TNC-M14-CNC Controller v1.1
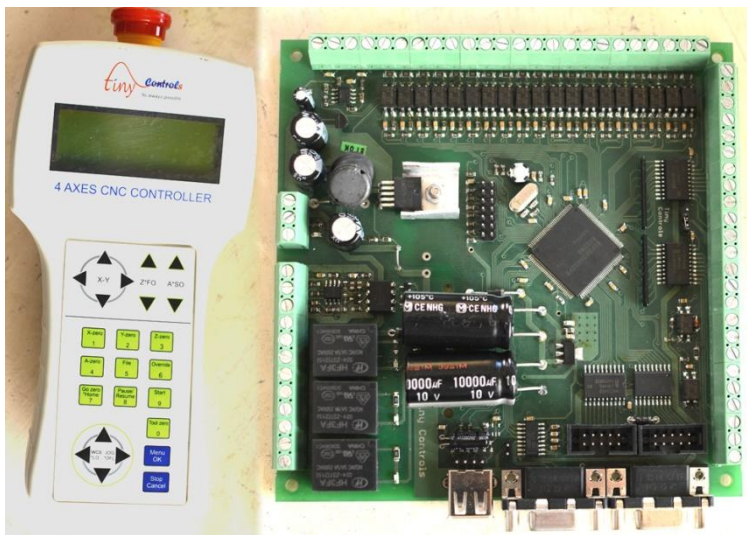
Document: Operation Manual
Document #: T12
Document Rev: 1.0

Product: CNC Controller
Product Rev: 2.0
Created: Oct-2013
Updated: Dec-2014

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

▪ **TNC-M14-CNC CONTROLLER v1.1**

"TINY CONTROLS" AND THE TINY CONTROLS COMPANY'S LOGO ARE COPYRIGHTS OF TINY CONTROLS PVT. LTD. 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.

CONTENTS

Warning:

When the warning icon appears, the user should know that it contains information which is essential for avoiding a safety hazard.

Caution:

When the caution icon appears, the user should know that it contains information which is necessary for avoiding a risk of damage to the product or other equipment.

Note:

When the Note icon appears, it indicates information which, if not heeded, can result in the CNC controller not operating to full efficiency. It also contains information concerning incorrect operations.

Tip:

When the Tip icon appears, it indicates information that can prove handy for performing certain settings or operations with the equipment.

Application:

▪ Do not use the CNC controller for a life support system or other purposes directly related to the human safety.

▪ Although CNC controller is manufactured under strict quality control, the user must install safety devices for applications, especially where serious accidents or material losses are foreseen in relation to its failure.

Installation:

▪ Install the control board on a non-inflammable material such as metal.

▪ Do not place inflammable matter nearby.

▪ Prevent lint, paper, fibers, sawdust, dust, metallic dust or other foreign material from accumulating onto the control board.

▪ Do not install or operate the CNC controller that is damaged or lacking parts. Doing so causes fire, accidents and/ or injuries.

Wiring:

▪ It is mandatory to connect the grounding wires.

▪ Only qualified operators should carry out the parts or the complete wiring process.

▪ Ensure that the power is turned off while performing the wiring of the whole system.

▪ Generally, the control signal wires are not reinforced during insulation. If they accidently touch any of the live parts in the main circuit, their insulation coat may break for any number of 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. Malfunction of the nearby devices should be taken care of immediately. Noise control measures need to be implemented to prevent the motor from malfunctioning.

Maintenance and Inspection, and Parts Replacement:

▪ Maintenance, inspection and parts replacement should be made by qualified persons only.

- Take off watches, rings and other metallic matter before starting any work on the machine.
- Use insulated tools only.

Operation:

- Do not operate the switches with wet hands.
- Although the hand pendant has an emergency button, the user needs to install an emergency button separately along with mounting of the control board.
- Do not turn the main circuit power on or off in order to start or stop CNC controller operation.
- In general, sheathes and covers of control signal cables and wires are not 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 do not come into contact live conductors of the main circuits.

Others:

- Do not modify the CNC control board or the handheld pendant device.

General Precautions:

Drawings in this manual for the connections may be illustrated without the shield cover to explain the parts in detail. Restore the shields in the original state and follow the description in the manual before starting any operation.

The ground terminal should be connected to the ground. Use ground wires whose size is greater than the power supply lines.

- Power supply should not be greater than 24 V; else it damages the Controller Board.
- Caution is necessary during mechanical and electrical installation. Poorly tightened cables cause many problems; it's also very difficult to detect such as defects while operating the system. To avoid electric shocks or equipment damages or burning of power cords, check loose connections or exposed extension cords which may cause tripping hazard.
- Unplug the controller power supply and other connection when the controller is going to be left unused for an extended period of time.

CNC CONTROLLER INCLUDES

1. Control Board

2. Handheld pendant

3. Serial cable (3 m) connected to the pendant.

MECHANICAL INSTALLATION
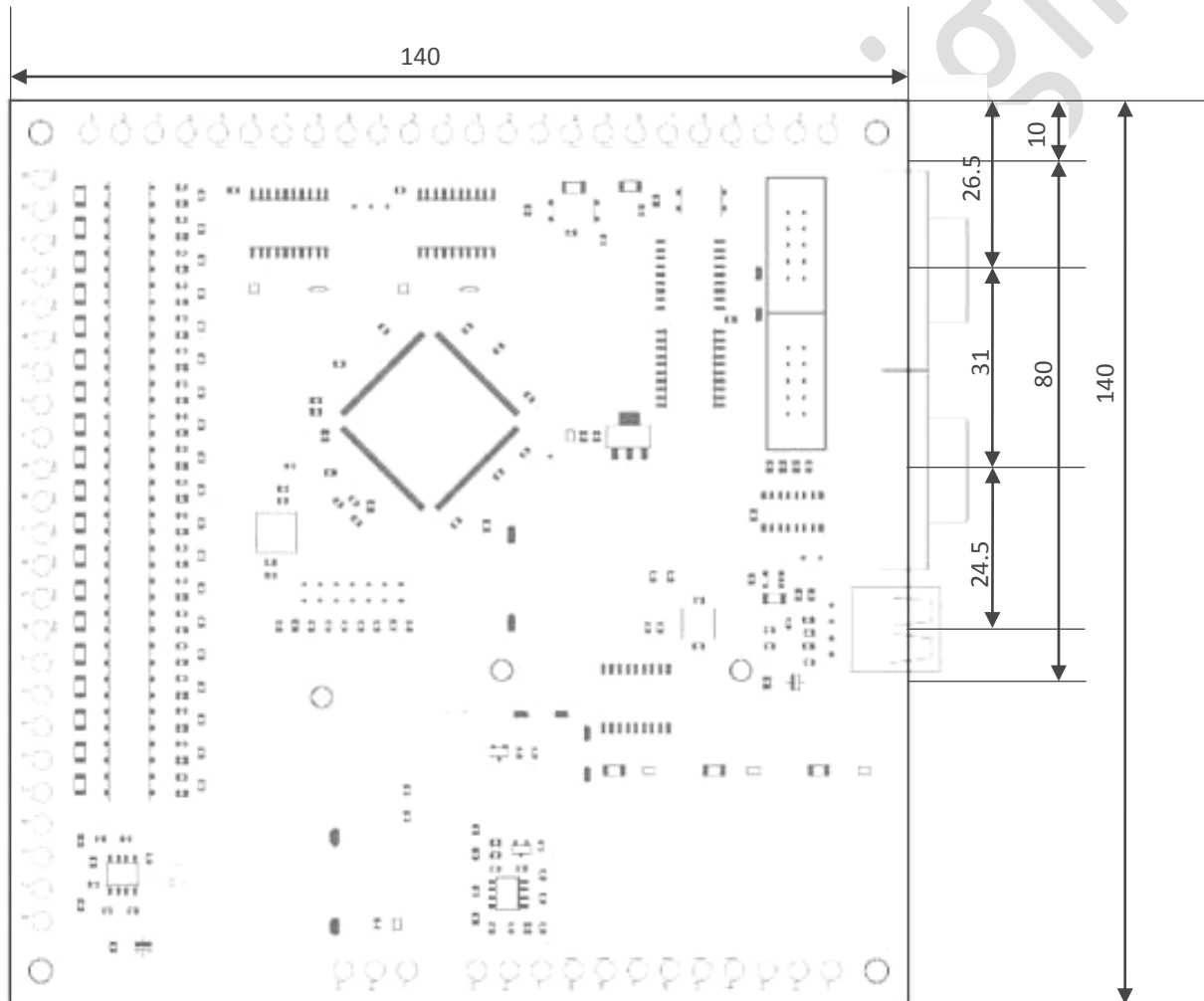
INTRODUCTION

The 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. A **Y-slave** is also included in the system enhancing the flexibility of CNC controller. All 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 2 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 it does not require a PC. **There is no limit on G-code file size**. The controller can resume the operation on any power interruption saving the current machine position, line of G-code and other status. The power up from interrupted cycle offers to recover from power fail, automatically 'homing' the machine, positioning to last location, and resuming from last saved line without losing a beat in the process. Traditional complicated NC operation becomes easier and simpler because of its user friendly interface.

The control board is the brain of the CNC controller. It handles the command reading function from the pen drive and the serial transmission function with the 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 the 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 positioned as 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. 19 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 Pendent 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 + Y-Slave axis
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	19
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	140x140 (in mm)
Weight	50g

Top terminals (left to right): X-DIR, X-STEP, X-COM, Y-DIR, Y-STEP, Y-COM, Z-DIR, Z-STEP, Z-COM, A-DIR, A-STEP, A-COM, Y-SLAVE DIR, Y-SLAVE STEP, Y-SLAVE COM, MISC OUTPUT 1, MISC OUTPUT 2, COM, MISC OUTPUT 3, MISC OUTPUT 4, COM, SPINDLE DIR, SPINDLE STEP, COM

Left terminals (top to bottom): +24V IN, MISC INPUT 4, MISC INPUT 3, MISC INPUT 4, Y SLAVE HOME IN, X HOME IN, Y HOME IN, Z HOME IN, A HOME IN, X LIMIT IN, Y LIMIT IN, Z LIMIT IN, A LIMIT IN, MISC INPUT 1, TOOL ZERO IN, BLOCK MODE IN, EMERGENCY, START, STOP, HOLD/ RESUME, ANALOG OUT 2, AGND, ANALOG OUT 1, AGND

Right callouts: CONNECTION TO PENDANT, MISC OUTPUTS 5-12, MISC OUTPUTS 13-20, CONNECTION TO PEN DRIVE

Bottom terminals (left to right): +24V IN, GND, +5V OUT, 10 V ANALOG IN, 0-10 V ANALOG OUT, 0V ANALOG IN, NO, COM SPINDLE OUTPUT, NC, NO, COM COOLANT MIST, NC, NO, COM COOLANT FLOOD, NC

Digital Outputs: The digital outputs are as following:

X-DIR OUT ○
X-STEP OUT ○
COM ○
Y-DIR OUT ○
Y-STEP OUT ○
COM ○
Z-DIR OUT ○
Z STEP OUT ○
COM ○
4-DIR OUT ○
4-STEP OUT ○
COM ○
Y-SLAVE STEP OUT ○
Y-SLAVE DIR OUT ○
COM ○
MISC OUTPUT 1 ○
MISC OUTPUT 2 ○
COM ○
MISC OUTPUT 3 ○
MISC OUTPUT 4 ○
COM ○
SPINDLE DIR ○
SPINDLE STEP ○
COM ○

a. Axes Outputs

a. Step outputs: 4

b. Direction outputs: 4

The CNC controller board has Steps/Dir outputs for four axes named X, Y, Z and 4th axis and one for Y-slave axis. The outputs are buffered outputs and can be disabled by disabling drive: Enable/Disable parameter in axis settings for axes which aren't required for an operation.

