



TNC-M14-CNC CONTROLLER



Document: Operation Manual
Document #: TNC-M14-m
Document Rev: 1.0

Product: CNC controller
Product Rev: 1.0
Created: Oct-2013

THIS MANUAL CONTAINS INFORMATION FOR INSTALLING AND OPERATING THE FOLLOWING PRODUCT:

- TNC-M14- CNC CONTROLLER

“TINY CONTROLS” AND THE TINY CONTROLS COMPANY’S LOGO ARE COPYRIGHT OF TINY CONTROLS (P) LTD. COMPANY. OTHER TRADEMARKS, TRADE NAMES, AND SERVICE MARKS OWNED OR REGISTERED BY ANY OTHER COMPANY AND USED IN THIS MANUAL ARE THE PROPERTY OF THEIR RESPECTIVE COMPANIES.

TINY CONTROLS PRIVATE LIMITED
C-55, NISHAT PARK, KAKROLA MOR, NEW DELHI, INDIA – 110078
WEB: <http://www.tinycontrols.com>
PHONE: +91-11-2533-1567, +91-991-119-3210

CONTENTS

1. SIGNS USED IN MANUAL	4
2. CNC CONTROLLER INCLUDES	6
3. MECHANICAL INSTALLATION	6
4. INTRODUCTION- TNC-M14- CNC CONTROLLER	7
5. FEATURES	8
6. CNC CONTROL BOARD SPECIFICATIONS	9
7. CNC CONTROLLER BOARD TERMINALS	10
8. CONNECTING OUTPUTS	11
9. CONNECTING INPUTS	13
10. CNC CONTROL BOARD CONNECTION WITH PEN DRIVE	15
11. CONNECTION DIAGRAM	16
12. AXES AND SPINDLE OUTPUTS CONNECTIONS WITH DRIVES	17
13. BLOCK MODE AND TOOL ZERO CONNECTIONS	18
14. HOME INPUTS CONNECTIONS WITH HOME SWITCHES	19
15. COOLANT, SPINDLE DRIVE AND VFD CONNECTIONS	19
16. LIMIT INPUTS CONNECTIONS WITH LIMIT SWITCHES	20
17. MOUNTING INSTRUCTIONS	21
18. INPUTS AND OUTPUTS SPECIFICATIONS FOR CONTROL BOARD	22
19. HAND HELD PENDANT	25
20. EMERGENCY BUTTON	28
21. BRIGHTNESS SETTINGS FOR LCD	28
22. KEYS FUNCTIONS IN VARIOUS MODES	29
23. CNC CONTROLLER BOARD CONNECTIONS WITH PENDANT	31
24. DESCRIPTION OF DISPLAYS IN VARIOUS STATES OF MACHINE	32
25. OPERATING THE CNC CONTROLLER	35
26. MAIN MENU	35
27. STRUCTURE TREE FOR MENU FUNCTIONS	36
28. GENERAL SETTINGS	37
29. ACTIVE PROFILE	41
30. AXIS SETTINGS	41
31. IO SETTING	44
a. SPINDLE SETTING	44
b. HOMING SETTING	46
c. HOME SWITCH SETTING	47
d. LIMIT SETTING	48
e. MISC OUTPUTS	49
32. TOOL SETTINGS	50

33. DATA FILE	52
34. DEFAULT PROFILES VALUES	54
35. KEY FUNCTIONS: NAVIGATION KEYS	55
36. NUMERIC KEYS	58
37. FEED RATE OVERRIDE AND SPINDLE SPEED OVERRIDE	59
38. G-CODE ALLOWED TO USE †	63
39. M CODES ALLOWED TO USE †	64
40. FORMAT OF A G-CODE LINE	65
41. TROUBLE SHOOTING	67
42. GLOSSARY	69
43. INDEX	71

SIGNS USED IN MANUAL

Warning:



A warning contains information, which is essential for avoiding a safety hazard.

Caution:



A caution contains information, which is necessary for avoiding a risk of damage to the product or other equipment.

Note:



This icon indicates information which, if not heeded, can result in the CNC controller not operating to full efficiency, as well as information concerning incorrect operations.

Tip:



This icon indicated information that can prove handy when performing certain settings or operations with the equipment.

Application:



- CNC controller may not be used for a life support system or other purposes directly related to the human safety.
- Although CNC controller is manufactured under strict quality control, install safety devices for applications where serious accidents or material losses are foreseen in relation to the failure of it.

Installation:



- Install the control board on a non inflammable material such as metal.
- Don't place inflammable matter nearby.
- Prevent lint, paper, fibers, sawdust, dust, metallic dust or other foreign material from getting accumulating onto the control board.



- Don't install or operate the CNC controller that is damaged or lacking parts. Doing so causes fire, accidents or injuries.

Wiring:



- Be sure to connect the grounding wires.
- Qualified operators should carry out the whole wiring process.
- Be sure to perform wiring of whole system after turning the power off.
- Generally control signal wires are not reinforced insulation. If they accidentally touch any of the live parts in the main circuit, their insulation coat may break for any reasons. In such a case, an extremely high voltage may be applied to the signal lines. Make a complete remedy to protect the signal line from contacting any hot high voltage lines.



- The wiring and the motors generate electric noise. Take care of malfunction of the nearby devices. To prevent the motor from malfunctioning, implement noise control measures.

Operation



- Don't operate the switches with wet hands.
- Although the hand pendant has an emergency button, install an emergency button separately along with mounting of control board.



- Don't turn the main circuit power on or off in order to start or stop CNC controller operation.
- In general, sheathes and covers of the control signal cables and wires are not specifically designed to withstand a high electric field (i.e. reinforced insulation is not applied). Therefore, if a control signal cable or wire comes into direct contact with a live conductor of the main circuit, the insulation of the sheath or the cover might break down, which would expose the signal wire to a high voltage of the main circuit. Make sure that the control signal cables and wires will not come into contact live conductors of the main circuits.



- Make sure that the control signal cables and wires will not come into contact live conductors of the main circuits.

Maintenance and Inspection, and Parts Replacement



- Maintenance, inspection and parts replacement should be made only by qualified persons.
- Take off the watch, rings and other metallic matter before starting the work.
- Use insulated tools.

Others



- Never attempt to modify the CNC control board or the handheld pendant device.

General Precautions

Drawing in this manual for the connections may be illustrated without the shield cover for explanation of the detail parts. Restore the shields in the original state and observe the description in the manual before starting operation.

The ground terminal should be connected to the ground. And make sure to use the ground wires whose size is greater than the power supply lines.



- Be sure that Power supply should not be greater than 24 V; else it damages the Controller Board.
- Caution is advised during the mechanical and electrical installation. Poorly tightened cables may cause many problems; it's also very difficult to detect such as defects while using the system. To avoid the electric shock or equipment damage or burning of power cords, check loose connections or exposed extension cords which may lead to tripping hazard.

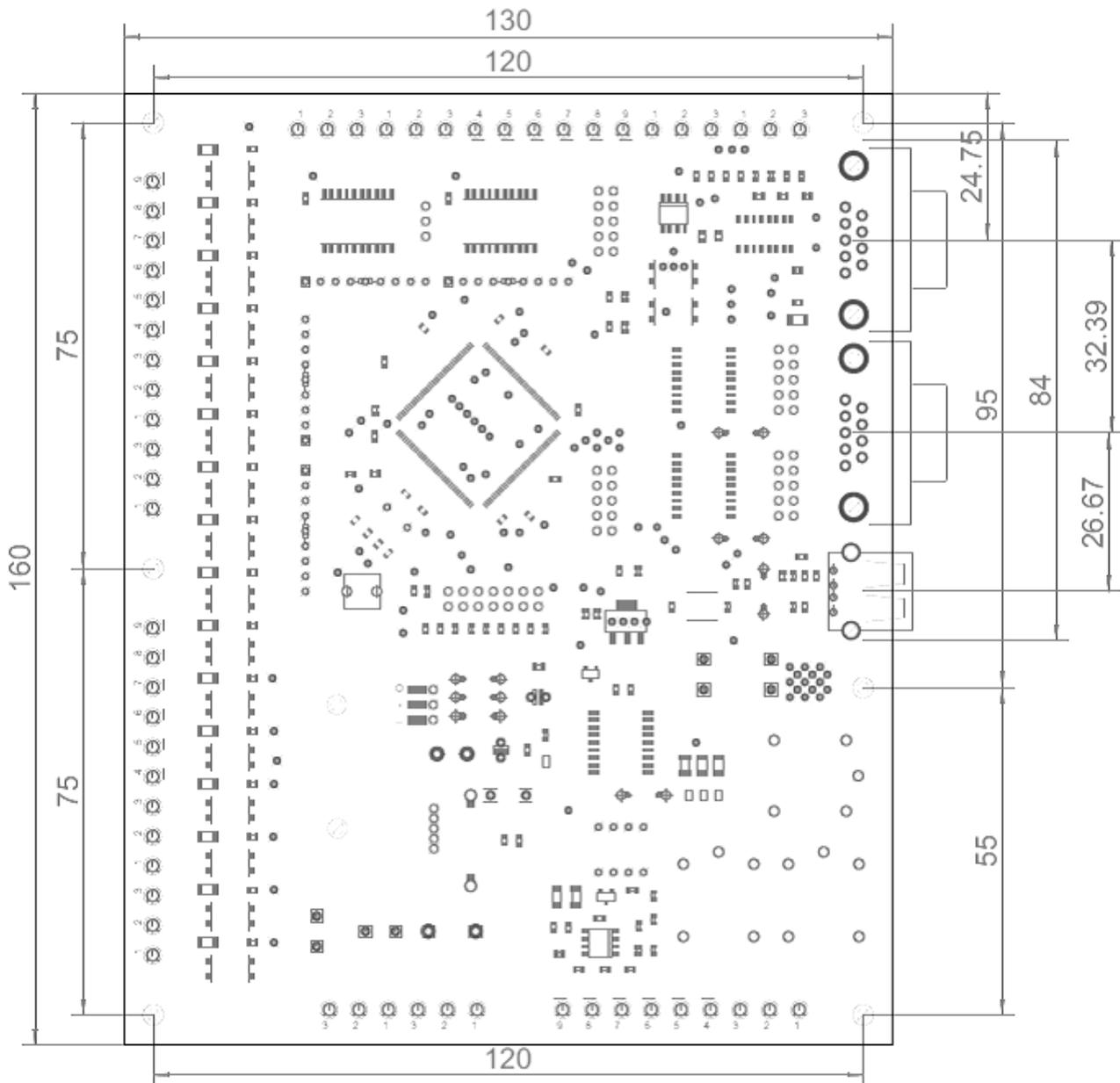


- Unplug the controller power supply and other connection when it is going to be left unused for an extended period of time.

CNC CONTROLLER INCLUDES

1. Control Board
2. Handheld pendant
3. 3mm Serial cable connected to the pendant.

MECHANICAL INSTALLATION:



INTRODUCTION- TNC-M14-CNC CONTROLLER

CNC controller is a G-code programmable controller with step and direction outputs for stepper drivers. The three axes **X, Y and Z** are **linear** whereas the **4th** axis is **rotary** and can be configured as either **A, B or C** axis, providing flexibility to system. All the four axes can be configured with independent adjustable maximum velocity and acceleration. An isolated analog **(0-10V) output** is provided for VFD to control spindle speed. Any two or all axes can be programmed for linear (line) interpolation, any 2 axes can be interpolated for curve (circular) Interpolation or any 2 axes curve with other axes line (helical) interpolation.

The USB host port provided is used to interface with a common pen (thumb) drive to read and execute G-code files and doesn't require a PC. **There is no limit on G-code file size**. Controller can resume the operation on any power interruption saving the current machine position, line of G-code and other status. Power up from interrupted cycle offers to recover from power fail, automatically 'homing' the machine, cursing to last location, and resuming from last saved line without losing a beat in the process. Traditional complicated NC operation become easy and simpler because of its user friendly interface.

Control board is the brain of CNC controller. It handles the command reading function from pen drive and the serial transmission function with pendant. It controls the CNC machine by giving outputs for steps and direction for stepper drives and spindle drive and receiving Limit, tool zero, block input and home inputs.

All the axes and spindle outputs are buffered and taken out through screw terminals from control board whereas all the inputs are optoisolated for noise immunity. A provision for 6 buffered and 16 open collector outputs is given on the controller board for other applications. These are termed as miscellaneous outputs and can be made On/Off in G-code file.

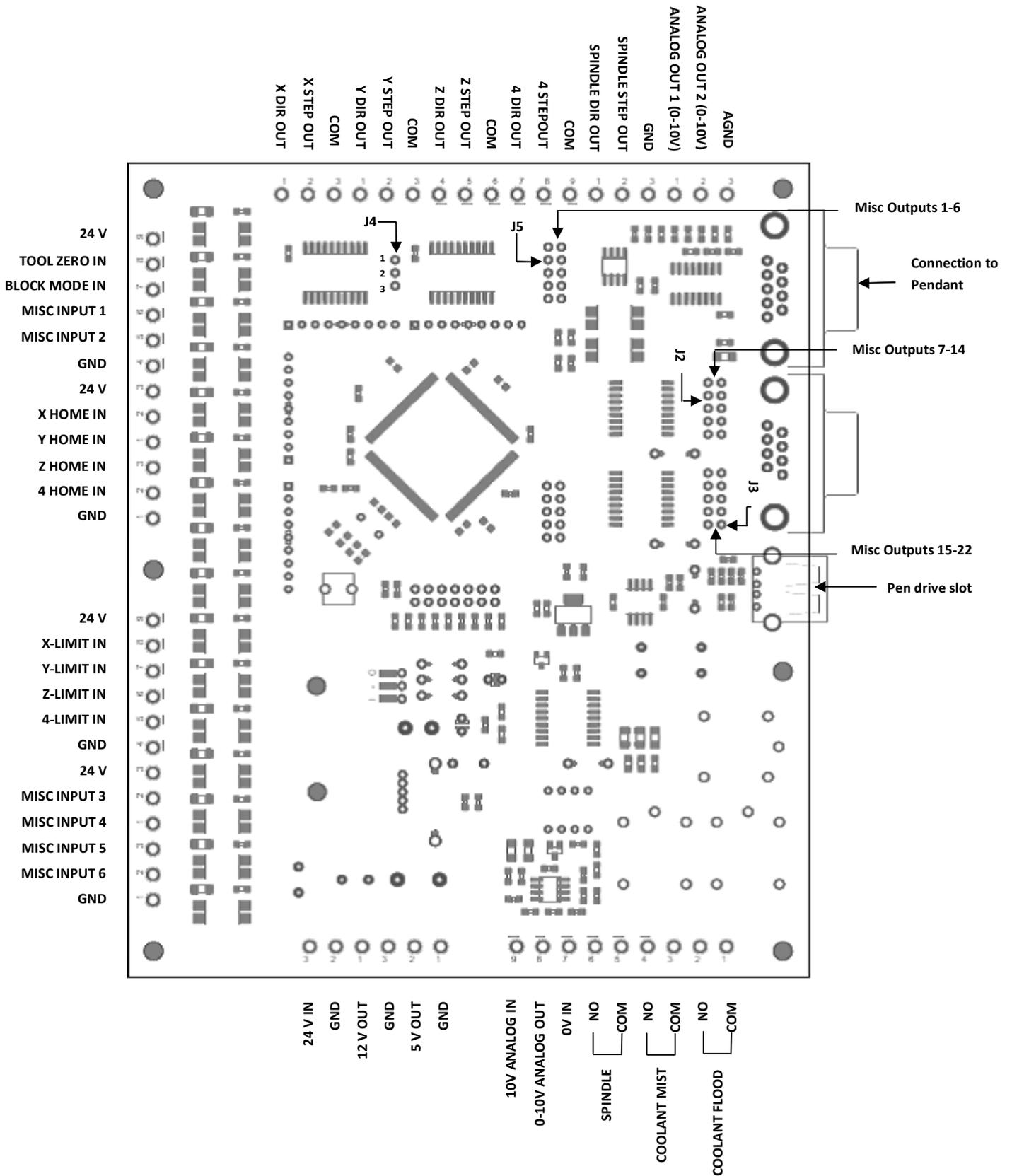
FEATURES

1. Totally independent single board solution.
2. 32 digital + 2 analog + 1 isolated analog + 3 relay outputs.
3. 16 isolated inputs.
4. Low jitter 150 KHz Pulse rate on each axis.
5. Constant velocity (with look ahead) and exact stop path modes.
6. Interpolation – linear, arc and helical.
7. Work offset (work coordinates).
8. Feed override option (Feed and Spindle speed).
9. Tool offset option (auto tool offset measurement).
10. Jogging via inbuilt keypad (Step, continues and fast mode).
11. Feed hold (pause), Cycle start (and cycle resume), Cycle stop (abort).
12. Home and limit inputs for all axes (can be individually enabled/disabled).
13. Block execution mode (menu / run time control)
14. Controlled feed hold with deceleration to ensure no skipped steps and loss of location.
15. G-code “**check**” option (Dry Run) and program limit check function.
16. Screw terminal connectors for connecting motor drive, input and outputs.
17. Help menu handy for first time users.
18. Handheld Pendant to display status/settings and control the machine.
19. Intuitive menus with easy to use smooth navigation.
20. Inputs for Limit and Home switches (or sensors).
21. Emergency Stop switch on handheld unit.

