EDGE® Connect

Shape Cutting Control

Installation and Setup Manual

809340 | Revision 4 | English

Register your new Hypertherm system

Benefits of registration

- ☑ **Safety:** Registration allows us to contact you in the unlikely event a safety or quality notification is required.
- ☑ **Education:** Registration gives you free access to online product training content via the Hypertherm Cutting Institute.
- ☑ **Confirmation of ownership:** Registration can serve as proof of purchase in case of an insurance loss.

Go to **www.hypertherm.com/registration** for easy and fast registration.

If you experience any problems with the product registration process, please contact registration@hypertherm.com.

For your records

Serial number: ______________________________

Purchase date: ______________________________

Distributor: ______________________________

____

____

Maintenance notes: ______________________________

____

____

____

____

EDGE, Phoenix, HPR, HPRXD, HPR130XD, HPR260XD, HPR400XD, CutPro, Remote Help, XPR, Powermax, and Hypertherm are trademarks of Hypertherm Inc. and may be registered in the United States and other countries. EtherCAT is a trademark of Beckhoff Automation. All other trademarks are the property of their respective holders.

Environmental stewardship is one of Hypertherm's core values, and it is critical to our success and our customers' success. We are striving to reduce the environmental impact of everything we do. For more information: www.hypertherm.com/environment.

© 2020 Hypertherm, Inc.

EDGE Connect

Installation and Setup Manual

809340
Revision 4

English
Original Instructions

October 2020

Hypertherm, Inc.
Hanover, NH 03755 USA
www.hypertherm.com

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 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
Soporte.Tecnico@hypertherm.com (Technical Service Email)

Hypertherm Plasmatechnik GmbH
Sophie-Scholl-Platz 5
63452 Hanau
Germany
00 800 33 24 97 37 Tel
00 800 49 73 73 29 Fax
31 (0) 165 596900 Tel (Technical Service)
00 800 4973 7843 Tel (Technical Service)
technicalservice.emea@hypertherm.com (Technical Service Email)

Hypertherm (Singapore) Pte Ltd.
82 Genting Lane
Media Centre
Annexe Block #A01-01
Singapore 349567, Republic of Singapore
65 6841 2489 Tel
65 6841 2490 Fax
Marketing.asia@hypertherm.com (Marketing Email)
TechSupportAPAC@hypertherm.com (Technical Service Email)

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
HTJapan.info@hypertherm.com (Main Office Email)
TechSupportAPAC@hypertherm.com (Technical Service Email)

Hypertherm Europe B.V.
Vaartveld 9, 4704 SE
Roosendaal, Nederland
31 165 596907 Tel
31 165 596901 Fax
31 165 596908 Tel (Marketing)
31 (0) 165 596900 Tel (Technical Service)
00 800 4973 7843 Tel (Technical Service)
technicalservice.emea@hypertherm.com
(Technical Service Email)

Hypertherm (Shanghai) Trading Co., Ltd.
B301, 495 ShangZhong Road
Shanghai, 200231
PR China
86-21-80231122 Tel
86-21-80231120 Fax
86-21-80231128 Tel (Technical Service)
techsupport.china@hypertherm.com
(Technical Service Email)

South America & Central America: Hypertherm Brasil Ltda.
Rua Bras Cubas, 231 – Jardim Maia
Guarulhos, SP – Brasil
CEP 07115-030
55 11 2409 2636 Tel
tecnico.sa@hypertherm.com (Technical Service Email)

Hypertherm Korea Branch
#3904. APEC-ro 17. Heaundae-gu. Busan.
Korea 48060
82 (0)51 747 0358 Tel
82 (0)51 701 0358 Fax
Marketing.korea@hypertherm.com (Marketing Email)
TechSupportAPAC@hypertherm.com
(Technical Service Email)

Hypertherm Pty Limited
GPO Box 4836
Sydney NSW 2001, Australia
61 (0) 437 606 995 Tel
61 7 3219 9010 Fax
au.sales@Hypertherm.com (Main Office Email)
TechSupportAPAC@hypertherm.com
(Technical Service Email)

Hypertherm (India) Thermal Cutting Pvt. Ltd
A-18 / B-1 Extension,
Mohan Co-Operative Industrial Estate,
Mathura Road, New Delhi 110044, India
91-11-40521201/ 2/ 3 Tel
91-11 40521204 Fax
HTIndia.info@hypertherm.com (Main Office Email)
TechSupportAPAC@hypertherm.com
(Technical Service Email)

For training and education resources, go to the Hypertherm Cutting Institute (HCI) online at www.hypertherm.com/hci.

ENGLISH

WARNING! Before operating any Hypertherm equipment, read the safety instructions in your product's manual, the *Safety and Compliance Manual* (80669C), *Waterjet Safety and Compliance Manual* (80943C), and *Radio Frequency Warning Manual* (80945C). Failure to follow safety instructions can result in personal injury or in damage to equipment.

Copies of the manuals may accompany the product in electronic and printed formats. You can also obtain copies of the manuals, in all languages available for each manual, from the "Documents library" at www.hypertherm.com.

BG (БЪЛГАРСКИ/BULGARIAN)

ПРЕДУПРЕЖДЕНИЕ! Преди да работите с което и да е оборудване Hypertherm, прочетете инструкциите за безопасност в ръководството на вашия продукт, „Инструкция за безопасност и съответствие“ (80669C), „Инструкция за безопасност и съответствие на Waterjet“ (80943C) и „Инструкция за предупреждение за радиочестота“ (80945C).

Копия на ръководствата може да придружават продукта в електронен и в печатен формат. Можете да получите копия на ръководствата, предлагани на всички езици, от „Documents library“ (Библиотека за документи) на адрес www.hypertherm.com.

CS (ČESKY/CZECH)

VAROVÁNÍ! Před uvedením jakéhokoli zařízení Hypertherm do provozu si přečtěte bezpečnostní pokyny v příručce k produktu a v *Manuálu pro bezpečnost a dodržování předpisů* (80669C), *Manuálu pro bezpečnost a dodržování předpisů při řezání vodním paprskem* (80943C) a *Manuálu varování ohledně rádiových frekvencí* (80945C).

Kopie příruček a manuálů mohou být součástí dodávky produktu, a to v elektronické i tištěné formě. Kopie příruček a manuálů ve všech jazykových verzích, v nichž byly dané příručky a manuály vytvořeny, naleznete v „Knihovně dokumentů“ na webových stránkách www.hypertherm.com.

DA (DANSK/DANISH)

ADVARSEL! Inden Hypertherm udstyr tages i brug skal sikkerhedsinstruktionerne i produktets manual og i *Manual om sikkerhed og overholdelse af krav* (80669C), *Manual om sikkerhed og overholdelse af krav for vandstråleskæring* (80943C), og *Manual om radiofrekvensadvarsel* (80945C), gennemlæses.

Kopier af manualerne kan ledsage produktet i elektroniske og trykte formater. Du kan også få kopier af manualer, på alle sprog der er til rådighed for hver manuel, fra "Dokumentbiblioteket" på www.hypertherm.com.

DE (DEUTSCH/GERMAN)

WARNUNG! Bevor Sie ein Hypertherm-Gerät in Betrieb nehmen, lesen Sie bitte die Sicherheitsanweisungen in Ihrer Bedienungsanleitung, das *Handbuch für Sicherheit und Übereinstimmung* (80669C), das *Handbuch für Sicherheit und Compliance bei Wasserstrahl-Schneidanlagen* (80943C) und das *Handbuch für Hochfrequenz-Warnung* (80945C).

Bedienungsanleitungen und Handbücher können dem Gerät in elektronischer Form oder als Druckversion beiliegen. Alle Handbücher und Anleitungen können in den jeweils verfügbaren Sprachen auch in der „Dokumente-Bibliothek“ unter www.hypertherm.com heruntergeladen werden.

ES (ESPAÑOL/SPANISH)

¡ADVERTENCIA! Antes de operar cualquier equipo Hypertherm, lea las instrucciones de seguridad del manual de su producto, del *Manual de seguridad y cumplimiento* (80669C), del *Manual de seguridad y cumplimiento en corte con chorro de agua* (80943C) y del *Manual de advertencias de radiofrecuencia* (80945C).

Pueden venir copias de los manuales en formato electrónico e impreso junto con el producto. También se pueden obtener copias de los manuales, en todos los idiomas disponibles para cada manual, de la "Biblioteca de documentos" en www.hypertherm.com.

ET (EESTI/ESTONIAN)

HOIATUS! Enne Hyperthermi mis tahes seadme kasutamist lugege läbi toote kasutusjuhendis olevad ohutusjuhised ning *Ohutus- ja vastavusjuhend* (80669C), *Veejoa ohutuse ja vastavuse juhend* (80943C) ja *Raadiosageduse hoiatusjuhend* (80945C). Ohutusjuhiste eiramine võib põhjustada vigastusi ja kahjustada seadmeid.

Juhiste koopiad võivad olla tootega kaasas elektroonilises ja trükivormingus. Juhiste koopiaid kõigis iga käsiraamatu jaoks saadaolevas keeles saate hankida ka „Documents library (dokumentide raamatukogust)" lehel www.hypertherm.com.

FI (SUOMI/FINNISH)

VAROITUS! Ennen minkään Hypertherm-laitteen käyttöä lue tuotteen käyttöoppaassa olevat turvallisuusohjeet, *turvallisuuden ja vaatimustenmukaisuuden käsikirja* (80669C), vesileikkauksen *turvallisuuden ja vaatimustenmukaisuuden käsikirja* (80943C) ja *radiotaajuusvaroitusten käsikirja* (80945C).

Käyttöoppaiden kopiot voivat olla tuotteen mukana elektronisessa ja tulostetussa muodossa. Voit saada käyttöoppaiden kopiot kaikilla kielillä "latauskirjastosta", joka on osoitteessa www.hypertherm.com.

FR (FRANÇAIS/FRENCH)

AVERTISSEMENT! Avant d'utiliser tout équipement Hypertherm, lire les consignes de sécurité du manuel de votre produit, du *Manuel de sécurité et de conformité* (80669C), du *Manuel de sécurité et de conformité du jet d'eau* (80943C) et du *Manuel d'avertissement relatif aux radiofréqunces* (80945C).

Des copies de ces manuels peuvent accompagner le produit en format électronique et papier. Vous pouvez également obtenir des copies de chaque manuel dans toutes les langues disponibles à partir de la « Bibliothèque de documents » sur www.hypertherm.com.

GR (ΕΛΛΗΝΙΚΆ/GREEK)

ΠΡΟΕΙΔΟΠΟΙΗΣΗ! Πριν θέσετε σε λειτουργία οποιονδήποτε εξοπλισμό της Hypertherm, διαβάστε τις οδηγίες ασφαλείας στο εγχειρίδιο του προϊόντος και στο *εγχειρίδιο ασφάλειας και συμμόρφωσης* (80669C), στο *εγχειρίδιο ασφάλειας και συμμόρφωσης του waterjet* (80943C) και στο *εγχειρίδιο προειδοποιήσεων για τις ραδιοσυχνότητες* (80945C).

Αντίγραφα των εγχειριδίων μπορεί να συνοδεύουν το προϊόν σε ηλεκτρονική και έντυπη μορφή. Μπορείτε, επίσης, να λάβετε αντίγραφα των εγχειριδίων σε όλες τις γλώσσες που διατίθενται για κάθε εγχειρίδιο από την ψηφιακή βιβλιοθήκη εγγράφων (Documents library) στη διαδικτυακή τοποθεσία www.hypertherm.com.

HU (MAGYAR/HUNGARIAN)

VIGYÁZAT! Mielőtt bármilyen Hypertherm berendezést üzemeltetne, olvassa el a biztonsági információkat a termék kézikönyvében, a *Biztonsági és szabálykövetési kézikönyvben* (80669C), a *Vízsugaras biztonsági és szabálykövetési kézikönyvben* (80943C) és a *Rádiófrekvenciás figyelmeztetéseket tartalmazó kézikönyvben* (80945C).

A termékhez a kézikönyv példányai elektronikus és nyomtatott formában is mellékelve lehetnek. A kézikönyvek példányai (minden nyelven) a www.hypertherm.com weboldalon a „Documents library" (Dokumentum könyvtár) részben is beszerezhetők.

ID (BAHASA INDONESIA/INDONESIAN)

PERINGATAN! Sebelum mengoperasikan peralatan Hypertherm, bacalah petunjuk keselamatan dalam manual produk Anda, *Manual Keselamatan dan Kepatuhan* (80669C), *Manual Keselamatan dan Kepatuhan Jet Air* (80943C), dan *Manual Peringatan Frekuensi Radio* (80945C). Kegagalan mengikuti petunjuk keselamatan dapat menyebabkan cedera pribadi atau kerusakan pada peralatan.

Produk mungkin disertai salinan manual dalam format elektronik maupun cetak. Anda juga dapat memperoleh salinan manual, dalam semua bahasa yang tersedia untuk setiap manual, dari "Perpustakaan dokumen" di www.hypertherm.com.

IT (ITALIANO/ITALIAN)

AVVERTENZA! Prima di usare un'attrezzatura Hypertherm, leggere le istruzioni sulla sicurezza nel manuale del prodotto, nel *Manuale sulla sicurezza e la conformità* (80669C), nel *Manuale sulla sicurezza e la conformità Waterjet* (80943C) e nel *Manuale di avvertenze sulla radiofrequenza* (80945C).

Il prodotto può essere accompagnato da copie elettroniche e cartacee del manuale. È anche possibile ottenere copie del manuale, in tutte le lingue disponibili per ogni manuale, dall'"Archivio documenti" all'indirizzo www.hypertherm.com.

JA (日 本語/JAPANESE)

警告！Hypertherm 機器を操作する前に、この製品説明書にある安全情報、「安全とコンプライアンスマニュアル」(80669C)、「ウォータージェットの安全とコンプライアンス」(80943C)、「高周波警告」(80945C) をお読みください。

説明書のコピーは、電子フォーマット、または印刷物として製品に同梱されています。各説明書は、www.hypertherm.com の「ドキュメントライブラリ」から各言語で入手できます。

KO (한국어/KOREAN)

경고! Hypertherm 장비를 사용하기 전에 제품 설명서와 안전 및 규정 준수 설명서(80669C), 워터젯 안전 및 규정 준수 설명서(80943C) 그리고 무선 주파수 경고 설명서(80945C)에 나와 있는 안전 지침을 읽으십시오.

전자 형식과 인쇄된 형식으로 설명서 사본이 제품과 함께 제공될 수 있습니다. www.hypertherm.com 의 'Documents library (문서 라이브러리)' 에서도 모든 언어로 이용할 수 있는 설명서 사본을 얻을수 있습니다.

NE (NEDERLANDS/DUTCH)

WAARSCHUWING! Lees voordat u Hypertherm-apparatuur gebruikt de veiligheidsinstructies in de producthandleiding, in de *Veiligheids- en nalevingshandleiding* (80669C) in de *Veiligheids- en nalevingshandleiding voor waterstralen* (80943C) en in de *Waarschuwingshandleiding radiofrequentie* (80945C).

De handleidingen kunnen in elektronische en gedrukte vorm met het product worden meegeleverd. De handleidingen, elke handleiding beschikbaar in alle talen, zijn ook verkrijgbaar via de "Documentenbibliotheek" op www.hypertherm.com.

NO (NORSK/NORWEGIAN)

ADVARSEL! Før du bruker noe Hypertherm-utstyr, må du lese sikkerhetsinstruksjonene i produktets håndbok, *håndboken om sikkerhet og samsvar* (80669C), *håndboken om vannjet sikkerhet og samsvar* (80943C), og *håndboken om radiofrekvensadvarsler* (80945C).

Eksemplarer av håndbøkene kan medfølge produktet i elektroniske og trykte utgaver. Du kan også få eksemplarer av håndbøkene i alle tilgjengelige språk for hver håndbok fra dokumentbiblioteket på www.hypertherm.com.

PL (POLSKI/POLISH)

OSTRZEŻENIE! Przed rozpoczęciem obsługi jakiegokolwiek systemu firmy Hypertherm należy się zapoznać z instrukcjami bezpieczeństwa zamieszczonymi w podręczniku produktu, w p*odręczniku bezpieczeństwa i zgodności* (80669C), *podręczniku bezpieczeństwa i zgodności systemów strumienia wody* (80943C) oraz *podręczniku z ostrzeżeniem o częstotliwości radiowej* (80945C).

Do produktu mogą być dołączone kopie podręczników w formacie elektronicznym i drukowanym. Kopie podręczników, w każdym udostępnionym języku, można również znaleźć w „Bibliotece dokumentów" pod adresem www.hypertherm.com.

PT (PORTUGUÊS/PORTUGUESE)

ADVERTÊNCIA! Antes de operar qualquer equipamento Hypertherm, leia as instruções de segurança no manual do seu produto, no *Manual de Segurança e de Conformidade* (80669C), no *Manual de Segurança e de Conformidade do Waterjet* (80943C) e no *Manual de Advertência de radiofrequência* (80945C).

Cópias dos manuais podem acompanhar os produtos nos formatos eletrônico e impresso. Também é possível obter cópias dos manuais em todos os idiomas disponíveis para cada manual na "Biblioteca de documentos" em www.hypertherm.com.

RO (ROMÂNĂ/ROMANIAN)

AVERTIZARE! Înainte de utilizarea oricărui echipament Hypertherm, citiți instrucțiunile de siguranță din manualul produsului, *manualul de siguranță și conformitate* (80669C), *manualul de siguranță și conformitate Waterjet* (80943C) și din *manualul de avertizare privind radiofrecvența* (80945C).

Produsul poate fi însoțit de copii ale manualului în format tipărit și electronic. De asemenea, dumneavoastră puteți obține copii ale manualelor, în toate limbile disponibile pentru fiecare manual, din cadrul secțiunii „Bibliotecă documente" aflată pe site-ul www.hypertherm.com.

RU (РУССКИЙ/RUSSIAN)

БЕРЕГИСЬ! Перед работой с любым оборудованием Hypertherm ознакомьтесь с инструкциями по безопасности, представленными в руководстве, которое поставляется вместе с продуктом, в *Руководстве по безопасности и соответствию* (80669C), в *Руководстве по безопасности и соответствию для водоструйной резки* (80943C) и *Руководстве по предупреждению о радиочастотном излучении* (80945C).

Копии руководств, которые поставляются вместе с продуктом, могут быть представлены в электронном и бумажном виде. Копии руководств на всех языках, на которые переведено то или иное руководство, можно также загрузить в разделе «Библиотека документов» на веб-сайте www.hypertherm.com.

SK (SLOVENČINA/SLOVAK)

VÝSTRAHA! Pred použitím akéhokoľvek zariadenia od spoločnosti Hypertherm si prečítajte bezpečnostné pokyny v návode na obsluhu vášho zariadenia a v *Manuáli o bezpečnosti a súlade s normami* (80669C), *Manuáli o bezpečnosti a súlade s normami pre systém rezania vodou* (80943C) a v *Manuáli s informáciami o rádiofrekvencii* (80945C).

Kópia návodu, ktorá je dodávaná s produktom, môže mať elektronickú alebo tlačenú podobu. Kópie návodov, vo všetkých dostupných jazykoch, sú k dispozícii aj v sekcii z „knižnice Dokumenty" na www.hypertherm.com.

SL (SLOVENŠČINA/SLOVENIAN)

OPOZORILO! Pred uporabo katerekoli Hyperthermove opreme preberite varnostna navodila v priročniku vašega izdelka, v *Priročniku za varnost in skladnost* (80669C), v *Priročniku za varnost in skladnost sistemov rezanja z vodnim curkom* (80943C) in v *Priročniku Opozorilo o radijskih frekvencah* (80945C).

Izdelku so lahko priloženi izvodi priročnikov v elektronski ali tiskani obliki. Izvode priročnikov v vseh razpoložljivih jezikih si lahko prenesete tudi iz knjižnice dokumentov "Documents library" na naslovu www.hypertherm.com.

SR (SRPSKI/SERBIAN)

UPOZORENJE! Pre rukovanja bilo kojom Hyperthermovom opremom pročitajte uputstva o bezbednosti u svom priručniku za proizvod, *Priručniku o bezbednosti i usaglašenosti* (80669C), *Priručniku o bezbednosti i usaglašenosti Waterjet tehnologije* (80943C) i *Priručniku sa upozorenjem o radio-frekvenciji* (80945C).

Može se dogoditi da kopije priručnika prate proizvod u elektronskom i štampanom formatu. Takođe možete da pronađete kopije priručnika, na svim jezicima koji su dostupni za svaki od priručnika, u "Biblioteci dokumenata" ("Documents library") na www.hypertherm.com.

SV (SVENSKA/SWEDISH)

VARNING! Läs häftet säkerhetsinformationen i din produkts *säkerhets- och efterlevnadsmanual* (80669C), *säkerhets- och efterlevnadsmanualen för Waterjet* (80943C) och *varningsmanualen för radiofrekvenser* (80945C) för viktig säkerhetsinformation innan du använder eller underhåller Hypertherm-utrustning.

Kopior av manualen kan medfölja produkten i elektronisk och tryckform. Du hittar även kopior av manualerna i alla tillgängliga språk i dokumentbiblioteket (Documents library) på www.hypertherm.com.

TH (ภาษาไทย/THAI)

คำเตือน! ก่อนการใช้งานอุปกรณ์ของ Hypertherm ทั้งหมด โปรดอ่านคำแนะนำด้านความปลอดภัยในคู่มือการใช้สินค้า คู่มือด้านความปลอดภัยและการปฏิบัติตาม (80669C), คู่มือด้านความปลอดภัยและการปฏิบัติตามสำหรับการใช้หัวตัดระบบวอเตอร์เจ็ต (80943C) และ คู่มือคำเตือนเกี่ยวกับความถี่วิทยุ (80945C) การไม่ปฏิบัติตามคำแนะนำด้านความปลอดภัยอาจส่งผลให้เกิดการบาดเจ็บหรือเกิดความเสียหายต่ออุปกรณ์

สินค้าอาจมีสำเนาคู่มือในรูปแบบอิเล็กทรอนิกส์และแบบสิ่งพิมพ์แนบมาด้วย นอกจากนี้ คุณสามารถขอรับสำเนาคู่มือแต่ละประเภทเป็นภาษาต่าง ๆ ที่มีให้ใช้งานได้ที่ “คลังเอกสาร” ในเว็บไซต์ www.hypertherm.com

TR (TÜRKÇE/TURKISH)

UYARI! Bir Hypertherm ekipmanını çalıştırmadan önce, ürününüzün kullanım kılavuzunda, *Güvenlik ve Uyumluluk Kılavuzu’nda* (80669C), *Su Jeti Güvenlik ve Uyumluluk Kılavuzu’nda* (80943C) ve *Radyo Frekansı Uyarısı Kılavuzu’nda* (80945C) yer alan güvenlik talimatlarını okuyun.

Kılavuzların kopyaları, elektronik ve basılı formatta ürünle birlikte verilebilir. Her biri tüm dillerde yayınlanan kılavuzların kopyalarını www.hypertherm.com adresindeki “Documents library” (Dosyalar kitaplığı) başlığından da elde edebilirsiniz.

VI (TIẾNG VIỆT/VIETNAMESE)

CẢNH BÁO! Trước khi vận hành bất kỳ thiết bị Hypertherm nào, hãy đọc các hướng dẫn an toàn trong hướng dẫn sử dụng sản phẩm của bạn, *Sổ tay An toàn và Tuân thủ* (80669C), *Sổ tay An toàn và Tuân thủ Tia nước* (80943C), và *Hướng dẫn Cảnh báo Tần số Vô tuyến* (80945C). Không tuân thủ các hướng dẫn an toàn có thể dẫn đến thương tích cá nhân hoặc hư hỏng thiết bị.

Bản sao của các hướng dẫn sử dụng có thể đi kèm sản phẩm ở định dạng điện tử và bản in. Bạn cũng có thể lấy bản sao của các hướng dẫn sử dụng, thuộc tất cả các ngôn ngữ hiện có cho từng hướng dẫn sử dụng, từ “Thư viện tài liệu” tại địa chỉ www.hypertherm.com.

ZH-CN (简 体中文/CHINESE SIMPLIFIED)

警告！ 在操作任何海宝设备之前，请阅读产品手册、《安全和法规遵守手册》 (80669C)、《水射流安全和法规遵守手册》 (80943C) 以及《射频警告手册》 (80945C) 中的安全操作说明。

随产品提供的手册可能提供电子版和印刷版两种格式。您也可从 “Documents library” （文档资料库）中获取每本手册所有可用语言的副本，网址为 www.hypertherm.com.

ZH-TW (繁 體中文/CHINESE TRADITIONAL)

警告！在操作任何 Hypertherm 設備前，請先閱讀您產品手冊內的安全指示，包括 《安全和法規遵從手冊》(80669C) 、《水刀安全和法規遵從手冊》(80943C)，以及 《無線電頻率警示訊號手冊》(80945C)。

手冊複本可能以電子和印刷格式隨附產品提供。您也可以在 www.hypertherm.com 的 「文檔資料庫」內獲取所有手冊的多語種複本。

Contents

Introduction

Hypertherm's CE-marked equipment is built in compliance with standard EN60974-10. The equipment should be installed and used in accordance with the information below to achieve electromagnetic compatibility.

The limits required by EN60974-10 may not be adequate to completely eliminate interference when the affected equipment is in close proximity or has a high degree of sensitivity. In such cases it may be necessary to use other measures to further reduce interference.

This cutting equipment is designed for use only in an industrial environment.

Installation and use

The user is responsible for installing and using the plasma equipment according to the manufacturer's instructions.

If electromagnetic disturbances are detected then it shall be the responsibility of the user to resolve the situation with the technical assistance of the manufacturer. In some cases this remedial action may be as simple as earthing the cutting circuit, see *Earthing of the workpiece*. In other cases, it could involve constructing an electromagnetic screen enclosing the power source and the work complete with associated input filters. In all cases, electromagnetic disturbances must be reduced to the point where they are no longer troublesome.

Assessment of area

Before installing the equipment, the user shall make an assessment of potential electromagnetic problems in the surrounding area. The following shall be taken into account:

- **a.** Other supply cables, control cables, signaling and telephone cables; above, below and adjacent to the cutting equipment.
- **b.** Radio and television transmitters and receivers.
- **c.** Computer and other control equipment.
- **d.** Safety critical equipment, for example guarding of industrial equipment.
- **e.** Health of the people around, for example the use of pacemakers and hearing aids.
- **f.** Equipment used for calibration or measurement.
- **g.** Immunity of other equipment in the environment. User shall ensure that other equipment being used in the environment is compatible. This may require additional protection measures.
- **h.** Time of day that cutting or other activities are to be carried out.

The size of the surrounding area to be considered will depend on the structure of the building and other activities that are taking place. The surrounding area may extend beyond the boundaries of the premises.

Methods of reducing emissions

Mains supply

Cutting equipment must be connected to the mains supply according to the manufacturer's recommendations. If interference occurs, it may be necessary to take additional precautions such as filtering of the mains supply.

Consideration should be given to shielding the supply cable of permanently installed cutting equipment, in metallic conduit or equivalent. Shielding should be electrically continuous throughout its length. The shielding should be connected to the cutting mains supply so that good electrical contact is maintained between the conduit and the cutting power source enclosure.

Maintenance of cutting equipment

The cutting equipment must be routinely maintained according to the manufacturer's recommendations. All access and service doors and covers should be closed and properly fastened when the cutting equipment is in operation. The cutting equipment should not be modified in any way, except as set forth in and in accordance with the manufacturer's written instructions. For example, the spark gaps of arc striking and stabilizing devices should be adjusted and maintained according to the manufacturer's recommendations.

Cutting cables

The cutting cables should be kept as short as possible and should be positioned close together, running at or close to the floor level.

Equipotential bonding

Bonding of all metallic components in the cutting installation and adjacent to it should be considered.

However, metallic components bonded to the workpiece will increase the risk that the operator could receive a shock by touching these metallic components and the electrode (nozzle for laser heads) at the same time.

The operator should be insulated from all such bonded metallic components.

Earthing of the workpiece

Where the workpiece is not bonded to earth for electrical safety, nor connected to earth because of its size and position, for example, ship's hull or building steel work, a connection bonding the workpiece to earth may reduce emissions in some, but not all instances. Care should be taken to prevent the earthing of the workpiece increasing the risk of injury to users, or damage to other electrical equipment. Where necessary, the connection of the workpiece to earth should be made by a direct connection to the workpiece, but in some countries where direct connection is not permitted, the bonding should be achieved by suitable capacitances selected according to national regulations.

Note: The cutting circuit may or may not be earthed for safety reasons. Changing the earthing arrangements should only be authorized by a person who is competent to assess whether the changes will increase the risk of injury, for example, by allowing parallel cutting current return paths which may damage the earth circuits of other equipment.
Further guidance is provided in IEC 60974-9, Arc Welding Equipment, Part 9: Installation and Use.

Screening and shielding

Selective screening and shielding of other cables and equipment in the surrounding area may alleviate problems of interference. Screening of the entire plasma cutting installation may be considered for special applications.

Attention

Genuine Hypertherm parts are the factory-recommended replacement parts for your Hypertherm system. Any damage or injury caused by the use of other than genuine Hypertherm parts may not be covered by the Hypertherm warranty, and will constitute misuse of the Hypertherm Product.

You are solely responsible for the safe use of the Product. Hypertherm does not and cannot make any guarantee or warranty regarding the safe use of the product in your environment.

General

Hypertherm, Inc. warrants that its Products shall be free from defects in materials and workmanship for the specific periods of time set forth herein and as follows: if Hypertherm is notified of a defect (i) with respect to the plasma power supply within a period of two (2) years from the date of its delivery to you, with the exception of Powermax brand power supplies, which shall be within a period of three (3) years from the date of delivery to you, and (ii) with respect to the torch and leads within a period of one (1) year from its date of delivery to you, with the exception of the HPRXD short torch with integrated lead, which shall be within a period of six (6) months from the date of delivery to you, and with respect to torch lifter assemblies within a period of one (1) year from its date of delivery to you, and with respect to Automation products one (1) year from its date of delivery to you, with the exception of the EDGE Connect CNC, EDGE Connect T CNC, EDGE Connect TC CNC, EDGE Pro CNC, EDGE Pro Ti CNC, MicroEDGE Pro CNC, and ArcGlide THC, which shall be within a period of two (2) years from the date of delivery to you, and (iii) with respect to HyIntensity fiber laser components within a period of two (2) years from the date of its delivery to you, with the exception of laser heads and beam delivery cables, which shall be within a period of one (1) year from its date of delivery to you.

All third-party engines, engine accessories, alternators, and alternator accessories are covered by the respective manufacturers' warranties and not covered by this warranty.

This warranty shall not apply to any Powermax brand power supplies that have been used with phase converters. In addition, Hypertherm does not warranty systems that have been damaged as a result of poor power quality, whether from phase converters or incoming line power. This warranty shall not apply to any product which has been incorrectly installed, modified, or otherwise damaged.

Hypertherm provides repair, replacement or adjustment of the Product as the sole and exclusive remedy, if and only if the warranty set forth herein properly is invoked and applies. Hypertherm, at its sole option, shall repair, replace, or adjust, free of charge, any defective Products covered by this warranty which shall be returned with Hypertherm's prior authorization (which shall not be unreasonably withheld), properly packed, to Hypertherm's place of business in Hanover, New Hampshire, or to an authorized Hypertherm repair facility, all costs, insurance and freight pre paid by the customer. Hypertherm shall not be liable for any repairs, replacement, or adjustments of Products covered by this warranty, except those made pursuant to this paragraph and with Hypertherm's prior written consent.

The warranty set forth above is exclusive and is in lieu of all other warranties, express, implied, statutory, or otherwise with respect to the Products or as to the results which may be obtained therefrom, and all implied warranties or conditions of quality or of merchantability or fitness for a particular purpose or against infringement. The foregoing shall constitute the sole and exclusive remedy for any breach by Hypertherm of its warranty.

Distributors/OEMs may offer different or additional warranties, but Distributors/OEMs are not authorized to give any additional warranty protection to you or make any representation to you purporting to be binding upon Hypertherm.

Patent indemnity

Except only in cases of products not manufactured by Hypertherm or manufactured by a person other than Hypertherm not in strict conformity with Hypertherm's specifications and in cases of designs, processes, formulae, or combinations not developed or purported to be developed by Hypertherm, Hypertherm will have the right to defend or settle, at its own expense, any suit or proceeding brought against you alleging that the use of the Hypertherm product, alone and not in combination with any other product not supplied by Hypertherm, infringes any patent of any third party. You shall notify Hypertherm promptly upon learning of any action or threatened action in connection with any such alleged infringement (and in any event no longer than fourteen (14) days after learning of any action or threat of action), and Hypertherm's obligation to defend shall be conditioned upon Hypertherm's sole control of, and the indemnified party's cooperation and assistance in, the defense of the claim.

Limitation of liability

In no event shall Hypertherm be liable to any person or entity for any incidental, consequential direct, indirect, punitive or exemplary damages (including but not limited to lost profits) regardless of whether such liability is based on breach of contract, tort, strict liability, breach of warranty, failure of essential purpose, or otherwise, and even if advised of the possibility of such damages. Hypertherm shall not be liable for any losses to Distributor based on down time, lost production or lost profits. It is the intention of the Distributor and Hypertherm that this provision be construed by a court as being the broadest limitation of liability consistent with applicable law.

National and local codes

National and local codes governing plumbing and electrical installation shall take precedence over any instructions contained in this manual. In no event shall Hypertherm be liable for injury to persons or property damage by reason of any code violation or poor work practices.

Liability cap

In no event shall Hypertherm's liability, if any, whether such liability is based on breach of contract, tort, strict liability, breach of warranties, failure of essential purpose or otherwise, for any claim, action, suit or proceeding (whether in court, arbitration, regulatory proceeding or otherwise) arising out of or relating to the use of the Products exceed in the aggregate the amount paid for the Products that gave rise to such claim.

Insurance

At all times you will have and maintain insurance in such quantities and types, and with coverage sufficient and appropriate to defend and to hold Hypertherm harmless in the event of any cause of action arising from the use of the products.

Transfer of rights

You may transfer any remaining rights you may have hereunder only in connection with the sale of all or substantially all of your assets or capital stock to a successor in interest who agrees to be bound by all of the terms and conditions of this Warranty. Within thirty (30) days before any such transfer occurs, you agree to notify in writing Hypertherm, which reserves the right of approval. Should you fail timely to notify Hypertherm and seek its approval as set forth herein, the Warranty set forth herein shall be null and void and you will have no further recourse against Hypertherm under the Warranty or otherwise.

Waterjet product warranty coverage

Product	Parts coverage
HyPrecision pumps	27 months from the ship date, or 24 months from the date of proven installation, or 4,000 hours, whichever occurs first
PowerDredge abrasive removal system	15 months from the ship date or 12 months from the date of proven installation, whichever occurs first
EcoSift abrasive recycling system	15 months from the ship date or 12 months from the date of proven installation, whichever occurs first
Abrasive metering devices	15 months from the ship date or 12 months from the date of proven installation, whichever occurs first
On/off valve air actuators	15 months from the ship date or 12 months from the date of proven installation, whichever occurs first
Diamond orifices	600 hours of use with the use of a thimble filter and compliance with Hypertherm's water quality requirements

Consumable parts are not covered by this warranty. Consumable parts include, but are not limited to, high-pressure water seals, check valves, cylinders, bleed-down valves, low-pressure seals, high-pressure tubing, low- and high-pressure water filters and abrasive collection bags. All third-party pumps, pump accessories, hoppers, hopper accessories, dryer boxes, dryer box accessories and plumbing accessories are covered by the respective manufacturers' warranties and not covered by this warranty.

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 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. "<u>Designated Hypertherm Patents</u>" shall mean United States Patent Nos. 8,338,739, 8,354,610, 8,354,609, 8,541,711, 8,710,395, and 8,436,270, including foreign equivalents, and any patents issuing therefrom; "<u>Hypertherm Plasma Systems</u>" shall mean Hypertherm HPR XD plasma systems, XPR plasma systems, or systems designated as True Hole enabled; "<u>Hypertherm Technology</u>" 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; "<u>Controller Platform</u>" shall mean Hypertherm computer numerical controller and/or Hypertherm software platform supplied with this license; and "<u>End User Customer(s)</u>" 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.

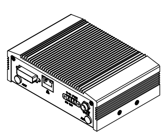
1 Specifications and Installation

Overview

The EDGE Connect is a computer numerical controller (CNC) designed as a shape cutting controller. EDGE Connect CNCs use Hypertherm's Phoenix® software and EtherCAT® communication for motion and cutting control.

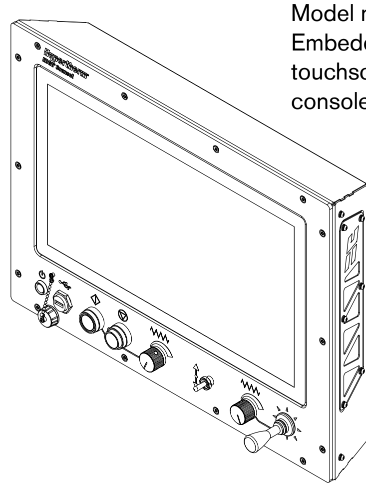
There are two models of the EDGE Connect CNC: the EDGE Connect and the EDGE Connect TC.

EDGE Connect



Model number 090184
Embedded-CNC only

EDGE Connect TC



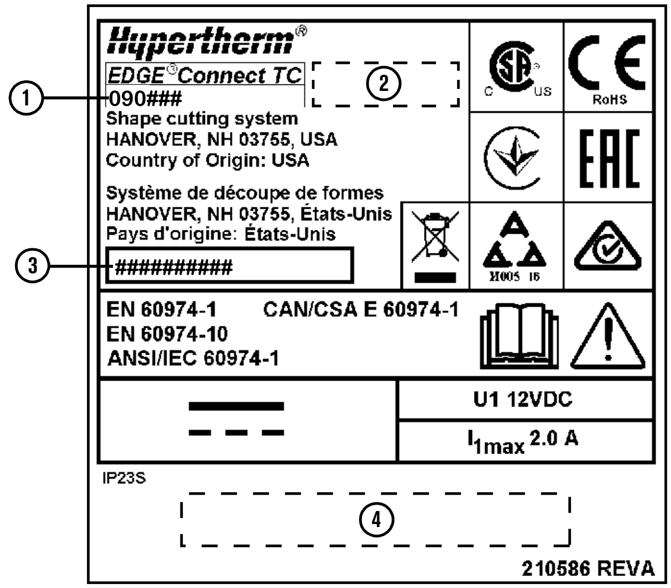
Model number 090198*
Embedded-CNC, 495.3 mm/19.5 inch touchscreen, and hardware operator console

* Model number 090198 (495.3 mm/19.5 inch touchscreen) replaced model number 090185 (469.9 mm/18.5 inch touchscreen).

Data plate

The data plate shows the CNC's part number and serial number. If you contact Hypertherm Technical Service, they ask for this information. You can find the data plate on the bottom of the EDGE Connect or on the rear of the EDGE Connect TC.

Figure 1



1 Part number
2 Part number bar code
3 Serial number (YYMM######, where YY is the year and MM is the month of manufacture)
4 Serial number bar code

Upon receipt

- Make sure that you received all items on your order in good condition. Contact your distributor if any parts are damaged or missing.
  - ❑ EDGE Connect/EDGE Connect TC
  - ❑ HASP software key (installed)
  - ❑ USB memory stick with product information
  - ❑ Wireless antennas
  - ❑ External power supply
  - ❑ Input power cord (United States IEC)
  - ❑ Pluggable terminal block (EDGE Connect only)
  - ❑ Other optional items that appear on your order
- Inspect the items for damage that may have occurred during shipment. If you find evidence of damage, see *Claims* below. All communications regarding this equipment must include the model number and serial number.
- Record your product information on the inside front cover of this manual, then use the instructions there to register your product's serial number at www.hypertherm.com.
- Before you set up this equipment, read the safety information included with your equipment. Failure to follow safety instructions can result in personal injury or in damage to equipment.

Claims

- Claims for damage during shipment – If your equipment was damaged during shipment, file a claim with the carrier. You can contact Hypertherm for a copy of the bill of lading. If you need additional assistance, call the nearest Hypertherm office listed in the front of this manual.
- Claims for defective or missing merchandise – If any component is missing or defective, contact your Hypertherm distributor. If you need additional assistance, call the nearest Hypertherm office listed in the front of this manual.

Installation requirements

All installation and service of the electrical systems must obey national and local electrical codes. A qualified person must do this work.

Contact the nearest Hypertherm Technical Service team listed in the front of this manual, or your authorized Hypertherm distributor with any technical questions.

Placement of system components

- Put all system components into position prior to making electrical, gas, and interface connections.
- Ground all system components to earth. See *Recommended grounding and shielding* on page 55 for details.

Embedded-CNC specifications

Processor	Intel® Celeron® J1900, Quad Core
Operating system	Windows® 10 embedded
RAM	4 GB
Hard drive	Solid-state, SATA drive, greater than or equal to120 GB
Wireless	Complies with 802.11G and 802.11N standards

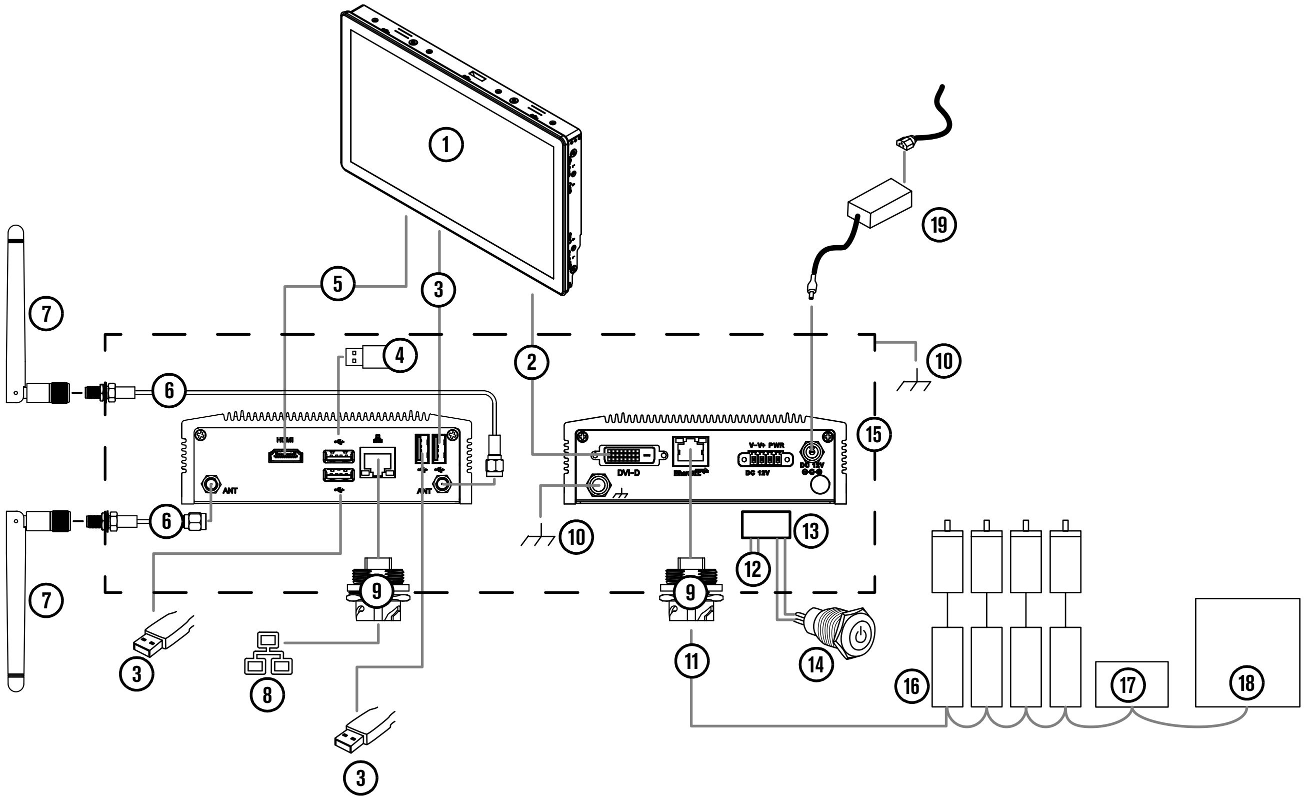
Environmental requirements

Temperature	-10°C to 40°C (14°F to 104°F)
Humidity	95% maximum relative humidity
Ingress protection EDGE Connect	IP2X Protect the equipment from exposure to excessive moisture.
Ingress protection EDGE Connect TC	IP23 Protect the equipment from exposure to excessive moisture.
Altitude	2,000 m (6,561 ft) maximum
Pollution	Pollution Degree 2

You must install the EDGE Connect in an enclosure to protect it from the environment. For enclosure recommendations, see *Enclosure* on page 32.

EDGE Connect (model number 090184)

Figure 2 – EDGE Connect system diagram



1	Monitor	**11**	EtherCAT cable***
2	DVI-D cable*	**12**	Terminal block power-in†
3	USB cable or device**	**13**	Terminal block
4	HASP software key	**14**	Power button
5	HDMI cable*	**15**	Metal enclosure
6	Wireless extensions	**16**	Drives and motors††
7	Wireless antennas	**17**	I/O module
8	Ethernet cable	**18**	Plasma power supply***
9	Bulkhead connector	**19**	External power supply and input power cord**
10	Chassis ground		

* Choose HDMI or DVI-D.

** If you use a USB hub, use only 1 USB hub.

*** If you want to connect a Powermax® or MAXPRO® to an EDGE Connect CNC, use a discrete connection through the drive I/O or an I/O module on the network.

† Connect the input power to the terminal block or barrel connector. Do not use either for power out.

†† Make sure that you can remove power from the drives separately from the CNC so that you can troubleshoot network faults or errors if necessary. Refer to the drive manufacturer's documentation for instructions on drive installation.

Before you begin

Enclosure

Make an enclosure for your CNC. You must install the EDGE Connect inside an enclosure. The enclosure is customer supplied and must meet the following specifications.

- The enclosure must be metal.
- The enclosure must meet all local and national codes.
- For the Ethernet and EtherCAT connections, you must use bulkhead connectors bonded to the enclosure to prevent noise. (See *Ethernet and EtherCAT bulkhead connector* on page 336.)

Monitor

Choose a monitor for your CNC. A touchscreen is suggested but not required. The monitor is customer supplied and must support the requirements listed in *Table 1*.

Table 1 – Required display settings

Monitor setting	Requirement	Notes
Screen resolution	1366 X 768	If you must choose a monitor with a higher native screen resolution, make sure that the monitor supports the 16:9 aspect ratio and lower resolutions including 1366 X 768.
Aspect ratio	16:9	
Refresh rate	60 Hz (recommendation)	A 60 Hz refresh rate is not optimal for all monitors. Make sure that you select a monitor that works well with a 60 Hz refresh rate.
Touchscreen	Minimum diagonal dimension: 469.9 mm (18.5 in) Single-touch touchscreen	
Connection	HDMI or DVI-D	See *Video cable* on page 35.

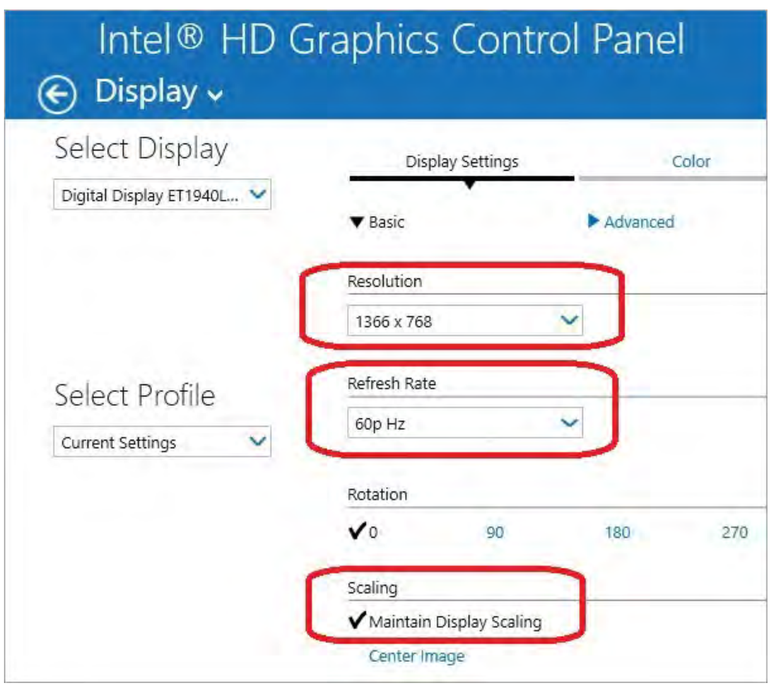
Phoenix and the Soft Op Con **must** be fully visible on the screen. A screen resolution of 1366 X 768 and an aspect ratio of 16:9 support this requirement.

Check the screen resolutions that the selected monitor supports

1. Connect a mouse to the CNC.
2. Close all the EDGE Connect software on the CNC.
   a. Click anywhere in the Main screen of Phoenix, and then press Alt+F4 to exit Phoenix.
   b. Click anywhere in the upper part of the Soft Op Con, and then press Alt+F4 to exit the upper part of the Soft Op Con.
   c. Click anywhere in the lower part of the Soft Op Con, and then press Alt+F4 to exit the lower part of the Soft Op Con.
3. Right-click on the Windows desktop, and then click **Graphics Properties**.
4. Click **Display**.
5. Compare the available options with the requirements in page 32.

Set the required display settings for the selected monitor

1. If the selected monitor can support the requirements, continue from *step 5* above.
2. On the Display screen, make the following selections:
   - **Resolution:** 1366 X 768
   - **Refresh rate:** 60 Hz
   - **Scaling:** Maintain Display Scaling



3. Click **Apply**.

4. After Windows applies the new settings, the message **Do you want to keep these settings?** appears. Click **Yes**.

5. Close the Display screen.

Troubleshoot common monitor display issues

Table 2

Display issue	Cause	Suggested Action
Phoenix and the Soft Op Con are not fully visible	The monitor has: ▪ A native resolution that is lower than 1366 X 768 **or** ▪ A 4:3 aspect ratio	Select a monitor that either: ▪ Has a native full-screen resolution of 1366 X 768 **or** ▪ Supports a higher resolution and a 16:9 aspect ratio, but also supports the Maintain Display Scaling option when a resolution of 1366 X 768 is selected
▪ Phoenix and the Soft Op Con do not fill the entire screen **or** ▪ Phoenix and the Soft Op Con are centered with black borders filling the outside portions of the screen	The monitor has a resolution that is higher than 1366 X 768	▪ Select the Maintain Display Scaling option in the monitor's display settings, if supported ▪ If the Maintain Display Scaling option is not supported at 60 Hz, try each of the other refresh rates to see if one of them supports Maintain Display Scaling ▪ If Maintain Display Scaling is not supported at any refresh rate, then select the Center Image option **or** ▪ Select a monitor that either has a native full-screen resolution of 1366 X 768 **or** supports a higher resolution and a 16:9 aspect ratio, but also supports the Maintain Display Scaling option when a resolution of 1366 X 768 is selected

To troubleshoot screen burn-in issues, see *The monitor displays a burn-in, ghost, or dim outline of a previously displayed image* on page 209.

See also *The warning message The display settings for this system are not at the optimal values appears* on page 209.

Video cable

Choose one of the following video cables to connect your CNC to the monitor. This cable is customer supplied.

HDMI cable	Maximum length: 5 m (16.4 ft)
DVI-D cable	Maximum length: 5 m (16.4 ft)

USB cable

This cable is customer supplied.

USB cable	Maximum length: 5 m (16.4 ft)

Ethernet cable

If you plan to use wireless, you do not need an Ethernet cable for your LAN connection.

This cable is customer supplied. Choose an Ethernet cable with the following specifications.

Specification	Cat5e, F/UTP type cable, braid and foil shielding minimum
Maximum length	61 m (200 ft)

EtherCAT cable

You need an internal and an external EtherCAT cable. Hypertherm sells EtherCAT cables. See *Parts* on page 333.

If you supply your own cables, choose EtherCAT cables that follow the Beckhoff® specification.

Type	Cat5e, 2-pair, 4-wire, double-shielded
Wire	Construction: Stranded tinned wire Diameter: 0.75 mm (7 X 0.25 mm), 22 AWG Insulation: Polyethylene, 1.5 mm (0.06 inch) diameter
Core	Construction: Filler as central element Layer 1: 4 wires, 2 pair Sequence of colors: White, yellow, blue, orange Layer 2: Plastic tape overlapped Inner jacket: Thermoplastic copolymer, 3.9 mm (0.04 inch) diameter Aluminum laminated foil overlapped Shield: Braided, tinned copper wires, 0.13 mm (0.005 inch) diameter, coverage about 85%, 4.7 mm (0.19 inch) diameter
Jacket	Material: Polyurethane Wall thickness: 0.9 mm (0.04 inch) Outer diameter: 6.5 mm (0.26 inch) ± 0.2 mm (0.008 inch)
Maximum length	61 m (200 ft)

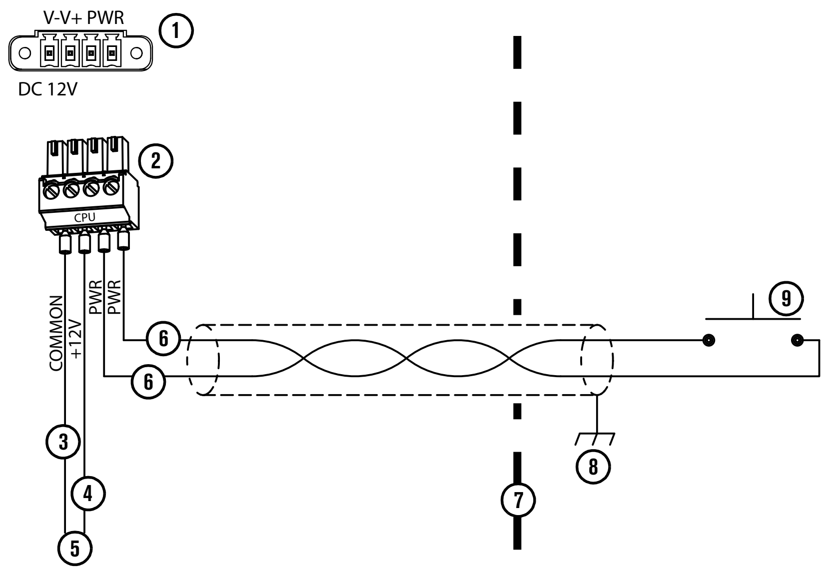
External power supply and power cord

Hypertherm's power supply is 5 A at 12 VDC or 60 W. It can be used to power the EDGE Connect and the EDGE Connect TC. If you buy your own external power supply and cord, choose a power supply with the following specifications.

Input voltage	100 VAC to 240 VAC at 50 Hz/60 Hz
Output voltage	12 VDC ± 5%
External power supply type	AC to DC switching
Minimum power supply capacity	36 W
External power supply to CNC connector	Switchcraft® 760K, barrel type Screw lock (use only locking connectors) 5.5 mm (0.22 inch) outer diameter 2.5 mm (0.1 inch) inner diameter 9.5 mm (0.37 inch) long

Alternatively, you can supply power to the EDGE Connect with the green terminal block. See *Figure 3*.

Figure 3



1	Connector on the EDGE Connect	**6**	Power
2	Terminal block (included)*	**7**	Metal enclosure wall
3	Common	**8**	Chassis ground
4	+12 V	**9**	Power button
5	Alternative power to the EDGE Connect		

* Terminal block can take 1.31 mm² – 0.08 mm² (16 AWG – 28 AWG) wires.

Power button and cable

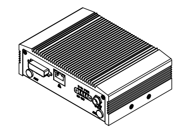
Connect the power button to the terminal block. See *Figure 3* on page 37.

Power button	Single-pole, single-throw (SPST), Normally Open, momentary switch with gold-plated contacts
Cable	Maximum length: 5 m (16 ft) Minimum 0.33 mm² (22 AWG) Shielded twisted pair

EtherCAT drives

Install all the drives before you install your CNC. For a list of supported EtherCAT drives and I/O modules, see the *EtherCAT Devices Supported by EDGE Connect CNCs* Application Note (809660). Technical documentation is available at www.hypertherm.com/docs.

EDGE Connect installation overview



These are the general steps to install the EDGE Connect. See *Figure 2* on page 31 for an example of how to install a system. For more details, see *Wiring Diagrams* on page 339.

1. Mount the EDGE Connect inside of an enclosure.
   - For enclosure requirements see *Enclosure* on page 32.
   - For mounting dimensions see *Mount the EDGE Connect* on page 39.
2. Install the wireless extensions and wireless antennas.

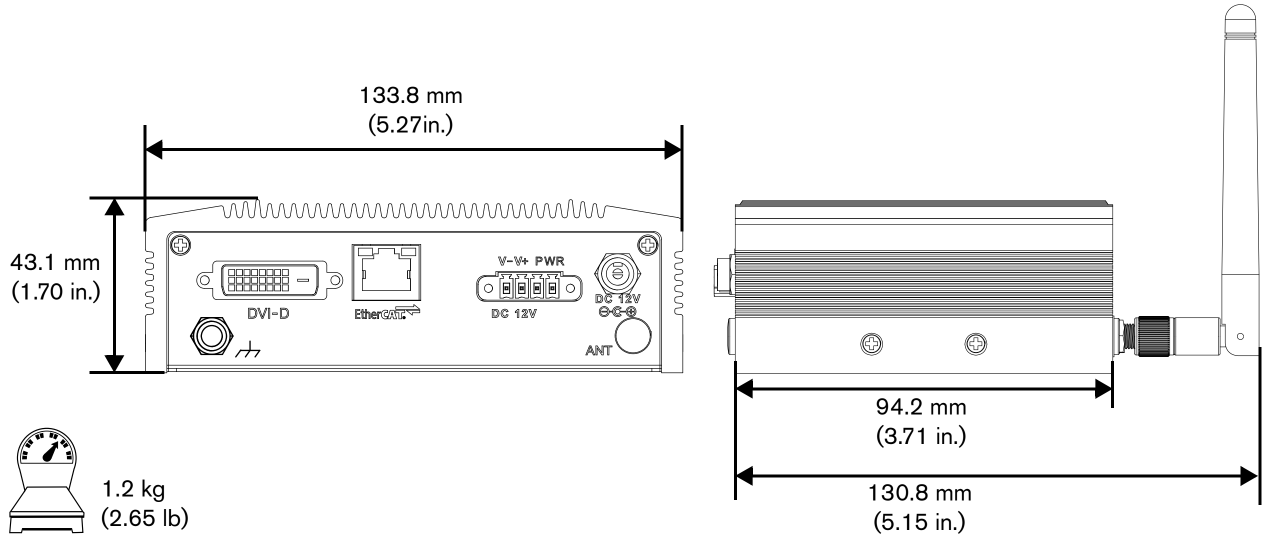
   Use an RP-SMA female to RP-SMA male extension cable so that the antennas can be on the exterior of the enclosure.

3. Ground the EDGE Connect. See *Ground the EDGE Connect* on page 40.
4. Connect a monitor to the EDGE Connect.
   - For monitor requirements see *Monitor* on page 32.
   - For cable requirements see *Video cable* on page 35 and *USB cable* on page 35.

   Turn off the CNC when not in use to prevent retained charge in the LCD crystals.

5. Install the bulkhead connectors into the enclosure for the LAN Ethernet and EtherCAT cables. The bulkhead connectors **must** be bonded to the enclosure to prevent noise.
6. Connect the LAN Ethernet cable and EtherCAT cable to the EDGE Connect.
   - For LAN Ethernet cable requirements see *Ethernet cable* on page 35.
   - For EtherCAT cable requirements see *EtherCAT cable* on page 36.
7. Connect a power button to the EDGE Connect.
   - For power button and cable requirements see *Power button and cable* on page 37.
8. Connect the power to the EDGE Connect.
   - Use the external power supply. See *External power supply and power cord* on page 36.
   - If necessary, use the pluggable terminal block. See *Figure 3* on page 37.
9. Connect a USB keyboard and USB mouse to the CNC to make software setup easier. This step is optional but strongly recommended.
10. Push the power button.

EDGE Connect dimensions and weight

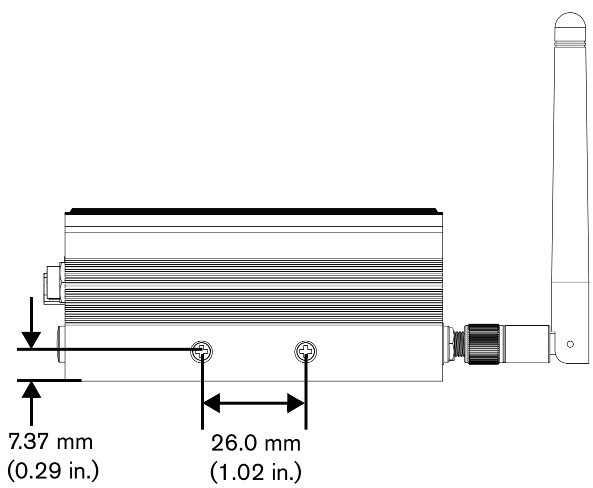


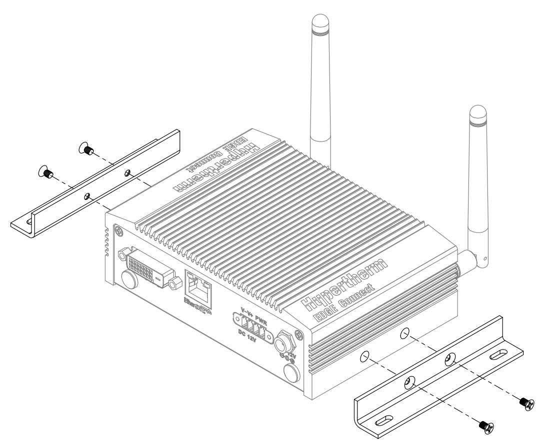
Mount the EDGE Connect

The EDGE Connect must be mounted inside an enclosure to protect it from the environment.

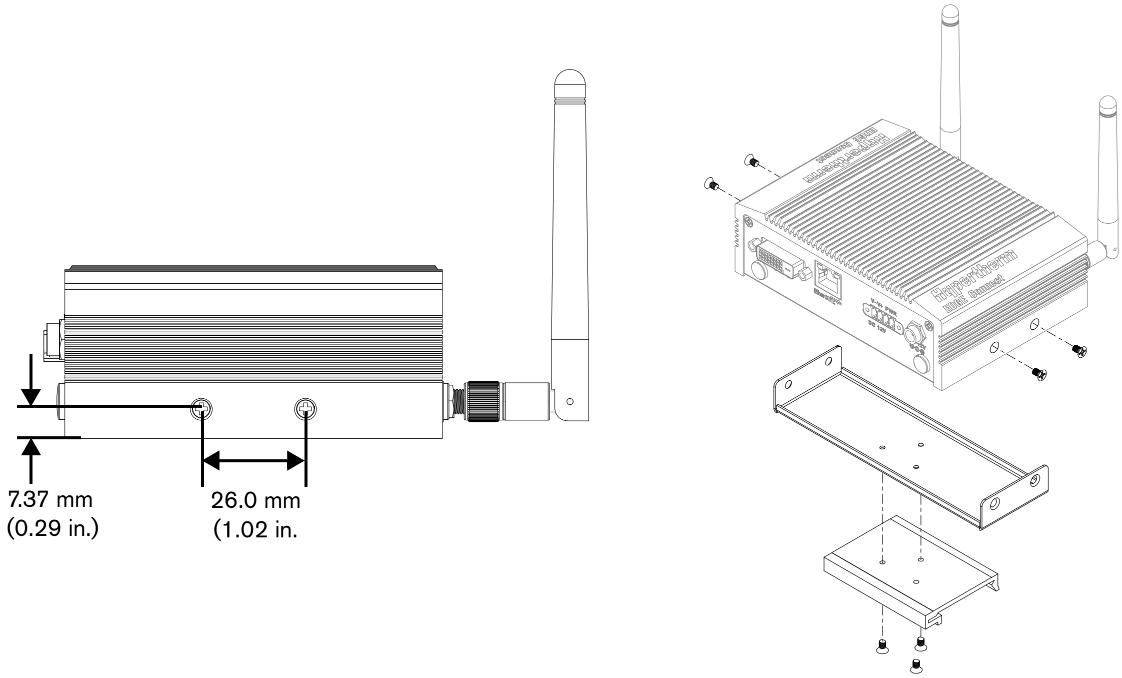
The EDGE Connect can be mounted with wall brackets, a DIN rail bracket, or a VESA bracket. Hypertherm sells wall brackets and a DIN rail bracket. See *Mounting options* on page 336.

