NX-Plasma Software Manual - Version 2026.4

NevexTech LTD

Contents

1 Legal Notice **6**
1.1 **IMPORTANT – PLEASE READ BEFORE USE** . . . 6
1.2 User Responsibility . . . 6
1.3 Required Technical Expertise . . . 6
1.4 Limitation of Liability . . . 7
1.5 Copyright and Rights . . . 7
1.6 Support and Contact . . . 7

2 NX-Plasma Software User Manual **8**
2.1 Introduction . . . 8
2.2 Key Features of NX-Plasma . . . 8
2.3 Getting Started - Installation and Setup . . . 10
2.3.1 Hardware Requirements . . . 10
2.3.2 Installation Guide . . . 11

3 Configuring Your Machine **13**
3.1 Setup and Configuration . . . 13
3.2 Axes configuration . . . 13
3.3 References Configuration . . . 14
3.3.1 Initial Referencing Procedure . . . 14
3.3.2 Importance of Correct Z Reference Settings . . . 14
3.3.3 Z Reference Validation Test . . . 14
3.4 Plasma Source Configuration . . . 15
3.4.1 Start Plasma Contacts . . . 15
3.4.2 Arc OK Signal . . . 15
3.4.3 Divided Arc Voltage . . . 15
3.5 CNC Workflow Enforcement . . . 16

4 Starting the NX-Plasma CNC Software **17**
4.1 Enabling the System . . . 18
4.1.1 Initial Status . . . 18
4.1.2 Connect to Motion Controller . . . 18
4.1.3 Bring Online . . . 18
4.2 Referencing the Machine . . . 18
4.2.1 Power-Up Reference . . . 18
4.2.2 Start Referencing . . . 18

4.3 Loading a Program . . . 19
4.3.1 Load Program . . . 19
4.3.2 Set Origin . . . 19
4.4 Jogging . . . 19
4.4.1 Jog Speed Control . . . 19
4.4.2 Full-Speed Jogging . . . 19
4.4.3 Jog Control . . . 20
4.4.4 Program Jogging . . . 21
4.5 Setting the Cutting Parameters . . . 21
4.6 Starting Cutting . . . 21

5 Main Screen Overview **22**
5.1 Navigating the User Interface . . . 23
5.1.1 General Interface Interaction . . . 23
5.1.2 Dual Operating Modes: CNC Mode vs. Editing Mode . . . 24
5.2 Toolbar . . . 24
5.2.1 Toolbar Buttons . . . 24
5.2.2 Switches . . . 25
5.2.3 Status Indicators . . . 25
5.3 Command Buttons . . . 25
5.4 Indicator Gauges . . . 26
5.5 Jog Control . . . 26
5.6 Coordinates (Digital Readout Position Indicator) . . . 26
5.7 The Z Axis Position Scale . . . 26
5.8 Status and Message Line . . . 27
5.8.1 States . . . 27
5.9 Menu Button . . . 28
5.9.1 Save and File Operations . . . 28
5.9.2 Save and Restore Work . . . 28
5.9.3 Statistics . . . 29
5.9.4 Diagnostics . . . 29
5.9.5 About NX-Plasma . . . 29
5.9.6 Help . . . 29
5.9.7 Exit . . . 29

6 Command Buttons **30**
6.1 Start Command Button . . . 30
6.1.1 Accessing Start Options . . . 30

6.1.2 Key Difference from Other CNC Systems . . . 31
6.1.3 Cutting Only Selected Profiles . . . 32
6.1.4 Example Workflow . . . 32
6.2 Stop Command Button . . . 32
6.3 Manual Command Button . . . 33
6.4 Origin Command Button . . . 33
6.4.1 Set Origin Command Button . . . 33
6.5 Align Command Button . . . 34
6.5.1 How to Use the Align Function . . . 34
6.5.2 Canceling Alignment Compensation . . . 35
6.6 Load Program Command Button . . . 35
6.6.1 G-Code Import Support . . . 36
6.7 Reference Command Button . . . 37
6.8 Connect / Reset Command Button . . . 38
6.8.1 Connect . . . 38
6.8.2 Reset . . . 38
6.8.3 Collision Detection Behavior . . . 38

7 Cutting Parameters Dialog 39

8 Part Wizard Dialog 42

9 Editing Mode Controls 44
9.1 Activating Editing Mode . . . 44
9.2 Mode Restrictions . . . 44
9.3 Mouse Controls in Editing Mode . . . 45
9.4 Editing mode Toolbar . . . 45
9.5 Selecting and Deselecting Parts in Editing Mode . . . 46
9.5.1 Basic Selection . . . 46
9.5.2 Multi-Select (Add to Selection) . . . 47
9.5.3 Selection Window . . . 47
9.5.4 Modifier Keys and Advanced Selection Logic . . . 47
9.6 Moving Parts . . . 47
9.6.1 How to Move Parts . . . 47
9.6.2 Collision Handling . . . 48
9.6.3 Shift-Key Move Strategy (Push Mode) . . . 48
9.6.4 Undo Support . . . 48
9.7 Changing the Part Cutting Order . . . 49
9.8 Editing Mode Shortcuts . . . 49

9.9 Nesting Tool for Layout Optimization . . . 50
9.10 Leads tool . . . 51
9.11 Options Dialog . . . 52
9.11.1 Display Settings . . . 52
9.11.2 Grid Configuration . . . 52
9.11.3 Behavior Settings . . . 53
9.11.4 Layout and Nesting Settings . . . 53
9.11.5 Unit settings . . . 53

10 Configuration Dialog **54**
10.1 Dual-Mode Configuration: Plasma and Oxy-fuel . . . 54
10.2 Dialog Structure . . . 55
10.2.1 General Configuration . . . 55
10.2.2 Inputs Configuration . . . 57
10.2.3 Outputs Configuration . . . 60
10.2.4 Axes Configuration . . . 63
10.2.5 Touch Configuration . . . 66
10.2.6 THC Configuration . . . 67
10.2.7 Encoders Configuration . . . 71
10.2.8 Other Configuration . . . 73

11 Shortcut Keys **75**

12 Troubleshooting **76**
12.1 Torch Collides with the Table . . . 76
12.2 Torch Keeps Moving Up . . . 77
12.3 Torch Not Firing or Cutting Properly . . . 78
12.4 Poor Cut Quality . . . 78
12.5 Lost Motion or Inaccurate Cuts . . . 78
12.6 Excessive Arc Lost Errors . . . 79

13 Frequently Asked Questions **80**
13.1 What is the `.nx` file format? . . . 80
13.2 Why save and load `.nx` files? . . . 80
13.3 Why Is It Important to Reference the Z Axis? . . . 81
13.4 Why Is It Important to Use a Touch Sensor? . . . 81
13.5 Common Issues and Recommendations . . . 81
13.6 How do I set cutting parameters for different materials and thicknesses? . . . 82
13.7 How do I enter Editing Mode and rearrange parts? . . . 82

13.8 How does the Part Wizard work for creating quick parts? . . . 82
13.9 How to Move the Part Origin to the Center of a Part . . . 82
13.10 Why Is the **Start** Button Disabled, and How Do I Fix It? . . . 83
13.11 How to Enable the Start Button . . . 83

14 Glossary of Plasma and Oxy-Fuel Cutting Terms **84**

15 Appendices **86**

16 SOFTWARE LICENSE AGREEMENT **87**
16.1 DEFINITIONS . . . 87
16.2 LICENSE GRANT . . . 87
16.3 INTELLECTUAL PROPERTY RIGHTS . . . 88
16.4 UPDATES AND SUPPORT . . . 88
16.5 PROHIBITED ACTIONS . . . 89
16.6 TERM AND TERMINATION . . . 89
16.7 WARRANTIES AND DISCLAIMERS . . . 89
16.8 LIMITATION OF LIABILITY . . . 89
16.9 GOVERNING LAW AND JURISDICTION . . . 90
16.10 MISCELLANEOUS . . . 90
16.11 AGREEMENT ACCEPTANCE . . . 90

1 Legal Notice

1.1 IMPORTANT – PLEASE READ BEFORE USE

The use of CNC (Computer Numerical Control) machines, including the NX-PLASMA system, involves **inherent risks**. By operating this equipment, you acknowledge and accept these risks and agree to the terms outlined below.

Nevex Tecnologia LTDA ("Nevex") shall **not be held liable** for any injury, damage, or loss resulting from the installation, use, or misuse of this device.

1.2 User Responsibility

As the operator of the NX-PLASMA device, **you are solely responsible** for:

- Ensuring the machine is operated **safely and correctly** at all times.
- Complying with **all applicable local, regional, and national laws and safety regulations**.
- Implementing **adequate safety precautions** to protect yourself, bystanders, and property.

1.3 Required Technical Expertise

The installation, operation, and maintenance of the NX-PLASMA system require **technical knowledge** in:

- Electricity and electrical systems.
- Electronics and wiring safety.
- CNC machinery and motion control principles.

If you do not possess sufficient expertise, **Nevex seriously recommends** consulting a qualified technician. The installation or operation of the equipment in an incorrect manner may lead to **malfunction, property damage, or serious personal injury**.

1.4 Limitation of Liability

Nevex does not accept liability for:

- Damage to the NX-PLASMA device, external equipment, or property.
- Injuries or harm to the operator or third parties.
- Operational disruptions, losses, or downtime caused by equipment malfunction or misuse.

IMPORTANT – USE THIS EQUIPMENT AT YOUR OWN RISK.

1.5 Copyright and Rights

1.6 Support and Contact

Please ensure you follow all safety guidelines outlined in this manual. **Your safety—and the safety of others—depends on proper use and understanding of this equipment.**

For questions, technical support, or authorized service, contact: support@nxplasma.com

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.

2.2 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 to define part geometry, and easily add lead-ins and lead-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.

2.3 Getting Started - 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.

2.3.1 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 64GB of Disk Space**
- **Mouse and Keyboard**
- **Windows 10 or 11 Operating System**
- **Video Card Supporting OpenGL 3.3 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.

2.3.2 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.

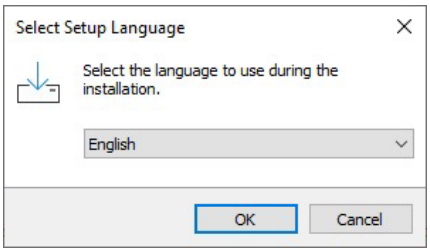

Figure 1: Select Installation Language

4. **Choose Installation Directory:** Utilize the browse button to designate the installation folder of your choice, then proceed by clicking the "Next" button.

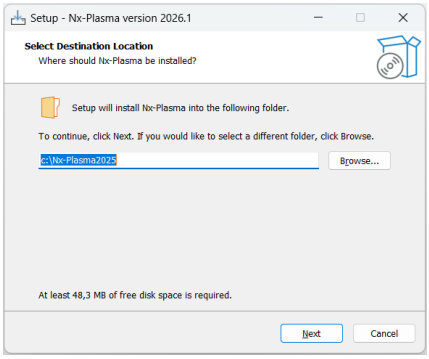

Figure 2: Select Installation Folder

5. **Confirm Installation:** Confirm your chosen installation folder by clicking the “Install” button.

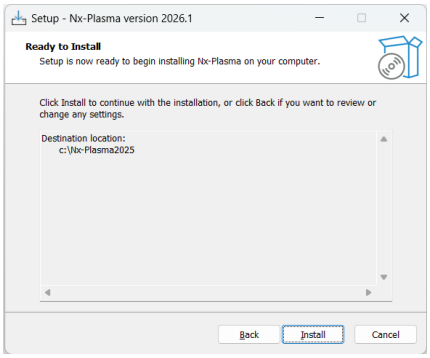

Figure 3: Confirm Installation Folder

6. **Completion and Finish:** Finalize the installation process by clicking the “Finish” button.

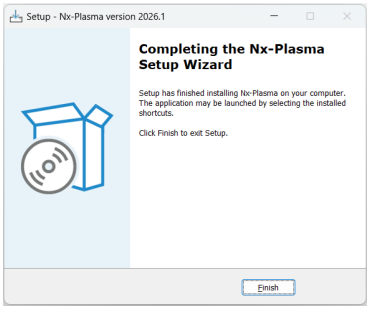

Figure 4: Installation Complete

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.

3.2 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.

3.3 References Configuration

Professional plasma cutting tables are typically equipped with reference sensors (limit switches) to detect the home position of each axis. The NX-PLASMA system supports referencing for all axes; however, **referencing the Z axis is essential** for proper operation. A reference switch must be installed at the top of the Z-axis travel to provide a consistent and reliable reference point for vertical positioning.

3.3.1 Initial Referencing Procedure

Upon powering up the machine, the **Z axis must be referenced** before any cutting operations can begin. This is done using the **Reference Z command**, which moves the Z axis upward until the **Z++ reference switch** is triggered. When the switch is activated:

- The motion stops.
- The system assigns the current position as the **"Z Reference Position"**, as defined in the Z-axis configuration settings.

