Starting with Mach 4

and ICNC 2.X

Before you begin, it's crucial to note that any Mach 4 build prior to version 5103 is not compatible with Soprolec Products, and ICNC2 cards need firmware version V5.38 CNC or later.

Mach4Hobby V5570 :
https://www.machsupport.com/ftp/Mach4/DevlopmentVersions/Mach4Hobby_Installer-4.2.0.5570.exe

Mach4Industrial V5570 :
https://www.machsupport.com/ftp/Mach4/DevlopmentVersions/Mach4Industrial_Installer-4.2.0.5570.exe

SOPROLEC   Digital control, **Automation** and Axis control

Table of contents

I – Plugin Setup

1 – Copy the following files:

ICNC2Plugin4.m4pw, ICNC2_VS.dll, and ICNC2Plugin4.sig to the Mach 4 installation directory. Typically, Mach 4 is installed on the local disk C in a folder named 'Mach4Hobby' or 'Mach4Industrial.' You'll then paste these files into the 'plugins' folder within the Mach 4 directory.

Disque local (C:) > Mach4Hobby

Nom	Modifié le	Type	Taille
Docs	14/03/2024 10:18	Dossier de fichiers	
GcodeFiles	14/03/2024 10:18	Dossier de fichiers	
Lang	14/03/2024 10:18	Dossier de fichiers	
Licenses	14/03/2024 10:18	Dossier de fichiers	
LuaExamples	14/03/2024 10:18	Dossier de fichiers	
Modules	14/03/2024 10:18	Dossier de fichiers	
Plugins	14/03/2024 10:24	Dossier de fichiers	
Pmc	14/03/2024 10:18	Dossier de fichiers	
Profiles	14/03/2024 10:18	Dossier de fichiers	
Screens	14/03/2024 10:18	Dossier de fichiers	
Subroutines	14/03/2024 10:18	Dossier de fichiers	
Tables	14/03/2024 10:18	Dossier de fichiers	
TraceIntermediary	14/03/2024 10:18	Dossier de fichiers	
Wizards	14/03/2024 10:18	Dossier de fichiers	
ZeroBraneStudio	14/03/2024 10:18	Dossier de fichiers	
concrt140.dll	13/02/2019 03:15	Extension de l'app...	244 Ko
CoreConf.dll	26/02/2024 21:33	Extension de l'app...	6 172 Ko
gcedit.exe	08/11/2018 01:43	Application	8 030 Ko

Digital control, **Automation** and Axis control

2 – Uncheck the simulator device :

Navigate to Menu -> Configure -> Select Motion Dev, then deselect the simulation device to prepare for adding our motion controller in the next step.
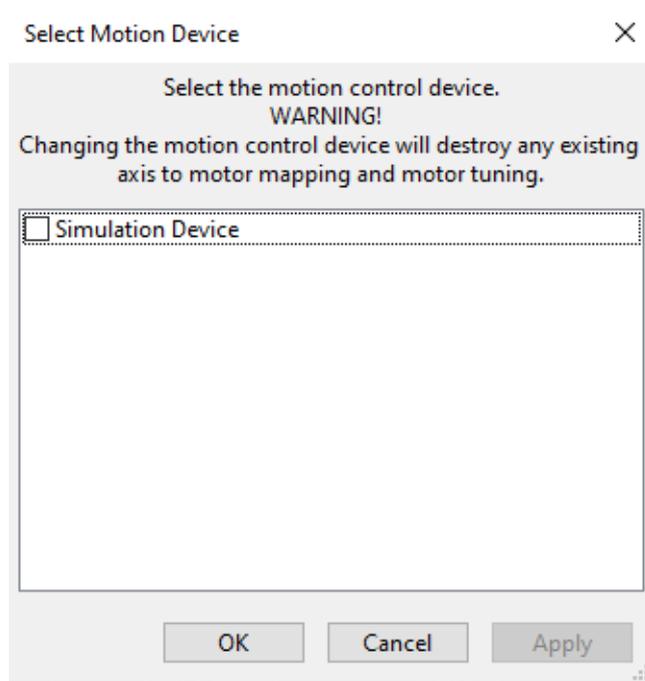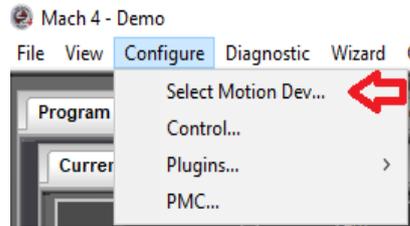
3 – Enable the Soprolec ICNC2 Plugin:

   - Navigate to Menu -> Configure -> Control -> Plugins tab.

   - Ensure that the "Soprolec – ICNC2" plugin is enabled with a green check-mark. If not, click on it to enable it. Remember, you'll need to restart Mach4 if you've made changes to enable it.

