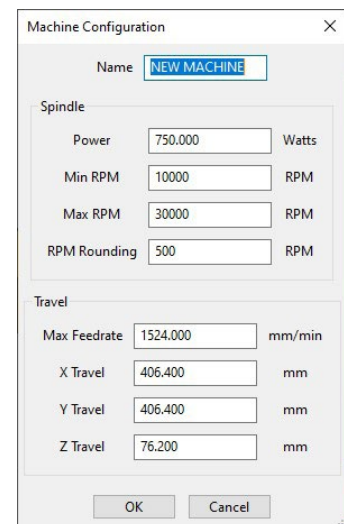
Travel

You will also be able to input the travel settings of your machine to make sure that MeshCAM does not output any machine instructions that would go outside of the physical properties that your machine is capable of.



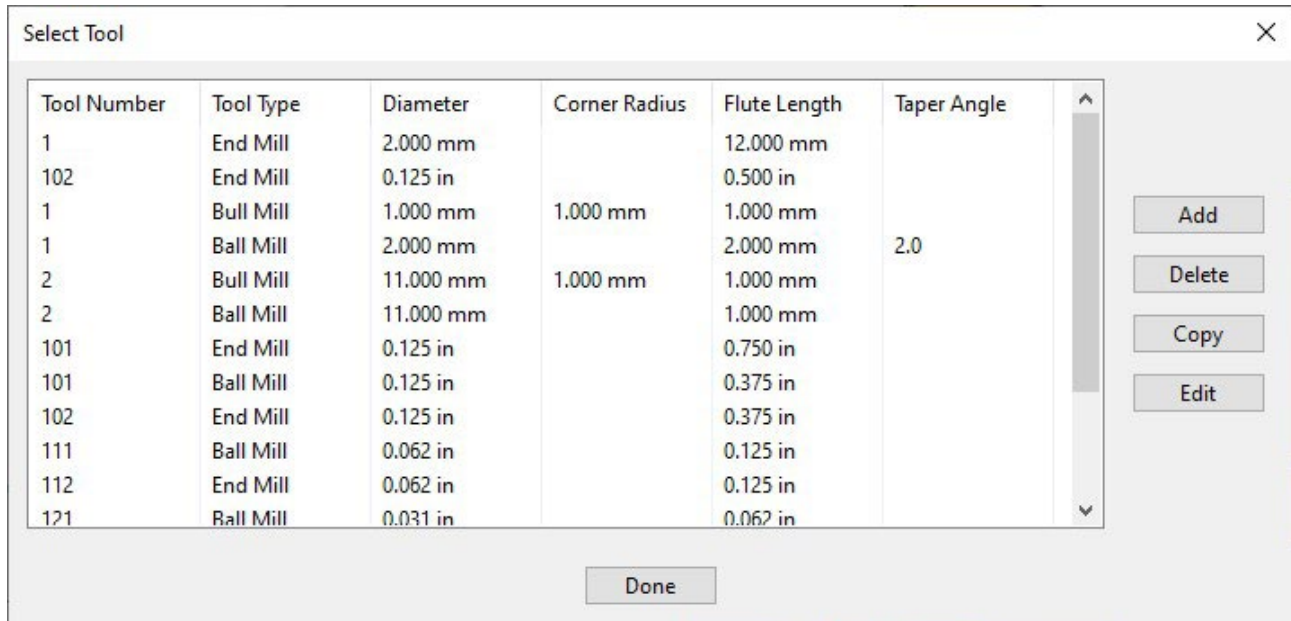
Select

The select button for the select machine prompt will allow you to select the highlighted machine from the list of machine types for your project.



List Tools

List tools will prompt a Select Tool pop up. This pop up will display a list of various tool types that are currently present into the software. The pop up will also display any tool types that have been added by you. The Select Tool pop up has four buttons on the right side and one at the bottom. The four buttons on the right are Add, Delete, Copy, and Edit.

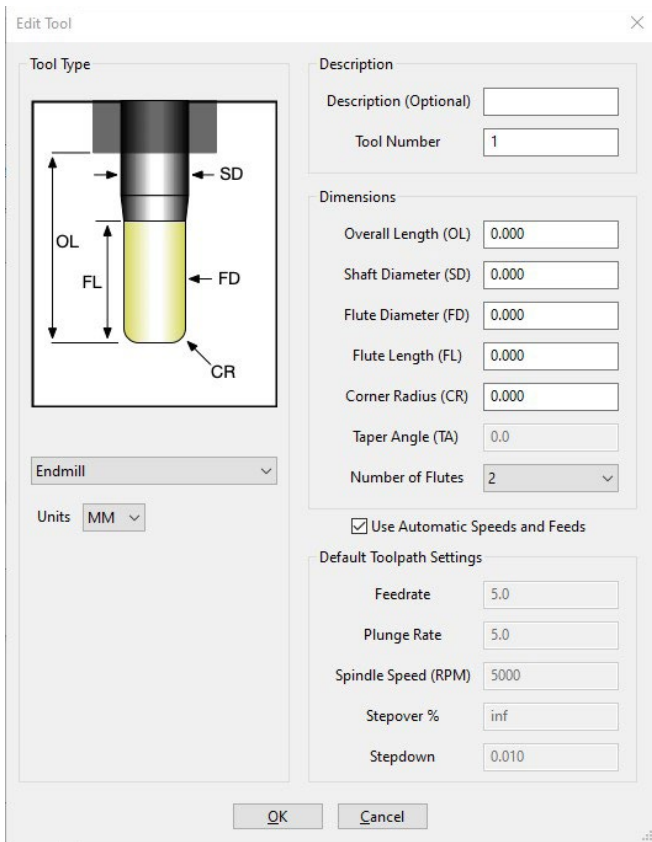


Add

The Add button will prompt and Edit Tool pop up where you will be able to input measurements of a new tool type. The Edit Tool pop up will allow you to change the Tool Type, Description, Dimensions, and Default Toolpath Settings. The Tool Type will display a diagram of the tool type displaying the different measurements needed to be inputted. It will also give you the option to choose between four different tool types Endmill, Ball Mill, Tapered Endmill, and Tapered Ball Mill. Selecting between the various tool types will change the diagram to display the correct measurements needed for each tool type. Tool Type will also allow you to change the units for the measurements that will be input into the pop up. The options of measurements are millimeter and inches.

Description

The description in this pop up will allow you to add an optional description to the tool type to input any notes of your own if needed as well as adding a tool number for the tool being added.



Toolpath

Under the check box will be the Default Toolpath Settings. These settings will give you the option to adjust the Feed Rate, Plunge Rate, Spindle Speed(RPM), Stepper %, and Stepdown. At the bottom of the edit tool type pop up there is a select button and a cancel button.

OK/Cancel

By selecting OK you will save the tool type and all the measurements into the list of various list of tool types that was prompted when selecting list tools. By selecting cancel, MeshCAM will force quit the edit tool pop up and will not save any data from the pop up to the tool types list.

Dimensions

The dimensions are the precise needed measurements for the tool being added. The list of various dimensions include Overall Length(OL), Shaft Diameter(SD), Flute Diameter(FD), Flute Length(FL), Corner Radius(CR), Taper Angle(TA), and Number of Flutes.

You will only be able to input the measurements for the corresponding tool type. For example, the Endmill and Ball Mill will not have the option to input a tapered angle because both tool types are not a tapered tool type. Also, both the Ball Mill and Tapered Ball Mill tool types will not be able to input a corner radius, but both the Endmill and Tapered Endmill will have a corner radius measurement because they have a measurable corner radius.

The number of flutes will allow you to choose the amount of flutes your tool type has. You will be able to choose between one and five flutes. Under the dimensions settings there will be a check box to use automatic speeds and feeds.

By toggling the check box you will be allowed to keep the default speeds and feeds MeshCAM has or adjust the default toolpath settings yourself. However, once adjusting any settings in the default toolpath settings and toggling the check box to the default it will remain the same and not undo the original default toolpath settings.

