Hypertherm®

Phoenix™ Software Version 9 Series

Installation and Setup Manual

806410 | Revision 11 | English

Phoenix Software® Version 9 Series

Installation and Setup Manual

806410
Revision 11

English

December 2014

Hypertherm Inc.
Hanover, NH 03755 USA

Hypertherm Inc.
Etna Road, P.O. Box 5010
Hanover, NH 03755 USA
603-643-3441 Tel (Main Office)
603-643-5352 Fax (All Departments)
info@hypertherm.com (Main Office Email)
800-643-9878 Tel (Technical Service)
technical.service@hypertherm.com (Technical Service Email)
800-737-2978 Tel (Customer Service)
customer.service@hypertherm.com (Customer Service Email)
866-643-7711 Tel (Return Materials Authorization)
877-371-2876 Fax (Return Materials Authorization)
return.materials@hypertherm.com (RMA email)

Hypertherm Plasmatechnik GmbH
Technologiepark Hanau
Rodenbacher Chaussee 6
D-63457 Hanau-Wolfgang, Deutschland
49 6181 58 2100 Tel
49 6181 58 2134 Fax
49 6181 58 2123 (Technical Service)

Hypertherm (S) Pte Ltd.
82 Genting Lane
Media Centre
Annexe Block #A01-01
Singapore 349567, Republic of Singapore
65 6841 2489 Tel
65 6841 2490 Fax
65 6841 2489 (Technical Service)

Hypertherm (Shanghai) Trading Co., Ltd.
Unit 301, South Building
495 ShangZhong Road
Shanghai, 200231
PR China
86-21-60740003 Tel
86-21-60740393 Fax

Hypertherm Europe B.V.
Vaartveld 9
4704 SE
Roosendaal, Nederland
31 165 596907 Tel
31 165 596901 Fax
31 165 596908 Tel (Marketing)
31 165 596900 Tel (Technical Service)
00 800 4973 7843 Tel (Technical Service)

Hypertherm Japan Ltd.
Level 9, Edobori Center Building
2-1-1 Edobori, Nishi-ku
Osaka 550-0002 Japan
81 6 6225 1183 Tel
81 6 6225 1184 Fax

Hypertherm Brasil Ltda.
Rua Bras Cubas, 231 – Jardim Maia
Guarulhos, SP - Brasil
CEP 07115-030
55 11 2409 2636 Tel
55 11 2408 0462 Fax

Hypertherm México, S.A. de C.V.
Avenida Toluca No. 444, Anexo 1,
Colonia Olivar de los Padres
Delegación Álvaro Obregón
México, D.F. C.P. 01780
52 55 5681 8109 Tel
52 55 5683 2127 Fax

Hypertherm Korea Branch
#3904 Centum Leaders Mark B/D,
1514 Woo-dong, Haeundae-gu, Busan
Korea, 612-889
82 51 747 0358 Tel
82 51 701 0358 Fax

Contents

RECOGNIZE SAFETY INFORMATION

The symbols shown in this section are used to identify potential hazards. When you see a safety symbol in this manual or on your machine, understand the potential for personal injury, and follow the related instructions to avoid the hazard.

FOLLOW SAFETY INSTRUCTIONS

Read carefully all safety messages in this manual and safety labels on your machine.

- Keep the safety labels on your machine in good condition. Replace missing or damaged labels immediately.
- Learn how to operate the machine and how to use the controls properly. Do not let anyone operate it without instruction.
- Keep your machine in proper working condition. Unauthorized modifications to the machine may affect safety and machine service life.

DANGER WARNING CAUTION

Hypertherm uses American National Standards Institute guidelines for safety signal words and symbols. A signal word DANGER or WARNING is used with a safety symbol. DANGER identifies the most serious hazards.

- DANGER and WARNING safety labels are located on your machine near specific hazards.
- DANGER safety messages precede related instructions in the manual that will result in serious injury or death if not followed correctly.
- WARNING safety messages precede related instructions in this manual that may result in injury or death if not followed correctly.
- CAUTION safety messages precede related instructions in this manual that may result in minor injury or damage to equipment if not followed correctly.

ELECTRICAL HAZARDS

- Only trained and authorized personnel may open this equipment.
- If the equipment is permanently connected, turn it off, and lock out/tag out power before the enclosure is opened.
- If power is supplied to the equipment with a cord, unplug the unit before the enclosure is opened.
- Lockable disconnects or lockable plug covers must be provided by others.
- Wait 5 minutes after removal of power before entering the enclosure to allow stored energy to discharge.
- If the equipment must have power when the enclosure is open for servicing, arc flash explosion hazards may exist. Follow ALL local requirements (NFPA 70E in the USA) for safe work practices and for Personal Protective Equipment when servicing energized equipment.
- The enclosure shall be closed and the proper earth ground continuity to the enclosure verified prior to operating the equipment after moving, opening, or servicing.
- Always follow these instructions for disconnecting power before inspecting or changing torch consumable parts.

ELECTRIC SHOCK CAN KILL

Touching live electrical parts can cause a fatal shock or severe burn.

- Operating the plasma system completes an electrical circuit between the torch and the workpiece. The workpiece and anything touching the workpiece are part of the electrical circuit.
- Never touch the torch body, workpiece or the water in a water table when the plasma system is operating.

Electric shock prevention

All Hypertherm plasma systems use high voltage in the cutting process (200 to 400 VDC are common). Take the following precautions when operating this system:

- Wear insulated gloves and boots, and keep your body and clothing dry.
- Do not stand, sit or lie on – or touch – any wet surface when using the plasma system.
- Insulate yourself from work and ground using dry insulating mats or covers big enough to prevent any physical contact with the work or ground. If you must work in or near a damp area, use extreme caution.
- Provide a disconnect switch close to the power supply with properly sized fuses. This switch allows the operator to turn off the power supply quickly in an emergency situation.
- When using a water table, be sure that it is correctly connected to earth ground.
- Install and ground this equipment according to the instruction manual and in accordance with national and local codes.
- Inspect the input power cord frequently for damage or cracking of the cover. Replace a damaged power cord immediately. **Bare wiring can kill.**
- Inspect and replace any worn or damaged torch leads.
- Do not pick up the workpiece, including the waste cutoff, while you cut. Leave the workpiece in place or on the workbench with the work cable attached during the cutting process.
- Before checking, cleaning or changing torch parts, disconnect the main power or unplug the power supply.
- Never bypass or shortcut the safety interlocks.
- Before removing any power supply or system enclosure cover, disconnect electrical input power. Wait 5 minutes after disconnecting the main power to allow capacitors to discharge.
- Never operate the plasma system unless the power supply covers are in place. Exposed power supply connections present a severe electrical hazard.
- When making input connections, attach proper grounding conductor first.
- Each Hypertherm plasma system is designed to be used only with specific Hypertherm torches. Do not substitute other torches which could overheat and present a safety hazard.

CUTTING CAN CAUSE FIRE OR EXPLOSION

Fire prevention

- Be sure the area is safe before doing any cutting. Keep a fire extinguisher nearby.
- Remove all flammables within 35 feet (10 m) of the cutting area.
- Quench hot metal or allow it to cool before handling or before letting it touch combustible materials.
- Never cut containers with potentially flammable materials inside – they must be emptied and properly cleaned first.
- Ventilate potentially flammable atmospheres before cutting.
- When cutting with oxygen as the plasma gas, an exhaust ventilation system is required.

Explosion prevention

- Do not use the plasma system if explosive dust or vapors may be present.
- Do not cut pressurized cylinders, pipes, or any closed container.
- Do not cut containers that have held combustible materials.

WARNING
Explosion Hazard
Argon-Hydrogen and Methane

Hydrogen and methane are flammable gases that present an explosion hazard. Keep flames away from cylinders and hoses that contain methane or hydrogen mixtures. Keep flames and sparks away from the torch when using methane or argon-hydrogen plasma.

WARNING
Hydrogen Detonation with
Aluminum Cutting

- Do not cut aluminum underwater or with water touching the underside of the aluminum.
- Cutting aluminum underwater or with the water touching the underside of the aluminum can result in an explosive condition that can detonate during plasma cutting operations.

WARNING
Explosion Hazard
Underwater Cutting with Fuel Gases

- Do not cut under water with fuel gases containing hydrogen.
- Cutting under water with fuel gases containing hydrogen can result in an explosive condition that can detonate during plasma cutting operations.

TOXIC FUMES CAN CAUSE INJURY OR DEATH

The plasma arc by itself is the heat source used for cutting. Accordingly, although the plasma arc has not been identified as a source of toxic fumes, the material being cut can be a source of toxic fumes or gases that deplete oxygen.

Fumes produced vary depending on the metal that is cut. Metals that may release toxic fumes include, but are not limited to, stainless steel, carbon steel, zinc (galvanized), and copper.

In some cases, the metal may be coated with a substance that could release toxic fumes. Toxic coatings include, but are not limited to, lead (in some paints), cadmium (in some paints and fillers), and beryllium.

Gases produced by plasma cutting vary based on the material to be cut and the method of cutting, but may include ozone, oxides of nitrogen, hexavalent chromium, hydrogen, and other substances if such are contained in or released by the material being cut.

Caution should be taken to minimize exposure to fumes produced by any industrial process. Depending upon the chemical composition and concentration of the fumes (as well as other factors, such as ventilation), there may be a risk of physical illness, such as birth defects or cancer.

It is the responsibility of the equipment and site owner to test the air quality in the area where the equipment is used and to ensure that the air quality in the workplace meets all local and national standards and regulations.

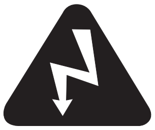
The air quality level in any relevant workplace depends on site-specific variables such as:

- Table design (wet, dry, underwater).
- Material composition, surface finish, and composition of coatings.
- Volume of material removed.
- Duration of cutting or gouging.
- Size, air volume, ventilation and filtration of the work area.
- Personal protective equipment.
- Number of welding and cutting systems in operation.
- Other site processes that may produce fumes.

If the workplace must conform to national or local regulations, only monitoring or testing done at the site can determine whether the site is above or below allowable levels.

To reduce the risk of exposure to fumes:

- Remove all coatings and solvents from the metal before cutting.
- Use local exhaust ventilation to remove fumes from the air.
- Do not inhale fumes. Wear an air-supplied respirator when cutting any metal coated with, containing, or suspected to contain toxic elements.
- Assure that those using welding or cutting equipment, as well as air-supplied respiration devices, are qualified and trained in the proper use of such equipment.
- Never cut containers with potentially toxic materials inside. Empty and properly clean the container first.
- Monitor or test the air quality at the site as needed.
- Consult with a local expert to implement a site plan to ensure safe air quality.



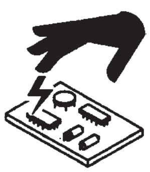
GROUNDING SAFETY

Work cable Attach the work cable securely to the workpiece or the work table with good metal-to-metal contact. Do not connect it to the piece that will fall away when the cut is complete.

Work table Connect the work table to an earth ground, in accordance with appropriate national and local electrical codes.

Input power

- Be sure to connect the power cord ground wire to the ground in the disconnect box.
- If installation of the plasma system involves connecting the power cord to the power supply, be sure to connect the power cord ground wire properly.
- Place the power cord's ground wire on the stud first, then place any other ground wires on top of the power cord ground. Fasten the retaining nut tightly.
- Tighten all electrical connections to avoid excessive heating.



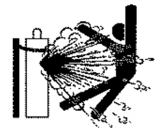
STATIC ELECTRICITY CAN DAMAGE CIRCUIT BOARDS

Use proper precautions when handling printed circuit boards:

- Store PC boards in anti-static containers.
- Wear a grounded wrist strap when handling PC boards.

COMPRESSED GAS EQUIPMENT SAFETY

- Never lubricate cylinder valves or regulators with oil or grease.
- Use only correct gas cylinders, regulators, hoses and fittings designed for the specific application.
- Maintain all compressed gas equipment and associated parts in good condition.
- Label and color-code all gas hoses to identify the type of gas in each hose. Consult applicable national and local codes.



GAS CYLINDERS CAN EXPLODE IF DAMAGED

Gas cylinders contain gas under high pressure.
If damaged, a cylinder can explode.

- Handle and use compressed gas cylinders in accordance with applicable national and local codes.
- Never use a cylinder that is not upright and secured in place.
- Keep the protective cap in place over valve except when the cylinder is in use or connected for use.
- Never allow electrical contact between the plasma arc and a cylinder.
- Never expose cylinders to excessive heat, sparks, slag or open flame.
- Never use a hammer, wrench or other tool to open a stuck cylinder valve.

A PLASMA ARC CAN CAUSE INJURY AND BURNS

Instant-on torches

Plasma arc comes on immediately when the torch switch is activated.

The plasma arc will cut quickly through gloves and skin.

- Keep away from the torch tip.
- Do not hold metal near the cutting path.
- Never point the torch toward yourself or others.

ARC RAYS CAN BURN EYES AND SKIN

Eye protection Plasma arc rays produce intense visible and invisible (ultraviolet and infrared) rays that can burn eyes and skin.

- Use eye protection in accordance with applicable national and local codes.
- Wear eye protection (safety glasses or goggles with side shields, and a welding helmet) with appropriate lens shading to protect your eyes from the arc's ultraviolet and infrared rays.

Skin protection Wear protective clothing to protect against burns caused by ultraviolet light, sparks, and hot metal.

- Gauntlet gloves, safety shoes and hat.
- Flame-retardant clothing to cover all exposed areas.
- Cuffless trousers to prevent entry of sparks and slag.
- Remove any combustibles, such as a butane lighter or matches, from your pockets before cutting.

Cutting area Prepare the cutting area to reduce reflection and transmission of ultraviolet light:

- Paint walls and other surfaces with dark colors to reduce reflection.
- Use protective screens or barriers to protect others from flash and glare.
- Warn others not to watch the arc. Use placards or signs.

Arc current (amps)	Minimum protective shade number (ANSI Z49.1:2005)	Suggested shade number for comfort (ANSI Z49.1:2005)	OSHA 29CFR 1910.133(a)(5)	Europe EN168:2002
Less than 40 A	5	5	8	9
41 to 60 A	6	6	8	9
61 to 80 A	8	8	8	9
81 to 125 A	8	9	8	9
126 to 150 A	8	9	8	10
151 to 175 A	8	9	8	11
176 to 250 A	8	9	8	12
251 to 300 A	8	9	8	13
301 to 400 A	9	12	9	13
401 to 800 A	10	14	10	N/A

PACEMAKER AND HEARING AID OPERATION

Pacemaker and hearing aid operation can be affected by magnetic fields from high currents.

Pacemaker and hearing aid wearers should consult a doctor before going near any plasma arc cutting and gouging operations.

To reduce magnetic field hazards:

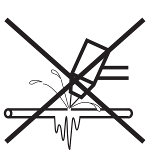
- Keep both the work cable and the torch lead to one side, away from your body.
- Route the torch leads as close as possible to the work cable.
- Do not wrap or drape the torch lead or work cable around your body.
- Keep as far away from the power supply as possible.

NOISE CAN DAMAGE HEARING

Cutting with a plasma arc can exceed acceptable noise levels as defined by local codes in many applications. Prolonged exposure to excessive noise can damage hearing. Always wear proper ear protection when cutting or gouging, unless sound pressure level measurements taken at the installed site have verified personal hearing protection is not necessary per relevant international, regional, and local codes.

Significant noise reduction can be obtained by adding simple engineering controls to cutting tables such as barriers or curtains positioned between the plasma arc and the workstation; and/or locating the workstation away from the plasma arc. Implement administrative controls in the workplace to restrict access, limit operator exposure time, screen off noisy working areas and/or take measures to reduce reverberation in working areas by putting up noise absorbers.

Use ear protectors if the noise is disruptive or if there is a risk of hearing damage after all other engineering and administrative controls have been implemented. If hearing protection is required, wear only approved personal protective devices such as ear muffs or ear plugs with a noise reduction rating appropriate for the situation. Warn others in the area of possible noise hazards. In addition, ear protection can prevent hot splatter from entering the ear.



A PLASMA ARC CAN DAMAGE FROZEN PIPES

Frozen pipes may be damaged or can burst if you attempt to thaw them with a plasma torch.

DRY DUST COLLECTION INFORMATION

At some sites, dry dust can represent a potential explosion hazard.

The U.S. National Fire Protection Association's 2007 edition of NFPA standard 68, "Explosion Protection by Deflagration Venting," provides requirements for the design, location, installation, maintenance, and use of devices and systems to vent combustion gases and pressures after any deflagration event. Consult with the manufacturer or installer of any dry dust collection system for applicable requirements before you install a new dry dust collection system or make significant changes in the process or materials used with an existing dry dust collection system.

Consult your local "Authority Having Jurisdiction" (AHJ) to determine whether any edition of NFPA 68 has been "adopted by reference" in your local building codes.

Refer to NFPA68 for definitions and explanations of regulatory terms such as deflagration, AHJ, adopted by reference, the Kst value, deflagration index, and other terms.

Note 1 – Hypertherm's interpretation of these new requirements is that unless a site-specific evaluation has been completed to determine that all dust generated is not combustible, the 2007 edition of NFPA 68 requires the use of explosion vents designed to the worst-case Kst value (see annex F) that could be generated from dust so that the explosion vent size and type can be designed. NFPA 68 does not specifically identify plasma cutting or other thermal cutting processes as requiring deflagration venting systems, but it does apply these new requirements to all dry dust collection systems.

Note 2 – Users of Hypertherm manuals should consult and comply with all applicable federal, state, and local laws and regulations. Hypertherm does not, by the publication of any Hypertherm manual, intend to urge action that is not in compliance with all applicable regulations and standards, and this manual may never be construed as doing so.

LASER RADIATION

Exposure to the laser output can result in serious eye injury. Avoid direct eye exposure.

For your convenience and safety, on Hypertherm products that use a laser, one of the following laser radiation labels has been applied on the product near where the laser beam exits the enclosure. The maximum output (mV), wavelength emitted (nM) and, if appropriate, the pulse duration is also provided.

Additional laser safety instructions:

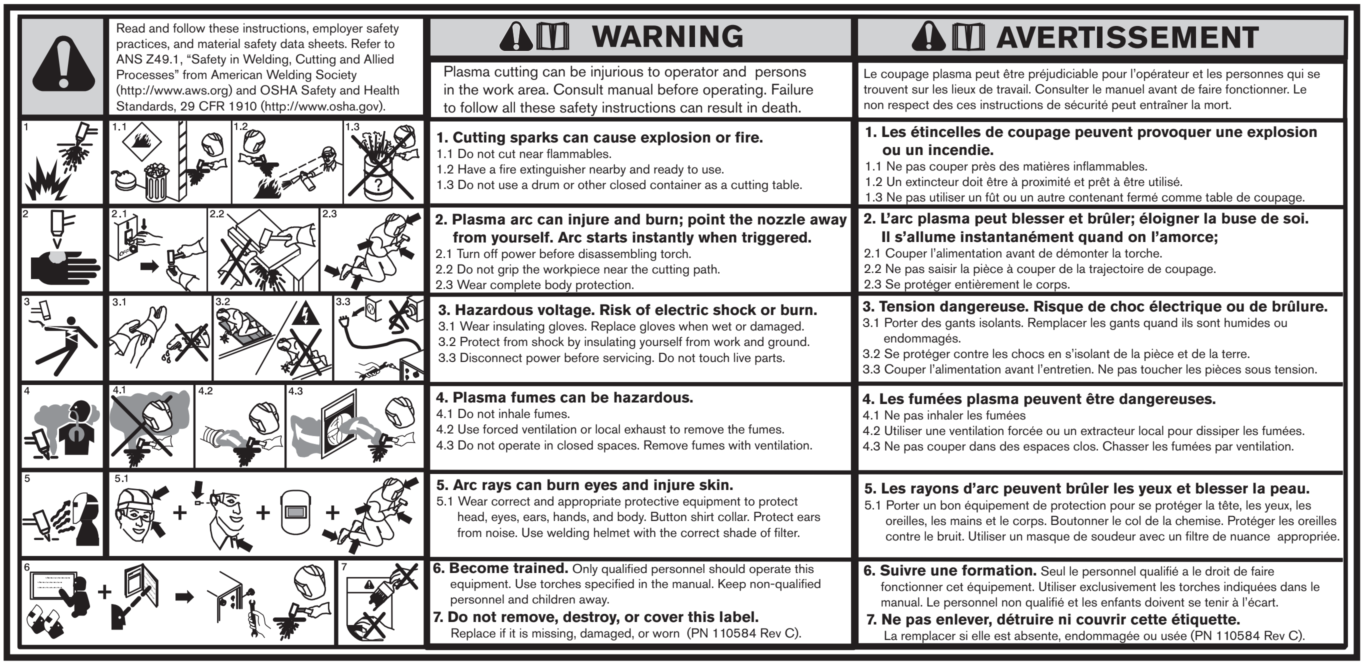
- Consult with an expert on local laser regulations. Laser safety training may be required.
- Do not allow untrained persons to operate the laser. Lasers can be dangerous in the hands of untrained users.
- Do not look into the laser aperture or beam at any time.
- Position the laser as instructed to avoid unintentional eye contact.
- Do not use the laser on reflective workpieces.
- Do not use optical tools to view or reflect the laser beam.
- Do not disassemble or remove the laser or aperture cover.
- Modifying the laser or product in any way can increase the risk of laser radiation.
- Use of adjustments or performance of procedures other than those specified in this manual may result in hazardous laser radiation exposure.
- Do not operate in explosive atmospheres, such as in the presence of flammable liquids, gases, or dust.
- Use only laser parts and accessories that are recommended or provided by the manufacturer for your model.
- Repairs and servicing MUST be performed by qualified personnel.
- Do not remove or deface the laser safety label.

ADDITIONAL SAFETY INFORMATION

1. ANSI Standard Z49.1, Safety in Welding and Cutting, American Welding Society, 550 LeJeune Road P.O. Box 351020, Miami, FL 33135
2. ANSI Standard Z49.2, Fire Prevention in the Use of Cutting and Welding Processes, American National Standards Institute 1430 Broadway, New York, NY 10018
3. ANSI Standard Z87.1, Safe Practices for Occupation and Educational Eye and Face Protection, American National Standards Institute, 1430 Broadway, New York, NY 10018
4. AWS F4.1, Recommended Safe Practices for the Preparation for Welding and Cutting of Containers and Piping That Have Held Hazardous Substances, American Welding Society 550 LeJeune Road, P.O. Box 351040, Miami, FL 33135
5. AWS F5.2, Recommended Safe Practices for Plasma Arc Cutting, American Welding Society 550 LeJeune Road, P.O. Box 351040, Miami, FL 33135
6. CGA Pamphlet P-1, Safe Handling of Compressed Gases in Cylinders, Compressed Gas Association 1235 Jefferson Davis Highway, Arlington, VA 22202
7. CSA Standard W117.2, Code for Safety in Welding and Cutting, Canadian Standards Association Standard Sales 178 Rexdale Boulevard, Rexdale, Ontario M9W 1R3, Canada
8. NFPA Standard 51B, Cutting and Welding Processes, National Fire Protection Association 470 Atlantic Avenue, Boston, MA 02210
9. NFPA Standard 70–1978, National Electrical Code, National Fire Protection Association, 470 Atlantic Avenue, Boston, MA 02210
10. OSHA, Safety and Health Standards, 29FR 1910 U.S. Government Printing Office, Washington, D.C. 20402
11. AWS Safety and Health Fact Sheets, American Welding Society 550 LeJeune Road, P.O. Box 351040, Miami, FL 33135 www.aws.org/technical/facts/

WARNING LABELS

This warning label is affixed to some power supplies. It is important that the operator and maintenance technician understand the intent of these warning symbols as described.



Warning labels

This warning label is affixed to some power supplies. It is important that the operator and maintenance technician understand the intent of these warning symbols as described. The numbered text corresponds to the numbered boxes on the label.

1. Cutting sparks can cause explosion or fire.

1.1 Do not cut near flammables.

1.2 Have a fire extinguisher nearby and ready to use.

1.3 Do not use a drum or other closed container as a cutting table.

2. Plasma arc can injure and burn; point the nozzle away from yourself. Arc starts instantly when triggered.

2.1 Turn off power before disassembling torch.

2.2 Do not grip the workpiece near the cutting path.

2.3 Wear complete body protection.

3. Hazardous voltage. Risk of electric shock or burn.

3.1 Wear insulating gloves. Replace gloves when wet or damaged.

3.2 Protect from shock by insulating yourself from work and ground.

3.3 Disconnect power before servicing. Do not touch live parts.

4. Plasma fumes can be hazardous.

4.1 Do not inhale fumes.

4.2 Use forced ventilation or local exhaust to remove the fumes.

4.3 Do not operate in closed spaces. Remove fumes with ventilation.

5. Arc rays can burn eyes and injure skin.

5.1 Wear correct and appropriate protective equipment to protect head, eyes, ears, hands, and body. Button shirt collar. Protect ears from noise. Use welding helmet with the correct shade of filter.

6. Become trained. Only qualified personnel should operate this equipment. Use torches specified in the manual. Keep non-qualified personnel and children away.

7. Do not remove, destroy, or cover this label. Replace if it is missing, damaged, or worn.

Symbols and marks

Your product may have one or more of the following markings on or near the data plate. Due to differences and conflicts in national regulations, not all marks are applied to every version of a product.

S mark

The S mark indicates that the power supply and torch are suitable for operations carried out in environments with increased hazard of electrical shock according to IEC 60974-1.

CSA mark

Products with a CSA mark meet the United States and Canadian regulations for product safety. The products were evaluated, tested, and certified by CSA-International. Alternatively, the product may have a mark by one of the other Nationally Recognized Testing Laboratories (NRTL) accredited in both the United States and Canada, such as UL or TÜV.

CE mark

The CE marking signifies the manufacturer's declaration of conformity to applicable European directives and standards. Only those versions of products with a CE marking located on or near the data plate have been tested for compliance with the European Low Voltage Directive and the European Electromagnetic Compatibility (EMC) Directive. EMC filters needed to comply with the European EMC Directive are incorporated within versions of the product with a CE marking.

Eurasian Customs Union (CU) mark

CE versions of products that include an EAC mark of conformity meet the product safety and EMC requirements for export to Russia, Belarus, and Kazakhstan.

GOST-TR mark

CE versions of products that include a GOST-TR mark of conformity meet the product safety and EMC requirements for export to the Russian Federation.

C-Tick mark

N30932

CE versions of products with a C-Tick mark comply with the EMC regulations required for sale in Australia and New Zealand.

CCC mark

The China Compulsory Certification (CCC) mark indicates that the product has been tested and found compliant with product safety regulations required for sale in China.

UkrSEPRO mark

The CE versions of products that include a UkrSEPRO mark of conformity meet the product safety and EMC requirements for export to the Ukraine.

Serbian AAA mark

И005 13

CE versions of products that include a AAA Serbian mark meet the product safety and EMC requirements for export to Serbia.

Introduction

Hypertherm maintains a global Regulatory Management System to ensure that products comply with regulatory and environmental requirements.

National and local safety regulations

National and Local safety regulations shall take precedence over any instructions provided with the product. The product shall be imported, installed, operated and disposed of in accordance with national and local regulations applicable to the installed site.

Certification test marks

Certified products are identified by one or more certification test marks from accredited testing laboratories. The certification test marks are located on or near the data plate.

Each certification test mark means that the product and its safety-critical components conform to the relevant national safety standards as reviewed and determined by that testing laboratory. Hypertherm places a certification test mark on its products only after that product is manufactured with safety-critical components that have been authorized by the accredited testing laboratory.

Once the product has left the Hypertherm factory, the certification test marks are invalidated if any of the following occurs:

- The product is modified in a manner that creates a hazard or non-conformance with the applicable standards.
- Safety-critical components are replaced with unauthorized spare parts.
- Any unauthorized assembly, or accessory that uses or generates a hazardous voltage is added.
- There is any tampering with a safety circuit or other feature that is designed into the product as part of the certification, or otherwise.

CE marking constitutes a manufacturer's declaration of conformity to applicable European directives and standards. Only those versions of Hypertherm products with a CE Marking located on or near the data plate have been tested for compliance with the European Low Voltage Directive and the European EMC Directive. EMC filters needed to comply with the European EMC Directive are incorporated within versions of the power supply with a CE Marking.

Certificates of compliance for Hypertherm products are available from the Downloads Library on the Hypertherm web site at https://www.hypertherm.com.

Differences in national standards

Nations may apply different performance, safety or other standards. National differences in standards include, but are not limited to:

- Voltages
- Plug and cord ratings
- Language requirements
- Electromagnetic compatibility requirements

These differences in national or other standards may make it impossible or impractical for all certification test marks to be placed on the same version of a product. For example, the CSA versions of Hypertherm's products do not comply with European EMC requirements, and therefore do not have a CE marking on the data plate.

Countries that require CE marking or have compulsory EMC regulations must use CE versions of Hypertherm products with the CE marking on the data plate. These include, but are not limited to:

- Australia
- New Zealand
- Countries in the European Union
- Russia

It is important that the product and its certification test mark be suitable for the end-use installation site. When Hypertherm products are shipped to one country for export to another country; the product must be configured and certified properly for the end-use site.

Safe installation and use of shape cutting equipment

IEC 60974-9, titled Arc Welding Equipment – Installation and use, provides guidance in the safe installation and use of shape cutting equipment and the safe performance of cutting operations. The requirements of national and local regulations shall be taken into consideration during installation, including, but not limited to, grounding or protective earth connections, fuses, supply disconnecting device, and type of supply circuit. Read these instructions before installing the equipment. The first and most important step is the safety assessment of the installation.

The safety assessment must be performed by an expert, and determines what steps are necessary to create a safe environment, and what precautions should be adopted during the actual installation and operation.

Procedures for periodic inspection and testing

Where required by local national regulations, IEC 60974-4 specifies test procedures for periodic inspection and after repair or maintenance, to ensure electrical safety for plasma cutting power sources built in conformity with IEC 60974-1. Hypertherm performs the continuity of the protective circuit and insulation resistance tests in the factory as non-operating tests. The tests are performed with the power and ground connections removed.

Hypertherm also removes some protective devices that would cause false test results. Where required by local national regulations, a label shall be attached to the equipment to indicate that it has passed the tests prescribed by IEC 60974-4. The repair report shall indicate the results of all tests unless an indication is made that a particular test has not been performed.

Qualification of test personnel

Electrical safety tests for shape cutting equipment can be hazardous and shall be carried out by an expert in the field of electrical repair, preferably someone also familiar with welding, cutting, and allied processes. The safety risks to personnel and equipment, when unqualified personnel are performing these tests, may be much greater than the benefit of periodic inspection and testing.

Hypertherm recommends that only visual inspection be performed unless the electrical safety tests are specifically required by local national regulations in the country where the equipment is installed.

Residual current devices (RCDs)

In Australia and some other countries, local codes may require the use of a Residual Current Devices (RCD) when portable electrical equipment is used in the workplace or at construction sites to protect operators from electrical faults in the equipment. RCDs are designed to safely disconnect the mains electrical supply when an imbalance is detected between the supply and return current (there is a leakage current to earth). RCDs are available with both fixed and adjustable trip currents between 6 to 40 milliamperes and a range of trip times up to 300 milliseconds selected for the equipment installation, application and intended use. Where RCDs are used, the trip current and trip time on RCDs should be selected or adjusted high enough to avoid nuisance tripping during normal operation of the plasma cutting equipment and low enough in the extremely unlikely event of an electrical fault in the equipment to disconnect the supply before the leakage current under a fault condition can pose a life threatening electrical hazard to operators.

To verify that the RCDs continue to function properly over time, both the trip current and the trip time should be tested periodically. Portable electrical equipment and RCDs used in commercial and industrial areas in Australia and New Zealand are tested to the Australian standard AS/NZS 3760. When you test the insulation of plasma cutting equipment to AS/NZS 3760, measure the insulation resistance according to Appendix B of the standard, at 250 VDC with the power switch in the ON position to verify proper testing and to avoid the false failure of the leakage current test. False failures are possible because the metal oxide varistors (MOVs) and electromagnetic compatibility (EMC) filters, used to reduce emissions and protect the equipment from power surges, may conduct up to 10 milliamperes leakage current to earth under normal conditions.

If you have any questions regarding the application or interpretation of any IEC standards described here, you are required to consult with an appropriate legal or other advisor familiar with the International Electrotechnical standards, and shall not rely on Hypertherm in any respect regarding the interpretation or application of such standards.

Higher-level systems

When a system integrator adds additional equipment; such as cutting tables, motor drives, motion controllers or robots; to a Hypertherm plasma cutting system, the combined system may be considered a higher-level system. A higher-level system with hazardous moving parts may constitute industrial machinery or robotic equipment, in which case the OEM or end-use customer may be subject to additional regulations and standards than those relevant to the plasma cutting system as manufactured by Hypertherm.

It is the responsibility of the end-use customer and the OEM to perform a risk assessment for the higher-level system, and to provide protection against hazardous moving parts. Unless the higher-level system is certified when the OEM incorporates Hypertherm products into it, the installation also may be subject to approval by local authorities. Seek advice from legal counsel and local regulatory experts if you are uncertain about compliance.

External interconnecting cables between component parts of the higher level system must be suitable for contaminants and movement as required by the final end use installation site. When the external interconnecting cables are subject to oil, dust, water, or other contaminants, hard usage ratings may be required.

When external interconnecting cables are subject to continuous movement, constant flexing ratings may be required. It is the responsibility of the end-use customer or the OEM to ensure the cables are suitable for the application. Since there are differences in the ratings and costs that can be required by local regulations for higher level systems, it is necessary to verify that any external interconnecting cables are suitable for the end-use installation site.

Introduction

The Hypertherm Environmental Specification requires RoHS, WEEE and REACH substance information to be provided by Hypertherm's suppliers.

Product environmental compliance does not address the indoor air quality or environmental release of fumes by the end user. Any materials that are cut by the end user are not provided by Hypertherm with the product. The end user is responsible for the materials being cut as well as for safety and air quality in the workplace. The end user must be aware of the potential health risks of the fumes released from the materials being cut and comply with all local regulations.

National and local environmental regulations

National and local environmental regulations shall take precedence over any instructions contained in this manual.

The product shall be imported, installed, operated and disposed of in accordance with all national and local environmental regulations applicable to the installed site.

The European Environmental regulations are discussed later in *The WEEE Directive*.

The RoHS directive

Hypertherm is committed to complying with all applicable laws and regulations, including the European Union Restriction of Hazardous Substances (RoHS) Directive that restricts the use of hazardous materials in electronics products. Hypertherm exceeds RoHS Directive compliance obligations on a global basis.

Hypertherm continues to work toward the reduction of RoHS materials in our products, which are subject to the RoHS Directive, except where it is widely recognized that there is no feasible alternative.

Declarations of RoHS Conformity have been prepared for the current CE versions of Powermax plasma cutting systems manufactured by Hypertherm. There is also a "RoHS mark" on the Powermax CE versions below the "CE Marking" on the data plate of CE versions of Powermax series units shipped since 2006. Parts used in CSA versions of Powermax and other products manufactured by Hypertherm that are either out of scope or exempt from RoHS are continuously being converted to RoHS compliance in anticipation of future requirements.

Proper disposal of Hypertherm products

Hypertherm plasma cutting systems, like all electronic products, may contain materials or components, such as printed circuit boards, that cannot be discarded with ordinary waste. It is your responsibility to dispose of any Hypertherm product or component part in an environmentally acceptable manner according to national and local codes.

- In the United States, check all federal, state, and local laws.
- In the European Union, check the EU directives, national, and local laws. For more information, visit www.hypertherm.com/weee.
- In other countries, check national and local laws.
- Consult with legal or other compliance experts when appropriate.

The WEEE directive

On January 27, 2003, the European Parliament and the Council of the European Union authorized Directive 2002/96/EC or WEEE (Waste Electrical and Electronic Equipment).

As required by the legislation, any Hypertherm product covered by the directive and sold in the EU after August 13, 2005 is marked with the WEEE symbol. This directive encourages and sets specific criteria for the collection, handling, and recycling of EEE waste. Consumer and business-to-business wastes are treated differently (all Hypertherm products are considered business-to-business). Disposal instructions for the CE versions of Powermax plasma systems can be found at www.hypertherm.com/weee.

The URL is printed on the symbol-only warning label for each of these CE version Powermax series units shipped since 2006. The CSA versions of Powermax and other products manufactured by Hypertherm are either out of scope or exempt from WEEE.

The REACH regulation

The REACH regulation (1907/2006), in force since June 1, 2007, has an impact on chemicals available to the European market. The REACH regulation requirements for component manufacturers states that the component shall not contain more than 0.1% by weight of the Substances of Very High Concern (SVHC).

Component manufacturers and other downstream users, such as Hypertherm, are obligated to obtain assurances from its suppliers that all chemicals used in or on Hypertherm products will have a European Chemical Agency (ECHA) registration number. To provide chemical information as required by the REACH regulation, Hypertherm requires suppliers to provide REACH declarations and identify any known use of REACH SVHC. Any use of SVHC in amounts exceeding 0.1% w/w of the parts has been eliminated. The MSDS contains a full disclosure of all substances in the chemical and can be used to verify REACH SVHC compliance.

The lubricants, sealants, coolants, adhesives, solvents, coatings and other preparations or mixtures used by Hypertherm in, on, for, or with its shape cutting equipment are used in very small quantities (except the coolant) and are commercially available with multiple sources that can and will be replaced in the event of a supplier problem associated with REACH Registration or REACH Authorization (SVHCs).

Proper handling and safe use of chemicals

Chemical Regulations in the USA, Europe, and other locations require that Material Safety Data Sheets (MSDS) be made available for all chemicals. The list of chemicals is provided by Hypertherm. The MSDS are for chemicals provided with the product and other chemicals used in or on the product. MSDS can be downloaded from the Downloads Library on the Hypertherm web site at https://www.hypertherm.com. On the Search screen, insert MSDS in the document title and click on Search.

In the USA, OSHA does not require Material Safety Data Sheets for articles such as electrodes, swirl rings, retaining caps, nozzles, shields, deflectors and other solid parts of the torch.

Hypertherm does not manufacture or provide the materials that are cut and has no knowledge whether the fumes released from materials that are cut will pose a physical hazard or health risk. Please consult with your supplier or other technical advisor if you need guidance concerning the properties of the material you will cut using a Hypertherm product.

Fumes emission and air quality

Note: The following information on air quality is intended for general information only and should not be used as a substitute for reviewing and implementing applicable government regulations or legal standards in the country where the cutting equipment will be installed and operated.

In the USA, the National Institute for Occupational Safety and Health (NIOSH) Manual of Analytical Methods (NMAM) is a collection of methods for sampling and analyzing contaminants in workplace air. Methods published by others, such as OSHA, MSHA, EPA, ASTM, ISO or commercial suppliers of sampling and analytical equipment, may have advantages over NIOSH methods.

For example, ASTM Practice D 4185 is a standard practice for the collection, dissolution, and determination of trace metals in workplace atmospheres. The sensitivity, detection limit, and optimum working concentrations for 23 metals are listed in ASTM D 4185. An industrial hygienist should be used to determine the optimum sampling protocol, considering analytical accuracy, cost, and optimum sample number. Hypertherm uses a third party industrial hygienist to perform and interpret air quality testing results taken by air sampling equipment positioned at operator stations in Hypertherm buildings where plasma cutting tables are installed and operated.

Where applicable, Hypertherm also uses a third party industrial hygienist to obtain air and water permits.

If you are not fully aware and up to date on all applicable government regulations and legal standards for the installation site, you should consult a local expert prior to purchasing, installing, and operating the equipment.

ENTERING INTO THE LICENSE AGREEMENT SET FORTH BELOW (THE "LICENSE AGREEMENT") GIVES YOU THE RIGHT TO USE THE HYPERTHERM TECHNOLOGY AND RELATED SOFTWARE AND EMBODIED THEREIN WITH HYPERTHERM HPR XD PLASMA SYSTEMS.

PLEASE READ THE LICENSE AGREEMENT CAREFULLY BEFORE USING THE SOFTWARE.

YOUR RIGHT TO USE THE HYPERTHERM TECHNOLOGY AND RELATED SOFTWARE EMBODIED THEREIN IS SUBJECT TO YOUR AGREEMENT TO BE BOUND BY THE TERMS AND CONDITIONS OF THE LICENSE AGREEMENT. BY ACTIVATING YOUR CONTROL PLATFORM AND/OR RELATED SOFTWARE PLATFORM, YOU ACKNOWLEDGE YOUR ACCEPTANCE OF THE LICENSE AGREEMENT AND REPRESENT THAT YOU ARE AUTHORIZED TO ENTER INTO THE LICENSE AGREEMENT ON BEHALF OF LICENSEE. IF YOU DO NOT AGREE TO THESE TERMS AND CONDITIONS, HYPERTHERM DOES NOT GRANT YOU THE RIGHT TO USE THE HYPERTHERM TECHNOLOGY OR RELATED SOFTWARE.

1. Certain definitions: "Designated Hypertherm Patents" shall mean United States Patent Application Nos. 12/341,731, 12/466,786, and 12/557,920, including foreign equivalents, and any patents issuing therefrom; "Hypertherm Plasma Systems" shall mean Hypertherm HPR XD plasma systems, including 130, 260 and 400 amp systems; "Hypertherm Technology" shall mean Hypertherm's proprietary hole cutting technology, including know-how, specifications, inventions, methods, procedures, algorithms, software, programs, works of authorship and other information, documentation and materials for use in programming and operating an automated high temperature thermal cutting system; "Controller Platform" shall mean Hypertherm computer numerical controller and/or MTC software platform supplied with this license; and "End User Customer(s)" shall mean an entity licensed to use the Hypertherm Technology for such entity's own internal business purposes and not for distribution to others.
2. The End User Customer shall be granted a non-exclusive, non-transferable, personal license, without the right to sublicense, to use the Hypertherm Technology, for internal business purposes only, solely as incorporated within the Controller Platform and solely for use in conjunction with Hypertherm Plasma Systems.
3. The End User Customer shall be granted a non-exclusive, non-transferable, personal, royalty-free license, without the right to sublicense, under the Designated Hypertherm Patents solely to the extent necessary to enable the End User Customer to exercise the rights granted under Paragraph 2, above. The License Agreement shall provide that, except for the rights expressly granted to the End User Customer in the License Agreement, the license under the Designated Hypertherm Patents shall not be deemed to grant any license or immunity for combining the Hypertherm Technology with other items or for the use of such combination.
4. The licenses granted to the End User Customer under Paragraphs 2 and 3, above, shall expressly be made subject to the following limitations and restrictions, and the End User Customer's agrees that it shall not (and shall not permit any third party to): (a) use or permit the use of the Hypertherm Technology in conjunction with any high temperature thermal cutting systems other than Hypertherm Plasma Systems; (b) remove, alter or obscure any copyright, trademark or other proprietary or restrictive notice or legend on or within the Hypertherm Technology; (c) disclose, sublicense, distribute or otherwise make available the Hypertherm Technology to any third party or permit others to use it; (d) provide timesharing, service bureau, data processing or other services to a third party whereby such third party would obtain the benefits of the Hypertherm Technology for its own end-user purposes through the End User Customer; (e) decompile, disassemble, or otherwise reverse engineer or attempt to deconstruct or discover any source code or underlying ideas or algorithms of the Hypertherm Technology by any means whatsoever; (f) assign, rent, lease, sell or otherwise transfer the Hypertherm Technology; or (g) modify or alter the Hypertherm Technology in any manner whatsoever or create derivative works thereof.
5. The License Agreement shall provide that nothing therein shall be construed as granting the End User Customer any right or license under any intellectual property right of Hypertherm or any of its licensors or suppliers by implication, estoppel or otherwise, except as expressly set forth in the License Agreement.

6. The License Agreement shall provide that Hypertherm shall retain sole and exclusive ownership of the Hypertherm Technology and that the End User Customer shall obtain no rights in the Hypertherm Technology, except for those expressly set forth in the sublicense agreement.
7. The License Agreement shall give Hypertherm the right to terminate the agreement effective immediately upon written notice if the End User Customer breaches any provision of the License Agreement and fails to cure such breach within five (5) days after receiving written notice thereof from Hypertherm.
8. HYPERTHERM, ITS LICENSORS AND SUPPLIERS MAKE NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED, WITH RESPECT TO THE HYPERTHERM TECHNOLOGY OR RELATED SOFTWARE EMBODIED THEREIN, AND DISCLAIM ALL IMPLIED WARRANTIES, INCLUDING, WITHOUT LIMITATION, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. WITHOUT LIMITING THE FOREGOING, NEITHER HYPERTHERM NOR ANY OF ITS LICENSORS OR SUPPLIERS MAKES ANY REPRESENTATION OR WARRANTY REGARDING THE FUNCTIONALITY, RELIABILITY OR PERFORMANCE OF THE HYPERTHERM TECHNOLOGY OR RELATED SOFTWARE EMBODIED THEREIN, OR THE RESULTS TO BE OBTAINED THROUGH THE USE OF THE HYPERTHERM TECHNOLOGY OR RELATED SOFTWARE, OR THAT THE OPERATION OF SUCH HYPERTHERM TECHNOLOGY OR RELATED SOFTWARE WILL BE UNINTERRUPTED OR ERROR-FREE.
9. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL HYPERTHERM, ITS LICENSORS OR SUPPLIERS BE LIABLE FOR ANY INDIRECT, EXEMPLARY, PUNITIVE, CONSEQUENTIAL, INCIDENTAL OR SPECIAL DAMAGES, INCLUDING LOST PROFITS, ARISING OUT OF OR IN CONNECTION WITH THE USE OF THE HYPERTHERM TECHNOLOGY OR RELATED SOFTWARE EMBODIED THEREIN, EVEN IF SUCH PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THE LIMITATION STATED IN THIS SECTION SHALL APPLY REGARDLESS OF THE FORM OF ACTION, WHETHER THE ASSERTED LIABILITY OR DAMAGES ARE BASED ON CONTRACT (INCLUDING, BUT NOT LIMITED TO, BREACH OF WARRANTY), TORT (INCLUDING, BUT NOT LIMITED TO, NEGLIGENCE), STATUTE, OR ANY OTHER LEGAL OR EQUITABLE THEORY.

Section 1

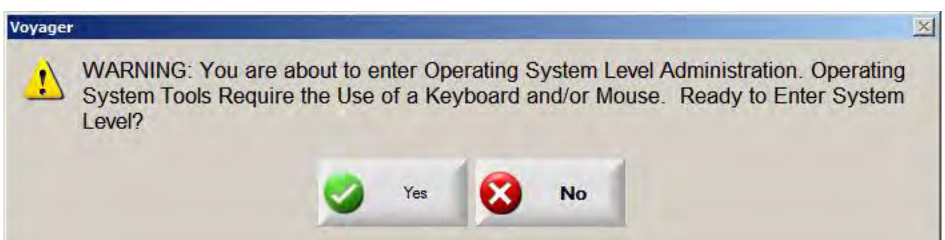
System Tools

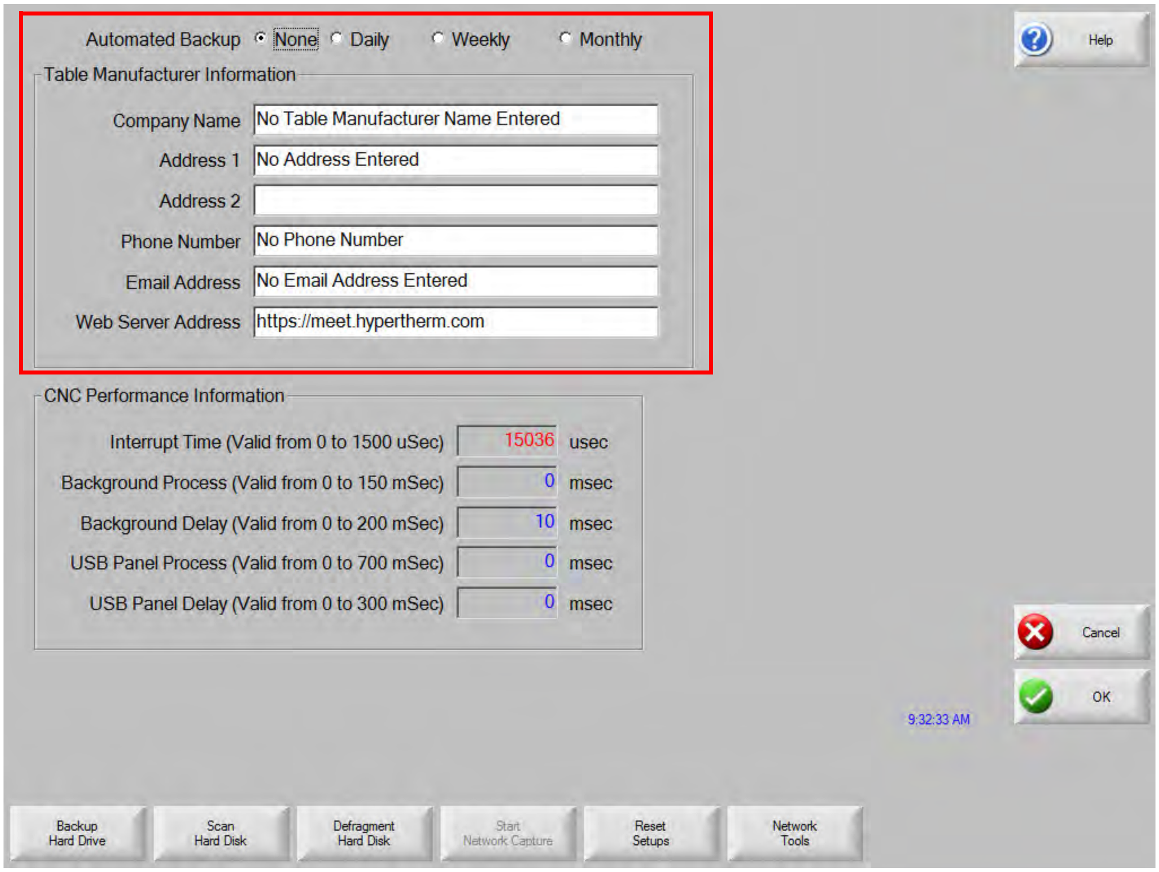
Introduction

The Phoenix software offers a suite of tools for performing system-level tasks such as backing up the CNC hard drive and connecting the CNC to a network. To access the Phoenix system tools, choose **Setups > Password > Special Setups > System.**

	CAUTION
⚠	**Using the System Tools requires operational knowledge of the Windows® operating system and should only be performed by qualified personnel.**

Before you begin, connect a mouse and keyboard to the CNC to use System Tools.

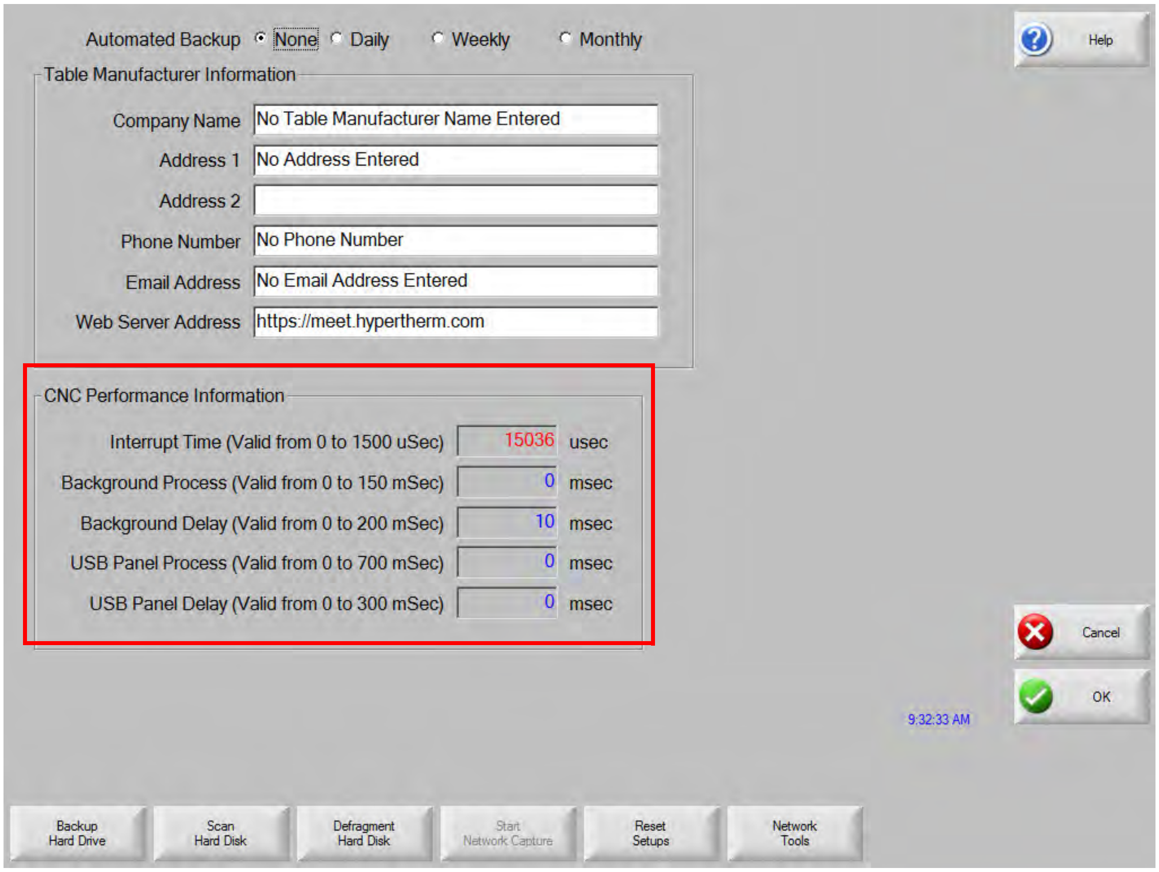




Automated Backup: Select the frequency of the automatic reminder for the backup of information on the hard drive. At the selected time, the system displays a prompt to back up the system when you turn it on.

Table Manufacturer Information: Enter the contact information for the cutting table manufacturer. This information displays when Remote Help starts. You can also change the web server address to show your own Remote Help server.

CNC performance information



The CNC Performance Information parameters display information about processes within the CNC that can contribute to unreliable motion control. Consult these parameters if the CNC takes longer than expected to load files or perform operations than it did when it was originally installed.

Each parameter provides the valid range for a process that operates within the CNC. Time values next to each parameter display the longest time that has been recorded for that process since the CNC was turned on. Values in blue are within the valid range; values in red are outside the valid range.

To verify CNC performance:

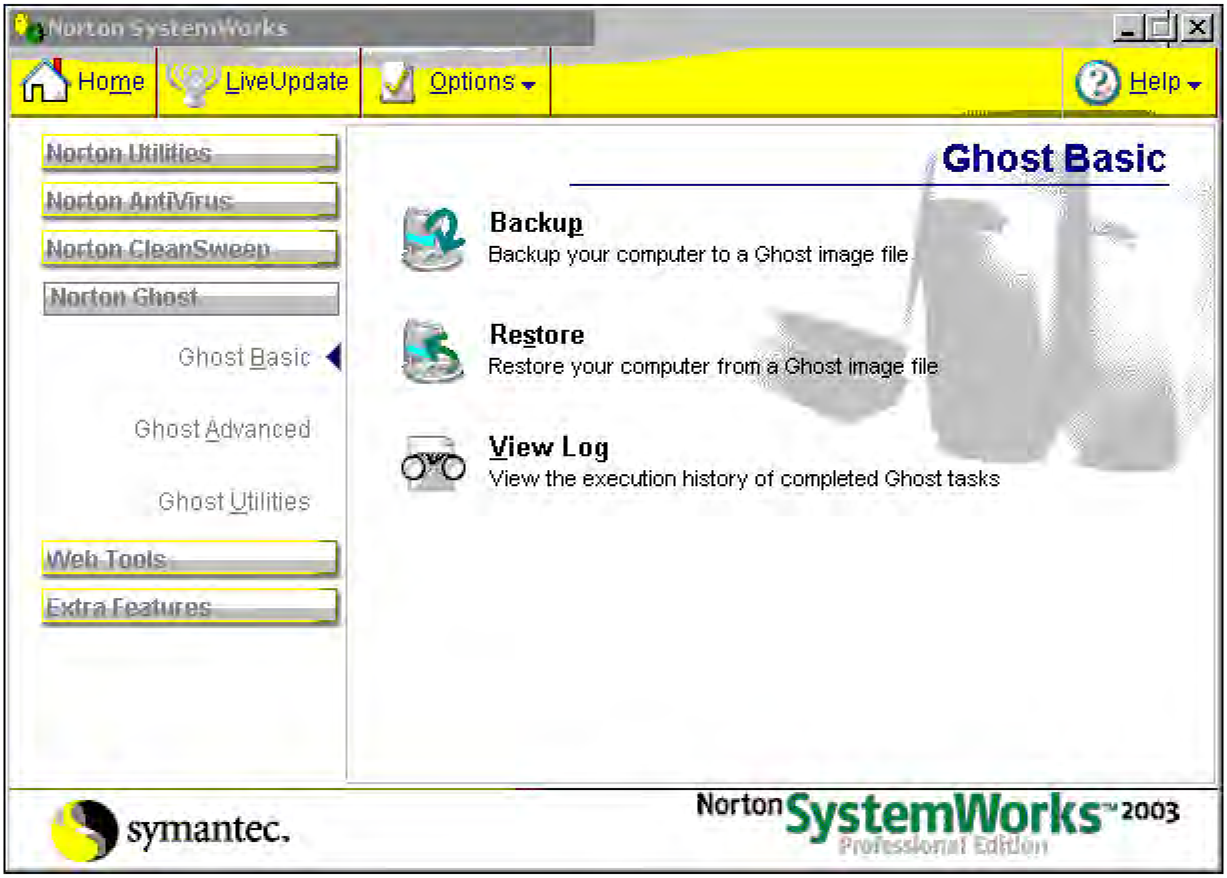
1. From the Main screen, select **Setups** > **Password** > **Special Setups**.
2. Choose the **System** soft key.
   - ❑ If the numbers for all parameters are blue, the CNC is operating normally.
   - ❑ If the number for any parameter is red, restart the CNC.
3. After restarting the CNC, check the System screen again.
   - ❑ If any number displays in red after the restart, and other programs are running, shut down any other programs so that Phoenix is the only program running.
   - ❑ If any number still displays in red after shutting down all other programs, contact Hypertherm Technical Service for assistance.

Back up the hard drive

1. From the Main screen, choose **Setups > Password > Special Setups**.

2. Choose the **System** soft key.

Press the **Backup Hard Drive** soft key to use the Norton™ Ghost Utility to save the contents of the main hard drive to a specified location on the CNC hard drive or to a mapped drive on the network.

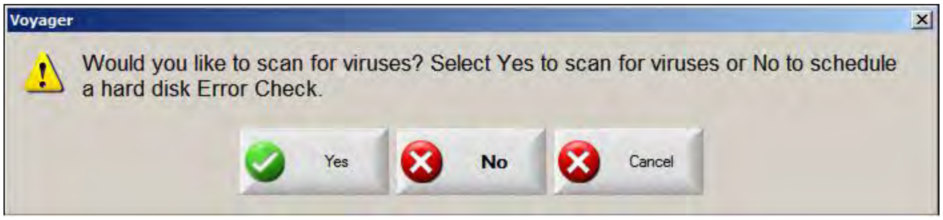


After using Norton Ghost, all files on Drive C: will be replaced. Make sure to reload the Phoenix.ini file (or other setup file) for the system.

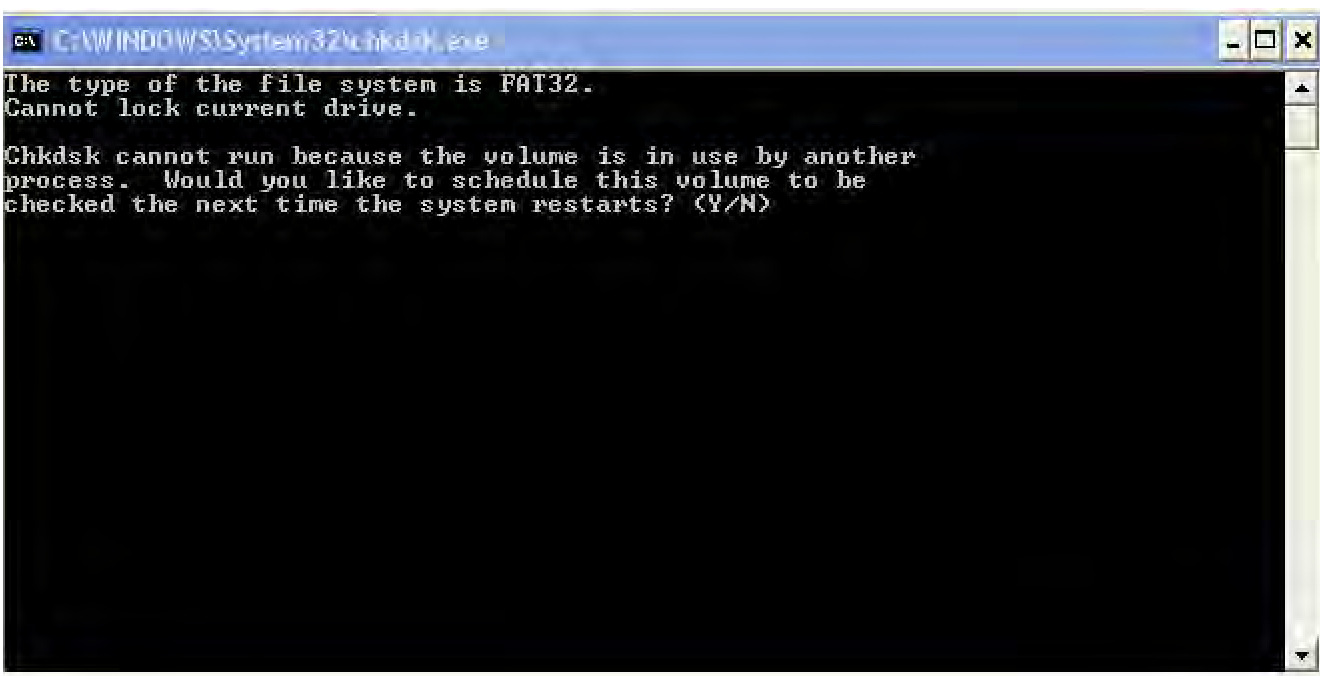
Scan Hard Disk

To scan the hard disk for viruses or disk errors using Norton Antivirus software:

1. From the Main screen, choose **Setups > Password > Special Setups**.
2. Choose **System > Scan Hard Disk.**

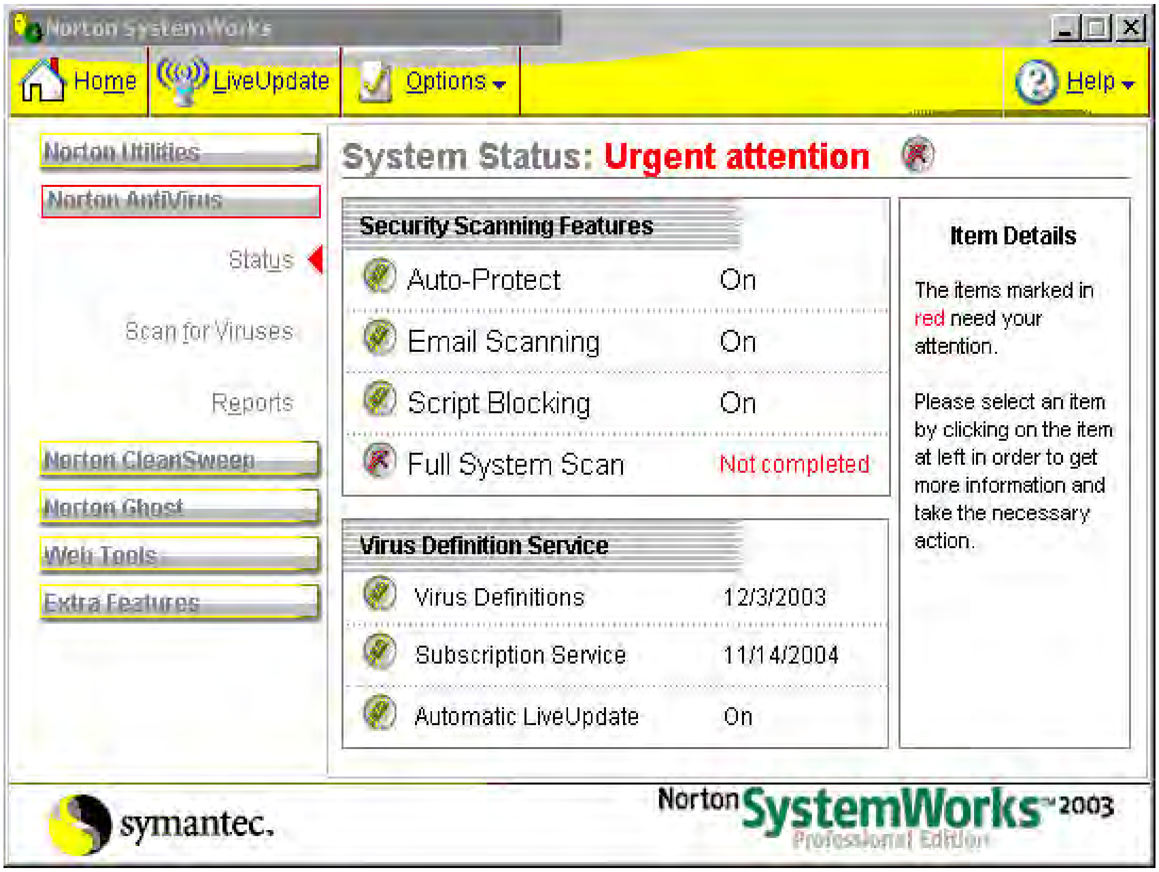


3. Select **Yes** on the verification popup to start the scan.
4. Select **No** to scan the hard drive for errors and correct them using the Windows XP CHKDSK (check disk) utility. This task should be performed approximately every three months depending on the number of files loaded to and removed from the CNC.
5. The Check Disk (CHKDSK) function cannot operate while the CNC software is in operation. Press “Y” (for yes) to check the disk the next time that the CNC boots up.



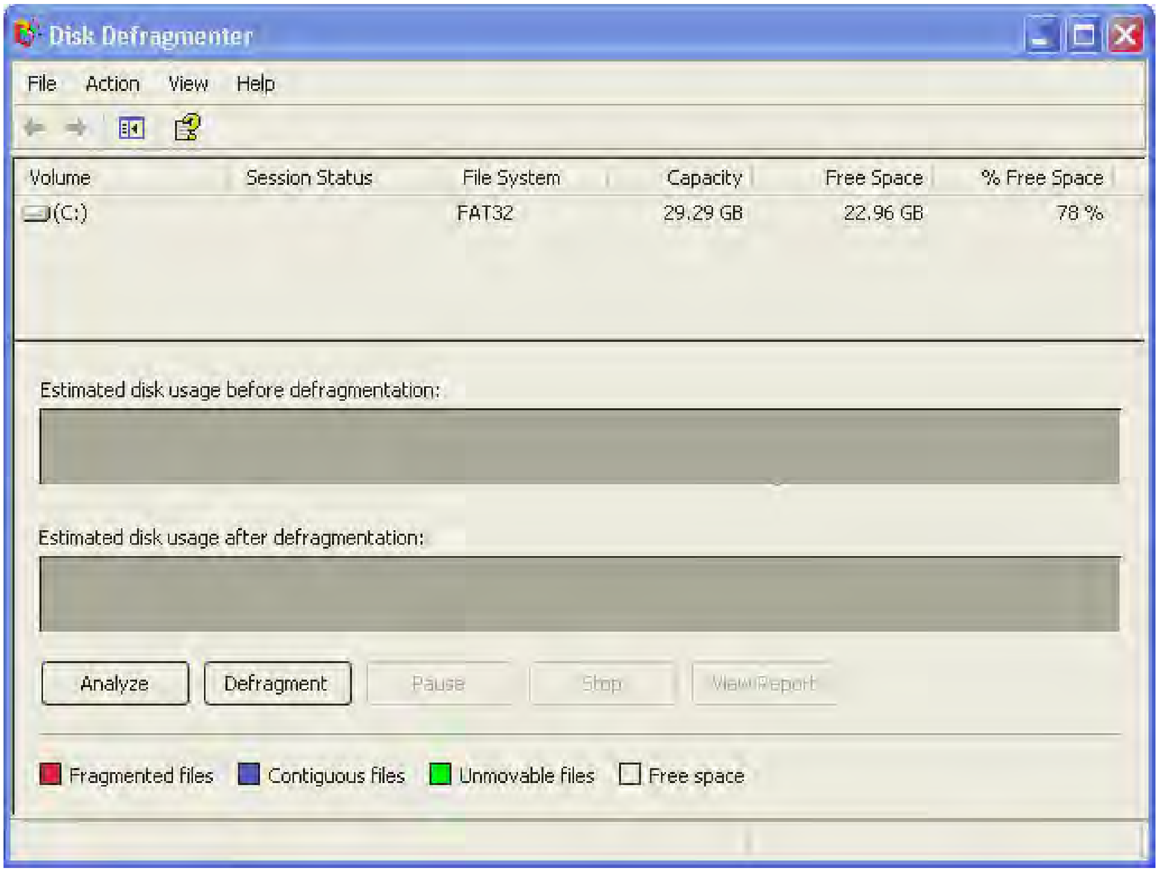
Antivirus

The Antivirus feature allows you to search files that are loaded on the CNC for known viruses.



Defragment hard disk

The defragmentation operation scans the CNC hard disk for file locations and rearranges them for optimum performance. This task should be performed approximately every three months.



To defragment the hard disk:

1. From the Main screen, choose **Setups > Password > Special Setups**.
2. Choose **System > Defragment Hard Disk.**

Launch an External Program

The Phoenix software provides a method for you to launch a program from a custom soft key on the Main screen. The following steps create a soft key that launches the Notepad text editor. You can modify the key label and path to launch a drive tuning, inventory, or other program needed in your environment.

1. Using a text (ASCII) editor such as Windows Notepad, create a file named "external.txt." This file will contain only two lines of text.
2. In the first line of text, enter the words that you want to appear on the soft key, in 60 characters or fewer.
3. In the second line of text, enter the path to the program on the Phoenix hard drive.
4. Save the file in the C:\Phoenix folder.

A sample external.txt file is shown below:

```
Notepad
C:\windows\system32\notepad.exe
```

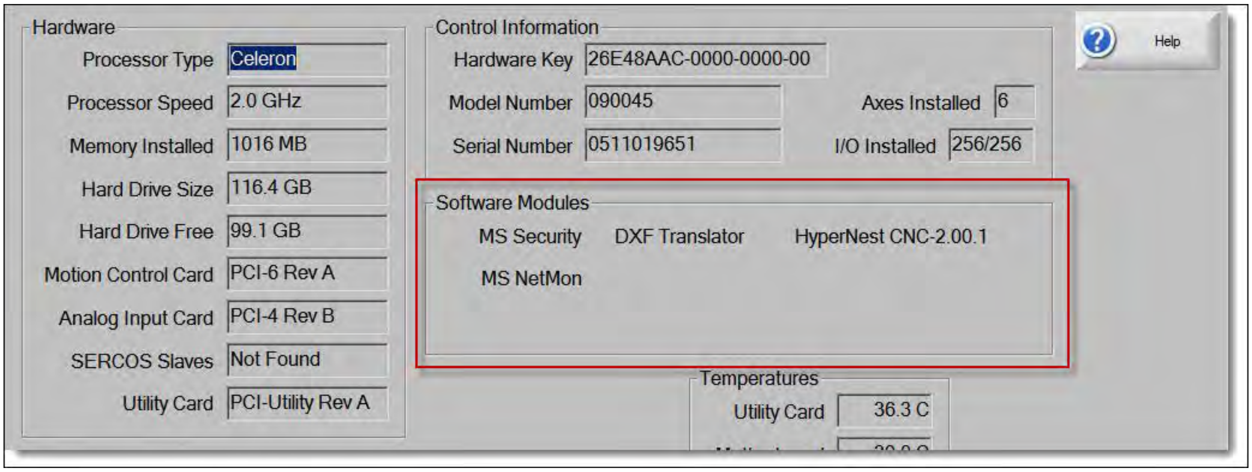
This file creates the Notepad soft key:

Notes:

- Phoenix can support only one custom soft key at a time.
- The path to the program must be valid or Phoenix will not display the custom soft key.
- The first line cannot exceed 60 characters, including spaces.
- The second line can contain spaces.

Software installed on the CNC

Hypertherm CNCs have several software modules pre-loaded at the factory. To view the default software modules, choose **Setups > Diagnostics > Control Information.** The picture below shows only a portion of the CNC information screen.



DXF Translator: Hypertherm CNCs can convert simple DXF files into .TXT files for cutting. To learn more about the DXF file translator, see *Chapter 3 Loading Parts* in the *Phoenix Operator Manual*.

Hypernest CNC: Hypertherm CNCs provide Manual and Automatic Nester options that allow the simple nesting of parts. To learn more about CNC nesting, see *Chapter 4 Arranging Parts* in the *Phoenix Operator Manual*.

Neither of the Nester options can nest parts that contain advanced feature codes such as G59 process selection codes.

MS Security: Microsoft Security Essentials is the virus protection built into the CNC. To learn more, see *Antivirus* on page 42.

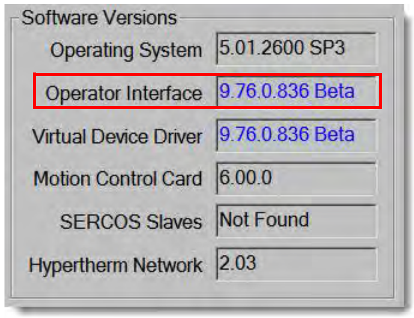
MS NetMon: Microsoft Network Monitor is the Ethernet monitoring tool built into the CNC. To start the Network Monitor, choose Setups > Password > Special Setups > System > Start Network Capture.

Phoenix software version number

Follow these steps if you need the Phoenix software version number for service.

1. Choose **Setups > Diagnostics > Control Information.**

2. Under Software versions, the Phoenix version number appears next to Operator Interface.



Phoenix setup files

Hypertherm CNCs save the values that you enter on each screen in the setup file named Phoenix.ini. This file is stored in the C:\Phoenix folder. The CNC saves the file when you make a change to a value and exit a screen. The CNC also saves a backup version of the setup file, Phoenix.bak. in the D:\Setups folder on the CNC hard drive.

Setup files can be copied to other Hypertherm CNCs. After configuring a CNC to run a particular cutting system, you can use that setup file for the next installation of a CNC, table, and tool. When you load a setup file into a CNC, that setup file overwrites both Phoenix.ini and Phoenix.bak.

EDGE Pro setup files should be used only on other EDGE Pro CNCs. MicroEDGE Pro and EDGE Pro Ti setup files have the same restriction.

About setup files

The CNC stores multiple copies of the Phoenix.ini setup file.

- Phoenix.ini is always the active setup file. Each time you make a change on a screen on the CNC, the changes are saved to Phoenix.ini. In the Special Setups screen you can save a copy of Phoenix.ini with a different name for a backup or to use on another CNC. When you load a setup file with another name, the contents of the file overwrite the contents of Phoenix.ini and Phoenix.bak. Phoenix.ini is stored in the **C:\Phoenix** folder.
- Phoenix.bak is an identical copy of Phoenix.ini. The CNC saves Phoenix.bak each time it saves Phoenix.ini. If Phoenix.ini becomes corrupted, the CNC loads the settings in Phoenix.bak and creates a new Phoenix.ini. Phoenix.bak is stored in the **D:\Setups** folder.
- Default.ini is an important backup of Phoenix.ini and is used to restore settings if both Phoenix.ini and Phoenix.bak become corrupted. Hypertherm recommends that you create the Default.ini when you commission a cutting system. Default.ini is stored in the hard drive root folder (**C:**).
- Factory settings represent the contents of the Phoenix.ini file as it was shipped from the factory. Each CNC has factory settings that represent the available features and axes on that CNC.
- Factory settings are stored internally in Phoenix software but are not stored in a specific file. Factory settings may be very different from the settings on a CNC when it is commissioned in a cutting system at a manufacturing site.

Setup files from previous versions of Phoenix

New versions of Phoenix software contain new features with settings that are saved to the Phoenix.ini file. When you load a setup file from a previous version of Phoenix, a message displays that shows you the new settings that are not in your setup file. Phoenix loads its factory settings for those settings.

After you load a setup file from a previous version of Phoenix, make sure that you **save a new version of the setup file** in the new version of Phoenix. By saving the Phoenix.ini in the new version, the you make sure that the setup file has settings for all new features in the Phoenix software.

Save and load the setup file

Hypertherm recommends saving the setup file three times when first commissioning a cutting system:

- Save the setup file to a memory stick with the name **EndUserDate.ini.**
- Save the setup file to the HD Backup location (**D:\Setups**) that you select the drop-down of the save dialog box, and name the file **EndUserDateBak.ini.**
- Save the setup file as **Default.ini** on the CNC hard drive root folder **C:\\.**

Save a setup file without using a password

The CNC can gather the Phoenix.ini, LastPart.txt, and SystemErrors.log, into a single zip file (Phoenix.zip). Use this procedure to save the setup file or to gather files to send to Hypertherm Technical Service. Since this method does not require a password, it provides a convenient way for an end user to save the Phoenix.ini without being exposed to password-protected settings.

1. Insert a memory stick into a USB port on the CNC.
2. From the Main screen, choose Files > Save to Disk > Save System files to Disk.
3. Choose Save All to Zip. The CNC creates Phoenix.zip and saves it on the memory stick.

Save a setup file with a different name

When you save a setup file under a different name, the CNC continues to use Phoenix.ini and does not update your saved setup file.

1. Choose **Setups > Password > Special Setups.**
2. Choose **Save Setups.**
3. Choose **Hard Drive, HD Backups,** or **Memory Stick** for the location.
4. Enter a file name.
5. Choose **OK.**

The CNC saves the file in the hard drive root folder (**C:**), **D:\Setups,** or the memory stick root folder.

Load a setup file

The setup file that you load becomes Phoenix.ini.

1. Choose **Setups > Password > Special Setups.**
2. Choose **Load Setups.**
3. Choose **Hard Drive** or **Memory Stick** for the location. The CNC lists only the *.ini files that are present in the root folder of the hard drive or the memory stick.
4. Enter a file name.
5. Choose **OK.**

Save and load default settings

The CNC uses Default.ini to restore settings if both Phoenix.ini and Phoenix.bak become corrupted.

Save Default.ini

1. Choose **Setups > Password > Special Setups.**
2. Choose **Save Setups.**
3. Choose **Hard Drive** for the location.
4. Enter the **Default** for the file name.
5. Choose **OK.**

The CNC creates Default.ini in the hard drive root folder (**C:**).

Load Default.ini

Default.ini must exist in **C:**. If no Default.ini exists, the CNC provides an option to reload factory settings instead.

1. Choose **Setups > Password > Special Setups > System.**
2. Choose the **Default Setups** soft key.

If you haven’t saved a Default.ini file, then the soft key reads Reset Setups and allows you to reset the CNC to factory settings instead.

Reload factory settings

If your system is experiencing motion or other problems for which you cannot find a solution, reloading the factory settings provides you with a known starting point for configuring the CNC to work with your cutting system.

CAUTION!

Reloading the factory settings changes the setup on the CNC. The cutting system may no longer be capable of motion and the I/O may not work. Only use this procedure if you do not have a backup setup file.

1. Choose **Setups > Password > Special Setups > System.**
2. Choose **Reset Setups**.

If the soft key label reads Default Setups, then the file Default.ini exists in the hard drive root folder. Reload the default settings instead of the factory settings if possible.

