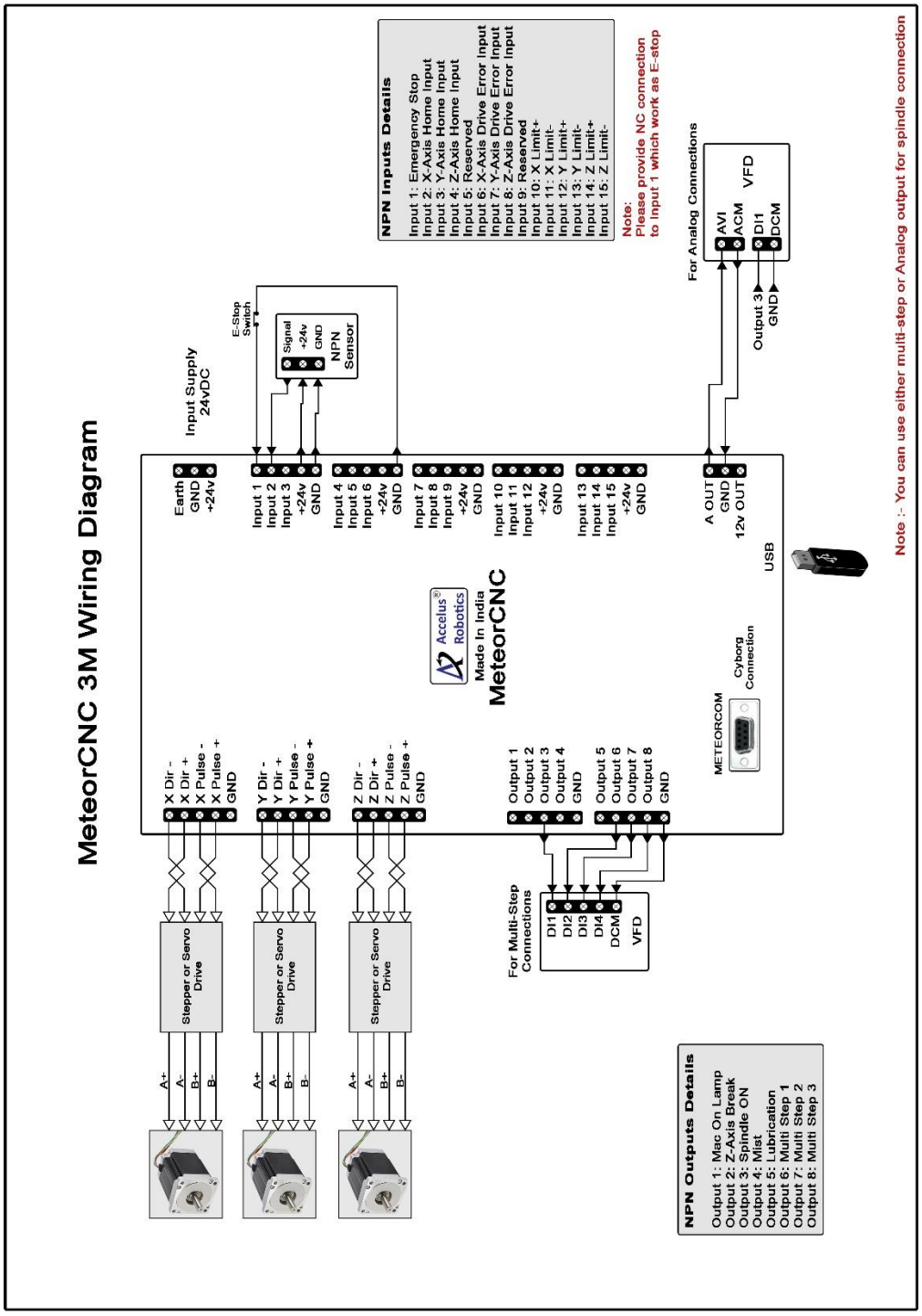
MeteorCNC
Manual V2.0

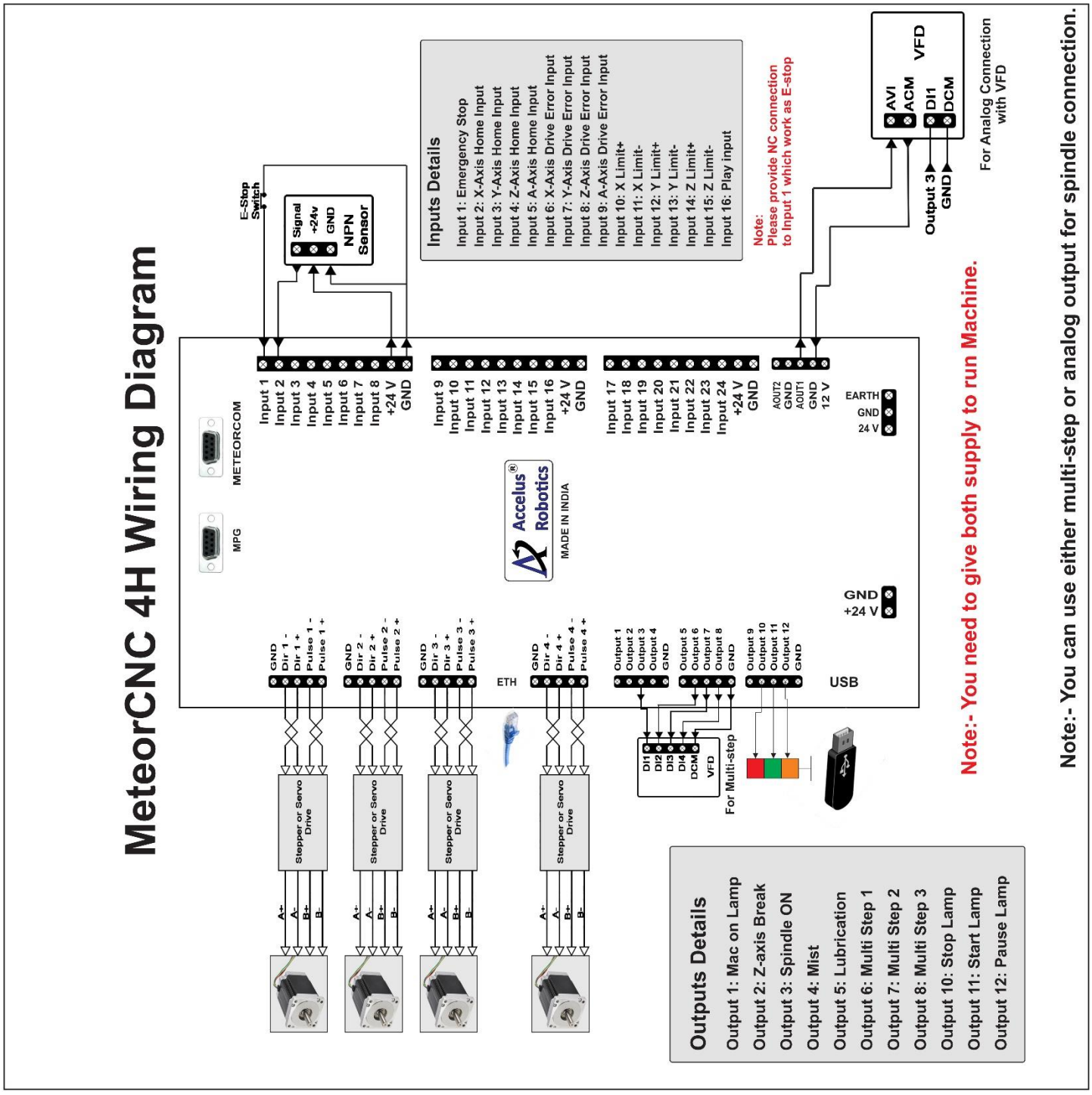
Index

