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

G-Code Editor Reference Guide

RhinoCAM 2026

Published: March 2026

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

MecSoft Corpotation
© Copyright 1998-2026

Table of Contents

Welcome 5

1 Overview ... 5
2 Features ... 6
3 Understanding ... 7
4 Work Flow ... 7

Quick Start 9

CAM Resource Guide 12

User Interface 13

1 Preferences ... 13
- User Interface ... 14
- Text Editor ... 17
- Toolpath ... 18
- Stock ... 20
- Cutting Tools ... 21
- Simulation ... 22

2 G-Code Browser ... 24
- Project Tab ... 27
- Project Tree ... 29
- Edit Tab ... 32
- Simulate Tab ... 35
- Online Help ... 38

Project Tab 40

1 Machine MCS Setup ... 42
2 Setup Pane ... 47
- Stock ... 47
  - Box Stock ... 48
  - Part Box Stock ... 51
  - Cylinder Stock ... 54
  - Part Cylinder Stock ... 56
  - Delete Stock ... 59
  - Stock from Selection ... 60
- Tool Library ... 62
- Tool Crib ... 65

3 G-Code Pane ... 67
- Load ... 67
- Info ... 68

4 File Operations Pane ... 69
- Merge Files ... 69
- Transform ... 73

© 2026 MecSoft Corporation

Z Instance ... 80
XY Instance ... 82
Save ... 84
Output ... 86
5 Right-Click Menu ... 87
Edit-G-Code-File ... 87
Simulate G-Code File ... 88
Info on G-Code File ... 90
Delete G-Code File ... 91

Edit Tab 93

1 Add Line Numbers ... 96
2 Insert Spaces ... 97
3 Make Upper Case ... 98
4 Delete Line Numbers ... 100
5 Delete Spaces ... 101
6 Make Lower Case ... 102
7 Cut Text ... 104
8 Copy Text ... 106
9 Paste Text ... 107
10 Find Text ... 109
11 Find & Replace Text ... 110
12 Undo Last Edit ... 112
13 Print G-Code ... 114
14 Save G-Code ... 115
15 Save As ... 116
16 Show G-Code Info ... 117
17 Select Full Line ... 119
18 Right-Click Menu ... 120
Go to Bottom ... 120
Go to Top ... 121
Go to Next Tool ... 122
Go to Next Spindle ... 124
Go to Next Feedrate ... 125
Cut Text ... 126
Copy Text ... 128
Paste Text ... 129
Undo Last Edit ... 131

Simulate Tab 133

1 Simulate Options ... 135
Simulation Preferences ... 135
Simulation Speed ... 136
Simulation Mode ... 137
2 Simulation Controls ... 138
Play Simulation ... 138

© 2026 MecSoft Corporation

Step Thru Simulation 139
Display Simulation Levels 140
Simulate to End 141
Pause Simulation 142
Stop Simulation 143

Functions from the MILL Module 145

Find More Resources 147

Index 148

© 2026 MecSoft Corporation

Welcome

G-CODE Editor Module 2026

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

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 RhinoCAMRhinoceros. 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

© 2026 MecSoft Corporation

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

© 2026 MecSoft Corporation

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

© 2026 MecSoft Corporation

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

© 2026 MecSoft Corporation

Quick Start

G-CODE Editor Module 2026

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

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

What's New!

What's New in RhinoCAM 2026

The Complete Quick Start Video Play List

Here is a link to the complete 2026 Video Play List

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.

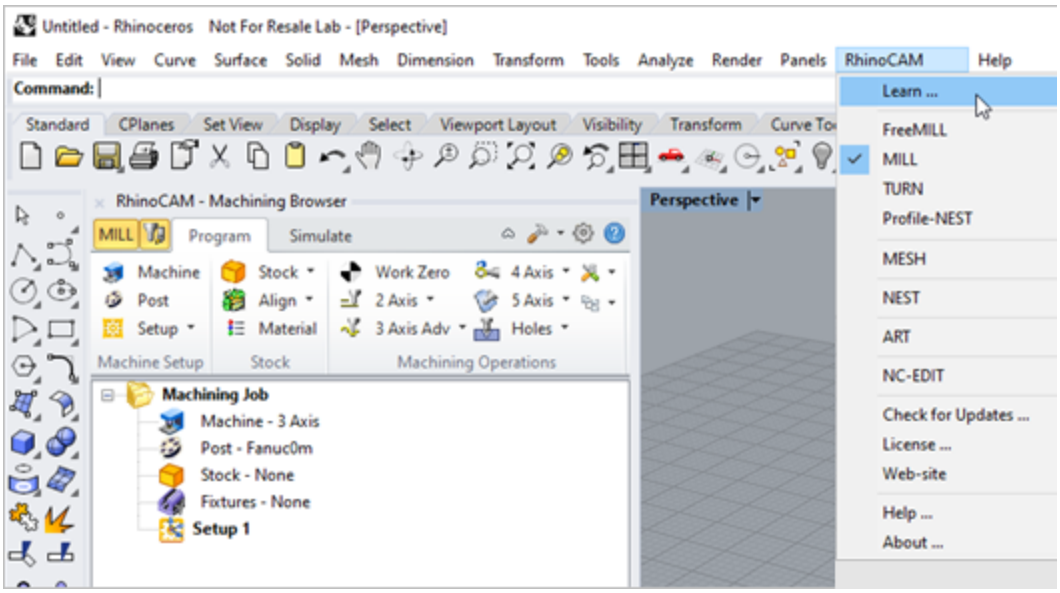
You will find:

- 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 ...

© 2026 MecSoft Corporation

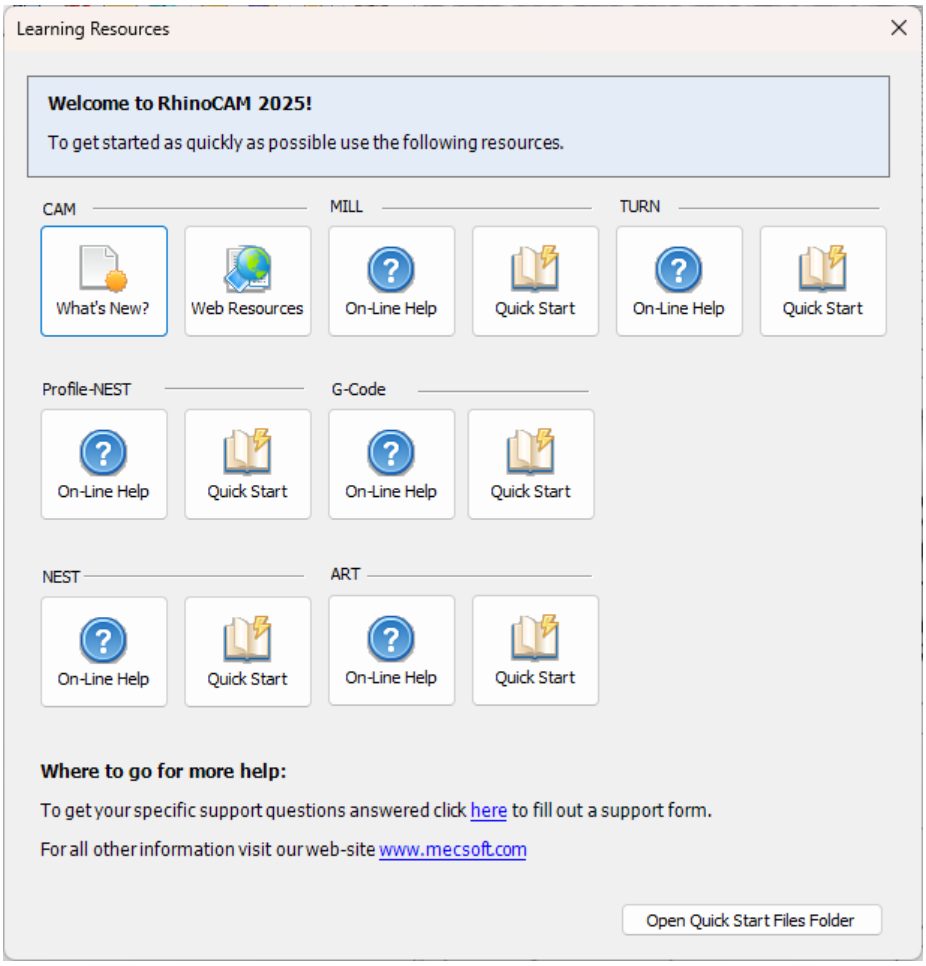


To access the Learning Resources dilog in RhinoCAM

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

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

© 2026 MecSoft Corporation



Learning Resources Dialog

Related Topics

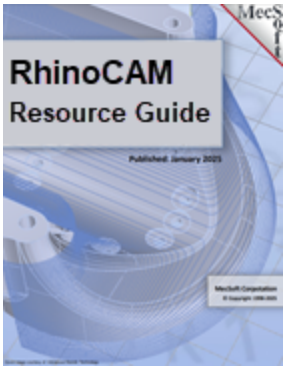
Find More Resources

© 2026 MecSoft Corporation

CAM Resource Guide

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

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Check Here!

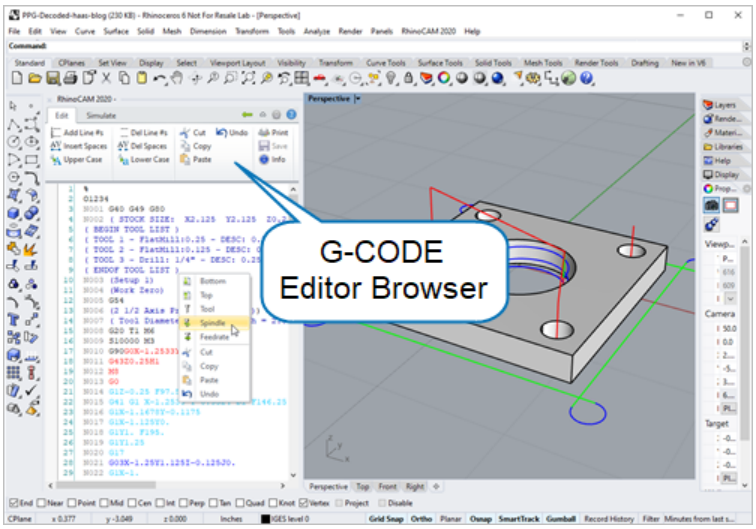
What's New | Quick Start Play List

© 2026 MecSoft Corporation

User Interface

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

The User Interface



RhinoCAM User Interface

Related Topics

Preferences

G-Code Browser

Project Tab

Edit Tab

Simulate Tab

4.1 Preferences

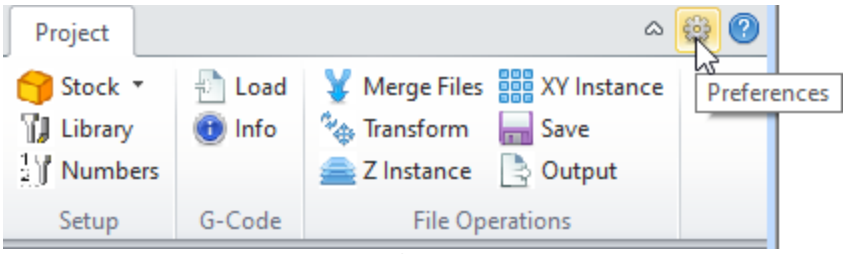
You can set various System Preferences that will be saved even after you exit the program. Select the Preferences icon from the G-Code Browser. When you install a new RhinoCAM update you can choose to import your Preferences from one version to the next.

To Access this Command:

click:

© 2026 MecSoft Corporation

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

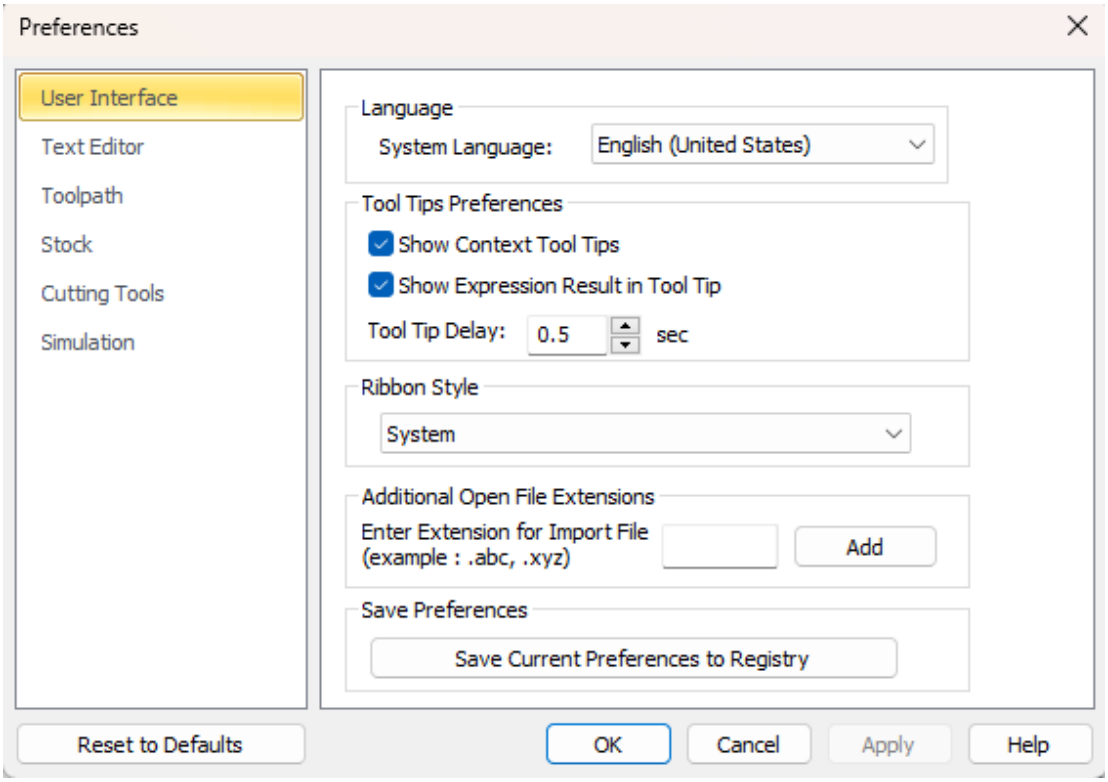
Here you can set preferences related to the g-code text editor including colors and associated programs and folders. You can use the Reset to defaults button if you want to revert to the default factory install settings.

To Access this Command:

click: > User Interface

Preferences > Text Editor

© 2026 MecSoft Corporation



Preferences > User Interface

Language

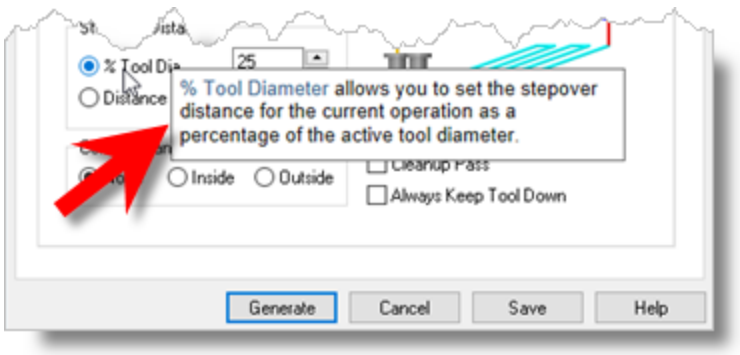
System language

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

Tool Tips Preferences

Show Context Tool Tips

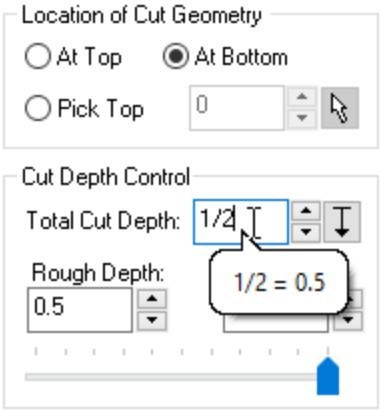
Check this box to display Context ToolTips when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. **Note** that Context ToolTips may not be available for ALL dialogs. You can also set the ToolTip Delay in seconds. This is the amount of time it takes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



© 2026 MecSoft Corporation

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.



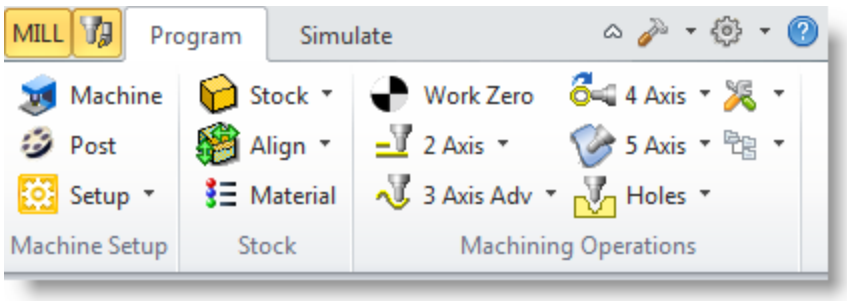
Show Expressions in ToolTip

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

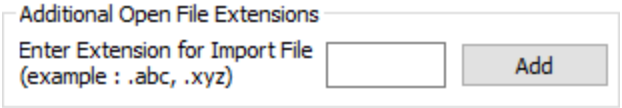
This allows the selection of different themes that change how the Browser windows appear. The borders, colors, highlighting, and shadowing of standard buttons, dialogs, and windows are controlled by which theme is selected.



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

© 2026 MecSoft Corporation

Save Current Preferences to Registry

If you have your preferences set the way you want them and do not want them top 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

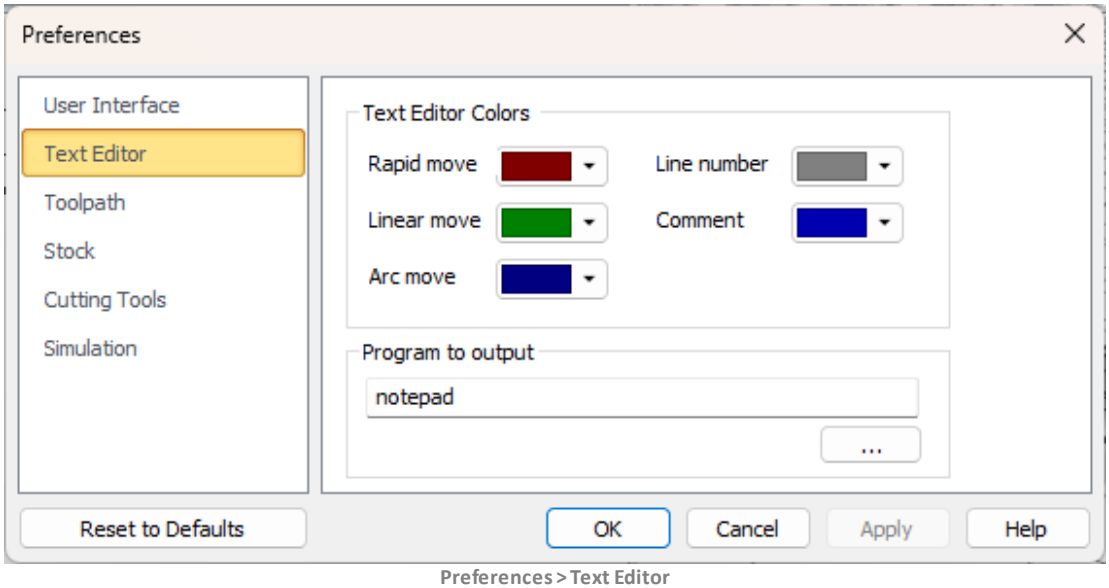
4.1.2 Text Editor

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

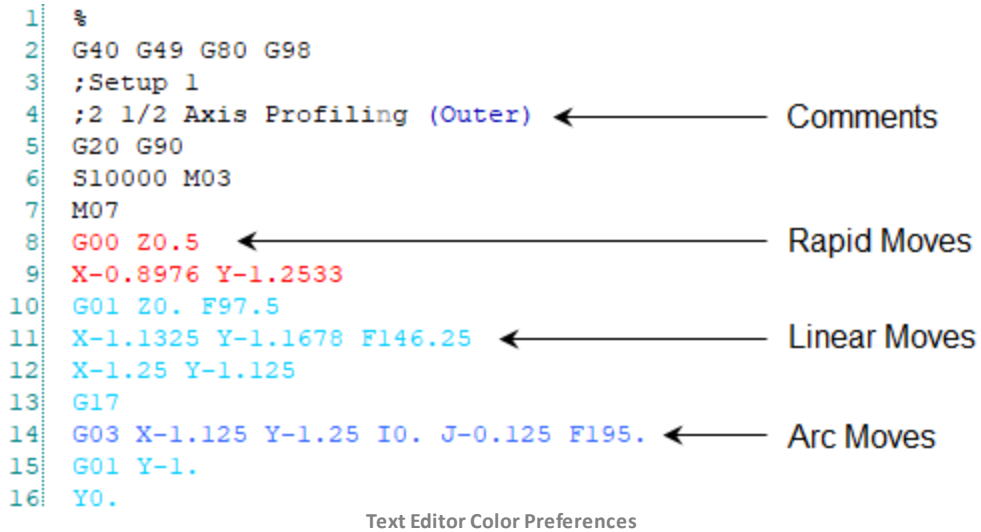


Preferences > Text Editor

Text Editor Colors

© 2026 MecSoft Corporation

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.



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

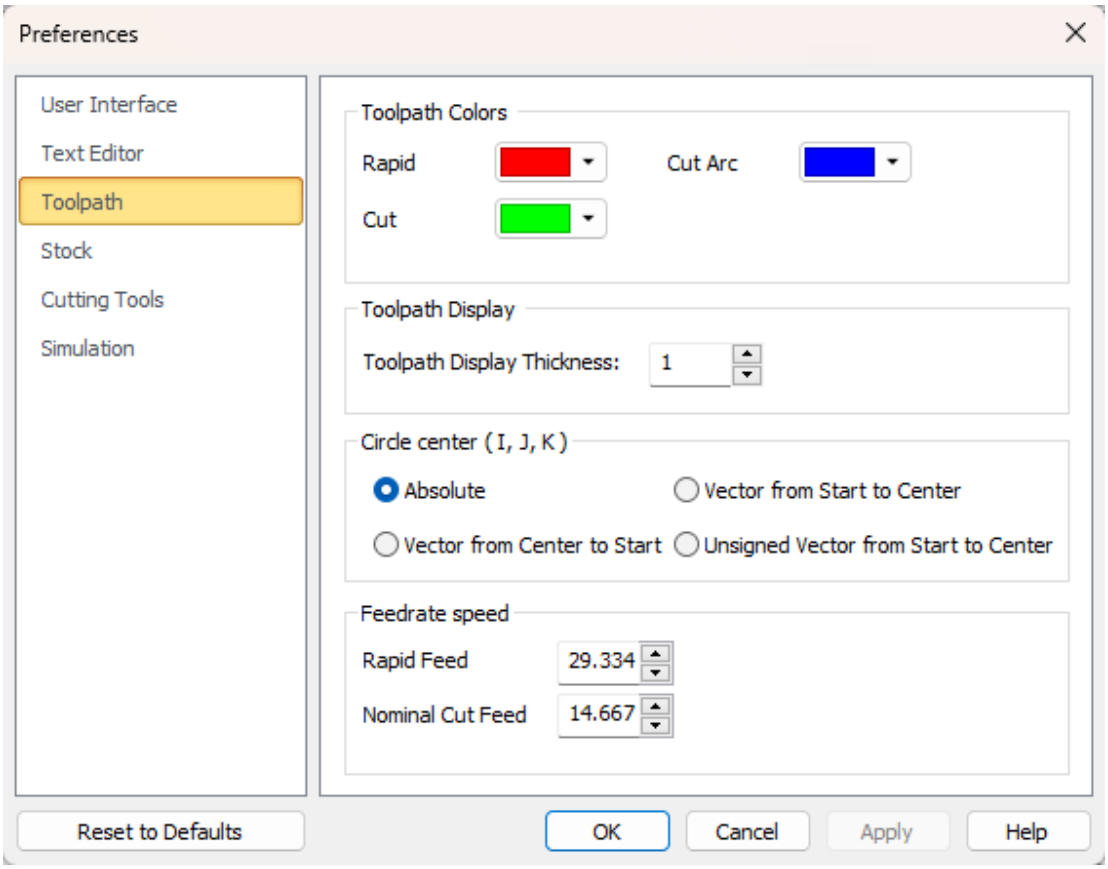
Here you can set Toolpath related preferences including colors, display, format and feedrate. You can use the Reset to defaults button if you want to revert to the default factory install settings.

To Access this Command:

click: > Toolpath

Preferences > Toolpath

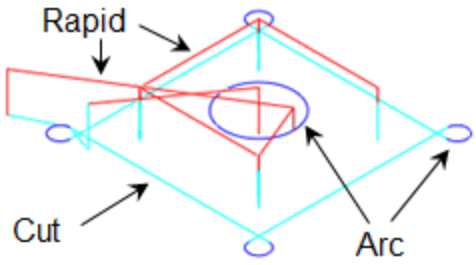
© 2026 MecSoft Corporation



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.

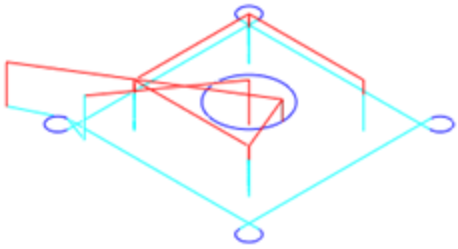


Toolpath Display Color Preferences

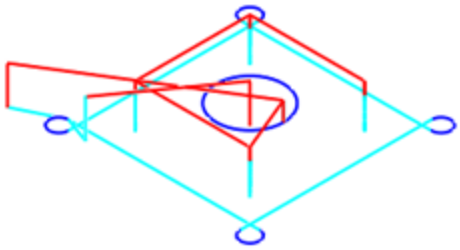
Toolpath Display

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

© 2026 MecSoft Corporation



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

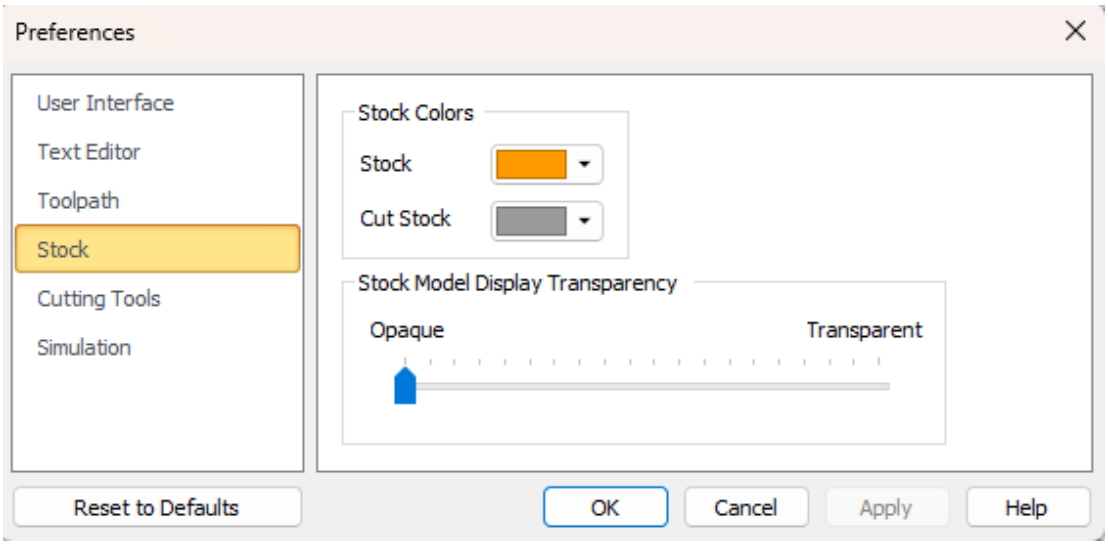
Here you can set Toolpath related preferences including colors, display, format and feedrate. You can use the Reset to defaults button if you want to revert to the default factory install settings.

To Access this Command:

© 2026 MecSoft Corporation

click: > Stock

Preferences > Stock



Preferences > Toolpath

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

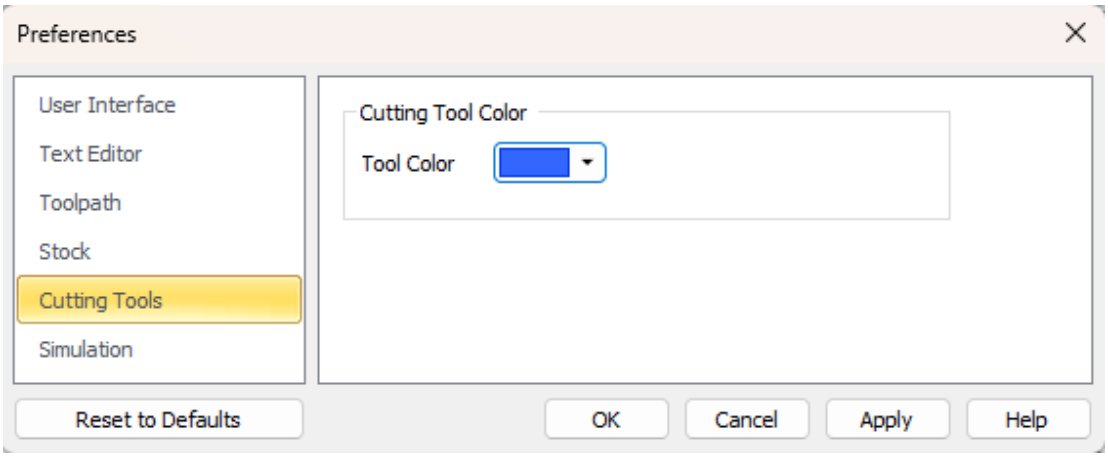
Here you can set Cutting Tool related preferences. You can use the Reset to defaults button if you want to revert to the default factory install settings.