COM Selection Jumper

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

Outputs for Spindle Drive

a. Step output: 1

b. Direction output: 1

The CNC Control board has a buffered Step/Dir output for spindle drive. The step frequency is user programmable and has a range of up to 20 KHz. These outputs can be disabled by disabling the Spindle**: Enable/Disable** parameter in Spindle settings of IO settings function.

c. Analog 0-10V Output:

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

10 V ANALOG IN
0-10V ANALG OUT
0V IN
NO
SPINDLE — **COM**
NC
NO
COOLANT MIST — **COM**
NC
NO
COOLANT FLOOD — **COM**
NC

J2

M54.05	M54.06
M54.07	M54.08
M54.09	M54.10
M54.11	M54.12
+ve CMN	GND

J3

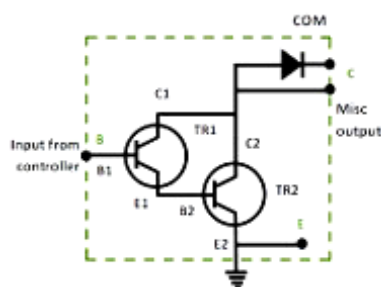| M54.13	M54.14
M54.11	M54.16
M54.17	M54.18
M54.19	M54.20
+5V	GND

d. Spindle Enable & Coolant Outputs:

Spindle NO output signal directly goes to VFD and enables the spindle. Spindle NO is a relay output. The relay switches to NO if **spindle** parameter is set **enable** in **IO Settings** and spindle is positioned as ON in G-code**.**

Coolant mist NO and coolant flood NO are the relay outputs. The relay switches to NO if these outputs are positioned as ON in the G-code.

e. Misc Outputs:

Misc outputs are miscellaneous outputs and are 20 in number. Required output can be positioned as ON in G-code file. Misc outputs 1-4 and 13-20 are buffered and the rest are open collector outputs. These can be made **OFF** collectively in G-code file but are to be positioned as **ON** individually using the respective M-code. The individual output terminals with its respective M-code are shown in the adjacent figure.

The adjacent figure shows the **connector J2** outputs (see the terminal diagram on page number 10 with respective M-code. These are 8 in number and are open collector outputs and can be positioned as on-off using G-code. These outputs generate Active high output when made ON in G-code.

The adjacent figure shows the **connector J3** outputs with respective M-code. These are 12 in number and are buffered outputs. Enable/Disable these outputs in the MISC outputs function of IO settings and position them as ON/OFF in G-code. These outputs can be set active high/active low in the Misc outputs function.

Use M54.01- M54.20 to make the Misc outputs on and M55.01-M55.20 to make them off. Use M56 to switch off all the miscellaneous outputs.

f. Analog outputs 1-2:

Analog Output 2 0-10 V ○
AGND ○
Analog Output 1 0-10 V ○
AGND ○

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

g. +5V Output:

○ ○ ○
+24 V IN   GND   +5 V OUT
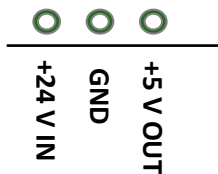
A +5V output can also be drawn from the controller board. It supplies 100mA current and remains active until the power continues to be supplied to the board.

CONNECTING INPUTS

○ ○ ○
+24 V IN   GND   +5 V OUT
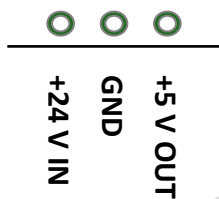
Inputs: The inputs are as following:

a. Power Supply

Connect a power supply of +24V DC at the input pin of the 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 recieves this signal by a wire whose other end is connected to the tool by a clamp.
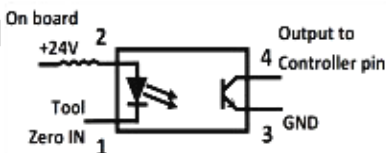
Terminal
24V COMMON SUPPLY (SEPARATE FROM THE POWER SUPPLY TO BOARD)
MISC INPUT 4
MISC INPUT 3
MISC INPUT 2
HOME Y-SLAVE IN
HOME X IN
HOME Y IN
HOME Z IN
HOME 4 IN
LIMIT X IN
LIMIT Y IN
LIMIT Z IN
LIMIT 4 IN
MISC INPUT 1
TOOL ZERO IN
BLOCK MODE IN
EMERGENCY
START
STOP
HOLD/ RESUME
ANALOG OUTPUT 2
AGND
ANALOG OUTPUT 1
AGND

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.

d. Homing inputs:

Homing inputs are input to **"HOME IN"** terminals of the CNC controller board from home switches of the CNC machine for respective axes. These inputs can be set as active high or active low. Usually all the axes of CNC machine consist of the Home switch. Disable the home switch function in Home Switch Settings menu for the axes which do not have Home switch or for the axes for which home switches are not required. A separate slot for Y-slave Home is given on the CNC board.

e. Limit switch inputs:

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

f. Misc Inputs:

Four terminals of misc inputs are provided on the board. These input terminals are misc input 1, misc input 2, misc input 3 and misc input 4. These are shown in terminal diagram (CNC Controller Board Terminals) on page number 10.

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 controller board.

Note: The USB flash drive should not be disconnected when the program is running on the G-code. Handle the en drive carefully.

Note: Connect the pen drive carefully. Reverse connections damage the pen drive. Do not connect the pen drive forcefully.

CONNECTION DIAGARM

Analog In

Gnd

Analog Out

Spindle On/ Off

Gnd

Spindle Step

Spindle Dir

Power supply to drive

Power supply to drive

Power supply to drive

Power supply to drive

Connection for pendant

Coolant Flood

Coolant Mist

Power Supply

Y-SLAVE HOME

HOME X	LIMIT X		Emergency
HOME Y	LIMIT Y	TOOL ZERO	Start
HOME Z	LIMIT Z	BLOCK MODE	Stop
HOME 4	LIMIT 4		Hold/ Resume

16
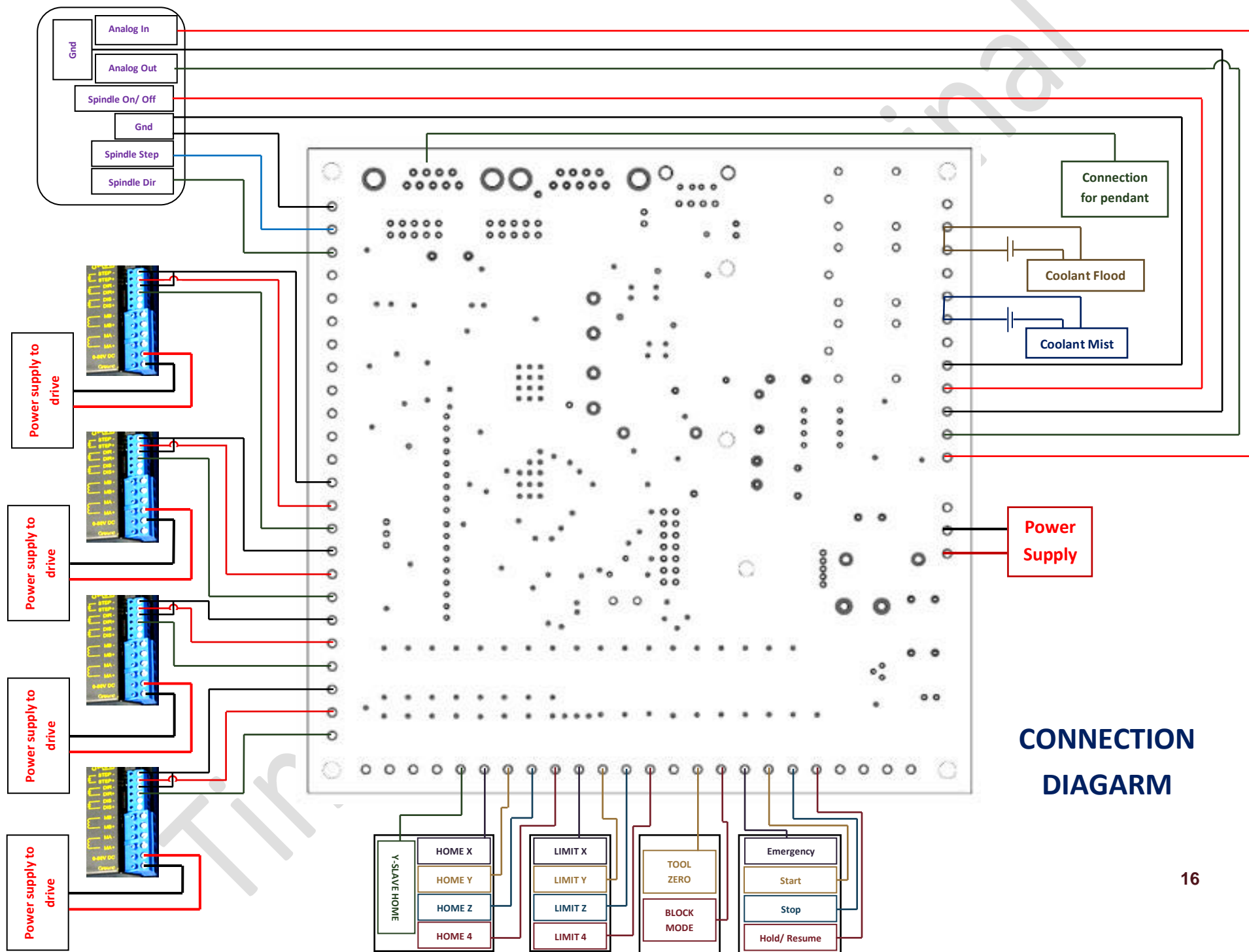
AXES AND SPINDLE OUTPUTS CONNECTIONS WITH DRIVES

X DIR OUT
X STEP OUT
COM
Y DIR OUT
Y STEP OUT
COM
Z DIR OUT
Z STEP OUT
COM
4 DIR OUT
4 STEP OUT
COM
Y-SLAVE DIR OUT
Y-SLAVE STEP OUT
COM
MISC OUTPUT 1
MISC OUTPUT 2
COM
MISC OUTPUT 3
MISC OUTPUT 4
COM
SPINDLE DIR
SPINDLE STEP
COM

