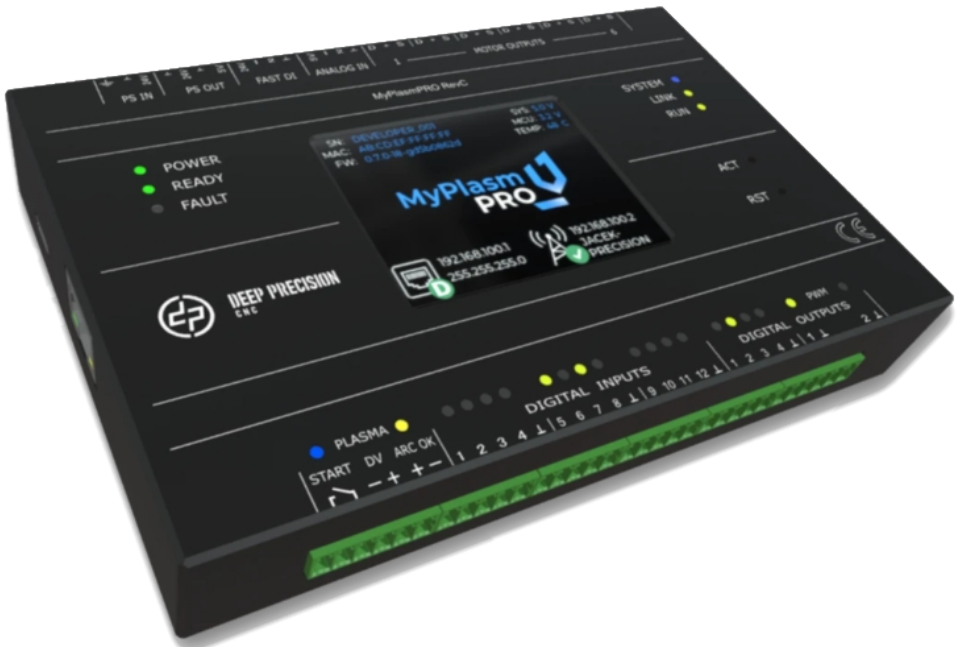
DEEP PRECISION
CNC

MyPlasm PRO 1.0



User Manual

Professional Plasma CNC Controller

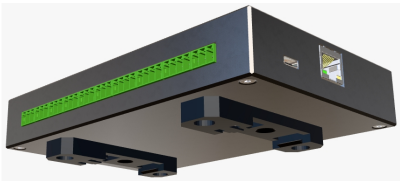
Table of Contents

1. Device Overview

MyPlasm PRO Controller is a professional CNC controller primarily designed for plasma cutting systems. Owing to its flexible control software architecture, it can also be adapted for other CNC applications to meet specific user requirements.
This manual applies to **software/firmware version 1.0 and default/basic Layout**.

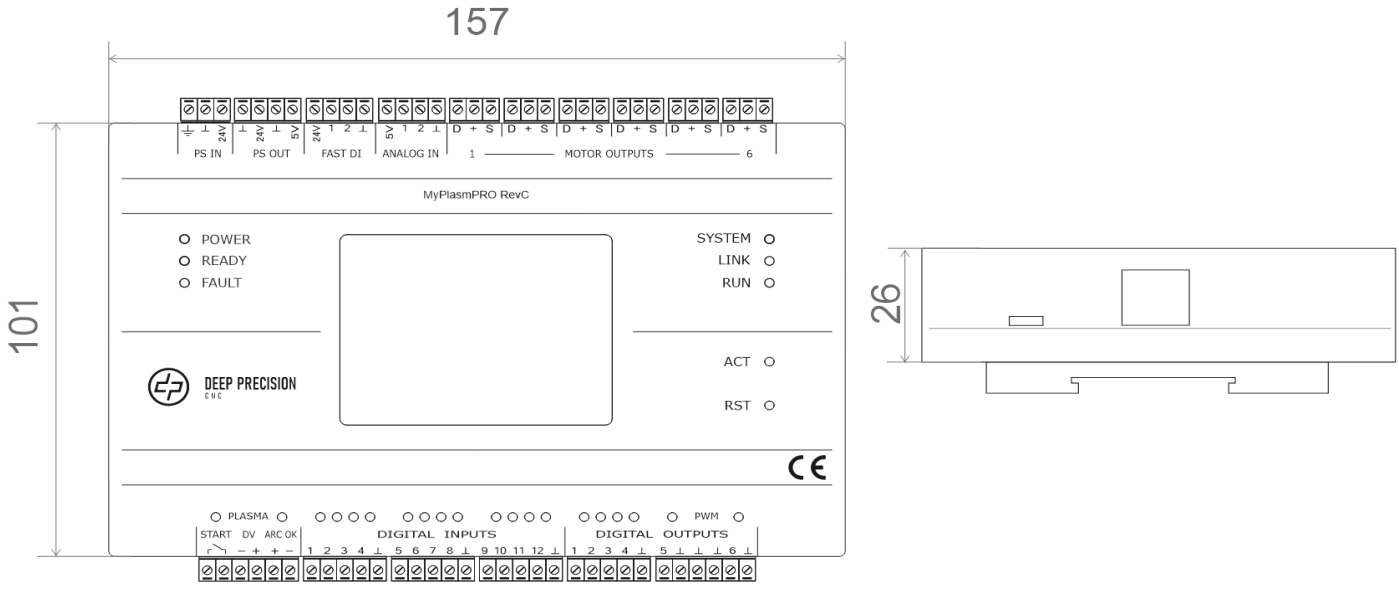
Due to the harsh operating conditions of CNC plasma cutting systems, including high levels of electromagnetic and electrical interference, the controller is built using high-grade components and enclosed in a robust aluminum housing that provides effective EMI shielding. Noise - resistant Ethernet communication and an integrated arc voltage measurement system with a built-in THC controller significantly simplify deployment in plasma cutting applications.

DIN-35 rail mounting brackets enable quick and convenient installation inside a control cabinet.



2. Technical Data

- Power supply : 24VDC / 300mA
- dimensions of Aluminium Cover : 157 x 101 x 26 [mm]
- Number of Axes : 6
- Signals for Serovos / Steppers : TTL PUL/DIR
- Step Freqency max : 2000kHz (2MHz)
- Communication interface : ETHERNET
- Expand interface : RS485
- Service interface : USB C
- Digital Inputs : 13
- Analog inputs (Differential) : 2 (1)
- Digital Outputs (PWM) : 6 (2)
- RelayOutput : 1
- Plasma Interface (THC) : Built-in
- Plasma Voltage (divided only) : 0-12V DC



3. PC/Laptop System and Hardware Requirements

- System Windows 10 , Windows 11
- Processor: Dual-core CPU, ≥ 2.0 GHz (Intel Core i3 / AMD equivalent or better)
- Memory: 8 GB RAM (4 GB minimum)
- Storage: 2 GB free disk space (SSD recommended)
- Graphics: Integrated GPU supporting DirectX 11 or higher
- Network: Ethernet port (10/100/1000 Mbps)
- Ports: USB port for serivce options
- Display: Minimum resolution* 1366 × 900 (1920 × 1080 recommended)

*The existing layout can be adjusted or a new layout can be created to support any screen resolution.

