MODIA Systems

MiniMiller MM100

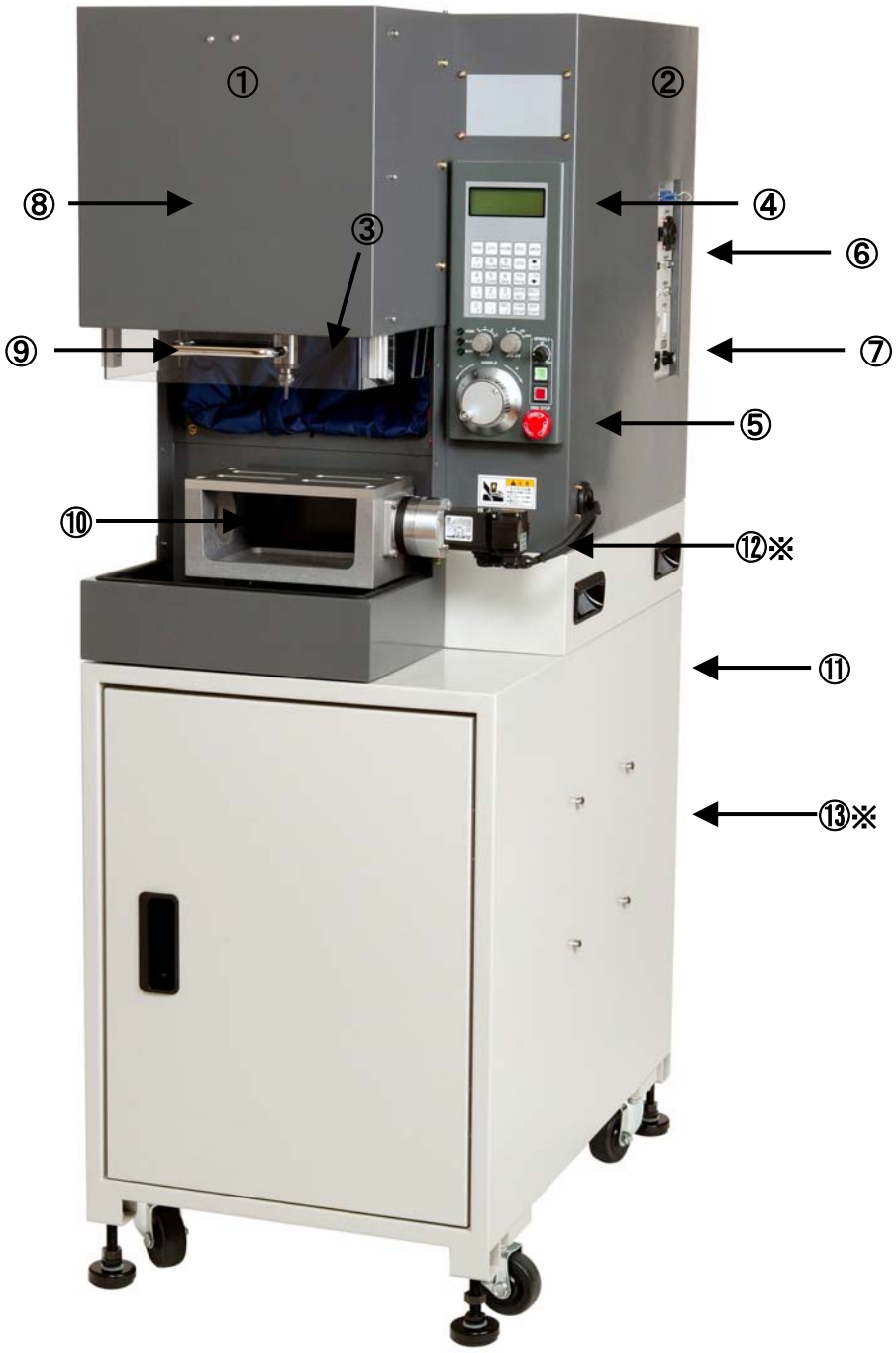
User's Guide Ver.2.4

Introduction

This guide provides required information needed to use the MiniMiller MM100 safely and efficiently. Please keep this guide in your records for future reference.

Specifications

Main Specifications

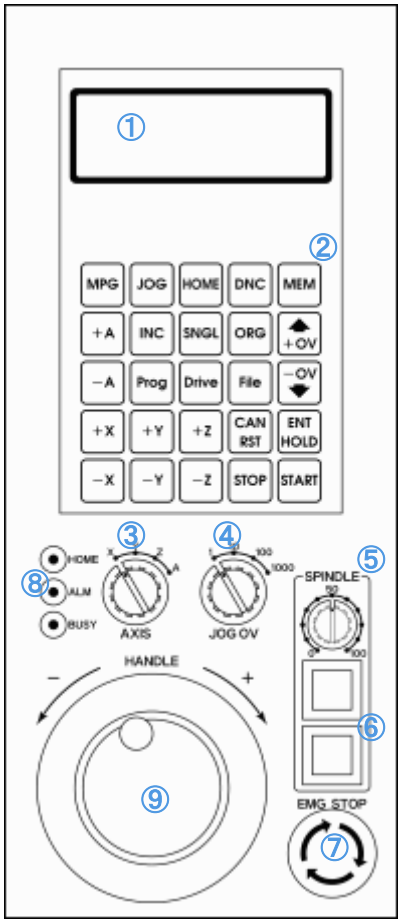


1. **X, Y, Z-axes**: Ball-screws and linear-guides comprise the X-axis which moves right and left, Y-axis which moves forward and backward, and the Z-axis which moves up and down.
2. **Servo Controller**: Houses the X, Y, Z-axes and all units which control the spindle motor and operational panel.
3. **Spindle Motor:** Rotating the tool for milling. (20000 - 60000rpm)
4. **Control Panel:** LCD display / Operating Keypad / Operation switches / Hand operated pulse handle (See page 6)
5. **EMG Stop Switch:** Incase of an emergency, immediately stops the unit. (The spindle motor cannot be stopped with this switch while in manual operation.)
6. **Connector Panel:** Contains various ports such as LAN, VGA, serial, and PS/2 as well as a circuit inlet.
7. **Power switch:** Turn on/off the main power supply.
8. **Spindle Cover:** Protects the spindle from harmful debris.
9. **Safety Cover:** User safety from the spindle and debris during operation.
10. **Work Table:** Holds the material to be milled.
11. **Optional Rack:** A special rack with casters for movement and adjusters to keep the unit level with its environment.

12. **4th Axis Unit:** A rotational axis unit parallel to the X-axis, and works with the other axes.
    *MM100-4 Type

13. **Coolant Unit:** Houses the water tank, filter, and pump. *Optional

Operation Panel



1. **LCD Display:** Displays unit status information.
2. **Operating Keypad:** Main operational functions.
3. **Axis Selector Switch:** Select which axis the pulse handle will control under MPG mode, and which axis to set the origin to under MPG/JOG/INC modes, and which axis to reset to the starting position in HOME mode.
4. **Axes Movement Setting Volume:** Multiplies the axis movement speed by 1/10/100/1000 with the pulse handle in MPG mode or with the axis movement keys in INC/JOG mode.
5. **Spindle Speed Manual Override:** Overrides the rotation rate of the main axis. Default is 100%, override may be adjusted below this for slower rotation or above for higher rotation up to a maximum rate of 20,000 RPM.
6. **Spindle and Coolant (option) Lamp:** While the spindle is ON, the lamp is blinked (**GREEN**). While the coolant unit is ON, the lamp is blinked (**ORENGE**)
7. **Emergency Stop Button:** Halts all unit activities and places the unit in ALARM status. (Axes movements, Spindle rotation and Coolant operation.) This button keeps being pressed until clicking the button off. To reset the alarm of system, press CANCEL key on the operation panel.
8. **Status LED's:**

   HOME: Lit once the axes are reset to the starting position.

   ALM: Blinks during unit has any alarm status.

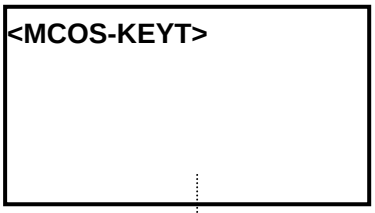
   BUSY: Blinks during automatic activity in DNC/MEM modes. If the mode is set to single-step action, the BUSY LED will blink only during operation.

   (Note: When the power is turned on and setting up the system, all LED's are blinked.
9. **Pulse Handle:** In MPG mode, commands the movement of each axis by a set interval for each pulse. This provides the user with the highest accuracy possible.

Display Panel

1. Start up

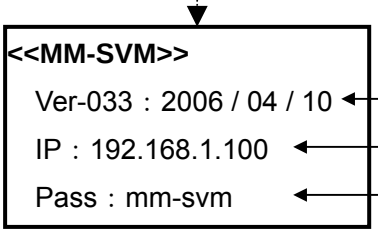
①Power on



20～30 seconds

*All LED's are blinked this moment

②System start up



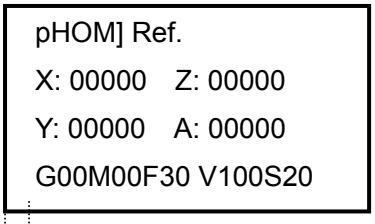
System Version

IP Address (Default)

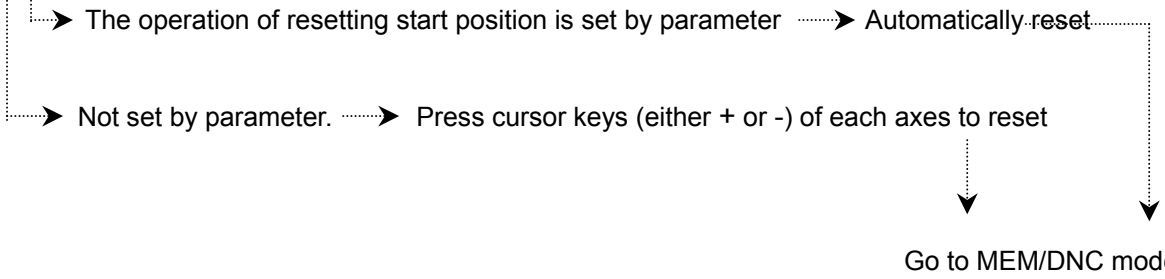
Password

2. HOME mode (Resetting start position)

●After system start up



*A-axis: Option



(Note: Mode can be specified in advance and set automatically by parameter.)

>> When this operation finished, HOME lamp will be blinked (GREEN).

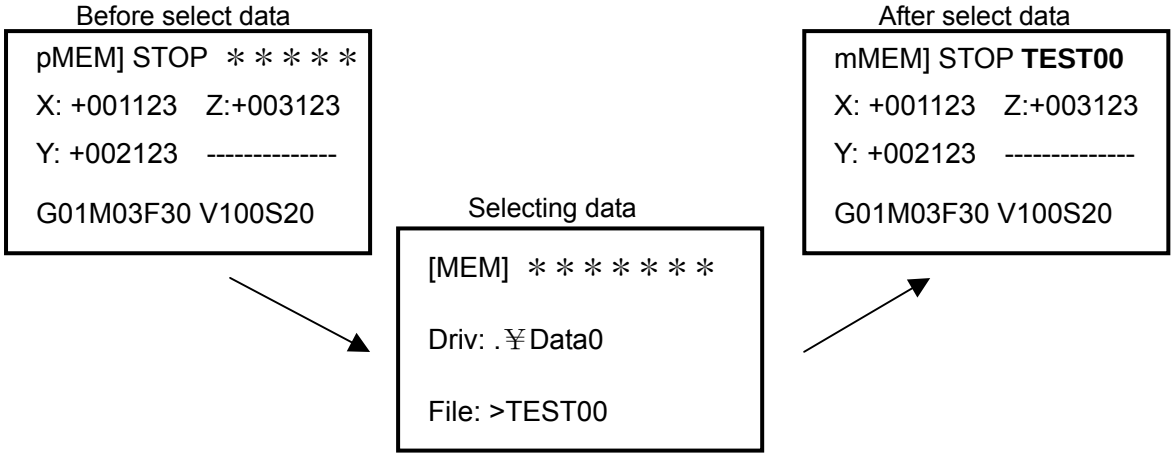
● When you want to reset the axes to starting position, press 

to see HOME mode on the display and cursor keys (either + or -) of each axis to reset to the starting position.

Note: Since the MM100 uses an incremental encoder, the unit must be reset after being turned on.

