

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

Reference 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 Corporation

© Copyright 1998-2025

Table of Contents

Welcome	5
1 Overview.....	5
2 Features.....	6
3 Understanding.....	7
4 Work Flow.....	7
Quick Start	9
CAM Resource Guide	11
User Interface	12
1 Preferences.....	12
User Interface	13
Text Editor	16
Toolpath	17
Stock	19
Cutting Tools	20
Simulation	21
2 G-Code Browser.....	23
Project Tab	26
Project Tree	28
Edit Tab	31
Simulate Tab.....	35
Online Help	37
Project Tab	39
1 Machine MCS Setup.....	41
2 Setup Pane.....	45
Stock	45
Box Stock.....	46
Part Box Stock.....	49
Cylinder Stock.....	51
Part Cylinder Stock.....	54
Delete Stock.....	57
Stock from Selection.....	58
Tool Library	60
Tool Crib	62
3 G-Code Pane.....	64
Load	64
Info	65
4 File Operations Pane.....	67
Merge Files	67
Transform	71

Z Instance	77
XY Instance	79
Save	81
Output	83
5 Right-Click Menu.....	84
Edit-G-Code-File	84
Simulate G-Code File	85
Info on G-Code File	87
Delete G-Code File	88

Edit Tab 90

1 Add Line Numbers.....	93
2 Insert Spaces.....	95
3 Make Upper Case.....	96
4 Delete Line Numbers.....	97
5 Delete Spaces.....	99
6 Make Lower Case	100
7 Cut Text.....	101
8 Copy Text.....	103
9 Paste Text.....	104
10 Find Text.....	106
11 Find & Replace Text.....	107
12 Undo Last Edit.....	109
13 Print G-Code	111
14 Save G-Code.....	112
15 Save As.....	113
16 Show G-Code Info.....	114
17 Select Full Line.....	116
18 Right-Click Menu.....	117
Go to Bottom	117
Go to Top	118
Go to Next Tool	119
Go to Next Spindle	121
Go to Next Feedrate	122
Cut Text	123
Copy Text	125
Paste Text	126
Undo Last Edit	128

Simulate Tab 130

1 Simulate Options.....	132
Simulation Preferences	132
Simulation Speed	134
Simulation Mode	134
2 Simulation Controls.....	135
Play Simulation	135

Step Thru Simulation	136
Display Simulation Levels	137
Simulate to End	138
Pause Simulation	139
Stop Simulation	140
Functions from the MILL Module	142
Find More Resources	144
Index	151

Welcome



G-CODE Editor Module 2025

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

Welcome to the [RhinoCAM-G-Code Editor Online Help](#)! This help system contains helpful information about every command and feature of the [RhinoCAM-G-Code Editor](#). [G-CODE Editor](#) can assist you with editing g-code files created with [RhinoCAM](#) or from other sources. It can also be used to navigate and perform cut material simulations of g-code files. Select a topic from the [Contents](#), [Index](#) or [Search](#) tabs located on the left. You can also select a Related Topic from below to get started. The [Features](#) topic contains links to every command in the [RhinoCAM-G-Code Editor](#)!

For additional assistance and resources we invite you to visit the [Find More Resources](#) topic. We hope you enjoy the [RhinoCAM-G-Code Editor](#)!



Related Topics

[Overview](#)

[Features](#)

[Understanding](#)

[Work Flow](#)

1.1 Overview

[G-CODE Editor](#) is a companion module that runs inside of [RhinoCAMRhinceros](#). [G-CODE Editor](#) assists users with editing g-code files created with [RhinoCAM](#) or from other sources. It can be used to navigate and perform cut material simulations of g-code files.



Related Topics

[RhinoCAM-G-Code Editor](#)

[Features](#)

[Understanding](#)

[Work Flow](#)

1.2 Features

The following features are included in the [RhinoCAM-G-Code Editor](#) module.



Project

- [Machine MCS Setup](#)
- [Preferences](#)
- [Stock](#)
- [Tool Library](#)
- [Tool Stations](#)
- [Load G-Code File](#)
- [Project Information](#)
- [Merge Files](#)
- [Transform Project](#)
- [Z Instance Project](#)
- [XY Instance Project](#)
- [Save Project](#)
- [Output Project](#)



Editing (Right-Click Menu)

- [Cut/Copy/Paste](#)
- [Undo Last Command](#)
- [Print G-Code File](#)
- [Save File](#)
- [Go to Bottom](#)
- [Go to Top](#)
- [Go to next Tool](#)
- [Go to next Spindle](#)
- [Go to next Feedrate](#)
- [Find & Replace Text](#)



Editing (Ribbon Bar Menu)

- [Add Line Numbers](#)
- [Delete Line Numbers](#)
- [Insert Delimiter Spaces](#)
- [Delete Delimiter Spaces](#)
- [Make all Upper Case](#)
- [Make all Lower Case](#)
- [G-Code Information](#)
- [Find & Replace Text](#)



Simulation

- [Simulation Preferences](#)
- [Simulation Speed](#)
- [Simulation Mode](#)
- [Play Simulation](#)
- [Display Simulation in Levels](#)
- [Simulate to End](#)
- [Pause Simulation](#)
- [Stop Simulation](#)



Related Topics

[Welcome](#)

[Overview](#)

[Understanding](#)

[Work Flow](#)

1.3 Understanding

The [RhinoCAM-G-Code Editor](#) module can edit g-code files that you create with [RhinoCAM](#). It can also be used to edit g-code files generated from other systems that produce ISO standard g-code files. You can also define tool libraries, tool cribs and perform cut material simulations.



Related Topics

[RhinoCAM-G-Code Editor](#)

[Overview](#)

[Features](#)

[Work Flow](#)

1.4 Work Flow

The following is a basic procedural work flow for using the [RhinoCAM G-CODE Editor](#) module:



Step 1: Load a G-Code Files

Run the [RhinoCAM G-CODE Editor](#) module and Load the g-code file or files using the commands on the [Project](#) tab. Your g-code files will be listed in the [Project](#) tree.



Step 2: Edit G-Code Files

Edit the G-Code file using the Edit tab functions. You can perform many automatic and manual editing tasks, save and print your G-Code files.



Step 3: Tool Motion Simulations (Optional)

To perform tool motion simulations, simply place the cursor at the top of the G-Code file and press the down arrow key and the tool motion will display on the graphics screen.



Step 4: Cut Material Simulations (Optional)

To perform a cut material simulations, load a [Tool Library](#) generated from [RhinoCAM](#), setup your [Tool Crib](#) with the tool numbers used in your g-code files. Then return to the [Project](#) tab, right-click on the G-Code file and select [Simulate](#).



Step 5: Save your G-Code Files

When you are done editing your g-code you can [Save](#) the g-code files or [Output](#) them directly to another program, of your choice such as a [DNC](#) program to communicate them directly to your CNC machine. See [Preferences](#) > [Text Editor](#) to define your output program.



Related Topics

[Welcome](#)

[Overview](#)

[Features](#)

[Understanding](#)

Quick Start

[Quick Start Guides](#) for each [RhinoCAM](#) module are available in both PDF and Video format. Refer to the following information to access these guides:



What's New!

[What's New in RhinoCAM 2025](#)



The Complete Quick Start Video Play List

[Here is a link to the complete 2025 Video Play List](#)



How to Access the Quick Start Guide Documents

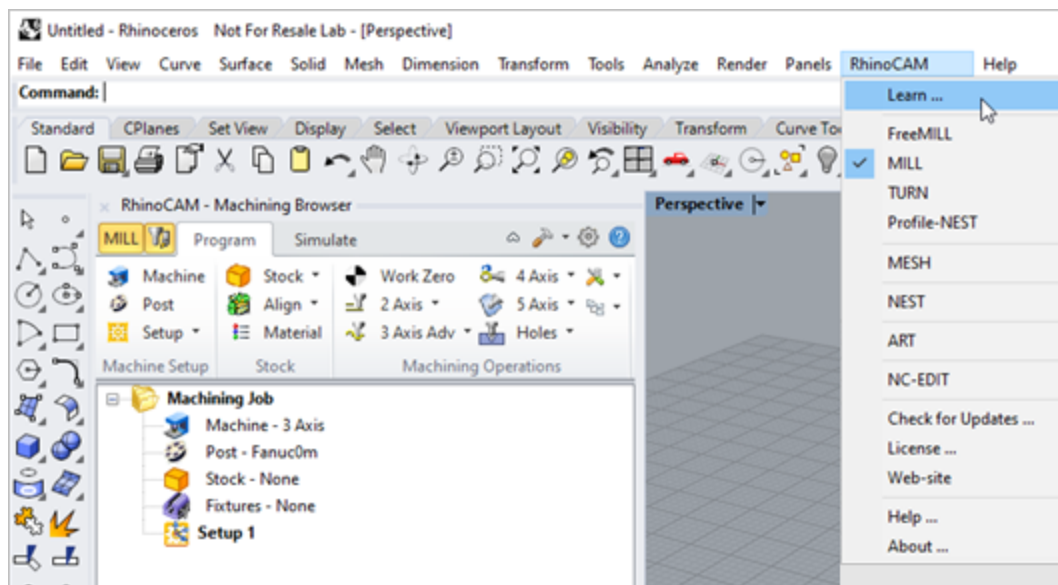
To help you quickly get started in working with each module, select one of the Help buttons located on the [RhinoCAM Learning Resources](#) dialog.

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:

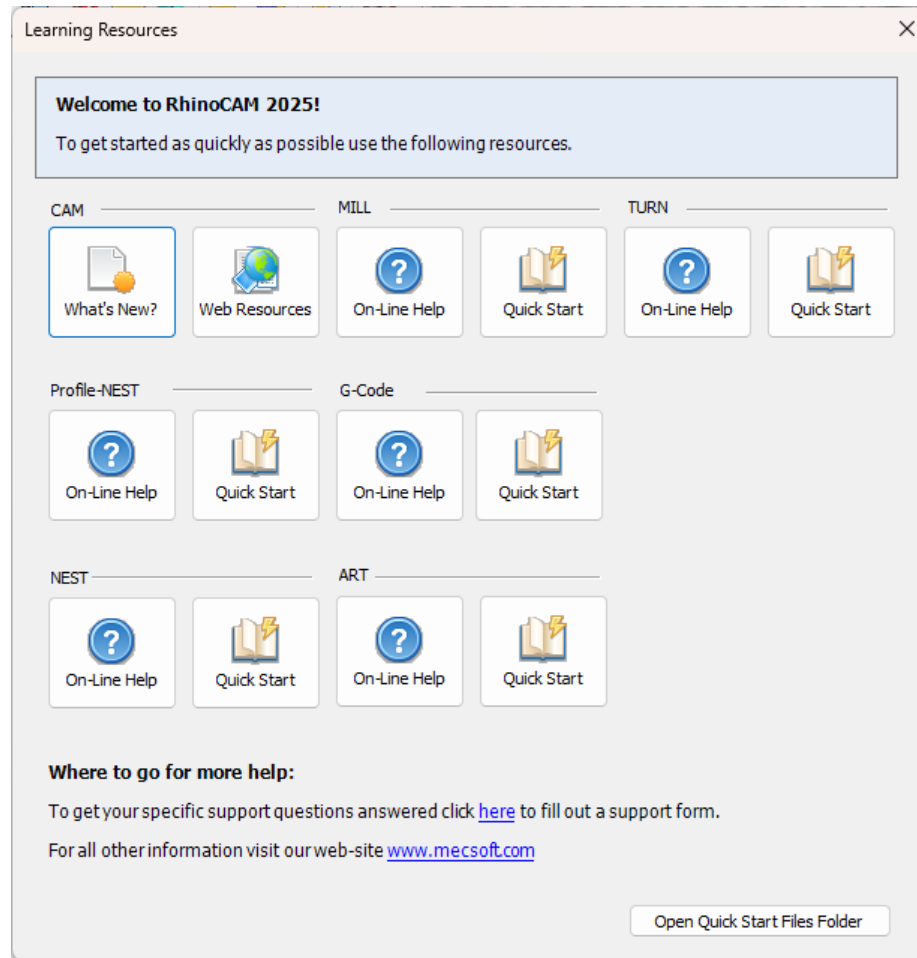
1. From the [Rhino Main Menu](#), drop down the Main menu and select [Learn ...](#)



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.



Learning Resources Dialog



Related Topics

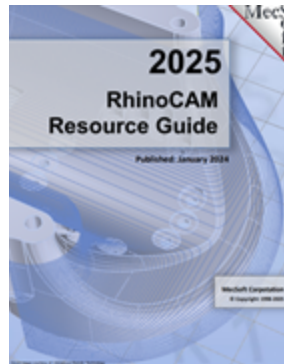
[Find More Resources](#)

CAM 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.

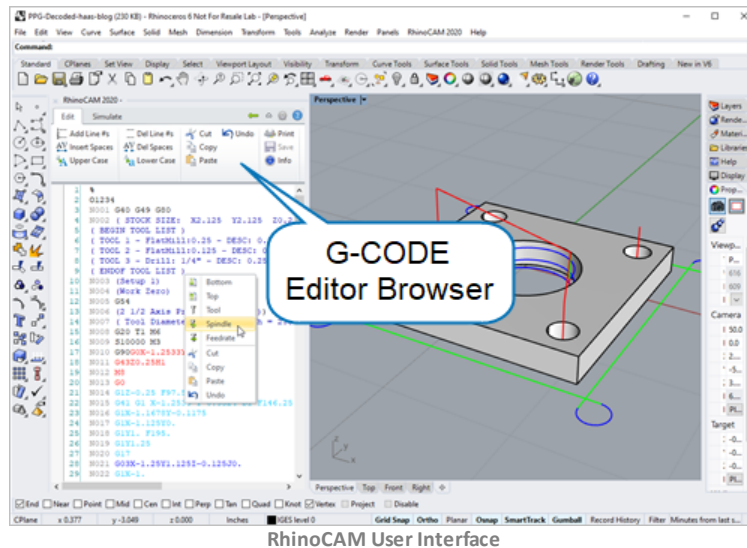
[Prefer Printed Documentation? Check Here!](#)

User Interface

RhinoCAM-G-CODE Editor adheres to the Windows standard for user interface design and is seamlessly integrated into the **Rhinoceros SolidWorksAlibre Design** screen layout. A screen image of **RhinoCAM-G-CODE Editor** running inside of **RhinocerosSolidWorksAlibre Design** is shown below:



The User Interface



RhinoCAM User Interface



Related Topics

[Preferences](#)

[G-Code Browser](#)

[Project Tab](#)

[Edit Tab](#)

[Simulate Tab](#)

4.1 Preferences



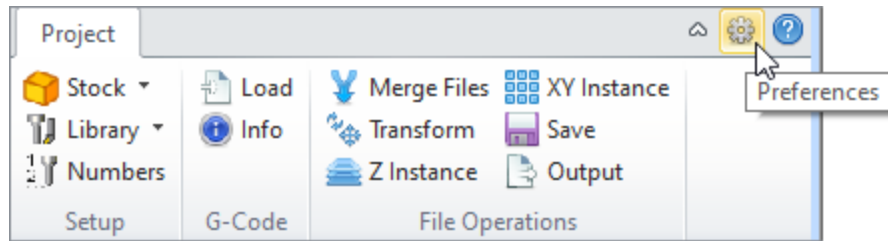
You can set various **System Preferences** that will be saved even after you exit the program. Select the **Preferences** icon from the **G-Code Browser**. When you install a new **RhinoCAM** update you can choose to import your **Preferences** from one version to the next.



To Access this Command:

click: 

The Preferences Icon



Set System Preferences menu item

Related Topics

[User Interface](#)

[System Preferences](#)

[Text Editor Preferences](#)

[Toolpath Preferences](#)

[Stock Preferences](#)

[Cutting Tools Preferences](#)

[Simulation Preferences](#)

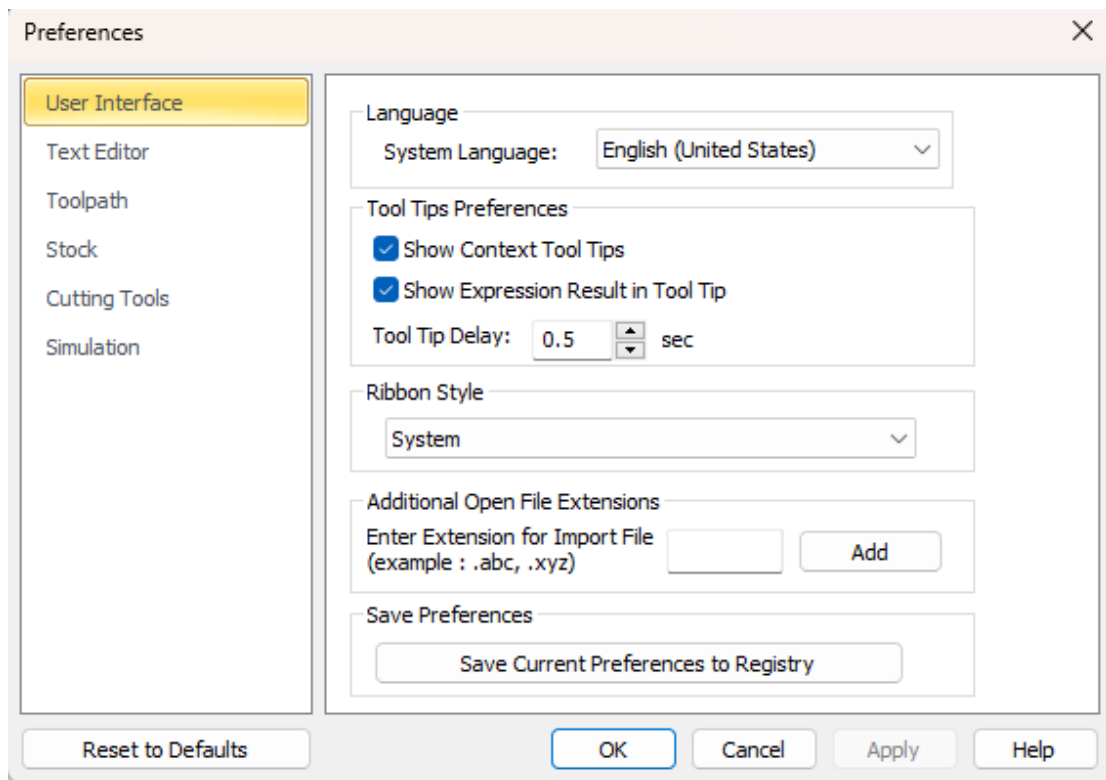
4.1.1 User Interface

Here you can set preferences related to the g-code text editor including colors and associated programs and folders. You can use the [Reset to defaults](#) button if you want to revert to the default factory install settings.

To Access this Command:

click:  > User Interface

Preferences > Text Editor



Preferences > User Interface



Language

System language

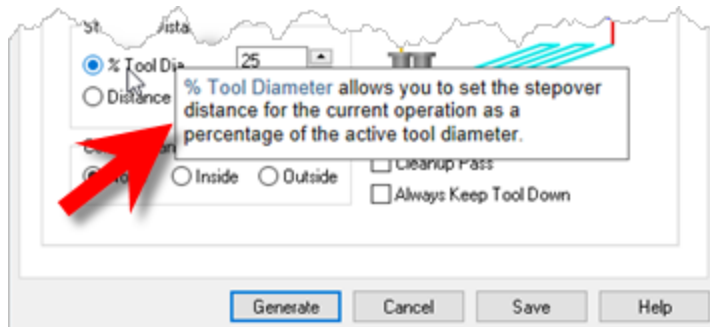
Use this to set the system language of the plugin's User Interface.



Tool Tips Preferences

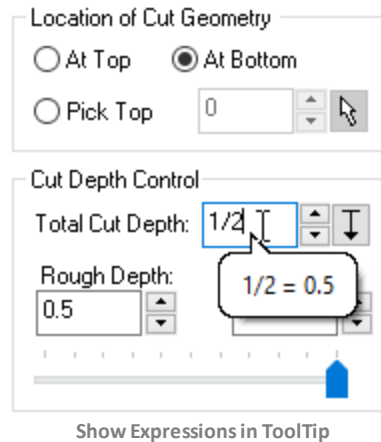
Show Context Tool Tips

Check this box to display [Context ToolTips](#) when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. **Note** that [Context ToolTips](#) may not be available for ALL dialogs. You can also set the [ToolTip Delay](#) in seconds. This is the amount of time it takes to display the [Context ToolTip](#) when the mouse activate it.



Show Expression Results in Tool Tips

You can enter expressions in any dialog field that expects a numerical value and the value will be computed and entered automatically. Check this box to pop-up the results of any expressions in a **ToolTip** balloon. An example is shown below.



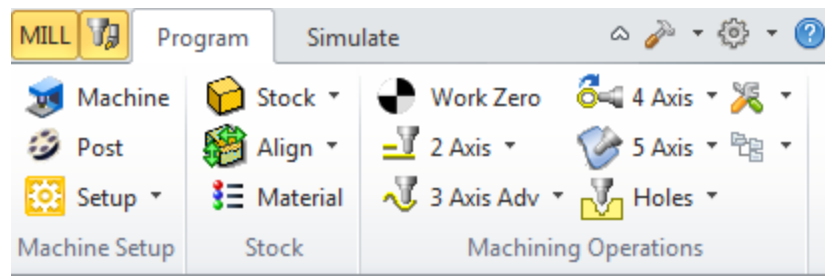
Tool Tip Delay

When **Show context ToolTips** is checked, this value defines the **Delay** in seconds it takes for a **ToolTip** to display once the cursor moves over it.



Ribbon Style

This allows the selection of different themes that change how the **Browser** windows appear. The borders, colors, highlighting, and shadowing of standard buttons, dialogs, and windows are controlled by which theme is selected.

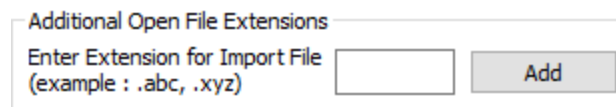


Example Ribbon Style: Office 2010 Silver



Additional Open File Extensions

If your g-code files have an extension other than ***.nc** you can add that extension here. Enter the extension in the field provided and then pick the **Add** button.



Save Preferences

Save Current Preferences to Registry

If you have your preferences set the way you want them and do not want them to change, select this button to save the current preferences to your Windows registry. Doing this will force them to be loaded when you create new files.



Related Topics

[Text Editor Preferences](#)

[Toolpath Preferences](#)

[Stock Preferences](#)

[Cutting Tools Preferences](#)


[Simulation Preferences](#)

4.1.2 Text Editor

Here you can set preferences related to the g-code text editor including colors and associated programs and folders. You can use the [Reset to defaults](#) button if you want to revert to the default factory install settings.

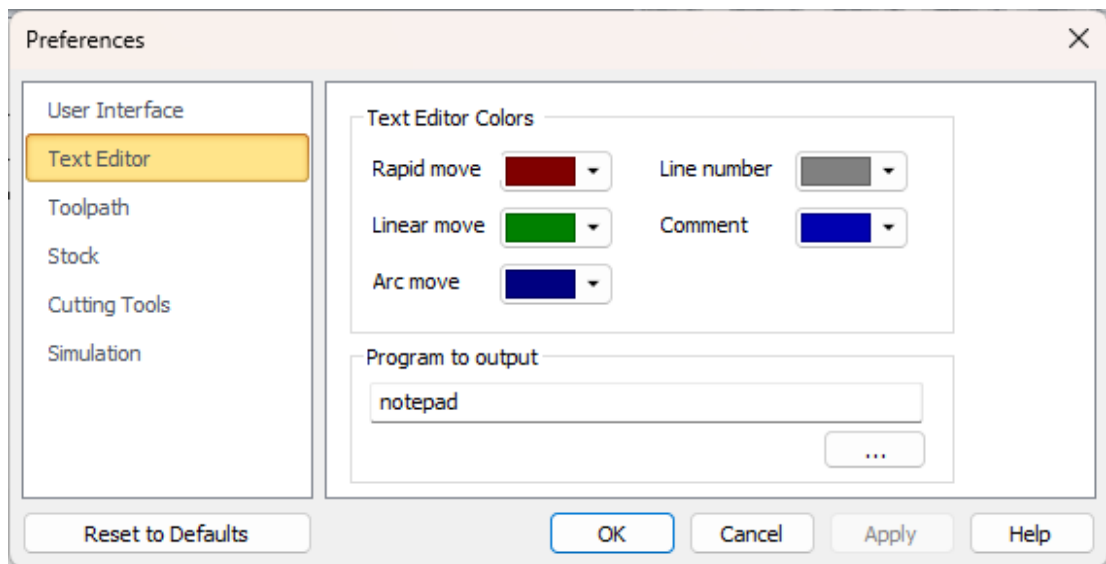


To Access this Command:

click:  > Text Editor



Preferences > Text Editor



Preferences > Text Editor



Text Editor Colors

You can specify the default colors to display for **Rapid**, **Linear** and **Arc** moves as well as for **Line Numbers** and **Comments** within the G-Code Editor. Use the color selector available for each.

```

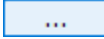
1  %
2  G40 G49 G80 G98
3  ;Setup 1
4  ;2 1/2 Axis Profiling (Outer) ← Comments
5  G20 G90
6  S10000 M03
7  M07
8  G00 Z0.5 ← Rapid Moves
9  X-0.8976 Y-1.2533
10 G01 Z0. F97.5
11 X-1.1325 Y-1.1678 F146.25 ← Linear Moves
12 X-1.25 Y-1.125
13 G17
14 G03 X-1.125 Y-1.25 I0. J-0.125 F195. ← Arc Moves
15 G01 Y-1.
16 Y0.

```

Text Editor Color Preferences



Save Directory

You can specify the default save directory. Pick the  button to select a different folder. When you pick **Save** from the **Project** tab, this is the folder where the file is saved. To overwrite the original g-code, use the **Save G-Code** command from the **Edit** tab.



Related Topics

[User Interface Preferences](#)

[Toolpath Preferences](#)

[Stock Preferences](#)

[Cutting Tools Preferences](#)

[Simulation Preferences](#)

4.1.3 Toolpath

Here you can set Toolpath related preferences including colors, display, format and feedrate. You can use the **Reset to defaults** button if you want to revert to the default factory install settings.

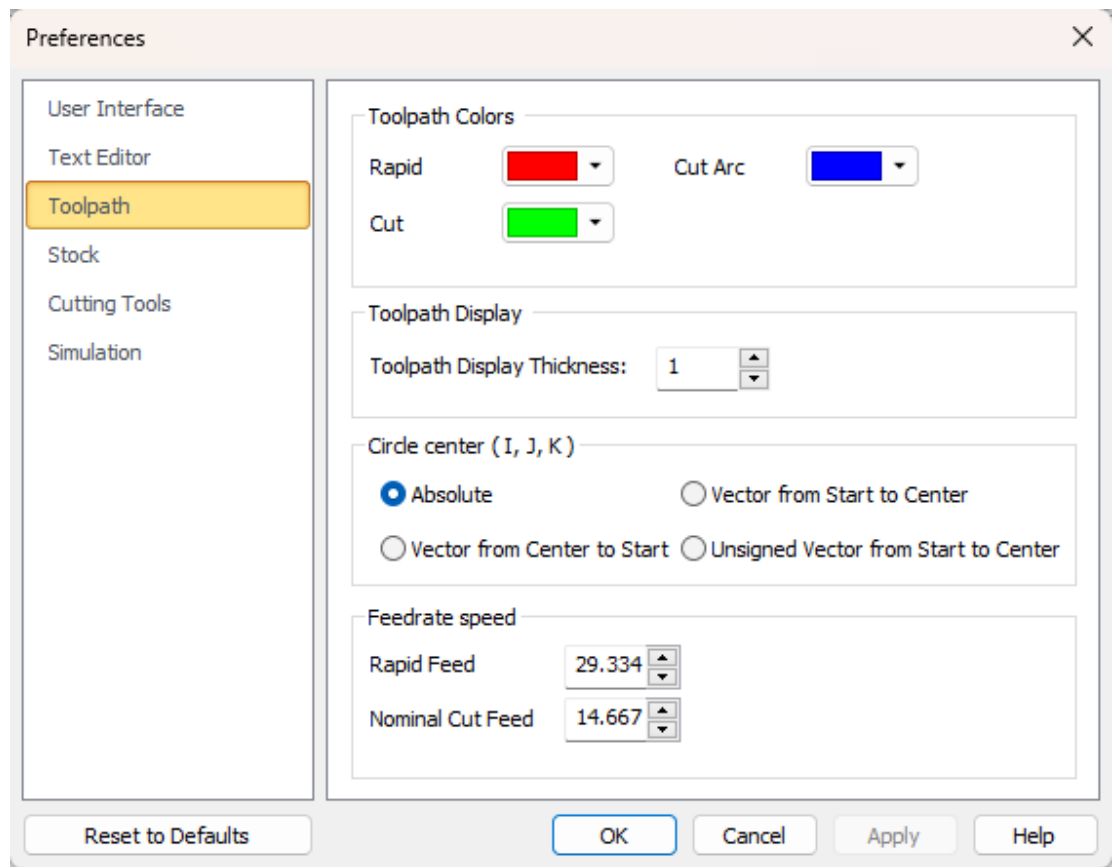


To Access this Command:

click:  > Toolpath



Preferences > Toolpath

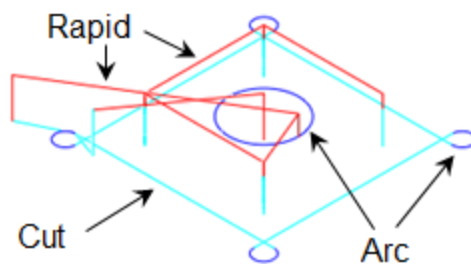


Preferences > Toolpath



Toolpath Colors

You can specify the default colors to graphically display **Rapid**, **Linear** and **Arc** moves. Use the color selector available for each.

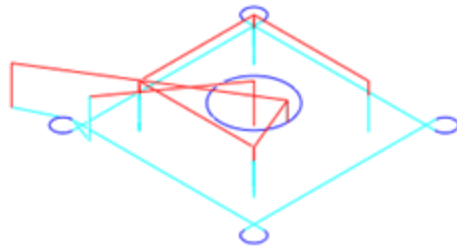


Toolpath Display Color Preferences

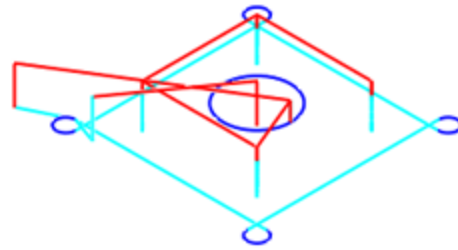


Toolpath Display

Here you can set the line thickness for the display of all toolpaths.



Toolpath Display = 1



Toolpath Display = 2



Circle Center (I,J,K)

I,J,K refers to the center point location for arc motions. In this section you can define how arc motions are formatted in the output G-Code file. Select from the following options:

- Absolute
- Vector from Center to Start
- Vector from Start to Center
- Unsigned Vector from Start to Center



Feedrate Speed

Here you can specify feedrate values for the G-Code file. Select from the following:

- Rapid Feed
- Nominal Cut Feed



Related Topics

[User Interface Preferences](#)

[Text Editor Preferences](#)

[Stock Preferences](#)

[Cutting Tools Preferences](#)

[Simulation Preferences](#)

4.1.4 Stock

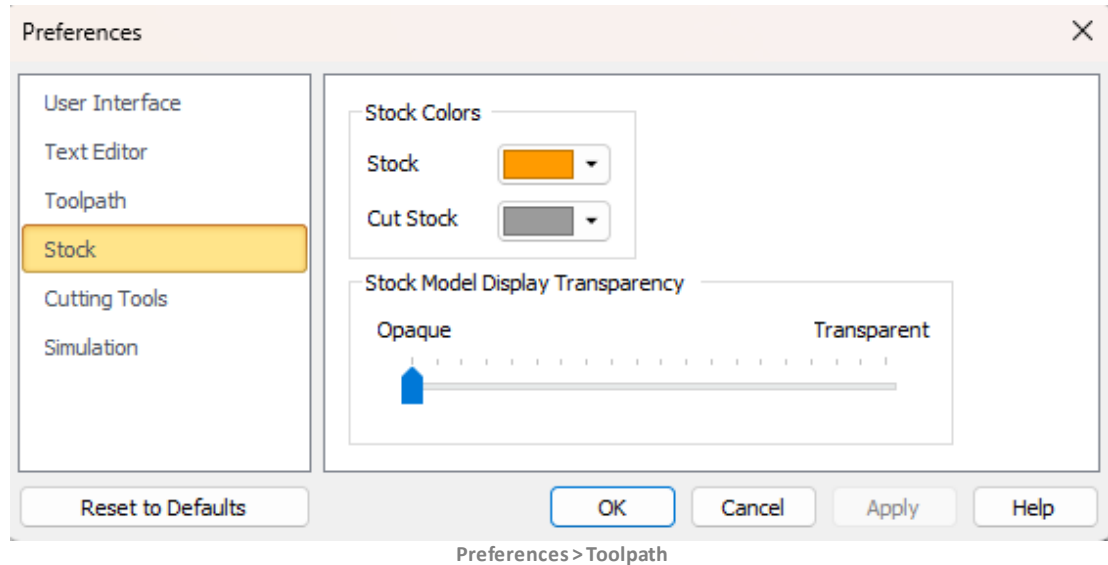
Here you can set **Toolpath** related preferences including colors, display, format and feedrate. You can use the **Reset to defaults** button if you want to revert to the default factory install settings.



To Access this Command:

click:  > Stock

Preferences > Stock



Stock Colors

Here you can set the default colors for [Stock](#) and [Cut Stock](#). Use the color selector available for each.

Stock Model Display Transparency

Use the slider to set the transparency of the [Stock Model Display](#).

Related Topics

[User Interface Preferences](#)

[Text Editor Preferences](#)

[Toolpath Preferences](#)


[Cutting Tools Preferences](#)

[Simulation Preferences](#)

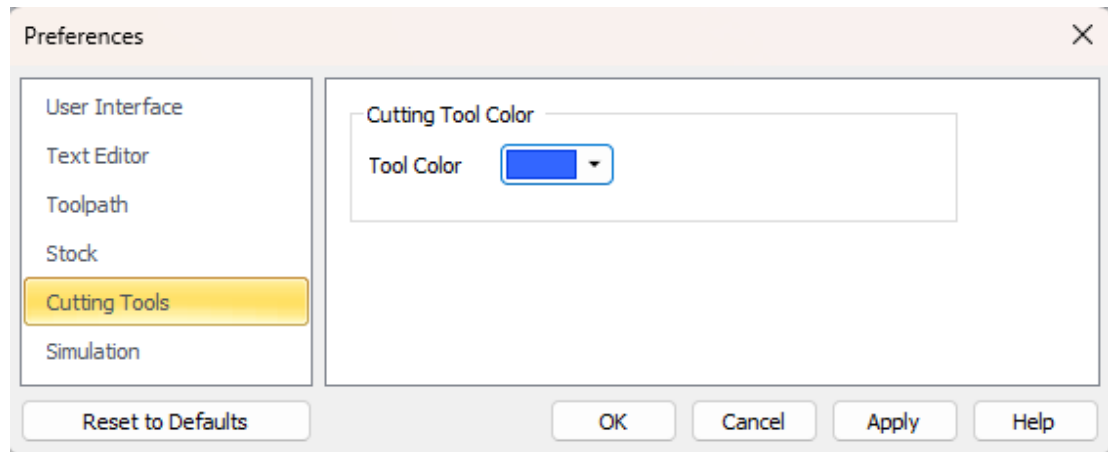
4.1.5 Cutting Tools

Here you can set [Cutting Tool](#) related preferences. You can use the [Reset to defaults](#) button if you want to revert to the default factory install settings.

To Access this Command:

click:  > Cutting Tools

Preferences > Cutting Tools



Preferences > Cutting Tools

Cutting Tool Color

Here you can set the default color for the [Cutting Tool](#) during simulations. Use the color selector available for each.

