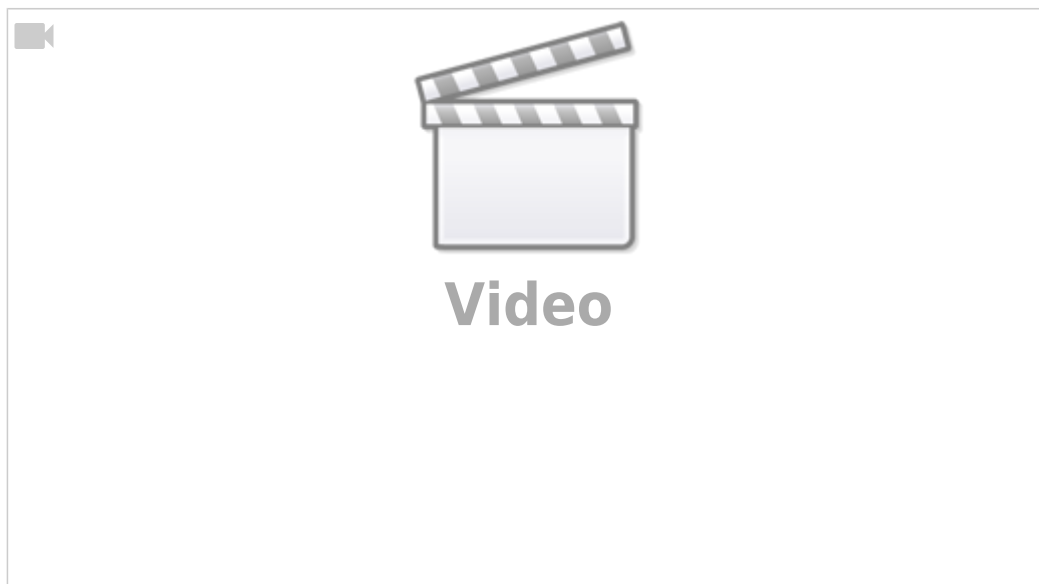


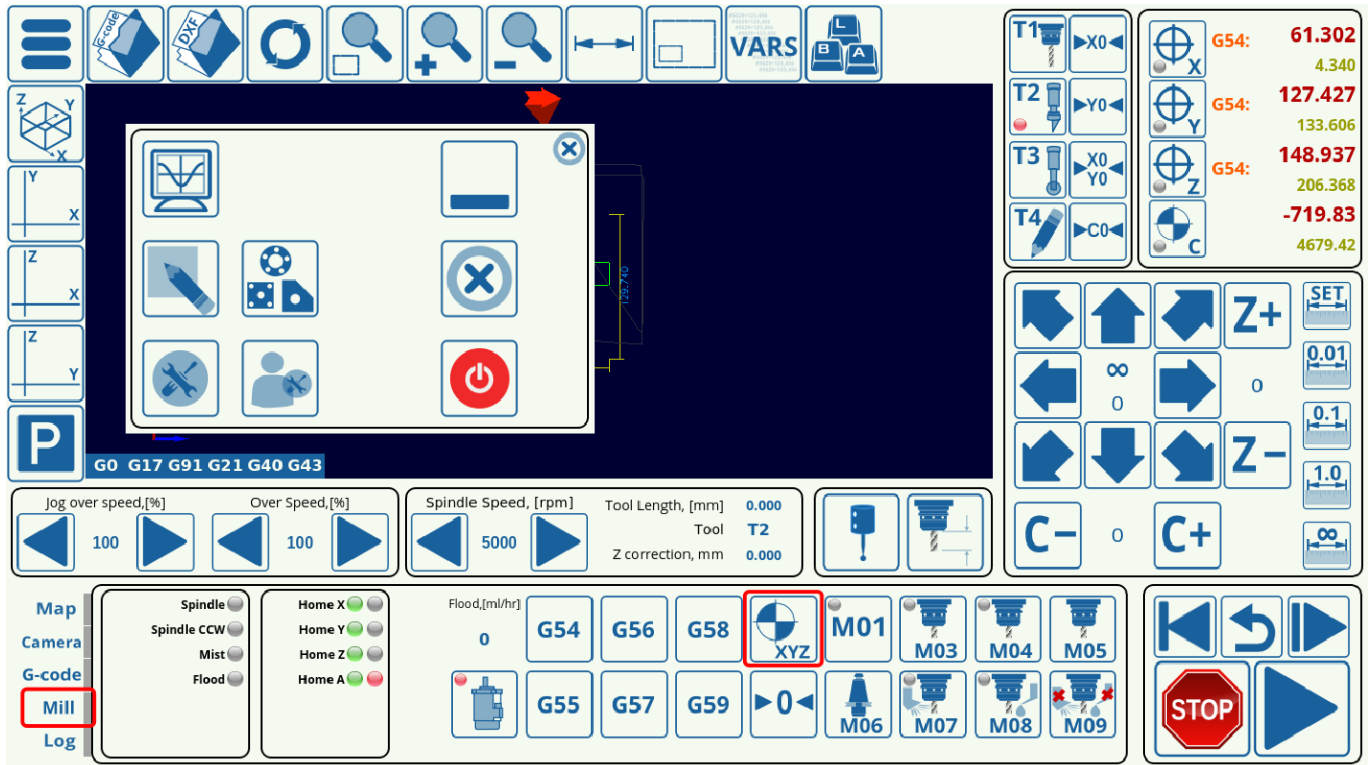
Homing Procedure Setup

This article is designed to introduce the reader to the basics of the homing procedure using myCNC software. The homing procedure is done using sensors (optical, mechanical, etc) that are set up on the edges of the machine's working surface in order to find the initial machine position (the Home position). This setup can be done for every axis that your machine is using.