Section 2

Machine Setup

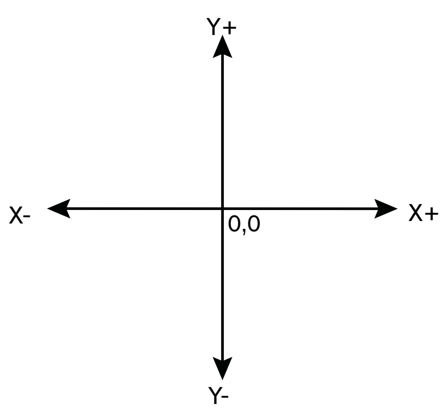
Overview

Phoenix provides several screens that you use to define your cutting system. The screens are accessed with the Machine Setups password provided to you by your table manufacturer. This section describes setting up your cutting system including:

- Defining axis orientation and positive motion
- Identifying all axes on the cutting system
- Setting machine speeds
- Assigning I/O points
- Setting up serial ports

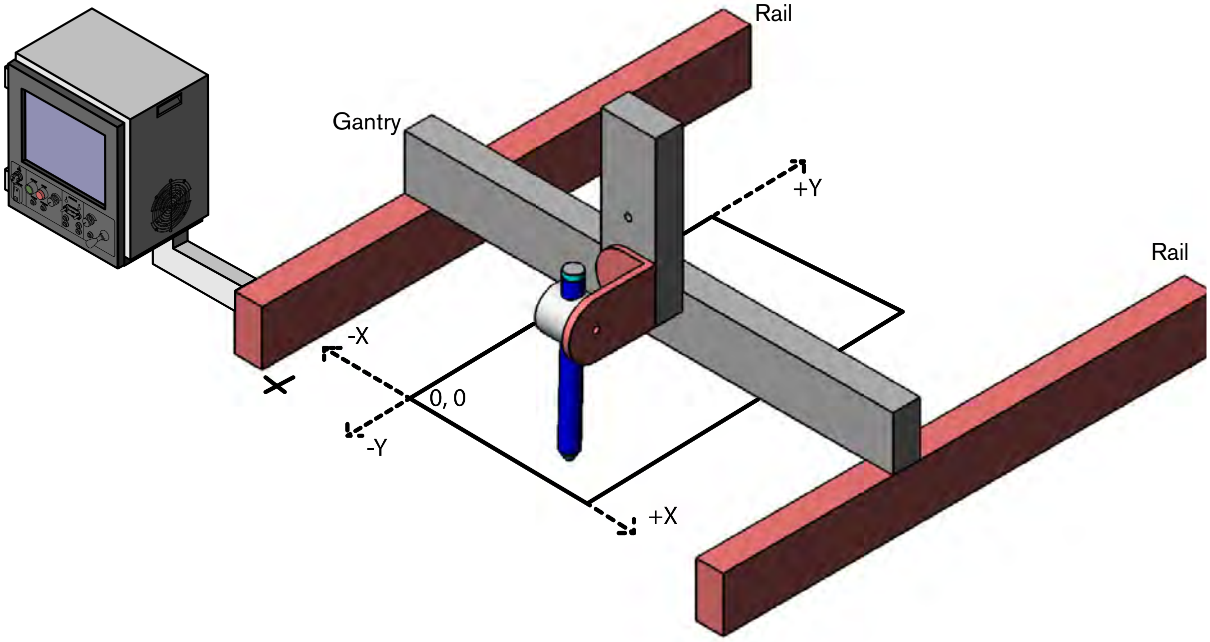
Axis orientation and positive motion

Gantry-table cutting systems are based on the Cartesian coordinate system. Phoenix requires you to define the X axis and positive motion relative to the table's 0,0, or absolute home, position. In the Cartesian coordinate system, the X+ and Y+ motion is represented like this:

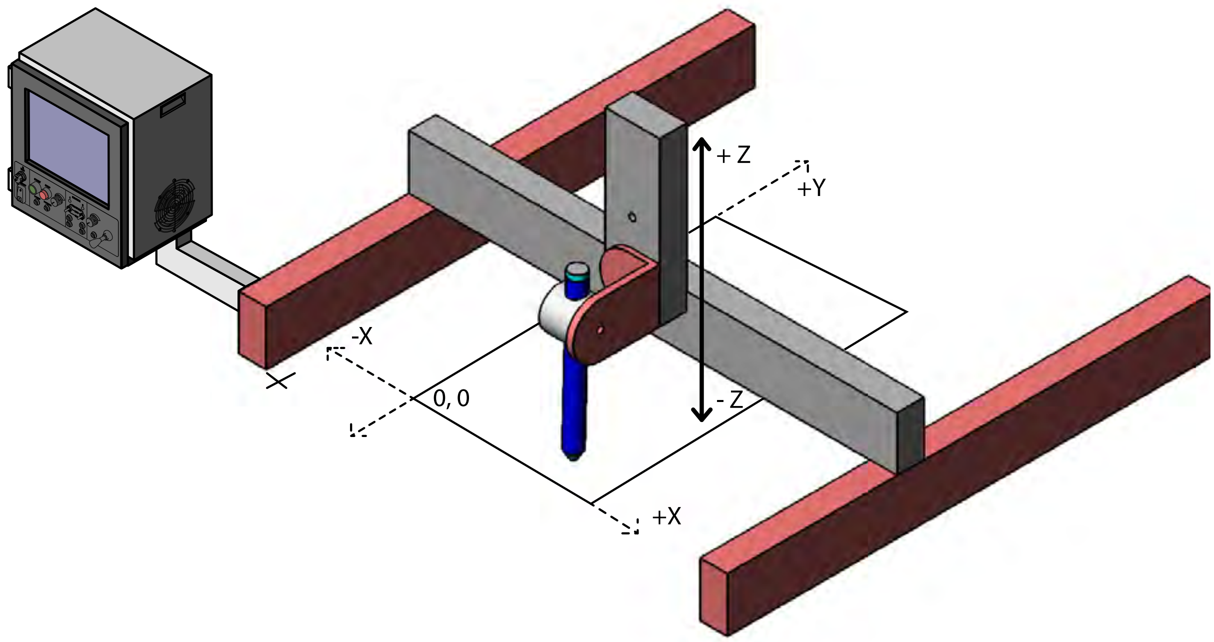


The X axis is the first axis Phoenix needs to know about. In a standard orientation, the X axis would be along the gantry. Phoenix calls the axis along the gantry the Transverse axis. The Y axis is always perpendicular to the X axis. In Phoenix, the Rail axis is perpendicular to the Transverse. In a standard orientation the Y axis would be along the rail. When a cutting system has a second rail motor, you select the Dual Gantry axis on the Machine Setups screen in Phoenix.

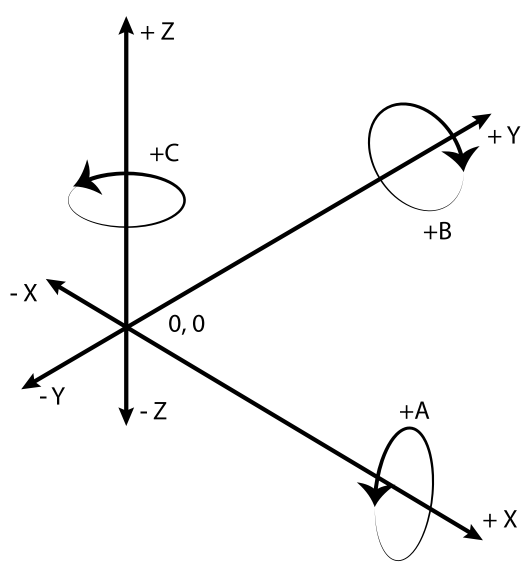
The following picture shows a cutting table where X is assigned to the gantry, also called Transverse, and Y is assigned to the Rail.



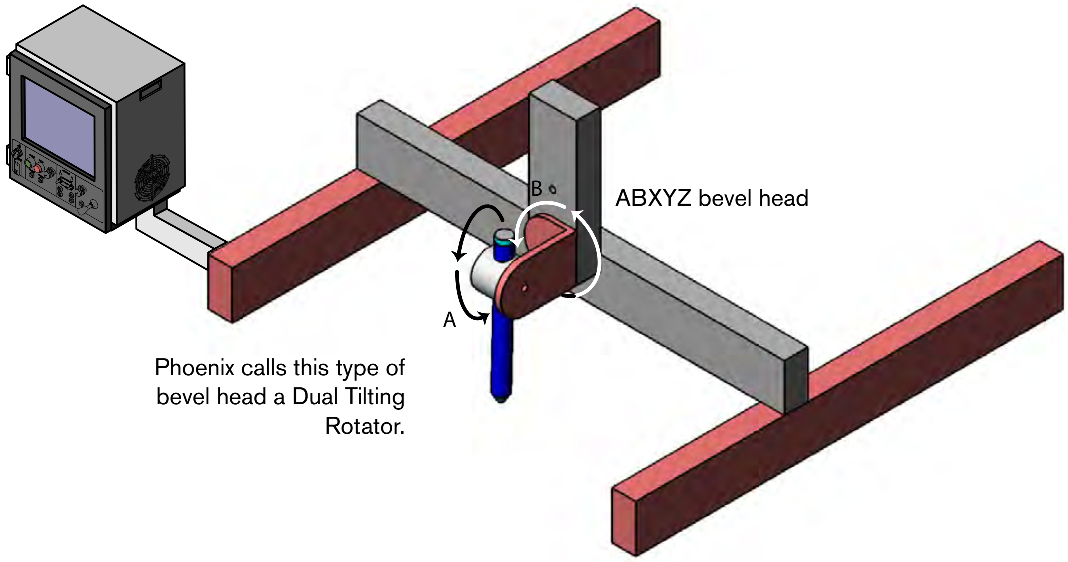
Most torch height controls require an axis on the CNC. The Sensor THC, integrated into many Hypertherm CNCs, is also an axis on the CNC. The Sensor THC axis, or lifter axis, is called a Z axis. Its positive direction is always up (skyward) and its negative direction is always down (earthward).

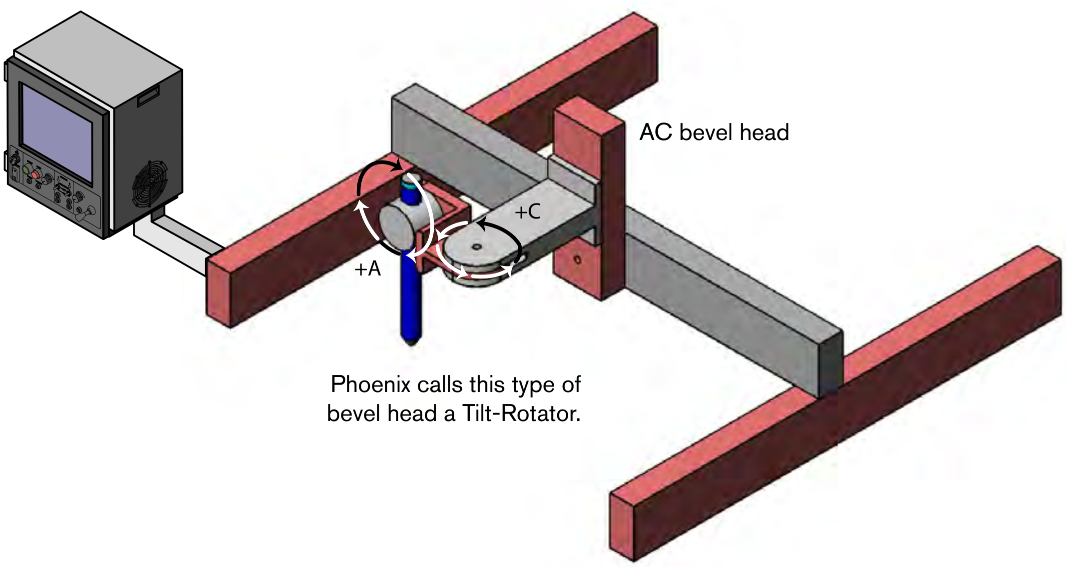


The X, Y, and Z axes are all linear axes. To perform angled cutting, the cutting system needs one or more rotational axes to position the tool or torch on an angle. These axes are commonly known as A, B, and C axes. Many bevel heads provide two axes of rotation that the CNC coordinates along with the X, Y, and Z motion to position the torch. In a standard orientation, the A axis rotates about the X axis, the B axis rotates about the Y axis, and the C axis rotates about the Z axis.



Most bevel heads provide two axes of rotation. These are known as ABXYZ and AC bevel heads.





About axis assignments

Hypertherm CNCs are hard-coded with the axis assignments. The table below shows the axis numbers that the CNC uses for 12 axes, and common letter identifiers for the axes.

Axis	Axis assignment
1	Transverse or Rail (X)
2	Rail or Transverse (Y)
3	Dual Gantry (W)
4	Sensor THC 1 (Z)
5	Rotate 1 (B or C)
6	Tilt 1 (A)
7	Transverse 2
8	Rotate 2
9	Tilt 2
10	Sensor THC 2
11	Sensor THC 3
12	Sensor THC 4

All cutting systems require both Transverse and Rail axes, but you can assign either to X or Y.

A Dual Gantry axis is parallel to the Rail axis (think of it as a "Rail 2" axis). In a dual gantry cutting system, the second rail axis is powered by its own motor using the same signal as the Rail axis.

Rotate and Tilt axes are used for bevel cutting.

Change the axis assignments with passwords

Using special passwords NRT and 1RT (No Rotate Tilt and 1 Rotate Tilt respectively) redirects certain axes signals to different axis numbers. A 2-station cutting system on a 6-axis CNC requires the NRT password so that the Transverse 2 and Sensor THC 2 signals are output on axes 5 and 6 instead of 7 and 10. (Previously the Transverse 2 axis was available only on a dual bevel-head cutting system).

Additional examples:

- To enable the Transverse 2 axis for pipe or tube cutting on a CNC with 6 or fewer axes, enter the NRT password.
- To set up a cutting system with both a straight torch and a bevel head, enter the 1RT password.

The following tables identify the application, password, and axis assignments for common cutting system axis configurations. Use these tables to identify the axis number, and for SERCOS cutting systems, the drive address where the CNC will output the axis signals.

4-axis I-cutting on pipe or tube, no Dual Gantry, NRT password

Axis/drive address	Axis assignment
1	Transverse or Rail
2	Rail or Transverse
3	Sensor THC
4	Transverse 2

5-axis I-cutting on pipe or tube, with Dual Gantry, NRT password

Axis/drive address	Axis assignment
1	Transverse or Rail
2	Rail or Transverse
3	Dual Gantry
4	Sensor THC
5	Transverse 2

6-axis, 2-station, I-cutting on flat plate, with Dual Gantry, NRT password

Axis/drive address	Axis assignment
1	Transverse or Rail
2	Rail or Transverse
3	Dual Gantry
4	Sensor THC
5	Transverse 2
6	Sensor THC 2

6-axis bevel cutting on pipe or tube, no Dual Grantry, 1RT password

Axis/drive address	Axis assignment
1	Transverse or Rail
2	Rail or Transverse
3	Sensor THC
4	Transverse 2
5	Rotate
6	Tilt

7-axis bevel cutting on pipe or tube, with Dual Gantry, 1RT password

Axis/drive address	Axis assignment
1	Transverse or Rail
2	Rail or Transverse
3	Dual Gantry
4	Sensor THC
5	Rotate
6	Tilt
7	Transverse 2

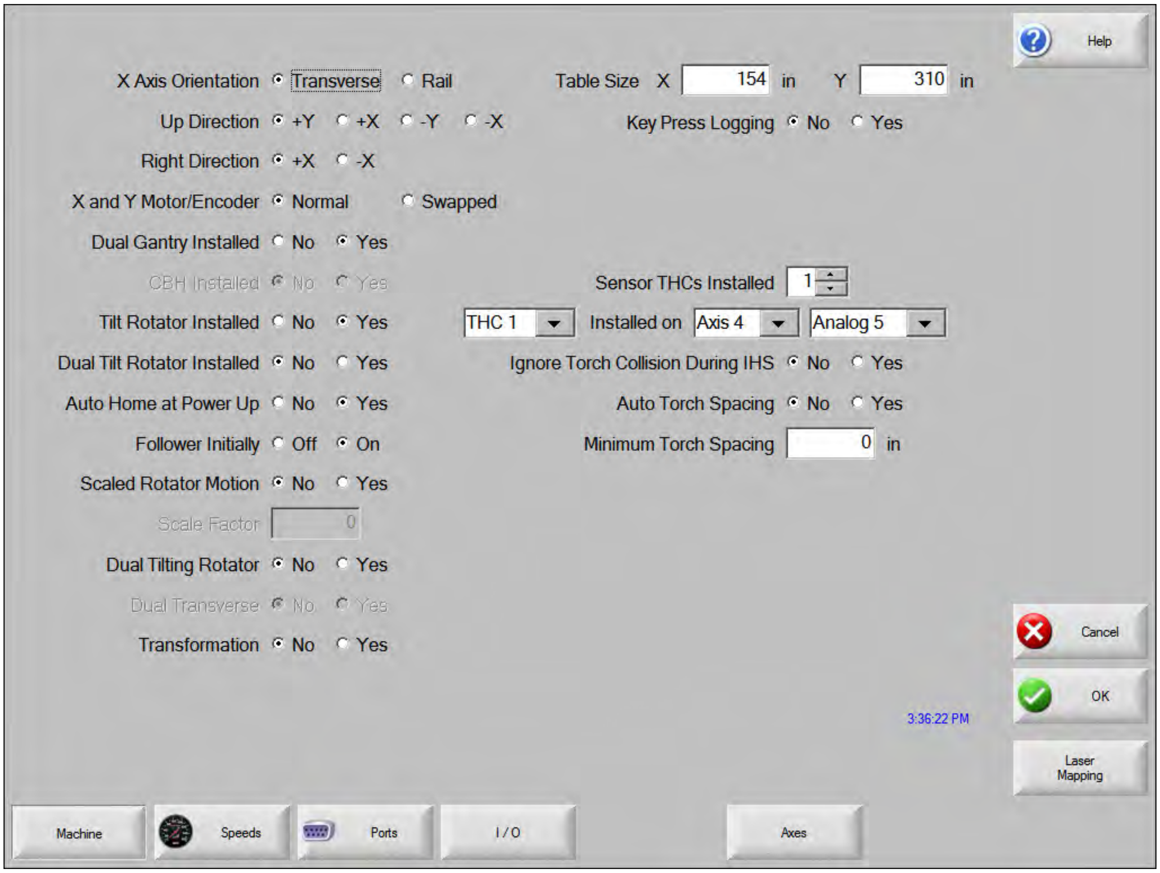
8-axis, 2-station, bevel and I-cutting on flat plate, 1RT password

Axis/drive address	Axis assignment
1	Transverse or Rail
2	Rail or Transverse
3	Dual Gantry
4	Sensor THC
5	Rotate
6	Tilt
7	Transverse 2
8	Sensor THC 2

System axis screens

Machine Setups screen

Use the Machine Setups screen to define the machine orientation, choose the axes you have on your cutting system, and enter the number of torch height controls. To open the Machine Setups screen, choose Setups > Password > Machine Setups.



X Axis Orientation: Specifies the X axis as the Transverse or Rail axis. The default setting is Transverse.

Up Direction: Defines the machine motion when the ↑ arrow key is pressed in manual mode.

Right Direction: Defines the machine motion when the → arrow key is pressed in manual mode.

X and Y Motor/Encoder: Select Swapped to swap the X and Y axis encoder signals internally. This feature simplifies the installation of systems with alternative wiring configurations.

Dual Gantry Installed: Select Yes to enable the Dual Gantry axis for a second side drive. The Dual Gantry axis is always parallel to the Rail axis.

CBH Installed: Select Yes to enable the contour bevel head axis and allow configuration of that axis.

Tilt Rotator Installed: Tilt Rotator describes an ACXYZ bevel head, which as a rotational axis (C) combined with a tilt axis (A). Choose this option when your cutting system has this type of bevel head installed.

Dual Tilt Rotator Installed: Choose this option when the cutting system has a second ACXYZ bevel head. Choosing this option also enables a second transverse axis that can be used for a second straight torch, or for a chuck to turn round pipe or rectangular tube stock.

Auto Home at Power Up: Determines whether the contour bevel head (CBH) and tilt or rotate axes automatically go into the homing routine when the CNC powers up.

Follower Initially: Determines whether the CBH and rotate features are always enabled after power up or are enabled through the part program.

Dual Tilting Rotator: Dual Tilting Rotator describes an ABXYZ bevel head which has two tilt axes (A and B). Choose this option when your cutting system has this type of bevel head installed. The Tilt Rotator Installed option must be selected to make Dual Tilting Rotator option available.

Dual Transverse: Enables the Transverse 2 axis for use with a second cutting station or as an axis for a chuck for holding pipe or tube stock.

Table Size: Enter between the X overtravel switches and the Y overtravel switches. The X and Y dimensions represent the cutting area of the table and limit the workpiece size dimensions set in the Cutting screen.

Key Press Logging: Records key press and other related information in a daily log file. Service technicians use this log to review events before a fault. When logging is enabled, the log file is saved to the hard drive so it can be retrieved by transfer to a USB memory stick. Usually this parameter is set to NO.

ArcGlide THCs Installed: Defines the number of ArcGlide THC axes assigned to the CNC.

Use Hypernet: Sets up the CNC to communicate with the ArcGlide over Hypernet.

Sensor THCs Installed: Defines the total number of Sensor THC axes assigned to the CNC.

THC Installed On Axis / Analog: Defines the torch height control axis. Select the THC, which axis it is installed on, and the analog input to use for arc voltage feedback.

For more information on the setup and use of the Sensor THC, refer to the other sections in this chapter, as well as *Torch Height Controls* in the *Phoenix Operator's Manual*.

Ignore Torch Collision During IHS: Allows the system to ignore input about a torch collision during the initial height sense cycle. This may be helpful for some THC mechanics.

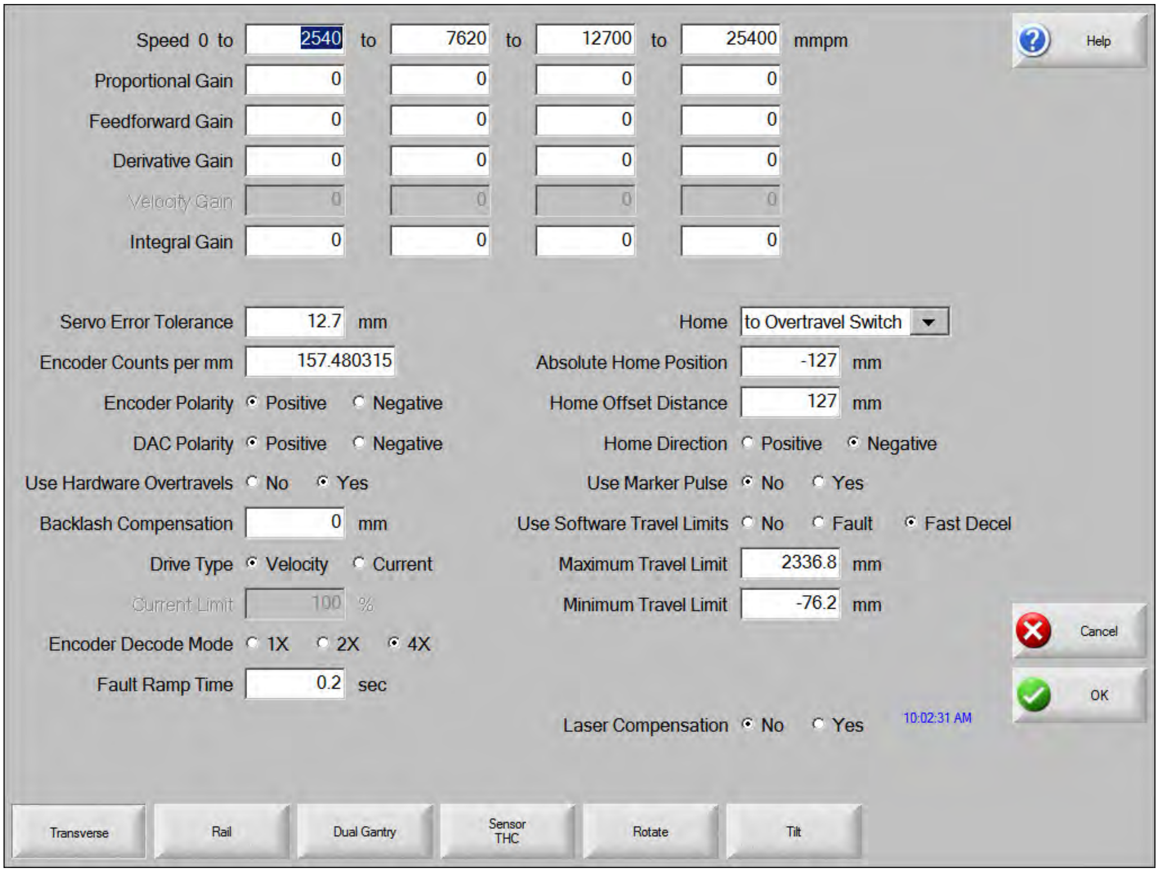
Command THCs Installed: Enables the Command THC serial link. For more information on this feature, refer to *Configuring Ports* on page 87. Shown in Type P mode only.

Auto Torch Spacing: Select Yes to enable the automatic torch spacing feature. This feature uses embedded part program codes and dedicated outputs to control individual torch stations to set spacing and clamping of direct and mirrored cutting. For more information, refer to the *Phoenix V9 Series Programmer's Reference*.

Transverse or Rail Axis

The following screen is identical for both the traverse and rail setup.

The CNC comes with both an advanced position and velocity servo loop. The following parameters are available to help configure the servo loops for your specific drive and mechanical system.



Speed 0 To: Enter speed ranges or breaks. You can set drive tuning parameters (gains) for each speed break. The speed breaks can be entered here or on the Speed screen. On the Speeds screen you can enter an Acceleration Rate for each speed break.

Proportional Gain: Proportional gain correlates to elastic stiffness in the control loop. The range of values for this field is 1 to 500, although setting from 40 to 60 are usual.

Increase the proportional gain to increase the static stiffness, but decrease the response of the servo loop.

Under a proportional loop control, the drive system applies a restoring torque to the motor in proportion to the position error of the axis. If proportional gain too high, the system and axis become unstable. During cutting, the torch can overshoot the path. This is also called a “hot” control loop.

If proportional gain is too low, the system response is not precise. This can be seen in the test pattern when the outside corners become rounded and not all the circle segments meet in the center.

Integral Gain: Integral gain improves the positioning accuracy of the control loop. Integral gain can be used to compensate for static friction or gravity. Excessive integral gain can result in system instability. For most shape cutting machines, this parameter should be set to zero (0).

Derivative Gain: Derivative gain helps to minimize sudden changes in velocity. The higher the derivative gain, the slower the response time to the control loop. For most velocity loop drives, this parameter is set to zero (0).

Feedforward Gain: Feedforward gain can be used to drive the following error to zero during machine motion. In all digital control loops there is a finite amount of error that is introduced by the velocity command. Increasing feedforward gain can reduce this introduced error.

Velocity Gain: When you use a current loop amplifier, you can use the internal velocity loop in the CNC to provide dampening without an external tachometer.

Use of the internal velocity loop with a current loop amplifier can result in higher static stiffness, smoother machine motion, and less overshoot.

Servo Error Tolerance: Servo error, also called following error, is the difference between the commanded motor position and the actual motor position. The servo error tolerance is the upper limit of the amount of following error allowed before the CNC faults.

The amount of servo error tolerance depends on the cutting system mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high could cause inaccurate motion or mechanical harm. Set the Following Error parameter in the Watch window and observe steady-state operation of the cutting system (some following error is normal). Set the servo error tolerance to a value slightly higher than the steady-state following error.

Encoder Counts per mm (inch): To determine the encoder counts per mm (inch), you will need to know the following measurements for your encoders:

- counts per revolution of the motor
- gear ratio
- distance of travel in one revolution of the pinion gear
- diameter of the pinion gear when it engages the rack

An example of the equation is shown below:

1000 line encoder multiplied by 4 (quadrature) per 1 motor revolution

10:1 gear ratio

Distance traveled in one revolution of the pinion

$$\frac{4000 \text{ counts}}{1 \text{ revolution}} \times \frac{10 \text{ rev of motor}}{1 \text{ rev of pinion}} \times \frac{1 \text{ rev}}{2\Pi \text{ inches}} = \frac{X \text{ counts}}{\text{inches}}$$

Encoder counts per unit (inches or mm)

Circumference of pinion (2 inch diameter multiplied by *pi)*

Fault Ramp Time: This parameter sets the motion deceleration time after a fault occurs. At the end of Fault Ramp Time the drives will be disabled.

Drive Type: This parameter is used to tell the CNC what type of control loop to run. If you are running an external velocity loop drive (indicated by having an integrated tachometer in the motor), select Velocity. If you are running in torque mode (no tachometer), select Current.

DAC Polarity: This parameter allows changing of the analog output polarity to establish proper control loop feedback without any wiring changes.

Encoder Polarity: This parameter allows changing of the encoder input polarity to establish proper counting for positive machine motion without any wiring changes.

Encoder Decode Mode: Currently, the CNC only supports 4X encoder decode mode. This has been done to increase positional accuracy.

Use Hardware Overtravels: Select whether the cutting system will be using hardware overtravel switches. If Hardware overtravel switches are used, the CNC will disable feedback and display an error message if the inputs become active. It is recommended that hardware overtravel switches be installed.

Backlash Compensation: The Backlash Compensation parameter is used to offset or compensate for any backlash in the mechanics of the drive system. The maximum value for backlash compensation is 12.7 mm (.5 in).

Home: The Home parameter is used to activate use of the Home feature. Depending on configuration, the table may be Homed to either a designated home switch or an overtravel switch.

The Home feature is used to set a known absolute physical position location on the cutting table that is used for referencing future manual Go to Home and other motion commands. This is generally performed through activation of a home switch positioned on the appropriate axis giving it a known physical location.

When you enter a homing command, the CNC moves the axis toward the home switches at the Fast Home Speed until the switches activate. Once the switches activate, motion stops and then the axis moves in the opposite direction off switch at the Slow Home Speed. The moment that the switch deactivates, the CNC records the position which provides an absolute reference point for future motion commands.

Absolute Home Position: Defines the position of the axis when the Home Limit Switch or Marker Pulse is detected.

Home Offset Distance: Allows the user to set an offset distance from the Home Limit Switch.

Home Direction: Determines which direction the axes will travel during phase one of the homing sequence.

Use Marker Pulse: When enabled, the absolute home position will be assigned at the instant the marker pulse is detected. It is recommended that the Marker Pulse be used for optimal homing repeatability.

Use Software Travel Limits: The CNC is capable of running with software travel limits that are based on position. When enabled, this feature allows you to select Fault or Fast Decel when active. Fault operates as hardware switches with immediate fault. Fast Decel uses the Fast Deceleration rate set on the Speeds screen to ramp down motion.

Maximum Travel Limit: If Software Overtravels are enabled, the user is prompted for the maximum travel of the cutting machine.

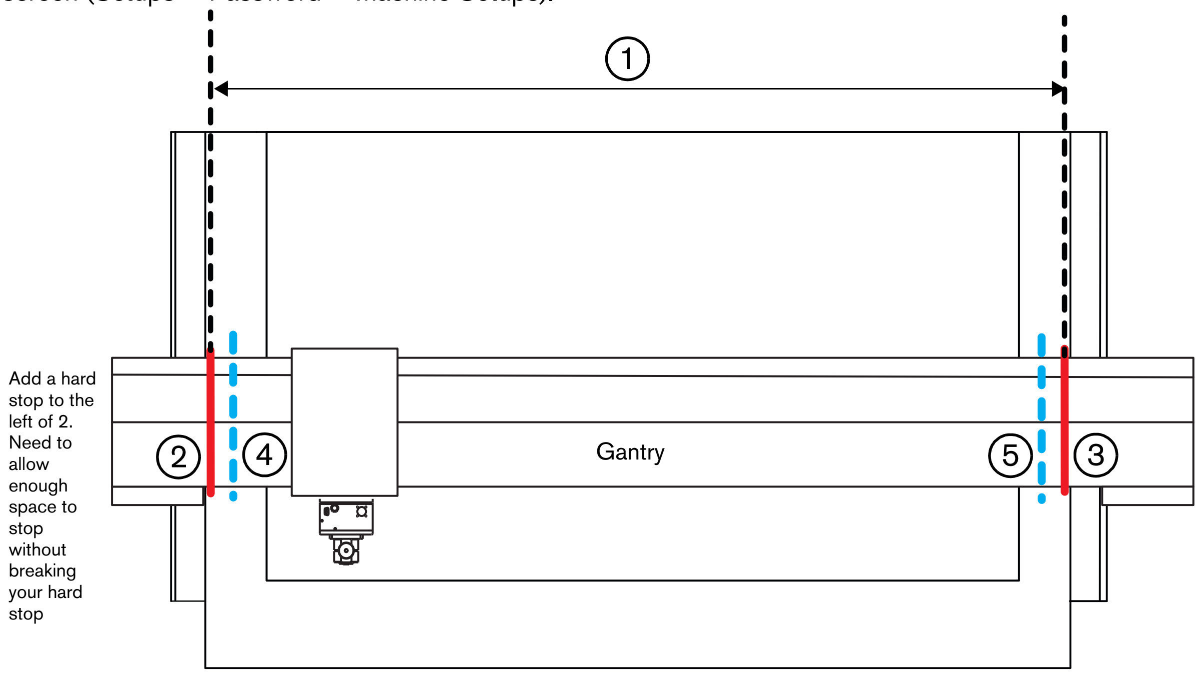
Minimum Travel Limit: If Software Overtravels are enabled, the user is prompted for the minimum travel of the cutting machine.

Laser Compensation: Uses position readings gathered with a laser interferometer to correct the actual position of the axes (Transverse, Dual Transverse, Rail, and Dual Gantry only).

Using software travel limits

A software travel limit (also called a *software overtravel*) is a position on the axis that can be reached before the system contacts a hardware limit switch or hard stop. The drawing below shows measurements from an example system with a gantry that has 154 inches (3912 mm) of travel on the Transverse axis between the overtravel switches.

The gantry travel distance of 154 inches (3912 mm) also equals the Table Size X dimension on the Machine Setups screen (Setups > Password > Machine Setups).



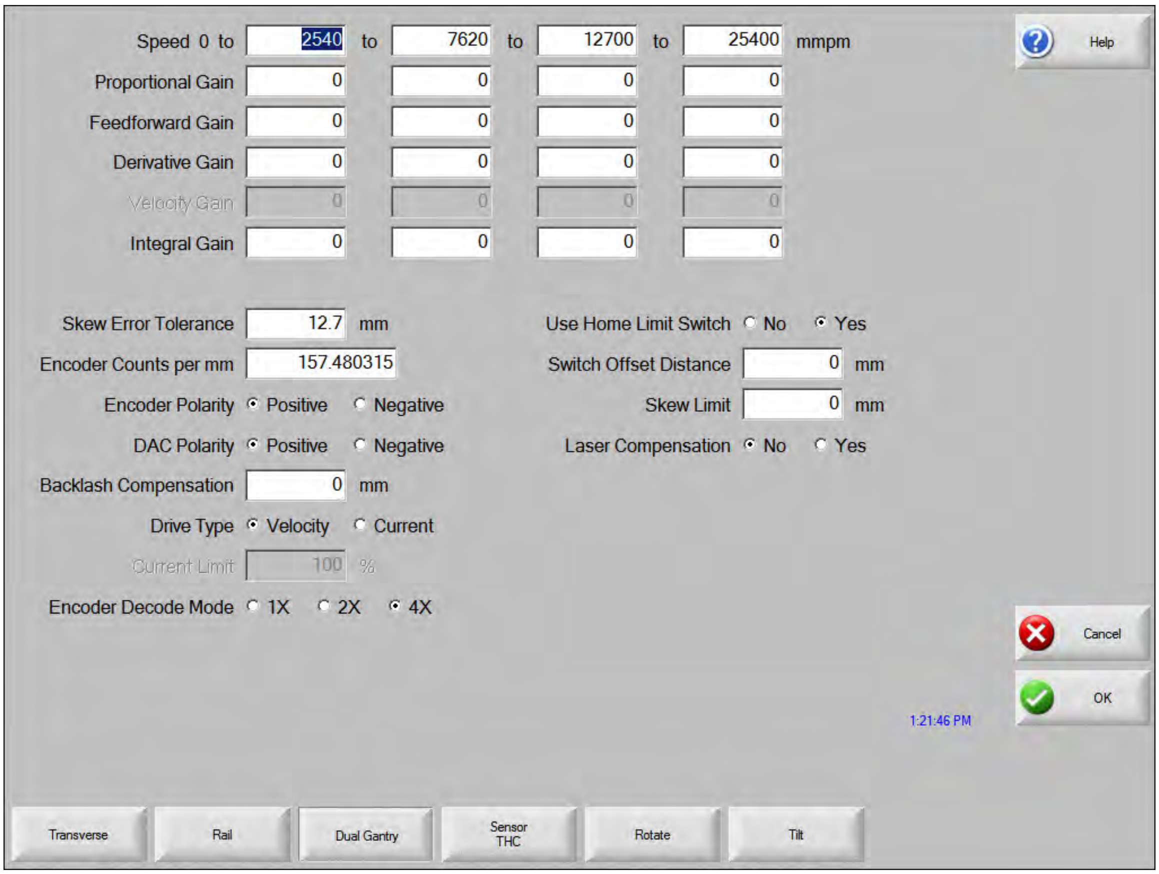
1. **1** Gantry distance between hardware overtravel switches 154 inches (3192 mm)
2. **2** X- overtravel switch at 0.
3. **3** X+ overtravel switch at154 inches (3912 mm)
4. **4** Software minimum travel limit at 3 inches (76 mm) to stop motion before contact with the hardware overtravel switch.
5. **5** Software maximum travel limit at 151 inches (3835 mm) to stop motion before contact with the hardware overtravel switch.

Dual Gantry Axis

The following setup screen is available when you have selected Dual Gantry on the Machine Setups screen.

The Dual Gantry Axis is commanded as a separate axis on the CNC that mirrors the output of the main Rail axis. Additionally, performance of the Dual Gantry Axis is compared to the main Rail Axis and additional output command is given to keep the axis in position.

The definitions for the setup parameters are the same as for the Transverse/Rail Axes. However, the number of selections are reduced because features for overtravels and homing do not apply.



Speed 0 To: This parameter allows the technician to input selected speed ranges to customize the gains for the various speeds selected.

These speed parameters are directly tied to the Speed To field for the acceleration rates previously noted on the Speeds setup screen. Changing these speed parameters will replace the Speed To values entered at the Speeds setup screen.

Proportional Gain: Proportional Gain correlates to Elastic Stiffness in the control loop. Increasing the proportional gain increases the static stiffness, but decreases response of the servo loop.

Under proportional loop control the drive system will apply a restoring torque to the motor in proportion the position error of the axis.

With a Proportional Gain too high, the system will be unstable which will result in overshoots, and a generally "nervous" and shaky axis. This is also referred to as a "hot" control loop.

With a proportional gain too low, the system will respond in a loose or sloppy manner. This can be seen in the test pattern when the outside corners become rounded and the circle segments do not all meet in the center.

Integral Gain: Integral Gain improves the positioning accuracy of the control loop.

Integral gain can be used to compensate for static friction or gravity. Excessive Integral Gain can result in system instability.

For most shape cutting machines, this parameter should be set to zero (0).

Derivative Gain: Derivative Gain helps to dampen sudden changes in velocity.

The higher the derivative gain, the slower the response time to the control loop.

For most velocity loop drives this parameter should be set to zero (0).

Feedforward Gain: Feedforward Gain can be used to drive the following error to zero during machine motion.

In all digital control loops there is a finite amount of error that is introduced by the velocity command. Increasing Feedforward Gain can reduce this introduced error.

Velocity Gain: When using a current loop amplifier, the internal velocity loop in the CNC can be used to provide dampening without an external tachometer.

Using the internal velocity loop with a current loop amplifier can result in higher static stiffness, smoother machine motion, and less overshoot.

Skew Error Tolerance: If the Dual Gantry Axis is installed, the user will be prompted for Skew Error Tolerance. This is the amount of position error allowed between the master and slave gantry drive axes prior to an error message being displayed.

This value should match the value used for the Rail for proper operation.

Encoder Counts per mm (inch): To determine the encoder counts per mm (inch), you will need to know the following measurements for your encoders:

- counts per revolution of the motor
- gear ratio
- distance of travel in one revolution of the pinion gear
- diameter of the pinion gear when it engages the rack

An example of the equation is shown below:

1000 line encoder multiplied by 4 (quadrature) per 1 motor revolution

10:1 gear ratio

Distance traveled in one revolution of the pinion

$$\frac{4000 \text{ counts}}{1 \text{ revolution}} \times \frac{10 \text{ rev of motor}}{1 \text{ rev of pinion}} \times \frac{1 \text{ rev}}{2\Pi \text{ inches}} = \frac{X \text{ counts}}{\text{inches}}$$

Encoder counts per unit (inches or mm)

Circumference of pinion (2 inch diameter multiplied by *pi*)

Drive Type: This parameter is used to tell the CNC what type of control loop to run. If you are running an external velocity loop drive (indicated by having an integrated tachometer in the motor), select Velocity. If you are running in torque mode (no tachometer), select Current.

DAC Polarity: This parameter allows changing the analog output polarity to establish proper control loop feedback without any wiring changes.

Encoder Polarity: This parameter allows changing the encoder input polarity to establish proper counting for positive machine motion without any wiring changes.

Encoder Decode Mode: Currently the CNC only supports 4X encoder decode mode. This has been done to increase positional accuracy.

Use Home Limit Switch: Selecting “Yes” will enable the Home feature for the Dual Gantry Axis.

The Z Home Switch must first be defined and mapped to an input location in the screen to enable this feature.

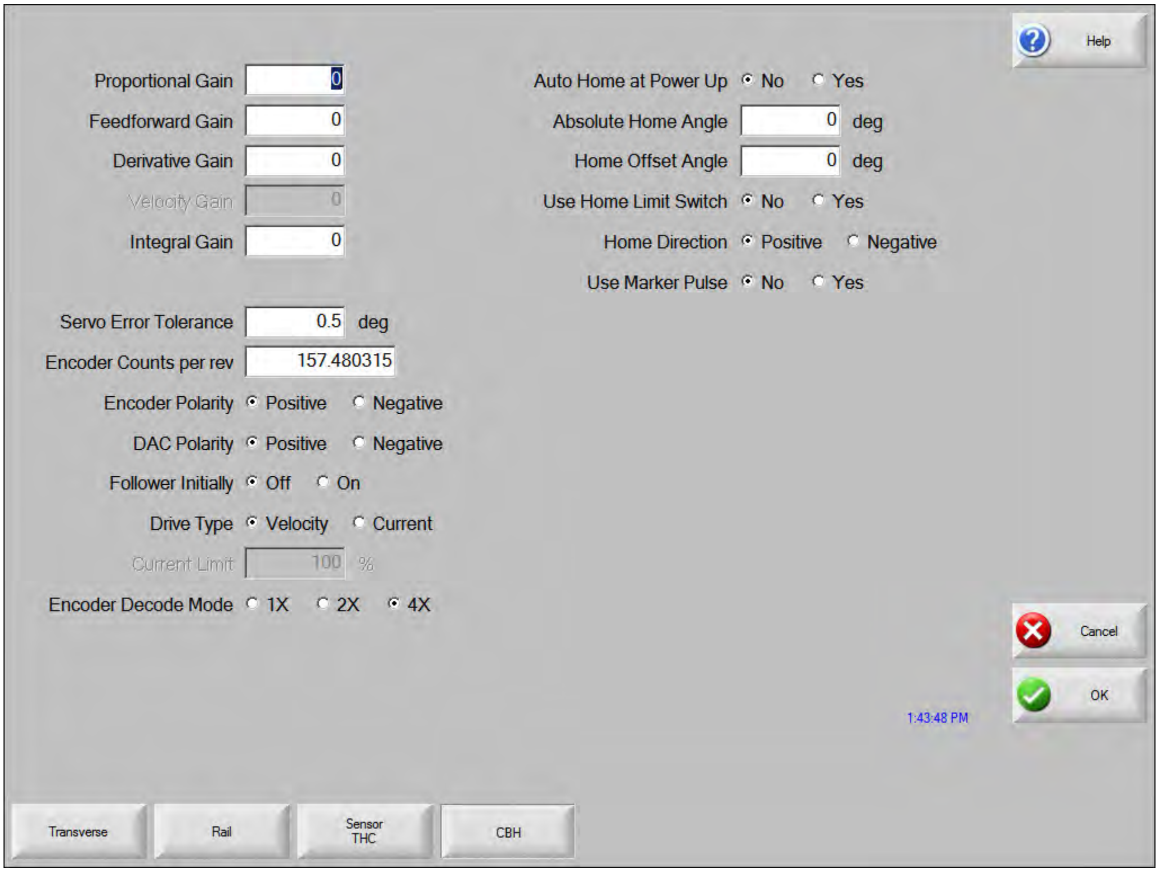
Switch Offset Distance: The Switch Offset Distance is used to specify any physical position offset between the Dual Gantry and Rail Home Switch positions. This allows the CNC to position the two axes accurately for operation and remove any skew of the gantry.

Backlash Compensation: The Backlash Compensation parameter is used to offset or compensate for any backlash in the mechanics of the drive system.

Laser Compensation: Uses position readings gathered with a laser interferometer to correct the actual position of the axes (Transverse, Dual Transverse, Rail, and Dual Gantry only).

CBH Axis

The following setup screen is available if the cutting system has for a contour bevel head.



Proportional Gain: Proportional Gain correlates to elastic stiffness in the control loop. Increasing the proportional gain increases the static stiffness, but decreases response of the servo loop.

Under proportional loop control, the drive system will apply a restoring torque to the motor in proportion to the position error of the axis.

With a Proportional Gain too high, the system will be unstable which will result in overshoots and a generally “nervous” and shaky axis. This is also referred to as a “hot” control loop.

With a proportional gain too low, the system will respond in a loose or sloppy manner. This can be seen in the test pattern when the outside corners become rounded and the circle segments do not all meet in the center.

Integral Gain: Integral Gain improves the positioning accuracy of the control loop.

Integral Gain can be used to compensate for static friction or gravity. Excessive Integral Gain can result in system instability.

For most shape cutting machines, this parameter should be set to zero (0).

Derivative Gain: Derivative Gain helps to dampen sudden changes in velocity. The higher the Derivative Gain, the slower the response time to the control loop.

For most velocity loop drives, this parameter will be set to zero (0).

Feedforward Gain: Feedforward Gain can be used to drive the following error to zero during machine motion. In all digital control loops there is a finite amount of error that is introduced by the velocity command.

Increasing Feedforward Gain can reduce this introduced error.

Velocity Gain: When using a current loop amplifier, the internal velocity loop in the CNC can be used to provide dampening without an external tachometer.

Using the internal velocity loop with a current loop amplifier can result in higher static stiffness, smoother machine motion, and less overshoot.

Servo Error Tolerance: Servo error, also called following error, is the difference between the commanded motor position and the actual motor position. The servo error tolerance is the upper limit of the amount of following error allowed before the CNC faults.

The amount of servo error tolerance depends on the cutting system mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high could cause inaccurate motion or mechanical harm. Set the Following Error parameter in the Watch window and observe steady-state operation of the cutting system (some following error is normal). Set the servo error tolerance to a value slightly higher than the steady-state following error.

Encoder Counts per rev: Enter a value that is the number of encoder edges per revolution of the Contour Bevel Head axis.

It is possible to enter fractional encoder units and the CNC will keep track of these fractions automatically. Encoder Counts per rev are equal to the resolution of the encoder multiplied by the encoder revolutions per rev.

For example, the resolution of a 4X – 1000 line encoder counts both edges (lines) of channel A and channel B to equal 4 counts per line time multiplied by the 1000 lines per revolution would equal 4000 counts per revolution.

Encoder Counts/Rev = 4 Counts/Line x 1000 Lines/Rev = 4000

Drive Type: This parameter is used to tell the CNC what type of control loop to run.

If you are running an external velocity loop drive (indicated by having an integrated tachometer in the motor), select Velocity. If you are running in torque mode (no tachometer), select Current.

DAC Polarity: This parameter allows changing of the analog output polarity to establish proper control loop feedback without any wiring changes.

Encoder Polarity: This parameter allows changing of the encoder input polarity to establish proper counting for positive machine motion without any wiring changes.

Encoder Decode Mode: Currently the CNC only supports 4X encoder decode mode. This has been done to increase positional accuracy.

Follower Initially: This parameter is used to determine if the CBH axis will be On when the CNC is first powered up.

Auto Home At Power Up: Determines if the contour bevel head will automatically go into the homing routine upon powering up the CNC.

Absolute Home Angle: Defines the position of the axis when the home limit switch or marker pulse is detected.

Home Offset Angle: Allows the user to set an offset angle from the home limit switch.

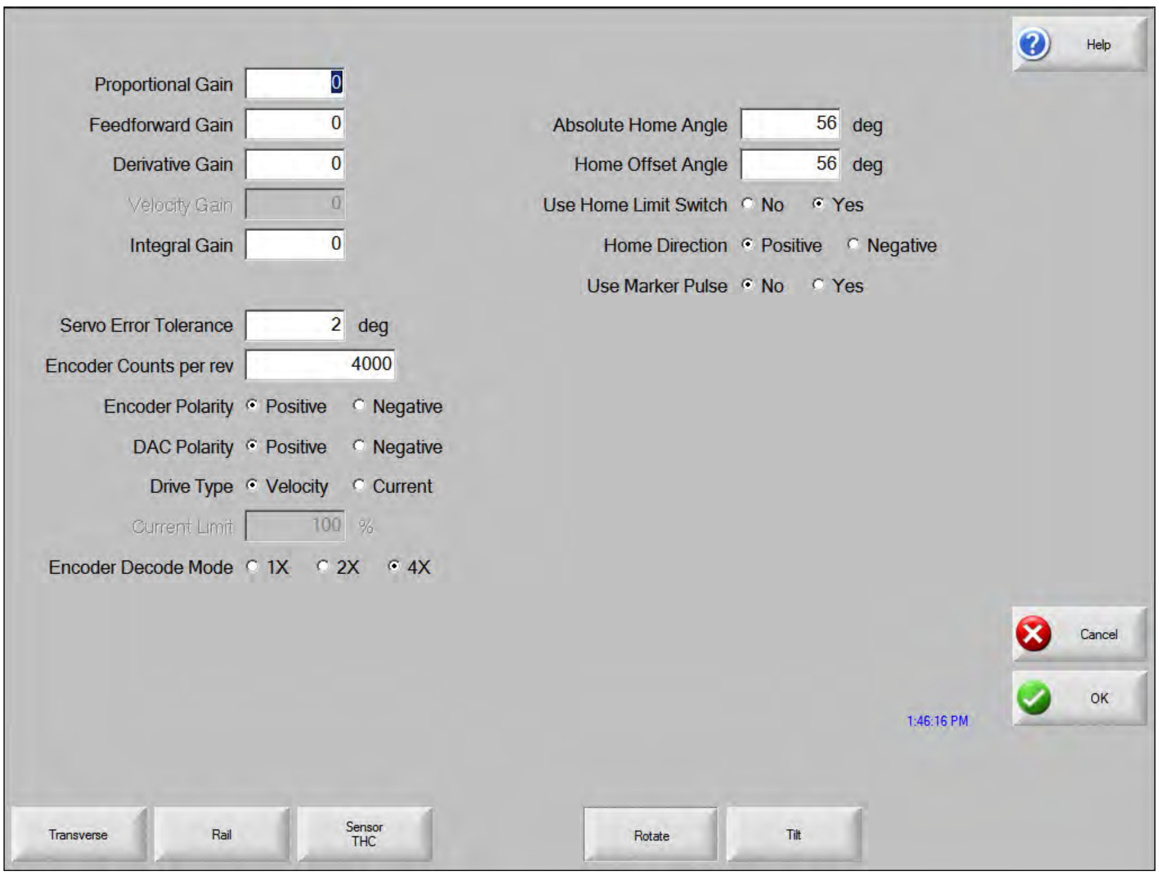
Use Home Limit Switch: Tells the CNC to look for a home limit switch during phase one of the homing sequence.

Home Direction: Determines which direction the axes will travel during phase one of the homing sequence.

Use Marker Pulse: When enabled, the absolute home position will be assigned at the instant the marker pulse is detected. It is recommended that the marker pulse be used for optimal homing repeatability.

Rotate Axis

In Phoenix, a Rotate Axis is equivalent to a C axis in the standard orientation (X is Transverse and Y is rail).



Proportional Gain: Proportional Gain correlates to Elastic Stiffness in the control loop. Increasing the proportional gain increases the static stiffness, but decreases response of the servo loop.

Under proportional loop control, the drive system will apply a restoring torque to the motor in proportion to the position error of the axis.

With a Proportional Gain too high, the system will be unstable which will result in overshoots and a generally “nervous” and shaky axis. This is also referred to as a “hot” control loop.

With a proportional gain too low, the system will respond in a loose or sloppy manner. This can be seen in the test pattern when the outside corners become rounded and the circle segments do not all meet in the center.

Integral Gain: Integral Gain improves the positioning accuracy of the control loop.

Integral Gain can be used to compensate for static friction or gravity. Excessive Integral Gain can result in system instability.

For most shape cutting machines, this parameter should be set to zero (0).

Derivative Gain: Derivative Gain helps to dampen sudden changes in velocity. The higher the Derivative Gain, the slower the response time to the control loop.

For most velocity loop drives, this parameter will be set to zero (0).

Feedforward Gain: Feedforward Gain can be used to drive the following error to zero during machine motion. In all digital control loops there is a finite amount of error that is introduced by the velocity command.

Increasing Feedforward Gain can reduce this introduced error.

Velocity Gain: When using a current loop amplifier, the internal velocity loop in the CNC can be used to provide dampening without an external tachometer.

Using the internal velocity loop with a current loop amplifier can result in higher static stiffness, smoother machine motion, and less overshoot.

Servo Error Tolerance: Servo error, also called following error, is the difference between the commanded motor position and the actual motor position. The servo error tolerance is the upper limit of the amount of following error allowed before the CNC faults.

The amount of servo error tolerance depends on the cutting system mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high could cause inaccurate motion or mechanical harm. Set the Following Error parameter in the Watch window and observe steady-state operation of the cutting system (some following error is normal). Set the servo error tolerance to a value slightly higher than the steady-state following error.

Encoder Counts per rev: Enter a value that is the number of encoder edges per revolution of the Contour Bevel Head axis.

It is possible to enter fractional encoder units and the CNC will keep track of these fractions automatically. Encoder Counts per rev are equal to the resolution of the encoder multiplied by the encoder revolutions per rev.

For example, the resolution of a 4X – 1000 line encoder counts both edges (lines) of channel A and channel B to equal 4 counts per line multiplied by the 1000 lines per revolution would equal 4000 counts per revolution.

Encoder Counts/Rev = 4 Counts/Line x 1000 Lines/Rev = 4000

Drive Type: This parameter is used to tell the CNC what type of control loop to run.

If you are running an external velocity loop drive (indicated by having an integrated tachometer in the motor), select Velocity. If you are running in torque mode (no tachometer), select Current.

DAC Polarity: This parameter allows changing the analog output polarity to establish proper control loop feedback without any wiring changes.

Encoder Polarity: This parameter allows changing of the encoder input polarity to establish proper counting for positive machine motion without any wiring changes.

Encoder Decode Mode: Currently the CNC only supports 4X encoder decode mode. This has been done to increase positional accuracy.

Absolute Home Angle: Defines the position of the axis when the home limit switch or marker pulse is detected.

Home Offset Angle: Allows the user to set an offset angle from the home limit switch.

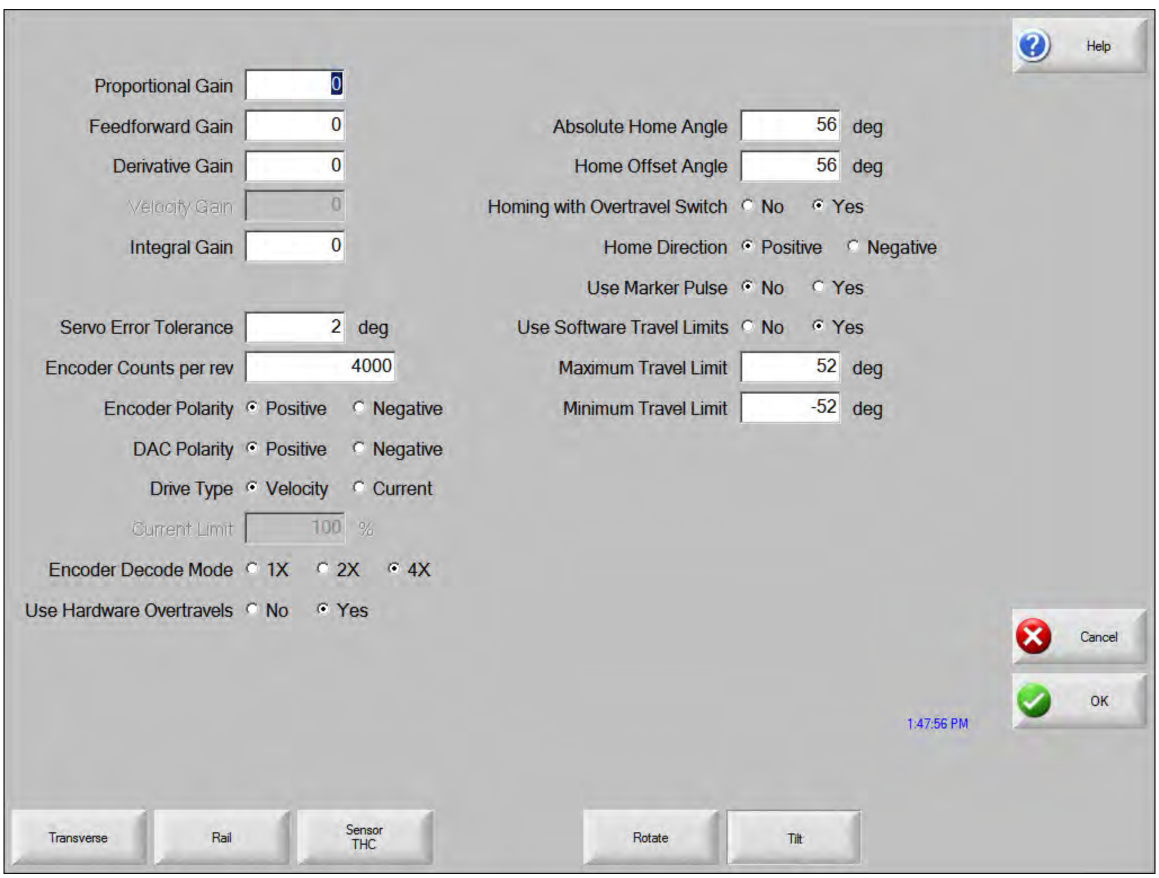
Use Home Limit Switch: Tells the CNC to look for a home limit switch during phase one of the homing sequence.

Home Direction: Determines which direction the axes will travel during phase one of the homing sequence.

Use Marker Pulse: When enabled, the absolute home position will be assigned at the instant the marker pulse is detected. It is recommended that the marker pulse be used for optimal homing repeatability.

Tilt Axis

In the Phoenix software, the Tilt axis is equivalent to the A axis in the standard orientation (X is transverse and Y is Rail).



Proportional Gain: Proportional Gain correlates to elastic stiffness in the control loop. Increasing the proportional gain increases the static stiffness, but decreases response of the servo loop.

Under proportional loop control, the drive system will apply a restoring torque to the motor in proportion to the position error of the axis.

With a Proportional Gain too high, the system will be unstable which will result in overshoots and a generally “nervous” and shaky axis. This is also referred to as a “hot” control loop.

With a proportional gain too low, the system will respond in a loose or sloppy manner. This can be seen in the test pattern when the outside corners become rounded and the circle segments do not all meet in the center.

Integral Gain: Integral Gain improves the positioning accuracy of the control loop.

Integral Gain can be used to compensate for static friction or gravity. Excessive Integral Gain can result in system instability.

For most shape cutting machines, this parameter should be set to zero (0).

Derivative Gain: Derivative Gain helps to dampen sudden changes in velocity. The higher the Derivative Gain, the slower the response time to the control loop.

For most velocity loop drives, this parameter will be set to zero (0).

Feedforward Gain: Feedforward Gain can be used to drive the following error to zero during machine motion. In all digital control loops there is a finite amount of error that is introduced by the velocity command.

Increasing Feedforward Gain can reduce this introduced error.

Velocity Gain: When using a current loop amplifier, the internal velocity loop in the CNC can be used to provide dampening without an external tachometer.

Using the internal velocity loop with a current loop amplifier can result in higher static stiffness, smoother machine motion, and less overshoot.

Servo Error Tolerance: Servo error, also called following error, is the difference between the commanded motor position and the actual motor position. The servo error tolerance is the upper limit of the amount of following error allowed before the CNC faults.

The amount of servo error tolerance depends on the cutting system mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high could cause inaccurate motion or mechanical harm. Set the Following Error parameter in the Watch window and observe steady-state operation of the cutting system (some following error is normal). Set the servo error tolerance to a value slightly higher than the steady-state following error.

Encoder Counts per rev: Enter a value that is the number of encoder edges per revolution of the Contour Bevel Head axis.

It is possible to enter fractional encoder units and the CNC will keep track of these fractions automatically. Encoder Counts per rev are equal to the resolution of the encoder multiplied by the encoder revolutions per rev.

For example, the resolution of a 4X – 1000 line encoder counts both edges (lines) of channel A and channel B to equal 4 counts per line time multiplied by the 1000 lines per revolution would equal 4000 counts per revolution.

Encoder Counts/Rev = 4 Counts/Line x 1000 Lines/Rev = 4000

Encoder Polarity: This parameter allows changing of the encoder input polarity to establish proper counting for positive machine motion without any wiring changes.

DAC Polarity: This parameter allows changing of the analog output polarity to establish proper control loop feedback without any wiring changes.

Drive Type: This parameter is used to tell the CNC what type of control loop to run.

If you are running an external velocity loop drive (indicated by having an integrated tachometer in the motor), select Velocity. If you are running in torque mode (no tachometer), select Current.

Current Limit: If you selected Current for Drive Type, set the maximum current output for operation of the lifter motor.

Encoder Decode Mode: Currently the CNC only supports 4X encoder decode mode. This has been done to increase positional accuracy.

Use Hardware Overtravels: Select whether the cutting system will be using hardware overtravel switches. If Hardware overtravel switches are used, the CNC will disable feedback and display an error message if the inputs become active. It is recommended that hardware overtravel switches be installed.

Absolute Home Angle: Defines the position of the axis when the home limit switch or marker pulse is detected.

Home Offset Angle: Allows the user to set an offset angle from the home limit switch.

Homing with Overtravel Switch: Activates the Home feature and uses the Overtravel Switch as its reference.

The Home feature is used to set a known absolute physical position location for the tilt axis.

Home Direction: Determines which direction the axes will travel during phase one of the homing sequence.

Use Marker Pulse: When enabled, the absolute home position will be assigned at the instant the marker pulse is detected. It is recommended that the marker pulse be used for optimal homing repeatability.

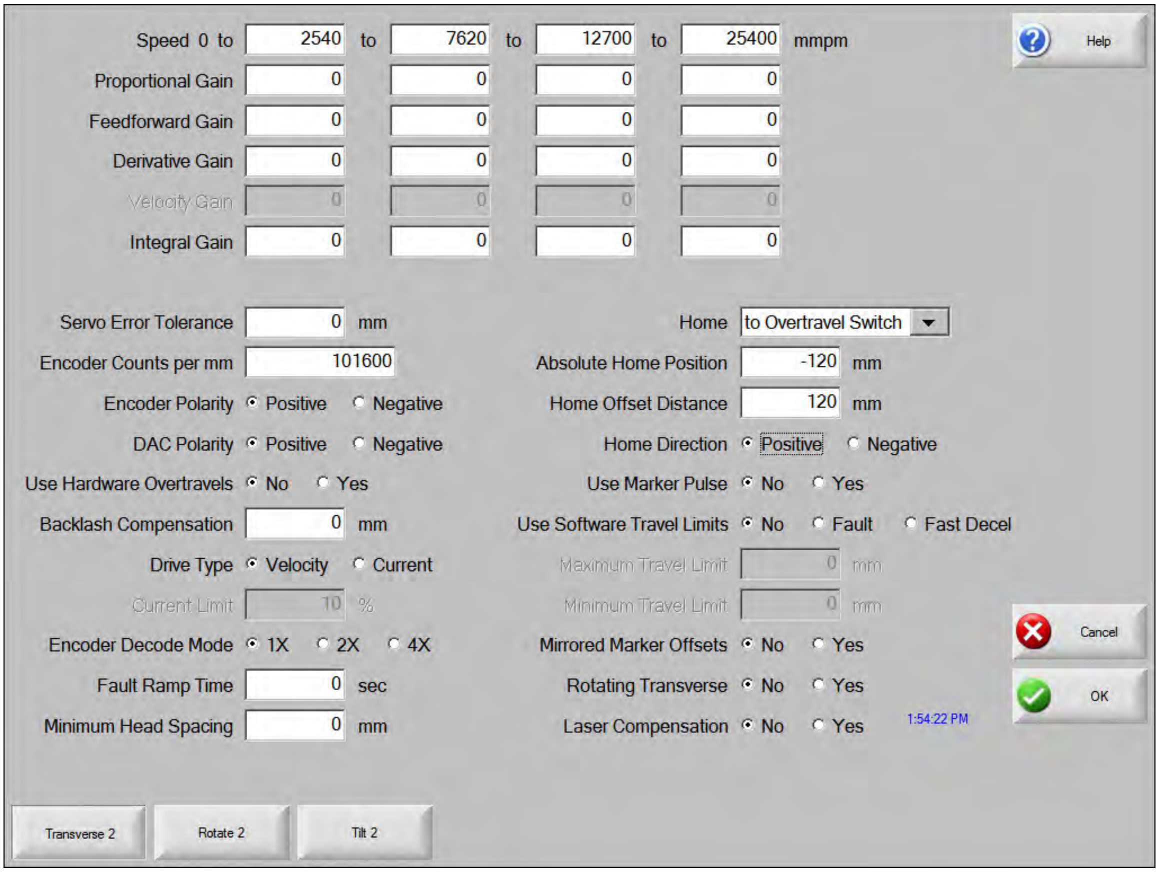
Use Software Travel Limits: The CNC is capable of running with software overtravel limits that are based on position. When enabled, this feature allows you to select Fault or Fast Decel when active. Fault operates as hardware switches with immediate fault. Fast Decel uses the fast decel value to ramp down motion.

Maximum Travel Limit: If Software Overtravels are enabled, the user is prompted for the maximum travel of the cutting machine.

Minimum Travel Limit: If Software Overtravels are enabled, the user is prompted for the minimum travel of the cutting machine.

Transverse 2 Axis

The Transverse 2 axis is also called a Dual Transverse axis. It can be used for either a second cutting station or as an axis that turns a chuck for pipe or rectangular tube cutting.



Speed 0 To: Specifies speed ranges and customizes the acceleration rates for these speeds. When a speed range that is lower than the maximum speed is entered, a new range at the maximum machine speed setting is created. You can set a maximum of 5 speed ranges.

To reduce the number of Speed To ranges, enter the maximum speed in the highest range and the additional range fields will be removed from the screen.

Proportional Gain: Proportional Gain correlates to elastic stiffness in the control loop. Increasing the proportional gain increases the static stiffness, but decreases response of the servo loop.

Under proportional loop control, the drive system will apply a restoring torque to the motor in proportion to the position error of the axis.

With a Proportional Gain too high, the system will be unstable which will result in overshoots and a generally “nervous” and shaky axis. This is also referred to as a “hot” control loop.

With a proportional gain too low, the system will respond in a loose or sloppy manner. This can be seen in the test pattern when the outside corners become rounded and the circle segments do not all meet in the center.

Integral Gain: Integral Gain improves the positioning accuracy of the control loop.

Integral Gain can be used to compensate for static friction or gravity. Excessive Integral Gain can result in system instability.

For most shape cutting machines, this parameter should be set to zero (0).

Derivative Gain: Derivative Gain helps to dampen sudden changes in velocity. The higher the Derivative Gain, the slower the response time to the control loop.

For most velocity loop drives, this parameter will be set to zero (0).

Feedforward Gain: Feedforward Gain can be used to drive the following error to zero during machine motion. In all digital control loops there is a finite amount of error that is introduced by the velocity command.

Increasing Feedforward Gain can reduce this introduced error.

Velocity Gain: When using a current loop amplifier, the internal velocity loop in the CNC can be used to provide dampening without an external tachometer.

Using the internal velocity loop with a current loop amplifier can result in higher static stiffness, smoother machine motion, and less overshoot.

Servo Error Tolerance: Servo error, also called following error, is the difference between the commanded motor position and the actual motor position. The servo error tolerance is the upper limit of the amount of following error allowed before the CNC faults.

The amount of servo error tolerance depends on the cutting system mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high could cause inaccurate motion or mechanical harm. Set the Following Error parameter in the Watch window and observe steady-state operation of the cutting system (some following error is normal). Set the servo error tolerance to a value slightly higher than the steady-state following error.

Encoder Counts per mm (inch): To determine the encoder counts per mm (inch), you will need to know the following measurements for your encoders:

- counts per revolution of the motor
- gear ratio
- distance of travel in one revolution of the pinion gear
- diameter of the pinion gear when it engages the rack

An example of the equation is shown below:

1000 line encoder multiplied by 4 (quadrature) per 1 motor revolution

10:1 gear ratio

Distance traveled in one revolution of the pinion

$$\frac{4000 \text{ counts}}{1 \text{ revolution}} \times \frac{10 \text{ rev of motor}}{1 \text{ rev of pinion}} \times \frac{1 \text{ rev}}{2\Pi \text{ inches}} = \frac{X \text{ counts}}{\text{inches}}$$

Encoder counts per unit (inches or mm)

Circumference of pinion (2 inch diameter multiplied by *pi)*

Fault Ramp Time: This parameter sets the motion deceleration time after a fault occurs. At the end of “Fault Ramp Time, The drives will be disabled.

Drive Type: This parameter is used to tell the CNC what type of control loop to run.

If you are running an external velocity loop drive (indicated by having an integrated tachometer in the motor), select Velocity. If you are running in torque mode (no tachometer), select Current.

DAC Polarity: This parameter allows changing of the analog output polarity to establish proper control loop feedback without any wiring changes.

Encoder Polarity: This parameter allows changing of the encoder input polarity to establish proper counting for positive machine motion without any wiring changes.

Encoder Decode Mode: Currently the CNC only supports 4X encoder decode mode. This has been done to increase positional accuracy.

Use Hardware Overtravels: Select whether the cutting system will be using hardware overtravel switches. If Hardware overtravel switches are used, the CNC will disable feedback and display an error message if the inputs become active. It is recommended that hardware overtravel switches be installed.

Backlash Compensation: The Backlash Compensation parameter is used to offset or compensate for any backlash in the mechanics of the drive system.

Minimum Head Spacing: Sets the minimum distance that is allowed between the Transverse 1 and Transverse 2 axes.

Home: The Home parameter is used to activate use of the Home feature. Depending on configuration, the table may be Homed to either a designated Home Switch or an Overtravel Switch.

The Home feature is used to set a known absolute physical position location on the cutting table that is used for referencing future manual Go to Home and other motion commands. This is generally performed through activation of a home switch positioned on the appropriate axis giving it a known physical location.

When you enter a homing command, the CNC moves the axis toward the home switches at the Fast Home Speed until the switches activate. Once the switches activate, motion stops and then the axis moves in the opposite direction off switch at the Slow Home Speed. The moment that the switch deactivates, the CNC records the position which provides an absolute reference point for future motion commands.

Absolute Home Position: Defines the position of the axis when the Home Limit Switch or Marker Pulse is detected.

Home Offset Distance: Allows the user to set an offset distance from the Home Limit Switch.

Home Direction: Determines which direction the axes will travel during phase one of the homing sequence.

Use Marker Pulse: When enabled, the absolute home position will be assigned at the instant the marker pulse is detected. It is recommended that the Marker Pulse be used for optimal homing repeatability.

Use Software Travel Limits: The CNC is capable of running with software Overtravel limit switches based on position. When enabled, this feature allows the user to select the fault logic of fault or Fast Decel when active. Fault operates as hardware switches with immediate fault. Fast Decel uses the fast Decel value to ramp down motion.

Maximum/Minimum Travel Limit: If Software Overtravels are enabled, the user is prompted for the maximum or minimum travel of the cutting machine.

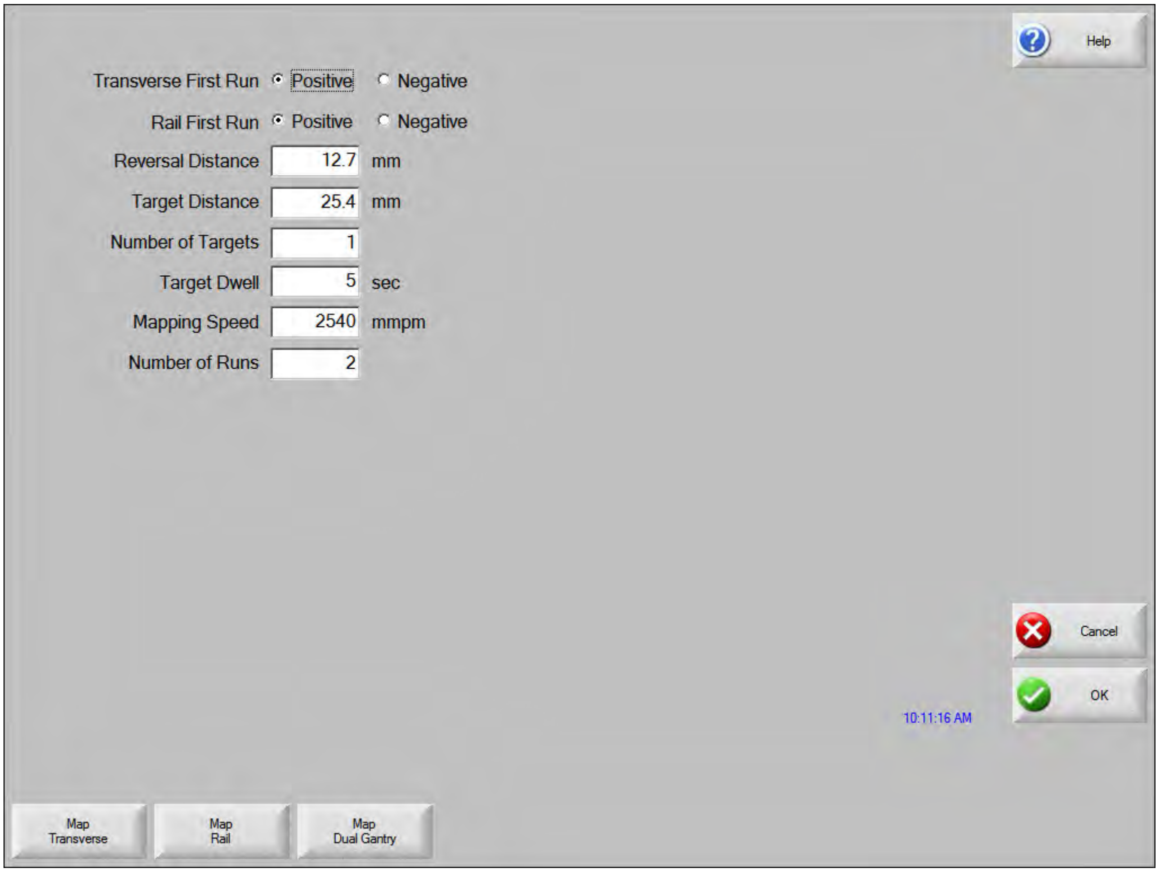
Mirrored Marker Offsets: Allows preset tool offsets to be performed as mirrored motion for the transverse 2 axis.

Laser Compensation: Uses position readings gathered with a laser interferometer to correct the actual position of the axes (Transverse, Dual Transverse, Rail, and Dual Gantry only).

Laser Mapping

The Laser Mapping screen provides an interface for the laser interferometer. The CNC uses the parameters in the Laser Mapping screen to generate a CNC part program that allows the laser interferometer to map the position of each drive axis (rail, transverse, and dual gantry) at specified points (or targets).

After the parameters are set and the laser interferometer is connected to the axis, press the mapping soft key for the appropriate axis. The CNC will automatically load and run the part program.



Rail/Transverse First Run: The direction of the first run on the Rail or Transverse axis. The selection should be based on the direction of motion from home position.

Reversal Distance: The distance of travel in the opposite direction at the beginning and end of a run.

Use: Reverse motion removes mechanical backlash before mapping the axis.

Target Distance: Sets the distance between targets.

Use: This value should be 20 inches or more.

Number of Targets: The number of pauses on each run where the laser interferometer measures the physical position of the axis. The part program that the CNC creates for mapping includes a pause (dwell) at each of these targets.

Use: Enter values between 2 and 1000.

Target Dwell: The amount of time that motion pauses at each target to allow the interferometer to take each measurement and record the results.

Use: Refer to the instruction manual for your laser interferometer to determine how to set this value.

Mapping Speed: The program speed of the part program that the CNC creates for mapping.

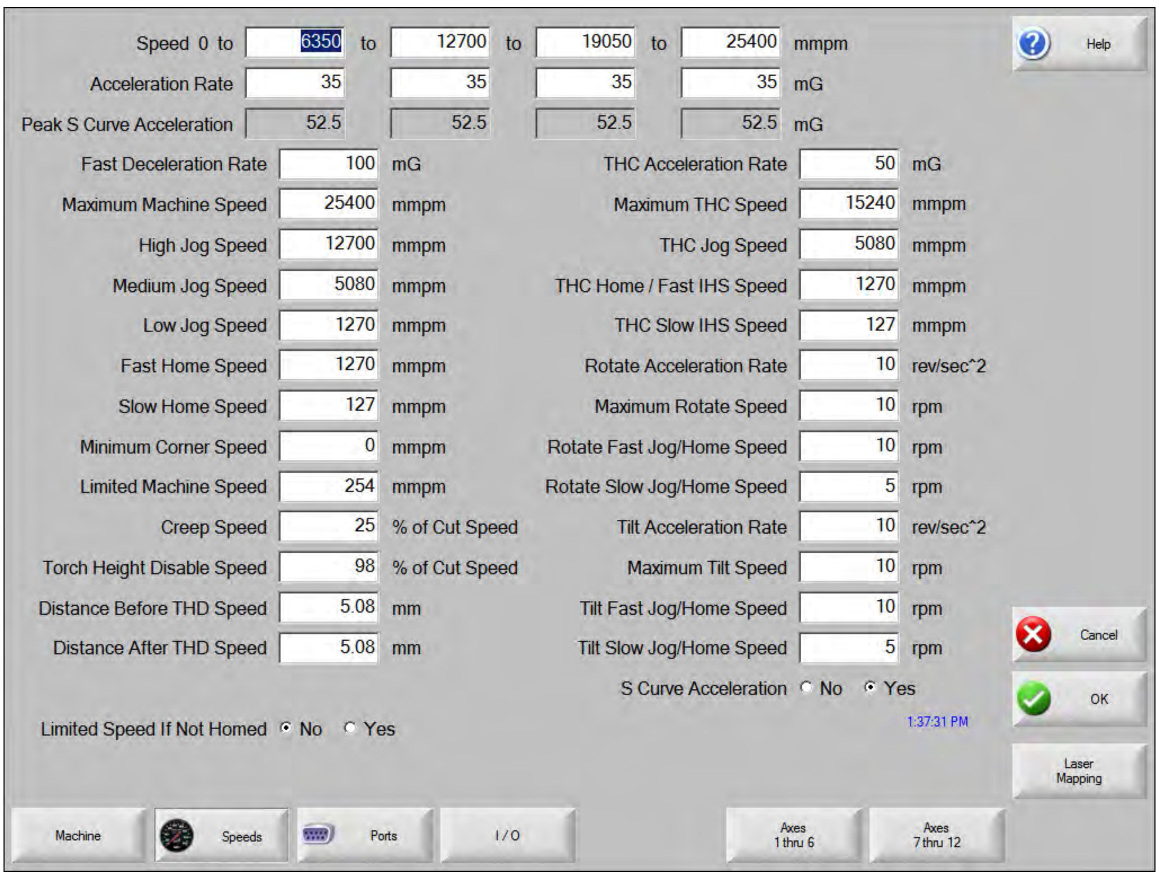
Number of Runs: The number of runs that the interferometer makes on the axis. Set this to a minimum of 2.

Use: Set this to an even number between 2 and 1000 because the interferometer must travel to the end of the axis and then return. The greater the number of runs, the more precise the motion compensation can be.

Setting Speeds

The Speeds setup screen is where you set the machine speeds for the operating modes.

The CNC is capable of operating over a wide range of speeds, depending on the drives, motors, gearboxes, and mechanics of the system.



Speed 0 To: Specifies speed ranges (or breaks) and customizes the acceleration rates for these speeds. When a speed range that is lower than the maximum speed is entered, a new range at the maximum machine speed setting is created. You can set a maximum of 5 speed ranges.

To reduce the number of Speed To ranges, enter the maximum speed in the highest range and the additional range fields will be removed from the screen.

Acceleration Rate: Determines both the acceleration and deceleration rate for all motion.

All mechanical systems have different acceleration and deceleration rates to move the cutting device smoothly. The higher the acceleration rate, the quicker the machine will get up to speed. The lower the acceleration rate, the smoother the machine will position the cutting device.

Enter a value in milliGs for the desired acceleration rate. One milliG is approximately 0.384 inches per second^2. A recommended, initial value for this field is 10 milliGs.

Peak S Curve Acceleration displays the maximum acceleration used during the beginning of a cut and represents a rate of acceleration that is 50% greater than the average rate of acceleration. This parameter appears when S Curve Acceleration (at the lower left in the Speeds Screen) is set to Yes.

Fast Deceleration Rate: This parameter is activated by the Fast Stop or Torch Collision inputs and specifies in milliGs how quickly the system will stop when the Fast Stop Input is active. Generally, this value is much higher than the acceleration rate.

One milliG is approximately 0.384 inches per second^2.

Maximum Machine Speed: Enter the maximum value at which the system is capable of contouring. This parameter limits the range of all subsequent speed entries. In addition, it scales the speedometer in the Watch Window during runtime operation.

The maximum speed at which the CNC can contour can be calculated with the following equation.

Maximum Contouring Rate (IPM) =

$$\frac{\text{60 seconds}}{\text{minute}} \times \frac{\text{1 inch}}{\text{user edges}} \times \frac{\text{100,000 edges}}{\text{second}}$$

For example: the CNC could command a cutting machine with 4000 edges per inch resolution at 1,500 IPM.

Limited Machine Speed: Defines maximum machine speed when the Limited Speed Input is active. This is commonly used to set a reduced machine speed for increased safety during machine testing or maintenance.

High Jog Speed: Defines the high speed for manual motion.

Medium Jog Speed: Defines the medium speed for manual motion.

Low Jog Speed: Defines the low speed for manual motion.

Minimum Corner Speed: Defines the minimum speed to use when negotiating corners. The CNC has centripetal velocity and acceleration contouring algorithms which provide optimal cut quality for most cut operations. In some situations, a minimum corner speed prevents over-burning at corners. In normal operations, this parameter should be set to zero.

Fast Home Speed: Defines the speed that the CNC uses during the first phase of the homing sequence. During the first phase, the cutting device moves toward the home limit switches at the fast rate.

Slow Home Speed: Defines the speed that the CNC uses during the second phase of the homing sequence. During the second phase, the cutting device moves off the home limit switch and proceeds to the marker pulse. When the CNC detects the marker pulse, it decelerates to a stop and performs a small move in the opposite direction to the marker pulse.

Creep Speed Percentage: Defines what percentage of the machine cut speed will be used when creeping. Generally, creep speed is 25% of the cut speed.

Torch Height Disable Speed: Specifies the percentage of the program cut speed at which the CNC disables the height control. For example, when the cutting system is slowed to cut a part feature such as a corner, the arc voltage increases which would cause the torch to lower into the workpiece. When the cutting system slows down to this percentage of the cut speed, the CNC disables THC voltage tracking.