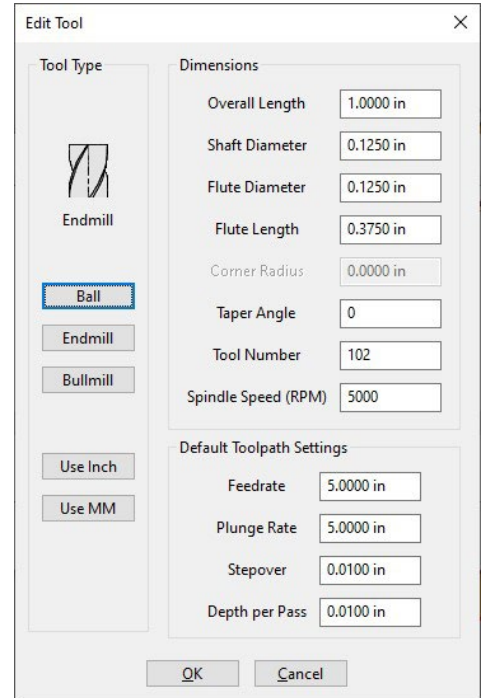
Delete

The Delete button on the select tool pop up will allow you to delete any of the tools from the select tool list.

Copy

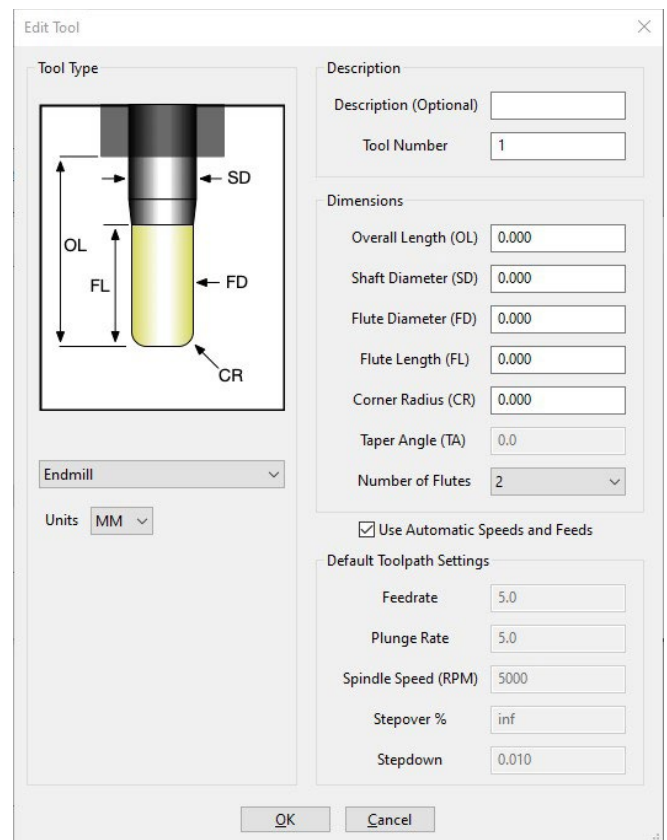
The copy button will allow you to make a duplicate of any of the tools listed in the select tool prompt. By pressing copy on a highlighted tool you will be prompted with an Edit Tool pop up different from the Add button Edit Tool pop up. This pop will duplicate all of the selected tool types settings and display them, allowing you to have control and the ability to make any changes to any of the settings.

You will be able to change the tool type to a Ball Mill, Endmill, or Bullmill. There will also be an option to change the unit of measurements to either inches or millimeters. The dimension settings will allow you to change the Overall Length, Shaft Diameter, Flute Diameter, Flute Length, Corner Radius (only possible if selecting a Bullmill tool type), Taper Angle, Tool Number, and Spindle Speed (RPM). Under the dimensions there will be the default toolpath settings. Under the toolpath settings you will be able to change the Feederate, Plunge Rate, Stepover, and Depth per pass.



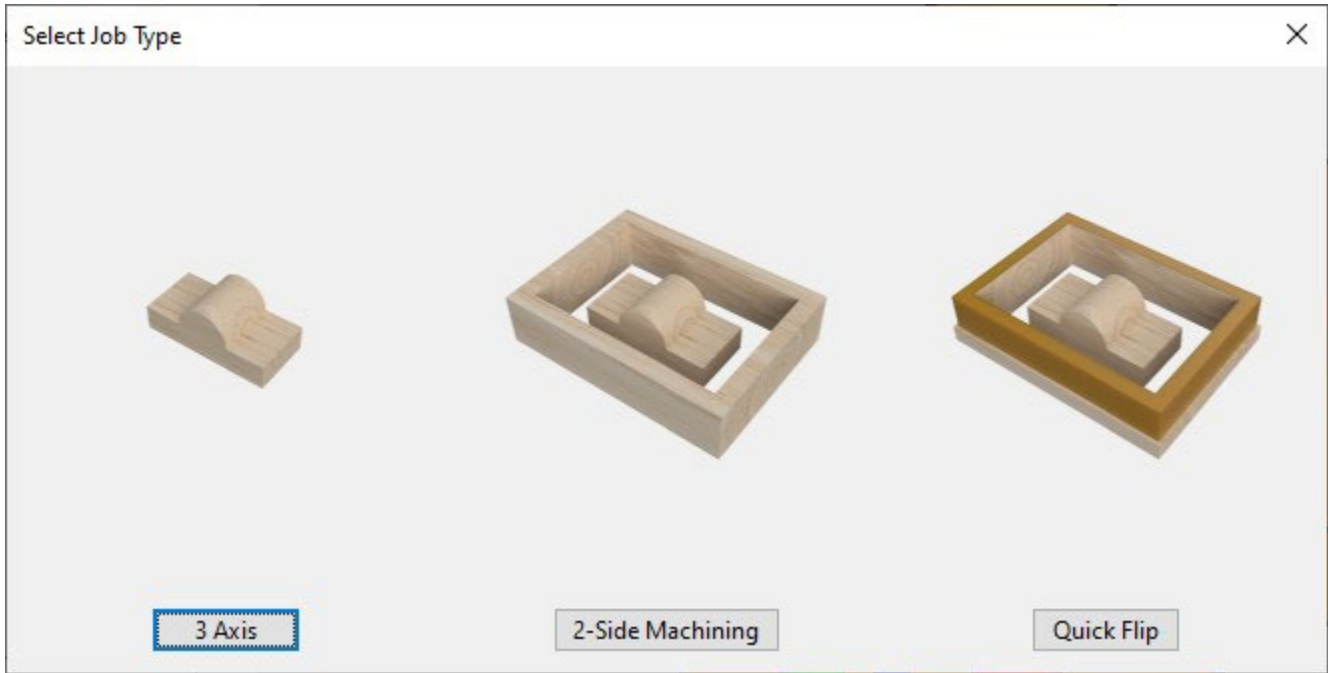
Edit

The fourth button in the select tool prompt is Edit. The Edit button will allow you to go in and adjust any information that was added when pressing the Add button in the Select Tool pop up. By selecting a tool type and pressing the Edit button you will be prompted with an Edit Tool pop up. This will be the same pop up that was prompted when adding a new tool type. However, once pressing Edit on the selected tool type that tool types information will already be input into the edit tool prompt allowing you to adjust any of the settings shown. The button at the bottom of the select tool pop up is Done. This button will simply close the pop up.



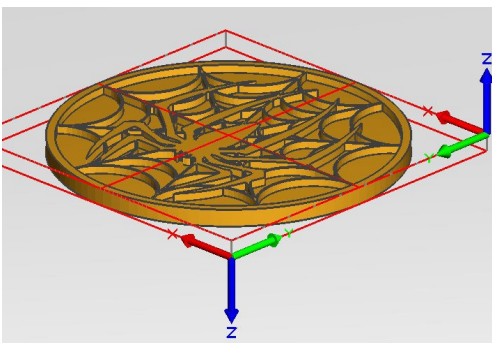
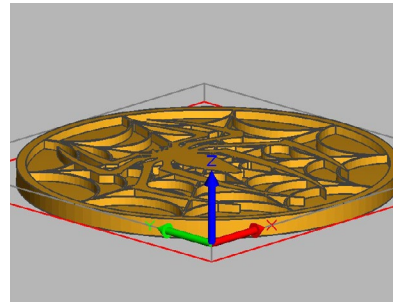
Set Job Type

Set Job Type will allow you to choose between a 3 Axis, 2-Side Machining, or Quick Flip job type. By pressing on select job type a Select Job type prompt will pop up with diagrams of the job type and their appropriate job type name under the diagram to be selected.