A quick video recap of the full manual is presented in this video:

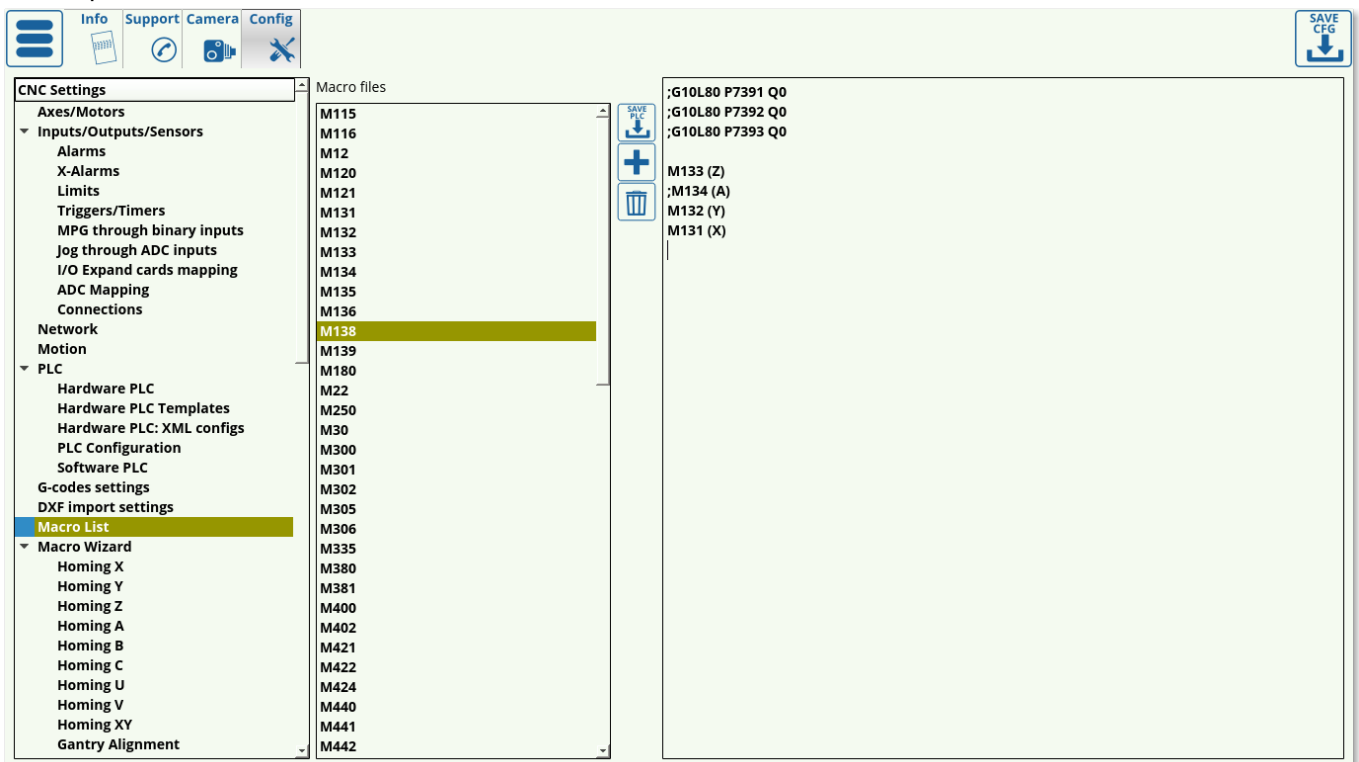


A lot of times, the homing procedure is initiated automatically upon program start, etc. To initiate the process manually, a Homing XYZ button is available in the Mill tab of the myCNC software (in X1366M, X1366M4, X1366M4E and X1366V profiles - available in the Plasma/Gas tabs in X1366P and X1366G profiles respectively):



In order to set up the homing procedure, do the following:

1. Go to **CNC Settings > Config > Macro List > M138**, and set up the homing procedure for the necessary axes by commenting out the axes and commands which are not required for your particular setup. This is done by adding ; in front of the commands, as shown in the screenshot below. For the configuration used in this example, the machine uses three axes (x, y and z), so the a-axis is not required, as well as the commands in the first half of the macro.



The code for a three-axes machine will therefore look like this:

```
;G10L80 P7391 Q0
```

```
;G10L80 P7392 Q0
;G10L80 P7393 Q0

M133 (Z)
;M134 (A)
M132 (Y)
M131 (X)
```

Note that the Z axis comes first, as the machine usually needs to lift the tool up before moving it in the xy-plane as to not crash into any obstacles.

