

Table of Contents	
Welcome and Congratulations	
Important Safety Instructions	
Parts of REVO M1A	
Installation steps	
MASSO Interface	
FAQ	

Welcome and Congratulations

You are now the proud owner of a REVO M1A, an incredibly powerful and easy-to-use CNC machine. In this guide we will walk you through everything you need to know to get started using your M1A

For those of you who are new to CNC machining, we recommend you to visit our youtube channel for basics of cnc.

Warranty

Every Revo product has 2year warrant against manufacturing faults. Visit our warranty site for M1A warranty details.

Glossary of Terms

As you're following the setup process in this guide, you may run into CNC terminology you're unfamiliar with.

Consult the Glossary of Terms at the end of this document for common CNC terms and definitions.

Important Call-Outs Used in This Guide

Throughout the guide, you will find information that we've called out for you to pay particular attention to. We use

three types of call-outs: Warnings, Notes, and Pro Tips:

WARNING: This is a warning. Information in these boxes is VERY important. Pay close attention.

NOTE: This is a note—information that points out critical steps or information for future reference.

PRO TIP: This is a Pro Tip. Anytime you see one of these, you will find helpful additional information.

Important Safety Instructions

REVO M1A is a serious machine that should be treated like any other power tool. Always follow safe-machining practices:

- Always wear safety glasses when your M1A is on.
- Always wear appropriate hearing protection. Hearing damage is cumulative and irreversible, so it is important to always err on the side of caution.
- Always keep the protective door closed while your M1A is machining.
- Never leave your M1A unattended while it is machining.
- Never reach into the machine while it is running.
- End mills are sharp and should be handled with care.
- Always ensure the end mill is securely held by the collet before machining.
- Always use common sense.

In the event that you need to stop the machine right away, press the E-STOP button and to turn off the M1A completely, use the Main Power Switch.

Additional Safety Recommendations

- Consider the possibility of a fire caused by friction from the spindle and take suitable fire prevention precautions (e.g., having a fire extinguisher handy and other suitable precautions).
- Plug M1A into an outlet that has a dedicated on/off switch. Be sure this is accessible while the machine is running, in case you should need to shut the machine off immediately.
- Recycle or safely dispose of milling debris and dust, keeping in mind flammability, (potential) spontaneous combustion, and chemical considerations. Even natural materials can have surprising disposal implications.

For example, walnut wood dust is allelopathic (it inhibits plant growth), an irritant to the skin and respiratory tract, and potentially poisonous to some animals. All of these possible disposal implications are in addition to the spontaneous combustion hazard posed by all types of sawdust.

Parts of REVO M1A

Buraya fotoğraf eklenecek teknik çizim, ölçüler, ağırlık, aksesuarlar vs

Power Cable

Pneumatic hose

Pneumatic Regulator

Coolant container

Masso Touchscreen

Hand wheel

Work Table

Spindle

Nut

Collet

Tool

Collet Wrench

Shaft Wrench

Tool Holder

Probe

Getting Started

Unpack the M1A, place it on your workspace, plug the power cable, plug the pneumatic hose, fill the cooling liquid, adjust the pneumatic pressure, turn the power lever and M1A is ready to work.

But first, there are preparations to make and checklists to fill. The following list contains necessary preparations for the installation of the REVO M1A. Check the list and make sure all the boxes are ticked before starting the installation process. For any missing item on the list, please read the related notes and take necessary actions.

	Transportation to the installation site is safe and usable
	Machine fits inside the installation site
	Doors, colons and elevators are wide enough to transport the machine
	Electric supply is ready
	Independent ground connection is available
	Pneumatic supply is ready
	An air dryer has been installed on pneumatic supply
	Internet connection is available
	Air conditioning and ventilation is available
	Dust collector is available (optional)

Transport

Control of the route for a smooth transport to the installation site should be done.

The width and load dimensions of each area that the machine will pass through please check the capacity is sufficient.

Check the following:

- Entry and parking of the transport vehicle
- The path to be followed inside the building (sills, doors, corridors)
- Stairs and/or Elevator
- Floor conditions and ground

Before you transport the M1A to the installation site, make sure there is no obstacle along the way.

Installation Site

If the machine is to be installed in a residential area, the building must be inspected by a civil engineer for the suitability of the bearing capacity of the ground.

- Check the weight of the machine.
- Take into account that M1A could create dynamic loads up to 500N in 0-5Hz.
- M1A should be installed on a flat surface.
- The ground must be sufficiently firm, rigid and smooth.
- M1A mustn't be screwed into the floor.

Pro tip: For ease of access and operational requirements, there must be enough empty space on all sides of the machines as a rule of thumb. Make sure there is enough clearance around the M1A. Please check the clearance requirements on Dimensions and Weight table.

Also check the occupational health and safety regulations regarding occupational hazards and emergency exit routes.

Note: Before you plant the machine, make sure installation site complies with the conditions described above.

Room Conditions

Note: The air of the workspace must be dry and dust free.

Requirements	Values
Temperature	15-30°C
Moisture	<65%

Make sure room temperature is sufficient. Ideal room temperature is 20-23°C.

Note: Workable room conditions must be established before commissioning the M1A.

Warning: If there is a possibility of frosting during storage and transport, all the liquid of the machine must be drained.

Ventilation

The installation site must have a good ventilation system. When processed, chips of processed materials that can produce harmful dust and must be collected with the dust extraction unit. Examine the OHS laws.

Type	Ventilation of the total room volume per hour
Workshops	3-6
Workshops working with solvents, chemicals	10-18
Laboratories	5-15
Garages	3-5

Connections

Open or loose machine connections can damage the machine and/or may affect its functions.

- * Make sure there are no open or loose connections.
- * Fix the cables and hose lines to be connected to the machine.
- * Place the machine as it can be accessed to the board and connection ports.

Electrical connection

For a trouble-free operation, the power line must comply with TS EN 60204-1 standards.

Voltage/Current	230V/11A
Frequency	50Hz. / 60Hz.
Energy Consumption	2,5 kw

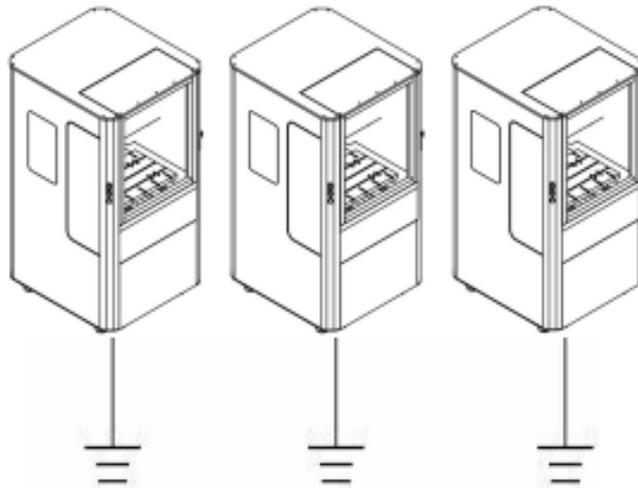
M1A may cause electromagnetic interference, which may affect other electrical equipment in the environment.

- If necessary, use noise suppressing devices in accordance with TS EN IEC 61800-3 standards.
- If the power supply has a residual current relay (RCD or FI), it must be type B.
- M1A must have a separate power line; no other device should be operated using the same power line. (e.g., Phone charger, electric heater, refrigerator, control computer)
- Working with electrical equipment requires certain technical knowledge and skill, otherwise there is a risk of electric shock and death.
- Electrical work should be done by a qualified electrician.

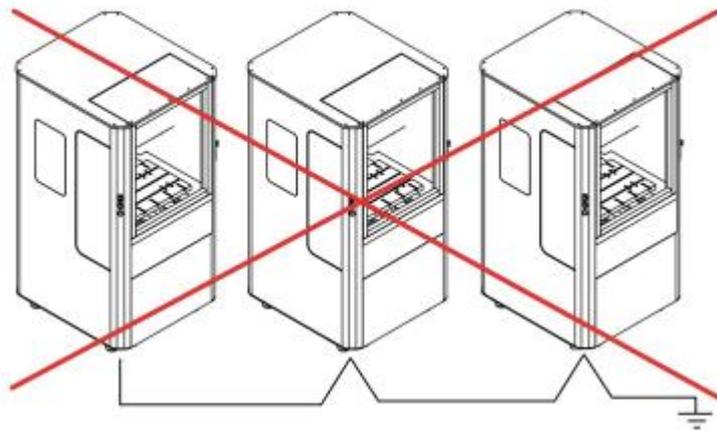
Grounding

- The REVO M1A requires 230V/11A electricity and operational grounding (Class 3) for operation.
- All electrical devices must be grounded to protect workers and the machine from electrical hazards.
- Grounding should be done in accordance with the standards of electrical appliances immediately after the machine is installed.

WARNING The grounding process should be left to the responsibility of the electrical authority. Failure to do so may result in serious injury, death or damage to the machine.



WARNING! Multiple ground conductor wires cannot be connected to a single terminal.



Note: M1A is using a standard 220V power connection. Before you plug the power cable, make sure electric line complies with the conditions described above.

Compressed air connection

REVO M1A uses compressed air for cooling and automatic tool changing. Even if your machine doesn't have a automatic tool changer, your machine needs the air supply.

- Make your compressor selection so that it can meet the needs of M1A continuously.
- Depending on the compressor type, the compressor capacity should be much higher than the machine requirement so that the compressor can cool down sufficiently.
- For compressor recommendation, please contact the relevant companies.

Air pressure	6-8 bar
Air consumption	200 L/hr max

Note: The air should be dry, clean and oil-free. Otherwise, damage to the components through which air passes may occur.

Air Purity	Class	Maximum Values
Solid particles	Class 3	5 μm , 5mg/m ³
Liquid content	Class 4	Dew pressure +3°C
Oil content	Class 3	1 mg/ m ³

You can use any air cleaning unit that can meet the conditions above.

Note: REVO highly recommends using an air filter system at the air inlet of the machine!

Warning: If the air coming into the machine doesn't comply with the industry standards (described above), the equipment using the airline will be out of warranty against damage that may occur as a result of interaction with the unpurified air.

REVO M1A has a 6mm compressed air inlet at the back panel of the machine. Before you connect the line, make sure air supply complies with the conditions described above.

Cooling Liquid

REVO recommends ethanol-based cooling liquid for better cutting speeds, longer tool life and better surface finishes on workpiece.

Select the refrigerant chemical suitable for the operation to be performed.

Pro Tip: Some processes require special coolants, while some processes may give better results without using a coolant. For detailed information, attend relevant trainings and seminars.

Use a different chemical in processes such as steel and similar materials that may generate sparks and that may cause fire from accumulated chips while being machined. (e.g., fatty alcohol)

Warning: The usage of Ethanol in machines with a dust extraction system carries the risk of explosion in the vacuum motor!

For best results when machining non-ferrite metals, use the following chemicals:

- Ethanol (99%)
- Ethyl Alcohol 642
- EEC Designation 200-578-6 (EINECS)

Pro Tip: Optimal coolant consumption is 50ml/h, maximum is 500ml/h which can be adjusted via coolant adjustment knob.

Note: Do not use drilling and cutting oils containing solids that tend to form flocculants. (e.g., cutting oil, Bohr oil)

Warning: REVO M1A is not designed for this type of chemicals and machine components may wear out or malfunction. Use of such chemicals will void your warranty.

Note: REVO machines are exported outside the European Economic Area without refrigerants and chemicals. In these cases, the customer must supply the appropriate chemicals.

When using a refrigerant chemical not recommended by REVO, review the chemical manufacturer's specifications for fire and explosion risks.

If in doubt, consult an OHS specialist. Storage containers of refrigerant chemicals must be labeled. Only use chemicals recommended by REVO, your machine is designed to work with these chemicals.

The use of chemicals that we do not recommend may cause bad results, damage the machine, and void your warranty.

- Make sure you have adequate ventilation when using refrigerant chemicals.
- Be sure to examine the technical documents of the refrigerant you will use.

Turn on The Main Power Switch

If you have followed the instructions and established required conditions so far, it is time to power on the M1A.

Main power switch is located at the back side of your M1A.

Turn on the main power switch, Touchscreen will turn on automatically and you will see the Masso interface in less than 10 seconds.

Masso Control Unit



M1A comes with Masso control unit as standard.

Following pages will guide you through the Masso interface.

When in doubt, check FAQ then consult REVO.

Check www.massoc.com.au for additional information.

F1 Screen

This screen allows you to configure your Masso settings.

When you enter this screen, you will be prompted for the Admin Password.

Note: Default Password is "HTG"

If you do not enter the password, you will enter read only mode where you will not be able to make changes. Use CTRL+L if you want to reenter the password.

The screenshot displays the F1 Setup screen with the following sections:

- Navigation Tabs:** F1 SETUP, F2 PROGRAM & MDI, F3 JOG & PROBING, F4 TOOLS & OFFSETS, F5 CONVERSATIONAL, F6 LOAD FILE.
- Machine Settings (Left Panel):**
 - General Settings (Selected)
 - Homing
 - Spindle
 - Lubrication
 - Tool Changer
 - X - Axis
 - Y - Axis
 - Z - Axis
 - A - Axis
 - B - Axis
 - Touch Probe
 - Auto Tool Zero
 - User Account
 - Save & Load Settings
- Serial Information (Bottom Left):**

MASSO Serial No: G3-6182
 Core: v2
 Software: v4.01.3a
 www.massco.com.au
 support@massco.com.au
- Inputs Table (Middle):**

Inputs	Function	Invert	Status
EStop	EStop	No	High
Encoder	Signal - A	No	Low
Encoder	Signal - B	No	Low
Encoder	Index	No	Low
MPG	Dial Signal - A	No	Low
MPG	Dial Signal - B	No	Low
MPG	Select X	No	Low
MPG	Select Y	No	Low
MPG	Select Z	No	Low
MPG	Select A	No	Low
MPG	Select B	No	Low
MPG	Resolution 1	No	Low
MPG	Resolution 2	No	Low
MPG	Resolution 3	No	Low
Analog	Input 1		0.00v
Analog	Input 2		0.00v
Input 1	Tool Setter Input	Yes	Low
Input 2	Probe Input Signal	Yes	Low
Input 3		Yes	Low
Input 4		Yes	Low
Input 5		Yes	Low
Input 6		No	Low
Input 7		No	Low
Input 8	B - Home Sensor Input	Yes	Low
Input 9	Y - Home Sensor Input	Yes	Low
- Outputs Table (Right Panel):**

Outputs	Function	Invert	Status
Spindle	CW	No	Low
Spindle	CCW	No	Low
Output 1		No	Low
Output 2		No	Low
Output 3		No	Low
Output 4		No	Low
Output 5		No	Low
Output 6		No	Low
Output 7		No	Low
Output 8		No	Low
Output 9		No	Low
Output 10		No	Low
Output 11	Auxiliary Output 1	No	Low
Output 12		No	Low
Output 13		No	Low
Output 14	Tower Light - Red	No	Low
Output 15	Tower Light - Yellow	No	Low
Output 16	Tower Light - Green	No	Low
Output 17		No	Low
Output 18		No	Low
- Virtual Keyboard (Bottom):** A standard QWERTY keyboard layout with function keys like !#1, -, ., and a back arrow.

F2 Screen

This is the Run screen where you run your G-Code files and monitor progress.

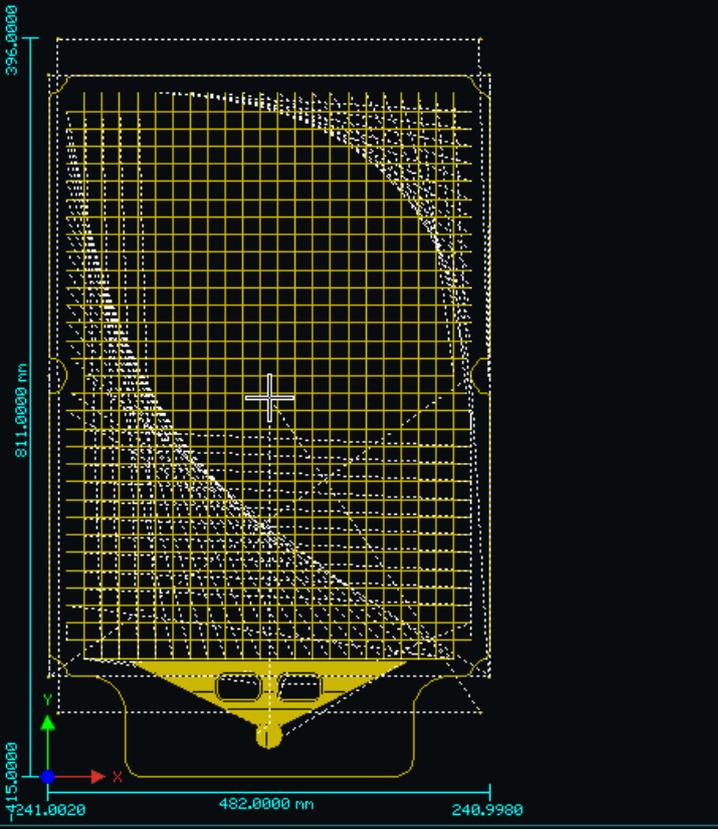
MASSO G3 Mill 5-Axis v4.01.3a Probe Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 1 Wi-Fi 11:09 PM

F1 SETUP F2 PROGRAM & MDI F3 JOG & PROBING F4 TOOLS & OFFSETS F5 CONVERSATIONAL F6 LOAD FILE

READY MACHINE READY X ZERO 0.000 mm A ZERO 0.000 deg B ZERO

DOOR DOOR CLOSED Y ZERO 0.000 mm Feed: 0, 100% mm/min

E-STOP Z ZERO 0.000 mm Tool: 3, T3



SPINDLE		MACHINE	
RPM: 0	X	0.000	mm
Req: 0, 100%	Y	0.000	mm
Direction: STOP	Z	0.000	mm
	A	0.000	deg
	B	0.000	deg

Vacuum Table(Vacuum Table Bottom.nc
 (Vacuum Table Bottom)
 (File created: Saturday February 22 2020 - 10:29 PM)
 (For Masso from Vectric VCarve Pro / Aspire 9.5)
 (Post Processor version Metric v1.2)
 (Material Size)
 (X= 567.000 , Y= 940.000 , Z= 19.000)
 ()
 (Toolpaths used in this file:)
 (Drill Mounting Holes)
 (Drill Top fixing Holes)
 (Seal)
 (V Top of hole)
 (Slots)
 (Vacuum Pocket)
 (Vacuum entry)
 (Tools used in this file:)
 (1 = End Mill {6 mm})
 (2 = V-Bit {90 deg 0.5"})
 (3 = End Mill Down cut {0.25 inch})
 N190 G00

Spindle CCW Spindle STOP Spindle CW

Single Block Coolant Flood Go to Work Origin Jump to Line

Cycle Start CTRL+S Feedhold Esc Rewind CTRL+R MDI CTRL+M

File Loaded

1 2 3 4 5 6 7 8 9 0
 q + w e r = t / y - u < i > o [p]
 a ! s @ d # f % g ^ h & j * k (l)
 ↑ z - x ' c " v : b ; n , m ? ↵
 !#1 - . ↵

- When you enter this screen, you will be prompted for the User Password. Default Password is "HTG". The user password can be changed or removed.
- Use CTRL+L to lock the Screen and prevent unauthorized use.
- The empty box next to the Spindle CW is where you can enter your spindle speed. Type the desired speed into the box and press enter.
- Click on Wi-Fi at the top of the screen to enter the Wi-Fi configuration page. If Wi-Fi is Orange it is connected.
- Click the Axis Zero buttons to zero out an Axis DRO
- Click on the DRO value to manually enter a new value
- Click Optional Stop at the top of the screen to toggle between on and off.

F3 Screen

This is the Jog Screen. From this screen you can Home and Jog your machine.

You can access MASSO probing functions by pressing the probing button on screen.



- Press the Home Button for 3 seconds to home the machine
- Jog and using the Axis + & - Buttons
- Select continuous or step jog mode as well as step distance
- Use the Slider to set continuous jog speed.
- Click the Axis Zero buttons to zero out an Axis DRO

F4 Screen

Tool Table, work offset screen and Park location.