3. MEM mode (Memory operation)

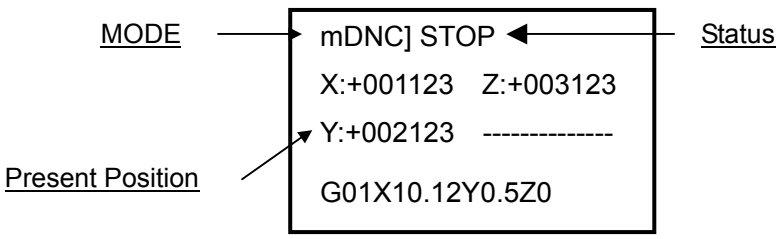
Press MEM >>



*See page9, how to select data.

4. DNC mode (DNC Operation)

Press DNC >>

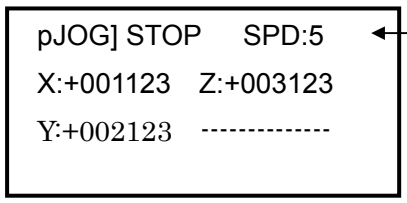


5. JOG mode (Manual Key Operation)

Press JOG >>

* Each press once, the display changes **JOG →INC →MPG** repeatedly.

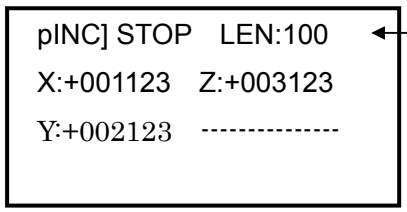
①JOG Mode



% of the JOG set speed

Set by 1/10/100/1000 with Axes movement setting Volume.

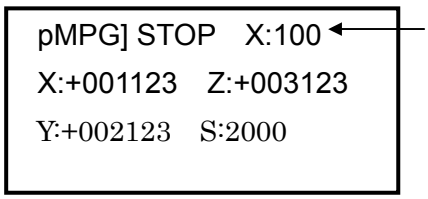
②INC Mode



Distance of movement(μ)

Set by 1/10/100/1000 with Axes movement setting Volume.

③MPG Mode

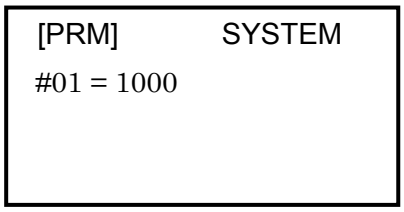


Selected axis / Speed by times

Select an axis by Axis Selector Switch, and set by 1/10/100/1000 with Axes Movement Setting Volume.

* 1000 is not available for safety sake.

6. Edit Parameter Mode



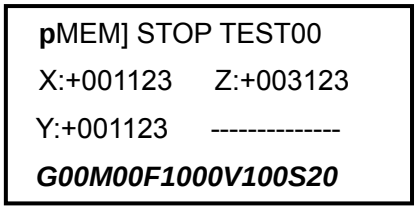
*See page 49, how to edit parameters.

7. Common Display

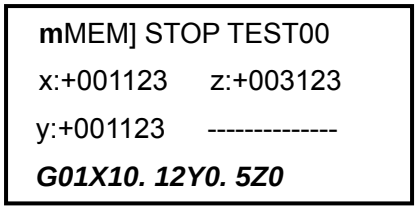
①Coordinates of Work／Coordinates of Machine

Press 6 Disp to switch the following three types of display >>

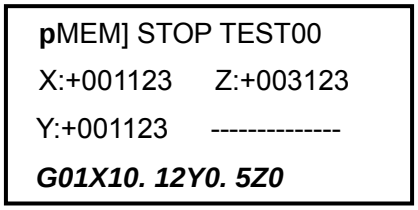
Coordinates of Work 1：display modal value



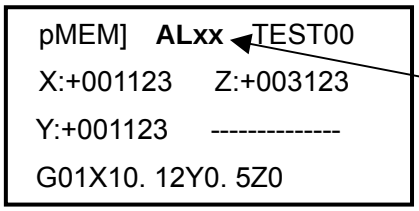
Coordinates of Machine：display next program



Coordinates of Work 2：display the next program to be operated



②While alarming



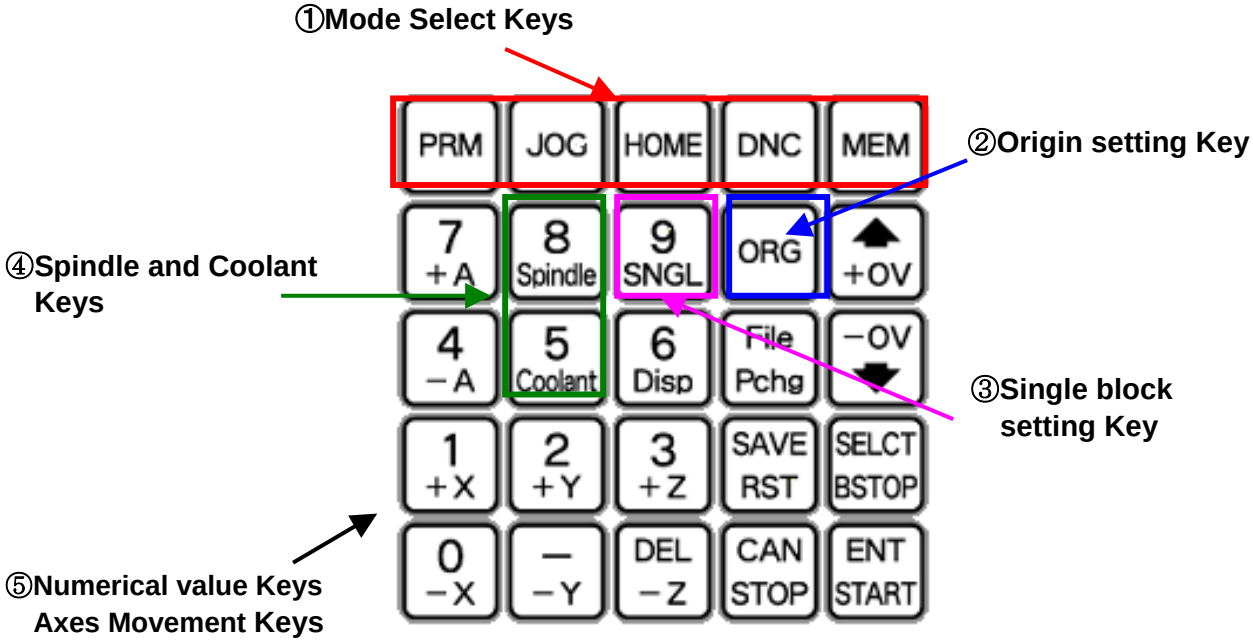
Alarm Codes will be displayed here.

Warning：WA01～WA99

Alarm：AL01～AL99

*See Page50-51, to check alarm codes.

Operating Keypad



①Mode Select Keys : PRM JOG HOME DNC MEM

1. Parameters Key : PRM Setting parameters

2. JOG Key : JOG Operates axes by hand with Pulse Handle.
   * Each press once, the display changes **JOG →INC →MPG** repeatedly.

●**JOG Mode** : Modes for maneuvering the unit's axes by axes cursor keys. While pressing cursor keys, the axes will move the distance set by Axes Movement Setting Volume.

●**INC Mode** : Modes for maneuvering the unit's axes by axes cursor keys. Each press once, the axes will move the distance set by Axes Movement Setting Volume.

●**MPG Mode** : Modes for maneuvering the unit's axes by hand-operated *Pulse Handle.* The axes will move the distance set by Axes Movement Setting Volume.

*See page19-20 for more detail about these modes.

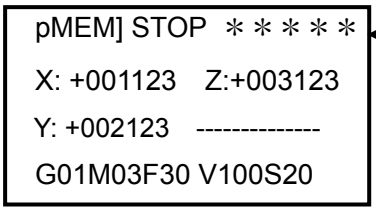
3. HOME Key : HOME Resetting the axes to the starting position.
   Press each cursor keys (either + or -) of axes to reset to the starting position.
   Display will show the axes are 00000 after they reset their position.

4. DNC Key : DNC DNC operation mode.
   A mode for sending and operating program/model data sent to the unit via the RS232C interface. There is no limit for the length of data, however, it will finish the operation when it meets M30 in the data at once.

5. MEM Key : [MEM] Memory operation mode.
MM100 has 30MB and stored 9 folders for program data.
Via FTP mode of network, input the program file (should be named ***txt.) from outside to the folders (named DATA**) and then select by operation panel.

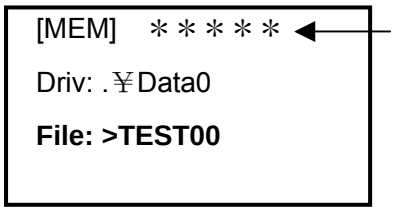
How to select the program file under MEM mode

Press [MEM] to switch the display to MEM mode.

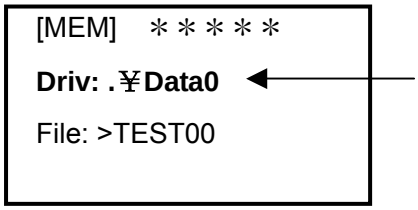


No selection yet.
* The selected file will be displayed.

Press [File Pchg] once, to switch the display to selection mode.



Use cursor keys [+OV] [−OV], to research files and select the file you want with press [SELCT BSTOP].



Press [File Pchg] once more, you can go back selecting the folders (Driv) which stores files.
Use cursor keys [+OV] [−OV], to research folders, and select the folder you want with press [SELCT BSTOP]. Then, select file.

>> After confirm if display has the file name you selected on the up right side, then press [ENT START] to start the operation.

>> When the operation starts, BUSY lamp will be blinked (**RED**).
*Under single block mode, it will blinked every step.

How to hold the operation after program operation start.

>> [CAN STOP] ・・・Hold the operation at once.

>> [SELCT BSTOP] ・・・Hold the operation after the block which is operating now is done.

To restart the program operation, press [ENT START]. On the other hand, to reset the program operation, press [SAVE RST], which cannot be back to get restart.

While program operation, axes' override of speed can be set with [+OV] [−OV] by every 10% (10-200%).

②Origin Setting Key： ORG

When setting the operation start position, each axis can be gotten origin set with this key if it needs. The axis which set origin, its start position will be kept for the programs that will start from then.
(Note: Same function as G92X0,Y0,Z0.)

Press ORG after set the axis to the proper position by JOG/INC/MPG modes, the display shows the axis position: 0. The axis which is selected by Axis Selector Switch is available to get Origin set.

③Single block Setting Key： 9 SNGL

This key set the program operation to single block operation in DNC/MEM modes.

In single step operation, each time press ENT START once, one block of program will be performed.

Displays small letter of S under set in single step operation mode.

```
pMEM] sSTOP TEST00
X: +001123   Z:+003123
Y: +002123   ----------
G01M03F30 V100S20
```

The letter is only displayed on Coordinates of Work Display1 (by modal value), not on Coordinates of Work Display2 or Coordinates of Machine Display.

*See page 9 about the displays.

④Spindle / Coolant Keys： 8 Spindle 5 Coolant

These keys are used for turning on the spindle and coolant.

>> 8 Spindle Press once, the spindle will be start the rotation, and press again, it will be stopped.

*Usually spindle operation follows the codes: M03 (to start) and M05 (to stop) of program. So this key is available to use if the program does not have these codes or operation by hand.

>> 5 Coolant Press once, the coolant unit will be start, and press again, it will be stopped.

*This key is for coolant unit which is option device ,and same function as the codes: M08 and M09.