3 Axis

Pressing on the 3 Axis button will change the job type to 3 Axis. A 3 Axis job type will show your project in the standard X, Y, and Z axis. A 3 Axis job type will keep the workspace in the same position while the machine operates using the three axes to carve out your project.



2-SideMachining

Pressing on the Flip Job button will change the job type to a Flip Job. A Flip Job will show your project in the standard X, Y, and Z axis. However, the axes will be displayed from the top and bottom of your project. A flip Job will allow you to use your from various sides if your project demands carving on different planes of your multidimensional project.

Quick Flip

Version 9 of MeshCAM adds a new "Quick Flip" job type to simplify the process of machining two sides of a model. It does this by enforcing a more rigid workflow for the user, but it also reduces the likelihood of mistakes.

Define Stock

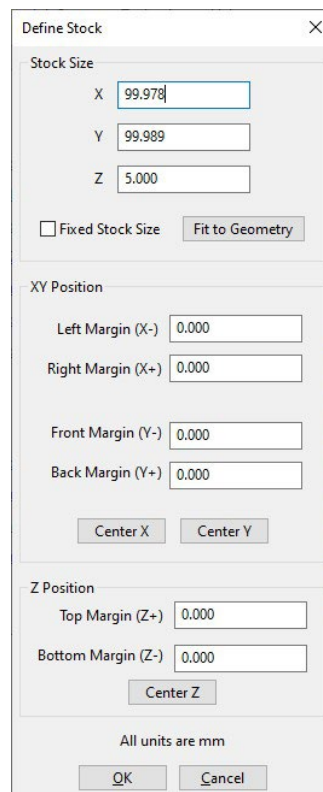
Define Stock will allow you to input all the specifications and measurements for your stock. Once pressing Define Stock you will be prompted with a Define Stock pop up. In this pop up you will be able to input the measurements and various settings for the Stock Size, XY Position, and Z Position. Under this there is an OK and Cancel button. The OK button will save any information that you have input and will apply it to your project. The Cancel button will force quit the pop up and not save any of your information inputted into your project.

Stock Size

Stock Size settings will allow you to change the X, Y, and Z measurements of your stock. Under this will be a Fixed Stock Size check box to toggle on and off. There is also a Fit to Geometry button that will set your stock size to match the exact measurements of your project.

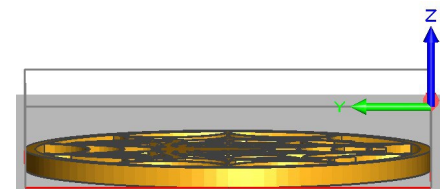
XY Position

The XY position settings will allow you to have control over where your project will be located on your stock. For example, I could choose to have the machine carve this spider coaster project picture below on the left hand corner of the stock leaving the bottom right hand corner untouched. Also, by moving your project on a Fit to Geometry stock size it will change the size of the stock measurements. In this setting you will be able to have control over the Left Margin (X-), Right Margin (X+), Front Margin (Y-), and Back Margin (Y+). Each of these margins allows you to move your project using the X and Y axis plane in a negative or positive direction. At the bottom of the XY position settings there will be a Center X and Center Y button that will center both axes based on your input. For example, if the Left Margin (X-) has an input of 30 and the Right Margin (X+) has an input of 0 the Center X button will center the x axis plane and meet both Left and Right Margins in the middle to both have an input of 15. The Center Y button would do the same, but for the Y axis plane only.



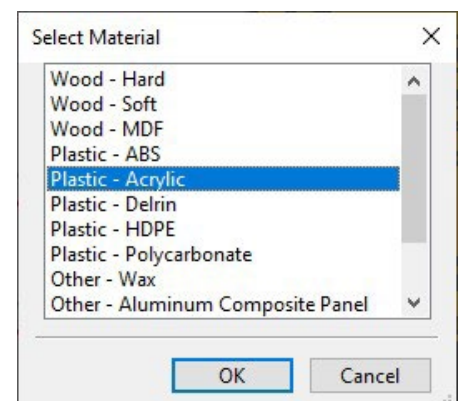
Z Position

The Z position settings will also allow you to have control over where your project will be located, but only through the Z axis. Again, by moving your project on a Fit to Geometry stock size it will change the size of the stock measurements. In this setting you will be able to have control over the Top Margin (Z+) and the Bottom Margin (Z-). These margins allow you to move your project through the Z axis plane in a negative or positive direction. The Z position settings also have a Center Z button that will allow you to center the position of your project on your stock on the Z axis plane using your inputted information on the Top and Bottom Margins.



Select Material

Selecting Material will prompt you with a Select Material pop up. This pop up will have a list of various material types that can be chose for your stock. You can scroll through the list using a scroll wheel on the mouse or by selecting the vertical scroll bar to go up or down the list. Left click on a material to select it. The selected material will be highlighted and can be saved to your project by pressing the OK button at the bottom of the pop up or double left clicking on the material. Pressing Cancel on the pop up will force quit the pop up and not save the selected material to your project.

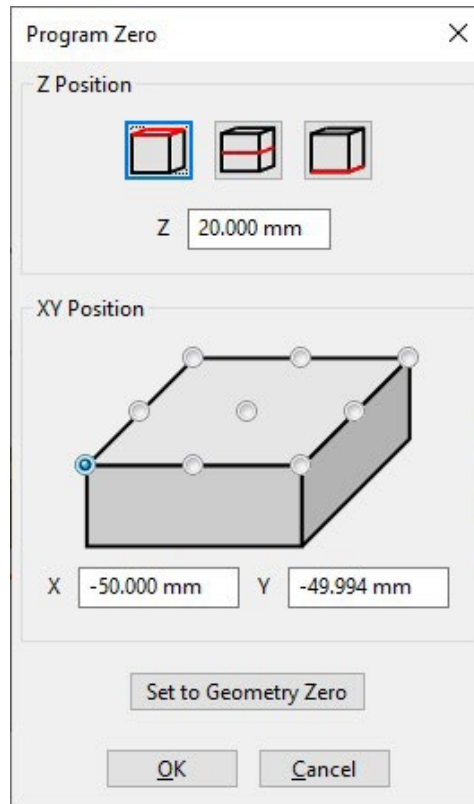


Set Program Zero

Set Program Zero will allow you to set your zero. Your zero is where your machine will guide itself and refer to as the point of origin or where it will reference when carving. Your zero can be seen on your project where the X, Y, and Z axes arrows intersect. When selecting Program Zero you will be prompted with a Program Zero pop up. In this pop up you will be able to change the zero in various Z position settings and XY position settings as well as setting your zero to Geometry Zero.

Z Position

The Z Position settings will have three diagrams of cubes and below the diagrams will be a text box to change Z Position settings manually. The three diagrams of cubes will all be identical except for a red outline at different Z positions. Each red outline will demonstrate on what Z position your zero will be located. The first cube to the far left will have the zero located at the highest Z position of your stock. The cube in the middle will have the zero located in the middle of your stock. The cube to the far right will have the zero located at the lowest Z position of your stock. There is a text box under the diagrams, in this text box you can input a specific Z position in the corresponding unit of measurements that you are using.

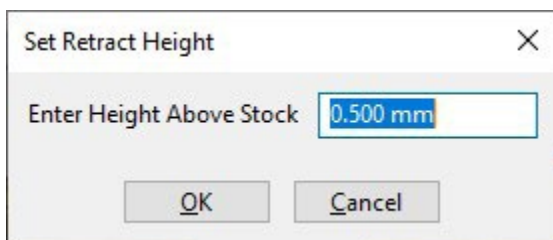


XY Position

The XY position settings will allow you to move your zero anywhere on the X and Y axes. There is a diagram of a rectangular prism and below there are text boxes where you can input an X and Y coordinate. The diagram will have nine different points on the top face of the rectangular prism to select from. The text box under the diagram will allow you to input a specific X and Y coordinate on your project to set your zero.

Set Retract Height

The Set Retract Height setting will allow you to set the retract height measurement you wish for your machine to perform. The retract height tells your machine how far to retract or pull up after performing an operation or in between subsequent cutting passes. Pressing on the Set Retract Height will prompt you with a Set Retract Height pop up.