EDGE Connect wall mount





EDGE Connect DIN rail mount

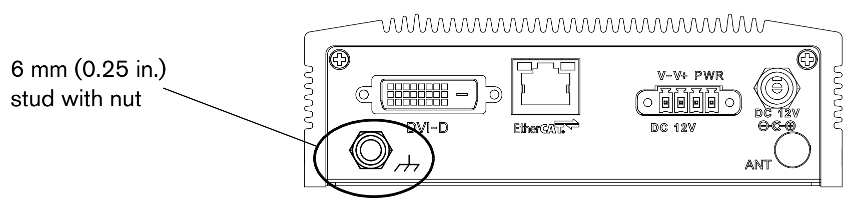


Ground the EDGE Connect

See *Recommended grounding and shielding* on page 55 for more information.

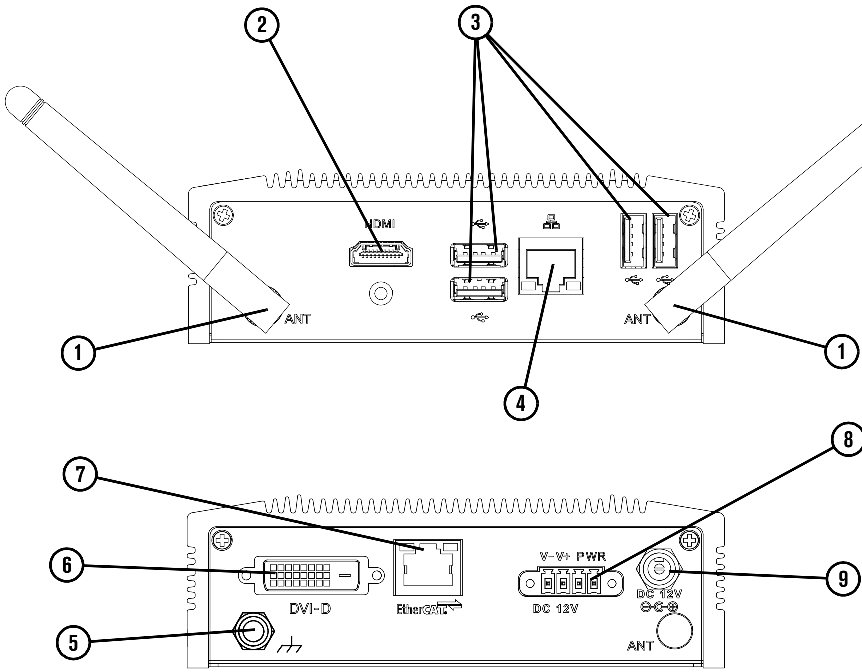
The ground wire is customer supplied. The minimum ground wire size is 1.3 mm² (16 AWG).

1. Remove the nut.



2. Install the ground terminal.
3. Install the nut.
4. Connect the ground wire to the metal enclosure.

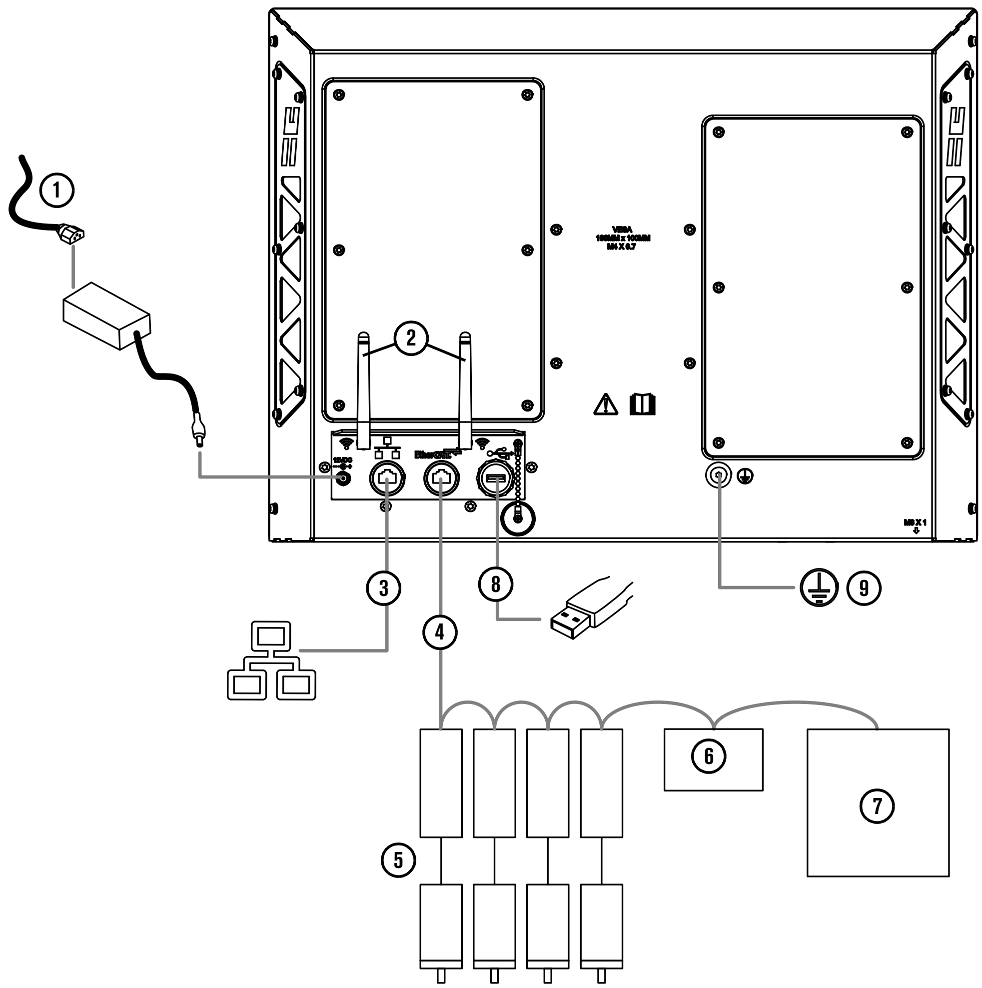
EDGE Connect connector locations



1	Wireless antenna connector	RP-SMA connector type
2	HDMI connector	Supports 1366 X 768 at 60 Hz
3	USB connector	4 hi-speed USB
4	Female RJ-45 connector	For LAN, 10/100/1000 Mbps
5	Chassis ground	
6	Female DVI-D connector	Supports 1366 X 768 at 60 Hz
7	Female RJ-45 connector	For EtherCAT, 10/100/1000 Mbps
8	Terminal block connector	For power-in, if the barrel connector is not used, and for a momentary switch
9	Female barrel connector	2.5 mm (0.1 inch) X 5.5 mm (0.22 inch),12 VDC

EDGE Connect TC (model number 090198)

Figure 4 – EDGE Connect TC system diagram



1 External power supply and input power cord

2 Wireless antennas

3 LAN Ethernet cable

4 EtherCAT cable*

5 Drives and motors**

6 I/O module

7 Plasma power supply*

8 USB device***

9 EMI chassis ground

* If you want to connect a Powermax or MAXPRO to an EDGE Connect CNC, use a discrete connection through the drive I/O or an I/O module on the network.

** Make sure that you can remove power from the drives separately from the CNC so that you can troubleshoot network faults or errors if necessary. Refer to the drive manufacturer's documentation for instructions on drive installation.

*** If you use a USB hub, use only 1 USB hub.

Before you begin

Ethernet cable

If you plan to use wireless, you do not need an Ethernet cable for your LAN connection.

This cable is customer supplied. Choose an Ethernet cable with the following specifications.

Specification	Cat5e, F/UTP type cable, braid and foil shielding minimum
Maximum length	61 m (200 ft)

EtherCAT cable

Hypertherm sells EtherCAT cables, see *External EtherCAT cables* on page 337.

If you supply your own cables, choose EtherCAT cables that follow the Beckhoff® specification.

Type	Cat5e, 2-pair, 4-wire, double-shielded
Wire	Construction: Stranded tinned wire Diameter: 0.75 mm (7 X 0.25 mm), 22 AWG Insulation: Polyethylene, 1.5 mm (0.06 inch) diameter
Core	Construction: Filler as central element Layer 1: 4 wires, 2 pair Sequence of colors: White, yellow, blue, orange Layer 2: Plastic tape overlapped Inner jacket: Thermoplastic copolymer, 3.9 mm (0.04 inch) diameter Aluminum laminated foil overlapped Shield: Braided, tinned copper wires, 0.13 mm (0.005 inch) diameter, coverage about 85%, 4.7 mm (0.19 inch) diameter
Jacket	Material: Polyurethane Wall thickness: 0.9 mm (0.04 inch) Outer diameter: 6.5 mm (0.26 inch) ± 0.2 mm (0.008 inch)
Maximum length	61 m (200 ft)

External power supply and power cord

Hypertherm's power supply is 5 A at 12 VDC or 60 W. It can be used to power the EDGE Connect and the EDGE Connect TC. If you choose to buy your own external power supply and cord, use the following specifications.

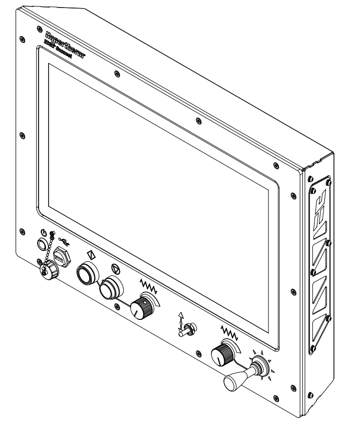
Input voltage	100 VAC to 240 VAC at 50 Hz/60 Hz
Output voltage	12 VDC ± 5%
External power supply type	AC to DC switching
Minimum power supply capacity	60 W
External power supply to CNC connector	Switchcraft® 760K, barrel type Screw lock (use only locking connectors) 5.5 mm (0.22 inch) outer diameter 2.5 mm (0.1 inch) inner diameter 9.5 mm (0.37 inch) long

EtherCAT drives

Install all the drives before you install your CNC. For a list of supported EtherCAT drives and I/O modules, see the *EtherCAT Devices Supported by EDGE Connect CNCs* Application Note (809660). Technical documentation is available at www.hypertherm.com/docs.

EDGE Connect TC installation overview

These are the general steps to install the EDGE Connect TC. See *Figure 4* on page 42 for an example of how to install this system. For more details, see *Wiring Diagrams* on page 339.



1. Mount the EDGE Connect TC.
   - For mounting dimensions see *Mount the EDGE Connect TC* on page 48.
2. Install the wireless extensions, if necessary, and wireless antennas.
3. Ground the EDGE Connect TC. See *Ground the EDGE Connect TC* on page 53.
4. Connect the LAN Ethernet cable and EtherCAT cable to the EDGE Connect TC.
   - For LAN Ethernet cable requirements see *Ethernet cable* on page 43.
   - For EtherCAT cable requirements see *EtherCAT cable* on page 43.
5. Connect the power to the EDGE Connect TC. For power supply requirements, see *External power supply and power cord* on page 44.
6. Connect a USB keyboard and USB mouse to the CNC to make software setup easier. This step is optional but strongly recommended.
7. Push the power button.

   Turn off the CNC when not in use to prevent retained charge in the LCD crystals.

EDGE Connect TC touchscreen specifications

Model number	Size of screen	Specifications
090198	495.3 mm (19.5 inches)	▪ Projected capacitive touch (PCAP) ▪ 16:9 aspect ratio ▪ 1366 X 768 resolution
090185	469.9 mm (18.5 inches)	

For more information about touchscreen resolution, see *Monitor* on page 32.

EDGE Connect TC dimensions and weight

Figure 5 – Model number 090198 (495.3 mm/19.5 in. touchscreen)

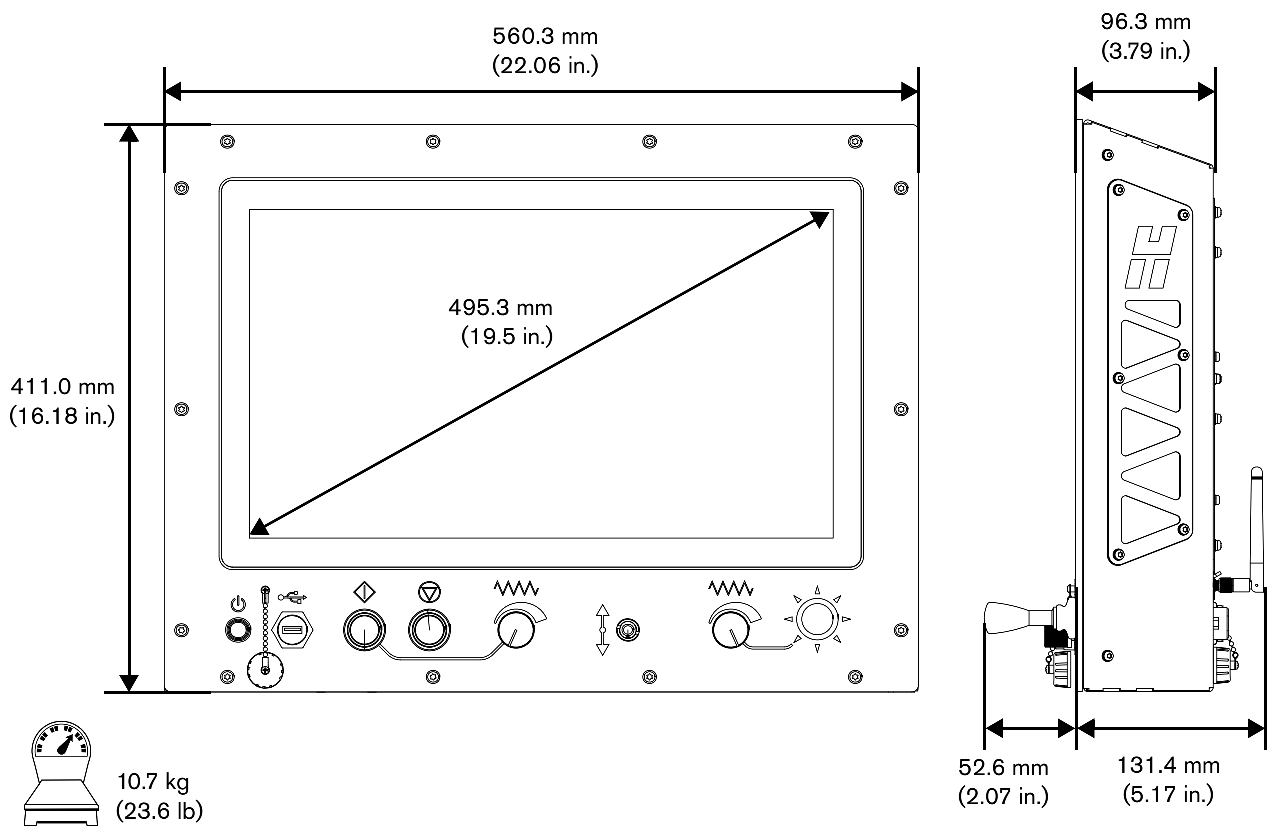
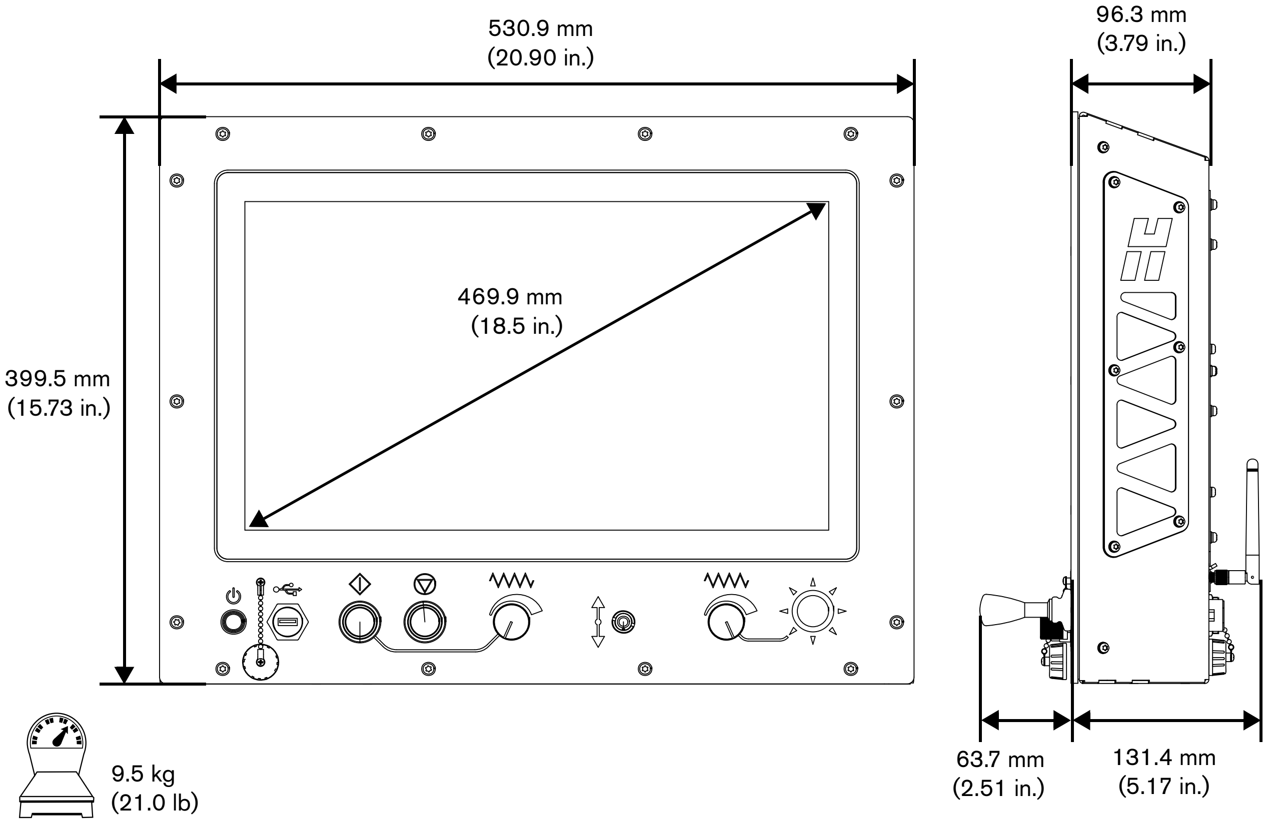


Figure 6 – Model number 090185 (469.9 mm/18.5 in. touchscreen)

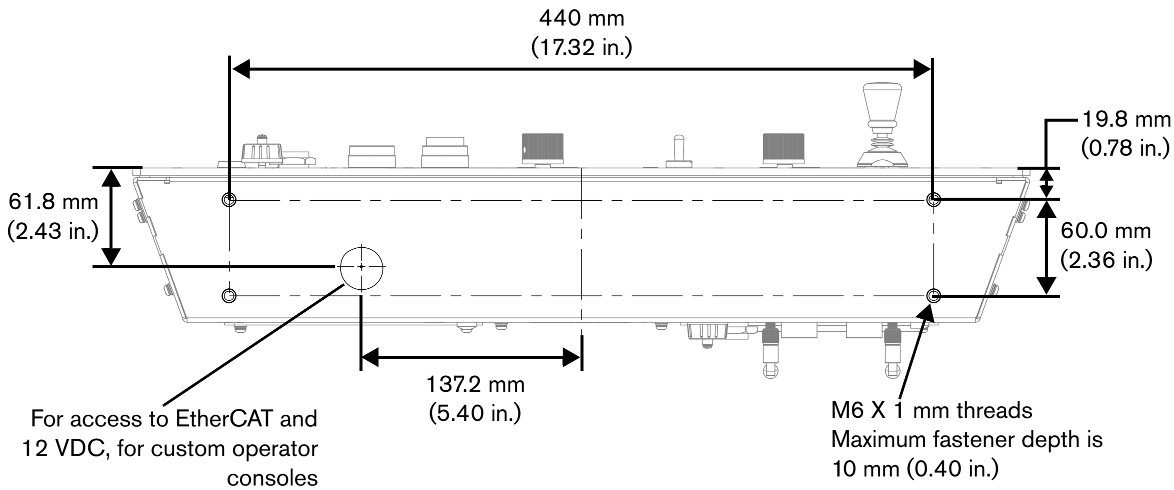


Mount the EDGE Connect TC

You must mount the CNC.

The EDGE Connect TC can be mounted from the bottom on a flat surface, with a VESA bracket, or with flush-mount brackets. Hypertherm sells mounting brackets. See *Mounting options* on page 336.

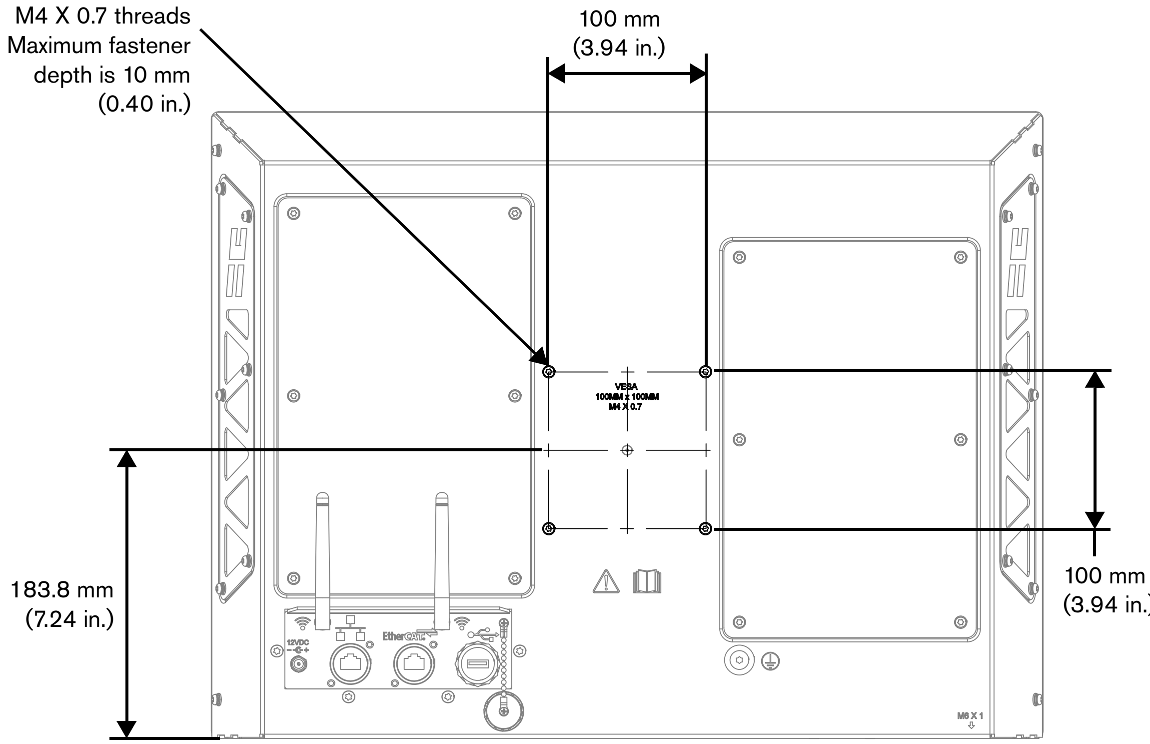
EDGE Connect TC bottom mount



The mounting pattern and dimensions are the same for model number 090198 (495.3 mm/19.5 in. touchscreen) and 090185 (469.9 mm/18.5 in. touchscreen).

EDGE Connect TC VESA mount

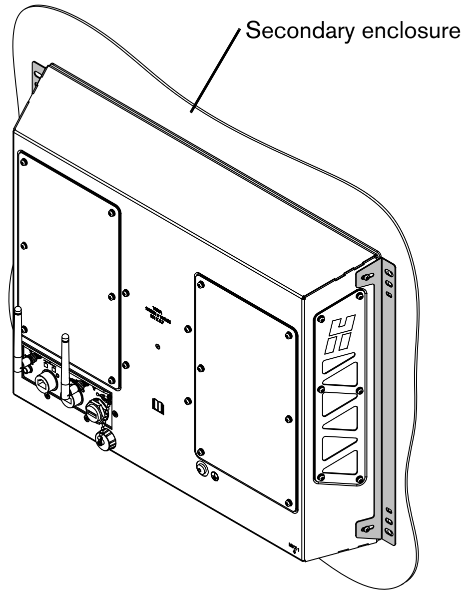
Hypertherm does not sell a VESA bracket. Use the dimensions below to buy a compatible bracket.



The mounting pattern and dimensions are the same for model number 090198 (495.3 mm/19.5 in. touchscreen) and 090185 (469.9 mm/18.5 in. touchscreen).

EDGE Connect TC flush-mount

Hypertherm sells a flush-mount bracket with screws. See *Mounting options* on page 336.



The maximum panel thickness is 10 mm (0.4 inch).

1. Select M4, M5, or M6 screws.
2. Use a CNC to cut the rectangle hole and screw holes. See *Figure 8* on page 52 for dimensions.

The rectangle hole dimensions allow 0.51 mm (0.02 inch) nominal clearance all around. You can adjust this clearance.

3. Use the brackets and screws to mount the EDGE Connect TC in the panel.

Figure 7 – Dimensions for model number 090198 (495.3 mm/19.5 in. touchscreen)

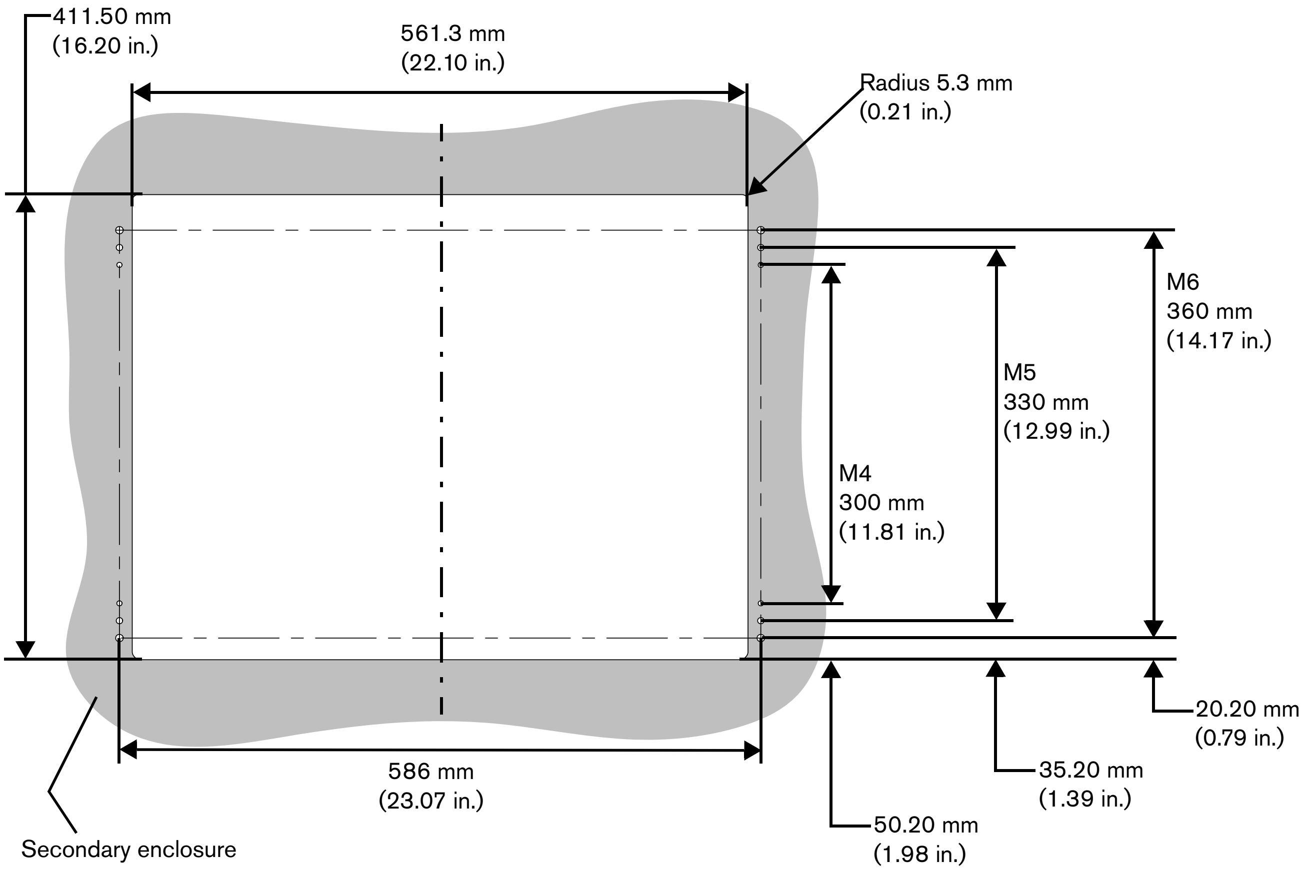
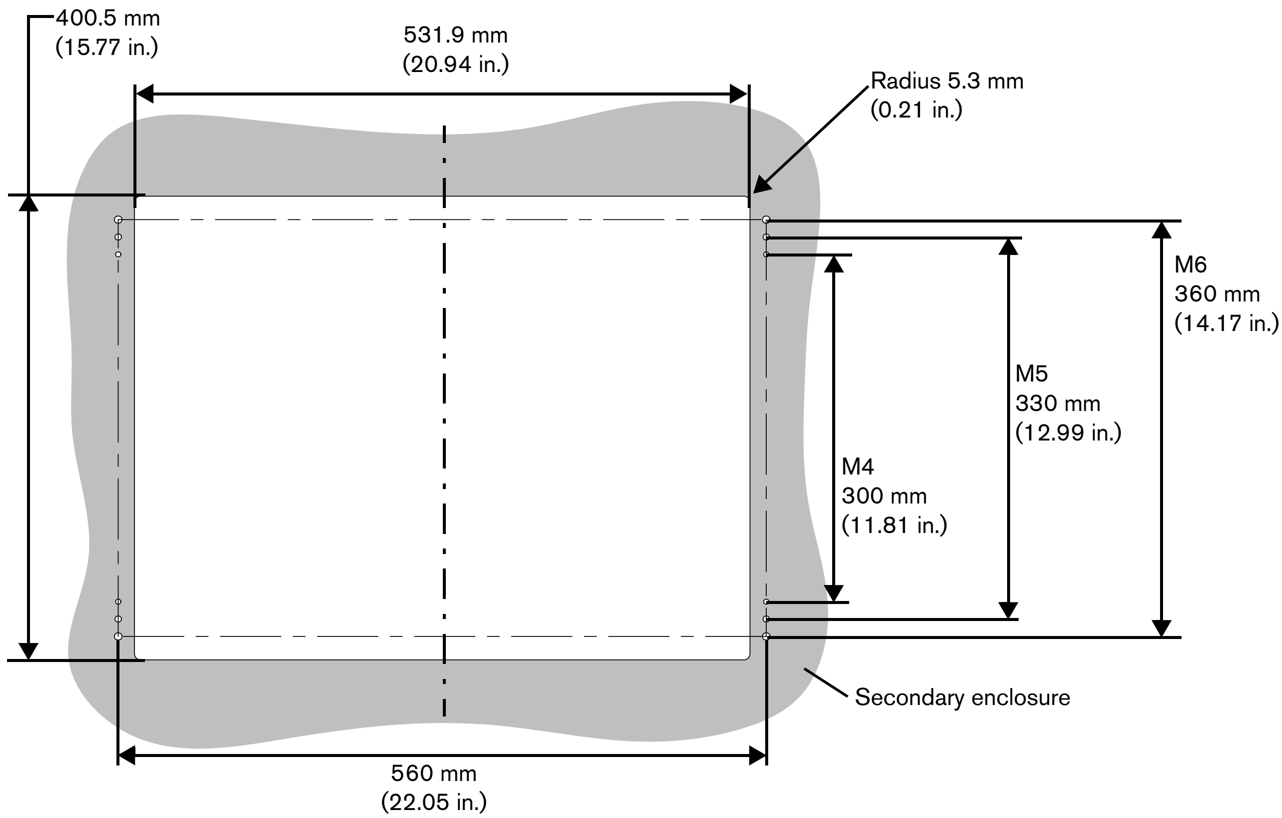


Figure 8 – Dimensions for model number 090185 (469.9 mm/18.5 in. touchscreen)

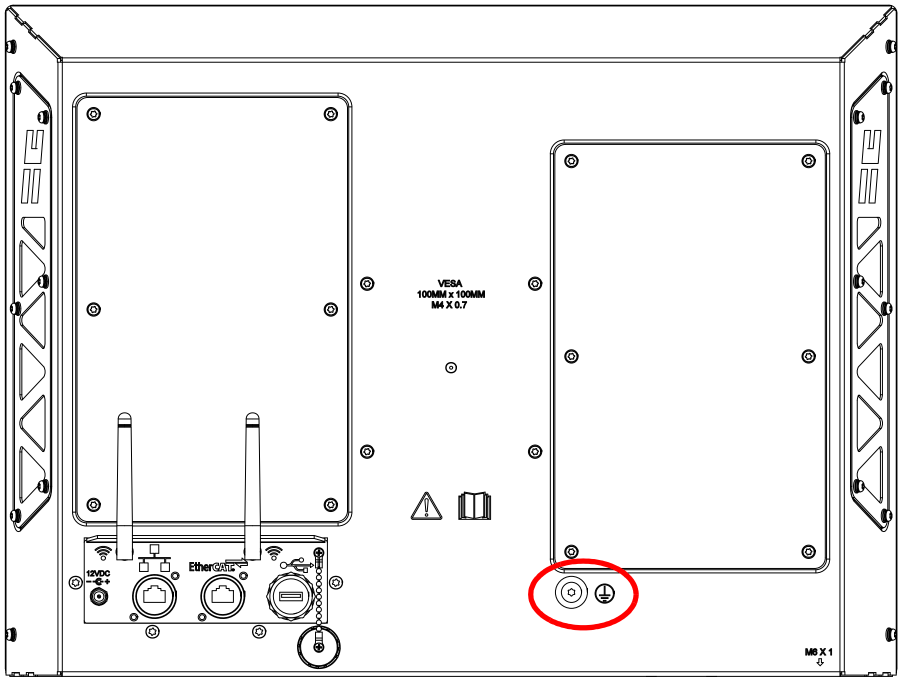


Ground the EDGE Connect TC

See *Recommended grounding and shielding* on page 55 for more information.

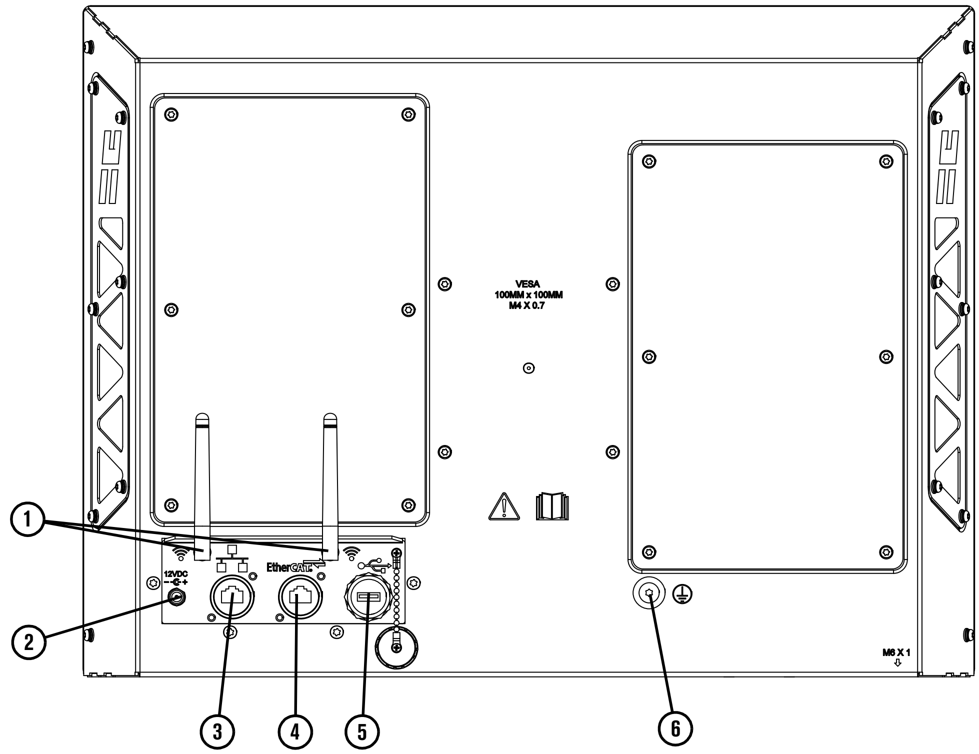
The ground wire is customer supplied. The minimum ground wire size is 1.3 mm^2 (16 AWG).

1. Remove the screw from the rear of the CNC.



2. Put the ground wire terminal on the screw.
3. Tighten the screw.
4. Connect the ground wire to the EMI chassis ground.

EDGE Connect TC rear I/O panel connector locations



1	Wireless antenna connector	RP-SMA connector type
2	Female power-in connector	2.5 mm X 5.5 mm,12 VDC
3	Female RJ-45 connector	For LAN, 10/100/1000 Mbps
4	Female RJ-45 connector	For EtherCAT, 10/100/1000 Mbps
5	USB connector	Hi-speed USB
6	EMI chassis ground	M6 thread insert

Recommended grounding and shielding

Introduction

This section describes practices for grounding and shielding a plasma cutting system to minimize its susceptibility to electromagnetic interference (EMI) (also known as **noise**). It also describes the service ground, protective earth (PE) ground, and DC power ground. The diagram at the end of this section shows these types of grounds in a plasma cutting system.

The grounding practices in this section have been used on many installations with excellent results, and Hypertherm recommends that these practices be a routine part of the installation process. The actual methods used to implement these practices may vary from system to system, but should remain as consistent as possible. However, due to the variation in equipment and installations, these grounding practices may not succeed in every case to eliminate EMI problems. Hypertherm recommends that you consult your local and national electrical codes to make sure that the grounding and shielding practices that you use satisfy the requirements for your location.

Types of grounding

Service ground (also known as safety ground) is the grounding system that applies to the incoming line voltage. It prevents a shock hazard to any personnel from any of the equipment or the cutting table. It includes the service ground coming into the plasma system and other systems, such as the CNC and the motor drives, as well as the supplemental ground rod connected to the cutting table. In the plasma circuits, the ground is carried from the plasma system chassis to the chassis of each separate console through the interconnecting cables.

Protective earth (PE) ground is the grounding system inside the electrical equipment. The PE ground, which connects to the service ground, provides electrical continuity between the equipment and the AC service.

DC power ground (also known as cutting current ground or work) is the grounding system that completes the path of the cutting current from the torch back to the plasma system. It requires that the positive lead from the plasma system be firmly connected to the cutting table ground bus with a properly sized cable. It also requires that the slats, on which the workpiece rests, make firm contact with the table and the workpiece.

EMI grounding and shielding is the grounding system that limits the amount of EMI emitted by the plasma and motor drive systems. It also limits the amount of EMI that is received by the CNC and other control and measurement circuits. The grounding practices described in this section mainly target EMI grounding and shielding.

Grounding practices

1. Unless noted, for HPR or MAXPRO200 cutting systems, use cables with a minimum gauge of 13.3 mm^2 (6 AWG) (047040) for the EMI ground cables shown in the *Example grounding diagram with an HPR or MAXPRO200 cutting system* on page 60. Unless noted, for XPR cutting systems, use cables with a minimum gauge of 21.2 mm^2 (4 AWG) (047031) for the EMI ground cables shown in the *Example grounding diagram with an XPR cutting system* on page 61.

2. The cutting table is used for the common, or star, EMI ground point and should have threaded studs welded to the table with a copper bus bar mounted on them. A separate bus bar should be mounted on the gantry as close to each motor as possible. If there are motors at each end of the gantry, run a separate EMI ground cable from the far motor to the gantry bus bar. The gantry bus bar should have a separate, heavy EMI ground cable 21.2 mm^2 (4 AWG; 047031) to the table bus bar. The EMI ground cables for the torch lifter and the RHF or combined ignition/gas connect console must each run separately to the table ground bus.

3. Inadequate grounding not only exposes operators to dangerous voltages, but inadequate grounding also increases the risk of equipment failure and unnecessary downtime. Ideally a ground should be zero ohms resistance, but field experience indicates under 1 ohm resistance is satisfactory for most applications. Hypertherm recommends that you consult your local and national electrical codes to make sure that the grounding and shielding practices that you use satisfy the requirements for your location.

4. A ground rod (a PE ground) that meets all applicable local and national electric codes must be installed within 6 m (20 ft) of the cutting table. For HPR or MAXPRO200 cutting systems, the PE ground must be connected to the cutting table ground bus bar using a minimum 13.3 mm^2 (6 AWG) green and yellow grounding cable (047121). For XPR cutting systems, the PE ground must be connected to the cutting table ground bus bar using a minimum 21.2 mm^2 (4 AWG) grounding cable (047031). Consult an electrician in your location to make sure that your grounding meets all local and national electric codes.

5. For the most effective shielding, use the Hypertherm CNC interface cables for I/O signals, serial communication signals, between plasma systems in multi-drop connections, and for interconnections between all parts of the Hypertherm system.

6. All hardware used in the ground system must be brass or copper. While you can use steel studs welded to the cutting table for mounting the ground bus, no other aluminum or steel hardware can be used in the ground system.

7. AC power, PE, and service grounds must be connected to all equipment according to local and national codes.

8. For a system with a remote high frequency (RHF) console or combined ignition/gas connect console, the positive, negative, and pilot arc leads should be bundled together for as long a distance as possible. The torch lead, work lead, and the pilot arc (nozzle) leads may be run parallel to other wires or cables only if they are separated by at least 150 mm (6 inches). If possible, run power and signal cables in separate cable tracks.

9. For a system with a RHF console or combined ignition/gas connect console, Hypertherm recommends that you mount this console as close as possible to the torch. This console also must have a separate ground cable that connects directly to the cutting table ground bus bar.

10. Each Hypertherm component, as well as any other CNC or motor drive cabinet or enclosure, must have a separate ground cable to the common (star) ground on the table. This includes the ignition/gas connect console, whether it is bolted to the plasma system or to the cutting table.

11. For HPR or MAXPRO200 cutting systems, the metal braided shield on the torch lead must be connected firmly to the ignition/gas connect console and to the torch. It is recommended to be electrically insulated from any metal and from any contact with the floor or building. The torch lead can be run in a plastic cable tray or track, or covered with a plastic or leather sheath.

12. For XPR cutting systems, the coupler on the pilot arc and coolant hose set assembly must be connected firmly to the gas connect console and torch connect console collars. Make sure to tighten the clamp. The collar on the torch lead must be connected firmly to the torch sleeve. Make sure to tighten the clamp. Connect a ground lead (10 AWG) to the flat terminal on the torch mounting sleeve.

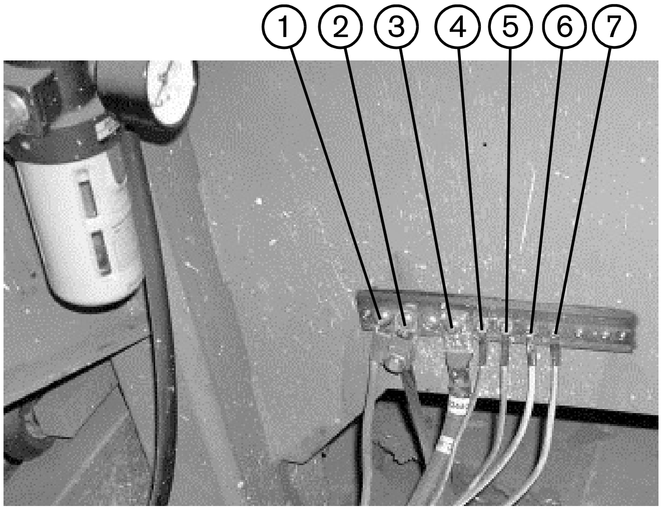
13. The torch holder and the torch breakaway mechanism – the part mounted to the lifter, not the part mounted to the torch – must be connected to the stationary part of the lifter with copper braid at least 12.7 mm (0.5 inches) wide. A separate cable must run from the lifter to the gantry ground bus bar. The valve assembly should also have a separate ground connection to the gantry ground bus bar.

14. If the gantry runs on rails that are not welded to the table, then each rail must be connected with a ground cable from the end of the rail to the table. The rail ground cables connect directly to the table and do not need to connect to the table ground bus bar.

15. If you are installing a voltage divider board, mount it as closely as possible to where the arc voltage is sampled. One recommended location is inside the plasma system enclosure. If a Hypertherm voltage divider board is used, the output signal is isolated from all other circuits. The processed signal should be run in twisted shielded cable (Belden 1800F or equivalent). Use a cable with a braided shield, not a foil shield. Connect the shield to the chassis of the plasma system and leave it unconnected at the other end.

16. All other signals (analog, digital, serial, and encoder) should run in twisted pairs inside a shielded cable. Connectors on these cables should have a metal housing. The shield, not the drain, should be connected to the metal housing of the connector at each end of the cable. Never run the shield or the drain through the connector on any of the pins.

The following picture shows an example of a cutting table ground bus with an HPR or MAXPRO200 cutting system. The components shown here may differ from your system.



1 Gantry ground bus

2 Ground rod

3 Plasma system lead (+)

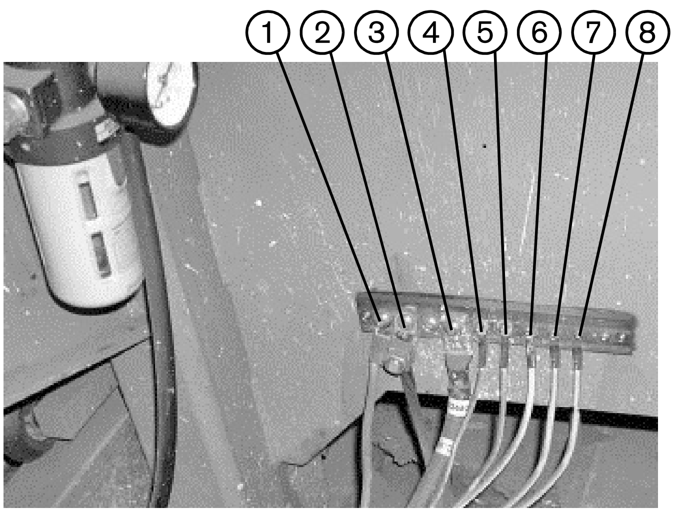
4 RHF console (if applicable, not on all systems)

5 CNC enclosure

6 Torch holder

7 Plasma system chassis

The following picture shows an example of a cutting table ground bus with an XPR cutting system. The components shown here may differ from your system.



1 Gantry ground bus

2 Ground rod

3 Plasma system lead (+)

4 Gas connect console

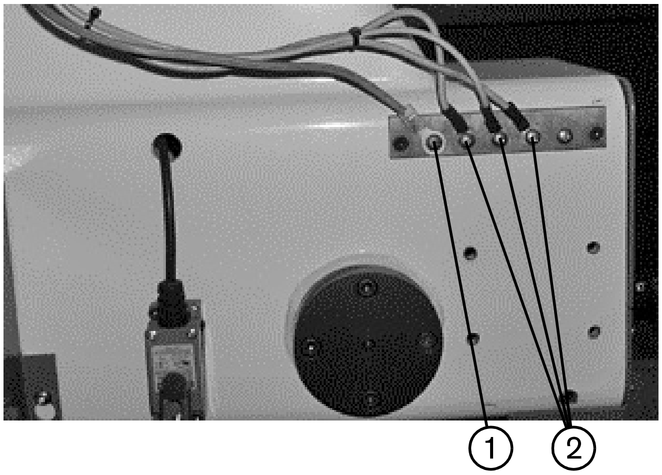
5 CNC enclosure

6 Torch holder

7 Plasma system chassis

8 Torch connect console

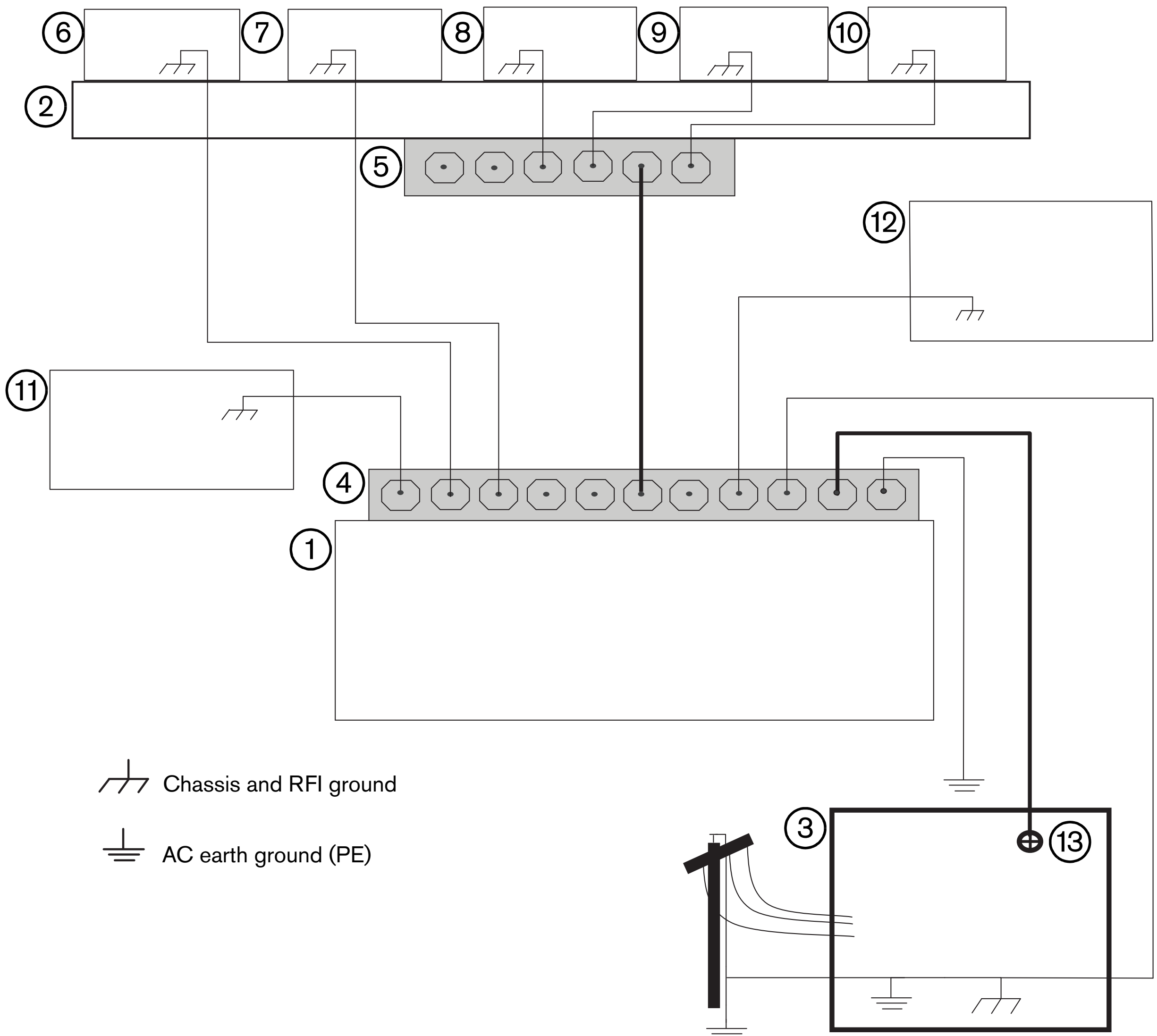
The following picture shows an example of a gantry ground bus. It is bolted to the gantry, close to the motor. All of the individual ground cables from the components mounted on the gantry connect to the bus. A single heavy cable then connects the gantry ground bus to the table ground bus.



1 Cable to the cutting table ground bus

2 Ground cables from components on the gantry

Example grounding diagram with an HPR or MAXPRO200 cutting system



1 Cutting table

2 Gantry

3 Plasma system

4 Table ground bus bar

5 Gantry ground bus bar

6 Torch height control lifter (ArcGlide, Sensor THC, Sensor PHC, or other)

7 RHF console (not on all systems). Connect to table ground bus bar.

8, 9 System-specific component such as metering console, gas console, or selection console

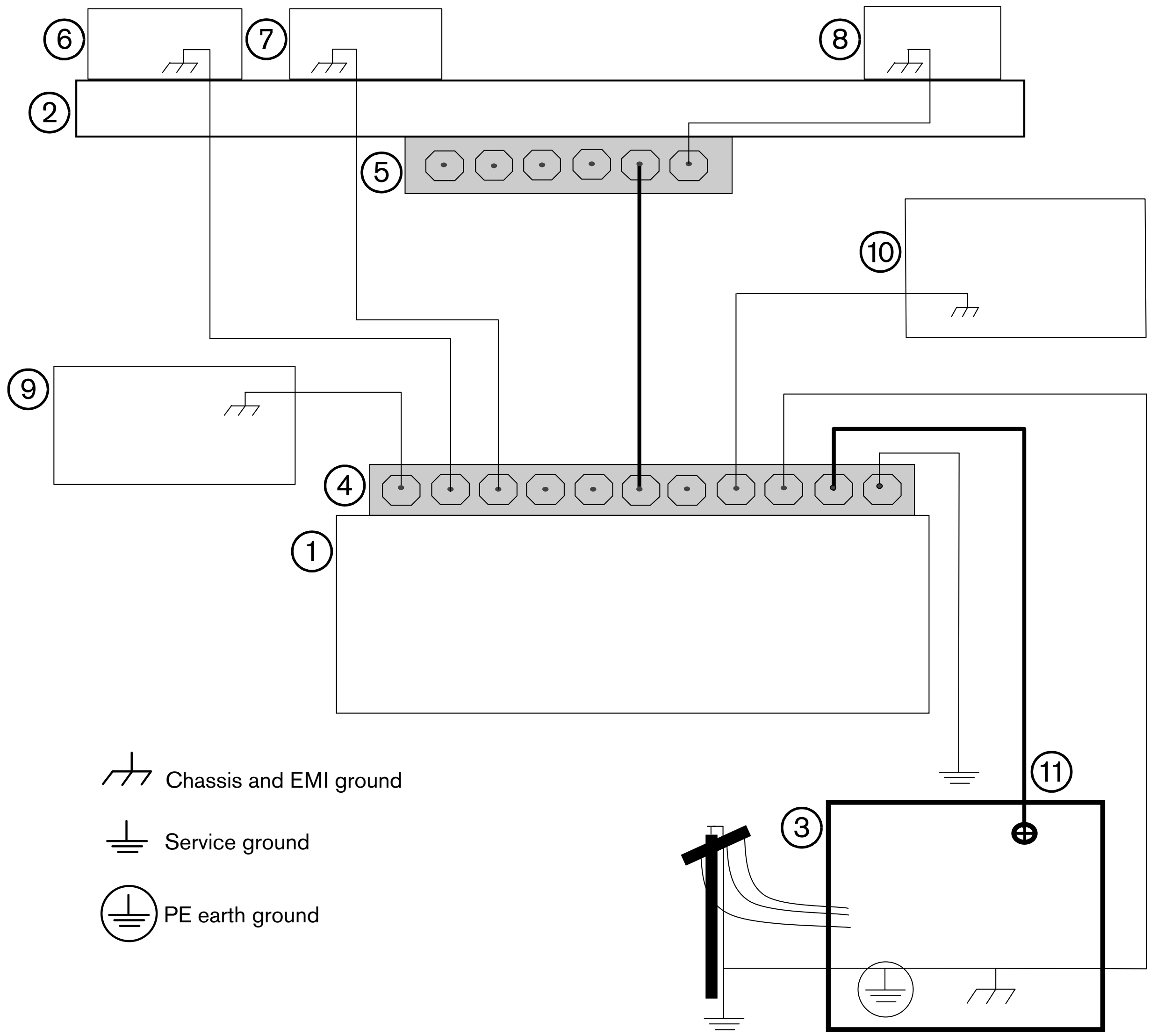
10 CNC chassis

11 Torch height control module (ArcGlide, Command THC)

12 System-specific component such as a cooler or chiller

13 DC power ground

Example grounding diagram with an XPR cutting system



1 Cutting table
2 Gantry
3 Plasma system
4 Table ground bus bar
5 Gantry ground bus bar
6 Torch height control lifter
7 Torch connect console
8 CNC controller
9 Torch height control module
10 Gas connect console. Connect to table ground bus bar.*
11 DC power ground (work)

* The ignition console is integrated into the gas connect console for XPR cutting systems.

This example is based on practices in North America. Other regions can have different local or national electrical codes. Hypertherm recommends that you consult your local and national electrical codes to make sure that the grounding and shielding practices that you use satisfy the requirements for your location.

Symbols and marks

Your product may have one or more of the following marks on or near the data plate. Because of 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 comply with European Directives. Applicable directives may include the European Low Voltage Directive, the European Electromagnetic Compatibility (EMC) Directive, the Radio Equipment Directive (RED), and the Restriction of Hazardous Substances (RoHS) Directive. See the European CE Declaration of Conformity for details.

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.

RCM mark

CE versions of products with a RCM mark comply with the EMC and safety 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.

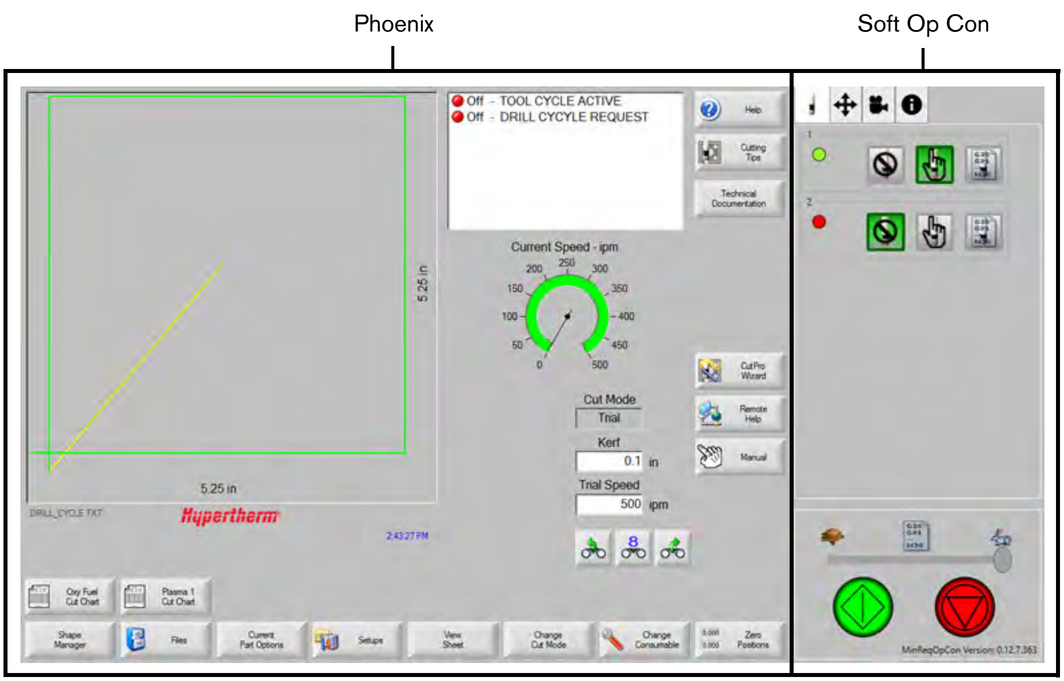
RoHS mark

The RoHS mark indicates that the product meets the requirements of the European Restriction of Hazardous Substances (RoHS) Directive.

2 Operate

When you turn on the CNC, the Phoenix software and the software operator console (Soft Op Con) open. Before using Phoenix, you must click **OK** on a dialog window to agree to Hypertherm's Terms of Use.

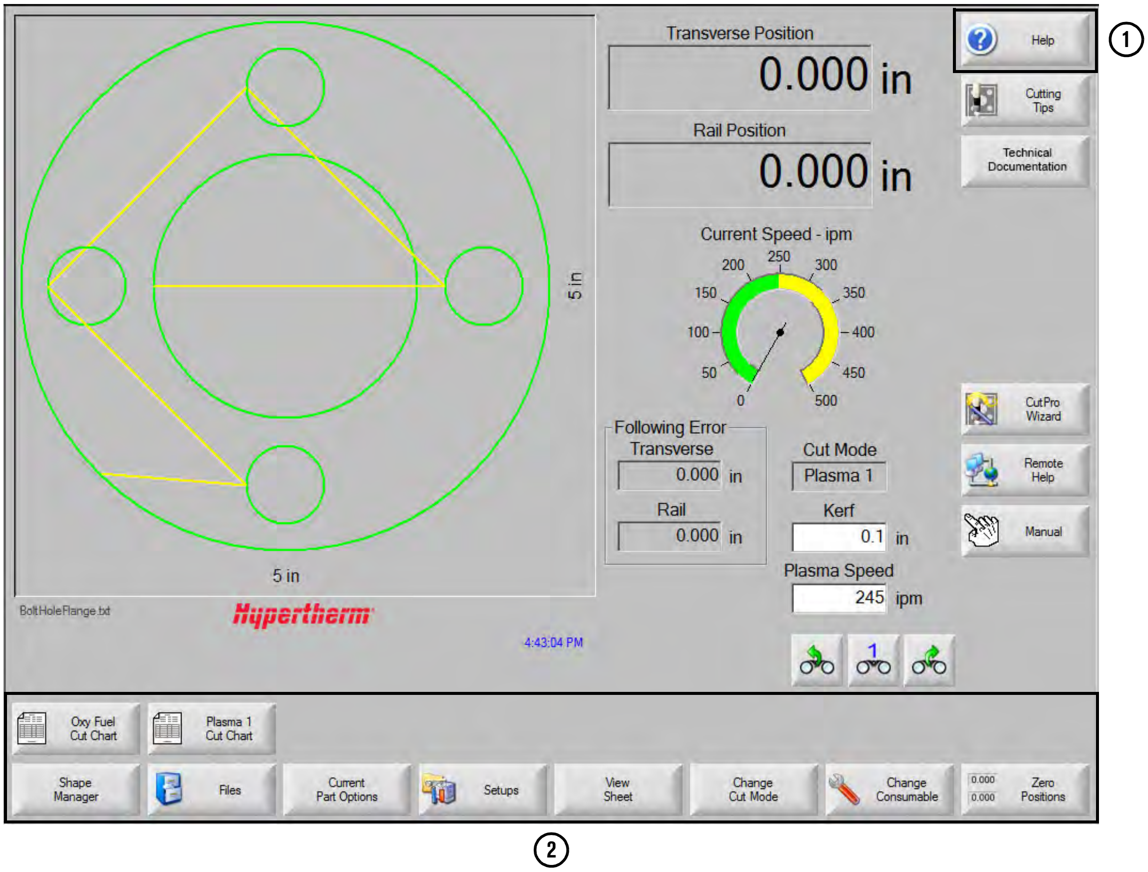
Figure 9



Phoenix is the core software interface for operating the CNC, and is common across all Hypertherm CNCs. The Soft Op Con is a fully integrated part of Phoenix. The standard Hypertherm Soft Op Con, shown in *Figure 9*, comes with the EDGE Connect and EDGE Connect TC CNCs.

Cutting system manufacturers can create their own custom software operator consoles using the Phoenix application programmer interface (API). If your cutting system has a custom software operator console, contact your cutting system manufacturer or see their documentation for more information. See also *Automatically open a custom software operator console* on page 70.

Operate Phoenix



In addition to the information in this manual, the Phoenix Help ① provides details on how to operate the Phoenix software. Phoenix Help is available for each screen in Phoenix. For more information, see *Phoenix Help and technical documentation* on page 73.

The buttons ② in Phoenix screens are called *soft keys*.

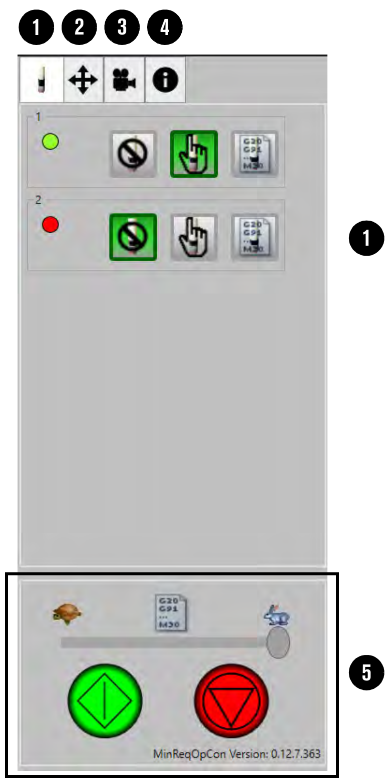
Some soft keys go to particular screens, and some soft keys do a particular action. The soft keys that show on a screen change based on installed components, enabled features, Phoenix setup options, passwords, and user levels (beginner, intermediate, or advanced).

This manual assumes the CNC is in advanced user level mode and shows all features.

