

Waterjet Control Operating Manual



1. Introduction

1.1 Overview

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



Figure 1 - 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. Jet and Abrasive control
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
 1. Prog Run: Primary screen for operations
 2. Tool Path: A larger tool path screen for more detailed tool path inspection
 3. Offsets: Detailed offset control
 4. MDI: Tab for controlling the machine using the MDI (Manual Data Input) line
 5. Diagnostics: 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 water jet, press the Home button on the sub menu.



Figure 2 - 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 3 - Homing Menu

3. Manual Movement

3.1 Operator Panel

1. Jog Buttons - Used to move the selected axis in either direction
2. Axis Selector Knob - Selects which axis the mill will jog in
3. Jog Control Knob - Controls the jog speed and mode (Step or Continuous)
4. Feedrate Override Knob - Controls the speed that G-Code files run at as a percentage of the commanded speed in the G-Code file
5. Emergency Stop Button - Stops all motion
6. Cycle Start Button - Starts the loaded G-Code file
7. Feed Hold Button - Pauses a running G-Code file

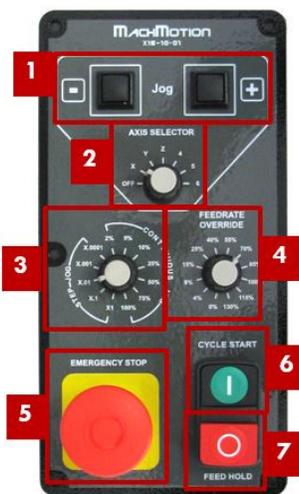


Figure 4 - Operator Panel

3.2 Pendant

1. Enable Button - Must be held down while jogging with the pendant
2. Jog Wheel - Controls jog direction and speed

3. Axis Selector Knob - Selects which axis the mill will jog in
4. Jog Control Knob - Controls the jog speed and mode (Step or Continuous)

Note: The Axis Selector Knob must be set to off to jog the water jet using the operator panel or keyboard

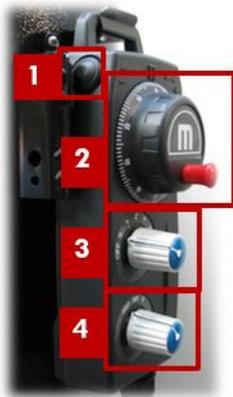


Figure 5 - Pendant

3.3 Keyboard and Mouse

3.3.1 Keyboard

Keyboard operation is just like any other computer.

3.3.2 Mouse

The ball of the mouse controls the cursor. Left and right click buttons are on either side of the mouse.

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

4. Programmed Movement

4.1 MDI

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



Figure 6 - MDI Menu Selection

Press the big black MDI button.



Figure 7 - MDI Control

Enter the desired movement or command into the field and press ENTER on the keyboard to initiate the command.
Example: G0 G91 X2



Figure 8 - Example MDI Command

4.2 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.

4.2.1 G-Code File Controls

The controls for G-Code files are shown below.



Figure 9 - G-Code Controls

1. Cycle Start - Starts a loading G-Code file
2. Feed Hold - Pauses a running G-Code file
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

4.2.2 Running a G-Code File Example

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

1. Press the LOAD GCODE button (Figure 9)
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 axis



Figure 10 - Offsets Sub Menu Selection



Figure 11 - Zero All

4. Press CYCLE START to run the program (Figure 4, Figure 9)
5. If it is necessary to stop in the middle of a program to inspect the part press FEED HOLD (Figure 4, Figure 9)
6. If it is necessary to end a program before it has completely run press STOP (Figure 9)

4.2.3 Tool Path Screen

Below are the controls to manipulate the tool path screen:

4. Pan - right click with the mouse and move the track ball to move the part around
5. Rotate - left click with the mouse and rotate the part with the track ball
6. 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.

5. 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 the G-Codes being run on the water jet are generated using a CAD/CAM package MachMotion recommends using the cutter comp in the CAM package and bypassing Mach3's cutter comp.

5.1 Tools and Tool Offsets

Water jets will only need one tool defined with the diameter set to the kerf width, and this is only necessary if Mach3 cutter compensation is being used.

5.2 Cutter Compensation

To use cutter comp 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.

6. Jet Control

6.1 G-Code Jet Control

The jet is controlled through G-Code using the M-Codes M480 (Jets On/Abrasive On) and M481 (Jets Off/Abrasive Off).

6.2 Select Jets/Abrasives

The operator can select the appropriate jets (up to three) and abrasive (up to three) outputs for operation. When the control executes M480 to turn on jets and abrasives, only the selected jets and abrasives will turn on. At least one jet must be selected for operation.

