

Router

- [2000 Series Router Operating Manual](#)

2000 Series Router Operating Manual



1. Introduction

1.1 Control Startup

To open the control software double-click on the profile icon on the desktop.



Control Icon

1.2 Overview

This manual gives the process for basic operation of a using the Mill / Router MachMotion control. The screen is shown below, followed by a brief summary of the different features of the screen. The number shown in the screenshot refers to a brief description below the image.

Control Screen Overview:

GCode Tab



1. Tool Path

- Regen ToolPath - Refresh the toolpath of the gcode
- View Top - Top view of the part
- View ISO - Side view of the part

2. File

- Recent - Load a recently loaded gcode program
- Load - Load a program from the computer or flash drive
- Edit - Edit the code that is loaded into the software
- Close - Close the gcode that is currently loaded in the software

3. Control

- Cycle Start - Starts the gcode from from the beginning of the part
- Feed Hold - Pauses the gcode program and keeps the spindle running
- Cycle Stop - Stops the gcode program from running
- Reset - Resets the alarm and also enables the machine

4. Status

- Status - Displays any current messages (Home All Pressed, Cycle Start Pressed, etc.)
- State - Displays the current state of the machine (Run, Feedhold, etc)
- Cycle Time - Displays how long the gcode has been running
- Date - Date and time of the timezone of the control

5. DROs (Axis Digital Readouts)

- MDI - Opens up a window that allows for gcode commands
- Viewing Part - Shows the part coordinates or machine coordinates of the machine

6. Active Modals

- Active Offset - Shows the current active fixture offset (G54, G55, etc)

7. Tool Display

- T - The current tool number selected
- Next Tool - The next tool that the gcode will need
- Diameter - Diameter of the tool
- Length - Length of the tool

8. Feedrate Display

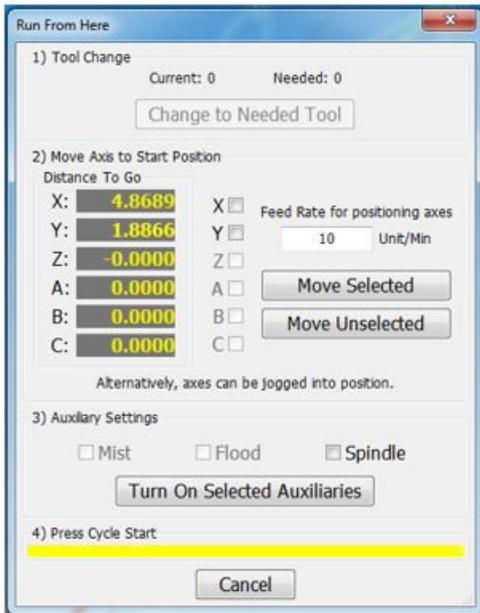
- F - The current feedrate commanded
- Feed OV - The current Feedrate Override utilizing the Feedrate Override knob on the operating panel (0-200%)
- Rapid OV - The current Rapid Override utilizing the Rapid Override knob on the operating panel (0-200%)

9. Spindle Display

- S - The current spindle speed
- TSpeed - The current spindle speed feedback
- FWD - Turns on if the spindle is moving Forward
- REV - Turns on if the spindle is moving in Reverse
- Spindle OV - The current spindle override utilizing the spindle override knob on the operating panel (0-200%)
- Spindle Load - The current spindle load utilizing the spindle speed feedback
- Range - Displays the current spindle range (spindle pulley)