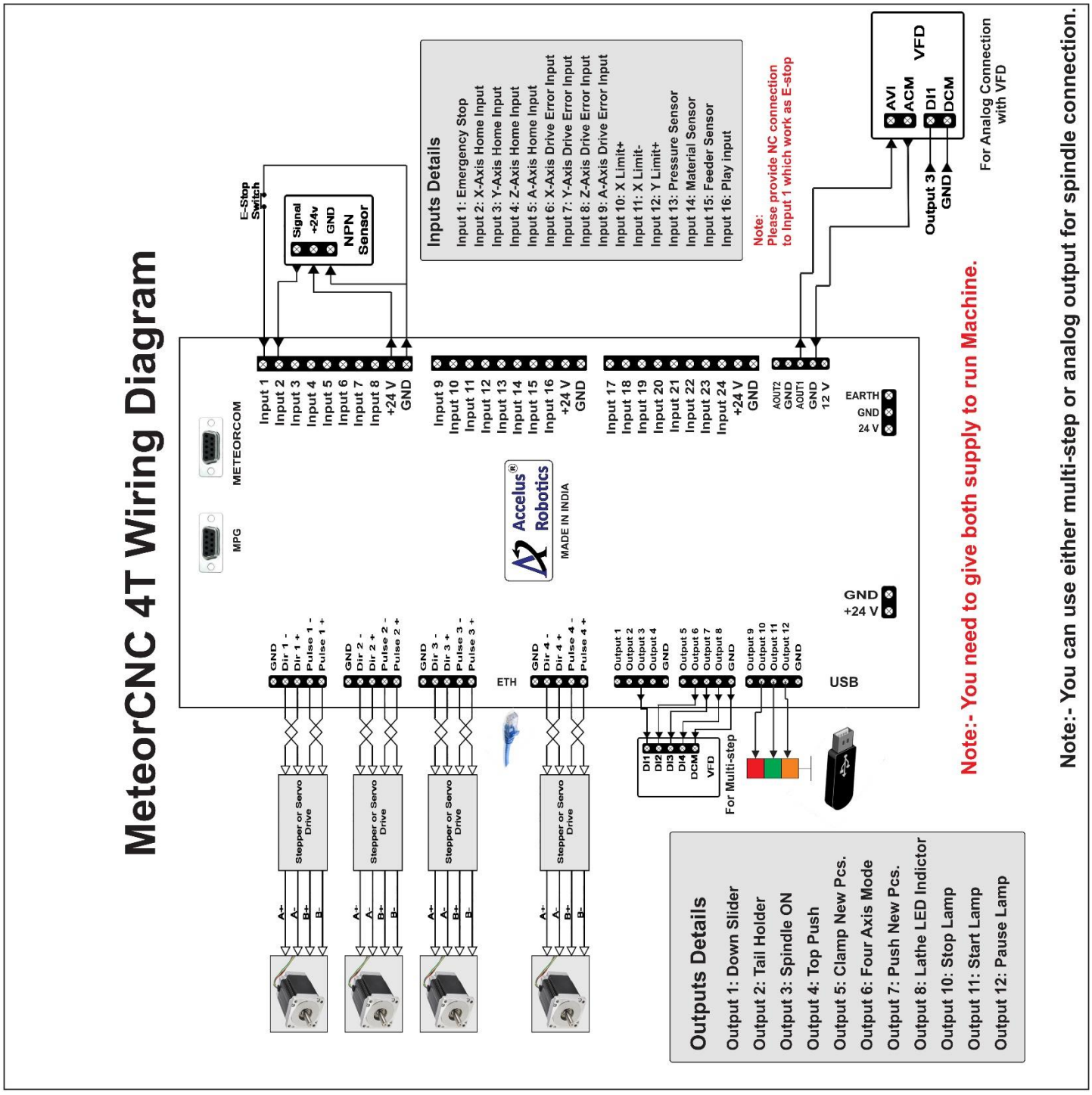
Sr.	Topic	Page Number
1	Wiring Diagram	3
2	Menu Systems	6
3	Keypad Description	15
4	Turmet (Lathe)	19

