

Mill

OPERATING MANUAL



7/1/14

G0107

This manual covers the operation of the Mill Control using Mach 3.

Formatting Overview:

- Menus, options, icons, fields, and text boxes on the screen will be bold (e.g. the **Help** icon).
- Clickable buttons will be bold and within brackets (e.g. the **[OK]** button).
- Directory names, commands, and examples of editing program files will appear in Courier New font

This manual as well as all other MachMotion manuals can be found at www.MachMotion.com

Copyright © 2014, MachMotion
All rights reserved.

Mill

G0107

Table of Contents

1	INTRODUCTION	4
1.1	Mach3 Startup	4
1.2	Overview	4
2	HOMING	5
3	PROGRAMMED MOVEMENT	6
3.1	MDI	6
3.2	G-Code Files.....	6
3.2.1	G-Code File Controls.....	7
3.2.2	Running a G-Code File Example	7
3.2.3	Tool Path Screen.....	8
3.3	Advanced Controls.....	8
3.3.1	Run From Here.....	9
3.3.2	Single Line.....	9
3.3.3	Mist.....	9
3.3.4	Flood	9
4	TOOLS, TOOL OFFSETS, AND CUTTER COMPENSATION	10
4.1	Tools and Tool Offsets.....	10
4.2	Cutter Compensation.....	11
5	SPINDLE CONTROL	11
5.1	G-Code Spindle Control.....	11
5.2	Manual Spindle Control.....	11
6	PART OFFSETS	12
7	APPENDIX A – STARTUP PROCEDURE	13
8	APPENDIX	14
8.1	Warranty Information	14
8.2	Additional Resources	14

1 INTRODUCTION

1.1 Mach3 Startup

To open Mach3 double click on the desired profile icon on the desktop or open the Mach3 Loader and choose a profile from the list.



FIGURE 1 - MACH3 ICONS

1.2 Overview

This manual gives the process for basic operation of a mill using the MachMotion Ultimate screen.



FIGURE 2 - ULTIMATE SCREEN OVERVIEW

For a quick reference, the description of each zone is located below.

1. The axis DROs show the current part location and homing state (The axis labels turn green when the axis is homed).
2. The tool path window shows the shows the path the tool will take when a G-Code file is loaded.
3. The axis DROs display the remaining travel distance to complete the current line of G-Code. The status LEDs give information on the status of different modes and features.
4. Spindle control and status
5. Feedrate control and status
6. The G-Code window displays the text of a loaded G-Code file.
7. G-Code control and other functions
8. Main screen tabs
 - a. Prog Run: Primary screen for operations
 - b. Tool Path: A larger tool path screen for more detailed tool path inspection
 - c. Offsets: Detailed offset control
 - d. MDI: Tab for controlling the machine using the MDI (Manual Data Input) line
 - e. Diagns: This contains input and output status and system level controls. It is primarily used for setup and troubleshooting.
9. Status bar shows errors, alarms and status messages
10. Sub menu with commonly used control menus
11. File window showing the name of a loaded G-Code file

2 HOMING

To home the mill, press the Home button on the sub menu.



FIGURE 3 - SUB MENU

To home all the axes press the HOME ALL button. The axes can also be homed individually using the individual axis buttons.



FIGURE 4 - HOMING MENU

3 PROGRAMMED MOVEMENT

3.1 MDI

To command a movement using the MDI feature, press the MDI button on the sub menu.



FIGURE 5 - MDI MENU SELECTION

Press the big black MDI button.



FIGURE 6 - MDI CONTROL

Enter the desired G-Code command into the field and press ENTER on the keyboard to initiate the command.
Example: G0 G91 X2



FIGURE 7 - EXAMPLE MDI COMMAND

3.2 G-Code Files

The primary method of controlling Mach3 is using G-Code files. G-Code files can be hand written, generated by a wizard, or generated from CAD files using a CAM program.

3.2.1 G-Code File Controls

The controls for G-Code files are shown below.

