

NX-Plasma Software Manual Version 5.1

NevexTech Ltda

Contents

1	Legal Notice	4
1.1	User Responsibility	4
1.2	Technical Expertise	4
1.3	Limited Liability	4
1.4	Copyright	4
2	NX-Plasma Software User Manual	5
2.1	Introduction	5
2.1.1	Key Features of NX-Plasma	5
2.2	Getting Started	7
2.2.1	Installation and Setup	7
3	Configuring Your Machine	10
3.1	Setup and Configuration	10
3.1.1	Axes configuration	10
3.1.2	References Configuration	11
3.1.3	Plasma Source Configuration	11
4	Starting the NX-Plasma CNC Software	13
4.1	Enabling the System	13
4.2	Referencing the Machine	14
4.3	Loading a Program	14
4.4	Jogging	14
4.4.1	Jog control	14
4.4.2	Keyboard Axes Jogging	15
4.4.3	Program Jogging	15
4.5	Setting the Cutting Parameters	15
4.6	Starting Cutting	16
5	Main Screen Overview	17
5.0.1	Navigating the User Interface	18
5.1	Toolbar	18
5.1.1	Toolbar Buttons	18
5.1.2	Switches	18
5.1.3	Status Indicators	19
5.2	Command Buttons	19
5.3	Indicator Gauges	20

5.4	Jog Control	20
5.5	Coordinates (Digital Readout Position Indicator)	20
5.6	The Z Axis Position Scale	20
5.7	Status and Message Line	21
5.7.1	States:	21
5.8	Menu Button	21
5.8.1	Save and File Operations	21
5.8.2	Save and Restore Work	22
5.8.3	Statistics	22
5.8.4	Diagnostics	22
5.8.5	About NX-Plasma	22
5.8.6	Help	22
5.8.7	Exit	22
6	Command Buttons	23
6.1	Start Command Button	23
6.2	Stop Command Button	23
6.3	Manual Command Button	24
6.4	Origin Command Button	24
6.5	Set Origin Command Button	24
6.6	Align Command Button	25
6.7	Load Program Command Button	25
6.8	Reference Command Button	25
6.9	Connect/Reset Command Button	26
7	DXF Import Dialog	26
8	Cutting Parameters Dialog	27
9	Axes Reference Dialog	31
10	Part Wizard Dialog	32
11	Editing Mode Controls	34
11.1	Nesting Tool for Layout Optimization	35
11.2	Leads tool	36
11.3	Selecting and Deselecting Parts in Editing Mode	37
11.4	Moving Parts	37
11.5	Editing mode shortcuts	37

12 Options Dialog	38
13 Configuration Dialog	39
13.1 General configuration	39
13.2 Inputs configuration	40
13.3 Outputs configuration	42
13.4 X, Y, Z, A, B, C, Axis configuration	43
13.4.1 Axis Inputs configuration	45
13.4.2 Axis reference configuration	46
13.5 Touch configuration	46
13.6 THC configuration	47
13.6.1 Advanced THC configuration	48
13.7 Other	50
14 Shortcut Keys	51
15 Troubleshooting	52
15.1 Torch Collides with the Table	52
15.1.1 Possible Causes and Solutions:	52
15.2 Torch Keeps Moving Up	53
15.3 Torch Not Firing or Cutting Properly	54
15.3.1 Possible Solutions:	54
15.4 Poor Cut Quality	54
15.4.1 Possible Solutions:	54
15.5 Lost Motion or Inaccurate Cuts	54
15.5.1 Possible Solutions:	55
15.6 Excessive Arc Lost Errors	55
16 Frequently Asked Questions	56
16.1 How to Move the Part Origin to the Center of a Part	56
17 Glossary of Plasma and Oxy-Fuel Cutting Terms	56
18 Appendices	57
18.1 User License Agreement	57

1 Legal Notice

IMPORTANT NOTICE

Please read this notice carefully before using the NX-PLASMA device. The use of numerical control machines involves inherent risks and potential hazards. Nevex Tecnologia LTDA (“Nevex”) cannot be held responsible for any accidents, damages, or harm caused by the operation of this device. By using this device, you agree to assume all risks and responsibilities associated with its use.

1.1 User Responsibility

You, as the user of the NX-PLASMA device, are solely responsible for:

- Ensuring the safe and proper operation of the machine.
- Complying with all relevant laws, regulations, and codes in your state/country.
- Implementing necessary safety measures to protect individuals and property.

1.2 Technical Expertise

The operation and maintenance of the NX-PLASMA device require a sound understanding of electricity, electronics, and CNC equipment. If you lack the necessary skills, Nevex strongly recommends seeking assistance from qualified technicians. Improper installation or operation may lead to serious accidents.

1.3 Limited Liability

NevexTech Ltd assumes no liability for:

- Any damage to the NX-PLASMA device or other equipment resulting from its use.
- Any injuries to you or third parties caused by operating the device.
- Any losses or disruptions arising from device malfunction.

1.4 Copyright

Copyright © 2018-2023 NevexTech Ltd All rights reserved.

Please use the NX-PLASMA device responsibly and adhere to the guidelines provided in this notice. Your safety and the safety of others depend on proper usage and understanding of the associated risks.

For inquiries or assistance, contact NevexTech Support.

2 NX-Plasma Software User Manual

2.1 Introduction

Welcome to the user manual for the NX-Plasma CNC system a user-friendly numerical control solution tailored for plasma and oxy-fuel cutting tables. In the following pages, you'll discover how this innovative software simplifies precision cutting through its intuitive interface and streamlined functionalities.

NX-Plasma isn't just another CNC system; it represents a shift in the way plasma cutting technology operates. Traditional CNC systems are primarily GCode followers, but NX-Plasma transforms this dynamic. It intelligently analyzes imported data, such as DXF files, creating a geometric model that represents parts and holes. Such a nuanced approach offers unmatched flexibility and control, surpassing conventional cutting methods.

This manual is your guide to effortlessly navigate the NX-Plasma CNC system, empowering you to unlock its full potential with ease. Whether you're an experienced professional or new to the world of cutting technology, you'll find the instructions here straightforward and approachable.

As you embark on this manual, you'll see how the NX-Plasma CNC system merges cutting-edge technology with a design focused on your needs. With step-by-step instructions, helpful tips, and practical examples, this manual is designed to be your companion in fully harnessing the system's capabilities.

Join us on this journey as we explore the features and functionalities that make the NX-Plasma CNC system an essential tool for efficient and precise cutting operations.

2.1.1 Key Features of NX-Plasma

- **Intuitive Modern Interface:** The software boasts a modern interface designed with didactic principles in mind, ensuring a user-friendly experience that accelerates your mastery of the system.
- **Efficient Workflow Control:** Our workflow control minimizes the need for constant operator oversight, streamlining the cutting process and enhancing efficiency.
- **No G-Code Knowledge Required:** You can dive into cutting operations without the need for G-Code language proficiency, making the system accessible to users of all skill levels.

- **Seamless DXF File Integration:** Import DXF files directly into the system and easily incorporate lead in/outs to your cutting profiles.
- **Quick Part Generation:** The integrated part wizard lets you swiftly create a variety of parts, simplifying the preparation process.
- **Optimized Nesting:** Maximize material usage by intelligently arranging parts for cutting, resulting in an optimized layout that reduces waste.
- **On-the-Fly Profile Selection:** Effortlessly choose profiles for immediate cutting, providing flexibility in adapting to changing requirements.
- **Precise Lead In/Out Generation:** Generate lead in/outs for part profiles with precision, enhancing the quality of your cuts.
- **Advanced Torch Height Control (THC):** Our top-notch THC ensures accurate cuts even on warped metal sheets, maintaining consistent quality.
- **Interruptible Work Sessions:** Save and restore your progress, allowing you to pause long cutting sessions and pick up right where you left off.
- **Accurate Jogging:** Navigate with accuracy using part profiles and nearest point on the profile, enabling precise starting points for your cuts.
- **Flexible Restart Options:** Choose to restart cuts from the beginning, the last stopped position, arc lost position, or nearest point on the profile for enhanced accuracy and time savings.
- **External Motion Control:** Our system is engineered from the ground up for plasma or oxy-fuel cutting machines, providing reliable external motion control with 5-axis capability.
- **Beginner to Expert Ease:** The modern didactic user interface ensures that regardless of your experience level, you can swiftly become proficient in operating the system.
- **Smart Hole Cutting:** Optimize hole cutting times by enabling a smart touch sequence that enhances efficiency.
- **Quick Manual Cutting:** Efficiently cut scrap metal pieces for reuse, adding a practical touch to your cutting operations.

Explore the power and convenience of NX-Plasma, where innovation meets user-centric design, making cutting operations seamless and productive for everyone.

2.2 Getting Started

2.2.1 Installation and Setup

Welcome to the software installation guide for NX-Plasma CNC. In this chapter, we will walk you through the process of installing the NX-Plasma CNC software on your computer. Before proceeding, it's essential to verify that your computer meets the necessary hardware requirements to ensure a seamless installation and smooth operation of the system.

2.2.1.1 Hardware Requirements Before you embark on the installation journey, please confirm that your system meets the following prerequisites:

- **IBM PC Compatible Computer**
- **Minimum 4GB of Main Memory**
- **Minimum 256MB of Video Memory**
- **At Least 32GB of Disk Space**
- **Mouse and Keyboard**
- **Windows 7, 8, or 10 Operating System**
- **Video Card Supporting OpenGL 3.0 or Higher**
- **100MB Network Interface**
- **Nx-Plasma Motion Controller**

Ensuring your hardware aligns with these specifications will pave the way for a successful installation and a satisfying experience using the NX-Plasma CNC software.

2.2.1.2 Installation Guide Follow these steps to seamlessly install NX-Plasma CNC software on your computer:

1. **Copy Installation Program:** Begin by copying the NX-Plasma installation program into a dedicated folder on your computer's hard drive.
2. **Initiate Installation:** Locate and double-click on the NX-PlasmaSetup.exe file to launch the installation process.
3. **Select Your Language:** Opt for your preferred installation language from the provided options and then click the "OK" button.
4. **Choose Installation Directory:** Utilize the browse button to designate the installation folder of your choice, then proceed by clicking the "Next" button.
5. **Confirm Installation:** Confirm your chosen installation folder by clicking the "Install" button.