To Access this Command:

© 2026 MecSoft Corporation

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

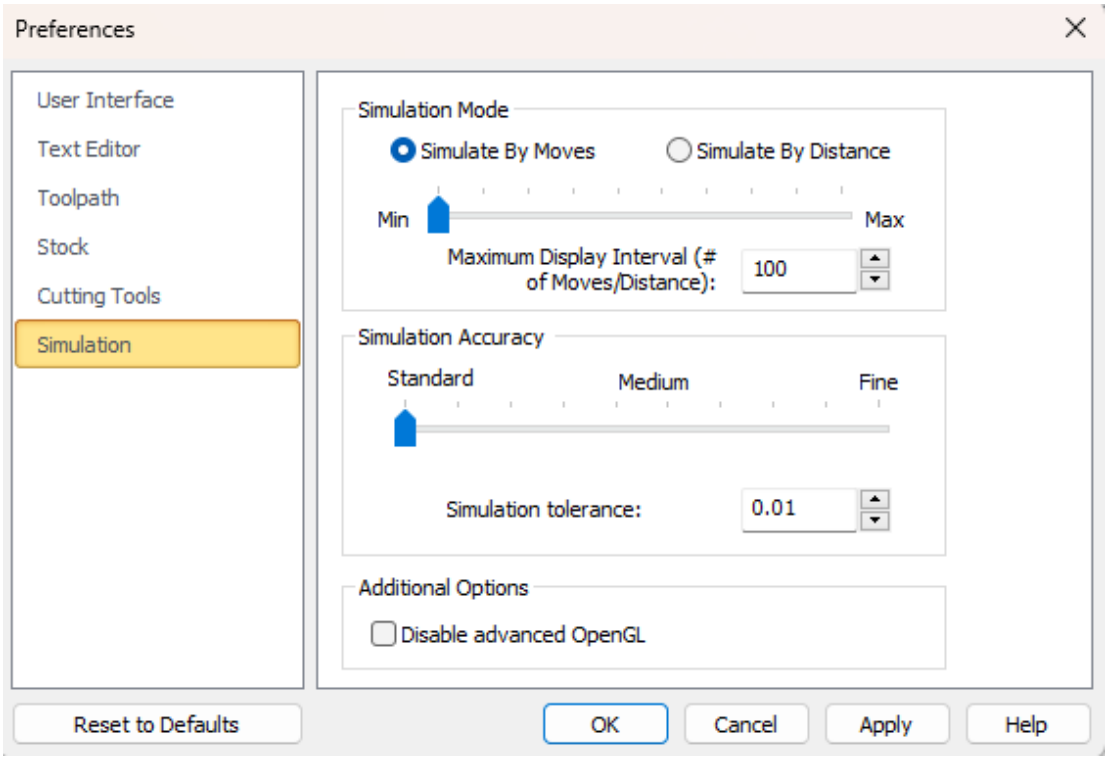
Here you can set Simulation related preferences. You can use the Reset to defaults button if you want to revert to the default factory install settings.

To Access this Command:

click: > Simulation

Preferences > Simulation

© 2026 MecSoft Corporation



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 un-checking 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 # of Motions / 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.

© 2026 MecSoft Corporation

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

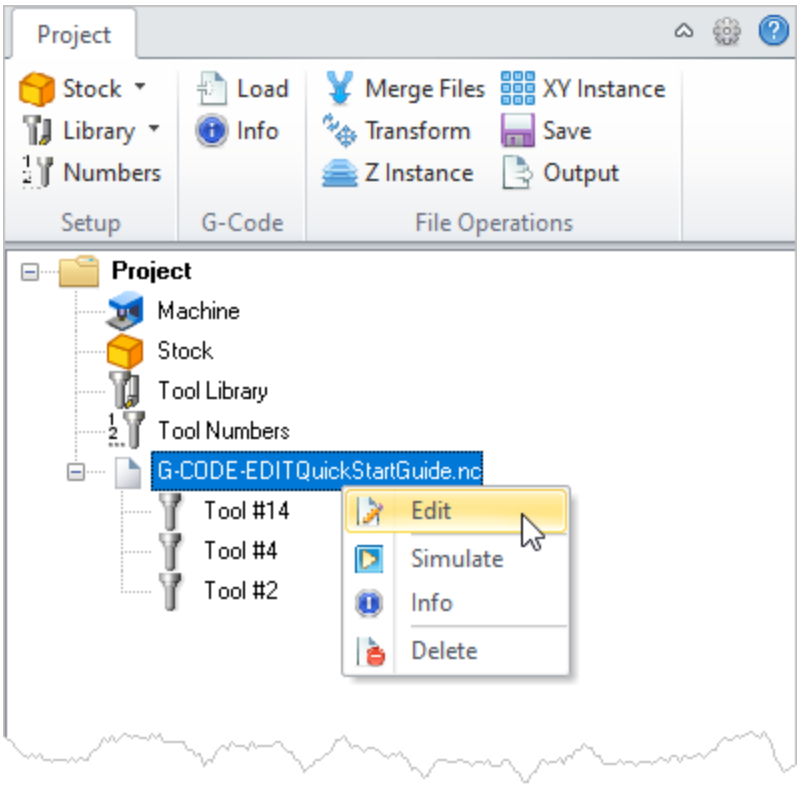
4.2 G-Code Browser

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

© 2026 MecSoft Corporation

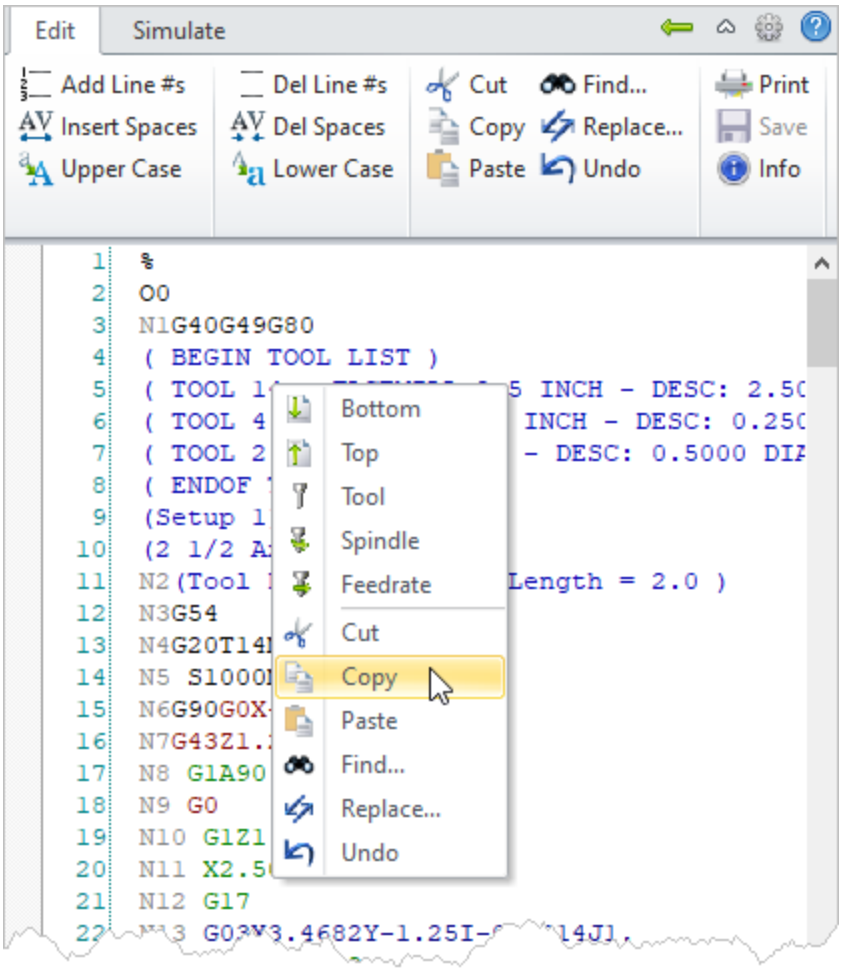


G-Code Browser, Project Tab

The Edit Tab

click: Project > Load > > Edit

© 2026 MecSoft Corporation

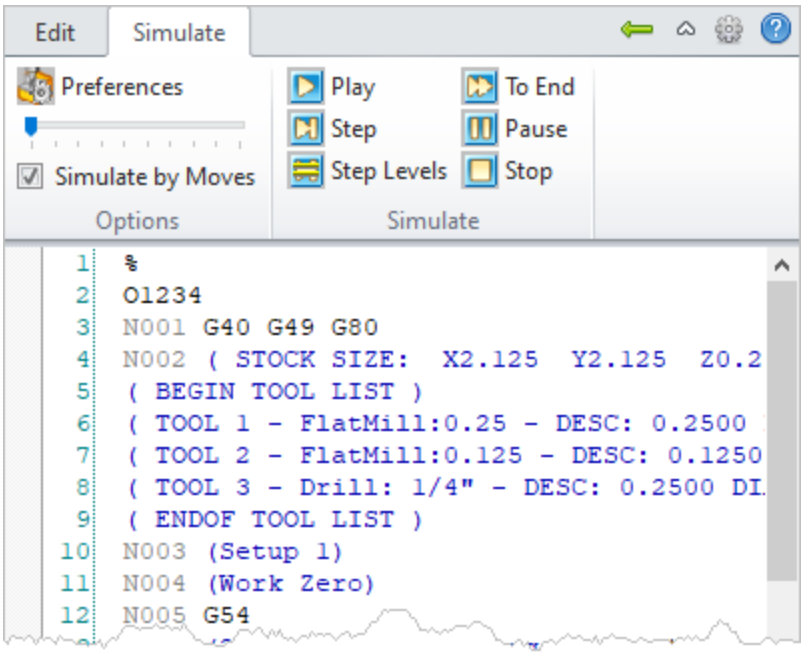


G-Code Browser, Edit Tab

The Simulate Tab

click: Project > Load > > Simulate (with Simulate Window displayed)

© 2026 MecSoft Corporation



G-Code Browser, Simulate Tab

Related Topics

User Interface

Project Tab

Project Tree

Edit Tab

Simulate Tab

4.2.1 Project Tab

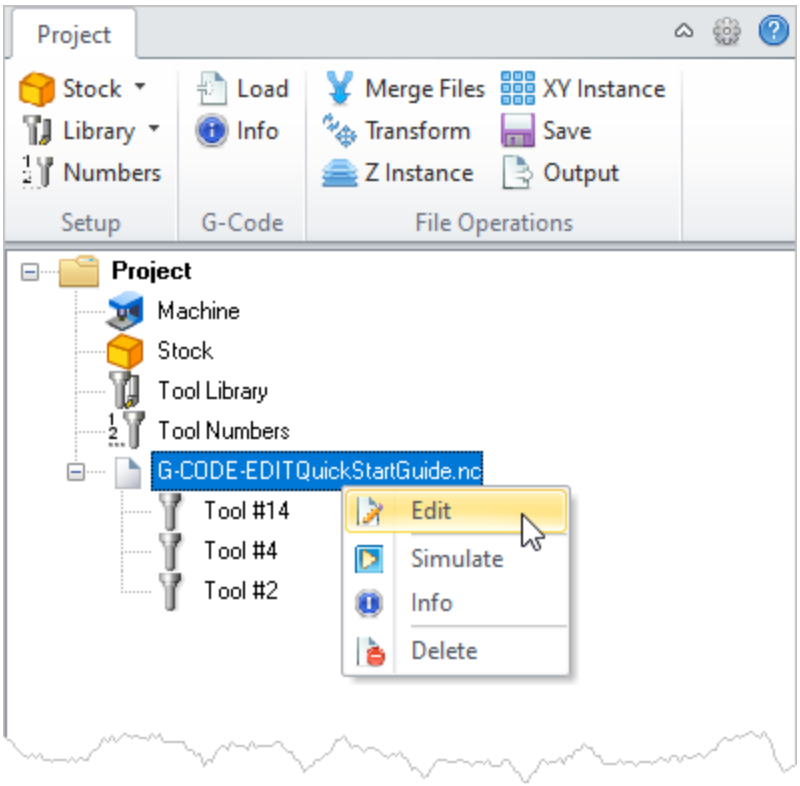
The Project tab is Selecting the Program tab in the Mops Browser shows 3 groups of menus that provide access for specifying Machine, Stock and defining Machining Operations.

To Access this Command:

click: Project

The Project Tab

© 2026 MecSoft Corporation



G-Code Browser, Project Tab

Ribbon Bar Options

Project 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

Stock Displays a menu of Stock definition commands.

Library Displays the Tool Library dialog.

Numbers Displays the Tool Crib dialog.

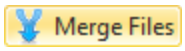
Project G-Code Pane

Load Load a g-code file into the Project tree.

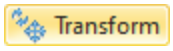
Info Display Information about the g-code files currently loaded into the Project.

© 2026 MecSoft Corporation

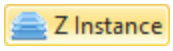
Project File Operations Pane



Merge two selected g-code files into one file.

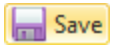


Displays the Transform g-code dialog.

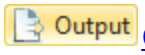


Displays the Z Instance g-code dialog.

Displays the XY Instance g-code dialog.

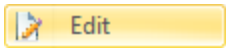


Saves all g-code files currently loaded into the Project.

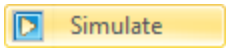


Outputs the g-code files currently loaded into the Project.

Project Tab, Right-Click Menu

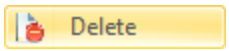


Edit the selected G-Code file.



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

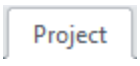
Simulate Tab

Online Help

4.2.2 Project Tree

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.

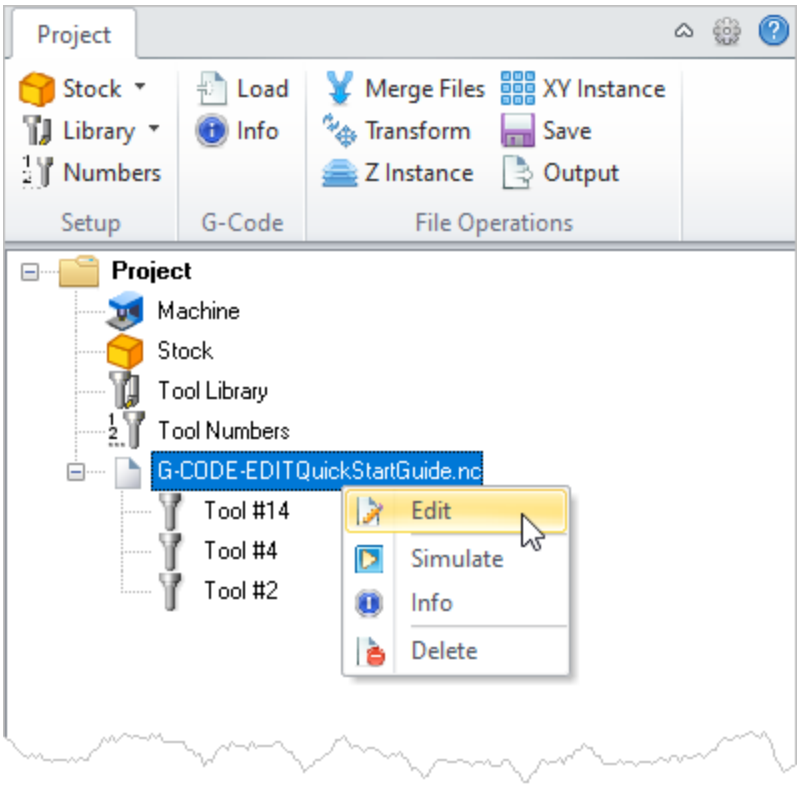
To Access this Command:

click: 

The Project Tab

You must have the Project tab selected in order to see the Project tree.

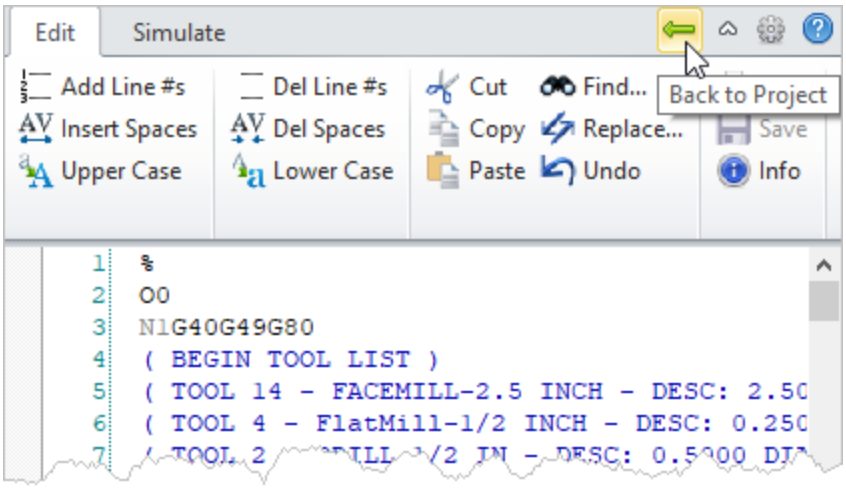
© 2026 MecSoft Corporation



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.

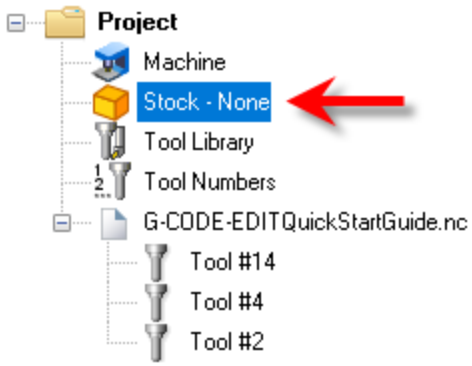


G-Code Browser, Return to Project Tab menu item

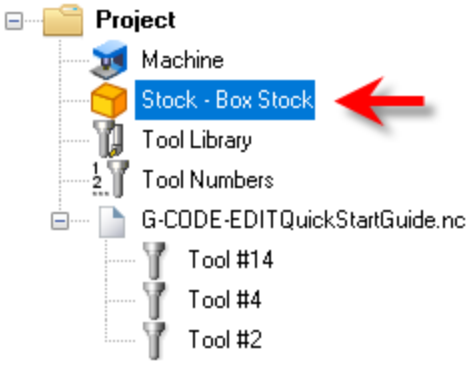
How Stock is Defined in the Project Tree

© 2026 MecSoft Corporation

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

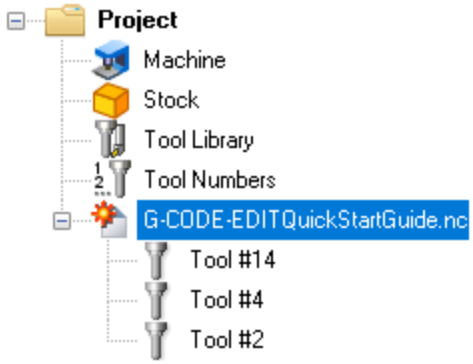


When Stock is defined

Red Flags in the Project Tree

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

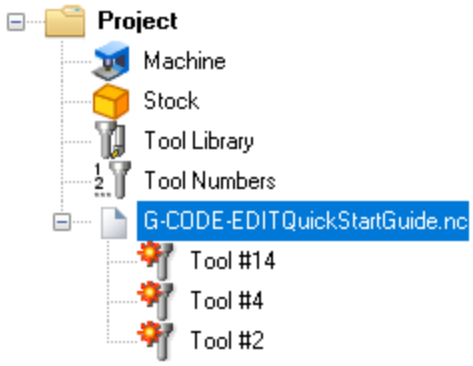
For example if a g-code file has been modified but has not yet been saved or output, it will be flagged as shown in the example below.



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.

© 2026 MecSoft Corporation



When a tool is flagged

Related Topics

Project Tab

Edit Tab

Simulate Tab

Online Help

4.2.3 Edit Tab

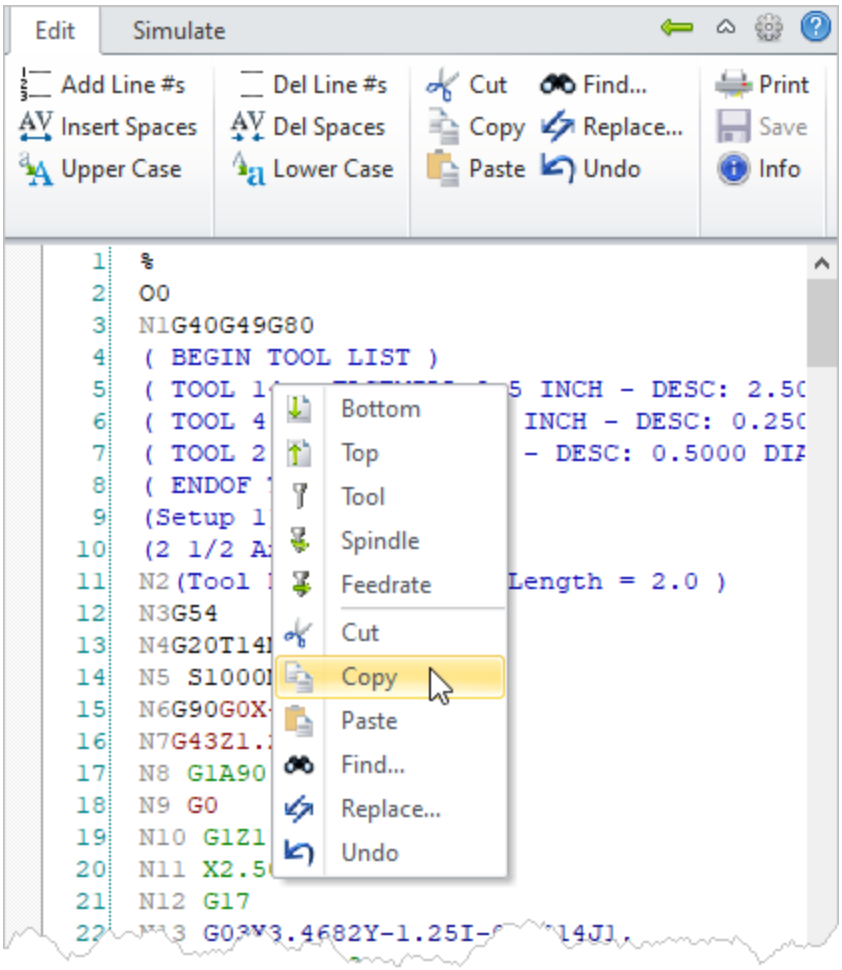
Edit When you double-left-click on a G-Code file from the Project tree, that file is loaded into the G-Code Editor. The Edit tab contains commands to Modify and Navigate the g-code file. The Edit tab and commands are listed below.

To Access this Command:

click: Project > Load > Edit

Edit tab

© 2026 MecSoft Corporation



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.

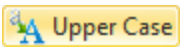
Displays the Online Help system.

Edit Tab

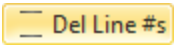
Add Line #s Add line numbers to the g-code file.

Insert Spaces Insert spaces (i.e., delimiters) between g-codes.

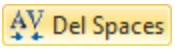
© 2026 MecSoft Corporation



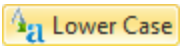
Convert g-code to all UPPERCASE text.



Delete line numbers from the g-code file.

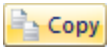


Delete spaces (i.e., delimiters) between g-codes.



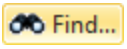
Convert g-code to all lowercase text.

Cut the selected text to the clipboard.

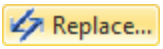


Copy the selected text to the clipboard.

Paste the selected text to the current cursor location.



Find text within the G-Code file.



Find & Replace text within the G-Code file.

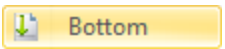
Undo the last command.

Print the G-Code file currently loaded in the editor.

Save the G-Code file currently loaded in the editor.

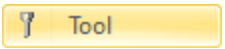
Display information about the current g-code file.

Edit Tab, Right-Click Menu

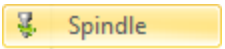


Go to the Bottom (End) of the G-Code file.

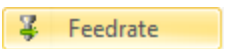
Go to the Top (Start) of the G-Code file.



Go to the next Tool Change code.

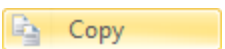


Go to the next Spindle code.

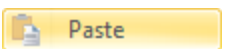


Go to the next Feedrate code.

Cut the selected text to the clipboard.

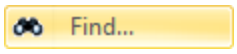


Copy the selected text to the clipboard.

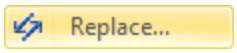


Paste the selected text to the current cursor location.

© 2026 MecSoft Corporation



Find text within the G-Code file.



Find & Replace text within the G-Code file.

Undo the last command.

Edit 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

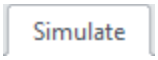
Project Tab

Project Tree

Simulate Tab

Online Help

4.2.4 Simulate Tab



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

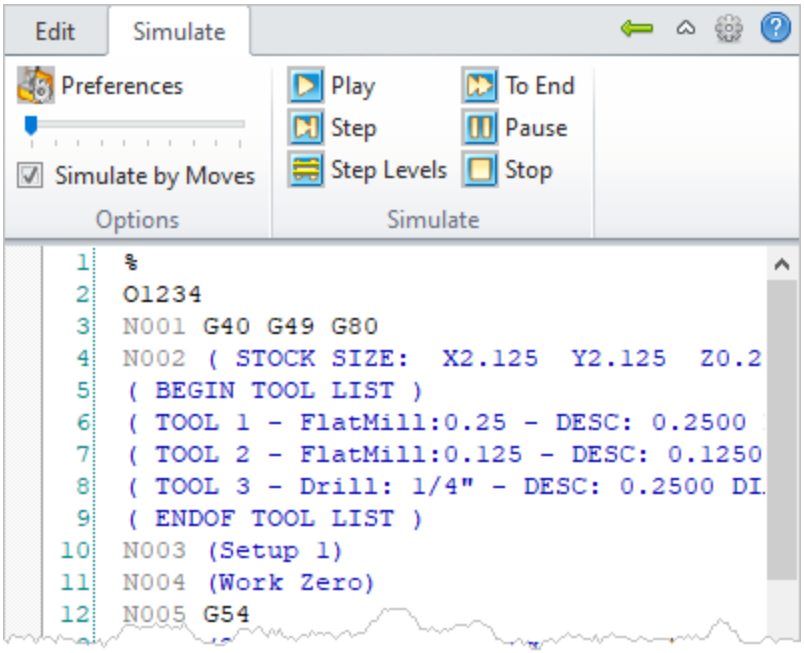
© 2026 MecSoft Corporation

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.

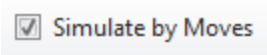
Displays the Online Help system.

Simulate Tab, Options Pane

Preferences Display Simulation Preferences.

© 2026 MecSoft Corporation

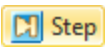
Simulation speed adjustment (Left: Slow, Right: Fast).



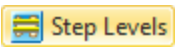
Check to simulate by moves. Uncheck to simulate by distance. See Preferences to set the default mode.

Simulate Tab, Simulate Pane

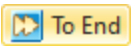
Play the simulation.



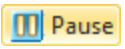
Step through the simulation by distance or by moves.



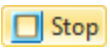
Display simulation in Levels.



Go direction to the End result of the simulation.



Pause the simulation.



Stop the simulation.

Simulate Tab, Display Toolbar

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

© 2026 MecSoft Corporation

Project Tab

Project Tree

Edit Tab

Online Help

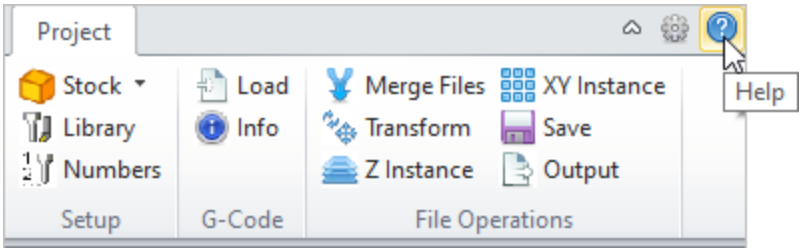
4.2.5 Online Help

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

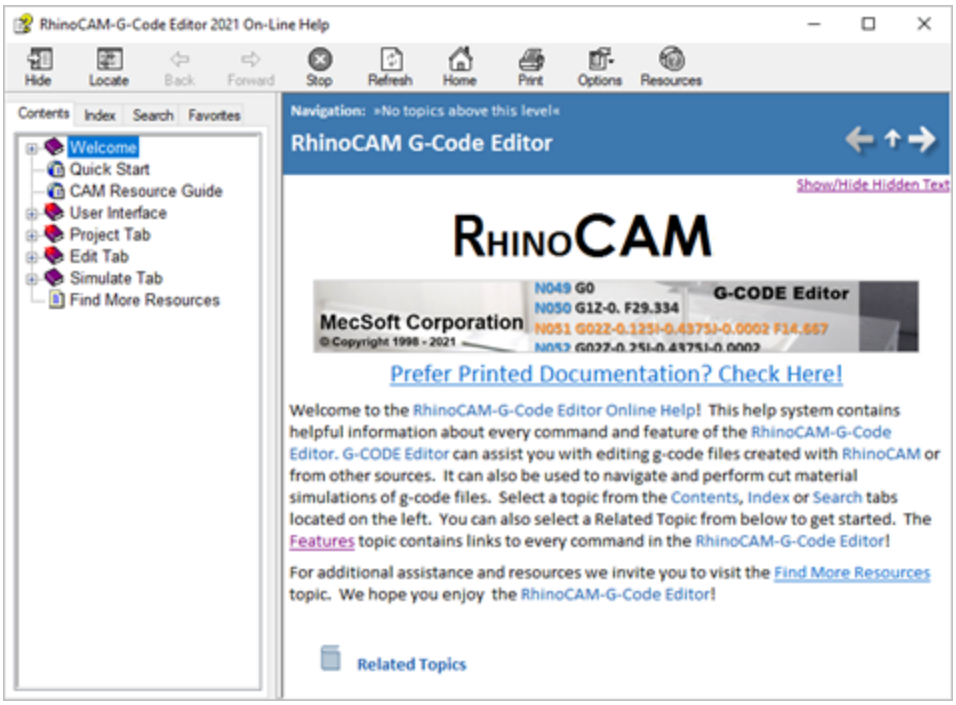
To Access this Command:

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

© 2026 MecSoft Corporation

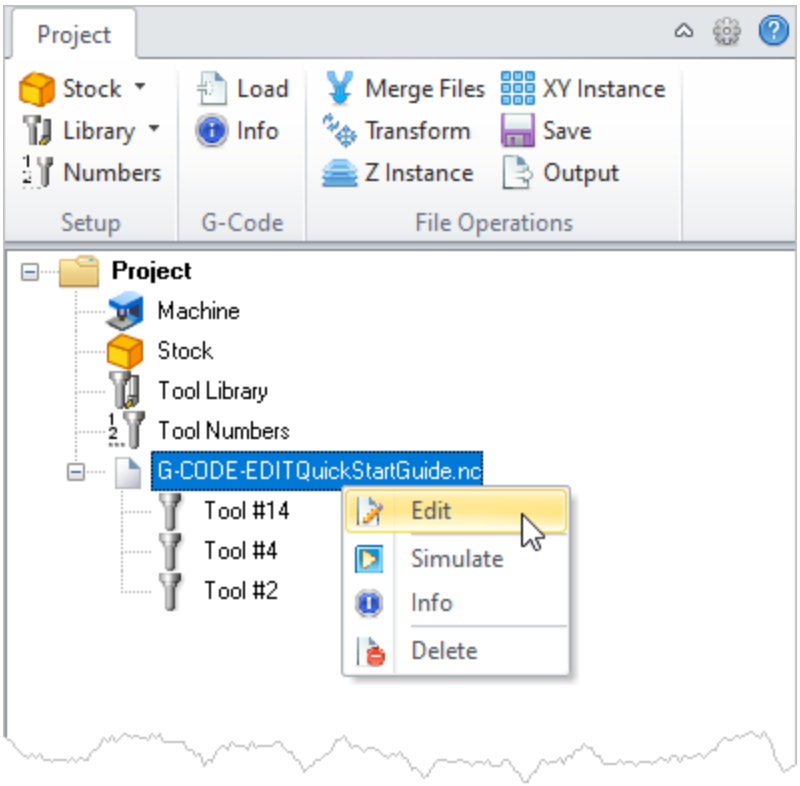
Project Tab

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: Project

The Project Tab



G-Code Browser, Project Tab

Ribbon Bar Options

Project Contains Project related commands.

