



GX SERIES CNC ROUTERS

Quick Start Guide



www.probotix.com

844-472-9262

HOW TO GET HELP

Online Support

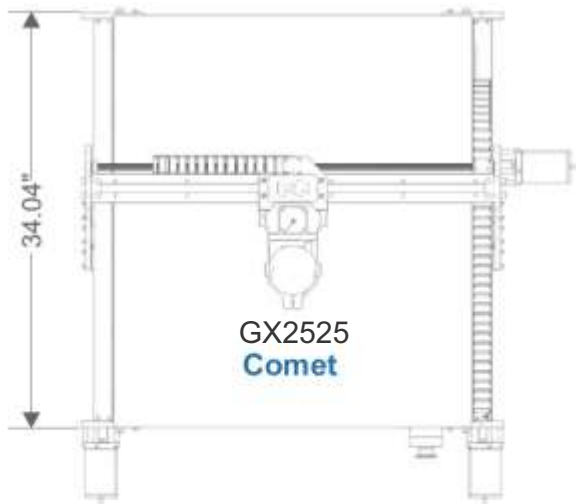
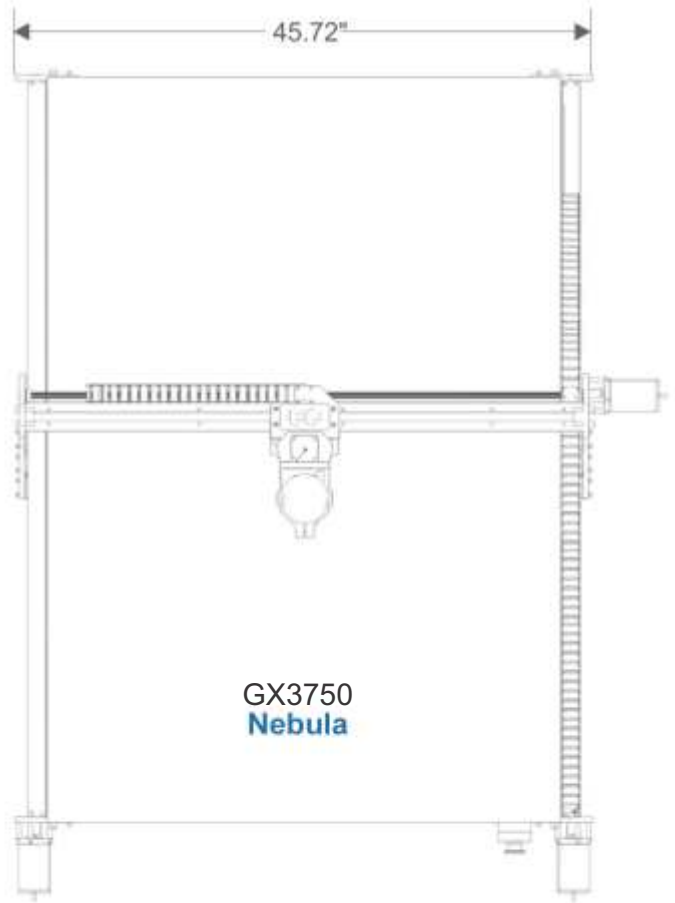
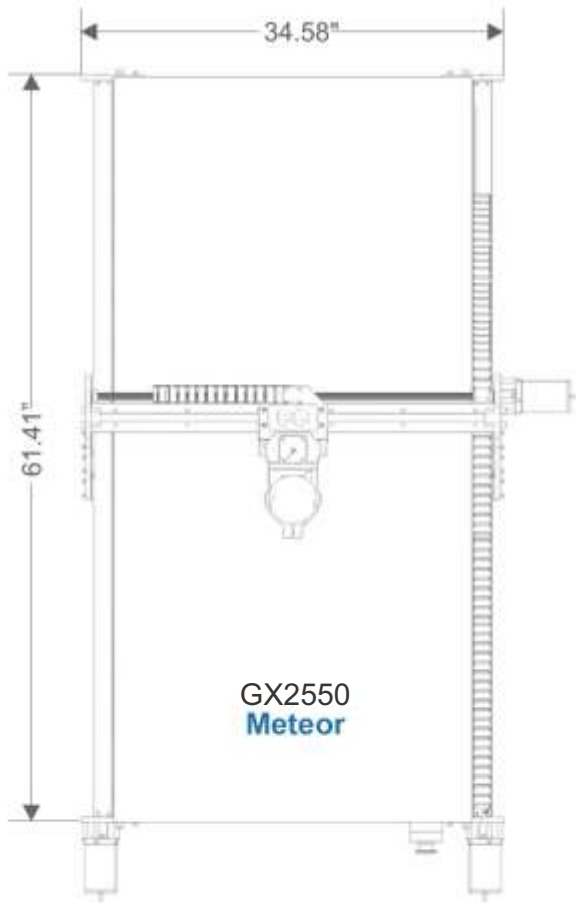
<http://www.probotix.com/wiki/>

<http://www.probotix.com/forum/>

Telephone Support

844-472-9262 Ext2

MACHINE FOOTPRINTS



Safety:

Safety glasses and hearing protection should be worn at all times while operating the machine, and long hair should be restrained with a cap or net when near the machines. Never wear loose clothing or jewelry when operating a CNC machine.

Operators should never leave the machine unattended during the cutting sequence, and labels should be observed at all times.

A CNC machine is controlled by the operator. The operator controls the machine either manually with the keyboard jog controls, with the handheld jog pendant, or automatically by means of G and M codes, or by running a g-code program. The operator needs to have at least a basic understanding of g-code and the machine coordinate system before operating the machine.