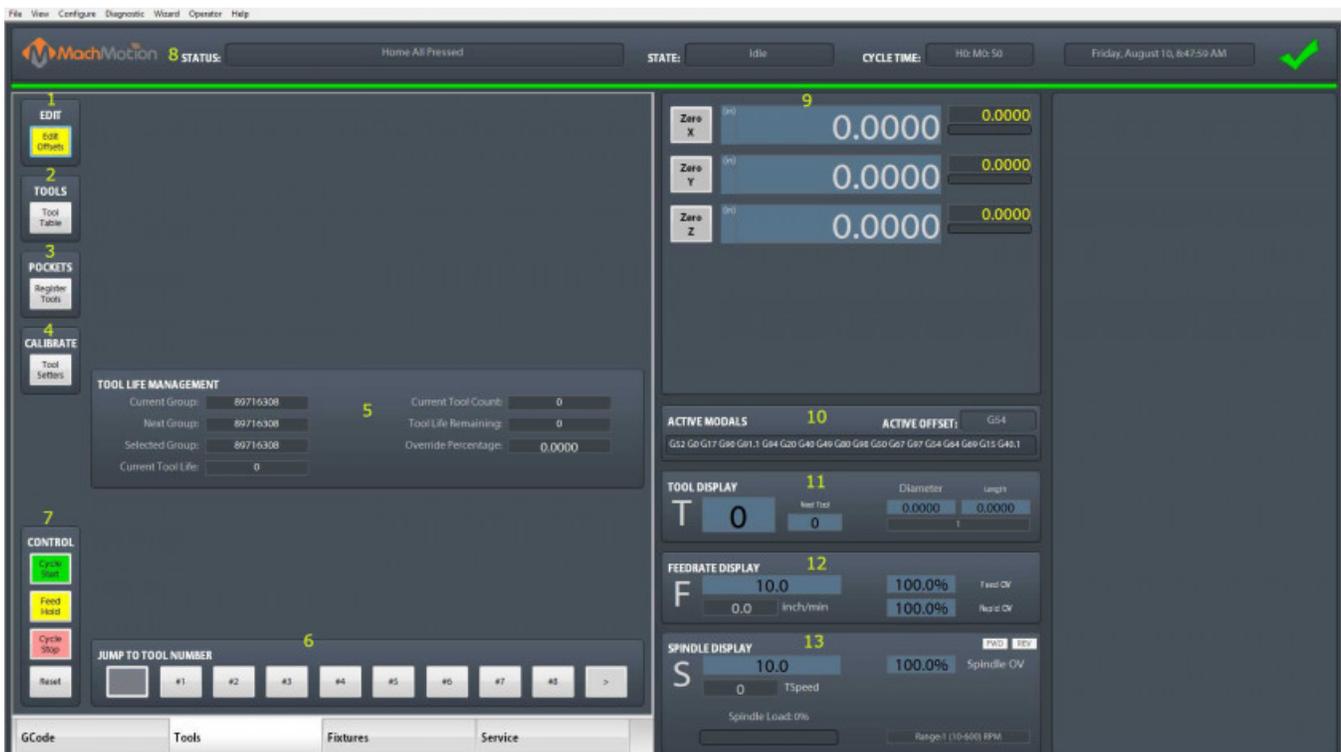
10. Advanced

- Single Block - If active the software will go line by line through the gcode when you press the cycle start button
- Block Delete - Deletes the block of gcode that is selected
- Part Counter - Displays the number of parts that the machine has produced
- M1 OPT Stop - If active the software will stop at any M1 commands in the gcode program and waits for cycle start
- Alexsys - Opens up the conversational assistant Alexsys in another window
- Dry Run - If active the software will ignore all mist or flood commands
- M-S-T Lock - If active the software will ignore all Macro codes, Spindle codes and Tool commands
- Run From Here/File Resume:
 1. Use the Up/Down arrows in the G-Code file window to select the line to start from
 2. Press the **[Run From Here]** or **[File Resume]** button
 3. Select the **[Change to Needed Tool]** button if applicable (See Image Below)
 4. Enter desired value in the Feed Rate field (See Image Below)
 5. Select axis to move and press **[Move Selected]** button (See Image Below) to move the axis into position
 6. Select desired Auxiliary Settings and press **[Turn On Selected Auxiliaries]** button
 7. Press **[Cycle Start]** to begin the file at selected starting line File Resume - If selected this will start the program at the selected gcode line and resumes the file



- Collapse v - If selected this will minimize the Advanced buttons

Tools Tab



1. Edit

- Edit Offsets - No offset changes can be made unless this value is yellow. It also displays the zero buttons near the DROs to the right.