CNC CONTROL BOARD SPECIFICATIONS

Contents	Specifications
Processor	32 bit processor
Working coordinates	6 working coordinates
Display	20x4 alphanumeric LCD
USB Host Port	USB-2.0 Port
Communication Terminal 1	Serial Terminal for pendant
Communication Terminal 2	Serial Terminal (PLC interface)
File Format	8.3 format (8 lettered name+3 lettered extension name)
Axes	4-axes
Interpolation function	Linear, curve, helical
Operating Interface	Friendly buttons & separate jog panel
Axes Drive Control type	Step/Dir (5V)
Max frequency of Step signal	150 KHz
Digital outputs	32
Digital inputs	20
Analog outputs (0-10 V) Isolated	1
Analog outputs (0-10 V) Non-Isolated	2
Relay outputs	3 (M3, M7, M8)
Interface Language	English
Supply voltage	24V DC
Max Power Consumption	24V/1A
Ambient temperature range	0°-55° Celsius
Relative Humidity	< 90% (without condensation)
Dimensions	160x130
Weight	50g

CNC CONTROLLER BOARD TERMINALS



CONNECTING OUTPUTS

Digital Outputs: The digital outputs are as following:

GND	○
SPINDLE STEP OUT	○
SPINDLE DIR OUT	○
COM	○
4 STEP OUT	○
4 DIR OUT	○
COM	○
Z STEP OUT	○
Z DIR OUT	○
COM	○
Y STEP OUT	○
Y DIR OUT	○
COM	○
X STEP OUT	○
X DIR OUT	○
COM	○

a. OUTPUTS FOR SPINDLE DRIVE

a. Step output: 1

b. Direction output: 1

CNC controller board has a buffered step and direction output for spindle drive. The step frequency is user programmable and ranges up to 20 KHz. These outputs can be disabled by disabling the **Spindle: Enable/Disable** parameter in **spindle settings** of **IO settings** function. Refer to IO settings on [page 44](#) for more details.

b. AXES OUTPUTS

a. Step outputs: 4

b. Direction outputs: 4

CNC Control board has Step/ Dir outputs for four axes named X, Y, Z and 4th axis. These outputs are buffered outputs and can be disabled by disabling **Drive: Enable/Disable** parameter in **Axis settings** for the axes which are not required in operation. Refer to axes settings on [page 41](#).

COM SELECTION JUMPER

User can change the COM signal configuration for axes outputs by changing the position of jumper J4. By default, jumper, J4 is connected to position 2 and 3, COM is grounded. To connect the COM to +5V, position the jumper at 1 and 2.

c. ANALOG 0-10V OUTPUT:

A 0-10V analog output signal directly goes to VFD to control the spindle. This circuit is completely isolated from rest of control board.

If **10V input** is fed from VFD to terminal **10V Analog IN**; an analog output in range of **0-10V** can be drawn from terminal **0-10V Analog OUT**.

10 V ANALOG IN
 0-10V ANALG OUT
 0V IN

SPINDLE [NO
 COM

COOLANT MIST [NO
 COM

COOLANT FLOOD [NO
 COM



If **5V input** is fed from VFD to **10V Analog IN**, an analog output in range of **0-5V** can be drawn from terminal **0-10V Analog OUT** terminal.

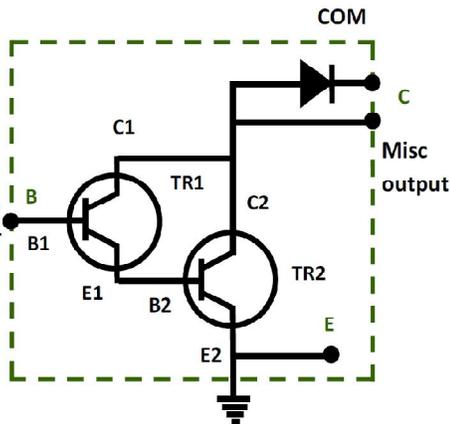
d. SPINDLE ENABLE & COOLANT OUTPUTS:

Spindle NO output signal directly goes to VFD and enables the spindle. Spindle NO is a relay output and the relay switches to NO if **spindle** parameter is set **enable** in **IO Settings** and spindle is made ON in Gcode.

Coolant mist NO and coolant flood NO are the relay outputs and the relay switches to NO if these outputs are made ON in the Gcode.

e. MISC OUTPUTS:

Misc outputs are miscellaneous outputs and 22 in number. Required output can be made ON in G-code file. Misc outputs 1-6 are buffered and the rest 16 are open collector outputs. These can be made **OFF** collectively in G-code file but are made **ON** individually using the respective Mcode. The individual output terminals with its respective Mcode are shown:



M54.02	○ ○	M54.03	J5
M54.06	○ ○	M54.01	
M54.04	○ ○	M54.05	

Figure shows the **connector 5** outputs (see the terminal diagram) with respective Mcode. These are 6 in number and are buffered outputs. Enable/ Disable these outputs in the MISC outputs function of IO settings and make them ON/OFF in Gcode. These outputs can be set active high/ active low in the Misc outputs function.

However, for the buffered output pin, there isn't any output if it is disabled in Misc output function and made ON in Gcode.

M54.07	○ ○	M54.08	J2
M54.09	○ ○	M54.10	
M54.13	○ ○	M54.12	
M54.11	○ ○	M54.14	

Figure shows the **connector 2** outputs with respective Mcode. These are open collector outputs and can be made ON/OFF in G-code. These outputs generate Active high output when made ON in Gcode.

- M54.15 ○ ○ M54.16 **J3**
- M54.17 ○ ○ M54.18
- M54.19 ○ ○ M54.20
- M54.21 ○ ○ M54.22

Figure shows the **connector 3** outputs with respective Mcode. These are open collector outputs and can be made ON/OFF in G-code. These outputs generate Active high output when made ON in Gcode.

M54.01-M54.22: To make the Misc output ON.

M55.01-M55.22: To make the Misc output OFF.

M56: Execution of this Mcode switches OFF all the Misc outputs.

ANALOG OUTPUTS 1-2:

- AGND
- ANALOG OUT2 0-10V
- ANALOG OUT1 0-10V
- GND

○ Two non-isolated analog 0-10V outputs are provided on board. These are auxiliary outputs and can be set to any value in range of **0-10V** using Mcode 57 for output 1 and Mcode 58 for output 2. The format of Mcode for these outputs is:

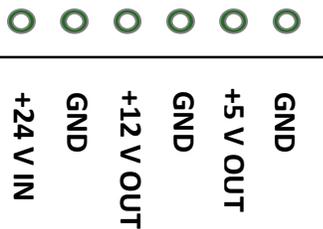
M57 V2000: Generates 2000mV at analog output channel: 1

M58 V5000: Generates 5000mV at analog output channel: 2

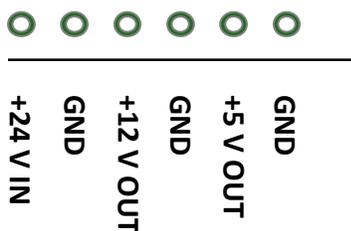
M57 V0: Generates 0mV at analog output channel: 1

12V AND 5V OUTPUTS:

A 12V and 5V output can also be drawn from the controller board. These two supply 100mA current and remain active until the power continues to the board.



CONNECTING INPUTS

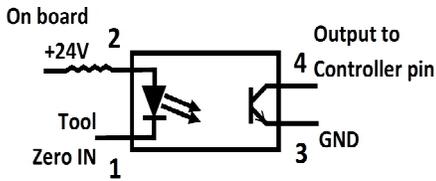


Inputs: The inputs are as following:

a. POWER SUPPLY

Connect a power supply of +24V DC at the input pin of CNC controller board for its operation.

Limit inputs, home inputs, tool zero input, block mode input and all the miscellaneous inputs are optoisolated.



b. TOOL ZERO INPUT:

An input signal is received on tool zero input pin when the tool touches the sensor plate. Tool Zero IN terminal receives this signal by a wire whose other end is connected to the tool by a clamp.

- 24 V
- TOOL ZERO IN
- BLOCK MODE IN
- MISC INPUT 1
- MISC INPUT 2
- GND
- 24 V
- X HOME IN
- Y HOME IN
- Z HOME IN
- 4TH HOME IN
- GND

-
-
-
-
-
-
-
-
-
-
-
-

c. BLOCK MODE INPUT :

An external input is given to the block mode input pin of the control board to execute the one line of G-code. In this mode, the motion stops at the execution of one programmed line and executes the next line on the trigger of an external input on this pin.

c. HOMING INPUTS:

Homing inputs are input to “HOME IN” terminals of CNC controller board from home switches of CNC machine for respective axes. These inputs can be set as active high or active low. Usually all the axes of CNC machine consists the Home switch. Disable the home switch function in Home Switch Settings menu for the axes which don’t have Home switch or for the axes for which home switches aren’t required.

- 24 V
- X LIMIT IN
- Y LIMIT IN
- Z LIMIT IN
- 4TH LIMIT IN

-
-
-
-
-

d. LIMIT SWITCH INPUTS:

Just like home switches, CNC machine can or can’t have limit switch. However, usually each axis of CNC machine have two limit switches, one mounted in +ve direction and other in –ve direction of axis. Axis limit input terminal shares the inputs from both switches of same axis. The limit switch can be set disabled if not required in limit settings function of IO Settings. Disable the limit switch function in Limit settings menu for the axes which don’t have the limit switch.

- GND
- 24 V
- MISC INPUT 3
- MISC INPUT 4
- MISC INPUT 5
- MISC INPUT 6
- GND

-
-
-
-
-
-
-

MISC INPUTS: Six terminals of misc inputs are provided on the board. Four input terminals are used and the rest two inputs terminals are made for future development

in board. These input terminals are Misc input terminal 3, 4, 5 and 6. And the function assigned to them are Hold/ resume, stop, start and emergency resectively. These functions are discussed further in manual.

CNC CONTROL BOARD CONNECTION WITH PEN DRIVE

Connect the USB flash drive to the USB slot on the control board as shown below:



The first LCD screen is displayed when the pen drive is detected by the controller.

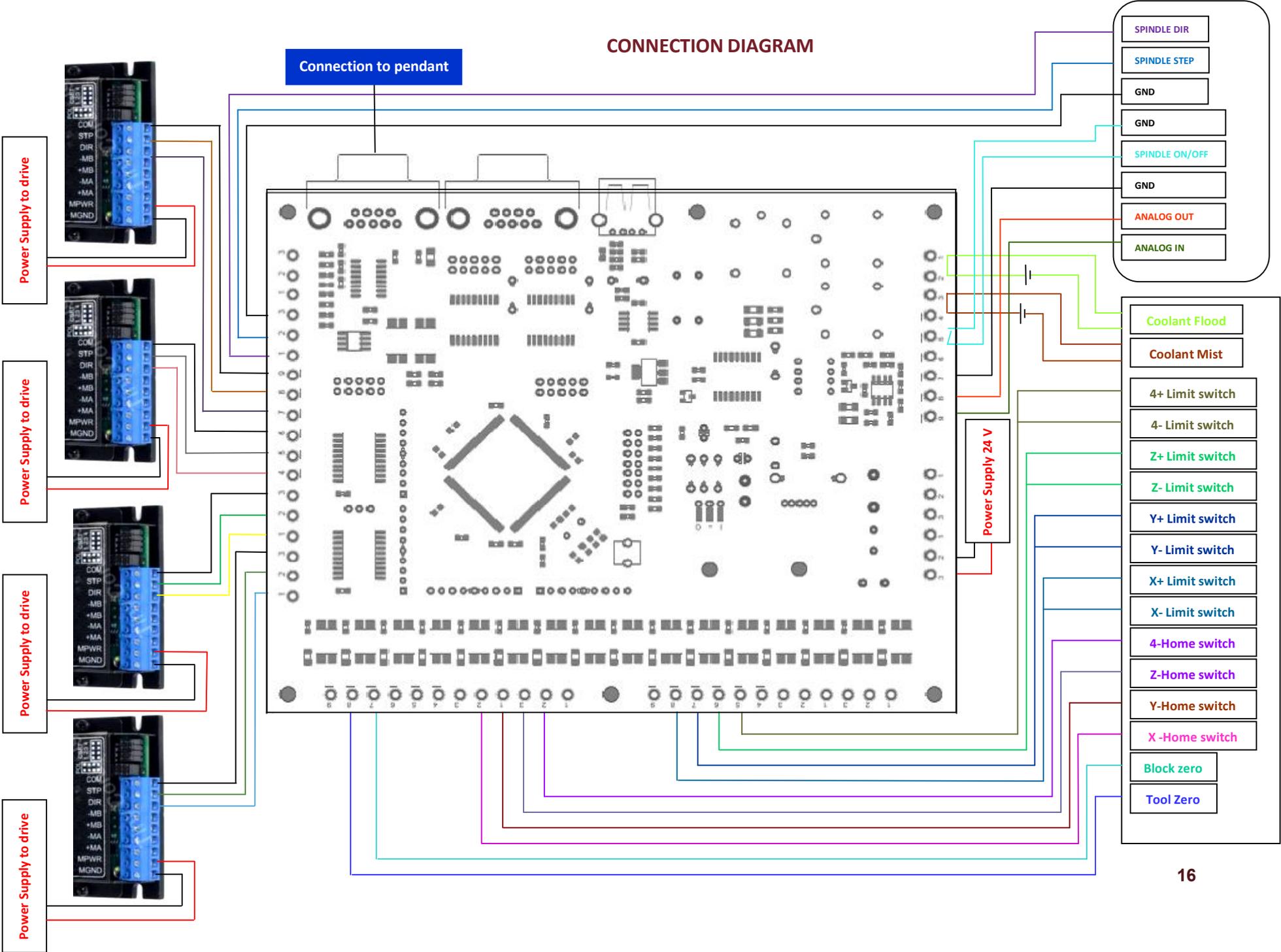


The second screen is displayed when the pen drive is disconnected from the control board.

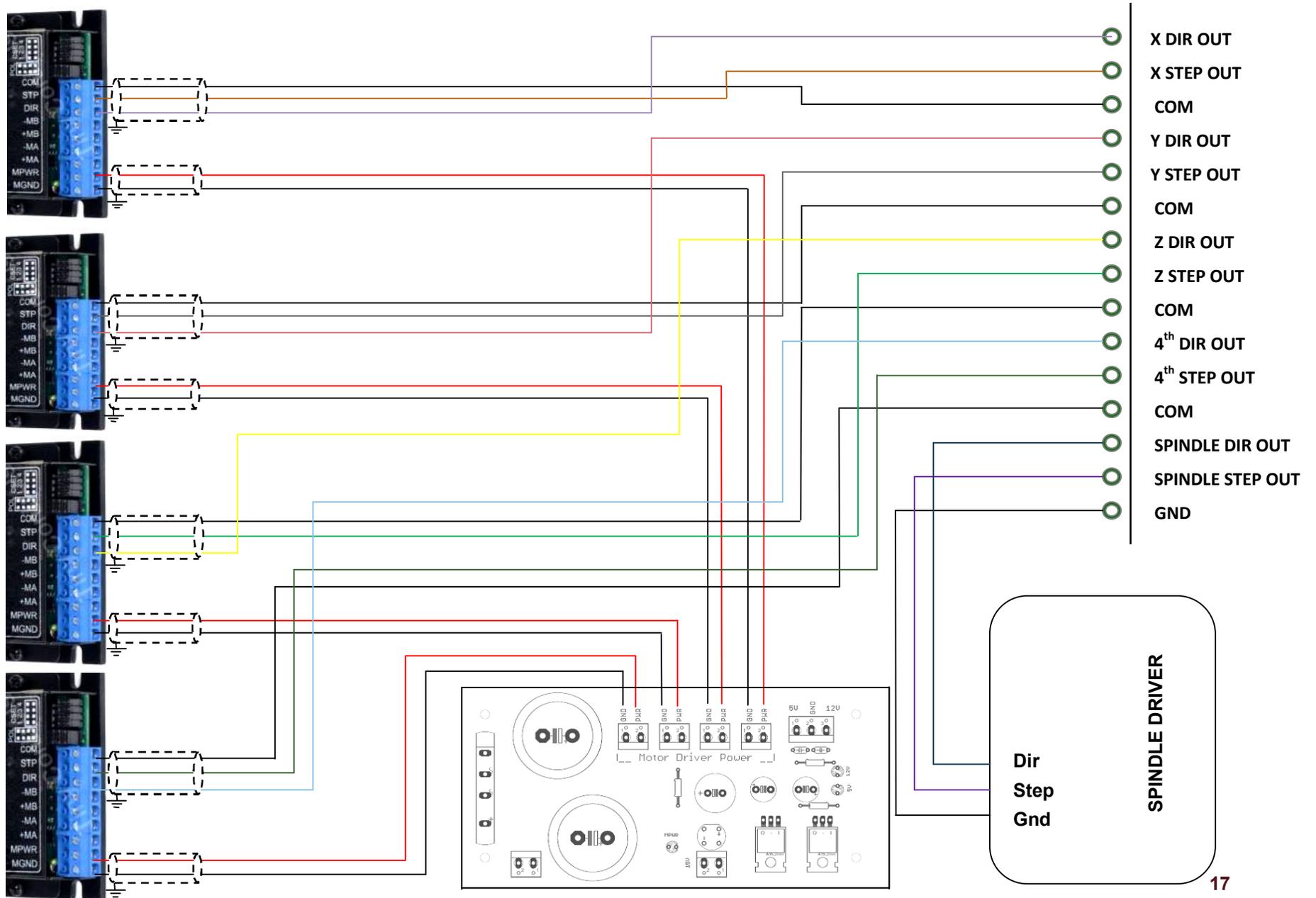
Note: Take care that the USB Flash drive should not be disconnected when the program is running on the Gcode. Handle the pen drive carefully.