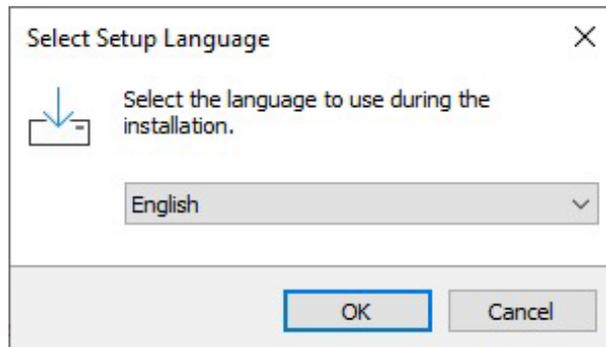


Figure 1: Select Installation Language

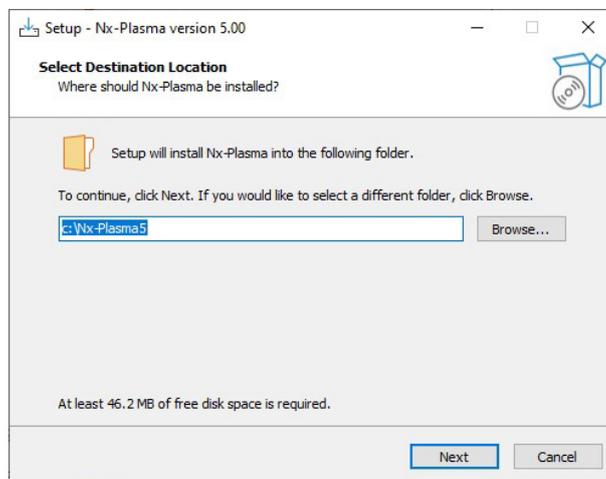


Figure 2: Select Installation Folder

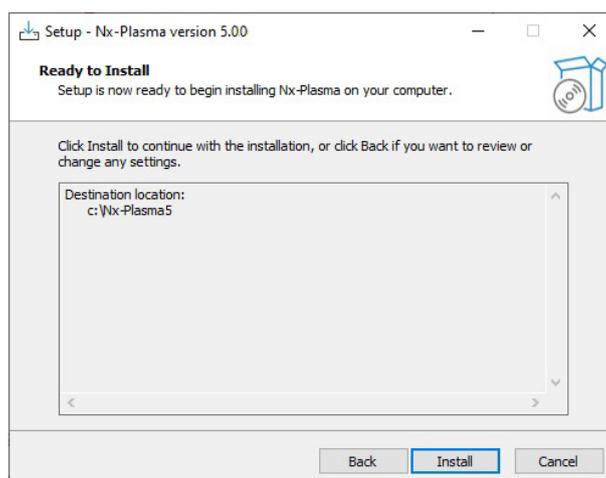


Figure 3: Confirm Installation Folder

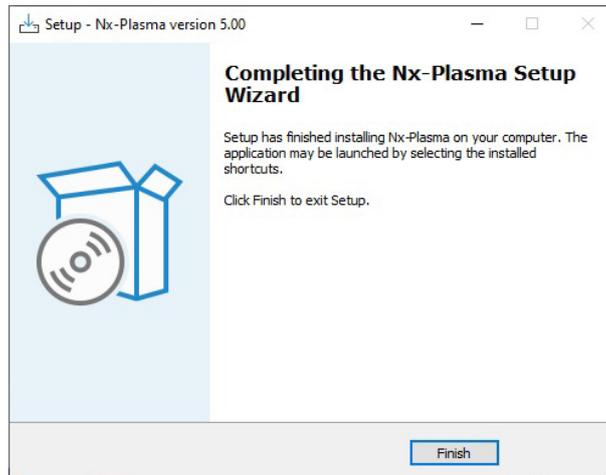


Figure 4: Installation Complete

6. **Completion and Finish:** Finalize the installation process by clicking the “Finish” button. Congratulations! You have successfully installed NX-Plasma CNC software on your computer.

3 Configuring Your Machine

3.1 Setup and Configuration

A plasma or oxy-fuel cutting table is an intricate machine that requires a thorough grasp of its components and functions. If you lack this expertise, it's advisable to seek professionals for the configuration and setup of the cutting table.

In this section, we provide a concise overview of the fundamental machine setup.

A typical plasma or oxy-fuel cutting table consists of the following components:

- **Table:** The work area of the machine that supports the material to be cut.
- **Y-Beams:** Long rails allowing forward and backward motion, these fixed beams provide support for the moving components of the machine.
- **Gantry:** Moves along the Y-beams and enables the left and right motion of the X-Carriage.
- **X-Carriage:** Moves along the gantry and supports the Z axis.
- **Z-Axis Head:** The Z-Axis assembly permits vertical motion of the torch.

3.1.1 Axes configuration

For the motion controller to effectively and accurately manage the cutting table, proper configuration is essential to align with the unique characteristics of your machine. These include:

- **X, Y, and Z Axis Configuration Parameters:**
 - **Steps/MM:** Calculating the number of pulses required to move the axes by one millimeter, accounting for any gearing or lead screws involved.
 - **Maximum Feed Rate:** Defining the highest permissible speed during axis movement.
 - **Maximum Acceleration:** Specifying the maximum allowable acceleration for axis motion.
 - **Direction of Rotation:** Designating the motor rotation direction (clockwise or anti-clockwise) to achieve precise table movement corresponding to CNC instructions.

These parameters must be entered in the Advanced Configuration dialog box, specifically under each axis. For Y-Beam motors, input the parameters for the Y axis while configuring the A axis as a slave to Y. Consequently, the A axis adopts Y's steps/mm, feed rate, and acceleration. Typically, the direction configuration for A should be opposite to that of Y, ensuring counter-directional movement for the motors.

This setup is standard, although alternative methods, such as wiring changes, can also achieve this effect.

3.1.2 References Configuration

A more advanced cutting table will likely include reference sensors (switches) that detect the home positions of each axis. The NX-Plasma system supports referencing in all axes, but it's essential, at a minimum, that the Z axis has a reference indicator positioned on top of the Z-Axis. This ensures a constant awareness of the precise Z axis position.

Upon initial machine startup, it's imperative to reference the Z axis. This involves using the reference command, which will move the Z axis until the Z++ reference switch is activated. At this point, the axis halts, and the "Z Reference Position" (configured within the Z axis settings) is designated as the current Z axis position. After this establishment, regular machine operations can commence. The value for this position should be determined by referencing the Z axis until it halts at the upper limit of its travel. Measure the distance from the torch tip to the tabletop of the cutting machine to obtain this value.

If the Z axis reference parameters are not correctly configured, the cutting process will not function as intended. The CNC system relies on accurate axis positioning information, and incorrect Z axis reference settings will lead to confusion about the actual axis positions.

3.1.3 Plasma Source Configuration

Each plasma cutting table is equipped with various plasma sources, each featuring a standard configuration comprising three distinct signals. These signals include:

- **Start Plasma Contacts:** Consisting of two wires, these contacts activate the plasma torch when short-circuited.
- **Arc Ok Signal:** Comprising two wires, this signal indicates that the plasma source has detected the successful transfer of the arc. It allows the CNC to initiate cutting movements. Additionally, the signal is employed to detect unexpected arc extinguishing, triggering an error and halting motion. If this signal remains enabled (which is recommended), motion control will only be activated upon the signal's detection from the plasma source.
- **Divided Arc Voltage:** This involves the division of the plasma arc tip voltage through a voltage divider. Nx-Plasma anticipates a 50:1 voltage division. It's crucial to ensure that the plasma source incorporates a voltage divider configured for this division ratio (50:1). Otherwise, the controller might incur severe damage if subjected to the full plasma voltage.

The default configuration values should function appropriately with standard controller wiring. However, if inputs other than the controller's standard ones are utilized, software configuration will be necessary.

Please note that NX-Plasma software enforces a work flows. For example, if the machine is not referenced or the part's origin has not been set, the "Start" button will be disabled, not allowing the cutting to start. This is done to help minimize operator oversight and in this case the CNC state will show "Not Referenced".

4 Starting the NX-Plasma CNC Software

To start the NX-Plasma CNC software, click the NX-Plasma desktop icon.



Figure 5: NX-Plasma Icon

Once loaded, the main screen will appear, serving as your workspace for control and management. Below, we outline the sequence of steps needed to initialize the software and start cutting operations.

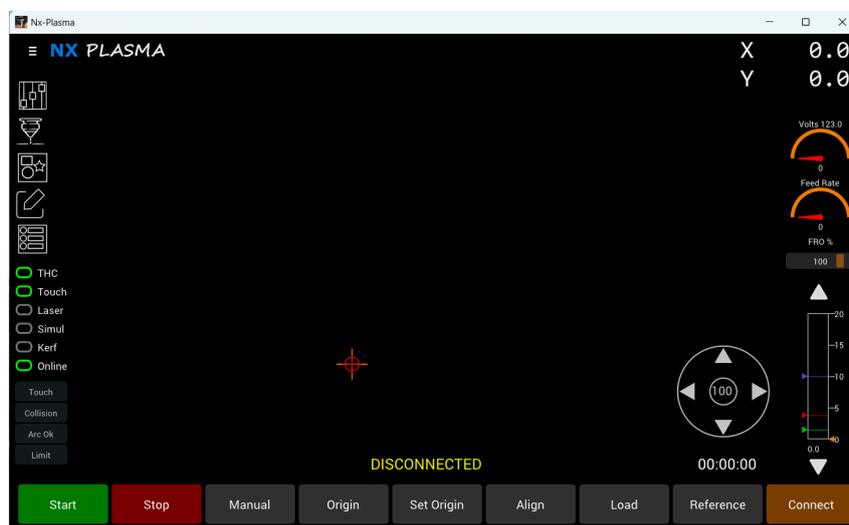


Figure 6: Initial Status displays a “DISCONNECTED”

Important: Please note that these instructions assume familiarity with plasma cutting systems and the ability to select appropriate parameters for cutting specific materials. Incorrect parameter usage or lacking understanding of the plasma cutting process can lead to cutting failure.

4.1 Enabling the System

1. **Initial Status:** Upon software startup, the system displays a “DISCONNECTED” status, indicating the control software has not connected to the motion controller.
2. **Connect to Motion Controller:** Establish the connection by clicking the “Connect” command button. This initiates communication between the CNC software and the motion controller. Ini-

tially, the status will change to “Offline” as a safety measure, preventing motion or plasma torch activation.

3. **Bring Online:** To enable the system, select the “Online” option button. The status changes to “Not Referenced,” indicating the need for machine referencng.

4.2 Referencing the Machine

1. **Power-Up Reference:** When powering up the CNC system, axes positions are unknown. Reference operations move the axes to known positions. Initiate referencng by clicking the “Reference” command button. A reference dialog appears for axis selection.
2. **Start Referencing:** Within the reference dialog, click “Start” to begin referencng. After completion, the dialog automatically closes.

4.3 Loading a Program

1. **Load Program:** Before cutting, load the numeric program file or create one using the Wizard. For detailed guidance, see the “Program” command button or the “Part Wizard” sections.
2. **Set Origin:** With a program loaded, the status line changes to “Set Origin.” Jog to the new origin position and click “Set Origin” to establish the part’s origin.

4.4 Jogging

To manually move the plasma torch, use the Jog Control or keyboard arrow keys.

4.4.1 Jog control

The Jog control enables you to move the machine or adjust the current jogging rate using the mouse.

- **Mouse Position in Outer Circle:** When you place the mouse within the outer circle and click, the machine will move in a direction corresponding to where on the circumference of the circle you clicked.
- **Mouse Click in Inner Circle:** Clicking within the inner circle increments the Jog speed percentage by 10% with each click. You can also use the ‘/’ or ‘**’ keys to increment or decrement the jog speed percentage.

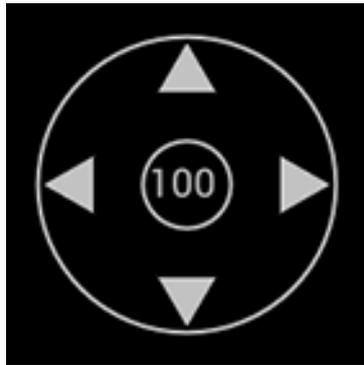


Figure 7: Jog Control

4.4.2 Keyboard Axes Jogging

- **X Axis:** Use left/right arrow keys.
- **Y Axis:** Use up/down arrow keys.
- **Z Axis:** Use PgUp/PgDn keys.
- **A Axis:** Use Home/End keys.
- **B Axis:** Use Insert/Delete keys.

Hold a key to jog an axis; release to stop. Shift + jog key achieves full-speed jogging.

4.4.3 Program Jogging

Program jogging is a feature that empowers you to navigate the plasma or oxy-fuel torch along the profile of the part. This functionality enables you to quickly position the torch at a specific point on the actual part being cut, streamlining the process of resuming cutting operations seamlessly.

To initiate a program jog, press and hold the control key while using the keyboard's left or right arrows.

A green circle is drawn around the nearest point on the part, indicating where the cutting process will commence when initiated from the current position.

4.5 Setting the Cutting Parameters

Prior to commencing cutting, it is crucial to configure the cutting parameters in alignment with the recommendations of the plasma source manufacturer:

Use the "Cutting Parameters Dialog" to select the appropriate cutting parameters that will be applied during the cutting process.

Within the “Cutting Parameters Dialog,” set the amperage, material type, and plate thickness to establish accurate cutting parameters.

For detailed guidance on utilizing the “Cutting Parameters Dialog,” please refer to the dedicated section titled “Cutting Parameters Dialog” for comprehensive information.