This procedure aligns the machine's internal position with its physical travel limits.

To determine the correct value for the "Z Reference Position," first execute the reference operation as described. Then, **physically measure** the distance between the **torch tip** and the **tabletop** (sacrificial plate). This measured value should be entered into the Z-axis configuration as the Z reference position.

3.3.2 Importance of Correct Z Reference Settings

Incorrect Z reference configuration will lead to **erroneous torch height control**, causing unreliable cutting or potential damage. Since all automatic torch height adjustments depend on an accurate Z reference, this setup step is critical.

3.3.3 Z Reference Validation Test

To verify the Z axis has been correctly referenced:

1. Perform a **Z axis reference operation**.
2. Use a ruler or gauge to **measure the vertical distance** from the torch tip to the top surface of the cutting table.

3. On the software interface, check the **Z value** displayed at the **bottom of the Z scale** on the right-hand side of the screen.
4. Confirm that the **displayed value matches the physical measurement**.
5. If the values match, your system is correctly configured.
6. If the values **do not match**, do the following:
    - Review and correct the **"Z Reference Position"** in the Z-axis configuration dialog.
    - Double-check the **steps per mm** configured for the Z axis to ensure proper motion scaling.

3.4 Plasma Source Configuration

Plasma cutting tables can be equipped with a variety of plasma sources, but most share a **standard signal interface** composed of three essential connections:

3.4.1 Start Plasma Contacts

This consists of **two wires** that, when short-circuited, **trigger the plasma torch to fire**. These contacts are controlled by the CNC system to start and stop the arc.

3.4.2 Arc OK Signal

Also using **two wires**, this signal is typically shorted by an internal relay inside the plasma power source once it detects that the arc has successfully transferred to the material.

- When the **Arc OK** signal is detected, the CNC controller is authorized to begin cutting motion.
- If the arc is unexpectedly extinguished (e.g., due to material gap or torch fault), the signal is lost, prompting the system to **immediately stop motion and raise an error**.
- It is strongly recommended to keep this signal **enabled** in the configuration. Doing so ensures the CNC will only move once the arc is confirmed as stable.

3.4.3 Divided Arc Voltage

The **tip voltage** of the plasma arc is made available to the CNC controller through a **voltage divider circuit**, typically built into the plasma source.

- The NX-PLASMA system expects a division ratio of **50:1**.
- It is **critical** that the plasma source is equipped with a compatible **voltage divider**.

- **Never connect full arc voltage directly** to the controller inputs—doing so can result in **severe hardware damage**.

If the plasma source uses standard signal outputs and wiring, the **default input configuration** of the controller should work without modification. However, if **custom wiring** or **non-standard input pins** are used, adjustments must be made in the **NX-Plasma software configuration**.

3.5 CNC Workflow Enforcement

To help prevent operator mistakes, the NX-PLASMA software **enforces a structured workflow**. For example:

- If the machine has **not been referenced**, or
- If the **part origin (zero point)** has not been set,

then the **“Start” button will remain disabled**, and the system status will show **“Not Referenced”**. This safeguard ensures the machine is always in a known and safe state before beginning any cutting operation.

4 Starting the NX-Plasma CNC Software

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

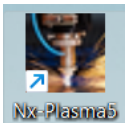

Figure 5: NX-Plasma Icon

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: NX-Plasma Main Screen

> **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

4.1.1 Initial Status

Upon software startup, the system displays a “DISCONNECTED” status, indicating the control software has not connected to the motion controller.

4.1.2 Connect to Motion Controller

Establish the connection by clicking the “Connect” command button. This initiates communication between the CNC software and the motion controller. Initially, the status will change to “Offline” as a safety measure, preventing motion or plasma torch activation.

4.1.3 Bring Online

To enable the system, select the “Online” option button. The status changes to “Not Referenced,” indicating the need for machine referencing.

4.2 Referencing the Machine

4.2.1 Power-Up Reference

When powering up the CNC system, axes positions are unknown. Reference operations move the axes to known positions. Initiate referencing by clicking the “Reference” command button. A reference dialog appears for axis selection.

4.2.2 Start Referencing

Within the reference dialog, click “Start” to begin referencing. After completion, the dialog automatically closes.

4.3 Loading a Program

4.3.1 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.

4.3.2 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, you can use either the **Jog Control interface** or the **keyboard**:

- **X Axis** – Left and Right arrow keys
- **Y Axis** – Up and Down arrow keys
- **Z Axis** – Page Up and Page Down keys
- **A Axis** – Home and End keys
- **B Axis** – Insert and Delete keys

4.4.1 Jog Speed Control

You can **adjust the jogging speed** using the **Jog Control** (see next section). Clicking the **center of the Jog Control dial** increases the jog speed in **10% increments**, wrapping back to **10% after reaching 100%**. This sets the jogging speed as a percentage of the **maximum jog speed** configured in the system.

4.4.2 Full-Speed Jogging

To **jog at 100% speed instantly**, **hold the SHIFT key** while pressing the corresponding jog direction keys (e.g., arrow keys, PageUp/PageDown, etc.). This allows quick full-speed movement without adjusting the jog percentage manually.

4.4.3 Jog Control

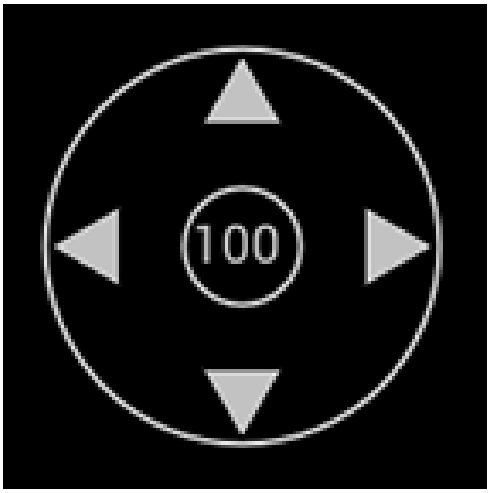

Figure 7: Jog Control

The **Jog Control** allows you to manually move the machine or adjust the current jog speed using the mouse or keyboard.

4.4.3.1 Outer Circle – Directional Movement Clicking and holding the mouse **anywhere along the outer circle** will move the machine in the corresponding direction. The movement direction is based on where you click along the circle's circumference.

4.4.3.2 Inner Circle – Speed Adjustment Clicking **inside the inner circle** increases the jog speed by **10% per click**. After reaching 100%, the speed wraps back to 10%.

4.4.3.3 Keyboard Shortcuts for Speed Control You can also adjust the jog speed using the keyboard:

- Press **'/'** to decrease jog speed by 10%
- Press **'*'** to increase jog speed by 10%

The jog speed setting determines the percentage of the **maximum jog speed** used during manual movements.

4.4.4 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:

1. Use the **"Cutting Parameters Dialog"** to select the appropriate cutting parameters that will be applied during the cutting process.
2. 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

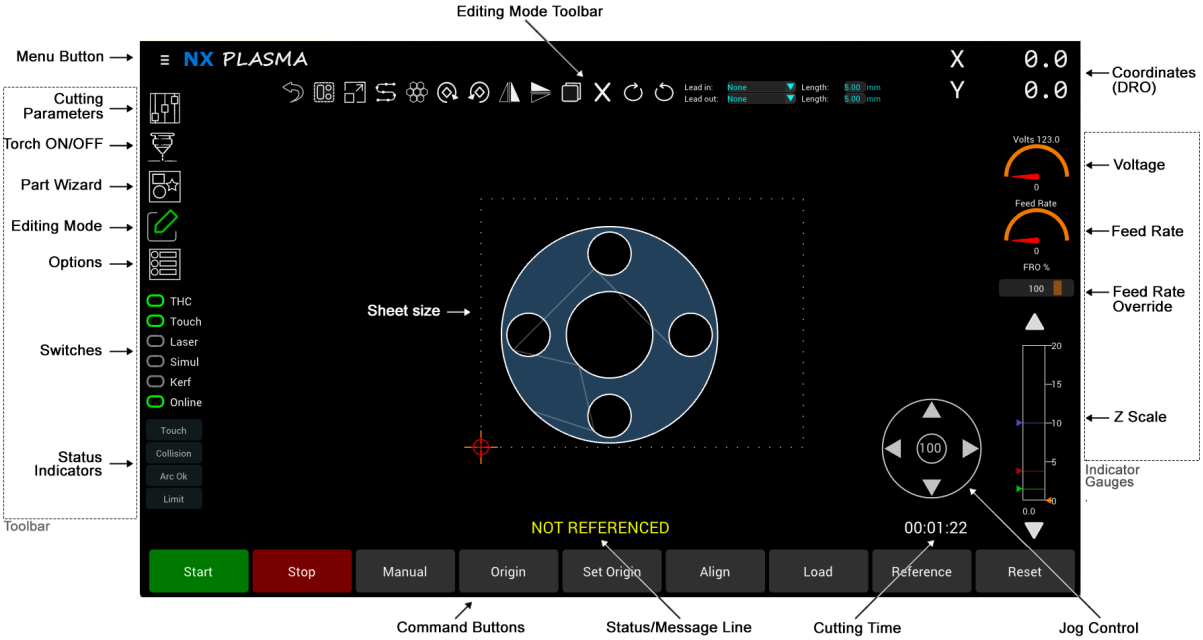

Figure 8: Main Screen

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:

- **Menu Button:** Access to file-related operations and software version information.
- **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.
- **Status and Message Line:** Updates, notifications, and warnings related to the machining process.
- **Cutting time:** Displays the estimated duration of active cutting operations.
- **Jog Control:** Incremental movement along different axes for precise adjustments.
- **Indicator Gauges:** Real-time data for monitoring plasma arc voltage, current feed rate and Z axis position.
- **Coordinates (DRO):** Accurate position information for maintaining machining location.

5.1 Navigating the User Interface

The NX-PLASMA software interface is designed to provide intuitive access to all CNC and editing operations. Below is a guide to navigating its key elements and understanding its dual-mode workflow.

5.1.1 General Interface Interaction

- **Mouse Navigation** Use the mouse to interact with **buttons**, **switches**, and **controls**. Click to open dialog boxes, toggle settings, or initiate actions.
- **Menu Button** Access file operations, machine configuration, and software version info.
- **Toolbar** Buttons are grouped by function—such as file operations, machine controls, and editing tools. Clicking a button may:
  - Open a dialog window.
  - Execute a specific action (e.g., start cut, set origin).
  - Toggle a parameter (e.g., enable/disable motors).
- **Status and Message Line** Continuously displays system messages, warnings, and notifications—keep an eye here for alerts and progress updates.
- **Jog Control** Enables precise manual movement of the torch. Use for positioning, alignment, or probing.
- **Indicator Gauges** Display key system values (e.g., feed rate, speed, voltage) for instant status feedback.
- **Digital Readout (DRO)** Displays **real-time machine position** for each axis.
- **Keyboard Shortcuts** Use hotkeys to speed up navigation and operation (see **Editing Mode Shortcuts**).

5.1.2 Dual Operating Modes: CNC Mode vs. Editing Mode

NX-PLASMA operates in **two distinct modes**, each with its own purpose and tools:

5.1.2.1 CNC Mode (Machine Control)

- Used for: **running cutting jobs**, jogging, probing, and interacting with the physical machine.
- All CNC operations are available, including torch control, referencing, and starting/stopping cuts.

5.1.2.2 Editing Mode (CAM/Layout Editor)

- Used for: **manipulating the layout of parts and profiles** to be cut.
- Allows on-machine editing, such as moving, rotating, deleting, or aligning parts—without needing external CAM software.
- Disables all machine motion controls for safety and focus.

> **Switching Modes**: Use the **Edit Mode icon** on the main toolbar or press **E** to toggle between CNC Mode and Editing Mode.

The current mode determines which controls are active and what interface tools are available.

5.2 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.

5.2.1 Toolbar Buttons

- **Cutting Parameters 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 display the status of the torch during cutting.
- **Part Wizard 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.
- **Options Button:** Display the options dialog allowing you to configure display preferences.

5.2.2 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.

5.2.3 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.

5.3 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 when it is not perfectly aligned with the cutting table, allowing the CNC to compensate for rotation before starting the cut.
- **Load Program Button:** Loads a CNC program.
- **Reference Button:** References the machine's axes.
- **Connect/Reset Button:** Connects or resets the motion controller.

5.4 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 (used mostly for oxyfuel cutting).
- **Z Position Indicator:** Display the current Z axis position.

5.5 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.

5.6 Coordinates (Digital Readout Position Indicator)

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

5.7 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.

5.8 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.