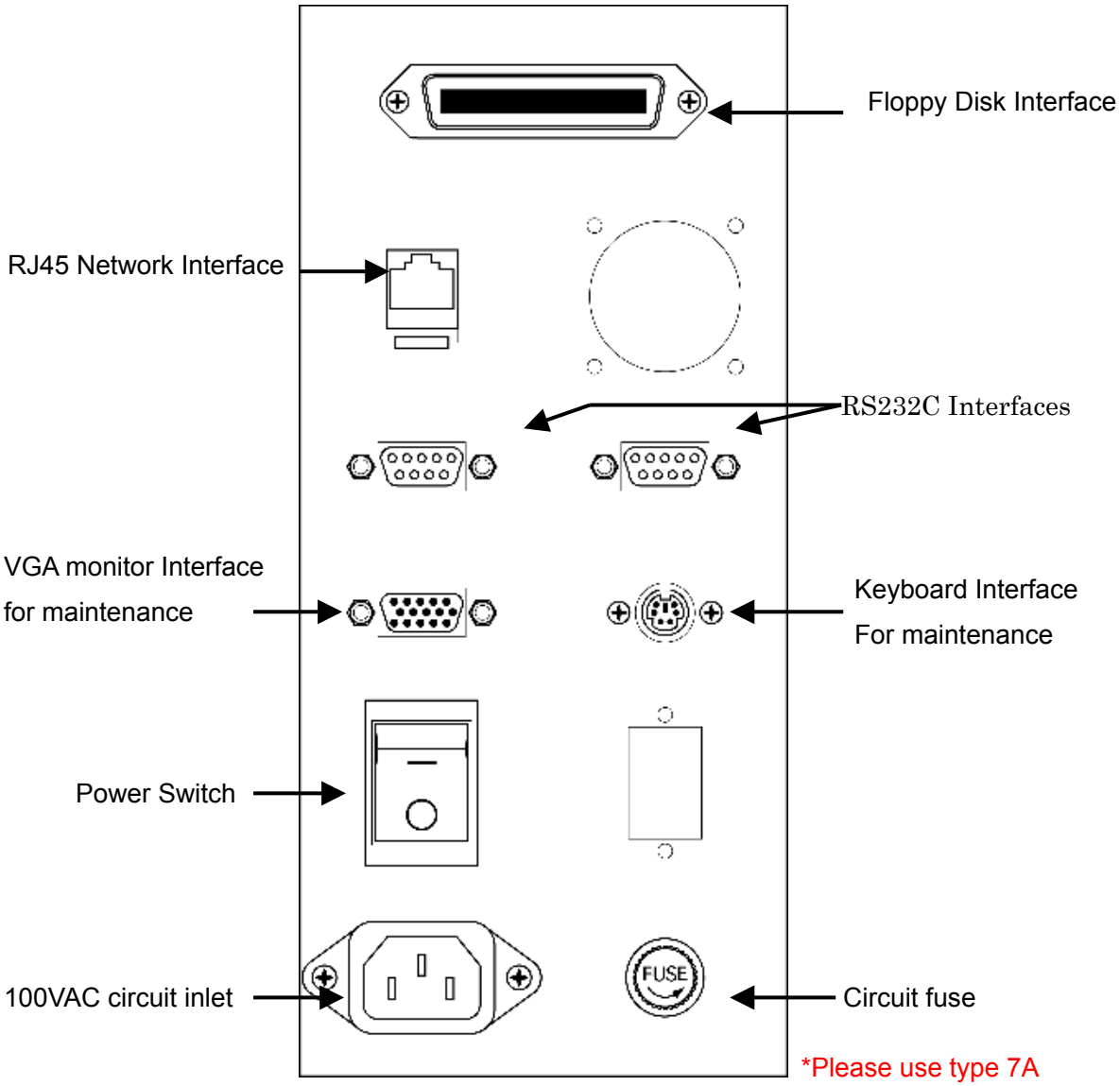
When these keys are pressed, spindle lamp will be blinked (**GREEN**) and coolant lamp will be blinked (**RED**).
And these operations will be stopped under any emergency stop or alarm happens.

⑤Numerical value Keys and Axes Movement Keys： 0 −X ～ 9 SNGL, − −Y DEL −Z

These keys are used for inputting the parameter values, and also used to move each axes in JOG/INC modes.

*See page 49, how to input the parameter values.

Connections Panel



Mode and Status

This section details the Modes and Statuses of the MiniMiller MM100 Unit. Please read this section carefully in order to understand each of the units many options.

Mode

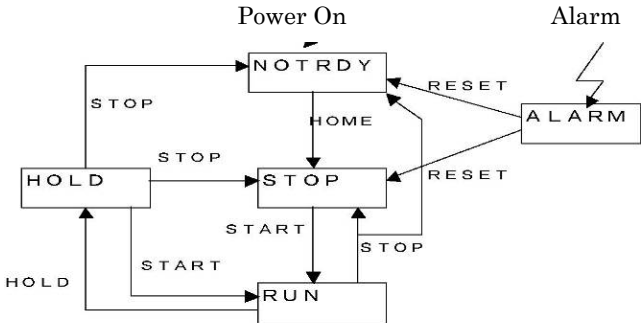
The MM100 has seven modes. Three operational modes: HOME/DNC/MEM and three axis movement modes: INC/JOG/MPG. These modes are changed by pressing any of the corresponding mode keys on the operating keypad. If an attempt is made to change the mode while the unit is being operated automatically, the unit will lock up.

- **HOME:** A mode for resetting the start position of the MM100s axes before operation. Since the MM100 uses an incremental encoder, the unit must be reset after being turned on before DNC and MEM operations can proceed.
- **DNC:** A mode for sending and operating program/model data sent to the unit via the RS232C interface.
- **MEM (MEMORY):** A mode for carrying out programs stored in the units internal flash memory (average: 7Mb max: 127Mb) or 3.5 inch floppy disks via the floppy disk interface. The operation of the MM100 this mode is the same as that of DNC mode.
- **INC/JOG/MPG:** Modes for maneuvering the unit's axes by keypad or hand-operated ***Pulse Handle.***
- **PRM:** A mode for inputting and changing the parameters.

Status

The status of the unit is always displayed in the display panel and shows following states.

Status	***Meaning***
ALARM-A	Alarm Servo OFF
ALARM-B	Alarm Stop/Servo ON
STOP	Stop
HOLD	Temporary Stop during operation
RUN	Operating

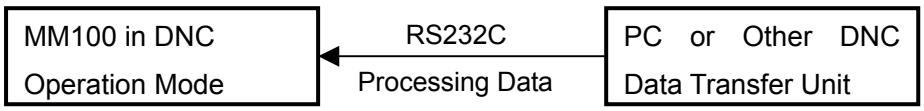


Processing Programs

Processing data is necessary for milling operations with the MM100. The following are ways to transfer such data to the MM100.

RS232C Interface (DNC Mode)

In DNC mode, users may transfer program data from a PC or other DNC data transfer device, such as Modia's ***LanTan LT120/LT330***, to the MM100 via the RS232C port located on the **Connections Panel** (see page 11).



- Adjust the interface settings for the serial connection between the MM100 and a transfer unit, such as a PC, to transfer the data. MM100 will handle the transfer of data in the order it is processed.
- To process smoothly without any interruptions from delays or lack of data, the MM100 can save data to an internal buffer.
- Depending on the amount of buffered data remaining, the MM100 continues to work as long as the proper quantity of data is supplied.

MEM Mode

In MEM mode, the MM100 can directly carry out processing data stored on various disk drives. Data files and the location of these files can be chosen via the Operating Keypad.

Operation

This chapter is designed to help you get started using the MiniMiller MM100. Please read this chapter thoroughly as it will help you understand the basic operation of the MM100.

Operational Outline

The following outline details the necessary procedures for operating and processing with the MM100. More detailed explanations will follow below.

1. **Preparatory Checks**
   - **A)** General appearance
   - **B)** Cables, optional devices

2. **Power:**
   - **A)** Position of power supply breaker switch.
   - **B)** Main unit power switch.

3. **Resetting Default Axes Positions**
   - **A)** Press the ENT START key while in HOME mode to reset axes positions.

     Note: *The unit will default to HOME mode after being turned ON.*

4. **Materials and Tools:**
   - **A)** Place the material to be processed on the worktable.
   - **B)** Set the tool that will be used for processing (Before/after power on)

5. **Set the Process Starting Point:**
   - **A)** Control the axes either via the ***Hand Operated Pulse Handle*** in MPG mode or via the ***Axes Movement Keys*** on the ***Operating Keypad*** in JOG/INC modes.
   - **B)** Set the Origin of each access for the particular process.

6. **Prepare the Unit for Processing:**
   - **A)** Choose an operation mode and select data file.
   - **B)** Confirm whether to use ***Single Step*** processing, and check the traverse speed override.
   - **C)** Press the ENT START key to begin processing.

- **Alarms:**
  1. Search for and solve the cause of the alarm.
  2. Release the system from alarm status.

- **Stopping Processes:**
  1. Stop processes temporarily with the 

 key.
  2. Stop processes completely with the 

 key.
  3. Place the system in alarm status with the ***Emergency Stop Button***.

7. **Finish the Operation:** If no problems arise and users have not initiated any manual operation on the unit, the operation will complete once all data has been processed.

Operation

The following section provides detailed information about the outline above.

1. **Check, Double Check, & Prepare:** To operate safely and efficiently, the following should be practiced each time the MM100 is used.
   - **A)** Power cables, communications cables, and any other cables such as LAN or floppy IDE cables should be connected securely.
   - **B)** Foreign objects that are or can be in the way of the spindle/axes operational range should be removed.
   - **C)** Make sure that anything you will be using (materials, tools, etc) are on hand and ready.
   - **D)** Any other safety precautions such as wearing gloves and protective lens should be practiced for user safety.
2. **Powering Up**
   - **A)** Please pull the safety cover down to prevent injury.

   - **B)** Turn on the unit via the main power switch located on the right side of the machine.

C) Once the unit is turned on, MM100 starts up with this follow.

- LCD display comes on.
- System version number is displayed on the LCD display.
- All LED on the operation panel are blinked.

D) The start up sequence takes about 1-2 minutes and is as follows:

- HOME mode is displayed on the LCD display.
- Servo turns on.
- All LED on the operation panel are faded away.

(Note: In case that the ***Emergency Stop Button*** or there is any system error, the alarm LED will be blinked.)

E) If the automatic reset default position of the axes are set by parameter already, the axes resetting will be begun at the same time the system turns on, and HOME LED will be blinked after the reset operation is completed. Then, the system goes to DNC/MEM modes automatically.

Note: *If the above conditions are not met, please confirm the power cable and all others are connected securely, the* ***Emergency Stop Button*** *is not depressed (reset if necessary), and reset the unit's power. If the unit continues failing to startup properly, please contact to the maker directly for support.*

3. Axes Resetting: The MM100 uses an incremental encoder and therefore axes positions must be reset after turning on the unit in order to prevent machine inaccuracies.

A) When the mode is set to HOME, please make sure no foreign objects are in the way of axes operational range and press the 

key. The unit will then reset each axis' position to factory default.

B) Axes Reset Sequence:

- Once the START is pressed, the Z-axis begins to move towards its home position.
- Upon completion, the LCD displays the Z-axis position as zero, and the X and Y-axes begin to move towards their home positions.
- Upon complete, all axes positions are displayed on the LCD as zero, and the HOME lamp on the operational panel is lit.

Display on LCD Panel

[HOM] NRDY X:+001123 Z:+003123 Y:+002123 A:+004123

[HOM] STOP X:+000000 Z:+000000 Y:+000000 A:+000000

***Note**: If the axes stop in the middle of resetting, or the system goes into alarm mode, please check and record the alarm display on the LCD and contact MODIA Systems for support.*

4. **Materials, Tools, & Programs:** The order in which to perform the following tasks will depend purely on how familiar users are with the MM100. Please consider your own safety when deciding which order is best for you.

   A) **Material Placement:** Place the material to be processed on the worktable and fasten with any tools that are necessary. The stability of the material is crucial to smooth processing, and more importantly in preventing accidents or damage to the unit. Therefore please pay careful attention when performing this task.

   B) **Tool Installation:** Processing tools such as drill bits must be fastened to the spindle in a secure fashion. Please use the included fastening tool to aid in securing any such tools. While performing this task, please pay attention to your fingers as well as the back and hollow of your hand in order to prevent injury to yourself. The following picture will illustrate this task.

   C) **Data Programs:** This task readies the program data that will be used to process your material. Users may choose one of two modes for this process, ***DNC*** or ***MEM***. In ***DNC*** mode, data is readied for use on an external device such as a PC or data transfer device. In ***MEM*** mode, users use the control panel to choose folders and files.

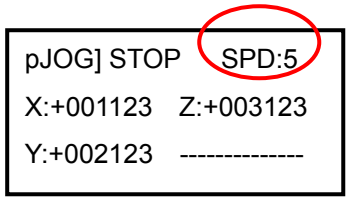
   *See page 9, how to choose program files.

5. **Process Start Point:** Each material and process has a different start point. For the sake of accuracy and continuity, it is necessary to set a new starting point each time a new process is chosen. Doing so will assure that materials will be processed as they were designed. To set the start point, please use the ***Pulse Handle*** while in MPG mode or the ***Axes Movement Keys*** in JOG/INC modes.

   A) JOG Mode: Press the 

key to switch the unit into JOG Mode. Move each axis by pressing each corresponding cursor keys. The movement speed of each axis can be set via the ***Axes Movement Setting Volume***, which is multiplier of the JOG speed set by parameter: 1500mm/min. Holding down a movement cursor key will keep the corresponding axis in transit.

Volume Dial	***Speed/Pulse***
1	5%
10	20%
100	50%
1000	100%

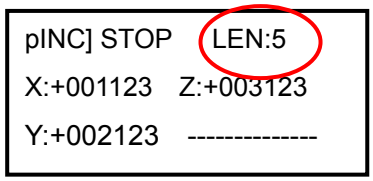


ex. Dial 10 → 20% 【1500(mm/min)x0.2=300mm/min】

   B) INC Mode: Press the 

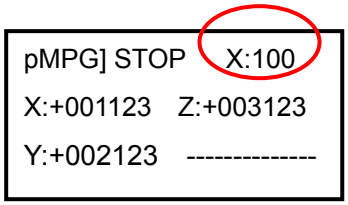
key to switch the unit into INC Mode. Move each axis by pressing each corresponding cursor keys. The movement distance (speed) of each axis can be set via the ***Axes Movement Setting Volume***. Each axis is will move a set distance when its corresponding movement keys are pressed. Holding down a movement key, however, will not cause the axis to keep moving.