4. Software

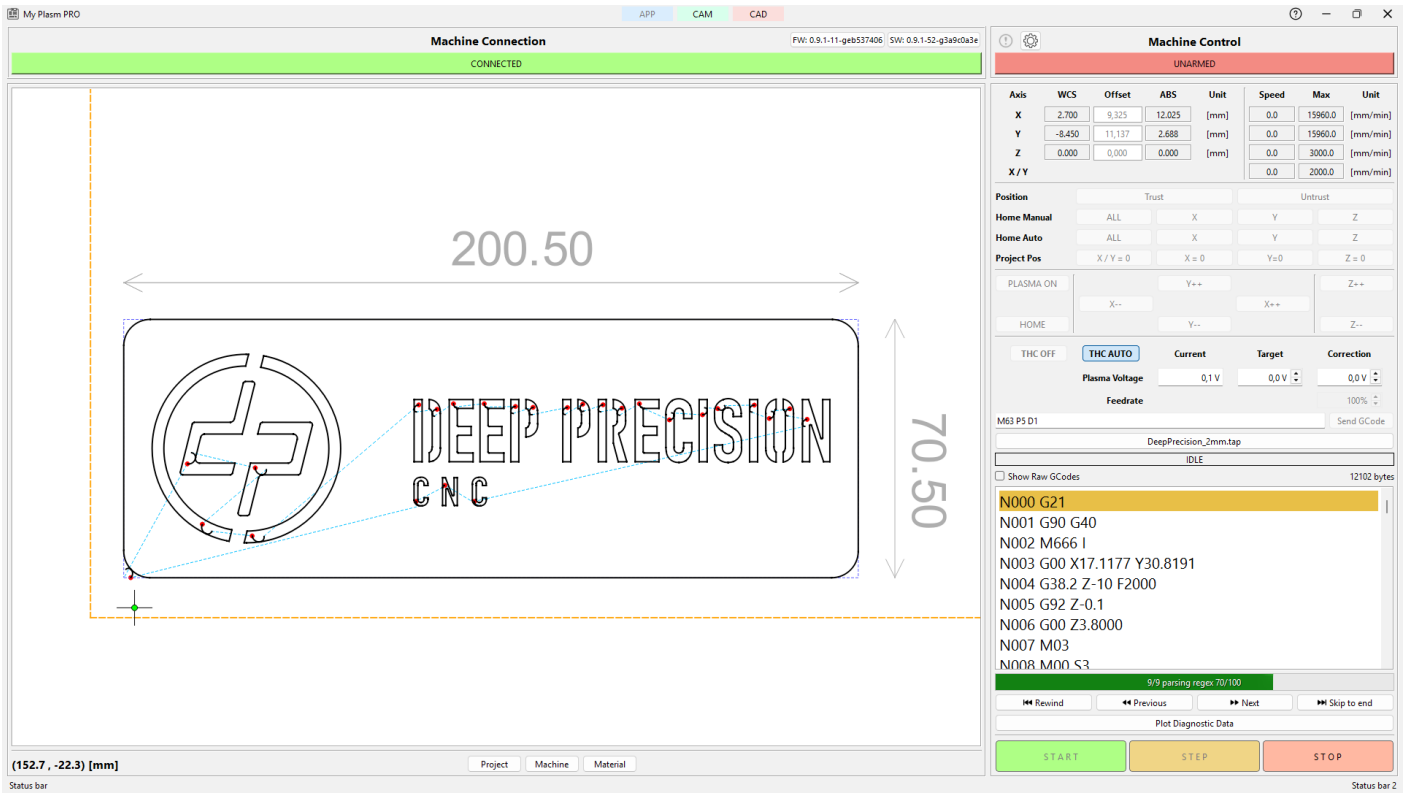
The controller operates exclusively with the free **MyPlasm CNC Pro** software, available for download from dpcnc.pro ,

The software suite consists of three modules:

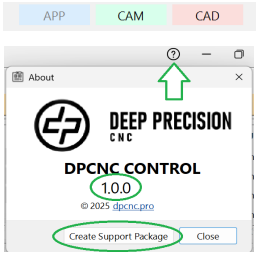
1. **CONTROL APP** – machine control module
2. **CAM module** – toolpath generation
3. **CAD module** – design and drawing environment

4.1 Controll APP

The control software is used for communication with the controller and for machine operation and control.



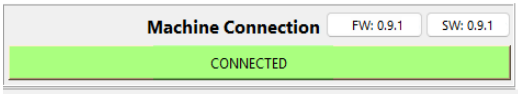
Tittle / TopBar :

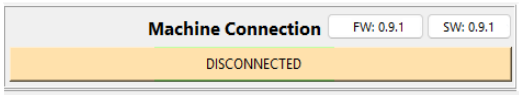


Quick Module Switching Buttons (**APP / CAM / CAD**)

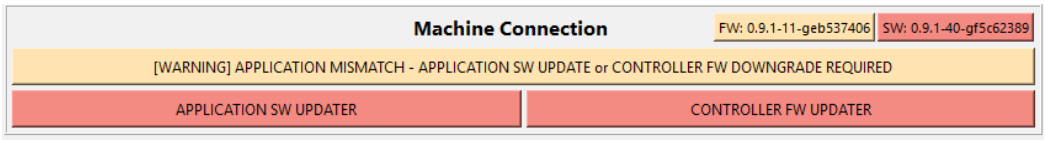
About – Provides application version information and allows generation of a diagnostic ZIP package for technical support purposes.

Machine Connection status:





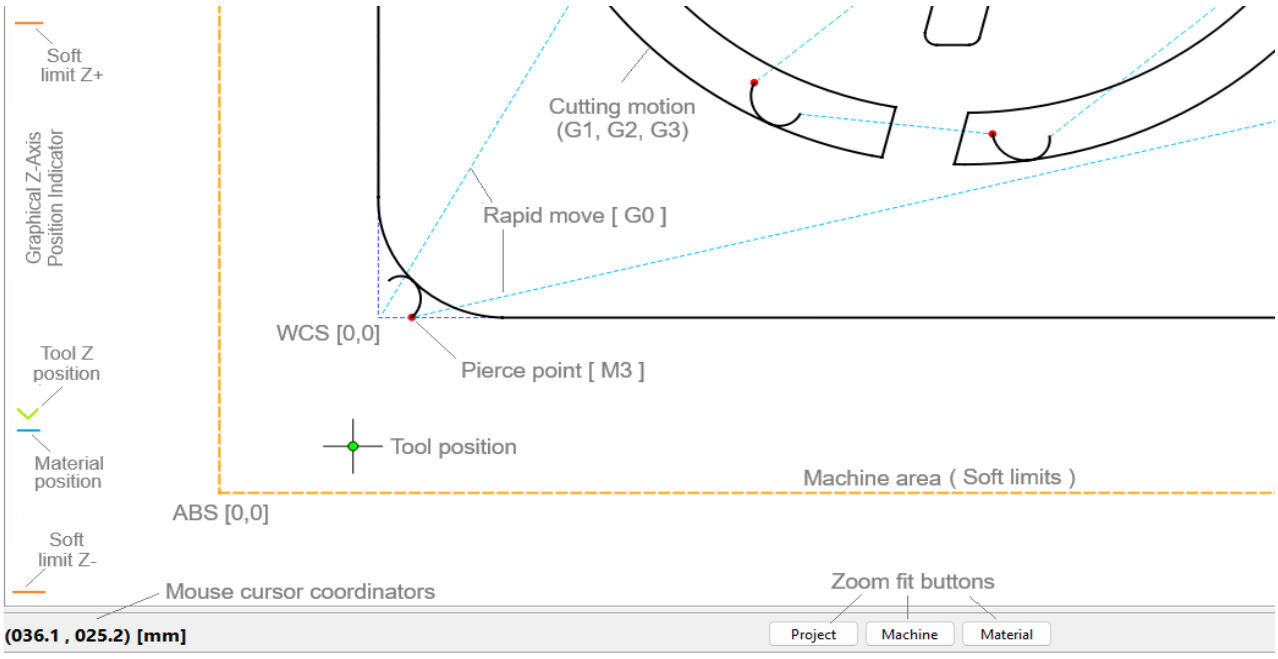
An interactive controller connection status indicator and software/firmware version display.
The information elements are clickable and provide access to advanced functions described later in this manual.