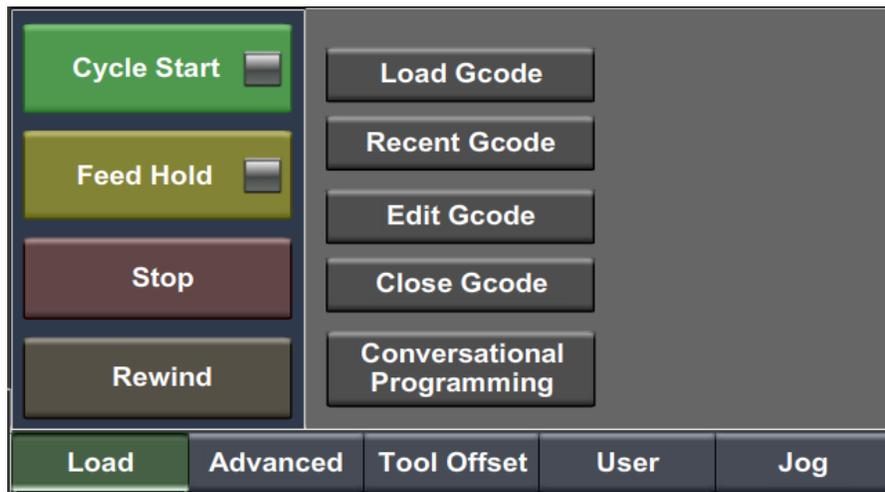


FIGURE 8 - G-CODE CONTROLS

1. Cycle Start – Starts a loading G-Code file
2. Feed Hold – Pauses a running G-Code file which can then be restarted by pressing Cycle Start
3. Stop – Stops a running G-Code file or other commanded movement
4. Rewind – Rewinds a loaded G-Code file to the beginning
5. Load Gcode – Opens a file browser to select an existing G-Code file
6. Recent Gcode – Opens a selection window with the ten most recent run G-Code files
7. Edit Gcode – Opens a loaded G-Code file in Notepad to allow easy editing
8. Close Gcode – Closes the loaded G-Code file
9. Conversational Programming – Opens the Newfangled Wizards

3.2.2 Running a G-Code File Example

To run a G-Code file, follow the steps below:

1. Press the LOAD GCODE button (Figure 8)
2. Select a G-Code file from the file menu window and press OK
3. Jog the machine to the work piece zero point and zero all the axes by selecting Offsets and then Zero All in the sub menu



FIGURE 9 – OFFSETS SUB MENU SELECTION



FIGURE 10 - ZERO ALL

4. Press CYCLE START to run the program (**Error! Reference source not found.**, Figure 8)
5. If it is necessary to stop in the middle of a program to inspect the part press FEED HOLD (**Error! eference source not found.**, Figure 8)
6. If it is necessary to end a program before it has completely run press STOP (Figure 8)

3.2.3 Tool Path Screen

Below are the controls to manipulate the tool path screen:

1. Pan – right click with the mouse and move the track ball to move the part around
2. Rotate – left click with the mouse and rotate the part with the track ball
3. Zoom – press shift on the keyboard and left click with the mouse, then zoom by rolling the track ball forward and backward

Note: Do not manipulate the tool path window or regenerate it while a G-Code file is running. It can adversely affect G-Code execution.

3.3 Advanced Controls

The *Advanced* tab has a number of features that can be used to control G-Code files beyond starting and stopping.