There is a latching mushroom emergency stop mounted on the front of the CNC machine. The [esc] key on the keyboard also acts as an emergency stop. The optional MPG jog pendant also has a latching e-stop button on it.

Maintenance:

All machines should be kept clean when not in use. The build-up of chips and dust on the system should be cleared daily with special attention focused on the lead screws, drive nuts, and bearing rails.

The machine should be cleaned and oiled every day before homing the machine. See page 15 for full maintenance instructions.

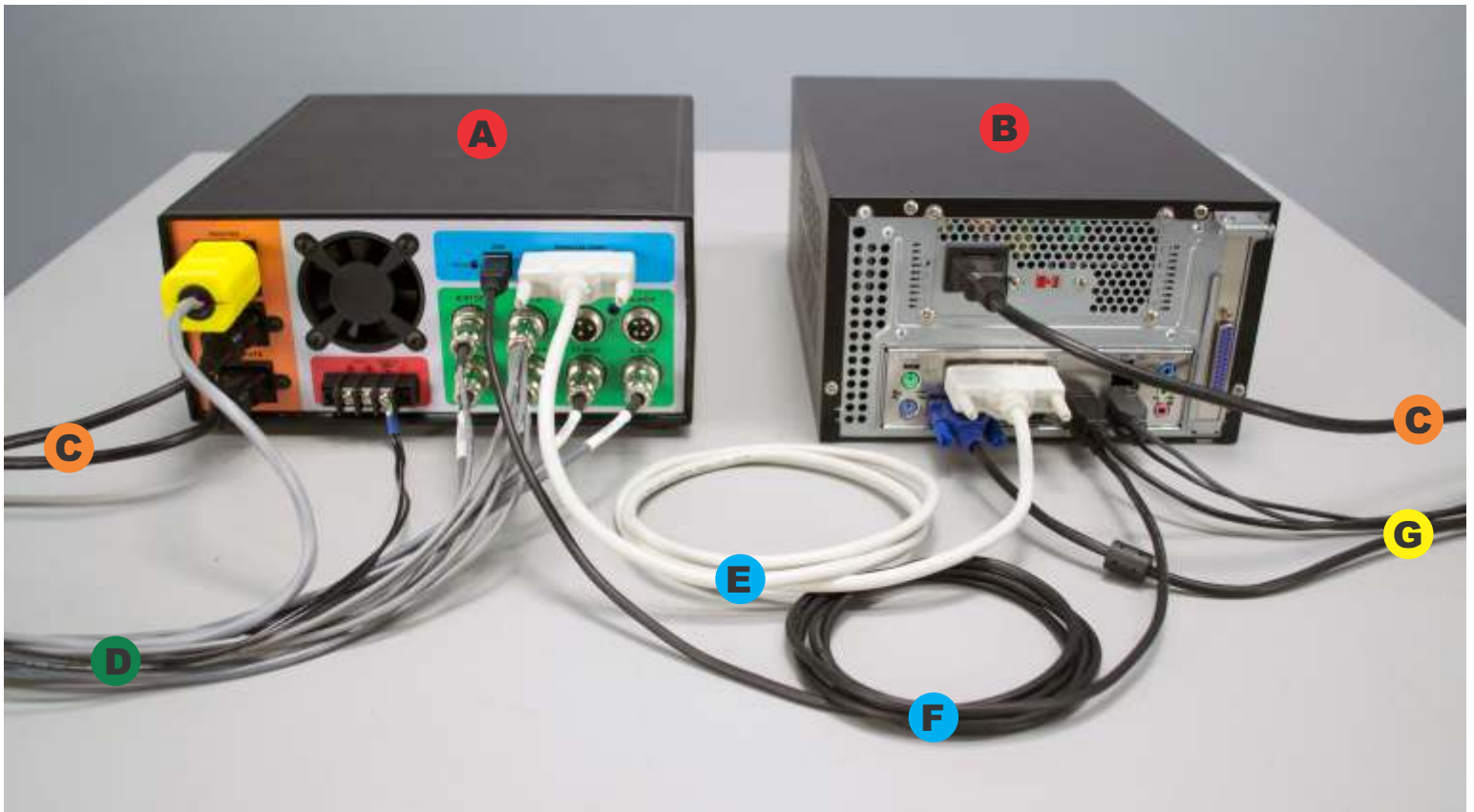
Electrical Requirements:

| | | |
|------------------|----------------|----------------------------------|
| Controller & PC: | 110VAC@10Amp | 6 outlet power strip recommended |
| Router: | 110VAC | 12 amps maximum |
| VFD Spindle: | 220VAC@20 Amps | NEMA6-20R receptacle |

If the machine is ordered with the customer supplied router option, a 110VAC@12Amp outlet is mounted on the gantry carriage. The controller will turn the outlet on and off with M codes in the software and the router speed is controlled by whatever speed control the router has.