A quick explanation of what each line of the above code represents:

- The first four lines (*G10L80 P7391 Q0*, etc) will simply set a zero for a variable that indicates the need for homing (variables #7391, etc). This will allow the user to simply forego the homing procedure by effectively indicating that the homing flag for each axis has been set to zero (so that no homing is needed).
- If the actual homing is required, the user needs to comment out the commands to set the flags to zero, and to run actual homing procedures (M133, M134, etc). Therefore, one or the other has to be commented out.
- The ; symbol comments out the line, anything in brackets such as (A) is also a comment

2. Open **CNC Settings > Config > Macro Wizard** and select the particular axis you would like to run the homing procedure for (Homing X in case of this example). The screen presented to you will look as follows:

The screenshot displays the 'Macro Wizard' configuration window for 'Homing X'. The left sidebar lists various settings categories, with 'Macro Wizard' expanded and 'Homing X' selected. The main configuration area includes the following fields:

- Homing direction:** - (dropdown)
- Sensor Number/Type:** 5 (input), Normally closed (dropdown)
- Encoder Index/Z home detection:** ☒ (checkbox)
- Index/Z Input number:** 99 (input), Normally opened (dropdown)
- Distance to Encoder Z Sensor:** (empty input field)
- Ignore Limits:** ☒ (checkbox)
- Soft stop:** ☒ (checkbox)
- Distance to Home Sensor:** 1000 (input)
- Gap:** 5 (input)
- Gap Speed:** 500 (input)
- Speed, Slow Speed:** 1000 (input), 60 (input)
- Position After Homing:** 0 (input)
- Reset Work position:** ☒ (checkbox)
- Macro filename:** M131 (dropdown), (*default is: M131)
- Macro header:** (M131 Homing X) (input)
- Macro footer:** (empty input field)

At the bottom of the main area are 'Generate' and 'Save macro' buttons. On the right, the 'Macro preview' window shows the following G-code:

```
G10 L80 P5521 Q1
G10 L80 P5525 Q1
M89 L1 P2(Quick stop when sensor triggered)
G91 G0 X -200.0000 F 600.00
G04 P0.1
M89 L0 P2(Quick stop when sensor triggered)
G91 G0 X 200.0000 F 30.00
G04 P0.1
G90 G10L70 P0 X #5451
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7391 Q0 (Homing Flag)
```

3. Select the **Homing direction** for the machine. This is selected depending on where your sensor is located in relation to the working bit of the machine. If the machine will have to move in the negative x, y or z direction to find it, select -. If the machine will have to move in the positive x, y or z direction, select +.

CNC Settings

- MPG through binary inputs
- Jog through ADC inputs
- I/O Expand cards mapping
- ADC Mapping
- Connections
- Network
- Motion
- PLC
 - Hardware PLC
 - Hardware PLC Templates
 - Hardware PLC: XML configs
 - PLC Configuration
 - Software PLC
- G-codes settings
- DXF import settings
- Macro List
- Macro Wizard
 - Homing X**
 - Homing Y
 - Homing Z
 - Homing A
 - Homing B
 - Homing C
 - Homing U
 - Homing V
 - Homing XY
 - Gantry Alignment
 - Back to Path
 - Surface Measure
 - Tool Length Measure
 - Tool Change
- Probing Wizard
- Probing Config

Homing direction -

Sensor Number/Type 5 **Normally closed**

Encoder Index/Z home detection ☒

Index/Z Input number 99 **Normally opened**

Distance to Encoder Z Sensor

Ignore Limits ☒

Soft stop ☒

Distance to Home Sensor 1000

Gap 5 Gap Speed 500

Speed, Slow Speed 1000 60

Position After Homing 0

Reset Work position ☒

Macro filename M131 (*)default is: M131

Macro header (M131 Homing X)

Macro footer

Generate **Save macro**

Macro preview

```
G10 L80 P5521 Q1
G10 L80 P5525 Q1
M89 L1 P2(Quick stop when sensor triggered)
G91 G0 X -200.0000 F 600.00
G04 P0.1
M89 L0 P2(Quick stop when sensor triggered)
G91 G0 X 200.0000 F 30.00
G04 P0.1
G90 G10L70 P0 X #5451
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7391 Q0 (Homing Flag)
```

4. Select your sensor number (depending on the input number to which you have connected the sensor) and its type (normally opened or normally closed).

CNC Settings

- Axes/Motors
- Inputs/Outputs/Sensors
 - Alarms
 - X-Alarms
 - Limits
 - Triggers/Timers
 - MPG through binary inputs
 - Jog through ADC inputs
 - I/O Expand cards mapping
 - ADC Mapping
 - Connections
- Network
- Motion
- PLC
 - Hardware PLC
 - Hardware PLC Templates
 - Hardware PLC: XML configs
 - PLC Configuration
 - Software PLC
- G-codes settings
- DXF import settings
- Macro List
- Macro Wizard
 - Homing X**
 - Homing Y
 - Homing Z
 - Homing A
 - Homing B
 - Homing C
 - Homing U
 - Homing V
 - Homing XY
 - Gantry Alignment

