Hypertherm®

Phoenix™ Software V9.76.0

Operator Manual

806400 | Revision 10 | English

Phoenix Software V9.76.0

Operator Manual

806400
Revision 10

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.

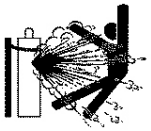
STATIC ELECTRICITY CAN DAMAGE CIRCUIT BOARDS

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.

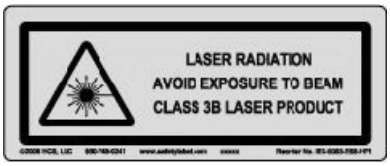
Note 1 – Hypertherm's interpretation of these new requirements is that unless a site-specific evaluation has been completed to determine that all dust generated is not combustible, the 2007 edition of NFPA 68 requires the use of explosion vents designed to the worst-case Kst value (see annex F) that 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:

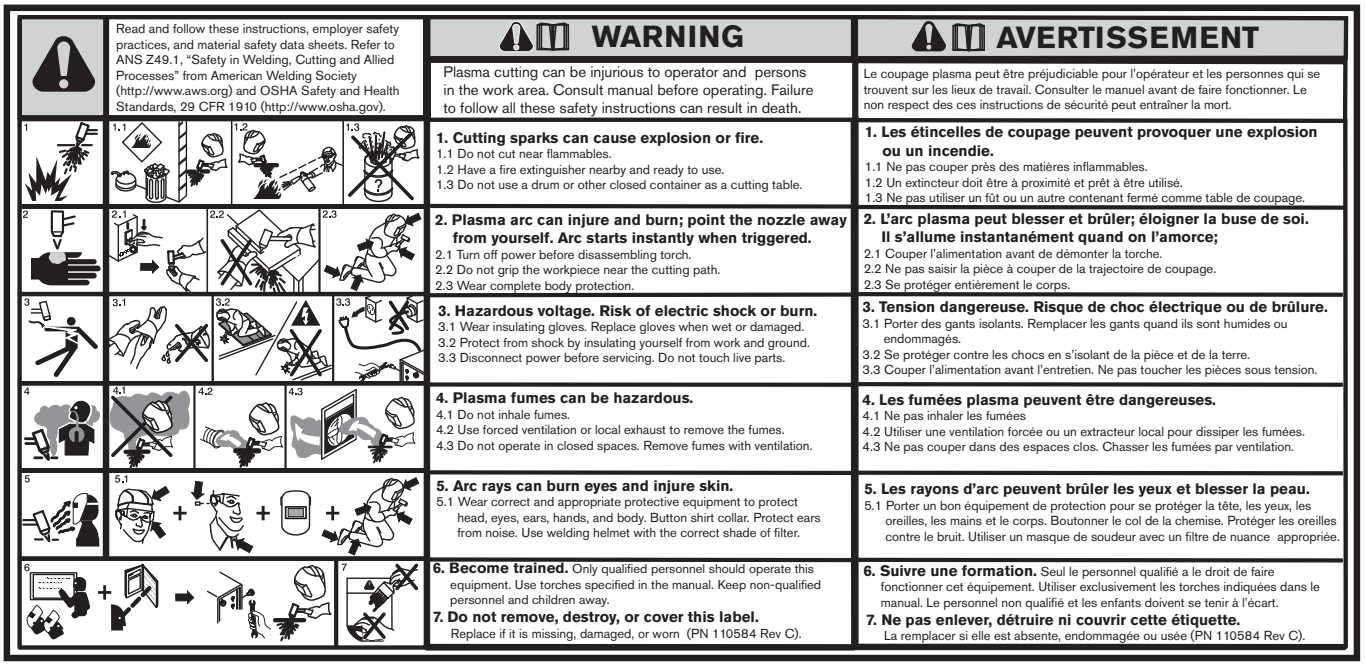
- Consult with an expert on local laser regulations. Laser safety training may be required.
- Do not allow untrained persons to operate the laser. Lasers can be dangerous in the hands of untrained users.
- Do not look into the laser aperture or beam at any time.
- Position the laser as instructed to 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.



Read and follow these instructions, employer safety practices, and material safety data sheets. Refer to ANS Z49.1, "Safety in Welding, Cutting and Allied Processes" from American Welding Society (http://www.aws.org) and OSHA Safety and Health Standards, 29 CFR 1910 (http://www.osha.gov).

WARNING

Plasma cutting can be injurious to operator and persons in the work area. Consult manual before operating. Failure to follow all these safety instructions can result in death.

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 (PN 110584 Rev C).

AVERTISSEMENT

Le coupage plasma peut être préjudiciable pour l'opérateur et les personnes qui se trouvent sur les lieux de travail. Consulter le manuel avant de faire fonctionner. Le non respect des ces instructions de sécurité peut entraîner la mort.

1. Les étincelles de coupage peuvent provoquer une explosion ou un incendie.
1.1 Ne pas couper près des matières inflammables.
1.2 Un extincteur doit être à proximité et prêt à être utilisé.
1.3 Ne pas utiliser un fût ou un autre contenant fermé comme table de coupage.

2. L'arc plasma peut blesser et brûler; éloigner la buse de soi. Il s'allume instantanément quand on l'amorce;
2.1 Couper l'alimentation avant de démonter la torche.
2.2 Ne pas saisir la pièce à couper de la trajectoire de coupage.
2.3 Se protéger entièrement le corps.

3. Tension dangereuse. Risque de choc électrique ou de brûlure.
3.1 Porter des gants isolants. Remplacer les gants quand ils sont humides ou endommagés.
3.2 Se protéger contre les chocs en s'isolant de la pièce et de la terre.
3.3 Couper l'alimentation avant l'entretien. Ne pas toucher les pièces sous tension.

4. Les fumées plasma peuvent être dangereuses.
4.1 Ne pas inhaler les fumées
4.2 Utiliser une ventilation forcée ou un extracteur local pour dissiper les fumées.
4.3 Ne pas couper dans des espaces clos. Chasser les fumées par ventilation.

5. Les rayons d'arc peuvent brûler les yeux et blesser la peau.
5.1 Porter un bon équipement de protection pour se protéger la tête, les yeux, les oreilles, les mains et le corps. Boutonner le col de la chemise. Protéger les oreilles contre le bruit. Utiliser un masque de soudeur avec un filtre de nuance appropriée.

6. Suivre une formation. Seul le personnel qualifié a le droit de faire fonctionner cet équipement. Utiliser exclusivement les torches indiquées dans le manual. Le personnel non qualifié et les enfants doivent se tenir à l'écart.

7. Ne pas enlever, détruire ni couvrir cette étiquette.
La remplacer si elle est absente, endommagée ou usée (PN 110584 Rev C).

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

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

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

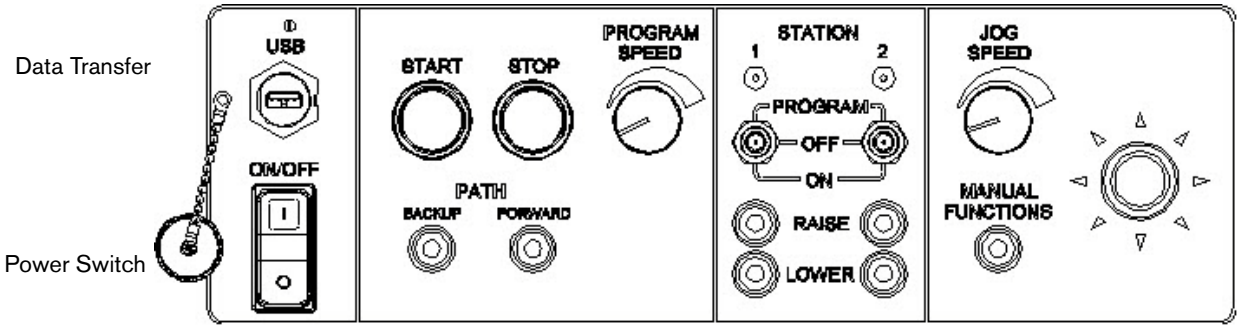
Operating the CNC

Phoenix software runs on the Hypertherm computer numerical controls (CNCs) including the EDGE® Pro and MicroEDGE® Pro, and EDGE® Pro Ti. Phoenix supports either a touch screen or LCD display with a USB-connected keyboard and mouse for entering information and navigating the software.

Operator Console

An optional operator console provided by Hypertherm, an OEM, or a system integrator powers up the CNC and controls machine motion such as station selection, raising or lowering the cutting tool, and positioning the cutting tool before starting a part program.

The EDGE Pro operator console is shown below. The operator console on your CNC may look different and have other controls than those shown here.



Touch Screen LCD

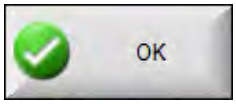
The Phoenix software is designed for 38 cm (15 inch) touch screens with 1024 x 768 or higher resolution. When your CNC is equipped with a touch screen, you can enter data into the software by touching the window controls and fields. Any field that requires data input automatically displays an onscreen keypad when you touch it.

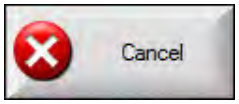
LCD Display

The MicroEDGE Pro can support an LCD display and requires 1024 x 768 resolution with a 4:3 aspect ratio.

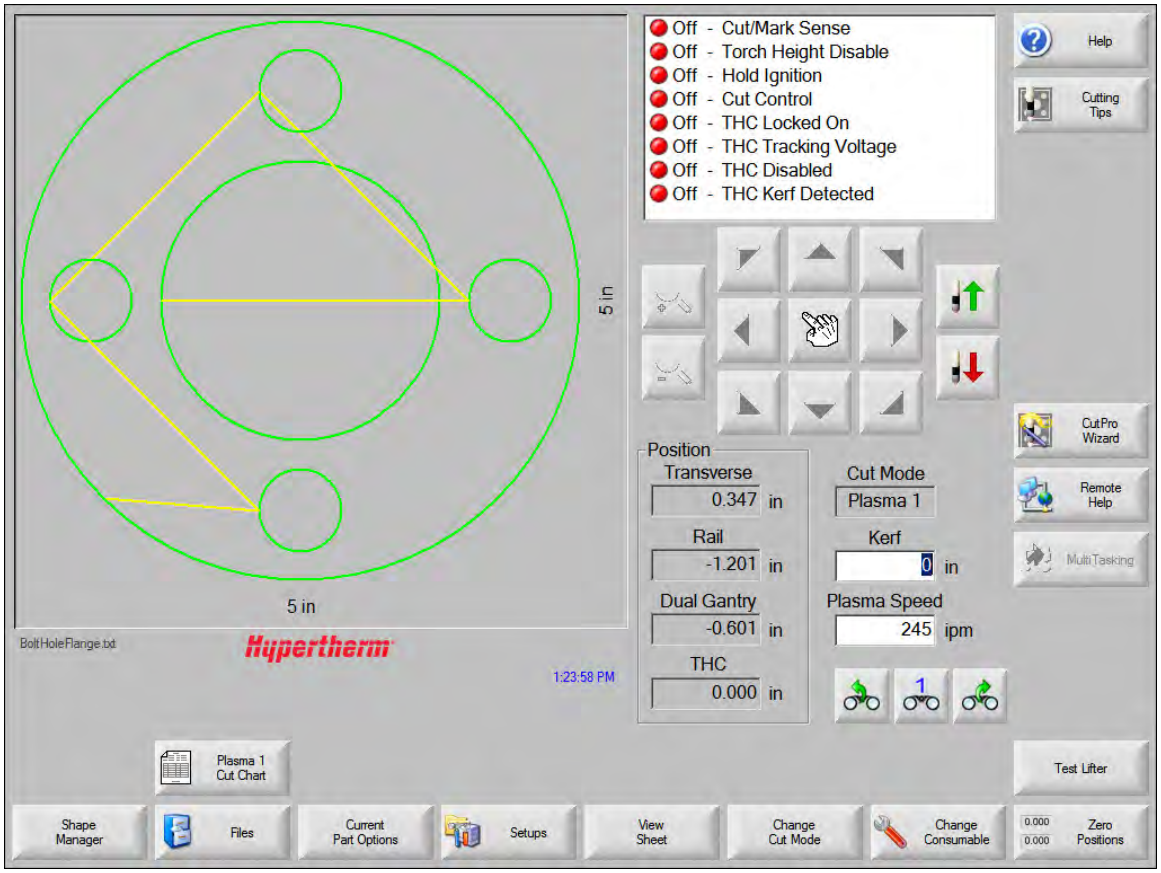
Screen Navigation

The eight keys located at the bottom of the screen are called *soft keys*. The soft keys correspond to the function keys on a PC keyboard. OK and Cancel soft keys let you save or cancel changes that you make in a screen.



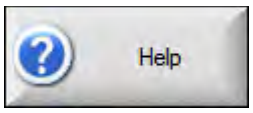


The features shown on each screen vary depending on the user level (Beginner, Intermediate, or Advanced) and the features enabled on the Special Setups and Station Configuration screens. This manual assumes the CNC is in Advanced Mode and shows all features.

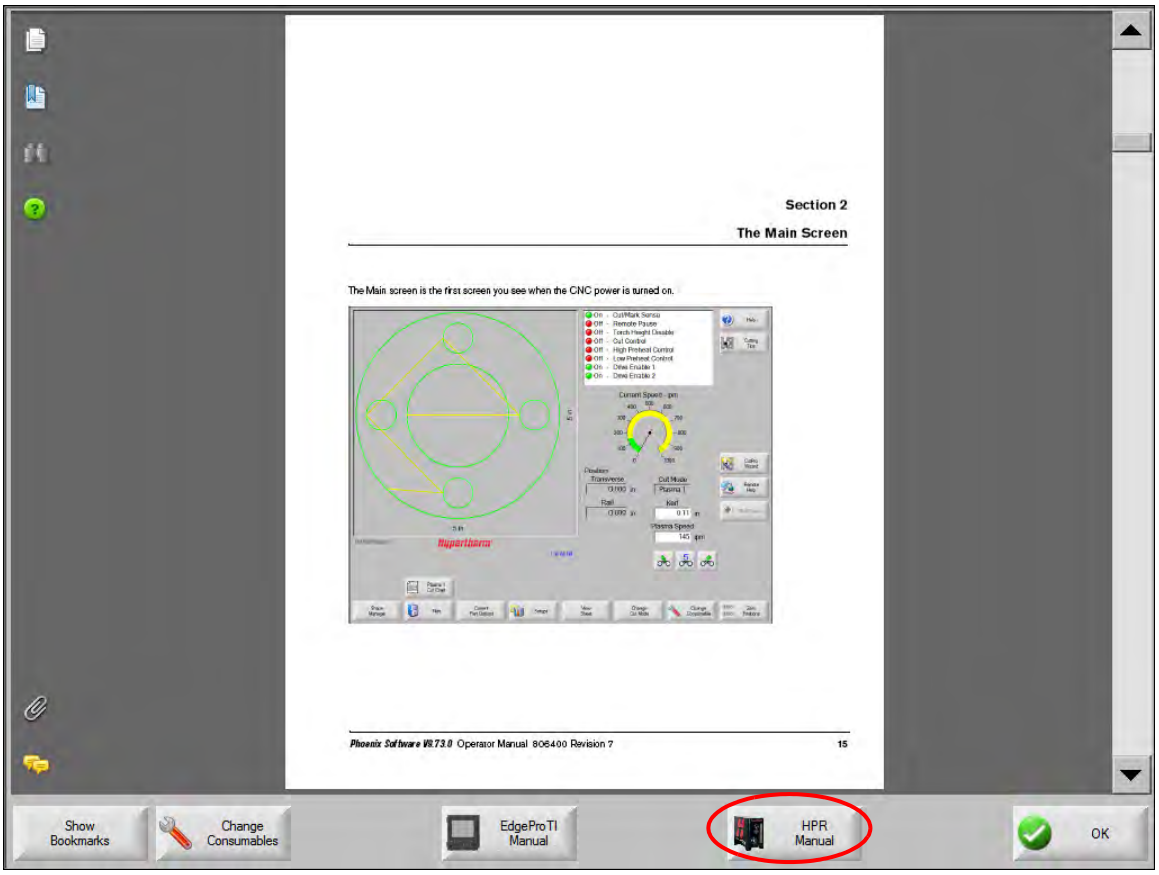


Help

Choose the Help soft key to display information about each screen.



Choose OK to exit the Help screen and return to the Main screen.



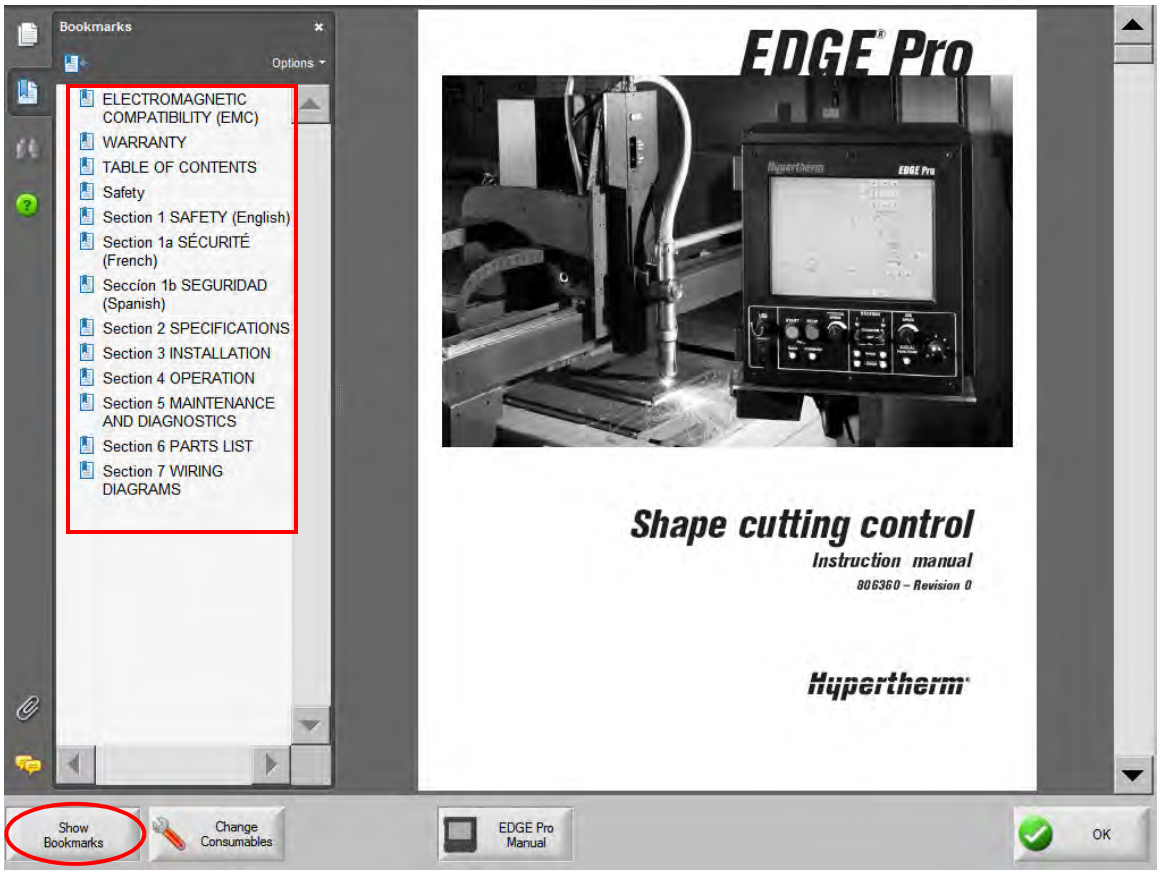
The Show Bookmarks soft key opens the navigation pane. Press Ctrl + F to use full text search.

The Help screen may also display buttons for other types of information. For example, the manual for the plasma system or torch height control installed on your system, or provided by your table manufacturer.

Show Bookmarks

Choose the Show Bookmarks soft key on the Help screen to view the contents of the help file. Click on a topic in the list to view it.

If you are operating the MicroEDGE Pro with a keyboard, use the Page Up/Page down keys to scroll through the document on screen.



Automated Operations

The Phoenix software includes two wizards that automate plate alignment and part cutting operations.

Align Wizard

The Align Wizard automates several tasks including aligning a nest on a plate, adjusting for a skewed plate, and positioning the torch at the program start location.

To start the Align Wizard, choose Shape Library on the Main screen, then choose Shape Wizard, Shape Options, Align. The Align Wizard may launch automatically. If not, choose the Align Wizard soft key.

For more information, see *Align Wizard* on page 70.

CutPro™ Wizard

The CutPro Wizard automates common cutting tasks including loading a part or nest, selecting the cutting process, aligning the part or nest on the plate, and starting the program.

The CutPro Wizard may launch automatically when you start the CNC. If not, choose the CutPro Wizard soft key on the Main screen to start the wizard. For more information on the CutPro Wizard see the *Cutting Parts* section.

Using Phoenix with a keyboard

Hypertherm CNCs can support a built-in keypad or a USB, PC keyboard instead of the touch screen to perform functions and data entry in the Phoenix software. To enable keyboard-only operation, choose Setups > Password > Special Setups and Touch Screen Not Installed.

IMPORTANT!

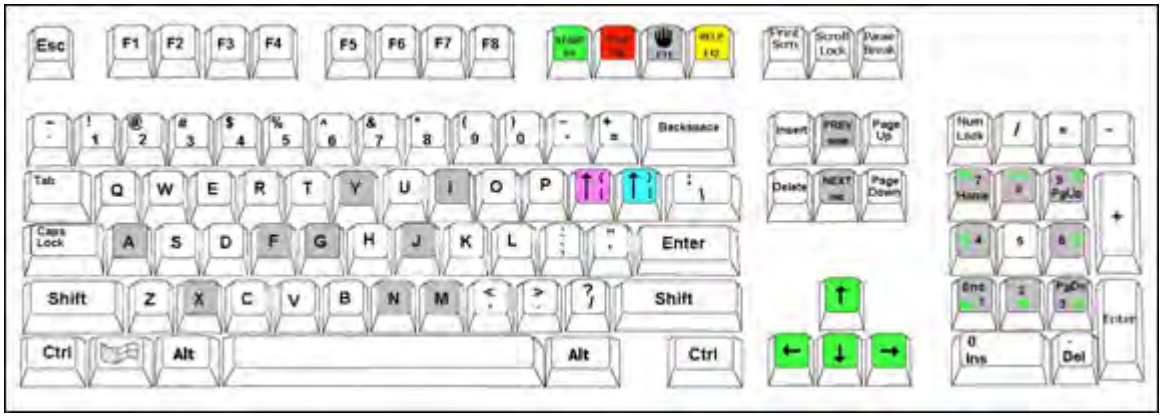
The following features are not supported when the touch screen is disabled:

- CutPro Wizard
- Align Wizard
- Interface diagnostics

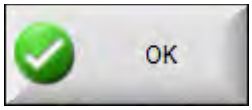
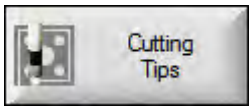
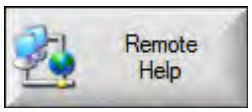
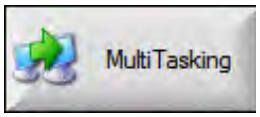
When the CNC is switched to keyboard operation, the soft keys display with pictures of the key combinations:

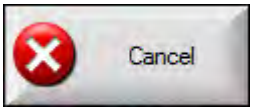
PC Keyboard

Hypertherm CNCs can support a USB, PC keyboard. You can use a keyboard to perform functions and data entry in the Phoenix software.



The following tables present common key combinations you will need to navigate and enter data into the CNC using only a keyboard.

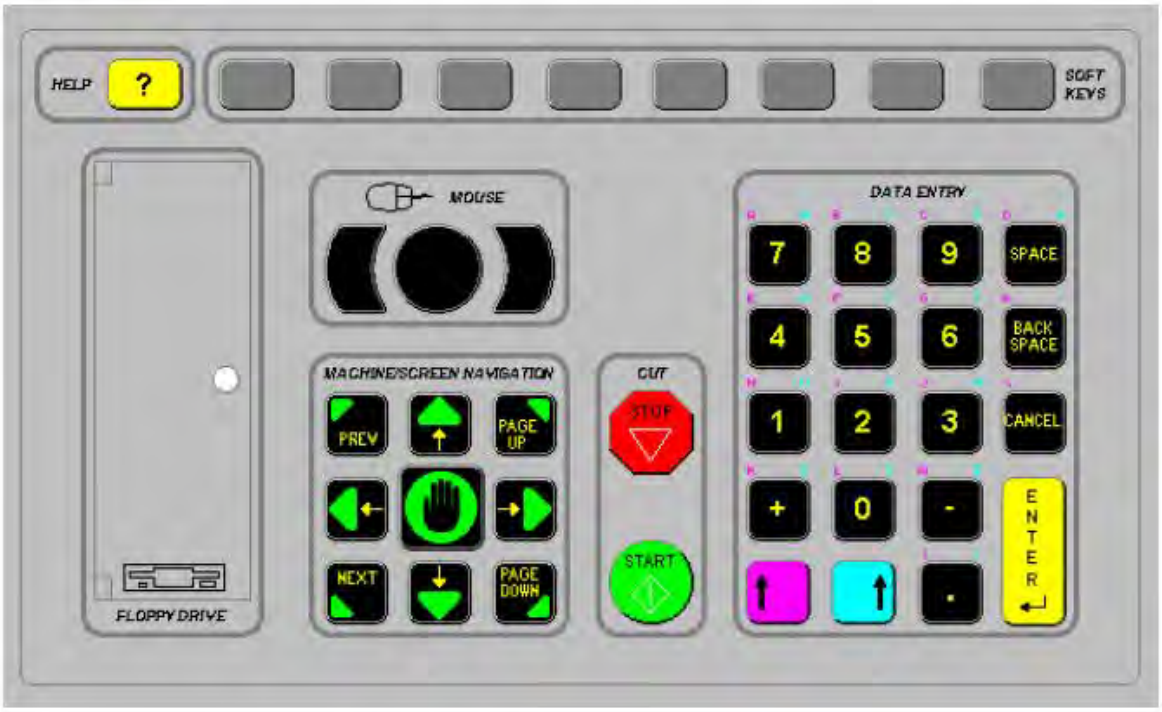
Key	Function
F1 to **F8**	Soft Keys **F1** to **F8** F1 through F8 activate the lower row of soft keys from left to right.
Shift **+** **Enter**	**Shift** + **Enter** accepts changes in a screen and equals the **OK** soft key. 
Enter	**Enter** moves from field to field on the screen the same way **Tab** does.
Left Bracket **[**	**[+ Function key** accesses the upper row of screen soft keys from left to right. For example, **[+ F2** opens the Plasma 1 cut chart screen.  **[+ F12** opens Cutting Tips 
Right Bracket **]**	**Right bracket** is equivalent to the **R-Shift** on onscreen messages. For example, in the message below, press **] + F8** to add a folder. **] + F4** opens remote Help.  **] + F2** opens MultiTasking  **] + 0 – 9** changes the Watch Window. 
Tab	**Tab** moves from field to field on the screen. **Shift + Tab** moves to the previous field.
F9	Program start
F10 **Pause**	Program stop
F11	Toggles between Manual and Main screens.

Key	Function
F12	Opens the help file. Press **F8** to exit the help file.
Arrow keys	In Manual mode, the arrow keys control manual motion. **Up arrow** and **Down Arrow** scroll through selections in a list. **Left Arrow** and **Right Arrow** select radio buttons. For example, use the Left and Right Arrow keys to select the On and Off radio buttons shown below. 
Esc	**Escape** exits a screen without saving changes and equals the Cancel soft key. 
+/-	**Plus** and **minus** on the numeric keypad Zoom In/Out in the part window.  Zoom In/Out is enabled when you choose View Sheet on the Main screen.
Backspace	**Backspace** deletes the last character entered.

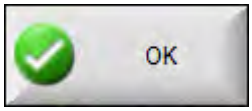
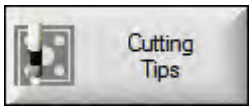
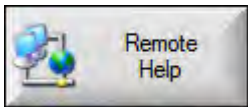
Custom Keypad

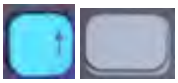
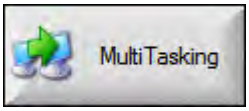
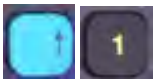
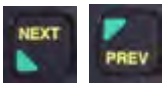
Many legacy Hypertherm CNCs are equipped with a custom keypad similar to the one below. The row of eight gray keys correspond to the Phoenix soft keys on screen. The following graphic shows the keypad from the EDGE® II CNC as an example. Phoenix Version 9.71 can be controlled by this and other keypads.

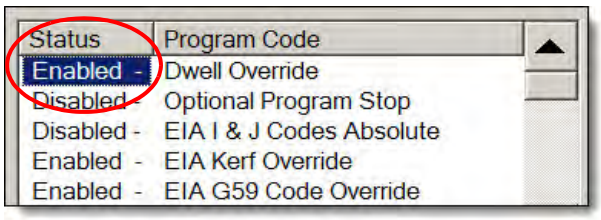
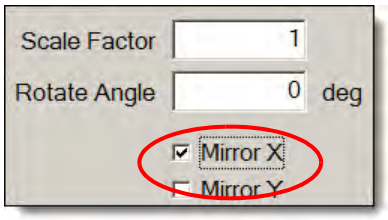
Front Panel



The following tables present common key combinations you will need to navigate and enter data using a keypad-equipped CNC.

Key	Description
	Screen soft keys F1 – F8, lower row, left to right.
	**Left Shift + Right Shift + Enter** Accepts changes in a screen and equals the OK soft key. 
	**Left Shift** (purple up arrow)
	**Left Shift** + **F1 – F8** access the upper row of screen soft keys, from left to right. For example, Left Shift + F2 opens the Plasma 1 Cut Chart. 
	When entering data, press Left Shift with a number to enter the purple characters on the keypad. For example, **Left Shift** + **7** enters **A**.
	**Left Shift** + **?** opens Cutting Tips. 
	Left Shift 0+ **Right Shift** + **Enter** accepts changes made to a screen.
	Left Shift is equivalent to the **Left Bracket** [.
	**Right Shift** (blue up arrow) **Right Shif**t + **F8** performs an action specified in the screen prompt "*Double-click to perform a function*".
	When entering data, press **Right Shift** with a number to access the blue characters on the keypad. For example, **Right Shift** + **7** enters **N**.
	Right Shift + **F4** opens Remote Help. 

Key	Description
	**Right Shift** + **F2** opens MultiTasking. 
	**Right Shift** + **0 – 9** changes the Watch window. 
	**Next/Prev** Moves from field to field on the screen. **Next** works the same as the **Tab** key on a PC keyboard.
	**Enter** Moves from field to field on the screen. **Enter** works the same as the **Tab** key on a PC keyboard.
	**Page Up/Page Down** pages through a list of options in a drop-down list.
	**Cancel** Exits a screen without saving changes. Same as **Escape** (Esc) on a PC keyboard and the **Cancel** soft key on screen.
	**Plus (+)** and **Minus (-)** control Zoom In and Zoom Out in the part window.  The Zoom soft keys are enabled when you choose View Sheet on the Main screen.
	The**?** key opens the Phoenix help file. Press **F8** to exit help.
	The **Manual** key toggles between the Main screen and Manual screen.

Key	Description
	**Arrow Keys** **Up Arrow** and **Down Arrow** scroll through selections in a list. **Left Arrow** and **Right Arrow** select radio buttons. For example, use the Left and Right Arrows to select the radio buttons Off and On. 
	**Space** changes state for a selection in a list. For example, on the Cutting Screen, use **Space** to toggle between Enabled and Disabled for program code status.  Space changes the state for a checkbox. 
	**Backspace** deletes the last character entered.
	**Program Start** and **Stop** perform only these functions on the keypad.

Updating the Phoenix Software

Hypertherm provides regular updates to the Phoenix software. You can download the most current software from *www.hypertherm.com*.

- Phoenix software update (update.exe)
- Phoenix Help file (Help.exe)
- Cut charts (CutChart.exe)

Follow the instructions on the web page to download the updates in your language.

Before you update the Phoenix software, follow these guidelines:

- Back up your system files: On the Main screen, choose Files> Save to Disk> Save System Files to Disk. See the section *Saving System Files* on page 250 for more information.
- Copy the files that you download from Hypertherm.com to the root folder of a USB memory stick.
- Be prepared to restart the CNC after you have updated the software.

Updating the software

1. At the CNC, insert the memory stick that contains the file update.exe into a USB port.

   Verify that update.exe resides in the root folder of the memory stick.

2. On the Main screen, choose Setups > Password. If you are not using a keyboard, double-tap the screen to display an onscreen keyboard.

3. Enter *updatesoftware* (all lower case, one word) and choose Enter. The Phoenix software automatically reads the memory stick and installs the new software.

Updating the cut charts

Hypertherm provides cut charts in two different file types: .fac and .usr. The .fac files are the factory-default cut charts. These cut charts cannot be changed. The .usr cut charts contain any changes you have made to a cut chart and saved with the Save Process soft key.

The cut chart update file (CutChart.exe) contains both .fac and .usr cut chart files. The update automatically overwrites all of the .usr cut charts. Before installing the update, back up your modified cut charts.

Hypertherm recommends saving modified cut charts as custom cut charts. When you create a custom cut chart, Phoenix creates a .usr file with a unique name. This prevents the custom cut charts from being overwritten by the .usr files in CutChart.exe. See *Creating a new cut chart* on page 172.

Backing up modified cut charts

1. At the CNC, connect a memory stick into a USB port.

2. On the Main screen, choose one of the cut chart soft keys, such as Plasma 1 Cut Chart.

3. Choose the Save Cut Charts soft key. Phoenix copies all the cut charts associated with the Plasma 1 Torch Type onto the memory stick.

Updating the cut charts

1. At the CNC, insert the memory stick that contains the file CutChart.exe into a USB port.

   Verify that CutChart.exe resides in the root folder of the memory stick.

2. On the Main screen, choose Process, and choose one of the cut chart soft keys such as Plasma 1 Cut Chart.

3. Choose the Load Cut Charts soft key, then choose Yes when prompted to load cut charts from the memory stick. Phoenix extracts the cut charts and copies them to the hard drive.

4. If you have modified cut charts to copy back onto the hard drive, you will need to exit Phoenix and use Windows® Explorer to copy the .usr files back onto the hard drive. The cut chart folder is c:\Phoenix\CutCharts.

Updating the help

1. At the CNC, insert the memory stick that contains the file Help.exe into a USB port.

   Verify that Help.exe resides in the root folder of the memory stick.

2. On the Main screen, choose Setups > Password. If you are not using a keyboard, double-tap the screen to display an onscreen keyboard.

3. Enter *updatehelp* (all lower case, one word) and choose Enter. The Phoenix software automatically reads the memory stick and installs the new help file.

Updating the manuals

Follow these steps to load new or updated manuals onto your CNC.

1. To obtain the newest manuals available from Hypertherm, visit www.hypertherm.com and choose the Downloads Library link.

2. In the Downloads Library, choose Product Type, then select a product name. For example, choose MAXPRO200 to display a list of manuals and other literature available for that product.

3. Choose the Manuals link and click to download a manual file.

4. Save the file in the root folder of a USB memory stick. Do not change the name of the file from its name in the Downloads Library. It will have a name like 807700r0.pdf.

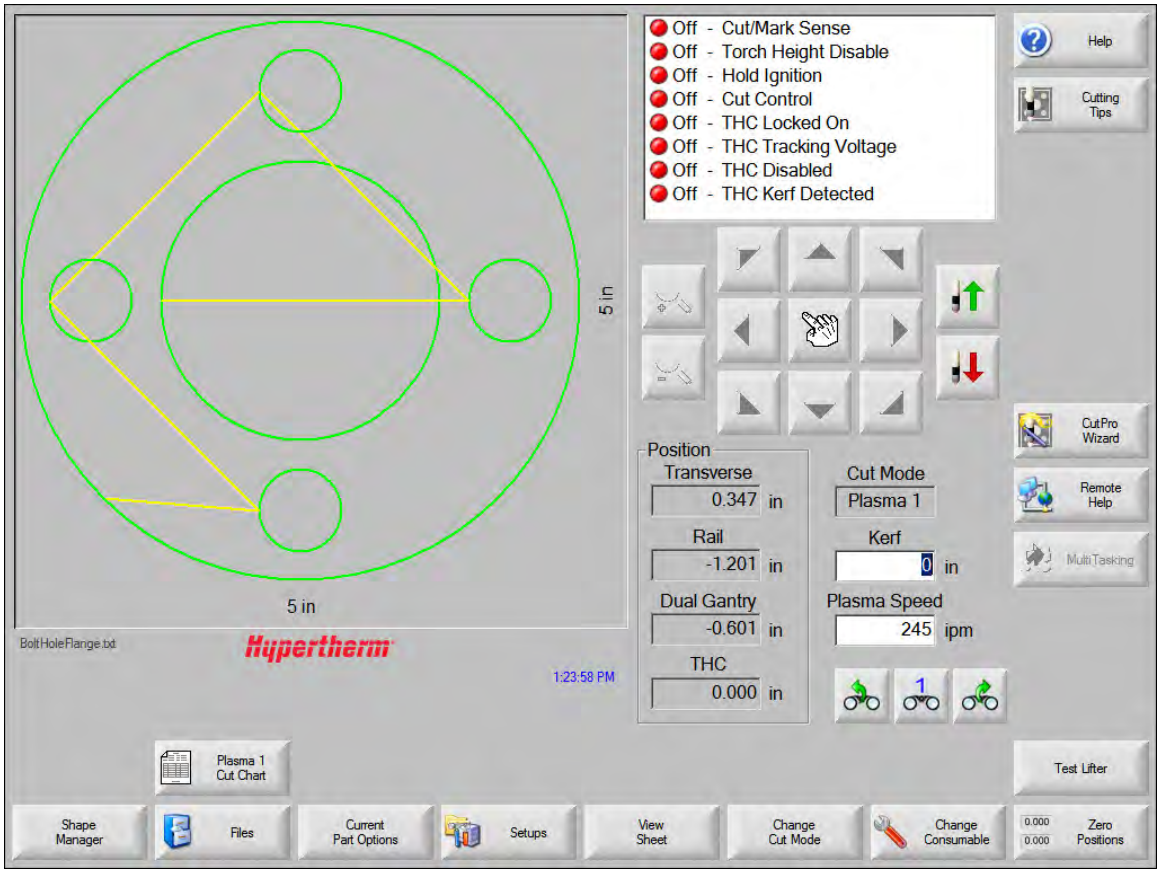
To load the manual onto your CNC, follow these steps. You can load more than one manual at a time onto the CNC as long as the files are in the root folder of the memory stick.

1. Insert the memory stick containing one or more Hypertherm product manuals into a USB port on the CNC.

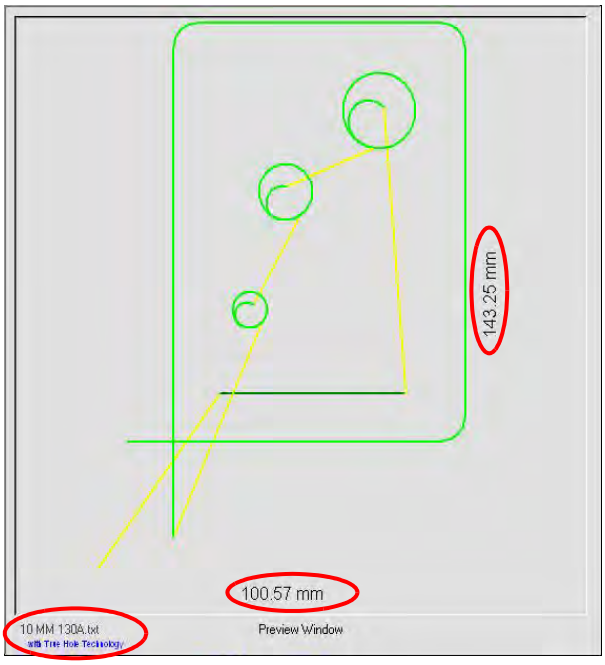
2. Choose Setups > Password, and enter *updatemanuals* (all lowercase, one word). The CNC copies the manuals from the memory stick to the hard drive.

Section 2
The Main Screen

The Main screen is the first screen you see when the CNC power is turned on.



Preview Window



The preview window shows the current part program and its dimensions. The name of the part program is displayed under this window, as well as the message, “with True Hole Technology” if the program uses this feature.

Watch Window

The watch window is the right part of the screen where monitoring features, such as a speedometer, job keys, positions indicators, cut mode, and time are displayed. You can configure this part of the screen, using the 10 different monitoring features in the Setups window. See *Watch Window Setup* on page 122 for more information.

Soft keys

Each of the Main Screen soft keys is described below:

Shape Manager opens the Shape Manager screen where you can load a simple shape, edit a part using the text editor or shape wizard or teach trace a part.

Files opens the Files screen where you can load, save, download or upload part files.

Current Part Options allows the part to be scaled, rotated, mirrored, and repeated.

Setups opens the Cutting screen which allows access to the Process, Watch Window setup, Diagnostics, and the password-protected setup screens.

View Part/View Sheet toggles the display of the part in the preview window. Phoenix displays the sheet dimensions that have been entered in Cutting screen.

Zoom +/- magnifies the part. After zooming out, the display can be zoomed in again by pressing the + key, which causes horizontal and vertical scroll bars to be displayed. Press the - key to zoom back out.

Manual Options lets you perform rip cuts, home the machine axes, and other manual operations.

Scroll Bars While the scroll bars are displayed and the control is not cutting, the view of the plate can be shifted horizontally and vertically by pressing and moving the scroll bar or by holding down a shift key and pressing the arrow keys on the keypad.

While the control is cutting, the view automatically shifts as the cut path reaches one of the edges of the view.

Change Cut Mode selects trial, oxyfuel, plasma, waterjet and laser cut modes, depending on the tooling selected in the Special Setups screen.

Change Consumable opens the Change Consumable screen.

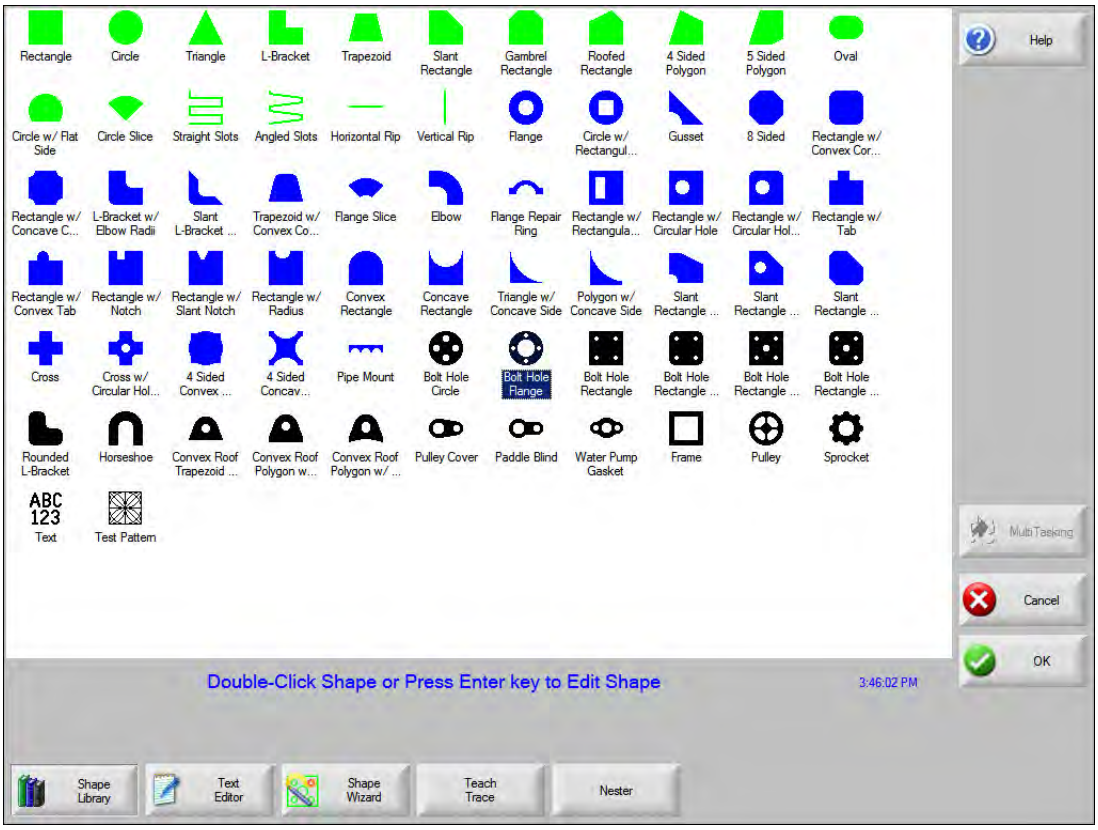
Zero Positions sets the current positions to zero on the Transverse, Rail, and Dual Gantry axes.

Section 3
Loading Parts

This section describes how to load a part from the Shape Library, a memory stick, or a host computer, save files, and import DXF files.

Loading a part from the Shape Library

The CNC contains a built-in Shape Library with more than 68 commonly used shapes. These shapes are *parametric*, that is, shapes whose size or geometry you can edit. The shapes in the library are color-coded from simplest (green) to most complex (black).



To select a simple shape:

1. On the Main screen, choose Shape Library.
2. Choose a shape.
3. Press OK.

Keypad operation:

1. Use the arrow keys to navigate to a shape.
2. Press Enter.

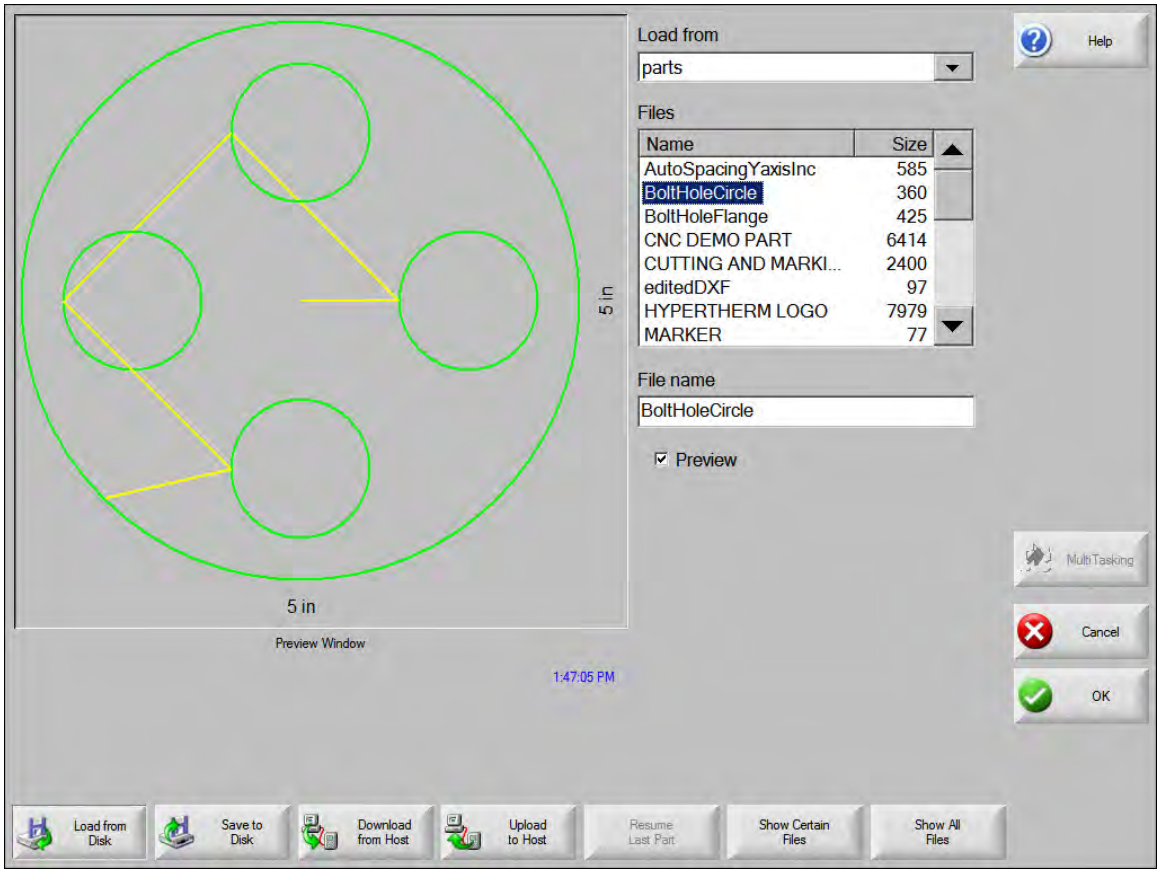
The shape is displayed with the default parameters or the parameters from the last time this shape was edited.

Loading a part

You can load part programs from the CNC hard drive, a USB memory stick or external mapped drives (network option) into working memory on the CNC.

The following screen is used to load a part from a USB memory stick or the hard drive. After all the parameters are set, press Enter on the keyboard to load the part

Permission to add or remove files and folders from the hard drive is assigned in the Status/Feature list on the password-protected Special Setups screen.



Load from Select the source from which to load a part: USB memory stick, or a folder on the hard drive. To add or remove a folder, double-click on the touch screen where indicated.

Keypad operation: To select a different folder, use the ↑ and ↓ keys. To add or remove new folder use the + or - key.

Files Lists the files in the folder you selected. Choose the name of the file that you want to load. You can select multiple files only if you are loading them from a USB memory stick to the hard drive.

Keypad operation: To scroll through different files, use the ↑, ↓, Page Up and Page Down keys. To remove a file, use the - key. To select multiple files to load, highlight the first file selection, then use the ↑ and ↓ keys while pressing the shift key to highlight the remaining files.

File name Displays the name of the file you selected. To remove a file, highlight the file name and double-click on the touch screen where indicated.

Keypad operation: To remove a file using the keyboard, use the - key.

Preview Check this box to preview the files you selected in the Preview Window.

Load to Select the destination for the part: load it for cutting or save it in a folder on the hard drive. To add or remove a folder, double-click on the touch screen where indicated. This selection is available only if you are loading the part from the USB memory stick.

Keypad operation: To select a different folder, use the ↑ and ↓ keys. To add a new folder, use the + key. To remove a folder, use the - key.

Hard drive file name Enter the name for the file that you are loading on the hard drive. This selection is only available if you are loading the part from the USB memory stick.

Show Certain Files Allows you to search the selected folder for specific part files by using wildcard searches with both the asterisk (*) and question mark (?).

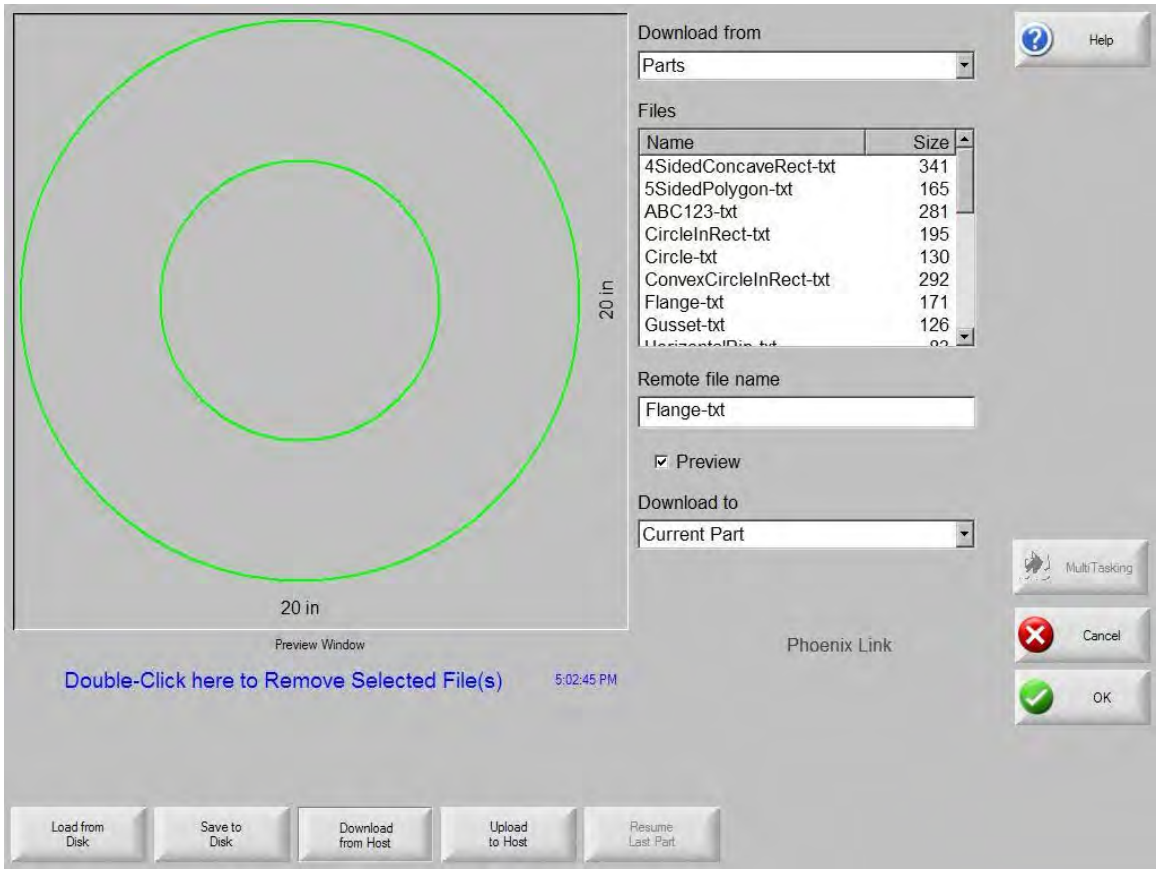
Keypad operation: To input the asterisk when using a keypad, press the left shift key and the backspace key. To input the question mark, press the right shift key and the backspace key.

Show All Files Allows you to switch from viewing the selected files to viewing all files with the predetermined file extensions selected in the Special Setups screen.

Downloading a part from a host computer

Use the screen below to download a part from a host computer through a RS-232C/ RS-422 serial port. After all the parameters below are set, press Enter on the keyboard to start the download.

Permission to add or remove files and folders from the hard drive is assigned in the Status/Feature list on the Special Setups screen.



Download from Selects the folder on the host computer from which you want to download a part. To add or remove a folder, double-click on the touch screen where indicated.

Keypad operation: To select a different folder, use the ↑ and ↓ keys on the keyboard. To add or remove new folder use the + or - key.

Files Lists the files in the download from folder that can be downloaded from the host computer.

Keypad operation: To scroll through different files, use the ↑, ↓, PAGE UP and PAGE DOWN keys. To select multiple files to download, highlight the first file selection, then use the ↑ and ↓ keys while pressing the shift key to highlight the remaining files.

Remote file name Enter the name of the remote file that will be downloaded from the host computer.

Preview Check this box to preview the file that you selected in the Files list box. To check or uncheck the box, press the Spacebar on the keyboard when the Preview box has the focus.

Download to Select where you want to download the part: to the current part in memory or to a folder on the local hard drive. If you select one of the local folders, the Local file name field displays.

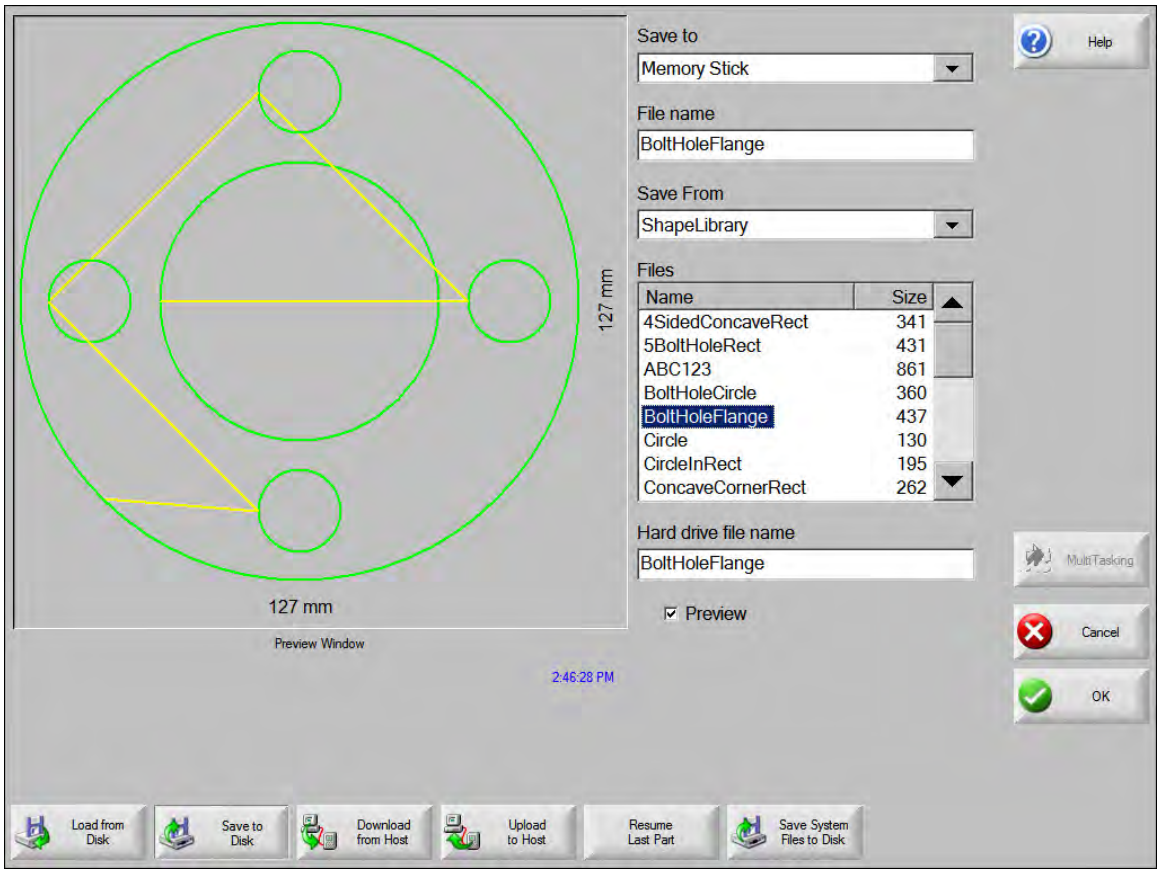
Keypad operation: To select a different folder, use the ↑ and ↓ keys. To add a new folder, use the + key. To remove a folder, use the - key.

Local file name The user-defined file name assigned to the file that is being downloaded to the hard drive.

Saving a part file

Use the following screen to save a part to a USB memory stick or the hard drive. After you have made all your selections and entries, choose OK to save the part.

Permission to add or remove files and folders from the hard drive is assigned in the Status/Feature list on the Special Setups screen.



Save to Select to save the file to the USB memory stick or to a folder on the hard drive. To add or remove a folder, double-click on the touch screen where indicated.

Keypad operation: To select a different folder, use the ↑ and ↓ keys on the keyboard. To add a new folder, use the + key on the keyboard. To remove a folder, use the - key on the keyboard.

File name Enter the file name that you are giving the file you are loading on the disk.

Save Original Text Hypertherm CNCs can import part files programmed for other CNCs. When you import one of these files, the Phoenix operating software translates the file into the format used by the Hypertherm CNC. The Save Original Text option saves the imported part file in its original format instead of the Hypertherm CNC format. This selection is not available if you are saving the file to the USB memory stick from the hard drive.

Save From Select whether you save from the current part or from a folder on the hard drive. To add or remove a folder, double-click on the touch screen where indicated. This selection is available only if saving to the USB memory stick from the hard drive.

Keypad operation: To select a different folder, use the ↑ and ↓ keys on the keyboard. To add a new folder, use the + key on the keyboard. To remove a folder, use the - key on the keyboard.

Files Select one or more part files from the list of all the files that are in the Load from folder that can be loaded from the disk. To remove a file, double-click on the touch screen where indicated. This selection and Multiple file selection are only available if you are saving files to the USB memory stick from the hard drive.

Keypad operation: To scroll through different files, use the ↑, ↓, PAGE UP and PAGE DOWN keys. To remove a file, use the - key. To select multiple files, highlight the first file selection, then use the ↑ and ↓ keys while pressing the shift key to highlight the remaining files.

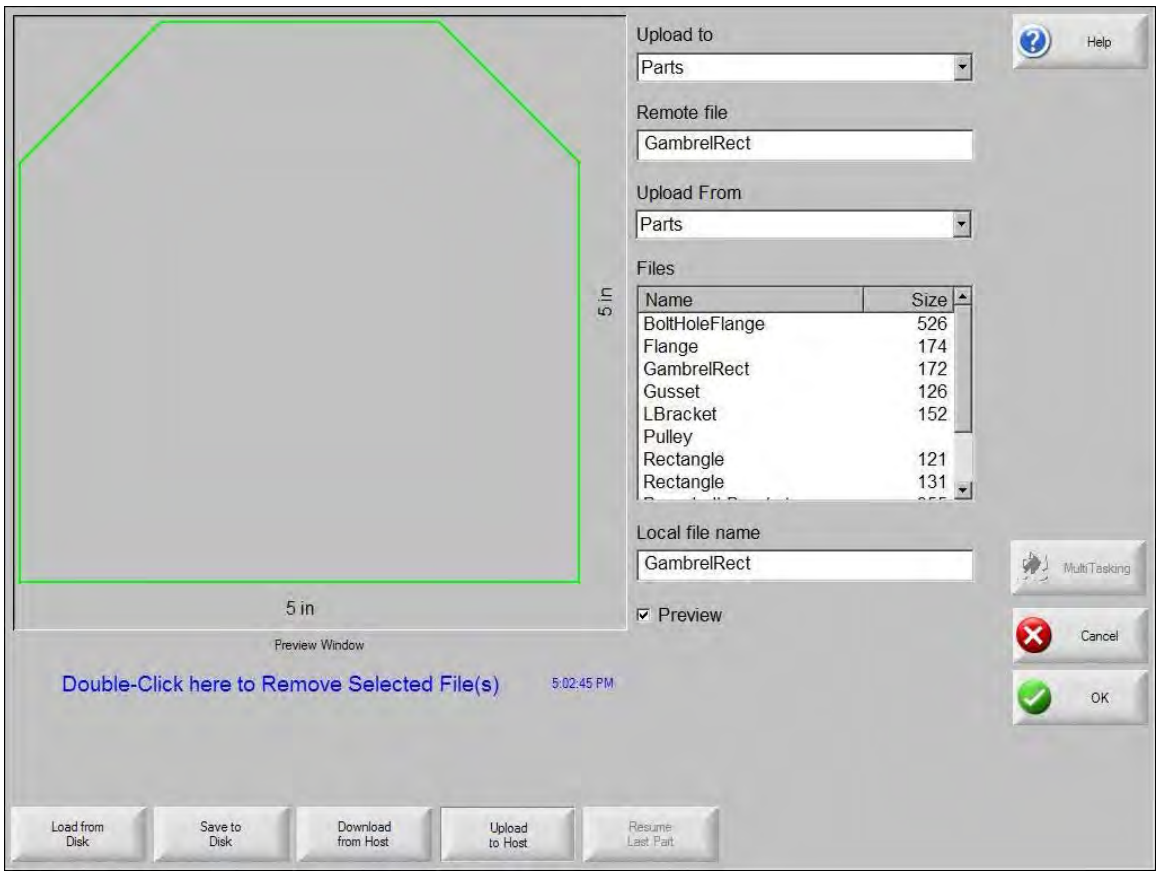
Hard drive file name Enter the name that you are giving to the file if you are loading it on the hard drive. This selection is available only if you are saving files to the USB memory stick from the hard drive.

Preview Check this box to preview the file that is selected in the Files list box. This selection is available only if you are saving files to the USB memory stick from the hard drive.

Keypad operation: To check or uncheck the box, press the Space key on the keypad when the Preview box has the focus.

Uploading part files to a host computer

Use this screen to upload a part to a host computer. After all the parameters are set, press Enter on the keyboard to start the upload.



Upload to Select the folder on the host computer to which you want to upload a file. To add or remove a folder, double-click on the touch screen where indicated.

Keypad operation: To select a different folder, use the ↑ and ↓ keys. To add a new folder, use the + key. To remove a folder, use the - key.

Remote file name Enter the name of the file that you are uploading to the host computer.

Upload from Select whether you upload the current part in memory or from a folder on the local hard drive. If you select one of the local directories, the Files, Local file name and Preview fields display. To add or remove a folder, double-click it on the touch screen. This selection is available only if you are saving a file to the USB memory stick from the hard drive.

Keypad operation: To select a different folder, use the ↑ and ↓ keys. To add a new folder, use the + key. To remove a folder, use the - key.

Files Lists all the files in the upload from folder that can be uploaded to the host computer. To remove a file, double-click on the touch screen.

Keypad operation: To scroll through different files, use the ↑, ↓, Page Up and Page Down keys. To remove a file, use the - key. To select multiple files to upload, highlight the first file selection, then use the ↑ and ↓ keys while pressing the shift key to highlight the remaining files.

Local file name The name of the local file that will be uploaded to the host computer.

Preview If you check this box, you can preview the file you selected in the Preview window.

Keypad operation: To check or uncheck the box, press the Spacebar on the keyboard when the Preview box has the focus.

Importing DXF files

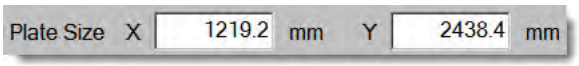
Hypertherm CNCs offer two styles of automated DXF import. The first DXF feature allows the CAD designer to prepare a DXF file that includes the location of pierces, pierce order and direction. When this file is loaded, the CNC translates the file into an EIA format part program.

The second type of DXF file is a fully automatic DXF import feature that allows the operator to select lead style and length. The CNC Auto DXF software automatically places the lead-in and lead-out based on the operator selections and creates an EIA format part program ready for use and the CNC.

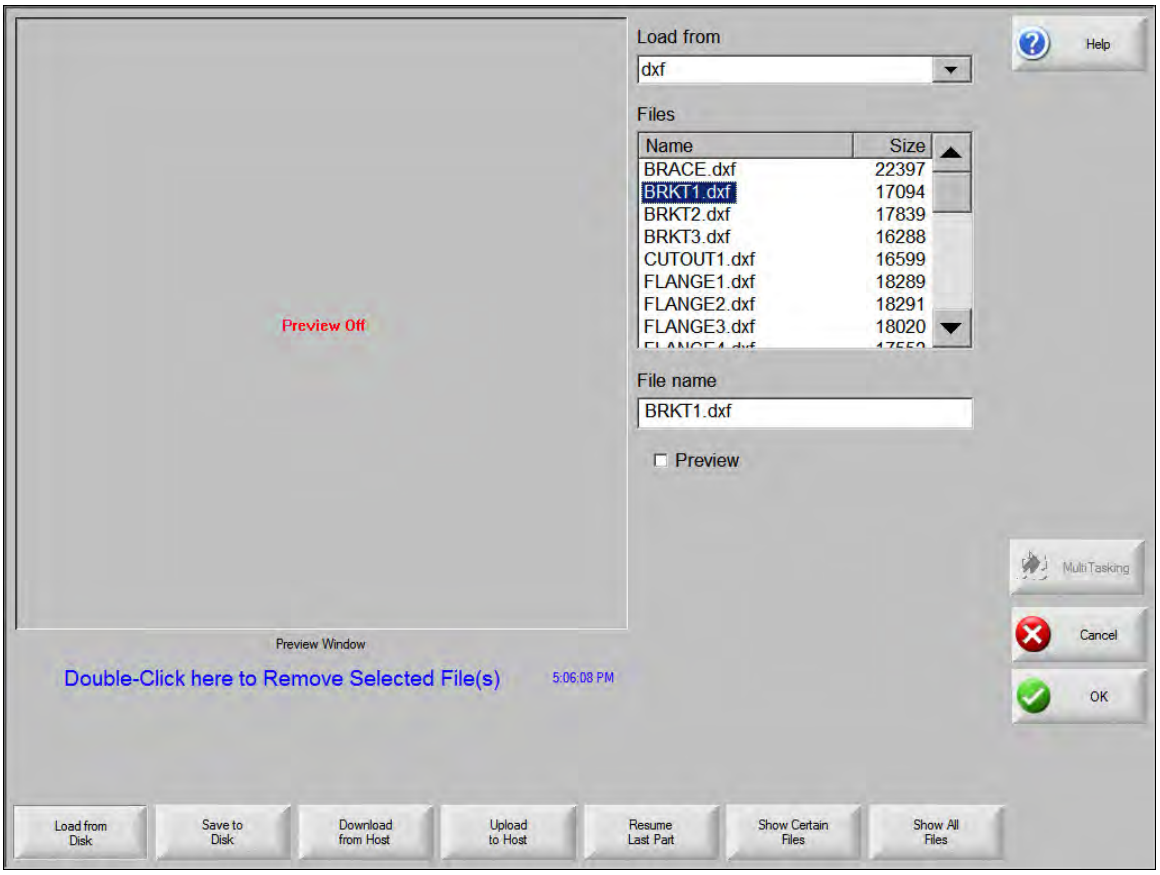
To load a DXF file, open the Files, Load from Disk screen and select the source location and file.

