

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)

(TOOL 2 - FlatMill:0.125 - DESC: 0.1250 DIA, 2 FLUTE, CARBIDE MAT)

(TOOL 3 - FlatMill:0.0625 - DESC: 0.0625 DIA, 2 FLUTE, CARBIDE MAT)

G-Code Editor Quick Start Guide

RhinoCAM 2026

Published: March 2026

(ENDOF TOOL LIST)

N4 (Setup 1)

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 Corporation

© Copyright 1998-2026

Table of Contents

Quick Start	5
Resource Guide	7
About this Guide	8
1 Useful Tips.....	8
2 About this Module.....	9
3 Using this Guide.....	9
4 Watch the Video!.....	11
Getting Ready	12
1 Running RhinoCAM.....	12
2 About the RhinoCAM Display.....	12
3 Load the G-Code Edit Module.....	13
4 Load the G-Code Files.....	14
5 G-Code Editing Strategy.....	19
Edit the G-Code File	21
1 How Tools are Used.....	21
2 Add Character Spaces.....	21
3 Load Your Tool Library.....	23
4 Define Your Tool Crib.....	25
5 Add Line Numbers.....	28
6 Saving Changes.....	30
G-Code Simulations	33
1 Tool Motion Simulation.....	33
2 The Simulate Tab.....	34
3 Define Your Stock.....	35
4 Cut Material Simulation.....	39
More G-Code Edits	41
1 Estimate Machining Time.....	41
2 More Edits.....	43
Where to go for more help	47

Index**49**

Quick Start



G-CODE Editor Module 2026

[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

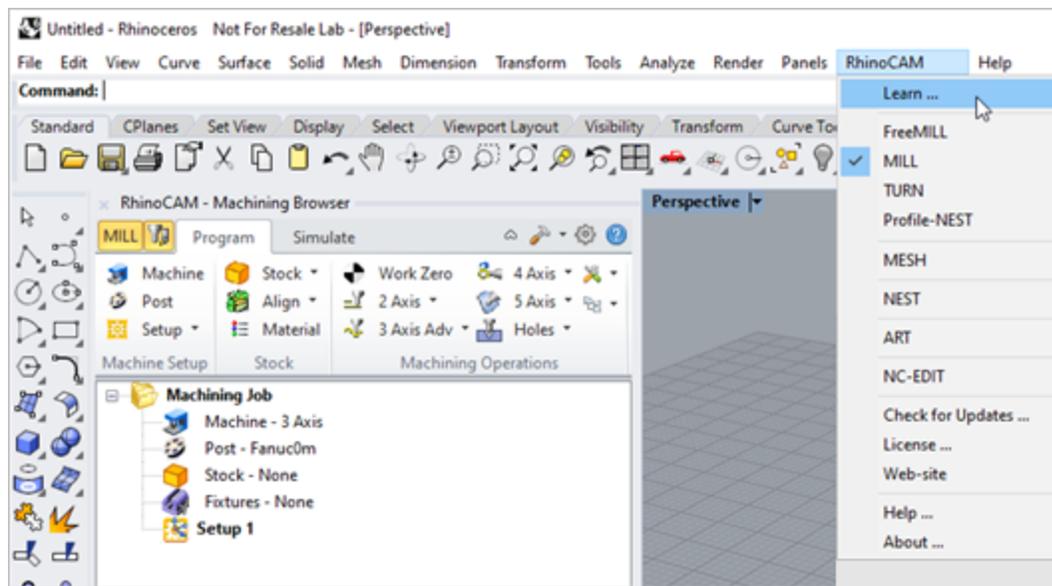
To help you quickly get started in working with each module, select one of the Help buttons located on the [RhinoCAM Learning Resources](#) dialog.

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:

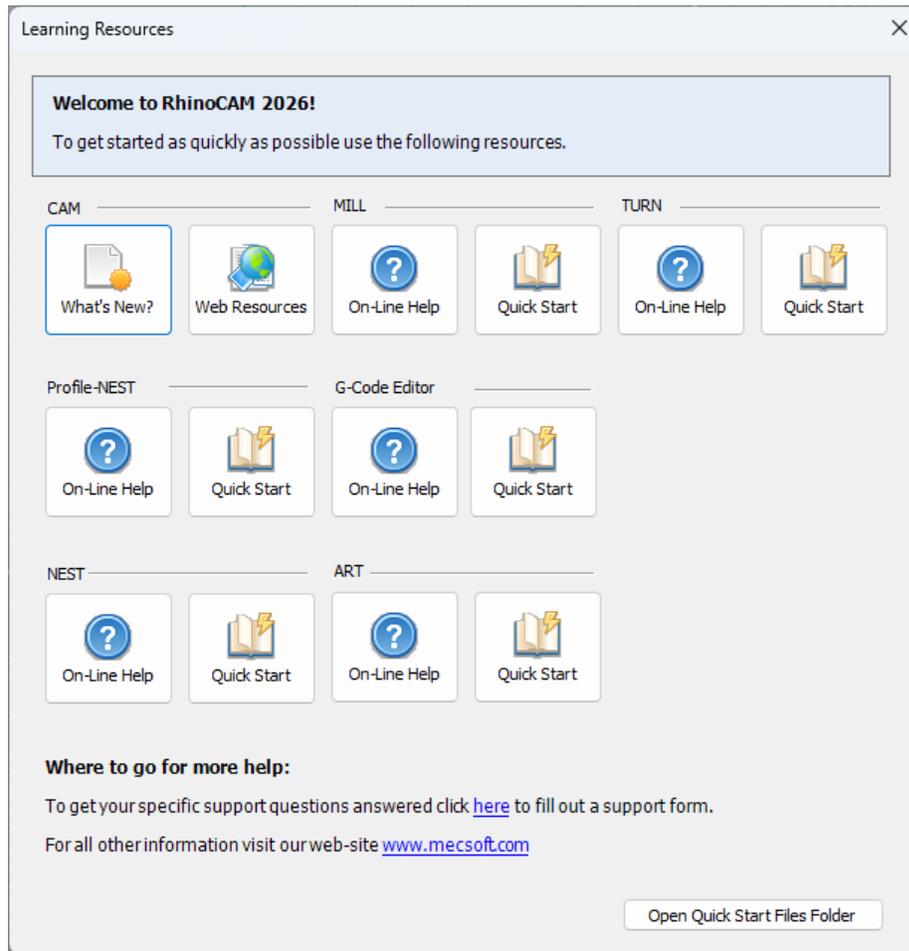
1. From the [Rhino Main Menu](#), drop down the Main menu and select [Learn ...](#)



To access the Learning Resources dialog 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.

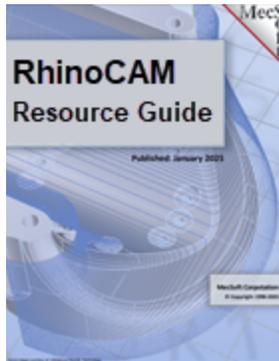


Resource Guide

Download this PDF Guide for a list of the available [RhinoCAM Resources](#).



2026 RhinoCAM Resource Guide



The 2026 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](#)

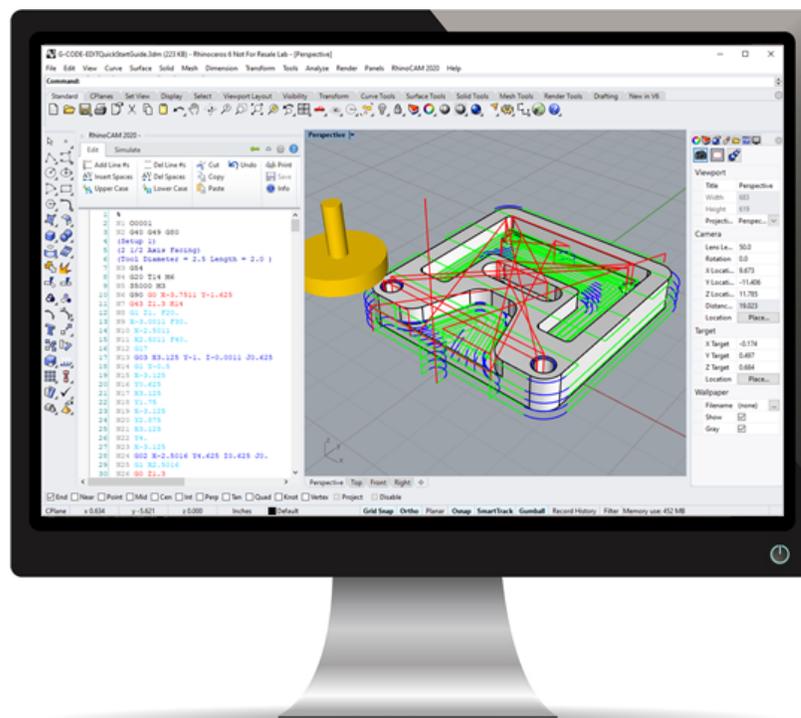
About this Guide



G-CODE Editor Module 2026

[Prefer Printed Documentation? Check Here!](#)

[What's New](#) | [Quick Start Play List](#)



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.

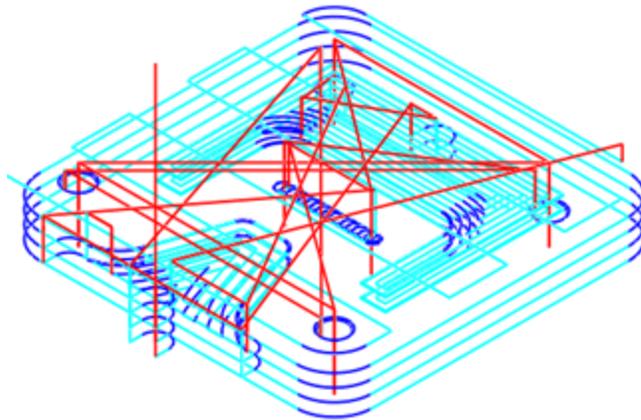


List of Included Source Files

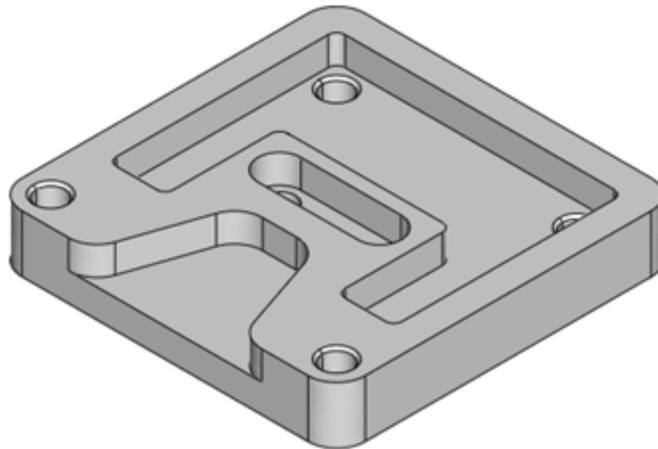
- Source G-Code File:
[G-CODE-EDITQuickStartGuide.nc](#)
- Source Part File:
[G-CODE-EDITQuickStartGuide.3dm](#)
- Source Tool Library File:
[G-CODE-EDITQuickStartGuide.csv](#)
- Source Part File (Completed):
[G-CODE-EDITQuickStartGuide_Completed.3dm](#)
- **Note:** Source part files are not required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



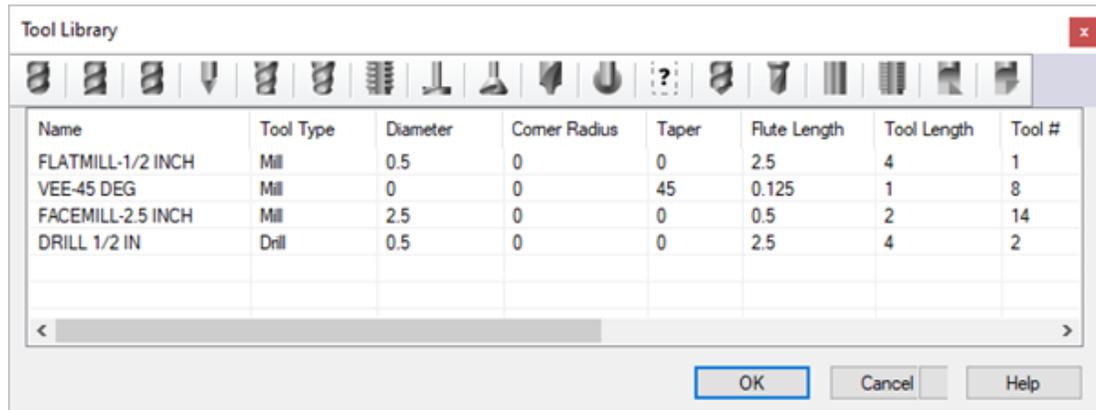
Included Source Files



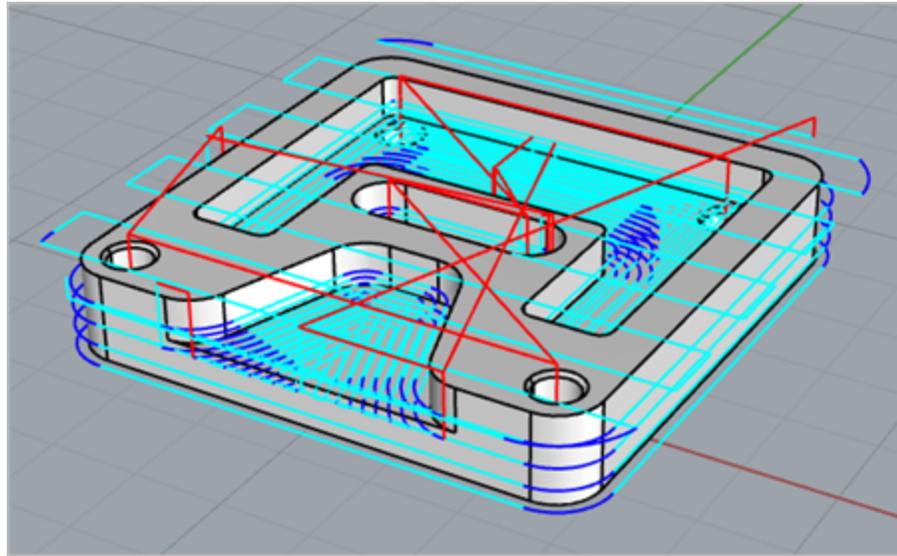
Source G-Code File:
G-CODE-EDITQuickStartGuide.nc



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



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



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.