Enter Height Above Stock

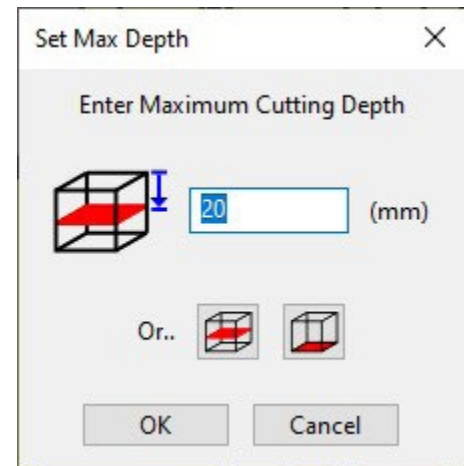
There is a text box where you can input height above stock measurement in the corresponding unit of measurement.

Set Max Depth

The Set Max Depth setting will allow you to input the lowest you wish your machine to carve. Pressing on the Set Max Depth setting will prompt you with a Set Max Depth pop up.

Enter Maximum Cutting Depth

This settings has a text box for inputting measurements and two diagram buttons at the bottom of the settings to select from. You will be able to input the measurements of how deep you wish your machine to carve in the text box. Keep in mind that the measurements go from the top of your stock downward. Under the text box there are two diagrams that are buttons. Both diagrams are transparent cubes with a red outline through a Z axis plane. The first diagram button is the option to set the max depth to the middle of your stock and the second diagram button is the option to set the max depth to the bottom of your stock.



Set Machine Region

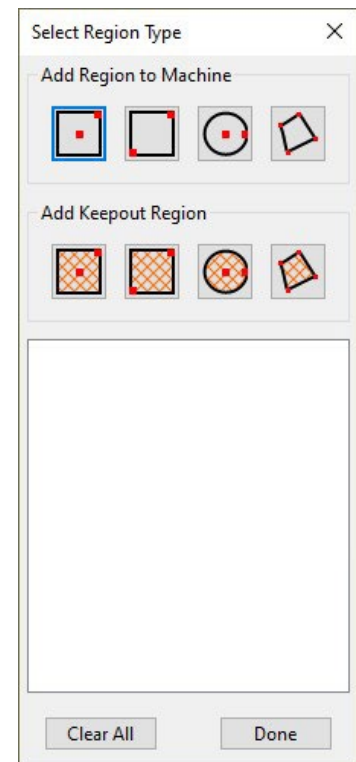
The Set Machine Region setting will allow you to pinpoint what area of your project to carve and what to avoid when carving. Pressing on the set machine region settings will prompt a Select Region type pop up and it will change your view of your project to a birds eye view fitting the entire project on your screen. This pop up will have an add region to the machine and add keep out region settings as well as a list of all regions that have been added to your project.

Add Region to Machine

The Add Region to Machine setting will allow you to add a specific region to your machine that you wish to carve for your project. There are four different shapes to choose from when adding a region. There is a rectangle (Center ,Corner), Rectangle(Corner, Corner), Circle(Center, Corner), and a Polygon. When pressing on a shape a small pop up will show in the corner to create your region and give directions on how to create your shape. There is also a cancel button that can be selected to cancel the action of the shape that has been selected.

Add Keep out Region

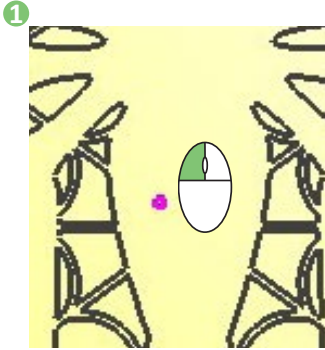
The Add Keep out Region setting will allow you to add specific regions in your project you do not want your machine to carve. There are four different shape options to choose from when adding a region. There is a rectangle (Center ,Corner), Rectangle(Corner, Corner), Circle(Center, Corner), and a Polygon. When pressing on a shape a small pop up will show in the corner to create your region and give directions on how to create your shape. There is also a cancel button that can be selected to cancel the action of the shape that has been selected.



Creating Region Shapes

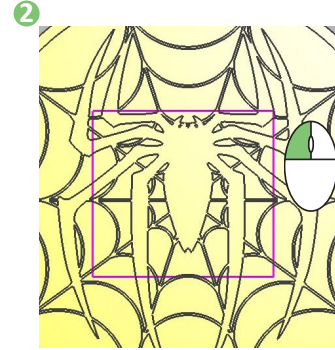
The first rectangle and circle are created by clicking center to corner. The second rectangle is created by clicking corner to corner. The last shape to add a region is a polygon. To create the polygon shape you will be able to freely click on various points of your project to create your desired shape and when done creating your shape you can right click anywhere on the project to connect the first and last click.

Click one will be the center of your shape.

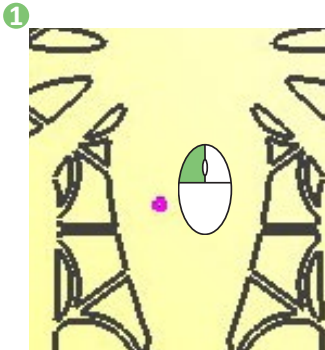


Center to Corner

Click two will be the corner or edge of your shape.

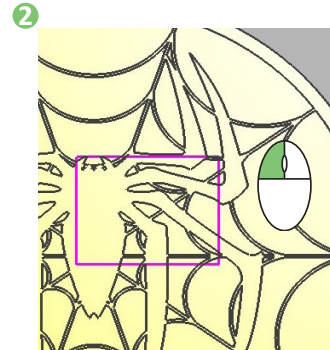


Click one will be the corner of your shape.

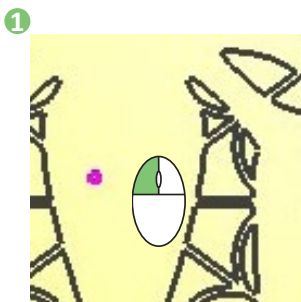


Corner to Corner

Click two will be the corner or edge of your shape.

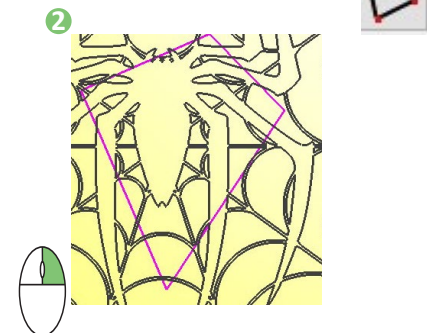


Click one will be the beginning of your polygon.



Polygon

You will be able to freely click on various points of your project to create your desired shape and when done creating your shape you can right click anywhere on the project to connect the first and last click.



Set Geometry Supports

The Set Geometry Supports settings will allow you to arrange any supports for your project. Selecting the set geometry supports settings will prompt you with a support dimensions pop up and set your view to a birds eye view fitting your entire project on your screen.

Support Height

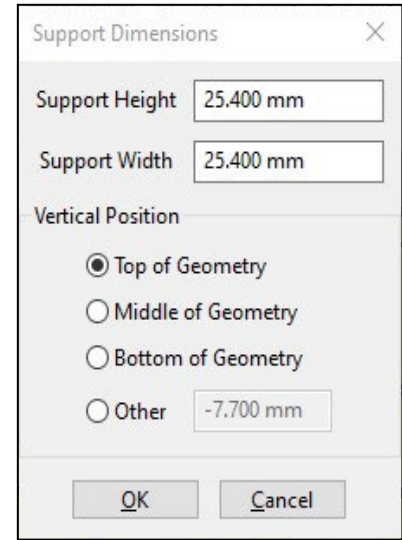
Here you will be able to input the support height in the corresponding unit of measurements.

Support Width

Here you will be able to input the support width in the corresponding unit of measurements.

Vertical Position

The Vertical Position settings will allow you to change the locations of your supports vertically. You will be able to set the supports at the Top of Geometry, Middle of Geometry, Bottom of Geometry, or Other. The first three options will allow you to set your supports at the top, middle, or bottom of your project. The other option when selected will allow you to type into a text box. This text box will allow you to input a specific measurement of the vertical location of your supports for your project.



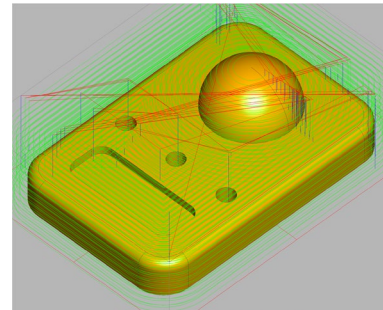
Understanding Toolpaths

The Tool Path settings will allow you to choose a various toolpath options for your machine to receive instructions and follow. Typical selection of toolpaths will be to first use a roughing toolpath, next a finishing toolpath for a nice precise finished surface, and finally any drilling toolpaths to create holes in your part.

