N2 G40 G49 G80
N3 (STOCK SIZE: X2.125 Y2.125 Z0.25)
(BEGIN TOOL LIST)
(TOOL 1 - FlatMill:0.25 - DESC: 0.2500 DIA, 2 FLUTE, CARBIDE MAT)

G-Code Editor Quick Start Guide

RhinoCAM 2025

Published: February 2025

N5 (Work Zero)
N6 G54
N7 (2 1/2 Axis Profiling (Outer))
N8 (Tool Diameter = 0.25 Length = 2.0)
N9 G20 T1 M6
N10 S10000 M3
N11 G90G0X-1.2533Y-0.3524
N12 G43Z0.25H1
N13 M8
N14 G0
N15 G1Z-0.25 F97.5
N16 G41 G1 X-1.2533 Y-0.3524 D1 F146.25
N17 G1X-1.1678Y-0.1175
N18 G1X-1.125Y0.
N19 G1Y1. F195.
N20 G1Y1.25
N21 G17
N22 G03X-1.25Y1.125I-0.125J0.
N23 G1X-1.
N24 G1X1.
N25 G1X1.25
N26 G03X1.125Y1.25I0.J0.125
N27 G1Y1.
N28 G1Y-1.

MecSoft Corpotation
© Copyright 1998-2025

Table of Contents

© MecSoft Corporation

© MecSoft Corporation

Quick Start

G-CODE Editor Module 2025

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

Quick Start Guides for each RhinoCAM module are available in both PDF and Video format. Refer to the following information to access these resources:

How to Access the Quick Start Guide Documents

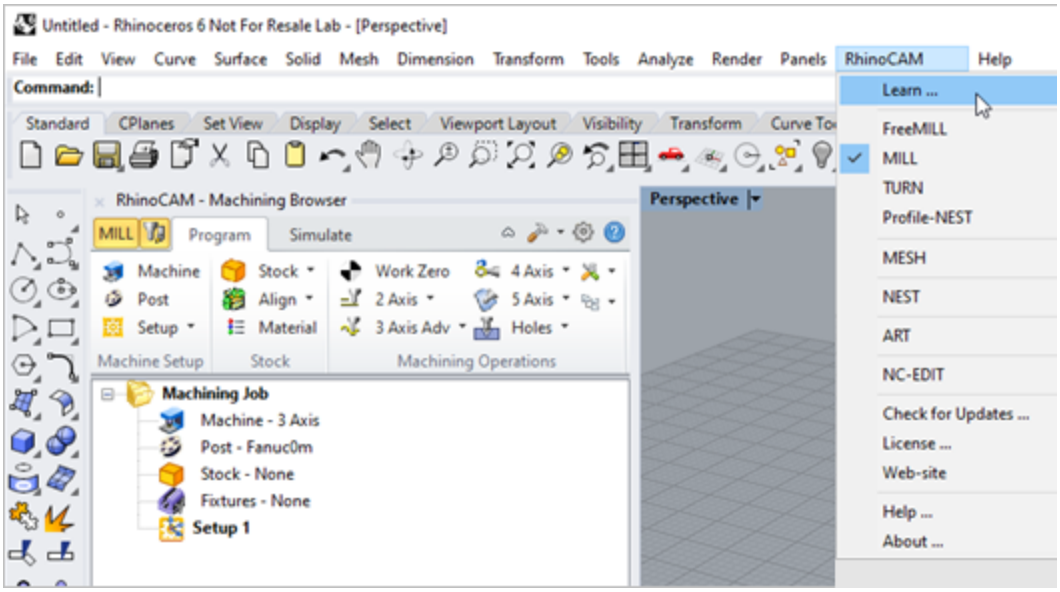
To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

You will find:

- Quick Start Guides
- What's New documents
- Online Help links

The Quick Start Guides will help you step through an example tutorial which will illustrate how to use the module. To access the Learning Resources dialog:

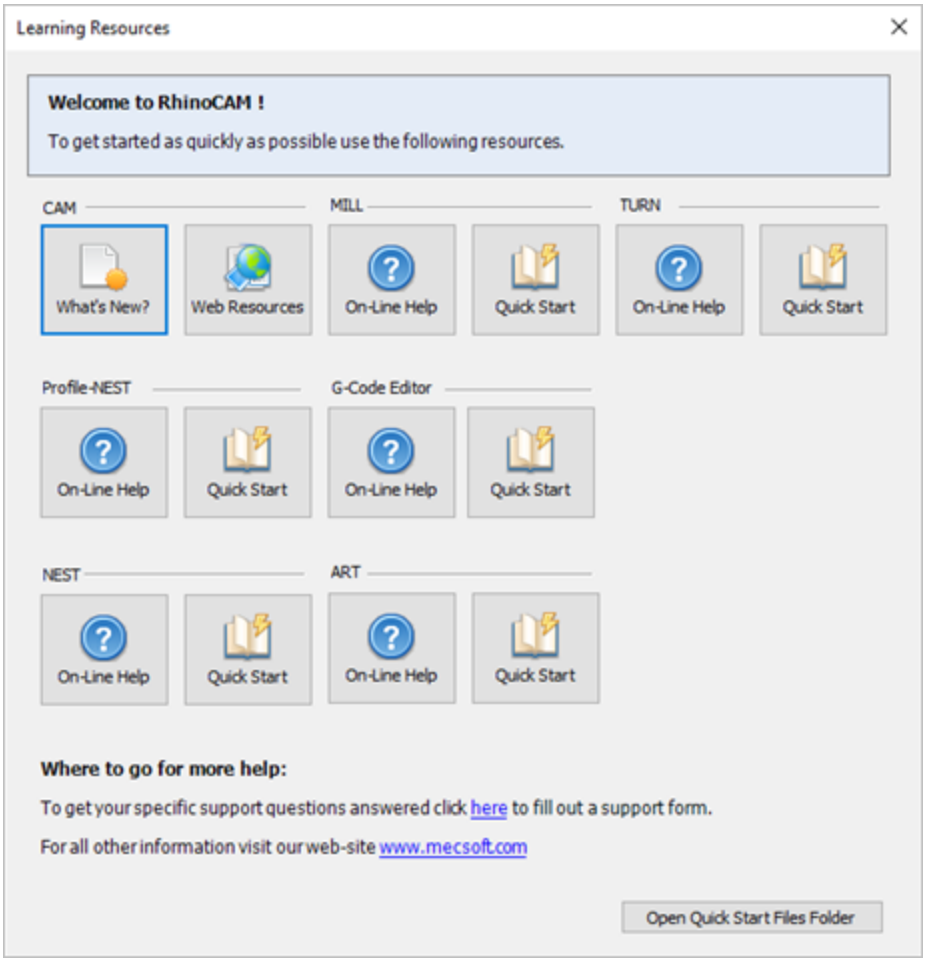
1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in RhinoCAM

2. Select a document from the Learning Resources dialog to get started using the module of your choice.

You can also select the Open Quick Start Files Folder button located at the bottom of the dialog to open the Quick Start folder where the source files (start and completed versions) are located.



Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide

The 2025 RhinoCAM Resource Guide!

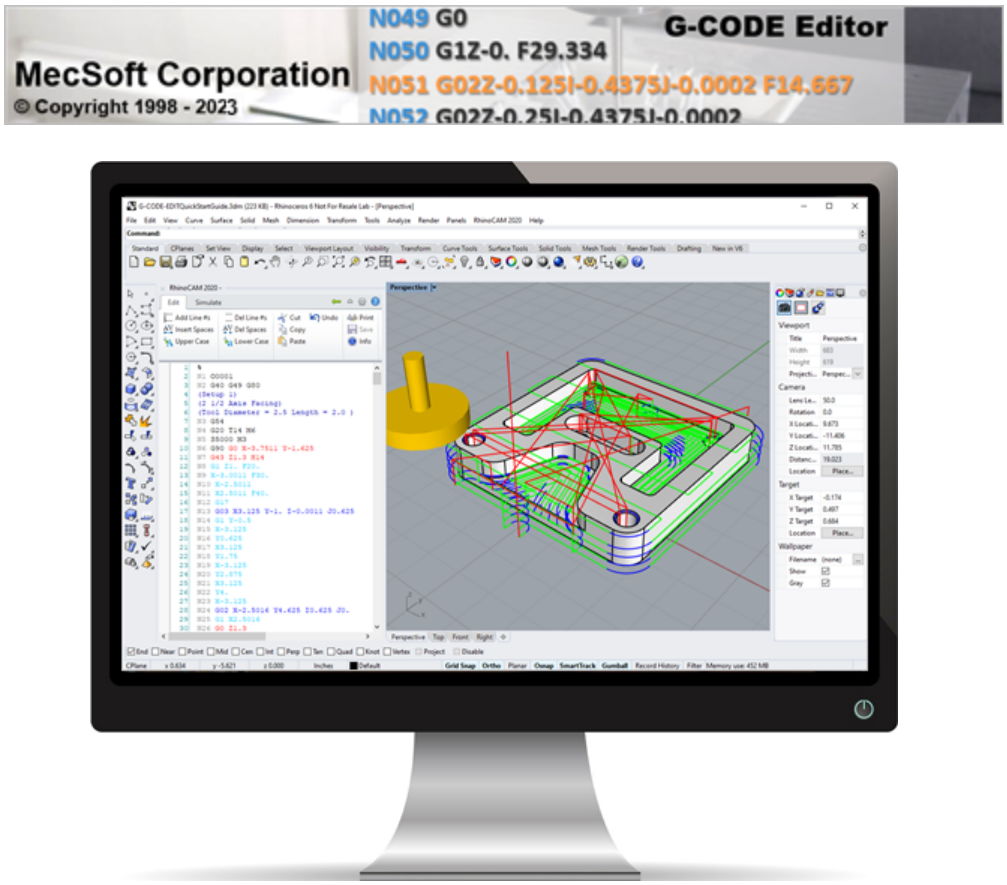
18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

About this Guide



3.1 Useful Tips

Here are some useful tips that will help you use this guide effectively.

1. For purposes of brevity, Rhino refers to both Rhinoceros 6 or Rhino 7.
2. Copy the tutorial files to a location other than the installation folder to make sure you have read/write privileges to the files.
3. Once you start working with the tutorial file, save your work periodically!
4. Don't stress out too much if you are having trouble with the tutorial. Call us or send us email and we can help you out.
5. Most of all have fun!

3.2 About this Module

G-CODE Editor is a companion module that runs inside of . G-CODE Editor assists users with editing g-code files created with RhinoCAM or from other systems that produce ISO standard g-code files. You can also define tool libraries, tool cribs to perform tool motion and cut material simulations.

3.3 Using this Guide

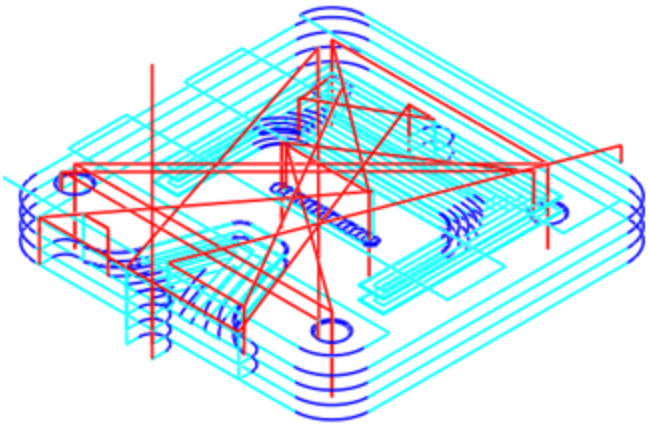
If you have installed RhinoCAM successfully on your computer and are now looking at the blank screen of Rhino and wondering what to do next, this is the guide for you. This guide will explain how to get started in using the RhinoCAM G-CODE EDITOR module to edit a simple g-code file through an example.

This guide has associated Rhino files that you can find located in the QuickStart folder under the installation folder of RhinoCAM. These files are shown and listed below.

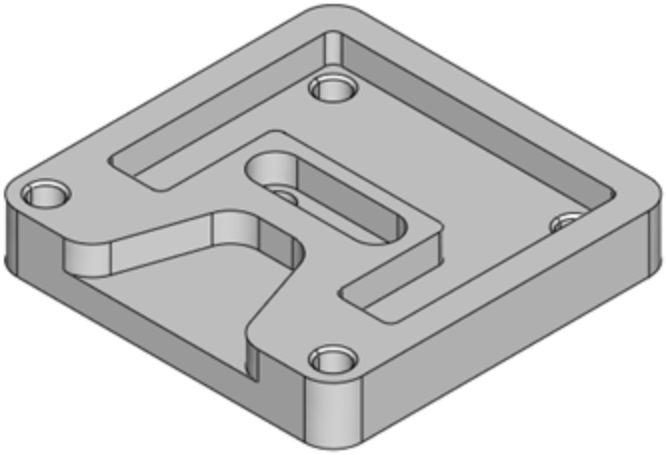
List of Included Source Files

- Source G-Code File:
  G-CODE-EDITQuickStartGuide.nc
- Source Part File:
  G-CODE-EDITQuickStartGuide.3dm
- Source Tool Library File:
  G-CODE-EDITQuickStartGuide.csv
- Source Part File (Completed):
  G-CODE-EDITQuickStartGuide_Completed.3dm
- **Note**: Source part files are not required to view or edit G-Code files in the G-Code Editor module.