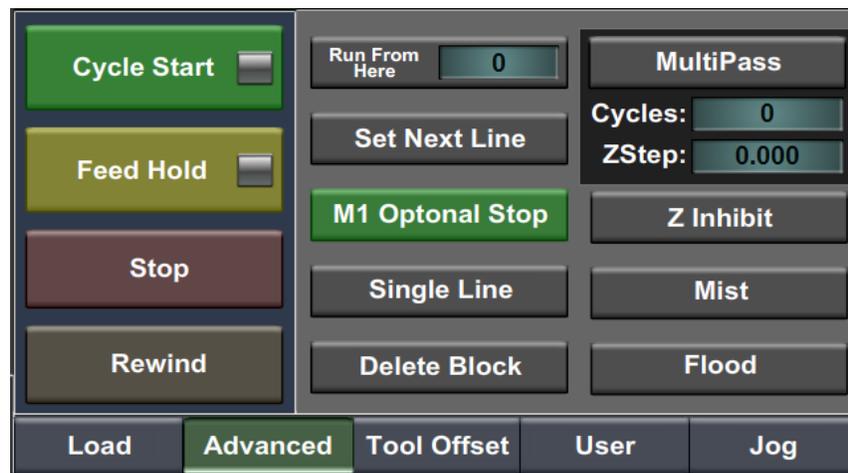


FIGURE 11 - ADVANCED CONTROLS

3.3.1 Run From Here

If for whatever reason a program needs to be started in the middle, the RUN FROM HERE button should be used. To use the feature follow these steps:

1. Use the up and down arrows underneath the G-Code window (See Figure 2) to select the line to start the program from
2. Press the RUN FROM HERE button
3. Press the CYCLE START button
4. Enter an appropriate value in the Rapid Height field (See Figure 12)

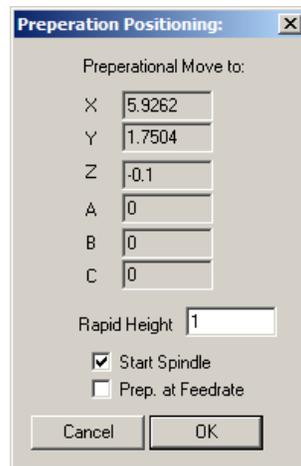


FIGURE 12 - PREPARATION MOVE WINDOW

5. Press OK to move to the start point and start the spindle
6. Press CYCLE START to begin the file

3.3.2 Single Line

To use the single line feature press the SINGLE LINE BUTTON and then press CYCLE START. The program will execute a single line of G-Code each time the CYCLE START button is pressed.

3.3.3 Mist

Mist turns on and off the mist control.

3.3.4 Flood

Flood turns on and off the flood control.

4 TOOLS, TOOL OFFSETS, AND CUTTER COMPENSATION

While Mach3 does have cutter comp and tool definitions, it has limitations. Mach3 is limited by part size and complexity. Due to this if a the G-Codes being run on the mill are generated using a CAD/CAM package, MachMotion recommends using the cutter comp in the CAM package and bypassing Mach3's cutter comp.

4.1 Tools and Tool Offsets

Defining tools has two main steps; setting up the master tool and entering additional tools.

To define the master tool, complete the following steps:

1. Set the tool number to 1
2. Set the tool length (H) to 0
3. Put the master tool in the spindle
4. Touch the tool off on a reference position, like the table or the top of a fixture
5. Zero the Z axis

Note: The master tool does not have to be an actual tool. It can be a blank rod or the bottom of the empty spindle. The main point is that it should be a specific and consistent reference point.

To define additional tools complete the following steps:

1. Define the master tool as stated above
2. Change the tool number to the number for the new tool
3. Put the new tool in the spindle
4. Touch the tool of on the same surface as the master tool
5. Go to the **Tool Offset** tab (See Figure 13)
6. Press the SET TOOL LENGTH button
7. Repeat steps 2-6 for each additional tool

Tool diameters can be set in the *D:* field (Figure 13) or in the *Tool Table* (Figure 14). Tool diameters are required only if cutter compensation is being used.



FIGURE 13 - TOOL WINDOW

Tool	Description	Diameter(D)	Height (H)	Diam. Wear	HeightWear
0	Ref. Tool	0.000000	0.000000	0.000000	0.000000
1		0.500000	1.000000	0.000000	0.000000
2		0.010000	5.000000	0.000000	0.000000
3		0.010000	5.000000	0.000000	0.000000
4		0.000000	0.000000	0.000000	0.000000
5		0.000000	0.000000	0.000000	0.000000

All Tool Entries are in your default setup measurement units regardless of G20/G1 modes.