SPINDLE

SPINDLE DIR

SPINDLE STEP

COM
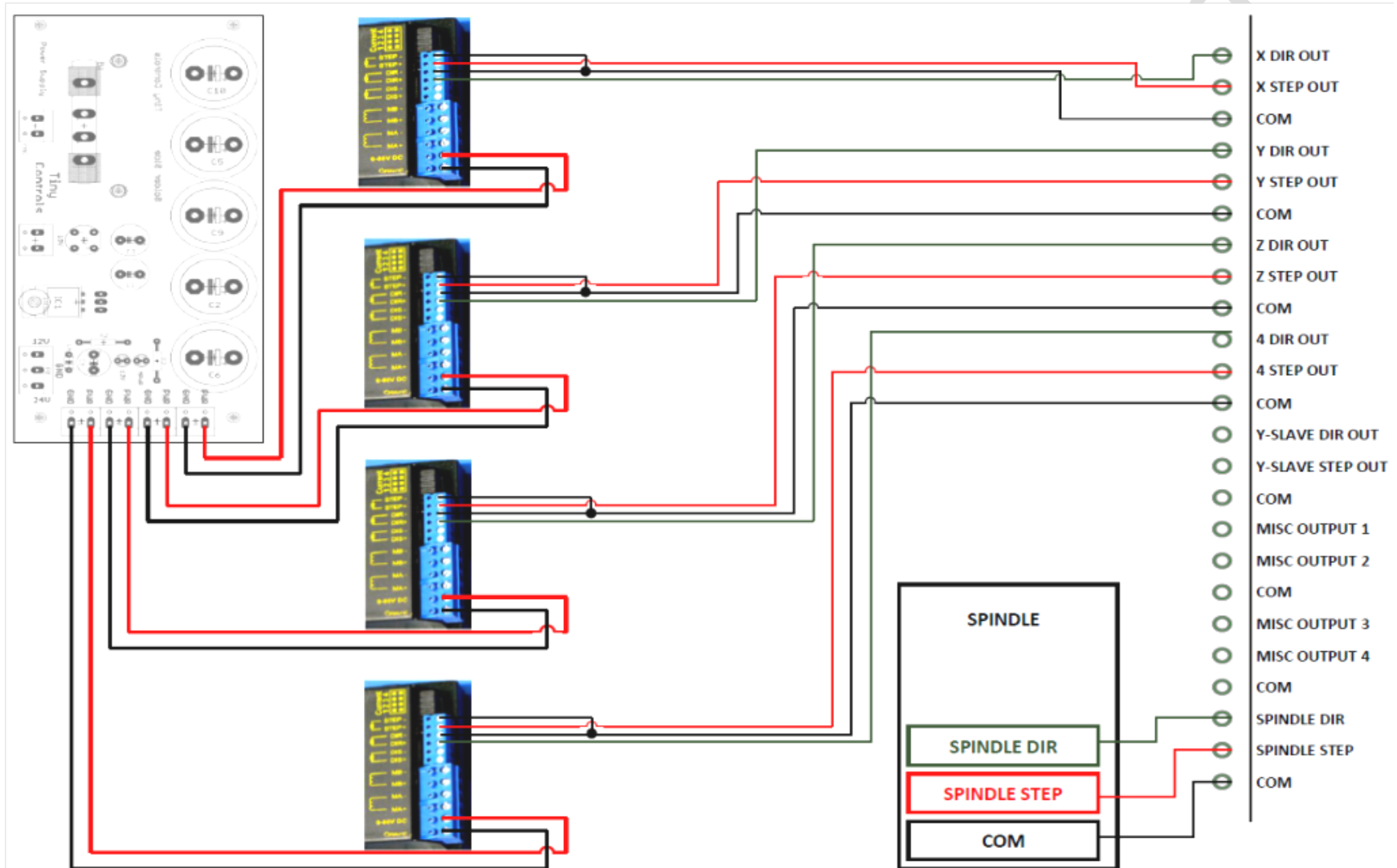
BLOCK MODE, TOOL ZERO AND MISC INPUTS CONNECTIONS

24V COMMON SUPPLY (SEPARATE FROM THE POWER SUPPLY TO BOARD)
MISC INPUT 4
MISC INPUT 3
MISC INPUT 2
HOME Y-SLAVE IN
HOME X IN
HOME Y IN
HOME Z IN
HOME 4 IN
LIMIT X IN
LIMIT Y IN
LIMIT Z IN
LIMIT 4 IN
MISC INPUT 1
TOOL ZERO IN
BLOCK MODE IN
EMERGENCY
START
STOP
HOLD/ RESUME
ANALOG OUTPUT 2
AGND
ANALOG OUTPUT 1
AGND

Spindle motor
Collet
Tool
Clip holding wire
Sensor Plate
Work bed
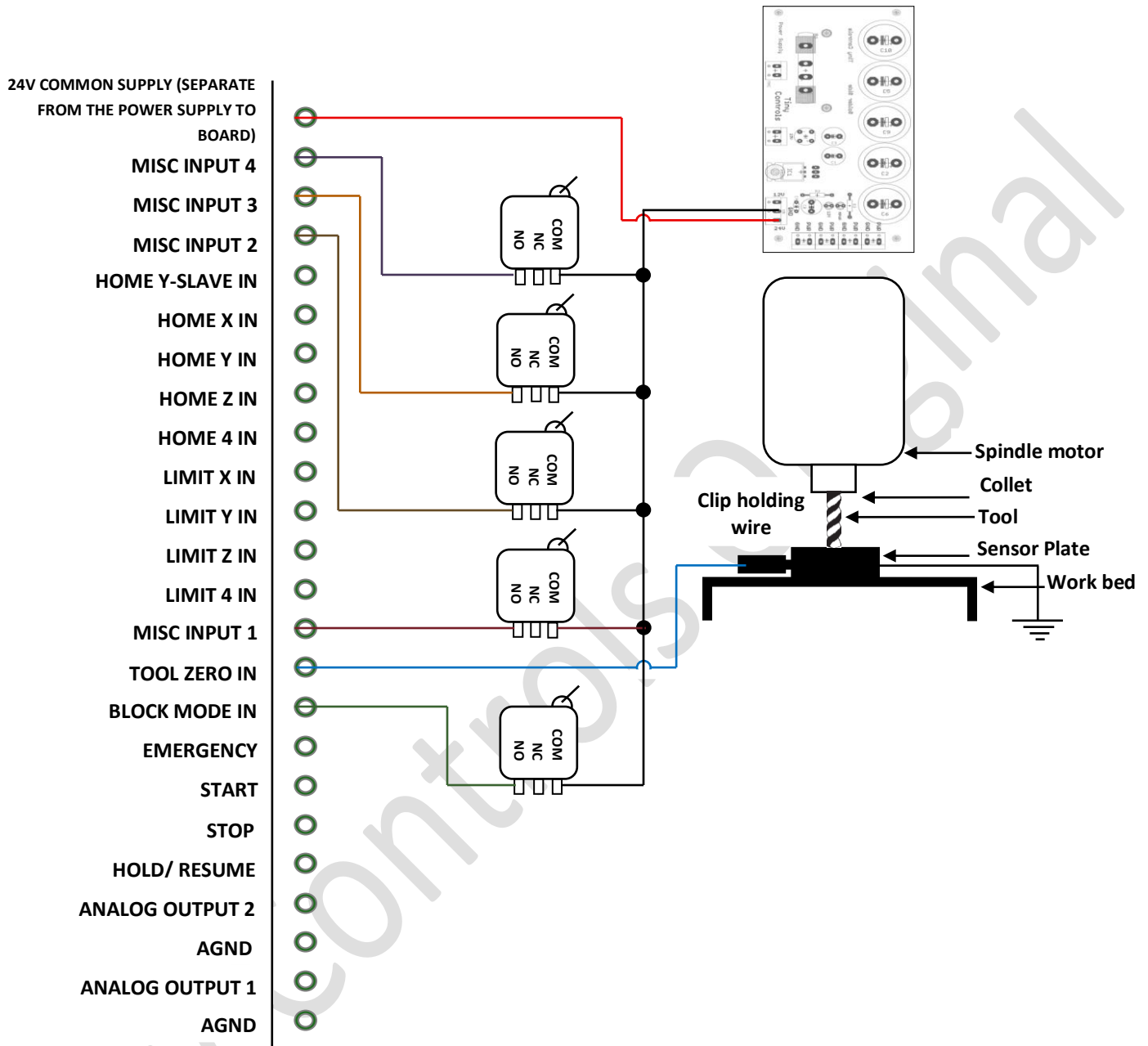
Note: The CNC control board should be installed in a dust free environment with suitable temperature and humidity conditions. Read the **CNC Controller Specification table** for these parameters.

HOME AND LIMIT INPUTS CONNECTIONS

24V COMMON SUPPLY (SEPARATE FROM THE POWER SUPPLY TO BOARD)
MISC INPUT 4
MISC INPUT 3
MISC INPUT 2
HOME Y-SLAVE IN
HOME X IN
HOME Y IN
HOME Z IN
HOME 4 IN
LIMIT X IN
LIMIT Y IN
LIMIT Z IN
LIMIT 4 IN
MISC INPUT 1
TOOL ZERO IN
BLOCK MODE IN
EMERGENCY
START
STOP
HOLD/ RESUME
ANALOG OUTPUT 2
AGND
ANALOG OUTPUT 1
AGND