4.6 Starting Cutting

Before you initiate the cutting process, it’s essential to verify the following:

- Confirm the air supply pressure is set correctly.
- Ensure the plasma source is turned on and the appropriate cutting amperage is set.
- Guarantee the safety of individuals in close proximity to the cutting area.

Once these safety measures are confirmed, click the “Start” button to initiate the cutting process. The machine will execute the programmed instructions, transitioning the CNC state to “Executing” until completion. Once the cutting is finished, the CNC state will return to “IDLE.”

Congratulations on a successful cut!

5 Main Screen Overview

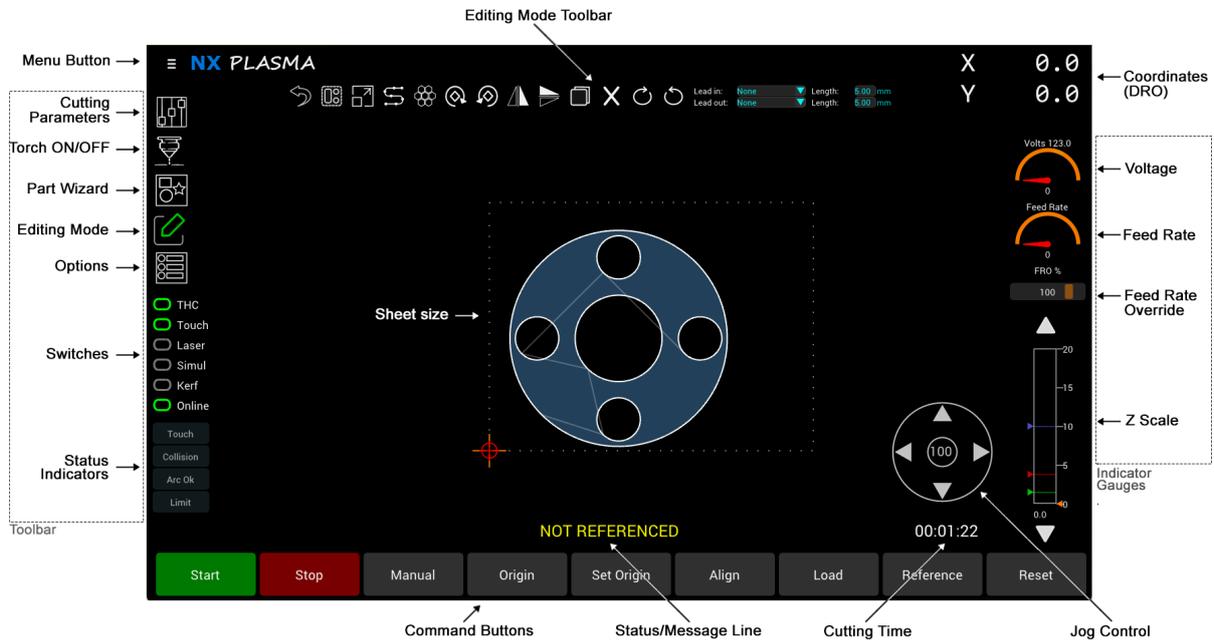


Figure 8: Main Screen

The main screen serves as the central hub for controlling and monitoring the CNC cutting operations. It provides a comprehensive view of the cutting process and essential controls, including:

- **Toolbar:** Quick access to functions such as cutting parameters, torch control, part wizard, edit mode and display options.
- **Command Buttons:** Actions like starting, stopping, manual control, loading programs, and more.
- **Indicator Gauges:** Real-time data for monitoring plasma arc voltage, current feed rate and Z axis position.
- **Jog Control:** Incremental movement along different axes for precise adjustments.
- **Coordinates (DRO):** Accurate position information for maintaining machining location.
- **Menu Button:** Access to file-related operations and software version information.
- **Status and Message Line:** Updates, notifications, and warnings related to the machining process.

5.0.1 Navigating the User Interface

- Use the mouse to click on buttons, switches, and controls for activation.
- Toolbar contains buttons grouped by functionality. Click to access dialogs, perform actions or switch configuration parameters on or off.
- Interpret indicator gauges for instant feedback on critical parameters.
- Utilize jog control for incremental movement and precise adjustments.
- Digital readout provides real-time machine position.
- Click menu button for file operations, machine configuration options and software version information.
- Monitor status and message line for updates and warnings.
- Utilize shortcut keys for faster navigation.

5.1 Toolbar

Positioned on the left-hand side of the screen, the toolbar is designed to provide quick and convenient access to crucial features and functionalities within the software. It is divided into three distinct sections, each serving a specific purpose to enhance your interaction with the software.

5.1.1 Toolbar Buttons

- **Cutting Parameters Dialog Button:** Accesses Cutting Parameters Dialog. Click this button to open the Cutting Parameters dialog and modify the current cutting parameters.
- **Torch On/Off Button:** Use this button to manually toggle the torch on or off. It also displays the status of the torch during cutting.
- **Part Wizard Dialog Button:** Accesses the Part Wizard Dialog, which provides a library of parameterized part.
- **Edit Mode Button** Switch between cutting mode and editing modes which allows you to manipulate the parts to be cut.
- **Display Options Popup Button:** Display the options dialog allowing you to configure display preferences.

5.1.2 Switches

- **THC Switch:** Allows you to activate or deactivate torch height control.

- **Touch Sequence Switch:** Toggle the use of the touch sequence during cutting. When On the CNC will perform a touch sequence before each cut.
- **Laser Pointer Switch:** Turns the laser pointer on or off for precision visual positioning of the torch's position.
- **Simulation Mode Switch:** Enables Simulation Mode for test runs. Activate this switch to engage the simulation mode, allowing you to conduct a test run without activating the plasma torch.
- **Kerf Switch** Turns on or off the kerf compensation.
- **Online Switch:** Allows you to place the CNC online or offline. in Offline mode no morots or torch commands will be issued.

5.1.3 Status Indicators

- **Touch Detected Indicators:** This indicator displays the status of the touch input.
- **Collision Detected Indicators:** When active it indicates that a collision has been detected.
- **Plasma Arc Indicators:** Show the status of the “Arc Ok”, input signal.
- **Limit Switch Indicators:** Shows whether a limit switch has been activated.

5.2 Command Buttons

The command buttons allow you to do various operations. Some buttons are enabled only during certain machine states. For example, “Start” can only be used when the machine has already been referenced, and the origin of the part established.

- **Start Button:** Starts the cutting process
- **Stop Button:** Stops any current movement of the machine.
- **Manual Button:** Launches the Manual Cut Dialog, enabling you to execute manual cuts or moves.
- **Origin Button:** Move to the part's origin.
- **Set Origin Button:** Set the part's origin.
- **Align Button:** Aligns the edge of the plate.
- **Load Program Button:** Loads a CNC program.
- **Reference Button:** References the machine's axes.
- **Connect/Reset Button:** Connects or resets the motion controller.

5.3 Indicator Gauges

- **Voltage Indicator:** Shows the plasma arc voltage during cutting.
- **Feed Rate Indicator:** Display the instantaneous feed-rate during cutting or jogging.
- **Feed Rate Override Indicator:** Allows you to vary the current feed rate by a percentage.
- **Z Position Indicator:** Display the current Z axis position.

5.4 Jog Control

The Jog control offers you the capability to manipulate the machine's movement and alter the jog speed seamlessly using your mouse. To relocate the machine, simply click within the circular area of the control and maintain the mouse button in a pressed state. As long as the button remains pressed, the machine will initiate movement in the designated direction. While the button is pressed, you have the freedom to maneuver the mouse in any direction within the circle, prompting the machine to jog correspondingly.

It's worth noting that utilizing the arrow indicators is unnecessary, as movement is unrestricted and possible in any direction, encompassing a full 360-degree range. Moreover, while keeping the mouse button pressed, you can rectify the direction of movement, enabling the machine to adopt a new trajectory without the requirement to halt its ongoing motion.

Should you desire to modify the pace of the jog, you can achieve this by clicking within the inner circle. Each click within this area will trigger a modification in jog speed. The percentage denoting the jog speed will progressively increase with each subsequent click and will eventually loop back to its initial value after reaching its maximum.

5.5 Coordinates (Digital Readout Position Indicator)

The DRO display the X and Y coordinates relative to the origin. When the laser pointer is switched on, it is displayed in red and its position adjusted by the laser offset from the torches position.

5.6 The Z Axis Position Scale

The Z-Axis scale serves to exhibit the present position of the Z axis. This scale is also equipped with markers for the approximate height, pierce height, and cutting height. The precise Z-coordinate is visually represented as a numerical value positioned beneath the Z scale.

For making alterations in the vertical alignment, you can utilize the arrow buttons situated at the top and bottom. These buttons facilitate the movement of the Z axis either in an upward or downward direction, allowing you to precisely adjust the vertical position.

5.7 Status and Message Line

The NX-Plasma CNC during its operation can be in one of various states. A descriptive text of the state of the CNC is displayed on the screen.

5.7.1 States:

- **Disconnected**
- **Idle**
- **Executing**
- **Jogging**
- **Not Referenced**
- **Set Origin**
- **Offline**

5.8 Menu Button

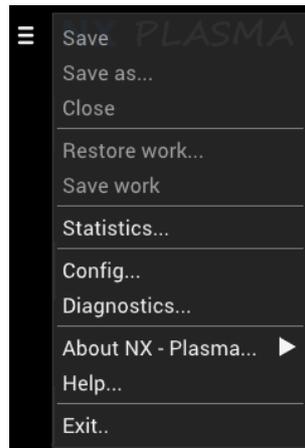


Figure 9: Menu Button

5.8.1 Save and File Operations

- **Save:** Save your workspace to a file. The file generated is of type “.nx”.
- **Save As:** Save with a new name or location.
- **Close:** Close the current file and clear all parts.

5.8.2 Save and Restore Work

- **Save work:** This feature allows you to save the current workspace along with its parameters. By doing so, you can power off the machine and, at a later time, restore it to the exact operating configuration you had when you saved your work. Clicking this option opens the save file dialog, where you can specify the desired file name for the work file, which will have the “.wks” extension.
- **Restore work:** Selecting this option opens the file load dialog, enabling you to load a “.wks” workfile. This action restores the operating environment to the state it was in when you originally saved the work. If your machine was turned off, you will need to reference it before fully restoring the work. All other parameters will be automatically restored from the work file. It’s important to note that if your machine lacks x and y reference sensors or you do not use them, you must avoid moving the machine once it’s turned off. In such cases, you will need to specifically reference the Z axis for accurate restoration.

5.8.3 Statistics

Display cutting statistics

5.8.4 Diagnostics

Display the diagnostics dialog, which allows you to monitor the status of all inputs and outputs. Clicking on an output will toggle that output on or off allowing you to perform tests on the output.

5.8.5 About NX-Plasma

Display the software and firmware versions, along with the copyright notice.

5.8.6 Help

Will display the keyboard shortcut help. You can also use the F1 key to display this help.

5.8.7 Exit

Will exit the program. Before it does so you will be required to confirm your action.

6 Command Buttons

The command bar enables you to initiate specific CNC actions.

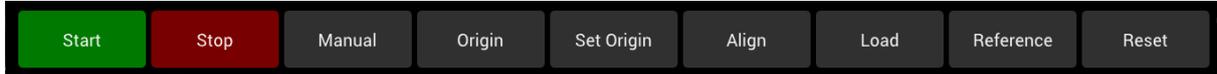


Figure 10: Menu Button

The command buttons allow you to do various operations. Some buttons are enabled only during certain machine states. For example, “Start” can only be used when the machine has already been referenced, and the origin of the part established.

6.1 Start Command Button

The start button begins the cutting process. Depending on the state of the cutting process, a menu will be shown allowing you to choose between starting from the beginning of the program, last stopped position, the arc position where the arc was lost or the nearest point on the cutting profile relative to where the machine is currently positioned.