IASSO G3 Mill 5-Axis v4.01.3a
Probe
Work Offset: G54
MPG AXIS: OFF
Optional Stop: On
Jobs: 1
11:08 PM

F1 SETUP
F2 PROGRAM & MDI
F3 JOG & PROBING
F4 TOOLS & OFFSETS
F5 CONVERSATIONAL
F6 LOAD FILE

Tool No	Slot No	Tool Name	Z Offset	Tool Diameter
0		T0	0.00000	0.00000
1		T1	-63.74525	0.00000
2		T2	-41.00856	0.00000
3		T3	0.00000	0.00000
4		T4	0.00000	0.00000
5		T5	0.00000	0.00000
6		T6	0.00000	0.00000
7		T7	0.00000	0.00000
8		T8	0.00000	0.00000

Work Offset	Work Offset Name	X	Y	Z	A
G 54		0.00000	0.00000	0.00000	0.00000
G 55		168.48037	85.00547	43.74394	-91.18800
G 56		0.00000	0.00000	0.00000	0.00000
G 57		0.00000	0.00000	0.00000	0.00000
G 58		0.00000	0.00000	0.00000	0.00000
G 59		0.00000	0.00000	0.00000	0.00000
Parking	PARKING POSITION	0.00000	1170.00000	4.00000	0.00000

1 2 3 4 5 6 7 8 9 0
 q + w e r = t / y - u < i > o [p]
 a ! s @ d # f % g ^ h & j * k (l)
 ↑ z - x ' c " v : b ; n , m ? ↵
 !#1 - . ↵

F5 Screen

Masso have built in wizards that will allow you to create basic G-code files. The wizards are intended for the most basic of jobs and CAM software is recommended. Additional information on the available [wizards](#).

MASSO G3 Mill 5-Axis v4.01.3a Probe Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 1 11:07 PM

F1 SETUP F2 PROGRAM & MDI F3 JOG & PROBING F4 TOOLS & OFFSETS F5 CONVERSATIONAL F6 LOAD FILE

Add/Edit Delete Move Up Move Down Copy

Wizard file Name:
Description Date Output Gcode

Wizard No.	Name	Start X	Start Y
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			

Save Wizard Post Gcode New Wizard

1 2 3 4 5 6 7 8 9 0
q + w e r = t / y - u < i > o [p]
a ! s @ d # f % g ^ h & j * k (l)
↑ z - x ' c " v : b ; n , m ?
!#1 - . ↵

F6 Screen

This screen is where you will select files from your USB Flash drive and load them into MASSO.

The screenshot displays the MASSO G3 F6 screen, which is used for loading files from a USB flash drive. The interface includes a top status bar, a function key menu, machine status indicators, coordinate readouts, and a central file management area.

Top Status Bar: MASSO G3 Mill 5-Axis v4.01.3a Probe Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 1 Wi-Fi 11:07 PM

Function Key Menu:

- F1 SETUP
- F2 PROGRAM & MDI
- F3 JOG & PROBING
- F4 TOOLS & OFFSETS
- F5 CONVERSATIONAL
- F6 LOAD FILE

Machine Status Indicators:

- READY MACHINE READY
- DOOR DOOR CLOSED
- E-STOP

Coordinate Readouts:

- X ZERO 0.000 mm
- Y ZERO 0.000 mm
- Z ZERO 0.000 mm
- A ZERO 0.000 deg
- B ZERO

Feed and Tool Information:

- Feed: 0, 100% mm/min
- Tool: 3, T3

SPINDLE and MACHINE Status:

SPINDLE		MACHINE	
RPM: 0		X	0.000 mm
Req: 0, 100%		Y	0.000 mm
Direction: STOP		Z	0.000 mm
		A	0.000 deg
		B	

NOTE: From software v4.0, CAPS lock will be in off state on system power up.

MASSO

Coordinate System:

File Management Icons:

- Load
- Edit
- New
- Delete

Virtual Keyboard:

1 2 3 4 5 6 7 8 9 0
q + w e r = t / y - u < i > o [p]
a ! s @ d # f % g ^ h & j * k (l)
↑ z - x ' c " v : b ; n , m ? ↵
!#1 - . ↵

- Double click the file you wish to load or select the file or press the Load key.
- Once the file loaded it will draw the toolpaths onto the screen for you to view.
- If you do not wish to wait for the screen print to load or wish to cancel the selected file from loading press the Escape key.
- To Delete a file, select and press Delete.
- To create a new File Press New.
- To edit a file, select and Press Edit

Controller Alarms

E-Stop alarm

An E-Stop alarm is given on startup of Masso. Press and release the E-Stop button to test the E-Stop is working and this will cancel the alarm.

E-Stop also occurs if the E-Stop button is pressed on the machine.

Homing alarm

This alarm indicates that the machine has not been homed. To clear the alarm home the machine.

If you get a homing error alarm it means something has gone wrong while homing. Rehome the machine to clear

Door Alarm

This alarm indicates that the cabinet door is open. Close the door to clear the alarm.

M1A is automatically set to stop when the door is opened.

Soft Limit

The extremes of travel of each axis is defined in each axis settings page.

If you execute a move that will exceed the limit of travel a soft limit will alarm will display.

To cancel issue a valid move on the axis and the alarm will clear.

Hard Limit

These alarms are triggered by the homing sensors. If a mechanical fault occurs that causes the axis to trigger a hard limit the Hard Limit alarm will be displayed.

To clear the alarm, find and clear the issue that caused the Hard limit alarm and Home the machine to clear the alarm.

Probing alarm

A Probing alarm occurs during a probing cycle where Masso does not receive a signal from the probing sensor within the specified distance.

You will also get a probing alarm if the probe is already in the active state when a probing cycle is started.

To clear issue a new probing command with a valid distance ensuring the probe is not already in an active state.

Tool Error Alarm

This occurs if you try and load a tool not supported by Masso.
(E.g., the tool number is greater than 99.)

Drive Alarm

When a Stepper or Servo drive goes into an alarm condition the Drive can signal Masso which will put the system into Feed-hold and stop the spindle.

The Drive alarm indication will be displayed on the screen. you drive must be capable of providing the required signal and they be connected to Masso.

To clear the alarm first clear the fault with the drive and home the machine to clear Alarm indication on Masso.

Lubricant Low alarm

Top up your lubricant vessel and the alarm will automatically clear.

Shortcuts

Most of the functions can now be accessed easily with mouse but will still require some keyboard shortcuts as below:



Operation	Key Combination	Valid in Screen
Lock System	CTRL + L	All Screens
Home Machine	CTRL + ALT + Home key	F2 & F3
Move to Parking Position	CTRL + ALT + P	F2 & F3
Cycle Start	CTRL + S	F2
Cycle Stop	Escape	F2
Restart Program	CTRL + R	F2
Jump to G-Code line	CTRL + J	F2
Jog Step Size	1,2,3 or 4	F3 (MASSO G3 Only)
X – Rapid	SHIFT+ Left or Right arrow key	F3
Y – Rapid	SHIFT + Up or Down arrow key	F3
Z – Rapid	SHIFT + “U” or “D” keys	F3
X – Jog	Left or Right arrow key	F3
Y – Jog	Up or Down arrow key	F3
Z – Jog	“U” or “D” keys	F3
Feed Override	select “F11”	F2 & F3
Spindle Override	select “F12”	F2
Override Increase	“+” (You can also use MPG)	F2 & F3
Override Decrease	“-” (You can also use MPG)	F2 & F3
Optional Stop On/Off	CTRL + O	All Screens
Open MDI Window	CTRL + M	F2 & F3
Create New Gcode File	CTRL + N	F2
Edit Gcode File	CTRL + E	F2
Set X-Axis to zero	CTRL + X	F3
Set Y-Axis to zero	CTRL + Y	F3
Set Z-Axis to zero	CTRL + Z	F3
Screen Print	Print Screen Key	All Screens
Screen Print	CTRL + P	All Screens

Setting Time

MASSO has a built-in real time clock and to set system time, a special command in the MDI window can be given in this format Time: HH: MM.

Note: If the MASSO controller is connected to a PC via Wi-Fi and using the MASSO Link software on the PC then the time is automatically synced with the PC time. You can find more information about MASSO Wi-Fi connectivity and MASSO Link software [here](#).

To manually reset the Time:

- Go to the **F2 - Program & MDI** screen.
- Press **CTRL + M** or click the MDI button to open the **MDI** Window.
- Type **Time: HH:MM** and press **Enter**.

Homing Your M1A

Homing your CNC machine is the process of sending it to a known, fixed, repeatable location. This means that every time you home the machine, it will move to EXACTLY the same position. Every. Single. Time.

This allows you to move your machine to ANOTHER position, relative to the home position, with great precision (0,01mm/m). On your M1A, the home position is the top left corner.

- To home the machine, first make sure that no alarms such as E-Stop are flashing on the screen.
- Next check that it's safe to run the homing cycle on the machine and the tool, work piece or clamps will not be hit.
- If you need to move the axis to a safe position before homing, jogging from the F3 screen or MPG is allowed.
- Once you are ready to home the machine, press CTRL+ALT+HOME key on the keyboard **OR** hold the HOME button in the F3 screen for 3 seconds.

Note: When homing is complete, your spindle will be in the top left corner and the table will be at the front of the machine.

Rapid/Jog

Press F3 to go to “F3 – Jog/Rapid” screen. For Jogging use Arrow keys and for Rapid movement press and hold the SHIFT key and use Arrow keys.



Note: MPG can also be used in the F1 screen to jog the axis.

Feed rate override

Feed rate override function allows the feed rate to be changed during machining. The range is from 20% to 100% of the specified feed rate and the current feed rate percentage is shown on the screen.

Note: This feature is available on the F2 & F3 Screens.

Note: Feed rate override ranges from 20% to 100% of the specified Feed rate. Feed rate override cannot exceed 100%

- Press the F11 Key on the keyboard
- Use the + & - keys on your keyboard to change the feed rate.
- You can also use the Pendant to change the feed rate by rotating the MPG dial in or - direction to increase or decrease the feed rate.

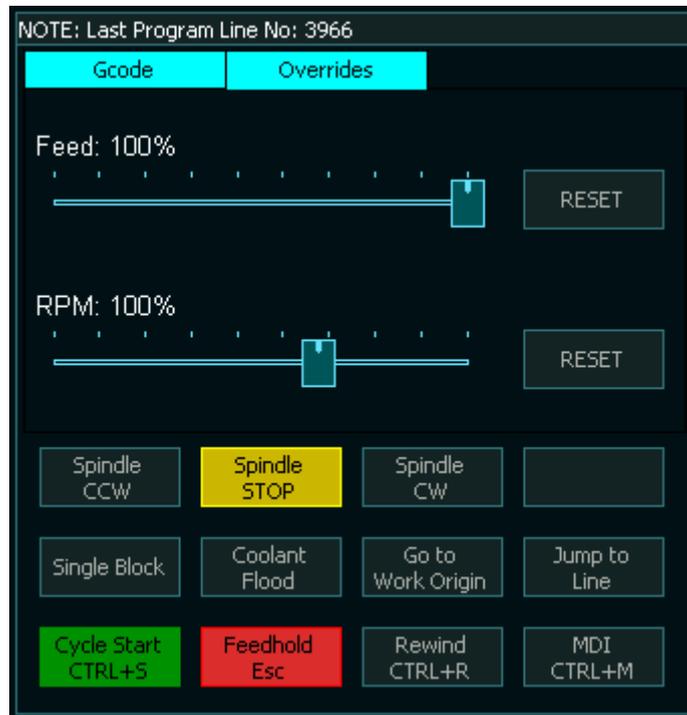
Feed: 0, 100% **mm/min**
Tool: 7,

In addition to the methods above, the MASSO G3 and MASSO Touch has an Overrides tab in the G-code window

The Feed override slider that can be used to change the feed rate between 20% & 100%.

Moving the slider will adjust the feed rate in real time while you are machining.

The Reset button will reset Feed to 100%



Spindle RPM Override

Note: This feature is available on the F2 & F3 Screens.

Speed override ranges from 10% to 150% of the specified spindle speed.

How to use

- Press the F12 Key on the keyboard
- Use the + & - keys on your keyboard to change the spindle RPM.

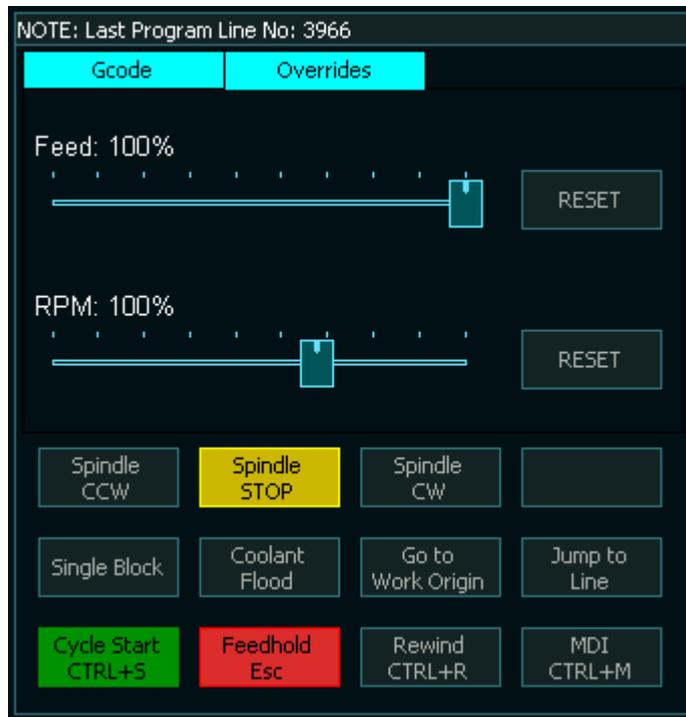
You can also use the Pendant to change the speed by rotating the MPG dial in either the + or - direction to increase or decrease the spindle RPM.



RPM Override set at 100%

In addition to the methods above the MASSO has an Overrides tab in the G-code window

- The RPM override slider that can be used to change the Spindle speed between 10% & 150%.
- Moving the slider will adjust the spindle speed in real time while you are machining.
- The Reset button will reset RPM to 100%

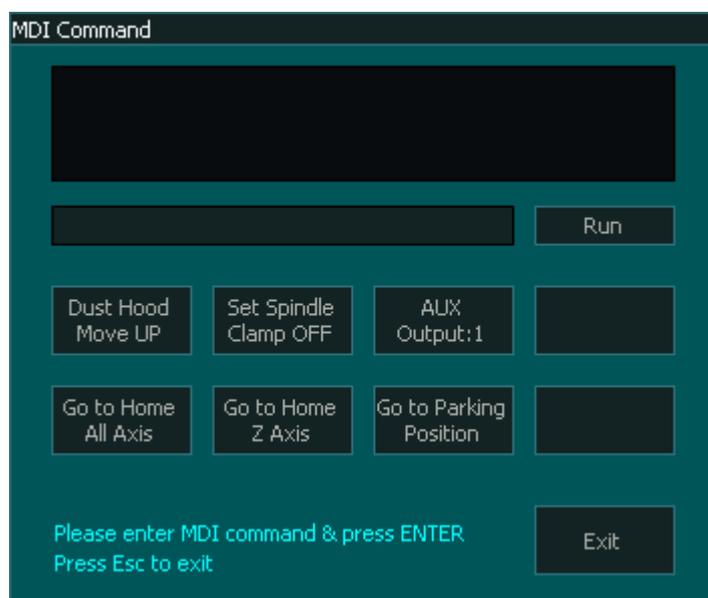


MDI command

MDI window can be opened from F2 - Program & MDI screen using CTRL+M or by clicking on the MDI button towards the bottom left of the screen.

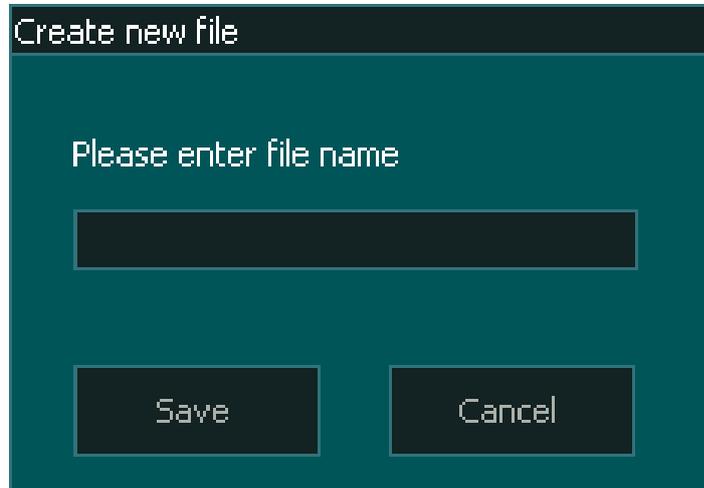
MDI window can be used to quickly run any g-code commands and the on-screen buttons can be used to control outputs or move the axis to the home position.

Pro Tip: In MASSO G3 the up and down arrow key can be used to select and execute commands from history. The last 20 commands are saved in memory.



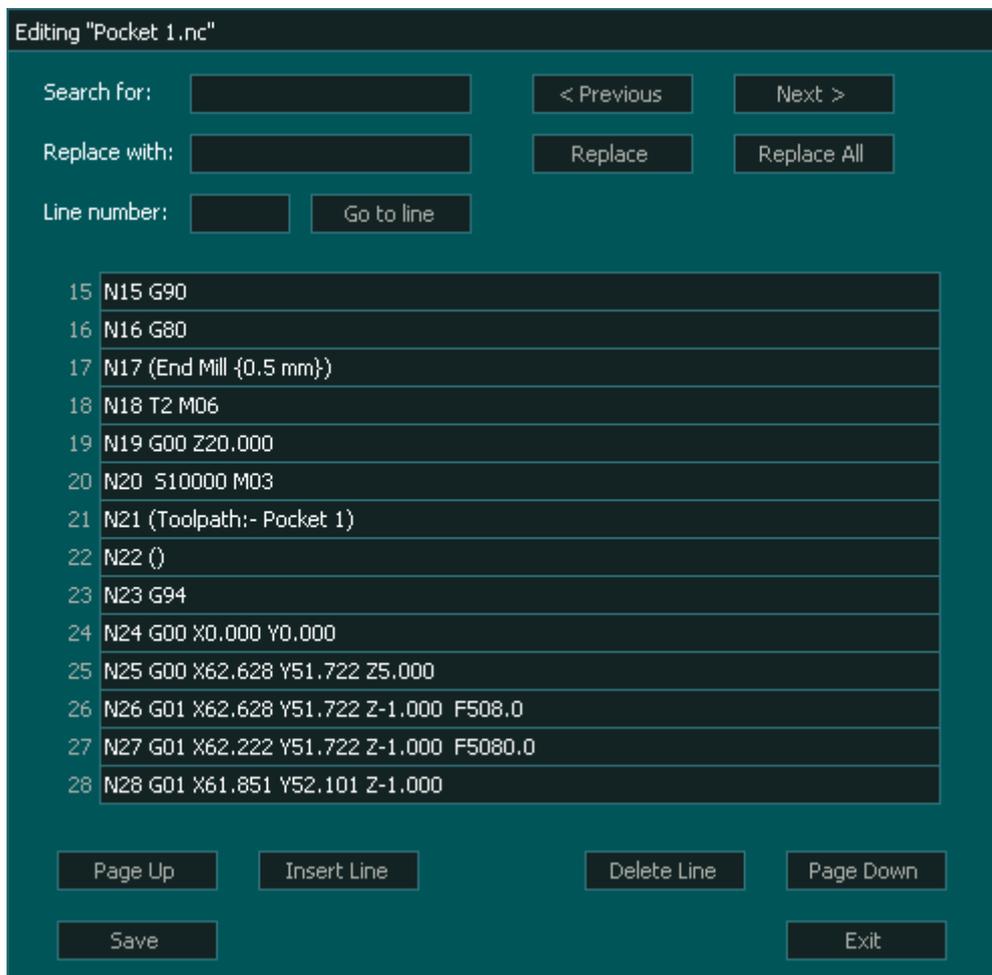
Creating New G-Code Files

- Press F2 to go to the F2 - Program & MDI screen.
- Next press CTRL + N to open a new file name window.



Editing G-Code

- Press F2 to go to the F2 - Program & MDI screen.
- Next press CTRL + E to open the edit file window.



Screen button functions

Search for: This allows you to search for specific text in the G-code file.

It is also used for the Replace with: function

Search for: Enter your text into the search box

Previous: Looks for the previous appearance of the text specified in the Search box

Next: Looks for the next appearance of the text specified in the Search box

Replace with:

This will replace the searched for text with the new text entered into the Replace with: box

Replace with: Enter the text you want to replace into the Replace with box

Replace: Replaces the text located by the Search for: function with the text in the Replace with: box

Replace All: Replaces all instances of the text in the Search for: box with the text in the Replace with: box

Line number

This will look for a specific line of G-code.

Line Number: Enter the line number of the G-Code file you wish to go to

Go to Line: Press Go to Line to move to the line in the Line number box

Note: It is not looking for the N number used in G-code as a line number you need to use the Search for: function instead.

Other Buttons

Page Up: This moves up one page of G-Code.

Page Down: This moves down one page of G-Code.

Insert Line: This inserts a new line above the current selected line. If your cursor is in the middle of the line of G-Code it will not split the line, just put an empty line above.

Delete Line: This deletes the current line with the cursor in it.

Save: This saves the changes you have made to the G-Code file.

Exit: This exits editing and reloads the G-Code file.

Note: If you press Exit without saving the file first all changes will be lost and the original file will be loaded.

Job Counter

MASSO has a built-in job counter that increments every time a G-Code file runs. This counter can be used to see number of parts made in large production work.