For more information, see the Torch Height Disable section in this chapter.

Distance Before THD Speed: This distance is active whenever the program cut speed drops below the Torch Height Disable Speed, or segments intersect at an angle greater than the tangent angle. Specifies the distance before turning a corner at which the CNCs disables the torch height control.

Distance After THD Speed: Specifies the distance after turning a corner to enable voltage tracking for torch height control. Distance After THD Speed is active whenever the speed drops below the Torch Height Disable Speed, or segments intersect at an angle greater than the tangent angle.

S-curve Acceleration: Select Yes to enable S-curve acceleration. Select No to use trapezoidal acceleration. Depending on the application and table mechanics, S-curve acceleration can reduce or eliminate "jerk" that results from trapezoidal motion. See *S-Curve acceleration* later in this section for more information.

CBH Speeds

The CBH speed setup parameters display only if they have been enabled.

CBH Acceleration Rate: Specifies the acceleration rate for smooth and stable movement of the contoured bevel head. The higher the acceleration rate, the quicker the CBH reaches cutting speed. The lower the acceleration rate, the smoother the machine will position the cutting device. Values are entered in revolutions/second^{2}.

Maximum CBH Speed: Specifies the maximum speed for the CBH rotation in revolutions/minute.

CBH High Jog / Home Speed: Specifies the manual CBH jog and fast Home speed.

CBH Low Jog / Home Speed: Specifies the CBH slow home speed.

THC Speeds

The THC speed setup parameters display only if a Sensor THC has been set up on the Machine Setups screen.

THC Acceleration Rate: Specifies the acceleration rate for smooth and stable movement of the Sensor THC. The higher the acceleration rate, the quicker the THC reaches cutting speed. The lower the acceleration rate, the smoother the machine will position the cutting device. Values are entered in milliGs.

Maximum THC Speed: Specifies the maximum speed for the Sensor THC in inches / minute.

THC Jog Speed: Specifies the THC jog speed.

THC Home/Fast IHS Speed: Specifies the speed that is used to home the torch at the top of the slide and to move the torch from the selected IHS height until it senses the workpiece.

THC Slow IHS Speed: Specifies the speed used to retract the torch after it senses the workpiece. The torch retracts at this speed until contact sense is lost. This determines the exact distance to the workpiece regardless of any flexing of the workpiece. A very low speed is recommended for optimal accuracy.

Tilt/Rotator Speeds

The following Speeds setup parameters are available when Tilt Rotator has been enabled.

Rotate Acceleration Rate: Specifies the acceleration rate for smooth and stable movement of the contour bevel head. The higher the acceleration rate, the quicker the Rotate Axis reaches cutting speed. The lower the acceleration rate, the smoother the machine will position the cutting device. Values are entered in revolutions/second^2.

Maximum Rotate Speed: Specifies the maximum speed for the Rotate Axis rotation in revolutions/minute.

Rotate High Jog Speed: Specifies the high speed for the rotate axis when it jogs.

Rotate Low Jog Speed: Specifies the low speed for the rotate axis when it jogs.

Tilt Acceleration Rate: Specifies the acceleration rate for smooth and stable movement of the contour bevel head. The higher the acceleration rate, the quicker the tilt axis reaches cutting speed. The lower the acceleration rate, the smoother the machine positions the cutting device. Values are entered in revolutions/second^2.

Maximum Tilt Speed: Specifies the maximum speed for the tilt axis, in revolutions/minute.

Tilt High Jog Speed: Specifies the high speed for tilt axis when it jogs.

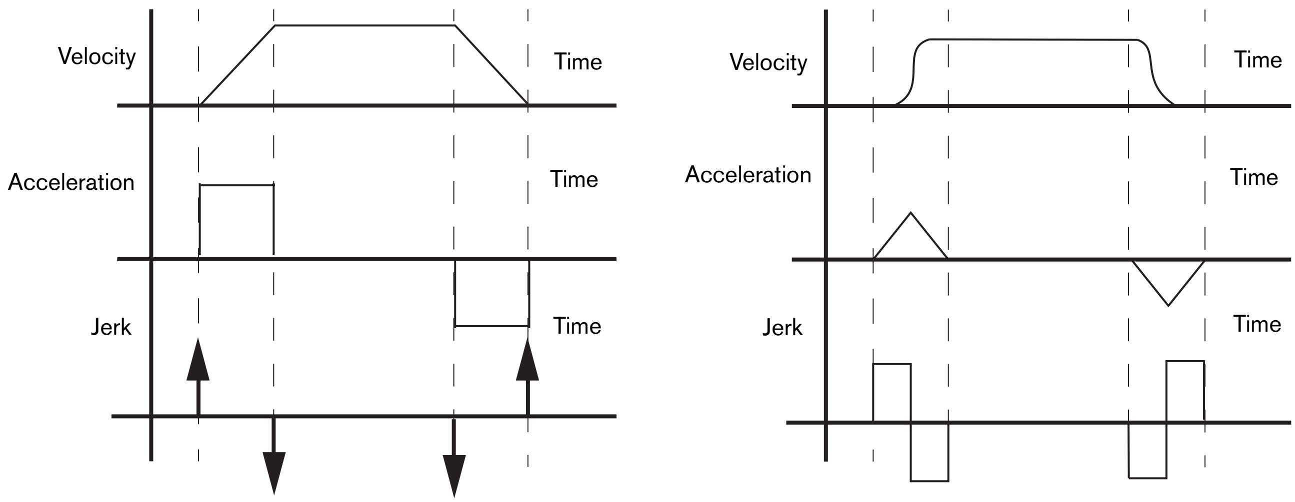
Tilt Low Jog Speed: Specifies the low speed for tilt axis when it jogs.

S-curve acceleration

S-curve motion is a feature of Hypertherm CNCs that allows smoother motion during acceleration than traditional, or trapezoidal, motion.

During trapezoidal motion, the CNC sends a command to the drive systems to accelerate immediately to the desired machine acceleration, as determined in the machine setups. This is the fastest way of reaching the maximum acceleration rate and achieving the desired cut speed. However, this rapid change in acceleration produces “jerk” in the motion of the table that can affect cut quality. Jerk is always greatest when the table is beginning to accelerate or decelerate at the beginning or end of a cut.

During S-curve motion, the CNC sends acceleration commands to the drive system in increments until the system reaches the maximum acceleration rate. The S-curve commands smooth out acceleration and reduce jerk for a given acceleration rate. This may allow the table to reach higher acceleration rates if jerk was the limiting factor when using trapezoidal motion.

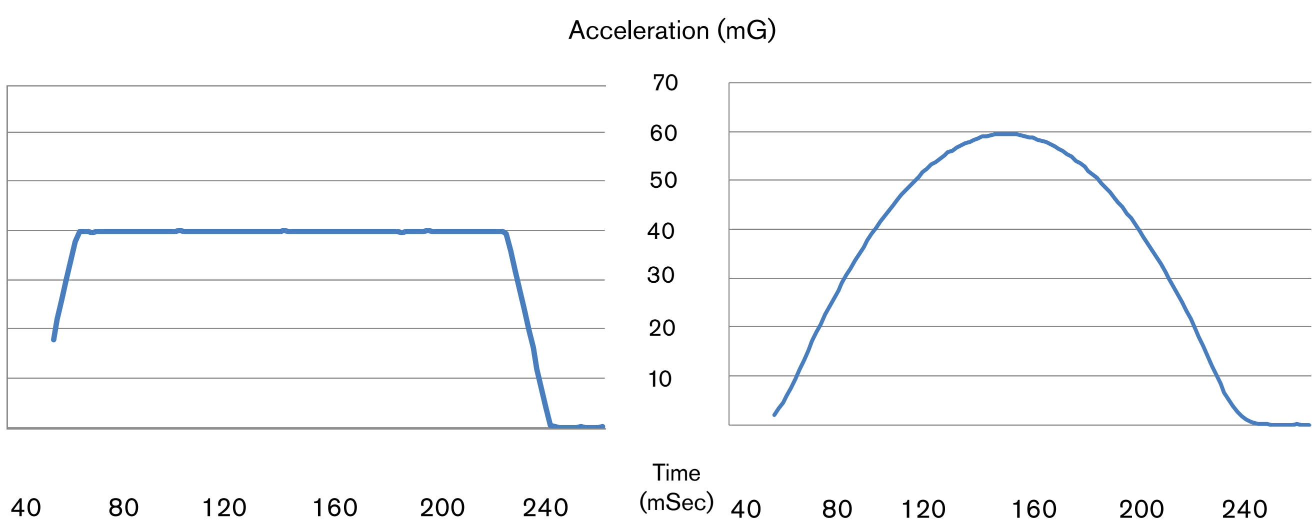


Trapezoidal S Curve

The typical implementation of S-curve motion uses the same peak acceleration as trapezoidal acceleration and results in smoother acceleration. However, this implementation takes more time to reach the desired acceleration rate and more time to complete a specific motion.

In Hypertherm's implementation of S-curve, the CNC uses the same average acceleration rate that was selected for trapezoidal acceleration but uses a peak acceleration that is 50% higher than the average acceleration setting. Increasing the peak acceleration allows S-curve acceleration to execute a segment in the same amount of time as trapezoidal acceleration.

As an example, if 40 mG average acceleration has been selected for trapezoidal motion, when S-curve motion is turned on, the table still accelerates at an average rate of 40 mG but the peak rate of acceleration will reach 60 mG.

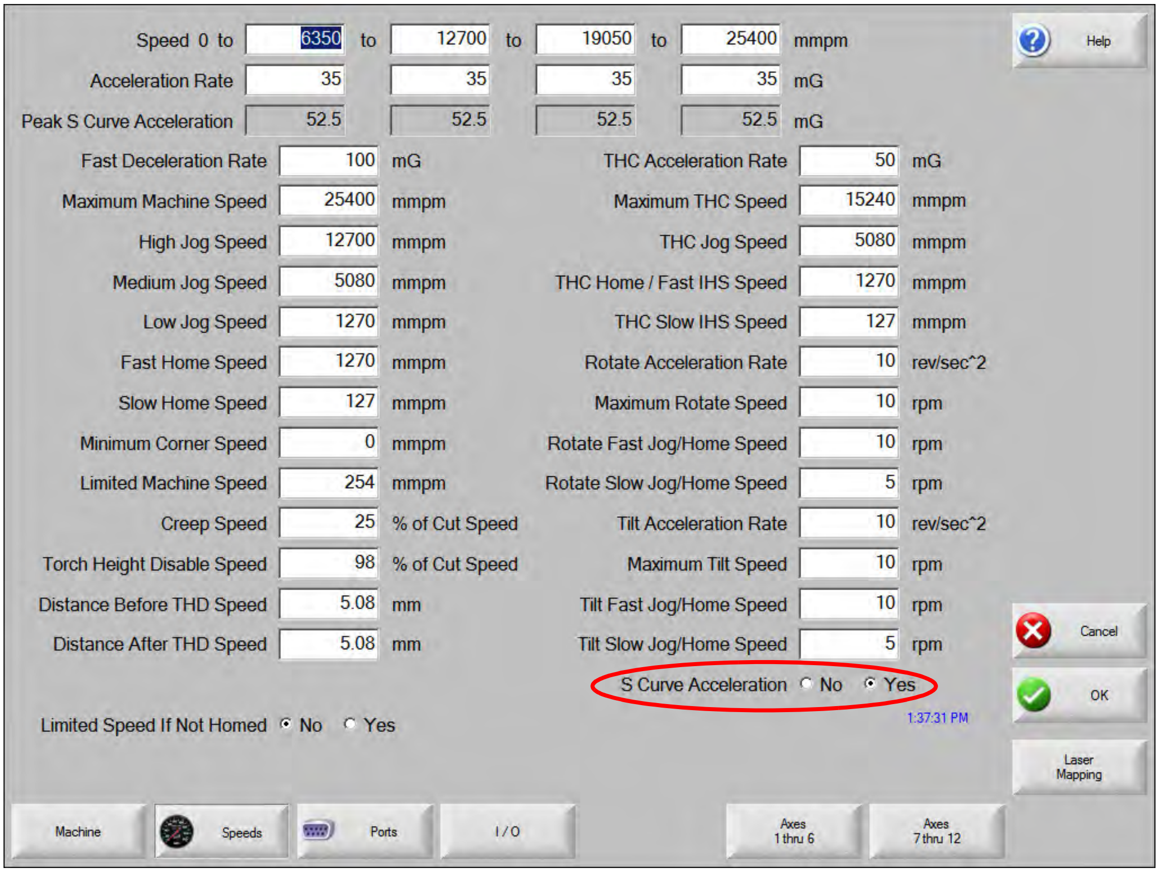


Trapezoidal motion S-curve motion

With a higher peak acceleration, S-curve allows the table to achieve the desired cut speed, with no jerk, in the same amount of time as trapezoidal motion.

S-curve setup

S-curve acceleration is enabled with a single parameter on the Machine Setups > Speeds setup screen.



Caution!	To prevent machine damage:
	If the existing trapezoidal acceleration parameter value is set to the maximum acceleration capability of the cutting table, reduce this acceleration parameter by 50% before enabling the S-curve feature

The Acceleration Rate parameter determines the average rate of acceleration for both S-curve and trapezoidal motion. When you choose S-curve, a new parameter, Peak S Curve Acceleration, displays on the screen. This parameter is for information only, and displays the maximum acceleration used during the beginning of a cut and represents a rate of acceleration that is 50% greater than the average rate of acceleration.

Torch Height Disable

The Torch Height Disable feature activates the Torch Height Disable output and freezes, or disables, the automatic adjustment of the automated torch height control system to prevent diving or crashing of the torch in corners, holes, or other part features which require the cutting system to slow down to preserve cut quality or system mechanics.

The CNC activates the Torch Height Disable output based on the values of the following parameters:

- Torch Height Disable Speed
- Distance before THD Speed
- Distance After THD Speed

When any one of these conditions is met, the Torch Height Disable feature is implemented. The definition of a corner is based on the tangent angle in the Special Setups screen and the value of the Torch Height Disable Speed.

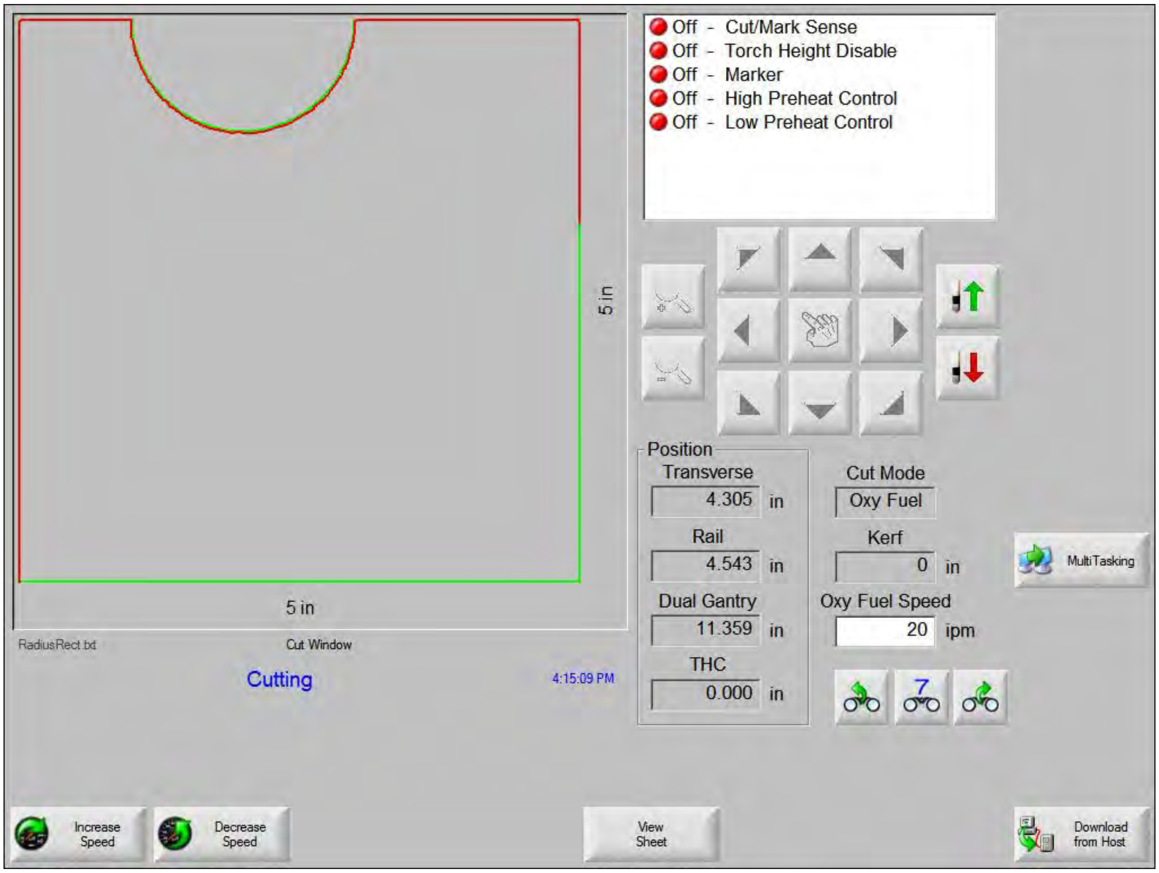
Tangent Angle: Specifies the degree of the tangent angle for motion control. Segments within a part that intersect at angles greater than the selected tangent angle will decelerate to zero or the minimum corner speed. Segments within a part that intersect at angles less than or equal to the selected tangent angle do not decelerate unless the next segment is a speed-limited arc.

Torch Height Disable Speed: Specifies the percentage of the program cut speed at which the CNC disables the height control. For example, when the cutting system is slowed to cut a part feature, such as a corner, the arc voltage increases which would cause the torch to lower into the workpiece. When the cutting system slows down to this percentage of the cut speed, the CNC disables THC voltage tracking.

Distance Before THD Speed: This distance is active whenever the program cut speed drops below the Torch Height Disable Speed, or segments intersect at an angle greater than the tangent angle. Specifies the distance before turning a corner at which the CNCs disables the torch height control.

Distance After THD Speed: Specifies the distance after turning a corner to enable voltage tracking for torch height control. Distance After THD Speed is active whenever the speed drops below the Torch Height Disable Speed, or segments intersect at an angle greater than the tangent angle.

In the following illustration, the Torch Height Disable output is activated based on the values of the setup parameters.



Parameter	Value
Torch Height Disable Speed Percentage value	90%
Distance Before THC Speed value	12.7 mm (0.5 inches)
Distance After THC Speed value	12.7 mm (0.5 inches)
Tangent Angle Value	20 degrees

For these values, the Torch Height Disable output is active 12.7 mm (0.5 inches) before and 2.7 mm (0.5 inches) after each 90 degree corner. This output is also activated as the torch accelerates away from the pierce and through the arc because the speed is below the Torch Height Disable Speed value. Finally, because the arc intersects at greater than the 20-degree tangent angle, the Torch Height Disable output is activated for 0.5 inches before and after the beginning of the arc, and is activated for 0.5 inches before and after the end of the arc.

Configuring Ports

Two serial ports are standard features on each CNC. Depending on the model, the serial ports are set for RS-232C or RS-422 communication. For more information about how to change the serial port setting between RS-232C and RS-422, see *Serial Ports* on page 241.

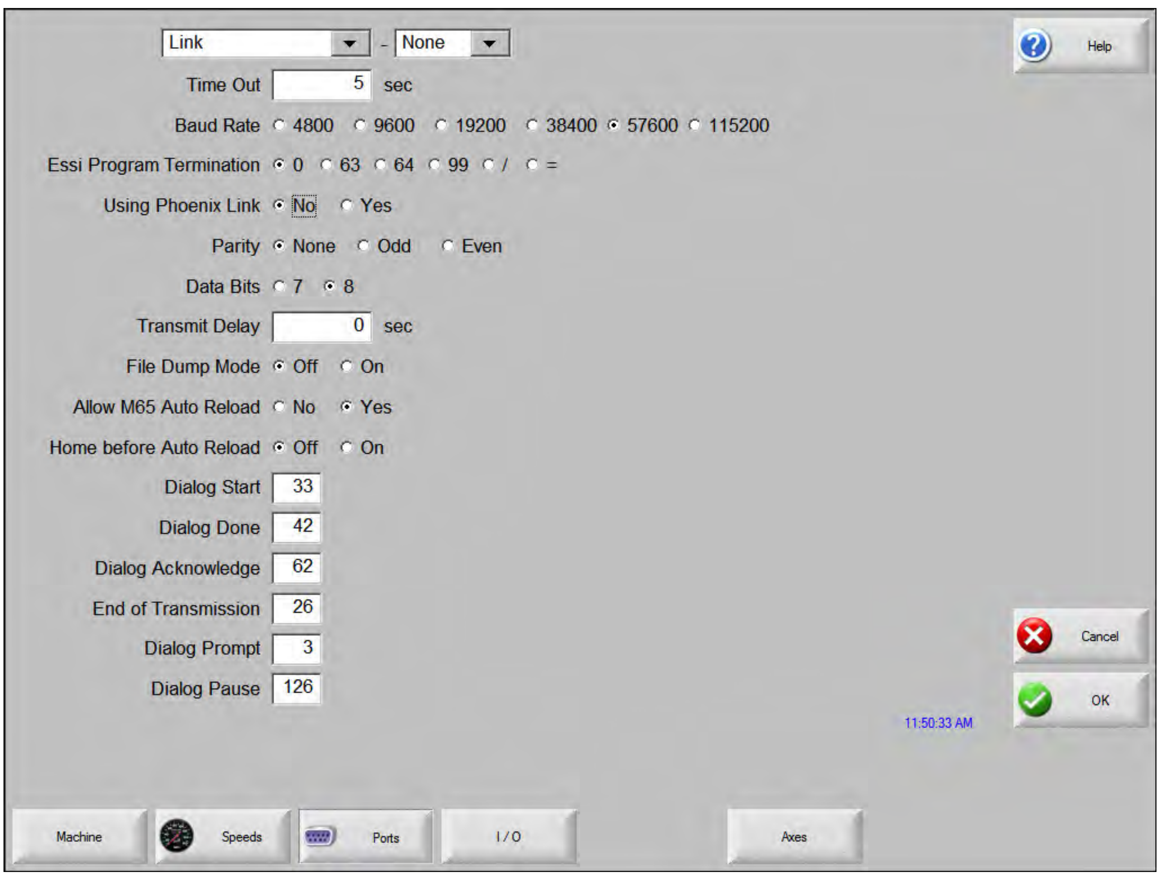
The serial ports are designed with opto-isolation and filtering to minimize EMI/RFI noise problems that are common in many cutting operations. Parameters are enabled and disabled depending on port feature selections.

The CNC can be configured to use the communication link provided on the CNC or the user can enter information to create a custom communication link on the Link screen to communicate with a host computer. Information on how to access the Phoenix software is provided later in this section.

A maximum of eight ports can be configured in software. Hardware provided with your CNC may support a lower number of communication ports.

The following parameters are configure the serial ports for your specific communications package and selected communication features.

Basic Configuration



Port Designation: Specifies which serial port is configured for link use with the following remote communication settings. Choose Link, Messaging, or a serial-capable cutting system such as Powermax, HPR, MAXPRO200, or HyPrecision™ waterjet.

Time Out: Specifies the amount of time the CNC waits before it displays a communications error if it is not able to establish a link with a remote device.

Baud Rate: Specifies the baud rate for the remote communication link. The available baud rates are shown. The CNC also incorporates a unique compression utility that allows the effective baud rate to be doubled. For example, for links configured to run at 9600 baud, the effective rate is 19.2K baud.

For serial messaging communication speeds from 1200 baud to 115200 baud are available. See *Serial Messaging* in the *Phoenix V9 Series Programmer's Reference* (806420) for more information.

ESSI Program Termination: This parameter allows the operator to tell the CNC which ESSI code (0, 63, 64 99, / or =) is used as machine stop.

Using Phoenix Link: Select Yes if you are using the Phoenix Communication link program that is supplied on the CNC. The CNC automatically enters the appropriate ASCII codes to the communication link dialog parameters.

Parity: Specifies the parity for the remote communication link.

Data Bits: Specifies the number of data bits the remote link uses. This value is 8 for the link software included with the CNC.

Transmit Delay: Specifies the amount of delay the CNC inserts between each character that is transmitted over the serial port. Some communication links require a small delay to avoid missing characters that have been sent to them.

This parameter should normally be set to 0 and should not exceed the value of the Time Out parameter.

The following parameters allow the user to create a custom communication link on the CNC by entering the ASCII code equivalent to the character or symbol used by the host computer for dialog start.

For example: The ASCII Decimal Code 33 is equal to the **!** symbol.

File Dump Mode: Select On to configure the link communication protocol to communicate with link programs that operate in File Dump Mode. This allows the CNC to accept part programs as a single uninterrupted stream of information, as with a tape-reader style link.

Allow M65 Auto Reload: Available when a generic link communication has been enabled and allows you to select whether the EIA M65 code is used as an auto-reload code or is ignored.

Home before Auto Reload: Automatically homes the machine before the next auto reload (EIA M65 code), either serial link or USB memory stick, is executed

Dialog Start: Enter the ASCII code equivalent to the character or symbol used by the host computer for dialog start.

Dialog Done: Enter the ASCII code equivalent to the character or symbol used by the host computer for dialog done.

Dialog Acknowledge: Enter the ASCII code equivalent to the character or symbol used by the host computer for dialog acknowledgement.

End of Transmission: Enter the ASCII code equivalent to the character or symbol used by the host computer for dialog acknowledge.

Dialog Prompt: Enter the ASCII code equivalent to the character or symbol used by the host computer for dialog prompt.

Dialog Pause: Enter the ASCII code equivalent to the character or symbol used by the host computer for dialog pause.

Rewind: This field is only enabled when Link File Dump Mode has been enabled. Enter the ASCII code equivalent to the character or symbol used by the host computer for tape rewind at the host tape drive system. Enter 0 to eliminate the rewind command.

The following additional parameters are available when Serial Messaging has been enabled. For more information see *Serial Messaging* in the *Phoenix V9 Series Programmer's Reference* (806420).

Flow control: Select None, Xon/Xoff or Hardware.

During Jog on Path: Select whether the CNC sends messages when jogging Forward or Backward on Path while at the Pause screen.

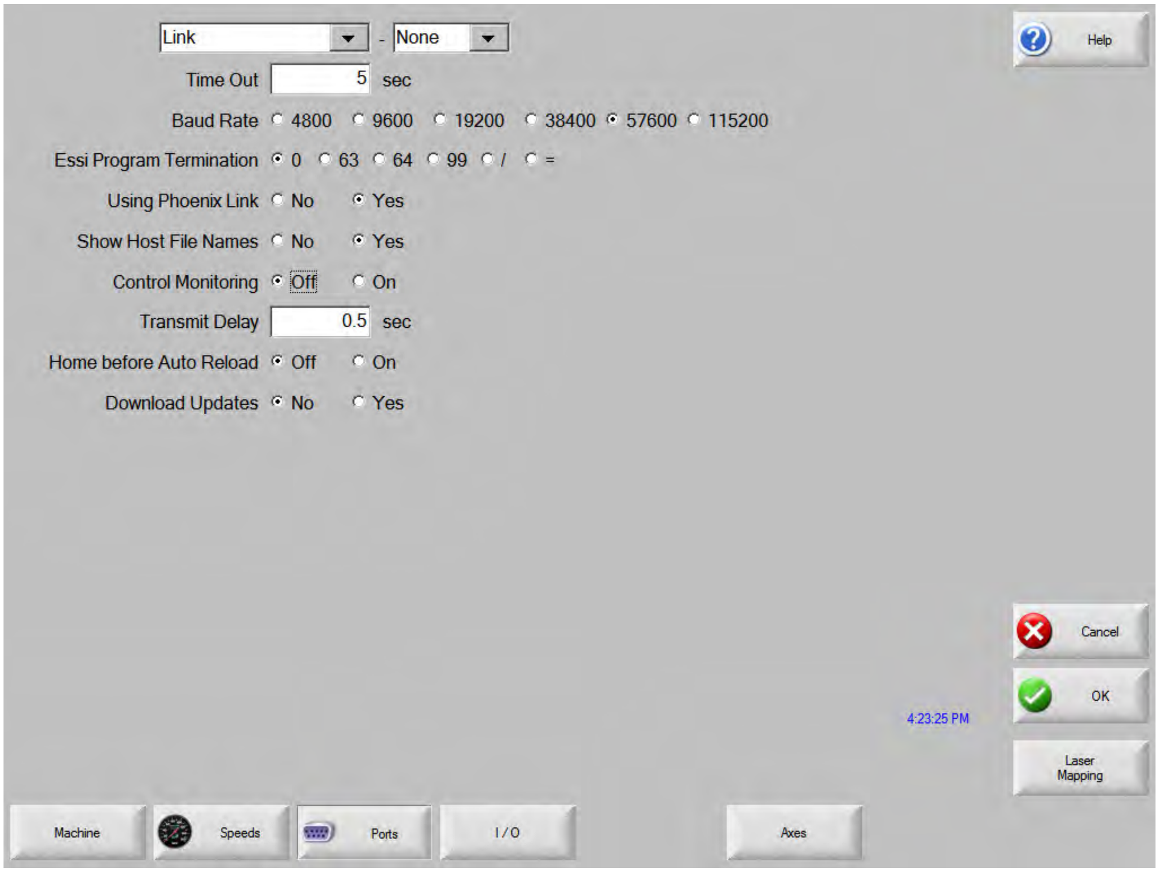
Notes:

- All messaging stops when the Stop Key has been pressed or when the Remote Pause input becomes active.
- The Message Type 21 is not executed for Backup on Path.

Retry on Time Out: Select the number of times the system automatically retries Message Type 22. After the system tries to send the message this many times it displays “Message Error” to indicate that the external device does not respond.

Time Out: This value is used for Message Type 22 if there is no time-out value used in the command string of the program code. For more information, refer to the Time Out/Transmit Delay parameters.

Phoenix Link Configuration



Port Designation: Specifies which serial port is configured for link use with the following remote communication settings. Choose Link, Messaging, or a serial-capable cutting system such as Powermax, HPR, MAXPRO200, or HyPrecision™ waterjet.

Time Out: Specifies the amount of time the CNC waits before it displays a communications error if it is not able to establish a link with a remote device.

Baud Rate: Specifies the baud rate for the remote communication link. The available baud rates are shown. The CNC also incorporates a unique compression utility that allows the effective baud rate to be doubled. For example, for links configured to run at 9600 baud, the effective rate is 19.2K baud.

For serial messaging communication speeds from 1200 baud to 115200 baud are available. See *Serial Messaging* in the *Phoenix V9 Series Programmer's Reference* (806420) for more information.

ESSI Program Termination: This parameter allows the operator to tell the CNC which ESSI code (0, 63, 64 99, / or =) is used as machine stop.

Using Phoenix Link: Select Yes if you are using the Phoenix Communication link program that is supplied on the CNC. The CNC automatically enters the appropriate ASCII codes to the communication link dialog parameters listed previously.

Show Host File Names: Select whether File Names are shown on the Download from Host screen. This is a time-saving feature for operators who have very large part folders at the host computer and know the exact file name of the part program they want to load.

The Using Phoenix Link parameter must be set to ON to enable this feature.

Control Monitoring: Allows the user to view CNC status at the host PC. Status for file name, position, cut mode and cut information is displayed. This feature is only available with the Phoenix Link communication software. For more information, see *Phoenix Link* on page 245.

Transmit Delay: Specifies the amount of delay the CNC inserts between each character that is transmitted over the serial port. Some communication links require a small delay to avoid missing characters that have been sent to them.

This parameter should normally be set to 0 and should not exceed timeout.

Auto Home before Auto Reload: Automatically homes the machine before the next auto reload (EIA M65 code), either serial link or USB memory stick, is executed

Download Updates: Allows the operator to download CNC software updates through the link communication to the host PC.

The “Using Phoenix Link” parameter must be set to YES. The new update file must be placed in the same folder location as the Link software. When the Update Software soft key is pressed on the Special Password screen, the CNC searches for the new software update through the link rather than the floppy drive.

Command THC Port Designation

WARNING

Configure the port for RS-422 operation before connecting to the Command THC.

The Command THC must be selected as a lifter in the Station Configuration screen.

Refer to the *Serial Ports* on page 241 for additional information on configuring the serial port for RS-422 communication.

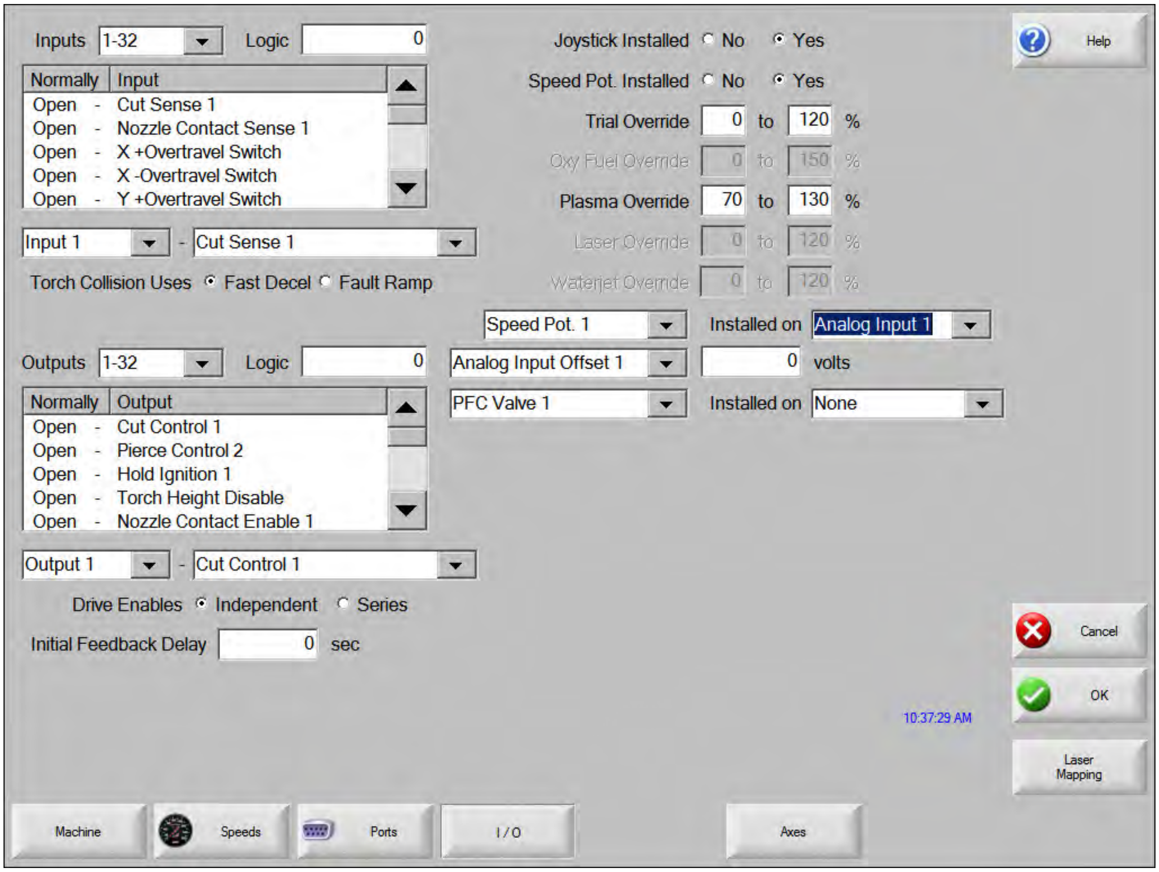
RS-422 Connections to Command THC with 25-pin D-type Connector

Command THC		CNC	
Signal Name	**DB-9 Pin**	**Signal Name**	**DB-9 Pin**
RxD -	1	TxD -	2
TxD -	2	RxD -	3
RxD +	20	TxD +	4
TxD +	21	RxD +	7

For improved noise immunity, cable shields for communication should be tied to ground. A ferrite bead with several wraps on the cable can also be used.

Configuring I/O

After you enter the Machine password and can view the Machine setup screen, press the I/O soft key to assign the inputs and outputs for the cutting system.



Inputs: When more than 64 I/O have been selected, the Input field becomes a drop-down box so you can configure I/O in the ranges of 1 to 32 and 33 to 64.

> When the optional USB front panel is used, the top 128 I/O are assigned to USB I/O to support the front panel.

Input Logic: Thirty two discrete digital inputs are standard on the CNC and are used as a mask to determine whether an input is treated as a normally open or closed contact. Entering a numerical value here sets all the Inputs to a predefined set of logic states for all of the inputs.

When Input Logic is set to zero (0), an active input is displayed with a green lamp in the input diagnostic screen.

Input Logic Selection Box: Determines whether the logic state for each input is normally open or closed. To select the logic state for an input, select the input and press the space key.

Input Selection: Defines which inputs are used by the CNC and their physical location on the CNC interface. All CNCs are shipped with the default selection of inputs and interface locations for I/O.

Torch Collision Uses: Determines the type of response that is used when the Torch Collision input is active. You can select a Fast Stop Decel value or the Fault Ramp time that has been selected for the individual axis.

Inputs Inverted: Defines which inputs are used by the CNC and their physical location on the CNC interface. All CNCs are shipped with the default selection of inputs and interface locations for I/O.

Analog Offset 1 – 12: The CNC constantly applies this offset to the incoming analog signal. The offset is subtracted from the incoming voltage and the CNC displays the voltage after this offset is subtracted.

Outputs: Thirty two discrete digital outputs are standard on the CNC and are used as a mask to determine whether the output is to be treated as a normally open or closed contact.

Outputs Logic: The Logic Selection Box allows the user to select the Logic state for each input to be either normally open or normally closed. The logic state for the inputs can be switched by selecting the output and pressing the space key.

Output Logic Selection Box: The Output Logic Selection box feature defines which inputs will be used by the CNC, and their physical location on the CNC interface. All CNCs are shipped with the default selection of outputs.

Drive Enables: Determines how the CNC responds to drive faults. If you select Series, all axes must complete their individual fault ramp times before the drives are disabled. If you select Independent, each drive is disabled as soon as its axis fault ramp time is finished.

Initial Feedback Delay: Allows the initial feedback from the drive system to be delayed after initial power up. This allows the drive system to become fully enabled before it enables the position loop.

Fume Extraction Delay: Enter the number of seconds after the end of a part program that the fume exhaust remains on.

Digital Input Definitions

Arc Error Counter 1 – 8: Used as part of consumable data tracking, this input triggers a counter on the Change Consumable screen. A Plasma Enable Input must be mapped to enable this input.

Cut/Mark Sense 1 & 2: Verifies that the plasma torch or marker tool has established the arc for the selected process. This indicates to the CNC that motion can begin. (Type "V" CNCs.)

Cut Sense: Indicates that the plasma arc has transferred to the work piece. If the Arc On Feedback setup parameter is ON, machine motion begins at the activation of this input. (Type "M" and "P" CNCs.)

Drive Disabled: Causes the CNC to stop all motion and generate a fault message. Position information is lost.

Enable Oxy Fuel 1 – 12: Used for tracking consumable data for the specified oxy torch. This input is now supported by the Station Select input.

Enable Plasma 1 – 8: Used for tracking consumable data for the specified plasma torch. This input is supported by the Station Select input.

Fast Stop: Used as an urgent stop input command to the CNC. When the Fast Stop input becomes active, the CNC decelerates motion using the Fast Deceleration Rate on the Speeds screen and forces the operator to the Pause screen. One second after the input becomes active, the Drive Enable output from the CNC turns off and disables motion. Position information and I/O points are maintained while the input is active. This allows the operator to recover the last position after the input has been cleared.

Fume Extraction Sense: If selected, this input confirms that an external Fume Extraction system on the cutting table is operational before beginning the cut. An option to override is displayed if the input has not been satisfied at the time of the cut.

Gas Control Error: Used as part of the HD3070 Automatic Gas Console communications to indicate an error from the Auto Gas Console.

Gas Control Read Comp.: Used as part of the HD3070 Automatic Gas Console communication to confirm that communication is complete.

Joystick (Direction): Command manual motion (up, down, left, and right) when using an external joystick.

Limit Switch: Indicates that the machine has traveled to its full positive travel of an axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion and generates a fault message. Motion is not re-enabled until the switch is deactivated. (Type "M" configuration only.)

Marker Select 1 & 2: Select which Marker process logic is used by the CNC. These inputs are generally run by external switches. (Type "V" CNCs.)

Nozzle Contact Sense: Used during Sensor THC IHS to detect the location of the cut surface. This input is returned to the CNC through the external voltage divider card.

Pierce Complete: Used for the laser cut process to confirm that the pierce is complete.

Plasma Select 1 & 2: Select which Plasma process logic is used by the CNC. These inputs are generally run by external switches. (Type "V" CNCs.)

Power Supply OK: Used for the optional Automated Process Controller (APC). It confirms a ready condition from the plasma power supply before beginning the cut process.

Preheat Sense: Indicates that the preheat input from the torch is active.

Program Inhibit: Forces the CNC to command the motion output to a zero (0) speed. This is generally used as a dwell to pause motion during a tool change or as a pause from a PLC interface.

Raise /Lower 4070 1 – 2: Used as a manual raise/lower command to the selected HD4070 integrated THC through the CNC serial power supply communication link. This input is exclusive to the HD4070 multiple power supply configuration.

Raise/Lower Torch: Operates multiple Sensor THCs when multiple Sensor THCs are commanded by the CNC as separate axes. The Torch Raise and Torch Lower commands can be issued with soft keys or through these external inputs to the CNC. (Type "V" CNCs.)

Ready to Fire PS 1 – 4: Ensures that all HD4070 power supplies have completed each individual Initial Height Sense and are ready for use. The torch ignition is held off by the Hold Ignition Input to the power supply from the CNC. This input is exclusive to the HD4070 multiple power supply configuration.

Remote Pause: Stops all CNC motion and displays the Pause screen. No motion is allowed until this input is deactivated.

Remote Start: Begins the selected program cycle as if the Start button on the CNC had been pressed.

Rotate +/-: Used for manual jog commands for the rotate axis.

Rotate Home Switch: Indicates that the machine has traveled to its rotate axis home position. When the rotate axis is homed, it moves in the specified home direction at the Fast Home Speed until the input is activated. The rotate axis decelerates to a stop, moves in the opposite direction at the Slow Home Speed until the switch is deactivated.

Sensor THC Enable: Indicates which Sensor THCs are active when multiple Sensor THCs are commanded by the CNC as separate axes. This input is now supported by the Station Select input.

Spare: Activated through the part program. If a spare input is located in the part program, the CNC pauses the process until the input state is detected. The spare inputs can be implemented with specific EIA "O" and "M" codes that indicate the input number and function. For more information, refer to the *Programmer's Reference*.

Speed Limit Input: Limits the machine speed for safety during machine testing and maintenance. When this input is active, motion is limited to the user-defined Limited Machine Speed selected in the password protected Speeds screen

Station Select Input: Indicates which Sensor THC is active when multiple Sensor THCs are commanded by the CNC as separate axes. Also tracks consumable data for the specified plasma or oxy torch. Stations 1 – 8 are designated for plasma; stations 9 – 20 are designated for oxyfuel.

Test Lifter: Performs a test IHS function with a Sensor THC.

THC Automatic: Used as an external input to switch the Sensor THC between automatic and manual operation.

THC Cut Sense: Verifies that the plasma torch or marker tool has established the arc for the selected Sensor THC and indicates to the CNC that motion can begin.

THC Homing to Limit: The THC Enable Input is used during Sensor THC Homing as the Current Limit input if the Home to Hard Stop Current Limited option is selected.

THC Homing to Switch: The Nozzle Contact Sense Input is used during Sensor THC Homing as the Home Switch input, if the Home to Switch option is selected.

Tilt +/-: Used for manual jog commands for the tilt axis.

Tilt Home Feature: Uses the Tilt Overtravel Switch input to indicate that the machine has traveled to its tilt axis home position. When the Tilt axis is homed, it moves in the specified home direction at the Fast Home Speed until the input is activated. The tilt axis decelerates to a stop and moves in the opposite direction at the Slow Home Speed until the switch is deactivated. See also Tilt Overtravel Switch.

Tilt +/- Overtravel Switch: Indicates that the machine has traveled to its full positive travel in the Tilt axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion, generates a fault message and only allows manual motion in the Tilt axis opposite direction. The Tilt Overtravel Input is also used for homing the tilt axis.

Tool Cycle Active: Activated from a PLC to pause motion until the input turns off. Used when a PLC is controlling other tool cycles such as drilling and tapping. After the input turns off, the program motion resumes.

Torch Collision: Used on torch systems with breakaway mounts. If a torch makes contact with the workpiece or an obstacle that causes the breakaway mount to release, an input for the mount is sent to the CNC indicating that a torch collision has occurred. The Pause screen displays. While the input is active, the Cut output is turned off and manual motion is enabled, allowing the operator to raise, lower, and move the torch position to clear the fault.

Position information, motion command, and I/O points are maintained and allow the operator to return to the cut path and resume cutting. This feature uses the Fast Stop Deceleration rate. Manual raise inputs can be used.

Torch Down Sense: Indicates that the torch is in the full down position.

Torch Up Sense: Indicates that the torch is in the full up position.

Tracer on Path: Used with the Teach Trace feature and indicates that the tracing system has detected the template line.

X Home Switch: Indicates that the machine has traveled to its X axis home position. If the X axis Use Home Limit Switch parameter is set to YES and a homing function is performed, the X axis moves in the specified home direction at the fast home speed until the input is activated. The X axis then decelerates to a stop and moves in the opposite direction at the slow home speed until the switch is deactivated. After the switch is deactivated, the X axis decelerates to a stop or, if the Use Marker Pulse parameter is set to YES, continues until the encoder marker pulse is detected.

X Overtravel Switch: Indicates that the machine has traveled to its full positive travel of an X axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion and generates a fault message. Motion is not re-enabled until the switch is deactivated.

X +Overtravel Switch: Indicates that the machine has traveled to its full positive travel on the X axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion, generates a fault message and only allows manual motion in the X axis negative direction.

X –Overtravel Switch: Indicates that the machine has traveled to its full negative travel on the X axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion, generates a fault message and only allows manual motion in the X axis positive direction.

Y Home Switch: Indicates that the machine has traveled to its Y axis home position. If the Y axis Use Home Limit Switch parameter is set to Yes and a homing function is performed, the Y axis moves in the specified home direction at the fast home speed until the input is activated. The Y axis then decelerates to a stop and moves in the opposite direction at the slow home speed until the switch is deactivated. After the switch is deactivated, the Y axis decelerates to a stop or, if the Use Marker Pulse parameter is set to Yes, continues until the encoder marker pulse is detected.

Y Overtravel Switch: Indicates that the machine has traveled to its full positive travel of a Y axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion and generates a fault message. Motion is not re-enabled until the switch is deactivated.

Y +Overtravel Switch: Indicates that the machine has traveled to its full positive travel on the Y axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion, generates a fault message, and only allows manual motion in the Y axis negative direction.

Y –Overtravel Switch: Indicates that the machine has traveled to its full negative travel on the Y axis. If hardware overtravels are enabled and this input is activated, the CNC stops all motion, generates a fault message, and only allows manual motion in the Y axis positive direction.

Z Home Switch: Indicates that the machine has traveled to its Z axis home position. If the Z axis Use Home Limit Switch parameter is set to Yes and a homing function is performed, the Z axis moves in the specified home direction at the fast home speed until the input is activated. The Z axis then decelerates to a stop and moves in the opposite direction at the slow home speed until the switch is deactivated. After the switch is deactivated, the Z axis decelerates to a stop or, if the Use Marker Pulse parameter is set to Yes, continues until the encoder marker pulse is detected.

Digital Output Definitions

Abrasive Control: Used exclusively with waterjet cut logic to activate the abrasive control, adding abrasive to the cut stream while cutting, or for charging the abrasive prior to cutting.

Assist Gas: Used exclusively with the laser cut logic to activate the assist gas.

Auto OHC 1 – 20: Instructs the Sensor OHC to operate the specified station in automatic mode

Bleed Off Gas: Used by the oxyfuel cut logic to send an output to the oxy torch to purge gases at the end of a cut. This output is on for the duration of the Bleed Off time selected in setups.

Change Consumable: Activates when a consumable data value on the Change Consumable screen has reached its preset maximum. Generally, this output is tied to an external indicator light or audible alarm to prompt the system operator to change the appropriate consumable. The affected consumable data must be reset on the Change Consumable screen to clear the output.

CNC/Tracer: Controls a CNC relay. The CNC relay determines whether the CNC, tracing system, or joystick is commanding the amplifiers. This output is always active unless the CNC is off, in Teach Trace mode, or is disabled.

Current Set BCD: Sets the current at the plasma power supply or marker supply. These outputs are tied to the BCD inputs. Combinations of the current values are used to attain the desired current set point.

Cut Control: Enables the cutting oxygen in oxyfuel mode or starts the plasma system in plasma mode. This output can also be used to activate a marking device.

Digital Beam: Used exclusively with the laser cut logic to activate the digital beam.

Drill Cycle: This output is activated with an M93 code in the part program. It activates an input on a PLC that is controlling tool cycles.

Drive Enable: Used to enable the drives during normal functions and to disable the drives during fault conditions.

Fume Extraction Control: When you assign this output, a Fume Extraction Delay timer appears on the I/O screen. The timer allows the fume extraction equipment to run for a period of time at the end of the part program when the CNC displays the Main screen. This output turns on at the beginning of a cut, at the start of a rip cut, or when resuming a cutting after pausing the program. The output turns off when the timer expires either at the end of the part program or when the operator pauses the part program.

Gas Control Write: Used as part of the HD3070 automatic gas console communications to configure the unit for operation.

Gas Flow Set 1 – 100: Used as part of the HD3070 automatic gas console communications (BCD) to set gas flow rates.

Hold Ignition: Is part of the Sensor THC operation to hold ignition of the external Plasma Supply. This output can be tied back to the CNC through the external voltage divider card or direct to the power supply depending on configuration.

Ignition Control: Turns on the igniters in oxyfuel mode or ignites the plasma system in plasma mode.

Key Press Indicator: Activated whenever a key is pressed on the CNC.

Laser Start: Used exclusively with the laser cut logic to activate the laser system.

Lifter Low Speed: Operates with the manual Raise/Lower inputs from the operator console to move an Oxyfuel lifter at a slow speed for manual jogging of the torch. This output is active during the period set by the Lifter Low Speed timer on the Oxyfuel Process screen.

Low Preheat Control: Enables the low preheat gas when cutting in oxyfuel mode. It is activated for the "Low Preheat Time" when initiating a cut.

Marker Control: Activates the marker tool during the marking process. Traditionally, the marker tool is activated through the combination of Marker Enable and Cut Control outputs. All the noted options are available and can be configured on the Marker Setup screen for the desired process timing and operational results.

Marker / Marker Enable: Activates an external marking device. It can only be activated by the appropriate "M" code in the part program. For more information, refer to the *Programmer's Reference*.

Marker Enables: Used for external logic. The appropriate output(s) is on during marking, based on the Marker Select inputs.

Motion Indicator: Activated whenever the CNC is commanding machine motion.

N_2 Select: Used by the Optional APC to enable the use of nitrogen while cutting.

Nozzle Contact Enable: Active during Sensor THC IHS. This input is tied back to the CNC through the external voltage divider card. This output can also be used to switch an external drive system to low output mode (if equipped) during IHS for stall force workpiece sensing.

Peck Drill Cycle: This output is activated with an M94 code in the part program. It activates an input on a PLC that is controlling a tool cycle.

Pierce Control: Used by the plasma and oxyfuel cut logic to send an output to the torch during the pierce. This output is on for the duration of the Pierce Time selected in setups.

Plasma Enables: Used for external logic. The appropriate output(s) is on during plasma cutting, based on the Plasma Select inputs.

Plasma Select: Activated whenever the CNC is in the Plasma cutting mode.

Preheat Control: Enables the high preheat gas when cutting in oxyfuel mode. It is activated for the "High Preheat Time" when initiating a cut.

Program Running: Active any time the CNC is operating within a part program.

Reduce Current: Used with the HD4070 to switch the plasma to low current mode.

Remote Air Plasma: Used as part of the HD3070 automatic gas console communications to signal the Air Plasma output.

Remote H35/ N_2 Plasma: Used as part of the HD3070 automatic gas console communications to signal the H35/N_2 output.

Remote O2 Plasma: Used as part of the HD3070 automatic gas console communications to signal the O2 Output.

Remote Test Operate: Used as part of the HD3070 automatic gas console communications to initiate the remote test.

Remote Test Preflow: Used as part of the HD3070 automatic gas console communications to test operation of preflow gases.

Shutter Open: Used exclusively with the laser cut logic to open the laser shutter for use.

Spare: Activated through the part program. If a spare output is located in the part program, the CNC turns on the output as directed. The Spare outputs can be implemented with specific EIA "W" and "M" codes that indicate the output number and function. For more information, refer to the *Programmer's Reference*.

Staged Pierce 1 – 4: Control oxygen regulators to increase the flow of oxygen to the flame in the torch and speed piercing. The timing of each output is determined by the values in the Staged Pierce Time 1-3 parameters on the Oxyfuel Process screen.

Station Clamp 1 – 19: Used as part of the Automated Torch Spacing feature. The Station Clamp is used to clamp the selected torch station to the transverse axis for standard cutting.

Station Enable 1 – 20: Activates any function specific to a torch station and are controlled through M37T and M38T codes within a part program. These codes generally enable a torch station for use. Usually, stations 1-8 are configured with plasma and stations 9-20 are configured with oxyfuel or other types of fuel. However, plasma can only be configured on stations 1-8.

Station Lock 1 – 19: Is part of the Automated Torch Spacing feature. The station lock locks the unused torch station to the gantry or beam when the torch is not in use.

Station Mirror 1 – 19: Used as part of the Automated Torch Spacing feature. The station mirror is used to clamp the selected torch station to the transverse axis for mirrored cutting.

Tap Cycle: This output is activated with an M95 code in the part program. It activates an input on a PLC that is controlling a tool cycle.

Tool Change: This output is activated with an M96 code in the part program. It activates an input on a PLC that is controlling a tool change.

Torch Down: Lowers the cutting torch. It is activated for the "Primary Torch Down Time" and the "Pierce Torch Down Time" in oxyfuel mode. It is activated for the "Torch Down Time" in plasma mode.

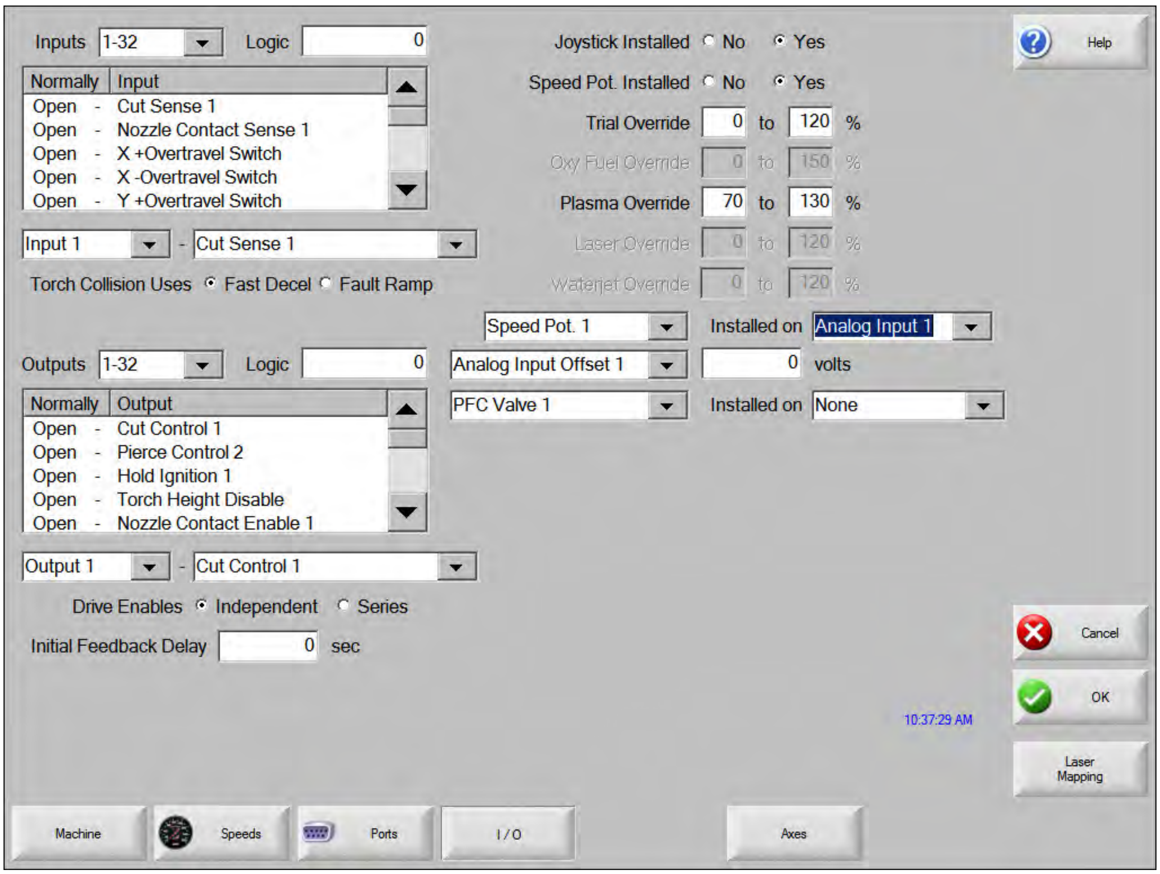
Torch Height Disable/Dual Grid Control: Disables the automatic torch height control in plasma mode. It is activated whenever the current axis position is within the "Plasma Distance To Corner" or the "Plasma Distance From Corner" parameters while cutting a part. It is also active whenever the current cutting speed drops below the result of (Programmed Cut Speed x (Plasma High/Lo Speed/100)).This output also reduces the plasma current in a switchable current plasma system when the machine speed is less than the Plasma Hi/Lo speed percentage.

Torch Up: Raises the cutting torch. It is activated for the "Primary Torch Up Time" and the "Pierce Torch Up Time" in oxyfuel mode. It is activated for the "Torch Up Time" in plasma mode.

Turn ON / Off Supply: Used by the Optional APC to turn on or off the plasma supply through the CNC.

Valve Select 1 – 8: Used as part of the HD3070 automatic gas console communications to select the appropriate valves for operation.

THC I/O

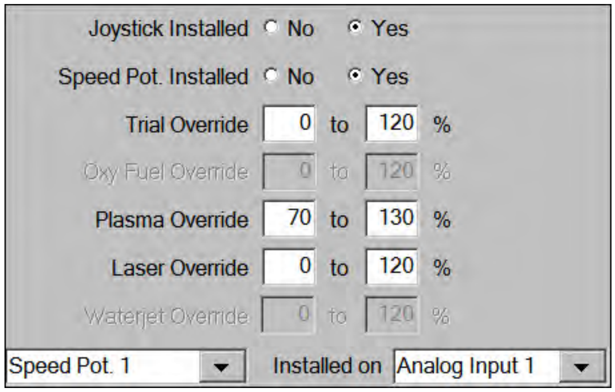


Analog Offset 1 – 12: The Analog Offsets 1 through 12 are used to correct an imbalance or to make the incoming analog voltage to the analog input card zero (0). Analog inputs are assigned to the THC Axis and the Analog Offsets for the appropriate THC Analog input may be adjusted here. To calibrate the Analog Inputs, place a jumper between the two pins specific to the input and view the incoming voltage at the Diagnostics Analog I/O screen. The incoming voltage should be zero. If any voltage is displayed at the diagnostic screen, an Offset Voltage equal to the incoming voltage being read can be entered here to make the incoming voltage equal to zero (0).

Speed Pot and Joystick Setup

Through the use of single-ended inputs to an optional analog input card, the CNC can be configured to support an external joystick or two speed potentiometers (speed pots).

These features are not available for all CNC types and optional CNC hardware may be required.



- The external joystick is used as a directional command signal when it is in manual motion.
- The optional speed potentiometer can be used to adjust the current motion speed.
- These features must first be enabled at the password-protected I/O screen.
- Speed pot 1 controls Plasma 2 process speeds.
- Speed pot 2 controls manual jog speeds with a range of 0 to 100.

Joystick Installed: Select Yes to enable the optional external joystick. This parameter is not available for all CNC types and optional CNC hardware is required.

Speed Pot Installed: Select Yes to enable the optional external speed potentiometer. This parameter is not available for all CNC types and optional CNC hardware is required.

Plasma Override%: Sets the minimum and maximum range for adjustment of the plasma speed potentiometer based on a percentage of the set cut speed. The speed potentiometer feature must be enabled to use this parameter.

Oxy Fuel Override%: Sets the minimum and maximum range for adjustment of the oxyfuel speed potentiometer based on a percentage of the set cut speed. The speed potentiometer feature must be enabled.

Trial Override%: Sets the minimum and maximum range for adjustment of the trial speed potentiometer based on a percentage of the set trial speed. The speed potentiometer feature must be enabled.

Speed Pot Installed On: Assigns speed potentiometer inputs to the desired Analog Input if a dedicated speed potentiometer input is not available on the analog input card.

Section 3

Torch Height Control Axis Setup

Hypertherm CNCs support an integrated Z axis called the Sensor THC that provides linear motion and automatic voltage tracking. The built-in electronics in the CNC can be used with the Sensor THC drive and lifter provided by Hypertherm (ASSY-0177) or with drives and lifters supplied by table manufacturers and system integrators.

The settings provided in the following sections apply to the Sensor THC lifter and drive supplied by Hypertherm (ASSY-0177). The settings can often be used for lifters and drives from other manufacturers.

This section describes the axis setup for the Sensor™ THC, Sensor™ Ti (page 116), and the ArcGlide® THC (page 118).

Sensor THC axis

Setting up the Sensor THC on the CNC requires these steps:

1. Choose the number of Sensor THCs installed on the cutting system, assign each THC to an axis and select an analog input for the arc voltage feedback
2. Set speeds and acceleration values for the Sensor THC axis.
3. Select the Sensor THC as a lifter on a station.
4. On the Sensor THC axis setup screen enter tuning values, encoder information, servo error tolerance, stall force tolerance, and other information about the THC axis.

Initial height sense

Phoenix software version 9.73.0 brought several enhancements to the Sensor THC. One of these enhancements changed the way the Sensor THC performs a first IHS. The first IHS detects the height of the workpiece so that the CNC can calculate the torch-to-work distance. The CNC uses the torch-to-work distance for each subsequent IHS which it can perform using much faster speeds since the height of the workpiece is known. The first IHS for the Sensor THC follows these steps:

1. Home the THC to top of slide.
2. Choose the Test Lifter soft key on the Main screen to perform the first IHS.

a. The THC moves at the maximum speed for 1/10th of the slide length then changes to Fast IHS speed.

b. At the IHS Start Height, the CNC begins monitoring the Nozzle Contact Sense input and the THC axis following error. The THC continues to move at the Fast IHS speed until the torch tip touches the workpiece.

c. The CNC records the height of the workpiece, then the THC retracts to the transfer height at the Slow IHS speed.

Any other IHS in a cutting job uses faster speeds: Maximum THC Speed to the Start IHS Height, then Fast IHS speed to the plate, then Slow IHS Speed to the Transfer Height.

Entering the slide length

The slide length is the amount of travel of the lifter. The CNC uses the slide length when changing speeds during the IHS. The CNC subtracts 5.08 mm (0.2 inches) from the slide length to ensure the lifter does not contact a hard stop when homing. Use the following test to determine the slide length.

Notes:

- Before running the test, remove the torch from the lifter to prevent any damage to the torch when you jog the lifter.
- This test requires that you jog the lifter to the hard stop. Be sure to change the THC Jog Speed to the Slow IHS speed, as stated in the following steps, to avoid any damage to the lifter when it contacts the hard stop.

1. Choose Setups > Watch, choose Position for the lower location, and choose THC Position.
2. Choose Machine Setups > Axes > Sensor THC Axis. Enter a slide length value that is larger than then the actual slide length.
3. From the Main screen, choose Manual (press F11 on the keyboard or choose the Manual icon) > Home Axes > THC. Choose OK to exit the screen.
4. Choose Manual Options.
5. Choose Slow IHS for the THC Jog Speed.
6. Lower the lifter until it reaches its full travel distance at the bottom of the slide.
7. Note the THC position. This value equals the total slide length.
8. Enter this value into the Sensor THC Axis screen.

Assigning the Sensor THC to an axis and selecting an analog input

1. Choose Setups > Password > Machine Setups.
2. Choose the number of Sensor THCs installed.
3. For each Sensor THC:

   a. Choose from THC1, THC2, and so on, to identify the torch height control.

   b. Choose the axis for the THC.

   The THC can be assigned to Axis 3 and higher. The CNC uses Axes 1 and 2 for Transverse and Rail. Axis 3 becomes Dual Gantry if one is used. If not, then the Sensor THC can be assigned to Axis 3.

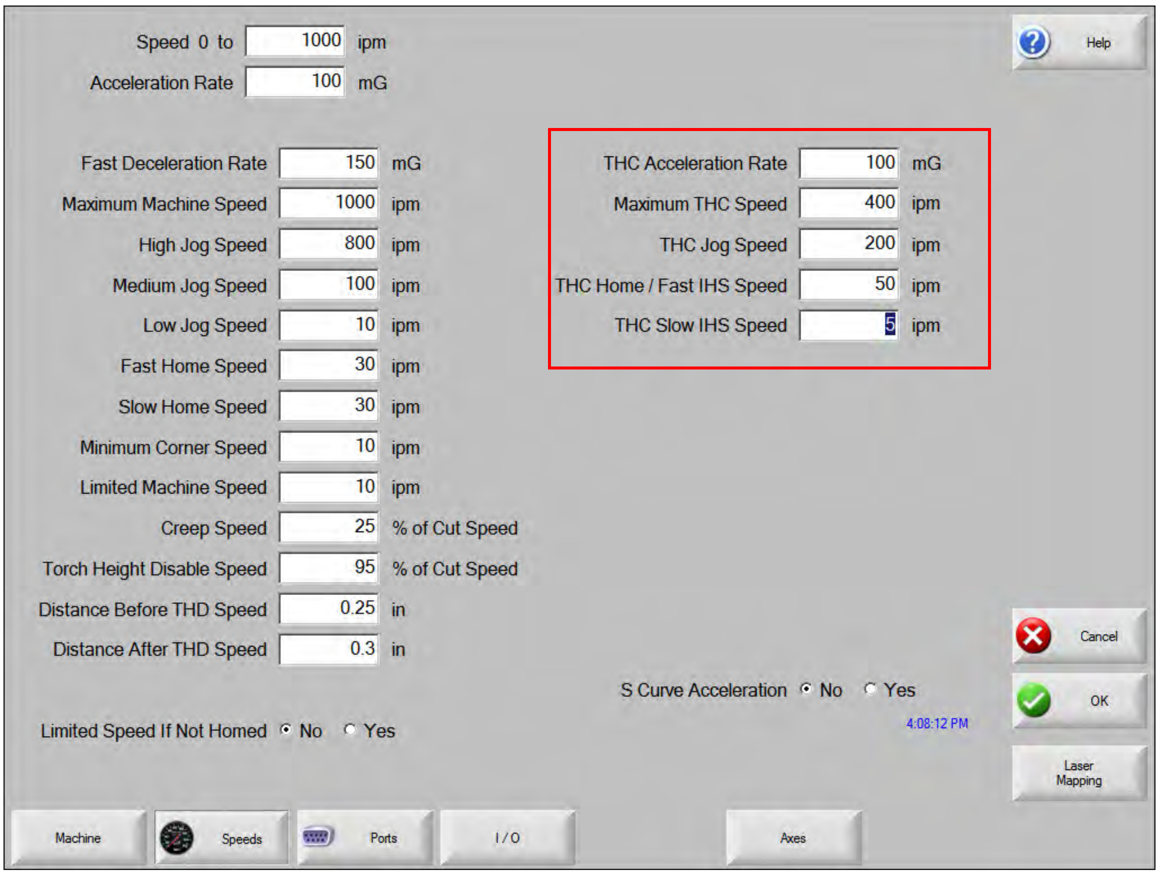
c. Choose the analog input to use for arc voltage feedback:

- EDGE Pro: Analog 5 for THC1, Analog 6 for THC2
- MicroEDGE Pro: Analog 5 for THC1, Analog 6 for THC2
- EDGE Pro Ti: Analog 3 for THC1

Note: If you have a SERCOS model of the EDGE Pro or MicroEDGE Pro, refer to the SERCOS Setup section in this manual for information on setting up the Sensor THC.

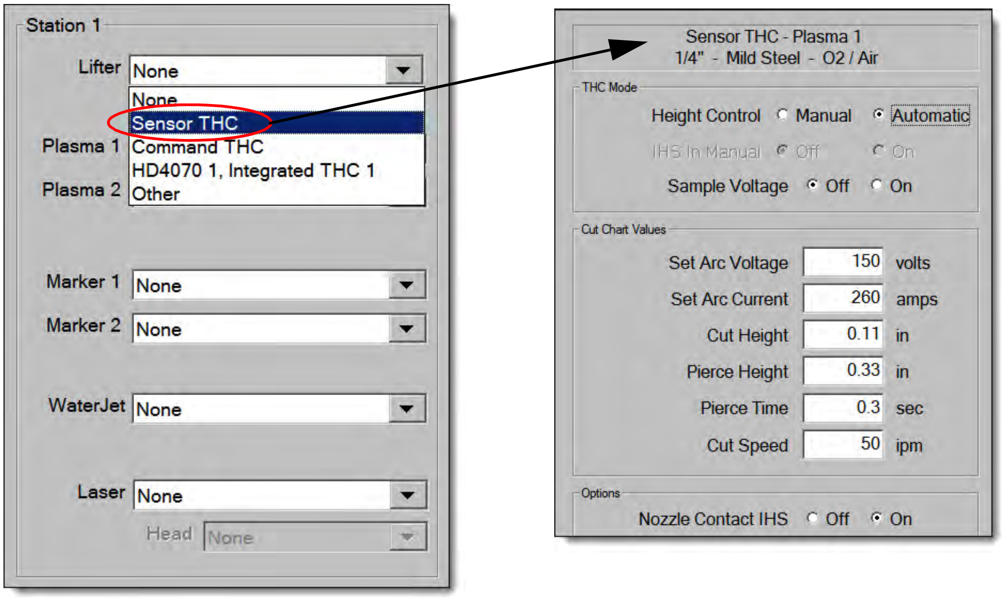
Setting speeds and acceleration

After you select the Sensor THC in the Machine screen, set the speed and acceleration settings on the Machine Setups > Speeds screen. The following screen shows the recommended speeds for the Hypertherm lifter and drive (ASSY-0177).



Assigning the Sensor THC as a lifter

To activate the Sensor THC in the Process screen, you must select it as a lifter for a station. Choose Setups > Password > Station Configuration and select the Sensor THC from the Lifter list.



Setting servo error tolerance and stall force tolerance

About following error

Following error is the difference between the commanded position and the actual position of the axis. In every cutting system, some following error is a normal part of operation. While you may want following error to be as low as possible, it will never be zero. A high following error, or one that keeps growing, can indicate an improperly controlled or mechanically binding axis.

In a well-tuned cutting system, following error reaches a steady state when the axis is moving at a given speed. Usually, the slower the speed, the lower the following error. If you see a following error that keeps increasing rather than leveling off at a steady state, the acceleration may be set too high. The recommended acceleration of the Sensor THC axis is 100mG (set in the Machine Setups > Speeds screen).

Servo error tolerance and stall force tolerance each require you to determine the steady-state following error at a given speed. Use the tests described in the following sections to determine the steady-state following error for that speed, then use the multiplier given to arrive at the optimum tolerance value. Servo error tolerance and stall force tolerance are both entered on the Sensor THC Axis screen (page 108).

Servo Error Tolerance

Servo Error Tolerance is the maximum amount of servo following error. If the cutting system accumulates following error that exceeds the Servo Error Tolerance them motion stops and the CNC displays an position error. Set the Servo Error Tolerance to three times the steady-state following error when traveling at the maximum speed. The maximum speed represents the most demanding moves for the lifter and should result in the highest following error.

Use the following test to determine the steady-state following error when traveling at the maximum THC speed.

Note: Before running the test, remove the torch from the lifter to prevent any damage to the torch when you jog the lifter up and down.

1. Choose Setups > Watch, then choose Following Error for the THC axis.
2. In Machine Setups > Speeds, set the THC Home/Fast IHS Speed to the same value as Maximum THC Speed.
3. Choose Manual (press F11 on the keyboard, or choose the Manual icon) > Manual Options, then select Fast IHS Speed for THC Jog Speed.
4. Jog the lifter from the top to the bottom of the slide several times and record the highest following error shown during the test.

   Note: If the following error continues to increase instead of reaching a steady-state, the acceleration setting for the THC axis may be set too high. On the Machine Setups > Speeds screen, enter a lower value for THC Acceleration Rate. Repeat step 4 and observe the following error again. If the following error does not level off to a steady state, decrease the THC Acceleration Rate again.
5. Multiply the following error by 3 and enter it as the value for Servo Error Tolerance in the Machine Setups > Sensor THC Axis screen.
6. Return to the Speeds screen and reset THC Home/Fast IHS Speed to its original value (For Sensor THC, 1270 mm/min or 50 inches/min).

Stall Force Tolerance

Stall force is a method of detecting the workpiece during initial height sense. Use stall force when cutting underwater or in other conditions in which ohmic contact is not feasible. Stall force is a measure of following error that accumulates when the torch tip contacts the workpiece and the lifter continues downward movement but is stopped by the workpiece. When the following error exceeds the stall force tolerance value, the CNC knows the position of the workpiece and retracts the torch.

For best operation, use a drive that provides a current limit input, such as the Yaskawa drive provided with Hypertherm's Sensor THC. Connect the THC Torque Limit output from the CNC to the current limit inputs of the drive. Using the software provided by the drive manufacturer, identify the current limit parameters and set them to reduce current when the THC Torque Limit output is on. The Yaskawa Sigma II drive has these parameters:

Code	Description	Value	Comment
CLMIF	Forward External Torque Limit	20%	Downward motion during IHS
CLMIR	Reverse External Torque Limit	200%	Upward motion during IHS and homing.

For drives from other manufacturers, set the torque limit to 10% of maximum for downward (or forward) motion, and 20% of maximum for upward (or reverse) motion

Use the following test to determine the steady-state following error for travel at Fast IHS Speed.

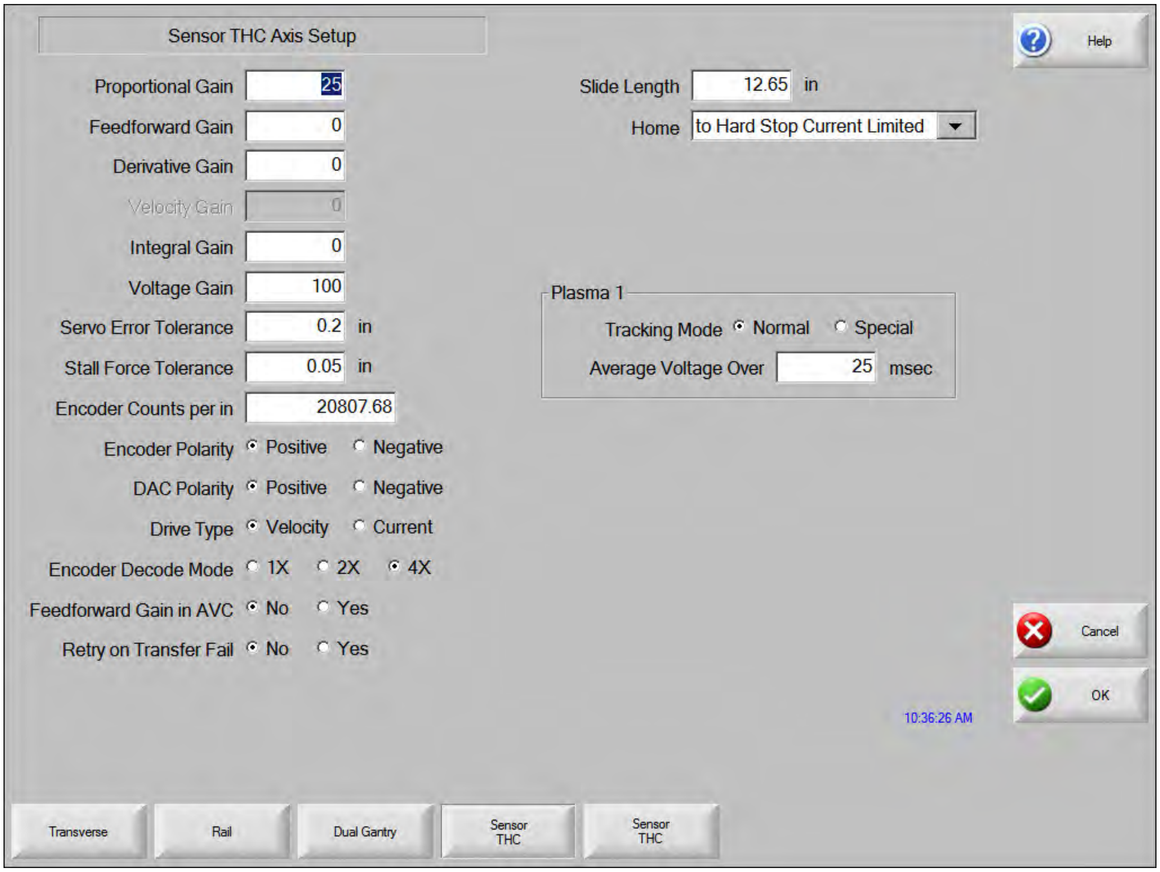
Note: Before running the test, remove the torch from the lifter to prevent any damage to the torch when you jog the lifter up and down.

1. Choose Setups > Watch and choose Following Error for the THC axis.
2. Choose Manual (press F11 on the keyboard, or choose the Manual icon) > Manual Options, then select Fast IHS Speed for THC Jog Speed.
3. Jog the lifter from the top to the bottom of the slide several times and record the highest following error shown during the test.
4. Multiply the following error by 2 and enter it as the value for Stall Force Tolerance in the Machine Setups > Sensor THC Axis screen.
5. Return to the Manual Options screen and return THC Jog Speed to its original value.

Sensor THC axis screen

The following setup screen becomes available when you select the Sensor THC on the Machine Setups screen.

1. From the Main screen, choose Setups > Password > Machine Setups.
2. On the Machine Setups screen, choose Axis > Sensor THC.



Tuning the axis with gains

Gains enable controlled speed and motion from a closed-loop servo system and reduce the effect of error in these systems. Gains can correct following error that occurs as a result of mechanical delays, friction and load, or uncontrolled speed.

Each gain value is a multiplier that modifies the output from the encoder and brings this output closer to the set point. Motion tuning involves adjusting gains to reduce error as much as possible and ensure reliable cutting performance.

Gains are often interrelated and the value for one gain parameter can have an influence on the performance of other gains. Therefore, motion tuning is experimental and requires multiple tests to reach a set of appropriate values for your system and application.

Most gains affect the raising and lowering of the torch, and improve the accuracy of the initial height sense motion. Voltage gain is the only gain that affects cutting.

The following definitions provide the default or recommended value for each parameter. These values are appropriate for the majority of applications.

Proportional Gain: The Proportional gain value determines how quickly the CNC adjusts its command to compensate for any positional error. If proportional gain is too high, the system and axis become unstable. During cutting, the torch can overshoot the required height. If proportional gain is too low, the system response is not precise.

Range: > 0 to 32,000

Recommended setting: 25

Feedforward Gain: The value for feedforward gain is a multiplier that reduces following error. In all digital control loops, the velocity command causes some following error. Increase feedforward gain to reduce this type of error.

When using stall force for workpiece sensing, feedforward gain can cause the torch to strike the workpiece with to much force. Set feedforward gain to 0 for this and most applications.

Note: The weight of the torch assembly can also be a factor in setting the value for this gain.

Range: 0 to 32,000

Recommended setting: 0 – 30; Sensor THC: 0

Derivative Gain: Derivative gain helps to minimize and smooth out the effect of sudden changes in speed. A higher value for derivative gain results in a slower response time to changes in speed in the control loop.

Derivative gain reduces future errors that are based on the current rate of change (acceleration or deceleration). For example, derivative gain prevents overshooting a corner by preventing unchecked acceleration as the torch approaches the corner.

For most velocity loop drives, this parameter is set to zero (0).

Range: 0 to 32,000

Recommended setting: 0 – 20; Sensor THC: 0

Velocity Gain: This parameter is available only if you select Current for Drive Type and you have a CNC, such as the EDGE® Pro Ti, that controls both position and velocity loops. Use this parameter to tune the internal velocity loop of the CNC to provide smoother machine motion and less overshoot.

Range: 0 to 32,000

Recommended setting: 0 for Sensor THC lifter which does not use Velocity Gain. 100 for other lifters such as the Sensor Ti that use current drives.

Integral Gain: Integral gain improves the positioning accuracy of the torch and reduces the cumulative effect of past errors that are caused by static friction or gravity. Excessive integral gain can result in system instability.

Range: 0 to 32,000

Recommended setting: 0; Sensor THC: 0

Voltage Gain: Controls the relative responsiveness of the entire Sensor THC to maintain the programmed cut height. The higher the value you enter, the more aggressive the response of the lifter. Voltage Gain is the only tuning parameter that affects cutting. The other gains affect raising and lowering the torch and initial height sense motion.

Range: 1 to 500

Recommended setting: 100; Sensor THC: 100

Servo Error Tolerance:

Servo error, also called following error, is the difference between the commanded motor position and the actual motor position. The servo error tolerance is the upper limit of the amount of following error allowed before the CNC faults.

The amount of servo error tolerance depends on the cutting system mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high could cause inaccurate motion or mechanical harm. Set the Following Error parameter in the Watch window and observe steady-state operation of the cutting system (some following error is normal). Set the servo error tolerance to a value slightly higher than the steady-state following error.