Homing direction -

Sensor Number/Type 5 **Normally opened**

Encoder Index/Z home detection ☒

Index/Z Input number 99 **Normally opened**

Distance to Encoder Z Sensor

Ignore Limits ☒

Soft stop ☒

Distance to Home Sensor 1000

Gap 5 Gap Speed 500

Speed, Slow Speed 1000 60

Position After Homing 0

Reset Work position ☒

Macro filename M131 (*)default is: M131

Macro header (M131 Homing X)

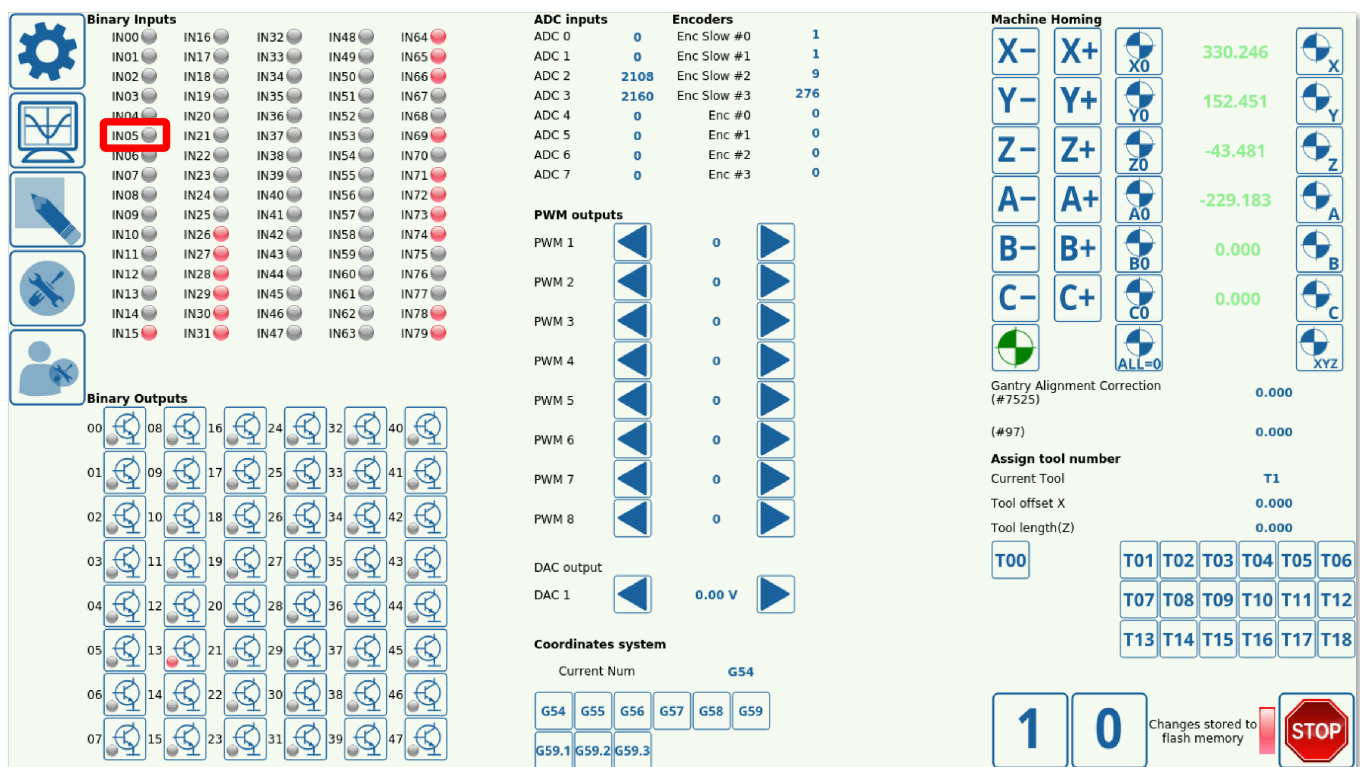
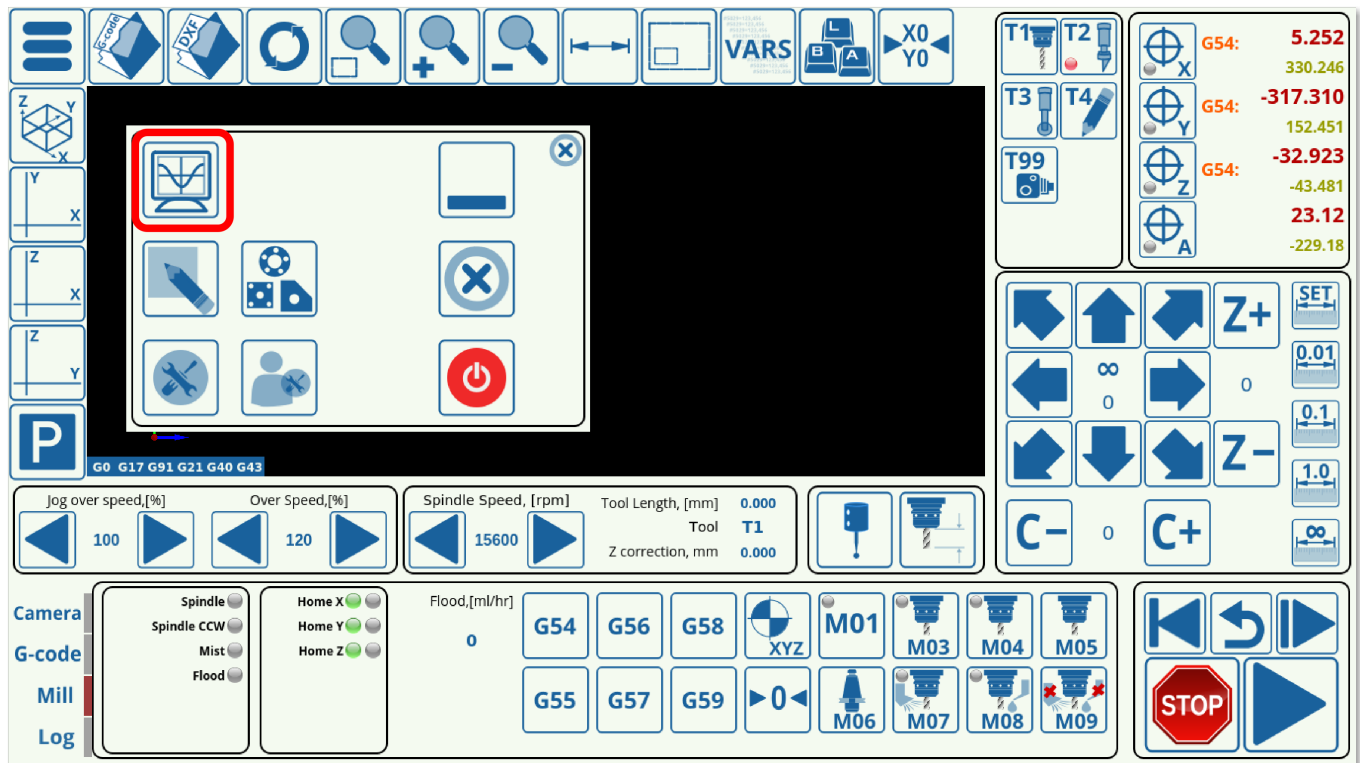
Macro footer