Home Y-Slave
Home X
Home Y
Home Z
Home 4
Limit X+
Limit X-
Limit Y+
Limit Y-
Limit Z+
Limit Z-
Limit 4+
Limit 4-
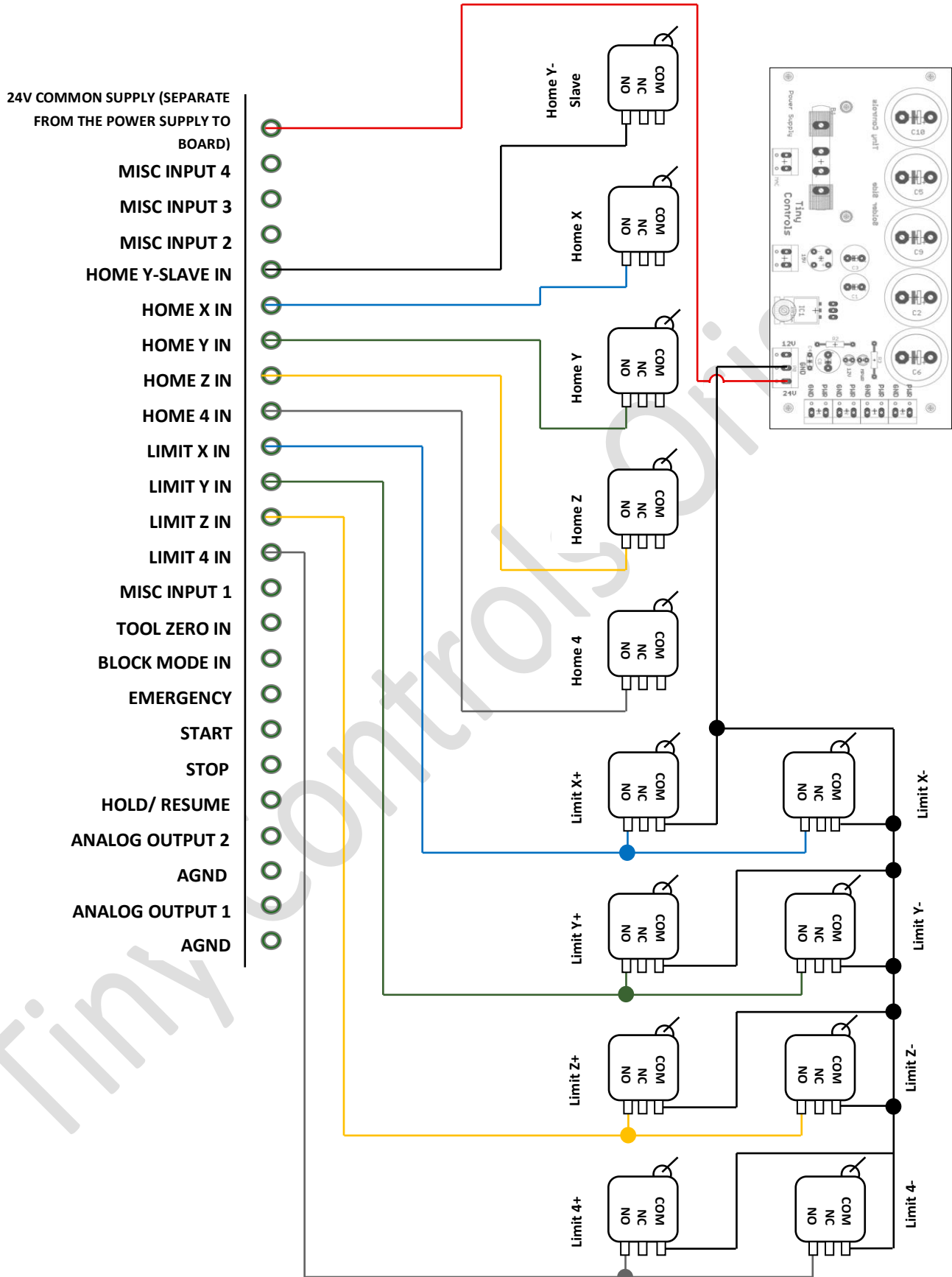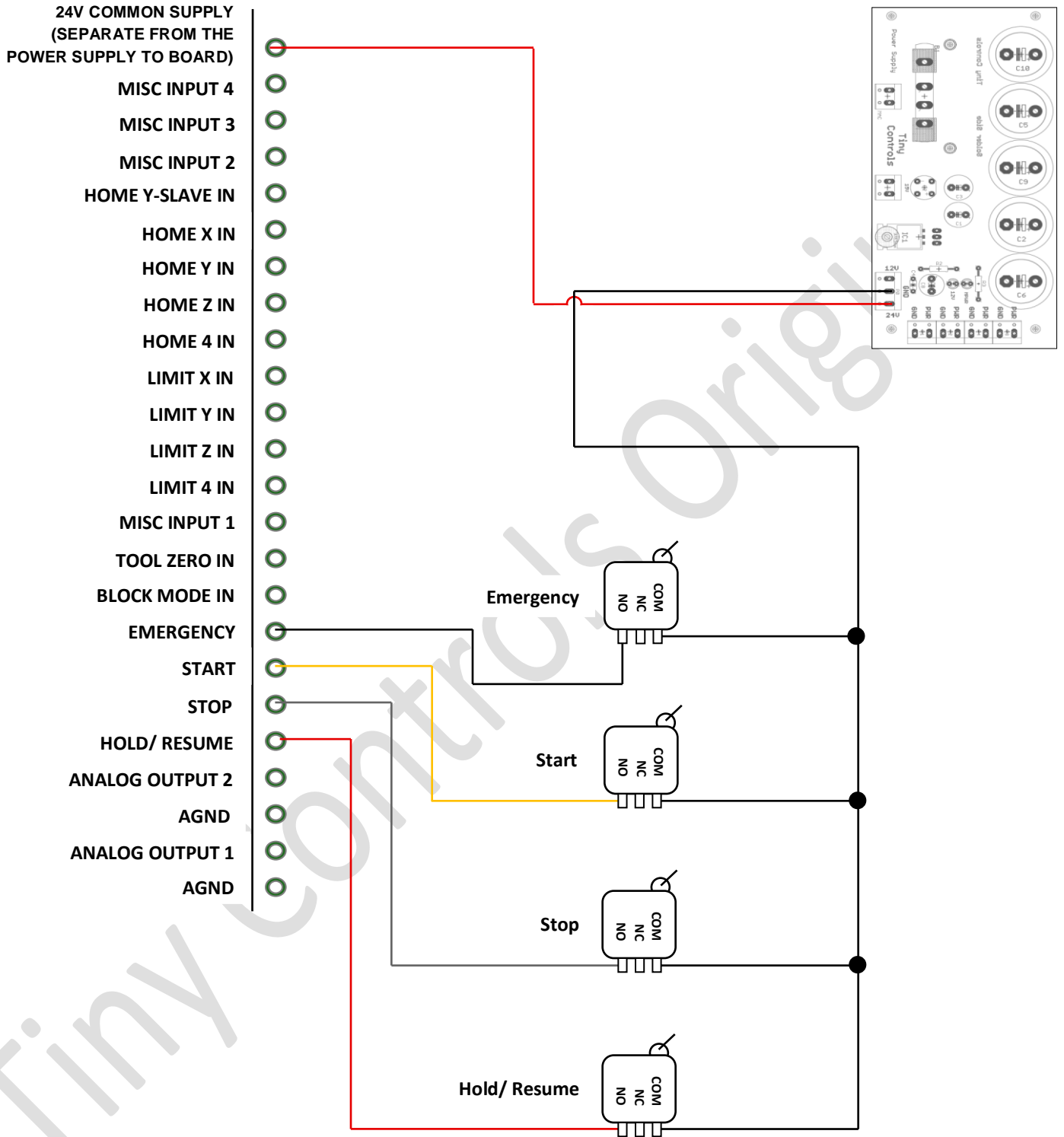
COOLANTS, SPINDLE DRIVE AND VFD CONNECTIONS
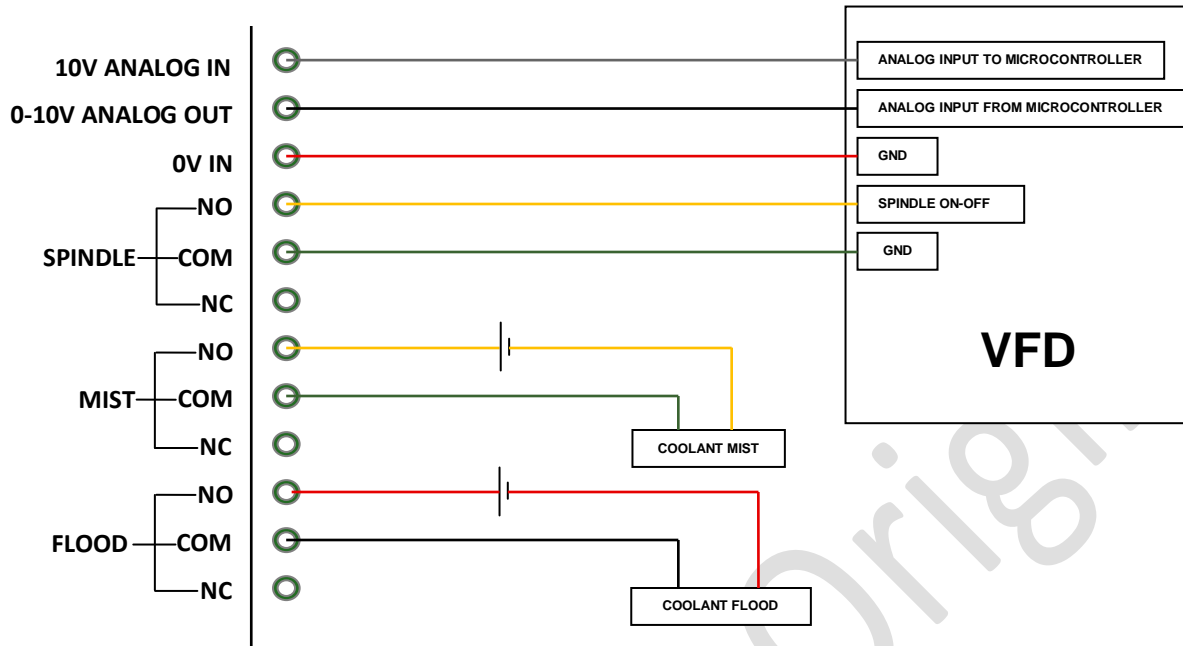
MOUNTING INSTRUCTIONS

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

1. Never use the controller 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 greater than 90% (non-condensing).
3. Do not use this product along with a strong magnetic field.
4. Protect it from dust, water and heat. Make sure that there are no conductive particles near the control board.
5. Do not allow any liquid or other foreign body to get on the controller board or on the pendant.
6. Do not 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, SPINDLES & COOLANT OUTPUTS:

Terminal Name	Type	Pin Function
X-axis Step pulse	Buffered Output	Pulse o/p for X axis, voltage=5V, current=20mA
X-axis Dir pulse	Buffered Output	Signal o/p for X axis, voltage=5V, current=20mA
Y-axis Step pulse	Buffered Output	Pulse o/p for Y axis, voltage=5V, current=20mA
Y-axis Dir pulse	Buffered Output	Signal o/p for Y axis, voltage=5V, current=20mA
Z-axis Step pulse	Buffered Output	Pulse o/p for Z axis, voltage=5V, current=20mA
Z-axis Dir pulse	Buffered Output	Signal o/p for Z axis, voltage=5V, current=20mA
4-axis Step pulse	Buffered Output	Pulse o/p for Z axis, voltage=5V, current=20mA
4-axis Dir pulse	Buffered Output	Signal o/p for Z axis, voltage=5V, current=20mA
Y-slave Step pulse	Buffered Output	Pulse o/p for Y-SLAVE, voltage=5V, current=20mA
Y-slave Dir pulse	Buffered Output	Signal o/p for Y-SLAVE, voltage=5V, current=20mA
Spindle Step Pulse	Buffered Output	Pulse o/p for spindle, voltage=5V, current=20mA
Spindle Dir signal	Buffered Output	Signal o/p for spindle, 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 o/p terminal	isolated	O/p voltage=0-10V (when I/P is 10V), current=20mA
Analog o/p terminal (0-10V)	Non-isolated	O/p voltage= 0 - 10V, current= 20mA
Analog o/p terminal (0-10V)	Non-isolated	O/p voltage= 0 - 10V, current= 20mA

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

Inputs: Power supply

Terminal Name	Type	Pin Function
System main power supply	24V DC	Power+
	Gnd	Power-

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

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

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

Tip: Refer to diagnosis function in **General settings** to analyze inputs and outputs of the CNC control board.

Inputs:

Terminal Name	Type	Pin Function
Y-Slave Home IN	Optoisolated I/P	Accept Y-Slave Homing Input
X Home IN	Optoisolated I/P	Accept X-axis Homing Input
Y Home IN	Optoisolated I/P	Accept Y-axis Homing Input
Z Home IN	Optoisolated I/P	Accept Z-axis Homing Input
4 Home IN	Optoisolated I/P	Accept 4-axis Homing Input
X Limit IN	Optoisolated I/P	Accept X-axis Limit Input
Y Limit IN	Optoisolated I/P	Accept Y-axis Limit Input
Z Limit IN	Optoisolated I/P	Accept Z-axis Limit Input
4th Limit IN	Optoisolated I/P	Accept Z-axis Limit Input
Tool Zero	Optoisolated I/P	Set Tool to Zero position
Block Mode	Optoisolated I/P	One line of G-code is executed when an input is given at this pin
Emergency	Optoisolated I/P	Accept Emergency input
Start	Optoisolated I/P	Accept start input
Stop	Optoisolated I/P	Accept stop input
Hold/ Resume	Optoisolated I/P	Accept hold/ resume input

The Pendant can be taken as hand held control for a machine tool. This device is used in conjunction with the CNC controller board to give the user complete access to the schematic and work area. The pendant and the CNC control board make 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 the user's 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** at page number 29 to know about the functions of the keys 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 different functions in different modes. All the changes done in numeric values of parameters are reflected on LCD.

2. OK & Esc:

OK button allows the user to enter in main menu when the controller is switched on. It also saves modified values in EEPROM when edited. Esc backs out of menus/modes. Pressing Esc repeatedly allows the 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 alternate functions in some modes.