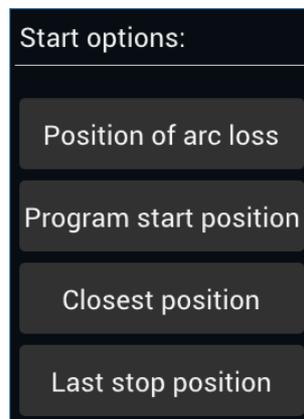


Figure 11: Menu Button

6.2 Stop Command Button

The stop button stops any current movement of the machine.



Figure 12: Manual Cut Dialog

6.3 Manual Command Button

The “Manual” command button opens a dialog that allows you to perform manual cuts in either the horizontal or vertical directions. Here’s how to use it:

1. Position the torch at the starting point using the keyboard controls.
2. Click the “Manual Cut” command button.
3. In the dialog box that appears, enter the desired values for both the X and Y directions, indicating how far you want the torch to move horizontally and vertically.
4. Press the “Cut” button. The CNC system will initiate a cutting operation, moving the torch by the specified X and Y displacement amounts from its current position.

If you only want to move the torch without cutting, click the “Move” button instead of “Cut.”

6.4 Origin Command Button

The Origin button commands the machine to move to the part's origin.

6.5 Set Origin Command Button

Each time you load a new program or generate a new part using the wizard, the CNC system requires you to define the origin point for the part.

Here’s how you can set the part’s origin:

1. Position the torch at the desired origin point for the cut.
2. Press the “Set Origin” command button.

By following these steps, the origin point for your drawing is now aligned with the current position of your torch. This ensures that subsequent cutting operations will reference this established origin point for accurate and consistent results.

6.6 Align Command Button

The Align button is used to effectively rotate the part so that it can be aligned to a metal sheet not aligned to the machine axes. To use this function, move the machine to a position on the edge of the plate and press align, now move the machine to another position on the same edge and press align a second time.

6.7 Load Program Command Button

The “Load” button facilitates the loading of a CNC program. Clicking this button triggers the appearance of the “Load Program” dialog box. Here, you have the option to either load an existing program or import a DXF file. If you select a DXF file for import, the DXF import dialog will be displayed. For a comprehensive understanding of the importation process when using DXF files, please refer to the “DXF Import Dialog” section, where you’ll find a detailed explanation.

The native file format for the Nx-Plasma CNC system is “.nx,” which can be saved or saved with a different name using the “Save” or “Save As” options in the main menu. Additionally, the CNC system supports the loading of G-Code files with the extensions “.nc,” “.tap,” or “.cnc.”

G-Code files compatible with this CNC system should adhere to the following GCodes exclusively:

G00: Used for rapid movements. G01: Employed for linear interpolation. G02 and G03: Utilized for circular interpolation. M03: Indicates the start of a cut. M05: Marks the end of a cut. Importantly, these GCodes should only include X and Y values, with no Z-axis movements. The CNC system takes control of Z-axis movements, including positioning for piercing or cutting height, as well as any necessary delays. Therefore, these elements should not be included in the GCode files intended for use with the Nx-Plasma CNC system.

6.8 Reference Command Button

When you first turn on the CNC, it does not know where the machine axes are positioned. In order to synchronize the positions, it requires Referencing the axes. During referencing, each of the axes will be moved until the reference switch is triggered. This will signal to the CNC that the axis is now in a known position.

To reference the machine, click on the Reference button on the command menu, displaying the Reference dialog box.

Select axes to be referenced and click the “Start” button. The CNC will reference each axes, starting with Z axis followed by the Y and X axis.

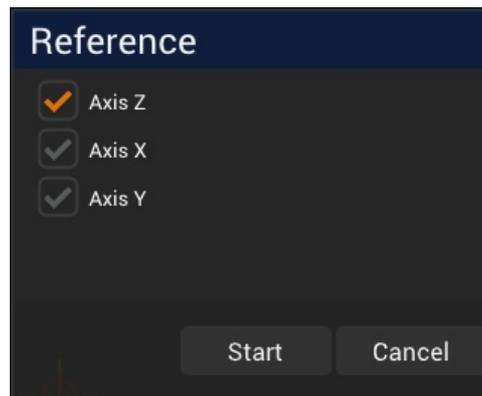


Figure 13: Reference Dialog Box

Notice that the only mandatory axis to be referenced is the Z axis. If you do not perform a reference operation, the CNC will not allow you to start cutting.

6.9 Connect/Reset Command Button

The Connect button connect the CNC to the motion controller. If already connected it changes to Reset, which is used reset the controller if needed.

7 DXF Import Dialog

When you select a DXF file in the “Load Program” dialog box. You will be presenedted with the DXF Import Dialog. This dialogs allows you to specify the parameters used to import a DXF drawing into the NX-Plasma CNC.

The dialog is divided into three distinct areas. The left pane display the graphical representation of the DXF contents. You can pan the view by pressing the left mouse button and dragging the mouse or zoom in/out using the mouse wheel. You can also change the where leads are placed in the parts by clicking on the segment you desired the leads to be places. Remember that not all places in a part are suitable for placing leads, and the system will not place the leads if it detects an problem in the placement.

In the right pane you can specify the various parameters to be used for the importation process:

1. Lead in lead out parameters to be added to the parts in the DXF, these are:
 - Lead in type
 - Lead in length

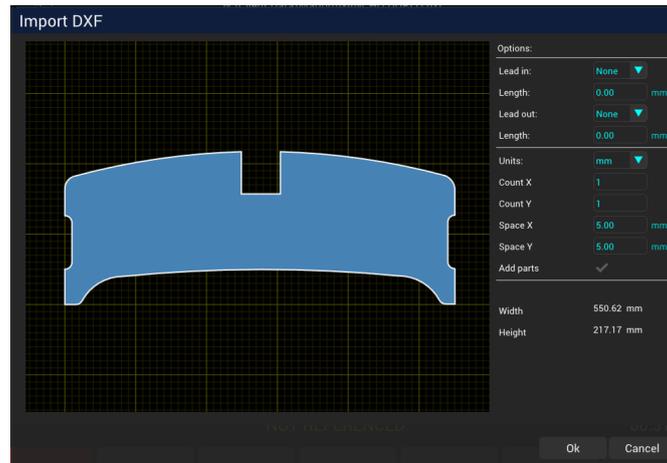


Figure 14: Dxf Import Dialog

- Lead out type
 - Lead out length
2. The units which the DXF was created (metric or imperial)
 3. The number of copies in the horizontal and vertical directions together with their separation distances. .

8 Cutting Parameters Dialog

The cutting parameters play a pivotal role in guiding the CNC machine during plasma cutting or oxy-fuel cutting processes. These parameters are subject to variations based on factors like the material being cut, its thickness, and the amperage employed. In the case of plasma cutting, each plasma source mandates distinct values, and these specifics are usually outlined in the plasma source user’s manual.

The Cut Parameters dialog is organized into four distinct sections:

1. Filter Bar:

- The filter bar provides options to select the cutting amperage and material type. This interactive feature enables you to precisely filter and display relevant cutting parameters based on your choices.
- Additionally, the filter bar includes three essential buttons: “Delete”, “Create New” and “Save”. These buttons empower you to efficiently manage cutting parameters, allowing deletion, creation and preservation of parameter sets.

Thickness	Parameters	Value	Unit
2mm	Plate thickness	5.00	mm
3mm	Cutting speed	750.00	mm/min
4mm	Volts	123.0	volts
6mm	Pierce height	3.80	mm
8mm	Pierce delay	0.20	sec
10mm	Cutting height	1.50	mm
12mm	Kerf	1.00	mm
16mm	Torch off delay	0.00	sec

Optimize touch Touch distance 50 mm
Safe height 20.0 mm Approximation height 10.0 mm

Figure 15: Cut Parameters Dialog

2. Material Thickness Selector Pane:

- Positioned on the left-hand side, the material thickness selector pane showcases a range of available cutting material thicknesses. By clicking on a specific thickness, you activate the display of corresponding cutting parameters for that particular thickness. This intuitive selection process simplifies the adjustment of settings based on material thickness.

3. Cutting Parameters Pane:

Situated on the right-hand side, the cutting parameters pane furnishes you with the specific cutting parameters pertinent to the chosen material thickness. These parameters directly influence the cutting process and include essential details such as:

- **Plate thickness:** The Material thickness.
- **Cutting speed:** Different materials, thicknesses, and cutting amperages necessitate varying the cutting speed, this should be in accordance with the cutting tables provided by your plasma source.
- **Volts:** Represents the optimal voltage setting for proficient cutting.
- **Pierce height:** Indicates the initial height at which the arc is ignited. This value significantly impacts the lifespan of consumables. If set too low, molten metal spatter during arc ignition can damage the nozzle.
- **Pierce delay:** Denotes the duration it takes for the plasma arc to pierce through the material. After this delay, the torch is elevated to the cutting height, marking the commencement of the

cutting movement.

- **Cutting height:** Refers to the height at which the actual cutting begins after the material is pierced. The CNC system transitions from the pierce height, after the pierce delay, to the designated cutting height.
- **Kerf:** Represents the compensation applied to the cutting path. The program is adjusted to account for the material lost during cutting. A “KERF” option, available in the options area on the main screen, enables you to activate or deactivate this compensation.
- **Torch off delay:** This feature enables you to define a delay period before the torch is deactivated upon completing the cutting profile. It proves particularly valuable when working with thicker metals, as it allows the arc at the plate’s bottom to fully catch up before shutting off.

4. Other Non-Material Dependent Parameters and Actions:

The bottom pane houses parameters that remain consistent across different materials. This is also where you’ll find the “OK” and “Cancel” buttons. The placement of these buttons facilitates the seamless acceptance or rejection of parameter changes.

- **Optimize touch:** Enables or disables touch optimization. This feature optimizes the cutting process by bypassing the need for a touch sequence when consecutive touches are situated within the specified distance outlined by the touch distance parameter.
- **Touch distance:** Specifies the maximum permissible separation between consecutive touches. If the distance between touches falls below this specified value, the need for a touch sequence is waived. It’s important to note that this option must be employed in conjunction with the “Optimize Touch” feature. This combined functionality reduces cutting times by intelligently managing touch sequences based on the defined touch distance threshold.
- **Safe height:** Establishes a secure Z-axis height that the machine can navigate without encountering obstructions. Adjusting this height strategically can lead to a reduction in overall Z-axis travel, effectively minimizing cutting times. By setting an optimal safe height, you enable the machine to move confidently within its designated workspace, enhancing both efficiency and safety during the cutting process.
- **Approximation height:** The approximation height marks the height above the material thickness at which the CNC transitions from rapid downward movement to a slower pace, facilitating precise touch execution without risking impact at high speeds. Skillful management of this height not only ensures accurate touches but also contributes to faster cutting operations. By strategically adjusting the approximation height, you can strike a balance between speed and precision, enhancing the efficiency of your cutting processes.

This structured layout of the Cutting Parameters Dialog guarantees an intuitive and streamlined experience, allowing you to efficiently tailor cutting parameters to suit the specific material and its thickness.

9 Axes Reference Dialog

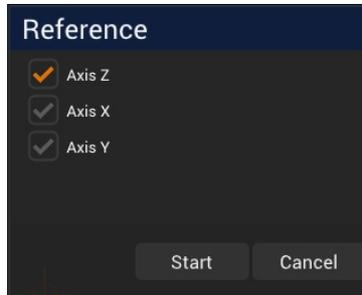


Figure 16: Reference Dialog

The reference dialog helps you set the starting positions for the cutting table's different axes. Referencing involves moving each axis to its initial position, where a sensor will be triggered. This sensor tells the CNC that the axis is in its starting position. The axis position is then marked as the "Reference Position" in the axis configuration.

Using the reference dialog, you can choose which axes to reference. It's essential to reference the Z axis at the very least to ensure accurate cutting.

The order in which the axes are referenced is determined by the setting in Configuration -> Other -> Reference Sequence. Typically, it starts by referencing the Z axis, followed by the X and Y axes.

10 Part Wizard Dialog

The Part Wizard is a collection of pre-designed components, each of which can be easily tailored to meet your specific needs through adjustable parameters.