Related Topics

[User Interface Preferences](#)

[Text Editor Preferences](#)

[Toolpath Preferences](#)


[Stock Preferences](#)

[Simulation Preferences](#)

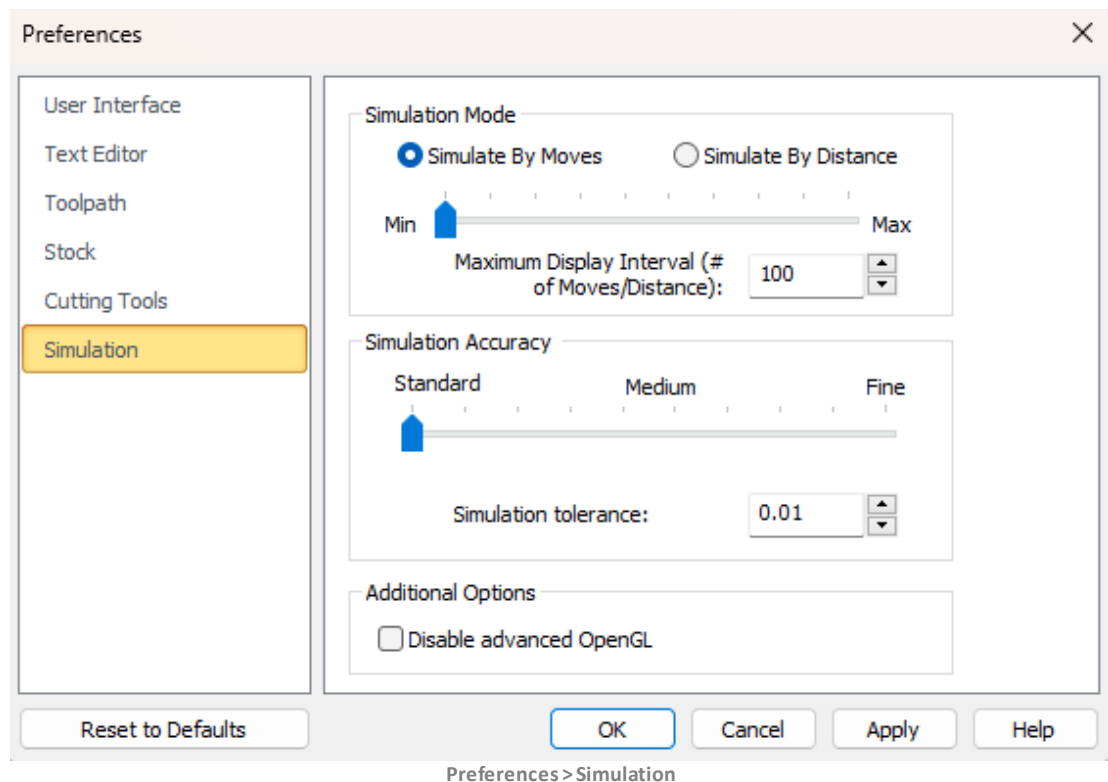
4.1.6 Simulation

Here you can set [Simulation](#) related preferences. You can use the [Reset to defaults](#) button if you want to revert to the default factory install settings.

To Access this Command:

click:  > Simulation

Preferences > Simulation



Simulation Mode

There are two simulation modes.

- Simulate by Moves**
 One pick of the [Step](#) icon on the [Simulate tab](#) moves the [Cutting Tool](#) one complete move (i.e., start to end). See [Maximum Display Interval](#) below. This option is the same as checking the [Simulate By Moves](#) box on the [Simulate](#) tab.
- Simulate by Distance**
 One pick of the [Step](#) icon on the [Simulate tab](#) moves the [Cutting Tool](#) a specified distance. See [Maximum Display Interval](#) below. This option is the same as unchecking the [Simulate By Moves](#) box on the [Simulate](#) tab.
- Maximum Display Interval**
 You can control the speed of the simulation using the slider bar and the [Maximum Display Interval](#). When using [Simulate by distance](#) mode, the speed is determined as $\# \text{ of Motions} / \text{Distance}$.



Simulation Accuracy

Use this slider to adjust the simulation display accuracy.

Simulation Tolerance

The [Simulation Accuracy](#) slider provides a high-level of adjustment and the [Simulation Tolerance](#) provides a more granular level of control. Note that the [Simulation Tolerance](#) will have a direct affect on simulation time.



Additional Options

Disable Advanced OpenGL

Check this box only if you have an older graphics card adapter that does not support advanced [OpenGL](#) (i.e., [OpenGL 2](#)). Some older cards may only support [OpenGL 1](#) for example. If you experience graphics instability checking this box may help resolve the issue.



Related Topics

[User Interface Preferences](#)

[Text Editor Preferences](#)

[Toolpath Preferences](#)

[Stock Preferences](#)

[Cutting Tools Preferences](#)

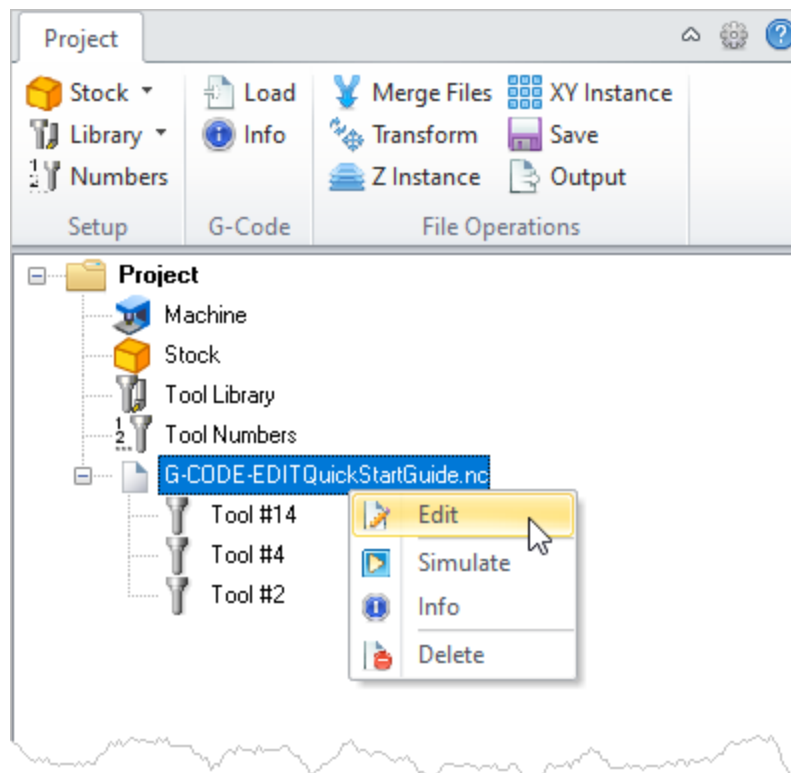
4.2 G-Code Browser

The [G-Code Browser](#) has four main modes of operation represented by tabs at the top of the browser window. These are [Project](#), [Edit](#), [G-Code Edit](#) and [Simulate](#). Each tabbed view also incorporates a ribbon toolbar of commands at the top. These toolbars group associated commands into panes.

Each menu tab is shown below:



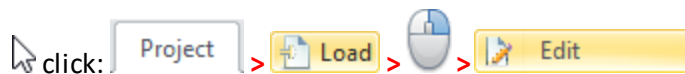
The Project Tab

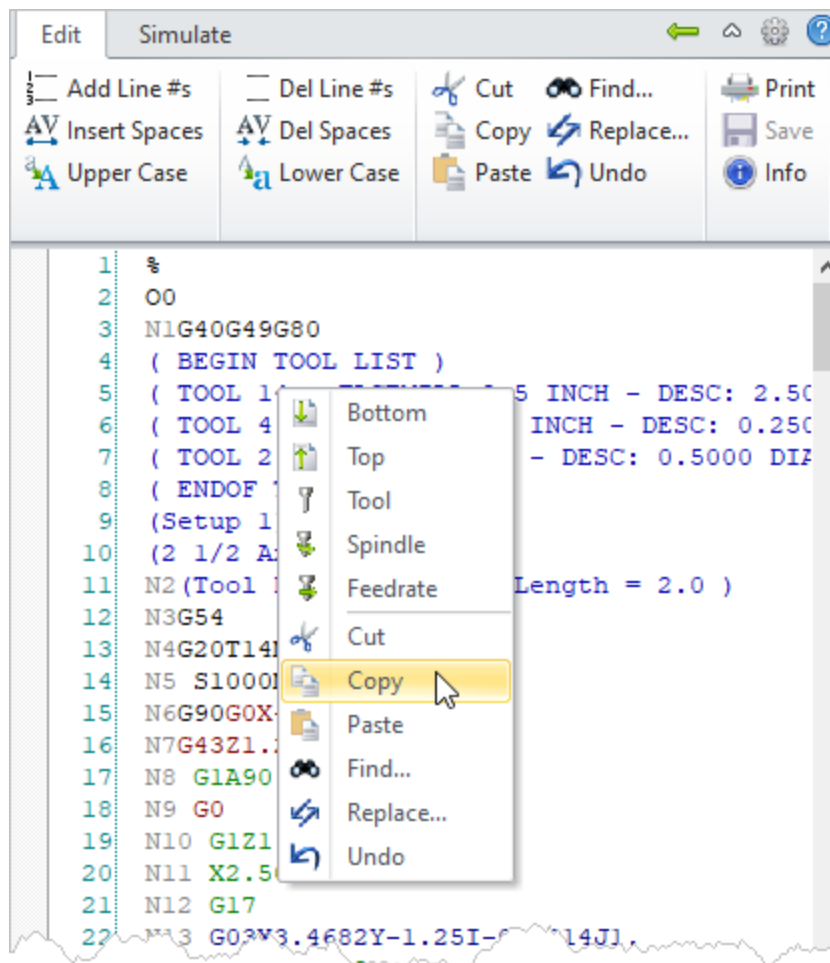


G-Code Browser, Project Tab



The Edit Tab

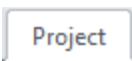




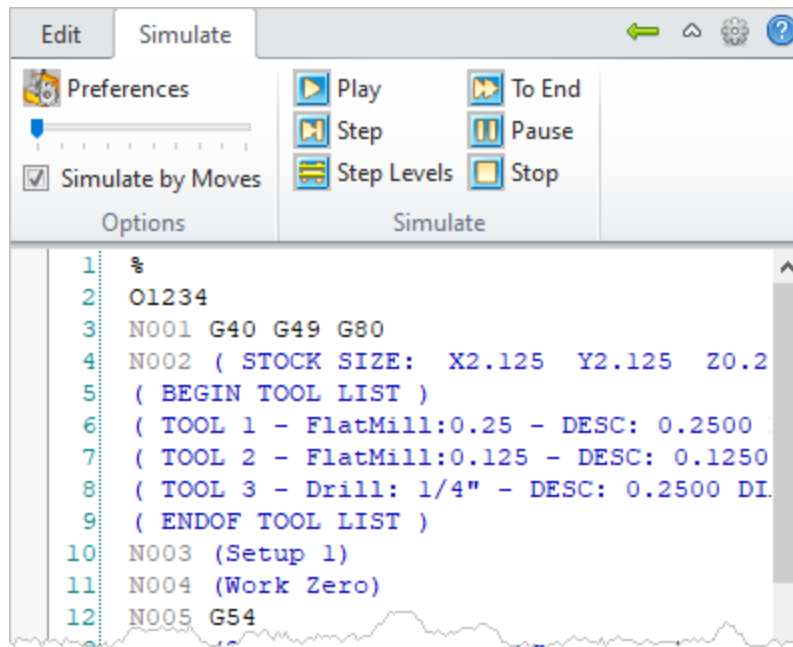


G-Code Browser, Edit Tab



The Simulate Tab

click:  >  >  (with [Simulate Window](#) displayed)



G-Code Browser, Simulate Tab



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Simulate Tab](#)

4.2.1 Project Tab

The Project tab is Selecting the [Program](#) tab in the [Mops Browser](#) shows 3 groups of menus that provide access for specifying [Machine](#), [Stock](#) and defining [Machining Operations](#).

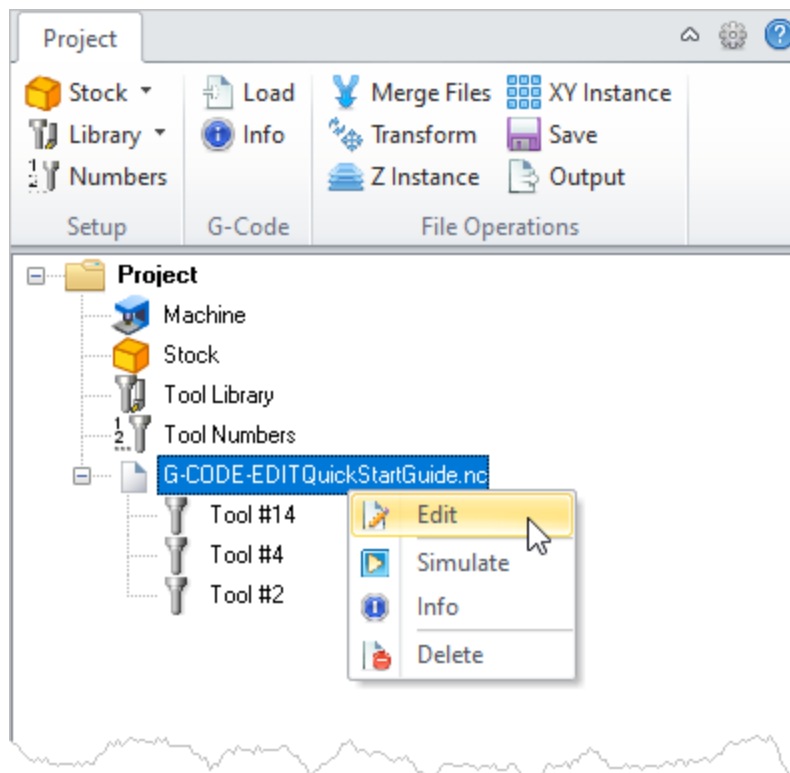


To Access this Command:

click: 



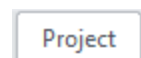
The Project Tab



G-Code Browser, Project Tab



Ribbon Bar Options



Contains [Project](#) related commands.



Use these icons to maximize and minimize the ribbon bar menus.



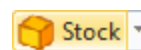
Display the [Preferences](#) dialog.



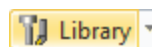
Display the [Online Help](#) system.



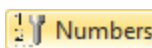
Project Setup Pane



Displays a menu of [Stock](#) definition commands.



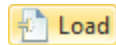
Displays the [Tool Library](#) dialog.



Displays the [Tool Crib](#) dialog.



Project G-Code Pane



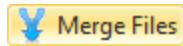
Load [Load](#) a g-code file into the Project tree.



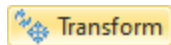
Info Display [Information](#) about the g-code files currently loaded into the Project.



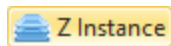
Project File Operations Pane



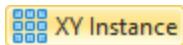
Merge Files [Merge](#) two selected g-code files into one file.



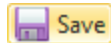
Transform Displays the [Transform](#) g-code dialog.



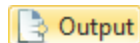
Z Instance Displays the [Z Instance](#) g-code dialog.



XY Instance Displays the [XY Instance](#) g-code dialog.



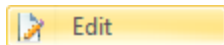
Save [Saves](#) all g-code files currently loaded into the Project.



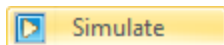
Output [Outputs](#) the g-code files currently loaded into the Project.



Project Tab, Right-Click Menu



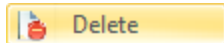
Edit Edit the selected G-Code file.



Simulate Simulate the selected G-Code file.



Info Display information about the selected G-Code file.



Delete Delete the selected G-Code file.



Related Topics

[Project Tree](#)

[Edit Tab](#)

[Simulate Tab](#)

[Online Help](#)

4.2.2 Project Tree




Project

The [Project](#) tree is located under the [Project](#) tab. It contains all of the elements needed for a specific project, including stock model, tool library, tool crib and g-

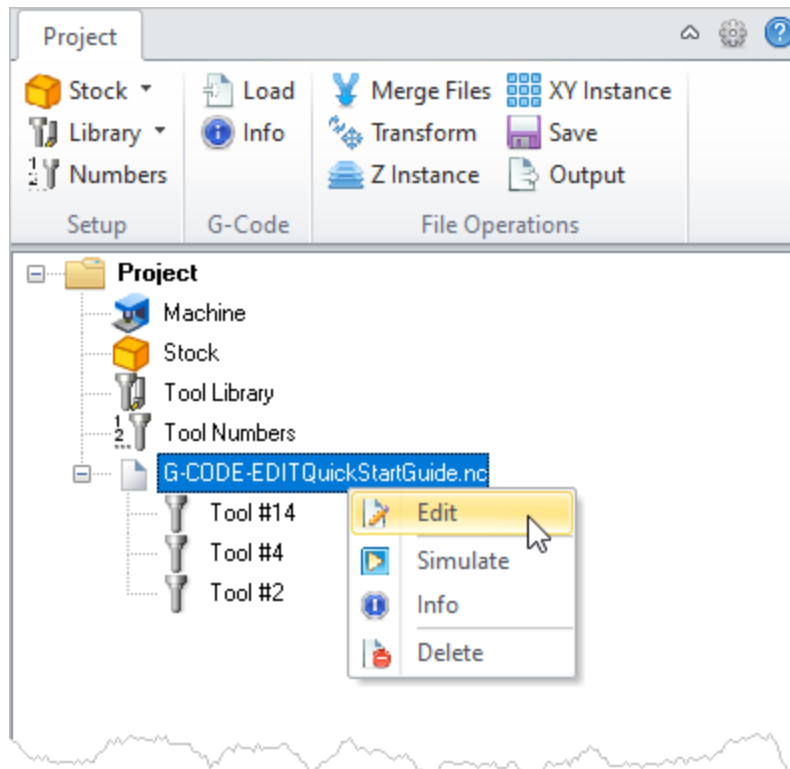
code files. The Project tree can also alert you to issues that need to be addressed by flagging items for attention.

To Access this Command:

click: 

The Project Tab

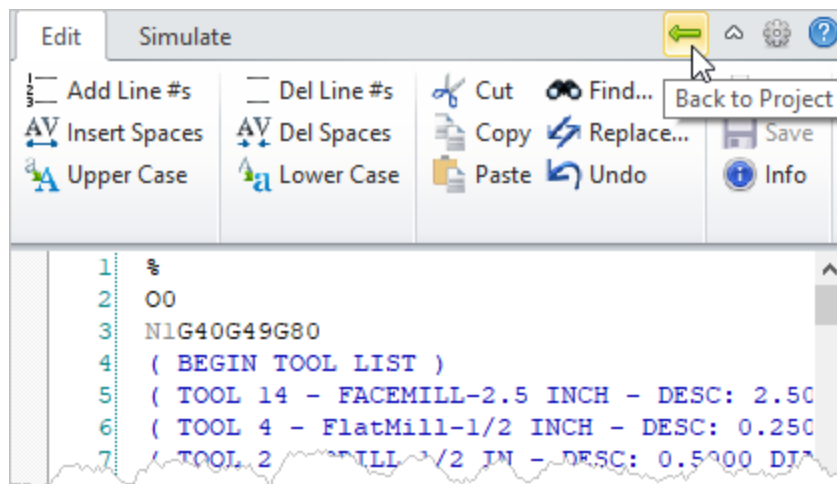
You must have the [Project](#) tab selected in order to see the [Project](#) tree.



G-Code Browser, Project Tab

Returning to the Project Tab

You can return to the [Project](#) tab by selecting this icon  located on the top [G-Code Browser](#) when either the [Edit](#), [G-Code Edit](#) or [Simulate](#) tabs are active.

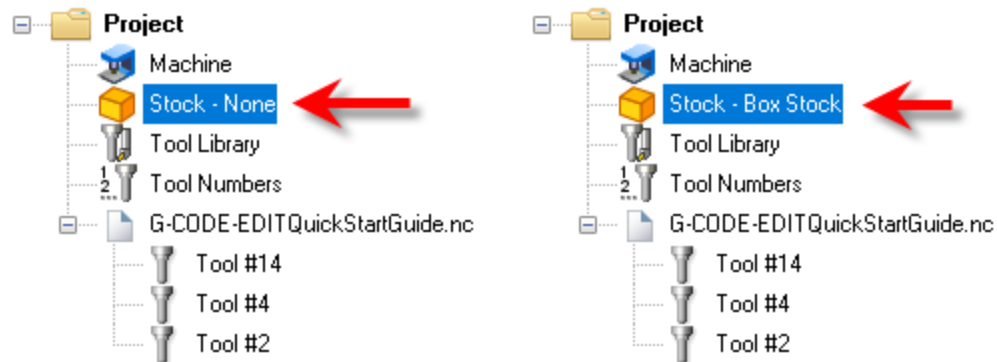


G-Code Browser, Return to Project Tab menu item



How Stock is Defined in the Project Tree

If there no stock yet defined, the **Stock** icon in the **Project** tree will say "Stock - None". Once a stock is defined, the Stock icon in the **Project** tree will indicate the type of stock that is defined. Refer to the two **Project** tree images. The image on the left has no **Stock** yet defined. The image on the right has **Box Stock** defined.



When Stock is not defined

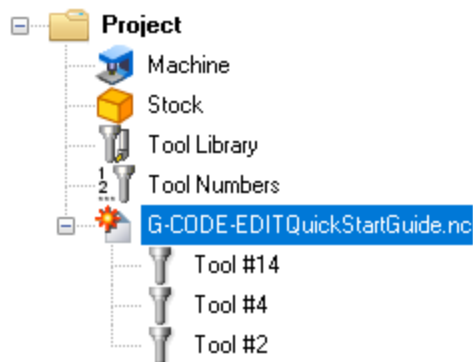
When Stock is defined



Red Flags in the Project Tree

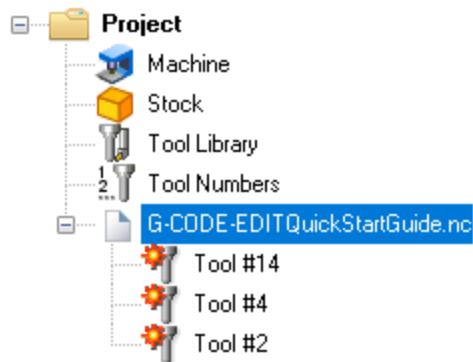
If items are flagged in the **Project** tree it means that they need your attention.

For example if a g-code file has been modified but has not yet been saved or output, it will be flagged as shown in the example below.



When a g-code file is flagged

If you load a g-code file that uses a tool number that is currently undefined in your tool crib, it will be flagged as shown below.



When a tool is flagged



Related Topics

[Project Tab](#)

[Edit Tab](#)

[Simulate Tab](#)

[Online Help](#)

4.2.3 Edit Tab

Edit When you double-left-click on a G-Code file from the **Project** tree, that file is loaded into the **G-Code Editor**. The **Edit** tab contains commands to **Modify** and **Navigate** the g-code file. The **Edit** tab and commands are listed below.

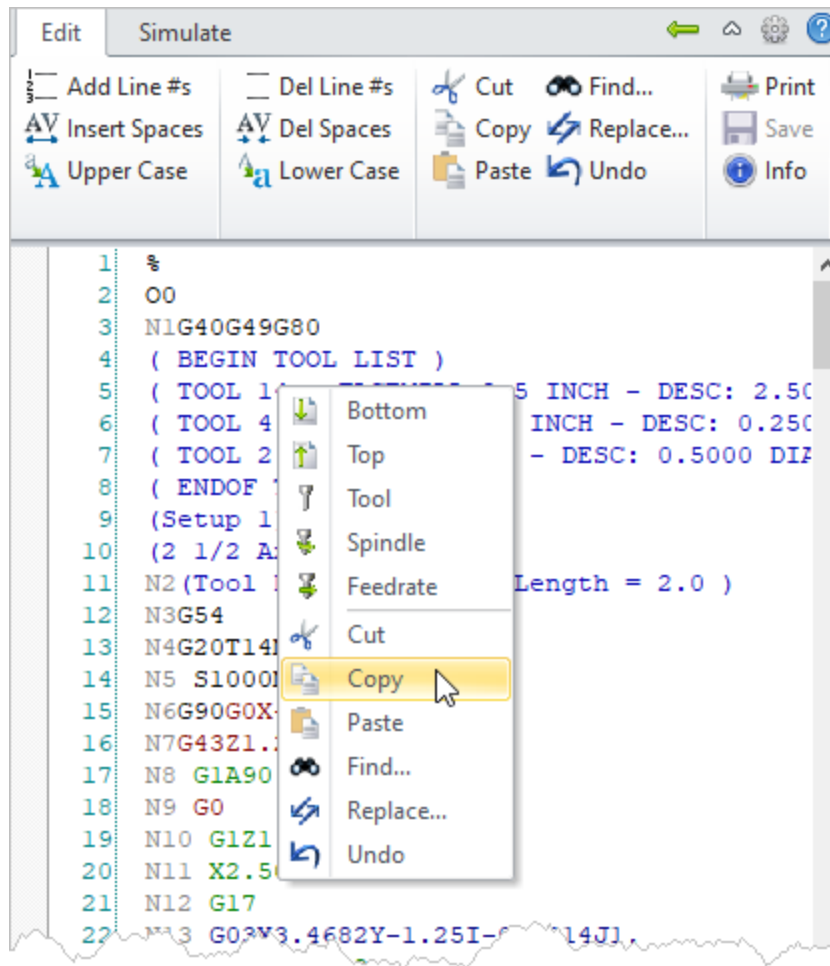


To Access this Command:

click: **Project** > **Load** > **Edit**



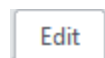
Edit tab



G-Code Browser, Edit Tab



Ribbon Bar Options



Contains file [Edit](#) related commands.



Returns to the [Project](#) tab.



Minimizes and maximizes the ribbon bar menu.



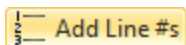
Displays the [Preferences](#) dialog.

















Displays the [Online Help](#) system.



Edit Tab














[Add line numbers](#) to the g-code file.

-  **Insert Spaces** [Insert spaces](#) (i.e., delimiters) between g-codes.
-  **Upper Case** [Convert g-code to all UPPERCASE text.](#)
-  **Del Line #s** [Delete line numbers](#) from the g-code file.
-  **Del Spaces** [Delete spaces](#) (i.e., delimiters) between g-codes.
-  **Lower Case** [Convert g-code to all lowercase text.](#)
-  **Cut** [Cut](#) the selected text to the clipboard.
-  **Copy** [Copy](#) the selected text to the clipboard.
-  **Paste** [Paste](#) the selected text to the current cursor location.
-  **Find...** [Find text](#) within the G-Code file.
-  **Replace...** [Find & Replace](#) text within the G-Code file.
-  **Undo** [Undo](#) the last command.
-  **Print** [Print](#) the G-Code file currently loaded in the editor.
-  **Save** [Save](#) the G-Code file currently loaded in the editor.
-  **Info** [Display information](#) about the current g-code file.








Edit Tab, Right-Click Menu

-  **Bottom** [Go to the Bottom](#) (End) of the G-Code file.
-  **Top** [Go to the Top](#) (Start) of the G-Code file.
-  **Tool** [Go to the next Tool Change](#) code.

 Spindle	Go to the next Spindle code.
 Feedrate	Go to the next Feedrate code.
 Cut	Cut the selected text to the clipboard.
 Copy	Copy the selected text to the clipboard.
 Paste	Paste the selected text to the current cursor location.
 Find...	Find text within the G-Code file.
 Replace...	Find & Replace text within the G-Code file.
 Undo	Undo the last command.

Edit Tab, Display Toolbar



-  **Stock Model Visibility:** Turn on/off stock model
-  **Part Model Visibility:** Turn on/off part model display during simulation.
-  **Toolpath Visibility:** Turn on/off toolpath display
-  **Follow Toolpath Display:** The toolpath is displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath after the tool passes).
-  **Trace Toolpath Display:** The toolpath is not displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath before the tool passes).



Segment Toolpath Display: The toolpath is only displayed for the segment that the tool is currently on.



Related Topics

[Project Tab](#)

[Project Tree](#)

[Simulate Tab](#)

[Online Help](#)

4.2.4 Simulate Tab

Simulate

When you double-left-click on a G-Code file from the [Project](#) tree, that file is loaded into the [G-Code Editor](#). The [Simulate](#) tab contains commands to perform tool motions and cut material simulations of the g-code file. The [Simulate](#) tab and commands are listed below. **Note:** You must have [Stock](#), [Tool Library](#) and [Tool Crib](#) defined before performing a [Simulation](#).

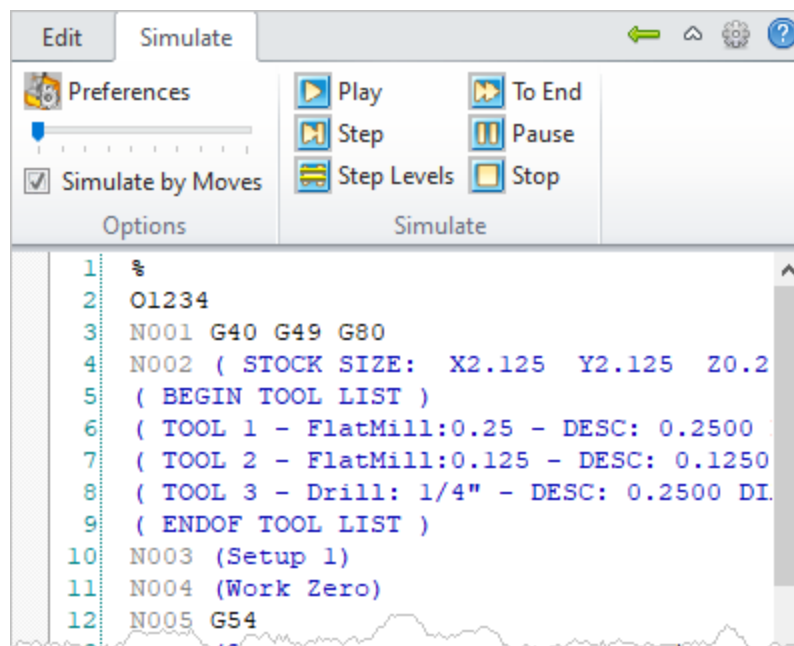


To Access this Command:

click: [Project](#) > [Load](#) > [Simulate](#)




Simulate tab





G-Code Browser, Simulate Tab

Ribbon Bar Options

 Contains file [Simulation](#) related commands.

 Returns to the [Project](#) tab.

 Minimizes and maximizes the ribbon bar menu.

 Displays the [Preferences](#) dialog.

 Displays the [Online Help](#) system.


Simulate Tab, Options Pane

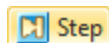
 Display [Simulation Preferences](#).

 [Simulation speed](#) adjustment (Left: Slow, Right: Fast).

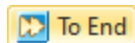
☒ [Simulate by Moves](#) Check to [simulate by moves](#). Uncheck to simulate by distance. See [Preferences](#) to set the default mode.

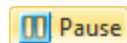
Simulate Tab, Simulate Pane

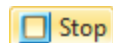
 [Play](#) the simulation.

 [Step](#) through the simulation by distance or by moves.

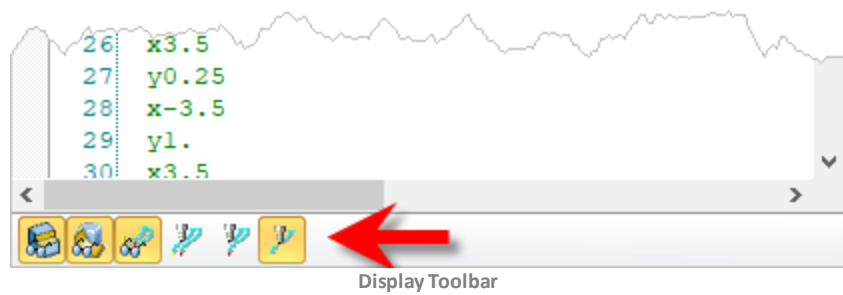
 [Display](#) simulation in [Levels](#).

 [Go direction to the End](#) result of the simulation.

 [Pause](#) the simulation.

 [Stop](#) the simulation.

Simulate Tab, Display Toolbar



Stock Model Visibility: Turn on/off stock model



Part Model Visibility: Turn on/off part model display during simulation.



Toolpath Visibility: Turn on/off toolpath display



Follow Toolpath Display: The toolpath is displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath after the tool passes).



Trace Toolpath Display: The toolpath is not displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath before the tool passes).



Segment Toolpath Display: The toolpath is only displayed for the segment that the tool is currently on.



Related Topics

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Online Help](#)

4.2.5 Online Help



You can access this online help system from the G-Code Browser.

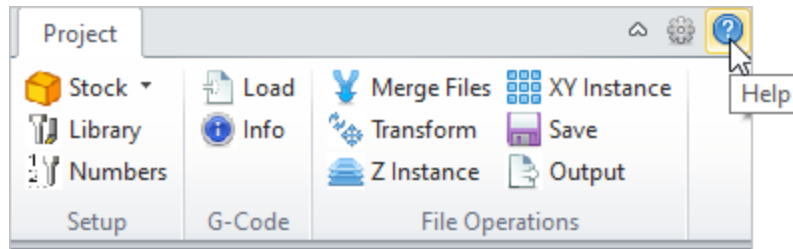


To Access this Command:

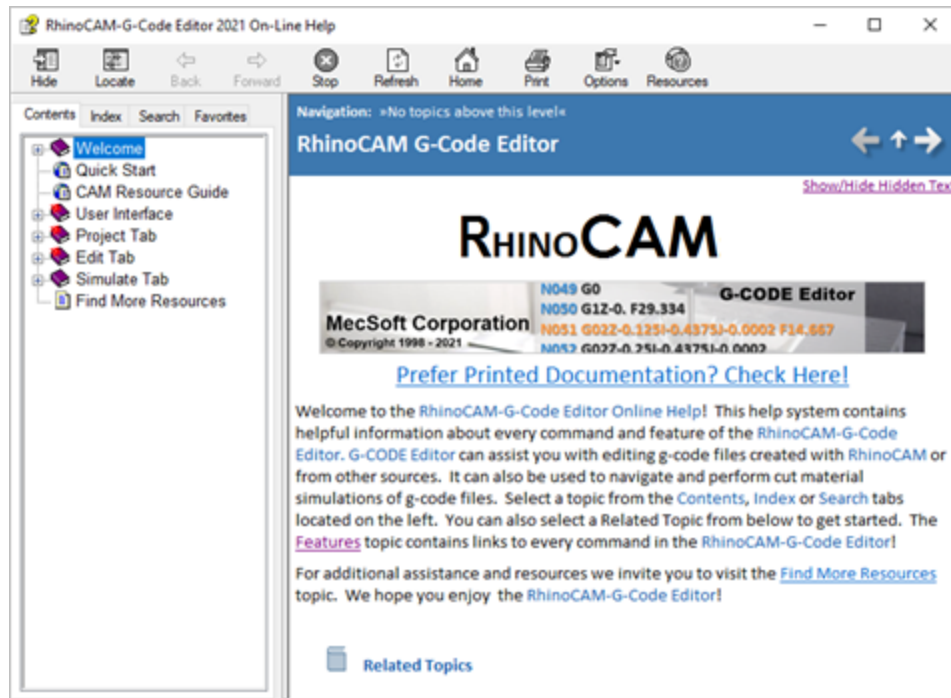
click:



The Online Help Icon



Online Help menu item



RhinoCAM-G-Code Editor Online Help System



Related Topics

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Simulate Tab](#)

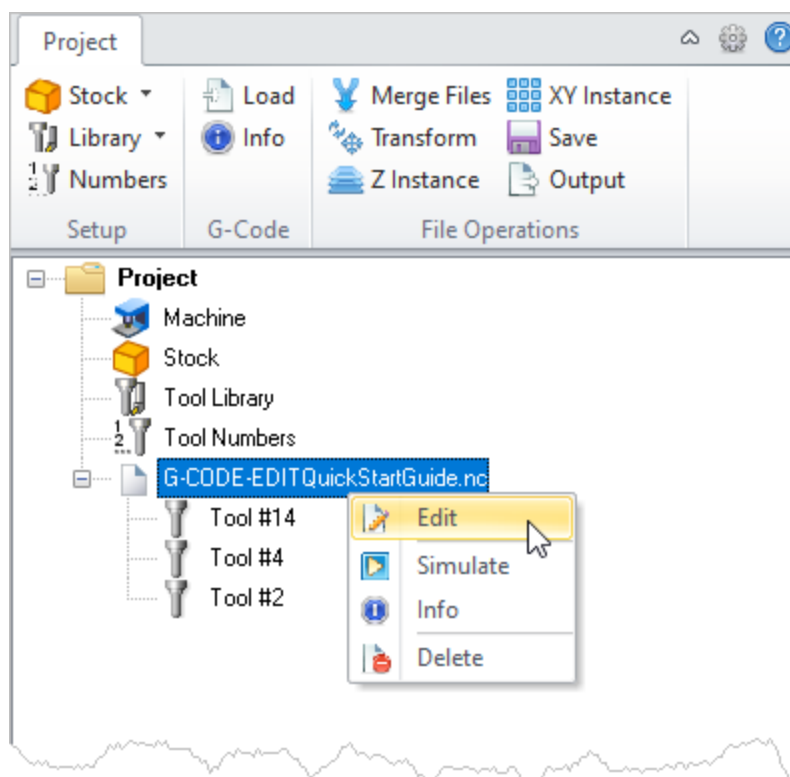
Project Tab

The Project tab is Selecting the [Program](#) tab in the [Mops Browser](#) shows 3 groups of menus that provide access for specifying [Machine](#), [Stock](#) and defining [Machining Operations](#).

To Access this Command:

click: 



The Project Tab



G-Code Browser, Project Tab

Ribbon Bar Options

 Contains [Project](#) related commands.

  Use these icons to maximize and minimize the ribbon bar menus.