Multiplier	***Speed/Pulse***
X1	1μ
X10	10μ
X100	100μ
X1000	1000μ



C) **MPG Mode:** Press the JOG key to switch the unit into MPG Mode. Choose an axis to move via the axis selector switch. Then move the chosen axis via the ***Pulse Handle***. Each click or pulse of the handle will move the chosen axis a set distance. Users may set how fast the axes move via the ***Axes Movement Setting Volume***.

Multiplier	***Speed/Pulse***
X1	1μ
X10	10μ
X100	100μ
X1000	1000μ



Note: Since the axes movement of 1000 μ is too big, 1000 μ is not available.

D) **Process Origin:** Once you have found each axis' start point, it is necessary to set the process' axis origins to each point. This means that for any particular program, the display will show each axis' starting position as zero. It does not mean that the unit's origins are being reset to different values.

To set the program's origins, choose an axis via the ***Axis Selector Switch*** (X/Y/Z/A). Then press the ORG key to set that particular point to the program's axis origins.

6. **Start/Stop Processing:** After performing and confirming the previous tasks, data processing may begin.

A) After choosing a processing program stored on disk or setting up a transfer device, pressing the START key in **DNC/MEM** modes will begin processing of program data. The MM100 operates on program data; however, it does not check the data before processing. Therefore, if data contain incorrect values or formats, the unit will halt and display an alarm status on the LCD Display.

To release the alarm (if possible), press the 

key. Whether or not the operation can continue depends on the type of alarm status (more details will be discussed later).

B) Processing speed (axes speed) can be adjusted by 10% increments from a minimum of 10% up to a maximum of 200% using the ***Override*** keys 

on the operating keypad. Overrides will go into effect on the next block of data after the present block is processed.

C) The 

key invokes ***Single Step Operation*** and allows users to progress data processing in single block increments. When the ***Single Step Operation*** is invoked, the unit switches from auto control to manual control. Pressing the START key in manual control will process one block of data. Subsequent presses will process successive blocks of data accordingly once the present block has finished.

D) The 

key changes the details on the on the LCD Display.

Coordinates of Work 1：display modal value

Coordinates of Machine：display next program to be operated

Coordinates of Work 2：display the next program to be operated

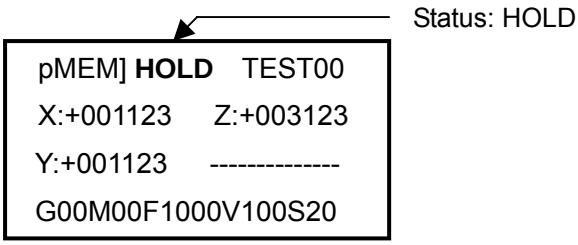
E) The STOP/BSTOP key suspends data processing, causing all axes to stop. Pressing the START key will resume processing previous action.

>> 

・・・Hold the operation at once.

>> 

・・・Hold the operation after the block which is operating now is done.



To restart the program operation, press ENT START. On the other hand, to reset the program operation, press SAVE RST, which will clear the buffer and can not be back to get restart.

* To hold a spindle, press 8 Spindle.

7. **Alarm Management:** When something happens to MM100 or the operation program, or Emergency switch is pressed, ALM LED will be lit on at the same time all operation stops. The MM100 supports a live alarming feature that halts all unit operations and displays an alarm status on the LCD Display.

   A) There are two kinds of alarms on the MM100, and different methods for remedying each one.

Alarm Type	***Reason***	***Result***
WA: Warning	Unit encounters data processing error; error receiving data via RS232C interface.	Axes slow down and stop; Servo remains ON
AL: Alarm	System or hardware error; ***EMG Stop Button*** is pressed.	All axes halt; Servo switches OFF (in auto control)

   B) **Releasing Alarms:** To release the unit from alarm status, press the SAVE RST key. Pressing this key once will release the alarm type of **WA**, pressing twice will release the alarm type of **AL**.
   After releasing alarms, the status of MM100 becomes RESET.

Note: *In DNC and MEM Mode, it is not possible to restart operations because data received to the unit's buffer is lost when the unit goes into alarm status.*

Programs

This section contains details regarding programs and their construction. Please read it carefully as it may help you to troubleshoot processing data errors.

Block Characters

One block contains a maximum of 64 letters. Flow control options for each block are CR/LF and are selected by parameter settings. The following characters are address codes that precede numerical values.

Address Code	***Function***
G	Preparatory Function
M	Auxiliary function
F	Feed Rate
S	Spindle Speed
T	T function
X	X-axis Move Instruction
Y	Y-axis Move Instruction
Z	Z-axis Move Instruction
A	A-axis Move Instruction
I	Central X-axis Coordinate of Circular Arc and Scaling
J	Central Y-axis Coordinate 〃
K	Central Z-axis Coordinate 〃
R	Circular Arc Radius
P	Scaling Coefficient
Q	
N	N-code
H	H-code for tool diameter fix

Note: *All characters other than the ones listed above are disregarded. More than one address may be written in one block; however, if there are more than two codes (excluding G), the former will be disregarded and the latter becomes effective.*

Data Format

Address	*Unit*	*Range*	*Decimal Point Input*
G	None	Effective G-code (figure)	Invalid
X	1mm	0～±999.999 mm	---
	1sec	0～999.999 sec	---
Y	1mm	0～±999.999 mm	---
Z	1mm	0～±999.999 mm	---
A	1mm	0～±999.999 mm	---
F	1mm/min	0～9999 mm/min	Invalid
M	None	0～99	Invalid
S	1rpm	0～20000 rpm	---
I	1mm	0～±999.999 mm	---
J	1mm	0～±999.999 mm	---
K	1mm	0～±999.999 mm	---
R	1mm	0～±999.999 mm	---
N	None	0～9999	Invalid
O	None	0～9999	Invalid
P	None	0～9999	---
Q	None	0～9999	Invalid

- Decimal points affect how the values for addresses **X, Y, Z, A, I, J, K, R, and P** are processed. If a decimal point is present, values are processed as direct millimeter values. Successive zeros are not required.

  Ex: X100. = X100.0

  If no decimal point is present, values are processed to 1/1000 mm. Leading zeros are also not required.

  Ex: X.56 = X0.56 or **G2 = G02**

- Addresses X, Y, Z, I, J, and K have limitations regarding their range.
  If a decimal point is included in an address value, the whole number value before the decimal must not exceed 4 characters.

  Ex: X12345.0 Wrong or **X1234. 0 Right**

  However, if a decimal is not present in an address value, the value must not exceed 8 characters.

  Ex: X123456789 Wrong or **X12345678 Right**

- Decimal values exceeding 3 decimal places are disregarded.

  Ex: X1.2345 = X1.234

Note: *Addresses utilizing decimal values may be constructed in the following formats. Note that the last two digits of the first example are ignored.*

XAAA.BBB**YY**	XAAA.BBB
X.BBB	X0.BBB
X1	X1.0
X1.	X1.0

- **Minimum Value:** If there is no decimal present in an address value, the minimum value becomes 1/1000mm of the specified figure. If a decimal is present then the minimum value is disregarded.

- It is possible to insert comments into a program if needed. Please use parentheses to denote comments.

Basic Functions

Coordinate System Setting

The unit's coordinate system is set to the units origins once all axes have been reset to HOME position. In order for a program to execute from its own specified origin, the unit must be instructed to carry out the G92 code.

Ex: G92 X000 Y000 Z000

When this block is executed, the coordinate system is set to the specified points and operates the program with the system thereafter. Any omitted axes, however, will remain unchanged.

Absolute and Incremental

There are two kinds of instruction that control axis movement in the MM100: Absolute and Incremental. Absolute instruction is executed through the **G90** code, which feeds the unit the coordinate values of the working coordinate system. Incremental instruction is executed through the **G91** code, which feeds the unit a specified move distance. The unit defaults to absolute instruction when it is turned on.

Axis Speed

Axes have two speeds controlled by different mechanisms: **Fast-forward** and **Cutting**.

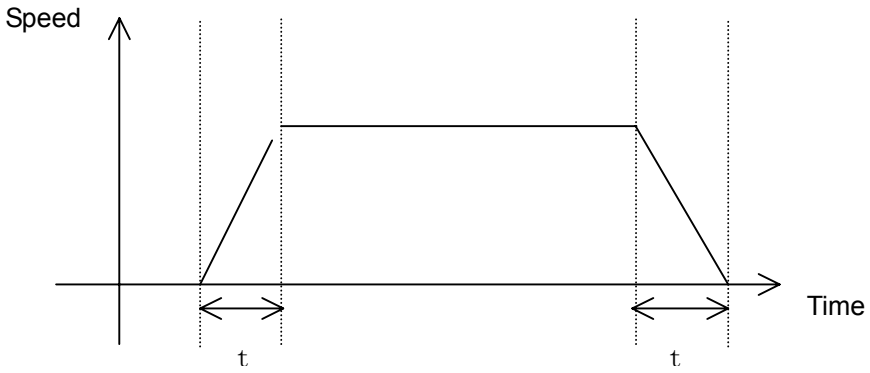
1. **Fast-forward Speed:** Controls the movement speed to a point specified by G00; the speed is specified in parameter settings. While fast-forwarding, each axis will reach the specified point simultaneously.
2. **Cutting Speed:** When a cutting instruction is sent by the G01, G02, or G03 codes the movement speed of the cutting tool is specified by the F function. The numerical value of the code can be up to 5 digits; the unit is mm/min.

   Ex: F3000 = 3000mm/min.

   Ex: F100 = 100mm/min.

Acceleration and Slowdown

Acceleration and slow down speeds can be diagramed in the shape of a trapezoid. The time period of movement is set in parameter settings.



Override

Override provides additional control over cutting and fast-forwarding speeds. It can be set from 10% to 200% of the present speed and can be changed while in motion.

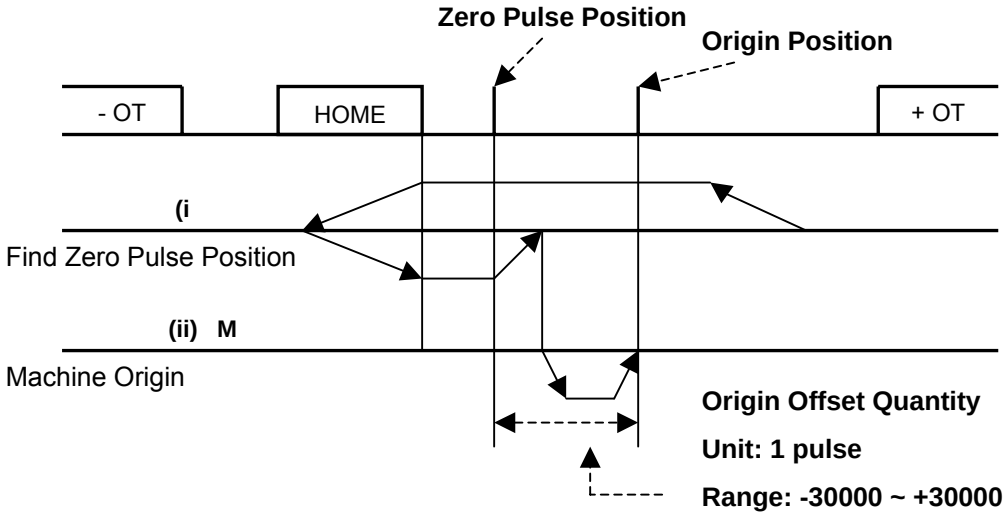
Hand Operation Settings

The amount of movement, speed, and magnification of hand operation can be set.

Reset axes to machine default point

The MM100 utilizes an incremental encoder and therefore, axes positions must be reset upon powering on the unit to avoid machine inaccuracies while carrying out a program. This need only be done at power on. The following process details how to reset the origin. Please note that the origin's Offset Quantity varies from machine to machine due of variations of the Home Limit Switch and Zero Pulse positions.

1. Find the Zero Pulse Position
2. Move the axes to the Home Position required by the program; this will become zero.
3. Reset Completion.



Preparatory Functions

The G-Function

The following table lists the various G-function codes and a general description of their functions.