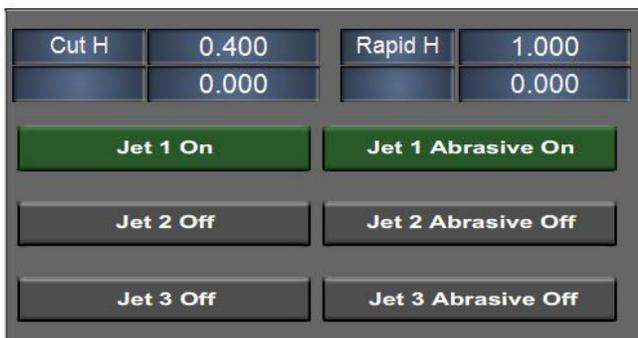


Figure 12 Jet/abrasive selection

6.3 Manual Jet Control

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



Figure 13 - Jet Control

1. Backup – See Section 6.3
2. Forward – See Section 6.3
3. Lost Cut Recovery – See Section 6.3
4. Pierce Delay: – Pierce delay value in seconds
5. Simulate On/Off – Controls simulate mode
6. Plate Alignment – See Section 6.4
7. Pump Control ON/OFF – Pump Control
8. Jets ON/OFF – Jet Control
9. Abrasive ON/OFF – Abrasive Control

6.4 Lost Cut Recovery

To use lost cut recovery follow the following steps:

1. Scroll through the G-Code to select the line to begin cutting on
2. Move the machine to the EXACT X and Y position where you want the machine to begin cutting using the **Forward** and **Backup**

*Note: The **Forward** and **Backup** buttons will move the machine to the final commanded position of a G-Code line.*

*Note: If you want to start in the middle of an arc or line segment you must turn Simulate on and press **Cycle Start**. The machine will begin moving along the profile. Press **Stop** when the cutter reaches the point that you want to begin cutting from.*

3. Press the **Lost Cut Recovery** button (See Figure 15)

6.5 Plate Alignment

The purpose of plate alignment is to compensate for the material not being exactly straight in relation to the table's axes.

To use Plate Alignment:

1. Locate jet along one edge of the material
2. Press the **Plate Alignment** button

Note: When it is pressed the first time a window will pop up asking if the operator wants to restore the last plate alignment. This allows the operator to restore the previous settings if a shutdown is required or if it got turned off accidentally.

3. After the dialog box is closed the LED on the **Plate Alignment** button will turn on
4. Move the jet to a point further down the same edge of the material
5. Press the **Plate Alignment** button a second time

7. 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.

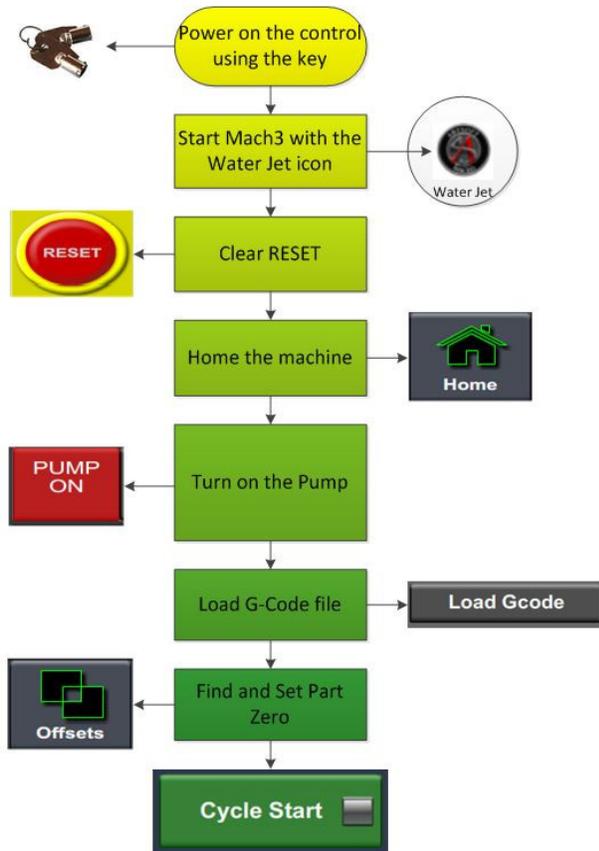
Part offsets can be defined and saved using G55-G59P253. The functionality is designed to allow different tooling setups to have predefined zero points to allow for streamlined setup.

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

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

8. Appendix A



9. Appendix B

Warranty Information

MachMotion warranty policy is subject to change. Updated information is available at our website:
<https://machmotion.com/warranty>

The MachMotion Team
<http://www.machmotion.com>
 14518 County Road 7240, Newburg, MO 65550
 (573) 368-7399 • Fax (573) 341-2672