What's New!

[What's New in RhinoCAM 2026](#)



The Complete Quick Start Video Play List

[Here is a link to the complete 2026 Video Play List](#)

Getting Ready

4.1 Running RhinoCAM

Locate the [RhinoCeros 8](#) 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 8](#). (The name of this program group will usually be called [RhinoCeros](#), unless you specified otherwise during setup.)

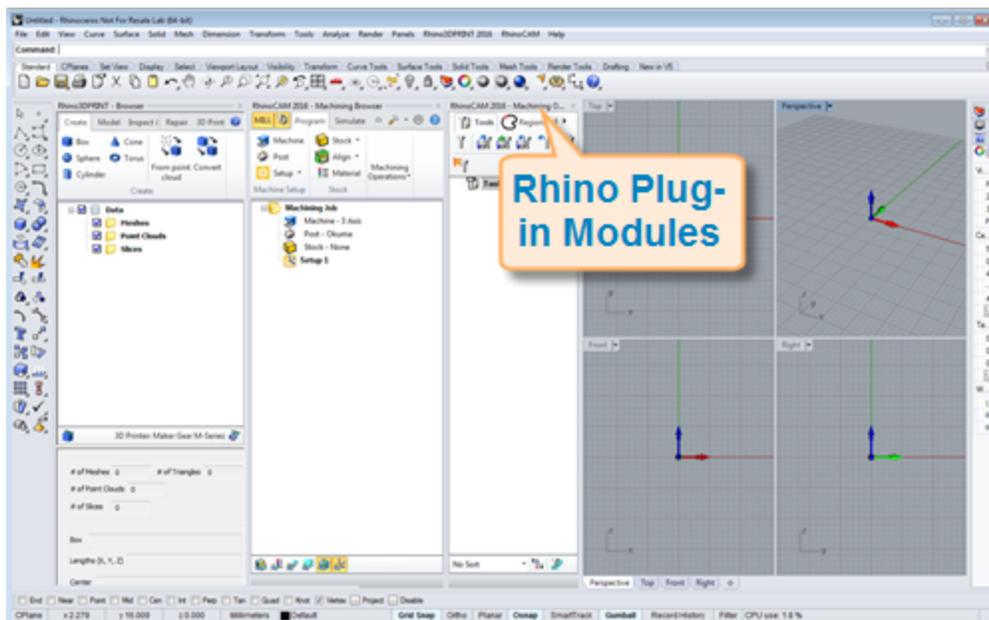
Once you locate the program group, select it and then select [RhinoCeros](#) to launch the application.

If the installation was successful, upon launching of [Rhino](#) you should observe a menu entry called [RhinoCAM 2026](#) in the main menu bar of [Rhino](#).

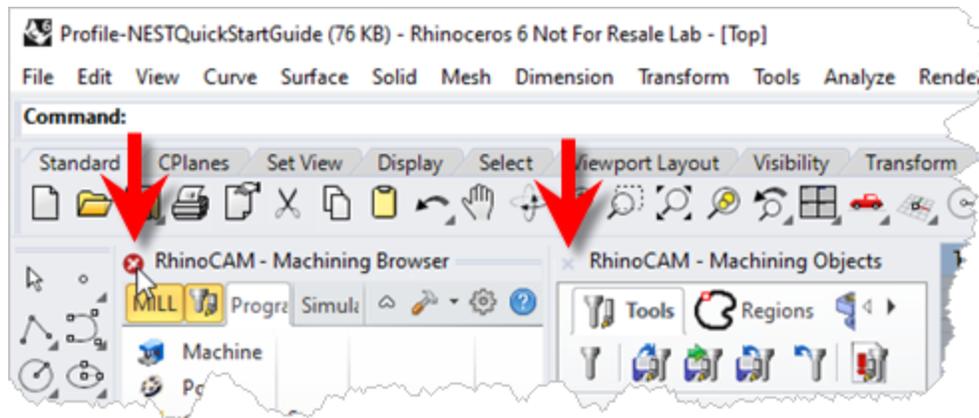
If you do not see this menu entry then please check the [On Line Help](#) document of the product (found in the installation folder) for help with trouble shooting the installation.

4.2 About the RhinoCAM Display

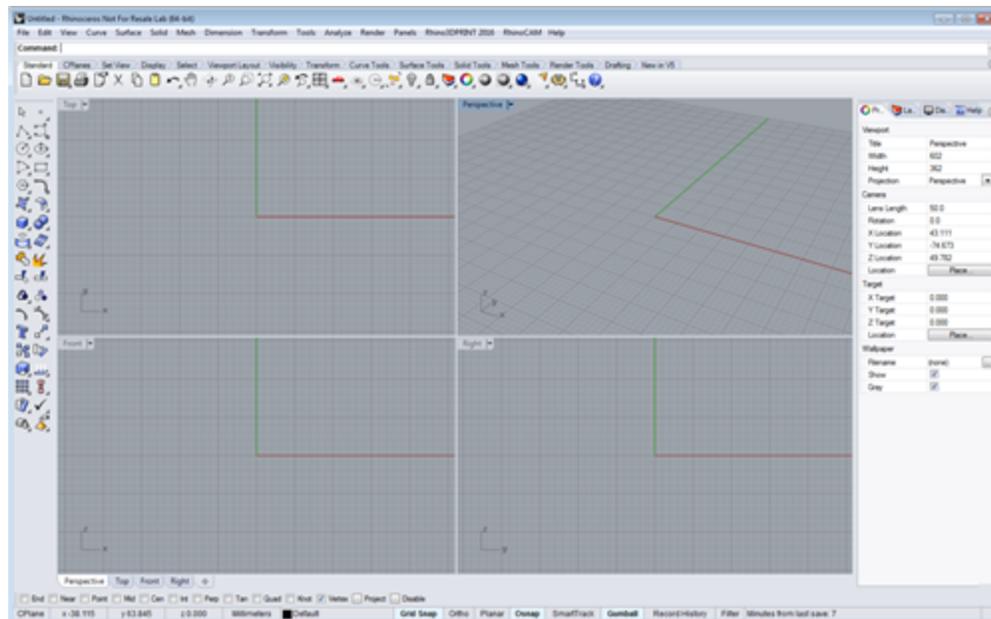
Before we begin, let's talk a bit about the [RhinoCAM](#) display. When you run [RhinoCAM](#) for the very first time, your screen may look this.



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



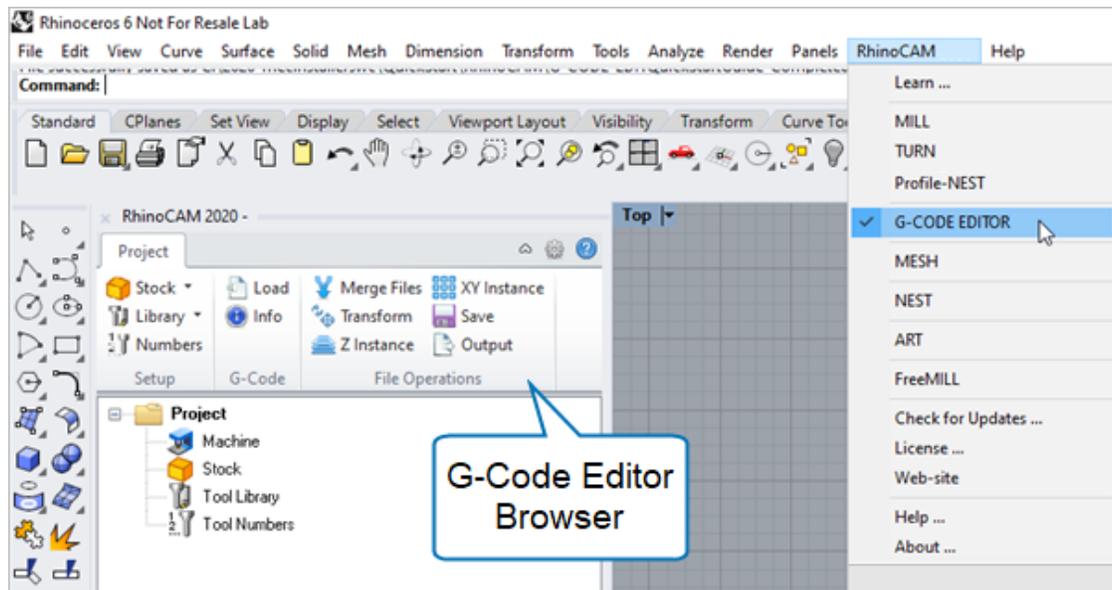
With all plug-in modules closed your screen will look like this:



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 2026](#) main menu and select [G-CODE EDITOR](#).



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!

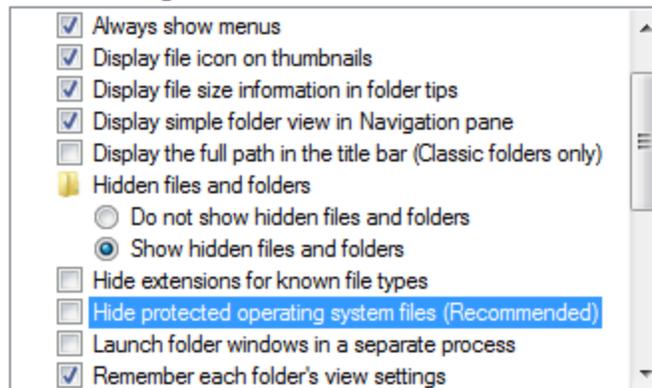
Load the Part File

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 2026 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:

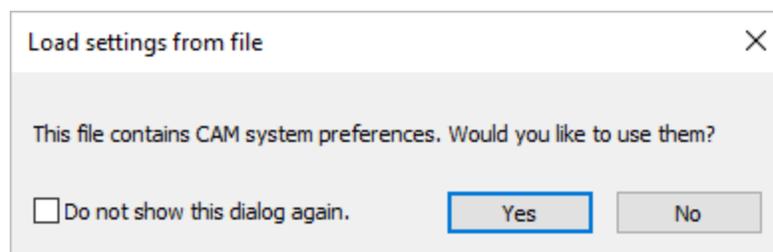
1. For **Windows 8** users: Go to **Control Panel > Appearance and Personalization > Folder Options**.
For **Windows 10** users: Go to **Control Panel > Appearance and Personalization > File Explorer Options**.
2. Select **View** tab and under advanced settings select **Show Hidden files and folders**, clear the check boxes for:
 - Hide extensions for known file types
 - Hide protected operating system files (Recommended)

Advanced settings:

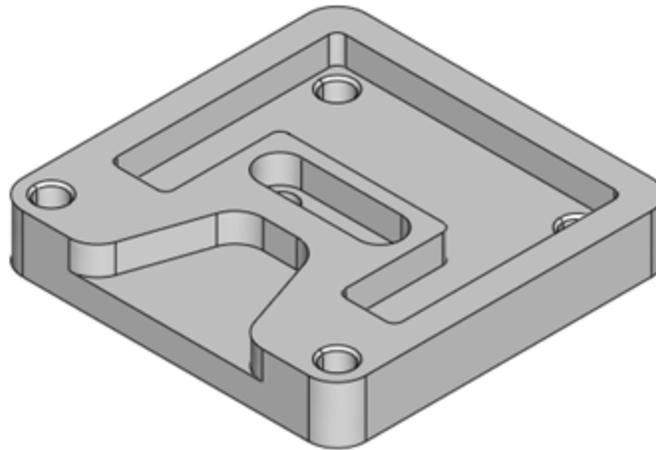


3. Click **Apply** and **OK**.

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



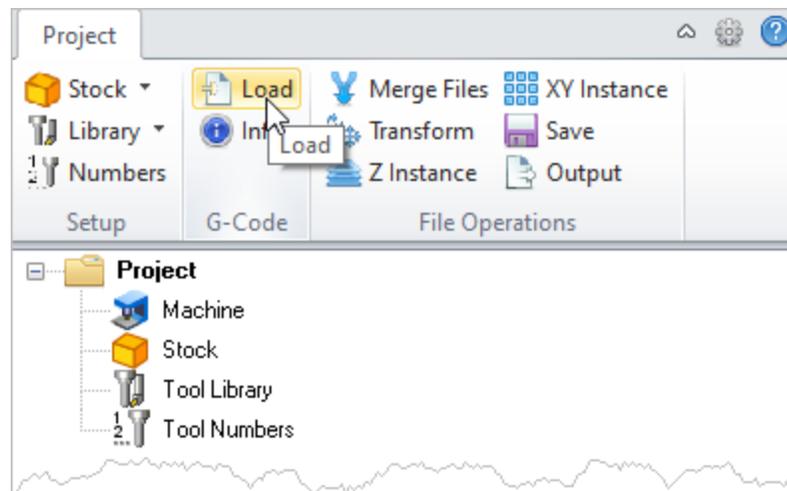
Source Part File:
G-CODE-EDITQuickStartGuide.3dm

3. From the [Rhino](#) display, double-left-click on the [Perspective View](#) tab to maximize it.



Load the G-Code File

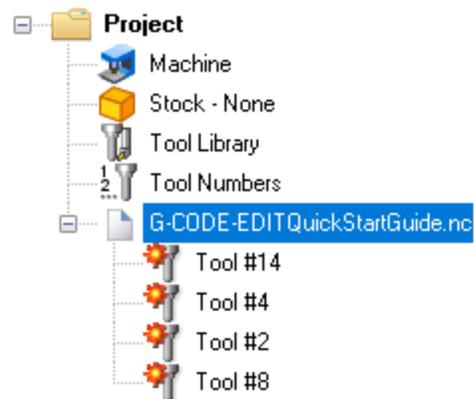
1. From the [Project](#) tab of the [G-CODE Editor Browser](#), select [Load](#).