Notes:

- Before you can load DXF files into the CNC, enter DXF as a file extension in the Setups > Password > Special Setups screen.
- Verify the plate size for the job in the Setups > Cutting screen:



  Then, in the DXF file, check the distance between the part origin and the part geometry. The CNC requires that the location of the part geometry in relation to the part origin be less than the plate size dimensions.
- When the CNC translates a DXF file, it saves the resulting EIA text file in the same location as the source DXF file. If you are retrieving your DXF files from a network location, the CNC must have read and write privileges for that network location. Either make sure the CNC can write to the network location, or transfer any DXF file that you expect to translate for use on the CNC to the CNC and avoid translating DXF files from network locations.



Load from Select DXF from the dropdown list.

File name Select a DXF file from the scroll box.

Preview Check this box to preview the selected file.

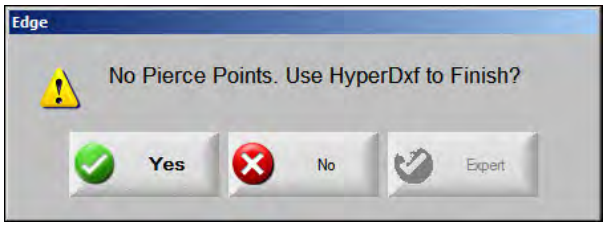
Show Certain Files This soft key allows you to show only certain files from the selected folder. Both the asterisk and question mark may be used in defining the files to show.

Keypad operation: The asterisk is generated by holding down the left shift key and pressing the backspace key. The question mark is generated by holding down the right shift key and pressing the backspace key.

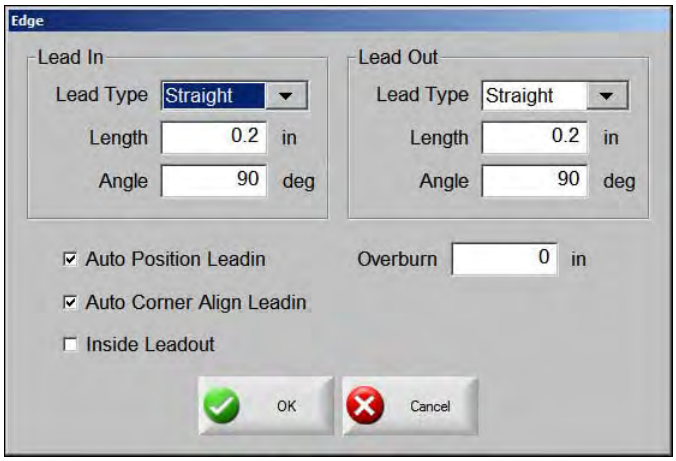
Show All Files This soft key allows the operator to undo Show Certain Files.

Raw DXF files

If the CNC does not detect pierce information in the DXF file, you have the option to use the Hyper DXF translation utility to import the file and add lead-in and lead-out information.



If you select Yes, a configuration screen displays fields to define the lead-in and lead-out format.



Lead In and Lead Out Select a Straight or Radius lead-in or lead-out.

Length and Radius Select the lead-in or lead-out length or radius.

Angle Select the angle, in degrees, for the lead-in or lead-out.

Auto Position Lead-in If this box is checked, the software attempts to find a suitable corner for the lead-in.

Auto Corner Align Lead-in If this box is checked, the software attempts to find a suitable corner for the lead-in.

Inside Lead-out If this box is checked, a lead-out is used on both internal and external cuts. If the box is not checked, lead-outs are added to external cuts only.

Overburn Overburn provides an overlapping cut in the lead-in and lead-out area of a hole.

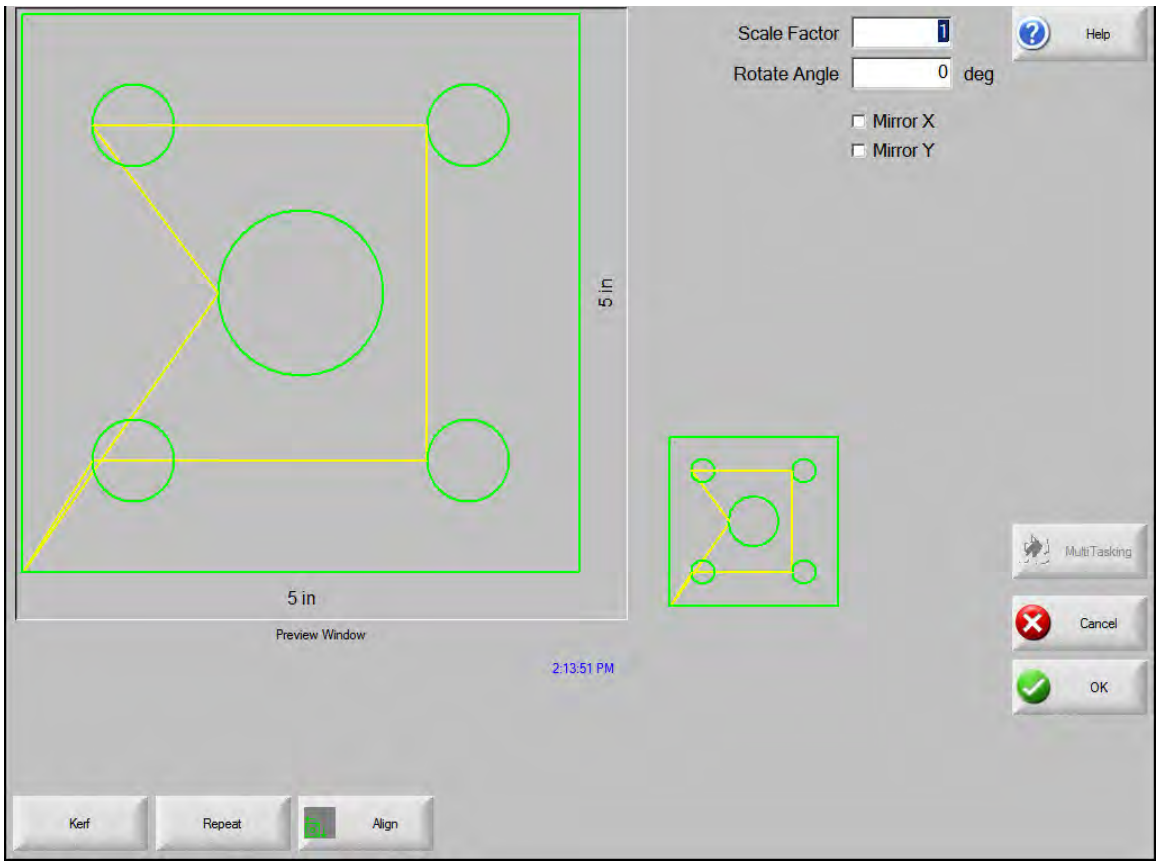
After import, an EIA part program with a .txt extension is created and placed in the source folder.

When the CNC translates a DXF file, it saves the resulting EIA text file in the same location as the source DXF file. If you are retrieving your DXF files from a network location, the CNC must have read and write privileges for that network location. Either make sure the CNC can write to the network location, or transfer any DXF file that you expect to translate for use on the CNC to the CNC and avoid translating DXF files from network locations.

Section 4

Arranging Parts

The Current Part Options screen lets you customize the layout of the current part. The Preview window shows the effects of each part option.



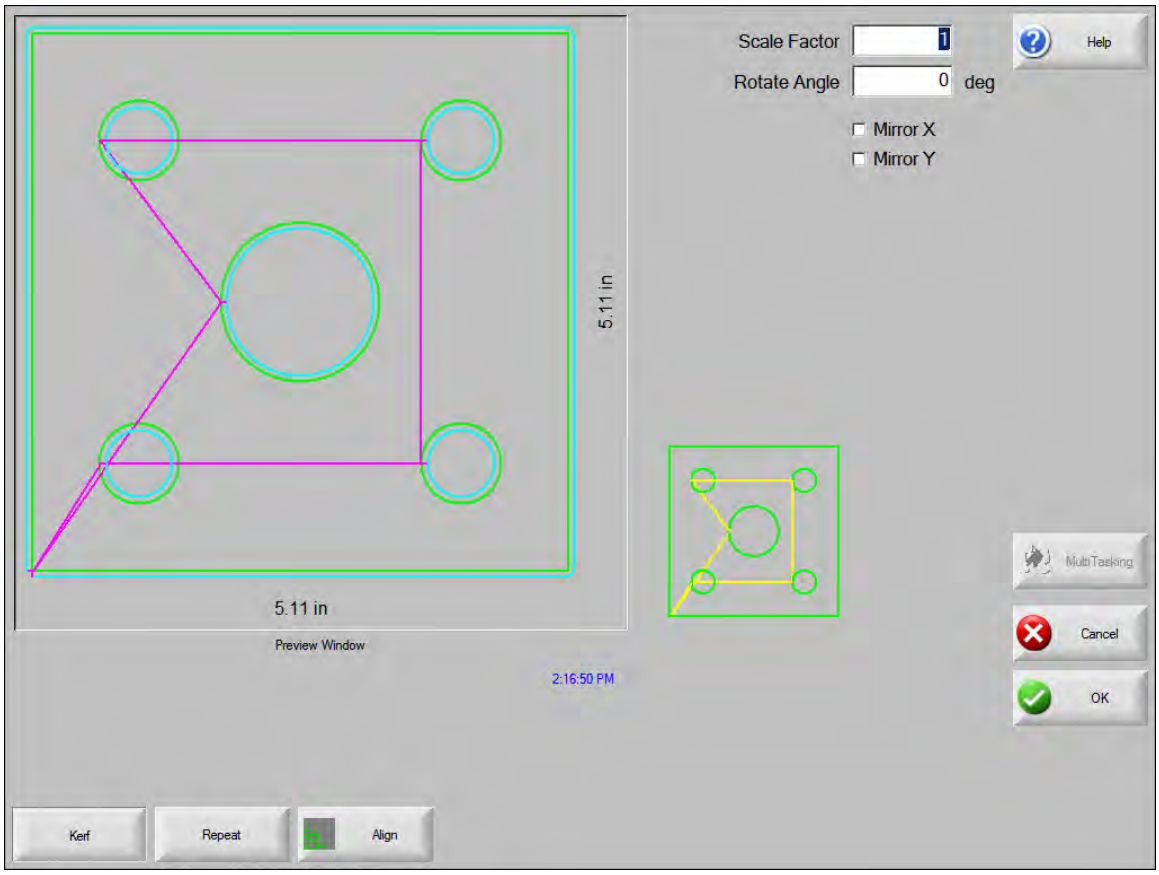
Scale Factor Allows the operator to scale the current part in memory by a programmed factor. After a new scale factor has been entered, the part is redrawn and its overall dimensions shown. The scale factor must be greater than zero.

Rotate Angle Allows the operator to rotate the current part in memory by a programmed value. After a new rotate angle has been entered, the new part is displayed in the Preview window. The rotate angle can be any positive or negative angle.

Mirror X/Mirror Y These check boxes make the X or Y dimensions negative. The result is a mirror image of the current part in memory.

Keypad operation: Press the Next or Enter key to toggle to the X or Y field. When the cursor is on the field, press the Space key to enter a check mark in the current highlighted field.

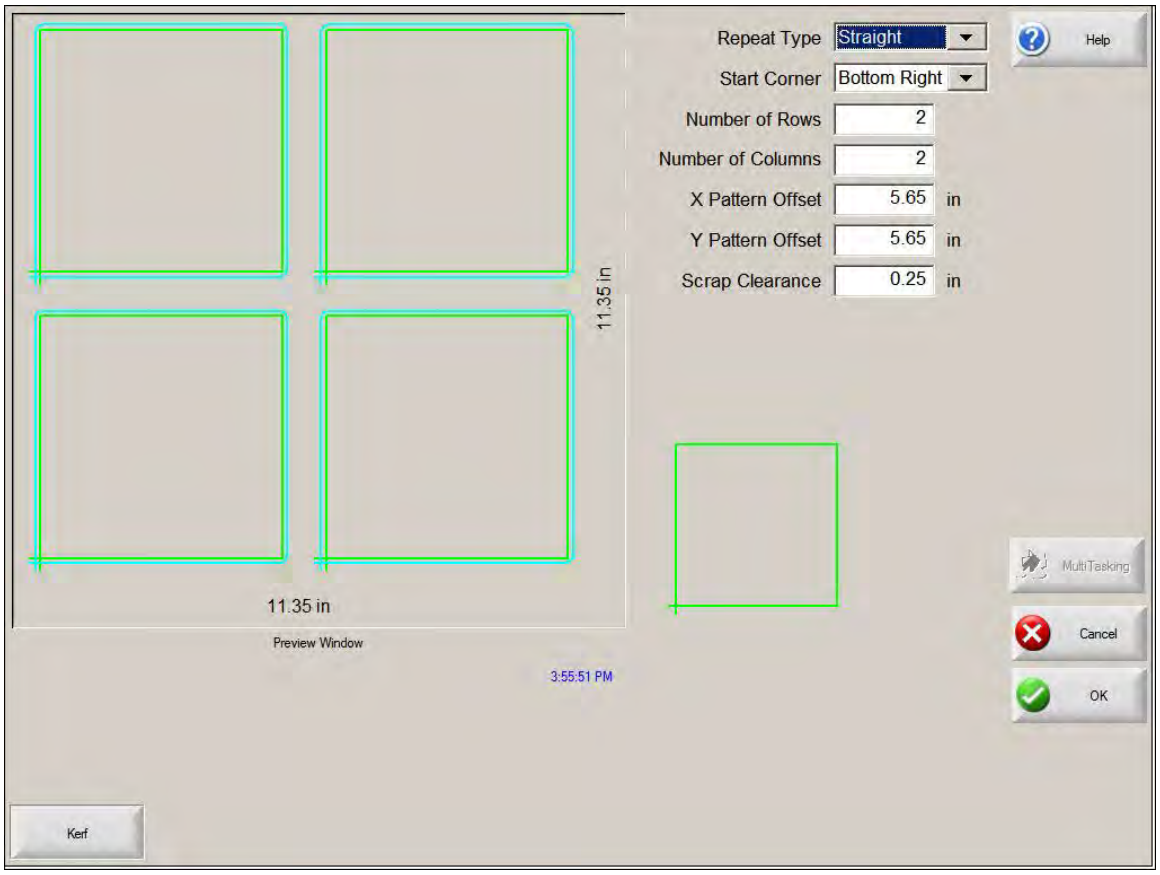
Kerf Press the Kerf soft key to show the kerf path in light blue. This helps you see the Kerf path before cutting. Press the button again to turn the Kerf part graphics off.



Repeating parts

The control has three built-in automatic repeat types: straight, staggered, and nested.

Straight repeat



Repeat Type Selects the type of the three repeat: Straight, Staggered or Nested.

Start Corner Selects the corner of the plate to start the shape repeat.

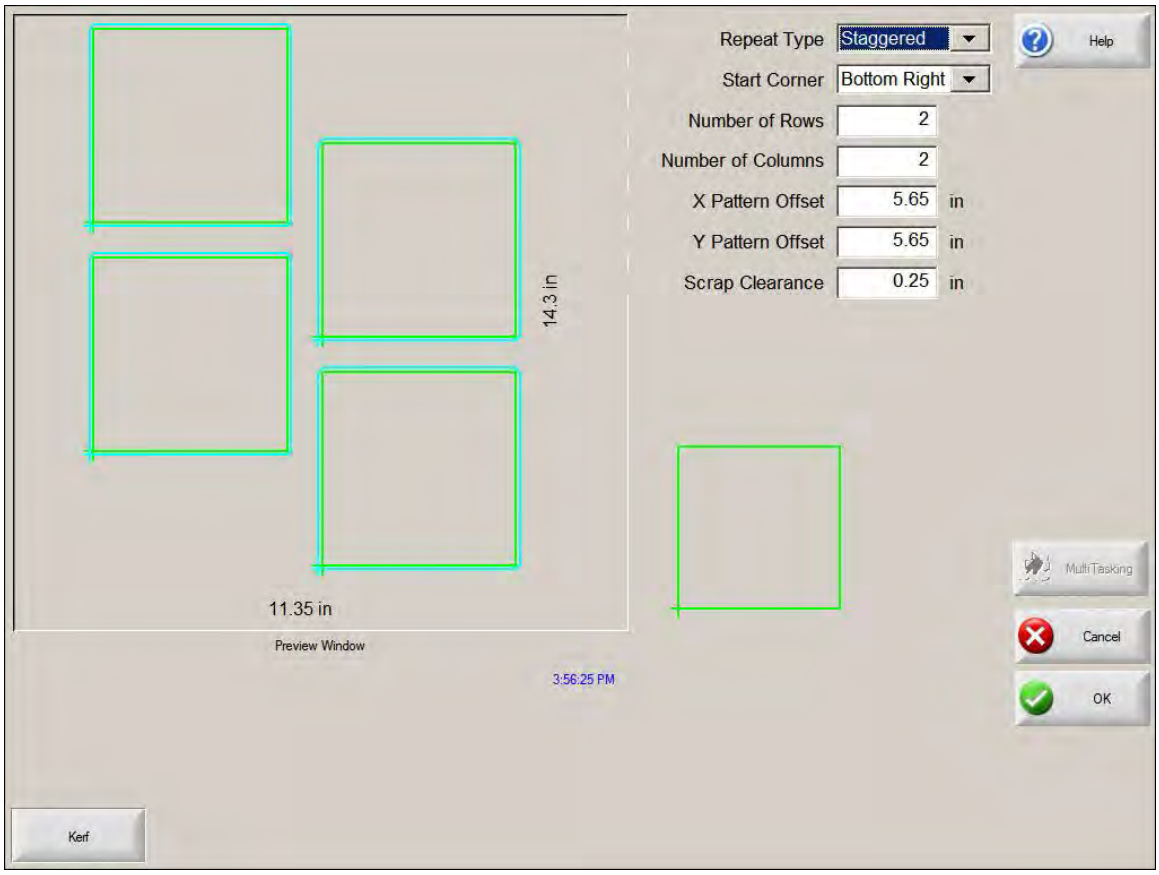
Number of Rows Enter the number of rows to cut.

Number of Columns Enter the number of columns to cut.

X Pattern Offset/Y Pattern Offset Automatically calculates the pattern offset based on the dimension of the current part in memory.

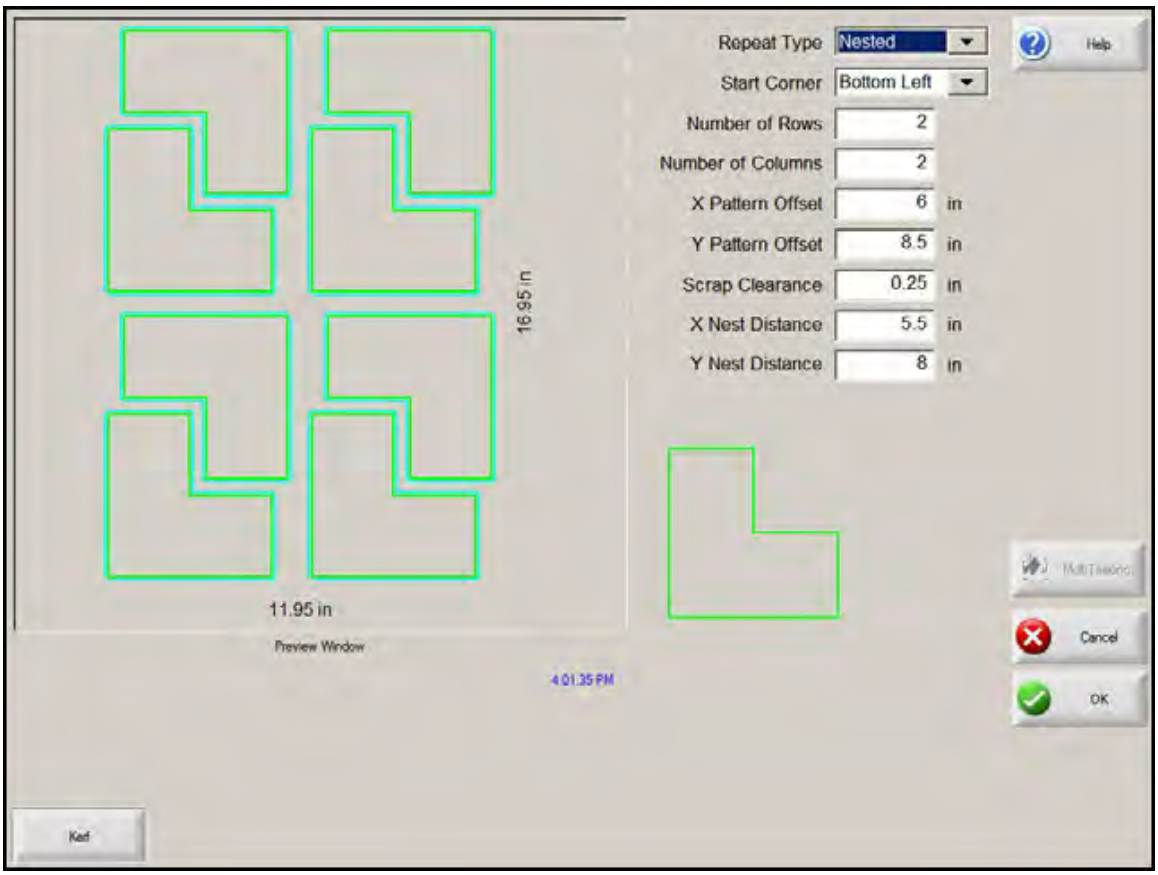
Scrap Clearance Inserts scrap clearance between parts in the grid pattern. The same value is used for X and Y dimensions.

Staggered repeat



X Nest Distance/Y Nest Distance Automatically calculates the nest offset based on the dimension of the current part in memory. This parameter is only available for the Nested type of repeat.

Nested repeat



Pattern Offsets Automatically calculates the minimum spacing required between repeated parts. The spacing is based on the size of the part (including lead-in and lead-out), the kerf value and scrap clearance. This calculated spacing allows the part to be repeated without overlapping.

You can use this pre-calculated value or select new values manually. If you enter new pattern offset values, Phoenix automatically draws the new pattern with the new values.

Nest Distance Automatically calculates the minimum spacing required between nested parts. The spacing is based on the size of the part (including lead-in and lead-out), the kerf value and scrap clearance. This calculated spacing allows the part to be repeated without overlapping.

You can choose to use this pre-calculated value or select new values manually. As you enter new offset values, the CNC automatically draws the new nested pattern with the new values.

Tip: If you change Nest Distance values manually, start with a simple nest (1 column, 1 row) and perform adjustments based on the display. The pattern in the Preview window changes as you change values. When the Nest Distance is what you want, increase the nest size to a 2-column, 2-row nest, then adjust the X and Y pattern offsets again. When you have the nest spacing you want, increase the nest size to the maximum that the plate allows.

Aligning parts

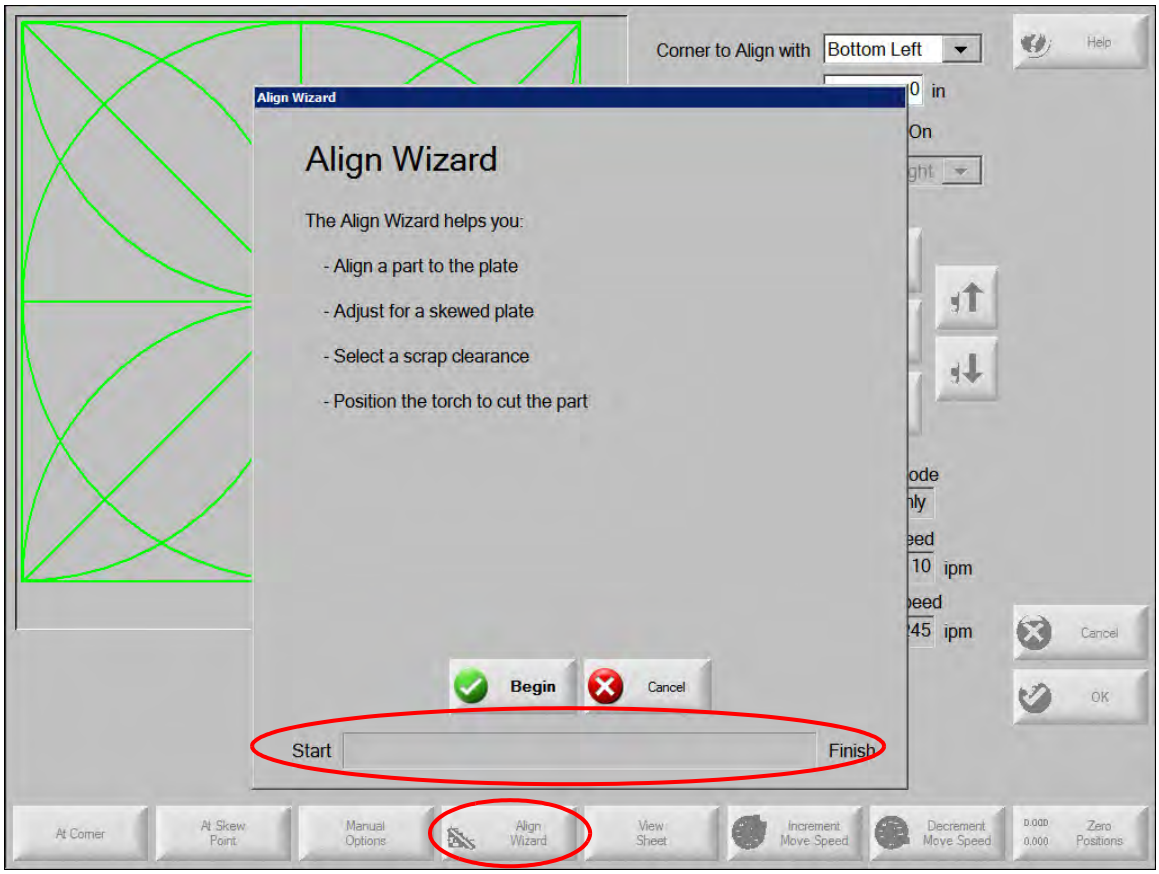
This screen allows you to:

- Launch the Align Wizard.
- Align the current part to one of the four corners of the plate. This is common with parts that have an internal pierce point such as a flange.
- Accommodate skewed plates when aligning the part. This is commonly used with a nest of parts that has a small margin of error for placement of the nest on the plate.

Align Wizard

The Align Wizard automates the sequence of operations to enter coordinates for a skewed plate on the table and to align parts to a skewed or aligned plate.

The Align Wizard opens automatically from the Align screen or you can press the Align Wizard soft key on the Align window.



As you work with the Align Wizard, it keeps track of your progress and displays it at the bottom of the wizard window in the progress bar.

You can choose to use the torch or a laser pointer to align the plate. If you choose the laser pointer, you must have a marker offset value of at least 1 entered for Marker Offset 10, 11, or 12 on the Special Setups screen.

Aligning parts manually

To align a part manually on the plate:

1. Set parameters that are needed to align your part in the upper right corner of the screen.
2. Move the torch to the first corner location (Corner to Align with) using the jog keys.
3. Press At Corner.
4. If you are aligning a part, go to step 7.
5. Move the torch to a point along the edge of the plate toward the selected Skew Reference.
6. Press At Skew Point.
7. Press OK. The machine will move to the start point for the part and return to the Main screen and be ready for cutting.

Corner to Align Selects the corner of the plate to align the part in.

Scrap Clearance This is the amount of clearance between the edge of the plate and the part the control will add in when moving to the start point of the part.

Skew Adjustment This determines if the control will adjust for plate skew when performing the align function.

Skew Reference This is the skew reference corner which you will move towards and mark a point along the edge. This is only available if Skew Adjustment is On.

At Corner Press this soft key when at the corner of the plate you want to align the part in.

At Skew Point Press this soft key when at the edge of the plate for skew adjustment. This is only available if Skew Adjustment is On.

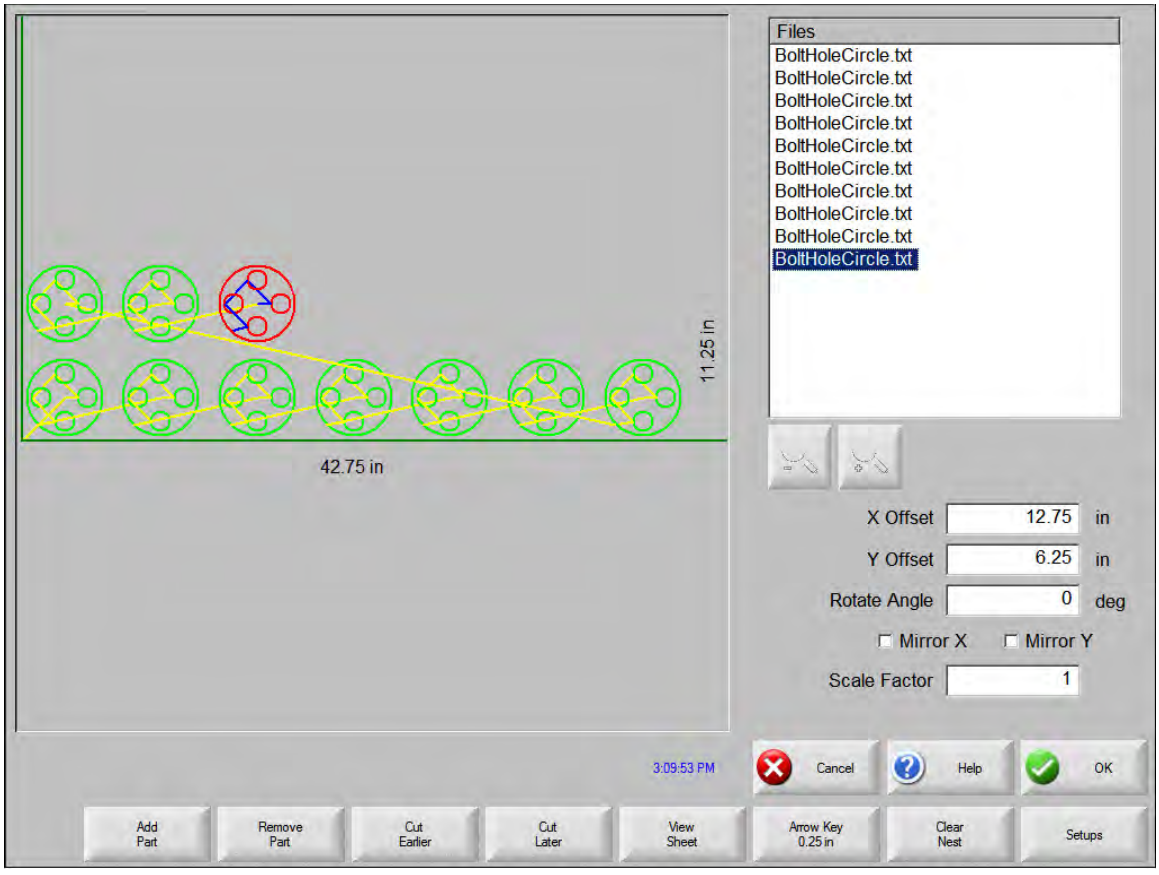
Nesting parts

Manual nesting

To open the Nesting screen, choose the Shape Manager soft key on the Main screen, then choose Nester.

The main viewing area is the largest area of the screen and is located in the upper left corner. The edge of the plate is displayed in dark green. The plate size displayed is based on plate information that has been selected in the Cutting screen (choose the Setups soft key to open the Cutting screen).

The upper right corner of the main screen displays the part program list for the nest in order of cut sequence. In the lower right, part position and orientation information for the selected part program is displayed and can be manipulated for use as new parts are added.



Add Part Allows you to select a part program from a selected source to add to the nest.

Remove Part Remove the selected part from the nest parts list.

Cut Earlier Pressing the Cut Earlier soft key will move the selected part program to an earlier position in the part cut list. The sequence in which the parts are cut changes, but not the selected part location in the nest.

Cut Later Pressing the Cut Later soft key will move the selected part program to a later position in the part cut list. The sequence in which the parts are cut changes, but not the selected part location in the nest.

View Sheet / View Part View Sheet allows the viewing of a part as it would appear on the plate. After pressing the View Sheet soft key, the display window zooms out to show the part in relationship to the entire plate.

After zooming out, the display can be zoomed in again by pressing the + key, which causes horizontal and vertical scroll bars to be displayed. Pressing the - key will zoom back out.

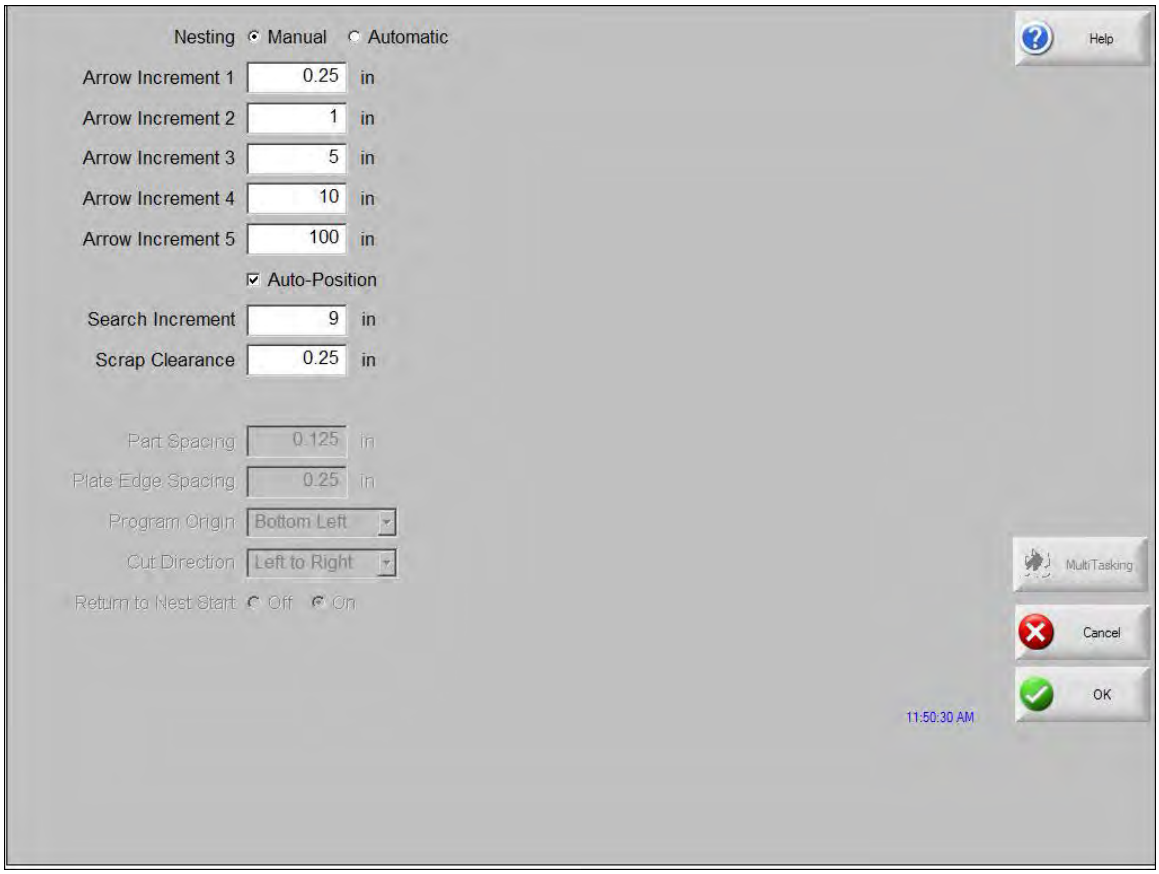
Arrow Key (Distance) The Arrow Key allows you to select one of five different preset move distances when the arrow keys are pressed to locate parts in the nest. These five distances are definable and are selected in the Nester setup screen.

Clear Nest Clear Nest will delete all the parts located in the nested parts list from temporary memory.

Setup Pressing the Setup soft key accesses the Nester setup screen for configuring the variable parameters when using Nester.

Nester setup

The following Setup parameters are used to configure the manual nesting process.



Nesting Select Manual.

Arrow Increment 1 – 5 At this screen, you can select different move increment dimensions. These dimensions are used as move distance references when the control arrow keys are pressed to place parts in position on the plate.

Autoposition Autoposition is an automated feature of the Nester software that enables block nesting. This type of nesting compares the overall block dimensions of the selected part and searches for the next available block on the plate that is large enough for the part.

Autoposition does not allow parts to be placed on top of other parts or inside other parts. However, it can be disabled if you want to add parts to scrap areas.

If Autoposition is not selected, imported parts are stacked in the lower left corner of the plate and must be manually arranged.

Search Increment The distance for the next available block on the plate that can be used for the next nested part.

Scrap Clearance The amount of space that is added to a block in the nest.

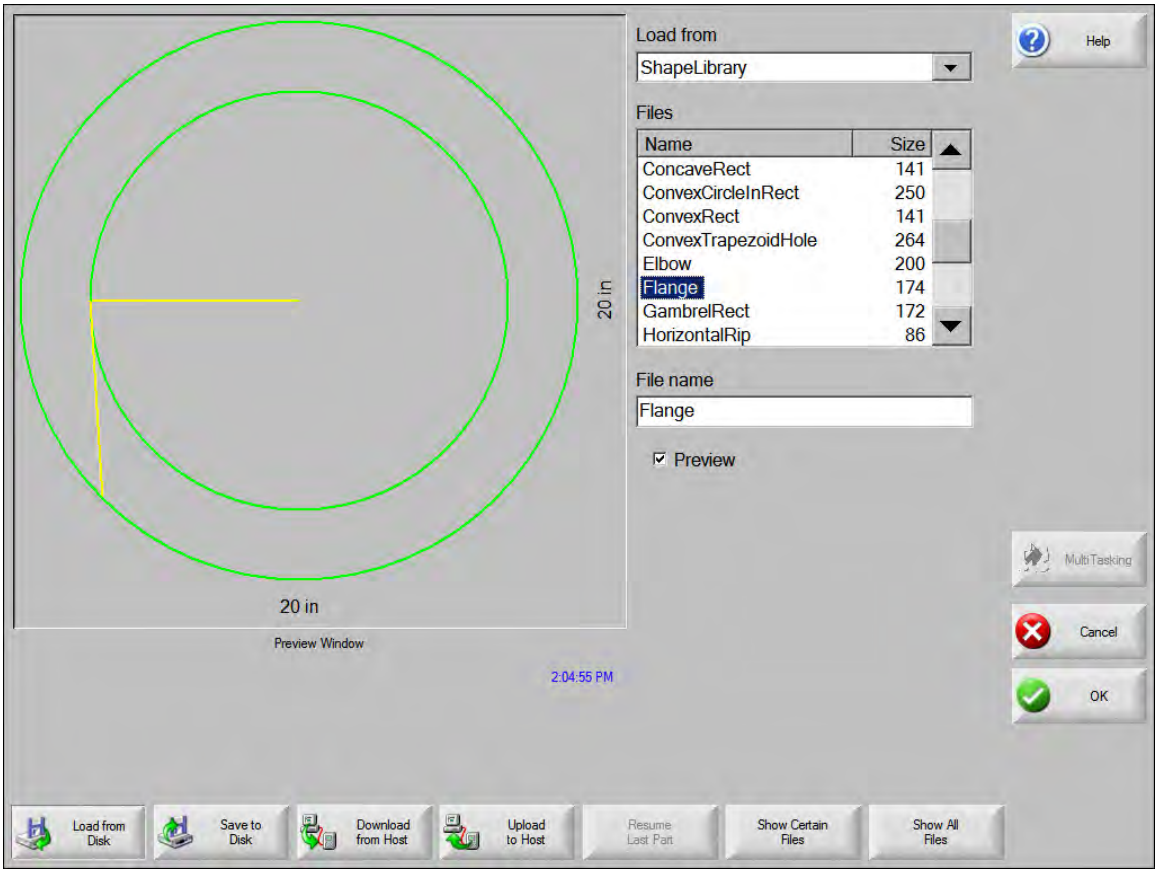
Using Manual Nester

To begin, first select the plate size requirements for the nest at the Cutting screen. This information is used to display the plate size and plate orientation at the main view screen to place parts on the plate. The plate information is retained with the nested part program when saved.

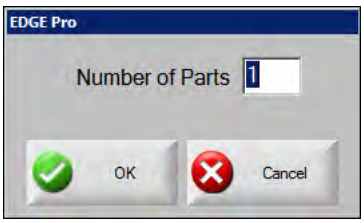
Press the Nester soft key from the Shape Manager screen then enter the Nester Setup Screen to configure the nesting software. Press OK to return to the Nester Main Screen to begin placing parts in the nest.

Adding parts

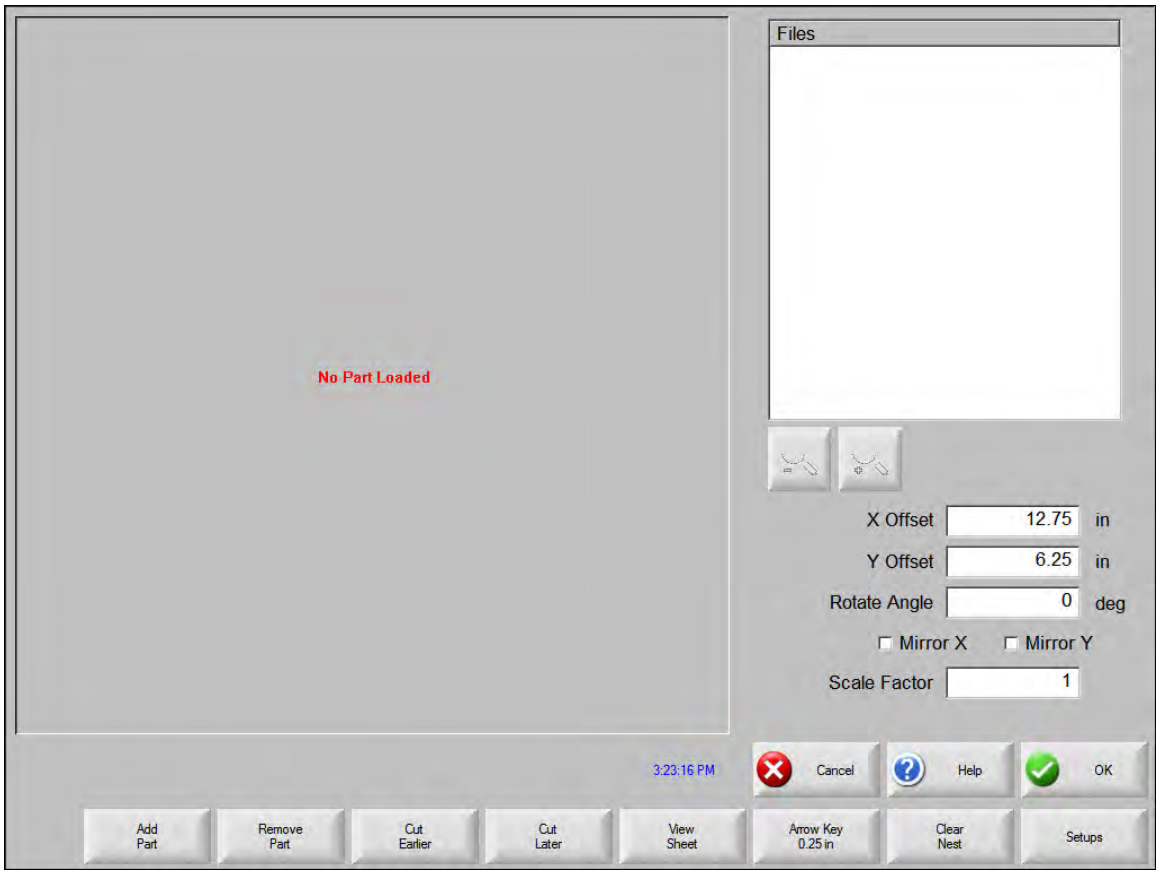
At the Nester Screen, press the Add Part soft key to add a new part to the nesting part list. The first screen viewed will allow you to select a part from the Simple Shape Library, from a disk or from the host computer via link communications.



After a part has been selected from either location, you will be prompted to select the number of pieces desired.



As new parts are added, they are displayed on the selected plate in preparation for final placement.



At this screen, the part can be oriented, scaled and moved to final position. This is done by moving to the Nester Part List and highlighting the file name. Then select the desired field to manually offset, rotate, mirror and scale the part.

To position the selected part, use the manual direction keys. The view screen will be outlined by a bold blue border indicating that the arrow keys are active. Pressing the arrow keys will allow movement of the part to the desired location on the plate. Each time the arrow keys are pressed, the selected part will move in the direction of the arrow by the increment set by the Arrow Key distance soft key. Use the arrow distance key, arrow keys and zooming in the view field, to place the part exactly as you need it.

Add more parts to the nest in the same fashion as described above. To customize your nest, you may decide to delete parts or add parts to the list and change the order in which they are cut by use of the soft keys displayed. When finished, press OK to return to the Main Cut screen and begin cutting the nest. The part nest is currently saved as a temporary file until another part is loaded.

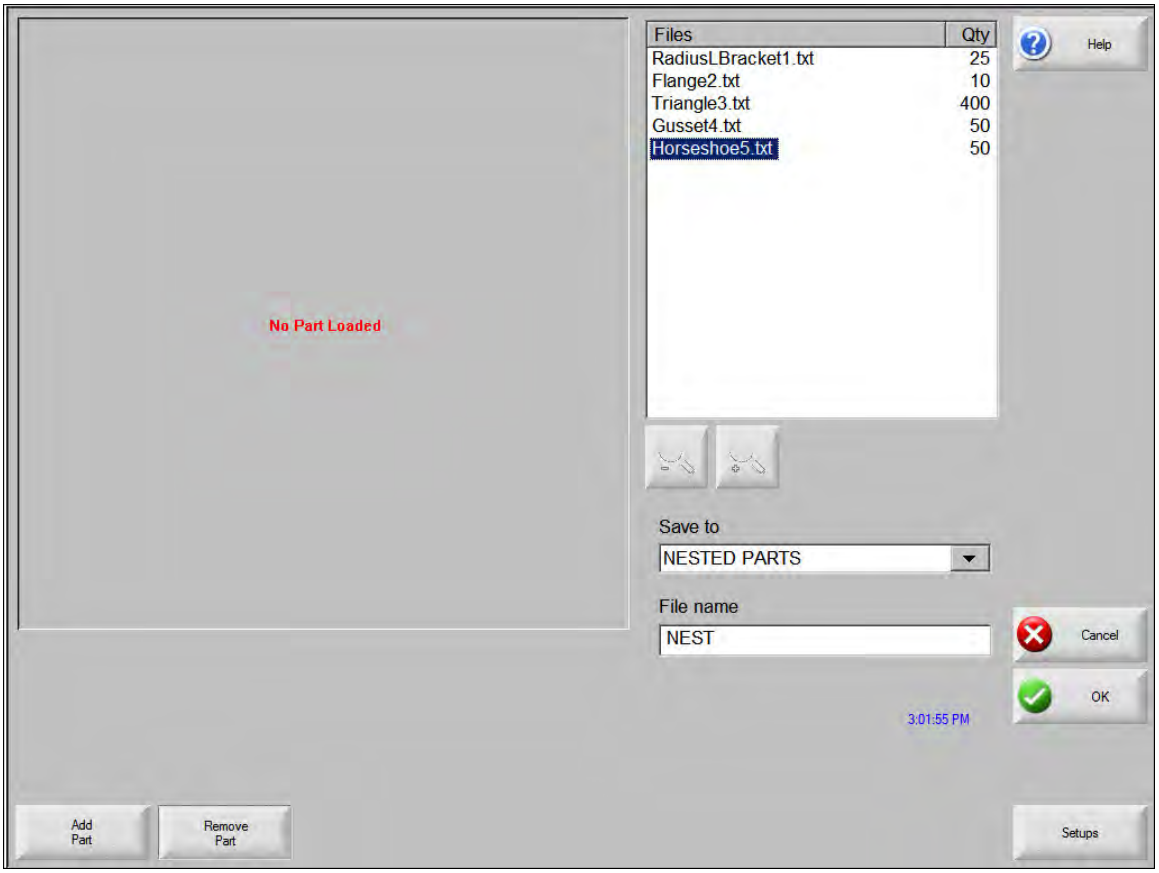
Saving a nest

From the Main screen, press the Files soft key then Save to Disk. From here the part can be saved to a hard disk folder on the CNC, Diskette or USB memory stick. The nested parts file may be saved as a nest or a part. Saving as a nest using the save as Nester File feature, will create a larger file which will allow future modification of the nest through Nester. Nested parts saved as a part file cannot be modified.

Hypernest® CNC automatic nesting

The main viewing area of the Nester screen is in the upper left corner and is used to preview manual nests. During an automatic nest, this area remains blank. The plate size used during Automatic nesting is based on plate information that has been selected at the main setup screen.

The upper right corner of the Main screen displays the list of part programs and quantities of parts that have been selected for nesting. In the lower right are fields for saving the nest with a name and to a folder.



This software feature is protected both by a software enable and a hardware key (dongle) installed on the CNC.

Setting up Hypernest on the CNC

Pressing the Setup soft key accesses the following setup parameters and can be used to configure the automatic nesting process for use.

If this feature is not available (grayed out), the feature has not been enabled on your CNC. Contact your CNC vendor for details on how to enable the Automatic Nesting feature.

Nesting Switching the Nesting Parameter to Automatic enables the feature.

Arrow Increment 1 – 5 At this screen, you can select different move increment dimensions. These dimensions are used as move distance references when the control arrow keys are pressed to place parts in position on the plate.

Autoposition Autoposition is an automated feature of the Nester software that enables block nesting. This type of nesting compares the overall block dimensions of the selected part and searches for the next available block on the plate that is large enough for the part.

Autoposition does not allow parts to be placed on top of other parts or inside other parts. However, it can be disabled if you want to add parts to scrap areas.

If Autoposition is not selected, imported parts are stacked in the lower left corner of the plate and must be manually arranged.

Search Increment The distance for the next available block on the plate that can be used for the next nested part.

Scrap Clearance The amount of space that is added to a block in the nest.

Part Spacing The Part Spacing feature sets the spacing between parts during the Automatic Nesting process.

Plate Edge Spacing This parameter allows you to set spacing around the edge of the plate to be used during the Autoposition Nesting process.

Program Origin The Program Origin (nest start location) may be set to lower left, upper left, lower right, upper right.

Cut Direction Cut Direction allows you to select the direction the parts will be placed in during the automatic nesting process. Options are Left or Right, Right to Left, Top to Bottom and Bottom to Top.

Nest Direction Select the direction in which nests are placed in the automatic nesting process.

Return to Nest Start When enabled, the Return to Nest Start feature will insert a traverse segment back to the start point at the end of the nest.

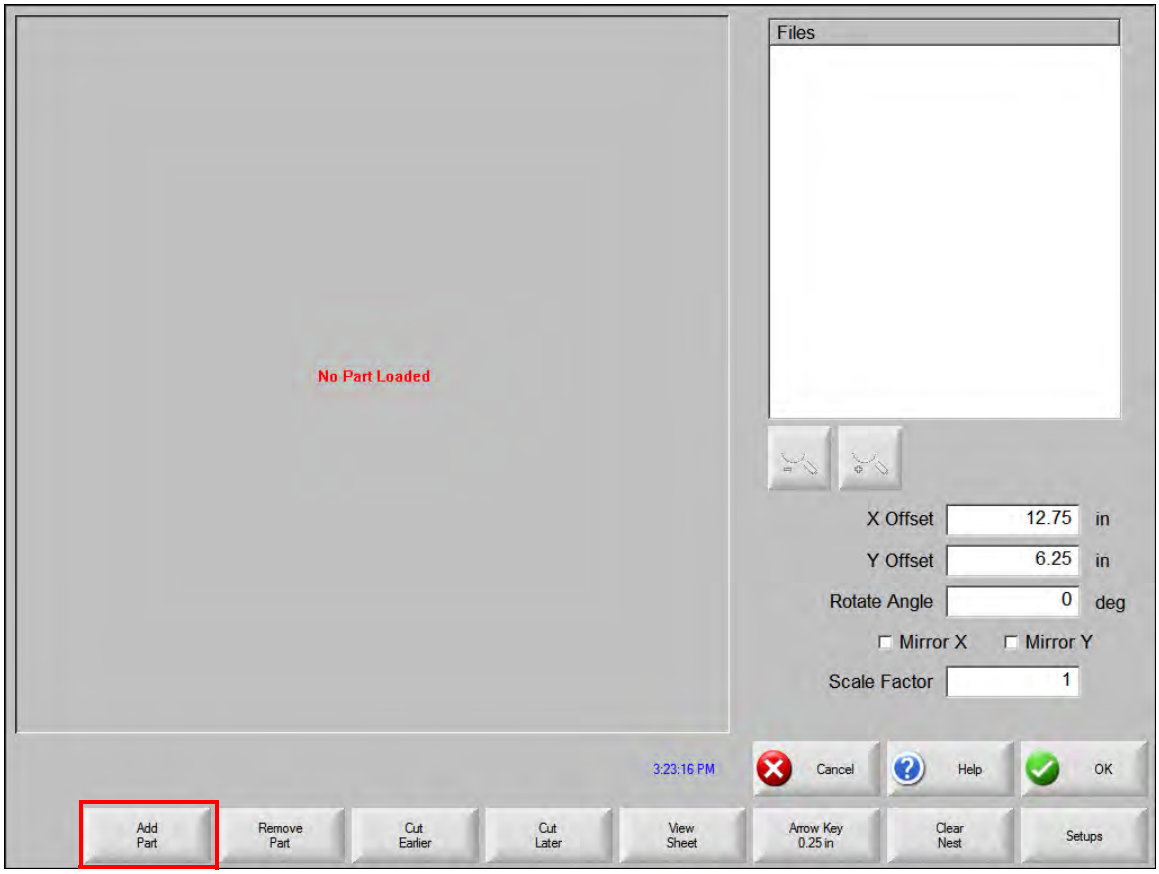
Use Remnant If remnants are created and saved for future use, select On to use one of these remnants for automatic nesting.

Generate and Cut Offcut Select On to generate offcuts for standard, rectangular nests. If this feature is enabled, offcuts are created when 30% or more of the sheet remains after nesting. The offcut is cut after a pause at the end of the nest on the last nested rectangular sheet.

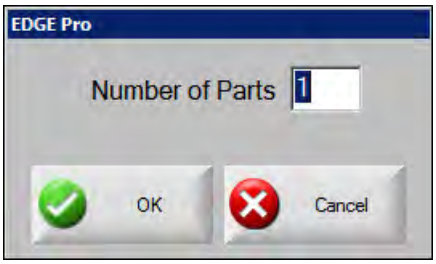
M65 Auto Reload Select On to allow new sheets to be reloaded automatically. When this feature is selected, there is a pause at the end of each sheet until the operator presses Start to Resume. Then, a new sheet automatically loads and runs. Auto reload works with standard, rectangular nests only.

Using Nesting

1. On the Main screen, choose Shape Manager > Nester.
2. On the Nester screen, choose the Add Part soft key to add a new part to the nesting part list.



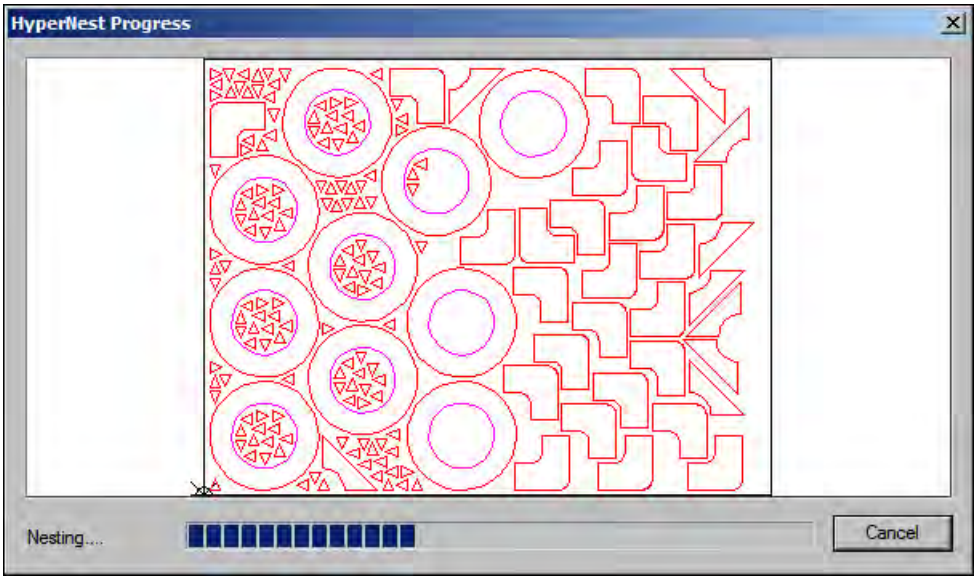
3. Select a part from the Shape Library, from a disk or from the host computer via link communications.
4. In the popup that displays, enter the number of pieces to be included in the nest.



5. As new parts are added, the part file name and quantity are listed in the Files window in preparation for final placement during the automatic nesting process.
6. Select a folder for the nest in the Save to dropdown list.
7. Enter the name of the nest in the File name field.

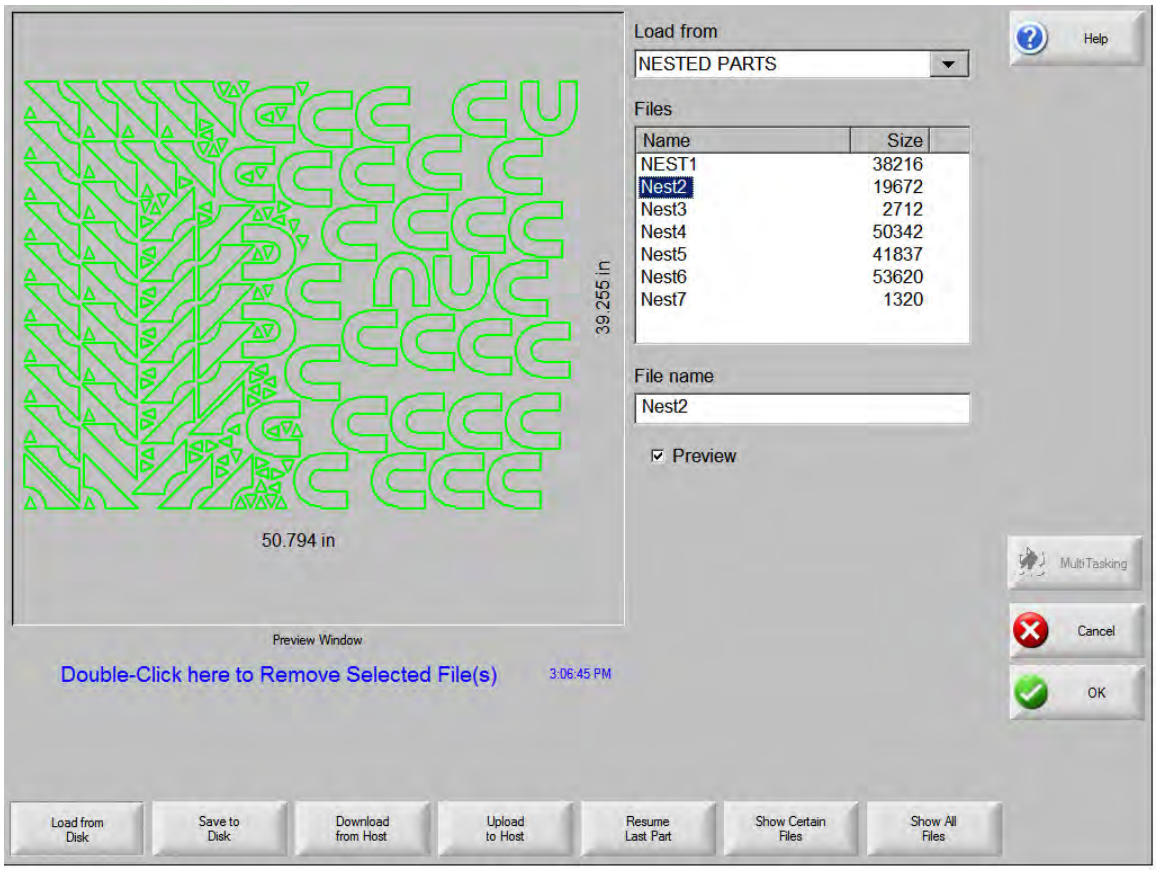
8. Press OK.

A progress window will be displayed during the nesting process.



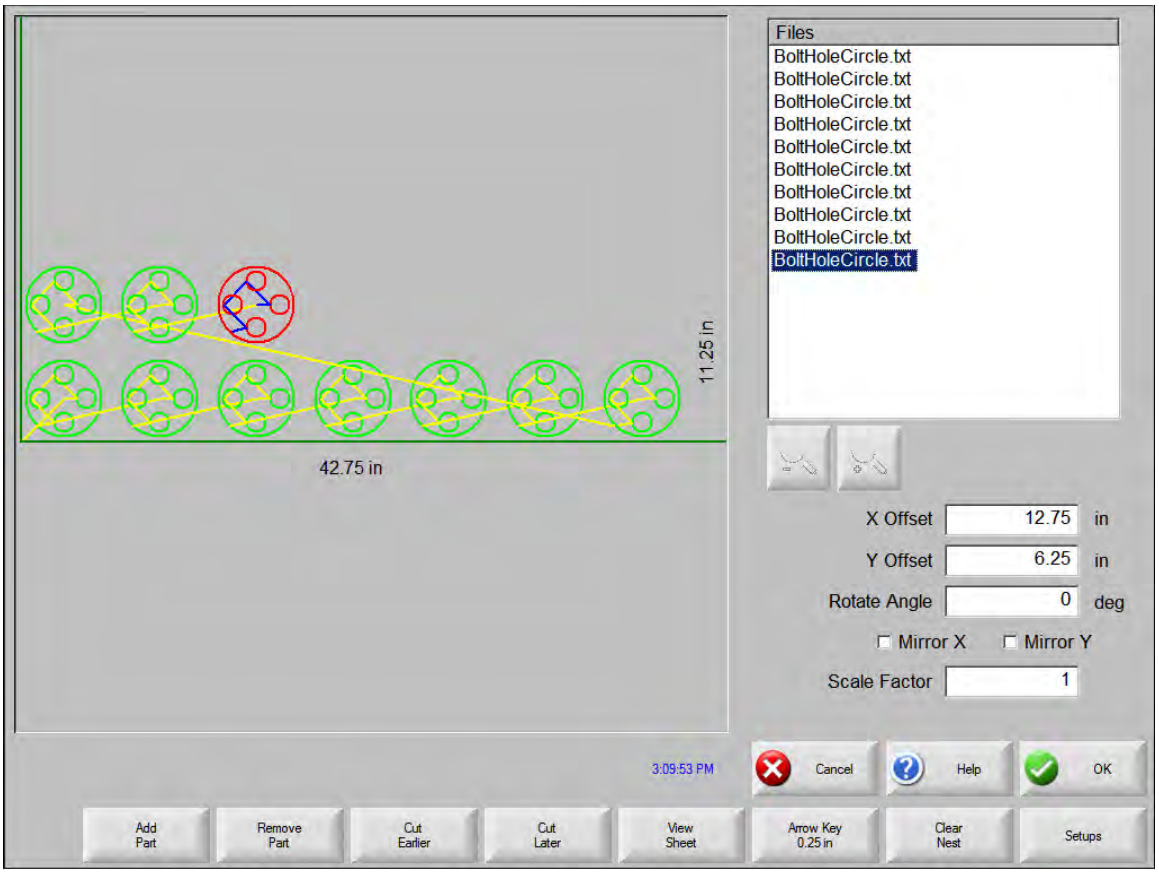
The nest process progresses quickly and not all shapes may be visible on screen or other drawing anomalies may be noted during the nesting process.

If more parts are selected than can fit on one plate, multiple plates or sheet (nested program) files will be generated and saved in the selected folder with the selected file name, but a numeric suffix will be added. For example, saving the part file as Nest may generate multiple part files named NEST1.txt, NEST2.txt, NEST3.txt, etc.

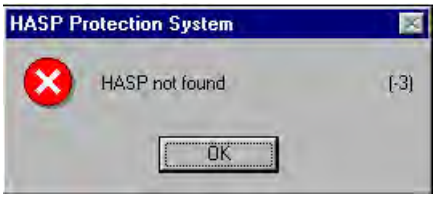


Removing a part from a nest

1. Highlight the selected part in the Files list.
2. Press the Remove Part soft key.

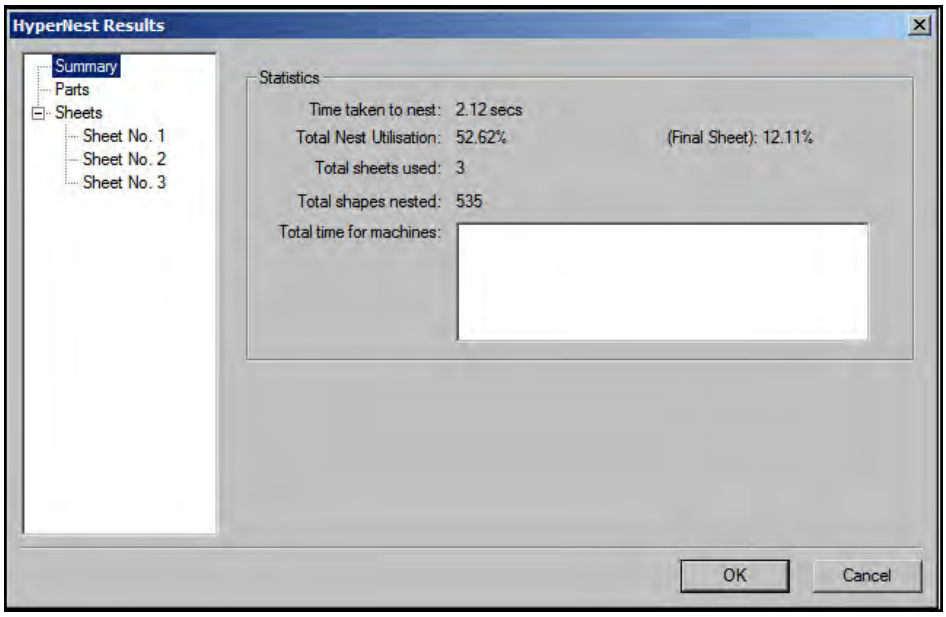


The software feature on the CNC is protected by a hardware key or dongle. If the hardware key has been removed from the CNC, the following message will appear when the Nest Parts soft key has been pressed.