Operate the standard Hypertherm Soft Op Con

The Soft Op Con can replace a hardware operator console. With an EDGE Connect TC, the Soft Op Con and the hardware operator console can be used together. For example, pressing Start in the Soft Op Con is the same as pressing the Start button on the hardware operator console.

Cutting system manufacturers can create their own custom software operator consoles using the Phoenix application programmer interface (API). If your cutting system has a custom software operator console, contact your cutting system manufacturer or see their documentation for more information.



1 Station control tab (shown)

2 Manual motion tab

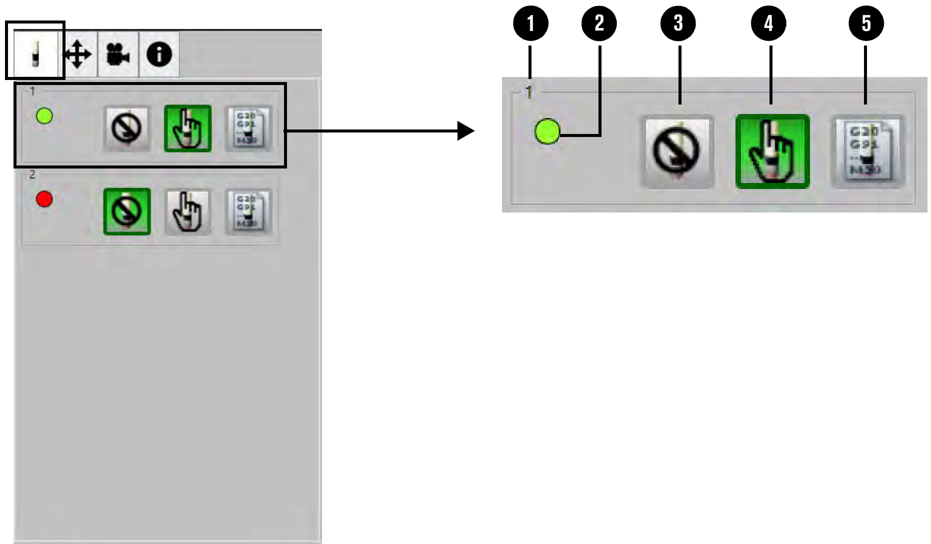
3 Screen capture and live video tab

4 Information tab

5 Program control (always shows)

The following sections explain each part of the Soft Op Con.

Station control tab



1 **Station number**
- As defined on the Station Configuration screen (stations 1 – 8) or the Digital I/O screen (stations 9 – 20)

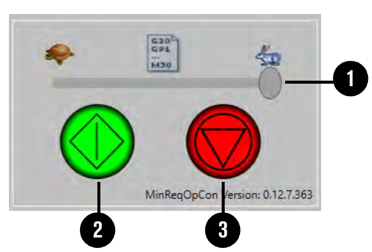
2 **Station status**
- Enabled (green)
- Disabled (red)

3 **Disable station key**
- Press to disable the station (green when selected)

4 **Manual mode key**
- Press to operate the station in manual mode (green when selected)

5 **Program (automatic) mode key**
- Press to operate the station in program mode (green when selected)

Program control

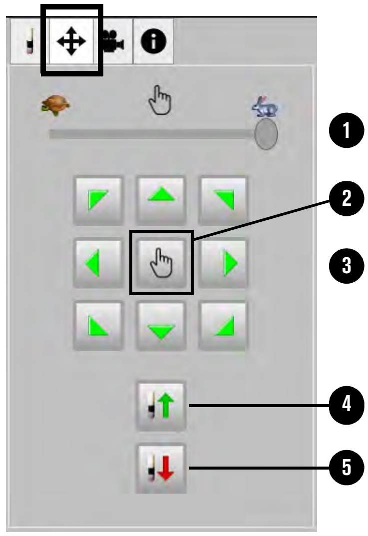


1 **Program speed** (if speed pots are enabled on the Analog I/O screen)
- Slide left for slower program speed
- Slide right for faster program speed

2 **Start cutting key**
- Press to start cutting

3 **Stop cutting key**
- Press to stop cutting

Manual motion tab



These soft keys control all of the stations that are currently enabled.

1 **Move speed** (if speed pots are enabled on the Analog I/O screen)
- Slide left for slower move speed
- Slide right for faster move speed

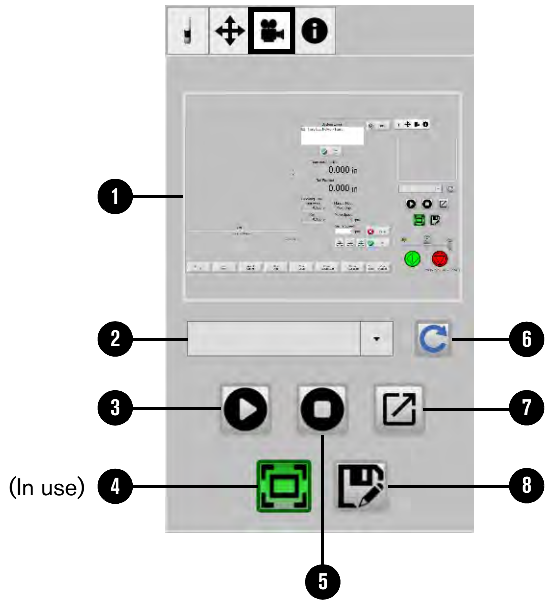
2 **Latch manual keys key**
- Press to turn ON latch manual keys (if enabled in the Special Setups screen)
- Press again to turn OFF latch manual keys

3 **Directional jog keys**
- Press an arrow key to move in that direction (if latch manual keys is enabled and ON)
- Press and hold an arrow key to move in that direction (if latch manual keys is OFF)

4 **Raise torch key**
- Press and hold to raise the torch or tool

5 **Lower torch key**
- Press and hold to lower the torch or tool

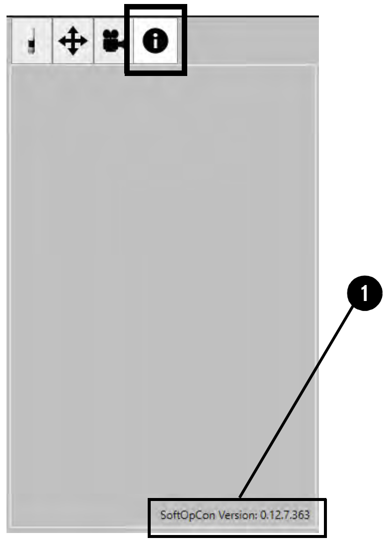
Screen capture and live video tab



1	**Screen capture and live video* view** This area shows the current screen capture or live video.*
2	**Select video source** If two or more video cameras are installed, choose a camera in the drop-down menu.
3	**Start** or **Pause** **live video**
4	**Capture the CNC screen**, including Phoenix and the Soft Op Con. This button appears green while in use.
5	**Clear screen capture/ Stop live video**
6	**Refresh video camera options**
7	**Enlarge image or video in a new window** ▪ Click and drag the new window, such as for display on an external monitor (if installed). ▪ To close the new window, click on the **X** in the upper-right corner of the window.
8	**Save screen capture** Open the **Save As** menu and save your screenshot as a file. Videos are not recorded on the CNC.

* Live video is supported if you have one or more supported USB video cameras installed. Video cameras are supplied by your cutting system manufacturer.

Information tab



1 **Soft Op Con version**

- Shows the version of the Soft Op Con on the CNC

Automatically open a custom software operator console

Cutting system manufacturers can create their own custom software operator consoles using the Phoenix application programmer interface (API). By default, the EDGE Connect Launcher automatically opens the standard Hypertherm Soft Op Con when an operator turns on the CNC. To automatically open the custom software operator console instead, modify the EDGE Connect Launcher.

Before you begin

- Make sure that the executable (.exe) files for the custom software operator console application are saved in the **C:\Phoenix** folder on the CNC. Typically there is one .exe file for the upper part of the Soft Op Con and one .exe file for the lower part of the Soft Op Con.
- Connect a USB keyboard and USB mouse to the CNC.

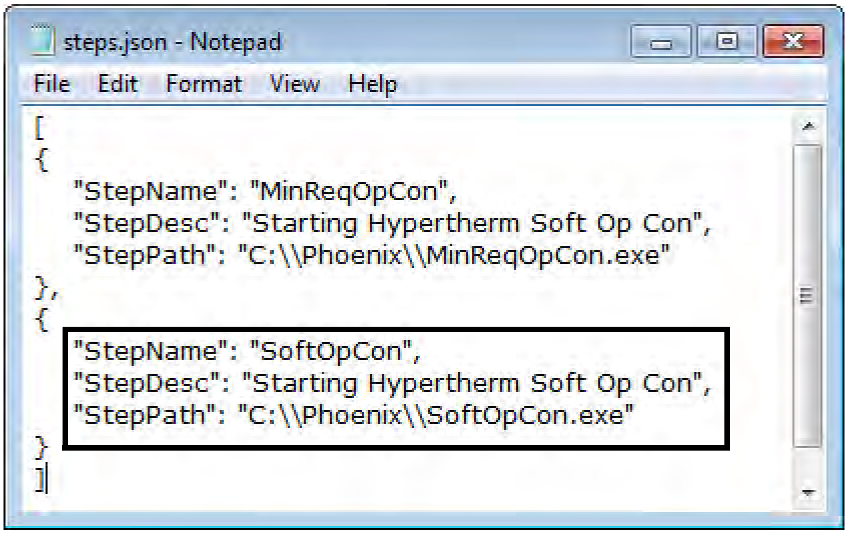
Modify the EDGE Connect Launcher

1. Close all the EDGE Connect software on the CNC.
   a. Click anywhere in the Main screen of Phoenix, and then press Alt+F4 to exit Phoenix.
   b. Click anywhere in the upper part of the Hypertherm Soft Op Con, and then press Alt+F4 to exit the upper part of the Hypertherm Soft Op Con.
   c. Click anywhere in the lower part of the Hypertherm Soft Op Con, and then press Alt+F4 to exit the lower part of the Hypertherm Soft Op Con.
2. Click the Windows Start button and then click **File Explorer**.
3. In the **C:\Phoenix** folder, double-click the **steps.json** file.

   The steps.json file should automatically open in Notepad. If it does not, open Notepad and then open the steps.json file.

4. To replace the upper part of the Hypertherm Soft Op Con, do the following in the steps.json file:

Figure 10



Only delete the specified characters. **Do NOT delete** quotation marks, commas, brackets, or other existing characters from the file.

a. Next to **StepName**, delete **SoftOpCon**.

b. Type the name of the upper part of the custom software operator console.

c. Next to **StepDesc**, delete **Starting Hypertherm Soft Op Con**.

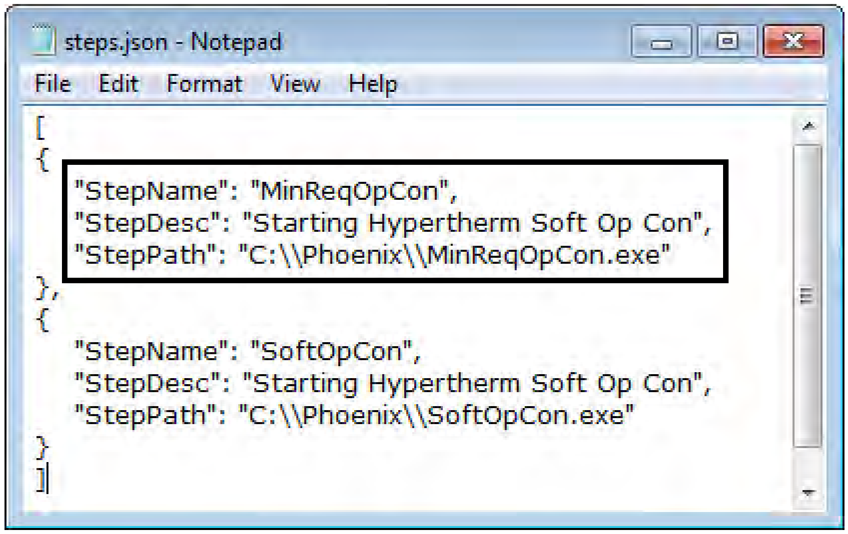
d. Type the text you want to appear while the custom software operator console opens.

e. Next to **StepPath**, delete **C:\\Phoenix\\SoftOpCon.exe**.

f. Type the absolute path of the executable (.exe) file for the upper part of the custom software operator console. For example: C:\\Phoenix\\CustomSoftOpCon.exe.

5. To replace the lower part of the Hypertherm Soft Op Con, do the following in the steps.json file:

Figure 11



Only delete the specified characters. **Do NOT delete** quotation marks, commas, brackets, or other existing characters from the file.

a. Next to **StepName**, delete **MinReqOpCon**.

b. Type the name of the lower part of the custom software operator console.

c. Next to **StepDesc**, delete **Starting Hypertherm Soft Op Con**.

d. Type the text you want to appear while the custom software operator console opens.

a. Next to **StepPath**, delete **C:\\Phoenix\\MinReqOpCon.exe**.

b. Type the absolute path of the executable (.exe) file for the lower part of the custom software operator console. For example: C:\\Phoenix\\CustomMinReqOpCon.exe.

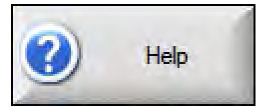
6. On the **File** menu in Notepad, click **Save** and then close the **steps.json** file.

7. Click the Windows Start button and then click **EDGE Connect Launcher** to open Phoenix and the custom Soft Op Con.

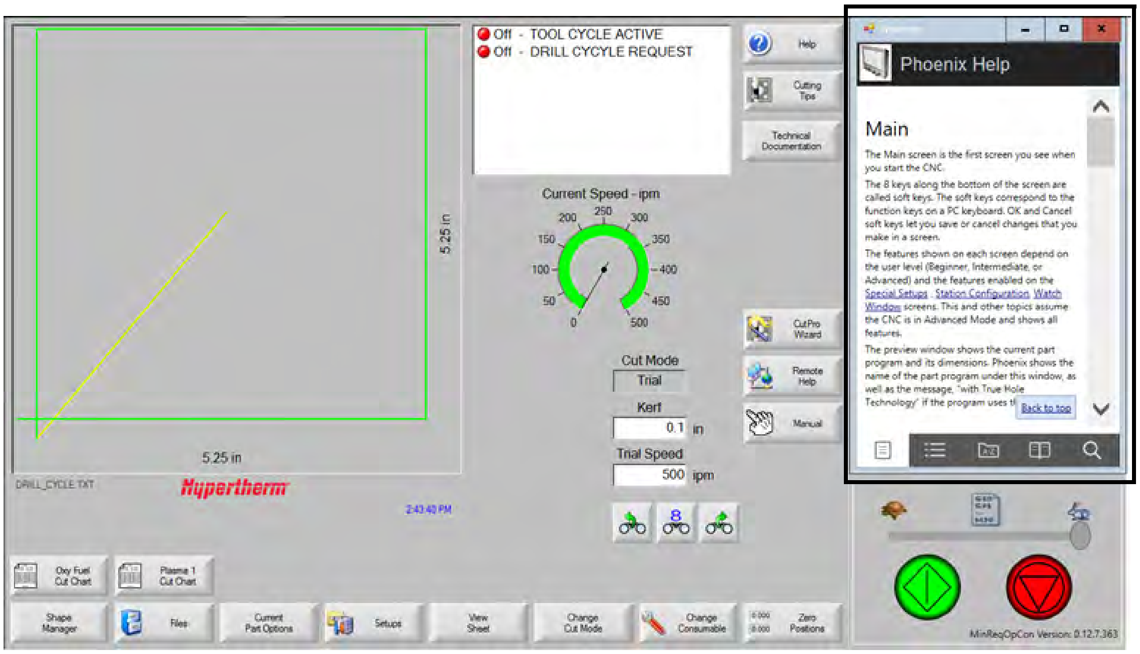
To get support using your custom Soft Op Con, contact your cutting system manufacturer.

Phoenix Help and technical documentation

Phoenix Help gives details on how to operate the Phoenix software. Phoenix Help is available for each screen in Phoenix.

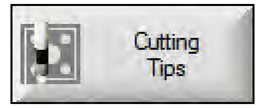


Choose the **Help** soft key to see information about each screen.

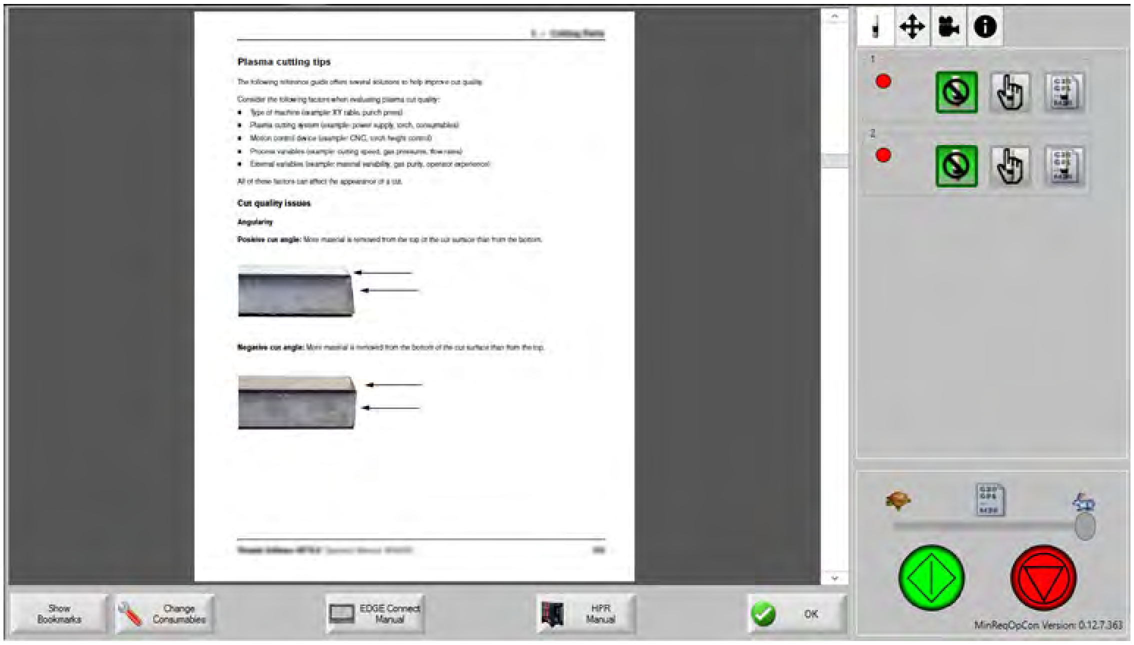


To close the Help window, press the **X** in the upper right corner.

Hypertherm also provides tips and solutions for common issues to help improve cut quality.



Choose the **Cutting Tips** soft key for this information.



To close the cutting tips, choose the **OK** soft key.

In addition, Hypertherm provides technical documentation for the CNC and any other installed Hypertherm components of your cutting system.

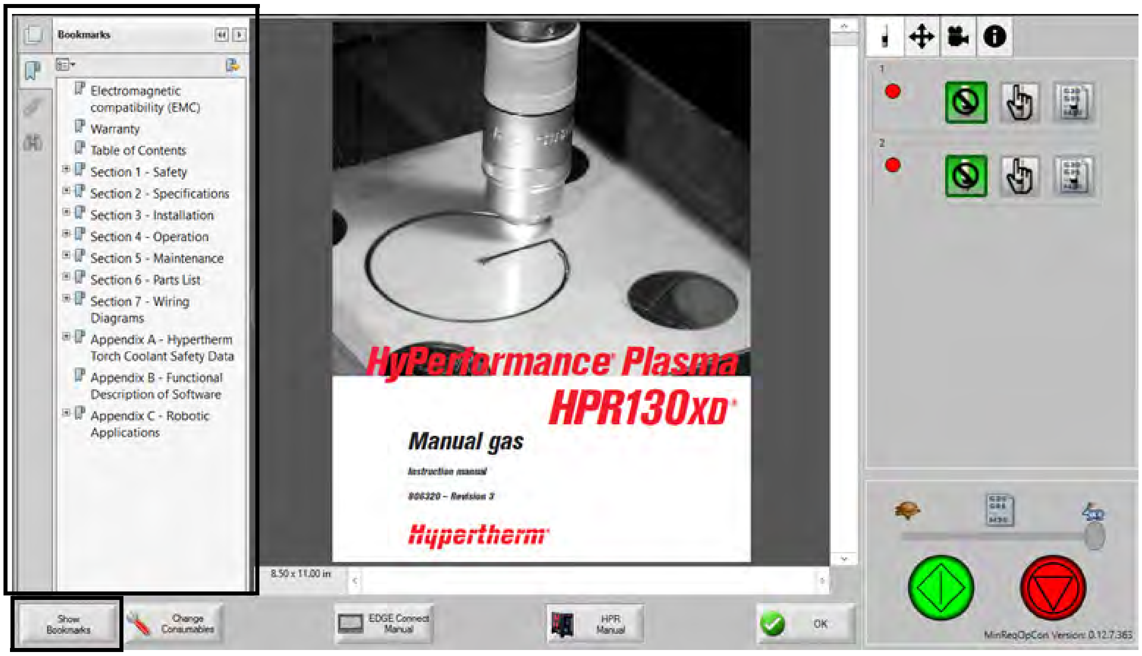
Choose the **Technical Documentation** soft key to get this documentation.

Figure 12

The *EDGE Connect Installation and Setup Manual* opens automatically.

Soft keys for all the technical documentation available for installed components appear at the bottom of the screen. For example, *Figure 12* shows an **EDGE Connect Manual** soft key and an **HPR Manual** soft key.

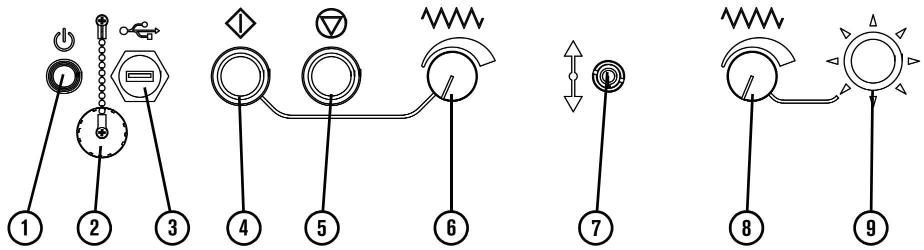
To see the contents of a manual, choose the **Show Bookmarks** soft key. Choose a topic in the list to go to that page in the manual.



To do a full-text search, press **Ctrl + F**.

Operate the EDGE Connect TC hardware operator console

The EDGE Connect TC comes with a hardware operator console.



1 Power button

2 USB connector cover with chain

3 Female USB connector

4 Start button (green)

5 Stop button (red)

6 Program speed potentiometer

7 Raise or lower station

8 Manual speed potentiometer

9 Joy stick

Operate the touchscreen

The EDGE Connect supports a touchscreen. The EDGE Connect TC comes with a touchscreen. Touch the window controls and fields to enter data into the software. Any field that requires data input automatically shows an on-screen keypad when you tap it twice.

Touchscreens, drops of water, and unintended motion

Drops of water can be a safety concern if you install the EDGE Connect on a water table. If water gets onto the touchscreen, the drops can activate a jog key and cause unintended motion on the table. Unintended motion can cause dangerous conditions for the operator and other people, damage to machinery, or faults in cutting. To prevent unintended motion:

- Dry your hands completely before you use the touchscreen.
- Install the CNC so that the touchscreen has protection from water splashes.
- If water could get on the touchscreen, do not display the jog keys in the Watch Window or the software operator console (Soft Op Con).

> Hypertherm recommends enabling the **Ready to Move?** dialog if there are any concerns that water could get onto the EDGE Connect touchscreen. The **Ready to Move?** dialog can be enabled on the Special Setups screen (Setups > Password > Special Setups).

For additional information, contact your nearest Hypertherm office listed in the front of this manual.

Optional keyboard and mouse

You can use a USB-connected keyboard and mouse to enter information and navigate the CNC software.

3 Machine Stop Strategies and Table Hardware

Machine stop strategies

Phoenix provides several options to ramp down motion to stop the cutting system in the case of a fault or error condition or an emergency. The option you choose depends on the mechanical design of the cutting system and your national and local electrical and safety codes. Consult your regulatory authority to make sure that your cutting system meets all codes.

Use the following information to implement the machine stop strategies that meet the needs of your cutting system and the electrical and safety codes for your location.

- How the CNC stops motion
  - Types of stops: Program stop/pause, Fast stop, and Fault ramp
- Safety circuit example
- About drive enable signals
- About the Drive Enable output and Drive Disabled input
- How the CNC enables and disables the drives
- Inputs and outputs provided by the CNC

How the CNC stops motion

Program pause/stop (non-urgent stop)

A non-urgent stop is used to pause or stop cutting or trialing. The CNC brings the machine to a stop using the acceleration rate set for the program speed or manual move speed at which the cutting system is moving.

Fast stop/fast deceleration (urgent stop)

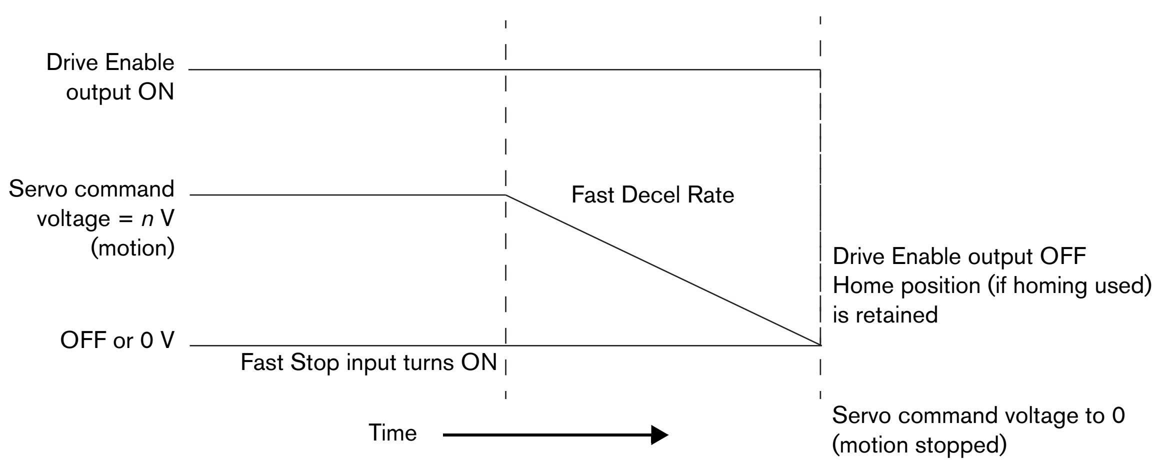
Fast stop brings all drives to an abrupt stop when a fault occurs and then disables the drives. A fast stop occurs when an operator activates the fast stop input or under these fault conditions:

Torch Collision (if Fast Decel is selected on the I/O setup screen)

- X and Y overtravel inputs are activated
- Software overtravel limits are activated (if Fast Decel is selected)

Fast stop should be used when safety is not a factor. The deceleration rate is entered on the Speeds screen in the **Fast Deceleration** field. The table's home position (if homing is used) and program position information are retained.

Figure 13 – Fast stop/fast deceleration



Fault ramp

Fault Ramp brings all drives to an abrupt stop and then disables the drives when one of these faults occur:

- Torch Collision (if Fault Ramp is selected on the I/O setup screen)
- X and Y overtravel inputs are activated
- Software overtravel limits are activated (if Fault Ramp is selected)

The deceleration rate uses the Fault Ramp Time entered on the Axis screens and disables the drives when the Fault Ramp time expires. The table's home position (if homing is used) is **not** retained and the table will need to be homed. An exception is a torch collision, which does not require homing again.

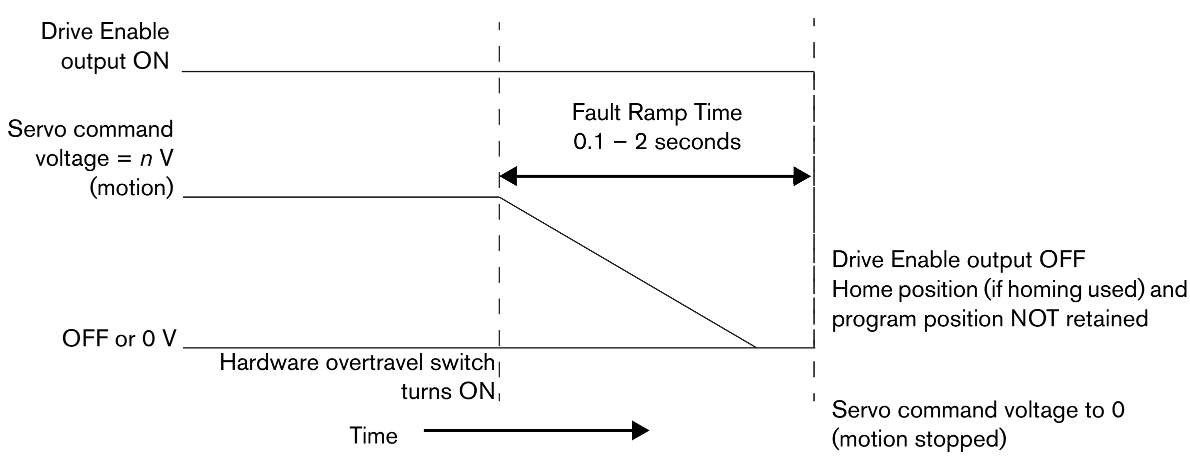
To meet immediate-disable safety standards, the Fault Ramp Time must be set to 0 seconds. When the Front Panel E-Stop input is activated, the drives will be disabled immediately. With no resistance from the motors, there is a potential for the table gantry to coast or move uncontrollably. How far the cutting machine coasts depends on the gantry weight, mechanics, the speed of the gantry when the Front Panel E-Stop input is activated, and the amount of friction present.

If a coasting gantry is a concern, adjust the Fault Ramp Time value to bring the gantry to a controlled stop before the drives are disabled. The available range for Fault Ramp Time is between 0.1 and 2 seconds.

In general, larger gantries require a longer Fault Ramp Time to stop the cutting system safely. Shorter Fault Ramp Times result in faster, or immediate, stop times which can cause damage to the cutting machine and the drive mechanics. Controlling the gantry to a stop may not meet certain safety regulations and may not meet other local codes or standards. Consult local safety regulations for a complete understanding of the requirements for your system.

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

Figure 14 – Fault Ramp



Emergency stop (E-stop) and Fast Stop compared

Fast Stop

Use the Fast Stop input when you want an urgent stop. DO NOT use the Fast Stop input for an E-stop. Connect the Fast Stop input to a separate switch on the operator console. When this input activates, the CNC decelerates motion for 1 second using the Fast Deceleration Rate on the Speeds screen, then turns off the individual drive enable signals and the Drive Enable output. The CNC retains position information so that you can restart the cutting job after clearing the condition that caused you to stop the system.

If the Fast Stop input is removed in less than 1 second, the drives may remain enabled.

Emergency stop

An emergency stop, or E-stop, is a physical switch on the cutting system or on a CNC custom operator console. The design and implementation of an E-stop switch and circuit depends on national and local electrical and safety codes.

The EDGE Connect/EDGE Connect TC CNC provides an input called Front Panel E-Stop for use when designing an E-stop circuit. When this input activates, the CNC turns off the individual drive enable signals and the Drive Enable output. The CNC also erases all position information from memory and requires that you re-home the cutting system. The CNC displays the status message **Emergency Stop Active** when the Front Panel E-stop input activates.

In Phoenix, you can enter a fault ramp time for each axis. The Fault Ramp Time delays the CNC from turning off the Drive Enable output so that power is maintained to the drives briefly (0.1 to 2 seconds) to stop the gantry without damage to the mechanics.

Normally local safety codes require AC power to be removed immediately from the drives. In this case, enter 0 for the fault ramp time for each axis.

CAUTION

Using a 0-second Fault Ramp Time could result in the gantry coasting, which can damage the cutting machine and the drive mechanics. Controlling the gantry to a stop may not meet certain safety codes and may not meet other local codes or standards. Consult local safety regulations for a complete understanding of the requirements for your system.

Enable the cutting machine after a power cycle or stop

After you turn ON the cutting machine and after an emergency or urgent stop, one or more error messages like the following appears.

These error messages are remaining from the interruption in EtherCAT network communications when the CNC was turned off or motion stopped. You need to re-enable the drives. Choose **Manual** to close each message. Phoenix continues to start the EtherCAT network.

If the error messages continue to appear, another method may be necessary. This is because how you re-enable the cutting machine depends on the wiring of the drives, the machine's safety circuit, the machine's reset mechanisms, the disable input assigned in the CNC, and the timing of the CNC's enable logic.

The following is another example sequence of re-enabling the drives. (The state of the machine is: E-stop depressed, no AC power to the drives, and the disable input is active to the CNC.) Turn ON the CNC and wait for the network to establish communications.

Deactivate the disable input to the CNC (Drive Disabled, Front Panel E-stop, or Fast Stop). Often this is accomplished by a reset mechanism or push-button, depending on the operator console and the drive box safety circuit.

After the input is deactivated, the CNC attempts to re-enable the drives for up to 2 seconds. After 2 seconds, the CNC stops trying. The drives must be in stand-by mode or the attempt fails after 2 seconds. If the CNC doesn't succeed in re-enabling the drives automatically, you can re-enable the drives manually by choosing the **Setups > Enable Control** soft key.

Safety circuit example

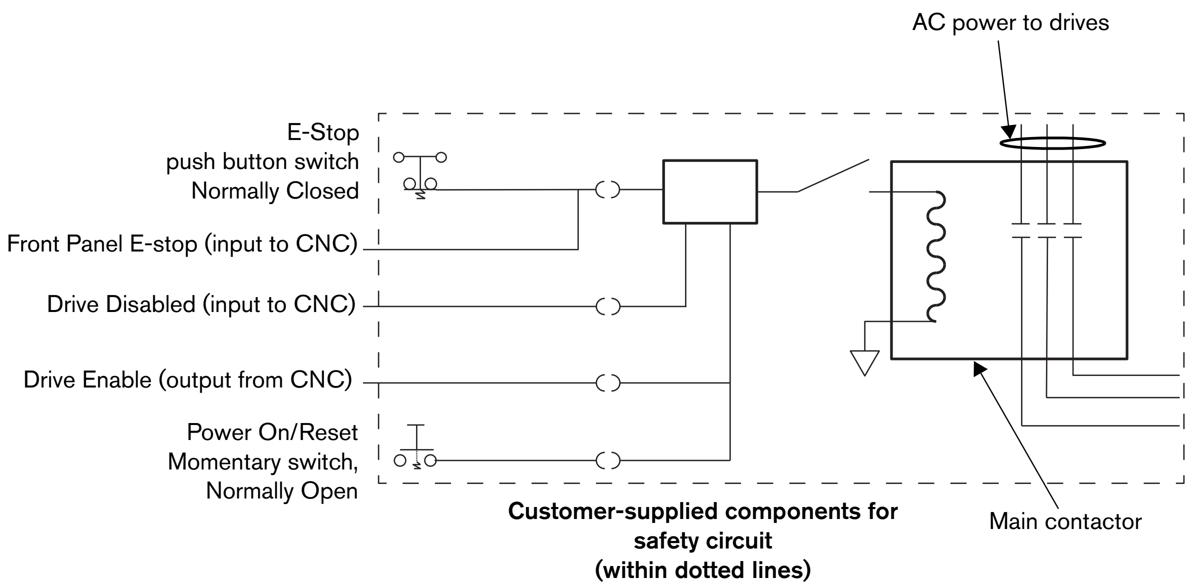
The following drawing shows a simplified example of the components and I/O of a cutting system safety circuit.

> The drawing is an example only. There are other ways to create a safety circuit. For instance, this example does not use the Safe Torque Off (STO) that is available on certain drives.

The CNC follows this sequence of operations:

- Power to the drives requires:
  - Front Panel E-stop input or Fast Stop input must be OFF, and
  - Drive Enable output must be ON, then
  - AC power can flow to the drives through the main contactor. A switch connects the main contactor to the safety circuit.

Figure 15 – Safety circuit logic example



When the E-stop push button is engaged, the Drive Disabled input turns ON and the Drive Enable output turns OFF. As long as Drive Disabled is ON, AC power is removed from the drives and no motion can occur. Use both of these inputs.

The Front Panel E-stop input indicates the E-stop switch is engaged. Drive Disabled indicates the safety circuit has not been reset. You have to re-enable with a push-button reset that turns off the Drive Disable input and maintains that status until the Drive Enable output becomes active.

After the stop condition is cleared, the Power On/Reset switch is used to restore the I/O and power to the drives.

Drive enable signals

When the CNC turns ON, with no safety inputs active (Drive Disabled, Fast Stop, or Front Panel E-stop), it immediately enables the drives.

Each axis has an axis enable signal. Phoenix activates this signal when it turns ON the Drive Enable # status signal where the # is the axis number. When the drives are wired in series, these signals activate or deactivate at the same time. When the drives are wired in parallel, the signals activate and deactivate in succession. After the Drive Enable bit is activated for each axis in the cutting system, the CNC turns ON the Drive Enable output.

In the **Setups > I/O** screen, select whether the drive enable signals should be activated in series or independently. If a fault or error occurs, drives in series all complete their ramp times before the drives are disabled. Drives that are wired in parallel or independently each disable at the end of the Fault Ramp Time.

Drive Enable output

A Drive Enable output must be assigned in the CNC and wired into the cutting system safety circuit. This output turns off after the individual drive enable signals turn off. When you require a controlled stop to the cutting system where motion ramps down before the drives are disabled, you want to include the Drive Enable output as part of your safety circuit logic and drive wiring scheme.

A 2-second delay is required before the Drive Enable output turns on. The operator may need to press and hold the reset switch for 2 seconds.

Drive Disabled input

Wire this input to the drive box safety circuit. The CNC doesn't know if a drive is disabled unless the Drive Disabled input is wired into the safety circuit. If a safety condition occurs, the Drive Disabled input opens (it is Normally Closed) and the CNC cannot enable the drives until the Drive Disabled input is closed again.

In a cutting system using an EDGE Connect CNC, messaging on the network tells the CNC whether or not the drives are enabled. If one or more drives are not enabled then the CNC changes to Control Disabled state. When the issue is resolved, the operator can toggle the Enable Control soft key in the Setups screen. This allows the CNC to re-enable the drives.

Hardware overtravel limits

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

Software travel limits

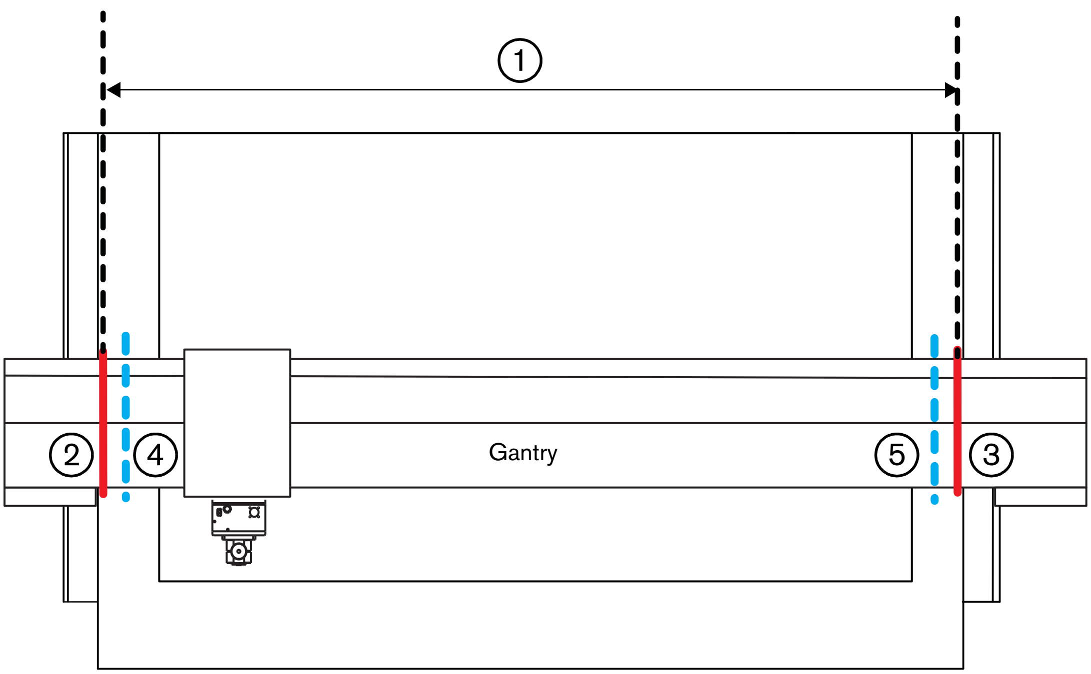
If Software Overtravels are enabled, the user is prompted for the minimum and maximum travel of the cutting machine.

A software travel limit (also called a *software overtravel*) is a position on the axis that can be reached before the system contacts a hardware limit switch or hard stop. *Figure 16* on page 86 shows measurements from an example system with a gantry that has 154 inches (3,912 mm) of travel on the Transverse axis between the overtravel switches.

The gantry travel distance of 154 inches (3,912 mm) also equals the Table Size X dimension on the Machine Setups screen (Setups > Password > Machine Setups).

For software travel limits to work correctly, the cutting system must first be homed so that the CNC can determine the absolute home position.

Figure 16



1 Gantry distance between hardware overtravel switches 154 inches (3,912 mm)

2 X- overtravel switch at 0 mm (0 inches) (this equals the Absolute Home position on the X axis setup screen). Add a hard stop to the left of ②. There needs to be enough space to stop without breaking your hard stop.

3 X+ overtravel switch at 154 inches (3,912 mm) (when the Absolute Home position is 0 mm [0 inches])

4 Software minimum travel limit at 3 inches (76 mm) to stop motion before contact with the hardware overtravel switch.

5 Software maximum travel limit at 151 inches (3,835 mm) to stop motion before contact with the hardware overtravel switch.

Home switches

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

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

When you enter a homing command, the CNC moves the axis toward the home switches at the Fast Home Speed until the switches activate. Once the switches activate, motion stops and then the axis moves in the opposite direction of the switch at the Slow Home Speed. The moment that the switch deactivates, the CNC resets the axis position to the absolute home position. Motion then continues for the Home Offset distance. When motion stops, the position is saved as the Home reference.

- The Absolute Home Position defines the position of the axis when the Home Limit Switch turns off or Marker Pulse is detected.
- The Home Offset Distance allows the user to set an offset distance from the Home Limit Switch.
- Home Direction determines which direction the axes will travel during phase one of the homing sequence.
- The Axes Home Position is the position of the axes when homing is complete.
- Use Marker Pulse, when enabled, assigns the absolute home position at the instant the marker pulse is detected. It is recommended that the Marker Pulse be used for optimal homing repeatability.
- The Dual Gantry Home switch offset value needs to change when the home switch is moved or replaced. Phoenix aligns the Rail and Dual Gantry axis based on the old Dual Gantry Home switch offset value. When you zero the switch offset value to measure the new switch offset value, Phoenix returns the wrong measured value because the original switch offset value is still being applied. There are 2 ways to fix this issue.
  - ❑ Zero the switch offset value and shutdown and re-start Phoenix.
  - ❑ Zero the switch offset value, disable and re-enable the control and then measure and enter the new switch offset value. A simple solution in Phoenix is to disable and then re-enable the axis each time the Dual Gantry Home switch offset value changes.

General recommendations

Follow these recommendations for ease of setup and best performance:

- Dedicate a single drive to each axis in the cutting system.
- Do not mix EtherCAT CoE and SoE drives on the same EtherCAT network.
- Connect the axis limit switches (also called overtravel switches) to the drive I/O, not to an inline I/O module. The drive and the CNC together provide the capabilities for the best possible machine deceleration in the event of a machine stop.
- When you choose Go to Home or attempt motion, a Ready to Move? message prompts you to confirm that you are ready to start motion. You can disable the Ready to Move? message in Special Setups.

4 Configure the EtherCAT Network

Hypertherm EtherCAT Studio is an EtherCAT network configuration tool. It includes an EtherCAT master and a library of EtherCAT slave information files for supported devices.

To configure the network, you use Hypertherm EtherCAT Studio to do the following:

1. Scan the slave devices to collect information about the network.
2. Enable and set up distributed clocks to synchronize communications.
3. Save the network information as a file and export it to the CNC.

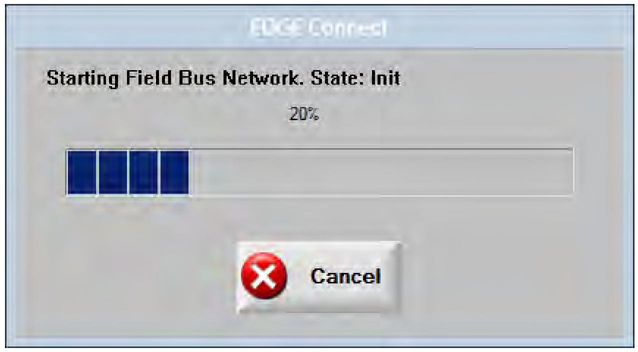
Before you begin

- Connect a USB keyboard and USB mouse to the CNC.
- Make sure that the cutting system is ready for motion:
  - All the slave devices are installed, configured, connected to the CNC with an EtherCAT cable, and energized.
  - The motors are connected to the drives.

Scan your slave devices

1. Turn ON the CNC.

   After Phoenix opens, it begins to start the EtherCAT network. The following message appears.



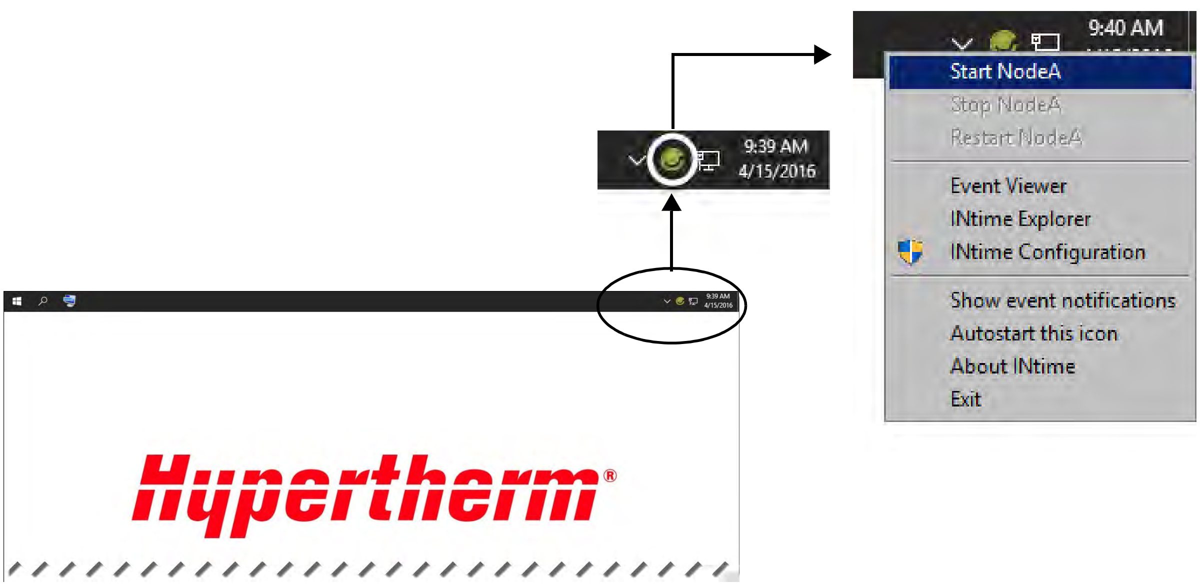
2. Choose **Cancel** to stop network start-up.

3. Click anywhere in the Main screen of Phoenix, and then press Alt+F4 to exit Phoenix.

4. Click anywhere in the upper part of Soft Op Con, and then press Alt+F4 to exit the upper part of Soft Op Con.

5. Click anywhere in the lower part of Soft Op Con, and then press Alt+F4 to exit the lower part of Soft Op Con.

6. In the notification area of the Windows task bar, right-click the INtime icon and then click **Start NodeA**.



7. Click the Windows Start button and then click **Master Server (INtime)**.

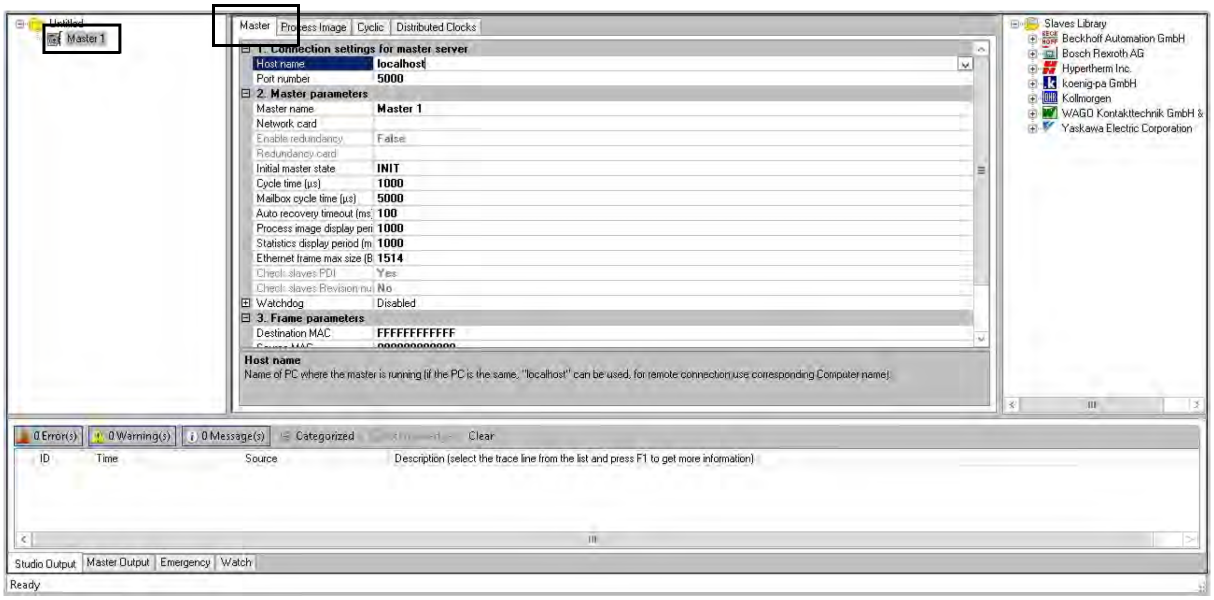
8. To make sure that INtime Server is running, click the Windows Start button and then click **Master Server (INtime)** again.

   The following message appears.

9. Click the Windows Start button and then click **HT EtherCAT Studio**.

   Hypertherm EtherCAT Studio appears. See *Figure 17* on page 92.

Figure 17 – Hypertherm EtherCAT Studio



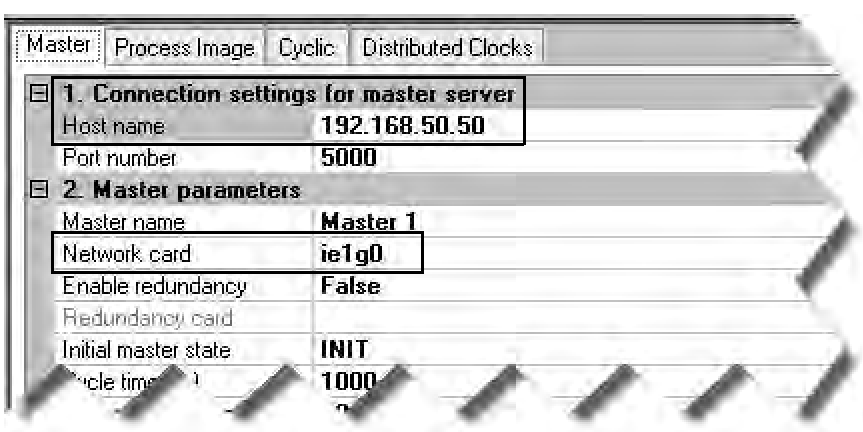
10. Make sure that **Master 1** is selected in the folder list on the left and that the **Master** tab is on top.

11. In **Host name** under **1. Connection settings for master server**, click the arrow to select **192.168.50.50:5000**. See *Figure 18*.

    Hypertherm EtherCAT Studio connects to the master.

    In **Network card** under **2. Master parameters, ie1g0** appears.

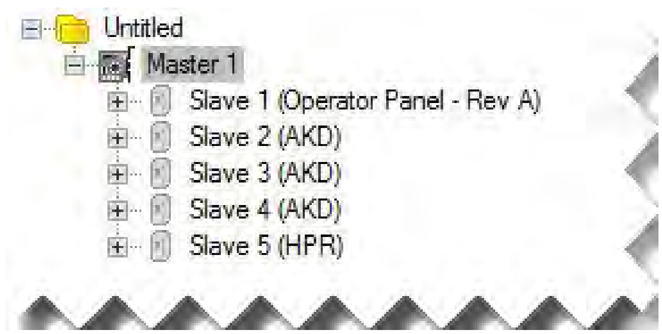
Figure 18 – Master settings



12. In the folder list on the left, right-click **Master 1** and then click **Scan configuration**.

Hypertherm EtherCAT Studio scans the slave devices on your network, and then shows a list of your slave devices.

Figure 19 – List of slave devices



As shown in the example in *Figure 19*, if you have an EDGE Connect TC, the operator console PCB (229827) is always the first slave device.

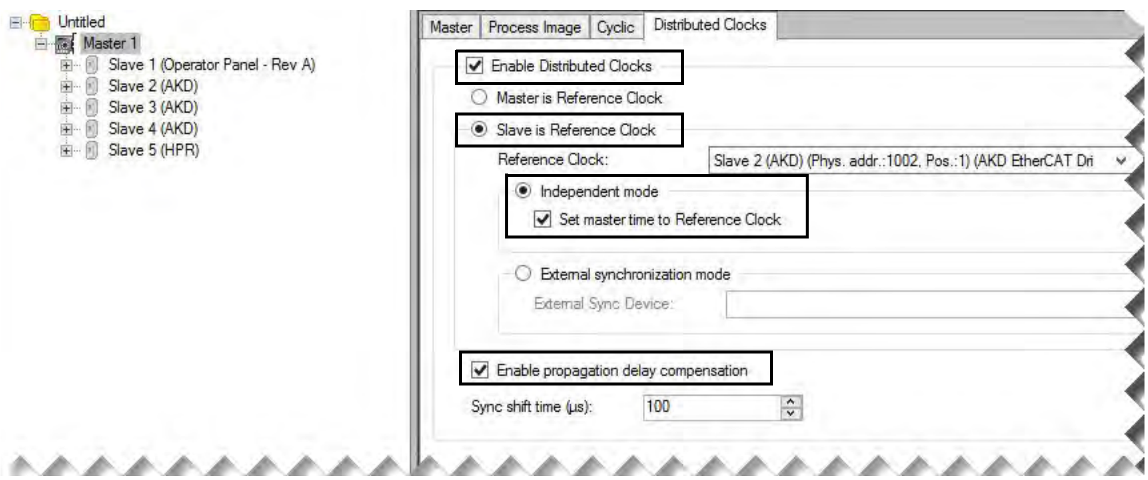
The HPR EtherCAT plasma interface PCBs (229829) are always the last slave devices (when installed as instructed).

Enable and set up distributed clocks

1. Make sure that **Master 1** is selected in the folder list on the left, and then click the **Distributed Clocks** tab.

2. Make the following selections. See *Figure 20*.
   - **Enable Distributed Clocks**
   - **Slave is Reference Clock**
   - **Independent mode**
   - **Set master time to Reference Clock**
   - **Enable propagation delay compensation**

Figure 20 – Distributed Clocks tab



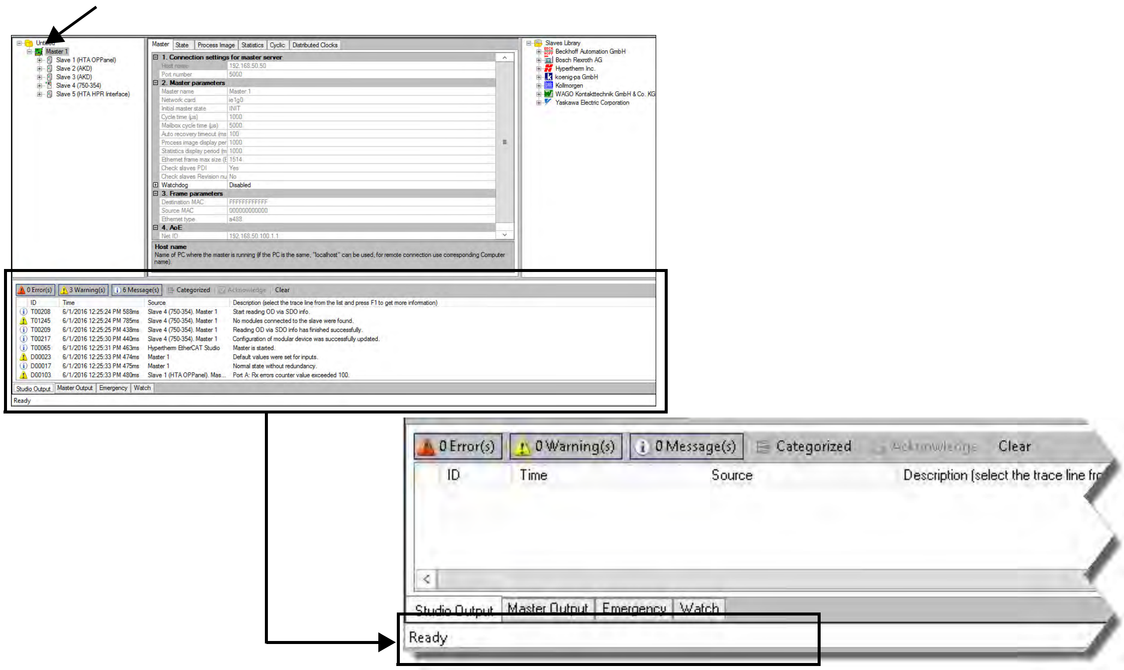
Hypertherm EtherCAT Studio automatically selects the first slave device that is capable of being the reference clock as the reference clock.

You will set the **Sync shift time** later. For now, leave the default value in this field.

3. In the folder list on the left, right-click **Master 1** and then click **Attach Master**.

Look at the status bar at the bottom of the screen to monitor this process.

Continue to the next step when the symbol to the left of Master 1 is green and the status is **Ready**.



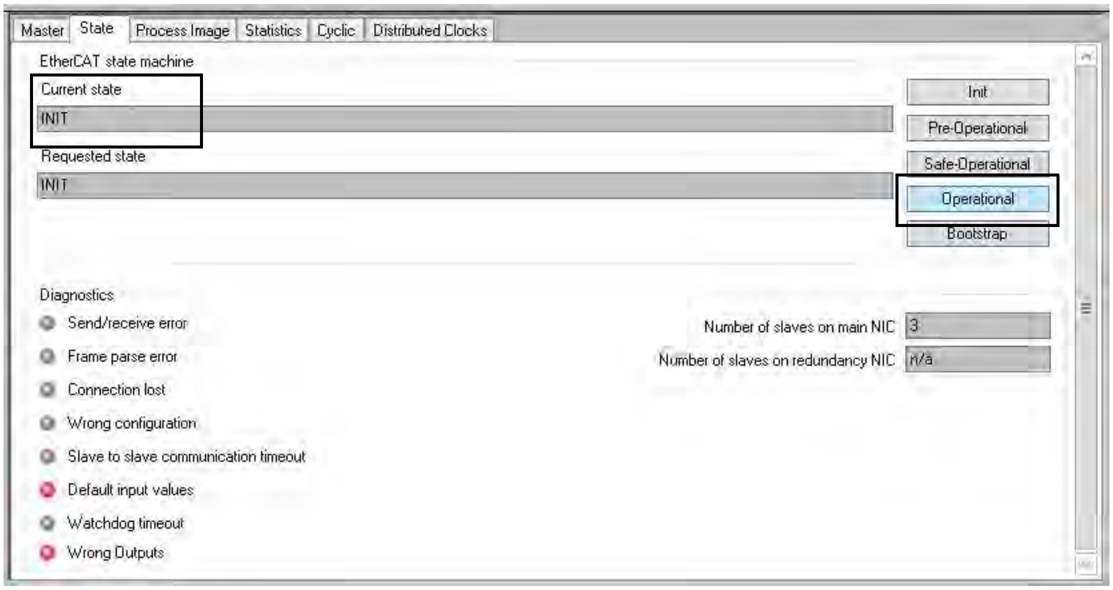
If Hypertherm EtherCAT Studio cannot attach the master, examine each slave device for errors. If you still do not know the cause of the problem, remove all the slave devices from the network except for the first slave device. Then go back to *step 1* on page 89 (see *Scan your slave devices*) and repeat these instructions. Continue adding one slave device at a time and repeating these instructions until you find the slave device that is causing the problem.

4. Make sure **Master 1** is selected in the folder list on the left and then click the **State** tab. See *Figure 21* on page 96.

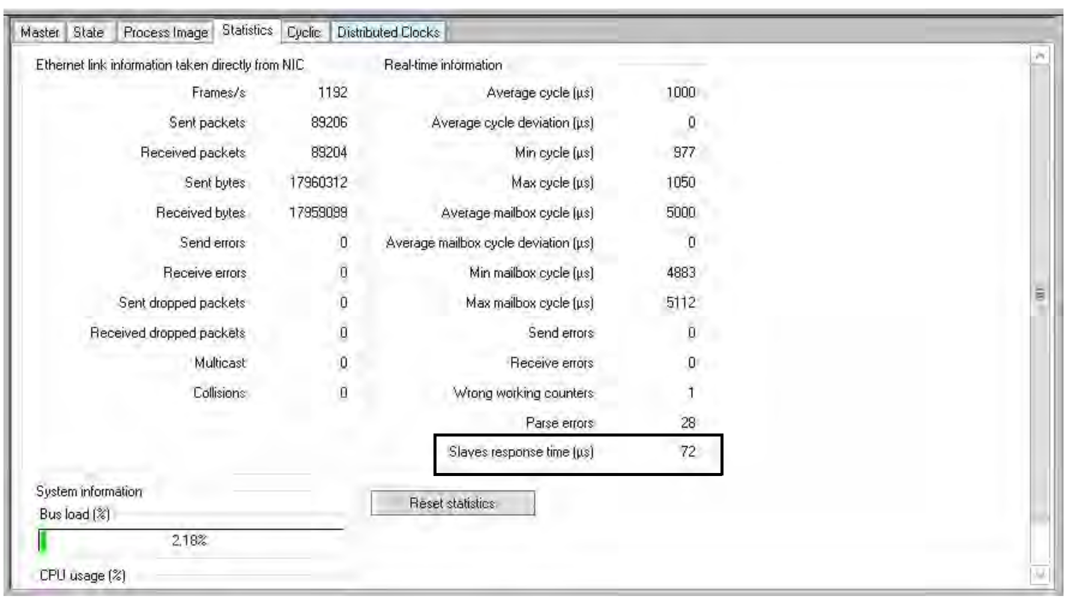
5. Click **Operational**.

Current state changes from **INIT** to **OPERATIONAL**.

Figure 21 – State tab



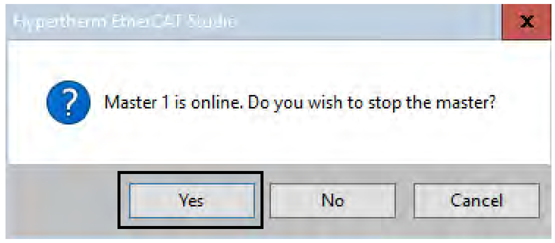
6. Click the **Statistics** tab. Monitor the number in **Slaves response time** for 15 to 30 seconds, and record the highest response time you see.



7. Select the **State** tab and click **INIT**. **Current state** changes from **OPERATIONAL** to **INIT.**

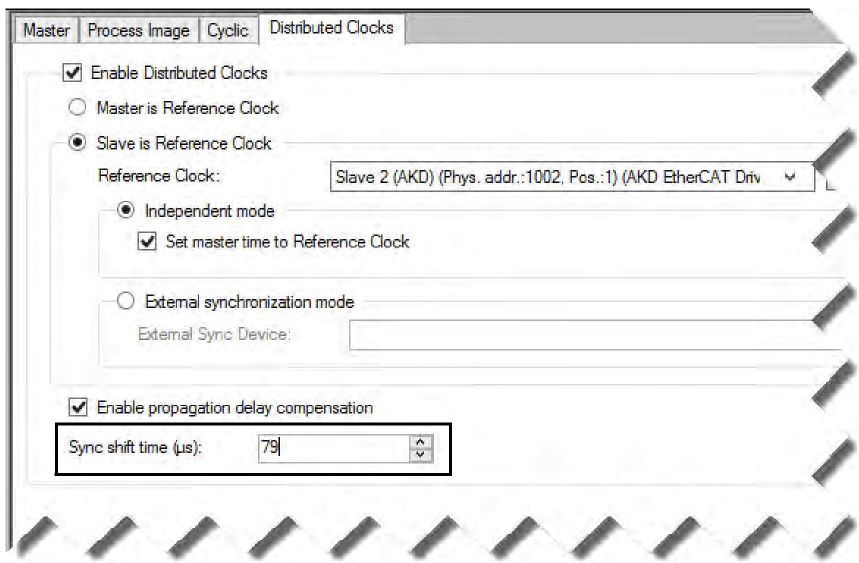
8. In the folder list on the left, right-click **Master 1** and then click **Detach Master**.