Set to the Servo Error Tolerance to three times the steady-state following error.

Range: 0 to 127 mm (5 inches)

Recommended setting: Sensor THC: 5.08 mm (.2 inch)

Stall Force Tolerance: Stall force is a method of detecting the workpiece during initial height sense. Use stall force when cutting underwater or in other conditions in which ohmic contact is not feasible. Stall force Tolerance is the maximum amount of following error on the THC axis. The CNC starts monitoring the THC axis following error when it reaches the Start IHS Height. The following error accumulates when the torch tip contacts the workpiece and the lifter continues downward movement but is stopped by the workpiece. When the following error exceeds the stall force tolerance value, the CNC knows the position of the workpiece motion reverses, Motion stops when the amount of reverse travel of the THC equals the Transfer Height.

Range: 0 to 127 mm (5 in.);

Recommended setting: Sensor THC: 1.27 mm (0.05 inch)

Encoder Counts per mm/inch: Sets the number of encoder counts per inch of axis travel.

Encoder counts per inch are equal to the resolution of the encoder, multiplied by the encoder revolutions per inch or mm (based on the machine drive gearing). Enter a value that is the number of encoder counts per inch of lifter travel. You can enter fractional encoder units and the CNC will keep track of these fractions automatically.

Range: Positive, non-zero values up to 39379.08 counts/mm (1,000,000 counts/inch)

Recommended settings: Sensor THC: 819.2 counts/mm (20807.68 counts/inch)

Encoder Polarity: This parameter allows you to change the encoder input polarity to ensure proper counting for positive machine motion without any wiring changes.

Select the option that ensures that positive axis motion results in a positive encoder count position.

To determine the existing encoder polarity:

- Disengage all axes so they spin freely.
- Perform a positive voltage test and watch the position value.
- If the value becomes negative, change the encoder polarity.

Options: Positive, Negative

Recommended setting: Sensor THC: Negative

DAC Polarity: Changes the output polarity of the servo command signal (or DAC output). The polarity can be changed to reverse the travel direction of the commanded axis.

Choose the option that ensures that a command to move the axis in the positive direction yields motion in the correct direction.

Options: Positive, Negative

Recommended setting: Sensor THC: Positive

Drive Type: Selects the type of control loop.

Select either Velocity or Current commands to be sent to the drive. Select Current for a lifter that is installed with an EDGE Pro Ti CNC.

Options: Velocity, Current

Recommended setting: Sensor THC: Velocity

Encoder Decode Mode: Select 4X for most applications to increase positional accuracy. The decode mode can be set lower for extremely high-speed applications.

Options: 4X, 2X, 1X

Recommended setting: Sensor THC: 4X

Feedforward Gain in AVC: This option is recommended for use in applications where voltage tracking is on *and* when the Z axis motion is part of a coordinated move with the other linear axes in the cutting system. For example, cutting rectangular tube requires the Sensor THC to lift while the tube turns, or cutting with an ABXYZ bevel head requires motion in X, Y, and Z axes to maintain the tool point when positioning the bevel head to an angle.

Set to No unless you are using the Sensor THC for bevel cutting with an ABXYZ-type bevel head, or in another application where coordinated Z motion is required.

Options: Yes, No

Recommended setting: Default: No

Retry on Transfer Fail: **Retry on Transfer Fail**: When IHS completes, the torch attempts ignition, the CNC starts a 10 second timer, waits for the Cut/Mark Sense input (or Cut Sense # input) to turn on, and displays the status message Waiting For Arc On, If the input does not activate within 10 seconds, the ignition process stops, the torch retracts, and the IHS starts again with another attempt to ignite the torch. The CNC repeats this sequence twice.

Options: Yes, No

Slide Length: Enter the physical length of the slide.

Recommended setting: Sensor THC: 321.31 mm (12.65 inches)

Home: Select the homing method for the THC Axis. If the Home to Switch option is selected, the CNC Nozzle Contact Sense input is the Home Switch input.

If you select the Hard Stop Current Limited option, the THC Torque Limit output should be connected to the current limit input of the amplifier to reduce torque during homing or initial height sense. Select the option that corresponds to the installed lifter and motor drive.

Options: Hard Stop, Hard Stop Current Limited, Home Switch

Recommended Setting: Sensor THC: Hard Stop Current Limited.

Home settings for current-type drives

Current Limit%: Set the maximum current output for the operation of the lifter motor.

Range: 0.001 to 100; Default: 50

Home Current Limit%: Enter the maximum current during the homing sequence to stall and detect the top of the slide. Set Homing Current Limit to 50% of drive capacity.

Range: 0.001 to 100; Default: 10

IHS Current Limit%: Set the maximum current during IHS to stall and detect the workpiece.

Range: 0.001 to 100; Default: 5

Tracking Mode

The CNC uses voltage tracking to lock on to the actual arc voltage and compare it to the Set Arc Voltage. You can choose individual tracking mode settings for Plasma 1, Plasma 2, Marker 1, Marker 2, and Laser processes.

Normal: For most applications, set tracking mode to Normal. Voltage tracking in Normal mode locks on within 2 volts of the Set Arc Voltage value. Adjust voltage gain to increase lifter responsiveness. In Normal mode, arc voltage accuracy is +/- 1.0 arc volt.

Average Voltage Over: Sets the arc voltage sampling rate. Phoenix records the arc voltage every millisecond and averages the arc voltage over the number of milliseconds entered. The average value is compared to the voltage set point and then the command to the lifter is calculated.

Recommended settings: Sensor THC: Choose Plasma 1 or 2 as the process, then choose Normal for tracking mode. Do not adjust voltage gain, and set Average Voltage Over to 25 ms. Do not use Special tracking mode with the Sensor THC.

HyIntensity® Fiber Laser: Choose Laser as the process, choose Normal for tracking mode, set Average Voltage Over to 1 ms, and set Corrections Made Every to 10 msec.

Special tracking mode

Use Special tracking mode only for applications that require an extra high-speed, high-accuracy response, such as diamond-plate cutting. In Special mode, arc voltage accuracy is +/- 0.25 volt.

Options: Normal, Special. The voltage gain range for Normal mode is 0 to 500 volts; the recommended value is 100 volts. The voltage gain range for Special mode is 0 to 10; the recommended value is 10.

Range: 1 to 100

Lock on Delta Limit: Creates a lock-on window above and below the lock-on value. The set arc voltage value, within this window, controls the operation of the THC. Enter the number of volts above and below the lock-on value to create the lock-on window.

Range: 0 to 10 volts; Default: 0

Lock on Rate Limit (for Special Tracking Mode): Sets the number of consecutive values within the lock-on window that are required to define lock-on. Enter the number of consecutive values within the lock-on window that must be recorded to remain at lock-on.

Range: 0 to 10 volts; Default: 10

Voltage Tracking Range: Sets the allowable variation in arc voltage from the set point. If the arc voltage exceeds this range, the CNC faults and pauses cutting. Enter the number of volts of allowable arc variation.

Range: 5 to 400 volts; Default: 20

Sensor THC I/O points

Following are typical I/O points assigned in the Machine Setups > I/O screen for Sensor THC.

THC Torque Limit: This output connects to the input /P-CL and /N-CL on the Yazkawa drive. This output limits the motor torque during IHS when using stall force, and during homing. The CNC turns this output off during cutting.

Nozzle Contact Enable: This output activates during the initial height sense and remains on while cutting when Nozzle Contact Cutting is set to ON in the Process screen.

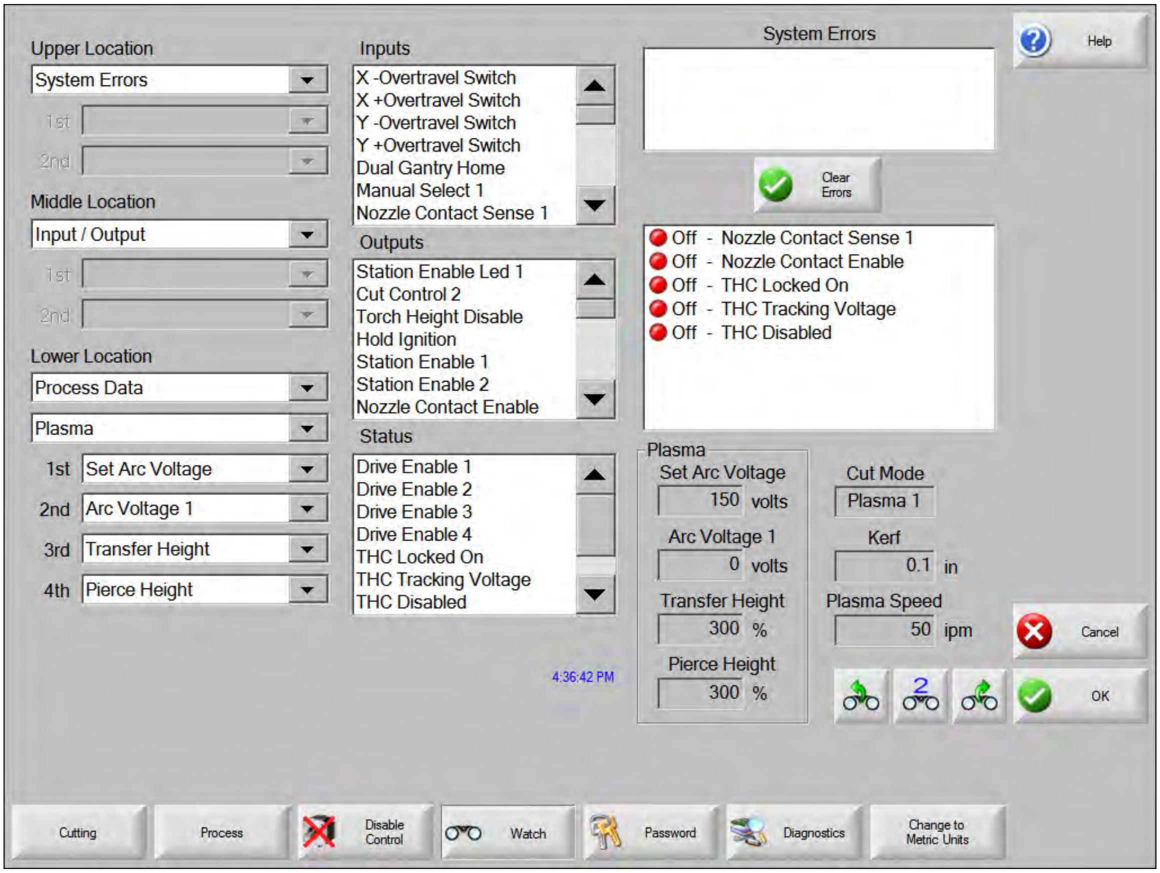
Nozzle Contact Sense: This input activates during initial height sense when the torch detects the workpiece and Nozzle Contact IHS is set to ON in the Process screen. Nozzle Contact Sense is also used as a homing input signal.

Nozzle Contact Enable and Nozzle Contact Sense are automatically assigned when the CNC supports the analog input board for the THC ports. You can select these I/O in the Watch Window, and the CNC displays them in the Analog Diagnostics screen. You cannot change their assignments.

Torch Collision: Determines the type of response that is used when the Torch Collision input is active. You can select to use a Fast Stop Decel value or the Fault Ramp time that has been selected for the individual axis.

Watch Window setup

An example Watch Window for the Sensor THC is shown below:



To set up this Watch Window:

1. Choose Setups > Watch.
2. Choose System Errors from Upper Location list.
3. Choose Input / Output from the Middle Location list. The Status list appears below the Inputs and Outputs lists. Select the following status bits and I/O from the list:

THC Locked On: This status bit turns ON when the arc voltage is within +/- 2 V of the set point.

THC Tracking Voltage: This status bit turns ON when the torch height control is using Automatic Voltage Control, When ON, the CNC makes adjustments to the position of the THC by comparing actual arc voltage to the set arc voltage. The torch height control adjusts the cutting height based on the arc voltage.

THC Disabled: This status bit turns on when the CNC disables the torch height control, usually as it approaches a corner in a part, (or a small radius, hole, or pierce), the cutting system slows down, and the actual cutting speed is slower than the program speed. This bit is controlled by the Torch Height Disable Speed along with the Distance Before, and Distance After settings on the Speeds screen, and by the M50 (Torch Height Disable) and M51 (Torch Height Enable) codes in a part program.

Nozzle Contact Sense: This input activates during initial height sense when the torch detects the workpiece.

Nozzle Contact Enable: This output activates during the initial height sense and while cutting if Nozzle Contact is on in the process screen.

Sensor Ti axis setup

The Sensor Ti lifter, used with the EDGE Pro Ti, requires different settings than the Sensor THC. Use the settings provided in the following sections for The Sensor Ti lifter. Refer to the previous sections about the Sensor THC for definitions of the settings.

Machine Setups screen

Parameter	Setting
Sensor THCs Installed	1
THC Installed on Axis	4
Analog input	3

The remaining parameters on the Machine Setups screen depend on the cutting system.

Speeds screen

Parameter	Setting
THC Acceleration Rate	100 mG
Maximum THC Speed	10160 mm/min (400 in/min)
THC Jog Speed	5080 mm/min (200 in/min)
THC Home / Fast IHS Speed	1270 mm/min (50 in/min)
THC Slow IHS Speed	127 mm/min (5 in/min)

The remaining parameters on the Speeds screen depend on the cutting system.

Sensor THC Axis screen

Parameter	Setting
Proportional Gain	50
feedforward Gain	0
Derivative Gain	0
Velocity Gain	150
Integral Gain	0
Voltage Gain	40
Servo Error Tolerance	5.08 mm (0.2 inch)
Stall Force Tolerance	0.51 mm (0.02 inch)
Encoder Counter mm/in	1028.35 counts/mm (26120 counts/inch)
Encoder Polarity	Positive
DAC Polarity	Negative
Drive Type	Current
Encoder Decode Mode	4X
Slide Length	165.1 mm (6,5 in)
Home	to Hard Stop Current Limited
Current Limit	25%
Home Current Limit	25%
IHS Current Limit	8%
Plasma Tracking Mode	Normal
Average Voltage Over	25 msec

Station Configuration screen

For Station 1, choose the Sensor THC as the lifter.

ArcGlide THC axis

The following setup screen is available if you have selected the ArcGlide THC in the Machine Setups screen. Refer to the *ArcGlide THC Instruction Manual* (806450) for information on installing the ArcGlide.

1. On the Main screen, choose Setups > Password > Machine Setups.
2. On the Machine Setup screen, choose the ArcGlide Axis soft key. If you have installed more than one ArcGlide axis, the screen displays a numbered soft key for each ArcGlide.
3. Choose the soft key for the ArcGlide axis you want to set up.

Gains

Gains enable controlled speed and motion from a closed-loop servo system and reduce the effect of position errors in these systems. Gains can correct following error that occurs as a result of mechanical delays, friction and load, or uncontrolled speed.

Each gain value is a multiplier that modifies the output from the encoder and brings this output closer to the set point. Motion tuning involves adjusting gains to reduce error as much as possible and to ensure reliable cutting performance.

Gains are often interrelated and the value for one gain parameter can have an influence on the performance of other gains. Therefore, motion tuning is experimental and requires multiple tests to reach a set of appropriate values for your system and application.

Most gains affect the raising and lowering of the torch, and improve the accuracy of the initial height sense motion. Voltage gain is the only gain that affects cutting.

The following definitions provide the default or recommended value for each parameter. These values are appropriate for the majority of applications.

Speed: This gain value regulates the speed of the lifter. Set Speed Gain after you enter the value for Maximum Speed.

Set the Maximum Speed *before* optimizing Speed Gain.

Optimize the Speed Gain value before attempting to change either the Position or Voltage gains. Manual motion and approaching the workpiece during automatic IHS require regulated speeds. If this value is too high, the lifter speed regulation will become unstable and prone to oscillation. If this value is too low, the speed regulation can become slow and inaccurate. Test this value by repeatedly raising and lowering the lifter in Manual mode. To optimize this gain, raise this value until there is a very slight oscillation during manual operation and then reduce the setting by 1.

Range: 1 to 10; ArcGlide: 5

Position: This gain is a multiplier for closed-loop positioning. Adjust Position Gain after you find the appropriate value for Speed Gain.

The THC uses position-based moves to move the torch to the IHS height and to the retract height at the end of a cut.

If this value is too high, the lifter positioning will become unstable and prone to oscillation. If this value is too low, the positioning can become slow and inaccurate.

Test this gain by repeatedly executing IHS TEST and checking that the lifter quickly and accurately reaches the IHS and Retract Heights.

To optimize this gain, raise the value until there is a very slight oscillation when the torch settles into the IHS or Retract positions and then reduce the setting by 1.

Range: 1 to 10; ArcGlide: 5.

Voltage: This gain is used when the THC operates with a closed-loop arc voltage control. Adjust this value after you find the appropriate value for Speed Gain.

If this value is set too high, the lifter position during closed-loop arc voltage control will become unstable and prone to oscillation. If this value is too low, the arc voltage control can become slow and inaccurate. Test voltage gain by repeatedly performing a test cut under closed-loop arc voltage control and verify that the THC quickly and accurately reaches the set arc voltage.

To optimize this gain, raise this value until there is a very slight oscillation during a cut and then reduce the setting by 1 or 2.

Range: 1 to 10; ArcGlide: 5

Speeds

Maximum: Sets the maximum linear speed that the THC can achieve. This value depends on the lifter motor speed at 45 VDC, the screw pitch, the weight of the lifter load, and the desired speed of operation.

This value is used as the 100% speed value when scaling the control equations. It is important that this value is set correctly because it affects the scaling of all the other control loops.

Set this parameter during installation, before optimizing Speed Gain.

Set this value to a speed that can be achieved easily. Use a conservative setting and assume a low, incoming AC line voltage. If this speed is set too high, the maximum available motor drive voltage will be insufficient to drive the motor to the desired maximum speed. This will make the top of the linear control range unavailable and result in sub-optimal operation.

If the Maximum Speed is set too high, it may be difficult for the THC to operate precisely at slow speeds when trying to control the arc voltage or to move to an accurate IHS distance. If this speed is set too low, the linear control range will only cover the low portion of the available mechanics range and the result may not be optimal.

For the Hypertherm 239 mm (9.40 inch) lifter, this value is 600 in/min (15240 mm/min). At the low input line voltage limit, the motor drive is capable of supplying about 45 VDC. The motor used in the Hypertherm 239 mm (9.40 inch) lifter will produce full-rated torque at about 3,300 RPM at this voltage. The Hypertherm 239 mm (9.40 inch) lifter has a 5-mm pitch ball-screw which requires 25.4 (in/mm) / 5 (mm) = 5.08 revolutions to travel one linear inch. This results in a value of 3,300 (RPM) / 5.08 (revolutions/inch) = 650 linear in/min of travel speed. This value was rounded down to the nearest even hundreds to guarantee a 100% value under all conditions.

ArcGlide: 15,240 mm/min (600 in/min)

Fast: This speed governs all automatic rapid moves such as the End-of-Cut-Retract or the Initial Approach to the workpiece.

Set this parameter to the value of Maximum Speed. However, you can set it lower if there is a heavy load on the lifter. With the Hypertherm 239 mm (9.40 inch) lifter, this value is set to 600 in/min (15,240 mm/min).

ArcGlide: 15,240 mm/min (600 in/min)

Slow: This speed is the Homing Speed and is the fastest speed for manual moves. It is also the maximum speed limit during arc voltage control.

ArcGlide: 3,810 mm/min (150 in/min)

IHS: This parameter sets the slow speed for the final approach to the workpiece during an IHS operation. It is also the slow speed for manual moves. To avoid poor regulation of speed, do not set this value below 10% of Maximum Speed.

ArcGlide: 1,524 mm/min (60 in/min)

Mechanical

Encoder Counts: This value scales the position feedback from the encoder.

The value is based on the ball-screw pitch and the encoder resolution. It is equal to the number of encoder counts per revolution, multiplied by the number of revolutions required to travel one inch. The encoder is operated in 4X mode, so the number of counts per revolution is equal to the number of pulses per revolution multiplied by four.

The Hypertherm 239 mm (9.40 inch) lifter has an encoder with 1,000 pulses per revolution which produces 4,000 counts per revolution in 4X mode. The lifter has a 5 mm pitch ball-screw which requires 25.4 (in/mm) / 5 (mm) = 5.08 revolutions to travel one linear inch. This results in a value of 5.08 (revolutions) x 4,000 (counts/revolution) = 20,320 counts per linear inch of travel.

ArcGlide: 800 counts/mm (20,320 counts/inch)

Slide Length: This value is the length of the lifter's usable travel. The ArcGlide lifter has 24.13 cm (9.45 inches) of usable travel.

Current Limit: This is the maximum continuous motor current for which the lifter motor is rated.

This value is the normal, continuous operating current limit but motor current can periodically exceed this value for a few seconds during extreme acceleration. The ArcGlide lifter uses a motor with a 3 A continuous current limit.

The motor can be damaged if the value for this parameter is set above the continuous current operating limit for the installed motor.

Range: 1 to 6 A; ArcGlide: 3 A

Stall Force: This value determines the amount of force for the Stall Force backup to the ohmic contact sensing. The values are relative values and start at 1 for the minimum force and 10 for the maximum force. This value should be set high enough to avoid false stall detection but not so high that the excess force causes a deflection of the workpiece and inaccurate IHS operation.

With the Hypertherm 239 mm (9.40 inch) lifter, the value of 5 is appropriate. It may be necessary to reduce this value if the THC is used with a thin workpiece. Use the Test IHS function to optimize this value.

Test this feature by disabling the ohmic contact by using the process menu or by removing the ohmic contact wire. The Stall Force value is optimum when there are no false workpiece detections and the IHS operation is accurate.

Range: 1 to 10; ArcGlide: 5

Miscellaneous

Pointer Offset Side: This value is the offset distance between the torch center and the laser pointer when looking from the side of the cutting machine. Use the precise distance between the laser spot and the center of the torch to fine-tune this value.

ArcGlide: 66.04 mm (2.600 inches)

Pointer Offset Front: This value is the offset distance between the torch center and the laser pointer when looking from the front of the cutting machine.

Use the precise distance between the laser spot and the center of the torch to fine-tune this value. To verify the laser offset, pierce a hole in the workpiece. Perform a manual offset using the laser pointer offset in Manual Options. If the offset is set correctly, the laser should be inside the pierce hole.

ArcGlide: 107.9246 mm (4.249 inches) (this value can vary slightly)

Voltage Calibration: This value is used to finely calibrate the measured arc voltage accuracy.

While this parameter is not normally required, it can be used when fine tuning multiple ArcGlide THCs on a cutting system.

The measured input value is multiplied by this constant before being used for either voltage control or display.

This calibration affects all of the ArcGlide hardware connected to this station including the HMI display and the HPR Hypernet interface. When an ArcGlide is shipped from the factory, the voltage has been calibrated in hardware and this value is set to 1.000. This parameter is normally only used to adjust for the same arc voltage displays on multiple ArcGlide installations.

Range: 0.900 to 1.100; ArcGlide: 1.000

Arc Voltage: This Arc Voltage value is for display only and cannot be edited. It is provided to check the results after changing the Arc Voltage Calibration above.

Range: 0 to 400

Retry on Transfer Fail: When IHS completes, the torch attempts ignition, the CNC starts a 10 second timer, waits for the Cut/Mark Sense input (or Cut Sense # input) to turn on, and displays the status message Waiting For Arc On, If the input does not activate within 10 seconds, the ignition process stops, the torch retracts, and the IHS starts again with another attempt to ignite the torch. The CNC repeats this sequence twice.

Options: Yes, No

Section 4

SERCOS setup

Hypertherm CNCs support both SERCOS II (fiber optics) and SERCOS III (Ethernet) communication protocols.

- See *SERCOS II support* on page 127 for information on SERCOS II.
- See the Field Service Bulletin *SERCOS III for Hypertherm CNCs* (808000) for setup information for SERCOS III drives. This Field Service Bulletin is provided with all SERCOS III-enabled Hypertherm CNCs and is available in the Downloads Library at Hypertherm.com.

SERCOS III support

SERCOS III is a real-time communication bus for digital servo drives designed for high-speed communications over industrial Ethernet cable. Hypertherm CNCs with Phoenix software version 9.74.1 or higher support the following SERCOS III equipment.

Drives

- Bosch IndraDrive Cs Basic with multi-Ethernet support, drive firmware type MPB, and firmware version 16V24, 17V14 or later, and 18V06 or later.

  For best performance, use the IndraDrive Cs Basic drive instead of the economy drive. The economy drive requires a 2 ms servo loop update rate (also called the *cyclic rate*) instead of the 1 ms update rate.

- Kollmorgen AKD drives with the following part number format: AKD-PXXXXX-***NBS3***-XXXX where ***NBS3*** designates the drive with firmware compatible with Phoenix 9.74.0.

Bus couplers

Bosch I/O bus coupler and I/O modules (listed by part number):

- R-IL S3 BK DI8 DO4-PAC – SERCOS III bus coupler, 8 digital inputs, 4 digital outputs, 500 mA
- R-IB IL AO 2/U/BP-PAC – Analog output module, 2 analog outputs, 2-wire connection
- R-IB IL AI 2/SF-PAC – Analog input module, 2 analog inputs, 2-, 3-wire connection

WAGO™ bus coupler and I/O modules (listed by part number):

WAGO Part Number	Description	Comments
750-459	Analog input module (4 inputs)	0–10 VDC (single ended)
750-351	SERCOS III bus coupler	
750-530	Output module (8 outputs)	24 VDC outputs
750-430	Input module (8 inputs)	24 VDC inputs
750-559	Analog output module (4 outputs)	0–10 VDC
750-627	Terminal bus extension	Allows connecting of remote I/O modules
750-628	Terminal bus extension coupler	
750-1500	Output module (16 outputs)	Ribbon cable interface
750-1400	Input module (16 inputs)	Ribbon cable interface
750-600	End module	No function (physical end cap)

Cabling

The SERCOS III ring requires Cat5e STP (twisted shielded pair) cables. While the ring can function through a single SERCOS port on the CNC, connecting a second cable to the second SERCOS port provides redundancy.

Hypertherm provides the following cables:

- 223380 – Ethernet Cat5E shielded cable, 5 ft (1.5 m)
- 223382 – Ethernet Cat5E shielded cable, 2 ft (0.6 m)
- 223381 – Ethernet Cat5E shielded cable, 1ft (0.3 m)

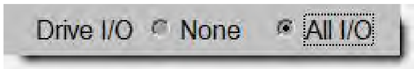
General recommendations

Follow these recommendations for ease of setup and best performance:

- Follow this order: set up the SERCOS ring, set up the CNC to communicate with the ring, and then third, phase-up the ring.
- Dedicate a single drive to each axis in the cutting system.
- Do not mix drive brands or types in the same ring. For example, the CNC can communicate with either the IndraDrive CS Basic drive and the IndraDrive CS Economy drive, but mixing the two types of drives on the ring will force CNC to use the higher servo loop update rate between the two drives and reduce system performance. For example, if one drive type uses a 1ms update rate, and another drive type uses a 2ms update rate, the CNC uses the 2ms update rate for *all* the drives.
- Connect the axis limit switches (also called overtravel switches) to the drive I/O, not to an inline I/O module. The drive and the CNC together provide the capabilities for the best possible machine deceleration in the event of a machine stop.

Drive I/O

The All I/O option for the Drive I/O parameter on the SERCOS screen (Setups > Password > Machine Setups > SERCOS) controls the use of the drive I/O. Set the Drive I/O parameter to All I/O.



Some I/O assignments may already be made in the drives. The CNC does not reset these assignments. However, if the drive is using the I/O, set the same I/O point in the CNC to Spare. Likewise, when the CNC uses an I/O point, set the same I/O point in the drive to "not assigned".

In most cases, assign the I/O points in the CNC and not in the drive. The only exception to this rule is the assignment of X and Y hardware overtravel (limit) switches because the drive provides the best-case deceleration of the cutting system if an overtravel switch is activated.

Inline I/O bus coupler

An inline I/O bus coupler can be used to handle miscellaneous I/O on the cutting system. Use the following settings to set up the CNC to use the I/O coupler in addition to the drive I/O:

1. Choose Setups > Machine Setups > SERCOS.
2. For Drive I/O choose **All I/O**.
3. For Bosch Inline I/O at Address 50 choose **Yes**.

The CNC assigns address 50 to the I/O coupler. Use the next available digital input, digital output, and analog input numbers. Using the example in the previous section, the I/O coupler would begin with digital input 22, digital output 4, and analog input 8.

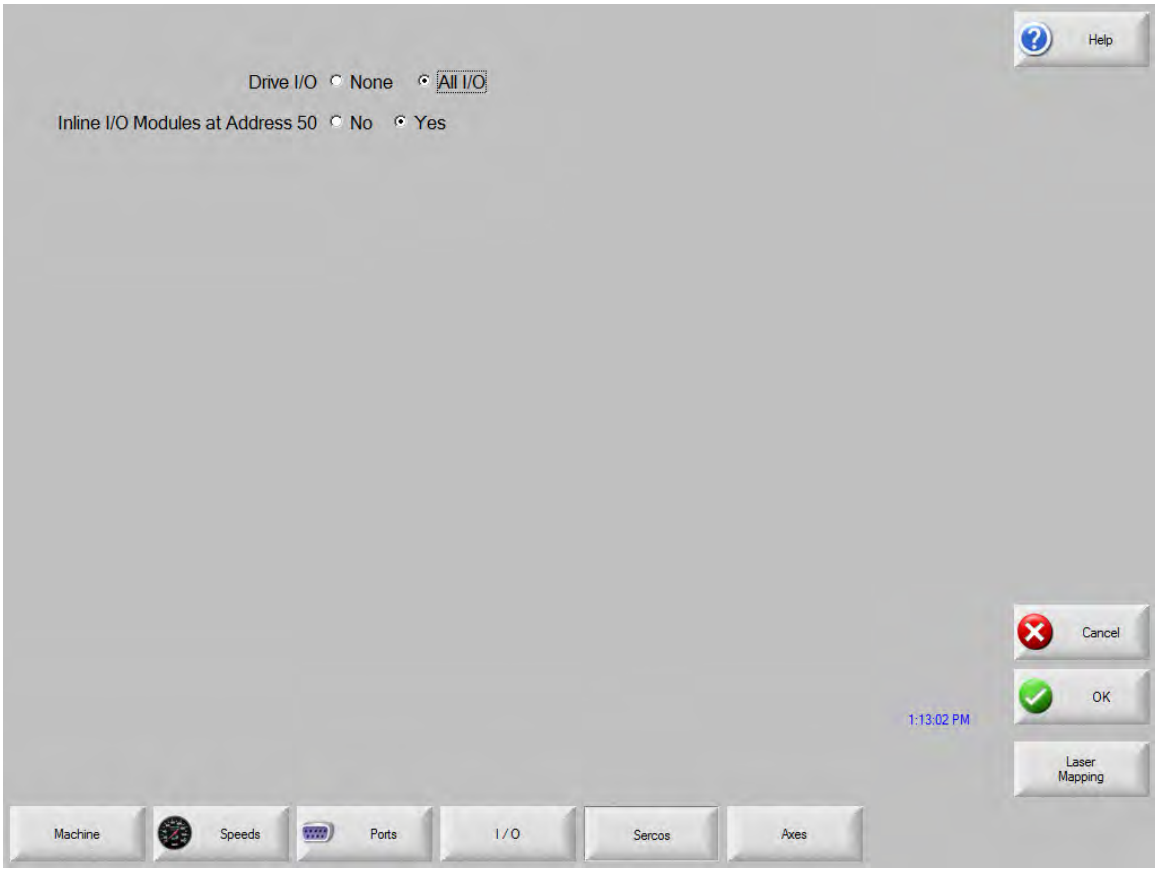
Troubleshooting tips

If you experience trouble phasing up the SERCOS ring, follow these guidelines.

- Check the cable type. SERCOS III requires CAT5e shielded cables. CAT5 cables are not capable of the faster transmission speeds required for SERCOS III.
- Check the cabling from P1 on the CNC to drive 1, and then from drive to drive. Be sure the cables are properly seated in their connectors on the drives and CNC and that there are no breaks in the cables.
- The CNC supports additional SERCOS passwords that can be used for troubleshooting or for preventing drives from phasing up.
  - The passwords, "1SA" through "12SA" must be used in sequence beginning with 1SA.
  - To start troubleshooting, enter the password 1SA to phase up only drive 1. If that is successful, enter the password 2SA, then 3SA and so on. When testing is complete, either enter the password SA, or cycle power to the CNC to re-enable all drives in the ring.
  - You can also use the passwords to phase up some, but not all of the drives. For example, in a six-drive cutting system, enter 4SA to phase up drives 1 through 4, but not 5 and 6.

SERCOS setup screen

The CNC requires that the drive I/O or inline bus coupler I/O be enabled. Choose Setups > Password > Machine Setups > SERCOS to open the setup screen.



Drive I/O: Select All I/O to enable the drive I/O (recommended). Select None if the drives do not have available I/O or you do not intend to use the drive I/O.

Bosch Inline I/O at Address 50: Select Yes if you are using an inline I/O bus coupler to expand the total number of I/O in the SERCOS ring.

SERCOS II support

The CNC supports either SERCOS II (fiber optic) or SERCOS III (Ethernet) communication interfaces. The previous section, *SERCOS III support* provides more information on SERCOS III.

Drives

- **Indramat/Bosch**
  - IndraDrive Mi
  - IndraDrive C basic drive
  - IndraDrive C basic drive with MA1 option
  - IndraDrive C advanced drive
  - IndraDrive C advanced drive with MA1 option
  - IndraDrive M Double Drive (I/O on drives is not supported)
  - EcoDrive 03
  - EcoDrive Cs
- **Kollmorgen**
  - ServoStar 300
- **Kollmorgen/Danaher**
  - ServoStar 600
  - SSCD
- **Yaskawa**
  - Sigma II with TRIO SERCOS adapter
- **PacSci**
  - PC840 Series

I/O bus couplers and I/O modules

I/O bus couplers

- Beckhoff SERCOS II BK7520 (has preferred higher baud rate)
- Beckhoff SERCOS II BK7500
- Bosch SERCOS II Reco
- Reco 02.2 SERCOS node
- 1 Beckhoff SERCOS node
- 1 Hypertherm Automation I/O SERCOS node

A single Hypertherm Automation I/O node can be configured with either 1 Reco or 1 Beckhoff node. No more than one Reco or Beckhoff I/O node can be enabled.

I/O modules

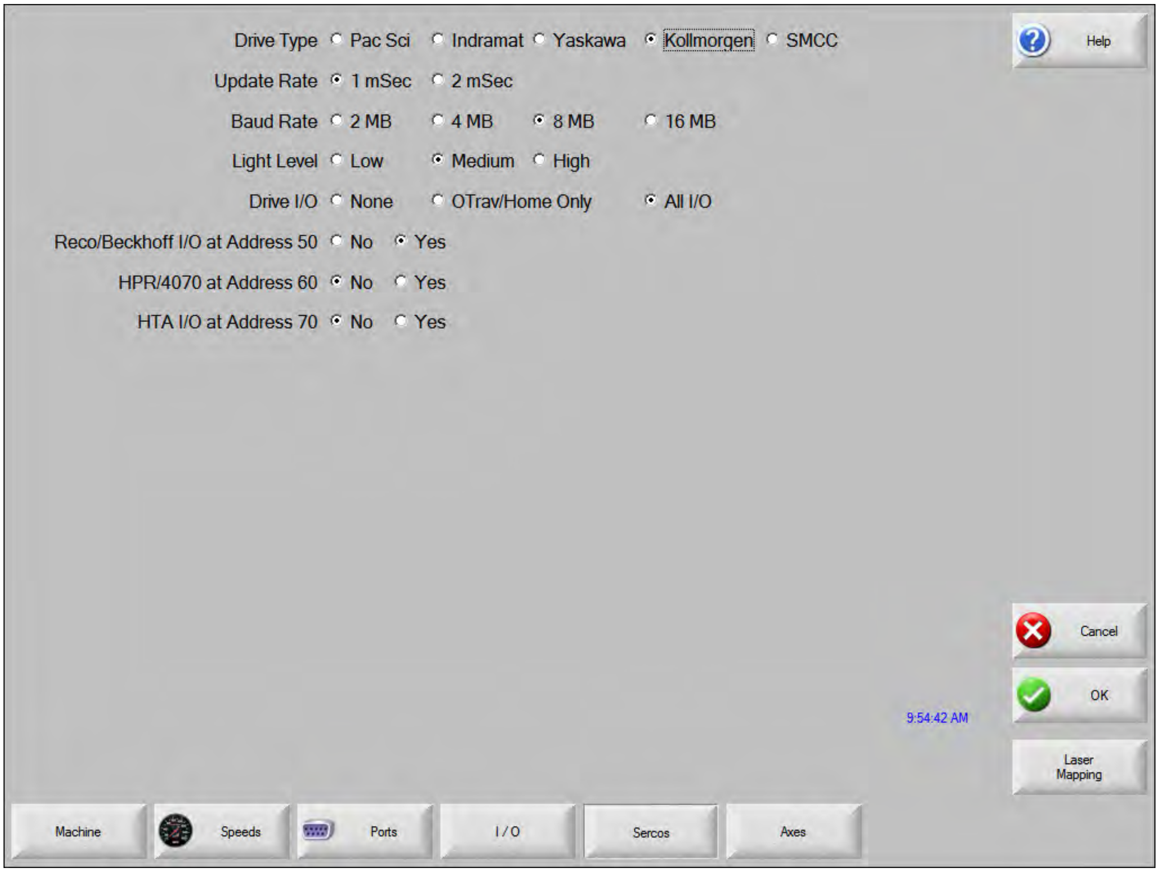
The following Beckhoff I/O modules are supported:

- KL4408 8 channel analog output
- KL4404 4 channel analog output
- KL3061 2 channel analog input
- KL9190 power feed terminal
- KL2488 8 channel digital output
- KL2622 relay output
- KM2002 16 channel digital output
- KM1002 16 channel digital input
- KL9010 end terminal

In addition, Reco I/O modules are supported, both analog and digital.

SERCSO II setup screen

Choose Setups > Password > Machine Setups > SERCOS to open the setup screen.



Drive Type: Select the manufacturer of the SERCOS style drive. This makes the proper operation and tuning parameters and software available.

Update Rate: Select the motion and I/O update rate for the SERCOS ring. One millisecond is recommended but the actual value depends on the capabilities of the hardware being used.

Baud Rate: Select the communication rate for the SERCOS ring. This is determined by the type of drive and I/O being used.

Light Level: Adjust the intensity of the light pulses used on the fiber optic ring. This feature compensates for diminished signal due to issues such as ring distance.

Drive I/O: Select the I/O options that are supported on the drive amplifier; None, Overtravel and Home Switches, or All I/O.

The drive I/O populates the general purpose I/O (both analog and digital) in order from the lowest SERCOS drive axis to the highest SERCOS drive axis. All I/O is the default setting for the SMCC interface.

RECO/Beckoff I/O at Address 50: Select Yes if the Indramat RECO or Beckoff I/O module is being used to expand the total number of I/O on the fiber optic ring.

The RECO I/O will populate the general purpose I/O (both analog and digital) following all drive I/O.

HPR/4070 at address 60: Select Yes if the optional SERCOS interface for HPR/4070 is being used.

HTA I/O at Address 70: Select Yes if the optional Hypertherm SERCOS interface card is being used.

Section 5

Station Setup

Overview

Hypertherm CNCs define a *station* as a physical tool on the cutting system along with its lifter (if one is used). The tool can be a plasma torch, marker, oxyfuel torch, laser, or water jet. Regardless of the tool, the CNC provides many of the same options for setup and operation of the stations.

The CNC supports multiple stations on a single cutting system. For the CNC to operate each cutting station independently, the stations must be set up with numbered inputs and outputs (I/O).

In this section you will find:

- Description of generic and numbered I/O
- Use of Auto Select and Manual Select inputs to activate either Program or Manual modes of operation
- How to assign the lifter, plasma supply (or other cutting supply such as marker or laser), and cut chart with each station (Station Configuration screen)

Generic and numbered I/O

The inputs and outputs for stations, available in Setups > Password > Machine Setups > I/O, can be used to control any of the cutting technologies supported by the CNC. The CNC supports two types of I/O for controlling the cutting stations:

- Generic I/O – used for any cutting process and to control a single cutting station
- Numbered I/O – used only for multiple cutting stations

Generic I/O

A standard group of inputs and outputs are used to control each cutting process. These I/O are called *generic* because all cutting sequences use them. Examples of generic I/O include Cut Control, Cut/Mark Sense, Torch Up, and Torch Down. Many generic I/O functions that affect stations have an equivalent numbered I/O.

Numbered I/O

When a cutting system has more than one station, you must use numbered I/O to identify the stations. Numbered I/O allow the operation of individual cutting stations through the CNC instead of through a programmable logic control (PLC) or relay logic interface. Each of the functions of generic inputs and outputs that affect stations is also available as numbered I/O.

It is recommended that you match the I/O number to the station number. For example, in a two-torch plasma cutting system, you would assign numbered I/O for each station:

Torch 1 (Station 1)	Torch 2 (Station 2)
Auto Select 1input	Auto Select 2 input
Manual Select 1 input	Manual Select 2 input
Station Enable LED 1 output	Station Enable LED 2 output
Cut Control 1 output	Cut Control 2 output
Cut Sense 1 input	Cut Sense 2 input

Enabling station I/O

Each station must be *enabled* before you can begin cutting. In many cutting systems, you enable a station with a switch on the operator console that activates an input (Manual Select input, for example). In other cases, the station may be enabled with the M37 code in the part program.

Auto Select and Manual Select inputs and Station Enable LED output

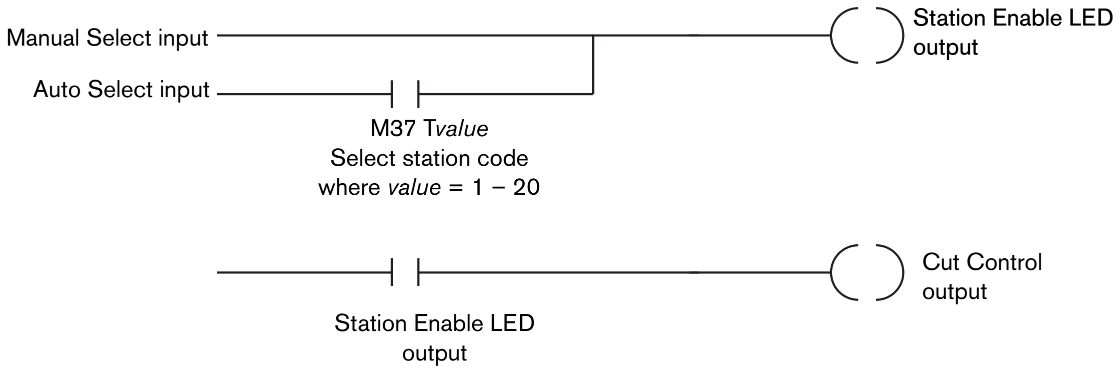
When creating an operator console that allows the cutting stations to operate in either manual mode or automatic mode (also called program mode), you must use the following inputs and outputs (as either generic or numbered I/O):

- **Manual Select** input: When Manual Select input is on, the station is in manual mode. Manual mode allows operations such as jogging, Go to Home, and rip cutting. Manual mode can also function as a program override for station selection. (The override function is described in more detail later in this section).
- **Auto Select** input: When the Auto Select input is on, the station is in automatic, or program, mode. For program mode to control the station, the part program must run the M37 T*value* code. The M37 code enables the station I/O.
- **Station Enable LED** output: This output must be ON for the CNC to activate the Cut Control output to tell the plasma supply to fire the torch. This output can be turned on in two different ways:
  - ❑ Manual Select input turns on the Station Enable LED output.

  Or

  - ❑ Auto Select input *and* the M37 T*value* in the part program must be read by the CNC. Both of these conditions must occur to turn on the Station Enable LED output.

The Station Enable LED output then activates the Cut Control output. The following picture shows the logic of the inputs and outputs that implement both manual and program modes in the operator's console. The EDGE Pro CNC operator console also uses this logic for the Station 1 and 2 switches.



Basic operating sequence

1. When preparing to cut, switch the station to manual mode or otherwise activate the Manual Select input. On the EDGE Pro CNC operator console, for example, move the Station 1 or 2 switch to the On position to activate the Manual Select input.

2. Load a part program and then align the part to the workpiece. Use the jog keys to position the torch for cutting.

3. Switch the station to automatic mode. On the EDGE Pro CNC operator console, switch the station to the Program position.

4. Press Start to begin cutting the part.

Using manual mode as an override

Another use of manual mode is to override the station selection in a part program. For example, the part program contains the M37 T1 code to select Station 1 for cutting the part. However, you want the program to use Station 2. On the EDGE Pro CNC operator console, switch Station 1 to Off and switch Station 2 to On. Load the part and press Start. When a station switch on the operator console is in the On (or manual mode) position, it overrides the station selection in the part program. In this example, the switch would cause the part to be cut with Station 2 instead of Station 1.

Summary

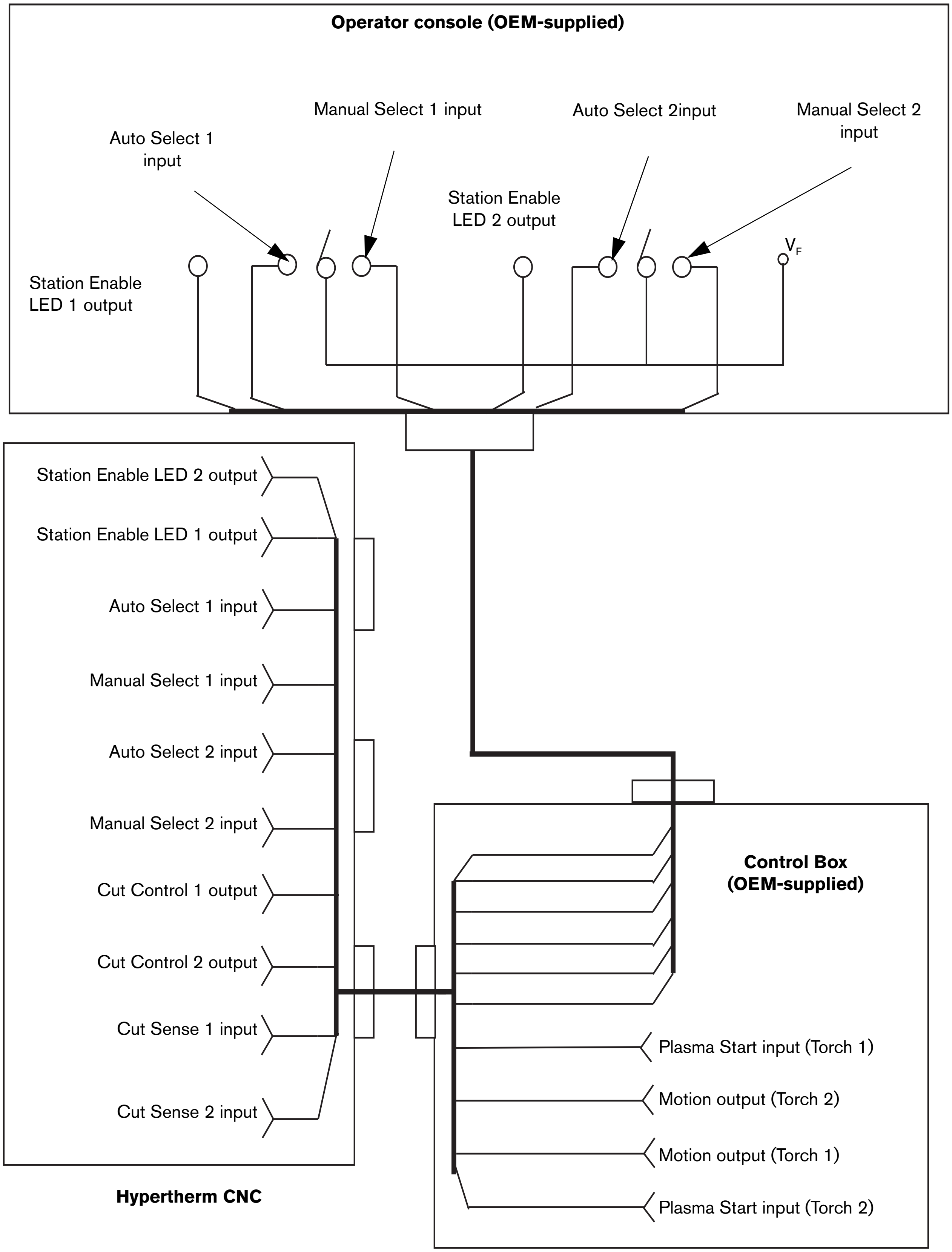
Auto Select and Manual Select inputs and the Station Enable LED output:

- Give the operator manual control over the stations, but also allow part programs that use the M37 code to select a station (Program mode).
- Are used on the EDGE Pro CNC for the Station switches on the operator console.
  - On = manual mode.
  - Program = automatic mode.
- Are an optimal method for controlling the stations when you have different tools on the same table (for example, plasma and laser).

  Note: Additional I/O may be required depending on the torch height controls and other mechanisms that are part of your cutting system.

The following picture shows an example of the Auto Select and Manual Select inputs, and Station Enable LED outputs and additional I/O for a two-torch operator console and control box.

Note: This is an example only and not intended as a recommendation for system design.



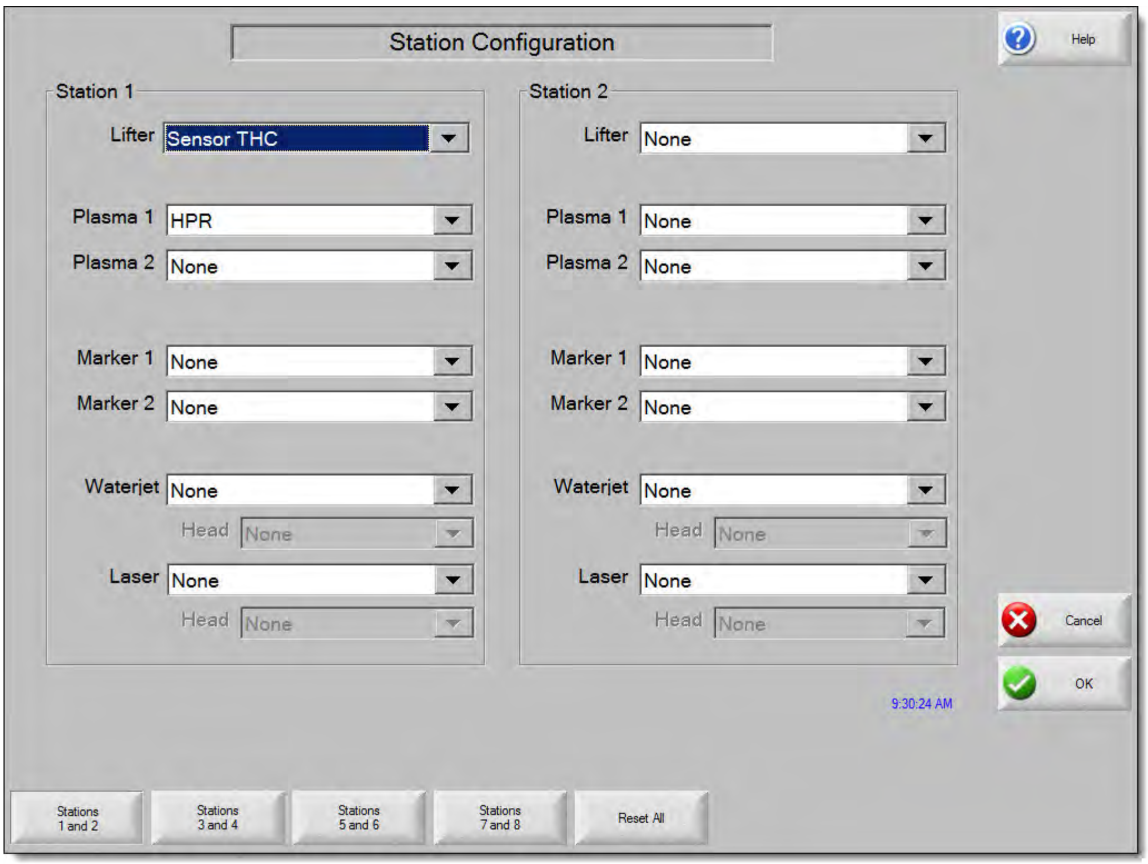
Operator console (OEM-supplied)
Manual Select 1 input
Auto Select 2input
Manual Select 2 input
Auto Select 1 input
Station Enable LED 2 output
Station Enable LED 1 output
V_F
Station Enable LED 2 output
Station Enable LED 1 output
Auto Select 1 input
Manual Select 1 input
Auto Select 2 input
Manual Select 2 input
Cut Control 1 output
Cut Control 2 output
Cut Sense 1 input
Cut Sense 2 input
Hypertherm CNC
Control Box (OEM-supplied)
Plasma Start input (Torch 1)
Motion output (Torch 2)
Motion output (Torch 1)
Plasma Start input (Torch 2)

Station configuration screen

The Station Configuration screen lets you assign the process to each station in the cutting system. In this screen, you select the type of torch height control and the model of the tool (plasma, marker, laser, or waterjet) for the station. The selections you make in this screen:

- Customize other screens in the CNC with features that are unique to the selected tool and torch height control
- Enable the cut charts for the selected tool

To open the Station Configuration screen, choose Setups > Password > Station Configuration.



Guidelines for using the Station Configuration screen

Follow these general guidelines to determine whether you need to make selections in the Station Configuration screen:

- Before using the Station Configuration screen, be sure you have made selections for Tools Installed in the Special Setups screen (Setups > Password > Special Setups). The Tools Installed selections define the processes used on the cutting system. For more information on Plasma 1 and Plasma 2 processes, see *Chapter 5 Plasma Setup.*
- When using the Sensor THC or ArcGlide THC for a lifter, be sure to select the lifter in this screen. Torch height control parameters specific to these two lifters become available in the Process screen so that you can optimize cutting performance.
- Select the process used by the tool on the station. For example, for Plasma 1 you may select HPR. This selection makes the cut charts available for the process.

Note: Oxyfuel systems DO NOT require a selection in the Station Configuration screen. Oxyfuel cut charts become available after you select Oxyfuel as a Tool Installed in the Special Setups screen (Setups > Password > Special Setups).

- For some plasma supplies, such as the HPR or Powermax, selecting the system enables serial communications to the plasma supply. Use Setups > Password > Machine Setups > Ports to set up serial communication.

  Note: Before the Sensor THC and ArcGlide THC become available in the Station Configuration screen, you need to select the torch height control in the Machine Setups screen (Setups > Password > Machine Setups). You also need to assign the Sensor THC to an axis.

- Use the soft keys to open the Station Configuration screen for Stations 1 – 2, 3 – 4, 5 – 6, 7 - 8.
- Use the Reset All soft key to return all settings for all stations to None.

  Note: See *Plasma Setup* on page 151 for more information on using the Station Configuration screen for a cutting system with multiple stations.

Conflicting process

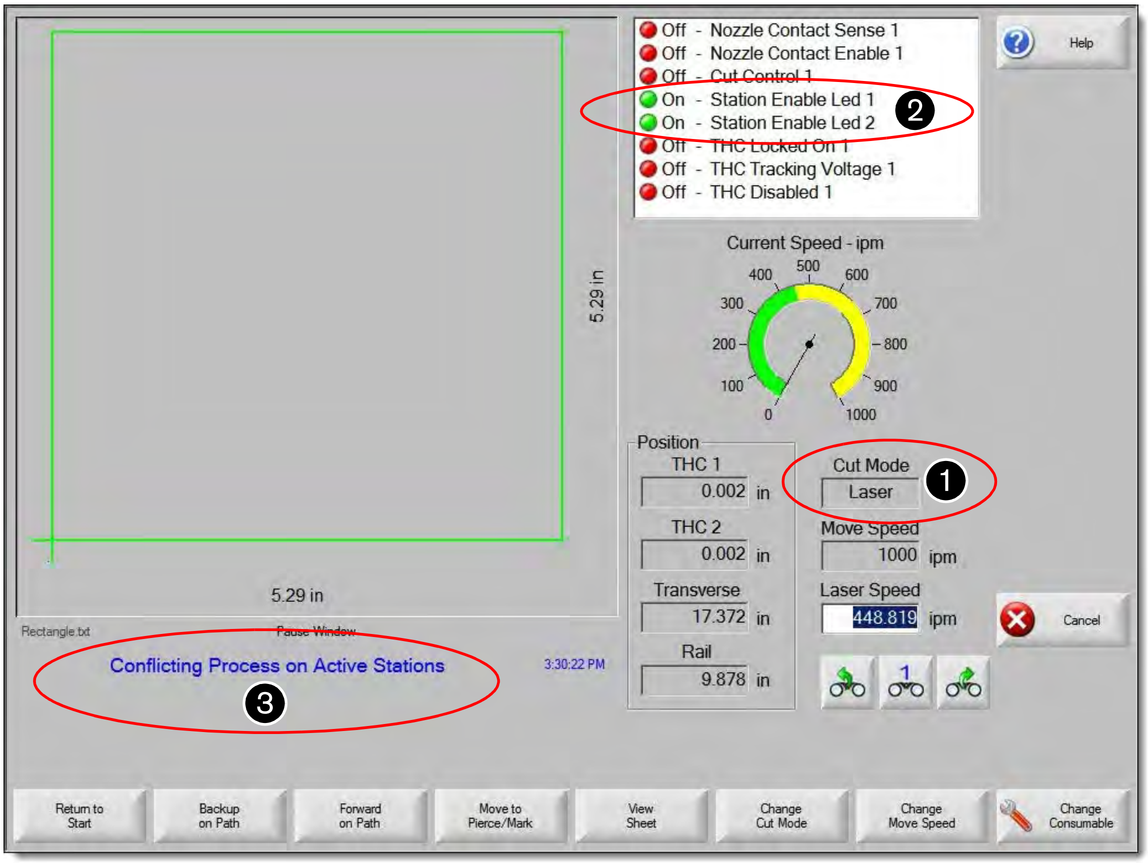
A conflicting process is an error condition that pauses a program or prevents a program from loading. A conflicting process occurs when the cut mode does not match the process (or processes) available for an active cutting station. The conflicting process routine in Phoenix compares the selected cut mode to the processes assigned to each active station and looks for a match. This happens automatically after you press Start on the controller. A station is active when the Station Enable LED output associated with it is active.

For example, if the cut mode is Plasma 1, then all active cutting stations require a Plasma 1 process or the program will pause with a conflicting process error message.

The check for conflicting process was added to ensure that dissimilar tools cannot operate if both of their stations are accidently left on by the operator. Conflicting process applies to all processes except Oxyfuel.

Example of a conflicting process

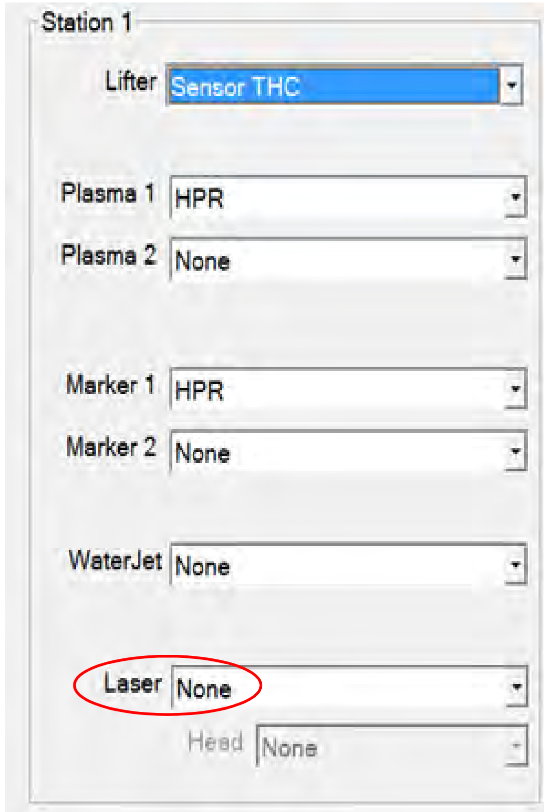
1. Cut Mode is set to Laser.
2. Station Enable LED 1 and Station Enable LED 2 are both active.
3. The status message indicates a conflicting process has occurred on active stations.



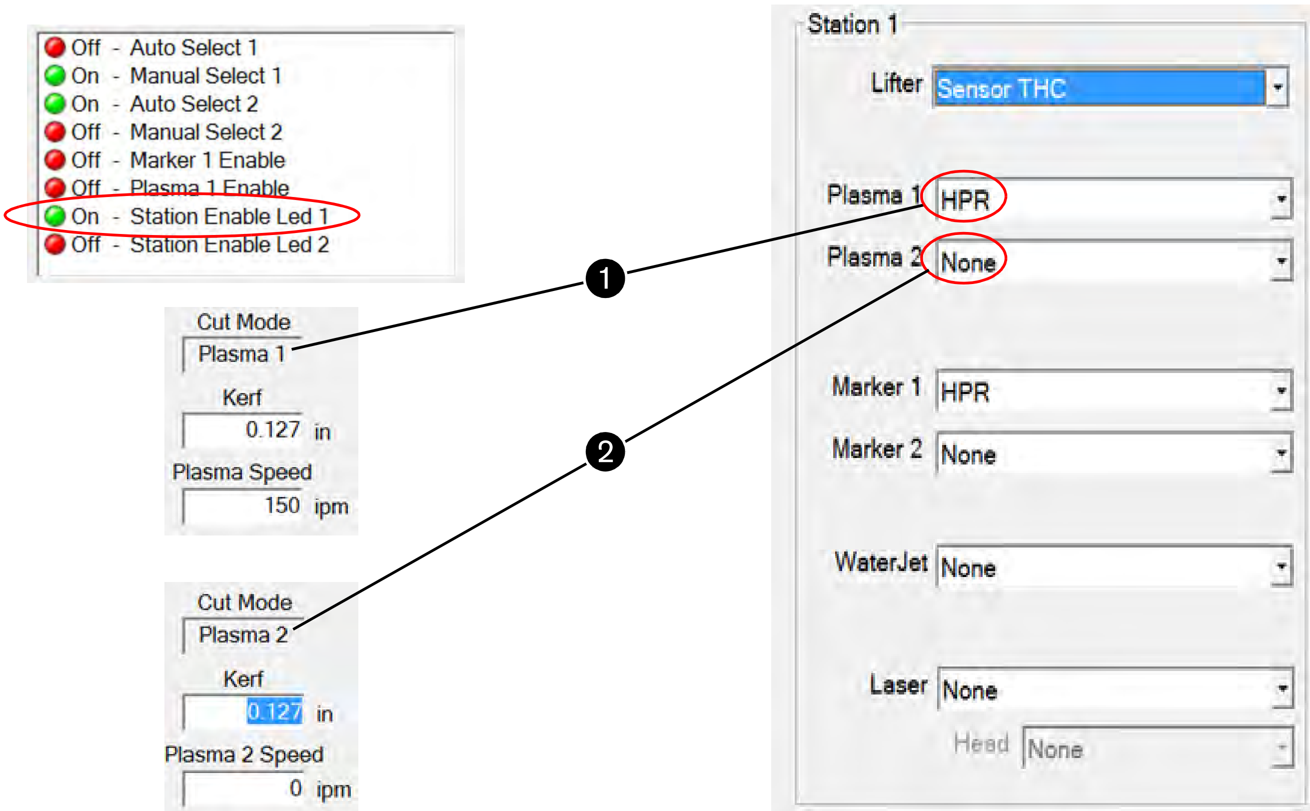
Processes that are assigned to a station on the Station Configuration screen are considered available processes. These processes include Plasma 1, Plasma 2, Marker 1, Marker 2, Waterjet, or Laser.

When Start is pressed Phoenix compares the processes assigned to the station against the selected Cut Mode. If they are not the same, the CNC pauses the program and displays the Conflicting Process message in the Status area.

The cause of the error in the this example is that Station 1 does not have a laser process assigned to the station and the station was active when attempting to cut with the Laser process. If station 1 was Off the error would not have occurred.



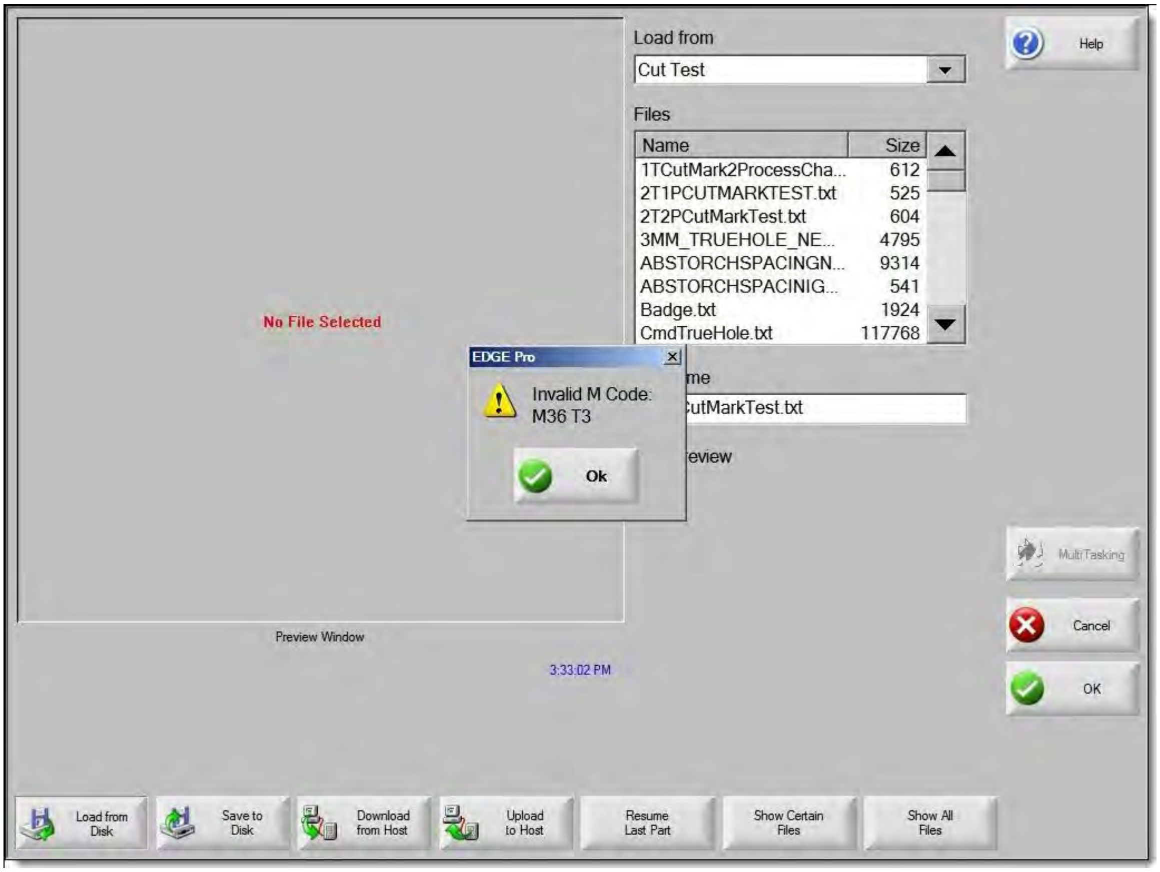
How a tool is associated with a station



In this example you can tell Station 1 is active because the Station Enable LED 1 is illuminated. When Start is pressed, Phoenix compares the cut mode with the processes assigned to the active station.

1. If the cut mode is Plasma 1, Phoenix checks for a valid Plasma 1 process on Station 1. HPR is a valid process.
2. If the cut mode is Plasma 2, Phoenix checks for a valid Plasma 2 process on station 1. There is no plasma process selected (None) so you get a conflicting process error.

A conflicting process can prevent a program from loading



The CNC displays a message indicating that an M-code is invalid. It is a legitimate M-code, but the Marker 1 process is not assigned to a station. For a process code to be a valid M code the process must be assigned to a station.

The part program in this example contains an M36 T3 (Marker 1 process select selection code) code, but a Marker 1 process is not assigned to a cutting station. However, Marker 1 is assigned as a tool. This type of scenario applies to all M36 process codes. In order to make use of an M36 code within a program, the process being called must be assigned to a station within the Station Configuration screen.

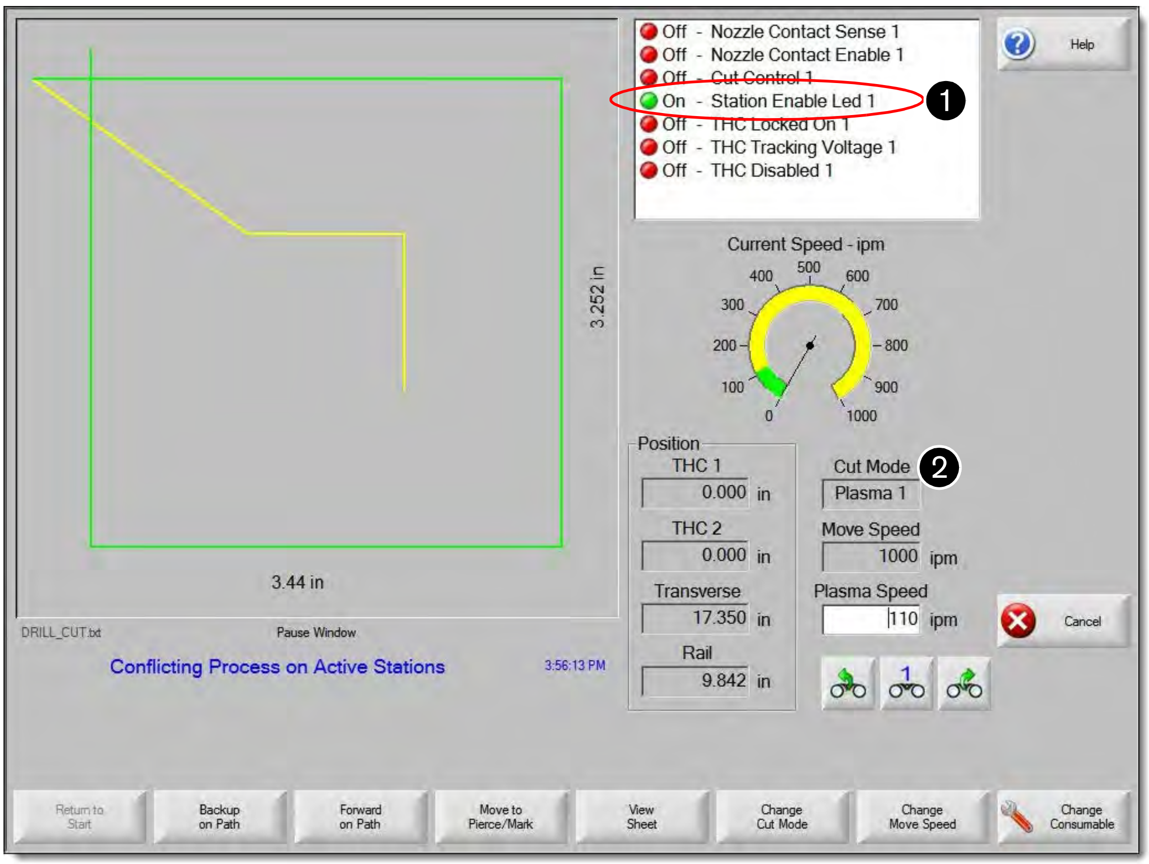
Process selection codes: M36 T1 (Plasma), T2 (Plasma 2), T3 (Marker 1), T4 (Marker 2), T5 (Laser), and T6 (Waterjet).

Adding a Marker 1 process to the Station Configuration screen resolves the issue.

Troubleshooting a conflicting process error

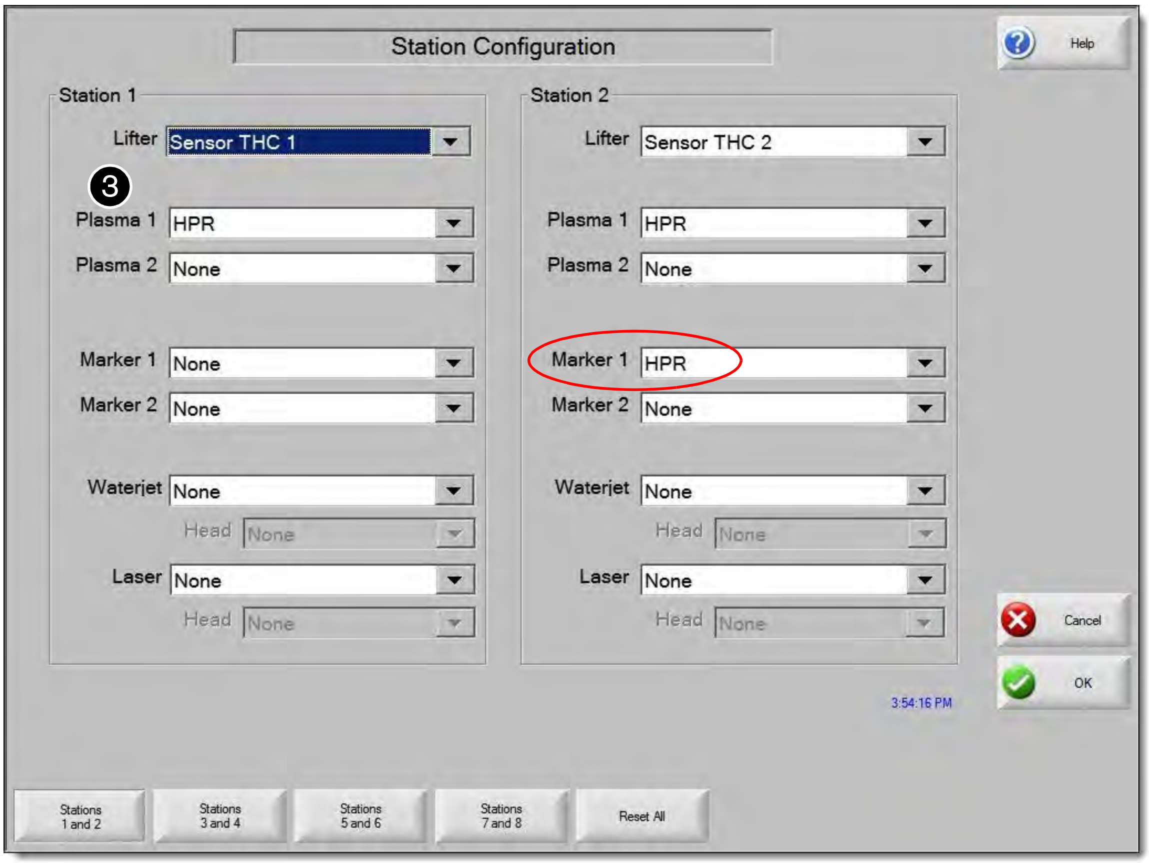
The settings look correct on the screen below

1. Station 1 is the active station.
2. The cut mode is Plasma 1.
3. Plasma 1 is defined on Station 1 on the Station Configuration screen. See the figure on the next page.



The cause of the error

Marker 1 is assigned to Station 2 in the Station Configuration screen and no station I/O is assigned for Station 2 which makes Station 2 active within Phoenix. The Station Enable LED 2 output becomes virtual and forced to ON if a process is assigned to a station and no station I/O is assigned.



Troubleshooting steps

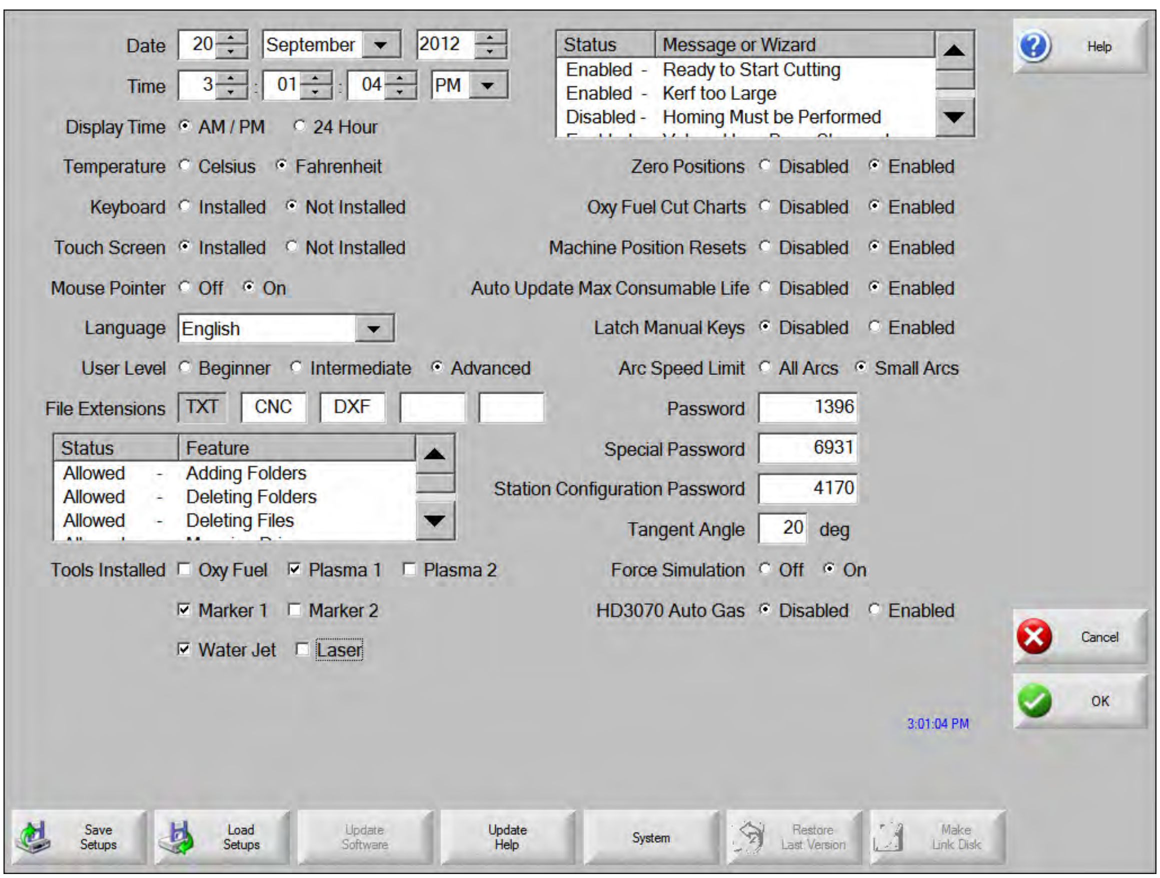
1. Review the settings in the Station Configuration screen. Does each station have:
   a. A station must have a process assigned that matches the cut mode.
   b. The processes assigned to the station must be valid.
   c. A physical tool with processes that match the station settings must be installed.
2. Review station I/O assignments.
   a. Verify that the station I/O is assigned properly.
   b. Each station should be: Auto Select # / Manual Select # / Station Enable LED #.
   c. If one station has station I/O, all stations need station I/O.
3. Verify the settings.
   a. Verify the Tools Installed on the Special Setups screen.
   b. Verify that all processes (for Tools Installed) are assigned on the Station Configuration screen.
   c. Verify that station I/O is assigned properly on the I/O screen.
   d. Verify that Station Select Override and Process Select Override are enabled on the Cutting screen if you are using M36 Tx codes to select a process from the part program.
4. Monitor the station I/O and processes (Watch Window).
   a. Monitor the Station Enable LED outputs.
   b. Monitor the Process Enable outputs.
   c. Monitor Process Data for each available process (preferably in one Watch Window).

Section 6

Special Setups

To open the Special Setups screen:

1. From the Main screen, choose Setups > Password.
2. Enter your password.
3. Choose OK.



Date: Enter the current date.

Time: Enter the current time. This is the time that is displayed on the main screen.

Display Time: Select whether the display time is shown in either AM/PM format or 24-hour clock format.

Mouse Pointer: Select On to use an external mouse or touchscreen.

Touchscreen Installed: Choose Installed if your control has a touchscreen. Choose Not Installed to enable keyboard control of the soft keys and other controls on the screen, and to display keyboard shortcuts on the screen.

Keyboard: Select Not Installed to use the on-screen keypad. Select Installed to use an external keyboard.

Language: Select the language that is displayed on the CNC from the available languages stored on the CNC.

Languages can be added to the CNC by performing the standard software update with a language-specific update file. Some languages also require the installation of a font viewer to properly display characters.

To change to a different language, select the desired language and press the OK soft key. You must restart the CNC for the new language to be displayed.

User Level: Select the user experience level. The level you select determines the screens and the features that are available. For example, the intermediate level provides a larger preview area and an ALT soft key to access additional screens.

File Extensions: Enter the file extensions of the part files that will be accepted at the control.

Status/Feature: Enable or disable software features. See the Status Feature List section later in this chapter.

Tools Installed: Selects the cut processes available to the CNC operator, for example, Plasma 1 and Marker 1. Tools Installed also enables the Process screens for the selected cut processes.

Status/Message or Wizard: Enable or disable system messages. See the Status/Message or Wizard List section later in this section.

Zero Positions: Determines whether the CNC operator has the ability to zero positions. When disabled, the zero positions soft key is grayed out and is unavailable to the operator.

Oxyfuel Cut Charts: Before Phoenix software version 9.72.0, the oxyfuel process did not use cut charts. In Phoenix 9.72.0 and higher, selecting oxyfuel as one of the Tools Installed on the Special Setups screen enables cut charts for oxyfuel processes. You can disable the use of oxyfuel cut charts so the your oxyfuel cut jobs will run as they did before Phoenix 9.72.0.

Machine Position Resets: If this setting is enabled, it resets the absolute machine position when the Zero Positions soft key is pressed. Only the current incremental motion (part) position is reset to zero. Position information based on homing is not lost.

Auto Update Max Consumable Life: If this feature is enabled, it tracks the consumable life values beyond the user-defined set point and assigns that maximum value as the new set point. If this feature is disabled, the user-defined set point for maximum consumable life is not updated.