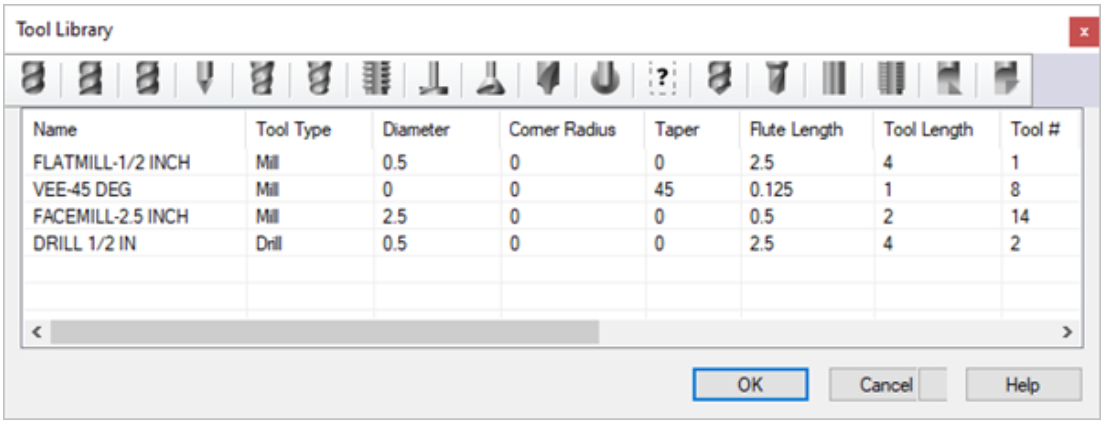
Included Source Files



Source G-Code File:
G-CODE-EDITQuickStartGuide.nc

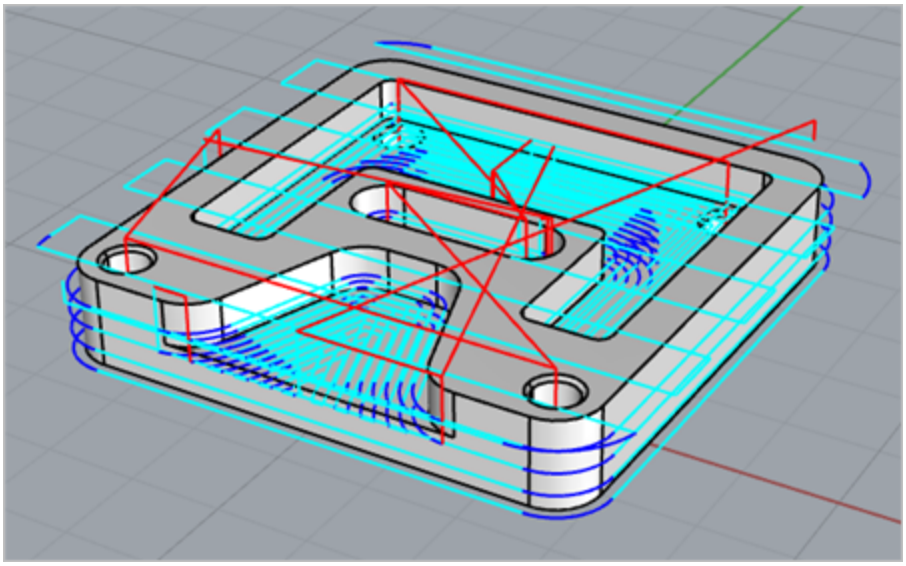


Source Part File:
G-CODE-EDITQuickStartGuide.3dm



Source Tool Library File:
G-CODE-EDITQuickStartGuide.csv

© MecSoft Corporation



Source Part File (Completed):
G-CODE-EDITQuickStartGuide_Completed.3dm

3.4 Watch the Video!

Want to see a video demonstration of this quick start guide? Just click on the play list below and watch the G-Code Editor Quick Start Guide video.

Here is a link to the complete 2023 Video Play List

© MecSoft Corporation

Getting Ready

4.1 Running RhinoCAM

Locate the Rhinoceros 6 (or Rhinoceros 7) shortcut on your desktop and double click to launch the application.

Alternatively you can also click on the Windows Start button and select All Programs. Go to the program group containing Rhinoceros 6 (or Rhinoceros 7). (The name of this program group will usually be called Rhinoceros, unless you specified otherwise during setup.)

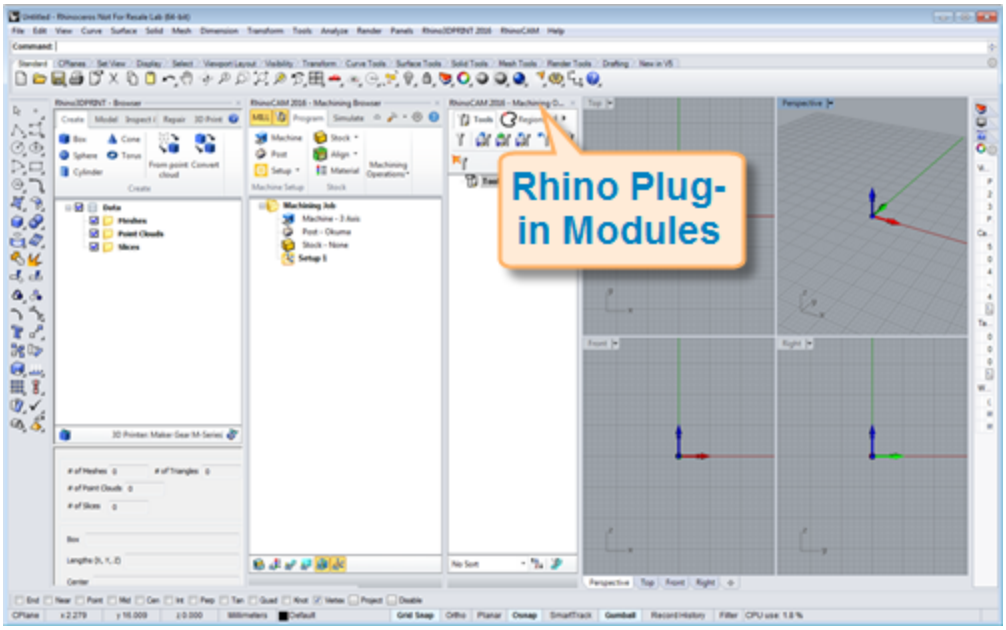
Once you locate the program group, select it and then select Rhinoceros to launch the application.

If the installation was successful, upon launching of Rhino you should observe a menu entry called RhinoCAM 2023 in the main menu bar of Rhino.

If you do not see this menu entry then please check the On Line Help document of the product (found in the installation folder) for help with trouble shooting the installation.

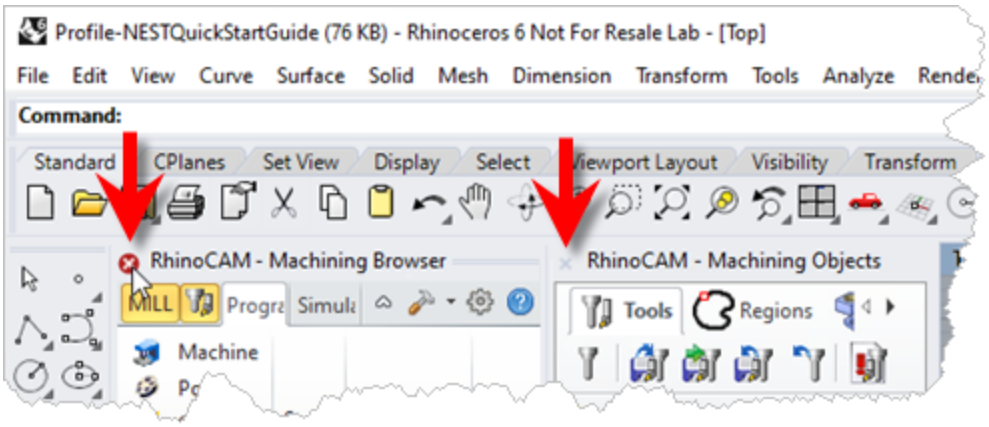
4.2 About the RhinoCAM Display

Before we begin, let's talk a bit about the RhinoCAM display. When you run RhinoCAM for the very first time, your screen may look this.

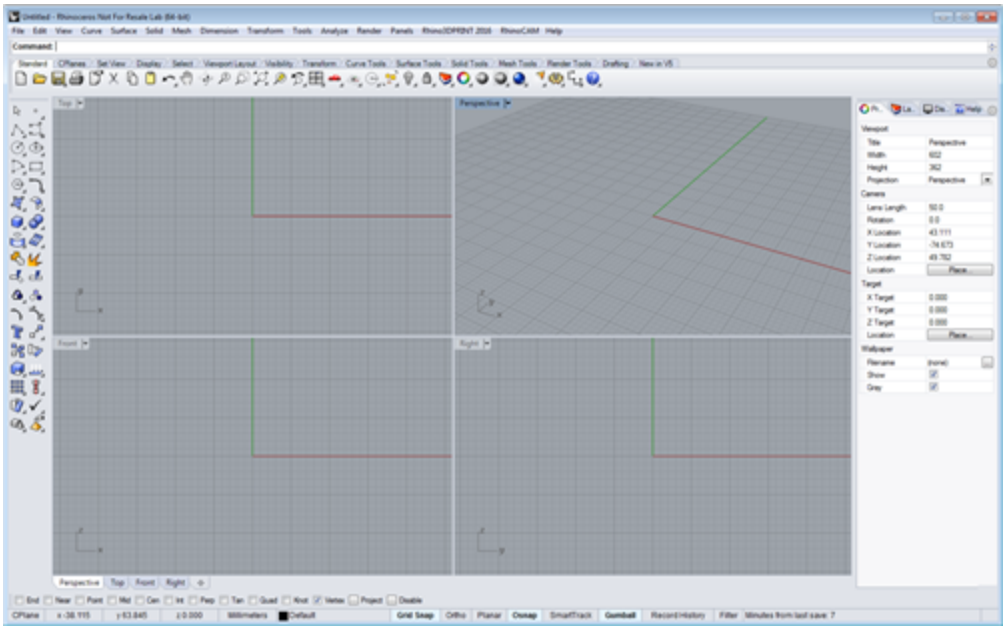


These windows on the left belong to plug-in modules that are currently loaded. For now, let's close all of them.

© MecSoft Corporation



With all plug-in modules closed your screen will look like this:

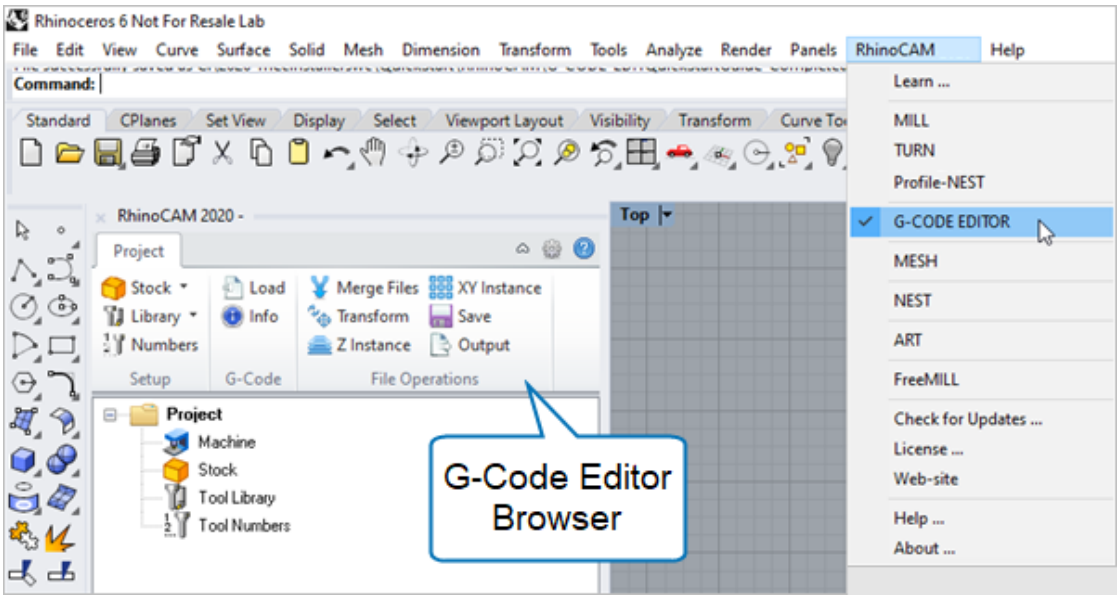


4.3 Load the G-Code Edit Module

Now, let's begin by launching the RhinoCAM G-CODE EDITOR module.

1. From the Rhino main menu bar, drop down the RhinoCAM 2023 main menu and select G-CODE EDITOR.

© MecSoft Corporation



2. Docked on the left you will see the G-CODE Editor Browser. You can move or re-size the browser as desired.

4.4 Load the G-Code Files

Your G-Code files can be created from RhinoCAM or you can load G-Code files created from any other CAM system that supports ISO standard g-code. For this guide we will be loading a part file and a G-Code file created from RhinoCAM.

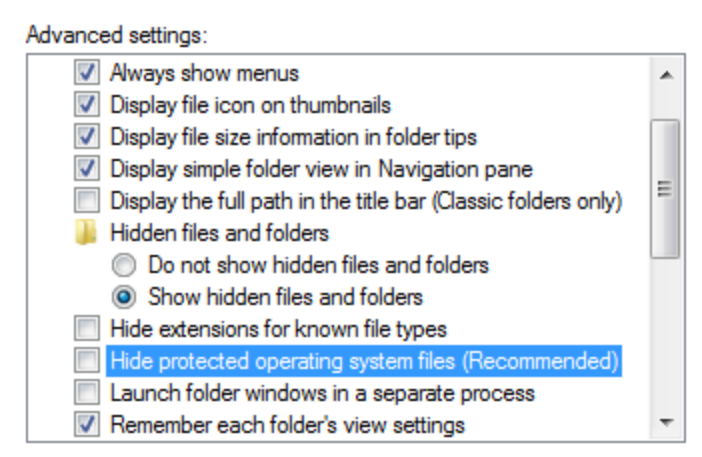
> You DO NOT need to have a part geometry file loaded in order to edit G-Code files! It is loaded in this guide because this is the part file the G-Code was posted from. You can load any standard ISO G-Code file into the G-CODE Editor and perform edits, tool simulations without a part file loaded!

Load the Part File

1. From the Rhino Main Menu, select File / Open.
2. From the Open dialog box, select the G-CODE-EDITQuickStartGuide.3dm file from the C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino x.x\QuickStart\ folder. As mentioned before, it is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges to modify the part.

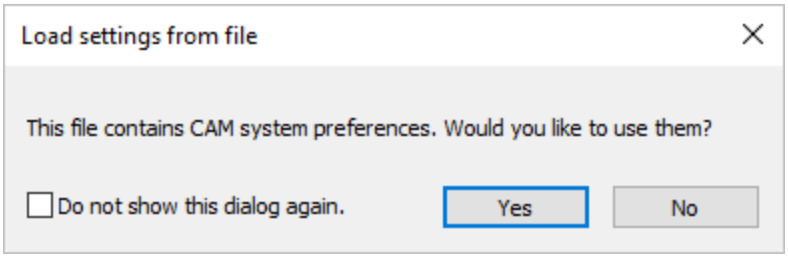
> By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For Windows 8 users: Go to Control Panel > Appearance and Personalization > Folder Options.
   For Windows10 users: Go to Control Panel > Appearance and Personalization > File Explorer Options.
2. Select View tab and under advanced settings select Show Hidden files and folders, clear the check boxes for:
   - Hide extensions for known file types
   - Hide protected operating system files (Recommended)



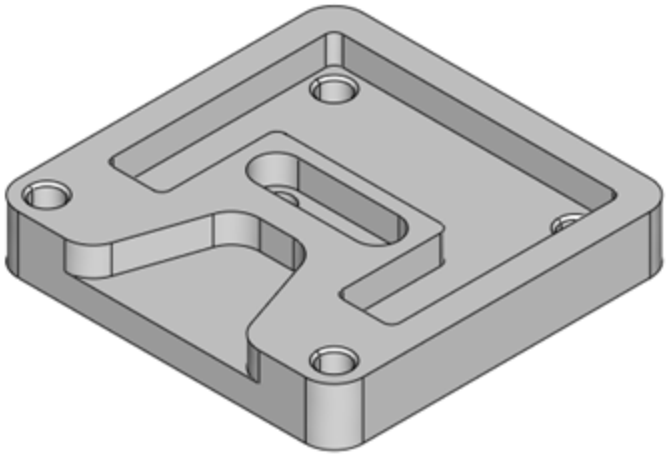
3. Click Apply and OK.