Chapter 1

Wiring Diagram




MeteorCNC 4H Wiring Diagram
E-Stop Switch
Signal
+24v
GND
NPN Sensor
Inputs Details
Input 1: Emergency Stop
Input 2: X-Axis Home Input
Input 3: Y-Axis Home Input
Input 4: Z-Axis Home Input
Input 5: A-Axis Home Input
Input 6: X-Axis Drive Error Input
Input 7: Y-Axis Drive Error Input
Input 8: Z-Axis Drive Error Input
Input 9: A-Axis Drive Error Input
Input 10: X Limit+
Input 11: X Limit-
Input 12: Y Limit+
Input 13: Y Limit-
Input 14: Z Limit+
Input 15: Z Limit-
Input 16: Play input
Note:
Please provide NC connection
to Input 1 which work as E-stop
AVI
ACM
VFD
DI1
DCM
Output 3
GND
For Analog Connection
with VFD
METEORCOM
MPG
Input 1
Input 2
Input 3
Input 4
Input 5
Input 6
Input 7
Input 8
+24 V
GND
Input 9
Input 10
Input 11
Input 12
Input 13
Input 14
Input 15
Input 16
+24 V
GND
Input 17
Input 18
Input 19
Input 20
Input 21
Input 22
Input 23
Input 24
+24 V
GND
AOUT2
GND
AOUT1
GND
12 V
EARTH
GND
24 V
Accelus® Robotics
MADE IN INDIA
GND
+24 V
ETH
USB
GND
Dir 1 -
Dir 1 +
Pulse 1 -
Pulse 1 +
GND
Dir 2 -
Dir 2 +
Pulse 2 -
Pulse 2 +
GND
Dir 3 -
Dir 3 +
Pulse 3 -
Pulse 3 +
GND
Dir 4 -
Dir 4 +
Pulse 4 -
Pulse 4 +
Output 1
Output 2
Output 3
Output 4
GND
Output 5
Output 6
Output 7
Output 8
GND
Output 9
Output 10
Output 11
Output 12
GND
Stepper or Servo Drive
Stepper or Servo Drive
Stepper or Servo Drive
Stepper or Servo Drive
A+
A-
B+
B-
DI1
DI2
DI3
DI4
DCM
VFD
For Multi-step
Note:- You need to give both supply to run Machine.
Outputs Details
Output 1: Mac on Lamp
Output 2: Z-axis Break
Output 3: Spindle ON
Output 4: Mist
Output 5: Lubrication
Output 6: Multi Step 1
Output 7: Multi Step 2
Output 8: Multi Step 3
Output 10: Stop Lamp
Output 11: Start Lamp
Output 12: Pause Lamp
Note:- You can use either multi-step or analog output for spindle connection.