FIGURE 14 - TOOL TABLE

4.2 Cutter Compensation

To use cutter compensation on a part the following G-Code is required:

1. Tool Definition – M6 T{desired tool number}
2. Cutter Comp On (Left/Right) – G41/G42
3. Cutter Comp Off – G40

Note: To use cutter comp on a contour it is necessary to have a lead in and lead out.

5 SPINDLE CONTROL

5.1 G-Code Spindle Control

The spindle is controlled through G-Code using the M-Codes M3 (Clockwise), M4 (Counterclockwise), and M5 (Off). To control the spindle speed in RPMs an S word is added.

Example: M3 S2000

This would turn the spindle on in the clockwise direction at 2000 RPM

5.2 Manual Spindle Control

To control the spindle separately from G-Code use the spindle control on the screen.

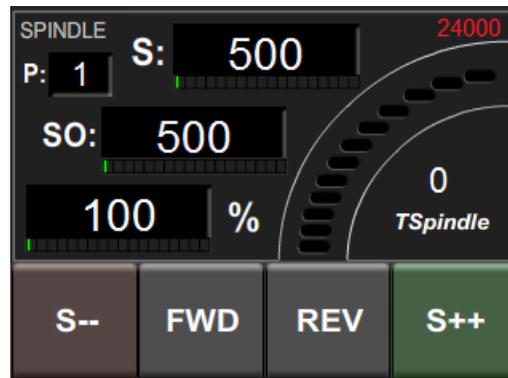


FIGURE 15 - SPINDLE CONTROL

1. S: – Commanded Speed
2. SO: – Spindle Override Speed
3. % – Override percentage
4. P: – Pulley number
5. TSpindle – Spindle speed (if feedback sensor is installed)
6. FWD – Turns spindle on clockwise
7. REV – Turns spindle on counterclockwise
8. S-- – Reduces override percentage by 10%
9. S++ – Increases override percentage by 10%
10. Red Number – Maximum configured spindle speed

6 PART OFFSETS

All G-Code files have their own coordinate system. In order to allow parts to be located on the table at any desired location the part offset can be defined to adjust the actual location of the part on the table. See Running a G-Code File Example for how to adjust it.

Part offsets can be defined and saved using G55-G59P253. The functionality is designed to allow different tooling setups to have predefined zero points to allow for streamlined setup.