To reset the job counter back to 0:

- Goto the F2 - Program & MDI screen.
- Press CTRL + M or click the MDI button to open the MDI Window.
- Type RESET_JOB_COUNTER and press Enter.

Note: The job counter can also be seen and reset remotely on a PC using the MASSO Link software.

Note: The job counter value is retained even after system power down. If the backup battery on the MASSO is removed, the job counter is reset to 0.

Note: For MASSO G3 model the job counter can count up to 4,294,967,295.

Loading & Running G-Code

MASSO runs G-Code files directly from the USB Flash drive. Please see the [video](#) for instructions on how to load and run G-Code files.

Usable G-Code files (**.nc .cnc .tap .wiz .txt .eia**) are displayed in the **F6 - Load File** screen.

After loading your G-Code file you must press the Rewind button before pressing Cycle Start to run your program.

Pro Tip: G-Code files can also be transferred wirelessly using Wi-Fi connection and transferring files from PC using the MASSO Link software.

Note: G-Code files as big as 4GB can be run on controller.

Note: If you are loading a very large G-Code and do not want to wait for the toolpath preview to be displayed you can press the Escape button on screen or the ESC key on the keyboard to cancel the preview.

Resuming Program or Jump to Line

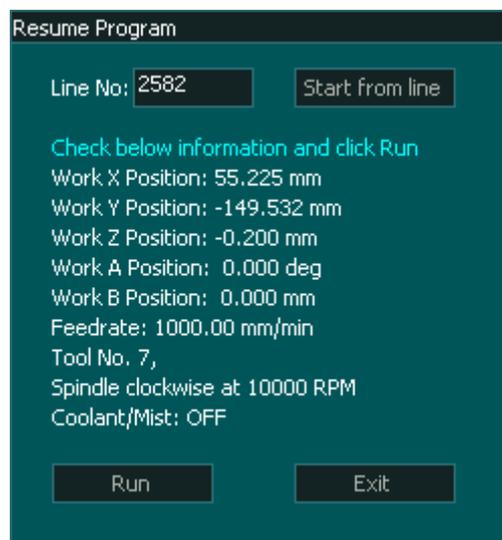
When machining complex jobs such as die molds or complex engraving jobs which can take a long time to machine, power failure or broken tool means that you need to start the entire G-Code file from start.

Restarting the G-Code file from start is very time and resource consuming. To be able to effectively restart your machining process from where you left, MASSO has a Resuming Program or Jump to Line feature.

With this feature the current G-Code line number is saved into a high-speed internal memory and even if the machine is powered off while machining, MASSO will automatically store that last line number in memory.

On power-up, a message is displayed on the G-Code list with the last line number that was running

- After this please load the G-Code file from the F6-Load File screen and go to F2-Program & MDI screen.
- Now press CTRL+J to open the Resuming Program window.
- The last line number will automatically be filled and depending on your program, please go back a few lines from the last running line so that the machining is resumed a few lines before it stopped.



- Next click the **Start from line** button and MASSO will process the G-Code file up to the line number entered by you. When done MASSO will calculate all the machining parameters from the G-Code file as below:
- Calculate the X, Y, and Z-axis positions to resume machining.
- Cutting feed rate as per G-Code.
- Tool Number.
- Spindle RPM and direction.
- Coolant/Mist status.
- Carefully check the information and click the **Run** button.
- Next, MASSO will resume the machining cycle as below:
- Move the Z axis up to the homing position.
- Change tool if a different tool is required.
- Sets the status of any output if used with M62 or M63 codes.
- Start spindle at the requested RPM and direction.
- Start Coolant/Mist as per G-Code.
- Go to X & Y resume position.
- Move the Z axis down to the cutting position
- Start machining at the feed rate as per G-Code file.

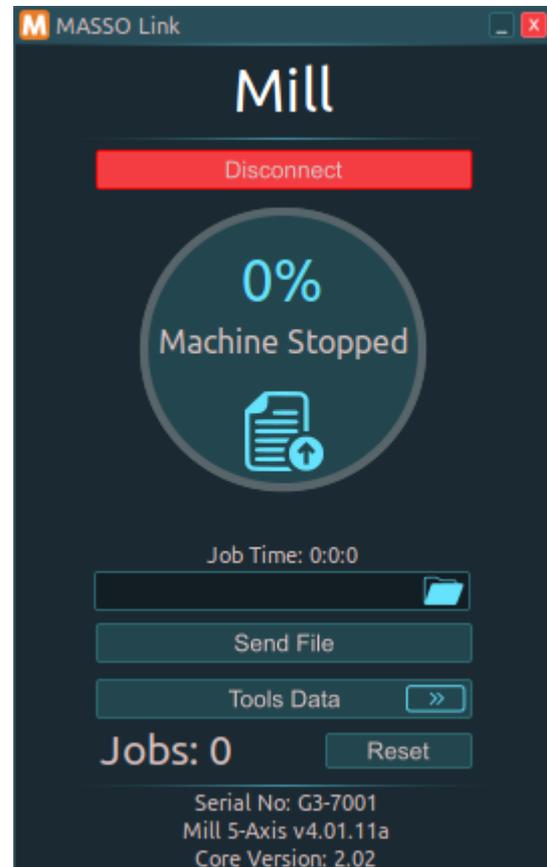
Wi-Fi Connectivity

MASSO comes with Wi-Fi hardware and software to easily connect your MASSO controller to a Wi-Fi network.

With the [MASSO Link Software](#), users can easily view the real-time status of your MASSO controller remotely, transfer G-Code files to MASSO from your PC, get tool data from MASSO's memory and generate tools list document that can be used with CAM software.

Running "MASSO Link" software on PC

Note: Masso link software uses UDP ports 11000 to 11050 for sending data and 65535 for receiving data.



Pro Tip: Additional installation information [Install and using MASSO Link](#)

Creating your own Wi-Fi network

If a Wi-Fi network is not available, MASSO can be used to create a Wi-Fi network and a PC can be connected directly to MASSO via this Wi-Fi network.

This feature is called My MASSO Network.

Tick the Make "My MASSO Network"

- Enter a network name (SSID) that you would like to call this network.
- Enter the security key.
- Click the Enable button.
- At this stage, the Wi-Fi network will show up on your PC in the Wi-Fi networks list and you can connect to this network and use the MASSO Link software.

Note: The security key must be between 8 to 32 characters long.

Transferring G-Code files to MASSO

Note: An USB Flash drive needs to be connected to MASSO to store files.

Once connected, G-Code files with .txt, .nc, .cnc, .tap, .eia extensions can be dragged and dropped on to the MASSO Link software window.

A folder name can be given and all the files sent to MASSO will be saved on the USB pen drive in this folder, else keeping this blank will save the files on the root folder of the USB pen drive.

Click Send File to send the file to MASSO.

Note: File transfer speed is up to 2Mb per minute and is dependent on several factors including Wi-Fi Signal and flash drive write speed. A poor Wi-Fi signal may slow down the file transfer process by requiring data to be resent. A quality, branded USB2 or USB3 flash drive is recommended as modern flash drives will have higher internal write speeds. While the transfer rate may be slower than your flash drive write speed the data is not written to the drive until a block of data is received at which time it will write the data and request the next block. Slow write speeds within the flash drive will become a determining factor in overall transfer speed. A slow flash drive can half the transfer speed of a file doubling the amount of time it needs to send.

Connecting to WiFi network

MASSO Wi-Fi module supports 2.4GHz frequency (IEEE 802.11 b/g/n).

- In the F2 - Program & MDI screen, click the Wi-Fi icon.
- Enter the name (SSID) and security key of the Wi-Fi network that you would like to connect to. If you would like to use dynamic IP as provided by your Wi-Fi network then leave the Fixed IP Address un-ticked and click the Connect button. Once connected the IP address provided by your Wi-Fi network is displayed.

Pro Tip: It's a good idea to use the Fixed IP Address option as every time you connect to your Wi-Fi network, the IP address might be changed by the router. As the IP address is required by MASSO Link software, the new IP address needs to be updated on the PC to be able to connect to the MASSO. Using the Fixed IP Address solves this issue.

Note: The maximum length of SSID is 30 characters and 60 characters for the security key.

Wi-Fi Settings

Make "My MASSO Network"

Network Name (SSID)

Security Key Show

Fixed IP Address

Status: Disconnected

MAC Address: CC:50:E3:27:69:F1

MASSO link on an existing Wi-Fi network

The [video](#) shows how to set up MASSO link on a PC however the method of setup and use is the same across all platforms.

Use this method if you have an existing Wi-Fi network

Make my MASSO network

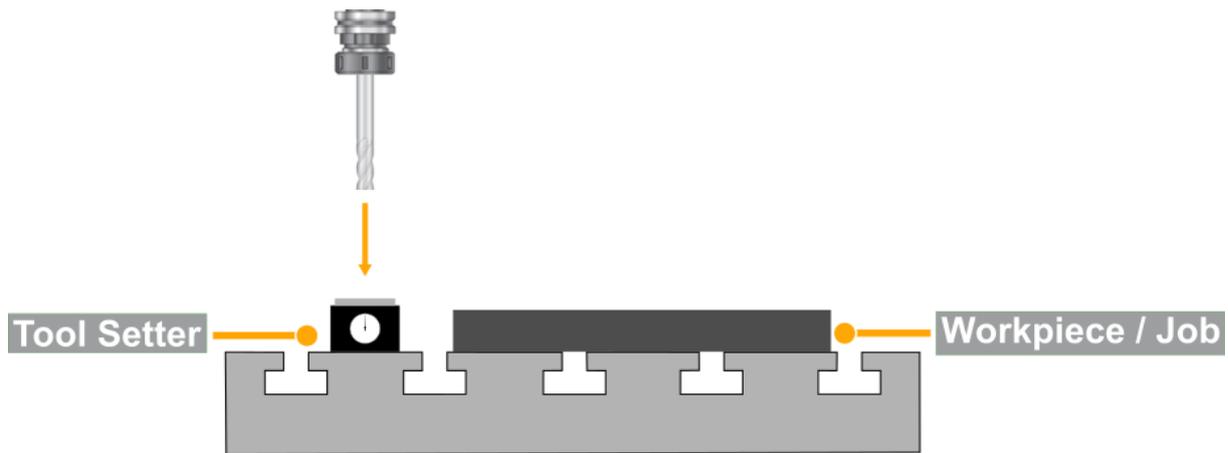
Use [this](#) method if you do not have an existing Wi-Fi network or you wish to use a direct point to point connection from MASSO to your PC.

Mill Tool Calibration Steps

For routers and milling machines, tool length can be calibrated for manually or automatically by using a tool setter or a simple touch plate.

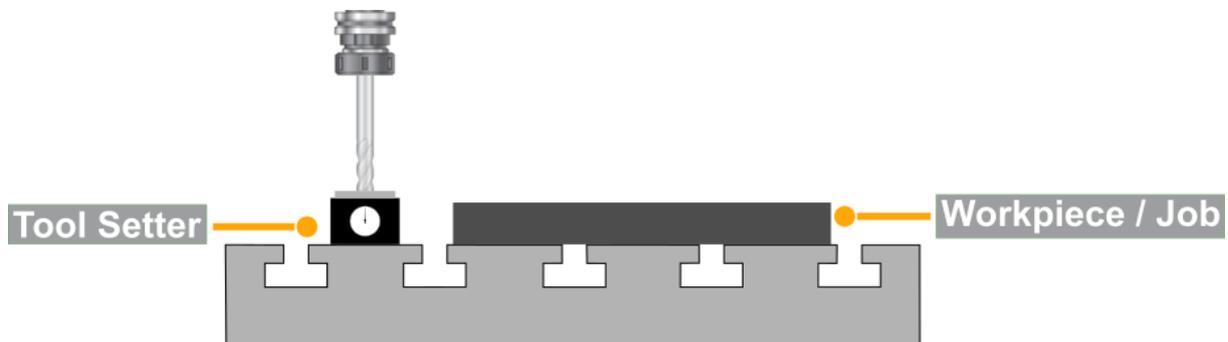
Pro Tip: More information about Touch Probe and Plates setup can be found on this [link](#)

Step 1: Place the tool setter at a predefined position on the machine.



Step 2: On MASSO go to F3-JOG screen, next move the tool in position on top of the tool setter and touch the tool till the tool setter shows exactly 0.00.

Warning: Once in position DO NOT MOVE the tool before completing the next step.



Step 3: On MASSO go to the **F4-Tools** screen and select the tool number you would like to assign to this tool. Now press the enter key to open the Edit Tool window.

MASSO G3 Mill 5-Axis v4.02.20a Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 175 a USB 2:07 AM

F1 SETUP F2 PROGRAM & MDI F3 JOG & PROBING **F4 TOOLS & OFFSETS** F5 CONVERSATIONAL F6 LOAD FILE

Current tool in use: 7,

Tool No	Slot No	Tool Name	Z Offset	Tool Diameter
0		10mm cutter	0.00000	0.00000
1		12mm cutter	0.00000	0.00000
2		15mm cutter	0.00000	0.00000
3		30mm cutter	-47.47000	0.00000
4		V tool 30-Deg	-47.43500	0.00000
5		8mm Drill	-47.45250	254.00050
6		6mm Drill	0.00000	508.00100
7		10mm Tool	5.00000	10.00000
8			-47.46250	0.00000

Work Offset	Work Offset Name	X	Y	Z	A	B
G 54		0.00000	202.42500	-5.44250	0.00000	0.00000
G 55		0.00000	0.00000	0.00000	0.00000	0.00000
G 56		0.00000	0.00000	0.00000	0.00000	0.00000
G 57		0.00000	0.00000	0.00000	0.00000	0.00000
G 58		0.00000	0.00000	0.00000	0.00000	0.00000
G 59		0.00000	0.00000	0.00000	0.00000	0.00000
Parking	CLEAR	100.00000	250.00000	-1.00000	2.00000	0.00000

Step 4: In the **Edit Tool** window give a tool name as per your requirement. Next move the click the **Zero** button to automatically calibrate the tool, MASSO will calculate the tool height and automatically fill the Z Offset value. Now the tool diameter can be entered if required and press **Save** button to save and complete the calibration process.

MASSO G3 Mill 5-Axis v4.02.20a Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 175 a USB 2:08 AM

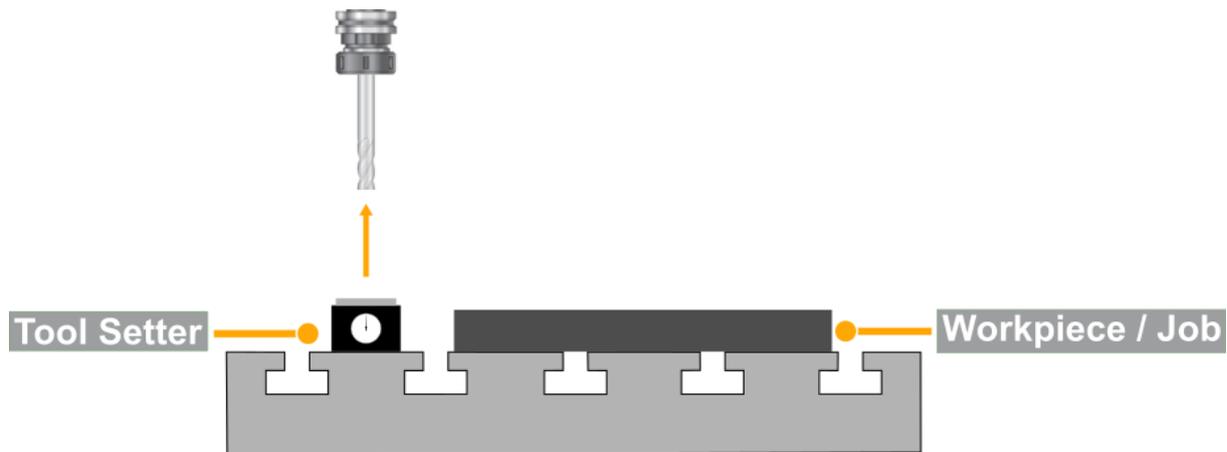
F1 SETUP F2 PROGRAM & MDI F3 JOG & PROBING **F4 TOOLS & OFFSETS** F5 CONVERSATIONAL F6 LOAD FILE

Current tool in use: 7,

Tool No	Slot No	Tool Name	Z Offset	Tool Diameter
0		10mm Tool	0.00000	0.00000
1			0.00000	0.00000
2			0.00000	0.00000
3			-47.47000	0.00000
4			-47.43500	0.00000
5			-47.45250	254.00050
6			0.00000	508.00100
7		10mm Tool	5.00000	10.00000
8			-47.46250	0.00000

Work Offset	Work Offset Name	X	Y	Z	A	B
G 54		0.00000	202.42500	-5.44250	0.00000	0.00000
G 55		0.00000	0.00000	0.00000	0.00000	0.00000

Step 5: Once the tool has been calibrated, please go to **F3-JOG** screen and move tool away from the tool setter.



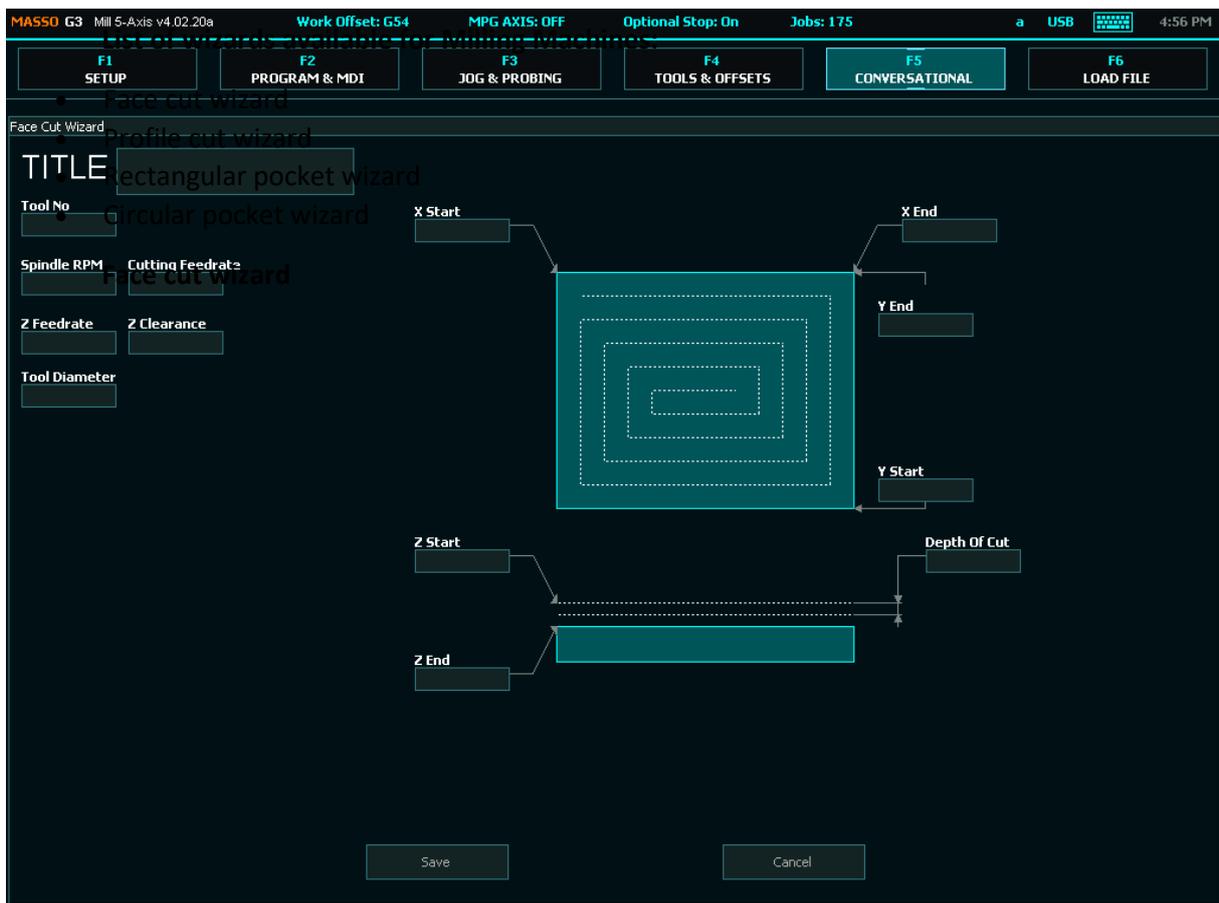
Work Offsets

Work offsets allows the user to position the work piece and cutting tool to allow cutting at the required position. There are different types of offsets such as work and temporary offsets, it's very important to understand these concepts as these will help you generate G-Code from a CAD/CAM software and then how to position the work piece on the machine.

Here is a great [video](#) from CNCnutz explaining the entire process

Mill Conversational Wizards

MASSO has built-in conversational wizards to easily generate G-Code programs for basic machining operations by entering basic information.