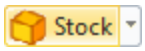
Use these icons to maximize and minimize the ribbon bar menus.

Display the Preferences dialog.

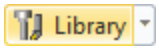
Display the Online Help system.

© 2026 MecSoft Corporation

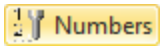
Project Setup Pane



Displays a menu of Stock definition commands.



Displays the Tool Library dialog.



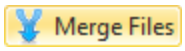
Displays the Tool Crib dialog.

Project G-Code Pane

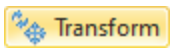
Load a g-code file into the Project tree.

Display Information about the g-code files currently loaded into the Project.

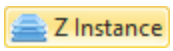
Project File Operations Pane



Merge two selected g-code files into one file.

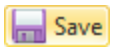


Displays the Transform g-code dialog.

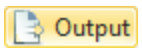


Displays the Z Instance g-code dialog.

Displays the XY Instance g-code dialog.

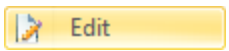


Saves all g-code files currently loaded into the Project.

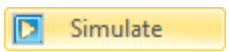


Outputs the g-code files currently loaded into the Project.

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

© 2026 MecSoft Corporation

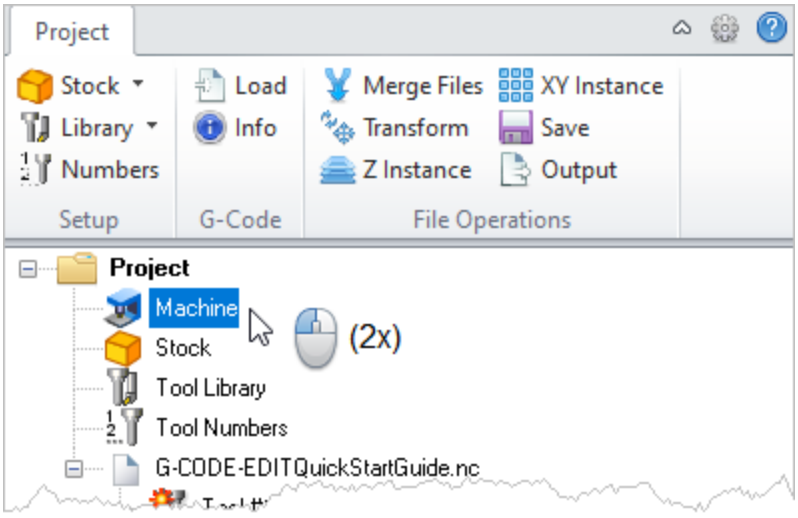
5.1 Machine MCS Setup

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:

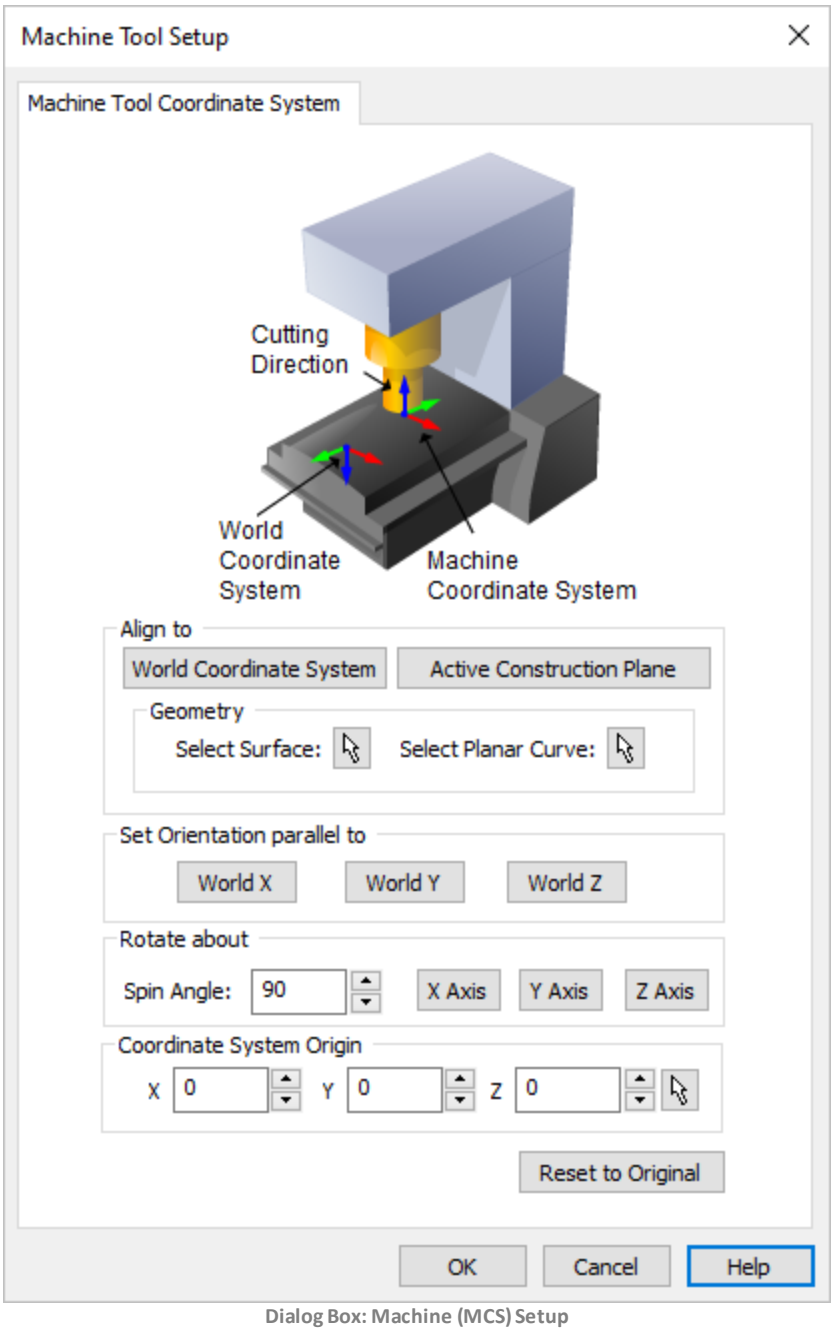
click: Project > double-click: Project / Machine

G-Code Browser: Project tab, Stock menu



Edit the Machine MCS

Dialog Box: Machine

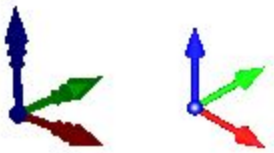


Dialog Box: Machine (MCS) Setup

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.

© 2026 MecSoft Corporation



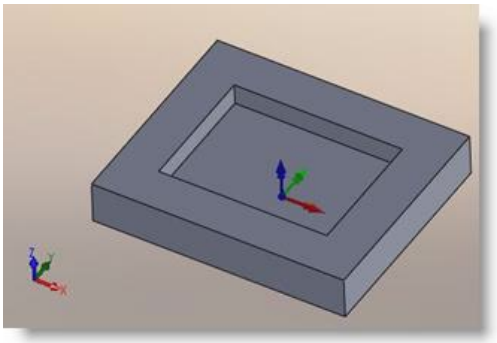
(MCS) (WCS)

Align to

This aligns the MCS orientation. Select from the following:

Machine Tool Coordinate System

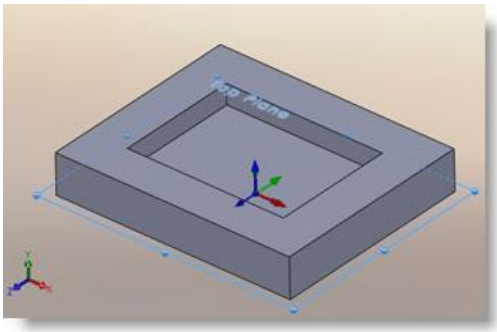
Aligns the Setup Coordinate System to Machine Tool Coordinate System.



Align To: World Coordinate System

Active View Construction Plane

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

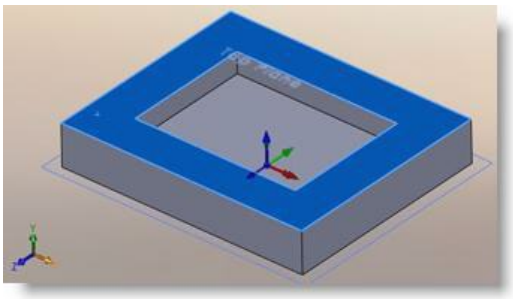


Align To: Active Construction Plane

Geometry: Part Surface

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

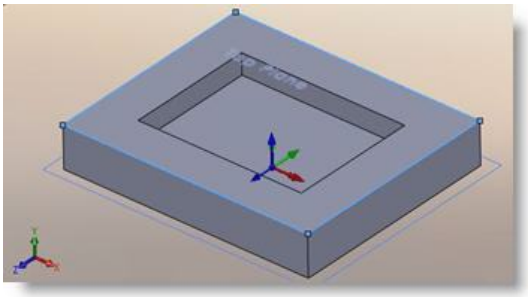
© 2026 MecSoft Corporation



Align To: Geometery, 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

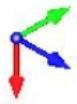
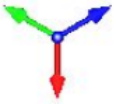
Allows you to set the Machine Coordinate System parallel to the World X Y or Z co-ordinate axis.

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.

© 2026 MecSoft Corporation

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

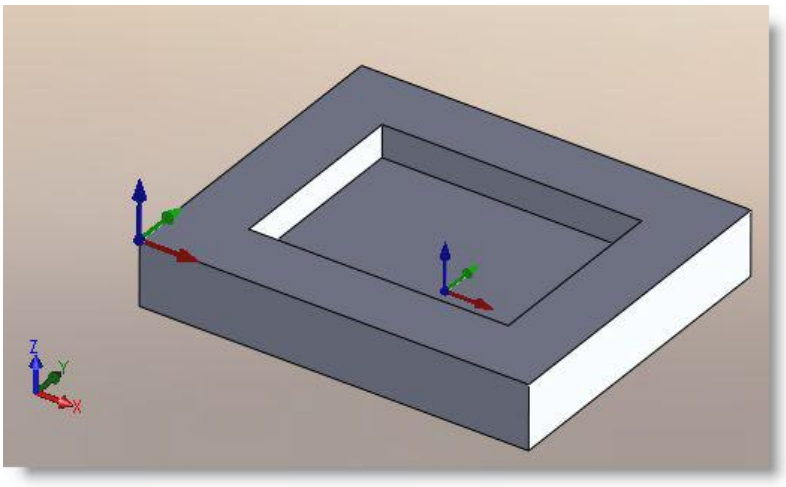
Rotate about

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.

© 2026 MecSoft Corporation



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

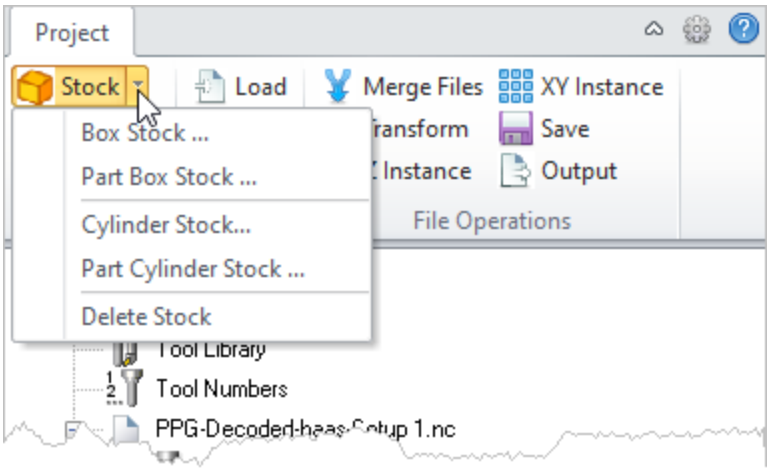
Stock This menu contains commands to create different types of Stock material in preparation for cut material Simulations. If there is no stock defined, Simulations will only show tool motions and not a cut material simulation.

To Access this Command:

click: Project > Stock

G-Code Browser: Project tab, Stock menu

© 2026 MecSoft Corporation



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

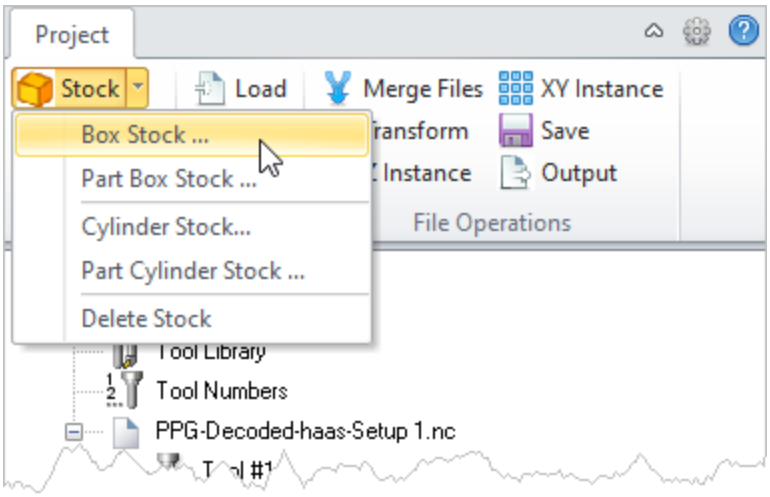
5.2.1.1 Box Stock

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: Project > Stock > Box Stock ...

Box Stock menu item

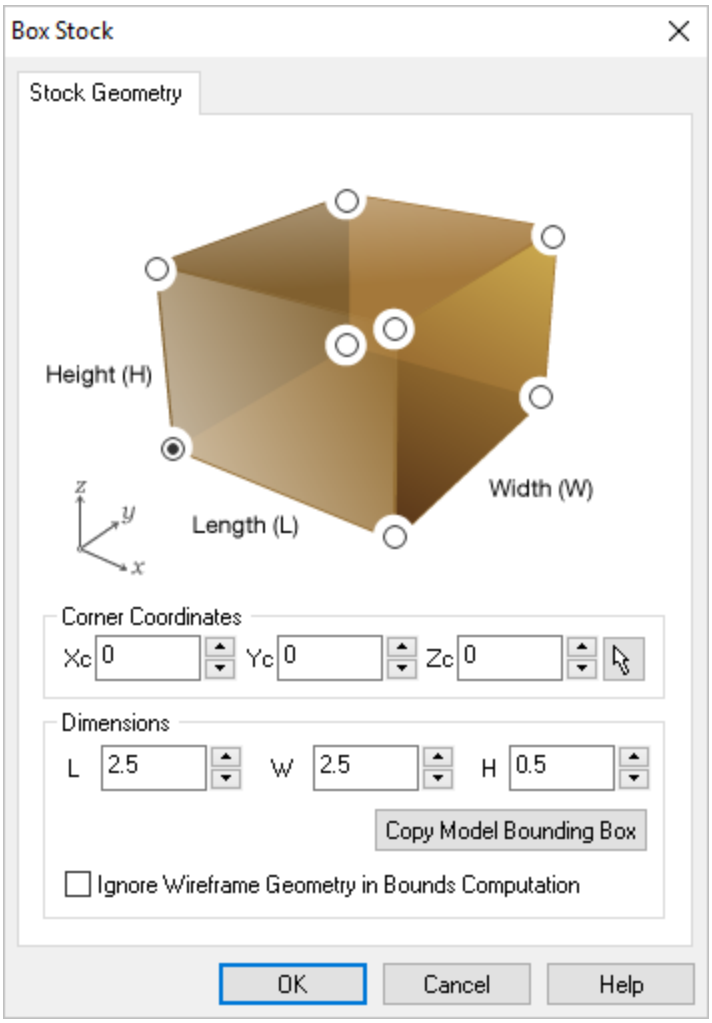


G-Code Browser: Box Stock menu item

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.

© 2026 MecSoft Corporation



Dialog Box: Box Stock

Copy Model Bounding Box

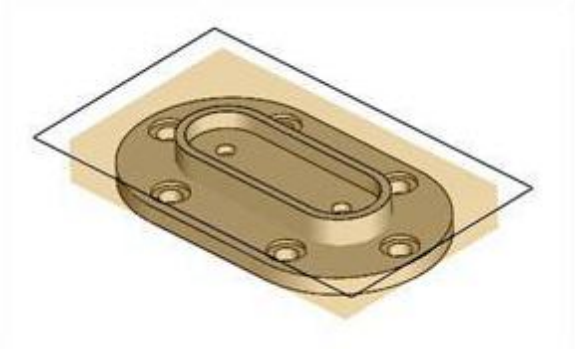
When Copy Model Bounding Box is selected, the system calculates the bounding box of the part model as the XYZ extents of geometry of the part model and displays the X Y Z Coordinate values under dimensions.

Ignore Wireframe Geometry in Bounds Computation

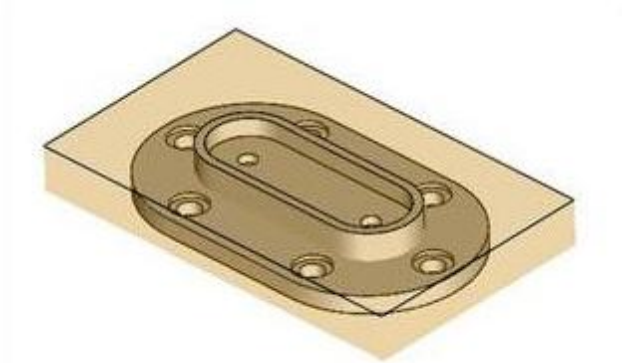
Check this box to ignores all 2D and 3D curve geometries present in the part from stock bounding box computation.

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

© 2026 MecSoft Corporation



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

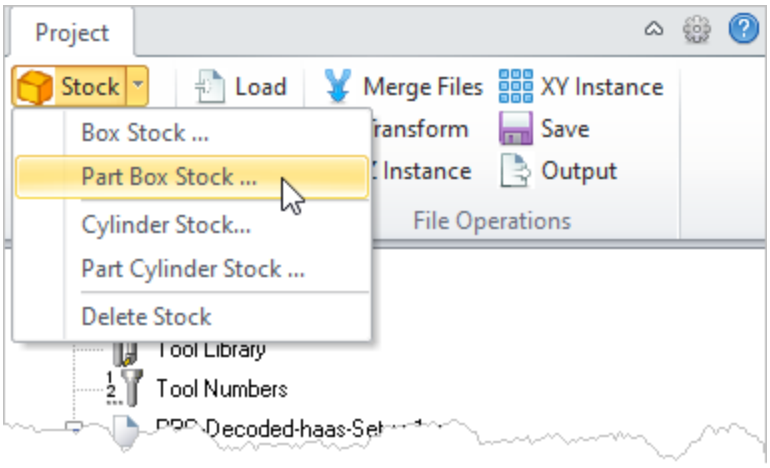
Stock You can define the raw stock model as a bounding box of the XYZ extents of geometry of the part model. You can then define offsets in any of the three coordinate directions to apply to the computed bounding box.

To Access this Command:

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

Part Box Stock menu item

© 2026 MecSoft Corporation

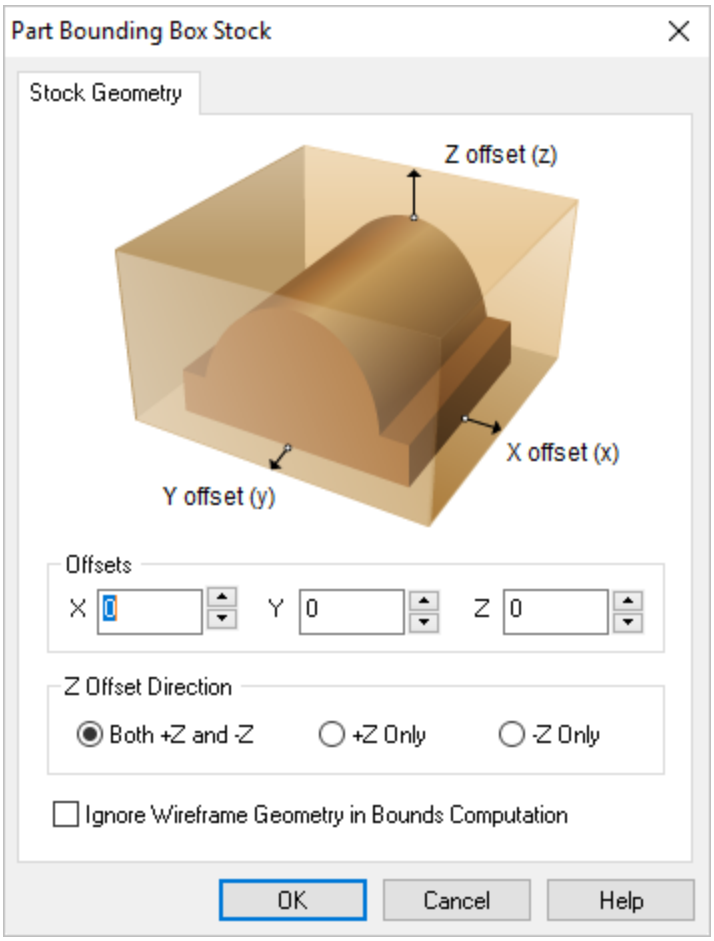


G-Code Browser: Part Box Stock menu item

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.

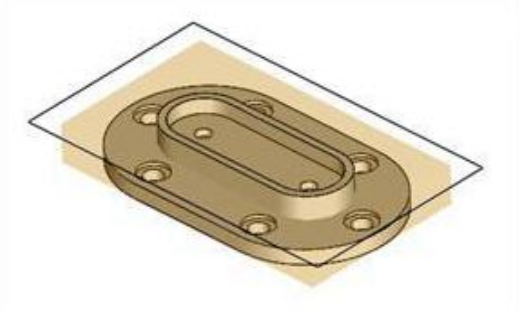
© 2026 MecSoft Corporation



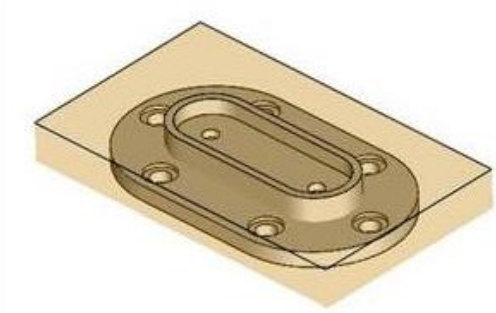
Dialog Box: Part Box Stock

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

© 2026 MecSoft Corporation

Create Box Stock

Create Cylinder Stock

Create Part Cylinder Stock

Stock from Selection

Delete Stock

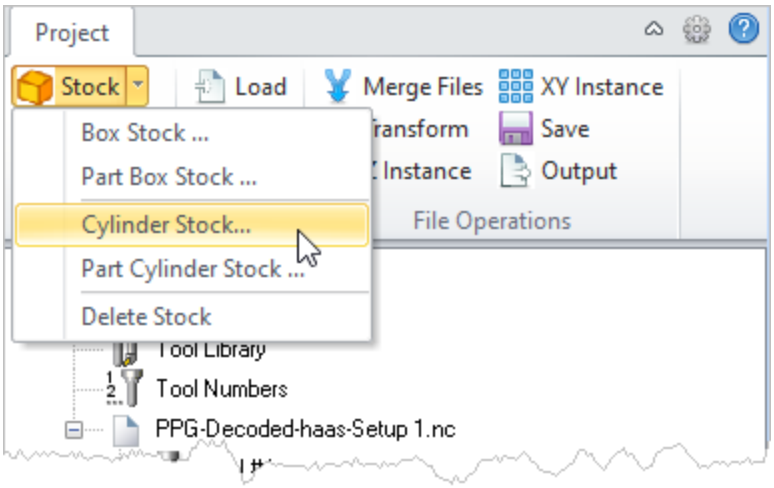
5.2.1.3 Cylinder Stock

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: Project > Stock > Cylinder Stock ...

Cylinder Stock menu item



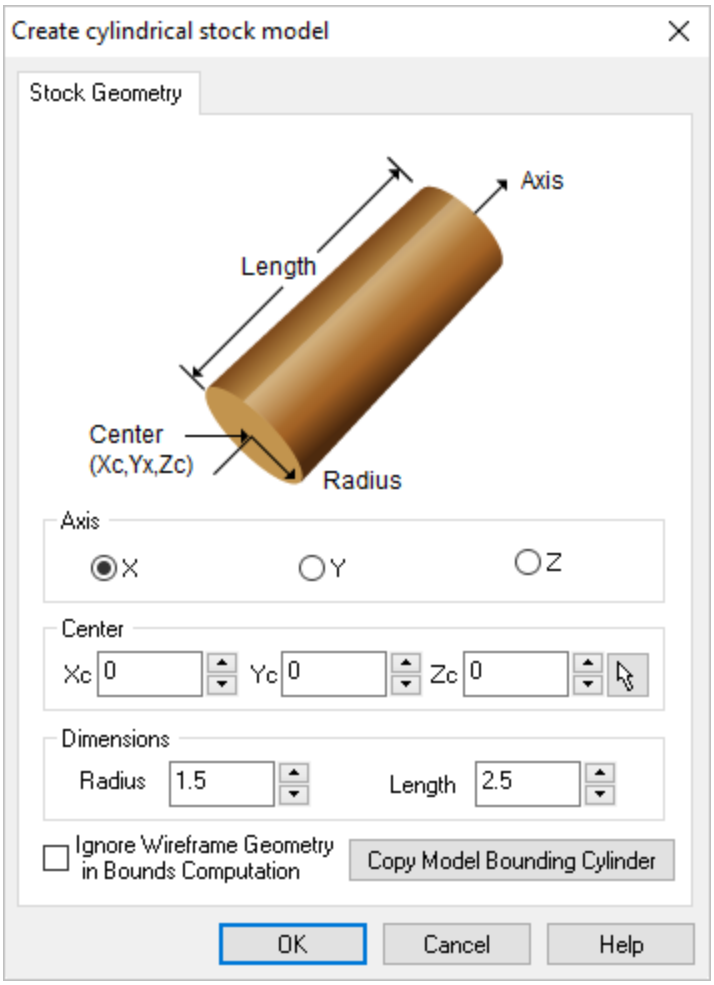
G-Code Browser: CylinderStock menu item

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.

© 2026 MecSoft Corporation



Dialog Box: Cylinder Stock

Copy Model Bounding Cylinder

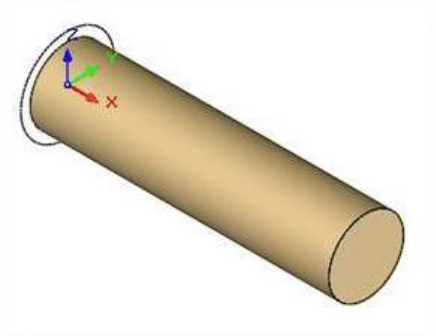
User can also select Copy Model Bounding Cylinder to determine the smaller possible stock model that fit the part geometry.

Ignore Wireframe Geometry in Bounds Computation

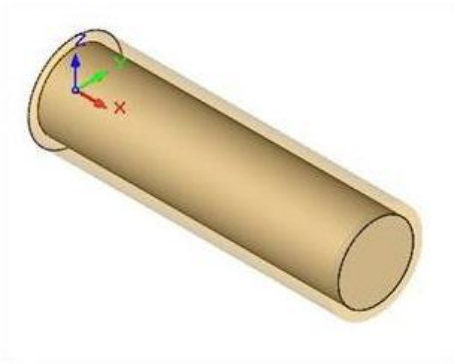
Check this ignore all 2D and 3D curve geometries present in the part from stock bounding cylinder computation.

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

© 2026 MecSoft Corporation



Ignore Wireframe geometry - checked



Ignore Wireframe geometry – un-checked

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

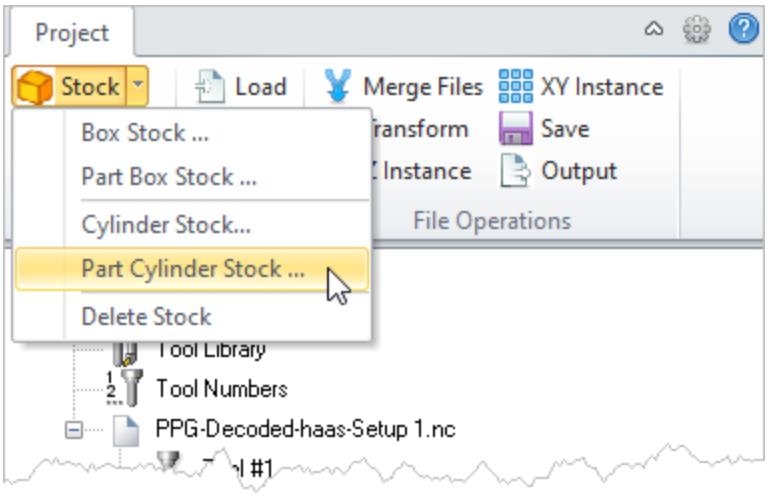
Stock You can define the raw stock model as a bounding cylinder of the XYZ extents of geometry of the part model. You can then define offsets to apply to the computed bounding box.

To Access this Command:

click: Project > Stock > Part Cylinder Stock ...

Part Cylinder Stock menu item

© 2026 MecSoft Corporation



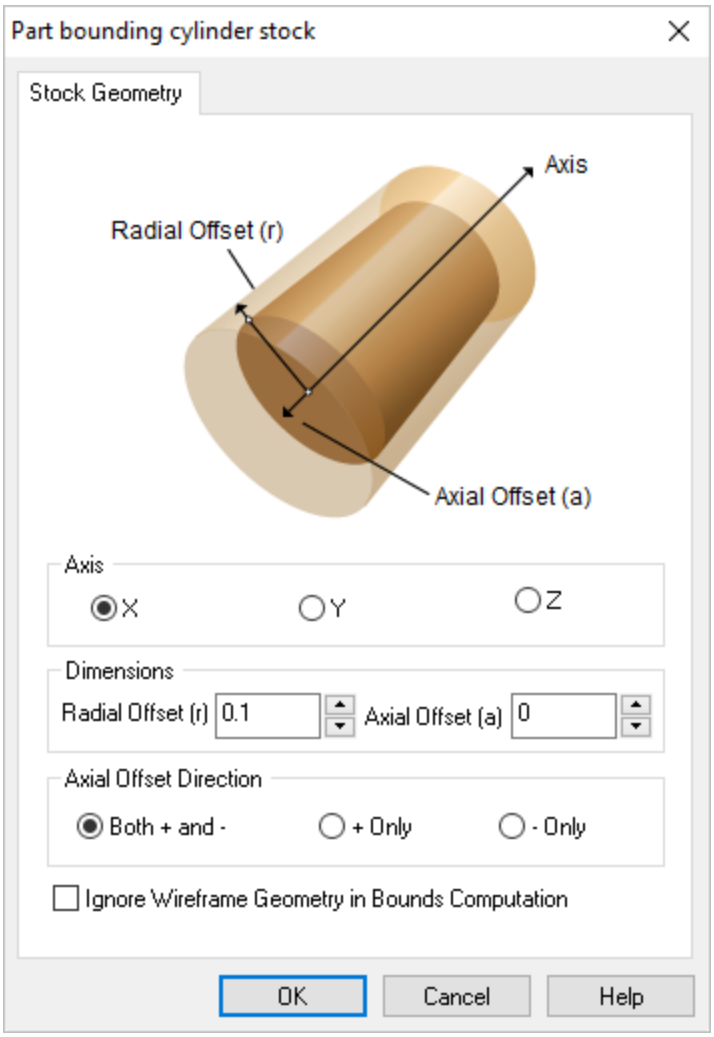
G-Code Browser: Part CylinderStock menu item

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.