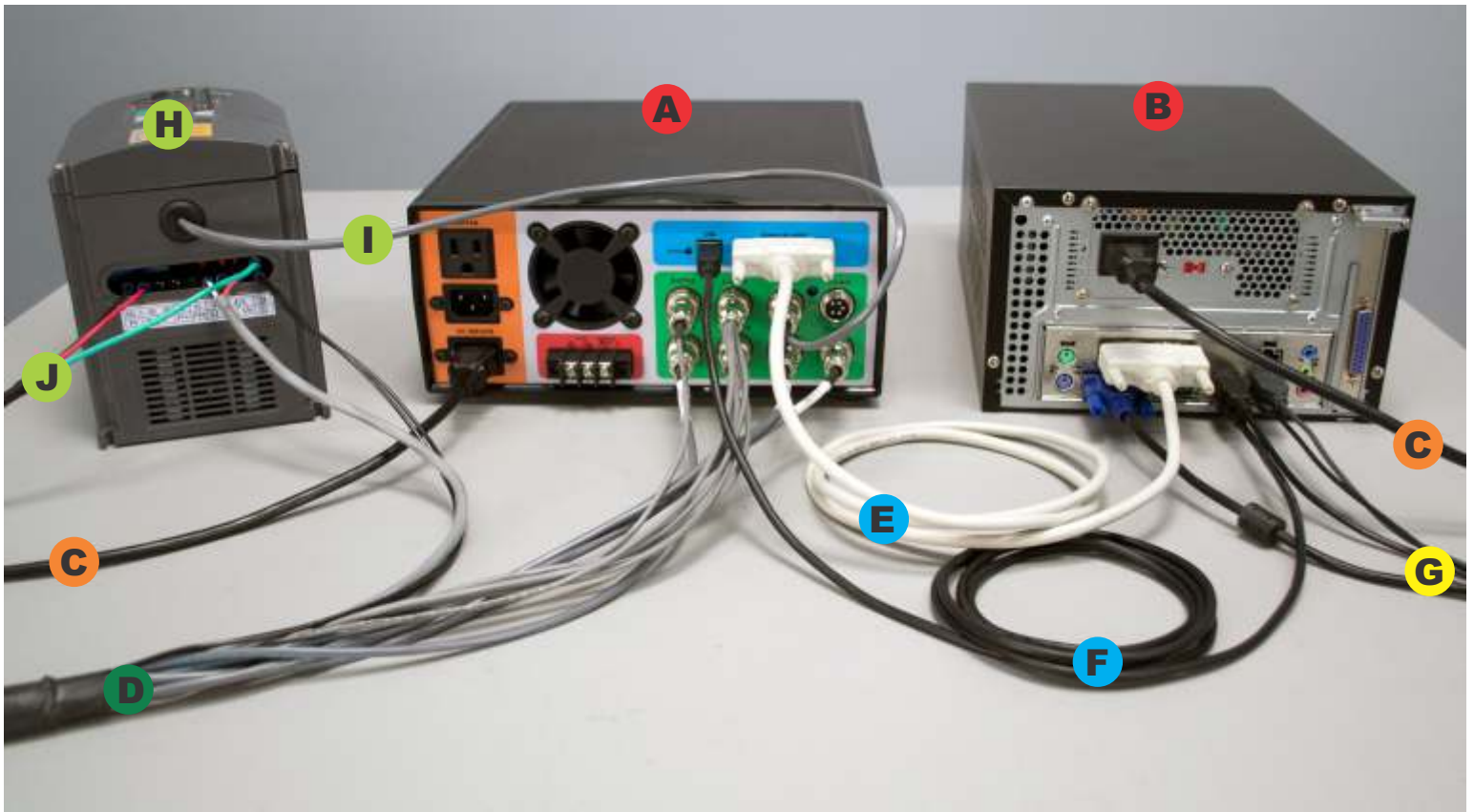
If the machine is ordered with the VFD spindle option, the VFD requires a separate 220VAC@20Amp circuit and spindle is turned on and off with M codes and the spindle speed is controlled with the S parameter. The controller's 110VAC@12Amp relay circuit can then be used to turn on and off a 110VAC@12Amp accessory such as a shop vac using M codes.

CONNECTION DIAGRAM (ROUTER)



- A** UNITY CONTROLLER
- B** COMPUTER
- C** 110VAC POWER
- D** MACHINE HARNESS
- E** PARALLEL CABLE
- F** USB CABLE
- G** KEYBOARD, MOUSE, MONITOR, JOG PENDANT