Group	Code	Function	Default
0	G00	Fast-forward	
	G01	Cutting-forward (Linear Movement)	x
	G02	Rotational Movement (Clockwise)	
	G03	Rotational Movement (Counter Clockwise)	
1	G90	Absolute Instruction	x
	G91	Incremental Instruction	
2	G61	Exact Stop Mode	x
	G64	Cutting Mode	
3	G17	Select X-Y Plane	
	G18	Select Z-X Plane	
	G19	Select Y-Z Plane	
4	G22	Stroke Limit ON	
	G23	Stroke Limit OFF	x
5	G98	Fixed Cycle (Initial level return)	
	G99	Fixed Cycle (R-point level return)	
6	G40	Tool diameter fix	
	G41	Tool diameter fix	
	G42	Tool diameter fix: chancel	x
7	G43	Tool diameter fix: +	
	G44	Tool diameter fix: -	
	G49	Tool diameter fix: chancel	x
8	G04	Dowel	
	G10		
	G11		
	G12		
	G13	Scheduling function	
	G92	Coordinate System Creation	
9	G50	Cancel Scaling	x
	G51	Scaling Command	
10	G53	Machine Coordinate System	
	G54	Work Coordinate System 1	
	G55	Work Coordinate System 2	
	G56	Work Coordinate System 3	
	G57	Work Coordinate System 4	
	G58	Work Coordinate System 5	
	G59	Work Coordinate System 6	

11	G15	Return to Origin	
	G28	Return to Reference Point	
	G31	Skip Z-axis movement	x
12	G73	Intermittent Forward	
	G80	Cancel Fixed Cycle	
	G81	Cutting Forward	
	G82	Cutting Forward	
	G83	Intermittent Forward	
	G85	Cutting Forward	
	G89	Cutting Forward	
13	G77	4^{th}-axes(A) Speed Setting	
	G78	4^{th}-axes(A) Millimeter Select	
	G79	4^{th}-axes(A) Angle Select	x
14	G65	Send External Input	x
	G66	Call Subprogram(s)	
	G67		
	G68		
	G69		

Note: *Codes belonging to the same group can not be written in the same program block. If a situation occurs, only the last code displayed will be run. "Default" means the default condition when the power on.*

a) G00: Fast-forward : Fast-forward to the target position.

- All axes reach to the position same time
- The speed of fast forward can be set by parameter
- The speed is synthesis speed of X, Y and Z-axes
- No imposition check

b) G01: Cutting Forward: Cutting-forward to the target position.

- All axes reach to the position same time
- The speed of cutting forward can be set by F-codes.
- The speed is synthesis speed of X, Y and Z-axes

c) G02/G03: Rotational Movement

- Rotational Movement on the plane are commanded by G17/G18/G19.

d) G04: Dowel: Wait

- The time of the wait is set by the address: G04 x -----

e) **G09: Exact Stop Check**: Cutting-forward to the target position.

- All axes reach to the position same time
- The speed of cutting forward can be set by F-codes.
- The speed is synthesis speed of X, Y and Z-axes
- Checks in-position values

f) **G28: Return to Reference Point: Resets machine coordinates to reference point**

- If a midway point is specified, the machine will return to its reference point once it passes through the midway point
- Operation speed can be set by parameter
- Does not check in-position values

g) **G54-G59: Work Coordinate System**: Changes the current system to a Work Coordinate system.

h) **G53: Machine Coordinate System**: Changes the current system to a Machine Coordinate system

i) **G61: Exact Stop Mode:** Changes operation from Cuting Forward Mode to In-position mode.

- In-position range can be set by parameter

j) **G64: Cutting Mode**: Changes from Cutting Forward to Consecutive Cutting Mode.

k) **G90: Absolute Instruction**: Set a command value as an absolute instruction.

l) **G91: Incremental Instruction**: Set a command value as incremental instruction. In this mode, incremental operation resumes from a blocks pause position.

m) **G92: Coordinate System Creation**: Create work coordinate system with the designated value.

n) **G13: Scheduling function**: Makes it possible to repeat (loop) NC Data

o) **G73-G89: Fixed Cycle**

G73: Intermittent Forward: High-speed drilling cycle to make a deep hole

Intermittently cut forward to the bottom of a hole while removing debris.

G 7 3 X____Y____Z____R____Q____F____K____

X____Y____ : Hole position data

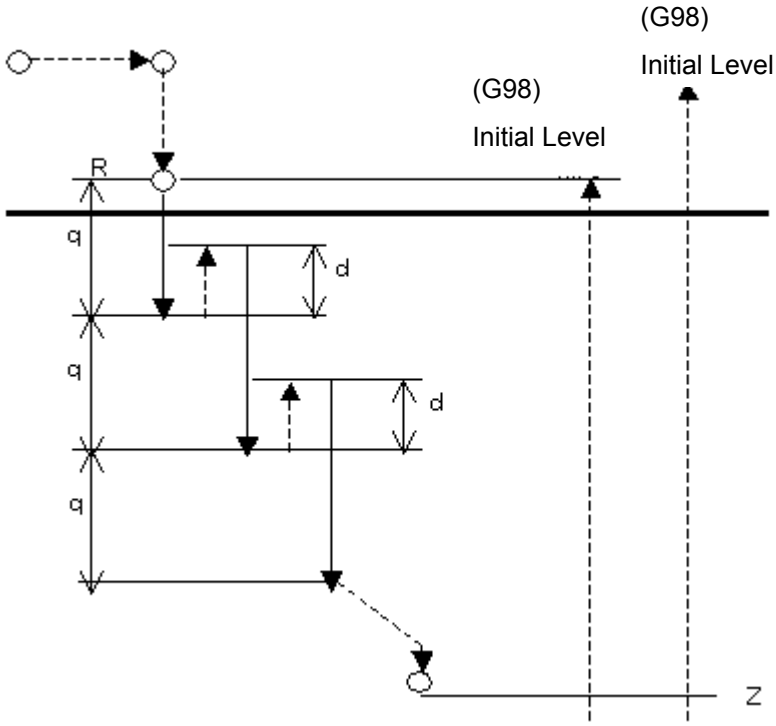
Z____ : Distance from R to the bottom of the hole

R____ : Distance from initial level to R

Q____ : Cutting amount each time

F____ : Cutting forward speed

K____ : Repeat times (only necessary to repeat)



・Intermittent Forward to Z way makes easier to exhaust the dust from deep hole.

・Because it can set evading amount precisely, efficient operation is possible.

・Evading amount is set by parameter.

・Evading moves by fast-forward.

・Before instruct G73, rotate the spindle by M code (M03/M04).

If M code is set in the same block as G73 in, M code will be excused after first positioning operation. (If K: Repeat times, is set, only first time. From second time, it will skip M code.)

・If tool long fix function (G43/G44/G49) is set during Intermittent forward, it will be offset when R point positioning.

・The data block for drilling has to have X, Y and Z axes data, otherwise the block will be ignored.

・Please set Q in the block which drilling is operated at. If Q is set at the blocks have no drilling operation, the data cannot be recognized as modal data

・Please NOT set G codes of group 01 in the block in which has G73, otherwise G73 will be canceled.

G81: Cutting Forward: Drill cycle spot drilling

For usual drilling. Cutting forward to the bottom of the hole and fast forward to evade from the bottom.

G８１X＿＿Y＿＿Z＿＿R＿＿F＿＿K＿＿

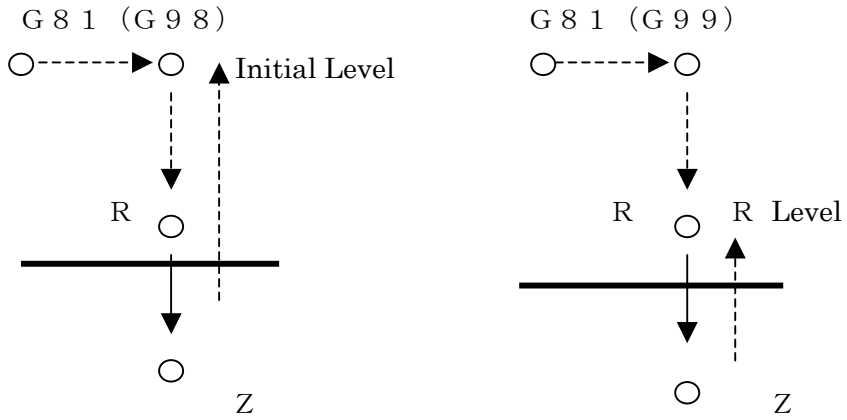
X＿＿Y＿＿ : Hole position data

Z＿＿ : Distance from R to the bottom of the hole

R＿＿ : Distance from initial level to R

F＿＿ : Cutting forward speed

K＿＿ : Repeat times (only necessary to repeat)



・After positioning X and Y axes, fast forward to R. Then drill from R level to Z.

・Evading moves by fast-forward.

・Before instruct G81, rotate the spindle by M code (M03/M04).
If M code is set in the same block as G81 in, M code will be excused after first positioning operation. (If K: Repeat times, is set, only first time. From second time, it will skip M code.)

・If tool long fix function (G43/G44/G49) is set during Intermittent forward, it will be offset when R point positioning.

・The data block for drilling has to have X, Y and Z axes data, otherwise the block will be ignored.

・Please NOT set G codes of group 01 in the block in which has G73, otherwise G73 will be canceled.

G82: Cutting Forward: Drill cycle counter boring

For usual drilling. Cutting forward to the bottom of hole and fast forward to evade from the bottom after dowelling at the bottom. This method can make better accuracy for the depth of the hole.

G 8 2 X___ Y___ Z___ R___ P___ F___ K___

X___ Y___ : Hole position data

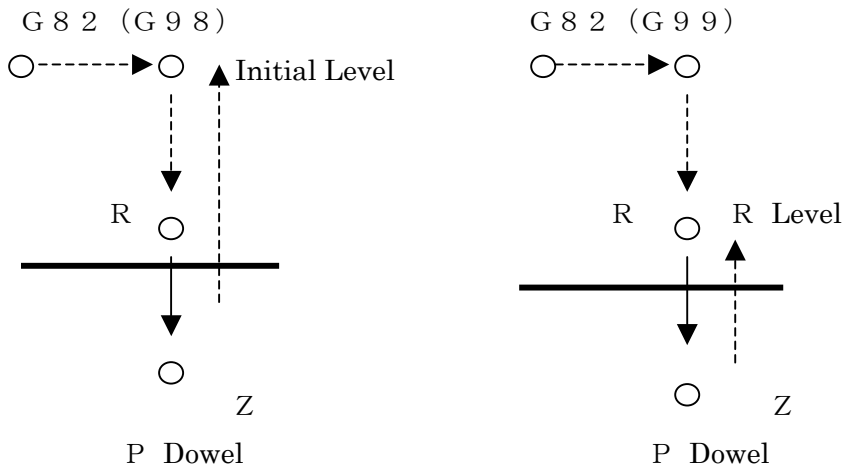
Z___ : Distance from R to the bottom of the hole

R___ : Distance from initial level to R

P___ : Doweling time at the bottom of the hole

F___ : Cutting forward speed

K___ : Repeat times (only necessary to repeat)



・After positioning X and Y axes, fast forward to R. Then drill from R level to Z.

・After dowelling at the bottom, evade by fast forward.

・Before instruct G82, rotate the spindle by M code (M03/M04).
If M code is set in the same block as G82 in, M code will be excused after first positioning operation. (If K: Repeat times, is set, only first time. From second time, it will skip M code.)

・If tool long fix function (G43/G44/G49) is set during Intermittent forward, it will be offset when R point positioning.

・The data block for drilling has to have X, Y and Z axes data, otherwise the block will be ignored.

・Please set P in the block which drilling is operated at. If P is set at the blocks have no drilling operation, the data cannot be recognized as modal data.

・Please NOT set G codes of group 01 in the block in which has G73, otherwise G73 will be canceled.

G83: Intermittent Forward: Deep drill cycle

Intermittently cutting forward to the bottom of the hole with exhausting the dust out of the hole.