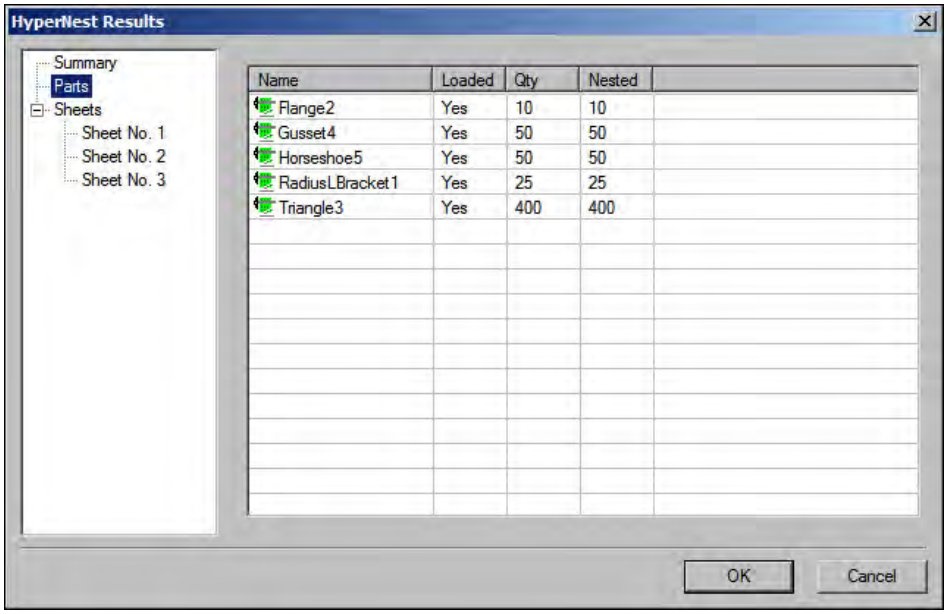
Nest summary

Upon completion of the nest, the software will provide a summary of the Automatic Nesting process.

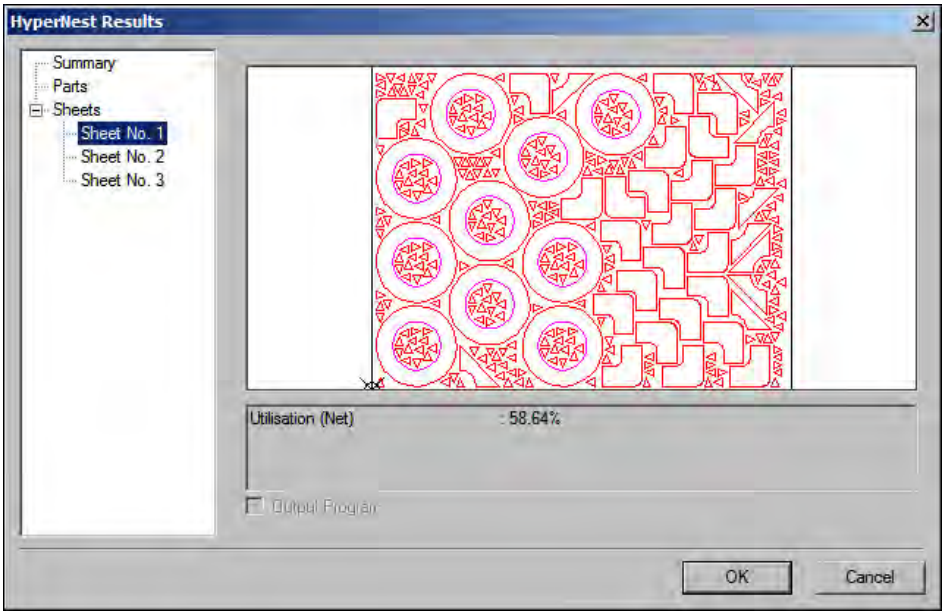


Statistical analysis of the process is provided for the number of sheets, time to execute nest, total nest utilization and total number of shapes nested.

Sheets that are generated with the exact same part configuration will be listed as “Sheet No. #”.

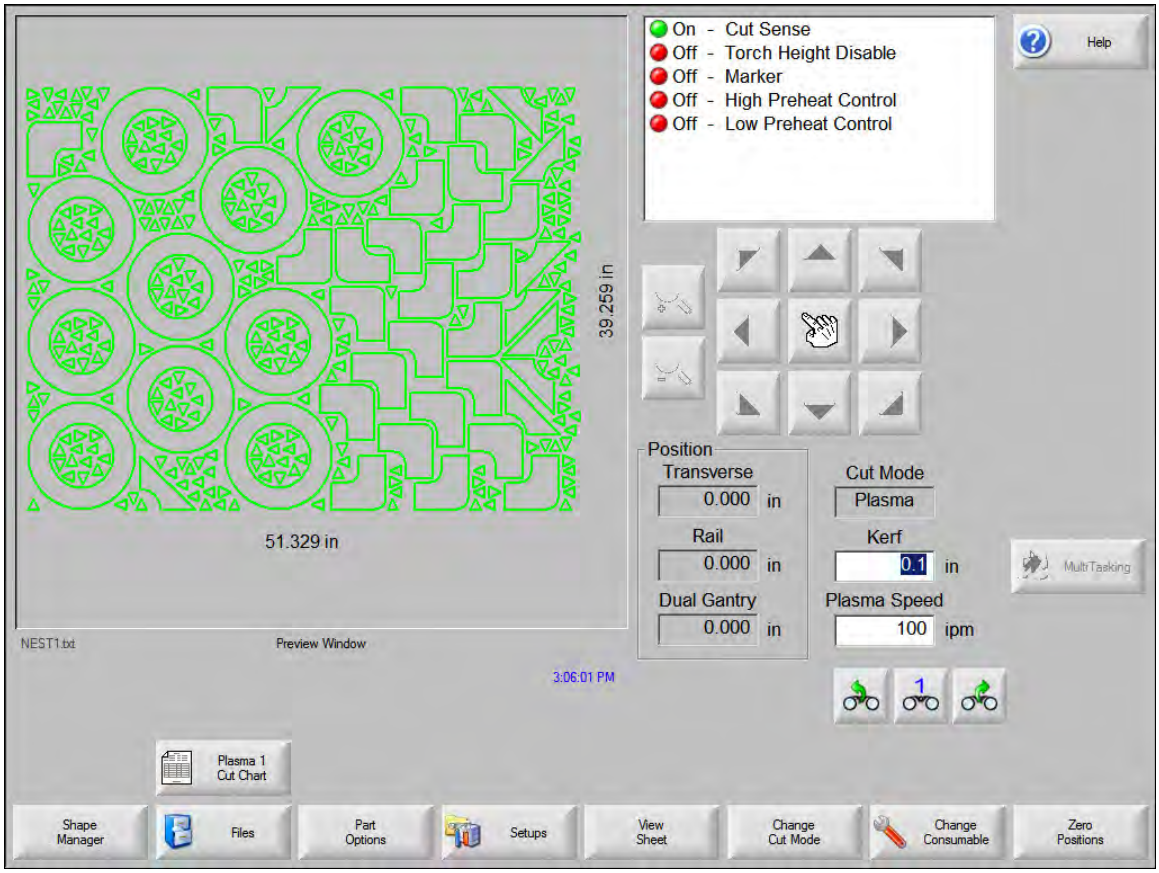


Scroll down to view an analysis of the parts used and to view the individual sheets produced and a listing of the net utilization for the specific sheet.



Press OK to accept the nest and have the first sheet become the current part. Press the Cancel key to reject the nest and return to the main nesting screen to add or remove parts from the nest.

Main screen view of nest



Parts with open loops or other invalid geometries may not be able to be automatically nested. It may be possible to manually nest parts which have been rejected by the Automatic Nesting function.

Section 5

Cutting Parts

CutPro™ Wizard

The CutPro Wizard automates the sequence of choices and selections required for cutting parts. If you have parts, nests, and cutting processes stored on your system, you can use the CutPro Wizard to simplify cutting operations.

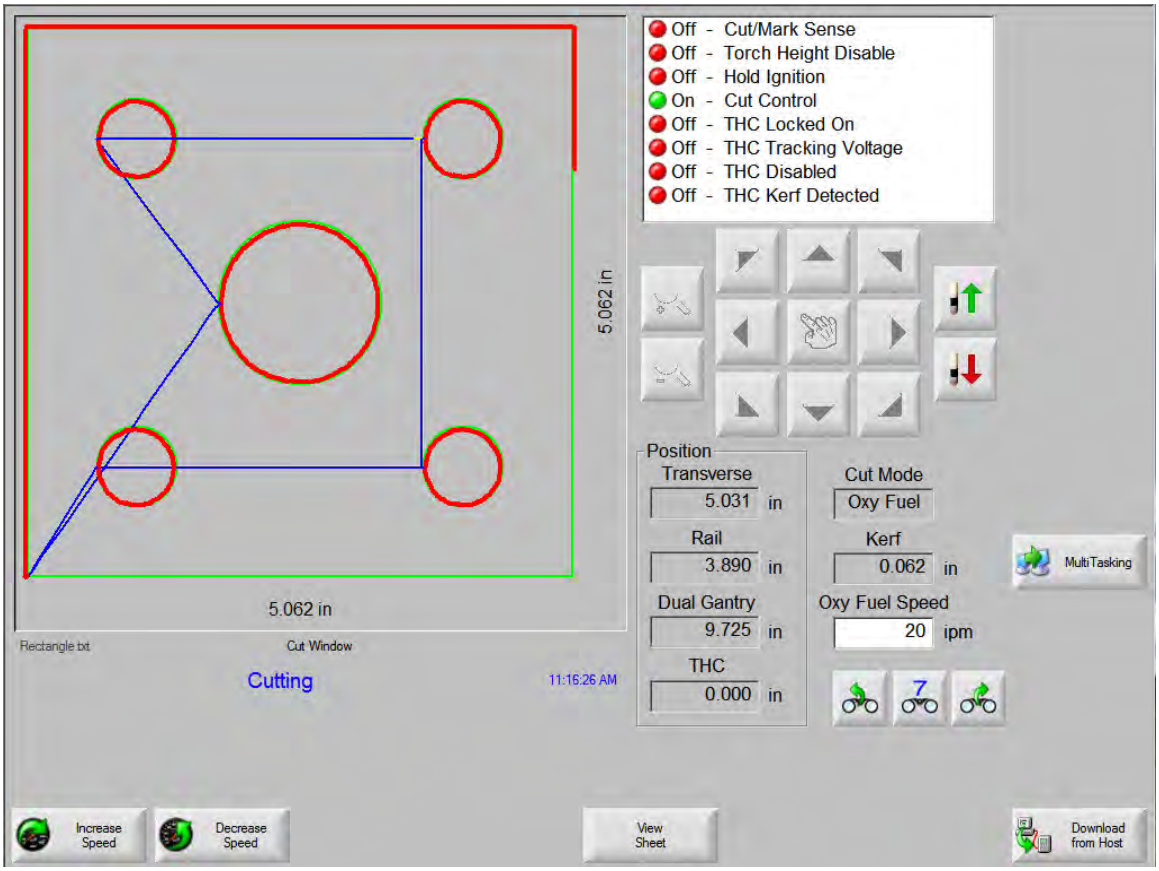
The CutPro Wizard also helps you to align parts and handle plate skew with the Align Wizard. For more information on the Align Wizard, see *Arranging Parts*.

The CutPro Wizard opens automatically from the Main screen or you can press the CutPro Wizard soft key on the Main screen. As you work with the CutPro Wizard, it keeps track of your progress and displays it at the bottom of the wizard window in the progress bar.

The CutPro Wizard is unavailable if you are operating the CNC in keyboard-only mode.

Cutting in manual mode

After you verify that the cut mode, cut speed and kerf values are set correctly, press the Start key on the Main or Manual window to cut a part. The following window is displayed.



To cut the part:

1. Verify that the Cut Mode is set to the selected cut type and that the Kerf and Cut Speed settings are correct.
2. Press the Start key on the front panel (or F9 on a keyboard). This starts cutting in the selected Cut Mode.

To preview the path:

1. Press the Change Cut Mode soft key until Trial appears in the Cut Mode window.
2. Press the Start key to have the cutting device follow the cut path without cutting. Motion is performed at the programmed speed.
3. Press the Stop key on the front panel to stop a cut. The machine decelerates to a smooth stop along the cut path. If the cutting process was on when you pressed the Stop key, it is deactivated according to the programmed cut logic.

During a cut, the Watch window displays information about the part being cut, such as the current cut speed, current axes positions, and path position.

In Sheet view, the control automatically scrolls to keep the cut location centered within the view screen. This feature is useful during normal cutting to zoom in and follow the cut path.

View Sheet is useful when proper plate size values have been entered in Cutting setups and when the machine has been homed. If you attempt to view large parts with a fully zoomed screen, the part may not be fully drawn on the screen before the next view location displays and the screen may flash. Zoom out to correct this and view a larger area.

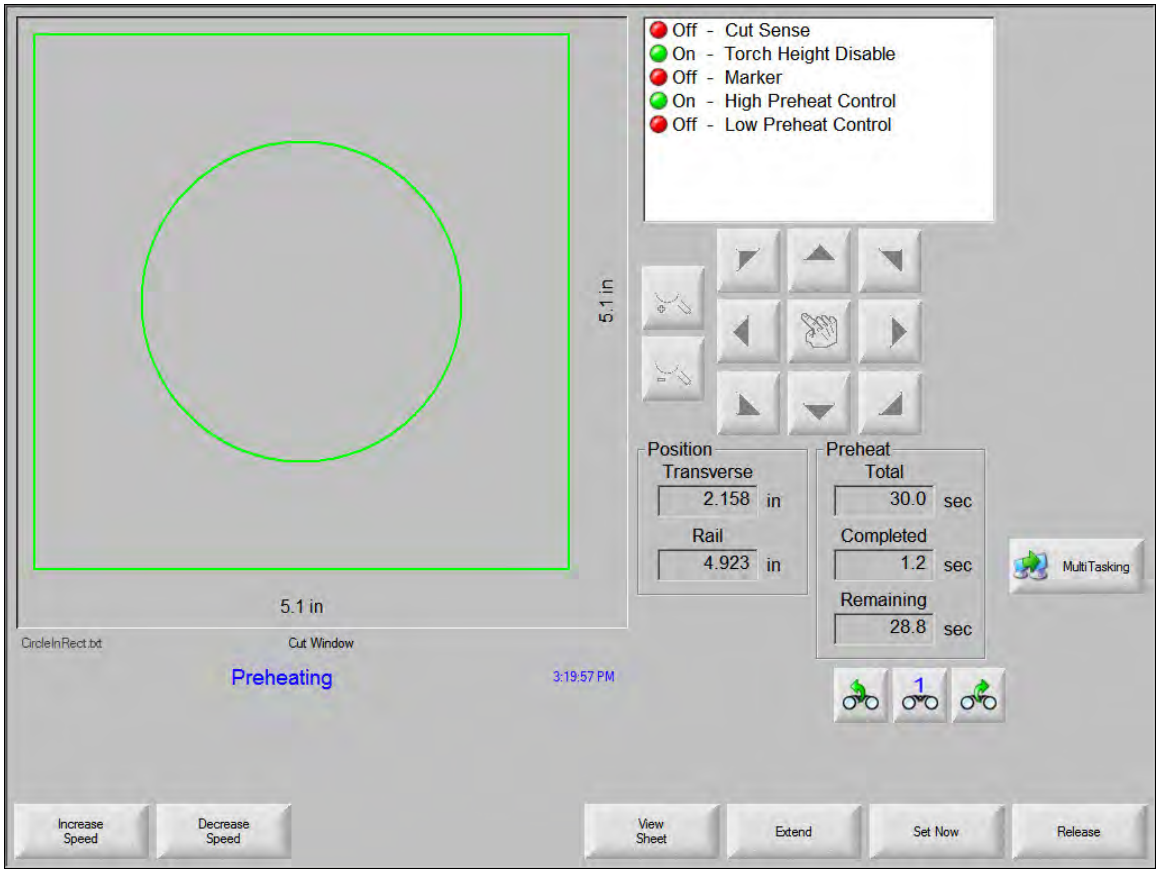
Increase Speed Increases the current cut speed by 3%.

Decrease Speed Decreases the current cut speed by 3%. Double-click the speed field to enter a new speed value.

Keyboard operation: To change the current cut speed while cutting a part, press the Enter key once to highlight the current cut speed, enter the new cut speed, and press Enter again.

Repeats If a Shape Repeat option has been enabled, press the Repeats soft key to see the number of rows and columns remaining to be cut. The Repeats soft key operates with the Extend soft key which is only active during initiation of the cut sequence.

Cut Delay Timers Cut Delay Timers define the timing logic for the cut and are available for both oxy fuel and plasma at the Cut Types setup screen. In cut mode, the control displays the preset delays as they are executed in the lower right corner of the screen. For certain delay times, such as Preheat and Pierce, a countdown timer shows the preset time and time remaining. An example of the Preheat Timer is illustrated below. The Preheat times, Total, Completed and Remaining, are shown to a tenth of a second.



When you activate the Cut Sense input, the Preheat delay time cycle ends. The time at the point of activation becomes the new Preheat time for subsequent cuts. Use the soft keys to change the Preheat cycle in progress:

Extend Extends the Preheat timer until it is stopped with either the Set-Now or Release soft key.

Set Now Ends the selected delay timer and saves the new set time. Use the Set Now soft key with the Extend soft key to modify the preset Preheat time.

Release Ends the selected delay timer, but does not modify the original delay time.

Press the Start key twice to bypass the Preheat and Pierce Time Delays and begin the cut in the Oxyfuel cut mode.

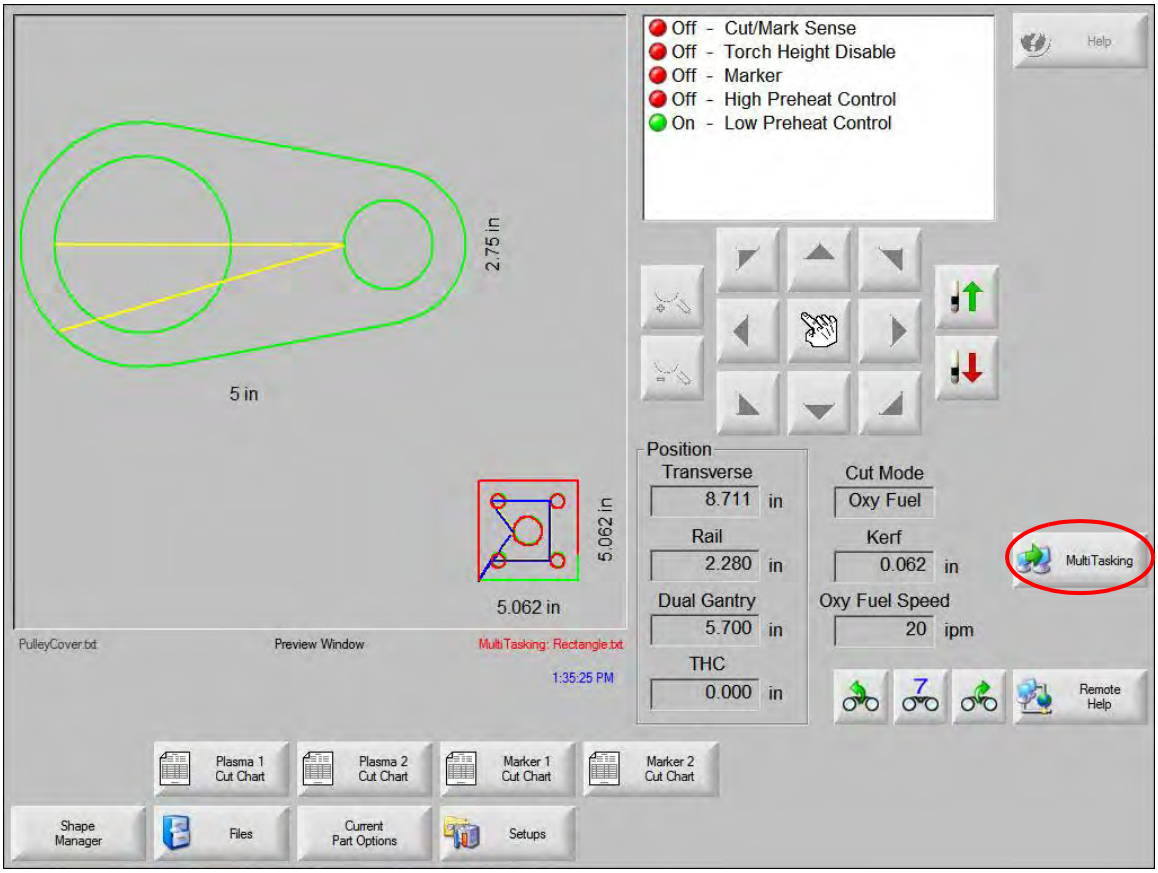
Multitasking

Multitasking allows you to load and configure a new part program while another part program is cutting. This feature is only available in advanced operation mode.

To use the multitasking feature:

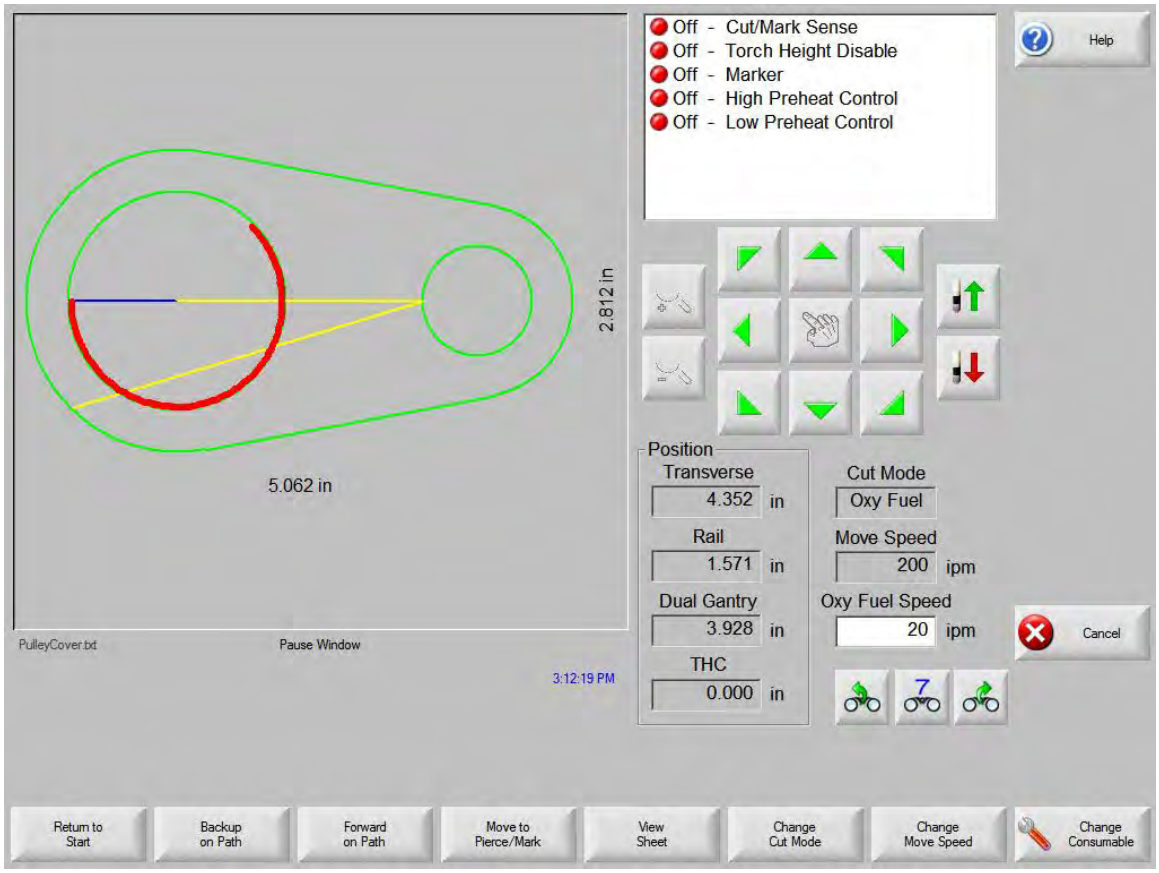
1. On the Shape Manager screen, press the Multitasking soft key. The current part program displays in the lower right corner of the Preview Window.

2. Select another part program from the Shape Library or a storage device. The new program displays in the Preview Window.



3. Press the multitasking soft key to switch between programs.

Pausing cutting



If the cut process fails, the CNC has the following recovery capabilities:

Cut Loss Recovery The Cut Loss Recovery features of the CNC are available from the Pause screen which displays when the operator presses Stop or if cut sense is lost. Press the Cancel key on the Pause screen to cancel the current part.

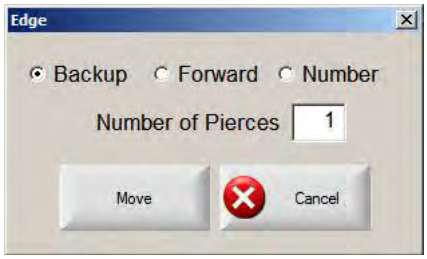
Return to Start This feature allows the operator to return to the initial starting point of the part program. If you use the Return to Start function after a cut loss, all information about the current position of the cutting device on the path is lost.

Backup and Forward on Path Use these two soft keys to move backward and forward along the cut path at the selected move speed to locate the pierce restart point. Press the Start key to resume the cut at the programmed cut speed. In addition to all segments of a standard part, the Backup and Forward on Path functions allow full movement through all sections of Shape Repeat part as well.

Like the Manual Mode functions, Backup and Forward on Path use the currently selected move speed. The different speeds allow moving rapidly along the path, or precisely positioning the cutting device.

When a cut loss occurs, the initial backup and forward speed is the one that was used last. To toggle between the move speeds, press the Change Move Speed soft key in the Pause window. The corresponding speed is displayed in the Move Speed window.

Move to Pierce/Mark Press the Move to Pierce/Mark soft key to move directly to any pierce point.



Enter the information for the pierce point and press Enter. The cutting device moves directly to the selected pierce point.

Change Cut Mode Alternates the restart mode between Cut and Trial. This allows the operator to move through the part partially as a trial cut and partially as an actual cut.

Change Move Speed Cycles through the four move speeds that are available: Maximum Machine Speed, High Jog Speed, Medium Jog Speed and Low Jog Speed from the Speeds setups.

On-Path Restart To restart the cut at the pierce point that you selected using On-Path Backup, press the START key. The cut speed and the cut mode are the same as they were before the motion was paused unless the values have been edited in the Watch window.

While the Pause Window displays, the manual arrow keys are fully functional so you can move the cutting device. This allows you to move the machine in any direction (not necessarily along the path) to inspect the partially cut piece. Once the cutting device is moved off the cut path, the Off-path Pause Window displays.

Return to Path Press the Return to Path soft key in the Off-path Pause window to return the cutting device to the point on the cut path from which it was jogged away. This feature is useful for inspecting or replacing components after cut loss and then returning to the cut loss point. When the cutting device is back on the cut path, the On-path Pause Window is restored and cutting can resume.

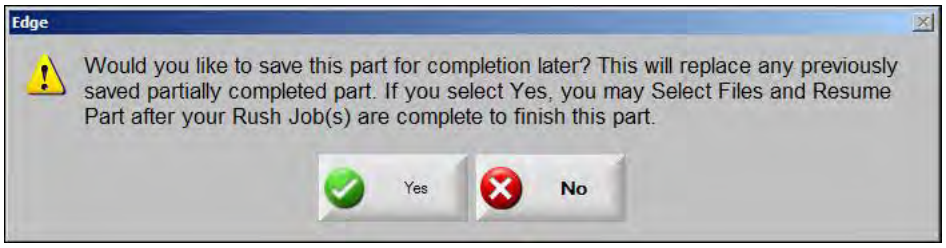
Move Part Moves the entire part on the plate. The point along the cut path to which the cutting device moves becomes the current position of the cutting device. The On-path Pause window displays again because the cutting device is on path.

Off-Path Restart Press the Start key from the Off-path Pause window, to construct a lead-in from the off-path point back to the original part.

In a cut loss situation, the operator can use the Backup on Path key in the On-path Pause window menu to position the cutting device on the cut path where the cut was lost. The operator can then use the manual arrow keys to jog the cutting device off path to a suitable pierce point.

Press Start at this point to cut a new lead-in from the off-path pierce point to the point along the path from which the cutting device was jogged away. When the cutting device is back on path, it continues along the path to cut the remainder of the part.

Rush Job Interrupt Allows you to pause the current part program and retain the part and current position information. On the Pause screen, press the Cancel key. A prompt displays on the screen to allow you to save the part information.

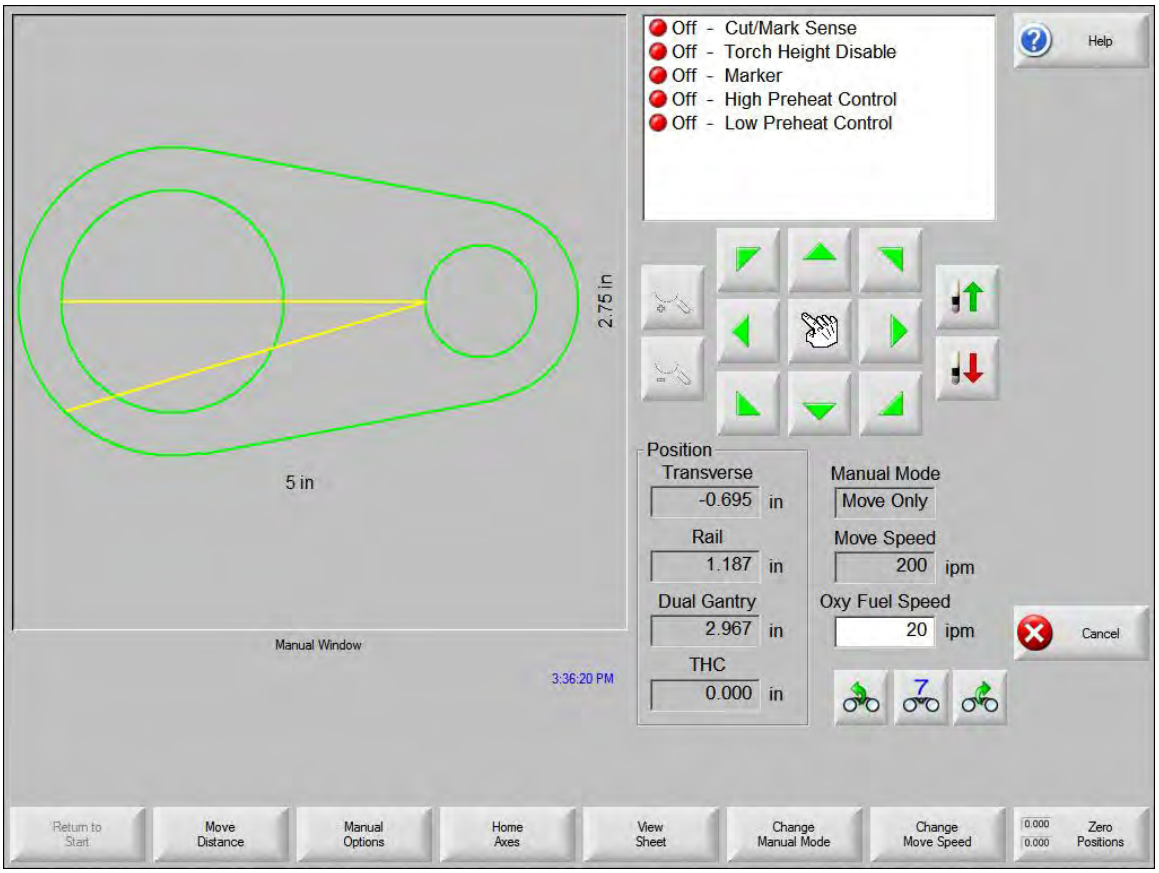


If you select Yes, the Resume Last Part button displays on the Files Screen. You can load and execute another part program and then return to the original part using the Resume Last Part soft key. The part program and position resumes.

Manual operations

The manual key is identified by the hand icon. If the manual key is not displayed, press [Shift + F11 or [+ F11 to show the manual operations screen.

Press the manual key on the screen to view the following screen. The directional jog keys are active when they are green.



Whenever the manual keys are active, the cursor icon in the graphics display window is the shape of a hand.

From the Manual window, the machine can be moved in one of eight directions using the arrow keys. The cutting device moves while you hold an arrow key down. When the key is released, the cutting device comes to a smooth stop.

If the latched manual key feature is enabled in the control setups, press the manual key a second time to allow motion to continue without holding down the arrow key.

This feature is available for the manual direction keys in the Manual, Align and Pause screens. When this feature is activated, the dialog “Latched Manual Keys On” will be displayed in red at the bottom right corner of the part window.

Motion can be paused by use of the Stop, Cancel or an arrow key. The latched manual key feature can be turned off by pressing the manual key again.

Return to Start Whenever the Manual window is opened, the transverse and rail positions at that point are saved.

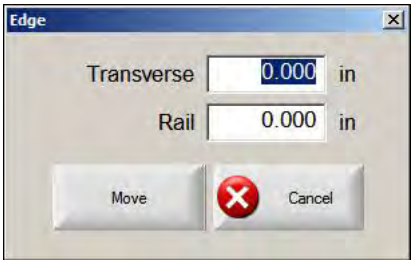
After rip cutting or other manual operations, it may be necessary to return to this “start” position.

Press the Return to Start soft key to generate motion in the transverse and rail axes from the machine’s current position to the position that was saved when the Manual window opened.

Move Distance When Move Only is displayed in the Manual Mode window, the second soft key from the left changes to Move Distance.

The Move Distance soft key allows you to perform moves over exact distances. After you press Move Distance, the CNC prompts you for traverse and rail distance values for the machine’s motion. Enter the appropriate values and press ENTER.

The cutting device moves the distance you entered in a straight line without executing any cut logic.

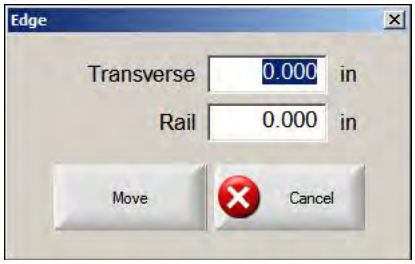


As with any automatic motion, you may press STOP on the front panel at any time to bring the machine smoothly to a stop before the programmed motion is complete.

Cut Distance When Rip Cut mode is selected in the Manual Mode window, the second soft key from the left changes to Cut Distance.

This soft key allows you to make rip cuts of an exact length. After you press Cut Distance, the control prompts you for traverse and rail distance values for machine motion. Enter the appropriate values and press ENTER.

After the cutting device executes the cut logic sequence, it moves the distance you entered in a straight line.



If you enter incorrect values, press the CANCEL key at any time.

After motion has begun, press STOP on the front panel to bring the machine smoothly to a stop before the programmed motion is complete.

Rip Cut mode is useful for making a cut along a specified linear path. Motion stops and cutting action ceases when the new position is reached or when the STOP key is pressed.

If you do not know the exact distance, enter a distance longer than needed in the right direction, and then press STOP to end the cut.

Manual Options Press the Manual Options soft key to access the Manual Options screen.

Home Axes Press the Home Axes soft key to access the Homes Axes screen.

View Sheet/View Part View Sheet allows you to view a part as it would appear on the plate. After you press the View Sheet soft key, the display window zooms out to show the part in relationship to the entire plate.

After the display zooms out, you can zoom in again by pressing the + key, which adds horizontal and vertical scroll bars. Press the - key to zoom out again.

While the scroll bars are displayed, you can hold down the scroll bar and move it to adjust the view of the machine horizontally and vertically. This mode is useful in normal cutting to closely follow the cut path while in zoom.

While cutting in sheet view, the control automatically scrolls to keep the cut location centered within the view screen. This feature is useful in normal cutting to follow the cut-path while zoomed in.

View Sheet is more useful when proper plate size values have been entered in cutting setups and when the machine has already been homed. If you are viewing large parts being cut with the display fully zoomed, the system may not be able to draw the part on screen before it has to move to the next view location. In this case, the view screen may flash but you can correct this by zooming out to a larger view area.

Change Manual Mode This soft key alternates the control Manual Mode between Move Only and Rip Cut.

If you press this soft key the second soft key from the left to changes function from Move Distance to Cut Distance. Rip Cutting is described in more detail below.

Change Move Speed This soft key toggles through the four Move Speeds: maximum machine speed, high jog speed, medium jog speed and low jog speed from the Speeds setups.

Zero Positions Press this soft key to return all axes positions to 0 (zero).

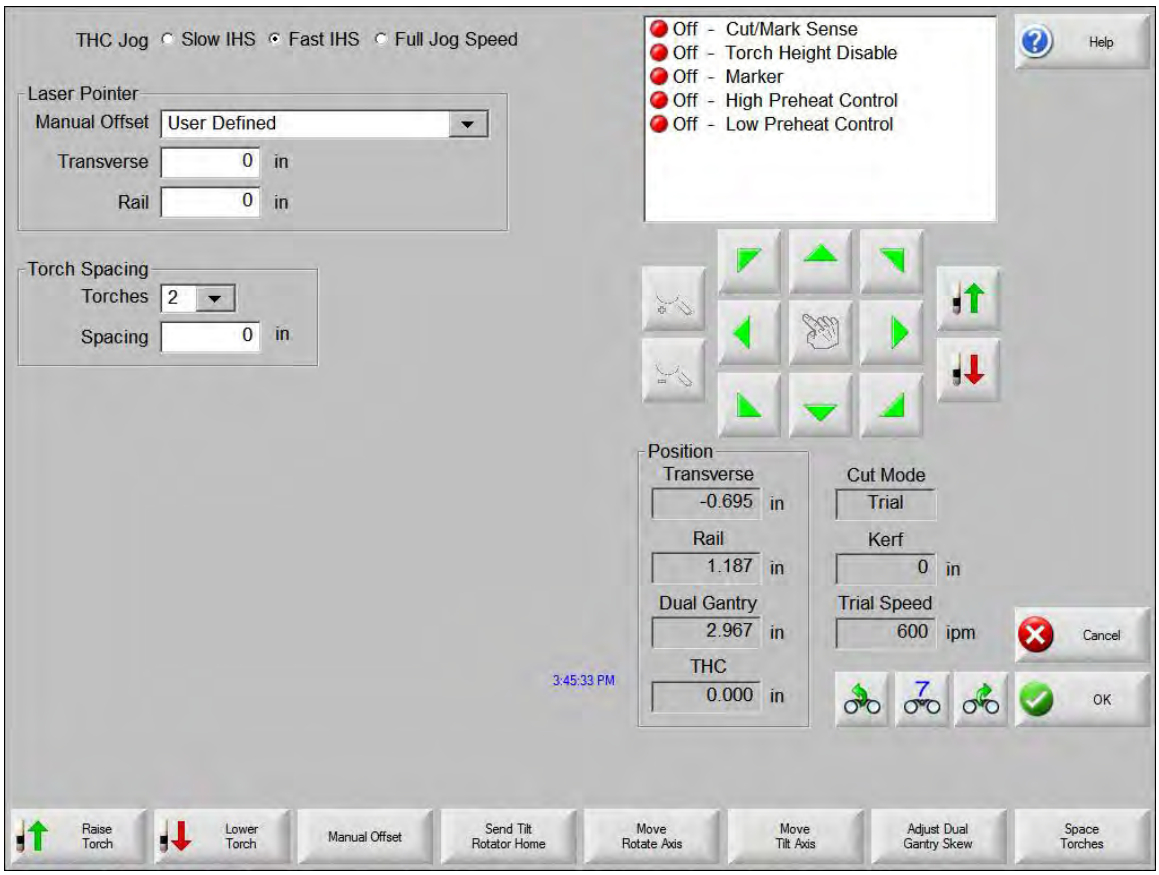
Rip cutting

When the Manual Mode window displays Rip Cut, you can use the arrow keys to begin a cut sequence and machine motion in the chosen direction.

To initiate a rip cut:

1. Verify that the correct cut mode has been selected.
2. Verify that the proper cut speed is displayed in the Cut Speed window (editable in Rip Cut mode).
3. Press the arrow key corresponding to the desired start direction for the cut.
   The cutting sequence proceeds even after the key is released; however, machine motion is generated only as long as an arrow key is held down, unless the latched manual key feature has been enabled.
4. Use the arrow keys to change direction.
5. Press Stop, Cancel, or Manual to stop the operation of the cutting device.

Manual options



Raise Torch Raises the cutting torch while the soft key is pressed or until the Torch Up sense input is activated. If a Sensor THC is installed, the CNC uses the THC Jog speed selections.

Lower Torch Lowers the cutting torch while the soft key is pressed or until the torch down sense input is activated. If the torch down output has been enabled to stay on during the cut in plasma setups, press the Lower Torch soft key to keep the torch in the down position until the Lower Torch soft key is depressed a second time. If a Sensor THC is installed, the CNC uses the THC Jog speed selections.

Manual Offset Manual Offset is useful for cutting tables that have a laser alignment tool attached and allows you to use your laser alignment tool for aligning a part to a plate.

The offset will stay in place until it is turned off from this screen or a part is cut in either oxyfuel or plasma cut mode.

You can select the following manual offsets:

- User Defined: Uses the selected X / Y Offset distance.
- Laser pointer to Plasma 1: Offset 10
- Laser pointer to Plasma 2: Offset 11
- Laser pointer to Oxy Fuel: Offset 12

This setting is displayed in the Laser Pointer group box on this screen.

Send Tilt Rotator Home Executes a move to the predefined rotate home position.

Move Rotate Axis Moves to a specified rotate axis position. Enter the position in the dialog box that displays after you press this key.

Move Tilt Axis Moves to a specified tilt axis position. Enter the position in the dialog box that displays after you press this key.

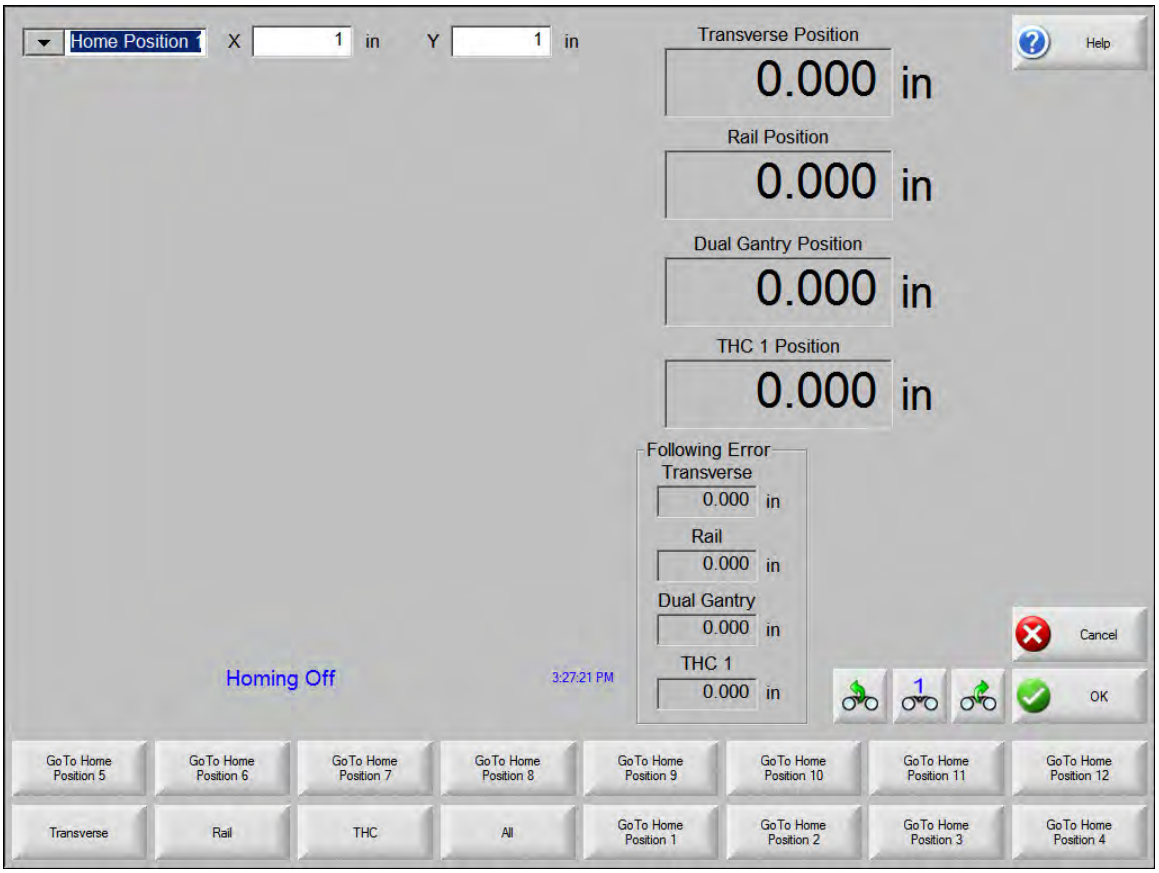
Adjust Dual Gantry Skew Allows you to move the master rail motor using the directional manual jog arrow keys to correct or realign gantry skew. This movement is only allowed after you enter a password.

Consult the table manufacturer's documentation to prevent damage to machinery.

Space Torches Performs the torch spacing routine. (This feature requires special program codes. See the Phoenix Software V9 Series Programmer's Reference for more information). The number of torches to space and the distance can be entered in this screen. Pressing Space Torches generates a .txt file and moves multiple torches to evenly spaced positions along the transverse axis.

Home axes

From the Home screen, each axis or all axes can be "homed". In addition, the transverse and rail axes can be sent to one of up to 12 programmed alternate home positions.



The home feature sets 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 the homing command is entered at the CNC, the CNC moves the axis toward the home switches at the fast home speed until the switches have been activated. After the switches have been activated, motion stops and the axis moves in the opposite direction off switch at the slow home speed.

When switch is deactivated, the position is recorded at the CNC and provides an absolute reference point for future motion commands.

Transverse Press the Transverse soft key to initiate the automated homing procedure. This procedure generally produces machine motion in the transverse axis, depending on the homing parameters set in the Setups.

Rail Press the Rail soft key to initiate the automated homing procedure. This procedure generally produces machine motion in the Rail axis, depending on the homing parameters set in the Setups.

CBH Press the CBH soft key to initiate the automated homing procedure. This procedure generally produces machine motion in the CBH axis, depending on the homing parameters set in the Setups.

THC Press the THC soft key to initiate the automated homing procedure for the Sensor THC. This procedure generally produces machine motion in the THC axis, depending on the homing parameters set in the Setups.

Tilt Press the Tilt soft key to initiate the automated homing procedure for the Tilt Axis.

Rotate Press the Rotate soft key to initiate the automated homing procedure for the Rotate Axis.

If tilt and rotate axes are enabled, the following window will appear and allow access to the Tilt/ Rotate or other axes.

Select Yes to access Homing for the Tilt and Rotate Axes.

Select No to access the Homing functions for the other axes.

All Press the All soft key to initiate the automated homing procedure. This procedure generally produces machine motion in one or more axes, depending on the homing parameters set in the Setups.

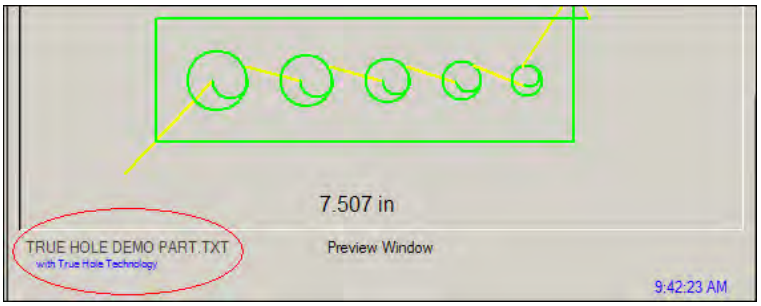
Go To Home Position Press one of the four Go To Home Position soft keys to move the transverse and rail axes to the predefined position set in the corresponding edit window. The Go To Positions are absolute and require that an automated home procedure already be executed.

True Hole verification

Hypertherm's True Hole technology is a specific combination of parameters that is linked to an amperage, material type, material thickness and hole size. True Hole technology requires a HyPerformance Plasma HPRXD auto gas system, True Hole enabled cutting table, nesting software, CNC, and torch height control.

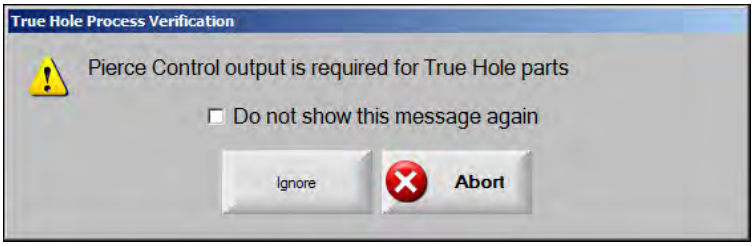
When the CNC prepares to cut a part with True Hole technology, it will check the settings that are specific to True Hole technology. The CNC then gives you the option to auto-correct these settings if it finds they are incorrect for cutting True Hole parts.

You can identify a True Hole part when you open the part file on the CNC.



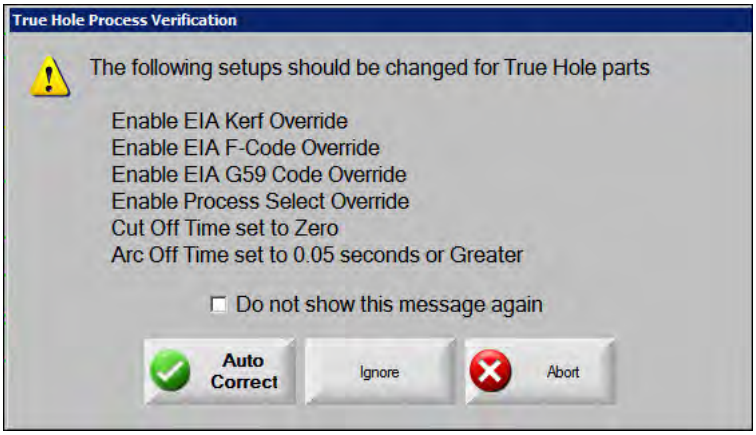
If the text "with True Hole Technology" does not appear below a True Hole part, the settings within the software may be incorrect.

1. Press Start on the front panel of the CNC to launch True Hole verification. First the CNC checks for the proper pierce control setting for the THC.



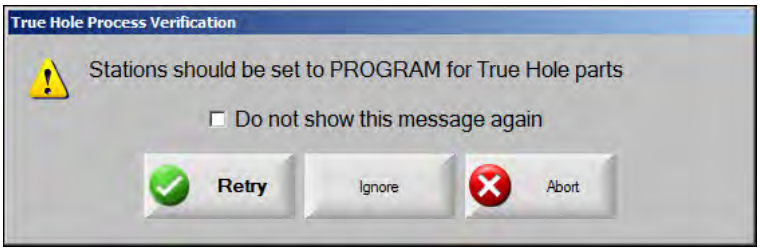
   Pierce control is assigned in the Setups > Password > Machine Setups > I/O screen.

2. The CNC checks for the program code settings and process settings. Choose Auto Correct to have the CNC correct these settings for you.



You can find the program code setups in the Setups, Cutting screen, and the process settings in the Setups, Process screen.

3. The CNC checks the position of the station switches and program speed control on the control operator console. At least one station switch must be in the Program position, and the speed must be set to 100%.



After you adjust the switches or speed control, press Retry to run the part program.

Plasma cutting tips

The following reference guide offers several solutions to help improve cut quality.

Consider the following factors when evaluating plasma cut quality:

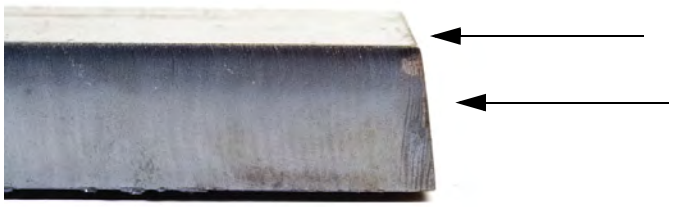
- Type of machine (example: XY table, punch press)
- Plasma cutting system (example: power supply, torch, consumables)
- Motion control device (example: CNC, torch height control)
- Process variables (example: cutting speed, gas pressures, flow rates)
- External variables (example: material variability, gas purity, operator experience)

All of these factors can affect the appearance of a cut.

Cut quality issues

Angularity

Positive cut angle: More material is removed from the top of the cut surface than from the bottom.

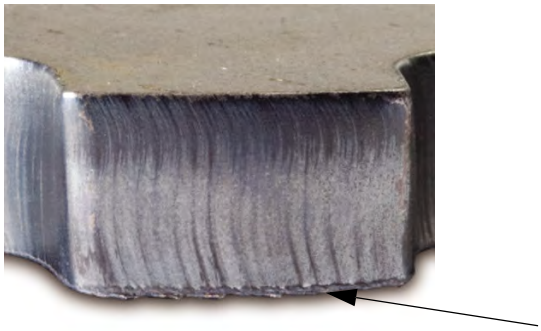


Negative cut angle: More material is removed from the bottom of the cut surface than from the top.

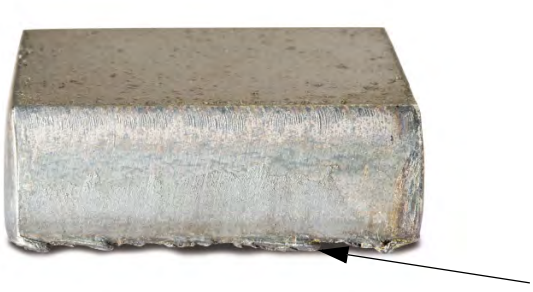
Top edge rounding: There is a slight rounding along the top edge of the cut surface.

Dross

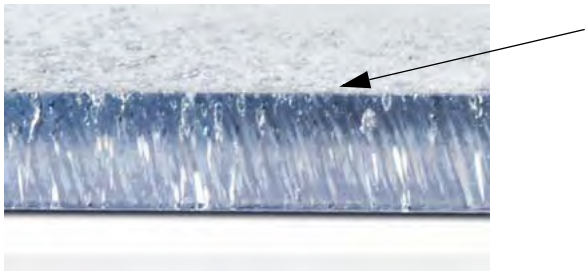
High-speed dross: A small, linear bead of molten material attaches and hardens along the bottom edge of the cut. In addition, S-shaped lag lines are present; dross is difficult to remove and requires grinding.



Low-speed dross: A bubbly or globular accumulation of molten material attaches and hardens along the bottom edge of the cut. In addition, vertical lag lines may be present; dross is easy to remove and flakes off in large chunks.



Top spatter: A light spatter of molten material collects on the top edges of the cut. Usually, this spatter is inconsequential and is most common with air plasma.



Surface finish

Roughness: Depending on the type of metal being cut, some roughness should be expected; “roughness” describes the texture of the cut face (the cut is not smooth).

Aluminum

Top: Air/Air

- Best for thin material under 1/8" (3 mm)

Bottom: H35/N_2

- Excellent edge quality
- Weldable edge

Mild steel

Top: Air/Air

- Clean cut
- Nitrided edge
- Increased surface hardness

Bottom: O_2

- Exceptional edge quality
- Weldable edge

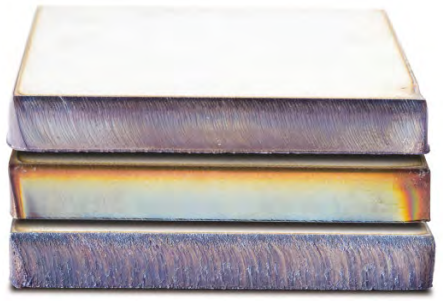
Color

Color results from a chemical reaction between a metal and the plasma gas that is used to cut it. Color changes are to be expected and vary most dramatically with stainless steel.

Top: N_2/N_2

Middle: $H35/N_2$

Bottom: Air/Air



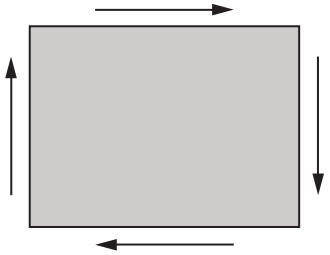
Basic steps to improve cut quality

Step 1: Is the plasma arc cutting in the appropriate direction?

- The squarest cut angles are always on the right side in relation to the forward motion of the torch.
- Verify the direction of the cut.
- Adjust the cutting direction, if necessary. The plasma arc typically spins clockwise with standard consumables.

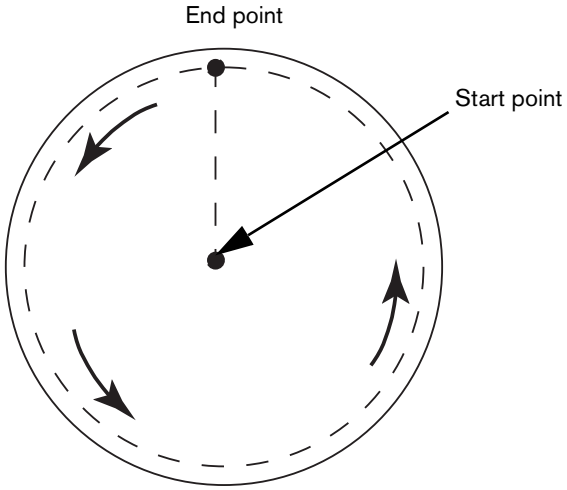
Contour

- The torch travels clockwise.
- The good side of the cut is to the right side of the torch, as it travels forward.



Internal feature (hole)

- Torch travels counterclockwise.
- Good side of the cut is to the right side of the torch as it travels forward.



Step 2: Was the correct process selected for the material and thickness being cut?

Refer to the cut charts in the Operation section of the Hypertherm Instruction Manual. On the CNC, choose the Cut Chart soft key on the Main Screen to view the cut chart for the selected torch type, material and thickness.

Follow the specifications in the cut charts:

- Select the appropriate process for:
  - Material type

  - Material thickness
  - Desired cut quality
  - Productivity goals
- Select the correct plasma and shield gas.
- Select correct parameters for:
  - Gas pressures (or flow rates)
  - Cut height and arc voltage
  - Cutting speed
- Confirm that the correct consumables are being used and verify the part numbers.

Generally, lower amperage processes offer better angularity and surface finish. However, cutting speeds are slower and dross levels are higher.

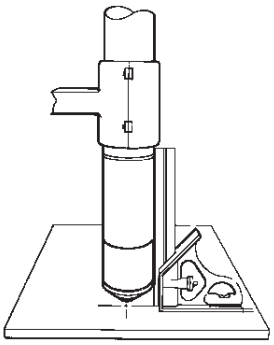
Step 3: Are the consumables worn?

- Inspect consumables for wear.
- Replace worn consumables.
- Always replace the nozzle and electrode at the same time.
- Avoid over-lubricating o-rings.

Use genuine Hypertherm consumables to ensure maximum cutting performance.

Step 4: Is the torch square to the workpiece?

- Level the workpiece.
- Square the torch to the workpiece, both from the front and side of the torch.



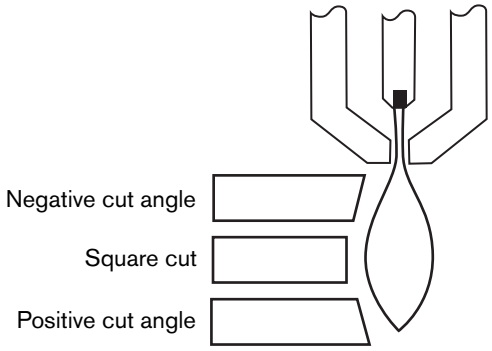
Inspect the material to see if it is bent or warped. In extreme cases this limitation cannot be corrected.

Step 5: Is the cut height set at the proper height?

- Adjust the cut height to the correct setting.
- If you are using arc voltage control, adjust the voltage.

As consumable parts wear, arc voltage settings need continual adjustment to maintain cut height.

- Cut height can impact angularity.



- Negative cut angle: torch too low; increase cut height.
- Positive cut angle: torch too high; decrease cut height.

A slight variation in cut angles may be normal if the variation is within tolerance.

Step 6: Is the cutting speed set too fast or too slow?

- Adjust cutting speed as needed.

Cutting speed may also impact your dross levels.

- High-speed dross: The cutting speed is too fast and the arc lags behind. Reduce the cutting speed.
- Low-speed dross: The cutting speed is too slow and the arc shoots ahead. Increase the cutting speed
- Top spatter: The cutting speed is too fast, reduce the cutting speed.

In addition to speed, both material chemistry and surface finish can impact dross levels. When the workpiece heats up, more dross may form on subsequent cuts.

Step 7: Are there problems with the gas delivery system?

- Identify and repair any leaks or restrictions.
- Use properly sized regulators and gas lines.
- Use pure, high-quality gas.
- If manual purge is required, such as with the MAX200, confirm that the purging cycle was completed.
- Consult the gas distributor.

Step 8: Is there torch vibration?

- Verify that the torch is securely attached to the table gantry.
- Consult the system builder, your table may require maintenance.

Step 9: Does the table need to be tuned?

- Check and ensure that the table is cutting at the specified speed.
- Consult the system builder; the table speed may need tuning.

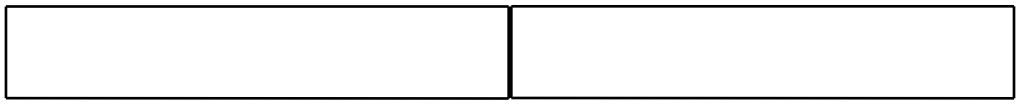
Bevel cutting tips

Cutting bevel angles with plasma requires specialized CAM software, process parameters, and a post-processor for the Hypertherm CNC. In some cases, cutting parts to specification may take several iterations. Use the following information to identify and solve cut-quality issues when cutting bevel angles.

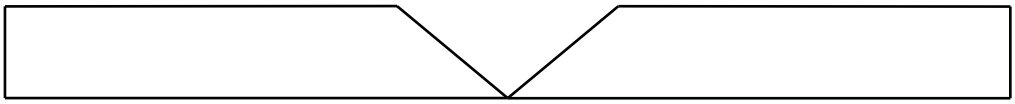
Types of bevel cuts

Plasma bevel cutting comprises six distinct cuts. Each cut is shown below from a side view and with another bevel-cut part. Straight, vertical cuts are referred to as I cuts. Refer to your CAM software for information on programming these cuts.

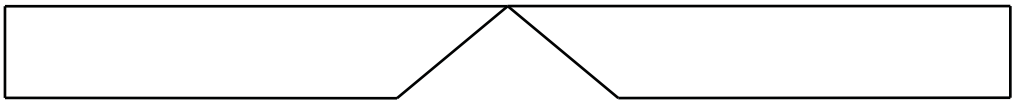
I cut



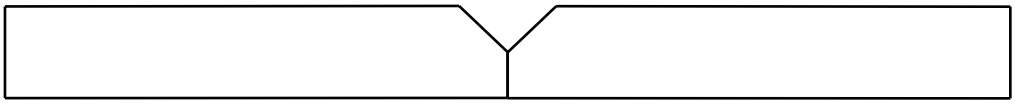
V cut



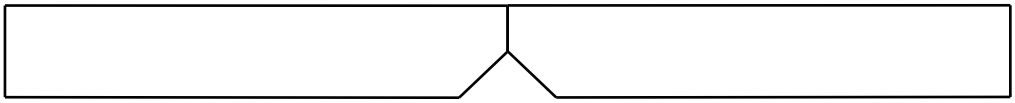
A cut



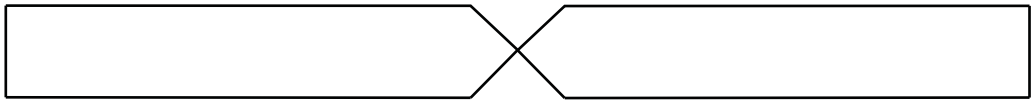
Y Top cut



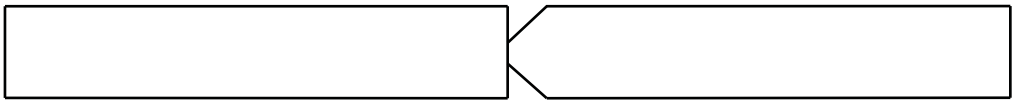
Y Bottom cut



X cut



K cut



Bevel cutting tips

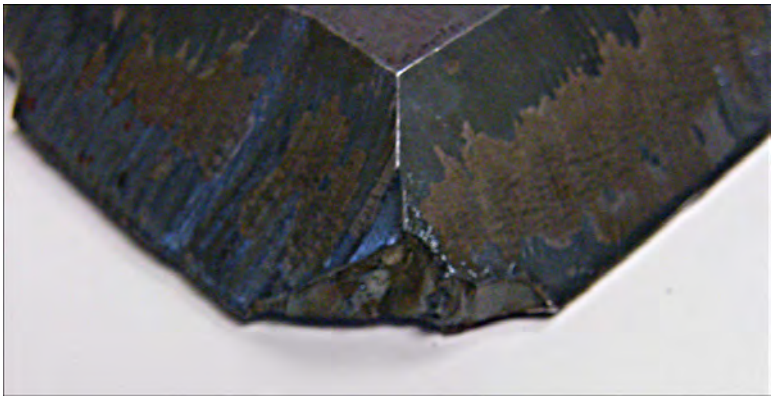
When troubleshooting a bevel-cut part, perform the following actions in order:

1. Measure and correct for bevel angle.
2. Measure and correct for land dimension if cutting Y Top cut parts.
3. Measure and correct for part dimension.

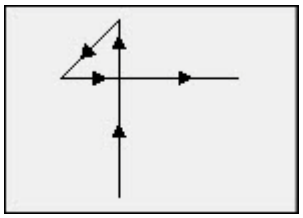
Quality bevel-cut parts result from a strong partnership between the part programmer and machine operator. The part programmer can take advantage of bevel parameters available in the CAM software used to produce the part program, and the operator can perform adjustments available on the CNC. Correcting a cut quality issue often requires that the part programmer make changes in the CAM software and generate a new part program.

Following are several common cut quality issues that can occur when bevel cutting, and suggestions for both the part programmer and machine operator to follow to eliminate the issues.