Latch Manual Keys: Enables or disables the Latch Manual Key feature which allows the manual motion keys to remain on with a single key press.

Arc Speed Limit: Allows the user to turn off the Speed Limit Check that is performed by *SoftMotion* for arcs larger than 254 mm (10 inches). This compensates for abrupt motion commands caused by non-tangent line arc segments and smooths out motion.

Password: Allows the user to enter a new password for Machine Setups. Numbers or letters can be used.

Special Password: Allows the user to enter a new password for Special Setups. Numbers or letters can be used.

Station Configuration Password: Allows the user to enter a new password for Station Configuration setups. Numbers or letters can be used.

Force Simulation: Disables the motion control card (or MCC) to simulate motion on the screen.

HD3070 Auto Gas: When enabled, this setting allows the CNC to communicate with the auto gas console for the HD3070 and give access to the auto gas setup screen.

Tangent Angle: Sets the degree of the tangent angle for motion control.

Segments within a part that intersect at angles greater than the selected tangent angle decelerate to zero or to the minimum corner speed.

Segments within a part that intersect at angles less than or equal to the selected tangent angle, do not decelerate unless the next segment is a speed-limited arc.

Touchscreen Installed: Enables full keyboard control of the software. Using keyboard control enables a set of key combinations for accessing the soft keys and other screen controls.

Status/Feature List

The Special Setups screen lets you enable or disable software features in the Status/Feature list. To change the status of a software feature, double-click or press Space.

Adding Folders: Select whether the CNC operator can add folders from the parts directory on the CNC or the host computer.

Deleting Folders: Select whether the CNC operator can delete folders from the parts directory on the CNC or the host computer.

Deleting Files: Select whether the CNC operator can delete folders from the parts directory on the CNC or the host computer.

Mapping Drives: Select whether the CNC operator can map to external drives from the CNC through the optional network.

Configuring Watch: Select whether the CNC operator can change the items in the Watch Window.

Add, remove, or change processes: Allows you to restrict the operator's ability to add, remove, or change a cut chart.

Torch Spacing: When enabled, turns on manual torch spacing in the Manual Options screen. This feature allows the operator to control torch positioning before running a part program in a multiple-torch system.

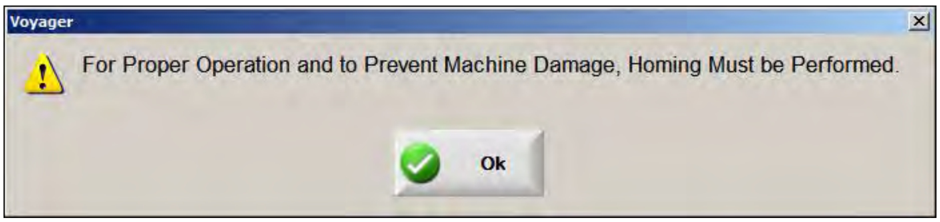
Status/Message or Wizard List

To change the status of a message, highlight the selected message and press the Space key.

Ready to Start Cutting: When enabled, the Ready Message feature will display a ready message when the Start button is pressed.

Kerf Too Large Warning: Disables the Kerf Too Large warning message. This message notifies the user that a conflict between the cut paths and the current Kerf value for a part has been detected and that some detail could be lost when the part is cut.

Homing Must Be Performed: Prompts the operator to home the selected axis before motion begins. The selected axis must have homing enabled on the axes setup screen.



Values Have been Changed: Displays as a confirmation when changes to values on the current configuration screen are detected. This ensures that changes are not made in error when exiting the screen.

Home Torch Height Control: Prompts the operator to home the Sensor THC axis at power up and before motion begins.

Ready to Final Align: Appears as part of the skew alignment function and displays just before final motion to position the tool begins.

Part Larger than Plate: Displays when the dimensions of the part that have been loaded exceed the selected plate dimensions.

Cut Chart Data Has Changed: Displays as a confirmation when changes to the values on the current Cut Chart screen are detected. This ensures that changes are not made in error when exiting the screen.

Save Part for Rush Job: Enables or disables the Resume Part prompt for Rush Job Interrupt.

Torch Already Raised/Lowered: Appears when the torch is fully raised or lowered.

Automatic Align Wizard: Enables or disables the automatic availability of the alignment wizard on the Align screen.

Automatic CutPro Wizard: Enables or disables the automatic availability of the Cut Pro wizard on the Main screen. If availability of the wizard is enabled, it appears 10 seconds after the Main screen is opened.

Start Cut from CutPro Wizard: Allows or prevents cutting to start directly from the Cut Pro Wizard. If this is disabled, the operator must use the Start button on the console to begin the cut.

True Hole Verification: On a True Hole cutting system, Phoenix can perform a series of checks to help ensure that the correct settings are made for True Hole cutting. You can enable or disable these checks individually. When enabled, Phoenix can automatically correct many of these settings.

True Hole I/O Verification: Checks for these outputs: Pierce Control, Pierce Control 1 – 20, or Pierce Control THC 1 – 4.

Torch Height Disable: Checks that no Station Select inputs are assigned.

True Hole Setups Verification: Checks for the following parameters and settings:

- EIA Kerf Override – Enabled
- EIA F-code Override – Enabled
- EIA G59 code Override – Enabled
- Process Select Override – Enabled
- Parallel Kerf Enabled – Disabled
- Arc Off Time ≥ .05 seconds
- Cut Off Time = 0
- If Auto Select Inputs AND Manual Select Inputs are assigned: Station Select Override – Enabled

For Command THC:

- Nozzle Ohmic Contact – Off
- Retract Delay > .22 seconds
- Pierce On with Cut On – Yes

For Sensor THC:

- Pierce on with Cut On – Yes

For Arc Glide THC:

- Use Pierce Complete – On

True Hole OpCon Verification: Checks the operator console switches:

- If any Auto Select inputs are assigned, then at least one station must be set to Program.
- Program speed set to 100%

Invalid process current selected: In the cut chart, you have selected a process current that is not valid for that is not supported.

Jog key watch warning: An option for Watch Window setup is to display manual-motion jog keys on the CNC touchscreen. However, if the CNC is installed in an environment where water droplets could come into contact with the touchscreen, unintended motion could occur. This warning displays when setting up a Watch Window with jog keys.

Verify lens installed and **Verify nozzle installed**: These messages may appear after a you have changed the cut chart and the new cut chart requires a different nozzle or lens.

Soft Keys

Save Setups: Press the Save Setups button to save the current CNC setting to the USB drive or hard drive. A window displays to select the drive and enter a file name.

Note: After installation of the CNC on the cutting table or if any setup parameters are changed, it is important to save the current setup file to both the hard drive and USB drive for future reference.

Load Setups: Press Load Setups to load the selected control setting from the floppy drive or hard drive on to the control. A window displays where you can select the drive and enter the file name.

Update Software: Press the Update Software button to update the CNC operating software from a floppy disk.

Update Help: Press the Update Help button to update the CNC help files from a floppy disk.

System Tools: Press the System Tools button to access core Windows features for system performance. Features include Update Registry, Scan Hard Disk, Defrag Hard Disk and Format Disk. System Tools also accesses features for virus scanning and adding special fonts for some languages.

Restore Last Version: Restores the CNC to the previous version of system software.

Make Link Disk: The CNC is shipped with proprietary communication link software for communication with a host computer. Press this soft key to transfer the link software to a floppy disk so it can be loaded onto a host computer. A text file is included with the software to instruct the user on setup and use of the link.

Section 7

Plasma Setup

This section describes the tasks you need to perform to set up the Hypertherm CNC with a plasma supply. Because of the variation between cutting systems, Hypertherm CNCs provide built-in flexibility for multiple methods of system setup, operation, and part programming. Since all cases cannot be described here, this section makes the following assumptions:

- The cutting operations are being controlled by the CNC, not by a programmable logic controller (PLC) or other external relay logic.
- The cutting system is equipped with a Hypertherm torch height control (THC).

In this section you will find:

- An overview of the Plasma 1 and Plasma 2 processes and guidelines for setting up the processes
- A detailed, step-by-step description of the plasma cut sequence
- Definitions of the inputs and outputs used for plasma cutting systems
- Plasma setup instructions for the CNC

Plasma part programs can use process selection variables to load the cut chart. The CutPro™ wizard provides additional automation. For more information about plasma process selection variables, see the *Phoenix Software V9 Series Programmer's Reference* (806420).

Plasma 1 and Plasma 2

Plasma 1 and Plasma 2 refer to two separate cutting processes. Marker 1 and Marker 2 likewise refer to two separate marking processes.

- Select Plasma 1 and Plasma 2 as Tools Installed on the Special Setups screen to enable the Plasma 1 and Plasma 2 Process screens.
- Assign the specific plasma supply to Plasma 1 and Plasma 2 on the Station Configuration screen to make the cut charts available along with other functionality specific to the cutting systems. In the Station Configuration screen you match up the process (Plasma 1 and Plasma 2) to the cutting station (Station 1, Station 2 and so on).

For more information on stations, see *Section 4 Station Setup*.

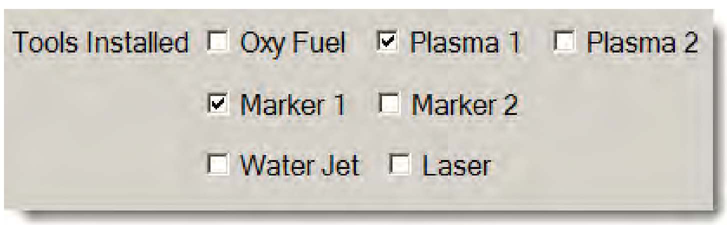
In general, follow these guidelines for using Plasma 1 and 2 on the CNC:

- In a single-torch cutting system, you need only Plasma 1 for Station 1.
- In a multiple-torch cutting system where the torches cut parts using the same process and cut chart, select only Plasma 1.
- In a multiple-torch system where the torches are from different plasma supplies, you need to select Plasma 1 and Plasma 2 so that the CNC makes a second cut process and cut chart available.
- Use both Plasma 1 and Plasma 2 when the part program calls for two different cutting processes. For example, the part program cuts the detail portions of a part with a low amperage consumable and then automatically switches to a higher amperage consumable to cut the part contour. This type of part program would require two torches: One torch has a low amperage consumable set while the second torch has a high amperage consumable set. Therefore, use Plasma 1 for the low amperage cut chart and Plasma 2 for the high amperage cut chart.

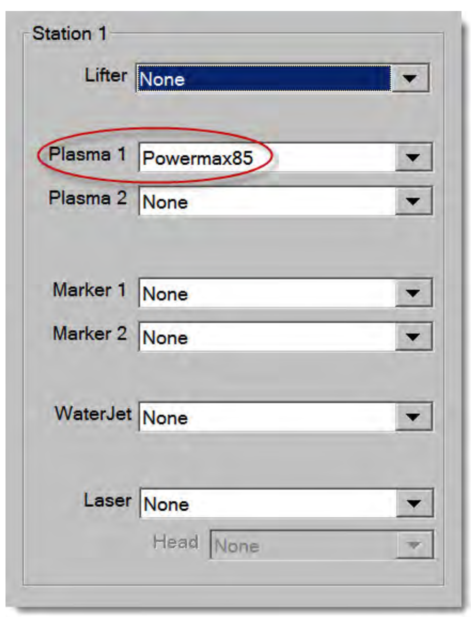
Examples using Plasma 1 and Plasma 2

To set up single torch Powermax85® system, follow these general steps on the CNC:

First, in the Special Setups screen, select only Plasma 1.



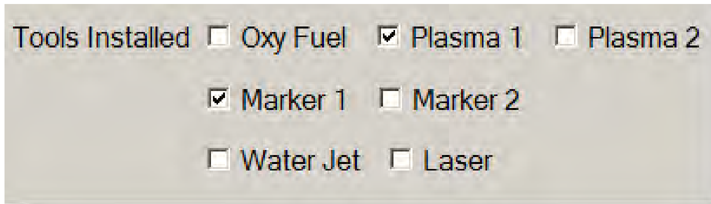
In the Station Configuration screen, choose the torch height control on the cutting station for the lifter. Next, assign the Powermax85 to Station 1.



Sample settings for a multiple-torch cutting system

To set up a multiple-torch HPR Auto Gas system that uses one cutting process and one marking process for all torches, follow these general steps on the CNC:

First, in the Special Setups screen, select Plasma 1 and Marker 1. Plasma 2 is a separate process and is not used in this example.

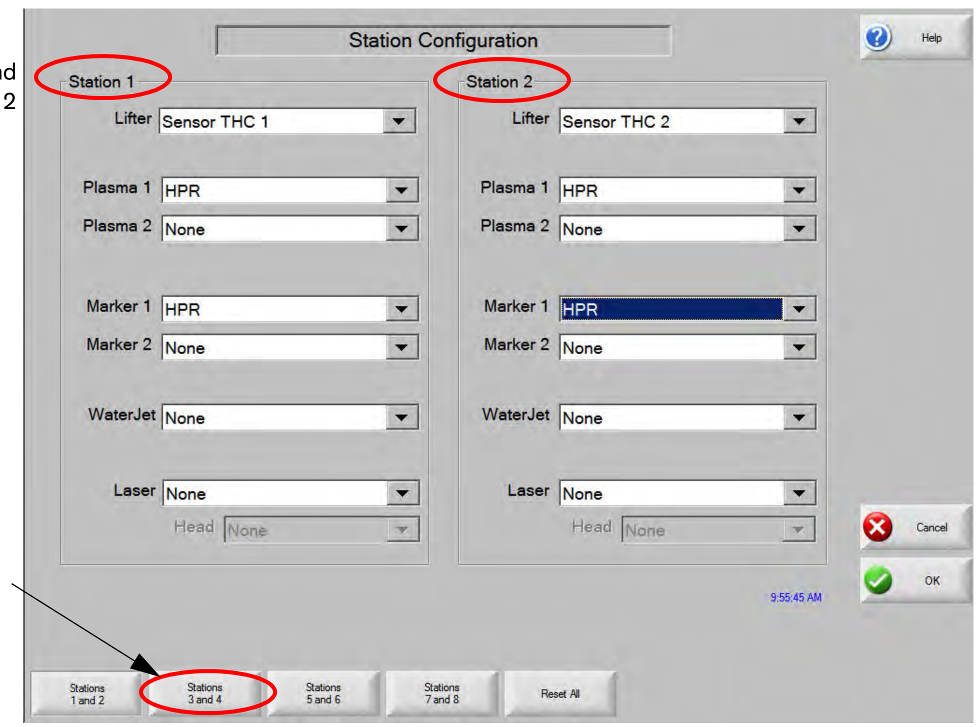


In the Station Configuration screens shown below, all four cutting stations have the HPR assigned to Plasma 1 and Marker 1. In this screen you are assigning the cutting and marking processes that will be used for each of the individual cutting stations.

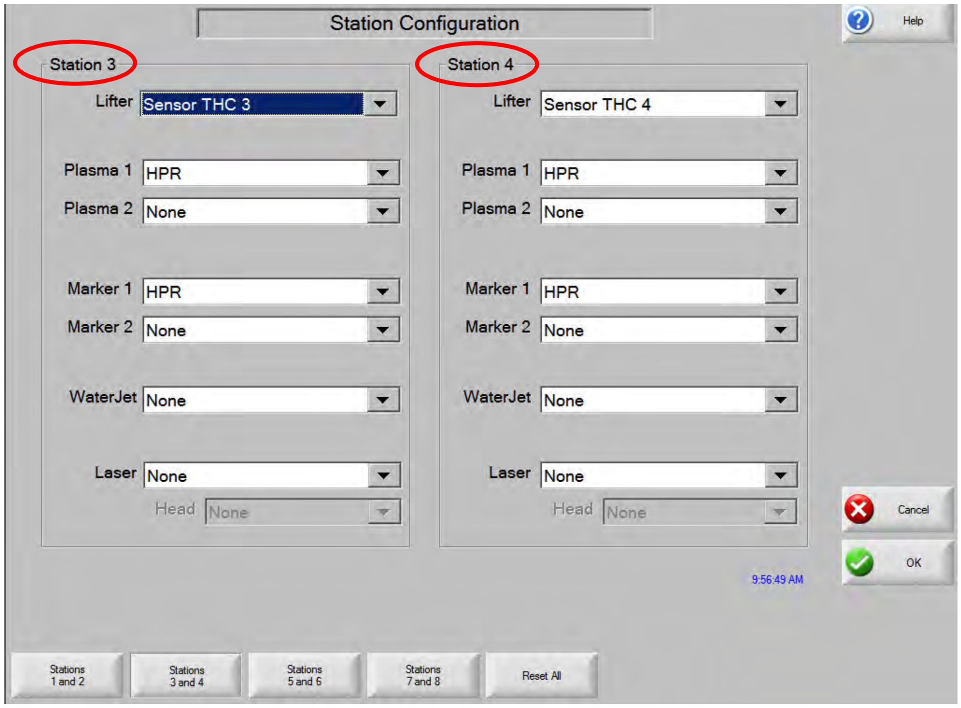
If all torches are always going to cut and mark with the same consumable sets, then the assignment is Plasma 1 and Marker 1 for all cutting tools.

Station 1 and Station 2

Choose this soft key to advance to the screen for Stations 3 and 4.



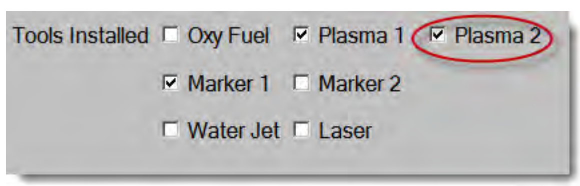
Station 3 and
Station 4



Sample settings for two-torch cutting system

To set up a two-torch cutting system that uses an HPR Auto Gas plasma supply with a Sensor THC and an HSD plasma supply with a Sensor PHC, follow these general steps on the CNC.

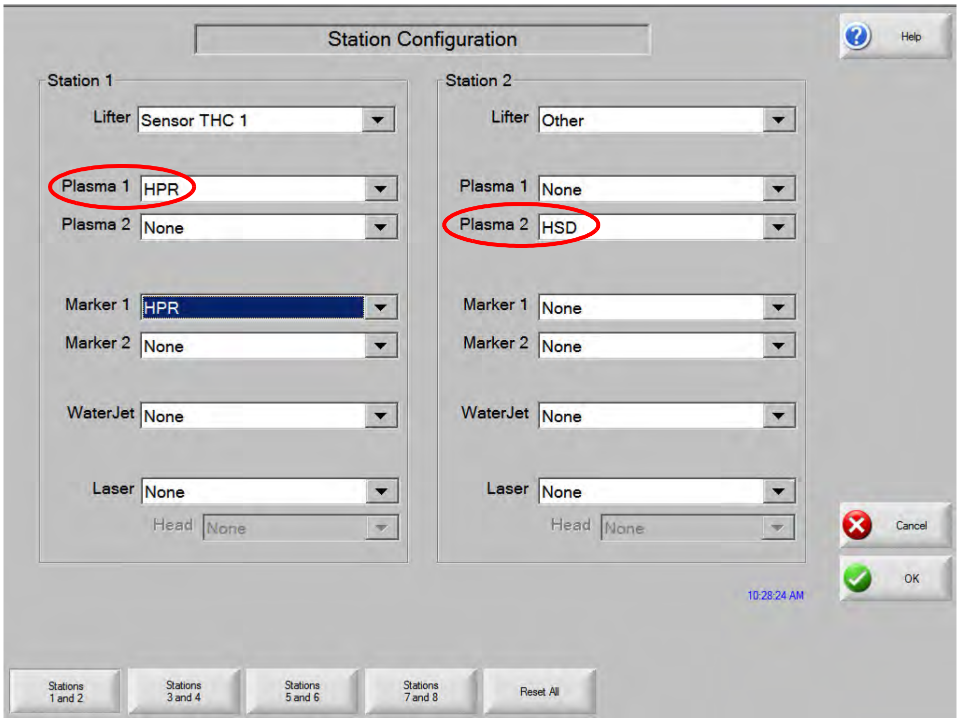
First, in the Special Setups screen, select Plasma 1 and Marker 1 *and* Plasma 2.



In the Station Configuration screen, both Plasma 1 and Plasma 2 are being used because the HPR and HSD plasma supplies use different cut processes and consumables. This is when the Plasma 2 selection is required.

Assigning the HSD on Station 2 and Plasma 2 provides the ability to use both the HPR torch and the HSD torch in the same part program.

Plasma 1 and Plasma 2 are required when the cutting system supports two different plasma supplies.



Plasma cut sequence

In most cutting systems, a torch height control (THC) manages the signals between the CNC and the plasma supply. The sequence that follows assumes a THC is part of the cutting system. When a signal is sent to the plasma supply, it typically passes through a THC, but the signal is the same.

Hypertherm CNCs execute the plasma cut sequence in these basic steps.

1. The CNC sends the Cut Control output signal to the plasma supply. Cut Control connects to the Start input on the plasma supply.
2. When the plasma supply transfers an arc, it sends the Motion output signal to the CNC. Motion is connected to the Cut Sense input on the CNC.
3. While waiting for the cut sense input to turn on, the CNC displays the message "Waiting for Arc On." When the Cut Sense input on the CNC turns on, the CNC initiates motion.

The Cut Control output and Cut Sense input are the minimum I/O signals needed in a plasma cutting system. The CNC provides additional I/O for torch height controls (THCs), for multiple-torch applications, and to enable features that reduce the cycle time between parts.

The following plasma sequence assumes the use of the Sensor THC. When using a torch height control from another manufacturer, the CNC provides a series of timers to control I/O to lower and raise the torch. For more information on using the timers for a torch height control, see *Section 7 Process setup* in the *Phoenix Operator's Manual* (806400).

The CNC identifies each state of the cut sequence with a status message below the part preview area of the Main screen.

The cut sequence begins when you press Cycle Start. At Cycle Start, the CNC begins reading and executing the part program.

State and Status message: *Lowering Torch*

Initiated when: The CNC reads the M07 code in the part program.

- The torch height control performs its initial height sense (IHS).
- Torch Height Disable output turns on and remains on until the cutting system reaches cutting speed.
- Cut Control and Hold Ignition turn on if Preflow During IHS is on in the Process screen.

State and Status Message: *Waiting for Arc On*

Initiated when: IHS completes.

- Cut Control output turns on. Cut Control activates the Plasma Start input to the plasma supply.
- Cut Control remains on until the CNC reads the M08 code (Cut Off) in the part program.
- The Pierce Control output turns on for an HPR plasma supply.

State and status message: *Piercing*

Initiated when: The plasma supply ignites an arc.

- The plasma supply turns on the Motion output.
- The Cut Sense input at the CNC turns on.
- The CNC delays motion until the Pierce Time set in the Process screen elapses.
- The Pierce Control output turns on until the Pierce Time elapses.

State and status message: *Creeping*

Initiated when: Creep motion begins.

- Creep motion continues until the Creep Time elapses. If Creep Time is not set in the Process screen, no creep motion will be used. Creep speed is a percentage of Cut Speed set on the Setups > Password > Machine Setups > Speeds screen.

State and status message: *Cutting*

Initiated when: Cutting system accelerates to Cut Speed.

- Torch Height Disable turns off after the cutting system reaches cutting speed.
- Torch Height Disable toggles on and off while cutting whenever the actual cut speed drops to a percentage below the set cut speed (the Torch Height Disable Speed that is set in the Speeds screen).

State and status message: *Raising Torch*

Initiated when: The CNC reads the M08 code (Cut Off) in the part program.

- Cut Control turns off, which turns off the Start input at the plasma supply.
- The Motion output at the plasma supply turns off, which turns off the Cut Sense input at the CNC.
    - For a non-HPR plasma supply, when Cut/Mark Sense input at the CNC turns off, it starts a retract delay timer and waits before retracting the torch because the arc may still be present.

- For an HPR plasma supply, when Cut/Mark Sense input at the CNC turns off, it immediately retracts the torch because the HPR has already extinguished the arc.

State and status message: *Stop Delay*

Initiated when: The Stop Time has been entered in the Process screen.

- A stop delay prevents the gantry from moving to the next pierce point until the Stop Time elapses. Setting the Stop Time ensures the torch clears any tip-ups before moving to the next pierce point.
- The THC raises the torch to the retract height.

State and status message: *Traversing*

Initiated when: The Stop Time elapses.

- The gantry moves to the next pierce point and the sequence repeats.

Setting up inputs and outputs for plasma

The following inputs and outputs are the minimum needed to control plasma cutting. You will also need to establish I/O for the cutting stations, hardware overtravel switches or other limit switches, and other cutting table peripherals. For cutting station I/O, see *Section 4 Station Setup*.

You can view the I/O assignments on the Diagnostics screen (Setups > Diagnostics > I/O).

Cut/Mark Sense / Cut Sense 1 – 20 This input notifies the CNC that the torch has transferred the arc to the workpiece. The input originates from the Motion output from the plasma supply or THC, and the CNC begins motion after this input activates.

Cut Sense 1 – 20 are the numbered versions of the Cut/Mark Sense input. In a multiple-torch cutting system, for example, you would assign the Cut Sense 1 and Cut Sense 2 inputs as part of the I/O for two torches.

Cut Control / Cut Control 1 – 20 This output activates the Start input of the torch height control or plasma supply. Cut Control turns on and remains on until the M08 (Cut Off) command is executed in the part program.

Pierce Control Activates during piercing and remains on until the Pierce Time elapses (set in the cut chart or Process screen). This output is used in some plasma supplies to keep the plasma supply from switching out of shield preflow while piercing. Pierce Control connects to the Pierce input at the plasma supply.

Torch Height Disable This output disables the THC while piercing or cutting corners or small holes. The output is used to disable an external THC from following arc voltage while cutting. The input at the THC is typically referred to as Corner Hold or Auto Height On/Off.

Hold Ignition Activates the IHS Sync input of the THC or the Hold input of a plasma supply. This output initiates Preflow During IHS. This output is also used in a multiple-torch cutting system to synchronize the ignition of all torches.

The following inputs and outputs are frequently used in multiple-torch cutting systems, or when using a torch height control or lifter from another manufacturer. These I/O settings are not needed for the Sensor THC.

Ready to Fire 1 – 20 Numbered I/O used only in multiple-torch systems to synchronize the ignition of all torches. This input connects to the IHS Complete output on the THC.

Lower Torch / Lower All Torches Activates the Torch Down output while it is on. The Lower Torch inputs can be connected to manual toggle switches on the operator console.

Raise Torch / Raise All Torches Activates the Torch Up output while it is on. The Raise Torch inputs can be connected to manual toggle switches on the operator console.

Torch Down Sense When this input turns on, the CNC turns off the Torch Down output. This input is not used with the Sensor THC.

Torch Up Sense When this input turns on, the CNC turns off the Torch Up output. This input is used as an upper limit switch for a torch lifter. This input is not used with the Sensor THC.

Torch Down This output signals the cutting station to lower the torch. Use the Torch Down Time on the Process screen to specify the number of seconds for the torch to be lowered. The Torch Down output turns off before the Torch Down Time elapses if the Torch Down Sense input activates. This output is not used with the Sensor THC.

Torch Up This output signals the cutting station to raise the torch. Use the Full Torch Up Time or Partial Torch Up Time on the Process screen to specify the number of seconds to raise the torch. The Torch up output turns off before the Full or Partial Torch Up Time elapses if the Torch Up Sense input activates. This output is not used with the Sensor THC.

Summary: setting up the plasma routine

1. Select the Plasma 1 and Plasma 2 processes in the Special Setups screen.
2. Select a marking process if the plasma torch will be used as a marker.
3. Assign the inputs and outputs for the plasma supply on the I/O setup screen.
   - ❑ The I/O assignments for the plasma supply depend on the number of torches and whether the CNC will be operating all the torches independently.
   - ❑ For the CNC to operate each torch independently, numbered outputs and station select inputs must be assigned in the I/O setup screen. Refer to *Section 4 Station Setup* for numbered I/O and Station Select inputs.
4. Define timer settings and other cutting parameters in the Process screen.

I/O and diagnostics

After serial link communication has been established between the power supply and the CNC, I/O and remote diagnostics screens are accessible through the standard Diagnostic screen. You can view status for the plasma supply software revision, gas pressure, usage, I/O and remote tools.

Status items for the supply are available on the Information screen.

HPR diagnostics

Test Preflow: Tests the preflow gases at the power supply. This feature is used to set the inlet gas pressures under normal flow conditions to the recommended level.

Test Cutflow: Tests the cutflow gases at the power supply. This feature is used to set the inlet gas pressures under normal flow conditions to the recommended level.

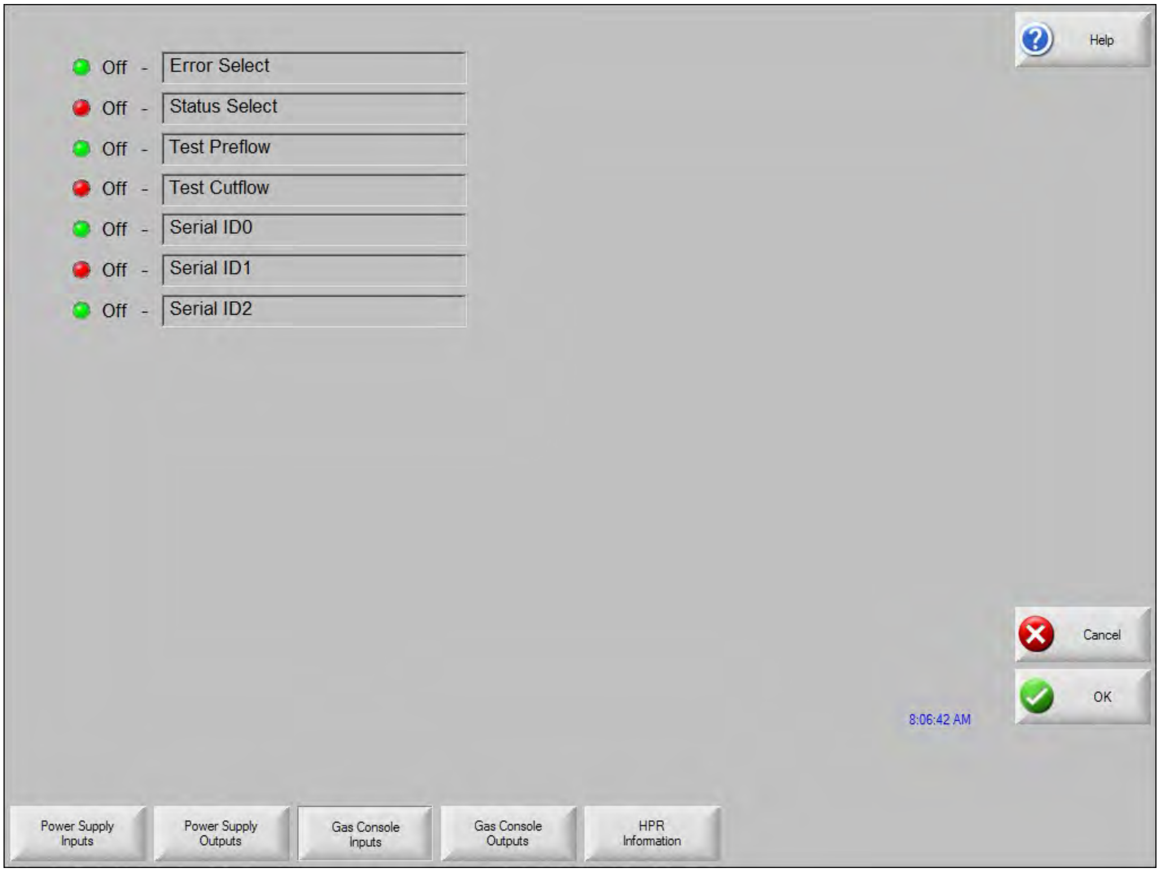
Test HPR Gas Console: Performs automated tests for the HPR Auto Gas console. Contact an authorized service agent for use of these tests.

Coolant Override: Overrides a coolant error and tests the coolant pump. This is useful for bypassing the error and purging the coolant line of air bubbles at initial power up.

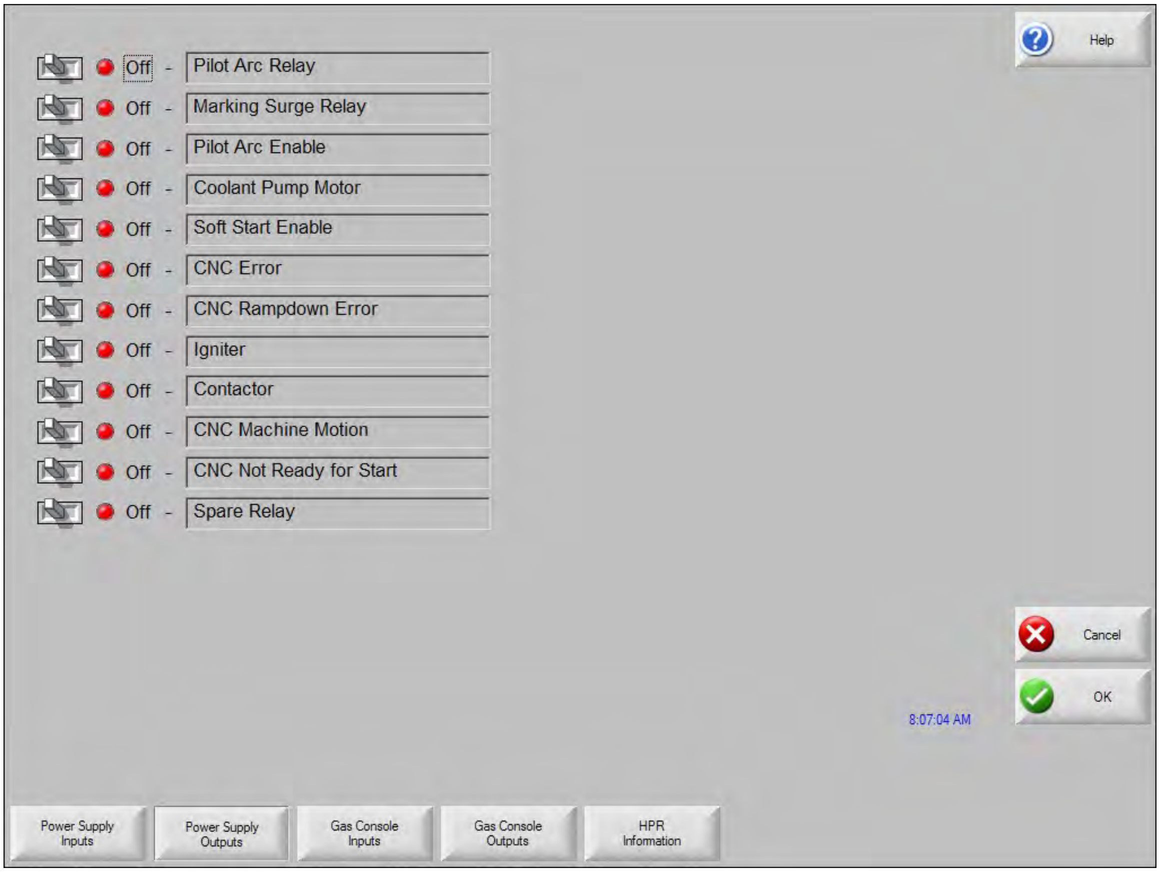
Update Software: Initiates the software update at the power supply. Step-by-step instructions are displayed on screen to guide you through the process.

Power Supply Inputs

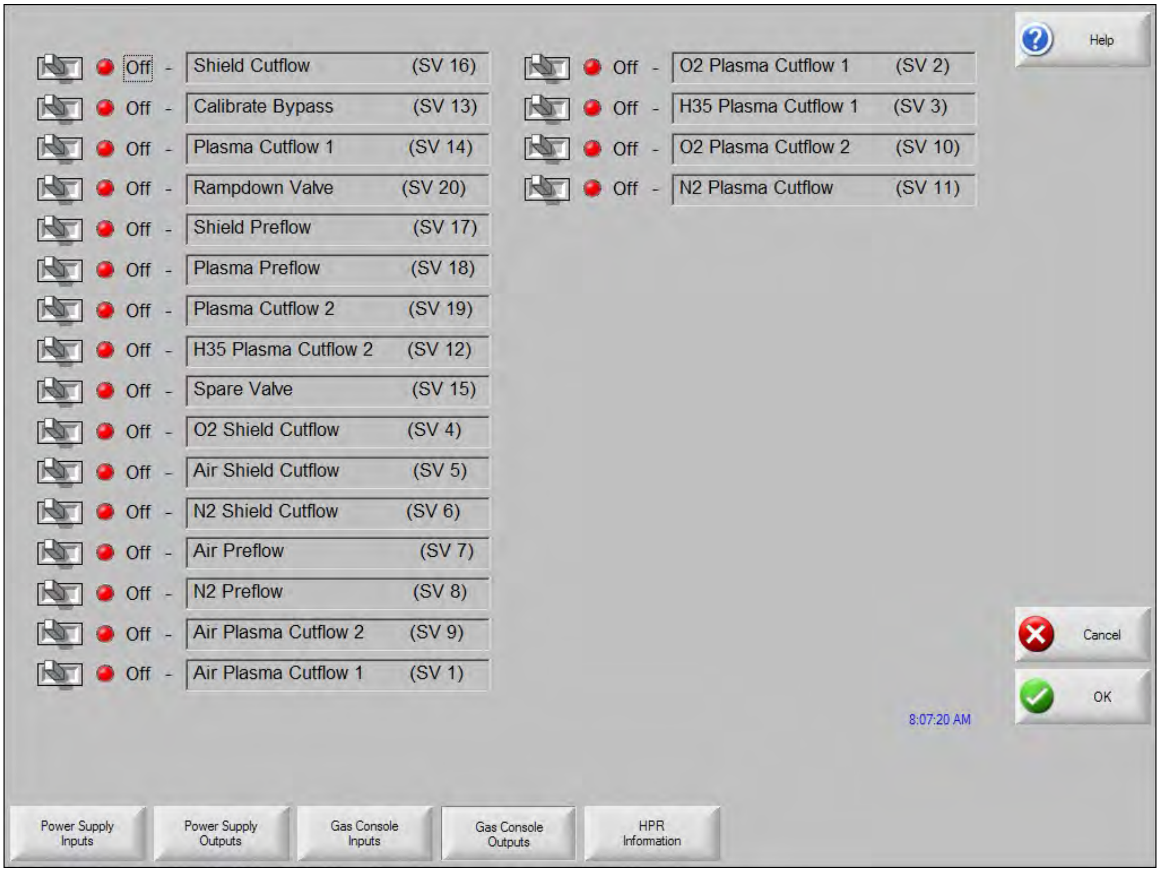
Gas Console Inputs



Power Supply Outputs



Gas Console Outputs



Notes:

- The output screen shows the current status of the listed supply outputs.
- Outputs for the supply cannot be activated through the diagnostic screen.

Serial communication interface

RS-422C connections to HPR CNC interface

Control (male)		Supply (male)	
Signal name	**DB-9 pin**	**Signal name**	**DB-37 pin**
TxD-	2	RxD-	1
RxD-	3	TxD-	2
TxD+	4	RxD+	20
RxD+	7	TxD+	21

Powermax plasma supply

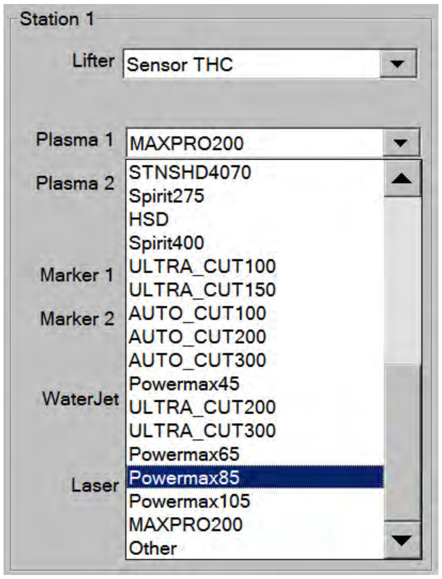
Hypertherm CNCs support a serial connection to the Powermax65®, Powermax85®, and Powermax105®. From the CNC you can set cut mode (normal cutting or continuous pilot arc), current, and gas pressure. A diagnostic screen helps you monitor the status of the Powermax system.

Setting up a Powermax system in Phoenix requires the following steps:

- Select the Powermax65 or Powermax85 in the Station Configuration screen.
- Assign the Powermax to a serial port.
- Assign I/O for the Powermax.

Selecting the Powermax in the Station Configuration screen

1. Choose Setups > Password and enter your password to open the Station Configuration screen.
2. For Station 1 (or the station you want to set up for the Powermax), choose your Powermax model from the drop-down list.



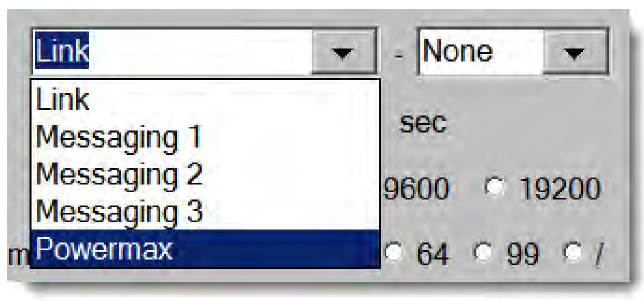
3. Complete additional selections for your cutting system and choose OK to exit, then choose OK again to save changes.

Assigning the Powermax to a serial port

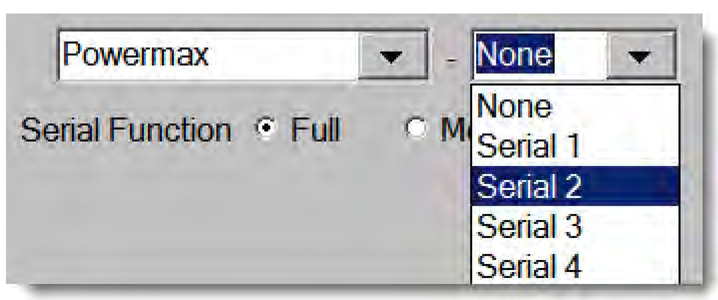
The CNC communicates with the Powermax through a serial port and RS-485 communications. Consult the operator manual for you CNC for the correct cabling to the serial port.

Note: Before connecting the Powermax to the Hypertherm EDGE Pro CNC, be sure to set the serial isolation board to RS-422. Refer to the *EDGE Pro Shape Cutting Control Instruction Manual* (806360) for more information.

1. Choose Setups > Password and enter your password to open the Machine Setups screen.
2. Choose the Ports soft key.
3. In the first drop-down list, choose the down arrow and scroll until you see Powermax, then select it.



4. Choose the serial port to which you have connected the Powermax.



5. For Serial Function, choose Full or Monitor.
   - **Full** mode allows the CNC to control the cut mode, current, and gas pressure. Choosing Full mode disables the front panel controls on the Powermax. The CNC also reads and reports information about the Powermax in the Diagnostics screen.
   - **Monitor** mode allows the CNC to read and report the information in the Diagnostics screen (described later in this section). The front panel controls remain active on the Powermax.
6. Choose OK to save the settings and exit the screen.

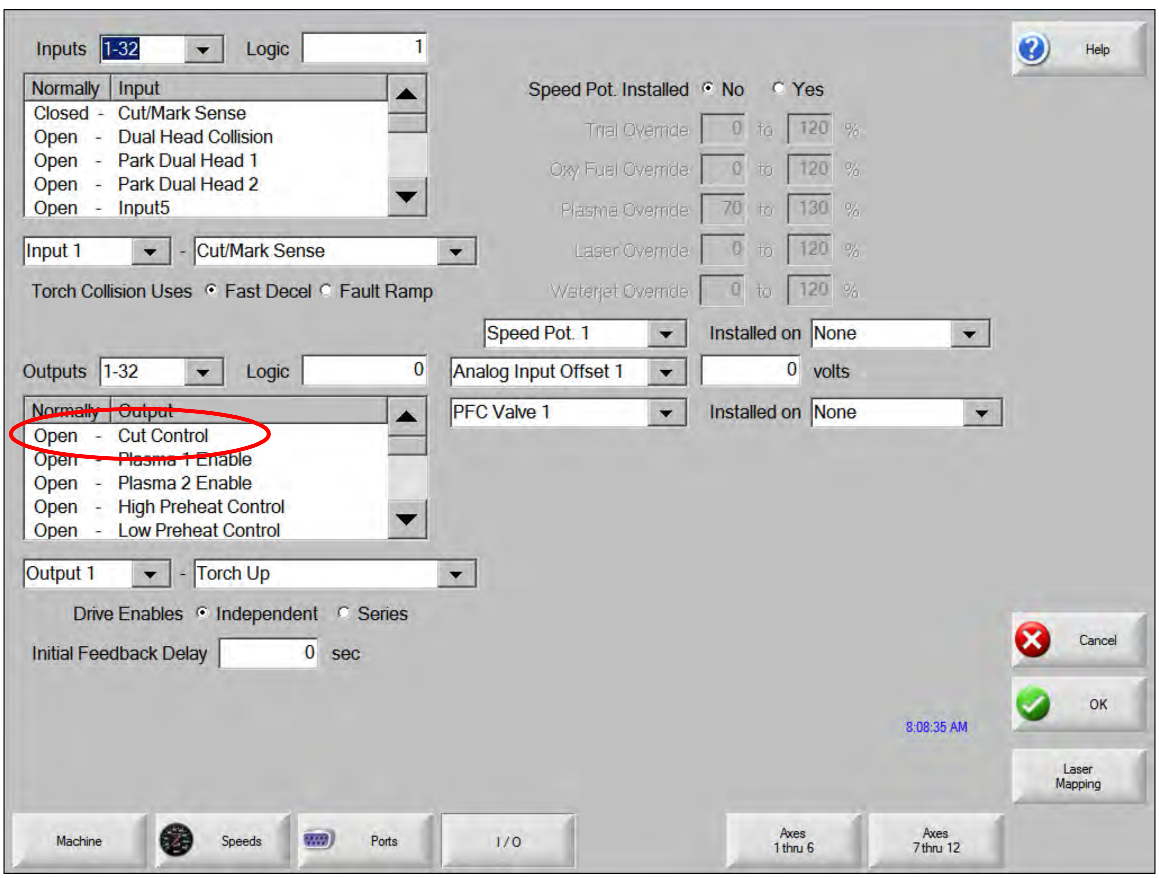
I/O selection for the Powermax

The CNC can control the Powermax signals for Start Plasma and Transfer (start machine motion). The table below shows the Powermax signals and the input and output to which the signals are assigned.

Powermax signal	Function	Input/output in Phoenix
Transfer (start machine motion)	Cut/Mark Sense	Input
Start Plasma	Cut Control	Output

1. Choose Setups > Machine Setups > I/O.
2. Under Inputs, choose the input number to which the Powermax Transfer (start machine motion) signal is wired.
3. Choose Cut/Mark Sense as the function for the input.
4. Under Outputs, choose the output number to which the Powermax Start Plasma signal is wired.
5. Choose Cut Control as the function for the output.
6. Choose OK to save changes and exit the screen.

   Note: Refer to the instruction manual provided with your plasma system for more information on wiring inputs and outputs.

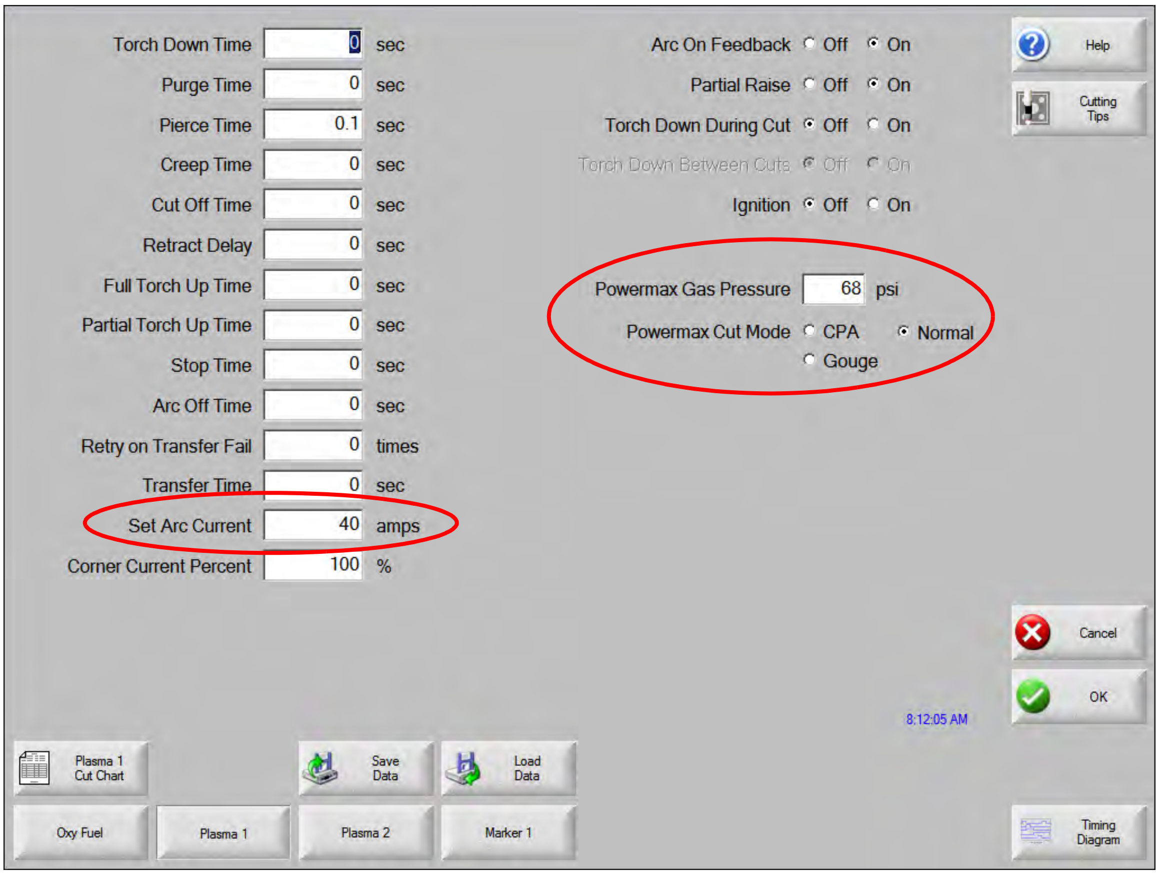


Setting cut mode, gas pressure, and current from the CNC

Once the Powermax is successfully connected to the CNC and set up in the Phoenix software, you can control the cut mode, gas pressure, and current from the Process screen in Phoenix.

To open the Process screen, choose Setups > Process.

To view the Process screen for the Powermax, the Powermax needs to be assigned to Plasma 1 or 2 in the Station Configuration screen. In the screen shown below, the Powermax is assigned as Plasma 1, so the Process screen appears when you choose the Process soft key.



You can adjust the settings for Arc Current, Gas Pressure, and Powermax Mode (Continuous Pilot Arc, Normal, or Gouge cutting) as needed for the operation. Any changes you make to this screen will be applied when you choose OK to exit the screen.

Powermax Diagnostic screen

The Powermax Diagnostic screen reports the status of several Powermax settings. Choose Setups > Diagnostics, then choose the Powermax soft key.

Cut Mode: Shows the cut mode set by the CNC and sent to the Powermax (Normal, Continuous Pilot Arc [CPA], or Gouge).

Set Arc Current: Shows the current level set by the CNC and sent to the Powermax.

Gas Pressure: Shows the gas pressure set by the CNC and sent to the Powermax. The CNC uses the gas pressure from the cut chart or the part program.

Torch Lead Length: The CNC uses the torch lead length to determine the correct range for the gas pressure. The gas pressure and lead lengths are stored in the Powermax65 and Powermax85 cut charts.

Control/DSP Revision: The firmware in the Powermax consists of two parts: the first is the control firmware and the second is the DSP.

Arc On Time: The time that the Powermax has been on and producing an arc.

AC Input Voltage: The supply voltage as measured by the Powermax sensors.

DC Buss Voltage: The internal DC voltage as measured by the Powermax sensors.

Last Fault: Shows either an operating fault or a system fault. The Powermax reports only system faults in the Fault Log. Most operating faults clear without operator intervention. For example, Low Gas Pressure, an operating fault, clears once gas pressure is restored.

Fault Log: Shows the four most recent system faults with a description and the Arc Time counter value at the time the error occurred.

MAXPRO200 plasma supply

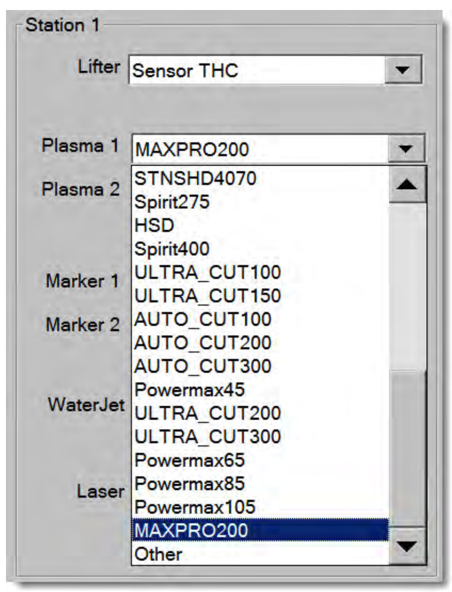
Hypertherm CNCs support a serial connection to the MAXPRO200 plasma supply. From the CNC you can set the gas pressure, and a diagnostic screen helps you monitor the status of the MAXPRO200.

Setting up a MAXPRO200 on the CNC requires the following steps:

- Select the MAXPRO200 in the Station Configuration screen.
- Assign the MAXPRO200 to a serial port.
- Assign I/O for the MAXPRO200.

Selecting the MAXPRO200 in the Station Configuration screen

1. Choose Setups > Password and enter your password to open the Station Configuration screen.
2. For Station 1 (or the station you want to set up for the MAXPRO200), choose MAXPRO200 from the Plasma 1 or Plasma 2 drop-down list.



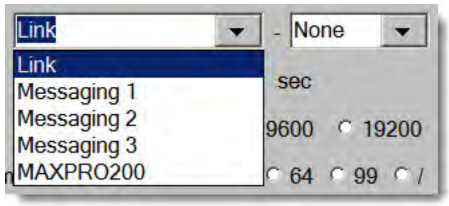
3. Complete additional selections for this station on your cutting system and choose OK to exit, then choose OK again to save changes.

Assigning the MAXPRO200 to a serial port

The CNC communicates with the MAXPRO200 through a serial port and RS-422 communications. Consult the operator manual for your CNC for the correct cabling to the serial port.

Note: Before connecting the MAXPRO200 to the Hypertherm EDGE Pro CNC, be sure to set the serial isolation board to RS-422. Refer to the *EDGE Pro Shape Cutting Control Instruction Manual* (806360) for more information.

1. Choose Setups > Password and enter your password to open the Machine Setups screen.
2. Choose the Ports soft key.
3. Choose MAXPRO200 from the left-most drop-down list at the top of the screen.



4. Choose the serial port for the MAXPRO200 in the right-most drop-down list.

5. Choose OK to save the settings and exit the screen.

I/O selection for the MAXPRO200

The following table shows the MAXPRO200 signals and the input and output to which the signals are assigned.

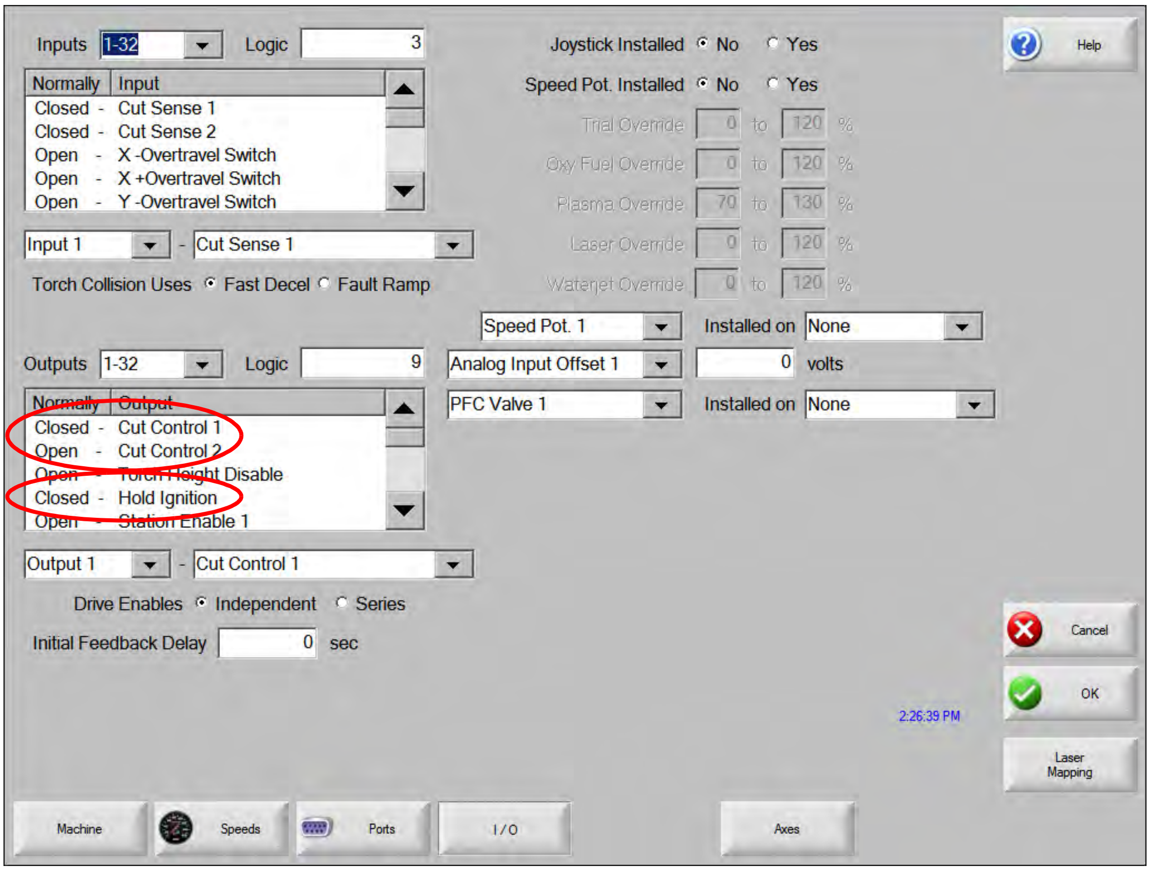
MAXPRO200 signal	Phoenix I/O function
Plasma start	Cut Sense 1 or 2 input
Machine motion	Cut Control 1 or 2 output
Hold ignition	Hold Ignition output
System errors	Not used by Hypertherm CNCs. MAXPRO200 communicates errors to the CNC through the serial port.

Phoenix can control the MAXPRO200 signals for start input, motion output, and hold ignition, as follows:

1. Choose Setups > Machine Setups > I/O.
2. Under Inputs, choose the input number to which the MAXPRO200 plasma start signal is wired.
3. Choose Cut Sense 1 (or Cut Sense 2) as the function for the input. This signal indicates the plasma system will fire an arc.
4. Under Outputs, choose the output number to which the MAXPRO200 machine motion signal is wired.

5. Choose the function for the output, as follows:
   - Choose Cut Control 1 (or Cut Control 2) to indicate the arc has transferred to the workpiece being cut.
   - Choose Hold Ignition to synchronize two or more torches on a single cutting table. For example, if you have two torches, Phoenix will apply the start and hold ignition at the same time to both systems. Once both torches have completed the IHS and are ready to fire, you should remove the hold signal from both systems.
6. Choose OK to save changes and exit the screen.

   Note: Refer to the instruction manual provided with your plasma system for more information on wiring inputs and outputs.



System error output

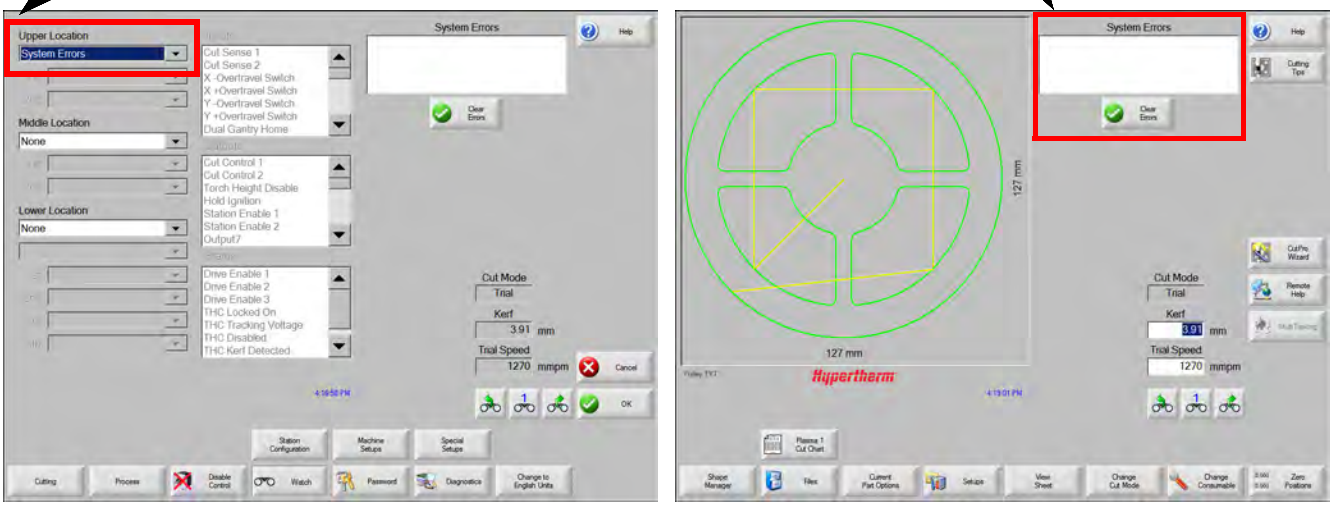
When the MAXPRO200 generates a system error, the error code displays on the interface of the MAXPRO200 itself. The system also communicates the error message to Phoenix through a serial connection. The error message may then display on the status line in Phoenix; however, because the status line can display only one message at a time, you may not see the error message for every system error that occurs. To ensure that you can see every MAXPRO200 error message in Phoenix, you can add system errors to a Watch Window.

Watch Window setup for the MAXPRO200

The Watch Window is the configurable part of the main screen that enables you to display up to 10 different monitoring setups for more convenient access to specific process-related parameters during cutting. Adding System Errors to the Upper Location or Middle Location of a Watch Window configuration enables you to see error messages in Phoenix when cutting with the MAXPRO200.

Add System Errors to the Watch Window setup here...

...to have them display on the Main screen here.

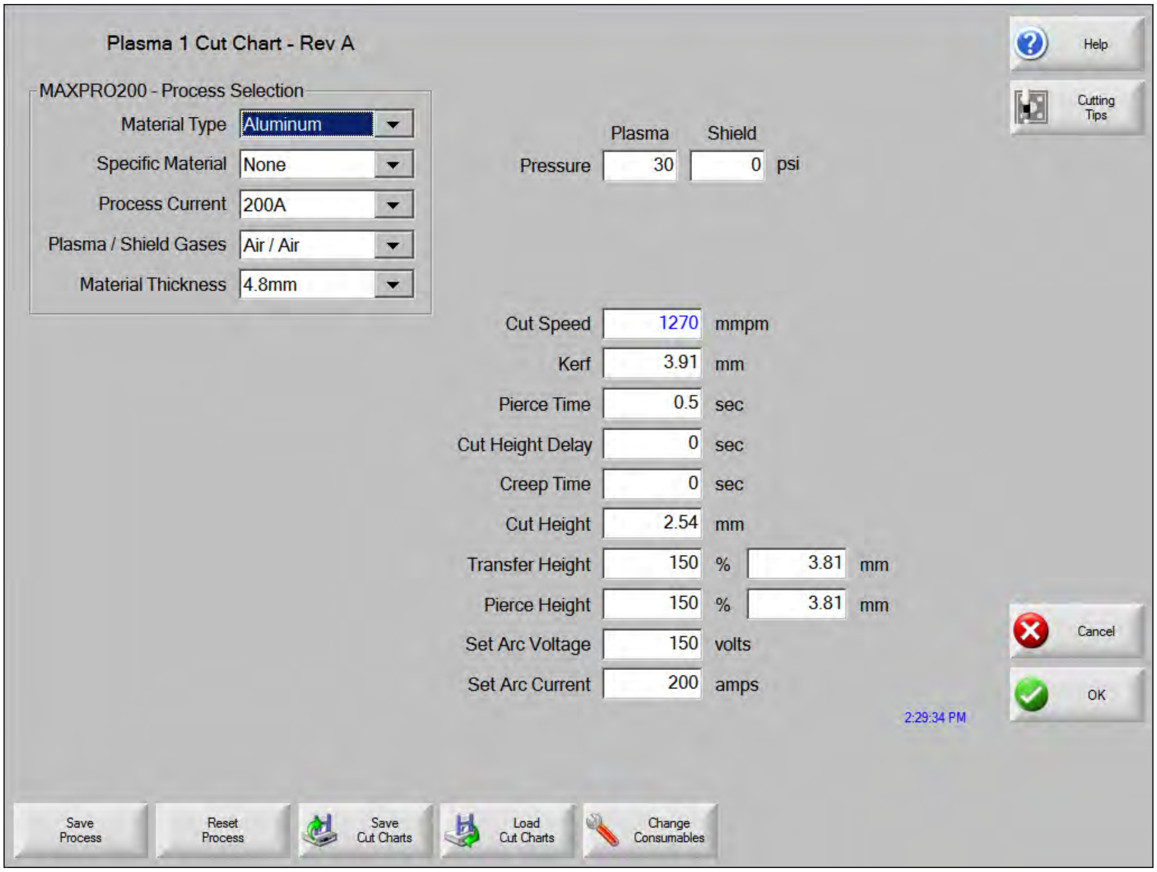


You can select the Watch soft key on the Setups screen to select the parameters you want to appear in the Watch Window. Refer to the “Cutting screen and Watch Window setup” section in the *Phoenix Operator’s Manual* (806400) for instructions on how to select your Watch Window parameters. Your Watch Window selections will vary slightly based on the CNC I/O configuration and machine options you have enabled.

Adjusting default cut chart settings from the CNC

After the MAXPRO200 is connected to the CNC and set up in the Phoenix software, you can control certain parameters from the MAXPRO200 cut chart, namely the current or the gas pressure, if you need to override the default values.

Choose the Plasma 1 or 2 Cut Chart soft key on the Main screen to open the cut chart.

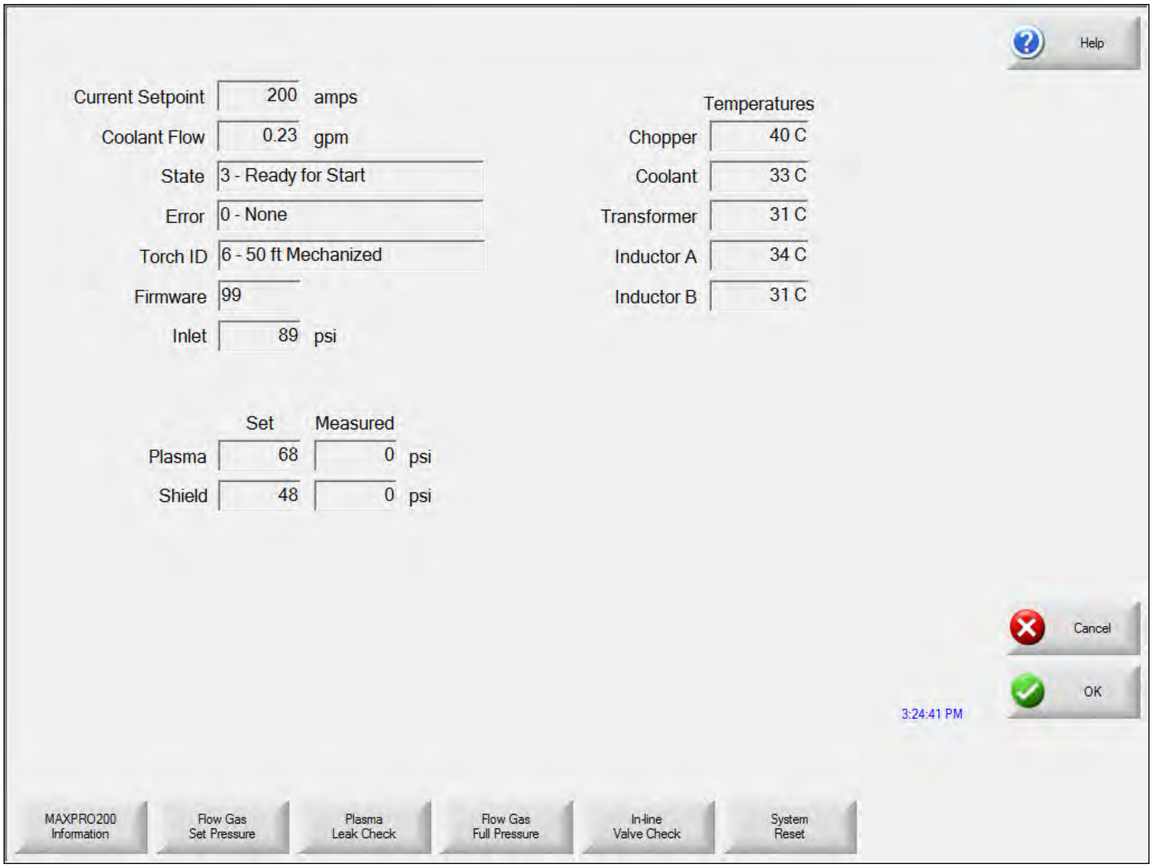


The cut chart processes you can select in Phoenix have been developed and tested to produce the greatest balance between cut quality and productivity. If you need to override these default values, you can enter or select the new values from the MAXPRO200 cut chart screen. Any changes you make to this screen will be applied when you choose OK to exit the screen.

The gas pressure values in each cut chart for the MAXPRO200 are based on the detected lead length currently in use. The default gas pressures will vary slightly for each process depending on your system's lead length.

MAXPRO200 Diagnostic screen

The MAXPRO200 Diagnostic screen reports the status of several MAXPRO200 conditions and enables you to perform certain diagnostic functions to aid troubleshooting. To display the screen, choose Setups > Diagnostics, then choose the MAXPRO200 soft key.



MAXPRO200 Information: Default range of system status settings that displays on the Diagnostic screen to help with troubleshooting. Use the other soft keys on the screen to start (or stop) certain diagnostic modes or to reset the system.

Current Setpoint: Amperage set for the MAXPRO200 power supply.

Coolant Flow: Rate at which the coolant is flowing.

State: The currently active state of the MAXPRO200 power supply.

Error: Code and description identifying the system error that has occurred, if any. For a more detailed description of each error and for possible corrective actions you can take, refer to the Troubleshooting table in the "Maintenance" section of the *MAXPRO200 Instruction Manual* (807700).

Torch ID: Number and description identifying the combination of lead length and torch type installed.

Firmware: Version of the firmware installed on the MAXPRO200 power supply.

Inlet: Initial measured inlet gas pressure.

Temperatures: Current temperature readings for the chopper, coolant, transformer, and inductors. If any one of these temperatures exceeds a maximum threshold, it will display in red. When this occurs, the plasma power supply cannot be operated until the condition is resolved.

Plasma: Plasma gas pressure. The Set value shows the gas pressure reported by the power supply. The Measured value displays as zero by default, but you can watch the values in this field to monitor the plasma gas pressure when you activate the diagnostic modes on the screen.

Shield: Shield gas pressure. The Set value shows the gas pressure reported by the power supply. The Measured value displays as zero by default, but you can watch the values in this field to monitor the shield gas pressure when you activate the diagnostic modes on the screen.

Flow Gas Set Pressure: Diagnostic mode used to determine if the gas pressure set for the power supply can be achieved and maintained. Select this soft key to activate the diagnostic mode, then select it again to deactivate it.

You can also select the Cancel or OK soft key at any time to end the current diagnostic mode and exit the Diagnostic screen.

Plasma Leak Check: Diagnostic mode used to determine if the valve in the plasma line is functioning properly to trap the gas in the line and maintain a steady pressure. Select this soft key to activate the diagnostic mode, then select it again to deactivate it.

Flow Gas Full Pressure: Diagnostic mode used to determine the highest gas pressure that can be maintained. Select this soft key to activate the diagnostic mode, then select it again to deactivate it.

For the Flow Gas Set Pressure and Flow Gas Full Pressure diagnostic modes, gas will continue to flow until you stop the diagnostic mode.

In-line Valve Check: Diagnostic mode used to determine if the valve in the plasma line is opening and closing properly and is allowing the gas to exit the line. Select this soft key to activate the diagnostic mode, then select it again to deactivate it.

System Reset: Reset key used to reset the power supply system, if needed.

For a more detailed explanation of the diagnostic modes on this screen and for possible corrective actions, refer to the “Operation” section in the *MAXPRO200 Instruction Manual* (807700).

Section 8

Diagnostics

Remote Help

Contact Hypertherm Technical Support or you OEM or system integrator for assistance with Remote Help.

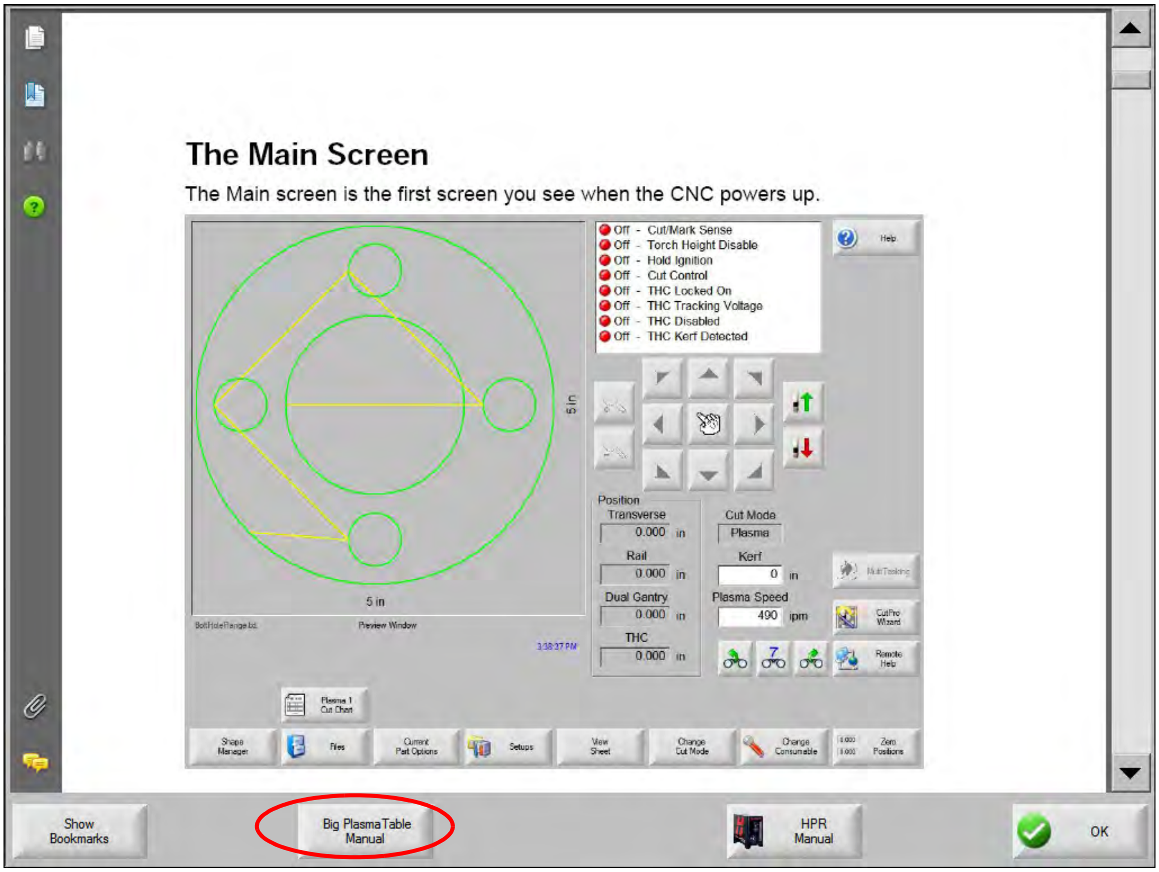
Load Additional Manuals

Table manufacturers can load manuals for their equipment on the CNC and make these manuals available to Phoenix users on the Help screen.

To add a manual:

1. Press Alt+F4 to exit Phoenix.
2. In Windows, open Windows Explorer.
3. Copy the .pdf version of your manual onto the hard drive in the following location: c:\PHOENIX\Help\OEM*filename*.pdf.
   - The first three letters in the file name must be OEM for example, c:\PHOENIX\Help\OEM Plasma Table.pdf.
   - The file name is displayed on a custom soft key and the word Manual is automatically appended to the file name.
4. When the user presses the Help button on any screen, a button with the name of the manual you loaded (Plasma Table Manual) displays at the bottom of the Help screen.
5. Press the button with the manual title to view this manual.

6. Repeat this procedure for each manual you want to add to the Help screen.



EDGE Pro Machine Interface Tests

The Phoenix software provides a series of diagnostic tests for the Hypertherm EDGE Pro CNC. These tests require the following test hardware:

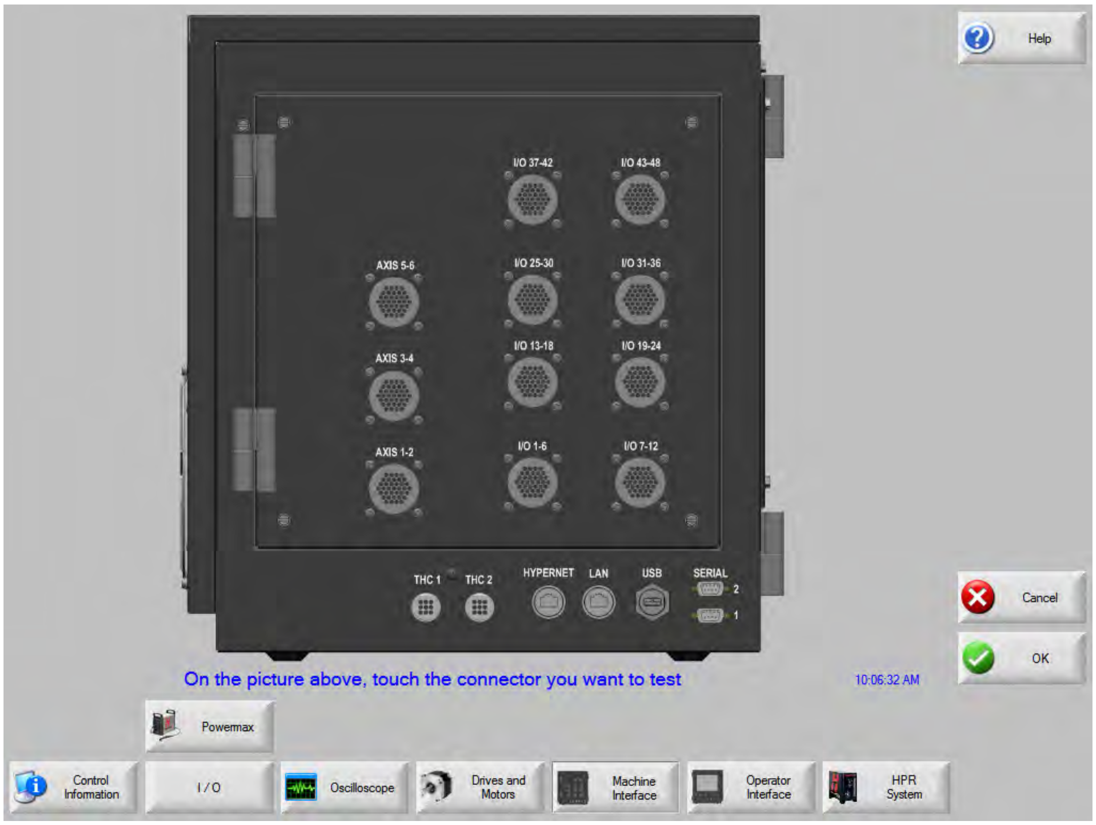
Connector test hardware	Kit part number
Serial port	228512
USB port	228512
LAN/HyperNet	228512
Picopath I/O – 1st tester (orange stripe) Picopath I/O – 2nd tester (white stripe)	228510
Picopath axes (blue)	228510
Picopath THC (yellow)	228510
HyPath I/O – 1st tester (green stripe) HyPath I/O – 2nd tester (red stripe)	228511
HyPath axes (green)	228511
HyPath THC (blue)	228511

Note:

- 228510 and 228511 both include the serial, USB, and LAN/Hypernet testers.
- If you are using the keyboard-only interface on the CNC (No Touchscreen Installed is selected in the Special Setups screen) you cannot perform the machine interface tests as described in this section unless you have a mouse installed on one of the USB ports.

To begin an interface test:

1. From the Main screen, choose Setups > Diagnostics > Machine Interface.
2. Enter the Machine password.
3. On the Machine Interface screen, press the port you want to test on the image of the CNC.
4. Follow the instructions in the sections below and on the screen.



Serial Test

The EDGE Pro is equipped with two serial ports that support RS-422 or RS-232 communications.

Conduct this test if:

- Processes or information communicated through the serial port are not operating properly.
- The CNC is unable to download files through the serial port.
- The serial link to the plasma supply fails.

To test serial communications:

1. On the Machine Interface screen, press the serial port you want to test.
2. On the CNC, plug the serial test hardware into the port you selected.

3. On the Machine Interface screen, press Test. A message informs you if the test is successful.

If the test fails, contact your OEM or system integrator to replace the serial isolation board.