When the Load Settings from File dialog appears, pick No for this file. In the future you may have older files whose CAM System Preferences you wish to use so leave the box *Do not display dialog again* unchecked for now.



The part appears as shown below

© MecSoft Corporation

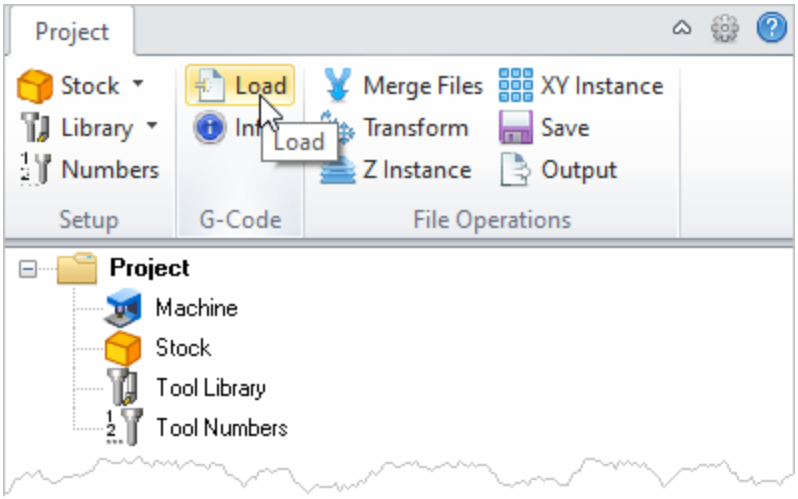


Source Part File:
G-CODE-EDITQuickStartGuide.3dm

3. From the Rhino display, double-left-click on the Perspective View tab to maximize it.

Load the G-Code File

1. From the Project tab of the G-CODE Editor Browser, select Load.

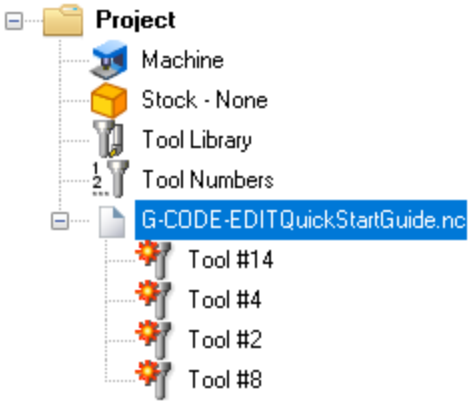


2. From the Open dialog box, select the G-CODE-EDITQuickStartGuide.nc file from the C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino x.x\QuickStart\ folder. As mentioned before, it is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges to modify the part.

   The G-Code file is loaded into the Project tree of the G-Code Editor Browser.

© MecSoft Corporation

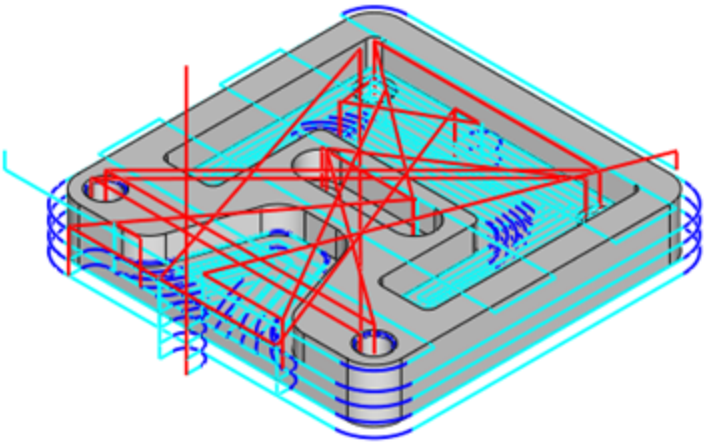
Notice that the G-Code file name appears in the Project tree with each tool referenced by the file listed below it.



Why are the tools flagged? The flag on the tool icons indicate that the tool numbers referenced within the loaded G-Code file could not be found in the Tool Crib. These flags do not need to be resolved in order to perform edits to your G-Code file!

Refer to the Load Your Tool Library section for more information.

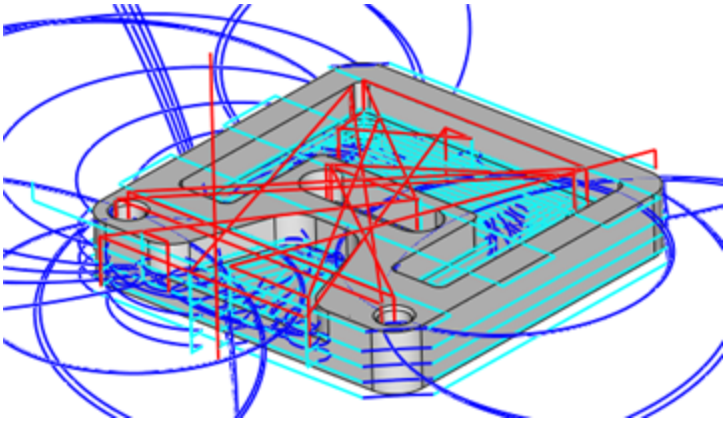
3. The backplot of the G-Code file is also displayed in the graphics screen coincident with the part geometry.

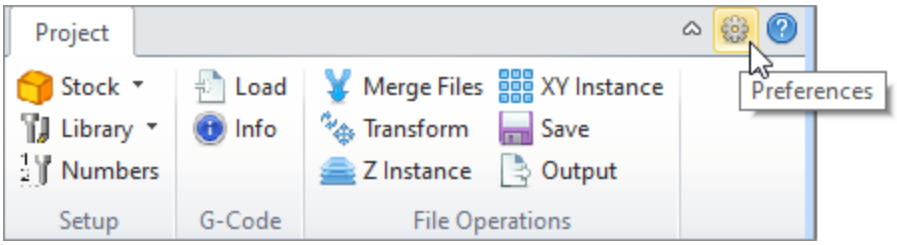


As we mentioned previously, you DO NOT need to have a part geometry file loaded in order to edit G-Code files! It is loaded in this guide because this is the part file the G-Code was posted from. You can load any standard ISO G-Code file into the G-CODE Editor and perform edits, tool motions and simulations without a part file loaded!

© MecSoft Corporation

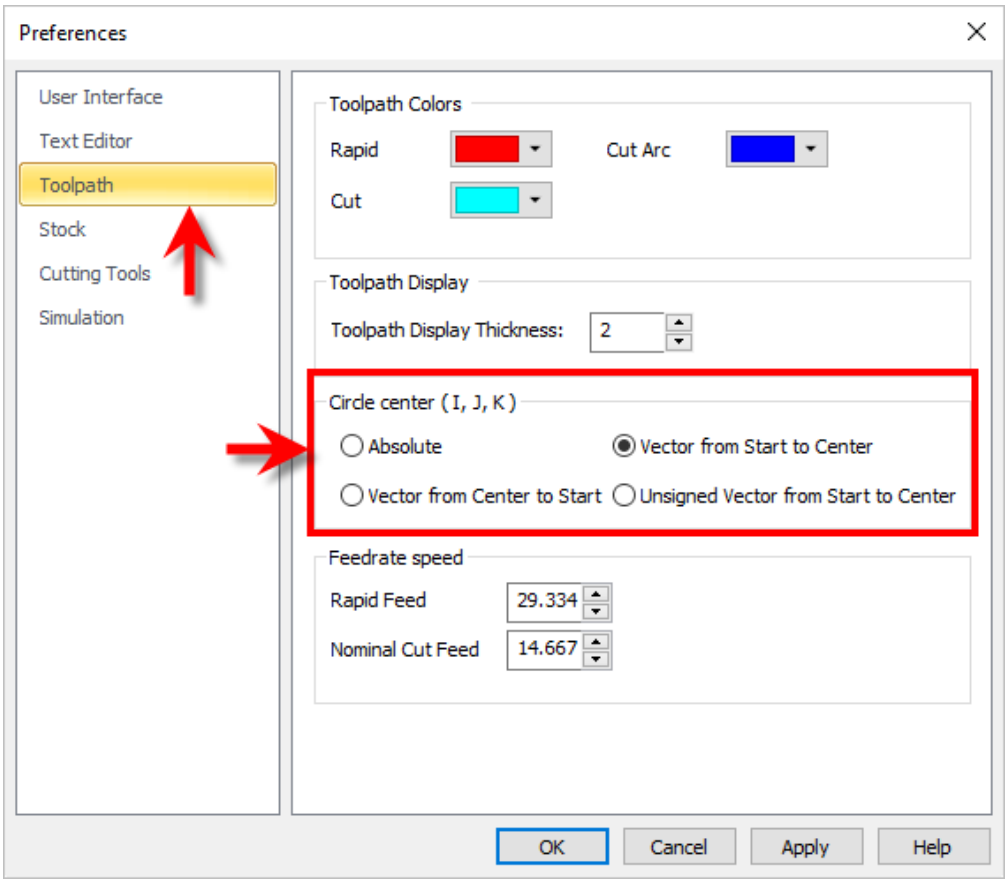
4. If the backplot of your G-Code file displays arc motions incorrectly as shown below, select the Preferences icon.



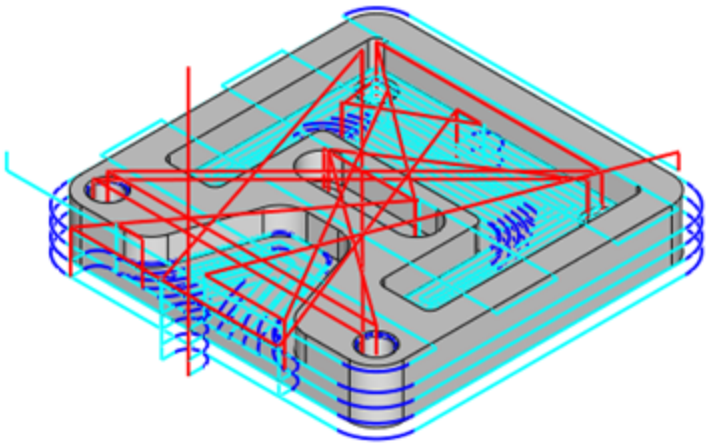


5. From the Preferences dialog select Toolpath from the left. Here you can set the Circle center (I,J,K) format to match the G-Code file that you load. For example if the CAM system that generated the G-Code has arc centers formatted using Vector from Start to Center, the backplot of the G-Code file may show arcs being displayed incorrectly. If this occurs you can set the same option here and pick OK or Apply and see that the arcs display properly.

© MecSoft Corporation



8. Your part and G-Code backplot should now look like this:



4.5 G-Code Editing Strategy

The G-Code editing strategy you employ at any given time may be dictated by your preferences or how your CNC machine controller expects the G-Code files to be formatted. The majority of the

© MecSoft Corporation

format adjustments can be addressed within the post definition file using the PPG (Post-Processor Generator) included with RhinoCAM.

However, there will be times when you may want or need to make manual edits to your G-Code files without having to revert back to the CAM system. Our editing strategy for this guide is outlines below.

G-Code Edit Strategy

We will follow the steps listed below for editing our G-Code file:

1. Load the G-Code file into the G-Code Editor Browser.
2. Perform some quick formatting edits and then output and save the G-Code file.
3. Load a Tool Library and define a Tool Crib in preparation for G-Code simulations.
4. Perform Tool Motion Simulations.
5. Define a Stock model and perform Cut Material Simulations.
6. Perform addition G-Code edits, estimate machine time and then save our changes to the G-Code file.

© MecSoft Corporation

Edit the G-Code File

5.1 How Tools are Used

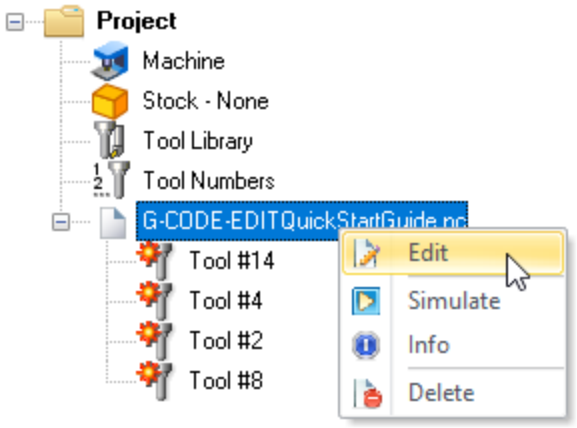
G-CODE Editor allows you to load a Tool Library and setup a Tool Crib. This allows you to match the tools that are referenced by the G-Code file. The Tool Library file (*.csv) can be generated and saved from within the MILL module while the part file with the CAM toolpaths is loaded. The Tool Crib is defined from within G-CODE Editor.

> You DO NOT need to have a Tool Library or a Tool Crib loaded in order to edit G-Code files! You can load and edit any standard ISO G-Code file with the G-CODE Editor.

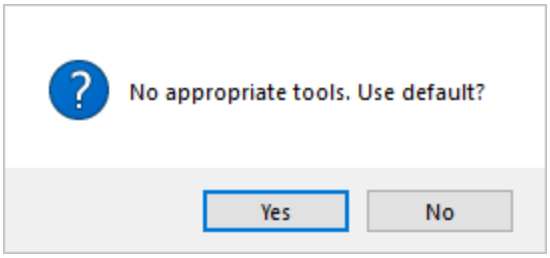
5.2 Add Character Spaces

You can make edits to your G-Code file as soon as it is loaded into the editor. As mention previously, you do not need a Tool Library or Tool Crib defined in order to edit the G-Code file. In this step we will add character spaces to the G-Code file.