2. From the [Open](#) dialog box, select the [G-CODE-EDITQuickStartGuide.nc](#) file from the [C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 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](#).

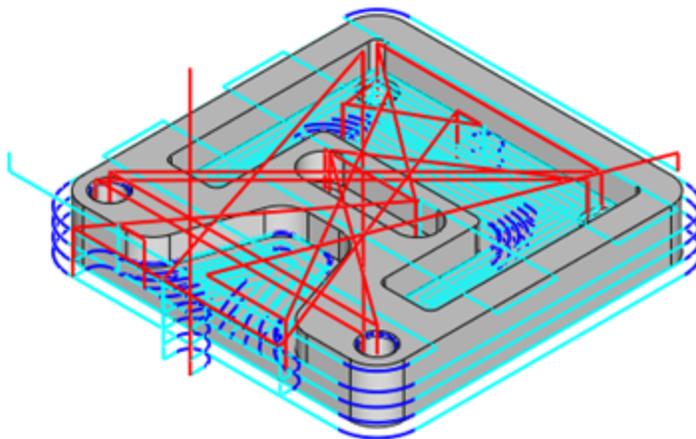
Notice that the G-Code file name appears in the **Project** tree with each tool referenced by the file listed below it.



! 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!

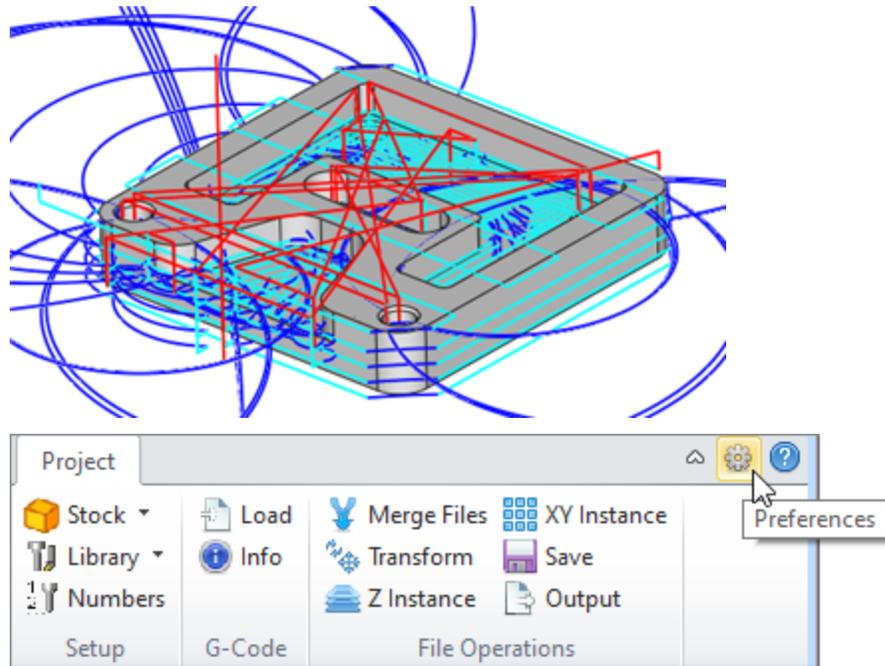
Refer to the [Load Your Tool Library](#) section for more information.

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

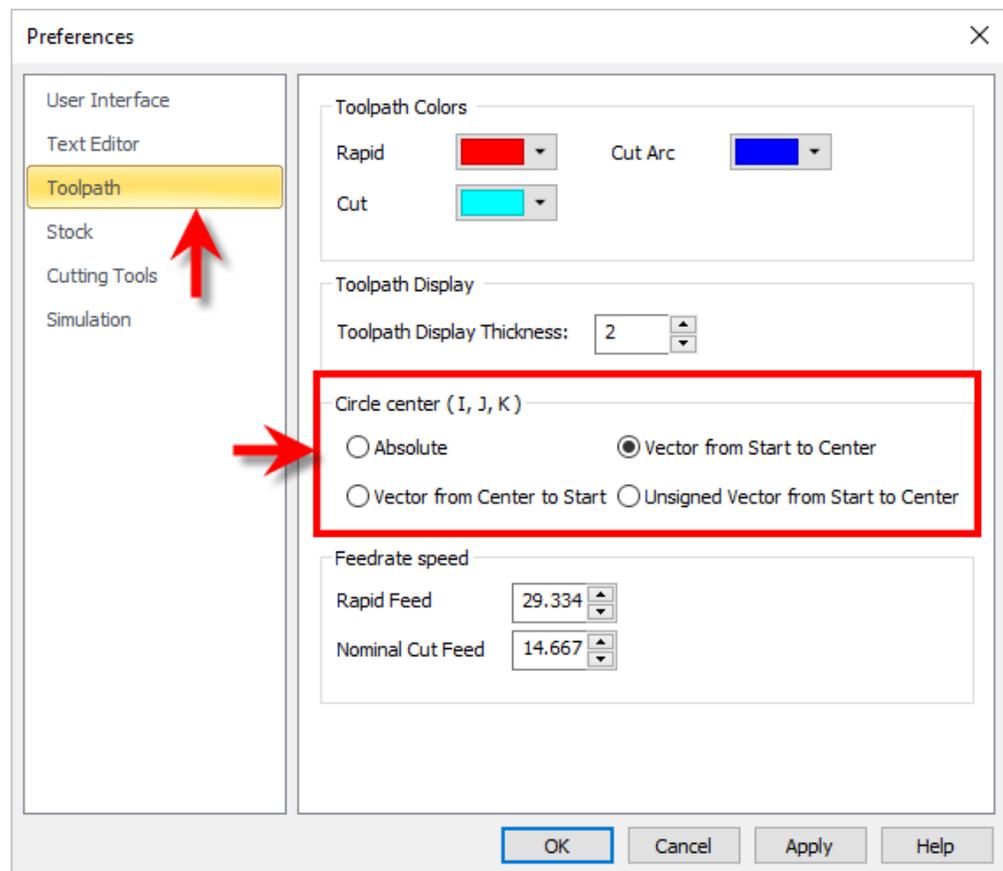


! As we mentioned previously, you **DO NOT** need to have a part geometry file loaded in order to edit G-Code files! It is loaded in this guide because this is the part file the G-Code was posted from. You can load any standard ISO G-Code file into the **G-CODE Editor** and perform edits, tool motions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

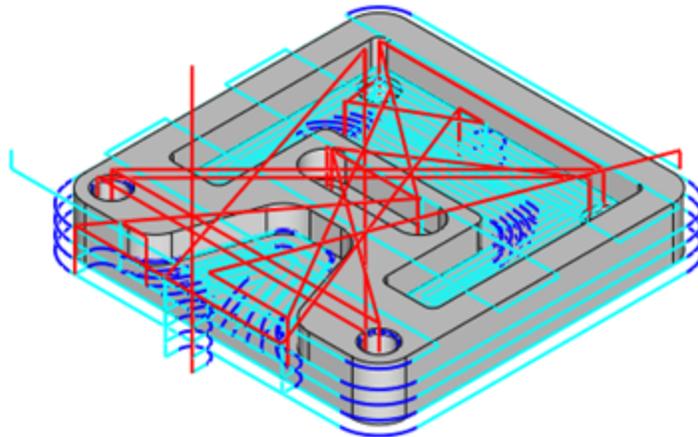
4. If the backplot of your G-Code file displays arc motions incorrectly as shown below, select the [Preferences](#) icon.



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.



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

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 outlined 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 additional G-Code edits, estimate machine time and then save our changes to the G-Code file.

Edit the G-Code File

5.1 How Tools are Used

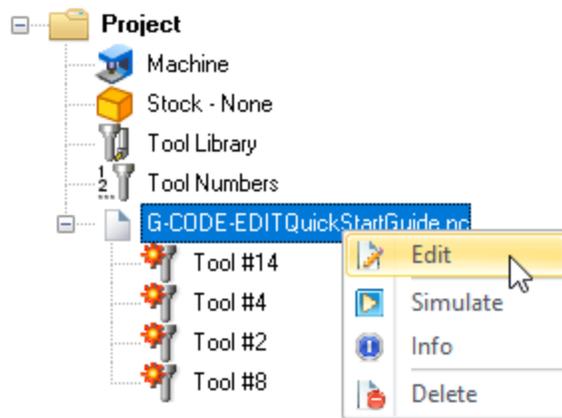
G-CODE Editor allows you to load a **Tool Library** and setup a **Tool Crib**. This allows you to match the tools that are referenced by the G-Code file. The **Tool Library** file (*.csv) can be generated and saved from within the **MILL** module while the part file with the CAM toolpaths is loaded.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

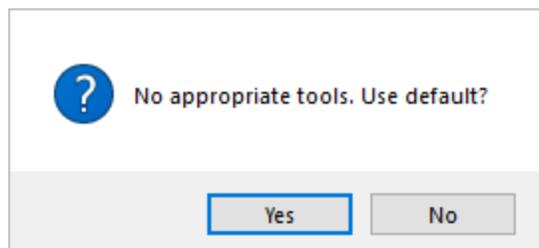
5.2 Add Character Spaces

You can make edits to your G-Code file as soon as it is loaded into the editor. As mentioned 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**.

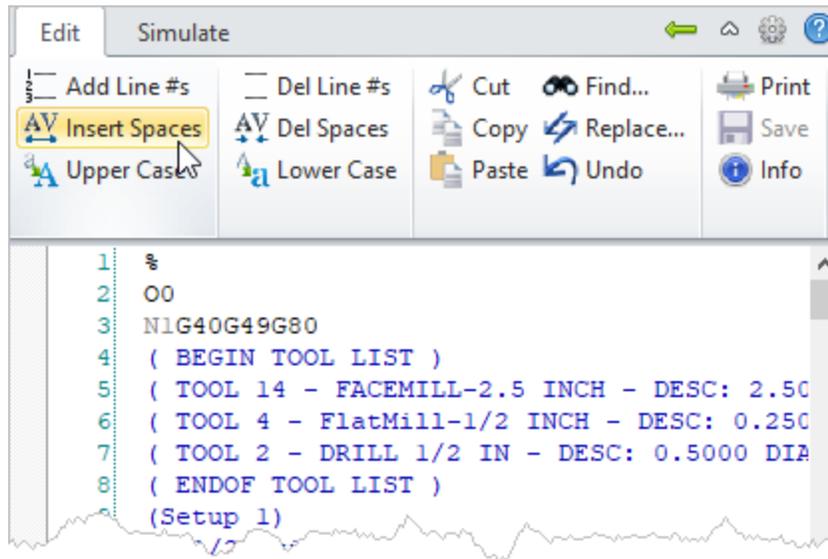


2. Pick **Yes** from the message that is displayed. Because no tools are defined yet, a default tool is used and displayed in the graphics window.

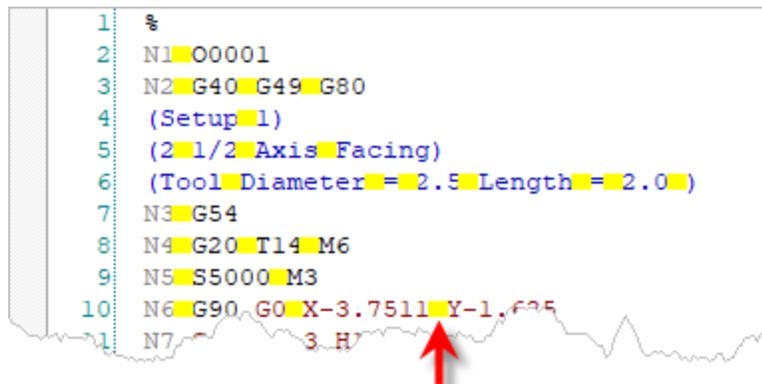


3. You will see that the **Project** tab is replaced with **Edit & Simulate** tabs and the G-Code file is loaded into the editor. The **Edit** tab contains all of the edit commands available in the current update of **RhinoCAM**.