5.8.1 States

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

5.9 Menu Button

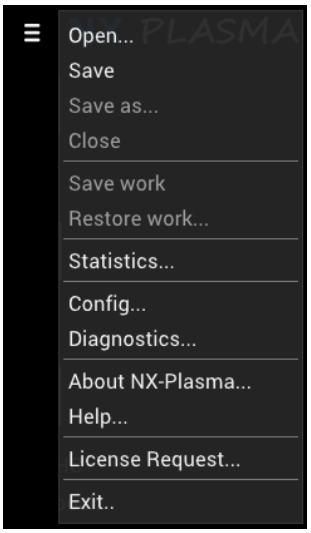

Figure 9: Menu Button

5.9.1 Save and File Operations

- **Open:** Open an existing cutting program in ".nx" format or import new parts from external files such as DXF or G-Code.
- **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.

5.9.2 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.

5.9.3 Statistics

Display cutting statistics

5.9.4 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.

5.9.5 About NX-Plasma

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

5.9.6 Help

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

5.9.7 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.

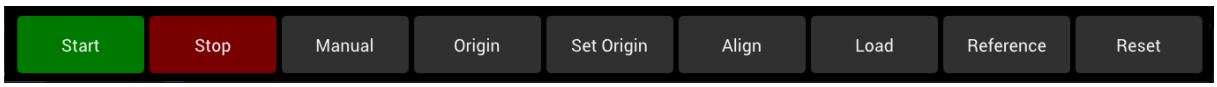

Figure 10: Command Buttons

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.

6.1 Start Command Button

The **Start** button begins the cutting process. Depending on the current system state, a context menu will appear with one or more available start options:

- Start from **Position of arc loss**
- Start from **Program start position**
- Start from **Closest position**
- Start from **Last stop position**

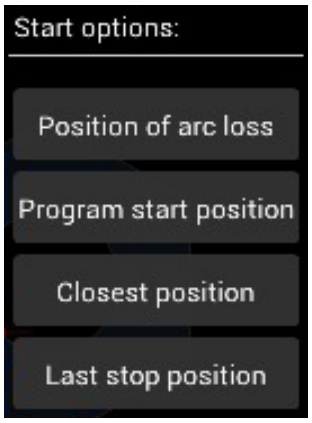

Figure 11: Start options menu

6.1.1 Accessing Start Options

To manually access these options, **hold the SHIFT key** while clicking the **Start** button. The button label will change to **“Start Options”**, and a menu will display the currently available choices:

- **Start from Program start position** Begins cutting from the start of the selected profile set.
- **Start from Closest position** Begins cutting from the point **on the profile** closest to the current torch location.
  - When you move the torch manually (e.g., using Jog), a **small green circle** appears, indicating the **nearest point on a selected profile**.
  - This is where the torch will move and start cutting if this option is chosen.
- **Start from Last stop position** Resumes cutting from the last position where the **Stop** button was pressed.
  - Marked by a **yellow circle**.
- **Start from Position of arc loss** If the system detected an arc loss during cutting, the exact position is stored and shown as a **red circle**.
  - You may inspect the area and resume cutting precisely from this saved location.

6.1.2 Key Difference from Other CNC Systems

> **IMPORTANT:** NX-Plasma only cuts from precise, valid positions *on the profile*—never from arbitrary torch positions.

Other systems may start cutting wherever the torch happens to be, even if misaligned, leading to ruined parts. NX-PLASMA avoids this by:

- Displaying a **visual marker** (green/yellow/red circle) for the calculated start position on the profile.
- **Automatically moving** the torch to that location before activating the arc.
- Ensuring all cuts begin **on a selected profile**, not just any geometry on screen.

This guarantees consistent, accurate cuts—even if the torch is only roughly positioned by the operator.

6.1.3 Cutting Only Selected Profiles

NX-Plasma will **only cut the profiles that are currently selected**.

- To **select or deselect** a profile, **hold the SHIFT key** and **click on the desired profile** in the layout view.
- Selected profiles are highlighted, and only these will be considered during the cut and when determining the nearest valid start point.

This gives you full control over which parts to cut in a given run and prevents unintentional cutting of nearby geometry.

6.1.4 Example Workflow

Want to start cutting from a specific place?

1. Use **SHIFT + click** to select the desired profile(s).
2. Jog the torch **near the area** where you want to start.
3. Press **Stop** to mark the approximate location (yellow circle).
4. Choose **"Start from last stop position."** The system will **automatically align and move** to the nearest valid start point **on the selected profile**, ensuring a clean, accurate cut.

6.2 Stop Command Button

The **Stop** button immediately halts any current machine movement, including cutting operations.

- When pressed during a cut, the system records the exact position where the torch was stopped.
- This position is visually marked with a **yellow circle** on the screen, placed **on the profile** at the closest valid point to the torch when the stop occurred.
- You can later resume cutting from this exact location using the **"Start from last stop position"** option (see **Start Command Button**).

This feature is useful for pausing the process to inspect, reposition, or perform minor corrections before resuming the job with precision.

6.3 Manual Command Button

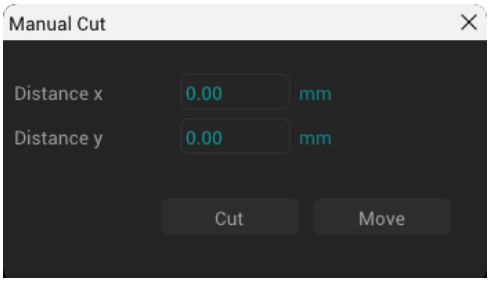

Figure 12: Manual Cut Dialog

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.

Here’s a refined and clearer version of your **Set Origin Command Button** section, with improved structure, consistent tone, and helpful detail:

6.4.1 Set Origin Command Button

Each time you load a new program or create a new part using the wizard, the CNC system requires you to define the **origin point** from which cutting operations will begin.

6.4.1.1 How to Set the Origin

1. **Position the torch** at the desired origin location on the cutting table.
2. Press the **"Set Origin"** button.

This operation assigns the current torch position as the **part origin** in both the **X and Y axes**. All cutting paths will now be referenced from this point, ensuring accurate and consistent part placement on the material.

6.4.1.2 Partial Origin Adjustment (X or Y Only) If you only want to adjust **one axis**, use the following keyboard shortcuts:

- **F6** – Set only the **X origin** to the current X position of the torch
- **SHIFT + F6** – Set only the **Y origin** to the current Y position

These options are useful when aligning a part along just one axis, such as when the sheet edge is skewed and you've used edge alignment.

6.5 Align Command Button

The **Align** button allows you to compensate for **sheet misalignment** by rotating the part layout to match the actual orientation of the material on the cutting table. This is especially useful when the metal sheet is not perfectly aligned with the machine's X/Y axes.

6.5.1 How to Use the Align Function

1. **Move the torch** to any point along the **edge of the metal sheet**.
2. Press the **Align** button.
3. Move the torch to **another point along the same edge**, at least **10 cm away** from the first point.
4. Press the **Align** button again.

After the second press, the system calculates the angle between the two points and automatically **rotates the part layout on screen** to match the detected edge angle. The CNC will now cut as if the sheet were properly aligned, without requiring physical repositioning of the material.

6.5.2 Canceling Alignment Compensation

To remove the alignment correction:

- **Hold the SHIFT key** – the **Align** button label will change to **“Cancel Align”**.
- Click it to restore the original orientation based on the machine’s axes.

You’re absolutely right—the original sentence blends two distinct actions: **loading** a native program versus **importing** external files. Here’s a revised and **clearly separated** version that reflects that distinction:

6.6 Load Program Command Button

The **Load** button allows you to bring files into the workspace in three ways:

1. **Open an existing NX-Plasma cutting program** (`.nx` files).
2. **Import part geometry** from DXF files.
3. **Import G-Code** files containing supported cutting instructions.

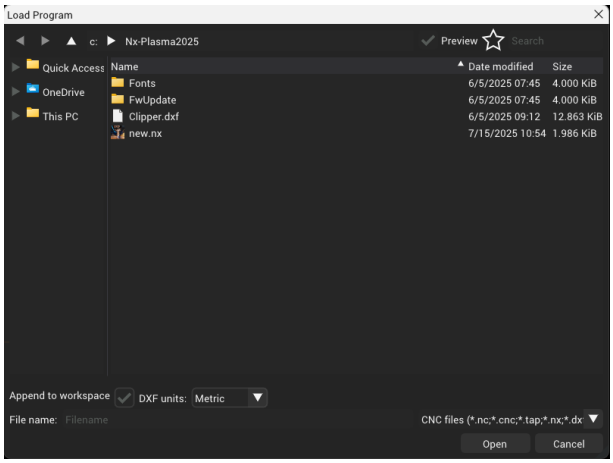

Figure 13: Load Program Dialog

When you click the **Load** button, a file selection dialog appears. You can choose from the following supported formats:

- **`.nx`** — NX-Plasma’s native format. Loads a complete cutting program, including parts, nesting layout, and machine settings.
- **`.dxf`** — Standard drawing file format used to import new part geometry.
- **`.nc`**, **`.tap`**, **`.cnc`** — G-Code formats. Only supported commands are recognized (see documentation for details).

> **Note:** NX-Plasma does **not** use G-Code or DXF files directly for cutting. Instead, when you load a G-Code or DXF file, the software **imports and converts** it into an internal **mathematical cutting model**. To preserve this model, you must **save the workspace** as a `.nx` file using the **Save** or **Save As** function. This is the correct workflow:
>
> 1. Import a DXF or G-Code file.
> 2. Save it as a `.nx` file to preserve the converted model.
> 3. Later, open the `.nx` file to resume or execute cutting, with all entries and settings ready.

When importing a **DXF file**, be sure to select the correct **unit system**—**Metric** or **Imperial**—from the dropdown menu. This ensures that the drawing is properly scaled.

Each part in the DXF file must have a **closed profile**—the start and end points must connect. If a profile is **not closed**, the software will **not recognize it as a part**. Instead, it will be treated as a regular cut path and displayed as an unfilled outline (just lines).

You can visually identify this in the workspace:

- **Valid parts** appear as **filled solids**.
- **Open profiles** appear as **simple outlines**, even if they include internal cutouts.

A common sign of an open profile is when **cutouts appear filled**, but the surrounding part does not.

> **Important:** DXF files should **not** contain lead-ins or lead-outs. A DXF file defines **what** part to cut — the **geometry** of the part — not **how** it should be cut. Lead-ins and lead-outs are added manually in NX-Plasma after importing the DXF.
>
> If you need to import parts that already include lead-ins and lead-outs (for example, from nesting or cutting software like **Pronest** or **SheetCam**), use a **G-Code file** (`.nc`, `.tap`, `.cnc`) instead. G-Code preserves the complete cutting instructions, including lead paths.

6.6.1 G-Code Import Support

The system supports G-Code files that follow a **minimal and compatible subset of commands**, as shown below:

G-Code	Meaning
G00	Rapid (non-cutting) movement
G01	Linear interpolation
G02	Clockwise circular interpolation

G-Code	Meaning
G03	Counter-clockwise circular interpolation
M03	Start cut (torch ON)
M05	End cut (torch OFF)

> **IMPORTANT:** G-Code files must include **only X and Y axis movements**. Do **not** include Z-axis commands. Z movement, pierce height, cutting height, and delay control are managed internally by NX-PLASMA. Including unsupported commands may result in **import failure** or unpredictable behavior.

6.7 Reference Command Button

When you first turn on the CNC, it does not know where the machine axes are positioned. To synchronize the positions, it is necessary to reference the axes. During referencing, each selected axis is moved until its reference switch is triggered, signaling to the CNC that the axis is now in a known position.

To perform this operation, click the **Reference button** on the command menu. This opens the Axes Reference Dialog:

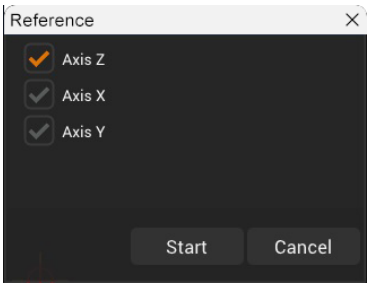

Figure 14: Reference Dialog

In the dialog, select the axes you wish to reference and click the Start button. The CNC will then move each selected axis to its reference position.

> **IMPORTANT:** The Z axis is mandatory. If the Z axis is not referenced, the CNC will not allow you to start cutting.

Referencing typically begins with the Z axis, followed by X and Y. This order can be changed via: **Configuration → Other → Reference Sequence**

Each axis that completes its movement and triggers the reference sensor is considered to be at its Reference Position, as configured in the system settings.

6.8 Connect / Reset Command Button

The **Connect** button is used to establish communication between the CNC software and the motion controller. Once a successful connection is made, the button changes its label to **Reset**.

6.8.1 Connect

- Clicking **Connect** initiates communication with the motion controller.
- This step is required before any machine operation can begin.

6.8.2 Reset

- When already connected, the button becomes **Reset**, which is used to **clear alarms and reinitialize the controller** after fault conditions.
- Typical situations requiring a reset include:
  - **Collision detected**
  - **Arc not detected**
  - Other controller-generated alarms

Resetting serves two purposes:

1. **It clears the alarm state** and returns the system to an operational state.
2. **It acknowledges the fault**, allowing the operator to take corrective action.