1. Right-click on the selected G-Code file located in the Project tree and select Edit.



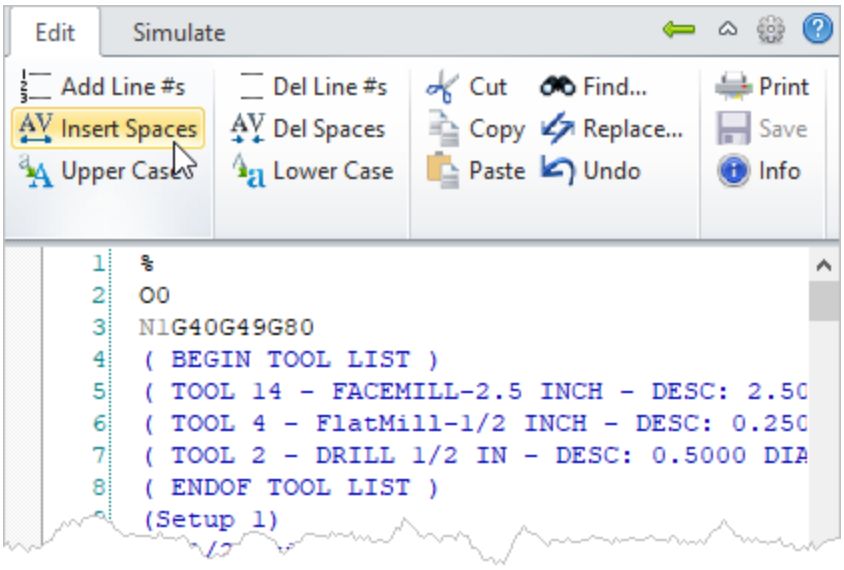
2. Pick Yes from the message that is displayed. Because no tools are defined yet, a default tool is used and displayed in the graphics window.



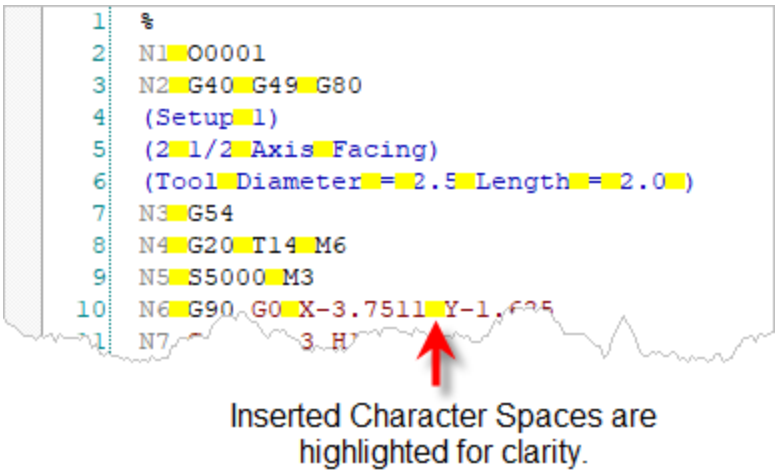
3. You will see that the Project tab is replaced with Edit & Simulate tabs and the G-Code file is loaded into the editor. The Edit tab contains all of the edit commands available in the current update of RhinoCAM.

© MecSoft Corporation

4. From the Edit tab, select Insert Spaces.

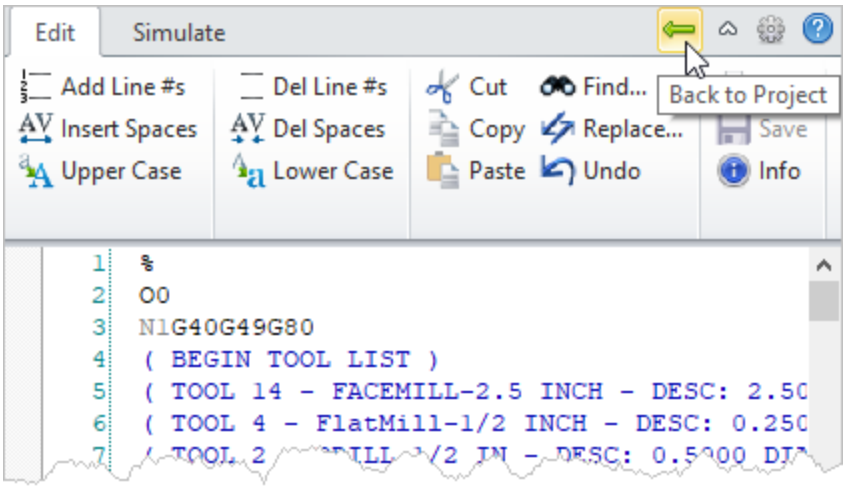


5. Character spaces are inserted into the G-Code file as shown.

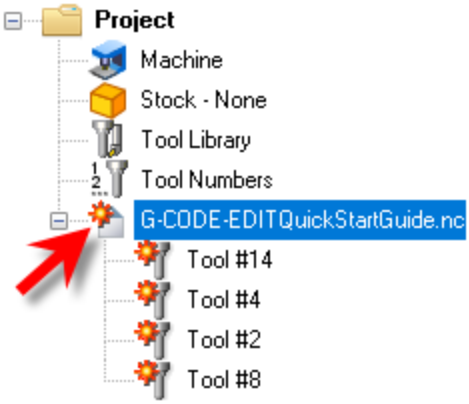


Inserted Character Spaces are highlighted for clarity.

6. When you want to return to the Project tab just select the icon shown below.



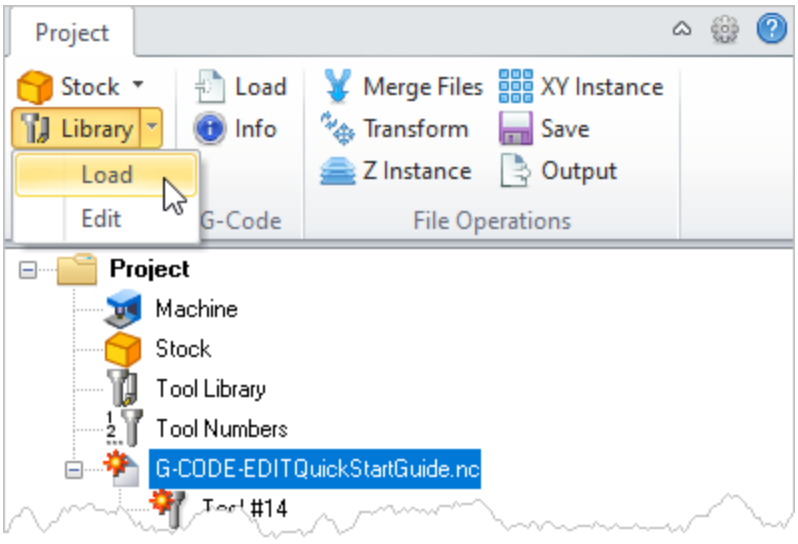
7. You see now that the G-Code file is also flagged. This indicates that an edit was made and that the file was not saved. We will make some additional edits so let's continue.



5.3 Load Your Tool Library

In this step we will load a Tool Library containing the tools that are referenced from the G-Code file. The Tool Library file for this guide was generated from the MILL module and saved in the QuickStart folder of your RhinoCAM install path.

1. From the Project tab select Library and then Load from the menu.

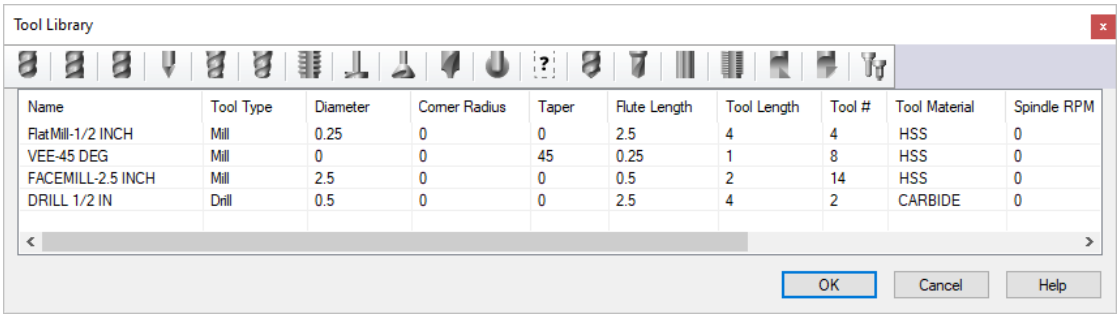


2. From the Open dialog box, select the G-CODE-EDITQuickStartGuide.csv file from the C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino x.x\QuickStart\ folder and then pick OK to load the file.

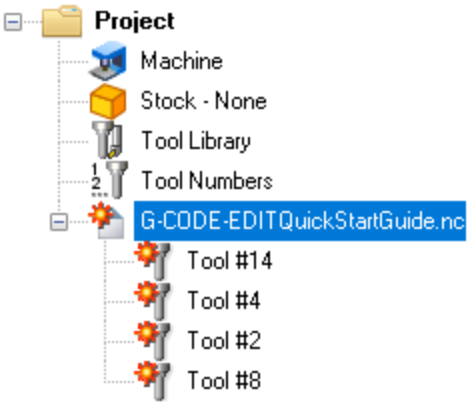
   As mentioned previously, it is advisable to make a copy of this file at a suitable alternative folder so that you have write privileges to modify it if needed.

   > The MILL module supports two types of Tool Library files (*.vkb and *.csv) G-CODE Editor requires the *.csv file format.

3. The Tool Library is then displayed allowing you to preview in the Tool information.



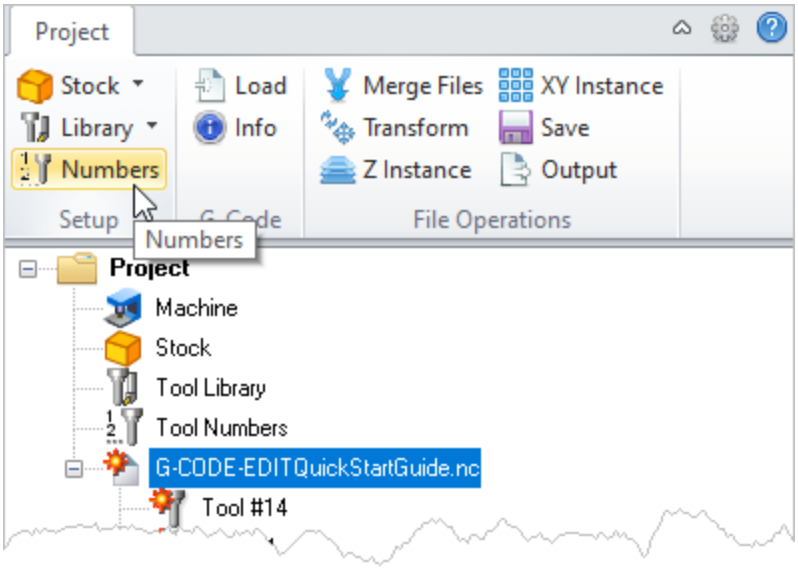
4. Now pick OK to close the Tool Library dialog. You will notice that the tools listed in the Project tree under the G-Code file and still flagged. That's because we need to setup the Tool Crib.



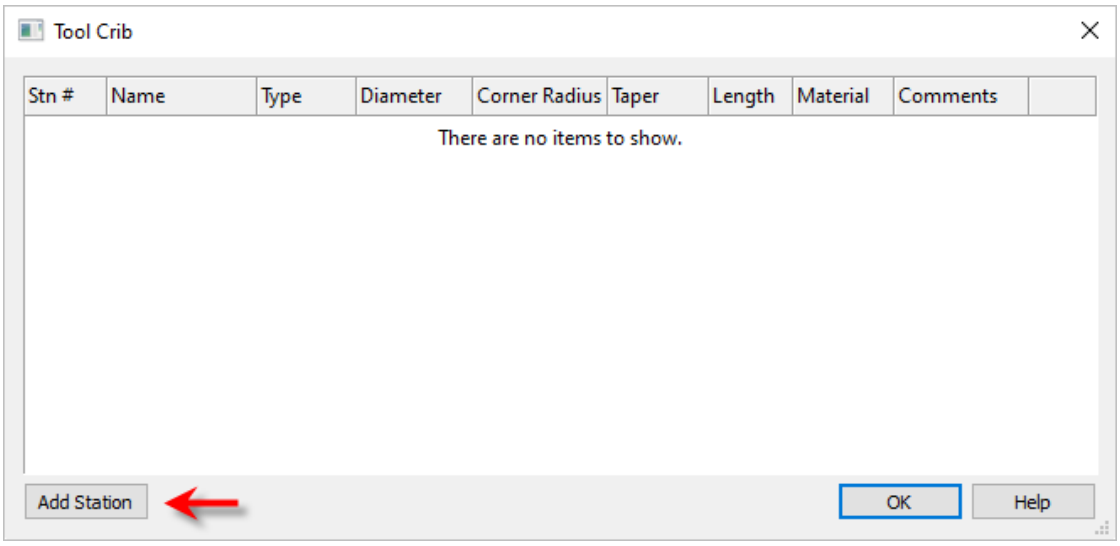
5.4 Define Your Tool Crib

In this step we will define our Tool Crib. The Tool Crib allows you to select the Tools from your Tool Library that are listed in the G-Code file. The reason for the Tool Crib is to two fold, **(1)** it mimics how tools are defined and selected for your CNC machine and **(2)** because a Tool Library can have multiple tools with the same Tool# , the Tool Crib allows you to define which tool to use for the current G-Code file(s).

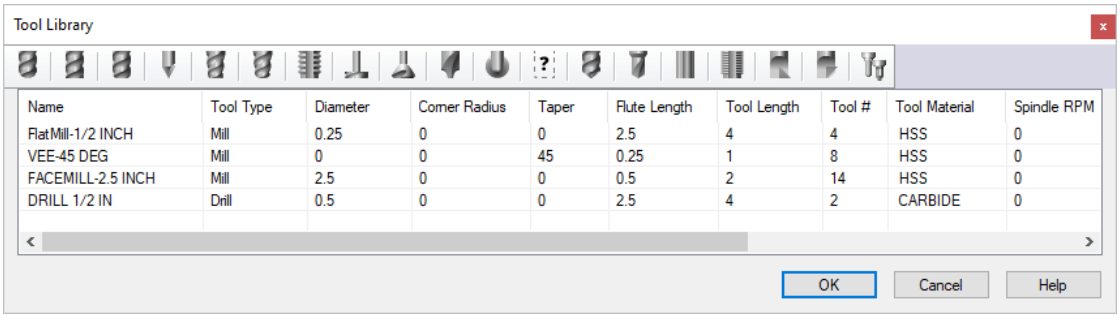
1. From the Project tab select Numbers to display the Tool Crib dialog.