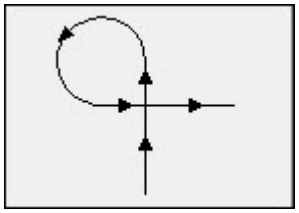
Clipped corner



In bevel cutting, a corner loop is used by the CAM software to reposition the bevel head when the cutting angle changes between two cuts. A clipped corner on a part could result when the corner loop is not large enough. To correct corner loop size, refer to the CAM software, then generate a new part program. Two types of corner loops are shown below:



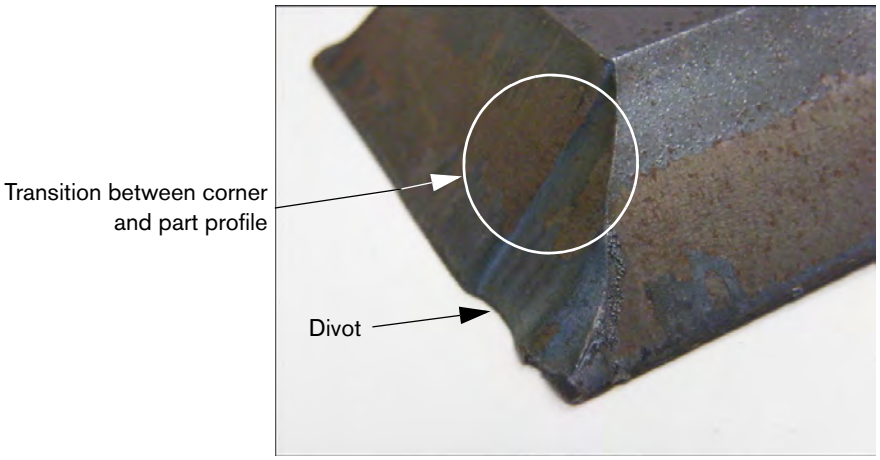
Linear corner loop



Arc corner loop

Inconsistent cut quality on a single part surface

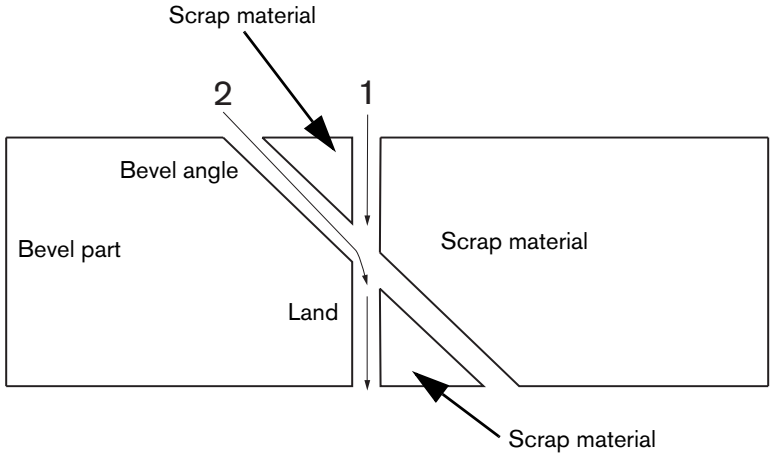
The graphic below shows two cut quality issues: a transition in cut quality from the part corner into the part profile, and a divot in the cut.



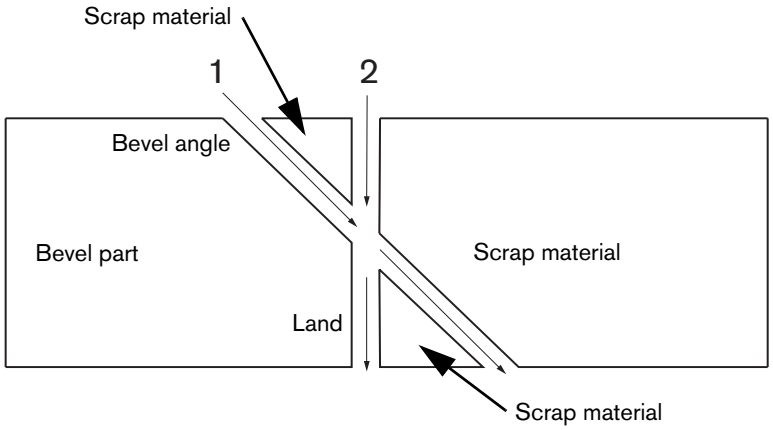
Increase the length of the lead-in segment. A longer lead-in allows the torch to lock on to the correct combination of arc voltage and cut height. To correct lead-in length, refer to the CAM software, then generate a new program.

Rounded edges on Y Top cuts

For Y Top cuts, a rounded edge can sometimes result if the land is cut before the bevel angle is cut. The examples below show the side view of the Y Top cut bevel part.



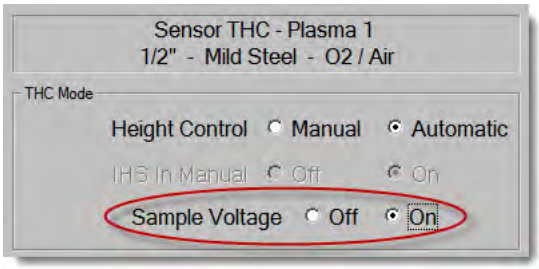
To change the order in which the passes (also called profiles) are cut, refer to your CAM software, then generate a new part program.



Part dimensions change within a nest

A change in part dimensions when cutting a nest of parts may result from in correct torch height due to an incorrect arc voltage setting, or worn consumables.

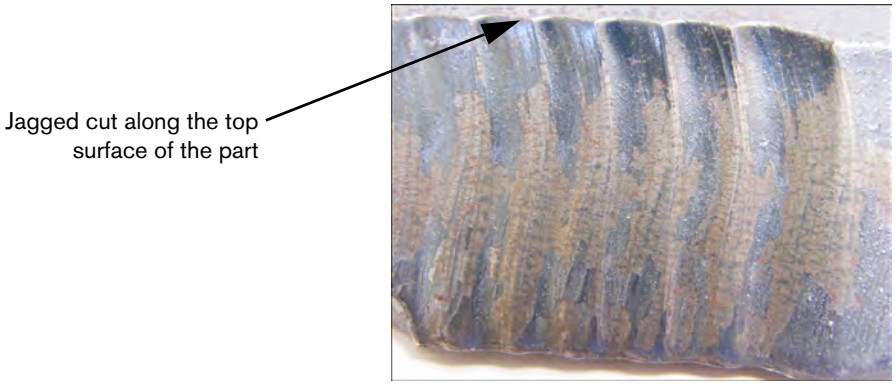
- Arc voltage is set in the cut chart but can be overridden on a job-by-job basis by adjusting the setting on the Process screen. To change the arc voltage setting, choose Setups > Process, and enter the new value for Set Arc Voltage. If you are using the torch height control in Manual mode, enter a new cut height in the Process screen.
- If your torch height control supports arc voltage sampling, be sure to turn it on in the Process screen (Setups > Process). Arc voltage sampling adjusts the arc voltage automatically as consumables wear.



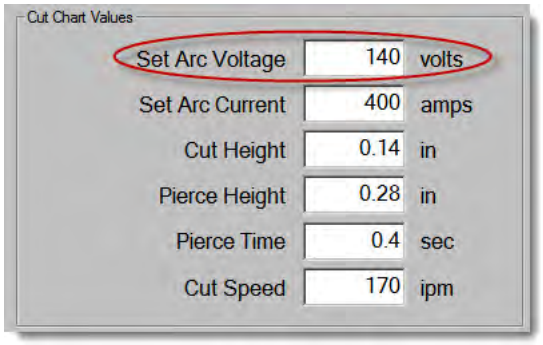
- Check the torch consumables and replace them if they are worn.

Jagged cut

A jagged cut can result from the torch repeatedly touching the plate.

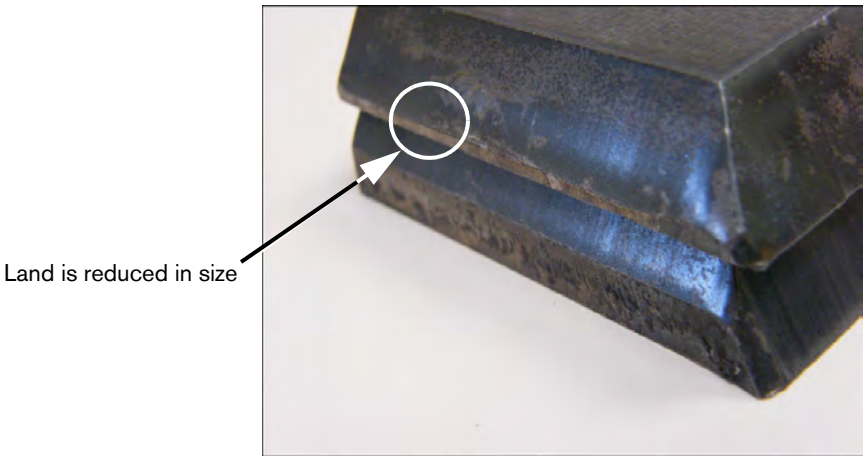


- Arc voltage may be set for the thickness of the material and not for the effective thickness of the material when cutting on an angle. To increase arc voltage on the CNC, choose Setups > Process, and adjust the arc voltage:



- Turn on arc voltage sampling. If arc voltage sampling is on, check the consumables and replace them if they are worn.
- Check and possibly increase the cut height in the part program. Since cut height affects the part dimension, you may also need to adjust additional process parameters that work with the cut height and affect part dimension. Changing these parameters requires you to generate a new part program.

Incorrect dimension for the land on a Y Top cut



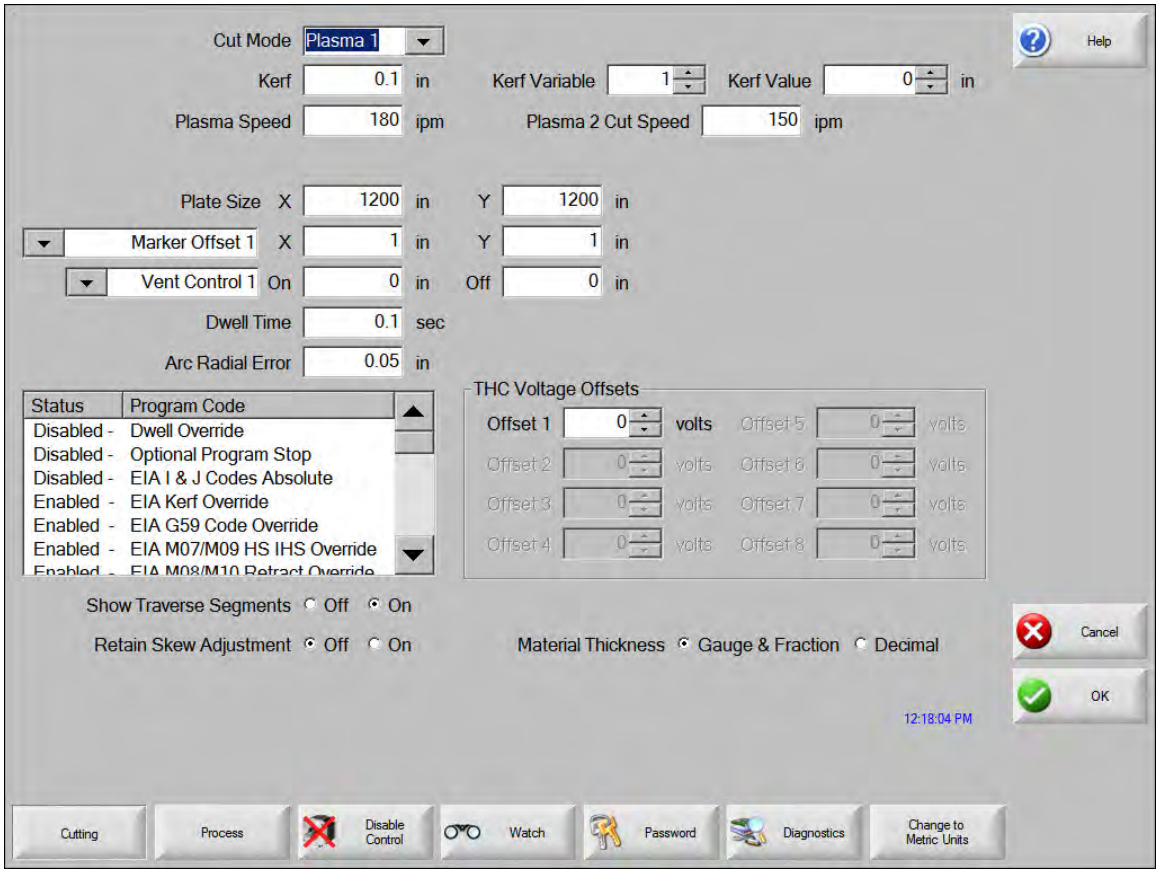
- Increase the Set Arc Voltage parameter in the Setups > Process screen to account for the change in material thickness because you are cutting at an angle.
- Turn on Sample Arc Voltage, also in the Setups > Process screen.
- Check and possibly increase the cut height in the Process screen.

Section 6

Cutting Screen and Watch Window Setup

Cutting setup

From the Main screen, choose the Setups soft key to open the Cutting screen. Here you can adjust the parameters for the cut mode you want to use: Trial motion (without cutting), Plasma, Laser, Oxyfuel, and Waterjet.



Cut Mode Specifies the current cut mode. Trial mode allows the operator to dry-run the current part program without cutting.

Kerf Specifies the amount of kerf that will be applied to the current part program. Care should be taken when selecting a kerf value as this parameter can cause invalid geometries to be generated. For example, adding a kerf of 12.7 mm (0.5 inch) to an arc with a radius of 6.35 mm (0.25 inch). After entering a kerf value, the kerf compensated cut path can be viewed by pressing the Kerf soft key under the Part Options menu.

Kerf Variable and **Kerf Value** Creates a kerf variable table that assigns a variable to a Kerf value. Up to 200 variables can be entered to create a reference table.

This kerf variable can be used within a part program to define the kerf value and as torch parts wear, the kerf value changes. If the kerf variable value is updated as the consumable wears and changes, the new value will be called by the kerf variable command with all programs loaded that use the variable.

The EIA-274D part code for left kerf variable is the G43 code. In the following example, G43 D1 X0.06:

- G43 is the kerf variable setting
- D1 is the kerf variable. Any number from 1 – 200 can be used.
- X0.06 is the selected kerf value.

Trial and **Cut Speed** Specifies the speed for the current cut mode. These speeds are saved independently for trialing and cutting. Both speeds are limited to the maximum machine speed. Cut and trial speeds can be executed at the embedded F-code speed within a part program.

Marker 1 and **Marker 2 Speed** Specifies the speed for the selected marker. These speeds are saved independently for each marker and are executed through the marker tool selection within a part program.

Marker 1 is activated by EIA RS-274D M09 and M10, or an ESSI 9 and 10.

Marker 2 is activated by EIA RS-274D M013 and M14, or an ESSI 13 and 14.

Plate Size Specifies the dimensions of the current plate. This dimension is used when loading a part to determine if it will fit on the plate. It is also used for viewing the part in screen view.

Vent Controls 1 – 50 Enter rail values for up to fifty programmable zones for fume extraction damper control. Based on the rail position, the vent controls digital outputs activate dampers at the selected zone for increased performance.

Marker Offsets 1 – 12 Enter values for up to twelve programmable marker offsets. The machine is offset by this amount at maximum speed when the appropriate marker code is detected.

Marker offset 9 Used only with the Offset IHS parameter on the Sensor THC/Plasma1 and 2 Process screens. By entering a value for Marker Offset 9, the torch will move from its current position to the marker offset distance, perform IHS, then return to its previous position. An Offset IHS is often used when cutting a pre-pierced workpiece so that the torch doesn't perform an IHS on a pierce point.

Marker offset 10, 11, 12 If values other than 1 are entered for marker offsets 10, 11 or 12, the Align wizard and CutPro Wizard will automatically allow you to select the laser pointer or the torch for alignment.

Dwell Time Specifies the amount of dwell (delay) that is inserted into the current part program when an appropriate RS-274D program block is reached. This time can be overridden in the part program. For example, in EIA programming a G04 X3 causes a three second dwell to be inserted at the current program block. A G04 with no X-code inserts a dwell with the current Dwell Time parameter.

Arc Radial Error Specifies the arc error tolerance to be used when checking the current segment for dimensional accuracy. All ESSI or EIA programs are comprised of lines, arcs, and circles. Arc Radial Error is used to make sure that the starting and ending radial vectors are within tolerance to describe a valid geometry.

Status/Program Code See the *Status/Program Code* section later in this section.

Show Traverse Segments Allows traverse segment lines (displayed in yellow) to be turned OFF or ON during all part preview displays.

Retain Skew Adjustment Retains the last calculated plate skew for all subsequent parts that are loaded. If disabled, any new part that is loaded will remove any previously calculated plate skew.

THC Voltage Offsets Voltage offsets add or subtract voltage from the arc voltage set point. Use a voltage offset to compensate for electrode wear in the torch. As the electrode wears, the arc voltage set point needs to increase to maintain a consistent cut height.

Phoenix can automatically increase the voltage offset if you have set the Volts/Min parameter on the Change Consumables screen. As time accumulates on the electrode on the Change Consumables screen, Phoenix increases the voltage offset value. To prevent a voltage offset from accumulating automatically, choose Change Consumables from the Main screen, and set Volts/Min to 0.

While cutting, pressing the Raise and Lower station buttons on the operator console of the EDGE Pro or EDGE Pro Ti also changes the voltage offset for the selected station by 0.5 per button press.

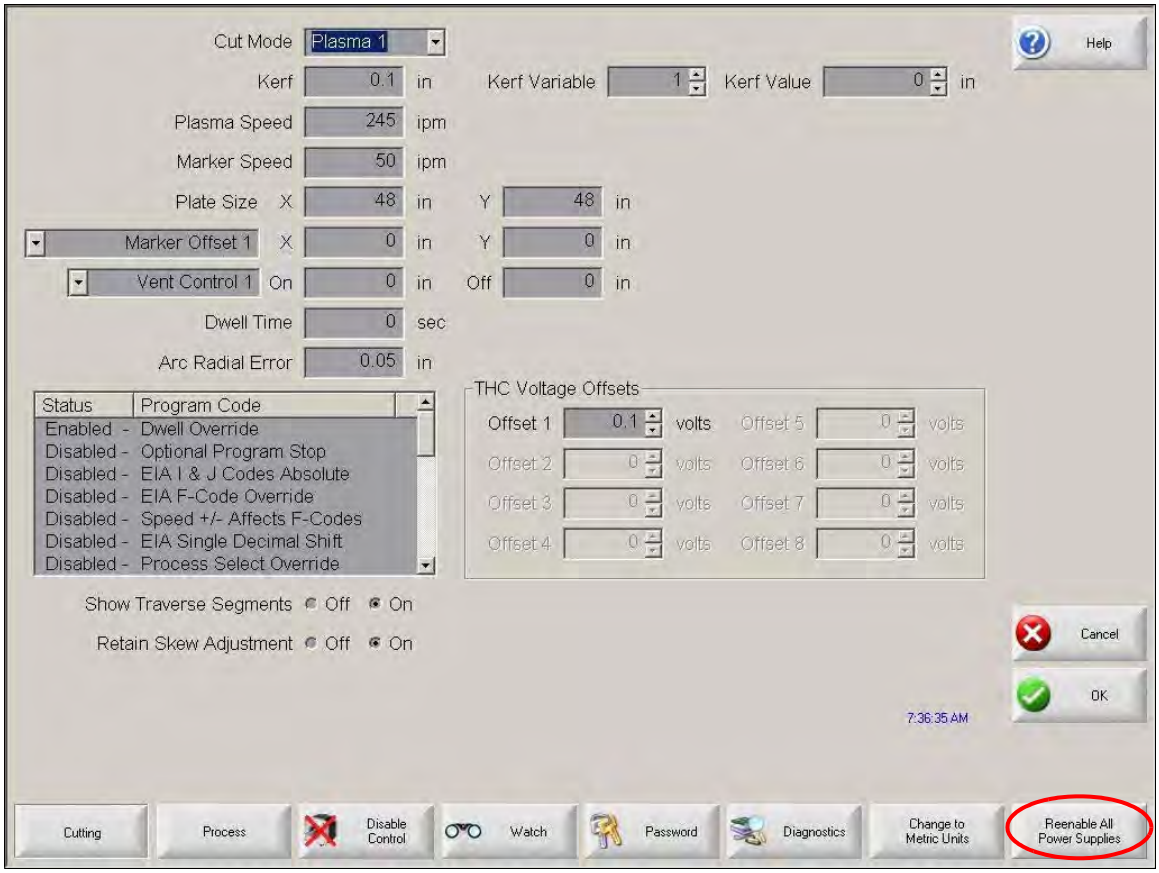
To reset the Voltage Offset, choose Setups from the Main Screen and set the voltage offset to 0.

Any value you enter for a voltage offset stays in effect until you change. The voltage offset does not reset to 0 if you change consumables or load a new cut chart.

Material Thickness Changes the display of material thickness in the cut chart screen between gauge and fraction and decimal. The CNC must be using English units to display this option.

Change to English/Metric Units Changes all measurements from English units to Metric, or from Metric units to English throughout the Phoenix software. The changes take place immediately after you choose the soft key, but you must choose Save Changes when you exit the Setups screen.

Re-enable All Power Supplies Press this soft key to re-enable any disabled HD4070 or HPR power supply with auto gas. This key is enabled only if a power supply has been disabled.



Status/Program Code

Dwell Override When this parameter is enabled, embedded dwell G04 X *value* codes in an RS-274D program override the operator-entered dwell time.

Optional Program Stop Allows overriding of the optional program stop code M01 in the current part program. If enabled, an M01 code operates identically to M00. If disabled, the M01 code is ignored.

EAI I & J Codes Selects absolute or incremental RS-274D programming mode. In incremental mode, all offsets for X, Y, I and J are relative to the current block. In absolute mode, all offsets for X, Y, I and J are relative to an absolute reference point unless they are changed by using a G92 (set axis presets) program code.

EIA F-Code Override When this parameter is enabled, embedded F-codes in an RS-274D program override any operator-entered cut speed.

Speed +/- Affects F-Codes When this parameter is enabled, the control applies the speed increase/decrease percentage to all embedded F-codes that are encountered in the part program.

EIA Decimal Shift Some programming styles are structured so that the decimal point in the EIA positioning affecting part sizing is assumed. The EIA decimal shift parameter allows the operator to select the location of the decimal point when translating parts by selecting normal or single for the correct translation. The selection should be set to Normal unless your part programs have only a single digit to the right of the decimal point.

Process Select Override When enabled, this feature allows the part program to override the process select input.

Station Select Override When enabled, this feature allows the part program to override the currently selected station select input.

Auto Torch Spacing Override When enabled, this feature allows the part program to override the manually selected torch spacing inputs.

G97 Loop Count Prompt When enabled, this feature will post a message on the screen to enter the number of loops or repeats to be selected when an EIA G97 code without a "T" value is encountered in the part program.

ESAB Multi Torch Support When enabled, this feature allows ESAB-style ESSI part programs to map codes to specific station selects.

ESSI Code	EIA Code	Description
7	M37 T1	Select station 1
8	M38 T1	Deselect station 1
13	M37 T2	Select station 2
14	M38 T2	Deselect station 2
15	M37 T3	Select station 3
16	M38 T3	Deselect station 3

Force G40 Kerf Disable In a part program, kerf is enabled and disabled using EIA G41 / G42 and G40 codes. Standard operation is to disable kerf at the cut off even if the G40 kerf disable is not in a program. With this parameter, you can turn off the "forced" G40 kerf disable if no G40 is used in the program by disabling the parameter.

G40 Used in Simple Shapes This parameter is used with the Force Kerf Disable parameter to allow the G40 code that is normally inserted in to a simple shape from the shape library to be omitted by disabling the parameter.

Auto Start after APA This parameter is used with the Automatic Plate Alignment feature to allow cutting to begin automatically after completion of the automatic plate alignment.

EIA Code 2 Decimal Shift Some programming styles are structured so that the decimal point in the EIA positioning affecting part sizing is assumed. The EIA Code 2 Decimal Shift parameter allows the operator to select the location of the decimal point when translating parts by selecting normal or single for correct translation. The selection should be set to Normal unless your part programs require two decimal shift to the right of the decimal point.

M17, M18 Used as Cut Codes This allows the EIA-274D M17 and M18 codes to be used for cut on and off commands when enabled.

M76 Rotary Shortest Path When enabled, this disables tilt and rotate software overtravels for dual tilt-style bevel heads and allows the EIA-274D M76 Rotate go to Home command to select shortest path. When disabled it allows motion by longest path when homing. This is advantageous for some bevel head designs.

EIA Kerf Override If this setting is disabled, all kerf value codes and Load Kerf Table variable are ignored. This parameter is enabled by default and cannot be changed while the part program is paused. This is useful when a process is used at a cutting machine that is different from the one that was used to create the part program.

EIA G59 Code Override When enabled, this option allows codes in the part program to select cut chart process variables (V500 codes) and cut chart process parameter overrides (V600 codes). If the G59 Code Override is disabled, the CNC ignores all G59 codes in the part program. This option is enabled by default.

EIA M07/M09 HS IHS Override When this option is enabled, M07 HS and M09 HS codes in the part program force an IHS when the pierce point is within the Skip IHS Distance set on the Process screen for the ArcGlide THC and Sensor THC.

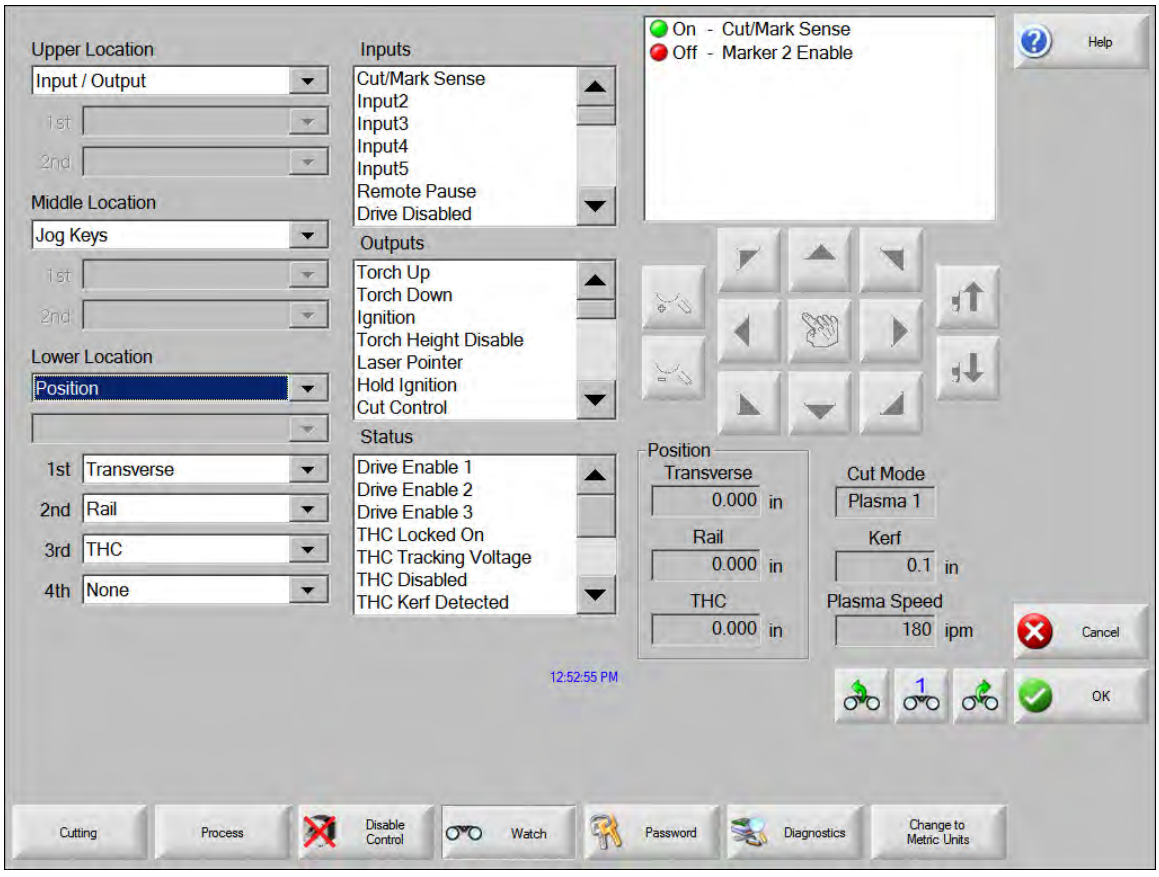
EIA M08/M10 Retract Override When this option is enabled, M08 RT and M10 RT codes in the part program cause the torch to retract to the transfer height when the cut is complete. When disabled, the torch retracts to the retract height.

Stop on Single Arc Lost If this setting is enabled, any cut sense input that is lost for longer than the arc off time during the cut pauses the part program with a Cut Sense Lost message.

Watch Window Setup

The Watch Window displays process-related parameters during cutting. You can customize a certain portion of the screen to display functions that are critical for your particular cutting operation. Whether it is Current Speed, Position, I/O status, or torch consumable life, you can display the information that you want to view during operations.

As these parameters are turned on or off, the Watch Window is updated.



Several options are available to personalize the Watch Window and not all options can be viewed at one time. The options are grouped into two sizes of widgets or icons. Large widgets may be placed into the upper position at the top of the Watch window or in the middle of the Watch Window. Small widgets are positioned in the lower left corner of the Watch Window next to the cut information and clock. The cut information and clock at the lower right may not be edited.

Selections in the Watch Window will change slightly based on the CNC I/O configuration and machine options that have been enabled.

The Watch positions will allow for the following selections:

None Select None to leave the position blank.

Input/Output Allows current state of selected Input, Outputs or Status information to be displayed during cutting. This can be especially useful in debugging gas control sequencing problems. To add or delete a desired Input, Output or Status point to the Input/Output list box, double-click an item or highlight an item and + (add) or – (delete) keys on the alpha-numeric keypad.

Digital Speedometer Allows the cut speed, maximum machine speed and current machine speed to be numerically displayed.

Position Allows the position for the selected axis to be displayed. Only two axes may be displayed at the Upper or Middle locations. The Lower location will allow up to four axes to be displayed.

Following Error Allows the Following Error to be displayed. Following Error is the distance between the position the control has calculated and the actual position of the torch. A large Following Error may indicate that the cut speed selected may be beyond the capability of the cutting system. Only two axes may be displayed at the Upper or Middle locations. The Lower location will allow up to four axes to be displayed.

Command Voltage Allows you to view directional motion command voltage being sent to the amplifier for velocity type drives. This displayed voltage also equates to current being commanded for motion in current type drives. Peak voltage can be displayed for a specified amount of time.

Temperature Selecting to add the Temperature information to the Watch window will display the current temperature inside the control in Fahrenheit or Celsius (selected at the Special Setups screen).

Specific control hardware is required.

Speedometer Allows cut speed, maximum machine speed and current machine speed to be graphically displayed while cutting.

Oxyfuel Torch Tip Allows the selected oxy fuel torch tip (1 – 12) consumable life to be graphically displayed while cutting. This helps determine when the torch tip should be replaced and keeping track of torch tip data for statistical process control (SPC).

Plasma Torch Tip Allows the selected plasma torch tip (1 – 8) consumable life to be graphically displayed while cutting. This helps determine when the torch tip should be replaced and keeping track of torch tip data for statistical process control (SPC).

Plasma Electrode Allows the selected plasma electrode (1 – 8) consumable life to be graphically displayed while cutting. This helps determine when the electrode should be replaced and keeping track of electrode data for statistical process control (SPC).

Jog Keys Selecting the jog keys option allows a directional keypad to be added to the watch window for manual motion directly from the touch screen. The operator can press the hand icon in the middle of the navigation pad to enable manual mode. Select a move speed and press an arrow for manual motion in the corresponding direction.

Laser Nozzle Laser Nozzle consumable life to be graphically displayed while cutting. This helps determine when the nozzle should be replaced and keeping track of nozzle data for statistical process control (SPC).

HPR Power Supply Allows you to view status for inputs, outputs and gas pressures for the HPR autogas console. Up to four power supplies may be monitored. This is generally used for service diagnostics only.

Cut Time Allows the operator to see an estimate of the amount of time it will take to cut the selected part or nest. This window also displays how much time has been completed and how much remains. A progress bar gives a graphical display of the cut time. The estimate of the cut time is based on the complexity of the parts or nest and the cut speed.

This window can help you optimize production plans and the use of resources.

Pierces Shows the operator how many pierces are needed for the part or nest that has been selected, how many pierces have been completed and how many remain.

Operators can use this window to plan consumable changes.

Process Data The Process Data option allows you to view up to four selected items for a selected cut or marking process. Process timers and status items for Oxy Fuel, Plasma, Marker, Waterjet and Laser may be selected.

The process data will only be displayed during the current cut process. Example: Plasma 1 process parameters will only be displayed in the Watch window at the main cut screen while cutting in Plasma 1 Mode.

System Errors Displays errors generated by the CNC, the plasma supply, or the ArcGlide torch height control. Each type of error has a different icon so you can identify the source of the error (CNC, plasma supply, or ArcGlide).

See *The following sections describe tools that you can use to diagnose and troubleshoot your CNC and Phoenix software.* for a list of the CNC errors. See the manual for the plasma supply or ArcGlide when either is the error source.

Multiple Watch Windows

Up to ten different Watch windows may be configured on the control for quick selection and viewing of the Watch icons:

To configure different Watch windows for viewing, first access the Watch setup screen. Press the number icon to enter a number or the Left/Right arrows to move up and down through the selections. The different Watch windows can be selected and viewed during operation using the same selection process.

Section 7
Cut Processes and Cut Charts

Process overview

A cut process is a combination of cutting parameters that have been tuned to provide the best cut quality for a type and thickness of material on a specific cutting system. Hypertherm CNCs provide processes for numerous plasma systems, and for marker, oxyfuel, laser, and waterjet systems.

Each process includes factory settings that you can customize for your jobs. Although each cutting process is unique, the way you set up the processes on the CNC is similar for each type of process.

The CNC stores the process information in the Process and Cut Chart screens. If you make a change to a cut chart, the new value will also be entered into the Process screen. However, changes you make in the process screen do not change the cut chart. A common practice is to load the cut chart for your cutting job, then make any one-time changes in the Process screen. However, if you will be using the modified information more than once, then you can make the changes in the cut chart and save the cut chart. See *Saving changes to a cut chart* on page 172 for more information.

Cut processes and part programs

In some cases, a part program may contain process information that loads when you run the part. When running the CutPro Wizard on the CNC, the wizard automatically loads the correct cut chart and prompts you to load the consumables for the process specified in the cut chart. CAD/CAM software such as Hypertherm's ProNest® provides the process information to the CNC in the part program so that you do not need to select the process or cut chart on the CNC. For programming parts that include process information, consult your CAD/CAM software or the *Phoenix V9 Series Programmer's Reference* (806420).

Before you begin

Before you can view the processes available on your CNC, the following steps must be completed. These steps are usually performed by your original equipment manufacturer (OEM), system integrator, or system administrator.

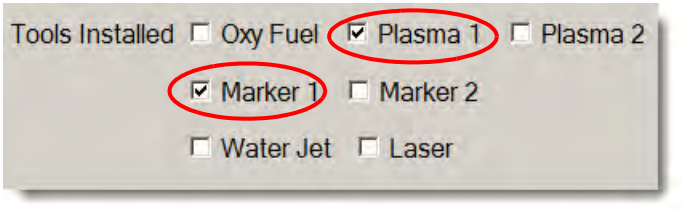
1. Select the processes under Tool Installed in Special Setups. This activates the Process screen for each type of cut process.

2. Select the lifter and cutting system models in the Station Configuration screen. This activates the cut charts for the specific models of cutting systems.

3. If needed, activate privileges in the Special Setups screen for adding, removing, or changing processes. See *Activate privileges in Special Setups* on page 128.

Processes selected in Special Setups

In the Special Setups screen (Setups > Password > Special Setups), select the type of cutting tools installed on your table. Selecting these tools enables the Process screens and the cut mode selections available to the operator.

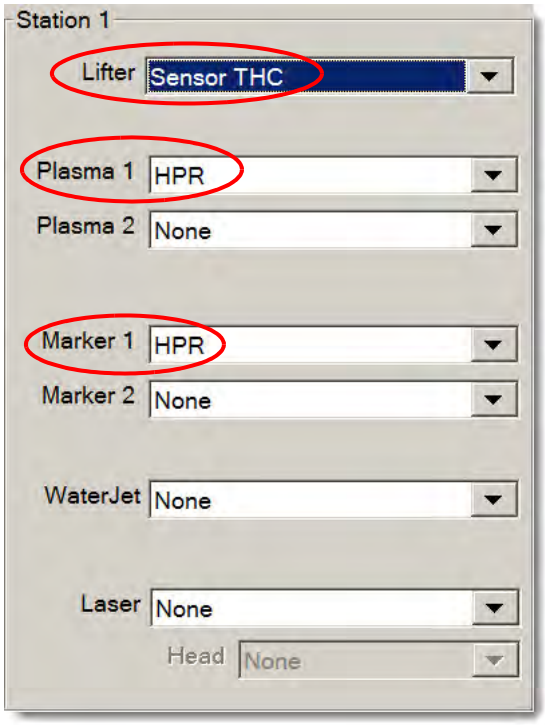


Selecting Plasma 1 and Marker 1 enables these soft keys on the Process screen:

System models selected in Station Configuration

The Station Configuration screen (Setups > Password > Station Configuration) enables the cut charts for the selected system model. For example, if you are using an HPR plasma system for cutting and marking, you would choose HPR for Plasma 1 and Marker 1, and these cut charts would become available.

You also need to select the torch lifter on your cutting table. The Process screen looks different depending on the torch lifter selected.



The CNC also provides cut charts for waterjet and laser systems. To enable these cut charts, you would select your waterjet or laser system from the lists provided in the Station Configuration screen.

The CNC provides cut charts for oxyfuel processes, but they become available when you select oxyfuel in the Tools Installed section of the Special Setups screen.

When to use Plasma 1 and 2 and Marker 1 and 2

Plasma 1 and Plasma 2, on the Special Setups screen, enable two separate cutting processes. Marker 1 and Marker 2 likewise enable two separate marking processes. Plasma 1 and Plasma 2 on the Station Configuration screen are used to make available separate cut charts and other functionality specific to the cutting system models selected on that screen.

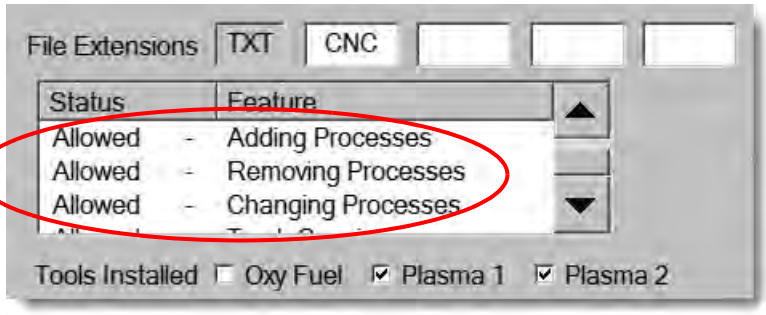
In general, follow these guidelines for using Plasma 1/2 and Marker 1/2 on the CNC:

- In a single-torch cutting system, you need only Plasma 1 and Marker 1 for Station 1.
- In a two-torch cutting system where both torches cut parts using the same process and cut chart, select Plasma 1 and Marker 1 for both Stations 1 and 2.
- In a two-torch system where the torches are from different types of plasma supplies you need to select Plasma 1 and Marker 1 for Station 1, and Plasma 2 and Marker 2 for Station 2 so that the CNC makes a second cut process and cut chart available.

For more information on setting up the CNC for a two-torch system, see the *Phoenix V9 Series Installation and Setup Manual* (806410).

Activate privileges in Special Setups

To enable the ability to change cut charts, choose Setups > Password > Special Setups. In the Status/Feature box, set Adding Processes, Removing Processes, and Changing Processes to Allowed.



Cutting screen, Process screen, and cut charts

For each cut process, you will have a Cutting screen, a Process screen, and in most cases, a cut chart. (The CNC provides cut charts for the cutting system models listed in the Station Configuration screen.) Selecting the cutting system in that screen enables the matching cut charts for the system. The CNC provides cut charts for plasma, laser, waterjet and oxyfuel cutting systems.

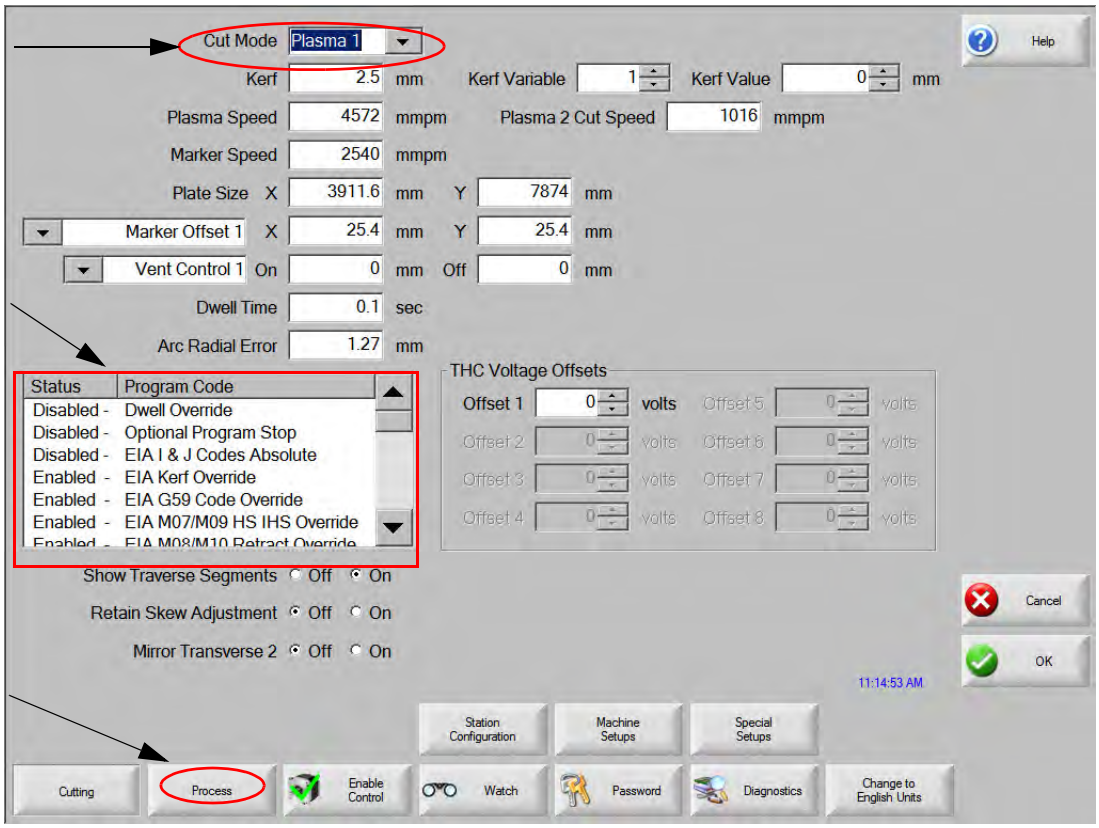
Cutting screen

Select Main > Setups to open the Cutting screen (shown below). In this screen, you can select the process you will be using (under Cut Mode), and set options on how the CNC handles codes from the part program. This screen is always available regardless of which cut process you will be using. To learn more about the Cutting screen, see *Cutting Screen and Watch Window Setup* on page 117. The options available on the Cutting screen may vary depending on the user level selected.

Select Cut Mode here or on the Main Screen.

These options govern how the CNC handles part program codes.

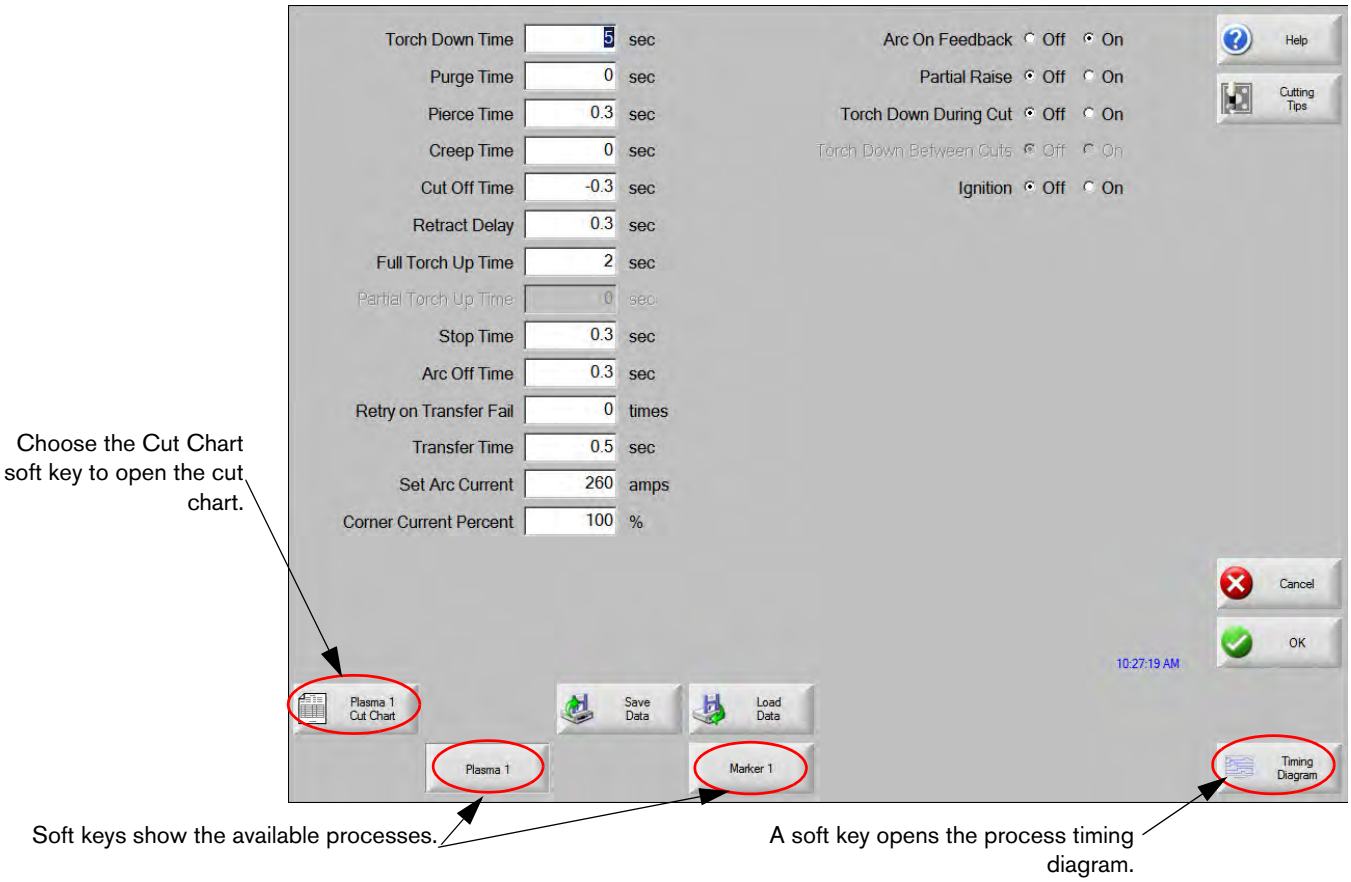
Choose the Process soft key to open the Process screen.



Process screen

Choose Main > Setups > Process to open the Process screen. This screen provides different options depending on the torch height control that was selected in the Station Configuration screen.

The Process screen shown below appears when you have Other selected as a lifter in the Station Configuration screen. To learn more about the Process screen for the ArcGlide THC or Sensor THC, see *Torch Height Controls* on page 175. To learn about the Process screen for a Command THC, see *Command THC Setup* on page 203.



The CNC provides a Process screen for the different cut processes. In the screen shown above, Plasma 1 and Marker 1 are the available cut processes.

Timing diagrams

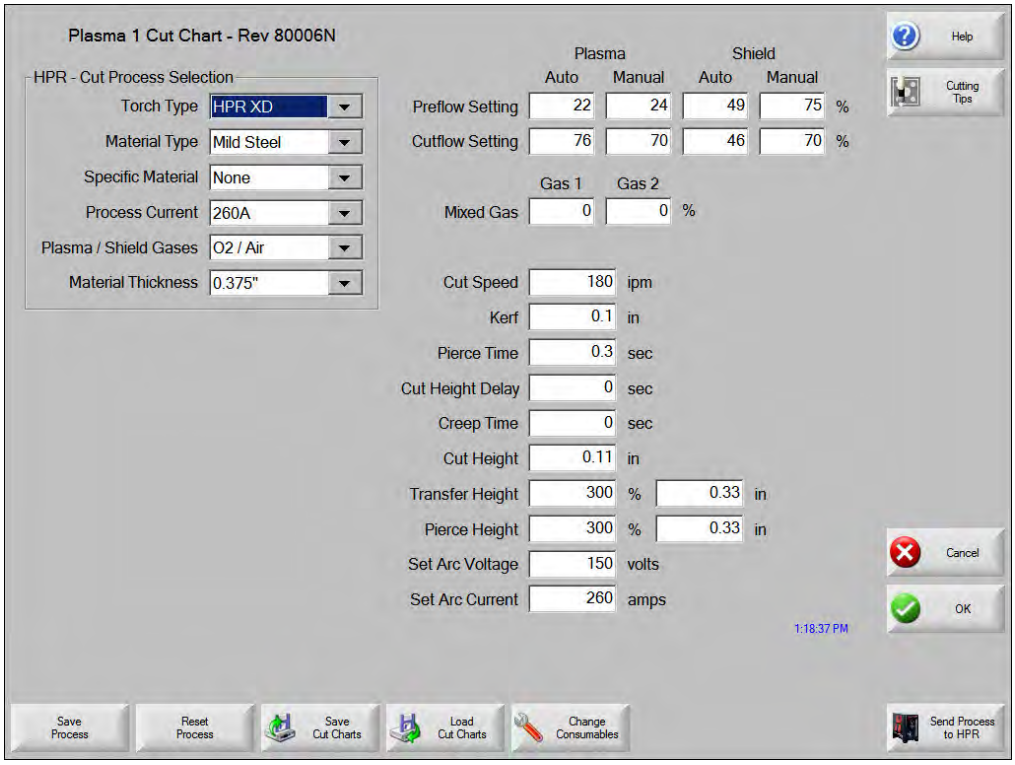
A timing diagram gives a map of the cutting process timing. It shows the activation and deactivation of the CNC outputs that send signals to the plasma, marker, oxyfuel, laser, or waterjet supply. The timing diagram for each type of process shows the outputs that are unique to that process. For example, the plasma and laser timing diagrams will show different outputs.

Saving a cut process

Use the Save Data and Load Data soft keys to save the settings from the Process screen. The CNC saves the file on either the CNC hard drive or a USB memory stick. You can then transfer this file to another CNC or save a copy as a backup.

Cut Chart screen

On the Process screen, choose the Cut Chart soft key to open the cut chart for the selected cut process. Cut charts are provided for different models of cutting systems and are made available for a cut process when you choose the cutting system in the Station Configuration screen. The CNC allows you to load two plasma cut charts (Plasma 1 and Plasma 2) and two marker cut charts (Marker 1 and Marker 2). You can load one oxyfuel, laser, or waterjet cut chart at a time.



Common soft keys

The cut chart screens for all processes have these soft keys:

Save Process Saves the current process settings to the hard drive.

Reset Process Press the Reset Process soft key to reset the current cut chart to factory defaults based on the process variables selected. The CNC retrieves these settings from the cut chart file ending in .fac.

Save Cut Charts Press the Save Cut Charts soft key to copy the current user and factory cut chart files to USB memory stick. User files have an .usr file extension and the factory files have a .fac file extension.

Examples of user and factory file names:

Mild Steel-HPR XD-HPR.usr

Mild Steel-HPR XD-HPR.fac

If you have changed any cut charts, be sure to use this feature to backup your cut charts before you load a cut chart update (cutchart.exe). Contact your OEM or system integrator if you need to update your cut charts.

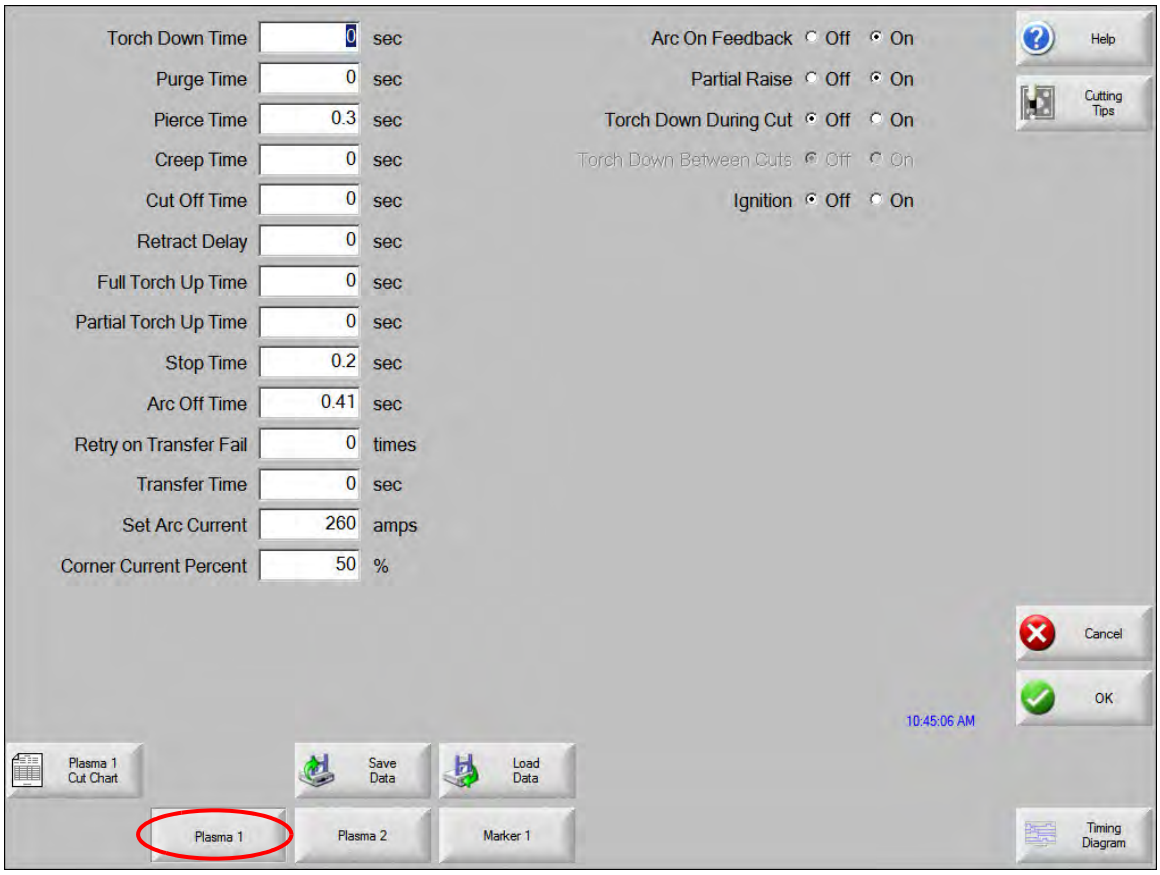
Load Cut Charts Press the Load Cut Charts soft key to load cut charts from the USB memory stick.

Change Consumables Press the Change Consumables soft key to view the torch consumables, their part numbers, and accumulated cutting time on the plasma nozzle and electrode or oxyfuel torch tip. For more information, see *Changing Consumables* on page 248.

Plasma process

Plasma 1 and Plasma 2 process screens

The CNC provides a Process screen for Plasma 1 and Plasma 2 processes. From the Main screen, choose Setups > Process > Plasma 1 or Plasma 2 to open the screen.



The Plasma 1 and Plasma 2 Process screens provide different options for the type of torch height control being used. Plasma 1 and Plasma 2 will show the name of the Hypertherm torch height control in the upper left corner. See *Torch Height Controls* on page 175 for information about the ArcGlide THC and Sensor THC. See the instruction manual for the plasma supply for setup and operation information.

When using a torch height control other than ones manufactured by Hypertherm, the Plasma 1 and Plasma 2 screens look like the one above. These screens provide a series of timers used to control outputs or motion of the torch lifter. The timers and motion start after the CNC reads an M07 (Cut on) code in the part program.

Torch Down Time: Sets the time to activate the Torch Down output to lower the torch to the workpiece. If you are using a torch height control system, set Torch Down Time to zero.

Purge Time: Sets the time delay from torch ignition until motion starts if Arc On Feedback is off. Set Purge Time to zero if Arc On Feedback is on.

Pierce Time: Specifies the time delay from when the torch completes lowering until motion starts at Creep Speed. The motion delay begins when the Cut Sense input turns on. Pierce Time allows the torch to completely pierce the material before motion starts.

Creep Time: Sets the time after piercing the part that the torch travels at Creep Speed. (See Setups > Machine Setups > Speeds for the Creep Speed.) The torch accelerates to cut speed after the Creep Time elapses.

Cut Off Time: The CNC turns off the Cut Control output after it reads an M08 in the part program. Use the Cut Off Time to change when the CNC turns off the Cut Control input. Use a positive value to keep the Cut Control input on after the end of the cut. Use a negative value (up to one second) to turn off Cut Control before the end of the cut.

Retract Delay: Sets a time delay at the end of the cut. The Retract Delay must elapse before the torch moves to the next the next pierce.

Full Torch Up Time: Sets the time to raise the torch to the travel limit of the lifter. If you are using an automated torch height control (such as the ArcGlide THC or Sensor THC), set Full Torch Up Time to zero.

Partial Torch Up Time: Sets a shorter time interval than Full Torch Up Time to raise the torch part way along the travel distance of the lifter. Select On for Partial Raise to enable Partial Torch Up Time. If you are using an automated torch height control (such as the ArcGlide THC or Sensor THC), set Partial Torch Up Time to zero.

Stop Time: Specifies the amount of time that X/Y motion pauses at the end of a cut. This pause allows the torch to completely raise and clear any cut pieces (tip-ups) before continuing to the next cut.

Arc Off Time: Sets an interval to allow motion to continue if the arc is lost while cutting.

Retry On Transfer Fail: Sets the number of times the CNC will attempt to fire the torch in the event that the torch fails to ignite.

Transfer Time: Specifies the amount of time used to attempt ignition of the torch. The CNC confirms ignition by the Arc Sense Input (Arc on Feedback).

Set Arc Current: This is the value for the plasma arc current. Enter the amperage needed to cut the material. This value originates in the cut chart and can be temporarily fine-tuned on this screen. This parameter can be used only with plasma systems that communicate with the CNC.

Corner Current Percent: Specifies a reduced current setting when cutting corners to improve cut quality. Equals a percentage of the Set Arc Current and activates the Torch Height Disable output.

Arc on Feedback: Set Arc On Feedback to On to use the Cut Sense input. The CNC doesn't start motion until the Cut Sense input activates.

Partial Raise: Raises the torch at the end of the cut for the time specified by Partial Up Time.

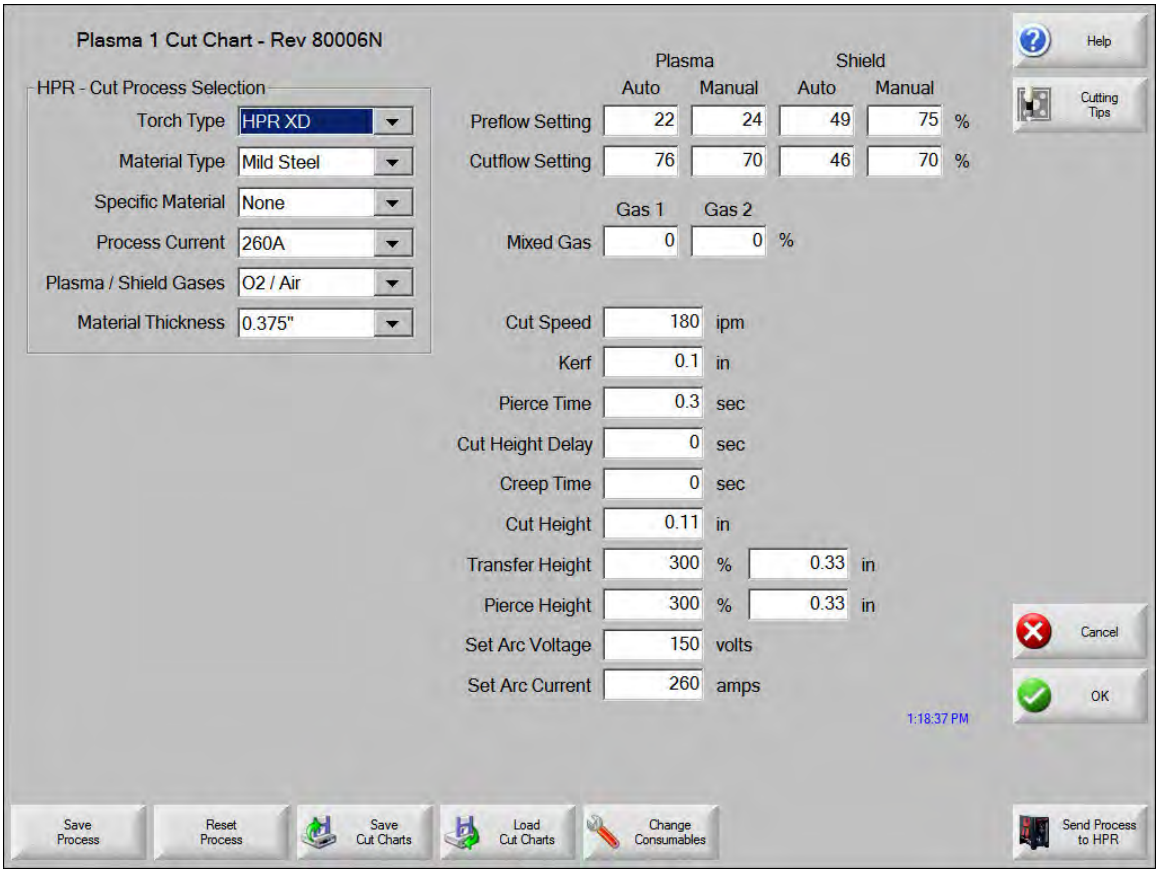
Torch Down During Cut: Forces the Torch Down output to remain on throughout the cut process.

Torch Down Between Cuts: Forces the Torch Down output to remain on while traversing between cuts.

Ignition: Enables use of the Ignition output for igniting the torch. If your plasma supply requires a separate ignition signal, set Ignition to On. If not, set Ignition Off. Typically, Hypertherm plasma supplies do not require this signal.

Plasma cut chart

Each cutting system has its own cut charts. The cut charts are provided for each cutting process: plasma, marker, laser, oxyfuel, and waterjet.



Each cut chart is based on the following process variables. Depending on the plasma supply that you have selected, other parameters may be available.

- Torch type
- Material type
- Specific material
- Process current
- Plasma and shield gases
- Material thickness

The default cut charts that are loaded into the system at the factory provide values for all remaining parameters for the cut chart and these are displayed on the right of the Cut Chart screen.

Torch Type: Select the torch being used on the cutting table, for example, HPR, HPR Bevel, HPR XD, and HPR XD Bevel. If a plasma supply has only one torch available for it, the Torch Type selection will not be available on the Cut Chart screen.

Material Type: Select the type of material for this cut chart: mild steel, stainless steel, or aluminum.

Specific Material: The Specific Material identifies a custom cut chart. For more information, see *Saving changes to a cut chart* on page 172.

Process Current: Enter the current set point for the material thickness and material type or specific material selected.

Plasma / Shield Gases: Select the gas types for the shield and for cutting.

Material Thickness: Select the thickness of the material.

The cut chart also includes the following parameters. The parameter values change depending on the process variables selected.

Preflow and Cutflow Settings: Set the plasma and shield gas preflow and cutflow. Use these settings for plasma systems that support an autogas console. For plasma systems without autogas, these values display only for reference.

Cut Speed: Sets the cutting speed (also known as feed rate) for the material type and material thickness.

Kerf: equals the width of the cut that the plasma arc, flame, laser, or waterjet removes as it cuts the material. The CNC automatically offsets the motion path by half the kerf dimension to ensure the part is cut at the correct size.

Pierce Time: Specifies the time it takes the plasma torch to pierce through the material, complete lowering, and begin motion at Creep Speed.

Cut Height Delay: Specifies, in seconds, the amount of time that the torch spends between pierce height and cut height while X and Y motion advances.

Creep Time: Specifies the amount of time that the torch travels at Creep Speed after piercing the part. Creep Speed is determined by a setup parameter at the Speeds setup screen and is a percentage of the programmed cut speed. After the creep time is completed, the control accelerates to full cut speed.

Cut Height: Determines the height at which the torch cuts the workpiece. This value can be temporarily fine-tuned on the Process screen.

Transfer Height: When the arc transfers to the workpiece, it can be “stretched” to the pierce height. The transfer height is lower than the pierce height because initiating arc transfer at a high pierce height may result in the arc not transferring to the workpiece at all. Enter the Transfer Height as a percentage of the cut height or as an actual transfer height distance.

Pierce Height: Specifies the height of the torch during pierces. This value can be entered as a percentage of the cut height or an actual pierce height distance. As common guideline is that thicker material requires a higher pierce height.

Set Arc Voltage: Enter the arc voltage for the material selected. This is part of automatic voltage control (AVC) Auto height. In general, the higher the arc voltage is set, the higher the torch will be from the plate while cutting.

Set Arc Current: This is the value for the plasma arc current. Enter the amperage needed to cut the material. This value also appears on the Process screen. This parameter can be used only with plasma systems that communicate with the CNC.

Send Process to HPR: Press the Send Process to HPR soft key to send the cut chart currently shown on screen immediately to the plasma supply. This soft key appears only when an HPR plasma supply is selected in the Station Configuration screen.

Cut charts for the HPRXD® Technology Advancements

Hypertherm has developed a range of cutting techniques designed to extend the capabilities of its existing HPRXD suite of plasma cutting systems.

- Thin stainless steel 60 A HyDefinition® inox (HDi) process (for both automatic and manual gas consoles)
- Fine Feature mild steel cut charts for 30–260 A processes (automatic gas console only)
- Underwater mild steel cut charts for 80–400 A processes (both automatic and manual gas consoles)
- 200 A bevel cut process for mild steel (both automatic and manual gas consoles)

While a few new consumables are needed for some processes, no system upgrades are required to use these cut charts.

The cut chart values are recommended to provide high quality cuts with minimal dross. Because of differences between installations and material composition, adjustments may be required to obtain desired results.

Use the following procedures to select the HPRXD cut charts. For more information refer to the *Phoenix Software V9 Series Programmer's Reference, Section 8: G59 Process Variables*.

Thin stainless steel inox (HDi)

Make the following selections to load the HDi cut chart.

1. On the Main Screen, choose the Plasma 1 or 2 Cut Chart soft key.
2. For Torch Type, choose HPRXD.
3. For Material Type, choose Stainless Steel.
4. For Specific Material, choose HDi.

To select the HDi cut chart from the part program, use one of the following codes:

- G59 V503 F2.99 – *Plasma 1 material type stainless steel, specific material HDi*
- G59 V513 F2.99 – *Plasma 2 material type stainless steel, specific material HDi*

The code for a specific material is the number following the decimal. F2.99 is for a stainless steel and .99 is for specific material HDi.

Fine Feature mild steel

Make the following selections to load the Fine Feature cut chart.

1. On the Main Screen, choose the Plasma 1 or 2 Cut Chart soft key.

2. For Torch Type, choose HPRXD.
3. For Material Type, choose Mild Steel.
4. For Specific Material, choose Fine Feature.

To select the Fine Feature cut chart from the part program, use one of the following codes:

- G59 V503 F1.97 – *Plasma 1 material type mild steel, Fine Feature specific material*
- G59 V513 F1.97 – *Plasma 2 material type mild steel, Fine Feature specific material*

The code for a specific material is the number following the decimal. F1 is for a mild steel, and .97 is for specific material Fine Feature.

Underwater mild steel

When cutting under water, be sure to disable ohmic sensing on the torch height control so that the torch height control uses stall force sensing instead to find the workpiece.

The True Hole process is not compatible with underwater cutting. If you are using a water table with the True Hole process, the water level should be at least 25 mm (1 inch) below the bottom surface of the workpiece.

Make the following selections to load the Underwater cut chart.

1. On the Main Screen, choose the Plasma 1 or 2 Cut Chart soft key.
2. For Torch Type, choose HPRXD.
3. For Material Type, choose Mild Steel.
4. For Specific Material, choose None.
5. For Cutting Surface, choose Underwater.