Profile cut wizard

MASSO G3 Mill 5-Axis v4.02.20a Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 175 a USB 4:57 PM

F1 SETUP F2 PROGRAM & MDI F3 JOG & PROBING F4 TOOLS & OFFSETS **F5 CONVERSATIONAL** F6 LOAD FILE

Profile Cut Wizard

TITLE

Tool No

Spindle RPM Cutting Feedrate

Z Feedrate Z Clearance

Tool Diameter

X Start Profile X Start Profile X End X End

Y End Profile Y End Profile Y Start Y Start

Z Start Depth Of Cut

Z End

Rectangular pocket wizard

MASSO G3 Mill 5-Axis v4.02.20a Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 175 a USB 4:57 PM

F1 SETUP F2 PROGRAM & MDI F3 JOG & PROBING F4 TOOLS & OFFSETS F5 CONVERSATIONAL F6 LOAD FILE

Rectangle Pocket Wizard

TITLE

Tool No

Spindle RPM Cutting Feedrate

Z Feedrate Z Clearance

Tool Diameter

X Start X End

Y End Y Start

Z Start Depth Of Cut

Z End

Save Cancel

The diagram shows a 3D perspective view of a rectangular pocket. The top surface is a light blue rectangle. The pocket is a rectangular hole cut into the top surface. The depth of the pocket is indicated by a vertical double-headed arrow labeled 'Depth Of Cut'. The top surface of the pocket is a darker blue rectangle. The bottom surface of the pocket is a hatched rectangle. The top surface of the pocket is labeled with 'X Start' and 'X End' at the left and right edges, and 'Y Start' and 'Y End' at the front and back edges. The bottom surface of the pocket is labeled with 'Z Start' at the top edge and 'Z End' at the bottom edge.

Circular pocket wizard

MASSO G3 Mill 5-Axis v4.02.20a Work Offset: G54 MPG AXIS: OFF Optional Stop: On Jobs: 175 a USB 4:58 PM

F1 SETUP F2 PROGRAM & MDI F3 JOG & PROBING F4 TOOLS & OFFSETS F5 CONVERSATIONAL F6 LOAD FILE

Circular Pocket Wizard

TITLE

Tool No

Spindle RPM Cutting Feedrate

Z Feedrate Z Clearance

Tool Diameter

The diagram shows a top-down view of a circular pocket on a rectangular workpiece. The pocket is defined by concentric circles. The center of the pocket is marked with 'X Center' and 'Y Center'. The diameter of the pocket is labeled 'Diameter'. The depth of the pocket is labeled 'Depth of Cut'. The starting and ending Z-axis positions are labeled 'Z Start' and 'Z End' respectively. The workpiece is shown in a cross-section view below the top-down view.

Z Start

Z End

Depth Of Cut

Diameter

X Center

Y Center

Save Cancel

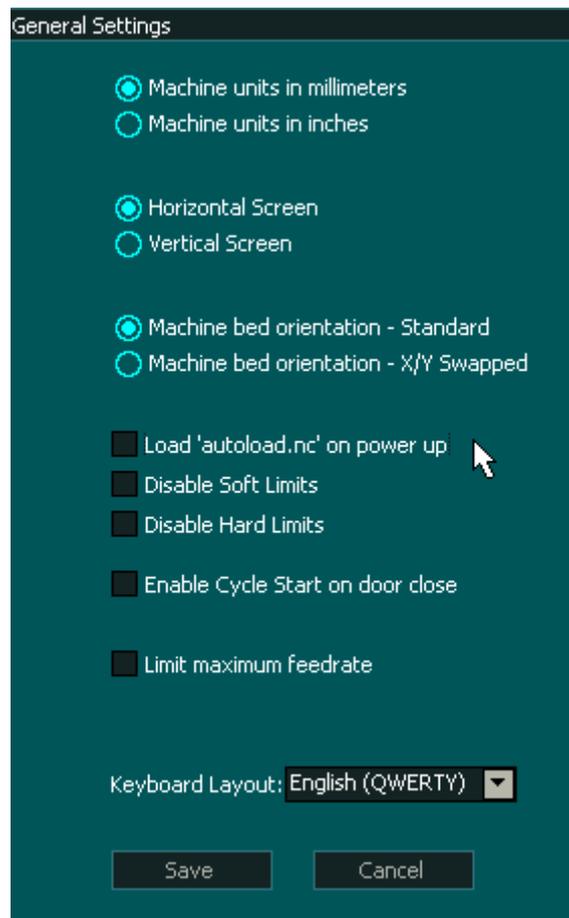
Auto Loading G-code

G-Code files can be automatically loaded by MASSO for special production runs and below are the different ways of automatically loading G-Code files.

Automatically loading G-Code file on power up or when connecting USB stick:

- Go to "F1-Setup" screen and open the "General Settings" window.
- Tick the Load 'autoload.nc' on power up option and click Save button.
- Now MASSO will look for a file autoload.nc on the USB Flash drive every time it powers up or if a USB Flash drive is connected.
- Once the file is found, its automatically loaded and pressing cycle start will start the job.

Note: The autoload files must be in root folder of the USB Flash drive.

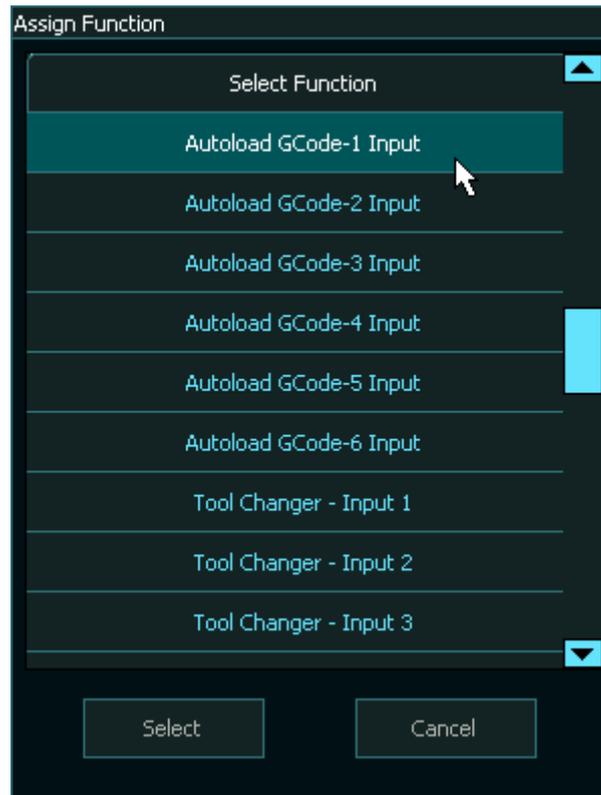


Loading G-Code file on Input signal going high:

- Go to "F1-Setup" screen and go to the "INPUTS" list.
- From the INPUTS list select one of the free inputs and select Autoload GCode-1 Input option and double click to assign the function to the input.
- When this input goes HIGH, MASSO will look for a file autoload1.nc on the USB pen drive, once the file is found, its automatically loaded and pressing cycle start will start the job.

Note: You can assign up to 6 autoload functions on inputs.

Note: The autoload files must be in root folder of the USB stick.

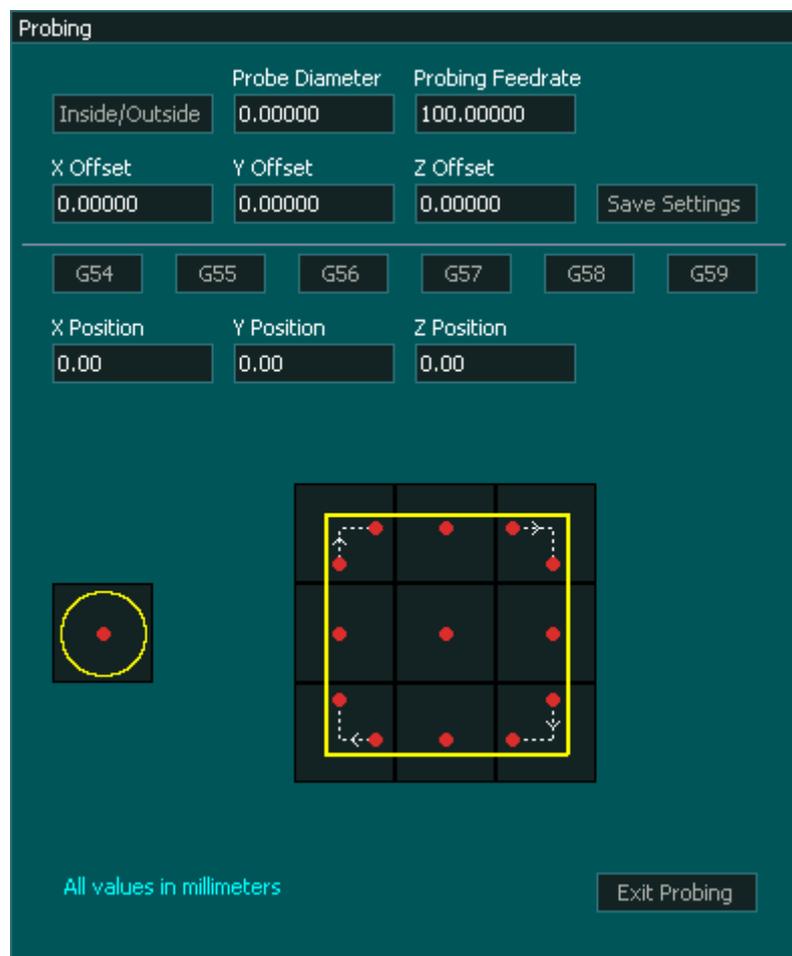


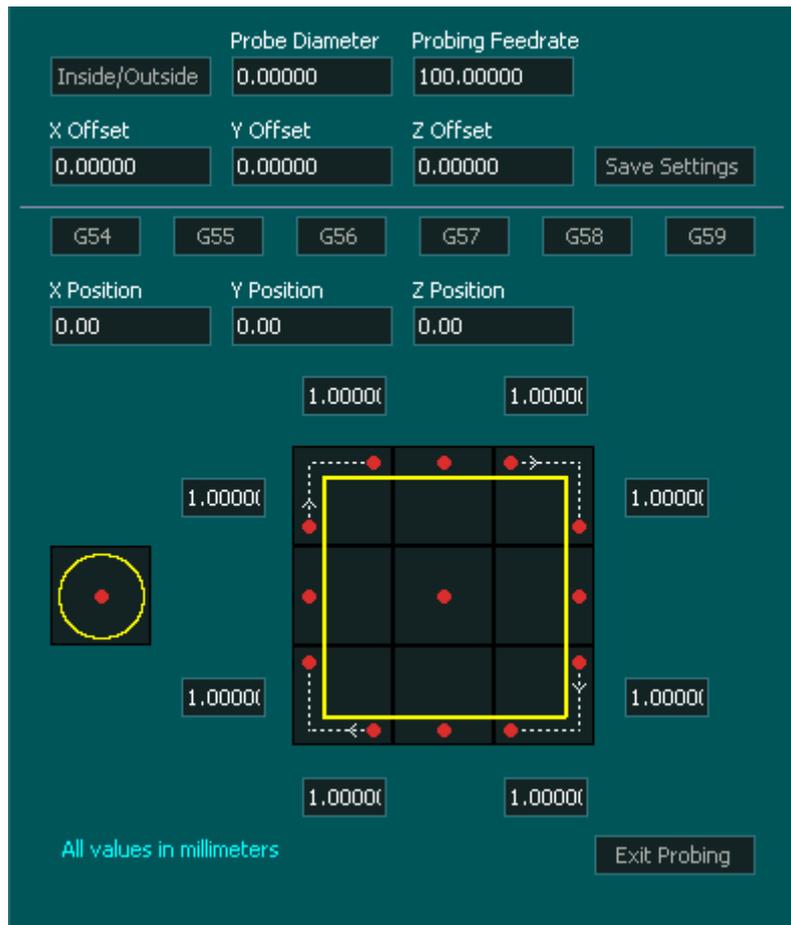
Probing



MASSO supports interactive part probing option that allows the user to probe parts and set work offsets. The following probing features are available:

- Top of part
- Sides of part
- Corners of part
- Auto find center of holes





Selecting a Work offset

There are 6 work offset buttons on the Probing screen. This allows the user to select which work offset the probing result will be save in.

The work offset needs to be selected before doing the probing cycle as the result is stored in the active offset.

The current work offset is displayed at the top of the screen

When you exit the probing screen M1A will remain in the work offset you last selected.

Top of Part Probe cycle

This Probing cycle is available for both inside and outside probing cycles and can be started by pressing the center square in the large probing block.

This will start the Z axis probing down at a the rate specified in the Probing **Feed rate** box.

- When the probe finds the surface the Z axis DRO coordinate will be updated with the value specified in the Z Position box.
- The Z probe offset value is most commonly used when using a probing touch off to compensate for the touch off thickness.
- The Z axis will then return to the starting position where probing began and display it's new coordinate based on its relative position to the probed surface.
- The probing cycle can be stopped by pressing the same or a different probing cycle button.

Side of Part Probe Cycles

There are 8 Side probing cycles. 4 are inside probing cycles and 4 are Outside probing cycles.

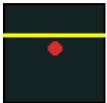
The one you choose will be determined by whether you are probing the inside or the outside of a part and also which axis you will be probing X or Y.

- First select Inside or outside by pressing the Inside/Outside button on the probing cycles screen.
- Next press the Save settings button to complete the change.
- X & Y offsets can be used to compensate for errors eg. your probe needs to move 5mm to register a touch off.

Note: The probing cycle can be stopped by pressing the same or a different probing cycle button.

Inside, Side of part probing

- The Red dot indicates the position of the probe at the start of the probing cycle and the probe will move towards the Yellow Line which represents the object being probed.
- The probing will occur at the speed entered in the Probing Feed rate box.
- After touch off the Axis will return to the start position at rapid speed.
- Enter the Probe diameter into the Probe Diameter box.



- Probe will start moving in the +Y direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the Y Position box.
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.



- Probe will start moving in the -Y direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the Y Position box.
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.



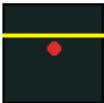
- Probe will start moving in the -X direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the X Position box.
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.



- Probe will start moving in the +X direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the X Position box.
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.

Outside, Side of part Probing

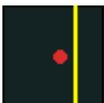
The Red dot indicates the position of the probe at the start of the probing cycle and the probe will move towards the yellow line which represents the object being probed. The probing will occur at the speed entered in the Probing Feed rate box. After touch off the Axis will return to the start position at rapid speed. Enter the Probe diameter into the Probe Diameter box



- Probe will start moving in the -Y direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the Y Position box
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.



- Probe will start moving in the +Y direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the Y Position box
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.



- Probe will start moving in the +X direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the X Position box
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.



- Probe will start moving in the -X direction at the specified Probing feed rate until it touches the part.
- The DRO will update with the value in the X Position box
- The Probe will return to its start position and display its new coordinate based on its relative position to the probed surface.

Corner Probing Cycles

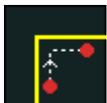
There are 8 Corner probing cycles. 4 are inside probing cycles and 4 are Outside probing cycles. The one you choose will be determined by whether you are probing the inside or the outside of a part.

- First select Inside or outside by pressing the Inside/Outside button on the probing cycles screen
- Next press the Save settings button to complete the change
- X & Y offsets can be used to compensate for errors e.g., your probe needs to move 5mm to register a touch off.
- Each corner probing button shows a dotted white line indicating the order in which probing will take place.
- The X & Y Position boxes are used to determine the coordinate of the surface at the point of touch.
- The probing cycle can be stopped by pressing the same or a different probing cycle button.

Inside Corner Probing

The Red dot indicates the position of the probe at the start of the probing cycle and the probe will move towards the yellow line which represents the object being probed.

- The dotted white line indicating the order in which probing will take place.
- Each probing cycle is an automatic two-part probing cycle.
- The probing will occur at the speed entered in the Probing Feed rate box.
- After touch off the Axis will return to the start position at rapid speed.
- Enter the Probe diameter into the Probe Diameter box



- Probe will start moving in the -X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- Probe will start moving in the +Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position and display its new coordinates based on its relative position to the probed surfaces.



- Probe will start moving in the +Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- Probe will start moving in the +X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position and display its new coordinates based on its relative position to the probed surfaces.



- Probe will start moving in the +X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- Probe will start moving in the -Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position and display its new coordinates based on its relative position to the probed surfaces.



- Probe will start moving in the -Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- Probe will start moving in the +X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position and display its new coordinates based on its relative position to the probed surfaces.

Outside Corner probing

When Outside corner probing is selected a set of 2 distance boxes are displayed on the screen at each corner.

These boxes represent the how far the X and Y axis will travel around the corner before starting the 2nd probe.

The Red dot indicates the position of the probe at the start of the probing cycle and the yellow line which represents the object being probed.

The White arrow represents the order in which probing will occur in the cycle and also the direction of travel the probe will take.

Each probing cycle is an automatic two part probing cycle.

The probing will occur at the speed entered in the Probing Feed rate box.

After touch off the Axis will return to the start position at rapid speed.

Enter the Probe diameter into the Probe Diameter box.



- Probe will start moving in the +X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- The probe will move at probing speed in the +Y direction for the distance shown.
- The probe will move at probing speed in the +X direction for the distance shown.
- Probe will start moving in the -Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to the start of the 2nd probing and display it's new coordinates based on its relative position to the probed surfaces.



- Probe will start moving in the -Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- The probe will move at probing speed in the +X direction for the distance shown.
- The probe will move at probing speed in the -Y direction for the distance shown.
- Probe will start moving in the -X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to the start of the 2nd probing and display its new coordinates based on its relative position to the probed surfaces.



- Probe will start moving in the -X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- The probe will move at probing speed in the -Y direction for the distance shown.
- The probe will move at probing speed in the -X direction for the distance shown.
- Probe will start moving in the +Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to the start of the 2nd probing and display its new coordinates based on its relative position to the probed surfaces.



- Probe will start moving in the -Y direction at the specified Probing feed rate until it touches the part.
- The Probe will return to its start position.
- The probe will move at probing speed in the -X direction for the distance shown.
- The probe will move at probing speed in the +Y direction for the distance shown.
- Probe will start moving in the +X direction at the specified Probing feed rate until it touches the part.
- The Probe will return to the start of the 2nd probing and display its new coordinates based on its relative position to the probed surfaces.

Center Probing cycles

This probing cycle is available from both inside and outside probing cycle pages.

The Red dot indicates the position of the probe at the start of the probing cycle and the yellow line which represents the side object being probed.

It is used to locate the center of an object such as a hole or a pocket.

The probing will occur at the speed entered in the Probing Feed rate box.

Probing the center of an object is an automatic 4 probe cycle process and after each touching the side the probe will rapid back to the starting position of the next probing before starting the next probe.

Enter the Probe diameter into the Probe Diameter box.

X & Y offsets can be used to compensate for errors e.g., your probe needs to move 5mm to register a touch off.

The probing cycle can be stopped by pressing the same or a different probing cycle button.

Pro Tip: You can only probe the inside of a part to find center. You cannot probe the outside of a part to find center.

Center Probing Cycle



- The probe will move in the -X direction at the specified Probing feed rate until it touches the part.
- The probe will return to its start position at rapid speed.
- The probe will move in the +X direction at the specified Probing feed rate until it touches the part.
- The probe will move to the X position in the Center of these two readings.
- The probe will move in the -Y direction at the specified Probing feed rate until it touches the part.
- The probe will return to its start position at rapid speed.
- The probe will move in the +Y direction at the specified Probing feed rate until it touches the part.
- The probe will move to the Y position in the Center of these two readings.
- The DRO will be updated with the values shown in the X & Y Position boxes

Supported G-Codes

G00 - Rapid Motion

This command is used to move one or more axis at the maximum feed rate to a specified location. If multiple axis is called, they will all move together to the desired location in a straight line and arrive at the same time. The axis can be linear, angular or a combination of both.

Syntax & Parameters

G00 followed by the axis you wish to move and its coordinate. Multiple axis may be specified on the same line.

X, Y, Z, A, B Value – specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.

U, V, W Value – specifies an incremental move of the X,Y or Z axis while the machine is in G90 Absolute mode. U moves the X axis, V moves the Y axis, W moves the Z axis All U, V & W moves are incremental regardless of whether the machine is in Absolute (G90) or Incremental (G91) mode

Example program

```
N10 G21
```

```
N20 G00 X10 Y20 Z30
```

In the above program the first line command G21 will first set the controller units to millimeters, then it will move the axis from its current location to next position of X 10mm, Y 20mm, & Z30 mm in a straight line at the maximum speed the machine axis will allow.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

Note: The motion can be either in G90 Absolute or G91 Incremental mode.

G01 - Linear Interpolation Motion

This command is used to move one or more axis at the specified feed rate to a specified location. If multiple axis is called, they will all move together to the desired location in a straight line and arrive at the same time. The axis can be linear, angular or a combination of both.

Syntax & Parameters

G01 followed by the axis you wish to move and its coordinate. Multiple axis may be specified on the same line.

X, Y, Z, A, B Value – specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.