2. Tools

- Tool Table - View and edit tool table

3. Pockets

- Register Tools - Register and save tools

4. Calibrate

- Tool Setters - Select this to add or remove a tool setter. For more information, follow this link: <https://support.machmotion.com/books/cnc-controls/page/creating-tool-setters---2000-series>

5. Tool Life Management - Will be added in the future

6. Jump to Tool Number

- 1,2,3,4 etc. - Tool number (Tool change). Just press the button and the tool will change.

For 7-13 see the [Gcode Tab] above for more details

Fixtures Tab - Manual Setup



1. Edit

- Edit Offsets - View and edit fixture offsets

2. Fixtures

- Fixtures Table - View and edit fixture table (G54, G55, G56, etc.)

3. X/Y Edge Finder Offset

- X+ - Zeros the X + offset
- X- - Zeros the X - offset
- Y+ - Zeros the X+ offset
- Y- - Zeros the Y- offset
- Edge Finder Diameter - This is the diameter of the edge finder used to calculate the new offset

based on the zero button pressed above.

4. Fixture Offsets

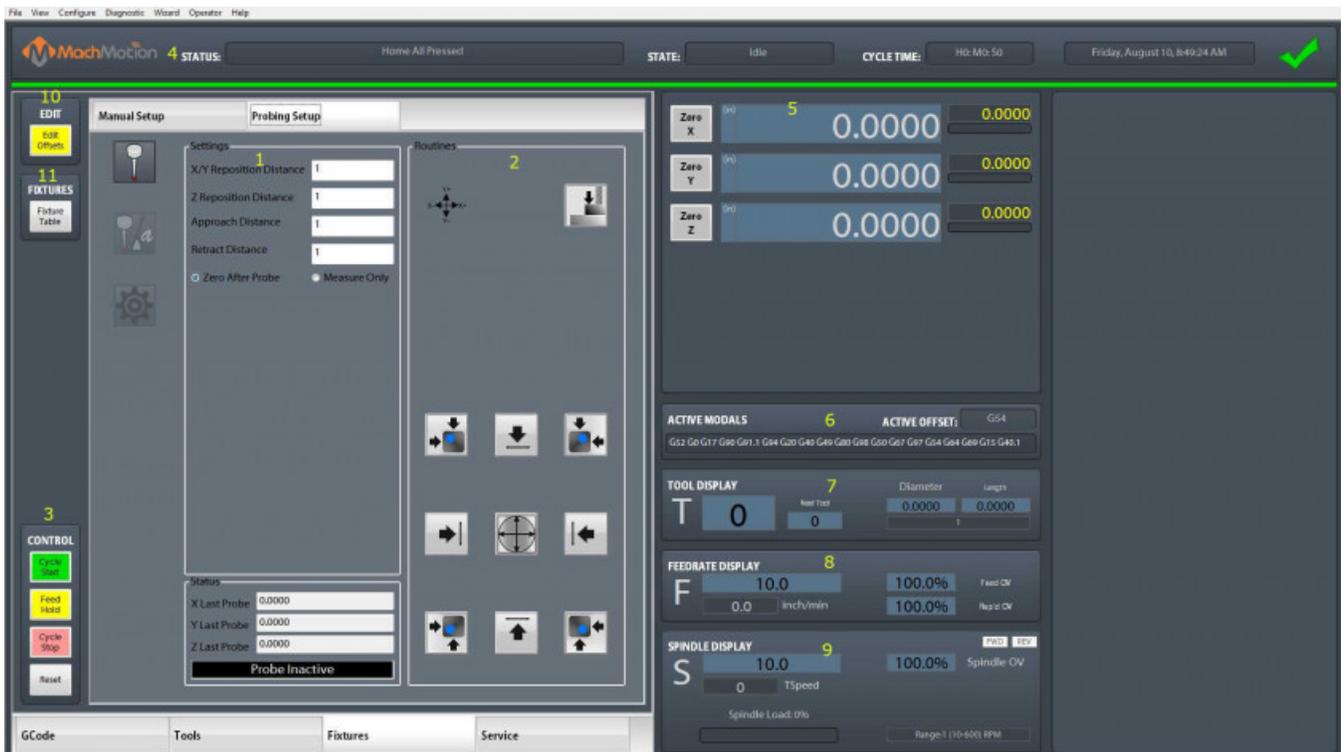
- X, Y, Z, etc. - Shows the currently loaded offset.