The last step will be a contour or cutout toolpath to make removing your part easier and make it ready for any post processing you may need to do. This section will explain the different type of toolpaths available and how they may be used. Since many of the tools have similar options, these options will be explained once and then referenced afterwards.

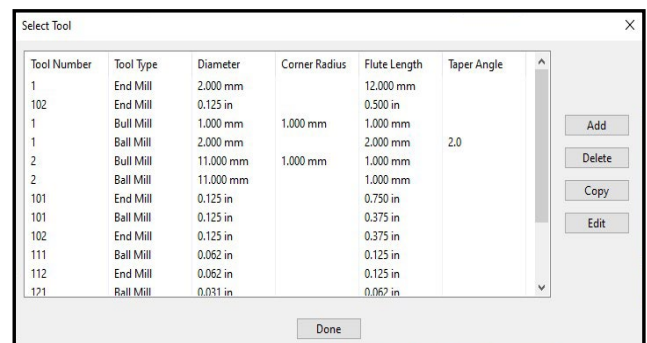
Roughing

The Roughing settings will allow Roughing toolpaths do the bulk of the material removal for your projects. MeshCAM lets you select two roughing strategies, contour offset and parallel roughing. Selecting the roughing settings will prompt you with a Roughing Toolpath pop up. This pop up will allow you to change what settings your rough toolpath will have when taking actions on your project.



Select Tool

The Select Tool settings will allow you to choose between the various tool types that are currently saved into MeshCAM. Selecting the select tool button will prompt a Select Tool pop up. In this pop up you will be able to select what tool type you wish to use for the Roughing Toolpath action. This pop up will also allow you to add, delete, copy or edit any tool types.



Speeds and Feeds

Speeds and Feeds will allow you to change any of the speed or velocity settings for the Roughing Toolpath instructions. Under these settings you can change the Feedrate, Plunge Rate, and RPM using a text box to input your desired settings. The Feedrate will dictate at what speed the machine will carve your project. The Plunge Rate dictates at what speed the tool bit is driven into your project. RPM dictates the revolutions per minute the spindle will complete.

Cutting Parameters

Cutting parameters will define your stepdown and stepover percentage. These options will define both the quality of the toolpath and the amount of time it will take to complete.

Stepdown defines how much depth the cutting tool will cut into per pass. Stepover % defines how far a toolpath's next cut will go laterally into the material. For harder materials a lighter load on these settings is advisable, and on softer materials you can increase the values to save time if your machine can adequately handle it.

Options

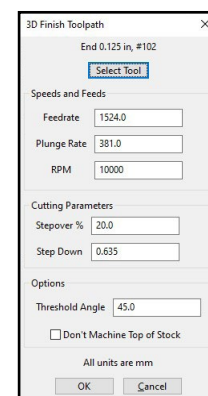
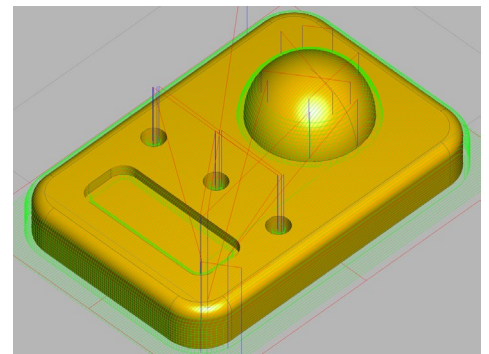
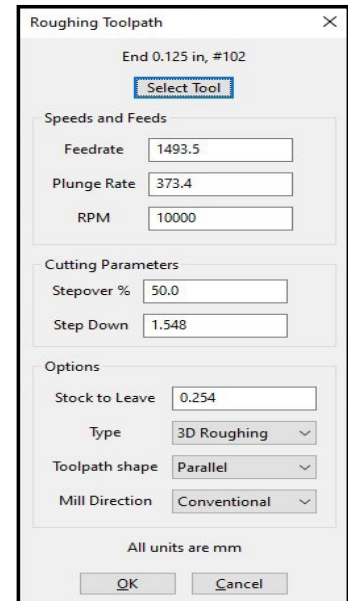
The Options settings will allow you to change any other specific settings for your Roughing Toolpath. The Options settings include Stock to Leave, Type, Toolpath Shape, and Mill Direction. The Stock to leave will have a text box where you can input the stock measurement that must remain on your project for subsequent toolpaths. Type, Toolpath Shape, and Mill Directions all have a drop down arrow with two options to choose from. The Type setting will allow you to select between 2D and 3D Roughing. Toolpath Shape settings allow you to choose between Parallel and Counter Offset. Mill Directions settings allow you to choose between conventional and climb.

Unified Finishing

Unified Finishing is the easiest way to finish your parts. This unique toolpath switches modes depending on the shape of your parts to create the best finish possible in the least amount of time. Selecting the Unified Finishing settings will prompt you with a 3D Finish Toolpath pop up. This pop up will allow you to change what settings your Unified Finishing toolpath will have when taking action on your project.

Options

The Options settings will allow you to change any other specific settings for your Unified Finishing Toolpath. The Options settings include the Threshold Angle and a check box to either Machine or Not Machine Top of Stock. The Threshold Angle settings has a text box next to it allowing you to input a specific angle degree. The check box will allow you to decide if you would like to machine the top of stock.

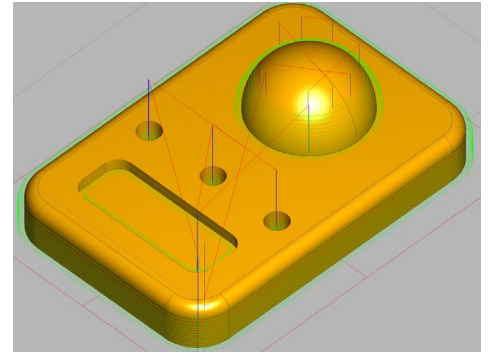


Waterline Finishing

Waterline Finishing, sometimes called Z-Level finishing, steps down your part to give a good finish on the steeper parts of your model. Selecting the Waterline Finishing settings will prompt you with a Waterline Finish toolpath pop up.

Options

The Options settings will allow you to change any other specific settings for your Waterline Finishing toolpath. The options settings have a Threshold Angle text box and a Full Retracts Only check box. The Threshold Angle text box will allow you to input a specific degree angle for the toolpath to follow. The Full Retracts Only check box can be toggled on and off by clicking the check box.



Waterline Finish Toolpath

End 0.125 in, #102

Select Tool

Speeds and Feeds

Feedrate 1493.5

Plunge Rate 373.4

RPM 10000

Cutting Parameters

Step Down 1.548

Options

Threshold Angle 45.000

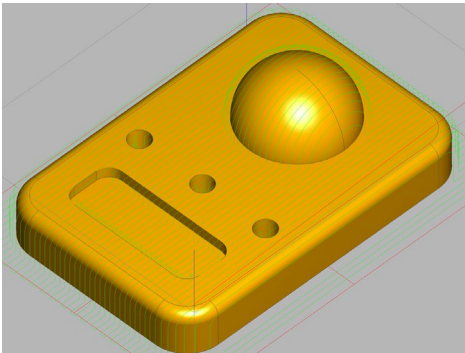
Full Retracts Only

All units are mm

OK Cancel

Parallel Finishing

Parallel Finishing takes a series of back-and-forth cuts across your model to machine the shallow, or flat, parts of your model. Selecting the Parallel Finishing settings will prompt you with a Parallel Finishing Toolpath pop up. This pop up will allow you to change what settings your Parallel Finishing Toolpath will have when taking action on your project.



Options

The Options settings will allow you to change any other specific settings for your Parallel Finishing Toolpath. The Options settings will include the Limit Toolpath to Shallow Areas, Threshold Angle, Mill Direction, and Don't Machine Top of Stock. The Limit Toolpath to Shallow Areas has a check box next to it to toggle these settings on and off, toggling the check box will allow you to input an angle into the Threshold Angle. The Threshold Angle has a check box next to it that can only be typed into when the Limit Toolpath to Shallow Areas check box is toggled on. The Mill Direction has a drop down arrow giving you the option to choose between an X Parallel and Y Parallel Mill Direction. Don't Machine Top of Stock has a check box, this check box will allow you to toggle to machine the top of the stock or not.