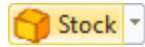
 Display the [Preferences](#) dialog.



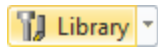
Display the [Online Help](#) system.



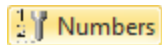
Project Setup Pane



Displays a menu of [Stock](#) definition commands.



Displays the [Tool Library](#) dialog.



Displays the [Tool Crib](#) dialog.



Project G-Code Pane



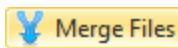
[Load](#) a g-code file into the Project tree.



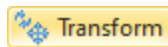
Display [Information](#) about the g-code files currently loaded into the Project.



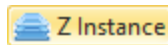
Project File Operations Pane



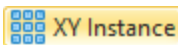
[Merge](#) two selected g-code files into one file.



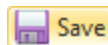
Displays the [Transform](#) g-code dialog.



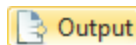
Displays the [Z Instance](#) g-code dialog.



Displays the [XY Instance](#) g-code dialog.



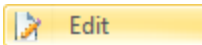
[Saves](#) all g-code files currently loaded into the Project.



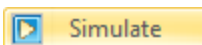
[Outputs](#) the g-code files currently loaded into the Project.



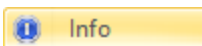
Project Tab, Right-Click Menu



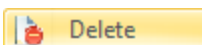
Edit the selected G-Code file.



Simulate the selected G-Code file.



Display information about the selected G-Code file.



Delete the selected G-Code file.



Related Topics

[Project Tree](#)

[Edit Tab](#)

[Simulate Tab](#)

[Online Help](#)

5.1 Machine MCS Setup



Project

Machine

You can set the cutting direction and alignment using the [Machine Coordinate System \(MCS\)](#) dialog. This orients the coordinates defined in the any g-code files that are currently loaded into the editor.



To Access this Command:

click:

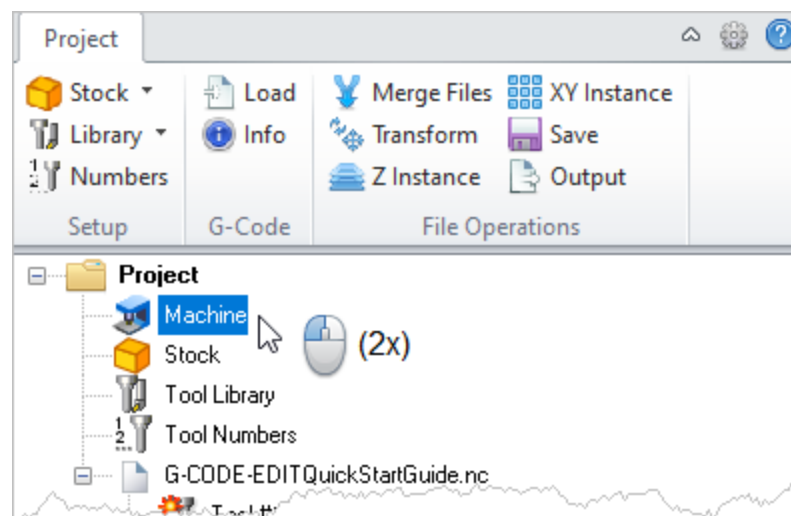
Project



double-click:



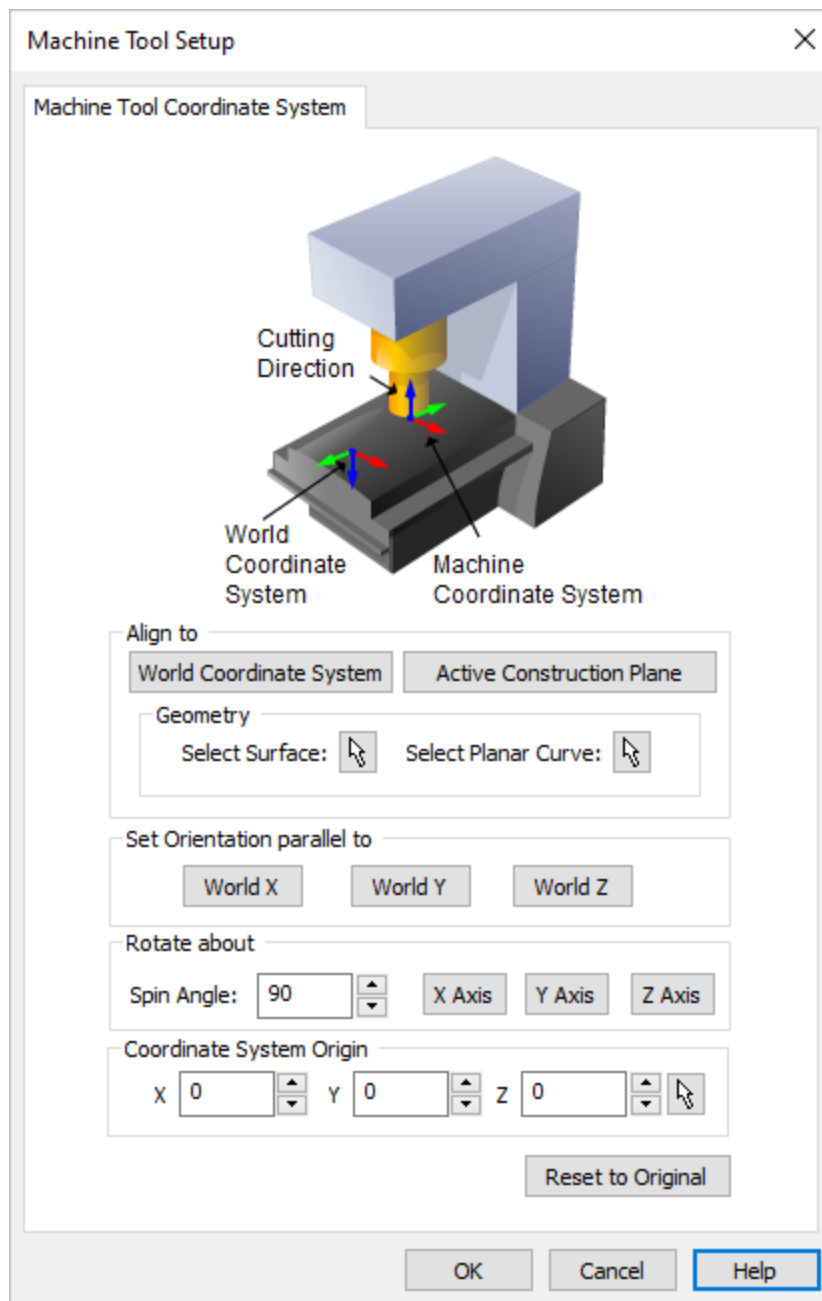
G-Code Browser: Project tab, Stock menu



Edit the Machine MCS



Dialog Box: Machine



Dialog Box: Machine (MCS) Setup

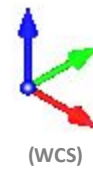


The Machine Coordinate System (MCS)

The **Machine Coordinate System (MCS)** is displayed as a triad with **Blue** line representing the Z-axis, **Red** representing X-axis and **Green** representing the Y-axis. The **WCS** is displayed the same way as **MCS** and is located at the origin. The lengths of the **WCS** arrows are shorter when compared to **MCS**.



(MCS)



(WCS)

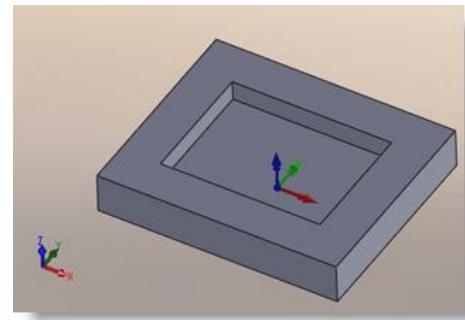


Align to

This aligns the **MCS** orientation. Select from the following:

Machine Tool Coordinate System

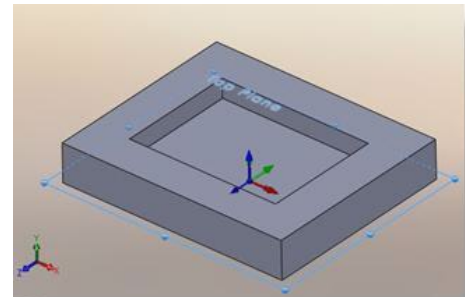
Aligns the Setup Coordinate System to Machine Tool Coordinate System.



Align To: World Coordinate System

Active View Construction Plane

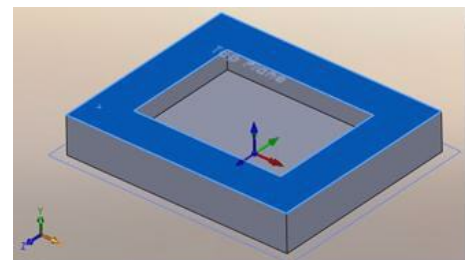
Setting the **MCS** to the orientation of the active viewport of the construction **Plane**.



Align To: Active Construction Plane

Geometry: Part Surface

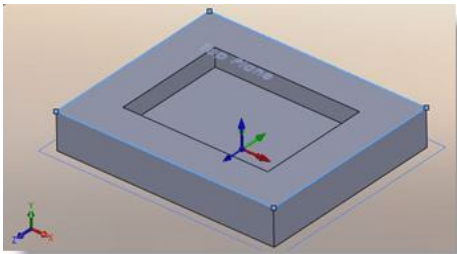
User can select a point on a surface determine the alignment of **MCS**.



Align To: Geometry, Part Surface

Geometry: Planar Curve

User can select a planar curve (2D sketch) to determine the alignment of MCS.



Align To: Geometry, Planar Curve



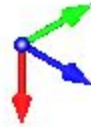








Set Orientation Parallel to

Allows you to set the Machine Coordinate System parallel to the World X Y or Z co-ordinate axis.

World X orients the Z axis of MCS parallel to World X axis.

World Y orients the Z axis of MCS parallel to World Y axis.

World Z orients the Z axis of MCS parallel to World Z axis.

Orientation Parallel to	Triad Display		
	MCS Visibility ON WCS Visibility OFF	WCS Visibility ON MCS Visibility OFF	MCS Visibility ON WCS Visibility ON
World X			
World Y			
World Z			



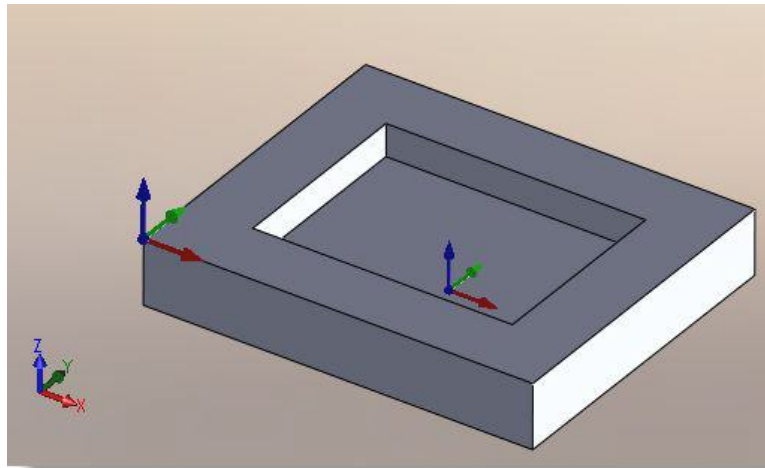
Rotate about

Allows you to rotate the **Machine Coordinate System** in X Y Z coordinate axis by any angle specified under **Spin Angle**. Specify **Spin Angle** and click the axis to rotate about. Clicking the same coordinate axis button multiple times rotates by the specified angle incrementally. For example if you set the **Spin Angle** = 90 and click **X Axis** button 2 times, the **MCS** is rotated about X coordinate axis by 180 degrees.



Coordinate System Origin

This translates the **MCS** origin to the desired location. This can be set to any location on the part geometry.



Use Coordinate System Origin to translate the MCS origin to the desired location



Reset to Original

Resets **MCS** orientation to current **MCS** orientation.



Related Topics

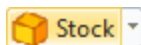
[G-Code Browser](#)

[Project Tab](#)

[Project Tree](#)

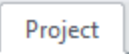
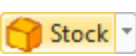
5.2 Setup Pane

5.2.1 Stock

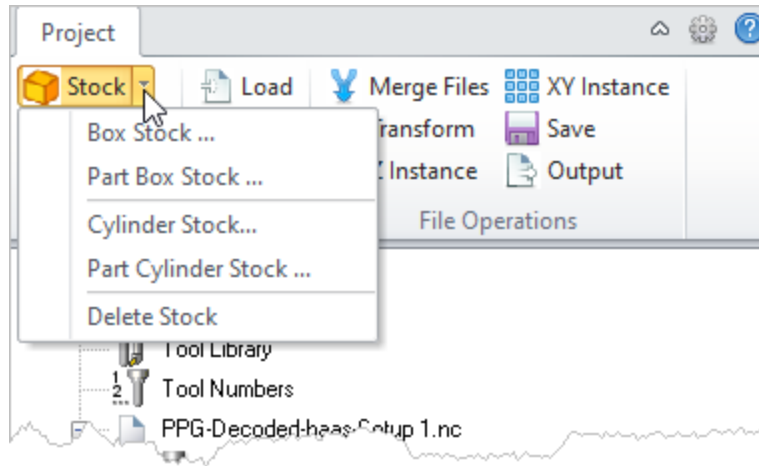


This menu contains commands to create different types of **Stock** material in preparation for cut material Simulations. If there is no stock defined, **Simulations** will only show tool motions and not a cut material simulation.

To Access this Command:

click:  > 

G-Code Browser: Project tab, Stock menu



The Stock Menu

Related Topics

[Stock Menu](#)

[Create Box Stock](#)

[Create Part Box Stock](#)

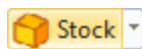
[Create Cylinder Stock](#)

[Create Part Cylinder Stock](#)

[Stock from Selection](#)

[Delete Stock](#)

5.2.1.1 Box Stock

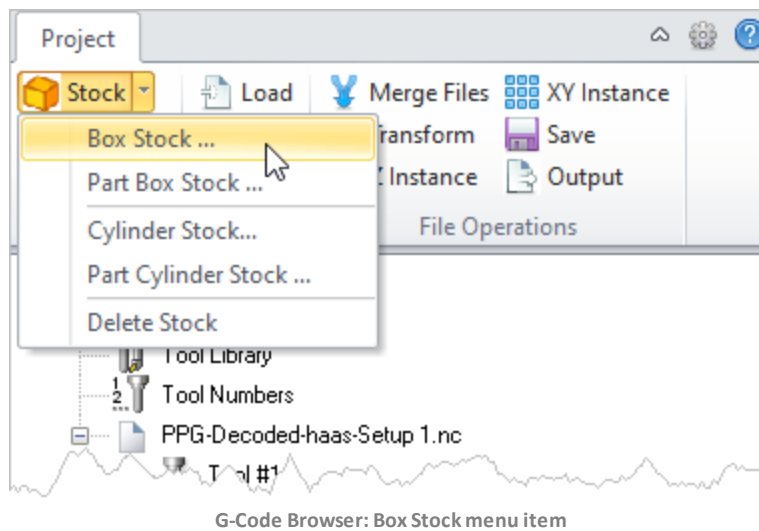


You can define the raw stock model as a simple box by selecting the [Box Stock](#) option from the [Stock](#) menu on the [Project](#) tab of the [G-Code Browser](#) and entering the [Stock](#) dimensions.

To Access this Command:

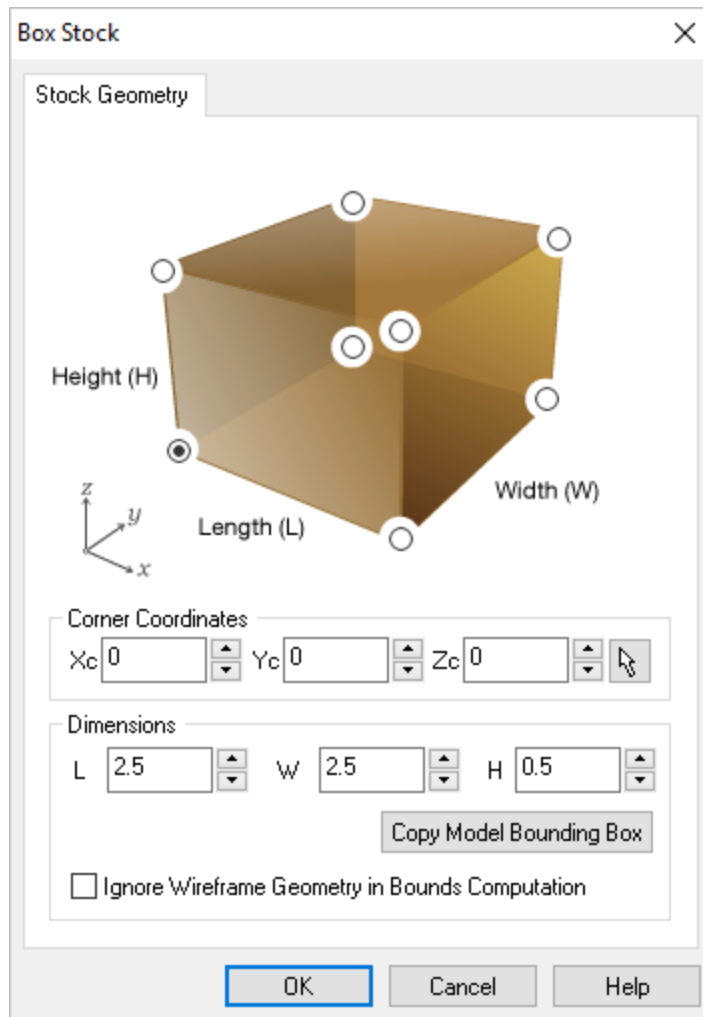
click:  >  > [Box Stock ...](#)

Box Stock menu item



Dialog Box: Box Stock

You can define the box by simply typing in the **Length**, **Width** and **Height** of the box in the corresponding edit boxes of the dialog. The corner position (origin) of the box can also be repositioned by picking the desired coordinate or by specifying the coordinate values in the corresponding edit boxes in the dialog (**Xc**, **Yc**, **Zc**). When you click on the **OK** button, a stock model based on your definition will be created and displayed.



Dialog Box: Box Stock

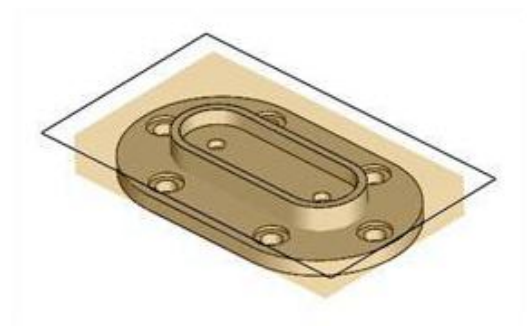
Copy Model Bounding Box

When [Copy Model Bounding Box](#) is selected, the system calculates the bounding box of the part model as the [XYZ](#) extents of geometry of the part model and displays the [XYZ Coordinate](#) values under dimensions.

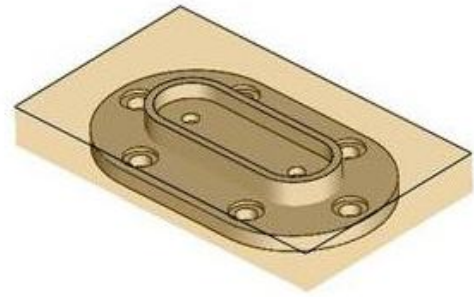
Ignore Wireframe Geometry in Bounds Computation

Check this box to ignores all 2D and 3D curve geometries present in the part from stock bounding box computation.

Make sure to click [Copy Model Bounding Box](#) after you check or uncheck [Ignore Wireframe Geometry in Bounds Computation](#).



Ignore Wireframe geometry - checked



Ignore Wireframe geometry – un-checked



Related Topics

[Stock Menu](#)

[Create Part Box Stock](#)

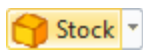
[Create Cylinder Stock](#)

[Create Part Cylinder Stock](#)

[Stock from Selection](#)

[Delete Stock](#)

5.2.1.2 Part Box Stock



Stock You can define the raw stock model as a bounding box of the XYZ extents of geometry of the part model. You can then define offsets in any of the three coordinate directions to apply to the computed bounding box.

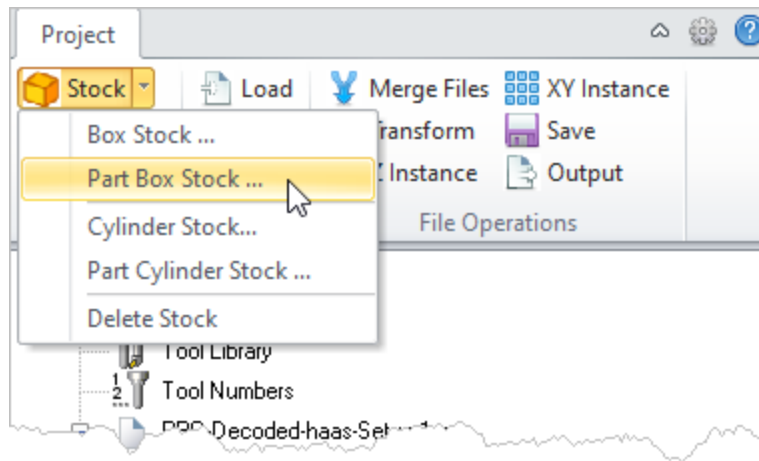


To Access this Command:

click: **Project** > **Stock** > **Part Box Stock ...**



Part Box Stock menu item

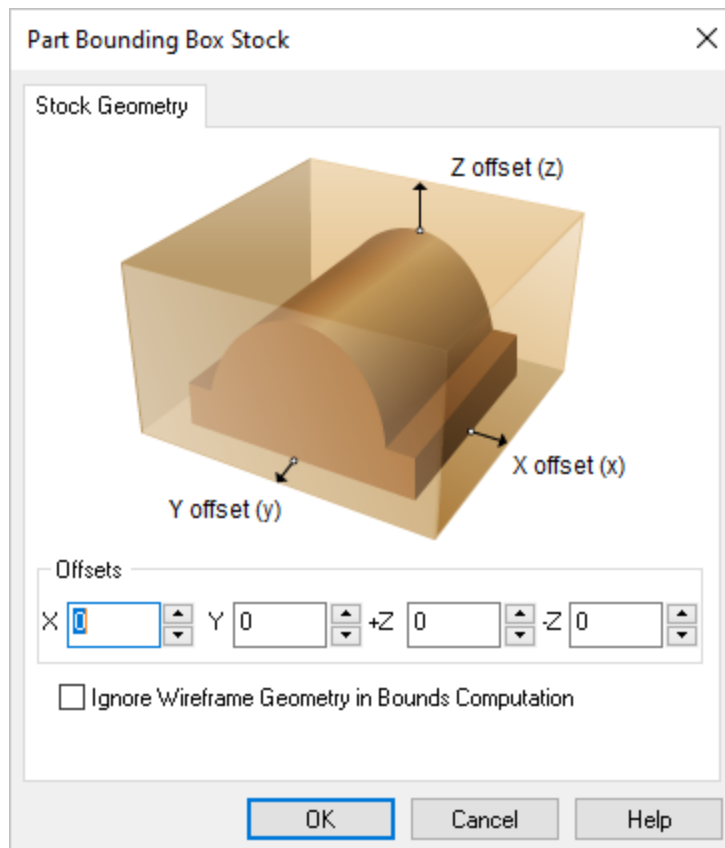


G-Code Browser: Part Box Stock menu item



Dialog Box: Part Box Stock

The system calculates the bounding box of the part model as the [XYZ](#) extents of geometry of the part model. You can then define offsets in any of the three coordinate directions to apply to the computed bounding box. The system will expand the bounding box by the offset amount in each of the coordinate directions. When you click on the [OK](#) button, a stock model based on your definition will be created and displayed.

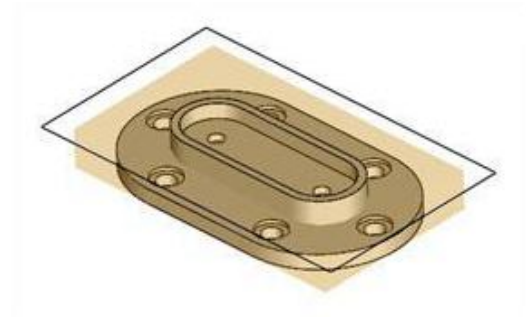


Dialog Box: Part Box Stock

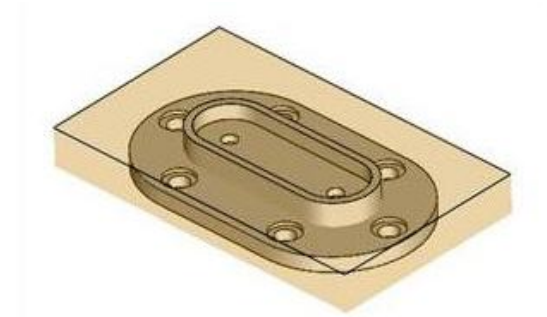


Ignore Wireframe Geometry in Bounds Computation

Check this to ignore all 2D and 3D curve geometries present in the part from stock bounding box computation.



Ignore Wireframe geometry - checked



Ignore Wireframe geometry – un-checked



Related Topics

[Stock Menu](#)

[Create Box Stock](#)

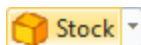
[Create Cylinder Stock](#)

[Create Part Cylinder Stock](#)

[Stock from Selection](#)

[Delete Stock](#)

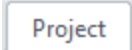
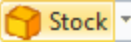
5.2.1.3 Cylinder Stock



You can define the raw stock model as a simple cylinder by selecting this option. When creating a cylinder stock, the axis of the cylinder is automatically set to the **World Z Axis** unless you select a different axis from the dialog.

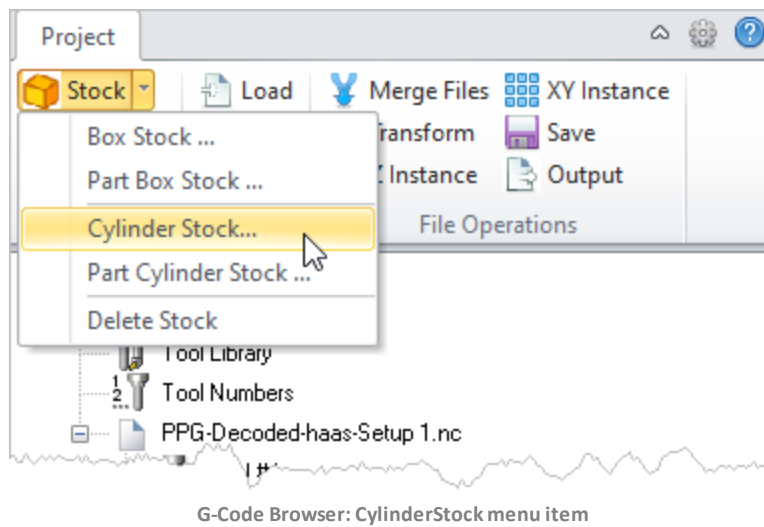


To Access this Command:

click:  >  > Cylinder Stock ...



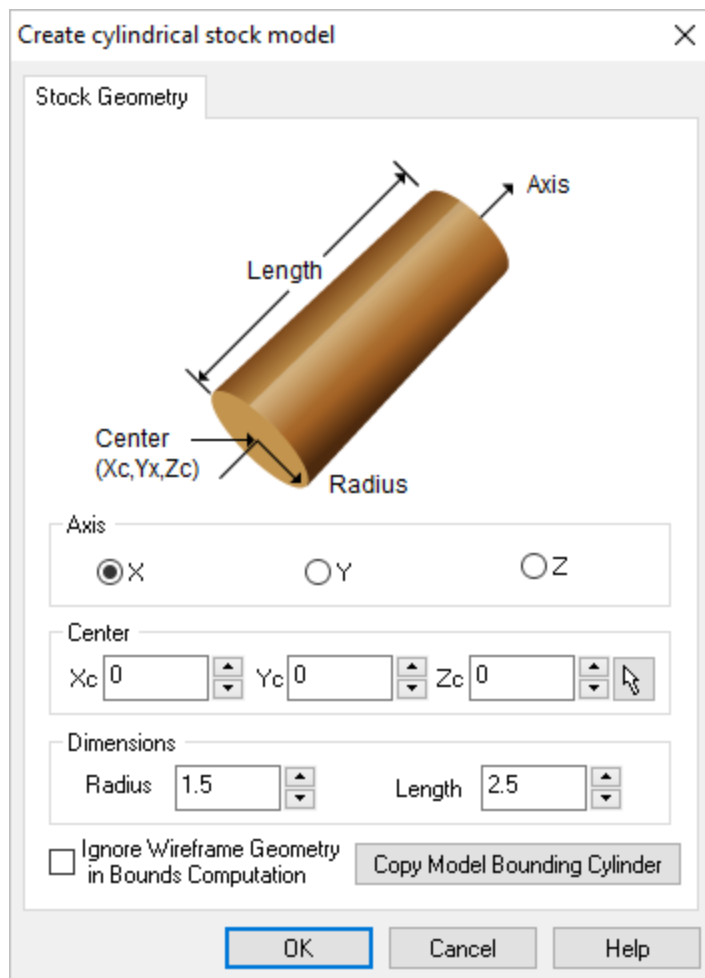
Cylinder Stock menu item



Dialog Box: Cylinder Stock

You can define the cylinder by simply selecting the axis of the cylinder (either **X** **Y** or **Z**), specifying the center coordinates by typing in the values or graphically picking the center and then by specifying the radius and the length of the cylinder.

The **Center** (origin) of the stock can also be repositioned by specifying the coordinate values in the corresponding edit boxes in the dialog (**Xc**, **Yc**, **Zc**). When you click on the **OK** button, a stock model based on your definition will be created and displayed. You can switch to the **Simulate** tab of the browser window to display the stock model that was created.



Copy Model Bounding Cylinder

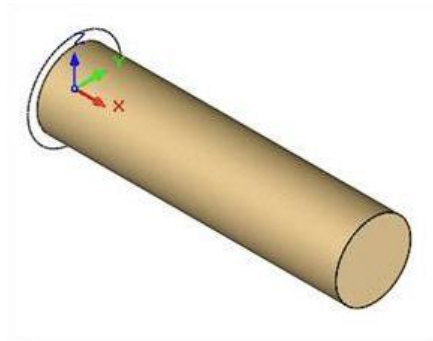
User can also select [Copy Model Bounding Cylinder](#) to determine the smaller possible stock model that fit the part geometry.



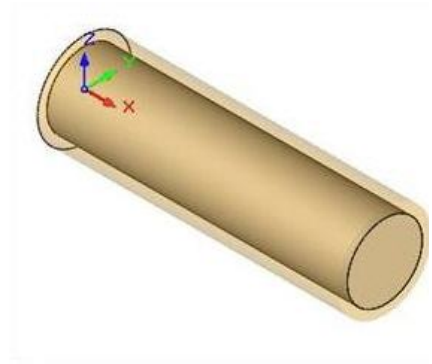
Ignore Wireframe Geometry in Bounds Computation

Check this ignore all 2D and 3D curve geometries present in the part from stock bounding cylinder computation.

Make sure to click [Copy Model Bounding Cylinder](#) after you check or uncheck Ignore Wireframe Geometry in Bounds Computation.



Ignore Wireframe geometry - checked



Ignore Wireframe geometry – unchecked



Related Topics

[Stock Menu](#)

[Create Box Stock](#)

[Create Part Box Stock](#)

[Create Part Cylinder Stock](#)

[Stock from Selection](#)

[Delete Stock](#)

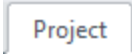

5.2.1.4 Part Cylinder Stock



You can define the raw stock model as a bounding cylinder of the **XYZ** extents of geometry of the part model. You can then define offsets to apply to the computed bounding box.

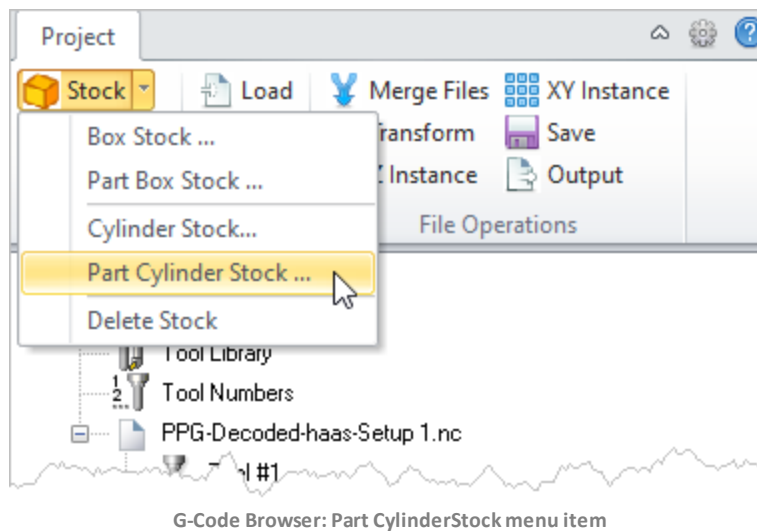


To Access this Command:

click:  >  > Part Cylinder Stock ...



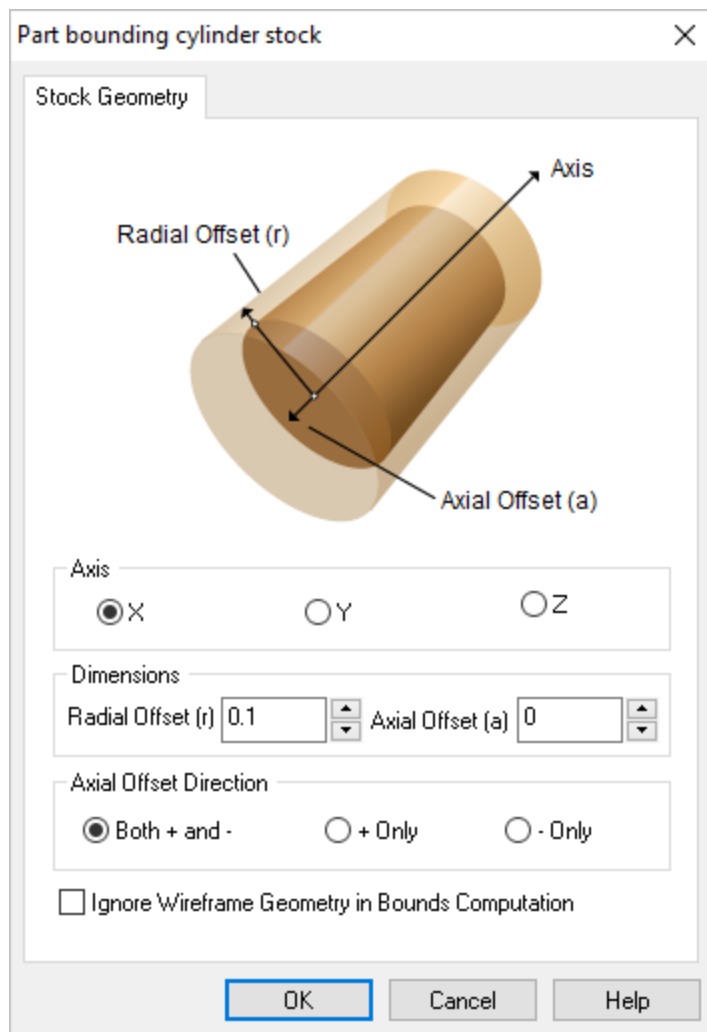
Part Cylinder Stock menu item



Dialog Box: Part Cylinder Stock

You can define the cylinder by simply selecting the axis of the cylinder (either **X**, **Y** or **Z**) and defining the radial and the axial offset distances. The bounding cylinder will be calculated and these offset values will be used to expand the cylinder in both the radial and axial direction.

Additionally you can specify the axial offset to be applied in both the positive and negative directions or only in the positive or only in the negative axial directions.



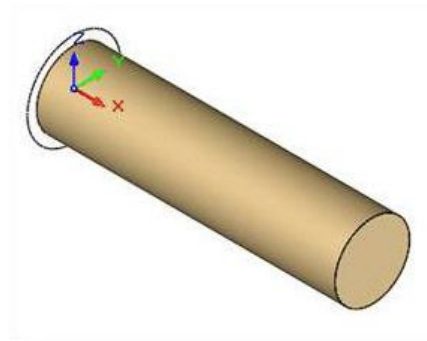
Dialog Box: Part Cylinder Stock



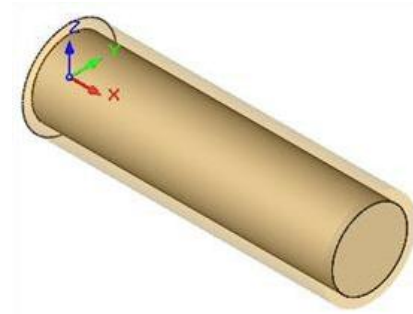
Ignore Wireframe Geometry in Bounds Computation

Check this to ignore all 2D and 3D curve geometries present in the part from stock bounding cylinder computation.