Up- down Navigation keys: ▲▼   **Up:** ▲▼   **Left:** ◀

Left-Right Navigation Keys: ◀▶   **Down:** ▼   **Right:** ▶

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. The 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:

X-Y JOG KEYS   **Z JOG KEYS**   **4TH JOG KEYS**
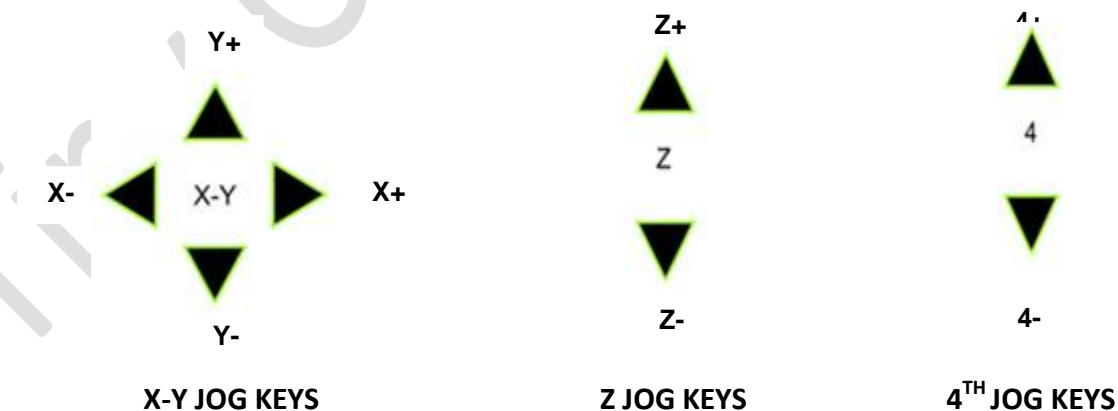
▪ **Z Jog keys:** These keys move the Z-axis up and down.

- **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 of 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 on the right side of the LCD screen shows a sign notifying the menu functions above.
- The ▼ symbol at lower corner on the right side of the LCD screen shows a sign notifying the user of the further menu functions below.

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

After **confirming** the changes have been made and before exiting to main menu, it is required for the user to **save** the modifications in settings in the desired profile else the controller retains the previous settings. Press Ok to save these settings.
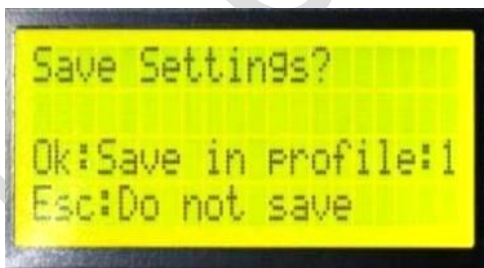
The adjacent screen appears when Esc is pressed for exiting back to the **Main Menu**.

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

Note:
Do not 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

The user can give an emergency input by a red color button present at the top of the pendant. When this button is pressed, the motion stops, however, it does not cut down the power supply to the 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 the 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, the machine remains at the same position after resuming. However, if any file is running and emergency button is pressed, the machine starts from its current position and moves to the home position on resuming. From home position, X-axis, Y-axis and Y-Slave move to the coordinate position at which the 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. The 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 the 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 the coordinate screen.

KEYS FUNCTIONS IN VARIOUS MODES

Most of the keys have different functions in different modes. The following table shows the functioning of the keys in various modes. Given below 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 the connection between the pendant and controller is lost when the machine is running according to the file loaded to the 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	Idle mode
Z-	FEED RATE-
Z+	FEED RATE-
4-	SPINDLE SPEED-
4+	SPINDLE SPEED+
Y+	FEED RATE RESET
Y-	SPINDLE SPEED RESET
5	SPINDLE ON-OFF

Table 1.2

CNC CONTROLLER BOARD CONNECTIONS WITH PENDANT

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

Each pendant is shipped with an already connected serial cable. The connection between the CNC control board and the pendant is made through an on-board DB-9 (F) connector and a DB-9 (M) connected at the other end of the serial cable. If the communication between the Pendant and the CNC controller board is lost, a message **"Controller disconnected"** appears on the LCD. When the connection resolves back, the 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 from2 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 or 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 the serial port and the pendant.
4. The cable should be checked before installing and also after installation to ensure that there are no cuts or damages.
5. There should not be any loose connections else it leads to poor transmission quality and difficulties in transmissions.
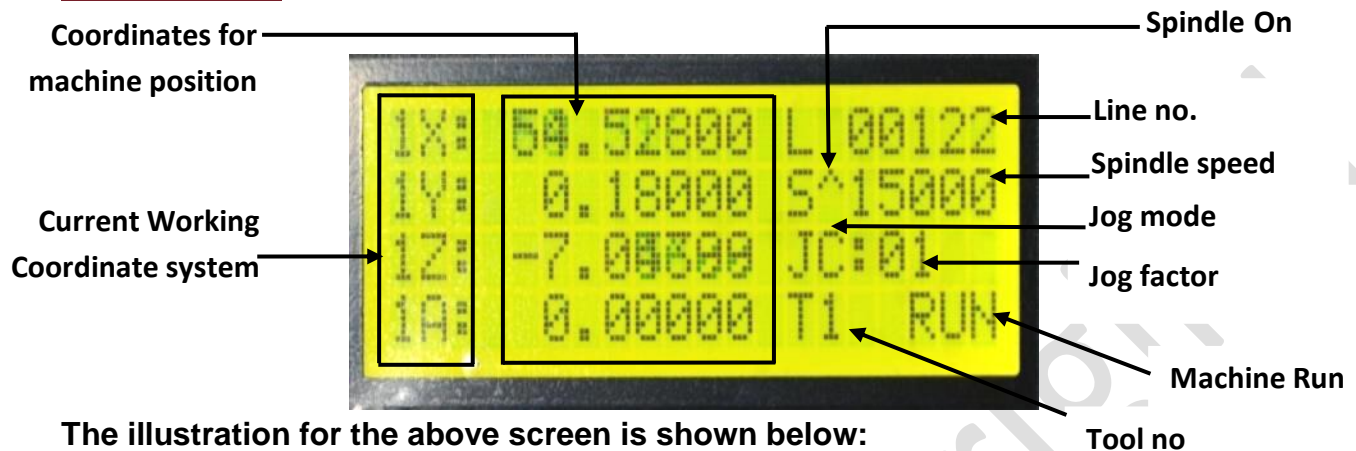
Note: Do not operate the controller or the pendant with wet hands. Ensure the connection of the pendant with the dedicated connector on the CNC Controller board.

Note: Do not drop the pendant. The LCD can be damaged due to external mechanical strokes. Handle the pendant carefully.

Note: Default values are stored in on the ROM chip of every controller and it cannot be altered by the user.

MACHINE RUN:

Coordinates for machine position

Current Working Coordinate system

Spindle On

Line no.

Spindle speed

Jog mode

Jog factor

Machine Run

Tool no

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". The CNC controller supports 6 incremental coordinate systems (G54-G59) and an absolute coordinate system (G53). For 3 axes system, LCD does not show rotary axes.

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. The spindle rotates if it is enabled and made ON in the G-code 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 the CNC controller.

L: L shows the number of the current line being in execution in the G-code file.

Tool number: T1 shows that tool number 1 is selected and being in operation. A provision of 6 tools is provided on the 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:
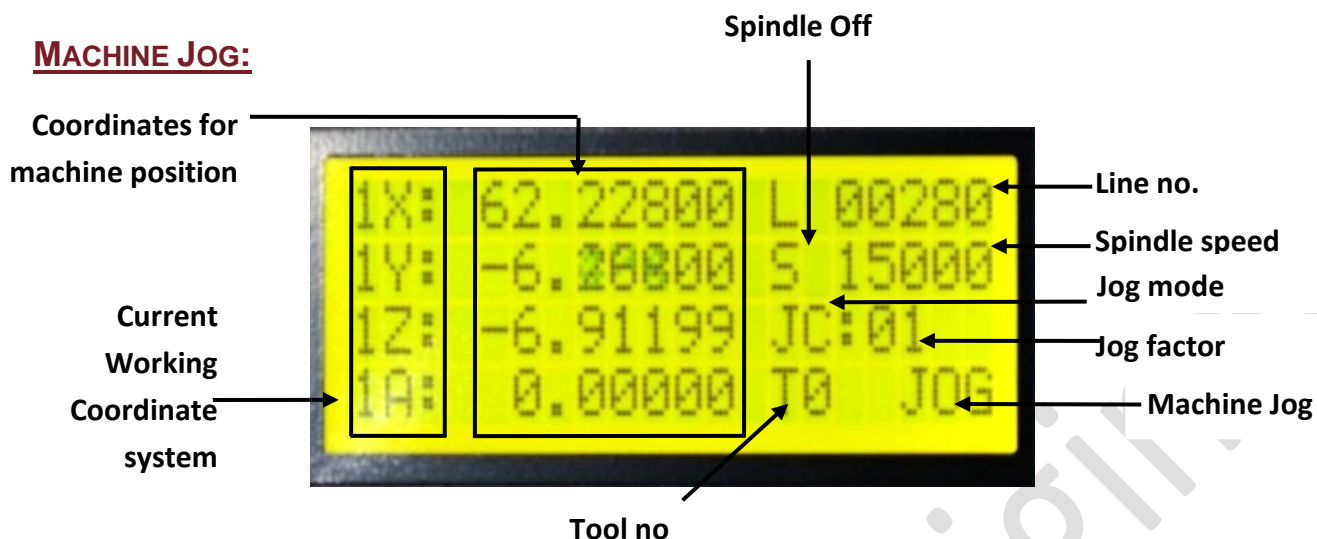
Spindle Off

Coordinates for machine position

Current Working Coordinate system

```
1X: 62.22800    L 00280
1Y: -6.28000    S 15000
1Z: -6.91199    JC:01
1A:  0.00000    T0    JOG
```
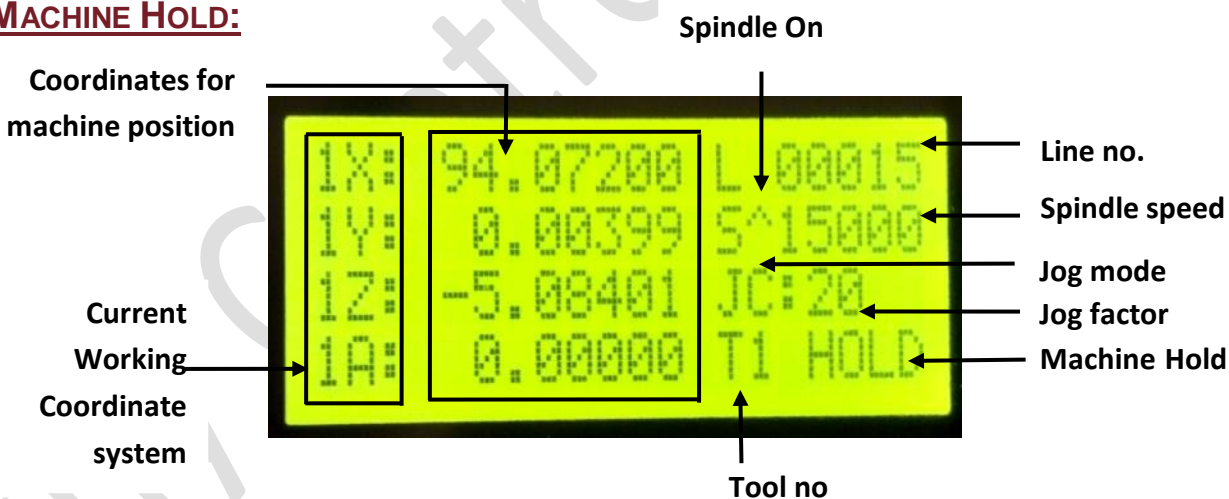
Line no.