9. When Hypertherm EtherCAT Studio asks you if you want to stop the master, click **Yes**.



10. Click the **Distributed Clocks** tab. In **Sync shift time**, type the response time noted in *step 6* on page 96 multiplied by 110% (1.10) and rounded up or down as necessary.

    For example, assume the response time noted in *step 6* was 72. 72 multiplied by 1.10 is 79.2. Round that down to 79, and type 79 in Sync shift time.



Export your network configuration to the CNC

1. In the folder list on the left, right-click **Master 1**, then click **Export**, and then click **Export Master Configuration KPA**.

2. In the **Save As** window, make sure you are in the **C:\Phoenix** folder. See *Figure 22*.

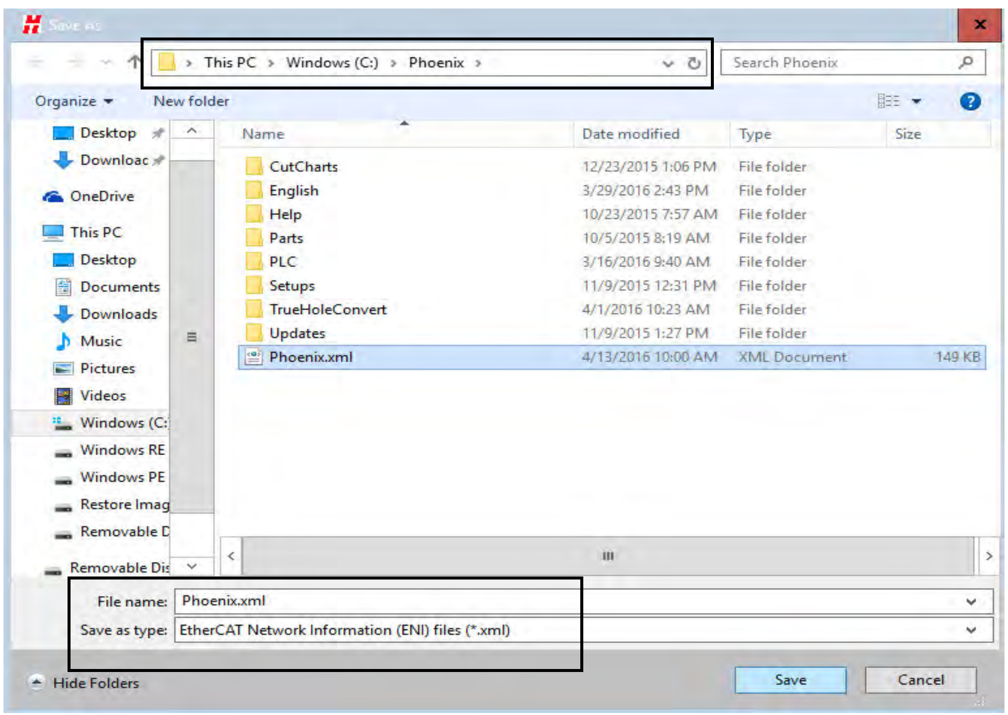
3. In the list of files, click **Phoenix.xml**. Make sure that **Save as type** is **EtherCAT Network Information (ENI) files (*.xml)**.

4. Click **Save**.

5. Click **Yes** to overwrite the existing XML file.

   The EtherCAT network is now configured.

Figure 22 – Save As window



6. On the **File** menu, click **Exit** to close Hypertherm EtherCAT Studio.

7. When Hypertherm EtherCAT Studio asks you if you want to stop the master, click **Yes**.

8. When Hypertherm EtherCAT Studio asks you if you want to save your project file, click **Yes**.

   Saving the project file is optional but strongly recommended, as a troubleshooting aid.

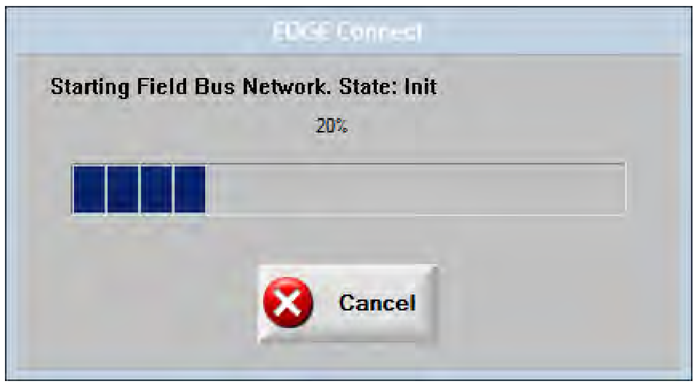
9. In the **Save As** window, make sure you are in the **C:\Phoenix** folder.

10. In **File name**, type **Phoenix.ecsn**. Make sure that **Save as type** is **EtherCAT Studio project files (*.ecsn)**.

11. Click **Save**.

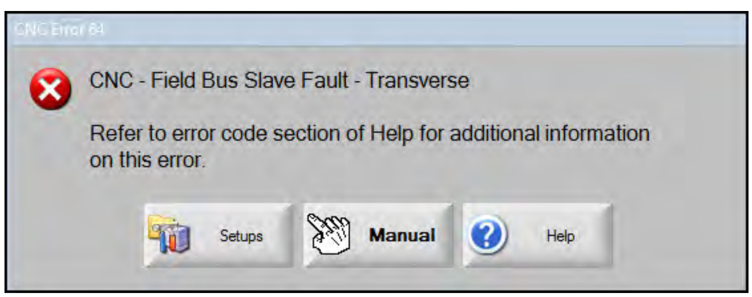
12. Click the Windows Start button and then click **EDGE Connect Launcher** to restart Phoenix.

After Phoenix opens, it begins to start the EtherCAT network. The following message appears.



This message shows the progress of network startup as it continues from **Init** to **Pre-Operational** to **Safe-Operational** to **Operational**.

One or more error messages like the following may appear.



Click **Manual** to close each message. Phoenix continues to start the EtherCAT network. These error messages are remaining from when you canceled network startup (see *step 2* on page 90).

If you continue to see error messages, disconnect the slave devices from power and then reconnect them to power. Then click **Manual** to restart the network.

5 Machine Axes

Overview

Phoenix has several screens that let you define your cutting system. The screens are accessed with the Machine Setups password provided by Hypertherm.

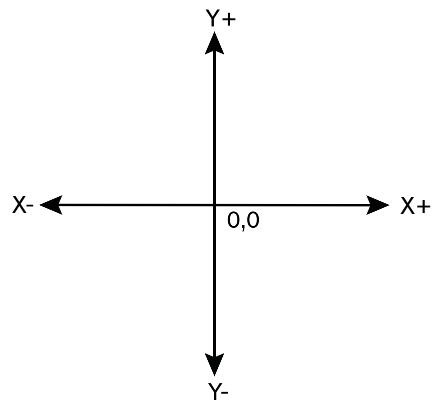
Access the screens by choosing Setups > Password > Machine Setups.

This section describes setting up your cutting system, including:

- Defining axis orientation and positive motion
- Identifying Transverse, Rail, and Dual Gantry axes on the cutting system (for Sensor THC axis setup see *Torch Height Control (THC)* on page 135)

Axis orientation and positive motion

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

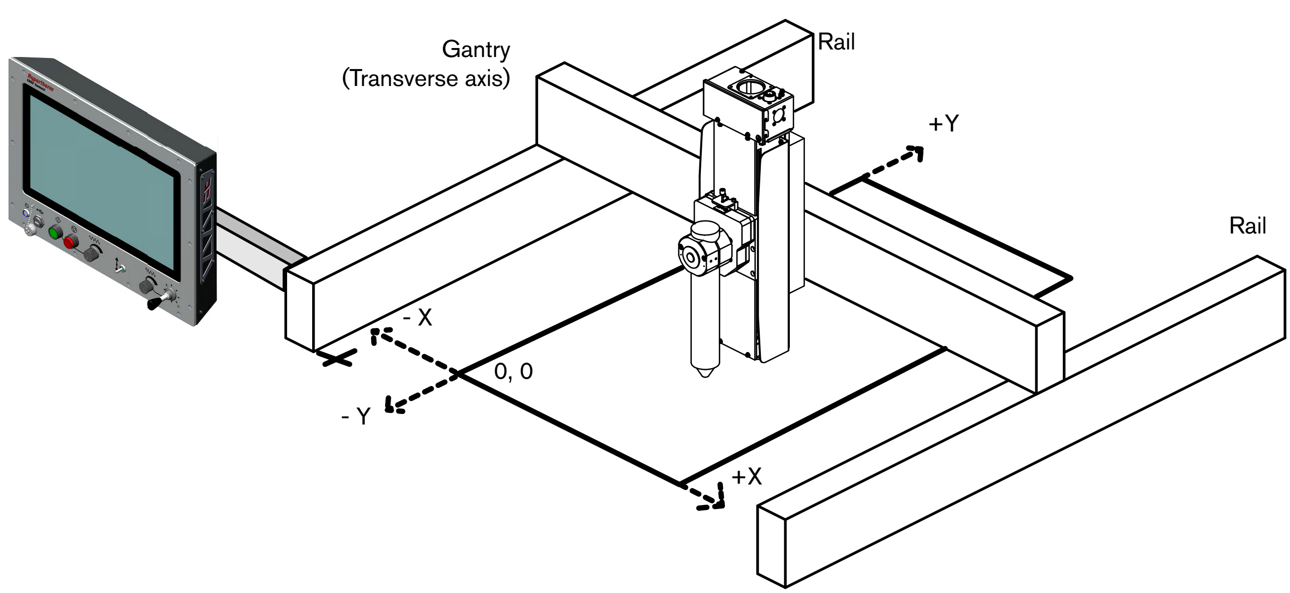


Axes assignments on the Machine screen are sequential, beginning with the first axis (X).

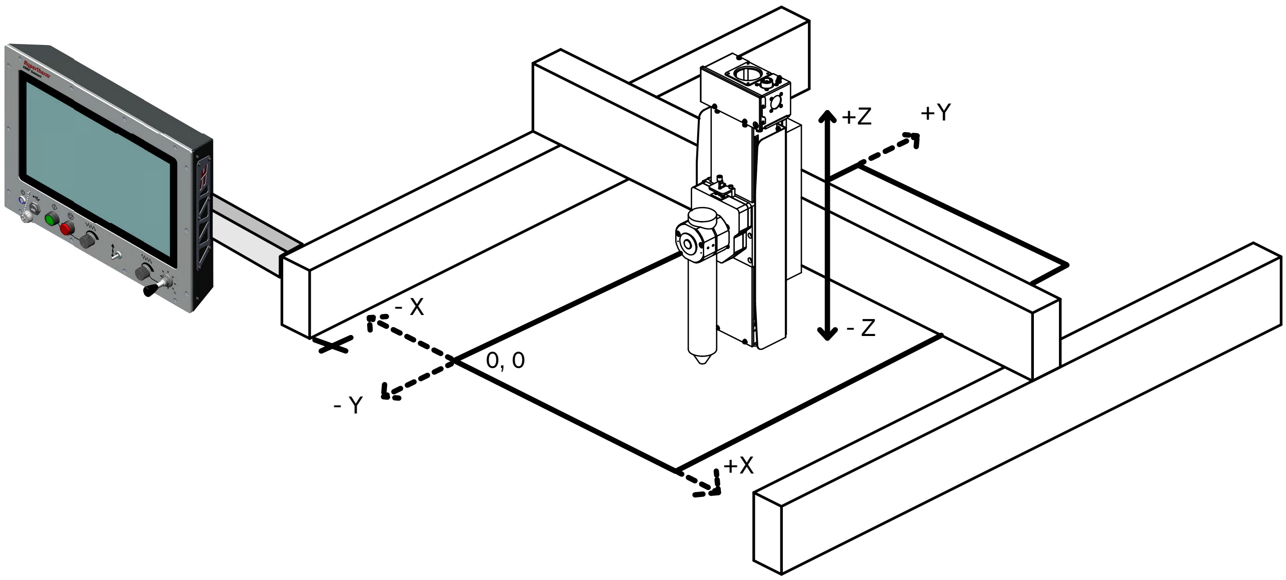
The X axis is the first axis Phoenix needs to know about. In a standard orientation, the X axis is along the gantry. Phoenix calls the axis along the gantry the Transverse axis. The Y axis is always perpendicular to the X axis. In Phoenix, the Rail axis is perpendicular to the Transverse. In a standard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

Figure 23 shows a cutting table where X is assigned to the gantry, also called Transverse, and Y is assigned to the Rail.

Figure 23



The Sensor THC, integrated into many Hypertherm CNCs, is also an axis on the CNC. The Sensor THC axis, or lifter axis, is called a Z axis. Its positive direction is always up (skyward) and its negative direction is always down (earthward). Its Z position increases in the positive direction for motion in the downward direction. If the Sensor THC is not used, an axis is not assigned to the CNC and Z-axis motion is not calculated.



Axis assignments

Hypertherm CNCs use fixed axis assignments for all axis types except the Sensor THC. The tables below show a sample of possible configurations. with the axis number and assignment that the CNC uses for up to 6 axes, and the common letter identifiers for the axes.

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

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

4-axis, I-cutting, no Dual Gantry

Axis number	Axis assignment
1	Transverse or Rail (X or Y)
2	Rail or Transverse (Y or X)
3	Sensor THC (Z)
4	Transverse 2

5-axis, I-cutting, with Dual Gantry

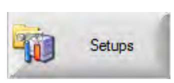
Axis number	Axis assignment
1	Transverse or Rail (X or Y)
2	Rail or Transverse (Y or X)
3	Dual Gantry (W)
4	Sensor THC (Z)
5	Transverse 2

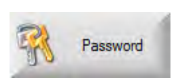
6-axis, 2-station, I-cutting, with Dual Gantry

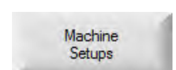
Axis number	Axis assignment
1	Transverse or Rail (X or Y)
2	Rail or Transverse (Y or X)
3	Dual Gantry (W)
4	Sensor THC 1 (Z)
5	Transverse 2
6	Sensor THC 2

Machine setups screen

Use the Machine Setups screen to define the axis assignments, machine motion direction, and table size.

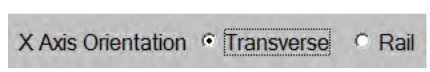
To open the Machine Setups screen, choose 

> 

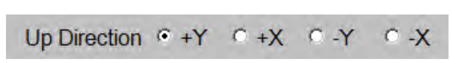
> 

.

The Machine Setups screen includes Sensor THC settings. For more information about those settings and Sensor THC axis setup, see *Torch Height Control (THC)* on page 135.

1. Choose the X axis as the **Transverse** or **Rail** axis 

. The default setting is **Transverse**.

2. Choose the **Up Direction** 

to define the machine motion when the up arrow soft key  is pressed on the Soft Op Con when the CNC is in manual mode. This choice also defines the machine motion when the joystick on the EDGE Connect TC hardware operator console is moved in the up direction when the CNC is in manual mode.

3. Choose the **Right Direction** Right Direction +X -X to define the machine motion when the right arrow soft key is pressed on the Soft Op Con when the CNC is in manual mode. This choice also defines the machine motion when the joystick on the EDGE Connect TC hardware operator console is moved in the right direction when the CNC is in manual mode.

4. Choose the **X and Y Motor/Encoder** mode X and Y Motor/Encoder Normal Swapped. Select **Swapped** to swap the X and Y axis encoder signals internally.

5. Choose **Yes** or **No** for **Dual Gantry Installed** Dual Gantry Installed No Yes. If yes is selected, the Dual Gantry will be assigned to axis 3 and the Z axis will be axis 4.

6. Enter the size of your cutting table. Enter the distance between the X overtravel switches and the Y overtravel switches. The X and Y dimensions represent the cutting area of the table and limit the workpiece size dimensions set in the Cutting screen.

   Table Size X 480 in Y 960 in

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

Transverse or Rail axis

The settings on the Transverse and Rail screens are identical.

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

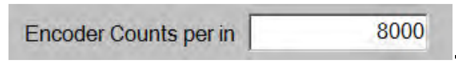
To get to the Transverse or Rail screens:

1. Choose Setups > Password > Machine Setups > Axes > Transverse or Rail.

2. Enter the **Servo Error Tolerance** Servo Error Tolerance 0.1 in.

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

   The amount of servo error tolerance depends on the cutting machine mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high may allow inaccurate motion or result in mechanical damage. Set the servo error tolerance to a value higher than the steady-state following error that is reported by the drive.

3. Enter the **Encoder Counts per mm (inch)** 

.

Encoder counts are a position scaling factor used by Phoenix. Refer to the drive manufacturer documentation for specific scaling information required by the drive.

In general, to determine the encoder counts per mm (inch), you need to know the following:

- Counts per revolution of the motor
- Gear ratio
- Distance of travel in one revolution of the pinion gear
- Diameter of the pinion gear when it engages the rack

The formula you use is shown below:

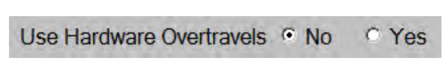
$$\frac{x \text{ encoder counts}}{1 \text{ motor revolution}} \text{ X } \frac{x \text{ motor revolutions}}{1 \text{ pinion revolution}} \text{ X } \frac{1 \text{ pinion revolution}}{x \text{ pinion circumference X } \Pi \text{ (pitch)}} = x \text{ encoder counts per mm (inch)}$$

Below is an example using the following sample data:

- Kollmorgen AKD drive with 65,536 encoder counts per revolution of the motor
- 10:1 gear ratio
- 150 mm (5.91 in.) pitch

$$\frac{65{,}536 \text{ encoder counts}}{1 \text{ motor revolution}} \text{ X } \frac{10 \text{ motor revolutions}}{1 \text{ pinion revolution}} \text{ X } \frac{1 \text{ pinion revolution}}{150 \text{ mm (5.91 in.) pitch}} = 4{,}369.067 \ (110{,}890.017) \text{ encoder counts per mm (inch)}$$

This is just an example. Refer to the drive manufacturer documentation for specific scaling information required by the drive. Some drive-specific information is also available in Hypertherm Application Notes. Technical documentation is available at www.hypertherm.com/docs.

4. Choose **Yes** or **No** for **Use Hardware Overtravels** 

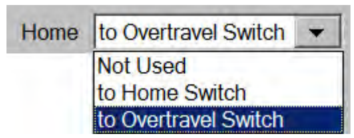
to select whether the cutting system will be using hardware overtravel switches. If hardware overtravel switches are used, the CNC disables feedback and shows an error message if the inputs become active. We recommend that hardware overtravel switches be installed.

5. Enter **Backlash Compensation** 

to offset or compensate for any backlash in the mechanics of the drive system. The maximum value for backlash compensation is 12.7 mm (0.5 inch).

6. Enter the **Fault Ramp Time** 

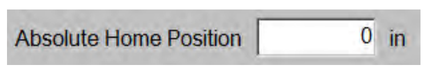
to set the motion deceleration time after a fault occurs. At the end of the Fault Ramp Time the drives will be disabled.

7. Choose a **Home** parameter 

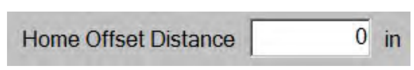
. This parameter is used to activate the Home feature. Depending on your configuration, the table may be homed to either a designated home switch or an overtravel switch.

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

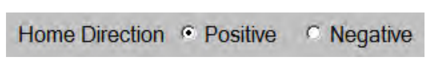
   When you home an individual axis or all axes, the CNC moves the axis toward the home switches at the Fast Home Speed until the switches activate. Once the switches activate, motion stops and then the axis moves in the opposite direction of the switch at the Slow Home Speed. The moment that the switch deactivates, the CNC sets the axis position equal to the **Absolute Home Position**.

8. Enter the **Absolute Home Position** 

to define the position of the axis when the Home Limit Switch turns OFF or Marker Pulse is detected.

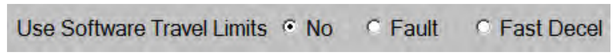
9. Enter the **Home Offset Distance** 

. This parameter allows you to set an offset distance from the Home Limit Switch or an offset from the marker pulse.

10. Choose a **Home Direction** 

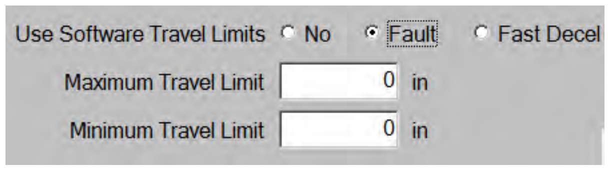
to determine which direction the axes will travel during phase one of the homing sequence.

11. Choose to disable or enable software travel limits. The CNC is capable of running with software travel limits that are based on position.

    a. Choose **No** to disable this feature 

.

    b. Choose **Fault** or **Fast Decel** to enable this feature. **Fault** performs a Fault Ramp to ramp down motion. The Fault Ramp Time is set on the Axis screen. **Fast Decel** uses the Fast Deceleration rate set on the Speeds screen to ramp down motion. If Software Overtravels are enabled, enter the maximum and minimum travel from the cutting machine's final home position.

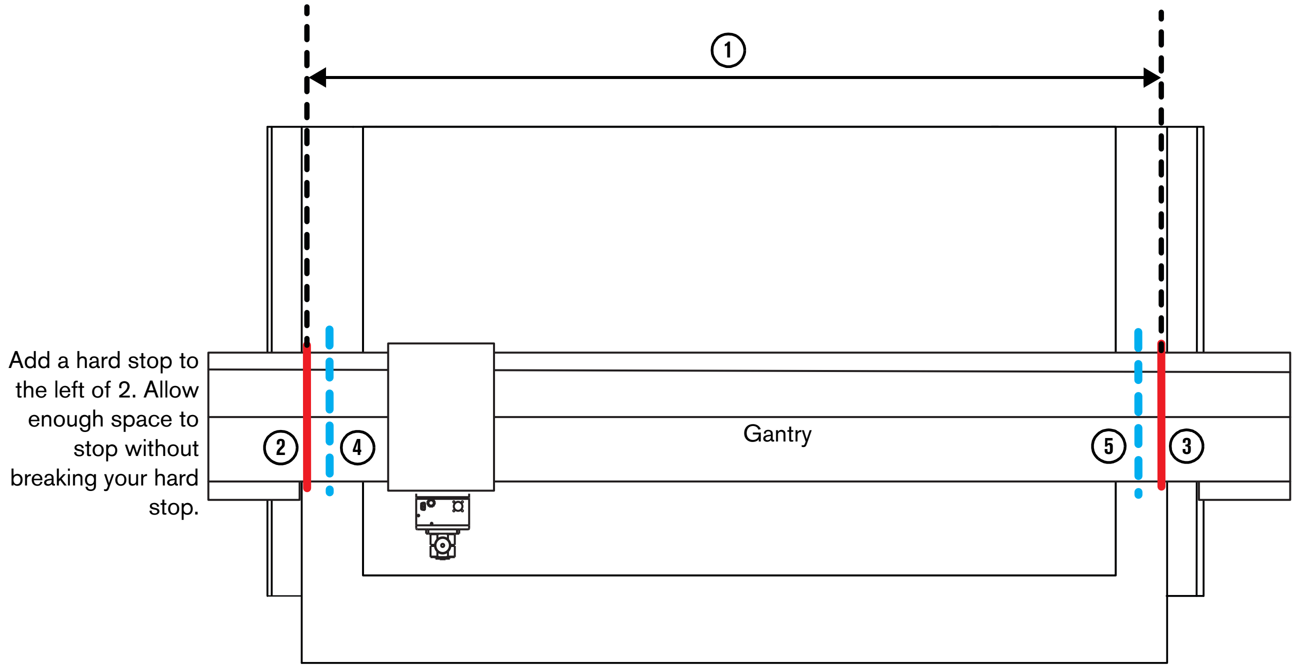


Using software travel limits

A software travel limit (also called a *software overtravel*) is a position on the axis that can be reached before the system contacts a hardware limit switch or hard stop. *Figure 24* on page 109 shows measurements from an example system with a gantry that has 154 inches (3,912 mm) of travel on the Transverse axis between the overtravel switches.

The gantry travel distance of 154 inches (3,912 mm) also equals the Table Size X dimension on the Machine Setups screen (Setups > Password > Machine Setups).

Figure 24 – Example measurements



1 The gantry distance between the hardware overtravel switches is 3,912 mm (154 inches).

2 The X- overtravel switch at 0.

3 The X+ overtravel switch at 3,912 mm (154 inches).

4 The software minimum travel limit is at 76 mm (3 inches) to stop motion before contact with the hardware overtravel switch.

5 The software maximum travel limit is at 3,835 mm (151 inches) to stop motion before contact with the hardware overtravel switch.

Dual Gantry axis

1. Choose Setups > Password > Machine Setups > Axes > Dual Gantry.

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

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

1. Enter the **Skew Error Tolerance** Skew Error Tolerance [0.1] in to set the amount of position error allowed between the master and slave gantry drive axes prior to an error.

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

2. Enter the **Encoder Counts per mm (inch)** Encoder Counts per in [8000]. See *step 3* on page 106 for more information.

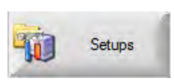
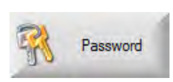
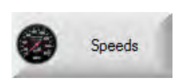
3. Enter the **Backlash Compensation** Backlash Compensation [0] in to offset or compensate for any backlash in the mechanics of the drive system. The maximum value for backlash compensation is 12.7 mm (0.5 inch).

6 Speeds

Set speeds

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

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

To get to the speeds screen choose  >  > .

Set speed ranges (speed breaks)

Enter parameters for the **Speed 0 to** settings.

These parameters specify speed ranges (or breaks) and customize the acceleration rates for these speeds. When a speed range that is lower than the maximum speed is entered, a new range setting is created. You can set a maximum of 5 speed ranges.

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

Acceleration rates

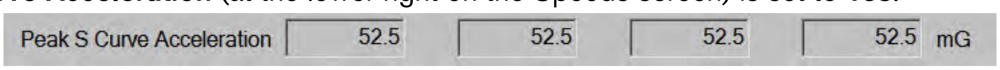
1. Enter parameters for the **Acceleration Rate** settings.These parameters determine both the acceleration and deceleration rate for all motion.

All mechanical systems have different acceleration and deceleration rates to move the cutting device smoothly. The higher the acceleration rate, the faster the machine will get up to speed and decelerate. Higher acceleration rates will be able to reach optimal cut speeds faster, resulting in better cut quality. The lower the acceleration rate, the slower the machine will get up to speed and decelerate.

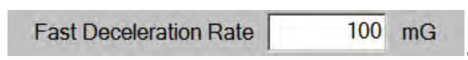
It is important to use the optimal acceleration for your cutting table's mechanical design and drive/motor capability. Excessive acceleration settings can cause issues with the performance of the cutting system.

Enter a value in mGs for the desired acceleration rate. 1 mG is approximately 0.386 inches per second^2. The drive, motor, gears, mass, and overall table design affect the acceleration capability. We recommend that for the initial tuning of a new design that you start at 10 mG before proceeding to your final acceleration values.

2. Enter parameters for the **Peak S Curve Acceleration** fields. This parameter appears when **S Curve Acceleration** (at the lower right on the Speeds screen) is set to **Yes**.



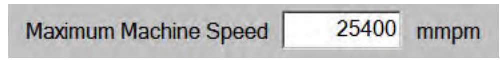
For more information, see *Trapezoidal motion compared to S-curve motion* on page 115.

3. Enter a parameter for **Fast Deceleration Rate** 

.
This parameter is activated by the Fast Stop, Software Overtravel, or Torch Collision inputs and specifies in mGs how quickly the system will stop when the Fast Stop input is active. Generally, this value is much higher than the acceleration rate.

1 mG is approximately 0.386 inches per second^2.

Maximum machine speed

Enter a parameter for the **Maximum Machine Speed** 

. This parameter limits the range of all subsequent speed entries. In addition, it scales the speedometer in the Watch Window.

Calculate the maximum machine speed using the following formula.

$$\frac{\underset{\text{(max. encoder counts per ms)}}{32{,}767} \times \underset{\text{(ms per minute)}}{60{,}000}}{\text{Encoder counts per mm (inch)}} = \text{Maximum machine speed per minute}$$

The maximum encoder counts per ms in Phoenix is fixed at 32,767.

For more information about calculating encoder counts per mm (inch), see *step 3* on page 106.

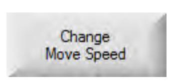
For example, assume the encoder counts per inch are 1,000,000.

$$\frac{32{,}767 \times 60{,}000}{1{,}000{,}000} = 1{,}966 \text{ inches per minute}$$

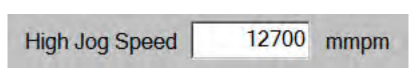
You would enter 1,966 in the Maximum Machine Speed parameter.

To use a higher machine speed, you must reduce the encoder counts per mm (inch) by reducing the position scaling in the drive.

Set the speed for manual moves

Choose the **Change Move Speed** soft key 

to cycle through the 4 move speeds that are available: Maximum Machine Speed, High Jog Speed, Medium Jog Speed, and Low Jog Speed.

1. Enter a parameter for the **High Jog Speed** 

. This parameter defines the maximum speed you can select for jog speed.

2. Enter a parameter for the **Medium Jog Speed** 

. This parameter defines the medium speed you can select for manual motion.

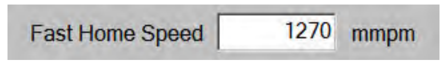
3. Enter a parameter for the **Low Jog Speed** 

. This parameter defines the lowest speed you can select for manual motion.

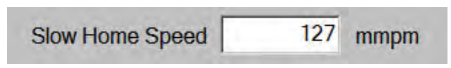
4. Select **Yes** or **No** for **Limited Speed If Not Homed** 

. If you select **Yes**, and the machine has not been homed, machine motion is limited to the value entered for **Limited Machine Speed**. See *Limited speeds*.

Home speeds

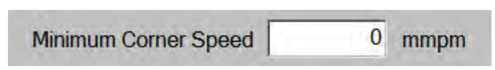
1. Enter a parameter for the **Fast Home Speed** 

. This parameter defines the speed that the CNC uses during the first phase of the homing sequence. During the first phase, the master station moves toward the home limit switches at the Fast Home speed and decelerates to a stop when the home input is activated.

2. Enter a parameter for the **Slow Home Speed** 

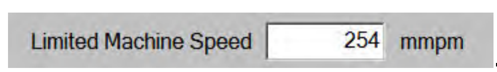
. This parameter defines the speed that the CNC uses during the second phase of the homing sequence. During the second phase, the master station reverses direction and moves at the slow home speed until the home input turns off. If marker pulse is used, motion continues until marker pulse occurs (or the home input turns off). Then the axis position is reset to the Absolute Home position and motion continues at the slow home speed, and the master station travels the Home Offset distance and comes to a stop. The position when homing is complete is the home position for that axis.

Corner speed

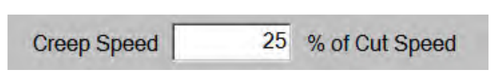
Enter a parameter for the **Minimum Corner Speed** 

. This parameter defines the minimum speed to use when negotiating corners. Corner speed is set by the centripetal limiting algorithms that use the program speed and the acceleration rate set for this speed range. For normal operations, this parameter should be set to zero to let the centripetal limiting algorithm determine the speed based on the acceleration rate.

Limited speeds

1. Enter a parameter for the **Limited Machine Speed** 

. This parameter defines the maximum machine speed when the **Limited Speed** input is active. This is commonly used to set a reduced machine speed for increased safety during machine testing or maintenance. If this value is set to 0, no motion will occur before homing (if homing is configured), and no motion will occur if the Limited Speed input is on.

2. Enter a parameter for the **Creep Speed** 

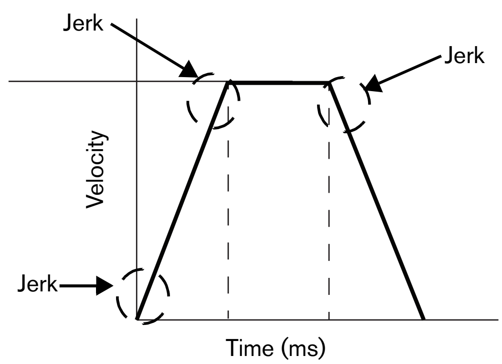
. The Creep Speed percentage defines what percentage of the machine cut speed is used when creeping. Generally, creep speed is 25% of the cut speed.

Trapezoidal motion compared to S-curve motion

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

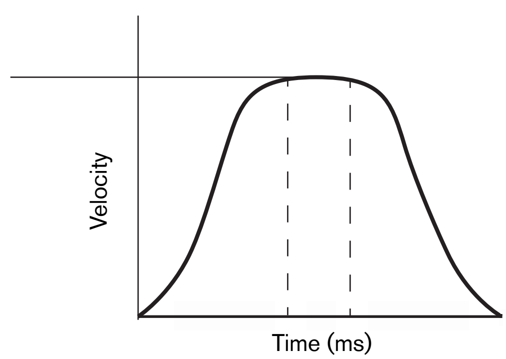
S-curve motion is a feature of Phoenix that allows smoother motion during acceleration than traditional, or trapezoidal, motion.

Figure 25 – Trapezoidal motion



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

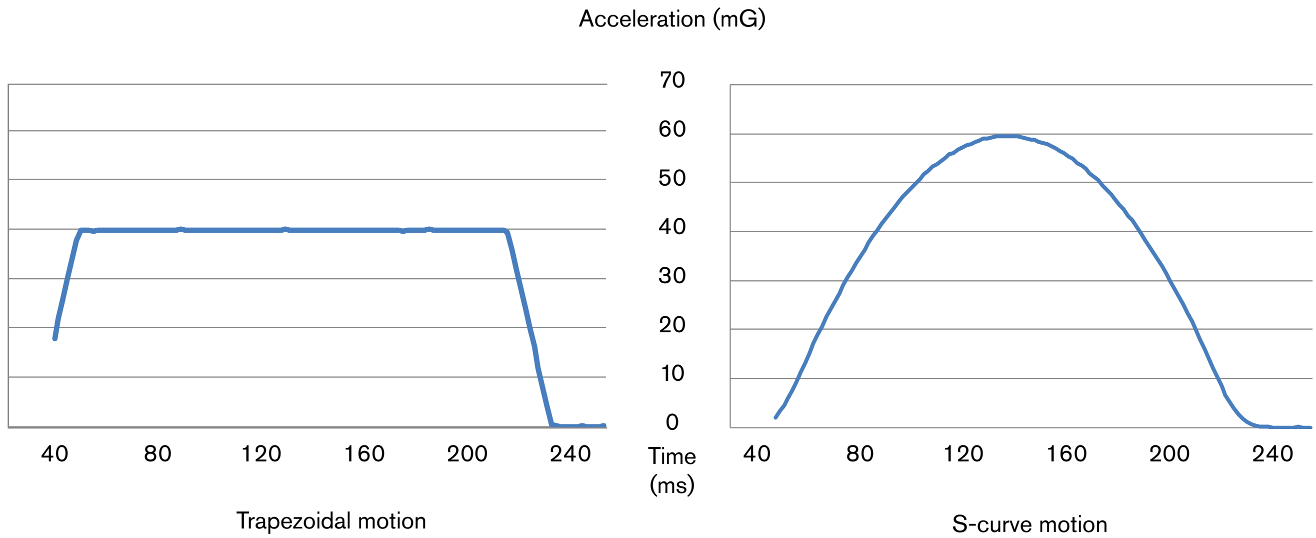
Figure 26 – S-curve motion



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

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

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

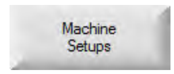


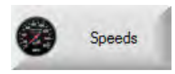
With a higher peak acceleration, S-curve allows the table to achieve the desired cut speed, with reduced jerk, in the same amount of time as trapezoidal motion.

S-curve setup

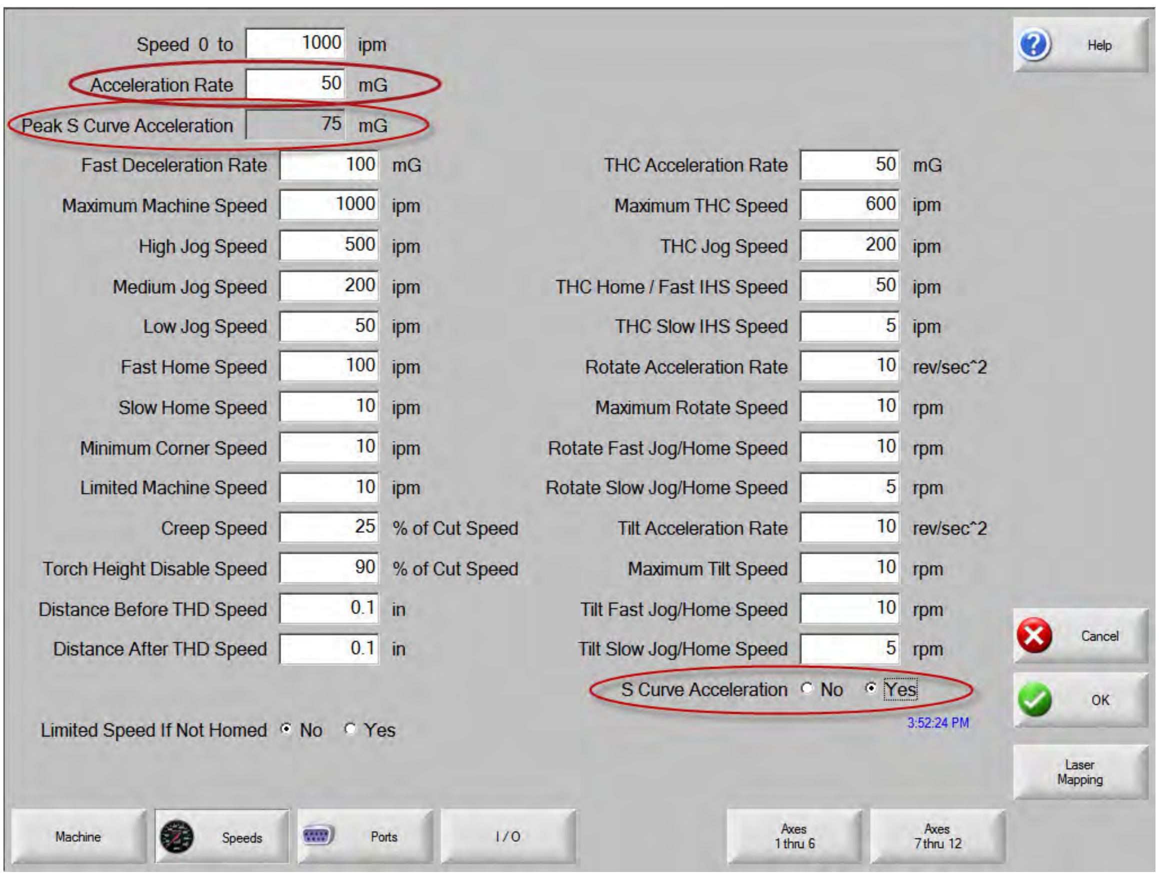
⚠	CAUTION!

Verify that the cutting table is capable of a 50% increase in acceleration before s-curve is enabled to avoid damage to mechanical components on the table.

S-curve acceleration is enabled with a single parameter on the 

> 

screen (**S Curve Acceleration**).



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

Change the cutting speed

The cut speed field on a cut chart screen sets the cutting speed (also known as the feed rate) for the material type and material thickness.

- If you change the default cut speed for a provided cut chart (like an HPR process) the new value will be shown in blue.
  - Double-tap on the Speed field. An on-screen keyboard shows, and you can enter a new speed value.
- If speed pots are not enabled, the **Increase Speed** and **Decrease Speed** soft keys show on the cutting screen.
  - Choose the **Increase Speed** soft key to increase the current cut speed by 3%.
  - Choose the **Decrease Speed** soft key to decrease the current cut speed by 3%.
- Keyboard operation: To change the current cut speed while cutting a part, press the Enter key once to highlight the current cut speed. Then enter the new cut speed, and press Enter again.

Cutting speed and acceleration affects cut quality

Here are some troubleshooting tips for cutting speed.

- High-speed dross occurs when the cutting speed is too fast and the arc lags behind.
  - Reduce the cutting speed.
- Low-speed dross occurs when the cutting speed is too slow and the arc shoots ahead.
  - Increase the cutting speed.
- Top spatter occurs when the cutting speed is too fast.
  - Reduce the cutting speed.
- Low-speed dross that occurs at the beginning or end of a cut segment can be caused by acceleration levels being too low.
  - Increase the acceleration rate.

7

I/O – Inputs and Outputs

Introduction

Phoenix supports up to 512 digital inputs and 512 digital outputs, and 32 analog inputs and 32 analog outputs.

General purpose I/O is 1 – 512. Station I/O is automatically assigned when you configure a station in the Station Configuration screen, and starts at 129 and above.

Use the plan you have created for your cutting system to assign I/O. See *Assign I/O* on page 122.

Types of I/O

Definitions for some of the most common I/O are available at the end of this section. See *Digital I/O definitions* on page 127.

Fixed function I/O (virtual)

Phoenix assigns fixed function I/O automatically. Fixed function I/O for an HPRXD plasma power supply is assigned when you choose an HPRXD as the cutting tool on the Station Configuration screen. Generally, fixed function I/O for the HPRXD is not shown as assigned I/O on the Digital I/O setup screen and does not appear on the I/O Diagnostics screen. But you can see the fixed function I/O that assigns a station for the Soft Op Con on the Digital I/O setup screen and the I/O Diagnostics screen. This I/O includes Auto Select 1 (2, 3, 4), Manual Select1 (2, 3, 4), Station Enable LED 1 (2, 3, 4), and Station Enable 1 (2, 3, 4).

How to view fixed function I/O

- To see the fixed function I/O that defines a station for the Soft Op Con on the Diagnostics I/O screen, choose **Setups>Diagnostics>IO**.
  - They start at 128. The HPRXD fixed function I/O is not shown.
- To see the fixed function I/O that defines a station for the Soft Op Con on the Digital I/O screen, choose **Setups>Machine Setups>Digital IO**.
  - They start at 128. The HPRXD fixed function I/O is not shown.
- To see both the fixed function I/O that defines a station for the Soft Op Con and the HPRXD fixed function I/O in the HPR Supply Watch Window or in the Oscilloscope, set up a Watch Window or choose **Setups>Diagnostics>Oscilloscope**.
  - The I/O start at 512.

Fixed function I/O for HPRXD plasma power supplies

When you have multiple HPRXDs the I/O will be numbered (HPR Cut Sense 1, HPR Cut Sense 2, and so on.)

Digital inputs
HPR Cut Sense
HPR Not Ready For Start
HPR Nozzle Contact Sense
HPR Process Ready

Digital outputs
HPR Cut Control
HPR Hold Ignition
HPR Pierce Control
HPR Nozzle Contact Enable
HPR Remote On

There is also a fixed function **analog** input: HPR Arc Voltage. This input is automatically assigned when you set up a Sensor THC on the Machine Setups screen (**Setups** > **Password** > **Machine Setups**).

Fixed function I/O for the EDGE Connect TC hardware operator console

Digital inputs
Op-Panel Joystick Up
Op-Panel Joystick Down
Op-Panel Joystick Left
Op-Panel Joystick Right
Op-Panel Forward On Path
Op-Panel Backward On Path
Op-Panel Start
Op-Panel Stop
Op-Panel Raise Tool
Op-Panel Lower Tool
Op-Panel DIP Option 1 (not used)
Op-Panel DIP Option 2 (not used)
Op-Panel DIP Option 3 (not used)
Op-Panel DIP Option 4 (not used)

Analog inputs
Op-Panel Cut Speed Pot
Op-Panel Jog Speed Pot

General purpose I/O

Use general purpose I/O for any cutting process to control cutting machine hardware such as overtravel switches, and to control a single cutting station.

Numbered I/O

Use numbered I/O to identify stations when the cutting system uses an operator console (with either a single torch or multiple torches) **or** has multiple stations. For example, assign the outputs Station Enable 1 and Cut Control 1 to station 1, Station Enable 2 and Cut Control 2 to station 2. The input and output names only appear after you have tools assigned in the Station Configuration screen. See *Station Setup* on page 155.

Assign I/O

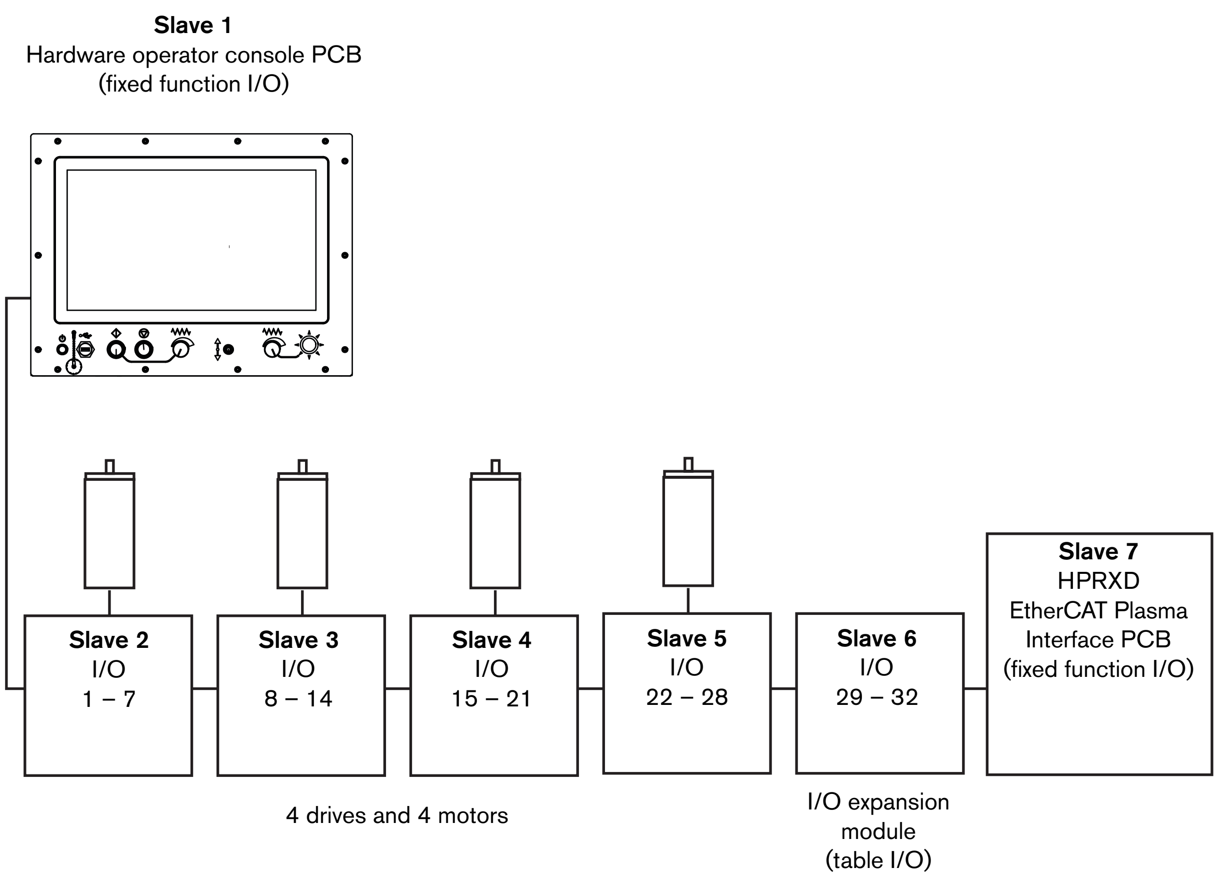
How Phoenix assigns I/O

An EtherCAT network can be configured with multiple slave devices that have various numbers of I/O available. Phoenix assigns I/O in order from 1 – 512, starting with the first slave device connected on the network. When you use the EDGE Connect TC, the PCB for the factory-installed hardware operator console is slave 1 on the EtherCAT network and uses virtual I/O. When you use the EDGE Connect CNC, slave 1 is the first device connected on the EtherCAT network. The HPRXD EtherCAT plasma interface PCB must be the last slave on the network and it uses fixed function I/O.

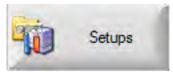
The following example includes:

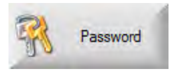
- Four drives, each with 7 digital inputs, 2 digital outputs, 1analog input, and 1 analog output
- One I/O module that has 4 digital inputs
- One HPRXD plasma power supply with a voltage divider card (VDC3)

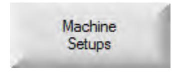
I/O assignment starts with the first drive physically connected to the network and continues from there.



Assign the digital I/O

1. From the Main screen, choose 

 > 

 > 

 > 

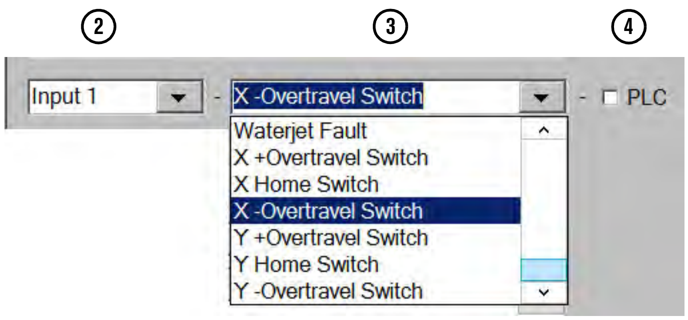
.

   Use a mouse to make it easier to scroll through the list of I/O.

   The Inputs window on the Digital I/O screen shows the inputs in groups of 32.

2. Select the input number from the list on the left.

3. Select the name of the input function from the list on the right. Select **Spare** if the name of the function is not shown in the list.



4. To assign the input for use with the PLC Connect programmable logic control, select the **PLC** box to the right of the input name selection window.

5. Select the output number from the list on the left.

6. Select the name of the output function from the list on the right.

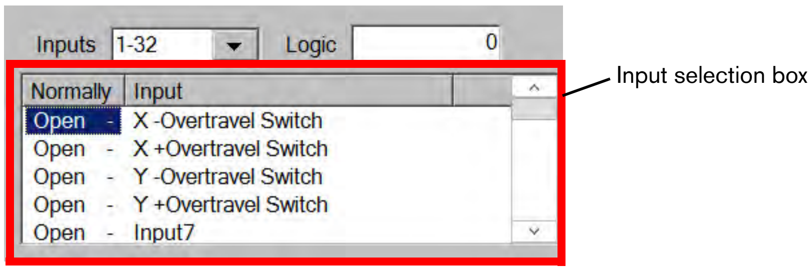
7. To assign the output for use with PLC Connect, select the **PLC** box to the right of the output name selection window.

8. Continue this process until you have assigned all of the digital I/O needed for the cutting system.

Logic

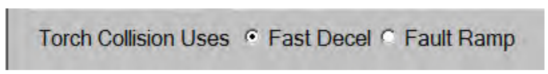
The value in this field represents I/O in the range selected that are set to Normally Closed. When Input Logic is set to zero (0), an active input is shown with a green lamp in the Diagnostics Inputs screen. The same process is used for output logic.

- Select an input in the input selection box and press the space key, or double-click to select the logic state for an input.



Torch Collision uses

Select the type of response that you want when the **Torch Collision** input is active. You can select a **Fast Decel** value or the **Fault Ramp** time that has been selected for the individual axis.



Drive Enables

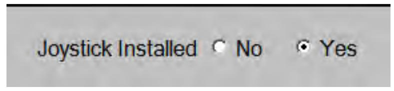
The **Drive Enables** selection determines how the CNC responds to drive faults. If you select **Independent**, each drive is disabled as soon as its axis fault ramp time is finished. If you select **Series**, all axes must complete their individual fault ramp times before the drives are disabled.

Initial Feedback Delay

Initial Feedback Delay allows the drive system to become fully enabled before the CNC enables the digital drives. The typical setting for digital drives is 0.

Joystick Installed

The joystick for the EDGE Connect TC is automatically enabled. If you are adding your own operator console you must enable the joystick on the Digital I/O screen as shown below.

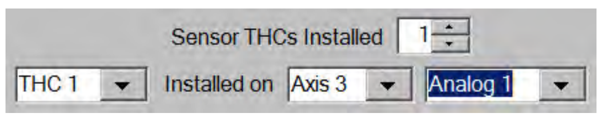


Analog I/O

If each drive in the network supports 1 analog input, then analog input 1 is on drive 1, analog input 2 is on drive 2, and so on. An I/O module follows in physical order after the drives (but before the plasma power supply). If there are 4 drives on the network, for example, then the I/O module analog inputs start at 5.

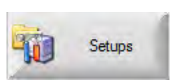
Arc voltage

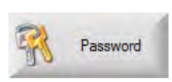
If you are using an HPRXD, Arc Voltage is automatically available and assigned as fixed on the machine screen. If you are using a Manual Gas Plasma system you need to assign Arc Voltage to an analog input on the Machine Setups screen along with the Sensor THC axis assignment.

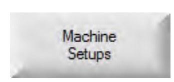


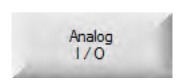
Enable speed pots

The EDGE Connect TC has a factory-installed hardware operator console with 2 speed pots, which are automatically enabled. If you have an EDGE Connect CNC to which you are adding your own hardware operator console, you must enable the speed pots on the Analog I/O screen and assign them to analog inputs.

1. From the Main screen, choose 

 > 

 > 

 > 

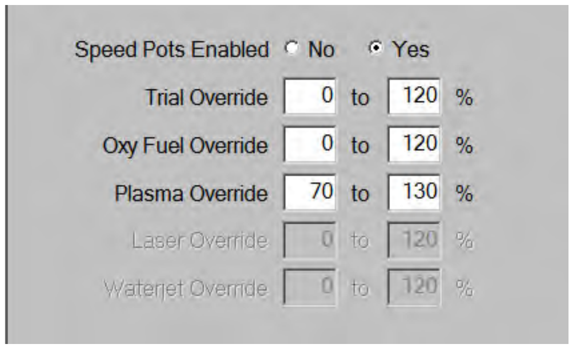
.
2. Select **Yes** to enable the speed pots. See *Figure 27* on page 126.

Also, you must enable speed pots to use the sliding speed control on the Soft Op Con.

Speed overrides

The speed override ranges control the speed pot setting when the knob is turned completely counterclockwise or clockwise. Instead of 0% (fully counterclockwise) to 100% (fully clockwise), the speed pot range can be set at 70% of the programmed cut speed (fully counterclockwise) to 130% (fully clockwise) of the programmed cut speed.

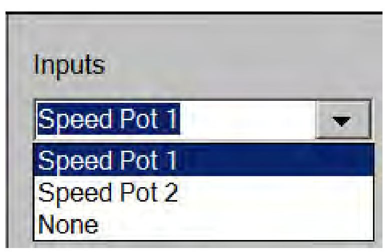
Figure 27



Assign analog inputs

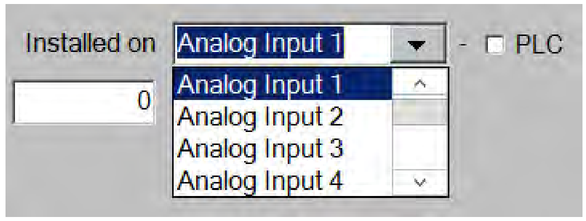
1. From the Main screen, choose Setups > Password > Machine Setups > Analog I/O.
2. Select the input name from the **Inputs** list.

If you have an EDGE Connect TC, analog 1 and analog 2 inputs are not available for use. They are automatically assigned to the speed pots in the hardware operator console.



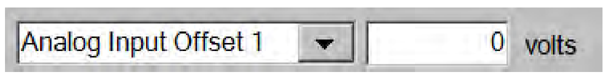
3. Select the input number from the **Installed on** list.

To assign the analog I/O for use with PLC Connect, select the **PLC** box to the right of the I/O name selection window.



Analog Input Offset 1 – 32

The Analog Input Offset corrects for any imbalance in or zeroes the incoming analog voltage to the CNC. The CNC constantly applies this offset to the incoming analog signal. The offset is subtracted from the incoming voltage and the CNC shows the voltage after this offset is subtracted.



Analog outputs

Phoenix provides analog outputs for controlling gas regulators in an oxyfuel cutting machine. Select the gas channel and assign an analog output and maximum pressure for it. The channels are listed in groups. Each channel in each group should be selected and matched with an analog output. Then Phoenix transfers these channels and pressures to the Oxyfuel Process screen.

Digital I/O definitions

Digital inputs

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

Cut Select – This input turns on the Cut Control and/or Cut Control outputs 1 – 20 while the switch remains in the ON position. The cut select input is primarily used with an oxyfuel hardware operator console to activate the cut and preheat channels, so the operator can adjust the regulators. However, the input will turn on the Cut Control and/or Cut Control outputs 1 – 20 for any cutting tool on any enabled station. Cut Select can also be used to test plasma torches by allowing the operator to bypass the CNC to fire the torch. It is the OEM's responsibility to wire this input correctly. Hypertherm recommends using a momentary switch so that an operator doesn't leave the input on.

Cut/Mark Sense 1 through 4 – Verifies that the plasma torch or marker tool has established the arc for the selected process. This indicates to the CNC that motion can begin.

Cut Sense – Indicates that the plasma arc has transferred to the work piece. If the Arc On Feedback setup parameter is ON, machine motion begins at the activation of this input.

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

Enable Plasma 1 through 8 – Formerly used for tracking consumable data for the specified plasma torch. The Station Select input now supports this input.

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

Fume Extraction Sense – Confirms that an external fume extraction system on the cutting table is operational before beginning the cut. An option to override appears if the input has not been satisfied at the time of the cut.

Use this input **only** when an external device, not the CNC, controls the fume extraction system. If the CNC controls the fume extraction system, use the Fume Extraction Control output instead. See page 132 for more information on that output. Do **not** use the Fume Extraction Sense input and the Fume Extraction Control output at the same time.

High Preheat Select – This input turns on the Preheat Control and/or the High Preheat Control outputs 1 – 20. With this input ON, high pressure fuel gas will be delivered to the torch and can be manually adjusted.

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

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

Low Preheat Select – This input turns on the Low Preheat output and/or the Low Preheat Control outputs 1 – 20 while the input remains in the ON position. With this input on, low pressure fuel gas will be delivered to the torch and can be manually adjusted.

Lower Torch / Lower All Torches – This input can be a toggle switch that turns on the Torch Down output. The Torch Down output signals the oxyfuel torch lifter to lower the torch.

Manual Ignition Select – This input turns on the Ignition output and/or the Torch Ignition outputs 1 – 20. The Ignition output remains on while the input is ON. This input can be used to manually turn on the igniters.

Marker Select 1 and 2 – Select which marker process logic is used by the CNC. These inputs are generally run by external switches.

Nozzle Contact Sense – Used during Sensor THC initial height sense (IHS) to detect the location of the cut surface. This input is returned to the CNC through the external voltage divider card.

Pierce Complete – Plasma Select 1 and 2 – Select which plasma process logic is used by the CNC. These inputs are generally run by external switches.

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

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

Remote Pause – Stops all CNC motion and shows the Pause screen. No motion is allowed until this input deactivates.

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

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

Rotate Home – Indicates that the machine has traveled to its Rotate axis home position. When the Rotate axis is homed, it moves in the specified home direction at the Fast Home Speed until the input activates. Then the Rotate axis decelerates to a stop, and then moves in the opposite direction at the Slow Home Speed until the switch deactivates.

Sensor THC Enable – Formerly indicated which Sensor THCs are active when multiple Sensor THCs are commanded by the CNC as separate axes. The Station Select input now supports this input.

Spare – Activates through the part program. If a spare input is located in the part program, the CNC pauses the process until the input state is detected. The spare inputs can be implemented with specific EIA "O" and "M" codes that indicate the input number and function. For more information, refer to the *EDGE® Connect Programmer Reference* (809550).

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

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

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

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

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

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

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

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

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

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

X Home – Indicates that the machine has traveled to its X axis home position. If the X axis Use Home Limit Switch parameter is set to Yes and a homing function is performed, the X axis moves in the specified home direction at the fast home speed until the input activates. The

X axis then decelerates to a stop and moves in the opposite direction at the slow home speed until the switch deactivates. After the switch deactivates, the X axis decelerates to a stop or, if the Use Marker Pulse parameter is set to Yes, continues until the encoder marker pulse is detected.

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

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

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

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

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

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

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

Digital outputs

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

Cut Control – Starts the plasma system in plasma mode, or enables the cutting oxygen in oxyfuel mode. This output can also be used to activate a marking device.

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

Error During Program – Activates when an Error or Fault occurs that pauses the machine or the part program. This output can be tied to an external indicator light or audible alarm to tell the operator when the cutting system is paused due to an Error or Fault condition.

Fume Extraction Control – When you assign this output, a Fume Extraction Delay timer appears on the I/O screen. The timer allows the fume extraction equipment to run for a period of time at the end of the part program when the CNC shows the Main screen. This output turns ON at the start of a cut, at the start of a rip cut, or when resuming a cut after pausing the program. Assuming the starting and ending positions in the part program are the same, this output turns OFF when the timer expires at the end of the part program. If the starting and ending positions in the part program are different, then this output turns OFF when the timer expires after the operator acknowledges the Ready to Start? prompt. This output also turns OFF when the operator pauses the part program.

> Use this output **only** when the CNC, not an external device, controls the fume extraction system. If an external device controls the fume extraction system, use the Fume Extraction Sense input instead. See page 128 for more information on that input. Do **not** use the Fume Extraction Control output and the Fume Extraction Sense input at the same time.

Hold Ignition – Part of the Sensor THC operation to hold ignition of the plasma power supply. This output can be tied back to the CNC through the external voltage divider card or direct to the plasma power supply depending on configuration.

Key Press Indicator – Activates when a key is pressed on the CNC.

Marker Control – Activates the marker tool during the marking process. The marker tool activates through the combination of Marker Enable and Cut Control outputs. Options are shown and can be configured on the Marker Setup screen for the desired process timing and operational results.

Marker / Marker Enable – Activates an external marking device. It can only be activated by the appropriate M code in the part program. For more information, refer to the *EDGE® Connect Programmer Reference* (809550).

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

Motion Indicator – Activates when the CNC is commanding machine motion.

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

Peck Drill Cycle – Activates with an M94 code in the part program. It activates an input in a PLC that is controlling a tool cycle.

Pierce Control – Used by the plasma cut logic to send an output to the torch during the pierce. This output is on for the duration of the Pierce Time selected on the Plasma Process screen.

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

Plasma Select – Activates when the CNC is in the plasma cutting mode.

Program Running – Active anytime the CNC is operating within a part program.

Spare – Activates through the part program. If a spare output is located in the part program, the CNC turns ON the output as directed. The Spare outputs can be implemented with specific EIA W and M codes that indicate the output number and function. For more information, refer to the *EDGE® Connect Programmer Reference* (809550).

Station Clamp 1 – 19 – Part of the Automated Torch Spacing feature. The Station Clamp is used to clamp the selected torch station to the Transverse axis for standard cutting.

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

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

Station Mirror 1 – 19 – Part of the Automated Torch Spacing feature. The station mirror is used to clamp the selected torch station to the Transverse axis for mirrored cutting.

Tap Cycle – Activates with an M95 code in the part program. It activates an input in a PLC that is controlling a tool cycle.

Tool Change – Activates with an M96 code in the part program. It activates an input in a PLC that is controlling a tool change.

Torch Down – Lowers the cutting torch. In plasma mode, this activates for the Torch Down Time in plasma mode.

Torch Height Disable – Disables the automatic torch height control in plasma mode. Torch Height Disabled is activated when the cut speed drops to the Torch Height Disabled Speed and remains active until the cut speed is greater than the Torch Height Disabled Speed. Torch Height Disabled remains active for a specified distance even when the cutting speed drops below the Torch Height Disable speed if a value has been entered for Distance Before THD Speed on the Speeds screen in Phoenix. Torch Height Disabled also remains active when the cutting speed is greater than the Torch Height Disabled Speed if a value has been entered for Distance After THD Speed on the Speeds screen in Phoenix. For more information see *Set up Torch Height Disable* on page 138.

Torch Up – Raises the cutting torch. In plasma mode, this activates for the Torch Up Time.

8 Torch Height Control (THC)

Sensor THC

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

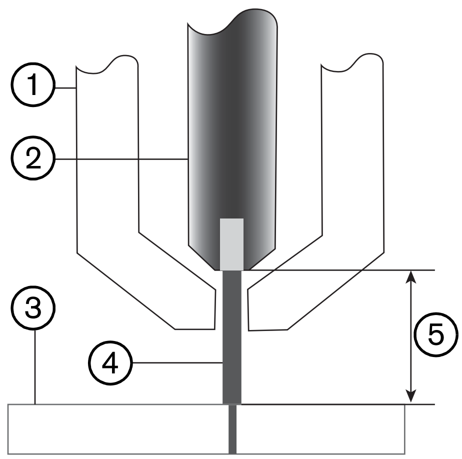
The settings provided in the following sections apply to the Sensor THC lifter and to drives supplied by table manufacturers and system integrators. The settings can often be used for lifters from other manufacturers.