Note: Connect the pen drive carefully. Reverse connections damages the pen drive. Don't connect the pen drive forcefully.

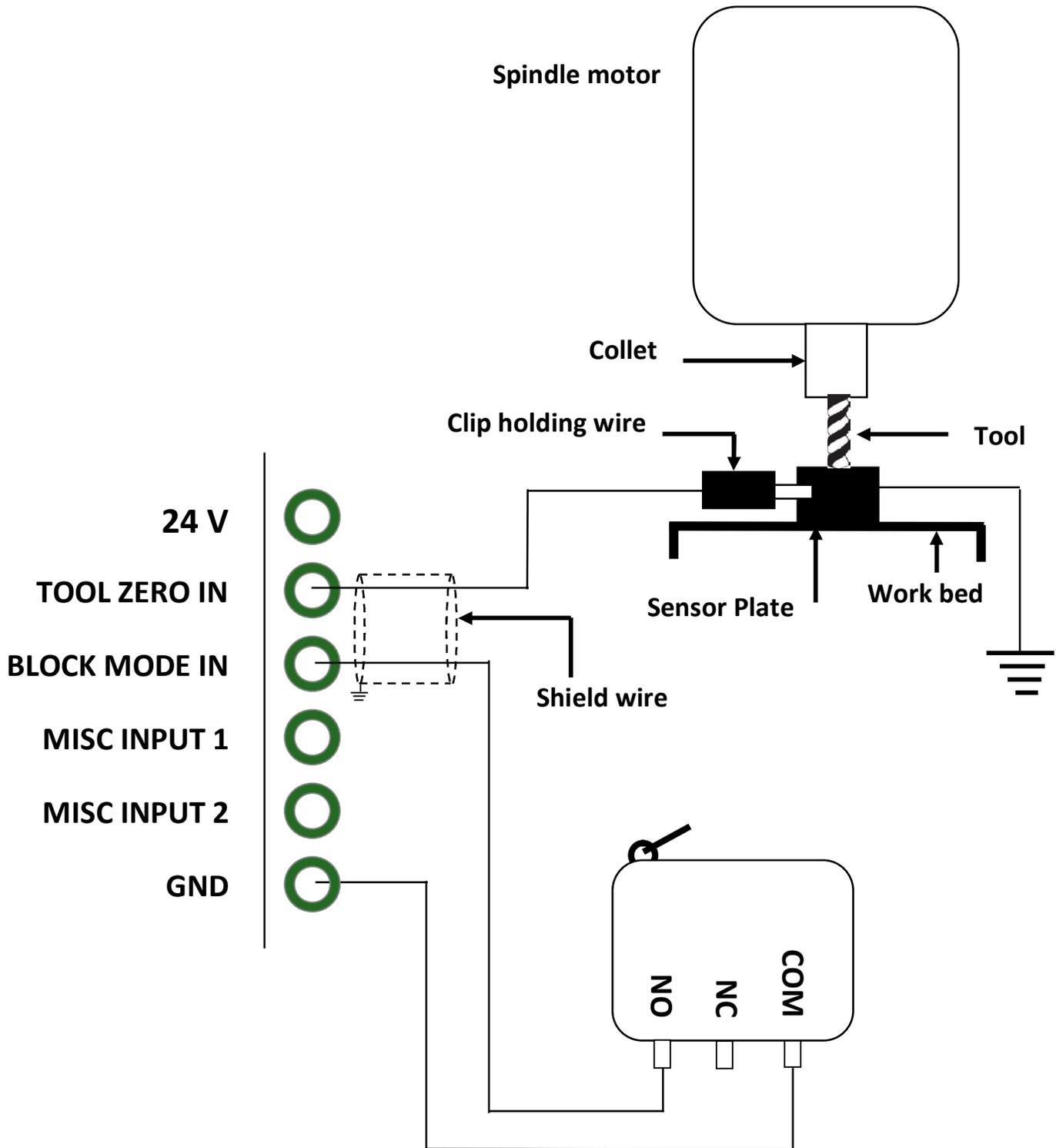
CONNECTION DIAGRAM



AXES AND SPINDLE OUTPUTS CONNECTIONS WITH DRIVES

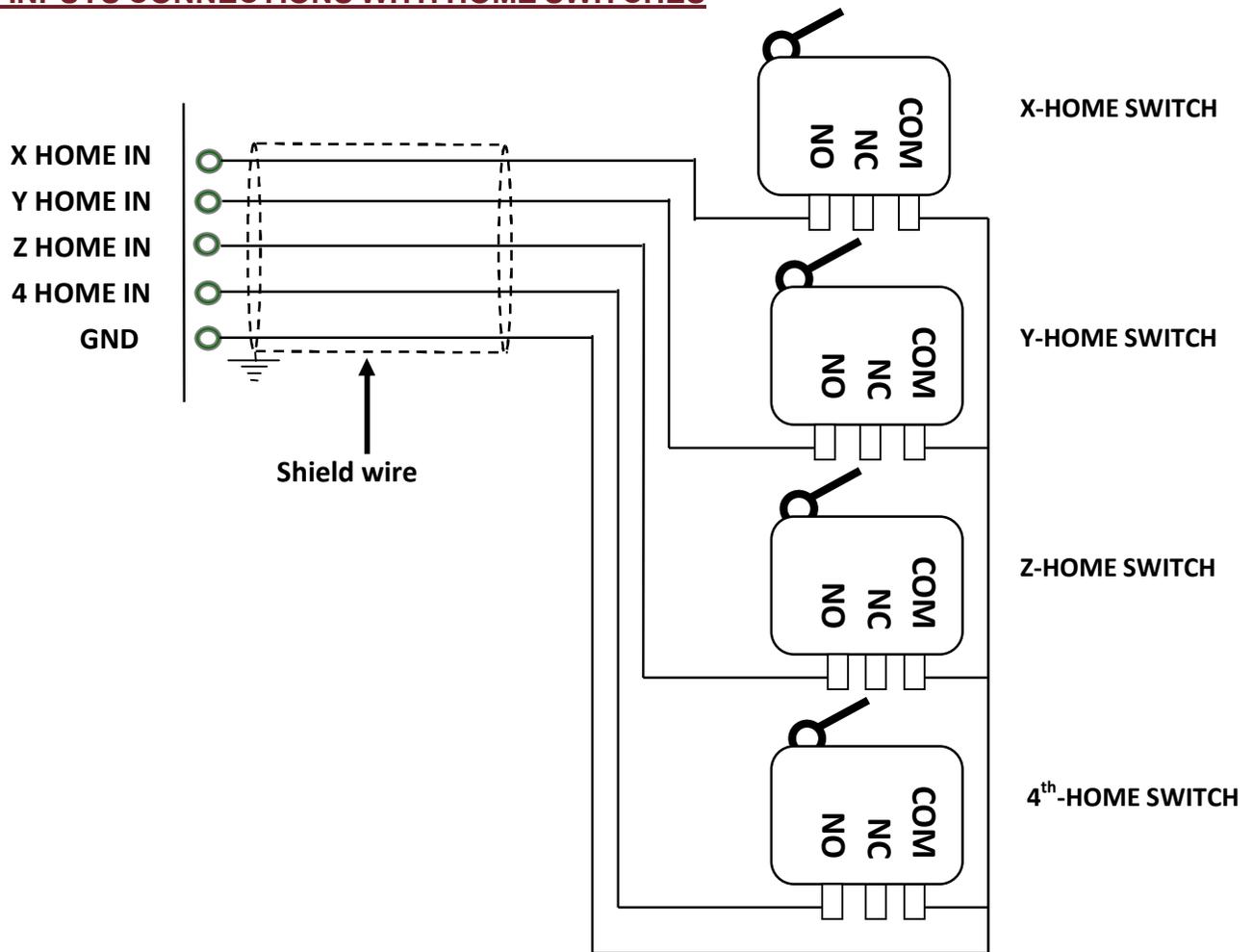


BLOCK MODE AND TOOL ZERO CONNECTIONS

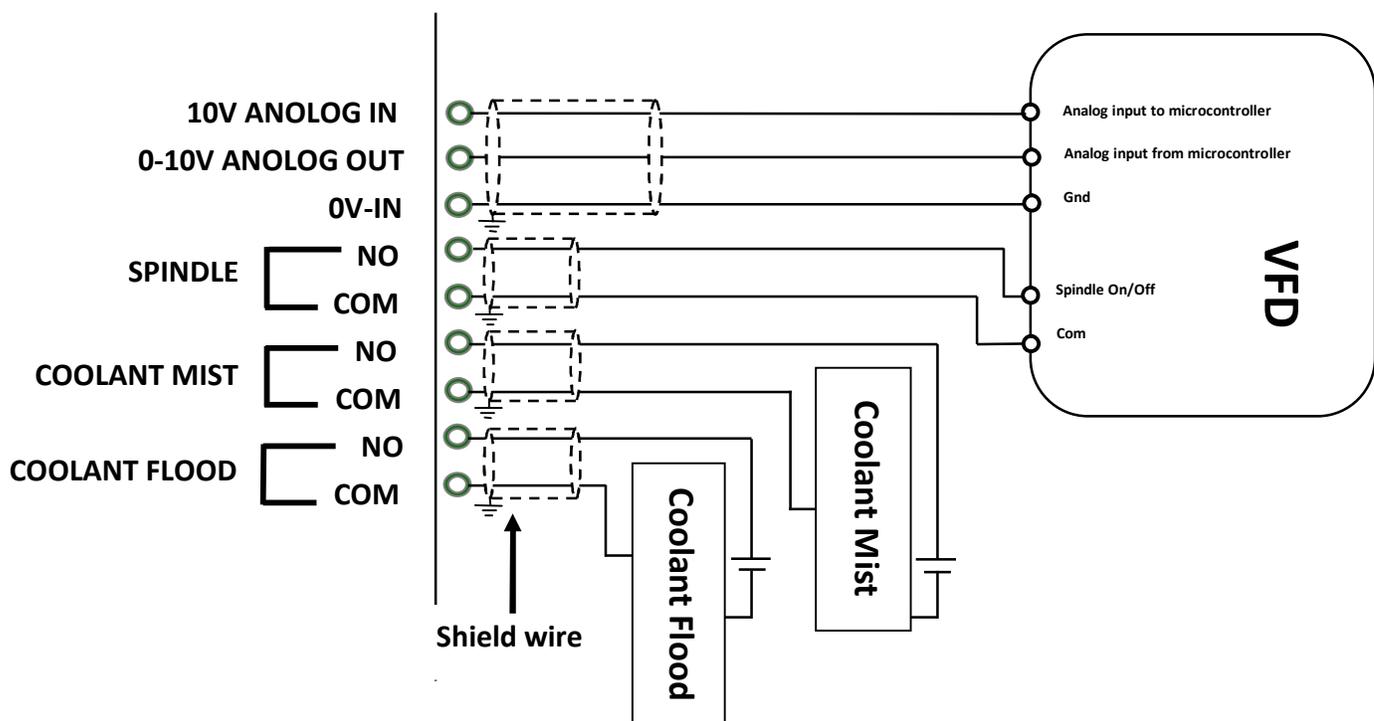


CNC control board should be installed in dust free environment with suitable temperature and humidity conditions. Read the **CNC Controller Specification table** for these parameters.

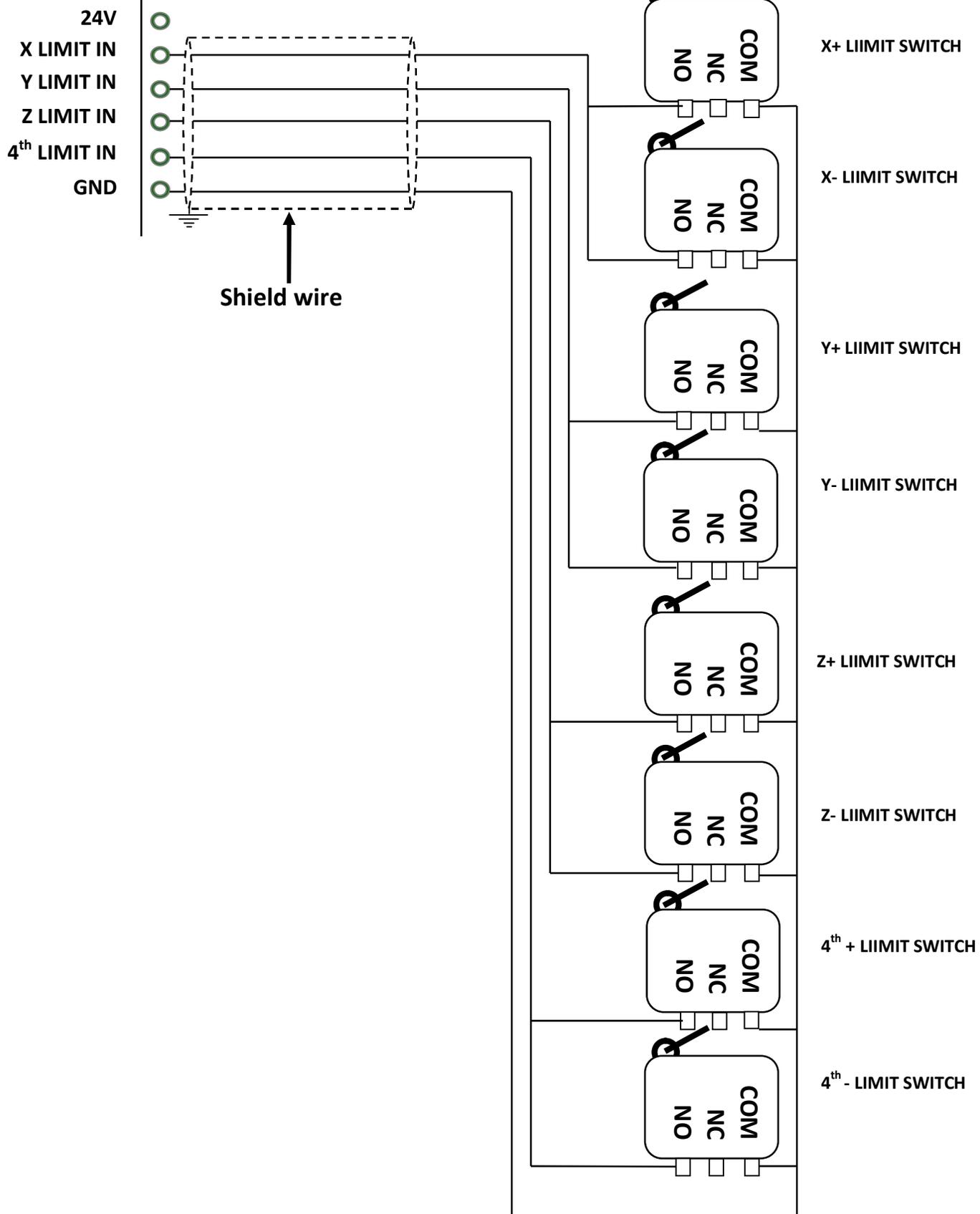
HOME INPUTS CONNECTIONS WITH HOME SWITCHES



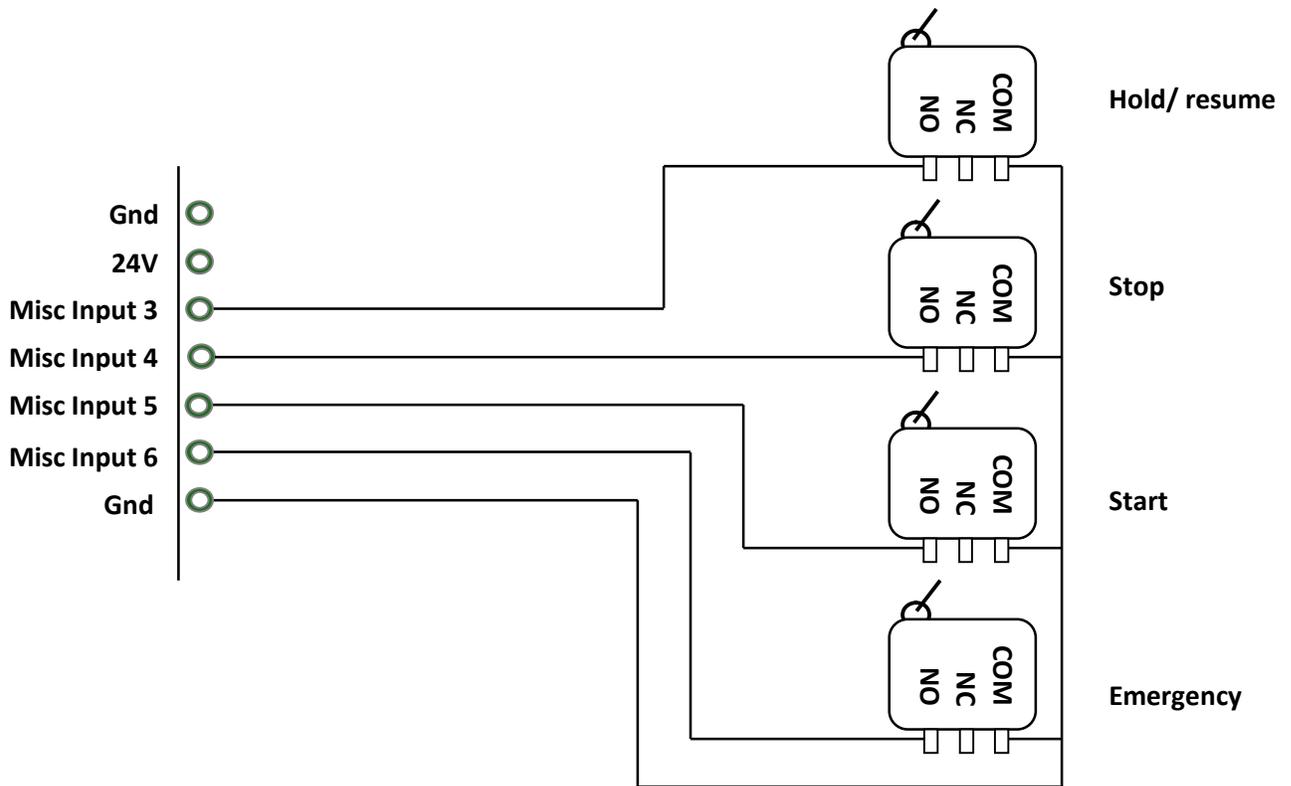
COOLANT, SPINDLE DRIVE AND VFD CONNECTIONS



LIMIT SWITCHES CONNECTIONS WITH CONTROLLER BOARD



MISC INPUTS CONNECTIONS WITH CONTROLLER BOARD



MOUNTING INSTRUCTIONS

CNC Controller board can be mounted in a box or can be used open. A proper arrangement of emergency button must be made in CNC controller board installation. If possible, board should be placed securely fastened to a smooth, flat surface. A suitable provision for the ventilation of heat (due to drives of steppers and spindle) should be installed else it can damage the controller.