© 2026 MecSoft Corporation



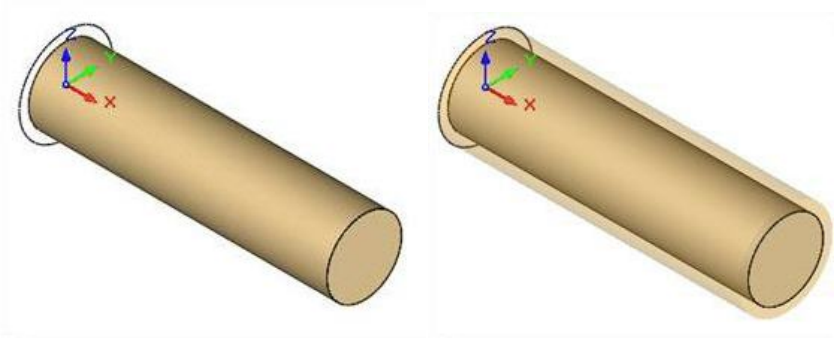
Dialog Box: Part Cylinder Stock

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.

© 2026 MecSoft Corporation



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

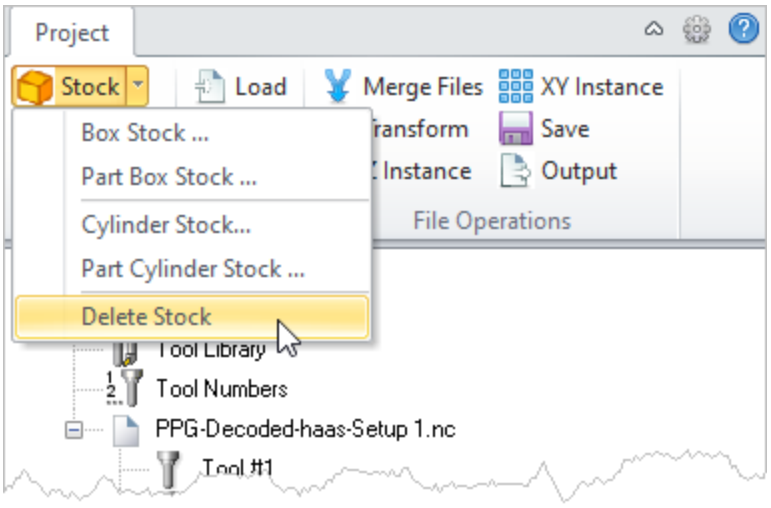
Stock Use this command to Delete the currently defined Stock model. If there is no stock defined, Simulations will only show tool motions and not a cut material simulation.

To Access this Command:

click: Project > Stock > Delete Stock

Delete Stock menu item

© 2026 MecSoft Corporation



G-Code Browser: Delete Stock menu item

Related Topics

Stock Menu

Create Box Stock

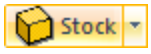
Create Part Box Stock

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 selects 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.

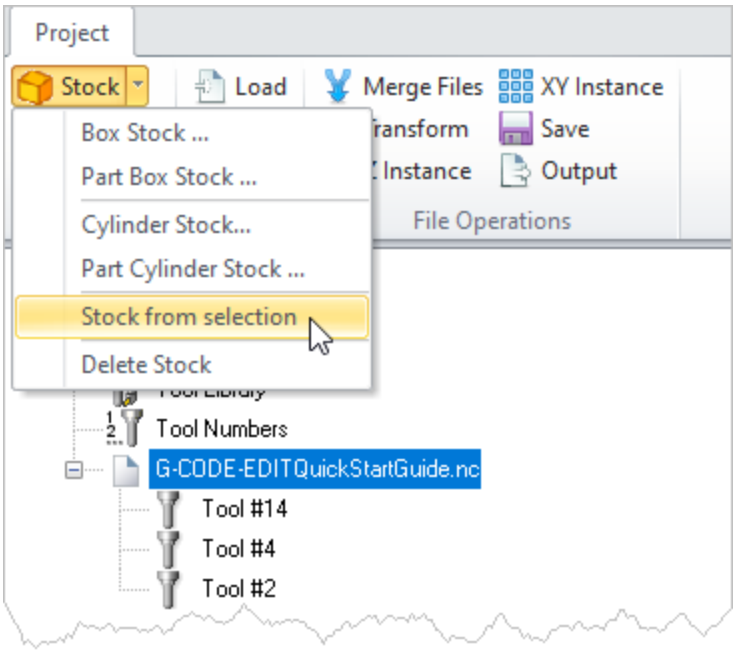
> Stock from Selection will fail to create stock if the selected 3D geometry is not a watertight model.

To Access this Command:

click: Project > Stock > Stock from Selection

Stock from Selection menu item

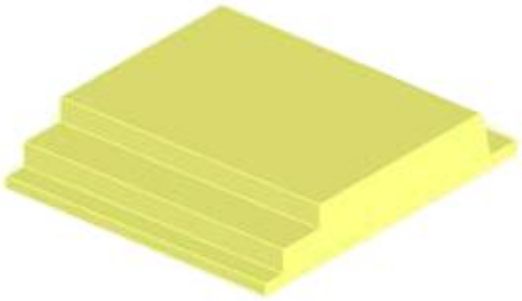
© 2026 MecSoft Corporation



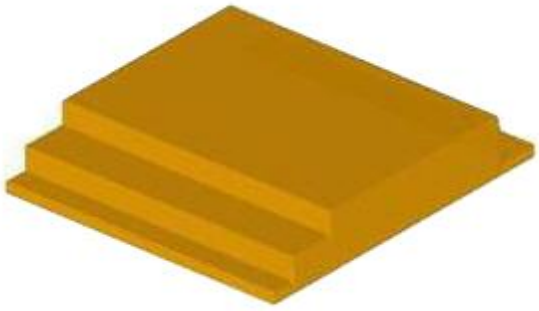
G-Code Browser: Stock from Selection menu item

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

© 2026 MecSoft Corporation

Stock Menu

Create Box Stock

Create Part Box Stock

Create Cylinder Stock

Create Part Cylinder Stock

Delete Stock

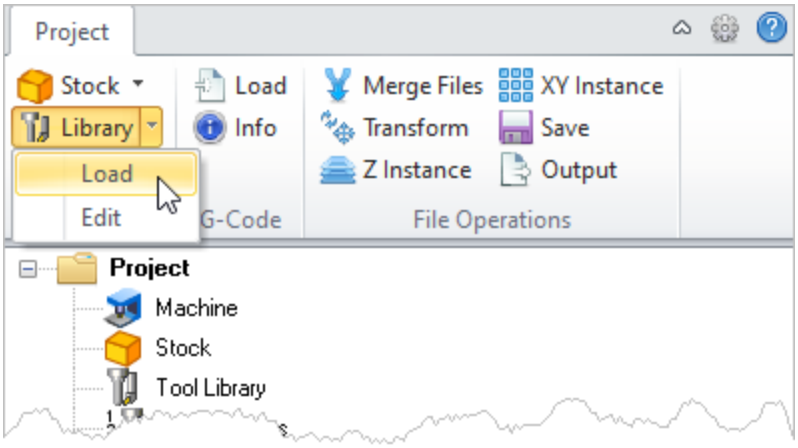
5.2.2 Tool Library

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: Project > Library > Load

Tool Library menu item

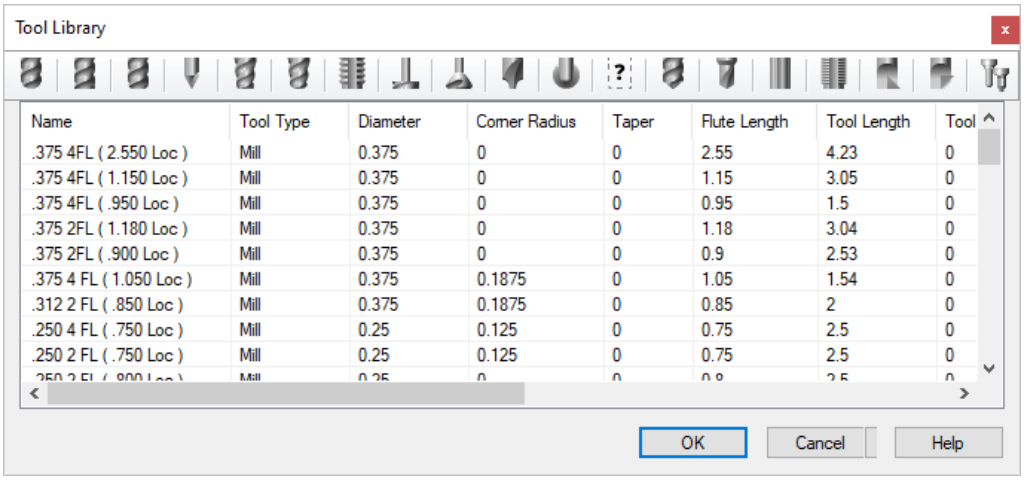


G-Code Browser: Tool Library menu item

Procedure

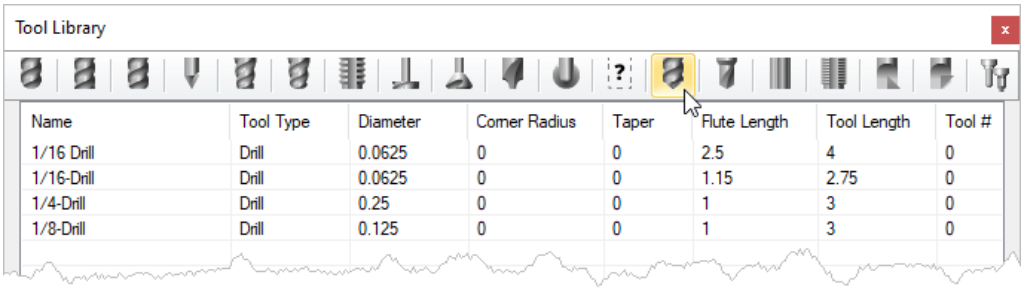
1. Select Library from the Project tab of the G-Code Browser and pick Load.
2. The File Open dialog will display. Navigate to the Tool Library file and pick Open to load it. The file extension must be *.csv. You can save a Tool Library (*.csv) file from the MILL module Save Tool Library command.
3. The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

© 2026 MecSoft Corporation



A Tool Library is loaded

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



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.Displays the Online Help system.

© 2026 MecSoft Corporation

Ball

Flat

Corner Rad

Vee

Chamfer

Taper

Thread

Face

Dove Tail

Fillet

Lollypop

User Defined

Drill

Center Drill

Ream

Tap

Bore

RBore

Show All

Related Topics

© 2026 MecSoft Corporation

User Interface

Project Tab

Stock Menu

Tool Crib

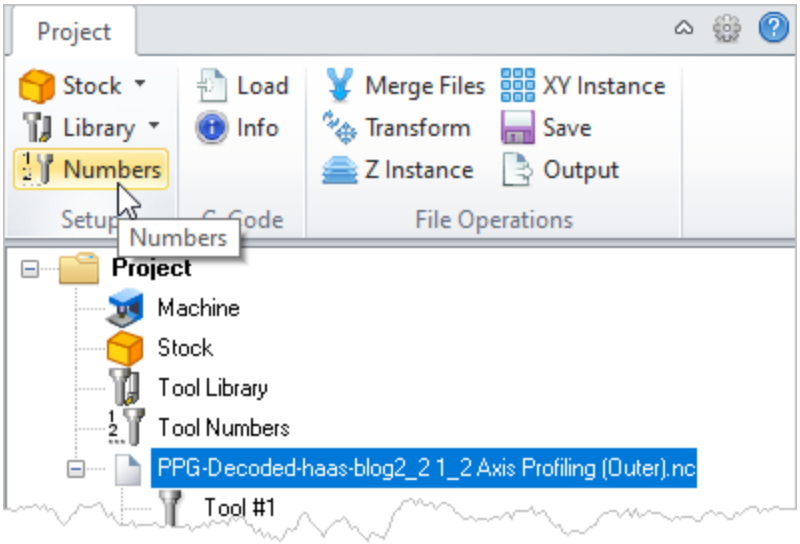
5.2.3 Tool Crib

Numbers 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: Project > Numbers

Tool Crib menu item

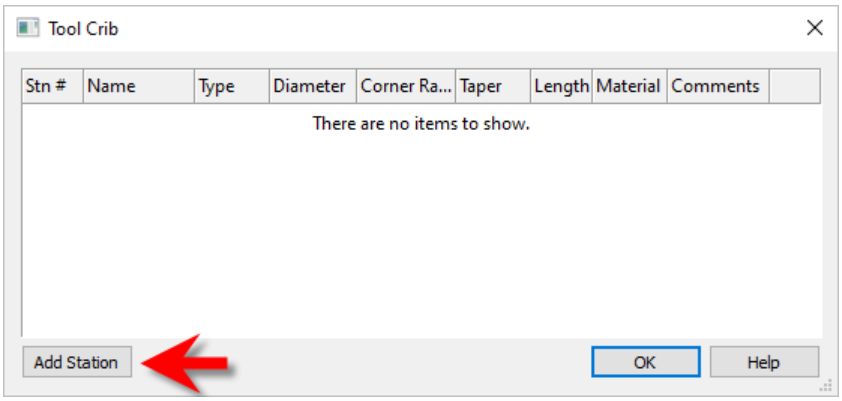


G-Code Browser: Tool Crib menu item

Procedure

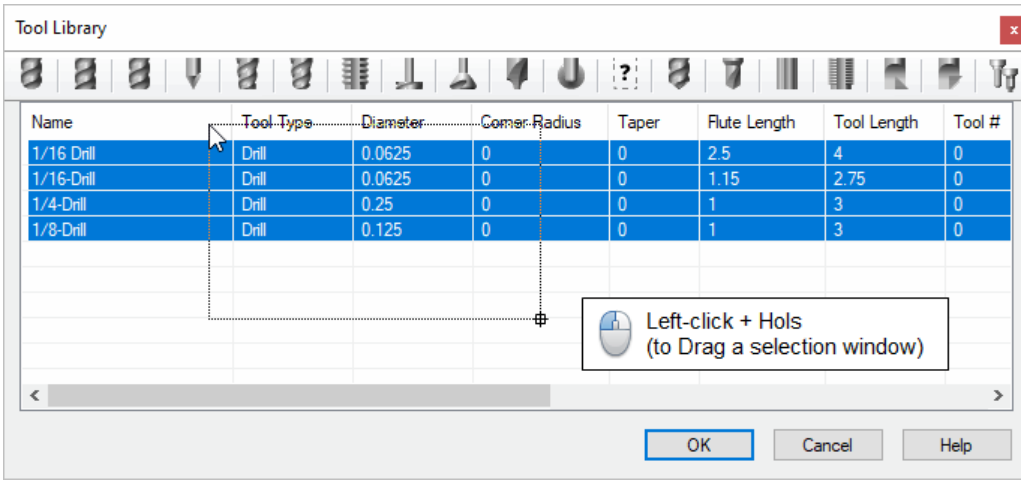
1. Select Numbers from the Project tab of the G-Code Browser.
2. The Tool Crib dialog will display.
   Select the Add Station button to grab a tool from the default Tool Library.

© 2026 MecSoft Corporation



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.

4. 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

© 2026 MecSoft Corporation

5.3 G-Code Pane

5.3.1 Load

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

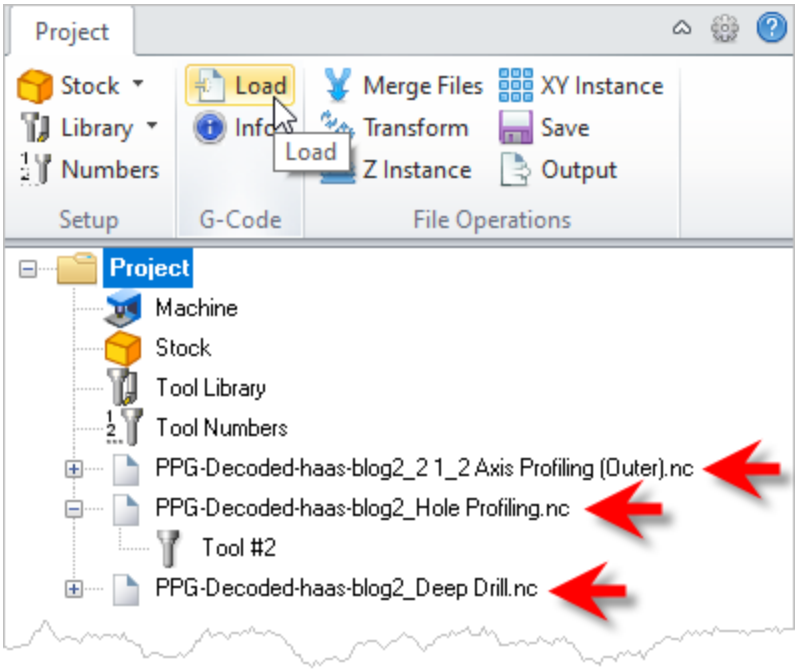
Then use the File Browser to select a G-Code file to load.

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

© 2026 MecSoft Corporation

Stock Menu

Tool Library

Tool Crib

Info

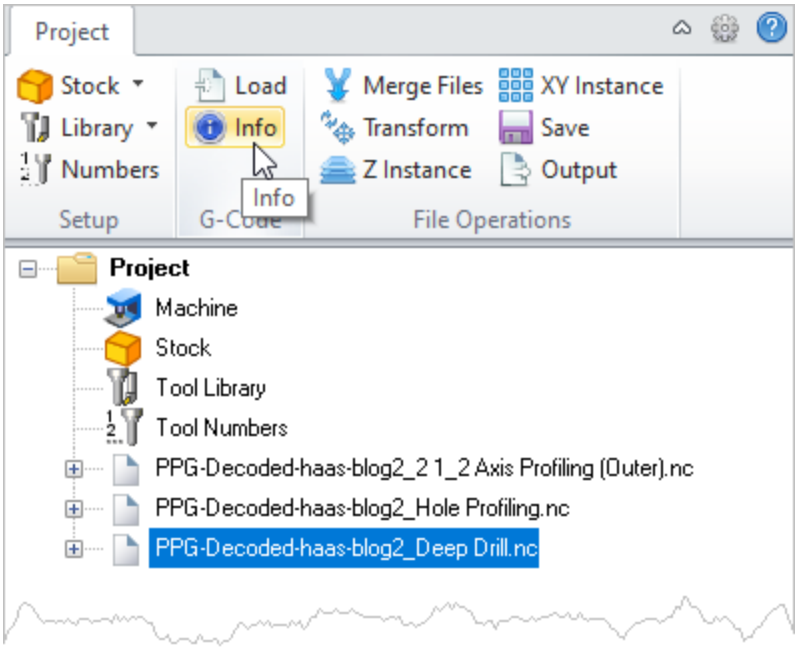
5.3.2 Info

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

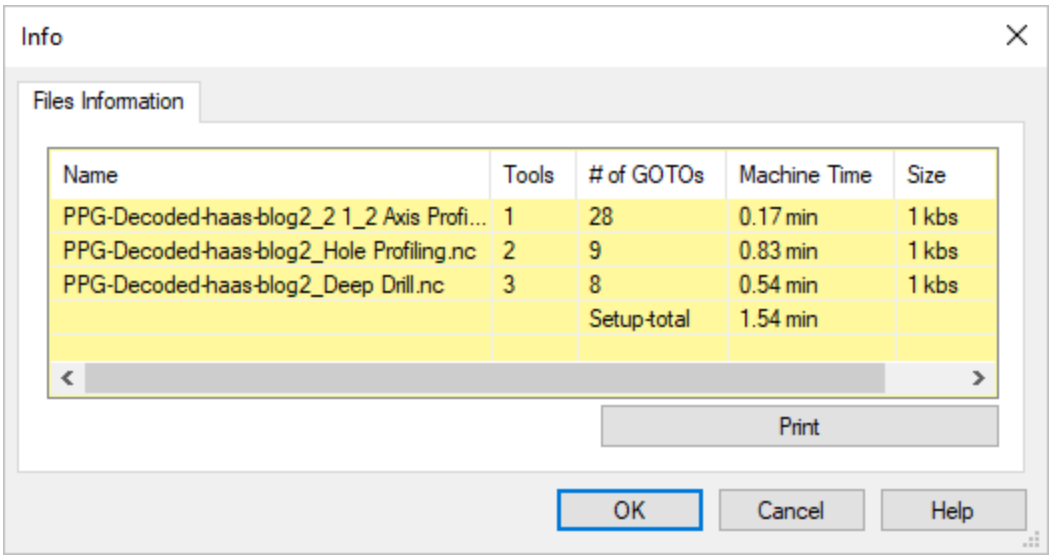
Info menu item



G-Code Browser: Info menu item

G-Code Files Information Dialog

© 2026 MecSoft Corporation



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

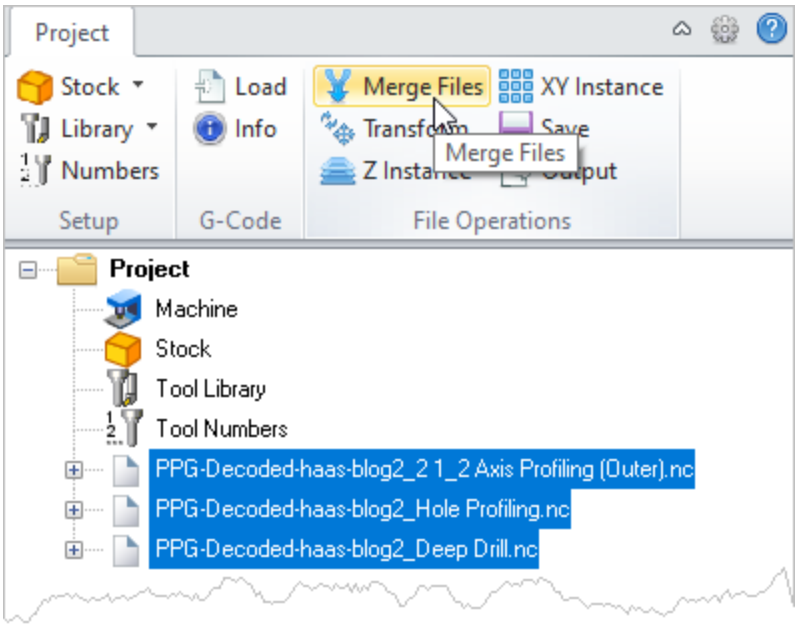
Merge Files Use this command to merge g-code files that are currently loaded into the Project tab of the G-Code Browser. Merged files can be output as one g-code file using the Output command.

To Access this Command:

click: Project > Merge Files

Merge Files menu item

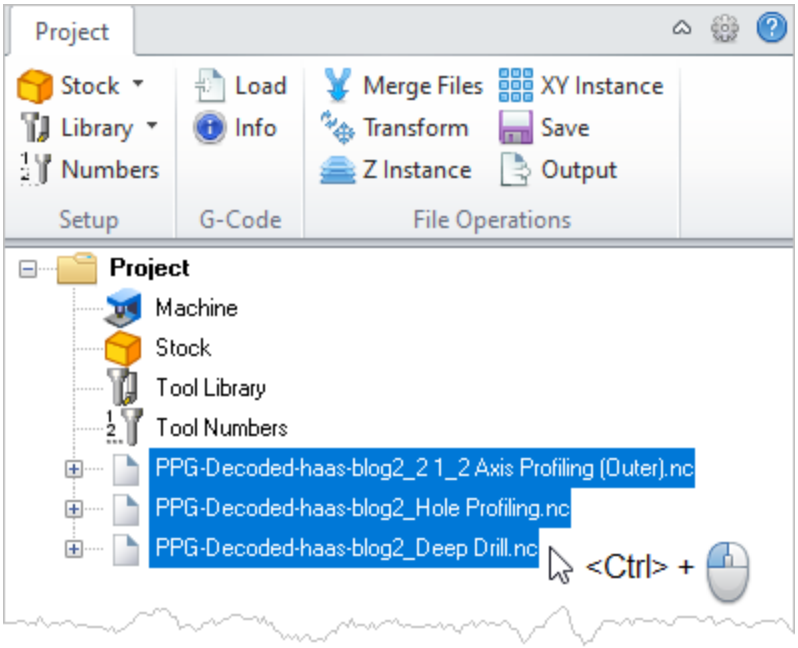
© 2026 MecSoft Corporation



G-Code Browser: Merge Files menu item

Procedure

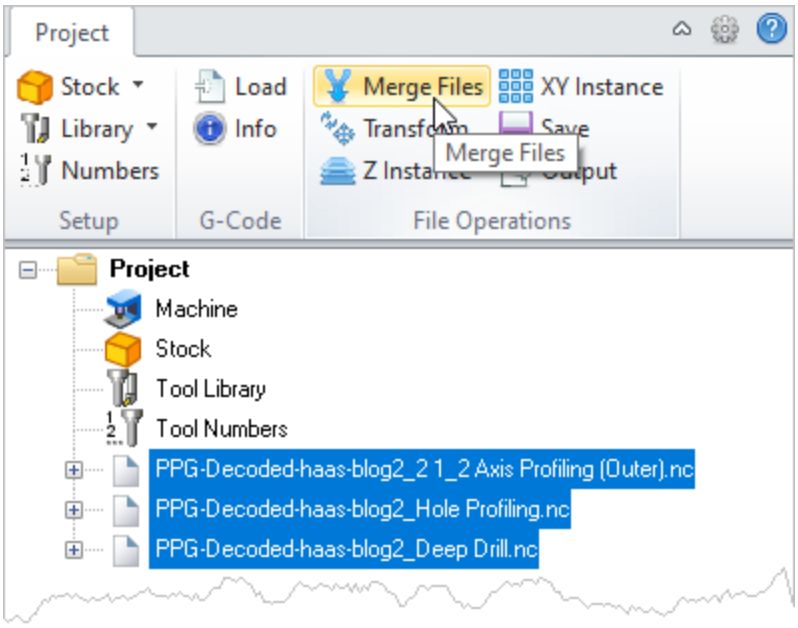
1. From the Project tab of the G-Code Browser, press and hold the <Ctrl> key while left-click selecting the g-code files that you want to merge.



G-Code Browser: Merge Files menu item

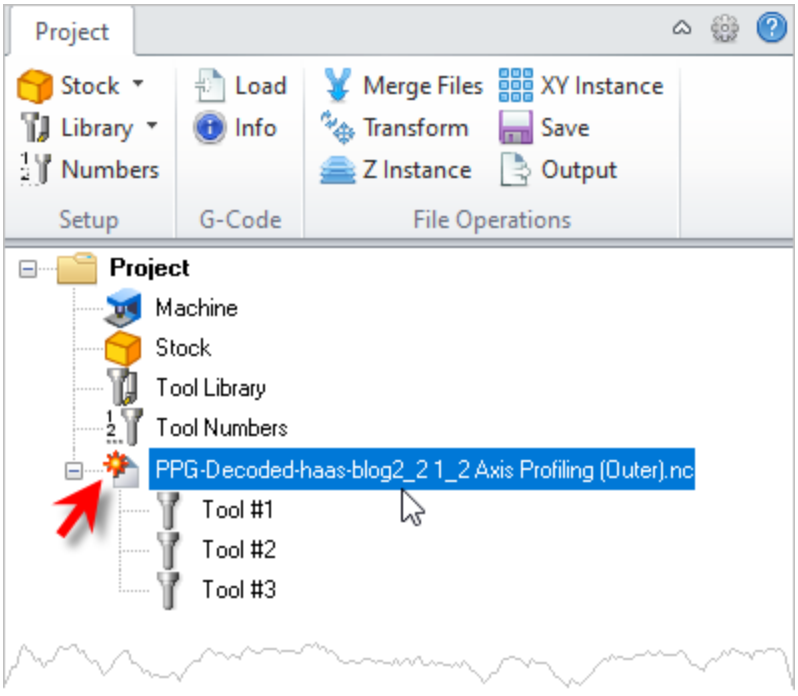
2. With the g-code files highlighted and selected, pick the Merge Files command from the Project tab.

© 2026 MecSoft Corporation



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.

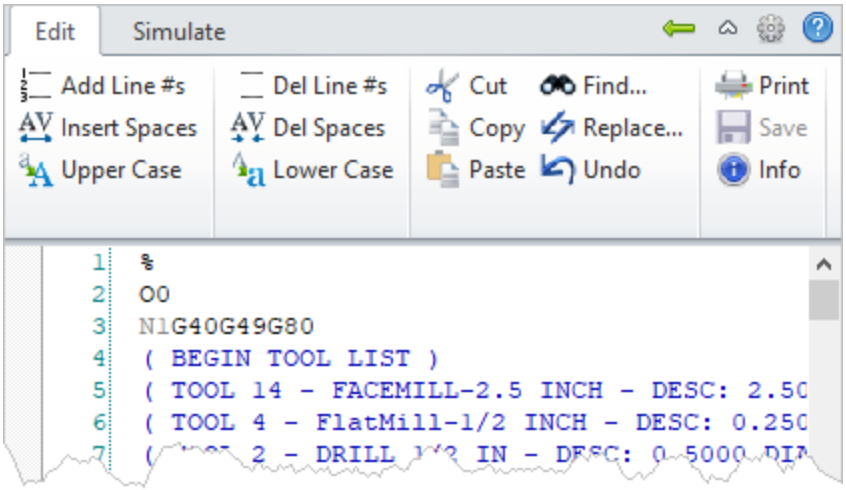


The second and third files are merged into the first selected file that appears in the Project.

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.