When you click on the **OK** button, a stock model based on the part geometry and your definition will be created. You can switch to the Stimulate tab of the browser window to display the stock model that was created.



Ignore Wireframe geometry - checked



Ignore Wireframe geometry – un-checked



Related Topics

[Stock Menu](#)

[Create Box Stock](#)

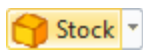
[Create Part Box Stock](#)

[Create Cylinder Stock](#)

[Stock from Selection](#)

[Delete Stock](#)

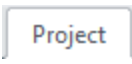
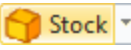
5.2.1.5 Delete Stock



Use this command to Delete the currently defined **Stock** model. If there is no stock defined, **Simulations** will only show tool motions and not a cut material simulation.

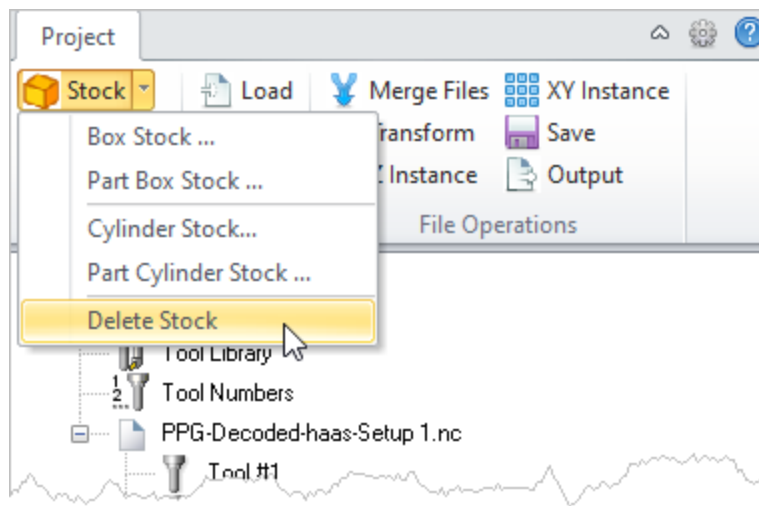


To Access this Command:

click:  >  > Delete Stock



Delete Stock menu item



G-Code Browser: Delete Stock menu item



Related Topics

[Stock Menu](#)

[Create Box Stock](#)

[Create Part Box Stock](#)

[Create Cylinder Stock](#)

[Create Part Cylinder Stock](#)

[Stock from Selection](#)

5.2.1.6 Stock from Selection



You can define the raw stock model from the currently active selected 3-D geometry. You can select 3D surfaces and/or meshes and then select [Stock from Selection](#) option from [Create Stock Model](#) under the [Program](#) tab in [Machining Browser](#).

No dialog will be invoked but the system will use the selected geometry and create a triangulated stock model. You can switch to the [Stock](#) tab of the browser window to display the stock model that was created.



[Stock from Selection](#) will fail to create stock if the selected 3D geometry is not a watertight model.

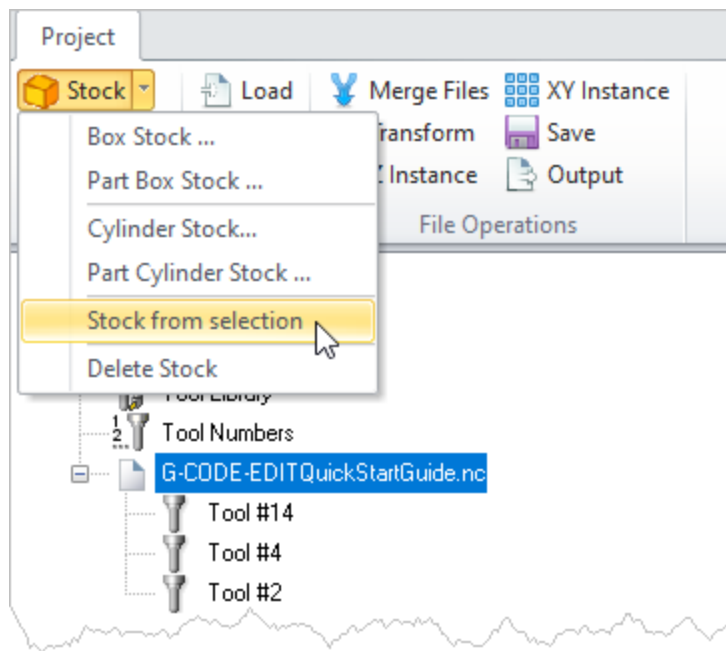


To Access this Command:

click: > > [Stock from Selection](#)



Stock from Selection menu item

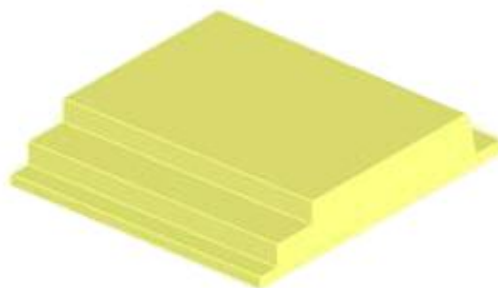


G-Code Browser: Stock from Selection menu item

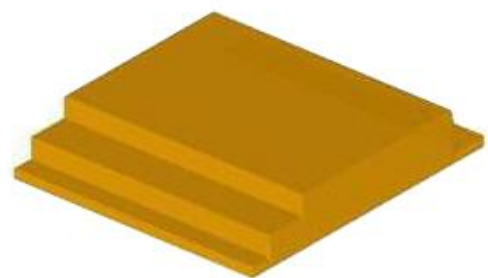


Stock from Selection Example

You can switch to the [Simulate](#) tab of the browser window to display the stock model that was created.



3D geometry selection for stock



Stock created from 3D geometry



Related Topics

[Stock Menu](#)

[Create Box Stock](#)

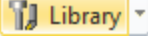
[Create Part Box Stock](#)

[Create Cylinder Stock](#)

[Create Part Cylinder Stock](#)

[Delete Stock](#)

5.2.2 Tool Library

 Use this command to load a pre-defined [Tool Library](#) file (*.csv). You can save a [Tool Library](#) (*.csv) file from the [MILL](#) module [Save Tool Library](#) command. When you create your [Tool Crib](#), the tools are selected from this [Tool Library](#).

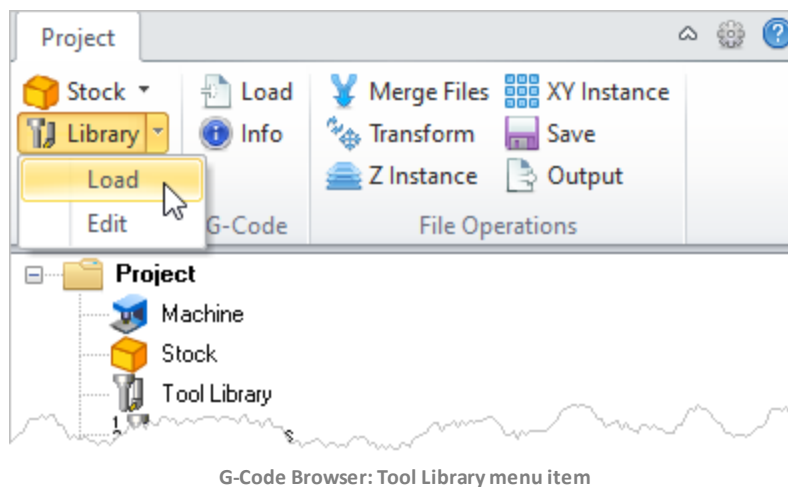


To Access this Command:

click:  >  > Load



Tool Library menu item



Procedure

1. Select [Library](#) from the [Project](#) tab of the [G-Code Browser](#) and pick [Load](#).
2. The [File Open](#) dialog will display. Navigate to the [Tool Library](#) file and pick [Open](#) to load it. The file extension must be *.csv. You can save a [Tool Library](#) (*.csv) file from the [MILL](#) module [Save Tool Library](#) command.
3. The [Tool Library](#) is loaded.

Tool Library

Name	Tool Type	Diameter	Comer Radius	Taper	Flute Length	Tool Length	Tool #
.375 4FL (2.550 Loc)	Mill	0.375	0	0	2.55	4.23	0
.375 4FL (1.150 Loc)	Mill	0.375	0	0	1.15	3.05	0
.375 4FL (.950 Loc)	Mill	0.375	0	0	0.95	1.5	0
.375 2FL (1.180 Loc)	Mill	0.375	0	0	1.18	3.04	0
.375 2FL (.900 Loc)	Mill	0.375	0	0	0.9	2.53	0
.375 4 FL (1.050 Loc)	Mill	0.375	0.1875	0	1.05	1.54	0
.312 2 FL (.850 Loc)	Mill	0.375	0.1875	0	0.85	2	0
.250 4 FL (.750 Loc)	Mill	0.25	0.125	0	0.75	2.5	0
.250 2 FL (.750 Loc)	Mill	0.25	0.125	0	0.75	2.5	0
.250 2 FL (.900 Loc)	Mill	0.25	0	0	0.9	2.5	0

OK

Cancel

Help

A Tool Library is loaded

4. Selecting a tool type icon from the menu at the top will display those tool types.

Tool Library

Name	Tool Type	Diameter	Comer Radius	Taper	Flute Length	Tool Length	Tool #
1/16 Drill	Drill	0.0625	0	0	2.5	4	0
1/16-Drill	Drill	0.0625	0	0	1.15	2.75	0
1/4-Drill	Drill	0.25	0	0	1	3	0
1/8-Drill	Drill	0.125	0	0	1	3	0

Listing only Drill Tools




















5. The default [Tool Library \(*.csv\)](#) files saved from the [MILL](#) module contains the following data fields as shown in each column of the dialog. You can edit *.csv files using an [Excel](#) compatible spreadsheet program.

Default [Tool](#) Data Fields Supported by the [MILL](#) Module are:

- [Name](#)
 - [Tool Type](#)
 - [Diameter](#)
 - [Corner Radius](#)
 - [Taper](#)
 - [Flute Length](#)
 - [Tool Length](#)
- [Tool Number](#)
 - [Tool Material](#)
 - [Spindle RPM](#)
 - [Cut Feed](#)
 - [Adjust Register](#)
 - [Cutcom Register](#)
 - [Comments](#)

Tool Library Toolbar

Select an icon from the toolbar to display only those tools.

Icon	Type	Icon	Type	Icon	Type
	Ball		Face		Ream
	Flat		Dove Tail		Tap
	Corner Rad		Fillet		Bore
	Vee		Lollypop		RBores
	Chamfer		User Defined		Show All
	Taper		Drill		
	Thread		Center Drill		



Related Topics

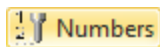
[User Interface](#)

[Project Tab](#)

[Stock Menu](#)

[Tool Crib](#)

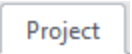
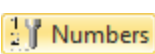
5.2.3 Tool Crib



Use this command to create a **Tool Crib** for the current **Project**. This will assign a tool station number to each tool you load into the **Tool Crib**. Be sure to assign the same **Tools** and **Tool Numbers** that are used in your g-code files.

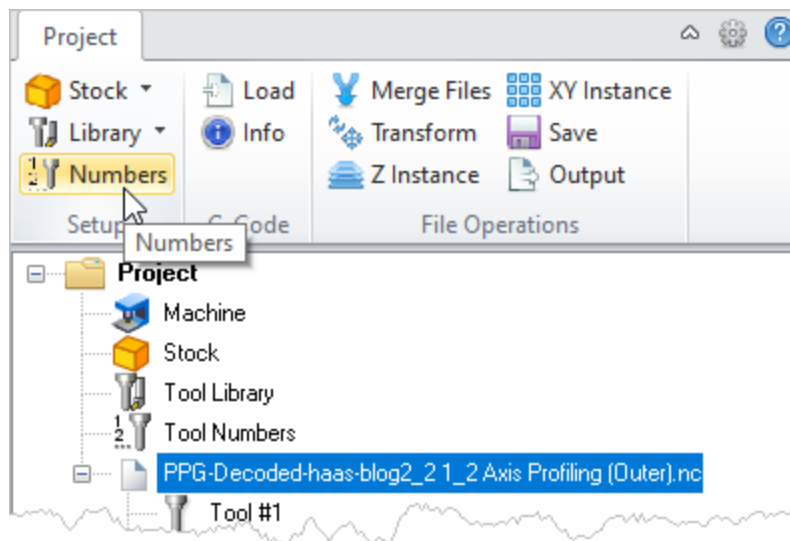


To Access this Command:

click:  > 



Tool Crib menu item

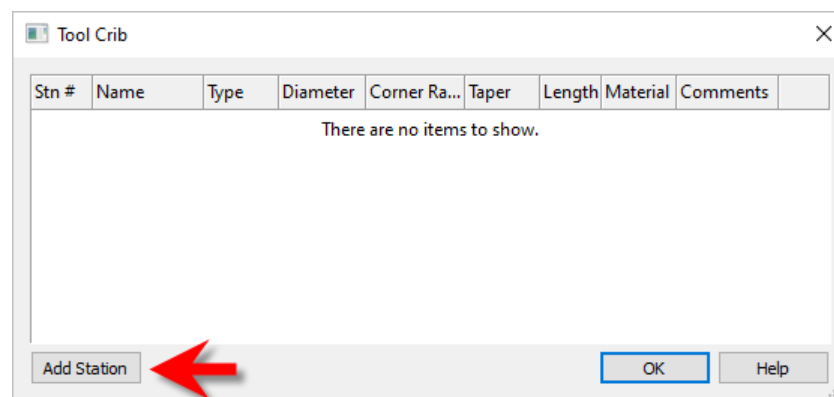


G-Code Browser: Tool Crib menu item



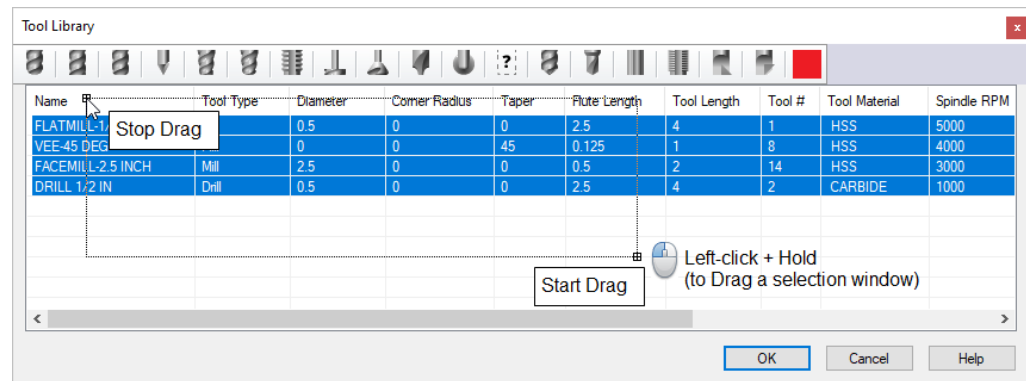
Procedure

1. Select **Numbers** from the **Project** tab of the G-Code Browser.
2. The **Tool Crib** dialog will display.
Select the **Add Station** button to grab a tool from the default **Tool Library**.



The Tool Crib dialog, Add Station button

3. The **Tool Crib** dialog will minimize and the **Tool Library** dialog will display. Select a tool from the library and then pick **OK**. The selected tool will be assigned as tool station #1 (i.e., Tool #1).



Select a Tool from the Library and pick OK.

- You can use the icons on the right side of each **Tool Station #** to delete that station or reassign a tool to that station. Pick **OK** to accept the **Tool Crib**.



Related Topics

[User Interface](#)

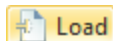
[Project Tab](#)

[Stock Menu](#)

[Tool Library](#)

5.3 G-Code Pane

5.3.1 Load



Use this command to Load a g-code file into the **G-Code Browser**. You can load as many g-code files as you wish and work with them from the **Project** tab of the **G-Code Browser**. The example below shows three files loaded.



To Access this Command:

click: **Project** > **Load**

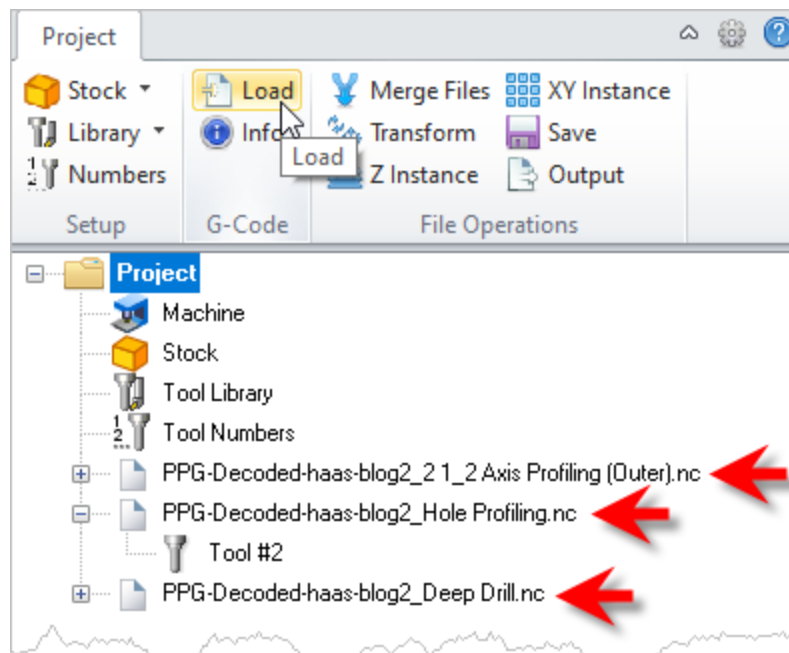
Then use the File Browser to select a G-Code file to load.

The following G-Code file type extensions are supported:

GCODE Files (*.nc)
PIM Files (*.pim)
All GCODE Files (*.nc;*.pim;*.tap;*.gc;*.txt;*.mpf;*.fnc;*.iso;*.cnc)
 File Extensions Supported by G-Code Editor



Load menu item



G-Code Browser: Load g-code file menu item



Related Topics

[User Interface](#)

[Project Tab](#)

[Stock Menu](#)

[Tool Library](#)

[Tool Crib](#)

[Info](#)

5.3.2 Info



Use this command to get Information about all of the g-code files that are currently loaded into the **Project** of the **G-Code Browser**. Information includes file **Name**, **Tool #**, **# of GoTo motions**, **Estimated Machining Time** and file **Size**.

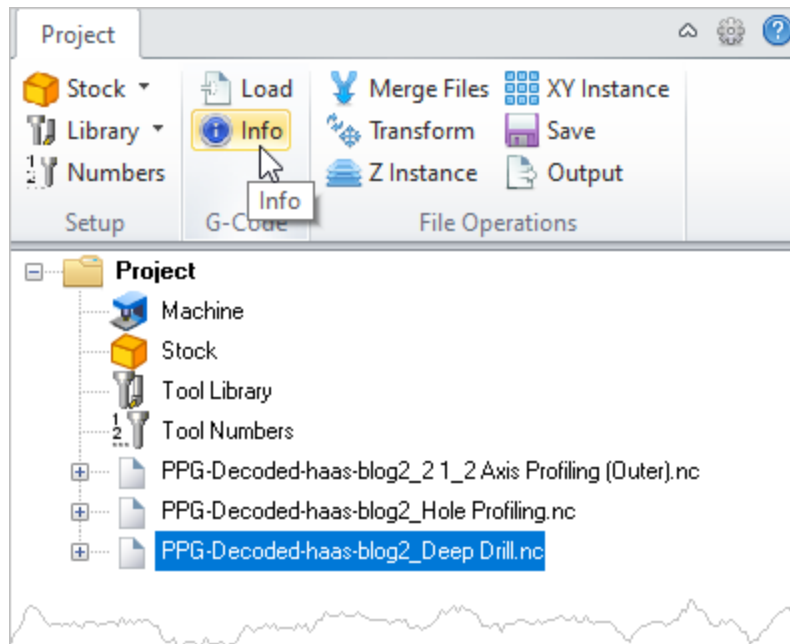


To Access this Command:

click: **Project** > **Info**



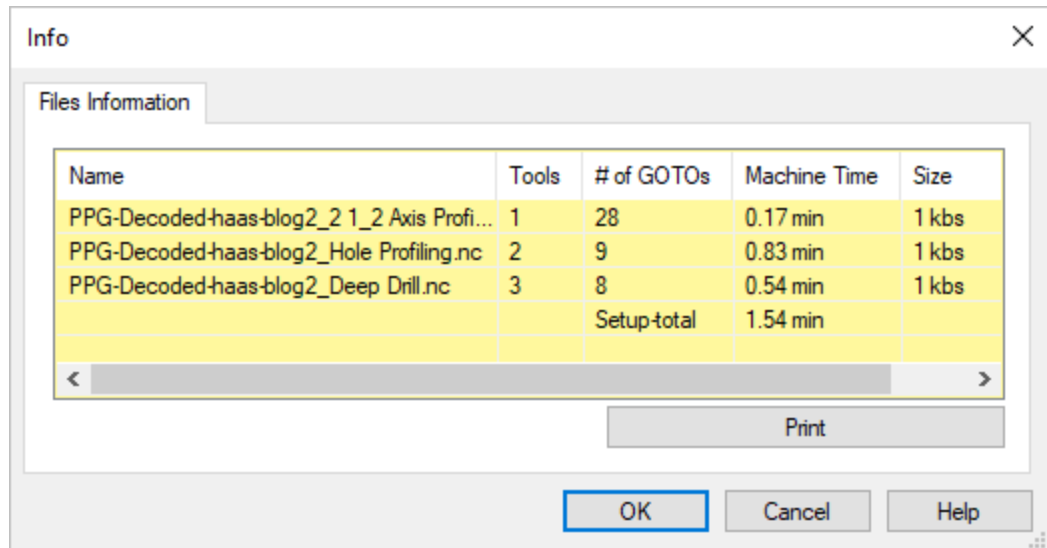
Info menu item



G-Code Browser: Info menu item



G-Code Files Information Dialog



Files Information Dialog



Related Topics

[User Interface](#)

[Project Tab](#)

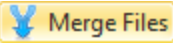
[Stock Menu](#)

[Tool Library](#)

[Tool Crib](#)[Load](#)

5.4 File Operations Pane

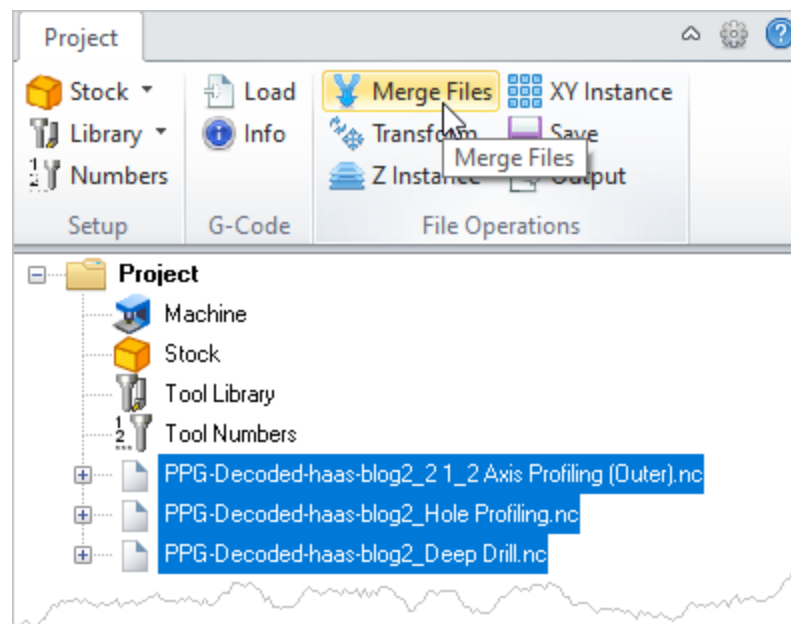
5.4.1 Merge Files

 **Merge Files** Use this command to merge g-code files that are currently loaded into the [Project](#) tab of the [G-Code Browser](#). Merged files can be output as one g-code file using the [Output](#) command.

 **To Access this Command:**

click:  > 

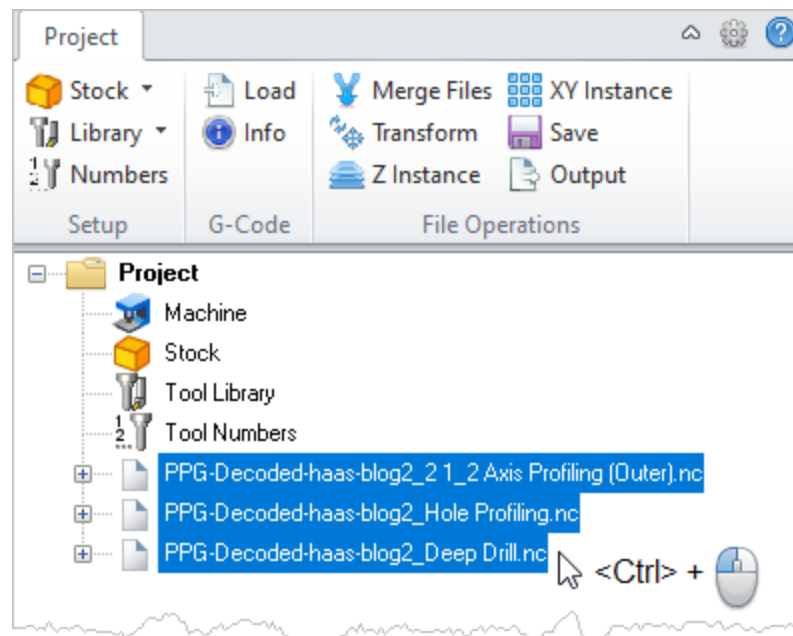
 **Merge Files menu item**



G-Code Browser: Merge Files menu item

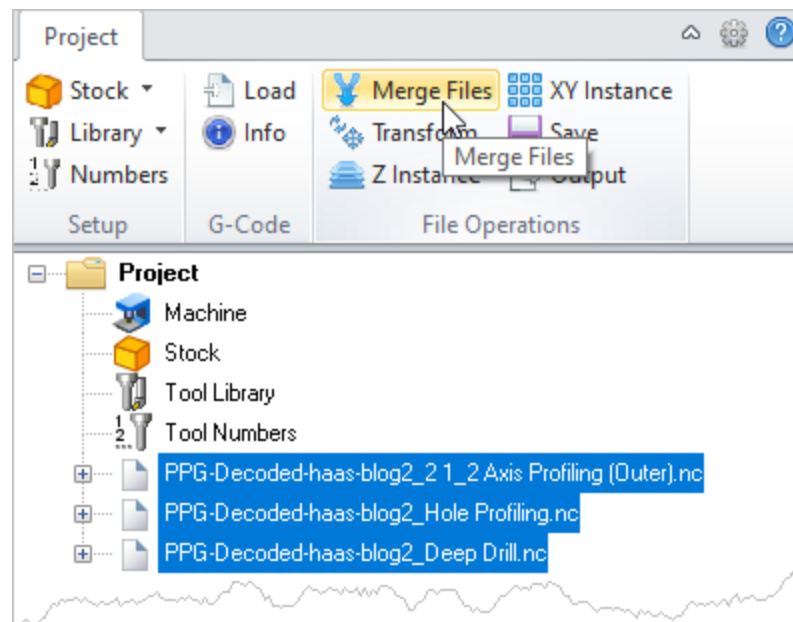
 **Procedure**

1. From the [Project](#) tab of the [G-Code Browser](#), press and hold the <Ctrl> key while left-click selecting the g-code files that you want to merge.



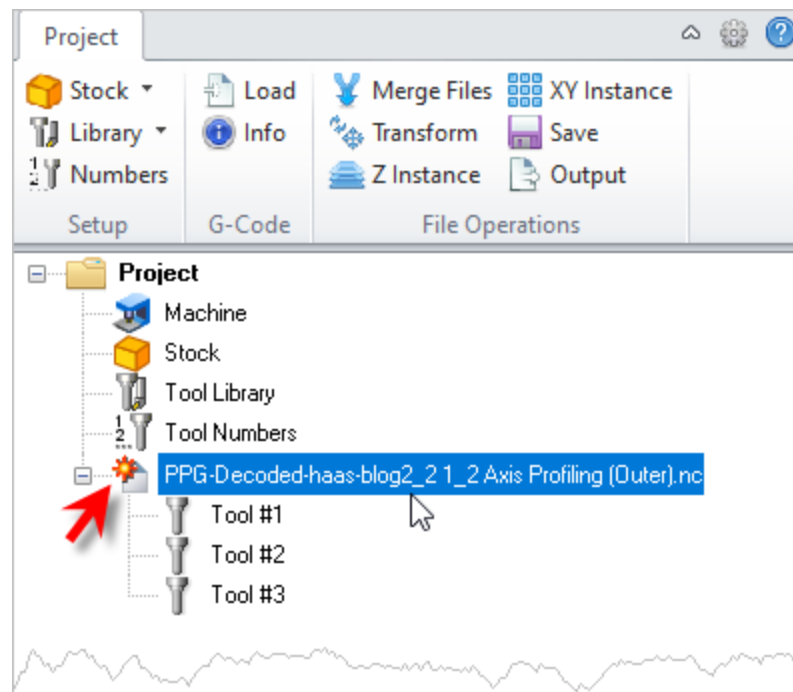
G-Code Browser: Merge Files menu item

2. With the g-code files highlighted and selected, pick the [Merge Files](#) command from the [Project](#) tab.



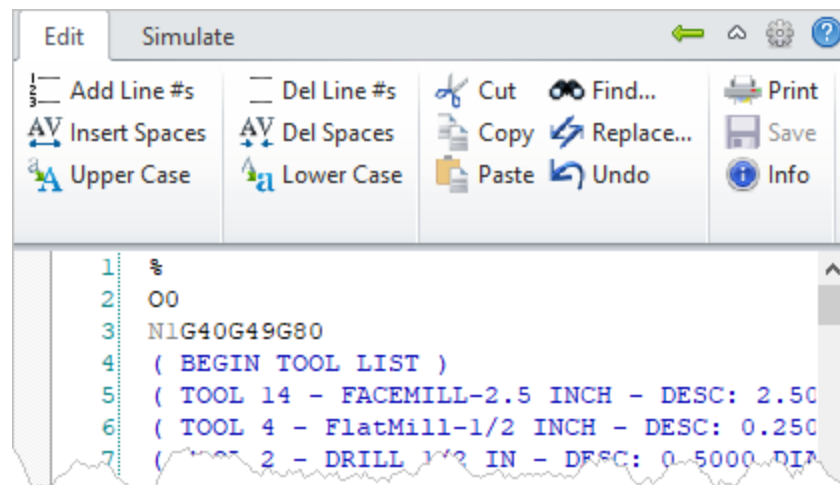
G-Code Browser: Merge Files menu item

3. In the example above we have selected three g-code files to merge. The second and third files are merged into the first selected file that appears in the Project.



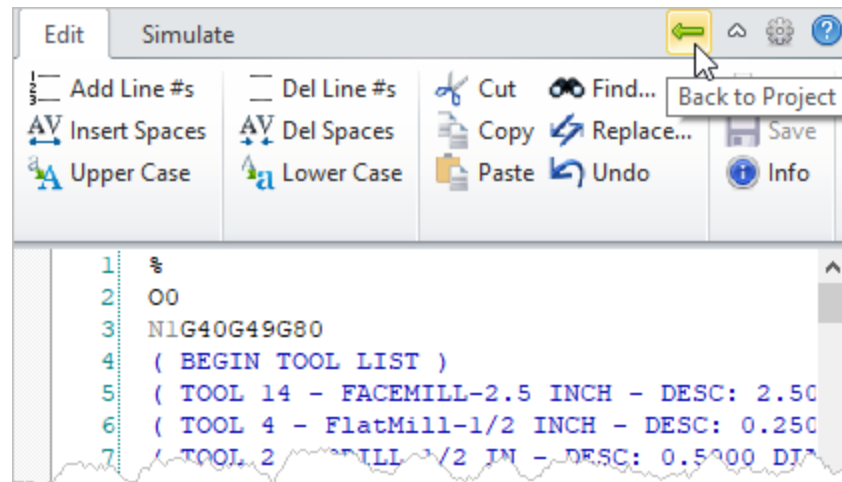
The second and third files are merged into the first selected file that appears in the Project.

4. Because the first g-code file has now changed, it is flagged (at the red arrow in the image above) to let you know that it has not yet been saved.
5. A double-left-click on the g-code file will load it into the [G-Code Editor](#) for viewing.



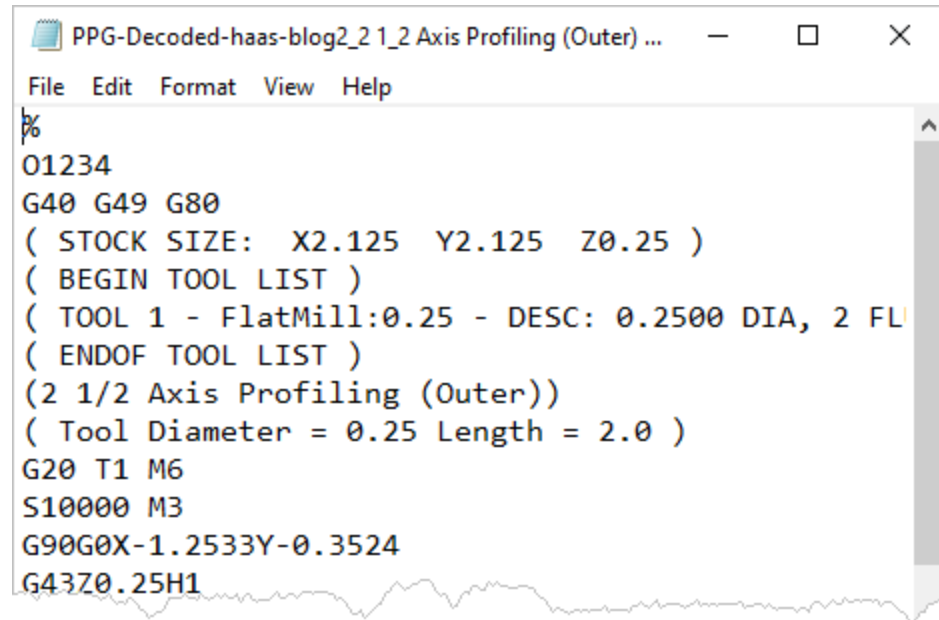
The merged file is loaded into the G-Code Editor

6. When done editing the g-code file, select the [Back to Project](#) icon to return to the [Project](#) tab.



Back to Project menu item

7. When satisfied, select the [Output](#) icon to save the new edited g-code file.



The merged g-code file is displayed



Related Topics

[User Interface](#)

[Project Tab](#)

[Load](#)

[Info](#)

[Transform](#)

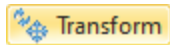
[Z Instance](#)

[XY Instance](#)

[Save](#)

[Output](#)

5.4.2 Transform



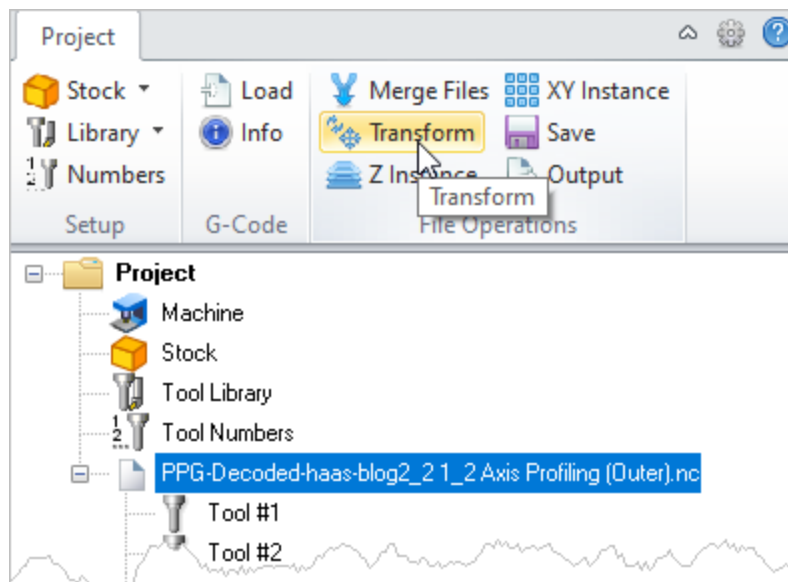
You can use this command to perform transformations on a selected g-code file. The transformations include [Move](#), [Rotate](#), [Mirror](#) and [Scale](#). Each method is documented in the topics below. Each tab of the [Transform](#) dialog below includes an example.