   - Additionally, enable the following plugins:

         - Keyboard Inputs (allows keyboard jogging)

         - Core – Newfangled Solutions
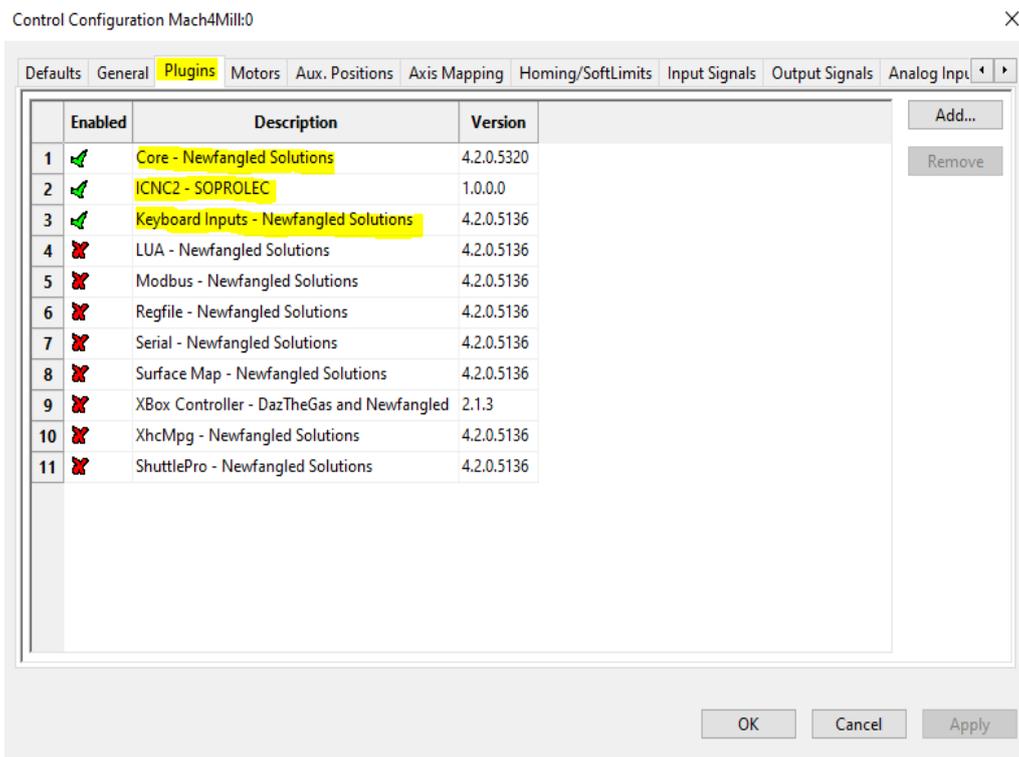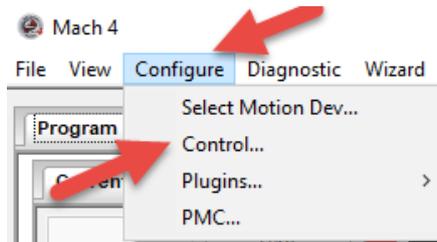
4 – Selecting the motion controller:

After restarting Mach 4, follow these steps:
- Go to Menu -> Configure -> Select Motion Dev.
- Check ICNC2-SOPROLEC from the list of available motion controllers.
- Press "OK" to confirm your selection.

Select Motion Device ✕

Select the motion control device.
WARNING!
Changing the motion control device will destroy any existing axis to motor mapping and motor tuning.

☑ ICNC2 - SOPROLEC

OK   Cancel   Apply
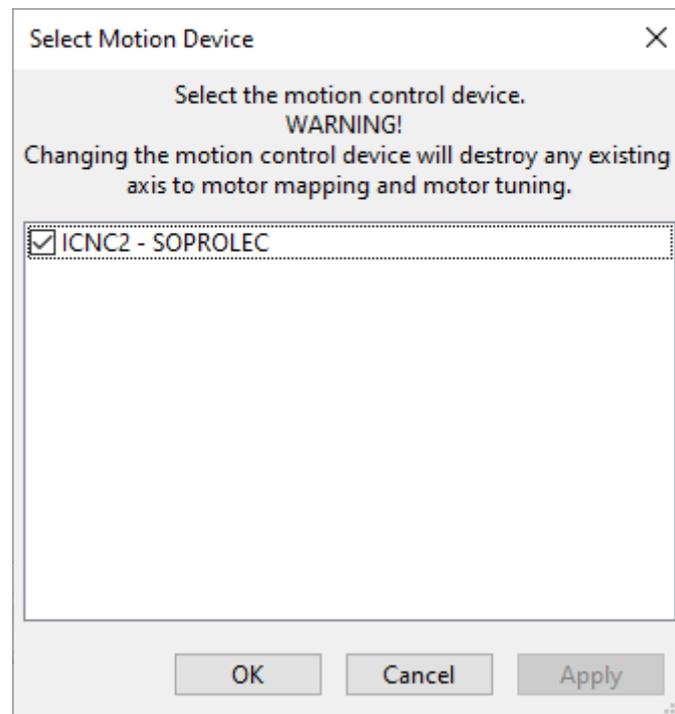
After this step, Mach 4 will be communicating with your ICNC2.X. You can verify this by checking the history button located at the lower left corner of the screen to visualize the received messages.