To select the underwater cut chart from the part program, use one of the following codes:

- G59 V506 F2 – *Plasma 1 cutting surface, 75 mm (3 inches) below water*
- G59 V516 F2 – *Plasma 2 cutting surface, 75 mm (3 inches) below water*

200 A bevel mild steel

Make the following selections to load the 200 A bevel cut chart.

1. On the Main Screen, choose the Plasma 1 or 2 Cut Chart soft key.
2. For Torch Type, choose HPRXD Bevel.
3. For Material Type, choose Mild Steel.
4. For Specific Material, choose None.
5. For Process Current, choose 200 A.

To select the 200 A bevel mild steel cut chart from the part program, use the following codes.

- G59 V502 F35 – *Plasma 1 torch type HPRXD Bevel*
- G59 V503 F2 – *Plasma 1 material type mild steel; no specific material*
- G59 V504 F200 – *Plasma 1 process current 200 A*

or

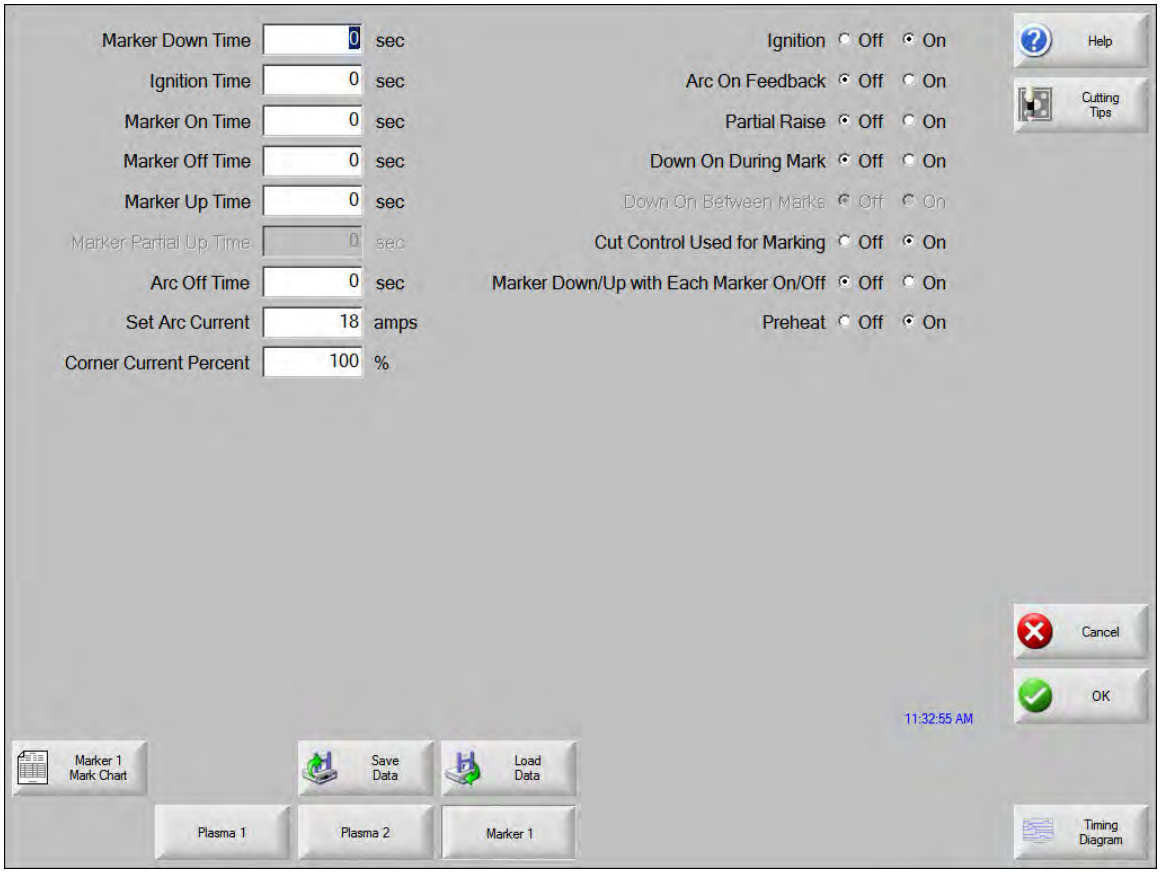
- G59 V512 F35 – *Plasma 2 torch type HPRXD Bevel*
- G59 V513 F2 – *Plasma 2 material type mild steel; no specific material*
- G59 V514 F200 – *Plasma 2 process current 200 A*

Marker process

Marker 1 and Marker 2 process screens

The CNC can support up to two markers on the cutting system. A marking tool is typically installed on the cutting system along with another cutting tool.

The screen below shows the Marker 1 process settings. From the Main screen, choose Setups > Process > Marker 1 or Marker 2 to open the screen.



The Marker 1 and Marker 2 Process screens provide different options for the type of torch height control being used.

When using a torch height control other than one manufactured by Hypertherm, the Marker 1 and Marker 2 screens look like the one above. These screens provide a set of timers used to control outputs or motion of the lifter. The timers and motion start after the CNC reads an M09 or M13 in the part program.

For information on the Process screens for Hypertherm torch height controls, see *Torch Height Controls* on page 175.

Running a marker process

To run the Marker 1 or Marker 2 process, the part program must contain the M36 T3 or M36 T4 process selection code. This code works like the Cut Mode parameter on the Main and Cutting screens. Marker part programs require this code because marker processes are not available in the Cut Mode selection.

In addition:

- A Marker Offset is set on the Cutting screen so the CNC can position the marker and then reposition the other cutting tool.
- Because a marking tool is always used with another tool, you will need to use numbered I/O to control the marker.

A marker font is available on the CNC. See the *Phoenix Software V9 Series Programmer's Reference* for more information.

Marker Down Time: Sets the length of time to lower the marking tool at the beginning of each mark. Activates the Torch Down output.

Ignition Time: Sets the length of time activate the Ignition output at each ignition point.

Marker On Time: Sets a time delay which before starting motion.

Marker Off Time: Sets a time delay before motion ends.

Marker Up Time: Activates the Torch Up output. Sets the time to raise the marking tool to the travel limit of the lifter.

Marker Partial Up Time: Activates the Torch Up output. Select On for Partial Raise to enable Marker Partial Up Time. Sets the time to raise the marking tool part way along the travel distance of the lifter.

Set Arc Current: This is the value for the plasma arc current. Enter the amperage needed to cut the material. This value originates in the cut chart and can be temporarily fine-tuned on this screen. This parameter can be used only with plasma systems that communicate with the CNC.

Corner Current Percent: Specifies a reduced current setting when cutting corners to improve cut quality. Equals a percentage of the Set Arc Current and activates when the Torch Height Disable output is on.

Ignition: Enables use of the Ignition output for lighting the torch. If your plasma supply requires a separate ignition signal, set Ignition to On. If not, set Ignition Off.

Arc On Feedback: Set Arc On Feedback to On to use the Cut/Mark Sense input. The CNC doesn't start motion until Cut/Mark Sense input activates.

Partial Raise: Raises the marking tool at the end of the mark for the time specified by Marker Partial Up Time.

Down On During Mark: Forces the Torch Down output to remain on throughout the marking process.

Down On Between Marks: Forces the Torch Down output to remain on while traversing between marking segments.

Cut Control Used for Marking: The CNC uses the Cut Control output to activate the marking tool. Set to On to use the Cut Control output. Set to Off to use the Marker Control output.

Marker Down/Up With Each Marker On/Off: Marker On/Off refers to these part program codes:

- M09 Marker 1 Enable and M10 Marker 1 Disable
- M13 Marker 2 Enable and M14 Marker 2 Disable

When the CNC reads the M09 in part program, it turns on the Torch Down output, and when it reads the M10 code it turns on the Torch Up output.

Preheat: When marking with plasma, turn off Preheat. This output is usually used with oxyfuel or zinc markers.

Marker cut chart

The CNC includes marker cut charts for the plasma supplies and the Arc Writer as listed in the Station Configuration screen.

Not all plasma systems support marking.

Material Type: Select the type of material for this cut chart: mild steel, stainless steel, or aluminum.

Specific Material: The Specific Material identifies a custom cut chart. For more information, see *Saving changes to a cut chart* on page 172.

Process Current: Enter the current set point for the material thickness and material type or specific material selected.

Marking Speed: Sets the marking speed (also known as feed rate) for the material type and material thickness.

Mark Width: Set Mark Width to zero. The width value is used only if you insert a G41 or G42 code to offset the width value.

Plasma / Shield Gases: Select the gas types for the shield and for cutting.

Cutflow Setting: Set the plasma and shield gas cutflow percentages for the process.

Marking Height: Sets the height above the workpiece to position the marker.

Start Height: Enter a percentage of the Marking Height to position the marker above the marking height before the start of the mark.

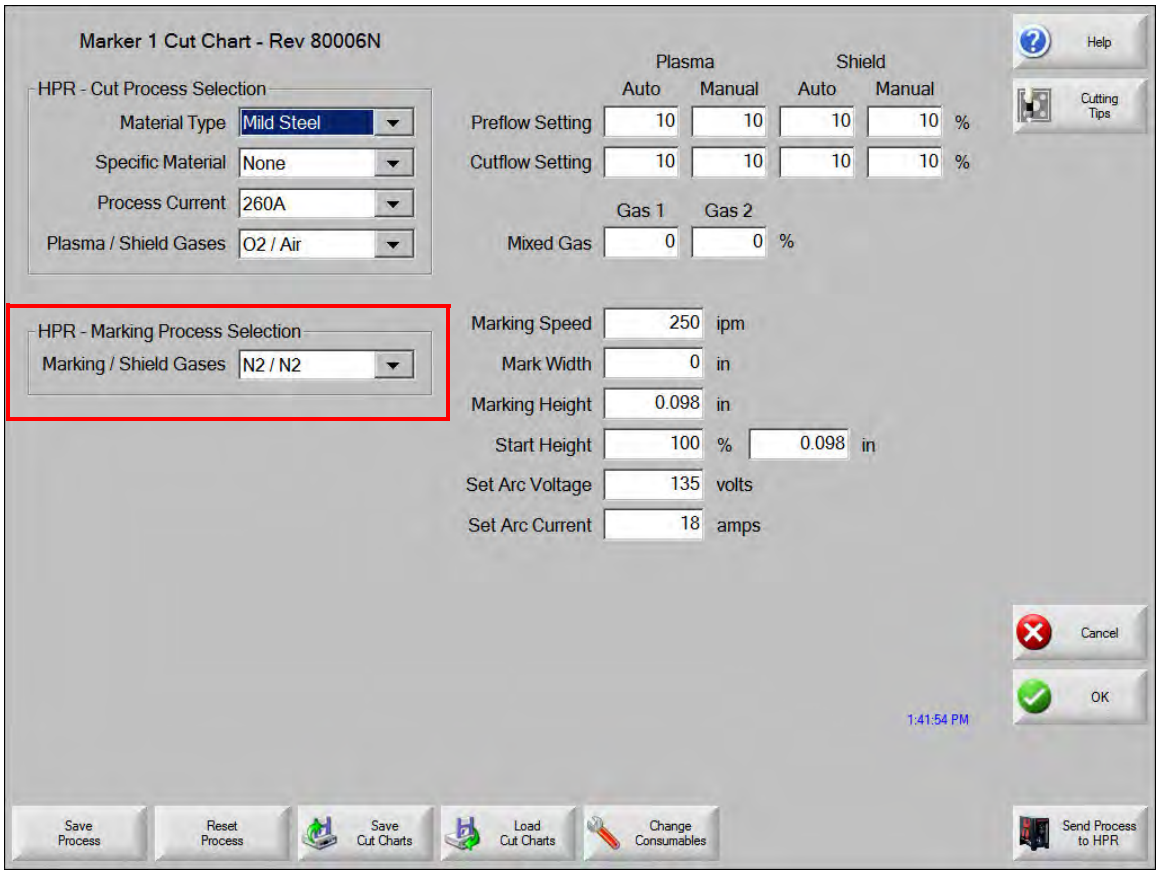
Set Arc Voltage: Enter the arc voltage for the material selected. This is part of automatic voltage control (AVC) Auto height. In general, the higher the arc voltage is set, the higher the torch will be from the plate while cutting.

Set Arc Current: This is the value for the plasma arc current. Enter the amperage needed to cut the material. This value also appears on the Process screen. This parameter can be used only with plasma systems that communicate with the CNC.

Using cut consumables for marking

For HPR and HPRXD plasma supplies, you can use the same consumables for cutting and marking. The CNC assumes that when you have the same plasma system selected for both a plasma and marker process that the same consumables will be used.

When you open the Marker cut chart, the same process variables are displayed under HPR – Cut Process Selection. You cannot change the Cut Process Selection variables for the marking process, but you can change the Marking/Shield gases and other marking parameters as needed for the marking process.



Oxyfuel process

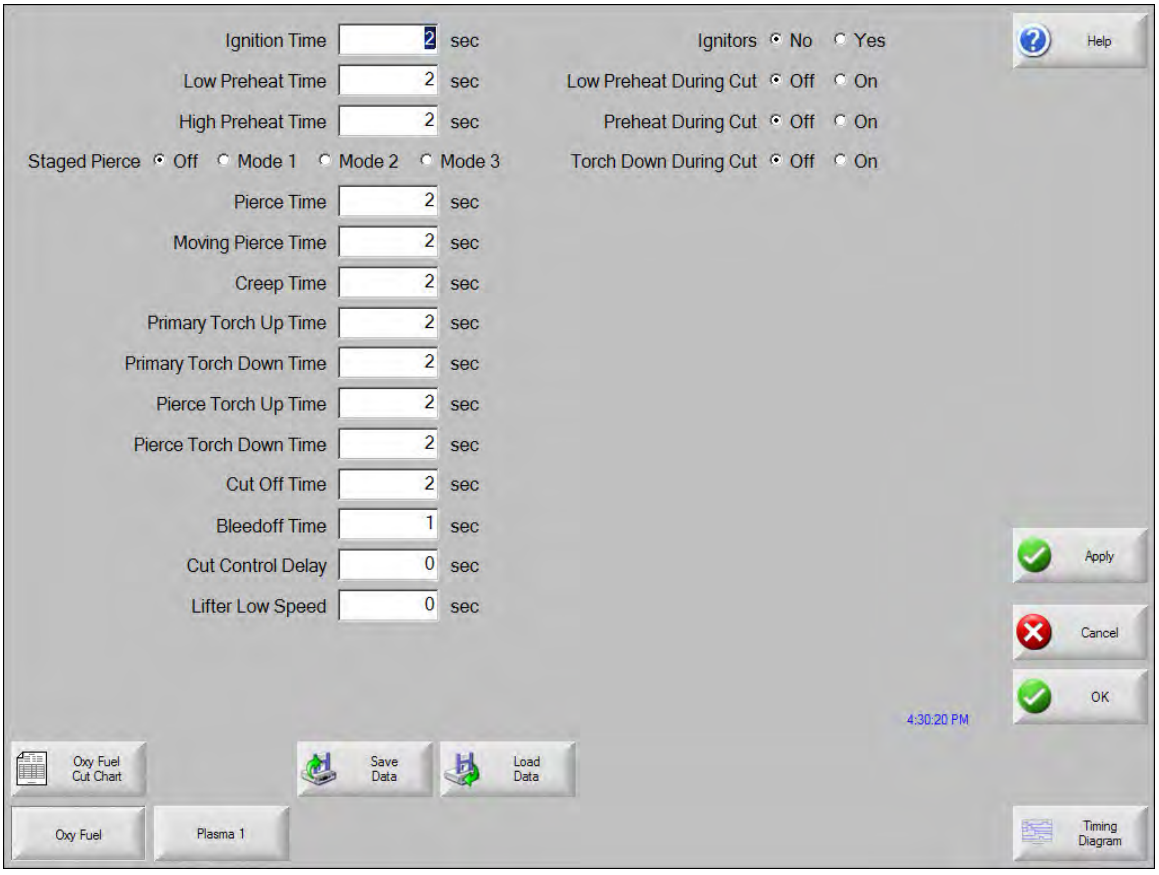
The CNC provides an oxyfuel process that operates three gas channels for one or multiple torches. The CNC controls the timing of the torch ignition, piercing, and torch up and down motion before during and after each.

Oxyfuel cutting with multiple torches is controlled with numbered I/O on the Setups > Password > Machine Setups > I/O screen. Use the generic (non-numbered I/O) for a single torch oxyfuel cutting system. Analog outputs, located on the same screen, can be used to control a gas console, although doing so requires a SERCOS interface. For more information on oxyfuel system setup, see *Oxyfuel Application* in the *Phoenix V9 Series Setup and Installation Manual*.

To enable oxyfuel process and cut chart screens, choose Setups > Password > Special Setups and select oxyfuel under Tools Installed. The CNC does not support oxyfuel in the Station Configuration screen.

Oxyfuel process screen

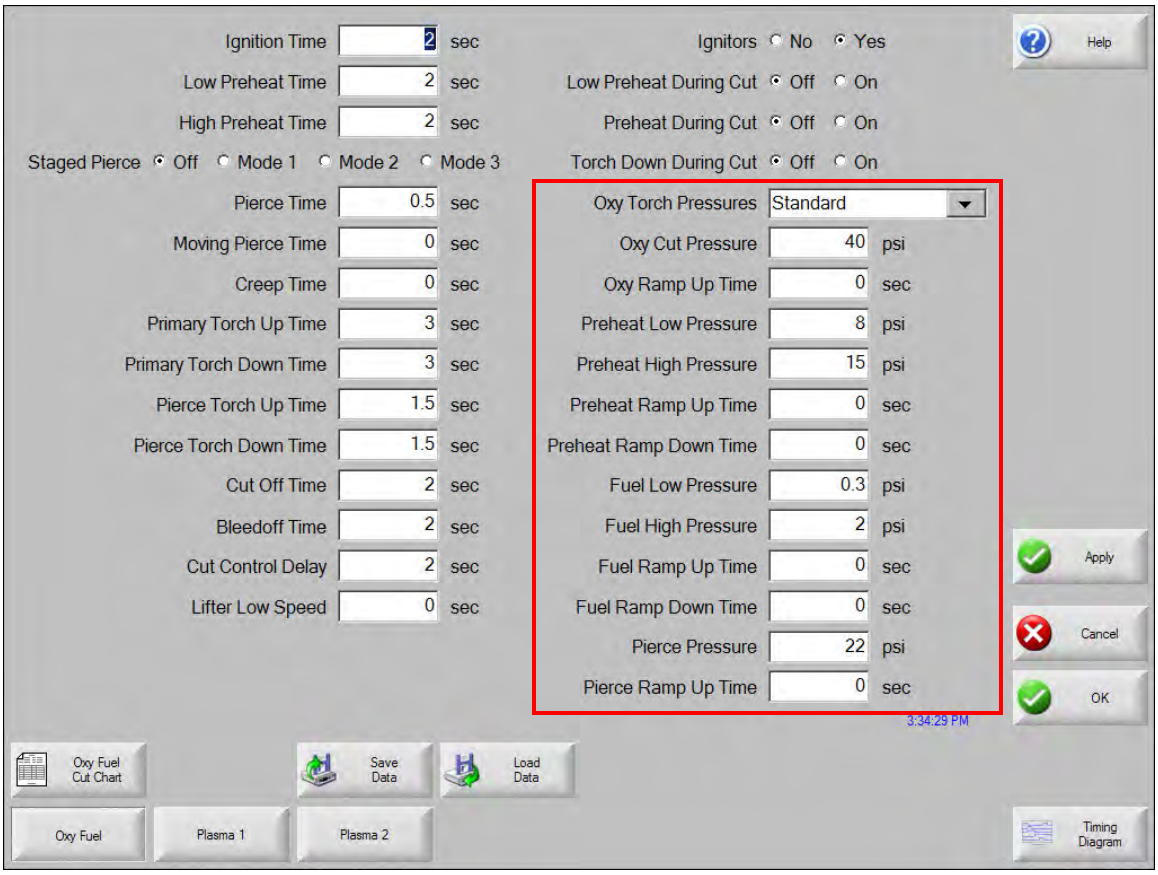
The CNC provides a Process screen for oxyfuel cutting. In this screen you can set timers for the outputs that control preheating the torch and workpiece, torch heights for piercing and cutting, and movement of the torch between cuts.



Parameter	Output controlled	Description
Ignition Time	Torch ignition	Specifies the time the oxyfuel igniter is held on at each ignition of the flame.
Low Preheat Time	Low preheat control	Turns on the low preheat gas valve. Low preheat time can be used during torch ignition.
High Preheat Time	High preheat control	Turns on the high preheat gas valve. Set the time for preheating the workpiece before piercing.
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.		
Staged Pierce	Staged pierce 1 – 4	Performs the pierce in a timed progression of outputs that can control oxygen pressure. When you select this option, the Staged Pierce Time 1 – 3 parameters replace the Pierce Time, Moving Pierce Time, and Creep Time parameters. Set the timer for each staged pierce output.
Pierce Time	Pierce control	Sets the amount of time the pierce control output is on before lowering the torch to the cut height.
Moving Pierce Time	Pierce control	Sets the amount of time that the pierce control output remains on and allows X/Y motion while piercing.
Creep time	none	Sets the amount of time after piercing the workpiece that the torch travels at Creep Speed. (See Setups > Machine Setups > Speeds for the Creep Speed.) The torch accelerates to cut speed after the Creep Time elapses.
Primary Torch Up Time	Torch Up	Sets the time to raise the torch after completing each cut. The torch continues raising until this time expires or the lifter reaches a limit switch that activates the Torch Up Sense input.
Primary Torch Down Time	Torch Down	Sets the time 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 Up Time	Torch Up	Sets the time for torch lift after piercing to clear a pierce puddle.
Pierce Torch Down Time	Torch Down	Sets the time to lower the torch for cutting. This timer should allow the torch to reach the cut height.
Cut Off Time	Cut Control	Sets the time 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.
Bleed-off Time	Bleed-off Gas	Sets the time 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.
Cut Control Delay	Cut Control	Sets the time the CNC waits before activating the Cut Control output during piercing.
Lifter Low Speed	Lifter Low Speed	Works only with multiple torch systems. This timer turns on with the Torch Up and Torch Down outputs and shuts off after the Lifter Low Speed time elapses. The value for the Lifter Low Speed timer should be less than the value for the Primary Torch Up Time and Primary Torch Down Time values.

Parameter	Output controlled	Description
Ignitors	Low Preheat Control	Set Ignitors to No and Ignition Time to 0 to activate the Low Preheat Control output at the end of a cut. 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	Low Preheat Control	Specifies whether the Low Preheat is left on during cutting.
Preheat During Cut	High Preheat Control	Specifies whether the Preheat is left on during cutting.
Torch Down During Cut	Torch Down	Specifies whether Torch Down is left on during cutting. Use this parameter for a pneumatic lifter.

Depending on your oxyfuel cutting system, you may have additional parameters on the Oxyfuel Process screen. These parameters activate when you set up analog outputs to control valves on a gas console. See *Oxyfuel Application* in the *Phoenix V9 Series Installation and Setup Manual* for information on using analog outputs to control gas valves in an oxyfuel cutting system.



The gas pressure settings in the process screen inherit the values in the oxyfuel cut chart. You can set timers for the gas valves to allow time for the gas pressure to ramp up to the preheat or cutting pressure. Use the Apply button to adjust the settings and test them on your system without exiting the Oxyfuel Process screen.

Oxy Torch Pressures: Select the type of oxyfuel torch for the process. These torches correspond to the analog output settings on the Machine Setups > I/O screen.

- Standard Triple Bevel 2
- Triple Bevel 3
- Triple Bevel Preheat

Oxy Cut Pressure: Enter the pressure, in bars (pounds per square inch), of the oxygen during cutting.

Oxy Ramp Up Time: Enter the time, in seconds, for the oxygen to reach cutting pressure.

Preheat Low Pressure: Enter the pressure of the oxygen at low pressure during preheat.

Preheat High Pressure: Enter the pressure of the oxygen at high pressure during preheat.

Preheat Pressure: Enter the preheat pressure for the triple bevel torch.

Preheat Ramp Up Time: Enter the number of seconds that the process takes to move from low to high pressure during preheat.

Preheat Ramp Down Time: Enter the number of seconds that the process takes to move from high to low pressure during preheat.

Fuel Low Pressure: Enter the pressure of the oxyfuel at low pressure during cutting.

Fuel High Pressure: Enter the pressure of the fuel gas at high pressure during cutting.

Fuel Pressure: Enter the fuel pressure for the triple bevel head.

Fuel Ramp Up Time: Enter the number of seconds that the process takes to move from low to high pressure during cutting.

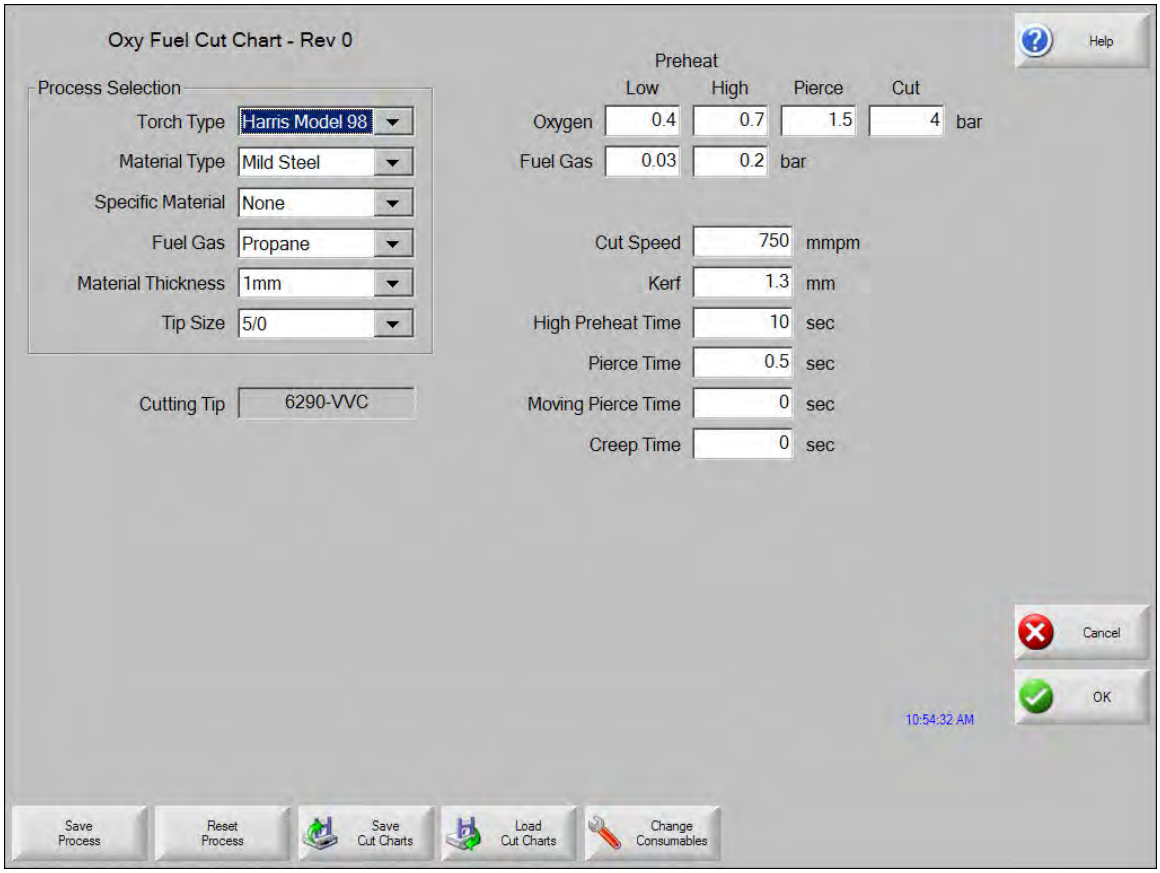
Fuel Ramp Down Time: Enter the number of seconds that the process takes to move from high to low pressure during cutting.

Pierce Pressure: Enter the pressure of the fuel gas during a pierce.

Pierce Ramp Up Time: Enter the number of seconds that the process takes to move to pierce pressure.

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, and the material type and thickness.



If your cutting system is using gas valves on analog outputs, the gas pressures in the cut chart are transferred to the Oxyfuel Process screen.

Torch Type: Select the name of the torch on your cutting system.

Material Type: Displays the type of material for this cut chart: mild steel, stainless steel, or aluminum.

Specific Material: The Specific Material identifies a custom cut chart. For more information, see *Saving changes to a cut chart* on page 172.

Fuel Gas: Shows the fuel gas for the process.

Material Thickness: Shows the workpiece thickness for the cut chart. Select a different material thickness to change the cut chart.

Tip Size: Shows the tip size needed for the torch. Select a different tip size to change the cut chart. The cutting tip part number displays below the Tip Size.

Cutting tip: Displays the model of cutting tip.

Oxygen and **Fuel Gas:** Each sets the gas pressures for preheating and cutting.

Cut speed: Sets the cutting speed (also known as feed rate) for the material type and material thickness.

Kerf: equals the width of the cut that the plasma arc, flame, laser, or waterjet removes as it cuts the material. The CNC automatically offsets the motion path by half the kerf dimension to ensure the part is cut at the correct size.

High Preheat Time: Turns on the high preheat gas valve. Set this time for preheating the workpiece before piercing. When you run the part, you can use the Set, Extend, or Release soft keys to change the preheat time.

Pierce Time: Sets the amount of time the pierce control output is on before lowering the torch to the cut height.

Moving Pierce Time: Sets the amount of time that the pierce control output remains on and allows X/Y motion while piercing. The moving pierce allows the molten material from the pierce to be ejected behind the torch.

Creep Time: Specifies the amount of time after piercing the part that the torch travels at Creep Speed, a percentage of the cut speed set on the Machine Setups > Speeds screen. After the creep time is elapses, the CNC accelerates to full cut speed.

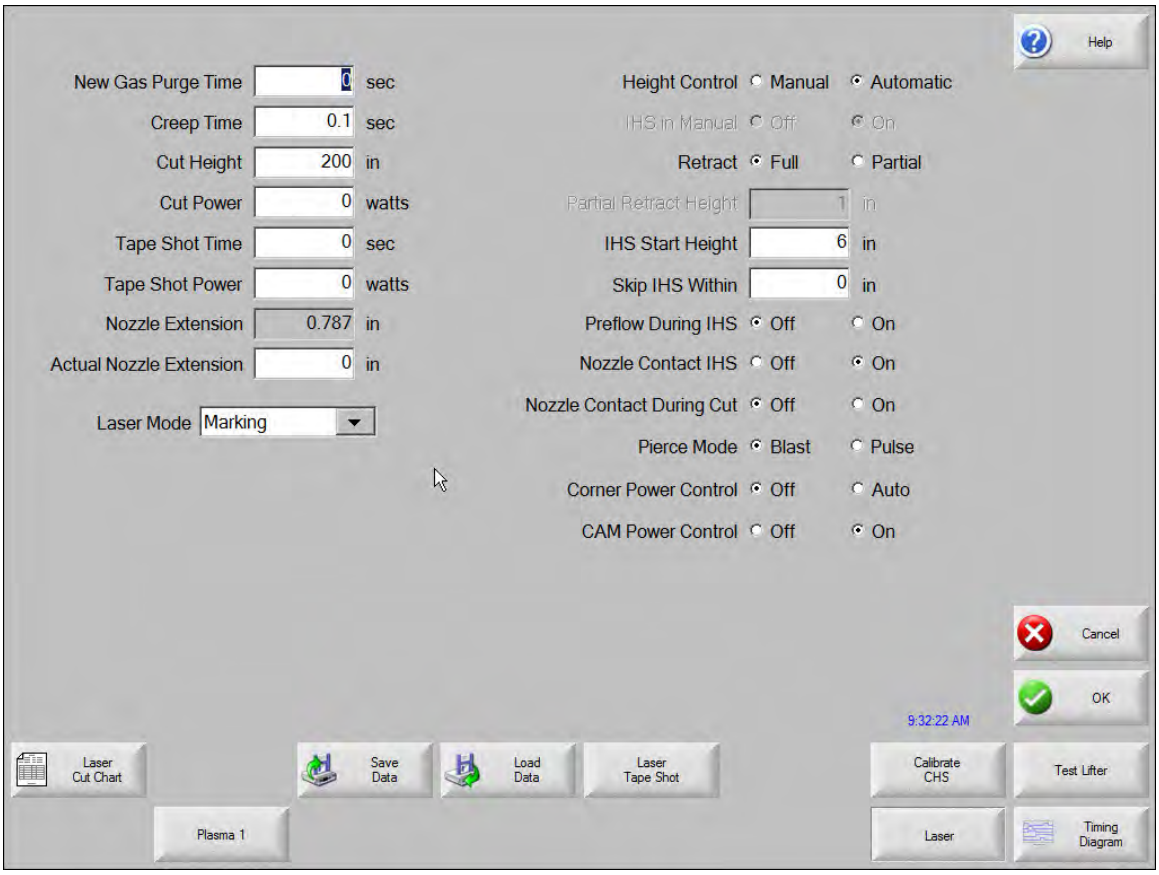
Fiber Laser process

Hypertherm CNCs support Hypertherm's HyIntensity Fiber Lasers® for cutting mild steel, stainless steel, aluminum and other materials. The CNC provides a unique process screen and cut chart screen for the fiber lasers.

Set up your fiber laser system within the CNC as described in *Before you begin* on page 125 and complete the installation and connections described in the laser system instruction manual.

Fiber laser process screen

The Fiber Laser Process screen lets you fine-tune the cutting process.



New Gas Purge Time: Sets the purge time in seconds when performing the first cut after power up and when switching from one cutting gas to another cutting gas. Set the purge time to a long enough interval to allow any impurities or previous cutting gasses to be cleared from the system before starting a new cutting process.

Creep Time: Specifies the amount of time after piercing the material that the laser head travels at Creep Speed for cutting. Creep Speed is determined by a setup parameter in the Speed Setup Screen and is a percentage of the programmed cut speed. After the Creep Time is completed, the control accelerates to full cut speed.

Cut Height: Sets the position of the laser nozzle above the workpiece.

Cut Power: Displays the laser power, in watts, for the job. This value originates in the cut chart. You can change the cut power for the current job in this screen.

Tape Shot Time: Sets the laser pulse time duration for a tape shot beam alignment.

Tape Shot Power: Sets the laser pulse power for a tape shot beam alignment.

Nozzle Extension: Displays the recommended distance between the nozzle and the lens for best results for the material and thickness.

Actual Nozzle Extension The Fiber Laser continuously monitors the actual nozzle extension of the Fiber laser cutting head and communicates this information to the CNC. If the actual nozzle extension distance varies more than 1 mm (above or below) the Nozzle Extension setting in the current cut chart, the CNC will display the Actual Nozzle Extension in red to show the operator that the Actual Nozzle Extension may not be set correctly.

Laser Mode: Select one of four laser modes from the cut chart: Cutting, Marking, Vaporization, or Fine Feature. See *Marking, Vaporization, and Fine Feature modes* on page 154 for more information.

Height Control Manual/Automatic: Select the type of height control for your cutting system. Choose Automatic for a Sensor THC height control.

IHS in Manual: If the cutting system has a manual height control, use the Initial Height Sense feature when operating the lifter in manual mode.

Retract Full/Partial: Selects the retract distance to be set at Full or Partial. In the Full retract mode, the laser head retracts to the Z-Axis Home position. In Partial retract mode, the laser head retracts to the Partial Retract Height.

IHS Start Height: Specifies the distance of travel for the height control to move the laser head at high speed before switching to low speed and beginning Initial Height Sense. Take caution when selecting this distance so that the laser head does not crash into the plate.

Skip IHS Within: Disables initial height sense at pierce points if IHS falls within the selected distance. This setting increases cutting productivity. The distance is measured from the end point of the cut segment to the next pierce point.

Preflow During IHS: Activates the preflow gases when the cutting system is performing initial height sense.

Nozzle Contact IHS: Select to use nozzle contact instead of capacitive height sense to detect the workpiece during initial height sense (the nozzle touches the workpiece).

Nozzle Contact During Cut: Uses the Nozzle Contact Sense input to detect contact with the workpiece during a cut.

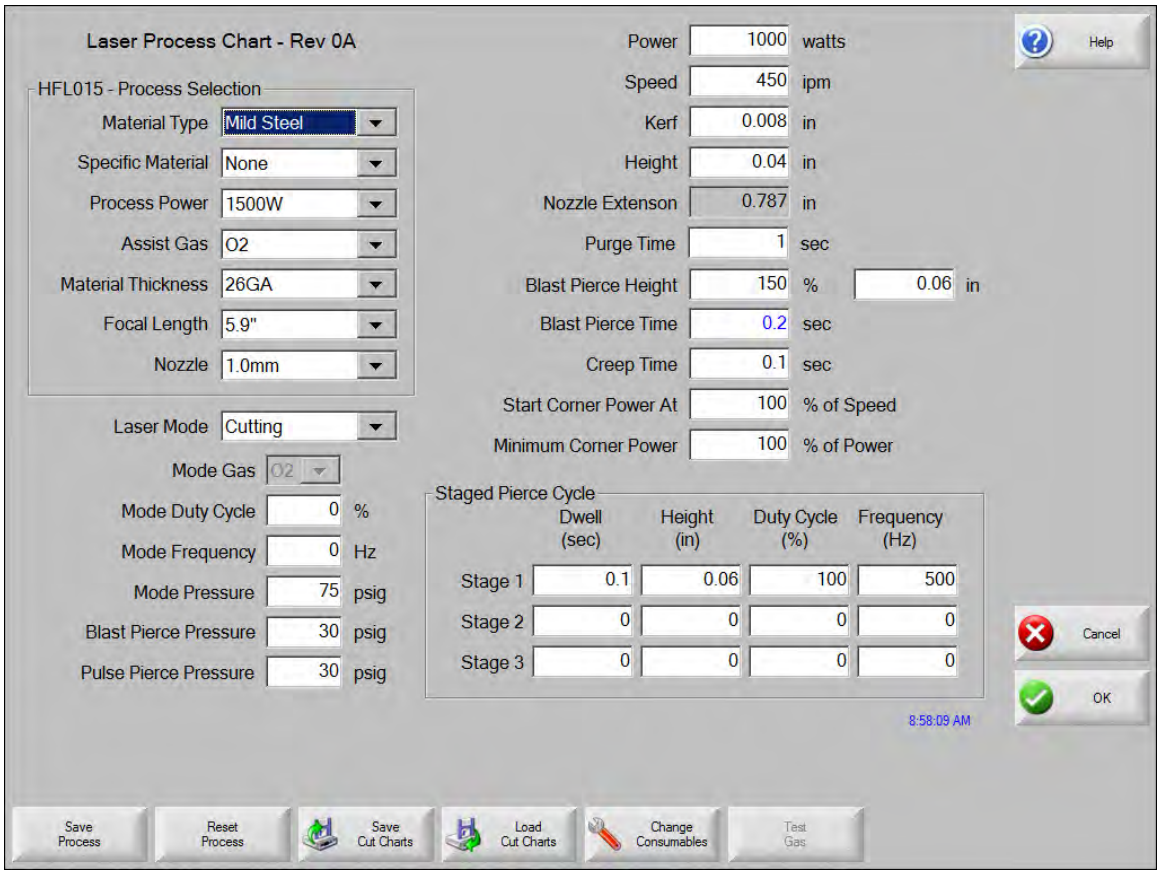
Pierce Mode Pulse/Blast: Select the type of piercing for the job. Pulse turns the beam on and off for a percentage of the pierce duty cycle. Use pulse to “peck” through the workpiece. Pulse can create a cleaner hole. If you use the Stage Pierce Cycles available in the Fiber laser cut chart, select Pulse for Pierce Mode.

Blast turns on the beam continuously. However, if piercing a thick workpiece, the Blast Pierce Mode can result in some spatter which could come into contact with the nozzle.

Corner Power Control: Set to Auto to reduce the laser power when cutting a corner on a part. Set to Off to cut the corner at the full programmed power as set in the cut chart.

CAM Power Control: Turns on and off the ability to use a V810 code in the part program to enable changing Duty Cycle (V808), and Modulation Rate (V809). See the *Phoenix Software V9 Series Programmer’s Reference* for more information on program codes used for laser cutting.

Fiber laser cut chart



The laser cut chart is based on the following process variables:

Material Type: Select the material type, such as mild steel, stainless steel, aluminum, brass, or copper.

Specific Material: The Specific Material identifies a custom cut chart. For more information, see *Saving changes to a cut chart* on page 172.

Process Power: Select the appropriate process power (wattage) for the material thickness and material type.

Assist Gas: Select the appropriate assist gas for the desired process.

Material Thickness: Select the material thickness for the material type.

Focal Length: Select the specific focal length lens that must be installed in the laser head.

Nozzle: Select the diameter and type of nozzle that must be installed for the process.

The cut chart also includes the following parameters. The parameter values change depending on the process variables selected.

Laser Mode: Select one of four laser modes from the cut chart: Cutting, Marking, Vaporization, or Fine Feature. See *Marking, Vaporization, and Fine Feature modes* on page 154 for more information.

Mode Gas: Activates for Marking and Vaporization modes, and is display-only for Cutting and Fine Feature modes. For Marking and Vaporization Modes, choose from N_2, O_2, or Air. Cutting and Fine Feature modes use the Assist Gas for the Mode Gas.

Mode Duty Cycle: When pulsing, the mode duty cycle equals the percentage of time the laser is ON. It also equals a percentage of the Cut Power. For example, when the Cut Power equals 2000 watts and the duty cycle is 50%, then the Fiber laser will cut at 1000 watts. The Cut Power is multiplied by the duty cycle, for example 50% or 0.50 x 2000 watts equals 1000 watts).

Mode Frequency: Equals the cycles per second the laser will pulse at the power level.

Mode Pressure: Shows the gas pressure for selected mode.

Blast / Pulse Pierce Pressure: Shows gas pressure values for blast or pulse piercing. Select the pierce mode on the Fiber Laser Process screen.

Power: Sets the power (watts) to be used during the cut process. This value can be less than the process power.

Speed: Specifies the speed for the selected mode.

Kerf: Equals the width of the cut that the plasma arc, flame, laser, or waterjet removes as it cuts the material. The CNC automatically offsets the motion path by half the kerf dimension to ensure the part is cut at the correct size.

Height: Sets the cut distance from the nozzle tip to the plate. Height is derived from the CHS signal and the calibration curve.

Nozzle extension: Displays the recommended distance between the nozzle and the lens for best results for the material and thickness.

Purge Time: Specifies the time delay from switching from one cutting gas type to another cutting gas type.

Blast Pierce Height: Equals a percentage of the Cut Height. Since blast pierce can cause molten metal to spatter, set the Blast Pierce Height to several times the Cut Height to keep spatter from coming into contact with the nozzle.

Blast Pierce Time: Sets the length of time for the blast pierce.

Creep Time: Specifies the period after piercing completes that the laser head travels at Creep Speed. Creep Speed is determined by a setup parameter in the Speeds setup screen and is a percentage of the programmed cut speed. After Creep Time is complete, the control accelerates to full cut speed.

Start Corner Power: defines a speed where the corner power analog signal will be used to start to decrease laser power. This is defined as a percentage of cut speed.

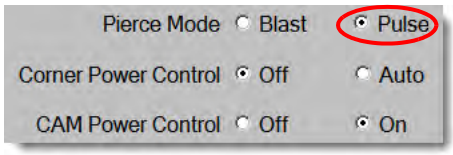
Minimum Corner Power: defines the minimum laser power the CNC will command when cutting through a corner. This is defined as a percentage of selected power (watts).

Setting up staged pierce

The Fiber Laser cut charts contain a multiple-stage pierce cycle. Using staged pierce allows the laser to create a small diameter pierce through thick materials. Staged pierce can be set up only through the cut chart on the CNC. G59 process codes cannot be used to select a staged pierce cycle. Staged pierce values are provided in the cut chart for materials 11 mm (1/2 inches) and thicker.

Follow these steps to set up a staged pierce cycle:

1. Choose Setups > Process to open the Fiber Laser Process screen.
2. Choose Pulse for Pierce Mode.



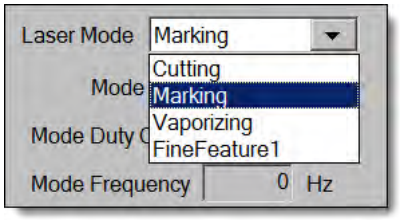
3. Choose Ok to save the change in the Fiber Laser Process screen.
4. Choose the Laser Cut Chart soft key on the Main screen to view the Staged Pierce Cycle parameters for the cut chart. The following example shows the staged pierce cycle from a 2000W cutting process.

- Each stage pierces for the **Dwell** time at the specified height.
- **Duty cycle** is a percentage of the Cut Power. For example, when the Cut Power equals 2000 watts and the duty cycle is 50%, then the Fiber laser will cut at 1000 watts (Cut Power multiplied by the duty cycle).
- **Frequency** equals the cycles per second the laser will pulse at the Staged Pierce power level.
- In the above example, the Fiber Laser will pulse at 1000 watts 500 times per second for 3 seconds at 7 mm above the workpiece for Stage 1.

Marking, Vaporization, and Fine Feature modes

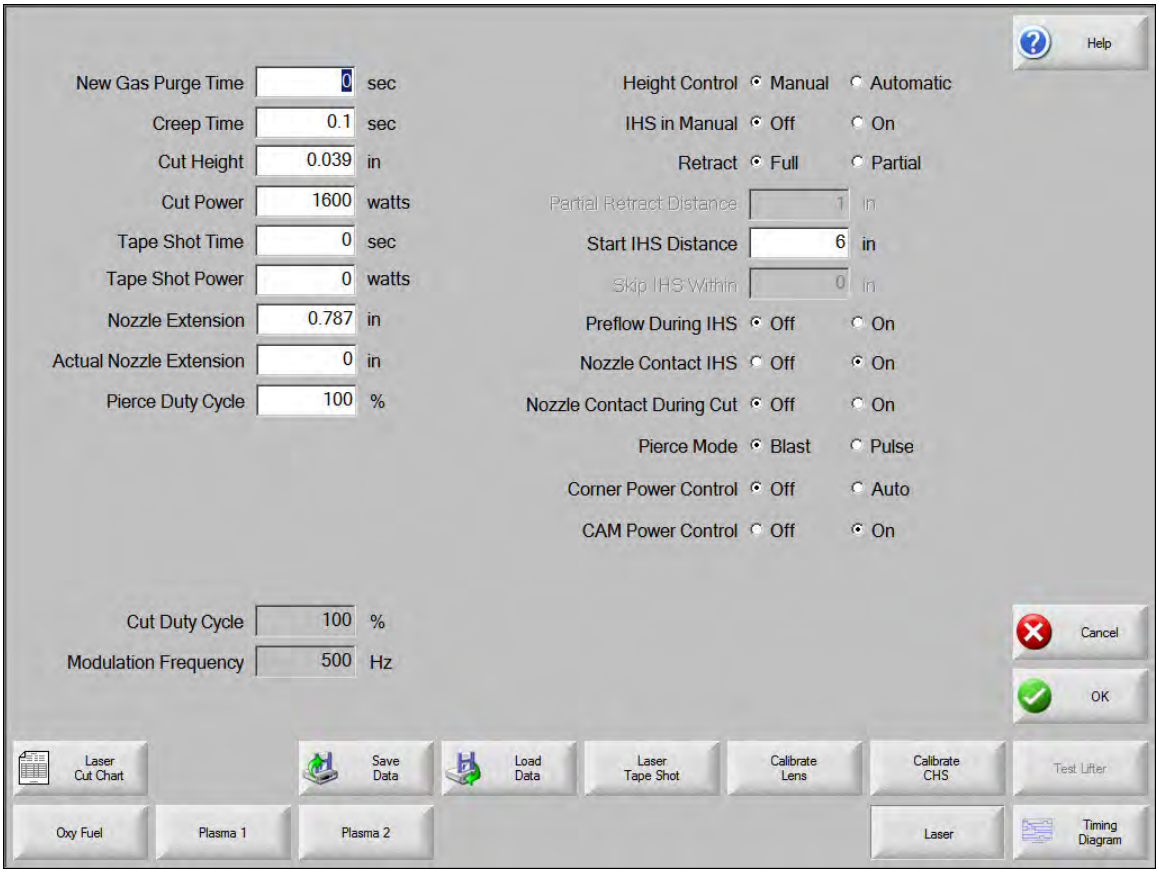
The Fiber Laser cut charts support marking, vaporization, and Fine Feature processes through the Laser Mode selection in the Cut Chart and Process screens. A part program using G59 process variables can change the Laser Mode as needed. If changing the Laser Mode from the Cut Chart or Process Screen, then only one mode can be used for the entire part program. See the *Phoenix Software V9 Series Programmer's Reference* for information on G59 process variables.

- Marking process uses lower cutting power to score the material surface. You can also use the M09 and M10 codes to turn marking on and off from the part program.
- Vaporization uses low power to remove a protective coating, such as plastic or oil, from the material surface before cutting. Vaporization can also be used to pre-treat rusty or scaled material to improve cutting consistency of these materials. For this application, you would run the part program once in Vaporization mode, then a second time in Cutting mode.
- Marking and vaporization processes do not require piercing. The Phoenix software now pre-sets the pierce parameters in the Fiber Laser cut chart to the following values:
  - Pierce Height: 100% of the Cut Height
  - Pierce Time: 0
  - Creep Time: 0
- Fine Feature uses low-frequency pulsing and low cut speed for part features that are smaller than the material thickness, or for sharp corners.



Laser process (non-fiber laser)

The options available on the Laser Process screen will vary depending on the laser system. This section describes all of the options even though some of them will not be available for your system.



Purge Time: Sets the time delay for cutting gas purge before motion starts.

New Gas Purge Time: Sets the purge time when switching from one cutting gas to another cutting gas.

Shutter Time: Sets the amount of time to open the shutter before the laser beam turns on.

Power Ramp Time: Sets the amount of time to ramp up the laser power prior to the laser pierce.

Pierce Time: Sets the time delay from when laser head completes lowering until motion is initiated at creep speed for cutting.

When Automatic Pierce Control is selected, this time is an additional delay after pierce is complete.

Pulse On Time/Pulse Off Time: When Automatic is selected for pierce control you can select Pulse On and Off Time to adjust duty cycle response of the sensor pulses from the laser cutting head.

Creep Time: Specifies the amount of time after piercing the material that the laser head travels at Creep Speed for cutting. Creep Speed is determined by a setup parameter in the Speed Setup Screen and is a percentage of the programmed cut speed. After the Creep Time is completed, the control accelerates to full cut speed.

Beam Off Time: Sets the time the beam output will be turned off before motion stops. Use this feature to create tabs on parts to keep them attached to the scrap material.

Postflow Time: Sets the time the cutting gas remains on after the cut is complete.

Cut Height: Sets the position of the laser nozzle above the workpiece.

Pierce Height: Sets the height of the nozzle for piercing. Enter a distance or a percentage of the Cut Height.

Lens Cut Position: Sets the focal lens position in the laser head for cutting.

Lens Pierce Position: Sets the focal lens pierce position in the laser head for cutting.

Laser Cut Power: Displays the laser power, in watts, for the job. This value originates in the cut chart. You can change the cut power for the current job in this screen.

Height Control Manual/Automatic: Select the type of height control for your cutting system. Choose Automatic for a Sensor THC height control.

IHS in Manual: If the cutting system has a manual height control, use the Initial Height Sense feature when operating the lifter in manual mode.

Retract Full/Partial: Selects the retract distance to be set at Full or Partial. In the Full retract mode, the laser head retracts to the Z-Axis Home position. In Partial retract mode, the laser head retracts to the Partial Retract Distance.

IHS Start Height: Specifies the distance of travel for the height control to move the laser head at high speed before switching to low speed and beginning Initial Height Sense.

Skip IHS Within: Disables initial height sense at pierce points if IHS falls within the selected distance. This setting increases cutting productivity. The distance is measured from the end point of the cut segment to the next pierce point.

Preflow During IHS: Activates the preflow gases when the cutting system is performing initial height sense.

Nozzle Contact IHS: Select to use nozzle contact (instead of capacitive height sense) to detect the workpiece during initial height sense (the nozzle touches the workpiece).

Nozzle Contact During Cut: Uses the Nozzle Contact Sense input to detect contact with the workpiece during a cut.

Pierce Control: Automatic pierce control uses sensors in the laser head to detect when the pierce is complete. Manual pierce control uses a preset pierce time and preset laser program.

Pierce Mode: Select the type of piercing for the job. Pulse turns the beam on and off for a percentage of the pierce duty cycle. Use pulse to “peck” through the workpiece. Pulse can create a cleaner hole.

Blast turns on the beam continuously. However, if piercing a thick workpiece, the Blast Pierce Mode can result in some spatter.

Pierce Complete: The Automatic pierce control monitors the voltage of sensors in the laser head and compares them to the value set by this parameter to detect the completion of the pierce.

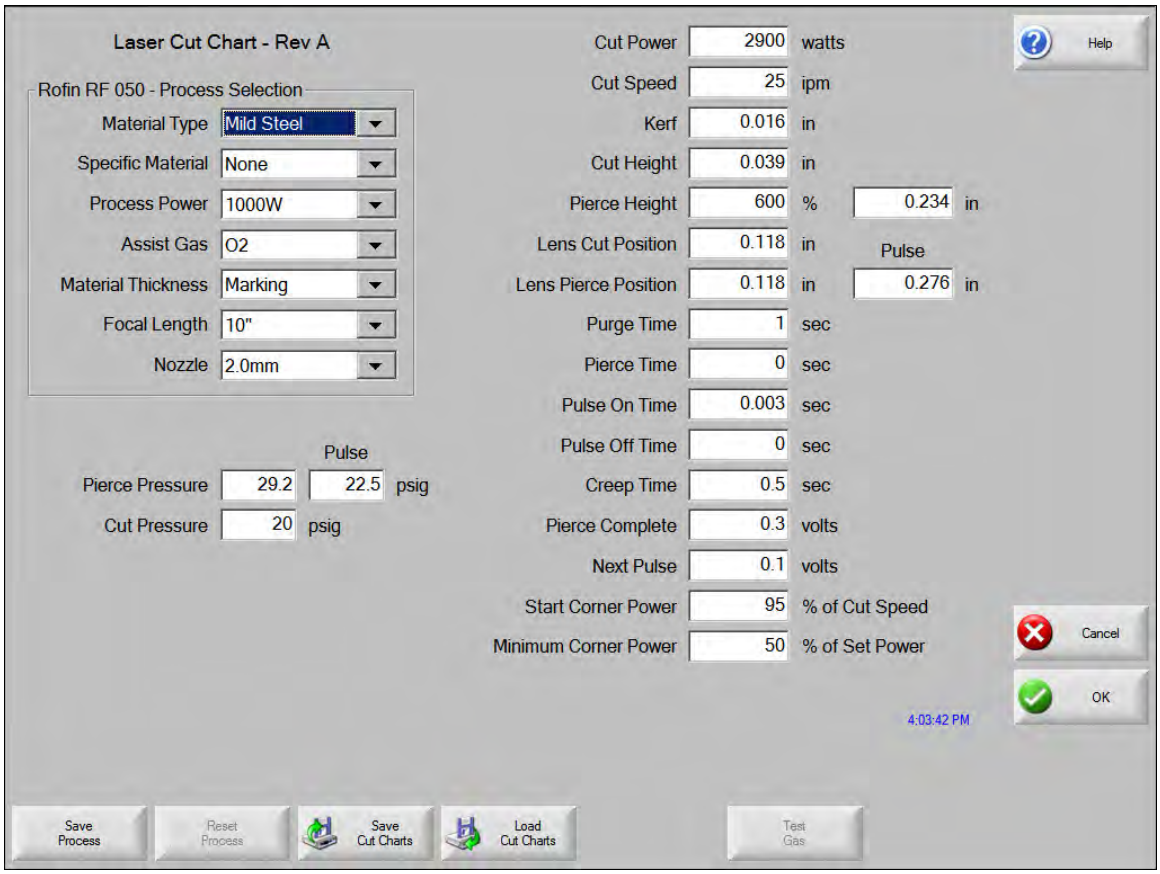
Next Pulse: Based on sensors in the laser head, the system can determine when the next laser pulse is delivered during Automatic pierce control. The voltage is derived from the feedback of the sensors in the laser cutting head.

Tape Shot Time: Sets the laser pulse time duration for a tape shot beam alignment.

Tape Shot Power: Sets the laser pulse power for a tape shot beam alignment.

Laser cut charts (non-fiber laser)

The cut charts provide factory-recommended settings for a material type and thickness. You can make changes to the cut charts using the Specific Material, Process Power, Assist Gases, Material Thickness, Focal Length, and Nozzle variables.



The laser cut chart is based on the following process variables:

Material Type: Select the material type, such as mild steel, stainless steel, or aluminum.

Specific Material: The Specific Material identifies a custom cut chart. For more information, see *Saving changes to a cut chart* on page 172.

Process Power: Select the appropriate process power (wattage) for the material thickness and material type.

Assist Gas: Select the appropriate assist gas for the desired process.

Material Thickness: Select the material thickness for the material type.

Focal Length: Select the specific focal length lens that must be installed in the laser head.

Nozzle: Select the diameter and type of nozzle that must be installed for the process.

The cut chart also includes the following parameters. The parameter values change depending on the process variables selected.

Pierce Pressure: Shows the gas pressure for piercing.

Cut Pressure: Shows the gas pressure for cutting.

Test Gas: Press the Test Gas soft key to perform the Test Gas function of the cutting assist gas delivery system.

Cut Power: Allows you to set the power (watts) to be used during the cut process. This value can be less than the process power.

Cut Speed: Specifies the Cut Speed for the selected material process.

Kerf: equals the width of the cut that the plasma arc, flame, laser, or waterjet removes as it cuts the material. The CNC automatically offsets the motion path by half the kerf dimension to ensure the part is cut at the correct size.

Cut Height: Select the cut distance from the nozzle tip to the plate. Cut Height is derived from the CHS signal and the calibration curve.

Pierce Height: Select the Pierce Height. This can be entered as a multiple factor that is a calculated value of the Cut Height or an actual Pierce Height distance.

Lens Cut Position: Sets the focal lens position in the laser head for cutting.

Lens Pierce Position: Sets the focal lens pierce position in the laser head for cutting.

Resonator On Time: Allows a specific time for the resonator to turn power ON.

Purge Time: Specifies the time delay from switching from one cutting gas type to another cutting gas type.

Pierce Time: Specifies the time delay from when laser head completes lowering until motion is initiated at creep speed for cutting. When Manual Pierce Control is selected, this is the total pierce time allowed. When Automatic Pierce Control is selected, this time is an additional delay after pierce is complete.

Pulse On/Off Time: When Automatic Pulse Mode is selected for pierce control you can select Pulse on and off time to adjust the pulse. The Off Time starts when the sensor signal falls below the next pulse threshold.

Creep Time: Specifies the period after pierce complete that the laser head travels at Creep Speed. Creep Speed is determined by a setup parameter in the Speeds setup screen and is a percentage of the programmed cut speed. After Creep Time is complete, the control accelerates to full cut speed.

Pierce Complete: The Automatic Pierce monitors voltage of sensors in the laser head to detect completion of the pierce. This is used in conjunction with Pulse On Time, Pulse Off Time and next pulse.

Next Pulse: Based on sensors in the laser head, the system can determine when the next pulse occurs. The Next Pulse will be delivered when the voltage drops below the Next Pulse setting.

Start Corner Power: Allows you to define a speed where the corner power analog signal will be used to start to decrease laser power. This is defined as a percentage of cut speed. The following graph shows Start Corner Power set to 80%.

Minimum Corner Power: Defines the minimum laser resonator power to switch when the cut speed reduces to zero in a corner. This is defined as a percentage of selected power (watts).

Waterjet process

Waterjet cutting uses highly-pressurized water, alone or with and abrasive material, to cut metal and non-conductive materials. The waterjet process described in this section applies only to Hypertherm HyPrecision™ intensifier pumps.

If you created waterjet cut charts using Phoenix 9.74.0 or earlier, those cut charts can no longer be used. Contact Hypertherm Technical Service or the Product Applications Engineer for your region for assistance. Hypertherm regional office locations can be found at the beginning of this manual.

The cut chart and process parameters can also be selected from the part program using the G59 process variables. See the *Phoenix Version 9 Series Programmer's Reference* for the format of the waterjet G59 codes.

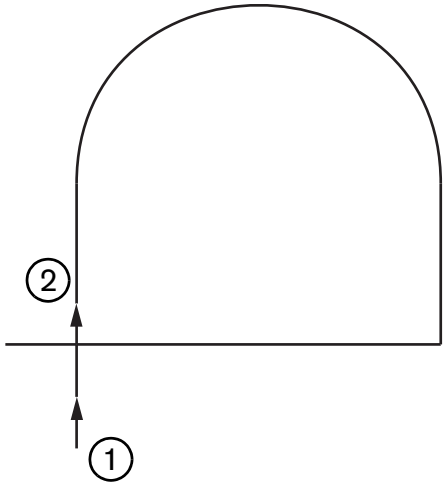
Waterjet pierce types

The CNC provides three moving pierces and a stationary pierce. Select the pierce type from the Waterjet Process screen, Waterjet Cut Chart screen, or from the part program using the G59 V825 code.

Additional pierce parameters can be changed in the Waterjet Cut Chart screen or through the part program. For many waterjet cutting applications, a moving pierce cuts through the material more quickly than a stationary pierce because the machine motion clears the cut of the abrasive and material debris.

Dynamic pierce

For the dynamic pierce, the waterjet moves along the part lead-in at the Pierce Speed for the Pierce Time. When the Pierce Time elapses, the waterjet changes to the Cut Speed. Be sure the part has a lead-in that is long enough to allow the waterjet to pierce the workpiece completely before changing to the Cut Speed.



1 Dynamic pierce starts at the beginning of the lead-in.

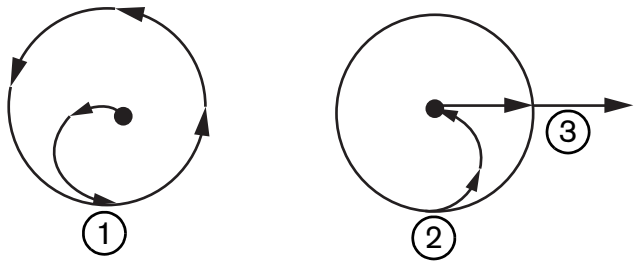
2 The waterjet changes to the Cut Speed after the Pierce Time elapses.

Circular pierce

For the circular pierce, the waterjet moves at the Pierce Speed for the Pierce Time in a circular motion. The Pierce Displacement shows the circle diameter. The circle diameter depends in part on the size nozzle being used.

- 0.03 inch nozzle size (0.76 mm) creates a circle diameter of 0.079 inches (2 mm).
- 0.04 inches nozzle size (1 mm) creates a circle diameter of 0.105 inches (2.7 mm).

When the Pierce Time elapses, the waterjet returns to the circle center point, then changes to the Cut Speed to cut the part.



1 Circular pierce begins at the centerpoint, then travels around the circle until the Pierce Time elapses.

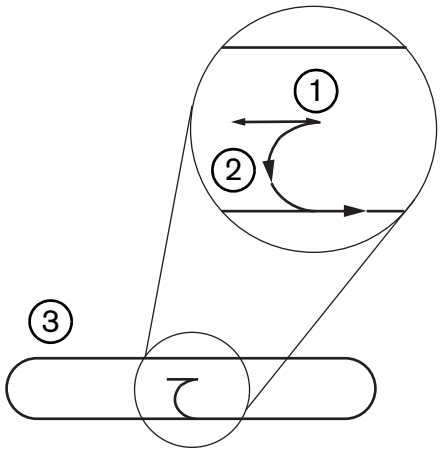
2 The waterjet returns to the circle center point, then travels at the cut speed as it approaches the cut path.

3 Part lead-in

A circular pierce may take longer than a dynamic or wiggle pierce but not as long as a stationary pierce. Use the circular pierce for small internal features in material thicker than 0.508 mm (0.2 inches).

Wiggle pierce

For the wiggle pierce, the waterjet moves back and forth over a segment for the Pierce Time at the Pierce Speed. The Pierce Displacement defines the length of the segment, and the segment is tangent to the part lead-in. When the Pierce Time elapses, the waterjet returns to the beginning of the pierce, then changes to the Cut Speed. Use the wiggle pierce for narrow features such as slots, closely nested parts, or when space constraints prevent the use of a circular or dynamic pierce. Use a wiggle pierce for material thicker than 38 mm (1.5 inches) where a dynamic pierce may become too long for internal features.



1 Wiggle pierce segment

2 Part lead-in. Arrows show cut direction.

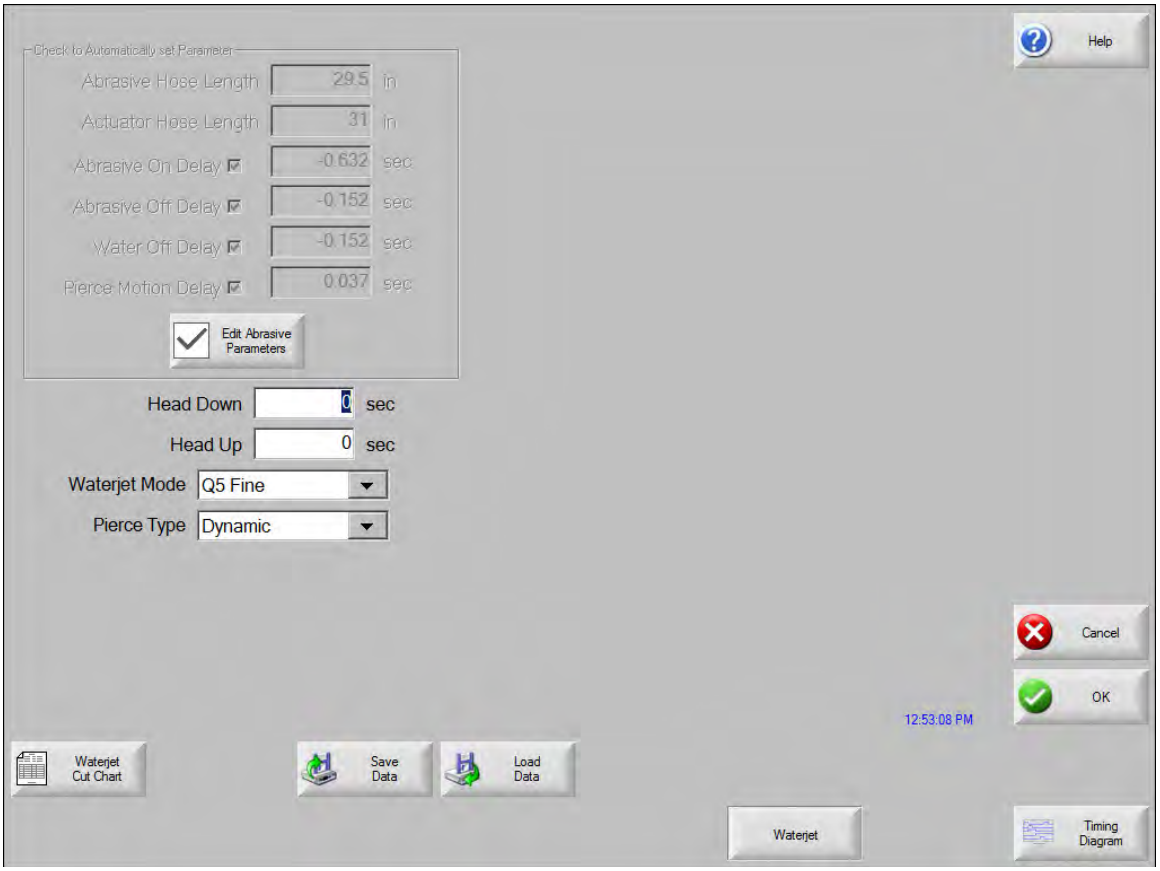
3 Internal part feature (slot)

Stationary pierce

For the stationary pierce, the waterjet stays at the pierce point until the Pierce Time elapses. Use the stationary pierce on material thinner than 0.508 mm (0.2 inches), or small internal part features on material thicker than 0.508 mm (0.2 inches).