To Access this Command:



Transform menu item



G-Code Browser: Transform menu item



Procedure

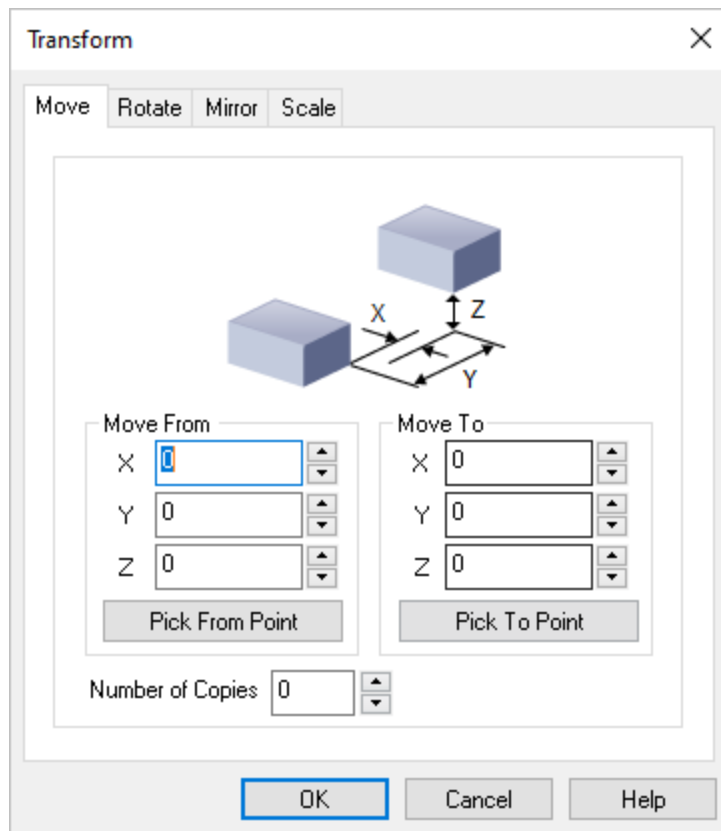
1. Select the g-code file from the [Project](#) tab that you wish to transform.
2. Select [Transform](#) from the [Project](#) tab menu. The [Transform](#) dialog will display.
3. Select the type of transformation to make by selecting the corresponding tab at the top of the dialog ([Move](#), [Rotate](#), [Mirror](#) or [Scale](#)).
4. Complete the necessary data fields and then pick **OK** from the dialog to perform the transformation.

- The transformation is displayed in the graphics screen and the g-code file is flagged for [Output](#).



Transform dialog, Move tab

The [Move](#) tab of the [Transform Toolpath](#) dialog allows you to translate the toolpath by specifying [From](#) and [To](#) points for the move. You can either specify these points by coordinate entry in the corresponding edit fields or could graphically pick the points by selecting the [Pick](#) buttons. Once the points are specified, selecting the apply button will perform the move operation.



Dialog Box: Transform Toolpath, Move tab

Move From

Enter the [X,Y,Z](#) coordinate values for the [Move From](#) point or select the [Pick From Part](#) button to select a point from the model.

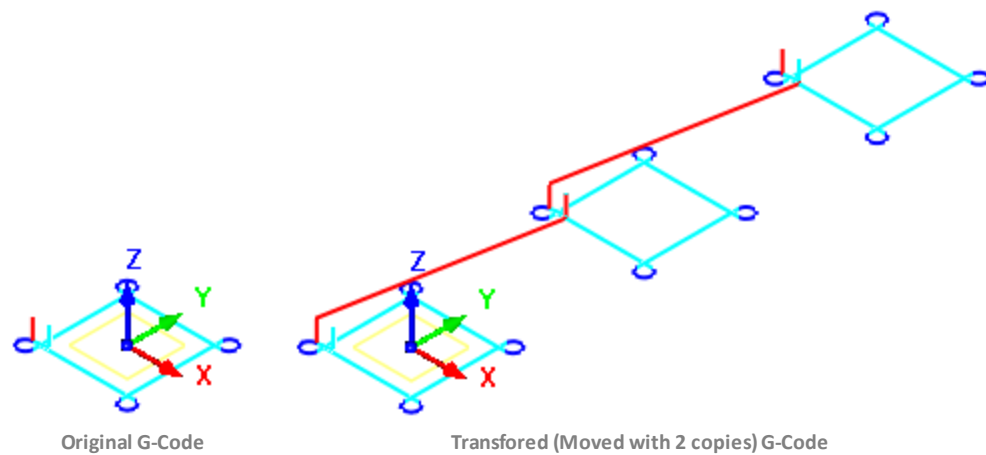
Move To

Enter the [X,Y,Z](#) coordinate values for the [Move To](#) point or select the [Pick To Part](#) button to select a point from the model.

Number of Copies

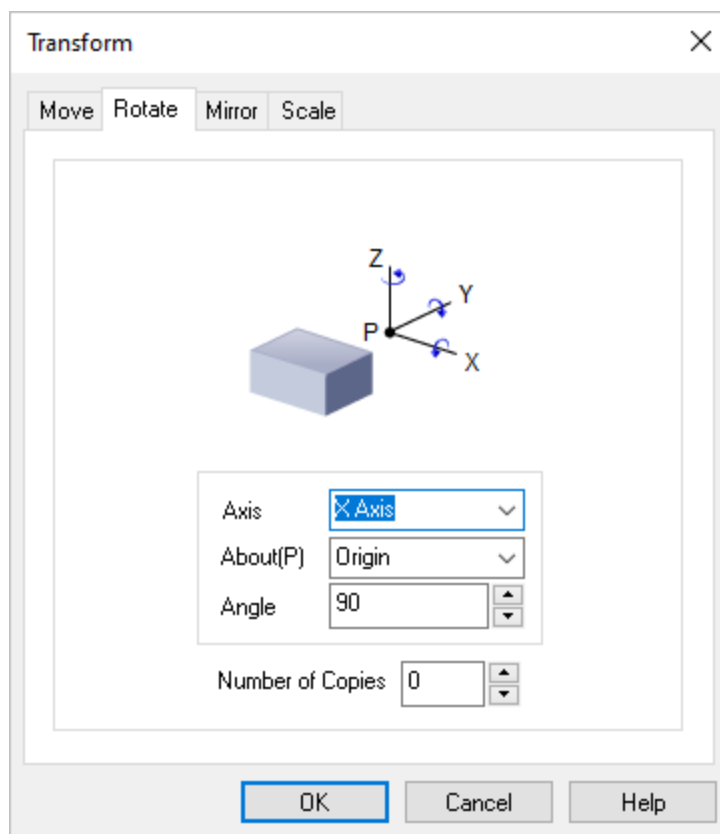
You can also [Copy](#) the toolpath by entering the number of copies.

Example: The original g-code file is moved along the Y axis making 3 copies.



Transform Dialog, Rotate tab

Selecting the **Rotate** tab allows you to rotate the toolpath.



Dialog Box: Transform Toolpath, Rotate tab

Axis / About(P) / Angle

Rotation can be performed around any of the three principal axes. The angle of rotation as well as the point and **Axis** of rotation can be specified.

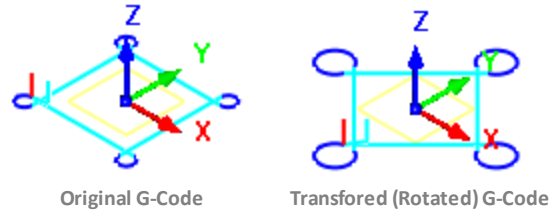
The point of rotation could be one of the **Origin**, **Center** of the part, **Minimum** point of the part and the **Maximum** point of the part.

The **Axis** of rotation could be the **X**, **Y** or the **Z Axis**. You can specify any arbitrary rotation **Angle**.

Number of Copies

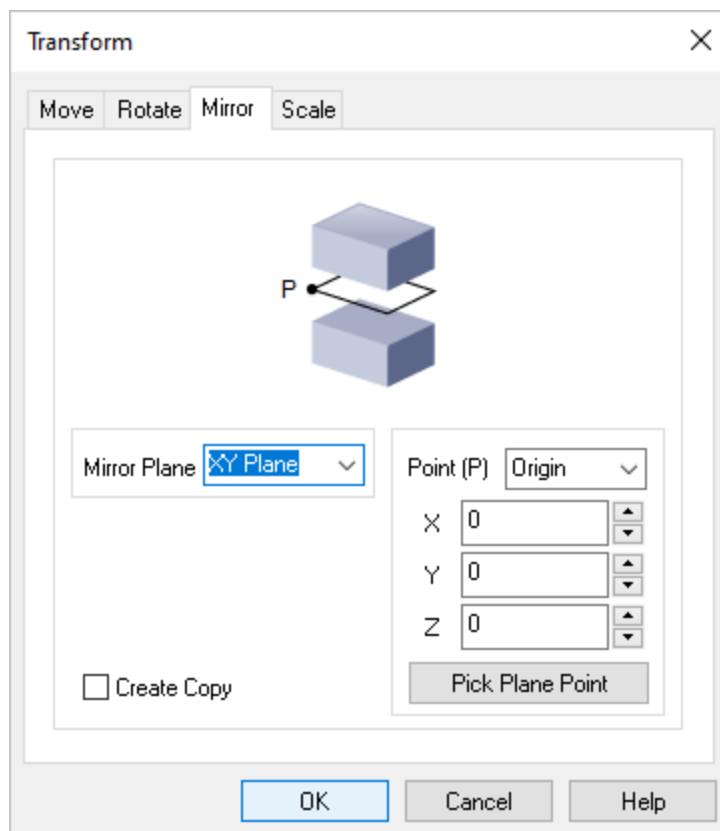
You can also **Copy** the toolpath by entering the number of copies.

Example: The original g-code file is rotated about the Z axis by 45 degrees.



Transform Dialog, Mirror tab

Selecting the **Mirror** tab allows you to Mirror the selected toolpath.



Dialog Box: Transform Toolpath, Mirror tab

Mirror Plane

Select the **Mirror Plane** from the list (**XY Plane**, **XZ Plane** or **YZ Plane**).

Point (P)

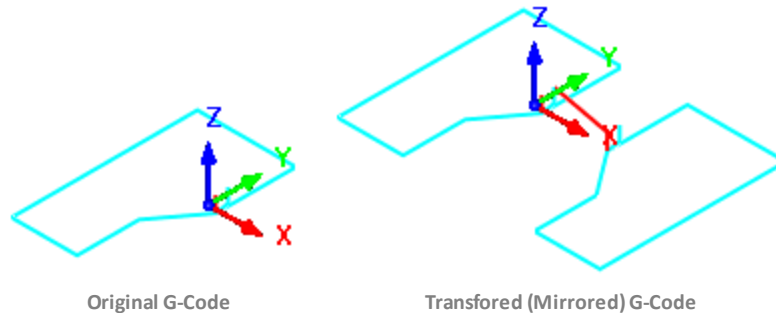
Select a **Mirror Point (P)** from the list (**Origin**, **Center** of the part, **Minimum** point of the part) or enter the **X,Y,Z** coordinate values of the desired point.

You can also select the **Pick Plane Point** button to select a point from the model.

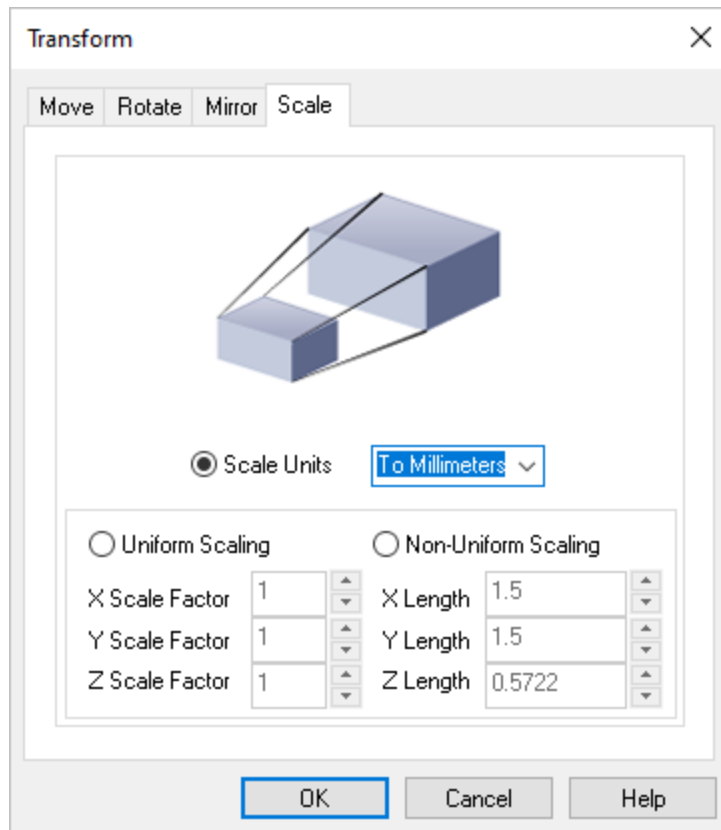
Create Copy

Check this box to create a **Copy** of the mirrored toolpath.

Example: The original g-code file is mirrored about the YZ plane keeping the original.

**Transform Dialog, Scale tab**

Selecting the **Scale** tab allows you to scale the selected toolpath. All scaling is performed about the zero of the coordinate axes. You can optionally scale from a mm to an inch or vice-versa or by specifying a scale factor. You can also scale the selections uniformly or perform non-uniform scaling in each of the three principal axes.



Dialog Box: Transform Toolpath, Scale tab

Scale Units

Select the **Scale Units** from the list (**To Millimeters** or **To Inches**).

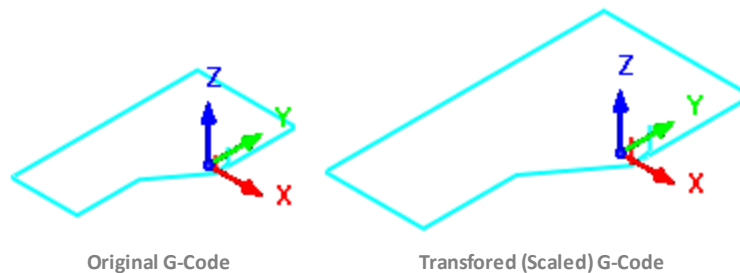
Uniform Scaling

Select this option and then enter the uniform **X**, **Y** and **Z** scale factors.

Non-Uniform Scaling

Select this option and then enter the actual non-uniform **X**, **Y** and **Z** length values.

Example: The original g-code file is scaled by 1.5 times the size of the original.



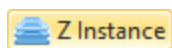
Related Topics

[User Interface](#)

[Project Tab](#)

[Load](#)[Info](#)[Merge Files](#)[Z Instance](#)[XY Instance](#)[Save](#)[Output](#)

5.4.3 Z Instance



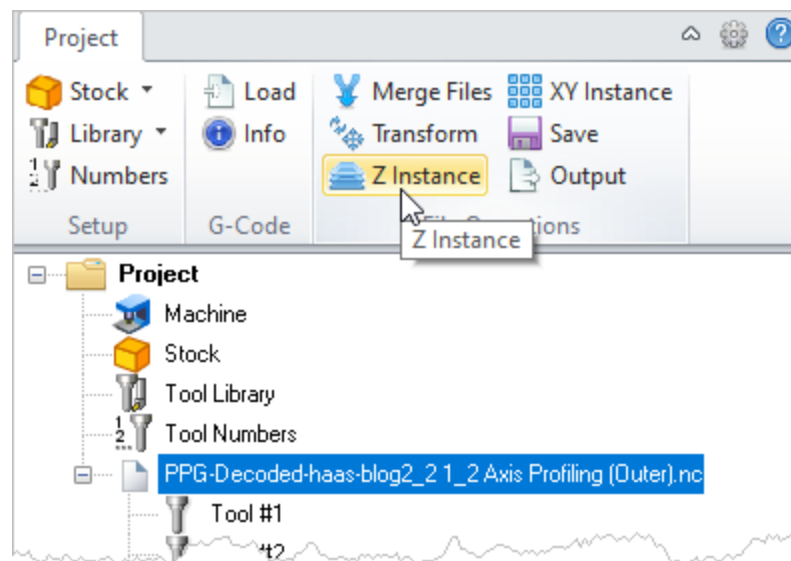
You can create multiple Z level instances of the toolpath via this dialog. This can be useful in situations where you are manufacturing multiple instances of the same part or want to create multiple z levels of the original path.



To Access this Command:



Z Instance menu item



G-Code Browser: Z Instance menu item



Procedure

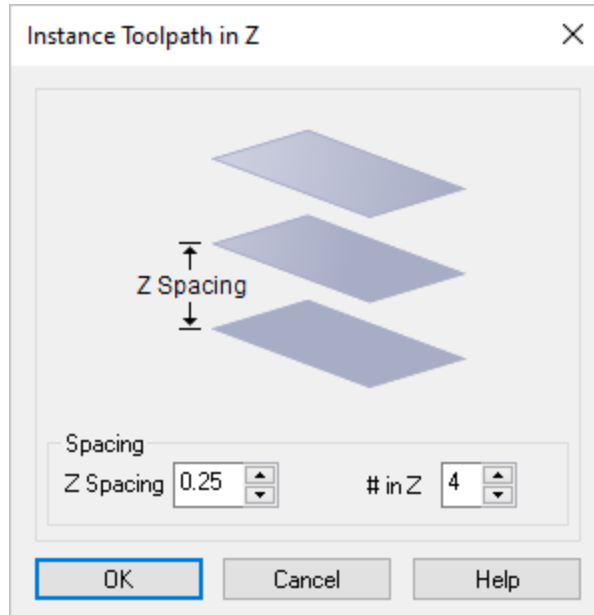
1. Select the g-code file from the [Project](#) tab that you wish to Z Instance.
2. Select [Z Instance](#) from the [Project](#) tab menu. The [Z Instance](#) dialog will display.

3. Enter the Z Spacing and # in Z values and then pick **OK** from the dialog.
4. The g-code is instanced in the Z axis.



Instance Toolpath in Z dialog

The g-code will be copied multiple times in the Z axis. Negative values can be specified in the Z spacing. In this case the instances will proceed below the current location of the toolpath.



Dialog Box: Instance Toolpath in Z Dialog

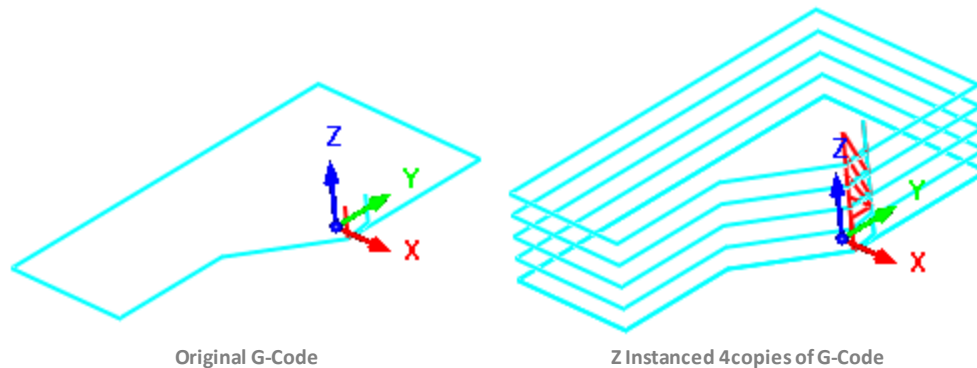
Spacing

Specify the Spacing between the toolpaths in the Z direction.

in Z

Specify the total number of instances of the toolpath to create.

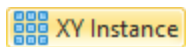
Example: The original g-code file is moved along the Y axis making 3 copies.



Related Topics

[User Interface](#)[Project Tab](#)[Load](#)[Info](#)[Merge Files](#)[Transform](#)[XY Instance](#)[Save](#)[Output](#)

5.4.4 XY Instance



You can create multiple instances of the g-code using this command. This can be useful in situations where you are manufacturing multiple instances of the same part.

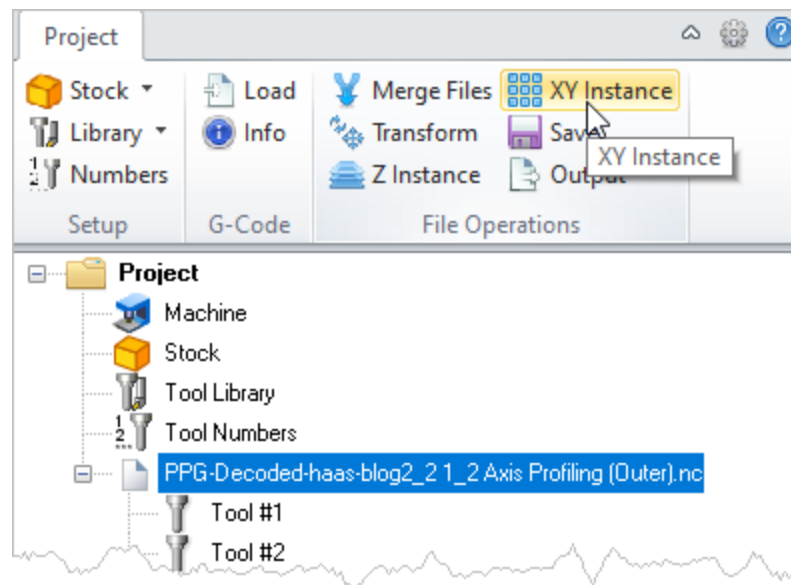


To Access this Command:

click: Project > XY Instance



XY Instance menu item



G-Code Browser: XY Instance menu item



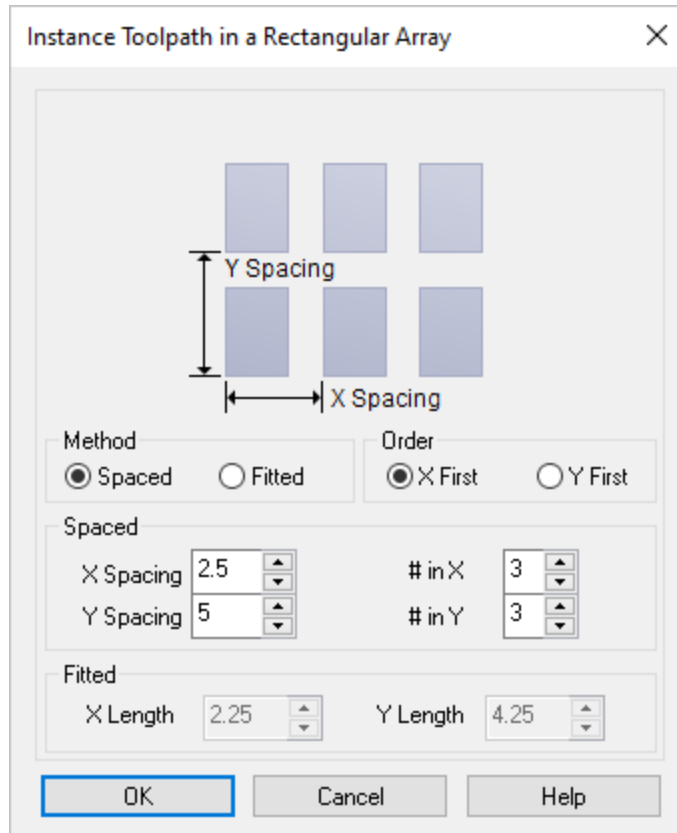
Procedure

1. Select the g-code file from the **Project** tab that you wish to Z Instance.
2. Select **Z Instance** from the **Project** tab menu. The **Z Instance** dialog will display.
3. Enter the Z Spacing and # in Z values and then pick **OK** from the dialog.
4. The g-code is instanced in the Z axis.



Instance Toolpath in a Rectangular Array dialog

You can specify multiple instances in the X and Y directions in two different ways.



Dialog Box: XY Instance Dialog



Method = Spaced

Using the **Spaced** method, you specify the independent **X Spacing** and **Y Spacing** values between the toolpaths.

The spacing is then used and applied to the bounding box of the part geometry as shown in the picture on the dialog.

Enter the number of copies of the instanced toolpath using the **# in X** and **# in Y** values.



Method = Fitted

Using the [Fitted](#) method, you specify a rectangle in which the toolpaths will be fitted by using the [X Length](#) and [Y Length](#) values.

The bounding box of the part geometry is considered when computing the number of toolpaths that can be fitted in the specified space.



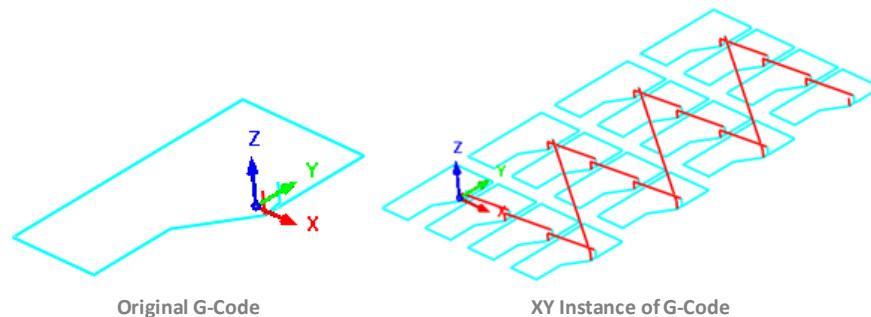
Order

The order of the instancing can also be specified by selecting either [X First](#) or [Y First](#).



Example

Example: The original g-code file is arrayed on the XY plane spaced 2.5 apart in X and 5.0 apart in Y with a total of 3 copies in each direction.



Related Topics

[User Interface](#)

[Project Tab](#)

[Load](#)

[Info](#)

[Merge Files](#)

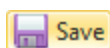
[Transform](#)

[Z Instance](#)

[Save](#)

[Output](#)

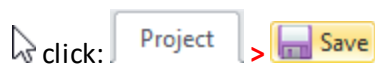
5.4.5 Save



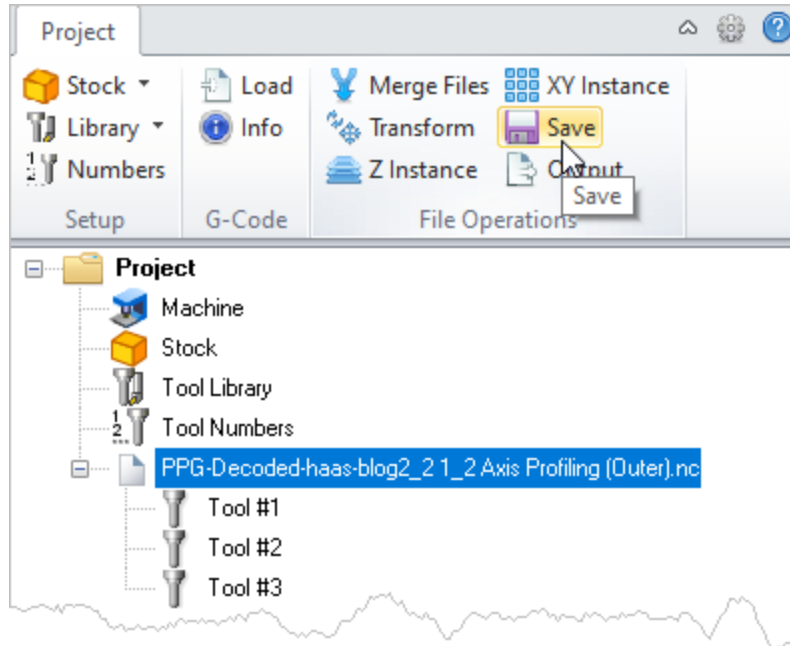
Use this command to [Save](#) a selected G-Code file to the [Save Directory](#) specified in the Preferences dialog under [Text Editor Preferences](#).



To Access this Command:



Save menu item



G-Code Browser: Save menu item



Procedure

1. Select one or more g-code files from the [Project](#) tab.
2. Select [Save](#) from the [Project](#) tab menu. The g-code files are saved to the [Save Directory](#) specified in the [Text Editor Preferences](#) dialog.

This is the save directly by default:

[C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino 7.0\Edited files\](#)



Related Topics

[User Interface](#)

[Project Tab](#)

[Load](#)

[Info](#)

[Merge Files](#)

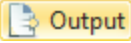
[Transform](#)

[Z Instance](#)

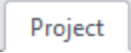
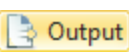
[XY Instance](#)

[Output](#)

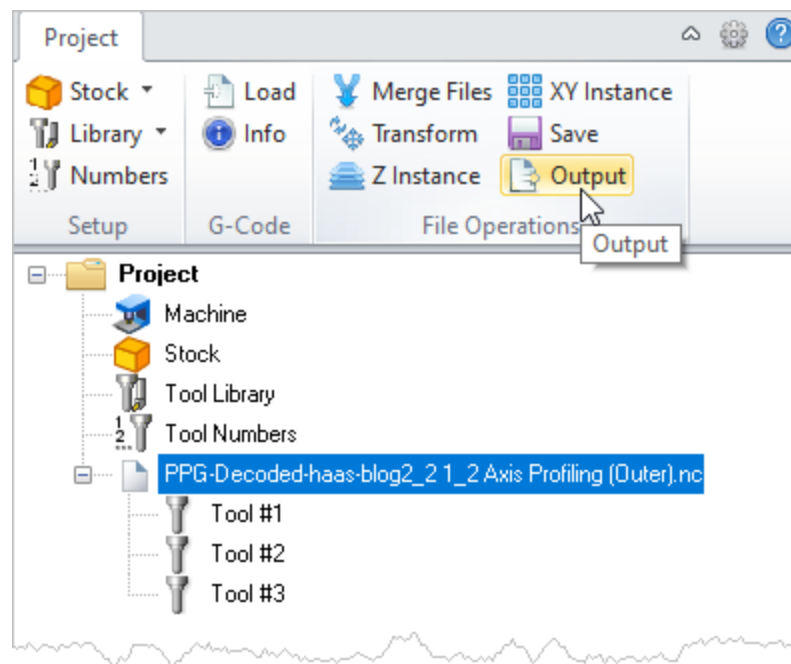
5.4.6 Output

 Use this command to [Output](#) a selected G-Code file to the [program](#) specified in the [Preferences](#) dialog under [Text Editor Preferences](#). For example it could be output to another text editor or a DNC program to go directly to your CNC machine.

 **To Access this Command:**

click:  > 

 **Output menu item**



G-Code Browser: Output menu item

 **Procedure**

1. Select a g-code files from the [Project](#) tab.
2. Select [Output](#) from the [Project](#) tab menu. The g-code file is sent to the [Program to Output](#) specified in the [Text Editor Preferences](#) dialog.

By default the [Program to Output](#) is set to [notepad](#).

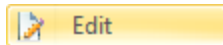
 **Related Topics**

[User Interface](#)

[Project Tab](#)[Load](#)[Info](#)[Merge Files](#)[Transform](#)[Z Instance](#)[XY Instance](#)[Save](#)

5.5 Right-Click Menu

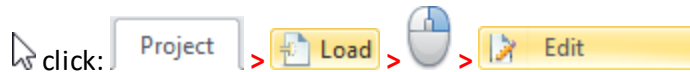
5.5.1 Edit-G-Code-File



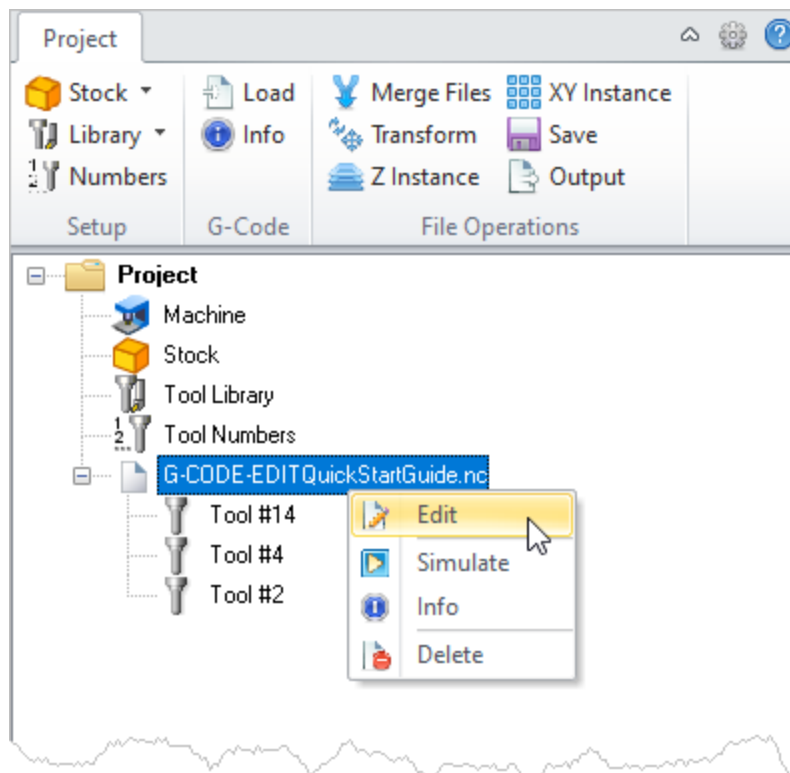
Use this command to [Edit](#) a G-Code file that is currently loaded into the [Project](#) tab of the [G-Code Browser](#).



To Access this Command:



Right-Click Menu



G-Code Browser, Project Tab



Procedure

1. Select a G-Code file that is currently loaded into the **Project** tab of the **G-Code Browser**.
2. Right-click and select **Edit** from the menu.
3. The G-Code file is loaded into the **Edit** tab of the **G-Code Browser**.



Related Topics

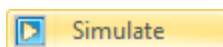
[Edit-G-Code-File](#)

[Simulate G-Code File](#)

[Info on G-Code File](#)

[Delete G-Code File](#)

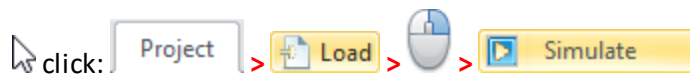
5.5.2 Simulate G-Code File



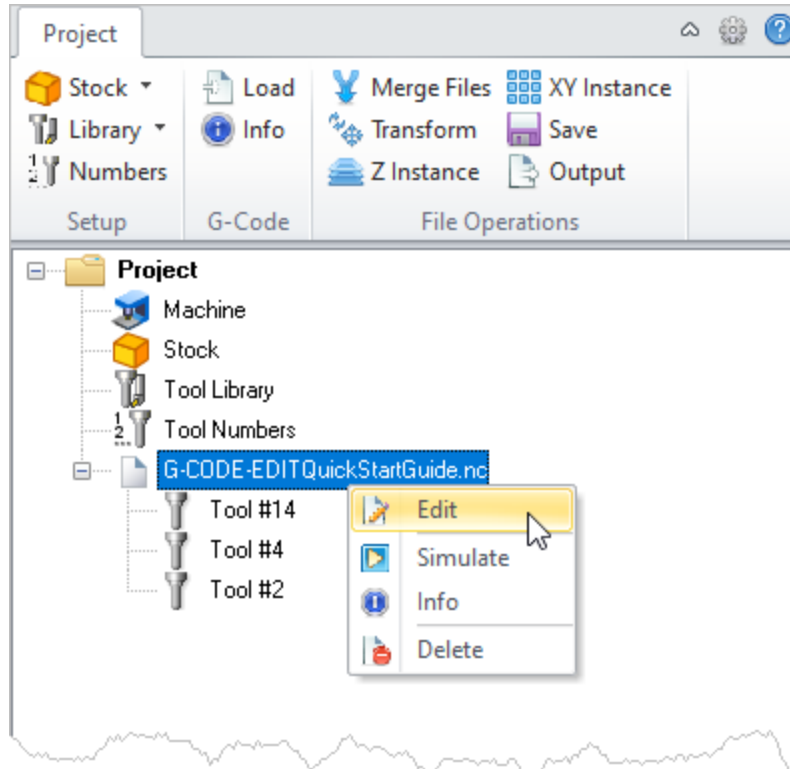
Use this command to **Simulate** a G-Code file that is currently loaded into the **Project** tab of the **G-Code Browser**.



To Access this Command:



Right-Click Menu



G-Code Browser, Project Tab



Procedure

1. Select a G-Code file that is currently loaded into the **Project** tab of the **G-Code Browser**.
2. Right-click and select **Simulate** from the menu.
3. The G-Code file is loaded into the **Simulate** tab of the **G-Code Browser**.