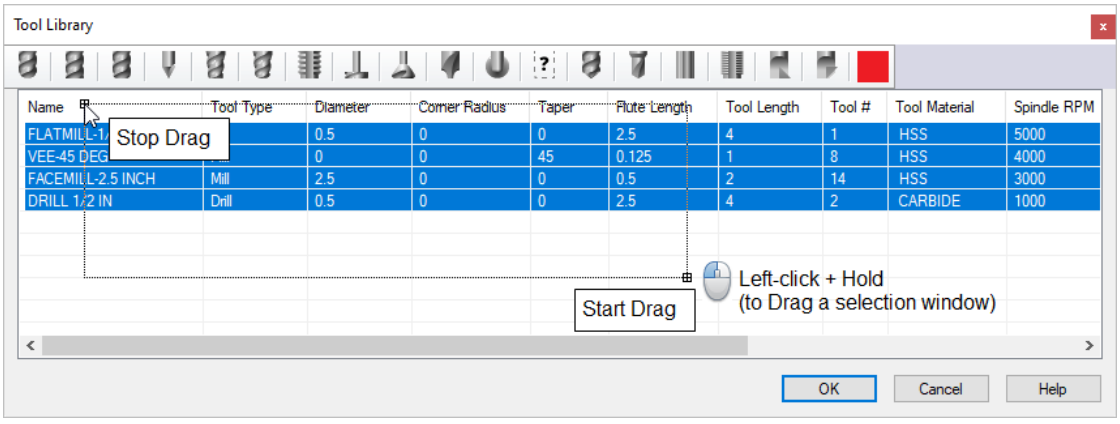
1. From the Tool Crib dialog select Add Station.



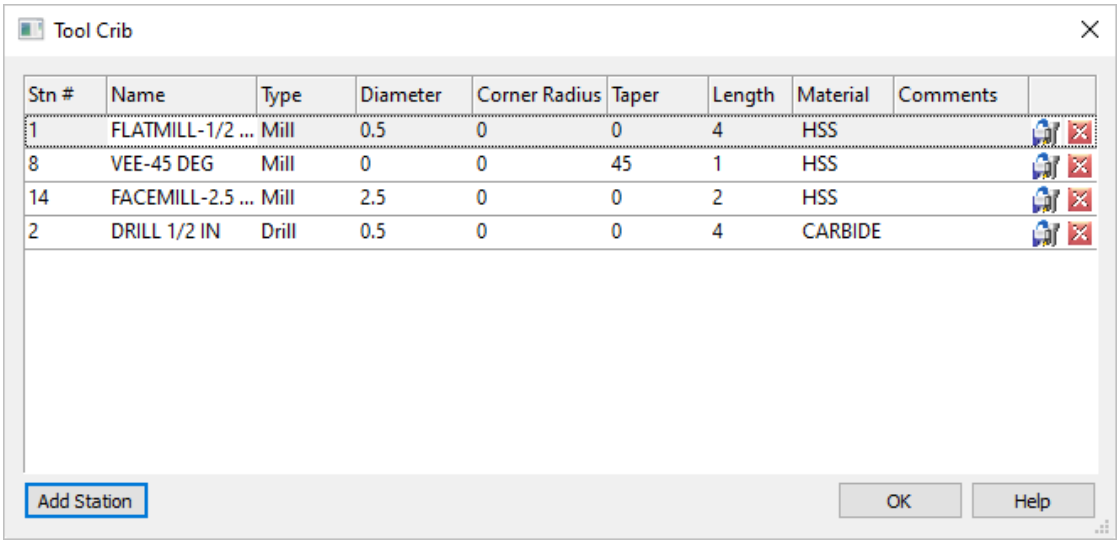
The Tool Library dialog will display.



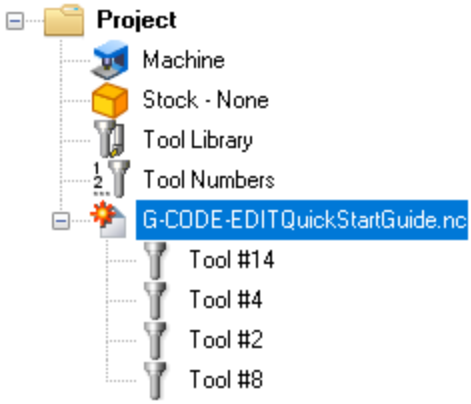
2. Our G-Code file references ALL of the tools in this library so you can drag a selection window encompassing all of the tool records until they highlight.



3. Now pick OK from the Tool Library dialog and the Tool Crib dialog will reappear with the tools listed. This will be the Tool Crib for this guide. Now pick OK from the Tool Crib dialog to close it.

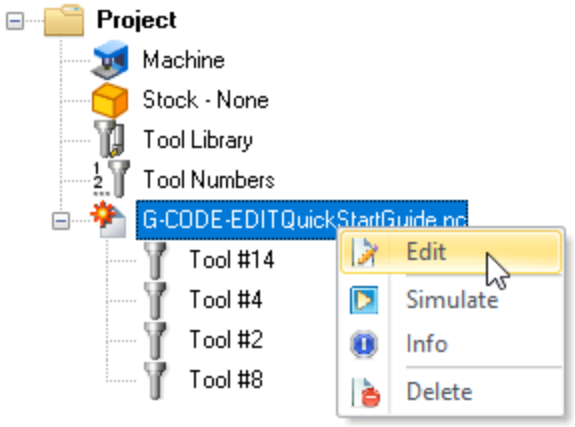


4. Now you will notice in the Project tab that all of the flags on each tool listed under the G-Code file are gone! This means that each tool was located in the Tool Crib.

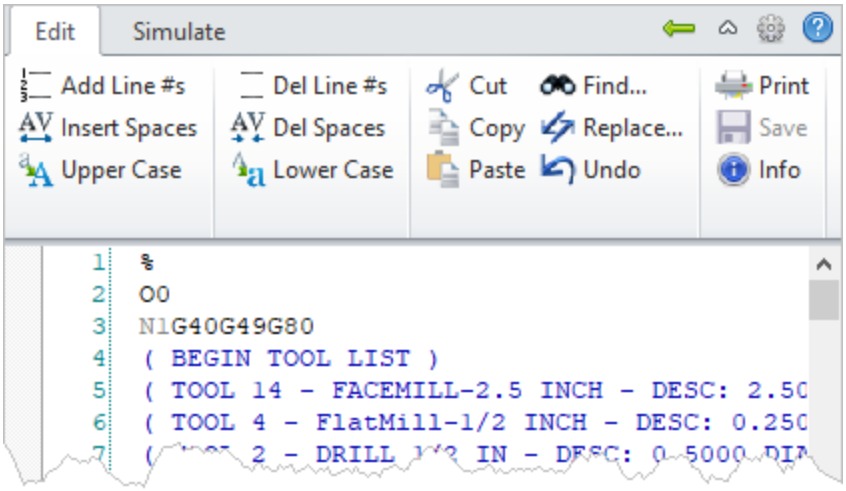


5.5 Add Line Numbers

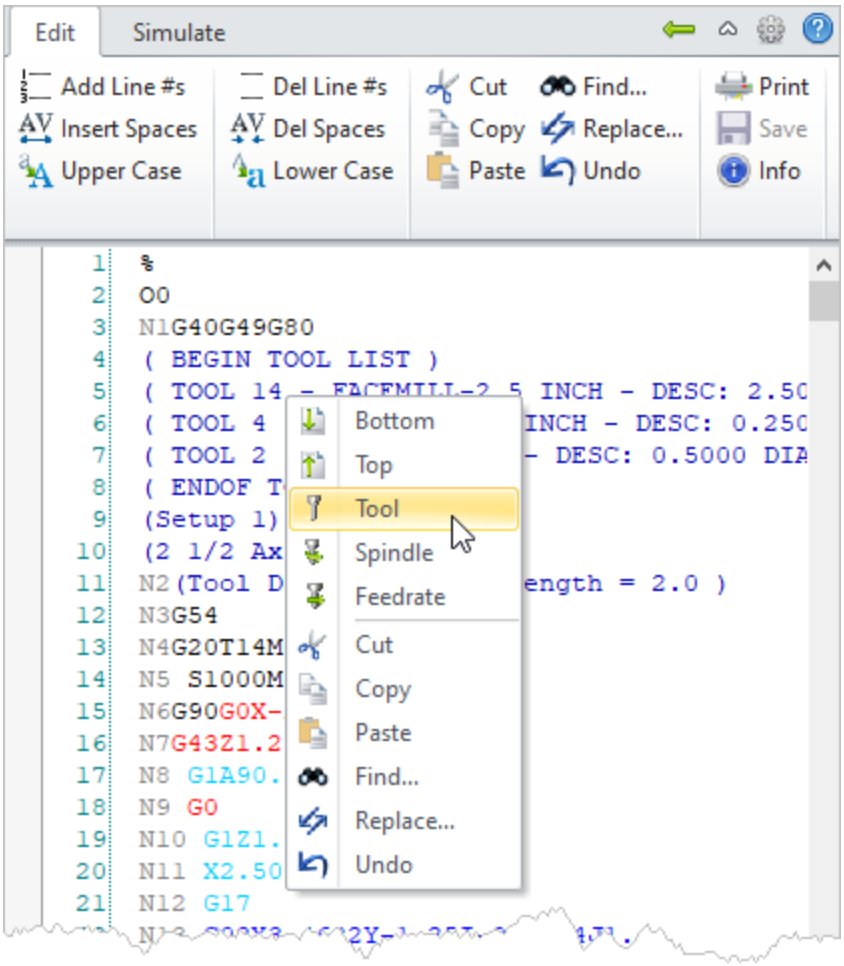
1. Right-click on the selected G-Code file located in the Project tree and select Edit.



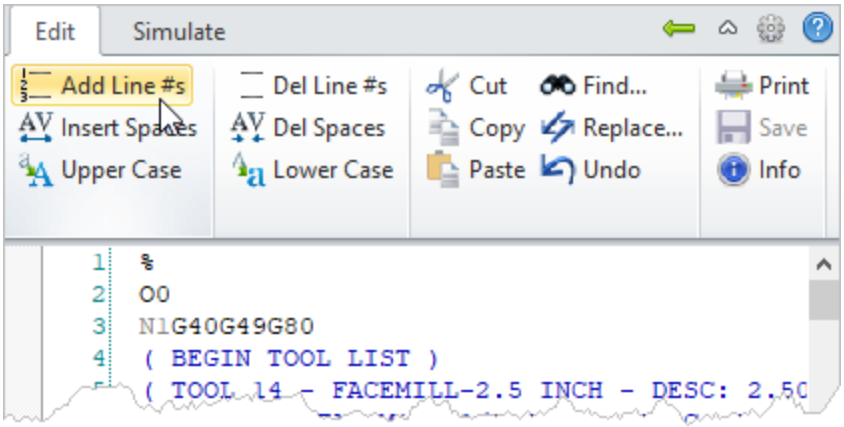
2. You will see that the Project tab is replaced with Edit & Simulate tabs and the G-Code file is loaded into the editor as shown below.



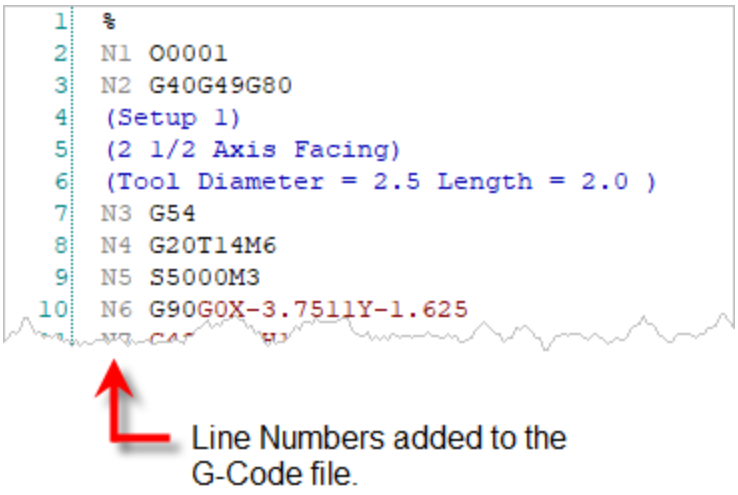
3. You can navigate quickly thru the G-Code by accessing the right-click menu shown below. The upper portion of the menu provides *go to* navigation commands. For example selecting Tool will move the cursor to the next line containing a Tool# call. You can also move quickly to the Top and Bottom of the file and to the next Feedrate and Spindle codes.



4. You will notice that line numbers are displayed in the left side column of the G-Code Editor. These are for reference only and are not in the G-Code file. To add Line Numbers to your G- Code file, select Add Line #s from the Edit tab.



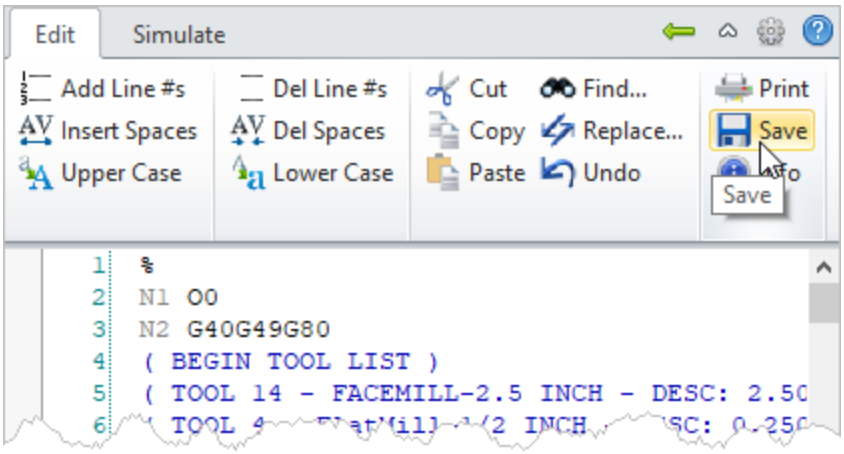
5. Line numbers are appear in the G-Code file as shown.



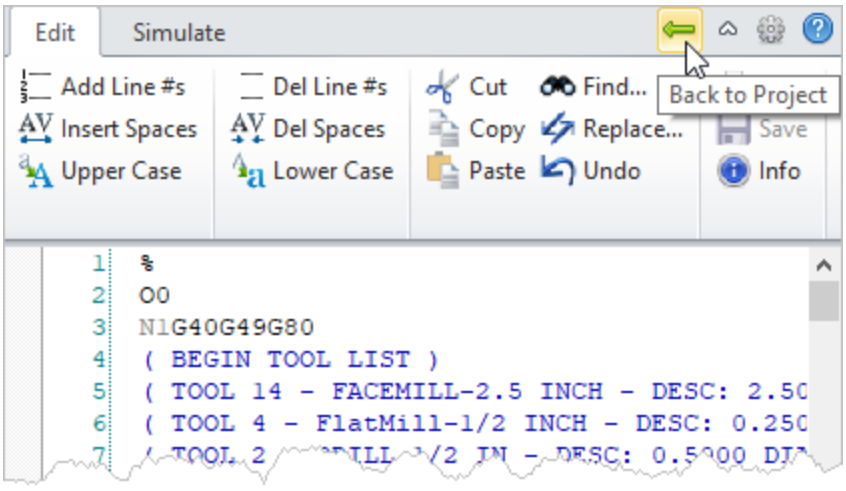
5.6 Saving Changes

Use the following techniques to save your G-Code files.