G 8 3 X____ Y____ Z____ R____ Q____ F____ K____

X____ Y____ : Hole position data

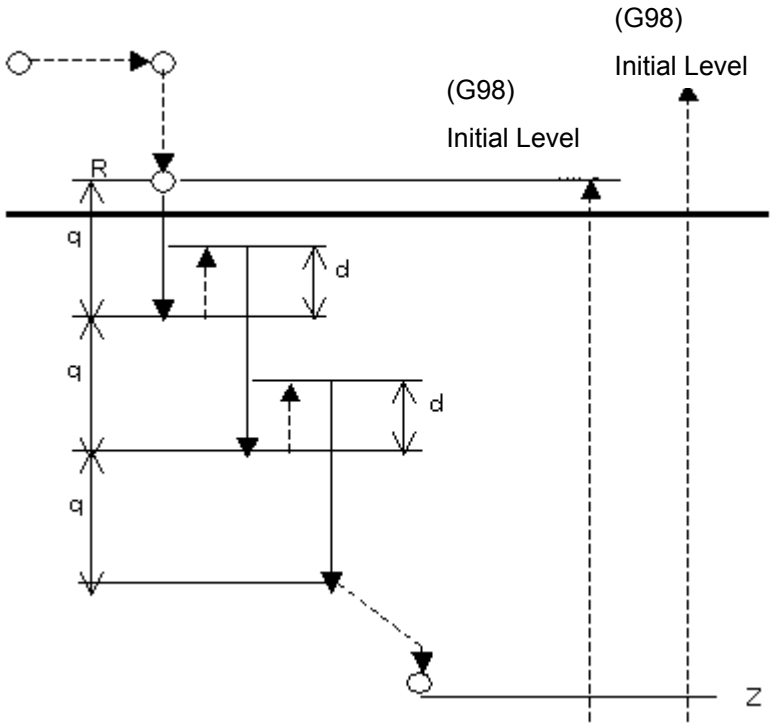
Z____ : Distance from R to the bottom of the hole

R____ : Distance from initial level to R

Q____ : Cutting amount each time

F____ : Cutting forward speed

K____ : Repeat times (only necessary to repeat)



- Q is the cutting amount by each once and is always set as incremental amount.
- After the second cutting, cutting operation will change to cutting forward from fast forward at the position "d".
  Position "d" is set by parameter.
- The instructed value for Q must be plus value. Minus value will be ignored.
- Before instruct G83, rotate the spindle by M code (M03/M04).
  If M code is set in the same block as G83 in, M code will be excused after first positioning operation. (If K: Repeat times, is set, only first time. From second time, it will skip M code.)
- If tool long fix function (G43/G44/G49) is set during Intermittent forward, it will be offset when R point positioning.

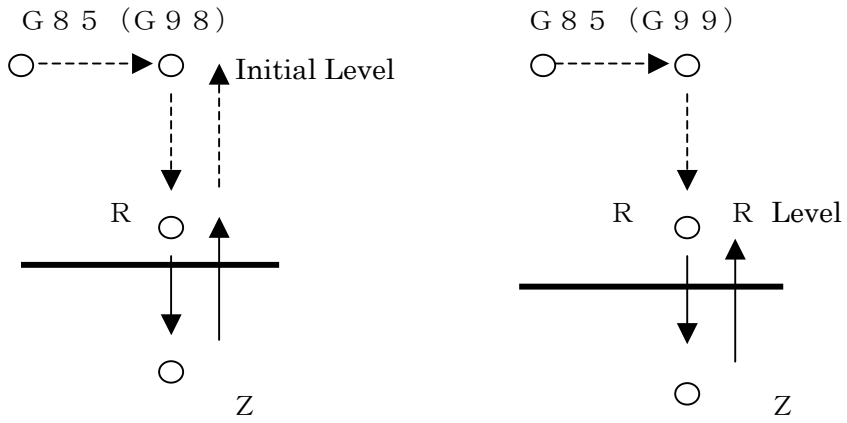
・The data block for drilling has to have X, Y and Z axes data, otherwise the block will be ignored.
・Please set P in the block which drilling is operated at. If P is set at the blocks have no drilling operation, the data cannot be recognized as modal data.
・Please NOT set G codes of group 01 in the block in which has G73, otherwise G73 will be canceled.

G85: Cutting Forward: Boring cycle

For boring operation.

G 8 5 X____Y____Z____R____F____K____

X____Y____ : Hole position data
Z____ : Distance from R to the bottom of the hole
R____ : Distance from initial level to R
F____ : Cutting forward speed
K____ : Repeat times (only necessary to repeat)



・After positioning X and Y axes, fast forward to R. Then drill from R level to Z.
・Before instruct G85, rotate the spindle by M code (M03/M04).
If M code is set in the same block as G85 in, M code will be excused after first positioning operation. (If K: Repeat times, is set, only first time. From second time, it will skip M code.)
・If tool long fix function (G43/G44/G49) is set during Intermittent forward, it will be offset when R point positioning.
・The data block for drilling has to have X, Y and Z axes data, otherwise the block will be ignored.
・Please NOT set G codes of group 01 in the block in which has G73, otherwise G73 will be canceled.

G89: Cutting Forward: Boring cycle

G８９X＿＿＿Y＿＿＿Z＿＿＿R＿＿＿F＿＿＿K＿＿＿

X＿＿＿Y＿＿＿ : Hole position data

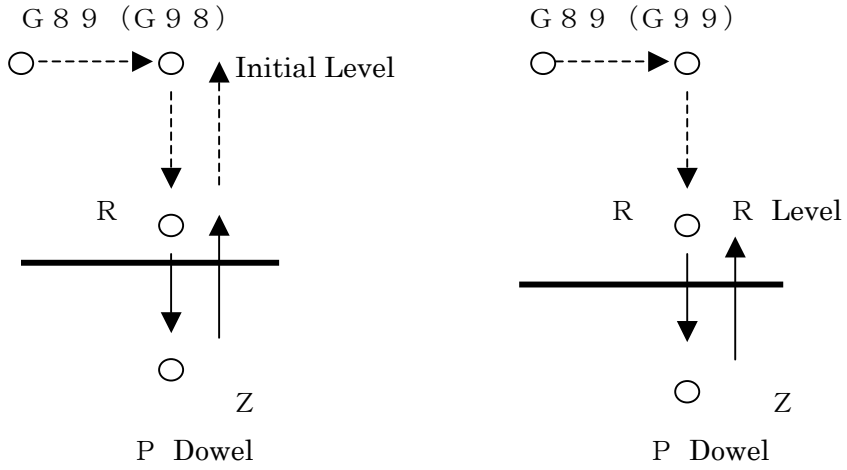
Z＿＿＿ : Distance from R to the bottom of the hole

R＿＿＿ : Distance from initial level to R

P＿＿＿ : Dowel time at the bottom of the hole

F＿＿＿ : Cutting forward speed

K＿＿＿ : Repeat times (only necessary to repeat)



・After positioning X and Y axes, fast forward to R. Then drill from R level to Z. After get to Z, dowel and then cutting forward to back to R.

・Before instruct G89, rotate the spindle by M code (M03/M04).
If M code is set in the same block as G89 in, M code will be excused after first positioning operation. (If K: Repeat times, is set, only first time. From second time, it will skip M code.)

・If tool long fix function (G43/G44/G49) is set during Intermittent forward, it will be offset when R point positioning.

・The data block for drilling has to have X, Y and Z axes data, otherwise the block will be ignored.

・Please NOT set G codes of group 01 in the block in which has G73, otherwise G73 will be canceled.

G80: Cancel Fixed Cycle

Cancel all fixed cycles and start usual operation all after. The data of R point level and Z point will be canceled. (Incremental instruction: R=0, Z=0)

The M-Function

M-Function codes are designed for use in special situations. The following table lists the M codes and their functions.

1. Program Control

Code	***Function***
M00	Stops the program and restarts from START position.
M02	Terminate the program/Delete the data buffer.
M30	Terminate the program (Rewind)/Delete the data buffer.
M98	Terminate subprograms
M99	Rewind program.

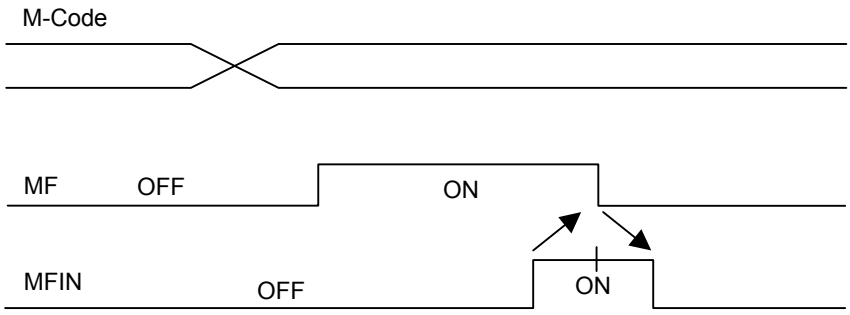
2. Spindle and Coolant

Code	***Function***
M03	Spindle CW ON
M04	Spindle CCW ON
M05	Spindle OFF
M08	Coolant ON
M09	Coolant OFF

The M-code is executed once a specified position in the program has been reached. Upon execution of the code, the MF status LED will turn ON (green), and the machine will verify that the MFIN signal input is ON; once verification is complete, the machine will move onto the next program block.

Output (Controller → Machine) M-Code, MF

Input (Controller ← Machine) MFIN



Note: *The M99 code does will not output to the display.*

The F-Function

The F-Function is used to set the machine's cutting speed; it is also possible to manually override the cutting speed via the Operation Panel

Linear Axis: 1mm/min. ~ 9999mm/min.

Rotational Axis: 1deg/min. ~ 99999mm/min.

Override: Cut Speed = F-Code x Percent Override (10% – 200%)

The S-Function

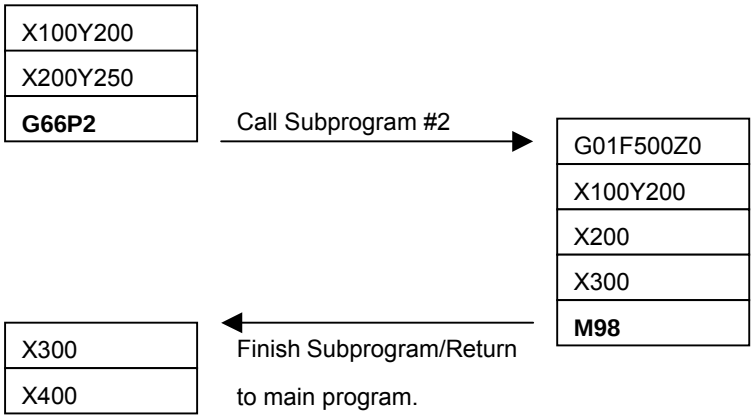
The F-Function is used to output voltage corresponding to the designated S code.

The Special Functions

USubprogram

The **G66** code is used for calling subprograms. It is used in the format **G66Pnn** where **P** indicates subprogram and **nn** indicates the desired subprogram. Should a subprogram be called that is does not exist, the machine will display the error message "Alarm: No Subprogram" Furthermore, you can not call a subprogram from within another subprogram.

Subprogram Execution Path



Subprogram Name

Ex) Subprogram#2 (G66P2)

The header and its extension of the subprogram are set by parameter.

Parameter File: HW-SYSTM.prm

Header: #34

Extension: #35

Ex) #34=S, #35=sub

Subprogram Name: S2.sub

Scheduling Function

The scheduling function is used for executing several NC data blocks consecutively

G13P1: Begin reading the NC data to be executed.

G13P2: Begin operating NC data blocks consecutively. The block after this code will be ignored.

A maximum of 8 NC data blocks may be registered.

Ex)

:

:

G13P1	Begin reading the NC data to be excised.
(TEST1)	1st registered NC data block
(TEST2)	2nd registered NC data bock
(TEST3)	3rd registered NC data block
(TEST4)	4th registered NC data block
G13P2	Begin executing NC data consecutively.
	Execute consecutively beginning with 1st NC data block.
	Only the NC data after this block are available.
	M30 is only available in the last NC data block. M30 is ignored in any previous blocks.

MM100 Technical Specifications

This section details the MM100's various mechanical and control specifications.