II – Configuration Example

The following steps outline the configuration process for your milling machine: we will configure our machine as a 3-axis milling machine with 3 homing sensors (NC), one for each axis. This machine does not have max limit sensors, so we will configure software soft limits and manage the emergency stop .

1 – Defaults:

As you can observe, the units have been configured in the metric system (mm).
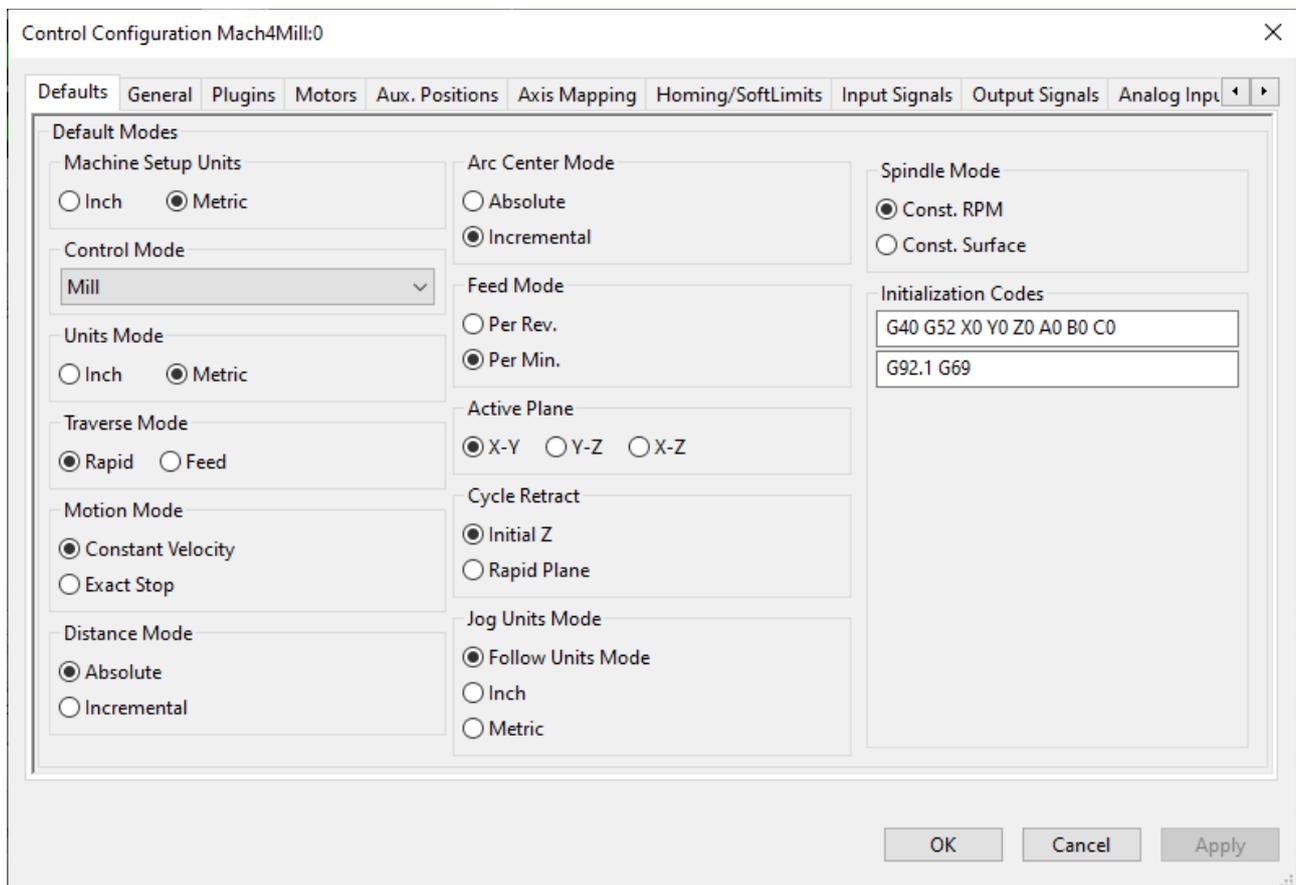
2 – Motors:

   In this section, we will configure the motors. It's crucial to differentiate each motor from its corresponding axis. For instance, while a motor can only be configured for one axis, an axis can accommodate multiple motors.