Related Topics

[Edit-G-Code-File](#)

[Simulate G-Code File](#)

[Info on G-Code File](#)

[Delete G-Code File](#)

5.5.3 Info on G-Code File

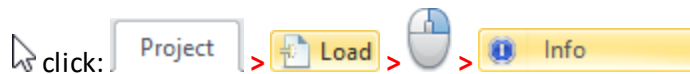


Info

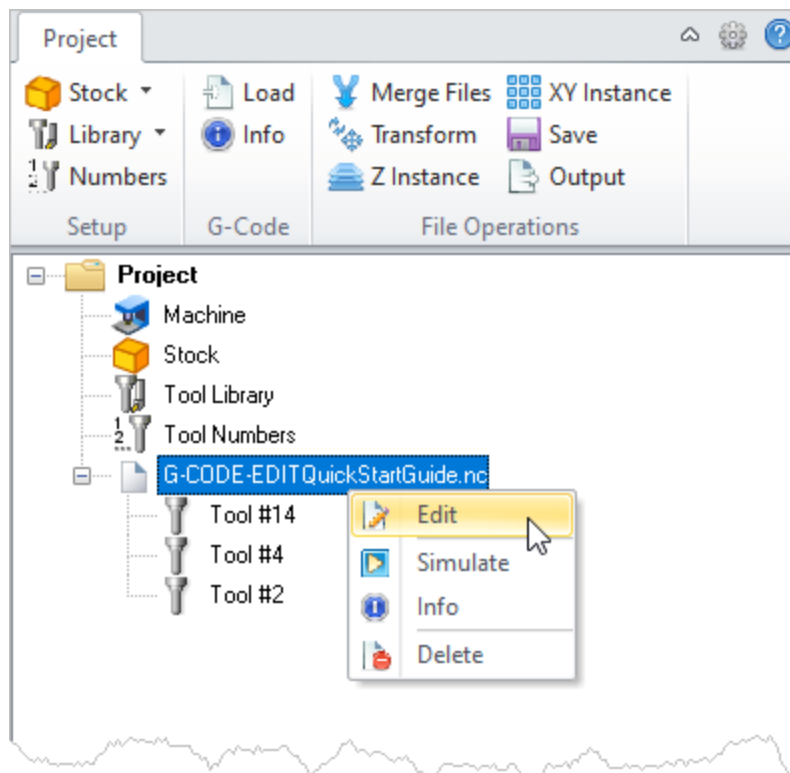
Use this command to display [Information](#) about a G-Code file that is currently loaded into the [Project](#) tab of the [G-Code Browser](#).



To Access this Command:



Right-Click Menu



G-Code Browser, Project Tab



Procedure

1. Select a G-Code file that is currently loaded into the [Project](#) tab of the [G-Code Browser](#).
2. Right-click and select [Info](#) from the menu.



Related Topics

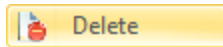
[Edit-G-Code-File](#)

[Simulate G-Code File](#)

[Info on G-Code File](#)

[Delete G-Code File](#)

5.5.4 Delete G-Code File

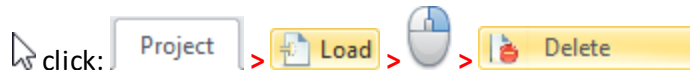


Delete

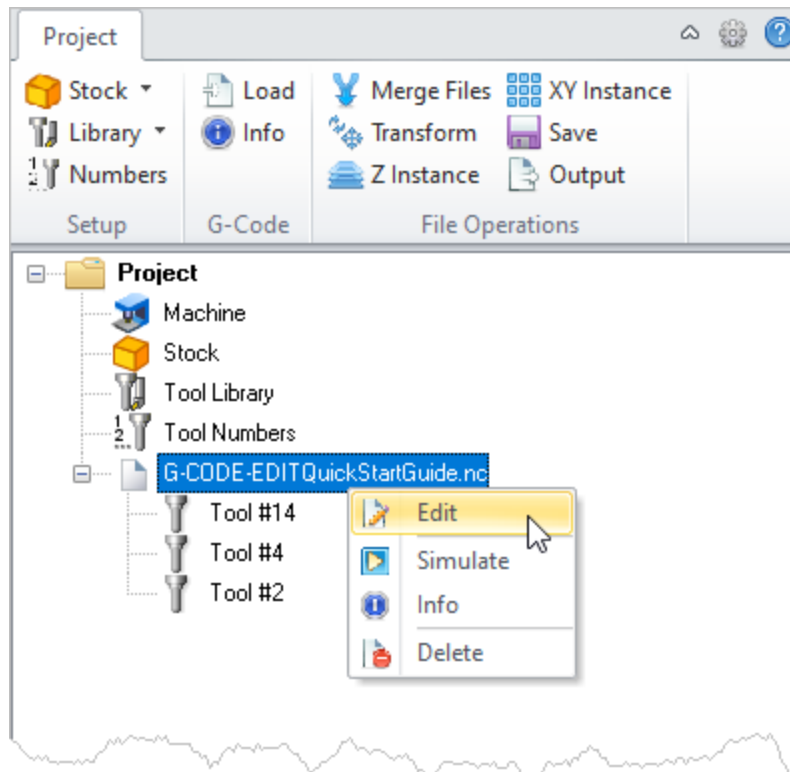
Use this command to [Delete](#) a G-Code file that is currently loaded into the [Project](#) tab of the [G-Code Browser](#).



To Access this Command:



Right-Click Menu



G-Code Browser, Project Tab



Procedure

1. Select a G-Code file that is currently loaded into the [Project](#) tab of the [G-Code Browser](#).
2. Right-click and select [Delete](#) from the menu.
3. The G-Code file is removed from the [Project](#) tab.



Related Topics

[Edit-G-Code-File](#)

[Simulate G-Code File](#)

[Info on G-Code File](#)

[Delete G-Code File](#)

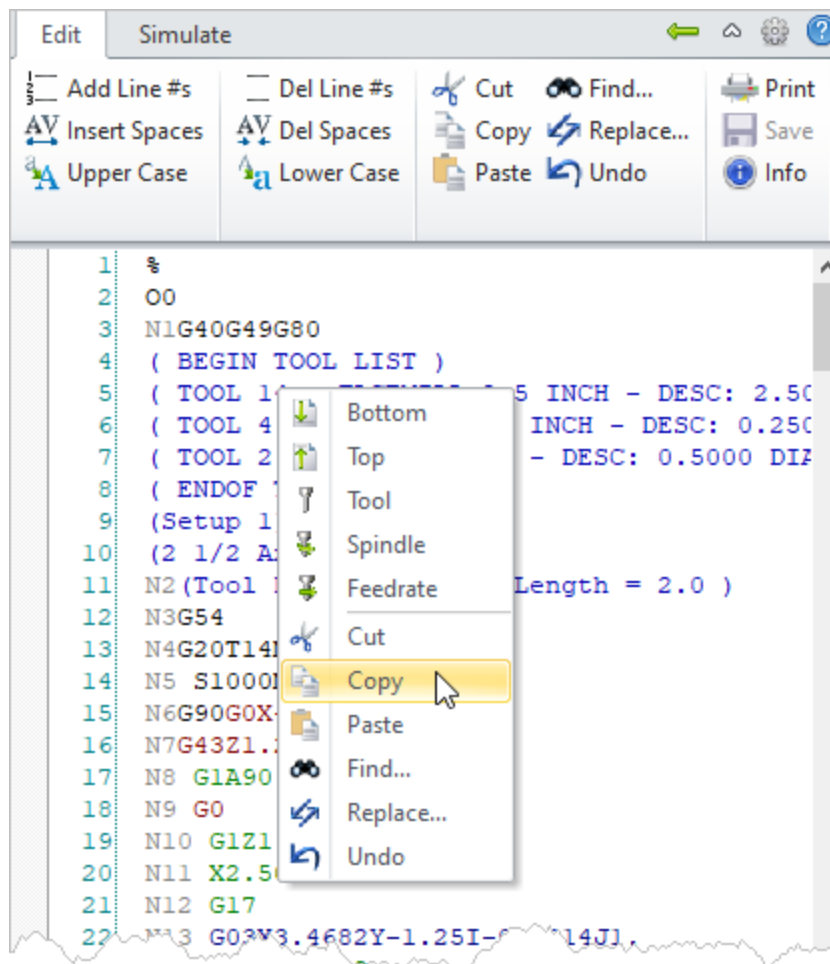
Edit Tab

Edit When you double-left-click on a G-Code file from the [Project](#) tree, that file is loaded into the [G-Code Editor](#). The [Edit](#) tab contains commands to [Modify](#) and [Navigate](#) the g-code file. The [Edit](#) tab and commands are listed below.

To Access this Command:

click: [Project](#) > [Load](#) > [Edit](#)

Edit tab



G-Code Browser, Edit Tab

Ribbon Bar Options

Edit Contains file [Edit](#) related commands.



Returns to the [Project](#) tab.



Minimizes and maximizes the ribbon bar menu.



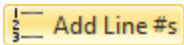
Displays the [Preferences](#) dialog.



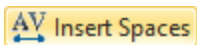
Displays the [Online Help](#) system.



Edit Tab



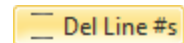
[Add line numbers](#) to the g-code file.



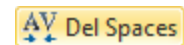
[Insert spaces](#) (i.e., delimiters) between g-codes.



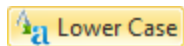
[Convert g-code to all UPPERCASE text.](#)



[Delete line numbers](#) from the g-code file.



[Delete spaces](#) (i.e., delimiters) between g-codes.



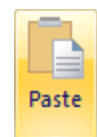
[Convert g-code to all lowercase text.](#)



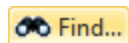
[Cut](#) the selected text to the clipboard.



[Copy](#) the selected text to the clipboard.



[Paste](#) the selected text to the current cursor location.



[Find text](#) within the G-Code file.



[Find & Replace](#) text within the G-Code file.



[Undo](#) the last command.



[Print](#) the G-Code file currently loaded in the editor.



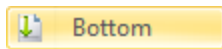
[Save](#) the G-Code file currently loaded in the editor.



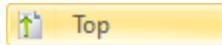
[Display information](#) about the current g-code file.



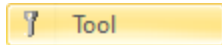
Edit Tab, Right-Click Menu



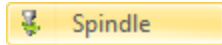
[Go to the Bottom](#) (End) of the G-Code file.



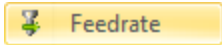
[Go to the Top](#) (Start) of the G-Code file.



[Go to the next Tool Change](#) code.



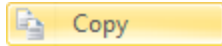
[Go to the next Spindle](#) code.



[Go to the next Feedrate](#) code.



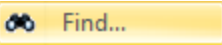
[Cut](#) the selected text to the clipboard.



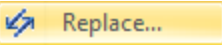
[Copy](#) the selected text to the clipboard.



[Paste](#) the selected text to the current cursor location.



[Find text](#) within the G-Code file.



[Find & Replace](#) text within the G-Code file.



[Undo](#) the last command.



Edit Tab, Display Toolbar



[Stock Model Visibility](#): Turn on/off stock model



Part Model Visibility: Turn on/off part model display during simulation.



Toolpath Visibility: Turn on/off toolpath display



Follow Toolpath Display: The toolpath is displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath after the tool passes.



Trace Toolpath Display: The toolpath is not displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath before the tool passes.



Segment Toolpath Display: The toolpath is only displayed for the segment that the tool is currently on.



Related Topics

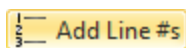
[Project Tab](#)

[Project Tree](#)

[Simulate Tab](#)

[Online Help](#)

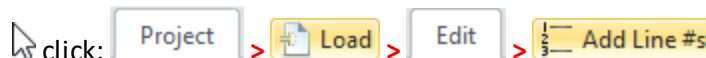
6.1 Add Line Numbers



Use this command to **Add Line Numbers** to the g-code file. Comments are not assigned line numbers. These line numbers are included when you save or output the g-code file. The line numbers you see to the far left are for reference within the **G-Code Editor** only and are not included in the g-code file.



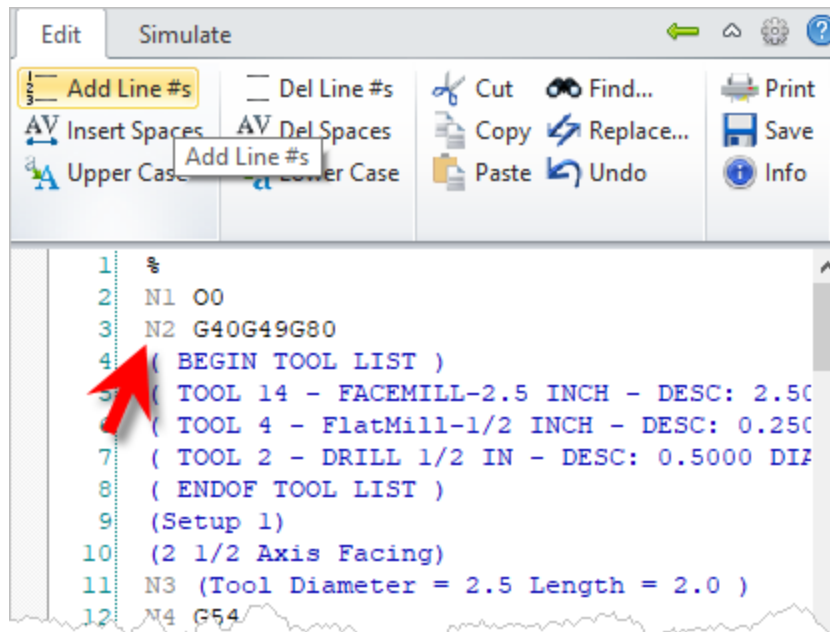
To Access this Command:



Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the **Project** tab.



Add Line # menu item



G-Code Browser: Edit G-Code tab, Add Line # menu item



Procedure

1. Select [Add Line #s](#) from the [Edit](#) tab. This will insert consecutive line numbers at the beginning of every line that contains g-code. Note that comments are not numbered.



Related Topics

[User Interface](#)

[G-Code Browser](#)

[G-Code Edit Tab](#)

[Delete Line Numbers](#)

[Insert Spaces](#)

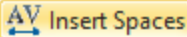
[Delete Spaces](#)

[Make Upper Case](#)

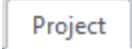


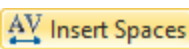
[Make Lower Case](#)

[Show G-Code Info](#)

6.2 Insert Spaces

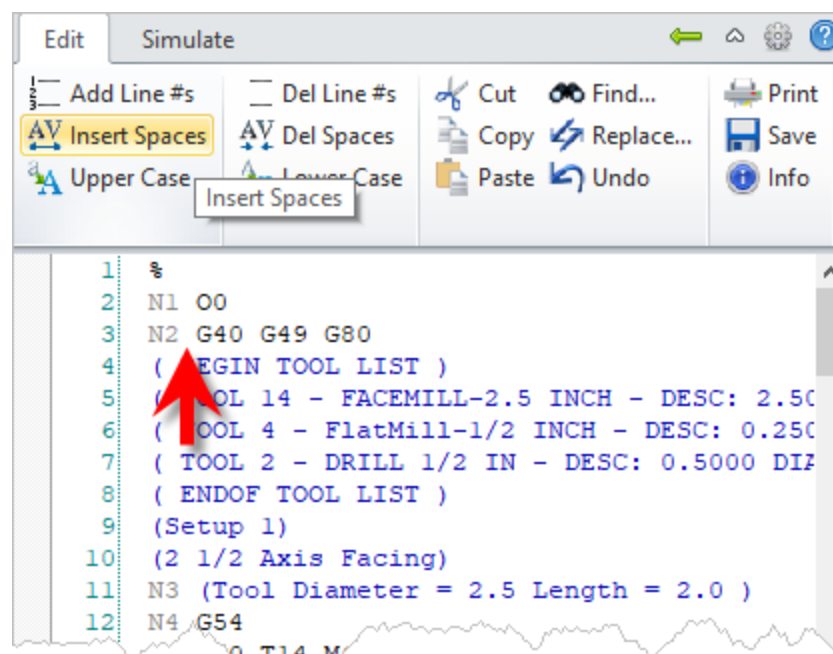
 Use this command to [Insert Spaces](#) between g-codes. These are also referred to as "[delimiters](#)". One space is inserted between each g-code. An example of inserted spaces is shown in the example below.

 **To Access this Command:**

click:  >  >  > 

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.

 **Insert Spaces menu item**



G-Code Browser: Edit G-Code tab, Insert Spaces menu item

 **Procedure**

1. Select [Insert Spaces](#) from the [Edit](#) tab. This will insert one a space character between all g-codes in the g-code file.

 **Related Topics**

[User Interface](#)

[G-Code Browser](#)

[G-Code Edit Tab](#)

[Add Line Numbers](#)

[Delete Line Numbers](#)

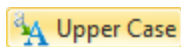
[Delete Spaces](#)

[Make Upper Case](#)

[Make Lower Case](#)

[Show G-Code Info](#)

6.3 Make Upper Case



Use this command to convert all alphabetic letters to **Upper Case** in the g-code file. An example of this is shown in the menu example below.



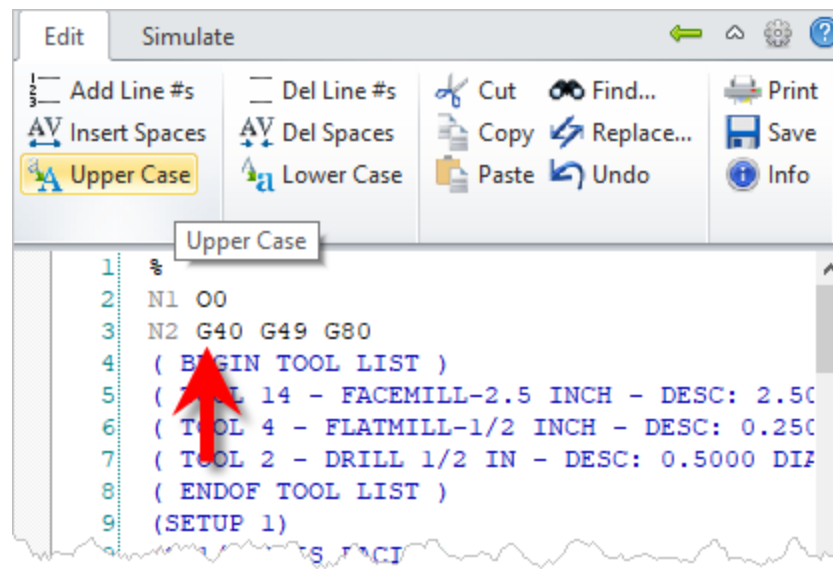
To Access this Command:

click: **Project** > **Load** > **Edit** > **Upper Case**

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the **Project** tab.



Upper Case menu item



G-Code Browser: Edit G-Code tab, Upper Case menu item



Procedure

1. Select **Upper Case** from the **Edit** tab. This will convert all alphabetic letters to **Upper Case** characters in the g-code file.



Related Topics

[User Interface](#)

[G-Code Browser](#)

[G-Code Edit Tab](#)

[Add Line Numbers](#)

[Delete Line Numbers](#)

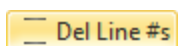
[Insert Spaces](#)

[Delete Spaces](#)

[Make Lower Case](#)

[Show G-Code Info](#)

6.4 Delete Line Numbers



Use this command to [Delete Line Numbers](#) in the g-code file. Comments are not assigned line numbers. Line numbers, if displayed, are included when you save or output the g-code file. The line numbers you see to the far left are for reference within the [G-Code Editor](#) only and are not included in the g-code file.



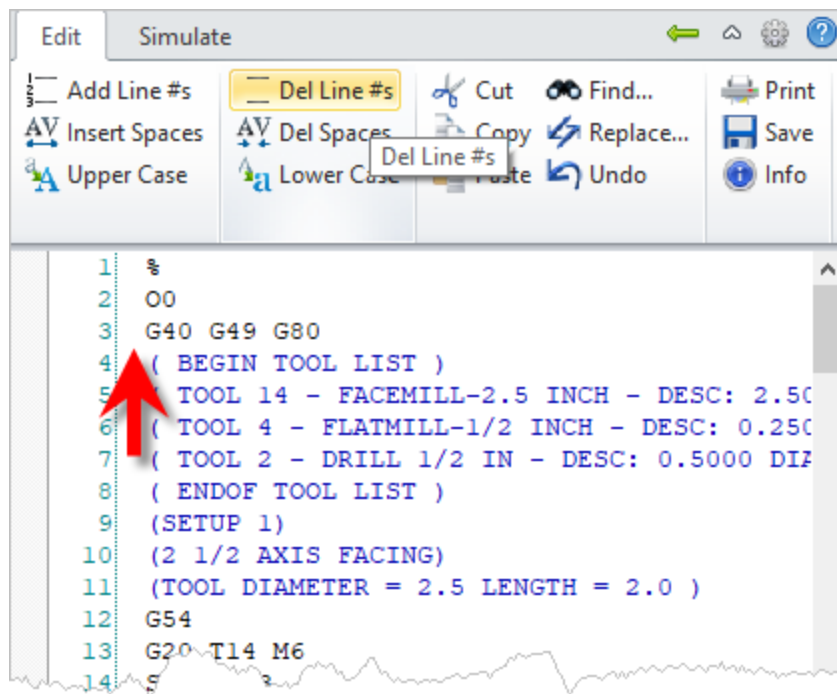
To Access this Command:

click:  >  >  > 

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



Delete Line # menu item



G-Code Browser: Edit G-Code tab, Delete Line # menu item



Procedure

1. Select **Del Line #s** from the **Edit** tab. This will remove all consecutive line numbers from the beginning of every line that contains g-code. Note that comments are not numbered.



Related Topics

[User Interface](#)

[G-Code Browser](#)

[G-Code Edit Tab](#)

[Add Line Numbers](#)

[Insert Spaces](#)

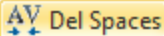
[Delete Spaces](#)

[Make Upper Case](#)

[Make Lower Case](#)

[Show G-Code Info](#)

6.5 Delete Spaces

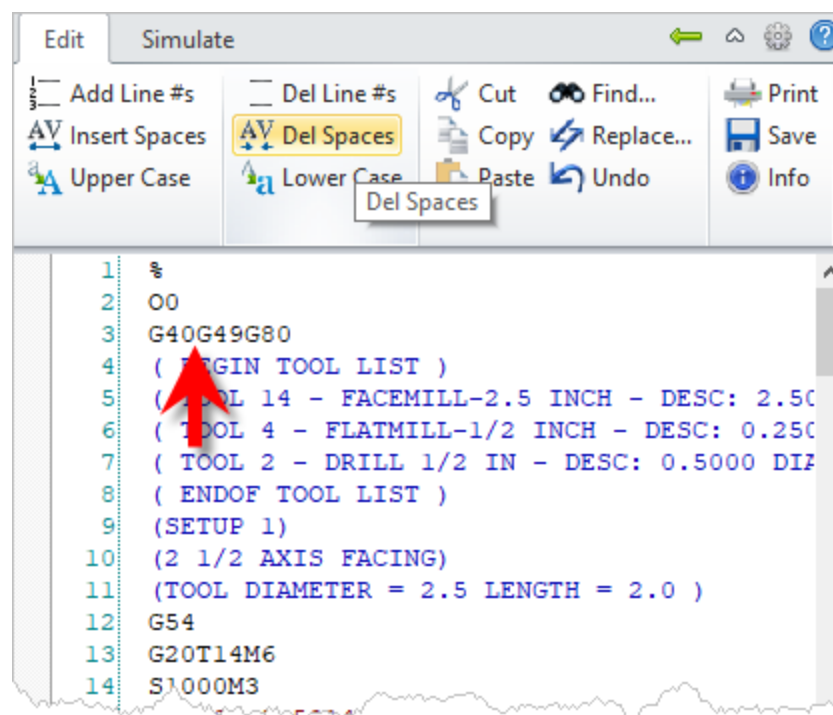
 Use this command to [Delete Spaces](#) between g-codes. These are also referred to as "delimiters". One space is deleted between each g-code. An example of deleted spaces is shown in the example below.

To Access this Command:

click:  >  >  > 

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.

Delete Spaces menu item



G-Code Browser: Edit G-Code tab, Delete Spaces menu item

Procedure

1. Select [Del Spaces](#) from the [Edit](#) tab. This will remove all space characters between all g-codes in the g-code file.

Related Topics

[User Interface](#)

[G-Code Browser](#)

[G-Code Edit Tab](#)

[Add Line Numbers](#)

[Delete Line Numbers](#)

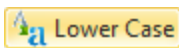
[Insert Spaces](#)

[Make Upper Case](#)

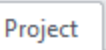

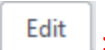
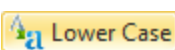
[Make Lower Case](#)

[Show G-Code Info](#)

6.6 Make Lower Case

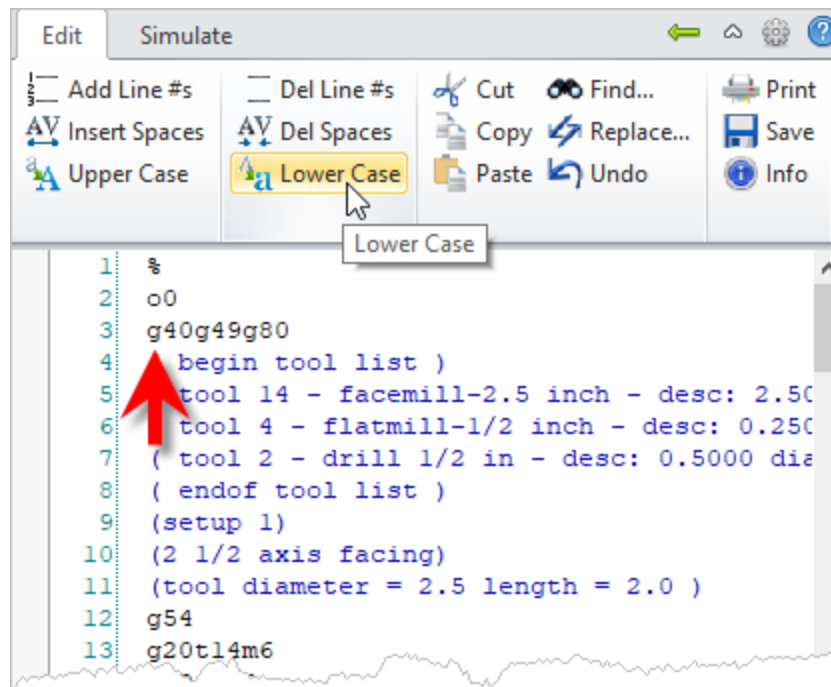
 Use this command to convert all alphabetic letters to **Lower Case** in the g-code file. An example of this is shown in the menu example below.

 **To Access this Command:**

click:  >  >  > 

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the **Project** tab.

 **Lower Case menu item**



G-Code Browser: Edit G-Code tab, Lower Case menu item



Procedure

1. Select [Lower Case](#) from the [Edit](#) tab. This will convert all alphabetic letters to [Lower Case](#) characters in the g-code file.



Related Topics

[User Interface](#)

[G-Code Browser](#)

[G-Code Edit Tab](#)

[Add Line Numbers](#)

[Delete Line Numbers](#)

[Insert Spaces](#)

[Delete Spaces](#)

[Make Upper Case](#)

[Show G-Code Info](#)

6.7 Cut Text



Use this edit tool to [Cut](#) (i.e., delete) the currently highlighted text. The cut text is added to your [Windows](#) clipboard and can be [Pasted](#) back into any g-code file within the [G-Code Editor](#) or other [Windows](#) program that can display text.



To Access this Command:

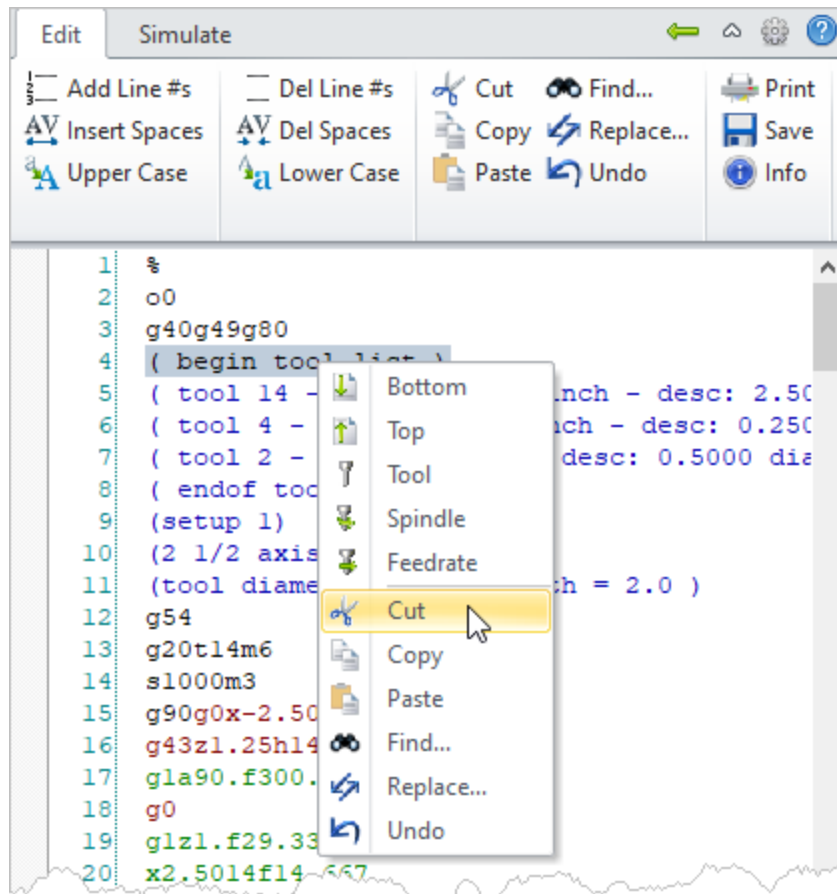
click: > > > >

or

click: > > >



Cut Text (Right-Click Menu)



G-Code Browser: Edit tab, Cut Text (Right-Click Menu)



Procedure

1. Use the mouse cursor left-click-drag to highlight the text you wish to remove.
2. Select **Cut** from the **Edit** tab or right-click and select **Cut** from the menu and the text is removed and added to the **Windows** clipboard.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Copy Text](#)


[Paste Text](#)

[Undo Last Edit](#)

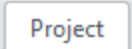


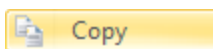
[Print G-Code](#)

[Save G-Code](#)

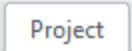



6.8 Copy Text

 Use this edit tool to [Copy](#) the currently highlighted text. The copied text is added to your [Windows](#) clipboard and can be [Pasted](#) back into any g-code file within the [G-Code Editor](#) or other [Windows](#) program that can display text.

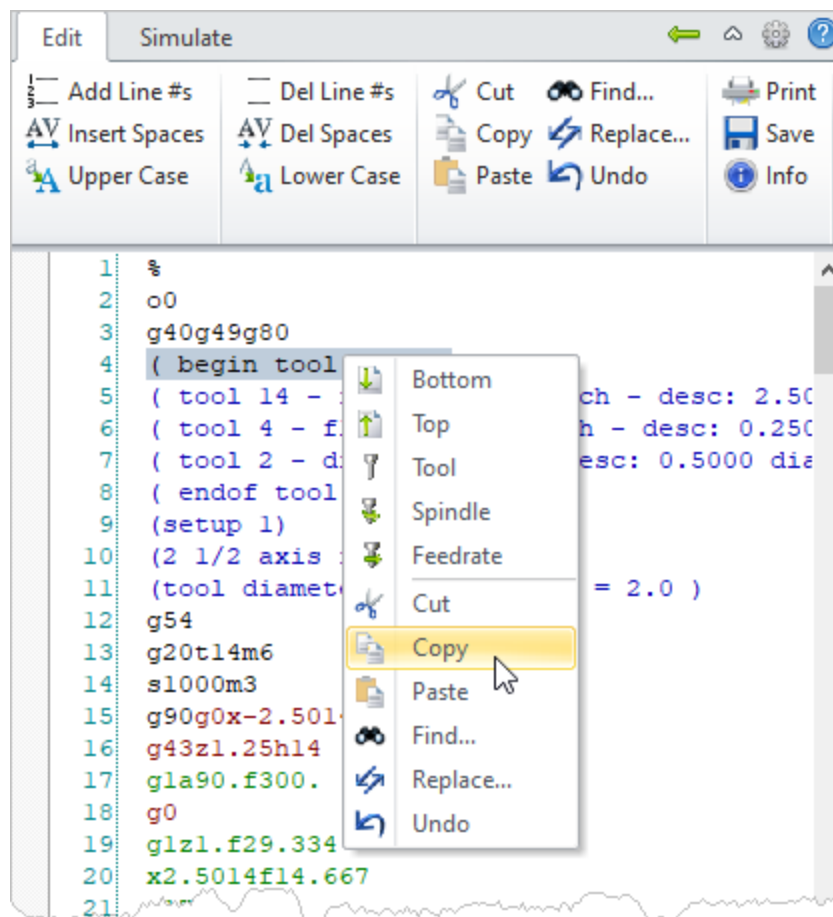
 **To Access this Command:**

click:  >  >  > 

or

click:  >  >  > 

 **Copy Text (Right-Click Menu)**



G-Code Browser: Edit tab, Copy Text (Right-Click Menu)



Procedure

1. Use the mouse cursor left-click-drag to highlight the text you wish to copy.
2. Select **Copy** from the **Edit** tab or right-click and select **Copy** from the menu and the text is copied to the **Windows** clipboard.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Paste Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

[Save G-Code](#)

6.9 Paste Text



Use this edit tool to **Paste** the contents of the **Windows** clipboard into the g-code file at the location of the cursor.



To Access this Command:

click: **Project** > **Load** > **Edit** > **Paste**

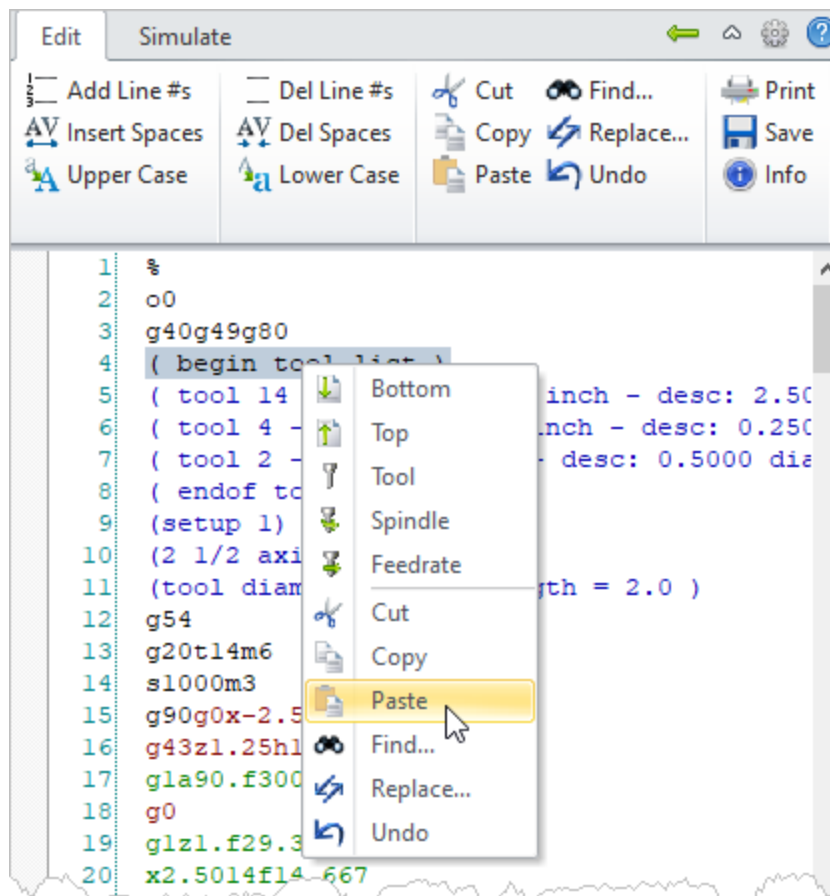
or

click: **Project** > **Load** > **Edit** > **Paste**

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file.



Paste Text (Right-Click Menu)



G-Code Browser: Edit tab, Paste Text (Right-Click Menu)



Procedure

1. Use the mouse to locate the cursor in the desired location.
2. Select **Paste** from the **Edit** tab or right-click and select **Paste** from the menu and the text is inserted into the G-Code file at the cursor location.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Copy Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

[Save G-Code](#)

6.10 Find Text



Use this dialog to [Find](#) text within your gcode file. You can use additional options in the [Find](#) dialog shown below.



To Access this Command:

click: > > >

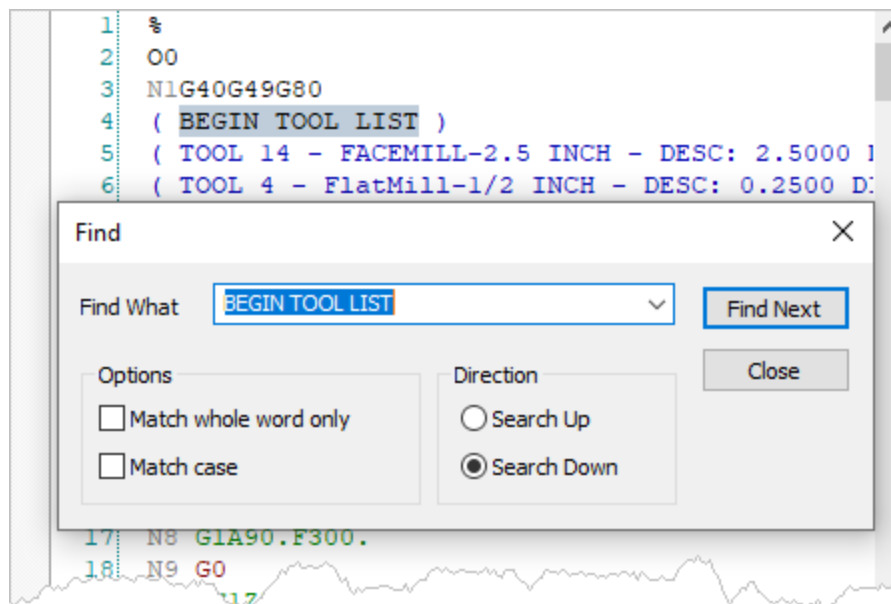
or

click: > > > >

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file.