1. Never use board in a space where there is no airflow or where ambient temperature exceeds 55 degree Celsius.
2. Never put board in conditions where humidity is 90% (non-condensing).
3. Don't use this product with strong magnetic field.
4. Protect it from dust, water and heat. Make sure that there should not be conductive particles near the control board.
5. Don't allow any liquid or other foreign body to get on controller board or on pendant.
6. Don't open the pendant for maintenance or modifications in its structure.
7. Be gentle when plugging in cables and pen drive.

INPUTS AND OUTPUTS SPECIFICATIONS FOR CONTROL BOARD

CONTROL OUTPUTS:

AXES, SPINDLE & COOLANT OUTPUTS:

Terminal Name	Type	Pin function
X-axis Step pulse	Buffered output	X-axis pulse output terminal output voltage = 5V current = 20mA
X-axis Dir signal	Buffered output	X-axis Signal output terminal output voltage = 5V current = 20mA
Y-axis Step pulse	Buffered output	Y-axis pulse output terminal output voltage = 5V current = 20mA
Y-axis Dir signal	Buffered output	Y-axis Signal output terminal output voltage = 5V current = 20mA
Z-axis Step pulse	Buffered output	Z-axis pulse output terminal output voltage = 5V current = 20mA
Z-axis Dir signal	Buffered output	Z-axis Signal output terminal output voltage = 5V current = 20mA
4-axis Step pulse	Buffered output	4-axis pulse output terminal output voltage = 5V current = 20mA
4-axis Dir signal	Buffered output	4-axis Signal output terminal output voltage = 5V current = 20mA
Spindle Step pulse	Buffered output	Spindle pulse output terminal output voltage = 5V current = 20mA
Spindle Dir signal	Buffered output	Spindle signal output terminal output voltage = 5V current = 20mA
Spindle Enable	Relay output	—
Coolant Mist Enable	Relay output	—
Coolant Flood Enable	Relay output	—

ANALOG OUTPUTS:

Terminal Name	Type	Pin function
Analog output terminal	Isolated	Output voltage =0–10V (When input is 10V) current = 20mA
Analog output terminal (0-10 V)	Non-Isolated	Output voltage = 0–10 V current = 20mA
Analog output terminal (0-10 V)	Non-Isolated	Output voltage = 0–10 V current = 20mA

A +12V and +5V output supplying 100mA current can also be drawn from the CNC controller.

INPUTS:

POWER SUPPLY:

Terminal Name	Pin	Pin function
System main power supply	24V DC	Power+
	GND	Power–



Tip:

+12V and +5V outputs are not controlled by microcontroller. These outputs remains active until the power supply continues to the board. Keep in mind that these can't be disabled by the pressing the emergency button.



Settings.tcs is a user made file in the USB Flash drive from which the settings for user programmable profile can be loaded to the profiles of the CNC controller.



Tip:

Input signal greater than 24V at input pins damages the CNC control board. Be sure about the polarity of the the power supply connections with control board.

Terminal Name	Type	Pin function
Tool zero	Optoisolated input	Set the tool to zero position
Block mode	Optoisolated input	One line of Gcode file is executed when an input is given at this pin
X-Home IN	Optoisolated input	Accept Homing input signal for X-axis
Y-Home IN	Optoisolated input	Accept Homing input signal for Y-axis
Z-Home IN	Optoisolated input	Accept Homing input signal for z-axis
4-Home IN	Optoisolated input	Accept Homing input signal for 4-axis
X-Limit IN	Optoisolated input	Accept Limit input signal for X-axis.
Y- Limit IN	Optoisolated input	Accept Limit input signal for Y-axis.
Z- Limit IN	Optoisolated input	Accept Limit input signal for Z-axis.
4- Limit IN	Optoisolated input	Accept Limit input signal for 4-axis.

MISC INPUTS: Six terminals of misc inputs are provided on the board. Four input terminals are used and the rest two inputs terminals are made for future development in board. These input terminals are Misc input terminal 3, 4, 5 and 6. And the function assigned to them are Hold/ resume, stop, start and emergency resectively. These functions are discussed further in manual.



Use the shield cable in connections to prevent the shorting of wires due to wear and tear of their insulation. The device should be operated in the vibration free conditions.

HAND HELD PENDANT



The Pendant can be taken as hand held control for a machine tool. This device is used in conjunction with CNC controller board to give user a complete access to the schematic and work area. The pendant and CNC control board makes a perfect combination of high speed and precise motion control. It allows the operator to control the axes upon which he/she wants the machine to move. It gives an ability to increase or decrease input feed rates and movement of the machine to help prevent the damage to the machine if it is over-performing.

It is fitted with an emergency button at the top to instantly stop the machine from continuing work if a large problem arises.

It is fitted with an array of keys for quick access in work process. A jog keys panel and separate navigation keys panel along with the numeric keys are provided for user access. It is embedded with an LCD to allow for details surrounding the work environment including machine parameters, offsets and locations. Refer to **table no 1.1** and **table no 1.2** to know about the keys functions in various modes. The description of these components is as following:

1. NUMERIC KEYS (0-9):

Numeric keys are to enter the numeric values for editing values of parameters. These keys also serve some different functions in different modes. All the changes done in numeric values of parameters are reflected on LCD.

2. OK & Esc:

OK button makes you to enter in main menu when controller is switched on. It also saves modified values in EEPROM when edited. Esc backs out of menus/modes.

Pressing Esc repeatedly makes control to return to the main menu. Pressing Esc in idle mode shows the product part number and the version of the CNC controller.

3. NAVIGATION KEYS PANEL (UP, DOWN, RIGHT, LEFT):

In editing, the navigation keys are used to navigate the cursor. These also serve the alternate function in some modes.

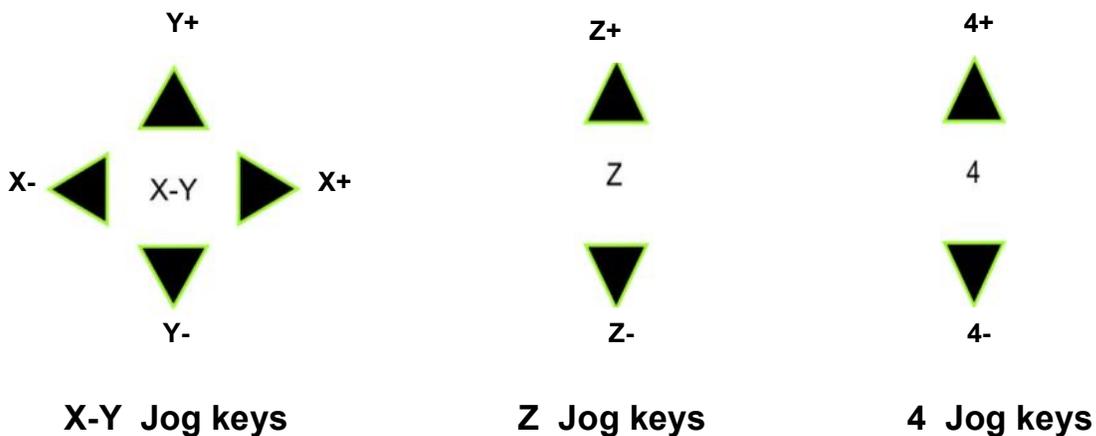


Up and down navigation keys move “*” up and down on LCD. Use left and right keys for changing the digit place. The navigation keys also perform some other functions in different modes.

4. JOG KEYS PANEL:

The Jog keys panel is given on the pendant for axes jogging. Z jog keys are also used for feed override purpose and 4th jog keys are used for speed override purpose.

- **X-Y JOG KEYS:** The X and Y jog keys are used to move position of the machine on the table work area. This is used to position the work piece suitably. The X-jog keys move the X-axis left and right. The Y-jog keys move the Y-axis front to back on the table. These are shown as:



- **Z JOG KEYS:** These keys move the Z-axis up and down.
- **4 JOG KEYS:** These keys move the 4th axis in clockwise and anticlockwise direction.

5. LCD DISPLAY (20x4):

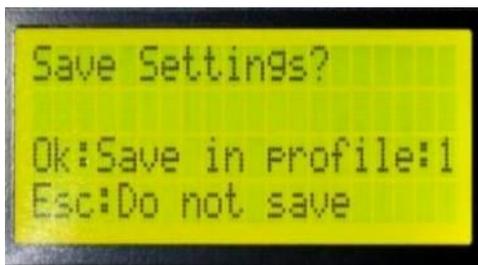


An LCD is incorporated into the design for the pendant to give the user an ability to access the information of parameters. It indicates all the information related to **MENUS** or **MODES**.

- The ‘*’ shows the current selected menu function. To enter in a menu function, move * to the function by using up-down navigation key and press Ok.
- The ▲ symbol at upper corner in right side of screen of LCD shows a sign notifying the menu functions above.
- The ▼ symbol at lower corner in right side of screen of LCD shows a sign notifying the further menu functions below.

After modifying the settings, Press Ok to **confirm** the settings and return to previous menu. In this condition only, modifications in settings are saved. Press Esc to **cancel** the modifications in settings and return to previous menu. In this condition, all the previous saved settings retain.

After **confirming** the changes made, before exiting to main menu, it is required to **save** the modifications in settings in the desired profile else controller retains the previous settings. Press Ok to save these settings.



Adjacent screen appears when Esc is pressed for exiting back to the **Main Menu**.

Select profile using **key no 1–4** or using **up-down** navigation keys. Press Ok to save these settings in required profile. Press Esc to exit to **Main Menu**.



Don't change the settings saved in settings.tcs by user on PC or any other media because their correctness is not checked while reading.

EMERGENCY BUTTON

User can give an emergency input by a red color button at the top of pendant. When this button is pressed, the motion stops however it does not cut down the power supply to CNC controller. On pressing the button, the machine instantly stops since all the “**control outputs**” from the controller board get disabled.

The following message is displayed if Emergency button is pressed.



Lock out the emergency button. A message “**Press key 8 to resume**” appears. Press key 8 to resume. If the controller was in idle mode before pressing the emergency button, machine remains at same position after resuming. However, if any file is running and emergency button is pressed, machine starts from its current position and moves to the home position on resuming. From home position, X-axis, Y-axis and 4th - axis move to the coordinate position at which emergency button was pressed. All the outputs which were enabled previously get enabled again and a message “**If Spindle On Press Ok**” appears. If the spindle is off, then press Esc key and cancel the process. If the spindle is on, press Ok key. Z axis moves to the coordinate position as before the emergency button press. Here, the controller starts to execute the code one line before in G-code at which emergency button was pressed and runs the file in same manner.

BRIGHTNESS SETTINGS FOR LCD

Adjust the brightness of CNC controller by following these steps:

- Press and hold key “1” when the CNC controller is **Switched ON** to enter in brightness setting function.
- Increase/ decrease the brightness by using “up/down” navigation keys in brightness setting function. Press “Ok” to return to coordinate screen.

KEYS FUNCTIONS IN VARIOUS MODES

Most of the keys have different functions in different modes. Following table shows the functioning of keys in various modes. Following are the functions of numeric and the jog keys.

KEY	IDLE MODE	HOLD MODE	RUN MODE
X-	Jog X-	Jog X-	-
X+	Jog X+	Jog X+	-
Y-	Jog Y-	Jog Y-	-
Y+	Jog Y+	Jog Y+	-
Z-	Jog Z-	Jog Z-	-
Z+	Jog Z+	Jog Z+	-
4-	Jog 4-	Jog 4-	-
4+	Jog 4+	Jog 4+	-
0	Tool zero	-	-
1	X-Zero	X-Zero	-
2	Y-Zero	Y-Zero	-
3	Z-Zero	Z-Zero	-
4	4-Zero	4-Zero	-
5	File select	Spindle ON/OFF	-
5 (Long press)	File check	-	-
6	Override screen	Override screen	Override screen
7	Go to Zero	-	-
7 (Long press)	Go to Home	-	-
8	Run from given line number	Resume	Pause
9	File Start	-	-

Table 1.1



Tip:

If connection between the pendant and controller is lost when the machine is running according to the file loaded to controller, the machine continues to run according to G-code file, Press the emergency button (to be attached by user) immediately else switch off the CNC machine.

KEY	IDLE MODE	HOLD MODE	RUN MODE
LEFT	Display WC/MC*	Display WC/MC*	Display WC/MC*
LEFT (Long Press)	Manual override mode	Manual override mode	-
RIGHT	Jog step/Jog Cont./Jog fast	Jog step/Jog Cont./Jog fast	Jog step/Jog Cont./Jog fast
RIGHT (Long Press)	Jog Offline	Jog Offline	-
UP	Jog Factor+/ Step-size change**	Jog Factor+/ Step-size change**	Jog Factor+/ Step-size change**
UP (Long press)	Jog Factor + (by 10)	Jog Factor + (by 10)	Jog Factor + (by 10)
DOWN	Jog Factor-/ Step-size change**	Jog Factor-/ Step-size change**	Jog Factor-/ Step-size change**
DOWN (Long press)	Jog Factor - (by 10)	Jog Factor - (by 10)	Jog Factor - (by 10)
OK	Menu	-	-
Esc	About Screen	Stop	Stop

*: Displays the Work coordinates and Machine coordinates.

** : Jog factor can be changed for Continuous jog and Fast jog mode. Step-size can be changed for Step Jog mode.

Override screen:

KEY	OVERRIDE SCREEN
Z-	Feed rate -
Z+	Feed rate +
4-	Spindle speed -
4+	Spindle speed +
Y+	Feed rate reset
Y-	Spindle speed reset

Table 1.2

CNC CONTROLLER BOARD CONNECTIONS WITH PENDANT

The controller makes communication with pendant through RS-232 serial port. There are two serial communication channels on the CNC controller board. One dedicated Serial communication channel is made between the hand held pendant and the CNC control board while the other channel can be made between the Controller board and any peripheral. Both serial ports of controller board can work on the same time.

Each pendant is shipped with an already connected serial cable. Connection between CNC control board and pendant is made through an on-board DB-9 (F) connector and a DB-9 (M) connected at other end of serial cable. If the communication between the Pendant and the CNC controller board lost, a message “Controller disconnected” appears on the LCD. When the connection resolves back, LCD shows the coordinate screen.

Notes: Some points to be kept in mind while using serial communications:

1. Operators should avoid installing serial cable next to high voltage lines and prevent any foot traffic from occurring over or across the serial cables.
2. The serial cable should not be involved in circumstances where damage is probable and operation over wire shouldn't exceed wire temperature beyond 100 degree Celsius.
3. The serial cable should not be operated next to high voltage or fluorescent lights which leads to error in data transmission between serial port and pendant.
4. The cable should be checked not to be cut or damaged.
5. There should not be loose connections else leads to poor transmission quality and difficulties in transmissions.