4. From the **Edit** tab, select **Insert Spaces**.

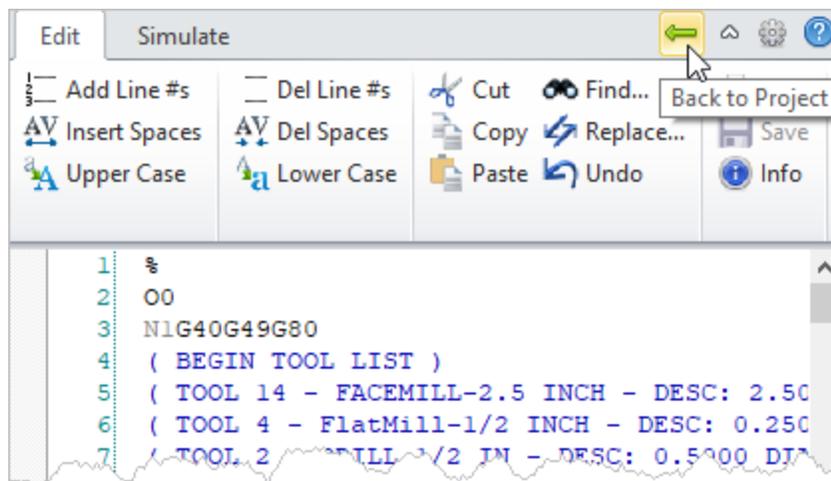


5. Character spaces are inserted into the G-Code file as shown.

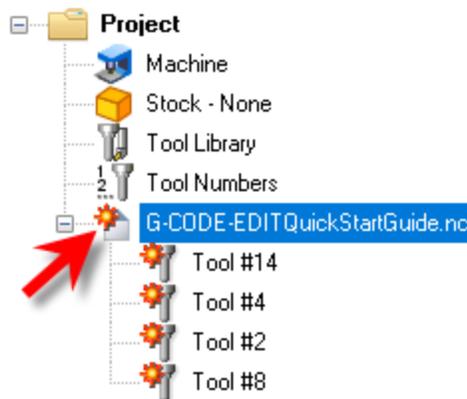


Inserted Character Spaces are highlighted for clarity.

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

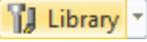


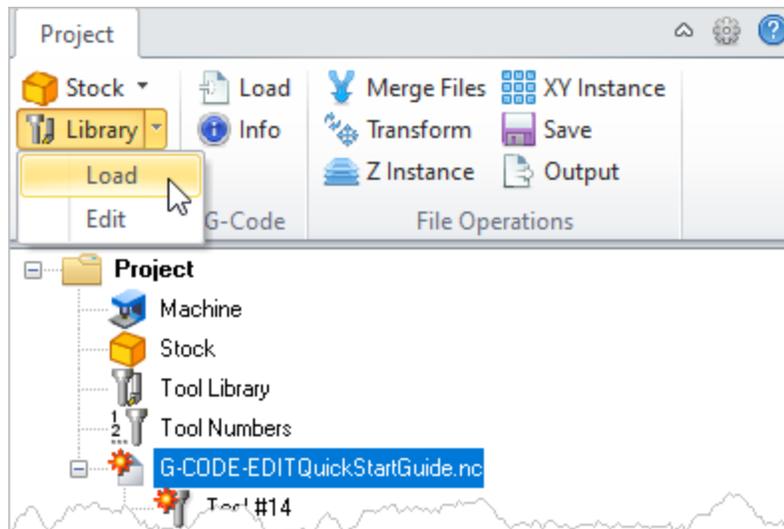
7. You see now that the G-Code file is also flagged. This indicates that an edit was made and that the file was not saved. We will make some additional edits so let's continue.



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.

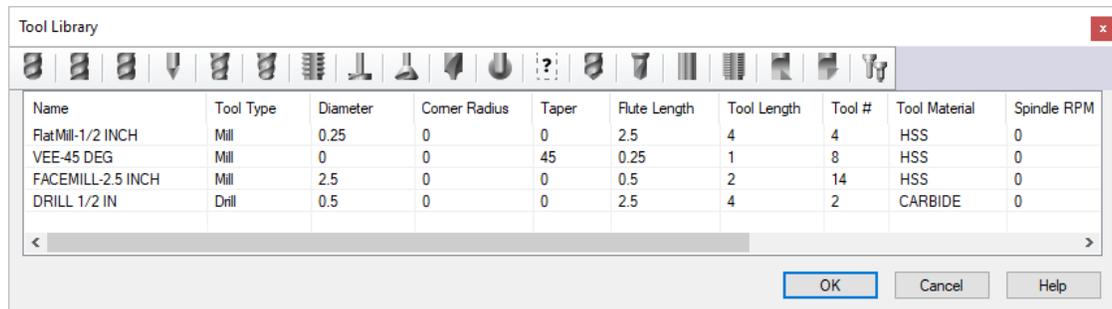


- From the **Open** dialog box, select the **G-CODE-EDITQuickStartGuide.csv** file from the **C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\QuickStart** folder and then pick **OK** to load the file.

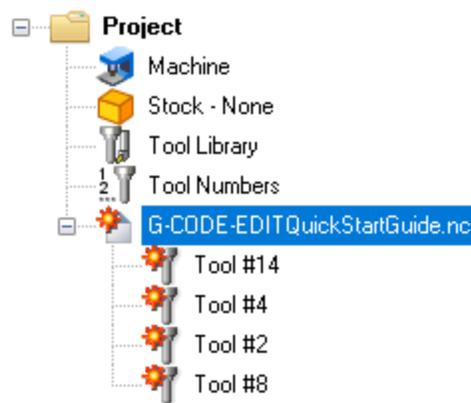
As mentioned previously, it is advisable to make a copy of this file at a suitable alternative folder so that you have write privileges to modify it if needed.

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

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

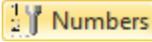


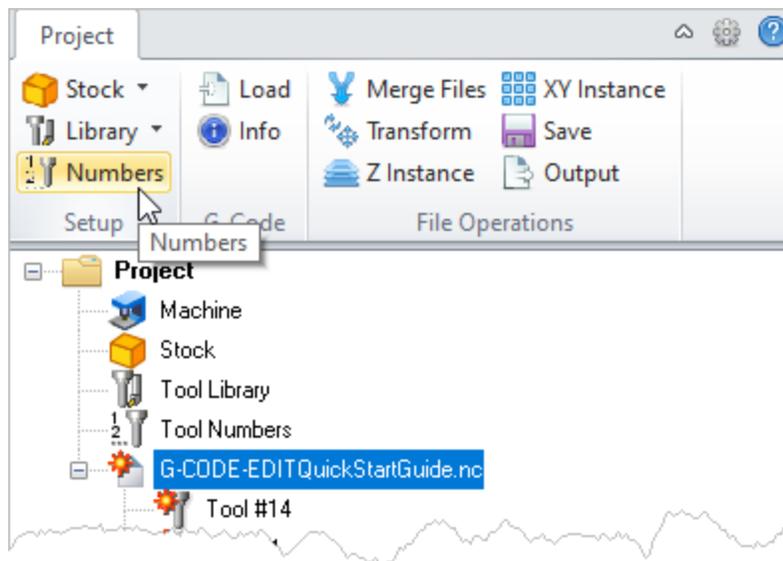
- Now pick **OK** to close the **Tool Library** dialog. You will notice that the tools listed in the Project tree under the G-Code file and still flagged. That's because we need to setup the **Tool Crib**.

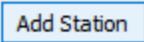


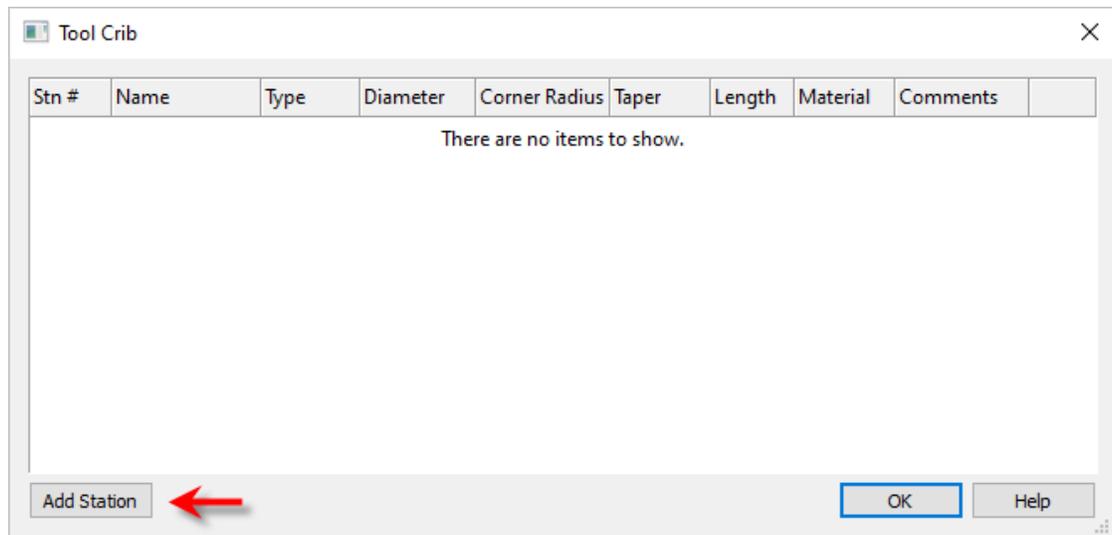
5.4 Define Your Tool Crib

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 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).

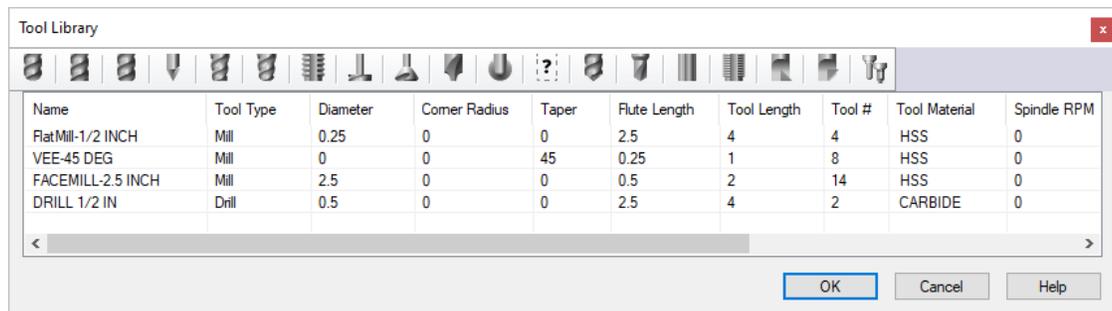
1. From the **Project** tab select  **Numbers** to display the **Tool Crib** dialog.



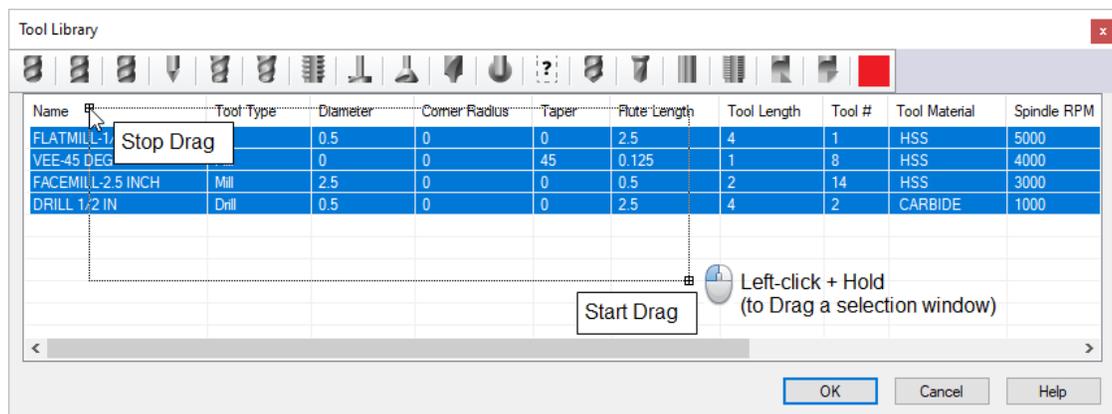
1. From the **Tool Crib** dialog select .