6.8.3 Collision Detection Behavior

In the case of a **collision detection alarm**, the Reset button provides additional behavior:

- After pressing **Reset**, the system **temporarily ignores the collision input signal for 15 seconds**.
- This allows you to move the machine (e.g., jog it away from the obstacle) without being immediately interrupted by the active collision input.
- After the 15-second bypass period, if the collision input is **still active**, the system will automatically **re-enter the alarm state** and notify the operator.

This mechanism ensures both safety and flexibility—allowing the operator to recover from collisions while still enforcing protection afterward.

7 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.

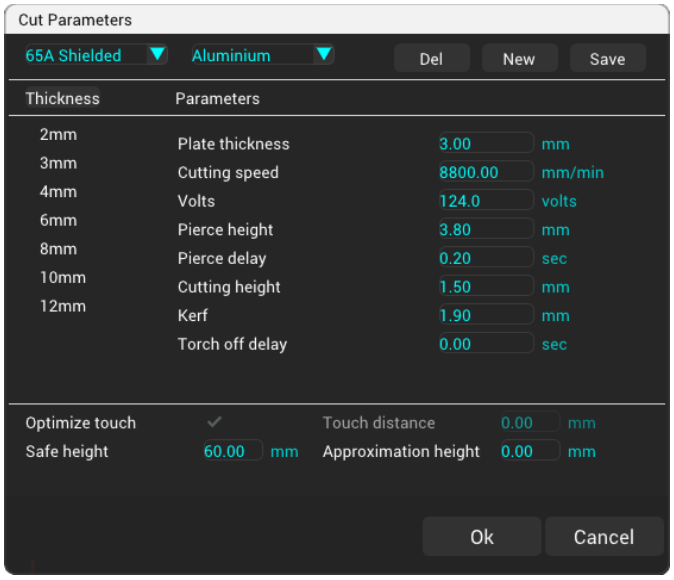

Figure 15: Cut Parameters Dialog

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.

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.
- Clicking the label of the column “Thickness” will also switch between metric and imperial sheet thickness units.

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.

8 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.

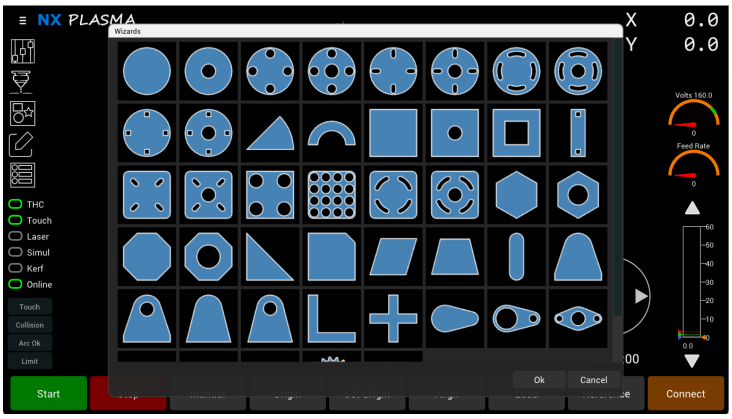

Figure 16: Wizard Dialog

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.

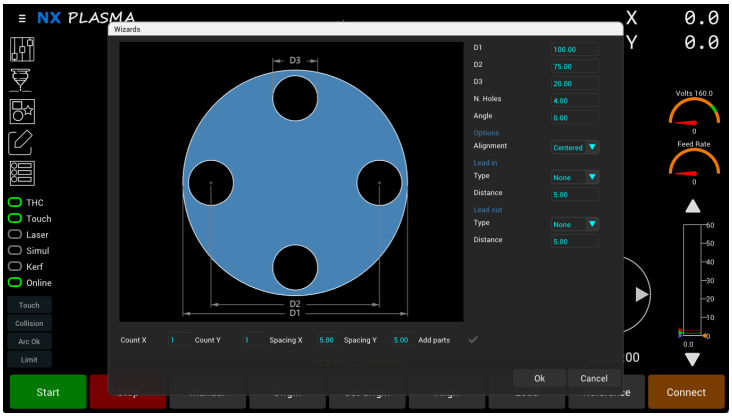

Figure 17: Wizard Dialog

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.

9 Editing Mode Controls

Editing Mode allows you to rearrange parts directly within the CNC system—**without needing external CAM software**. This powerful tool gives you the flexibility to **optimize part placement**, adjust spacing, rotate parts, and fine-tune your layout **in real time**, right at the machine.

9.1 Activating Editing Mode

You can enter Editing Mode in either of the following ways:

- Click the **"Edit Mode"** button on the main toolbar
- Press the **E key** on your keyboard

Once activated, a **dedicated toolbar** will appear, providing access to layout editing commands such as move, rotate, delete, and alignment tools.

9.2 Mode Restrictions

> **IMPORTANT:** While in Editing Mode, **all CNC machine controls are disabled**.

This includes:

- Jogging the machine
- Starting or stopping cuts
- Running macros or touch operations

This restriction is intentional and serves two purposes:

1. **Safety** – Prevents unintended motion while editing the layout
2. **Focus** – Ensures your attention remains on optimizing part placement

To resume machine operation, simply **exit Editing Mode**.

9.3 Mouse Controls in Editing Mode

Mouse behavior is adapted to suit layout editing. Here's how the mouse functions work:

- **Left Mouse Button**
  - Click to **select** or **deselect** individual parts or profiles
  - Hold SHIFT or CTRL for multi-select behavior (see **Selection Rules**)
- **Right Mouse Button**
  - Click and drag to **rotate** the currently selected part(s)
  - Rotation is centered around the part's current origin
- **Mouse Wheel**
  - Scroll to **zoom in or out** of the workspace
  - Click (press down) the wheel to perform **Zoom All**, which automatically adjusts the zoom level so that **all parts are visible on screen**

9.4 Editing mode Toolbar

Editing Mode transforms the CNC system into a lightweight layout editor—enabling quick, on-machine modifications that save time and reduce the need for repetitive CAM workflows. Its safety-first design ensures editing is isolated from active CNC operations, while the intuitive mouse controls provide a fast, responsive experience.

> **NOTE:** Combine Editing Mode with the Touch tools and **Set Origin** to rapidly adjust and cut parts on-the-fly—even with reused scrap material.

Figure 18: Editing Mode Toolbar

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.

Redo: This button restores the changes that were previously undone. Use it to reapply a command that was reverted using the Undo button.

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.

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.

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.

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.

9.5 Selecting and Deselecting Parts in Editing Mode

In **Editing Mode**, you can select or deselect parts—including internal features like holes—using intuitive mouse actions and keyboard modifiers. The system offers flexible and powerful selection tools to help manage even complex layouts efficiently.

9.5.1 Basic Selection

- **Click on a part** to select it. This selects both the **external contour** and all **internal cutouts or holes**, treating them as a single group.
- **Click again on the same part** to **deselect all internal cutouts**, leaving **only the outer profile selected**. This is particularly useful when you want to **apply or modify lead-ins and lead-outs exclusively on the external contour**, without affecting the entry/exit paths for internal holes. It helps avoid placing lead-ins inside small cutouts where space is limited, or when internal cuts are meant to be processed separately.

9.5.2 Multi-Select (Add to Selection)

- **Hold SHIFT** and click additional parts to **add them** to the current selection set. This allows you to build a selection incrementally without deselecting previously selected parts.

9.5.3 Selection Window

- **Click and drag** from an empty area to create a **selection rectangle**. All parts **completely enclosed** by the rectangle will be selected.

9.5.4 Modifier Keys and Advanced Selection Logic

You can control selection behavior with keyboard modifiers:

Keys Pressed	Result
SHIFT	Adds newly selected parts to the current selection
CONTROL	Removes selected parts from the current selection
SHIFT + CONTROL	**Toggles** selection state: – Selected items become deselected – Deselected items become selected

This combination of click behavior and keyboard modifiers gives you full control when working with nested parts, grouped elements, or when preparing specific toolpath strategies.

9.6 Moving Parts

You can reposition selected parts directly using the mouse. The system offers **collision-aware movement** to help maintain proper part spacing and prevent overlaps during manual layout adjustments.

9.6.1 How to Move Parts

1. **Select the part(s)** you want to move using the methods described in the **Selection section**.
2. **Click and hold** the left mouse button on any selected part.
3. **Drag the mouse** to the desired location.
4. Release the button to complete the move.

The selected parts will move by the same distance as the mouse's displacement.

- If **all parts are selected**, the **entire layout moves as a group**.
- If only **one or several parts** are selected, they will move **until they encounter a collision** with other parts or boundaries in the workspace.

9.6.2 Collision Handling

The system includes an automatic collision detection feature:

- When a moving part collides with another part, **movement stops immediately**, and a **yellow circle** is shown at the collision point.
- This ensures that selected parts do not overlap or violate the configured **separation distances**.

> **Note:** If parts were loaded with smaller separation values than the current setting, a collision might occur immediately—even with small movements. In such cases, the system prevents the move to maintain valid spacing.

9.6.3 Shift-Key Move Strategy (Push Mode)

There are **two move strategies**, depending on whether the **SHIFT key** is held:

Strategy	Description
Default Mode	*(SHIFT not pressed)* Selected parts move **until a collision is detected**. Movement stops at the collision point.
Push Mode	*(SHIFT pressed)* Selected parts **push** any colliding parts out of the way to make room. This can trigger a chain reaction where other parts are displaced.

While **Push Mode** can be useful for tightly packing parts or resolving blocked moves, it may cause **unexpected rearrangement** of nearby parts. Use it with care.

9.6.4 Undo Support

If an unwanted arrangement occurs—especially after using Push Mode—simply press the **Undo** button to restore the previous part layout.

9.7 Changing the Part Cutting Order

You can change the order in which parts are cut.

To change the cutting order:

1. Make sure the system is in **Edit Mode**.
2. Press **s** on the keyboard to display the cutting sequence numbers. Each profile will show its current cutting sequence number.
3. To change a part's cutting position, click the part while holding down the **Alt** key.

When no parts are selected, the first part you Alt-click becomes the first part to be cut. Its cutting sequence number changes to **1**.

Each additional part you Alt-click is placed immediately after the last selected part in the cutting sequence. For example, the next part you Alt-click becomes number **2**, the next becomes number **3**, and so on.

To move a group of parts in the cutting order, select the parts using drag selection. The newly selected parts will be moved to the position immediately after the currently last selected part.

In other words, Alt-clicking a part moves that part to the position immediately after the currently last selected part. If no parts are selected, the Alt-clicked part becomes the first part in the cutting sequence.

Press **s** again to hide the cutting sequence numbers.

You can also press **a** once or twice to select all parts or deselect all parts.

9.8 Editing Mode Shortcuts

While in **Editing Mode**, you can use the following keyboard shortcuts for faster layout control:

- **Toggle Grid Display** – Press **G** Toggles the visibility of the layout grid. This can help with manual part alignment and spacing.
- **Move Origin to Part Center** – Press **O** Moves the origin point (0,0) of the current job to the **center of the selected part**. Useful when you want to align or rotate parts around their midpoint.
- **Move Origin to Lower-Left Corner** – Press **SHIFT + O** Moves the origin point to the **lower-left corner** of the selected part, often used for alignment with material edges or sheet corners.

> **IMPORTANT:** These shortcuts affect the visual and positional reference used by the CNC system for layout, not the machine's physical coordinate origin.

9.9 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:

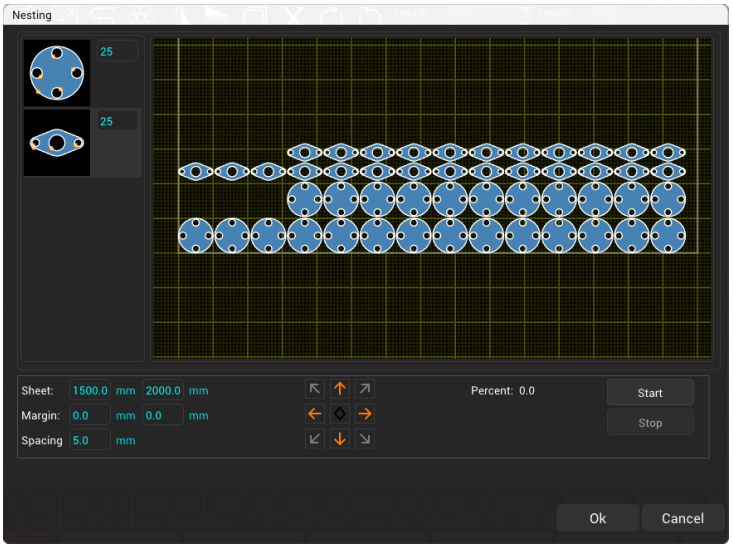

Figure 19: Nesting Dialog Box

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

- **Sheet:** 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.
- **Margin:** 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.