Generate **Save macro**

Macro preview

```
G10 L80 P5521 Q1
G10 L80 P5525 Q1
M89 L1 P2(Quick stop when sensor triggered)
G91 G0 X -200.0000 F 600.00
G04 P0.1
M89 L0 P2(Quick stop when sensor triggered)
G91 G0 X 200.0000 F 30.00
G04 P0.1
G90 G10L70 P0 X #5451
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7391 Q0 (Homing Flag)
```

Note that the normally opened or normally closed does not depend on the particular sensor or sensor type you are using, but rather on the way it has been set up in myCNC software. In case of this example, Sensor 5 is used, with it being normally opened as can be seen in the System Diagnostics window:



As can be seen from the System Diagnostics window, Sensor 5 is normally open as the indicator light is not on. However, that behaviour can be inverted in **CNC Settings > Config > Hardware > Common Hardware Settings**, so it is always recommended to check the System Diagnostics window to find out what the normal behaviour of the sensor has been set up as.

5. Choose your Limits behaviour in the **Ignore Limits** field. Set the check mark to ON if the sensor you are using for homing is the same as the one you are using for your limits (most common case), and set it to OFF if you are using multiple sensors.

The screenshot shows the 'CNC Settings' window with the 'Macro Wizard' section selected. Under 'Homing X', the 'Ignore Limits' checkbox is checked and highlighted with a red box. Other settings include: Homing direction: -, Sensor Number/Type: 5, Encoder Index/Z home detection: [X], Index/Z Input number: 99, Distance to Encoder Z Sensor: [empty], Soft stop: [checked], Distance to Home Sensor: 1000, Gap: 5, Gap Speed: 500, Speed, Slow Speed: 1000 / 60, Position After Homing: 0, Reset Work position: [checked], Macro filename: M131, Macro header: (M131 Homing X), and Macro footer: [empty]. The 'Macro preview' on the right shows a list of G-code commands.

6. The **Soft Stop** setting allows for a gradual stop after the sensor has been passed (especially useful with optical sensors). It is highly beneficial to use the soft stop setting in order to not damage the machine moving at a high homing speed that's coming to an abrupt stop due to triggering the sensor.

This screenshot is similar to the one above, but the 'Soft stop' checkbox is now checked and highlighted with a red box. The 'Ignore Limits' checkbox remains checked. All other settings and the 'Macro preview' are identical to the previous screenshot.

The Soft Stop time can be specified in **CNC Settings > Config > Preferences > Start/Stop**. Note that there must be some clearing after the sensor has been passed, as to allow for the Soft Stop to take place without crashing into an obstacle (such as a wall).