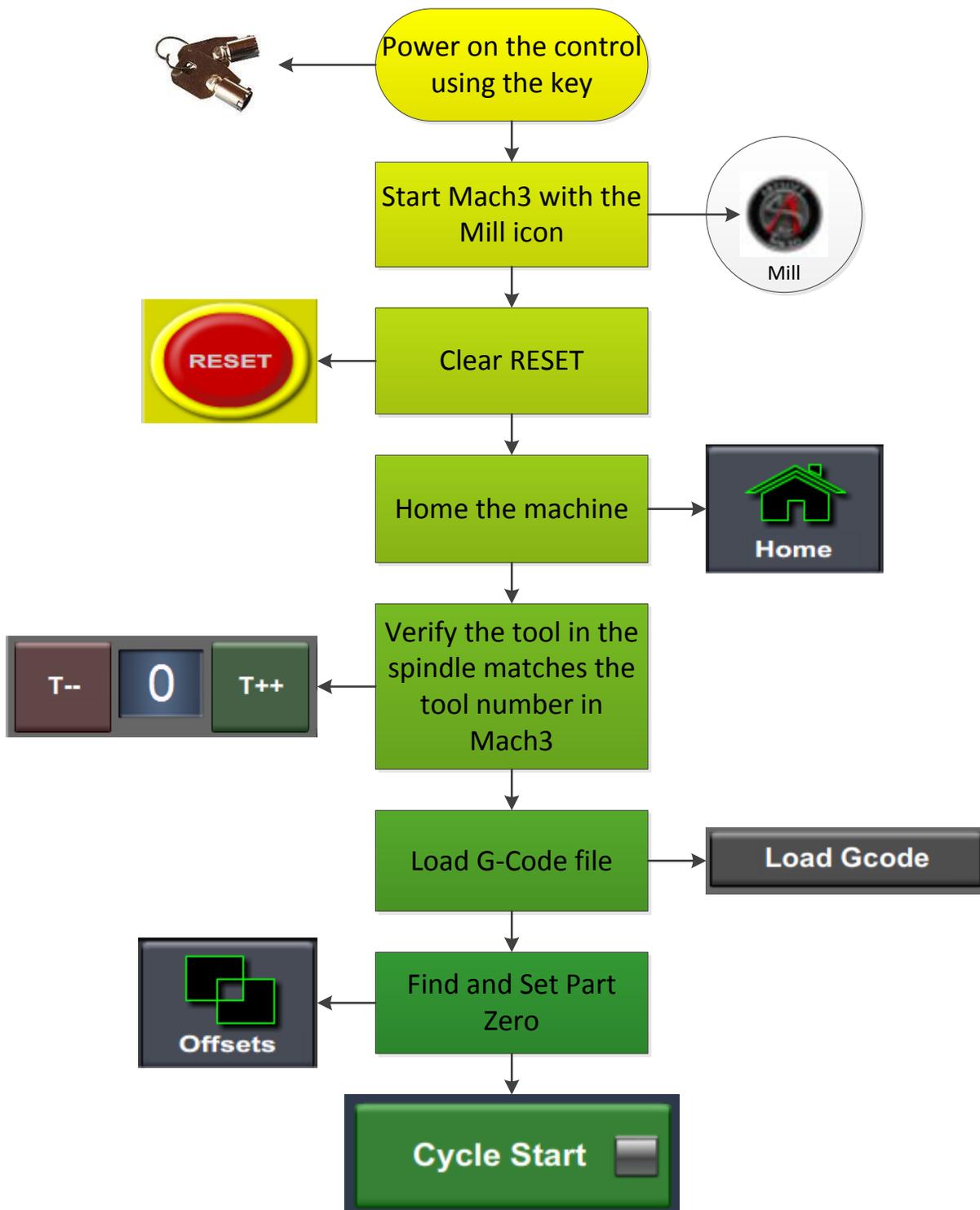
To set up saved part offsets do the following:

1. Go to the **Offsets->Work Offsets** tab
2. Set the desired offset in the *Active Work Offset* field at the top of the page
3. Jog the machine to the desired zero position
4. Set all axes to zero from the **Offsets** menu on the sub menu

*Note: the **Offsets->Work Offsets** tab shows the current program coordinates, part offsets, and machine coordinates.*

See the G-Code User Guide for more details on using offsets in G-Code.

7 APPENDIX A – STARTUP PROCEDURE



8 APPENDIX

8.1 Warranty Information

MachMotion guarantees all products to be free from manufacturer defects for a period of one year from the date of purchase. Products which prove to be defective under normal conditions and proper use, during the warranty period, will be repaired or exchanged free of charge. For warranty service the customer must contact MachMotion for an RMA number and then return the defective product to MachMotion. If a product is sent to MachMotion without an RMA number, the product may be misdirected or delayed. When a product or part is exchanged, any replacement item becomes the customer's property and the replaced item becomes MachMotion's property.

If the defect is found to be caused by improper use or installation, the warranty is void. Otherwise the product will be repaired or exchanged and returned to the address located on the Product Return/Repair Form.

MachMotion will cover ground shipping cost for the replacement/repaired product being returned to the customer. MachMotion does offer expedited shipping at the customer's expense.

If a replacement product is needed quickly, a replacement can be sent immediately. In this case the customer will be charged for the replacement part at the time of the order and be refunded that charge when the defective component is returned to MachMotion, assuming the defective item falls under the warranty guidelines. MachMotion will issue a refund within two work weeks after receiving the faulty component.

8.2 Additional Resources

Additional manuals and resources can be found at MachMotion.com

The Mach Motion Team

<http://www.machmotion.com>

14518 County Road 7240, Newburg, MO 65550

(573) 368-7399 • Fax (573) 341-2672