U, V, W Value – specifies an incremental move of the X, Y or Z axis while the machine is in G90 Absolute mode. U moves the X axis, V moves the Y axis, W moves the Z axis All U, V & W moves are incremental regardless of whether the machine is in Absolute (G90) or Incremental (G91) mode

F Value – The F value defines the feedrate at which the axis will move at. If the F value is not specified then the feed rate used from the last G01 command is used.

Example program

```
N10 G21  
N20 G01 X10 Y20 Z30 F100  
N30 G01 X0 Y0 Z0
```

In above program the first line command G21 will first set the controller units to millimeters, then it will move the axis to next position of X 10mm, Y 20mm, & Z 30mm in a straight line at 100mm/min feed rate. In the last line the axis will move to X 0mm, Y 0mm, & Z 0mm at 100mm/min feed rate.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

Note: The motion can be either in G90 Absolute or G91 Incremental mode.

G02 – Circular Interpolation (Clockwise)

This command runs a circular motion in a clockwise direction. The arc starts at the current position and the center is defined by the I, J, K values as an offset of the current position or R for radius. The X, Y & Z values define the finish point. Masso uses Incremental I, J, K values for Arcs

Syntax & Parameters

- G02 starts an arc movement beginning at the X, Y, Z coordinate location where the G02 was issued.
- X Value – This is the X coordinate defining the end point
- Y Value – This is the Y coordinate defining the end point
- Z Value - This is the Z coordinate defining the end point
- I Value – are relative coordinates and define the X center coordinate as an offset of the starting point X coordinate. (Valid in G17 & G18 planes)
- J Value – are relative coordinates and define the Y center coordinate as an offset of the starting point Y coordinate. (Valid in G17 & G19 planes)
- K Value – are relative coordinates and define the Z center coordinate as an offset of the starting point Y coordinate. (Valid in G18 & G19 planes)
- R Value - Is used to define the arc radius for arcs up to 90 degrees.

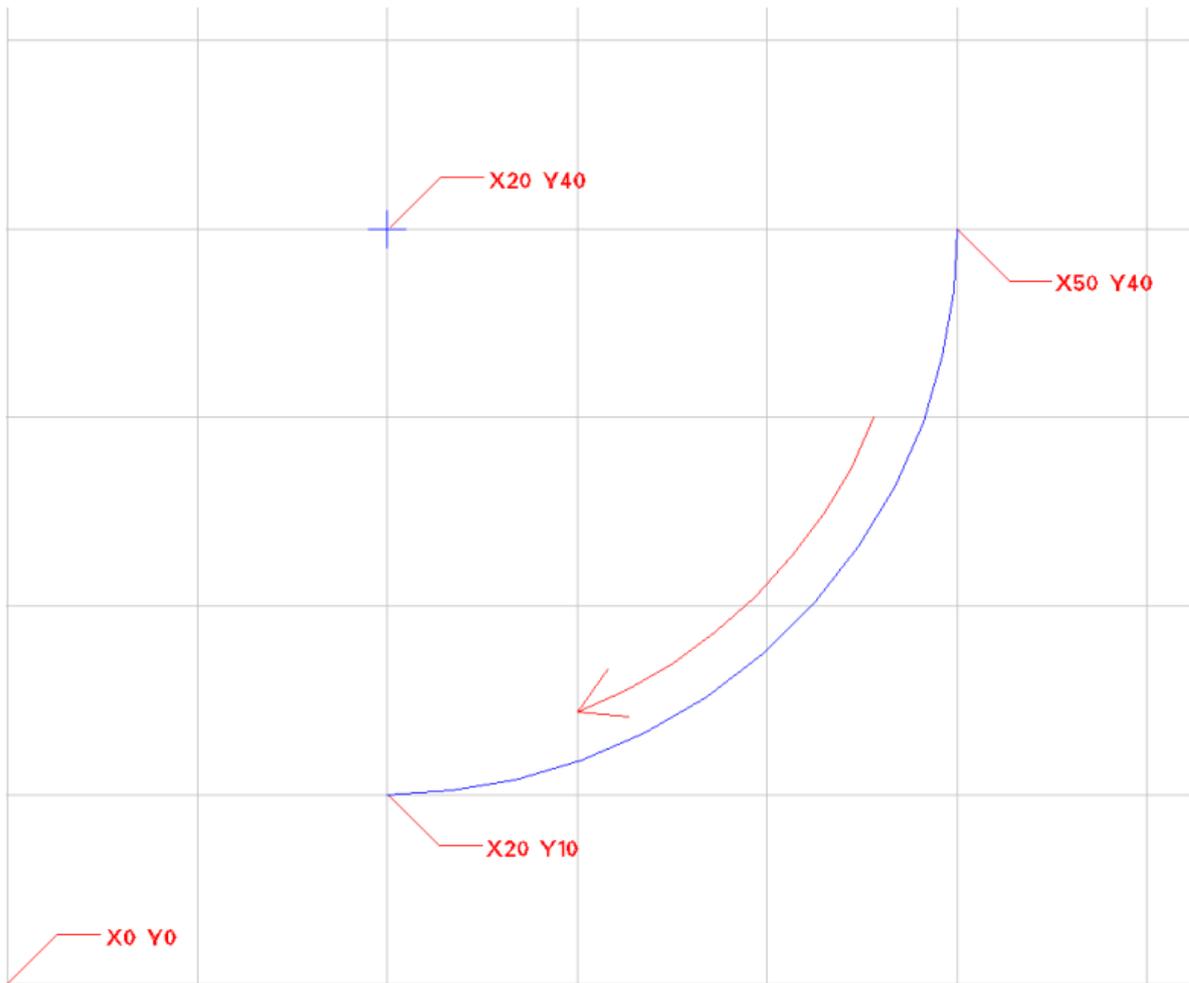
Warning: The use of R Value for arcs greater than 90 degrees do not work as it defines multiple center points. For arcs greater that 90 degrees use IJK values for G02 commands.

Example program for making an Arc

```
N10 G00 X50 Y40 Z-1
```

```
N20 G02 X20 Y10 I-30 J0
```

- In the above example in the first line moves to position X50 Y40 Z-1
- An arc is made starting at position X50 Y40 and ending at coordinate X20 Y10 with center at position X20 Y40 moving in a clockwise direction. The Z height does not change.
- The Arc center is calculated as follows
- Starting position X50 + I-30 = Center X coordinate X20
- Starting position Y40 + J0 = Center Y coordinate Y40



Example program for making a Spiral

```
N10 G00 X50 Y40 Z0
```

```
N20 G02 X20 Y10 Z-3 I-30 J0
```

- By adding a Z height into G02 you can create a spiral arc.
- In the above example in the first line moves to position X20 Y10 Z0
- An arc is made starting at position X50 Y40 and ending at coordinate X50 Y40 with center at position X20 Y40 moving in a counter clockwise direction. The arc is the same as in the diagram above but the Z axis will descend from its current height to -3 units at a uniform rate over the arc to form a spiral path.
- The Arc center is calculated as follows
- Starting position $X50 + I-30 =$ Center X coordinate X20
- Starting position $Y40 + J0 =$ Center Y coordinate Y40

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G03 – Circular Interpolation (Counter Clockwise)

This command runs a circular motion in a counter clockwise direction. The arc starts at the current position and the center is defined by the I, J, K values as an offset of the current position or R for radius. The X, Y & Z values define the finish point. Masso uses Incremental I, J, K values for Arcs

Syntax & Parameters

- G03 starts an arc movement beginning at the X, Y, Z coordinate location where the G03 was issued.
- X Value – This is the X coordinate defining the end point
- Y Value – This is the Y coordinate defining the end point
- Z Value - This is the Z coordinate defining the end point
- I Value – are relative coordinates and define the X center coordinate as an offset of the starting point X coordinate. (Valid in G17 & G18 planes)
- J Value – are relative coordinates and define the Y center coordinate as an offset of the starting point Y coordinate. (Valid in G17 & G19 planes)
- K Value – are relative coordinates and define the Z center coordinate as an offset of the starting point Y coordinate. (Valid in G18 & G19 planes)
- R Value - Is used to define the arc radius for arcs up to 90 degrees.

Warning: The use of R Value for arcs greater than 90 degrees do not work as it defines multiple center points. For arcs greater that 90 degrees use IJK values for G03 commands.

Example program for making an Arc

```
N10 G00 X20 Y10 Z-1
```

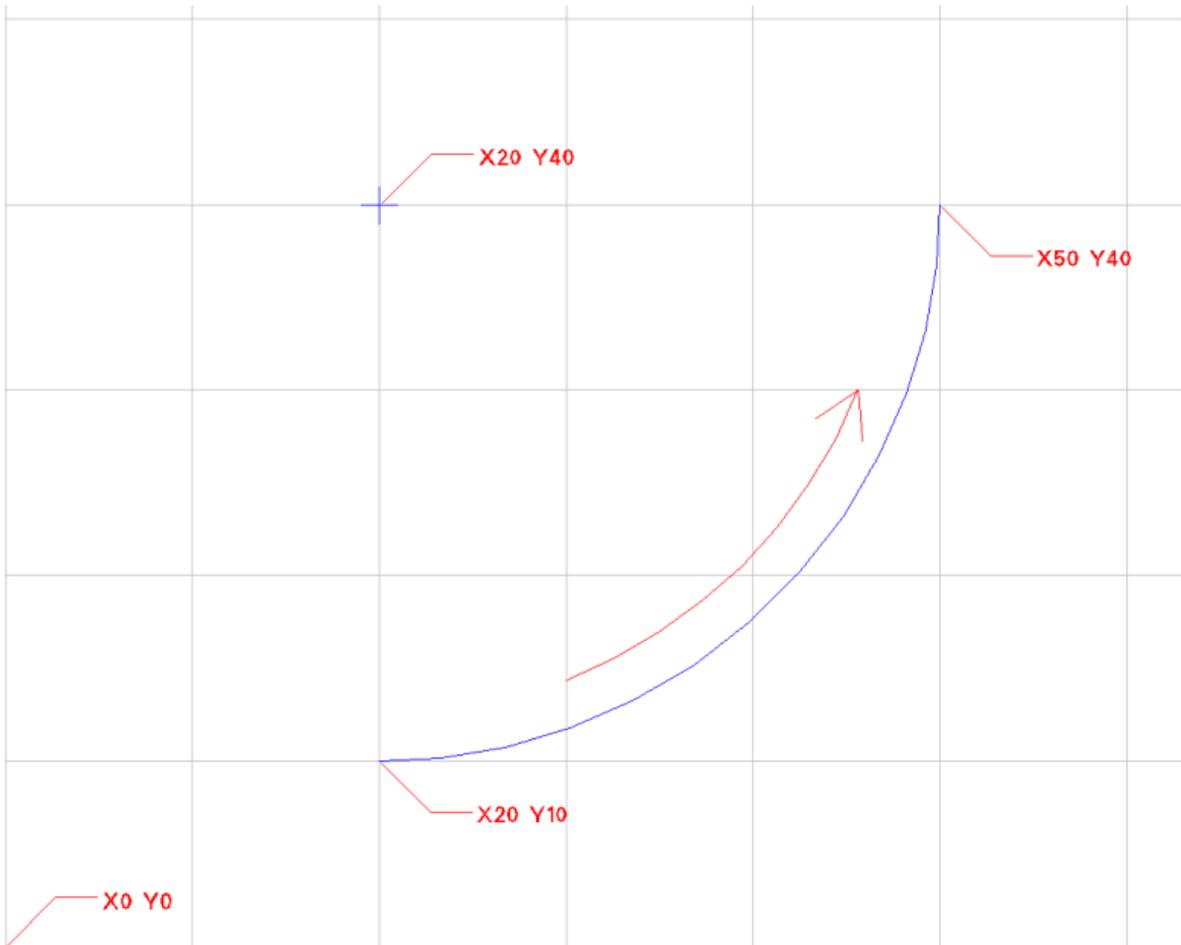
```
N20 G03 X50 Y40 I0 J30
```

In the above example in the first line moves to position X20 Y10 Z-1

- An arc is made starting at position X20 Y10 and ending at coordinate X50 Y40 with center at position X20 Y40 moving in a counter clockwise direction.
- The Z height does not change.

The Arc center is calculated as follows

- Starting position X20 + I0 = Center X coordinate X20
- Starting position Y10 + J30 = Center Y coordinate Y40



Example program for making a Spiral

```
N10 G00 X20 Y10 Z0
```

```
N20 G03 X50 Y40 Z-3 I0 J30
```

By adding a Z height into G03 you can create a spiral arc.

In the above example in the first line moves to position X20 Y10 Z0

- An arc is made starting at position X20 Y10 and ending at coordinate X50 Y40 with center at position X20 Y40 moving in a counter clockwise direction.
- The arc is the same as in the diagram above but the Z axis will descend from its current height to -3 units at a uniform rate over the arc to form a spiral path.

The Arc center is calculated as follows

- Starting position X20 + I0 = Center X coordinate X20
- Starting position Y10 + J30 = Center Y coordinate Y40

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G04 – Dwell

This command is used to Dwell (pause) the execution of the next CNC program block by the specified time.

A program dwell time can be created at any point within a program.

Syntax & Parameters

P Value - The P value is required with G04 and it defines the time to dwell/pause. This value is in milliseconds.

Example program

```
N10 G04 P2000
```

The program above program will pause the program for 2 seconds.

G10 – Set Work Offset Values

This command is used to set work offsets values to different offset locations.

Syntax & Parameters

L2 Value - Must be issued with G10.

P Value - The P value is required with G10 and it defines the work offset number to change between G54 and G59.

- P0 is used to set the current active work offset.
- P1 is used to set G54 work offset
- P2 is used to set G55 work offset.
- P3 is used to set G56 work offset.
- P4 is used to set G57 work offset.
- P5 is used to set G58 work offset.
- P6 is used to set G59 work offset.

X, Y, Z, A, B Value - specifies the axis you wish to offset following the distance value. The distance value will be the current machine units in use.

Example program

```
N10 G10 L2 P1 X10 Y10
```

The program line above will set the X and Y axis G54 offset to 10 millimeters.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

Note: This function does not overwrite G54 - G59 offset values saved to backup memory through the F4 screen, although the F4 screen will display the current offset values.

G17 – XY Plane Selection

This command is used to set current work plane to XY. This is the default work plane on power up.

Syntax & Parameters

- G17

Example program

```
N10 G17
```

The above G-Code will set the system to XY work plane.

Note: This is the default work plane on power up.

G18 – ZX Plane Selection

This command is used to set current work plane to ZX.

Syntax & Parameters

- G18

Example program

```
N10 G18
```

The above G-Code will set the system to ZX work plane.

G19 – YZ Plane Selection

This command is used to set current work plane to YZ.

Syntax & Parameters

- G19

Example program

```
N10 G18
```

The G-Code above will set the system to YZ work plane.

G20 – Set Machine Units to Inches

This command is used to set machine units to Inches. All G-Code values after this command will be processed as inches.

Syntax & Parameters

- G20

Example program

```
N10 G20
```

The G-Code above will set the machine units to Inches.

G21 – Set Machine Units to Millimeters

This command is used to set machine units to Millimeters. All G-Code values after this command will be processed as Millimeters.

Syntax & Parameters

- G21

Example program

```
N10 G21
```

The G-Code above will set the machine units to Millimeters.

G28 – Return to Machine Home

WARNING: This command can be used in different combinations and wrong command can result in unexpected rapid motion. Depending if the machine is in Absolute or Incremental mode the behavior of G28 command will be very different, extra caution should be used when using this command.

This command is used to move the axis back to the home position of the axis after the machine was homed. Further axis commands can also be combined with G28 to achieve intermediate position.

Syntax & Parameters

- G28 - Only G28 can be used, this will move all axis at rapid back to the home position.
- **X, Y, Z, A, B Value** - specifies the intermediate position you wish to move following the distance to move. The distance value will be the current machine units in use.
- Combining the G28 with a rotary axis in incremental mode G91 will allow the axis to unwind in 1 revolution or less. See example below.

Example program for moving all together axis to home position

```
N10 G28
```

The G-Code above will move all axis of the machine at rapid back to the home position.

Example program all axis to machine 0.00

```
N10 G28 X0 Y0 Z0
```

The G-Code above will move all axis to working coordinates X0 Y0 Z0 before moving to home position.

Example program all axis to machine 0.00

```
N10 G91 G28 X0 Y0 Z0
```

```
N20 G90
```

The G-Code above will move all axis to home position as there is no intermediate position to go to first.

Example program to move Z axis first

```
N10 G91 G28 X0 Y0 Z8
```

```
N20 G90
```

The G-Code above will first move the Z axis 8.00 units and then move all axis to their home positions

Rotary Axis Unwind within one rotation

```
G00 A900 (Rapid the A axis to A900 (2.5 turns))
```

```
G91 (Change to Incremental mode)
```

```
G28 A0 (Moves Axis by 0 degrees then moves to A0 within one rotation)
```

```
G90 (Return to Absolute mode)
```

In the example above, if you specify another coordinate e.g., G28 A360 it will move an additional 360 degrees taking the axis to A1260 (3.5 turns) then move to A0 in less than 1 rotation.

In this case it would be an additional 1/2 rotation to be back to home.

Note: If you do not change to incremental mode before the G28 A0 the axis will unwind the full 2.5 revolutions to A0 and then move to the home position. In this case no further move is needed.

G30 - Move to Parking Position

Warning: This command can be used in different combinations and wrong command can result in unexpected rapid motion. Depending if the machine is in Absolute or Incremental mode the behavior of G30 command will be very different, extra caution should be used when using this command.

This command is used to move the Machine to the defined Parking position

- At times users may wish to have a dedicated park location to move the Spindle to, so that it is out of the way when loading and unloading material.

Syntax & Parameters

- **G30** - This first rapids the Z axis to its specified parking position and then the remaining axis will simultaneously move to their parking positions.
- The Park position is defined in the F4 Screen

Example program for moving all axis to Parking position

N10 G30

The G-Code above will move all axis to defined position in the F4 Screen.

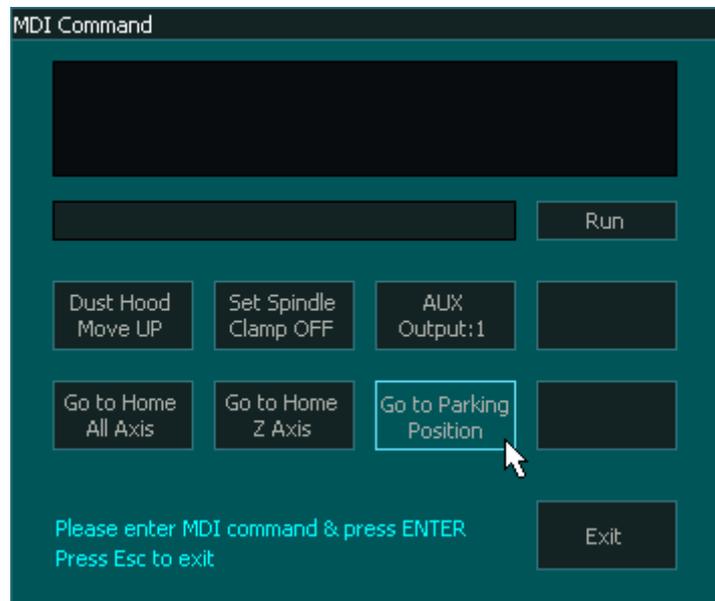
Defining the Parking Position

The Parking Position is defined in the F4 Screen using Machine Coordinates.

Work Offset	Work Offset Name	X	Y	Z	A	B
G 54		0.00000	202.42500	-5.44250	0.00000	0.00000
G 55		0.00000	0.00000	0.00000	0.00000	0.00000
G 56		0.00000	0.00000	0.00000	0.00000	0.00000
G 57		0.00000	0.00000	0.00000	0.00000	0.00000
G 58		0.00000	0.00000	0.00000	0.00000	0.00000
G 59		0.00000	0.00000	0.00000	0.00000	0.00000
Parking	Parking Position	100.00000	250.00000	-1.00000	2.00000	0.00000

Parking Commands

- G30
- CTRL + ALT + P
- Click the Go to Parking Position button in the MDI Screen
- Go to Parking Position Button



G32 – Threading Cycle

This command is used on Lathe machines for running threading cycle synchronized with the spindle.

Note: Taper threading is not supported.

Syntax & Parameters

Z Value (required) - specifies the end location of the Z axis for threading. The distance value will be the current machine units in use.

F Value (required) - this is the thread pitch value. The distance value will be the current machine units in use.

Example program

```
N10 G21  
N20 G00 Z0  
N30 G32 Z-10 F1.5
```

In the above program the first line sets the machine units to millimeters.

The second line command moved the Z axis to work 0.00 millimeters.

The third line starts the threading cycle to thread to Z -10 millimeters location while threading at 1.5-millimeter pitch.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G38.2 – Straight Probe Cycle

This command is used for probing parts or fixtures. The axis specified in the G38.2 command will move until the probe touches, if the probe does not touch within the specified distance, then the program stops and an alarm is displayed on the screen.