CNC Settings

- G-codes settings
- DXF import settings
- Macro List
- Macro Wizard
 - Homing X
 - Homing Y
 - Homing Z
 - Homing A
 - Homing B
 - Homing C
 - Homing U
 - Homing V
 - Homing XY
 - Gantry Alignment
 - Back to Path
 - Surface Measure
 - Tool Length Measure
 - Tool Change
- Probing Wizard
 - Probing Config
 - Probing Macro Wizard
- Preferences
 - Common
 - Start/Stop**
 - Shape Library Settings
- Screen
 - Colors
 - Popup Messages
 - 3D Visualisation
 - Work Offsets
 - Parking Coordinates
- Technology
 - Plasma Cutting

Cutting ON commands: M20; M71; M03;
 Cutting OFF commands: M21; M74; M05;
 Soft stop time,s: **0.05**
 Limit stop time,s: 0.05
 Deceleration time, If "Stop" pressed: Not defined
 After Stop Handler: Not defined
On Start
 Check Soft Limits for the full toolpath: ☒
 Toolpath Position checking, axes: ☒ x ☒ y ☒ z ☒ a ☒ b ☒ c ☒ u ☒ v
 Current position & Toolpath position should be equal in given axes
 Goto Toolpath Position for axes: ☒ x ☒ y ☒ z ☒ a ☒ b ☒ c ☒ u ☒ v
 Cutting/Spindle ON & tool moves to toolpath position on Start for given axes
 Direct Move: Direct Move
 Lift Height: 10
 Move-to-Toolpath speed: 6000
 Current position as Start ((*Disabled by default): ☒
 Reset work Position on "Reset-NC", "Tie" pressed: ☒

7. The Distance to Home Sensor should be set to be slightly higher than the maximum length that the working piece will have to travel to find the homing sensor. This depends on the particular size of your machine.

CNC Settings

- Axes/Motors
- Inputs/Outputs/Sensors
 - Alarms
 - X-Alarms
 - Limits
 - Triggers/Timers
 - MPG through binary inputs
 - Jog through ADC inputs
 - I/O Expand cards mapping
 - ADC Mapping
 - Connections
- Network
- Motion
- PLC
 - Hardware PLC
 - Hardware PLC Templates
 - Hardware PLC: XML configs
 - PLC Configuration
 - Software PLC
- G-codes settings
- DXF import settings
- Macro List
- Macro Wizard
 - Homing X**
 - Homing Y
 - Homing Z
 - Homing A
 - Homing B
 - Homing C
 - Homing U
 - Homing V
 - Homing XY
 - Gantry Alignment

Homing direction: -
 Sensor Number/Type: 5 Normally opened
 Encoder Index/Z home detection: ☒
 Index/Z Input number: 99 Normally opened
 Distance to Encoder Z Sensor:
 Ignore Limits: ☒
 Soft stop: ☒
Distance to Home Sensor: 1000
 Gap: 5 Gap Speed: 500
 Speed, Slow Speed: 1000 60
 Position After Homing: 0
 Reset Work position: ☒
 Macro filename: M131 (*default is: M131)
 Macro header: (M131 Homing X)
 Macro footer:
 Generate Save macro
Macro preview:
 G10 L80 P5521 Q1
 G10 L80 P5525 Q1
 M89 L1 P2(Quick stop when sensor triggered)
 G91 G0 X -200.0000 F 600.00
 G04 P0.1
 M89 L0 P2(Quick stop when sensor triggered)
 G91 G0 X 200.0000 F 30.00
 G04 P0.1
 G90 G10L70 P0 X #5451
 G10 L80 P5521 Q0
 G10 L80 P5525 Q0
 G10 L80 P7391 Q0 (Homing Flag)

8. It is sometimes desirable to set the homing position to be slightly further away from the edge at which the sensor is located, so that the Home position is not right at the very edge of the working space. Using the Gap setting, the user can add a gap between the sensor position and the new software Home position (5 mm in case of this example). The Gap Speed will describe the speed with which the machine moves between the sensor (located at the edge) and the gap end (located closer to the centre):

The screenshot shows the 'Homing X' configuration in the CNC Settings window. The left sidebar lists various settings categories, with 'Homing X' selected. The main configuration area includes fields for 'Homing direction' (set to '-'), 'Sensor Number/Type' (set to '5' and 'Normally opened'), 'Encoder Index/Z home detection' (checked), 'Index/Z Input number' (set to '99' and 'Normally opened'), 'Distance to Encoder Z Sensor' (empty), 'Ignore Limits' (checked), 'Soft stop' (checked), 'Distance to Home Sensor' (set to '1000'), 'Gap' (set to '5', highlighted with a red box), 'Gap Speed' (set to '500'), 'Speed, Slow Speed' (set to '1000' and '60'), 'Position After Homing' (set to '0'), 'Reset Work position' (checked), 'Macro filename' (set to 'M131'), 'Macro header' (set to '(M131 Homing X)'), and 'Macro footer' (empty). A 'Generate' button and a 'Save macro' button are at the bottom. The 'Macro preview' section on the right shows a list of G-code commands: G10 L80 P5521 Q1, G10 L80 P5525 Q1, M89 L1 P2(Quick stop when sensor triggered), G91 G0 X -200.0000 F 600.00, G04 P0.1, M89 L0 P2(Quick stop when sensor triggered), G91 G0 X 200.0000 F 30.00, G04 P0.1, G90 G10L70 P0 X #5451, G10 L80 P5521 Q0, G10 L80 P5525 Q0, and G10 L80 P7391 Q0 (Homing Flag).

9. The **Speed** field specifies the speed at which the machine is moving during its initial movement towards the sensor. After the machine triggers the sensor, it starts to move back in order to find the precise spot at which the sensor has been triggered, at the **Slow Speed**.