Waterjet process screen

The timers in the Waterjet Process screen begin after the CNC executes the M07 code (Cut On) at the beginning of a cut. Choose Setups > Process > Waterjet to open the waterjet process screen.



Abrasive On Delay: Sets a time for abrasive flow to start before or after the water flow starts. Enter a negative Abrasive On Delay to start the abrasive flow before the water flow starts, up to 1 second (a value of -1). Enter a positive Abrasive On Delay to start the abrasive flow after the water flow starts, up to 5 seconds. When the Abrasive On Delay elapses, the Pierce Motion Delay starts.

Abrasive Off Delay: Sets a timer to stop the abrasive flow before or after the end of the cut. Enter a negative Abrasive Off Delay to stop the abrasive flow up to 1 second before the end of the cut (-1). Enter a positive Abrasive Off Delay to stop the abrasive flow up to 9.9 seconds after the end of the cut.

Water Off Delay: Sets a timer to stop the water flow before or after the end of the cut. Enter a negative Water Off Delay to stop the water flow up to 1 second before the end of the cut (-1). Enter a positive Water Off Delay to stop the water flow up to 9.9 seconds after the end of the cut.

The Abrasive Off Delay and Water Off Delay run concurrently.

Pierce Motion Delay: The time before the pierce starts after the CNC executes the M07 code in the part program. When the Abrasive Flow Delay elapses, the Pierce Motion Delay starts. Enter 0 (no delay) to 9.9 seconds for the Pierce Motion Delay.

Head Down: The Head Down time starts when the CNC executes the M07 code and activates the Torch Down output. Torch Down remains activated until either the Torch Down Sense input activates, or the Head Down time elapses. The CNC displays the Head Down status message on the Main screen for the duration of the Head Down time. When the Head Down timer is greater than 0, the Torch Down output activates before Cut Control output activates.

Head Up: The Head Up time starts when the CNC executes the M08 code and activates the Torch Up Output. (The CNC also activates Torch Up when the operator presses Stop on the operator console or F10 on the keyboard.) Torch Up remains activated until either the Torch Up Sense input activates or the Head Up time elapses. The CNC displays the Head Up status message on the Main screen for the duration of the Head Up time. When the Head Up timer is greater than 0, the Torch Up output activates after Cut Control turns off.

Setting Head Down and Head Up to 0 disables them.

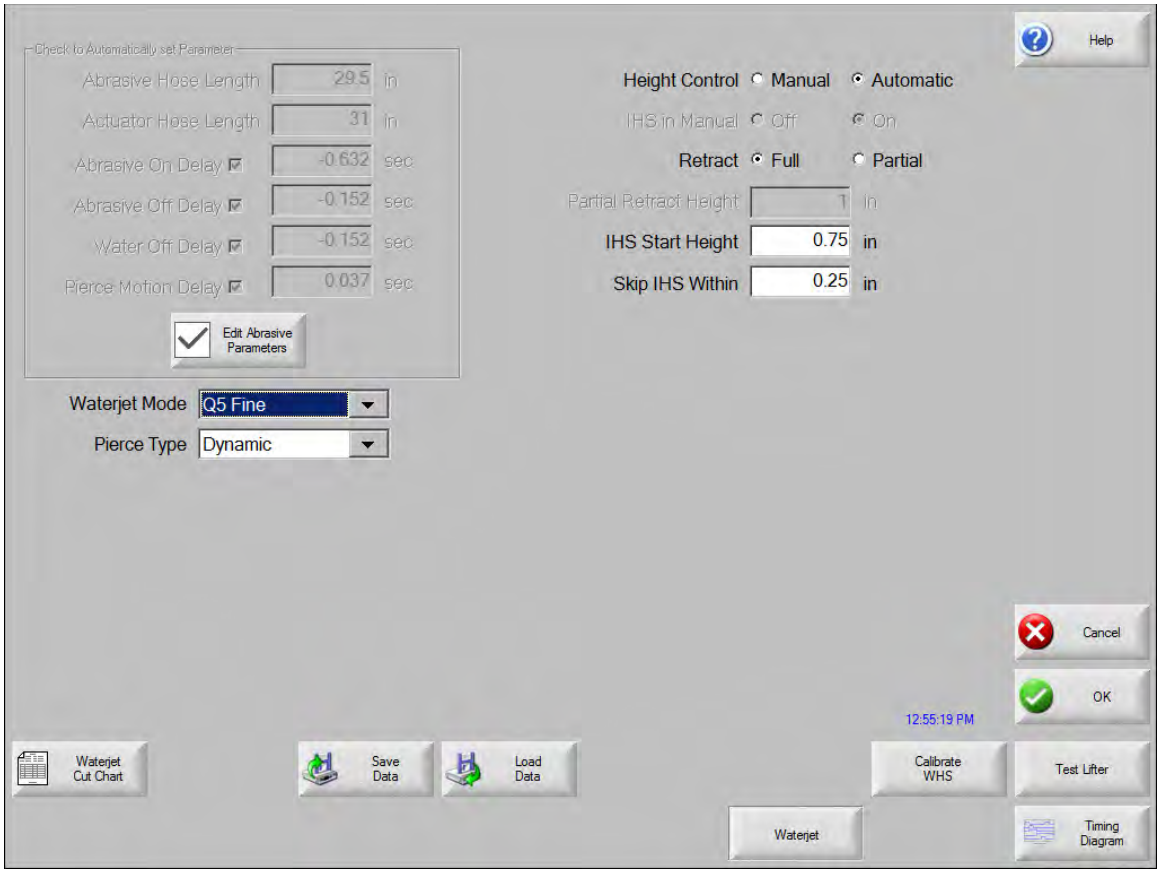
Head Down time occurs before the Abrasive On Delay. Head Up time occurs after Abrasive or Water Off Delays. Head Down and Head Up do not run concurrently with any other timers.

On a cutting system with multiple stations, The Torch Down output remains activated until all stations activate their Torch Up Sense inputs, or the Head Down or Head Up time elapses.

Waterjet Mode: Select the edge surface finish for all of the cuts in the part program. Q1 has the fastest cut speed, but the edge surface finish will be more rough, while Q5 has the slowest cut speed but a smoother edge finish. Q6, Wet Run, etches the metal by cutting at a high feed rate without abrasive.

Pierce Type: Choose from these moving pierce techniques: Dynamic, Circular, Wiggle; or choose a Stationary pierce. All cuts in the part program use this pierce type. Moving pierces cut through the material more quickly because the machine motion clears the cut of the abrasive and material debris. See *Waterjet pierce types* on page 159 for more information.

Waterjet process screen (with Sensor height control)



Abrasive Hose Length: The length of the hose from the abrasive regulator to the cutting head. This length is one factor used to calculate the Abrasive On Delay and the Abrasive Off Delay.

Actuator Hose Length: The length of the air hose from the cut control solenoid to the actuator valve on the cutting head. This length is one factor used to calculate the Water Off Delay.

Abrasive On Delay: Sets a time for abrasive flow to start before or after the water flow starts. Enter a negative Abrasive On Delay to start the abrasive flow before the water flow starts, up to 1 second (a value of -1). Enter a positive Abrasive On Delay to start the abrasive flow after the water flow starts, up to 5 seconds. When the Abrasive On Delay elapses, the Pierce Motion Delay starts.

Abrasive Off Delay: Sets a timer to stop the abrasive flow before or after the end of the cut. Enter a negative Abrasive Off Delay to stop the abrasive flow up to 1 second before the end of the cut (-1). Enter a positive Abrasive Off Delay to stop the abrasive flow up to 9.9 seconds after the end of the cut.

Water Off Delay: Sets a timer to stop the water flow before or after the end of the cut. Enter a negative Water Off Delay to stop the water flow up to 1 second before the end of the cut (-1). Enter a positive Water Off Delay to stop the water flow up to 9.9 seconds after the end of the cut.

The Abrasive Off Delay and Water Off Delay run concurrently.

Pierce Motion Delay: The time before the pierce starts after the CNC executes the M07 code in the part program. When the Abrasive Flow Delay elapses, the Pierce Motion Delay starts. Enter 0 (no delay) to 9.9 seconds for the Pierce Motion Delay.

Waterjet Mode: Select the edge surface finish for all of the cuts in the part program. Q1 has the fastest cut speed, but the edge surface finish will be more rough, while Q5 has the slowest cut speed but a smoother edge finish. Q6, Wet Run, etches the metal by cutting at a high feed rate without abrasive.

Pierce Type: Choose from these moving pierce techniques: Dynamic, Circular, Wiggle; or choose a Stationary pierce. All cuts in the part program use this pierce type. Moving pierces cut through the material more quickly because the machine motion clears the cut of the abrasive and material debris. See *Waterjet pierce types* on page 159 for more information.

Height Control Manual/Automatic: Select the type of height control for your cutting system and that is best for the material being cut. Choose Automatic for the Sensor height control except for materials that need to be cut using manual height control.

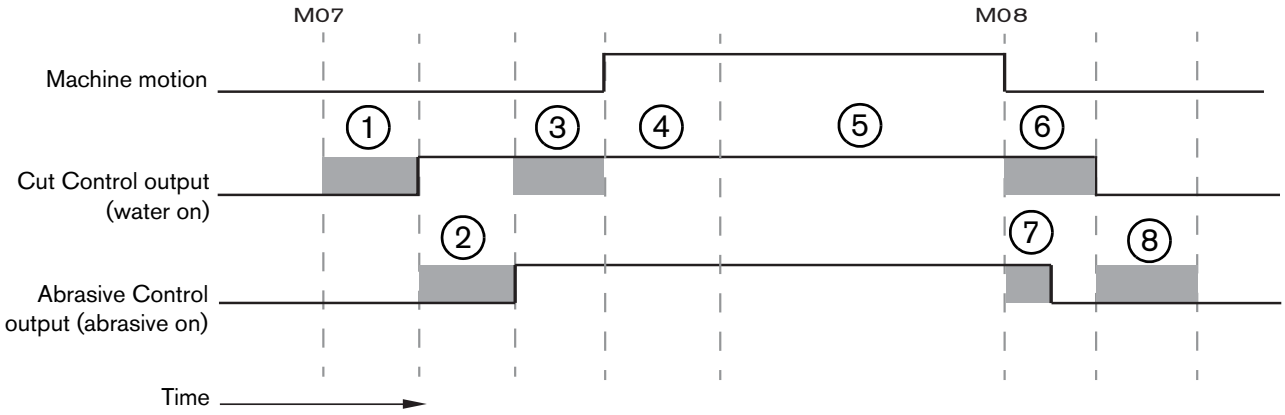
IHS in Manual: If the cutting system has a manual height control, use the Initial Height Sense feature when operating the lifter in manual mode.

Retract Full/Partial: Selects the retract distance to be set at Full or Partial. In the Full retract mode, the cutting head retracts to the Z-Axis Home position. In Partial retract mode, the cutting head retracts to the Partial Retract Height.

IHS Start Height: Specifies the distance of travel for the height control to move the cutting head at Fast IHS speed before switching to Slow IHS speed and beginning Initial Height Sense. Take caution when selecting this distance so that the cutting head does not crash into the plate.

Skip IHS Within: Disables initial height sense at pierce points if IHS falls within the selected distance. This setting increases cutting productivity. The distance is measured from the end point of the cut segment to the next pierce point.

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 the abrasive).



1 Head down time

2 Abrasive On Delay time (+)

3 Pierce Motion Delay time

4 Pierce Time (set on the cut chart)

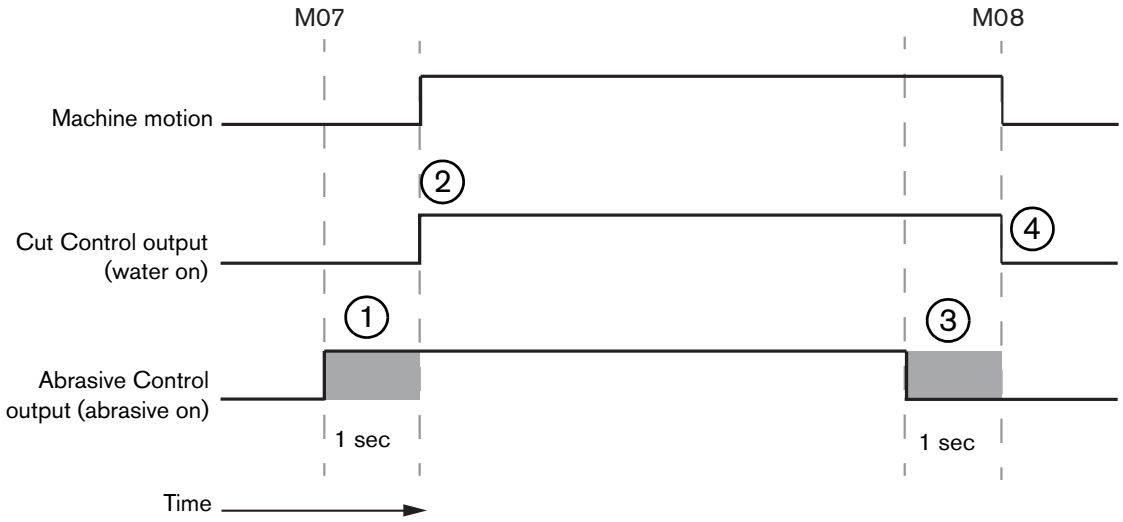
5 Cutting motion

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.

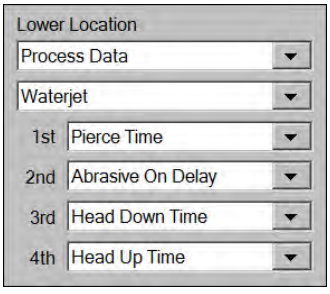


1. Abrasive On Delay (-) starts abrasive before water flow.
2. Cut Control (water flow) turns on after Abrasive On Delay elapses.
3. Abrasive off Delay (-) turns off abrasive before the end of cut (M08)
4. Cut Control turns off after Abrasive Off Delay elapses.

Waterjet Watch Window

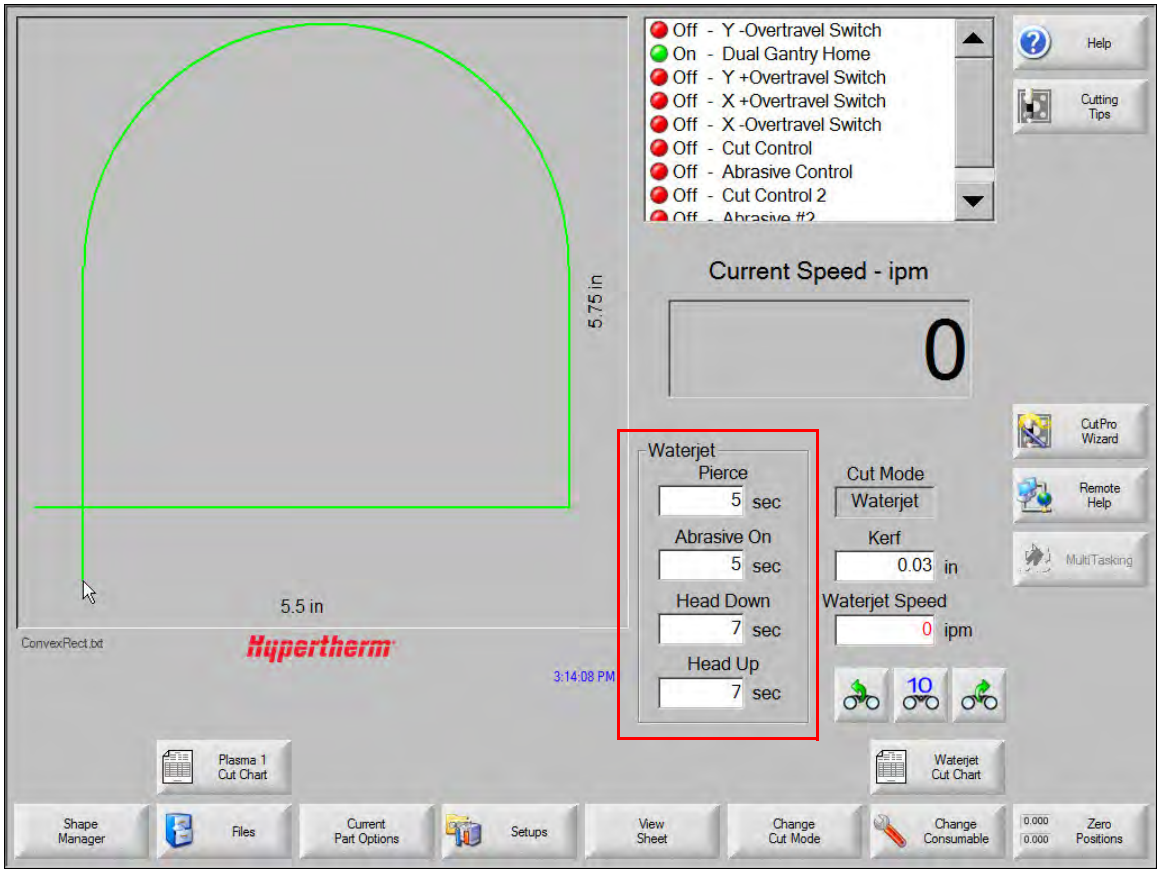
You can set up a Watch Window to view the timers while cutting parts.

1. Choose Setups > Watch.
2. Choose Process Data for Lower Location.
3. Choose Waterjet below Process Data.
4. The Watch Window provides 4 fields for Process Data. From each field, choose a timer or parameter.



5. Choose OK to save the Watch Window. The CNC displays the Process Data on the Main Screen. You can also change some process parameter values when they display in the Watch Window.

If you would like to display additional Process Data parameters, you can set up additional Watch Windows. The CNC provides 10 Watch Windows that you can customize.



Adjusting the pierce time

You can override the Pierce Time set in the cut chart or part program while the pierce timer is running. The pierce timer automatically displays on the Main screen when the Pierce Time starts. After the part program starts, the following three soft keys appear on the screen for the duration of the pierce time. As soon as the pierce time elapses, the soft keys disappear from the screen.

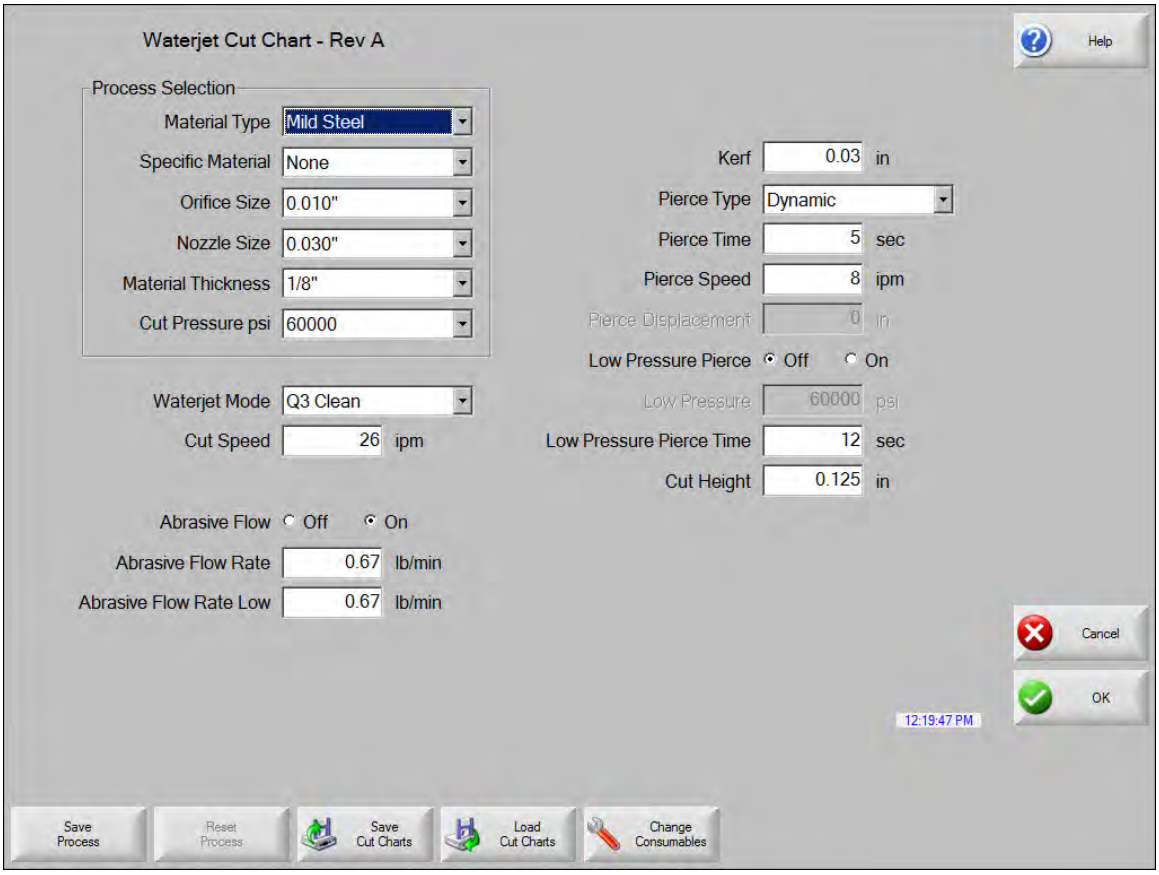
Extend: Lengthens the Pierce Time. To end the Pierce Time, press the Set Now or Release soft key.

Set Now: Ends the Pierce Time and stores the new pierce time in the setup file. Use Set Now with Extend to change and save a new Pierce Time. The CNC uses the new pierce time until you load a different cut chart, or change the Pierce Time on the Waterjet Cut Chart screen.

Release: Ends the Pierce Time without saving a new Pierce Time.

Waterjet cut chart

The cut charts provide factory-recommended settings for a material type and thickness. From the Main screen, choose the Waterjet Cut Chart soft key.



The waterjet cut chart is based on the process variables shown below. When you select a process variable, the CNC retrieves the cutting parameters from the cut chart.

Material Type: Selects the material type, such as mild steel, stainless steel, or aluminum. Choose Other to load a generic cut chart that you can customize and save for a different material than the ones provided.

Specific Material: The Specific Material identifies a custom cut chart. See *Saving a waterjet cut chart* on page 170 for more information.

Orifice Size: A waterjet consumable, this defines the orifice diameter.

Nozzle Size: A waterjet consumable, this defines the nozzle diameter.

Material Thickness: Select the thickness for the material type.

Cut Pressure: The commanded water pressure setpoint for the process.

For the CNC to send the water pressure setting to the pump requires that the pump be equipped with electronic proportional pressure control. If the pump has dual manual pressure control, then the Cut Pressure shows the recommended setting that must be manually set at the pump. Refer to the pump operator manual for more information.

The cut chart also includes the following parameters. The parameter values change depending on the process variables selected.

Waterjet Mode: Select the edge surface finish for all of the cuts in the part program. Q1 has the fastest cut speed, but the edge surface finish will be more rough, while Q5 has the slowest cut speed but a smoother edge finish. Q6, Wet Run, etches the metal by cutting at a high feed rate without abrasive.

Cut Speed: Sets the cutting speed (also known as feed rate) for the material type and material thickness. The Waterjet Mode also affects the Cut Speed.

Abrasive Flow: Turns on abrasive for the cut. You can use the Abrasive On Delay and Abrasive Off Delay timers on the Waterjet Process screen to control when the abrasive flow starts or stops in relation to the water flow.

Abrasive Flow Rate: Sets an abrasive flow regulator. The value may be for display-only. To support an abrasive flow regulator requires an analog output on the SERCOS ring, or the analog signals from an axis on the CNC. When this value is set to 0, the Abrasive Control output remains off. The flow rate is calculated by the CNC from 0 to 1 kg per minute (0 to 2.2 lbs per minute) and proportionally output from 0 to +10 volts.

- EDGE Pro Hypath or MicroEDGE Pro Hypath: Use Axis DAC +10 V output and Analog common signals on the drive/encoder connector.
- EDGE Pro Picopath or MicroEDGE Pro Picopath: Use the Servo output (+10 VDC) and Servo output common signals on the drive/encoder connector.

In addition, the Hypath or Picopath CNC must have an available axis. For example, if your CNC supports two axes, a third axis must be enabled on the CNC hardware key to support an abrasive flow regulator. Contact Hypertherm Technical Service or the Product Applications Engineer for your region for assistance. Hypertherm regional office locations can be found at the beginning of this manual. For more information on connecting the abrasive flow regulator to the CNC, see the *Phoenix V9 Series Installation and Setup Manual*.

Abrasive Flow Rate Low: The abrasive flow rate used during low pressure piercing. This activates the abrasive flow rate analog output.

Kerf: Equals the width of the cut that the waterjet removes as it cuts the material. The CNC automatically offsets the motion path by half the kerf dimension to ensure the part is cut at the correct size.

Pierce Type: Choose from these moving pierce techniques: Dynamic, Circular, Wiggle; or choose a Stationary pierce. All cuts in the part program use this pierce type. Moving pierces cut through the material more quickly because the machine motion clears the cut of the abrasive and material debris. See *Waterjet pierce types* on page 159 to learn about all the pierce types.

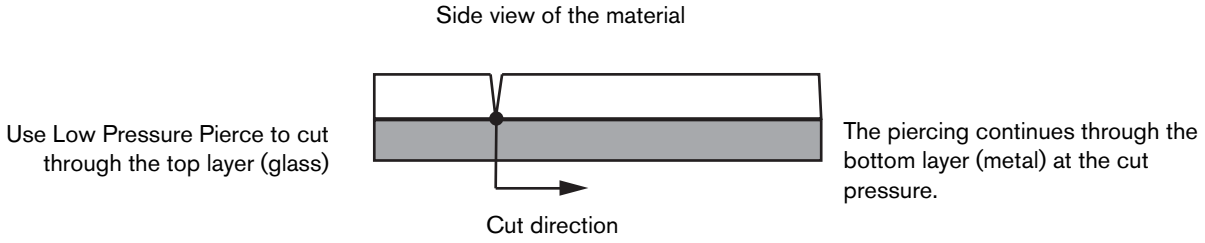
Pierce Time: Sets the time that the Pierce Type is in use. The Pierce Time begins after the CNC executes the M07 (Cut On) code in the part program and both the Abrasive Flow Delay and the Pierce Motion Delay elapse.

Pierce Speed: Sets the speed for the moving pierces: Dynamic, Circular, and Wiggle. The Pierce Speed is usually much slower than the Cut Speed.

Pierce Displacement: Defines diameter of the circular pierce or the length of the wiggle pierce.

Low Pressure Pierce: The commanded pierce water pressure setpoint. Low Pressure Pierce uses a lower cut pressure during all or part of the pierce. Any pierce type can use low pressure pierce. The system uses the Low Pressure (usually around 25% of the Cut Pressure) for the duration of the Low Pressure Pierce Time. When Low Pressure Pierce is set to ON, the CNC activates the Low Pressure Pierce output for the duration of the Low Pressure Pierce Time.

Use the Low Pressure Pierce for cutting into a layered workpiece, for example, a mirror that has a layer of glass adhered to a layer of metal. This allows the system to pierce the glass at the low pressure, then pierce the metal with the cut pressure.



For the CNC to send the Low Pressure Pierce setting to the pump requires that the pump be equipped with electronic proportional pressure control. If the pump has dual manual pressure control, then the Cut Pressure shows the recommended setting that must be manually set at the pump. Refer to the pump operator manual for more information.

Low Pressure Pierce Time: This timer begins at the same time as the Pierce Time and measures the amount of time the pump operates at the Low Pressure Pierce set point for the water pressure.

Cut Height: Equals the height of the nozzle tip above the workpiece. Set the cut height in one of these ways:

- Using a feeler gage, manually jog the cutting head to cut height for the job.
- Use the Head Down timer to lower the cutting head to the cut height. You may have to fine-tune the Head Down timer to reach the correct height.
- Use the Torch Down Sense input to signal the CNC that the cutting head has reached the cut height.

Saving a waterjet cut chart

Follow these steps to save a custom waterjet cut chart for a material other than mild steel, stainless steel, or aluminum.

Before you can create a new cut chart, you need to have the Adding Processes feature set to Allowed in the Status/Feature list on the Special Setups screen.

See *Saving changes to a cut chart* on page 172 for information on saving a plasma cut chart.

1. On the Waterjet Cut Chart screen, choose Other for material type.
2. Choose Specific Material.

3. Double-click the blue message at the bottom of the Cut Chart screen.
4. Choose Add.
5. Enter the material name and choose OK. The name is available in the Specific Material list when the Other material type is selected.
6. Enter settings for the cut chart as needed.
7. Choose Save Process to save the cut chart.

The CNC creates two copies of the cut chart and names them like the following examples:

Other DialLine300-HyPrecision.fac

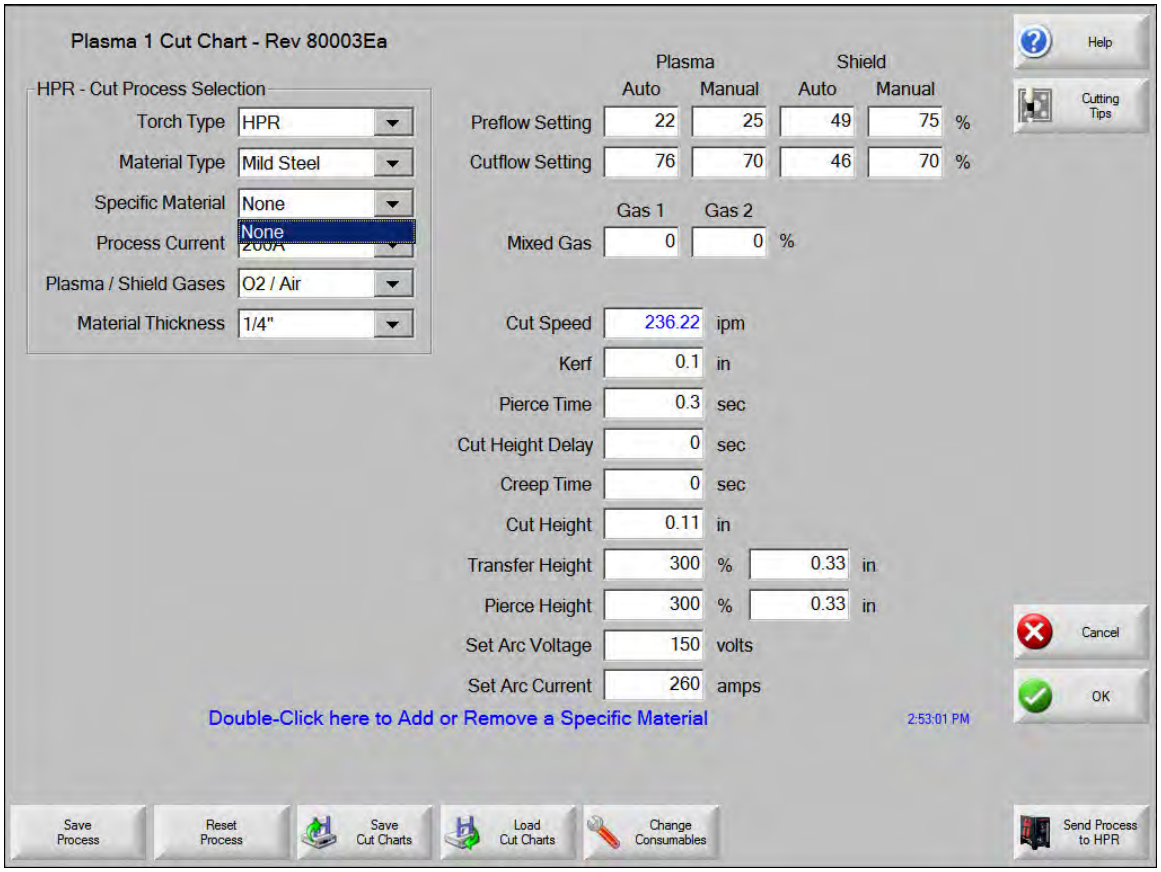
Other DiaLine300-HyPrecision.usr

The CNC preserves the original settings in a .fac, or factory cut chart. Any time you make changes to the cut chart, the CNC saves those changes to the .usr, or user, cut chart.

Saving changes to a cut chart

The CNC provides cut charts for mild steel, stainless steel, and aluminum. You can change the cut charts provided on the CNC by entering a new value into the cut chart and answering Yes to save the changes when you exit the cut chart screen. The CNC saves the changes in the .usr version of the cut chart. You can always revert to the factory cut chart settings by choosing the Reset Process soft key on the cut chart screen. The factory cut charts, the .fac versions, do not get overwritten by the CNC.

If you will be cutting a different material, or have a special process for cutting a material such as mild steel, you can save the cut process in its own cut chart. Phoenix identifies a custom cut chart by the Specific Material process variable. Choose Specific Material, and then double-click the blue message at the bottom of the screen (or press the right bracket] + F8) to add or remove a specific material. The CNC allows you to save up to 98 custom cut charts.



Creating a new cut chart

Before you can create a new cut chart, you need to have the Adding Processes feature set to Allowed in the Status/Feature list on the Special Setups screen.

1. Select the Torch Type and Material Type that is similar to the process you want to create.
2. Choose the Specific Material process variable. None, the factory default, may be the only option for Specific Material.
3. Double-click on the blue message at the bottom of the screen.

4. Choose Add on the message popup.

5. Enter the name of the new specific material and choose OK.

6. Phoenix saves the new material in the Specific Material list, and copies all the variables and parameters into a cut chart identified by the new material. Phoenix then resets the cut chart screen to the first torch type in the list. You may need to reselect the torch type and material to see your new specific material entry in the list.

   You can also add or remove a process current, a pair of process/shield gases, or a material thickness. Be sure to have the Specific Material displayed before adding a new process variable.

7. After you have selected the process variables for your cut chart, adjust the parameter values on the right of the screen to accommodate the process variables.

8. Choose Save Cut Chart and choose Yes on the next two confirmation messages. Phoenix saves the cut chart as both a .fac and .usr file. It names the files with the material type, a number, torch type, plasma system. For example:
   - Mild Steel 2-HPR-HPR.usr
   - Mild Steel 2-HPR-HPR.fac

Retrieving the new cut chart

1. Choose the Torch Type, Material Type, and Specific Material.

2. Choose the Process Current and Material Thickness. The CNC displays the parameters for your custom cut chart.

Section 8

Torch Height Controls

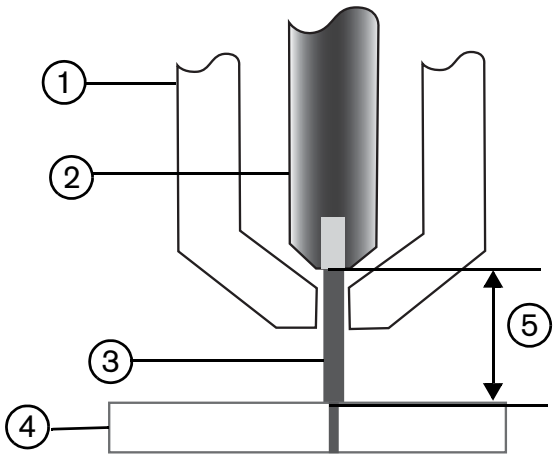
Hypertherm CNCs support the Sensor THC and ArcGlide THC, as well as torch height controls from other manufacturers.

About plasma torch height control

A circuit board inside the plasma supply (a *plasma interface board* or a *voltage divider card*) measures the voltage drop across the plasma arc. This measurement is the raw arc voltage and can range from 0 VDC to 400 VDC. The circuit board then reduces this measurement into an analog signal (0 VDC to 10 VDC) that is sent to the CNC. This signal represents the actual arc voltage when cutting.

In the CNC, each plasma process has an arc voltage set point, called the *Set Arc Voltage*, for a given material thickness, cut height, cut speed, gas type, and current. When cutting starts, the CNC tracks the actual voltage drop across the arc and compares it to the Set Arc Voltage. When the actual arc voltage is higher or lower than the Set Arc Voltage, the CNC commands the lifter to move the torch down or up.

- When the actual arc voltage is *higher* than the arc voltage set point, the torch moves *down*.
- When the arc voltage is *lower* than the set point, the torch moves *up*.
- The higher the arc voltage set point, the higher the cut height.



1 Torch

2 Electrode

3 Plasma arc

4 Workpiece

5 Voltage drop is measured over the plasma arc between the electrode and the workpiece

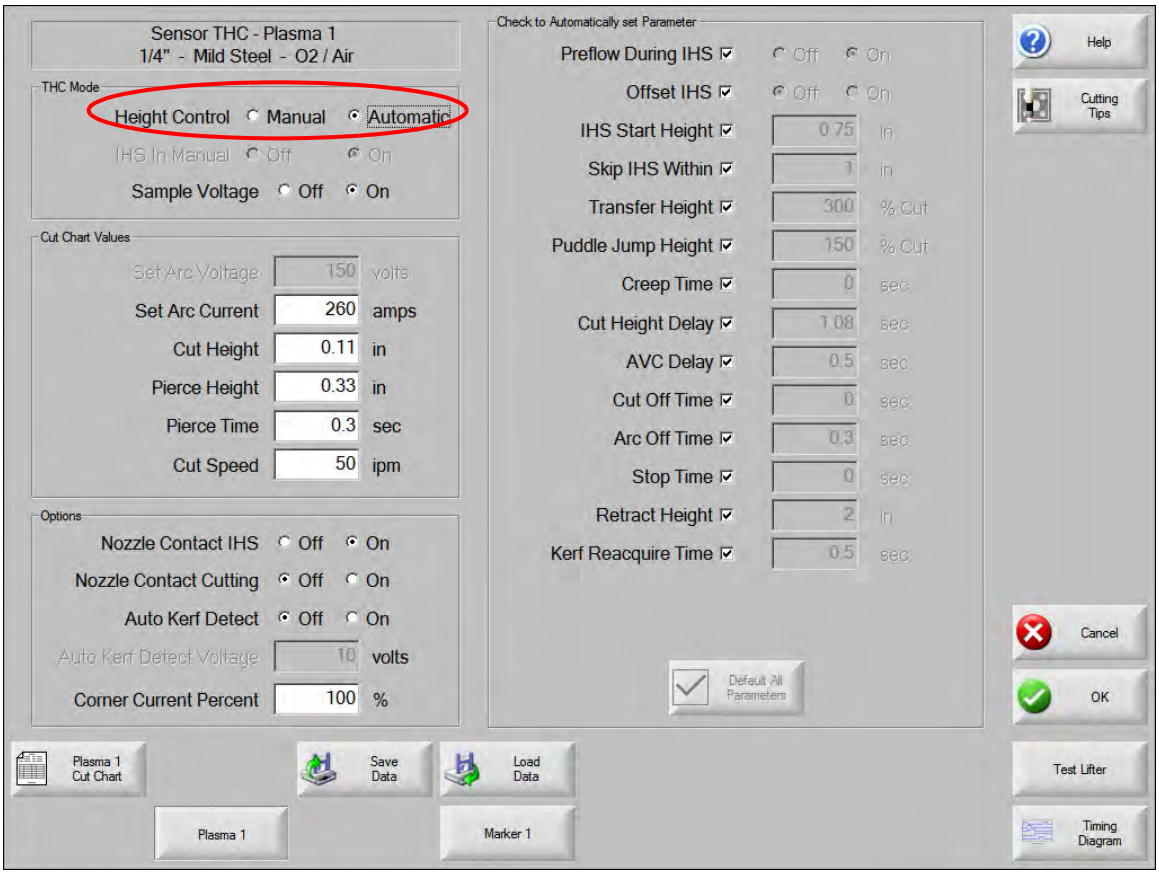
This section describes the operation of the Sensor THC and ArcGlide THC. In it you will find:

- ArcGlide THC and Sensor THC operating modes
- Methods for changing the arc voltage set point
- Initial height sense
- THC sequence of operations
- THC process screen
- THC marker screen
- Watch Window setup for THCs
- Status messages
- ArcGlide Diagnostic screen

For information about settings for a THC that is not manufactured by Hypertherm, refer to *Cut Processes and Cut Charts* on page 125.

ArcGlide THC and Sensor THC operating modes

Choose Setups > Process, and select the operating mode on the Plasma Process screen. The selection of Automatic or Manual mode is the same for the Sensor THC.



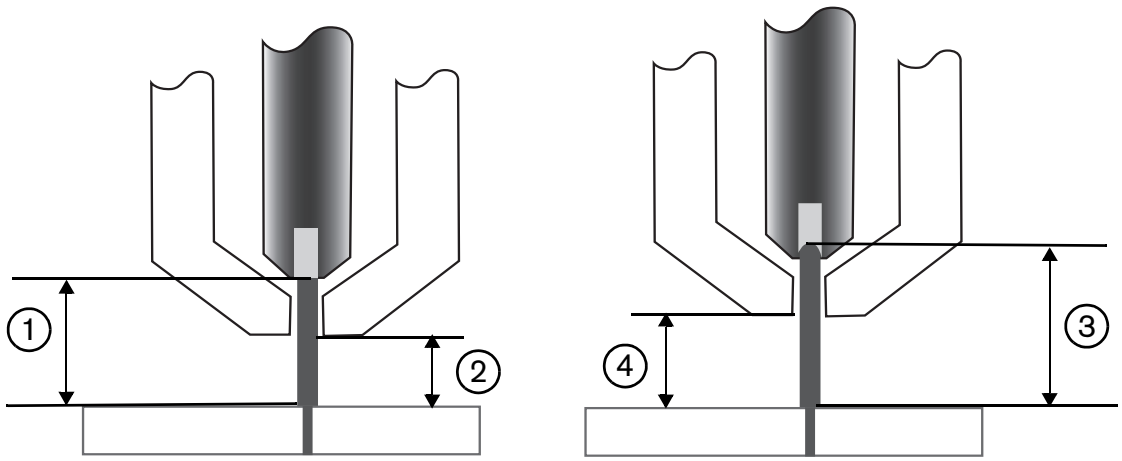
Automatic modes

In all of the automatic modes, the THC performs an initial height sense (see page 182), then retracts to the Transfer Height. After torch ignition, the plasma arc transfers to the workpiece, then the torch moves to the Pierce Height for the duration of the Pierce Time. During this pre-cutting sequence, the torch height control is disabled and the CNC is not tracking the arc voltage. When the Pierce Time elapses, motion begins and the CNC begins tracking arc voltage after the AVC (Automatic Voltage Control) Delay time elapses and the cutting speed is equal to the program speed.

Sample Arc Voltage mode

Use Sample Arc Voltage mode as much as possible to achieve a consistent cut quality over the life of the consumables. When cutting begins, the CNC takes several samples of the arc voltage and averages these samples. It then uses the sample average as the Set Arc Voltage instead of the value in the Process screen, and compares the sample to the actual arc voltage. If the actual voltage is higher than the sample value, the torch moves down. If the actual arc voltage is lower than the sample value, the torch moves up.

The advantage of Sample Arc Voltage mode is that the voltage sample is the result of many readings of the actual arc voltage under steady-state cutting conditions at the correct speed and cut height for the active cut process. Instead of you having to change the Set Arc Voltage as the consumables begin to wear, the CNC re-calculates the voltage sample for each cut in the program and corrects the torch height automatically to maintain the ideal cut height for the process over the life of the consumables.



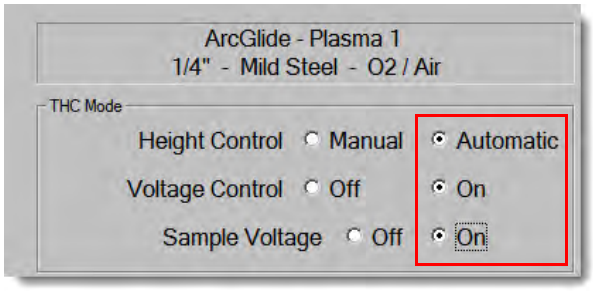
1 Plasma arc from a new electrode

2 Cut height from cut chart

3 The plasma arc lengthens as the electrode wears and the arc voltage increases.

4 When the cut height increases because of electrode wear, the CNC lowers the torch to maintain consistent cut height ②. When not using Sample Arc Voltage, the torch moves closer and closer to the workpiece as the electrode wears.

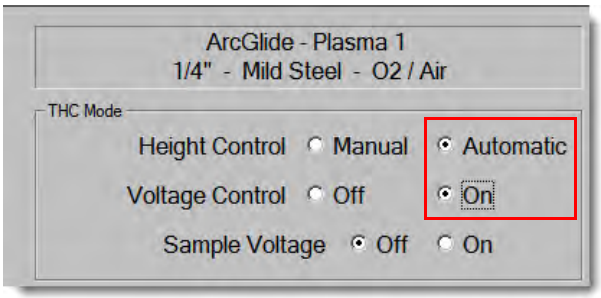
Turn on Sample Arc Voltage mode by making the following selections on the Plasma Process screen. The selections are the same for the Sensor THC.



If the sample arc voltage sample suddenly changes the CNC will stop cutting and show a warning. For example, if the sample average was 100 V and on the next sample, the CNC recorded 115 V. An increase of 15 V indicates that material or slag could be interfering with the arc. The CNC stops so that you can correct the issue, and you can resume cutting by pressing Start.

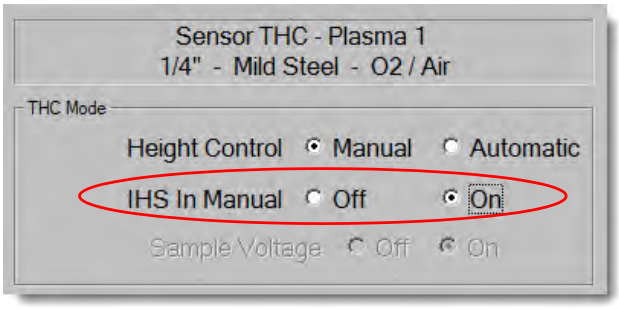
Set Arc Voltage

On the Plasma Process screen, choose Automatic for Height Control and On for Voltage Control to use Set Arc Voltage mode. This mode is recommended for cutting or marking thin material at low cut height, workpieces that are dirty, rusty, oiled or painted, or cutting on a water table or with water injection. When cutting begins, the CNC uses the Set Arc Voltage value from the cut chart and compares it to the actual arc voltage. If the actual voltage is higher than the Set Arc Voltage, the torch moves down. If the actual arc voltage is lower than the Set Arc Voltage, the torch moves up. In Set Arc Voltage mode, you can change the Set Arc Voltage while cutting, or apply voltage offsets for each station in the cutting system. See *Methods for changing the Set Arc Voltage* on page 180 for more information.



Voltage control off – ArcGlide THC or IHS in Manual – Sensor THC

This mode is recommended for rip cuts, or for cutting single, simple parts, or remnants. The THC performs the pre-cutting positioning sequence, then moves to the cut height called for in the cut chart. When cutting begins, the THC maintains the cut height without using arc voltage tracking.



Manual mode

This mode is recommended for rip cuts, for calibrating arc voltage, or for cuts where cut quality is not a primary concern. Manual mode does not use arc voltage tracking or any provide automatic movement of the lifter. You can move the lifter only with the UP and DOWN switch on the ArcGlide THC HMI, the Raise and Lower buttons on the EDGE Pro CNC operator console, or by activating the Raise Torch # or Lower Torch # inputs. Using one of these methods, position the torch at the height above the workpiece to you want use as a cut height. Make sure to be close enough to the workpiece for arc transfer.

Methods for changing the Set Arc Voltage

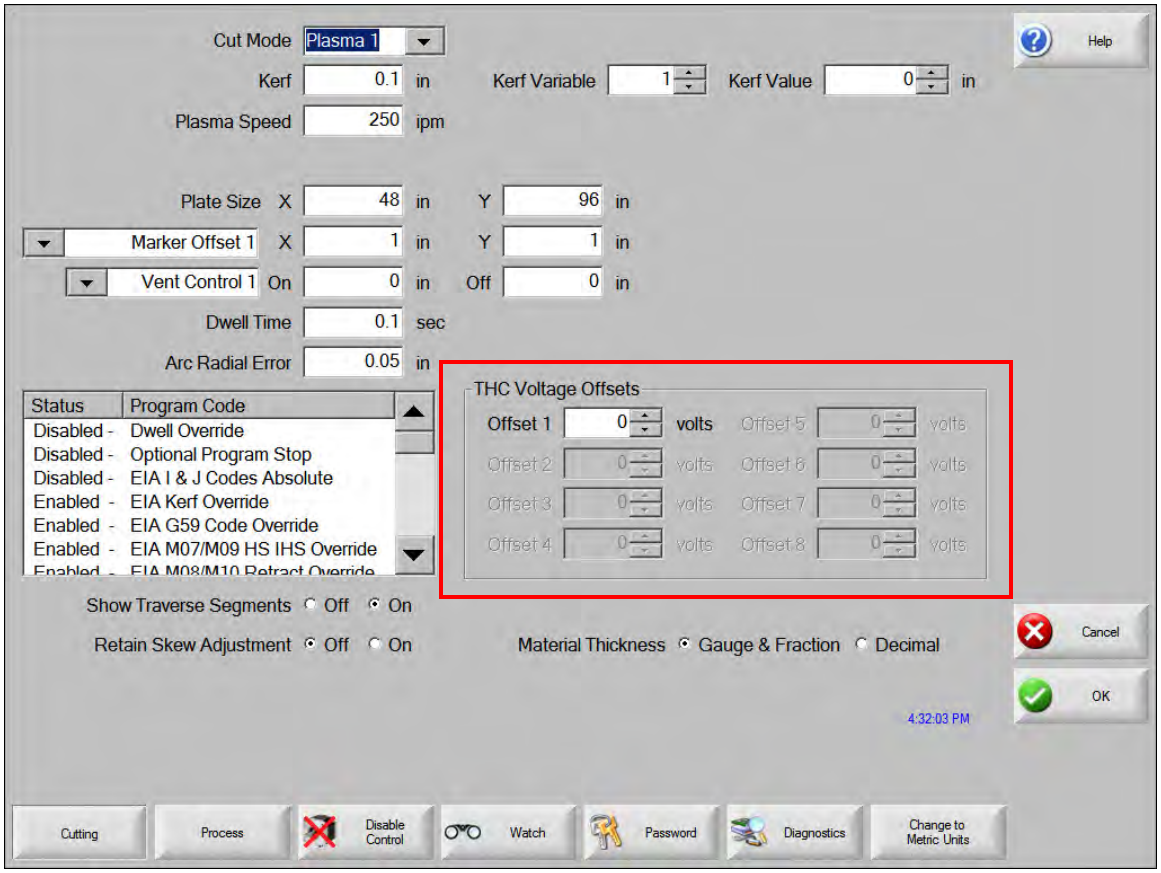
When Sample Voltage is OFF, changing the Set Arc Voltage moves the torch up or down. The CNC reads the Set Arc Voltage parameter (also called the *arc voltage set point*) from the Plasma Process screen (which is set by the cut chart). Some methods for changing the Set Arc Voltage include:

- Issue a G59 V600 F*value* command in the part program for Plasma 1 where F*value* where *value* is the new Set Arc Voltage. (Use G59 V625 F*value* to change the Set Arc Voltage for Plasma 2).
- Enter THC voltage offsets.
- Press the Increase Arc Voltage or Decrease Arc Voltage soft keys on the Main screen while the system is cutting.
- Change the Set Arc Voltage in the Process screen or cut chart.

THC voltage offsets

THC voltage offsets provide a method for changing the Set Arc Voltage value called for in the cut chart. When you enter a positive voltage offset, the CNC adds the voltage offset to the Set Arc Voltage. When you enter a negative voltage offset, the CNC subtracts the voltage offset from the Set Arc Voltage. Voltage offsets are used only when the torch height control is in Automatic mode with Sample Arc Voltage turned off. Sample Arc Voltage mode does not use the THC Voltage Offsets. Both Sensor THC and ArcGlide can use the THC Voltage Offsets feature.

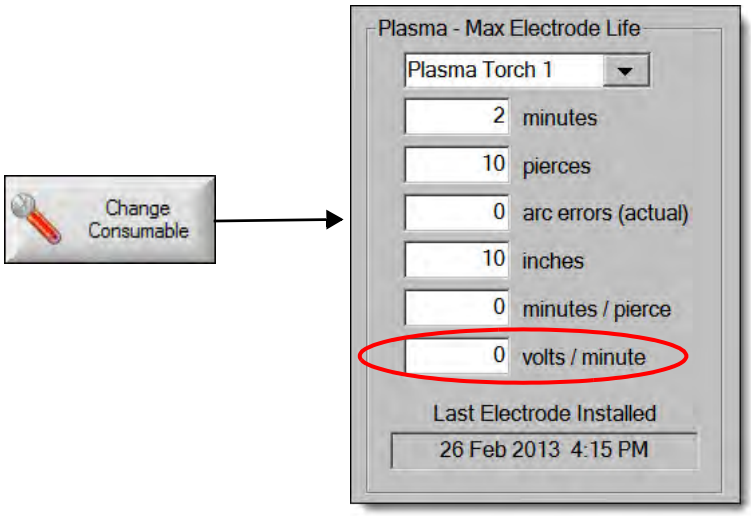
Choose Setups to view the THC Voltage Offsets on the Cutting screen. Offset 1 applies to Station 1, Offset 2 applies to Station 2, and so on.



The THC voltage offsets are saved to the system setup file and are used with every cutting job, even if you load a different cut chart or change the consumables. **The THC voltage offset value will remain at a value unless you change it to zero.** To reset a voltage offset to zero, choose Setup to open the Cutting screen and change the THC voltage offset.

The CNC allows only one arc voltage set point, even when your cutting system has more than one torch. You can use the voltage offset to change the height of an individual torch in a multi-torch system by adding additional voltage to the arc voltage set point for that torch.

For the Sensor THC, the THC voltage offsets can be changed automatically and continually by entering a value for the Volts per Minute parameter on the Change Consumable screen (on the Main screen, choose the Change Consumable soft key). The amount that to enter depends on each customer's consumable use history. For more information see *Changing Consumables* on page 248.



If you have entered a value for Volts/Min, when you change consumables, reset the THC Voltage Offset to zero to allow the CNC to gradually increase the offset using the Volts per Minute parameter. Otherwise, the THC Voltage Offset could be too large and cause torch movement or cut quality issues when applied to the Set Arc Voltage when you are cutting with new consumables.

When using Sample Arc Voltage mode, set the Volts per Minute to 0.

Increase or decrease voltage soft keys

After cutting begins in Automatic mode, the CNC displays Increase Arc Voltage and Decrease Arc Voltage soft keys on the Main screen. Press these keys to change the arc voltage while cutting.

- For the ArcGlide THC, these keys increase or decrease the THC Voltage Offset by 0.5 V per key press.
- For the Sensor THC, these keys increase or decrease the Set Arc Voltage parameter by 0.5 V per key press.

Sometimes the voltage offset or Set Arc Voltage parameters change by more than 0.5 V depending on the length of the key press.

Raise and Lower buttons or inputs

The EDGE Pro CNC operator console has Raise and Lower buttons for each of two stations. These buttons activate the Raise Torch # and Lower Torch # inputs. Activating these inputs using the buttons on the EDGE Pro CNC or on a custom operator console affects the THC voltage offsets in the following ways when the system is cutting:

- For the ArcGlide THC, Raise and Lower change the THC voltage offset for that station by 0.5 V per press. If you display the THC voltage offset in the Watch window, the change to the THC voltage offset appears when cutting completes.
- For the Sensor THC, Raise and Lower change the THC voltage offset for that station by 0.5 V per press. If you display the THC voltage offset in the Watch window, you will see the value increase after your press the button.

Raise and Lower only change the voltage offset when the system is cutting. When the system is not cutting, Raise and Lower move the lifter up and down.

Process screen or cut chart

- If you want a voltage change for a single cutting job, change the Set Arc Voltage in the Process screen.
- To change the Set Arc Voltage for a process, change the value in the cut chart and save it as a custom cut chart.

Initial height sense

Hypertherm THCs use a sequence called *initial height sense*, or IHS, for detecting the workpiece. You perform a first initial height sense after powering up the cutting system and before each cutting job. A 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 all subsequent IHSs which it can perform using much faster speeds since the height of the workpiece is known.

The IHS begins at the Start IHS Height set in the Process screen. When the torch reaches this distance above the workpiece the following actions occur:

- Speed slows from Maximum THC Speed to Fast IHS Speed.
- THC Torque Limit and Nozzle Contact Enable outputs turn on.
- The CNC monitors the Nozzle Contact Sense input. This input activates when the torch touches the workpiece, so the CNC knows the height of the workpiece.
- The CNC monitors the axis following error which the CNC compares to the stall force. When the following error exceeds the stall force, the CNC knows the height of workpiece.
- After sensing the workpiece, the torch retracts at the Slow IHS speed to the Transfer Height.
  - When using Nozzle Contact Sense to sense the workpiece, the CNC measures the Transfer Height from the point the Nozzle Contact Sense turns off during the retract.
  - When using Stall Force to sense the workpiece, the CNC measures the Transfer height from the point where the following error exceeded the Stall Force.

Performing a first IHS

1. Home the THC 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. Choose the Test Lifter soft key. The THC performs the initial height sense starting from the THC axis home position.

The Sensor THC and the ArcGlide perform the IHS using different speeds.

Sensor THCs speeds are set in Setups > Machine Setups > Speeds screen. ArcGlide speeds are set in the Setups > Machine Setups > ArcGlide axis screen.

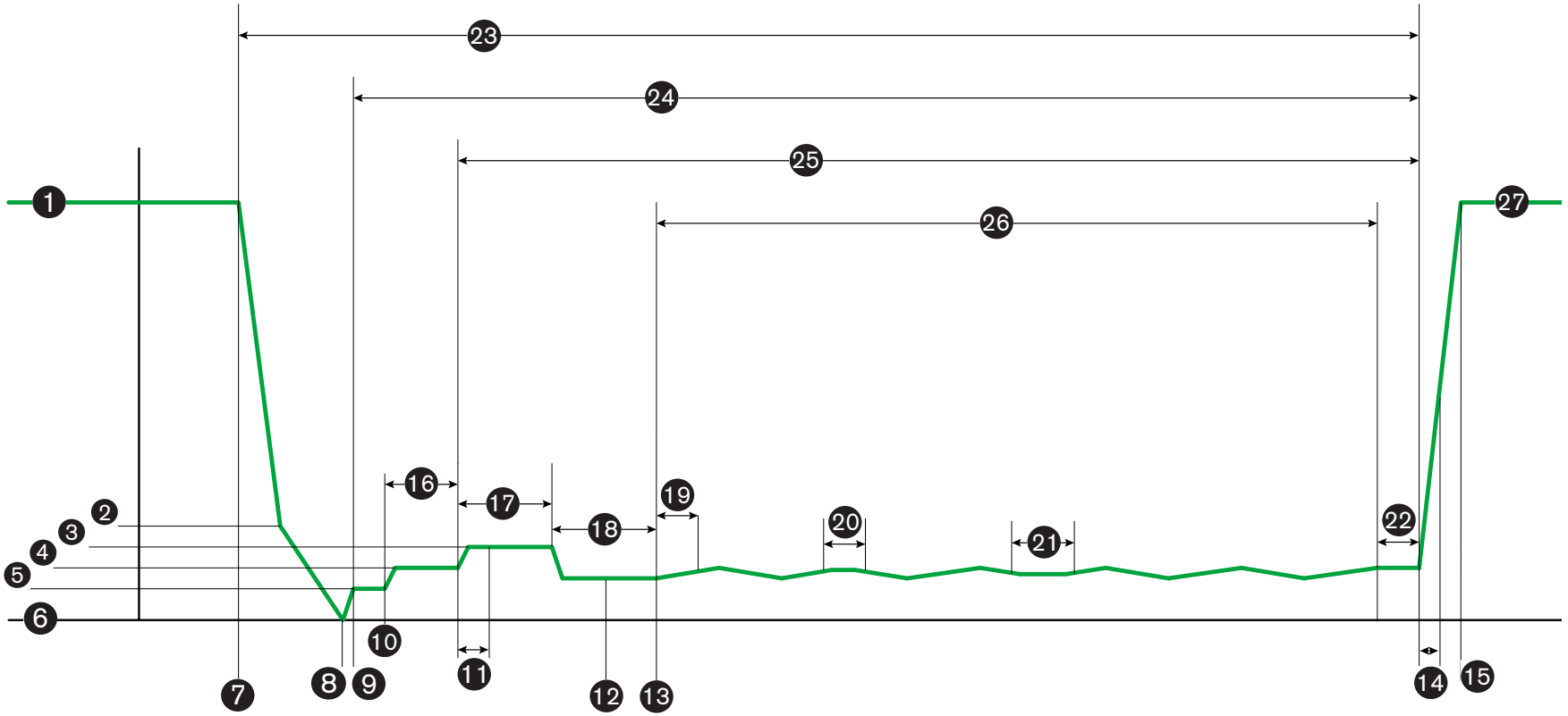
Sensor THC moves 1/10th of the slide length at the maximum speed, then changes to the Fast IHS speed until it reaches the IHS Start Height (set on the Process screen). The THC changes to the Slow IHS speed until it detects the workpiece. It then raises to the Transfer Height (also set in the Process screen).

ArcGlide moves all the way to the workpiece at the Slow IHS speed. On subsequent IHSs, the ArcGlide uses the Fast Speed until it reaches the Start IHS height. Then it switches to Slow speed. After the torch contacts the workpiece, the ArcGlide raises the torch to the Transfer Height.

A THC error, a manual move, an idle timeout of 30 seconds, or a power cycle all result in the next IHS occurring at the Slow Speed to find the workpiece height again.

THC sequence of operations

The following diagram shows the heights and timers used by THCs during cutting in automatic mode.



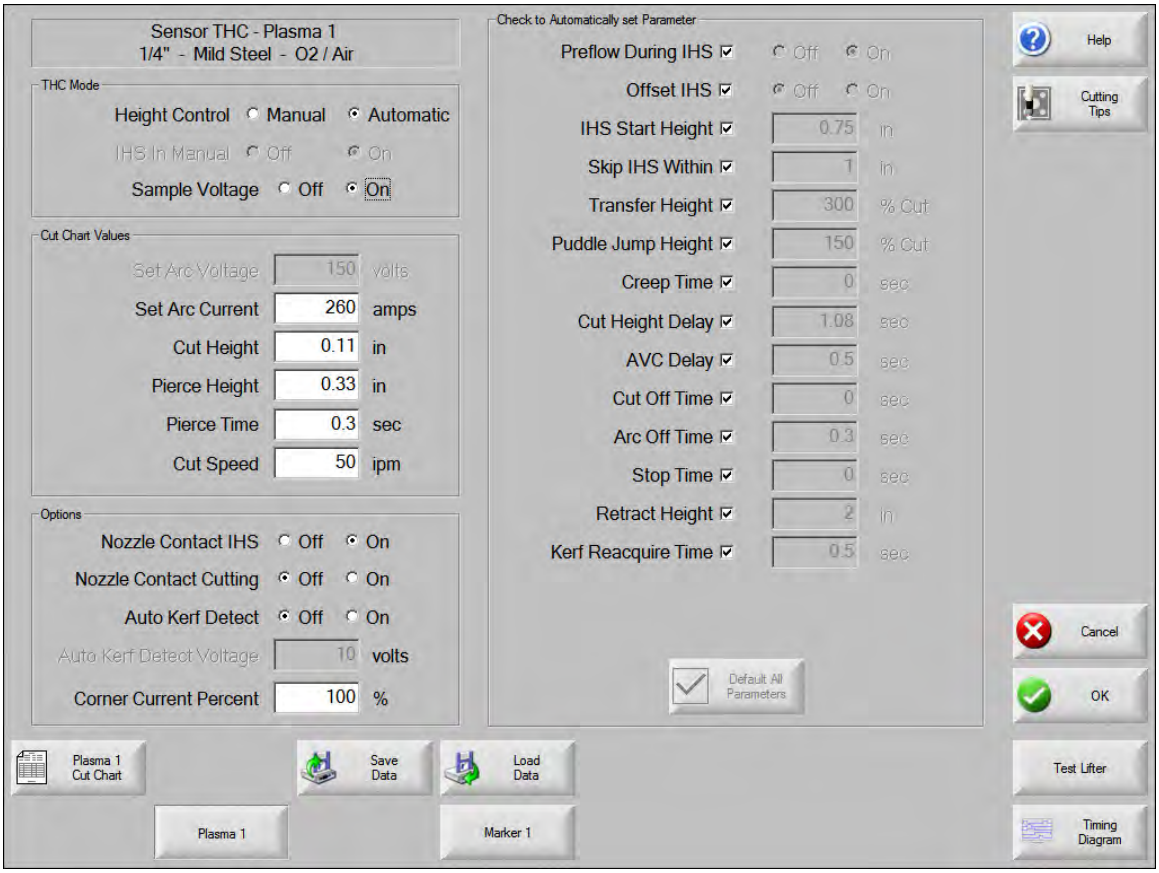
1	Torch path	8	Tip touch	15	Retract complete	22	Cut Off time
2	IHS start height	9	Plasma start	16	Pierce delay	23	CNC cut control active
3	Puddle jump height	10	Arc transfer	17	Cut height delay	24	Plasma torch active
4	Pierce height	11	Creep time	18	AVC delay	25	Cutting X/Y motion
5	Transfer height	12	Cut height	19	AVC sampling	26	AVC active
6	Workpiece	13	AVC begins	20	Auto kerf detect	27	Retract height
7	CNC cycle start (Cut Control)	14	Stop time	21	Torch height disable		

THC sequence of operation in automatic mode

THC Process screen

The Process screen contains a combination of plasma process and THC parameters that control the THC operations. In this screen you can customize the operation for the duration of a single cutting operation. After a part or nest program is complete, if the cut chart is accessed, the selections on this screen return to the values of the selected cut chart.

To open the THC Process screen, choose Main > Setups > then choose the plasma process soft key that corresponds to the process you want to use.



There are four sets of parameters on the Process screen:

- THC mode
- Cut chart
- Options
- Automatically-set

THC Mode

Height Control

Manual Mode: When you set the THC into Manual Mode, you control the THC with the Raise and Lower station controls on the CNC operator console, or the jog keys on the CNC screen. Using Manual Mode in this way requires that the torch be close enough to the workpiece to transfer the arc. Once cutting starts, the torch stays at the height where you positioned it.

Automatic Mode: The torch moves through its programmed sequence, based on values set on the Process screen.

Setting: Manual/**Automatic**

Voltage Control (ArcGlide THC) Voltage control is available only in Automatic Mode. When Voltage Control is ON, the torch height is controlled by the measured arc voltage. When Voltage Control is OFF, the torch maintains a constant position during the cut that is independent of the arc voltage.

Setting: Off/**On**

IHS in Manual (Sensor THC): The THC must be in Manual Mode. If IHS in Manual is ON, IHS and the sequence of operations is automatic but the torch height is not controlled by measured arc voltage. If IHS in Manual is OFF, all operations are controlled manually.

Setting: Off/**On**

Sample Voltage: The Height Control must be in Automatic Mode and the Voltage Control must be ON. When Sample Voltage is ON, the THC measures the voltage at the end of the AVC Delay and uses it as a set point for the remainder of the cut. When Sample Voltage is OFF the Set Arc Voltage is used as the set point for torch height control.

Setting: Off/**On**

Cut Chart Values

These fields display the values in the cut chart that is active for the process. These values can be changed here, for this job, and the values for the estimated parameters will be automatically recalculated and displayed. However, these changes are not saved in the cut chart.