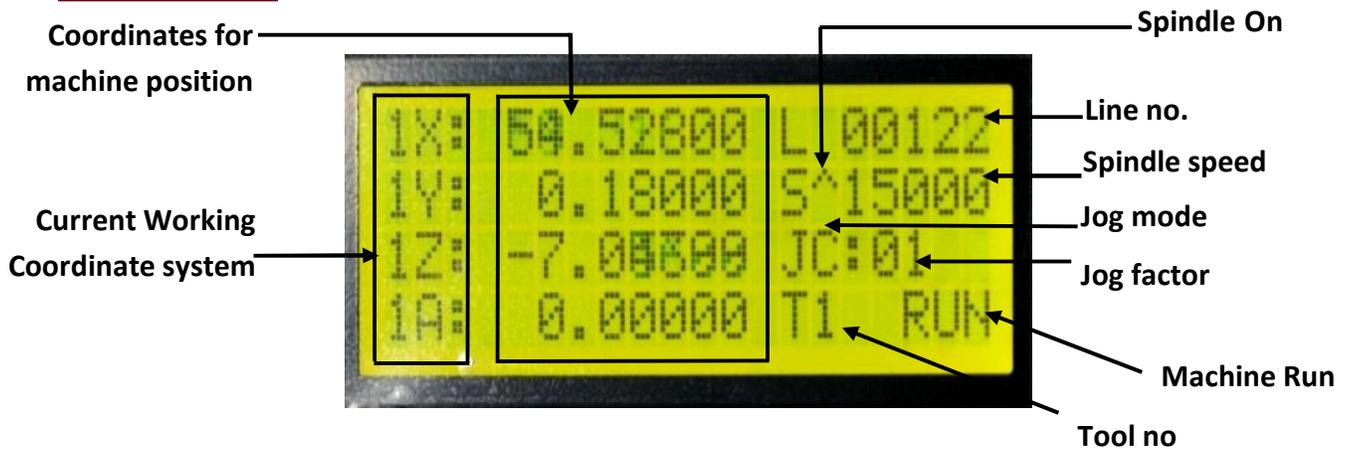
 **Note:** Don't operate the controller or the pendant with wet hands. Be sure about the connection of pendant with dedicated connector on the CNC Controller board.

 **Note:** Take care that pendant should not be dropped. The LCD can be damaged due to the external mechanical strokes. Handle the pendant carefully.

 **Note:** Default values are stored in on chip ROM of every controller and it can't be altered by user.

DESCRIPTION OF DISPLAYS IN VARIOUS STATES OF MACHINE

MACHINE RUN:



The illustration for the above screen is shown below:

Current working Coordinate system: Here “1” with 1X, 1Y, 1Z and 1A show that Current working coordinate system is “coordinate system 1”. CNC controller supports 6 incremental coordinate systems (G54-G59) and an absolute coordinate system (G53).

Coordinates for machine position: These coordinates shows the machine’s current position w. r. t. current WCS.

S: It shows the spindle speed at which the spindle is rotating. Spindle rotates if it is enabled and made ON in the Gcode file. Refer to spindle settings in **IO Setting** for method of enabling the spindle.

S^: ^ shows that spindle is on. Absence of this symbol with “S” shows that spindle is off.

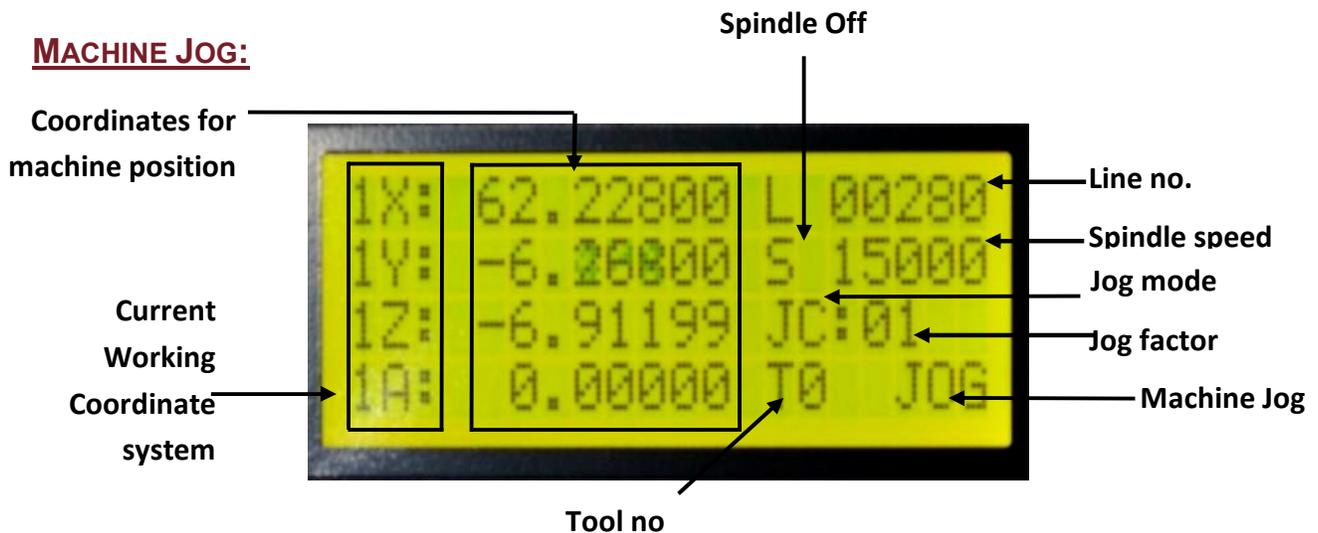
Jog mode: This term shows the current Jog mode set (Jog continuous mode for above screen). Its value shows the jog factor or jog step set as per jog mode selected. A provision of three modes is given in CNC controller.

L: L shows the number of current line being in execution in the Gcode file.

Tool number: T1 shows that tool number 1 is selected and being in operation. A provision of 6 tools is provided in CNC controller. Refer to **Tool Settings** menu for selecting the tool.

Machine Run: Machine run shows that the machine is in Run Mode.

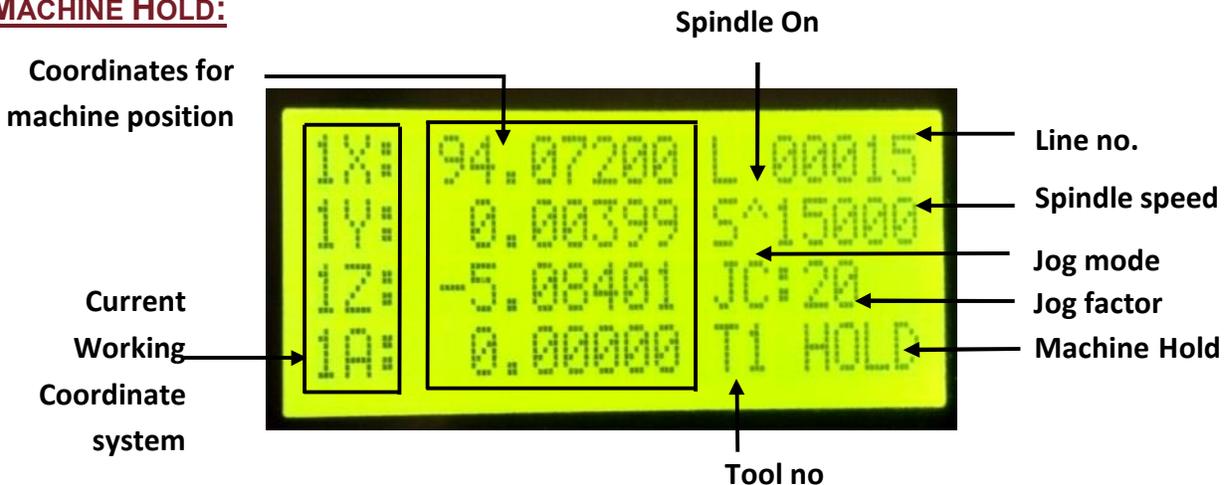
MACHINE JOG:



The current working coordinate system, Coordinates for machine position, Tool number, Line number, Jog mode, Jog factor, Spindle Speed, Spindle On/Off have their usual meaning.

Machine Jog: Machine jog shows that the machine is jogging according to jog mode selected. A provision for three jog modes is given. Refer to [page 55](#) for jog modes and [table no 1.1](#) for the jog keys and other keys for modifying the jog value.

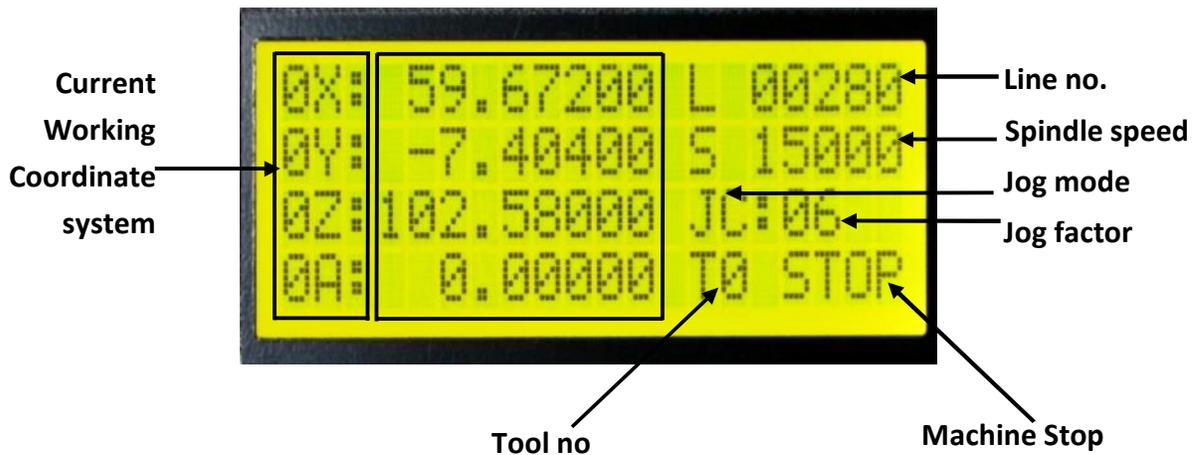
MACHINE HOLD:



The Current working coordinate system, Coordinates for machine position, Tool number, Line number, Jog mode, Jog factor, Spindle Speed, Spindle On/Off have their usual meaning.

Machine Hold: Machine Hold shows that the machine is in Hold mode.

MACHINE STOP:



Current working Coordinate system: Here “0” shows that machine produces the coordinated motion to the programmed point in “Absolute coordinate system”.

The Coordinates for machine position, Tool number, Line number, Jog mode, Jog factor, Spindle Speed, Spindle On/Off have their usual meaning.

Machine Stop: Machine stop shows that the machine is in idle mode.

Tip: Use jog fast mode if the required jog factor is around 30 to 80. It is advised to keep machine at a safe height to avoid any damage.

Tip: It is also possible to perform simultaneous jogging in multiple axes by pressing respective keys in **continuous** and **fast jog mode**.

OPERATING THE CNC CONTROLLER

MAIN MENU:



When the power is applied to the control board, the “ON” status is displayed by the glowing of red LED on the control board and a splash screen appears on LCD showing version and the product part number. If the pen drive is not mounted, a message appears as shown by the screen adjacent.

Pressing any key shows the splash screen for a second and then shows coordinate screen. Mount the pen drive at the pen drive slot. Screen shows the message “**USB DEVICE DETECTED**”.



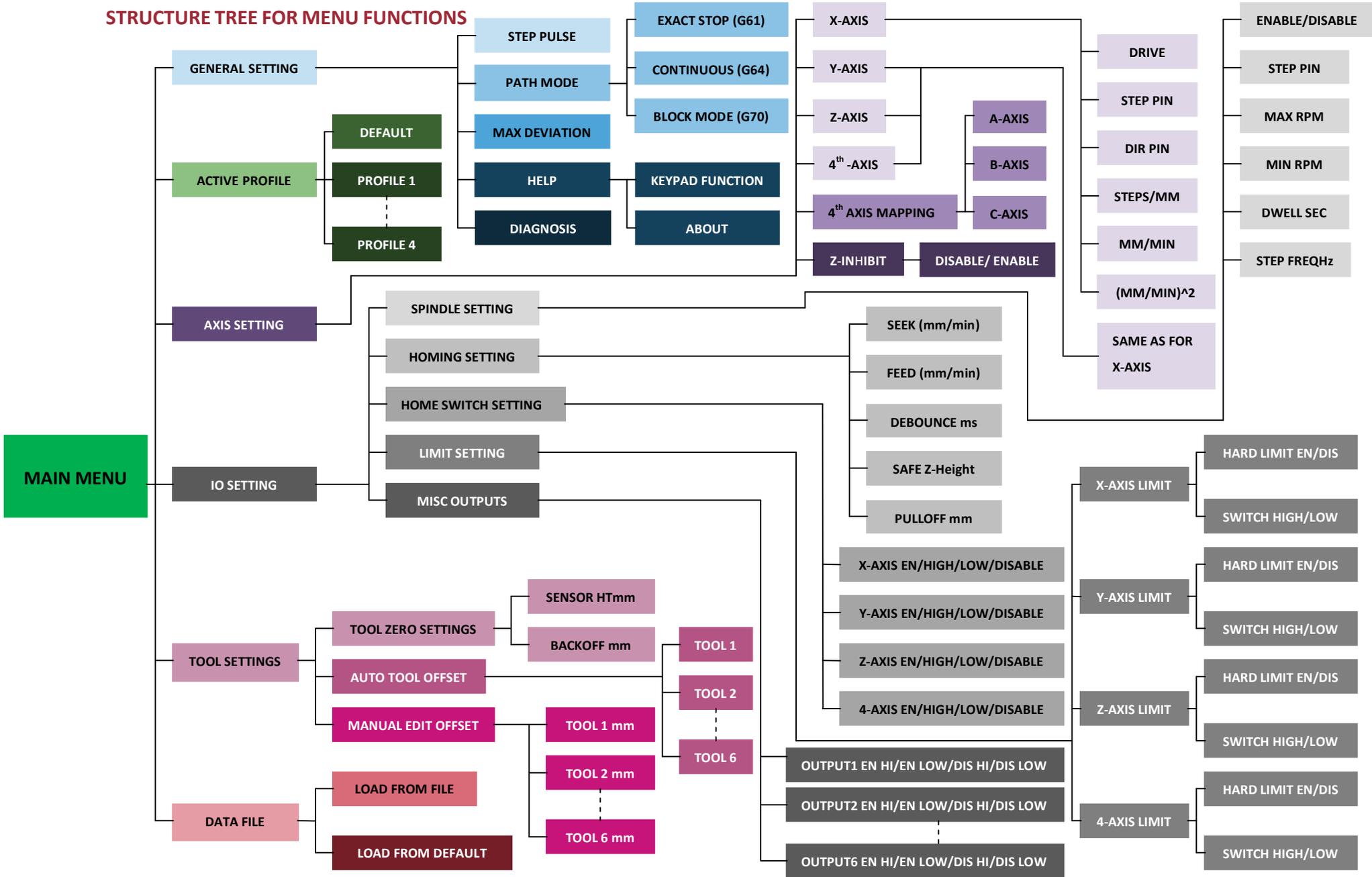
After complete detection of the USB flash drive, a message “**Press key 8 to resume**” appears. Pressing Esc shows the coordinate screen. Adjacent screen results on pressing key 8.

Pressing Esc shows coordinate screen on LCD. Press Ok to enter in **MAIN MENU**. Functions in Main Menu help operator to handle the operation of CNC. These functions are discussed in the subsequent sections on the manual.



Note: Be sure about the polarity of external power input. Refer to terminal diagram for input polarity check.

STRUCTURE TREE FOR MENU FUNCTIONS



GENERAL SETTINGS

The **GENERAL SETTINGS** function adjusts **Step pulse width**, **Path mode** & **Max deviate**. **Keypad function** shows the function of all the keys on pendant in various modes. **Diagnosis function** allows the operator to diagnose inputs and outputs of the CNC controller.

STEP PULSE: This function adjust the width of step pulse for stepper drives of stepper motor. Step drives for all axes operate with same step pulse width. The unit for step pulse width is uS. The maximum Step pulse width is:

$$\text{Step pulse width max} = (4 * 1, 50, 000 * 60) / ((\text{Steps/mm}) * \text{velocity})$$

Where, $((\text{Steps/mm}) * \text{velocity})_{\text{max}}$ is the product of steps/mm and velocity of an axis for which this product value is maximum in contrast to rest three axes.

PATH MODE: Path modes are set to control machine when it changes the path (for example at junctures). CNC controller supports following three path modes:

- **EXACT STOP (G 61):** In exact stop mode, machine follows a programmed path as exactly as possible, slowing at sharp corners of the path.
- **CONTINUOUS (G 64):** In the continuous mode, sharp corners of the path can be rounded slightly so that speed can be kept up. In continuous mode, control tries to keep the velocity constant and doesn't try to keep controlled point exactly on the path. Rather, at junctures in between the motion where direction changes sharply, the corner is rounded.
- **BLOCK MODE (G 70):** In the BLOCK MODE, the motion stops at the execution of one programmed line and executes the next line on the trigger of an external input at pin "**BLOCK IN**". In the BLOCK MODE, the control stops motion at the end of one programmed move exactly after the completing the execution of G-code line. The control begins motion at the trigger of external input and again stops after executing subsequent G-code line. The process continues until the G-code ends. The stop is preceded by deceleration so that motion is kept at the feed rate for as long as possible.

- **MAX DEVIATE:** Maximum Deviate function forms the curve at the junction according to the deviation factor set. Range for max deviate factor is 1-999. If it is set 1, a curve is formed with the deviation of 0.01mm from the junction point. Higher the value of max deviate, greater is the distance of curve from the junction point.

HELP: screen shows the Keypad function and about function. Their description is as below:

- **ABOUT:** 'ABOUT' screen shows product part number and version of the controller.
- **KEYPAD FUNCTION:** 'KEYPAD FUNCTION' screen shows the keypad function in various modes of CNC controller. The keypad function allows the operator to learn the function of all keys in a particular mode. The function of the keys in the entire above mode is discussed in **Key Functions in various Modes**.