Parallel Finish Toolpath

End 0.125 in, #102

Select Tool

Speeds and Feeds

Feedrate 1493.5

Plunge Rate 373.4

RPM 10000

Cutting Parameters

Stepover % 50.0

Options

Limit Toolpath to Shallow Areas

Threshold Angle -1.000

Mill Direction X Parallel

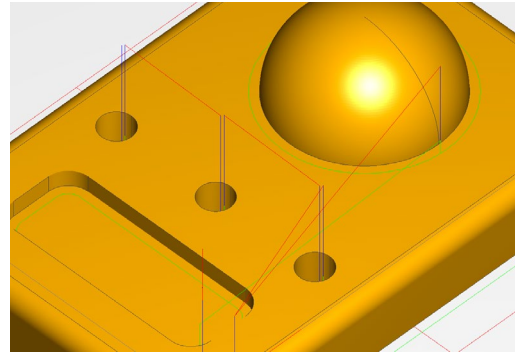
Don't Machine Top of Stock

All units are mm

OK Cancel

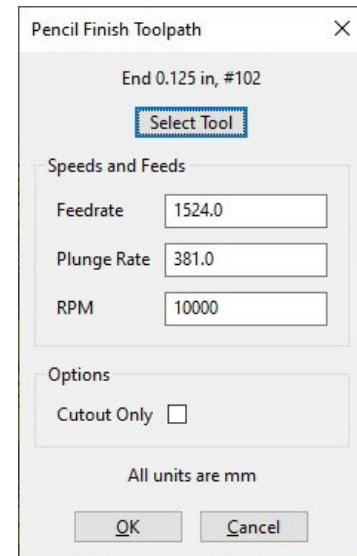
Pencil Finishing

Pencil Finishing is a cleanup toolpath that traces around the corners of your model to remove any leftover stock that might be difficult to remove by hand. Selecting the Pencil Finishing settings will prompt you with a Pencil Finish Toolpath pop up. This pop up will allow you to change what settings your Waterline Finishing toolpath will have when taking action on your project.



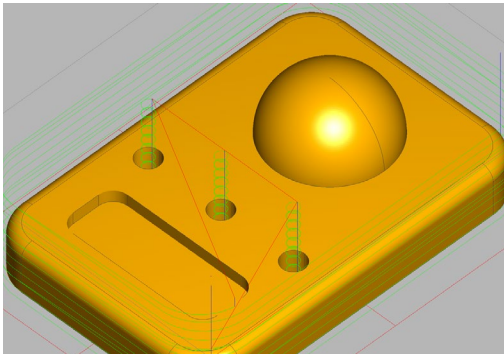
Options

The Options settings for the Pencil Finishing toolpath only has a Cutout Only check box. This check box can be clicked to be toggled on or off.



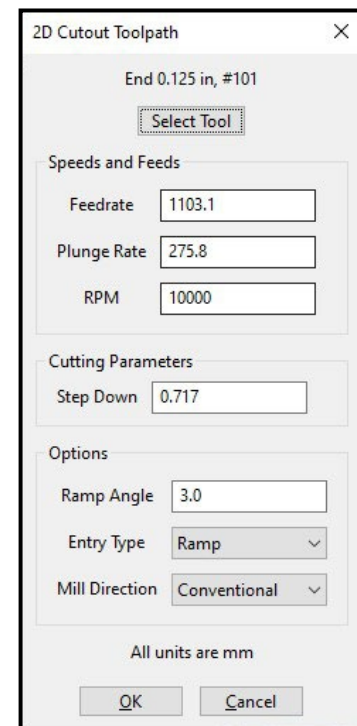
Cutout

The Cutout settings will allow you to simply cutout or remove your project from the stock. Selecting the Cutout settings will prompt you with a 2D Cutout Toolpath pop up. This pop up will allow you to change what settings your Cutout Toolpath will have when taking action on your project.



Options

The Options settings will allow you to change any other specific settings for your Cutout Toolpath. The Options settings include the Ramp Angle, Entry Type, and Mill Direction. The Ramp Angle has a text box next to it allowing you to input a specific angle degree. The Entry Type will have a drop down arrow giving you the option to choose between a Ramp or Plunge Entry Type. The Mill Direction has a drop down arrow giving you the option to choose between a Climb or Conventional Mill Direction.

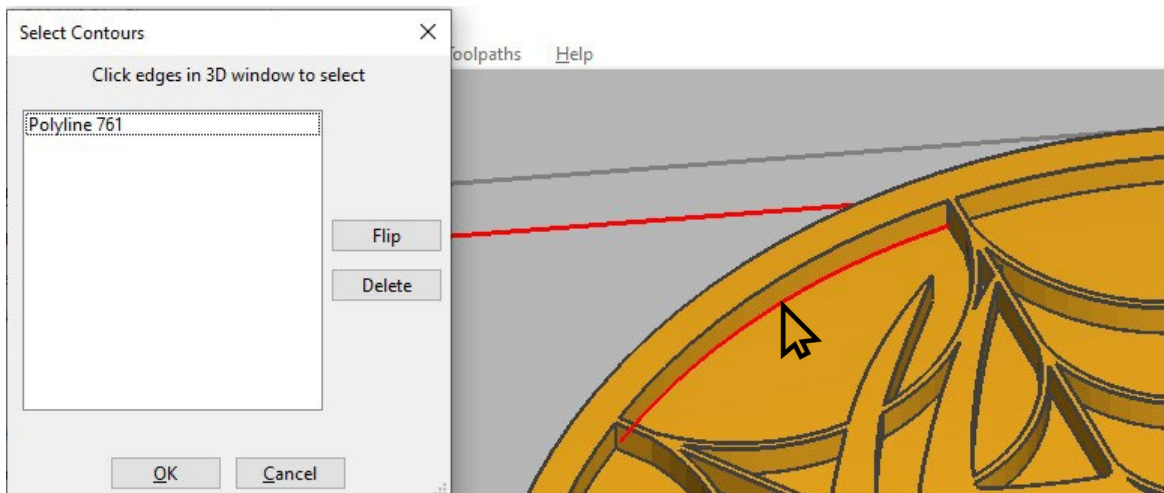
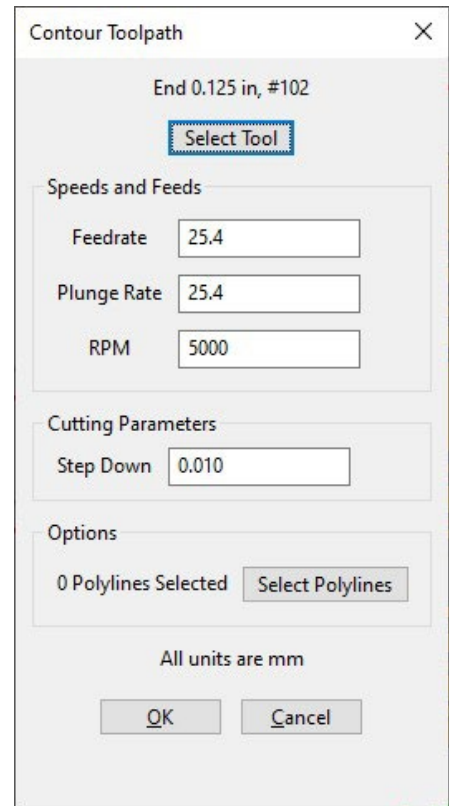


Contour (2D)

The Contour (2D) settings will allow you to change the settings your Contour (2D) toolpath will have when taking action on your project. Selecting the Contour (2D) settings will prompt you with a Contour Toolpath pop up.