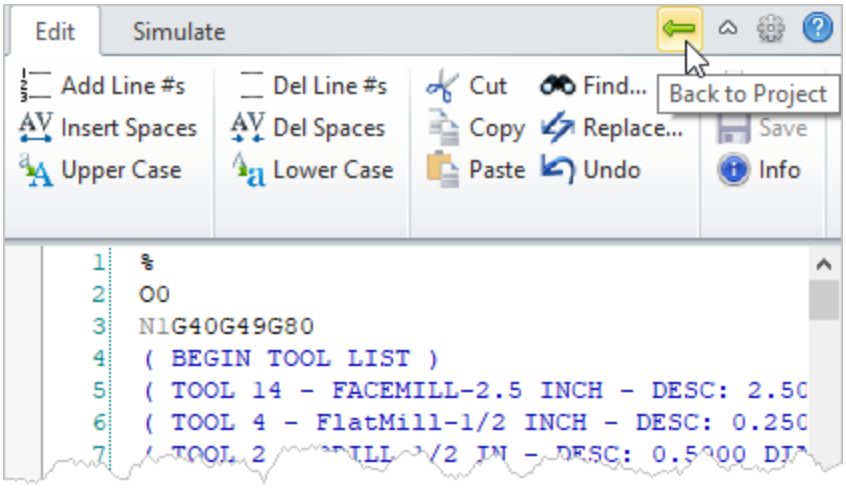
© 2026 MecSoft Corporation

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

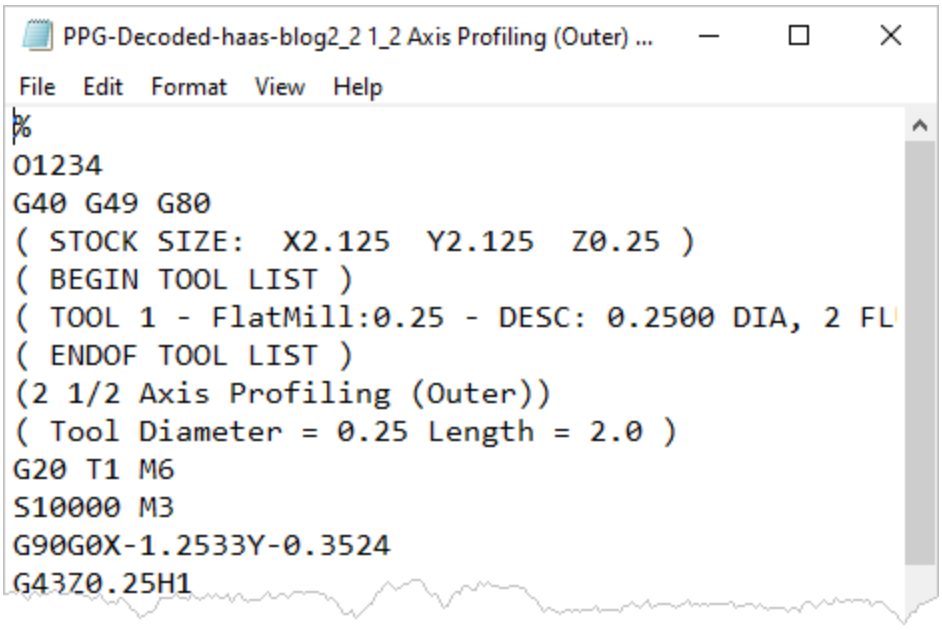
6. When done editing the g-code file, select the Back to Project icon to return to the Project tab.



Back to Project menu item

7. When satisfied, select the Output icon to save the new edited g-code file.

© 2026 MecSoft Corporation



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

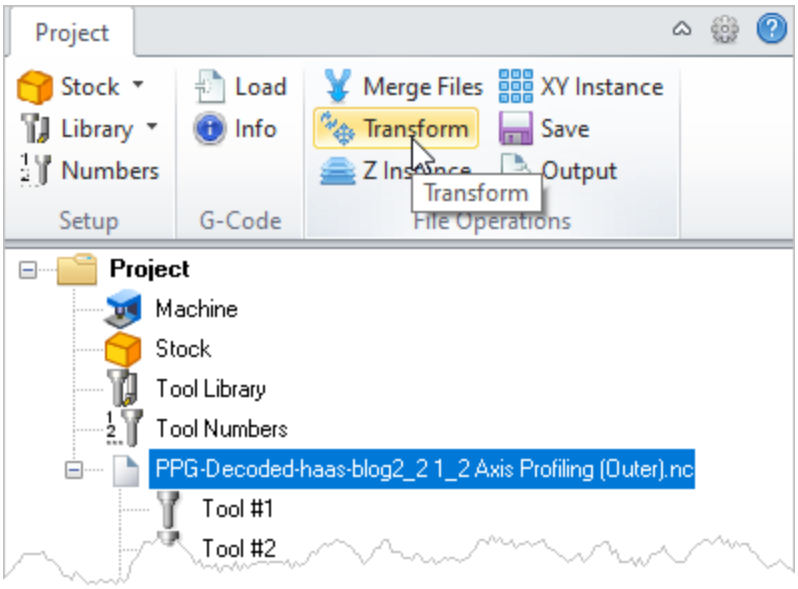
Transform You can use this command to perform transformations on a selected g-code file. The transformations include Move, Rotate, Mirror and Scale. Each method is documented in the topics below. Each tab of the Transform dialog below includes an example.

To Access this Command:

click: Project > Transform

© 2026 MecSoft Corporation

Transform menu item



G-Code Browser: Transform menu item

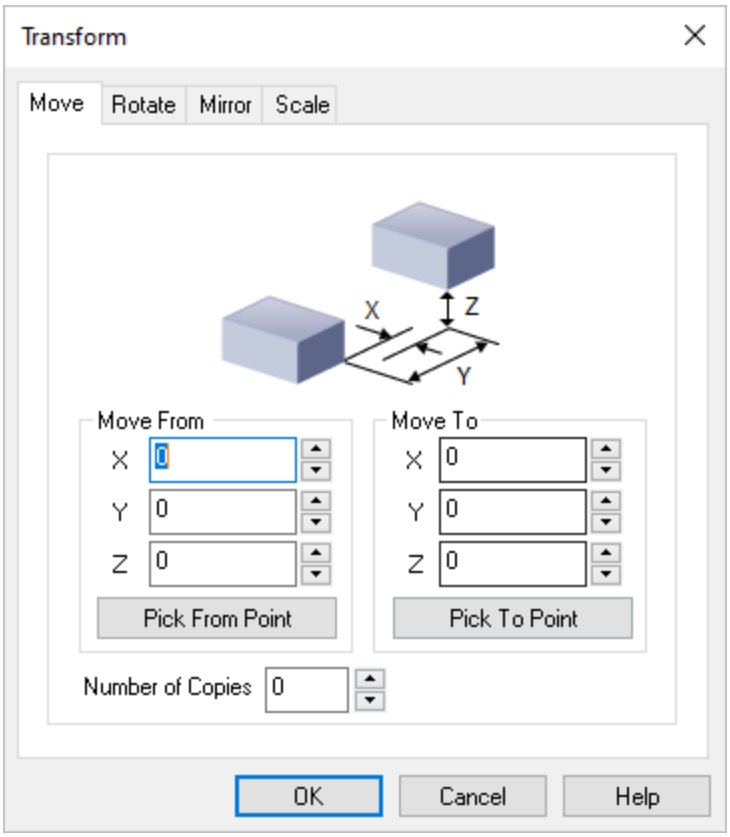
Procedure

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 tap 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.
5. 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.

© 2026 MecSoft Corporation



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

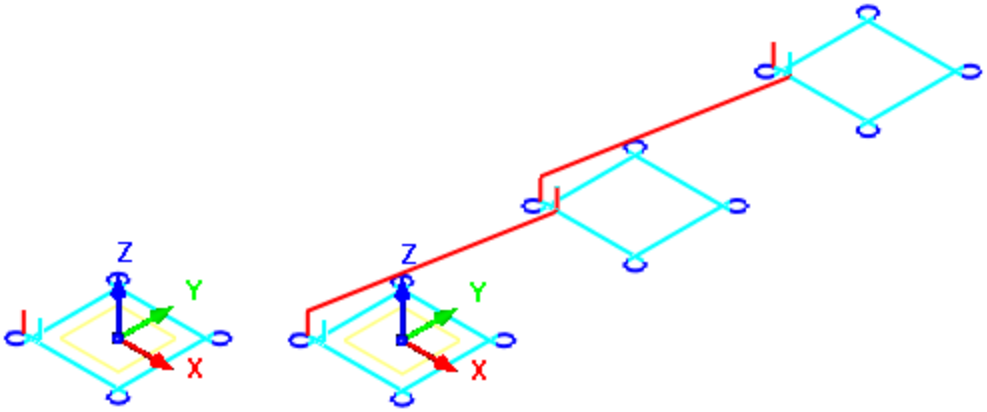
Enter the X,Y,Z coordinate values for the Move To point or select the Pick To Part button to select a point from the model.

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.

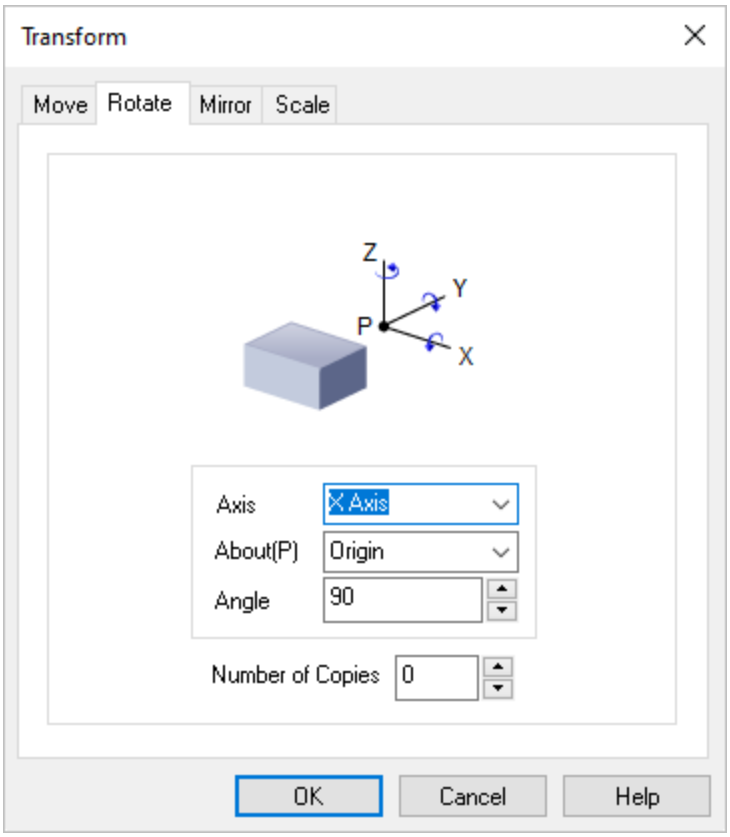
© 2026 MecSoft Corporation



Original G-Code Transfored (Moved with 2 copies) G-Code

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.

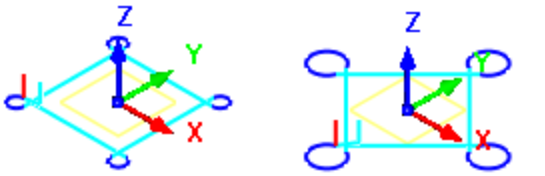
© 2026 MecSoft Corporation

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.

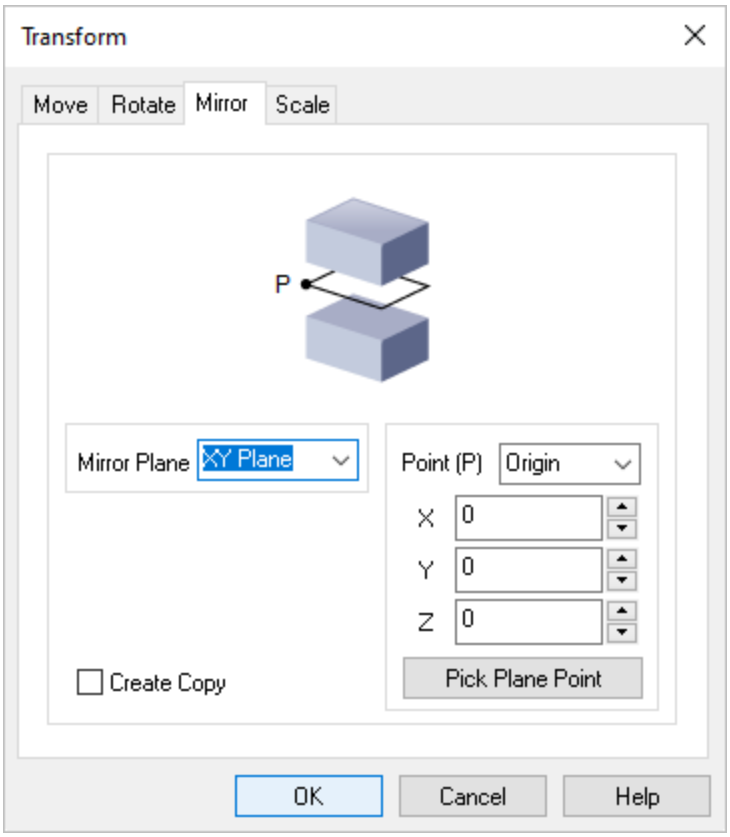
Example: The original g-code file is rotated about the Z axis by 45 degrees.



Original G-Code Transfored (Rotated) G-Code

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)

© 2026 MecSoft Corporation

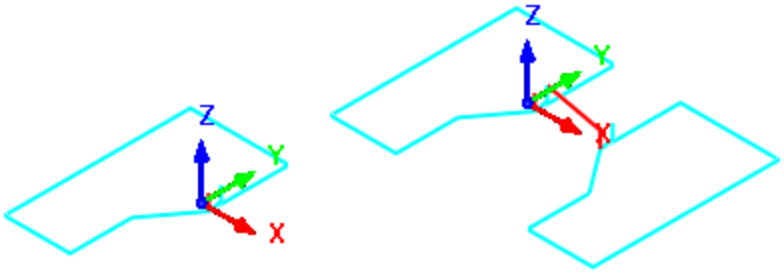
Select a Mirror Point (P) from the list (Origin, Center of the part, Minimum point of the part) or enter the X,Y,Z coordinate values of the desired point.

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.

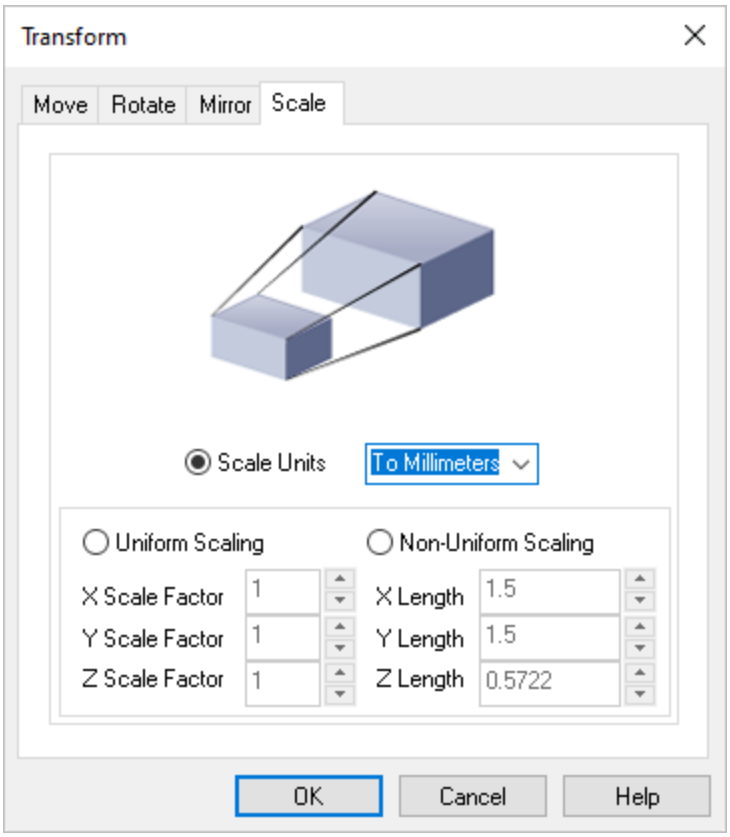


Original G-Code Transfored (Mirrored) G-Code

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.

© 2026 MecSoft Corporation



Dialog Box: Transform Toolpath, Scale tab

Scale Units

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

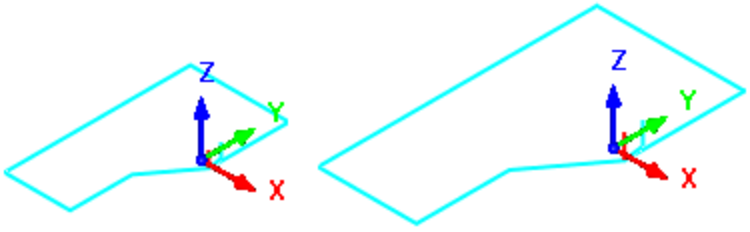
Uniform Scaling

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.



Original G-Code Transfored (Scaled) G-Code

Related Topics

User Interface

Project Tab

© 2026 MecSoft Corporation

Load

Info

Merge Files

Z Instance

XY Instance

Save

Output

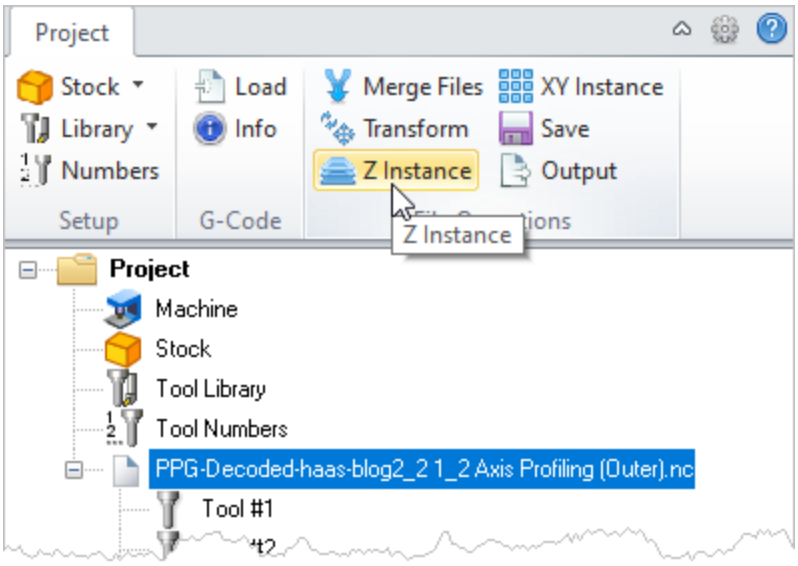
5.4.3 Z Instance

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:

Project > Z Instance

Z Instance menu item



G-Code Browser: Z Instance menu item

Procedure

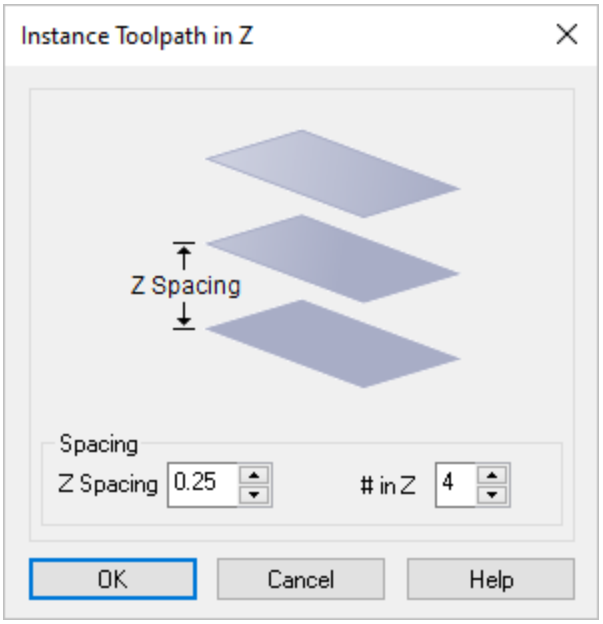
1. Select the g-code file from the Project tab that you wish to Z Instance.
2. Select Z Instance from the Project tab menu. The Z Instance dialog will display.

© 2026 MecSoft Corporation

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

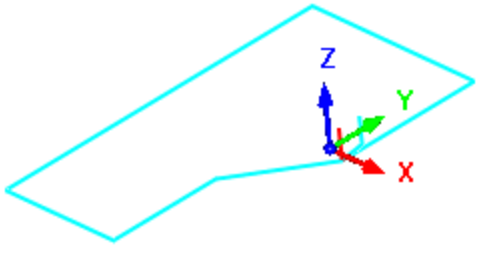
Spacing

Specify the Spacing between the toolpaths in the Z direction.

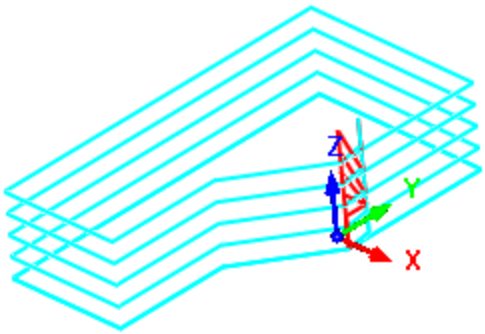
in Z

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.



Original G-Code



Z Instanced 4copies of G-Code

Related Topics

© 2026 MecSoft Corporation

User Interface
Project Tab
Load
Info
Merge Files
Transform
XY Instance
Save
Output

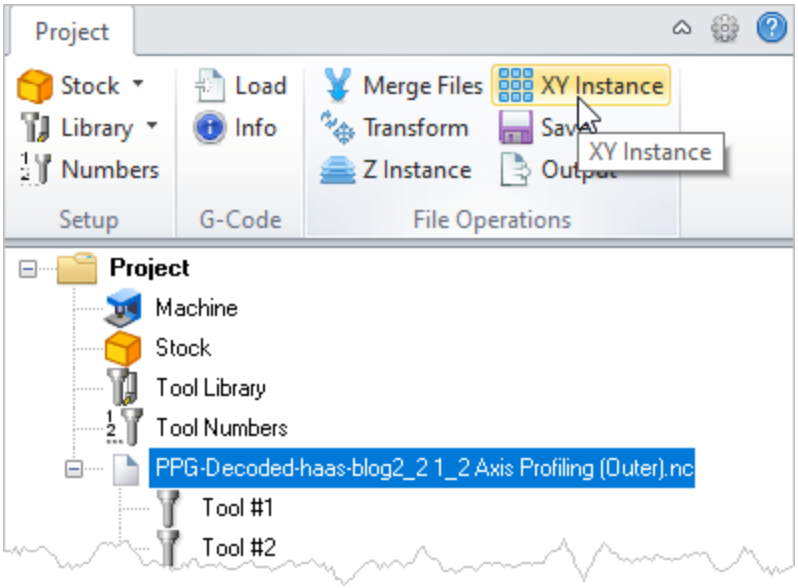
5.4.4 XY Instance

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

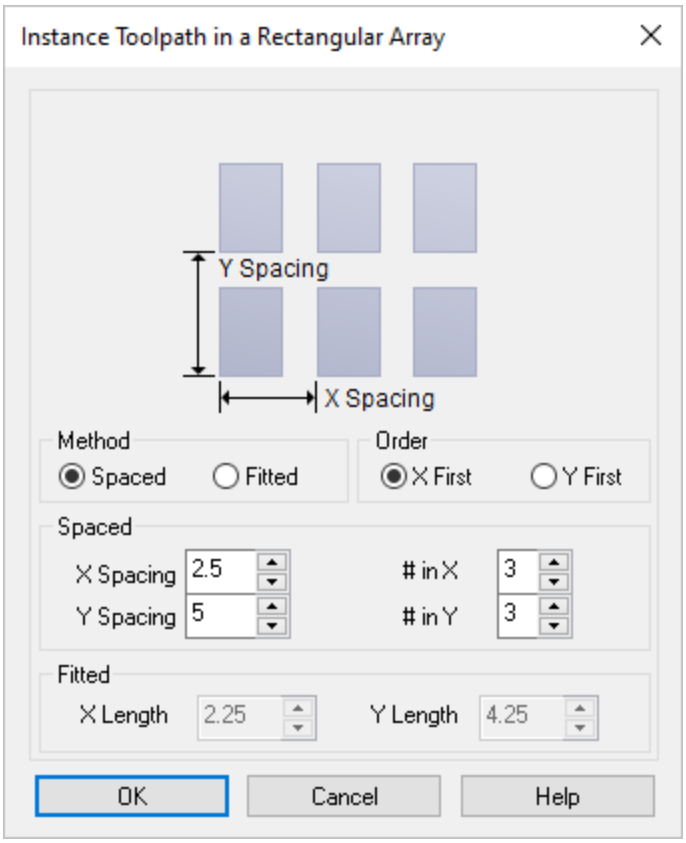
Procedure

© 2026 MecSoft Corporation

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

© 2026 MecSoft Corporation

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

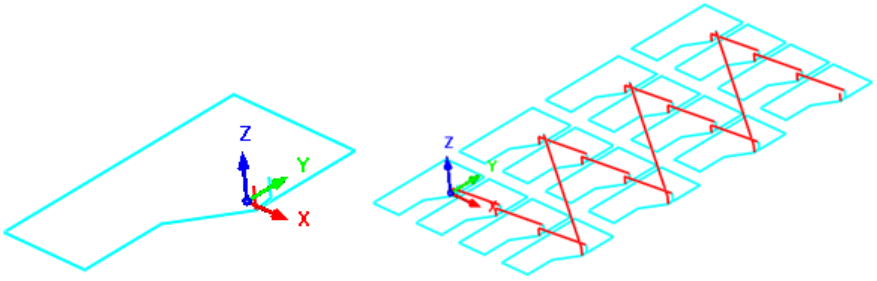
The bounding box of the part geometry is considered when computing the number of toolpaths that can be fitted in the specified space.

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.



Original G-Code XY Instance of G-Code

Related Topics

User Interface

Project Tab

Load

Info

Merge Files

Transform

Z Instance

Save

Output

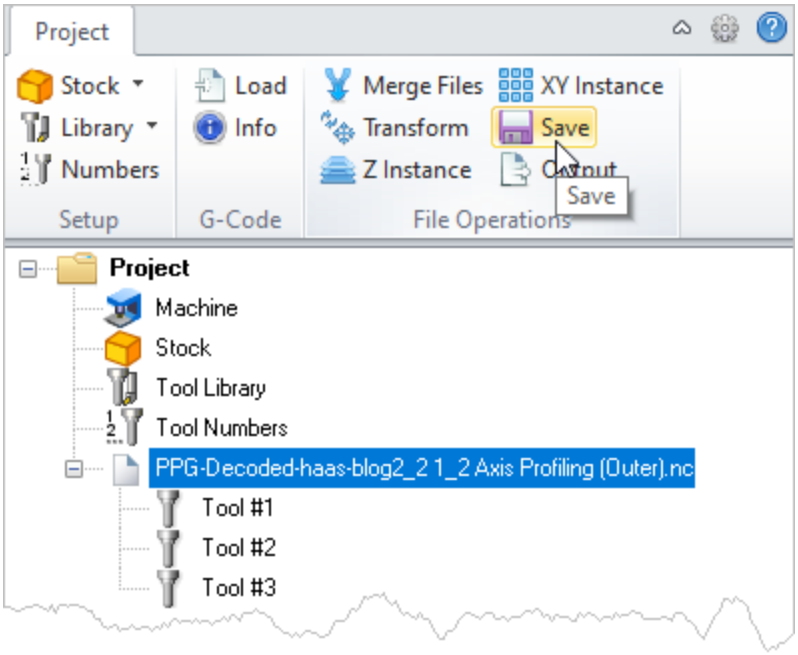
5.4.5 Save

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:

click: Project > Save

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

© 2026 MecSoft Corporation

Output

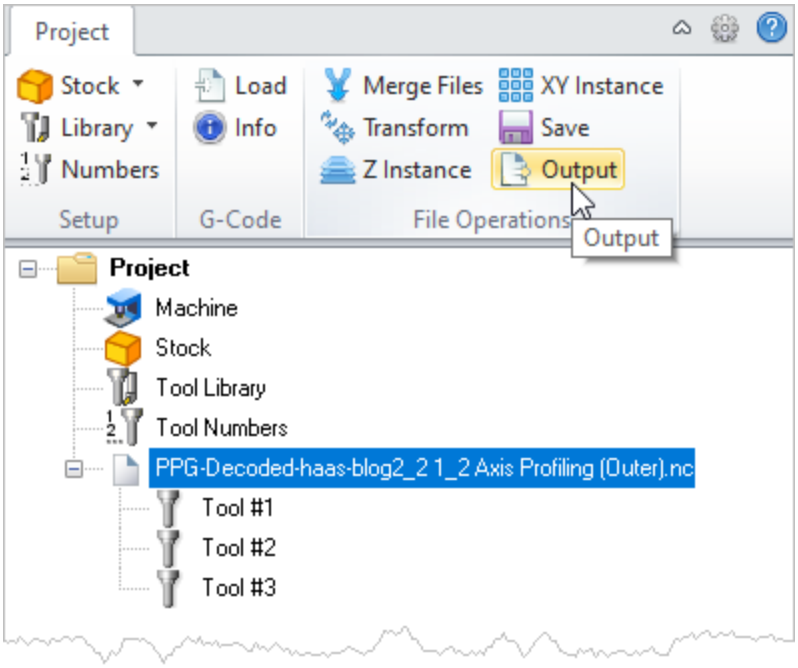
5.4.6 Output

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: Project > Output

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

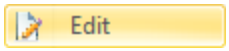
User Interface

© 2026 MecSoft Corporation

Project Tab
Load
Info
Merge Files
Transform
Z Instance
XY Instance
Save

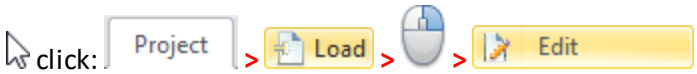
5.5 Right-Click Menu

5.5.1 Edit-G-Code-File



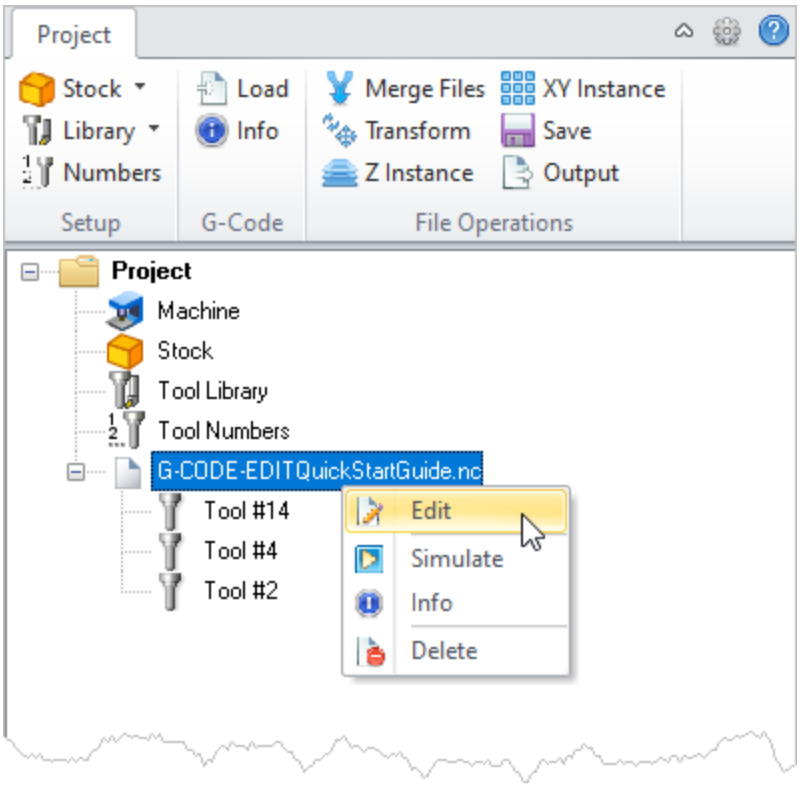
Use this command to Edit a G-Code file that is currently loaded into the Project tab of the G-Code Browser.