   First, we need to identify and check the first three motors since we are configuring a 3-axis machine. Then, for each motor, proceed to configuration. Begin by setting the counts per unit. Since we've configured the system in the metric system, this corresponds to pulses per mm. In my case, it's 320 because my drivers are set to 1600 pulses per rotation, and the screw has a pitch of 5mm per step, resulting in 1600/5 = 320 pulses per mm. Next, specify the velocity and acceleration parameters. Repeat this process for all three axes.

Digital control, **Automation** and Axis control
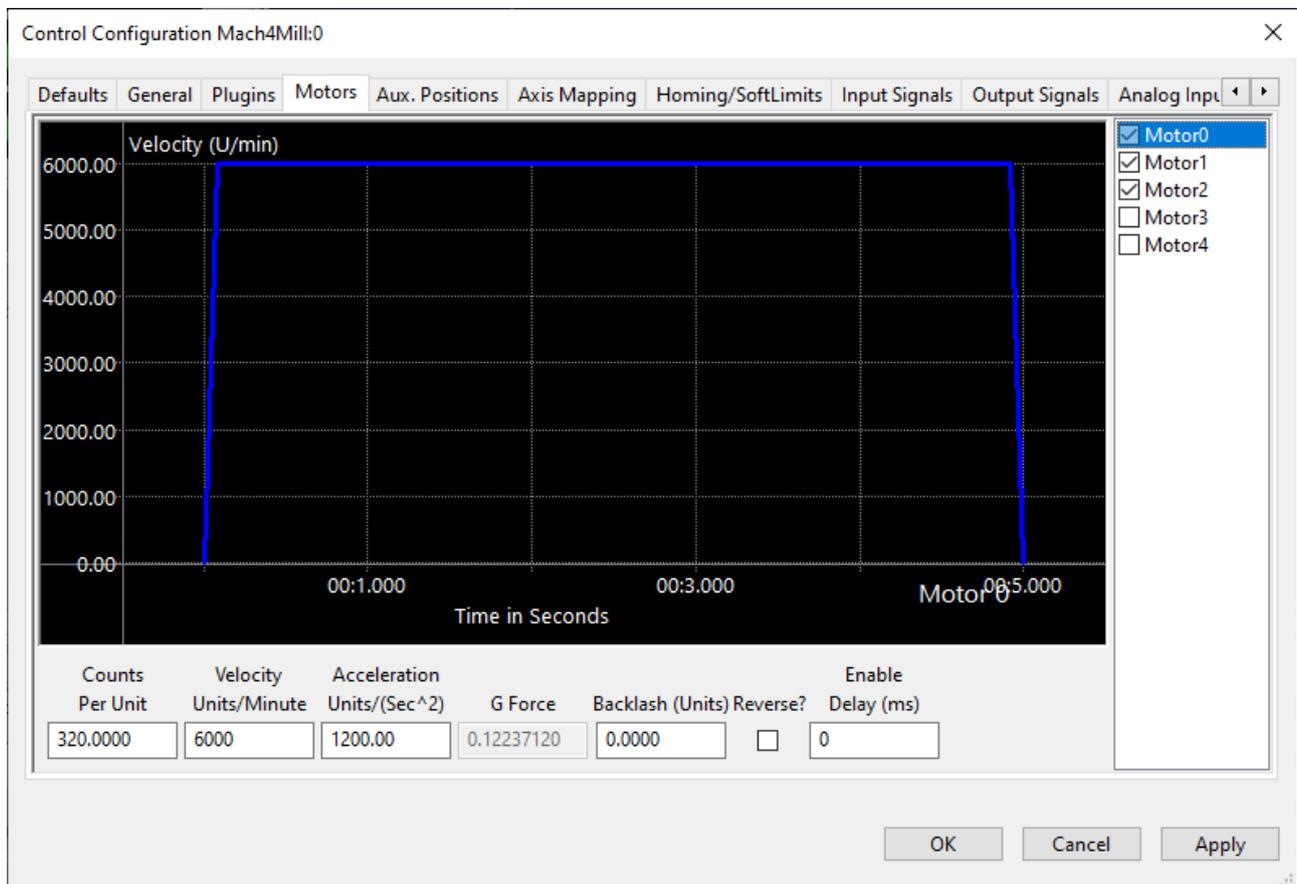
3 – Axis Mapping:

In this section, we will enable the axes and assign motors to them. (Don't forget, after completing this step, return to the Motors tab and reverse the motor assignment for the Z-axis if applicable.)