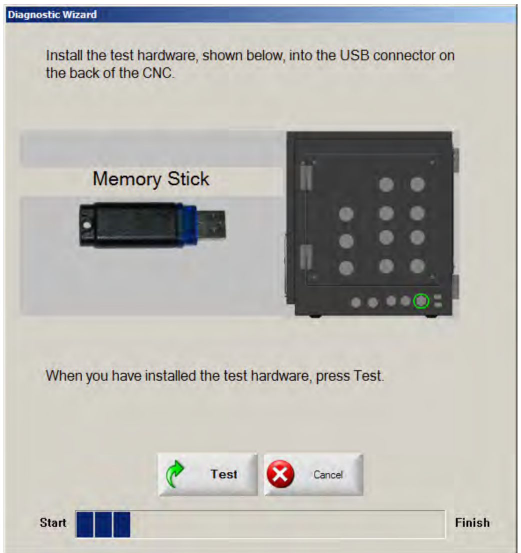
USB Test

Conduct this test if the memory stick is not shown as an option when you try to load part programs or update cut charts, software, or Help.

To test the USB port:

1. On the Machine Interface screen, press the USB port.

2. Follow the instructions on the screen to insert the memory stick in the USB port on the back of the CNC.



3. On the Machine Interface screen, press Test. A message informs you if the test is successful.

If the test fails, contact your OEM or system integrator to replace the motherboard.

I/O Test

Conduct this test if:

- An I/O point is malfunctioning.
- You need to eliminate CNC I/O operation as a problem in the system.
- A continuous fault is occurring, such as a limit switch that is not turning on or clearing.

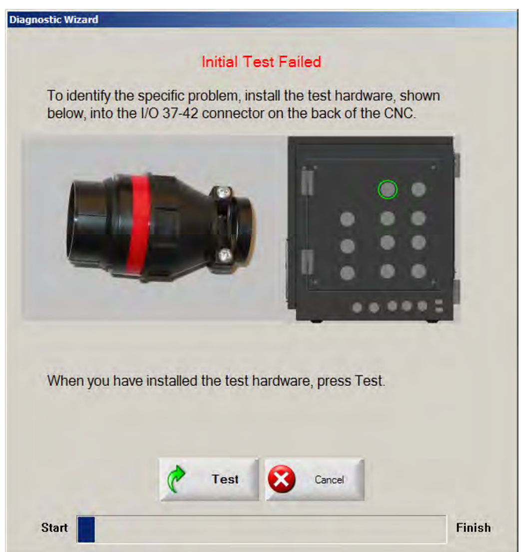
To test I/O:

1. On the Machine Interface screen, press the I/O port you want to test.

2. Follow the instructions on the screen to connect the HyPath (green) or Picopath (orange) test plug to the I/O port you selected on the back of the CNC.

3. On the Machine Interface screen, press Test. A message informs you if the test is successful.

4. If the test fails, follow the instructions on the screen to connect the test HyPath (red) or Picopath (white) test plug to isolate the input or output point that is causing the problem.



5. On the Machine Interface screen, press Test. The screen reports the number of the failed input or output.

- Picopath: Rewire the I/O to a few I/O point, then reassign the input function in the phoenix software.
- Hypath:
  - If an input has failed, rewire the input to a free input point, then reassign the input function in the Phoenix software.
  - If an output has failed, you can remove a relay from a spare output and swap it with the relay at the failed output. No rewiring is required.

Axis Test

> **WARNING**
>
> **To prevent motion on the table, disconnect all the axis cables from the CNC before conducting this test.**

Conduct this test if there is:

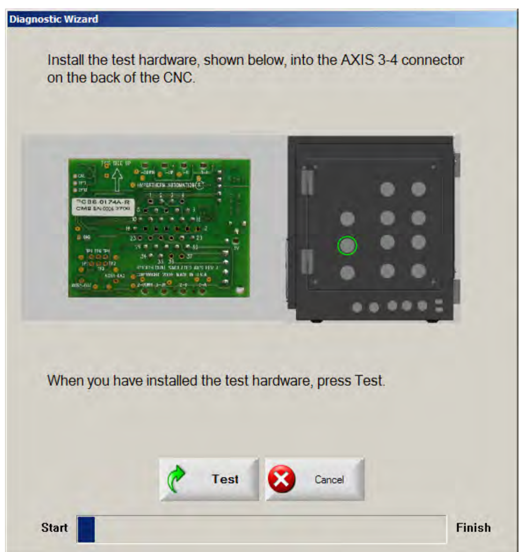
- A runaway axis
- No motion
- Erratic motion
- A high number of position errors
- Multiple parts are not cut the correct size

For all axes to pass, the following conditions need to be met.

- You can only test the number of axes that are enabled on the hardware key (axes installed in Control Information kit).
  - If you have a 2-axis CNC, you still have 4 or 6 physical axes
  - The test will not allow you to test the other axes. Nothing will happen when you click on the 3rd/4th axis connector for example.
- Axes are tested by their assignment in Phoenix.
  - Axis switching, like Dual Transverse, CBH if no 3rd axis, or Rotate/Tilt on a 4 axis CNC, will fail the axis diagnostic test.
  - If you are not using the 3rd or 4th axis, or have an assignment that puts the axis at 5 and higher in the application, you can test the hardware by temporarily assigning the 3rd and 4th axes to Sensor THCs.

To test an axis:

1. On the Machine Interface screen, press the axis port you want to test.
2. Follow the instructions on the screen to connect the Hypath (green) or Picopath (blue) axis simulator board to the CNC at the axis port you selected.



3. On the Machine Interface screen, press Test. A message informs you if the test is successful.

If the test fails, contact your OEM or system integrator to check or replace these parts in the following order:

1. Servo field interface
2. Ribbon cable
3. MCC card

THC Test

⚠	**WARNING** **To prevent motion on the table, disconnect all axis cables from the CNC before conducting this test.**

Conduct this test if:

- Nozzle contact is not functioning properly. For example, IHS using ohmic contact is not sensing the workpiece or is not accurate, the torch is running into the workpiece during cutting without retracting, or the torch is firing in the air.
- Arc voltage feedback is not functioning properly.
- The torch is rising off the workpiece or driving into the workpiece during the first part of a cut after piercing.

To test a THC port:

1. On the Machine Interface screen, press the THC port you want to test.
2. Follow the instructions on the screen to connect the THC test cable to the THC port you selected and to the Axis 1-2 port (Hypath) or the X/Y port (Picopath). Use the blue tester for Hypath and the yellow tester (not shown) for Picopath.

3. On the Machine Interface screen, press Test. A message informs you if the test is successful.

If the test fails, contact your OEM or system integrator check or replace these parts in the following order:

1. Analog field interface card
2. Ribbon cable
3. Analog input card

LAN and Hypernet Tests

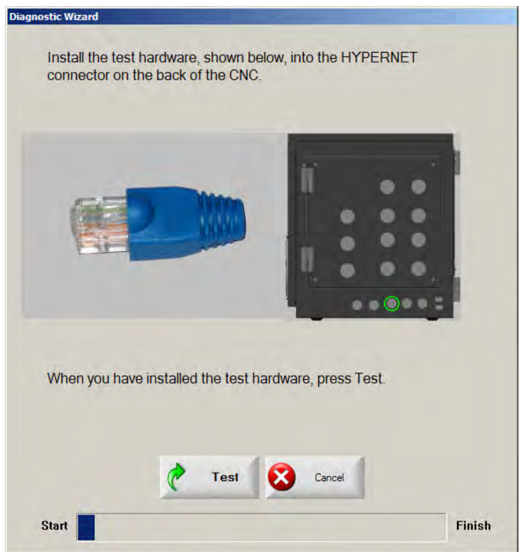
The LAN and Hypernet tests both use the same tester.

Conduct this test if:

- The CNC is not communicating to the ArcGlide or plasma system.
- The CNC is not communicating to the local area network.

To test a LAN port:

1. On the Machine Interface screen, choose the LAN or Hypernet port.
2. Follow the instructions on the screen to plug in the tester.



3. On the Machine Interface screen, choose Test. A message informs you if the test is successful.

If either test fails, contact your OEM or system integrator to replace the motherboard.

Operator Interface Test

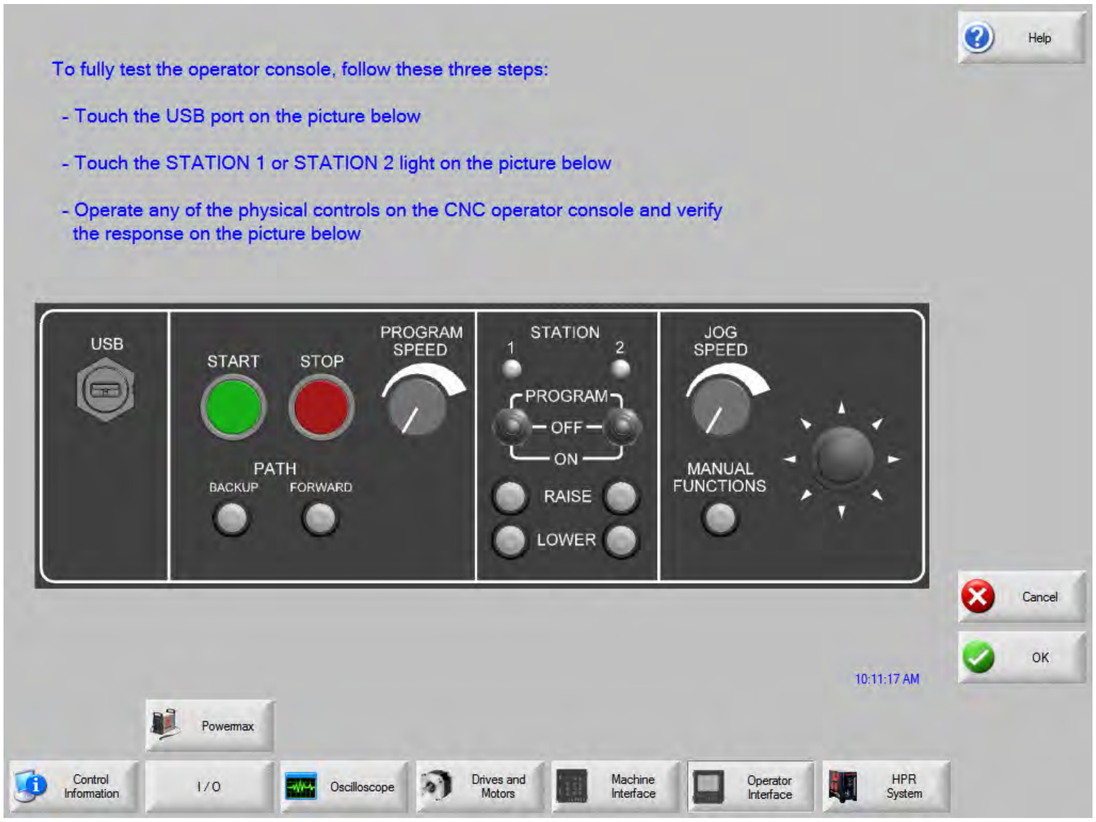
The operator interface test verifies that the front panel of an EDGE Pro CNC is operating correctly.

Perform this test if any function on the integrated operator console is not working as expected.

> **Note:** None of the components on the screen allow motion or any action to take place on the CNC or cutting table.

To test the operator console:

1. From the Main screen, select Setups > Diagnostics > Operator Interface.
2. To test the USB port, touch it on the picture on the Operator Interface screen then follow the instructions on the screen.



3. To test the lights for Station 1 or 2, touch the corresponding light on the picture on the Operator Interface screen then follow the instructions on the screen.
4. You can also operate any component on the front panel of the CNC and watch the corresponding component on the screen. Verify that the operation of the component on the screen matches what you do on the panel

If the test for a single component fails, replace that component. If multiple components fail, contact your OEM or system integrator to replace the ribbon cable to the front panel board, the front panel board, or both.

MicroEDGE Pro Machine Interface Tests

The Phoenix software provides a series of diagnostic tests for the Hypertherm MicroEDGE Pro CNC. These tests require the following test hardware:

Connector test hardware	**Kit part number**
Serial port – RS-232 (motherboard) Serial port – RS-422/RS-232 (serial isolation board)	228831
USB port	228831
LAN/HyperNet	228831
HyPath I/O – 1st tester (green stripe) HyPath I/O – 2nd tester (red stripe)	228832
HyPath axes (green)	228832
HyPath THC (blue)	228832
HyPath joystick/speedpot (purple)	228832
Picopath I/O – 1st tester (orange stripe) Picopath I/O – 2nd tester (white stripe)	228833
Picopath axes (blue)	228833
Picopath THC (yellow)	228833
Picopath joystick/speedpot (brown stripe)	228833

Note:

- For MicroEDGE Pro with SERCOS, use kit 228831.
- Kits 228832 and 228833 both include the serial, USB, and LAN/Hypernet testers.
- If you are using the keyboard-only interface on the CNC (No Touchscreen Installed is selected in the Special Setups screen) you cannot perform the machine interface tests as described in this section unless you have a mouse installed on one of the USB ports.

To begin an interface test:

1. From the Main screen, choose Setups > Diagnostics > Machine Interface.
2. Enter the Machine password.
3. On the Machine Interface screen, choose the connector you want to test on the image of the CNC.

4. Follow the instructions in the sections below and on the screen.

Serial Test

The MicroEDGE Pro is equipped with four serial ports. Serial 1 and 2 support RS-232 communications. Serial 3 and 4 support RS-422 or RS-232 communications.

Conduct this test if:

- Processes or information communicated through the serial port are not operating properly.
- The CNC is unable to download files through the serial port.
- The serial link to the plasma supply fails.

To test serial communications:

1. On the Machine Interface screen, choose the serial port you want to test.
2. On the CNC, plug the serial test hardware into the port you selected.

3. On the Machine Interface screen, press Test. A message informs you if the test is successful.

In the event of a test failure, contact your OEM or system integrator with the following information:

- Serial 1 or 2 failure: replace motherboard.
- Serial 3 or 4 failure: replace serial isolation card.

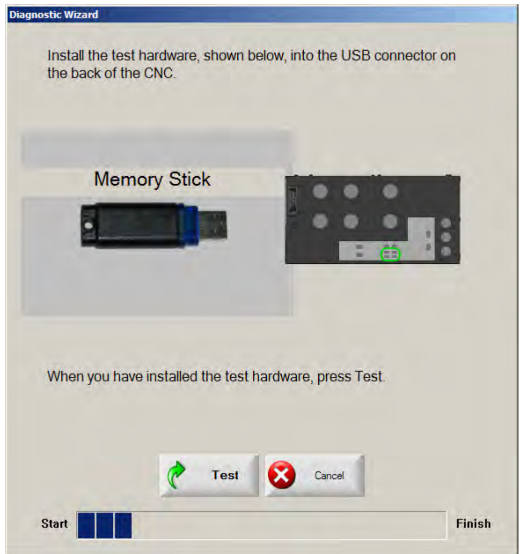
USB Test

Conduct this test if the memory stick is not shown as an option when you try to load part programs or update cut charts, software, or help. Use this test to check all the USB ports: the four on the back and the one on the front of the MicroEDGE Pro.

To test the USB port:

1. On the Machine Interface screen, choose a USB port.

2. Follow the instructions on the screen to insert the memory stick in the USB port on the back of the CNC.



3. On the Machine Interface screen, press Test. A message informs you if the test is successful.
4. Repeat the test on the remaining USB ports.

If the test fails on multiple USB ports, contact your OEM or system integrator to replace the motherboard.

I/O Test

Conduct this test if:

- An I/O point is malfunctioning.
- You need to eliminate CNC I/O operation as a problem in the system.
- A continuous fault is occurring, such as a limit switch that is not turning on or clearing.

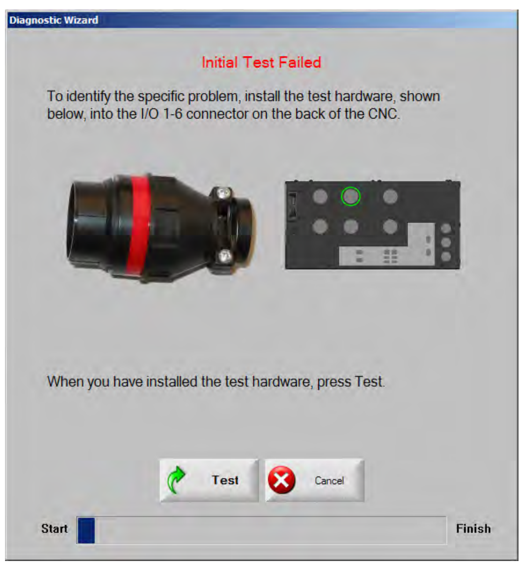
To test I/O:

1. On the Machine Interface screen, choose the I/O port you want to test.

2. Follow the instructions on the screen to connect the Picopath (orange) or HyPath (green) test plug into the I/O port you selected on the back of the MicroEDGE Pro.

3. On the Machine Interface screen, press Test. A message informs you if the test is successful.

4. If the test fails, follow the instructions on the screen to connect the second Picopath (white) or HyPath (red) test plug to isolate the I/O point that is causing the problem.



5. On the Machine Interface screen, press Test. The screen reports the number of the failed input or output.

Picopath: Rewire the I/O to a free I/O point and reassign the I/O function in the Phoenix software.

Hypath:

- If an input has failed, rewire the input to a free input point, then reassign the input function in the Phoenix software.
- If an output has failed, you can remove a relay from a spare output and swap it with the relay at the failed output. No rewiring is required.

Axis Test

WARNING

To prevent motion on the table, disconnect all axis cables from the CNC before conducting this test.

This test verifies axis operation. Conduct this test if there is:

- A runaway axis
- No motion
- Erratic motion
- A high number of position errors
- Multiple parts are not cut the correct size

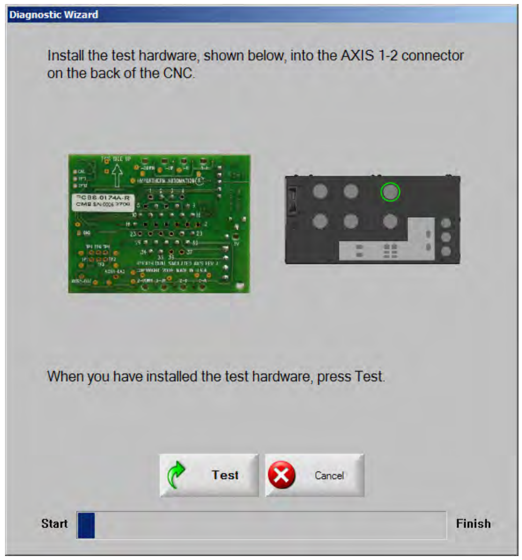
In order for all axes to pass, the following conditions need to be met.

- You can only test the amount of axes that are enabled on the hardware key (axes installed in Control Information kit).
  - If you have a 2-axis CNC, you still have 4 or 6 physical axes
  - The test will not allow you to test the other axes. Nothing will happen when you click on the 3rd/4th axis connector for example.
- Axes are tested by their assignment in Phoenix.
  - Axis switching, like Dual Transverse, CBH if no 3rd axis, or Rotate/Tilt on a 4 axis CNC, will fail the axis diagnostic test.
  - If you are not using the 3rd or 4th axis, or have an assignment that puts the axis at 5 and higher in the application, you can test the hardware by temporarily assigning the 3rd and 4th axes to Sensor THCs.

To test an axis:

1. On the Machine Interface screen, choose the axis port you want to test.

2. Follow the instructions on the screen to connect the Hypath (green) or Picopath (blue) axis simulator board to the CNC at the axis port you selected.



3. On the Machine Interface screen, choose Test. A message informs you if the test is successful.

If the test fails, contact your OEM or system integrator to check or replace these parts in the following order:

1. 4-axis servo field interface
2. Ribbon cable
3. MCC card

THC Test

WARNING

To prevent motion on the table, disconnect all axis cables from the CNC before conducting this test.

Conduct this test if:

- Nozzle contact is not functioning properly. For example, IHS using ohmic contact is not sensing the workpiece or is not accurate, the torch is running into the workpiece during cutting without retracting, or the torch is firing in the air.
- Arc voltage feedback is not functioning properly.
- The torch is rising off the workpiece or driving into the workpiece during the first part of a cut after piercing.

To test a THC port:

1. On the Machine Interface screen, choose the THC port you want to test.

2. Follow the instructions on the screen to connect the THC test cable to the THC port you selected and to the Axis 1 – 2 port (Hypath) or X/Y port (Picopath). Use the blue tester for Hypath and the yellow tester (not shown) for Picopath.

3. On the Machine Interface screen, choose Test. A message informs you if the test is successful.

If the test fails, contact your OEM or system integrator to check or replace these parts in the following order:

1. Analog field interface card
2. Ribbon cable
3. Analog input card

LAN and Hypernet Tests

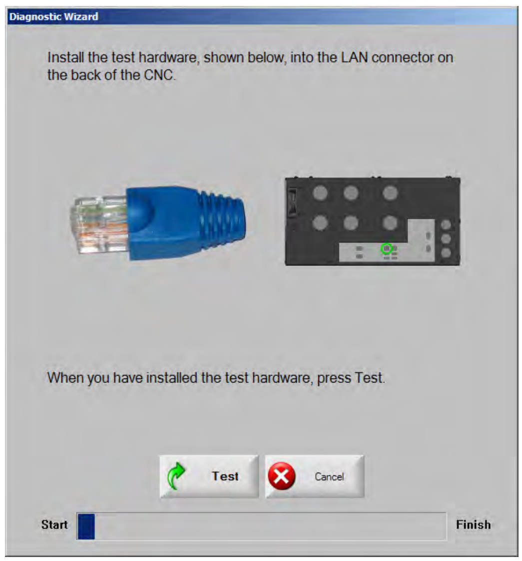
The LAN and Hypernet tests both use the same tester.

Conduct this test if:

- The CNC is not communicating to the ArcGlide or plasma system.
- The CNC is not communicating to the local area network.

To test a LAN port:

1. On the Machine Interface screen, choose the LAN or Hypernet port.
2. Follow the instructions on the screen to plug in the tester.



3. On the Machine Interface screen, choose Test. A message informs you if the test is successful.

If either test fails, contact your OEM or system integrator to replace the motherboard.

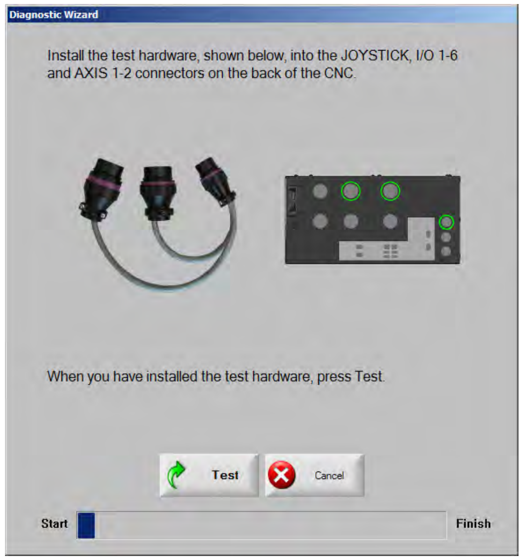
Joystick and Speedpot Test

⚠	**WARNING** **To prevent motion on the table, disconnect all axis cables from the CNC before conducting this test.**

To test the joystick and speedpot port:

1. On the Machine Interface screen, choose the joystick/speedpot port.
2. Follow the instructions on the screen to connect the Hypath (purple) tester to the Joystick/Speedpot port, Axis 1 – 2 port and the I/O 1 – 6 port.

For a Picopath system, connect the brown tester to the Joystick/Speedpot port, the X/Y port and the I/O port.



3. On the Machine Interface screen, choose Test. A message informs you if the test is successful.

If the test fails, contact your OEM or system integrator to check or replace these parts in the following order:

1. Analog field interface card
2. Ribbon cable
3. Analog interface card

I/O

Inputs

This test continuously monitors and displays the state of all of the optically-isolated discrete inputs in the CNC. The current state of each input, ON or OFF, is displayed next to the input name. You can also use a Watch Window to monitor selected inputs.

You can change the name of inputs that are displayed with a white background. To change the input name, use the Previous/Next arrow key to highlight the input and type the new input name. The input can be connected to the CNC and used as commanded in the part program. For example, the EIA W7 S1 code pauses the program and waits for input 7 to become active.

To open the I/O diagnostics screen, choose Setups, Diagnostics, I/O, and enter the Machine password.

Note: The number and name of the inputs depends on the CNC software and hardware configurations.

Outputs

This test allows the activation of all optically-isolated discrete outputs in the CNC. To change the state of an output, select it with the PREV and NEXT keys and press the SPACE key. The current state of each output, ON or OFF, is displayed next to the output name. You can also use a Watch Window to monitor selected outputs. An output screen with all of the outputs OFF is displayed below.

WARNING
When you activate the CNC's outputs manually, the machine, torch and marker can move and the cutting device can activate.

You can change the name of outputs that are displayed with a white background. To change the output name, use the Previous/Next arrow key to highlight an output and type the new output name. The output can be connected to the CNC and used as commanded in the part program. For example, the EIA M22 and M23 code will turn on and off output 12.

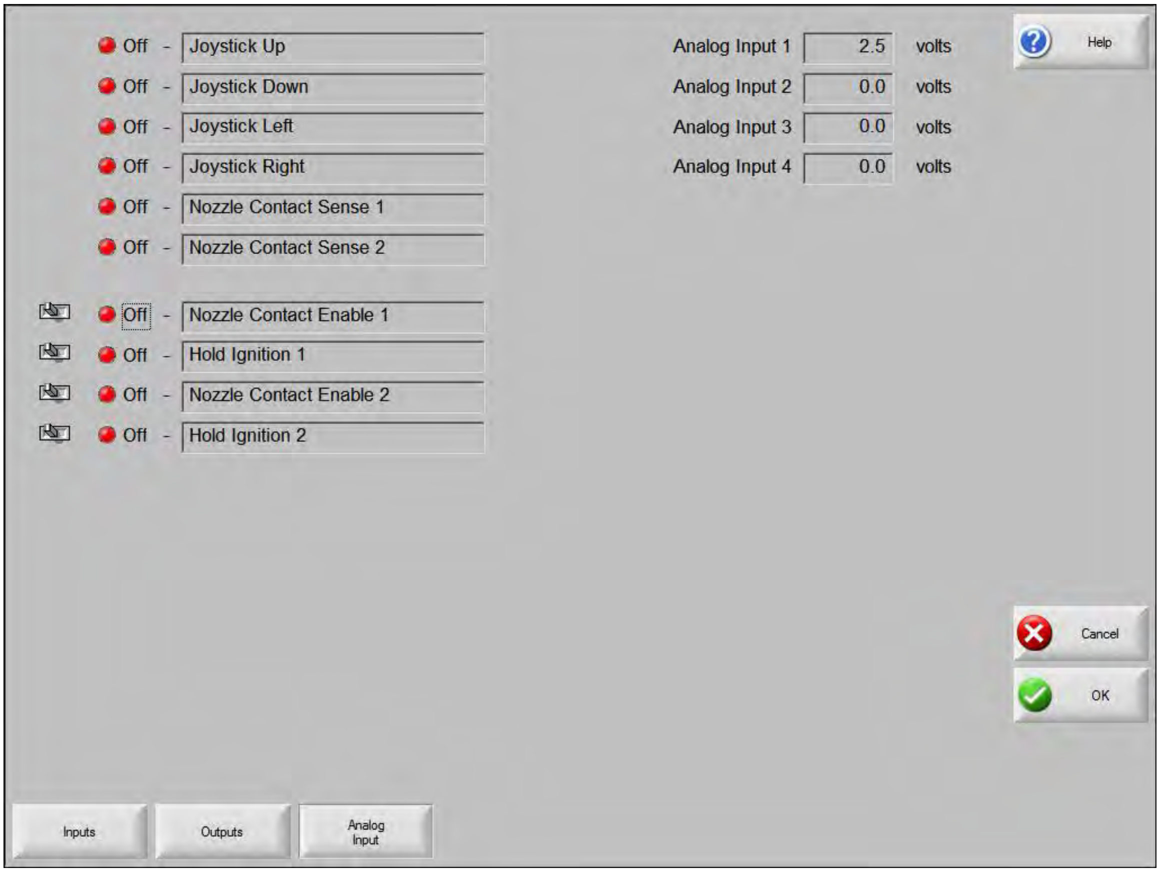
Note: The number and name of the outputs depends on the CNC software and hardware configurations.

Expanded I/O

For CNCs equipped with more than 64 I/O, a series of selection soft keys are available to access the required inputs, outputs and analog I/O.

Note: The optional USB Front Panel I/O is not accessible from the I/O Diagnostics screen. Functionality for the USB Front Panel can be tested through the Keyboard Diagnostics.

Analog Input Diagnostics



Note: The appearance of the View screen depends on the type of Analog Input Card that is installed.

Inputs

The Analog Input Card is required to operate the optional joystick and speed pots is equipped with six low voltage inputs that are optically isolated and can operate at 24 – 120 VAC/DC. The first four inputs are used for the optional joystick feature. Inputs 5 and 6 are used to support THC1 and THC2 Nozzle Contact Sense respectively. These inputs are normally-open.

Analog Inputs

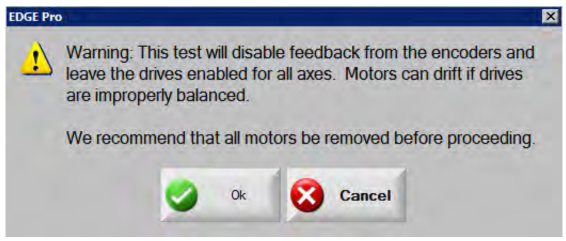
The Analog Input values shown in the upper right hand of the screen are the voltages coming into the analog card. Analog Inputs 1 and 2 are used for THC1 and THC2. Analog inputs 3 and 4 are used for optional external speed pots.

Drives and Motors

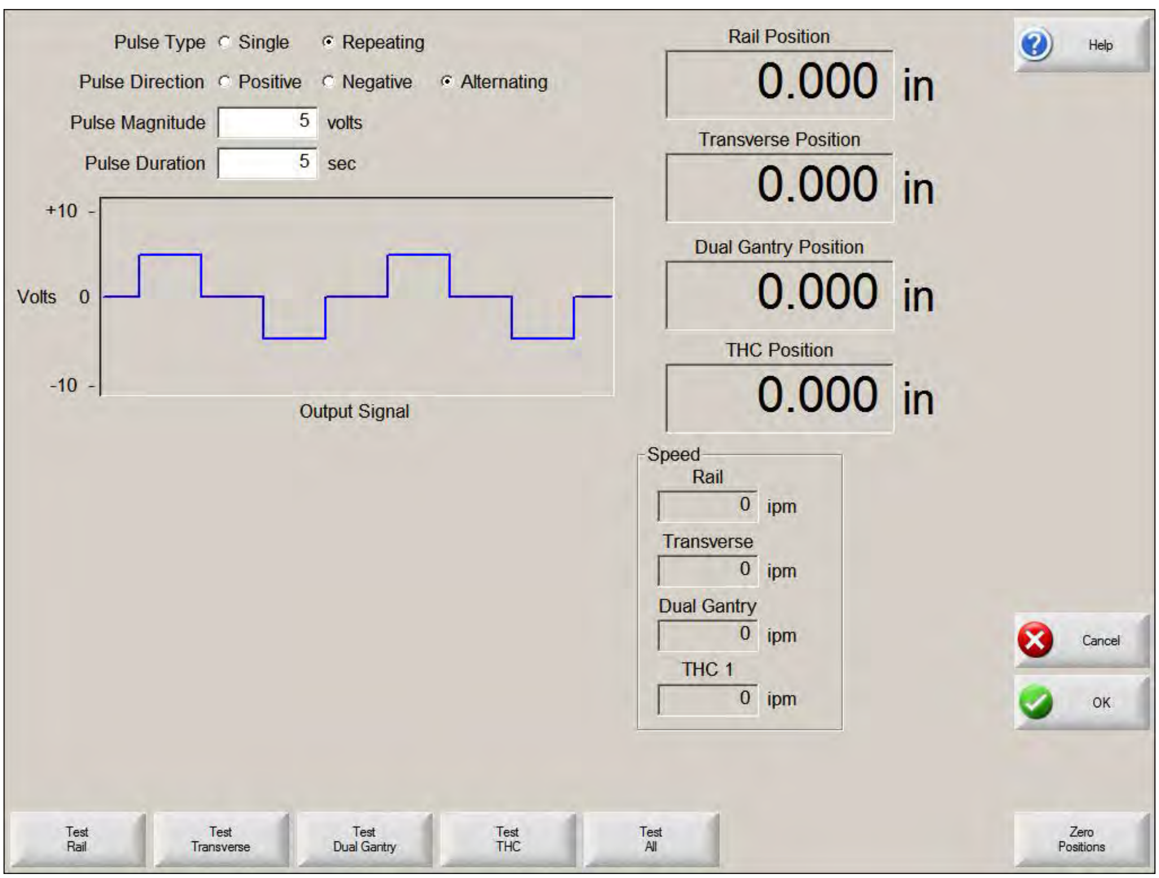
The Drives and Motors diagnostic screen allows a technician to send a direct signal to the drive amplifiers for testing, without enabling the servo loop.

	WARNING
	Drive motors must be removed from the table to prevent unplanned motion.

Select OK on the warning window:



Enter the machine password to view the Drives and Motors screen:



Pulse Type: Select the kind of pulse, single or repeated, that is sent to the drive amplifier. If you select a repeated pulse, the signal continues until the Test soft key for that axis is pressed again.

Pulse Direction: Selects the direction of the pulse; positive, negative, or alternating, that is sent to the drive amplifier.

Pulse Magnitude: Sets the voltage of the signal that is sent to the drive amplifier.

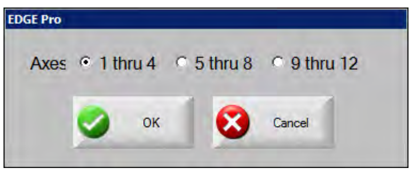
Pulse Duration: Sets the amount of time for each segment of the signal that is sent to the drive amplifier.

Test Transverse/Rail/Dual Gantry/CBH: The Test buttons send the selected output signal to the appropriate drive amplifier. If a repeating signal has been selected, any combination of available test buttons can be used for the test. The output signal is sent to the drive amplifier until the output soft key has been pressed a second time to end the test.

Test All: Sends the selected output signal to all drive amplifiers.

Speed: The Speed window displays the current speed for each axis.

Note: When more than four axes are selected, the user must select which group of axes will be tested on the Drives and Motors Diagnostic screen:



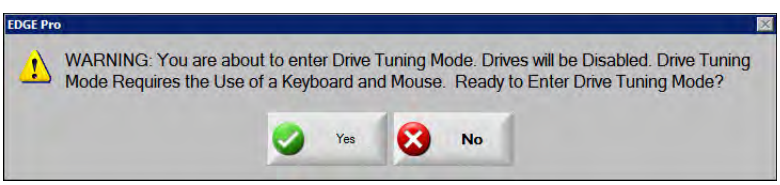
SERCOS Drives and Motors Test

Use the software provided by the drive manufacturer to perform diagnostics on the drives and motors in a SERCOS ring.

WARNING

Use extreme caution when you send direct signals to the drive amplifiers. Unexpected motion on the cutting table can occur.

Press Yes on the warning message window:



When the Drive and Motors Diagnostics screen displays, the appropriate drive communications software is launched for the drive that you selected. Refer to information supplied by the drive manufacturer to use the drive setup software.

Using Norton Ghost Utility

If a disaster occurs and prevents the CNC from operating, the Norton Ghost™ Utility is available on specified CNCs to replace core system files and re-enable operation of the CNC. Norton Ghost must be installed on the CNC hard drive with an appropriate ghost image file.

The Norton Ghost Utility works by allowing you to retrieve an image file from the drive D: partition of the hard drive to replace all data on drive C: The Ghost Image can be a factory default or custom, user-created image files.

All current information on Drive C: will be lost. You should create a unique user image after table configuration which includes all the important table settings.

This feature may be executed from the System Tools screen or through use of a Ghost image utility. This tool is available from the CNC supplier and transmitted over the Internet.

- Create Utility from E-mail – To create a Norton Ghost tool utility from the compressed software sent via the Internet.
- USB Memory – Instructions to download USB format tools and files are available to create a bootable USB device. Specific batch files to run the utility are included.

Creating a Ghost Recovery File

To create a Custom Image File:

1. Place the Norton USB memory device into the CNC.
2. Turn the CNC on. Norton Ghost displays:

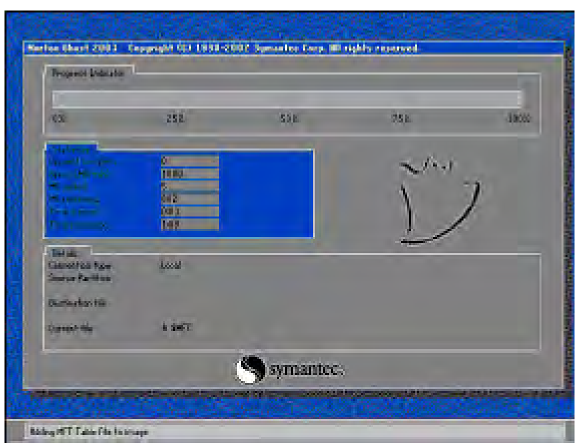
```
Starting PC DOS.......

Select 1 to make a New Backup Image or 2 to restore existing [1,2]
```

3. Press 1 to create a new back up image.

   You can create up to three custom images. They are created as “newest” to “oldest” image and are automatically assigned. The “original” ghost image is the factory default and cannot be over written. Back up begins automatically:

```
--------------Backing Up Drive, Please Wait ---------------
```



The following Message appears when the backup is complete:

```
---- Back Up Complete, Please Remove Disk and Reboot -----
```

1. Remove the USB memory device.
2. Turn the CNC off and then on. You may need to load a setup file and update the Phoenix software if the image is not current.

Retrieving an Image File

To retrieve an image file:

1. Place the Norton USB memory device into the CNC.

2. Turn the CNC on. Norton Ghost displays:

```
Starting PC DOS.......
Select 1 to make a New Backup Image or 2 to restore existing [1,2]
```

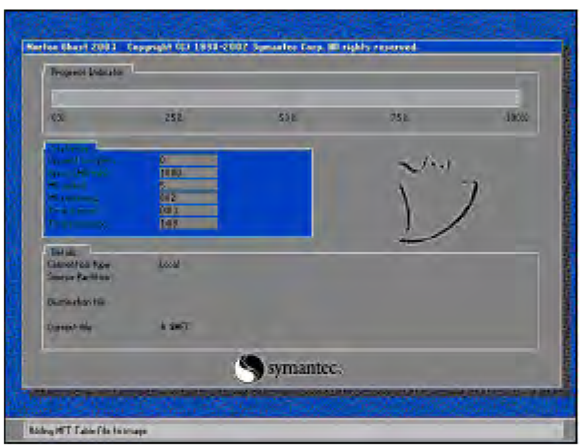
3. Press 2 to retrieve the backup image.

   You can create up to three custom images. They are created as newest to oldest image. The original ghost image is the factory default. The next message allows the user to select which version to use when restoring:

```
1. Newest.gho
2. Older.gho
3. Oldest.gho
4. Original.gho
Select the Number of the Image to restore from the choices above [1,2,3.4 ]
```

4. Press the number for the image that you want to retrieve.

```
--------------------Restoring, Please Wait ----------------
```



Norton Ghost launches and displays messages to indicate the progress of the restore. The following message displays when the restore is complete:

```
---------End Restoring, Please Remove Disk and Reboot--------
```

Remove the USB memory device. Turn the CNC off and then on. Table setups and software updates may need to be reloaded if the image is not current.

Section 9

Motion Control

This section provides an introduction to the following topics on motion control theory:

- Closed loop servo control
- Encoders
- Following error
- Position and servo error
- Edge per inch parameter
- Gain
- Recommended tuning procedures

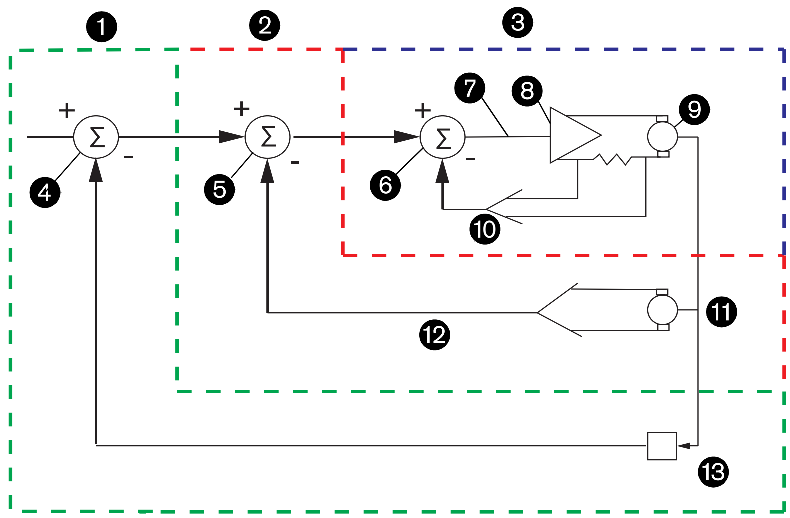
⚠	**WARNING** **System tuning should be performed by a qualified service technician. Improper tuning can cause personal injury or damage to the system.**

Closed Loop Servo Control

A servo system is the means of accurately controlling electrical motors to create force. The motor output is connected to a gear system to translate the rotational force of the motor into linear motion. In addition, this gear system modifies the strength and speed of the motion.

Closed loop servo control is the continuous process of monitoring position, or velocity commands, or both compared to actual position and velocity and adjusting the output accordingly. A servo system without feedback devices and automatic adjustment capabilities is called an open loop servo control.

The drawing below outlines a typical velocity and position loop system. The most important aspect for motion control is the position loop. The process starts with the motion control providing a motion command (voltage) to the motor to move at a specific speed to a position. Position is tracked during the motion by means of a feedback device, or encoder, that provides both directional and distance information. Based on this feedback, the CNC adjusts its motion command, or voltage, to the motor to ensure that the motor is accurately positioned on the designated motion path at the correct speed.



1 Position loop (green)

2 Velocity loop (red)

3 Current loop (blue

4 Position command

5 Velocity command signal

6 Current command or velocity error

7 Current error signal

8 Current amplifier

9 Servo motor

10 Current feedback signal

11 Tachometer

12 Velocity feedback signal

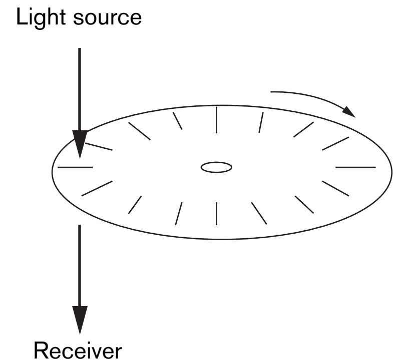
13 Encoder

Typical Velocity and Position Loop System

The motion command starts as digital output within the CNC and is converted to a +/- 10 VDC analog output for use by the motors. This conversion of the motion command within the control is referred to as the digital to analog converter (DAC) output and is performed by the motion control card. After the analog output leaves the control, it travels to a drive amplifier that increases the voltage output to the motor and creates motion. Also, there is usually a linear relationship between the voltage sent and the machine speed. For example, if 10 volts is the maximum machine speed, 5 volts is half maximum machine speed. Additionally, the polarity of the output (DAC polarity +/-) to the amplifier dictates the direction of the motor rotation. Typically, this is known as the Velocity command. In most applications and in the following application description, the feedback device is an encoder.

Encoders

An encoder is a feedback device that provides signal pulses as the motor turns. The diagram at the right illustrates the basic concept of an encoder. Although this does not represent all encoder feedback device technology, this illustration provides a visual aid to help understand the process.

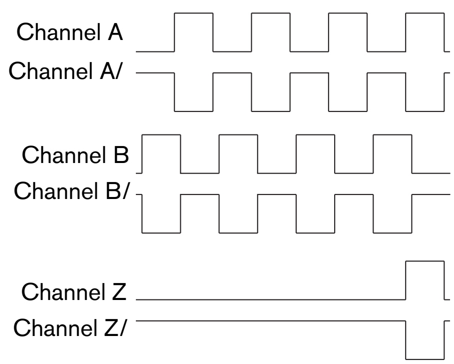


The illustration shows a disk with small holes cut out along the outer edge. The light source projects a beam of light downward through the holes in the disk. As the disk turns on the end of the motor shaft, the light passes through the disk and creates pulses. The receiver below the disk picks up the light pulses and sends that feedback to the control.

There is a direct relationship between the rotation of the motor shaft, the encoder light pulses, and the distance of the motion. Therefore, the control is able to calculate distance by counting the encoder pulses it receives, which closes the position loop. This relationship, shown as a simplified formula is:

encoder pulses x motor revs = distance

The encoder generates a square wave signal as illustrated in the Encoder Signal diagram to the right. Most encoders provide two main signals, A and B, and the complements, A/ and B/. These signals are also referred to as channels. The compliment channels are not always used but if they are used, they can provide increased noise immunity. The rotational direction, or encoder polarity, can be determined by the signal that is received, either ABABA or BABAB.



The channel Z signal is produced only once a revolution and is called the *marker pulse*. This marker pulse is quite often used for accuracy in homing routines.

The pulses are called *counts*. The holes in the disk are also called *lines*. The pulses that the receiver picks up may actually be the beginning and end of each pulse for a line on both channels (A&B) so that the receiver picks up four pulses for each line. This is called a 4x mode encoder. Thus, a 1000 line encoder in 4x mode would produce 4000 counts for each revolution of the motor. The more counts an encoder produces, the more accurate the motion is.

Following Error

Following error or servo error is the distance between the position that the CNC commands and the actual position of motion. Some following error is normal because the resistance of the load prevents the response of the motor to reach the ideal command of the system. This latent response is similar to driving a car away from a stop light. It takes time for the engine to produce the force that is required to move the weight of the car to the desired speed. Following error for each axis can be selected to be viewed in the Watch Window / Status Window of the CNC. This is a quick reference tool for monitoring the performance of the actual machine motion as compared to the CNC command.

The important consideration for X/Y coordinate motion is that the response for the X and Y motion is similar. If the response is dissimilar, poor results for commanded motion will result. An example of a dissimilar tuned response would be that when a circular motion is commanded, an oval or elliptical motion will result as one axis out-performs the other.

Position and Servo Error

A position, or servo, error occurs when the distance between the ideal motion position and the actual motion position exceeds a pre-defined servo error tolerance value. A position error indicates that the response from the motion command was not executed, was executed too slowly or was executed incorrectly. The Servo Error Tolerance is a user-definable value within the CNC and is usually set to a value twice the following error during normal operation. This allows the system to operate without nuisance faults caused by temporary resistance to the motion, such as dirt on motion rails or temporary tension from motor cabling.

It is important to remember that in the closed position loop, the output command adjusts to maintain speed and positioning. Thus, a loss of encoder feedback causes the control to send its maximum motion command to the motor so the motor “runs away” without control. Conversely, if the motor fails to turn while feedback is enabled, a position error results and the command output stops when the fault occurs.

Encoder Counts and Maximum Machine Speed

Remembering that the position loop uses the pulses or counts from the encoder to calculate distance, it is important to determine how many encoder pulses are equal to a specific distance of machine motion. The control uses the encoder counts per inch/mm value as the constant to calculate distance and speed.

The encoder counts per inch/mm is a calculation based on the number of pulses generated by the encoder for one revolution of the motor and how much distance is traveled from that motor motion through the gearing being used. The following formula illustrates this relationship:

counts/line x counts/rev x 1 rev/inch = encoder counts/inch

For example, the resolution of a 4X – 1000 line encoder counts both edges (lines) of channel A and channel B to equal 4 counts per line multiplied by the 1000 lines per revolution equaling 4000 counts per revolution. If the encoder revolutions per inch of travel are 1:1, we would have 4000 encoder counts per inch of travel.

4 counts/line x 1000 lines/rev x 1 rev/inch = 4000 encoder counts/inch

Determining Maximum Machine Speed

The following formula is used to establish the proper value for determining maximum machine speed.

max RPM of the motor x number of inches of travel per revolution = maximum machine speed in inches per minute.

For example:

4000 RPM Motors x 0.125 inches per rev = 500 inches per minute

Through test or calculation it is determined that the table moves 1/8 inches per revolution of the encoder.

With the maximum speed, encoder counts per inch, DAC and encoder polarities determined and entered into control setups, a simple test of machine motion can be made in the control Drive Diagnostics screen. This test should be made with the motors disengaged for safety. This is a basic motion test of the system and does not use gain terms or the position loop for motion. For more information on maximum machine speeds, refer to the Setting Speeds chapter in the Machine Setup chapter.

Gain

Proportional Gain

In a feedback control system, the error term is acted on by the control system and it alters the output. Proportional gain is amplification of the error term. In a closed loop control system, this is proportional to the error signal. Thus, the output is proportional gain multiplied by error.

In most systems, proportional gain is the primary tuning parameter for improving the response of the position loop.

Integral Gain

Proportional gain cannot completely eliminate error. The system can become unstable if only proportional gain is used to eliminate error in a system. When the response of a system is satisfactory, but steady state error is excessive, the error can be further reduced by increasing system gain only for long term accumulations of error over time. Integral gain is sometimes used to compensate for static load disturbances like torque loading, gravity bias, and offset.

Note: Integral gain can cause instability in a system because it has a more instant effect at the beginning of a move profile. Therefore, it is rarely used for velocity drives. However, integral gain can provide improved response with current drives. Only small adjustments to integral gain should be made if indications of steady state error exist. In almost all applications, this value is set to zero. Use the feed forward gain for closing following error that occurs during a move profile.

Derivative Gain

Derivative gain responds to the rate of change of the signal and can produce corrections before the error term becomes large, therefore it is useful in improving the transient response of a system. Since it opposes change in the controlled output, it can produce a stabilizing effect by dampening a tendency toward oscillation. As a rule of thumb, tune by following error to the maximum proportional gain then set derivative gain at 10% of proportional gain. Do not overdo derivative gain as it can have a detrimental effect on the overall response of the system.

Feedforward

Feed forward “pushes” the commanded output ahead to reduce or eliminate the difference between the actual position and the commanded position during motion, called dynamic following error. Feed forward corrects for a “lag” in the system and has an effect similar to proportional gain. Do not make large increases in feed forward gain because it can cause positive following error and excessive overshoot.

Velocity Gain

When using a current loop amplifier, the internal velocity loop in the control can provide dampening without an external tachometer.

Using the internal velocity loop with a current loop amplifier can result in higher static stiffness, smoother machine motion, and less overshoot.

Tuning Procedures

Using the information previously provided as a core understanding of the position loop process, we can now look at tuning motion for the desired motion performance. Due to potential safety hazards and the potential possibility for machine damage from incorrect tuning, it is recommended that this operation be performed by trained and experienced personnel.

Prior to performing motion tuning, values for DAC polarity, encoder polarity and maximum machine speed should be calculated and entered into control setups. Servo Error Tolerance for the axes should be set to maximum (5 inch max for X/Y motion).

Initially, you should start basic tuning with the motors disengaged from the machine to allow you to confirm controlled motion and the response of the motion axes. An initial acceleration rate of 10 Mg can be used to begin the motion tuning. It is important to remember that after the motors are engaged to the machine for motion, the load requirements for each axis will change and require additional tuning. Additionally, if the acceleration rate is increased after the initial tuning, additional adjustments to gain tuning may be required for proper machine motion.

Tuning for Velocity Drives

You should tune the system by performing manual moves at a slow speed with all gains set to zero with the exception of proportional gain. Proportional gain should start with a small value of 10. Engage the motors to the machine frame and beginning raising the proportional gain to allow movement. If the axes start to vibrate or oscillate, this indicates the selected gain value is too high and should be lowered. It may also require de-tuning the response of the drive amplifier.

Now that you have some basic control of the system, set up the Watch Window to display position and following error for the selected axes. Load and automatically cut the Test Pattern simple shape from the shape library in trial mode. You will be able to follow the response of the system by watching the performance of the motor /encoder feedback and its ability to maintain path in the preview window.

Systematically increase the value for proportional gain until it has little or no effect on the following error or until instability becomes noticeable. Then reduce the value for proportional gain to eliminate instability. Derivative gain may be used to provide a dampening effect. For a velocity loop servo drive, integral gain should be set to zero. After tuning for the best response, increase feed forward gain systematically to further reduce following error if necessary. You do not have to eliminate following error but you should ensure that following error for each axis is similar.

Feed forward gain reduces steady state following error, but may cause overshoot.

It is important to note that some following error is desirable, following error terms should be nearly the same from one axis to another, and the polarity of the following error term must be the same as the direction of travel.

Finally, check the system tuning by placing a pen on the system. “Draw” the test pattern provided in the simple shape library and observe the cornering capability, arc/ circle contouring and the ability to position at the center crossings. When troubleshooting tuning issues, vary the size and trial speed to observe the mark of the pen.

After the system has been successfully tuned, adjust the servo error tolerance to equal twice the normal following error for the axes.

This is a common approach to setting the servo error tolerance but is up to the discretion of the service technician. This adjustment allows the system to operate without the nuisance faults that are caused by temporary resistance to the motion, such as dirt on motion rails or temporary tension from motor cabling.

Tuning for Current Drives

Tuning for current drives is similar to tuning for velocity drives because the recommended tuning procedure focuses on the response of the motion. The process begins with low initial values that are increased to get the necessary response. If the axes start to vibrate or oscillate, this indicates the selected gain value is too high and should be lowered. Basic tuning should start with the motors disengaged from the machine to allow you to confirm the controlled motion and response of the motion axes. Motors can then be engaged for fine tuning their performance.

Set up the Watch Window to display position and following error for the selected axes. Load and automatically cut the Test Pattern simple shape from the shape library in trial mode. You can follow the response of the system by watching the performance of the Motor/Encoder feedback and its ability to maintain path in the preview window.

Systematically increase the velocity gain until it has little or no effect on the following error or until instability becomes noticeable. Proportional gain can then be used to increase the “stiffness” of the response. Integral gain can be used to improve the steady-state performance. Derivative gain can be used to provide a dampening effect.

After tuning for the best response, increase feed forward gain systematically to further reduce following error if necessary. You do not have to eliminate following error but it is wise to ensure that following error for each axis is similar.

Feed forward gain reduces steady state following error, but may cause overshoot.

Note that some following error is desirable. Following error terms should be nearly the same from one axis to another, and the polarity of the following error term must be the same as the direction of travel.

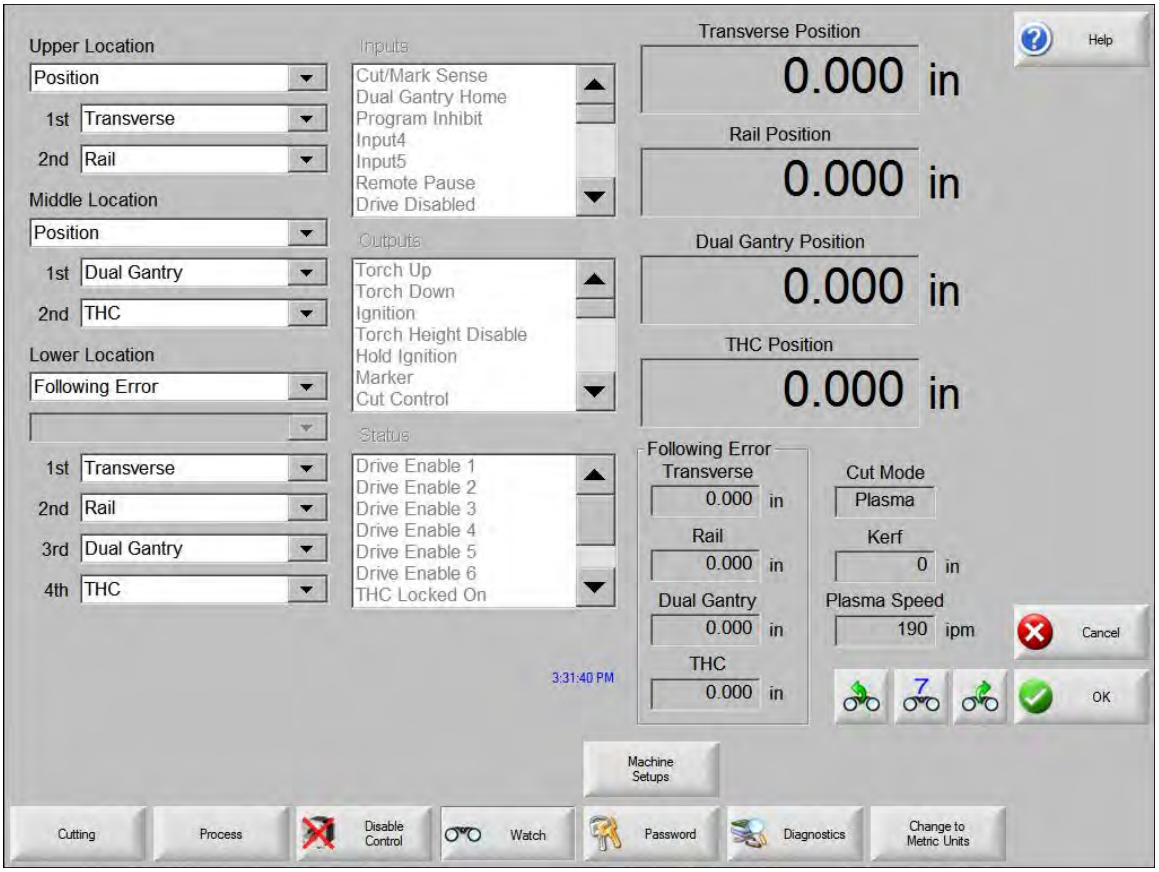
Finally, check the system tuning by placing a pen on the system. Draw the test pattern provided in the simple shape library and observe the cornering capability, arc/ circle contouring and the ability to position at the center crossings. When troubleshooting tuning issues, varying the size and trial speed can often provide valuable information by observing the pen marks.

After the system has been tuned successfully, adjust the servo error tolerance to equal twice the normal following error for the axes.

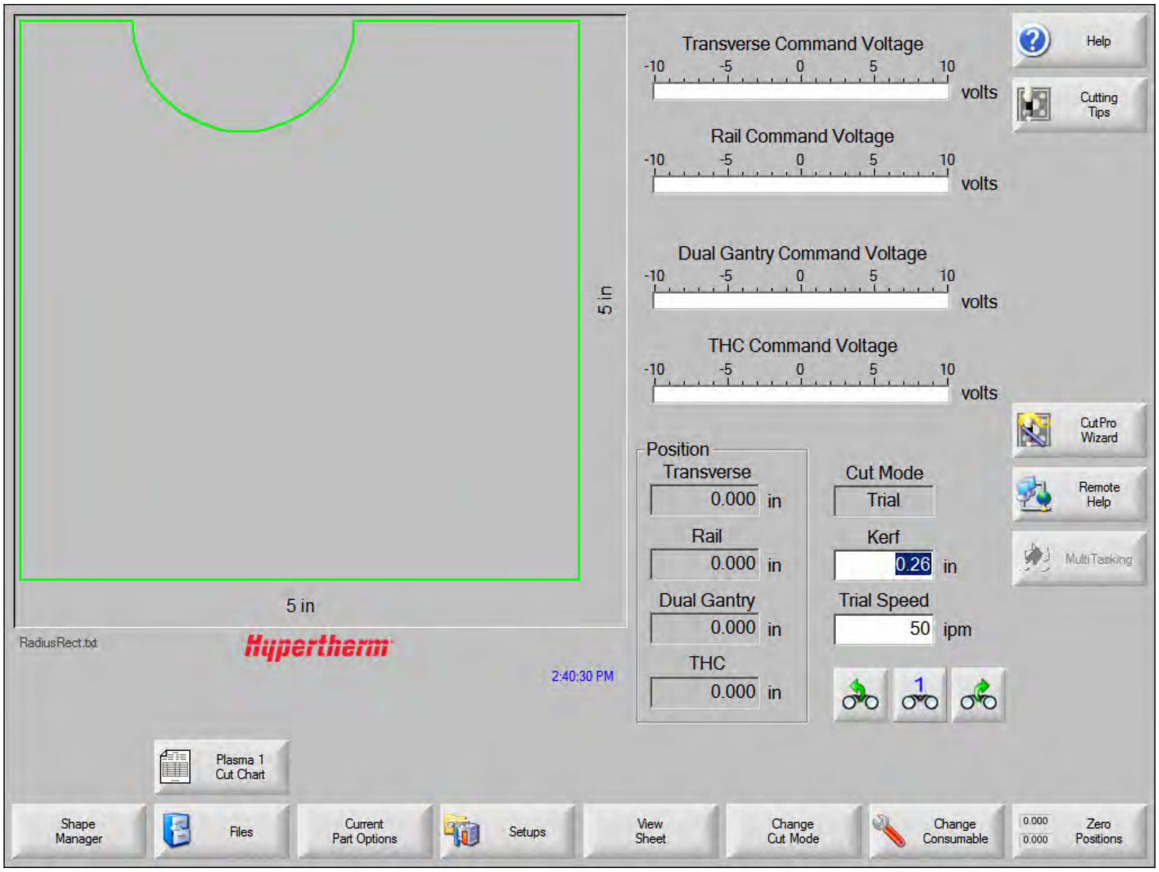
This is a common approach to setting the servo error tolerance but is up to the discretion of the service technician. This adjustment allows the system to operate without nuisance faults caused by temporary resistance to the motion, such as dirt on motion rails or temporary tension from motor cabling.

Motion Tuning Watch Windows

Position and following error can be displayed in the Watch Window.



Command Voltage allows the user to view directional motion command voltage being sent to the amplifier for velocity drives. This displayed voltage also equates to current being commanded for motion in current drives. Peak voltage can be displayed for a specified amount of time.



Section 10

Motion Compensation

Motion compensation is a feature of Phoenix software that improves the linear positioning accuracy of a cutting table if there are mechanical inaccuracies on the axes. A laser interferometer records a number of positions along the rail, transverse, and dual gantry axes. The CNC uses the data from the interferometer to compensate the commanded position to each drive axis and reduce position errors to 0.

The compensation calculation is based on the position deviations between the CNC's commanded position and the data collected by the laser interferometer at the same points, or targets, along each axis.

Motion compensation does not improve the following error on an axis. The only way to reduce following error is to ensure that system drives are tuned properly.

Hardware and Software Requirements

The following equipment and software are required to implement motion compensation:

- A Hypertherm CNC
- Phoenix software version 9.7 or later
- Laser interferometer, such as the Renishaw XL-80

Overview

Position data from the laser interferometer are based on target locations along each axis. The CNC uses these positions to create a part program for each axis.

Because motion compensation is based on position, it is important that all measurements and compensation start at a consistent location. The table must be homed to the same location for each axis before laser mapping and before motion starts.

In addition, parameters such as Home Offset, Absolute Home, Home Direction, Up Direction, and Right Direction affect motion on the table. If any of these parameters are changed after compensation data is entered into the Phoenix software, axes must be remapped to capture the new settings.

Calculating Compensation Data

Phoenix determines the amount of compensation to apply to servo commands before commanding motion along an axis. The amount of compensation that Phoenix applies is equal to the size of the position error between two targets. Phoenix distributes this compensation along the distance between two targets

The basis for motion compensation is the assumption that the cutting system is at the same starting position as when the axes were mapped. If the start locations are not the same, then Phoenix cannot apply motion compensation correctly. For this reason, Homing should be enabled so that the system is homed after the system is turned on or after an error to ensure that motion starts from the same point that was recorded in the data files.

Phoenix determines the amount of compensation to apply to the servo command before it commands motion along an axis. Compensation is calculated and applied in the following steps:

1. The laser interferometer calculates the average position deviation at each target. The position deviation is the difference between the target's position along the axis and the actual position of the axis at the target.

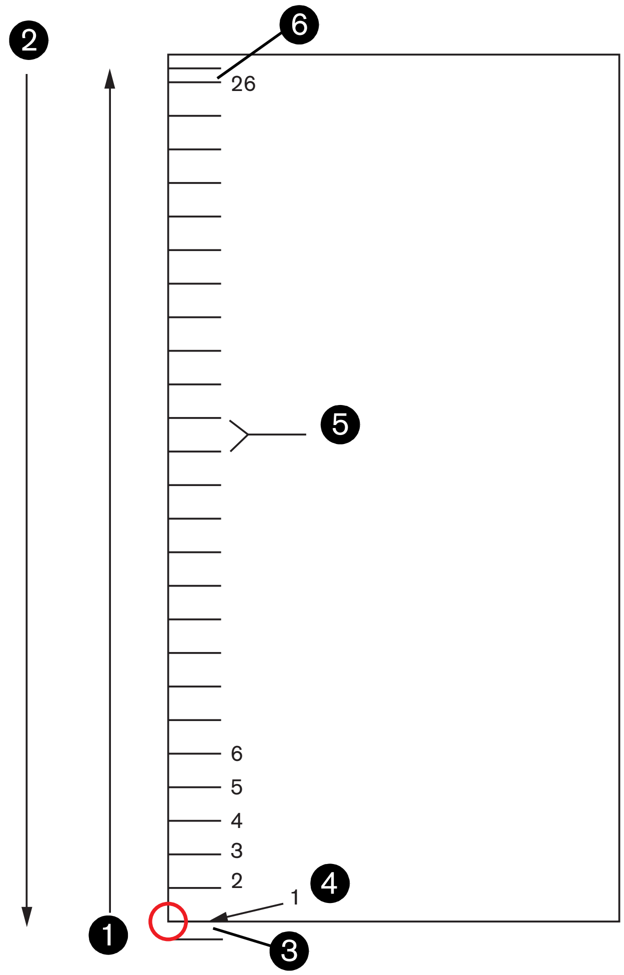
2. The laser interferometer calculates the position deviation between two corresponding targets and records this data in a text data file. This data file is loaded into the EDGE Pro so that Phoenix software can use it to compensate servo commands for the position deviations.

3. Phoenix software distributes (averages) the position deviation over the length of each target distance along the axis.

4. Phoenix compensates the servo command along each target distance on the axis so that when the axis reaches each target, the actual position error is 0. If the average position deviation is negative (the average position deviation is short of the target position), compensation is added to servo commands. If the average position deviation is positive (the average position deviation is past the target position), the compensation is subtracted from servo commands.

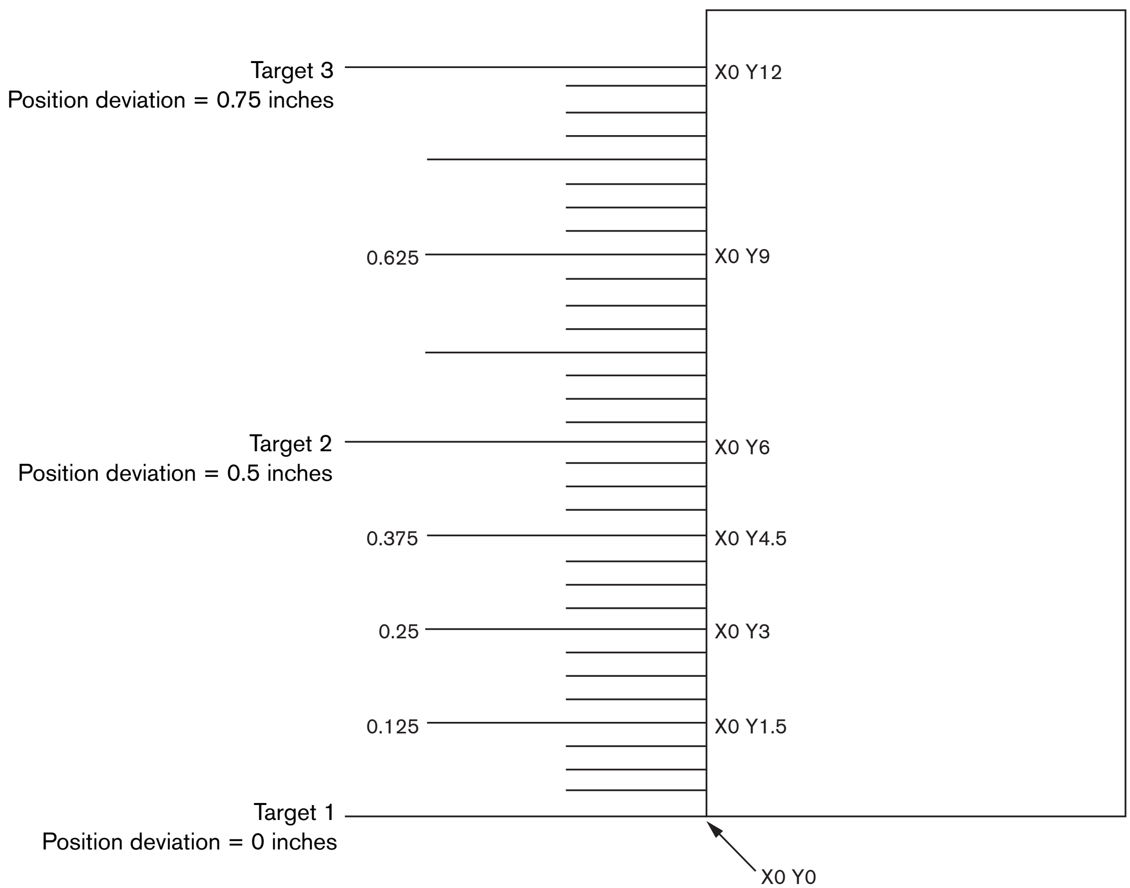
The following figure illustrates a cutting table with target points arranged along two runs. The first target is at home position, shown as position 0,0. Each target is equally spaced at 152.4 mm (6 inches), the target distance. When the user presses the soft key to map the axis, Phoenix creates a CNC program with traverses equal to the target distance and pauses after each traverse.

In this example, the traverse segments are 152.4 mm (6 inches) long. The first move in the program is a reverse move to remove any backlash. The next move is toward the first target. At the first target, the program pauses motion and prompts the user to begin mapping the axis. The laser interferometer records the position deviation at each target along the run. In this example, the laser interferometer will record the position deviation at 52 targets, or 26 targets on each run.



1	Run 1: Rail + (Y + axis)
2	Run 2: Rail - (Y - axis)
3	Run 1 reversal distance
4	First target at Y0, home position
5	Target distance
6	Run 2 reversal distance
Number of targets: 26 each run	
Target distance: 152.4 mm (6 inches)	

The next figure is based on the previous example. The laser interferometer calculates the position deviation listed at the 3 targets while mapping the rail axis and then saves the data to a file. In this example, exaggerated deviations are used to clarify how the data are used.



Phoenix software compensates for these position deviations as it commands motion up and down the axis. The amount of compensation is based on the current position in relation to the previous and next target and the deviations between the targets.

Target #	Position	Motion compensation
1	0.0 in	0.0 in
	1.5 in	0.125 in
	3.0 in	0.25 in
	4.5 in	0.375 in
2	6.0 in	0.5 in
	9.0 in	0.625 in
3	12.0 in	0.75 in

Calculating Backlash Compensation

In addition to calculating position deviation from the data in the files from the laser interferometer, Phoenix software calculates mechanical backlash compensation. The backlash calculation assumes a reverse move at the beginning of each run.

Phoenix calculates backlash compensation using the following steps:

1. Calculates the difference between the first target and the last target for each run.
2. Adds together the results for each run to obtain the total backlash for both runs.
3. Divides this sum by the number of runs.

The result is the average backlash across all runs at the first and last targets.

The following example data file provides the data for the backlash compensation calculation that is outlined below it.

Run	Target	Data
1	1	0.40 mm
1	2	0.30 mm
1	3	0.40 mm
2	3	0.60 mm
2	2	0.25 mm
2	1	0.60 mm

The calculation for the average backlash is:

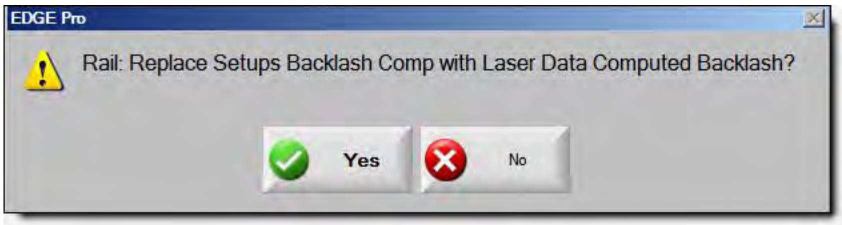
((0.40-0.60) + (0.40-0.60)) / 2 =

-0.20 + -0.20 / 2 =

-0.40 / 2 =

0.20 mm (0.008 inches)

If the backlash compensation calculated with values from the data file is different from the value that is entered on the Axis screen, Phoenix prompts the user to use the calculated backlash compensation value.

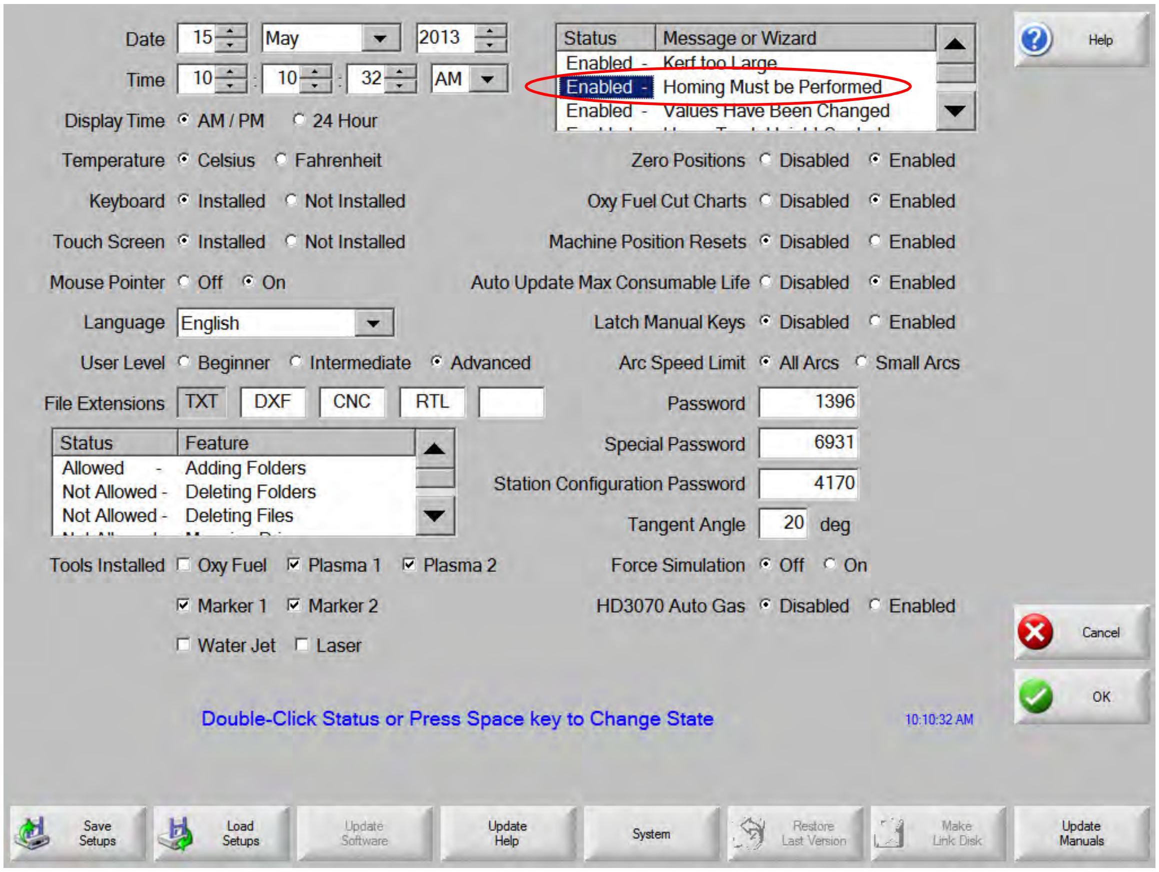


Capturing and Using Motion Data in Phoenix

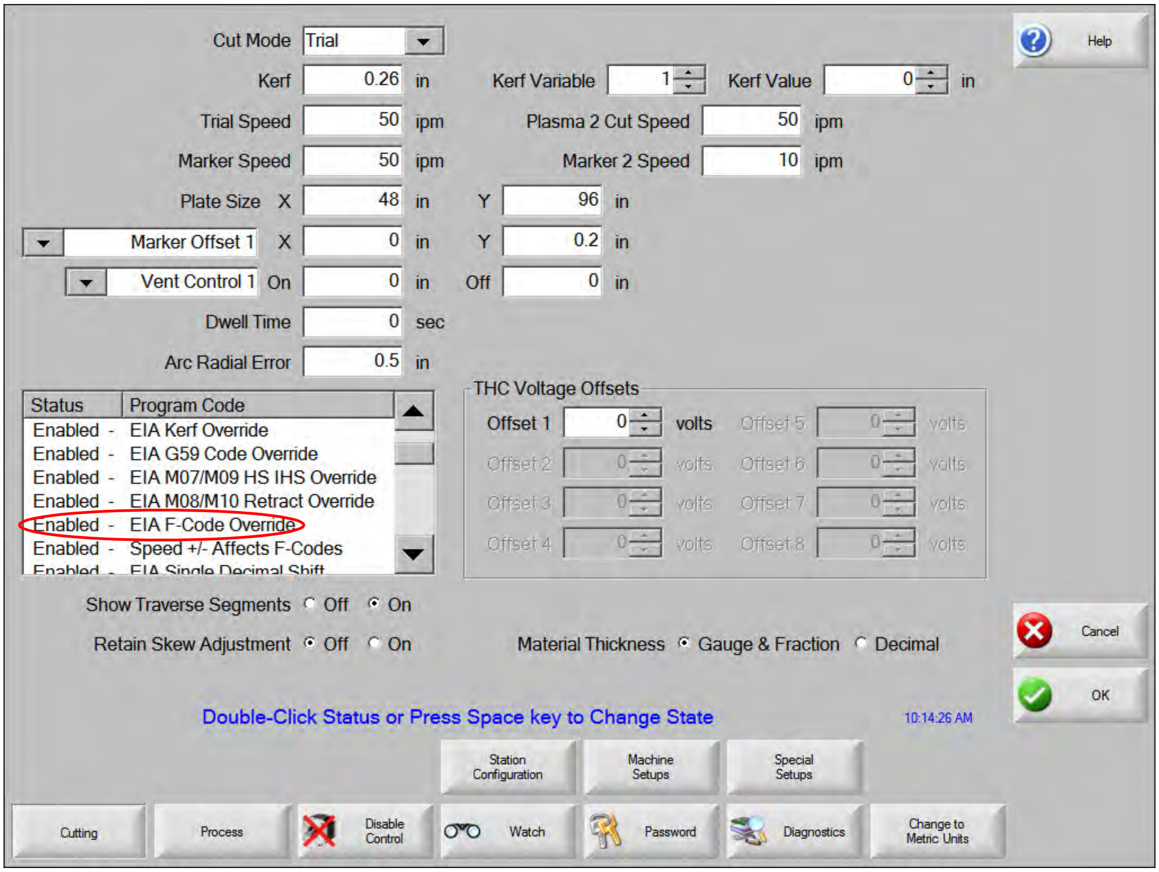
Before you map an axis or enable laser compensation, you must:

- Enable Homing Must Be Performed in the Message or Wizard list on the Special Setup screen.
- Disable Machine Position Resets in Special Setup screen.

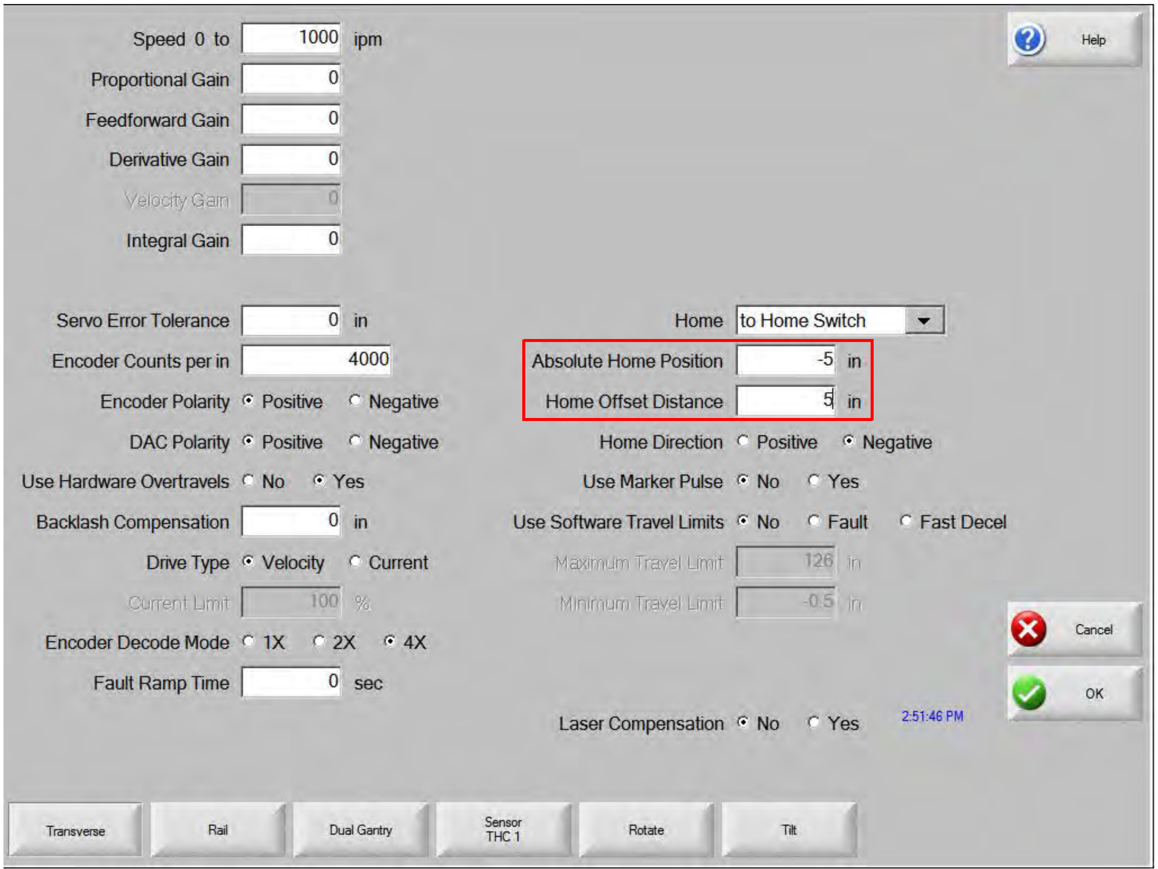
- Add RTL to the list of File Extensions in the Special Setup screen.