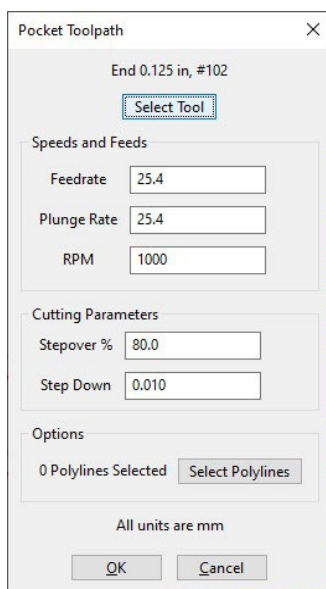
Options

The Options settings will allow you to change any other specific settings for your Contour (2D) toolpath. The Options settings have a Polylines Selected setting with a Select Polylines button. Selecting the button will prompt a Select Contours pop up. This pop up will have a white panel where a list of polylines selected will be displayed. The pop up gives instructions to click on the edges of your project to select those polylines. You can select the polylines by hovering over any of the polylines on your project. Hovering over polylines will be highlighted in red to display which polyline you would be selecting. Left Clicking on a polyline will display the name of the polyline in the Select Contours pop up. The pop up has two buttons to the right, a Flip and Delete button. To select these buttons you must first select a polyline. To select a polyline left click a polyline from the displayed list in the pop up. Selecting a polyline from the list will highlight the polyline. Selecting flip on a highlighted polyline will flip the polyline and will display "(Flipped)" next to the polyline name in the list. Selecting the Delete button while a polyline is highlighted will delete the selected polyline from the list. The OK button at the bottom of the Select Contours pop up will save the selected polylines to the Contour (2D) toolpath settings. The Cancel button at the bottom of the Select Contours pop up will force quit the Select Contours pop up and will not save any of the selected polylines to the Contour (2D) toolpath settings.



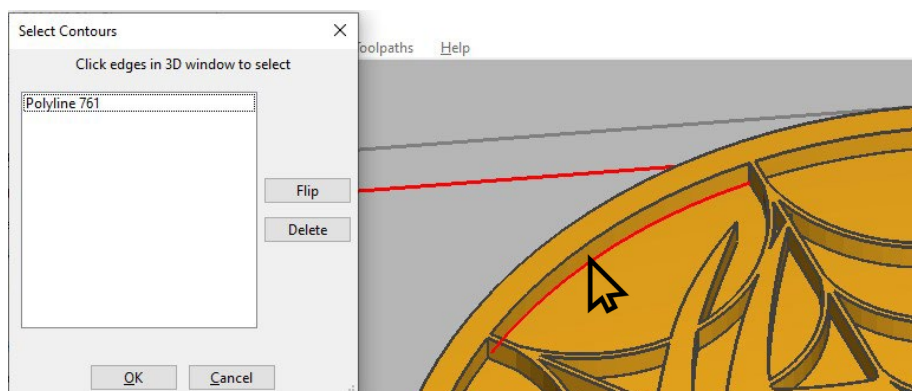
Pocket (2D)

The Pocket (2D) settings will allow you to change what settings your Pocket (2D) toolpath will have when taking action on your project. Selecting the Pocket (2D) settings will prompt you with a Pocket Toolpath pop up. All of the measurements in this setting will be in millimeters.



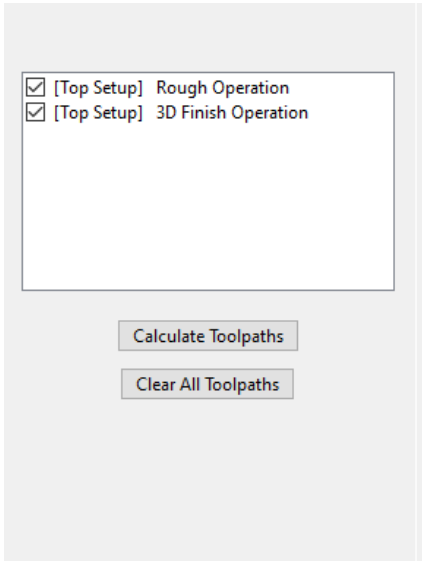
Options

The Options settings will allow you to change any other specific settings for your Pocket (2D) toolpath. The Options settings have a Polylines Selected setting with a Select Polylines button. Selecting the button will prompt a Select Contours pop up. This pop up will have a white panel where a list of polylines selected will be displayed. The pop up gives instructions to click on the edges of your project to select those polylines. You can select the polylines by hovering over any of the polylines on your project. Hovering over a polyline will be highlighted in red to display which polyline you would be selecting. Left Clicking on a polyline will display the name of the polyline in the Select Contours pop up. The pop up has two buttons to the right, a Flip and Delete button. To select these buttons you must first select a polyline. To select a polyline left click a polyline from the displayed list in the pop up. Selecting a polyline from the list will highlight the polyline. Selecting flip on a highlighted polyline will flip the polyline and will display "(Flipped)" next to the polyline name in the list. Selecting the Delete button while a polyline is highlighted will delete the selected polyline from the list. The OK button at the bottom of the Select Contours pop up will save the selected polylines to the Contour (2D) toolpath settings. The Cancel button at the bottom of the Select Contours pop up will force quit the Select Contours pop up and will not save any of the selected polylines to the Contour (2D) toolpath settings.



Simulating your CNC job

As you built a list of toolpaths it is a great idea to start to calculate and if you have the Pro version simulate those toolpaths. Taking this step will make sure that you have all of the correct settings to create your part. This process will help make sure that you have your workholding and supports for your stock in place. When simulating toolpaths you will be able to see an approximation of what your final part will resemble. This can help rule out any re-work you may have to do after sending a job that needs extra fine tuning. In the standard version you will be able to calculate a view toolpaths which can also aid in making sure there has not been a mis-typed parameter so far in the process.



Calculate toolpaths

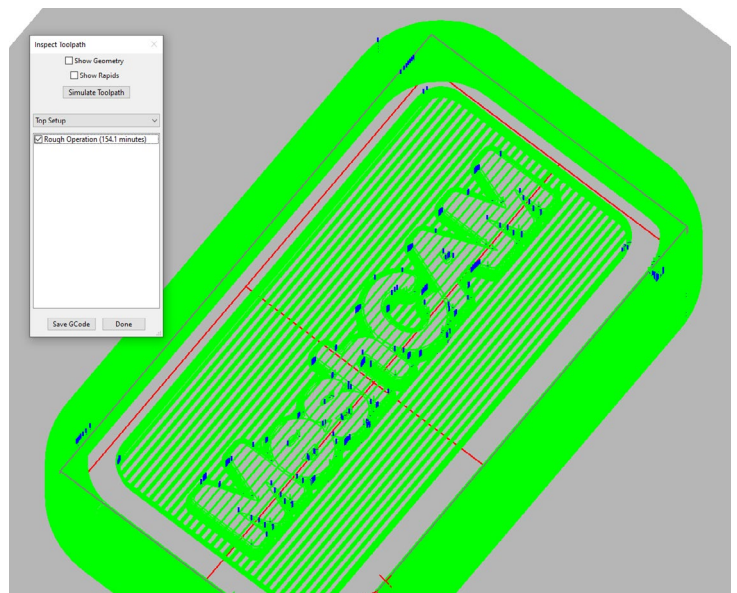
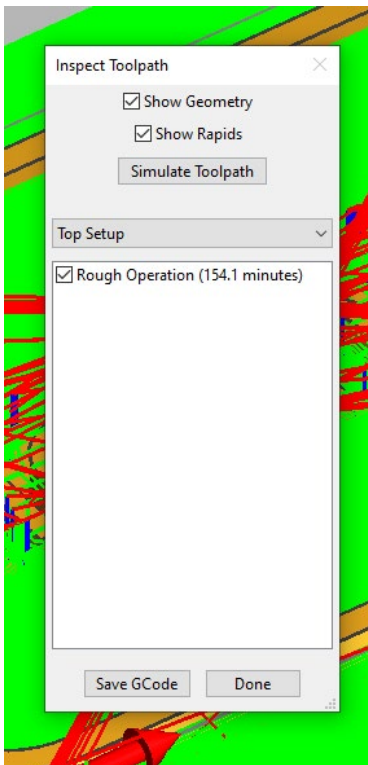
After you have created your first toolpath, it is a great idea to go ahead and calculate it. This will allow you to see the amount of time it will take to machine and also make sure that the toolpath will not crash into any workholding or fixturing that you have set in place. To calculate your toolpaths simply check all toolpaths you would like to calculate and click on calculate toolpaths.

Inspect toolpaths

Once you have calculated your toolpaths you will be able to see how much time each individual toolpath will take and toggle visibility of your model geometry, rapids, and the separate toolpaths themselves. This is all you will be able to see in the standard version of MeshCAM.