5. Rotary Axis

- X,Y, Z -

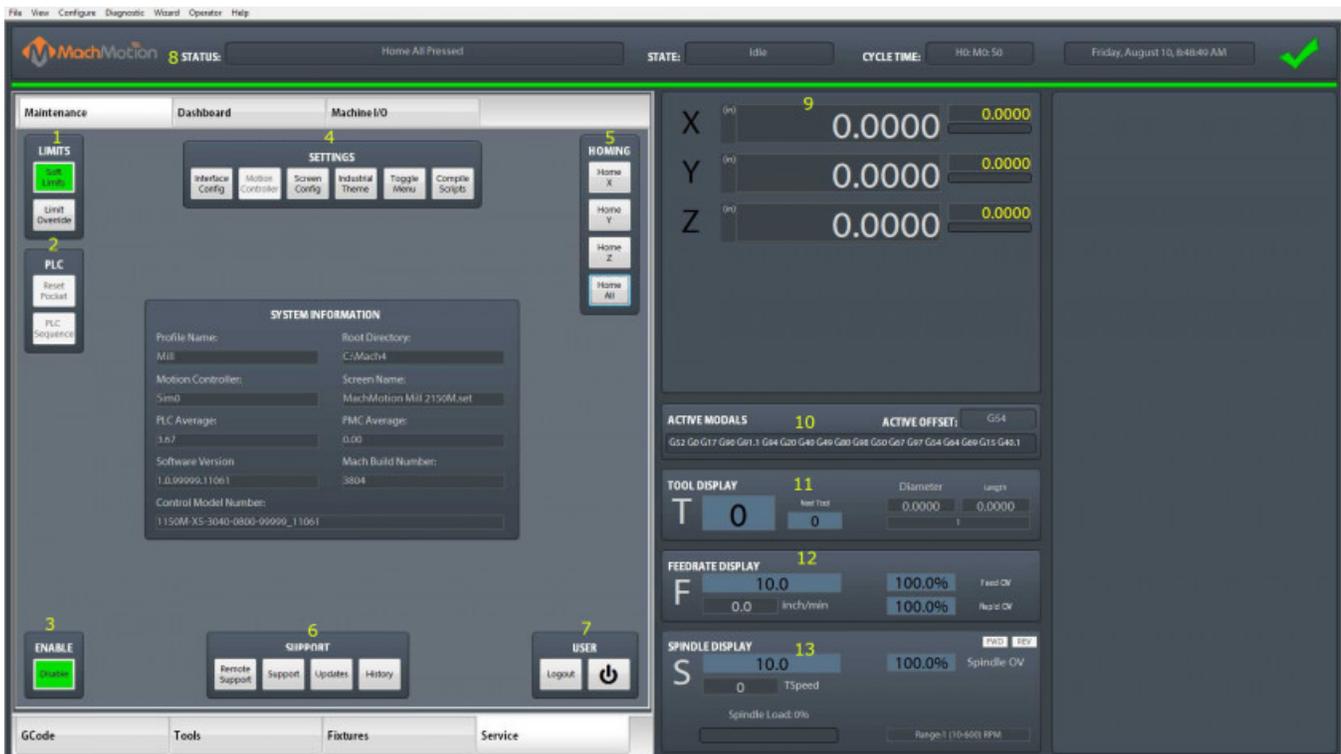
For 7-13 see the [[Gcode Tab](#)] above for more details

Fixtures Tab - Probing Setup



This tab is used to setup your part or measure a part. See this link for more information: <https://support.machmotion.com/books/software/page/machmotion-probing-wizard> .