Syntax & Parameters

X, Y, Z, A, B Value - specifies the axis you wish to move for probing and the maximum machine coordinate to move towards when probing. The distance value will be the current machine units in use.

F Value - The F value defines the feed rate at which the axis will move at.

Example program

```
N10 G38.2 Z-10 F100
```

In the program above the Z axis will move towards machine coordinate Z -10 and wait for the probe to touch. Once the probe touches, the Z axis will stop and the program will move to the next G-code line. If the probe is not touched before it reaches Z -10 location then the program stops and an alarm is displayed on the screen.

Pro Tip: The coordinate value specified is a machine coordinate.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G38.6 - Digitizing Probing Cycle

WARNING: The incorrect use of this G-Code can cause damage to your probing equipment; the item being probed and personal injury. Please understand and exercise with care.

This command is used for digitizing parts using a touch probe or similar. Typically used to create a cloud point capture of a 3d object or to map uneven material surface for engraving with the use of auto leveling software.

- The axis specified in the G38.6 command will move until the probe touches.
- The X, Y & Z touchpoint data is saved to "MASSO Probe data.txt"
- If the "MASSO Probe data.txt" does not exist then the MASSO will create a new file.
- If the "MASSO Probe data.txt" exists then the data will be added to the end of the existing file.
- If the probe does not touch within the specified distance, then a "Probing error at:" entry is made in the "MASSO Probe data.txt" file, the program will Stop and a Probing Error Alarm will be displayed.
- All coordinates used with G38.6 are working coordinates.
- All recorded coordinates in the "MASSO Probe data.txt" file are working coordinates.

Syntax & Parameters

X, Y, Z Value - specifies the axis you wish to move for probing following the coordinate to move to. The coordinate value is a working coordinate.

F Value - The F value defines the feed rate at which the axis will move.

Example program

```
N10 G1 X20 Y10 F1000
```

```
N20 G38.6 Z-3 F100
```

```
N30 G0 Z2
```

In the example above, the machine moves to X20 Y10

The Z axis will probe down to working coordinate -3 and if the probe detects a touch it will stop and record the X, Y & Z working coordinates in the "MASSO Probe data.txt" file.

If the probe does not record by the time it reaches Z -3, it will stop and record Probing error at: 20.00000, 10.00000 in the "MASSO Probe data.txt" file. A Probing error Alarm will display in the screen.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

Sample Data File

0.00000, 0.00000, -0.88750

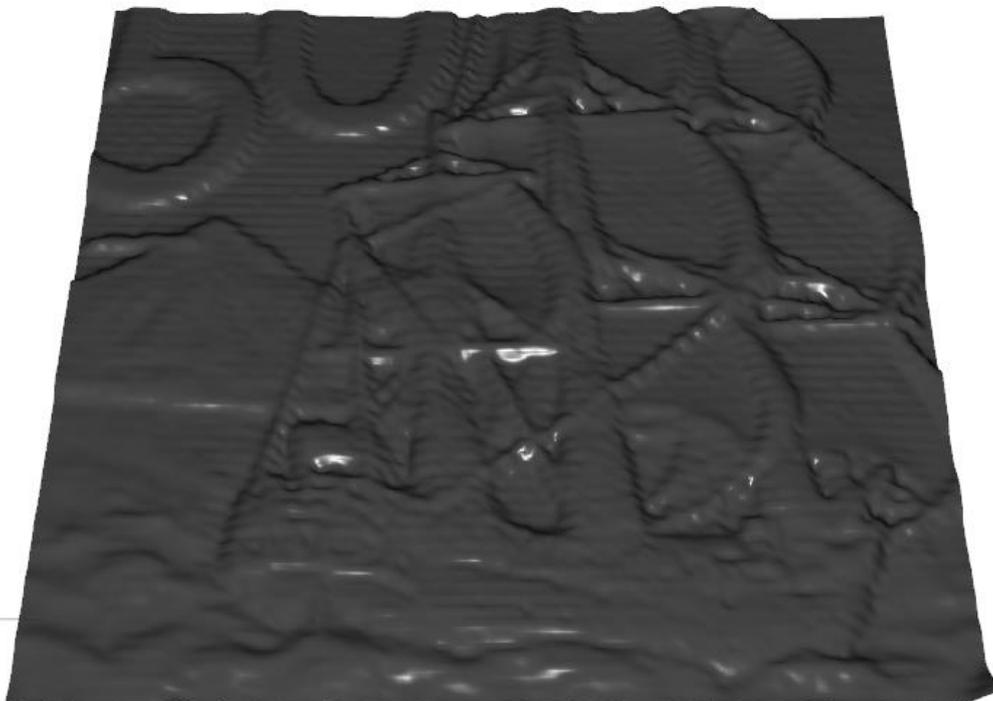
0.13310, 0.00000, -0.90749

0.26619, 0.00000, -0.91750

0.38597, 0.00000, -0.92249

Probing error at: 0.00000, -0.92000

Sample Digitized item



Auto Levelling using G38.6

Auto leveling allows you to map the surface of your material and then by probing the surface using the G38.6 digitizing probing cycle and then applying the resulting height map to adjust your original G-Code file.

This is done using 3rd party software such as AutolevellerAE.

This is an open-source software that can be used to generate the G-Code file to do the initial probing of the material and then the resultant MASSO Probe data.txt file can be loaded in to the software along with your project G-Code file to create a new G-Code file with the coordinates adjusted to suit the material surface.

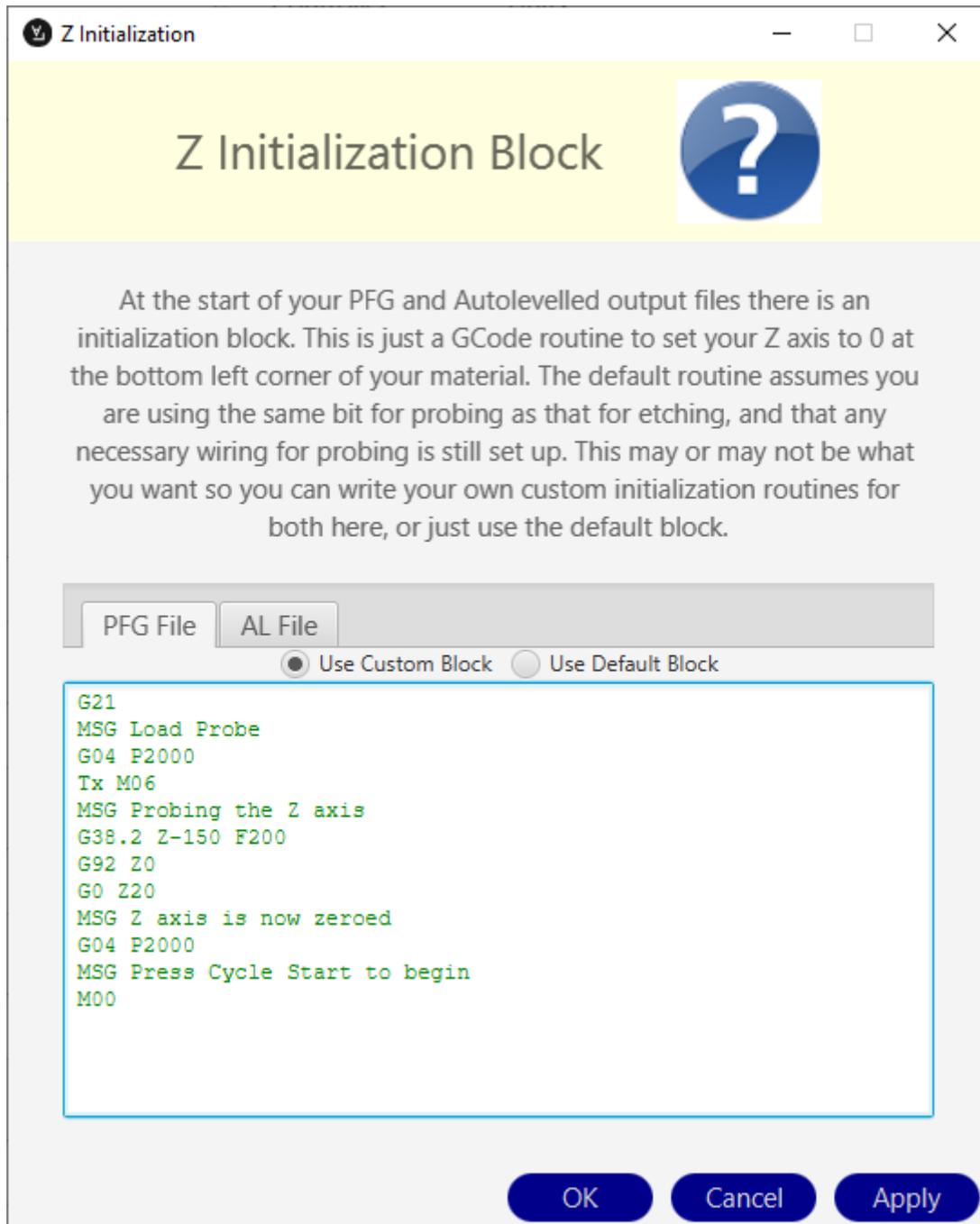
AutolevellerAE can be down loaded [here](#).

The following does not seek to teach you how to use this Auto-Leveler software but how to set up the custom parameters to use it with MASSO.

Note: REVO and MASSO has no association with this software.

Setting up the Z initialization routine

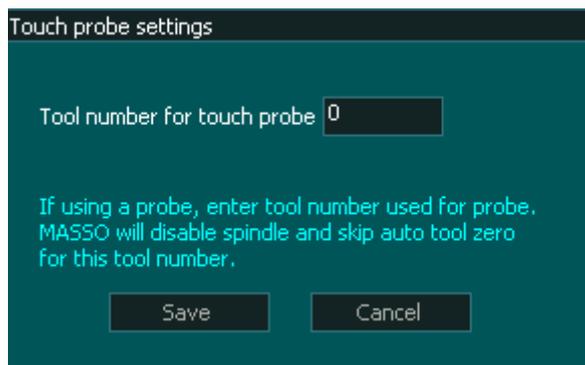
Under Options menus at the top of the screen select Z initialization routine.



- Under Options menus at the top of the screen select Z initialization routine
- Select Use Custom Block
- Enter your start up routine in here.

In this example

- Machine is set to Metric (G21)
- Message is displayed "Load Probe"
- 2 second delay
- Tool change command to executed.
- Replace Tx with the tool number defined in the F1 screen Touch Probe setting. In the example below the touch probe is Tool 0 so you would replace Tx M06 with T0 M06
- Message is displayed "Probing the Z axis"
- The Z axis probes down to the surface at 200mm/min. When the probe touches the surface, it stops. (You may need to change the Z-150 value to suit your machine. The Z value in the G38.2 probing cycle is a Machine Coordinate)
- The Z axis coordinates are set to Z0 using G92 temporary work offset.
- The Z axis moves 20mm up from the surface to Z20
- Message is displayed "Z axis is now zeroed"
- 2 second delay
- Message is displayed "Press Cycle Start to begin"
- M00 Program stop and wait for cycle start.
- After you have a entered your initialization G-Code press Apply followed by OK.



Note: It is important to define a touch probe tool and use this tool while probing even if you do not have an auto tool changer. The Touch Probe tool disables the spindle while it is selected and is an important safety feature.

Custom Z Initialization Block

Please change to suit your machine.

This routine will automatically change your tool to the Probing tool and will automatically use the G38.2 probing routine to zero the Z axis to the correct starting height.

G21

MSG Load Probe

G04 P2000

Tx M06

MSG Probing the Z axis

G38.2 Z-150 F200

G92 Z0

G0 Z20

MSG Z axis now zeroed

G04 P2000

MSG Press Cycle Start to begin

M00

Note: The auto zero probing routine uses G92 to set Z zero. If you stop the file before it finishes you can remove the temporary work offset by going to MDI and issue a G92.1

Custom Controller Options

From the Options Menu at the top of the screen select Custom Controller Options.

Custom Controller Options

Pre-probe Command:

Probe Word:

Post-probe Command:

Pre-zero Command:

Zero Word:

Post-zero Command:

Current Z Parameter:

Starting Probe Parameter:

Open Log Command:

Close Log Command:

File Extension:

Example GCode:

```
M0 (Attach probe wires and clips that need attaching)
(Initialize probe routine)
G0 Z25 (Move clear of the board first)
G1 X12.042 Y10.1 F600 (Move to bottom left corner)
G0 Z1 (Quick move to probe clearance height)
G38.6 Z-1 F100
G0 Z1 (Move Z to above the contact point)
G38.6 Z-1 F100
G0 Z1

MSG Starting probing cycle
G0 Z1
G1 X12.042 Y10.1 F600
```

OK Cancel Apply

- From the Options Menu at the top of the screen select Custom Controller Options.
- Probe Word: G38.6
- Open Log Command: MSG Probing Complete
- Close Log Command: G92.1
- File Extension: Enter nc

Note: If you have values showing in the parameter fields that are grayed out, they can be ignored.

Create a Probing Mesh

AutolevellerAE 0.9.5u2

File Options Tools Help

Mesh Probe (RPF)

*Starting X: 0

*Starting Y: 0

*X Width: 100

*Y Length: 100

XY Feed: 2000

Z Feed: 300

Probe Depth: -3

Probe Clearance: 1

Point Spacing: 10

Probe Safe Height: 20

* - Required

Note: Mesh fields display the Mesh size without insets

Total Points	Points per Row	Rows
121	11	11

Controller: Custom Units: Millimeters Insets: 0 0 0 0

Mesh/Probe Area: Probe Point: Linear Toolpath: Traversal Move: Arc Toolpath: Drill Cycle: X

Job/OGF Area: Not set Mesh/Probe Area: Units=Millimeters X=0 Y=0 Width=100 Height=100

Level Options: Maximum Segment Length=5.0, Init Block=Custom, PFG Init Block=Custom

Autolevel Generate PFG

- **Starting X:** Enter the X working coordinate you wish to start probing from.
- **Starting Y:** Enter the Y working coordinate you wish to start probing from.
- **X Width:** Enter the X working coordinate you wish to finish probing.
- **Y Length:** Enter the Y working coordinate you wish to finish probing.
- **XY Feed:** Enter the feed rate you wish to travel between probing locations.
- **Z Feed:** Enter the feed rate you wish to prove the Z axis.
- **Probe Depth:** Enter the Z axis working coordinate for the end of probing. This is the depth that probing will stop and a "Probing error at:" entry is made in the "MASSO Probe data.txt" file before moving to the next coordinate.
- **Probe Clearance:** Enter the Z height working coordinate that the Z axis will rise up to after each probe. If you make this too high probing will take longer than needed and if you make it too low you may hit the part while moving to the next probing position.
- **Point Spacing:** Enter the distance between the probing locations.
- **Probe Safe Height:** This is used at the end of the probing cycle to rise the Z axis above the surface.

Once you are happy with the parameters press Generate PFG to create the probing G-Code file.

Pro Tip: This software can also be used to create cloud point captures of 3D objects using the same process. Select a point spacing that suits the item you are probing. A finer Point Spacing will render a more detailed data capture which you can pass to other software for rendering to a 3D model.

Create an Auto Leveled file

Basic overview

- Load your G-Code file Called OGF in the software.
- Load the Captured MASSO Probe Data.txt File saved to your MASSO Flash drive under the File Menu Raw Data File (RPF)
- Adjust parameters as needed. Please see the Autoleveller software manual for full details on how to use this aspect of the software.
- Press Autolevel to generate adjusted G-Code file to run on MASSO.

WARNING: The incorrect use of this G-Code can cause damage to your probing equipment; the item being probed and personal injury. Please understand and exercise with care.

Probing Check sheet

Setting up the PFG File

- Set your X & Y start coordinates of the probe to match the X & Y position where the cutting takes place or you will scan the wrong position.
- Set the Probing depth to be sufficient to probe the full depth of the surface. Where possible stay within the travel of your touch probe to reduce the chance of damage to the probe.
- Set the Probe clearance height to the minimum needed to clear any part of the surface being probed for the most efficient probing cycle.
- Set the Point Spacing to suit the type of surface you are mapping. Course for Auto levelling and fine for 3d rendering.
- Set the Safe probe height high enough to clear clamps and other obstructions at the end of the file.

Getting Ready to probe

- Load the probing PFG File
- Load the Probing tool. Tx M06 where Tx is the tool number defined in the MASSO F1 Screen Touch probe parameter.
- Set your X & Y zero position of the probe to match the X & Y zero point of the cutter or you will scan the wrong position.
- Move to the Start X Y position you selected for probing. This may not be X0 Y0
- Set the Z zero point of the probe above the surface of the subject being probed if you are using the auto zero probing routing shown in the example above.
- Run the G-Code file.

G53 – Move In Absolute Machine Coordinates

This command is used move one or more axis to a specified location. If multiple axis is called, they will all move together to the desired location in a straight line and arrive at the same time. The axis can be linear, angular or a combination of both.

Syntax & Parameters

- G53 followed by the axis you wish to move and its coordinate. Multiple axis may be specified on the same line.
- X, Y, Z, A, B Value - specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.
- G00 - G00 can also be used in combination to move at rapid feed rate.

Example program to move at rapid feed rate

```
N10 G00 G53 X10 Y20
```

In the program above, the axis will move to X 10 and Y 20 of the absolute machine coordinates at rapid feed rate.

Example program to move at a specified feed rate

```
N10 G53 X10 Y20 F100
```

In the program above, the axis will move to X 10 and Y 20 of the absolute machine coordinates at 100 millimeters/minute feed rate.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G54 to G59 – Select Work Offset Coordinate System

The G54 to G59 commands are used to select the current work offset for use. The user can use the F4 screen or G10 command to set the offset values.

Syntax & Parameters

G54 to G59

Example program

```
N10 G55
```

In the G-Code above, the work offset G55 is selected to offset the machining position.

G73 – High Speed Peck Drilling

This command is Canned Cycle used for high-speed peck drilling.

Syntax & Parameters

- G73 followed by axis, R, Q, K & F values
- X, Y, Z, A, B Value - specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.
- R Value - This is the retract position in the Z axis. This value must be specified.
- Q Value - The Q value is peck (incremental) value. A positive non zero Q value must be specified.
- K Value - This is number of times the cycle needs to be repeated.
- F Value - The F value defines the feed rate at which the axis will move at.

Example program

```
N10 G99 G73 X10 Y10 Z-8 R2 Q1 F100  
N20 X20  
N30 X30
```

- The first line moves the X & Y axis to 10mm position with retract plane set to 2mm, drilling to Z -8mm, peck of 1mm at 100 mm/minute feedrate and starts drilling.
- The second line moves X axis to 20mm position and drill a hole as per the same values.
- The third line moves X axis to 30mm position and drill a hole as per the same values.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G80 – Cancel Modal Motion

This command is used to cancel Canned Cycles such as G73, G81, G82 & G83.

Syntax & Parameters

- G80

Example program

```
N10 G99 G73 X10 Y10 Z-8 R2 Q1 F100  
N20 X20  
N30 G80
```

- The first line moves the X & Y axis to 10mm position with retract plane set to 2mm, drilling to Z -8mm, peck of 1mm at 100 mm/minute feed rate and starts drilling.
- The second line moves X axis to 20mm position and drill a hole as per the same values.
- The third line cancels the G73 canned cycle.

Note: Units are defined as either inches or mm depending on your machine's setup or if G20 or G21 command in use.

Note: After using G80 the current mode is cancelled and replaced with G00 Rapid motion until it is changed in G-Code.

G81 – Drilling Cycle

This command is Canned Cycle used for drilling cycle.

Syntax & Parameters

- G81 followed by axis, R & F value.
- **X, Y, Z, A, B Value** - specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.
- **R Value** - This is the retract position in the Z axis. This value must be specified.
- **F Value** - The F value defines the feed rate at which the axis will move at.

Example program

```
N10 G99 G81 X10 Y10 Z-8 R2 F100
```

```
N20 X20
```

```
N30 X30
```

- The first line moves the X & Y axis to 10mm position with retract plane set to 2mm, drilling to Z -8mm at 100 mm/minute feed rate and starts drilling.
- The second line moves X axis to 20mm position and drill a hole as per the same values.
- The third line moves X axis to 30mm position and drill a hole as per the same values.

Note: Units are defined as either inches or mm depending on your machine's setup or if G20 or G21 command in use.

G82 – Drilling Canned Cycle With Dwell

This command is Canned Cycle used for drilling cycle with dwell (pause) at the bottom of hole.

Syntax & Parameters

- G82 followed by axis, R, P & F values.
- **X, Y, Z, A, B Value** - specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.
- **R Value** - This is the retract position in the Z axis. This value must be specified.
- **P Value** - This is the dwell (pause) at the bottom of hole. This value is in milliseconds.
- **F Value** - The F value defines the feed rate at which the axis will move at.