Main scene

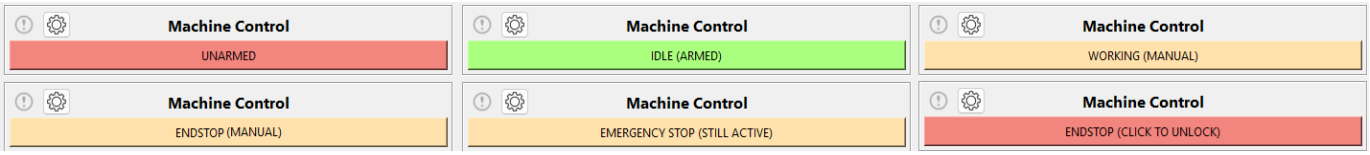
On the main scene, the following are displayed:

- The project, including cutting paths, rapid move paths, pierce points
- Machine area [soft limits]
- Tool position



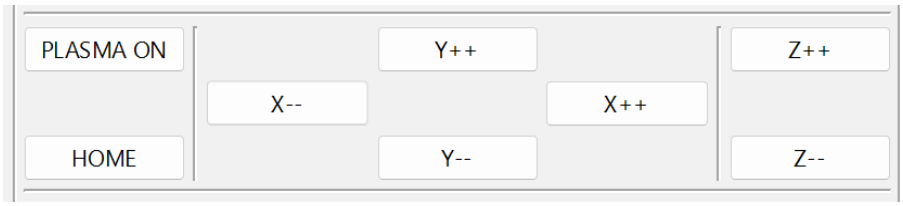
Machine controll

Controller Status :



An interactive controller status indicator showing whether the controller is disarmed (click it to ARM) and displaying current controller states such as Idle, Manual, Auto, Emergency Stop etc..
It also allows machine unlocking after a limit switch is triggered, enabling manual axis movement.

Manual Move (JOG)



Manual axis control buttons. The JOG feed rate is configured in the settings under **Jog Feedrate**. Holding the **SHIFT** key forces maximum speed, while holding **CTRL** enables slow, precise movement.

Default JOG ShourtCut :

Left (X--) : [A], [Left]
Right (X++) : [D], [Right]
Backward (Y--) : [S], [Down]
Forward (X++) : [W], [Up]
Up (Z--) : [Q], [PgDown]
Up (Z++) : [E], [PgUp]

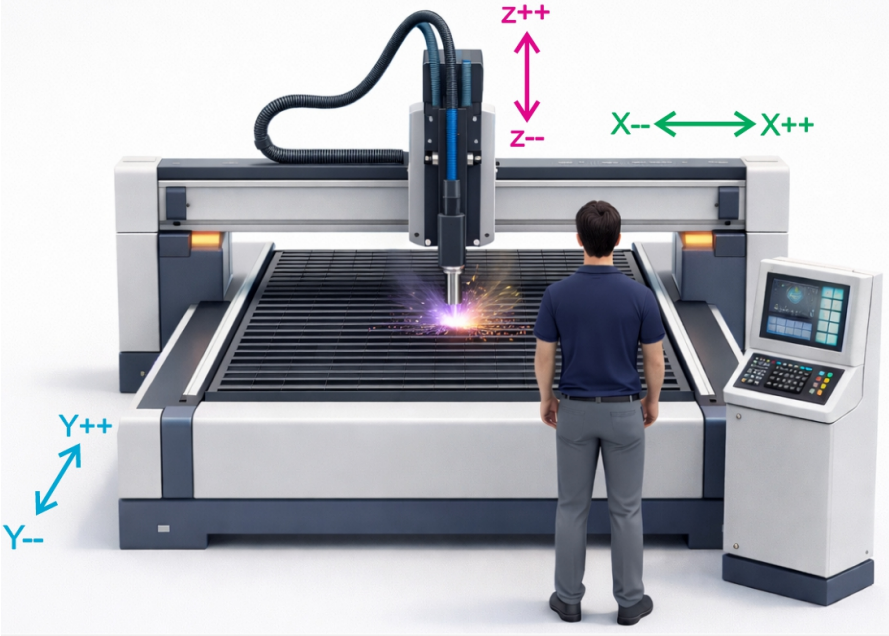
Keyboard shortcuts can be modified in the app/**keyboard_shortcuts.json** file.

GameController

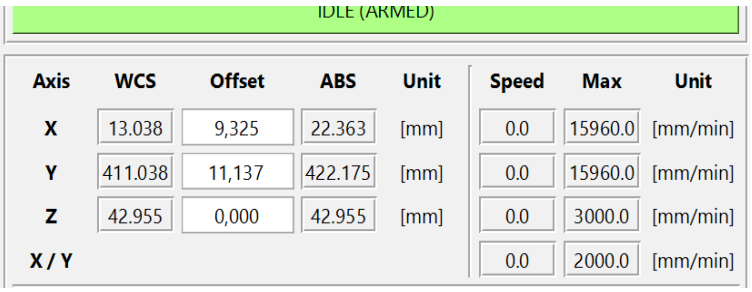
JOG functions can also be operated using game controllers (wired or wireless) compatible with the Windows operating system. If the controller is correctly recognized by Windows, it will work automatically after the software is launched.

Controller support will be extended in future software versions.
In version 1.0, buttons mapping and assignment of functions other than JOG are not available.