Service Tab - Maintenance



1. Limits

- Soft Limits - Toggles software limits on or off
- Limit Override - Toggles to allow for the machine to move off a limit switch

2. PLC

- Reset Pocket - Only used for tool changers.
- PLC Sequence -

3. Enable

- Disable - Toggles if the machine is enabled or disabled

4. Settings

- Interface Config -
- Motion Controller - Opens the plugin for the Motion Controller
- Screen Config - Edit the screen layout
- Industrial Theme -
- Toggle Menu - Turns the menu on or off
- Compile Scripts - Refreshes (recompiles) programming scripts

5. Homing

- Home X - Homes the X axis
- Home Y - Homes the Y axis
- Home Z - Homes the Z axis
- Home All - Homes all axes

6. Support

- Remote Support - Starts a remote support session with MachMotion Technical Support
- Support - Opens up the online MachMotion Support Library
- Updates - Checks for updates for MachMotion Software
- History - Views status history and alarms

7. User

- Logout - Logs out of the Windows username
- Power - Turns off the computer

For 7-13 see the [[Gcode Tab](#)] above for more details

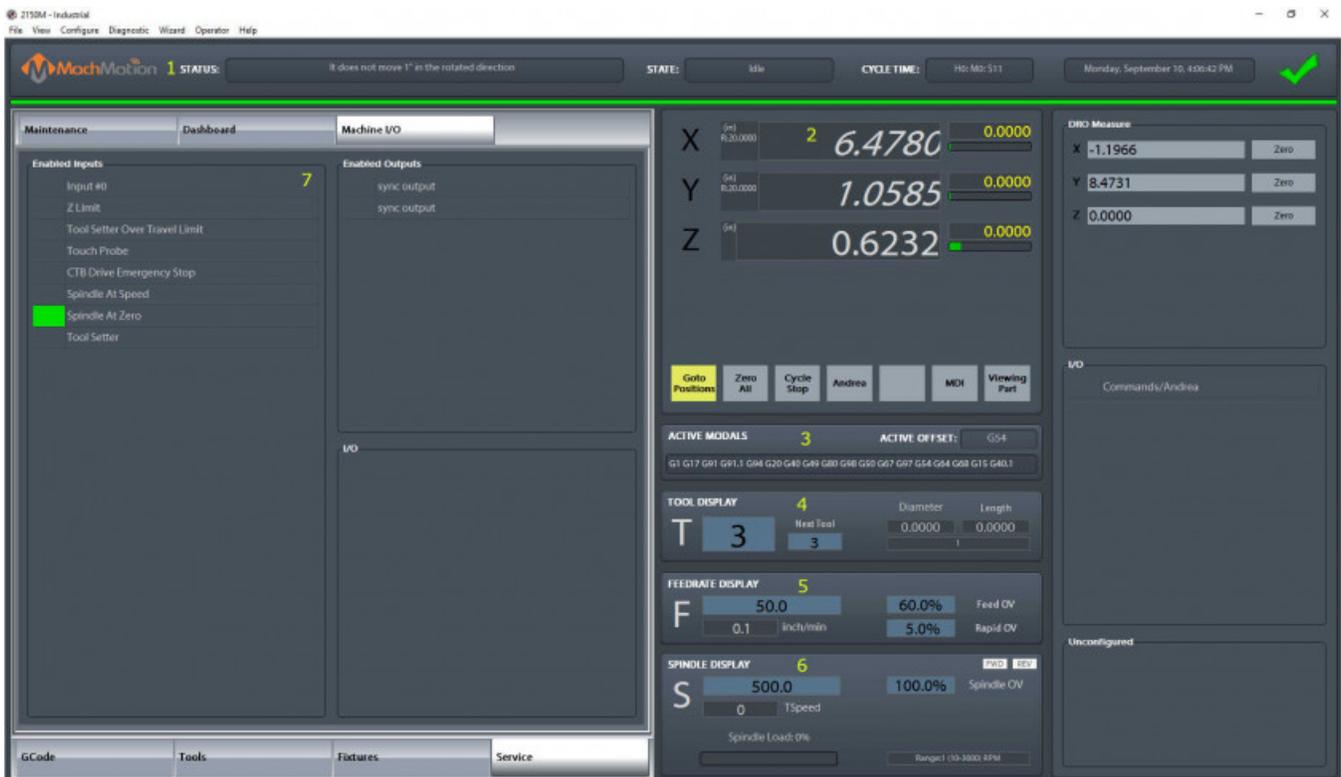
Service Tab - Dashboard

The screenshot displays the MachMotion 1 Service Tab Dashboard. The interface is organized into several sections:

- Top Bar:** Includes the MachMotion logo, a status indicator (1), a message "It does not move 1\" in the rotated direction", STATE: Idle, CYCLE TIME: HD:Mo:5:11, and the date Monday, September 10, 4:05:41 PM.
- Maintenance Tab:** A table showing G92 Offsets for X, Y, Z, A, B, and C axes, all set to 0.0000. A yellow '7' is displayed next to the offsets.