Example program

```
N10 G99 G82 X10 Y10 Z-8 R2 P1000 F100
```

```
N20 X20
```

```
N30 X30
```

- The first line moves the X & Y axis to 10mm position with retract plane set to 2mm, drilling to Z -8mm with 1 second dwell at bottom of the hole at 100 mm/minute feedrate and starts drilling.
- The second line moves X axis to 20mm position and drill a hole as per the same values.
- The third line moves X axis to 30mm position and drill a hole as per the same values.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G83 – Peck Drilling For Deeper Holes

This command is Canned Cycle used for peck drilling deeper holes.

Syntax & Parameters

- G83 followed by R, Q, K & F values.
- **X, Y, Z, A, B Value** - specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.
- **R Value** - This is the retract position in the Z axis. This value must be specified.
- **Q Value** - The Q value is peck (incremental) value. A positive non zero Q value must be specified.
- **F Value** - The F value defines the feedrate at which the axis will move at.

Example program

```
N10 G99 G83 X10 Y10 Z-8 R2 Q1 F100
```

```
N20 X20
```

```
N30 X30
```

- The first line moves the X & Y axis to 10mm position with retract plane set to 2mm, drilling to Z -8mm, peck of 1mm at 100 mm/minute feed rate and starts drilling.
- The second line moves X axis to 20mm position and drill a hole as per the same values.
- The third line moves X axis to 30mm position and drill a hole as per the same values.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

G90 – Set Distance Mode To Absolute

This command is used to set mode to absolute. This is the default mode on power up.

Syntax & Parameters

- G90

Example program

```
N10 G90
```

The G-Code above will set the machine units to absolute.

G91 – Set Distance Mode To Incremental

This command is used to set mode to incremental.

Syntax & Parameters

- G91

Example program

```
N10 G91
```

The G-Code above will set the machine units to incremental.

G92 – Temporary Work Offset

This command is used set temporary work offset values. The temporary work offsets are used in combination to the main work offsets G54 to G59.

Syntax & Parameters

- **G92** followed by axis values.
- **X, Y, Z, A, B Value** - specifies the axis you wish to move following the distance to move. The distance value will be the current machine units in use.

Example program

```
N10 G92 X10 Y20
```

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

Note: When A G92 work offset is used and a DRO is zeroed the offset value will show on the DRO instead of 0. Use G92.1 to clear the temporary work offset.

Note: G92 work offsets can be cancelled by using G92.1 in G-Code or using MDI.

G92.1 – Cancel Temporary Work Offset

This command is used cancel the G92 temporary work offset values.

Syntax & Parameters

- G92.1

Example program

```
N10 G92.1
```

The G-Code above cancels the temporary work offset values.

G93 – Inverse Time Mode

This command is used set the current feed rate mode to Inverse Time Mode.

Syntax & Parameters

- **G93**
- **X, Y, Z, A & B** These are the destination coordinates of the axis.
- **F Value** This is the feed rate. The move will be completed it 1/F minutes or 60/F seconds.
- **G01, G02 and G03** It is Mandatory to include a Feed rate for these commands
- **G00** No Feed rate is required

Example program

```
N10 G93
```

```
N20 G01 X1 A1 F30 #This will move TO position X1 A1 in 2 seconds
```

```
N30 G00 X0 A0 #This will Rapid to X0 A0
```

The program above sets the feed rate mode to Inverse Time Mode.

G94 – Units Per Minute Mode

This command is used set the current feed rate mode to Units per Minute Mode. When in this mode the feed rate F value is in units per minute.

Syntax & Parameters

- G94

Example program

```
N10 G94
```

The program above sets the feed rate mode to Units per Minute Mode.

G95 - Feed Per Revolution

G95 (Feed Per Revolution) is a modal G-code that instructs the control to interpret feed commands as mm per revolution (mm/rev) or inches per revolution of the spindle.

Syntax & Parameters

- **G95**
- **F Value:** The F value specifies the distance travelled per revolution of the spindle used to calculate the G95 feed rate.
- **S Value:** This specifies the spindle speed used to calculate the G95 feed rate

Example program

```
N10 G21
```

```
N20 S800 M3
```

```
N30 G95
```

```
N40 G01 X20 F0.1
```

The program above would cause the X axis to advance 10mm for every revolution of the spindle until the X axis reaches 20mm. The Feed rate will be 80mm per minute.

G95 feed rate above is calculated as follows

(Spindle speed) x (Specified feed rate) = G95 feed rate

800 x 0.1 = 80mm

Using G95 with G96

Warning: Be aware that the CSS spindles speed does not work with G95 Feed Per Revolution.

- Using G95 & G96 together may cause unexpected results if used incorrectly.
- When G95 is used with G96 an S value must be specified for G95 to work.
- The S value must be given before the G96.
- The S value as part of the G96 command is not used by G95.
- The spindle speed calculated by G96 is not used by G95.

G96 – Turn on Constant Surface Speed (CSS)

This command is used to enable Constant Surface Speed (CSS) mode in lathe version.

Syntax & Parameters

- **G96** followed by **D & S** values.
- **D Value** - This is to set the maximum spindle RPM that the spindle can achieve in CSS mode as the tool gets closer to the center.
- **S Value** - To set surface speed value. The units will be the current machine units in use. Metric uses meters per minute. Imperial uses feet per minute.

Example program

```
N10 G21  
N20 G0 X50  
N30 G96 D12000 S200  
N40 M3  
N50 G1 X1 F80
```

- In the example above, CSS mode is enabled with a surface speed of 200 meters per second and the maximum RPM is set to 12000.
- The X axis starts at X 20 from the center and moves towards the center at a feed rate of 80mm/min. Spindle speed starts at a calculated 3185 rpm and the speed increases the closer it moves to the spindle center. It reaches the specified maximum of 12000 rpm 5.2mm from the center at which time the speed remains at 12000rpm until the X axis reaches the final position of 1mm.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

Warning: Please be aware that the CSS spindle speed does not work with G95 Feed Per Revolution. Using G96 & G95 together may cause unexpected results if used incorrectly. For more information, please see G95.

G97 – Turn off Constant Surface Speed (CSS)

This command is used to disable Constant Surface Speed (CSS) mode and set to RPM mode in lathe version.

Syntax & Parameters

- G97

Example program

```
N10 G97
```

In the example above, CSS mode is disabled and the system is set back to RPM mode.

G98 – Canned Cycle – Retract Back To The Initial Z

This command is used in combination with canned cycles to retract Z axis to the position before canned cycles was started.

Syntax & Parameters

- G98

Example program

```
N10 G98 G73 X10 Y10 Z-8 R2 Q1 F100
```

In the program above, after finishing the drilling cycle the Z axis will retract back to the position before the drilling cycle.

G99 – Canned Cycle – Retract Back To R Plane

This command is used in combination with canned cycles to retract the Z axis to the R plane value.

Syntax & Parameters

- G99

Example program

```
N10 G99 G73 X10 Y10 Z-8 R2 Q1 F100
```

In the program above, after finishing the drilling cycle the Z axis will retract to the value defined by the R word.

MSG - Print message to screen

This command allows the user to add messages to their G-Code which will be displayed on the screen in the tool path area.

These can be used as training aids, instructions or reminders of when to do something.

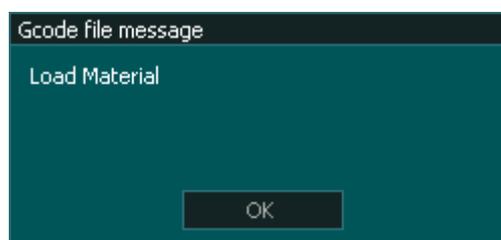
Syntax & Parameters

- MSG - followed by a space and then the message to be displayed on screen
- MSG - when no message follows the MSG command the current on screen, message will be removed from the screen.

Example program to display the message "Load Material " on the screen

```
N10 MSG Load Material
```

- Message length is limited to a single line of 34 characters and messages that exceed this length will only display the first 34 characters.
- Messages can be cancelled by pressing the OK button on the message display or using the MSG G-Code by itself.
- Messages will remain on screen until cancelled, overwritten by the next message or a System message.



N10 MSG Load Material

N20 G04 P3000

N30 MSG

- Displays "Load Material" on screen
- Waits for 3 seconds
- Removes message from screen.

Pro Tip: Messages can be used with M00 & M01 to stop the machine so that the user can read the message before proceeding if required.

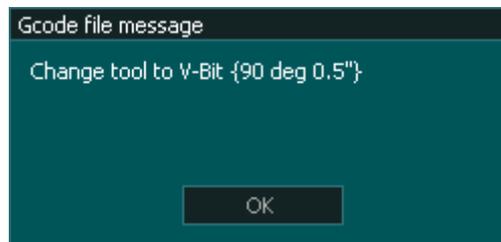
Example program to display the message "Change tool to V-Bit {90 deg 0.5}" on the screen before a tool change:

N10 MSG Change tool to V-Bit {90 deg 0.5"}

N20 M01

N30 T3 M06

- Messages can be written into the post processor to automatically add this type of message.
- The M01 in the example will stop the machine until the Cycle Start is pressed allowing the user time to read the message when they are ready. There is no need to press OK in this instance because it will immediately be overwritten by the system tool change message.
- If Optional Stop CTRL+O is turned off then the machine will not stop and the user will not see the message as it will immediately be overwritten by the System message to Change tool



Note: Multiple messages can be used with G04 Pauses to display messages in a timed sequence.

Example program to display multiple messages on the screen

N10 MSG READ DIAL FOR BACKLASH MEASUREMENT

N20 G04 P3000

N30 MSG ENTER THIS VALUE INTO MASSO BACKLASH

N40 G04 P3000

N50 MSG TEST COMPLETE

In some applications it may be desirable to provide instructions in a timed sequence. Using a G04 pause between messages will allow them to be displayed for the specified time before it is overwritten by the next message.

The [video clip](#) shows an example of using the MASSO MSG G-Code to create a backlash test sequence with instructions for the tester to carry out at each step of the process.

F - Feed rate

This G-Code is used to set the feed rate.

It can be used as part of another G-Code command or by itself.

The specified feed rate remains in operation until overwritten

The feed rate does not override the maximum feed rate specified for any given axis.

Syntax & Parameters

- **F Value** - This defines the feed rate the machine will use for the specified operation.

Example program

```
N10 F1500
```

The machine will use a feed rate of 1500 units per minute.

Note: Units are defined as either inches or mm depending on your machine's setup or G20 or G21 command in use.

N - Number

This command is used as a number within the G-Code.

It performs no function within the G-Code other than an identifier of a particular line of G-Code

This is not to be confused with the line number that MASSO uses when running a program.

Syntax & Parameters

- **N Value** - The number that follows the N is used to denote the line number

Example program

```
N10
```

```
N20
```

```
N30
```

S - Speed / Intensity

This command is used to specify a Spindle Speed or if used with Laser specifies the Laser intensity

Syntax & Parameters

S Value - This specifies the spindle speed in RPM

Example program

```
N10 S10000
```

This sets a spindle speed of 10000 rpm and is usually followed by an M3 or M4 on the same line

() - Comments

This command is used to allow the user to add comments into their G-Code file. Anything written between the open and closed bracket is ignored.

Syntax & Parameters

- (- The open bracket is used to define the start of a comment.
-) - The close bracket is used to define the end of a comment.

Example program

```
N10 (My comment)
```

Invalid Gcode

WARNING: If a line of G-Code contains unsupported G-Code commands, incorrect syntax or hidden unprintable characters, the entire line of code is ignored.

When MASSO ignores a line of code the program will move to the next valid line of G-Code and continue execution.

WARNING: Running incorrectly written or formatted G-Code may cause unpredictable or unwanted moves and may result in damage to the work or machine.

M00 – Program Stop

This command stops the program. Pressing cycle start will resume the program.

Syntax & Parameters

- M00

Example program

```
N10 G00 X0 Y0
```

```
N20 G00 X10
```

```
N30 M00
```

```
N40 G00 Y10
```

- The first line will move both X and Y axis to 0.00 position.
- In line two the X axis will move to X 10 position.
- In line three the program will stop and wait for user to press cycle start.

M01 – Optional Program Stop

This command stops the program if optional stop is turned on. Pressing cycle start will resume the program. You can press CTRL + O when on the F2-Program & MDI screen page to toggle between Optional Stop ON/OFF and the status is displayed on the bottom status bar.

Syntax & Parameters

- M01

Example program

```
N10 G00 X0 Y0
```

```
N20 G00 X10
```

```
N30 M01
```

```
N40 G00 Y10
```

- The first line will move both X and Y axis to 0.00 position.
- In line two the X axis will move to X 10 position.
- In line three the program will stop and wait for user to press cycle start if optional stop is set to On, else will run line four and move Y axis to 10.

M02 – Program End

This command ends the program.

Syntax & Parameters

- M02

Example program

```
N10 G00 X0 Y0
```

```
N20 G00 X10
```

```
N30 M02
```

```
N40 G00 Y10
```

- The first line will move both X and Y axis to 0.00 position.
- In line two the X axis will move to X 10 position.
- In line three the program will stop and will not run line four.

M03 – Spindle ON (Clockwise)

This command starts the spindle in clockwise direction at the RPM set by the S command

Syntax & Parameters

- **M03** - Clockwise command followed with or without S value.
- **S** - The S value defines the required RPM. If the S value is not given then the last S value is automatically used.

Example program

```
N10 M03 S1200
```

- In this example the spindle is started in clockwise direction at 1200 RPM.

M04 – Spindle ON (Counter Clockwise)

This command starts the spindle in counter clockwise direction at the RPM set by the S command

Syntax & Parameters

- **M04** - Clockwise command followed with or without S value.
- **S** - The S value defines the required RPM. If the S value is not given then the last S value is automatically used.

Example program

```
N10 M04 S1200
```

- In this example the spindle is started in counter clockwise direction at 1200 RPM.

M05 – Spindle OFF

This command stops the spindle.

Syntax & Parameters

- **M05**

Example program

```
N10 M05
```

- In this example the spindle will be stopped.

M06 – Tool Change

WARNING: When using an auto tool changer, the M06 must be preceded by an M05 or the drawbar will not release and can result in damage to the tool changer.

CAUTION: This command can be used in different combinations and wrong command can result in unexpected loading of tool.

Valid tool numbers are T0 to T99.

This command is used to change tool immediately and can be used with T value.

Syntax & Parameters

- **M06**
- **T Value** – specifies the tool number to change, this value can be used before M06 or after M06 but will have a very different process of tool loading.

Example program

```
N10 M05
```

```
N20 T5 M06
```

In the program above, M05 stops the spindle then T5 tells the system that we would like to load tool number 5 and M06 is used to tell the system to load the tool.

M07 – Turn Mist Coolant On

This command turns On the Mist Coolant output.

Syntax & Parameters

- **M07**

Example program

```
N10 M07
```

In this example the Mist Coolant is turned On.

M08 – Turn Flood Coolant On

This command turns On the Flood Coolant output.

Syntax & Parameters

- **M08**

Example program

```
N10 M08
```

In this example the Flood Coolant is turned On.

M09 – To Turn All Coolant Off

This command turns Off both Mist Coolant and Flood Coolant output.

Syntax & Parameters

- M09

Example program

```
N10 M09
```

In this example if any of the Mist Coolant or Flood Coolant was On, then this code switches them Off.

M10 – Chuck Or Rotary Table Clamp closed

This command turns the Chuck Clamp output to LOW.

This is considered in MASSO logic as the Chuck Clamp being closed.

CAUTION: For safety, output LOW means the clamp is closed and an interlock signal from the spindle drive must be wired for safety so that the clamp does not open while the spindle/chuck is spinning.

Syntax & Parameters

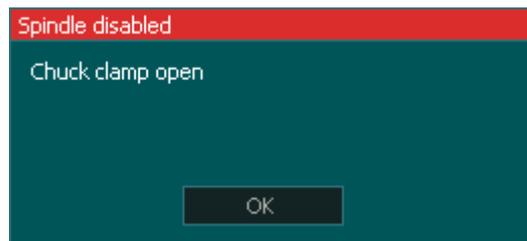
- M10

Example program

```
N10 M10
```

In this example the Chuck Clamp is closed.

If you try and start the spindle while the Chuck Clamp is open you will see the following Message.



Pressing OK will remove the message.

To continue you need to close the Chuck Clamp and manually start the Spindle before pressing Cycle Start if you are running G-Code program.

CAUTION: If you run your program with the Chuck clamp open the spindle will not start and the program will continue with the spindle off. This may cause damage to your machine or work.

M11 – Chuck Or Rotary Table Clamp Open

This command sets the Chuck Clamp output to High.

This is considered in MASSO logic as the Chuck Clamp being open.

CAUTION: For safety this output HIGH means the clamp is open and an interlock signal from the spindle drive must be wired for safety so that the clamp does not open while the chuck is spinning.

Syntax & Parameters

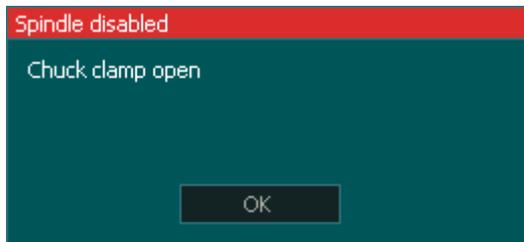
- M11

Example program

N10 M11

In this example the Chuck Clamp is opened.

If you try and start the spindle while the Chuck Clamp is open you will see the following Message.



Pressing OK will remove the message.

To continue you need to close the Chuck Clamp and manually start the Spindle before pressing Cycle Start if you are running G-Code program.

CAUTION: If you run your program with the Chuck clamp open the spindle will not start and the program will continue with the spindle off. This may cause damage to your machine or work.

M30 – End The Program And Rewind

This command ends the program and moves back to the first line of the G-Code file.

Syntax & Parameters

- **M30**
- **L Value** When added to the M30 command causes the program to be repeated for the specified number of cycles. L10 will run the Gcode 10 times before rewinding stopping
- **L0 Value** When L0 is added to M30 the Program will rewind and repeat on an infinite loop.

Example program

```
N10 G00 X0 Y0  
N20 G00 X10  
N30 M30  
N40 G00 Y10
```

- The N10 line will move both X and Y axis to 0.00 position.
- In line N20 the X axis will move to X 10 position.
- In line N30 the program will stop and the file will rewind back to line N10, pressing cycle start will start the program from N10 line.

Infinite Loop

```
N10 G00 X0 Y0  
N20 G00 X10  
N30 M30 L0
```

- This program executes lines N10 to N30 before rewinding back to N10 and starting again automatically. It will continue until it is manually stopped.

Repeat 20 times

```
N10 G00 X0 Y0  
N20 G00 X10  
N30 M30 L20
```

- This program executes lines N10 to N30 before rewinding back to N10 and starting again until it has completed 20 cycles at which time it will rewind to N10 and stop.

M62 – Turn On Digital Output Synchronized With Motion

This command is used to switched ON any of the 16 auxiliary outputs synchronized with the start of the next motion command.

Note: If no motion is commanded, the output will not turn ON. It is best to specify motion immediately following the M62 command.

Syntax & Parameters

- **M62**
- **P Value** - The P value is required and defines the output number to switch On. This value can be between 1 to 16.

Example program

```
N10 G0 X0  
N20 M62 P1  
N30 X10  
N40 M63 P1  
N50 X20
```

In the example above, the X axis moves to X0 position, then auxiliary output 1 is turned ON when the X axis starts to move towards X10 position. Once the axis reaches X10 position and starts moving to X20 position, the auxiliary output 1 is turned OFF.

Note: For M62 P# to work you must assign an output as an auxiliary output in F1-Setup screen under OUTPUTS list.

M63 – Turn Off Digital Output Synchronized With Motion

This command is used to switched OFF any of the 16 auxiliary outputs synchronized with the start of the next motion command.

Note: If no motion is commanded, the output will not turn OFF. It is best to specify motion immediately following the M63 command.

Syntax & Parameters

- **M63**
- **P Value** - The P value is required and defines the output number to switch On. This value can be between 1 to 16.

Example program

```
N10 G0 X0  
N20 M62 P1  
N30 X10  
N40 M63 P1  
N50 X20
```

In the example above, the X axis moves to X0 position, then auxiliary output 1 is turned ON when the X axis starts to move towards X10 position. Once the axis reaches X10 position and starts moving to X20 position, the auxiliary output 1 is turned OFF.

Note: For M63 P# to work you must assign an output as an auxiliary output in F1-Setup screen under OUTPUTS list.

M64 – Turn On Digital Output Immediately

This command is used to switched ON any of the 16 auxiliary outputs. If this command is used between motion commands, then the motion will come to full stop before switching ON the output.

Syntax & Parameters

- M64
- **P Value** - The P value is required and defines the output number to switch On. This value can be between 1 to 16.

Example program

N10 M64 P1

N20 M64 P4

- The N10 line will switch ON the auxiliary output 1.
- The N20 line will switch ON the auxiliary output 4.

Note: For M64 P# to work you must assign an output as an auxiliary output in F1-Setup screen under OUTPUTS list.