MeteorCNC 4T Wiring Diagram
E-Stop Switch
Signal
+24v
GND
NPN Sensor
Inputs Details
Input 1: Emergency Stop
Input 2: X-Axis Home Input
Input 3: Y-Axis Home Input
Input 4: Z-Axis Home Input
Input 5: A-Axis Home Input
Input 6: X-Axis Drive Error Input
Input 7: Y-Axis Drive Error Input
Input 8: Z-Axis Drive Error Input
Input 9: A-Axis Drive Error Input
Input 10: X Limit+
Input 11: X Limit-
Input 12: Y Limit+
Input 13: Pressure Sensor
Input 14: Material Sensor
Input 15: Feeder Sensor
Input 16: Play input
Note:
Please provide NC connection
to Input 1 which work as E-stop
AVI
ACM
VFD
DI1
DCM
Output 3
GND
For Analog Connection
with VFD
METEORCOM
MPG
Input 1
Input 2
Input 3
Input 4
Input 5
Input 6
Input 7
Input 8
+24 V
GND
Input 9
Input 10
Input 11
Input 12
Input 13
Input 14
Input 15
Input 16
+24 V
GND
Input 17
Input 18
Input 19
Input 20
Input 21
Input 22
Input 23
Input 24
+24 V
GND
AOUT2
GND
AOUT1
GND
12 V
EARTH
GND
24 V
Accelus® Robotics
MADE IN INDIA
GND
+24 V
ETH
USB
GND
Dir 1 -
Dir 1 +
Pulse 1 -
Pulse 1 +
GND
Dir 2 -
Dir 2 +
Pulse 2 -
Pulse 2 +
GND
Dir 3 -
Dir 3 +
Pulse 3 -
Pulse 3 +
GND
Dir 4 -
Dir 4 +
Pulse 4 -
Pulse 4 +
Output 1
Output 2
Output 3
Output 4
GND
Output 5
Output 6
Output 7
Output 8
GND
Output 9
Output 10
Output 11
Output 12
GND
Stepper or Servo Drive
A+
A-
B+
B-
DI1
DI2
DI3
DI4
DCM
VFD
For Multi-step
Outputs Details
Output 1: Down Slider
Output 2: Tail Holder
Output 3: Spindle ON
Output 4: Top Push
Output 5: Clamp New Pcs.
Output 6: Four Axis Mode
Output 7: Push New Pcs.
Output 8: Lathe LED Indictor
Output 10: Stop Lamp
Output 11: Start Lamp
Output 12: Pause Lamp
Note:- You need to give both supply to run Machine.
Note:- You can use either multi-step or analog output for spindle connection.

Chapter 2

Menu Systems

1. **MeteorCNC**

   It is showing the machine status. With position of all axis.

2. **Operations**
   1. Home Now
      i. Home all : Homing of all axis will be done on the basis of Homing Sequence set in machine setup.
      ii. Home Z : Homing of Z-axis
      iii. Home XY : Homing of XY-axis
      iv. Home X : Homing of X axis.
      v. Home Y : Homing of Y axis.
      vi. Home A : Homing of A axis.

      Note : Shortcut of this screen is **Shift + Number 5**.

   2. Int. Memory
      In this program files are stored. Format can be .nc , .ngc , .tap. if controller is 4T series then user can store .dxf file also.

   3. USB Files
      In this user can see the file from inserted USB pen drive. User can copy .nc,.ngc,.tap & .dxf files from pen drive & paste it in internal memory of controller.
      Note : Use FAT32 or NTFS USB

   4. Goto WCS org
      i. Goto WCS all : This will send X ,Y, Z & A to origin of work Co-ordinate systems (WCS).
      ii. Goto WCS X : This will Send X-axis to origin of WCS.
      iii. Goto WCS Y : This will Send Y-axis to origin of WCS.
      iv. Goto WCS Z : This will Send Z-axis to origin of WCS.
      v. Goto WCS A : This will Send A-axis to origin of WCS.

   5. Goto MCS org
      i. Goto MCS all : This will send X, Y, Z & A to origin of Machine Co-ordinate systems (MCS).
      ii. Goto MCS X : This will send X-axis to origin of MCS
      iii. Goto MCS Y : This will send Y-axis to origin of MCS
      iv. Goto MCS Z : This will send Z-axis to origin of MCS
      v. Goto MCS A : This will send A-axis to origin of MCS

6. Select WCS
   User can Select total 9 WCS (G54 , G55 , G56 , G57 , G58 ,G59 , G59.1 ,G59.2 ,G59.3)

7. OTA USB Files
   This Function can be used for update the controller. User need to insert the pen drive to controller with latest file version & follow the instruction given on screen.

3. Brakept Rstr (BreakPoint Restore)

This functionality used to saved line number & axis position when power goes off. MeteorCNC will store the data in following conditions

- During power failure in between running the program.
- User stop the design with stop button from hand handle or By pressing Emergency Stop (Input 1).

Note : Shortcut : **Shift + Enter**

4. Job Setup

1. Z safe height
   In this , Z-axis will move to safe position on Go to Z=0.

2. Park Position
   i. Park On /Off : By turning ON this function, machine's all Axis will go to defined parked position after design end.
   ii. Set MCS : It will set MCS to machine park position.
   iii. Park Position : machine will parked at given park position of X,Y,Z & A. on display it will show MCS value only.

3. Shift offset
   i. X work shift : This will shift X axis work offset & its unit in mm.
   ii. Y work shift : This will shift Y axis work offset & its unit in mm.
   iii. Z work shift : This will shift Z axis work offset & its unit in mm.
   iv. A work shift : This will shift A axis work offset & its unit in mm.

4. Jog Speed & Distance
   i. Step Distance : User can defined Step Distance of X,Y,Z & A axis.
   ii. Jog speed : User can define High & Low Jog speed as well as Z feed percentage.

5. Mirror G-code
   This will make mirror of X-axis or Y-axis G-code. This can be set by user.

6. Local Speed
   i. Control feed : This will bypass the program feed (F).
   ii. Control speed : This will bypass the program's spindle speed (S).

iii. Local speed : In this user can define feed & spindle speed.

7. Spindle Setup
   i. Spindle Speed : User can define spindle speed.
   ii. Delay Time : Spindle will take define delay time to reach its desired speed in design play. Program will halt during this operation.
   iii. Max.Speed : This is the max RPM at which spindle will run.