Coordinators System

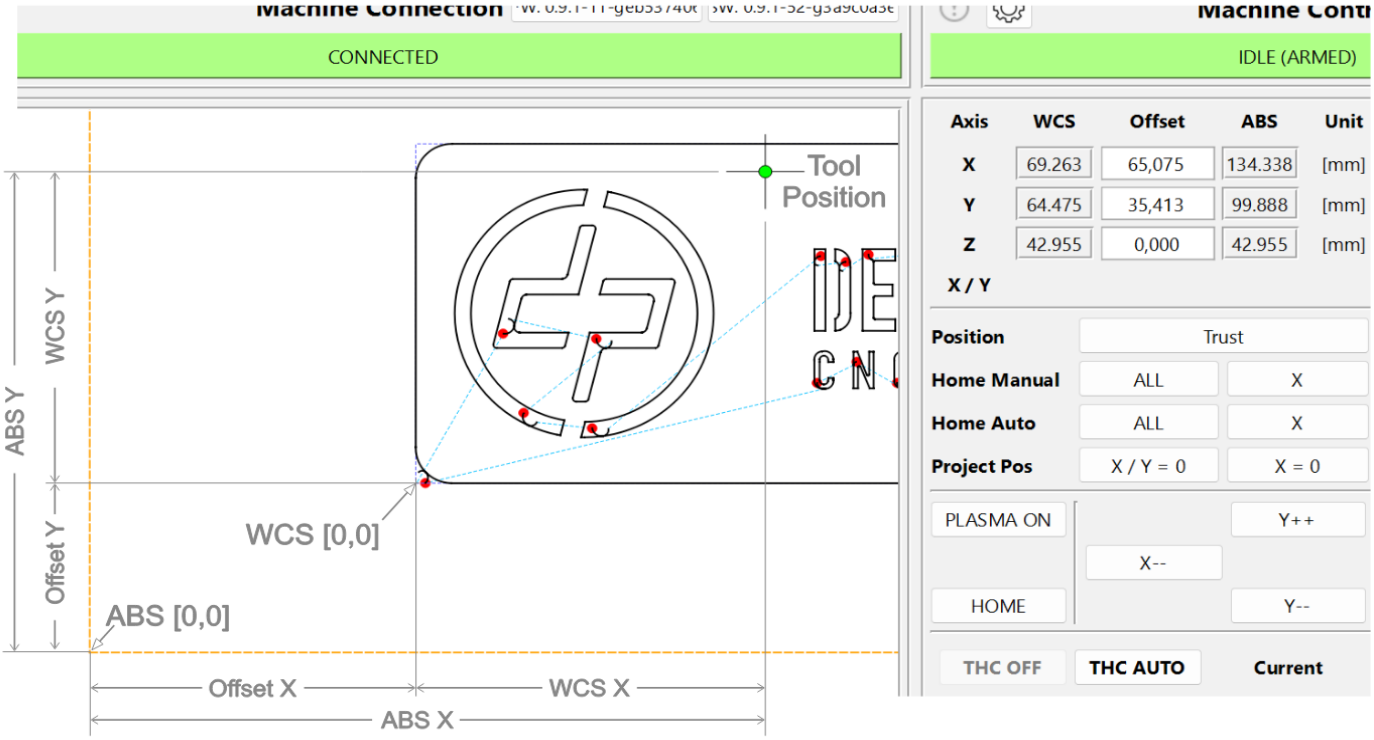


Displays the current tool position relative to the project (**WCS – Work Coordinate System**) and to the machine (**ABS – Absolute Position**). The **Offset** defines the shift of the work coordinate system relative to the absolute machine coordinates (project position on the machine).



Axis speeds are displayed in real time for each axis, along with the current XY feed rate.

EXAMPLE :



Homing / Machine Reference Points

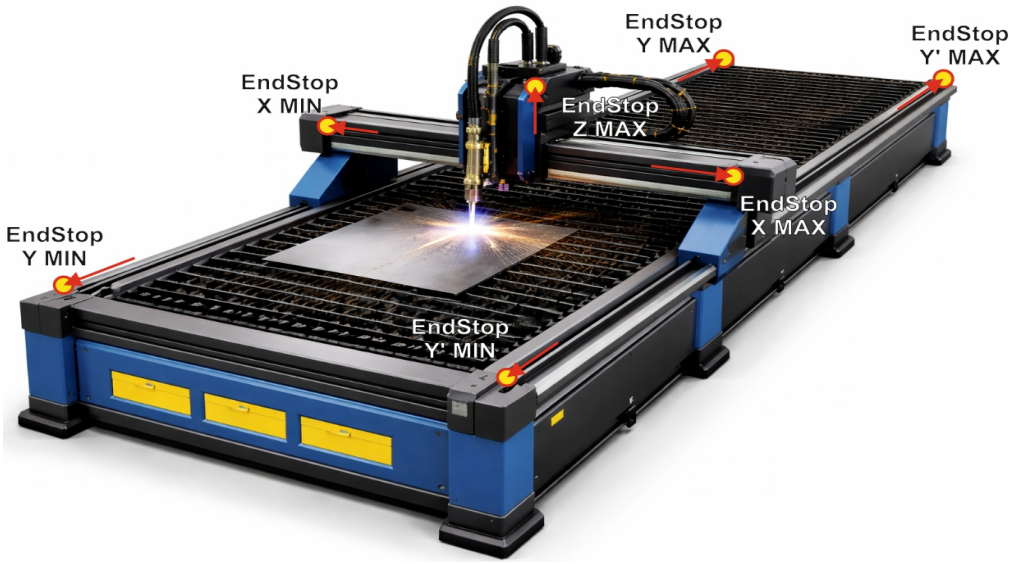
To allow the controller to determine the exact tool position on the machine, reference points must be defined and a homing sequence must be performed after each machine power-up.

If homing is not executed, the controller cannot verify whether the physical machine position matches the internal coordinates. In this case, the position is marked as **UNTRUST** and displayed in red on the screen.

Axis	WCS	Offset	ABS	Unit
X	-7.050	65,075	58.025	[mm]
Y	-8.263	35,413	27.150	[mm]
Z	0.000	19,510	19.510	[mm]
X / Y				

This does not prevent normal operation; however, software limits are not enforced. It is strongly recommended to perform homing (reference move) as the first action after powering on the machine.

Automatic homing uses limit switches, which can be installed at either end of an axis and configured accordingly by setting the axis homing direction.

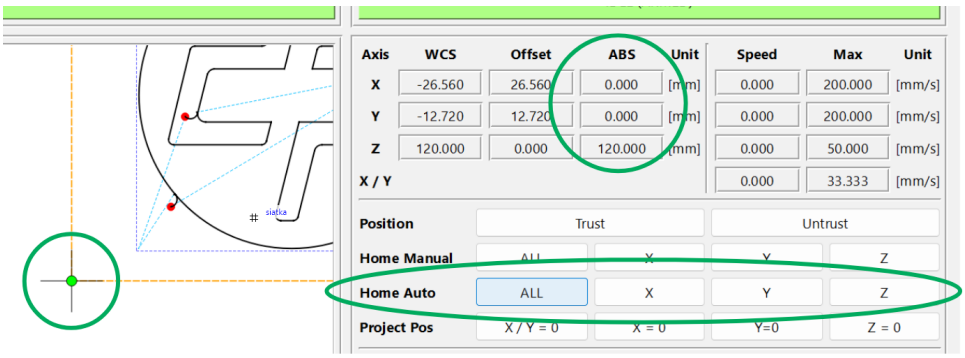


Below is an example configuration using a single limit switch per axis. The X and Y axes are configured as **MIN endstops** (lower-left corner of the machine), while the Z axis is configured as a **MAX endstop** (upper position). With this configuration, activating **Automatic Homing** moves the Z axis upward to its maximum position and assigns **ABS Z MAX**. The X axis moves left and the Y axis moves toward the operator, zeroing the **ABS X** and **ABS Y** coordinates.

Example SETTINGS:

Stepper Out	Axies	Invert	Min Endstop	Max Endstop	Home Dir
M1	X	☐	DI1	-	MIN
M2	Y	☐	DI2	-	MIN
M3	Y	☑	DI3	-	MIN
M4	Z	☐	-	DI4	MAX

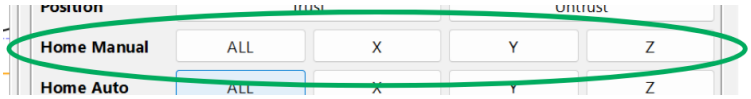
POS. AFTER AUTO HOME (REFERNECE)



Note: In addition to the homing (reference move) function, endstop sensors also operate as limit switches, triggering an emergency stop.

Manual Homing

If manual homing is required (without EndStop sensors), reference points can be set manually. In this case, according to the configured **HomeDir** (MIN or MAX), each axis must be moved to its end position using **JOG**, and then the assigned **Manual Reference (Home Manual)** button must be pressed.