To Access this Command:

click: 

Right-Click Menu

© 2026 MecSoft Corporation



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

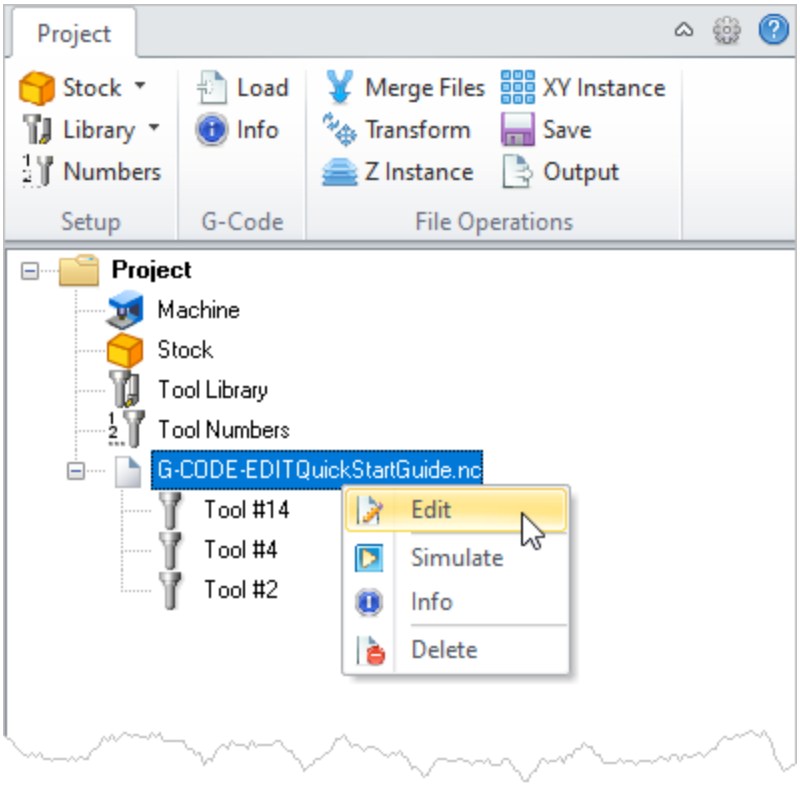
5.5.2 Simulate G-Code File

Simulate 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:

click: Project > Load > > Simulate

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

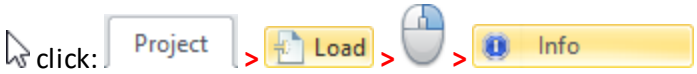
Delete G-Code File

© 2026 MecSoft Corporation

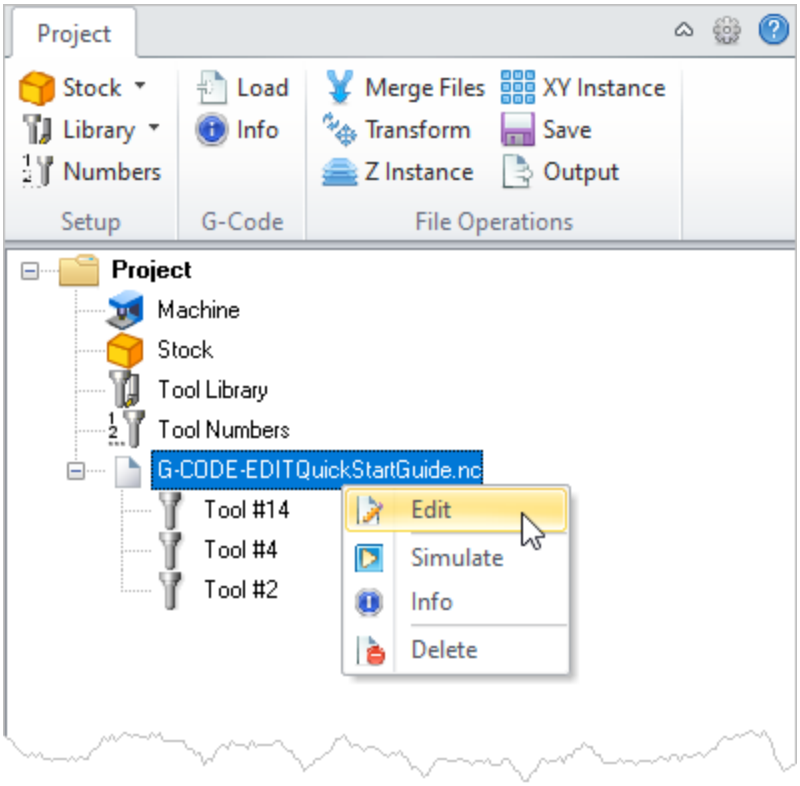
5.5.3 Info on G-Code File

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:

click: 

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

© 2026 MecSoft Corporation

Info on G-Code File

Delete G-Code File

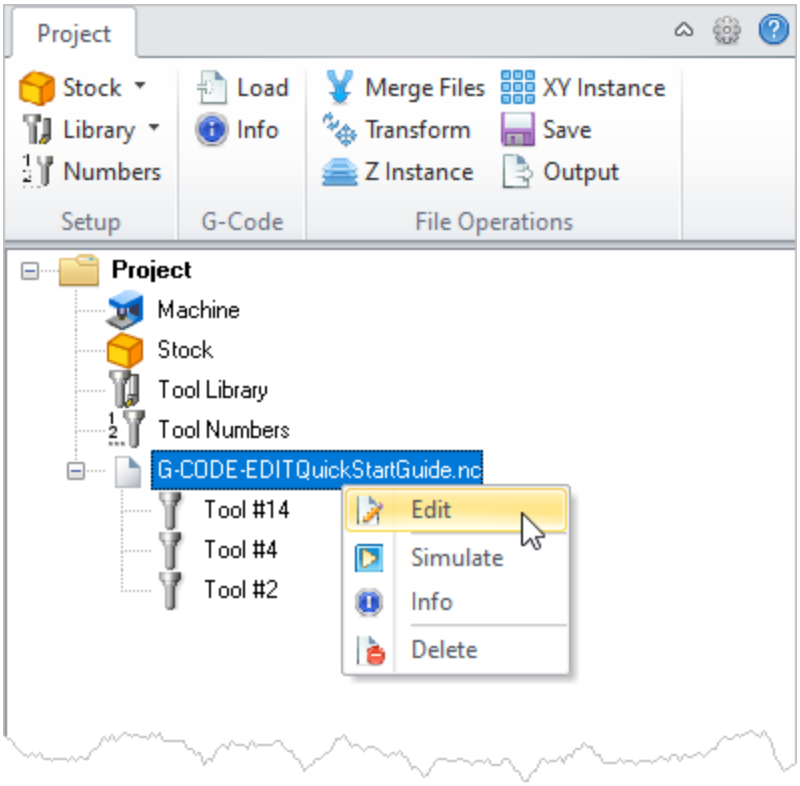
5.5.4 Delete G-Code File

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:

click: Project > Load > > Delete

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.

© 2026 MecSoft Corporation

Related Topics

Edit-G-Code-File

Simulate G-Code File

Info on G-Code File

Delete G-Code File

© 2026 MecSoft Corporation

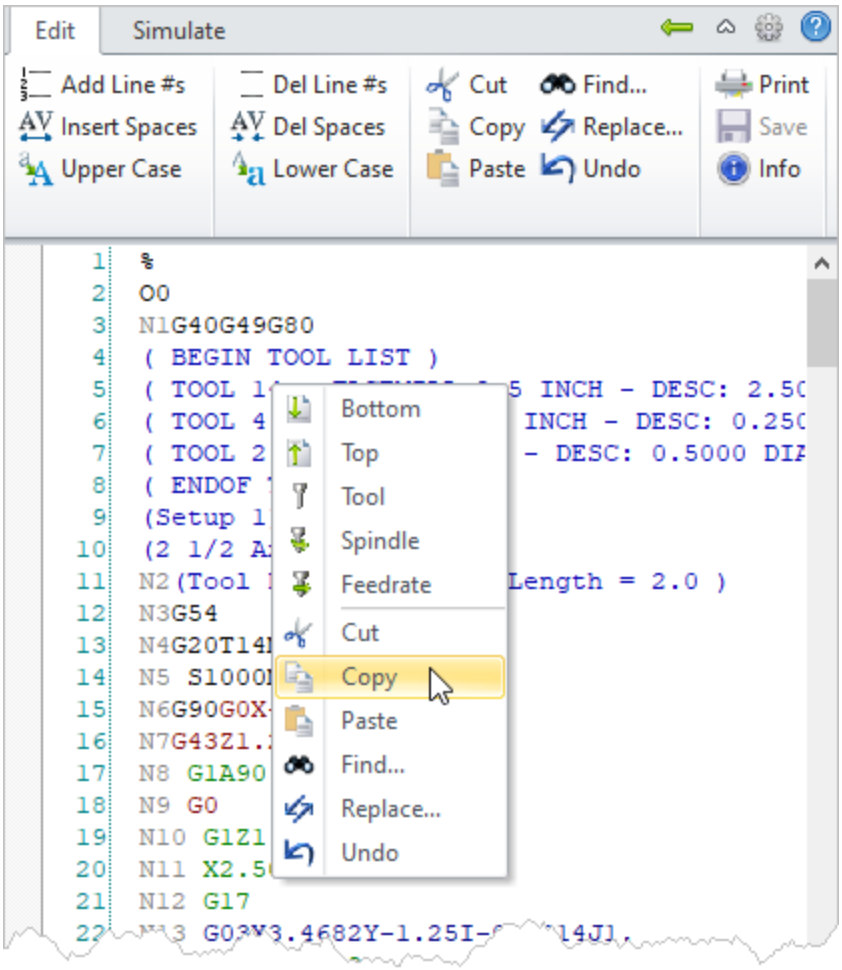
Edit Tab

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.

© 2026 MecSoft Corporation

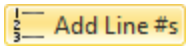
Returns to the Project tab.

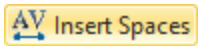
Minimizes and maximizes the ribbon bar menu.

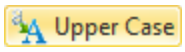
Displays the Preferences dialog.

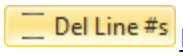
Displays the Online Help system.

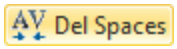
Edit Tab

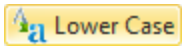
 Add line numbers to the g-code file.

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

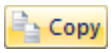
 Convert g-code to all UPPERCASE text.

 Delete line numbers from the g-code file.

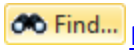
 Delete spaces (i.e., delimiters) between g-codes.

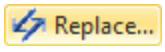
 Convert g-code to all lowercase text.

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.

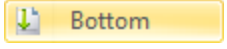
Undo the last command.

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.

© 2026 MecSoft Corporation

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

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

© 2026 MecSoft Corporation

Project Tab

Project Tree

Simulate Tab

Online Help

6.1 Add Line Numbers

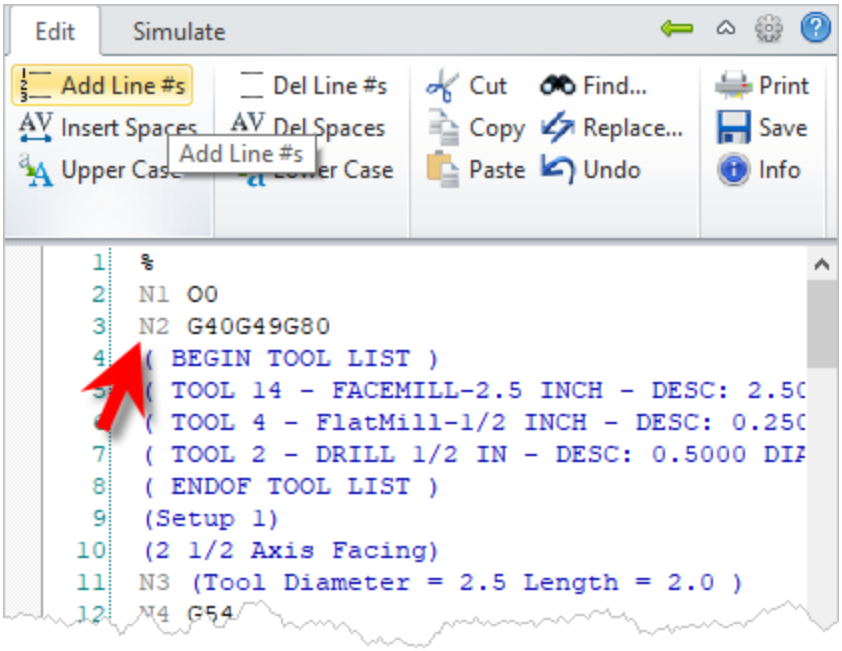
Add Line #s Use this command to Add Line Numbers to the g-code file. Comments are not assigned line numbers. These line numbers are included when you save or output the g-code file. The line numbers you see to the far left are for reference within the G-Code Editor only and are not included in the g-code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

To Access this Command:

click: Project > Load > Edit > Add Line #s

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

© 2026 MecSoft Corporation

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

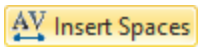
Delete Spaces

Make Upper Case

Make Lower Case

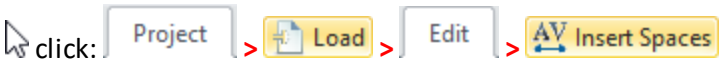
Show G-Code Info

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.

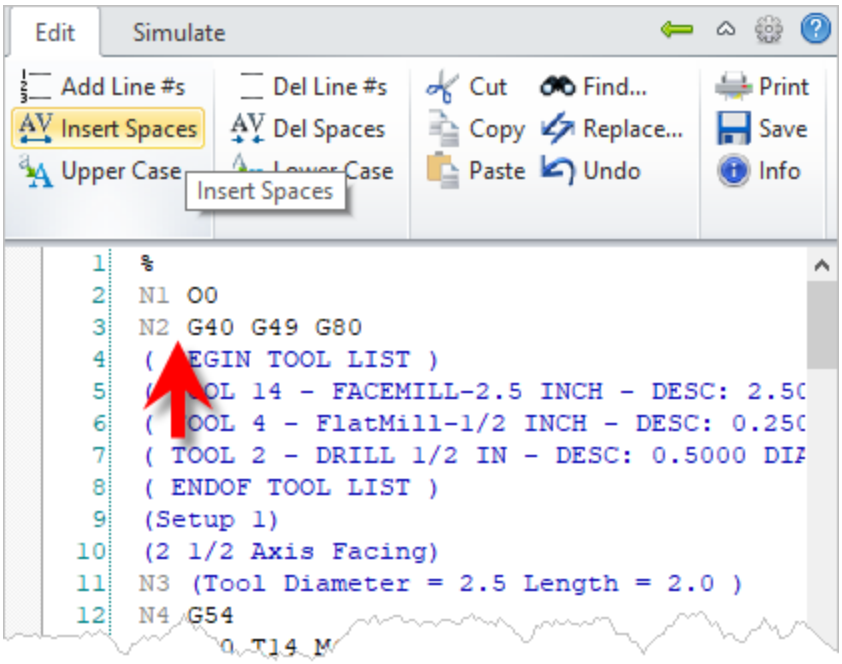
To Access this Command:



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

© 2026 MecSoft Corporation



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

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.

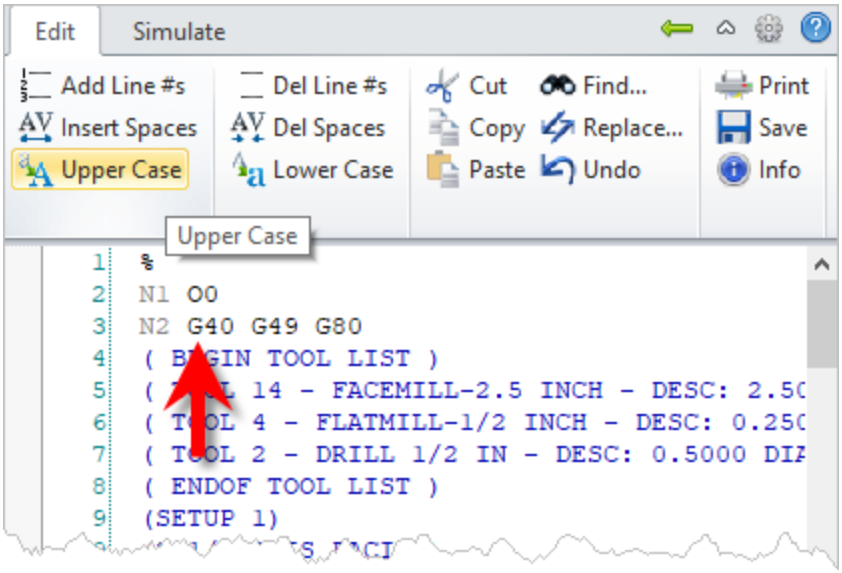
© 2026 MecSoft Corporation

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

© 2026 MecSoft Corporation

6.4 Delete Line Numbers

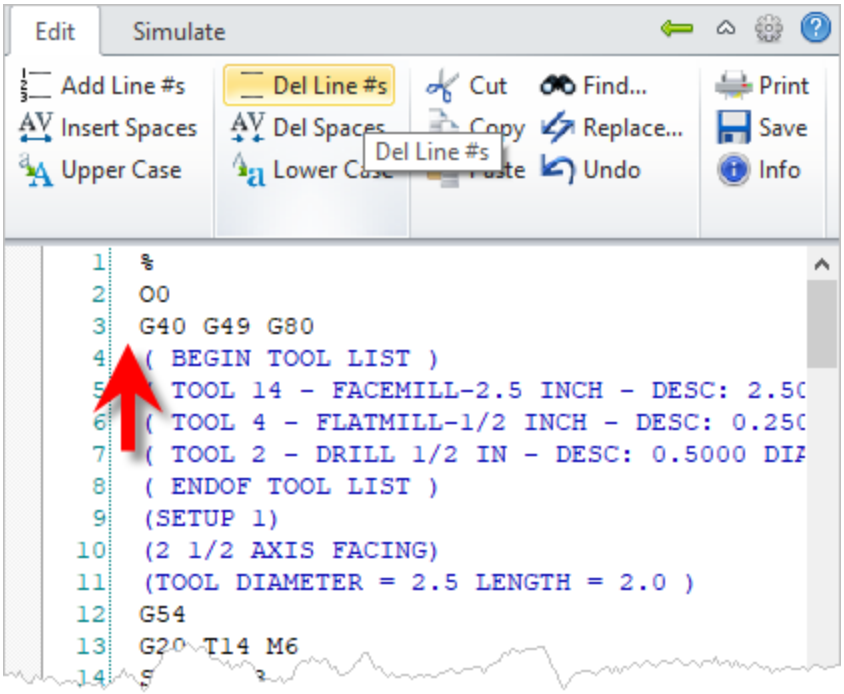
Del Line #s Use this command to Delete Line Numbers in the g-code file. Comments are not assigned line numbers. Line numbers, if displayed, are included when you save or output the g-code file. The line numbers you see to the far left are for reference within the G-Code Editor only and are not included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

To Access this Command:

click: Project > Load > Edit > Del Line #s

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

© 2026 MecSoft Corporation

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

Del 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.

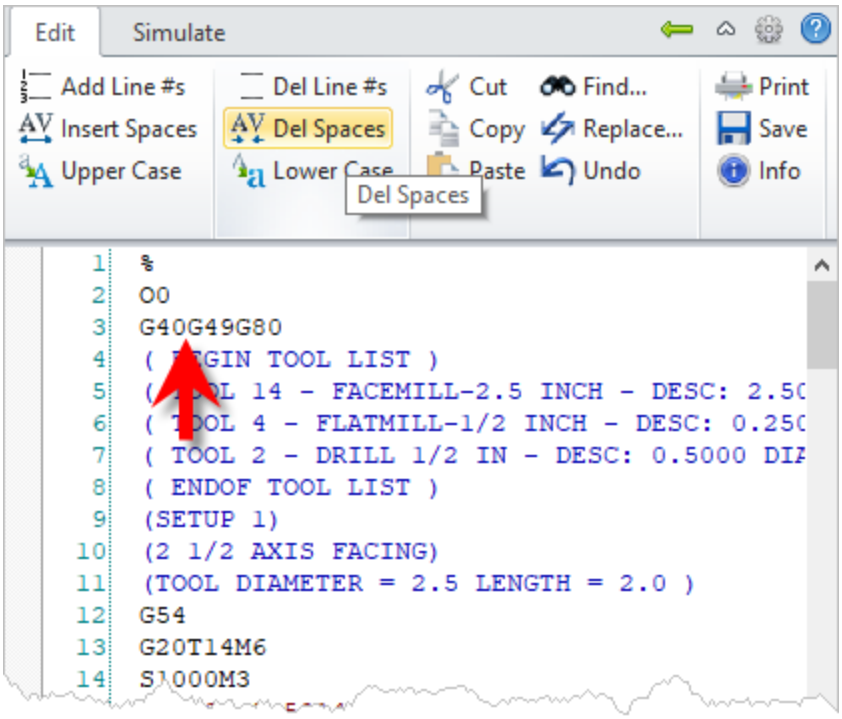
To Access this Command:

click: Project > Load > Edit > Del Spaces

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

© 2026 MecSoft Corporation



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

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.

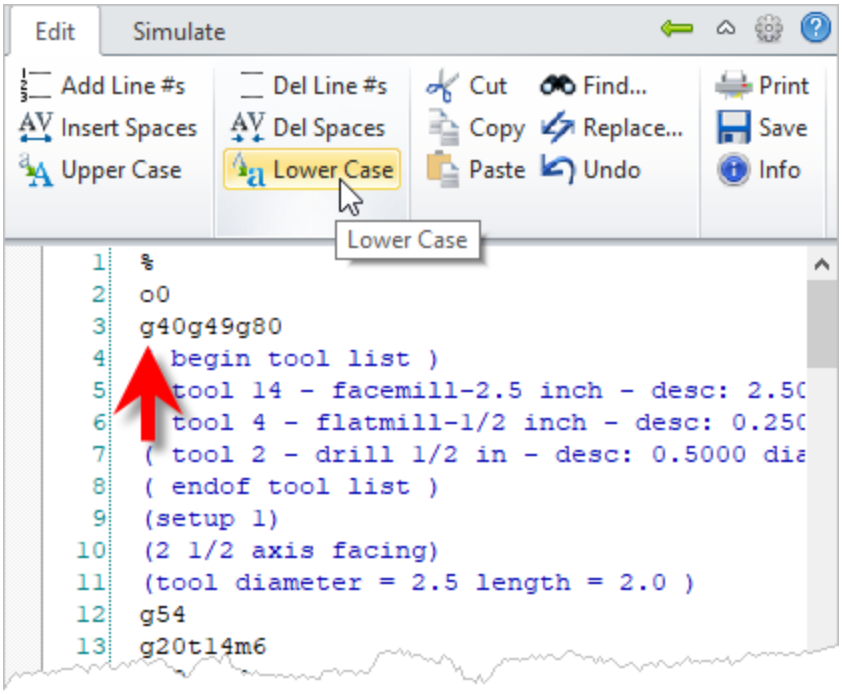
© 2026 MecSoft Corporation

To Access this Command:

click: Project > Load > Edit > Lower Case

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

© 2026 MecSoft Corporation

Delete Spaces

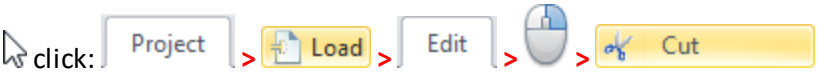
Make Upper Case

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:

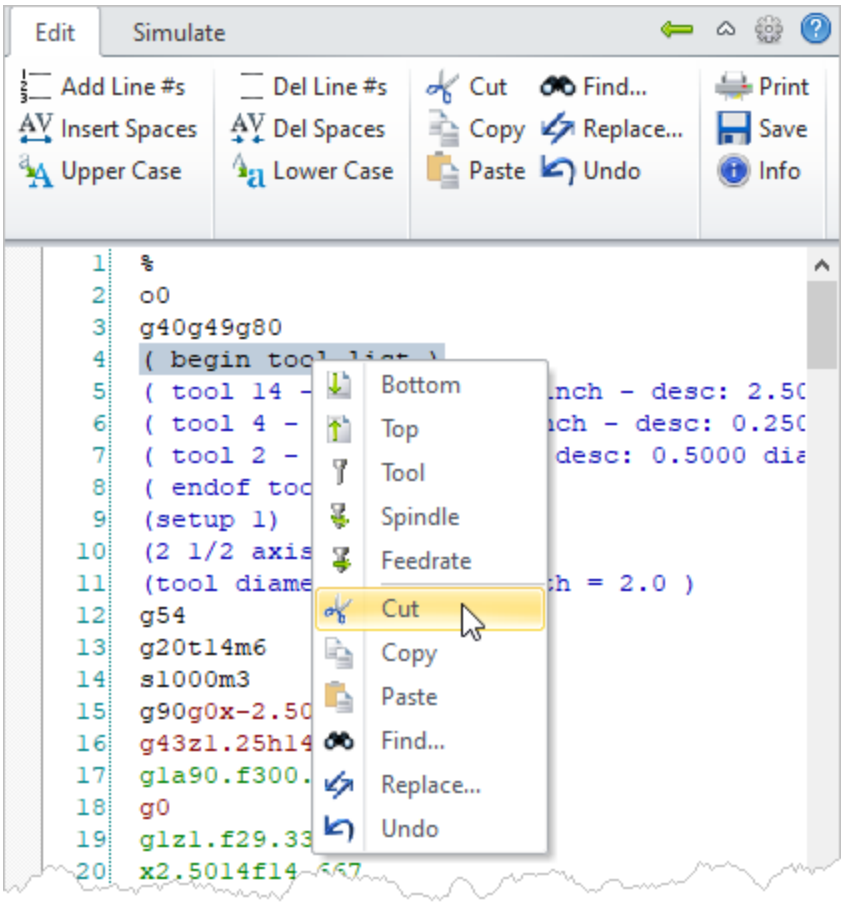
click: 

or

click:

Cut Text (Right-Click Menu)

© 2026 MecSoft Corporation



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

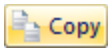
Undo Last Edit

Print G-Code

© 2026 MecSoft Corporation

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.

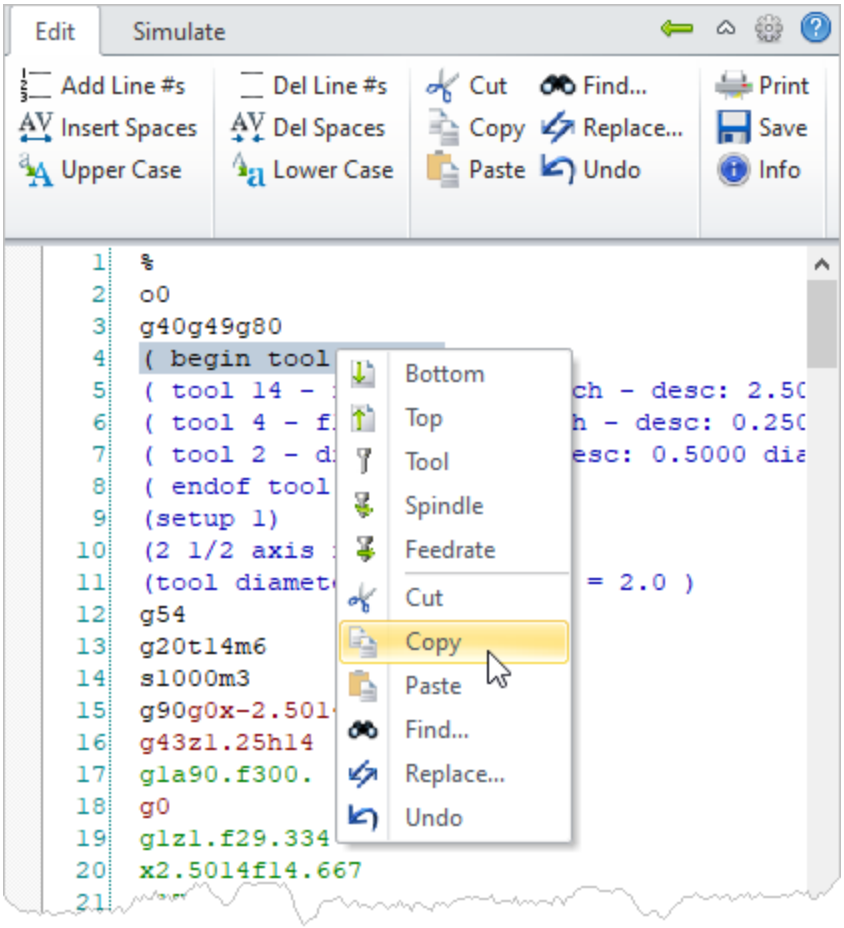
To Access this Command:

click: Project > Load > Edit > > Copy

or

click: Project > Load > Edit > Copy

Copy Text (Right-Click Menu)



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

© 2026 MecSoft Corporation

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

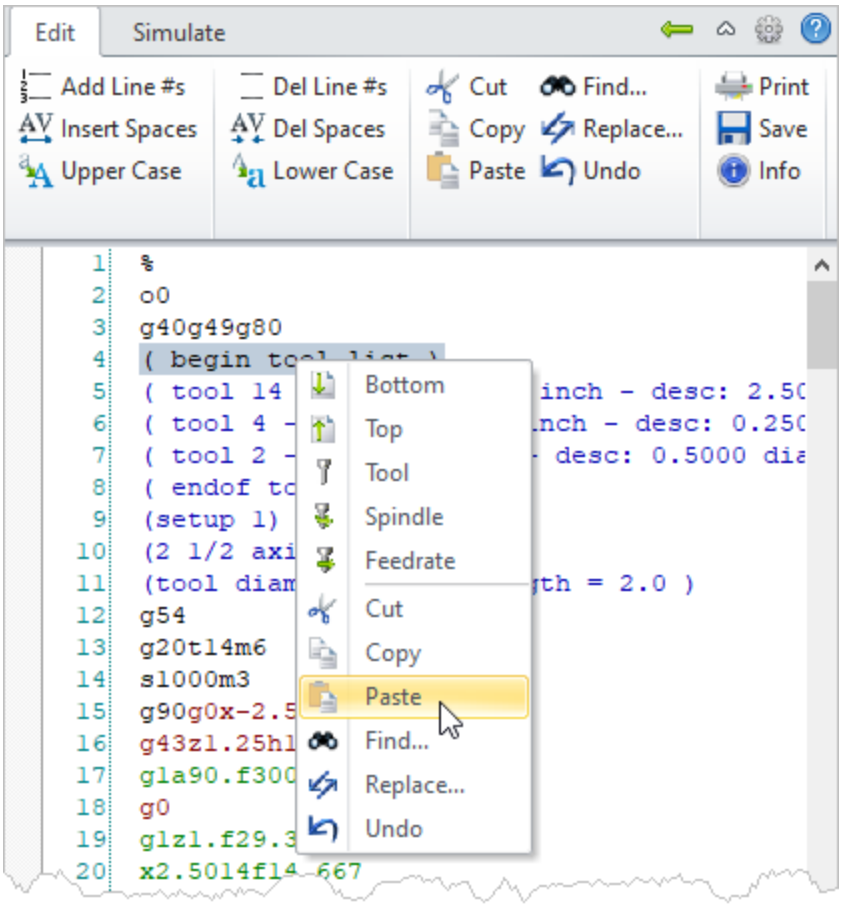
or

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)

© 2026 MecSoft Corporation



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

© 2026 MecSoft Corporation

Save G-Code

6.10 Find Text

Find... 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:

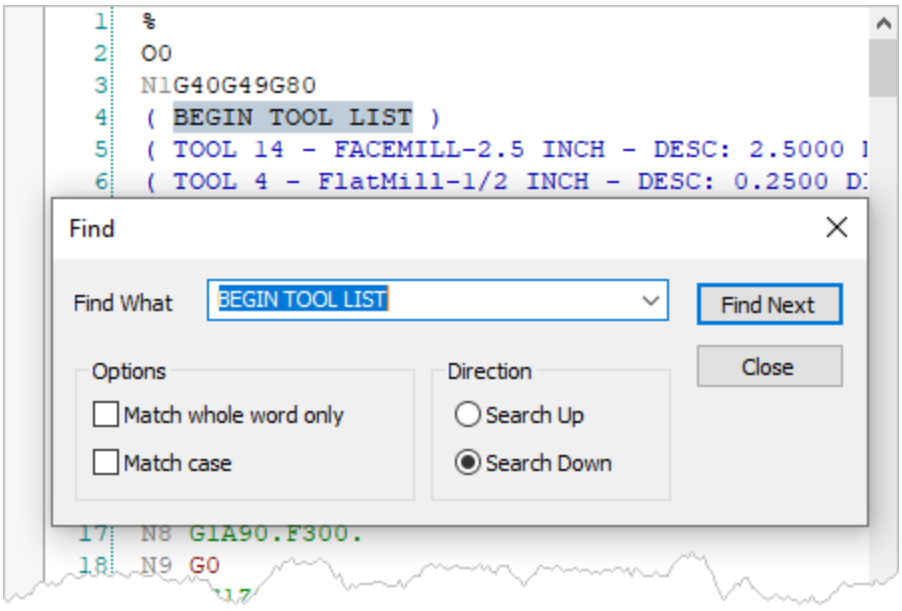
click: Project > Load > Edit > Find...

or

click: Project > Load > Edit > > Find...