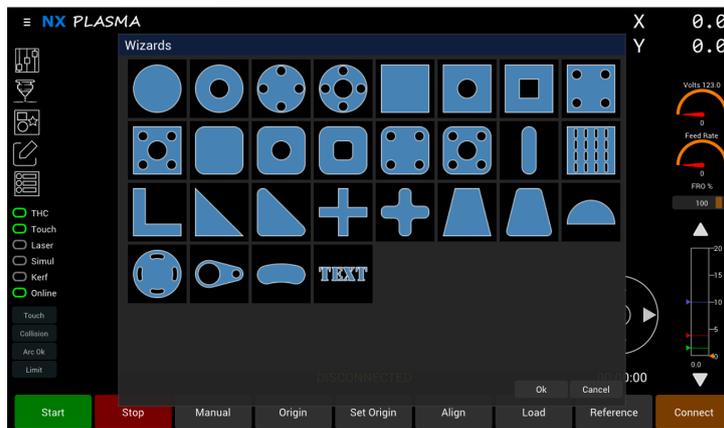


Figure 17: Wizard Dialog

When you access the Part Wizard dialog, you will find a list of available part types. By selecting a desired part through a simple click, you can access the corresponding part configuration dialog.

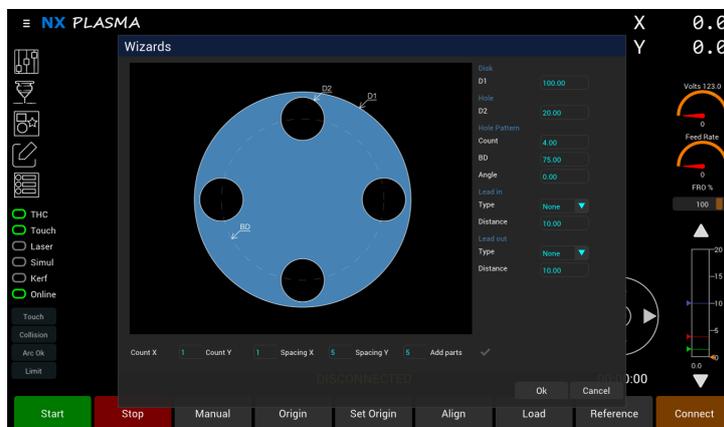


Figure 18: Wizard Dialog

In this configuration dialog, the selected part is displayed on the left pane, while the adjustable parameters are presented on the right pane. These parameters enable you to fine-tune various characteristics of the chosen part. For instance, if you are configuring a flange, you can adjust parameters such as the flange diameter, hole diameter, the number of holes, and more.

Additionally, all parts within the wizard include lead-in and lead-out parameters. These settings allow you to specify the type of lead-ins and lead-outs you prefer, along with their respective lengths,

ensuring precise customization for the selected part.

The lower pane provides the functionality to generate multiple copies of the parts in a grid pattern. This can be achieved by specifying the number of columns and rows, as well as defining the spacing between the parts in both the horizontal (x) and vertical (y) directions.

Additionally, there is an option labeled “Add parts.” When checked, this option permits you to append the newly generated parts to any existing ones already present in the workspace, facilitating efficient organization and management of your design.

11 Editing Mode Controls

The Editing mode empowers you to make real-time adjustments to the arrangement of parts within your current workspace. This valuable feature enables you to optimize your cutting process on the fly, eliminating the need for a separate CAM software, thus saving you both time and effort. With Editing mode, you have the flexibility to directly adjust your layout right on the CNC system.

When you activate Editing mode, a dedicated toolbar containing various editing commands becomes available for your use. You can enter Editing mode by either selecting the “Edit Mode” button on the main toolbar or by pressing the ‘e’ key on your keyboard.

It’s important to note that during Editing mode, no CNC control functions are permitted. You won’t be able to jog your machine, initiate cutting, or perform any CNC-related actions while in this mode. This restriction is a safety feature and also ensures that your focus remains solely on optimizing the layout of your parts.



Figure 19: Wizard Dialog

Here’s an overview of the available tools in Editing mode:

- **Undo:** This button reverts your workspace parts to their previous state. Use it when you’ve executed a command and wish to return to the previous workspace layout.
- **Select/Deselect:** Clicking this tool selects or deselects all parts, depending on their previous state.
- **Zoom All:** This tool adjusts the view to display all parts in the workspace simultaneously.
- **Optimize Paths:** This tool reorders the cutting sequence of parts in the workspace, optimizing it to begin cutting from the lower left corner. You can use the ‘s’ key on your keyboard during cutting mode to display the cutting sequence number for each part.
- **Nesting:** Opens the nesting dialog, which will be further explained in this chapter.
- **Rotate Clockwise:** Rotates one or more selected parts clockwise by 90 degrees. Holding the shift key while clicking this tool reduces the rotation to 45 degrees.
- **Rotate Counter-Clockwise:** Rotates one or more selected parts counterclockwise by 90 degrees. Holding the shift key while clicking this tool reduces the rotation to 45 degrees.
- **Flip Vertically:** Allows you to mirror selected parts on the vertical axis.
- **Flip Horizontally:** Allows you to mirror selected parts on the horizontal axis.

- **Copy:** Copies the selected parts, placing the copies to the right of the original parts in the workspace.
- **Delete:** Deletes selected parts from the workspace.
- **Leads Tool:** The next icons enable you to change the position of a lead by moving it to the next segment in the selected outline. Further details on leads manipulation can be found in this chapter.

11.1 Nesting Tool for Layout Optimization

The Nesting tool is a valuable feature in Editing mode, allowing you to optimize the arrangement of parts within the workspace by fitting them into a defined area referred to as a “sheet.” When you select this tool from the Editing mode toolbar, it will open the Nesting Dialog, as shown below:



Figure 20: Wizard Dialog

This dialog provides you with the flexibility to adjust parameters for the nesting process. The critical elements are as follows:

Sheet Size: This represents the dimensions of the area within which the nesting should occur, specified by width and height. The nesting operation aims to place parts starting from the lower left corner

and moving towards the upper right. When the specified width is exceeded, parts are placed on an upper row.

Part Fitment: If parts cannot fit within the defined sheet size, the nesting process will halt, and an error message will be displayed. The width parameter is particularly crucial, and by specifying a sufficiently large height, you can ensure a successful nesting operation.

Spacing: This parameter determines the minimum space to be maintained between individual parts during the nesting process.

Once you have configured these parameters to your liking, you can initiate the nesting operation by clicking the “Start” button. As the software evaluates various layouts, the view will continuously update to reflect the evolving arrangement. This process continues until the nesting algorithm finds an optimal solution.

To halt the nesting process, you can press the “Stop” button or click outside of the Nesting Dialog.

11.2 Leads tool

These tools are located on the right side of the editing toolbar and are activated only when a part is selected.

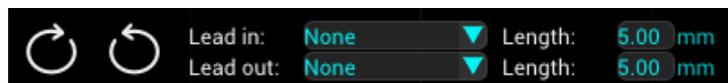


Figure 21: Wizard Dialog

When a part is selected, the leads will be applied according to the outer outline of the part. If you select a hole, the leads will be applied specifically to that hole.

The Lead Tools empower you to manipulate the leads of a chosen outline (profile). Each cutting outline consists of both lines and arcs. You can adjust the placement of the leads (in and out) along the outline segments by clicking either the “Displace Lead Clockwise” or “Displace Lead Counter-Clockwise” button. With each press of these tools, the lead position moves to the next or previous segment.

However, please note that leads cannot always be placed on every segment, as it could result in an invalid lead geometry. In such cases, the algorithm will attempt to select another segment where the lead can be properly placed.

If no suitable segment is found, the leads will not be applied.

For both lead-in and lead-out, the “Lead Type” control allows you to define the geometry of the lead. You can choose from available types such as Arc, Tangent Perpendicular, or None. The “Distance”

parameter (also applicable to both lead-in and lead-out) determines the length of the lead.

11.3 Selecting and Deselecting Parts in Editing Mode

In Editing mode, you have the option to select or deselect parts, including their holes, by simply clicking within them. Here's how it works:

Selecting a Part: When you click on a part, it becomes the active selection, and any previously selected parts are automatically deselected.

Appending to the Selection: To add a part to the currently selected set without deselecting others, hold down the Shift key on your keyboard while clicking on the desired part. This allows you to incrementally build your selection while keeping the existing selections intact.

11.4 Moving Parts

You can relocate selected parts by following these steps:

Click on a part using the left mouse button.

While keeping the mouse button pressed, move the mouse to the desired new position.

If all parts are selected, they will move together as a group.

However, if only one or two parts are selected, they will move until they encounter a collision with other parts in the workspace.

When a collision occurs, the moving parts will come to a halt, and a visual indicator in the form of a yellow circle will be displayed at the collision point. This feature prevents parts from overlapping and ensures that their separation distances are maintained.

It's worth noting that sometimes you may have parts loaded with separation distances smaller than the current set value. In such cases, a collision will be detected immediately, and the part will not move. However, you can still move the part by pressing and holding the Shift key. This action will push the colliding part away from the moving part. Be cautious with this method, as it can lead to unexpected results in the arrangement of other parts.

If an undesirable arrangement occurs, remember that you can always use the undo button to revert to the previous layout of the parts.

11.5 Editing mode shortcuts

- **Grid on/off:** Use the 'g' key to enable display of the grid.
- **Move origin:** Use the 'o' key to place the origin in the middle of the selected part.

12 Options Dialog

This dialog provides you with control over various options:

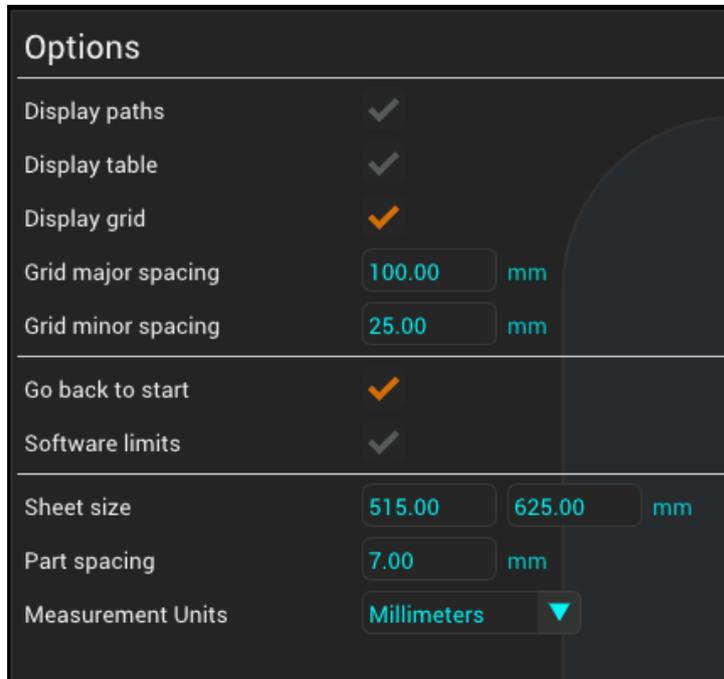


Figure 22: Wizard Dialog

- **Display Paths:** Toggles the display of motion paths on and off.
- **Display Table:** Enables or disables the display of the table area.
- **Display Grid:** Activates or deactivates the grid display.
- **Grid Major Spacing:** Specifies the spacing for major grid lines.
- **Grid Minor Spacing:** Determines the spacing for minor grid lines.
- **Go Back to Start:** Forces the CNC system to return the torch to the origin once cutting has concluded.
- **Software Limits:** Enables or disables software-imposed limits.
- **Part Spacing:** Sets the distance used for part separation during collision detection when moving parts in edit mode.
- **Measurement Units:** Allows you to choose between metric or imperial measurement units.

13 Configuration Dialog

The configuration dialog offers access to all adjustable settings within the NX-Plasma CNC system. To configure these parameters effectively, a technical understanding of conventional CNC systems and the Plasma/Oxy-fuel cutting process is beneficial. This knowledge ensures optimal setup and performance.