Spindle speed

Jog mode

Jog factor

Machine Jog

Tool no

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 36** for jog modes and **table no 1.1** on page 29 for the jog keys and other keys for modifying the jog value.

MACHINE HOLD:

Spindle On

Coordinates for machine position

Current Working Coordinate system

```
1X: 94.07200    L 00015
1Y:  0.00399    S^15000
1Z: -5.08401    JC:20
1A:  0.00000    T1   HOLD
```

Line no.

Spindle speed

Jog mode

Jog factor

Machine Hold

Tool no
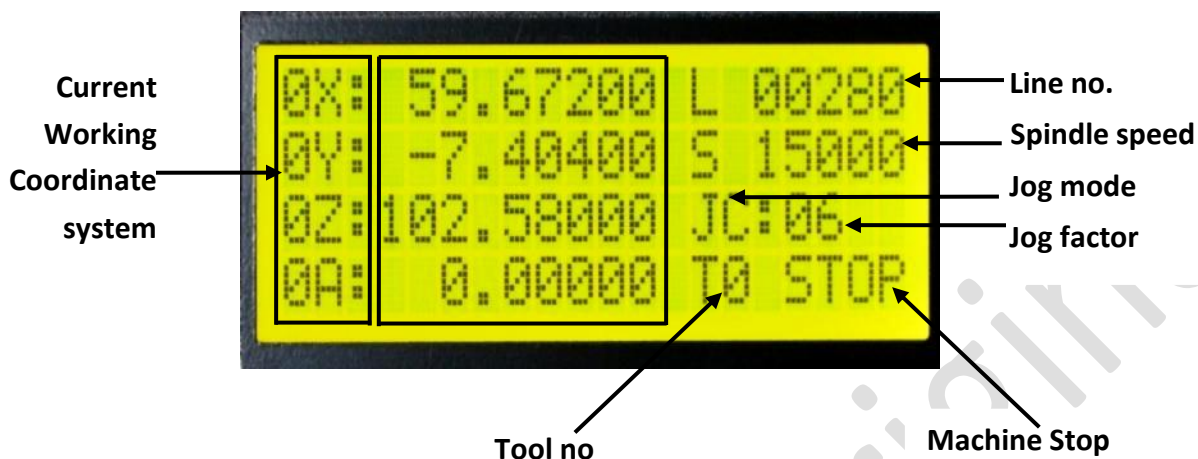
The Current working coordinate system, Coordinates for machine position, Tool number, Line number, Jog mode, Jog factor, Spindle Speed, Spindle On/Off have their usual meaning.

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

MACHINE STOP:

Current Working Coordinate system →	0X: 59.67200 L 00280 ← Line no.
	0Y: -7.40400 S 15000 ← Spindle speed
	0Z: 102.58000 JC:06 ← Jog mode / Jog factor
	0A: 0.00000 T0 STOP

Tool no   Machine Stop

Current working Coordinate system: Here "0" shows that the 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**.

The 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, Digital read Out (DRO) shows all the coordinates with respect to G54 working coordinate system, which can be identified by the number shown at the left column.
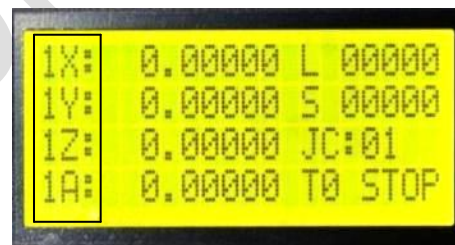
In other cases,

0 – Absolute coordinate

1 – G54, 2 – G55, 3 - G56, 4 - G57, 5 - G58, 6 –G59

This key enables the user to view coordinates in absolute mode in all the three modes. It displays 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 the 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 the LCD screen.

Manual override has been also discussed in the 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 is triggered

again; "hard limit/home" is not checked then the 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 for the travel of the CNC machine carriage such as the CNC machine axis movement. Jogging allows for manual operation of tools by using Jog keys. (The functions of jog keys have been discussed under jog keys panel of **the Hand held pendant** section). Jog mode can be selected in **run, idle** and **hold mode.** However, **jogging** can be operated in **idle** and in **hold mode** only.

During machine jogging, the DRO shows the adjacent screen. The CNC Controller provides the three modes for jogging. These are discussed below:

a. Continuous Mode: This mode allows for 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. The user can change the jog factor by up-down navigation keys. Jogging velocity for each axis can be calculated by following the given formula which shows the calculation of jog velocity for X-axis:

*Jogging velocity (**Axis**) = (Max Velocity for Axis/100) x 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 the user 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 the jog factor for Jog fast mode is 40 and the 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 gets changed and the whole WCS shifts to a new position. Due to jogging, position of the machine changes and the coordinates for current position of machine gets changed w.r.t. absolute coordinates. However, changed coordinates are not reflected by the screen after exiting from this mode. The LCD shows same coordinates as before entering in jog offline mode for machine current position. The absolute coordinates for WCS get updated to a position in such a way 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 the LCD for example remains (10, 10, 10). But the absolute coordinates for WCS 1 is changed in such a way that the 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.

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: 4th 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 the USB drive if USB device is properly mounted. Otherwise, the LCD shows the message of unavailability of USB Flash drive.

It is advised to keep file name at a maximum of **8 letters name** and **3 letters 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, the 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 at page 41. Spindle turns off if Esc button is pressed.

KEY 5 Long Pressing: G-code File Check

Pressing **KEY 5 long** leads the controller to check G-code file selected from the pen drive in **idle mode** for the safety of the CNC machine and its proper motion. The controller reads the complete G-code file and detects the errors and shows the respective error messages. The 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 tool change in the G-code during the execution of G-code file, the adjacent screen appears.

There can be miscellaneous errors in the G-code 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

The 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 G-code in the range of minimum to maximum velocity whereas the minimum value is the feed rate set in G-code and maximum value for velocity is the minimum value of MM/MIN function among all axes set by user in Axes settings menu.

The 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 G-code file in the range of MIN RPM to MAX RPM function set by the user in spindle setting menu.

KEY 6: OVERRIDE SCREEN

When the machine is in any of the 3 modes, pressing key 6 shows the override screen on the LCD. The 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, the 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, the maximum value for spindle speed override is 10% of the spindle speed and the maximum value of spindle speed override is 200% of the programmed spindle speed in G code file. However, the 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 a 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: The 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 the machine at a known location. Although this controller is capable of restoring location of the machine even after the power shut down and shows the machine's current location all the time, there can be some situations during the machine's 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, 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.
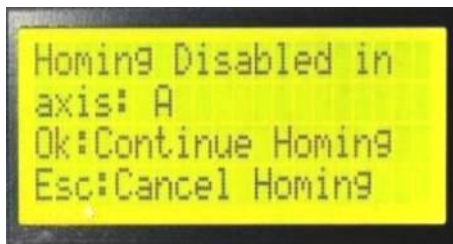
Above message appears for the disabled axis.

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 G-code file. All of this process can be carried out in the 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 G-code 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, the 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 are met satisfactorily.

Condition for resumption:

1. The USB drive must be mounted.

2. A file with the same name must exist in USB drive.

3. The contents of G-code file should not be changed.

**Operations during resumption in Hold Mode:**

For hold mode, the 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, the machine follows the steps given below 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 step number 2 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
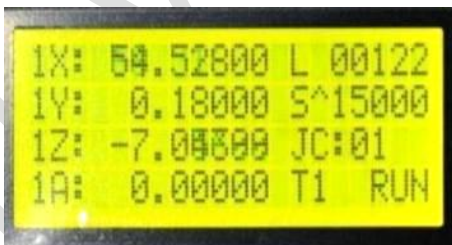
In **idle mode**, pressing key 9 starts execution of selected G-code file. An appropriate message appears upon non-existence of file or USB drive.

The screen shows 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 key 0 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.
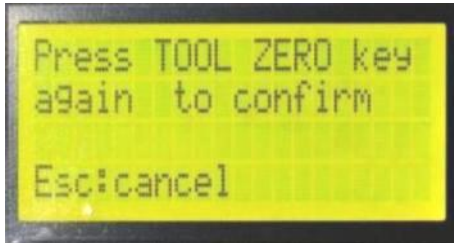
Pressing key 0 sets selected tool to tool zero. The user can press Esc to cancel the tool zero process and stop the tool at that position. The tool does not touch the sensor plate in this condition. The adjacent message appears when tool 0 is pressed.

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

OPERATING THE CNC CONTROLLER

MAIN MENU

When power is applied to the control board, the "ON" status is displayed by the glowing of red LED on the control board. A splash screen appears on the LCD showing version and the product part number.

If the pen drive is not mounted, a message appears as shown by the adjacent screen. Pressing any key shows the splash screen for a second and then shows the coordinate screen. Mount the pen drive at the pen drive slot. The screen displays 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. The adjacent screen appears on pressing key 8.

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

- **General Settings**
- **Active profile**
- **Axes settings**
- **IO settings**
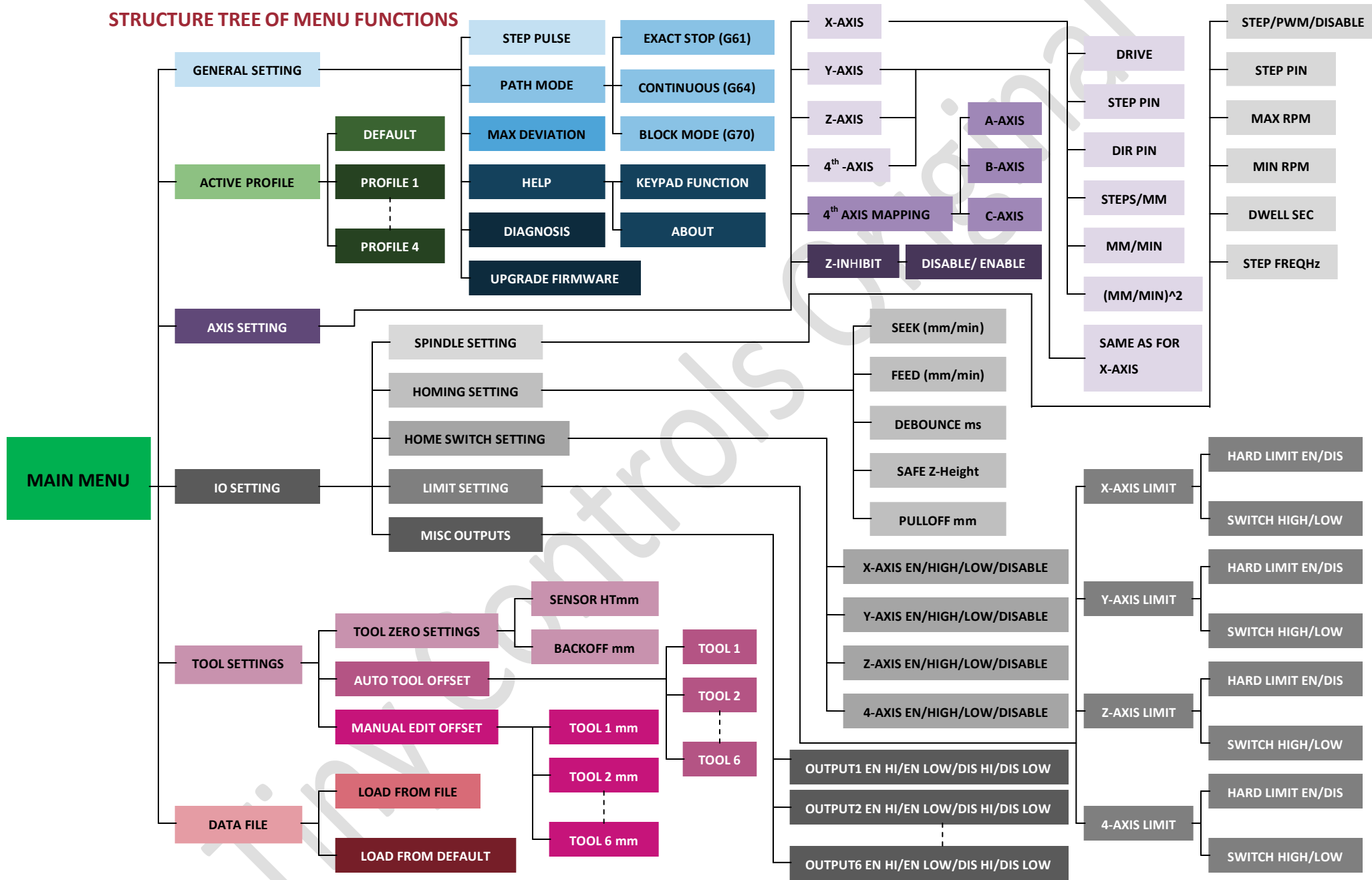- **Tool Settings**
- **Data file**