The screenshot shows the 'Homing X' configuration in the CNC Settings window, similar to the previous one. The 'Speed, Slow Speed' field is now highlighted with a red box, showing '1000' and '60'. The 'Gap' field is no longer highlighted. The 'Macro preview' section on the right shows the same list of G-code commands as before.

10. The **Position After Homing** field specifies the value assigned to the position after the homing. For example, if the homing for x-axis took place in the left corner of the workstation, then this x-position will be 0. If, however, the homing sensor was on the right, and the machine is 1000 mm wide, then the homing position will be specified to be 1000.

CNC Settings

- Axes/Motors
 - Inputs/Outputs/Sensors
 - Alarms
 - X-Alarms
 - Limits
 - Triggers/Timers
 - MPG through binary inputs
 - Jog through ADC inputs
 - I/O Expand cards mapping
 - ADC Mapping
 - Connections
 - Network
 - Motion
 - PLC
 - Hardware PLC
 - Hardware PLC Templates
 - Hardware PLC: XML configs
 - PLC Configuration
 - Software PLC
 - G-codes settings
 - DXF import settings
 - Macro List
 - Macro Wizard
 - Homing X**
 - Homing Y
 - Homing Z
 - Homing A
 - Homing B
 - Homing C
 - Homing U
 - Homing V
 - Homing XY
 - Gantry Alignment

Homing direction

Sensor Number/Type

Encoder Index/Z home detection ☒

Index/Z input number

Distance to Encoder Z Sensor

Ignore Limits ☒

Soft stop ☒

Distance to Home Sensor

Gap **Gap Speed**

Speed, Slow Speed

Position After Homing

Reset Work position ☒

Macro filename (*)default is: M131

Macro header

Macro footer

Generate **Save macro**

Macro preview

```

G10 L80 P5521 Q1
G10 L80 P5525 Q1
M89 L1 P2(Quick stop when sensor triggered)
G91 G0 X -200.0000 F 600.00
G04 P0.1
M89 L0 P2(Quick stop when sensor triggered)
G91 G0 X 200.0000 F 30.00
G04 P0.1
G90 G10L70 P0 X #5451
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7391 Q0 (Homing Flag)
  
```

11. Choose whether you want to reset your Work Position or not after the homing is done.

12. Choose your Macro filename (it is recommended to keep the default macro filenames unless otherwise required). The default macro filenames are as follows:

- M131 for the Homing X procedure
- M132 for the Homing Y procedure
- M133 for the Homing Z procedure
- M138 for Homing XYZ

13. Add any necessary macros to the Macro Header and Macro Footer if so desired. These will be added in the beginning or in the end of the Homing procedure respectively. THESE FIELDS ARE USUALLY LEFT BLANK.

The screenshot shows the CNC Settings application with the 'Homing X' configuration selected in the left sidebar. The main configuration area includes fields for Homing direction, Sensor Number/Type, Encoder Index/Z home detection, Index/Z input number, Distance to Encoder Z Sensor, Ignore Limits, Soft stop, Distance to Home Sensor, Gap, Gap Speed, Speed, Slow Speed, Position After Homing, Reset Work position, Macro filename, Macro header, and Macro footer. The 'Macro header' field is highlighted with a red box and contains the text '(M131 Homing X)'. The 'Macro footer' field is empty. The 'Macro preview' section on the right shows a list of G-code macros including G10, M89, G91, G04, M89, G90, G10, and G10.

Any macro in brackets such as this:

(M131 Homing X)

is "commented out", disabling its addition to the Homing procedure.
Add the necessary macros without brackets, separated by a semicolon such as this:

M5;M9

14. Press **Save Configuration**, then press **Generate** and **Save macro**.

CNC Settings

- Axes/Motors
 - Inputs/Outputs/Sensors
 - Alarms
 - X-Alarms
 - Limits
 - Triggers/Timers
 - MPG through binary inputs
 - Jog through ADC inputs
 - I/O Expand cards mapping
 - ADC Mapping
 - Connections
 - Network
 - Motion
 - PLC
 - Hardware PLC
 - Hardware PLC Templates
 - Hardware PLC: XML configs
 - PLC Configuration
 - Software PLC
 - G-codes settings
 - DXF import settings
 - Macro List
 - Macro Wizard
 - Homing X**
 - Homing Y
 - Homing Z
 - Homing A
 - Homing B
 - Homing C
 - Homing U
 - Homing V
 - Homing XY
 - Gantry Alignment

Homing direction: -

Sensor Number/Type: 5 Normally opened

Encoder Index/Z home detection: ☐ (Red box)

Index/Z Input number: 99 Normally opened

Distance to Encoder Z Sensor:

Ignore Limits: ☒

Soft stop: ☒

Distance to Home Sensor: 1000

Gap: 5 Gap Speed: 500

Speed, Slow Speed: 1000 60

Position After Homing: 0

Reset Work position: ☒

Macro filename: M131 (*default is: M131)

Macro header: (M131 Homing X)

Macro footer:

Generate (Red box) **Save macro** (Red box)