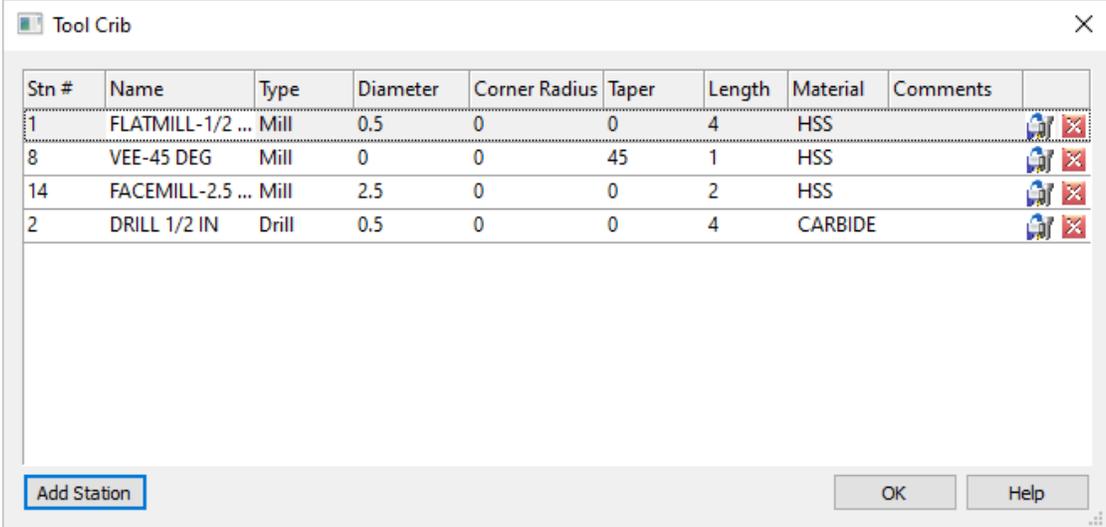
The **Tool Library** dialog will display.



- 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.



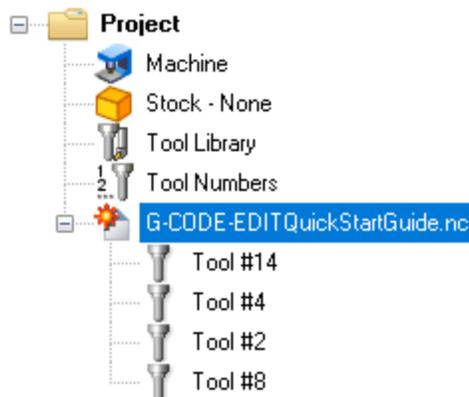
- Now pick **OK** from the **Tool Library** dialog and the **Tool Crib** dialog will reappear with the tools listed. This will be the **Tool Crib** for this guide. Now pick **OK** from the **Tool Crib** dialog to close it.



The screenshot shows a dialog box titled "Tool Crib" with a close button (X) in the top right corner. It contains a table with the following columns: Stn #, Name, Type, Diameter, Corner Radius, Taper, Length, Material, and Comments. There are four rows of tool data. Below the table is an "Add Station" button, and at the bottom right are "OK" and "Help" buttons.

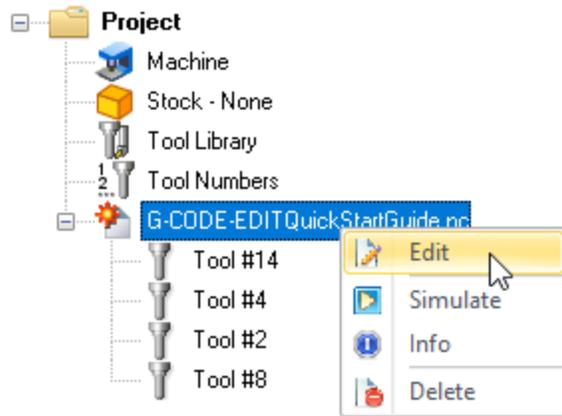
Stn #	Name	Type	Diameter	Corner Radius	Taper	Length	Material	Comments
1	FLATMILL-1/2 ...	Mill	0.5	0	0	4	HSS	
8	VEE-45 DEG	Mill	0	0	45	1	HSS	
14	FACEMILL-2.5 ...	Mill	2.5	0	0	2	HSS	
2	DRILL 1/2 IN	Drill	0.5	0	0	4	CARBIDE	

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](#).

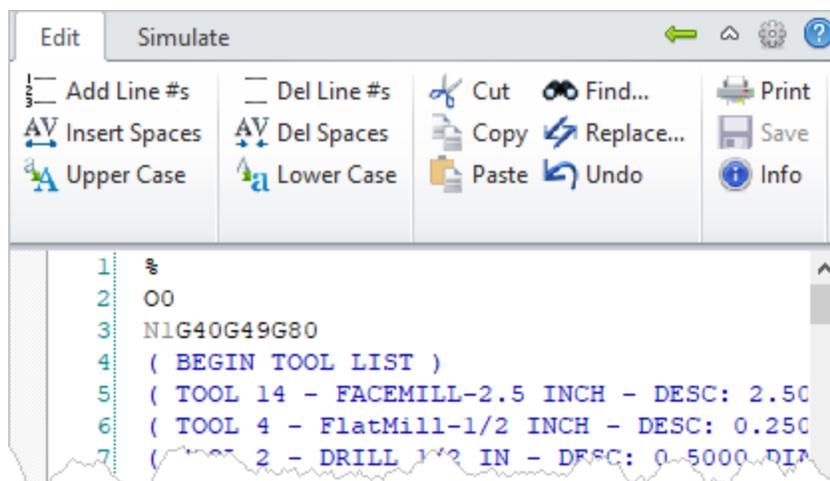


5.5 Add Line Numbers

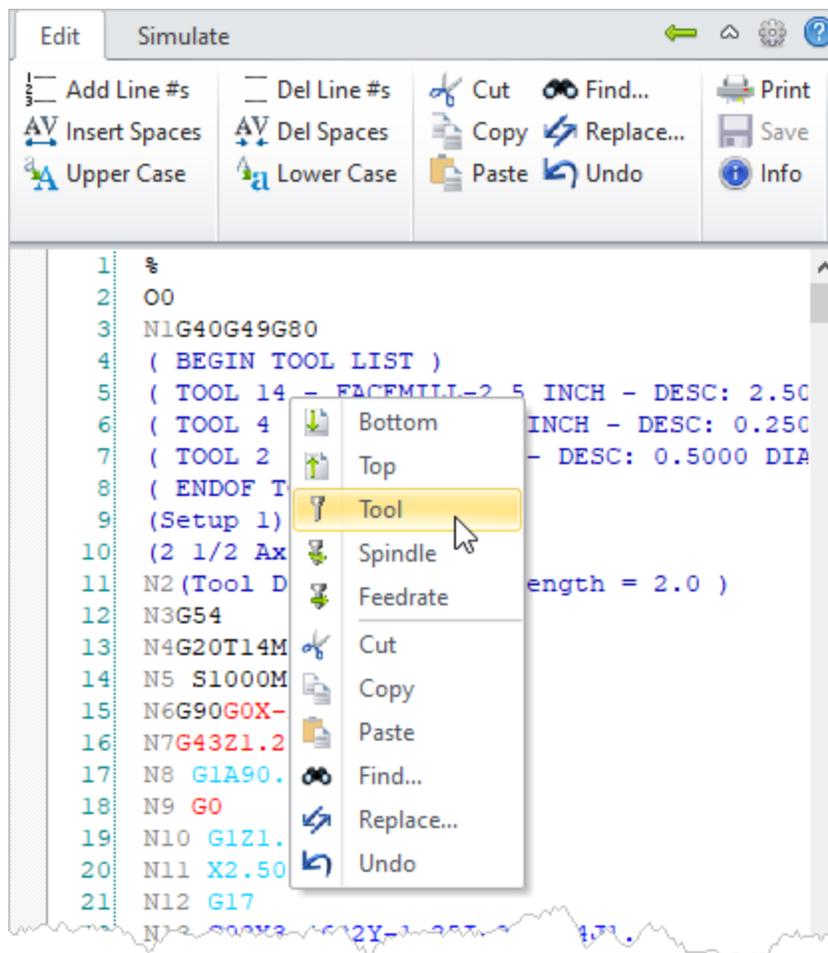
1. Right-click on the selected G-Code file located in the [Project](#) tree and select [Edit](#).



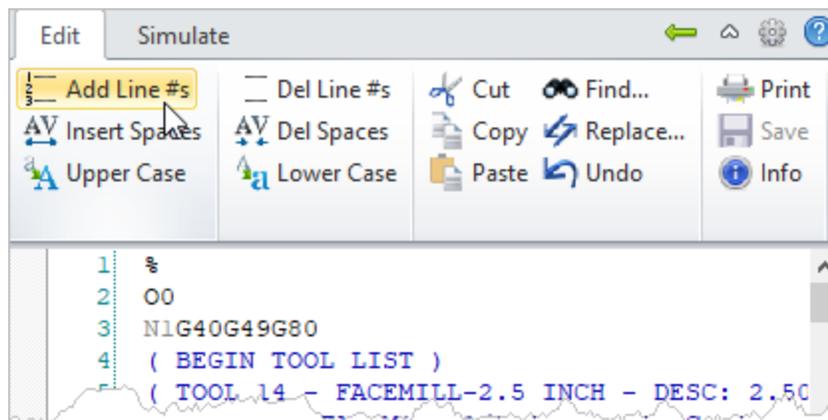
2. You will see that the [Project](#) tab is replaced with [Edit](#) & [Simulate](#) tabs and the G-Code file is loaded into the editor as shown below.



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.

```

1  %
2  N1 O0001
3  N2 G40G49G80
4  (Setup 1)
5  (2 1/2 Axis Facing)
6  (Tool Diameter = 2.5 Length = 2.0 )
7  N3 G54
8  N4 G20T14M6
9  N5 S5000M3
10 N6 G90GOX-3.7511Y-1.625

```

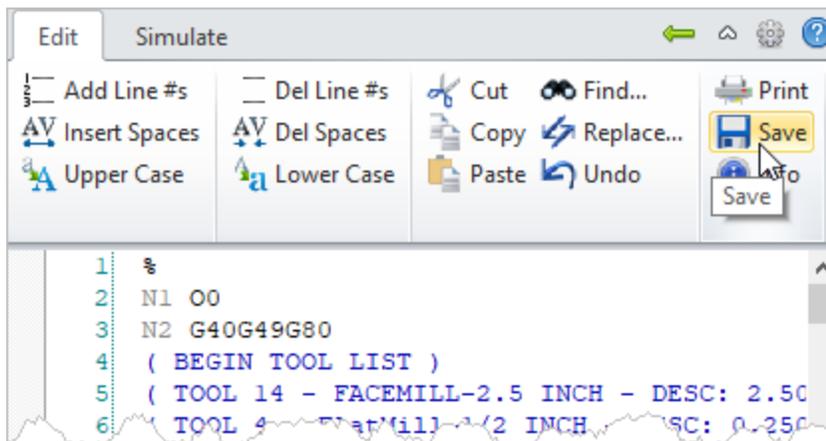


Line Numbers added to the G-Code file.

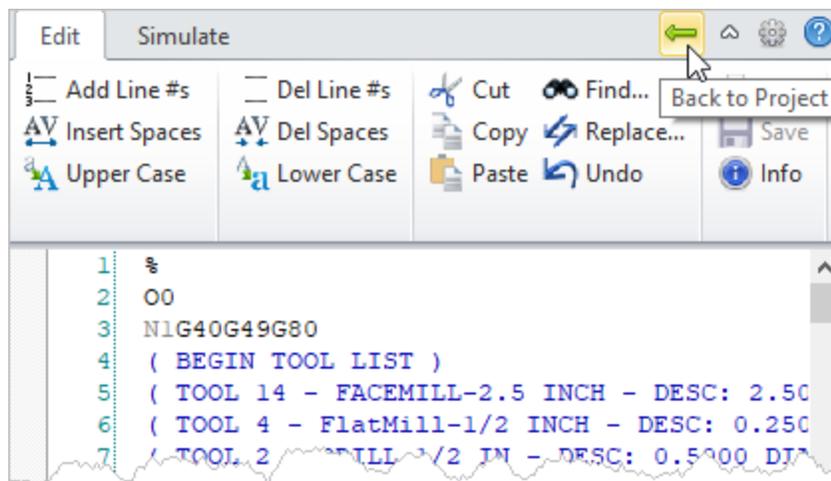
5.6 Saving Changes

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

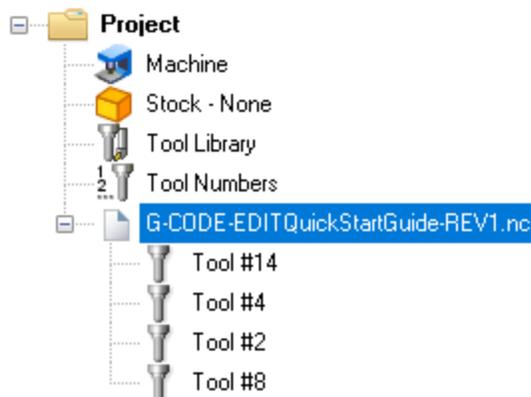
1. To save and write over your original G-Code file, select  Save from the [Edit](#) tab.