9.10 Leads tool

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

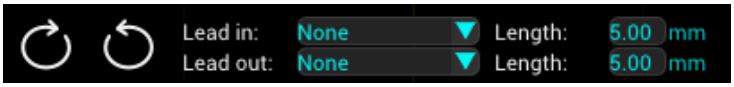

Figure 20: Leads tool

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 None, Arc, Tangent, Perpendicular or Multiple. The “Length” parameter (also applicable to both lead-in and lead-out) determines the length of the lead.

9.11 Options Dialog

The **Options Dialog** lets you configure layout display, machine behavior, unit settings, and movement preferences in the NX-PLASMA CNC system:

Figure 21: Options Dialog

9.11.1 Display Settings

- **Display paths** Toggles the visibility of cutting and rapid motion paths on screen.
- **Display table** Shows or hides the representation of the machine's cutting table in the layout area.
- **Display grid** Enables or disables the on-screen grid used for visual alignment and spacing.

9.11.2 Grid Configuration

- **Grid major spacing** Sets the spacing between major grid lines.
- **Grid minor spacing** Sets the spacing for finer, secondary grid lines.

9.11.3 Behavior Settings

- **Go back to start**

  If enabled, the machine returns the torch to the origin (X0, Y0) after the cutting process finishes.

- **Software limits**

  Activates or deactivates software-imposed movement boundaries to keep the torch within the safe working area.

- **Jog speed**

  Sets the **default jogging speed** as a percentage of the machine's maximum speed. This value affects manual torch movement (e.g., via Jog Control or arrow keys) when not using the **SHIFT key**, which always jogs at 100%.

9.11.4 Layout and Nesting Settings

- **Sheet size**

  Defines the **width and height of the metal sheet** used for nesting and part layout.

- **Part spacing**

  Determines the **minimum allowed distance between parts**, used in both manual movement and automatic nesting to prevent collisions.

9.11.5 Unit settings

- **Measurement Units**

  Select between **Metric (mm)** or **Imperial (inches)** as the working units for dimensions, part sizing, grid spacing, and sheet layout.

10 Configuration Dialog

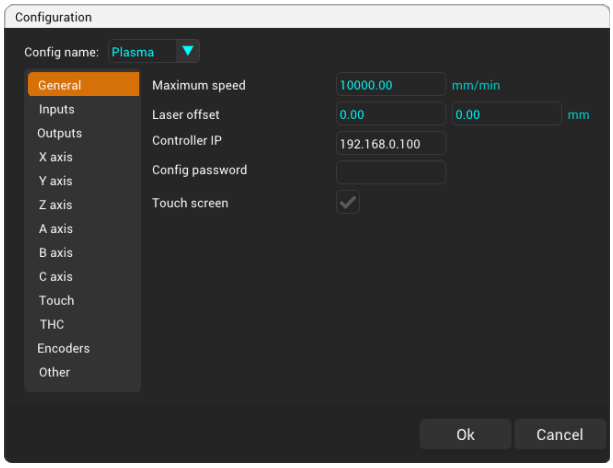

Figure 22: Configuration Dialog

The **Configuration Dialog** provides access to **all adjustable system settings** within the NX-PLASMA CNC software. To configure parameters effectively, a basic understanding of **CNC system principles** and the **Plasma or Oxy-fuel cutting process** is recommended. Proper configuration ensures accurate operation and optimal cutting performance.

10.1 Dual-Mode Configuration: Plasma and Oxy-fuel

NX-PLASMA supports **independent configuration for Plasma and Oxy-fuel modes**. This is especially useful if your machine is equipped with **both a plasma torch and an oxy-fuel torch**, as these tools often have different physical mounting positions and operational parameters.

For example:

- You can configure the **laser pointer offset** for the **Plasma torch** by switching to **Plasma mode**, entering the appropriate X/Y offsets, and saving the values.
- Then, switch to **Oxy-fuel mode** and enter a different set of offsets corresponding to the **Oxy-fuel torch**.
- When switching between modes, the dialog will automatically show and apply the correct values for each torch type.

> **IMPORTANT:**
>
> **Shared Parameters:** If a setting is **not explicitly configured in Oxy-fuel mode**, the system will

> automatically use the corresponding value defined in Plasma mode as a fallback.

This dual-configuration capability allows seamless mode switching without manual reconfiguration every time.

10.2 Dialog Structure

The configuration dialog is divided into two panes for clarity:

- **Left Pane:** Displays a list of configuration sections (e.g., General, Axis, Torch, I/O). Select a section to view its parameters.
- **Right Pane:** Shows the settings relevant to the selected section, with editable fields, dropdowns, or toggles depending on the parameter type.

This layout provides an intuitive way to navigate and adjust only the parameters you need, organized by topic or function.

10.2.1 General Configuration

The **General** section contains essential system-wide parameters that affect movement behavior, positioning tools, communication with the controller, and configuration security.

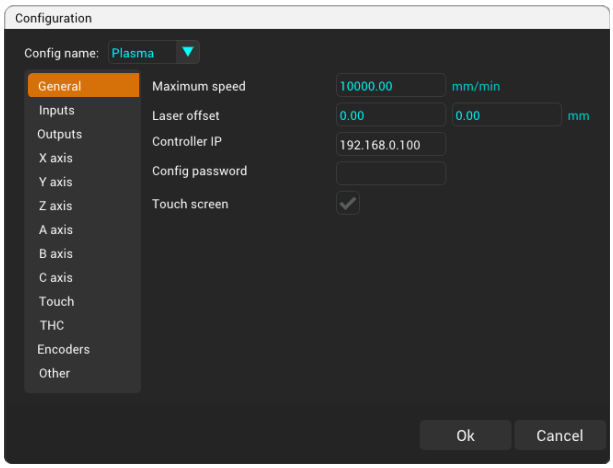

Figure 23: General Configuration Dialog

- **Maximum speed**

  Defines the **maximum speed** at which the CNC can move the torch. Even if individual axes are configured for higher speeds, this global limit will be enforced to ensure safe operation.

- **Laser offset**

  Specifies the **X and Y offset** between the **laser pointer** and the **actual torch position**. Since precise mechanical alignment between the laser and the torch is not always possible, this setting allows the system to compensate for the difference.

  - When the laser is activated, the **Digital Read Out (DRO)** displays the torch coordinates adjusted by this offset.
  - These adjusted values are shown in **red** to indicate that the torch location is being visually represented via the laser pointer.
  - If you position the torch using the laser (e.g., to set part origin), the system will automatically apply the offset, ensuring that the **cut begins at the true torch position**.

> **IMPORTANT:** You can configure **different laser offsets for Plasma and Oxy-fuel modes** if both tools are installed. The system will switch offsets automatically when changing modes.

- **Controller IP**

  Specifies the **IP address of the motion controller**. The default value is `192.168.0.100`. This must match the actual IP assigned to the controller for communication to succeed.

  To test connectivity:

  1. Open a **Windows Command Prompt**.
  2. Type: `ping 192.168.0.100` (or the configured IP address).
  3. If you receive packet loss or no response, check:
     - The controller's power status
     - Ethernet cable connection
     - Controller's IP configuration (it may differ from the expected address)

- **Configuration password**

  Allows you to **lock the configuration dialog** to prevent unauthorized changes.

  - To set a password: enter your desired password in this field.
  - Once set, this password will be required to access the configuration dialog.

> **IMPORTANT:** To **remove the password**, simply **clear the field** and save. If the password is lost and cannot be retrieved, you may need to **reinstall the software** to regain access.

> **IMPORTANT – Configuration Backup**
>
> All CNC configuration settings are stored in a single file named **NxCnc.db**, located in the NX-Plasma installation directory. This file contains every parameter configured through the Configuration Dialog — axis settings, I/O mappings, touch/THC parameters, cutting tables, and more.
>
> It is **strongly recommended** that you back up this file regularly, especially after making configuration changes. In the event of a system failure, software reinstall, or hardware replacement, restoring this file will recover your complete machine setup without the need to reconfigure everything from scratch.
>
> **To back up:** Simply copy NxCnc.db to a safe location (e.g., USB drive, network share, or cloud storage). To restore, replace the file in the installation directory with your backup.

10.2.2 Inputs Configuration

The **Inputs Settings** section allows you to assign physical input lines on the motion controller to specific CNC functions. These include sensor inputs, safety triggers, and user-initiated control switches.

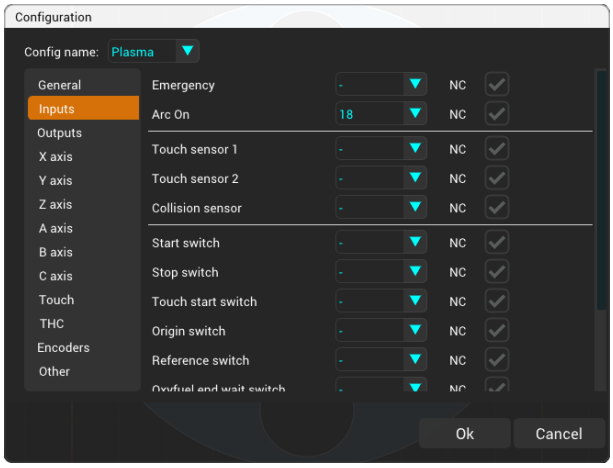

Figure 24: Inputs Configuration Dialog

Each function can be mapped to a controller input via the drop-down list:

- Select a pin number to assign it.
- Select "–" to leave it unassigned (disabled).

You can also define whether each input is **Normally Closed (NC)** by checking the NC box:

- **Unchecked**: Input is **Normally Open** — signal is active when the switch closes.
- **Checked**: Input is **Normally Closed** — signal is active when the switch opens.

10.2.2.1 Primary Inputs (Sensors & Safety) These are inputs typically connected to machine sensors, switches or safety devices:

- **Emergency** Connect to an **emergency stop switch**. When triggered, all machine motion stops immediately. This is a critical safety input.
- **Arc On (Arc OK)** Connected to the **plasma source's Arc Transfer signal**. When active, it indicates that a stable arc is present and cutting can begin.
  - If **not configured**, arc transfer is ignored, and arc loss cannot be detected. On a NX-Cnc motion controller it should be set to virtual input pin 18.
  - This signal is critical for reliable and safe cutting.

> **Note:** Different plasma sources have different arc signal behavior. If you're experiencing frequent **false arc-loss alarms**, try **increasing the Arc debounce time** to allow for slower or unstable arc detection. Conversely, if your plasma reacts quickly and stably, a lower debounce time may improve responsiveness.

- **Touch sensor 1** (Ohmic Contact Sensor) Used with **Ohmic-type sensors** that detect electrical contact with the metal surface. These sensors are precise and trigger immediately when the torch touches the material.
  - **No offset compensation is applied.**
- **Touch sensor 2** (Floating Head Sensor) Used with **mechanical or inductive switches**, typically installed on a **floating head**. Due to mechanical hysteresis, these sensors trigger slightly **before actual contact** with the plate.
  - The CNC applies an adjustable **"Touch 2 Offset"**, configured in the **Touch Settings**, to compensate for this delay.

> **Note: You can connect both Touch Sensor 1 and Touch Sensor 2 at the same time**. The system will automatically use **whichever sensor triggers first**, allowing for a robust and flexible probing setup that combines the speed of Ohmic sensing with the reliability of a floating head fallback.

  - **Collision sensor** Connected to a **torch collision detection switch**. If triggered (e.g., torch hits an obstacle or tips over), motion is halted to prevent damage.

> **Note:** To recover from a collision, press the **RESET** command button. This temporarily disables the collision input, allowing you to jog the machine. After **15 seconds**, the collision input is automatically reactivated for continued protection.

10.2.2.2 Action Switches (Manual Controls) These inputs allow connection to **external hardware switches** for common user commands:

- **Start switch** – Begins the cutting process.
- **Stop switch** – Immediately halts all motion and active operations.
- **Touch start switch** – Initiates a torch height sensing (touch) cycle.
- **Origin switch** – Sends the torch to the part origin (X0, Y0).
- **Reference switch** – Triggers the machine's **homing (referencing)** sequence.

10.2.2.3 Input Filtering (Debounce) Debounce settings help prevent false triggering caused by noise or mechanical bouncing:

- **Debounce Time** Global debounce delay for **all inputs** (in milliseconds). Inputs must remain stable for this duration before being accepted.
- **Arc Debounce Time** Specifically applies to the **Arc OK** input.
  - Use a **higher value** for plasma sources with slower arc transfer or unstable initial arcs.
  - Use a **lower value** for plasma systems with fast, clean arc starts.

10.2.3 Outputs Configuration

The **Outputs Configuration** section allows you to assign controller output pins to specific system functions, enabling external devices (e.g., torches, gas valves, relays, or indicators) to respond automatically to CNC events and machine states.

Each output function can be mapped to a physical output pin using the drop-down list. If left unassigned, some functions may default to internal relays, where applicable.