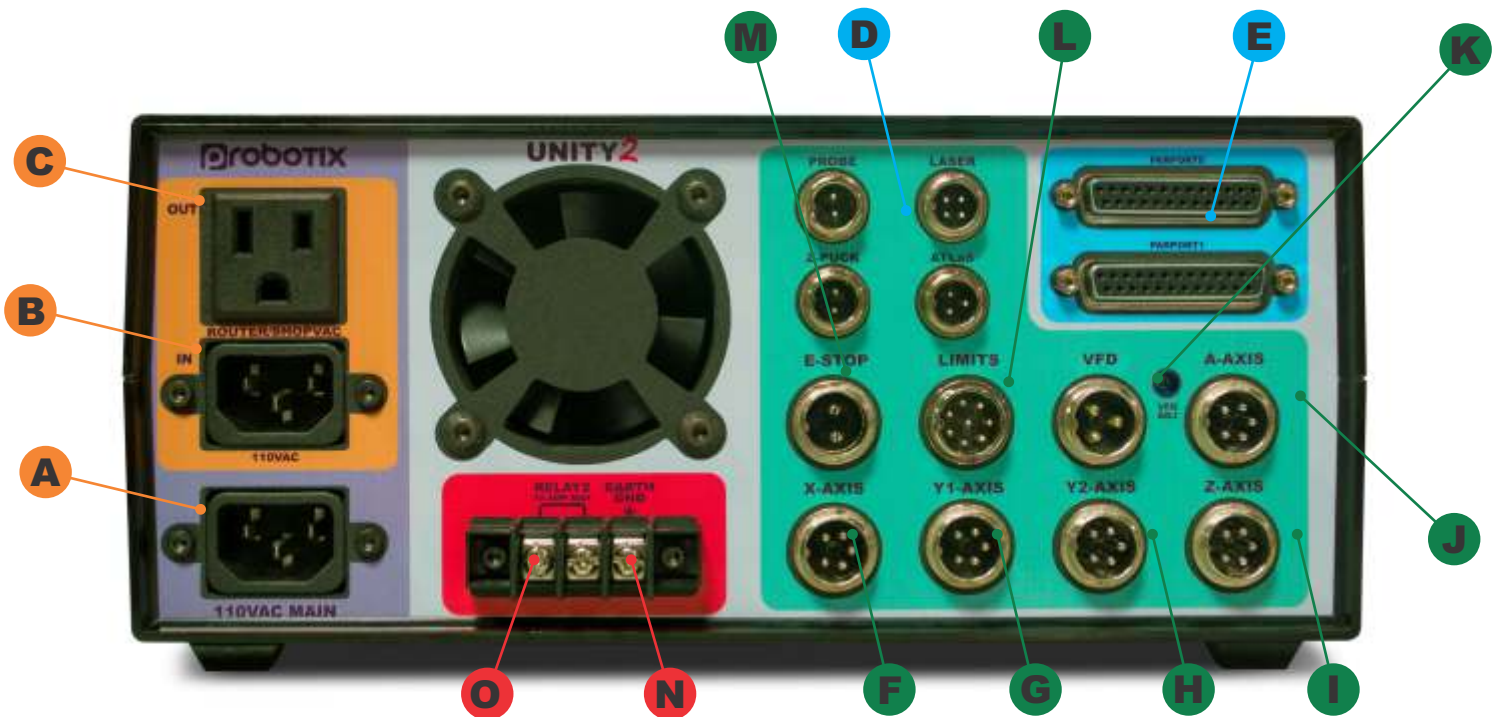
CONNECTION DIAGRAM (VFD SPINDLE)



- A** UNITY CONTROLLER
- B** COMPUTER
- C** 110VAC POWER
- D** MACHINE HARNESS
- E** PARALLEL CABLE
- F** USB CABLE
- G** KEYBOARD, MOUSE, MONITOR, JOG PENDANT
- H** VFD
- I** VFD CONTROL CABLE
- J** 220VAC

Unity has 2 Db25 connections

UNITY2 CONTROLLER PORTS



A: AC INPUT - Primary power input for power supply.

B: AC INPUT - Power input here is switched by relay to the (C) ROUTER output.

C: ROUTER - Connect to the green power cable from the machine harness. On-screen spindle controls, as well as M3/M5 g-codes will control this output. When a VFD spindle is used instead of a router, this outlet can be used for a shop vac and is controlled by M8/M9.

D: USB - Connect to USB connector on PC.

E: PARALLEL PORT - Connect to the primary (built onto the motherboard) DB-25 connector on the PC.

F, G, H, I, J: MOTOR CONNECTIONS - Connect to each of the labeled motor connections from the machine harness.

J: A-AXIS - For optional rotary axis.

K: VFD - Connect to 3-pin VFD pigtail for speed control of a VFD spindle.

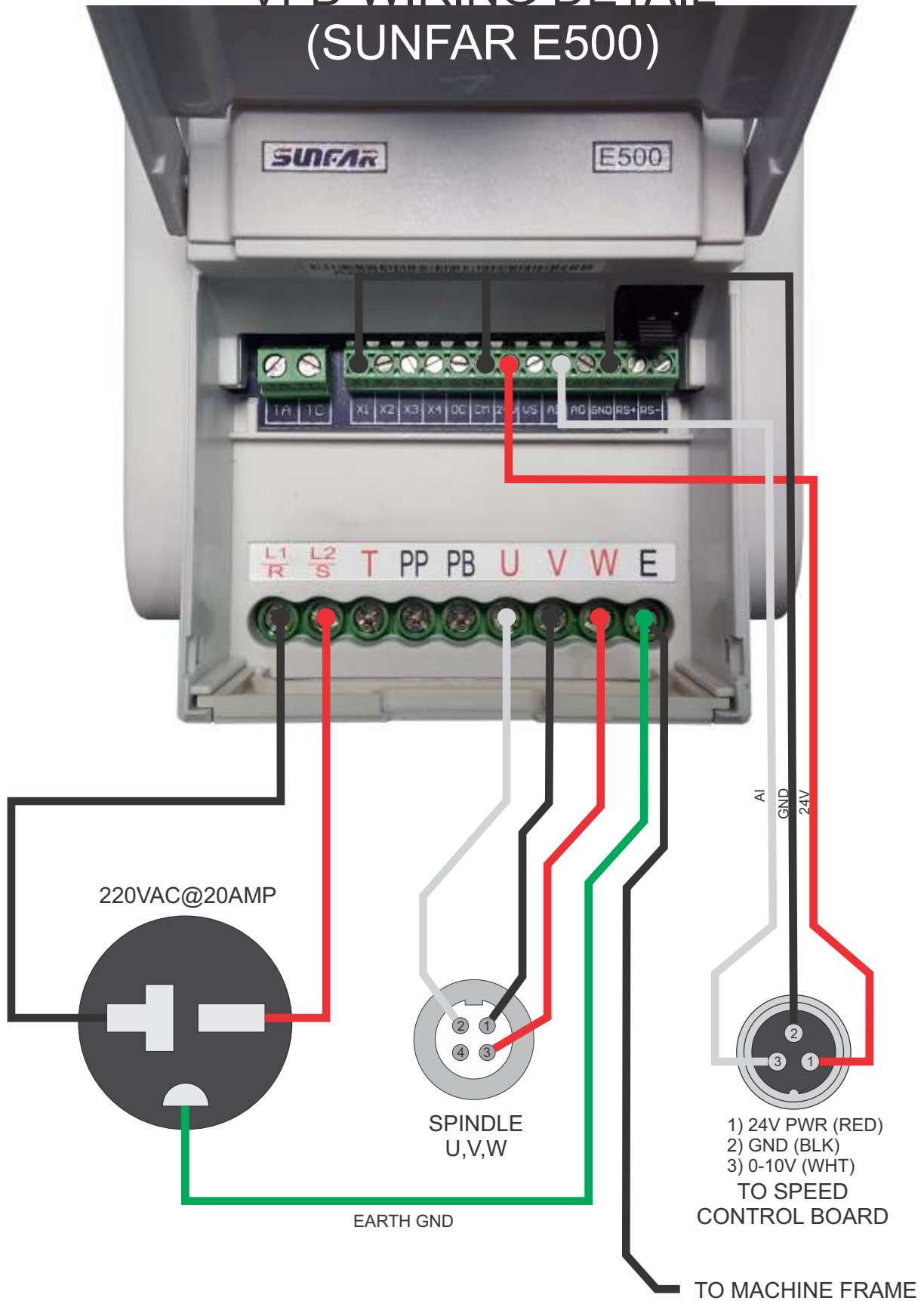
L: LIMITS - Connect to the 8-pin limit switch connector from the machine harness.