Mechanical Specifications

No	*Item*	*Specification*
1	Axes Operational Ranges	X-axis: 110mm Y-axis: 85mm Z-axis: 100mm A-axis: 360°X-25 rotation (Optional)
2	Axes Mechanisms	X, Y, Z-axes: Precision ball screw and linear-guide A-axis: Non-recoil precision rotary mechanism
3	Axes Motors	100W AC Servo Motor (Z-axis has a brake attachment)
4	Position Detection	Incremental encoder
5	Axes Speeds	9999mm/min (Cut and Fast-forward) **Note:** Speed varies with parameter settings
6	Positioning accuracy	0.02mm/100mm
7	Repeat Position Accuracy	0.006mm
8	Spindle Motor	100W AC Servo Motor
9	Spindle Rotations	20000 RPM (60000 RPM Optional)
10	Spindle Colette Size	1-6mm
11	Work Table Area	200mm (X) x 110mm (Y)
12	Work Fixation Method	Table center x T-slot (axes direction) x 1
13	Operation Panel	Key switch and manual pulse handle
14	Display	LCD with backlight (20 characters X 4 lines)
15	Program Storage	Internal flash memory (30MB Standard) External 3.5-inch FDD (Optional)
16	Interface	RS232C, LAN (FTP server), 3.5-inch FDD
17	Power-supply	AC 90-110V (AC 180-220V Optional)
18	Current Rating	450VA (800VA Max)
19	Size	H1320mm X W440mm X D700mm
20	Weight	~ 92 kg (220 lbs)

Control Specifications

No.	*Item*	*Specification*
1	Control Axis	X, Y, Z and optional A-axis
2	Simultaneous Control Axes	X, Y, Z axes and optional A-axis
3	Axes Unit of Movement	Linear Axes: 0.001mm Rotational A-axis: 0.001deg
4	Position Detection	Incremental Encoder
5	Maximum Instruction Value	Linear Axes: ±999.999mm Rotational A-axis: ±999.999deg
6	Data Input	File transferring via FTP
7	Data Code	ASCII
8	Decimal Input	Possible
9	Fast-forward Speed	Parameter: Fast forward Speed
10	Cutting-forward Speed	F code Override: 10-200%
11	Acceleration and slow down speeds	Trapezoidal
12	Absolute / incremental Instructions	G90: Absolute Instruction G91: Incremental Instruction
13	Coordinate System Creation	G92: Create coordinate system
14	Fast-forward	G00: Fast-forward (Fast-forward speed)
15	Linear Movement	G01: Linear Movement (Cutting-forward speed)
16	Circle Movement	G17: X-Y Plane G02: CW G18: Z-X Plane G03: CCW G19: Y-Z Plane
17	Dowel	G04: Wait for the designated time
18	Exact stop Check	G09: In-position check (one shot)
19	Exact stop Mode / Cutting Mode	G61: In-position check Mode G64: Continuous Cutting Mode
20	M-Function Codes	M03/05: Spindle ON/OFF M08/09: Coolant ON/OFF Control for external output
21	Single Block	Step by step program execution
22	Skip External Input	G31: Stop movement and set Z axis to zero.
23	Emergency Stop	Emergency Stop Button or external input
24	Over-travel	When the unit encounters a stroke-end during operation, an over-travel alarm will occur

25	Software Limiting	Axes limits may be set in parameter settings
26	Manual Axes Operation	Via +/- keys on the operation panel or the manual pulse handle
27	Self-diagnostic Function	Memory is checked upon power ON. Various alarms are simultaneously checked during operation.
28	Override	Cutting speed can be overridden from 100-200%
29	Return to Reference Point	Return to the reference point set by G28.
30	Scaling	Scale between 0.001-9.999 times by G51 (Scaling) and G50 (Cancel Scaling)
31	S-Function Codes	Set spindle speed
32		
33	Manual pulse handle input	Operate each axis manually by manual pulse handle.
34	Tool diameter fix	G43, G44, G49: Tool diameter fix
35	Fixed Cycle	Fixed Cycle
36		

Parameter

This sections details important information regarding parameter settings needed to operate the MM100. You may set and change any settings during PRM Mode. (***P.49: Edit Parameter***)

System Parameters

System parameters are used to set up the MM100 for operation and are stored in the **SYSTEM.prm** file. Though the values will be displayed as decimals, please enter them into the parameter file as whole numbers. The following table displays the parameter file line by line. Please use it as a reference when setting your system parameters.

No.	*Meaning*	*Input method*	*Note*
1	Parameter Version		
2	Fast-forward	1000	1mm/min
3	F-code Initial Value	600	1mm/min
4	JOG speed	600	1mm/min
5	INC Speed	600	1mm/min
6	Return to Origin Speed	600	
7	Range of Imposition	50	0.001mm
8	S-code Initial Value	2000	1rpm
9		0	0
10	M03/M04/M05 Spindle acceleration and slow down speeds	0-100	0: 4 sec 1-100: 0.1/sec
11	F1 1	User Data	mm/min
12	F1 2	User Data	mm/min
13	F1 3	User Data	mm/min
14	F1 4	User Data	mm/min
15	F1 5	User Data	mm/min
16	F1 6	User Data	mm/min
17	F1 7	User Data	mm/min
18	F1 8	User Data	mm/min
19	F1 9	User Data	mm/min
20	Fixed cycle D-code	0	0.001mm
21			
22	DNC COM1 baud rate	9600	9600bps
23	DNC COM1 switch	0 x 1a	0x1a: 7bits, Even
24	DNC EOB	0	0: LF/1 : CR

25	DNC Tr-Time	0	0.1sec
26	Return to Origin Speed at Low speed	120	1mm/min
27	Return to Origin Speed at High speed	600	1mm/min
28	Return to Origin Speed for Evading	600	1mm/min
29			

Work coordinate Parameters

Work coordinate parameters are used to set up the work coordinate #1-5 and are stored in the **WKOFFSET.prm** file.

No.	Meaning/Parameter Function	Input method	Note
1	Work Coordinate #1 X-axis	User Data	0.001mm
2	Y-axis	User Data	0.001mm
3	Z-axis	User Data	0.001mm
4	A-axis	User Data	0.001mm
5	Work Coordinate #2 X-axis	User Data	0.001mm
6	Y-axis	User Data	0.001mm
7	Z-axis	User Data	0.001mm
8	A-axis	User Data	0.001mm
9	Work Coordinate #3 X-axis	User Data	0.001mm
10	Y-axis	User Data	0.001mm
11	Z-axis	User Data	0.001mm
12	A-axis	User Data	0.001mm
13	Work Coordinate #4 X-axis	User Data	0.001mm
14	Y-axis	User Data	0.001mm
15	Z-axis	User Data	0.001mm
16	A-axis	User Data	0.001mm
17	Work Coordinate #5 X-axis	User Data	0.001mm
18	Y-axis	User Data	0.001mm
19	Z-axis	User Data	0.001mm
20	A-axis	User Data	0.001mm
21	Work Coordinate #6 X-axis	User Data	0.001mm
22	Y-axis	User Data	0.001mm
23	Z-axis	User Data	0.001mm
24	A-axis	User Data	0.001mm

Machine Parameters

Machine Parameters are used to control the operation of each axis and to set up program files, and are stored in the **HW-SET.prm** file.

No.	*Meaning/Parameter Function*	*Input method*	*Note*
1	Type	0	0: Normal 1000: Manual operation
2	X-axis Pulse Rate	8192	1 Pulse
3	X-axis Movement per Pulse	5000	0.001 mm
4	Y-axis Pulse Rate	8192	1 Pulse
5	Y-axis Movement per Pulse	5000	0.001 mm
6	Z-axis Pulse Rate	8192	1 Pulse
7	Z-axis Movement per Pulse	5000	0.001 mm
8	A-axis Pulse Rate	1	1 Pulse
9	A-axis Movement per Pulse	1	1 Pulse
10	A-axis Function	0	0: None
11	G31 Type	0	Fixed
12	Acceleration and slow down speeds buffer	20	1-99 (msec)
13	MCOS-ACM board Address	0	0-15
14	Machine default point Offset : X-axis	0	0.001mm
15	Y-axis	0	0.001mm
16	Z-axis	0	0.001mm
17	A-axis	0	0.001mm
18	Reference Point : X-axis	User Data	0.001mm
19	Y-axis	User Data	0.001mm
20	Z-axis	User Data	0.001mm
21	A-axis	User Data	0.001mm
22	Software Limit + X	User Data	0.001mm
23	Software Limit – X	User Data	0.001mm
24	Software Limit + Y	User Data	0.001mm
25	Software Limit – Y	User Data	0.001mm
26	Software Limit + Z	User Data	0.001mm
27	Software Limit – Z	User Data	0.001mm
28	Software Limit + A	User Data	0.001mm
29	Software Limit – A	User Data	0.001mm

30	Stroke Limit + X	User Data	0.001mm
31	Stroke Limit - X	User Data	0.001mm
32	Stroke Limit + Y	User Data	0.001mm
33	Stroke Limit - Y	User Data	0.001mm
34	Stroke Limit + Z	User Data	0.001mm
35	Stroke Limit - Z	User Data	0.001mm
36	Stroke Limit + A	User Data	0.001mm
37	Stroke Limit - A	User Data	0.001mm
38	M-Code Mode	0 x 003d	Normal: 0x 003f
39	External Limit	0 x 0080	
40	Return to origin Mode	0	0 x 0001: X- axis 0 x 0002: Y- axis 0 x 0004: Z- axis 0 x 0008: A- axis
41	Return to origin Way	0 x 07	0: Minus Way 1: Plus Way
42	Automatically return to origin Mode	0	0: Not set automatic return 1: MEM mode after return to origin point. 2: DNC mode after return to origin point.
43	Hardware Type	0	Fixed
44	Extension Control	0	Fixed
45	External input	0	Fixed
46		0	
47		0	
48		0	
49		0	
50	TCP Port Number	6000	
51	Spindle Max. rotation	9000	rpm
52	Spindle fix amount	100	0.1%
53	Delay for Servo break	500	1msec
54	Special External output	0	0: ON by M30

Directory Parameters

Directory Parameters are used to save the program during MEM mode, and are stored in the **DIR-NAME.prm** file. ***Please NOT change any parameters in this file.***

No.	*Meaning/Parameter Function*	*Input Method*	*Notes*
1	Temp file name	Character String	8 Characters or less
2	Data file extension		3 Characters or less
3	Subprogram header		4 Characters or less
4	Subprogram extension		3 Characters or less
5	External program instruction header		6 Characters or less
10	Temporary directory		60 Characters or less
11	Data directory: 1		
12	Data directory: 2		
13	Data directory: 3		
14	Data directory: 4		
15	Data directory: 5		
16	Data directory: 6		
17	Data directory: 7		
18	Data directory: 8		
19	Data directory: 9		

Edit Parameters

① Press PRM to go parameter display.

```
[PRM]          SYSTEM
#01 = 1000
Article IX.
```

② These parameter types are changed by pressing any of the corresponding on File Pchg keys on the operating keypad.

SYSTEM >> **WKOFFSET** >> **H-CODE** >> **D-CODE** >> **HW-SET** >> SYSTEM

Note: Currently H-CODE and D-CODE are NOT available.

③ Use +OV −OV to choose the parameter number and press SELCT BSTOP to confirm.

```
[PRM]          SYSTEM
#03＝200
Article VIII.      >>>>>
```

Ex. Choose system parameter #3.

④ Input the parameter value after “>>>>>” using the numeric keys.

```
[PRM]          SYSTEM
#03＝200
Article VII.       >>>>> 100
```

Ex. Changed system parameter #3 from 200 to 100.

Change From

Change To

⑤ After press ENT START to confirm, the value will be changed and “#” will be displayed right next to the parameter name.

```
[PRM]          #SYSTEM
#03＝100
Article VI.
```

⑥ To reload the parameter, press SAVE RST, and “#” will disappear.

```
[PRM]          SYSTEM
#03＝100
Article V.
```

Alarm

When an alarm occurs, it will be displayed on the unit's LCD screen. Alarms are released by pressing the RESET key on the control panel.

There are also two levels of alarms and the Process/Cancellation conditions are different for each.

Level 1: Warnings

The alarms are from an executing error or RS232c error.

Alarm condition: All axes stop and servo ON.

Level 2: Alarms

These alarms are from servo errors and emergency stop input.

Alarm condition: All axes servo OFF and break ON.

To release the alarms, press 

key and MM100 becomes RESET condition. It is not possible to restart operations automatically because data received to the unit's buffer is lost when the unit goes into alarm status.

Alarm Codes

Warnings (WA01-WA99)

No.	Description
WA01	A Parameter Has Changed
WA02	
WA03	HOST Device Not Ready
WA04	NC Program Error
WA05	RS232C Error
WA06	G31 Error
WA07	Subroutine is Being Processed
WA08	No Instructed Subroutine
WA09	
WA10	X-axis + Software Limit
WA11	Y-axis + Software Limit
WA12	Z-axis + Software Limit
WA13	A-axis + Software Limit
WA14	X-axis - Software Limit
WA15	Y-axis - Software Limit
WA16	Z-axis - Software Limit
WA17	A-axis - Software Limit

WA18	
WA19	
WA20	X-axis + Stroke Limit
WA21	Y-axis + Stroke Limit
WA22	Z-axis + Stroke Limit
WA23	A-axis + Stroke Limit
WA24	X-axis - Stroke Limit
WA25	Y-axis - Stroke Limit
WA26	Z-axis - Stroke Limit
WA27	A-axis - Stroke Limit
:	:
:	:
WA60	Fixed Cycle Error
WA81	Tool Change Error
WA82	Tool Change Error
WA83	Tool Change Error
WA84	Tool Change Error

Alarms (AL01-)

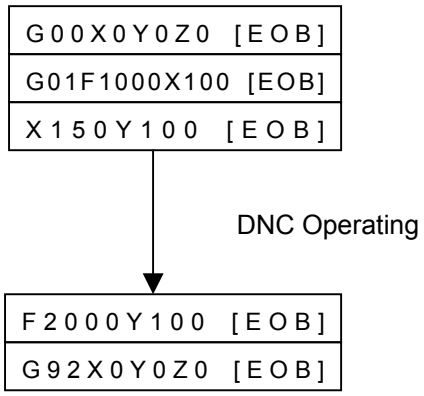
No.	Description
AL01	Failed Servo ON
AL02	X-axis Driver Error
AL03	Y-axis Driver Error
AL04	Z-axis Driver Error
AL05	A-axis Driver Error
AL11	X-axis +OT Limit
AL12	Y-axis +OT Limit
AL13	Z-axis +OT Limit
AL14	A-axis +OT Limit
AL15	X-axis -OT Limit
AL16	Y-axis -OT Limit
AL17	Z-axis -OT Limit
AL18	A-axis -OT Limit
AL20	EM Switch
AL21	Parameter Error
AL22	Fixed Cycle Error

Communication Protocol

DNC Mode

In DNC mode, the controller receives data from a host when the servo is ON and processes the data block upon reception.