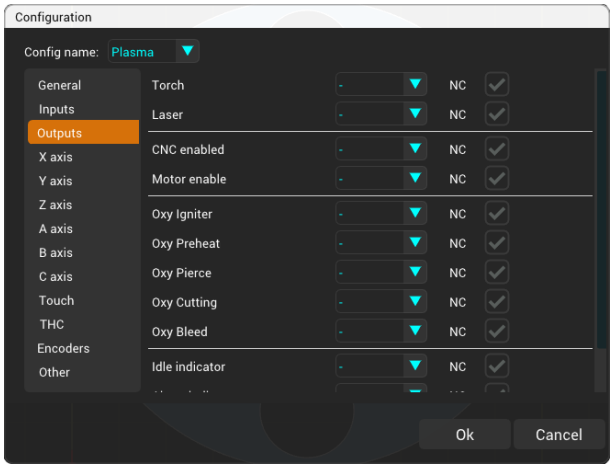

Figure 25: Outputs Configuration Dialog

10.2.3.1 Torch and Laser Control

- **Torch** Activates the **plasma torch** output when a cut is started.

> **Note:** If unassigned, the system uses the **default internal relay** (virtual output 7).

- **Laser** Activates the output for the **laser pointer**, used for visual alignment or part location.

10.2.3.2 System Status Outputs

- **CNC enabled** Becomes active when the CNC software is **connected and ready**.
- **Motor enabled** This signal activates when the CNC system goes online. It enables motor control by default through **Virtual Output 6**, which is internally connected to the **Enable pin** of each axis connector.

  If you assign a specific output pin to this function, that pin will be used **instead of Virtual Output 6**.

> **Note:** Use a custom output pin only if you need to route the motor enable signal through external logic or hardware. Otherwise, Virtual Output 6 will automatically control all axis enable lines.

- **Touch in Progress** This output becomes active when a touch sequence is about to be executed in plasma mode, and turns off when the touch sequence ends.

  It is typically used to control an isolation relay that enables the ohmic sensor input only while the touch sequence is being performed. When the touch sequence finishes, the relay is switched off again.

  This is especially useful with high-frequency start plasma sources, because it helps prevent the high-frequency start signal from reaching the ohmic sensor input, protecting the sensor circuitry.

10.2.3.3 Oxy-Fuel Phase Outputs

To simplify integration with various **oxy-fuel torch manufacturers**, NX-PLASMA defines outputs that correspond to the **phases of an oxy-fuel cut cycle**.

Manufacturers can wire these outputs to **relays or solenoid valves** to control the appropriate gas flows and ignition mechanisms based on their specific requirements.

10.2.3.4 Cut Cycle Phases and Outputs

1. **Preheating Phase**
   - **Oxy Igniter Output** Activated **at the start of preheating** to enable the igniter (e.g., spark plug or pilot flame).
   - **Oxy Preheat Output** Activated **together with the igniter** to open preheat gas valves (e.g., acetylene or propane).
   - **Both outputs remain active** until the end of the cut.
2. **Pierce Phase**
   - **Oxy Pierce Output** Activated **when the system transitions to piercing**. Typically used to open the **high-pressure oxygen valve** required for piercing.
   - Remains active for the duration of the cut.
3. **Cutting Phase**
   - **Oxy Cutting Output** Activated when entering the **main cutting phase**. Typically used to control the **cutting oxygen valve**.
   - Remains active until the cut completes.
4. **Post-Cut Bleed**

- **Oxy Bleed Output** Activated **only briefly at the end of each cut** to purge lines or release gas pressure. Useful for maintaining safety and prolonging torch component life.

> **Note Manufacturers can assign these outputs to different relays** that open or close valves for gases (e.g., O_2, C_2H_2, air) depending on their hardware setup. This flexible phase-based output system avoids the need for rigid, gas-specific mappings, and supports diverse torch designs.

10.2.3.5 Indicator Outputs These outputs can be connected to LEDs, signal lamps, or dashboard panels to reflect controller state:

- **Idle indicator** Active when the system is powered and idle — no motion or execution in progress.
- **Alarm indicator** Active when the system enters an alarm state (e.g., emergency stop, collision, arc loss).
- **Busy indicator** Active while the controller is executing commands, moving, or cutting.

10.2.3.6 Summary Assigning outputs to these predefined functions allows the CNC system to **seamlessly coordinate plasma or oxy-fuel torch control**, external signaling, and hardware automation. The **oxy-fuel phase outputs** are especially powerful for torch manufacturers, providing a clean, phase-based interface to **trigger valves, igniters, and relays** in any desired combination — without requiring internal software customization.

10.2.4 Axes Configuration

The Axes Configuration section defines how each CNC axis behaves, including motion parameters, software travel limits, signal polarity, and hardware mappings. These settings are essential for ensuring accurate movement, smooth acceleration, safe operating boundaries, and correct signal routing between the controller and motor drivers.

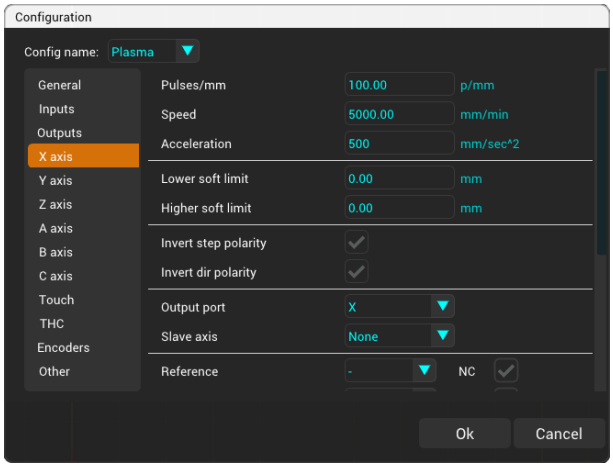

Figure 26: Axes Configuration Dialog

The **Axes Configuration** section defines motion parameters, limits, direction polarity, and hardware mappings for each CNC axis. Proper configuration ensures safe, smooth, and accurate motion.

10.2.4.1 Motion Parameters

- **Pulses/Unit (Pulses per mm or inch)** Sets how many electrical pulses the controller sends to move the axis by 1 unit (mm or inch, based on system units). This value must match your stepper driver and mechanical setup.
- **Speed** Defines the **maximum movement speed** for the axis. The controller will never exceed this speed during execution.
- **Acceleration** Specifies how quickly the axis ramps up or down in speed. Higher acceleration yields faster response but may cause mechanical vibration. Choose values that balance performance and smoothness.

10.2.4.2 Software Limits

- **Lower software limit** Defines the **minimum coordinate** the axis can move to. Movements beyond this limit are restricted to avoid collisions or over-travel.
- **Higher software limit** Sets the **maximum coordinate** allowed for the axis. Like the lower limit, it acts as a virtual boundary for safe movement.

> **Note:** Software limits are enforced during job execution. If the toolpath attempts to exceed these limits, motion will be stopped and an alarm may be raised.

10.2.4.3 Signal Inversion

- **Invert step polarity** Inverts the polarity of the step signal sent to the motor driver. Use this only if required by your driver.
- **Invert dir polarity** Inverts the direction signal. Useful if the axis moves in the wrong direction (e.g., pressing right moves X negative).

10.2.4.4 Output Port

- **Output port** Selects the physical port or connector used for sending motion signals for this axis. You can redirect an axis to another motor connector (e.g., assign X to B) to accommodate custom wiring or hardware layouts.

10.2.4.5 Slave axis

- A **slave axis** mirrors the movement of another axis. This is useful in gantry systems with dual motors or when two axes must move in sync. > **Note:** Slave axes do not have independent motion; they replicate the primary axis's step signal. However, their direction can be inverted if needed.
- **Output port:** The hardware port through which this axis's step and direction signals are sent (e.g., Y-axis port).
- **Slave axis:** Replicates the movement of the primary (e.g., A), using the same step signal. The **direction** can be independently inverted if required.

> **Note:** Slave axes ensure synchronized movement but do not have independent motion. Direction inversion allows mirrored motion when needed.

10.2.4.6 Axis Inputs Configuration Each axis can be associated with specific physical input signals that control essential functions such as homing (referencing), limit detection, jogging, and servo fault monitoring. These input signals are connected to sensors or switches on the machine and allow the CNC system to respond appropriately to mechanical boundaries, operator commands, and error conditions.

Properly configuring these inputs is crucial for ensuring safe operation, preventing over-travel or collisions, and enabling precise manual control during setup, calibration, or troubleshooting.

- **Reference** Input pin for the **reference/home sensor**. Used during homing to define the zero position.
- **Limit minus** Pin for the **negative travel limit sensor**. Prevents the axis from moving past its lower boundary.
- **Limit plus** Pin for the **positive travel limit sensor**. Prevents over-travel beyond the upper coordinate.
- **Servo motor error** Pin for the **servo alarm signal**. Triggers a controller alarm if the motor reports an error or fault.
- **Jog Minus / Jog Plus** Optional inputs that allow the operator to manually move the axis using external push-button switches.

10.2.4.7 Axis Referencing Settings These parameters fine-tune the homing behavior of each axis:

- **Position** Sets the coordinate value that the controller assigns when the reference sensor is triggered.
- **Speed** Defines the **homing speed** as a percentage of maximum axis speed. Slower speeds increase homing precision and reduce mechanical stress.
- **Invert direction** Changes the direction the axis moves to seek the reference sensor. Default is negative; enable this if your home switch is on the positive side.

> **Note:** For safety reasons, the **Z axis should always be referenced using a switch at the top (positive) position**, ensuring the torch lifts away from the material during homing.

10.2.4.8 Summary A well-configured axis system ensures accurate motion, proper software limits, and safe homing. Support for step/dir inversion, custom port routing, and slave axes gives you full flexibility to adapt the system to your mechanical layout and wiring preferences.

10.2.5 Touch Configuration

The **touch process** is a critical step in determining the exact height of the metal plate prior to cutting. During this sequence, the controller follows a precise procedure to detect when the torch makes contact with the plate, establishing a reference height for the cut.

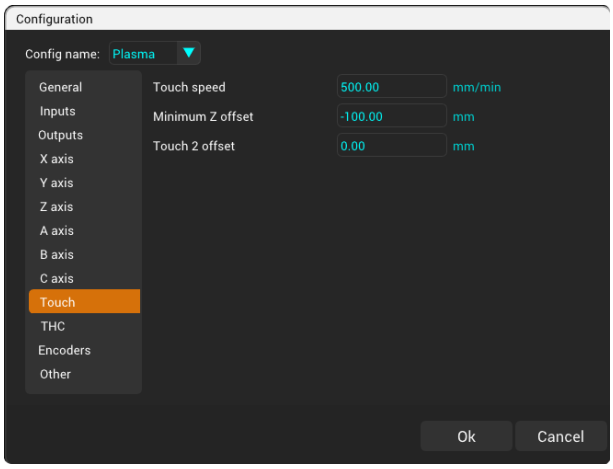

Figure 27: Touch Configuration Dialog

Before reviewing the configuration fields, here's how the touch sequence operates:

- **Initial Movement** The controller moves the Z-axis rapidly toward the plate until it reaches a height defined by the **Approximation Distance** above the expected **plate thickness** (both set in the Cutting Parameters dialog).
- **Speed Reduction** When the torch is within the approximation distance, the descent slows to the defined **Touch Speed**, allowing for precise detection with minimal mechanical impact.
- **Touch Detection** The Z-axis continues downward at reduced speed until a change in the **Touch Input** signal is detected, indicating contact between the torch and the metal surface. At this point, the controller sets the current Z position as the zero reference for cutting.

This process is typically repeated before every cut unless the **Optimize Touch** option is enabled. When optimization is active, the touch operation is skipped if the distance from the last touch point is below the **Touch Optimize Distance** threshold.

10.2.5.1 Key Parameters

- **Touch Speed** Defines the speed (in mm/s) used during the final descent of the touch sequence. Lower values reduce impact and wear, improving accuracy and sensor lifespan.
- **Minimum Z Offset** A safety threshold: if the torch reaches this absolute Z height without detecting touch, the controller triggers an alarm. This protects against failed detection due to wiring or mechanical faults.
- **Touch 2 Offset** Used when employing a **floating head sensor**. Because of mechanical hysteresis, the touch signal may trigger before the actual plate contact. This offset compensates for that discrepancy, ensuring accurate height calibration.

> **Note:** Accurate touch configuration is essential for reliable piercing and consistent cut quality, especially when using thin sheets or uneven surfaces.

10.2.6 THC Configuration

Torch Height Control (THC) is a key feature in plasma cutting systems that dynamically adjusts the Z-axis during the cut to maintain a consistent torch-to-material distance. It does this by monitoring the arc voltage and making real-time height corrections, ensuring clean cuts, extending consumable life, and improving overall quality—especially on warped or uneven sheets.

The parameters below allow you to fine-tune how THC behaves in response to voltage changes, feed rate, and cutting scenarios like hole cutting and slowdowns.