8. Tool setup : User can Set up to 9 tools. User can define X,Y,Z offset & Tool Diameter also.

9. Tool Select : User can Select up to 9 tools as per requirement.

10. Touch probe

    i. Prob Pos : User can define Position of probe (X ,Y & Z position where probe is situated). This value is from MCS i.e. from Home sensors.
    ii. Start probe : User can start the probing using this option. (Shift + Tool)
    iii. Probe Only Z : This option allow only Z-axis probing operation.

    Note : To start the probe operation Z-axis negative limit should be -1000 or more.

    Probe Operation Explanation.

    1. In probe Operation only MCS values are coming in count.
    2. **Tool 0,** is not applicable for Probe operation.
    3. User have two option for probing operation
       i. Operation 1: Only Z-axis probing.
       This kind of operation is running when probe switch is not fixed at one position.
       select **probe only Z** option & turn it ON then start probe operation. This will move Z-axis till probe input get turn On & again it will go back to the sensor.

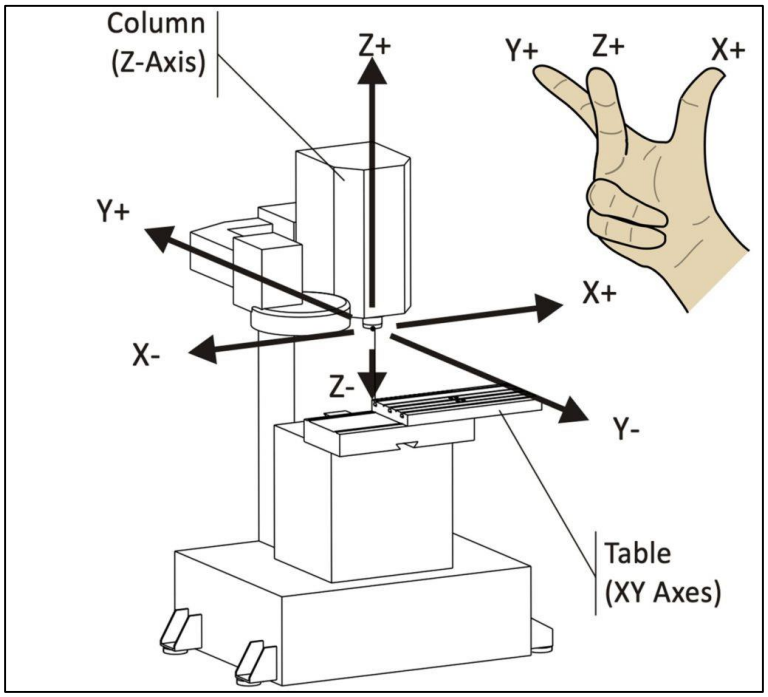
       e.g. if probe switch height is 5 mm & Z-axis is coming down upto 80 mm & Tool 1 is selected then its showing 85mm in start probe option for Z-axis & In Tool 1 Z-axis offset value will be -85 mm.

       ii. Operation 2: X ,Y,Z probing
       This kind of operation is running when probe switch is fixed.
       e.g. In Prob pos ,X=100 , Y=200 & Z=5.
       These defined position is probe switch position from Homing Sensor. When user perform probe operation that time it will travel (X, Y) distance & Z-axis will travel until it gets input from probe. In start probe option it will show position w.r.t. set zero position (WCS).

11. Play Timer
    During play time, On main screen user can show either file name or Design running time or both.
    When user select both file name & timer that time Timer will show for 30 sec & for 2 sec file name will display.

12. Display Feed
    This option will allow user to show running feed or Feed override on the main screen.

5. Machine Setup



An Easy way to understand the Co-ordinate system in relation to CNC is using Right Hand Rule(RHR).
RHR : Hold your hand out palm up with your thumb & index finger pointed outwards & your middle finger pointed upwards. Place your hand in front of CNC machine, aligned with the machine's spindle & you will see the axes line up perfectly.

When user enter in this menu, system will ask Password for further access. User can ask the password to manufacturer or distributer or support service.

1. Axis Setup
   i. Pulse / mm
   This is scaling factor of number of pulses to be generated by the MeteorCNC to move axis by 1mm. please refer below examples for better understanding.

   Eg.1 :
   Drive PPR : 4000 & Ball screw pitch : 5 mm
   Pulse/mm (scale) = Drive PPR / Ball screw pitch = 4000/5 = 800

   Eg.2 :
   Drive PPR : 4000 & Pinion Dia : 10
   Pulse / mm (scale) = Drive PPR / (pi*pinion dia) = 4000 / (3.14*10)
   =127.3888

   Consider gear box of 10 is connected after above mechanism than
   formula will be
   Pulse / mm (scale) = [Drive PPR / (pi*pinion dia)] * gear box