M: E-STOP - Connect to the 2-pin e-stop connector from the machine harness.

N: EARTH GND - It is recommended that the machine is grounded to earth. Connect the two black wires from the machine harness to this connection. If using a VFD, ground the machine frame to the VFD earth ground terminal instead. Additional earth grounding may be necessary when working with certain spindles or certain types of materials that may cause excess EMI or static electricity.

O: RELAY2 - This is a set of dry contacts to control a circuit up to 240V@12Amp. This signal is controlled by the digital 0 controls in the software (M64 P0, M65 P0).

VFD WIRING DETAIL (SUNFAR E500)

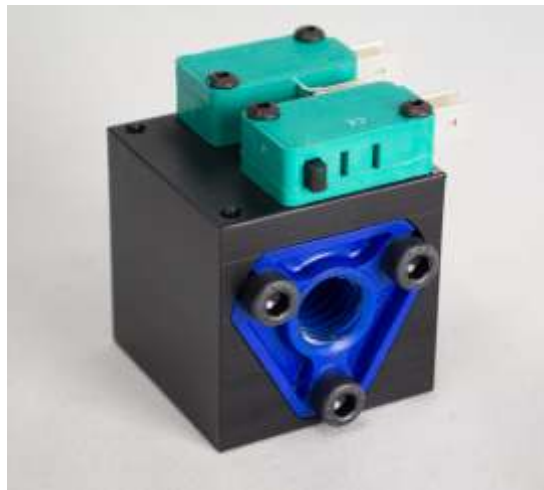


NOTICE: Internal wiring of Chinese spindles is not consistent. If spindle spins in reverse, swap U & V. You MUST verify visually or physically the proper rotation of the spindle to ensure the machine cuts properly!

LIMIT SWITCHES











The Y1 and Y2 limit switches have been removed to protect them during transport. Reinstall them as seen above before running the machine.



GETTING STARTED

Simple Start Up Procedure:

- Start computer
- Launch LinuxCNC from icon on desktop 
- Turn power on Unity Controller
- Verify red E-Stop indicator follows physical e-stop 
- Click orange Machine Power button 
- Click the Home All button 
- Click folder icon to open g-code file 
- Load tool into spindle or router
- Mount stock on the table
- Jog machine to place tool on origin of part
- Perform touch-off sequence (all 3 axis) 
- Verify spindle power switch is on 
- Hit the Run button 
- Watch in awe

*There are additional instruction in the wiki for using the Z Touch-off puck and the ATLaS Automatic Tool Length Sensor.

CAM SOFTWARE

You can use any CAM software to generate g-code for LinuxCNC. Most CAM software programs will have appropriate post processors named either LinuxCNC or EMC2. You can use generic g-code post processors as well. We also have post processors on our wiki specifically for our machines. If using the ATLaS Automatic Tool Length Sensor, you need to make sure the post processor you are using is inserting the proper tool change routine.

G54 COORDINATE SYSTEM

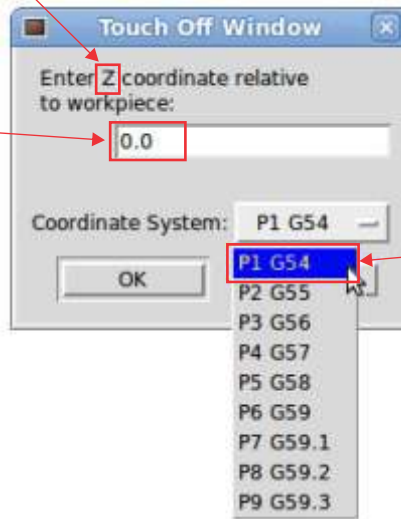
CNC machines use a numbered coordinate system. G54 is the default coordinate system. Some post processors carelessly do not insert a coordinate system. The results of not specifying a coordinate system in the g-code is unpredictable and dangerous. Be sure the coordinate system is being inserted into the g-code preamble.

TOUCHING OFF

After you mount your stock to the table and insert the tool you will be cutting with, use the jog controls to move the tool to the origin of the part. This is the same location you told the CAM software you wanted the origin to be. It could be on one of the corners of the part, or top center of the part. Then choose each axis and click the **Touch Off** button.