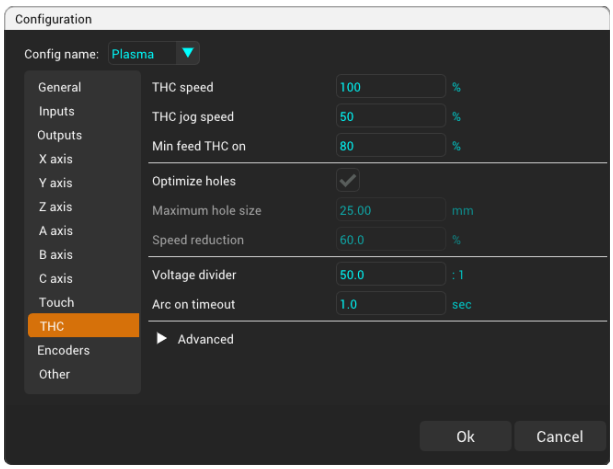

Figure 28: THC Configuration Dialog

10.2.6.1 Main Parameters

- **THC Speed** Sets the maximum rate (as a percentage of Z-axis capability) at which THC is allowed to adjust the torch height. A value of **100%** provides full responsiveness.
- **THC Jog speed**
- **Min Feed – THC On** Defines the minimum cutting speed (as a % of current feed rate) required for THC to remain active. Below this threshold (e.g., during corners or small features), THC is temporarily paused to prevent erratic Z-axis behavior.
- **Optimize Holes** Enables feed rate reduction for smaller holes, improving roundness and reducing dross during hole cutting.
- **Maximum Hole Size** Holes smaller than or equal to this diameter will be affected by hole optimization settings.
- **Speed Reduction** Sets how much the cutting speed should be reduced (in %) when cutting small holes. Lower speeds result in smoother, more precise holes.
- **Voltage Divider** This parameter defines the expected voltage division ratio from your plasma source. The NX-Plasma system is designed for a standard **50:1** voltage divider, which is the required and supported configuration. Setting this to a different ratio (e.g., **25:1**) will **not work** and will result in incorrect voltage readings and improper THC behavior.

  That said, fabrication tolerances, cable length, connector resistance, or minor variations in your plasma source's internal divider may cause the actual division ratio to deviate slightly from 50:1. To compensate for this, you can fine-tune this setting — for example, entering **49.5** instead of 50 — to achieve more accurate voltage readings and improve THC responsiveness.

  > **Tip:** If you notice consistent discrepancies between expected and actual arc voltage during cutting, measuring the real divided voltage and adjusting this value accordingly can improve system precision.

- **Arc On Timeout** Defines how long (in milliseconds) the controller will wait for the **Arc OK** signal after issuing a torch-on command. If the arc doesn't stabilize in time, the cut is canceled to prevent errors or poor quality starts.

> **Tip:** Fine-tune THC parameters based on your plasma source's characteristics and cutting speed ranges to optimize cut quality and system responsiveness.

10.2.6.2 Advanced THC Configuration In most cases, the default values in this section are sufficient and optimized for reliable operation. However, advanced users may adjust these settings to tailor THC response during specific segments of the cut.

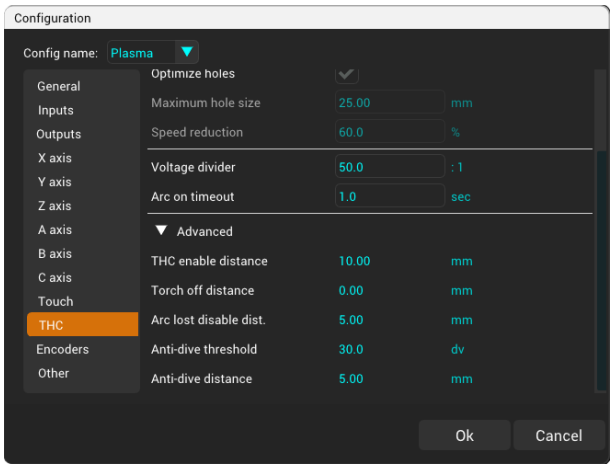

Figure 29: Advanced THC Configuration Dialog

10.2.6.2.1 Advanced Parameters

- **THC Enable Distance** Specifies how far the torch must travel after piercing before THC becomes active. This delay allows the arc voltage to stabilize before corrections begin.
- **Torch Off Distance** Defines how many millimeters before the end of the cut the torch should be turned off. This is often set to **0**, but can be increased to ensure a cleaner separation at the end of the path.
- **Arc Lost Disable Distance** Disables arc loss detection near the end of a cut to avoid false alarms caused by arc extinguishing as the part drops or when exiting thin material.
- **Anti-Dive Threshold** This parameter defines the **sensitivity of the anti-dive system**, which monitors the arc voltage in real time for sudden, unexpected **spikes**. These voltage surges often occur when the torch passes over a **hole**, a **previous cut**, or **molten residue** (*borra*)—causing the arc to stretch and the voltage to rise sharply.

  When such a **rapid upward voltage deviation** is detected—exceeding the configured threshold—the system **freezes THC movement** immediately. This is done to prevent the torch from making a reactive downward correction (a “dive”), which could lead to tip damage, scrap parts, or a collision with the material.

  This detection is **not based on a simple voltage limit**, but rather on a complex algorithm that evaluates the **rate of change and deviation** relative to the current cutting voltage. Once the

spike is detected, THC height control is suspended until the torch travels a specified distance, defined by the **Anti-Dive Distance** parameter. After that distance is completed, normal THC behavior resumes.

> **Note: Do not change this setting unless you are an expert user** and have tested the new values under a variety of real cutting conditions.
>
> - If the value is set **too low**, the torch may fail to recognize genuine anti-dive events—resulting in **torch dives**, especially near the end of a cut where arc conditions change abruptly.
> - If the value is set **too high**, THC movement may be **inhibited too often**, giving the impression that **THC is not working** and leading to inconsistent cutting height.
>
> The default value of **300** is carefully chosen to balance responsiveness and reliability for most machines and materials.

- **Anti-Dive Distance** Specifies a segment near the start or end of the cut where THC is temporarily disabled to prevent instability during acceleration/deceleration phases.

> **Note:** These advanced settings are intended for experienced users. Incorrect configuration may lead to inconsistent cuts or reduced system safety.

10.2.6.3 THC Summary

- **Torch Height Control (THC)** is a core feature of the NX-Plasma system, designed to automatically maintain the optimal torch-to-material distance during cutting. By continuously adjusting the Z-axis based on arc voltage feedback, THC helps deliver consistent cut quality, reduces dross, extends consumable life, and compensates for warping, material unevenness, or plate imperfections.

> **Recommended Use:** THC should **always remain enabled**, unless you are performing a very specific cut on **extremely thick materials** or are an expert operator with a precise reason to disable it. Disabling THC in standard operations can lead to inconsistent cut depth, reduced quality, and unnecessary wear on consumables.

> **IMPORTANT Clarification:** If the torch collides with the material during a cut, this **does not mean THC is malfunctioning**. This issue is almost always caused by an **incorrect arc voltage setpoint**, not by a THC failure. THC may be working properly, but it is following a reference voltage that doesn't correspond to actual arc conditions—causing it to move the torch too low or too high.

> **Key Advantage of NX-Plasma:** Unlike many CNC systems where users must manually adjust the **THC reaction speed** (or Z-axis speed) depending on whether they are cutting **thin or thick plates**, the NX-Plasma system **automatically handles this for you**. You will **never need to adjust Z-axis speed or THC speed manually** based on material thickness or cutting conditions. This simplifies setup, eliminates guesswork, and ensures optimal torch response in all scenarios.

With proper configuration, the NX-Plasma THC system delivers precise, hands-free height control that adapts seamlessly to your cutting needs—regardless of material size or thickness.

10.2.7 Encoders Configuration

The **Encoders** section allows configuration of up to four independent encoder channels. These encoder inputs can be used to connect external **digital rotary encoders** to the controller's input pins. They enable real-time manual control of key parameters, such as adjusting the **THC setpoint voltage**, **feed rate override percentage**, or moving the **Z-axis up or down** in **OxyCut mode** using physical knobs or dials.

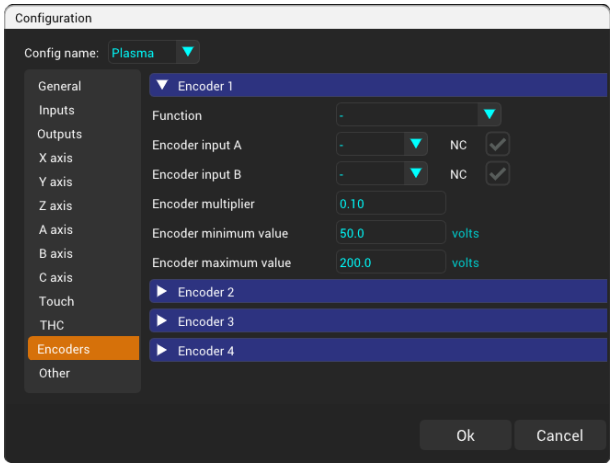

Figure 30: Encoders Configuration Dialog

10.2.7.1 Configuration Parameters (per encoder)

- **Function** Defines how the encoder input is interpreted by the CNC system. Supported functions include:
  - **Voltage** – Allows real-time adjustment of the plasma **THC setpoint voltage** during cutting.
  - **Feed Override** – Controls the **cutting feed rate override percentage** via an external encoder, typically within a range like 0–120%.
  - **OxyCut Z Height Adjustment** – Enables manual control of the **Z-axis position** during oxy-fuel cutting.
- **Encoder Input A / B** Assigns the controller's digital input pins for the encoder signal:
  - **Input A** – Main encoder signal (typically the channel A pulse).
  - **Input B** – Secondary signal (channel B) for quadrature decoding, allowing direction sensing.
  - The **NC checkbox** allows you to **invert the signal polarity** if needed, depending on encoder wiring or logic level.
- **Encoder Multiplier** Adjusts the encoder's sensitivity. A **larger multiplier** means fewer steps (or less rotation) are needed to affect the output. A **smaller multiplier** makes the encoder less sensitive, requiring more rotation to produce the same change.
  - For **Voltage**, this adjusts how encoder steps translate into voltage increments.
  - For **Feed Override**, it defines how pulses map to percentage changes.
  - For **OxyCut Z**, it determines how fast the Z position changes per encoder step.
- **Encoder Minimum Value** Sets the **lowest allowed output value** for the selected function:
  - For **Voltage**, this defines the minimum THC voltage setpoint (e.g., 50.0V).
  - For **Feed Override**, this sets the minimum override percentage (e.g., 50%).
- **Encoder Maximum Value** Sets the **upper limit** of the encoder's effect:
  - For **Voltage**, this defines the maximum allowable setpoint (e.g., 200.0V).
  - For **Feed Override**, this caps the override percentage (e.g., 120%).

> **Note:** These encoder functions provide powerful, real-time control during cutting. They are recommended for advanced users who require live adjustments without interrupting the cutting process.

10.2.8 Other Configuration

The **Other** section contains essential system-wide configuration options that define physical machine dimensions, axis referencing behavior, pulse timing for motion control, and plasma system presets.

These settings play a crucial role in adapting the software to your specific CNC hardware and ensuring reliable, precise operation.

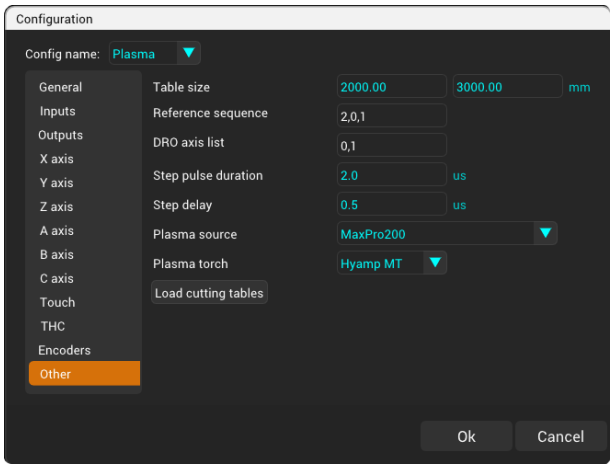

Figure 31: Other Configuration Dialog

10.2.8.1 Configuration Parameters

- **Table Size** Specifies the physical dimensions of the machine's working area. This value defines the boundaries within which cutting operations are allowed.
- **Reference Sequence** Defines the order in which the axes are homed during the machine's referencing (homing) cycle. The default value is `"2,0,1"` (Z first, then X, then Y), where:
  - `0 = X-axis`
  - `1 = Y-axis`
  - `2 = Z-axis` You may customize this order using a comma-separated list of axis numbers to suit your machine layout or mechanical structure.
- **DRO Axis List** Controls which axes appear in the **Digital Readout (DRO)** on the user interface. For example:
  - `"0,1"` displays only X and Y.
  - `"0,1,2"` displays X, Y, and Z. This allows a cleaner UI for simpler machines while enabling full readout for multi-axis setups.