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.

© 2026 MecSoft Corporation

- 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: Project > Load > Edit > Replace...

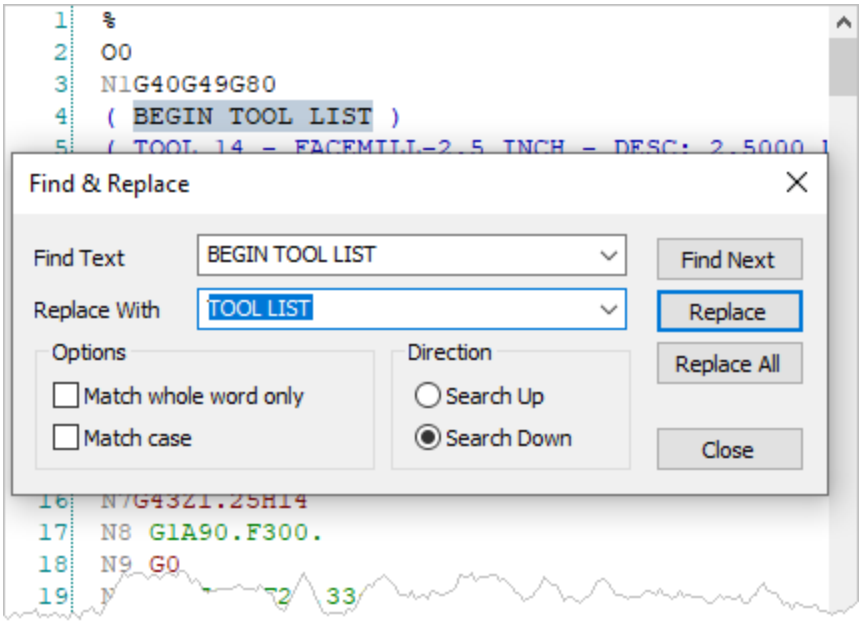
or

click: Project > Load > Edit > > Replace...

© 2026 MecSoft Corporation

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.

© 2026 MecSoft Corporation

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:

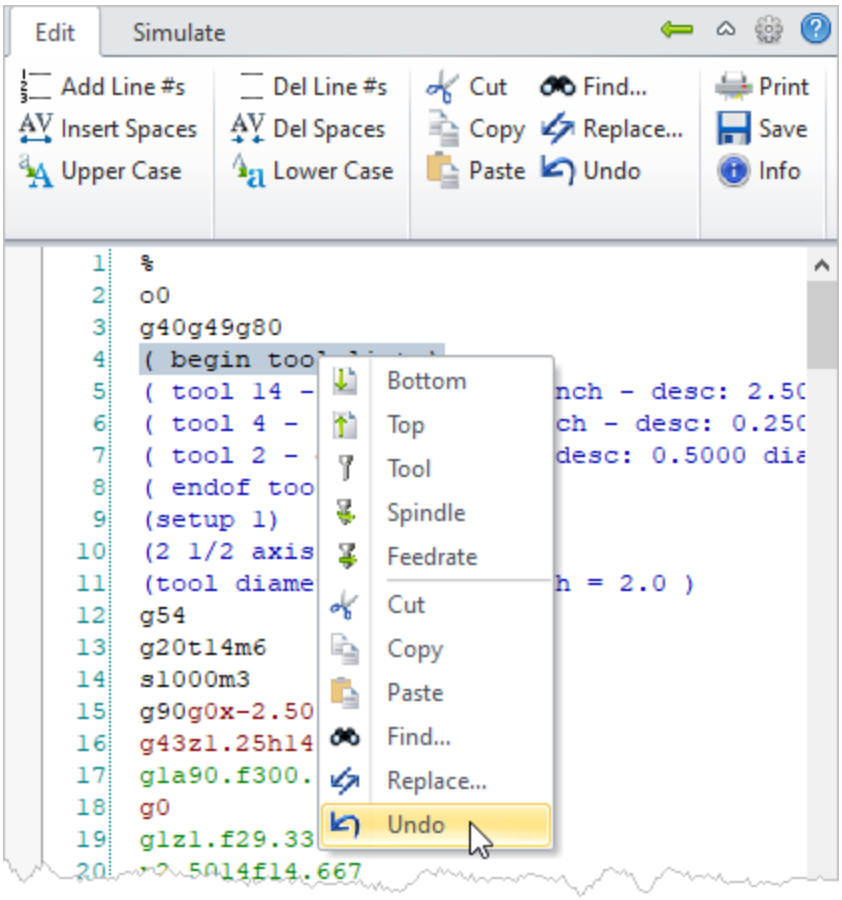
click: Project > Load > Edit > > Undo

or

click: Project > Load > Edit > Undo

Undo Last Edit (Right-Click Menu)

© 2026 MecSoft Corporation



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

© 2026 MecSoft Corporation

Print G-Code

Save G-Code

6.13 Print G-Code

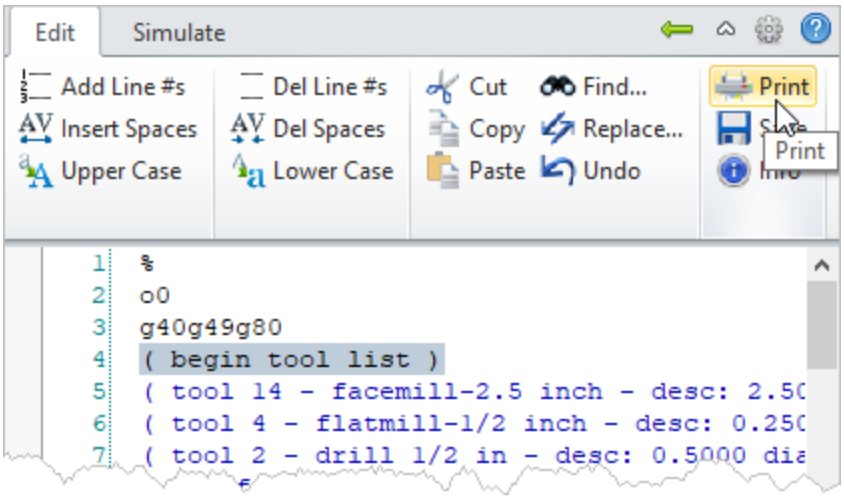
Print Use this command to Print the current g-code file. The default Windows Print dialog will display allowing you to select a printing device and print preferences.

To Access this Command:

click: Project > Load > Edit > Print

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

© 2026 MecSoft Corporation

Edit Tab
Cut Text
Copy Text
Paste Text
Undo Last Edit
Save G-Code

6.14 Save G-Code

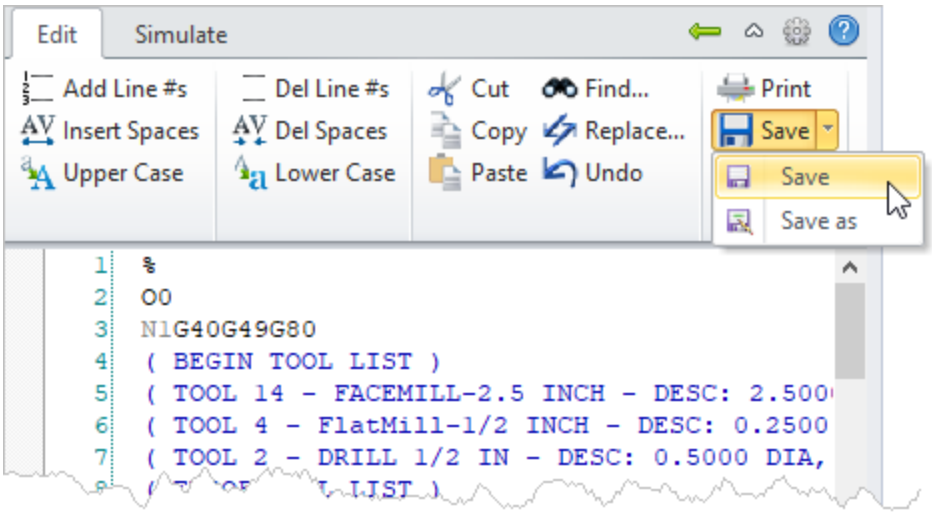
Save Select this command to Save the g-code file to the default Save Directory defined in the Text Editor Preferences dialog. You original g-code file IS NOT overwritten.

To Access this Command:

click: Project > Load > Edit > Save > Save

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. You original g-code file IS NOT overwritten.

© 2026 MecSoft Corporation

Related Topics

User Interface

Project Tab

Project Tree

Edit Tab

Cut Text

Copy Text

Paste Text

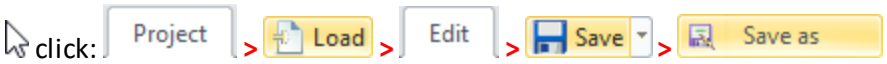
Undo Last Edit

Print G-Code

6.15 Save As

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.

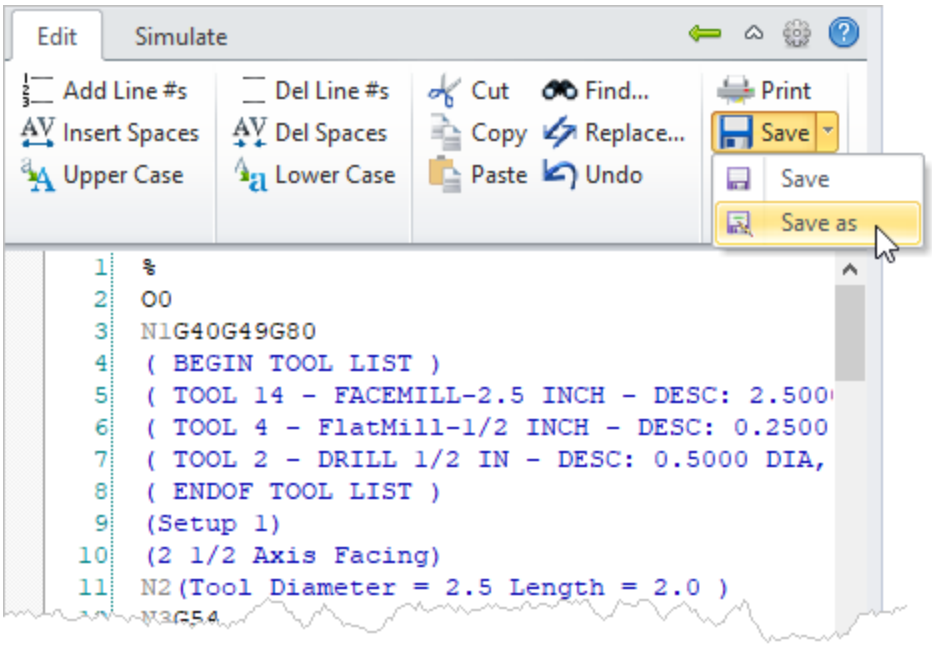
To Access this Command:



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

Save menu item

© 2026 MecSoft Corporation



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

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

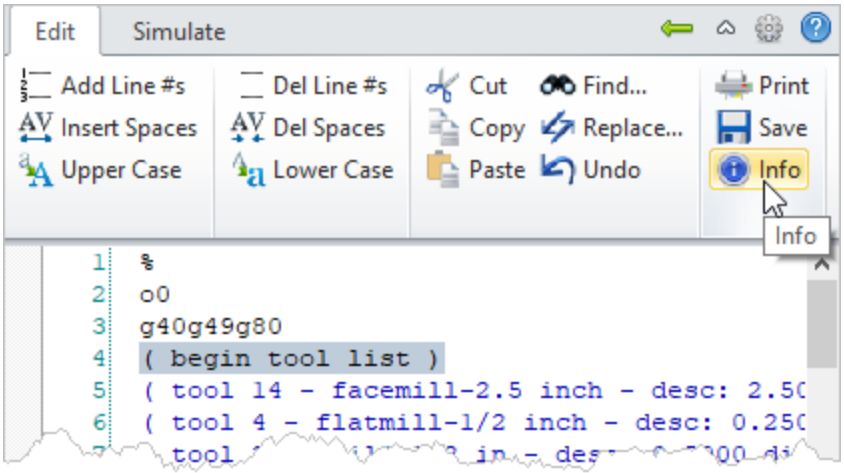
© 2026 MecSoft Corporation

To Access this Command:

click: Project > Load > Edit > Info

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

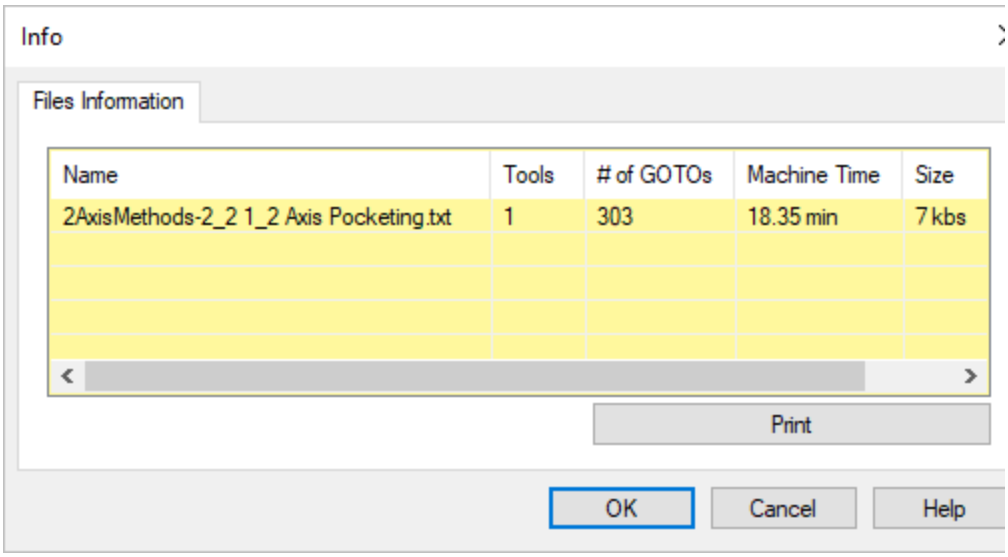
Info menu item



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

© 2026 MecSoft Corporation

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.

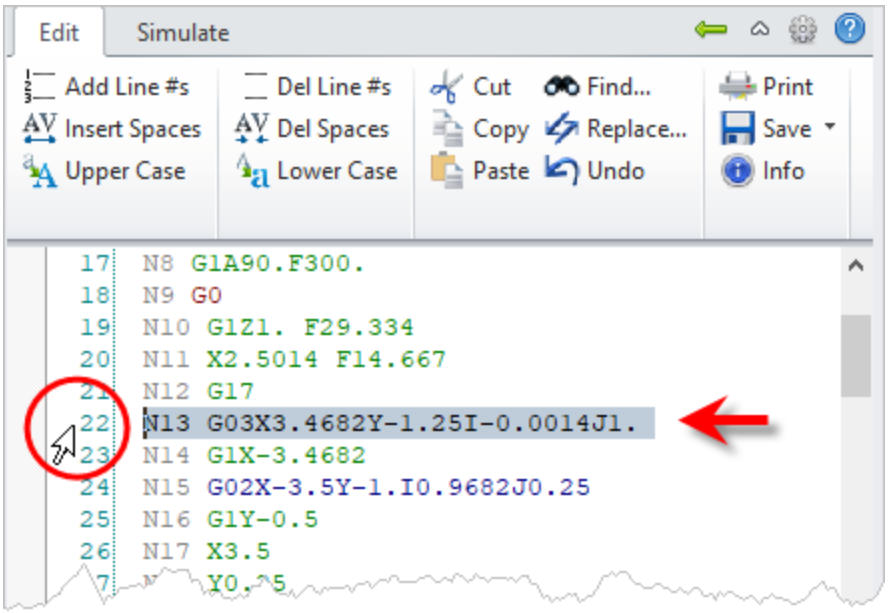
To Access this Command:

click: Project > Load > Edit > Add Line #s

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

Info menu item

© 2026 MecSoft Corporation



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

6.18 Right-Click Menu

6.18.1 Go to Bottom

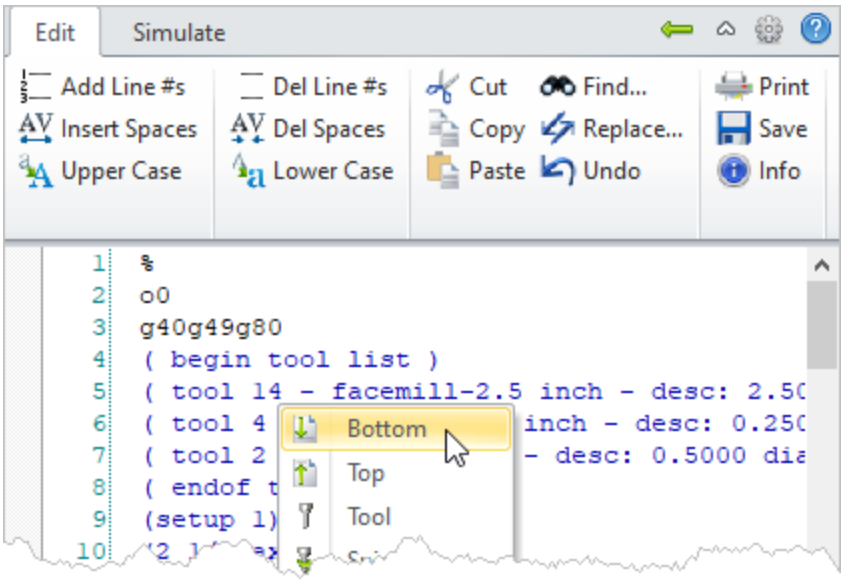
Bottom Use this command to move the cursor to the last g-code numbered line in the g-code file.

To Access this Command:

click: Project > Load > Edit > > Bottom

Bottom (Right-Click Menu)

© 2026 MecSoft Corporation



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

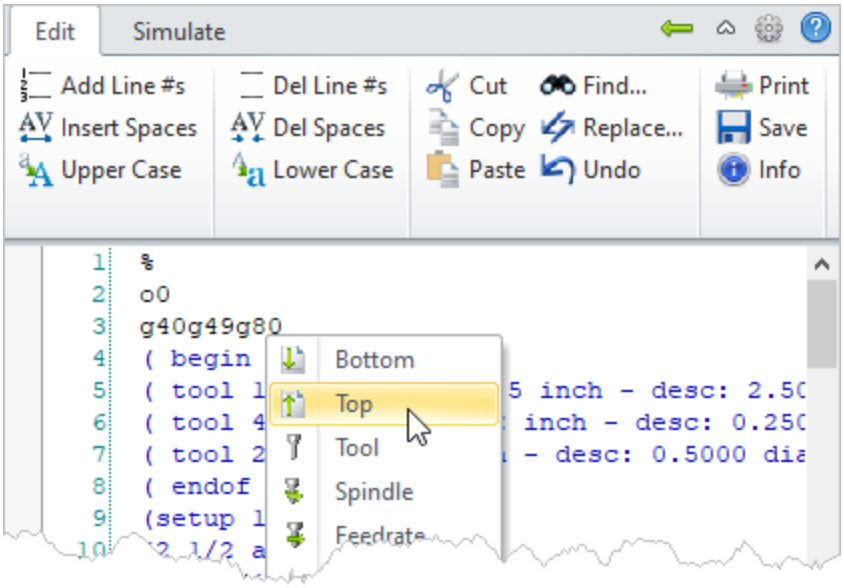
Top Use this command to move the cursor to the first line in the g-code file.

To Access this Command:

© 2026 MecSoft Corporation

click: Project > Load > Edit > Top

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

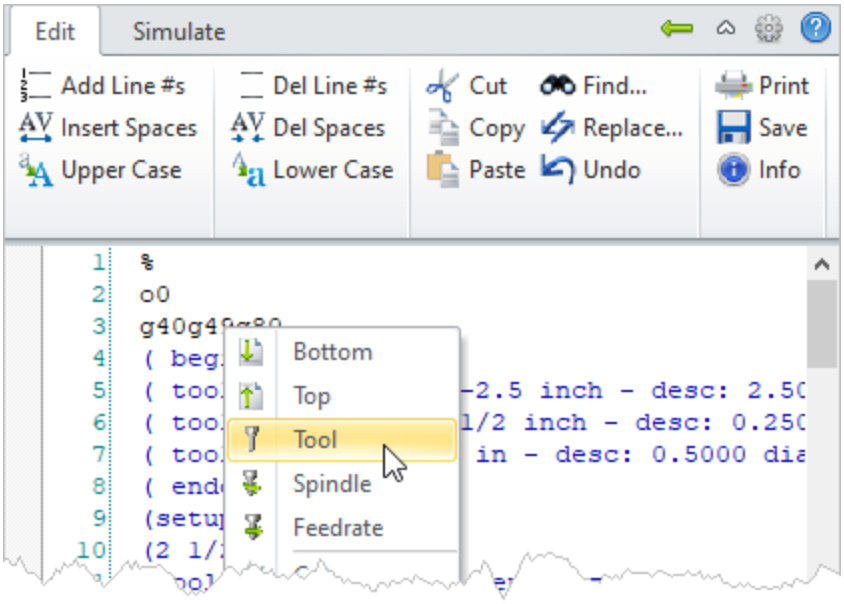
Tool Use this command to move the cursor to the next line in the g-code file that contains a tool number.

© 2026 MecSoft Corporation

To Access this Command:

click: Project > Load > Edit > > Tool

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

© 2026 MecSoft Corporation

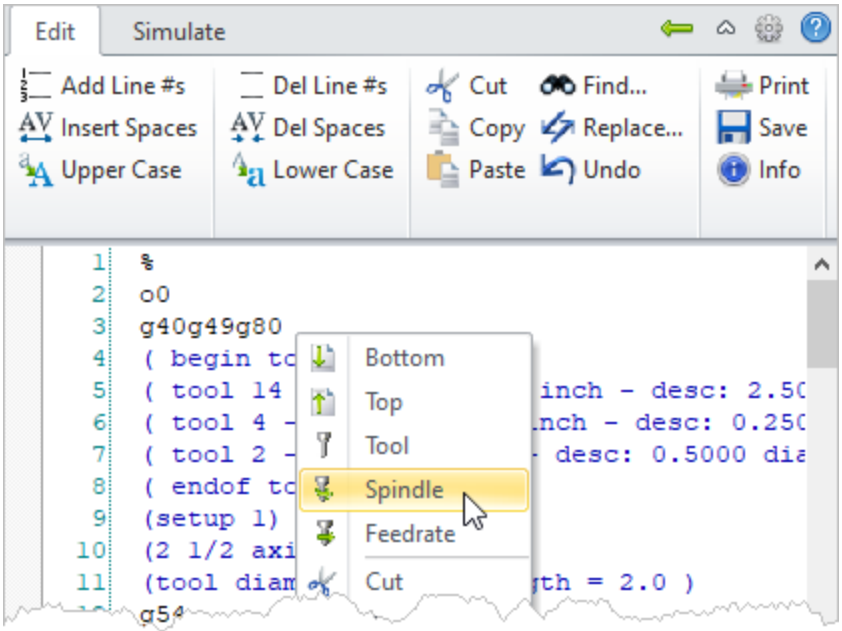
6.18.4 Go to Next Spindle

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:

click: Project > Load > Edit > > Spindle

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

© 2026 MecSoft Corporation

Go to Next Tool

Go to Next Feedrate

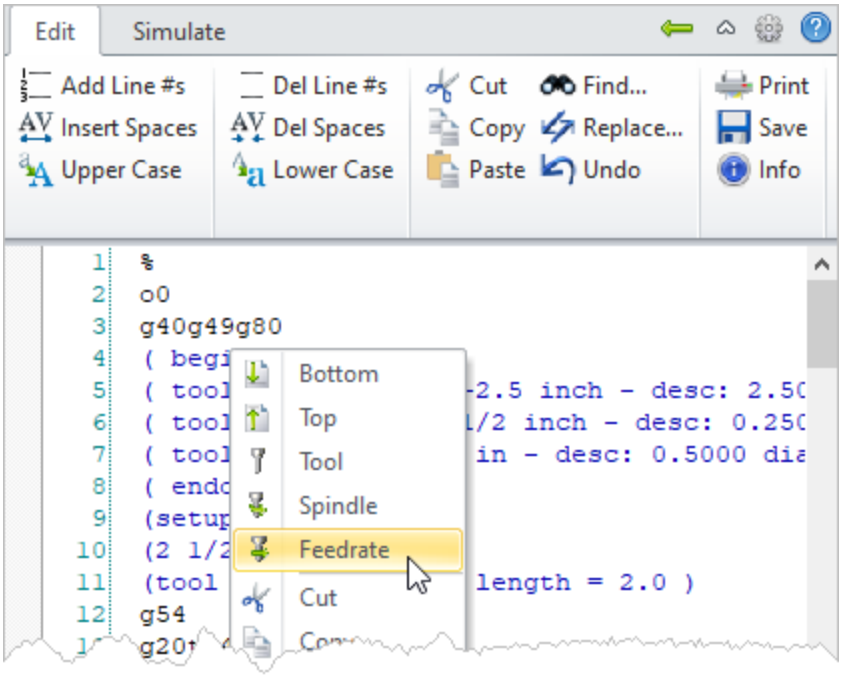
6.18.5 Go to Next Feedrate

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: Project > Load > Edit > > Feedrate

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

© 2026 MecSoft Corporation

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:

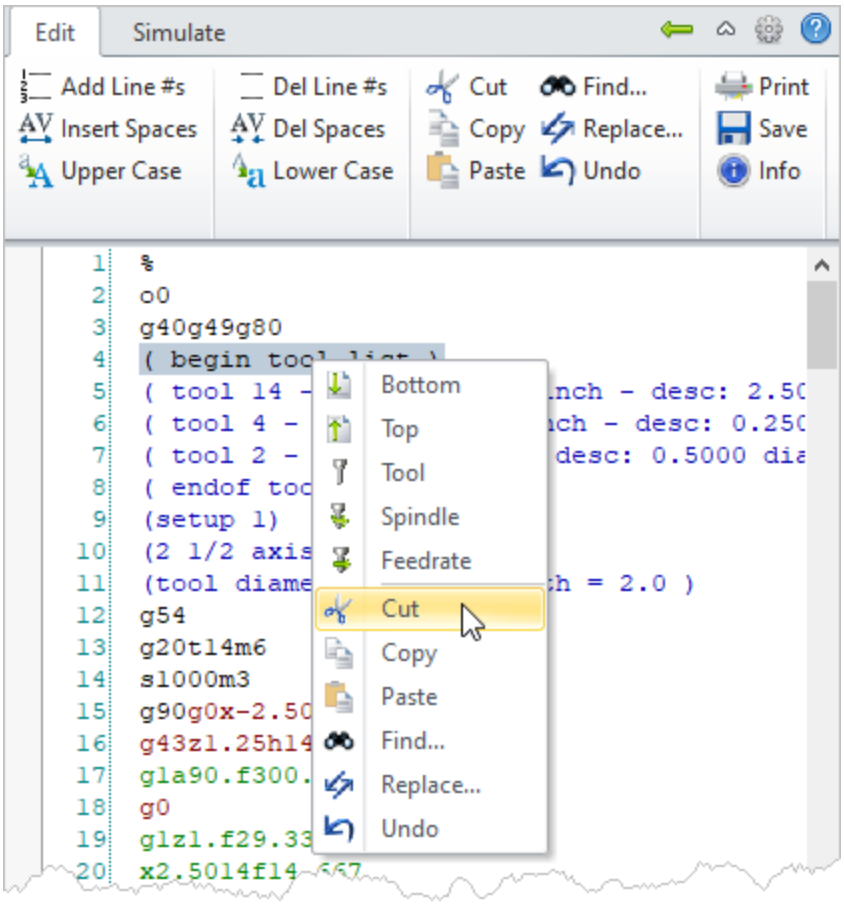
click: Project > Load > Edit > > Cut

or

click: Project > Load > Edit > Cut

Cut Text (Right-Click Menu)

© 2026 MecSoft Corporation



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

© 2026 MecSoft Corporation

Save G-Code

6.18.7 Copy Text

Copy 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.