DNC Data



Note: *[EOB] = CR/LF*

Reset

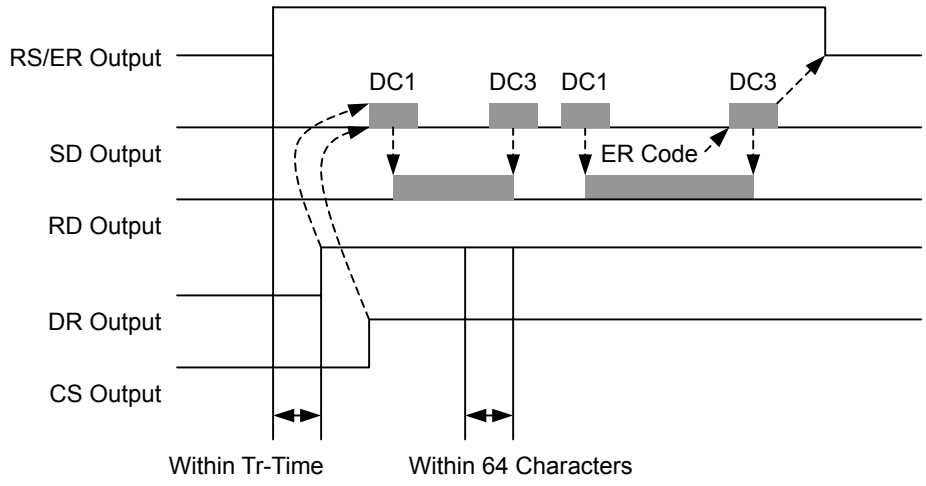
Sending the Reset Code **CAN: 18A** to the controller will reset it. The following once the controller begins to reset.

1. DNC operation will decelerate to a complete stop.
2. The DNC Buffer will be cleared.
3. Further data being sent will be ignored for a given amount of time once the reset code has been received.
4. Alarms that may have occurred during operation will be reset.
5. Program processing will decelerate to a stop and rewind??
6. The reset cod **CAN: 18A** will be sent to the host.
7. Work coordinates offset is set to zero, the origin reference point.

Communication Protocol

When the START key is pressed, communication undergoes the following sequence.

1. ER and RS signals are set to “H”
2. The DR signal is checked for “H” designation. An alarm will occur if the signal has not been set to “H” over a given amount of time; this time may be set under **Tr-Time** in parameter settings. Please note that should the parameter be set to zero, no alarm will occur and instead the protocol will continue to wait for the check to pass.
3. Once the DR signal has been set to “H”, the protocol performs the same check on the CS signal. Once both signals have passed the check, the protocol will send out a DC-1 code.
4. Upon receiving the DC1 code “11H”, external equipment may begin sending data.
5. While data is being sent, should the buffer not have adequate space to receive data, the protocol will send out the DC3 code “13H”.
6. Upon receiving the DC3 code, external equipment must be stopped within 64 characters to avoid an error alarm.
7. Once the buffer has recovered adequate space, the protocol will again send out the DC1 code and any external equipment may begin sending data.
8. When all communications are complete, the protocol will send out a DC3 code and the RS signal will be designated “L”.



RS-232C Parameter

These parameters are stored in the **SYSTEM.PRM** file.

1. Baud Rate: 9600/19200/36000
2. Data: 7/8 bit
3. Parity Check: Odd/Even/None
4. Stop Bit: 1/2 bit

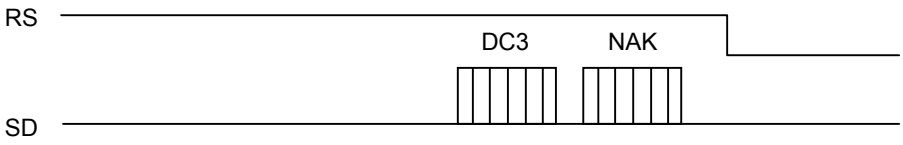
Data Form

1. Data Format: ASCII
2. EOB Code: CR/CF
3. Max. Characters in a single line: 64

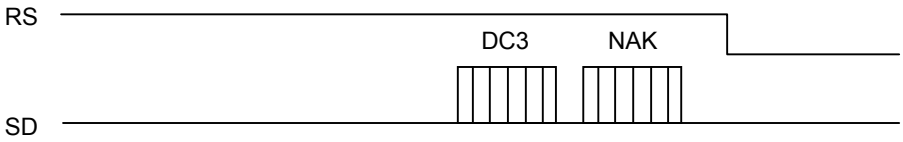
Termination

The data transmitting process terminates under the following circumstances.

- Forwarding the NAK code while processing M02 and M30



- While the machine is stopped, pressing the RESET button will forward the SYN code and terminate the process. The SYN code can also terminate the process during a system alarm.

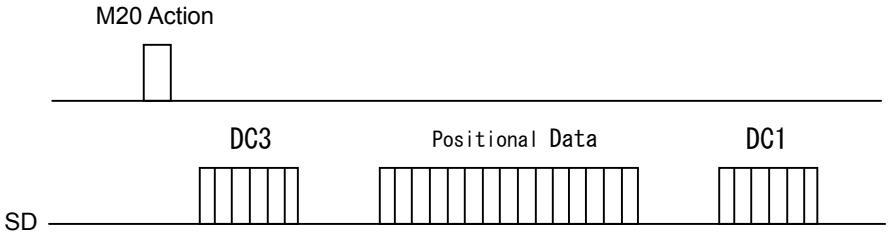


- Upon receiving the second % code after start up (the first % code is ignored).

Note: Please note that under the above circumstances, all data within the data buffer will be eliminated

Position Reporting

When M20 is executed, the program's present position will be sent to the host. A DC3 code will be sent first, followed by the positional data. Once received, a DC1 code is sent out and resumption of data processing begins.



Position Data Format

X	(S)	*	*	*	*	.	*	*	*	Y	(S)	*	*	*	*

X	(S)	*	*	*	*	.	*	*	*	Y	(S)	*	*	*	*

(S) = Minus symbol with **Negative**; Space when **Positive**

* = Numbers; no omission of leading zeros

NC Status Transmission

The controller may send its status information to a HOST on demand with the Controller Status Request **BEL: 07H**.

Statuses

Formatted **Sn** where **n** corresponds to one of the following numbers.

0: Return to Origin Incomplete

1: Return to Origin Complete

2: Resetting Origin

3: DNC Operating

4: Processing Test Program

5: Temporarily Stopped

6: Pulse Mode

7: Servo OFF

8: Alarm 1

9: Alarm 2

Return Statuses

H0: Return to Origin Incomplete

H1: Return to Origin Complete

Alarm Statuses

Formatted **Enn** where **nn** corresponds with one of the following numbers.

0: No Alarm

1 – 99: Corresponding Alarm Number

Sequence Number

Formatted **Nnnnn** where the sequence ranges from **1 – 999999** (should be 99999?)

Present Position

Xnnnnnn: Current position of X-axis (work coordinate value) in 0.001mm units.

Ynnnnnn: Current position of Y-axis (work coordinate value) in 0.001mm units.

Znnnnnn: Current position of Z-axis (work coordinate value) in 0.001mm units.

Znnnnnn: Current position of A-axis (work coordinate value) in 0.001deg units or spindle rotation in Rounds per Minute.

External Output Status

Onnn where **nnn** is designated in HEX (0 – F)

OUT1 ~ OUT12

OT-Switch Status

Tnnnn: where 0 = OFF / 1 = +OT / 2 = -OT (X, Y, Z, A axes)

External Input Status

Innnnn where **nnnnnn** is designatd in HEX (0 – F)

IN1 ~ IN24)

The following is an example of what a complete status output my look like, please make note of the bolded characters and make use of the above descriptions to help you understand the following character string:

Ex. **S**3**H**1**E**0**N**1010**X**1001**Y**106**Z**5128**A**10000**T**0000**I**1238AF

External Interface

External Output

External Output: CN12

External Output	Name / Function	Connector
OUT1	Servo ON	– 1
OUT2	RUN	– 2
OUT3	Alarm	– 3
OUT4	Completed Returning to home position	– 4
OUT5	Spindle ON	– 5
OUT6	Coolant ON	– 6
OUT7	M30 Output	– 7
OUT8	Completed Instructing External Program	– 8
P24	OUT-COM (DC24V)	– 9
N24	GND	– 10
OUT9	M-Code 01H	– 11
OUT10	M-Code 02H	– 12
OUT11	M-Code 04H	– 13
OUT12	M-Code 08H	– 14
OUT13	M-Code 10H	– 15
OUT14	M-Code 20H	– 16
OUT15	M-Code 40H	– 17
OUT16	M-Code MF Output	– 18
P24	OUT-COM (DC24V)	– 19
N24	GND	– 20

Power Quality

Open Collector Output

Photo Coupler: Toshiba TLP521 or equivalent

Transistor: Toshiba ‘TD62082

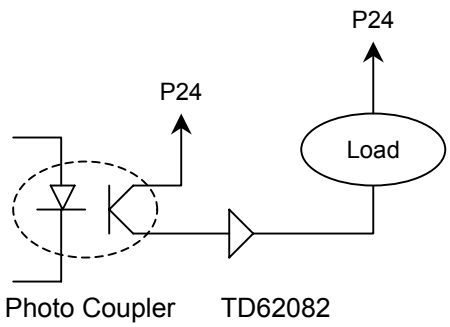
Output System: NPN Open Collector Output

Regular Load Voltage: DC24V

Regular Load Current: 100mA

Total Load Current: 400mA

Leakage Current: less than 1mA



Relay Output

Break Output

Relay: G6B1114P (Omron: 5A30VDC)

Regular Load Voltage: DC24V

Regular Load Current: 2.0A

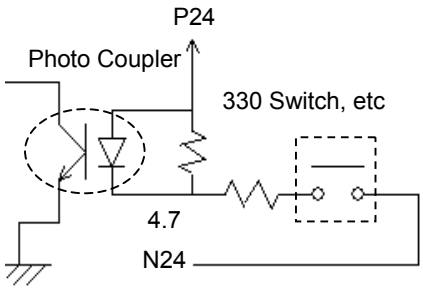
External Output: CN13

External Output	Name / Function	Connector
IN1	Control Signal: START	– 1
IN2	Control Signal: STOP	– 2
IN3	Control Signal: RESET	– 3
IN4		– 4
IN5	M-Code MFIN	– 5
IN6		– 6
IN7		– 7
IN8		– 8
P24	DC24V	– 9
N24	IN-COM (GND)	– 10
IN9	External Program Number 01H	– 11
IN10	External Program Number 02H	– 12
IN11	External Program Number 04H	– 13
IN12	External Program Number 08H	– 14
IN13	External Program Number 10H	– 15
IN14	External Program Number 20H	– 16
IN15	External Program Number 40H	– 17
IN16	External Program Number Select Program	– 18
P24	DC24V	– 19
N24	IN-COM (GND)	– 20
EM	Input EM	– 21
N24	IN-COM (GND)	– 22
SYSIN	Input SYSIN	– 23
N24	IN-COM (GND)	– 24
P24	DC24V	– 25
N24	IN-COM (GND)	– 26

Power Quality

Photo Coupler Input

Regular Input Voltage: DC24V

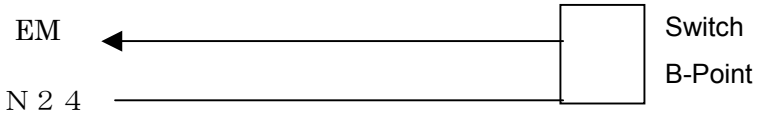


Input Signal Source

By non-voltage point or open collector

ON/OFF Input

40V Durable Pressure and open-and-shut capacity with 17mA or more electric current are necessary.



RS232C Interface

The RS232C interface can be used for communication between the unit and any other equipment. However, please do not use any other pins except the ones displayed below.