About plasma torch height control

A circuit board inside the plasma power supply (the VDC3 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 (-10 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 Workpiece

4 Plasma arc

5 Voltage drop is measured over the plasma arc between the electrode and the workpiece

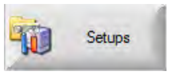
Sensor THC setup

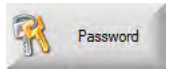
Set up the Sensor THC before you select a cutting tool in the Station Configuration screen or you will not see Sensor THC as a selection when assigning the cutting tool.

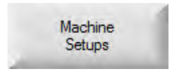
Setting up the Sensor THC on the CNC requires the following:

- Choose the number of Sensor THCs installed on the cutting system, and assign each THC to an axis.
- Set speeds and acceleration values for the Sensor THC axis.
- Enter encoder information, servo error tolerance, stall force tolerance, and other information about the THC axis.
- Select the Sensor THC as a lifter on a station.

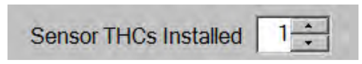
Assign the Sensor THC to an axis

1. Choose 

 > 

 > 

.

2. Choose the number of Sensor THCs installed 

.

3. For each Sensor THC:

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

.

   b. Choose the axis for the THC 

.

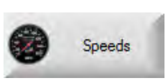
   The THC can be assigned to Axis 3 and higher. The CNC uses Axes 1 and 2 for Transverse or Rail. Dual Gantry is assigned to Axis 3 if one is used. If Dual Gantry is not being used, then the Sensor THC can be assigned to Axis 3.

   If you assign an HPR in the Station Configuration screen, the input for arc voltage (shown as Analog 1 above) will automatically change to Fixed. If you are using a generic plasma cutting tool, you will need to assign an analog input for arc voltage.

Set speeds and acceleration

After you select the Sensor THC on the Machine screen, set the speed and acceleration settings.

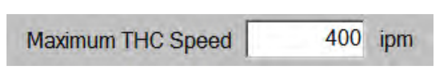
The values shown are recommended initial settings. You may need to change these values based on the mechanics of your cutting system.

1. Choose 

.

2. Enter the **THC Acceleration Rate** 

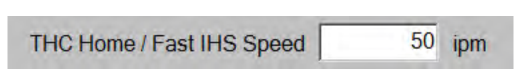
.

3. Enter the **Maximum THC Speed** 

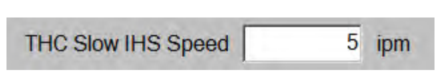
 (10,160 mmpm).

4. Enter the **THC Jog Speed** 

 (5,080 mmpm).

5. Enter the **THC Home / Fast IHS Speed** 

 (1,270 mmpm).

6. Enter the **THC Slow IHS Speed** 

 (127 mmpm).

Set up Torch Height Disable

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

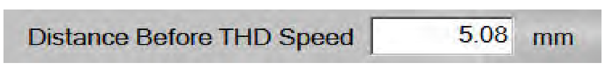
The CNC activates the Torch Height Disable output based on the values of the following parameters:

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

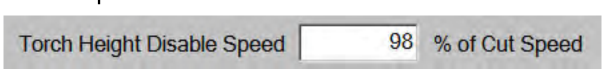
When any one of these conditions is met, the Torch Height Disable feature is implemented. The definition of a corner is based on the tangent angle in the Special Setups screen and the value of the Torch Height Disable Speed.

1. Choose 

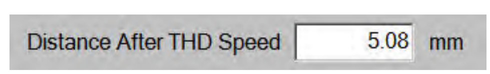
.

2. Enter a parameter for the **Torch Height Disable Speed** 

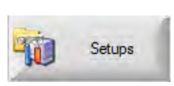
. Specifies the percentage of the program cut speed at which the CNC disables the height control. For example, when the cutting system is slowed to cut a part feature such as a corner, the arc voltage increases, which causes the torch to lower toward the workpiece. When the cutting system slows down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

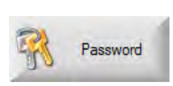
3. Enter a parameter for the **Distance Before THD Speed** 

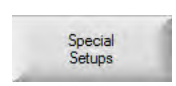
. This distance is active whenever the program cut speed drops below the Torch Height Disable Speed, or segments intersect at an angle greater than the Tangent Angle. Specifies the distance before turning a corner at which the CNCs disables the torch height control.

4. Enter a parameter for the **Distance After THD Speed** 

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

5. To set the Tangent Angle, choose 

> 

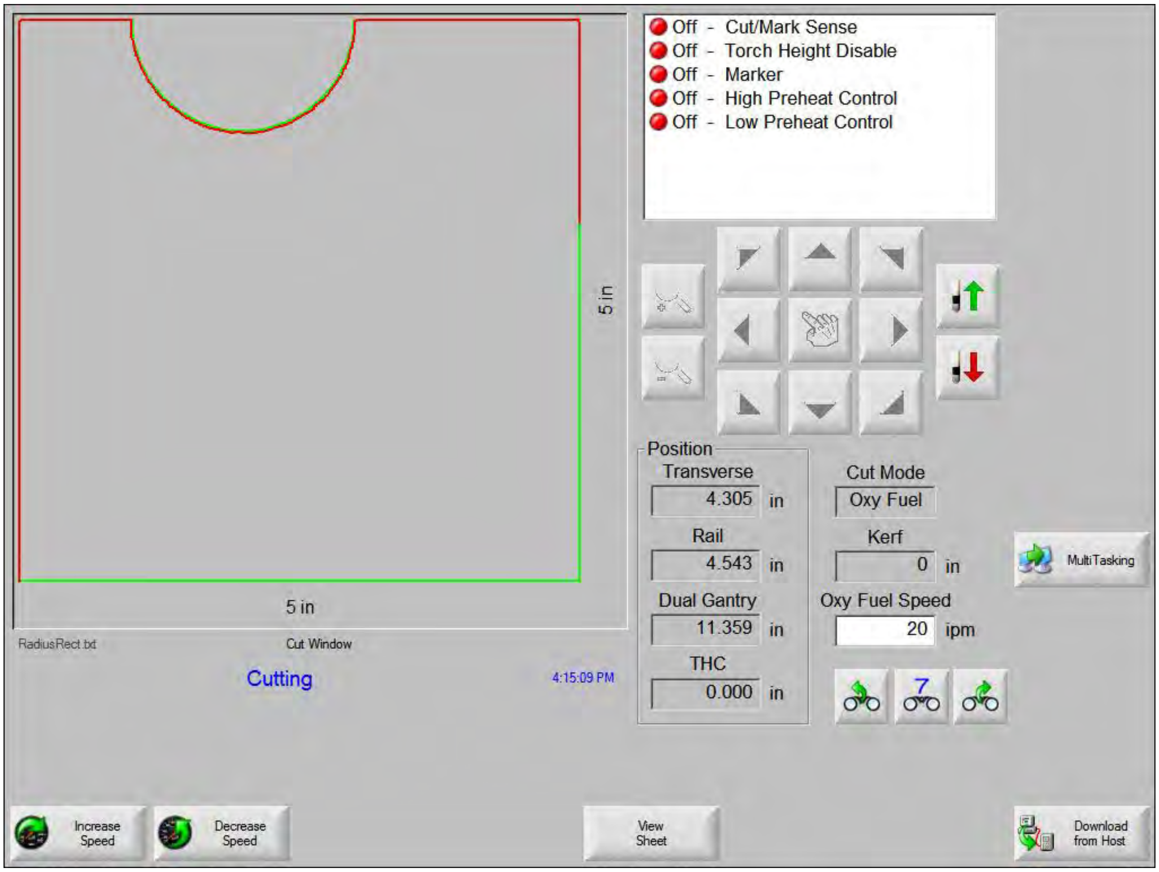
> 

.

6. Enter a parameter for the **Tangent Angle** 

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

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

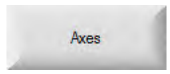


Parameter	Value
Torch Height Disable Speed Percentage value	90%
Distance Before THC Speed value	12.7 mm (0.5 inches)
Distance After THC Speed value	12.7 mm (0.5 inches)
Tangent Angle Value	20 degrees

For these values, the Torch Height Disable output is active 12.7 mm (0.5 inches) before and 12.7 mm (0.5 inches) after each 90 degree corner. This output is also active as the cutting tool accelerates after a pierce. It remains active until the cut speed is greater than the Torch Height

Disable Speed. Finally, because the arc intersects at greater than the 20-degree tangent angle, the Torch Height Disable output is activated for 12.7 mm (0.5 inches) before and after the beginning of the arc, and is activated for 12.7 mm (0.5 inches) before and after the end of the arc.

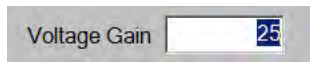
Define the THC axis

On the Machine Setups screen, choose 

> 

.

Voltage Gain



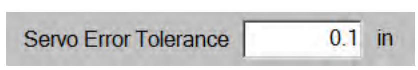
This gain is used when the THC operates with a closed-loop arc voltage control. Adjust this value after you find the appropriate value for Speed Gain.

If this value is set too high, the lifter position during closed-loop arc voltage control becomes unstable and prone to oscillation. If this value is too low, the arc voltage control can become slow and inaccurate. Test voltage gain by repeatedly performing a test cut under closed-loop arc voltage control, and make sure that the THC quickly and accurately reaches the set arc voltage.

To optimize this gain, raise this value until there is a very slight oscillation during a cut, and then reduce the setting by 1 or 2.

The voltage gain range for Normal mode is 0 to 500 volts.

Servo Error Tolerance

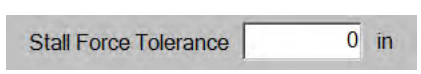


Servo error, also called following error, is the difference between the commanded motor position and the actual motor position as calculated by the drive. The servo error tolerance is the upper limit of the amount of following error allowed before the CNC faults.

The amount of servo error tolerance depends on the cutting machine mechanics. Setting the servo error tolerance too low could cause the CNC to fault repeatedly. Setting it too high may allow inaccurate motion or result in mechanical damage. Set the servo error tolerance to a value higher than the steady-state following error which is reported by the drive.

Range: 0 mm to 127 mm (5 inches)

Stall Force Tolerance

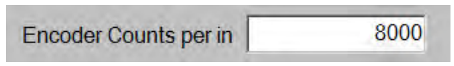


Stall force is a method of detecting the workpiece during initial height sense. Use stall force when cutting underwater or in other conditions in which ohmic contact is not feasible. Stall force tolerance is the maximum amount of following error on the THC axis. The CNC starts monitoring the THC axis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

In general, stall force tolerance should be set to twice the following error observed during Fast IHS speed.

Range: 0 mm to 127 mm (5 inches)

Encoder Counts per mm (inch)



Encoder counts are a position scaling factor used by Phoenix. Refer to the drive manufacturer documentation for specific scaling information required by the THC drive.

In general, to determine encoder counts per mm (inch) for a THC drive, you need to know the following:

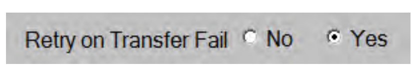
- Counts per revolution of the motor
- Gear ratio
- Distance of travel in one revolution of the lead screw

An example of the equation you use is shown below:

$$\frac{\text{65,536 counts}}{\text{1 motor revolution}} \times \frac{\text{1 lead screw revolution}}{\text{5 mm (0.197 in.)}} = \text{13,107.20 (332,670.05) encoder counts per mm (inch)}$$

Range: Positive, non-zero values up to 39,379.08 counts/mm (1,000,000 counts/inch)

Retry on Transfer Fail



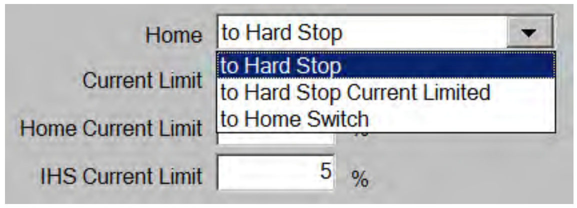
When IHS completes, the torch attempts ignition, the CNC starts a 10-second timer, waits for the Cut/Mark Sense input (or Cut Sense # input) to turn on, and shows the status message Waiting For Arc On. If the input does not activate within 10 seconds, the ignition process stops, the torch retracts, and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

Slide Length

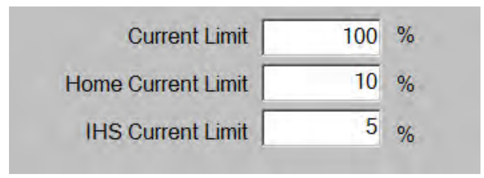
The slide length is the amount of travel of the lifter. The CNC uses the slide length when changing speeds during the IHS. The CNC subtracts 5.08 mm (0.2 inches) from the slide length to make sure the lifter does not contact a hard stop when homing.

Home setting for current-type drives

1. Choose the method you want to use to home the torch from the list.



2. Set the **Current Limit %** to the maximum current output for the operation of the lifter motor.
   Range: 0.001 to 100



3. Set the **Home Current Limit %** to the maximum current during the homing sequence to stall and detect the top of the slide. Set Homing Current Limit to 50% of drive capacity.
   Range: 0.001 to 100

4. Set the **IHS Current Limit %** to the maximum current during IHS to stall and detect the workpiece.
   Range: 0.001 to 100

See *Home the axis* on page 143 for more information.

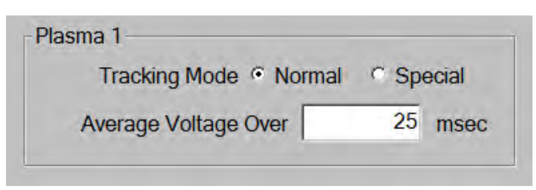
Tracking Mode

The CNC uses voltage tracking to lock on to the actual arc voltage and compare it to the Set Arc Voltage. You can choose individual tracking mode settings for Plasma 1, Plasma 2, Marker 1, and Marker 2. Do not use Special tracking mode with the Sensor THC. Special tracking mode is only used for extreme material tracking situations.

1. For almost all applications we strongly recommend that you set tracking mode to Normal. Voltage tracking in Normal mode locks on within 2 volts of the Set Arc Voltage value. Adjust voltage gain to increase lifter responsiveness. In Normal mode, arc voltage accuracy is +/- 1.0 arc volt.

   Recommended settings: For the Sensor THC, choose Plasma 1 or 2 as the process, then choose Normal for tracking mode. Set Average Voltage Over to 25 msec.

2. Use **Average Voltage Over** to set the arc voltage sampling rate. Phoenix records the arc voltage every millisecond and averages the arc voltage over the number of milliseconds entered. The average value is compared to the voltage set point and then the command to the lifter is calculated.

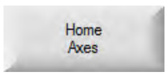


Home the axis

Manual

1. On the Main screen choose 

.

2. Choose 

.

3. Choose 

.

Automatic

If the **Home Torch Height Control** status message is enabled in Special Setups, then Phoenix automatically prompts you to home the THC axis when you turn ON the CNC and before motion begins.

Set the Sensor THC operating modes

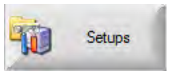
Automatic modes

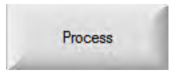
In all of the automatic modes, the THC performs an initial height sense (IHS), then retracts to the Transfer Height. After torch ignition, the plasma arc transfers to the workpiece, then the torch moves to the Pierce Height for the duration of the Pierce Time. During this pre-cutting sequence, the torch 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.

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 provide any automatic movement of the lifter. You can move the lifter only with the UP and DOWN soft keys on the EDGE Connect Soft Op Con, the toggle switch on the EDGE Connect TC hardware 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 you want to use as a cut height.

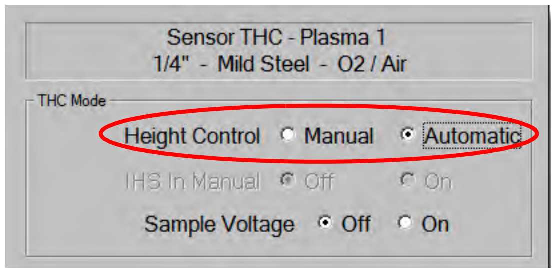
To set the mode:

1. Choose 

 > 

.

2. Select **Manual** or **Automatic** for the operating mode on the Plasma Process screen.



Sample Arc Voltage mode

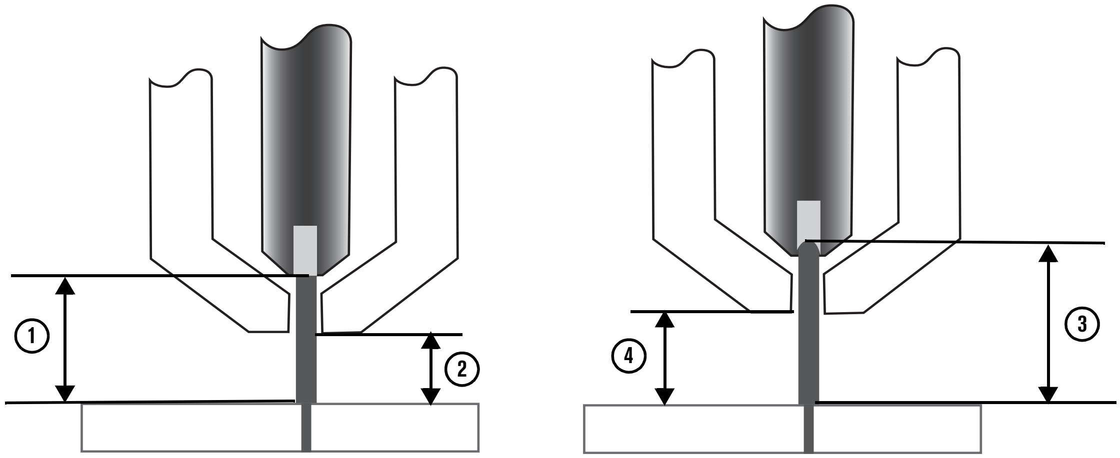
Use Sample Arc Voltage mode as much as possible to achieve a consistent cut quality over the life of the consumables. When cutting begins, the CNC takes several samples of the arc voltage and averages these samples. It then uses the sample average as the Set Arc Voltage instead of the value in the Process 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.

If the 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 a sample of 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.

Sample arc voltage should be used for “I” cutting and bevel cutting.

Figure 28



1 Plasma arc from a new electrode

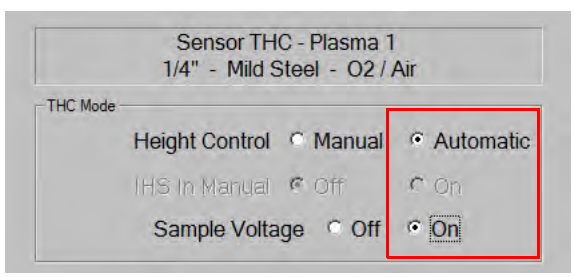
2 Cut height from cut chart

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. When not using Sample Arc Voltage, the torch moves closer and closer to the workpiece as the electrode wears.

To use Sample Arc Voltage mode:

- Make the following selections on the Plasma Process screen.

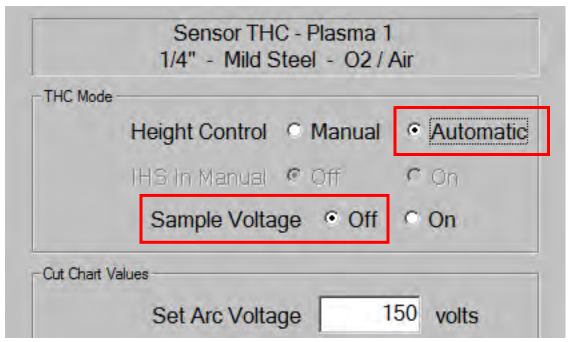


Set Arc Voltage mode

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.

To use Set Arc Voltage mode:

- On the Plasma Process screen, choose **Automatic** for **Height Control** and **Off** for **Sample Voltage** to use Set Arc Voltage mode.



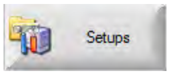
Methods for changing the Set Arc Voltage

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:

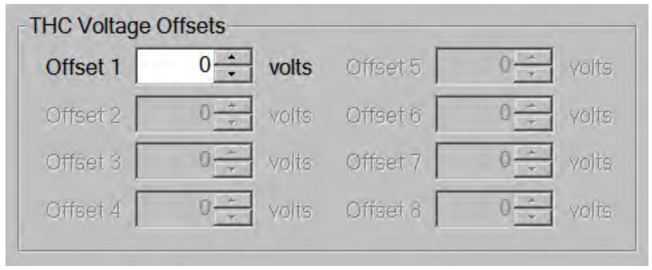
- Issue a G59 V600 Fvalue command in the part program for Plasma 1 where the value is the new Set Arc Voltage. (Use G59 V625 Fvalue to change the Set Arc Voltage for Plasma 2).
- Enter THC voltage offsets.
- When **Automatic** is selected for **Height Control**, press the **Increase Arc Voltage** or **Decrease Arc Voltage** soft keys on the Main screen while the system is cutting.
- When **Manual** is selected for **Height Control**, press the **Raise Manual Cut Height** or **Lower Manual Cut Height** 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.

Choose 

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 change the process, load a different cut chart, or change the consumables. **The THC voltage offset value will remain at a value unless you change it.**

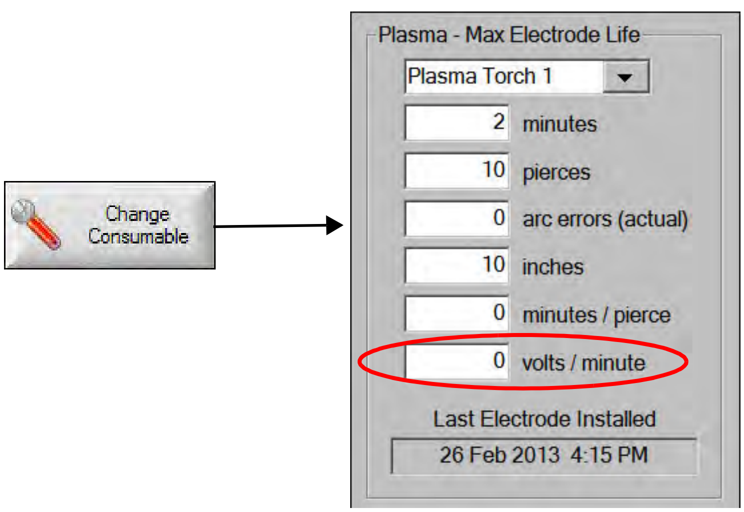
To reset a voltage offset to zero, choose Setup to open the Cutting screen and change the THC voltage offset

- When using Sample Arc Voltage mode, set the THC voltage offset to zero
- After you load new consumables, set the THC voltage offset to zero.

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.

Change offsets automatically for Sensor THC

For the Sensor THC, the THC voltage offsets can be changed automatically and continually by entering a value for the **Volts / Minute** parameter on the Change Consumable screen (on the Main screen, choose the **Change Consumable** soft key). The amount to enter depends on each system's consumable use history. For more information see *Change consumables* on page 275.



If you entered a value for Volts / Minute, when you change consumables, reset the THC Voltage Offset to zero to allow the CNC to gradually increase the offset using the Volts / 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 / Minute to 0.

Increase or decrease arc voltage in Automatic mode

After cutting begins in Automatic mode, the CNC shows **Increase Arc Voltage** and **Decrease Arc Voltage** soft keys on the Main screen. Press these keys to change the arc voltage while cutting.

These keys increase or decrease the Set Arc Voltage parameter by 0.5 V per soft 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 cut height in Manual mode

After cutting begins in Manual mode, the CNC shows **Raise Manual Cut Height** and **Lower Manual Cut Height** soft keys on the Main screen.

These soft keys change the cut height by 0.2 mm (0.01 inch) per soft key press. These soft keys also affect the THC voltage offset by 0.5 V per soft key press.

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.

Change arc voltage in the 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 or on the process Watch Window (if Set Arc Voltage is included on the Watch Window).
- 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

Phoenix uses a sequence called initial height sense, or IHS, for detecting the workpiece. 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.

IHS sequence

The IHS begins at the **Start IHS Height** set in the Process screen. When the torch (or external sensor if using Offset IHS) 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 (or an external sensor for Offset IHS) touches the workpiece, so the CNC knows the height of the workpiece.
- In case the Nozzle Contact Sense input does not activate, the CNC also 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 (or sensor) 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.

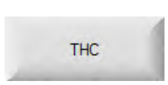
IHS in Manual mode

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.

Perform a first IHS

1. Choose 

or press F11 on the Main screen to home the THC axis.
2. Choose 

.
3. Choose 

.
4. Choose OK twice to return to the Main screen.

5. Choose 

. The THC performs the initial height sense starting 12.7 mm (0.5 inches) from the THC axis home position.

Sensor THC moves 12.7 mm (0.5 inches) from the THC axis home position 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).

Sensor THC's speeds are set in the Setups > Machine Setups > Speeds screen.

Offset IHS

Offset IHS enables an external sensor (or probe) to detect the workpiece and perform an initial height sense. Offset IHS is useful when cutting a pre-pierced workpiece to prevent the torch from performing an IHS on a pierce point.

The sensing device and the mechanism to control the device are determined by the OEM.

To set up Offset IHS in Phoenix:

1. Select **Setups** > **Process** to go the **Sensor THC - Plasma 1 or 2 Process** screen.

2. Set the **Offset IHS** parameter to **On**.

3. Review the **IHS Start Height** value. To use the default value, make sure that the value's checkbox is selected.

   This height must be a distance above the workpiece that will allow the sensor to fully extend. For example, if the sensor extends 50mm below the shield of the torch, the IHS Start Height must be greater than 50mm.

4. Click **OK** to save your settings.

5. On the **Cutting** screen (select **Main** > **Setups** > **Cutting**), set the offset values for **Marker Offset 9**.

   The Z value is the difference in height between the torch and the external sensor. This distance is determined by the OEM who installs the external sensor. This value should be less than the IHS Start Height (set in step 3).

When Offset IHS is performed, the torch moves to the position of Marker Offset 9, performs the IHS according to the *IHS sequence* on page 149, and returns to the pierce location.

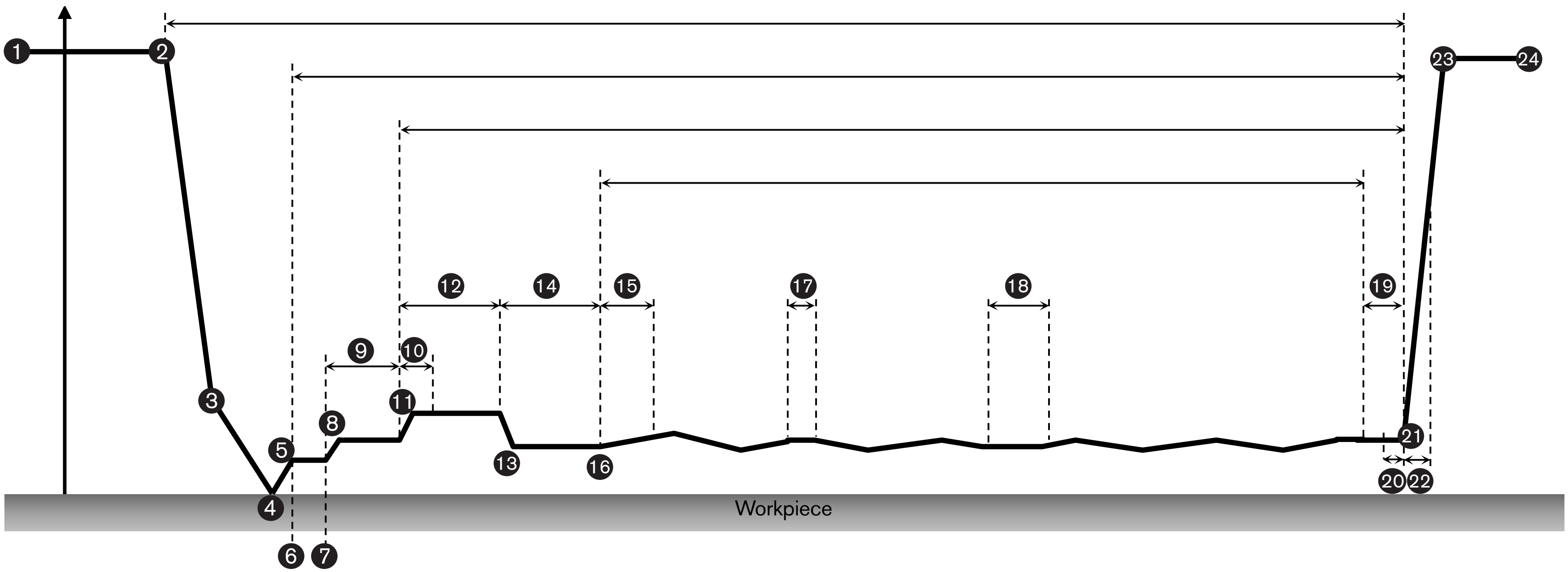
When to disable height control

The THC Disabled 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. The reason is that when the speed slows down, the arc voltage increases. This causes the torch to lower in order to reach the arc voltage set point. The torch will continue to lower until there is a collision with the workpiece, which causes a fault. To prevent the collision and fault, you can program the speeds for the torch height control so that the THC disables when the cutting speed slows down.

THC sequence of operations

The diagram on the next page shows the heights and timers used by THCs during cutting in Automatic mode. See the following notes for the diagram below:

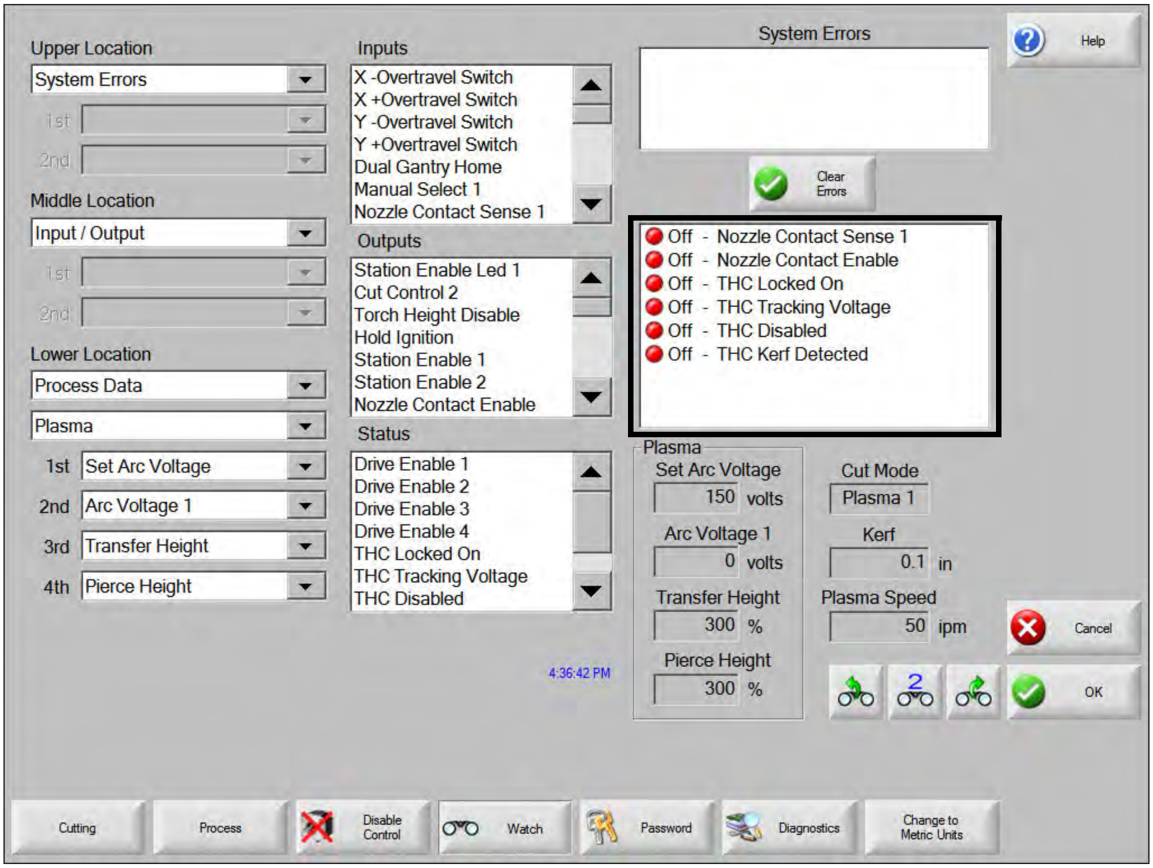
- For ⓯: The voltage tracking begins using the average of samples: one sample per cut up to 6 samples.
- For ⓰: The THC tracks arc voltage as long as AVC is not disabled by an M50, auto-kerf, or due to the torch height disable settings in Phoenix.
- For ⓱: Momentarily stops tracking until either the actual arc voltage comes back to its previous reading (before kerf was detected) or kerf re-acquire time expires.



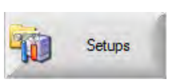
1	Torch path	8	Pierce height	15	AVC sampling (see note)	22	Stop time
2	M07 Command – cut on	9	Pierce delay	16	AVC begins (see note)	23	Retract complete
3	IHS start height	10	Creep time (does not affect THC states)	17	Auto kerf detect (see note)	24	Retract height
4	Tip touch (ohmic contact/stall force)	11	Puddle jump height	18	Torch height disable		CNC cut control active (not shown)
5	Transfer height	12	Cut height delay	19	Cut off time (this example assumes (-) cut off time)		Plasma torch active (not shown)
6	Cut control	13	Cut height	20	Arc off time (this limits the nuisance trip arc lost at end of cut. Max 2 sec.)		Cutting X/Y motion (not shown)
7	Arc transfers	14	AVC delay	21	M07 Command – cut off		AVC active (voltage tracking state) (not shown)

Set up a Watch Window

An example Watch Window for the Sensor THC is shown below:

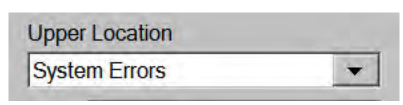


To set up this Watch Window:

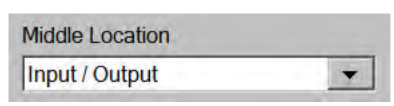
1. Choose 

 > 

.

2. Choose **System Errors** from the **Upper Location** list 

.

3. Choose **Input / Output** from the **Middle Location** list 

.

4. The **Status** list appears below the **Inputs** and **Outputs** lists. Select the following status bits and I/O from the lists.

   Nozzle Contact Sense

   This input activates during IHS when the torch detects the workpiece.

   Nozzle Contact Enable

   This output activates during the IHS and while cutting if Nozzle Contact is on.

   THC Locked On

   This status bit turns ON when the arc voltage is within +/- 2 V of the set point.

   THC Tracking Voltage

   This status bit turns ON when the torch height control is using Automatic Voltage Control (AVC). When ON, the CNC makes adjustments to the position of the THC by comparing actual arc voltage to the set arc voltage. The torch height control adjusts the cutting height based on the arc voltage.

   If the 50 V default tracking voltage is exceeded, the THC will not track voltage. This can happen, for example, with an incorrect Pierce Height or Cut Height setting.

   THC Disabled

   This status bit turns ON when the CNC disables the torch height control, usually as it approaches a corner in a part (or a small radius, hole, or pierce), the cutting system slows down, and the actual cutting speed is slower than the program speed. This bit is controlled by the Torch Height Disable Speed along with the Distance Before and Distance After settings on the Speeds screen, and by the M50 (Torch Height Disable) and M51 (Torch Height Enable) codes in a part program.

   THC Kerf Detected

   This status bit turns ON when the THC detects a rapid rise in the measured arc voltage, which indicates that the torch is cutting across a previously cut kerf. This bit is controlled by the Auto Kerf Detect and Auto Kerf Detect Voltage parameters. Auto Kerf Detect must be active and the THC must be in Automatic mode. When kerf is detected, the CNC temporarily disables the AVC and prevents the torch from diving into the workpiece.

9 Station Setup

Overview

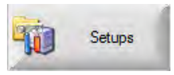
Hypertherm CNCs define a *station* as a physical tool on the cutting system along with its lifter (if one is used). The tool can be a plasma torch, marker, or oxyfuel torch. Regardless of the tool, the CNC provides many of the same options for setup and operation of the stations.

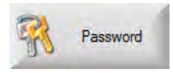
The CNC supports multiple stations on a single cutting system. For the CNC to operate each cutting station independently, the stations must be set up with numbered inputs and outputs (I/O).

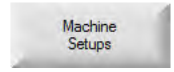
In this section you will find the following:

- A description of generic and numbered I/O
- How to use Auto Select and Manual Select inputs to activate either Program or Manual modes of operation
- How to assign the lifter and plasma power supply (or other cutting supply such as marker), and enable the cut charts for each station on the Station Configuration screen

Generic and numbered I/O

The inputs and outputs for stations, available in 

> 

> 

> 

, can be used to control any of the cutting technologies supported by the CNC. The CNC supports 2 types of I/O for controlling the cutting stations:

- Generic I/O – used for any cutting process and to control a single cutting station
- Numbered I/O – used for multiple cutting stations or with an operator console

Generic I/O

A standard group of inputs and outputs are used to control each cutting process. These I/O are called *generic* because all cutting sequences use them. Examples of generic I/O include Cut Control, Cut/Mark Sense, Torch Up, and Torch Down. Many generic I/O functions that affect stations have an equivalent numbered I/O.

Numbered I/O

You must use numbered I/O to identify stations when the cutting system has more than 1 station **or** uses an operator console (with either a single torch or multiple torches). Numbered I/O allow the operation of individual cutting stations through the CNC instead of through a programmable logic control (PLC) or relay logic interface. Each of the functions of generic inputs and outputs that affect stations are also available as numbered I/O.

We recommend that you match the I/O number to the station number. For example, in a 2-torch plasma cutting system, you would assign numbered I/O for each station as follows.

Torch 1 (Station 1)	Torch 2 (Station 2)
Auto Select 1 input	Auto Select 2 input
Manual Select 1 input	Manual Select 2 input
Station Enable LED 1 output	Station Enable LED 2 output
Cut Control 1 output	Cut Control 2 output
Cut Sense 1 input	Cut Sense 2 input

If you have an HPRXD plasma power supply, the fixed function I/O HPR Cut Control and HPR Cut Sense are automatically assigned. For more information see *Fixed function I/O (virtual)* on page 119.

Enable station I/O

Each station must be *enabled* before you can start operation. In some cutting systems, you enable a station with a switch on a hardware operator console that activates an input (the Manual Select input, for example). In other cases, the station is enabled with the M37 code in the part program. (For more information about codes and part programs, see *EDGE® Connect Programmer Reference* (809550).)

On the CNC's standard Software Operator Console (Soft Op Con) you can choose one of 3 modes:

Disable Station	Manual Mode	Program (Automatic) Mode
		

The color of the mode you select changes from gray to green when you select it.

For more information about Soft Op Con, see *Operate the standard Hypertherm Soft Op Con* on page 65.

Auto Select and Manual Select inputs and Station Enable LED output

When creating an operator console that allows the cutting stations to operate in either Manual mode or Automatic mode (also called program mode), you must use the following inputs and outputs (as either generic or numbered I/O).

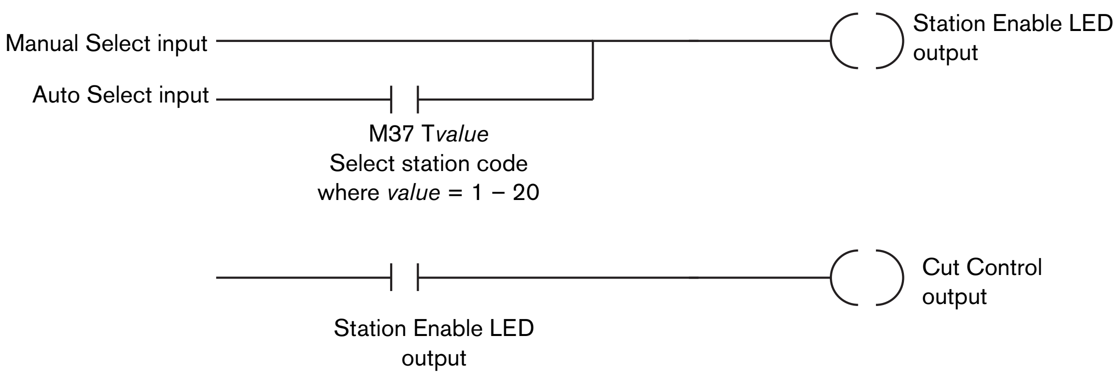
Once these I/O are assigned, the cutting stations show in the Soft Op Con.

- **Manual Select input:** When the Manual Select input is on, the station is in Manual mode. Manual mode allows operations such as jogging, Go to Home, and rip cutting. Manual mode can also function as a program override for station selection. (The override function is described in more detail in *Use Manual mode as an override* on page 159.)
- **Auto Select input:** When the Auto Select input is on, the station is in Automatic, or program, mode. For Automatic mode to control the station, the part program must run the M37 Tvalue code. The M37 code enables the station I/O. For more information on program codes see the *EDGE® Connect Programmer Reference* (809550).

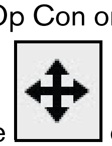
- **Station Enable LED output:** When the Station Enable LED output is ON, the station is enabled and the Cut Control output can be activated. This output can be turned ON in 2 different ways:
  - The Manual Select input turns on the Station Enable LED output.
  - The CNC reads the Auto Select input **and** the M37 Tvalue in the part program. Both of these conditions must occur to turn on the Station Enable LED output.

  The Station Enable LED output then activates the Cut Control output. *Figure 29* shows the logic of the inputs and outputs that implement both manual and program mode in the operator console. The Soft Op Con also uses this logic for Station 1 and 2.

Figure 29



Basic operating sequence:

1. Load a part program.
2. Choose a mode on the Soft Op Con.
   - Automatic mode for a part program with automatic station selection and activation.
   - Manual mode for a part program without automatic station selection, or when you want to override the automatic station selection (see *Use Manual mode as an override* on page 159).
   - Disable mode for a station that will not be used in the part program.
3. Align the part to the workpiece. Use the jog arrows on the Soft Op Con or a joystick if available to position the torch for cutting. To access the jog arrows choose  on the Soft Op Con.

   A *Ready to Move?* message displays when you use the jog keys in the Soft Op Con, unless the message is disabled in Special Setups.

4. Choose Start to start cutting the part.

Use Manual mode as an override

Another use of Manual mode is to override the station selection in a part program. For example, the part program contains the M37 T1 code to select Station 1 for cutting the part. However, you want the program to use Station 2. On the Soft Op Con, choose Station Disable for Station 1 and choose Manual mode for Station 2. Load the part and choose Start. When you choose Manual mode, it overrides the station selection in the part program. In this example, choosing Manual mode would cause the part to be cut with Station 2 instead of Station 1.

Summary

The Auto Select and Manual Select inputs and the Station Enable LED output do the following.

- Give the operator manual control over the stations, but also allow part programs that use the M37 code to select a station (program mode).
- Are used for the station modes on the Soft Op Con.
- Once assigned, make the cutting stations show in the Soft Op Con.
- Are an optimal method for controlling the stations when you have different tools on the same table (for example, plasma and oxyfuel).

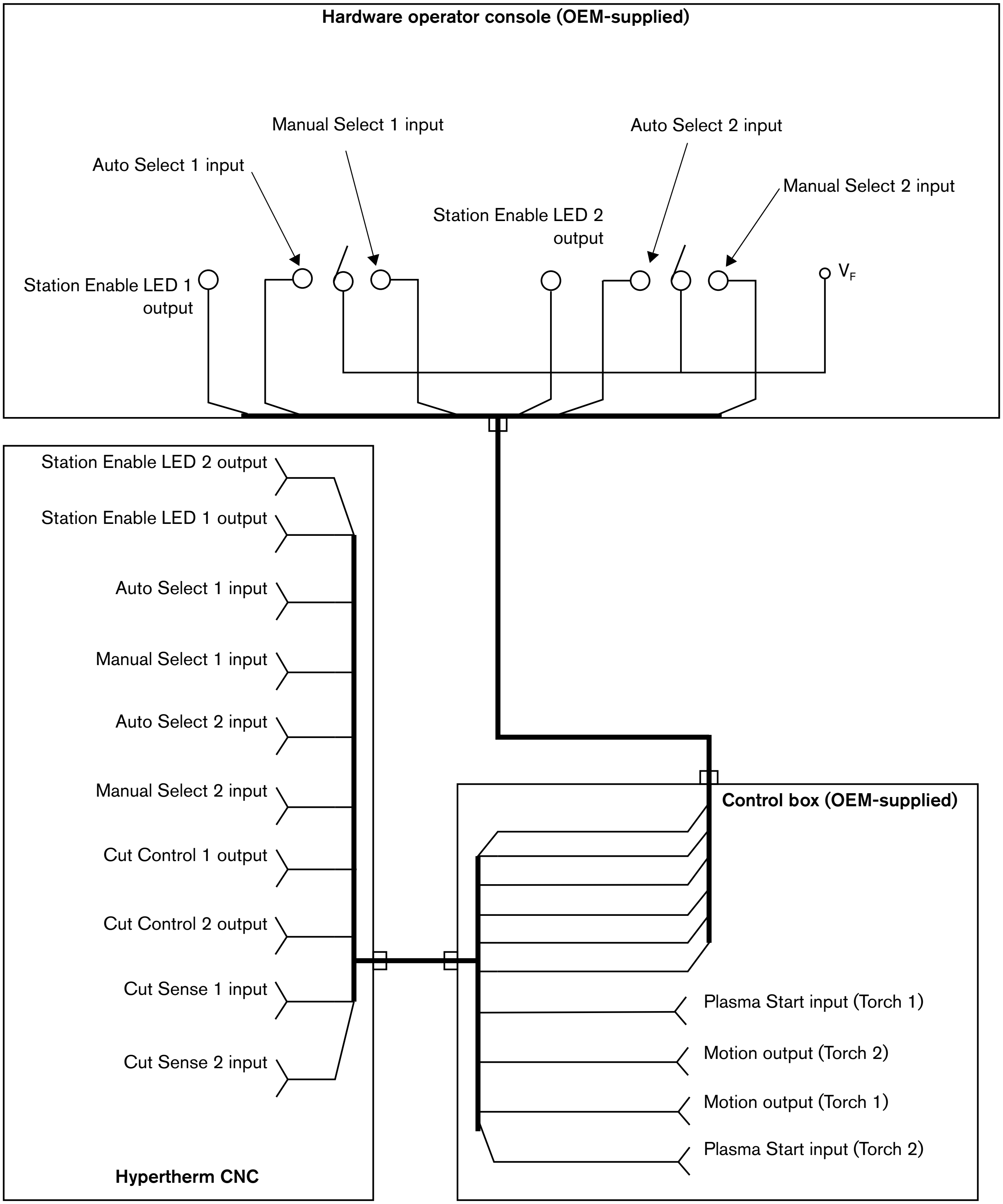
Additional I/O may be necessary depending on the torch height controls and other mechanisms that are part of your cutting system.

Figure 30 on page 160 shows an example of the Auto Select and Manual Select inputs, Station Enable LED outputs, and additional I/O for a 2-torch operator console and control box.

This is an example only and not intended as a recommendation for system design.

If you have an HPRXD plasma power supply, the fixed function I/O HPR Cut Control and HPR Cut Sense are automatically assigned and used instead of Cut Control 1 and 2 and Cut Sense 1 and 2. For more information see *Fixed function I/O (virtual)* on page 119.

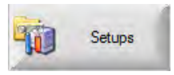
Figure 30

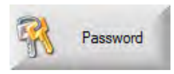

Hardware operator console (OEM-supplied)
Manual Select 1 input
Auto Select 2 input
Auto Select 1 input
Manual Select 2 input
Station Enable LED 2 output
Station Enable LED 1 output
V_F
Station Enable LED 2 output
Station Enable LED 1 output
Auto Select 1 input
Manual Select 1 input
Auto Select 2 input
Manual Select 2 input
Cut Control 1 output
Cut Control 2 output
Cut Sense 1 input
Cut Sense 2 input
Hypertherm CNC
Control box (OEM-supplied)
Plasma Start input (Torch 1)
Motion output (Torch 2)
Motion output (Torch 1)
Plasma Start input (Torch 2)

THC and cut chart setup on the Station Configuration screen

The Station Configuration screen lets you define each station in the cutting system. In this screen, you select the type of torch height control and the model of the tool (plasma or marker) for the station. The selections you make in this screen do the following.

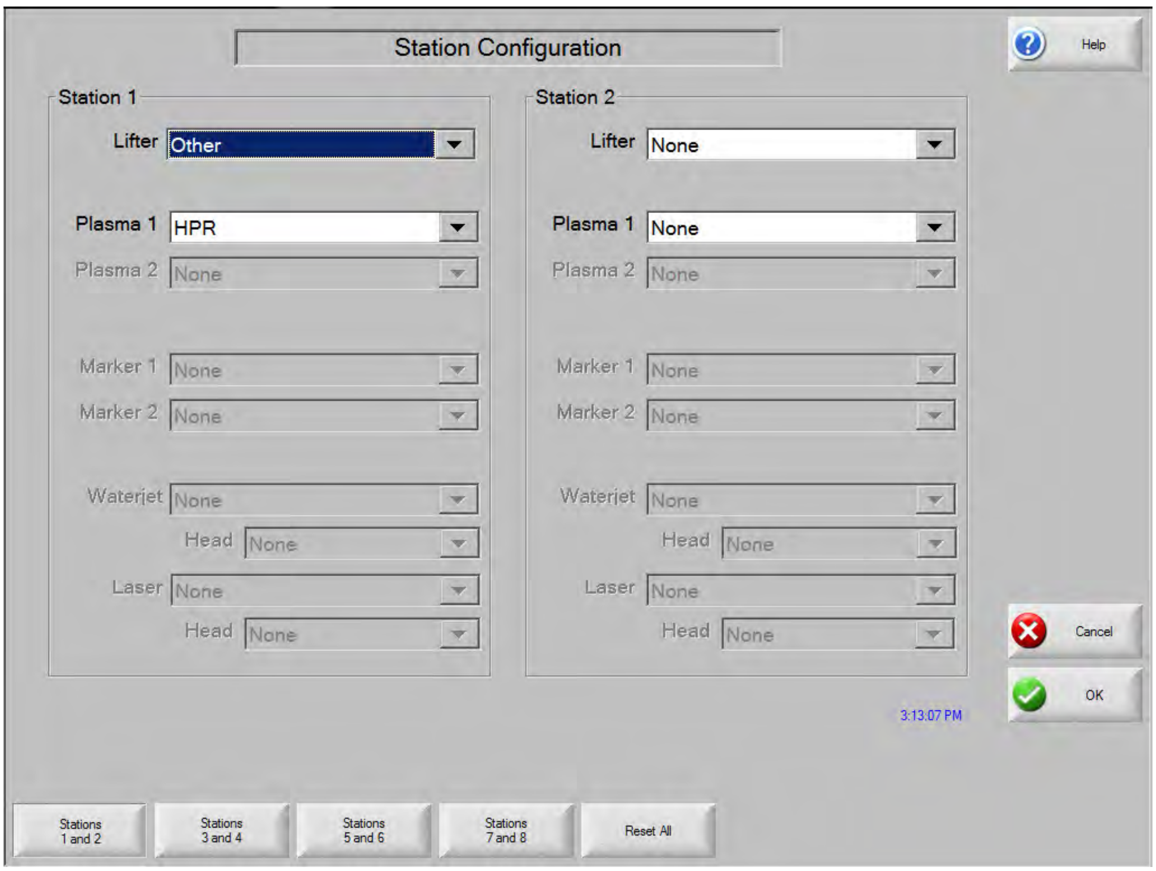
- Customize other screens in the CNC with features that are unique to the selected tool and torch height control
- Enable the cut charts that are available for the selected tool

To open the **Station Configuration** screen, choose 

 > 

 > 

.



Guidelines for using the Station Configuration screen

- Before you use the Station Configuration screen, make sure that you have made selections for Tools Installed in the Special Setups screen (Setups > Password > Special Setups). The Tools Installed selections define the processes used on the cutting system. For more information on Plasma 1 and Plasma 2 processes, see *Plasma Setup* on page 169.
- When you use the Sensor THC for a lifter, make sure that you select the lifter in this screen. Torch height control parameters specific to this lifter become available in the Process screen so that you can optimize cutting performance.

  Before the Sensor THC is available in the Station Configuration screen, you need to select it as the torch height control in the Machine Setups screen (Setups > Password > Machine Setups). You also need to assign the Sensor THC to an axis.

- Select the process used by the tool on the station. For example, for Plasma 1 you can select HPR. This selection makes the cut charts available for the process.

  Oxyfuel systems DO NOT require a selection in the Station Configuration screen. Oxyfuel cut charts become available after you select Oxyfuel as a Tool Installed and enable Oxy Fuel Cut Charts in the Special Setups screen (Setups > Password > Special Setups).

- Use the soft keys to open the Station Configuration screen for Stations 1 – 2, 3 – 4, 5 – 6, and 7 – 8.
- Use the Reset All soft key to return all settings for all stations to None.

  See *Plasma Setup* on page 169 for more information on using the Station Configuration screen for a cutting system with multiple stations.

Conflicting process

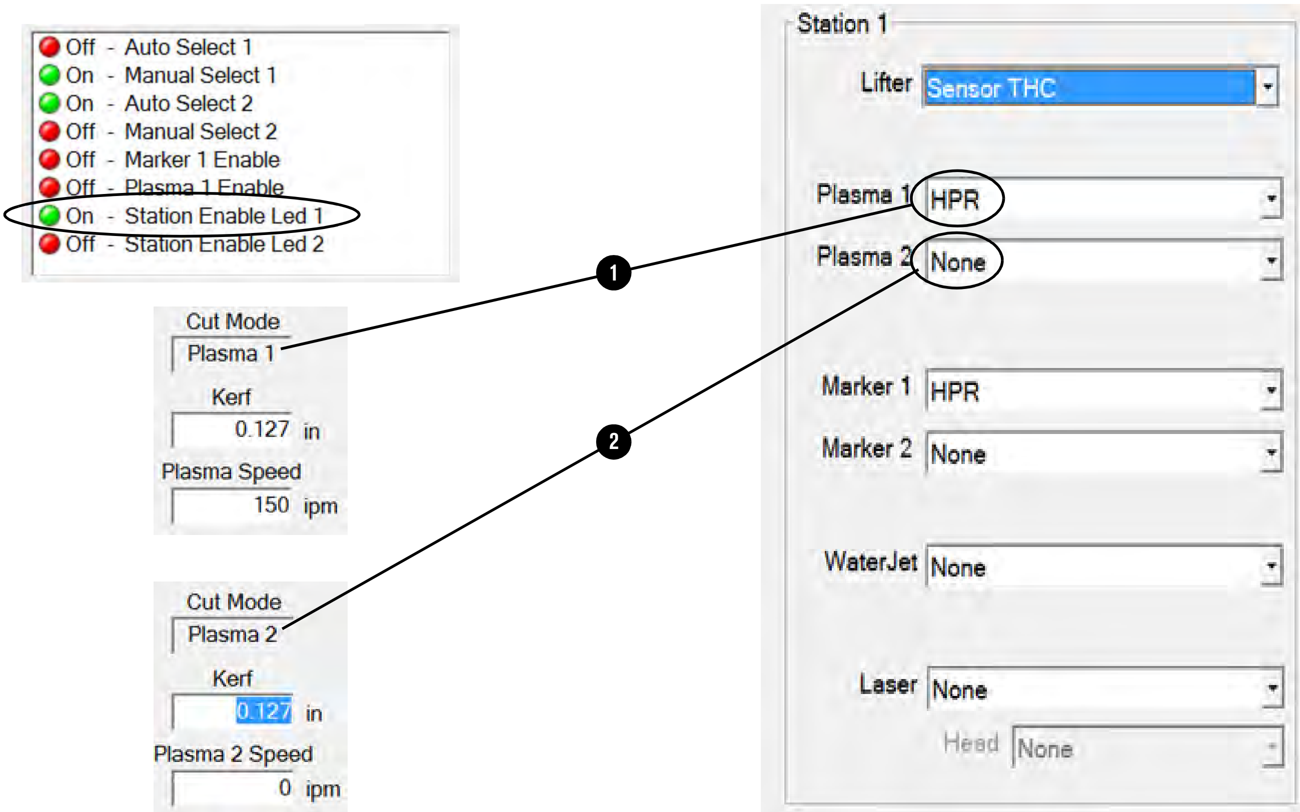
A *conflicting process* is an error condition that pauses a program or prevents a program from loading. A conflicting process occurs when the cut mode does not match the process (or processes) available for an active cutting station. The conflicting process feature in Phoenix compares the selected cut mode to the processes assigned to each active station, and looks for a match. This happens automatically after you choose Start on the controller. A station is active when the Station Enable LED output associated with it is active.

For example, if the cut mode is Plasma 1, then all active cutting stations require a Plasma 1 process or the program will pause with a conflicting process error message.

The conflicting process feature makes sure that different tools cannot operate if both of their stations are accidentally left on by the operator. Conflicting process applies to all processes except oxyfuel.

Example of a conflicting process

How a tool is associated with a station

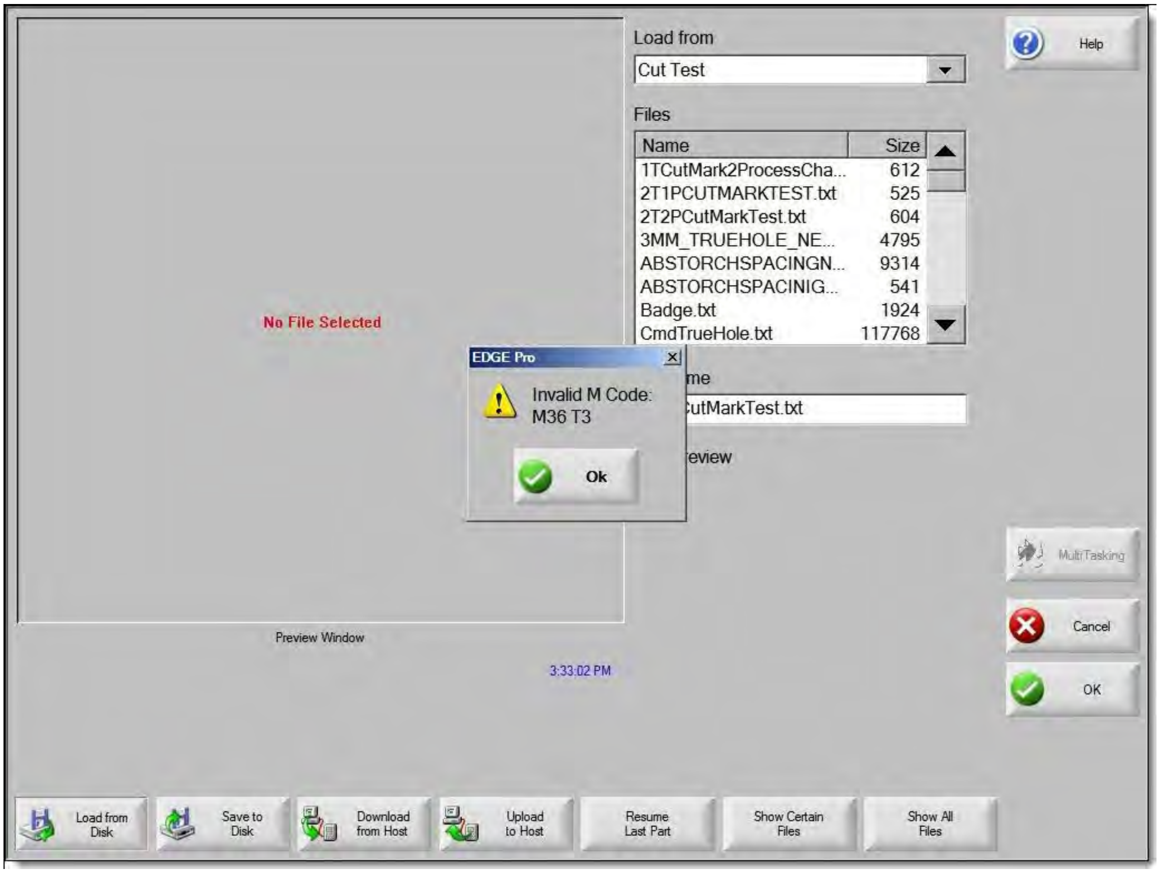


In this example you can tell Station 1 is active because the Station Enable LED 1 is illuminated. When you choose Start, Phoenix compares the cut mode with the processes assigned to the active station.

1. If the cut mode is Plasma 1, Phoenix checks for a valid Plasma 1 process on Station 1. HPR is a valid process.

2. If the cut mode is Plasma 2, Phoenix checks for a valid Plasma 2 process on Station 1. There is no plasma process selected (None) so you get a conflicting process error.

A conflicting process example that prevents a program from loading



In this example, the CNC shows a message indicating that an M-code is invalid. It is a legitimate M-code, but the Marker 1 process is not assigned to a station. For an M-code to be valid the associated process must be assigned to a station.

The part program in this example contains an M36 T3 code (Marker 1 process select selection code), but a Marker 1 process is not assigned to a cutting station. However, Marker 1 is assigned as a tool. In order to make use of an M36 code within a program, the process being called must be assigned to a station within the Station Configuration screen. This type of scenario applies to all M36 process codes.

- M36 T1 (Plasma)
- M36 T2 (Plasma 2)
- M36 T3 (Marker 1)
- M36 T4 (Marker 2)

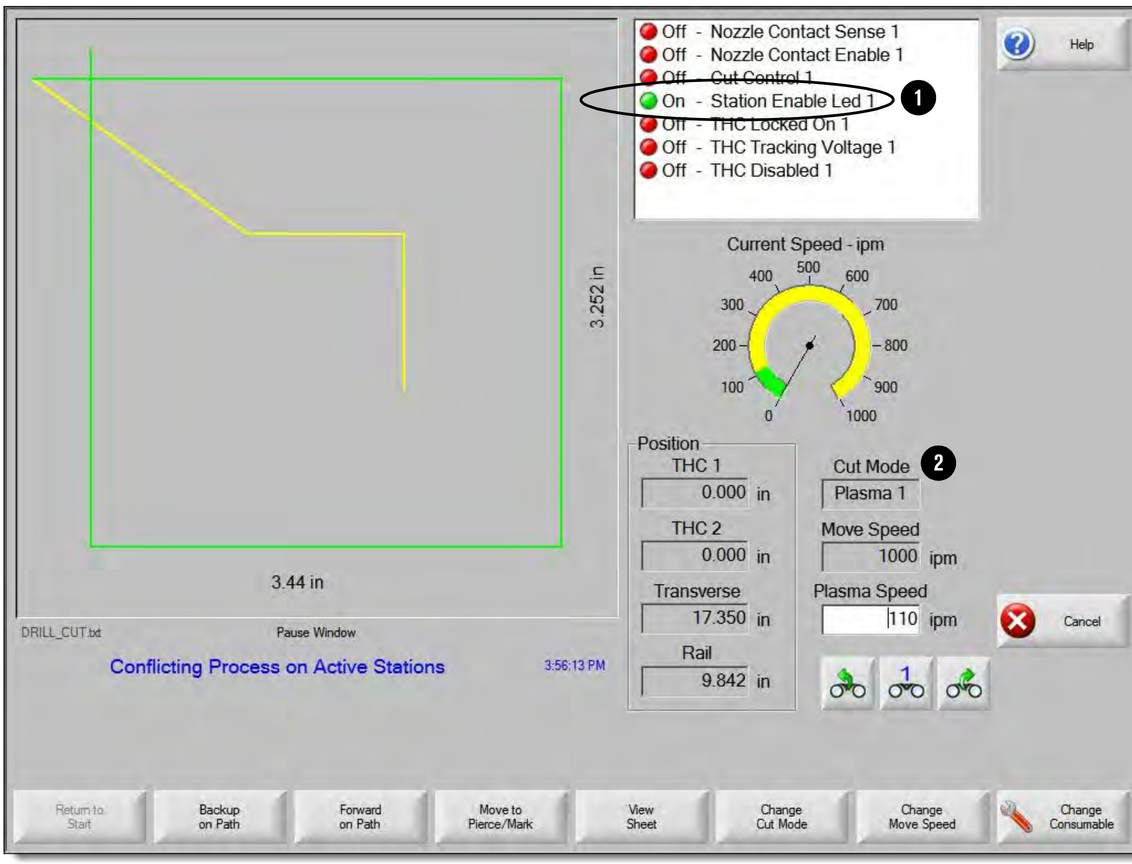
Adding a Marker 1 process to the Station Configuration screen resolves the issue.