The axis that we are setting the origin for.

The current position of the tool relative to the stock at this moment. May be zero, may be the thickness of the piece of paper you are sliding under the tool.



This needs to match the coordinate system that the CAM software is inserting into the g-code.

G64 PATH OPTIMIZATION

Except for in the case of full 90 degree arcs, CAM software will break any curves in your designs up into a bunch of tiny segments. This will create jittery motion, will create tool marks in the part, shorten your tool life, and can extend the cut times significantly. Fortunately, LinuxCNC has a mode called G64 path optimization.

When G64 is turned on, LinuxCNC will perform a look-ahead and calculates the angle of the next segment(s), only slowing down as little as possible to round the corner(s). It tries to maintain the programmed velocity as close as possible, while not violating the velocity and acceleration capabilities of the machine. If G64 is programmed without a P parameter, LinuxCNC will take its liberty and round over those corners as much as it needs, the faster the feedrate, the greater the deviation from the programmed location. While cutting a square at only 10IPM will be a near perfect square, cutting that same square at 100IPM will result in 3 extremely rounded corners. In LinuxCNC, G64 is turned on by default (look for it under active g-codes under the MDI tab), but it does not have this P parameter (tolerance) by default.

The simple solution is to type G64 P0.010 into the MDI command box and hit go. This will tell LinuxCNC to yes, turn on path optimization, but do not deviate more than 0.010" from where I told you to go. The tolerance you use will depend upon the types of parts you are making. You will want to set it to 0.010 or more for decorative type things, especially while 3D carving. For more precise parts, You will use 0.001. G61 cancels G64 and will cause the machine to come to a complete stop at the end of every segment.

One more thing... You can put that G64 line in the preamble of your post processor so that it inserts it into every g-code file. It is modal, so it stays the same unless explicitly changed with another G64 Pxxx, or canceled with a G61. It also resets when the software is restarted. The most efficient way to manage this is to create separate post processor for each tolerance level you want to use.

Do not confuse G64 Path Optimization with the G54 Coordinate System

More information about this and other topics can be found on the wiki:
<http://www.probotix.com/wiki/>

MAINTENANCE

Daily Use:

Put a couple of drops of 3-in-1 oil on each of the six linear rails before homing the machine each day. This will allow a small amount of oil to be sucked up into the bearing housings before cutting any parts.

Use WD-40 and an air nozzle to remove debris from the screw. Apply a light amount of 3-in-1 oil daily.

All of the other bearings are sealed. The most important thing is to keep any dust cleaned off of the machine, especially if you are cutting abrasive materials such as MDF, fiberglass, or carbon fiber.

High Humidity Environments & Storage:

If you use your machine in a high humidity environment, ie in a shed or other non-airconditioned environment, or if you plan to store it without use, rust can form on the linear rails, leadscrews, and the black-oxide machine fasteners. Coat these surfaces with grease or WD-40 often. Again, be sure to dry the screws and rails of the WD-40 before re-applying grease and oil before use. The black-oxide machine fasteners will not need to be wiped dry.

Bearing Replacements:

Depending on the amount of use and the loads on the machine for your application, the linear bearings and leadscrew ball bearings will need to be replaced eventually. You will notice excess slop, marks on the rails, or jerky motion on the affected axis. Replace them annually for worry free operation.

Anti-backlash Nuts:

The Delrin anti-backlash nuts are designed to wear and may need to be replaced periodically, depending upon the amount of use and abuse your machine receives. When they fail, you will notice backlash on the affected axis. You can check them by pulling and pushing on each axis and observing slop between the leadscrew and the drive nuts. To replace them, you will need to remove the leadscrews. Replace them annually for worry free operation.

PC and Controller:

The electronics enclosure and PC enclosure can be blown out with dry air.

LINUXCNC KEYBOARD SHORTCUTS

| | |
|------------------|--|
| ESCAPE | E-STOP |
| F1 | TOGGLE E-STOP |
| F2 | TOGGLE MACHINE POWER |
| F3 | MANUAL MODE |
| F4 | AUTO MODE |
| F5 | MDI MODE |
| F6 | RESET INTERPRETER |
| F7 | TOGGLE MIST |
| F8 | TOGGLE FLOOD |
| F9 | TOGGLE SPINDLE FORWARD |
| F10 | TOGGLE SPINDLE REVERSE |
| F11 | DECREASE SPINDLE SPEED |
| F12 | INCREASE SPINDLE SPEED |
| X | SELECT X-AXIS |
| Y | SELECT Y-AXIS |
| Z | SELECT Z-AXIS |
| LEFT/RIGHT ARROW | JOG X-AXIS |
| UP/DOWN ARROW | JOG Y-AXIS |
| PAGE UP/DOWN | JOG Z-AXIS |
| HOME | HOME SELECTED AXIS |
| END | TOUCH-OFF SELECTED AXIS |
| </> | DECREASE/INCREASE JOG SPEED |
| C | SELECT CONTINUOUS JOGGING |
| I | SELECT INCREMENTAL JOGGING AND TOGGLE BETWEEN INCREMENTS |
| 1-9,0 | FEED OVERRIDE 10%-90%, 0 IS 100% |
| @ | TOGGLE COMMANDED/ACTUAL POSITION DISPLAY |
| # | TOGGLE ABSOLUTE/RELATIVE POSITION DISPLAY |
| O | OPEN A PROGRAM |
| R | RUN THE OPENED PROGRAM |
| P | PAUSE PROGRAM |
| S | RESUME PAUSED PROGRAM |
| A | STEP ONE LINE IN PAUSED PROGRAM |
| B | TAKE SPINDLE BREAK OFF |
| ALT-B | PUT SPINDLE BRAKE ON |
| ALT-S | START LOGGING |
| ALT-P | PAUSE LOGGING |
| ALT-F | SAVE LOG FILE |
| CTRL-P | PRINT LOG FILE (ONLY WITH XGRAPH) |

MPG JOG PENDANT



A: ENABLE SWITCH - Must be held in while using.