The configuration dialog is divided into two panes. On the left, you can choose a configuration theme or section, while the right pane displays the settings that are pertinent to the selected section. This layout makes it easy to navigate and adjust the specific parameters you need.

13.1 General configuration

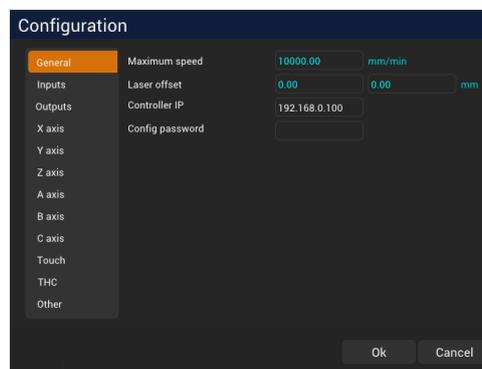


Figure 23: General Configuration Dialog

- **Maximum Speed:** This setting defines the highest speed at which the CNC will maneuver the torch.
- **Laser Offset:** Defines the X and Y offset for the laser pointer marker. Given that precise alignment of the laser marker with the torch position isn't always achievable, this parameter permits you to designate the X and Y offset from the torch location to the laser marker position.

Upon activating the laser switch, the coordinates on the Digital Read Out (DRO) will be adjusted by these values and presented in red. This serves as a visual reminder that the coordinates have been offset.

- **Controller IP:** This is the IP address of the external motion controller. By default, this value is set to 192.168.0.100. It must match the IP address currently assigned to the controller.

To verify the correctness of the address, open a Windows Command Prompt and utilize the “ping” command to assess communication with the controller. If the ping command indicates

packet loss, it signifies that the controller is unresponsive. This could be due to reasons such as the controller being powered off, a disconnection or defect in the Ethernet cable between the PC and controller, or the controller having a different IP configuration.

Example:

c:> ping 192.168.0.100

- **Config password:** To restrict unauthorized modifications to the configuration dialog, you have the option to set a password. This password must be entered to gain access to the configuration dialog.

To remove the password, simply clear the contents of this field.

13.2 Inputs configuration

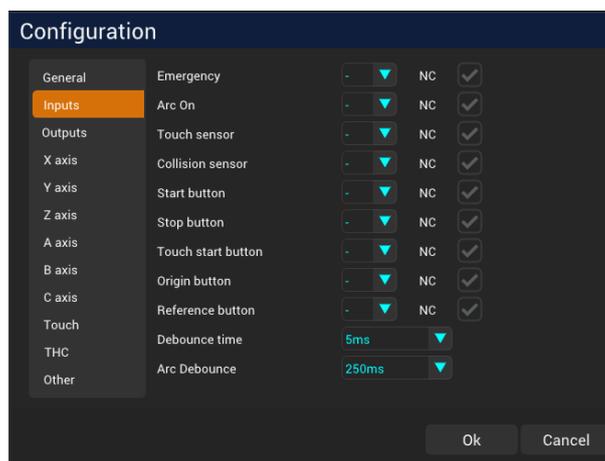


Figure 24: Inputs Configuration Dialog

The Inputs Settings section allows you to specify the physical inputs on the motion controller and their assignments to various functions. You can use the drop-down list next to each input function to select the desired input pin or choose the first entry (“-”) to indicate no assignment. Additionally, you can specify whether the inputs are “normally open” or “normally closed” by selecting the NC (Normally Closed) checkbox.

- **Emergency:** Assign this input to the emergency mode trigger. Typically, it is connected to a button that, when pressed, activates the CNC’s emergency mode, bringing all motion to an immediate halt.
- **Arc On:** This input is connected to the plasma source and acknowledges the transfer of the arc to the metal plate, signaling that CNC cutting may commence. In cases where this input is not

explicitly defined, the CNC system will disregard the signal. Furthermore, without the specific signal, it will be unable to detect the loss of the arc during the cutting process and may continue to cut, even if the arc is not successfully transferred.

- **Touch sensor:** Specify the input that indicates to the CNC when the torch has made contact with the top of the metal plate. This input is commonly connected to the Ohmic sensor or floating head sensor.
- **Collision sensor:** This input is wired to the collision sensor, which is used to detect a torch collision or unintended contact.
- **Start button:** Connect this input to the start button, which initiates the cutting process when triggered.
- **Stop button:** When activated, this input halts all ongoing motion and processes.
- **Touch start button:** Assign this input to the button that initiates the touch sequence, ensuring proper initial positioning of the torch.
- **Origin button:** Use this input to trigger the CNC to move to its designated origin position.
- **Reference button:** Assign this input to trigger the machine's referencing sequence, establishing known reference points.
- **Debounce time:** This parameter specifies the amount of time an input must remain stable before the CNC recognizes it as a valid signal.
- **Arc Debounce:** Specify the duration that the "Arc Ok" input must remain stable before the CNC acknowledges it as a valid signal.

Configuring these input assignments ensures that the CNC system responds correctly to various commands and sensor inputs, enabling safe and efficient operation during cutting processes.

13.3 Outputs configuration

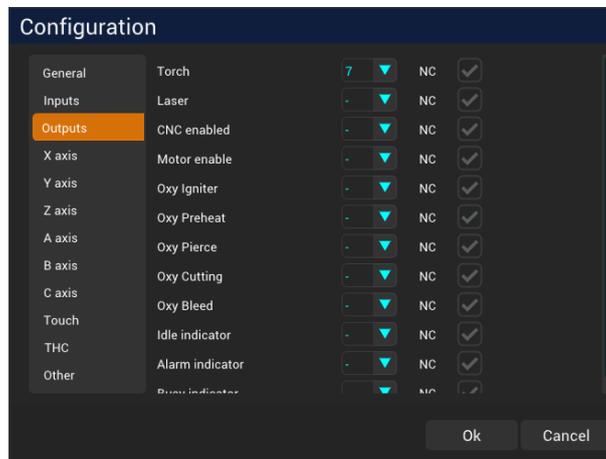


Figure 25: Outputs Configuration Dialog

The outputs configuration section allows you to specify how various output pins are assigned to specific functions within your CNC system:

- **Torch:** This entry permits you to specify the output pin number used to activate the torch. You have the flexibility to assign any output pin, but if left unspecified (default value), it will utilize the output connected to the internal relay of the motion controller.
- **Laser:** Determines the output activated when the laser pointer switch is engaged.
- **CNC enabled:** Activates the specified output when the CNC is in connected mode, indicating that it's ready for operation.
- **Motor enable:** Enables the designated output when the CNC is online, indicating that the motors are operational.
- **Oxy Igniter:** This function activates the output responsible for the oxy igniter, which is used during ignition.
- **Oxy Preheat:** This function activates the output during the oxy-fuel preheat phase, ensuring the necessary preheating process is initiated.
- **Oxy Pierce:** This function triggers the output during the oxy-pierce phase, which is critical for piercing through materials.
- **Oxy Cutting:** This function is activated at the beginning of oxy-fuel cutting and stays active until the cut is completed.
- **Oxy Bleed:** Activates the output during oxy-fuel bleed mode, ensuring proper gas flow.

- **Idle indicator:** This function signals the output when the controller is in idle mode, indicating that it's not actively processing commands.
- **Alarm indicator:** This function is activated when the controller encounters an alarm condition, indicating an issue that requires attention.
- **Busy indicator:** This function is activated when the controller is busy executing commands or processes.

Configuring these output assignments ensures that your CNC system functions efficiently, with outputs appropriately responding to different operational states and requirements.

13.4 X, Y, Z, A, B, C, Axis configuration

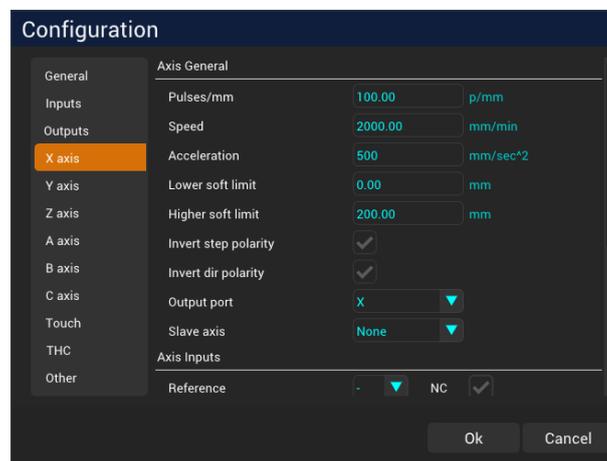


Figure 26: Axis Configuration Dialog

- **Pulses/Unit (Pulses per Unit):** This setting determines how many electrical pulses the controller will send to the motor driver to move an axis by 1 unit of measurement, this will be pulser per millimeter or pulses per inch, depending on you units setting.
- **Speed:** The speed parameter sets the maximum speed at which an axis can move. The controller will ensure that the axis does not exceed this speed limit during its movements.
- **Acceleration:** Acceleration defines the maximum rate at which an axis can change its speed. It's the rate at which the velocity increases or decreases. This parameter is important to prevent sudden jerky movements and ensure smooth transitions between speeds.
- **Lower soft limit:** The lower soft limit specifies the minimum position or coordinate to which the controller will allow an axis to move. If the axis reaches or attempts to move beyond this

limit, the controller may limit further movement or trigger an alarm if reached while cutting a part. It acts as a safety feature to prevent over travel or collisions.

- **Higher soft limit:** Similarly, the higher soft limit sets the maximum position or coordinate for an axis. It helps prevent the tool or torch from moving too far in a particular direction.
- **Invert step polarity:** This option allows you to invert the polarity of the step pulse signals sent to the motor driver.
- **Invert dir polarity:** Similar to the step polarity, this parameter allows you to invert the polarity of the direction signals sent to the motor driver. Inverting the direction polarity changes the axis's perceived positive and negative directions. For example, if pressing the right arrow normally moves the X-axis in the positive direction, but your machine moves in the opposite direction, you can use this option to correct it.
- **Output port:** This setting determines the physical connector or port where the control signals for the axis will be outputted. It allows you to assign a specific output port for a particular axis. This can be useful in situations where you want to customize the physical wiring of your control system. For example, you might want to connect the X-axis to a different motor connector (e.g., B motor connector) for specific reasons, and this parameter allows you to make that configuration.
- **Slave axis:** A "Slave Axis" is a concept in motion control systems where one axis replicates the movement of another axis. This is typically used in applications where two or more axes need to move synchronously or in coordination. Here's how it works:
 - **Primary Axis:** The primary axis is the axis that receives the control signals, such as step pulses and direction signals, from the controller. In your example, this would be the Y axis.
 - **Slave Axis:** The slave axis is the axis that replicates the movement of the primary axis. In your example, this would be the A axis.
 - **Replication of Signals:** All control signals sent to the primary axis (Y) are replicated in the slave axis (A), except for the direction signal. The direction signal can be inverted if specified in the configuration of the slave axis (A). This means that when you move the primary axis (Y) in one direction, the slave axis (A) will move in the same direction, following the same position and velocity profiles.
 - **Synchronization:** Using a slave axis allows for synchronized or mirrored movement of multiple axes. This is useful in applications where you want two axes to move together, maintain a fixed distance from each other, or perform coordinated actions.
 - **Direction Inversion**:** In some cases, you might need the slave axis to move in the opposite direction of the primary axis. To achieve this, you can configure the direction signal

inversion in the slave axis's settings.

Overall, using slave axes is a valuable feature in motion control systems, especially in scenarios where precise synchronization or mirroring of movements is required. It simplifies the control of multiple axes and ensures that they move in tandem, improving the accuracy and efficiency of the system.

13.4.1 Axis Inputs configuration

This section of the configuration enables you to specify the input pin number and signal status for various functions linked to the assigned axis.

Properly configuring these axis inputs is important in ensuring the secure and precise operation of your motion control system. It allows you to establish control over axis movement, define reference points accurately, prevent unintended over travel, and respond promptly and effectively to errors or alarms.