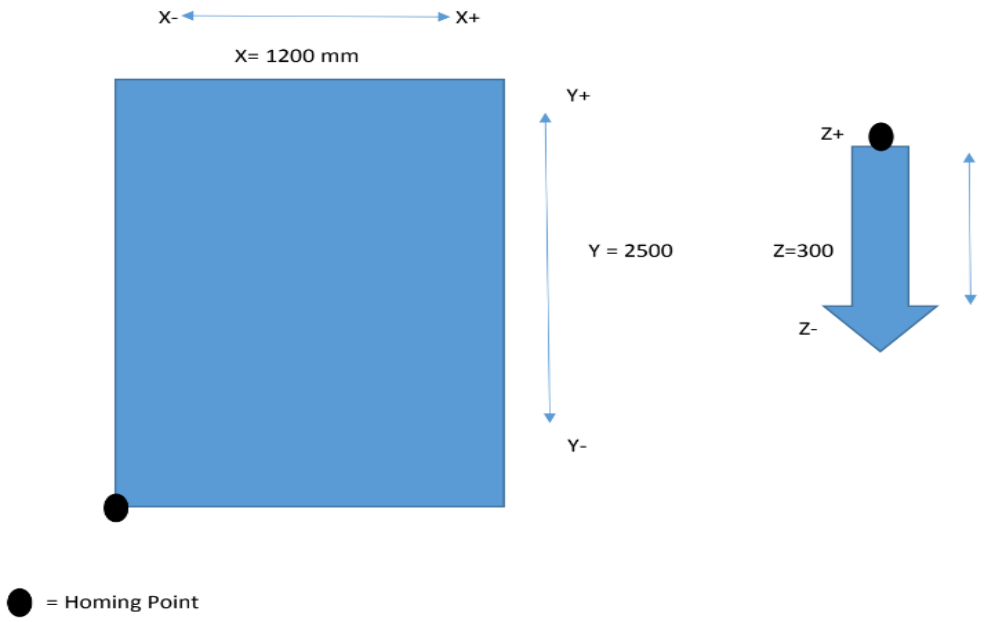
   = [4000/3.14*10] * 10
   = 1273.88

   ii. Backlash
   Backlash compensation value cab be used to make up for small deficiencies in the hardware used to drive an axis.

   iii. Pos Soft Limit (Positive Soft Limit)
   The maximum positive soft limit for axis motion, when this limit is exceeded the controller aborted the operation.

   iv. Neg Soft Limit (Negative Soft Limit)
   The maximum negative soft limit for axis motion, when this limit is exceeded the controller aborted the operation.

Read the following example for better understanding



Consider X & Y homing is done at extreme negative position as shown in above image. To utilize full bed area set X pos limit : 1200 mm & X neg limit : -1. Same for Y axis, set Y pos limit :2500 & Y neg limit : -1. & Z pos lim : 1 & Z neg lim : -300

Note : X & Y value can be vary as per bed size.

v. Maximum Acceleration (mm/s2)
Maximum acceleration for this axis is machine per unit second squared. Set the acceleration According to machine capacity & user requirement.

vi. Maximum Speed
Maximum velocity for this axis in machine unit per second.

For better understanding refer the example

E.g.1
Motor Max Speed : 3000 RPM
Pitch : 5 mm
Motor Max Speed (rps) = 3000/ 60 = 50 rps

So , Axis Max Speed (rps) = 50 x 5 = 250 rps

Axis Max Speed (rpm) = 250 x 60 = 15,000 rpm

E.g. 2
Motor Max Speed : 1200 RPM
Pinion Dia (D): 5 mm
Motor Max Speed (rps) = 1200/ 60 = 20 rps

So. Axis Max Speed (rps) = 20 x pi x 5 = 251.2 rps
Axis Max Speed (rpm) = 251.2 x 60 = 15,072 rpm

vii. Homing Speed
Initial homing velocity in machine per unit minute. sign denotes the direction of travel. A value '0' means assume that the current location is home position for the machine.

viii. Homing Sense Speed
Initial homing velocity in machine per unit minute. sign denotes the direction of travel. A value '0' means assume that the current location is home position for the machine.

ix. Homing Sequence
This function is used to defined Homing sequence of the axes. When you pressed "Home all " Command according to sequence, axis will do homing. If user set -1 in sequence then then that axis will be skipped from homing process.

x. No Force Home : By using this option user can start operation without doing homing.

Default parameter Setting For MeterCNC is given in below Table

	X- Axis		Y-Axis		Z-Axis	
	Stepper Motor	Servo Motor	Stepper Motor	Servo Motor	Stepper Motor	Servo Motor
Pulse /mm						
Pos soft lim						
Neg soft lim						
Max.Acc	500	1500	500	1500	400	1000
Max.Spd	12000	25000	12000	25000	6000	10000
Homing Spd	-3600	-3600	-3600	-3600	1800	1800
Homing Sens	60	60	60	60	-60	-60
Homing Seq	1	1	1	1	0	0

Note : pulse / mm & limits values depends on mechanical & calculation for same is given in above explanation.

2. Override Setup
   i. Max Feed Override : This maximum feed override the user may select.
   e.g. if user is selecting 1.2 means 120% of program's feed rate.
   ii. Max spindle Override : The maximum spindle override the user may select.
   e.g. if user is selecting 1.0 means 100% of the program's spindle speed.
   iii. Min spindle Override : The minimum spindle override the user may select.
   e.g. if user is selecting 0.1 means 50% of the program's spindle speed.

3. Feed Override Increment
   User can define Increment value of feed.

4. Set rapid Override
   This is percentage value of G0.

5. Startup Home
   This function will give you option that for startup homing screen.

6. Lubrication
   i. On Delay : This will keep lubrication ON for specified time in seconds during cycle run.
   ii. Off Delay : This will keep lubrication OFF for specified time in seconds during cycle run.