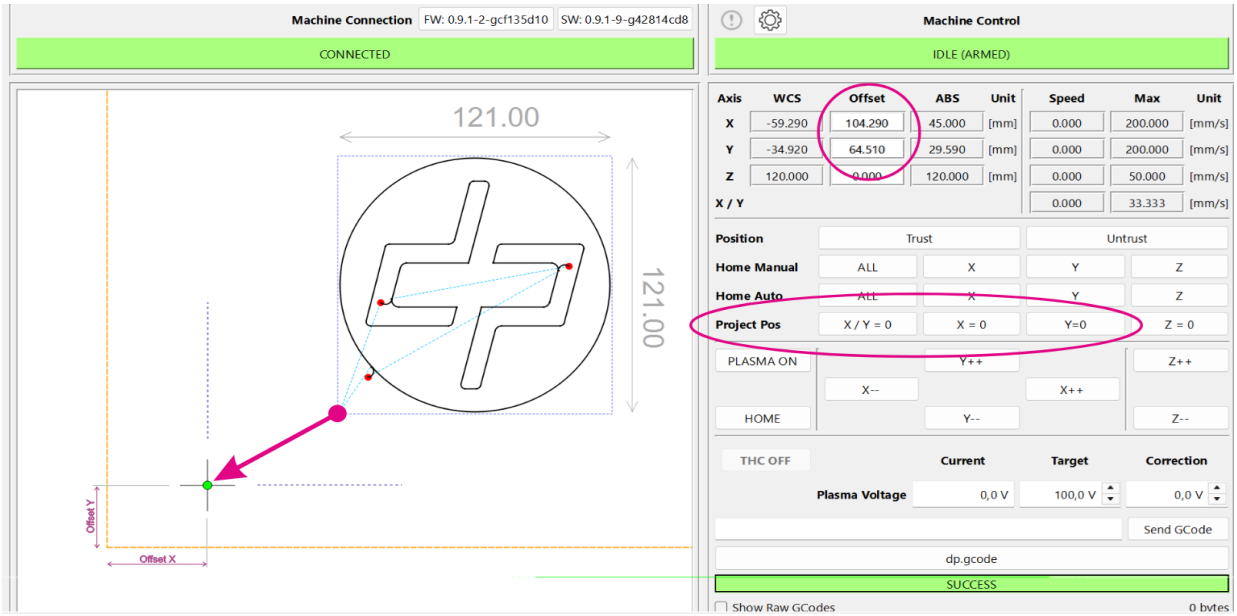
This function assigns the reference positions in the same way as during automatic homing.

After homing (automatic or manual), the machine position is considered trusted, and software limits restrict axis motion to the defined boundaries for each axis.

Project Positioning on the Machine

Project placement on the machine is defined by setting an offset of the **Work Coordinate System (WCS)** relative to the machine reference point **ABS [0,0]**.

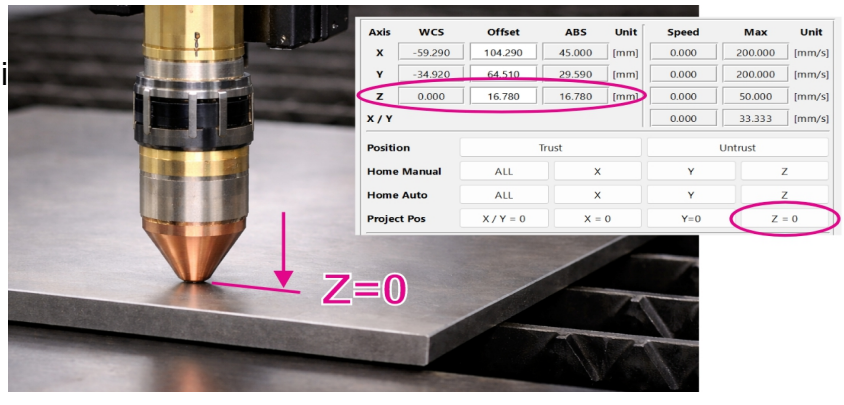
The **X/Y positioning buttons** assign the project **[0,0]** to the current tool position, allowing quick and precise alignment of the project origin with the tool location.



The offset value can also be entered manually directly in the **Offset** fields.

WARNING!

This function sets the zero point for G-code execution in the **WCS**. It does not necessarily represent the **Left-Down** corner of the project; it defines the reference point for **XY = [0,0] only**.

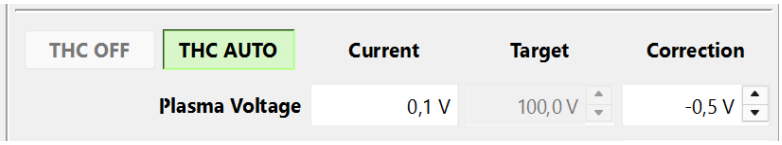


The material position on the Z axis s defined by its top surface.

The simplest method is to touch the material surface with the nozzle and press [**Z = 0] button**.

As with X and Y, the Z offset value can also be entered manually.

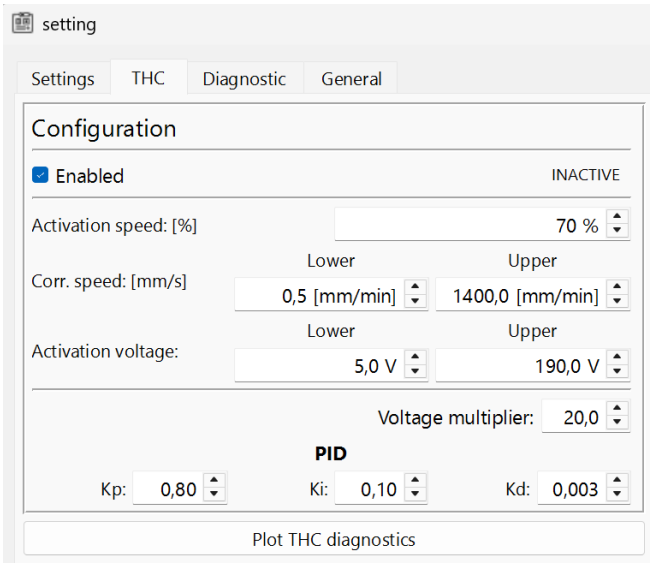
THC - Torch Height Control



Main THC Controller Parameters

- **THC Enable Indicator** – shows whether the THC controller is active.
- **THC AUTO Function** – automatically assigns the **Target Voltage** and provides a convenient method for initial voltage setup when the plasma source manufacturer does not specify this value.
- **Target Voltage** – a key parameter for maintaining the correct torch height above the material during cutting. The THC height controller raises or lowers the torch based on the difference between the measured arc voltage and the target value.
    - Higher value = greater torch height
    - Lower value = smaller torch height
- **Correction Value** – allows fine adjustment of the torch height if the actual height does not meet expectations. This value can be modified during cutting.

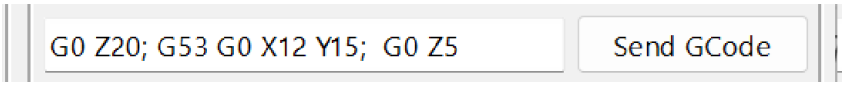
Advanced THC controller parameters are available in the **THC** tab of the configuration window and are described in detail later in this manual.