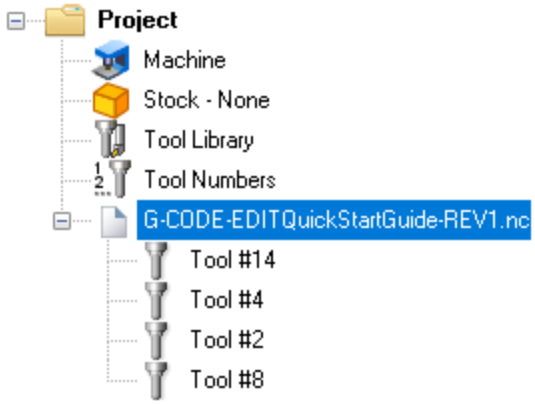
1. To save and write over your original G-Code file, select Save from the Edit tab.

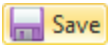


2. Pick OK to save over the original G-Code file. If you do not want to overwrite your original file, select Save As from the Save menu and enter a different file name and pick OK.
3. Now return back to the Project tab.

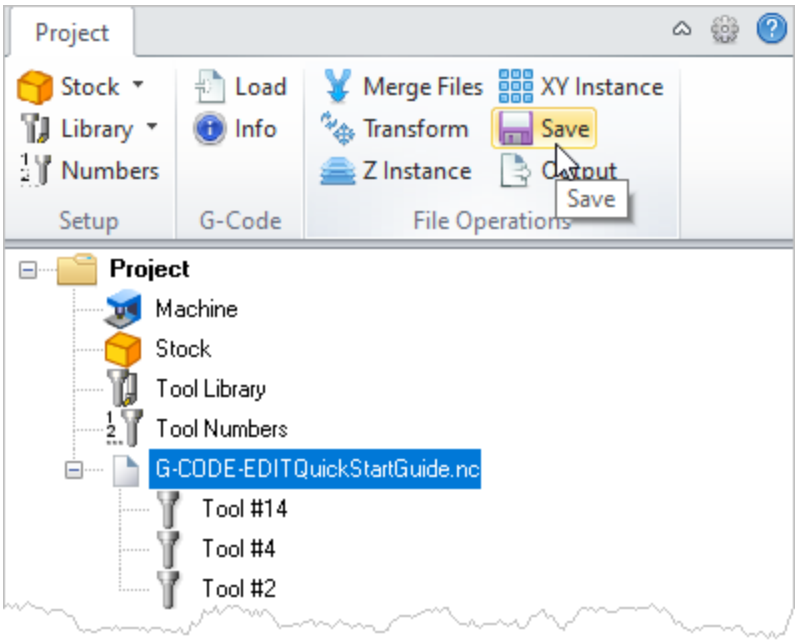


4. You will notice that the G-Code file is flagged because it has not been saved.



5. You can also save your G-Code file from the Project tab by selecting 

from the Project tab menu.



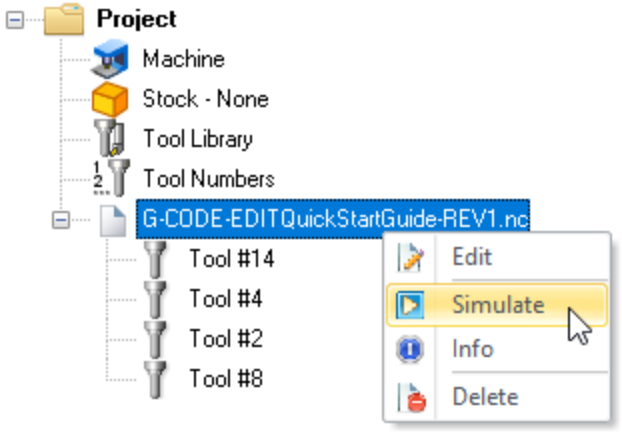
© MecSoft Corporation

G-Code Simulations

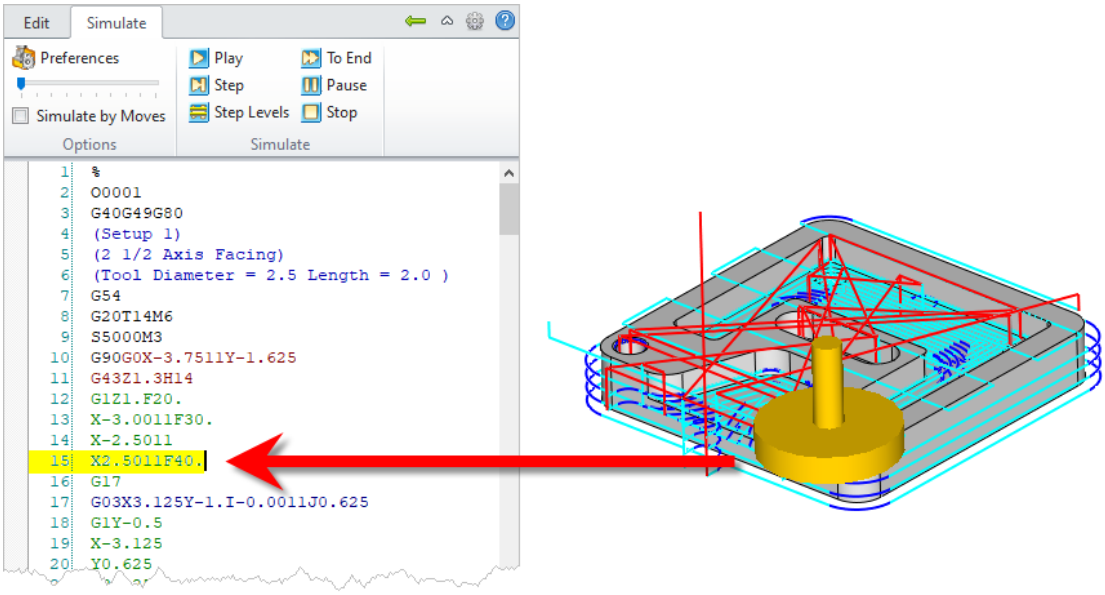
6.1 Tool Motion Simulation

Now that we have a Tool Library loaded and a Tool Crib defined all simulations that we do will display the correct tool. Our first simulation will be a Tool Motion simulation. The tool will be displayed in the graphics screen as it follows each line in the G-Code file.

1. Select the G-Code file from the Project tab, right-click and select Simulate.



2. The G-Code file is loaded into the Simulate tab and the cursor begins at the top of the file and proceeds to the bottom of the file while a graphical display of the tool is shown in the graphics screen at each line in the G-Code file.



3. Here are some operations tips when running a Tool Motion Simulation:

   A. You can toggle the display of the stock using the icon provided at the bottom of the Simulate tab.

© MecSoft Corporation

B. During the tool motion simulation, you can use the icons on the Simulate tab to Pause, Stop and Start the simulation.

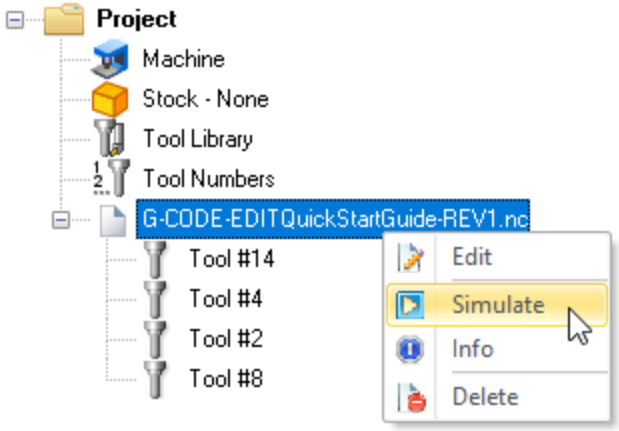
C. Once the simulation is Paused, Stopped or reaches the end of the G-Code file, you can then manually move the cursor to any line in the G-Code file to see the exact tool position at that line. This is very helpful for locating the exact line of code that is causing a problem.

D. If you are getting an error message from your CNC machine controller and it reports a line # where the error occurred, you can analyze the G-Code and tool position at that line number. This is very helpful for understanding the error and how best to take corrective action.

6.2 The Simulate Tab

The Simulate tab provides commands for running simulations and access to simulation preferences. Here is a brief description of the Simulate tab. These commands work in the same manner as those in the MILL module.

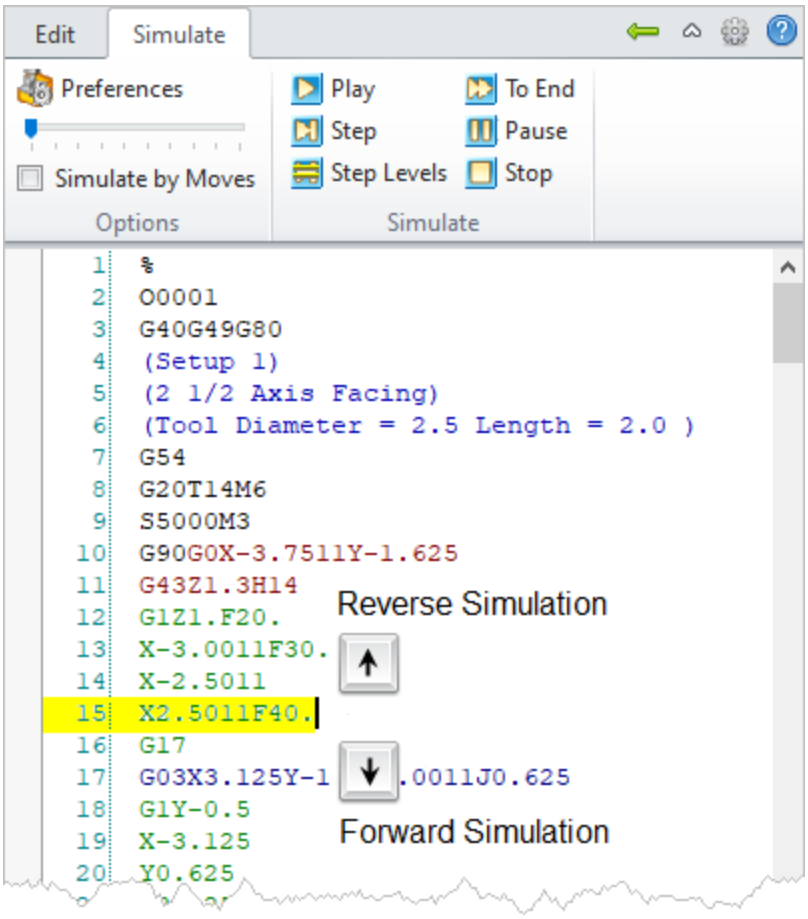
1. Select the G-Code file from the Project tab, right-click and select Simulate.



2. While the Tool Motion Simulation is running you can select the Pause button to stop the simulation at the current line in the G-Code file. Selecting Play will continue the simulation. Step will run the simulation incrementally. You can also select Preferences to access the Simulation Preferences. These are the same Preferences located in the Preferences dialog.

   Optionally you can Pause the simulation and then use the Up and Down Arrow Keys on your keyboard to move the simulation Forward or Reverse line by line.

© MecSoft Corporation

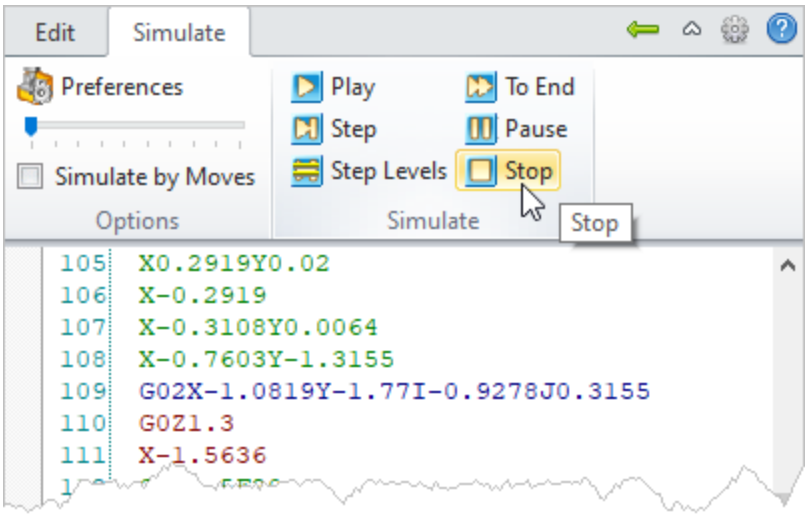


6.3 Define Your Stock

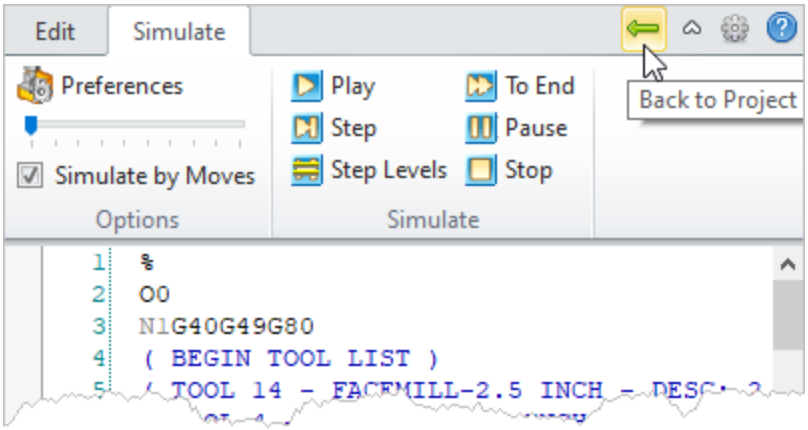
In order to run a Cut Material Simulation you must have a Stock defined.

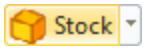
1. Pause or Stop any simulation that is currently running by selecting the appropriate command from the Simulate tab.