DIAGNOSIS FUNCTION: Diagnosis function makes the user to analyze the proper functioning of inputs and outputs of the CNC controller board.

Method to access GENERAL SETTINGS is following:

1. Enter in **MAIN MENU**.
2. Enter in "**General settings**".

Method to access STEP PULSE:



- a. Enter in Step Pulse.
- b. Edit the "**SET STEP PULSE WIDTH**" and confirm or cancel the settings and exit to previous menu.

Method to access PATH MODE:

EXACT STOP (G61):

- a. Select and enter in the **PATH MODE**.
- b. Select the **Exact stop** and confirm or cancel it to exit to previous menu.

CONTINUOUS MODE (G64):



- a. Enter in path mode and select Continuous. Adjacent screen appears.
- b. Confirm or cancel it and exit to previous menu.

BLOCK MODE (G70):

- a. Enter in path mode and select **Block mode**.
- b. Confirm or cancel it and exit to previous menu.

Method to access Max Deviate menu:

- a. Enter in Max deviate.
- b. Edit the **Max DEVIATE FACTOR** and confirm or cancel the settings and exit to previous menu.

Method to access HELP menu>>Keypad menu:



- a. Select **HELP** and enter.
- b. Select the Keypad function in **HELP menu** and enter.
- c. Select any mode and enter. It shows functions of all keys in selected mode. Exit to previous menu.

Refer to **table no 1.1** and **table no 1.2** for the keys function in the various modes.

Method to access HELP menu>> ABOUT menu:

- a. Select the about function in HELP menu and enter. About screen appears showing the product part number and the version of CNC controller.
- b. Press Esc to exit to previous menu.
3. Save the confirmed settings in required profile or exit without saving.

 **Tip:** Don't change the name of the file in pen drive.

Method to access Diagnosis menu:

INPUT ANALYSIS: When any input is received by the CNC controller, it shows the corresponding input name on the LCD screen.

Following table shows the method to diagnose the outputs:

	Output	Key to be pressed	Action to be performed by CNC
Axes outputs	X-axis outputs	Key X+	X-axis moves in X+ direction.
		Key X-	X-axis moves in X- direction.
	Y-axis outputs	Key Y+	Y-axis moves in Y+ direction.
		Key Y-	Y-axis moves in Y- direction.
	Z-axis outputs	Key Z+	Z-axis moves in Z+ direction.
		Key Z-	Z-axis moves in Z- direction.
	4-axis outputs	Key 4+	4 th -axis moves in 4 th + direction.
	Key 4-	4 th -axis moves in 4 th - direction.	
Spindle outputs	Spindle Enable relay	Key 1	Green LED glows for spindle relay, isolated output= +5V
	Spindle start	Key 4	Spindle rotates in CW direction
		Left Navigation key	Spindle starts to rotate CCW
		Right navigation key	Spindle starts to rotate CW
	Isolated analog output	Up navigation key	Spindle speed increases, isolated output>+5V
	Down navigation key	Spindle speed decreases, isolated output <+5V	
	Non isolated analog output 1-2	Key 5	+5V at both the output terminals
Non isolated analog outputs		Up navigation key	Non-isolated outputs 1&2 >+5V
		Down navigation key	Non-isolated outputs 1&2 <+5V
Coolant outputs	Coolant Flood	Key 2	Green LED glows for Coolant flood relay
	Coolant Mist	Key 3	Green LED glows for Coolant mist relay
Misc outputs	Misc output 1-6	Key 6	Misc output 1-6 ON
	Misc output 7-14	Key 7	Misc output 7-14 ON
	Misc output 15-22	Key 8	Misc output 15-22 ON

Analyze all the inputs and outputs One by one carefully. Don't forget to save the changes made in settings for each function/menu in the required profile after confirming them else controller retains the previous settings.

All the outputs of control board can be made OFF by pressing key 0. Outputs can't be made OFF individually. Press Esc to exit. A message for "**Diagnosis complete: System Reboot**" appears.

ACTIVE PROFILE

ACTIVE PROFILE menu make the operator to select a profile and store the General settings, Axis settings, IO settings for spindle, homing and Tool settings for the operation to be controlled by CNC Controller. These settings are loaded through a file "settings.tcs" stored in pen drive to the active profile.

A provision for four user programmable profiles is given in CNC controller and a default profile is given. All user programmable profiles work same as default profile if no parameters are changed in them.

Settings.tcs is a user created file in USB Flash drive and it stores the settings for all the profile. These settings can be loaded to all the profiles simultaneously.

All the default settings are stored in on-chip ROM of every controller at the time of shipment and can't be altered or no other settings can be loaded to default profiles.

Method to access ACTIVE PROFILE:



1. Enter in **MAIN MENU** and select **Active profile** menu item.
2. Enter in **Active profile**. Select the required profile and save it.

AXIS SETTINGS

AXIS SETTINGS menu functions make the user to modify settings of the axes parameters in profiles. CNC controller controls an independent mechanism of machine which produces relative linear motion of tool and workpiece in three mutually orthogonal directions called X, Y and Z axes and an additional mechanism producing the relative rotation of the workpiece and tool around an axis. It is the rotational axis and can be set along any of the linear axes and labeled as A, B and C. A-axis

describes the motion around X-axis, B-axis describes the motion around Y-axis and C-axis describes the motion around Z-axis.

For **AXIS SETTINGS**, make changes in the parameters discussed in this section and save them in desired profile.

FOR X-AXIS, Y-AXIS, Z-AXIS & 4TH AXIS: The functioning of 3 linear axes and a rotational axis depends on the following parameters. Make the following settings for all required axes, one by one. These settings are:

- **DRIVE: ENABLE/DISABLE: ENABLE/DISABLE** is for enabling or disabling the axis drive. The drive mechanics of CNC machines convert torque provided by the electric motors into linear motion of the tool head. Step and Dir outputs are disabled for the axis whose drive has been disabled.

- **STEP PIN: LOW/HIGH:** The **STEP PIN** selects the state of the step pulse given to drive of stepper motor (connected to axes). All axes have their dedicated Step output pins on the control board; however step pulse width is same for all the axes. When this pin is active high, an active high step pulse is driven to respective axis drive. When the pin is active low, an active low step pulse is driven to the axis drive.

- **DIR PIN: LOW/HIGH:** This **DIR PIN** selects the state of direction signal given to the drive of motor attached to axis. All axes have their dedicated Direction output pins on the control board and axes can have different direction settings according to the job to be performed. When this pin is active high, the machine/tool head moves in positive or negative direction according to the commands given in G-code file or by jog keys.

And when the pin is active low, machine/tool head moves in the opposite direction to the one in active high state. The direction for X axis on the work bed of the machine can be either left or right, for Y axis is Back or Front whereas for Z axis is either up or down. However for rotary axis, it is either clockwise or anticlockwise.

- **STEP/MM: STEPS/MM** shows the number of steps the motor must turn in order for the CNC machine to move 1 mm on the particular axis. This is machine dependent and remains same for a particular axis of machine. Different machines can have different values of steps/mm. The axes of the machine can have different “step/mm” number.

- **MM/MIN:** It shows the maximum velocity of the axis. Different axes can have different velocities. The maximum achievable velocity for all axes is:

$$\text{Velocity (max)} = (1, 50, 000 * 60) / (\text{Steps/mm})$$

Steps/mm remains fixed for axis. The pulse rate varies in accordance with velocity (**MM/MIN**) for particular axis. Maximum achievable pulse rate for all axes is 150 KHz.

- **MM/SEC²:** It is the increment of the processing velocity from initial velocity to the highest one for the motor connected to axis. The value of the acceleration is user programmable and can vary for all axes. The range of acceleration for all axes is 1 to 99999.

Set all these parameters for all the axes, individually.

4TH AXIS MAPPING: 4th axis mapping decides whether the rotary axis is A-axis, B-axis or C-axis. Set the parameters for the rotational axis in the same way as for the linear axes.

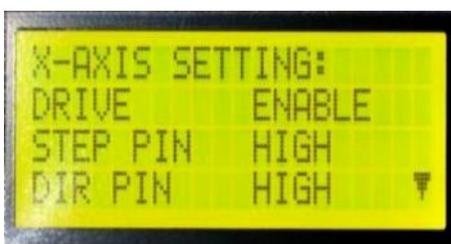
Z-INHIBIT ENABLE/DISABLE: Z-inhibit function allows the user to inhibit the Z axis. Z axis does not move if Z-inhibit function is set enabled. Any command line written for Z axis motion in G-code file does not work if Z inhibit is enabled.



Following is discussed the way to access the Axis setting:

1. Enter in **MAIN MENU** and select **AXIS SETTINGS**.
2. Enter in **AXIS SETTINGS**.

Method to access Axis settings>> X-axis:



- a. Select required axis (Here **X-axis** is selected) and enter.
- b. Select **DRIVE** and enable or disable it.
- c. Select **STEP PIN** and set it as active high or active low.
- d. Select **DIR PIN** and change it to active low or active high.
- e. Select **STEPS/MM** and edit its value.
- f. Select **MM/MIN** (velocity) and edit the value.

- g. Select **(MM/sec) ^2** and edit the value.
- h. Confirm or cancel the modifications made in settings and exit to previous menu.
- i. Modify the axes settings for rest 3 axes.

FOR 4TH AXIS MAPPING:



- a. Select and enter in **4th AXIS MAPPING**.
- b. Any one axis can be selected here which user wants to be the rotational axis. Confirm or cancel the settings made and Exit to previous menu.

FOR Z INHIBIT:



- a. Select **Z-INHIBIT** function and enter in it.
- b. Select enable or disable and exit to previous menu.
- c. Confirm or cancel the settings made and Exit to previous menu.

- 3. Save the confirmed settings in required profile or exit without saving.

IO SETTING

IO setting provides an interface between CNC Controller and CNC machine by making settings for limit and home inputs and for the spindle outputs. These settings make the settings for home and limit switches and also for the buffered Misc outputs. The functions under IO setting menu are discussed below:

SPINDLE SETTING

Spindle holds the cutting tool. Spindle can rotate in CW/CCW direction with the velocity set in Gcode file. The axis of spindle is kept parallel to the Z-axis and is coincident with Z-axis; except in machines where the spindle can be rotated by rotational axis. The functions for spindle settings decides Spindle Enable/Disable, Step frequency for spindle drive and state of step pulse, the maximum revolution per minute at which spindle needs to spin. These are given below:

- **SPINDLE (ENABLE/DISABLE):** The Spindle Enable/Disable function allows operator to enable or disable the spindle by giving respective signal to the spindle drive.

Spindle rotates if the spindle drive receives enable signal and spindle is made ON in Gcode file. If spindle is disabled in IO setting and spindle is made ON in G-code file, then spindle does not work. “**Spindle enable/disable**” is the relay output from CNC Control Board and the relay remains off if spindle is disabled.

- **STEP PIN HIGH/LOW:** It sets the state of the step pulse driven to spindle drive. This pin, if selected active high, drives an active high pulse and if active low, drives an active low signal to spindle drive. This output pin get disabled if Spindle is set disabled.
- **MAX RPM:** The Max RPM refers to maximum number of revolutions per minute the spindle rotates. Spindle can be set to rotate at or below the maximum velocity set but not beyond Max RPM.
- **MIN RPM:** The Min RPM refers to minimum number of revolutions per minute the spindle rotates. Spindle can be set to rotate at or above the minimum velocity set but not below Min RPM.
- **DWELL SEC:** Some delay is set before the spindle starts to rotate and at the time when the spindle stops after rotation. This delay is dwell time. Spindle remains idle for this duration of time. It is a user programmable and is in seconds.
- **STEP FREQ HZ:** It decides the frequency of step pulse at step pin output on CNC controller board. By default, it is 1 KHz and maximum is defined by following formula and is user programmable.

$$\text{Step freq (max)} = (1\text{MHz}) / (2 * \text{Pulse width})$$

Method to access IO Setting:

1. Select and enter in **IO SETTING** function in **MAIN MENU**.

Method to access Spindle Setting:



- a. Select **SPINDLE SETTING** and enter.
- b. Set the Spindle “**Enable or Disable**”.
- c. Select **STEP PIN** and make it High or Low.
- d. Select **MAX RPM** and edit its value.
- e. Select **MIN RPM** and edit its value.
- f. Select **DWELL sec** and modify its value.

- g. Select **STEP FREQ** and modify the values.
- h. Confirm or cancel the settings made and Exit to previous menu.

HOMING SETTING

HOMING SETTING provides settings for an advanced homing cycle with speed configurable options to return to home position. Homing ensures the safety of the machine. Homing can be done in the conditions where the motion got skipped for any axis. In such case, machine should be placed somewhere at the known place. For this purpose, home switches should be installed for all axes on every machine.

For homing, CNC controller follows a sequence of cycles before reaching to machine zero. In first cycle, Z-axis moves upward till finding and triggering Z-axis home switch. In second cycle, rest three axis moves together in independent way and trigger their home switches. In both cycles, they move at set “**homing seek rate**”. In proximity of all home switches, machine leaves and approaches switches in two cycles to precisely “**home**” at the machine zero location at set “**homing feed rate**”. In the last cycle machine moves to a position determined by set “**pull off distance**” away from machine zero location for all enabled axes in coordinated way at homing seek rate. Finally machine stops, where DRO shows the current known location.

- **SEEK (MM/MIN):** The rate at which an axis traverses from its current position to its Home switch on pressing Key 7 long is called homing seek rate. It is user programmable and range for seek rate is 1-9999mm/min.
- **FEED (MM/MIN):** The rate at which home switches are triggered and released (twice) to precisely home at the machine zero location is called homing feed rate. It is also user programmable and maximum range for feed rate is 9999 mm/min.
- **PULL OFF MM:** The machine moves to a position away from machine zero location for all enabled axes in coordinated way at seek rate after homing. The distance of this position from machine zero is set by “Pull off” function. Pull off function is programmable and the maximum range is size of machine.
- **SAFE Z- HEIGHT:** This function ensures the safety of the machine for which Home switch for Z-axis either has been disabled or not installed. When key 7 is pressed long,

Z-axis uplifts to a height set in this function and then homing of other axes takes place. By this, the tool installed at Z-axis remains safe. The maximum height can be the height of the Z-axis of the machine.

- **DEBOUNCE mS:** This is the delay settled for debouncing of the Home Switch, when pressed. By default, it is 100 ms. However; it is user programmable. Maximum value of debounce delay is 1 second. The units for debounce delay are ms.

Here is the method to access HOMING SETTING:



- Select and enter in **HOMING SETTING**.
- Select **SEEK mm/min** and edit its value.
- Select **FEED mm/min** and modify its value.
- Select **PULL OFF mm** and modify its value.
- Select **SAFE Z-Height** and edit its value.
- Select **DEBOUNCE us** and edit its value.
- Confirm or cancel the settings made and Exit to previous menu.

HOME SWITCH SETTING

Home Switch Setting allows the operator to enable or disable the home switches for all axes. Disable the home switches for axes for which switches are not installed. An input signal is driven from the home switch to its axis Home-In terminal on CNC control board when it got triggered.

When the machine is in Run/Jog mode and Home switch for an axis got triggered, an input is generated to the axis Home-IN pin and motion of an entire axes system ceases. A message of alarm for **Hard Home/Limit** is displayed on the screen. Press Ok key to continue. None of the jog key works and entire motion system stops in this condition. Press the left navigation key long to enter in manual override mode and to release the switch.

All the keys are disabled except the jog and navigation keys in manual override mode. Jog the machine reverse using jog keys to release the switch and set the machine at required position. At any situation, if the limit/ home switch got triggered, hard limit/home is not checked and user need to jog the machine reverse. Press left navigation key long again to exit from manual override mode.

For all axes, it is required to make some settings for home switches. These are:

- **ENABLE:** sets the home switch enabled for the axis. It makes the homing of the axis when key 7 is pressed long.
- **HIGH/ LOW:** Active high sets the home switch to generate an active low pulse to axis Home-In terminal on the board when the switch got triggered. Active low sets the home switch to generate an active high pulse when the switch got triggered.
- **DISABLE:** sets Home switch disabled for the axis. Homing of the axis got ceased when key 7 is pressed long. However the Z-axis if set disabled, moves to Z-safe height (discussed in Homing setting function) when homing command is given.

Access the Home switch settings as following:



- a. Select and enter in **Home Switch Setting** function in **IO Setting Menu**.
- b. Enable or disable the home switch for the axis as per requirement. Set the switch high or low if enabled. Make the modifications for all the axes.
- c. Confirm or cancel the settings made and Exit to previous menu.

LIMIT SETTING

Limit Setting allows the operator to enable/disable the limit switches for all axes. Usually, in all the machines, each axis consists of a pair of hard limit switches installed in its negative and positive direction. Axis Limit-IN terminal at the CNC Control board shares the inputs from both switches. There is no Limit-In input terminal for Y-Slave.

When the machine is in Run/Jog mode and Limit switch for an axis got triggered, an input is generated to the axis Limit-IN pin and motion of an entire axes system ceases. An alarm message of **Hard Home/Limit** is displayed on the screen. Press Ok key to continue. None of the jog key works and entire motion system stops in this condition. Press the left navigation key long to enter in manual override mode and to release the switch.