- **Pulse Duration** Sets the **width of the step pulses** in microseconds. This value must be compatible with the specifications of your motor drivers (stepper or servo). A typical value might be **2–10 µs**, but always refer to your driver's datasheet.
- **Step Delay** Defines the **delay between direction change and the first step pulse**, in microseconds. This ensures that motor drivers receive a stable direction signal before stepping begins. Like pulse duration, this should match your hardware requirements.
- **Plasma Source** Select the model of your **plasma power supply**. This selection is used to determine the correct set of **cutting parameter tables**.
- **Plasma Torch** Select the model of your **plasma torch** (e.g., mechanized torch variant) to match the specifications in the cutting tables.
- **Load Cutting Tables** Click this button to load the appropriate **cutting tables** based on the selected plasma source and torch. These tables define parameters such as cutting speed, pierce time, arc voltage, and other plasma-specific settings.

> **Note:** This section is key to aligning the software with your **mechanical setup**, **motion control timing**, and **plasma equipment**. Proper configuration ensures smooth axis movement, accurate homing, and correct cut parameter selection for your material and tool combination.

11 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

12 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.

12.1 Torch Collides with the Table

Torch keeps moving down until it collides with the table.

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.

12.2 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.

12.3 Torch Not Firing or Cutting Properly

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

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.

12.4 Poor Cut Quality

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

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.

12.5 Lost Motion or Inaccurate Cuts

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

Possible Solutions:

1. Verify motor and driver performance on all axes.
2. Inspect for mechanical interference, loose couplings, or debris.
3. Ensure correct steps/unit (steps/mm) calibration.
4. Check that acceleration values are no too high for the machines motors.
5. If using servo motor check servo motors internal configuration.
6. Check for wear or binding in lead screws or linear guides.
7. Confirm current and microstepping settings in motor drivers.
8. Lubricate motion components as per machine maintenance schedule.

12.6 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.

13 Frequently Asked Questions

13.1 What is the .nx file format?

The .nx file format is the **native and proprietary format** used by NX-Plasma CNC software. It stores the **entire cutting program**, including:

- Imported or created part geometry (converted into an internal **mathematical cutting model**).
- Nesting layouts.
- Lead-in and lead-out paths.
- Machine settings.
- Other workspace parameters.

Unlike DXF or G-code files, which must be **imported and converted** each time, .nx files allow NX-Plasma to **load and resume operations immediately**, with no need to reprocess or reconfigure the data.

13.2 Why save and load .nx files?

Saving your workspace as a .nx file ensures that all aspects of the cutting setup are preserved, including:

- Converted profiles and closed contours.
- Lead-ins and cut sequencing.
- Nesting positions and arrangements.
- Custom settings and machine parameters.

This provides several advantages:

- **Faster workflow** – instantly resume cutting without re-importing or adjusting files.
- **Accuracy** – eliminates discrepancies from repeated imports or manual setup.
- **Convenience** – store fully-prepared jobs for reuse, editing, or reference.

In short, .nx files are the **recommended way to save and manage your cutting projects** in NX-Plasma.

13.3 Why Is It Important to Reference the Z Axis?

Referencing the **Z axis** is essential for the CNC machine to correctly determine its vertical position. While referencing the X and Y axes is not strictly necessary in plasma cutting—since cuts are made relative to the position where the **"Set Origin"** command was issued—the **Z axis must always be referenced** after powering up the machine.

This is because the machine has **no way of knowing the current height of the Z axis** when it starts. Without referencing, any movements along the Z axis (such as descending to the sheet surface) could be inaccurate or even dangerous. Performing a Z-axis reference ensures proper synchronization between the machine's internal position and the actual physical position of the torch.

13.4 Why Is It Important to Use a Touch Sensor?

The **touch sensor operation** plays a critical role in plasma cutting accuracy by detecting the **exact height of the sheet surface**. This allows the machine to establish the correct **Z = 0** position at the starting point of the cut.

Maintaining an accurate **cutting distance**—the gap between the torch tip and the material—is key to achieving clean, precise cuts. After the cut begins, the system uses **THC (Torch Height Control)** to automatically adjust the torch height by **monitoring the cutting voltage**.

Here's how it works:

- If the **monitored voltage is higher** than the configured value, it means the torch is **too far** from the sheet. The CNC lowers the torch to reduce the voltage.
- If the **voltage is too low**, the torch is **too close**, so the CNC raises it to increase the gap.

13.5 Common Issues and Recommendations

- If you configure the cutting voltage **too low**, the CNC will continually try to lower the torch to match it—eventually **colliding with the sheet** if the surface is reached and the voltage is still too high.
  - **Solution:** Raise the configured voltage slightly (by 0.5 to 1 volt) to prevent collisions.
- If the cutting voltage is set **too high**, the CNC will raise the torch trying to reach it. This can eventually trigger a **Z-axis upper limit fault** when the torch reaches the top.
- This behavior can also indicate **incorrect configuration**, such as:
  - Mismatched **steps per millimeter**

- Incorrect **Z reference height** in the machine settings

> **Note:** We **do not recommend using software limits** on the Z axis. Software limits may interfere with the THC operation and **will not trigger any alarms**. Instead, the torch may simply stop moving when limits are exceeded—causing loss of height control without warning.

13.6 How do I set cutting parameters for different materials and thicknesses?

Open the Cutting Parameters Dialog from the toolbar, select amperage and material type, then choose thickness. Adjust values like speed, volts, pierce height/delay, and kerf based on your plasma source's manual. Save custom presets for reuse. Incorrect settings can cause poor cuts or consumable damage.

13.7 How do I enter Editing Mode and rearrange parts?

Press the "E" key or click the Edit Mode button on the toolbar. This disables CNC controls for safety. Select parts (hold SHIFT to multi-select), then use tools to move, rotate, flip, copy, delete, or nest them. Exit Editing Mode to resume machine operations. Use mouse wheel for zoom and shortcuts like "O" to set origin to lower-left.

13.8 How does the Part Wizard work for creating quick parts?

Click the Part Wizard button to access a library of parameterized shapes (e.g., flanges). Select a part, adjust dimensions, holes, and lead-in/out types on the right pane. Generate grids of copies by setting rows/columns and spacing. Check "Add parts" to append to your workspace without overwriting existing ones.

13.9 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.

13.10 Why Is the Start Button Disabled, and How Do I Fix It?

The **Start** button may remain disabled due to one or more safety-related conditions:

- The machine is in **editing mode** — CNC controls are disabled while editing.
- One or more axes are **not referenced** — especially the Z axis, which must be referenced before cutting.
- The **part origin has not been set** — the system requires a defined start point for the cut.

13.11 How to Enable the Start Button

1. **Exit editing mode** and return to **CNC mode**.
2. **Reference the machine**, especially the Z axis.
3. **Load a cutting program** or import a part (DXF or G-Code).
4. **Jog** to the desired origin position.
5. Click **“Set Origin”** to define the start point.
6. Ensure the system is **online and free of errors**.

If the Start button is still disabled, check the status bar for messages such as:

- **“Not Referenced”**
- **“Offline”**
- **“No Origin Set”**

These messages will indicate what still needs to be addressed before cutting can begin.

14 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 machines, including plasma cutting and oxy-fuel cutting machines, ensuring 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.

15 Appendices

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

16 SOFTWARE LICENSE AGREEMENT

Effective Date: 01/01/2024

This Software License Agreement (“Agreement”) is entered into by and between **Nevex Tecnologia Ltda.**, a company organized under the laws of Brazil, with its principal mailing address at **P.O. Box 24913, CEP 80410-170, Curitiba, Paraná, Brazil** (“Licensor”), and the individual or entity using the software (“Licensee”).

By checking the “I agree” box below, Licensee agrees to the terms of this Agreement. If Licensee does not agree to the terms, they should not install or use the Software.

16.1 DEFINITIONS

- **Software**: Refers to the program known as NX-Plasma, including any updates, documentation, and materials provided by Licensor.
- **Updates**: Refers to improvements, bug fixes, patches, or enhancements provided by Licensor.
- **License**: The non-exclusive, non-transferable, limited right to use the Software as defined in this Agreement.
- **Evaluation License**: A temporary, limited license intended solely for non-commercial trial purposes, which may include feature limitations or time-based expiration.
- **Proof of License**: A valid invoice or purchase confirmation from the Licensor or an authorized reseller confirming license ownership.

16.2 LICENSE GRANT

Scope of Use:

- Licensee is granted the right to install and use the Software on one CNC plasma cutting table or one computer system unless otherwise agreed in writing.
- The Software is licensed, not sold, and Licensee may not transfer or assign the License to another entity without prior written consent from the Licensor.
- If the Software is used in Evaluation License mode, it may only be used for testing or demonstration purposes and not for production or commercial use.

Restrictions:

- Licensee may not modify, reverse engineer, decompile, disassemble, or create derivative works based on the Software.
- Licensee may not sublicense, rent, lease, or otherwise redistribute the Software or any part thereof.
- Licensee may not remove or alter any copyright, trademark, or other proprietary notices.
- Licensee may not attempt to bypass, disable, or interfere with any security features, licensing mechanisms, or access restrictions implemented in the Software.
- Licensee may not use emulators, virtual machines, or other means to circumvent hardware verification.

Hardware Binding: The Software may be licensed for use only when connected to a licensed NX-Plasma controller. Attempting to bypass or emulate the hardware connection is strictly prohibited.

Proof of License: Licensee must retain Proof of License as evidence of a valid license. Lack of such documentation may render the License invalid.

16.3 INTELLECTUAL PROPERTY RIGHTS

- **Ownership**: All rights, title, and interest in and to the Software, including but not limited to copyrights, trademarks, and trade secrets, are and shall remain the exclusive property of Licensor.
- **No Implied Rights**: Licensee acknowledges that no rights are granted under this Agreement other than those explicitly set forth herein. This includes, but is not limited to, any patent or proprietary rights related to the Software.

16.4 UPDATES AND SUPPORT

- **Updates**: The Licensor may, at its sole discretion, provide updates to the Software free of charge for a period of one (1) year from the date of purchase.
- **Restrictions on Updates**: The Licensee may not charge third parties for any updates provided by the Licensor, nor may they represent such updates as their own work. Updates are intended solely to improve functionality, correct defects, or maintain compatibility.
- **Post-Update Period**: After the initial one-year period, Licensor may offer further updates as paid upgrades or as part of a new licensing agreement.

16.5 PROHIBITED ACTIONS

Licensee agrees not to:

- Modify the Software's user interface, branding, or executable files, including changing the displayed name of the Software or altering any accompanying documentation or marketing materials.
- Misrepresent the Software as a product created by anyone other than Licensor.
- Redistribute or resell the Software under a different name or license.

16.6 TERM AND TERMINATION

- **Term**: This Agreement is effective until terminated.
- **Termination by Licensor**: Licensor may terminate this Agreement immediately if Licensee breaches any of its terms.
- **Effect of Termination**: Upon termination, Licensee must immediately stop using the Software and delete all copies from its systems, including backup copies, and certify to Licensor in writing that all copies have been destroyed.
- **No Refund**: Upon termination, Licensee shall not be entitled to any refund of license fees or other compensation.

16.7 WARRANTIES AND DISCLAIMERS

- **No Warranty**: The Software is provided "AS IS" without warranties of any kind, express or implied, including but not limited to merchantability, fitness for a particular purpose, or non-infringement.
- **No Guarantee**: Licensor does not guarantee uninterrupted or error-free operation of the Software.
- **No Suitability Guarantee**: Licensor makes no representation that the Software will meet Licensee's specific requirements or perform at any particular level of reliability or performance.

16.8 LIMITATION OF LIABILITY

- Licensor's liability for any claims arising out of or related to this Agreement shall be limited to the amount paid by Licensee for the Software during the 12 months prior to the claim.
- Licensor shall not be liable for any indirect, incidental, consequential, or punitive damages, including but not limited to loss of revenue, data, or business opportunities.

16.9 GOVERNING LAW AND JURISDICTION

- **Governing Law**: This Agreement shall be governed by the laws of Brazil.
- **Jurisdiction**: Any disputes arising out of or related to this Agreement shall be subject to the exclusive jurisdiction of the courts located in Curitiba, Paraná, Brazil.
- **Dispute Resolution**: Before resorting to litigation, the parties agree to attempt to resolve any disputes through mediation or arbitration as may be appropriate under Brazilian law.

16.10 MISCELLANEOUS

- **Entire Agreement**: This Agreement constitutes the entire understanding between the parties and supersedes all prior agreements, whether oral or written, relating to the subject matter of this Agreement.
- **Amendments**: Any amendments to this Agreement must be made in writing and signed by both parties. Any such amendments shall be communicated to the Licensee via email or within the Software itself.
- **Severability**: If any provision of this Agreement is found to be unenforceable, the remaining provisions shall remain in full force and effect.

16.11 AGREEMENT ACCEPTANCE

By checking the “I agree” box, Licensee acknowledges and agrees to be bound by the terms of this Software License Agreement. If Licensee does not agree to the terms, they should not install or use the Software.

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@nxplasma.com****.*