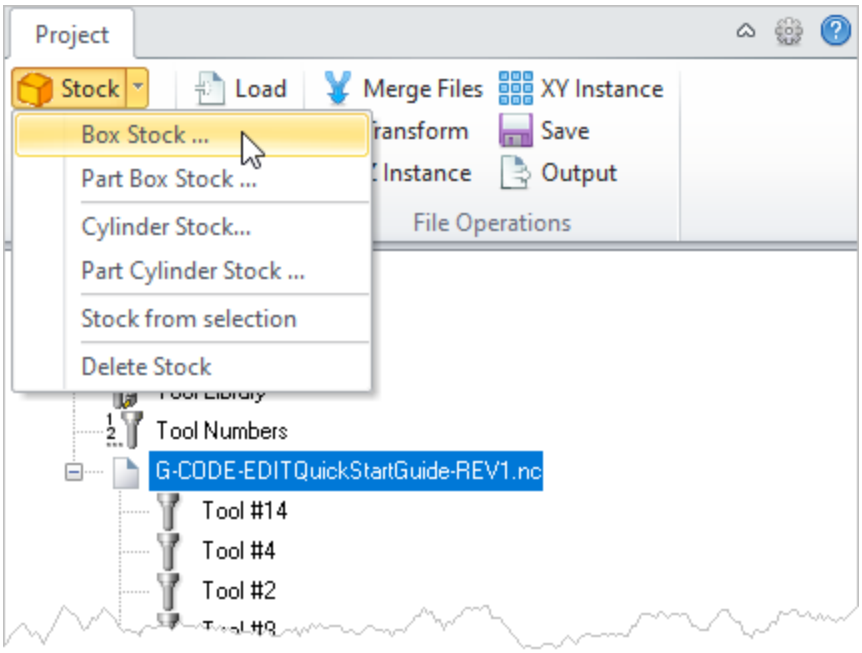
© MecSoft Corporation



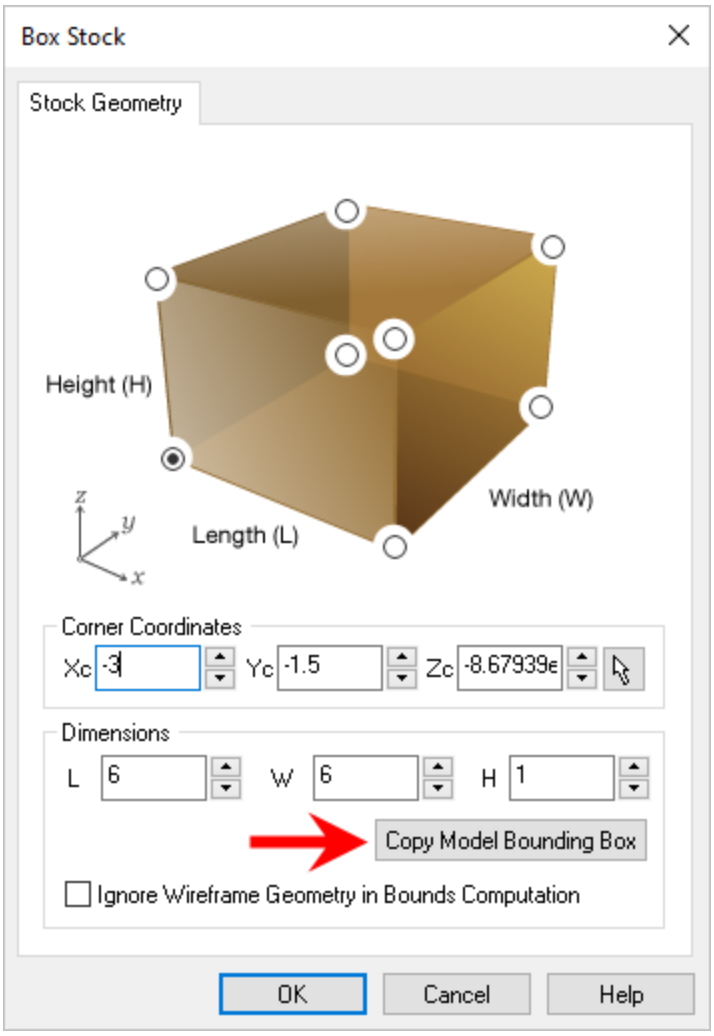
2. Return to the Project tab.



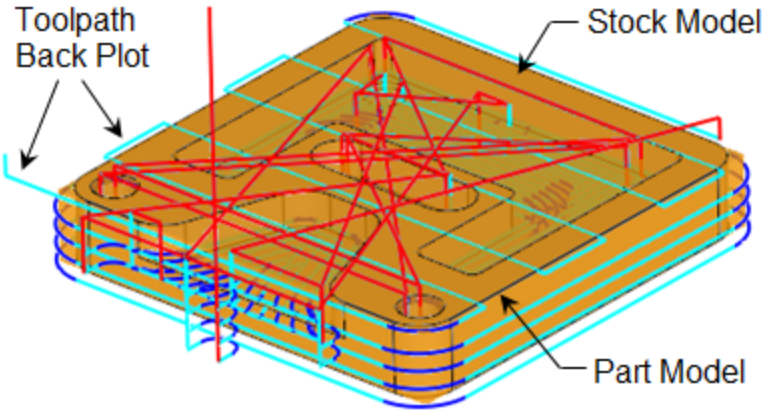
3. From the Project tab select  and then select Box Stock ... from the menu.



4. This will display the Box Stock dialog similar to the MILL module.

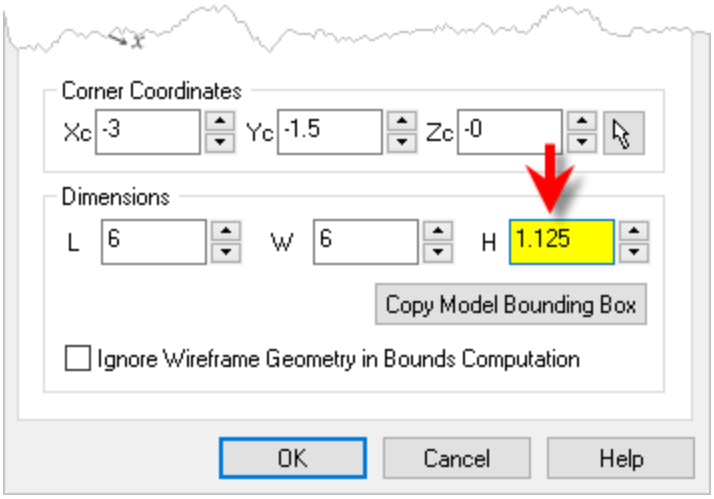


5. Select the Copy Model Bounding Box button. The stock model will be displayed in the graphics window that matches the size of the part model.

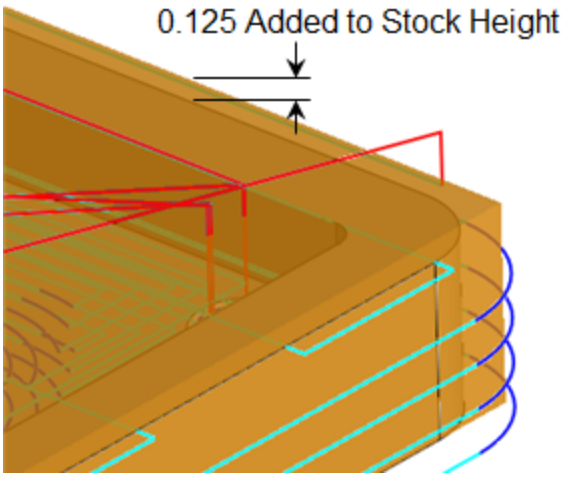


© MecSoft Corporation

6. Now enter 1.125 for the Height (H) dimension in the Box Stock dialog. This will add 0.125 to the height of the Stock model.



7. Now pick OK to close the Box Stock dialog.



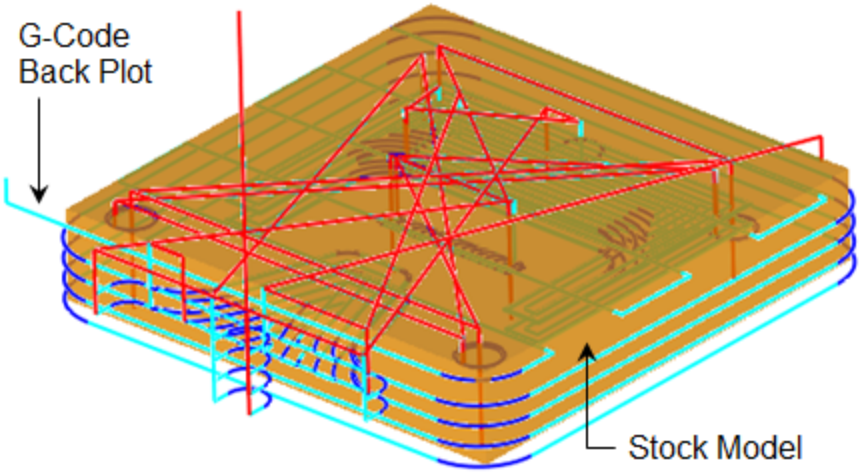
6.4 Cut Material Simulation

With our Stock defined we can now perform a Cut Material Simulation. With Stock defined, selecting Simulate will now display the in-process cut stock at each line in the G-Code file!

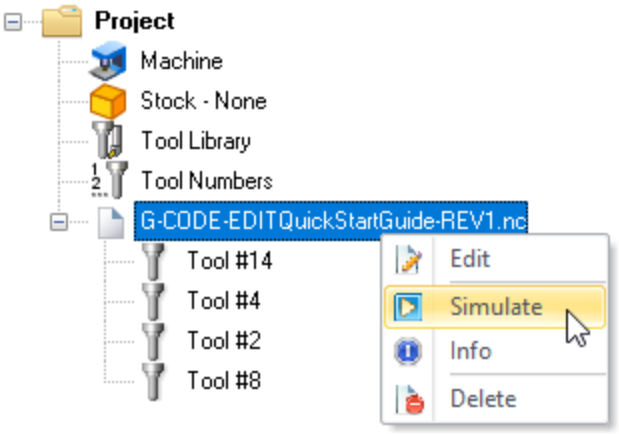
1. First let's hide the display of the Part model to see the Cut Material Simulation more clearly.
   Select the Part model from the graphics screen and then press the (Ctrl + H) keyboard short cut to hide the Part model.

   Now only the Stock model and the G-Code backplot are displayed in the graphics window.

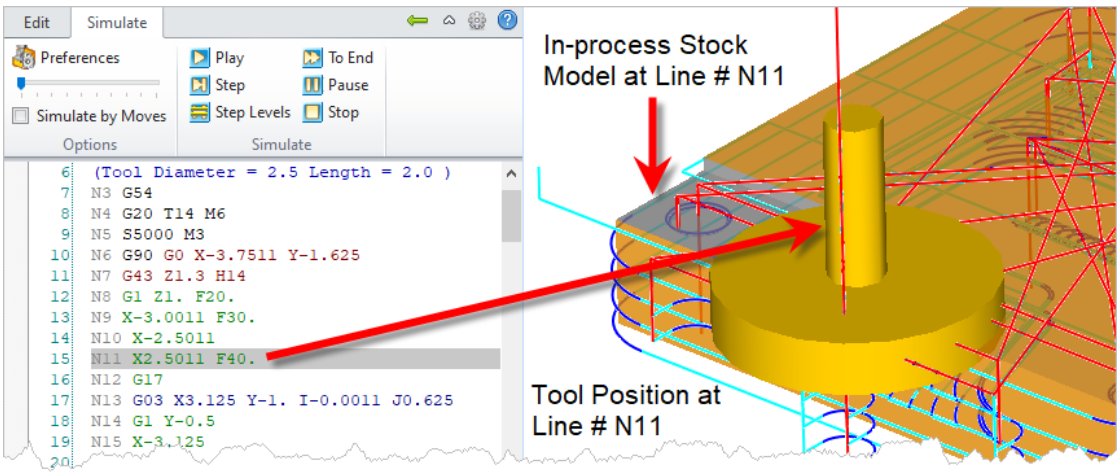
© MecSoft Corporation



2. Now select the G-Code file from the Project tab, right-click and select Simulate.



3. The G-Code file is loaded into the Simulate tab and the cursor begins at the top of the file and proceeds to the bottom of the file while a graphical display of the tool is shown in the graphics screen at each line in the G-Code file.

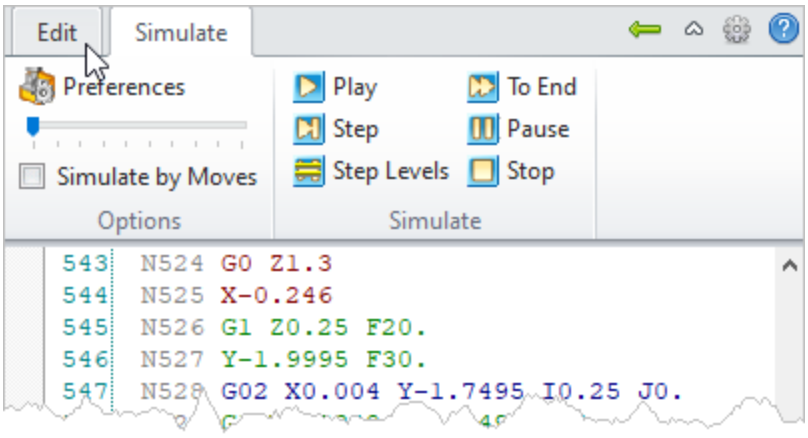


More G-Code Edits

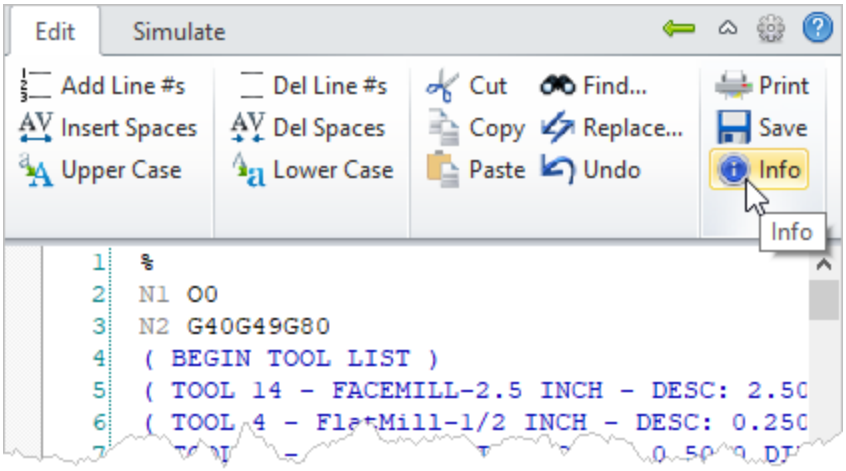
7.1 Estimate Machining Time

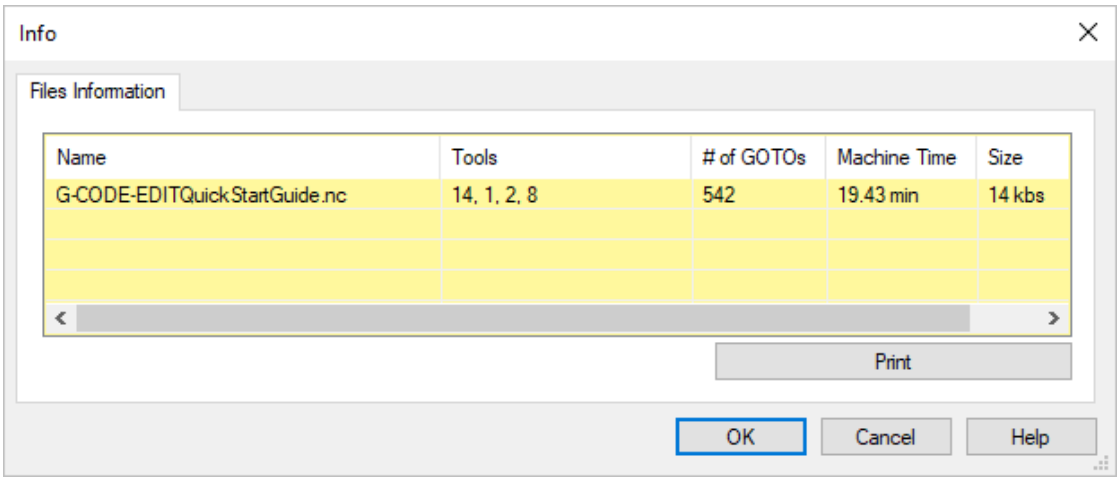
You can get additional Information about the G-Code files currently loaded into the G-CODE Editor.