- Positioning:** X, Y, and Z coordinates are shown with their respective ranges and zero offsets. X: 6.4780 (range 0 to 9.20000), Y: 1.0585 (range 0 to 8.20000), Z: 0.6232 (range 0 to 0.0000).
- Buttons:** A row of buttons for Goto Positions, Zero All, Cycle Stop, Andrea, MDI, and Viewing Part.
- ACTIVE MODALS:** Shows 3 active modals and ACTIVE OFFSET: G54. A list of modal codes (G1-G15) is visible below.
- TOOL DISPLAY:** Shows tool T3 with a next tool of 3. Diameter and Length are both 0.0000.
- FEEDRATE DISPLAY:** Shows feedrate F50.0 (0.1 inch/min) with 60.0% Feed OV and 5.0% Rapid OV.
- SPINDLE DISPLAY:** Shows spindle speed S500.0 (0 TSpeed) with 100.0% Spindle OV. Spindle Load is 0% and Range is 00-3000 RPM.
- DIRC Measure:** Shows X: -1.1966, Y: 8.4731, and Z: 0.0000, all with Zero offsets.
- I/O:** Shows Commands/Andrea and Unconfigured sections.
- Bottom Bar:** Includes tabs for GCode, Tools, Fixtures, and Service.

The Dashboard is used to make the control just the way you want it! For more information, see this link: <https://support.machmotion.com/books/mach4-control-features/page/2000-series-commands-function-buttons-and-dashboard> .

Service Tab - Machine I/O



This tab is used for diagnostics and shows all your machine IO.

2. Homing

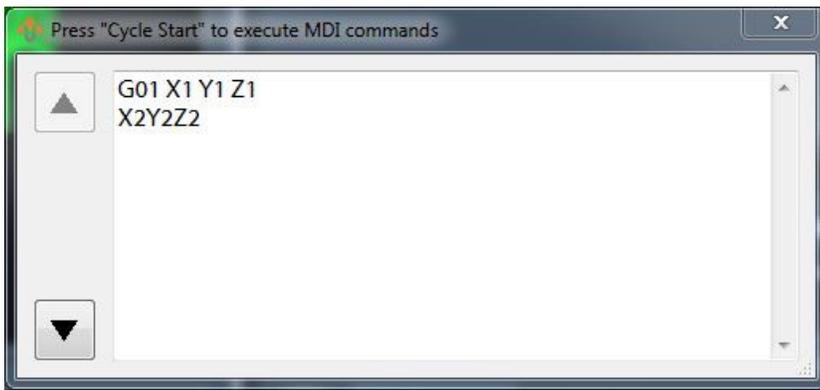
To home the Machine, begin by click the [**Reset**] button and then the [**Service**] tab and click on [**Home All**].

3. Programmed Movement

3.1 MDI

To command a movement using the MDI feature, press the [**MDI**] button.