- Enable EIA F-Code Override and Optional Program Stop under Program Code in Cutting screen.



- On the Axis screen, enter -5 for Absolute Home Position and 5 for Home Offset Distance.



To allow Phoenix to compensate for position inaccuracies on an axis, complete the following steps:

- Map the axis.
- Create a compensation data file for the axis.
- Load the data file into the CNC.
- Turn on motion compensation in Phoenix.

Repeat these steps for each axis you want to map.

Map Axes

The CNC uses the parameters in the Laser Mapping screen to generate a part program that allows the laser interferometer to map the position of each drive axis (rail, transverse, and dual gantry) at specified positions (or targets).

Before you begin mapping an axis, you must install the laser interferometer on the axis that you want to map. Refer to the manufacturer's documentation for details.

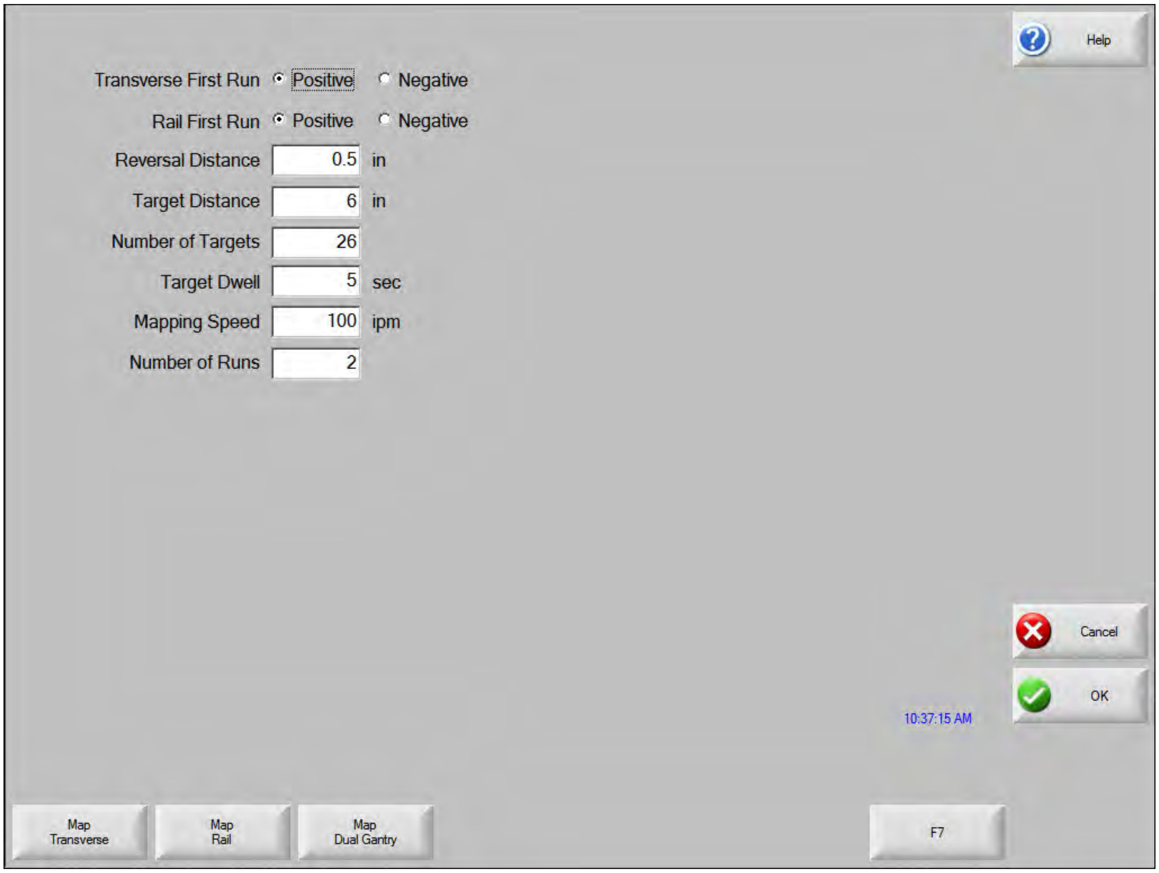
Map the entire length of each axis with the laser interferometer and start the home position.

1. Home the gantry.

2. In Phoenix, open the Cutting screen and set the Cut Mode to Plasma or Oxyfuel.

3. Set the Plate Size dimensions for the area that will be mapped.

4. Open the Machine screen using the password assigned to it, and choose Laser Mapping.



5. Use the parameters on the screen to define targets along the axis and how the laser will move between them. The CNC uses these parameters to create a part program to control motion on each axis. These parameters are described below.

   Rail/Transverse First Run: The direction of the first run on the axis. Base the selection on the direction of motion from home position.

   Reversal Distance: The distance of travel in the opposite direction at the beginning and end of a run to remove mechanical backlash before mapping. Reverse motion removes mechanical backlash before mapping the axis.

   Target Distance: Sets the distance between targets. Set this value in larger distances (508 mm, 20 inches) for a rough mapping and in much smaller distances (25 – 50 mm. 1 – 2 inches) for actual mapping. A smaller distance between targets will permit more precise motion compensation.

   Number of Targets: The number of positions on each run where the laser interferometer measures the physical position of the axis. The part program that the CNC creates for mapping includes a pause at each of these positions. Enter values between 2 and 1000. A larger number of targets will permit more precise motion compensation.

Target Dwell: The amount of time that motion pauses at each target to allow the interferometer to take each measurement and record the results. Refer to the instruction manual for your laser interferometer to determine how to set this value.

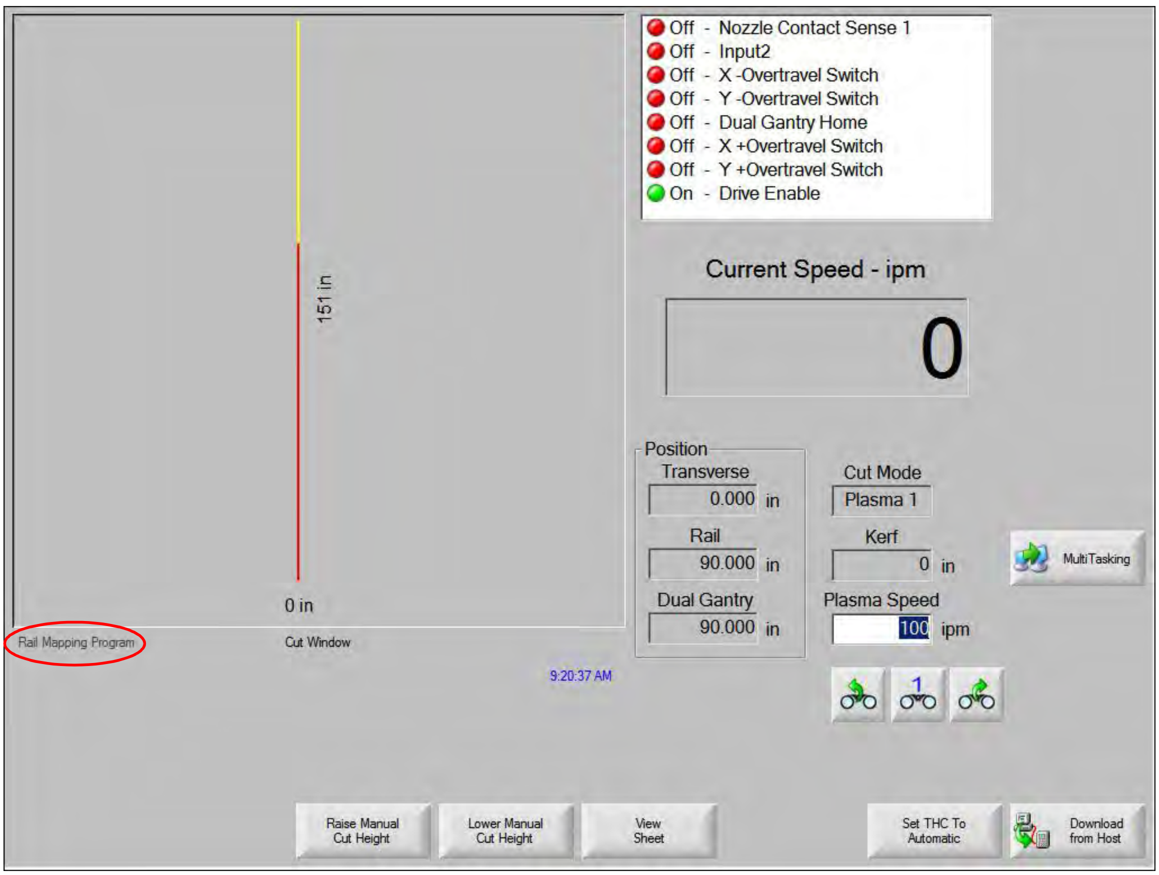
Mapping Speed: The program speed of the part program that the CNC creates for mapping.

Number of Runs: The number of times that the part program runs on an axis.

Set this to an even number between 2 and 1000 because the interferometer must travel to the end of the axis and then return. The greater the number of runs, the more precise the motion compensation can be.

6. Choose the appropriate soft key at the bottom of the screen to map the rail, transverse, or dual gantry axis.

   After you press the soft key to map an axis, the CNC loads and runs a part program based on the values you entered in the Laser Mapping screen. You can view the run motion and pauses of the axis in the Preview Window on the Main screen.



7. After you have finished mapping the axis, save the data from the interferometer and format it according to the instructions in *Create the Motion Compensation Data File*.

Repeat this procedure for each axis that you want to map.

To map both rail axes on a dual gantry cutting table, perform the following steps:

1. Create a "rough" map of the rail axis by entering 508 mm (20 inches) or more for Target Distance on the Laser Mapping screen, then map this axis, create the data file (see *Create the Motion Compensation Data File* on page 228), load the data file (see *Load the Data File* on page 229), and turn on laser compensation for this axis (see *Turn on Motion Compensation* on page 229).

2. Create a "fine" map of the dual gantry axis by entering a normal (much smaller) distance for Target Distance on the Laser Mapping screen, then map this axis, create and load the data file, and on turn laser compensation for this axis.

3. Turn off laser compensation for the rail axis.

4. Create a "fine" map of the rail axis, using the procedure described in Step 2.

Create the Motion Compensation Data File

The motion compensation data file incorporates the data that the laser interferometer collects. The name of the compensation data file for each axis must be different so that the CNC can apply motion compensation to the appropriate axis. Names for the motion data files must use the following conventions:

*name*A1U.rtl for the X axis (transverse or rail)

*name*A2U.rtl for the Y axis (transverse or rail)

*name*A3U.rtl for the dual gantry axis

Note that:

- *name* can be any valid filename and is not case sensitive.
- The letters A and U must be uppercase.
- The file extension must be rtl.
- The X and Y axes can be transverse or rail. Verify your table's orientation in the Machine screen.

The contents of the data file must use the following format:

Header::

Target-count: 3

Targets:

0.000000 508.000000 1016.000000

Axis: x

Run-count: 2

Run	Target	Data
1	1	0.000
1	2	-3.048
1	3	21.590
2	3	48.260
2	2	42.926
2	3	42.164

The CNC can read the data with any number of spaces or tabs between data types. The following table describes the types of data in the file.

Field	Description	Format
Header::	This must be the first line of the file. It identifies the file as a laser compensation file.	As shown
Target-count	The number of targets in each run.	2 to 1000
Targets	The target positions along the axis.	Position along the axis, in mm, with 6 significant decimal places, where the laser interferometer measures the axis position and the discrepancy from the commanded position.
Axis	The letter for the axis being measured.	X, Y, Z (dual gantry)
Run-count	The number of runs along the axis to each target location.	An even number, between 2 and 1000
Run	Identifies the run during which the data was collected.	1 to the value of Run-count.
Target	Identifies the target for which the data was collected.	Number in the sequence of positions identified in Targets.
Data	The position deviation between the target position and the axes actual position.	A number, in mm/1000, with 3 significant decimal places.

Load the Data File

After you create the data files, you must load them into the CNC so that Phoenix can access them.

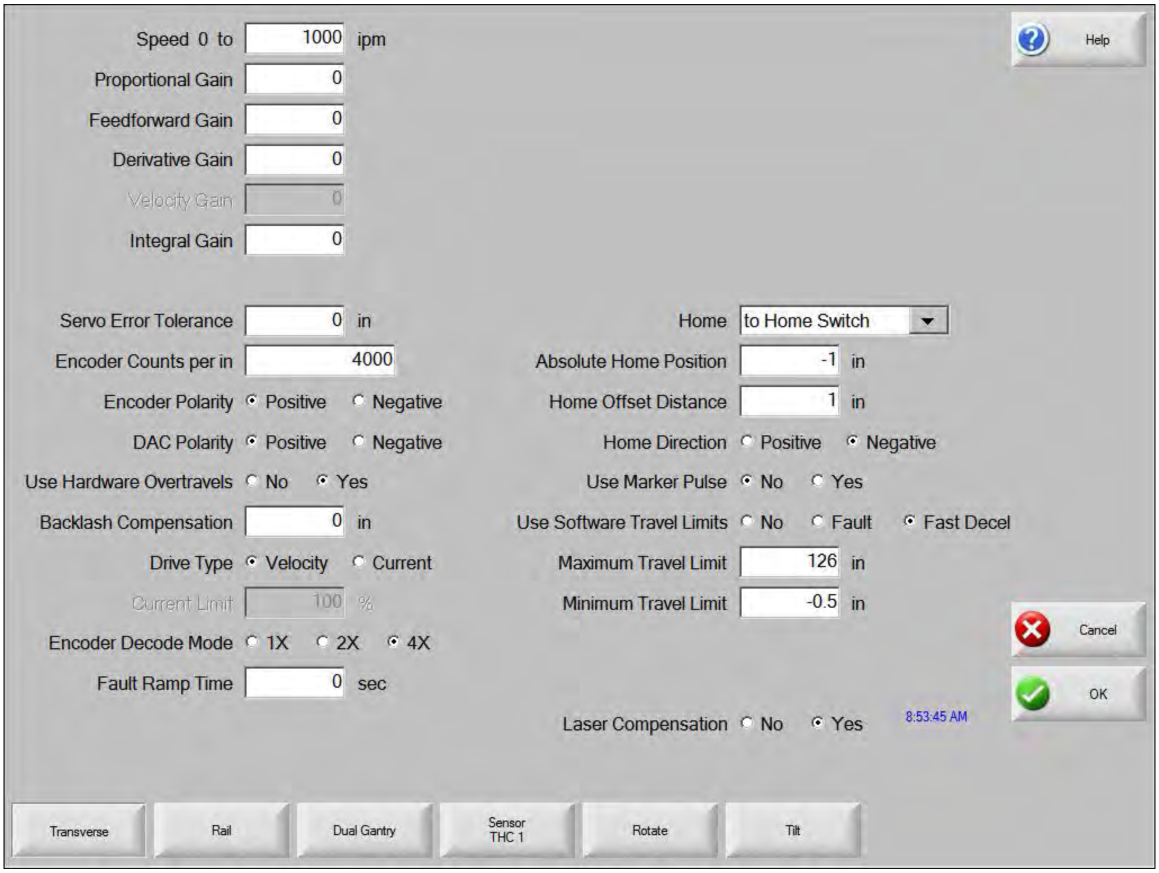
1. Save the compensation data files that you created on a memory stick.
2. Insert the memory stick into one of the USB ports in the CNC.
3. In Phoenix, choose Files on the Main screen, and Load from Disk.
4. Choose Memory Stick in the Load from list.
5. Choose the file that you want to load and choose OK.
6. Repeat Steps 3 through 5 for every data file you want to load.

Turn on Motion Compensation

To allow Phoenix to use the compensation data during cutting operations:

1. Open the Machine screen with the password assigned to it.
2. Choose the axis (Transverse, Rail, Dual Gantry) on which you want to enable motion compensation.

3. On the screen for the axis, choose Yes for Laser Compensation.



4. Repeat Steps 3 and 4 for each of the remaining two axes on which you want to activate motion compensation.

5. Choose OK.

When the CNC issues Servo commands to direct motion, Phoenix uses the compensation data that the laser interferometer collected to correct the differences between the measured and commanded motion.

Save the Setup File

When you have finished motion measurements and have saved and loaded the RTL files that you want the system to use for compensation, save the Setup.ini file so the system will also use any new settings. Save these files as:

- Default.ini on the hard drive.
- *Today's date.ini* on the hard drive, on a backup hard drive, and on a removable storage device.

Section 11

Networking

Hypertherm CNCs can be connected to a local area network (LAN) for part program download and Remote Help. Connecting the CNC to a network requires knowledge of the Windows XP operating system and networking features. Contact your network system administrator for assistance in creating an account for the CNC on a file server and a workgroup for the CNC to join.

Before You Begin

These following sections provide the background you will need to achieve optimal performance using the CNC on a network. For all procedures in this chapter, a keyboard and mouse connected to the CNC provide ease of use.

Dynamic Host Configuration Protocol

On the CNC, both the wired LAN connection and the optional wireless interface card are set up to use Dynamic Host Configuration Protocol (DHCP). DHCP automatically communicates to the host network to get the address and other internet protocol settings when the CNC is connected to the network. Some networks, however, do not use a DHCP server, and the network setup on the CNC differs between the two. Refer to the appropriate section in this chapter:

- For a DHCP network, refer to *Connecting the CNC to a Network (DHCP)*.
- For a non-DHCP network, refer to *Connect the CNC to a Network (non-DHCP)*.

Using the CNC in a Domain-based Network

Most Windows XP networks are domain-based networks. In an office where the computers are members of a domain, each computer has security credentials – a user name and password – that you are required to enter to log into the domain.

Because of the performance requirements of the CNC to maintain the motion control of a cutting system, the CNC ***cannot be joined to or logged into a domain***. Joining a domain creates a user account on the CNC and this will result in performance issues. Instead, the CNC can be a *member* of the domain without logging in.

- The system administrator creates a user name and password for the CNC on the file server.
- When a connection to the network is established (either through the wired LAN connector or through a wireless connection), the CNC is visible to the network without being logged in.

- When mapping drives for part program download to the CNC, the server asks for the user name and password before allowing the CNC access to files.
- If the system administrator has created workgroups on the network, the CNC can be a member of a workgroup. Refer to *Connecting the CNC to a Workgroup* later in this chapter for instructions.

Administrator and User Accounts on the CNC

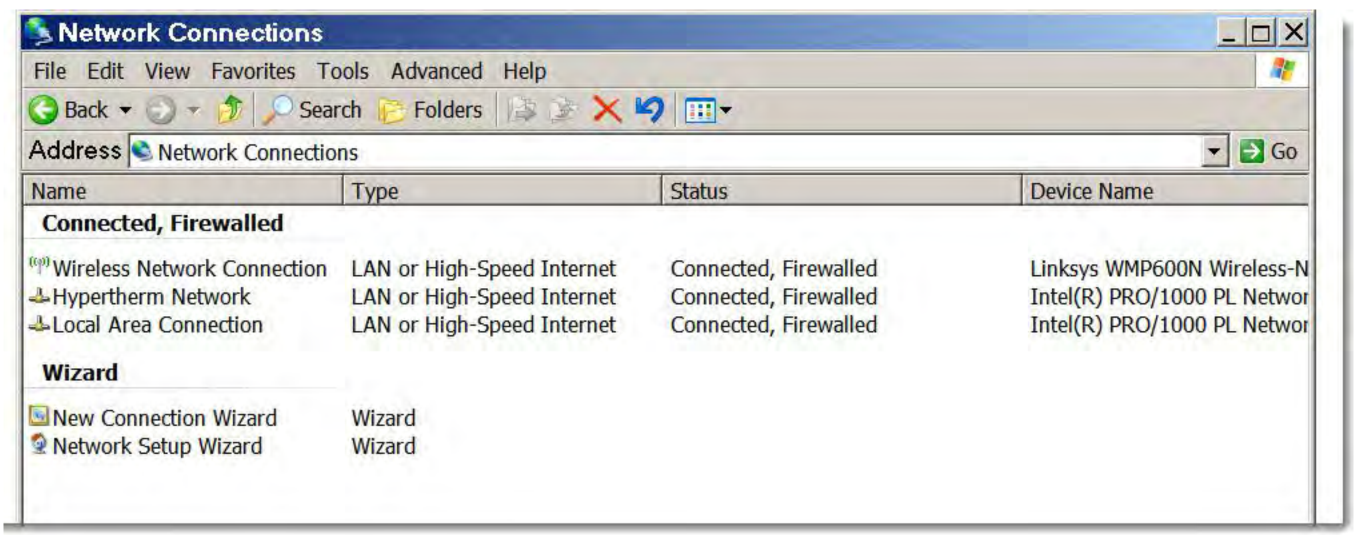
The CNC is set up by default with a networking account with administrator rights. This is a local account and is used only on the CNC by the Windows XP operating system. When the CNC powers up, it automatically logs into Windows XP with this administrator account.

In most networking situations, user accounts are used to log into a computer, and into a network. However, user accounts will create performance issues on the CNC. If the CNC were to be joined to the domain, Windows XP would automatically create a new user account and it would cause performance issues. For these reasons, we strongly advise that you do not create user accounts on the CNC, and do not directly log into the domain-based network.

About Network Connections

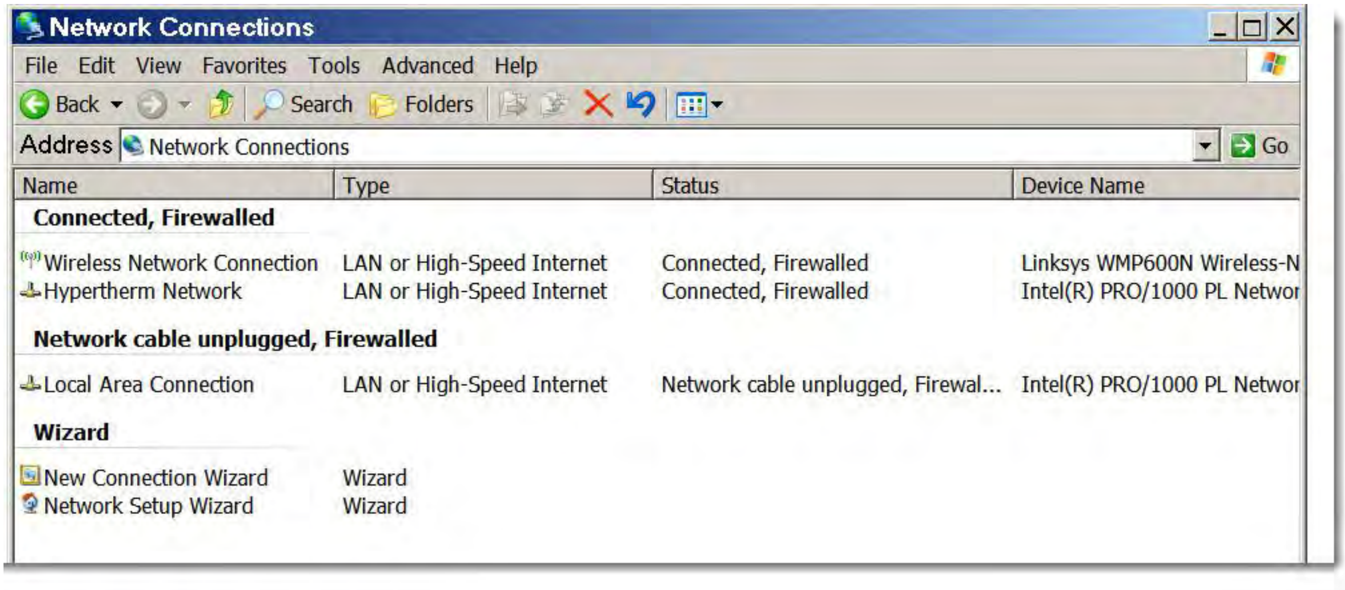
The CNC can have multiple network connections. You can view the network connections from the Phoenix software through the Network Tools function. To access Network tools:

1. From the Main screen, choose Setups > Password and enter the Special Setups password.
2. On the Special Setups screen, choose System > Network tools to open the Network Connections window.



- **Wireless Network Connection** appears if the optional wireless card is installed in the CNC.
- **Hypertherm Network** appears if you have a device such as the ArcGlide or an HPR plasma system communicating with the CNC using Hypernet.
- **Local Area Connection** is the wired LAN connection to your local network.

You can disconnect the network connections that you aren't using. For example, if you have wireless card, you can unplug the cable from the LAN connector to disconnect the unused network connection.



In the picture above, the LAN cable has been unplugged. The CNC is using the wireless network connection to the office network, and the Hypernet connection to communicate with other devices in the cutting system.

Connecting the CNC to a Network (DHCP)

The Windows operating system on the CNC is set up to use DHCP to obtain the address and other internet protocol settings for the CNC.

To connect the CNC to a network that uses DHCP:

1. The system administrator creates an account for the CNC on the network file server. The account requires:
   - A user name, for example: *operator*
   - A password for the user name, for example: *1234*
2. If you are using the wired LAN interface, connect the network cable to the LAN connector on the CNC.
3. Turn on the power to the CNC. After power up, the CNC connects to the network.

The wireless interface automatically searches for and connects to the LAN when you turn on the CNC.

Notes:

- You will be logged into the CNC as the local administrator. This is not the same as being an administrator on the LAN.
- In a DHCP network, the CNC automatically connects to the network and obtains an address. The CNC does not log in to the network. DO NOT attempt to have the CNC log in to the network.
- You will be asked for the user name and password provided by your system administrator when you map a network drive (see *Mapping a Network Drive* later in this chapter). The user name and password are security credentials that will be needed to access drives on the network. The server will recognize the CNC as having access rights to files on the server without requiring the CNC to be a member of the domain.

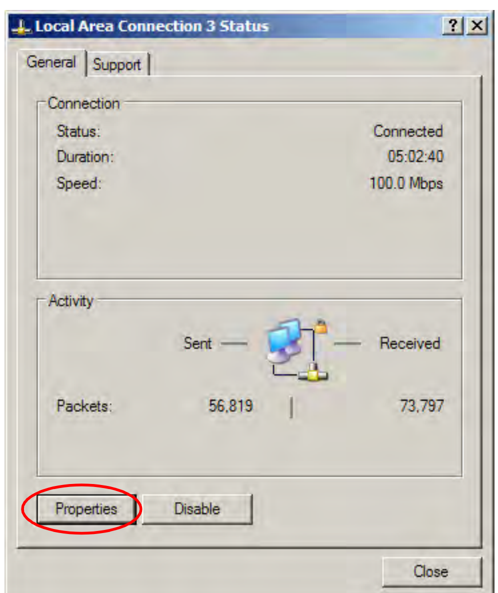
Connecting the CNC to a Network (non-DHCP)

If you are not using a DHCP Server, you will need the following internet protocol network settings. Enter them in the table below for future reference:

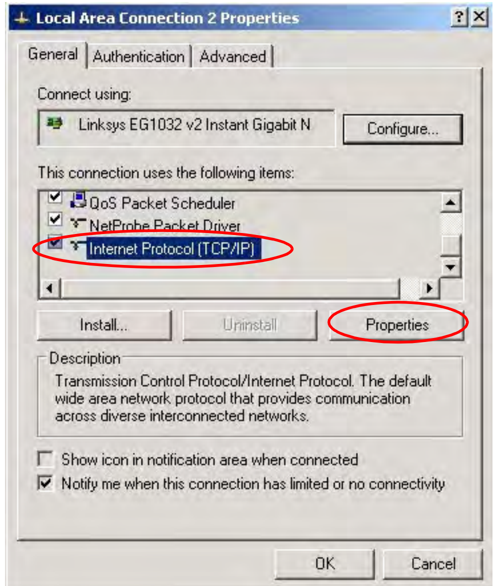
IP Address	
Subnet Mask	
Default gateway	
Preferred DNS server	
Alternate DNS server	
Will the CNC be part of a workgroup?	
Name of the workgroup	
Has the operator's user name and password been added to the server?	

In a non-DHCP network you will need to access the network setting screen and change the internet protocol settings to make the CNC compatible with the network.

1. Press Alt+F4 to exit Phoenix.
2. Choose Start > Settings > Network Connections.
3. Double-click the active network connection in the list. The status window for the connection opens.



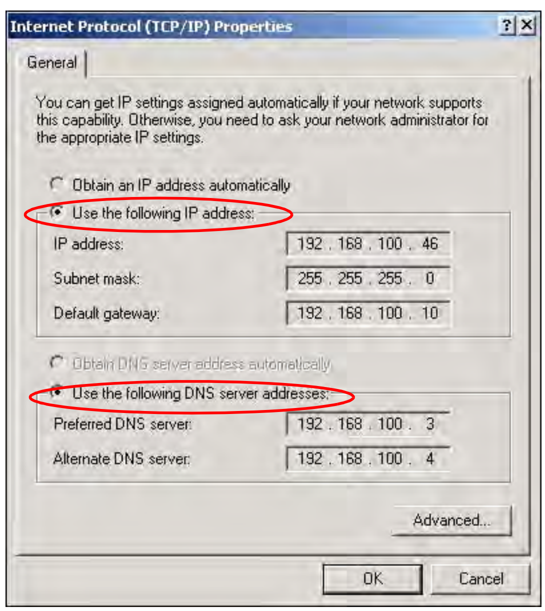
4. Choose Properties. The Connection Properties screen for the adapter shows the features for its network connection.



5. Highlight Internet Protocol (TCP/IP) and choose Properties.

6. Enter the TCP/IP address information in the IP address, Subnet mask, and Default gateway fields. Note that you must provide a valid IP address that has not been used elsewhere in the network.

7. Enter the DNS server address information in the Preferred DNS server and Alternate DNS server fields.



8. Choose OK, then OK to exit the screens.

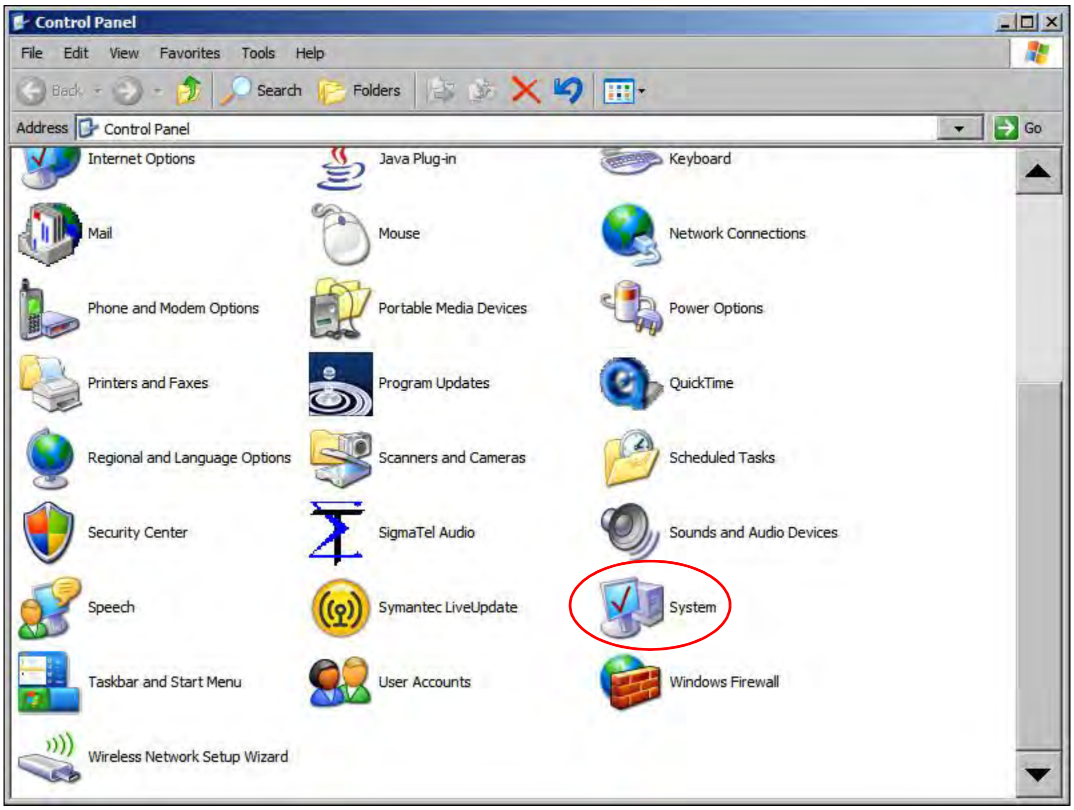
9. Restart the CNC.

Connecting the CNC to a Workgroup

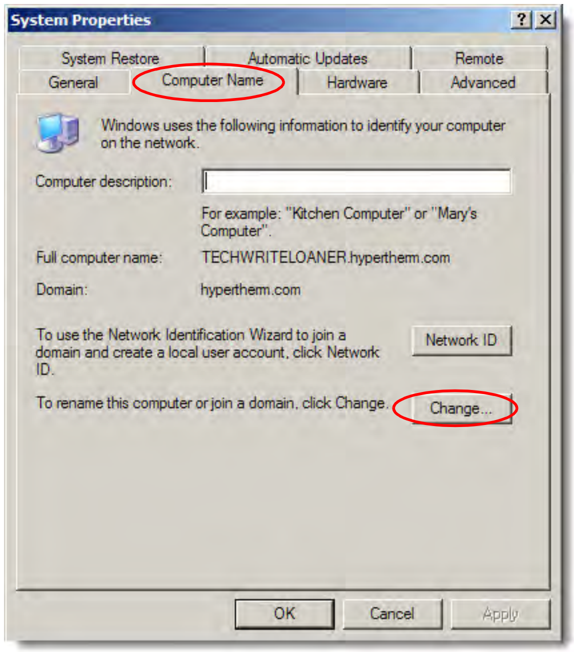
The CNC can be set up as a member of a workgroup. If you are going to connect the CNC to a workgroup, you will need to enter the workgroup name in the System Properties of Windows XP.

To connect to a workgroup:

1. Press Alt + F4 to return to the Windows desktop.
2. Select Start > Settings > Control Panel.
3. Double-click on the System icon.



4. Choose the Computer Name tab.

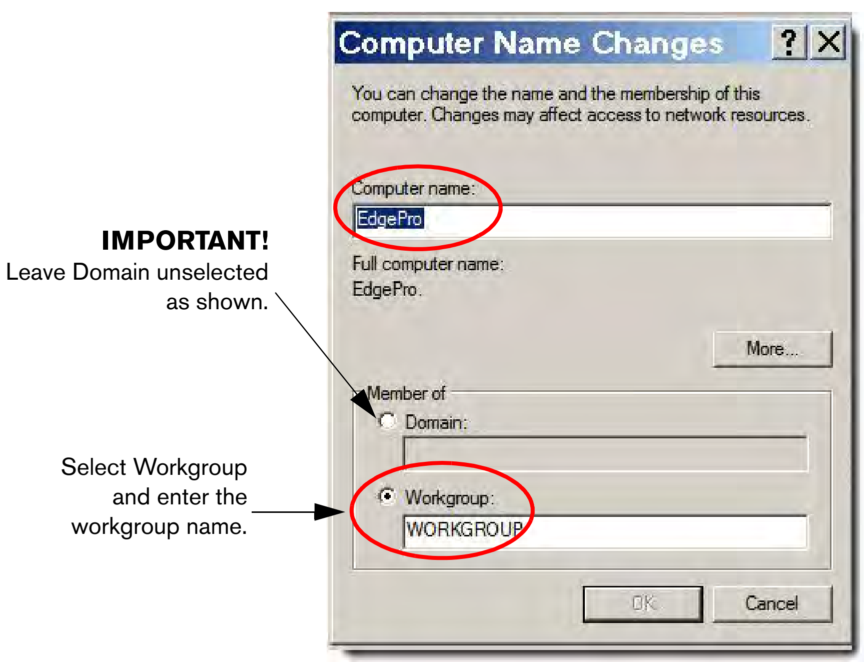


5. Click Change.

6. Enter the name of your CNC in the Computer name field.

7. In the Member of group box, select Workgroup.

8. Enter the workgroup name of the computer with which you share files.



9. Choose OK, then OK.

10. Restart the CNC.

Mapping a Network Drive

Phoenix requires a network drive to be mapped in Windows. After you map the network drive, you can add it as a folder in Phoenix.

1. Press Alt+F4 to exit the Phoenix software.

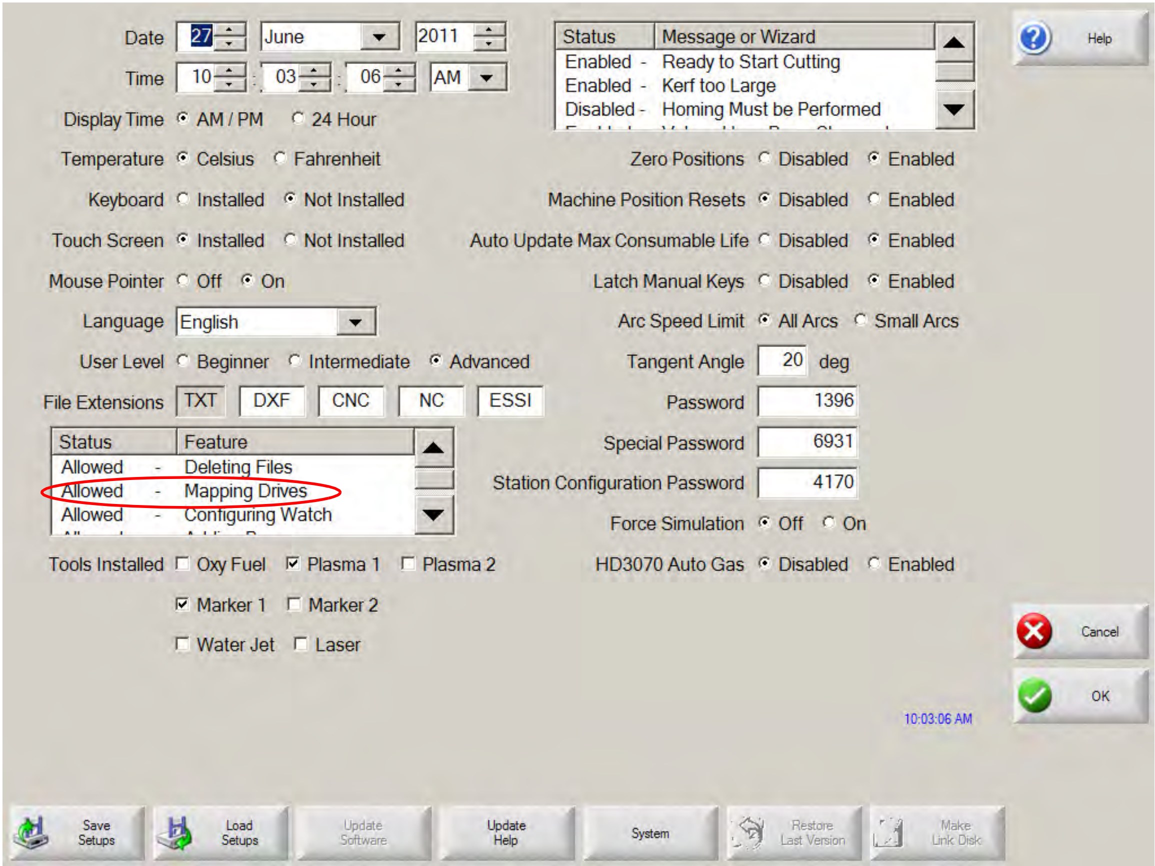
2. Choose Start Menu > Windows Explorer > Tools > Map Network Drive.

3. Choose a drive letter, then the folder. Make note of the folder path. When you connect to the folder in Phoenix, you will need to enter the folder path.

4. Choose Reconnect at Logon.

5. Choose Finish to save the mapped network drive.

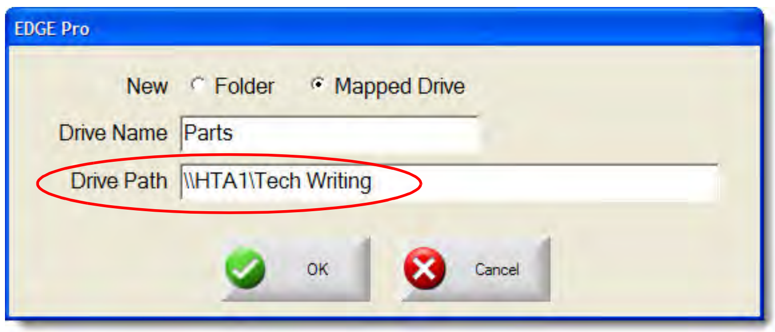
Mapping Drives and Adding Folders must both be set to Allowed in the Status/Feature list in the Special Setups screen.



Adding a Folder in Phoenix

1. From the Main screen, choose the Files soft key.
2. Double-click the blue message or press Right-Shift+F8 to add a folder.
3. Choose Mapped Drive.
4. Enter a Drive Name. This is the name that appears in the Load Files list.
5. Enter the actual path to the drive, not a drive letter. The drive path is formatted \\servername\foldername.

   Windows maintains a more consistent connection to the drive when you use the \\servername\foldername format than when you use a drive letter.
6. Choose OK.



Section 12

Serial Ports

Hypertherm CNCs support two serial ports that can be set for either RS-422 or RS-232 communications. Both serial ports are set to RS-422 at the factory. To change either serial port to RS-232, you must change a jumper setting on the serial isolation board that is inside the CNC. The jumper positions are clearly marked on the board.

CAUTION

Set the port for the chosen type of serial operation (RS-422 or RS-232) before connecting compatible devices.

The serial ports in the CNC are specifically designed to operate with a standard nine-pin serial port connector. The following table provides specifications for these ports:

Channel Type	Optically isolated RS-232C or RS-422
Information Code	ASCII
Baud Rate	User-selectable up to 115.2K baud
Number of Start Bits	1
Number or Stop Bits	1
Word Length	User-selectable 7 or 8 bits
Parity	User-selectable none, even or odd
Data Synchronization	XON (Control-Q) / XOFF (Ctrl/S)
Time Out	User-selectable in one-second increments
Transmit Delay	User-selectable in 0.01-second increments
Rear Panel Connector	IBM-PC/AT compatible 9-pin D-type female

Control RS-232C DB-9 Pinout

Pin	Signal Name	Description
1	Shield	Chassis ground
2	TxD	Transmit data to external device
3	RxD	Receive data from external device
4		No connection
5	Common	Ground
6		No connection
7		No connection
8		No connection
9		No connection

RS-232C Connections to Host PC with 9-pin D-type Connector

Host PC		Control	
Signal Name	**DB-9 Pin**	**Signal Name**	**DB-9 Pin**
Shield	1	Shield	1
RxD	2	TxD	2
TxD	3	RxD	3
Common	5	Common	5

RS-232C Connections to Host PC with 25-pin D-type Connector

Host PC		Control	
Signal Name	**DB-25 Pin**	**Signal Name**	**DB-9 Pin**
Shield	1	Shield	N/C
RxD	3	TxD	2
TxD	2	RxD	3
Common	7	Common	5

Control RS-422 DB-9 Pinout

Pin	Signal Name	Description
1	Shield	Chassis ground
2	TxD-	Transmit data - to external device
3	RxD-	Receive data - from external device
4	TxD+	Transmit data + to external device
5	Common	Ground
6		No connection
7	RxD+	Receive data + from external device
8		No connection
9		No connection

RS-422 Connections to Host PC with 9-pin D-type Connector

Host PC		Control	
Signal Name	**DB-9 Pin**	**Signal Name**	**DB-9 Pin**
Shield	1	Shield	Not connected
RxD-	3	TxD-	2
TxD-	2	RxD-	3
RxD+	7	TxD+	4
TxD+	4	RxD+	7
Common	5	Common	5

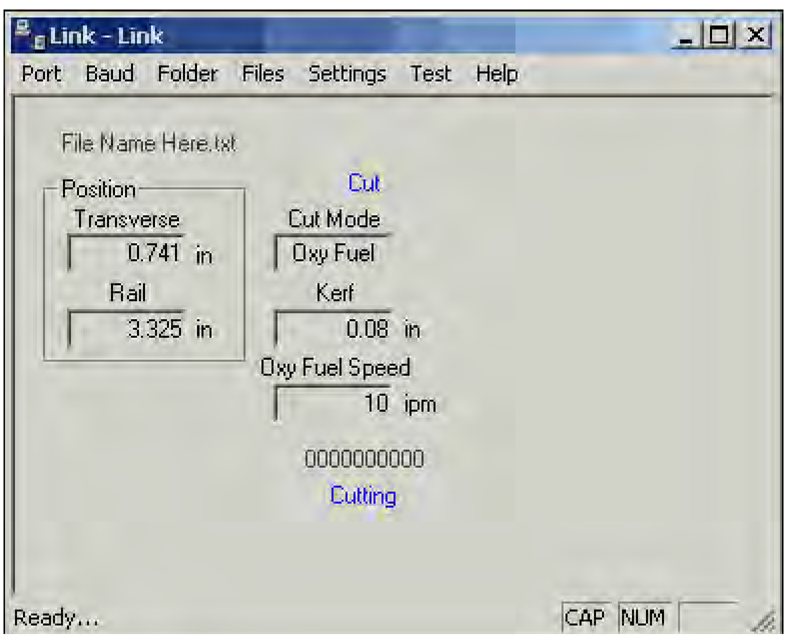
RS-422 Connections to Host PC with 25-pin D-type Connector

Host PC		Control	
Signal Name	**DB-25 Pin**	**Signal Name**	**DB-9 Pin**
Shield	1	Shield	Not connected
RxD-	3	TxD-	2
TxD-	2	RxD-	3
RxD+	14	TxD+	4
TxD+	16	RxD+	7
Common	7	Common	5

Section 13
Phoenix Link

The Phoenix Link communication software is used on the CNC for transmission of part and consumable database files between the CNC and the host computer. The program uses a 2X compression feature that allows the communication system to operate at speeds up to 230 K baud.

When the Phoenix Link communication software is operating, the following window displays at the host computer for configuration. After the software is running, the window can be minimized.



Port: Allows you to select the communication port on the host computer.

Baud: Allows you to select the baud rate for communication on the host computer. The minimum baud rate is 9600 bps; the maximum baud rate is 115,200 bps.

Folder: Allows you to change the path of the Master folder (in the Parts folder). Subfolders and the part files within this folder are viewed at the CNC.

Files: Allows you to indicate which file extensions are acceptable for part program files that are downloaded by the link software and viewed at the CNC.

Settings: Allows you to configure the link to allow the Auto Reload (M65) feature to be enabled and recognized during communication. In addition, you can also select and configure communication with multiple CNCs using Multi-Drop.

Test: Tests the operation of the communication port. To test, simply connect the send and receive signals on the selected communication port by inserting a paper clip and click on the port to test. The status of the test is displayed in the lower left corner of the window.

Help: Displays the current version of the link software.

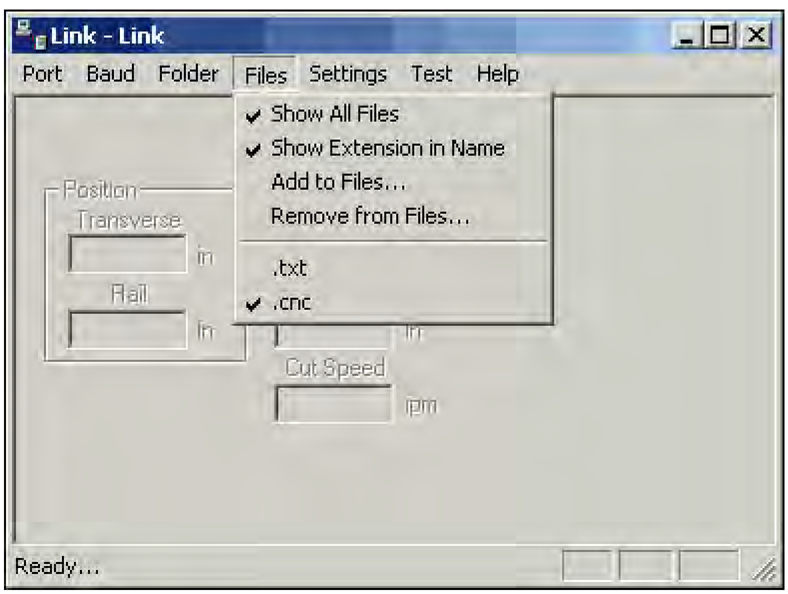
Files Menu

The Files setup parameter allows you to indicate which file extensions are acceptable for part program files that are downloaded by the link software and viewed at the CNC.

After a file has been saved on the CNC, it is assigned the .txt file extension.

Note: Take care when downloading files. Part programs with the same name, but different file extensions, are overwritten during multiple file downloads. A warning is displayed at the CNC before a file is overwritten.

The list of acceptable part program types is listed in the lower half of the of the Files option box. A checkmark (√) before the file extension indicates which selected file type will be displayed at the CNC. Note that the checkmark (√) in the file extension listing also indicates the file extension that is placed on a part program when uploading a file to the host.



The following options are available to define what part programs can be viewed at the CNC.

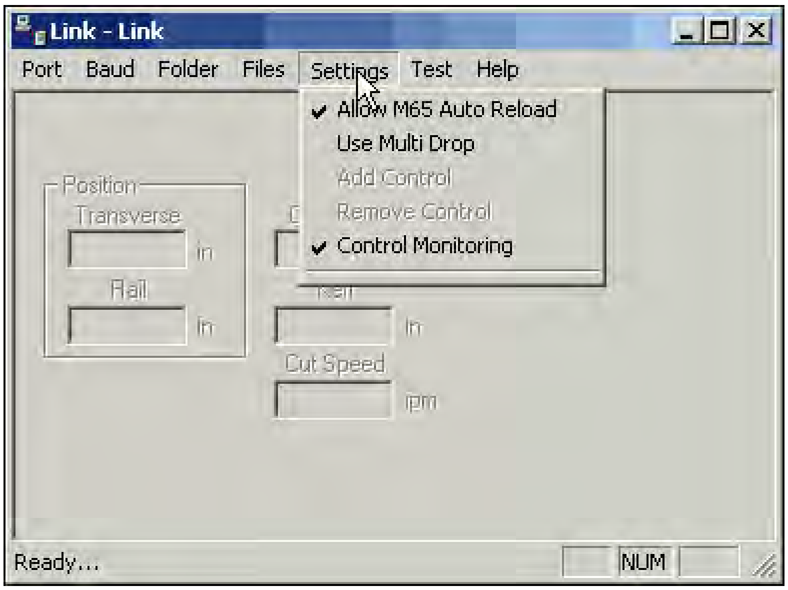
Show All Files: Allows all file types in the selected parts folder to be viewed at the CNC. A checkmark (√) before this feature indicates that it has been enabled.

Show Extension in Name: Allows the user to view the file extension as part of the file name at the CNC. For the file name *Job123.CNC*, the CNC displays *Job123_CNC* in the filename location of the download screen. The CNC displays *Job 123* if this feature is not enabled. This feature is helpful if multiple files that have the same file name but different file extensions are being used. A checkmark (√) before this feature indicates that it has been enabled.

Add to Files: Allows the user to add to the list of acceptable part program types to be viewed at the CNC. Up to four file types (extensions) can be added to the default .txt extension. A checkmark (√) before a file extension indicates which file extension will be added to any part file that is uploaded to the host.

Remove from Files: Allows the user to remove file extensions from the list of acceptable part program types.

Settings Menu



Allow M65 Auto Reload: Select this option to allow part programs to be partitioned into smaller part programs separated by the M65 code. During the download, the CNC downloads the individual sections of the part program and allows the operator to execute that section of the part. When that section of the program has been completed, the next section of the program is automatically downloaded for execution. The downloads continue until an end-of-program (M02) code is detected.

Use Multi Drop: Allows the link to be configured for communication with multiple CNCs through one communication port on the host PC. Up to eight CNCs can be supported with the Multi-Drop feature.

Note: The Using Phoenix Link parameter must be set to Yes at the CNC to enable this feature. In addition, specific hardware for the CNC (serial communication board Rev D or higher) and host PC may be required to support this feature. Contact your CNC supplier for more details.

Add Control: Adds a new CNC to the list of CNCs to communicate with. The corresponding CNC number must be assigned to the new CNC in the link set-up screen.

Remove Control: Removes a CNC from the list of CNCs with which the host PC communicates.

Control Monitoring: Allows you to view CNC status at the host PC. Status for file name, position, cut mode and cut information is displayed. This feature is only available with the Phoenix Link communication software.

Installation

The list below outlines the step-by-step procedure for the communication setup of the CNC and installation of the Phoenix Link communication software:

Minimum System Requirements

The system requirements for Phoenix Link are:

- Processor: Pentium 100MHZ
- Hard Drive Space: 0.5 MB
- Memory: 4MB
- Display: VGA
- Floppy Drive: 3.5 inches
- Operating System: MS Windows 95, 98, NT, 2000, ME or XP
- Serial Port: One RS-232 or RS422 serial port is required for each CNC

Software

1. At the CNC, access the Special Password screen.
2. Insert a USB memory device into the floppy drive or USB port.
3. Press the Make Link Disk soft key. Three files are transferred to the memory device:
   - Link.exe
   - Setup.exe
   - Readme.txt

   The Setup file creates a Link folder on the root directory of the host PC and copies the Link software into it. The Readme file contains additional information about the installation and setup of the Link software on the host computer.
4. When the light goes out on the floppy drive, transfer the floppy disk to the host computer and place the disk into the floppy (A) drive.
5. Click the Windows Start button and select Run.
6. Enter *A:\Setup.bat* in the Open field and click OK.

   The setup file expands the files and creates a Link folder on the root directory. The Link.exe file is copied to this folder.

   The Link software must be located and run on the PC that is connected to the communication cable coming from the CNC.
7. Click the Windows Start button and click Run.
8. Enter *A:\Link\Link.exe* in the Open field and click OK.

   After the link software has been launched, two other files are created. A Link.ini file, which contains the Link software configuration information, and a Parts folder, which is the default Master parts folder.

9. Create a shortcut to the Link.exe file on the desktop of the host PC.

10. At the host computer, create one or more folders in the Parts directory. Copy any part programs that will be accessed by the CNC in these folders.

    For example, to add the folders Workfile 1 and Workfile 2 to the parts directory, the hierarchy of the files would resemble the following structure:

    Path C:\Link\Parts\Workfile 1 and C:\Link\Parts\Workfile 2

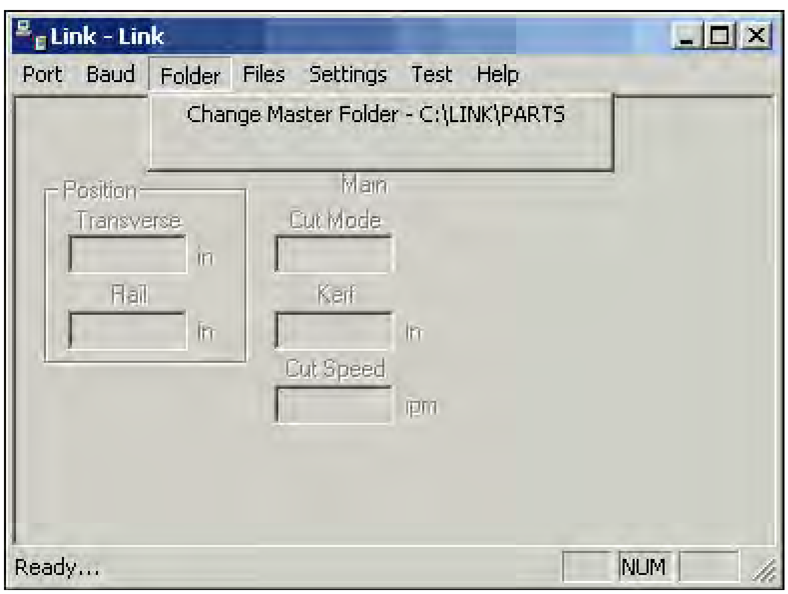
Any part programs that will be viewed at and downloaded to the CNC should be placed here. Only the sub folders Workfile 1 and Workfile 2 and the program files located within them are viewed at the CNC.

Change Master Folder

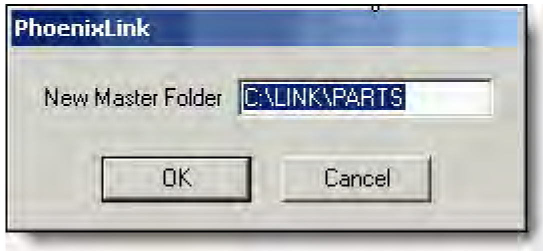
If the host computer currently has existing part programs and folders, the Link software can be configured to operate with the current folder names. This master part program folder can be located on the host PC or on another PC that is networked to the host PC.

The Link software must be located and run on the PC that is connected to the communication cable coming from the CNC. This is accomplished by configuring the Link software through the Change Master Folder feature to point the link to the folder that contains the required part programs.

1. Launch the Link software at the host computer to view the Link window.



2. Select Files > Change Master Folder.

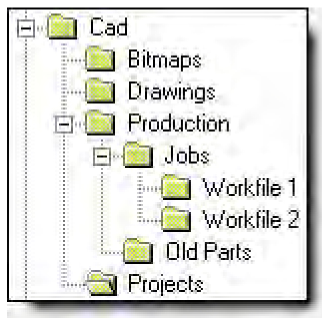


3. To change the master file from the current folder, enter the selected path in the New Master Folder field and click OK.

4. Use the same procedure to select a master folder on a network.

   For example, if the folders that contain the part programs are currently located in a folder named Jobs that is located on the PC named R drive, the Link.exe could be configured with the Change Master folder option to access these files.

   Example path: R:\Cad\Production\Jobs



Again, start by launching the Link Software at the host computer. Select Files\Change Master Folder.

The Master file may now be changed from the current folder to the selected “Jobs” folder by typing in the path R:\Cad\Production\Jobs. Select OK to accept the change.

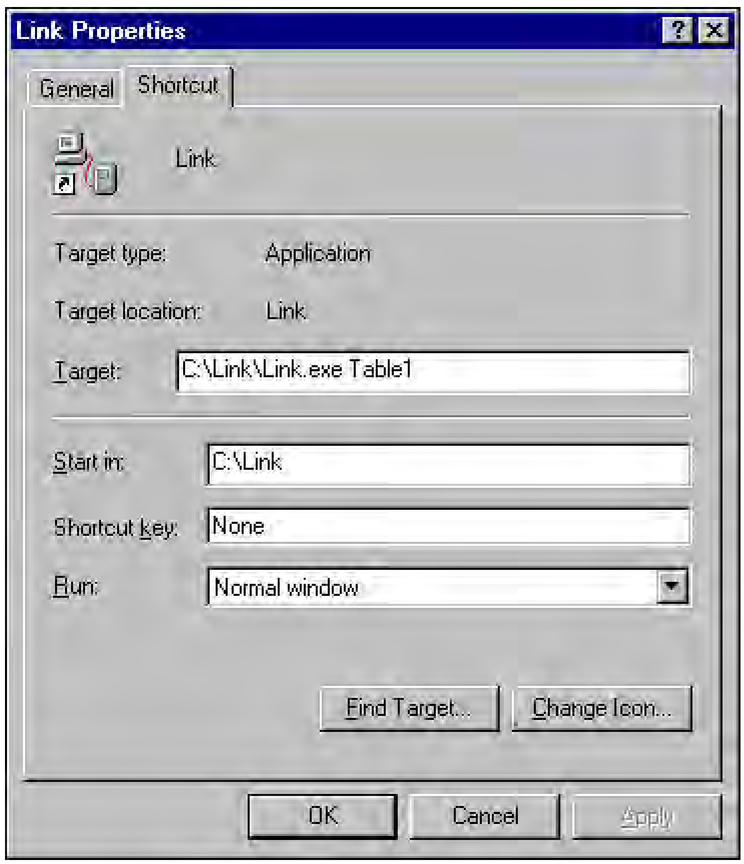
The final step to setup the Link software is to configure the Link software for Port, Baud, Files and Settings to match those being used at the CNC. It is recommended that a shortcut to the Link.exe be placed on the desktop of the host PC for ease of use.

Operating Multiple Links

To connect more than one CNC to the host PC without the use of the Multi-Drop feature, you can install Link software specifically for each CNC. Each version of Link that operates requires its own, dedicated communication port on the host PC.

1. Create individual folders for each cutting table.
2. Copy the Link.exe file into each folder.
3. Create a shortcut for each Link.exe on the desktop.
4. Right-click on each shortcut to open the shortcut properties dialog box.
5. Add to each shortcut the target command line information to indicate with which table the Link will be communicating.

In the target command line example below, *Table1* has been added to the end of the command line to indicate the Link will be specific to communications with Table 1.



This “table” information, added to the Target command line, is added to the title bar of the Link window. The Link can now be launched from the shortcut and configured specifically for this “table.” When the Link is closed, a “Table1.ini” with the specific Link setup file will be saved in the folder.

Continue to configure each additional Link for communication to each CNC. The separate Link files can be configured to point to the same or different master parts folders.

Hardware

To set up communication, place the CNC next to the host computer and connect them with a short communication cable. After communication is successfully established, the CNC can be moved to the cutting table and connected to the host computer with the appropriate cable or short-distance modem system.

1. Configure the selected RS232/RS422 serial port on the CNC for operation, as described in the Serial Port information of this guide.

2. Test the CNC's communication port in the Diagnostic screen to confirm proper operation of the serial port. Directions for testing of the serial port are described in the Diagnostic section of this guide.

3. Enter the CNC's Link setup screen to select the use of the desired communication port, select the baud rate and select to use Phoenix Link. Start at the lowest baud rate and increase it until the maximum is achieved. If you select a baud rate that is too high characters or information can be lost and error messages will be displayed.

4. Enter the CNC's Special Password screen to add the file extension of the part programs that are being used to the current list of acceptable file extensions.

5. Load and configure the Link software at the host computer as described in the Software section.

6. Test the host computer's selected serial port with the "Test Port" feature of the Link software as outlined in the Phoenix Link software overview.

7. Connect the CNC to the host computer's selected communications port with the appropriate cable or modem system.

Operating Phoenix Link

The Phoenix Link software must be running on the host computer to communicate with your CNC.

1. Start the Link software at the host PC from the desktop shortcut or select the Windows Start Button, then select Run.

2. From the opening prompt, enter C:\Link\Parts\Link.exe (or the appropriate path to the Link.exe) and click OK.

   The Phoenix Link window appears. The Link software window can be minimized during operation.

3. At the CNC, press the Files soft key and select to Upload or Download to Host. As the CNC is connecting to the host computer, a status display in blue text for the communication can be viewed in the lower right corner of the screen.

4. The CNC will initialize the port, open the port, and try to contact the host. If this attempt at communication fails, an error message is displayed in red. If the attempt is successful, the CNC downloads the available folder and part file names to be viewed on screen.

   Note: If you select No in the Show Host File Names field at the CNC, only folder names can be viewed.

Common Errors

The following list describes common errors that you can encounter when you install a communication system.

- The port selected at the host computer has already been assigned to another software program.
- The host and CNC have been configured to different setup values.
- The cable has been connected to the incorrect port or software has been configured to the incorrect port.
- The selected modem or cabling system being used has swapped the transit and received signals, causing a communications failure.
- Trying to transmit over too long a cable without a modem or using incorrect wire for the distance being traveled.
- Trying to communicate at too high a baud rate for the modem system selected.
- The correct file extension has not been entered at the CNC's Special Password screen, preventing part programs from being displayed at the CNC.

Error Messages

The following list provides descriptions of common error messages and possible causes.

Unable to Open Port (CNC)

The selected communications port could not be opened. The wrong port has been selected or the communications port has failed.

Unable to Open Port (host)

The selected communications port could not be opened. The wrong port has been selected, the communications port has failed or another software application is using the port.

Unable to Initialize Port

The selected communications port could not be initialized. The wrong port has been selected, the communications port has failed or another software application is using the port.

Port Failed

The Port Failed error message is displayed if the communications port test has failed for the selected port. There could be a failure of the communications port or a fault in the test wire connection that is incorrectly connected to the proper send and receive pins location.

Host Not Responding

This message is displayed if the communication port has successfully opened the port but has failed to establish communication with the Link software. There could be a fault with the cable connection, cable configuration, or the host computer's communications port.

Communication Failed

This message is displayed if, after establishing communication with the host, an expected message from the host is not correct.

Communications Time Out

This message is displayed if the specified amount of time allowed for a response from the host computer expires. This amount of time is configured in the Link setup screen. There could be a fault in the cable connection, cable configuration, or the host computer's communications port.

Checksum Error

This message is displayed at the CNC after a part has been uploaded or downloaded if the checksum calculated by the host and the CNC do not match. The checksum calculation is performed by adding the numeric values of the ASCII codes that are used. Possible reasons for this error are incorrect cabling or transmission at a speed that is too high for the selected communications system.

Warning: The Master Folder Selected does not Contain any Folders

This error message is displayed at the host computer if the selected master folder does not contain any subfolders. You can choose to add subfolders if they do not exist. If subfolders do exist, the path to the master folder may be incorrect.

For additional assistance in trouble shooting the Phoenix Link in the event of a failure, contact your CNC vendor.

Section 14

Aligning Plates

The Automatic Plate Alignment (APA) feature detects the edges of a rectangular plate and calculates the degree of skew to align a part on the plate. Two types of APA sequences are available:

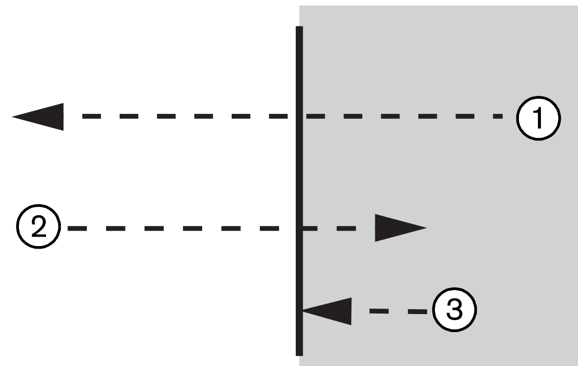
- **Five-point alignment** is the default type of APA and detects plate alignment, skew, and size.
- **Three-point alignment** detects plate alignment and skew.

Notes about APA

- The APA sensors must be assigned in I/O setups. The logic states must be set so that the input is active when the sensor is over the plate.
- The sensor moves fast to the outward edge of the plate until the sensor detects the edge and switches to OFF. For accuracy, motion reverses until the sensor is activated again and switches to ON. See Sensing Sequence.
- If User Defined is selected as the Manual Offset value on the Manual Options screen, an X/Y tool offset value from the Setup screen is required to compensate for the distance between the master torch and the APA sensor.
- Scrap clearance, if required, is entered on the Align screen. In addition, Corner to Align With on the Align screen sets the start corner location of the program.
- If your machine requires homing, this function should be performed before APA.
- Some sensors may detect the table slats as an extension of the plate. If this is a problem in your application, configure 2 sensors. The first sensor to detect the edge of the plate (switch to OFF) is used for all plate detection at that location.
- An appropriate X/Y offset for the two sensors is recommended to minimize the possibility that both sensors could detect the slats as an extension of the plate. The distance between the first and second sensor is programmed in tool offset #11 on the Setup screen.
- Sensing radius should be programmed according to the radius of the sensor. If dual sensors are used, the radius applies to both sensors.
- You can insert program codes into the part program to automatically configure 3-part APA for that part. For more information, see *Program Code*, later in this section.

Sensing Sequence

1. The sensor moves off the plate at high speed (status = OFF).
2. The sensor returns to the plate at medium speed (status = ON).
3. The sensor moves off the plate at slow speed (status = OFF). The edge of the plate is detected and recorded.



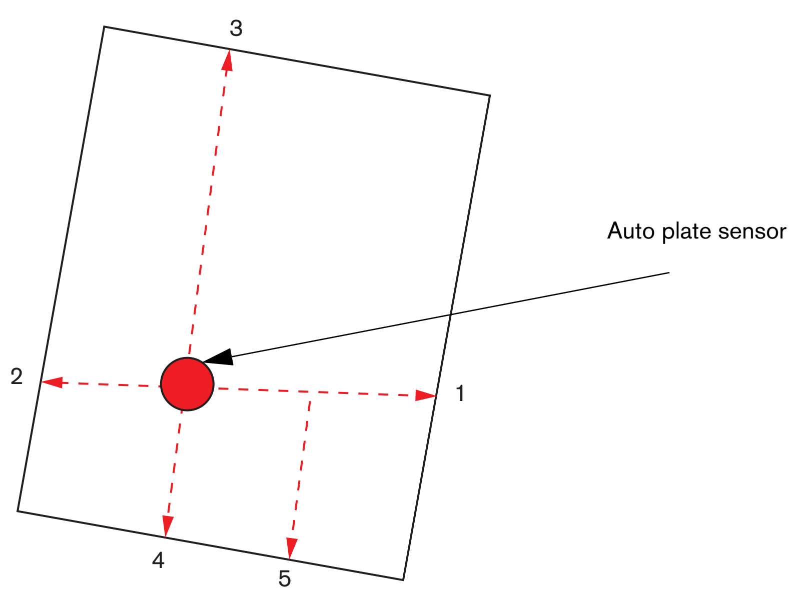
Five-Point Alignment

5-point alignment uses 5 reference points to calculate skew and plate size. The CNC commands the Sensor PHC to move around the plate in search of the edge at 5 points. After the 5 reference points have been detected, the CNC positions the torch in the correct location to start the part program with the calculated skew.

To start 5-point alignment:

1. Position the Plate Sensor above the plate manually. The Auto Plate Align Sensor Input should be ACTIVE.
2. Select Part Options > Align. Verify that the correct corner has been selected. Edit parameters for Skew alignment.
3. Press the Automatic Plate Align soft key on the Part Options screen.

The sensor moves across the plate to detect the reference edges of the plate. Each position is recorded as one of the reference points. Motion continues until all 5 points are detected.



After the sensor detects all 5 reference points, the CNC positions the sensor or the torch at the starting point of the part program.

The values for Scrap Clearance and Start Corner on the Align screen contribute to the calculation of the final position for the alignment and the start point of the part program. The APA function also enters the values for the dimensions of the plate in the appropriate fields in the Setup screen.

The part program can be executed automatically at the end of the alignment if Auto Start after APA is enabled on the process Setup screen or it can wait for the start of the cycle.

Three Point Alignment

During a 3-point alignment, the CNC commands motion of the sensor around the plate and searches for the edges of the plate at 3 reference points After the sensor detects the 3 reference points, the CNC positions the sensor or the torch at the starting point of the part program with the calculated alignment and skew.

Program Code

To configure APA within a part program, you must add EIA program codes to the beginning for the part program. The parameters remain in effect until they are changed or the CNC is restarted.

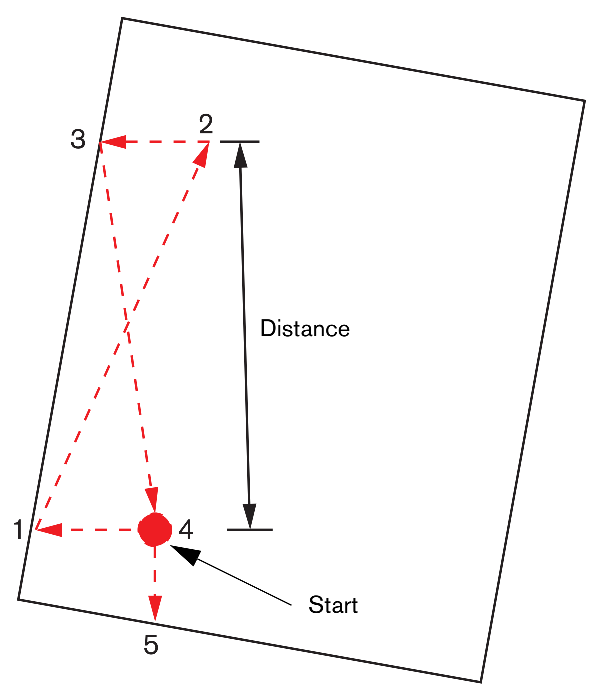
For more information on configuring APA in a part program, see the *Programmer's Reference*.

Motion Path

The following drawing shows a typical edge detection sequence for APA with the following selections on the Align screen:

- Corner to Align: Lower left
- Skew Point: Upper left

Change the values for these fields to specify a different starting point and sequence.



To start the 3-point alignment:

1. Load a part program with the appropriate program codes included at the beginning of the program.

2. Position the plate sensor above the plate manually. The Auto Plate Align Sensor Input should be ACTIVE.

3. Select Part Options > Align. Verify that the correct corner has been selected. Edit parameters for Skew alignment.

4. Press Auto Plate Alignment.

The sensor moves across the plate to detect the reference edges of the plate. Each position is recorded as one of the reference points. Motion continues until all 3 points are detected.

After the sensor detects all 3 reference points, the CNC positions the sensor or the torch at the starting point of the part program.

The values for Scrap Clearance and Start Corner on the Align screen contribute to the calculation of the final position for the alignment and the start point of the part program. The APA function also enters the values for the dimensions of the plate in the appropriate fields in the Setup screen.

The part program can be executed automatically at the end of the alignment, or it can wait for the start of the cycle, as specified in Program Code selection list on the Setup screen.

Section 15

Oxyfuel Application

This section describes the operation, layout, and setup of a sample oxyfuel cutting system. Because of the variation between cutting systems, Hypertherm CNCs provide built-in flexibility for multiple methods of system setup, operation, and part programming. Since all cases cannot be described here, this section makes the following assumptions:

- The cutting operations are being controlled by the CNC, not by a PLC or other external logic.
- The cutting system supports both low preheat and high preheat fuel gas channels.
- The CNC supports either the Hypath or SERCOS interface to provide sufficient I/O for the oxyfuel application.

In this section you will find:

- An overview of how the CNC executes the oxyfuel process
- An illustration of a sample two-torch system
- Station setup for oxyfuel
- A detailed, step-by-step description of the oxyfuel cut sequence
- Definitions of the inputs and outputs used for oxyfuel cutting systems, and a ladder logic diagram showing the I/O
- Oxyfuel setup instructions for the CNC
- Advanced features for oxyfuel cutting (analog outputs for gas control, process overrides, and staged pierce function)

Oxyfuel part programs can use process selection variables to load the cut chart. However, oxyfuel cut charts are not supported in the Cut Pro Wizard. For more information about oxyfuel process selection variables, see the *Phoenix Software V9 Series Programmer's Reference* (806420).

For more information on generic, numbered, and station I/O, see *Chapter 4 Station Setup*.

Oxyfuel overview

Hypertherm CNCs provide an oxyfuel process that controls fuel gas, pierce-oxygen, and cut-oxygen in several stages. These stages activate outputs to run an oxyfuel cutting system. The CNC executes the oxyfuel cut sequence in this order for each cut. The stages are controlled with timers set on the Oxyfuel Process screen.

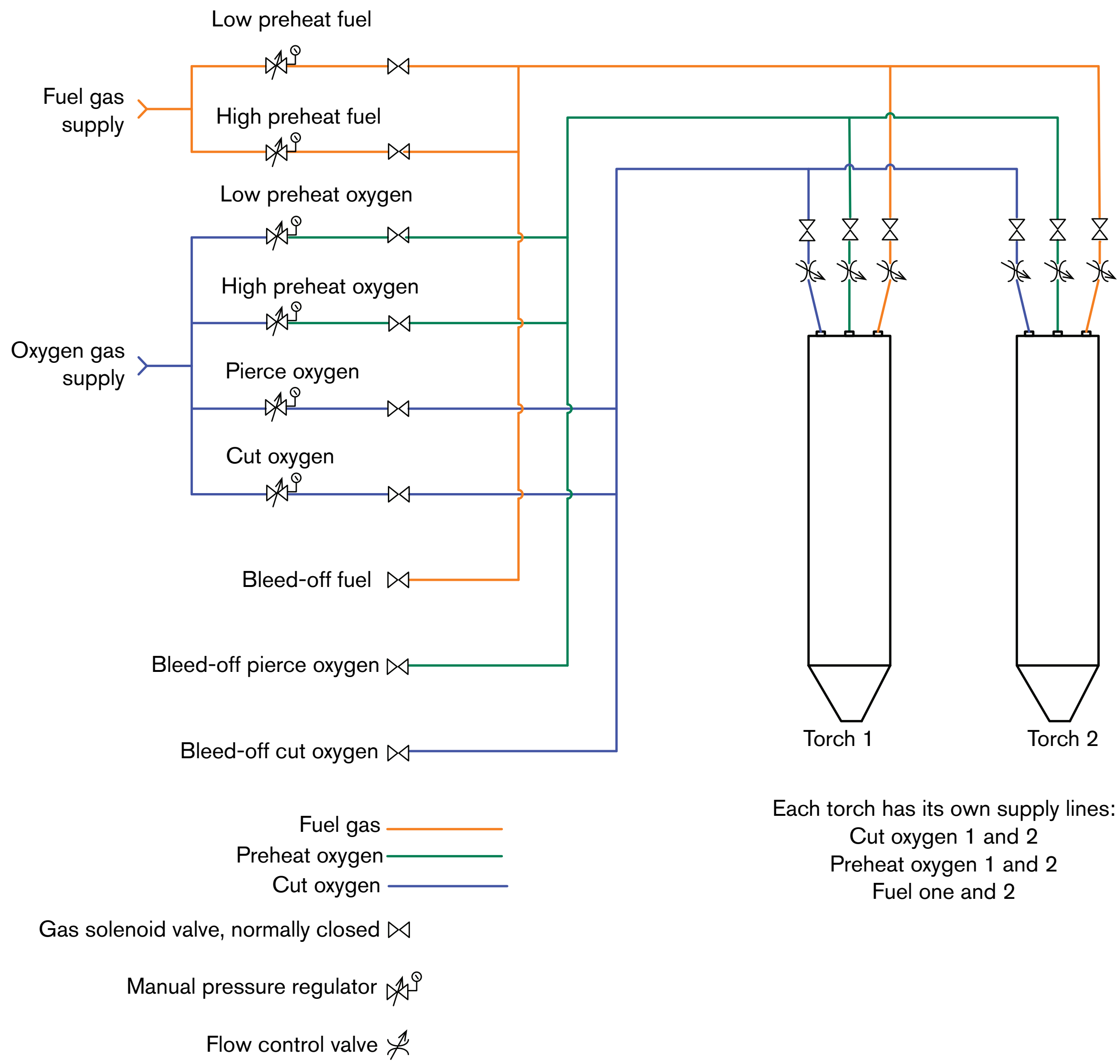
Stage	Output	Definition
Cycle Start	None	Starts the oxyfuel sequence. Upon cycle start, the torch moves to the first pierce point in the part program.
Ignite torch.	Torch Ignition	Operates an igniter circuit used to ignite the torch.
Preheat the workpiece.	Low Preheat Control	Turns on the low pressure fuel gas used to ignite the torch.
	High Preheat Control	Turns on the high pressure fuel gas used to preheat the workpiece and assist in cutting.
Pierce the workpiece.	Pierce Control	Turns on the pierce oxygen, a low pressure oxygen channel that accelerates the preheat process.
Cut the workpiece.	Cut Control	Turns on the cut oxygen, a high pressure oxygen channel for cutting the workpiece.

Note: The Preheat Control output is for use in a system where a low preheat fuel gas channel is not available and one output controls the preheat fuel gas. Use Preheat Control instead of High Preheat Control in this type of system. See *Low preheat fuel gas options* later in this section for more information.

Two-torch oxyfuel system diagram

The following picture shows a sample two-torch oxyfuel system with three gas channels.

> **Note:** This picture is a functional drawing and is not intended as a recommendation for system design. You will need to procure components that are rated for your individual cutting system and your production needs.



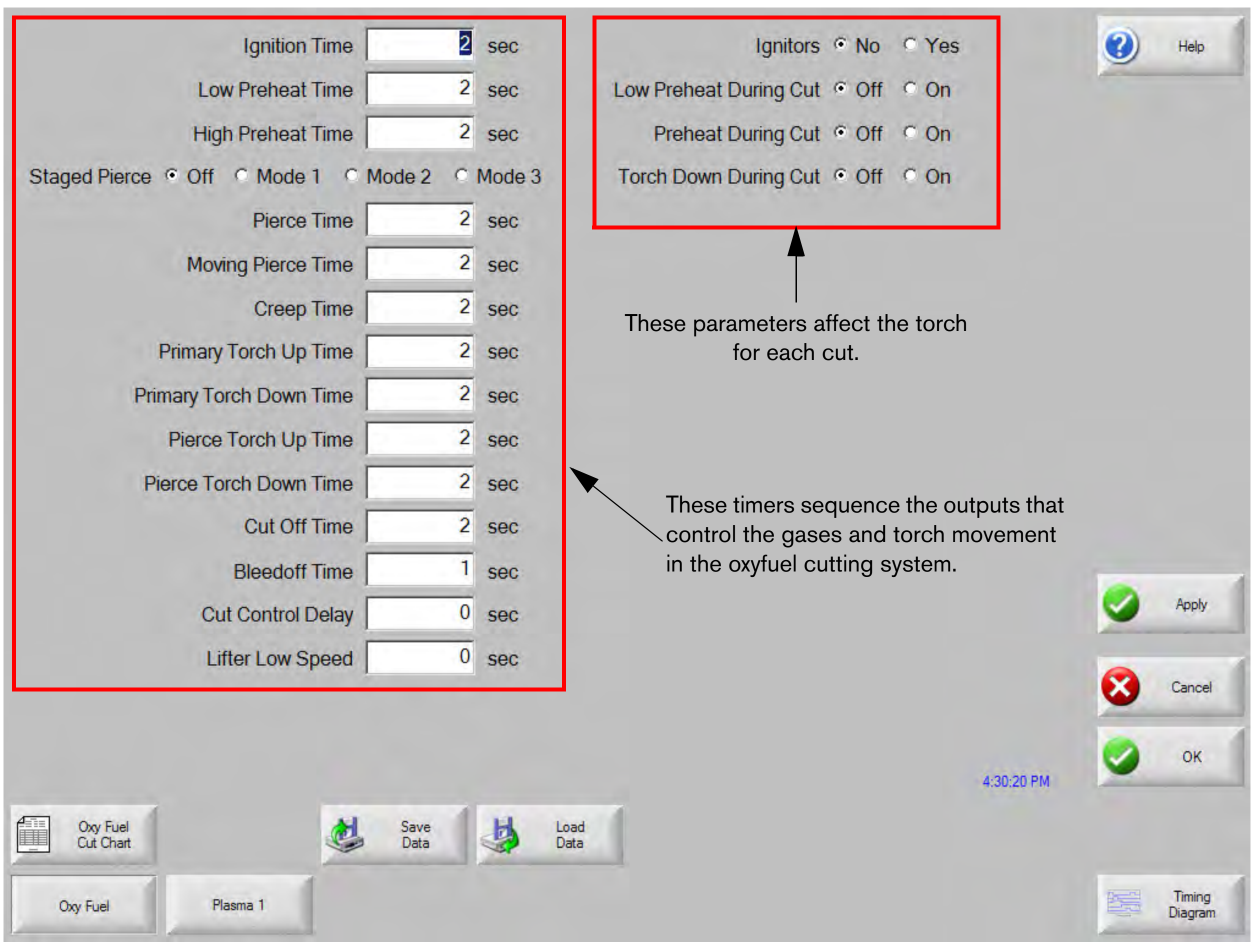
Low preheat fuel gas options

Hypertherm CNCs provide the ability to control a low preheat fuel gas channel. This channel can be used in different ways.