MDI - Manual Data Input

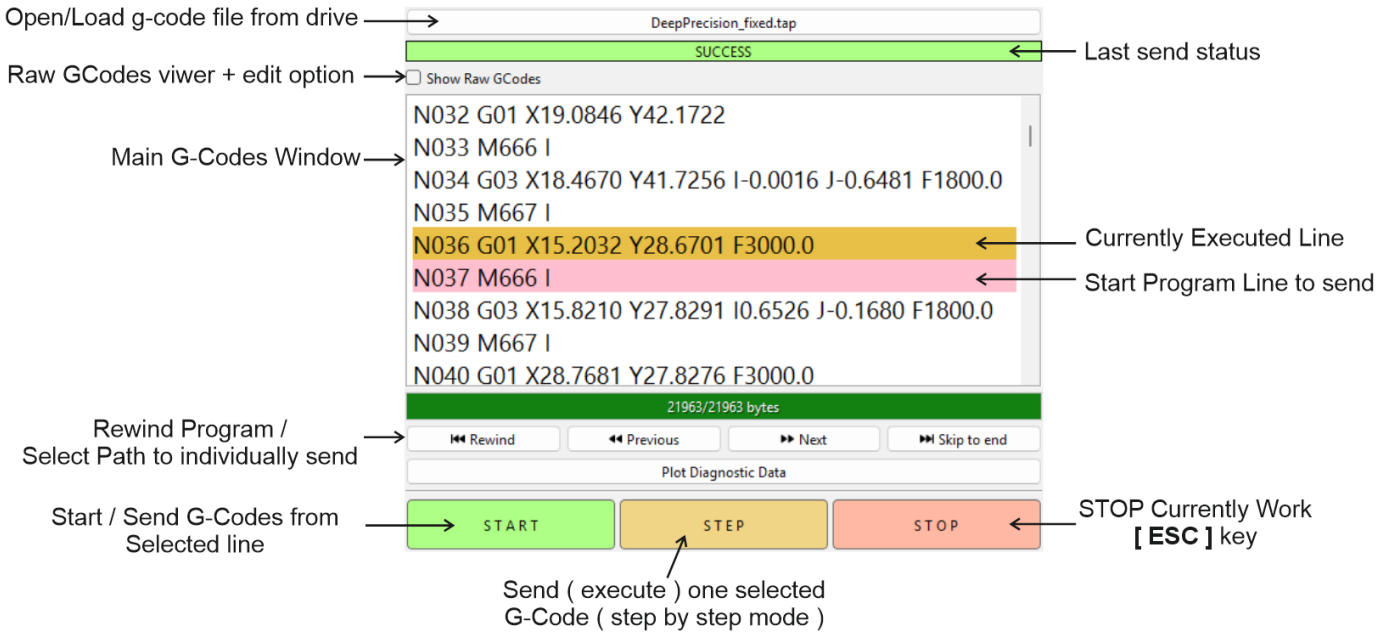
This function allows sending a single manually entered G-code command.

Multiple commands can also be sent at once by separating them with semicolons „ ; ” .

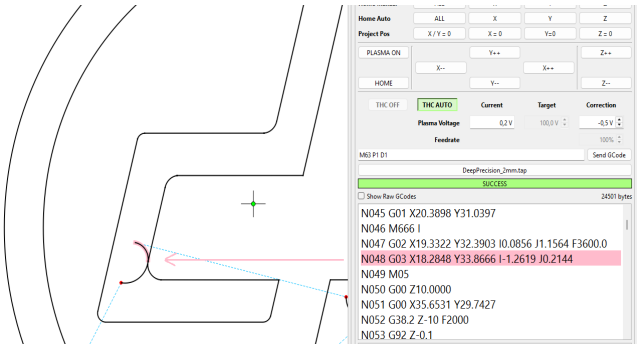


Automatic Program Execution

The function starts automatic operation by transmitting the G-code command list to the controller.

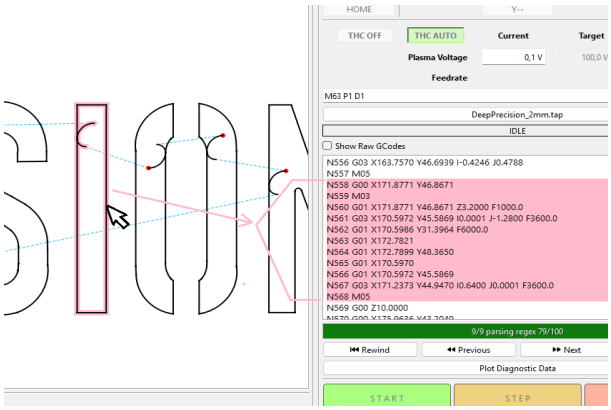


The selected G-code line is highlighted on the graphical toolpath.
Multiple G-code lines can be selected by holding the **SHIFT** key.



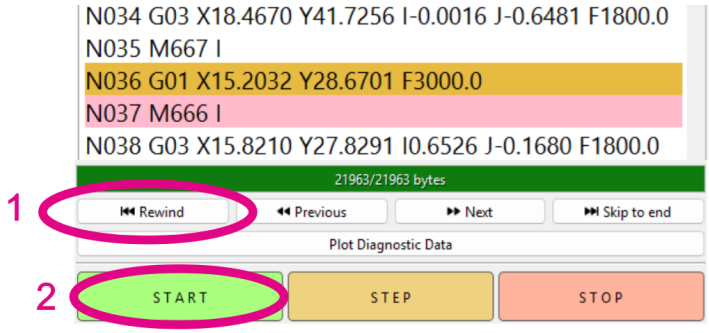
If a single G-code line is selected, **START** executes the program beginning from that line.

Selecting a toolpath on the scene with the mouse highlights the corresponding G-code lines.

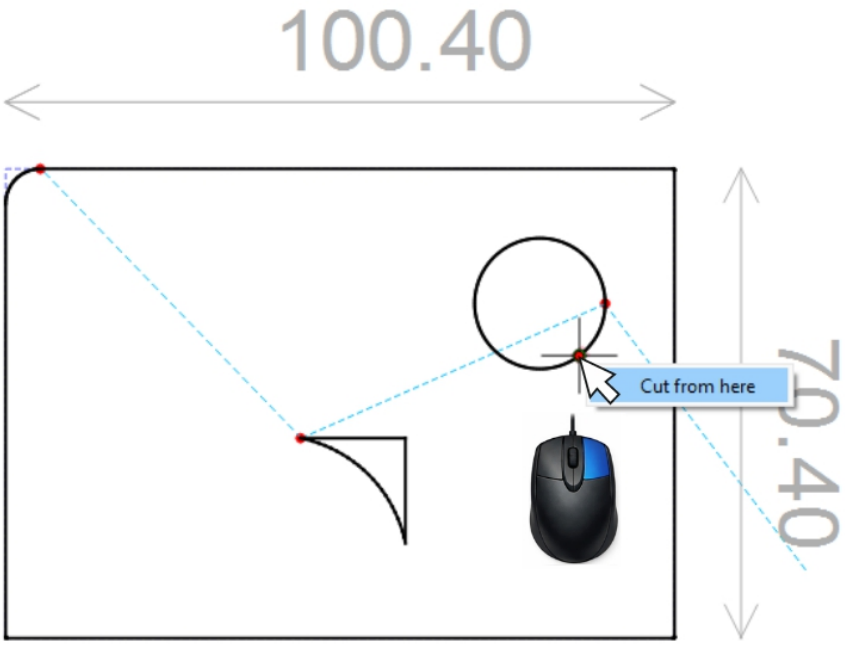


If a range of lines is selected, **START** sends and executes only the G-code commands within the selected range (e.g. cutting a single toolpath).

To start/send a gcodes from the beginning, rewind the program to the begin or clear the selection by clicking an empty area outside the toolpath. An automatic start from the beginning also occurs after program completion.



To start a toolpath from any point, right-click the desired start point and select **“Cut From Here”** from the context menu.