Troubleshooting a conflicting process error

The settings look correct on the screen below

1. Station 1 is the active station.
2. The cut mode is Plasma 1.
3. Plasma 1 is defined on Station 1 on the Station Configuration screen. See *Figure 32* on page 166.

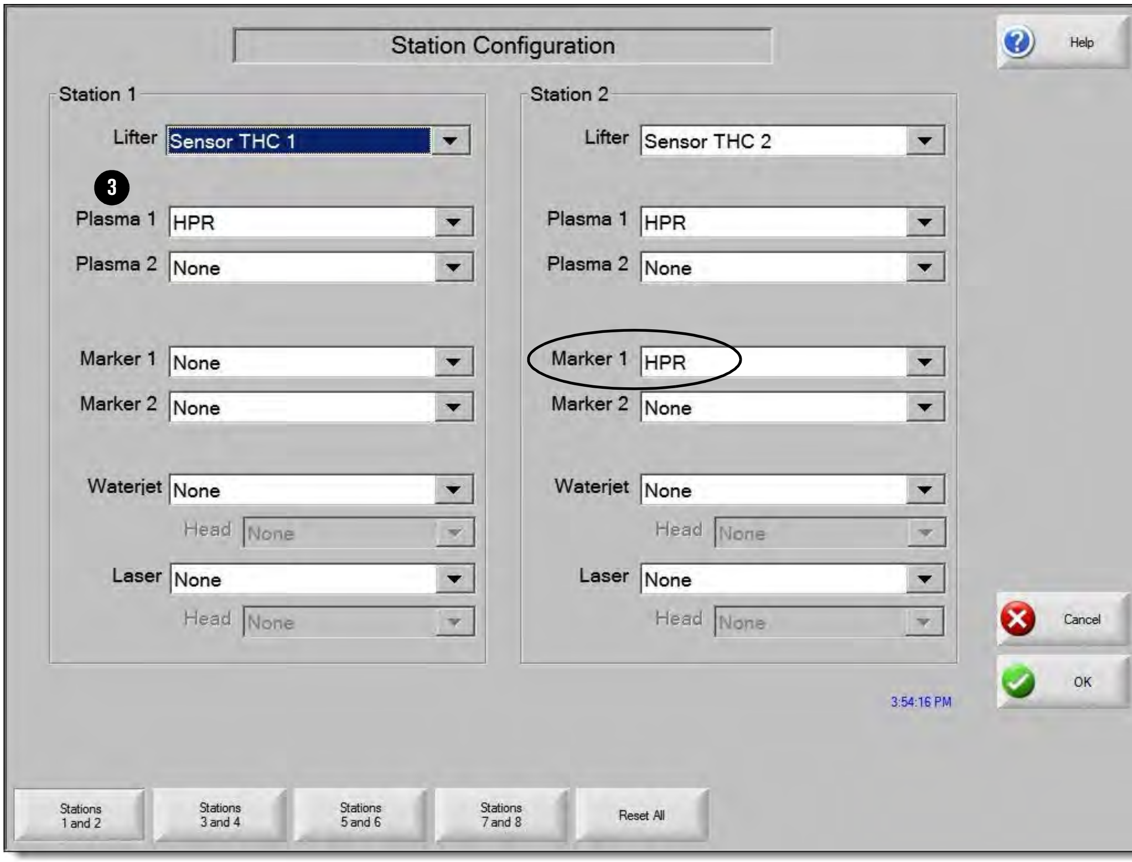
Figure 31



The cause of the error

Marker 1 is assigned to Station 2 in the Station Configuration screen and no station I/O is assigned for Station 2, which makes Station 2 active in Phoenix.

Figure 32



Troubleshooting steps

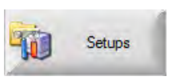
1. Review the settings in the Station Configuration screen.
   a. A station must have a process assigned that matches the cut mode.
   b. The processes assigned to the station must be valid.
   c. A physical tool with processes that match the station settings must be installed.
2. Review station I/O assignments.
   a. Make sure that the station I/O is assigned correctly.
   b. Each station should have the following I/O assigned: Auto Select # / Manual Select # / Station Enable LED # (and same # for each station).
   c. If one station has station I/O, all stations need station I/O.

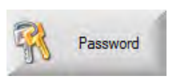
3. Review the following settings.
   a. Make sure that the Tools Installed on the Special Setups screen are correct.
   b. Make sure that all processes (for Tools Installed) are assigned on the Station Configuration screen.
   c. Make sure that station I/O is assigned correctly on the I/O screen.
   d. Make sure that Station Select Override and Process Select Override are enabled on the Cutting screen if you are using M36 T*value* codes to select a process from the part program.

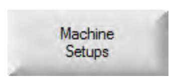
4. Monitor the station I/O and processes (on a Watch Window).
   a. Monitor the Station Enable LED outputs.
   b. Monitor the Process Enable outputs.
   c. Monitor Process Data for each available process (preferably in one Watch Window).

Remove a station from the Soft Op Con

Removing a station on the Station Configuration screen does not automatically remove the station from the Soft Op Con. This is because the Auto Select and Manual Select inputs and the Station Enable LED output are what make stations show in the Soft Op Con. To remove a station from the Soft Op Con, you must also remove these I/O.

1. Choose 

 > 

 > 

 > 

.

2. Set the I/O for the station that you want to remove to **Spare**.

10

Plasma Setup

Overview

This section describes the tasks you need to perform to set up the Hypertherm CNC with a plasma power supply. Because of the variation between cutting systems, Hypertherm CNCs provide built-in flexibility for multiple methods of system setup, operation, and part programming. Since all cases cannot be described here, this section makes the following assumptions.

- The cutting operations are being controlled by the CNC, not by a programmable logic controller (PLC) or other external relay logic.
- The cutting system is equipped with a Sensor THC.

In this section you will find the following.

- An overview of the Plasma 1 and Plasma 2 processes and guidelines for setting up the processes
- A detailed, step-by-step description of the plasma cut sequence
- Definitions of the inputs and outputs used for plasma cutting systems
- Plasma setup instructions for the CNC

Plasma part programs can use process selection variables to load the cut chart. The CutPro™ Wizard provides additional automation. For more information about plasma process selection variables, see the *EDGE® Connect Programmer Reference* (809550).

Plasma 1 and Plasma 2

Plasma 1 and Plasma 2 refer to two separate cutting processes. Marker 1 and Marker 2 likewise refer to two separate marking processes.

- Select Plasma 1 and Plasma 2 as Tools Installed on the Special Setups screen to enable the Plasma 1 and Plasma 2 Process screens.
- Assign the specific plasma power supply to Plasma 1 and Plasma 2 on the Station Configuration screen to make the cut charts available along with other functionality specific to the cutting systems. In the Station Configuration screen you match up the process (Plasma 1 and Plasma 2) to the cutting station (Station 1, Station 2, and so on).

For more information on stations, see *Station Setup* on page 155.

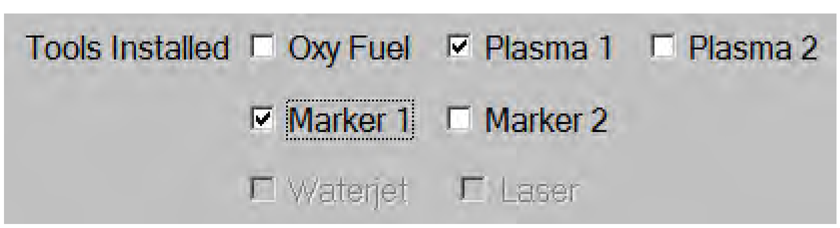
In general, follow these guidelines for using Plasma 1 and 2 on the CNC.

- In a single-torch cutting system, you need only Plasma 1 for Station 1.
- In a multiple-torch cutting system where the torches cut parts using the same process and cut chart, select only Plasma 1.
- In a multiple-torch system where the torches are from different plasma supplies, you need to select Plasma 1 and Plasma 2 so that the CNC makes a second cut process and cut chart available.
- Use both Plasma 1 and Plasma 2 when the part program calls for two different cutting processes. For example, the part program cuts the detail portions of a part with a low amperage consumable and then automatically switches to a higher amperage consumable to cut the part contour. This type of part program would require two torches: One torch has a low amperage consumable set while the second torch has a high amperage consumable set. Therefore, use Plasma 1 for the low amperage cut chart and Plasma 2 for the high amperage cut chart.

Sample settings for a single-torch cutting system

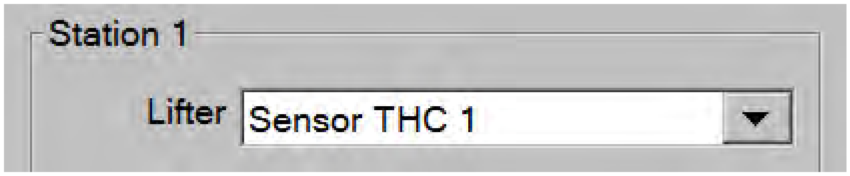
To set up a single-torch HPRXD system, follow these general steps on the CNC.

1. On the **Setups > Password > Special Setups** screen, select only **Plasma 1** and **Marker 1**.

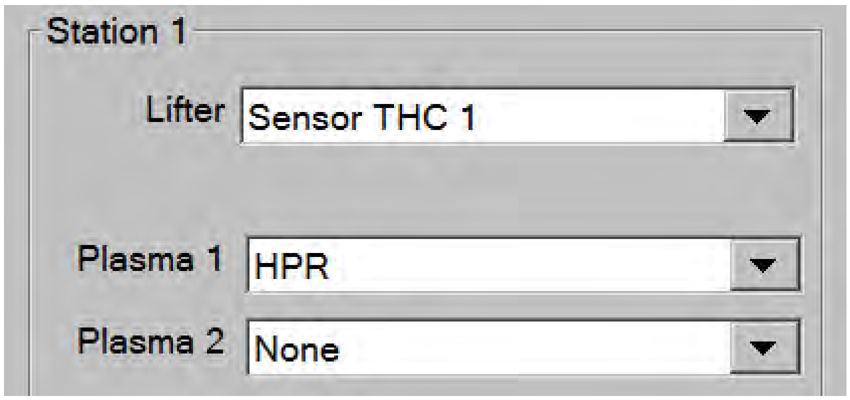


THC selections must also be made on the Machine Setups screen before selecting them on the Station Configuration screen. See *Torch Height Control (THC)* on page 135.

2. On the **Setups > Password > Station Configuration** screen, choose the torch height control on **Station 1** for the lifter.



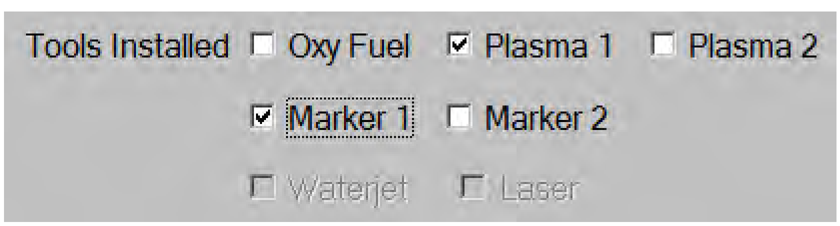
3. Choose **HPR** for **Plasma 1**.



Sample settings for a multiple-torch cutting system

To set up a multiple-torch HPR Auto Gas system that uses one cutting process and one marking process for all torches, follow these general steps on the CNC:

1. On the **Setups > Password > Special Setups** screen, select **Plasma 1** and **Marker 1**. Plasma 2 is a separate process and is not used in this example.

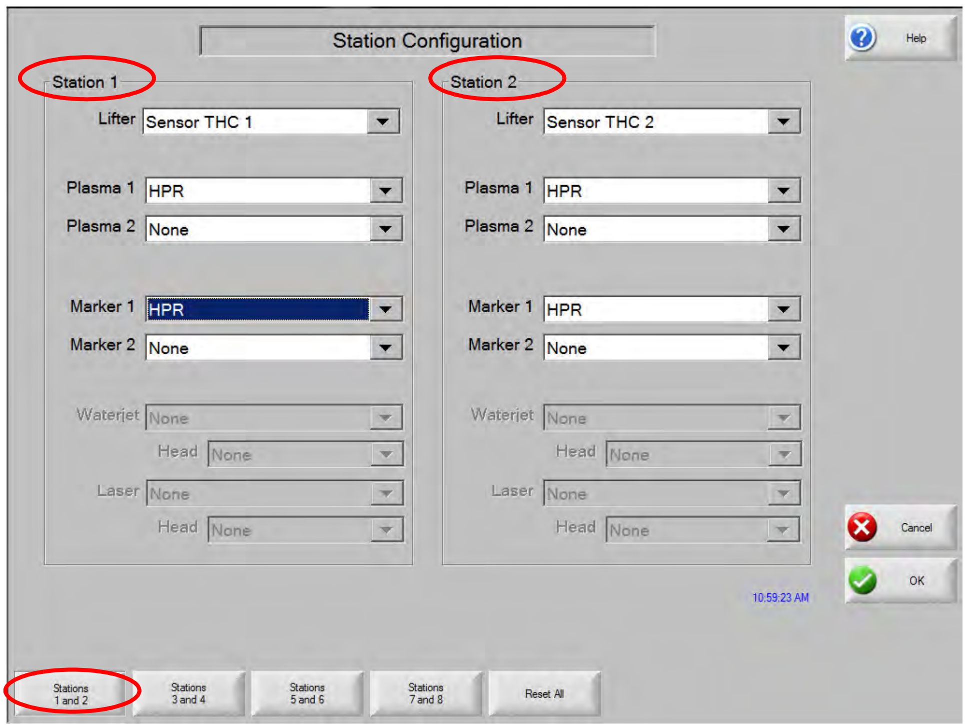


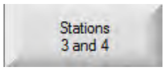
2. On the **Setups > Password > Station Configuration** screen, choose **HPR** for **Plasma 1** and **Marker 1** for **Station 1** and **Station 2**.

3. On this screen you are assigning the cutting and marking processes that will be used for each of the individual cutting stations.

   If all torches are always going to cut and mark with the same consumable sets, then the assignment is Plasma 1 and Marker 1 for all cutting tools.

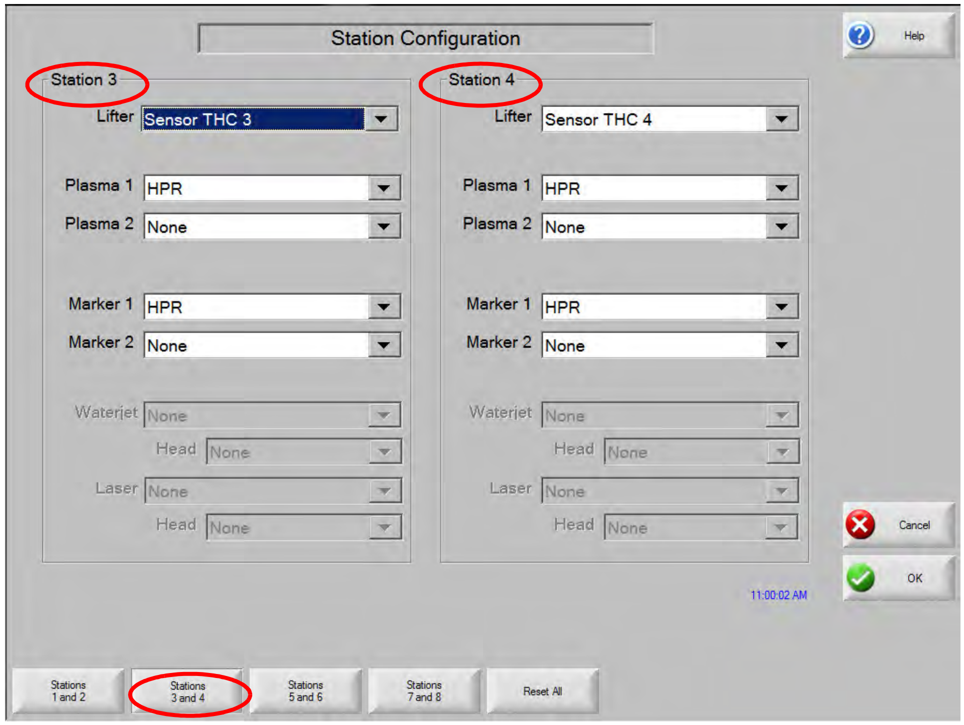
Station 1 and Station 2



4. Choose 

to advance to the screen for Stations 3 and 4, and repeat the assignment of the HPR to **Plasma 1** and **Marker 1** for these stations.

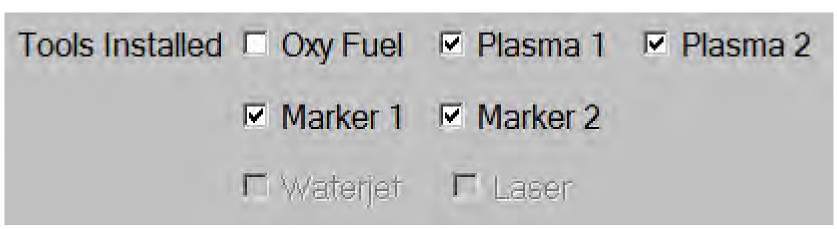
Station 3 and Station 4



Sample settings for a two-torch cutting system

This example uses an **HPR130XD** Auto Gas plasma power supply on one station and an **HPR400XD** Auto Gas plasma power supply on the other station. To set up a two-torch cutting system where two different processes are being used, follow these general steps on the CNC.

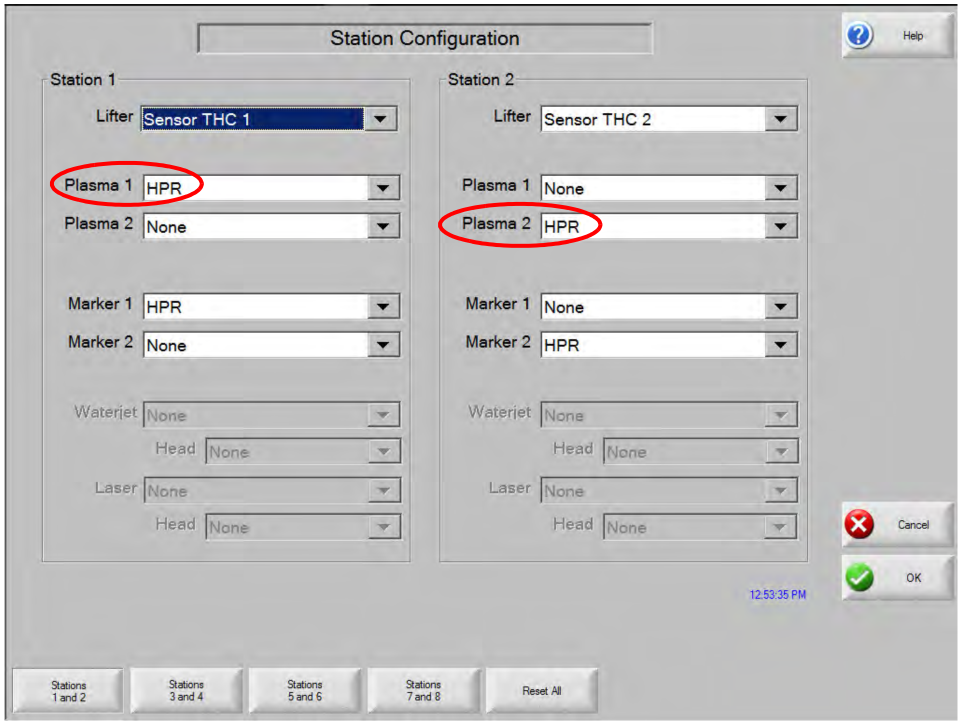
1. On the **Setups > Password > Special Setups** screen, select **Plasma 1** and **Marker 1** *and* **Plasma 2** and **Marker 2**.



2. On the **Setups > Password > Station Configuration** screen, select HPR in both **Plasma 1** and **Plasma 2**. Both are being used because the HPR130XD and HPR400XD plasma power supplies use different cut processes and consumables. This is when the Plasma 2 selection is required.

Assigning the HPR400XD on Station 2 and Plasma 2 provides the ability to use two HPRXD torches with different consumables for different cutting processes in the same part program.

Plasma 1 and Plasma 2 are required when the cutting system supports different cutting processes in the same part program.



Plasma cut sequence

Also see the *THC sequence of operations* timing diagram on page 151.

Hypertherm CNCs execute the plasma cut sequence in these basic steps.

1. The CNC sends the Cut Control output signal to the plasma power supply. Cut Control connects to the Start input on the plasma power supply.
2. When the plasma power supply transfers an arc, it sends the Motion output signal to the CNC. Motion is connected to the Cut Sense input on the CNC.
3. While waiting for the cut sense input to turn on, the CNC displays the message “Waiting for Arc On.” When the Cut Sense input on the CNC turns on, the CNC initiates motion.

The Cut Control output and Cut Sense input are the minimum I/O signals needed in a plasma cutting system. The CNC provides additional I/O for torch height controls (THCs), for multiple-torch applications, and to enable features that reduce the cycle time between parts.

In most cutting systems, a THC manages the signals between the CNC and the plasma power supply. When a signal is sent to the plasma power supply, it typically passes through a THC, but the signal stays the same.

The following plasma sequence assumes the use of the Sensor THC. When using a THC from another manufacturer, the CNC provides a series of timers to control I/O to lower and raise the torch. For more information on using the timers for a THC, see *Torch Height Control (THC)* on page 135.

The CNC identifies each state of the cut sequence with a status message below the part preview area of the Main screen.

The cut sequence begins when you press Cycle Start. At Cycle Start, the CNC begins reading and executing the part program.

State and Status message: *Lowering Torch*

Initiated when: The CNC reads the M07 code in the part program.

- The THC performs its initial height sense (IHS).
- Torch Height Disable output turns on and remains on until the cutting system reaches cutting speed.
- If Preflow During IHS is on in the Process screen (and you have an HPRXD plasma power supply), then Cut Control and Hold Ignition turn on.

State and Status Message: *Waiting for Arc On*

Initiated when: IHS completes and the torch is at the Transfer Height.

- Cut Control output turns on. Cut Control activates the Plasma Start input to the plasma power supply.
- Hold Ignition turns OFF.
- Cut Control remains on until the CNC reads the M08 code (Cut Off) in the part program.
- The Pierce Control output turns ON (if you have an HPRXD plasma power supply).

State and status message: *Piercing*

Initiated when: The plasma power supply ignites an arc.

- The plasma power supply turns ON the Motion output.
- The Cut Sense input at the CNC turns ON.
- The CNC delays motion until the Pierce Time set in the Process screen elapses.
- The Pierce Control output turns ON until the Pierce Time elapses.

State and status message: *Creeping*

Initiated when: Creep motion begins.

- Creep motion continues until the Creep Time elapses. If Creep Time is not set in the Process screen, no creep motion will be used. Creep speed is a percentage of the Cut Speed set on the Setups > Password > Machine Setups > Speeds screen.

State and status message: *Cutting*

Initiated when: Cutting system accelerates to Cut Speed.

- Torch Height Disable turns OFF after the cutting system reaches cutting speed.
- Torch Height Disable toggles ON and OFF while cutting whenever the actual cut speed drops to a percentage below the set cut speed (the Torch Height Disable Speed is set in the Speeds screen).

State and status message: *Raising Torch*

Initiated when: The CNC reads the M08 code (Cut Off) in the part program.

- Cut Control turns OFF, which turns OFF the Start input at the plasma power supply.
- The Motion output at the plasma power supply turns OFF, which turns OFF the Cut Sense input at the CNC.
  - **For an HPRXD plasma power supply:** when the Cut/Mark Sense input at the CNC turns OFF, it immediately retracts the torch because the HPRXD has already extinguished the arc.
  - **For a non-HPRXD plasma power supply:** when the Cut/Mark Sense input at the CNC turns OFF, it starts a retract delay timer and waits before retracting the torch because the arc may still be present.

State and status message: *Stop Delay*

Initiated when: A Stop Time has been entered in the Process screen.

- A stop delay prevents the gantry from moving to the next pierce point until the Stop Time elapses. Setting a Stop Time makes sure the torch clears any tip-ups before moving to the next pierce point.
- The THC raises the torch to the retract height.

State and status message: *Traversing*

Initiated when: The Stop Time elapses.

- The gantry moves to the next pierce point and the sequence repeats.

Setting up inputs and outputs for plasma

Fixed function I/O for HPRXD

Phoenix assigns fixed function I/O automatically when HPR is selected on the Station Configuration screen. Fixed function I/O for the HPRXD is not shown as assigned to any I/O points in the Inputs and Outputs lists on the I/O screen and does not appear in the I/O Diagnostics screen.

The fixed function I/O for HPRXD plasma power supplies is shown below.

Digital inputs
HPR Cut Sense
HPR Not Ready For Start
HPR Nozzle Contact Sense
HPR Process Ready

Digital outputs
HPR Cut Control
HPR Hold Ignition
HPR Pierce Control
HPR Nozzle Contact Enable
HPR Remote On

Station I/O for the Soft Op Con

The I/O that makes a station show on the CNC's standard Soft Op Con (Auto Select and Manual Select inputs, and the Station Enable LED output) are automatically assigned by Phoenix when you select HPR on the Station Configuration screen. See *Station Setup* on page 155.

Fixed function I/O descriptions

HPR Cut Sense / HPR Cut Sense 1 – 4 – These inputs notify the CNC that the torch has transferred the arc to the workpiece. The input originates from the Motion output from the plasma power supply or THC, and the CNC begins motion after this input activates and the Pierce Time expires.

HPR Cut Sense 1 – 4 are the numbered versions of the HPR Cut Sense input. In a multiple-torch cutting system, the HPR Cut Sense would be automatically assigned for each station that you configure with an HPR in the Station Configuration screen.

HPR Not Ready for Start – When enabled, the torch will not fire.

HPR Nozzle Contact Sense – Used during IHS to detect the surface of the workpiece. With Sensor THC, this input is returned to the CNC through the external voltage divider card.

HPR Process Ready – This input turns on at the CNC when the HPR EtherCAT board has finished handling the process update. This input does **not** mean that the HPR plasma power supply has finished handling the process update or that the HPR is ready or waiting for a plasma start signal.

HPR Cut Control 1 – 4 – These outputs activate the Start input of the plasma power supply. Cut Control turns ON and remains ON until the M08 (Cut Off) command is executed in the part program.

HPR Hold Ignition – Activates the IHS Sync input of the THC or the Hold input of a plasma power supply. This output initiates Preflow During IHS and, if needed, a process change. This output is also used in a multiple-torch cutting system to synchronize the ignition of all torches.

HPR Pierce Control – Activates during piercing and remains ON until the Pierce Time elapses (set in the cut chart or Process screen). This output is used in some plasma power supplies to keep the plasma power supply from switching out of shield preflow while piercing. Pierce Control connects to the Pierce input at the plasma power supply.

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

HPR Remote on 1 – 4 – Applies power to the plasma power supply from a remote location.

Summary: setting up the plasma routine

1. Select the Plasma 1 and Plasma 2 processes in the Special Setups screen.
2. Select a marking process if the plasma torch will be used as a marker.
3. Assign the inputs and outputs for the plasma power supply on the I/O setup screen. For more information, see *I/O – Inputs and Outputs* on page 119.
    a. The I/O assignments for the plasma power supply depend on the number of torches and whether the CNC will be operating all the torches independently.
    b. For the CNC to operate each torch independently, numbered outputs and Station Select inputs must be assigned in the I/O setup screen. See *Station Setup* on page 155 for numbered I/O and Station Select inputs.
4. Define timer settings and other cutting parameters in the Process screen.

I/O, plasma power supply status, and diagnostics information

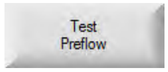
You can view status items for the plasma power supply, arc-on statistics, software revisions, temperatures, gas types, gas pressures, remote tools, and I/O.

Choose Setups > Diagnostics > HPR System.

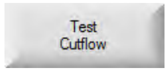
Plasma power supply status

Choose HPR Information to see status items for the plasma power supply on the HPR Information screen.

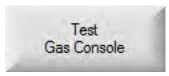
Remote tools (HPR diagnostics)

Test Preflow: Choose 

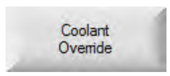
to test the preflow gases at the plasma power supply. This feature is used to set the inlet gas pressures under normal flow conditions to the recommended level.

Test Cutflow: Choose 

to test the cutflow gases at the plasma power supply. This feature is used to set the inlet gas pressures under normal flow conditions to the recommended level.

Test Gas Console: Choose 

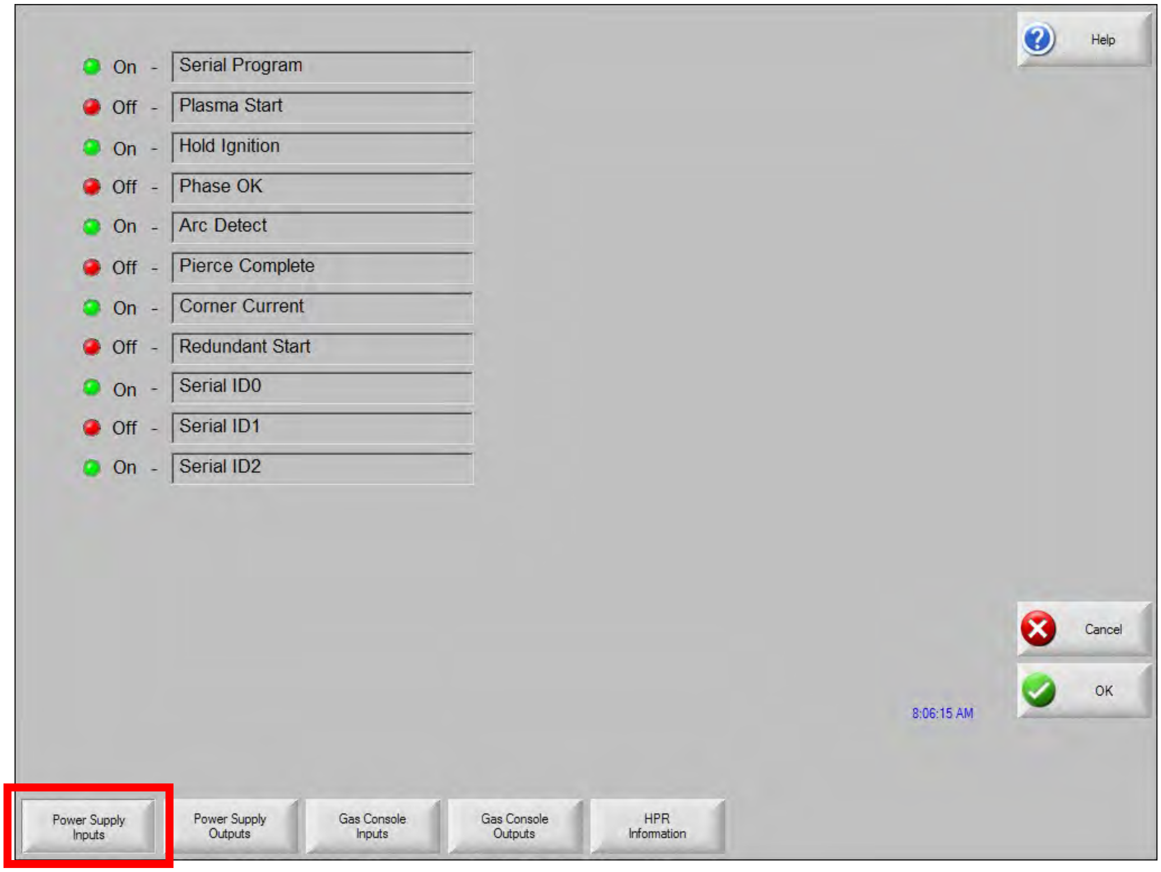
to perform automated tests for the HPR Auto Gas console. Contact an authorized service agent for use of these tests.

Coolant Override: Choose 

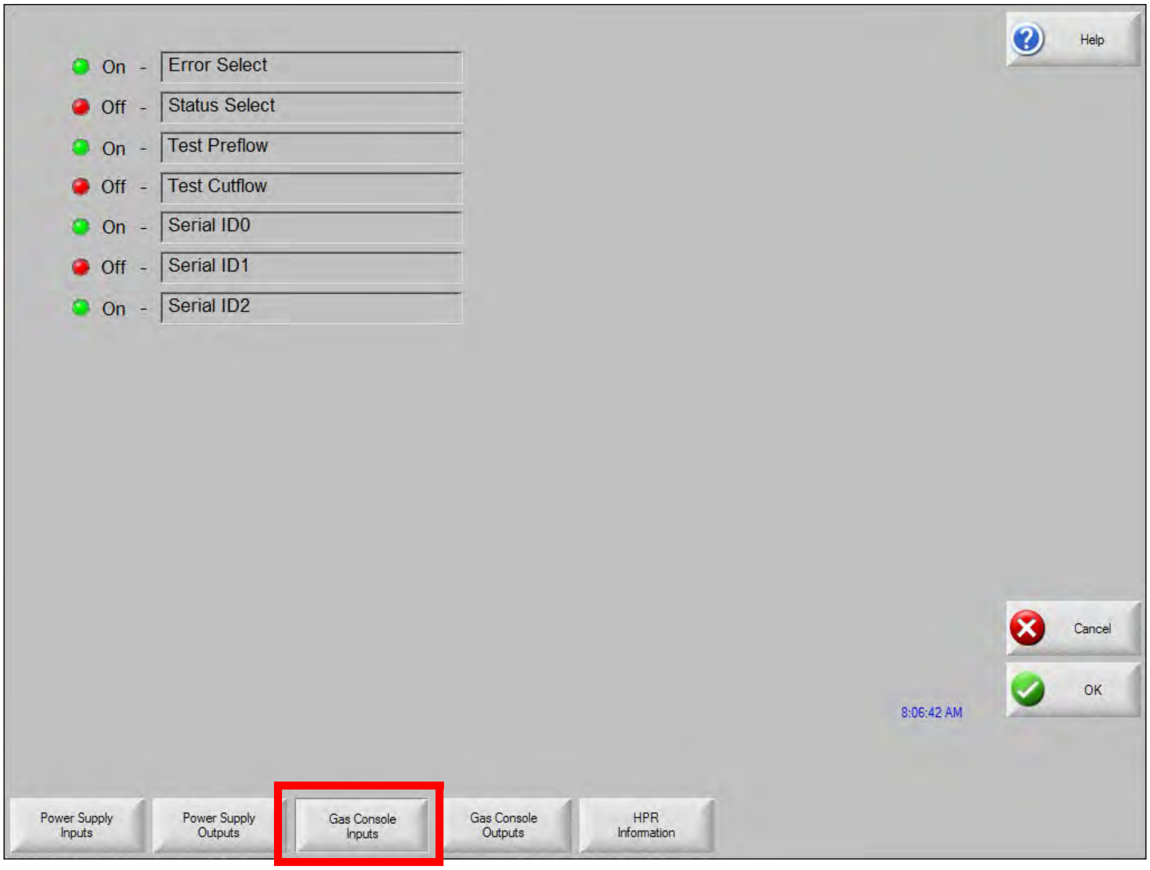
to override a coolant error and to test the coolant pump. This is useful for bypassing the error and purging the coolant line of air bubbles when you turn ON the plasma power supply.

Plasma power supply and gas console I/O

Plasma power supply inputs

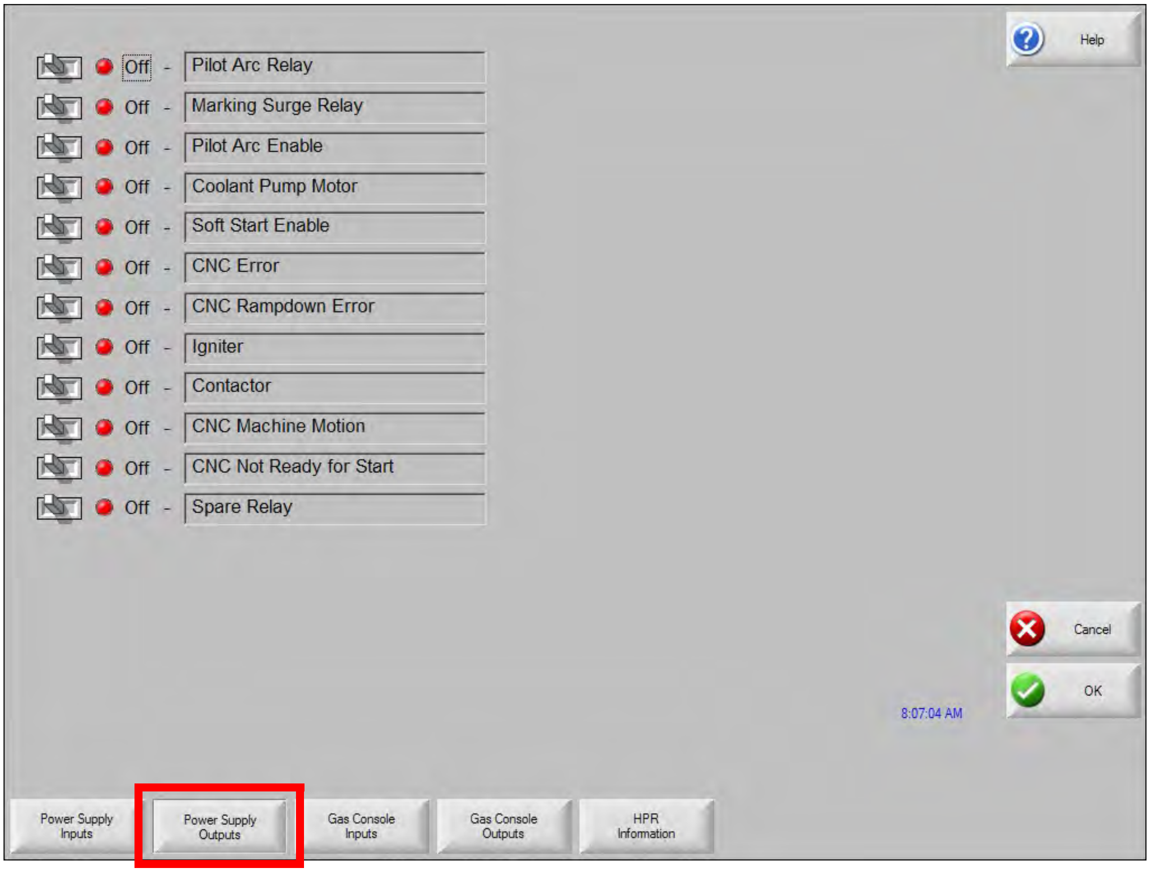


Gas console inputs



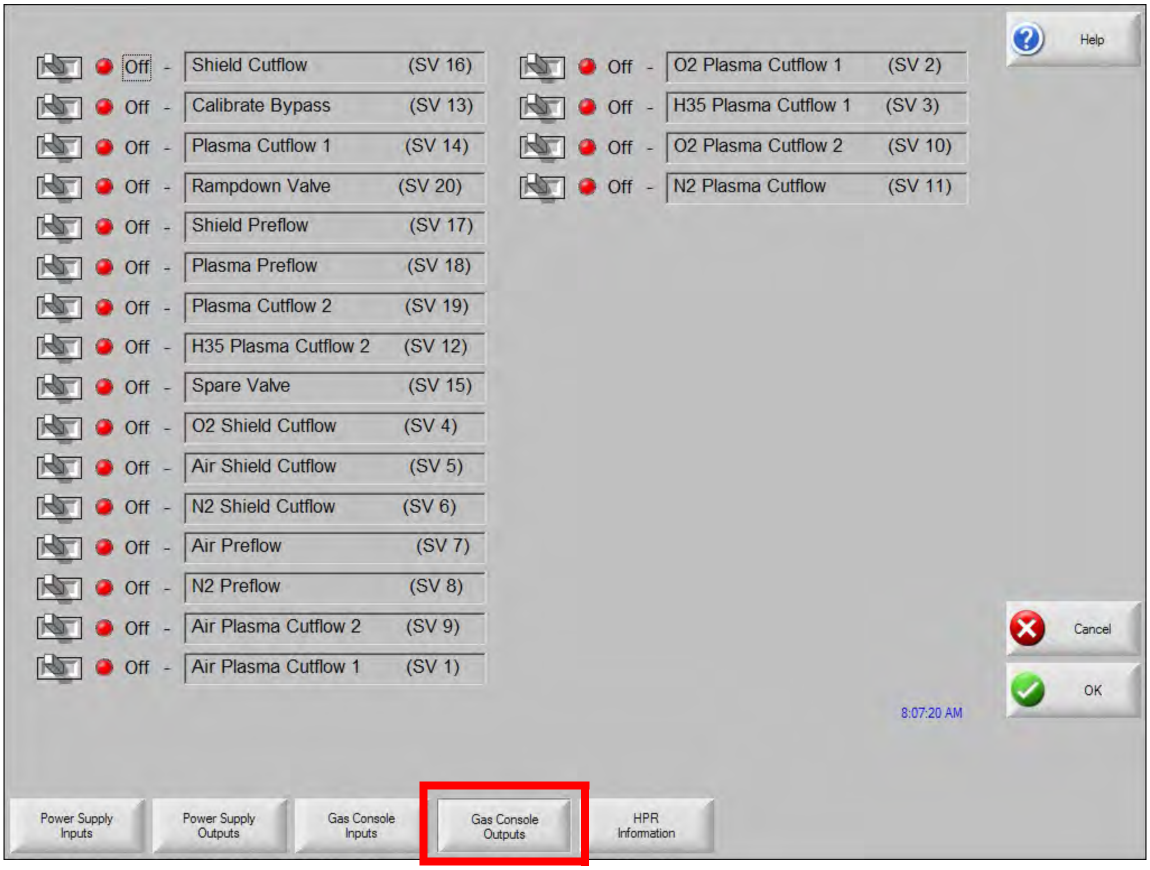
Plasma power supply outputs

A password is required to see the outputs screens.



Gas console outputs

A password is required to see the outputs screens.



The outputs screen shows the current status of the listed plasma power supply outputs.

Outputs for the plasma power supply cannot be activated through the diagnostic screen.

11

Phoenix Setup Files

About setup files

Hypertherm CNCs save the values that you enter on each screen in a setup file named Phoenix.ini.

Setup files can be copied to other EDGE Connect/EDGE Connect TC CNCs. For example, after configuring a CNC to run a particular cutting system, you can copy that setup file for the next installation of an EDGE Connect/EDGE Connect TC CNC, table, and tool. When you load a setup file into a CNC, that setup file that you load overwrites both the Phoenix.ini and Phoenix.bak.

CAUTION

Do not load a setup file from an EDGE Pro, MicroEDGE Pro or EDGE Pro Ti onto the EDGE Connect/EDGE Connect TC. Doing so could cause the cutting machine to become inoperable.

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

- **Phoenix.ini** is always the active setup file. Each time you make a change on a screen in Phoenix, the changes are saved to Phoenix.ini. In the Special Setups screen you can save a copy of Phoenix.ini with a different name for a backup or to use on another CNC. When you load a setup file with another name, the contents of that file overwrite the contents of Phoenix.ini and Phoenix.bak. Phoenix.ini is stored in the **C:\Phoenix** folder.
- **Phoenix.bak** is an identical copy of Phoenix.ini. The CNC saves Phoenix.bak each time it saves Phoenix.ini. If Phoenix.ini becomes corrupted, the CNC loads the settings in Phoenix.bak and creates a new Phoenix.ini. Phoenix.bak is stored in the **C:\Phoenix** folder.

- **Default.ini** is an important backup of Phoenix.ini and is used to restore settings if both Phoenix.ini and Phoenix.bak become corrupted. We recommend that you create the Default.ini when you commission a cutting system. Default.ini is stored in the **C:\Phoenix\Setups** folder.
- **Factory settings** represent the contents of the Phoenix.ini file as it was shipped from the factory. Each CNC has factory settings that represent the available features and axes on that CNC.

  Factory settings are stored internally in Phoenix but are not stored in a specific file. Factory settings can be very different from the settings on a CNC when it is commissioned in a cutting system at a manufacturing site.

Setup files from previous versions of Phoenix

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

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

⚠ CAUTION

Do not load a setup file from an EDGE Pro, MicroEDGE Pro or EDGE Pro Ti onto the EDGE Connect/EDGE Connect TC. Doing so could cause the cutting machine to become inoperable.

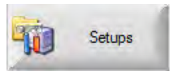
Save and load the setup file

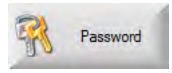
We recommend that you save the setup file 3 times when first commissioning a cutting system:

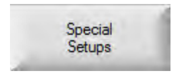
- Save the setup file to a USB memory stick with the name EndUserDate.ini. We also recommend that you save a copy to the hard drive on the CNC.
- Save the setup file using the Backup feature. See *Back Up and Restore the System* on page 197.
- Save the setup file with the name Default.ini on the CNC’s hard drive in the **C:\Phoenix\Setups** folder. This is the default location.

Save a setup file with a different name

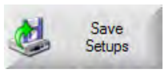
When you save a setup file with a different name, the CNC continues to use Phoenix.ini and does not update your saved setup file. You would do this to save a particular configuration.

1. From the Main screen, choose 

 > 

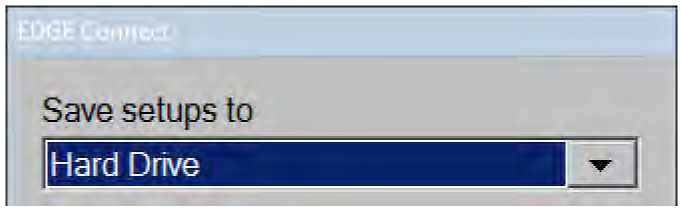
 > 

.

2. Choose 

.

3. Choose **Hard Drive**, **HD Backups**, or **Memory Stick** for the location.



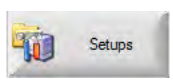
4. Enter a file name.

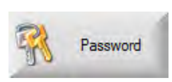
5. Choose **OK**.

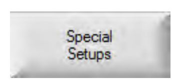
The CNC saves the file in the selected location.

Load a setup file

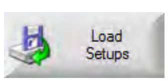
The setup file that you load becomes the active setup file (Phoenix.ini).

1. From the Main screen, choose 

 > 

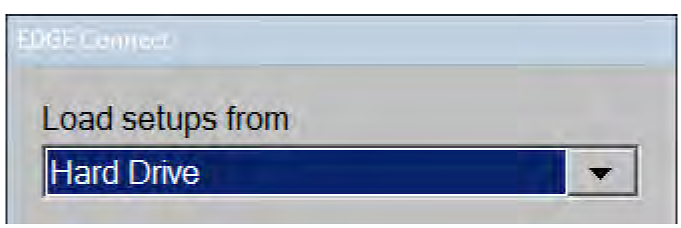
 > 

.

2. Choose 

.

3. Choose **Hard Drive** or **Memory Stick** for the location. The CNC lists the *.ini files that are in the root folder of the hard drive or the USB memory stick.



4. Enter a setup file name.

5. Choose **OK**.

Save system files for troubleshooting

The CNC can gather the Phoenix.ini, LastPart.txt, Phoenix.xml, Network.xml, SystemErrors.log, and other files to a USB memory stick or to a .zip file on a USB memory stick. These files have important information for troubleshooting the CNC. For more information, see *Save system files for troubleshooting* on page 204.

Also see *Back Up and Restore the System* on page 197 for other options.

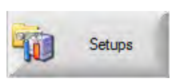
Save and load default settings

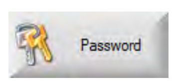
The CNC uses the Default.ini file to restore settings if both Phoenix.ini and Phoenix.bak become corrupted.

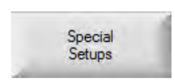
⚠ CAUTION

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

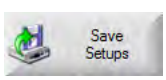
Save Default.ini

1. From the Main screen, choose 

 > 

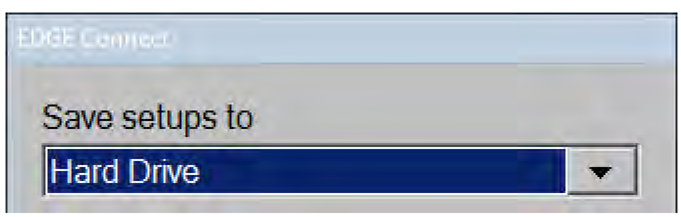
 > 

.

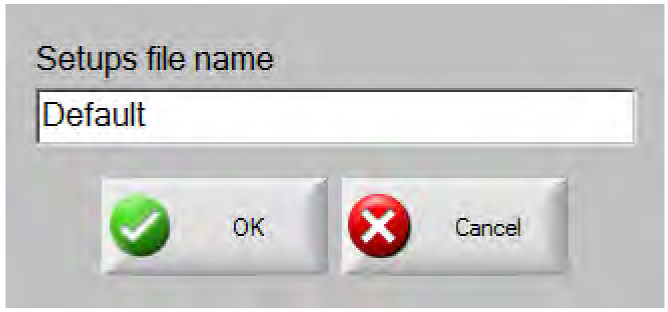
2. Choose 

.

3. Choose **Hard Drive** for the location.



4. Enter **Default** for the file name.

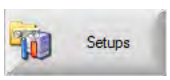


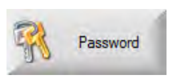
5. Choose **OK**.

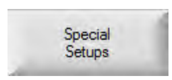
The CNC saves the Default.ini file in the **C:\Phoenix\Setups** folder.

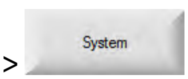
Load Default.ini

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

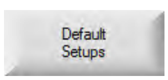
1. Choose 

> 

> 

> 

.

2. Choose 

.

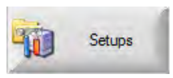
If you haven't saved a Default.ini file, then the soft key is labeled **Reset Setups** instead and lets you reset the CNC to factory settings.

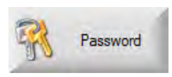
Reload factory settings

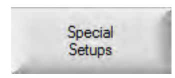
If your system is experiencing motion or other problems for which you cannot find a solution, reloading the factory settings gives you a known starting point for configuring the CNC to work with your cutting system.

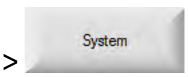
⚠ CAUTION

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

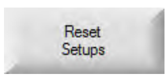
1. From the Main screen, choose 

> 

> 

> 

.

2. Choose 

.

   If the soft key is labeled **Default Setups** instead, then there is a Default.ini file in **C:\Phoenix\Setups**. Reload the default settings instead of the factory settings if possible.

12 Local Area Networking

Connect to a local area network with wireless or Ethernet

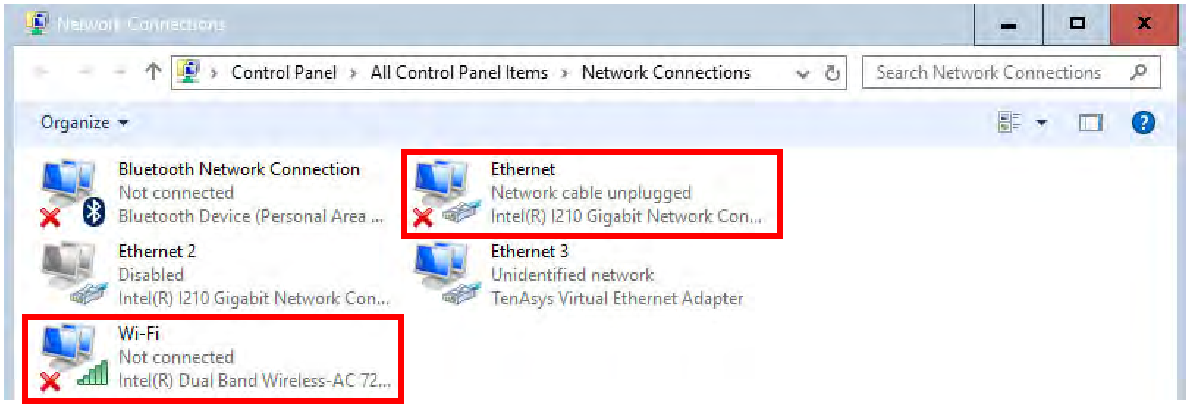
Use this procedure to connect to a local area network (LAN) with Ethernet or wireless for the first time.

Dynamic Host Configuration Protocol (DHCP) connections are expected for the purpose of this document. With a DHCP connection, the EDGE Connect/EDGE Connect TC automatically connects to the LAN and automatically acquires the Internet Protocol (IP) address.

Hypertherm cannot troubleshoot issues with password-protected network domains. If you must connect the CNC to a password protected network domain: **1.)** It is your own responsibility to manage your network credentials. **2.)** Your IT department should automate the network login process for the CNC operator. **For best results, connect the EDGE Connect/EDGE Connect TC to a trusted network workgroup that does not require login credentials.**

1. Connect a USB keyboard and USB mouse to the CNC to make LAN setup easier.
2. From the **Main** screen, choose **Setups** > **Password**.
3. Enter the Special Setups password.

4. On the **Special Setups** screen, choose **System** > **Network Tools** to open the **Network Connections** screen in Windows.

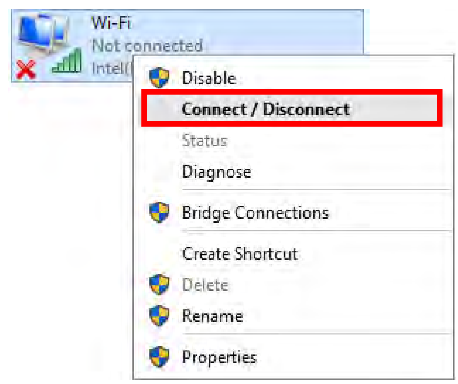


The EDGE Connect does not use the Bluetooth connection.

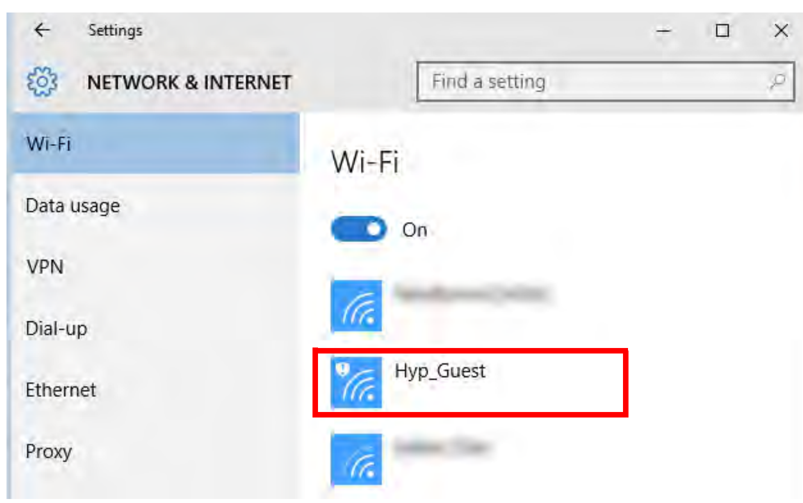
Do not change the Ethernet 2 (Intel) or Ethernet 3 (TenAsys) connections.

5. To connect to a LAN with Ethernet connect the Ethernet cable to the LAN connector on the CNC. To connect to the LAN with wireless continue to *step 6*.

6. To connect to the LAN with wireless, in the **Network Connections** screen, right-click the wireless connection, and then choose **Connect/Disconnect**.

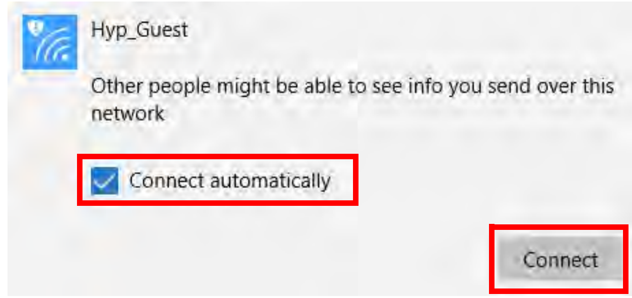


7. Choose the wireless network that you want to connect to.



8. If you want to connect automatically in the future, select **Connect automatically**.

9. Choose **Connect**.



You can also use the Windows 10 Network Connections Manager to locate your router or wireless network access point. Choose **Start Menu** > **Settings** > **Network & Internet**.

Notes

- The wireless network card in the EDGE Connect conforms to 802.11AC, 802.11N, and 802.11G standards and connects to any AC, N, or G wireless network. It provides at least wireless G or better performance for part program downloads, Remote Help™, or other network tasks. For best performance, use the fastest connection (802.11AC).
- The LAN must be set to broadcast its SSID for it to be visible as a connection. If you know the SSID, Windows 10 can also connect to hidden wireless networks.
- If the network connection wizard does not list any internal wireless networks, use another wireless device to make sure that the LAN is available and that the router or access point connection is available.
- Wireless signal strength decreases with distance.
- A wireless connection does not require 100% signal strength (shown as 4 solid bars on the screen) to operate correctly. If the Network Connections Manager shows 2 or 3 signal strength bars, the network connection is satisfactory.
- If the wireless signal strength is below 2 bars, move the wireless router or access point closer to the CNC, or add another access point closer to the CNC.
- For best wireless network performance, minimize the number of clients on the wireless network.
- Avoid connecting slower wireless devices to the network (for example, 802.11B devices). The performance of the entire network can decrease to that level. Contact your network administrator for additional support.

Map a drive

To access part programs from the LAN, you must map a network drive and folder in both Windows 10 File Explorer and Phoenix.

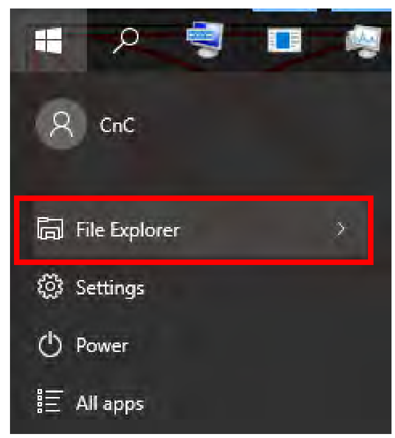
Part programs must be the only type of files in the mapped folder.

You must map the root folder. Phoenix does not navigate subfolders.

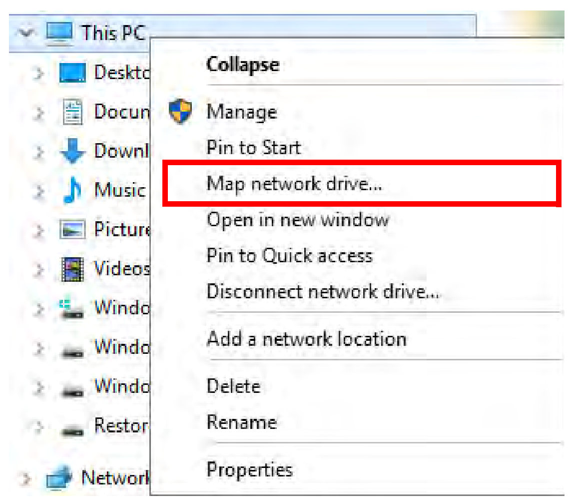
Make sure that the part program extension you are trying to access is in the file extensions area of the Special Setups screen.

1. Map the drive in Windows 10:

   a. Choose **Start Menu** > **File Explorer**.



   b. Right-click **This PC**, and choose **Map Network Drive...**.



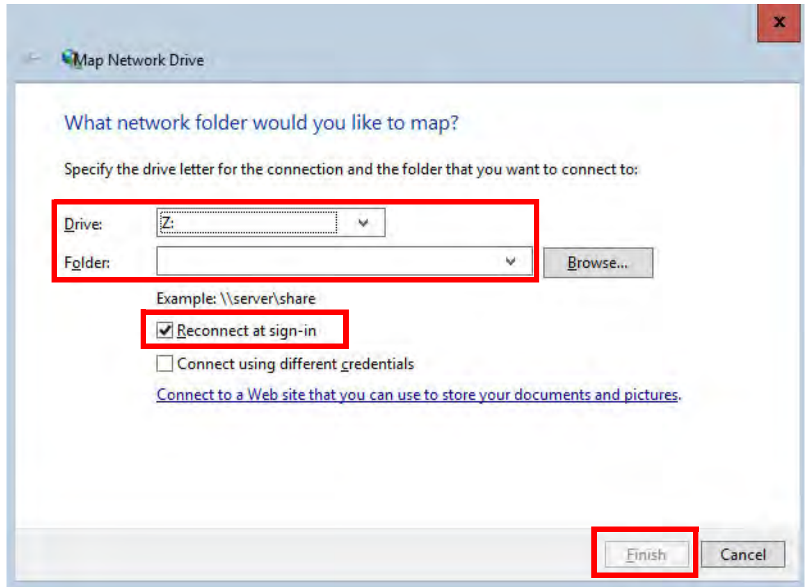
c. Choose a drive letter.

d. Choose the folder.

Make note of the folder path. When you connect to the folder in Phoenix, you need to enter the folder path **starting with the drive letter**.

e. Select **Reconnect at sign-in**.

f. Choose **Finish**.



2. Map the drive in Phoenix:

Before you begin, make sure that **Mapping Drives** and **Adding Folders** are set to **Allowed** in the Special Setups screen. (**Tip:** To avoid accidentally deleting files from your network, also set **Deleting Folders** to **Not Allowed**.)

ProNest CNC can use the mapped folders from Windows and Phoenix.

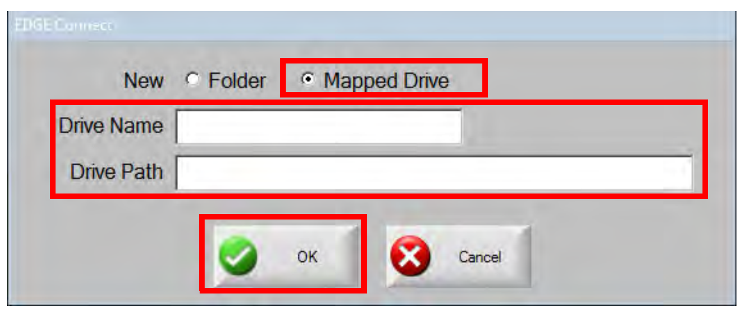
a. On the **Main** screen, choose **Files**.

b. Double-click or tap twice below the preview window where indicated to add the mapped drive folder that you created in Windows.

c. Choose **Mapped Drive**.

d. Type a **Drive Name**.

This is the name that appears in the **Load Files** list.



e. Type the absolute **Drive Path** to the folder that has the files using this format: [*DriveLetter*]:\[*DirectoryName*]\[*FolderName*] (**Example:** Q:\MyDirectory\CustomerParts)

Do NOT provide the UNC drive path (\\servername\foldername\).

f. Choose **OK**.

13

Back Up and Restore the System

Introduction

It is very important to create a backup so you can restore the cutting system to a previous operational state if files become corrupted or there is some other issue that requires you to restore the cutting system.

A backup file is a snapshot of the software installed on your CNC. The backup file includes the operating system software and the Phoenix software and configuration files.

What is backed up:

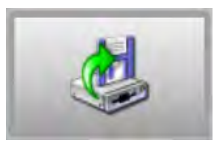
- Operating system software
  - Licensing information
  - Windows 10 operating system and configuration files
  - INtime operating system and configuration files
  - Hypertherm EtherCAT Studio
- The C:\Phoenix folder (Phoenix software and configuration files)
  - Phoenix software
  - Soft Op Con
  - Phoenix configuration files
  - Network configuration file
  - Launcher

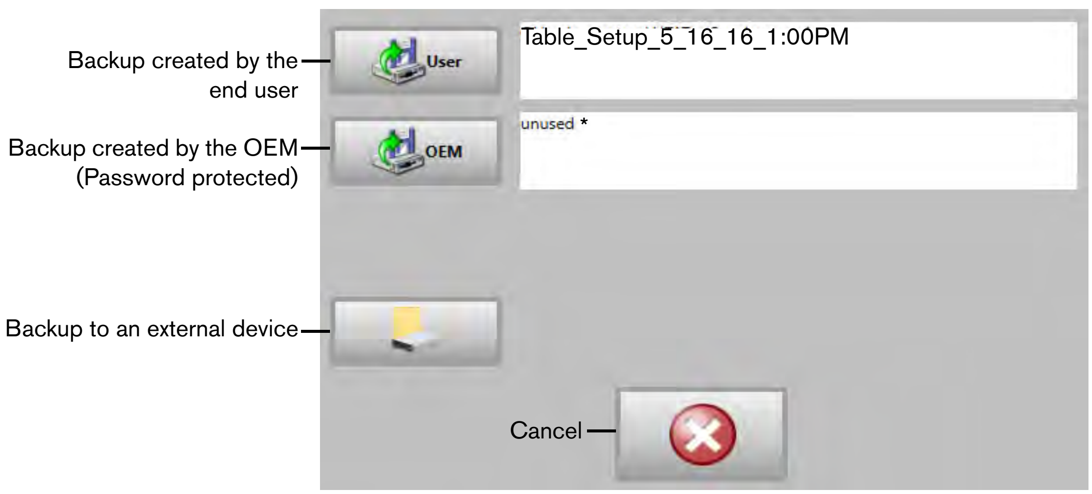
Backup

A new Backup and Restore utility has been created for the EDGE Connect/EDGE Connect TC CNCs. The new process is very similar to the previous method.