M65 – Turn Off Digital Output Immediately

This command is used to switched OFF any of the 16 auxiliary outputs. If this command is used between motion commands, then the motion will come to full stop before switching OFF the output.

Syntax & Parameters

- M65
- **P Value** - The P value is required and defines the output number to switch On. This value can be between 1 to 16.

Example program

N10 M65 P1

N20 M65 P4

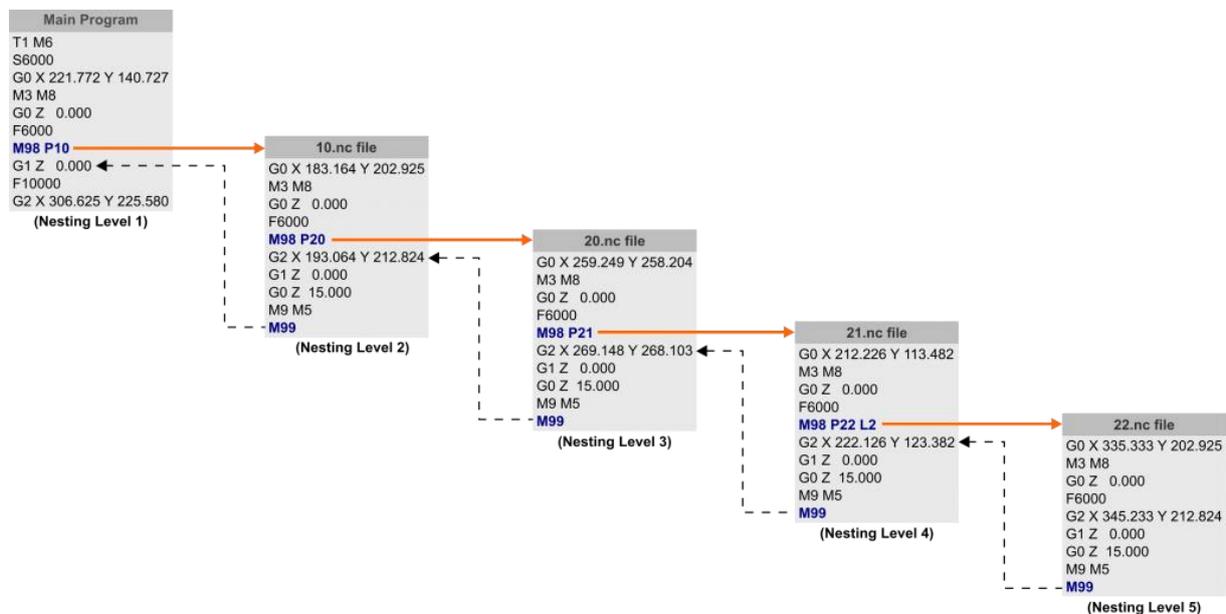
- The N10 line will switch OFF the auxiliary output 1.
- The N20 line will switch OFF the auxiliary output 4.

Note: For M65 P# to work you must assign an output as an auxiliary output in F1-Setup screen under OUTPUTS list.

M98 & M99 – Sub Program Call

MASSO supports Sub-Program Call using M98 & M99 codes. Subprograms with up to 5 levels of nesting is supported, with this feature sub programs can be called from the main program without the need to rewrite the same program again and again.

When a sub program is called, MASSO looks for a separate ".nc" file and runs that as a sub-program. Having a separate file approach means that programs that have common features can now be used with other programs.



Note: All sub-program files must be on the root folder of the USB Flash drive.

Note: Loading a subprogram directly, not via a sub-program call, will cause MASSO to hang.

Note: Subroutine toolpaths are not drawn on the screen during program load.

M98 Sub-Program Call

Syntax & Parameters

- **M98**
- **P Value** - The P value is used to define the sub program file name.
- **L Value** - The L value can be used to run the sub program multiple times.

Example program

N10 M98 P10 L5

The program above will look for the 10.nc file as set by the P value and will run it 5 times as set by L value.

NOTE: If the 10.nc file is not found an error message on the screen is displayed and program goes into feed-hold. When a program is loaded from F6 screen, MASSO checks that all sub-program files are on the USB and if a file is missing an error is shown to the user

M99 End Sub-Program or Return

M99 command is used to End Sub-Program or Return back to the main program.

Syntax & Parameters

- M99

Example program

```
N10 M99
```

When the line above is found in a sub program file, the control returns back to the main program.

CAM Post Processors

Artcam [2010](#) [2015](#) [2018](#)

[BobCAD-CAM](#)

[Fusion 360](#)

Other CAM Software

Q: Why is my CAM software not mentioned?

A: There are numerous CAM software makers and it is not possible to list them all. If you do not see your CAM software supplier listed, please contact them directly and ask for a Post processor for your MASSO controller. If they do not have one available, they should be able to make one for you. Should they need assistance or have questions they can contact MASSO support and we will be happy to assist.

Q: Who makes MASSO Post Processors?

A: Post Processors are made by the CAM Software supplier.

As writers of the CAM software, they have the expertise in writing Post Processors to suit their product. Should they need assistance or have questions they can contact MASSO support and we will be happy to assist.

Q: Why does MASSO not write Post Processors?

Post Processors are made by the Cam software writers as they have the expertise in their own products. In the past we have approached various CAM software makers and requested Post Processors for MASSO but as we do not have their product or a support contract they are not interested. MASSO support is happy to work with CAM software makers to help with the development of Post Processors.

Q: Why are there links to some Post processors and not others?

A: The Post Processors were provided by users of the various software or by the CAM manufacture themselves and linked to for convenience. If you have a post processor you have made and wish to share, please post it in the Forums Post Processor section. The person who makes the post processor is responsible for maintaining it.

Contact MASSO support and a page with link will be added the new post processor.

Before turning on the machine

- Make sure the training and programming manuals are fully understood.
- Cables with damaged insulation cause current leakage and electric shock. Strictly check these before all operations.
- Use oil-proof safety shoes, closed-sided safety glasses and other safety equipment.
- Before turning on the machine, turn on the workshop feed switch, the machine main switch and the switch on the operator panel, respectively.
- Check the coolant level and top up if necessary.
- Check that the motor and slides are lubricated normally.

Preparations

- Tooling should be done in accordance with the standards, dimensions and type of the machine.
- Worn, dull tools cause serious damage. First, replace all worn tools with new ones.
- The work area should be illuminated in order to carry out safety checks.
- Equipment and tools around the machine should be placed in a way that allows easy movement.
- Tools and other apparatus should not be placed in tool holders, protective covers and similar places.
- Be sure to do a trial run whenever a new tool is installed.

First Run

When you power up your M1A, Masso screen will turn on automatically and you will see the interface in seconds.

First thing you need to do every time you turn on M1A is homing.

To perform this, Touch **HOME** button for 3 seconds on **F3 screen**, or use **CTRL+ALT+HOME** key combination on keyboard.

Machining A Workpiece

- In order not to be caught by the moving parts and working parts of the machine, the hair of the operators should be collected, their clothes should be suitable in terms of OHS, and jewelry such as earrings, rings, necklaces, bracelets should not be used.
- When handling a very heavy item, work with two or more personnel to reduce the risk of accident.
- Securely attach the workpieces to the workbench.
- Never touch the rotating workpiece or spindle.
- While M1A is machining a workpiece, do not attempt to clean or deburr that part with a cloth or by hand. Always stop your machine first and then use a brush or an air gun.
- Use a brush to remove the sawdust from the tip of the cutting tool, never with bare hands.
- When processing magnesium alloy materials, wear a protective mask.
- Never open the machine cover during automatic operation.

- When machining heavy workpieces, take the necessary precautions by remembering that the burrs may splash and cause any ignition or fire.
- When the machine is idle, the sound level is below 65 dB(A).
- Machine noise may exceed 75 dB(A) while machining a part.
- Check the technical specification of the raw material to be processed.
- Use a personal dust mask if there is a possibility of harmful dust generation.

Stopping The Machining

- When you want to turn off the machine after completing a workpiece, first turn off the power cut-off button on the machine operation panel and then the machine main switch.
- Remove the tool from the spindle before long machine downtimes.

Finishing The Machining

- Return each moving part of the machine to its starting position.
- Press E-Stop Button
- Remove the machined workpiece.
- Clean up the machine table from dusts and chips.
- Check the coolant, hydraulic and lubricating oil levels and top up if necessary.
- Before leaving the machine at the end of the shift, turn off the power button on the operator panel, turn off the main switch and the factory supply switch, respectively.

Door Trigger and Enclosure

M1A has a door trigger. When the spindle is running, opening the door will trigger the sensor which causes the project to pause, and the spindle to pause and move up to a park position.

When the sensor is triggered, Masso will display the pause screen.

Once the door is closed again, you'll need to click the Resume button on the pause screen to continue the job.

The enclosure and protective acrylic door are there for two reasons: to contain the mess and to protect you from broken cutters and flying stock.

- End mills, or cutters, are made of carbide, a very hard, very brittle metal. When they break, they snap suddenly and without warning. The window will help protect you from any broken cutters.
- Stock material can break away during cutting. This can happen because you failed to clamp the workpiece properly, or the zeroing is wrong, or because you were cutting too fast.

If either of these happens during a job, power the machine down (hit the emergency-stop if one is available on your setup) and fix the problem.

In summary, always close the door when the M1A is machining.

NOTE: We also strongly recommend keeping the door closed when the machine is not in use. Leaving the door open for long periods of time could weaken the door hinge.

Door Error Messages

When the door is not fully closed, it cannot disengage the interlock. If this happens, Masso will prompt you to check the door and yellow warning light will blink. If you see a “Please close machine door” or “Machine not idle” message, or if the screen says “BUSY” for a long period of time, make sure the door is appropriately closed.

Install the Collet and Nut

What Is a Collet?

A collet is used to secure an end mill in the spindle. A collet is sized to hold one diameter of cutter. M1A comes with an ER-16 collet which is capable of holding 1 to 16mm cutters. You can buy additional ER-16 collets to hold different diameter cutters from REVO or other vendors.

How to Install the Collet and Nut

To install the collet and nut:

1. Snap the collet into the collet nut:

- a. Set the collet nut on the table with the threads facing up.
- b. Use your thumb to press the collet into the nut until you hear a “click.”
Once the collet is clicked into the nut it is secured in place.

2. Insert the collet into the shaft of the spindle and loosely thread the nut onto the spindle shaft. Do not use the collet wrenches to tighten the collet nut at this time.

WARNING: Never fully tighten the collet nut without first inserting a tool or the collet could break

Install a Probing Pin

After homing the machine, you will be prompted to insert a tool. At this time, you can either install a probe or you can install an end mill. Typically, you will install an end mill only after finding job zero.

To get ready to find job zero in the first project, you’ll need to install the probing pin:

1. Insert the probing pin into the loose collet a minimum of 20mm

— the entire length of the collet and nut.

2. Hold the shaft of the spindle (above the threads) with one hand, while holding the probing pin in position

and tightening the collet nut with your other hand.

3. Tighten the collet nut with your fingers, just until the probing pin will not fall out.

4. Use the shaft wrench to hold the shaft of the spindle in place, while using the collet wrench to tighten the collet nut and secure the probing pin.

5. Follow the probing protocol on Masso Screen. Check the details on relevant page of this guide.

General Tips and Safety Precautions

REVO M1A is a CNC mill.

The milling process takes place when the cutting tool, which is made of very hard materials, attached to the milling motor, rotating at a very high speed, contacts the workpiece and removes chips from the workpiece.

As it can be understood from here, this is an important operation that will not accept carelessness.

Otherwise, situations that may put human life at risk and/or disruption to the structure of the machine may occur.

The M1A is equipped with various safety measures to protect its operator, itself and its equipment from accidents and damage.

All machine operators are required to operate the machine after reading and understanding the reminders given in the machine manuals and taking the necessary special precautions.

Using the machine without completing the maintenance and usage training given by REVO officials may cause dangerous results.

- Find out the location of the E-Stop button, do not hesitate to use it even in the slightest mishap and inform the authorized persons.
- In case of sudden power cuts, turn off the main switch immediately.
- Water or oil spilled on the floor may cause the floor to become slippery. To prevent this type of accident, the place must always be dry and clean.
- Before pressing the buttons, check that you have pressed the correct button.
- Never press a button whose function you do not know.
- Workpieces placed next to the workbench should be placed in a way that does not cause any accident.
- Necessary precautions should be taken to prevent the work pieces from slipping over each other.
- Protect control units, operator panels, electrical control panel from sudden electrical shocks.
- If you are experiencing electrical problems in your area, it is recommended to use UPS (uninterruptible power supply). Otherwise, such shocks may cause malfunctions.
- Do not change parameters and other electrical settings unless it's absolutely necessary.
- Do not remove or wipe the warning signs.
- Isolate your machine from excessive residual dust, corrosive and corrosive excessive humidity, and an environment with chemical vapors.
- Make sure that the water is removed from the air circuit and use an air dryer.
- The machine must be used by competent and OHS trained personnel.
- DO NOT rush to use the machine. It is better not to do a job in CNC milling than to try to do it carelessly and quickly. Review all operation steps before you start, considering the possible damage.
- Observe the items specified in the machine's operating instructions. The specified rotation and advance speeds are of great importance in terms of occupational safety.
- Do not approach the machine, do not open the machine cover, do not touch the buttons when there is someone using the control unit of the machine or a program is running automatically.

- Do not touch the control computer if the machine cover is open and someone is connecting parts to the machine.
- Likewise, do not let anyone else touch the control computer while you are working on the machine.
- Do not use the machine cooperatively. Do not use collaborative methods such as "One person connects the part and another person deals with the computer".
- Handle the whole process yourself, from connecting parts to computer control.
- As soon as you turn on the machine, send it to the HOME location first.
- Do not operate the machine manually unless there is a job that cannot be done with pre-made programs.
- Get support from our company against any situation before working manually.
- If you are going to use it manually, check the spindle speed at the beginning of each job. Incorrect speed can cause the tool to break, fly off or damage the machine.
- If you have a problem of inattention due to drugs, drugs, insomnia, illness, etc., do not use the machine and report the situation to your supervisor.
- If you have done anything that could damage yourself or the machine while using the machine, notify the responsible persons immediately. Failure to report this could magnify the damage that may have occurred.
- Be sure to tighten the collet nut sufficiently when connecting the cutting tools.
- The required tightening force for the ER20 HEX nut in the machine is 55-75Nm.

FAQ

Which End Mills Can I Use?

REVO M1A uses a spindle with an ER-20 collet system as standard.

That means you can use every end mill compatible with ER20 standards in M1A as long as you have the relevant collet.

The machine ships with a 5mm collet and nut, which allows you to use any end mill, also known as a cutter, that has an 5mm diameter shank. The shank diameter determines what size collet should be used to hold the cutter in the spindle.

Which Materials Can the REVO M1A cut?

M1A is designed to machine in accurately materials up through the hardness of non-ferrous alloys such as brass and aerospace-grade aluminum.

- Hardwoods, such as maple, ash, basswood, birch, cherry, walnut, poplar, and oak.
- Softwoods, such as pine, cedar, balsa, and fir.
- Plastics, such as ABS, Delrin (Acetal), Polycarbonate (PC), Acrylic (PMMA), Nylon, HDPE, LDPE, PP, PET, and PVC, Polyethylene, and Polyurethanes.
- Synthetics and composites, such as Renshape and other Polyurethane resins with fillers, as well as foams.
- Metals such as copper, brass, and aluminum, including 6061 and 7075 alloys for example.

What Are Toolpaths and G-code?

When your CNC machine is running, it is executing G-code, which was created by defining toolpaths. Think of toolpaths as your visual guide to “programming” your CNC machine.

Toolpaths tell the machine how to cut your design into the material. They tell the machine what tool to use, what area of the material to cut, and how fast to cut.

Toolpaths tell the machine how deep to cut each pass and how deep to cut when the feature is finished.

The machine has both depth per pass and max depth parameters because the end mill cuts toolpath depth a little at a time, if the end mill tried to cut out the entire depth at once, it would either get stuck, or cause a tear out (a rip in the material).

You never have to look at G-code if you don't want to. It's not vital to know or understand how it works or how to read it. But it is important is to understand that G-code is made from toolpaths.

Machine Operating Checklist

1. Be Safe

Always follow the safety guidelines listed at the beginning of this document. Always wear appropriate safety equipment, especially safety glasses/goggles and hearing protection.

2. Check the Machine

Check that all screws are tight, rails are in good condition with no nicks or other damage, wiring is in good condition with continuity and securely fastened, and that nothing is frayed or broken and everything is clear and safe.

3. Secure the Workpiece

Secure the workpiece (the material being milled) right-side up and in the desired orientation to the worksurface using a technique appropriate to the material.

4. Clear the Work Area

Ensure the work area is clear and the spindle can move without interference.

5. Close the Door.

6. Home Your Machine

7. Set Job Zero

8. Double Check the Work Area

Ensure that nothing has been left in the work area. Make sure the door is closed.

9. Examine and Install the End Mill

Examine each end mill prior to use to ensure that it is sharp, in good condition, and not chipped (this is best done with a loupe or magnifying glass).

10. Load the G-code File

Load the NC or EGC file (the G-code which gives the machine its instructions) which you have already simulated. Follow all prompts for tool changes.

11. Monitor the Machine During Operation

Monitor the machine during operation. Ensure there is no build-up of dust, debris, or fumes, and that nothing works loose. Do not reach into the machine's working envelope or insert any object into it while the machine is operating. If you attempt to open the protective door during machining, the job will pause and the spindle will stop and park.

12. Finish and Clean Up

Once the job is complete, the spindle will automatically stop and return to its parked position. Remove the finished piece. Remove any waste and vacuum the inside of the REVO M1A.

Store end mills carefully when not in use to protect the cutting edges. Collets and end mills should be cleaned when they accumulate milling debris. The easiest way to clean end mills and collets is by dipping them into a suitable solvent such as isopropyl alcohol. Make sure to let the alcohol completely evaporate before using again.

CNC Glossary

CAD (Computer Aided Design): CAD is where you turn your idea into a digital design! CAD is software that allows you to draw on your computer. This can be anything from a full-blown parametric 3D modeling package like Solidworks, to something as simple as Adobe Illustrator. Don't let the word CAD scare you. All it means is 'a program to draw in'.

CAM (Computer Aided Manufacturing): CAM is where you turn your CAD design into something your CNC machine can understand. CAM is software that lets you specify HOW your design is going to be made on your CNC machine. Your CAM program will output G-code. Like most terminology around CNC, CAM sounds scarier than it really is.

Collet: A collet is a cone-shaped sleeve that holds an end mill in place in the spindle.

End Mill / Cutter / Tool: End mills are the cutting tools used by your M1A. End mills are similar to drill bits, though, typically they can cut in all directions. They come in several varieties, such as square, ball nose, and V-bit, and many sizes.

G-Code: G-code (general, or geometric, code) is a CNC programming language that controls when, where, and how the machine moves across the workpiece (for example, when to turn on or off, how fast to travel to a particular location, what paths to take, etc.).

HOME: Homing your machine is the process of sending it to a known, fixed, repeatable location. This means that every time you home, the machine will move to exactly the same position allowing you to move your machine to positions relative to the home position, with great precision. On your M1A, the home position is the back-left corner.

Interlock: M1A has an interlock which prevents the door from being opened while the machine is operating. When the door is opened, the current job will pause and the spindle will stop and move to a parked location.

Interlock Key: The interlock key is a small magnetic key that allows you to disengage the M1A's safety interlock. The interlock key is provided so you can set job zero with the door open. Never use the interlock key while the spindle is on.

Job Zero / Job Origin / Toolpath Zero / Program Zero: It may have many names, but job zero basically tells the machine where to begin running the job. Job zero is a point in your design where all of the toolpaths will be based from (the X, Y, and Z coordinates). Job zero is most commonly set in the lower-left corner of your stock.

Jog: Move the spindle to a specific position (a set of X, Y, Z coordinates) in the work area.

Spindle: The spindle is the part of the cnc machines that turns the end mills. The M1A spindle has a 1,5kW motor with 9– 36k RPM and angular contact bearings for greater rigidity.

Stock / Workpiece / Material: The sheet of material (wood, plastic, composite/synthetic, metal) being machined.

Table: The M1A's aluminum table holds your wasteboard and your material for machining. The table is your Y-Axis and it moves forward and backward.

Toolpath: A toolpath is the "route" the cutting tool will follow as it shapes the workpiece.

Tool Change: When running a job that requires multiple tools, Masso will prompt you each time you need to change tools. After each tool change, measure the length of the new tool.

Wasteboard: The wasteboard is a sacrificial surface that is installed to your machine's table. It allows for easy through-cutting operations (e.g. projects in which a part is cut completely from the surrounding stock) without the worry of damaging your end mill or table. Wasteboards are often customized with a grid of drilled holes which allow for workholding options such as top clamps.

Working Envelope: A working envelope is the CNC machine's range of movement across its three axes: X, Y, Z. **Workholding / Hold-Down Solution:** These are the options for securing your stock material to the machine table.