Control Configuration Mach4Mill:0 ✕

	Enabled	Master	Slave 1	Slave 2	Slave 3	Slave 4	Slave 5
X (0)	✔	Motor0					
Y (1)	✔	Motor1					
Z (2)	✔	Motor2					
A (3)	✗						
B (4)	✗						
C (5)	✗						
OB1 (6)	✗						
OB2 (7)	✗						
OB3 (8)	✗						
OB4 (9)	✗						
OB5 (10)	✗						
OB6 (11)	✗						

Tabs: Defaults | General | Plugins | Motors | Aux. Positions | Axis Mapping | Homing/SoftLimits | Input Signals | Output Signals | Analog Inpu ◄ ►

OK   Cancel   Apply

Digital control, **Automation** and Axis control

4 – Homing/Soft Limits :

On this tab, it is essential to adjust your machine dimensions under the parameter soft limits, both plus and minus. Additionally, you can customize the homing direction for each axis and arrange the order of homing for your axes.

Control Configuration Mach4Mill:0 ✕

Defaults | General | Plugins | Motors | Aux. Positions | Axis Mapping | Homing/SoftLimits | Input Signals | Output Signals | Analog Inp ◄ ►

	Home Dir	Home Order	Home Offset	Home Speed%	Home In Place	Soft Enable	Soft Min	Soft Max	Ref On Start
X (0)	Neg	2	0.0000	40.00	✗	✔	0.0000	580.0000	✔
Y (1)	Neg	2	0.0000	40.00	✗	✔	0.0000	320.0000	✔
Z (2)	Pos	1	0.0000	20.00	✗	✔	-95.0000	0.0000	✔
A (3)	Neg	2	0.0000	40.00	✗	✗	0.0000	0.0000	✗
B (4)	Pos	3	0.0000	20.00	✗	✗	0.0000	0.0000	✗
C (5)	Pos	0	0.0000	20.00	✗	✗	0.0000	0.0000	✗
OB1 (6)	Pos	0	0.0000	20.00	✗	✗	0.0000	0.0000	✗
OB2 (7)	Pos	0	0.0000	20.00	✗	✗	0.0000	0.0000	✗
OB3 (8)	Pos	0	0.0000	20.00	✗	✗	0.0000	0.0000	✗
OB4 (9)	Pos	0	0.0000	20.00	✗	✗	0.0000	0.0000	✗
OB5 (10)	Pos	0	0.0000	20.00	✗	✗	0.0000	0.0000	✗
OB6 (11)	Pos	0	0.0000	20.00	✗	✗	0.0000	0.0000	✗

OK | Cancel | Apply

5 – Input Signals :

Among the crucial inputs in our setup are the three homing sensors and our emergency stop (e-stop) input. The e-stop input is a simulated input generated by our plugin, mirroring the Enable state of your ICNC2.X. This input operates as an active high input.

Control Configuration Mach4Mill:0	Mapping Enabled	Device	Input Name	Active Low	Log Enabled	User Description
Input #63	✗			✗	✓	
Motor 0 Home	✓	ICNC2_0	Input1	✓	✓	
Motor 1 Home	✓	ICNC2_0	Input2	✓	✓	
Motor 2 Home	✓	ICNC2_0	Input3	✓	✓	
Motor 3 Home	✗			✗	✓	
Motor 4 Home	✗			✗	✓	
Motor 5 Home	✗			✗	✓	
Motor 6 Home	✗			✗	✓	
Motor 7 Home	✗			✗	✓	
Motor 8 Home	✗			✗	✓	
Motor 9 Home	✗			✗	✓	
Motor 10 Home	✗			✗	✓	
Motor 11 Home	✗			✗	✓	
Motor 12 Home	✗			✗	✓	
Motor 13 Home	✗			✗	✓	

Tabs: Defaults | General | Plugins | Motors | Aux. Positions | Axis Mapping | Homing/SoftLimits | Input Signals | Output Signals | Analog Input

OK   Cancel   Apply

6 – Output Signals :

In our example, we will configure one enable output for our drivers and another for our spindle.

7 – Analog Inputs :

« Numerator » is the maximum voltage value admissible by the input.
« Denominator » is the resolution of the used Digital to Analog converter (1024 stands for 10 bits).

Control Configuration Mach4Mill:0

General | Plugins | Motors | Aux. Positions | Axis Mapping | Homing/SoftLimits | Input Signals | Output Signals | Analog Inputs | Analo

	Device	Analog Input Name	Numerator	Denominator	Offset	User Description
Analog Input #0			0.000000	0.000000	0.000000	
Analog Input #1	ICNC2_0	AIN2	10	1024		
Analog Input #2	ICNC2_0	AIN3	10	1024		
Analog Input #3	ICNC2_0	AIN4	10	1024		
Analog Input #4						
Analog Input #5						
Analog Input #6						
Analog Input #7						
Analog Input #8						
Analog Input #9						
Analog Input #10						
Analog Input #11						
Analog Input #12						
Analog Input #13						
Analog Input #14						
Analog Input #15						
Analog Input #16						
Analog Input #17						

OK | Cancel | Apply

8 – Spindle Speed Control:

To control the spindle speed via an analog output of the **ICNC2.X** board, follow these steps:

1. **Configuring Speed Values**

   - Open the configuration window and go to the **Spindle** tab.