- **Reference:** This setting specifies the input pin number to which the reference sensor for the axis is connected. The reference sensor is used to establish a known reference point or home position for the axis. It helps the controller determine the initial position of the axis accurately.
- **Limit minus:** Specifies the input pin number for the negative direction limit sensor. When this sensor is activated, it indicates that the axis has reached its lower limit in the negative direction. The controller can use this information to prevent the axis from moving further in that direction, preventing over travel or collisions.
- **Limit plus:** Similar to the “Limit Minus” input, this setting specifies the input pin number for the positive direction movement limit sensor. When activated, it signals that the axis has reached its upper limit in the positive direction, allowing the controller to take appropriate action to prevent over travel.
- **Servo motor error:** This input allows you to connect the error or alarm signal from the servo motor. When the servo motor detects an error or fault, it can activate this input pin. In response, the CNC controller can generate an alarm, notifying operators of the servo error. This feature helps maintain system safety and reliability.
- **Jog minus:** Specifies an input pin number that, when activated (usually by a switch or button), causes the controller to initiate a jog movement of the machine axis in the negative direction. Jogging is a manual control method used for fine positioning and setup.
- **Jog plus:** Similar to the “Jog Minus” input, this setting specifies an input pin number that, when activated (usually by a switch or button), causes the controller to initiate a jog movement of the machine axis in the positive direction.

13.4.2 Axis reference configuration

These settings are essential for establishing accurate reference points for your axis, optimizing positioning precision, and ensuring safety during the homing process.

- **Position:** This parameter specifies the value that the axis coordinate will assume when the reference sensor is activated. It serves as the reference point or home position for the axis.
- **Speed:** The speed setting indicates the percentage of the maximum axis speed that the controller will utilize when maneuvering the axis towards the reference sensor. Lowering this speed percentage enhances precision in determining the position and increases safety in the event of a failure of the corresponding sensor.
- **Invert Direction:** By default, the normal direction for moving towards the sensor is in the negative axis direction. However, selecting this option will invert the movement to the positive direction. This option allows you to customize the reference approach direction to align with your specific system requirements.

13.5 Touch configuration

The “Touch” process is a critical step in determining the height of the metal plate to be cut. During the “Touch” sequence, the controller follows a specific procedure:

- **Initial Movement:** The controller begins by moving the Z axis at the maximum permitted speed towards the metal plate until it reaches the “approximation distance” (set in the cutting parameters dialog) above the configured plate thickness (also configured in the cutting parameters dialog).
- **Speed Adjustment:** Once the Z axis is within the specified approximation distance above the configured plate thickness (also configured in the cutting parameters dialog), its speed is reduced to the touch speed parameter.
- **Touch Detection:** The Z axis continues to move at this reduced velocity until it detects a change in state from the touch input. This change indicates that the torch tip has made contact with the metal sheet. This contact sets the relative zero height for the current cut.

This touch sequence is typically executed at the beginning of each cut, unless the “**Optimize Touch**” option in the cutting parameters is active. If the “**Optimize Touch**” option is enabled, the touch operation will only occur when the distance from the last cut exceeds the “Touch Optimize Distance” parameter.

Here are the key touch-related settings:

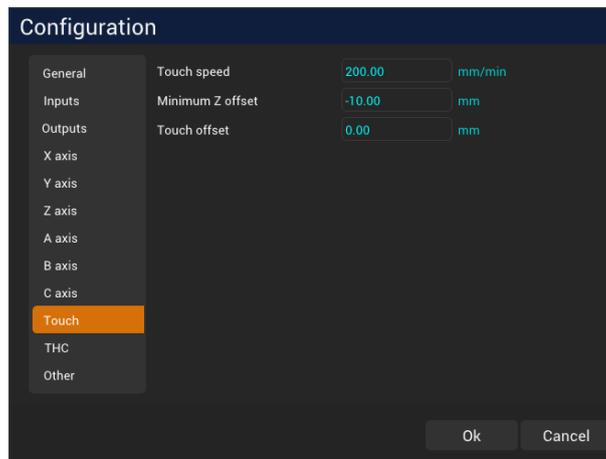


Figure 27: Touch Configuration Dialog

- **Touch speed:** This parameter specifies the speed to be used during the approximation phase of the touch sequence. Lowering this speed helps reduce the impact the torch experiences when contacting the metal plate.
- **Minimum Z offset:** This is an absolute Z height value. If the Z axis reaches this height without detecting a touch input change in state, it triggers an alarm, alerting the operator to this condition.
- **Touch offset:** When using a floating head for touch detection, there is often a discrepancy between the height indicated by the sensor for a touch and the actual plate height. By specifying this offset, you can compensate for this error and ensure accurate touch detection.

These touch settings are critical for achieving precise and safe metal cutting operations, as they help establish the correct cutting height and provide safeguards against potential issues during the process.

13.6 THC configuration

The THC (Torch Height Control) setting determines the behavior of torch height control in the NX-Plasma CNC system:

- **Voltage divider:** The NX-Plasma CNC utilizes a 50:1 voltage divider. Adjusting this value allows you to compensate for errors in the input voltage, ensuring accurate THC operation.
- **THC speed:** This parameter controls the maximum percentage velocity at which the Z-axis can be adjusted by the THC. Higher values make the THC more responsive. You can leave this at 100%, and the THC will automatically adjust itself as needed.

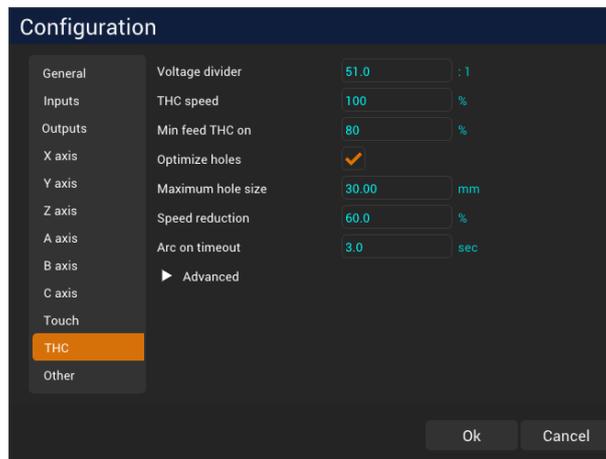


Figure 28: THC Configuration Dialog

- **Min feed THC on:** This setting defines the minimum percentage of the current cutting feed-rate that the controller must achieve before THC control is activated. This feature is particularly helpful during cornering or hole cutting, where acceleration constraints of the Z-axis may prevent the feed rate from being reached. When the speed falls below this threshold, THC is temporarily disabled, and torch height adjustments will resume once the speed surpasses the threshold again.
- **Optimize holes:** Enabling this option enhances hole cutting quality. Holes smaller or equal to the “Maximum Hole Size” (specified below) will have their cutting speeds reduced based on the “Speed Reduction” parameter.
- **Maximum hole size:** Specifies the maximum hole size for which hole optimization speed reduction will be applied.
- **Speed reduction:** This parameter allows you to specify a percentage by which the current feed-rate will be reduced during hole optimization, contributing to better hole cutting quality.
- **Arc on timeout:** This parameter defines the amount of time from issuing the torch-on command by the CNC to detecting the “Arc Ok” signal. Until the “Arc Ok” signal is detected, the CNC will not initiate the cut. This feature ensures that the cutting process starts only when the arc is stable and ready for cutting.

These THC settings are crucial for optimizing cutting quality, ensuring accurate torch height control, and enhancing the overall performance of the CNC plasma cutting system.

13.6.1 Advanced THC configuration

This section provides advanced customization options for the THC (Torch Height Control). If you are initially setting up the machine, it is advisable to keep these parameters as they are, as they have been

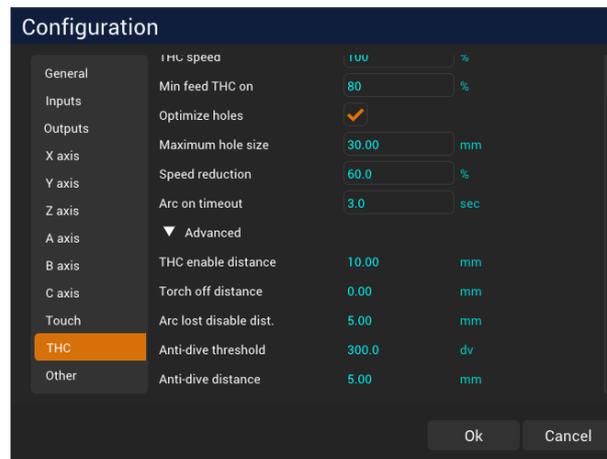


Figure 29: Advanced THC Configuration Dialog

selected for optimal performance.

- **THC enable distance:** This parameter determines the distance the torch must travel through a cut before the THC is enabled. This allows the torch to gain speed and stabilizes the voltage before using it to control the torch height.
- **Torch off distance:** This setting allows you to specify a predetermined distance before the end of the cut at which the torch will be switched off. Typically, this parameter is set to zero.
- **Arc lost disable dist.:** You can use this parameter to disable the arc lost signal a certain distance before the end of the cut. This precaution eliminates the possibility of an arc loss occurring right at the end of the cut. It's particularly useful in situations where the part being cut falls off the metal sheet, causing the plasma source to interrupt the arc, which can trigger an "Arc Lost" alarm.
- **Anti-dive threshold:** This is an advanced parameter, and it is recommended to leave it at the current default value of 300. It likely won't require adjustment under normal operating conditions.
- **Anti-dive distance:** Another advanced parameter that may require specific tuning based on your machine's characteristics and cutting requirements. Detailed adjustments should be made by experienced operators or technicians familiar with the system.

These advanced THC configuration options provide fine-tuning capabilities for optimizing the behavior of the torch height control system to suit specific cutting scenarios. If you're new to the machine, it's generally best to retain the default settings for optimal performance.

13.7 Other

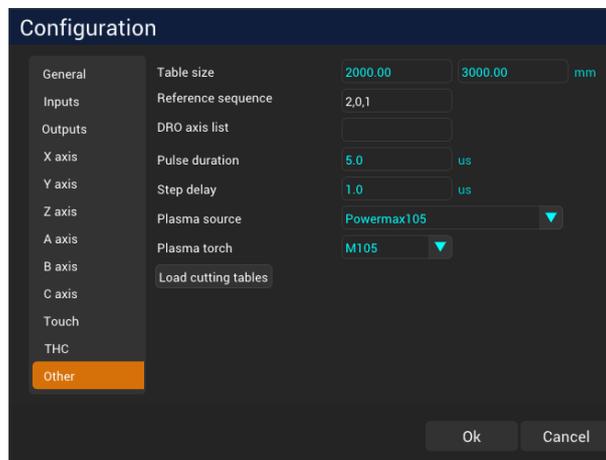


Figure 30: Other Configuration Dialog

- **Table size:** Specifies the dimensions of the CNC table. This setting determines the available workspace for your CNC machine.
- **Reference sequence:** During a referencing sequence, you can define the order in which the axes are sequenced. The default value is “2,0,1,” which means that the Z-axis is referenced first, followed by the X-axis, and then the Y-axis. You can customize the sequence by providing a comma-separated list of axis numbers.
- **DRO axis list:** This option allows you to customize the axes displayed in the Digital Readout (DRO). By default, the DRO displays the X and Y axes. If you want to include the Z-axis as well, simply specify “0,1,2” in this field.
- **Pulse duration:** Specifies the width or duration of the step pulse in microseconds. This value should be compatible with the specifications of your stepper or servo motor driver.
- **Step delay:** This parameter represents the delay between the change of the direction signal and the next step pulse. It’s also a parameter specified in your stepper or servo driver’s specifications.
- **Plasma source:** Here, you can select the cutting parameters tables that correspond to your specific plasma source model and torch model. This ensures that the CNC machine uses the appropriate cutting settings.
- **Plasma torch:** Specify the model of your plasma torch in this field.
- **Load cutting tables button:** Clicking this button will load the CNC plasma cutting tables based on the plasma source model and torch model you’ve specified above. This step is essential for

configuring the machine with the correct cutting parameters.