1. Restart the CNC.
2. Windows allows you to choose an operating environment during startup. When the **Choose an Operating System** screen shows, touch the screen anywhere.

   The Choose an Operating System screen shows for 5 to 15 seconds before Phoenix starts. Touch the screen anywhere to stop the timer.

3. Choose the second option, **Windows Preinstallation Environment**, to show the Backup and Restore screen.
4. Choose the Backup soft key  to show the Backup and Restore screen.

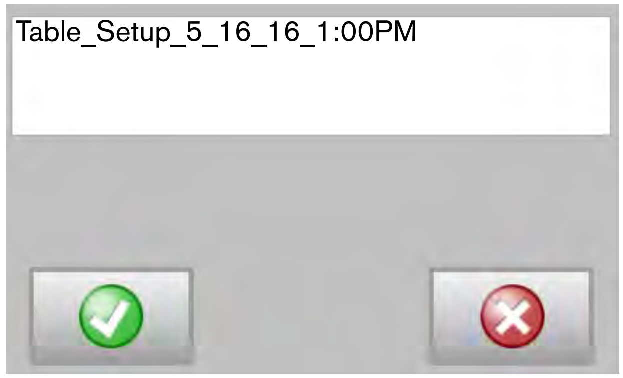


**unused* is the description you see until a backup has been created.

5. Choose the type of backup you want to create.
   - An OEM backup requires the Special Setups password.
   - Use the USB port to backup to an external device.

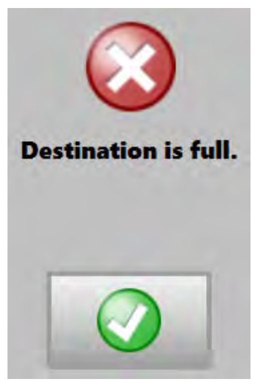
   To limit the amount of hard drive space that is used, only one OEM backup and one User backup can be saved to the hard drive. This process will overwrite a backup file if one was already on the hard drive.

6. Type a description, and then choose the OK soft key .



7. The CNC restarts when it completes the backup.

   If there is not enough space on the hard drive to create a backup, the following message appears. You will have to delete files to make space on the hard drive if you get this message.



Restore

The system can be restored from an end-user backup, OEM backup, factory default, or external device.

To restore the system:

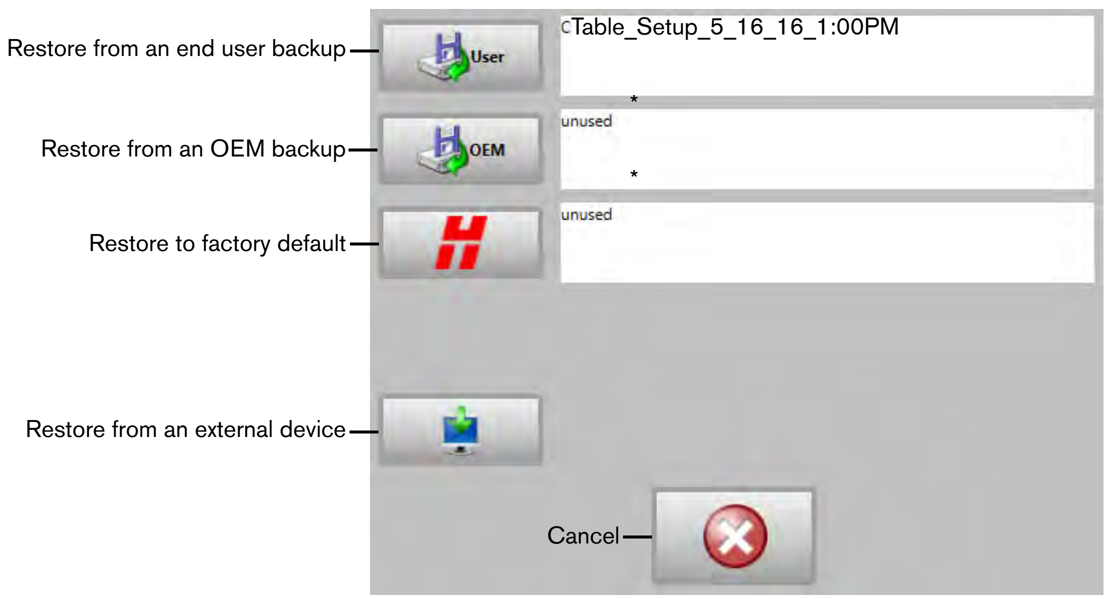
1. Restart the CNC.

2. Windows allows you to choose an operating environment during startup. When the **Choose an Operating System** screen shows, touch the screen anywhere.

   The Choose an Operating System screen shows for 5 to 15 seconds before Phoenix starts. Touch the screen anywhere to stop the timer.

3. Choose the second option, **Windows Preinstallation Environment**, to show the Backup and Restore screen.

4. Choose the Restore soft key  to show the Backup and Restore screen.

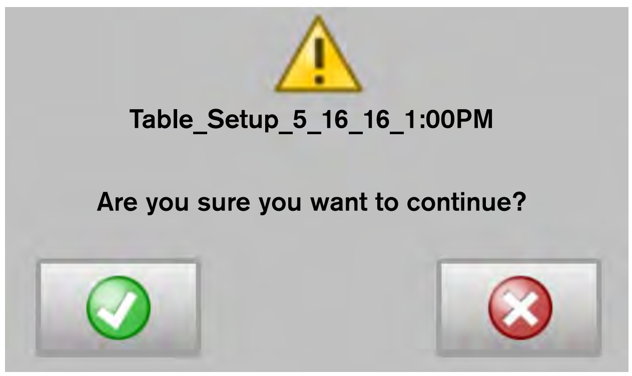


**unused* is the description you see until a backup has been created.

5. Choose the type of restore you want to do.
   - The C:\Phoenix directory is overwritten if you choose to restore from an OEM backup. The C:\Phoenix directory is restored to the state it was in when the backup was created.
   - The C:\Phoenix directory is maintained if you choose to restore from an end user backup. The current C:\Phoenix directory is maintained.
   - The C:\Phoenix directory is overwritten if you choose to restore from an external device through the USB port.

   To limit the amount of hard drive space that is used, only one OEM backup and one end user backup can be saved to the hard drive. This process will overwrite a backup file if one was already on the hard drive.

6. A message asks you to confirm that you want to restore. Choose the OK soft key to continue.



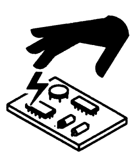
7. The CNC restarts when it completes the restore process.

14 Troubleshooting and Diagnostics

Hypertherm assumes that the service personnel that do troubleshooting and diagnostics are qualified technicians. Knowledge of final isolation troubleshooting techniques is also assumed.

In addition to being technically qualified, service personnel must do all tests with safety in mind. Read the safety instructions in your product's manual, the *Safety and Compliance Manual* (80669C), the *Waterjet Safety and Compliance Manual* (80943C), and the *Radio Frequency Warning Manual* (80945C).

CAUTION



Static electricity can damage circuit boards. Use precautions when handling printed circuit boards.

Wear a grounded wrist strap when handling PC boards.

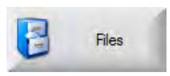
If you have questions or problems during servicing, call the Hypertherm Technical Service team listed in the front of the manual.

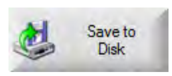
Save system files for troubleshooting

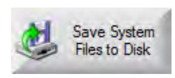
The CNC can gather the Phoenix.ini, LastPart.txt, Phoenix.xml, Network.xml, SystemErrors.log, and other files to a USB memory stick or to a .zip file. These files have important information for troubleshooting the CNC. Use this procedure to save the CNC's setup files or to gather files to send to Hypertherm Technical Service.

Because this method does not require a password, it is a convenient procedure for an end user to save the Phoenix.ini without being exposed to password-protected settings.

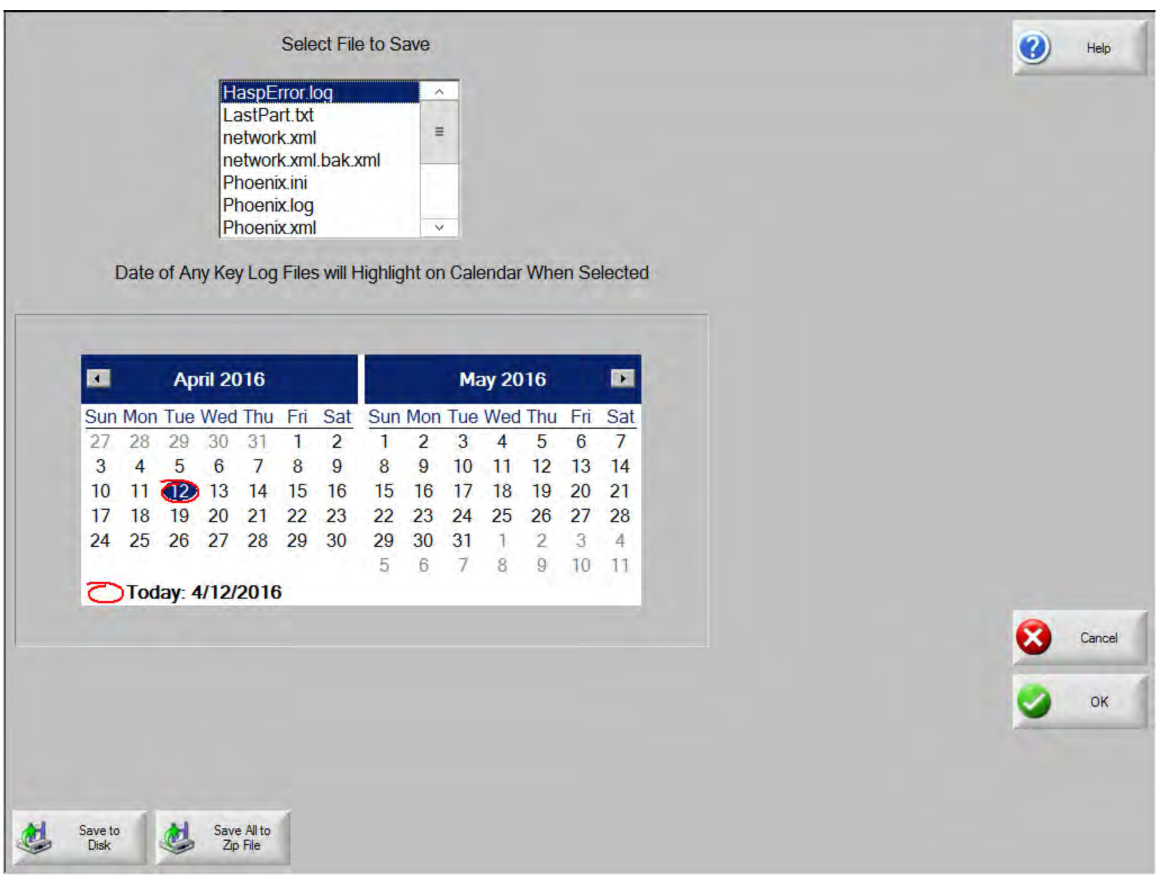
Also see *Back Up and Restore the System* on page 197 for other options.

1. Put a USB memory stick in a USB connector on the CNC.
2. From the Main screen, choose 

 > 

 > 

.
3. Select one or more files from the **Select File to Save** list.
4. Choose **Save to Disk**. Or, choose **Save All to Zip File** to save all of the files as **Phoenix.zip** on the USB memory stick.



Remote Help™ from a technician

Remote Help is an Internet-based diagnostic tool that uses screen-sharing to let a remote technician and a CNC operator work together to troubleshoot issues.

For more information about Remote Help see the *Remote Help™ for Hypertherm CNCs* Application Note (807560). Technical documentation is available at www.hypertherm.com/docs.

CNC troubleshooting

CNC messages

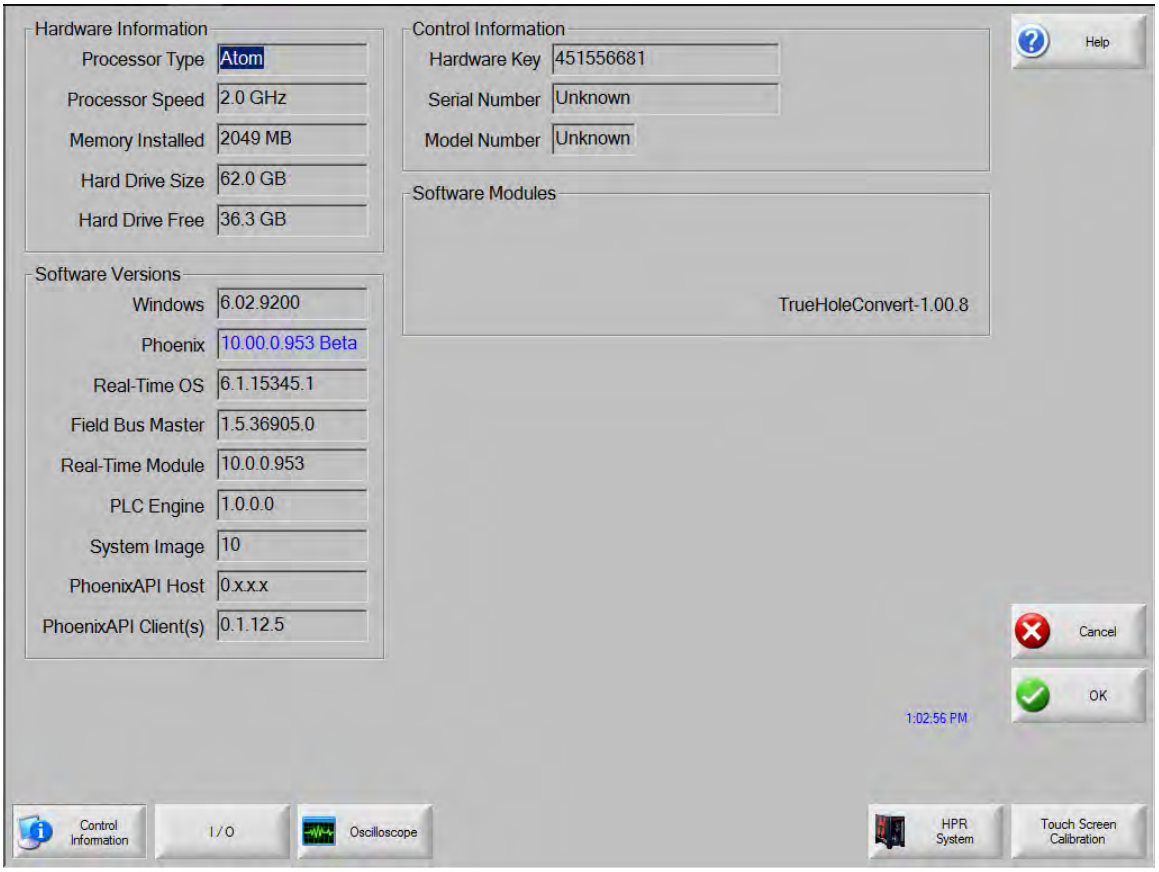
For help with CNC status messages, see *Status messages* on page 224.

For help with CNC pause, fault, and error messages, *CNC pauses, faults, and errors* on page 227.

CNC information screen

This Control Information screen displays the current software versions, the hardware configuration, and the software modules installed on the CNC. You must provide this information if you contact Hypertherm Technical Service for support.

From the Main screen, choose **Setups > Diagnostics > Control Information**.



Hardware Information: The Hardware Information section shows the current hardware configuration which includes the processor type, processor speed, memory installed, hard drive size, and hard drive free space.

Software Versions: The Software Versions section shows the current version of Windows®, Phoenix, the real-time operating system, the field bus master, the real-time module, the PLC engine, the system image, the Phoenix API Host, and the Phoenix API Client.

Control Information: The Control Information section shows the hardware key number, the CNC serial number, and the CNC model number.

Software Modules: The Software Modules section shows the optional software that has been installed, such as DXF-DWG Translator, McAfee® VirusScan® 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 shows you how many days or uses remain.

Limited Version: The Limited Version section shows only if the CNC is operating a trial version of software. The trial version of software is available for 90 days. Contact your cutting system manufacturer to have this timer reset if necessary.

Control Days Left: The Control Days Left section only shows if a timer is 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 cutting system manufacturer to have this timer reset if necessary.

OEM Days Left: The OEM Days Left section only shows if your cutting system manufacturer set up a timer to limit the number of days that the Phoenix software is valid.

> After the timer expires, a warning appears and you can no longer use the Phoenix software. For more information, see *The warning message Use of this control has expired appears* on page 259.

Touch Screen Calibration: Launches the touch screen calibration utility for tuning screen response.

General CNC troubleshooting

When you push the power button, the CNC does not power ON

Possible causes

- The external power supply or power cord are not connected to the CNC or power source.
- The external power supply is underrated.
- The external power supply connector is not compatible with the connector on the CNC.
- The terminal block is loose.
- The wall circuit breaker is not active.
- The power button and harness are defective.

Suggested actions

- Make sure that the external power supply and power cord are connected to the CNC and a power source.
- Use the external power supply that comes with the CNC.
- Make sure that the terminal block is tightly in the connector on the embedded-CNC and the wires are tight.
- Make sure that the wall circuit breaker is active.
- Remove **all** USB devices and USB cables inside and outside the CNC, such as:
  - USB extension cables
  - USB cable to the monitor
  - USB hard drives
  - USB memory sticks
  - USB hubs
  - Any other USB extension cables and devices

  Then turn off the CNC, and turn it on again. Install one USB device and cable at a time until you determine the cause of the problem.

- EDGE Connect TC only: Make sure that all of the power LEDs are ON on the PCB inside of the CNC. See *LEDs and test points for EDGE Connect TC* on page 211.
- EDGE Connect TC only: Push the auxiliary power button on the PCB in the CNC. See *Figure 33* on page 212. If the CNC starts, replace the power button and harness.
- EDGE Connect TC only: The power distribution or operator console PCB in the CNC is defective.

The power button is illuminated, but the touchscreen or monitor is black

Possible causes

- The DVI or HDMI cable is not connected to the embedded-CNC or touchscreen or monitor.
- Power is not connected to the touchscreen or monitor.
- The touchscreen or monitor is OFF.
- The touchscreen or monitor is in sleep mode.
- The power harness is defective. (EDGE Connect TC only)
- The power cord to your monitor is defective. (EDGE Connect only)
- The ambient temperature is out of the acceptable range.

Suggested actions

- Make sure that the DVI or HDMI cable is connected to the touchscreen or monitor and the embedded-CNC.
- Make sure that power is connected to the touchscreen or monitor.
- Make sure that the touchscreen or monitor is ON.

  For EDGE Connect TC, use the OSD remote control to make sure that the touchscreen is ON.

- Disconnect and reconnect power to the touchscreen or monitor to see if it comes out of sleep mode.
- Replace the DVI or HDMI cable.
- Replace the power harness. (EDGE Connect TC only)
- Replace the power cord to your monitor. (EDGE Connect only)
- Make sure that the ambient temperature is within the acceptable range. See *Environmental requirements* on page 30.

The warning message *The display settings for this system are not at the optimal values* appears

Possible causes

- A screen resolution and aspect ratio that makes Phoenix and the Soft Op Con fully visible on the screen is required. This warning message means that the monitor's current display settings do not meet this requirement.

Suggested actions

- See *Table 1* on page 32 for a list of required display settings.
  - To check the display settings that the monitor can support, see the instructions on page 33.
- If the monitor can support the requirements, choose **Yes**. For more instructions, see page 33.

If you choose **No**, Phoenix keeps the current display settings and does not show this warning message again.

For more troubleshooting information for display issues, see page 34.

The monitor displays a burn-in, ghost, or dim outline of a previously displayed image

Possible cause

The CNC was turned ON when not in use. Monitors retain charge in their LCD crystals when power has been ON for long periods of time.

Suggested action

- Turn OFF the CNC when not in use.

Previously mapped local network drives do not show when you try to load a part

Possible causes

- You don't have a network connection. See *Connect to a local area network with wireless or Ethernet* on page 191.
- The network drives are not mapped or are not a valid location. See *Map a drive* on page 194.
- You restored from an image that did not have mapped network drives.

Suggested actions

- Connect to a local network in Windows. See *Connect to a local area network with wireless or Ethernet* on page 191.
- Try to open the mapped network drive through the file explorer in Windows.
- Map the drives. See *Map a drive* on page 194.
- Contact your local network administrator.

You cannot download part programs over the local network (LAN or wireless)

Possible causes

- You don't have a network connection.
- You do not have the correct file location address mapped.
- You are trying to load part programs that are not supported.
- You do not have the file extension in the Special Setups screen in Phoenix.
- Your mapped location includes sub folders, and the part program is not in the root folder.

Suggested actions

- Connect to a network in Windows.
- Contact your local network administrator.
- Make sure that the correct file location address is mapped.
- Make sure that the part programs you are trying to load are supported.
- Make sure that you include the correct file extensions in the Special Setups screen in Phoenix. (Examples: CNC, DXF)
- Make sure that the mapped location for part programs is the root folder.

The CNC does not recognize a USB device

Possible causes

- The USB device is defective.
- The USB cable inside of the metal enclosure is disconnected from the embedded-CNC.
- The load from location is incorrect in Phoenix.
- Phoenix is slow to recognize the USB device.
- The USB connector on the CNC is defective.

Suggested actions

- Make sure that the USB device operates correctly in another computer.
- Make sure that the USB cable is connected to the embedded-CNC.
- Make sure that you select the correct load from location in Phoenix.
- Try the USB device in the embedded-CNC. See *EDGE Connect connector locations* on page 41 for USB connector locations.
- If the device can be seen in Windows, make sure that you wait for Phoenix to load completely before you plug in the USB device.
- Replace the defective USB connector or use a different USB device.

Test the PC board and connectors

LEDs and test points for EDGE Connect TC

The following LEDs illuminate as soon as 12 VDC is applied to the CNC. It is not necessary to push the power button.

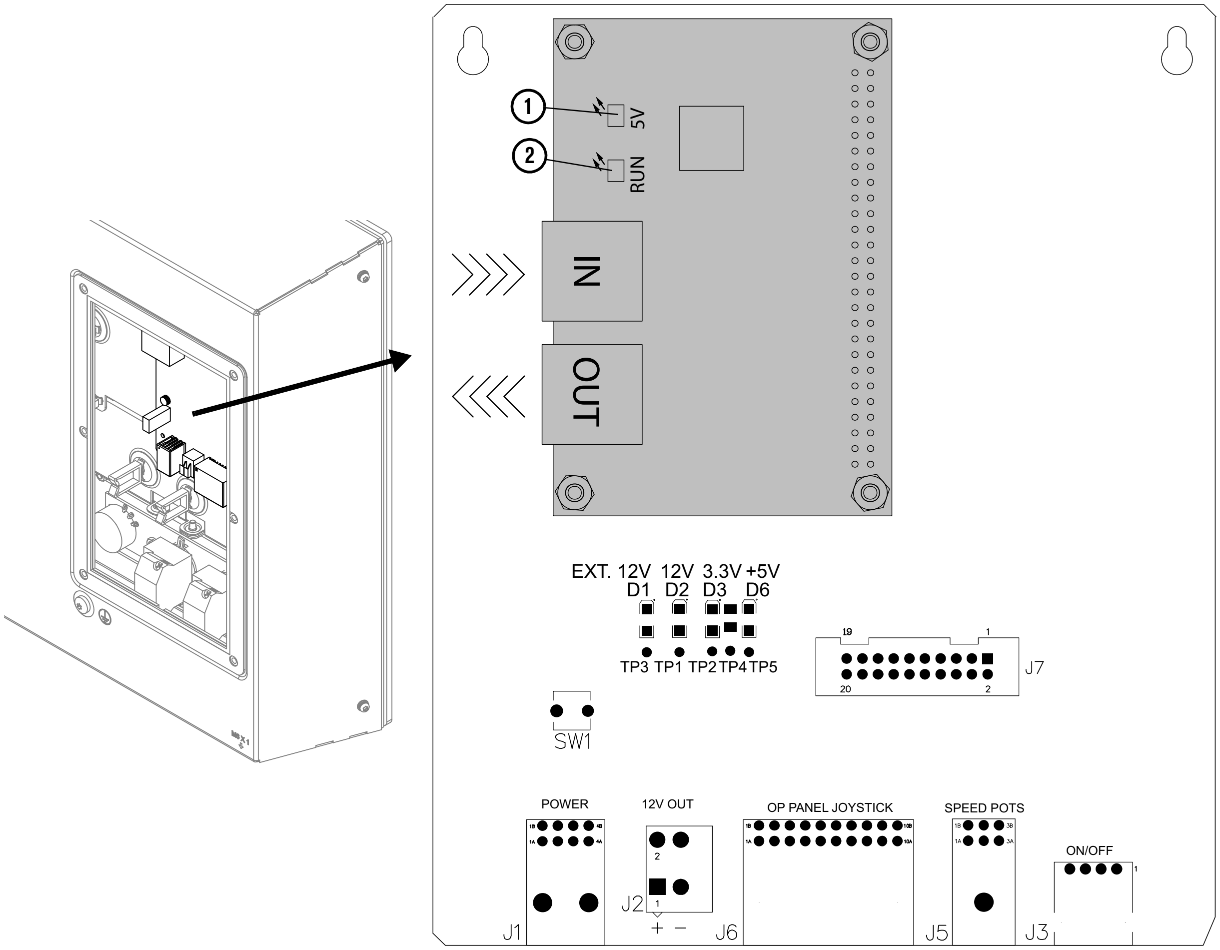
- D1
- D2
- D3
- D6
- 5 V (① in *Figure 33* on page 212)

The RUN LED stays OFF when you apply power to the CNC. When you push the power button on the CNC, the RUN LED blinks for a few seconds, and then remains illuminated when the field-bus has been established.

If none of the LEDs are on, make sure that the external power supply is working. Disconnect the cable from the CNC and look for 12 VDC on the power supply connector.

Alternatively you can measure the voltages between the test points. See *Table 3* on page 212.

Figure 33 – EDGE Connect TC PCB LEDs



1	+5 VDC	**D6**	+5 VDC
2	Run	**J1**	Power
SW1	Auxiliary power button	**J2**	12 VDC out (0.7 A maximum)
D1	External 12 VDC	**J6**	Operator panel joystick
D2	12 VDC	**J5**	Speed potentiometers
D3	3.3 VDC	**J3**	On/off

Table 3 – Test points

Test points	Voltage
TP1 (+) to TP4 (-)	12 VDC
TP5 (+) to TP4 (-)	5 VDC
TP2 (+) to TP4 (-)	3.3 VDC
TP3 (+) to TP4 (-)	12 VDC
J2-1 (+) to J2-2 (-)	12 VDC

Test the USB, EtherCAT, and LAN connectors and hardware operator console

Anything that is not connected will fail the test even if the connector on the CNC is functional.

1. Install USB memory sticks in all open connectors.

   Disconnect all USB hubs. Install single USB memory sticks only.

2. Make sure that an EtherCAT slave or the EtherCAT test PCB is connected.
3. Make sure that the LAN cable is connected to a wireless network.
4. Make sure that the wireless LAN is connected.
5. Choose **Setups** > **Passwords**.
6. Enter the **TESTHARDWARE** password.
7. Follow the instructions on the screen.

   For the hardware operator console test, you have 10 seconds to respond before the test times out.

Hardware operator console for EDGE Connect TC only

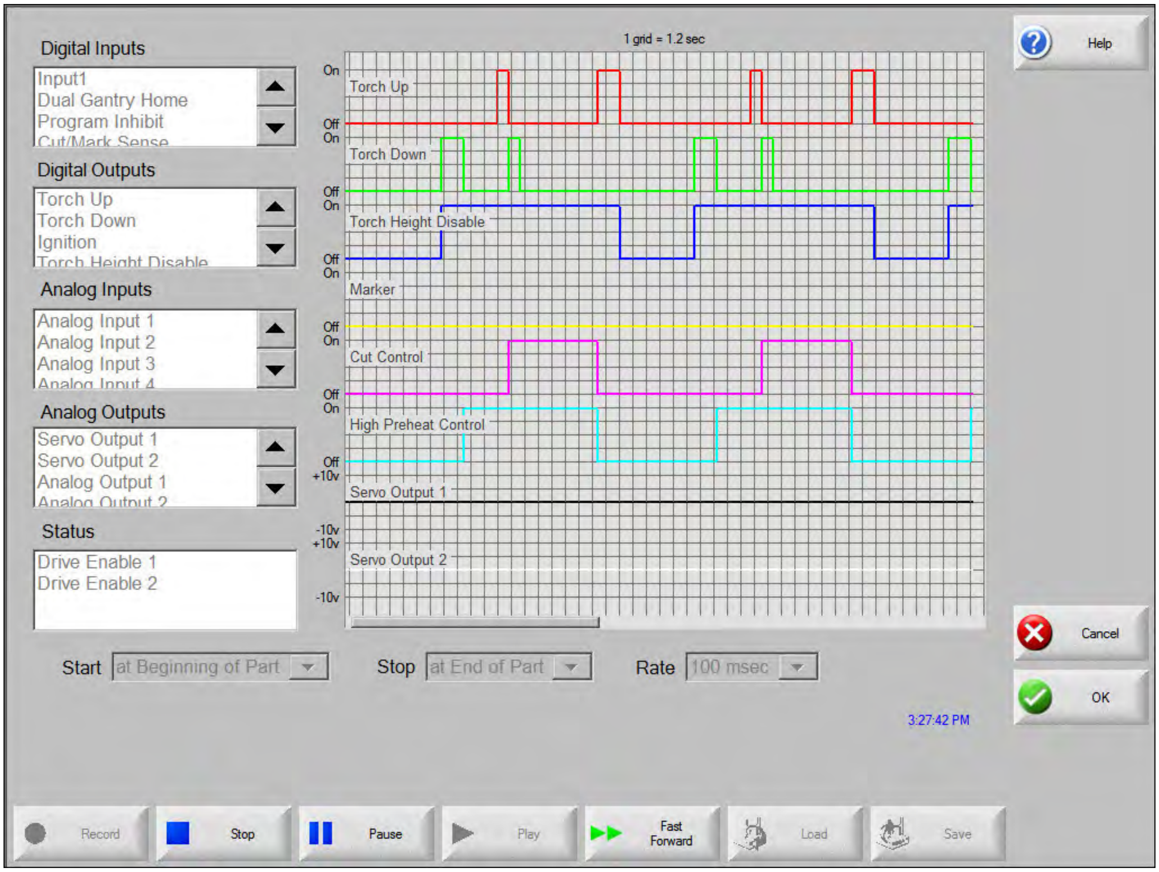
If any of the hardware operator console tests fail:

- Make sure that the I/O for the hardware operator console is set up correctly in Phoenix. See *I/O – Inputs and Outputs* on page 119.
- Replace the component or wire harness that is defective. See *Operator console parts* on page 335.

Log data with the oscilloscope

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. Go to **Setups > Diagnostics > Oscilloscope**.
2. Double-click an item in the boxes on the left of the screen to add it to the oscilloscope grid. You can add up to 8 items.
3. To remove an item from the grid, double-click it in the appropriate box.
4. In the **Start** list, select when you want the oscilloscope to begin recording.
5. In the **Stop** list, select when you want the oscilloscope to end recording.
6. In the **Rate** list, select the intervals at which the oscilloscope should record the data you selected.

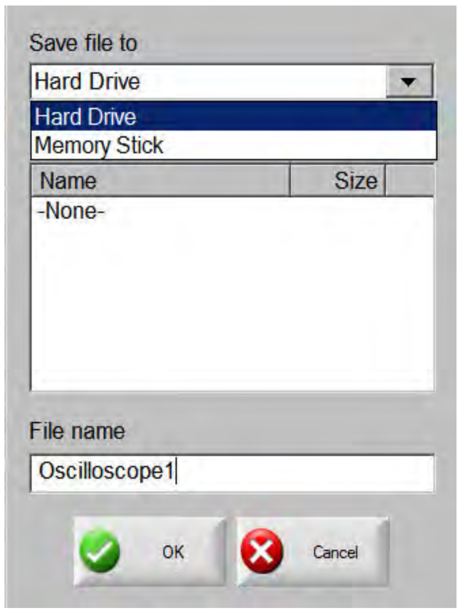
Save an oscilloscope file

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 to record 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 you execute the next nest.

To save the log file:

1. Choose the **Save** soft key.
2. Select the device where you want to save the file from the **Save file to** drop-down list.
3. Enter a name for the file in the **File name** field.
4. Choose **OK**.



Load an oscilloscope file

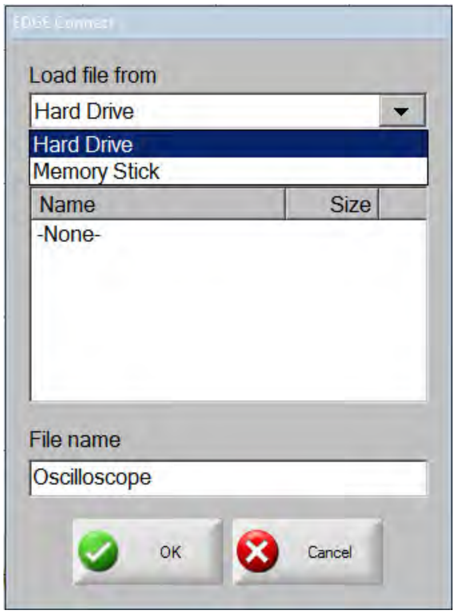
After you have saved an oscilloscope file, you can reload it and play it back on the CNC. This is the only way you can view this type of file.

In addition, Hypertherm Technical Service 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. Choose **Load** on the oscilloscope screen.
2. Select a device from the **Load file from** list.
3. Enter the name of the file you want to load in the **File name** field.

4. Choose **OK**.



View 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. Load the file. See *Load an oscilloscope file*.
2. Use the soft keys at the bottom of the screen to control the file.
   - Choose **Play** to start the file.
   - Choose **Stop** to end the file.
   - Choose **Pause** to stop the file temporarily.
   - Choose **Fast Forward** to speed up the file.

HPRXD plasma power supply troubleshooting

HPR error help

For help troubleshooting an HPR error message, see *Tool dialog box messages* on page 226.

HPR information screens

After communication has been established between the plasma power supply and the CNC, I/O and remote diagnostics screens are accessible through the Diagnostics screen. You can view status for the plasma power 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 plasma 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 plasma 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 power supply or gas console.

Outputs: Displays the current status of the outputs from the plasma power supply or gas console, but the outputs can't be activated in this screen. The Machine Setups password is required to access the Outputs screens.

Bypass an HPRXD on the EtherCAT network

When you turn ON the CNC, Phoenix opens and then starts the EtherCAT network. The EtherCAT network becomes operational once network communications are established among all of the slave devices defined in your network configuration. Your HPRXD plasma power supply is a slave device on the network. If you turn OFF the HPRXD, network communications will fail and Phoenix will show a field bus fault until you turn ON the HPRXD again.

Restarting the CNC will generate an EtherCAT communications error on the slave devices on the network. When the CNC starts again a communication fault message will be displayed until the field bus is *manually started* and can establish communication with the devices on the network.

If you *need* to keep an HPRXD plasma power supply turned OFF temporarily (for example, for maintenance), you can temporarily remove the HPRXD from the EtherCAT network configuration. Removing an HPRXD from the network configuration, known as *bypassing* an HPRXD, lets you continue to use the cutting system without field bus faults.

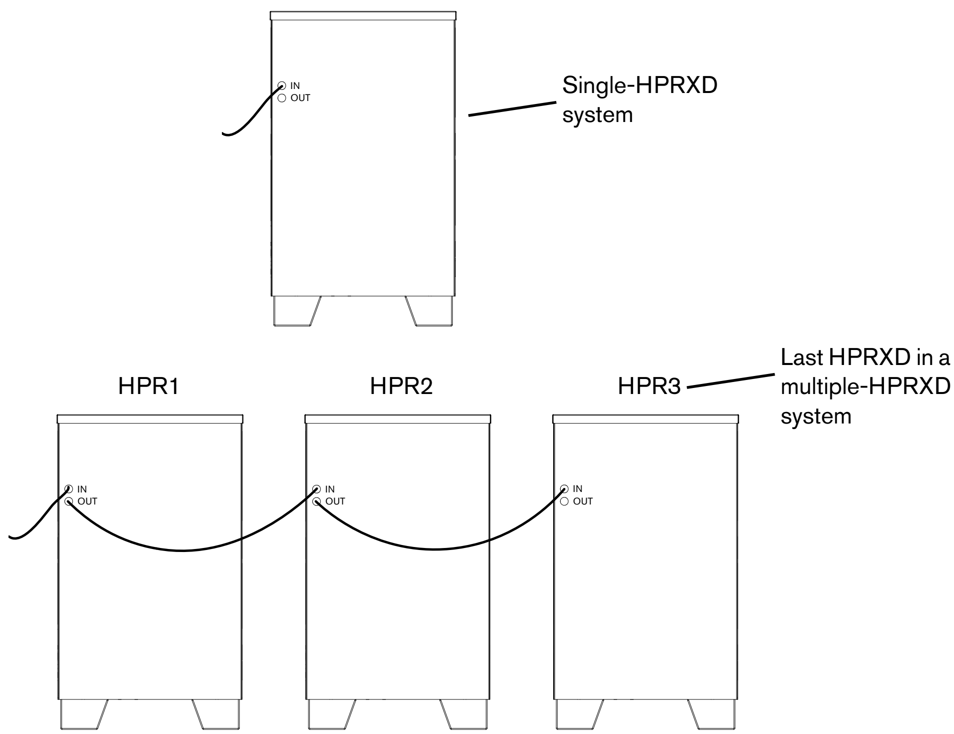
To bypass an HPRXD, you use the inline EtherCAT cable connector (208366) and bulkhead connector covers (108866) included in the HPRXD interface EtherCAT adapter kits (428489 and 428447). For more information, see the *HPR130XD®, HPR260XD®, and HPR400XD® EtherCAT® and VDC3 Board Installation* Field Service Bulletin (809260).

Only one HPRXD can be removed from the network at a time. If you need to remove more than one HPRXD, contact your cutting system manufacturer. Your network will need to be reconfigured without these HPRXDs, and then reconfigured again when you are ready to put the HPRXDs back on the network. For more information, see *Configure the EtherCAT Network* on page 89.

Bypass the HPRXD in a single-HPRXD cutting system

These instructions are also applicable if you have multiple HPRXDs and the HPRXD you want to bypass is the **last** HPRXD in physical order on the network. See *Figure 34*. To the EtherCAT network, they are the same.

Figure 34

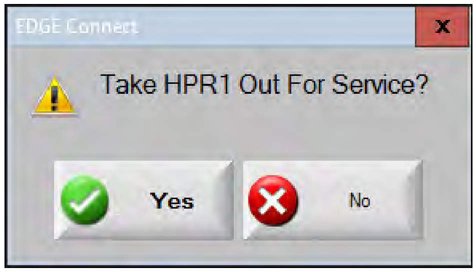


1. Turn OFF the CNC.

2. Turn OFF main power to the HPRXD.

3. Turn ON the CNC.

   After Phoenix opens, it begins to start the EtherCAT network. Then the following message appears.



The message shows the HPRXD you are bypassing. In this example, it is HPR1.

4. Choose **Yes**.

   Phoenix removes the HPRXD from the network configuration and starts the EtherCAT network.

If you restart the CNC while the HPRXD is bypassed, you will need to respond to the **Take HPR# Out for Service?** message again. Choose **Yes**.

Put the HPRXD back on the network in a single-HPRXD cutting system

These instructions are also applicable if you have multiple HPRXDs and the HPRXD you want to put back is the **last** HPRXD in physical order on the network. See *Figure 34* on page 219.

1. Turn OFF the CNC.

2. Turn ON main power to the bypassed HPRXD.

3. Turn ON the CNC.

   Phoenix adds the HPRXD to the network configuration and starts the EtherCAT network.

Bypass an HPRXD in a multiple-HPRXD cutting system

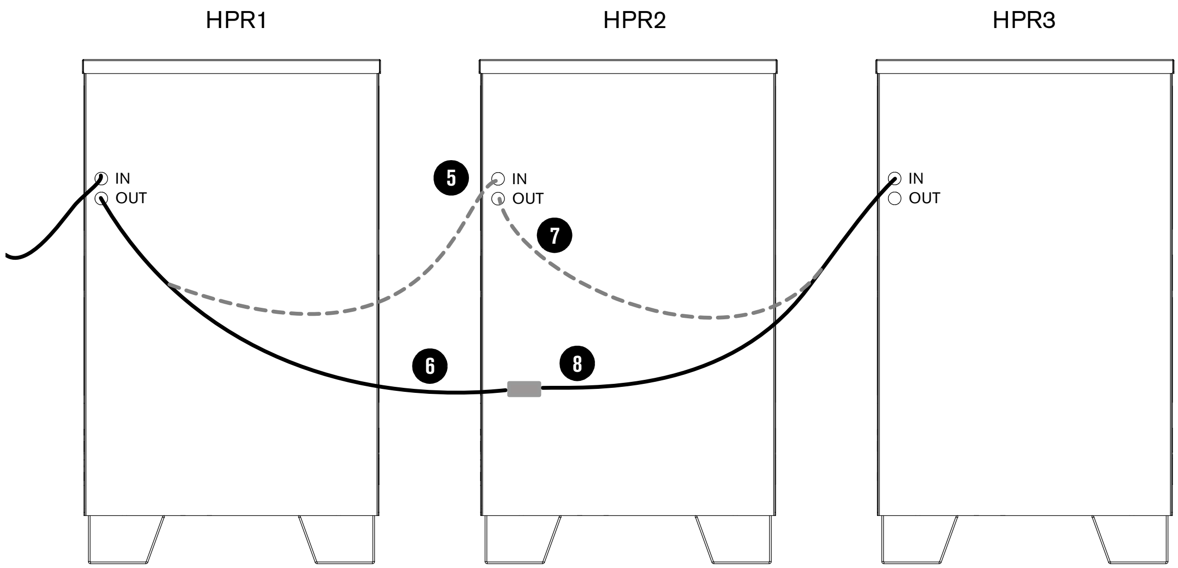
If the HPRXD you want to bypass is the **last** HPRXD in physical order on the network (see *Figure 34* on page 219), follow the instructions for a single-HPRXD cutting system instead. For more information, see *Bypass the HPRXD in a single-HPRXD cutting system* on page 219.

1. Turn OFF the CNC.

2. Turn OFF main power to the HPRXD you want to bypass.

3. Make sure that the bulkhead EtherCAT cable connectors on the rear panel of the HPRXD you want to bypass are clearly labeled. The top connector is IN and the bottom connector is OUT (when installed as instructed). See *Figure 35*.

Figure 35



4. Remove the inline EtherCAT cable connector (208366) from the rear panel of the HPRXD.

   Keep the two cable ties (343005).

5. Disconnect the IN EtherCAT cable from the HPRXD you want to bypass. See *Figure 35*.

6. Connect the IN EtherCAT cable to one end of the inline EtherCAT cable connector.

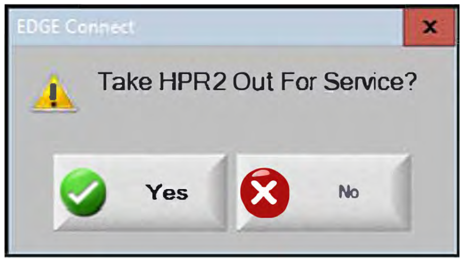
7. Disconnect the OUT EtherCAT cable from the HPRXD you want to bypass.

8. Connect the OUT EtherCAT cable to the other end of the inline EtherCAT cable connector.

9. Put a bulkhead connector cover (108866) on each of the bulkhead EtherCAT cable connectors to keep metal dust and other material out of the connectors.

10. Turn ON the CNC.

    After Phoenix opens, it begins to start the EtherCAT network. Then the following message appears.



> The message shows the HPRXD from which you disconnected the EtherCAT cables. In this example, it is HPR2.

11. Choose **Yes**.

    Phoenix removes the HPRXD from the network configuration and starts the EtherCAT network.

> If you restart the CNC while this HPRXD is bypassed, you will need to respond to the **Take HPR# Out for Service?** message again.
> Choose **Yes**.

Put an HPRXD back on the network in a multiple-HPRXD cutting system

> If the HPRXD you want to put back is the **last** HPRXD in physical order on the network (see *Figure 34* on page 219), follow the instructions for a single-HPRXD cutting system instead. For more information, see *Put the HPRXD back on the network in a single-HPRXD cutting system* on page 220.

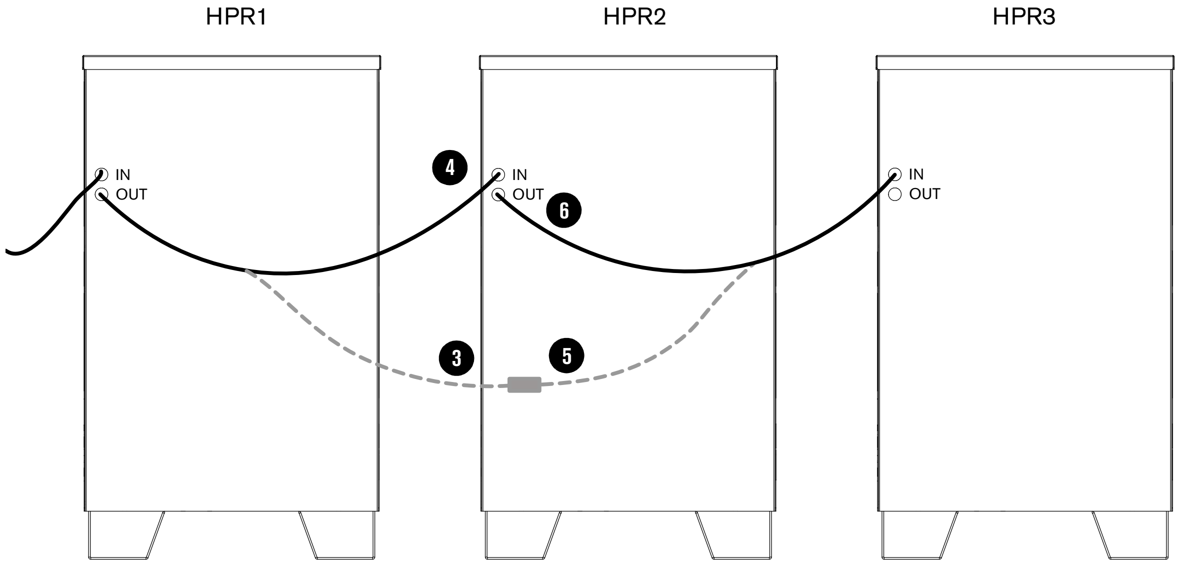
1. Turn OFF the CNC.

2. Remove the bulkhead connector covers from the bulkhead EtherCAT cable connectors on the rear panel of the HPRXD.

> Keep them for future use.

3. Disconnect the IN EtherCAT cable from the inline EtherCAT cable connector. See *Figure 36* on page 223.

Figure 36



4. Connect the IN EtherCAT cable to the IN bulkhead EtherCAT cable connector on the rear panel of the HPRXD.

5. Disconnect the OUT EtherCAT cable from the inline EtherCAT cable connector.

6. Connect the OUT EtherCAT cable to the OUT bulkhead EtherCAT cable connector on the rear panel of the HPRXD.

   Attach the inline EtherCAT cable connector (208366) to the rear panel of the plasma power supply again, using the two cable ties (343005) you kept earlier.

7. Turn ON main power to the HPRXD.

8. Turn ON the CNC.

   Phoenix adds the HPRXD to the network configuration and starts the EtherCAT network.

Status messages

Status messages appear in blue text within the status area below the part preview on the Main screen. See *Figure 37* on page 225. Status messages appear in order of priority to indicate the current sequence of events on the cutting system.

Plasma cut sequence status messages

These messages appear after you start a part program, and tell you the current status of the cut sequence. Many of the messages are informational, but some messages identify an error and require you to take action.

Status Message	Description	Occurs during a part program…	If the program pauses…
Traversing	The torch is moving to the next pierce point.	After Cycle Start and after each cut.	Take no action.
Lowering torch	The torch is lowering at the pierce point and the Torch Down output is activated.	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	The CNC is waiting for the Cut Sense input. Cut Sense is the Arc Transfer output from the plasma power supply or the Motion output from the THC.	After IHS is complete.	▪ Add the Cut Sense input to a Watch Window. ▪ Test the Cut Sense input at the CNC to make sure that it is working.
Piercing	The Pierce Control output is active.	During pierce.	Take no action.
Creeping	Creep Motion is occurring, after the pierce delay.	After the Pierce timer expires, and indicates the beginning of motion codes.	Take no action.

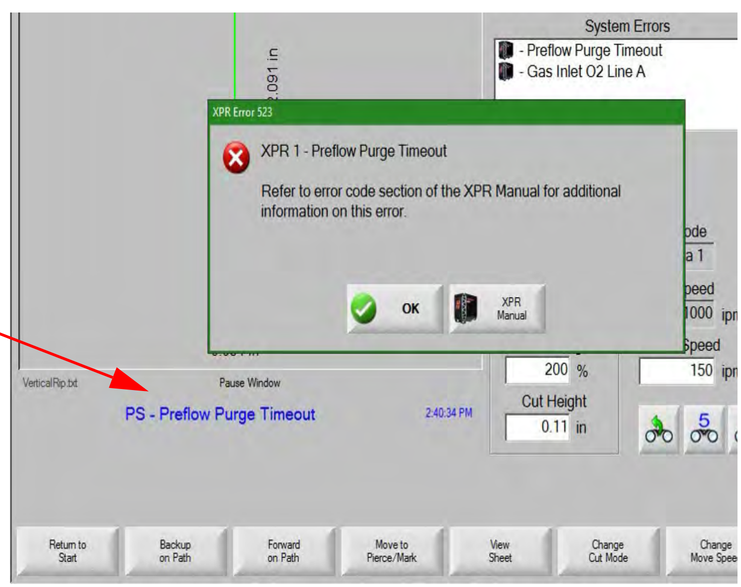
Status Message	Description	Occurs during a part program…	If the program pauses…
Cutting	Torch is cutting and motion is occurring.	Execution of the motion.	No action
Cut Sense Lost	The arc was lost while cutting.	The status message appears if a cut sense lost occurs during program execution (after an M07) or before executing the next M08.	▪ The arc lost electrical connection to the workpiece while cutting. ▪ For suggested actions, see *Cut Sense Lost* on page 232.
Raising Torch	Torch has reached the end of the cut and starts retracting from the workpiece.	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

Tool status messages

Like the plasma cut sequence, a tool's state can also appear in the status area. Since only one status message can display at once, **any errors that cause the tool to stop cutting have the highest priority to appear in the status area**. See *Figure 37*.

Figure 37 – Example of a tool error status message

Status message



To learn more about errors, see *Dialog box messages: pauses, faults, and errors* on page 226.

Dialog box messages: pauses, faults, and errors

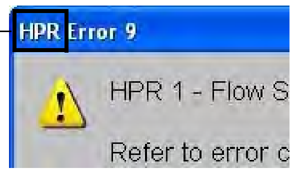
A dialog box displays on the CNC screen when the cutting system is not operating correctly. Faults and errors cause the arc to go out and cutting motion to pause. This prevents damage to the cutting system, unsatisfactory cut quality, or a decline in productivity.

Dialog box messages look different depending on whether they are generated by the CNC or by an external tool.

Tool dialog box messages

If the top left corner of the dialog box message on the CNC screen shows the name of an active external tool (such as HPR, XPR, Powermax, etc.), then that tool is not operating correctly.

Indicates that the message is from a tool (HPR)



Tool error dialog box example

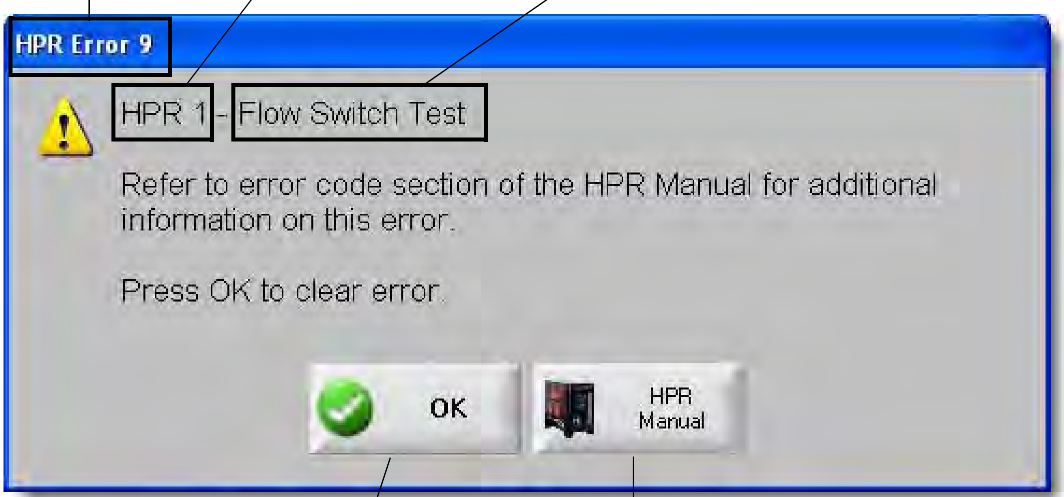
An example of a tool dialog box message is shown in *Figure 38*.

Figure 38 – Example of a tool error dialog box

Tool name and error number

Tool and station number

Error summary



Closes the dialog box

Shows the tool's instruction manual for troubleshooting steps

Troubleshooting tool errors

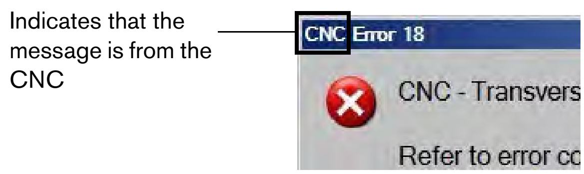
1. See the tool and error number in the top left corner of the tool error dialog box. See *Figure 38* on page 226.
2. Choose the tool's instruction manual button to view troubleshooting information. See *Figure 38* on page 226.

   If the tool was not manufactured by Hypertherm, refer to the troubleshooting guide provided by the tool's manufacturer.

3. Follow the troubleshooting steps for the error number from *step 1*.
4. Close the instruction manual screen.
5. Choose **OK** to close the dialog box.

CNC pauses, faults, and errors

Dialog box messages for CNC pauses, faults, and errors have **EDGE Connect** or **CNC** in the top left corner.



Troubleshooting CNC pauses, faults, and errors

CNC pauses, faults, and errors pause cutting motion for different reasons and have different effects. For help troubleshooting CNC dialog box error messages, see *Table 4* on page 228.

Table 4 – Differences between CNC pauses, faults, and errors

	Pauses	CNC faults	CNC errors
Example	See *Figure 39* on page 229.	See *Figure 40* on page 229.	See *Figure 41* on page 230.
Cause	▪ A problem occurred during cutting or marking, but no critical errors are present. ▪ Cause is summarized on the dialog box.	▪ A problem requires resolution to prevent damage to the cutting system, unsatisfactory cut quality, or a decline in productivity. ▪ Cause is summarized on the dialog box.	▪ A critical problem requires immediate resolution to prevent damage to the cutting system, unsatisfactory cut quality, or a decline in productivity. ▪ Cause is summarized on the dialog box.
Effects on motion and part programs	▪ Motion decelerates to a stop at a normal rate. ▪ The CNC maintains the current position and the part program pauses. ▪ Motion and program execution can continue without action, but the **fault could occur again if it is not fully resolved.**	▪ Motion decelerates to a stop at a fast or normal rate. ▪ The CNC maintains the current position. ▪ Motion and program execution **cannot continue until the fault is resolved**.	▪ Motion decelerates to an urgent stop prior to the CNC disabling the drive. The part program is canceled. ▪ The CNC clears the current position of all axes. Homing may be required after clearing the error. ▪ Motion and program execution **cannot continue until the error is resolved**.
Resolution	▪ Follow the instructions on the Reason Paused dialog box. ▪ Choose **OK** to close the dialog box message.	▪ For troubleshooting steps, see *CNC fault message reference* on page 231. ▪ Choose **OK** to close the dialog box message.	▪ For troubleshooting steps, see *CNC error message reference* on page 238. ▪ To clear the error with manual motion, choose **Manual** on the dialog box. ▪ To resolve a possible setup issue, choose **Setups** on the dialog box.
Next steps	If the problem has been resolved without canceling the part program, choose **Cycle Start** again to resume cutting or marking.	After the fault is cleared, choose **Cycle Start** again to resume cutting or marking.	Recover the part program by choosing **Resume Last Part** on the Main > Files screen.

CNC dialog box message examples

Figure 39 – Example of a pause dialog box

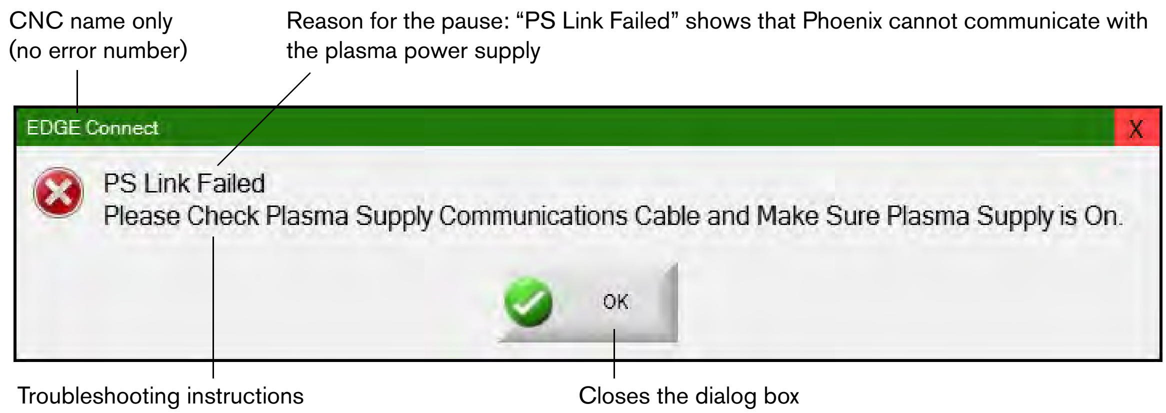


Figure 40 – Example of a fault dialog box

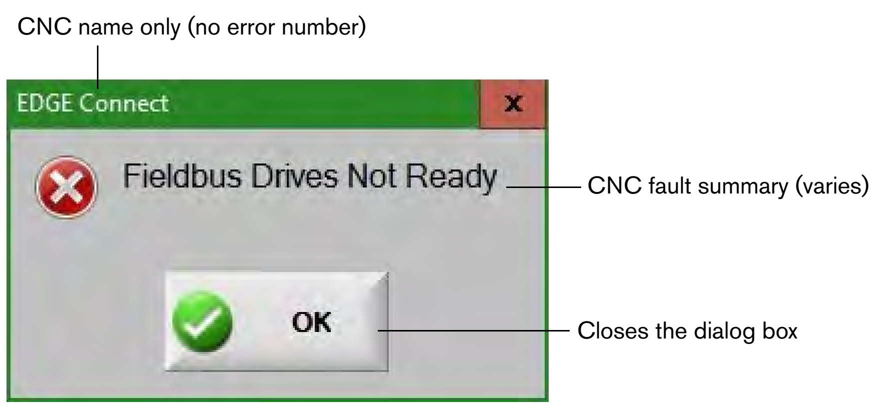
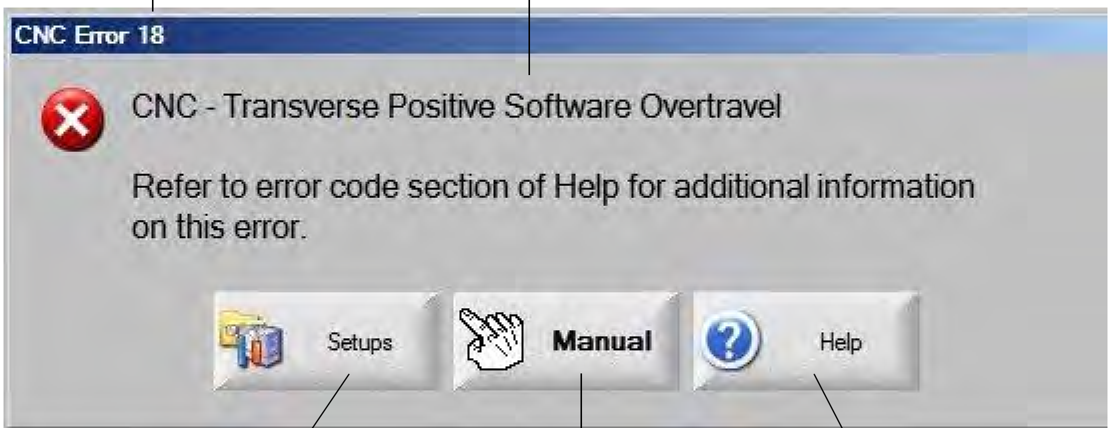


Figure 41 – Example of a CNC error dialog box

CNC error number

CNC error summary



Opens the Setups screen to let you resolve a possible setups issue

Lets you do manual motion to clear the fault

Opens the Error Message section in the online Help

CNC fault message reference

For all faults that result from input logic, refer to the Diagnostics Inputs screen for the location of the input and to make sure that the input is working correctly.

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 turned ON.
- There is a faulty cable or electrical connection between the Fast Stop input to the drive or I/O module 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 drive or I/O module.
- 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).

HPR Fault

The HPRXD reports errors to the CNC by sending a message over the EtherCAT network. Then the CNC prompts you with a dialog box message or a status message. These messages include an HPR Manual soft key so you can open the Help screen and view the troubleshooting information in the HPR instruction manual.

Possible cause

- The HPR has a fault and is reporting an error message to the CNC.

Suggested actions

- HPR errors are stored and displayed in the following areas of Phoenix:
    - Within the HPR Watch Window or within the HPR Diagnostic screen, the parameter is listed as Last Err (Last Error).
    - In the System Errors Watch window.

Cut Gas Lost

Possible causes

- The cut gas pressure dropped while cutting.
- There is a low pressure regulator setting.
- The cut gas source is 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.
- Make sure that the cut gas can flow through the torch when doing a test cut flow.

Cut Sense Lost

The arc lost electrical connection to the workpiece while cutting and before the execution of the M08 (or the end of the lead-out).

Possible causes

- The CNC part program has an excessive lead-out length.
- One or more cut sense inputs turned off while cutting and the Arc Off time expired before a cut sense input could turn on.

> For multiple torch applications, a Cut Sense Lost message only shows when a single torch loses its arc if the Stop On Single Arc Lost setting is enabled. If the Stop On Single Arc Lost is disabled, the program will continue to cut without showing a Cut Sense Lost message until all plasma torches lose their arcs while cutting.

Suggested actions

- Correct the conditions that caused the arc to lose electrical connection to the workpiece.
- Use the computer-aided manufacturing (CAM) software to decrease the lead-out length and generate a new program.
- If these actions do not resolve the issue, consider increasing the Arc Off time in the Process screen.

> Increasing Arc Off time can decrease consumable life. The maximum Arc Off time is 2 seconds.

Invalid Process Requested in Part Program

Possible causes

- The CNC part program has a cutting process (M36) or station code (M37) that Phoenix does not recognize.
- The CNC part program is calling a process variable value in the cut chart that does not exist. Examples include:
  - Material Thickness
  - Plasma/Shield Gases
  - Process Current
- A programming code is disabled in the Cutting setup screen. Examples include:
  - G59 Process codes
  - M07 HS/M08 RT
  - Process Select (M36)
  - Station Select (M37)

Suggested actions

- Make sure that the station is in Program mode in the Soft Op Con.
- Make sure that the Station Configuration screen is correctly configured for your cutting system.
- Update Phoenix and cut charts.
- If the CNC part program includes G59 V5xx Fvalue process override codes, make sure that the parameters match the values in the cut chart.
  - Material Thickness
  - Torch Type
  - Plasma/Shield Gas Type
  - Cutting Current
- Make sure 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.
- Make sure 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.

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 and the drive or I/O module.

Suggested actions

- Check the external device that turns on the Remote Pause input.
- Check cables and wiring between the Remote Pause input and the drive or I/O module.
- 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.
- There is a faulty cable or electrical connection between the Safety Mat input and the drive or I/O module.

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 drive or I/O module.
- 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 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 be active. If the Rail limit was reached after the Transverse limit, then motion will only occur in the opposite direction of the Rail axis first.
- If problem persists, check the Minimum and Maximum Travel Limit settings in the Axis 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 and the drive or I/O module.

Suggested actions

- Raise the torch and reinstall the torch collision device used on the THC lifter.
- Check the torch collision device to make sure that the device is working correctly.
- 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, make sure that the switch is functioning correctly.
- Check the cables and wiring between the torch collision device and the drive or I/O module.
- 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).

Field Bus Drives Not Ready

A *field bus* is sometimes referred to as a *fieldbus* or a *network*.

Possible cause

- The current drive status is not valid.