   - Adjust the **minRPM** and **maxRPM** values on line 0.

   - The analog output will provide a maximum voltage of **10V** when the spindle reaches **maxRPM**.

2. **Configuring Outputs**

   - In the **Outputs** window, the **Spindle** output will be used for turning the spindle **on/off**.

   - The **Analog** output will be used for speed control.

3. **Selecting the Analog Output**

   - An additional parameter must be configured in the **Plug-in** settings.

   - Select the appropriate analog output.

   - Refer to **Section III** of the Plug-in settings for more details.

III – Plugin and ICNC2 parameters

1 – Milling:

If you wish to use Mach4 to command a milling machine (without a plasma torch), the only parameters that matter for you are:

Probe Acceleration: Acceleration while performing a probe.

Probe Deceleration: This value must be high so the axis stops immediately when the input changes state.

Start/Stop Frequency: The default value is 100 Hz.

Buffer Size: This parameter is crucial as it defines the buffer size of your ICNC2. To find the perfect value for your setup, start with the lowest value (20 ms). After saving the parameter by clicking on the write button, jog one of your axes at high speed. If the jogging is smooth without any blockage in the middle of the movement, you have chosen the perfect value. If not, try a larger value. Note: if the parameter is too large, it will reduce your machine's reactivity during jogging movements.

Soft Limits: When this parameter is on, your machine will respect the programmed machine bounds at every start (you will not be able to move beyond the programmed limits).

Disable When Disconnected: If this parameter is on and your ICNC2 USB connection gets disconnected, it will automatically disable Mach4. We recommend enabling this function.

Spindle Speed: This parameter is by default OFF. It is only useful if your spindle speed is commanded by an analog output on your ICNC2. If this is the case, you will need to choose an analog output. Then, the actual spindle speed will be scaled between 0 and 10 V based on your maximum programmed RPM in the Mach4 configuration.

THC: This parameter should always be OFF if your machine is not a plasma cutting table.

2 – Plasma:

If you wish to use Mach4 to operate a plasma cutting machine, all the parameters mentioned above are useful, along with the THC parameters:

 THC Ain Number: This is the number of the THC analog input. It needs to be between 1 and 4. If this input is not valid, the THC will not start.

THC Activation Delay: Delay that occurs after the activation of the THC via the macro script. If you use auto-calculate THC Value, then the value of this parameter needs to be 0.

THC Proportional Gain: This parameter determines the reactivity of THC movements.

15

THC Time to Calculate Target: This value is only useful when the auto-calculate THC Value is ON. It determines the time of inactivity of the THC at the beginning of the cutting, then calculates the best THC target for your material based on the cutting height. The calculated value can be visualized in the THC Diagnostics tab under Target voltage.

THC Real Voltage For 0V: The real voltage of the generator when the analog input is at 0V.

THC Real Voltage For 10V: The real voltage of the generator when the analog input is at 10V.

Arc OK Mach #Input Number: The Mach4 digital input number for your generator's digital output arc OK.

THC: This parameter must be enabled for plasma cutting.

Auto Calc THC Value: When enabled, this parameter automatically calculates your THC cutting voltage based on your cutting height. If disabled, you will need to manually input this value as explained below in the V-Plasma cutting parameters.

Using Arc OK: This parameter allows you to choose whether to wait for the Arc OK signal from your generator or to use a delay after turning on your torch.

High-Speed Clearance Height: The clearance height used when executing the M6 command after plasma cutting.

AIN THC Adjust Target / THC Target AIN Adjust / AIN N°:

This setting, expressed as a percentage, works together with **THC Target AIN Adjust** and the selected **AIN°**.

- If **THC Target AIN Adjust** is enabled, this percentage value determines the maximum fine adjustment applied through an analog input.
- Example: If the fine adjustment is assigned to **AIN2**:
    - When **AIN2 = 5V (on a 0-10V scale)** → No adjustment (0%).
    - When **AIN2 = 0V (on a 0-10V scale)** → The target voltage will be reduced by **X%**.
    - When **AIN2 = 10V (on a 0-10V scale)** → The target voltage will be increased by **X %**.

Where **X** is the percentage value set in this parameter.

IV−Plasma procedure and THC activation via G-code

There are multiple methods for executing plasma cutting with Torch Height Control (THC).

First Method: Using M47

One approach is to use the **M47** command. If you choose this method, the probing parameters will be those displayed in **Mach4**. This command runs a macro script that considers the defined parameters (probe offset, delay time, heights, etc.) to automatically generate the corresponding G-code instructions. If **THC** is enabled in the plugin settings, this command will also activate it.