The Find dialog



Use this dialog to Find text within your gcode file.

Find What

Enter the text you wish to find and then pick Find Next to locate it.

Find Next

Go to the next location of the text.

Options

- [Match whole word only](#): Only complete words will be located.

- **Match case: Use case sensitivity:** Only words that match the upper/lower case will be located.

Direction

- **Search Up:** Search upwards in the g-code file.
- **Search Down:** Search downwards in the g-code file.



Procedure

1. Select **Find** from the **Edit** tab or right-click and select **Find** from the menu. The **Find** dialog will display.
2. Enter the text to find and then pick **Find** Next.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Copy Text](#)

[Paste Text](#)

[Find & Replace Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

[Save G-Code](#)

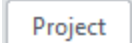



6.11 Find & Replace Text



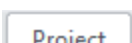



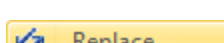
Use this dialog to **Find & Replace** text within your gcode file. You can find and replace all occurrences of text and use additional options in the Find & Replace dialog shown below.



To Access this Command:

click:  >  >  > 

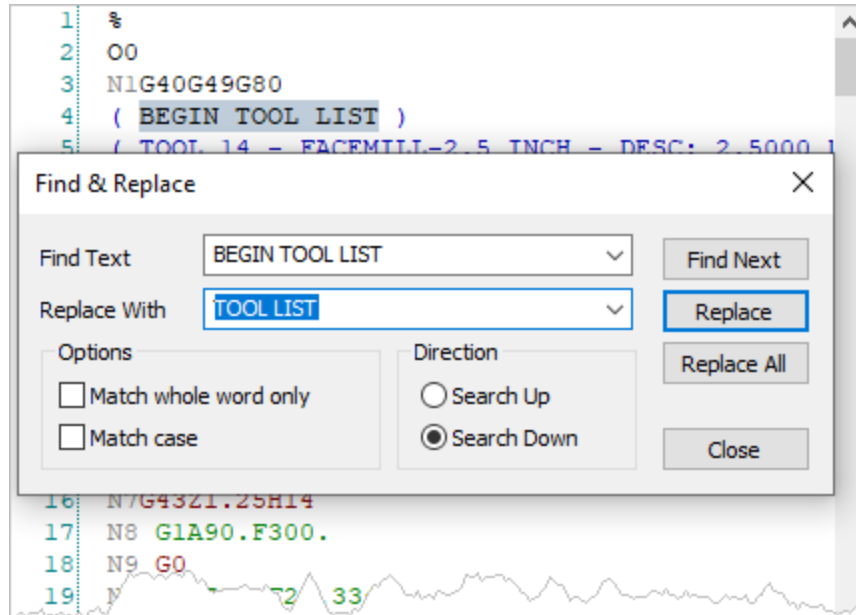
or

click:  >  >  >  > 

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file.



The Find & Replace dialog



Use this dialog to Find & Replace text within your gcode file.

Find Text

Enter the text you wish to find and then pick Find Next to locate it.

Find Next

Go to the next location of the text.

Replace With

Enter the text you wish to replace with.

Replace

Replace the currently highlighted text with the text entered into the [Replace With](#) field.

Replace All

Search the entire g-code file and replace all occurrences of the [Find Text](#) with the text entered into the [Replace With](#) field.

Options

- [Match whole word only](#): Only complete words will be located.
- [Match case: Use case sensitivity](#): Only words that match the upper/lower case will be located.

Direction

- [Search Up](#): Search upwards in the g-code file.
- [Search Down](#): Search downwards in the g-code file.



Procedure

1. Select **Find & Replace** from the **Edit** tab or right-click and select **Replace** from the menu. The **Find & Replace** dialog will display.
2. Use the **Find & Replace** dialog shown above to complete the process.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Copy Text](#)

[Paste Text](#)

[Find Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

[Save G-Code](#)


6.12 Undo Last Edit




Use this command to **Undo** the last edit made to the g-code file. You can repeat the selection to **Undo** all edits made in the current session.



To Access this Command:

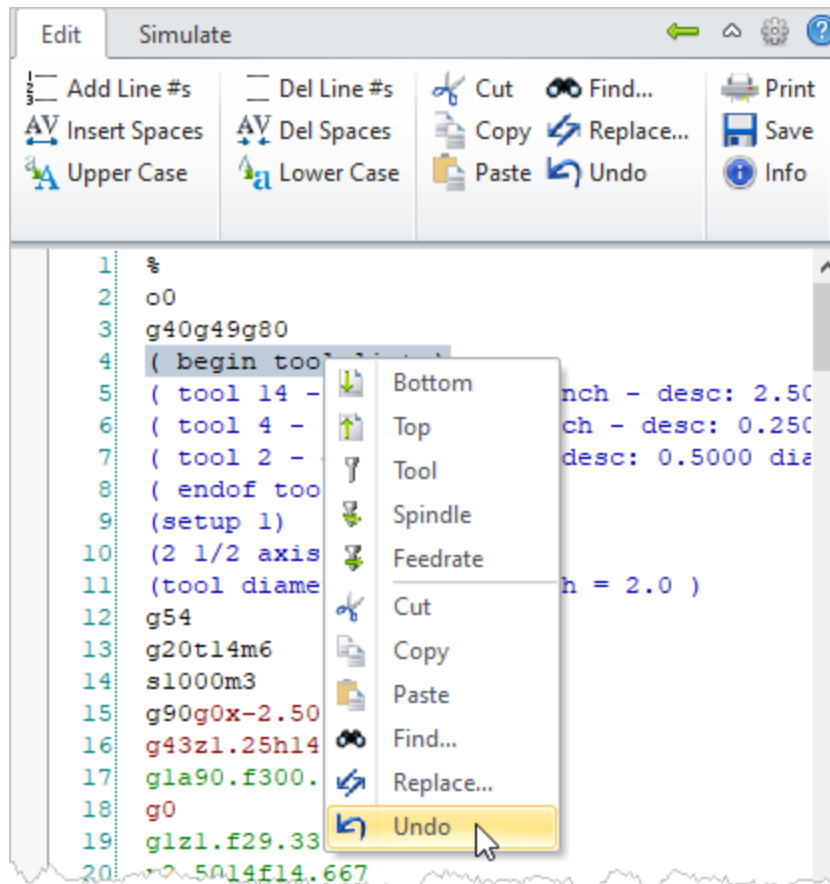
click: **Project** > **Load** > **Edit** >  > **Undo**

or

click: **Project** > **Load** > **Edit** > 



Undo Last Edit (Right-Click Menu)



G-Code Browser: Edit tab, Undo Last Edit (Right-Click Menu)



Procedure

1. Select **Undo** from the **Edit** tab or right-click and select **Undo** from the menu. The last edit will be undone.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Copy Text](#)

[Paste Text](#)

[Find Text](#)

[Find & Replace Text](#)

[Print G-Code](#)

[Save G-Code](#)

6.13 Print G-Code



Use this command to [Print](#) the current g-code file. The default [Windows Print](#) dialog will display allowing you to select a printing device and print preferences.



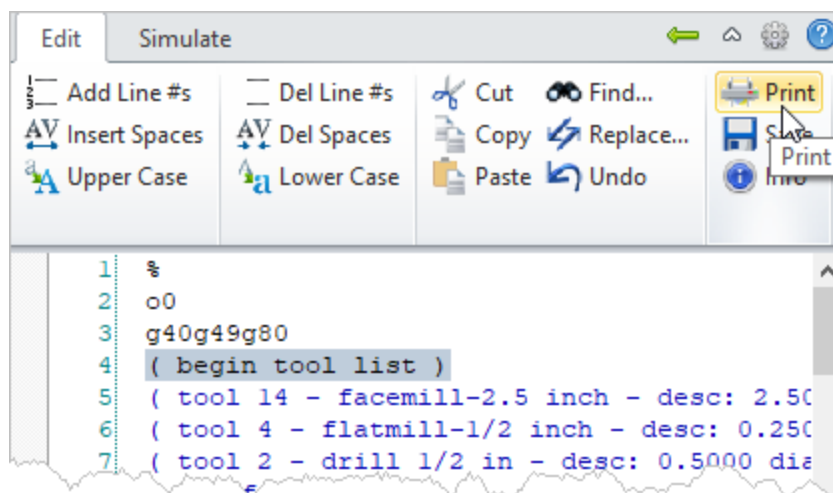
To Access this Command:

click: > > >

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



Print menu item



G-Code Browser: Edit tab, Print menu item



Procedure

1. Select [Print](#) from the [Edit](#) tab. This will display the [Windows Print](#) dialog allowing you to select a printer or device to print to.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

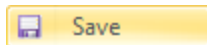
[Copy Text](#)

[Paste Text](#)

[Undo Last Edit](#)

[Save G-Code](#)

6.14 Save G-Code

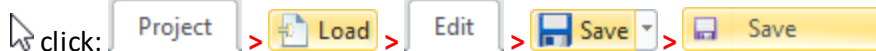


Save

Select this command to [Save](#) the g-code file to the default [Save Directory](#) defined in the [Text Editor Preferences](#) dialog. Your original g-code file IS NOT overwritten.



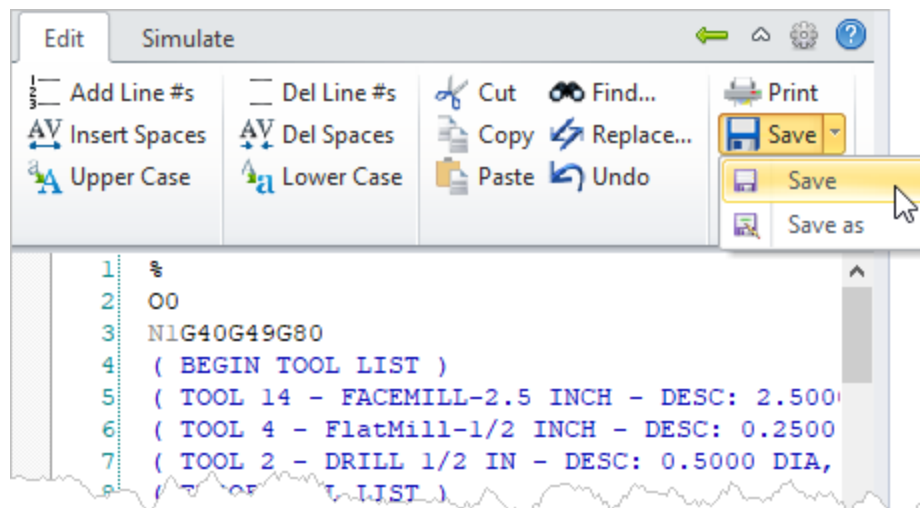
To Access this Command:



Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



Save menu item



G-Code Browser: Edit tab, Save menu item



Procedure

1. Select [Save](#) from the [Edit](#) tab. This will save the g-code file to the default [Save Directory](#) defined in the [Text Editor Preferences](#) dialog. Your original g-code file IS NOT overwritten.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

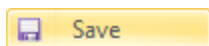
[Copy Text](#)

[Paste Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

6.15 Save As



Save

Select this command to save the g-code file as another name. The Save As dialog will display allowing you to select the save as folder and the name of the saved file.



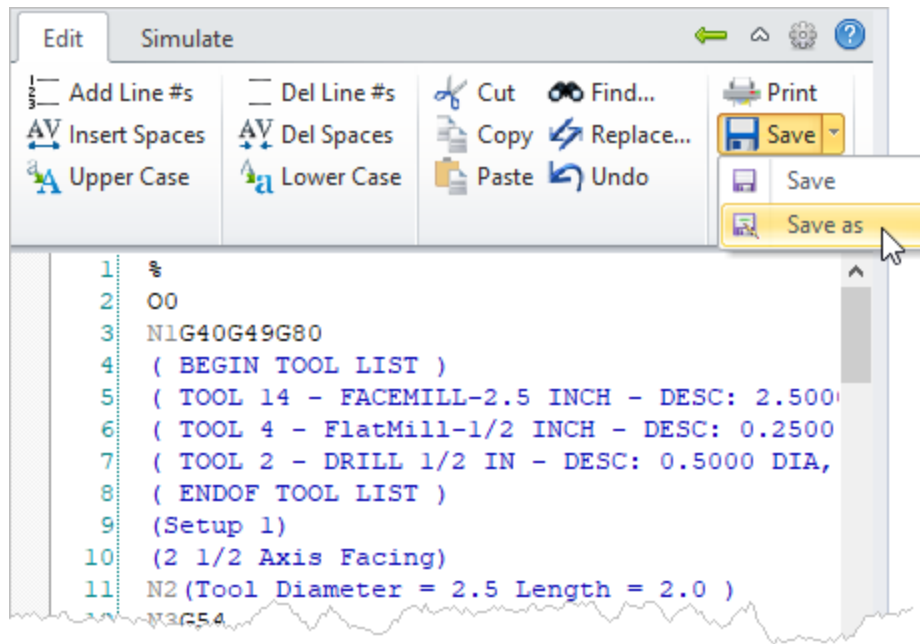
To Access this Command:

click: Project > Load > Edit > Save > Save as

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



Save menu item



G-Code Browser: Edit tab, Save As menu item



Procedure

1. Select [Save](#) from the [Edit](#) tab and then Save As. This will save the g-code file to the folder of your choice.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Copy Text](#)

[Paste Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

6.16 Show G-Code Info



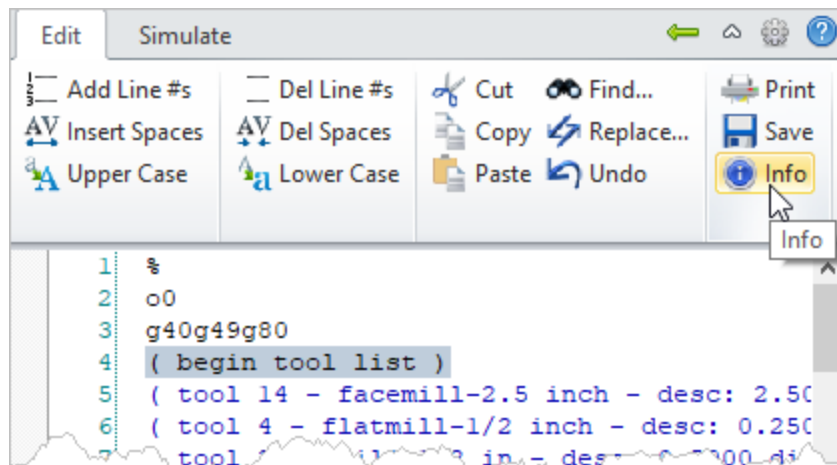
Use this command to display an information window of the current contents of the G-Code Editor.

To Access this Command:

click: > > >

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.

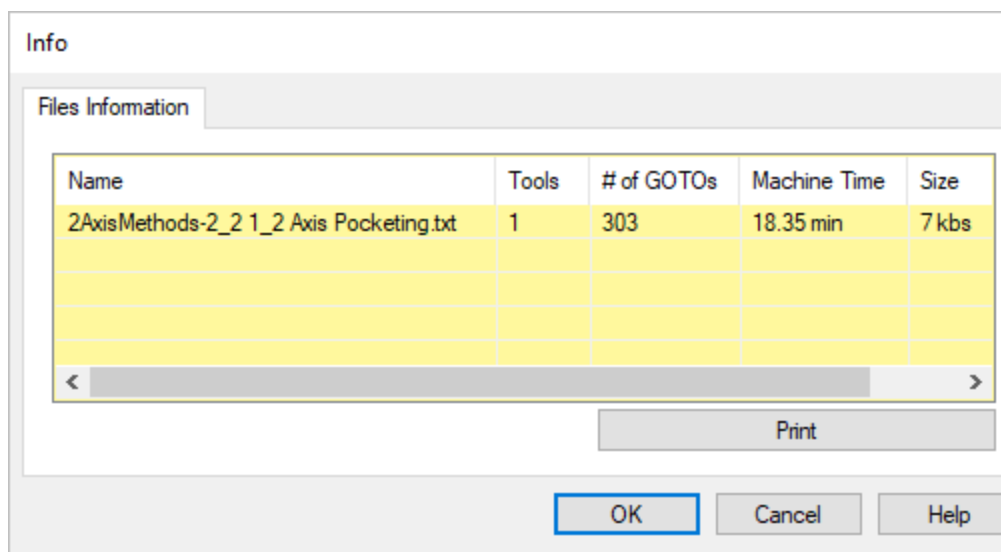
Info menu item



G-Code Browser: Edit G-Code tab, Info menu item

Procedure

1. Select [Info](#) from the [Edit](#) tab.
2. This will display the Information dialog containing information related to the current g-code file.



Information dialog



Related Topics

[User Interface](#)

[G-Code Browser](#)

[G-Code Edit Tab](#)

[Add Line Numbers](#)

[Delete Line Numbers](#)

[Insert Spaces](#)

[Delete Spaces](#)

[Make Upper Case](#)

[Make Lower Case](#)

6.17 Select Full Line

You can select the full line of code. To do so, first make sure Line Numbers are displayed by selecting [Add line #s](#) from the [Edit](#) tab menu. Then select the line number displayed in the line number column. This will select the full line of code. Refer to the example below.



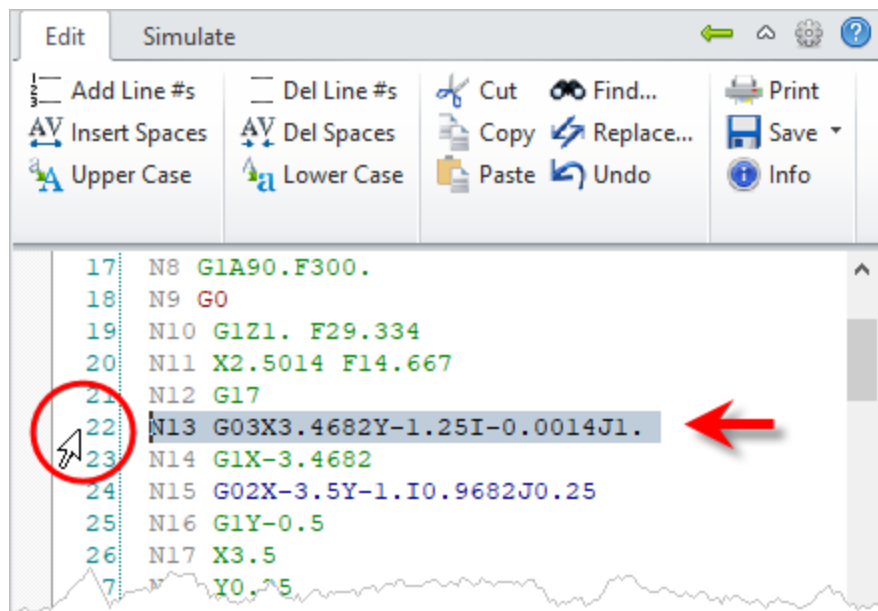
To Access this Command:



Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



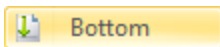
Info menu item



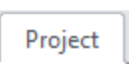
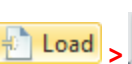

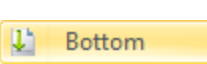
G-Code Browser: Edit G-Code tab, Select Whole Line

6.18 Right-Click Menu

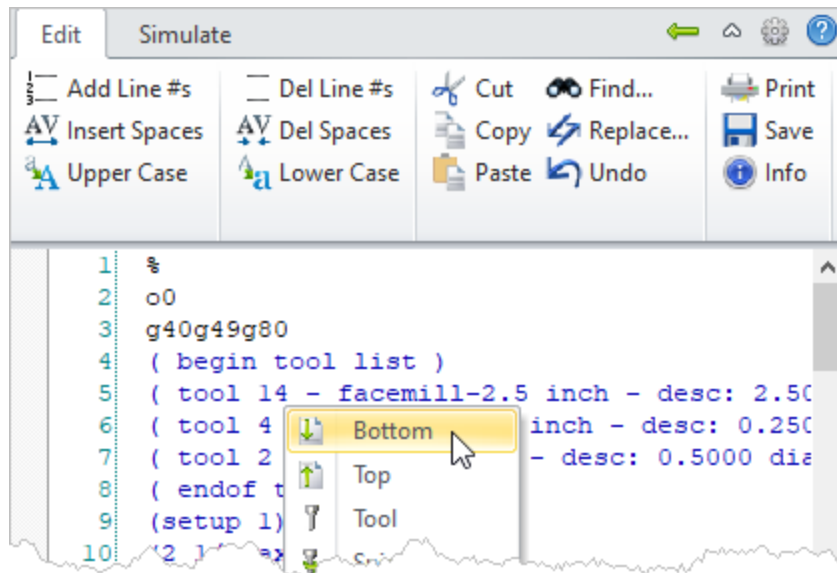
6.18.1 Go to Bottom

 **Bottom** Use this command to move the cursor to the last g-code numbered line in the g-code file.

 **To Access this Command:**

click:  >  >  > 

 **Bottom (Right-Click Menu)**



G-Code Browser: Edit tab, Bottom (Right-Click Menu)



Procedure

1. Select **Bottom** from the **Edit** tab. This will move the cursor to the last g-code numbered line in the g-code file.



Related Topics

[Go To Bottom](#)

[Go to Top](#)

[Go to Next Tool](#)

[Go to Next Spindle](#)

[Go to Next Feedrate](#)

[Cut Text](#)

[Copy Text](#)

[Paste Text](#)

[Undo Last Edit](#)

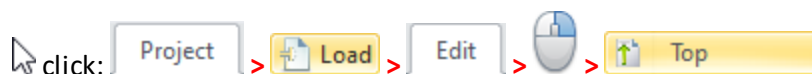
6.18.2 Go to Top



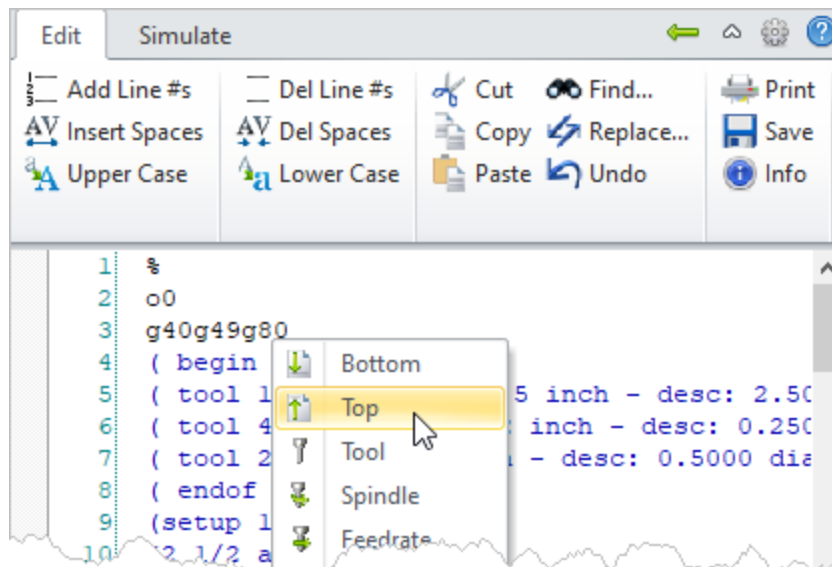
Use this command to move the cursor to the first line in the g-code file.



To Access this Command:



Top (Right-Click Menu)



G-Code Browser: Edit tab, Top (Right-Click Menu)

Procedure

1. Select **Top** from the **Edit** tab. This will move the cursor to the first line in the g-code file.

Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

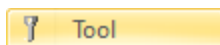
[Go to Bottom](#)

[Go to Next Tool](#)

[Go to Next Spindle](#)

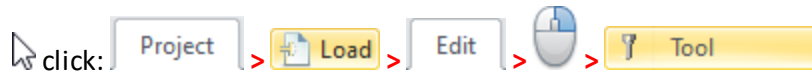
[Go to Next Feedrate](#)

6.18.3 Go to Next Tool

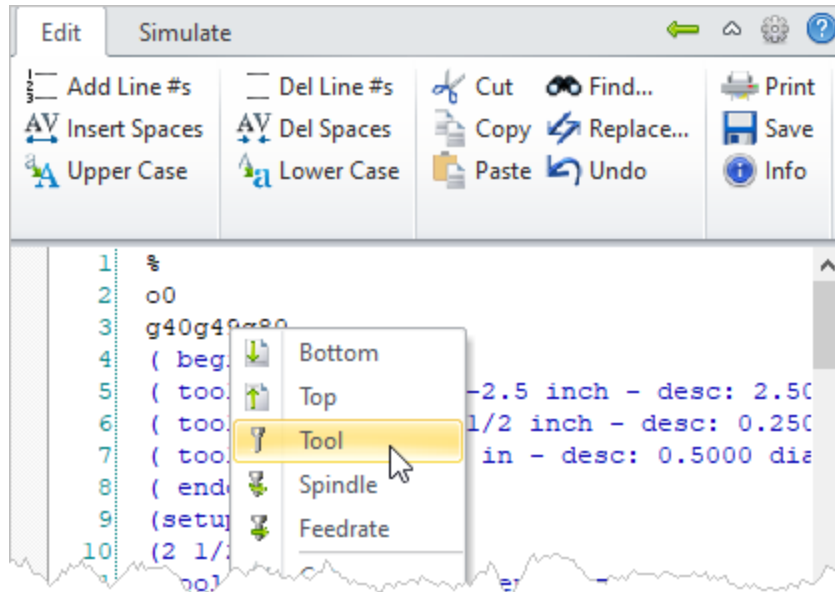


Use this command to move the cursor to the next line in the g-code file that contains a tool number.

To Access this Command:



Next Tool (Right-Click Menu)



G-Code Browser: Edit tab, Next Tool (Right-Click Menu)

Procedure

1. Select **Tool** from the **Edit** tab. This will move the cursor to the next line in the g-code file that contains a tool number.

Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

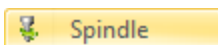
[Go to Bottom](#)

[Go to Top](#)

[Go to Next Spindle](#)

[Go to Next Feedrate](#)

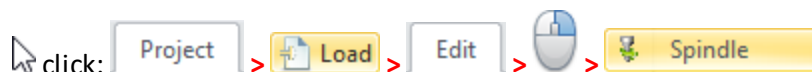
6.18.4 Go to Next Spindle

**Spindle**

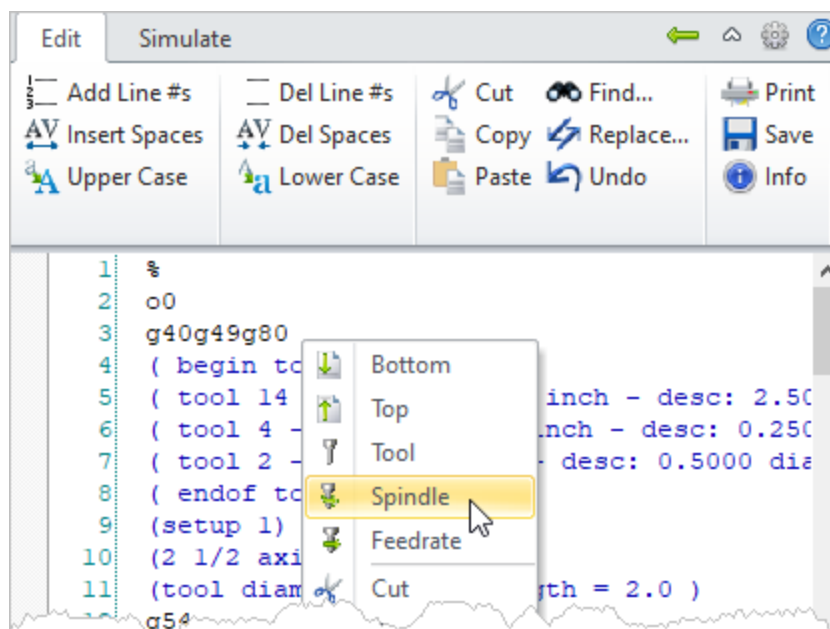
Use this command to move the cursor to the next line in the g-code file that contains a **Spindle** code.



To Access this Command:



Next Spindle (Right-Click Menu)



G-Code Browser: Edit tab, Next Spindle (Right-Click Menu)



Procedure

1. Select **Spindle** from the **Edit** tab. This will move the cursor to the next line in the g-code file that contains a **Spindle** code.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)


[Go to Bottom](#)

[Go to Top](#)

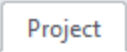

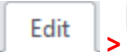
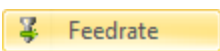
[Go to Next Tool](#)

[Go to Next Feedrate](#)

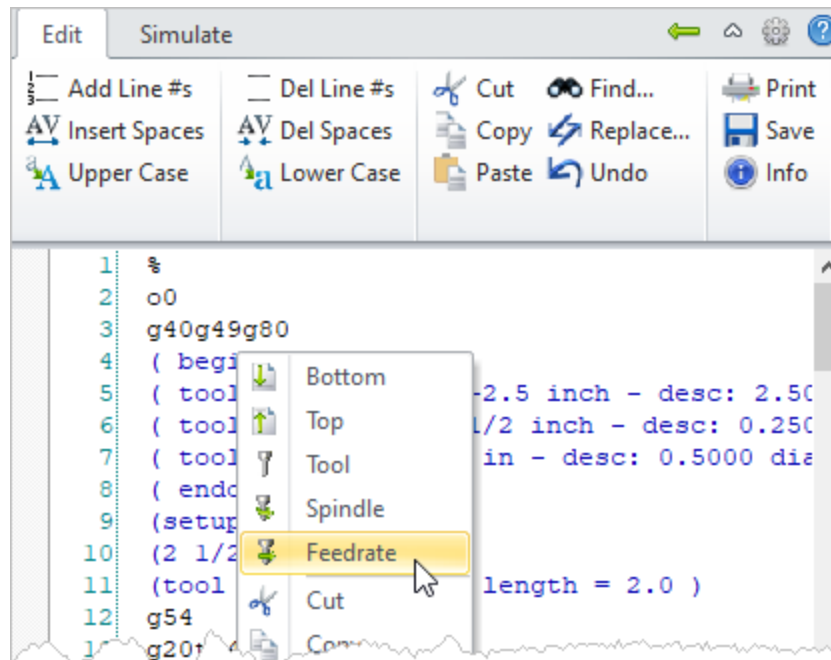
6.18.5 Go to Next Feedrate

 **Feedrate** Use this command to move the cursor to the next line in the g-code file that contains a **Feedrate** code.

 **To Access this Command:**

click:  >  >  > 

 **Next Feedrate (Right-Click Menu)**



G-Code Browser: Edit tab, Next Feedrate (Right-Click Menu)

 **Procedure**

1. Select **Spindle** from the **Edit** tab. This will move the cursor to the next line in the g-code file that contains a **Feedrate** code.

 **Related Topics**

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Go to Bottom](#)

[Go to Top](#)

[Go to Next Tool](#)

[Go to Next Spindle](#)

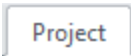
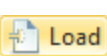


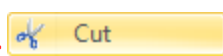
6.18.6 Cut Text



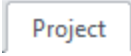



Use this edit tool to **Cut** (i.e., delete) the currently highlighted text. The cut text is added to your **Windows** clipboard and can be **Pasted** back into any g-code file within the **G-Code Editor** or other **Windows** program that can display text.



To Access this Command:

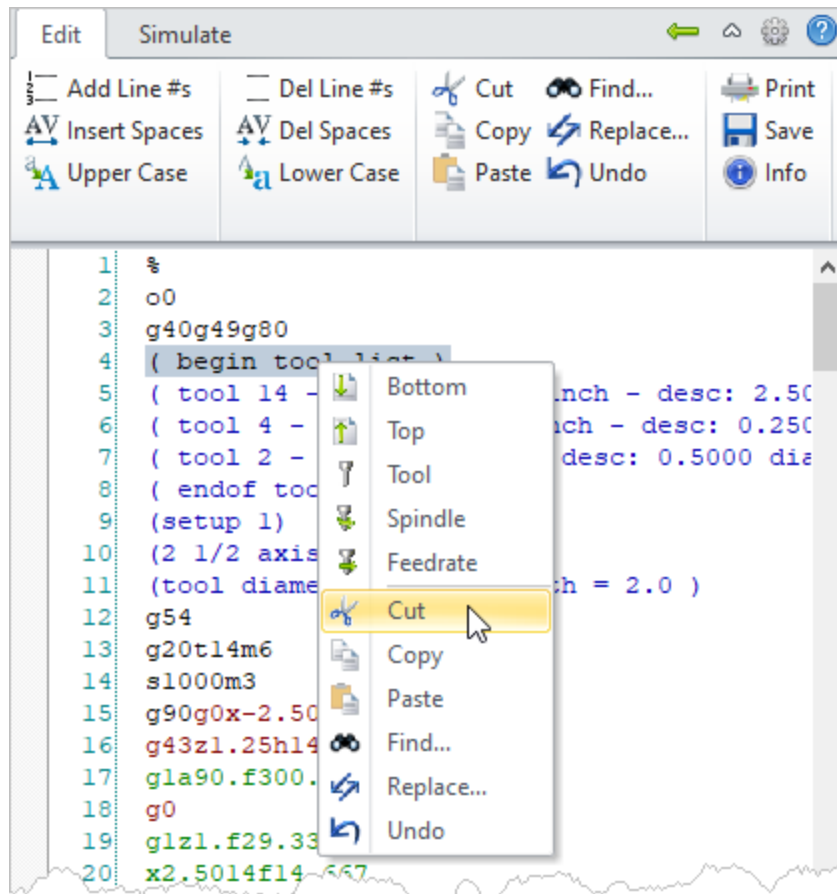
click:  >  >  >  > 

or

click:  >  >  > 



Cut Text (Right-Click Menu)



G-Code Browser: Edit tab, Cut Text (Right-Click Menu)



Procedure

1. Use the mouse cursor left-click-drag to highlight the text you wish to remove.
2. Select **Cut** from the **Edit** tab or right-click and select **Cut** from the menu and the text is removed and added to the **Windows** clipboard.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Copy Text](#)

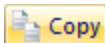
[Paste Text](#)

[Undo Last Edit](#)


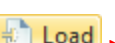

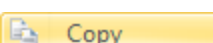
[Print G-Code](#)

[Save G-Code](#)

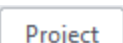
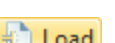


6.18.7 Copy Text

 Use this edit tool to [Copy](#) the currently highlighted text. The copied text is added to your [Windows](#) clipboard and can be [Pasted](#) back into any g-code file within the [G-Code Editor](#) or other [Windows](#) program that can display text.

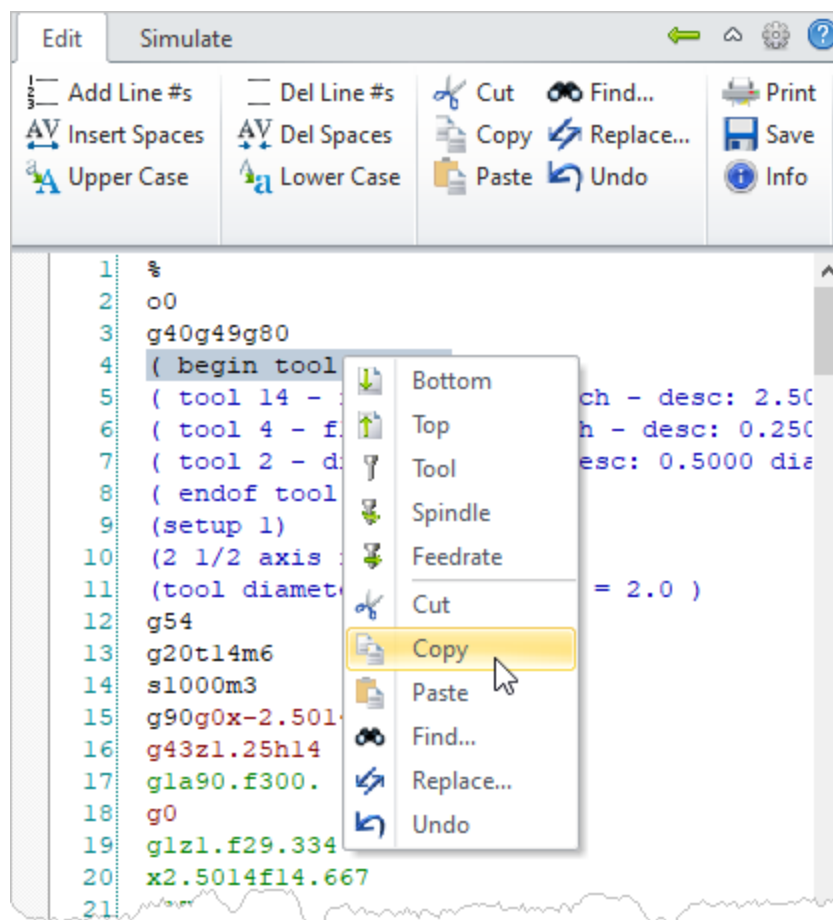
 **To Access this Command:**

click:  >  >  > 

or

click:  >  >  > 

 **Copy Text (Right-Click Menu)**



G-Code Browser: Edit tab, Copy Text (Right-Click Menu)



Procedure

1. Use the mouse cursor left-click-drag to highlight the text you wish to copy.
2. Select **Copy** from the **Edit** tab or right-click and select **Copy** from the menu and the text is copied to the **Windows** clipboard.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Paste Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

[Save G-Code](#)

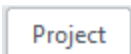
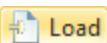

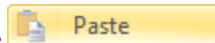
6.18.8 Paste Text



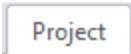
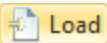


Use this edit tool to **Paste** the contents of the **Windows** clipboard into the g-code file at the location of the cursor.