Note: Be sure about the polarity of external power input.

STRUCTURE TREE OF MENU FUNCTIONS FOR 3-AXES CNC CONTROLLER

MAIN MENU

- GENERAL SETTING
  - STEP PULSE
  - PATH MODE
    - EXACT STOP (G61)
    - CONTINUOUS (G64)
    - BLOCK MODE (G70)
  - MAX DEVIATION
  - HELP
    - KEYPAD FUNCTION
    - ABOUT
  - DIAGNOSIS
  - UPGRADE FIRMWARE

- ACTIVE PROFILE
  - DEFAULT
  - PROFILE 1
  - PROFILE 4

- AXIS SETTING
  - X-AXIS
    - DRIVE
    - STEP PIN
    - DIR PIN
    - STEPS/MM
    - MM/MIN
    - (MM/MIN)^2
  - Y-AXIS
  - Z-AXIS
    - SAME AS FOR X-AXIS
  - Z-INHIBIT
    - DISABLE/ ENABLE

- IO SETTING
  - SPINDLE SETTING
    - DRIVE
      - STEP/PWM/DISABLE
    - STEP PIN
    - MAX RPM
    - MIN RPM
    - DWELL SEC
    - STEP FREQHz
  - HOMING SETTING
    - SEEK (mm/min)
    - FEED (mm/min)
    - DEBOUNCE ms
    - SAFE Z-Height
    - PULLOFF mm
  - HOME SWITCH SETTING
    - X-AXIS EN/HIGH/LOW/DISABLE
    - Y-AXIS EN/HIGH/LOW/DISABLE
    - Z-AXIS EN/HIGH/LOW/DISABLE
    - 4-AXIS EN/HIGH/LOW/DISABLE
  - LIMIT SETTING
    - X-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
    - Y-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
    - Z-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
    - 4-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
  - MISC OUTPUTS
    - OUTPUT1 EN HI/EN LOW/DIS HI/DIS LOW
    - OUTPUT2 EN HI/EN LOW/DIS HI/DIS LOW
    - OUTPUT6 EN HI/EN LOW/DIS HI/DIS LOW

- TOOL SETTINGS
  - TOOL ZERO SETTINGS
    - SENSOR HTmm
    - BACKOFF mm
  - AUTO TOOL OFFSET
    - TOOL 1
    - TOOL 2
    - TOOL 6
  - MANUAL EDIT OFFSET
    - TOOL 1 mm
    - TOOL 2 mm
    - TOOL 6 mm

- DATA FILE
  - LOAD FROM FILE
  - LOAD FROM DEFAULT

45

STRUCTURE TREE OF MENU FUNCTIONS

MAIN MENU

- GENERAL SETTING
  - STEP PULSE
  - PATH MODE
  - MAX DEVIATION
  - HELP
  - DIAGNOSIS
  - UPGRADE FIRMWARE
  - EXACT STOP (G61)
  - CONTINUOUS (G64)
  - BLOCK MODE (G70)
  - KEYPAD FUNCTION
  - ABOUT

- ACTIVE PROFILE
  - DEFAULT
  - PROFILE 1
  - PROFILE 4

- AXIS SETTING
  - X-AXIS
  - Y-AXIS
  - Z-AXIS
  - 4th-AXIS
  - 4th AXIS MAPPING
    - A-AXIS
    - B-AXIS
    - C-AXIS
  - Z-INHIBIT — DISABLE/ ENABLE
  - DRIVE
    - STEP/PWM/DISABLE
    - STEP PIN
    - MAX RPM
    - MIN RPM
    - DWELL SEC
    - STEP FREQHz
  - STEP PIN
  - DIR PIN
  - STEPS/MM
  - MM/MIN
  - (MM/MIN)^2
  - SAME AS FOR X-AXIS

- IO SETTING
  - SPINDLE SETTING
  - HOMING SETTING
    - SEEK (mm/min)
    - FEED (mm/min)
    - DEBOUNCE ms
    - SAFE Z-Height
    - PULLOFF mm
  - HOME SWITCH SETTING
    - X-AXIS EN/HIGH/LOW/DISABLE
    - Y-AXIS EN/HIGH/LOW/DISABLE
    - Z-AXIS EN/HIGH/LOW/DISABLE
    - 4-AXIS EN/HIGH/LOW/DISABLE
  - LIMIT SETTING
    - X-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
    - Y-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
    - Z-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
    - 4-AXIS LIMIT
      - HARD LIMIT EN/DIS
      - SWITCH HIGH/LOW
  - MISC OUTPUTS
    - OUTPUT1 EN HI/EN LOW/DIS HI/DIS LOW
    - OUTPUT2 EN HI/EN LOW/DIS HI/DIS LOW
    - OUTPUT6 EN HI/EN LOW/DIS HI/DIS LOW

- TOOL SETTINGS
  - TOOL ZERO SETTINGS
    - SENSOR HTmm
    - BACKOFF mm
  - AUTO TOOL OFFSET
    - TOOL 1
    - TOOL 2
    - TOOL 6
  - MANUAL EDIT OFFSET
    - TOOL 1 mm
    - TOOL 2 mm
    - TOOL 6 mm

- DATA FILE
  - LOAD FROM FILE
  - LOAD FROM DEFAULT

46

GENERAL SETTINGS

The **GENERAL SETTINGS** function adjusts **step pulse width**, **path mode & max deviate**. **Keypad** function shows the function of all the keys on the pendant in various modes. **Diagnosis** function allows the operator to diagnose inputs and outputs of the CNC controller. **Update firmware** function updates the program.

STEP PULSE

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

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

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

PATH MODE

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

▪ **Exact Stop (G 61):** In exact stop mode, the machine follows a programmed path as exactly as possible, slowing at sharp corners of the path.

▪ **Continuous (G 64):** In continuous mode, sharp corners of a path can be rounded slightly so that speed can be kept up. In continuous mode, control tries to keep velocity constant and does not try to keep controlled point exactly on the path. Rather, at junctures in between motion where direction changes sharply, corner is rounded.

▪ **Block Mode (G 70):** In 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 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 the 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 set deviation factor. The 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. The higher the value of max deviate, the greater is the distance of curve from the junction point.

HELP

Help: The screen shows the keypad function and about function. Their description is given below:

▪ **About:** The **'ABOUT'** screen shows product part number and version of the controller.

▪ **Keypad function:** The **'KEYPAD FUNCTION'** screen shows the keypad function in various modes of the 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: The Diagnosis function allows the user to analyze the proper functioning of inputs and outputs of the CNC controller board.

UPGRADE FIRMWARE

The Update firmware function upgrades the program.

Method to access GENERAL SETTINGS:

a. Enter in **MAIN MENU**.

b. 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**. The above screen appears.

b. Select the **Exact stop** and confirm or cancel the settings to exit to previous menu.

Continuous Mode (G64):

a. Enter in path mode and select Continuous.

b. Confirm or cancel the settings and exit to previous menu.

Block Mode (G70):

a. Enter in path mode and select **Block mode**.

b. Confirm or cancel the settings 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 Keypad function 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 the CNC controller.

b. Press Esc to exit to previous menu.

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

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 processed	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.
	4th-axis outputs	Key 4+	Z-axis moves in 4th + direction.
		Key 4-	Z-axis moves in 4th - direction.
Spindle outputs	Spindle Enable relay	Key 1	Green LED glows for spindle relay, isolated output= +5V
	Spindle Start	Key 4	Spindle starts 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 O/p> +5V
		Down navigation key	Spindle speed decreases, isolated O/p< +5V
Non-Isolated analog output	Non-Isolated analog outputs 1-2	Key 5	+5V at both output terminals.
		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 o/p 1-6	Key 6	Misc O/p 1-6 ON
	Misc o/p 7-14	Key 7	Misc O/p 7-14 ON
	Misc o/p 15-20	Key 8	Misc O/p 15-22 ON

Carefully analyze all the inputs and outputs one by one. Do not forget to save the changes made in settings for each function/menu in the required profile after confirming them; else the controller retains the previous settings. All the outputs of the control board can be set OFF by pressing key 0. Outputs cannot be set OFF

individually. Press Esc to exit. A message for "**Diagnosis complete: System Reboot"** appears.

Method to access UPGRADE FIRMWARE menu:

a. Enter in Upgrade Firmware submenu from General settings menu.

b. Press OK key to upgrade the firmware, else press Esc key.

c. On pressing the Ok Key to upgrade the firmware, a message **"Attach/Re-insert USB drive"** appears if the pen drive is not connected. The pen drive must contain the file **Download.enc** which is used to upgrade the program. Press Ok after inserting pen drive for up-gradation process, else press Esc key.

d. However, if the pen drive is already connected, a message **"Long Press External Start/ Stop"** appears. Press Ok key to confirm the up-gradation process, else press Esc key. Ground the start and stop pin of the controller board.

e. Message of **"Updating Firmware"** appears on the screen when the process starts. Keep the start-stop terminal connected to ground for 2 seconds. Red LED glows on the board when the updating process starts and it continues to glow until the process completes itself.

f. When the process is completed, Red LED does not glow. Remove the connection of start stop and ground terminals. All the system gets reset after up-grading the firmware. Default values get loaded and on pressing Esc key, new version will be reflected on the LCD.

⊂Tip: If file download.enc isn't available in the pen drive, a message of "Download.enc file not found" appears on the board.

ACTIVE PROFILE

ACTIVE PROFILE menu allows 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 the 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 on the CNC controller and a default profile is given.

All user programmable profiles work the same as the 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 profiles. 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 cannot 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 the settings.