A **post-processor for SheetCam** is available to generate G-code compatible with this mode.

Second Method: G-Code with Embedded Probing Parameters

Another approach is available where the **G-code itself contains all the probing parameters**. A separate **post-processor for SheetCam** is also available to generate G-code compatible with this method.

Important THC Settings

For proper **THC operation**, the following parameters must be correctly set in this mode:

- **Cut height**
- **Maximum height**
- **Minimum height**

In this mode, **THC is activated using the M48 command**, and it automatically deactivates when the plasma torch turns off.

⚠ **Very Important**: In both modes, it is **essential to disable the machine's soft limits** to ensure proper probing functionality.

V – Plasma cutting parameters
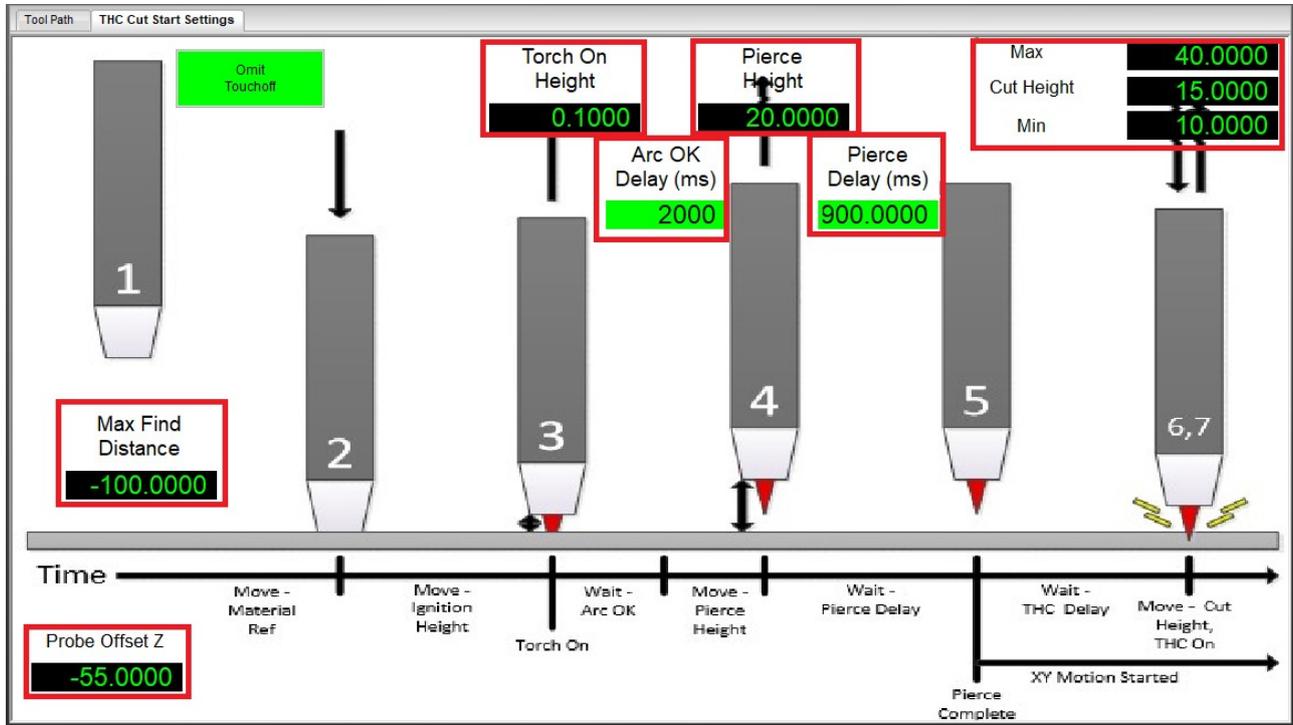
This diagram represents the plasma cutting procedure, outlining the necessary steps before starting a plasma cut. All parameters highlighted in red are crucial for this operation.

Procedure explanation:

1. The torch will first probe to find the sheet's position.
2. Once probing ends, it will adjust according to the offset position to account for deceleration delay.
3. The torch will then move to the height required to turn it on.
4. After activation, it will wait for an "arc OK" signal (if the "Using arc OK" parameter is enabled in the plugin settings).
5. Following this, the torch will reposition to the pierce height and wait for a given delay.
6. Finally, it will move to the cutting height and start the cut. on the THC

Max Find Distance: The maximum distance the probe can travel. If this distance is reached and the probe input does not detect the torch, a probe error will be generated, stopping the probing process.

Probe Offset Z: The movement after probing to compensate for the deceleration delay, relative to the probe position.

Torch On Height: The height at which the torch will be turned on, relative to the height after the probe offset.