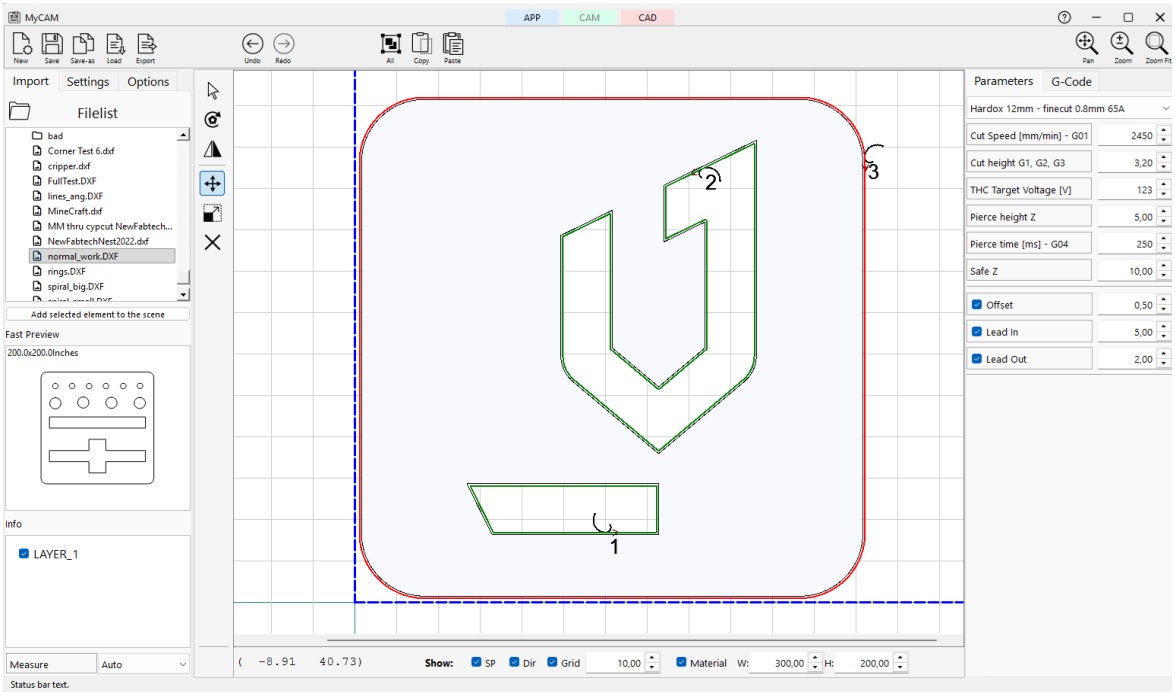
Before starting the working (cutting) motion, the **M3** command is executed by default.

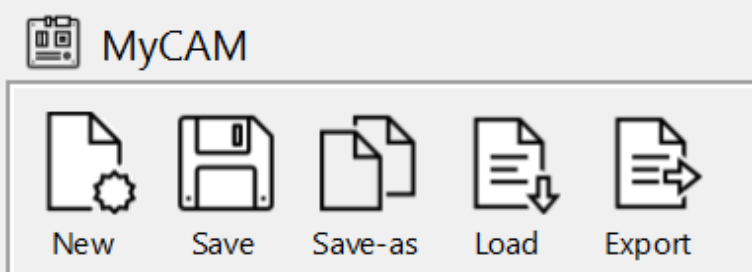
The command or command sequence can be freely customized as required in the **app/user_settings.toml** file by separating commands with the **`\n`** delimiter. For example :

```
continue_gcode = "M3\nM0 S2\nG4 P200"
```

4.2 CAM Module

The CAM module is used to prepare toolpaths and generate G-code based on the input drawing (e.g. DXF file).

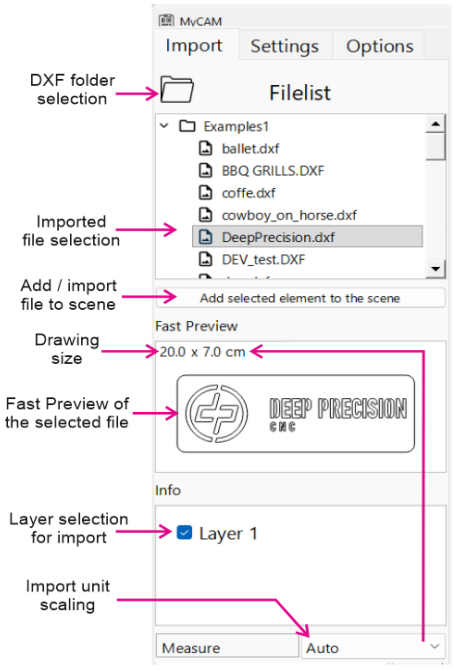




New – Clear Main Scene (New work)
Save – Save all currency work as .cam file
SaveAs – Save as new filename
Load – Open Saved .cam from disk

Export – Auto Generate G-Code and Send it to Controll Application (Will Auto Load)

DXF file import window



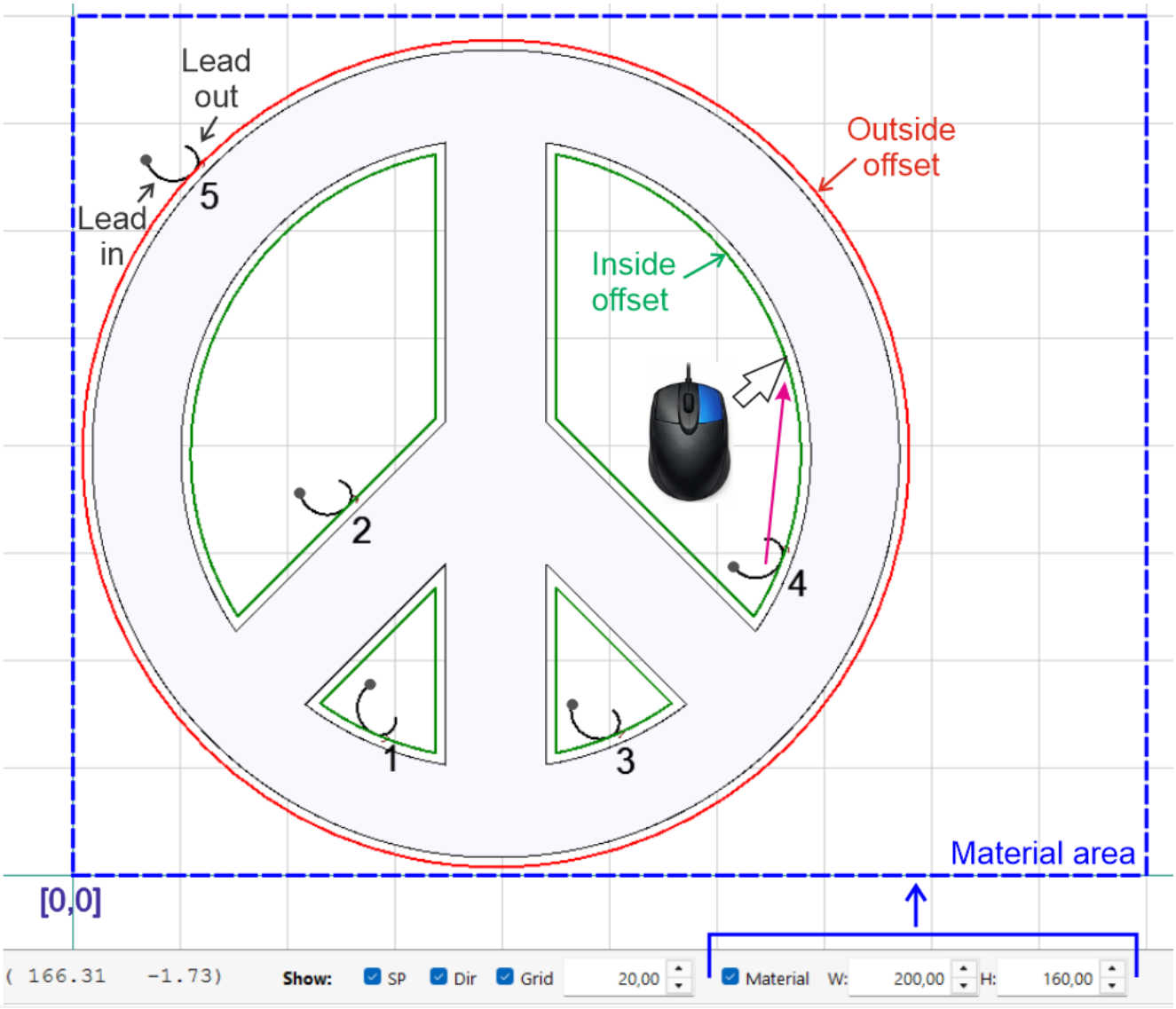
The **DXF Quick Preview** window allows browsing graphic files stored on the disk.