Set Arc Voltage: The ArcGlide THC must be in Automatic Mode, the Voltage Control must be ON, and the Sample Voltage must be OFF. The Sensor THC must be in Automatic Mode and Sample Voltage must be OFF.

When Sample Voltage is OFF, the Set Arc Voltage is used as the set point for torch height control.

Setting: 50 to 300 VDC

Set Arc Current: This is the value for the plasma arc current. Enter the amperage needed to cut the material. This parameter can be used only with plasma systems that communicate with the CNC.

Setting: 5 to 1000 Amps

Cut Height: Determines the height at which the torch cuts the workpiece.

Setting: 0.25 to 25.4 mm (0.01 to 1.0 in)

Pierce Height: Determines the height at which the torch pierces the workpiece. The torch moves to this height after Transfer Height.

Setting: 0.25 to 25.4 mm (0.01 to 1.0 in)

Pierce Time: This is the value for the Pierce Delay. During this time, the X/Y cutting motion is delayed to allow the plasma to fully pierce the workpiece.

Setting: 0 to 10 seconds

Cut Speed: This value specifies the cutting speed.

Setting: 50 mm/min (2 in/min) to Maximum Machine Speed

Options

Nozzle Contact IHS: To set this parameter, the Height Control must be in Automatic Mode. When Nozzle Contact IHS is ON, the THC uses electrical Ohmic contact to sense the workpiece. When this parameter is OFF, the THC uses stall force to sense the workpiece. This setting is normally disabled on water tables or painted workpiece due to the unreliability of the electrical contact.

Setting: Off/On

Nozzle Contact Cutting: The THC uses ohmic contact to sense and retract from the workpiece during the cut. This may be disabled for use with water tables, a dirty workpiece, or a cutting or marking process with a very low cut or mark height.

Setting: Off/On

Auto Kerf Detect: To set this parameter, the THC must be in Automatic Mode. When Auto Kerf Detect is active, the THC looks for a rapid rise in the measured arc voltage that indicates that the torch is cutting across a previously cut kerf. This parameter temporarily disables the AVC and prevents the torch from diving into the workpiece.

Setting: Off/On

Auto Kerf Detect Voltage: This parameter is active if Auto Kerf Detect is turned ON. The lower the voltage, the higher the detection sensitivity. This value should be set high enough to detect normal kerf crossings but low enough to prevent false kerf detection.

Setting: 1 to 10 volts

Setting: 1 to 10 volts

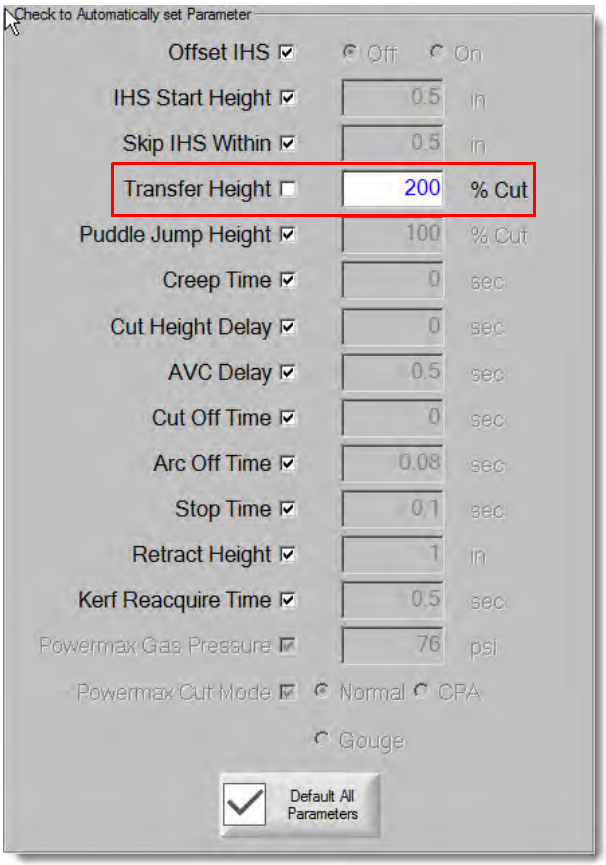
Corner Current Percent: Specifies a reduced current setting when cutting corners to improve cut quality.

The ArcGlide DOES NOT support this feature. Corner Current percent applies ONLY to Sensor THC.

Setting: 50% to 100% of Set Arc Current

Automatically-set

The CNC can automatically estimate these values for the current plasma process. Choose the Default All Parameters soft key to load the estimated values. Under most conditions, these estimated values yield good results. However, you can override any of these values for special conditions. To override an estimated value, clear the check box for that parameter and enter a new value. The CNC displays your entered value in blue:



If a part program reloads the cut chart, the Transfer Height, Creep Time, and Cut Height Delay values will replaced with the cut chart values for these parameters.

Preflow During IHS: This parameter is used for Rapid Part cutting. When this parameter is active (On), the CNC issues the Start and Hold Ignition signals to the plasma system early to allow the gas preflow to occur while the THC is performing an IHS operation. This reduces the time required to move to the next part and start cutting.

Setting: Off/On

Offset IHS (Sensor THC): This parameter enables a remote probe for workpiece detection and initial height sense. If this feature is used, the CNC reads the offset values for Marker Offset 9, set on the Cutting screen (select Main > Setups > Cutting). An Offset IHS is often used when cutting a pre-pierced workpiece so that the torch doesn't perform an IHS on a pierce point. The torch moves the distance of the offset, performs the IHS and returns to the pierce location. The Z position of the marker offset coordinates height differences between the torch and the probe.

Setting: Off/On

IHS Start Height: This is the height above the workpiece where the THC starts the initial height sense process. When the torch reaches this distance above the workpiece the following actions occur:

- Speed slows from Maximum THC Speed to Fast IHS Speed.
- THC Torque Limit and Nozzle Contact Enable outputs turn on.
- The CNC monitors the Nozzle Contact Sense input. This input activates when the torch touches the workpiece, so the CNC knows the height of the workpiece.
- The CNC monitors the axis following error which the CNC compares to the stall force. When the following error exceeds the stall force, the CNC knows the height of workpiece.

Setting: 2.54 to 50.8 mm (0.1 to 2.0 in)

Skip IHS Within: This parameter optimizes production by reducing the time between cuts. If the next starting point is within this distance of the end of the previous cut, the THC skips the IHS. When this happens, the torch goes directly to the Transfer Height and skips contact with the workpiece. This setting can improve the overall machine production rate. Set this parameter to 0 to disable this feature. Skip IHS will be ignored if:

- An M07 HS command is in the part program for that pierce (see the *Phoenix Series 9 Programmer's Reference* for more information).
- Sampled arc voltage mode is active and an IHS is needed for arc voltage sampling (six arc voltage samples are required before IHS can be skipped).
- The THC is locked out by the M50 (Height Sensor Disable) command.
- The THC is not in Automatic mode.
- The ArcGlide is not connected to a CNC with Hypernet.

Setting: 0 to table size (mm or inch)

Transfer Height: When the arc transfers to the workpiece, it can be "stretched" to the pierce height. The transfer height is lower than the pierce height because initiating arc transfer at a high pierce height may result in the arc not transferring to the workpiece at all. Enter the Transfer Height as a percentage of the cut height or as an actual transfer height distance. The Transfer Height originates from the cut chart.

Setting: 50% to 400% of Cut Height, commonly 150%

Puddle Jump Height: This value determines the height above the workpiece where the torch is raised after pierce and before lowering to Cut Height so that the torch clears the top dross puddle that can form during the pierce. Enter a percentage of the Cut Height. The torch remains at this height until the Cut Height Delay has elapsed. If you are not using Puddle Jump Height, set this parameter to 100%.

Setting: 50% to 500% of Cut Height

Creep Time: Specifies the amount of time after piercing the workpiece that the torch travels at Creep Speed. (See Setups > Machine Setups > Speeds for the Creep Speed.) The torch accelerates to cut speed after the Creep Time elapses. Creep Speed can help stabilize the arc during the transition to Cut Speed. The Creep Time originates from the cut chart.

Setting: 0 to 10 seconds

Cut Height Delay: This value sets the number of seconds to hold the torch at the Puddle Jump Height before transitioning to the Cut Height so that the torch clears the top dross puddle that can form during a pierce. If you are not using Puddle Jump Height, set this parameter to 0. The Cut Height Delay originates from the cut chart.

Setting: 0 to 10 seconds

AVC Delay: This value sets a number of seconds to allow the plasma system to achieve steady-state operation at the cut height before automatic voltage control begins. After this delay, the AVC is enabled for the remainder of the cut. If the THC is in Sampled Voltage mode, the arc voltage sample is taken after this delay.

Setting: 0 to 10 seconds

Cut Off Time: This value turns off the plasma arc before or after the end of the programmed cut to improve edge quality. If the value is negative, the torch turns off before the end of the cutting machine motion. With positive values, the plasma arc turns off after motion stops. This parameter minimizes notches in part edges that can occur when motion stops with the arc on.

Setting: -1 to 2 seconds

Arc Off Time: This value defines the number of seconds to wait before sending a lost arc signal. This setting allows arc loss during the remainder of the part to be ignored so the CNC can move to the next pierce point.

Setting: 0 to 2 seconds

Stop Time: This parameter allows a pause at the end of a cut and delays X/Y motion to the next pierce point. This delay can be used to retract the torch to avoid tip-ups.

Setting: 0 to 10 seconds

Retract Height: This parameter specifies the height above the workpiece to which the torch retracts at the end of a cut.

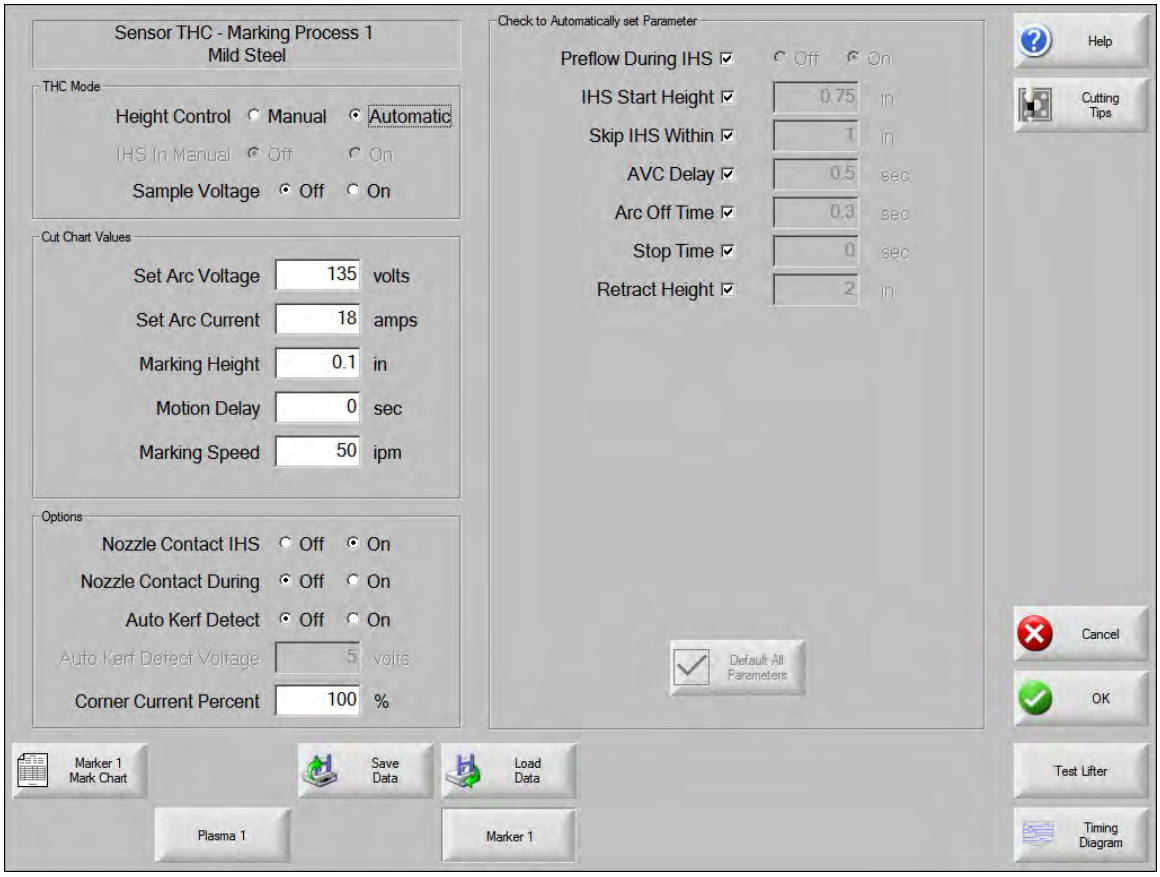
Setting: 2.54 mm (0.1 in) to the maximum lifter length.

Kerf Reacquire Time (Sensor THC) When Auto Kerf Detect reads a sudden drop in voltage, it activates the Torch Height Disable output for the Kerf Reacquire time. When the Kerf Reacquire Time elapses, the CNC turns off the Torch Height Disable output and the THC starts tracking voltage again.

Marker Setup

The Marking Process screen contains parameters that control the operation of the THC and its sequence of operations. On this screen you can customize a single marking operation. After a part program is complete, if the cut chart is accessed the selections on this screen return to the values of the selected cut chart.

To open the THC Marking screen, choose Setups > Marker 1 or 2.



THC Mode

Height Control: When you set the THC into Manual Mode, you control the THC with the Raise and Lower station controls on the CNC operator console, or the jog keys on the CNC screen. Using Manual Mode in this way requires that the torch be close enough to the workpiece to transfer the arc. Once cutting starts, the torch stays at the height where you positioned it.

For the Sensor THC, when you select IHS in Manual and press Start, the torch performs an initial height sense then moves the to cut height and stays there for the job.

Setting: Manual/**Automatic**

Voltage Control (ArcGlide THC): The THC must be in Manual Mode. If Voltage Control is ON, the torch height is controlled by the measured arc voltage. If Voltage Control is OFF, the torch maintains a constant position during the cut that is independent of the arc voltage.

Setting: Off/**On**

IHS in Manual (Sensor THC): The THC must be in Automatic Mode. If IHS in Manual is ON, IHS and the sequence of operations is automatic but the torch height is not controlled by measured arc voltage. If IHS in Manual is OFF, all operations are controlled manually.

Setting: Off/**On**

Sample Voltage: The Height Control must be in Automatic Mode and the Voltage Control must be ON. When Sample Voltage is ON, the THC measures the voltage at the end of the AVC Delay and uses it as a set point for the remainder of the cut. When Sample Voltage is OFF the Set Arc Voltage is used as the set point for torch height control.

Setting: Off/**On**

Cut Chart Values

These fields display the values in the cut chart that is active for the process. These values can be changed here, for the current part, and the values for the estimated parameters will be automatically recalculated and displayed. However, these values are not saved in the cut chart.

Set Arc Voltage: The ArcGlide THC must be in Automatic Mode, the Voltage Control must be ON, and the Sample Voltage must be OFF.

The Sensor THC must be in Automatic Mode and Sample Voltage must be OFF.

When Sample Voltage is OFF, the Set Arc Voltage is used as the set point for torch height control.

Setting: 50 to 300 VDC

Set Arc Current: This is the value for the plasma arc current. Enter the amperage needed to cut the material. This value originates in the cut chart and can be temporarily fine-tuned on this screen. This parameter can only be used with plasma systems that communicate with the CNC.

Setting: 5 to 999 Amps

Marking Height: Determines the height at which the torch marks the workpiece. This value originates in the cut chart and can be temporarily fine-tuned on this screen.

Setting: 0.25 to 25.4 mm (0.01 to 1.0 in)

Motion Delay: Delay from firing the torch to X/Y marking motion. Normally set to zero.

Setting: 0 to 10 seconds

Marking Speed This value specifies the marking speed. It originates in the plasma process cut charts and can be temporarily fine-tuned on this screen.

Setting: 50 mm/min (2 in/min) to Maximum Machine Speed

Options

Nozzle Contact IHS: To set this parameter, the Height Control must be in Automatic Mode. When Nozzle Contact IHS is ON, the THC uses electrical Ohmic contact to sense the workpiece. Set this option to OFF when cutting on a water table.

Setting: Off/On

Nozzle Contact Marking: The THC uses Ohmic contact to sense and retract from the workpiece during marking. This may be disabled for use with water tables, a dirty workpiece, or a cutting or marking process with a very low standoff.

Setting: Off/On

Auto Kerf Detect: To set this parameter, Height Control must be in Automatic Mode. When Auto Kerf Detect is active, the THC looks for a spike in the measured arc voltage when the torch is marking across a previously cut kerf. This parameter temporarily disables AVC and prevents the torch from diving into the workpiece.

Setting: Off/On

Auto Kerf Detect Voltage: This parameter is active if Auto Kerf Detect is turned ON. Enter the voltage change required to detect a kerf crossing.

Setting: 0 to 10 volts

Corner Current Percent: Specifies a reduced current setting when cutting corners to improve cut quality. Equals a percentage of the Set Arc Current and activates when the Torch Height Disable Speed output is on. The Torch Height Disable Speed is set on the Machine Setups > Speeds screen. This parameter can only be used with plasma systems that communicate with the CNC.

The ArcGlide DOES NOT support this feature. Corner Current percent applies ONLY to Sensor THC.

Setting: 50% to 100% of Set Arc Current

Automatically-set

The CNC automatically calculates these values for the current marking process. The calculated values are visible in the fields next to each parameter. Click the Default All Parameters soft key to select the calculated values for all parameters. Under most conditions, the calculated values yield good results. However, you can override any of these values for special conditions. To override a calculated value, remove the check in the check box for that parameter and enter a new value.

Preflow During IHS: This parameter is used for Rapid Part marking. When this parameter is active (On), the CNC issues the Start and Hold Ignition signals to the plasma system early to allow the gas preflow to occur while the THC is performing an IHS operation. This reduces the time required to move to the next part and start marking.

Setting: Off/On

IHS Start Height: This is the height above the workpiece where the THC starts the initial height sense process. When the torch reaches this distance above the workpiece the following actions occur:

- Speed slows from Maximum THC Speed to Fast IHS Speed.
- THC Torque Limit and Nozzle Contact Enable outputs turn on.
- The CNC monitors the Nozzle Contact Sense input. This input activates when the torch touches the workpiece, so the CNC knows the height of the workpiece.
- The CNC monitors the axis following error which the CNC compares to the stall force. When the following error exceeds the stall force, the CNC knows the height of workpiece.

Setting: 2.54 to 50.8 mm (0.1 to 2.0 in)

Skip IHS Within: This parameter optimizes production. If the next starting point is within this distance of the end of the previous cut, the THC skips the IHS. When this happens, the torch goes directly to the Transfer Height and skips contact with the workpiece. This setting can improve the overall machine production rate.

Skip IHS will be ignored if:

- The ArcGlide is not connected to a CNC with Hypernet.
- An M07 HS command is in the part program for that pierce (see the *Phoenix Series 9 Programmer's Reference* for more information).
- Sampled arc voltage mode is active and an IHS is needed for arc voltage sampling (six arc voltage samples are required before IHS can be skipped).
- The THC is locked out by the M50 (Height Sensor Disable) command.
- The THC is not in Automatic mode.
- Set this parameter to 0 to disable this feature.

Setting: 0 to table size (mm or inch)

AVC Delay: This value sets the number of seconds that are required for the plasma system to achieve steady-state operation at the mark height. After this delay, the automatic voltage control is enabled for the remainder of the mark. If the THC is in Sampled Voltage mode, the arc voltage sample is taken after this delay.

Setting: 0 to 10 seconds

Arc Off Time: This value defines the number of seconds to wait before sending a lost arc signal. This setting allows arc loss during the remainder of the part to be ignored so the CNC can move to the next pierce point.

Setting: 0 to 2 seconds

Stop Time: This parameter allows a pause at the end of a mark and delays X/Y motion to the next pierce point.

Setting: 0 to 10 second

Retract Height: This parameter specifies the height above the workpiece to which the torch retracts at the end of a mark.

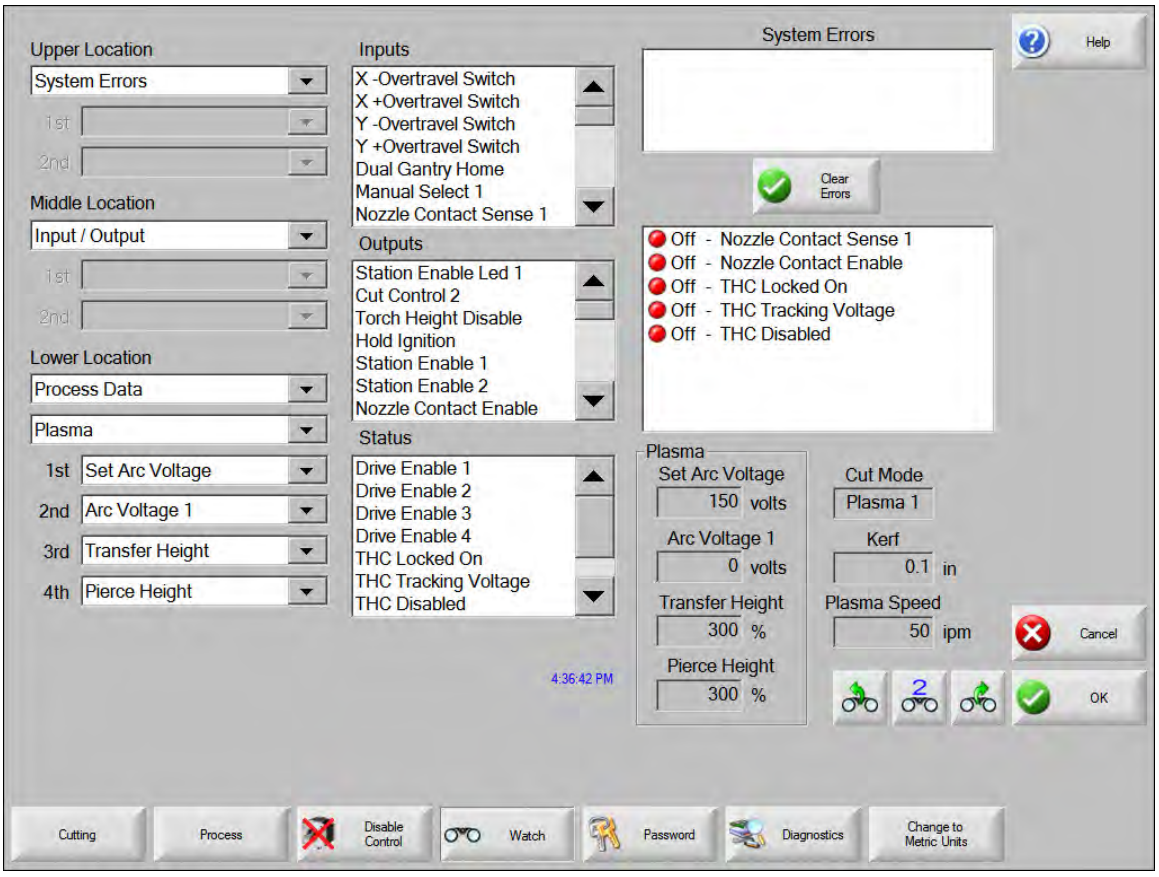
Setting: 2.54 mm (0.1 in) to the maximum lifter length

Watch Window

You can set up a Watch Window to monitor the torch height control.

Sensor THC

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 from the list:

THC Locked On: This status bit turns to ON when the torch height control, using Set Arc Voltage or Sampled Arc Voltage modes, is reading and transmitting the arc voltage to the CNC.

THC Tracking Voltage: This status bit turns ON as 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 and slows down to cut the corner. When the speed slows down, the arc voltage increases and could exceed the arc voltage set point which could cause a fault. You can program the speeds for the torch height control so that the THC disables when the cutting speed slows down.

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.

4. Choose Process Data from the Lower Location list.

5. Choose Plasma from the list below Process Data.

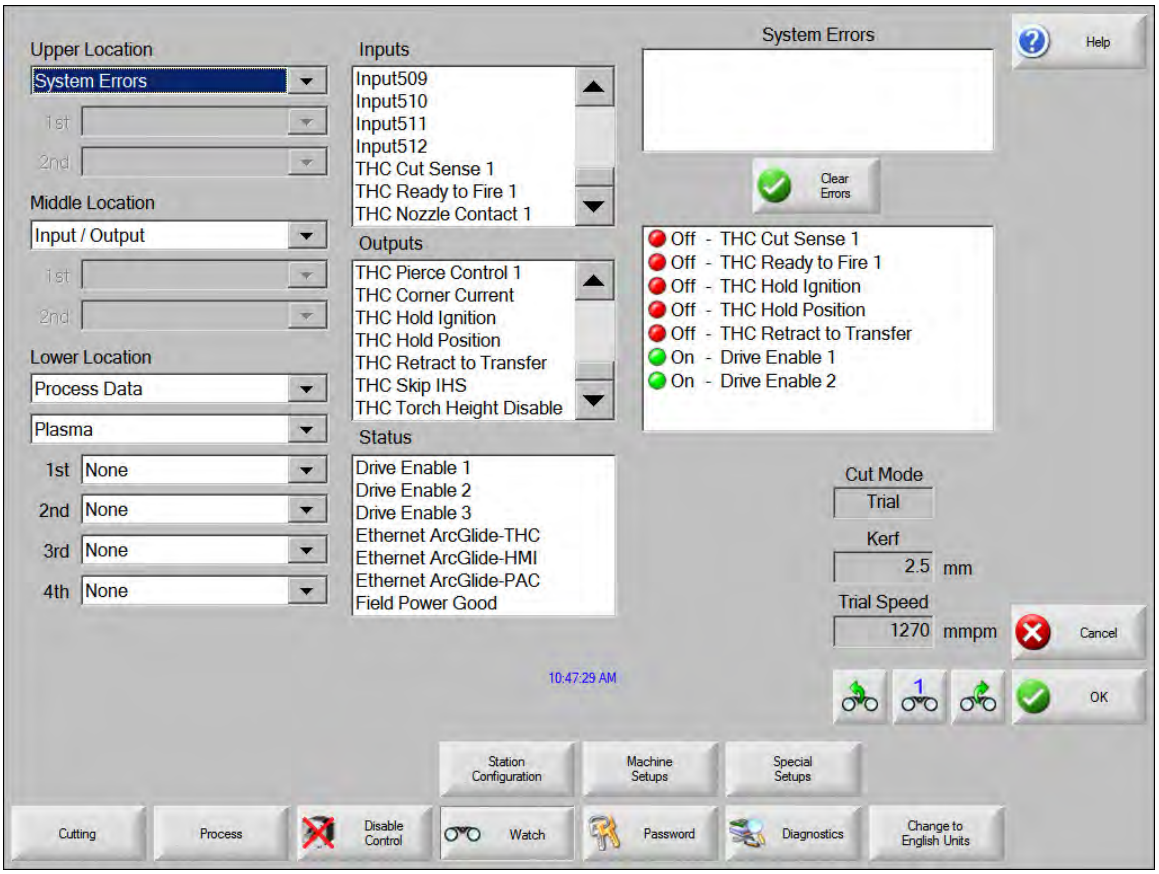
6. Choose Set Arc Voltage for the 1st and Arc Voltage 1 for the 2nd to compare the set arc voltage to the actual arc voltage.

   Process data parameters available in the Watch Window are listed in the following table. See *THC Process screen* on page 185 for parameter definitions.

Set Arc Voltage	Skip IHS Within
Transfer Height	Creep Time
Cut Off Time	Stop Time
Arc Voltage 1	Voltage Offset 1

ArcGlide

An example Watch Window for the ArcGlide 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.
4. Scroll to the bottom of the Inputs and Outputs lists to view the ArcGlide I/O. These signals have THC at the beginning of their names.
5. Choose the I/O signals to display in the Watch Window.

Status messages

The following table lists the status messages that the CNC displays on the Main screen during the operation of the torch height control. The table also describes the action of the THC when each status message appears during a part program, and what to do if the part program pauses during the display of any of these messages.

Status Message	Meaning	Occurs during a part program...	If the program pauses...
Traversing	Torch is moving to next pierce point	After Cycle Start and after each cut.	No action
Lowering torch	Torch is at pierce point and torch down output is activated.	Occurs at execution of Cut On (M07). The status message remains until the IHS is complete.	■ Press Stop, then Start. ■ If the message persists, check for a fault input such as Torch Collision, Fast Stop, Drive Disable, or Remote Pause. Set up I/O in the Watch Window to view these I/O while the part program runs.
Waiting for Arc On	CNC is waiting for the Cut Sense Input. Cut Sense is the Arc Transfer output from the plasma system or the Motion output from the THC.	Occurs after IHS is complete.	■ Add the Cut Sense input to a Watch Window. ■ Test the Cut Sense input at the CNC to verify that it is working.
Piercing	Pierce Control output is active	Occurs during pierce.	No action
Creeping	Creep Motion is occurring, after pierce delay.	Occurs after the Pierce timer expires and indicates the beginning of motion codes.	No action
Cutting	Torch is cutting and motion is occurring.	Execution of the motion.	No action

Status Message	Meaning	Occurs during a part program...	If the program pauses...
Arc Off	The arc is off.	If motion stops before the execution of M08 (or end of lead-out), a Cut Sense Lost message is posted. This message may occur during the execution of the motion codes (G-codes).	■ The arc lost electrical connection to the workpiece while cutting. ■ If the message occurs at the end of a cut, check the lead-out length or increase the value of Arc Off Time in the Process screen.
Raising Torch	Torch has reached the end of the cut.	Occurs at the execution of Cut Off (M08).	No action
Stop Delay	Motion is delayed before the torch's rapid traverse to the next pierce point.	The status message appears after the torch has reached its retract position.	No action
PS Link Failed	The CNC did not receive a serial response from the plasma supply or a checksum error occurred when attempting to communicate with the plasma supply.	The program is paused if this message occurs during a cut.	■ Verify that the power supply power is ON. ■ Check the serial cable connections. ■ Check the Hypernet connections and the Ethernet switch. ■ On an HPR, check the termination jumpers, J106/J107 or J104/J105 on the control board. They should not be used with a Hypernet interface. ■ In a multi-torch system, be sure Manual or Auto Station Select inputs are used and active. ■ If you are prompted when pressing Cycle Start (multi-torch system), the prompt asks you to proceed without link. This is a normal prompt. Press NO if one of the torches is purposely powered down. ■ Faulty RS-422 serial device or Hypernet interface (contact your OEM).

ArcGlide Diagnostics screen

The ArcGlide Diagnostics screen displays the software version and status for all Hypernet components of the cutting system.

To view the ArcGlide Diagnostics screen, choose Setups > Diagnostics > ArcGlide.

Raise Torch Press the Raise Torch key to raise the torch.

Lower Torch Press the Lower Torch key to lower the torch.

IHS Test Press this soft key to test the IHS function.

	**WARNING**
	**The Hold to Test Fire soft key will fire the torch. Observe all safety precautions before firing the torch.**

Hold to Test Fire Press this soft key to test-fire the torch and verify that the system is connected properly.

Hold to Test Fire Press this soft key to test-fire the torch and verify that the system is connected properly.

Disable Drive and Brake This soft key allows an operator to move the lifter by hand to check for mechanical binding issues.

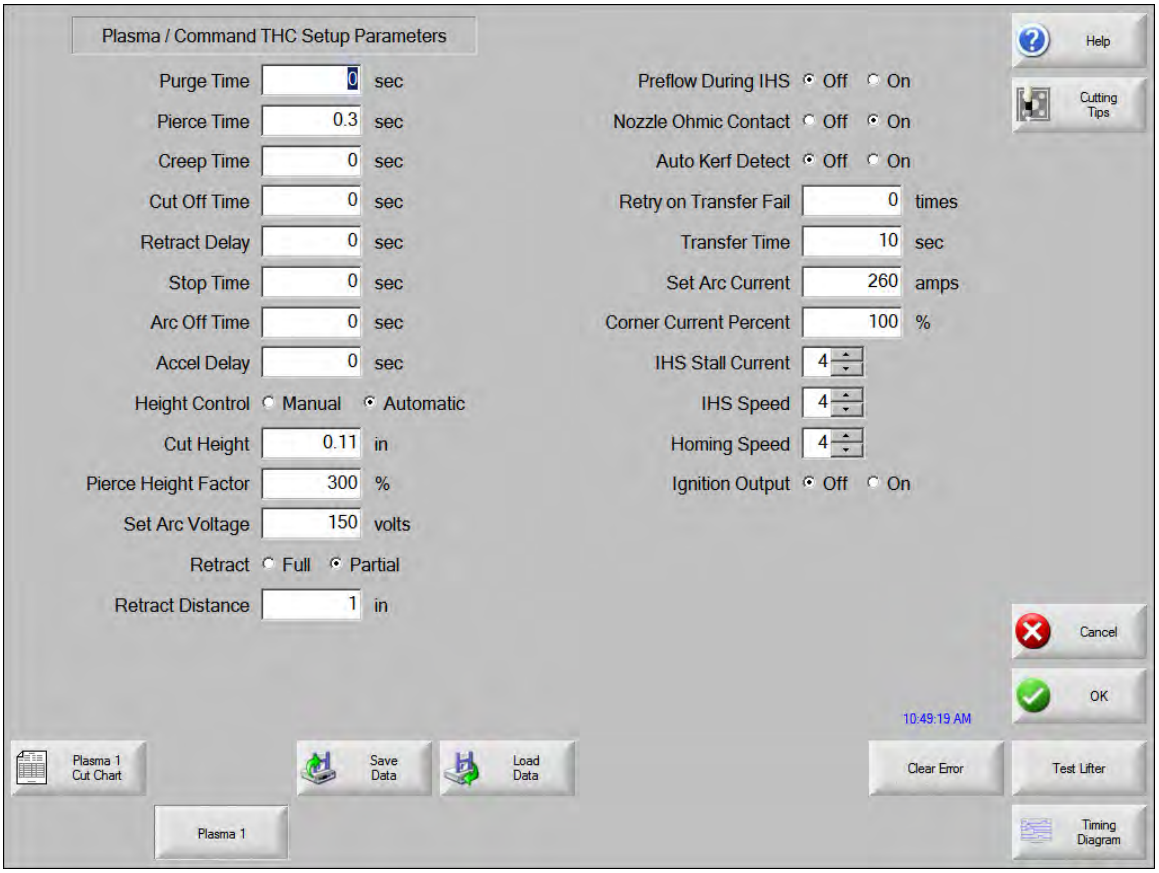
Remote Disable PAC Press this soft key to power off the plasma system.

ArcGlide 1 to 4 A soft key is available for each ArcGlide THC that is configured in your system. Press one of these keys to view diagnostics information, and to operate controls, for the corresponding THC.

Section 9
Command THC Setup

The Command THC is an automated torch height control system that adjusts the distance between the plasma torch and the work surface to provide improved cut quality. After the Command THC is set up using the password-protected parameters, you can set up operational parameters for the Command THC on the plasma Process screen.

For additional information on the use of the Command THC, refer to the operating instructions provided with the Command THC system.



Purge Time Specifies the time delay from torch ignition until motion is enabled, if the value for Arc On Feedback is Off. Enter 0 (zero) for the Purge Time if the value for Arc On Feedback is On.

Pierce Time Specifies the delay between the time that the torch is fully lowered until motion is initiated at Creep Speed. This value allows the plasma torch to pierce the material completely before moving.

Creep Time Specifies the amount of time that the torch travels at creep speed after piercing the material. Creep Speed is a percentage of the programmed cut speed and is determined by a setup parameter on the Speed Setup screen. After Creep Time has elapsed, the CNC accelerates to full cut speed.

Arc Off Time Specifies the amount of time to wait before indicating a lost cut signal. This delay helps minimize nuisance trips when the torch travels over previously cut paths in complex nested layouts.

Stop Time Specifies the amount of time that motion pauses at the end of a cut. This pause allows the torch to raise completely and clear cut irregularities before advancing to the next cut segment.

Accel Delay Delays the activation of the Automatic Voltage Control so the cutting table can reach a steady cutting speed. This parameter should be set as low as possible without allowing the torch to dive excessively at the beginning of a cut.

Retract Delay Time Specifies the delay between the end of the cut signal and retracting the torch.

Ignition Output Enables the Ignition output to ignite the plasma torch. If your plasma system requires a separate ignition signal, select On. If your system does not require a separate ignition signal, select Off.

Height Control Allows the operator to select manual or automatic mode for the Command THC. Manual mode disables the torch height control and allows the torch to cut at the specified cut height and voltage. Automatic mode allows the THC to command the torch up and down to maintain the voltage at a specified set point.

Retract Full/Partial Selects a full or partial retract distance for the torch. In the Full retract mode, the torch retracts to the Home position. In Partial retract mode, the torch retracts to the set retract distance.

Retry On Transfer Fail Specifies the number of times the CNC attempts to fire the torch if the torch fails to ignite.

Transfer Time Specifies the amount of time to attempt ignition of the torch. The ignition is confirmed by the Arc Sense Input (Arc on Feedback) to the CNC.

Set Arc Current Allows you to set the arc current at the plasma supply. This feature uses the Set Current BCD output from the CNC to activate the BCD inputs at the plasma supply and supports EIA RS-274D part program code G59 V*value* F*value* for setting current.

Corner Current Percent Allows the operator to improve cut quality at corners by selecting a reduced current setting for cutting corners. This value is a percentage of the Set Current (above) and is active when the Torch Height Disable output is on.

Set Arc Voltage Selects the necessary arc voltage for the material being cut.

Cut Height Selects the desired cut distance from the plate and sets the initial cut height before Arc Voltage control is activated.

Retract Distance Selects the THC Retract Distance when partial retract mode is configured.

Pierce Height Factor Factor that is multiplied by the Cut Height value to set the distance for the pierce height.

IHS Stall Current Sets the lifter downward force to detect when the torch makes contact with the plate during the IHS cycle. This is a relative factor between 1 and 10. The limited stall force is always used if nozzle ohmic sense is turned off.

IHS Speed Sets the lifter downward speed during the IHS cycle. This is a relative factor between 1 and 10.

Homing Speed Determines the retract or homing speed. This is a relative factor between 1 and 10.

Nozzle Ohmic Contact Select On for the Command THC when using Ohmic Contact Sense to detect the plate during the IHS cycle.

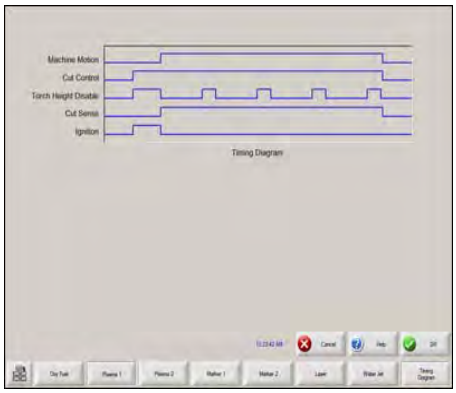
Preflow During IHS Select On to activate preflow during the IHS cycle.

Auto Kerf Detect Select On to reduce the possibility of the torch diving into the plate. When this feature is enabled, the THC detects sudden changes in arc voltage when it crosses a kerf path and freezes the THC.

Clear Error The Clear Error soft key allows you to clear an error at the Command THC control box. After the soft key is pressed, a message is posted on the CNC displaying a description of the error.

Test Lifter Press the Test Lifter soft key to command the torch lifter to lower to the plate, sense the plate and retract to its pierce height.

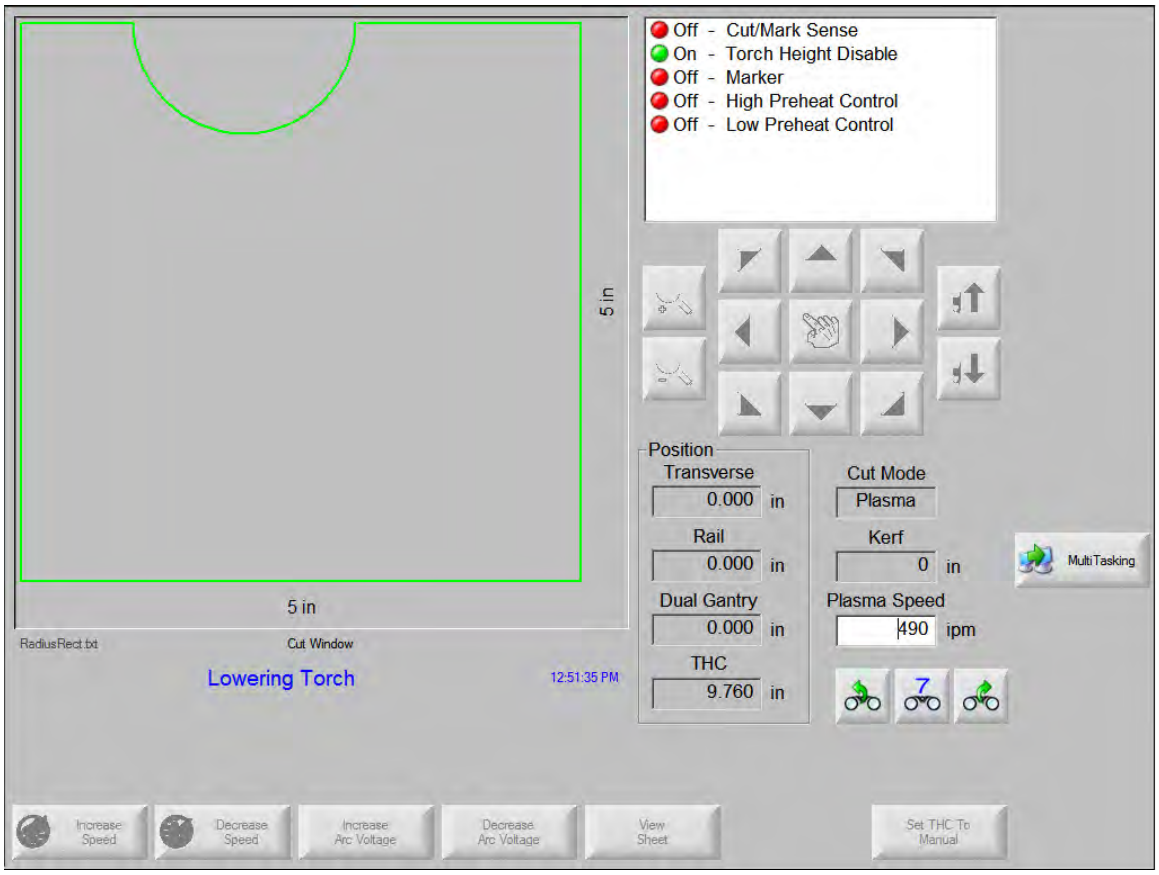
Timing Diagram Press the Timing Diagram soft key to view the timing diagram from the process parameters.



Command THC Main Cut Screen

You can operate the Command THC in automatic or manual mode.

Automatic THC Mode



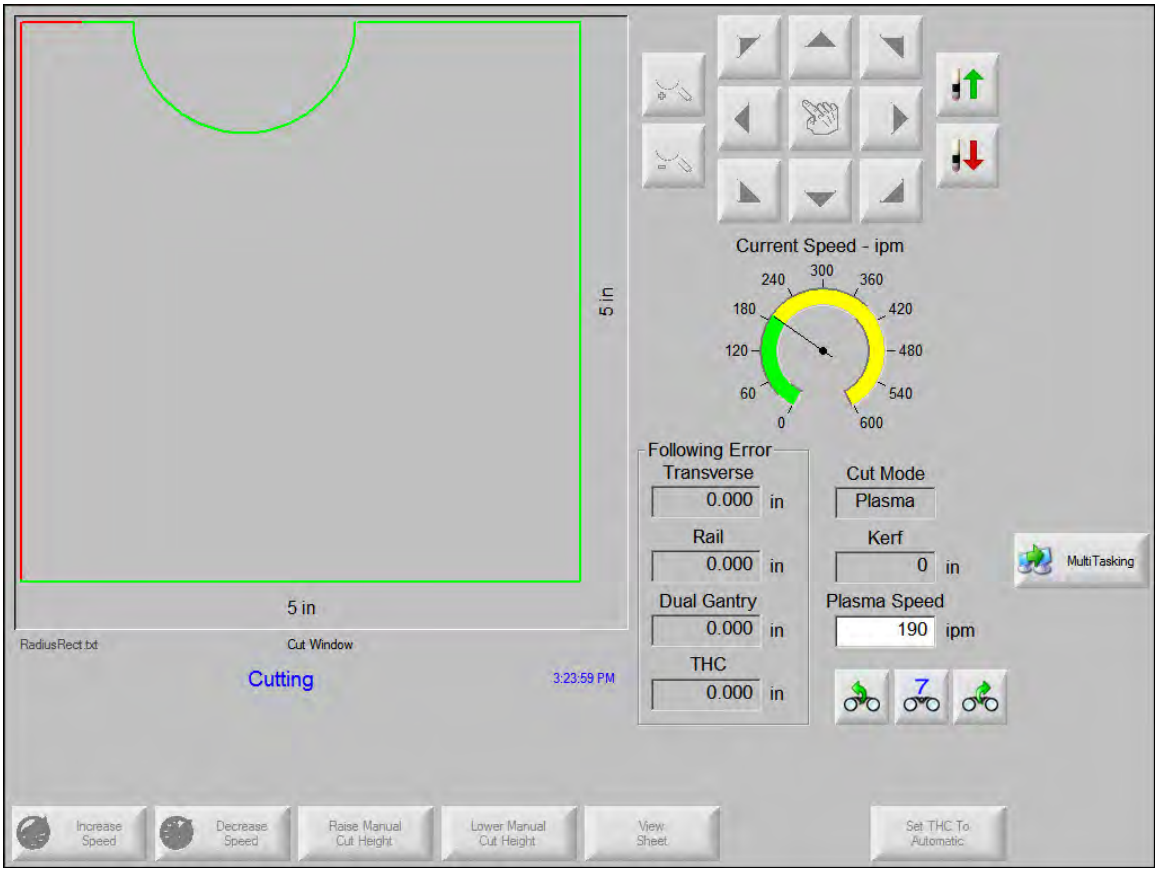
Increase/Decrease Arc Voltage These two soft keys display on the main cut screen while the Command THC is operating in automatic mode. These soft keys allow you to increase and decrease the Arc Voltage for the cut.

Extend Press this soft key during the pierce cycle to extend the pierce timer until it is stopped either by the Set-Now or Release soft key.

Set Now Press the Set Now soft key to end the pierce cycle and save the new pierce time. The Set Now soft key is often used in with the Extend soft key to modify the preset pierce time.

Release Ends a pierce cycle without modifying the original pierce time. The original pierce time is saved for the remaining pierces.

Manual THC Mode



Raise/Lower Torch These two soft keys display on the main cut screen while the Command THC is operating in Manual mode. These soft keys allow you to raise and lower the torch for the cut.

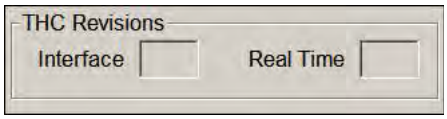
Extend Press this soft key during the pierce cycle to extend the pierce timer. To stop the timer, press either the Set-Now or Release soft key.

Set Now Press the Set Now soft key to end the pierce cycle and save the new pierce time. The Set Now soft key operates with the Extend soft key to modify the preset pierce time.

Release Press the Release soft key to end a pierce cycle but keep the original pierce time.

Machine Interface

The current Command THC Interface and Real Time Revision Levels are displayed at the Control Information screen when it is enabled.



CAUTION

Configure the port for RS-422 operation before connecting to the Command THC. The Command THC Link must first be enabled on the Machine Setups > Ports screen, and selected as the lifter in the Station Configuration screen. For more information on configuring the serial port for RS-422 communication, refer to the Serial Ports section of the *Phoenix Software V9 Series Installation and Setup Manual*.

Section 10

Diagnostics and Troubleshooting

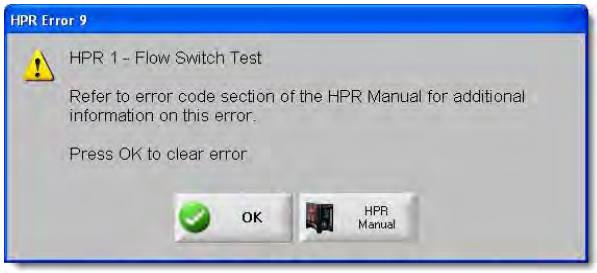
The following sections describe tools that you can use to diagnose and troubleshoot your CNC and Phoenix software.

Remote Help

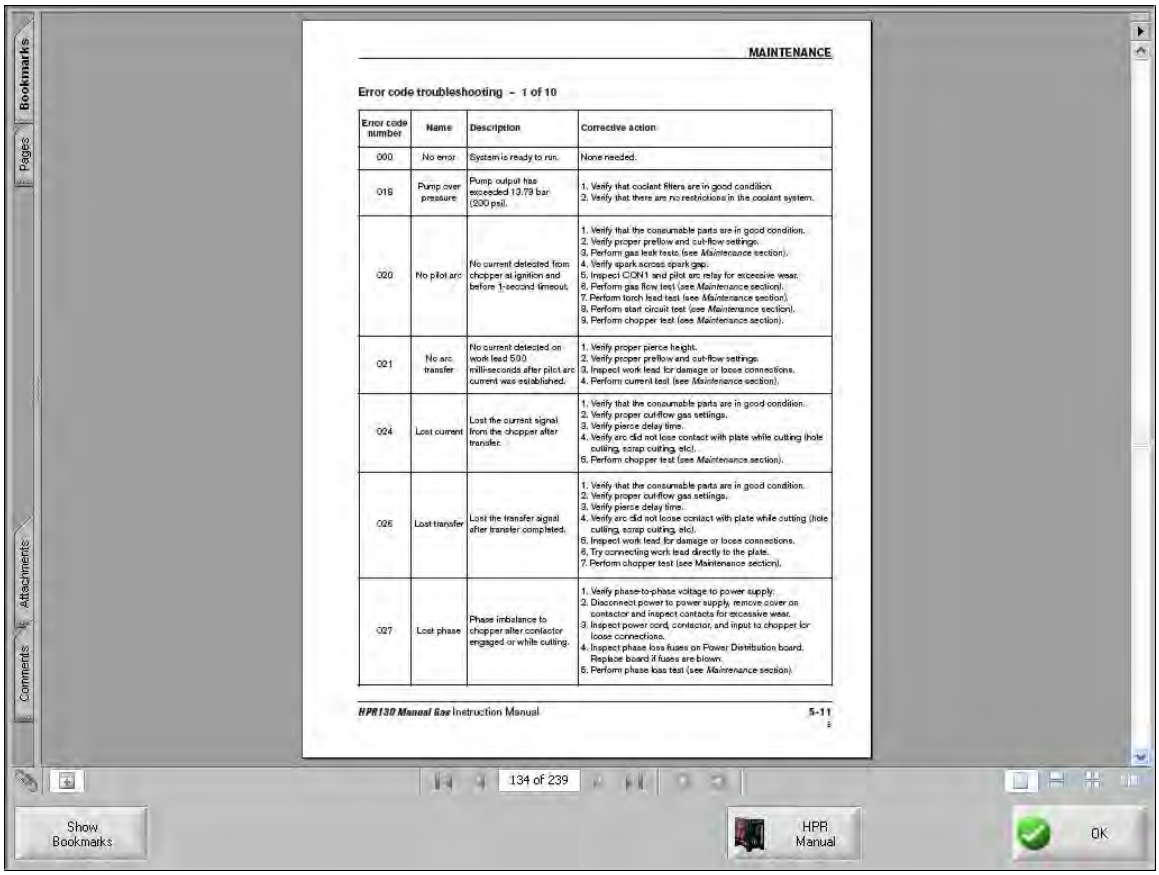
Contact Hypertherm Technical Support or your OEM or system integrator for assistance with Remote Help.

HPR Error Help

If an HPR error message displays on the CNC screen, you can click on the HPR Manual button to open the Help screen and view the troubleshooting information from the appropriate manual.



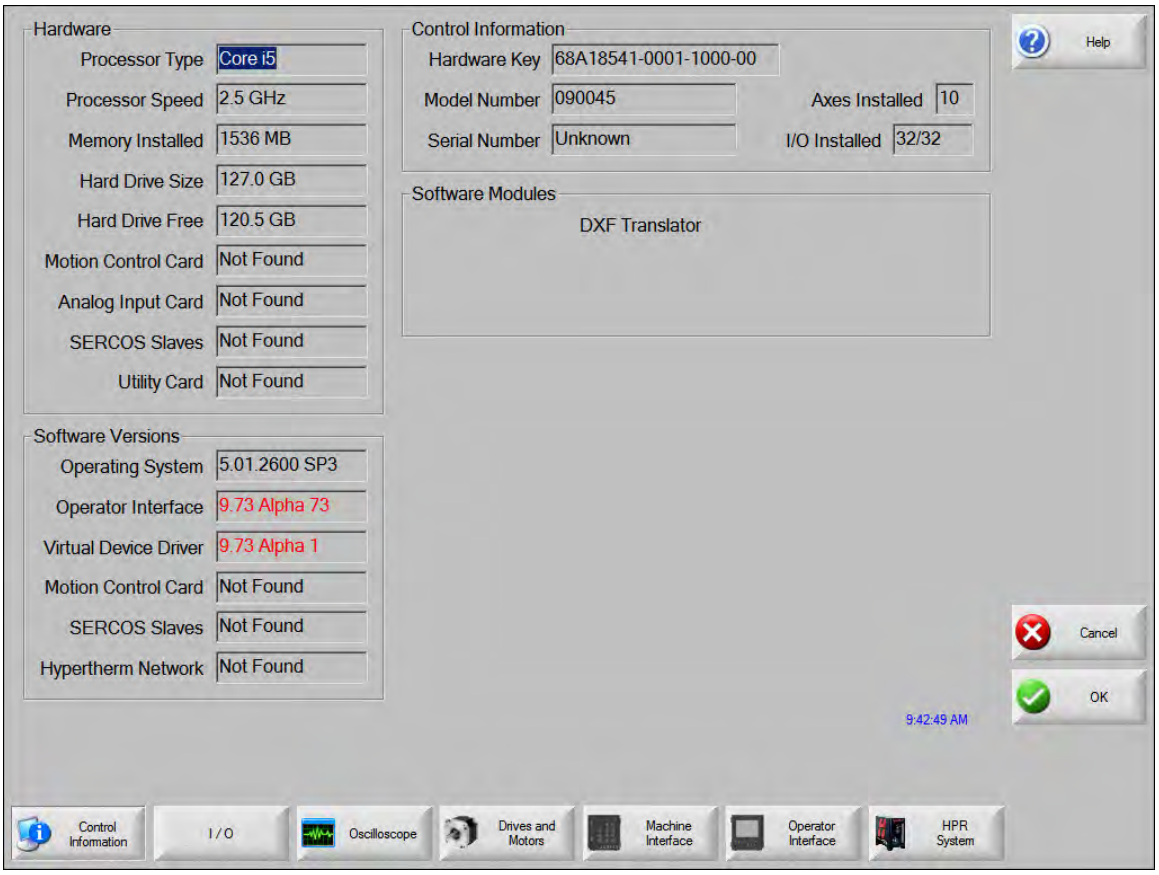
1. On the error message popup, click or press on the HPR Manual button.
2. On the Help screen, scroll through the troubleshooting information.
3. Click OK on the Help screen to close it.
4. Click OK on the Error message to clear the error.



CNC Information

This screen displays the current software versions and hardware configuration of the CNC. You must provide this information if you contact the factory for support.

From the Main screen, select Setups > Diagnostics > Control Information.



Hardware: The Hardware section shows the current hardware configuration which includes the Processor Type, Processor Speed, Memory Installed, Hard Drive Size, Hard Drive Free Space and Motion Control Card revision.

Software Versions: The Software Versions section shows the current version of the CNC's operating system, operator interface (software version), virtual device driver and the motion control card software.

Control Information: The control Information section displays the hardware key number, the CNC model number, serial number, control I/O type, axes enabled and I/O enabled.

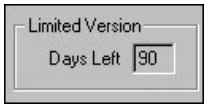
Software Modules: The Software Modules information displays the optional software that has been installed, such as DXF Translator, McAfee Virus Scan Software, or NJWIN font Viewer. If a number appears after the name of a software option, a timer is associated with this software and the number lets you know how many days/uses remain.

Voltage: System Voltages at the motherboard are displayed for motherboards equipped with this monitoring feature.

Temperature: Temperature at the motherboard is displayed for motherboards equipped with this monitoring feature.

Fans: Fan speeds at the motherboard are displayed for motherboards equipped with this monitoring feature.

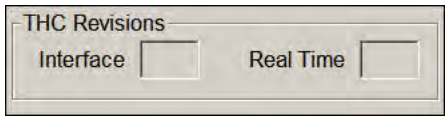
Limited Version: The Limited Version information is displayed if the CNC is operating on a trial version of the software. The version of software is available for 90 days. Contact your CNC vendor to have this timer reset.



Control Days Left: Appears only when a timer has been set up on the Control Information screen to limit the number of days that the Phoenix software is valid, for example when a limited version upgrade has been installed. Contact your OEM to have this timer reset.

OEM Days Left: A timer that the OEM can set on the Phoenix software on the Control Information screen. Contact your OEM to have this timer reset.

THC Revision: The current Command THC Interface and Real Time Revision Levels are displayed at the Control Information screen when enabled.



Touch screen calibration: Launches the touch screen calibration utility for tuning screen response.

I/O, Drives and Motors, Machine Interface

These screens require that you enter a password to open them from the Control Information screen.

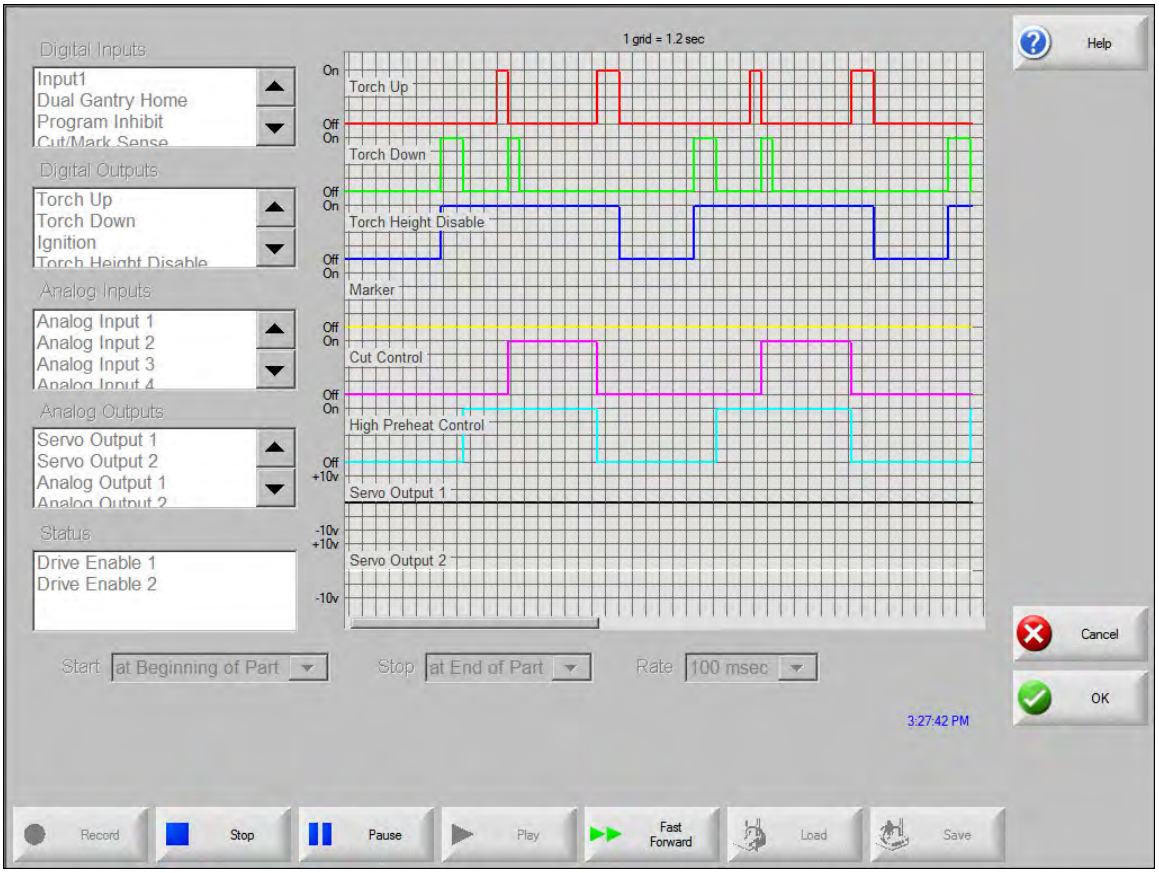
1. Choose Setups > Diagnostics.
2. From the Control Information screen, choose I/O, Drives and Motors, or Machine Interface.
3. Enter 7235.
4. Refer to the *Phoenix Software V9 Series Installation and Setup Manual* (806410) for information on these screens, or follow the on-screen instructions.

You must re-enter the password whenever you need to access either the I/O or Drives and Motors screens.

Using the Oscilloscope Function

You can use the Oscilloscope to log I/O, servo output voltage to the drive amplifiers, analog inputs, and drive status while the CNC is operating. The grid represents the time rate in which the function is recording data.

You can set up the Oscilloscope to help you understand an issue with an input or output or to log a function and then provide a visual representation of the log file.



To create an Oscilloscope log:

1. Double click on an item in the scroll boxes on the left of the screen to add it to the Oscilloscope grid. You can add up to eight items.
2. To remove an item from the grid, double-click on it in the appropriate scroll box.
3. In the Start drop-down list, select when you want the Oscilloscope to begin recording.
4. In the Stop drop-down list, select when you want the Oscilloscope to end recording.
5. In the Rate drop-down list, select the intervals at which the Oscilloscope to record the data you selected.

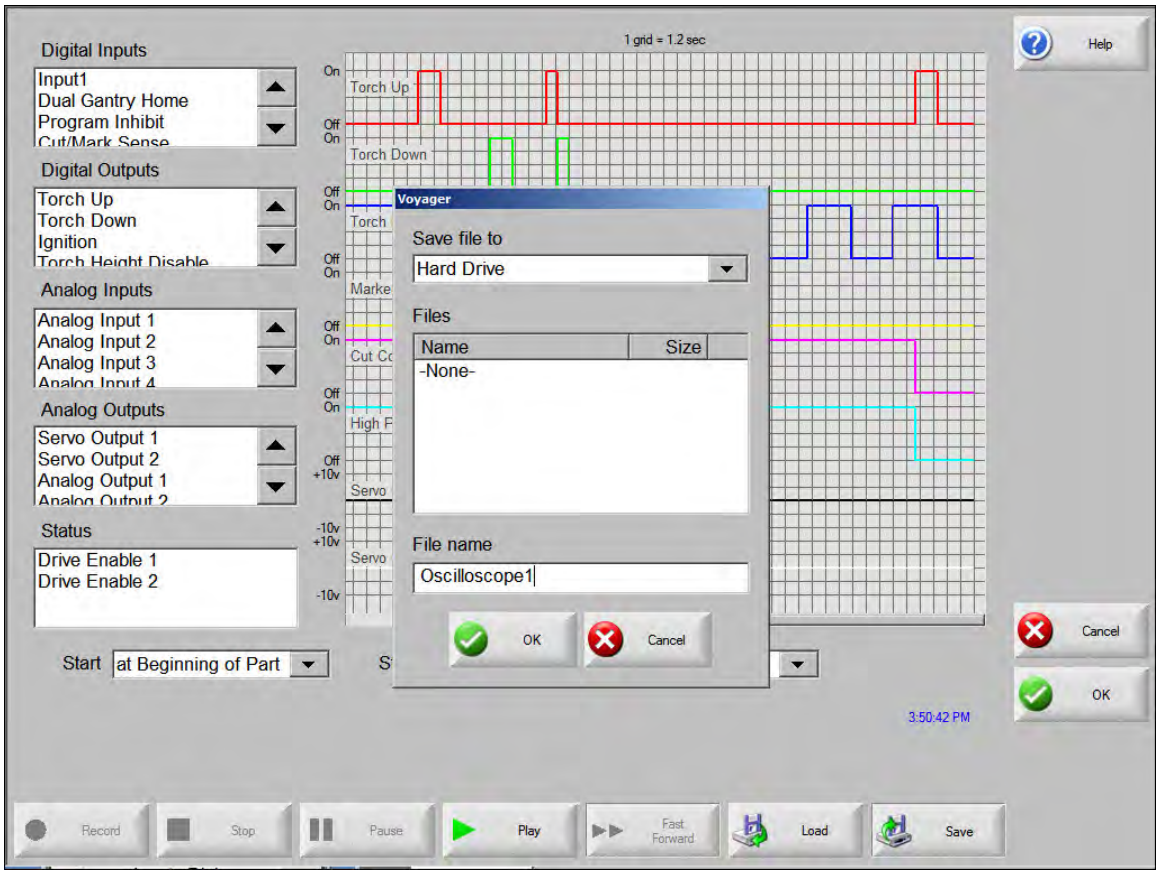
Saving the Oscilloscope File

When you are finished with a test, you can save the log file so you can use it later.

If you have created a function that starts recording at the beginning of a part program and ends at the last cut off, the file will be overwritten when the next part program is started. Be sure to save the file before executing the next nest.

To save the log file:

1. Press the Save soft key. The window where you enter file information opens.
2. Select the device where you want to save the file from the Save file to dropdown list.
3. Enter a name for the file in the File name field.
4. Press or click OK.



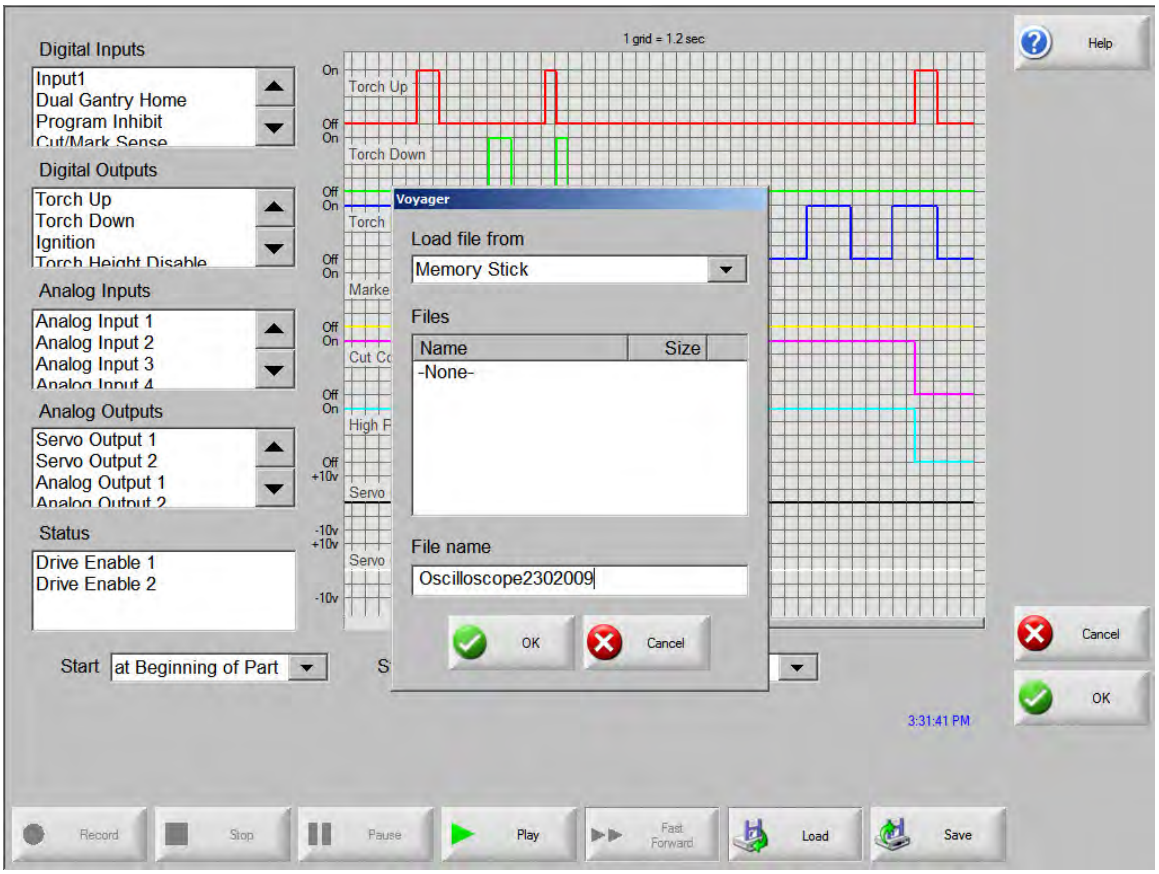
Loading an Oscilloscope File

After you have saved an Oscilloscope file, you can reload it and play it back on the CNC. This is the only way you can view this type of file.