- Wire the Low Preheat Select input to the operator console. This input can function as an on/off switch for the oxyfuel system. When Low Preheat Select turns on, the CNC turns on the Low Preheat Control output and opens the low pressure fuel gas channel. In some oxyfuel cutting systems the low pressure fuel gas remains on throughout operation so that the operator can manually change the fuel gas pressure or the torch can remain lit between cuts.
- On the Oxyfuel Process screen, the Low Preheat during Cut parameter forces the Low Preheat Control output to stay on. You can use this method as an alternative to having the Low Preheat Select input wired to the operator console.

Oxyfuel cut sequence

The oxyfuel cut sequence follows these steps. All timers referred to below are located on the Oxyfuel Process screen (Setups > Process > Oxyfuel).



To start the oxyfuel cut sequence, select Oxyfuel Cut Mode on the Main screen, load a part file, and press Cycle Start. The torch moves to the first pierce point in the program.

The M07 code (Cut On) runs in the part program. Torch Down output turns on.

- Torch Down output remains on until either the Primary Torch Down Time elapses or the Torch Down Sense input turns on.
- Status Message: *Lowering Torch*

Ignition output turns on.

- Ignition output remains on until the Ignition Time expires.
- Status Message: *Igniting Torch*

Low Preheat Control output turns on if Low Preheat Time is used.

- This output may already be on because the Low Preheat Select input was turned on at the beginning of the sequence.
- Low Preheat Control output remains on until the Low Preheat Time elapses.
- Status Message: *Low Preheat*

High Preheat Control output turns on.

- The preheating of the workpiece occurs to prepare for piercing.
- High Preheat Control output remains in this state until the High Preheat Time elapses or you press Cycle Start. Cycle Start bypasses preheats and allows manual control of piercing.
- To have the High Preheat Control output remain on while piercing *and* cutting, set Preheat During Cut to Yes on the Oxyfuel Process screen. Use this option if the oxyfuel torch requires high pressure fuel gas for cutting.
- Status Message: *High Preheat*

Pierce Control Output turns on.

- Pierce Control output activates the pierce-oxygen channel.
- The Pierce Control output can be used to decrease the preheating time of the workpiece.
- Pierce Control remains on until the Pierce Time elapses.
- High Preheat Control remains on if Preheat During Cut is active.
- Low Preheat remains on.
- Status message: *Piercing*

Cut Control output turns on during piercing.

- The Cut Control output activates the cut-oxygen channel.
- The Cut Control output turns on after the Cut Control Delay time elapses. This timer allows the Pierce Control output time to start piercing the workpiece. Then Cut Control turns on to provide a boost by turning on the cut-oxygen channel.
- Status message: *Piercing*

Creep motion begins when piercing completes.

- Creep motion is a percentage of travel speed set in the Machine Setups > Speeds screen.
- Creep motion continues until the Creep Time elapses.
- Cut Control remains on, Pierce Control turns off, High Preheat remains on if Preheat During Cut is on, and Low Preheat remains on.
- Status Message: *Creeping*

Machine accelerates to the oxyfuel speed (cut speed).

- Cut Control remains on, High Preheat remains on if Preheat During Cut is active, and Low Preheat remains on.
- Status Message: *Cutting*

M08 code (Cut Off) runs in the part program.

- Cut Control shuts off, High Preheat shuts if Preheat During Cut is active, and Low Preheat remains on.
- Low Preheat prevents the flame from extinguishing.

Torch Up output turns on.

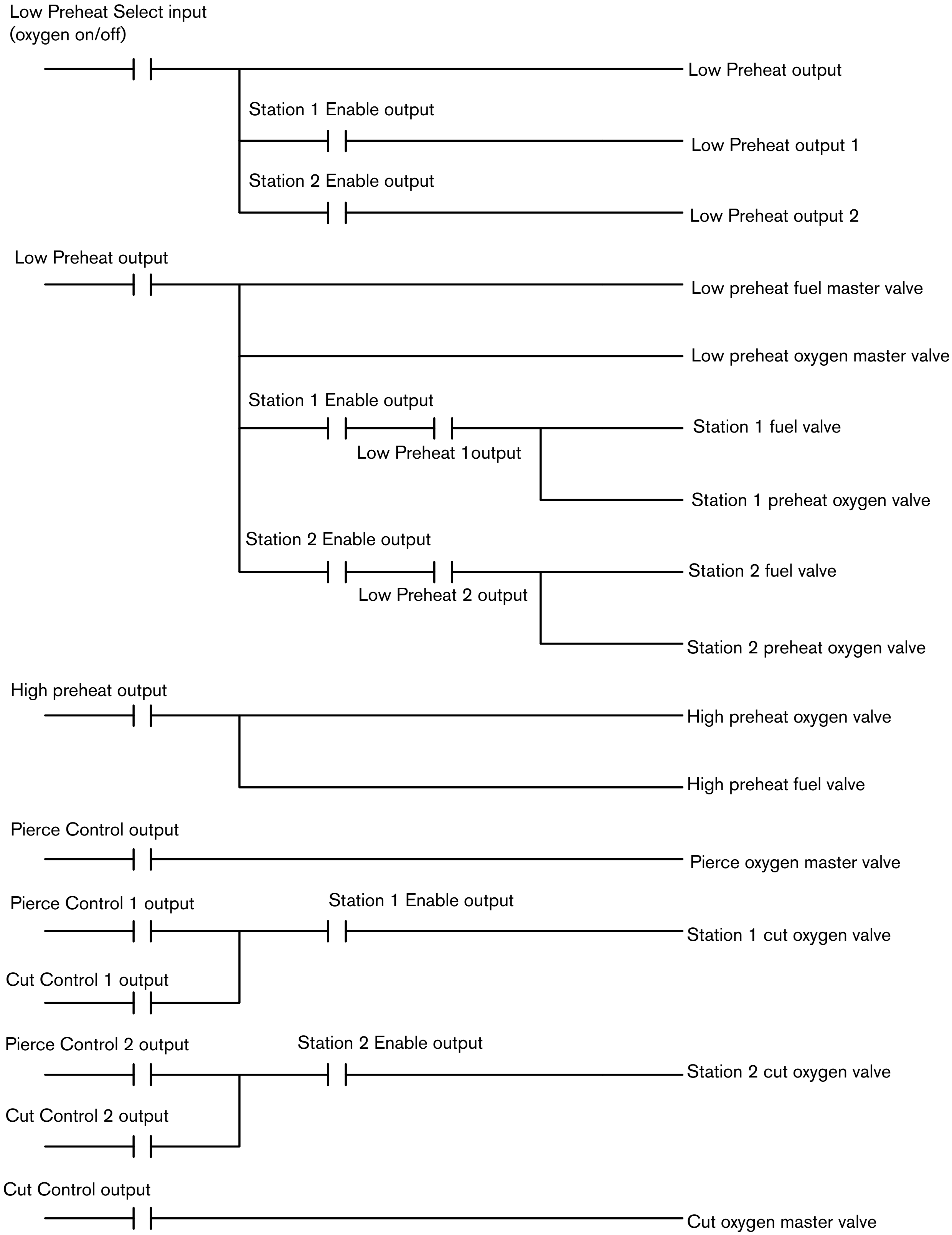
- Torch Up output remains on until the Primary Torch Up Time elapses or until Torch Up Sense input activates, whichever occurs first.
- Status Message: *Raising Torch*

Torch rapid traverses to the next pierce point.

- Low Preheat is still active due to Low Preheat Select input.
- Status Message: *Traversing*

The entire process repeats when the torch moves to the next pierce point.

An oxyfuel ladder logic diagram is shown below. Each "rung" in the ladder shows the internal logic of the CNC I/O and how it would control the fuel gas and oxygen valves in an oxyfuel cutting system.


Low Preheat Select input
(oxygen on/off)
Low Preheat output
Station 1 Enable output
Low Preheat output 1
Station 2 Enable output
Low Preheat output 2
Low Preheat output
Low preheat fuel master valve
Low preheat oxygen master valve
Station 1 Enable output
Station 1 fuel valve
Low Preheat 1output
Station 1 preheat oxygen valve
Station 2 Enable output
Station 2 fuel valve
Low Preheat 2 output
Station 2 preheat oxygen valve
High preheat output
High preheat oxygen valve
High preheat fuel valve
Pierce Control output
Pierce oxygen master valve
Pierce Control 1 output
Station 1 Enable output
Station 1 cut oxygen valve
Cut Control 1 output
Pierce Control 2 output
Station 2 Enable output
Station 2 cut oxygen valve
Cut Control 2 output
Cut Control output
Cut oxygen master valve

Oxyfuel inputs

Oxyfuel inputs can be used for switches on an operator console to control the oxyfuel cutting system. The screen below shows an example of input assignments for an oxyfuel cutting system with two torches.

Low Preheat Select: This input can be used as an oxyfuel on/off switch on the operator console of the CNC (a momentary push-button or a single-pole, single-throw (SPST) toggle switch). This input turns on the Low Preheat output while the switch remains in the ON position. With this input on, low pressure fuel gas will be delivered to the torch, and can be manually adjusted.

Cut Select: This input can be used as an ON/OFF switch on the operator console of the CNC (a momentary push-button or SPST toggle switch). This input turns on the Cut Control output while the switch remains in the ON position. With this input on, high pressure oxygen will be delivered to the torch and can be manually adjusted.

High Preheat Select: This input can be used as an ON/OFF switch on the operator console of the CNC (a momentary push-button or SPST toggle switch). This input turns on the High Preheat output while the switch remains in the ON position With this input ON, high pressure fuel gas will be delivered to the torch and can be manually adjusted.

Manual Ignition Select: This input can be used as an ignition switch (a momentary push-button) on the operator console of the CNC. It turns on the ignition output while the input is ON. This input can also be used to manually turn on the ignitors.

Lower Torch / Lower All Torches: This input can be a toggle switch that turns on the Torch Down output. The Torch Down output signals the oxyfuel torch lifter to lower the torch.

Raise Torch / Raise All Torches: This input can be used as a toggle switch that turns on the Torch Up output. The Torch Up output signals the oxyfuel torch lifter to raise the torch.

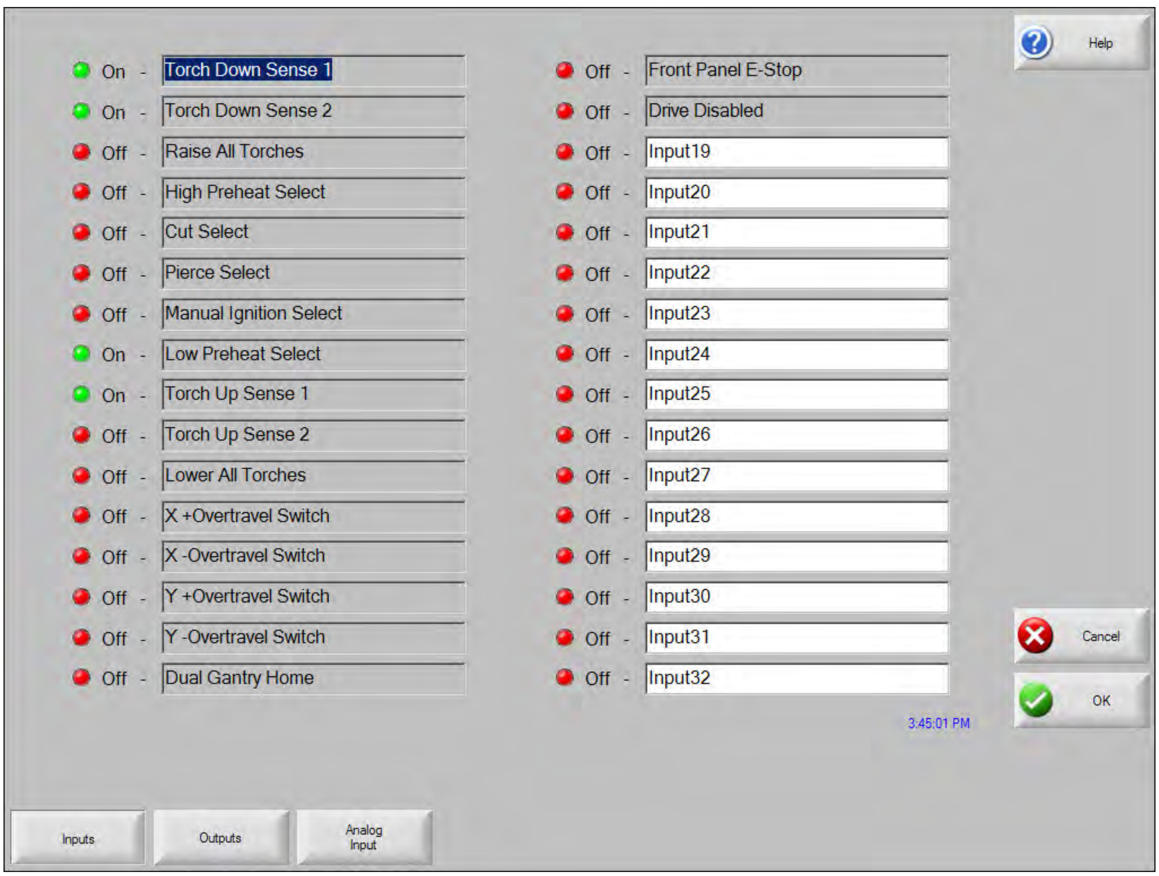
Torch Down Sense: When this input turns on, the CNC turns off the Torch Down output. This input is used with a lower limit switch or proximity switch on the oxyfuel torch lifter, but not as a safety switch. The Torch Down Sense input does not cause a fault on the CNC.

Torch Up Sense: When this input turns on, the CNC turns off the Torch Up output. This input is used as an upper limit switch proximity switch on the oxyfuel torch lifter, but not as a safety switch. The Torch Down Sense input does not cause a fault on the CNC.

Oxyfuel outputs

The oxyfuel outputs are controlled by timers on the Oxyfuel Process screen. The screen below shows an example of output assignments for an oxyfuel cutting system with two torches.

Note: In a cutting system that combines plasma and oxyfuel, when assigning I/O for oxyfuel cutting stations, use numbered I/O 9 – 20.



Torch ignition: Activates torch ignitors.

Timer: Ignition Time specifies the time the oxyfuel ignitor is turned on when igniting the flame.

Low preheat control: Activates the low pressure fuel gas channel.

Timers:

- Low Preheat Time specifies the number of seconds to preheat the workpiece before piercing using the low pressure fuel gas channel.
- Ignitors: Set Ignitors to No and Ignition Time to 0 to activate the Low Preheat Control output at the end of a cut and keep the torch lit during rapid traverse. Set Ignitors to Yes to turn off the Low Preheat Control output and re-ignite the flame at each next pierce point.
- Low Preheat During Cut specifies whether the Low Preheat is left on during cutting.

High preheat control: Activates the high pressure fuel gas channel.

Timers:

- High Preheat Time specifies the number of seconds to preheat the workpiece before piercing using the high pressure fuel gas channel.
- Preheat During Cut specifies whether the Preheat is left on during cutting.

Notes:

- When you run the part, you can use the Set, Extend, or Release soft keys to change either the Low or High Preheat time. To bypass the preheat timers completely, press Cycle Start twice.
- If the cutting system does not support the Low Preheat Control output, then High Preheat Control is used to keep the torch lit during rapid traverse when Ignitors is set to No and Ignition Time to 0.

Pierce control: Turns on the pierce oxygen, a low pressure oxygen channel that accelerates the preheat process.

Timers:

- Pierce Time specifies the number of seconds the Pierce Control output is on before lowering the torch to the cut height.
- Moving Pierce Time allows X/Y motion and specifies the number of seconds the Pierce Control output remains on.

Torch Up: Activates a relay for a non-servo motor to move the torch up.

Timers:

- Primary Torch Up Time specifies the number of seconds to raise the torch after completing each cut. The torch continues raising until this time elapses or the lifter reaches a limit switch that activates the Torch Up Sense input.
- Pierce Torch Up Time Sets the time for torch lift after piercing to clear a pierce puddle.

Torch Down: Activates a relay for a non-servo motor to move the torch up.

Timers:

- Primary Torch Down Time specifies the number of seconds to lower the torch at the beginning of each cut after torch ignition. The torch continues to lower until this time expires or the lifter reaches a limit switch that activates the Torch Down Sense input.
- Pierce Torch Down Time specifies the number of seconds to lower the torch for cutting. This timer should allow the torch to reach the cut height.

- Torch Down During Cut specifies whether the Torch Down output is left on during cutting. This parameter is can be used with a pneumatic lifter.

Cut Control: Activates the cut oxygen channel, a high pressure oxygen channel for cutting the workpiece.

Timers:

- Cut Off Time specifies the number of seconds for the Cut Control output to remain on at the end of a cut. Allows the torch to finish its cut and removes any lag (a slight angle that is created when the flame meets metal and bends). Using the Cut Off time provides time for the flame to become perpendicular before it is turned off.
- Cut Control Delay specifies the number of seconds the CNC waits before turning on the Cut Control output during piercing.

Bleed-off Gas: Controls a valve used to purge gas from the torch.

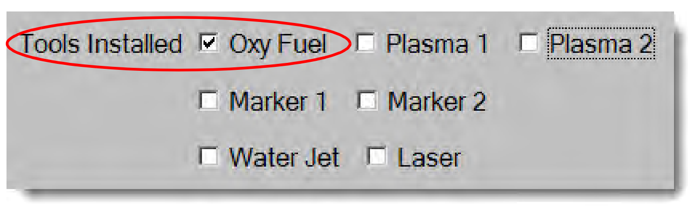
Timer: Bleed-off Time specifies the number of seconds the torch pauses to purge gas at the end of a cut before traversing to the next cut. This timer can overlap the Primary Torch Up timer.

Lifter Low Speed: Turns on with the Torch Down and Torch Up numbered I/O and allows a fine jog adjustment when the torch is near the workpiece. Lifter Low Speed turns off when the Lifter Low Speed time elapses.

Timer: Lifter Low Speed specifies the number of seconds for the Lifter Low Speed output to be on. The value for this timer should be less than the value for the Primary Torch Up Time and Primary Torch Down Time values.

Setting up oxyfuel

1. In the Special Setups screen (Setups > Password > Special Setups) select Oxyfuel as a Tool Installed.



   This selection activates oxyfuel as a Cut Mode, and activates the Oxyfuel Process screen and the Oxyfuel Cut Chart.

2. In the I/O screen (Setups > Password > Machine Setups > I/O), assign the inputs and outputs for your oxyfuel cutting system.

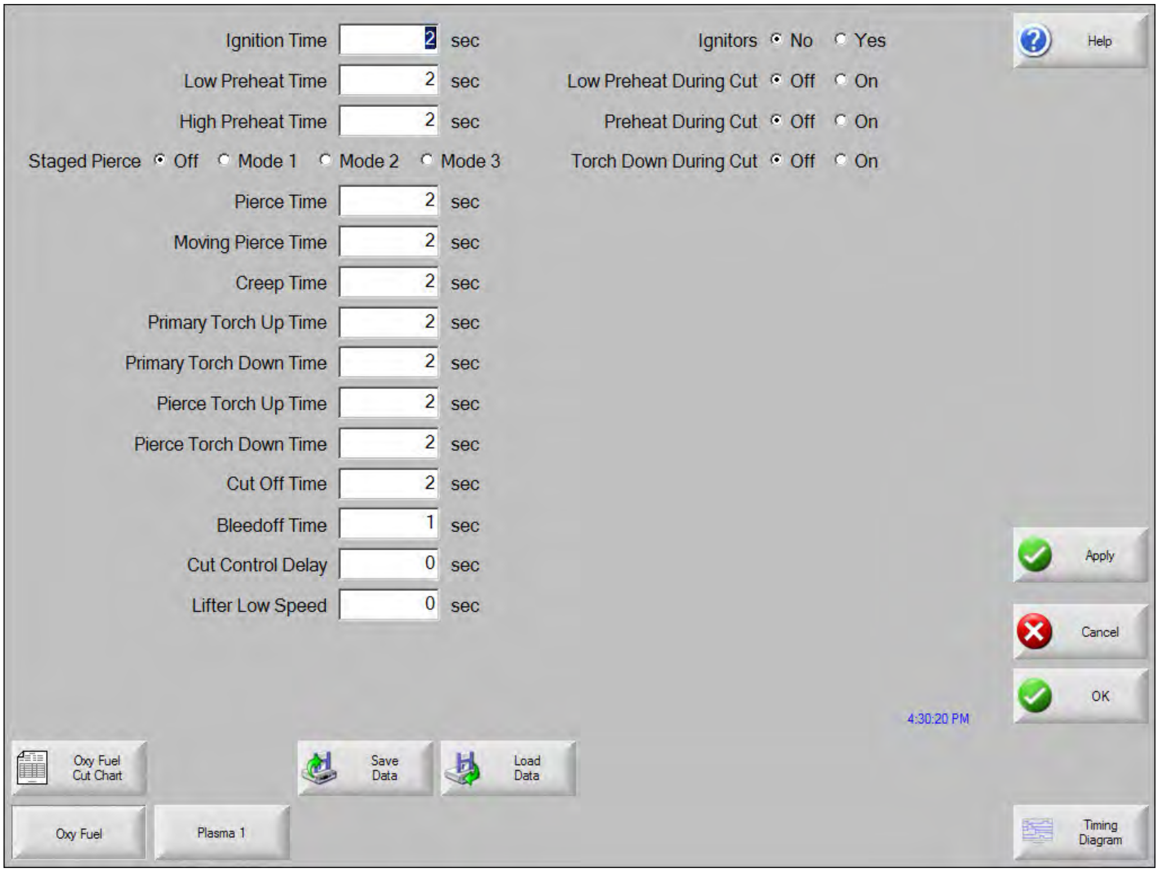
   The I/O assignments for the system depend on the number of oxyfuel torches and whether the CNC will be operating all the torches independently. Use the numbered I/O to operate each oxyfuel torch independently. Assign Station Select inputs if numbered I/O are used, one input for each station.

   Note: If the system does not use ignitors to light the oxyfuel torches, do not assign the Ignition output.

3. In the Oxyfuel Process screen, enter timer values for all preheats, piercing, creeping, and torch up/down movement. (Setups > Process > Oxyfuel).

   Select Yes to Ignitors if you will use the Ignitor output at every pierce point. If you will be using the Manual Ignition Select input to ignite all torches, select No to Ignitors.

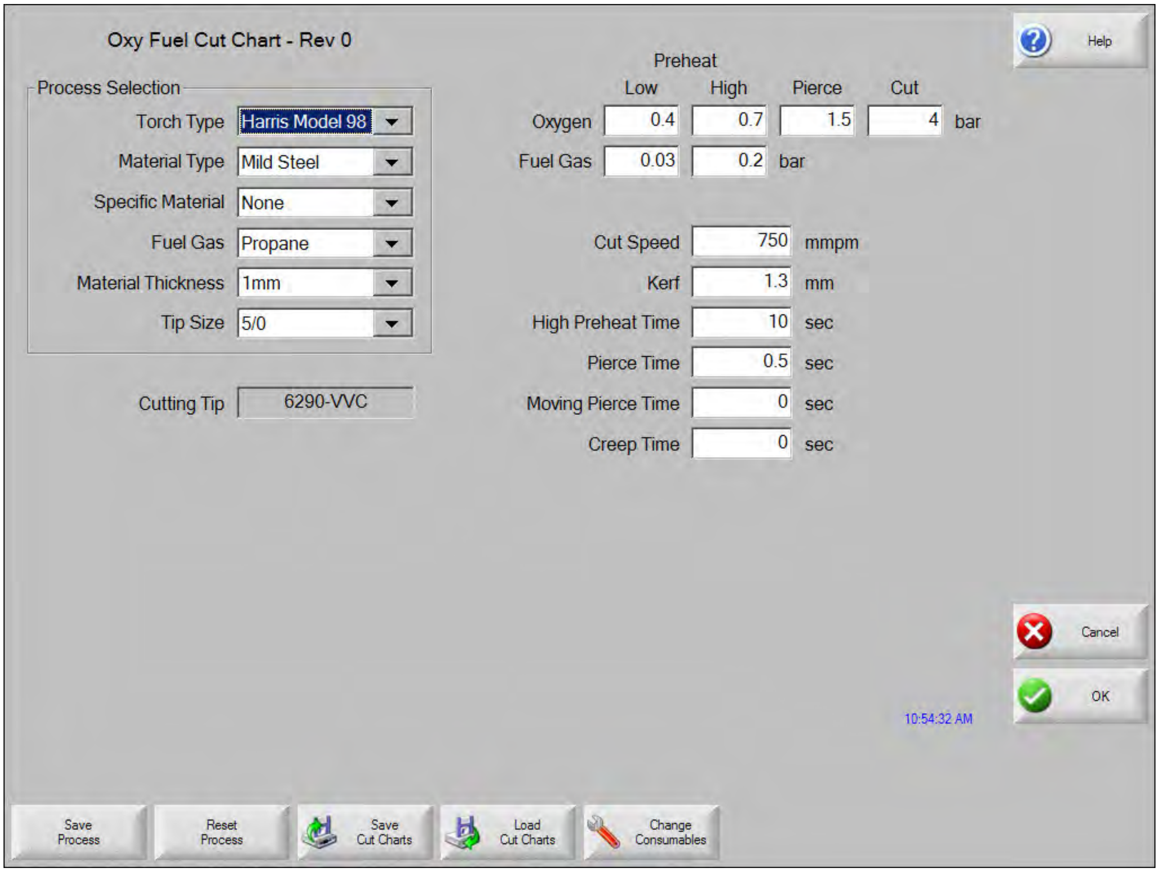
The Oxyfuel Process screen is shown below.



Oxyfuel cut chart

The CNC provides cut charts for oxyfuel cutting systems. The cut charts are specific to the type of torch used on the oxyfuel cutting system.

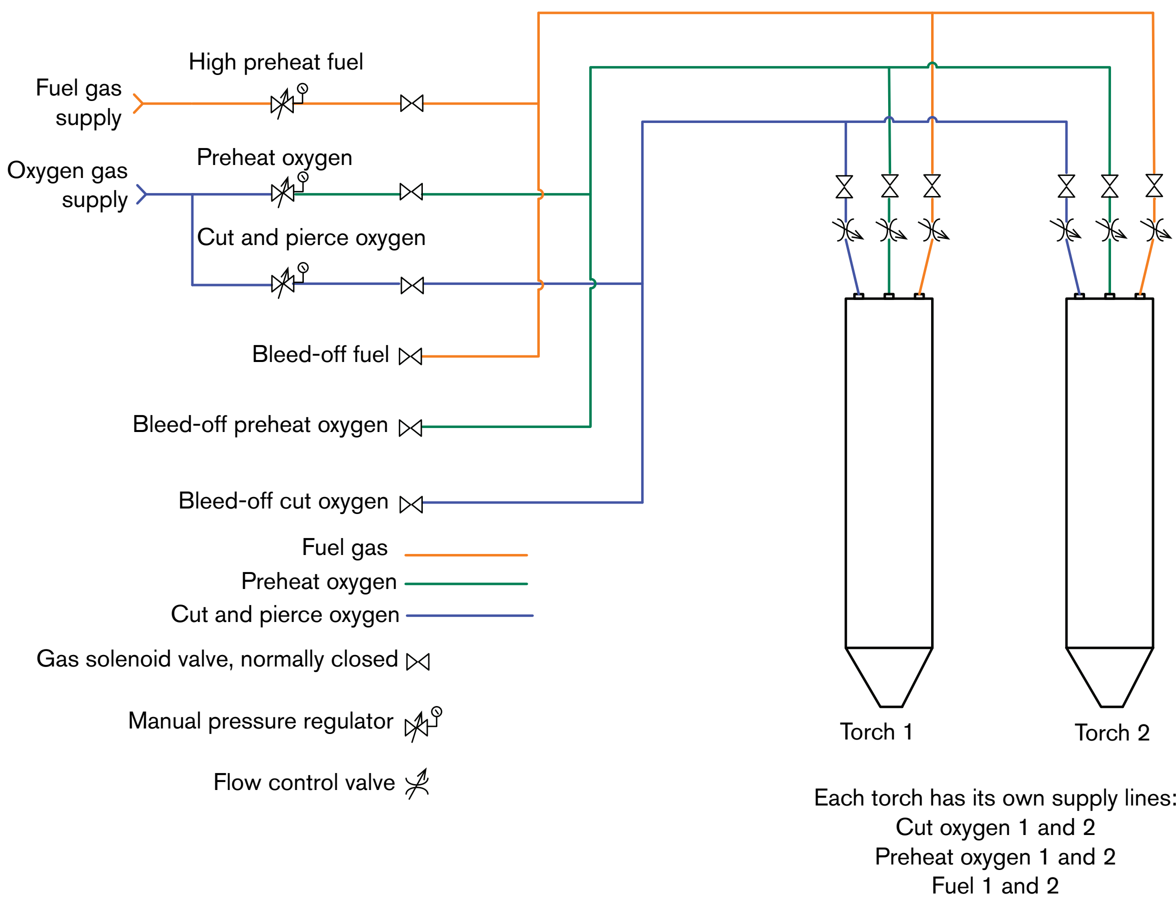
If your cutting system is using proportional regulators on analog inputs (described in the next section), the gas pressures in the cut chart are transferred to the Oxyfuel Process screen.



Controlling proportional gas regulators with analog outputs

Hypertherm CNCs provide analog outputs that can control proportional gas regulators. Using this strategy allows you to reduce the number of regulators needed for the gas delivery system from six as shown earlier in this section, to three as shown below. Analog outputs are available for three channels: fuel gas, preheat oxygen, and pierce/cut oxygen.

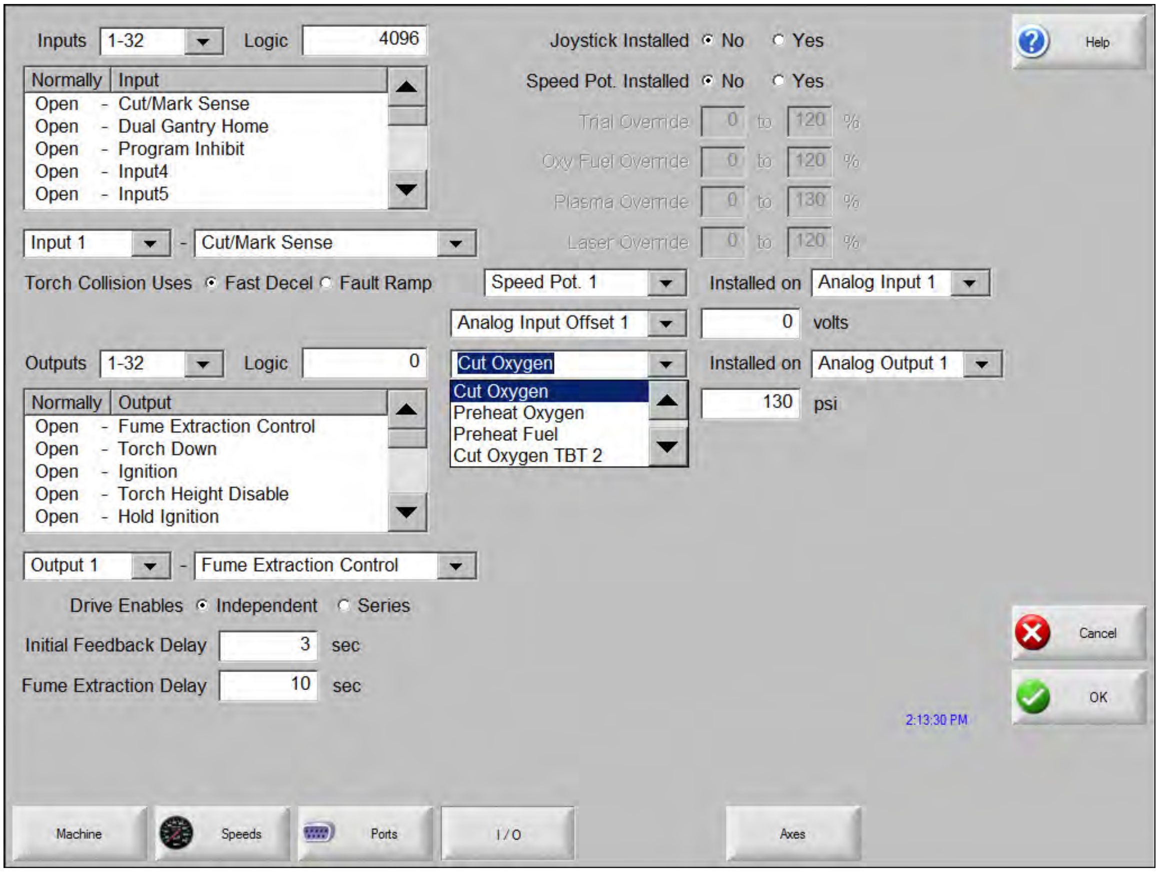
Setting up your oxyfuel cutting system in this manner requires a SERCOS drive system to control the extended number of analog outputs.



Setting up analog outputs

The CNC provides 0 to 10 VDC reference voltage for the analog outputs. For each analog output you will need to enter the maximum regulator pressure. The CNC then calculates the voltage to correspond to pressure settings from 0 to the maximum pressure. There is no feedback to the CNC to control accuracy.

1. Select Setups > Password > Machine Setups > I/O.



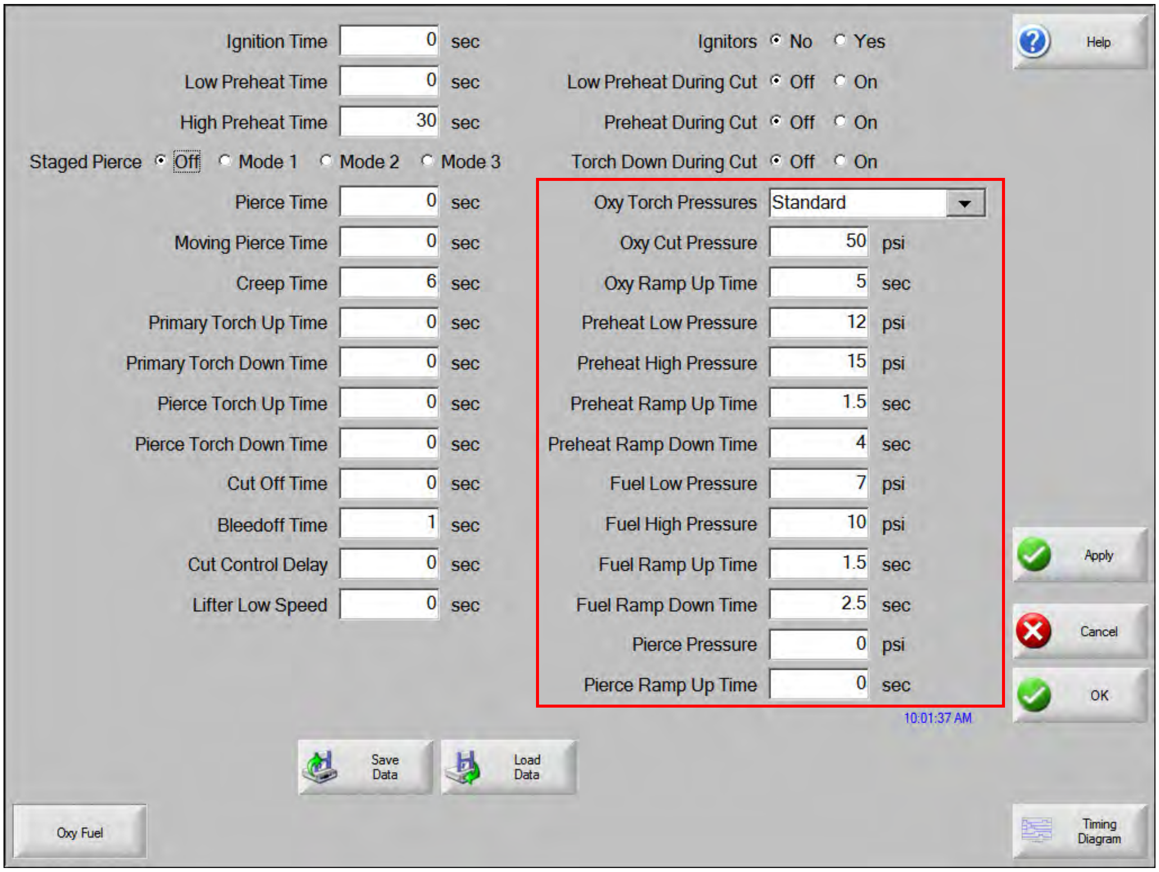
2. Select the gas channel and assign an analog output and maximum pressure for it. The channels are listed in groups. Each channel in each group should be selected and matched with an analog output. The CNC transfers these channels and pressures to the Oxyfuel Process screen.

Oxyfuel torch pressures	**Analog outputs to assign**
Standard	Cut Oxygen Preheat Oxygen Preheat Fuel
Triple bevel torch 2	Cut Oxygen TBT 2 Preheat Oxygen TBT 2 Preheat Fuel TBT 2
Triple bevel torch 3	Cut Oxygen TBT 3 Preheat Oxygen TBT 3 Preheat Fuel TBT 3
Triple bevel preheat torch	Preheat Oxygen TBPT Preheat Fuel TBPT

Setting gas pressures from the CNC

You can adjust the pressures on the Oxyfuel Process screen. Each pressure has a ramp-up time. The oxyfuel cut chart provides the starting set of values.

When adjusting the gas pressures or timers, use the Apply soft key to send the pressures to the cutting system. As you are fine-tuning the system for your requirements, you can select Apply, change the pressures on the screen, and select Apply again without exiting the screen.

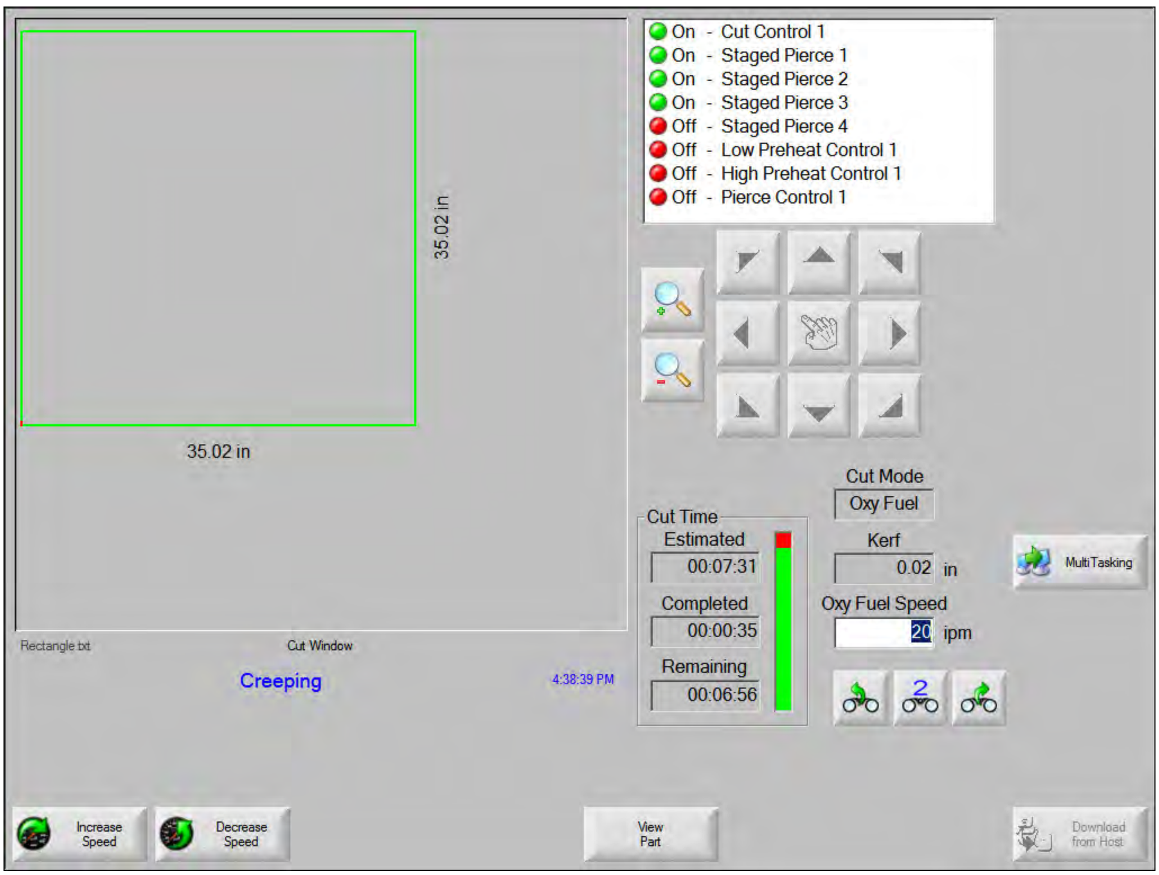


Staged pierce for oxyfuel cutting

Staged pierce increases the oxygen pressure while piercing and helps the torch penetrate the material more quickly. It is also a moving pierce that replaces the stationary pierce. During the staged pierce, the CNC begins moving at the creep speed as it performs the pierce.

Staged pierce requires four outputs called Staged Pierce 1 – 4. The outputs can control four separate oxygen regulators or one proportional regulator. On the oxyfuel process screen, you can select one of three modes for the staged pierce. Each mode activates the staged pierce outputs in a different order. You can enter the duration for each output to be activated.

- Staged Pierce Time 1 controls Staged Pierce 1 output.
- Staged Pierce Time 2 controls Staged Pierce 2 output.
- Staged Pierce Time 3 controls Staged Pierce 3 output.
- Staged Pierce 4 activates after Staged Pierce Time 3 elapses.



The following table illustrates the output activation for the three modes. When you run the part, the Creeping message displays as the status when the I/O that are associated with these parameters activate.

	Mode 1	Mode 2	Mode 3
	Staged Pierce Output (number) = (state)	Staged Pierce Output (number) = (state)	Staged Pierce Output (number) = (state)
Torch reaches creep speed	1 = On 2 = Off 3 = Off 4 = Off	1 = On 2 = Off 3 = Off 4 = Off	1 = On 2 = Off 3 = Off 4 = Off
Staged Pierce Time 1 expires	1 = On 2 = On 3 = Off 4 = Off	1 = On 2 = On 3 = Off 4 = Off	1 = Off 2 = On 3 = Off 4 = Off
Staged Pierce Time 2 expires	1 = On 2 = On 3 = On 4 = Off	1 = On 2 = On 3 = On 4 = Off	1 = Off 2 = Off 3 = On 4 = Off
Staged Pierce Time 3 expires	1 = On 2 = On 3 = On 4 = On Torch accelerates to cut speed	1 = Off 2 = Off 3 = Off 4 = On Torch accelerates to cut speed	1 = Off 2 = Off 3 = Off 4 = On Torch accelerates to cut speed
At end of Staged Pierce	All Staged Pierce outputs remain on until the end of the cut (M08)	Staged Pierce output 4 remains on until the end of the cut (M08)	Staged Pierce output 4 remains on until the end of the cut (M08)

Section 16

Waterjet Application

This section describes the HyPrecision™ waterjet setup for Hypertherm CNCs. It includes:

- System overview and the connections between the CNC and the waterjet pump, cutting head, and abrasive regulator
- Enabling the waterjet process on the CNC
- Description of the waterjet cut sequence
- Definitions of the inputs and outputs used for waterjet cutting systems
- Cutting a part
- Waterjet cut speed calculator
- System messages

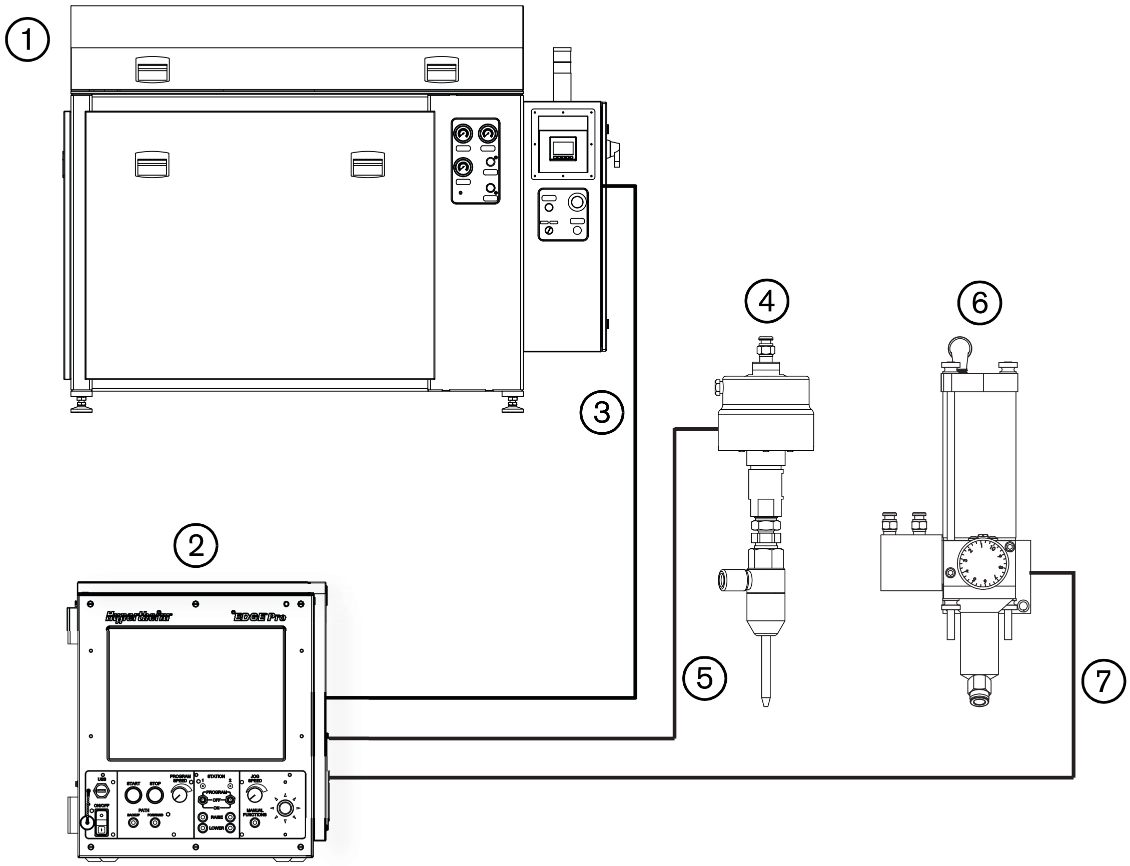
Connecting the HyPrecision pump to the CNC requires that the pump be equipped with the remote communications option. For further assistance with the waterjet pump, contact your table manufacturer.

Consult the following additional resources for more information about using a waterjet with a Hypertherm CNC:

- *The Phoenix Operator Manual* provides information on the waterjet Process and Cut Chart screens, piercing, timing diagrams and Watch Window.
- *The Phoenix V9 Series Programmer's Reference* provides the cut chart and process parameter variables for controlling waterjet process selection through a part program.
- The CNC provides manuals for HyPrecision waterjet pumps and instructions for changing the consumables on the cutting head.

Waterjet system overview

The CNC connects to the waterjet pump, cutting head, and abrasive regulator.



1 HyPrecision waterjet pump

2 CNC

3 Serial connection from CNC to PLC on waterjet pump

4 Waterjet cutting head

5 Cut Control output from CNC to solenoid valve on cutting head to turn on water flow

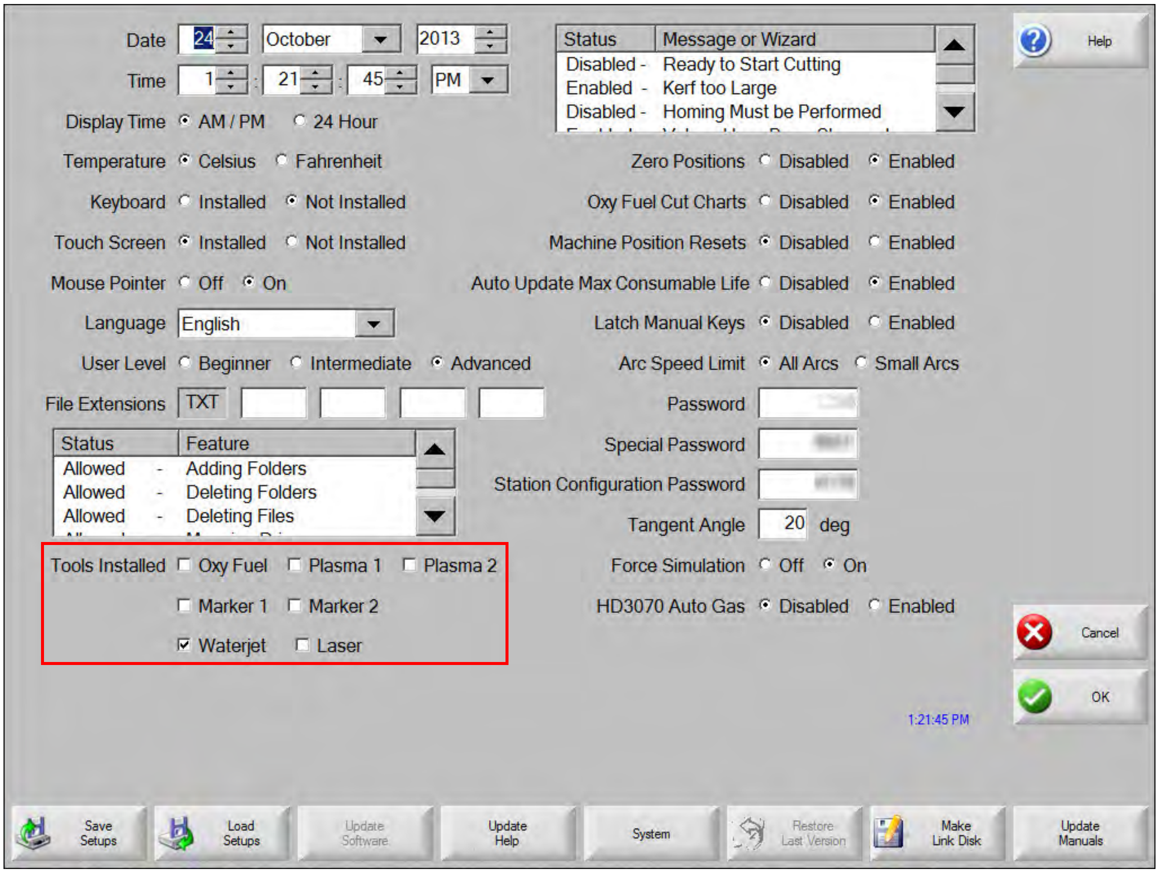
6 Abrasive regulator

7 Abrasive Control output from CNC to solenoid valve on abrasive regulator to turn on abrasive flow

Enabling the waterjet process on the CNC

Use the Special Setups screen to enable the waterjet process and make the Waterjet Process screen available.

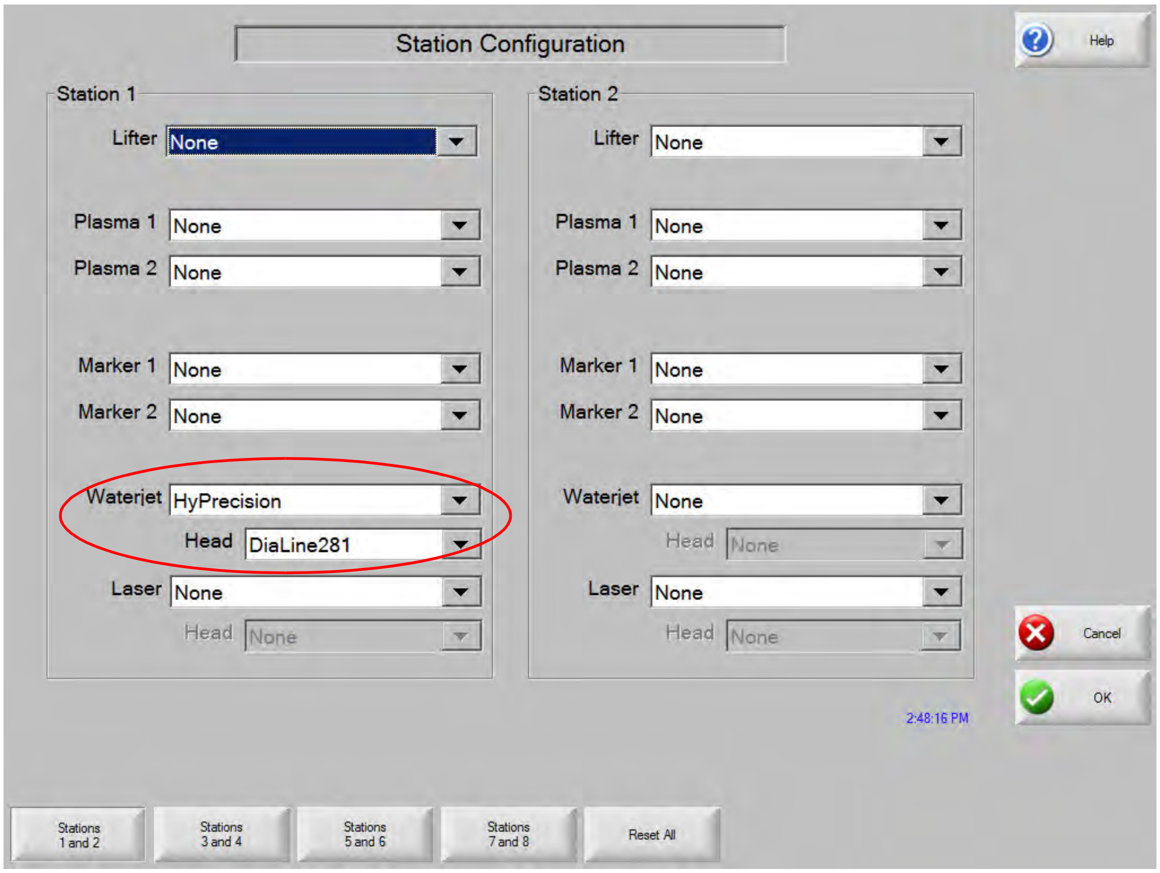
1. Choose Setups > Password > Special Setups.
2. Select Waterjet under Tools Installed.
3. Choose OK.



Selecting the waterjet pump model

Select the waterjet pump model to enable the cut chart and activate the pump manual on the help screen.

1. Choose Setups > Password > Station Configuration.
2. For Waterjet, choose HyPrecision.
3. Choose the cutting head supported by your waterjet system.
4. Choose OK.



Serial communications

If the waterjet pump is equipped with the remote communication option, the CNC can communicate with the pump PLC through an RS-422 serial link. These types of messages are exchanged between the CNC and the PLC:

- Cut pressure
- Low pressure (used for piercing certain materials)
- Pump warnings
- Pump errors

Assigning a serial port

After connecting the CNC to the waterjet pump, assign the serial port to the pump.

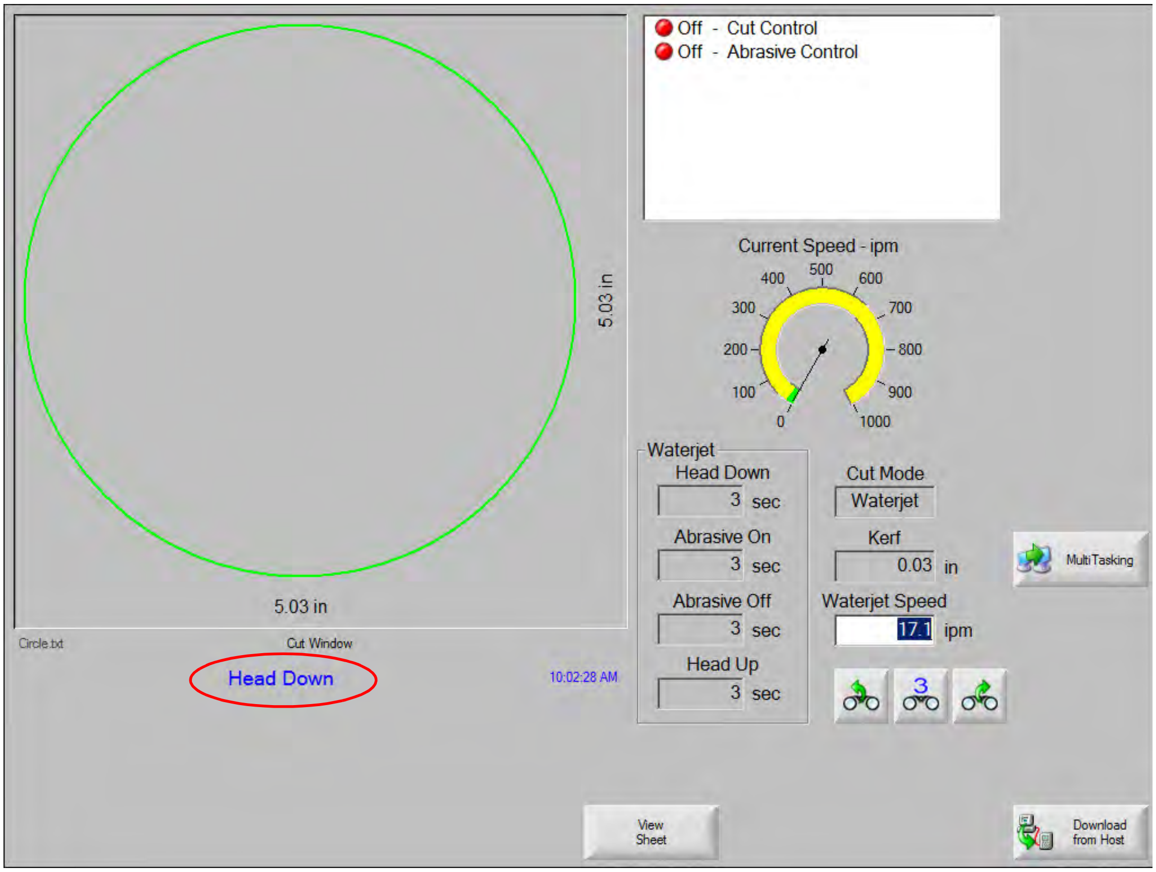
1. Choose Setups > Password > Machine Setups > Ports.
2. Choose HyPrecision from the drop down list.
3. Choose the CNC serial port number to which you have connected the waterjet pump. The serial port numbers are marked on the back of the CNC.

Sequence of operations

Hypertherm CNCs execute the waterjet cut sequence in these basic steps.

1. The operator enables a station on the CNC to turn on the pump. For example, on the EDGE Pro CNC, moving the Station 1 switch to either the ON or PROGRAM position enables the station.
2. The operator presses Start to begin executing the part program. When the CNC executes the M07 code (Cut On) in the part program, the CNC turns on the Cut Control output signal which activates the solenoid valve in the cutting head and starts water flow to the cutting head.
3. When the CNC executes the M07 command, it also turns on the Abrasive Control output (unless you have set an Abrasive On Delay timer). The Abrasive Control output connects to a solenoid valve on the abrasive regulator.
4. Motion starts after the CNC runs the Abrasive On Delay and Pierce Motion Delay timers. These timers ensure the water and abrasive flow are both stabilized at the cutting head.

The CNC identifies each state of the cut sequence with a blue status message below the part preview area on the Main screen.



State and status message: *Head Down*

> **Initiated when:** The CNC reads the M07 code (Cut On) in the part program and the Head Down timer is set in the Waterjet Process screen. The Head Down timer elapses before Cut Control output turns on.

State and status message: *Abrasive Delay*

> **Initiated when:** The CNC turns on the Cut Control output and when the Abrasive On Delay timer, which is set in the Waterjet Process screen, elapses, the Abrasive Control output turns on.

State and status message: *Pierce Motion Delay*

> **Initiated when:** The CNC starts the Pierce Motion Delay timer after the Abrasive On Delay elapses.

State and status message: *Piercing*

Initiated when: The CNC begins the pierce routine after the Pierce Motion Delay elapses. The pierce routine is specified in either the part program or in the cut chart. The CNC continues the pierce routine until the Pierce Time elapses.

The HyPrecision waterjet cut charts provided on the CNC include four styles of piercing: Dynamic, Wiggle, Circular, and Stationary. Dynamic, Wiggle, and Circular pierces are all moving pierces. Dynamic pierce cuts along the part lead-in for the pierce time. Wiggle and Circular pierces cut along a displacement distance or diameter, then move to the beginning of the lead-in or part cut path. See the *Phoenix Operator Manual* (806400) for more information about piercing.

State and status message: *Cutting*

Initiated when: The Pierce Time elapses and the cutting system accelerates to Cut Speed.

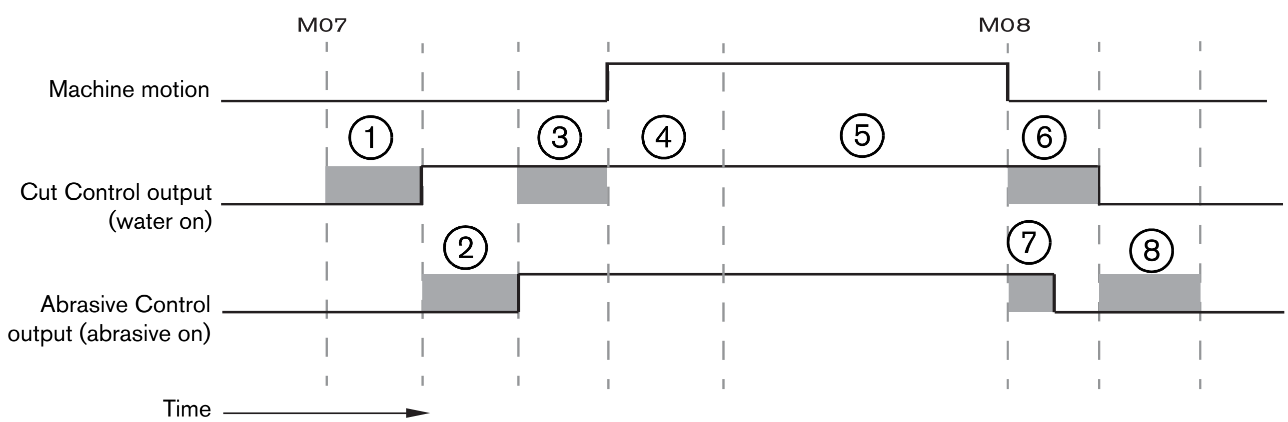
State and status message: *Abrasive Off Delay or Water Off Delay.*

Initiated when: The CNC executes the M08 code (Cut Off) in the part program. Abrasive Off Delay and Water Off Delay timers run concurrently. Abrasive Control output turns off when the Abrasive Off Delay elapses. Cut Control output turns off when Water Off delay elapses.

State and Status message: *Head Up*

Initiated when: The CNC turns off the Cut Control output.

The timing diagram below shows the relationship of the timers to the Cut Control output (which turns on water flow), and the Abrasive Control output (which turns on abrasive flow).



1 Head down time

2 Abrasive On Delay time (+)

3 Pierce Motion Delay time

4 Pierce Time (set on the cut chart)

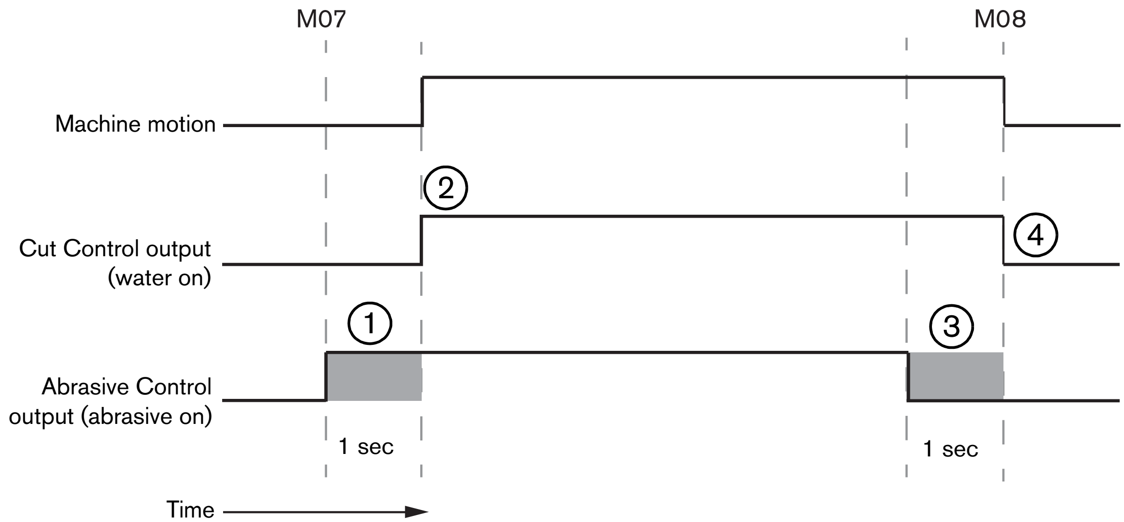
5 Cutting motion

6 Water Off Delay (+) (runs concurrently with the Abrasive Off Delay).

7 Abrasive Off Delay (+)

8 Head Up time

The next timing diagram shows an example of a negative Abrasive On Delay and negative Abrasive Off Delay of one second each. Use the negative Abrasive On/Off delays for special applications that require the abrasive flow to start or stop before the water flow.



1 Abrasive On Delay (-) starts abrasive before water flow.

2 Cut Control (water flow) turns on after Abrasive On Delay elapses.

3 Abrasive off Delay (-) turns off abrasive before the end of cut (M08)

4 Cut Control turns off after Abrasive Off Delay elapses.

Waterjet I/O

Waterjet cutting uses the following I/O points. Assign the I/O points in the I/O screen (Setups > Password > Machine Setups > I/O).

Cut Control: This output starts water flow to the cutting head and is activated by the CNC when it executes the M07 code (Cut On) in the part program (after the Head Down timer elapses). Connect an output from the CNC to a solenoid valve on the waterjet cutting head.

Abrasive Control: This output starts abrasive flowing to the cutting head and is activated when:

- Abrasive Flow is set to ON in the Waterjet Cut Chart screen
- M07 executes in the part program.
- Abrasive On Delay timer elapses if set.

Connect an output from the CNC (HyPath or Picopath) or the SERCOS drive to a solenoid valve on the abrasive regulator.

Torch Down and **Torch Up:** These outputs connect to relays on the lifter and are controlled by the Head Down and Head up timers on the Waterjet Process screen.

Automatically setting abrasive delays

The CNC can automatically calculate the Abrasive On and Off Delays, the Water Off Delay, and the Pierce Motion Delay. The CNC requires that you enter lengths for the abrasive and actuator hoses on the Waterjet Process screen.

1. From the Main screen, choose **Setups > Process.**
2. Choose **Edit Abrasive Parameters.**
3. Enter the **Machine Setups password.**
4. Enter the **hose lengths.**

Waterjet height control

Hypertherm CNCs provide software support for a height control for use with waterjet cutting. The height control mechanism — for example a foot-sensor — must provide a voltage output to the CNC that ranges from 0 to 10 V. The output voltage from the height sensing device should decrease as the cutting head moves toward the workpiece. When the input voltage to the CNC is 0 V, the CNC uses this as the location of the workpiece surface.

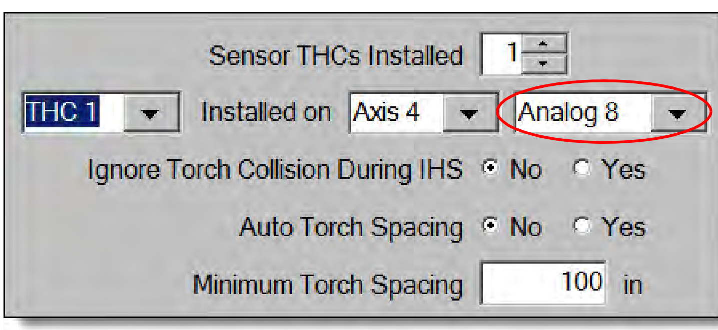
Waterjet height control has these requirements:

- Supported on SERCOS III cutting systems using Kollmorgen AKD drives. Hypath and Picopath CNCs are not currently supported.
- For the height sensing device, use a foot-sensor design.

Setting up the waterjet height control follows these general steps.

1. The analog output from the height control device must be wired to an analog input in the waterjet height control axis drive.

   Assign the analog input in the **Setups > Password > Machine Setups** screen. The analog input number that you select depends on the location of the axis on the SERCOS ring and the number of analog inputs on each drive in the ring. In the example below, if the height control axis is Axis 4, and each drive has on analog input, the analog input would be Analog 8. (Analog inputs 1 – 4 are reserved by the CNC.)



For more information on assigning analog inputs in SERCOS III drives, see *SERCOS III Setup for Hypertherm CNCs* (808000) available in the Downloads Library on www.hypertherm.com.

2. After installing the height sensing device with analog voltage output to the CNC, set up the Sensor WHC axis. See *Torch Height Control Axis Setup* on page 103 for information on setting up the waterjet height control axis. The waterjet height control axis is like the Sensor THC axis that is used for plasma cutting.

3. Monitor incoming voltage from the height sensing device. Carefully lower the waterjet nozzle to the surface of the workpiece. When the nozzle touches the workpiece, the voltage should be 0 V. Choose **Setups > Diagnostics > I/O** and enter the Machine Setups password. Choose **Analog Input** to view the incoming voltage.

   You may need to use a voltage offset provided by your hardware; for example, in the SERCOS III drive. Adjust the voltage offset so that the height sensing device is outputting 0 V when the waterjet nozzle touches the workpiece.

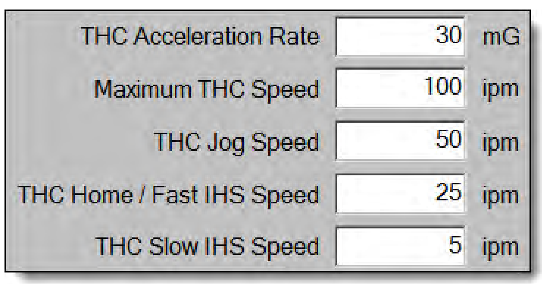
   Changing the length of the nozzle installed in the waterjet cutting head would also require you to re-check the voltage input to ensure that 0 V is being output when the nozzle touches the workpiece.

4. Calibrate the waterjet height control. After you have the 0 V signal at the workpiece surface, run the Calibrate WHS routine. Home the height control, then choose **Setups > Process > Calibrate WHS.**

5. Perform the first initial height sense. Before starting a cutting job, choose **Test Lifter** from the **Main** screen. The CNC lowers the cutting head to the workpiece, then moves to the cut height for the cutting job.

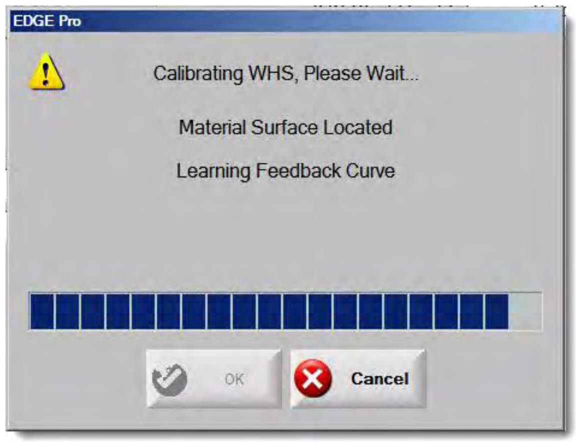
Speeds for waterjet height control

Choose **Setups > Password > Machine Setups > Speeds** to set the travel speeds for the waterjet height control. Below are some suggested speeds for a foot-sensor height sensing device.



Calibration

The waterjet height control calibration feature provides the CNC with the voltage input at different height increments. To begin calibration, choose **Setups > Process > Calibrate WHS.** The CNC lowers the height sensing device to the workpiece surface, then moves it up from the workpiece in increments and measures the voltage at each increment. The CNC then calculates the height at different voltages and uses this information to control cutting height. Perform calibration whenever you change the waterjet nozzle to a different length.



Foot Sensor Up input

The CNC provides an input called Foot Sensor Up which activates when the height control device is fully retracted from the workpiece surface. When this input activates, the CNC prevents waterjet height control calibration, use of the test lifter routine (waterjet initial height sense), running of a part program, or rip cutting. Blocking motion when Foot Sensor Up is activated protects the waterjet nozzle from damage that could occur if it contacts the workpiece surface.

Initial height sense (IHS)

Waterjet height control uses a sequence called initial height sense, or IHS, for detecting the workpiece. You perform a first IHS after powering up the cutting system and before each cutting job. A first IHS detects the location of the workpiece so that the CNC can calculate the cut height. The CNC uses the cut height for all subsequent IHSs, which it can perform using much faster speeds since the position of the workpiece surface is known.

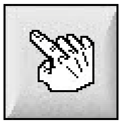
Be sure you have performed the waterjet height control calibration **before** performing the first IHS.

The IHS begins at the **IHS Start Height** set in the **Setups > Process** screen. When the cutting head reaches this distance above the workpiece the following actions occur:

- Speed slows from **Maximum THC Speed** to **Fast IHS Speed.**
- The CNC monitors the incoming analog voltage from the height control until it reads 0 V.
- After the CNC determines the height of the workpiece, it moves the cutting head to the **Cut Height** at the **Slow IHS Speed.**

Performing a first IHS

1. Home the Waterjet Height Control axis by pressing **F11**, or choosing the **Manual** soft key.



2. Choose the **Home Axes** soft key.
3. Choose the **THC** soft key.
4. Choose **OK** twice to return to the **Main** screen.
5. On the **Main** screen, choose the **Test Lifter** soft key. The CNC performs the IHS starting from the waterjet height control axis home position.

A height control error, a manual move, an idle timeout of 30 seconds, or a power cycle on the CNC all result in the next IHS occurring at the Slow IHS Speed to find the workpiece height again.

Skip IHS

Skip IHS optimizes production by reducing the time between cuts. If the next pierce point is within the **Skip IHS Distance** set on the **Process** screen, from the end of the previous cut, the CNC does not perform an IHS before starting the next cut.

1. Choose **Setups > Process**.
2. Enter a value for **Skip IHS Distance**. Enter **0** to disable Skip IHS.

Low pressure piercing

The CNC supports low pressure piercing using the G59 V827 F2 waterjet variable.

Code	Description
G59 V827 F2	Low pressure pierce, maintain (F2) until next G59 V827, or a new cut chart is selected, or a new part program is loaded. Include P XXXXXX for pressure if there is serial communication. Set the pressure at the pump if there is no serial communication.
G04 X*x*	Dwell for *x* seconds to allow the waterjet pump to transition to low pressure setting.

All other G59 variables in the part program must come before G59 V827 F2 and G04.

The CNC also provides a Low Pressure Pierce output which can be connected to an input on the pump PLC to switch the pump to low pressure mode. You can view the Low Pressure Pierce output in the I/O section of the Watch Window.

Cutting a part

Enabling a station

Before cutting, the pump must reach cut pressure. To start pressurizing the pump, the CNC must have a station enabled in one of the following ways:

- Move a station switch on the operator console into the ON position. Use the ON position when preparing to cut parts that do not include the M36 T6 code, such as the simple shapes included on the CNC.

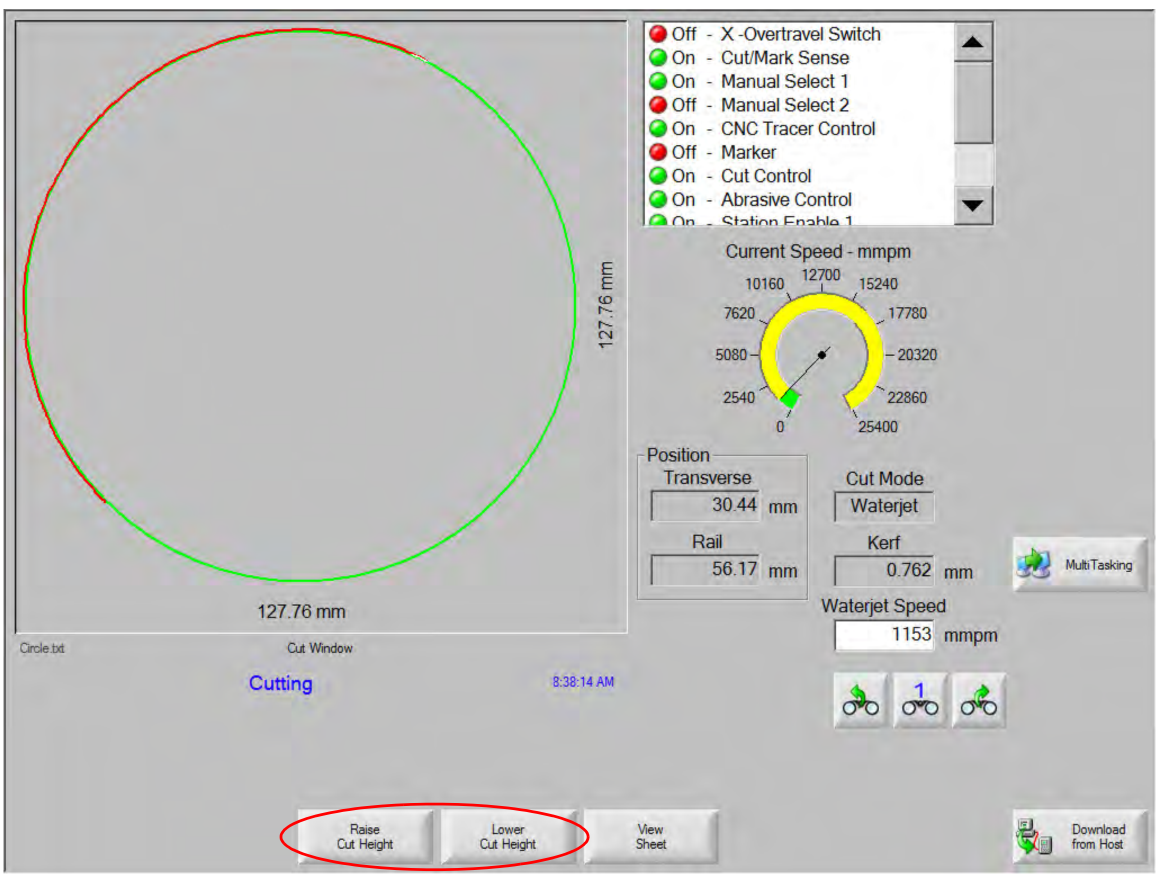
  Or

- Move a station switch into the PROGRAM position and load a part program that has the M36 T6 code that selects the Waterjet Process.

Setting the cut height without height control

Manually lower the cutting head to the correct height by pressing the Lower (or equivalent) switch on the CNC operator console. Many operators use a feeler gage to set the position of the nozzle above the material. Using a servo drive for the lifter provides position feedback to the CNC Main screen.

For cutting systems that use the Sensor THC for height control, you can also adjust the cut height while the waterjet is cutting. During cutting, the CNC displays Raise Cut Height and Lower Cut Height soft keys.



Cut speed calculator

Hypertherm provides a cut speed calculator that helps you to plan and estimate your cutting jobs, and to generate values for additional waterjet cut charts.

To launch the calculator, from the Main screen, choose the Waterjet Cut Chart soft key, then choose the Cut Speed Calculator soft key. For more information, choose the Help menu in the Cut Speed Calculator.

Waterjet system messages

System messages display on the CNC and warn the operator of status and faults at the waterjet pump (for serially-connected waterjet pumps only). Refer to the manual for the waterjet pump for troubleshooting information.

- **Warning:** CNC continues cutting until it finishes the part program, then it displays the error message in red in the status area of the Main screen. You must correct the error condition at the waterjet pump before you can restart cutting.

Error number	Message
2	Oil temperature > 55° C (130° F)
5	Dirty hydraulic filter
7	Excess motor starts

- **Shutdown level 1:** The CNC displays the message in the System Errors Watch Window and pauses cutting. The waterjet intensifier stops but the pump motor continues to run. You must correct the error condition at the waterjet pump before you can restart cutting.

Error number	Message
3	Oil temperature > 65° C (150° F)
4	Low inlet water pressure
8	Intensifier 1 overstroke to right
9	Intensifier 1 overstroke to left
10	Intensifier 2 overstroke to right
11	Intensifier 2 overstroke to left

- **Shutdown level 2:** CNC displays the error in the System Errors Watch Window and pauses cutting. The waterjet intensifier and pump motor both stop. You must correct the error condition at the waterjet pump before you can restart cutting.

Error number	Message
1	Oil level low
6	Main motor fault
12	Boost pump motor fault
13	Fan cooler motor fault