All the keys are disabled except the jog and navigation keys in manual override mode. Jog the machine reverse using jog keys to release the switch and set the machine at required position. At any situation, if the limit/ home switch got triggered, hard

limit/home is not checked and user need to jog the machine reverse. Press left navigation key long again to exit from manual override mode.

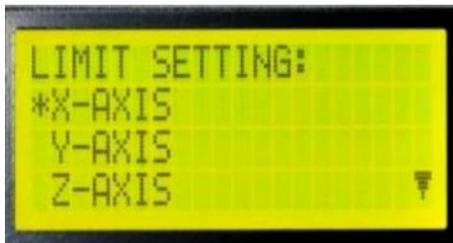
Disable the Limit switches for the axes which don't have the home switches. For all the three axes, it is required to make the following settings for Limit switches.

- **HARD LIMIT ENABLE/DISABLE:** Hard limit if enabled generates the alarm message when hard limit switch got triggered. Hard limit if disabled does not generate any message when limit switch got triggered however the input pulse is driven to the axis Limit-IN pin of CNC control board when the switch got triggered and the machine get blocked in the same way.

- **HARD LIMIT SWITCH (ACTIVE HIGH):** Active high sets the Limit switch to generate an active low pulse to axis Limit-In terminal on the CNC control board when the switch got triggered.

- **HARD LIMIT SWITCH (ACTIVE LOW):** Active low sets the Limit switch to generate an active high pulse to axis Limit-In terminal on the CNC control board when the switch got triggered.

Steps for Limit Setting:



- a. Select and enter in the **LIMIT SETTING** in **IO SETTING**.
- b. Select the required axis. Select **X-axis** and enter.
- c. Select **HARD LIMIT** and enable or disable it.
- d. Select the **SWITCH** and make it high or low.
- e. Confirm or cancel the settings made and Exit to previous menu.

MISC OUTPUTS

Misc Outputs are miscellaneous outputs and 22 in number. Misc outputs 1-6 are buffered outputs and rest 16 are open collector outputs. User can make them ON/OFF in Gcode only. All these outputs can be made OFF collectively but are made ON individually in Gcode file. Then open collector outputs are active high outputs whereas the buffered outputs can be set as active high/active low outputs. Buffered outputs can be enabled/ disabled in MISC Outputs function. Buffered output if disabled don't give

any output on the output pin even made ON in Gcode file. Misc output can be set in any of the following state:

- **EN/HI:** If output is set enable high, the output pin generates an active high pulse if output is made ON in Gcode.
- **EN/LOW:** If output is set enable low, the output pin generates an active low pulse if output is made ON in Gcode.
- **DIS/HI:** If output is set Disable high, the output pin does not generate any output pulse even made ON in Gcode file. However, the pin state is high in this condition.
- **DIS/LOW:** If output is set Disable low, the output pin does not generate any output pulse even made ON in Gcode file. However, the pin state is low in this condition.

Method to access Misc outputs:



- a. Select and enter in **MISC OUTPUTS**.
- b. Enter in Misc settings and select an output and change relevant mode for output signal.
- c. Do same for number of outputs required.
- d. Confirm or cancel the settings made and Exit to previous menu.

2. Save the confirmed settings in required profile or exit without saving.

TOOL SETTINGS

Tool settings are made to set the tool zero related parameters, measure the offset of tools automatically and edit the offsets in tool table, if measured manually. A provision for 6 tools change is given.

In order to calculate offset of tools, it is required to perform tool zero process for each tool. This process can be initiated by pressing Z-Zero (key '0') when the system is in idle mode.

Upon initiating the tool zero process, tool starts moving downward along Z-axis with set "**homing feed rate**" and tries to find out sensor metal plate kept on the top of work piece. As it touches the top of sensor plate, controller gets a signal through its tool

zero input pin and it stops any further movement instantly to avoid any damage to the tool. At the same time, controller moves the tool upward to a set **Back off Distance**.

At the moment, tool touches sensor metal plate, controller sets Z=0 at the top of work piece and updates current working co-ordinate system as well as DRO (digital readout); for the reference tool (tool0). For doing this **height of the sensor plate** must be known to the controller. It must be measured and set accurately by the user. In case of other tools i.e. tool no. 1 to tool no. 6, same process takes place to calculate their offset automatically, but there are no changes to the current working co-ordinates. It should be noted that tool offsets are calculated with respect to the reference tool (tool0) and it is required to perform tool zero process for reference tool first. Tool offset for tool0 is zero always and can't be modified even manually. All these highlighted parameters can be set in menu. These are discussed below:

TOOL ZERO SETTINGS

There are two parameters must be set before performing the tool zero process. Value of both the parameters must be set within machine's Z-axis limit. These are:

- **SENSOR HT MM:** Height of sensor metal plate can be set in mm.
- **BACKOFF MM:** This is the back off distance that machine moves back after touching the sensor plate at rapid rate. This is the value that DRO shows when machine stops after this move.

Steps to access Tool settings>>Tool zero settings:



- a. Select and enter in Tool Settings.
- b. Enter in **Tool Zero Settings**.
- c. Edit "**Sensor HTmm**" value.
- d. Edit the "**Back off mm**".
- e. Confirm or cancel the settings made and Exit to previous menu.

TOOL OFFSET: Tool offset is a stored value that compensates for variations in tool length. Each tool requires an offset, which is measured from spindle position on Z-axis. For example, suppose that Tool No. 0 is taken as the reference tool and it is set

for tool zero position. Now replace the Tool No. 0 with Tool No. 1 which is 2mm longer than Tool No. 0. Now offset value is 2mm. Tool No. 1 moves 2 mm less distance than Tool No. 0. This offset value can be calculated automatically by **AUTO MEASURE OFFSET** or manually by **MANUAL EDIT OFFSET**.

AUTO MEASURE OFFSET

AUTO MEASURE OFFSET function calculates the offset i.e. the difference in reference tool length and selected tool length automatically. User need to make the Tool Zero Settings for reference tool first and then offset for selected tool is automatically calculated.

Steps to access AUTO MEASURE OFFSET:

- a. Select **AUTO MEASURE OFFSET**.
- b. Select the tool required.
- c. Confirm or cancel the settings made and Exit to previous menu.

MANUAL EDIT OFFSET

In **MANUAL EDIT OFFSET** function, operator has to calculate the offset function between tools manually and input that value in this function. Firstly, tool zero for the reference tool is set and then offset value for selected tool is calculated by user. Controller not only saves this value but also retains the previous offset value stored for the tools.

Steps to access MANUAL EDIT OFFSET:



- a. Enter in **MANUAL EDIT OFFSET** function in tool settings.
- b. Select the tool required and enter the offset calculated. Change the sign of value from +ve /-ve by long pressing the **KEY '0'**.
- c. Confirm or cancel the settings made and Exit to previous menu.

Save the confirmed settings in required profile or exit without saving.

DATA FILE

Data file function allows loading the General settings, Axes settings, IO settings and Tool settings chose by the user to all the profiles including the active profile. These settings are to be saved in a file named as **Settings.tcs** which the user creates in the USB Flash drive and can be loaded to all the profiles by “load from file” function discussed further.

It should be noted that file name **Settings.tcs** should not be changed. While reading the settings for the profiles, this file must be present on the USB Flash drive.

Besides reading settings from Settings.tcs with “load from file”, a menu for loading the default settings to all profiles is given in data file function. All default values are stored in on-chip ROM of every controller at the time of shipment and can't be altered. Here is shown the method to perform these operations:

LOAD FROM FILE: Load from file menu allows loading settings to the respective profiles from user made file named “Settings.tcs” in the thumb drive.

Method to access data file >> load from file:

- a. Select and enter in “Data File” function in Main Menu.
- b. Enter in “Load from File” menu. A message of “Settings loaded to all profiles from settings.tcs” appears. Press any key to exit.

LOAD FROM DEFAULT: Load from default allows saving the default values of settings into all the profiles. All the default values loads from controller itself.

Method to access load from default:



- a. Enter in “Load from Default”.
- b. A message of “default settings loaded to all files” appears. Press any key to exit.

ADVANCED SETTINGS: User can't access this function. Only manufacturer can handle these settings.

DEFAULT PROFILES VALUES

1. GENERAL SETTINGS

- **Step pulse width:** 4us
- **Path mode:** Continuous mode
- **Max Deviation:** 5
- **Help:** Keypad function and About

2. AXIS SETTINGS:

- **Drive:** Axes X, Y and Z enabled. Axis A disabled
- **Step pin:** Active High
- **Dir pin:** Active High
- **Steps/mm:** 250 steps (X, Y and Z)
- **Velocity (mm/min):** 10000 mm/min
- **Acceleration:** 100 mm/sec².
- **4th Axis mapping:** Along A
- **Z inhibit:** Disable

3. IO SETTINGS:

SPINDLE SETTINGS:

- **Spindle:** Enable
- **Step pin:** Active High
- **MAX RPM:** 20000
- **MIN RPM:** 6000
- **Dwell time:** 4 seconds
- **Step frequency:** 1 KHz

HOME SETTINGS:

- **Seek rate:** 750.00mm/min
- **Feed rate:** 100.00mm/min

- **Pull off:** 1 mm
- **Safe Z Height:** 10mm
- **Debounce delay:** 100mS

HOME SWITCH SETTINGS:

- **Home switch Enable (all axes):** Active High

LIMIT SETTINGS:

- **Hard Limit Switch (all axes):** Active High

MISC OUTPUTS: All disable high

4. TOOL ZERO SETTINGS:

TOOL ZERO SETTINGS:

- **Sensor Height:** 2mm
- **Back off:** 20mm

KEY FUNCTIONS: NAVIGATION KEYS

Functions of keys in Various Modes are shown in table 1.1 and 1.2. This section describes the function of navigation and numeric keys in detail.

KEY LEFT: WCS TOGGLE SCREEN

By default, DRO shows all the coordinates with respect to G54 working coordinate system, which can be identified by the number shown at the left most column of it.

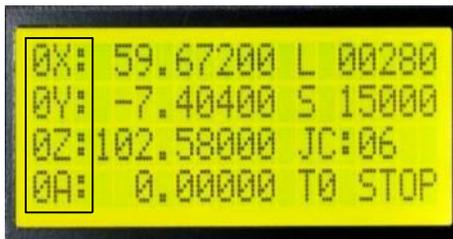
In other cases,

0 – Absolute coordinate

1 – G54, 2 – G55, 3 - G56, 4 - G57, 5 - G58, 6 – 59

This key enables user to view coordinates in absolute mode in all the three modes. It shows back the coordinates with respect to current working coordinate system selected in G-code file when pressed again.

ABSOLUTE COORDINATES



G54 COORDINATES



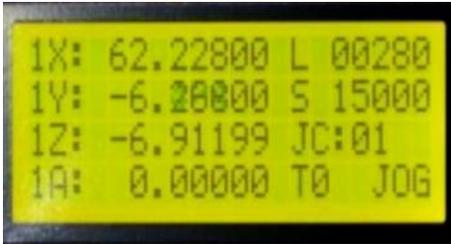
KEY LEFT LONG PRESS: MANUAL OVERRIDE MODE

Manual override mode leads the user to release the home switch or the limit switch if triggered when machine is in hold or in idle mode. Only jog keys and navigation keys work in manual override mode and rest of the keypad fails. Manual override mode is shown by * symbol along with jog mode displayed on LCD screen.

Manual override has been also discussed in Home switch settings function of IO settings. After entering in manual override mode, jog the machine to the reverse direction of switch triggering and release the switch. Then jog the machine at required position. However, if at any position during jogging, limit or the home switch got triggered again; “hard limit/ home” is not checked and user needs to jog the machine in reverse direction. Press long the same key to exit from the manual override mode.

KEY RIGHT: JOG MODE MENU

Jogging is used to travel the CNC machine carriage such as CNC machine axis movement. Jogging allows for manual operation of tool by using Jog keys. (The functions of jog keys have been discussed under jog keys panel of **Hand held pendant** section). Jog mode can be selected in **run**, **idle** and **Hold mode**. However, **jogging** can be done in **idle** and in **hold mode** only.



DRO shows the adjacent screen during machine jogging. CNC Controller provides the three modes for jogging. These are discussed as below:

a. CONTINUOUS MODE: This mode allows continuous jogging of the axis in required direction by pressing respective jog key. A jog factor can be set in range of 1-99 which determines the rate of jogging of machine. User can change the jog factor by up-down navigation keys. Jogging velocity for each axis can be calculated by following formula. Here is shown the calculation of jog velocity for X-axis:

$$\text{Jogging velocity (Axis)} = (\text{Max Velocity for Axis} / 100) \times \text{Jog Factor}$$

The maximum velocity is the MM/MIN set for X-axis in **Axis Settings** menu. Calculate the jogging velocity for each axis in the same way.

b. STEP MODE: In order to precisely place the machine at required location, jog the machine in step jog mode. It allows users to select a step size out of the four **pre-programmed** step sizes. Select the step-size using up-down navigation keys and step-sizes available are 1mm, 0.1mm, 0.01mm and 0.001 mm.

c. FAST MODE: In order to jog the machine at required location comparatively faster than the above two modes, jog the machine in fast jog mode. The default value of jog factor for Jog fast mode is 40 and user can increase or decrease it in same manner as for the continuous jog mode.

STEPS FOR JOGGING:

- Use **Right navigation key** to select the Jog mode in **idle/run** or in **hold mode**.

- Put machine in **idle or hold mode**. Press **Jog keys** of required axis to jog machine.

KEY RIGHT LONG PRESS: JOG OFFLINE MODE

In jog offline mode, absolute coordinates got changed and whole WCS shifts to a new position. Due to jogging, position of the machine changes and the coordinates for current position of machine got changed w.r.t. absolute coordinates. However, changed coordinates aren't reflected by screen after exiting from this mode. LCD shows same coordinated as before entering in jog offline mode for machine current position. The absolute coordinates for WCS got updated to a position such that the coordinates appearing on screen are w.r.t. new absolute coordinates. For example:

Press right navigation key long to enter in jog offline mode.

Jog the machine to required position and exit from jog offline mode.

The coordinates on LCD remains (10, 10, 10). But the absolute coordinates for WCS 1 got changed such that machine position is (10, 10, 10) w.r.t. absolute coordinates updated.

KEY UP: JOG FACTOR INCREMENT/STEP SIZE CHANGE

If Continuous/Fast jog mode is selected, press this key to increase the jog factor by 1 in **Idle, Hold** and **Run mode**. Long pressing this key increases the jog factor by 10. Press this key to change the step-size if Step jog mode is selected.

KEY DOWN: JOG FACTOR DECREMENT/ STEP SIZE CHANGE

If Continuous/Fast jog mode is selected, press this key to decrease the jog factor by 1 in **Idle, Hold** and **Run mode**. Long pressing this key decreases the jog factor by 10. Press this key to change the step-size if Step jog mode is selected.

ESC KEY: ABOUT SCREEN FLASH

Pressing **Esc** in **idle mode** shows the about screen. "About" screen shows product part number and version of the controller. After few seconds, the screen shows the current coordinates in idle mode. Pressing **Esc** in **Hold** and **Run mode stops** the machine.

NUMERIC KEYS

COORDINATE ZERO KEYS

These keys are used to update the current working coordinate system (WCS) (G54-G59) and sets machine at zero position for the selected axis in current WCS. Zeroing all the axes sets new origin of current working coordinate system in **Idle** and **Hold mode**. These keys do not have any effect while the machine is in run mode. For example:

To change current working coordinates, Suppose:

Current working coordinate system w.r.t machine zero coordinates = (100, 100,100).

Machine current position w.r.t. current working coordinate system = (110, 0, 0).

Then, on pressing Key 1 i.e. X coordinate zero key: Current working coordinates w.r.t machine zero = (110, 100, 100).

- **KEY 1: X-COORDINATE ZERO:** Press **KEY 1** for X coordinate zero.
- **KEY 2: Y-COORDINATE ZERO:** Press **KEY 2** for Y coordinate zero.
- **KEY 3: Z-COORDINATE ZERO:** Press **KEY 3** for Z coordinate zero.
- **KEY 4: 4 COORDINATE ZERO:** Press **KEY 4** for 4th coordinate zero.

KEY 5: G-CODE FILE SELECT



Key 5 works for **File select** in **idle mode**. Upon pressing this key, it lists maximum of fifty files available on USB drive if USB device is properly mounted. Otherwise, LCD shows the message of unavailability of USB Flash drive.

It is advised to keep file name of maximum **8 lettered name** and **3 lettered extension**. Otherwise, it does not display full file name properly.

Key 5 works for **Spindle On/Off** in **Hold mode**. On pressing Key 5 in hold mode, spindle stops if it was rotating and rotates if it was stopped previously.

A message "waiting for dwell period" appears. After the dwell time, coordinate screen appears on the display. Press key 5 again to resume to previous state. If the

coordinates are not changed, machine resumes from same position on pressing key 8 for resumption. However if coordinates have been changed, a different procedure for resumption is to be employed. The complete process for resumption is discussed further in this manual. Spindle turns off if Esc button is pressed.

KEY 5 LONG PRESS: GCODE FILE CHECK

Pressing **KEY 5 long** leads the controller to check G-code file selected from pen drive in **idle mode** for the safety of CNC machine and its proper motion. Controller reads the complete G-code file and detects the errors and shows the respective error messages. Entire G-code is checked line by line.