Tip: Right-click to change the sorting method or refresh the file list.

Double-clicking an item in the list or clicking **"Add selected item to scene"** imports the file.

Before importing or adding a file to the main scene, ensure that the scale and units are correct. The displayed size in the preview window should be accurate; otherwise, adjust the import units.

CAM Main Scene



On the main scene, the workpiece (imported drawing) and the toolpath visualization are displayed.
Internal paths (for holes), shown in green by default, use an internal offset, while external paths (contours), shown in red by default, are offset outward.

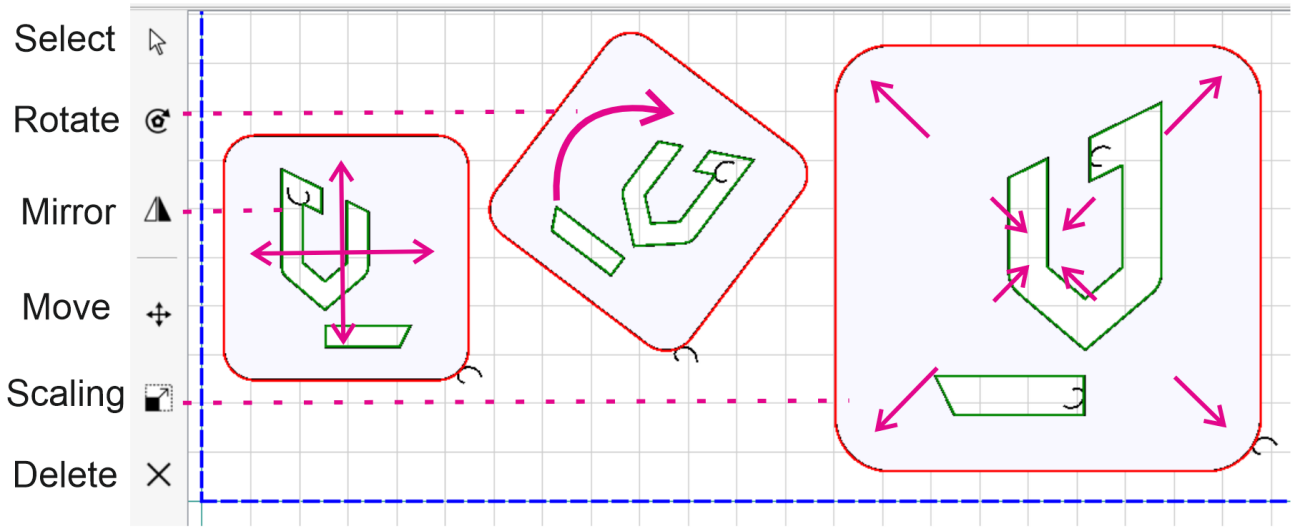
Pierce points (including optional lead-in and lead-out paths) can be repositioned by right-clicking on the toolpath.

To display a symbolic material area, enter the material dimensions and enable the corresponding checkbox.

Special attention should be paid to the **[0,0]** point of the scene/material, as it defines the reference point for the generated G-code.

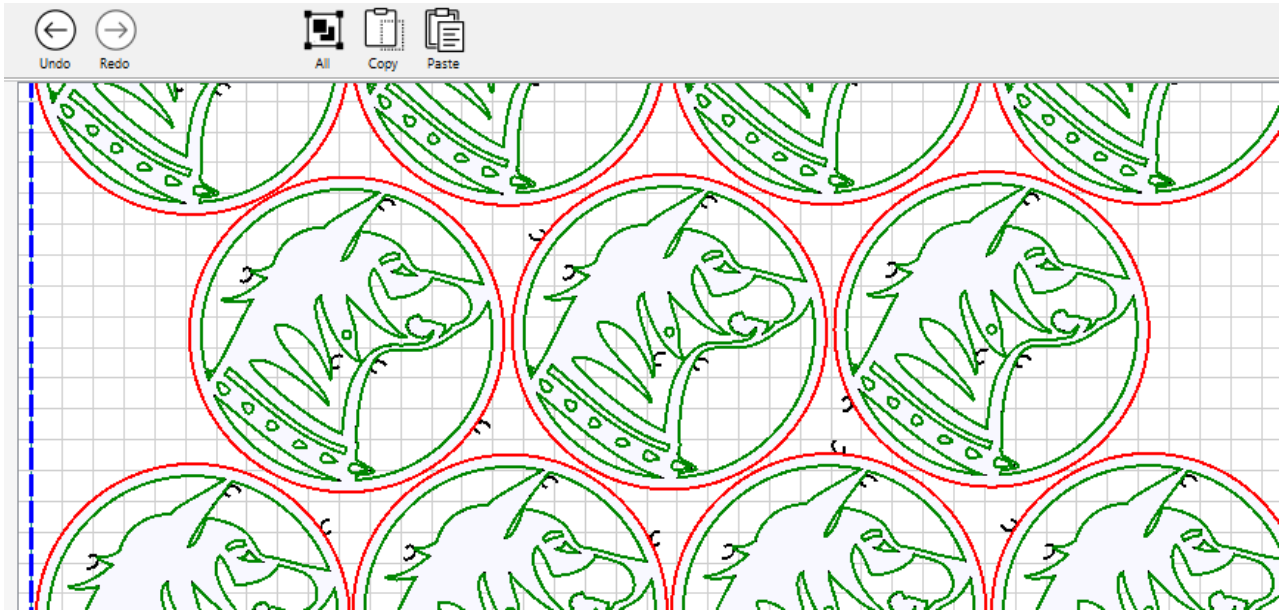
Options at the bottom of the screen allow enabling or disabling displayed elements such as start points, toolpath direction arrows, the material area, and the grid. The grid spacing can also be configured there.

Geometry Transformations



Tip: Holding the CTRL key allows selecting and modifying multiple elements at once.

Undo last operation [Ctrl+Z], Redo [Ctrl +Y], Select All [Ctrl+A], Copy [Ctrl+C], Paste [Ctrl+V]



Cutting parameters TBC

4.3 CAD Module - TBC

5. Connections TBC

6. Configuration TBC

7. G-Codes TBC

8. Advanced TBC

8.1 GUI Modyfications TBC

8.2 Create own function buttons TBC

8.3 Create own postprocessors TBC