To Access this Command:

click:  >  >  >  > 

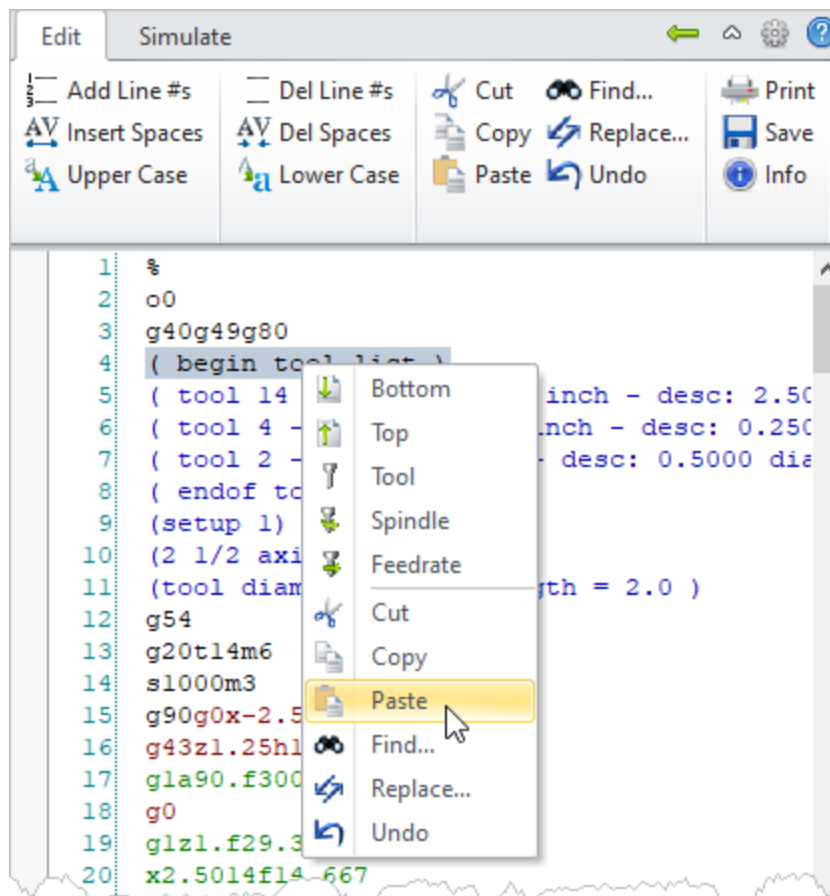
or

click:  >  >  > 

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file.



Paste Text (Right-Click Menu)



G-Code Browser: Edit tab, Paste Text (Right-Click Menu)



Procedure

1. Use the mouse to locate the cursor in the desired location.
2. Select **Paste** from the **Edit** tab or right-click and select **Paste** from the menu and the text is inserted into the G-Code file at the cursor location.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Copy Text](#)

[Undo Last Edit](#)

[Print G-Code](#)

[Save G-Code](#)

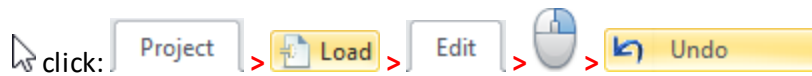
6.18.9 Undo Last Edit



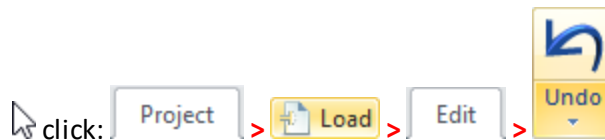
Use this command to [Undo](#) the last edit made to the g-code file. You can repeat the selection to [Undo](#) all edits made in the current session.



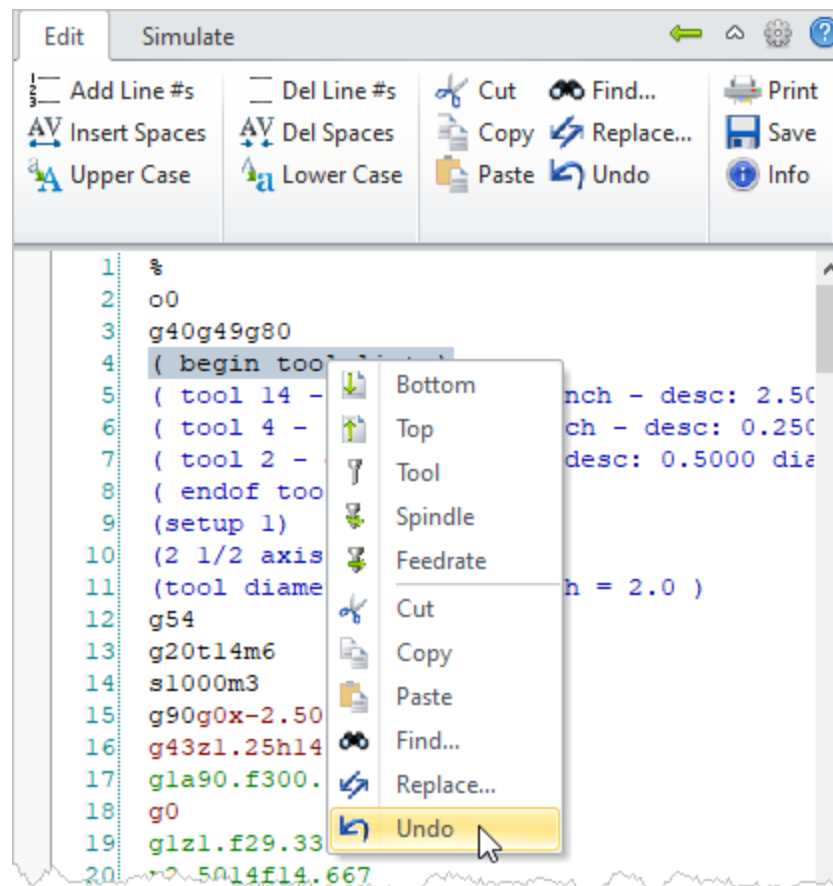
To Access this Command:



or



Undo Last Edit (Right-Click Menu)



G-Code Browser: Edit tab, Undo Last Edit (Right-Click Menu)



Procedure

1. Select [Undo](#) from the [Edit](#) tab or right-click and select Undo from the menu. The last edit will be undone.



Related Topics

[User Interface](#)

[Project Tab](#)

[Project Tree](#)

[Edit Tab](#)

[Cut Text](#)

[Copy Text](#)

[Paste Text](#)

[Find Text](#)

[Find & Replace Text](#)

[Print G-Code](#)

[Save G-Code](#)

Simulate Tab

Simulate

When you double-left-click on a G-Code file from the [Project](#) tree, that file is loaded into the [G-Code Editor](#). The [Simulate](#) tab contains commands to perform tool motions and cut material simulations of the g-code file. The [Simulate](#) tab and commands are listed below. **Note:** You must have [Stock](#), [Tool Library](#) and [Tool Crib](#) defined before performing a [Simulation](#).

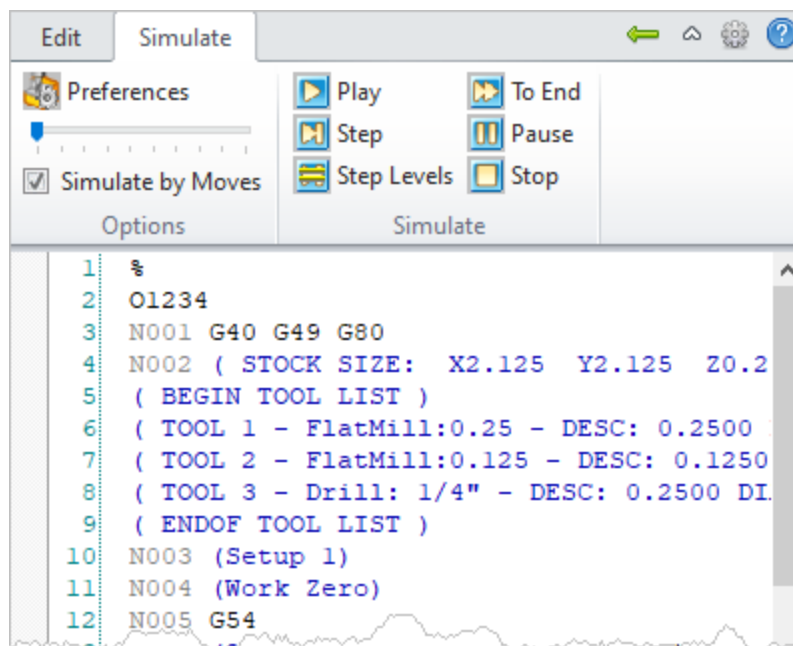


To Access this Command:

click: [Project](#) > [Load](#) > [Simulate](#)



Simulate tab



G-Code Browser, Simulate Tab



Ribbon Bar Options

Simulate

Contains file [Simulation](#) related commands.



Returns to the [Project](#) tab.



Minimizes and maximizes the ribbon bar menu.



Displays the [Preferences](#) dialog.



Displays the [Online Help](#) system.

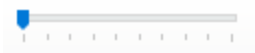


Simulate Tab, Options Pane

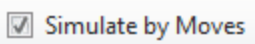


Preferences

Display [Simulation Preferences](#).



[Simulation speed](#) adjustment (Left: Slow, Right: Fast).

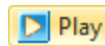


☒ Simulate by Moves

Check to [simulate by moves](#). Uncheck to simulate by distance. See [Preferences](#) to set the default mode.

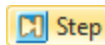


Simulate Tab, Simulate Pane



Play

[Play](#) the simulation.



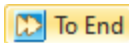
Step

[Step](#) through the simulation by distance or by moves.



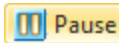
Step Levels

[Display](#) simulation in [Levels](#).



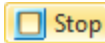
To End

[Go direction to the End](#) result of the simulation.



Pause

[Pause](#) the simulation.



Stop

[Stop](#) the simulation.



Simulate Tab, Display Toolbar



Display Toolbar



[Stock Model Visibility](#): Turn on/off stock model



Part Model Visibility: Turn on/off part model display during simulation.



Toolpath Visibility: Turn on/off toolpath display



Follow Toolpath Display: The toolpath is displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath after the tool passes.



Trace Toolpath Display: The toolpath is not displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath before the tool passes.



Segment Toolpath Display: The toolpath is only displayed for the segment that the tool is currently on.



Related Topics

[Project Tab](#)

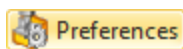
[Project Tree](#)


[Edit Tab](#)

[Online Help](#)

7.1 Simulate Options

7.1.1 Simulation Preferences

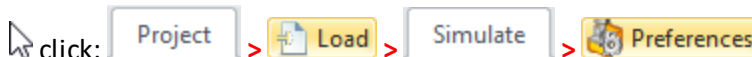


Select this button to display the [Simulation Preferences](#) dialog. This is the same as selecting  > [Simulation](#).



To Access this Command:

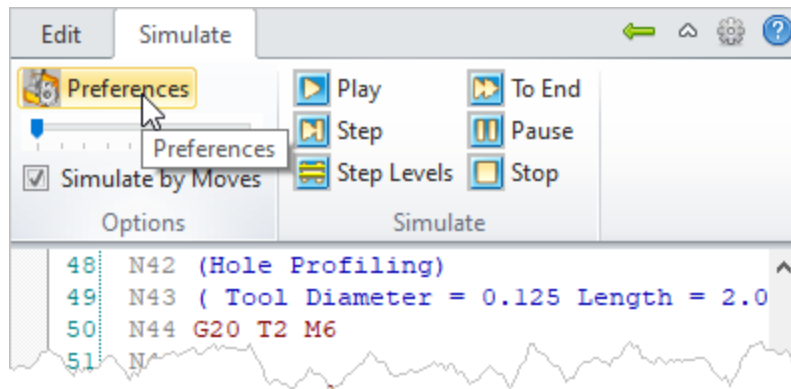
click:



Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



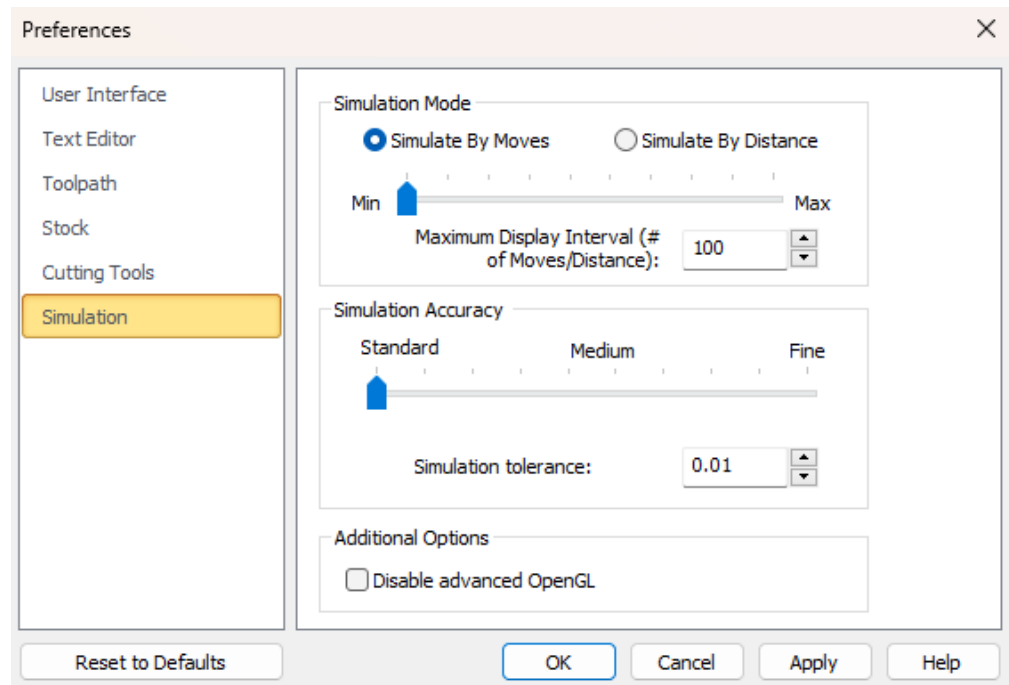
G-Code Browser: [Simulate tab](#), [Preferences menu item](#)



G-Code Browser: Simulate, Preferences menu item

Procedure

1. Select [Preferences](#) from the [Simulate](#) tab. This will display the [Simulate Preferences](#) dialog.



Preferences > Simulation

2. Make desired changes and then pick [Apply](#). Pick [OK](#) to close the dialog.

Related Topics

[Simulate Tab](#)

[Simulation Speed](#)

[Simulation Mode](#)

7.1.2 Simulation Speed



You can use this slider to adjust the simulation speed. Left is slower and right is faster. You can set the default speed for simulations from the [Simulation Preferences](#) dialog.



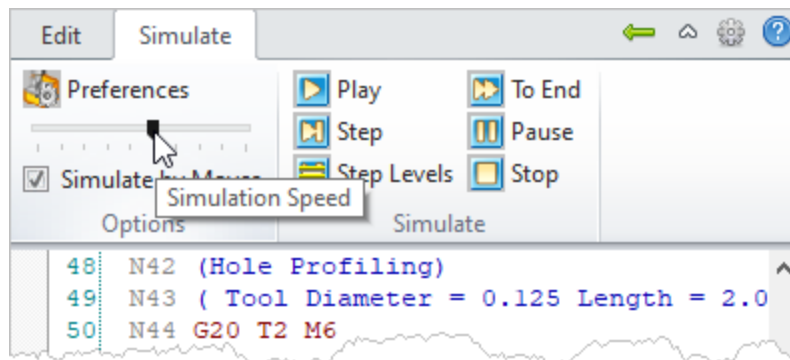
To Access this Command:



Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



G-Code Browser: Simulate, Speed menu item



G-Code Browser: Simulate, Speed menu item



Procedure

1. Left-click and drag the slider to adjust the [Simulation Speed](#). Left is slower and right is faster.



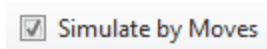
Related Topics

[Simulate Tab](#)

[Simulation Preferences](#)

[Simulation Mode](#)

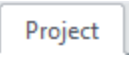
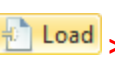
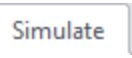

7.1.3 Simulation Mode



You can use this checkbox to [Simulate by Moves](#). If unchecked, the simulation is calculated by distance as defined in the [Simulation Preferences](#) dialog.

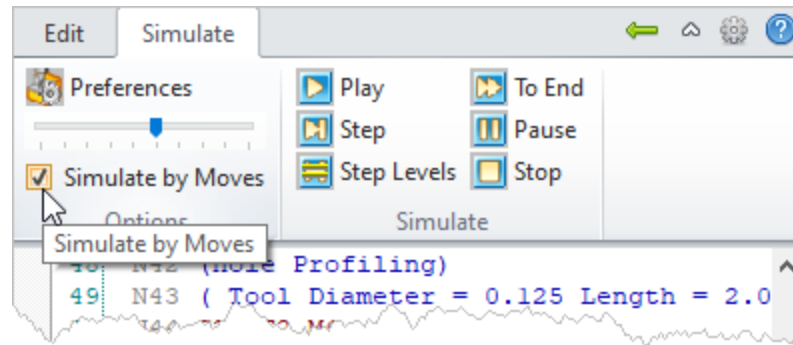


To Access this Command:

click:  >  >  >  Simulate by Moves

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.

G-Code Browser: Simulate, Mode menu item



G-Code Browser: Simulate, Mode menu item

Procedure

1. Check the box to Simulate by Moves. Uncheck the box to Simulate by Distance. The increment for moves and distance can be controlled from the [Simulation Preferences](#) dialog.

Related Topics


[Simulate Tab](#)

[Simulation Preferences](#)

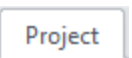
[Simulation Speed](#)

7.2 Simulation Controls

7.2.1 Play Simulation

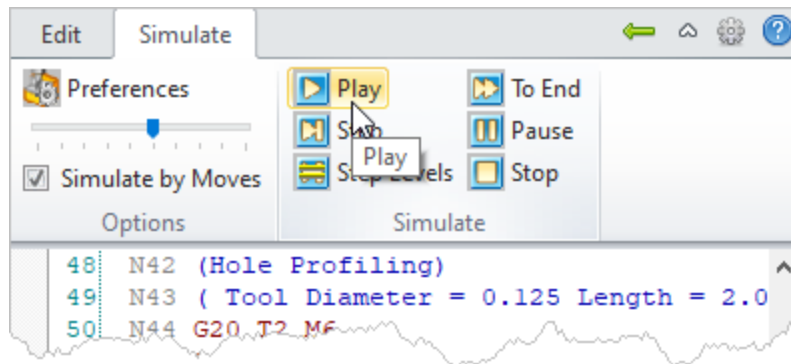
 Select this command to [Play](#) the simulation. If [Stock](#) is defined, a cut material simulation is displayed. If no [Stock](#) is defined only the tool motions is simulated.

To Access this Command:

click:  >  >  > 

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.

G-Code Browser: Simulate, Play menu item



G-Code Browser: Simulate, Play menu item



Procedure

1. Select **Play** from the **Simulate** tab and the simulation will begin from the current tool location.



Related Topics

[Simulate Tab](#)

[Step Thru Simulation](#)

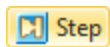
[Display Simulation Levels](#)

[Simulate to End](#)

[Pause Simulation](#)

[Stop Simulation](#)

7.2.2 Step Thru Simulation



Select this command to **Step** through the simulation. Each step is determined by the [Simulation Mode](#). You can control this from the [Simulation Preferences](#) dialog.



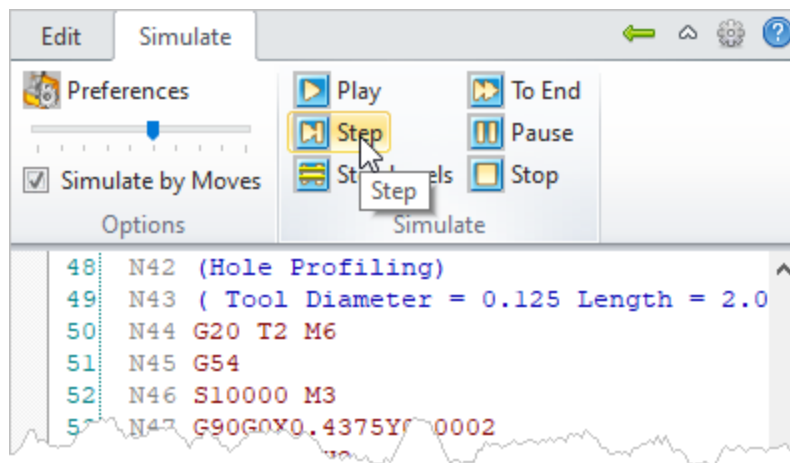
To Access this Command:

click: **Project** > **Load** > **Simulate** > **Step** (with [Simulation Window](#) displayed)

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the **Project** tab.



G-Code Browser: Simulate, Step menu item



G-Code Browser: Simulate, Step menu item



Procedure

1. Select **Step** from the **Simulate** tab and the simulation will advance by one increment. The increment is determined by the [Simulation Mode](#). You can control this from the [Simulation Preferences](#) dialog.



Related Topics

[Simulate Tab](#)

[Play Simulation](#)

[Display Simulation Levels](#)

[Simulate to End](#)

[Pause Simulation](#)

[Stop Simulation](#)

7.2.3 Display Simulation Levels



Step Levels If your toolpath has multiple Z levels, you can select this command to move the tool to the end of each level. If [Stock](#) is defined, the cut material simulation at each Z level is displayed.



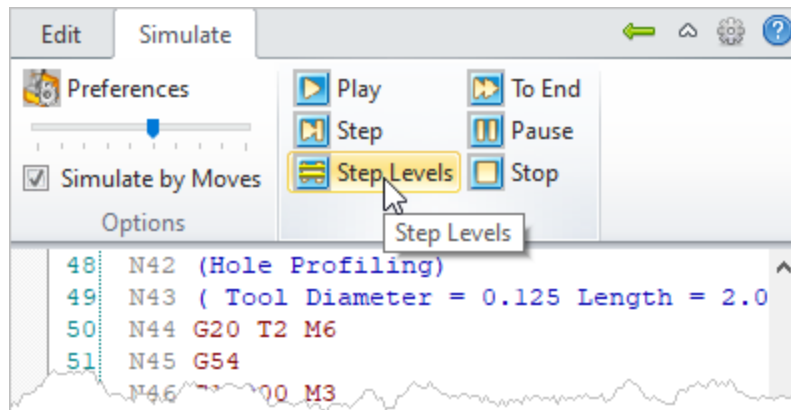
To Access this Command:

click: **Project** > **Load** > **Simulate** > **Step Levels**

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the **Project** tab.



G-Code Browser: Simulate, Step Levels menu item



G-Code Browser: Simulate, Step Levels menu item



Procedure

1. Select **Step Levels** from the **Simulate** tab and the simulation will advance to the end of the current Z level.



Related Topics

[Simulate Tab](#)

[Play Simulation](#)

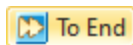
[Step Thru Simulation](#)

[Simulate to End](#)

[Pause Simulation](#)

[Stop Simulation](#)

7.2.4 Simulate to End



You can select this command to move the **End** of the simulation. If [Stock](#) is defined, the final cut material simulation is displayed.



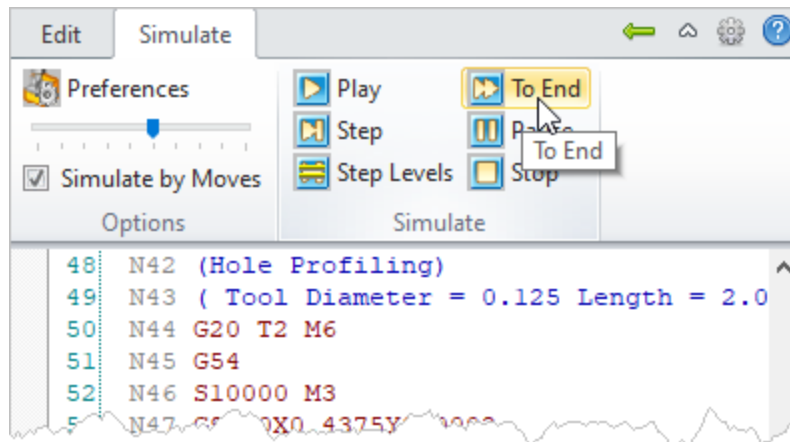
To Access this Command:

click: **Project** > **Load** > **Simulate** > **To End** (with [Simulation Window](#) displayed)

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the **Project** tab.



G-Code Browser: Simulate, To End menu item



G-Code Browser: Simulate, To End menu item

**Procedure**

1. Select **To End** from the **Simulate** tab to move the **End** of the simulation.

**Related Topics**

[Simulate Tab](#)

[Play Simulation](#)

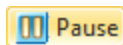
[Step Thru Simulation](#)

[Display Simulation Levels](#)

[Pause Simulation](#)

[Stop Simulation](#)

7.2.5 Pause Simulation



You can select this command to **Pause** the simulation at the current tool location. If **Stock** is defined, the cut material simulation is also paused.

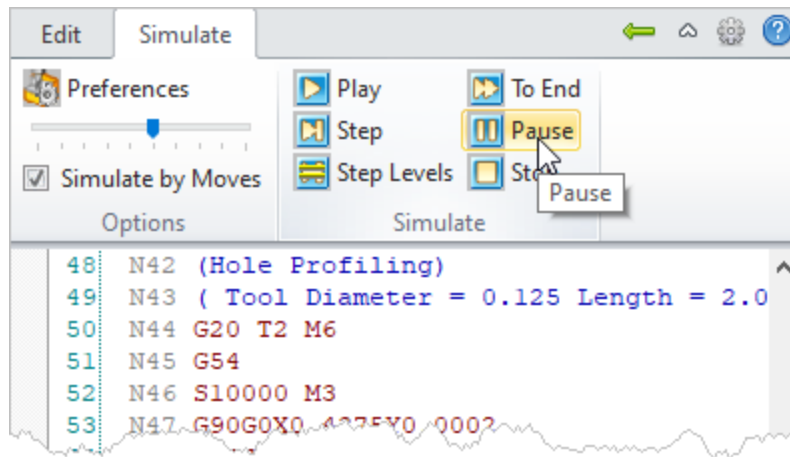
**To Access this Command:**

click: **Project** > **Load** > **Simulate** > **Pause**

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the **Project** tab.



G-Code Browser: Simulate, Pause menu item



G-Code Browser: Simulate, Pause menu item



Procedure

1. Select **Pause** from the **Simulate** tab to **Pause** the simulation at the current tool location.



Related Topics

[Simulate Tab](#)

[Play Simulation](#)

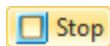
[Step Thru Simulation](#)

[Display Simulation Levels](#)

[Simulate to End](#)

[Stop Simulation](#)

7.2.6 Stop Simulation



You can select this command to **Stop** the simulation at the current tool location. If [Stock](#) is defined, the cut material simulation is also stopped.



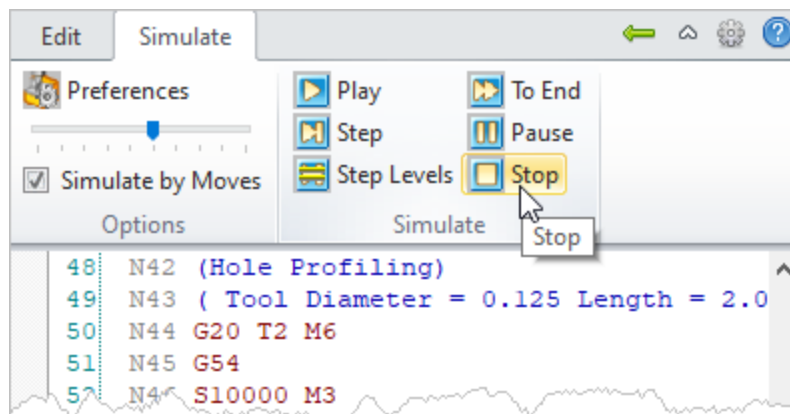
To Access this Command:

click: **Project** > **Load** > **Simulate** > **Stop**

Note: To access this tab activate (i.e., double-left-click) on a loaded g-code file from the [Project](#) tab.



G-Code Browser: Simulate, Stop menu item



G-Code Browser: Simulate, Stop menu item



Related Topics

[Simulate Tab](#)

[Play Simulation](#)

[Step Thru Simulation](#)

[Display Simulation Levels](#)

[Simulate to End](#)

[Pause Simulation](#)

Functions from the MILL Module

You can access the [G-Code Editor](#) module from within the [MILL](#) module by clicking on the G-Code icon located in each machining operation (Mop) folder as shown below if the icon is not flagged. However, ONLY the [Edit](#) tab is available and you cannot simulate G-Code.



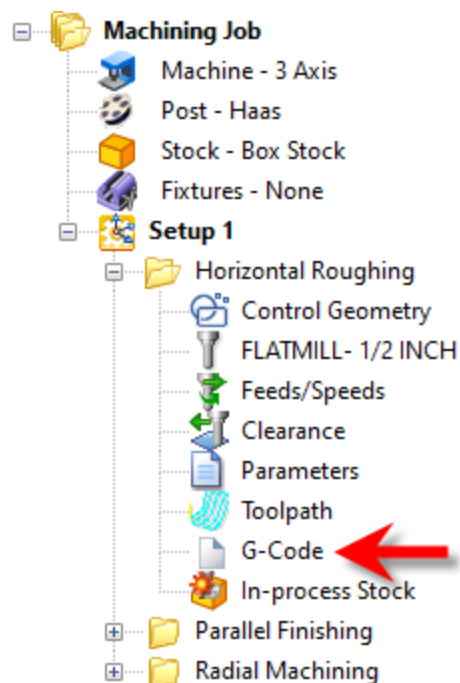
How G-Code is Stored

Your G-Code data is stored with your CAM part files. When you generate a toolpath operation in the MILL module the G-Code related to that operation is stored with the CAM part file. This means that you will always have access to the latest G-Code generated for any operation.



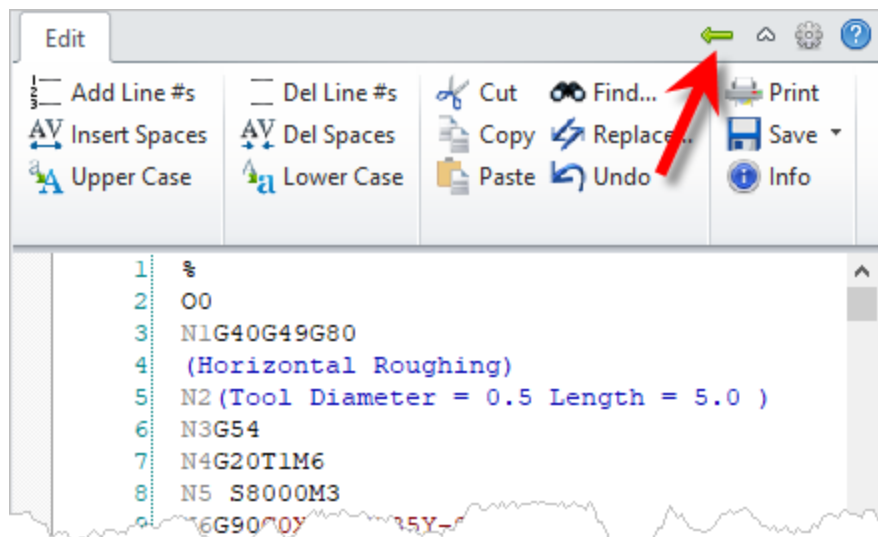
To Access G-Code from the MILL Module

To access the [G-Code Editor](#) from an operation that you have generated within the [MILL](#) module simply click on the [G-Code](#) icon within the mop folder as shown below:



G-Code Browser: Tool Crib menu item

The [Machining Browser](#) will be replaced with the [Edit](#) tab of the [G-Code Editor](#) as shown below. To return to the [MILL](#) module pick the left arrow icon at the top of the [G-Code Editor](#) browser as shown below.



G-Code Browser: Tool Crib menu item

Find More Resources

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



Index

- A -

Add Line Numbers 93

- C -

Colors

Cut Stock 19

Cutting Tool 20

Stock 19

Text Editor 16

Tool Motion Display 17

Copy Text 103, 125

Cut Text 101, 123

Cutting Tools

Crib 62

Library 60

Preferences 20

- D -

Delete

a G-Code File from the Project 88

Delete Line Numbers 97

Delete Spaces 99

Display Toolbar

Toggle Follow Toolpath Display 31, 90

Toggle Part Visibility 31, 90

Toggle Segment Toolpath Display 31, 90

Toggle Stock Visibility 31, 90

Toggle Toolpath Visibility 31, 90

Toggle Trace Toolpath Display 31, 90

- E -

Edit

G-Code File 84

Edit > Select Full Line 116

Edit Tab 31, 90

- F -

Find Text 106

- G -

G-Code

Add Line Numbers 93

Copy Text 103, 125

Cut Text 101, 123

Delete Line Numbers 97

Delete Spaces 99

Find Text 106

From the MILL Module 142

Go to Bottom 117

Go to Next Feedrate 122

Go to Next Spindle 121

Go to Next Tool 119

Go to Top 118

Information 65

Insert Spaces 95

Load 64

Make Lower Case 100

Make Upper Case 96

Merge Files 67

Output 83

Paste Text 104, 126

Print G-Code 111

Replace Text 107

Save 81

Save As 113

Save G-Code 112

Show Info 114

Transform 71

Undo Last Edit 109, 128

XY Instance 79

Z Instance 77

G-Code Editor

About 5

Features of 6

Overview 5

Understanding 7

Workflow 7

- H -

Help 37

- I -

Information

G-Code 65

on a G-Code File 87

Insert Spaces 95

- L -

Load G-Code 64

- M -

Machine Tool Setup 41

Make Lower Case 100

Make Upper Case 96

MCS Setup 41

Merge Files 67

MILL Module 142

Mirror G-Code 71

Move G-Code 71

- N -

Navigate

Go to Bottom 117

Go to Next Feedrate 122

Go to Next Spindle 121

Go to Next Tool 119

Go to Top 118

- O -

Output G-Code 83

- P -

Part

Toggle Visibility 31, 90

Paste Text 104, 126

Preferences 12

Cutting Tools 20

Simulation 21, 132

Stock 19

Text Editor 16

Toolpath 17

User Interface 13

Print G-Code 111

Project

Tab 26, 39

Tree 28

- R -

Replace Text 107

Rotate G-Code 71

- S -

Save As 113

Save G-Code 81, 112

Scale G-Code 71

Select

Full Line 116

Show G-Code Info 114

Simulate

G-Code File 85

Tab 35, 130

Simulation

Display Levels 137

Mode 134

Pause 139

Play 135

Preferences 21, 132

Speed 134

Step Thru 136

Stop 140

To End 138

Stock

Box 46

Cylinder 51

Delete 57

from Selection 58

Menu 45

Part Box 49

Part Cylinder 54

Preferences 19

Stock

Toggle Visibility 31, 90

Transparency 19

- T -

Text Editor Preferences 16

Toggle Follow Toolpath Display 31, 90

Toggle Part Visibility 31, 90

Toggle Segment Toolpath Display 31, 90

Toggle Stock Visibility 31, 90

Toggle Toolpath Visibility 31, 90

Toggle Trace Toolpath Display 31, 90

Tool Crib 62

Tool Library 60

Toolpath

Display width 17

Follow Display 31, 90

Preferences 17

Segment Display 31, 90

Toggle Visibility 31, 90

Trace Display 31, 90

Tools (See Cutting Tools) 60

Transform G-Code

Mirror 71

Move 71

Rotate 71

Scale 71

- U -

Undo Last Edit 109, 128

User Interface 12

Display Toolbar 31, 90

Edit Tab 31, 90

G-Code Browser 23

Preferences 13

Project Tab 26, 39

- W -

Where to go for more help 144

- X -

XY Instance G-Code 79

- Z -

Z Instance G-Code 77