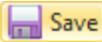


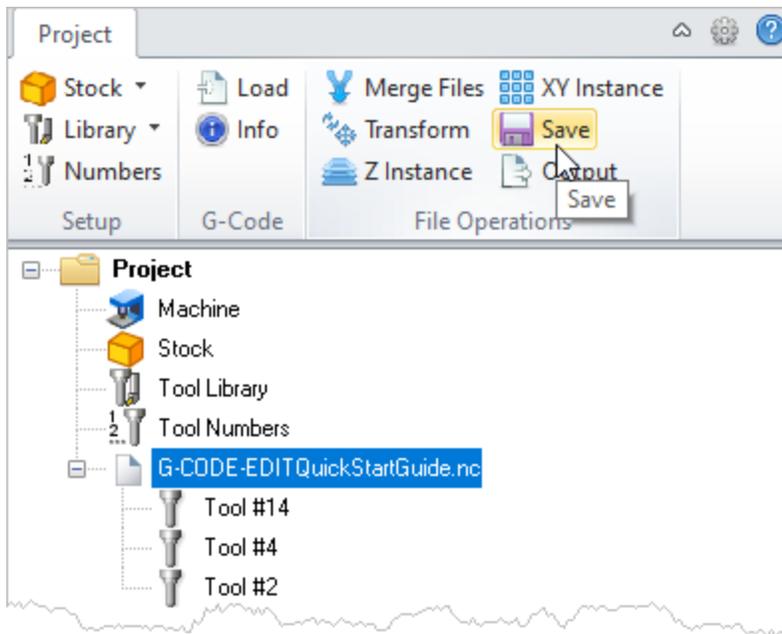
2. Pick [OK](#) to save over the original G-Code file. If you do not want to overwrite your original file, select [Save As](#) from the Save menu and enter a different file name and pick [OK](#).
3. Now return back to the [Project](#) tab.



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  Save from the Project tab menu.

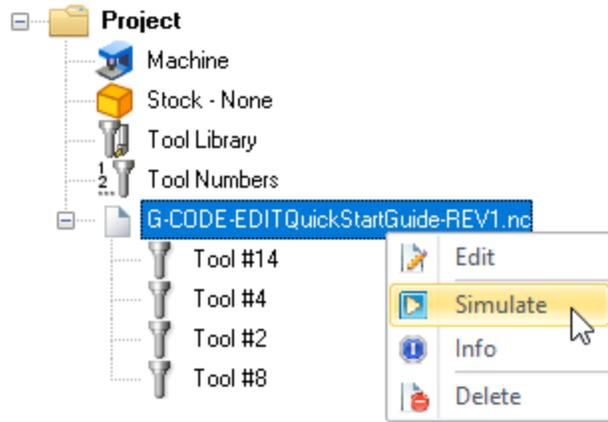


G-Code Simulations

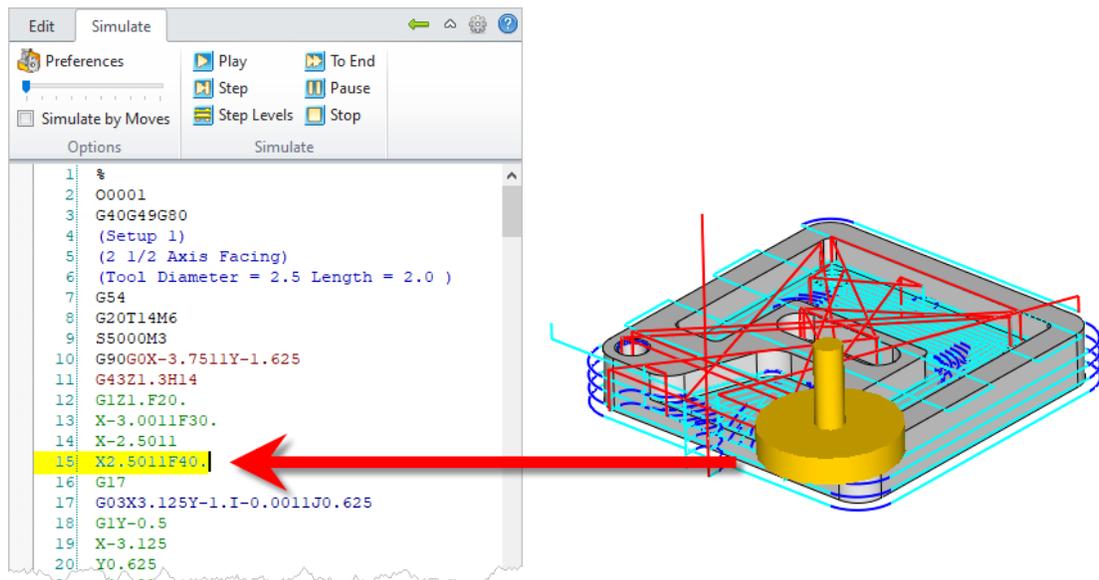
6.1 Tool Motion Simulation

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.

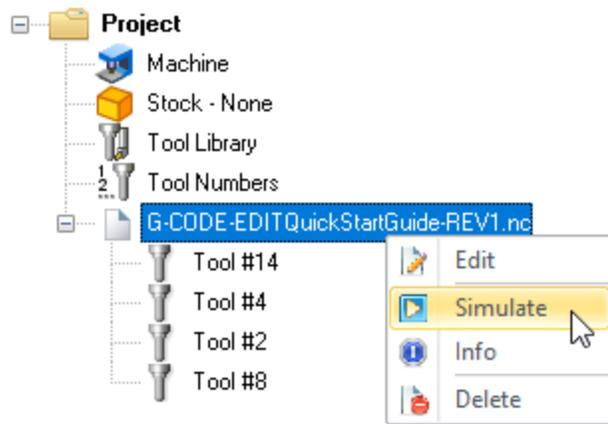


- B. During the tool motion simulation, you can use the icons on the [Simulate](#) tab to [Pause](#), [Stop](#) and [Start](#) the simulation.
- C. Once the simulation is [Paused](#), [Stopped](#) or reaches the end of the G-Code file, you can then manually move the cursor to any line in the G-Code file 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

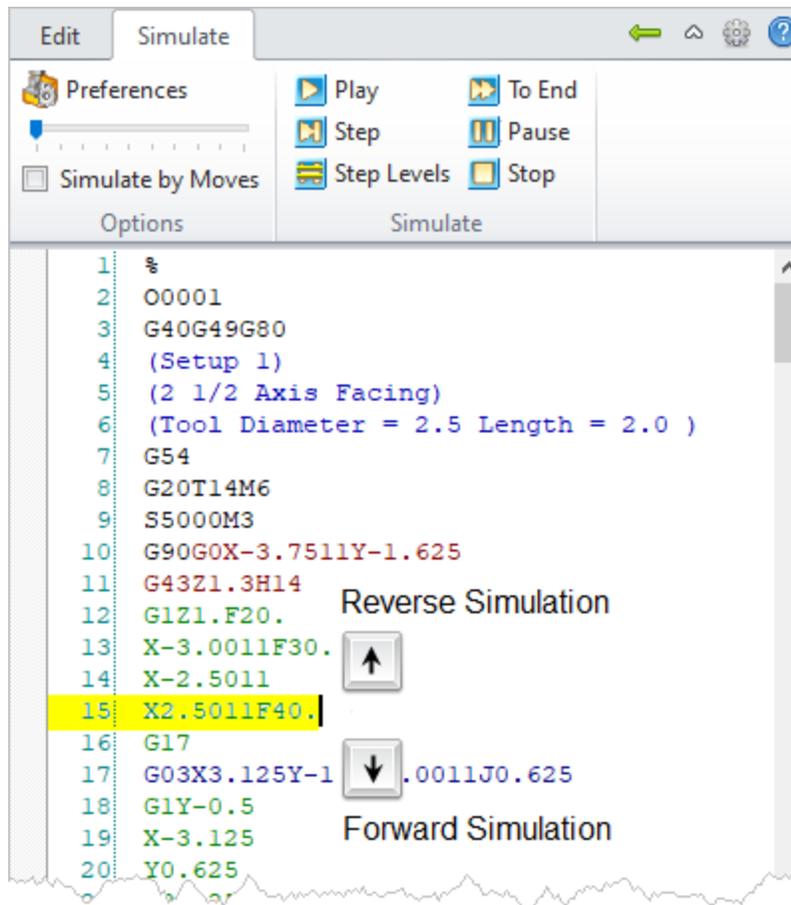
The Simulate tab provides commands for running simulations and access to simulation preferences. Here is a brief description of the [Simulate](#) tab. These commands work in the same manner as those in the [MILL](#) module.

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.

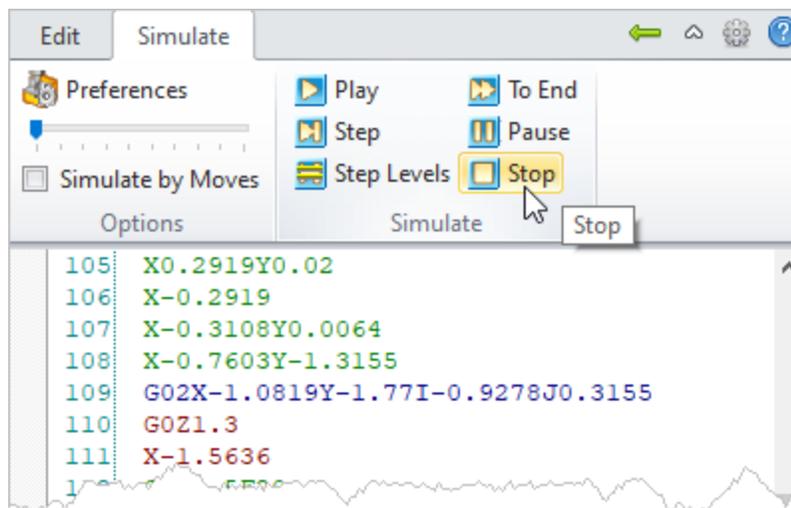
Optionally you can [Pause](#) the simulation and then use the [Up](#) and [Down Arrow Keys](#) on your keyboard to move the simulation [Forward](#) or [Reverse](#) line by line.



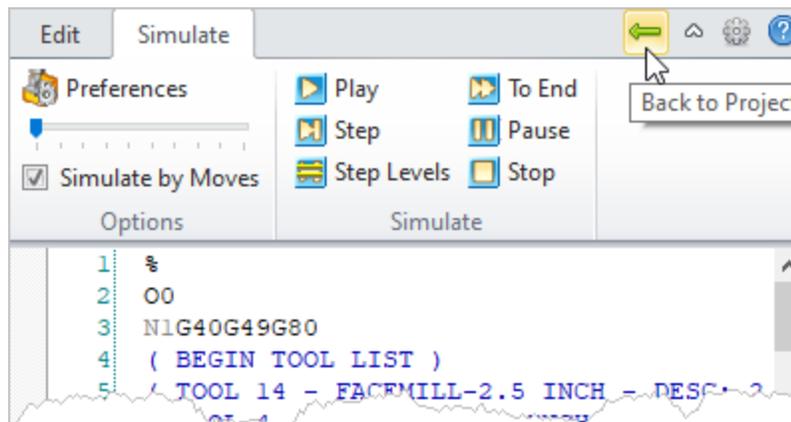
6.3 Define Your Stock

In order to run a [Cut Material Simulation](#) you must have a Stock defined.

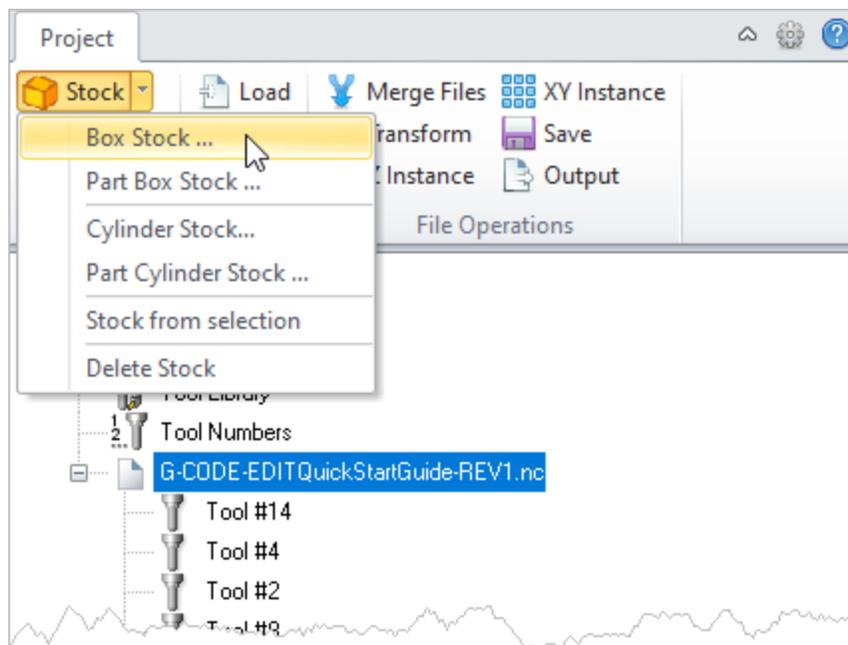
1. [Pause](#) or [Stop](#) any simulation that is currently running by selecting the appropriate command from the Simulate tab.