B: AXIS SELECTOR - Selects which axis to jog

C: MPG - Weighted wheel with detents. When Spinning fast, sets the speed of jogging, when spun slowly, each detent moves the increment determined by the increment selector switch.

D: INCREMENT SELECT - X1 = 0.100 in, X2 = 0.010 in, X3 = 0.001 in

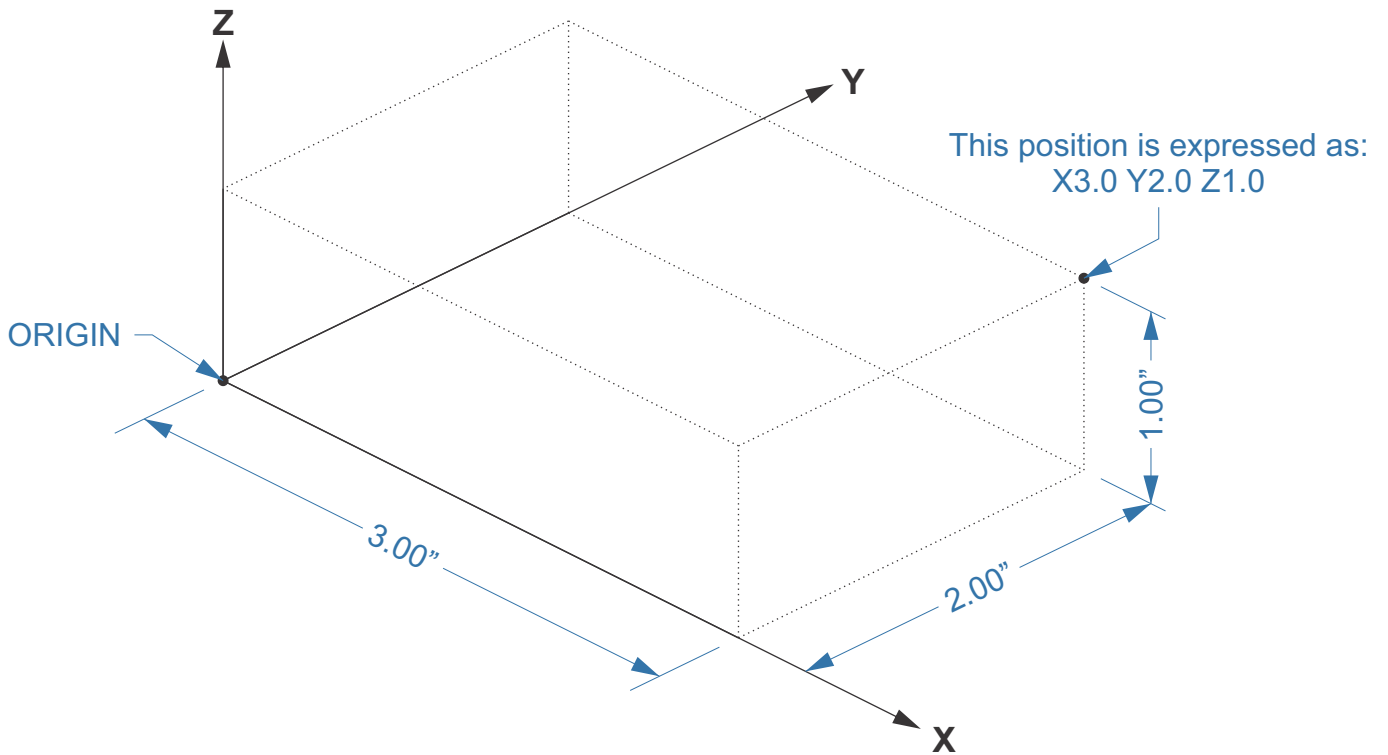
E: EMERGENCY STOP - Latching emergency stop, twist to unlatch