Enter the desired G-Code command into the field and press [**Cycle Start**] to execute the command(s). The up/down arrow buttons will scroll through the history of cycled commands. Click the Red [**X**] to close the MDI window.



Example MDI Command

3.2 G-Code

The primary method of commanding motion 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.

3.2.6 Tool Path Screen

Below are the controls to manipulate the tool path screen:

- Zoom - Right click with the mouse and move mouse up/down or using the scroll wheel on the mouse
- Rotate - Left click with the mouse and rotate the part by moving the mouse
- Pan - Press and hold [**Ctrl**] on the keyboard and left click with the mouse, then pan by moving the mouse (one-hand control option is to use left and right mouse click and move the mouse. No [**Ctrl**] press needed)

4. Tools and Tool Offsets

See the Offset tab above.

5. Spindle Control

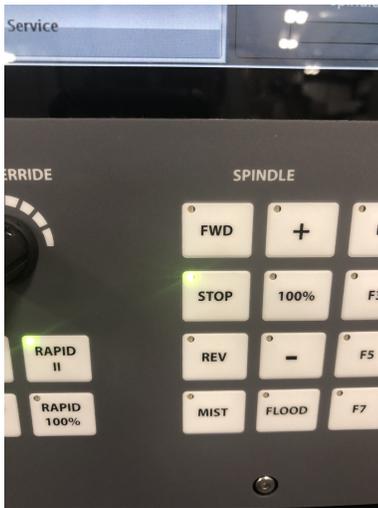
5.1 G-Code Spindle Control

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.

For example, M3 S2000 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 operating panel. The [**Spindle FWD**] turns the spindle on clockwise and the [**Spindle REV**] turns the spindle on counterclockwise.



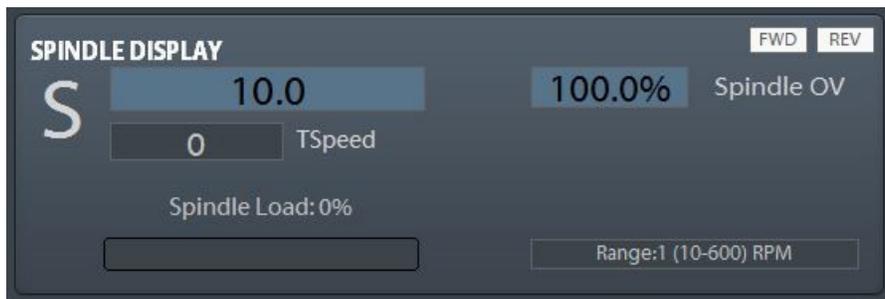
Spindle Control

The following spindle settings are also shown on the Spindle tab located in Configure -> Control -> Spindle:

1. G50 Speed Limit - the maximum RPM the spindle can move with the current G50 setting
2. Range - Pulley number selected and speed range

5.3 Spindle Display

The current spindle settings are shown in the main Spindle Display.



Spindle Display

1. S - Commanded Speed
2. Spindle OV - Spindle Override Percentage
3. Spindle Load - % of the load of the spindle.
4. Range - Current Pulley Selected

6. Machine Input / Output Control

For setting up machine specific IO, see this link: <https://support.machmotion.com/books/mach4-control-features/page/2000-series-commands-function-buttons-and-dashboard> .