Press Ok to allow the controller to check the next line in G-code file if an error is found. Press Esc to exit. If there is any command of the tool change in the Gcode during the execution of Gcode file, adjacent screen appears.

There can be miscellaneous errors in the Gcode lines for example **Bad number format, unsupported statement, Expected command letter** etc.

Press down navigation key to continue else press Esc to exit.

FEED RATE OVERRIDE AND SPINDLE SPEED OVERRIDE

Feed rate override function allows the operator to adjust the feed rate of the tool during operation. The feed rate override functions as a percentage of programmed value of feed rate “F” in Gcode in the range of minimum to maximum velocity whereas the minimum value is the feed rate set in Gcode and maximum value for velocity is the minimum value of MM/MIN function among all axes set by user in Axes settings menu.

Spindle speed override function allows the operator to adjust the speed of spindle during operation. The override functions as a percentage of the programmed value of spindle speed “S” in Gcode file in the range of MIN RPM to MAX RPM function set by user in spindle setting menu.

KEY 6: OVERRIDE SCREEN

When the machine is in either of 3 modes, pressing key 6 shows the override screen on the LCD. User can change the percentage of feed rate override using Z jog keys and percentage of spindle speed override using the 4th jog keys. Press 'Z+' key to increase feed rate override and 'Z-' key to decrease the feed rate override. Press 'Y+' to reset the feed rate. Press '4+' Key to increase the spindle speed override and '4-' key to decrease the spindle speed override. Press 'Y+' to reset the Spindle speed. Press Ok key to exit from override screen.

However, minimum value for feed override is 10% of the programmed feed rate and maximum value is 200% of the programmed feed rate. However, maximum achievable feed rate for the tool can't reach beyond MM/MIN.

Similarly, maximum value for spindle speed override is 10% of the spindle speed and maximum value of spindle speed override is 200% of the programmed spindle speed in G code file. However, maximum reachable spindle can't cross the MAX RPM set in spindle setting function of IO settings.

KEY 7: GO TO ZERO

Press key 7 to make the entire axes system to go to Zero position set for that working coordinate system in **idle mode**. When key 7 is pressed, **Go to Zero** action takes place in following steps:

Step 1: When key 7 is pressed, Z-axis moves to Z-safe height. However, if Z-axis is already at or at height more than Z-safe height, this step gets omitted and next step is accomplished directly.

Step 2: Coordinated motion of X-axis, Y-axis and 4th-axis to their respective zero-position w.r.t. WCS takes place.

Step 3: After step 2, the motion of Z-axis to its respective zero position takes place.



Tip: User can increment or decrement the jog factor in continuous or jog fast mode by 10 rather than 1 by long pressing the jog increment/decrement key.

KEY 7 LONG PRESS: HOMING

Homing is the process of finding home switches and bringing machine at a known location. Although this controller is capable of restoring location of machine even after power shut down and shows machine's current location all the time, there can be some situations during machine operations when its location can't be identified accurately. For this purpose, home switches should be installed on every machine for desired axes. The homing procedure has been discussed in homing setting section of IO setting menu. Homing can be done in **idle mode**.

In case, if homing is not required for any particular axis, disable its home switch.

By default, axis A has been disabled. Following message appears when key 7 is pressed long.



Above message appears for the disabled axis.

Upon pressing Ok, homing process starts and message "**Going Home**" appears.

KEY 8: FEED HOLD/ RESUME

When the machine is in **idle mode**, pressing key 8 shows a screen in which the line number is needed to be input by operator from which he wants the machine to start execution of Gcode file. All this process can be carried out in following steps:

Step1: Press key 8 in idle mode and enter the line number. Press Ok key after entering the line number.

Step2: Pressing Ok key results in homing of machine and then execution of Gcode file from input line number starts.

While machine is in **Run mode**, pressing this key **stops/pauses** the machine with set deceleration and keeps it in **Hold mode**. In this state, spindle can be switched **ON/OFF** using key 5.

In **hold mode**, this key can be used to resume execution of G-code file, if all conditions for resumption satisfy.

Condition for resumption:

1. USB drive must be mounted.
2. File with same name must exist in USB drive.
3. Contents of Gcode file should not be changed.

Operations during resumption in Hold Mode:

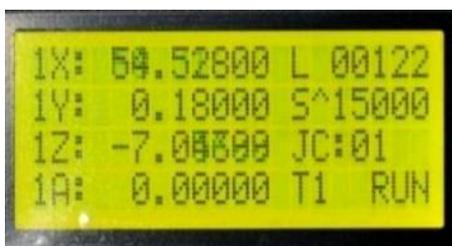
For hold mode, machine starts moving from current point as normal execution, if there is no change in position, when resumed. On the other hand, it is possible to change machine location by jogging. In this situation, machine follows following steps in order to attain a known location on specified tool path by G-codes. This location is the target location specified in last line of G-code file.

Step1: When key 8 is pressed in Hold mode, Z-axis moves to Z-safe height set in Home settings function of IO settings menu. However, if Z-axis is already at or a height greater than Z-safe height, step number 1 is omitted and next step is executed directly.

Step2: Coordinated motion of X-axis, Y-axis and 4th-axis to the target location takes place.

Step 3: After step 2, the motion of Z-axis to its respective target location takes place. Then continue executing next lines in file.

KEY 9: FILE START



In **idle mode**, pressing this key starts execution of selected G-code file. Appropriate message appears upon non-existence of file or USB drive. Screen show the DRO status in Run mode. Here T1 indicates tool number 1 and ^ indicates spindle's ON state, where others having their usual meaning.

KEY 0: TOOL 0

Pressing this key starts tool zero process when machine is in **idle mode**. As the screen shows, this process can be terminated at any time by pressing 'Esc' key.



Pressing key 0 sets selected tool to tool zero. User can press Esc to cancel tool zero process and stop the tool at that position. Tool doesn't touch the sensor plate in this condition. Adjacent message appears when tool 0 is pressed.

Press key 0 again to continue tool 0 process else press Esc key to cancel.

G-CODE ALLOWED TO USE †

- G0:** Rapid Positioning
- G1:** Linear Interpolation
- G2:** Circular/Helical Interpolation (Clockwise)
- G3:** Circular/Helical Interpolation (Counterclockwise)
- G4:** Dwell
- G10:** Coordinate System Origin Setting
- G17:** XY-Plane Selection
- G18:** XZ-Plane Selection
- G19:** YZ-Plane Selection
- G20:** Inch System Selection
- G21:** Millimeter System Selection
- G28:** Return to Home
- G30:** Return to Secondary Home
- G53:** Motion in Machine Coordinate System
- G54:** Use Preset Work Coordinate System 1
- G55:** Use Preset Work Coordinate System 2
- G56:** Use Preset Work Coordinate System 3
- G57:** Use Preset Work Coordinate System 4
- G58:** Use Preset Work Coordinate System 5
- G59:** Use Preset Work Coordinate System 6

- G61:** Exact Stop mode
- G64:** Continuous mode
- G70:** Enable Block mode
- G71:** Disable Block mode
- G80:** Cancel Motion Mode
- G90:** Absolute Distance Mode
- G91:** Incremental Distance Mode
- G92:** 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
- G93:** Inverse Time Feed Rate Mode
- G94:** Units per Minute Feed Rate Mode

M CODES ALLOWED TO USE †

- M0:** program stop
- M1:** optional program stop
- M2:** program end
- M3:** turn spindle clockwise
- M4:** turn spindle counterclockwise
- M5:** stop spindle turning
- M6:** Tool Change
- M7:** Mist coolant on
- M8:** flood coolant on
- M9:** mist and flood coolant off
- M30:** program end, pallet shuttle, and reset
- M54.1- M54.22:** Digital output ON.
- M55.1-M55.22:** Digital output OFF.
- M56:** All Digital output OFF.
- M57, M58:** Analog output (0-10V).
- M75:** for comments transfer through serial port (in double quotes).

† Not all these codes are based on NIST RS274/NGC Interpreter.

Format of a G-code Line

Lines of G-code can be collected in a file to make a program or run a block at a time. Each line can include commands to a machining center to do several different things.

G1X5.991Y21.259Z-0.500

For CNC controller, a typical line of code consists of one or more "words". An example for the code line is shown above. A permissible line of input code consists of the following, in order, with the restriction of max 50 character allowed on a line.

1. Number of words, parameter settings and comments.
2. At the end of line marker (carriage return or line feed or both). Any other inputs not explicitly allowed are illegal.
3. Spaces and tabs are allowed anywhere on a line of code and doesn't change the meaning of the line, inside except inside comments. This makes some strange-looking input legal.
4. Blank lines are allowed in the inputs. They are to be ignored. Inputs are case insensitive, except in comments i.e. any letter outside a comment can be in upper or lower case without changing the meaning of line.

WORD:

A word is a letter followed by a real value. A word consists of a letter followed by a number (or something that evaluates a number). A word may either give a command or provide an arrangement to a command.

NUMBER:

The following rules are used for (explicit) numbers. In these rules a digit is a single character between 0 and 9.

- A number consists of an optional plus or minus sign, followed by zero to many digits, followed, possibly, by one decimal point, followed by zero to many digits provided there is at least one digit somewhere in the number.
- There are two kinds of numbers: integers and decimals.
- Numbers can have any number of digits, subject to the limitation on line length. Only about seven significant figures can be retained.
- A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required. A number written with initial or trailing zeros have the same value when it is read as if the extra zeros were not there.

COMMENTS AND MESSAGES:

Printable characters and white space inside quotation is a comment. A left quotation always starts a comment. The comment ends at the first right quotation found thereafter. Once a left quotation is placed on a line, a matching right quotation must appear before the end of the line. Comments cannot be nested; it is an error if a left quotation is found after the start of a comment and before the end of the comment. Here is an example of a line containing a comment: G80 M5 'stop motion'. Comments do not cause a machining center to do anything.

A comment contains a message if "MSG" appears after the left quotation and before any other printing characters. Variants of "MSG" which include white space and lower case characters are allowed. The rest of the characters before the right quotation are considered to be a message. Messages should be displayed on the message display device. Comments not containing messages need not be displayed there. Comment can be written in any of quotation either in single quotes or in double quotes. For example:

M75 'tiny controls'

M75 "tiny controls"

TROUBLE SHOOTING

1. What happens if there is a power cut in between the operation of CNC machine?

Ans: Controller can resume the operation on any power interruption saving the current line of G-code and 'homing' the machine on power up from last location.

2. What happens on pressing the Emergency button?

Ans: When this button is pressed, the motion of axes and spindle stops instantly however it does not cut down the power to control board or CNC machine. All the control outputs are disabled. Lock out the emergency button when the machine is needed to be resumed.

3. Communication trouble shooting:

Serial Communication:

Connection between CNC controller and handheld pendant is made through serial cable. Occasionally, operators experience difficulties when the pendant or controller board loses the connection, faces problems to access connections etc. If control works for some time but experiences random problems, then machine might be experiencing some sort of communication failures.

Note that the serial port of the board could be damaged by a bad electrical storm or any comparable natural occurrence and this damage is not always fixed the same way for all machines.

Some common remedies to remove the communication problems are:

Cable testing: Cable testing can be started with unplugging the serial cable from the side of control board and plug it back.

- If the above process doesn't work then it may happen that connectors have been damaged or chip for Rs-232 have been failed and needed to be replaced.
- The internal configuration of cable has been damaged.
- If the length of wire is greater than 3m, use line amplifiers.

USB Port:

The CNC controller's USB port connection uses a common pen drive through which the G-code file is input to controller. There may be many issues due to which controller experiences random problems:

- **Pen Drive not found:** Insert the Pen drive and try again.
- **File doesn't exist:** Selected file doesn't appear on pen drive. Check whether pen drive contains the file or not.
- **Wrong format of file:** CNC controller supports the 8.3 formats (8 lettered filename +3 lettered extensions). Long filenames are not supported.

4. What happens if the pen drive is removed (by mistake) in between the program?

Ans: Ans: When pen drive get removed in between the code execution, reconnect the pen drive, resume the machine from the current position to Home position by pressing key 8. From home position, X-axis, Y-axis and Y-Slave move to the coordinate position at which emergency button was pressed. All the outputs which were enabled previously get enabled again and a message “**If Spindle On Press Ok**” appears. If the spindle is off, then press Esc key and cancel the process. If the spindle is on, press Ok key. Z axis moves to the coordinate position as before the emergency button press. Here, the controller starts to execute the code one line before in G-code at which emergency button was pressed and runs the file in same manner.

Same process happens if the emergency button is pressed.

5. What happens if the connection between the pendant and controller is lost in between the execution of G-code file?

Ans: If connection between the pendant and controller is lost, the machine continues to run according to G-code file, Press the emergency button on the control board (to be attached by user) immediately else switch off the CNC machine.

GLOSSARY

Axis motor: A motor that causes motion in a particular axis.

Absolute coordinate system: In an absolute coordinate system, all the references are made to the origin of coordinate system. All the commands of motion are defined by the absolute coordinate referred to origin.

Block mode: In the block mode, the machine motion stops at the execution of one programmed line and execute the next line on the trigger of an external input called block mode input.

Circular interpolation: A translation of linear axis positions into curved tool motions. Circular interpolation requires an endpoint, a feed rate, a center, a radius, and a direction of movement.

Coolant: A machining center has components to provide mist or flood coolant. Coolant output is taken out from the control board and it activates the coolant of the machine.

CNC: Computer numerical control.

Dwell: A programmed time delay.

Emergency Stop: The red color button located on the top of the handheld controller, used to stop the machine immediately by disabling the axes and spindle motor and other control outputs.

Feed rate: The rate at which the cutting tool is advanced into the workpiece.

Gcode: A program code that determines the type of operation performed on the machine. G codes used for CNC motion controller are based on NIST RS274/NGC Interpreter-v3 with some exceptions/extensions.

Home: The origin of the machine coordinate system. The unchangeable home position is also known as the machine zero point.

Homing: The process of sending the machine to its home or machine zero position.

Hand held pendant: A handheld pendant is an interface device between CNC machine and the CNC controller board.

Jog Control: Manual movement modes for the machine axes, using very small or large predefined movement.

Keypad: The part of the handheld pendant with a grid of keys.

Linear interpolation: A translation of linear axis positions into straight vertical, horizontal, or diagonal tool movements. Linear interpolation requires an endpoint and a feed rate.

Machine zero point: A fixed zero reference point set by the machine manufacturer. The machine zero is used to define the coordinate based grid system of the CNC machine. All machining coordinate originate from this point.

Mcodes: Miscellaneous modes function in a CNC program to indicate an auxiliary function (i.e. coolant on, tool change etc).

Offset: The offsets are used to shift parts of the three dimensional coordinate based grid systems, used by the CNC machine.

Pause: The key on the keypad that put the machine into pause mode where action is momentarily stopped until the machine is resumed back.

Resume: The key put the machine back into the run state from the pause state.

Relative coordinate system: is type of control always uses a reference to the preceding point in a sequence of points. The disadvantages of this system are that if an error occurs, it accumulates in further motion.

Rotational direction/axis: A tool data specification that accounts for the direction of turning spindle. The rotational direction is indicated by a curved arrow.

Sensor plate: The sensor plate is a device that is placed on the top of the workpiece. It is of known thickness and on having a contact with the spindle, sends an input to the tool zero on the control board.

Spindle: The rotating motor that holds the cutting tool.

Stepper Drive: An amplifier that controls the action of the stepper motor.

Stepper motor: An axis motor that uses the stepper technology.

Tool Offset: When machining, allowances must be made for the size of the tools being used, since they all differ in length. The tool offset is the amount the Z value must be moved (or offset). Sometimes also called tool length offset.

X-axis: The axis that refers to the width of the table left to right. It is generally the longest axis of movement of the work piece or tool. It should be perpendicular to Z-axis and always parallel to the surface of holding device.

Y-axis: The axis that refers to the length of the table. Front to back.

Z-axis: The axis that refers to the accessible range of area above the table.

INDEX

Connection Diagrams:

CNC control board connection diagram with pen drive, 15
Overall connection diagram, 16
Axes and spindle drive connections, 17
Block mode & tool zero connection, 18
Home input connections, 19
Coolant, spindle drive & VFD connections, 19
Limit inputs connections, 20

Gcode File:

Format of Gcode line, 65
Gcode allowed to use, 63-64
Mcode allowed to use, 64

Handheld pendant:

Pendant description, 25-27
Emergency button, 28
Brightness of LCD, 28
CNC control board connections with pendant, 31

Keys functions:

Navigation keys:

Key left: WCS Toggle, 55
Key right: Jog mode menu, 55
Key right long press: Jog offline mode, 56
Key UP: Jog factor increment, 57

Key Down: Jog factor decrement, 57

Numeric keys:

Key 0: Tool 0, 62
Key 1-4: Coordinate zero keys, 57-58
Key 5: Gcode file select, 59
Key 5 long press: Gcode file check, 59
Key 6: Override screen, 59
Key 7: Go to zero, 60
Key 7 long press: Homing, 60
Key 8: Feed fold/ Resume, 61
Key 9: Cycle start, 61
Keys functions in various modes-29-30

Main menu:

Structure tree for menu, 36
General Settings, 37-40
Active profile, 41
Axis Settings, 41-44
IO settings, 44-49
Tool settings, 50-52
Data File 52

Specifications:

CNC Control board specification, 9
Inputs & outputs specifications for control board, 22-24