The dropdown selection that says Top Setup will allow you to flip between Top Setup and Bottom setup if you are doing a two-sided or quick-flip job.

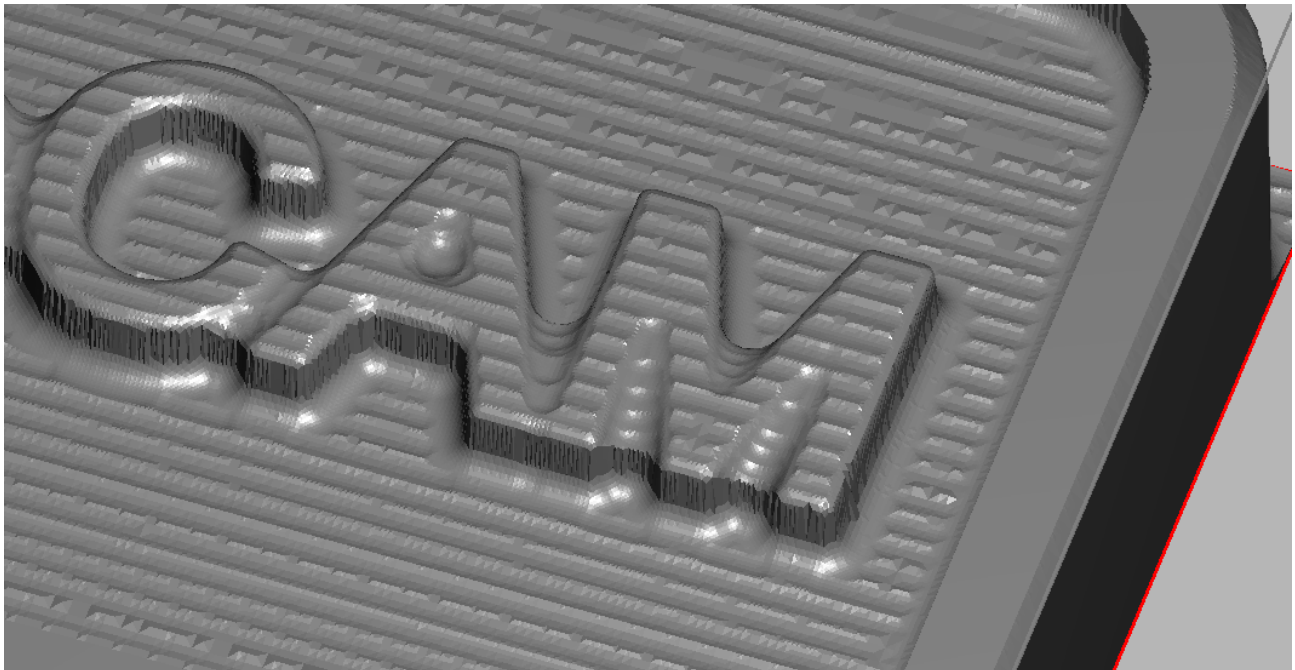
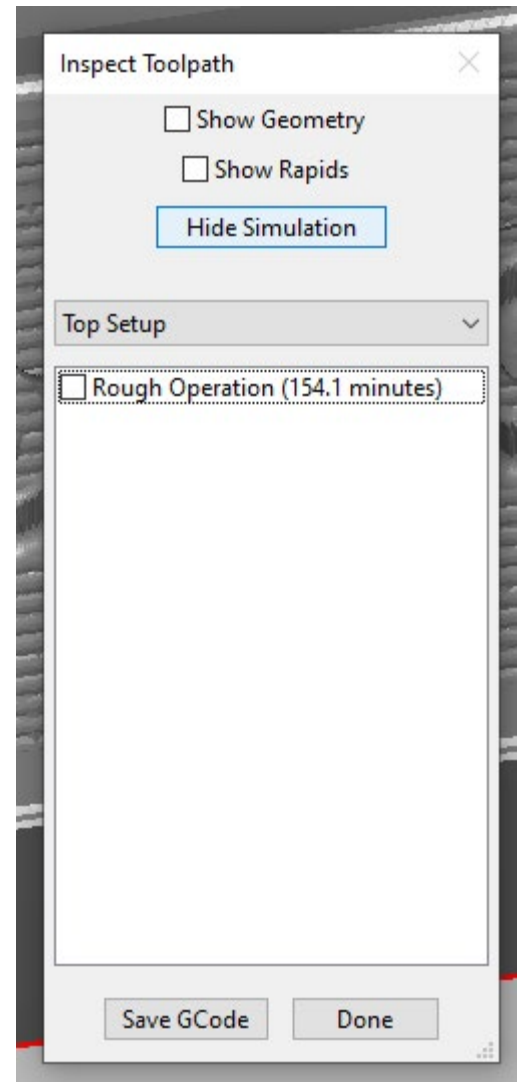
While this dialogue is enabled you can simply move it out of the way and inspect your toolpath as necessary.



Simulate toolpaths

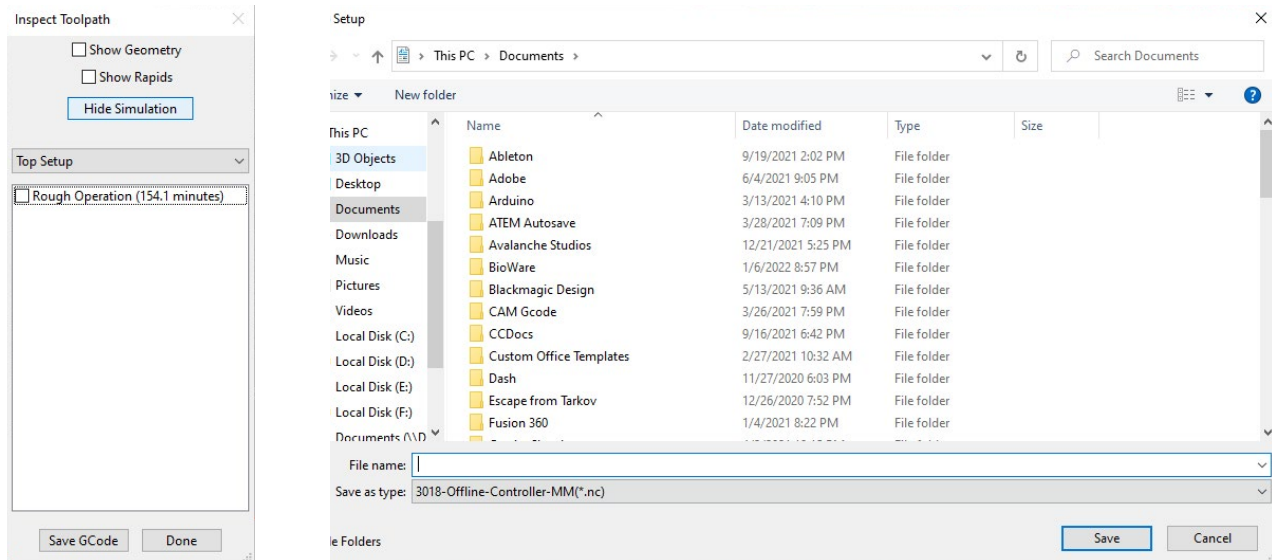
If you have purchased the Pro version of MeshCAM you can also simulate your toolpaths. To do this simply click on the simulate toolpaths button after you have checked the toolpaths you would like to simulate. This will then simulate what your finished part will look like. Please note that there may be slight imperfections in detailed areas as this is just a simulation. This operation is extremely useful to see what the result of your job will be without wasting stock and material.

Once you have simulated your toolpaths it is helpful to deselect the toolpaths as seen in this screenshot. This will allow you to zoom in and see how your part will look after machining. Note the ridges in this example. This is a great use of simulation as you can see that because a ball end mill was used, there are visible machining marks across the workpiece.



Saving your G-code

After the toolpaths have been calculated, and simulated with the pro version, it is time to save G-code that you will use with your machine. To do this select the Save GCode button. This will open an explorer or finder window to save your GCode in the location of your choice. The most important selection of this process is selecting the type of machine you are using. The Save as type selection will apply the post processor of your choice to match your machine. If you are unsure of which option to pick make sure to look at your machine's documentation for guidance.



Saving your project

Once you have saved your GCode make sure that you then save your project. This step will allow you to edit and refine your process and continue to access the project in the future.