1. Select the Edit tab to return to the Edit menu.

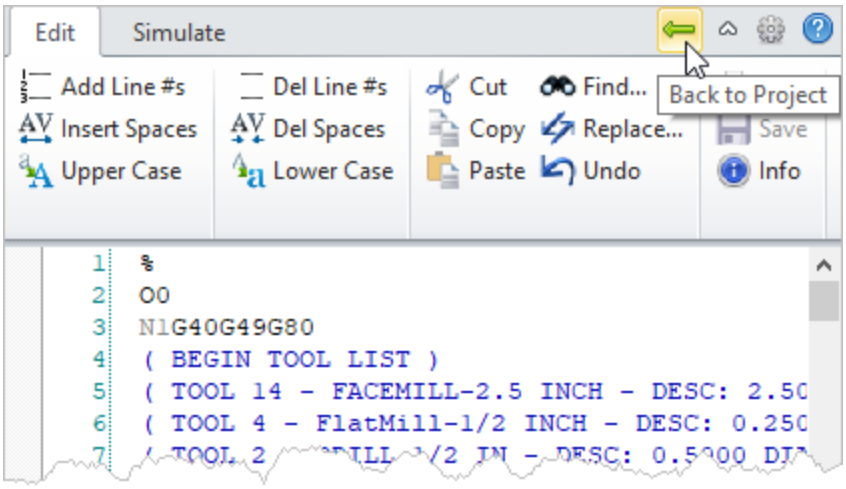


2. Now select Info from the Edit menu to display the Information dialog.

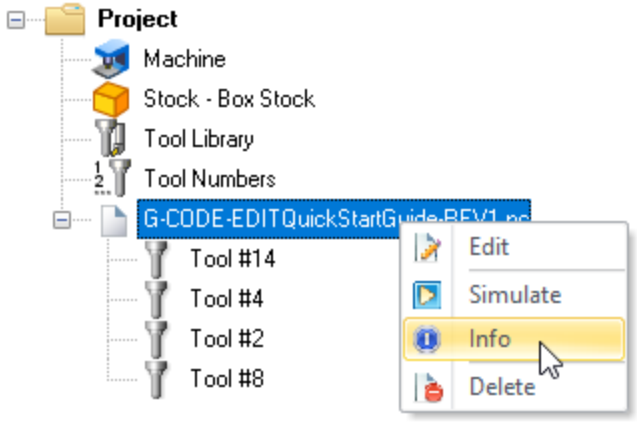




3. You can also perform this task from the Project tab. Return to the Project tab, select the G-Code file and then select Info from the Project menu to display the Information dialog.



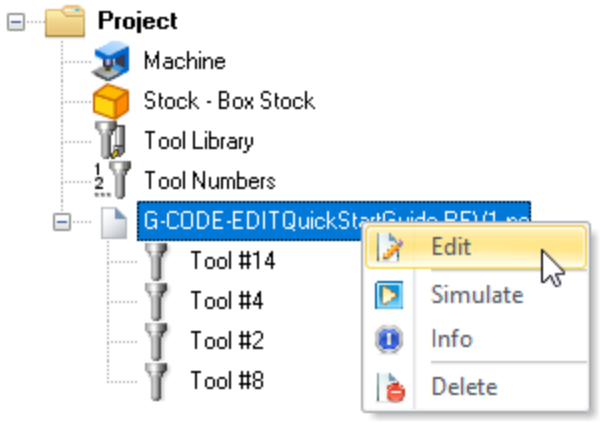
You can also right-click on the G-Code file and select Info to display the Information dialog.



7.2 More Edits

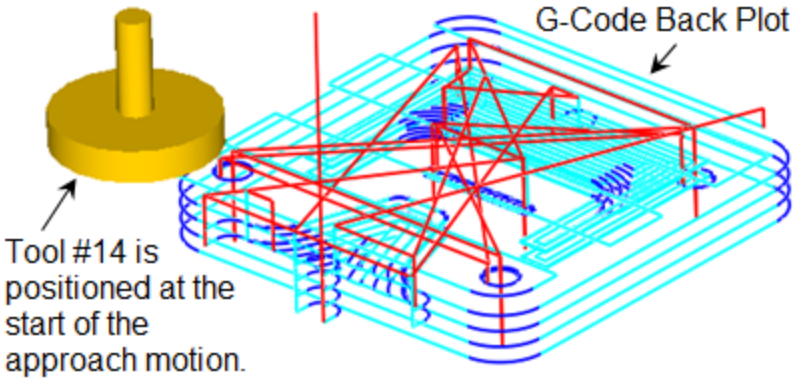
Let's perform one additional edit before we complete this guide.

1. Right-click on the selected G-Code file located in the Project tree and select Edit.



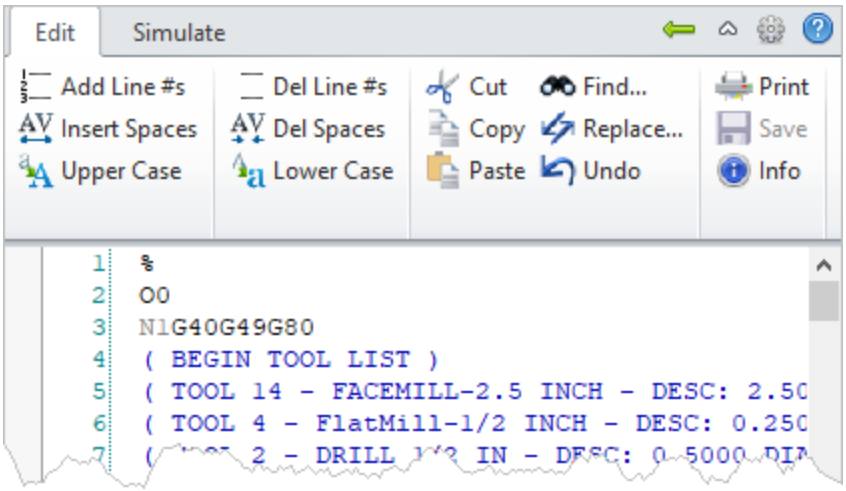
1. If you recall back in the section Add Line Numbers, you were warned that no appropriate tools were found. That's because we did not have a Tool Library and Tool Crib loaded yet.

   Now when you go to the Edit tab, the first tool # is loaded and displayed in the graphics screen at the beginning of the first tool motion.

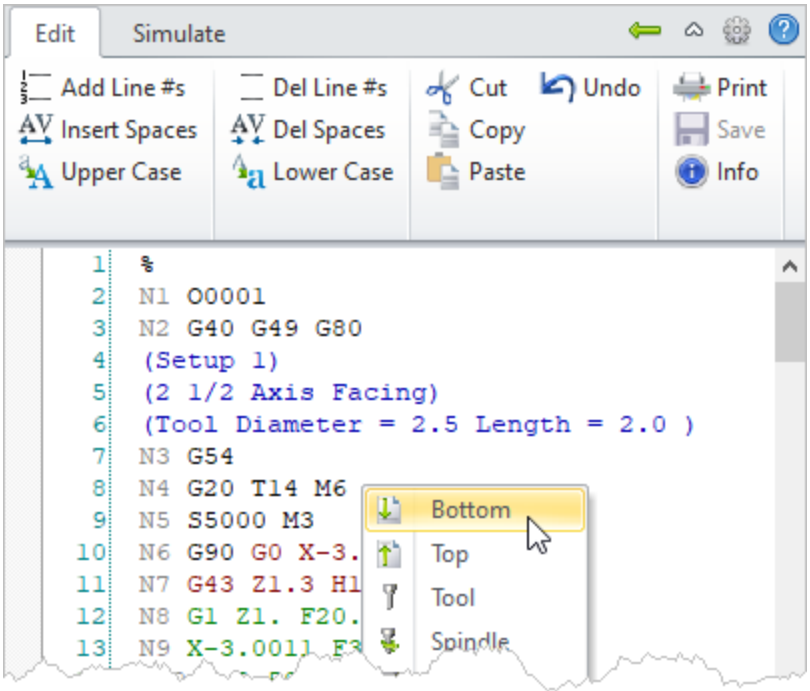


2. You will see that the Project tab is replaced with Edit & Simulate tabs and the G-Code file is loaded into the editor as shown below.

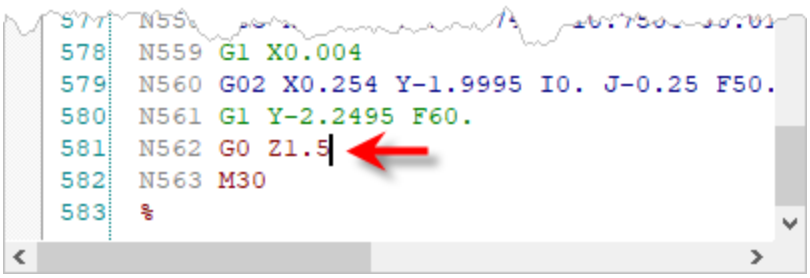
© MecSoft Corporation

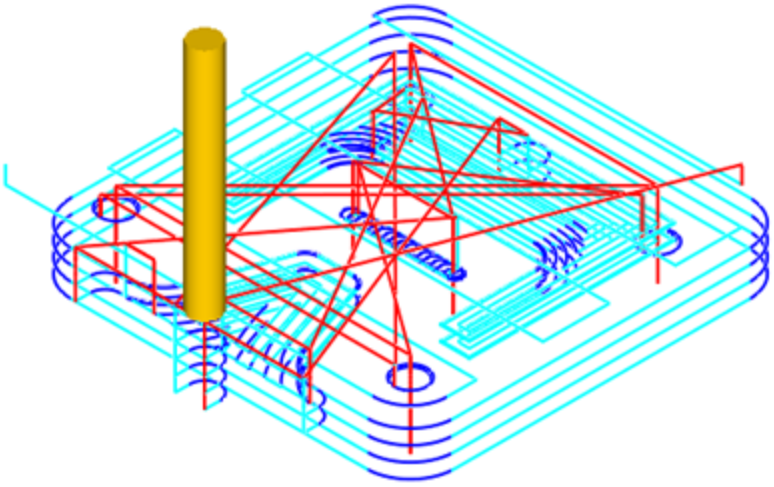


3. One edit than many users find useful is to retract the tool to a higher Z height once cutting is completed.
   From the Edit tab right-click on any line in the G-Code file and select Bottom to go directly to the end of the file.

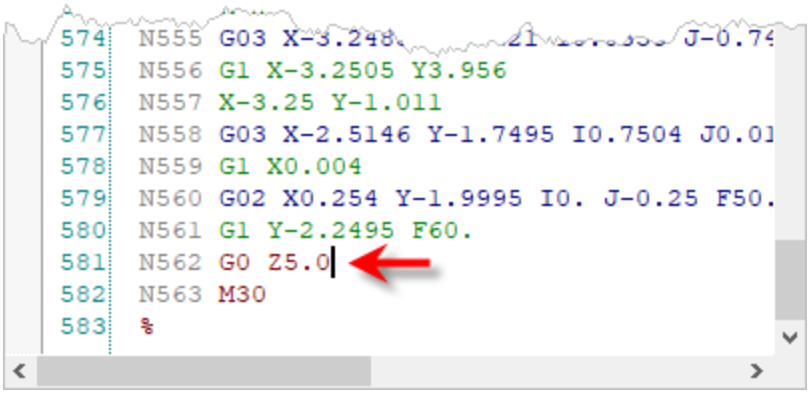


4. Now place the cursor on the last G0 line and notice where the tool is position in the graphics screen.

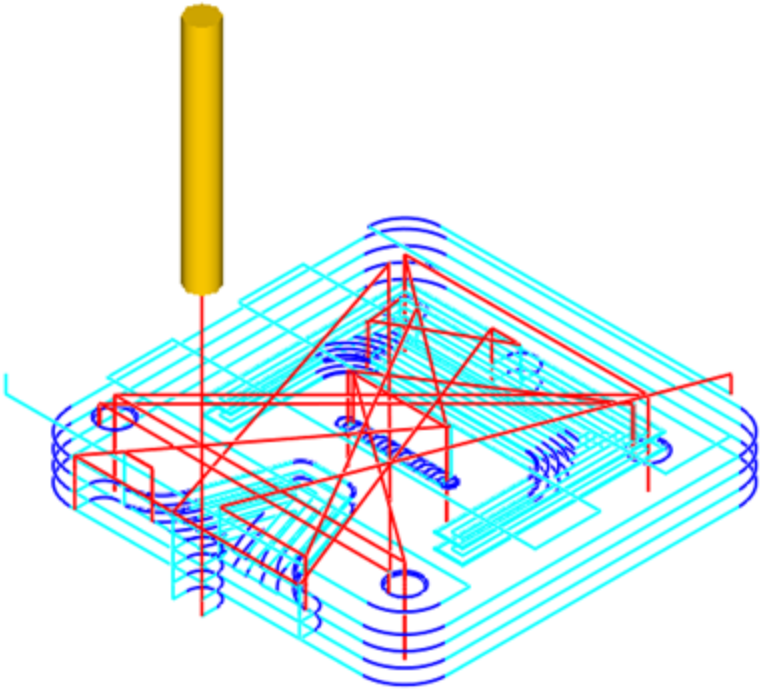




5. Now change Z1.5 to Z5.0 and notice that the tool rises up to the Z5.0 location.



© MecSoft Corporation



6. Refer to the section Saving Changes to update your external G-Code file will all of the latest changes.

© MecSoft Corporation

Where to go for more help

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide

The 2025 RhinoCAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

Index

- A -

- E -

- L -

- M -

- Q -

- R -

- S -

- T -

- U -

- W -

© MecSoft Corporation