AXIS SETTINGS

AXIS SETTINGS menu functions allow the user to modify settings of the axes parameters in profiles. The CNC controller controls an independent mechanism of the machine which produces relative linear motion of tool and work piece in three mutually orthogonal directions called X, Y and Z axes.

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

X-AXIS (OR Y-AXIS, Z-AXIS AND 4TH AXIS)

The functioning of the 3 linear axes depends on some parameters. Make the 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 the 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 the 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: The **DIR PIN** selects the state of direction signal given to the drive of the motor attached to the axis. All axes have their dedicated direction output pins on the control board and axes can have different direction settings in accordance to the job to be performed. When this pin is in active high state, the machine/tool head moves in positive or negative direction according to the commands used in G-code file or by jog keys.

And when pin is active low, machine/tool head moves in the opposite direction to 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.

Step/MM: **STEPS/MM** shows the number of steps that the motor must turn in order to allow the CNC machine to move 1 mm on a particular axis. This is machine dependent and remains the same for a particular axis of the 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:

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

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

MM/sec²: It is the increment of processing velocity from initial velocity to highest one for a motor connected to an 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.

The user needs to 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. This option is unavailable in 3 axes CNC controller system.

Z-INHIBIT

The Z-inhibit function allows the user to inhibit Z axis. The 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 are set enabled.

Dicussed below is the way to access the Axes setting:

1. Enter in Main Menu and select 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 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 Axes Settings for the rest of the 3 axes.
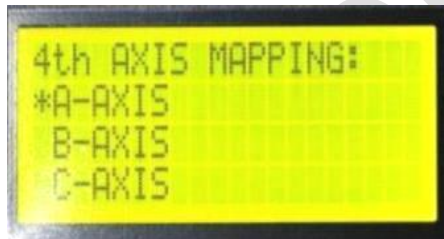
For 4th Axis Mapping:

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 the CNC Controller and the CNC machine by making settings for limit and home inputs and for spindle outputs. This function makes settings for Home and Limit switches and also for buffered Misc outputs. The functions under IO setting menu are discussed next:

SPINDLE SETTING

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

▪ **Spindle (Step/ PWM/Disable):** The Spindle Step/ PWM/Disable function allows the operator to drive Step/PWM signal to Spindle drive or disable spindle by giving respective signal to spindle drive. The spindle rotates if spindle drive receives Step or PWM signal (as selected for application) and spindle is set as ON in G-code file. If the spindle is disabled in IO setting and is set as ON in G-code file, then the spindle does not work. **"Spindle Step/ PWM/ Disable"** is the relay output from the 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 the spindle drive. This pin, if selected as active high, drives an active high pulse and if selected as active low, drives an active low signal to the spindle drive. This output pin gets disabled if Spindle is set disabled.

▪ **Max RPM:** The Max RPM refers to the maximum number of revolutions per minute that the spindle rotates. The 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 the minimum number of revolutions per minute that the spindle rotates. The 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. The spindle remains idle for this duration of time. It is user programmable and is set in seconds.

▪ **Freq Hz:** It decides frequency of step pulse or PWM signal at step pin output on the CNC controller board. If Spindle PWM is selected, it shows the frequency set for PWM. By default, it shows the frequency for Spindle Step function. By default, frequency is 1 KHz for both Step and PWM. The minimum value of this function is 45 Hz and the maximum value for this function is 60 KHz.

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 "PWM/ Step/ Disable".

c. Select Step Pin and set 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 **FREQ Hz** and modify the values as per the value at which the drive is set at PWM or Step.

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 is skipped for any axis. In such cases, the machine should be positioned at the known coordinates. For this purpose, home switches should be installed for all axes on every machine.

For homing, the CNC controller follows a sequence of cycles before reaching to machine zero. In first cycle, the Z-axis moves upward till finding and triggering Z-axis home switch. In second cycle, the rest of the three axes move together in an

independent way and trigger their home switches. In both cycles, they move at set "**homing seek rate"**. In proximity of all the home switches, the 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, the machine moves to a position determined by set "**pull off distance"** away from machine zero location for all enabled axes in a coordinated way at homing seek rate. Finally the 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 the 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 the 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 the size of the machine.

▪ **Safe Z-Height:** This function ensures the safety of the machine for which Home switch for Z-axis has either 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. The maximum value of debounce delay is 1 second. The units for debounce delay are ms.

Here is the method to access Homing Setting:

a. Select and enter in Homing Setting.

b. Select Seek mm/min and edit its value.

c. Select Feed mm/min and modify its value.

d. Select Pull Off mm and modify its value.

e. Select Safe Z-Height and edit its value.

f. Select Debounce us and edit its value.

g. 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 of axes for which switches are not installed. An input signal is driven from the home switch to its axis Home-In terminal on the CNC control board when it triggers.

When the machine is in Run/Jog mode and Home switch for an axis triggers, an input is generated to the axis Home IN pin and the motion of an entire axes system ceases. A message of alarm for **Hard Home/Limit** is dispalyed on the LCD screen. Press Ok key to continue. None of the jog key works and the 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 disbaled except the jog and navigation keys in manual override mode. Jog the machine reverse using jog keys to release the switch and sets the machine at required position. In any situation, if the limit/home switch triggers and hard limit/home is not checked then the user needs to jog the machine reverse. Press left navigation key long again to exit from manual override mode.

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

▪ **Enable:** Set the home switch enabled for the respective axis**.** It allows for 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 is triggered. Active low sets the home switch to generate an active high pulse when the switch is triggered.

▪ **Disable:** Set Home switch disabled for the respective axis. Homing of the axis ceases when key 7 is pressed long. However if the Z-axis set is disabled, moves to Z-safe height (discussed in Homing setting function) when homing command is given.

Access the Home switch settings are given next:
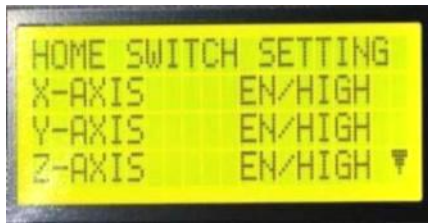
a. Select and enter in Home Switch Setting function in IO Setting Menu.

b. Enable or disable the home switch for an axis as per requirement. Set the switch high or low if enabled. Modify 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 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 the Limit switch for an axis is 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 dispalyed on the LCD screen. Press Ok key to continue. All the jog key stops working and the entire motion system stops in this condition. Press the left navigation key long to enter in manual override mode and to relaese the switch.

All the keys are disbaled 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. In any situation, if the limit/home switch is triggered and hard limit/home is not checked, then the user needs 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 do not have the home switches. For all the three axes, it is required to program the following settings for Limit switches.

Hard Limit Enable/Disable: Hard limit if enabled generates the alarm message when hard limit switch is triggered. Hard limit if disabled does not generate any message when the limit switch is triggered. However the input pulse is driven to the axis Limit-IN pin of the CNC control board when the switch is triggered and the machine gets blocked in the same way.

Hard Limit Switch (Active High): Active high sets Limit switch to generate an active low pulse to axis Limit-In on the CNC control board when the switch is 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 is triggered.

Steps for Limit Setting:
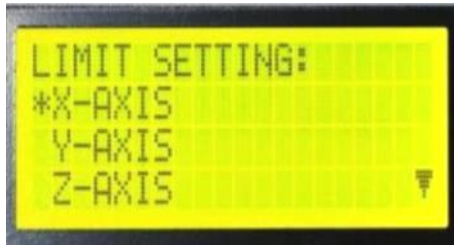
a. Select and enter in the **LIMIT SETTING** in **IO SETTING**.

b. Select the required axis. For example, select **X-axis** and enter.

c. Select **HARD LIMIT** and enable or disable it.

d. Select the **SWITCH** and set it as high or low.

e. Confirm or cancel the settings made and Exit to previous menu.

MISC OUTPUTS

Misc Outputs are miscellaneous outputs and are 20 in number. Misc outputs 1-4 and 13-20 are buffered outputs and 5-12 are open collector outputs. The user can set them as ON/OFF in G-code only. All these outputs can be set as OFF collectively but are made ON individually in G-code 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 does not give any output on the output pin even when set as ON in G-code file. Misc outputs 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 set as ON in G-code.

▪ **En/LOW:** If output is set enable low, the output pin generates an active low pulse if output is set as ON in G-code.

▪ **DIS/HI:** If output is set Disable high, the output pin does not generate any output pulse even set as ON in G-code 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 set as ON in G-code file. However, the pin state is low in this condition.

Here is discussed method to use 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. Follow the same procedure for the 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 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 the system.

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, the tool starts moving downward along Z-axis with set "**homing feed rate**" and tries to find out where the sensor metal plate is kept on the top of work piece. As it touches the top of sensor plate, the 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, the controller moves the tool upward to a set **Back off Distance**.

At the moment that the tool touches the sensor metal plate, the controller sets Z=0 at the top of work piece and updates current working co-ordinate system as well as DRO; for the reference tool (tool0). For this purpose, the **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, the 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 cannot be modified, not even manually. All these highlighted parameters which can be set in menu are discussed below:

TOOL ZERO SETTINGS

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

SENSOR HT мм: Height of sensor metal plate can be set in mm.

BACKOFF мм: This is the back off distance or the distance that the machine moves back after touching the sensor plate at rapid rate. This is the value that DRO shows when the 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 HT mm**" 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, the Tool No. 0 is taken as the reference tool and it is set for tool zero position. Now when the user replaces the Tool No. 0 with Tool No. 1 which is 2mm longer than Tool No. 0 then the 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 between the reference tool length and selected tool length automatically. The user needs 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, the operator has to calculate the offset function between tools manually and input that value in this function. First, tool zero for the reference tool is set and then offset value for selected tool is calculated by the user. The 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.

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

DATA FILE

The 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 in this section.

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 cannot be altered. Given below is the method to be followed to perform these operations:

LOAD FROM FILE: Load from file menu allows loading settings to the respective profiles from the 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 load from the 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 the manufacturer has the authority to 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:** Step
- **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

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

▪ Number of words, parameter settings and comments.

▪ At the end of line marker (carriage return or line feed or both). Any other inputs not explicitly allowed are illegal.

▪ Spaces and tabs are allowed anywhere on a line of code and does not change the meaning of the line, except in inside comments.

▪ Blank lines are allowed in 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 and then many digits, followed, possibly, by one decimal point, followed by zero and then 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 read as 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 form of quotation either in single quotes or in double quotes. For example:

M75 'tiny controls'

M75 "tiny controls"

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

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

2. What happens on pressing the Emergency button?

When the emergency button is pressed, the spindle and machine axes 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 the CNC controller and the handheld pendant is made through serial cable. Occasionally, the 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 the 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 by unplugging the serial cable from the side of the control board and then plugging it back.

▪ If the above process doesn't work then the problem may be that the connectors have been damaged or chip for Rs-232 has failed and needs to be replaced.

▪ The internal configuration of the 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 file names are not supported.

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

When the pen drive is 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.

The same process will take place if the emergency button is pressed.

5. What happens if the connection between the pendant and the controller is lost while executing on of G-code file?

If connection between the pendant and the 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 on 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 executes 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. It is 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 work piece.

G-code: A program code that determines the type of operation performed on the machine. The G codes used for the 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: The 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.

Mcode: 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 puts the machine into pause mode where an action is momentarily stopped until the machine resumes.

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

Relative coordinate system: This 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.

Sensor plate: The sensor plate is a device that is placed on the top of the work piece. It is of known thickness and on any 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 it is 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

USER NOTES