G-CODE QUICK REFERENCE

| Motion | | |
|--|---------------------|--|
| G0 | | Rapid motion |
| G1 | | Coordinated motion ("Straight feed") |
| G2, G3 | I J K or R | Coordinated helical motion ("Arc feed") CW or CCW |
| G38.2 | | Straight Probe |
| G80 | | Cancel motion mode |
| G81 | R L P | Drilling Cycle |
| G82...G89 | R L P Q | Other canned cycles |
| G33 | K | Spindle-synchronized motion |
| G33.1 | K | Rigid Tapping |
| G76 | P Z I J R K Q H L E | Multipass lathe threading cycle |
| Plane Selection (affects G2, G3, G81...G89, G40...G42) | | |
| G17 | | Select XY plane |
| G18 | | Select XZ plane |
| G19 | | Select YZ plane |
| Distance Mode | | |
| G90 | | Absolute distance mode |
| G91 | | Incremental distance mode |
| Feed Rate Mode | | |
| G93 | | Inverse time feed rate |
| G94 | | Units per minute feed rate |
| G95 | | Units per revolution |
| Units | | |
| G20 | | Inches |
| G21 | | Millimeters |
| Cutter Radius Compensation | | |
| G41, G42 | D | Start cutter radius compensation left or right |
| G41.1, G42.1 | D L | Start cutter radius compensation left or right, transient tool |
| G40 | | Cancel cutter radius compensation |
| Tool Length Offset | | |
| G43 | H | Use tool length offset from tool table |
| G43.1 | I K | Use specified tool length offset for transient tool |
| G49 | | Cancel tool length offset |
| Return Mode in Canned Cycles | | |
| G98 | | Retract to R position |
| G99 | | Retract to prior position |
| Path Control Mode | | |
| G61 | | Exact Path mode |
| G61.1 | | Exact Stop mode |
| G64 | P | Continuous mode with optional path tolerance |
| Stopping | | |
| M0 | | Pause Program |
| M2 | | End Program |
| M1, M30, M60 | | Other stop codes |
| Spindle Control | | |
| M3, M4 | S | Turn spindle clockwise or counterclockwise |
| M5 | | Stop spindle |
| G96 | D S | Constant surface speed mode (foot/minute or meter/minute) with top speed |
| G97 | | RPM mode |
| Coolant | | |
| M7 | | Turn mist on |
| M8 | | Turn flood on |
| M9 | | Turn all coolant off |
| Other Modal Codes | | |
| F | | Set Feed Rate |
| S | | Set Spindle Speed |
| T | | Select Tool |
| M50...M53 | P0 (off) or P1 (on) | Feed Override, Spindle Override, Adaptive Feed, Feed Hold |
| G54...G59, G59.1...G59.3 | | Select coordinate system |
| Flow-control Codes | | |
| O ... | | sub/endsub, while/endwhile, if/else/endif, do/while, call, break/continue/return |
| Non-modal Codes | | |
| M6 | T | Change tool |
| G4 | P | Dwell (seconds) |
| G10 L2 | P X Y Z A B C | Coordinate system origin setting |
| G28 | | Return to home |
| G30 | | Return to secondary home |
| G53 | | Motion in machine coordinate system |
| G92 | X Y Z A B C | Offset coordinate systems and set parameters |
| G92.1 | | Cancel offset coordinate systems and set parameters to zero |
| G92.2 | | Cancel offset coordinate systems but do not reset parameters |
| G92.3 | | Apply parameters to offset coordinate systems |
| M101...M199 | P Q | User-defined M-codes |
| (...) | | A comment "... " to the user |
| (MSG,...) | | Display the message "... " to the user (e.g., in a popup) |
| (DEBUG,...#123...#<foo>) | | Display the message (with variables substituted) like MSG |
| (PRINT,...#123...#<foo>) | | Display the message (with variables substituted) to stdout |

Table 1. Coordinate System

| P Value | Coordinate System | G code |
|---------|-------------------|--------|
| 0 | Active | n/a |
| 1 | 1 | G54 |
| 2 | 2 | G55 |
| 3 | 3 | G56 |
| 4 | 4 | G57 |
| 5 | 5 | G58 |
| 6 | 6 | G59 |
| 7 | 7 | G59.1 |
| 8 | 8 | G59.2 |
| 9 | 9 | G59.3 |



LINUXCNC NUMBERED PARAMETERS

31-5000 - G code user parameters. These parameters are global in the G code file, and available for general use. Volatile.

5061-5069 - Coordinates of a "G38.2" Probe result of X, Y, Z, A, B, C, U, V & W. Volatile.

5070 - "G38" probe result - 1 if success, 0 if probe failed to close. Used with G38.3 and G38.5. Volatile.

5161-5169 - "G28" Home for X, Y, Z, A, B, C, U, V & W. Persistent.

5181-5189 - "G30" Home for X, Y, Z, A, B, C, U, V & W. Persistent.

5211-5219 - "G92" offset for X, Y, Z, A, B, C, U, V & W. Persistent.

5210 - 1 if "G92" offset is currently applied, 0 otherwise. Persistent.

5211-5219 - G92 offset (X Y Z A B C U V W).

5220 - Coordinate System number 1 - 9 for G54 - G59.3. Persistent.

5221-5230 - Coordinate System 1, G54 for X, Y, Z, A, B, C, U, V, W & R. R denotes the XY rotation angle around the Z axis. Persistent.

5241-5250 - Coordinate System 2, G55 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5261-5270 - Coordinate System 3, G56 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5281-5290 - Coordinate System 4, G57 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5301-5310 - Coordinate System 5, G58 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5321-5330 - Coordinate System 6, G59 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5341-5350 - Coordinate System 7, G59.1 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5361-5370 - Coordinate System 8, G59.2 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5381-5390 - Coordinate System 9, G59.3 for X, Y, Z, A, B, C, U, V, W & R. Persistent.

5399 - Result of M66 - Check or wait for input. Volatile.

5400 - Tool Number. Volatile.

5401-5409 - Tool Offsets for X, Y, Z, A, B, C, U, V & W. Volatile.

5410 - Tool Diameter. Volatile.

5411 - Tool Front Angle. Volatile.

5412 - Tool Back Angle. Volatile.

5413 - Tool Orientation. Volatile.

5420-5428 - Current relative position in the active coordinate system including all offsets and in the current program units for X, Y, Z, A, B, C, U, V & W, volatile.

5599 - Flag for controlling the output of (DEBUG,) statements. 1=output, 0=no output; default=1. Volatile.

5600 - Toolchanger fault indicator. Used with the iocontrol-v2 component. 1: toolchanger faulted, 0: normal. Volatile.

5601 - Toolchanger fault code. Used with the iocontrol-v2 component. Reflects the value of the toolchanger-reason HAL pin if a fault occurred. Volatile.