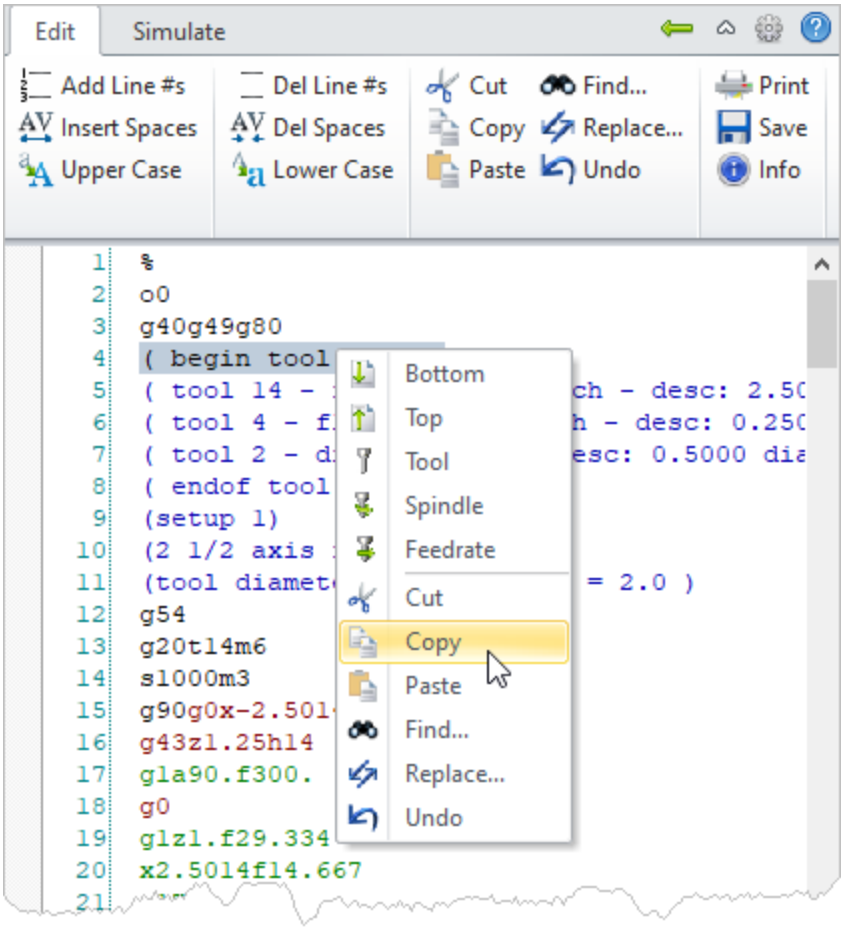
To Access this Command:

click: Project > Load > Edit > > Copy

or

click: Project > Load > Edit > Copy

Copy Text (Right-Click Menu)



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

© 2026 MecSoft Corporation

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: Project > Load > Edit > > Paste

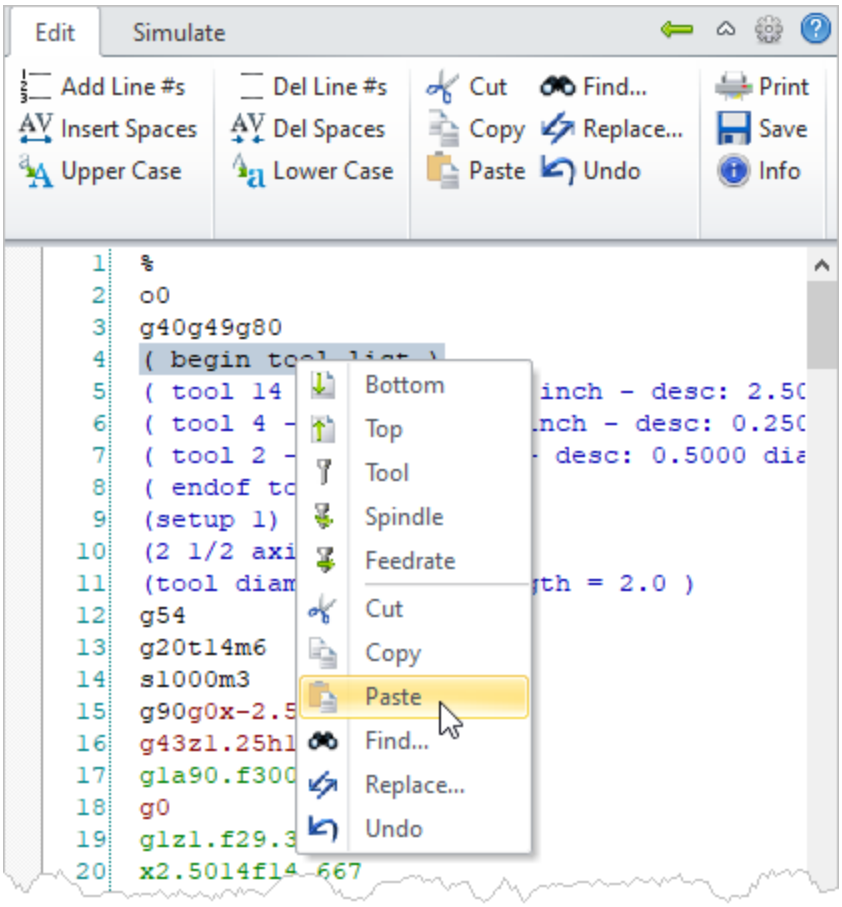
or

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)

© 2026 MecSoft Corporation



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

© 2026 MecSoft Corporation

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.

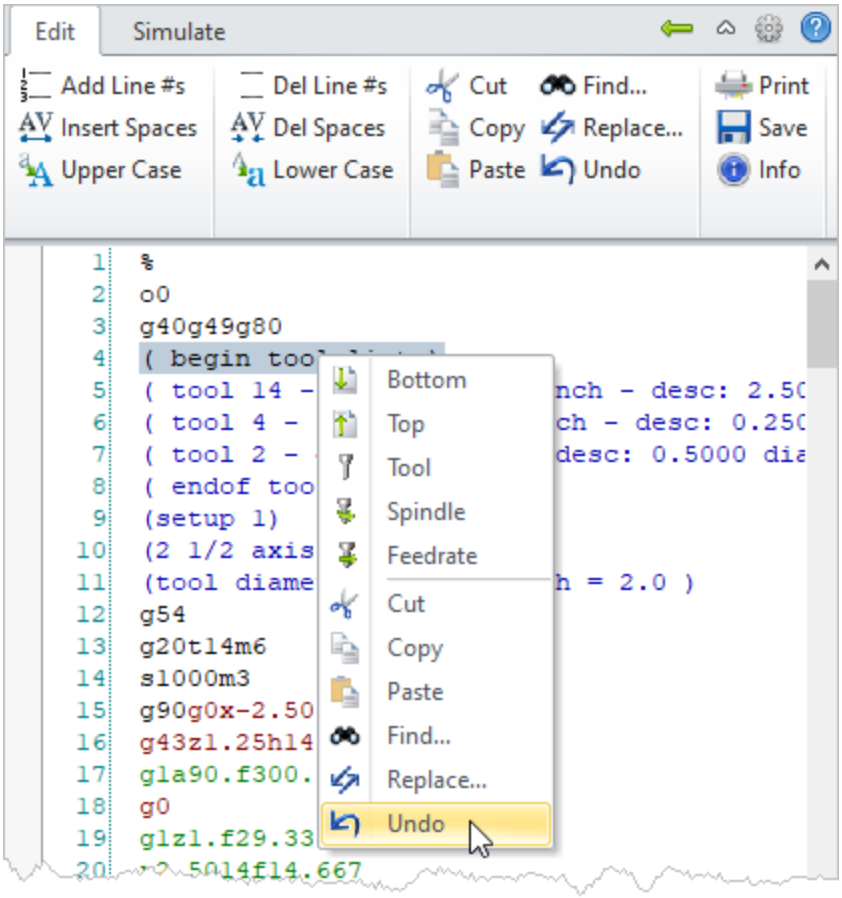
To Access this Command:

click: Project > Load > Edit > > Undo

or

click: Project > Load > Edit > Undo

Undo Last Edit (Right-Click Menu)



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

© 2026 MecSoft Corporation

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

© 2026 MecSoft Corporation

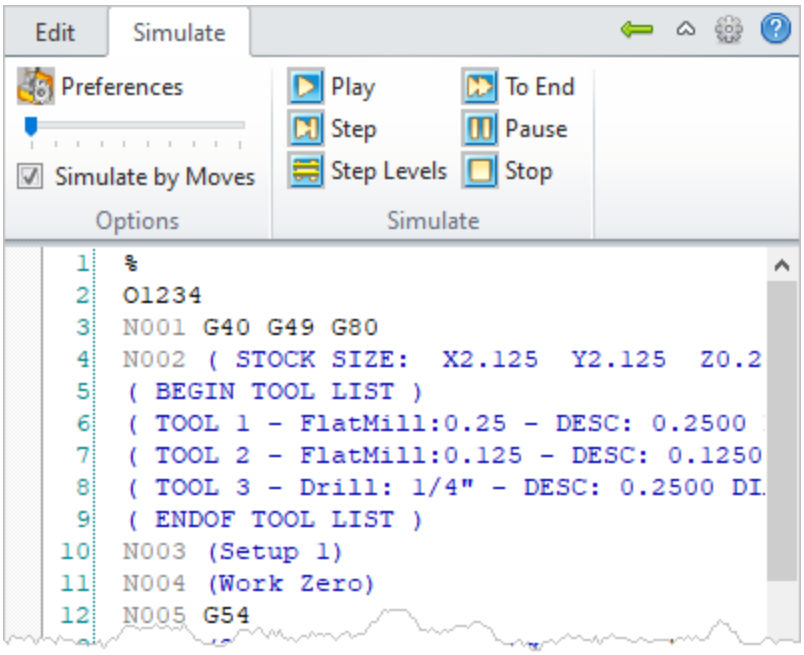
Simulate Tab

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.

© 2026 MecSoft Corporation

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.

© 2026 MecSoft Corporation

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

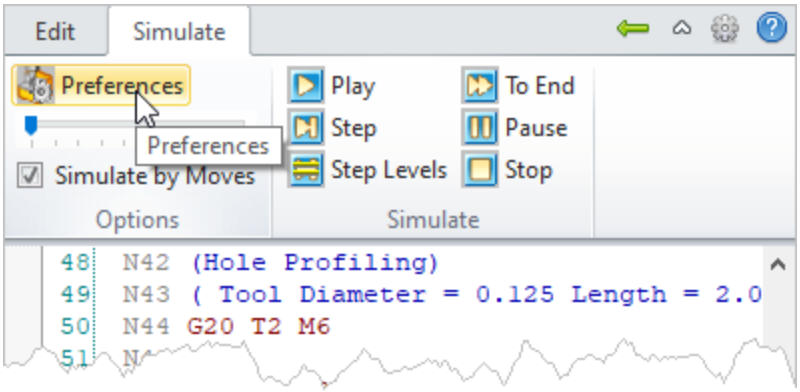
Preferences Select this button to display the Simulation Preferences dialog. This is the same as selecting > Simulation.

To Access this Command:

click: Project > Load > Simulate > Preferences

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

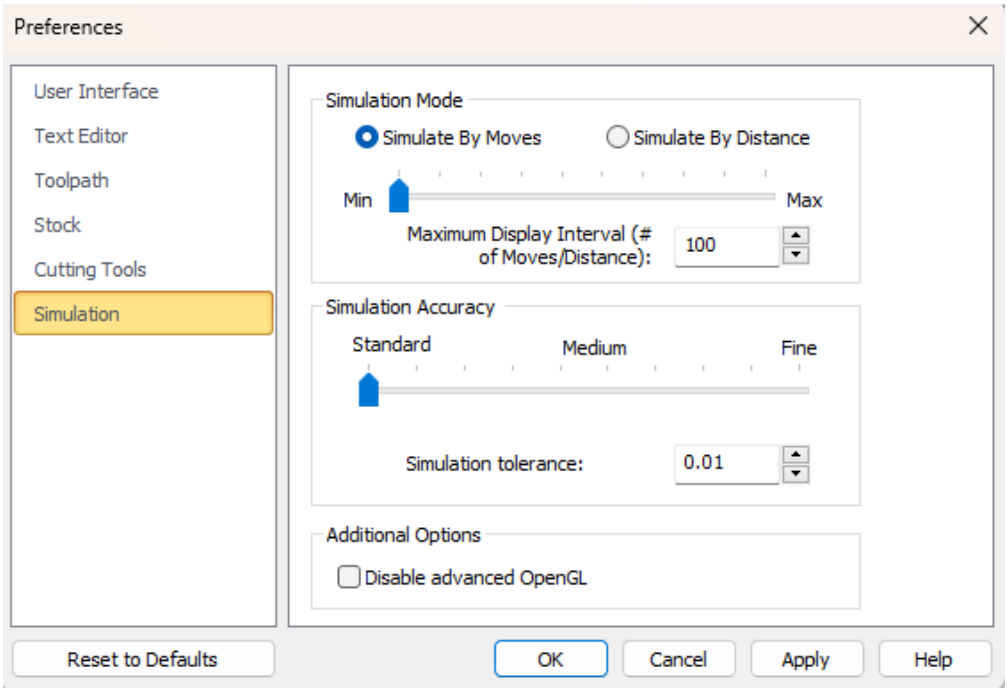


G-Code Browser: Simulate, Preferences menu item

Procedure

© 2026 MecSoft Corporation

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.

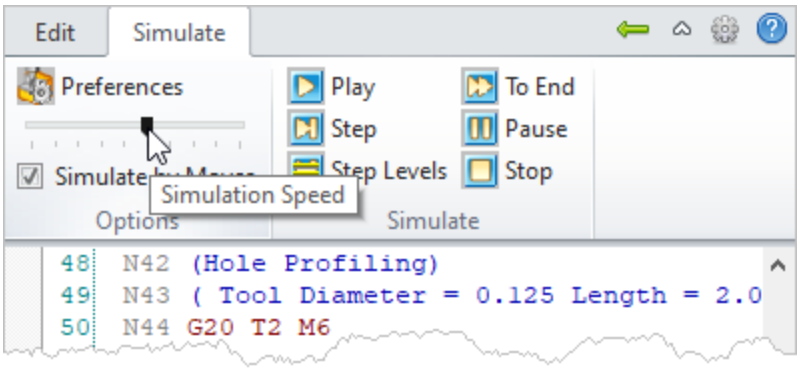
To Access this Command:

click: Project > Load > Simulate >

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

© 2026 MecSoft Corporation



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

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

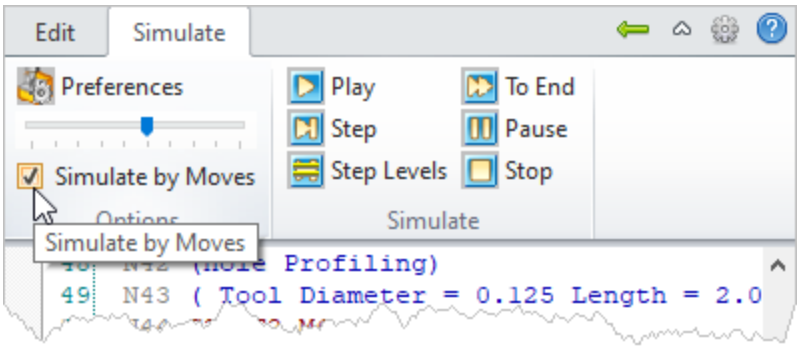
To Access this Command:

click: Project > Load > Simulate > 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

© 2026 MecSoft Corporation



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

Play Select this commend 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.

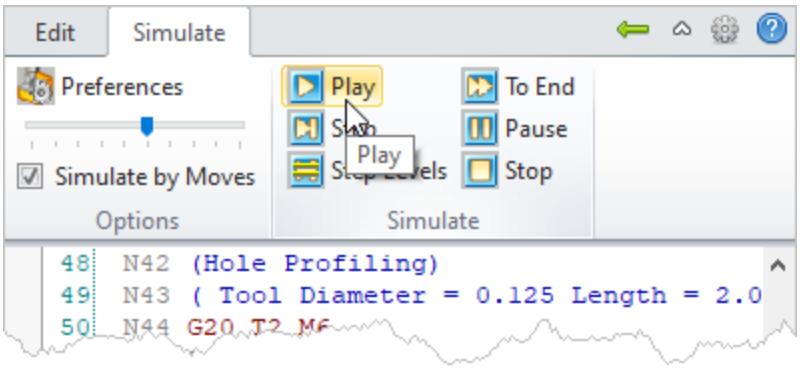
To Access this Command:

click: Project > Load > Simulate > Play

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

© 2026 MecSoft Corporation



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

Step 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.

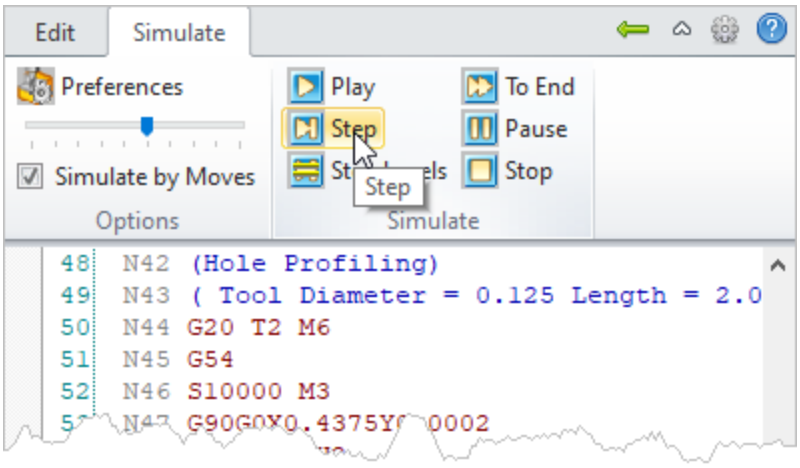
To Access this Command:

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

© 2026 MecSoft Corporation



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.

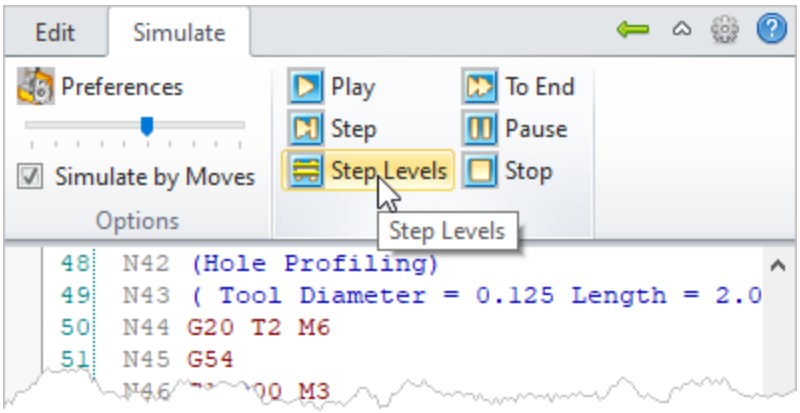
To Access this Command:

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

© 2026 MecSoft Corporation



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

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.

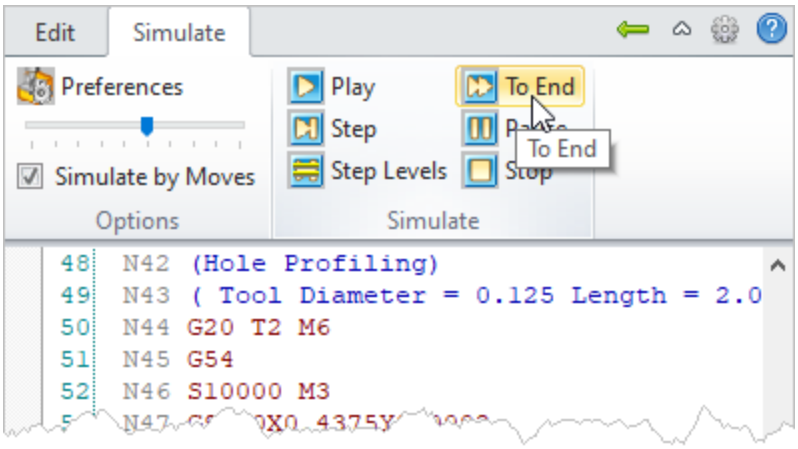
To Access this Command:

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

© 2026 MecSoft Corporation



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

Pause 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.

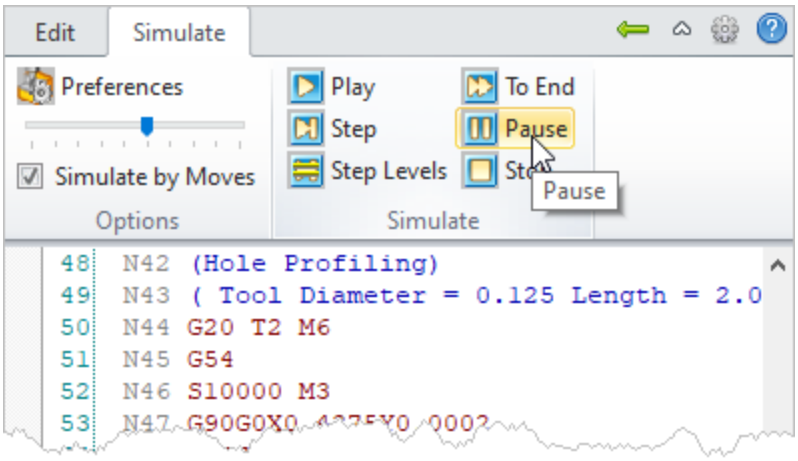
To Access this Command:

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

© 2026 MecSoft Corporation



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

Stop 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.

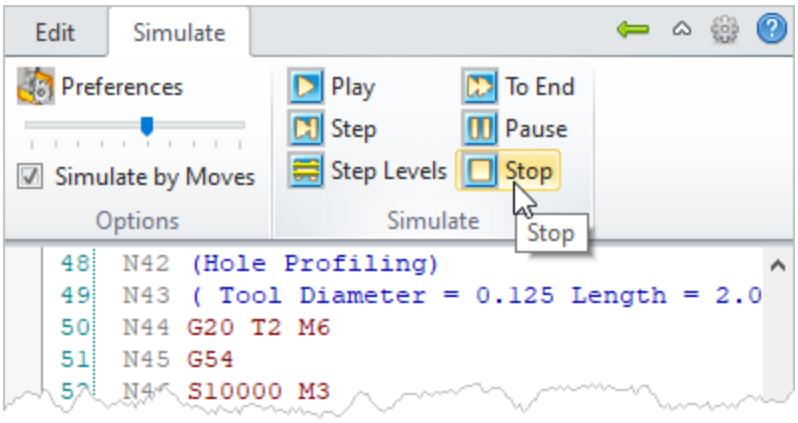
To Access this Command:

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

© 2026 MecSoft Corporation



G-Code Browser: Simulate, Stop menu item

Related Topics

Simulate Tab

Play Simulation

Step Thru Simulation

Display Simulation Levels

Simulate to End

Pause Simulation

© 2026 MecSoft Corporation

Functions from the MILL Module

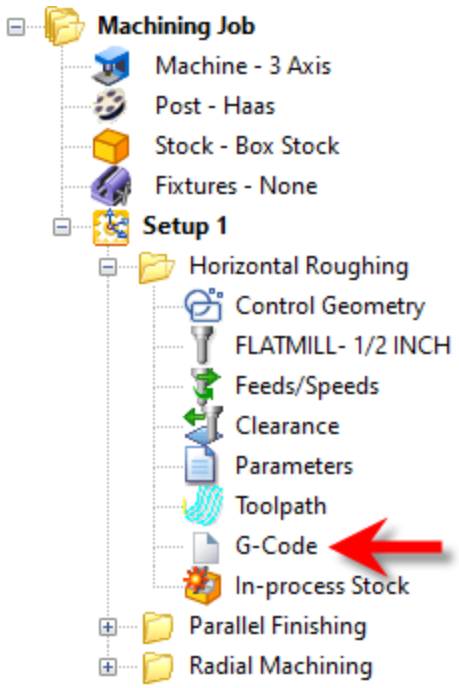
You can access the G-Code Editor module from within the MILL module by clicking on the G-Code icon located in each machining operation (Mop) folder as shown below if the icon is not flagged. However, ONLY the Edit tab is available and you cannot simulate G-Code.

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

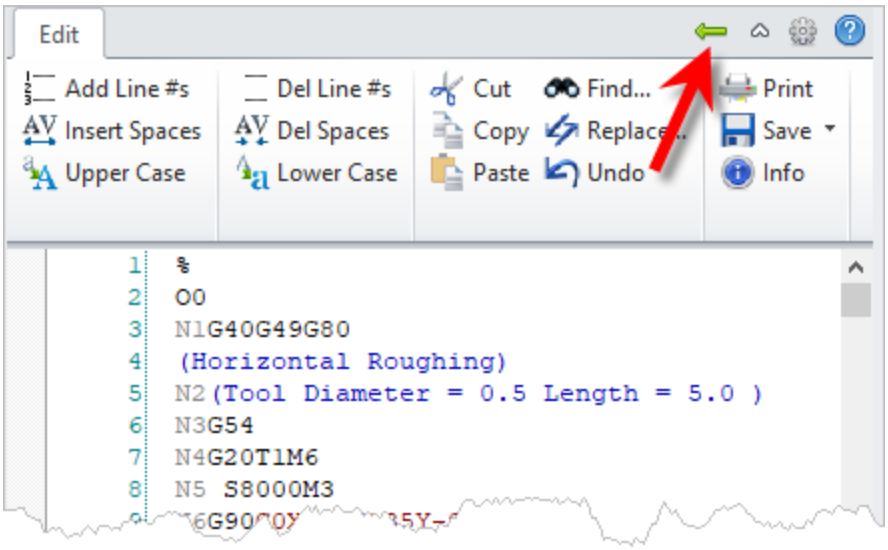
To access the G-Code Editor from an operation that you have generated within the MILL module simply click on the G-Code icon within the mop folder as shown below:



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.

© 2026 MecSoft Corporation



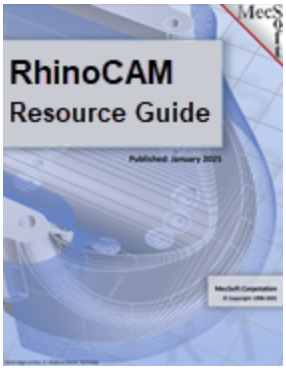
G-Code Browser: Tool Crib menu item

© 2026 MecSoft Corporation

Find More Resources

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

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

© 2026 MecSoft Corporation

Index

- A -

Add Line Numbers 96

- C -

Colors
- Cut Stock 20
- Cutting Tool 21
- Stock 20
- Text Editor 17
- Tool Motion Display 18

Copy Text 106, 128
Cut Text 104, 126
Cutting Tools
- Crib 65
- Library 62
- Preferences 21

- D -

Delete
- a G-Code File from the Project 91

Delete Line Numbers 100
Delete Spaces 101
Display Toolbar
- Toggle Follow Toolpath Display 32, 93
- Toggle Part Visibility 32, 93
- Toggle Segment Toolpath Display 32, 93
- Toggle Stock Visibility 32, 93
- Toggle Toolpath Visibility 32, 93
- Toggle Trace Toolpath Display 32, 93

- E -

Edit
- G-Code File 87

Edit > Select Full Line 119
Edit Tab 32, 93

- F -

Find Text 109

- G -

G-Code
- Add Line Numbers 96
- Copy Text 106, 128
- Cut Text 104, 126
- Delete Line Numbers 100
- Delete Spaces 101
- Find Text 109
- From the MILL Module 145
- Go to Bottom 120
- Go to Next Feedrate 125
- Go to Next Spindle 124
- Go to Next Tool 122
- Go to Top 121
- Information 68
- Insert Spaces 97
- Load 67
- Make Lower Case 102
- Make Upper Case 98
- Merge Files 69
- Output 86
- Paste Text 107, 129
- Print G-Code 114
- Replace Text 110
- Save 84
- Save As 116
- Save G-Code 115
- Show Info 117
- Transform 73
- Undo Last Edit 112, 131
- XY Instance 82
- Z Instance 80

G-Code Editor
- About 5
- Features of 6
- Overview 5
- Understanding 7
- Workflow 7

© 2026 MecSoft Corporation

- H -

Help 38

- I -

Information
G-Code 68
on a G-Code File 90
Insert Spaces 97

- L -

Load G-Code 67

- M -

Machine Tool Setup 42
Make Lower Case 102
Make Upper Case 98
MCS Setup 42
Merge Files 69
MILL Module 145
Mirror G-Code 73
Move G-Code 73

- N -

Navigate
Go to Bottom 120
Go to Next Feedrate 125
Go to Next Spindle 124
Go to Next Tool 122
Go to Top 121

- O -

Output G-Code 86

- P -

Part
Toggle Visibility 32, 93
Paste Text 107, 129
Preferences 13
Cutting Tools 21
Simulation 22, 135
Stock 20
Text Editor 17
Toolpath 18
User Interface 14
Print G-Code 114
Project
Tab 27, 40
Tree 29

- R -

Replace Text 110
Rotate G-Code 73

- S -

Save As 116
Save G-Code 84, 115
Scale G-Code 73
Select
Full Line 119
Show G-Code Info 117
Simulate
G-Code File 88
Tab 35, 133
Simulation
Display Levels 140
Mode 137
Pause 142
Play 138
Preferences 22, 135
Speed 136
Step Thru 139
Stop 143
To End 141
Stock
Box 48
Cylinder 54
Delete 59
from Selection 60
Menu 47
Part Box 51
Part Cylinder 56
Preferences 20

© 2026 MecSoft Corporation

Stock
Toggle Visibility 32, 93
Transparency 20

- T -

Text Editor Preferences 17
Toggle Follow Toolpath Display 32, 93
Toggle Part Visibility 32, 93
Toggle Segment Toolpath Display 32, 93
Toggle Stock Visibility 32, 93
Toggle Toolpath Visibility 32, 93
Toggle Trace Toolpath Display 32, 93
Tool Crib 65
Tool Library 62
Toolpath
Display width 18
Follow Display 32, 93
Preferences 18
Segment Display 32, 93
Toggle Visibility 32, 93
Trace Display 32, 93
Tools (See Cutting Tools) 62
Transform G-Code
Mirror 73
Move 73
Rotate 73
Scale 73

- U -

Undo Last Edit 112, 131
User Interface 13
Display Toolbar 32, 93
Edit Tab 32, 93
G-Code Browser 24
Preferences 14
Project Tab 27, 40

- W -

Where to go for more help 147

- X -

XY Instance G-Code 82

- Z -

Z Instance G-Code 80

© 2026 MecSoft Corporation