In addition, a technical support organization can develop a customized log file for your operation, save it, and email it to you. You can load this customized file onto your CNC, and execute the function.

To load an oscilloscope log file:

1. Press Load on the Oscilloscope screen. The window where you enter file information opens.
2. Select a device from the Load File list.
3. Enter the name of the file you want to load in the File name field.
4. Click or press OK.



Viewing an Oscilloscope File

After you create and save a log file, you can replay it for diagnostics and troubleshooting.

To play an Oscilloscope log file:

1. Reload the file using the steps in the previous procedure.
2. Use the soft keys at the bottom of the screen to control the file:
   - Press Play to start the file.
   - Press Stop to end the file.
   - Press Pause to stop the file temporarily.
   - Press Fast Forward to speed up the file.

HPR plasma system

After serial link communication has been established between the power supply and the CNC, I/O and remote diagnostics screens are accessible through the Diagnostics screen. You can view status for the plasma supply software revision, gas pressure, usage, I/O and remote tools. The screen below shows the information screens for an HPR system.

Test Preflow: Tests the preflow gases at the power supply. This feature sets the inlet gas pressures under normal flow conditions to the recommended level.

Test Cutflow: Tests the cutflow gases at the power supply. This feature sets the inlet gas pressures under normal flow conditions to the recommended level.

Test Gas Console: Performs automated tests for the AutoGas console. Contact an authorized service agent for use of these tests.

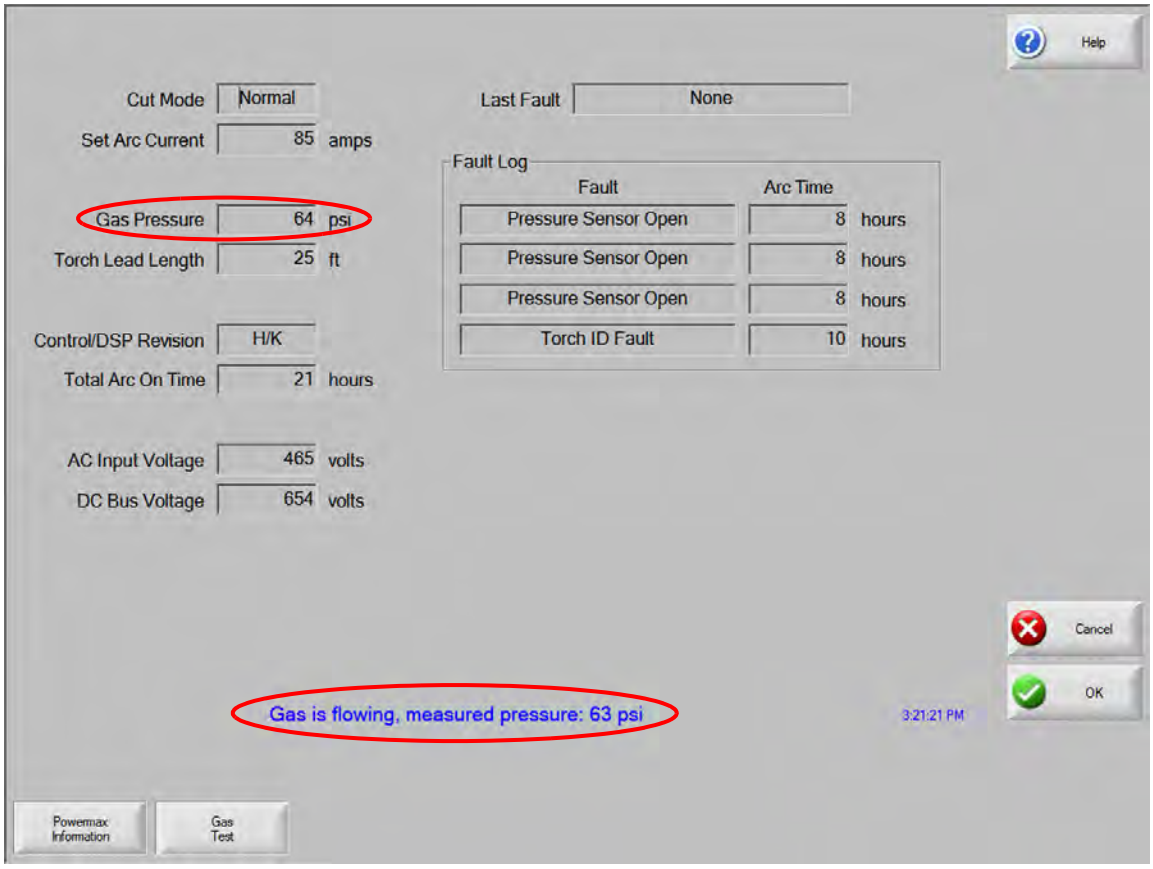
Coolant Override: Overrides a coolant error and tests the coolant pump. This is useful for bypassing the error and purging air bubbles from the coolant line when power is turned ON.

Inputs: Displays inputs to the plasma supply or gas console.

Outputs: Displays the current status of the outputs from the plasma supply or gas console, but the outputs can't be activated in this screen.

Powermax 65, 85, 105, and 125 systems

When using Powermax plasma systems, the CNC shows this screen when you press the Powermax soft key from the Diagnostics screen.



Gas Test: Turns on the gas flow and shows the actual gas pressure in blue text above the Gas Test soft key. This reading can be compared to the Gas Pressure set by the CNC to determine if there is a problem with the gas flow. 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.

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 Powermax 65 and Powermax 85 cut charts.

Control/DSP Revision: The firmware in the Powermax consists of two parts: the first is the control firmware and 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.

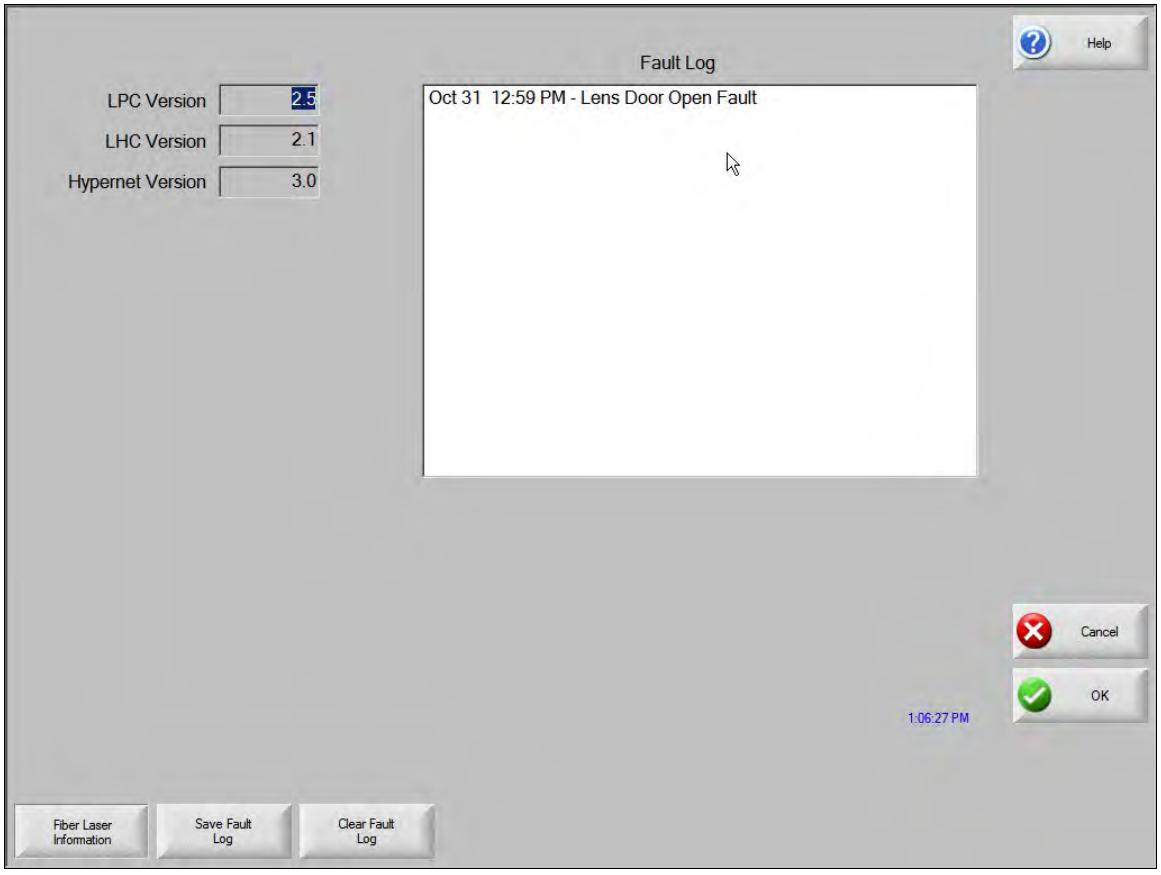
DC Bus Voltage: The internal DC voltage as measured by the Powermax sensors.

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.

Fiber laser diagnostics screen

When using the HFL010, HFL015, HFL020, or HFL030 fiber laser system, the CNC shows this screen when you press the Fiber Laser soft key on the Diagnostics screen.



LPC version: Shows the firmware version of the fiber laser power control.

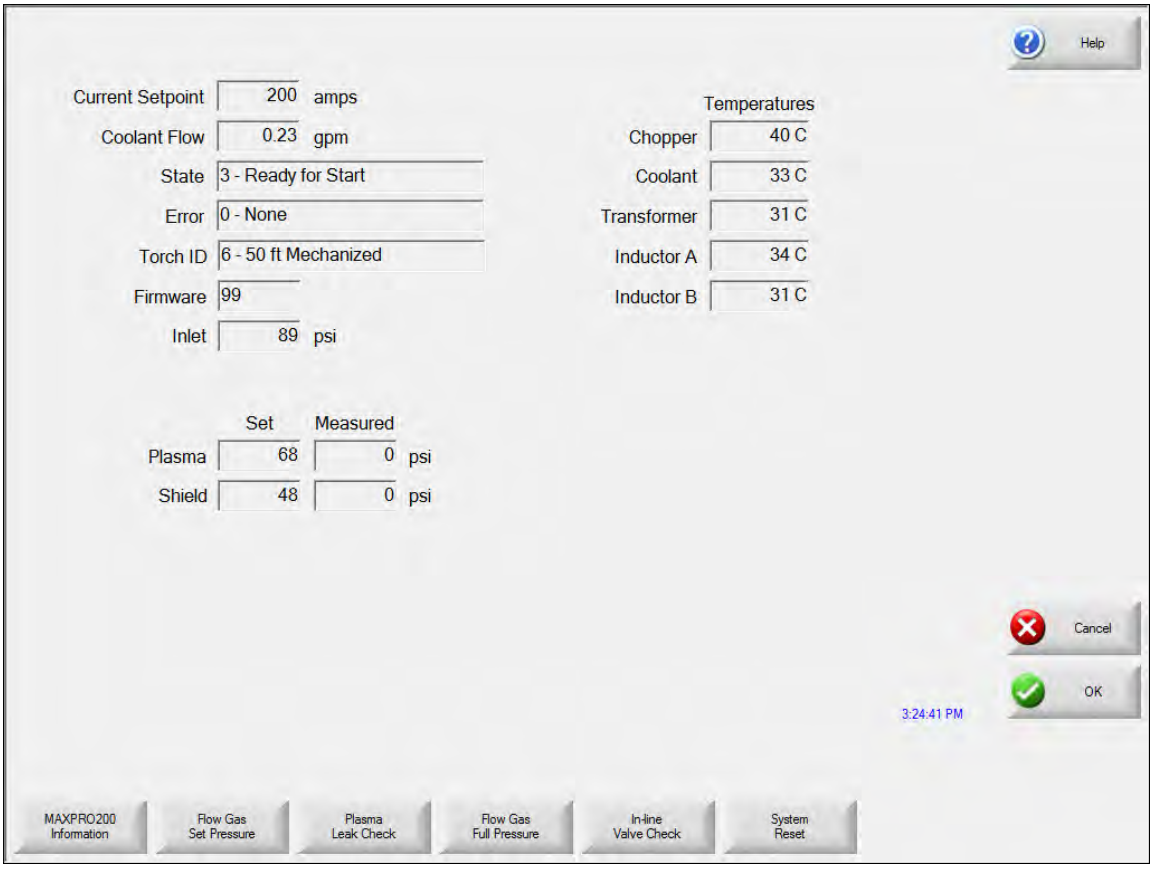
LHC version: Shows the firmware version of the fiber laser head controller.

Hypernet version: Shows the firmware version of the Hypernet communications used by the fiber laser.

Fault log: Shows the most recent fiber laser faults.

MAXPRO200 diagnostics 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* (807770).

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.

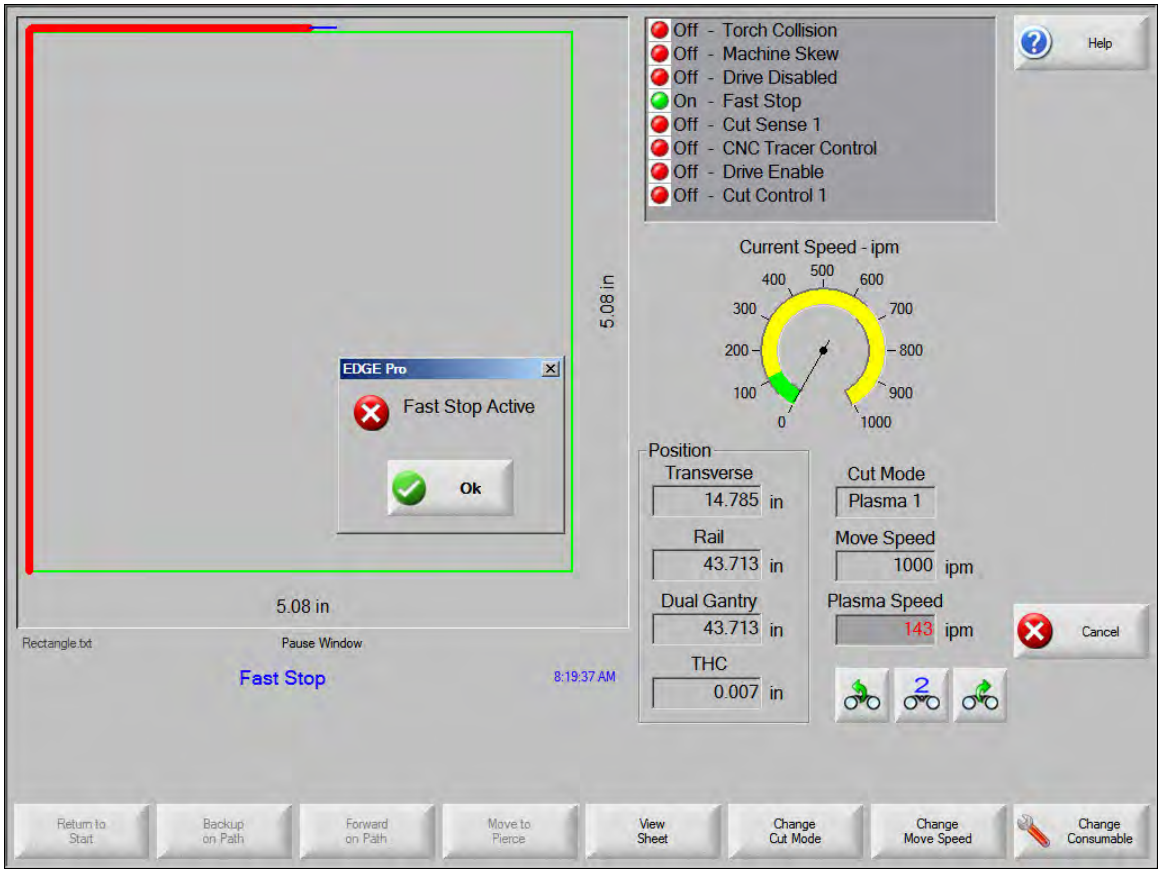
For a more detailed explanation of the diagnostic modes on this screen and for possible corrective actions, refer to the “Operation” section in the *MAXPRO200 Instruction Manual* (807770).

Fault and error messages

Phoenix software produces a number of dialog messages that cause motion and cutting to stop.

Faults

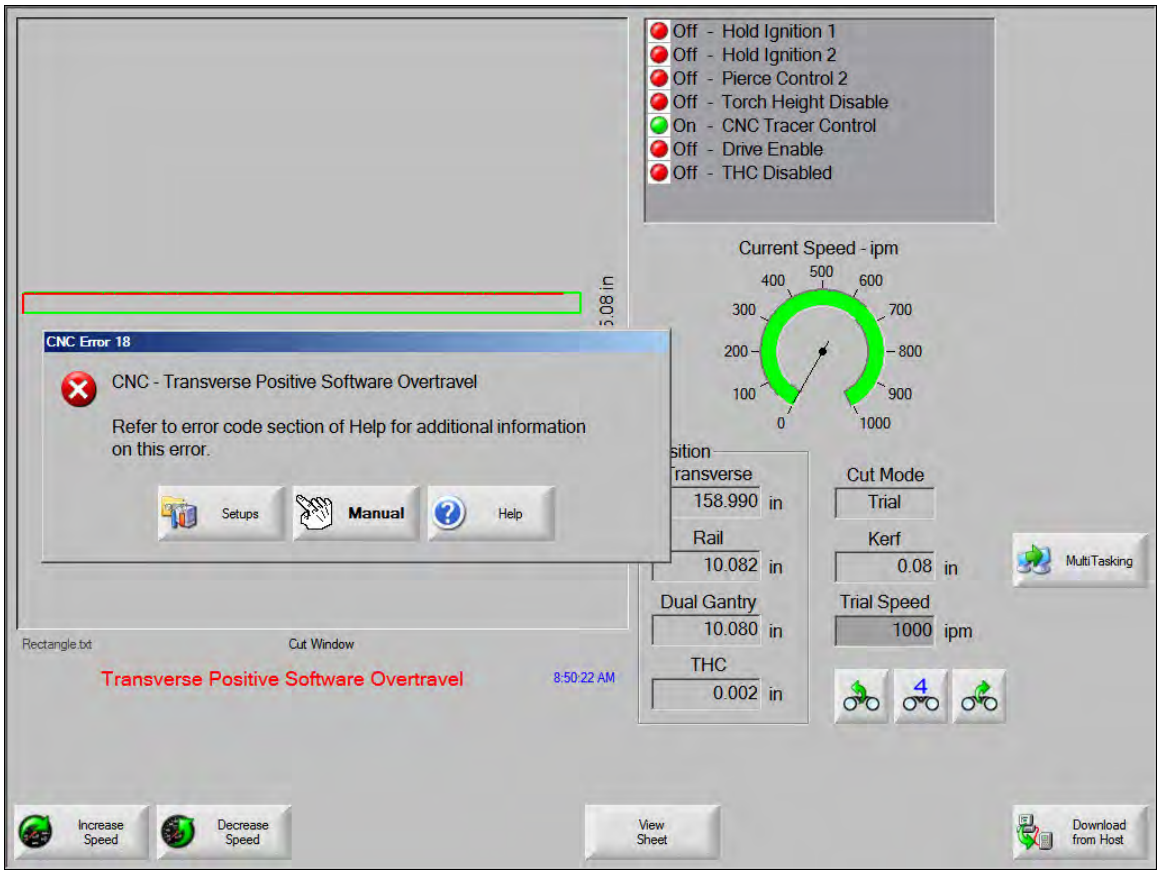
A fault brings CNC motion to a controlled stop and holds the position of all axes. After the fault is cleared, motion can continue from the current position on the table. If a fault occurs while running a CNC part program, the part program is paused and position in the program is not lost. Drive Disabled and Front Panel E-stop are the only exceptions; these faults cancel the part program.



Press OK on the Fault dialog to acknowledge the fault, fix the fault in the Manual pause screen, and continue the program. A description of each fault is provided in the *Fault Messages* section.

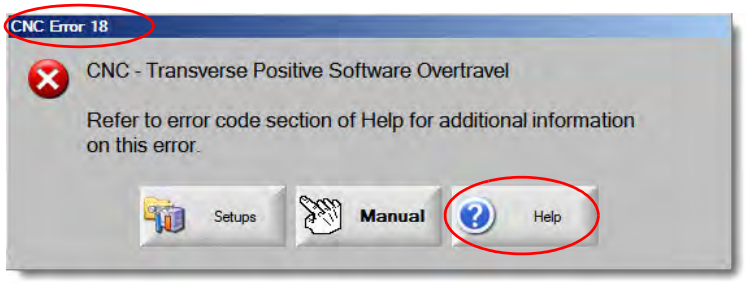
Errors

An error brings CNC motion to a controlled stop and then zeroes the positions of all axes. If an error occurs while running a CNC part program, the part program is canceled and the program position is zeroed.



You must home the table after an error because the program position is lost. Features such as Resume Last Part are affected after an error occurs and if the table is not homed, the torch may not resume at the correct location on the table. If Homing Must Be Performed is enabled, the operator sees a dialog message to home the machine when attempting to move the gantry after the error is cleared.

The CNC Error message window displays the error number and has a Help soft key that launches the Error Message section within online Help. The error message window also includes a Setups soft key, that returns you to the Setups screen, and a Manual soft key that allows you to perform manual motion to clear the fault.



A description of each error is provided in the *Error Message Reference* section.

Fault messages

For all faults that result from input logic, refer to the EDGE Pro Diagnostic Input screen for the location of the input and to verify that the input is working properly.

0.079 FAST Nozzle called for in part program. Insure correct Nozzle is installed before proceeding.

Possible cause

- The CNC part program is calling for a 0.079 Fast Nozzle.

Suggested action

- Verify that the correct nozzle is installed in the laser head before proceeding.

10" Focal length called for in part program. Insure correct lens is installed.

Possible cause

- The CNC part program is calling for a 10 inch focal length.

Suggested action

- Verify that the correct lens is installed in the laser head before proceeding.

10" Focal Length and 0.079" FAST Nozzle called for in part program. Insure correct Lens and Nozzle are installed before proceeding. The message appears after pressing Cycle Start.

Possible cause

- The CNC part program is calling for a 10 inch focal length and a 0.079 Fast Nozzle.

Suggested action

- Verify that the correct nozzle and lens are installed in the laser head before proceeding.

ArcGlide Fault See the *ArcGlide Instruction Manual* (806450) for a list of error messages. The ArcGlide reports errors to the EDGE Pro by sending the message through Hypernet communications. The EDGE Pro prompts you with a dialog message or a status message.

Possible cause

- The ArcGlide has a fault and is reporting an error message to the EDGE Pro.

Suggested actions

- ArcGlide faults are stored and displayed in the following areas of the EDGE Pro:
  - In the ArcGlide Diagnostic screen.
  - In System Errors Watch window of the EDGE Pro.
  - In the ArcGlide Manual; there is a description of the fault and information on how to resolve the problem.

Beam Path Interlock Active This is typically a normally closed input.

Possible causes

- The input device that insures that the beam is on path has activated the Beam Path Interlock input.
- There is a faulty cable or electrical connection between the Safety Mat input to the EDGE Pro and the device that activates the input.

Suggested actions

- Check the external device that turns on the Beam Path Interlock input.
- Check the cables and wiring between the Beam Path Interlock input and the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).

Buffer Timeout

Possible causes

- The position data stored on the Motion Control Card (MCC) was not accessible.
- This error may occur after replacing the MCC or after a software update.
- The MCC may be faulty.

Suggested actions

- This message may appear after replacing the MCC within the EDGE Pro or after a software update.
- Restart the EDGE Pro. If the problem persists, there could be a failure with the MCC board.

Cut Gas Lost

Possible causes

- The cut gas pressure dropped while cutting.
- There is a low pressure regulator setting.
- The cut gas source may be low or empty.
- There is a gas leak or restriction in one of the gas lines.
- There is a faulty solenoid valve.

Suggested actions

- Check the pressure regulator setting while purging the cut gas.
- Check the volume in the cut gas supply.
- Check for loose or damaged gas lines.
- Verify that the cut gas can flow through the torch when doing a test cut flow.

Cut Height Exceeds Maximum Detectable CHS Height

Possible causes

- The Cut Height value in the Laser Process Screen or within the CNC part program exceeds the capabilities of the Capacitive Height Sensor (CHS).
- The CHS, after calibration, will not be able to properly sense the cut height while cutting.

Suggested actions

- Verify that the correct cut height value was set in the Laser Process screen or within the CNC part program.
- Recalibrate the CHS if the cut height value is a reasonable value.

Fast Stop Active This input is typically a normally closed input.

Possible causes

- An E-stop on the cutting table is depressed.
- The servo amplifiers are not powered up.
- There is a faulty cable or electrical connection between the Fast Stop input to the EDGE Pro and the device that activates the input.

Suggested actions

- Check the external device that turns on the Fast Stop input.
- Check the cables and wiring between the Fast Stop input and the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).

Fatal Error – HD4070 The HD4070 reports errors to the EDGE Pro by sending a message through serial communication and prompts you with a dialog message or a status message.

Possible cause

- The HD4070 has a fault and is reporting an error message to the EDGE Pro.

Suggested actions

- HD4070 errors are stored and displayed in the following areas of the EDGE Pro:
  - In System Errors Watch window of the EDGE Pro.
  - In the HD4070 diagnostic screen.
- Check the HPR Manual for a description of the error and information on how to resolve the problem.

HPR Fault See the *HPR Instruction Manual* (various part numbers) for a list of error codes. The HPR reports errors to the EDGE Pro by sending a message through serial communication and prompts you with a dialog message or a status message.

Possible cause

- The HPR has a fault and is reporting an error message to the EDGE Pro.

Suggested actions

- HPR errors are stored and displayed in the following areas of the EDGE Pro:
  - Within the HPR Watch Window or within the HPR Diagnostic screen, the parameter is listed as Last Err (Last Error).
  - In System Errors Watch window of the EDGE Pro.

Invalid Process Requested in Part Program

Possible causes

- The CNC part program has a cutting process (M36) or station code (M37) that the EDGE Pro does not recognize.
- The CNC part program is calling out a cut chart that does not exist in the EDGE Pro.
- Material Thickness
- Plasma/Shield Gas
- Amperage
- A programming code is disabled in the Cutting setup screen.
- G59 Process codes
- M07 HS/M08 RT
- Process enable
- Station enable

Suggested actions

- Verify that the Station switches on the EDGE Pro are in the Program Position.
- Verify that the Station Configuration screen is properly configured for your cutting system.
- Update the software and cut charts.
- If the CNC part program includes G59 V5xx Fvalue process override codes, verify that the parameters match the values in the cut chart.
  - Material Thickness
  - Torch Type
  - Plasma/Shield Gas Type
  - Cutting Current
- Verify that these parameters exist in the Plasma/Marker Cut Charts. If one of these values does not exist in the cut chart, create a custom cut chart to resolve the problem.
- Verify that the correct parameters are Enabled and/or Disabled within the Program Code section in the Cutting Screen.
- Contact your table manufacturer if you are not sure which codes should be enabled or disabled.

Pierce Height Exceeds Maximum Detectable CHS Height

Possible causes

- The Pierce Height value in the Laser Process Screen or within the CNC part program exceeds the capabilities of the Capacitive Height Sensor (CHS).
- The CHS, after calibration, will not be able to sense the Pierce height after an IHS (Initial Height Sensing).

Suggested actions

- Verify that the correct pierce height value was set in the Laser Process screen or within the CNC part program.
- Re-calibrate the CHS if the pierce height value is a reasonable value.

Remote Pause Active Remote Pause is typically a Normally Closed input.

Possible causes

- The Remote Pause input has been activated by an external device.
- There is a faulty cable or electrical connection between the Remote Pause input to the EDGE Pro and the device that activates the input.

Suggested actions

- Check the external device that turns on the remote pause input.
- Check cables and wiring between the Remote Pause input and the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (Normally Closed or Normally Open).

Safety Mat Active This input is typically a Normally Closed Input

Possible causes

- The light curtain or safety mat or some other type of external device that activates when a person is within a restricted area around the cutting table was activated.
- A faulty cable or electrical connection between the Safety Mat input to the EDGE Pro and the device that activates the input.

Suggested actions

- Check the external device that turns on the Safety Mat input.
- Check the cables and wiring between the Safety Mat input and the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (Normally Closed or Normally Open).

Software Limit Active

Possible cause

- Motion in either the rail or transverse (or both) axes has reached either the Maximum or Minimum software travel limit.

Suggested actions

- Motion is only allowed in the opposite direction of the limit that is active (Transverse or Rail).
- If both software limits are active concurrently, motion is allowed from the axis that reached the limit last. For instance, motion results in both the rail and transverse software limits to active. If the Rail limit was reached after the Transverse limit, then motion will only occur in the opposite direction on the rail axis first.
- If problem persists, check the minimum and maximum soft limit settings in the Axes setup screen.

Torch Collision Active Torch Collision is typically a Normally Closed input.

Possible causes

- The torch collided with the plate and activated a momentary or sustained torch collision input.
- There is a faulty cable or electrical connection between the Torch Collision input to the EDGE Pro and the device that activates the input.

Suggested actions

- Raise the torch and reseat the torch collision device used on the THC lifter.
- Check the torch collision device to insure that the device is working properly.
- If there is a magnetic breakaway, check the proximity switch to see if it switches on and off when tripped manually.
- If there is a pneumatic breakaway, verify that the switch is functioning properly.
- Check the cables and wiring between the torch collision device and the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (Normally Closed or Normally Open).

Error message reference

1 **Transverse position error** The position of the Transverse axis is behind the commanded position by more than the Servo Error Tolerance value.

Possible causes

- The following error of the Transverse axis exceeded the Servo Error Tolerance value.
- There is a mechanical bind in the axis.
- The drive amplifier has produced a fault.
- There is a faulty motor/encoder cable, either from the motor or from the servo amplifier.
- There is no encoder feedback from the Transverse motor or servo amplifier.
- There is a faulty axis interface board in the EDGE Pro.
- If this is a new installation:
  - The encoder and/or DAC (command voltage) polarity is set incorrectly.
  - The Servo Error Tolerance is set too low.
  - The gains are set to low.
  - The Maximum Machine Speed is set too high.
  - The Acceleration is too high.

Suggested actions

- Use the EDGE Pro diagnostic tool kit to test the functionality of each axis. Refer to the EDGE Pro manual for instructions on how to test the axis interface board.
- Use the Drives & Motors Diagnostic screen to establish motion control in a new installation. Determine:
  - Maximum Machine Speed
  - DAC Polarity
  - Encoder Polarity
- Use the Drives & Motors Diagnostic screen to test the actual servo amplifier, motor, and cables.
- Perform the following tests:
  - Encoder Feedback to EDGE Pro
  - Command Voltage to Servo amplifier
  - Maximum Speed of motor

2 **Rail Position Error** The position of the Rail axis is behind the commanded position by more than the Servo Error Tolerance value.

Possible causes

- The following error of the Rail axis exceeded the Servo Error Tolerance value.
- There is a mechanical bind in the axis.
- The drive amplifier has produced a fault.
- There is a faulty motor/encoder cable either from the motor or from the servo amplifier.
- There is no encoder feedback from the Rail motor or servo amplifier.
- There is a faulty axis interface board in the EDGE Pro.
- If this is a new installation:
- The encoder and/or DAC (command voltage) polarity is set incorrectly.
  - The Servo Error Tolerance is set too low.
  - The gains are set too low.
  - The Maximum Machine Speed is set too high.
  - The Acceleration is too high

Suggested actions

- Use the EDGE Pro diagnostic tool kit to test the functionality of each axis. Refer to the EDGE Pro manual for instructions on how to test the axis interface board.
- Use the Drives & Motors Diagnostic screen to establish motion control in a new installation.
- Determine:
  - Maximum Machine Speed
  - DAC Polarity
  - Encoder Polarity
- Use the Drives & Motors Diagnostic screen to test the actual servo amplifier, motor, and cables.
- Perform the following tests:
  - Encoder Feedback to EDGE Pro
  - Command Voltage to Servo amplifier
  - Maximum Speed of motor

3 **Dual Gantry Position Error** The position of the Dual Gantry axis is behind the commanded position by more than the Servo Error Tolerance value.

Possible causes

- The following error of the Dual Gantry axis exceeded the Servo Error Tolerance value.
- There is a mechanical bind in the axis.
- The drive amplifier has produced a fault.
- There is a faulty motor/encoder cable either from the motor or from the servo amplifier.
- There is no encoder feedback from the Dual Gantry motor or servo amplifier.
- There is a faulty axis interface board in the EDGE Pro.
- If this is a new installation:
  - The encoder and/or DAC (command voltage) polarity is set incorrectly.
  - The Servo Error Tolerance is set too low.
  - The gains are set too low.
  - The Maximum Machine Speed is set too high.
  - The Acceleration is too high.

Suggested Actions

- Use the EDGE Pro diagnostic tool kit to test the functionality of each axis. Refer to the EDGE Pro manual for instructions on how to test the axis interface board.
- Use the Drives & Motors Diagnostic screen to establish motion control in a new installation.
- Determine:
  - Maximum Machine Speed
  - DAC Polarity
  - Encoder Polarity
- Use the Drives & Motors Diagnostic screen to test the actual servo amplifier, motor, and cables.
- Perform the following tests:
  - Encoder Feedback to EDGE Pro
  - Command Voltage to Servo amplifier
  - Maximum Speed of motor

---

4 **Rotate Position Error** The position of the Rotate axis is behind the commanded position by more than the Servo Error Tolerance value. The following error of the Rotate axis exceeded the Servo Error Tolerance value.

Possible cause

- Refer to the Transverse Position Error (Error 1) for information on possible causes.

Suggested action

- Refer to the Transverse Position Error (Error 1) for information on suggested actions.

---

5 **Tilt Position Error** The position of the Tilt axis is behind the commanded position by more than the Servo Error Tolerance value. The following error of the tilt axis exceeded the Servo Error Tolerance value.

Possible cause

- Refer to the Transverse Position Error (Error 1) for information on possible causes.

Suggested action

- Refer to the Transverse Position Error (Error 1) for information on suggested actions.

6 **CBH Position Error** The position of the CBH axis is behind the commanded position by more than the ServoError Tolerance. The following error of the CBH axis exceeded the Servo Error Tolerance value.

Possible cause

- Refer to the Transverse Position Error (Error 1) for information on possible causes.

Suggested action

- Refer to the Transverse Position Error (Error 1) for information on suggested actions.

7 **THC Position Error** The position of the THC axis is behind the commanded position by more than the Servo Error Tolerance value. The following error of the THC axis exceeded the Servo Error Tolerance value.

Possible causes

- There is a mechanical bind in the lifter.
- The drive amplifier has produced a fault.
- There is a faulty motor/encoder cable either from the motor or from the servo amplifier.
- There is no encoder feedback from the Transverse motor or servo amplifier.
- In a Yaskawa drive, the Forward/Reverse External Torque Limit input is active all the time.
- There is a faulty axis interface board in the EDGE Pro.
- If this is a new installation, the encoder or DAC (command voltage) polarity is set incorrectly.
- The Servo Error Tolerance, gain, or torque values are set too low.
- The Maximum Machine Speed, Acceleration, lifter speed, or stall force values are set too high.

Suggested actions

- Use the EDGE Pro diagnostic tool kit to test the functionality of each axis. Refer to the EDGE Pro manual for instructions on how to test the axis interface board.
- Use the Drives & Motors Diagnostic screen to establish motion control in a new installation (disengage the motor from the lead screw or ball screw).
- Determine:
  - Maximum Machine Speed
  - DAC Polarity
  - Encoder Polarity
- Positive motion of the THC axis will lower the torch.

8 **Transverse Positive Hardware Overtravel** This input is typically a normally closed input. The Transverse Positive Hardware Overtravel switch is named either +X Overtravel or + Y Overtravel. The switch is located on the far positive end of the Transverse axis.

Possible causes

- The cutting station is engaging one of the two limit switches located along the Transverse axis.
- There is a faulty limit switch.
- There is a damaged cable or loose electrical connection between the limit switch and the where the input connects to the EDGE Pro.
- There is no DC voltage to the machines inputs.
- There is a faulty input to the EDGE Pro.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch.

Suggested actions

- Jog the cutting station in the opposite direction to move off the limit switch.
- Check the hardware limit switch on the gantry for damage and verify that the switch is functioning properly.
- Test the limit switch inputs in the Diagnostic Inputs screen.
- Check the cables and wiring between the limit switch and the EDGE Pro.
- Make sure the cutting table is powered up.
- Make sure all cables are properly installed at the back of the EDGE Pro.
- If this is a new installation:
  - Check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).
  - Refer to the EDGE Pro Diagnostic Input screen for location of the Transverse Positive Hardware Overtravel switch and to verify that the input is working properly.

---

9 **Rail Positive Hardware Overtravel** This input is typically a normally closed input. The Rail Positive Hardware Overtravel switch is named either +X Overtravel or + Y Overtravel. The switch is located on the far positive end of the Rail axis.

Possible causes

- The cutting station is engaging one of the two limit switches located along the Rail axis.
- There is a faulty limit switch.
- There is a damaged cable or loose electrical connection between the limit switch and the where the input connects to the EDGE Pro.
- There is no DC voltage to the machine's inputs.
- There is a faulty input to the EDGE Pro.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch.

Suggested actions

- Jog the cutting station in the opposite direction to move off the limit switch.
- Check the hardware limit switch on the gantry for damage and verify that the switch is functioning properly. Test the limit switch inputs in the Diagnostic Inputs screen.
- Check the cables and wiring between the limit switch and the EDGE Pro.
- Make sure the cutting table is powered up.
- Make sure all cables are properly installed at the back of the EDGE Pro.
- If this is a new installation:
  - Check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).
  - Refer to the EDGE Pro Diagnostic Input screen for location of the Rail Positive Hardware Overtravel switch and to verify that the input is working properly.

---

13 **Transverse Negative Hardware Overtravel** This input is typically a normally closed input.The Transverse Negative Hardware Overtravel switch is named either -X Overtravel or -Y Overtravel. The switch is located on the far negative end of the Transverse axis.

Possible causes

- The cutting station is engaging one of the two limit switches located along the Transverse axis.
- There is a faulty limit switch.
- There is a damaged cable or loose electrical connection between the limit switch and the where the input connects to the EDGE Pro.
- There is no DC voltage to the machines inputs.
- There is a faulty input to the EDGE Pro.
- If this is a new installation, verify that the logic of the overtravel inputs does not match the logic of the actual limit switch.

Suggested actions

- Jog the cutting station in the opposite direction to move off of the limit switch.
- Check the hardware limit switch on the gantry for damage and verify that the switch is functioning properly. Test the limit switch inputs in the Diagnostic Inputs screen.
- Check the cables and wiring between the limit switch and the EDGE Pro.
- Make sure the cutting table is powered up.
- Make sure all cables are properly installed at the back of the EDGE Pro.
- If this is a new installation:
  - Check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).
  - Refer to the EDGE Pro Diagnostic Input screen for location of the Transverse Negative Hardware Overtravel switch and to verify that the input is working properly.

---

14 **Rail Negative Hardware Overtravel** This input is typically a normally closed input. The Rail Negative Hardware Overtravel switch is named either -X Overtravel or -Y Overtravel. The switch is located on the far negative end of the Rail axis.

Possible causes

- The cutting station is engaging one of the two limit switches located along the Rail axis.
- There is a faulty limit switch.
- There is a damaged cable or loose electrical connection between the limit switch and the where the input connects to the EDGE Pro.
- There is no DC voltage to the machines inputs.
- There is a faulty input to the EDGE Pro.
- If this is a new installation, verify that the logic of the overtravel inputs does not match the logic of the actual limit switch.

Suggested actions

- Jog the cutting station in the opposite direction to move off the limit switch.
- Check the hardware limit switch on the gantry for damage and verify that the switch is functioning properly.
- Test the limit switch inputs in the Diagnostic Inputs screen.
- Check the cables and wiring between the limit switch and the EDGE Pro.
- Make sure the cutting table is powered up.
- Make sure all cables are properly installed at the back of the EDGE Pro.
- If this is a new installation:
  - Check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (Normally Closed or Normally Open).
  - Refer to the EDGE Pro Diagnostic Input screen for location of the Rail Negative Hardware Overtravel switch and to verify that the input is working properly.

---

18 **Transverse Positive Software Overtravel** Software Overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible causes

- Motion along the transverse axis (positive direction) has reached the preset software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Suggested action

- Jog the cutting station in the opposite direction to move off the software limit.

---

19 **Rail Positive Software Overtravel** Software overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible causes

- Motion along the Rail axis (positive direction) has reached the preset software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Suggested action

- Jog the cutting station in the opposite direction to move off the software limit.

---

23 **Transverse Negative Software Overtravel** Software Overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible causes

- Motion along the transverse axis (negative direction) has reached the preset software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Suggested action

- Jog the cutting station in the opposite direction to move off the software limit.

---

24 **Rail Negative Software Overtravel** Software Overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible causes

- Motion along the Rail axis (negative direction) has reached the preset software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Suggested action

- Jog the cutting station in the opposite direction to move off of the software limit.

---

28 **Tilt Positive Hardware Overtravel** This input is typically a normally closed input. The Tilt Positive Hardware Overtravel switch is named Tilt + Overtravel. The switch is located on the far positive end of the Tilt axis.

Possible causes

- The cutting station is engaging one of the two limit switches located along the Tilt axis.
- There is a faulty limit switch.
- There is a damaged cable or loose electrical connection between the limit switch and the where the input connects to the EDGE Pro.
- There is no DC voltage to the machines inputs.
- There is a faulty input to the EDGE Pro.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch.

Suggested actions

- Jog the tilt axis in the opposite direction to move off the limit switch.
- Check the hardware limit switch on the bevel head for damaged and verify that the switch is functioning properly.
- Test the limit switch inputs in the Diagnostic Inputs screen.
- Check the cables and wiring between the limit switch and the EDGE Pro.
- Make sure the cutting table is powered up.
- Make sure all cables are properly installed at the back of the EDGE Pro.
- If this is a new installation:
  - Check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (Normally Closed or Normally Open).
  - Refer to the EDGE Pro Diagnostic Input screen for location of the Tilt Positive Hardware Overtravel switch and to verify that the input is working properly.

---

29 Tilt Negative Hardware Overtravel This input is typically a normally closed input. The Tilt Negative Hardware Overtravel switch is named Tilt – Overtravel. The switch is located on the far negative end of the Tilt axis.

Possible causes

- The cutting station is engaging one of the two limit switches located along the Tilt axis.
- There is a faulty limit switch.
- There is a damaged cable or loose electrical connection between the limit switch and where the input connects to the EDGE Pro.
- There is no DC voltage to the machine inputs.
- There is a faulty input to the EDGE Pro.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch.

Suggested actions

- Jog the tilt axis in the opposite direction to move off the limit switch.
- Check the hardware limit switch on the bevel head for damage and verify that the switch is functioning properly.
- Test the limit switch inputs in the Diagnostic Inputs screen.
- Check the cables and wiring between the limit switch and the EDGE Pro.
- Make sure the cutting table is powered ON.
- Make sure all cables are properly installed at the back of the EDGE Pro.
- If this is a new installation:
  - Check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).
  - Refer to the EDGE Pro Diagnostic Input screen for location of the Tilt Positive Hardware Overtravel switch and to verify that the input is working properly.

---

30 **Tilt Positive Software Overtravel** Software Overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible cause

- Motion along the Tilt axis (positive direction) has reached the preset software limit.

Suggested actions

- Jog the cutting station in the opposite direction to move off the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

31 **Tilt Negative Software Overtravel** Software Overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible cause

- Motion along the Tilt axis (negative direction) has reached the preset software limit.

Suggested actions

- Jog the cutting station in the opposite direction to move off the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

34 **Rotate Positive Software Overtravel** Software Overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible cause

- Motion along the Rotate axis (positive direction) has reached the preset software limit.

Suggested actions

- Jog the cutting station in the opposite direction to move off the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

35 **Rotate Negative Software Overtravel** Software Overtravels minimum and maximum values are based on position from home and should stop motion prior to engaging a hardware overtravel.

Possible cause

- Motion along the Rotate axis (negative direction) has reached the preset software limit.

Suggested actions

- Jog the cutting station in the opposite direction to move off the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

36 Dual Gantry Skew Error A Dual Gantry Skew would occur after homing the Rail/Dual Gantry.

Possible causes

- The Dual Gantry skew has exceeded the Dual Gantry Skew Limit.
- There is a mechanical bind in the axis.
- The Dual Gantry switch offset has changed or was incorrectly.
- If this is a new installation:
  - The skew limit is set to low.
  - Check the switch offset.

Suggested actions

- Ensure that the gantry is square and there are no mechanical binds on the rail and dual gantry axes.
- Check the Dual Gantry and Rail home switches for damaged or a loose engagement block.
- Verify that the switch offset setting is correct.
- Check the skew limit setting.
- The skew limit should be some value higher than the following error while homing and not high enough to cause mechanical damage.

---

37 Collision Fault The Collision input is typically normally closed.

Possible causes

- The Collision Fault input is activated.
- The Collision Fault device is damaged.
- There is a faulty cable or electrical connection between the Collision input to the EDGE Pro and the device that activates the input.

Suggested actions

- Clear the error and resume cutting.
- Check the collision fault device to insure that the device is working properly.
- Check the cables and wiring between the collision fault device and the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).
- Refer to the EDGE Pro Diagnostic Input screen for location of the Torch Collision input and to verify that the input is working properly.

---

38 Excessive Mechanical Skew The difference in position between the Rail and the Dual Gantry is greater than 2 inches when the EDGE Pro was powered up.

Possible causes

- The position of the Dual Gantry axis changed prior to booting up the EDGE Pro.
- There is a mechanical bind in the axis.
- The Dual Gantry or the Rail pinion gears were removed from the rack and then rotated prior to engaging the pinion gear back into the rack.

In a SERCOS system with Absolute encoders the Dual Gantry position was changed within the drive amplifier.

Suggested actions

- Verify that the gantry is square.
- Check the gantry for mechanical binds.
- If the pinion gears were disengaged from the drive rack when the EDGE Pro was powered down:
  - Power ON the EDGE Pro with the pinion gears still disengaged. If there is excessive difference in position, the Rail will rotate to equalize position.
  - Power OFF the cutting machine and then re-engage the pinion gears. This will prevent this fault from occurring.
- If the CNC is a SERCOS machine with absolute encoders check the position and encoder settings in the drive amplifier.
- If the dual gantry position changed, power OFF the cutting machine, disengage the pinion gear, and rotate the pinion until the dual gantry position matches the rail position.

41 Dual Head Collision Fault Torch Collision is typically a normally closed input.

Possible causes

- The two cutting stations were jogged too close to each other and activated the Dual Head Collision Fault switch.
- There is a faulty switch.
- There is a faulty cable or electrical connection between the Dual Head Collision Fault input to the EDGE Pro and the device that activates the input.
- There is a faulty input on the EDGE Pro.

Suggested actions

- Park one of the stations and jog the other station to clear the fault.
- Check the Dual Head Collision Fault device to insure that the device is working properly.
- Check the cables and wiring between the Dual Head Collision Fault device and the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (Normally Closed or Normally Open).
- Refer to the EDGE Pro Diagnostic Input screen for location of the Torch Collision input and to verify that the input is working properly.

42 **Torch Collision** Torch Collision is typically a normally closed input.

Possible causes

- The torch collided with the plate and caused the torch collision input to activate, either momentarily or maintained.
- There is a faulty cable or electrical connection between the Torch Collision input to the EDGE Pro and the device that activates the input.
- If this is a new installation:
  - Check the Torch Collision fault setting in the I/O setup screen.
  - Fast Decel will result in a Fault.
  - Fault Ramp will result in an Error.

Suggested actions

- Raise the torch and reseat the torch collision device used on the THC lifter.
- Check the torch collision device to insure that the device is working properly.
  - If it is a magnetic breakaway, check the proximity switch to see if it activates on and off when tripped manually.
  - If it is a pneumatic breakaway verify that the switch is functioning properly.
- Check the cables and wiring between the torch collision device and the back of the EDGE Pro
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (Normally Closed or Normally Open).
- Refer to the EDGE Pro Diagnostic Input screen for location of the Torch Collision input and to verify that the input is working properly.

---

43 **Field Power Failure** The EDGE Pro has +5, +12, -12, and +24VDC available for external use. These voltages are accessible on the Drive/Encoder connectors and the I/O connectors on the back of the EDGE Pro.

Possible causes

- One of the field voltages dropped below its nominal operating range.
- One of the field voltages is shorted to ground or to common.
- There is a damaged I/O cable or Drive/Encoder cable.
- There is a faulty power supply inside the EDGE Pro.
- If this is a new installation, one of the field voltages is being loaded down due to excessive load or an improper connection to the EDGE Pro.

Suggested actions

- Power OFF the EDGE Pro and disconnect all the cables to the back door of the EDGE Pro. Power ON the EDGE Pro and verify that the Field Power Failure message continues to appear. The Field Power Failure can be viewed from the watch window.
- If the fault continues to occur, contact your table manufacturer.
- If the problem is resolved, plug in each cable, one at a time, until the problem returns.
- If the problem is outside the EDGE Pro:
  - Check the EDGE Pro manual for the maximum available power for the field voltages and insure that the devices driven by these voltages do not exceed this rating.
  - Check all cables for damage.
- If an external source is used to activate the EDGE Pro inputs:
  - Ensure this source is not connected to the +24VDC of the EDGE Pro.
  - Ensure that the commons on this source is connected to the commons of the EDGE Pro.

---

44 **Hardware Fault or Failure** This fault occurs when two or more opposing inputs are activated at the same instance in time, for example, Raise Torch 1, Lower Torch 1, Raise Torch 2, and Lower Torch 2 are all turned on at the same time, or two opposing Joystick inputs are turned on at the same time (Left and Right or Up and Down).

Possible causes

- High-frequency noise is causing multiple jog inputs to suddenly turn on.
- A setup file is loaded that has inverted input logic for either the Raise/Lower Torch inputs or the Joystick inputs.
- There is a faulty joystick.
- There is a faulty raise or lower torch input.
- There is a faulty board within the EDGE Pro.
- If this is a new installation:
  - Check the logic of all jog inputs.
  - Check the wiring of all jog inputs.

Suggested actions

- Restart the EDGE Pro to clear the fault. If the inputs were activated in error, the problem should not re-occur.
- If the problem continues:
  - Test the Joystick within the Front Panel Diagnostic screen
  - If the joystick has a sticking or damaged switch, the joystick will not be shown in the center position.
  - Go to the Inputs Diagnostic screen and check the state and operation of the joystick and THC raise/lower inputs.

---

45 Dual Transverse Positive Hardware Overtravel

Possible cause

- In a dual transverse system, the second cutting station has engaged the hardware overtravel switch on the gantry.

Suggested actions

- Jog the cutting station in the opposite direction to move off the limit switch.
- Check the hardware limit switch on the gantry for damage and verify that the switch is functioning properly.
- Test the limit switch inputs in the Diagnostic Inputs screen.
- Check the cables and wiring between the limit switch and the EDGE Pro.
- Make sure the cutting table is powered up.
- Make sure all cables are properly installed at the back of the EDGE Pro.
- If this is a new installation, check the logic of this input in the I/O setup screen. The logic should match the logic of the device that activates the input (normally closed or normally open).
- Refer to the EDGE Pro Diagnostic Input screen for location of the Transverse Positive Hardware Overtravel switch and to verify that the input is working properly.

---

46 Dual Transverse Negative Software Overtravel

Possible cause

- In a dual transverse system, the second cutting station has reached the position of the Minimum or Maximum Travel Limit set in the Transverse 2 axis screen.

Suggested action

- Jog the cutting station in the opposite direction to move off the software limit.

---

47 Dual Transverse Positive Software Overtravel

Possible cause

- In a dual transverse system, the second cutting station has reached the position of the Minimum or Maximum Travel Limit set in the Transverse 2 axis screen.

Suggested action

- Jog the cutting station in the opposite direction to move off the software limit.

---

48 Dual Transverse Negative Software Overtravel

Possible cause

- In a dual transverse system, the second cutting station has reached the position of the Minimum Travel Limit set in the Transverse 2 axis screen.

Suggested action

- Jog the cutting station in the opposite direction to move off the software limit.

---

49 SERCOS Ring Fault

Possible causes

- SERCOS II: Improperly sanded or seated fiber optic cables.
- SERCOS III: Defective or incorrect Ethernet cables.

Suggested actions

- SERCOS II: Make sure all cables in the ring are seated properly. Check connectors for dust or dirt which can interfere with the fiber optic signal. Refer to the manufacturer's instructions for sanding fiber optic cables.
- SERCOS III requires Cat5e Ethernet cables.

52 SERCOS Drive Fault

Possible causes

- The SERCOS drive has generated a fault.
- The Phoenix software reports the fault number from the drive.

Suggested actions

- Check the documentation provided by drive manufacturer to identify the error condition.
- Perform the corrective action suggested by the drive manufacturer.
- At the CNC, use the displayed soft key to clear the fault. The soft key performs a reset at the drive.

53 Lost Connection to ArcGlide

Possible causes

- The Hypernet setting in the CNC is set to No.
- The Ethernet cable is not plugged into the Hypernet port on the CNC or the ArcGlide HMI.
- The ArcGlide Ethernet switch is not properly powered.
- The control module, the Hypernet plasma interface board, and the HMI are not addressed correctly.

Suggested actions

- Verify that the Hypernet setting in the CNC is set to On in the Machine Setups screen (Setups>Password>Machine Setups).
- Verify all ArcGlide setup screens.
- Check the Ethernet cables connecting to the HMI, the CNC, and the Hypernet plasma interface board.
- Verify that the Ethernet switch is properly powered and addressed to the same unit number.
- Check the Ethernet cables for damage.

54 Lost Connection to Laser

Possible cause

- The Ethernet cable is not plugged into the Hypernet port on the CNC or HyIntensity Fiber Laser (HFL).

Suggested action

- Check the Ethernet cables connecting the CNC and the HFL.

Errors 55 through 59 are specific to the Edge Pro Ti. The first step for any of these errors is to verify that you have the latest software revision installed.

55 Axis 1 drive fault (EDGE Pro Ti)

Possible cause

- A short circuit in the wiring.
- Servo power over voltage.
- Servo over temperature.
- No cable connected to motor.
- DIP switch on the DC servo PCB for brush/brushless motor is set incorrectly.

Suggested action

- Verify that the LED on the EdgePro Ti DC servo PCB (141281) is illuminated to indicate a problem.
  - Axis 1 fault LED – D21
- Remove the motors from the racks and then move the motor and encoder cables for one axis to another axis. If you get the same error code again, the fault is probably due the PCB or another internal issue. If you get a different error code the problem is due to external cabling or motor faults. This error could also be caused by very high ambient temperatures in the work area.
  - Verify that the wiring is not damaged and the connections are correct and secure if the error code changes.
  - Verify the voltage across the screws of J3 on the back of the PCB is at 60 volts (+/- 5%).
- Verify that the internal temperature shown in the watch window is within the specified operating range -10° C to 40° C (14° F to 104° F).
- Set the DIP switch to the correct setting for the motor being used.

56 Axis 2 drive fault (EDGE Pro Ti)

Possible cause

- A short circuit in the wiring.
- Servo power over voltage.
- Servo over temperature.
- No cable connected to motor.
- DIP switch on the DC servo PCB for brush/brushless motor is set incorrectly.

Suggested action

- Verify that the LED on the EdgePro Ti DC servo PCB (141281) illuminates to indicate a problem.
  - Axis 2 fault LED – D17
- Remove the motors from the racks and then move the motor and encoder cables for one axis to another axis. If you get the same error code again, the fault is probably due the PCB or another internal issue. If you get a different error code the problem is due to external cabling or motor faults. This error could also be caused by very high ambient temperatures in the work area.
  - Verify that the wiring is not damaged and the connections are correct and secure if the error code changes.
  - Verify the voltage across the screws of J3 on the back of the PCB is at 60 volts (+/- 5%).
- Verify that the internal temperature shown in the watch window is within the specified operating range -10° C to 40° C (14° F to 104° F).
- Set the DIP switch to the correct setting for the motor being used.

---

57 Axis 3 drive fault (EDGE Pro Ti)

Possible cause

- A short circuit in the wiring.
- Servo power over voltage.
- Servo over temperature.
- No cable connected to motor.
- DIP switch on the DC servo PCB for brush/brushless motor is set incorrectly.

Suggested action

- Verify that the LED on the EdgePro Ti DC servo PCB (141281) illuminates to indicate a problem.
  - Axis 3 fault LED – D16
- Verify that the wiring is not damaged and the connections are correct and secure.
- Remove the motors from the racks and then move the motor and encoder cables for one axis to another axis. If you get the same error code again, the fault is probably due the PCB or another internal issue. If you get a different error code the problem is due to external cabling or motor faults. This error could also be caused by very high ambient temperatures in the work area.
  - Verify that the wiring is not damaged and the connections are correct and secure if the error code changes.
  - Verify the voltage across the screws of J3 on the back of the PCB is at 60 volts (+/- 5%).
- Verify that the internal temperature shown in the watch window is within the specified operating range -10° C to 40° C (14° F to 104° F).
- Set the DIP switch to the correct setting for the motor being used.

---

58 Axis 4 drive fault (EDGE Pro Ti)

Possible cause

- A short circuit in the wiring.
- Servo power over voltage.
- Servo over temperature.
- No cable connected to motor.
- DIP switch on the DC servo PCB for brush/brushless motor is set incorrectly.

Suggested action

- Verify that the LED on the EdgePro Ti DC servo PCB (141281) illuminates to indicate a problem.
  - Axis 4 fault LED – D13
- Verify that the wiring is not damaged and the connections are correct and secure.
- Remove the motors from the racks and then move the motor and encoder cables for one axis to another axis. If you get the same error code again, the fault is probably due the PCB or another internal issue. If you get a different error code the problem is due to external cabling or motor faults. This error could also be caused by very high ambient temperatures in the work area.
  - Verify that the wiring is not damaged and the connections are correct and secure if the error code changes.
  - Verify the voltage across the screws of J3 on the back of the PCB is at 60 volts (+/- 5%).
- Verify that the internal temperature shown in the watch window is within the specified operating range -10° C to 40° C (14° F to 104° F).
- Set the DIP switch to the correct setting for the motor being used.

59 Servo power failure (EDGE Pro Ti)

Possible cause

- The servo power supply failed.
- The fan inside the servo power supply failed.
- The servo voltage is 20% or more below the expected 60 volts.

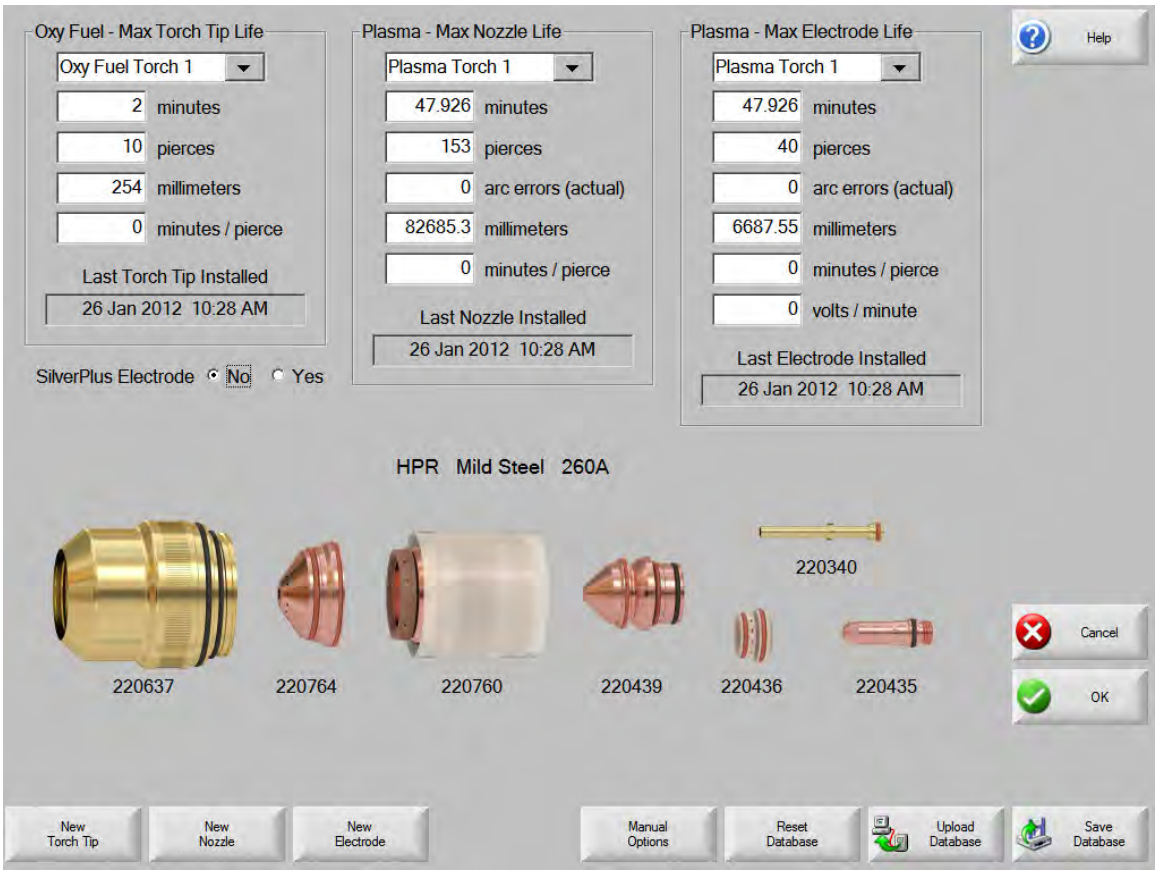
Suggested action

- Verify that the emergency stop wiring is not damaged, the connections are correct and secure, and that the E-stop is functioning properly.
- Verify that the wiring is not damaged and the connections are correct and secure.
- Verify the voltage across the screws of J3 on the back of the PCB is at 60 volts (+/- 5%).
- Replace the servo power supply.

Changing Consumables

This screen tracks consumable life data for statistical analysis. In addition, this feature can prompt the operator that a consumable has reached its expected life by using an output from the CNC to activate an indicator such as a light or alarm. This feature allows the operator to change the consumable and avoid a consumable failure that can affect cut quality or damage the torch.

This Change Consumable feature can only track consumable life data and offer features related to that data. The CNC cannot detect consumable condition or failures.



If the New Torch Tip or New Electrode soft key is pressed every time a torch tip or electrode is changed, the last information for the corresponding consumable will be added to a database. This database shows the date a consumable was changed and how long it lasted in minutes, pierces, and millimeters or inches.

To reset the current consumable value, press the corresponding soft key. The CNC resets the tracking information to zero and starts counting down from the user-defined set point as you cut in the selected mode. The installation date for the selected consumable is updated and the current values for the selected consumable are recorded, with the date, in a database that can be saved to a USB memory stick.

You can set up a Watch Window to view this data during cutting. See *Watch Window Setup* on page 122.

The consumable information that is updated (Oxyfuel torch 1–12 / Plasma torch 1–8) is determined by the Station Select 1–20 inputs.

For example, Plasma Torch 1 torch tip has a limit of 5000 minutes of operation. After 5000 minutes, the Change Consumable output becomes active and is indicated with a lamp or audible alarm. The intent is to set the limits at an expected life value of the consumable so that the operator is reminded to change the consumable when it has reached its expected life.

Auto update max consumable life: When this feature is enabled it tracks the consumable life 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 remains the same until the user changes it manually. This auto update feature can be disabled in the password protected Special Setups screen.

Minutes: The estimated life in time that the torch tip, nozzle, or electrode lasts. This value increases to the maximum life achieved or a maximum value can be entered.

Pierces: The estimated life in pierces that the torch tip, nozzle, or electrode lasts. This value increases to the maximum life achieved or a maximum value can be entered.

Inches or **Millimeters:** The estimated life in distance that the torch tip, nozzle, or electrode lasts. This value increases to the maximum life achieved or a maximum value can be entered.

Minutes per Pierce: Piercing causes additional wear on the consumables. This parameter allows you to enter a value which is added to the overall minutes value for each pierce, providing a more accurate representation of overall consumable wear.

Arc Errors: Arc errors can be tracked using the Arc Error Counter input to the CNC from the plasma supply. The power supply indicates an arc error when the plasma arc does not achieve a long-life ramp down.

Volts per Minute: The Volts per Minute parameter changes the THC Voltage Offset based on the number of minutes that have elapsed while cutting in Plasma 1 or Plasma 2 cut mode. By adding a small fraction of a volt per minute of cutting to the THC voltage offset, the CNC compensates for consumable wear. Volts per Minute applies only to Station 1 or Station 2.

The Volts per Minute will continue to increase the THC voltage offset until you reset the Volts per Minute to 0 and the THC voltage offset to 0.

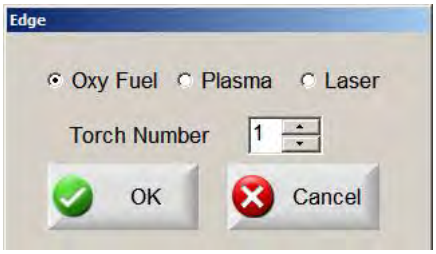
This parameter is available for use only with the Sensor THC while cutting in Set Arc Voltage mode. If cutting in Sample Arc Voltage mode, set the Volts per minute to 0.

Last Torch Tip Installed: Displays the date and time when the selected tip was installed.

Last Electrode Installed: Displays the date and time when the selected electrode was installed.

SilverPlus® Electrode: Choose Yes if you are using the SilverPlus electrode in the torch. The screen updates with the correct part number for the SilverPlus electrode.

New Torch Tip: Press the New Torch Tip soft key to select which Torch Tip has been replaced and to update the database.



New Electrode: Press the New Electrode soft key to select which Electrode has been replaced and to update the database.

Manual Options: Opens the Manual Options screen so that you can reposition the torch to change the consumables.

Reset Database: Resets the values in the database on the CNC and clears the torch tip, nozzle, or electrode information after uploading or saving the database.

Upload Database: Uploads the current database to a host computer.

Save Database: Saves the current database to the USB memory stick.

Saving System Files

You can save the following files to a memory stick or to a .zip file:

- The last part file
- Setups.ini
- Error messages
- Key logs

To save key logs, the Key Logging feature must be set to Yes in the Machine Setups screen.

To save system files:

1. Install a memory stick in one of the USB ports in your CNC.
2. From the Main screen, select Files > Save to Disk > Save System Files to Disk.
3. On the System Files screen, select one or more files from the Select File to Save list.

4. Choose Save to Disk. If preferred, choose Save All to Zip File to have the files saved as Phoenix.zip on the memory stick.