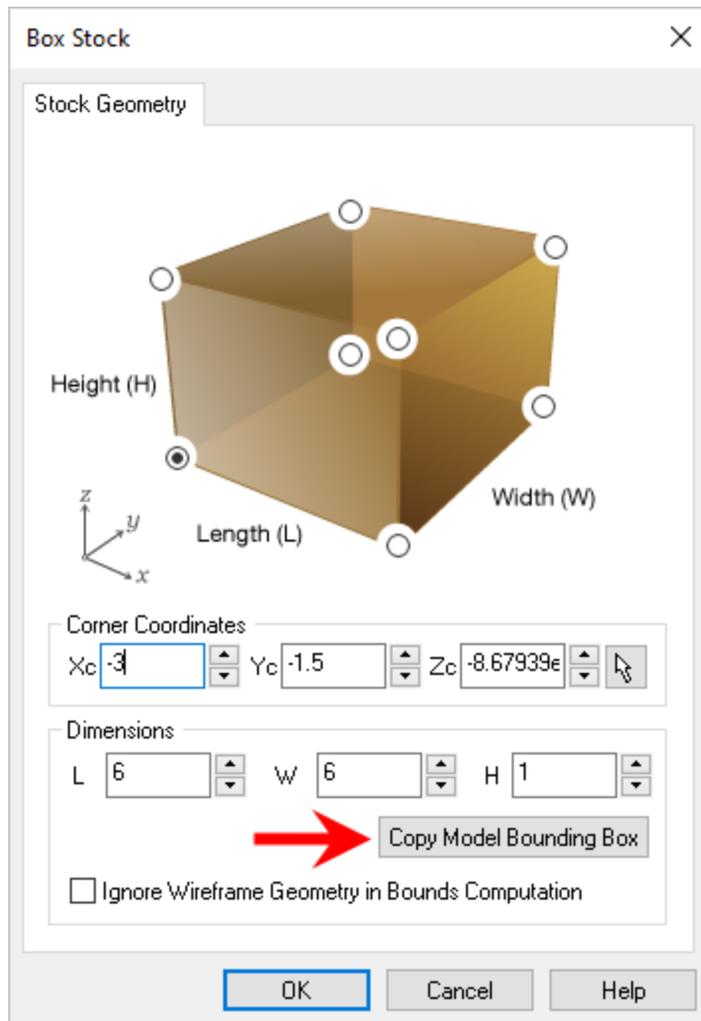
2. Return to the [Project](#) tab.



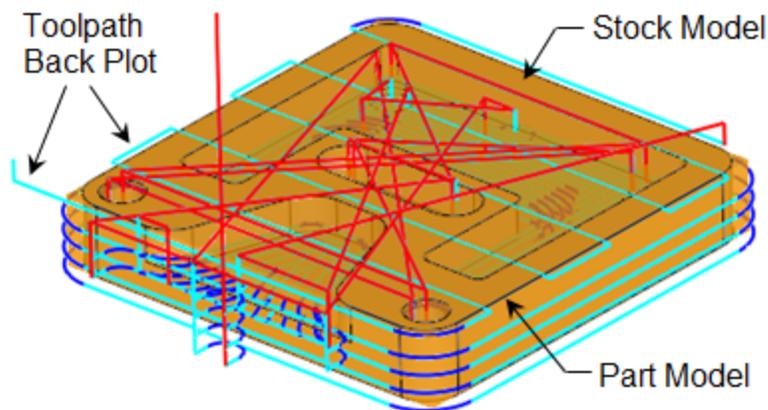
3. From the [Project](#) tab select  **Stock** and then select **Box Stock ...** from the menu.



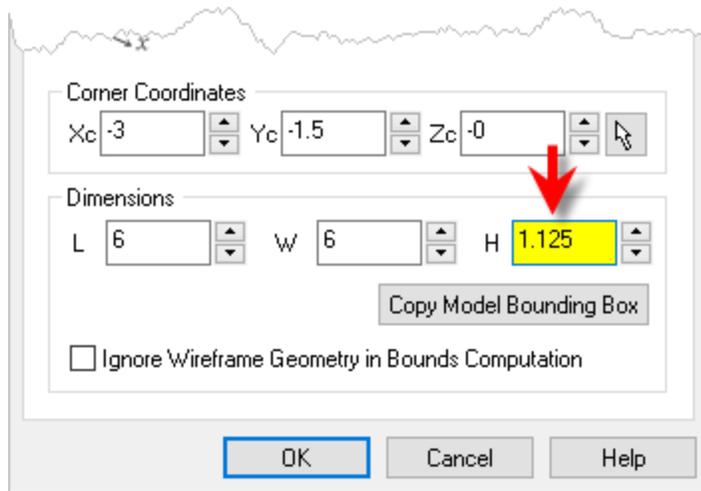
4. This will display the Box Stock dialog similar to the MILL module.



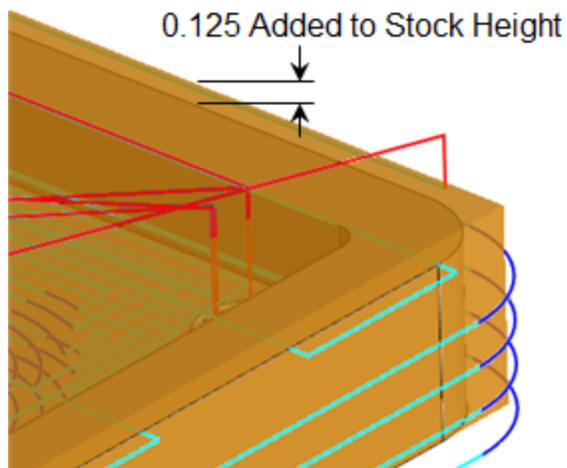
5. Select the **Copy Model Bounding Box** button. The stock model will be displayed in the graphics window that matches the size of the part model.



- 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.



- Now pick OK to close the Box Stock dialog.

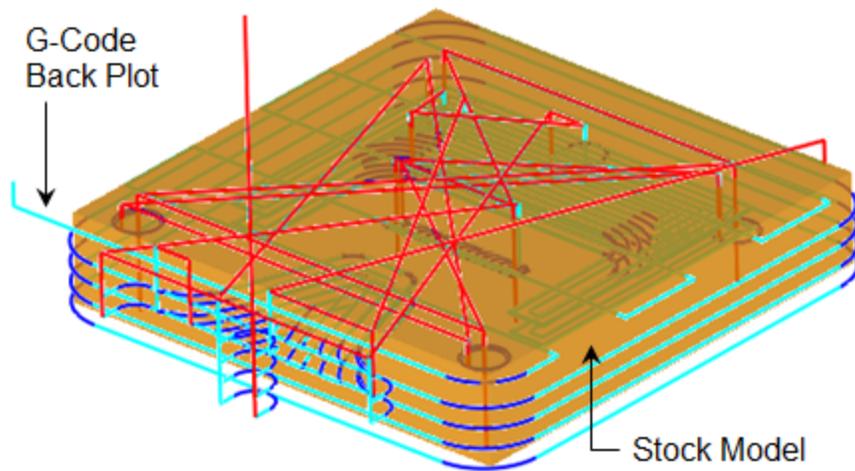


6.4 Cut Material Simulation

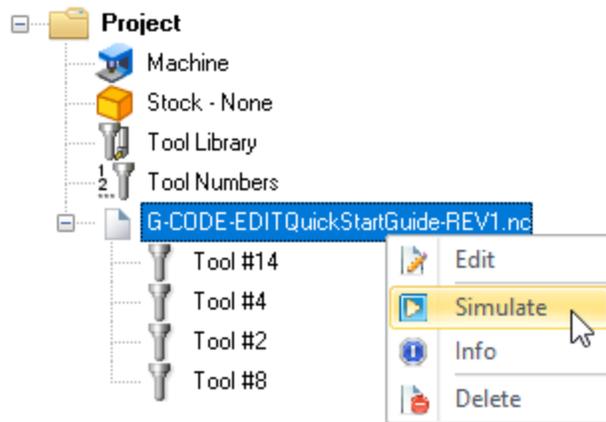
With our Stock defined we can now perform a Cut Material Simulation. With Stock defined, selecting Simulate will now display the in-process cut stock at each line in the G-Code file!

- 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.

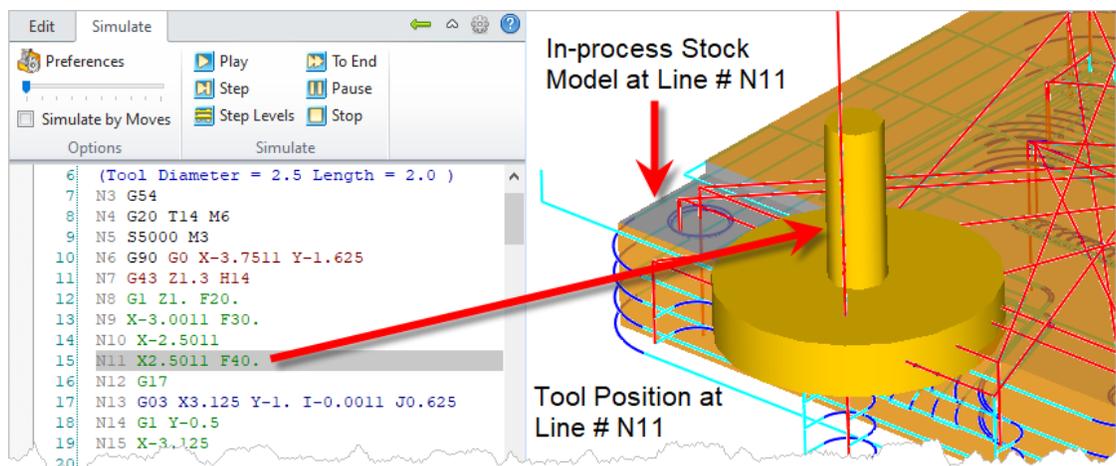
Now only the Stock model and the G-Code backplot are displayed in the graphics window.



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



- 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.

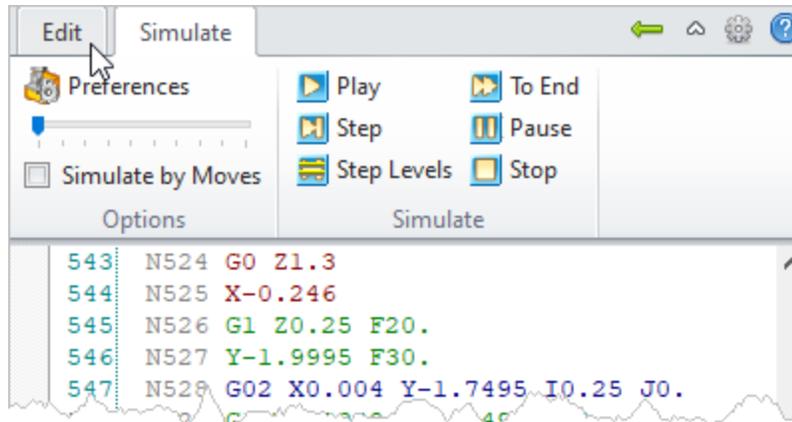


More G-Code Edits

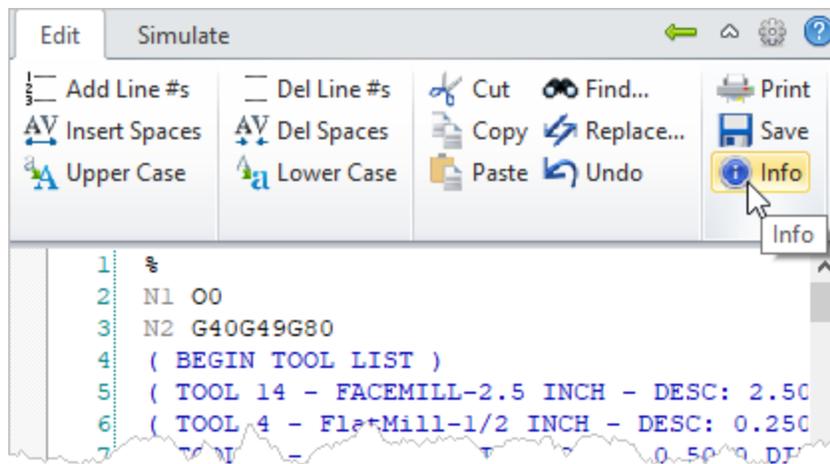
7.1 Estimate Machining Time

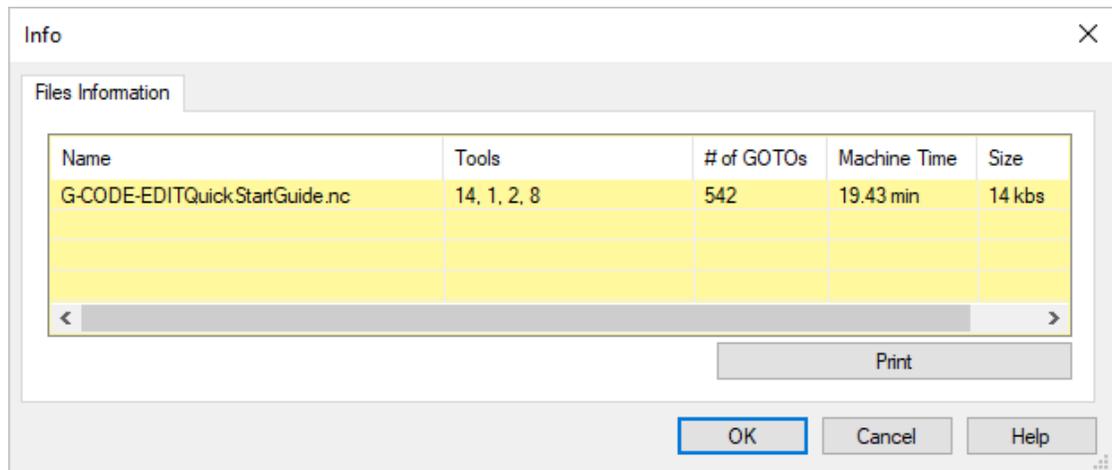
You can get additional Information about the G-Code files currently loaded into the [G-CODE Editor](#).