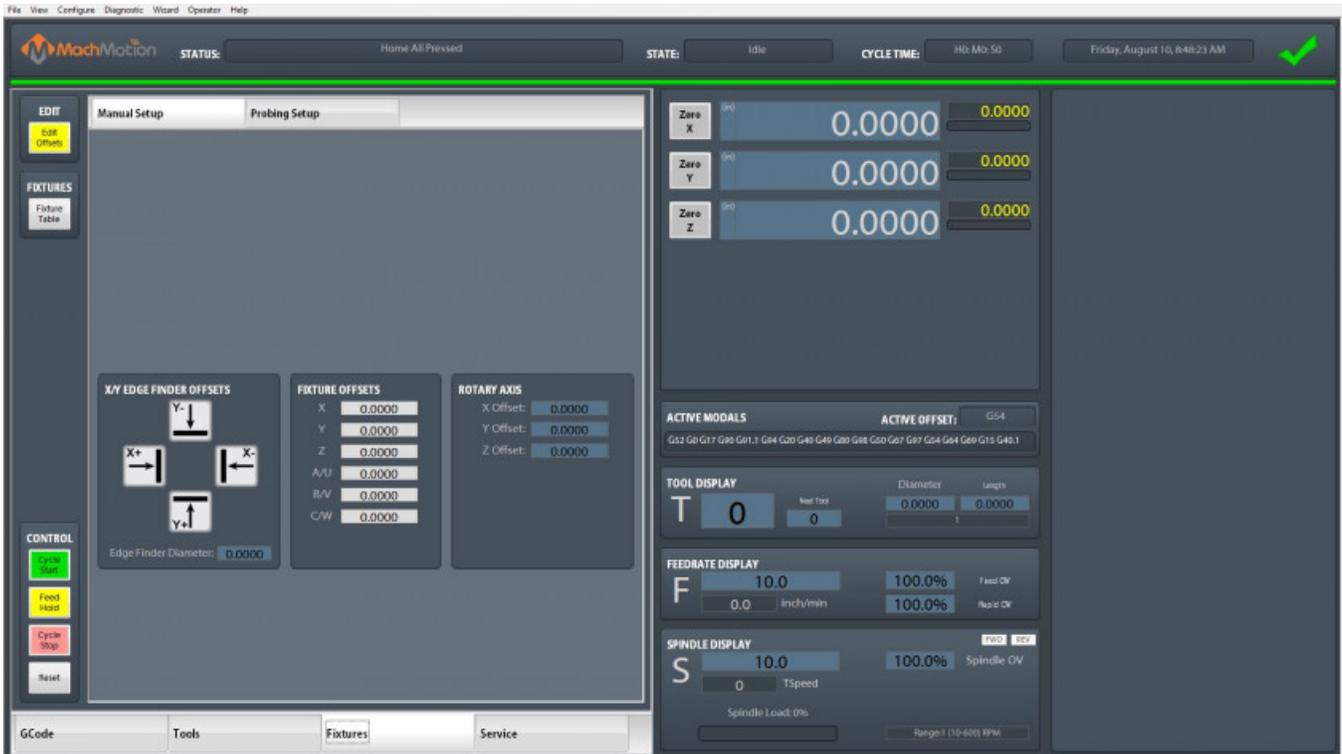
7. Fixture Offsets

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.

Part offsets can be defined and saved using G54-G59P120. The functionality is designed to allow different tooling setups to have predefined zero points to allow for streamlined setup.

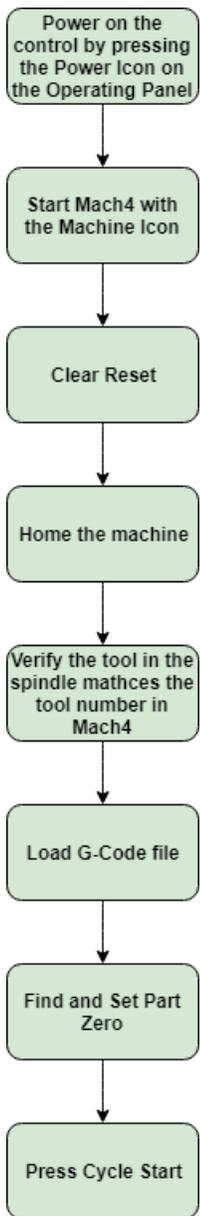
You can view the fixture table and change the values directly by clicking the [**Fixtures**] tab. The values can also be set by using the MDI command to select the G-Code number for the fixture offsets to be stored in. Once the machine is at the desired zero position, zero Z by pressing the [**Zero Z**] button.



Fixture Offsets

8. Appendix

8.1 Start-Up Procedure



Start-Up Procedure

8.2

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