Digital control, **Automation** and Axis control

**Arc OK Delay:** The delay, in milliseconds, after receiving the "arc OK" signal.

Pierce Height: The height at which the material will be pierced, relative to the height after the probe offset.

Pierce Delay: The delay, in milliseconds, for piercing the material.

Cut Height: The height at which the material will be cut, relative to the height after the probe offset.

Max: The maximum height for cutting the material with the THC, relative to the height after the probe offset.

Min: The minimum height for cutting the material with the THC, relative to the height after the probe offset.
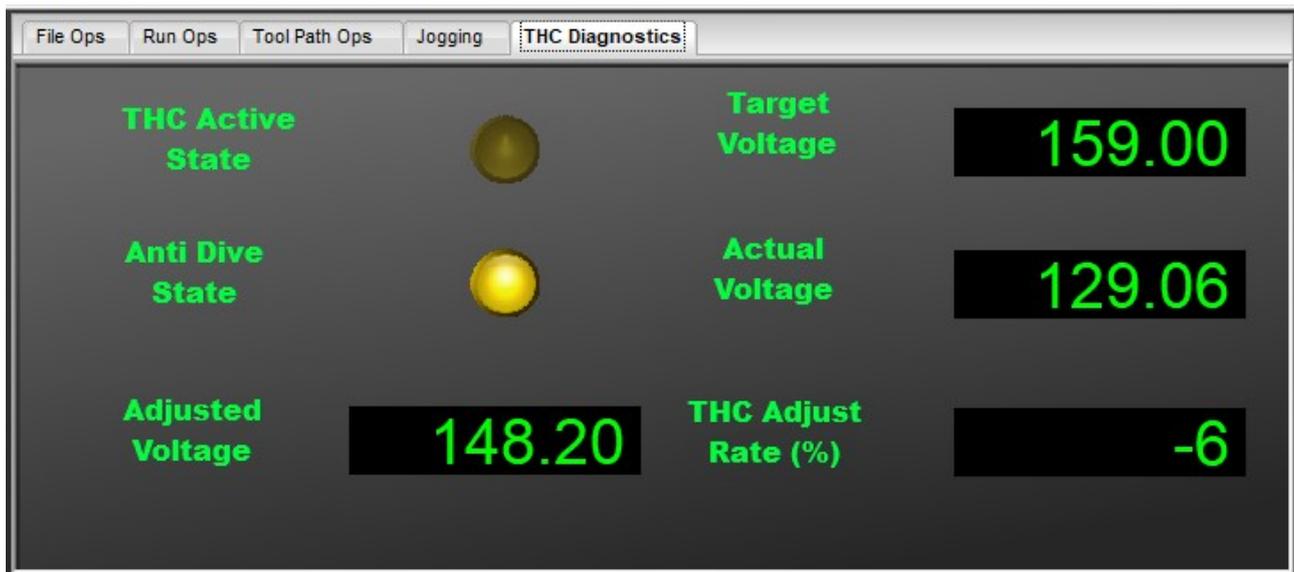
Target Voltage: This is the THC (Torch Height Control) target voltage. If the "Auto Calculate THC Target" parameter is enabled, this target voltage will be calculated at the beginning of the cut based on the cutting height. If this parameter is disabled, you will need to input the target voltage manually.

Actual Voltage: This is the actual voltage measured on the analog input, scaled appropriately.

THC Adjust Rate: This refers to the real-time percentage, adjusted via the analog command, for fine-tuning the THC control. Please refer to the plugin settings for more details.

THC Active State : THC activated or deactivated.

Adjusted Voltage : The target voltage adjusted using the THC Adjust Rate (AIN).

VI – Plasma Complete profile Soprolec

To simplify your machine configuration, you can download the complete plasma profile from our website. The package includes a plasma screen and the full configuration for the machine. Within the ZIP file, you will find folders that correspond to those in the Mach4 installation. To set it up, copy each file from the respective folder in the ZIP file and paste it into the corresponding folder in your Mach4 installation directory. Start by copying the folder **Plasma_Soprolec** into the **Profiles** folder, then copy the **Plasma_Soprolec.set** file into the **Screens** folder.

VII – Restriction of our plugin

1 – Probe :

      -Only the input probe, designated as G31, should be used with our plugin. Probe Inputs labeled as probe 1, 2, or 3 are not compatible. Specifically, using G31,1 or G31,2 will yield the same result as G31 alone, focusing solely on the input probe.
      -The G-code line featuring G31 for probing can only specify one axis at a time. If multiple axes are mentioned, only the axis associated with the lowest motor number will be probed.
      -Probing is only effective on axes that are not linked to slave axes. If a linked axis is probed, it will result in an error message, and the probing action will not take effect.

2 – Homing :

      -If your homing input is not assigned, the homing process will not take effect. If you initiate homing for an axis that has a slave axis, both homings will be launched simultaneously.