1. Select the [Edit](#) tab to return to the [Edit](#) menu.

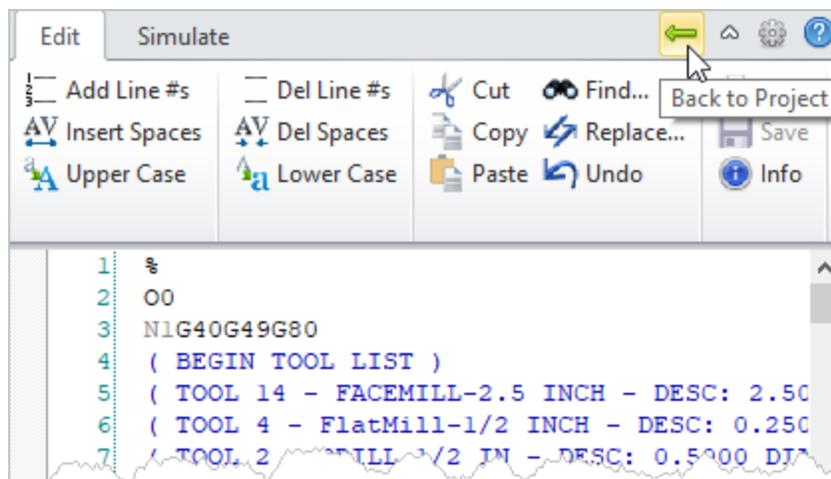


2. Now select  from the [Edit](#) menu to display the [Information](#) dialog.

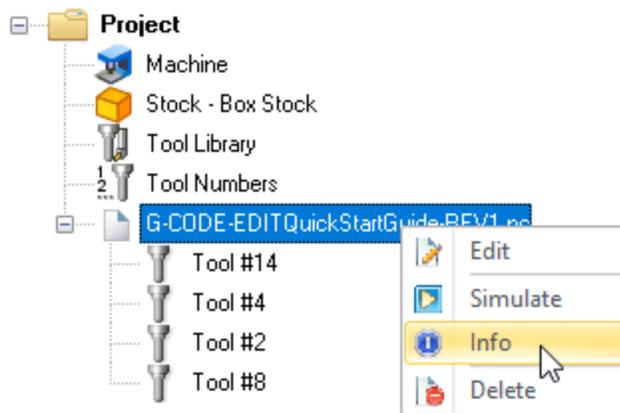




3. You can also perform this task from the **Project** tab. Return to the **Project** tab, select the G-Code file and then select **Info** from the **Project** menu to display the **Information** dialog.



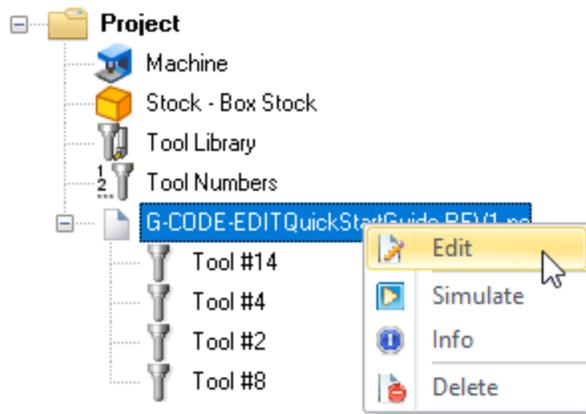
You can also right-click on the G-Code file and select **Info** to display the **Information** dialog.



7.2 More Edits

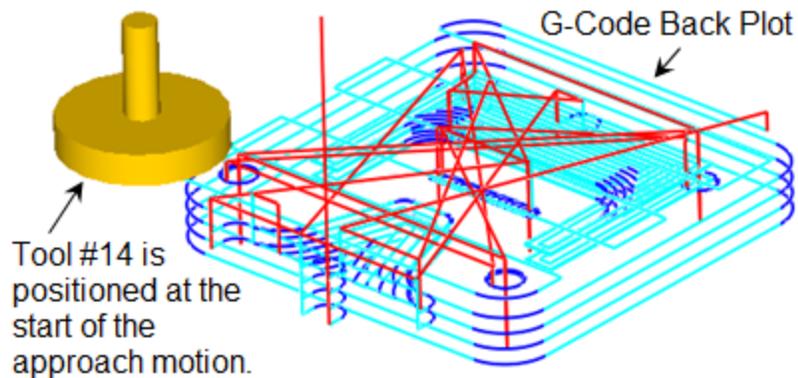
Let's perform one additional edit before we complete this guide.

1. Right-click on the selected G-Code file located in the [Project](#) tree and select [Edit](#).

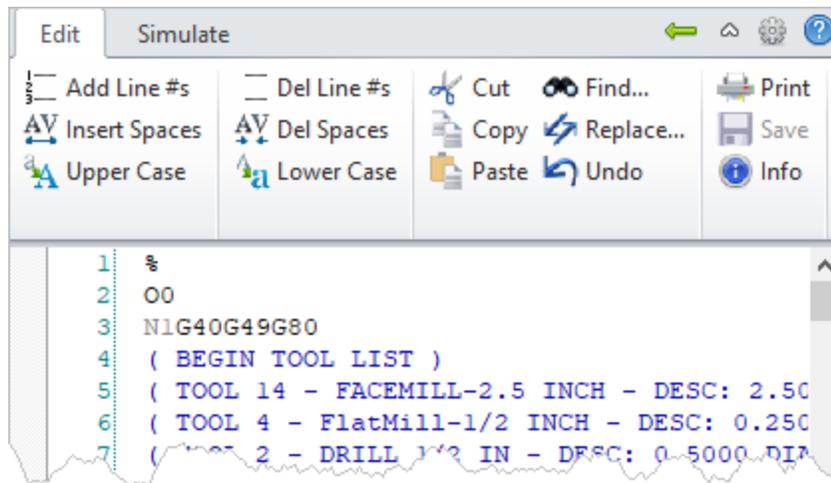


1. If you recall back in the section [Add Line Numbers](#), you were warned that no appropriate tools were found. That's because we did not have a [Tool Library](#) and [Tool Crib](#) loaded yet.

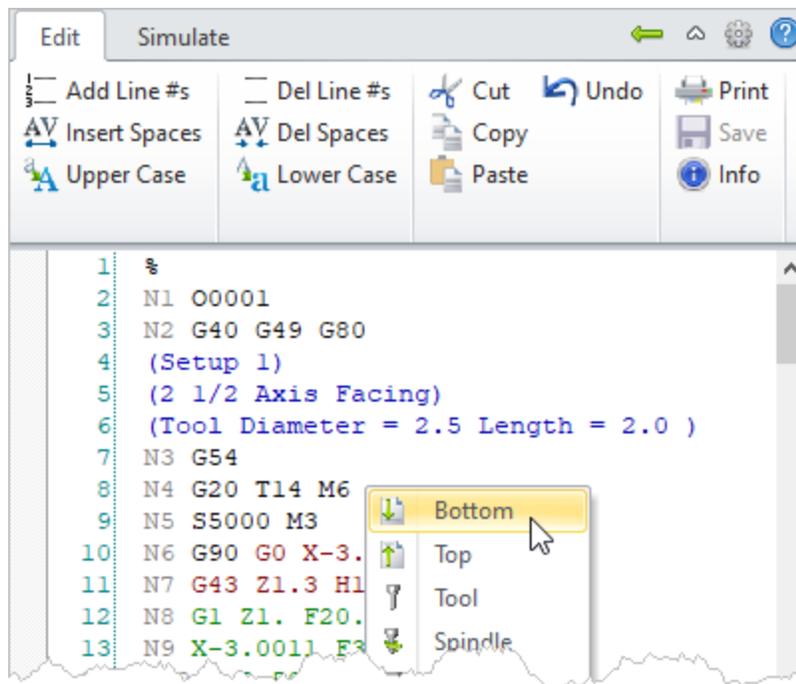
Now when you go to the [Edit](#) tab, the first tool # is loaded and displayed in the graphics screen at the beginning of the first tool motion.



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.

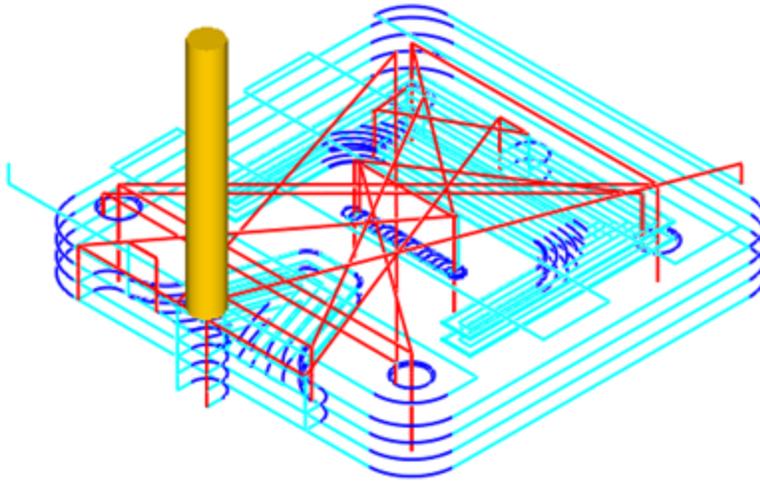


- One edit that many users find useful is to retract the tool to a higher Z height once cutting is completed. From the **Edit** tab right-click on any line in the G-Code file and select **Bottom** to go directly to the end of the file.



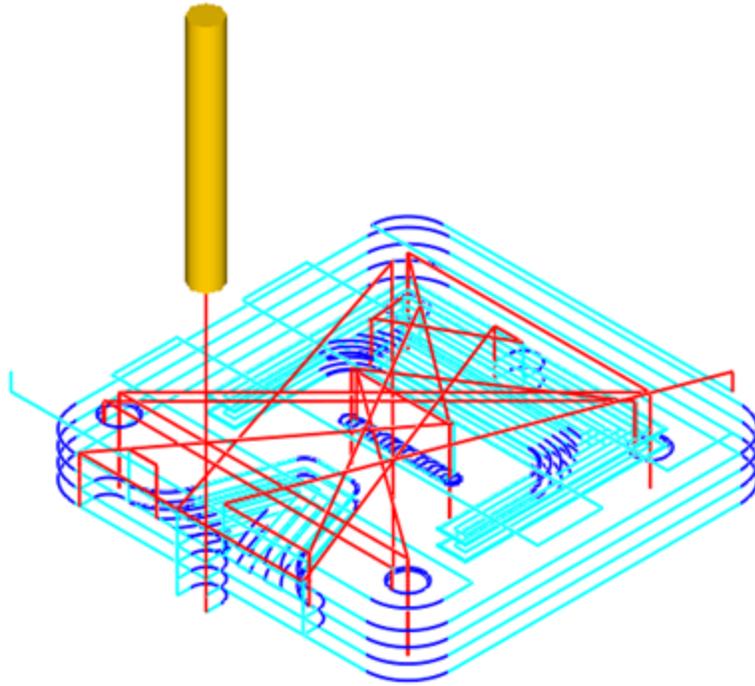
- Now place the cursor on the last G0 line and notice where the tool is position in the graphics screen.

```
577 N555 G03 X-3.248 Y3.956 I0.7504 J0.011
578 N559 G1 X0.004
579 N560 G02 X0.254 Y-1.9995 I0. J-0.25 F50.
580 N561 G1 Y-2.2495 F60.
581 N562 G0 Z1.5
582 N563 M30
583 %
```



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

```
574 N555 G03 X-3.248 Y3.956 I0.7504 J0.011
575 N556 G1 X-3.2505 Y3.956
576 N557 X-3.25 Y-1.011
577 N558 G03 X-2.5146 Y-1.7495 I0.7504 J0.011
578 N559 G1 X0.004
579 N560 G02 X0.254 Y-1.9995 I0. J-0.25 F50.
580 N561 G1 Y-2.2495 F60.
581 N562 G0 Z5.0
582 N563 M30
583 %
```



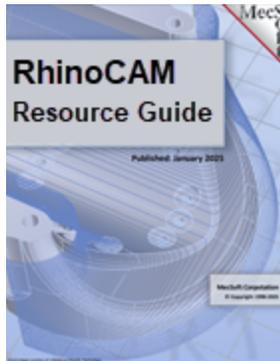
6. Refer to the section [Saving Changes](#) to update your external G-Code file with all of the latest changes.

Where to go for more help

Download this PDF Guide for a list of the available [RhinoCAM Resources](#).



2026 RhinoCAM Resource Guide



The 2026 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 -

About

- the G-Code Editor Module 9
- the RhinoCAM Display 12
- This Guide 8
- using this Guide 9

Add Character Spaces 21

Add Line Numbers 28

- E -

Edit

- Add Character Spaces 21
- Add Line Numbers 28
- Tool Retract Z Height 43

- L -

Load

- the G-Code Edit Module 13
- the G-Code Files 14

- M -

Machining Time Estimates 41

- Q -

Quick Start 5

- R -

Resource Guide 7

Running RhinoCAM 12

- S -

Saving Changes 30

Simulations

- Cut Material Simulations 39

Define Stock for 35

In-Process Stock During 39

The Simulate Tab 34

Tool Motion 33

Strategy 19

- T -

Tools

Define a Tool Crib 25

How that are Used 21

Load a Tool Library 23

- U -

Useful Tips 8

Using this Guide and Associated Part Files 9

- W -

Where to go for more help 47