These CNC configuration settings are crucial for tailoring the machine's behavior to your specific equipment and requirements. Properly configuring these parameters ensures accurate operation and optimal performance during cutting tasks.

14 Shortcut Keys

The following short cut keys are available:

- **F1:** Help
- **F2:** Start
- **Ctrl + F2:** Start program options menu
- **F3:** Stop
- **F4:** Torch On/Off
- **F5:** Move to origin
- **F6:** Set part origin
- **F7:** Align
- **F8:** Load a program
- **F9:** Reference the cutting table
- **F10:** Reset
- **F12:** Set plate height for oxy-fuel cutting
- **Ctrl + F6:** Set part origin X value
- **Shift + F6:** Set part origin Y value
- **Right arrow:** Jog in X+ direction
- **Left arrow:** Jog in X- direction
- **Up arrow:** Jog in Y+ direction
- **Down arrow:** Jog in Y- direction
- **Page Up:** Jog in Z+ direction
- **Page Down:** Jog in Z- direction
- **Ctrl + Right arrow:** Profile jogging +
- **Ctrl + Left arrow:** Profile jogging -
- **Keypad Plus:** Increment cutting voltage by 0.5 volts
- **Keypad Minus:** Decrement cutting voltage by 0.5 volts
- **Ctrl + Keypad Plus:** Increase voltage by 1 volt
- **Ctrl + Keypad Minus:** Decrease voltage by 1 volt
- **Keypad Multiply:** Increase jog speed
- **Keypad Divide:** Decrease jog speed

- **E:** Toggle edit mode
- **o:** Set part origin to selected part's center
- **O:** Set part origin to selected part's lower-left position
- **1:** Display parts
- **2:** Display motion path
- **3:** Display the cutting table

15 Troubleshooting

This section provides a list of common issues that you may encounter while using your plasma cutting CNC system, along with possible solutions to address them.

15.1 Torch Collides with the Table

Torch keeps moving down until it collides with the table.

15.1.1 Possible Causes and Solutions:

1. Wrong Cutting Voltage

- **Cause:** The specified cutting voltage is set too low. The torch height control system attempts to lower the current voltage to reach the specified cutting voltage, which might be unrealistically low.
- **Solution:** Adjust the cutting voltage settings to match the requirements of the material being cut. Ensure that the specified voltage is within a feasible range for the plasma cutting process.

2. Improper Cutting Height Specification

- **Cause:** The cutting height parameter is incorrectly set too low in the CNC software. The system believes the torch should be closer to the material than it should, causing it to descend too far.
- **Solution:** Review and adjust the cutting height parameter in the CNC software to accurately reflect the material thickness and cutting requirements. This will help maintain an appropriate distance between the torch and the material.

3. Incorrect Touch Sequence

- **Cause:** The touch sequence, which involves the torch making initial contact with the material before cutting, might not be executed correctly. This could lead to inaccurate height measurements and improper torch positioning during cutting.
- **Solution:** Verify that the touch sequence is properly configured in the CNC software. Ensure that the torch makes accurate initial contact with the material surface before commencing the cutting operation.

4. Z Axis Reference Value Error

- **Cause:** The Z axis reference value, which establishes the home position of the Z axis, might be improperly set. This can cause the system to misinterpret the torch's actual position.
- **Solution:** Recheck and recalibrate the Z axis reference value to accurately correspond to the actual home position of the Z axis. This will prevent the system from assuming incorrect positions during cutting.

5. Wiring Issues Impacting Voltage Reporting

- **Cause:** Problems with wiring, connectors, or sensors can prevent the actual cutting voltage from being accurately reported to the motion controller. This can lead to incorrect decisions being made by the torch height control system.
- **Solution:** Thoroughly inspect all wiring connections, sensors, and connectors related to the torch height control system. Ensure that they are securely connected and functioning as intended.

15.2 Torch Keeps Moving Up

An issue where the torch continues to move upward until a limit switch is triggered.

1. Excessive Cutting Voltage

- **Cause:** Setting the cutting voltage too high can prompt the torch height control system to raise the voltage beyond the required level, resulting in unintended upward torch movement.
- **Solution:** Carefully adjust the cutting voltage settings to align with the material's specifications. This prevents unnecessary upward motion of the torch.

2. Incorrect Z Axis Reference

- **Cause:** A misconfigured Z axis reference can lead to inaccurate position detection. This misinterpretation can cause the torch to ascend beyond the intended height.
- **Solution:** Refer to the section on Z axis reference and re-calibrate the reference value to ensure precise positioning.

3. Incorrect Touch Sequence

- **Cause:** If the initial touch sequence isn't executed correctly, the system might inaccurately measure the material height. This miscalculation can trigger the torch to move excessively upward.
- **Solution:** Review the touch sequence configuration, following the guidelines from the previous section, to guarantee accurate execution.

15.3 Torch Not Firing or Cutting Properly

Issue: The plasma torch fails to ignite or doesn't cut material effectively.

15.3.1 Possible Solutions:

1. Ensure that the torch leads and connections are securely attached.
2. Check the gas supply and pressure to ensure sufficient flow for ignition.
3. Verify the cleanliness of the torch consumables and replace them if necessary.
4. Check the torch height control settings to ensure proper distance from the material.
5. Inspect the ignition system for any loose wires or damaged components.
6. Adjust the cutting parameters in the software to match the material and thickness.

15.4 Poor Cut Quality

Issue: The cut edges are uneven, with excessive dross or slag.

15.4.1 Possible Solutions:

1. Check the torch consumables for wear or damage, and replace them if needed.
2. Ensure that the correct cutting parameters are configured for the material thickness.
3. Verify the torch height control settings to maintain consistent distance from the material.
4. Check the gas pressure and flow rate to ensure proper cut quality.
5. Adjust the pierce delay and cut speed settings for optimal results.
6. Inspect the torch alignment to ensure it's perpendicular to the material.

15.5 Lost Motion or Inaccurate Cuts

Issue: The CNC system exhibits lost motion or produces inaccurate cuts.

15.5.1 Possible Solutions:

1. Verify that all axis motors are functioning correctly and no belts or gears are loose.
2. Check for any mechanical obstructions or debris that may interfere with motion.
3. Ensure the axis calibration settings are accurate and properly configured.
4. Inspect the lead screws or ball screws for wear and lubricate them if necessary.
5. Verify the stepper motor drivers or servo drives for proper voltage and current settings.
6. Calibrate the axis steps per unit (steps/mm) settings in the software.

15.6 Excessive Arc Lost Errors

Issue: Frequent arc lost errors during cutting operations.

Possible Solutions:

1. Check the integrity of the torch leads and connections.
2. Ensure proper grounding of the cutting table and components.
3. Verify that the plasma source is receiving stable power and voltage.
4. Inspect the torch consumables for damage or contamination.
5. Adjust the arc voltage settings to match the material and cutting conditions.
6. Check for interference from nearby electromagnetic sources.

Remember that troubleshooting can be complex and may require careful observation and testing. Always follow safety precautions and consult your CNC system's manual and technical support resources for detailed guidance.

Note: This troubleshooting section aims to provide general guidance for addressing common issues. For specific problems or complex scenarios, always consult your CNC system's documentation and seek professional assistance if needed.

16 Frequently Asked Questions

16.1 How to Move the Part Origin to the Center of a Part

To relocate the origin of a part to its center, follow these steps:

1. **Select the Part:** Highlight the part you want to work with.
2. **Press the “o” Key:** Once the desired part is selected, press the “o” key on the keyboard. This action will adjust the origin to the center of the chosen part.

Additionally, if you prefer to position the origin at the lower-left corner of the part, you can follow this alternative step:

3. **Press the “O” Key:** Instead of pressing “o,” press the “O” key. This will shift the origin to the lower-left corner of the selected part.

By using these key commands, you can effortlessly modify the origin of the part to align with either its center or lower-left corner, according to your requirements.

17 Glossary of Plasma and Oxy-Fuel Cutting Terms

Plasma Cutting: A metal cutting process that uses a high-velocity jet of ionized gas (plasma) to melt and remove material. Commonly used for metals and other conductive materials.

Torch: The handheld or mechanized device that holds and directs the plasma jet during cutting.

Arc: A discharge of electrical energy between two electrodes, producing intense heat to ionize the gas and create plasma.

CNC (Computer Numerical Control): A system that uses computerized controls and programming to automate machinery, including plasma cutters, for precise and repeatable cutting movements.

Consumables: Components of the plasma torch that wear out during cutting, including electrodes, nozzles, and shields.

Nozzle: A component at the end of the torch that focuses the plasma jet and provides cooling for efficient cutting.

Electrode: The component that creates the electrical arc necessary to produce plasma.

Cutting Speed: The rate at which the plasma torch travels along the material, determining the speed of the cutting process.

Kerf: The width of the material removed by the plasma jet during cutting.

Heat-Affected Zone (HAZ): The portion of the material that experiences thermal changes due to cutting, affecting its properties.

Piercing: The initial penetration of the material by the plasma jet, typically performed before cutting begins.

Kerf Width: The width of the cut produced by the plasma jet, which may vary depending on material thickness and cutting speed.

Backsplash: Molten metal that can splash back onto the workpiece during cutting, affecting cut quality.

Plasma Arc: The electrically conductive channel formed when plasma is generated, carrying electrical current.

Cutting Table: The surface on which the material to be cut is placed. It may include features like water or fume extraction systems.

Torch Height Control: A system that maintains a consistent distance between the torch and the material during cutting for optimal results.

Oxy-fuel Cutting: A cutting process that uses a flame generated by the reaction of oxygen with a fuel gas, suitable for thicker materials.

Kerf Angle: The angle of the cut edge of the material resulting from the plasma jet's movement.

Dross: Molten metal that solidifies along the edge of the cut, often requiring post-cutting cleanup.

18 Appendices

- **Technical Support:** Contact information. If you have problems or any questions in the installation or operation of this product, please contact NevexTech's Technical assistance department by email support@nxplasma.com

18.1 User License Agreement

Software and Hardware User License Agreement

This Software User Manual is provided by NevexTech Ltda ("Licensor") for use with the NX-Plasma CNC ("Software") and NX-Plasma CNC Controller ("Hardware"). By using the Software or Hardware, you ("User") implicitly agree to the following terms and conditions:

1. Grant of License

Subject to the terms and conditions of this Agreement, Licensor hereby grants User a non-exclusive, non-transferable, limited license to use the Manual for personal or internal business purposes only.

2. Restrictions

User agrees not to:

a. Modify, adapt, translate, reverse engineer, decompile, or disassemble the Software, Hardware or Manual. b. Reproduce, distribute, or sublicense the Manual to any third party. c. Remove or alter any copyright or proprietary notices contained in the Software or Manual. d. Use the Manual for any unlawful or prohibited purpose.

3. Ownership

User acknowledges that Licensor owns all right, title, and interest in and to the Software, Hardware or Manual, including all intellectual property rights.

4. Termination

Licensor may terminate this Agreement at any time if User breaches any of its terms. Upon termination, User must cease using the Manual and destroy all copies.

5. Warranty Disclaimer

THE MANUAL IS PROVIDED "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NONINFRINGEMENT.

6. Limitation of Liability

IN NO EVENT SHALL LICENSOR BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE MANUAL, EVEN IF LICENSOR HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

7. General

a. This Agreement constitutes the entire agreement between the parties concerning the subject matter hereof. b. This Agreement may not be amended except in writing and signed by both parties. c. This Agreement shall be governed by and construed in accordance with the laws of [Your Jurisdiction]. d. Any disputes arising under or in connection with this Agreement shall be subject to the exclusive jurisdiction of the courts of [Your Jurisdiction].

By using the Software, Hardware and accessing this Manual, User implicitly agrees to be bound by the terms and conditions of this Agreement.

Thank you for using the CNC Software User Manual. For further assistance, contact our technical support team at support@nevextech.com.