Macro preview

```
G10 L80 P5521 Q1
G10 L80 P5525 Q1
M89 L1 P2(Quick stop when sensor triggered)
G91 G0 X -200.0000 F 600.00
G04 P0.1
M89 L0 P2(Quick stop when sensor triggered)
G91 G0 X 200.0000 F 30.00
G04 P0.1
G90 G10L70 P0 X #5451
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7391 Q0 (Homing Flag)
```

SAVE CFG (Red box)

Using Incremental Encoders

These settings allow for a higher accuracy during the homing procedure, however they require using an incremental encoder which might not always be feasible. These settings are located on the same screen, as shown below:

CNC Settings

- Axes/Motors
 - Inputs/Outputs/Sensors
 - Alarms
 - X-Alarms
 - Limits
 - Triggers/Timers
 - MPG through binary inputs
 - Jog through ADC inputs
 - I/O Expand cards mapping
 - ADC Mapping
 - Connections
 - Network
 - Motion
 - PLC
 - Hardware PLC
 - Hardware PLC Templates
 - Hardware PLC: XML configs
 - PLC Configuration
 - Software PLC
 - G-codes settings
 - DXF import settings
 - Macro List
 - Macro Wizard
 - Homing X**
 - Homing Y
 - Homing Z
 - Homing A
 - Homing B
 - Homing C
 - Homing U
 - Homing V
 - Homing XY
 - Gantry Alignment

Homing direction: -

Sensor Number/Type: 5 Normally opened

Encoder Index/Z home detection: ☐ (Red box)

Index/Z Input number: 99 Normally opened (Red box)

Distance to Encoder Z Sensor:

Ignore Limits: ☒

Soft stop: ☒

Distance to Home Sensor: 1000

Gap: 5 Gap Speed: 500

Speed, Slow Speed: 1000 60

Position After Homing: 0

Reset Work position: ☒

Macro filename: M131 (*default is: M131)

Macro header: (M131 Homing X)

Macro footer:

Generate **Save macro**

Macro preview

```
G10 L80 P5521 Q1
G10 L80 P5525 Q1
M89 L1 P2(Quick stop when sensor triggered)
G91 G0 X -200.0000 F 600.00
G04 P0.1
M89 L0 P2(Quick stop when sensor triggered)
G91 G0 X 200.0000 F 30.00
G04 P0.1
G90 G10L70 P0 X #5451
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7391 Q0 (Homing Flag)
```

SAVE CFG

1. Set the checkmark to ON for the **Encoder Index/Z Home Detection** in order to use this setting.
2. As with the sensor from Step 4 of the regular procedure, set your sensor number and its normal

behaviour.

3. Set the **Distance to Encoder Z Sensor** (usually used on Linear Encoders). This is useful when a linear encoder is located some distance away from the sensor, and is set to be slightly smaller (on the scale of nearly a millimeter) than the actual distance to the encoder. This setting is not necessary, however it does allow the user to lower the required time that the machine spends on the Encoder Z Home Detection.

Checking sensor contact at homing start

In certain situations, homing may start while the homing sensor is already “engaged” (i.e. the sensor is in a state that indicates that contact has been made already). This may be due to a faulty sensor/setup, or it may be due to the fact that the sensor is touching the machine. In that case, it is not desirable to continue moving the sensor any further, as this may physically break it.

To prevent this, a Hardware PLC can be used. An example of such a PLC is M286, which is generated automatically for tasks such as Surface Measure. For the Surface Measure procedure, the macro code that includes M286 will look the following way:

```
M286 P[35] L[256+2] (lift up sensor activated, sensor normally opened)
```

or

```
M286 P[256+35] L[256+2] (lift up sensor activated, sensor normally closed)
```

Note that the 35 part of the code refers to port 35 - this value must be customized depending on the input port used.

The code for M286 Hardware PLC itself is shown below:

```
//////// FILE GENERATED AUTOMATICALLY. DO NOT EDIT IT. //////////
#define input  var00
#define state  var01

main()
{
    input=eparam & 0xFF;           //P-parameter lo byte
    state=(eparam>>8) & 0xff;      //P-parameter high byte
    axis= (eparam>>16) & 0xff;      //L-parameter lo byte
    dir=   (eparam>>24) & 0xff;      //L-parameter high byte

    axis_mask=0;
    if (axis<6) {axis_mask=1<<axis;}
    else { exit(99); };

    length=100; //1mm
    if (dir==0) { length=0-length; };

    gvarset(7080,1000); //set speed
```

```
do {  
  a=portget(input);  
  if (state==0) {    if (a==0)  { exit(99); }; };  
  if (state!=0) {    if (a!=0)  { exit(99); }; };  
  
  g0moveA(0, axis_mask, length);           //incremental programming;  
  timer=200; do{timer--;} while (timer>0); //wait motion started  
  //wait motion stopped  
  do { code=gvarget(6060); } while (code!=0x4d);  
  }while (1);  
  
  exit(99);  
};
```

This PLC will check if the input port is “triggered”, and will move the machine in 1mm increments until the the conditions are met.

If the port is in its “correct” state, the PLC will exit immediately.

From:

<http://cnc42.com/> - myCNC Online Documentation

Permanent link:

http://cnc42.com/quickstart/mycnc-quick-start/homing_setup

Last update: **2022/10/21 13:54**