7. Change Password
   User can change the machine setup password if he knows the old password.

6. Diagnosis

1. Version : This is showing Firmware version of Controller.

2. Pendent Version : This is showing Firmware version of handhandle.

3. Keypad check : This is used for checking Handhandle's key.

4. Shortcut keys

Shortcut keys	Functions
Shift + Enter	Breakpoint Restore
Shift + Esc	Reset
Shift + Down	Feed
Shift + 0	Set XYA=0
Shift + 4	Jog Speed
Shift + 6	Mirror
Shift + 8	Select WCS
Shift + dot	Goto Z=0

Shift + Menu	Error Screen
Shift + 5	Homing
Shift + File	MPG
Shift + Up	Feed 100
Shift + Spindle 100	Spindle 100

5. Machine Status
   This is used to check controller's status. User can check status of Emergency Stop,Machine Status, current mode,WCS,Tool Number,Tool offset,Turmet.

6. G52 offset
   G52 offset of X,Y,Z axis for selected WCS.

7. G43 offset
   Length offset of X ,Y ,Z axis.

8. Axes Frequency
   It will shows the frequency of X,Y,Z, A axis.

9. Running Error
   It is showing error list.it will show upto 15 errors.

10. Disk space
    It is showing total space in the controller.

11. Clear Log
    This will clear the log of controller

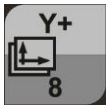
12. Input
    It is showing status of inputs.

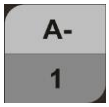
13. Output
    By using this user can check the controller's outputs.

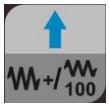
14. Unlock cycles
    After finishing trial cycles , user can ask for further cycles from dealer or manufacturer.User can see controller number in this screen.

Chapter 3

Keypad Description

No.	Key Icon	Key Name	Key Description
1		Menu / Error Screen	Entering this key will show you Menu of the system. By entering Shift + key shows error screen
2		File / MPG	Entering File key will show you programs stored in Internal memory of controller. By Entering Shift + MPG will enable the MPG Operations.
3		X+ / Number 6 / Mirror G-code	X-axis Positive Jog. Input Number 6. Entering Shift + key Shows G-code mirror options.
4		X- / Number 4 / Jog Speed	X-axis Negative Jog. Input number 4. Entering Shift + key show jog speed & distance option.
5		Y+ / Number 8 / Work Co-ordinates	Y-axis Positive Jog. Input number 8. Entering Shift + key will change work co-ordinate systems.
6		Y- / Number 2	Y-axis Negative Jog. Input Number 2

8		Z- / Number 3	Z-axis Negative Jog. Input Number 3
9		A + / Number 7	A-axis Positive Jog. Input Number 7.
10		A - / Number 1	A-axis Negative Jog. Input Number 1.
11		Jog Speed / Number 5 / Home Screen	It will change the Jog Speed. Input Number 5. Entering Shift + key shows Homing Screen.
12		Stop / Back	This key will stop the running program. This key will use as back option.
13		Play / Pause / Enter	This key will play the program. This key will pause the running program. This key will use as Enter & will ensure the value as well as it will use to edit the parameters.
14		Tool Selection / Touch Probe	This key will show you the tool list. Entering Shift + Key shows probe options
15		Spindle Override + / Spindle Override 100%	This will increase the Spindle Override. Entering Shift + key will set the Spindle override 100%

16		Spindle Override - / Auto Tool Changer	This will decrease the spindle override. Entering Shift + Key shows Auto tool Changer.
17		Shift	Auxiliary Key
18		Spindle Start & Stop	By Entering this key Spindle start & stop.
19		Up key / Feed Override + / Feed Override 100%	Up arrow key for menu & file browsing. Feed override increment. Entering Shift + key will set the Feed override 100%
20		Down key / Feed Override -	Down arrow key for menu & file browsing. Feed override decrement.
21		Go to XYA = 0 / Go to Z=0 / . & - sign	Go to origin position of WCS for XYA. Entering Shift + Key will Go to origin position (safe height) for Z . & - (shift + key)
22		Set XYA= 0 / Set Z = 0 / Number 0	Set origin position of WCS for XYA. Entering Shift + Key will set origin position for Z Input Number 0.

No.	Led Indication	Description
1		This will Indicate power in Hand handle.
2		This led will glow when user enter shift button
3		This led is continuously blink show that communication is running between hand handle & controller.

1		For MPG use

Chapter 4

Turmet (Lathe)

Brief Introduction

1. Wood working lathe machine consist of 3 axis & 4 axis.
2. If machine is having 3 axis than Axis Name : Y axis, Z axis & A axis.
If machine is having 4 axis than Axis Name : X axis,Y axis ,Z axis & A axis.
3. In our MeteorCNC, the axis on which Spindle is attached consider that axis as Z axis.
4. A- axis is having Servo spindle, in lathe mode it will work as spindle & in other modes it will work as servo motor.
5. Do not confused between multihead spindle machine which is attached on same axis & Multiaxis machine whose tool is different.

Sensor positions (Home switches)

1. In y axis, sensor position has in opposite side of A-axis..Toward sensor side consider negative & away from sensor is positive side.
2. In Z-axis, sensor is away from the job, towards sensor position is positive & away from the sensor position is negative.
3. In x-axis towards sensor side consider as negative & away from sensor is positive.

Set Zero position

1. The set zero position of Z- axis & X-axis is corners of job.
2. The setzero position of y-axis is below flat surface of the job.

1. Mode Change
   i. Lathe mode : if it is not in lathe mode then it will switch to lathe mode. In this mode rotary axis will act as a spindle in Speed mode.
   ii. Spiral mode : if it is not in spiral mode then it will switch to spiral mode. In this mode rotary axis will act as a 4^{th} Axis in Position mode.
   iii. Mill mode : if it is not in mill mode then it will switch to mill mode. In this mode rotary axis will act as a 4^{th} Axis in Position mode.
   iv. 4X mill mode : if it is not in 4x mill mode then it will switch to 4xmill mode. In this mode rotary axis will act as a 4^{th} Axis in Position mode.

   Whenever mode change is triggered X & Z axis will move to home position before mode change will happen.

2. Play mode
   i. Lathe : it will accept 2d dxf file, it will convert it to Gcode program automatically & run the machine.

ii. Spiral : it will accept 2d dxf file, it will convert it to Gcode program automatically & run the machine.
iii. Mill : Program will accept Gcode file with .nc,.ngc extension.
iv. 4x mill : Program will accept Gcode file with .nc,.ngc extension. with running X,Y,Z & rotary axis together like router to do milling & engraving on each face of the wood.
v. Lathe spiral : It is combine mode , need to select two files to run this mode , select one file for lathe(.dxf file) & other for spiral (.dxf file).

First lathe program will run. Once lathe program is completed, automatically it will be switched to spiral mode for running spiral program back to back without any halt. On completion of both the program, it will be switch back to lathe mode again and ready for next cycle.

vi. Lathe mill: It is combine mode , need to select two files to run this mode , select one file for lathe(.dxf file) & other for mill (.nc or .ngc file).

First lathe program will run. Once lathe program is completed, automatically it will be switched to mill mode for running mill program back to back without any halt. On completion of both the program, it will be switch back to lathe mode again and ready for next cycle.

Note : During Combined mode (lathe –spiral ,lathe-mill) when user can select file that time controller is asking for which operation file is selected.

3. Lathe Speed
   i. Cutting feed : It will cut the deign with defined cutting feed . this feed will be applied after cutter is plunged in the material. Its unit is mm/min
   ii. Plunge speed : This speed defined Plunge speed of Z axis & X axis. Unit is mm/min
   iii. Retract Speed : This speed defined Retract speed of Z axis & X axis. Unit is mm/min
   iv. Line Speed : This speed is for cleaning the surface of the Job.

4. Spiral Speed
   i. Cutting feed : it will cut the design with defined cutting feed. this feed will be applied after cutter is Plunge ed in the material. its unit is mm/min.
   ii. Plunge speed : This speed defined Plunge speed of Z axis & X axis. Unit is mm/min
   iii. Retract speed : This speed defined retract speed of Z axis & X axis. Unit is mm/min.

5. Job Setup
   i. Y-pitch : it will break the curve in to y pitch to find X & Z axis position.it is in mm.
   e.g. if curve is 50mm big & y pitch is 5 mm then it will break curves into 10 parts & will give X & Z axis position.
   ii. Axis follow : it is in mm. X axis plug at y axis 0 & Z axis will Plunge e after completing Axis follow distance.

iii. Auto feeder : To enable auto feeder or not.
iv. Slide Delay : This is slide delay.
v. Push Delay : This is push delay for auto feeder.
vi. Hold Delay : This is hold delay in auto feeder process.
vii. Feeder Delay : This is feeder delay in auto feeder process.
viii. Clamp delay : This is delay for clamping the job.
ix. Bypass feeder : It will enable or disable the feeder option.
x. Bypass material : It will enable or disable the feeder option.
xi. X goto Safe : After design play if user want to send X-axis to safe position.
xii. Total Cycle : User can define no.of cycles
xiii. Cycle Done : This will show no. of cycles are done.

6. Lathe Setup
   i. Total Depth : it is a final depth from the job's edge. Unit mm
   ii. Cut pass
      Finish pass (%) : it is set by User for finishing pass.
      Rough pass (%) : It is set by user for rough pass.
   iii. Pass number : only in roughing
   E.g. if pass number is 2 than it will do 80/2 = 40 % of two rough pass,if pass number is 3 & rough pass= 90 than it will do 90/3 = 30 % of rough pass
   iv. Sanding : It is user for sanding cycle (cleaning cycle).
   v. Line Cut : It is use for line cutting
   vi. Sand Z dist : it is define Z-axis position for sanding cycle from sensor.
   vii. Linning :
      Continuous increment : User can define Incremental depth.
      Offset :
   viii. UpDn Delay : This delay is use for making Up & Down sanding motor. Down before sanding cycle start & Up after sanding cycle finish.
   ix. Motor Delay : This delay is use to start Sanding Motor.
   x. Lathe Tool : User can select lathe tool.
   xi. X- Direction : It is defined X-axis safe position. Forward & reverse can be set according to mounting of X-axis. If X-axis & Z-axis mounted on same side then direction of both axes are same but if X-axis & Z-axis are mounted opposite then Direction will be opposite.

7. Spiral Setup
   i. Number pattern : it will define number of pattern
   ii. Spiral Angle : it will define angle of pattern
   iii. Total depth: it will define total depth.
   iv. Pass number: it will define total number of pass during operation
   v. Spiral Tool : User Can Select Spiral tool.