Suggested actions

- Look at the drive for any errors, then disable and re-enable the control in Phoenix to reestablish communication.
- Remove power from all of the slaves on the network, then restore power to all of the slaves and restart the network.

Take HPR# Out for Service?

Possible causes

- The EtherCAT cable to the HPRXD plasma power supply shown in the message is disconnected or damaged.
- Main power to the HPRXD plasma power supply shown in the message is OFF.
- There is a problem with the EtherCAT plasma interface board inside the HPRXD.

Suggested actions

If you *want* to remove the HPRXD from the EtherCAT network, see *Bypass an HPRXD on the EtherCAT network* on page 218.

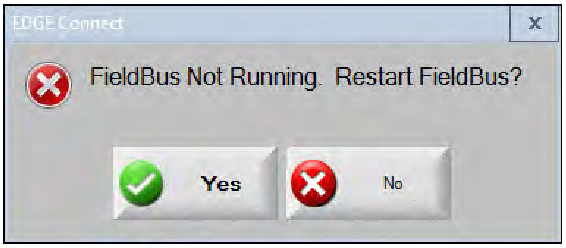
If you *do not want* to remove the HPRXD from the EtherCAT network:

a. Choose **No**. Phoenix begins to start the EtherCAT network.

b. Choose **Cancel** to stop network start-up.

c. As necessary, reconnect or replace the EtherCAT cable. Or, turn ON main power to the HPRXD.

d. On the Main screen in Phoenix, choose **Manual**. The following message appears.



e. Choose **Yes**.

f. Phoenix starts the EtherCAT network.

If the HPRXD error caused other errors, one or more error messages appear before Phoenix finishes starting up the EtherCAT network. Click **Manual** to close each message. Phoenix starts the EtherCAT network.

If Phoenix continues to show the **Take HPR# Out for Service?** message, the problem may be the EtherCAT plasma interface board. Contact your cutting system manufacturer.

CNC error message reference

Error 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.
- If this is a new installation:
  - The Servo Error Tolerance is set too low.
  - The gains are set too low or too high in the drives.
  - The Maximum Machine Speed is set too high.
  - The Acceleration is set too high.
  - There are other incorrect settings in the drive.

Suggested actions

- Correct the setting causing the error.
- Contact the drive manufacturer for further troubleshooting.

Error 2) Rail Position Error

The position of the Rail axis is behind the commanded position by more than 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.

Error 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

- Refer to the Transverse Position Error (Error 1) for information on possible causes.

Suggested actions

- Refer to the Transverse Position Error (Error 1) for information on suggested actions.

Error 4) Rotate Position Error

The position of the Rotate axis is behind the commanded position by more than 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.

Error 5) Tilt Position Error

The position of the Tilt axis is behind the commanded position by more than 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.

Error 6) CBH Position Error

The position of the CBH axis is behind the commanded position by more than 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.

Error 7) THC Position Error

The position of the THC axis is behind the commanded position by more than the Servo Error Tolerance value.

Possible causes

- There is a mechanical bind in the lifter.
- The drive amplifier has produced a fault.
- In a Yaskawa drive, the Forward/Reverse External Torque Limit input is active all of the time.
- 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 action

- Positive motion of the THC axis will lower the torch.

Error 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 1 of the 2 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 drive or I/O module.
- There is no DC voltage to the machine's inputs.
- The drive or I/O module output is latched on.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch (Normally Closed or Normally Open).

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 make sure that the switch is functioning correctly.
- Test the limit switch inputs in the Diagnostics Inputs screen.
- Check the cables and wiring between the limit switch and the drive or I/O module.
- Make sure that the cutting table is turned ON.
- 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 Diagnostics Inputs screen for the location of the Transverse Positive Hardware Overtravel switch and to make sure that the input is working correctly.
  - Make sure that Use Hardware Overtravels is set to Yes on the Transverse axis screen.

Error 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 drive or I/O module.
- There is no DC voltage to the machine's inputs.
- There is a faulty input to the drive or I/O module.
- The drive or I/O module output is latched on.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch (Normally Closed or Normally Open).

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 make sure that the switch is functioning correctly. Test the limit switch inputs in the Diagnostics Inputs screen.
- Check the cables and wiring between the limit switch and the drive or I/O module.
- Make sure that the cutting table is turned ON.
- Make sure all cables are correctly installed at the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Rail Positive Hardware Overtravel switch and to make sure that the input is working correctly.

Error 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 drive or I/O module.
- There is no DC voltage to the machine's inputs.
- There is a faulty input to the drive or I/O module.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch (Normally Closed or Normally Open).

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 make sure that the switch is functioning correctly. Test the limit switch inputs in the Diagnostics Inputs screen.
- Check the cables and wiring between the limit switch and the drive or I/O module.
- Make sure that the cutting table is turned ON.
- Make sure all cables are correctly installed at the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Transverse Negative Hardware Overtravel switch and to make sure that the input is working correctly.

Error 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 drive or I/O module.
- There is no DC voltage to the machine's inputs.
- There is a faulty input to the drive or I/O module.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch (Normally Closed or Normally Open).

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 make sure that the switch is functioning correctly.
- Test the limit switch inputs in the Diagnostics Inputs screen.
- Check the cables and wiring between the limit switch and the drive or I/O module.
- Make sure that the cutting table is turned ON.
- Make sure all cables are correctly installed at the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Rail Negative Hardware Overtravel switch and to make sure that the input is working correctly.

Error 18) Transverse Positive Software Overtravel

Software overtravel 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 of the software limit.

Error 19) Rail Positive Software Overtravel

Software overtravel 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 of the software limit.

Error 23) Transverse Negative Software Overtravel

Software overtravel 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 of the software limit.

Error 24) Rail Negative Software Overtravel

Software overtravel 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.

Error 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 drive or I/O module.
- There is no DC voltage to the machine's inputs.
- There is a faulty input to the drive or I/O module.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch (Normally Closed or Normally Open).

Suggested actions

- Jog the Tilt axis in the opposite direction to move off of the limit switch.
- Check the hardware limit switch on the bevel head for damage and make sure that the switch is functioning correctly.
- Test the limit switch inputs in the Diagnostics Inputs screen.
- Check the cables and wiring between the limit switch and the drive or I/O module.
- Make sure that the cutting table is turned ON.
- Make sure all cables are correctly installed at the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Tilt Positive Hardware Overtravel switch and to make sure that the input is working correctly.

Error 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 the drive or I/O module.
- There is no DC voltage to the machine inputs.
- There is a faulty input to the drive or I/O module.
- If this is a new installation, the logic of the overtravel inputs does not match the logic of the actual limit switch (Normally Closed or Normally Open).

Suggested actions

- Jog the Tilt axis in the opposite direction to move off of the limit switch.
- Check the hardware limit switch on the bevel head for damage and make sure that the switch is functioning correctly.
- Test the limit switch inputs in the Diagnostics Inputs screen.
- Check the cables and wiring between the limit switch and the drive or I/O module.
- Make sure that the cutting table is turned ON.
- Make sure all cables are correctly installed at the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Tilt Positive Hardware Overtravel switch and to make sure that the input is working correctly.

Error 30) Tilt Positive Software Overtravel

Software overtravel 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 of the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Error 31) Tilt Negative Software Overtravel

Software overtravel 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 of the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Error 34) Rotate Positive Software Overtravel

Software overtravel 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 of the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Error 35) Rotate Negative Software Overtravel

Software overtravel 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 of the software limit.
- If this is a new installation, check the minimum, maximum, and fault settings in the Axis setup screen.

Error 36) Dual Gantry Skew Error

A Dual Gantry Skew can 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 set incorrectly.
- If this is a new installation:
  - The skew limit is set too low.
  - Check the switch offset.

Suggested actions

- Make sure that the gantry is square and that there are no mechanical binds on the Rail and Dual Gantry axes.
- Check the Dual Gantry and Rail home switches for damage or a loose engagement block.
- Make sure 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.

Error 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 drive or I/O module and the device that activates the input.

Suggested actions

- Clear the error and resume cutting.
- Check the collision fault device to make sure that the device is working correctly.
- Check the cables and wiring between the collision fault device and the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Torch Collision input and to make sure that the input is working correctly.

Error 38) Excessive Mechanical Skew

The difference in position between the Rail and the Dual Gantry was greater than 2 inches when the CNC was turned ON.

Possible causes

- The position of the Dual Gantry axis changed prior to starting the CNC.
- 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.

Suggested actions

- Make sure that the gantry is square.
- Check the gantry for mechanical binds.
- If the pinion gears were disengaged from the drive rack when the CNC was turned OFF:
  - Start the CNC with the pinion gears still disengaged. If there is excessive difference in position, the Rail will rotate to equalize position.
  - Turn OFF the cutting machine and then re-engage the pinion gears. This will prevent this fault from occurring.
- If the Dual Gantry position changed, turn OFF the cutting machine, disengage the pinion gear, and rotate the pinion until the Dual Gantry position matches the Rail position.

Error 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 this 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 drive or I/O module and the device that activates the input.
- There is a faulty input on the drive or I/O module.

Suggested actions

- Park one of the stations and jog the other station to clear the fault.
- Check the Dual Head Collision Fault device to make sure that the device is working correctly.
- Check the cables and wiring between the Dual Head Collision Fault device and the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Torch Collision input and to make sure that the input is working correctly.

Error 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 sustained.
- There is a faulty cable or electrical connection between the Torch Collision input to the drive or I/O module 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 make sure that the device is working correctly.
  - 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, make sure that the switch is functioning correctly.
- Check the cables and wiring between the torch collision device and the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Torch Collision input and to make sure that the input is working correctly.

Error 44) Hardware Fault or Failure

This fault occurs when two or more opposing inputs are activated at the same 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.
- If this is a new installation:
  - Check the logic of all jog inputs.
  - Check the wiring of all jog inputs.

Suggested actions

- Restart the CNC to clear the fault. If the inputs were activated in error, the problem should not re-occur.
- If the problem continues:
  - Test the joystick hardware.
  - If the joystick has a sticking or damaged switch, the hardware test will fail.
  - To test the inputs, create a Watch Window, and assign the Op Con inputs for the joystick (left and right, up and down) and the Op Con (raise and lower tool).
  - Replace the joystick.

Error 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 of the limit switch.
- Check the hardware limit switch on the gantry for damage and make sure that the switch is functioning correctly.
- Test the limit switch inputs in the Diagnostics Inputs screen.
- Check the cables and wiring between the limit switch and the drive or I/O module.
- Make sure that the cutting table is turned ON.
- Make sure all cables are correctly installed at the drive or I/O module.
- 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 Diagnostics Inputs screen for the location of the Transverse Positive Hardware Overtravel switch and to make sure that the input is working correctly.

Error 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 of the software limit.

Error 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 of the software limit.

Error 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 of the software limit.

Error 60) Field Bus Configuration Fault

This fault occurs when there is a problem with the EtherCAT network configuration (Phoenix.xml) file. A *field bus* is sometimes referred to as a *fieldbus* or a *network*.

Possible causes

- There are incorrect or missing settings in a non-Hypertherm slave device's EtherCAT slave information (ESI) file.
- There is an unsupported slave device on the EtherCAT network.
- Distributed clocks is set up incorrectly.

Suggested actions

- Create a new XML file. See *Configure the EtherCAT Network* on page 89.
- Contact the manufacturer of the non-Hypertherm slave device for a corrected ESI file.
- Replace the unsupported slave device with a supported slave device.
- Set up distributed clocks correctly. See *Configure the EtherCAT Network* on page 89.
- Create a new XML file. See *Configure the EtherCAT Network* on page 89.

Error 61) Field Bus Device Fault

This fault occurs when there is an operational problem with a slave device that is active on the EtherCAT network. The fault message shows the device with the problem. A *field bus* is sometimes referred to as a *fieldbus* or a *network*.

Phoenix can only report a Field Bus Device Fault if the device communicates the error to Phoenix. Some devices may not be able to communicate errors to Phoenix, or may be able to communicate some but not all errors. If this is the case, you will need to check the drive itself.

This fault is different from a Field Bus Slave Fault (error 64), which indicates a network communications error rather than an operational error.

Possible cause

- These errors are device- and manufacturer-specific. See the documentation supplied by the drive manufacturer for more information.

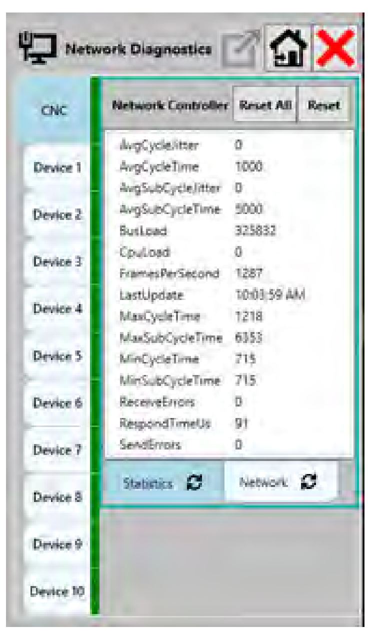
Suggested action

- Check the status of the drive. Then see the documentation supplied by the drive manufacturer for more information.

Network diagnostic screen

The Network Diagnostic screen provides information about the CNC and devices on the EtherCAT network. This information is read only and can be used for troubleshooting errors on the network or communication problems with one of the devices.

To get to the Network Diagnostic screen from the Main screen select the following soft keys: Setups > Diagnostics > Network Diagnostics. The Network Diagnostic screen is displayed in the upper right corner of the screen.



Error 62) Field Bus Network Fault

Error 62 is broken down into 13 sub-categories (62-1 through 62-13)

This fault occurs when the EtherCAT network master (the CNC) has a non-recoverable EtherCAT network fault that stops network communications. A *field bus* is sometimes referred to as a *fieldbus* or a *network*.

Error code	Fault	Description	Possible causes	Solutions
62 - 1	Unrecognized Fault	An unhandled EtherCAT field bus fault occurred.	▪ Noise on the network ▪ Cables that do not follow the Beckhoff EtherCAT specifications. ▪ Other errors specific to the EtherCAT network protocol. ▪ One or more devices on the network is in an error state and must have the power removed to clear the error. ▪ Loose, disconnected, or damaged cable (Pulling a cable with too much force damages the shielding inside).	▪ Make sure the drive firmware is up to update. ▪ Remove power from all the devices on the network, including the CNC. Turning OFF the power is not sufficient to clear an error. Physically disconnect the power from the devices to clear an error. Then restore power to all those devices to restart the cutting system and Phoenix. ▪ Replace the cables. See EtherCAT cable on page 34. ▪ Clear the error and restart the EtherCAT network.
62 - 2	Missed Update	Data arrived too early or too late on one or more connected devices. The master did not receive the data from the connected devices in the time allowed.		
62 - 3	Network Down	The network physical link is disconnected. The state of the EtherCAT field bus does not match what Phoenix expected. For example, Phoenix is trying to initiate a move to one of the drives and the device is not operational.		
62 - 4	Configuration	The Phoenix.xml configuration file does not match the discovered devices on the EtherCAT field bus.	▪ EtherCAT field bus devices have changed or the wrong Phoenix.xml was installed. ▪ The Phoenix.xml file may be corrupted.	▪ Recreate the Phoenix.xml file using Hypertherm Studio. ▪ Clear the error and restart the EtherCAT network.

Error code	Fault	Description	Possible causes	Solutions
62 - 5	Watchdog Timeout	EtherCAT real-time process timing constraints were not satisfied.	Windows or INtime configuration issue	▪ Make sure the drive firmware is up to update. ▪ Remove power from all the devices on the network, including the CNC. Turning OFF the power is not sufficient to clear an error. Physically disconnect the power from the devices to clear an error. Then restore power to all those devices to restart the cutting system and Phoenix. ▪ Replace the cables. See EtherCAT cable on page 34. ▪ Clear the error and restart the EtherCAT network.
62 - 6	Parse	There was a problem parsing data from the network.	▪ Noise on the network ▪ Cables that do not follow the Beckhoff EtherCAT specifications. ▪ Other errors specific to the EtherCAT network protocol. ▪ One or more devices on the network is in an error state. ▪ Loose, disconnected, or damaged cable (Pulling a cable with too much force damages the shielding inside).	
62 - 7	Send Receive	There was a problem sending or receiving data on the network.		
62 - 8	Slave to Slave Timeout	A timeout occurred during Slave to Slave communications.		
62 - 9	Unknown Field Bus Network Fault	An unhandled EtherCAT field bus fault occurred.		
62 - 10	Default Input Data	An error on the EtherCAT field bus caused default data to be used.		
62 - 11	Outputs Not Delivered	EtherCAT field bus data was not delivered to one or more devices.		
62 - 12	Internal Sync Lost	EtherCAT field bus internal synchronization was not successful.		
62 - 13	Propagation Delay Lost	EtherCAT field bus propagation delay was not successful.		
62 - 15	Fieldbus: Timing Alert	Communication errors between the CNC and connected devices.		

Error 64) Field Bus Slave Fault

This fault occurs when a slave device that is active on the EtherCAT network has an EtherCAT network protocol error and cannot communicate with the EtherCAT network. The fault message shows the slave device with the problem. A *field bus* is sometimes referred to as a *fieldbus* or a *network*.

This fault is different from a Field Bus Device Fault (error 61), which indicates an operational error rather than a network communications error.

Possible causes

- These faults are caused by errors specific to the EtherCAT network protocol.
- The physical location of one or more slave devices does not match the location described in the EtherCAT network configuration (Phoenix.xml) file.

Suggested actions

- See the documentation supplied by the drive manufacturer for more information.

- Clear the error and restart the EtherCAT network.
- Turn OFF the slave device, and then turn ON the slave device to restart it. Then restart the EtherCAT network.

Error 66) RTOS Watchdog Fault

Possible cause

- These faults are caused when Windows does not respond to the real-time operating system (RTOS).

Suggested action

- Clear the error and restart the CNC.

Error 67) PLC application

Possible cause

- The cutting system manufacturer has introduced a fault in the CNC using the PLC's Create Fault function block. The fault shows an error number that is specific to the cutting system manufacturer's application that is running in the PLC.

Suggested actions

- Restart the CNC.
- See the documentation supplied by the cutting system manufacturer.

Error 68) PLC

Possible cause

- The PLC has encountered an operational error and stopped running the PLC program.

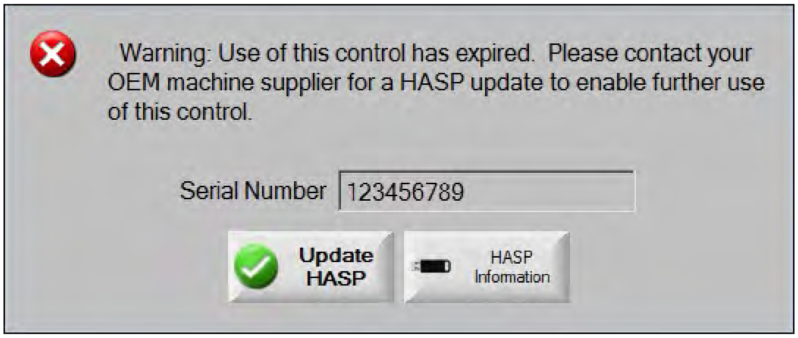
Suggested action

- Restart the CNC to restart the PLC.

HASP warning and error message reference

The warning message *Use of this control has expired* appears

Figure 42



Possible causes

- Your cutting system manufacturer (OEM) set up a timer to limit the number of days that the Phoenix software is valid, and the timer has expired. You can no longer use the Phoenix software.

Suggested actions

- Contact your OEM to have the timer extended or removed. The OEM will email you a *.V2C file to use to update the HASP in the EDGE Connect.

  Then do the following.

  - Save the *.V2C file to the root directory of a USB memory stick.
  - Put the USB memory stick in a USB connector on the CNC.
  - On the warning dialog box shown in *Figure 42*, choose **Update HASP**.
  - In **Load HASP update from**, choose **Memory Stick**.
  - Choose the *.V2C file, and then choose **OK**. The HASP is updated.
- If you get an error, then there is a problem with the *.V2C file. Do the following.
  - Make sure that the USB memory stick is still in a USB connector on the CNC.
  - Choose **HASP Information** on the warning dialog box shown in *Figure 42*.
  - In **Save HASP information to**, choose **Memory Stick**.
  - Keep the default name for the *.C2V file, and then choose **OK**.
  - Contact your OEM, and email the *.C2V file to them.

See also *Error 54) HASP Update Too Old* on page 260 and *Error 55) HASP Update Too New* on page 261.

*.V2C files must be installed in the order in which they were generated for your CNC, and all **.V2C files that have been generated must be applied. (You cannot skip **.V2C files.) The *.C2V file that you email to your OEM shows the current **.V2C file, so that they can troubleshoot the problem with Hypertherm Technical Service.

Error 18) HASP key ID not found

Possible cause

- The HASP is not physically connected to the CNC.
- The HASP is not detected by Windows.

Suggested action

- Install the HASP if it is not connected.
- Disconnect and reconnect the HASP if it is connected.

Error 19) HASP invalid update data

Possible cause

- Using the **UPDATEFEATURES** password with a corrupted V2C file.

Suggested action

- Make sure you have selected the correct *.V2C file
- Contact your OEM and email them the *.V2C file so that they can troubleshoot the problem with Hypertherm Technical Service.

Error 54) HASP Update Too Old

Possible cause

- You tried to update the HASP on the EDGE Connect CNC, but the selected *.V2C file is invalid. The reason is that *.V2C files must be installed in the order in which they were generated for your CNC, and all *.V2C files that have been generated must be applied. This error means that one or more later *.V2C files have not yet been installed.

Suggested action

- Contact your OEM and email them a *.C2V file containing your information so that they can troubleshoot the problem with Hypertherm Technical Service. To get the *.C2V file, do the following.
  - Make sure that the USB memory stick is still in a USB connector on the CNC.
  - Choose **Setups > Password > HASPINFORMATION**.
  - In **Save HASP information to**, choose **Memory Stick**.
  - Keep the default name for the *.C2V file, and then choose **OK**.

See also *The warning message Use of this control has expired appears* on page 259.

Error 55) HASP Update Too New

Possible cause

- You tried to update the HASP on the EDGE Connect CNC, but the selected *.V2C file is invalid. The reason is that *.V2C files must be installed in the order in which they were generated for your CNC, and all *.V2C files that have been generated must be applied. This error means that one or more earlier *.V2C files have not yet been installed.

Suggested action

- Contact your OEM and email them a *.C2V file containing your HASP information so that they can troubleshoot the problem with Hypertherm Technical Service. To get the *.C2V file, do the following.
  - Make sure that the USB memory stick is still in a USB connector on the CNC.
  - Choose **Setups > Password > HASPINFORMATION**.
  - In **Save HASP information to**, choose **Memory Stick**.
  - Keep the default name for the *.C2V file, and then choose **OK**.

See also *The warning message Use of this control has expired appears* on page 259.

Plasma cutting tips

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

Consider the following factors when evaluating plasma cut quality.

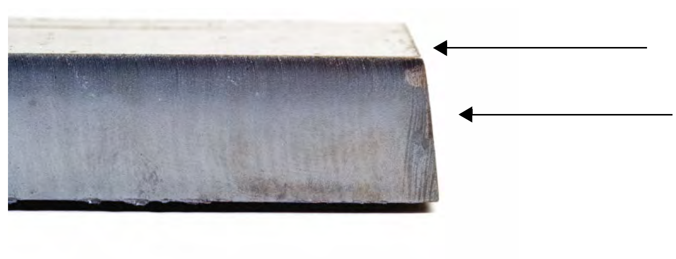
- Type of machine (example: XY table, punch press)
- Plasma cutting system (example: plasma power supply, torch, consumables)
- Motion control device (example: CNC, torch height control)
- Process variables (example: cutting speed, gas pressures, flow rates)
- External variables (example: material variability,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

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

Cut quality problems

Angularity

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

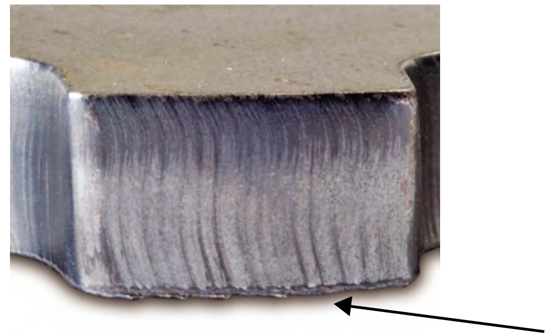


Negative cut angle: More material is removed from the bottom of the cut surface than from the top.

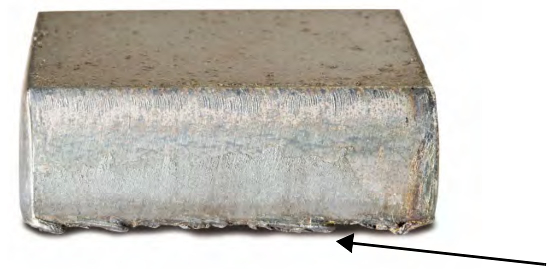
Top edge rounding: There is a slight rounding along the top edge of the cut surface.

Dross

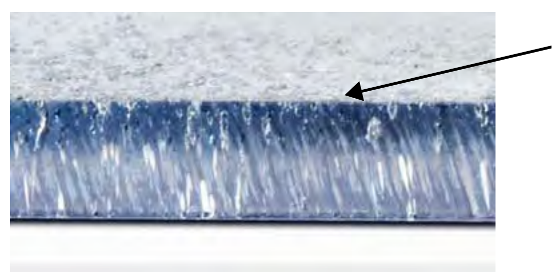
High-speed dross: A small, linear bead of molten material attaches and hardens along the bottom edge of the cut. In addition, S-shaped lag lines are present; dross is difficult to remove and requires grinding.



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 can be expected; “roughness” describes the texture of the cut face (the cut is not smooth).

Aluminum

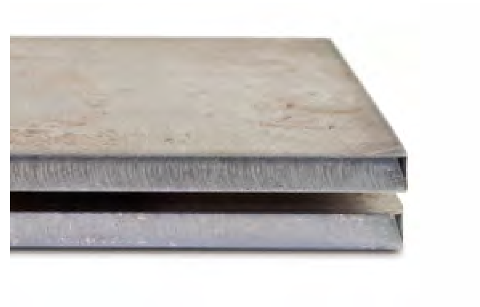
Top: Air/Air

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

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 expected and vary most dramatically with stainless steel.

Top: N_2/N_2

Middle: H35/N_2

Bottom: Air/Air

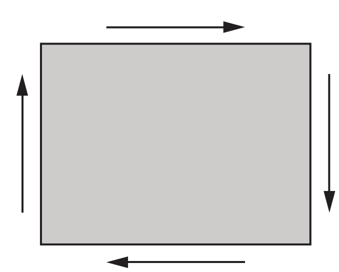
Basic steps to improve cut quality

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.
- Make sure that the direction of the cut is correct.
- Adjust the cutting direction, if necessary. The plasma arc typically spins clockwise with standard consumables.

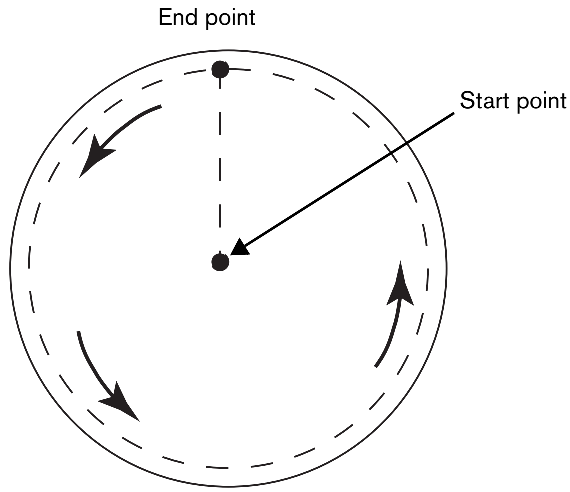
Contour

- 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?

Refer to the cut charts. 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
- Make sure that the correct consumables are being used, and make sure that the part numbers are correct.

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?

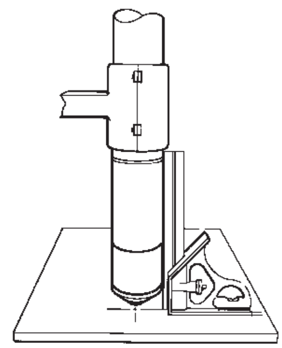
- Inspect consumables for wear.
- Replace worn consumables.
- Always replace the nozzle and electrode at the same time.
- Avoid over-lubricating O-rings.

Use genuine Hypertherm consumables to maximize 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.

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



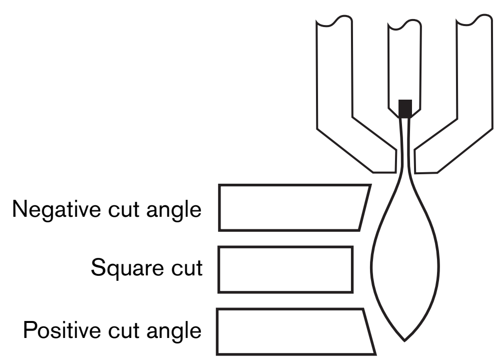
Step 5: Is the cut height set at the proper height?

- 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 can be normal if the variation is within tolerance.



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

- Adjust the cutting speed as needed.

  Cutting speed can 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 can form on subsequent cuts.

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

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

Step 8: Is there torch vibration?

- Make sure that the torch is tightly attached to the table gantry.
- Consult the cutting system manufacturer. Your table may require maintenance.

Step 9: Does the table need to be tuned?

- Make sure that the table is cutting at the specified speed.
- Consult the cutting system manufacturer. The table speed may need tuning.

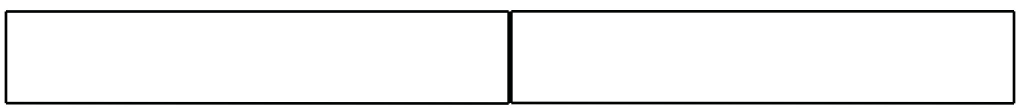
Bevel cutting tips

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

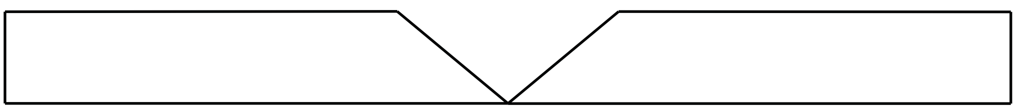
Types of bevel cuts

Plasma bevel cutting has 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.

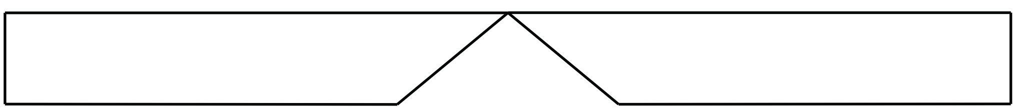
I cut



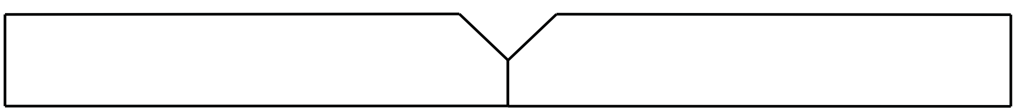
V cut



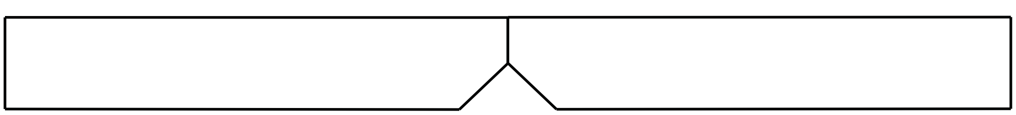
A cut



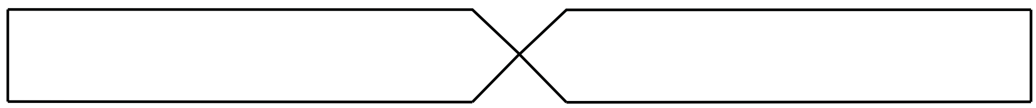
Y Top cut



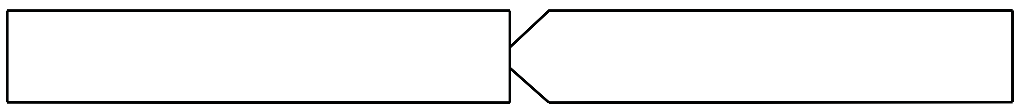
Y Bottom cut



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.

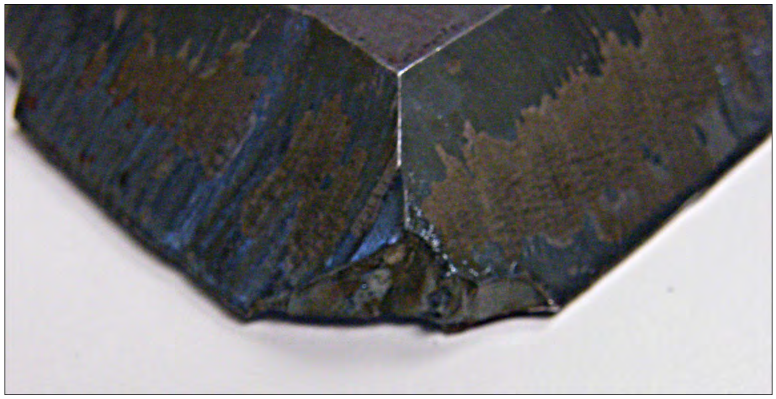
2. Measure and correct for land dimension if cutting Y Top cut parts.

3. Measure and correct for part dimension.

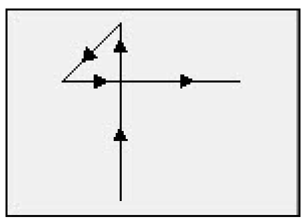
Quality bevel-cut parts result from a strong partnership between the part programmer and machine operator. The part programmer can take advantage of bevel parameters available in the CAM software used to produce the part program, and the operator can perform adjustments available on the CNC. Correcting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

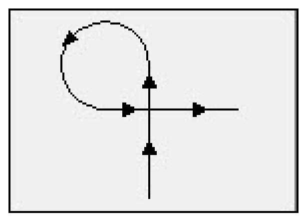
Clipped corner



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.



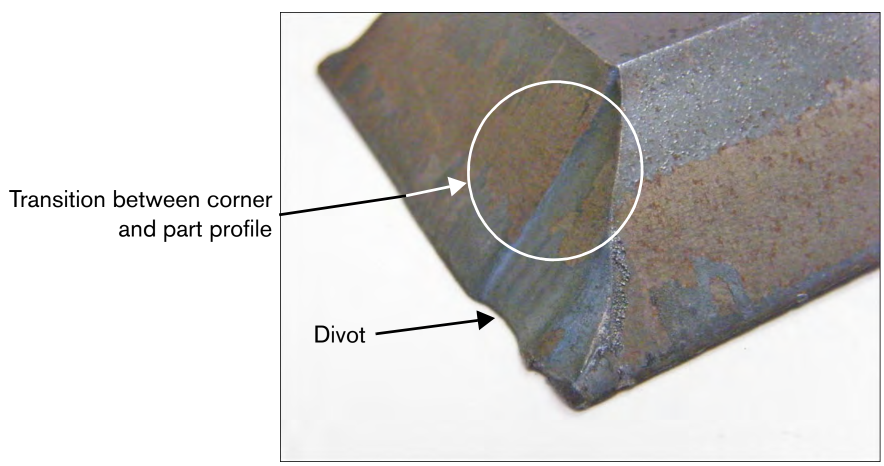
Linear corner loop



Arc corner loop

Inconsistent cut quality on a single part surface

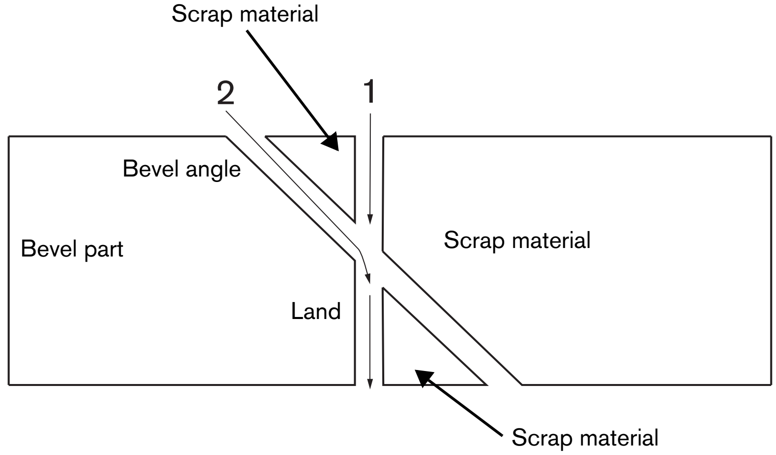
The graphic below shows two cut quality issues: a transition in cut quality from the part corner into the part profile, and a divot in the cut.



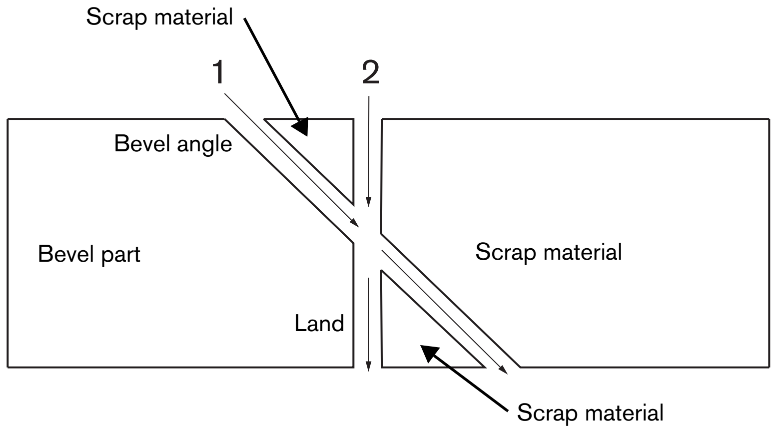
Increase the length of the lead-in segment. A longer lead-in allows the torch to lock on to the correct combination of arc voltage and cut height. To correct lead-in length, refer to the CAM software, then generate a new program.

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.



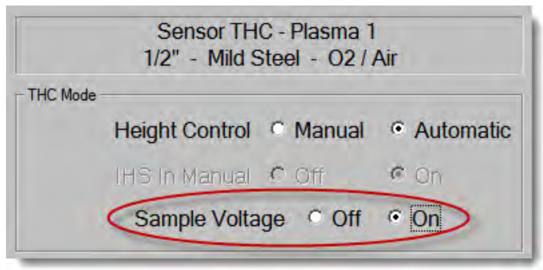
To change the order in which the passes (also called profiles) are cut, refer to your CAM software, then generate a new part program.



Part dimensions change within a nest

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

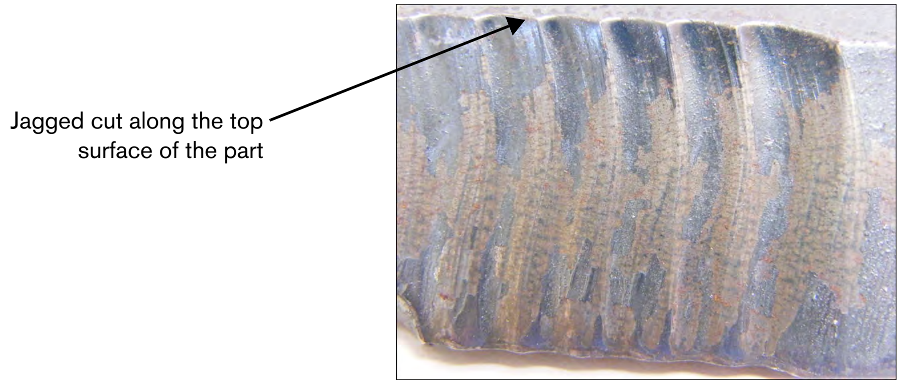
- Arc voltage is set in the cut chart but can be overridden on a job-by-job basis by adjusting the setting on the **Process** screen. To change the arc voltage setting, choose **Setups** > **Process**, and enter the new value for **Set Arc Voltage**. If you are using the torch height control in Manual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



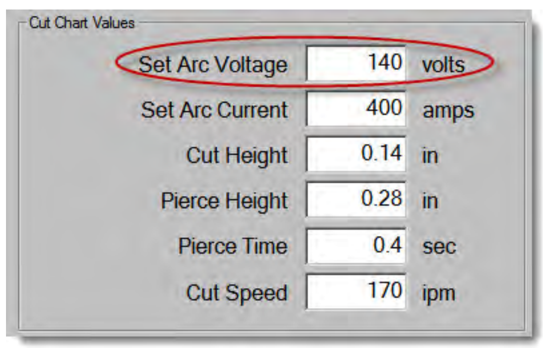
- Check the torch consumables and replace them if they are worn.

Jagged cut

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

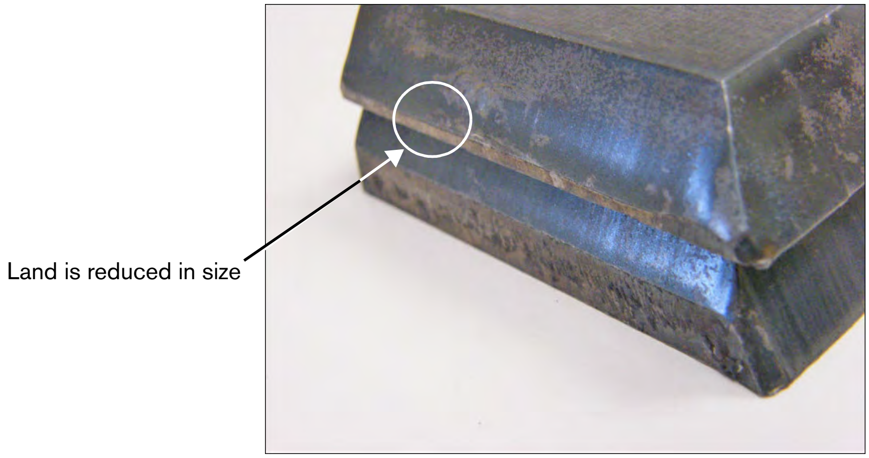


- Arc voltage can be set for the thickness of the material and not for the effective thickness of the material when cutting on an angle. To increase arc voltage on the CNC, choose **Setups** > **Process**, and adjust the arc voltage.



- Turn on arc voltage sampling (**Setups** > **Process**). 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

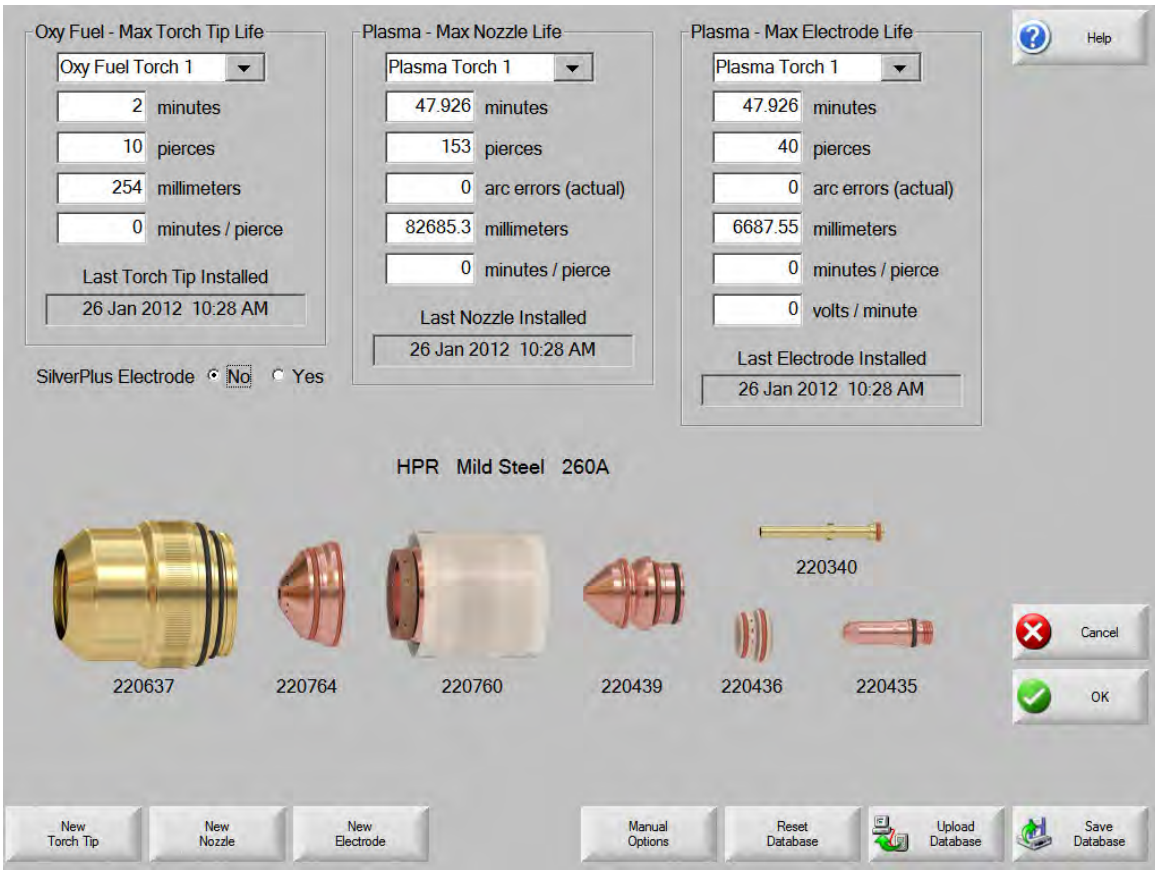


- Increase the **Set Arc Voltage** parameter in the **Setups** > **Process** screen to account for the change in material thickness because you are cutting at an angle.
- Turn on **Sample Arc Voltage**, also in the **Setups** > **Process** screen.
- Check and possibly increase the cut height in the Process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

Change 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 **New Torch Tip**, **New Nozzle**, or **New Electrode** is chosen 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:

- Choose 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. This can be saved to a USB memory stick.

You can set up a Watch Window to view this data during cutting. See *Set up a Watch Window* on page 153.

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, the Plasma Torch 1 torch tip has a limit of 5,000 minutes of operation. After 5,000 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.

If **Auto update max consumable life** is enabled on the Special Setups screen, consumable life is tracked beyond the user-defined set point and then assigned that 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.

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 power supply. The plasma 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.

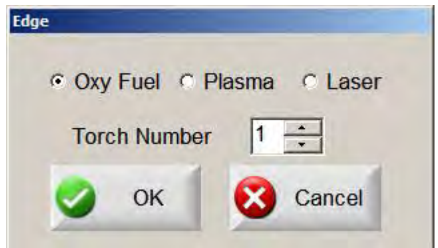
This parameter is available for use only with the Sensor THC while cutting in Set Arc Voltage mode. If cutting in Sample Arc Voltage mode, set the Volts per minute to 0.

Last Torch Tip Installed: The date and time when the selected tip was installed.

Last Electrode Installed: 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 soft key: Choose the **New Torch Tip** soft key to select which Torch Tip has been replaced and to update the database.



New Electrode soft key: Choose the **New Electrode** soft key to select which electrode has been replaced and to update the database.

New Nozzle soft key: Choose the **New Nozzle** soft key to select which nozzle has been replaced and to update the database.

Manual Options soft key: Opens the **Manual Options** screen so that you can reposition the torch to change the consumables.

Reset Database soft key: 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 soft key: Uploads the current database to a host computer.

Save Database soft key: Saves the current database to the USB memory stick.

15

Part Replacement

For a list of replacement parts, see *Parts* on page 333.

CAUTION

Static electricity can damage circuit boards. Use precautions when handling printed circuit boards.

Wear a grounded wrist strap when handling PC boards.

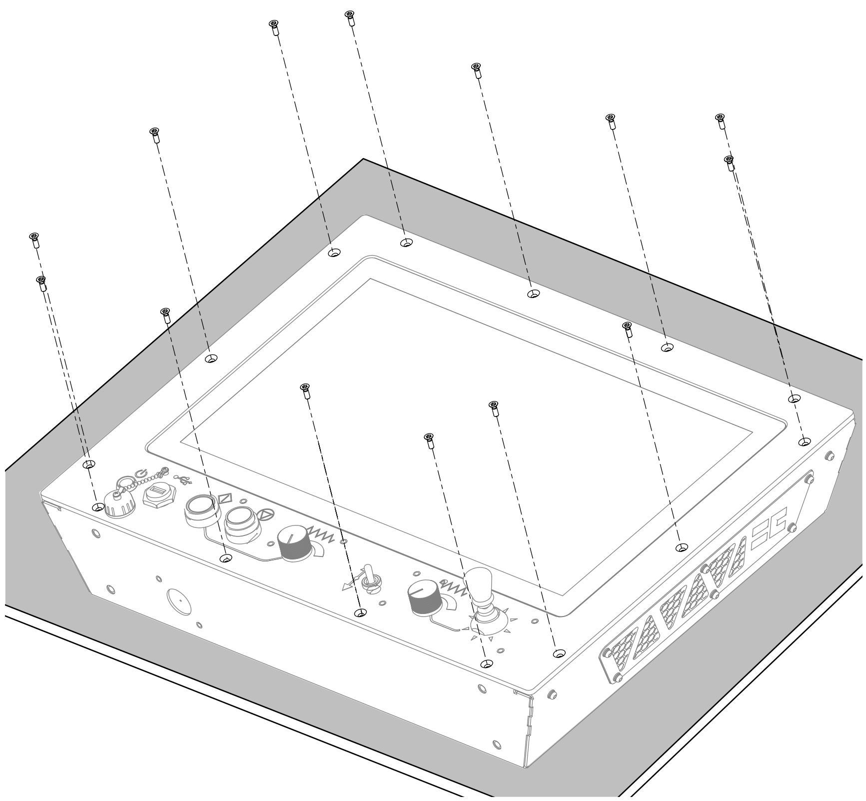
Required tools

These tools are used in most of the procedures in this section.

- T20 driver
- T15 driver
- Adjustable wrench

Remove the rear enclosure

1. Turn OFF the CNC.
2. Disconnect the power cord from the CNC.
3. Disconnect the remaining cables from the front and rear of the CNC.
4. Remove the wireless antennas and any extension cables.
5. Put the CNC on a clean, flat surface with the front panel assembly face up.
6. Remove the screws from the front panel assembly.



After you remove the screws from the front panel assembly, the front panel assembly will be loose but still connected by the screws in the rear I/O panel.

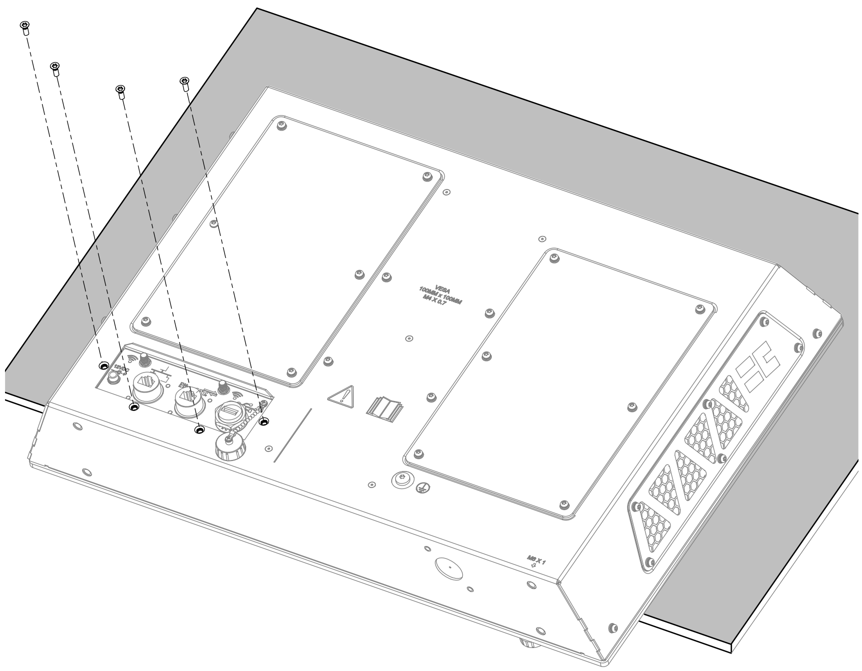
7. Pick up the CNC and turn it over, with the front panel assembly face down on the clean, flat surface. Leave the joystick hanging off the side of the surface.

⚠ CAUTION

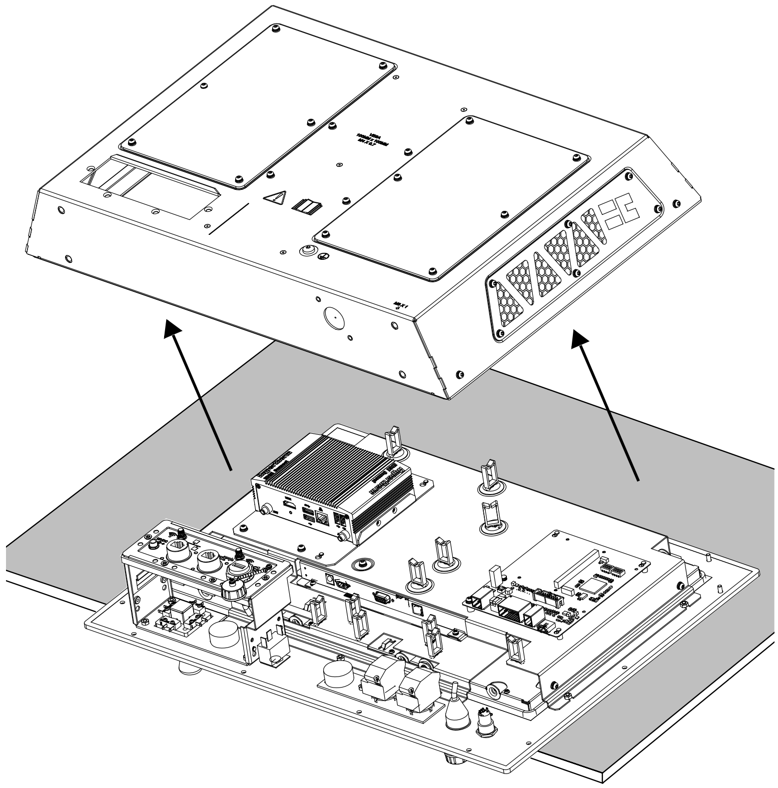
Any debris under the CNC could cause damage to the touchscreen.

Do not push down on the part of the CNC that is off the surface.

8. Remove the screws from the rear I/O panel.



9. Lift the rear enclosure away from the front panel assembly.



Install the rear enclosure

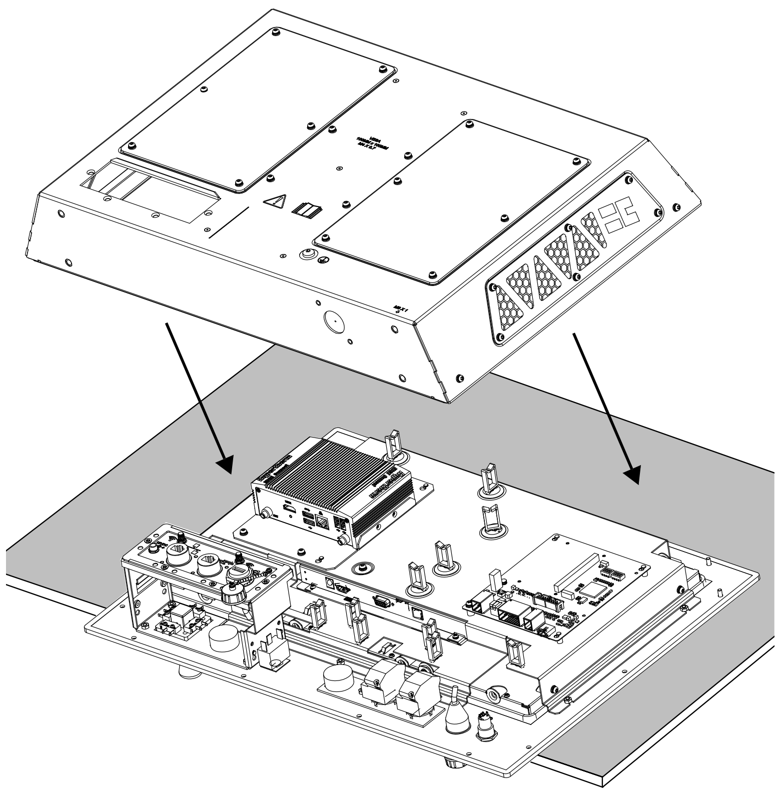
1. Put the CNC on a clean, flat surface with the front panel assembly face down. Leave the joystick hanging off the side of the surface.

⚠ CAUTION

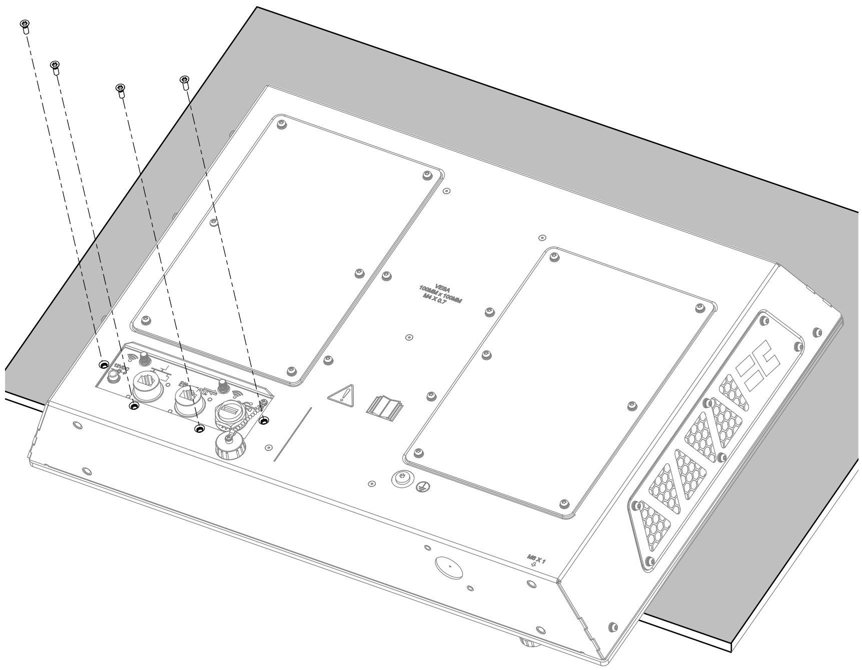
Any debris under the CNC could cause damage to the touchscreen.

Do not push down on the part of the CNC that is off the surface.

2. Put the rear enclosure on the front panel assembly.



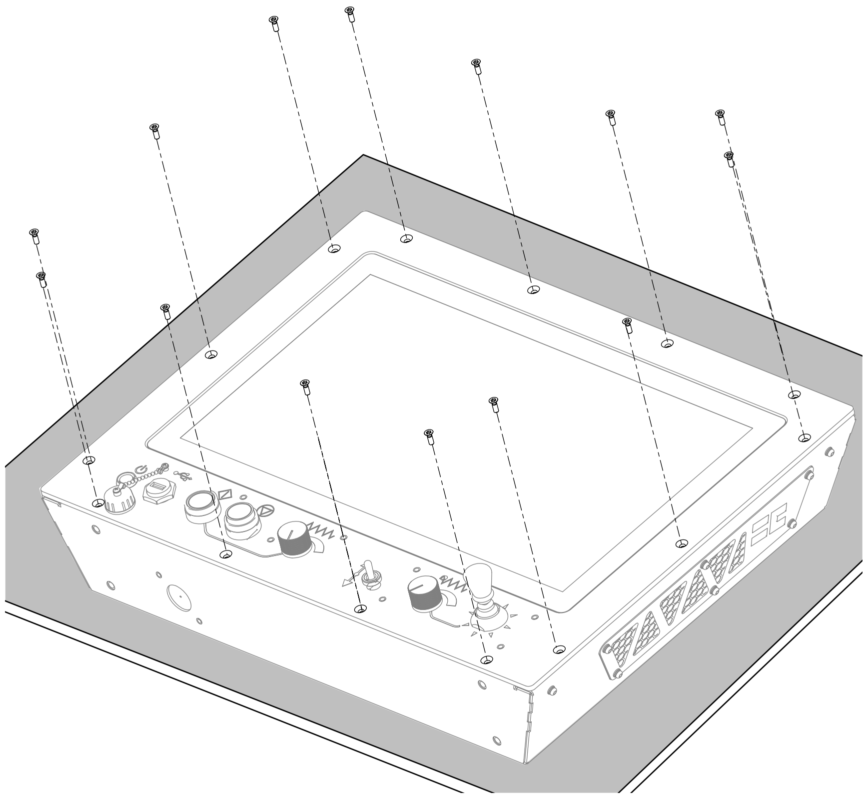
3. Install the screws on the rear I/O panel.



4. Pick up the CNC and turn it over, with the front panel assembly face up.

The front panel assembly will be loose but connected to the rear enclosure by the screws in the rear I/O panel.

5. Install the screws on the front panel assembly. Tighten the screws to 1.1 N∙m (10 in∙lb).



6. Connect all of the cables to the front and rear of the CNC.

7. If necessary, install the wireless antennas.

Replace the touchscreen

1. Make sure that you have the correct touchscreen kit for your CNC model (see *Table 5*). You can find the CNC model (part) number on the data plate on the rear of the CNC. See *Data plate* on page 28 for more information about the data plate.

Table 5 – Touchscreen kits

Model number	Kit number	Kit contents
090198	428765	▪ 495.3 mm (19.5 inch) touchscreen ▪ HDMI cable ▪ USB cable ▪ 9 screws
090185	428632	▪ 469.9 mm (18.5 inch) touchscreen ▪ DVI cable ▪ USB cable ▪ 9 screws
The touchscreens also come with an on-screen-display (OSD) remote control. You do **not** need the OSD remote control to install the touchscreen. You may want to use the OSD remote control to adjust the screen brightness or turn on the touchscreen during troubleshooting if it does not turn on automatically.		

2. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
3. Cut the cable tie on the yellow/black power connector wires.
4. Disconnect the power connector from the touchscreen. (Leave the other end of the power connector connected to the printed circuit board.)
5. (Model number 090198 only) Cut the cable tie on the USB cable.
6. Remove the USB cable and video cable from between the touchscreen and the embedded-CNC.

Figure 43 – Remove cables from model number 090198 (495.3 mm/19.5 in. touchscreen)

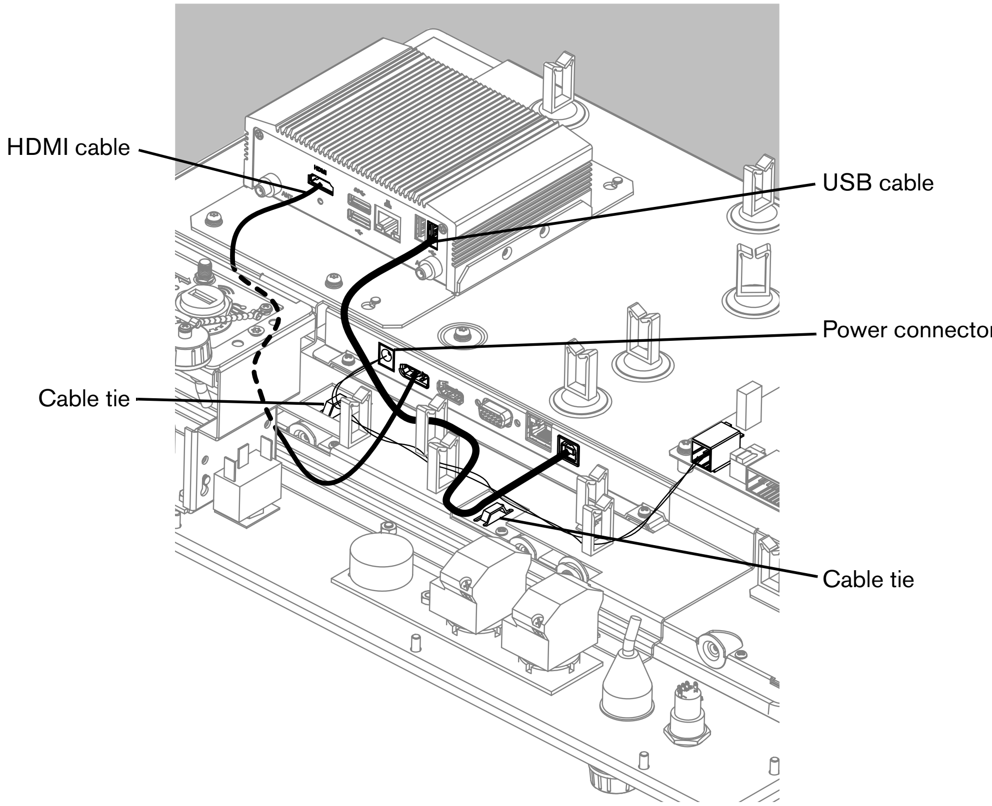
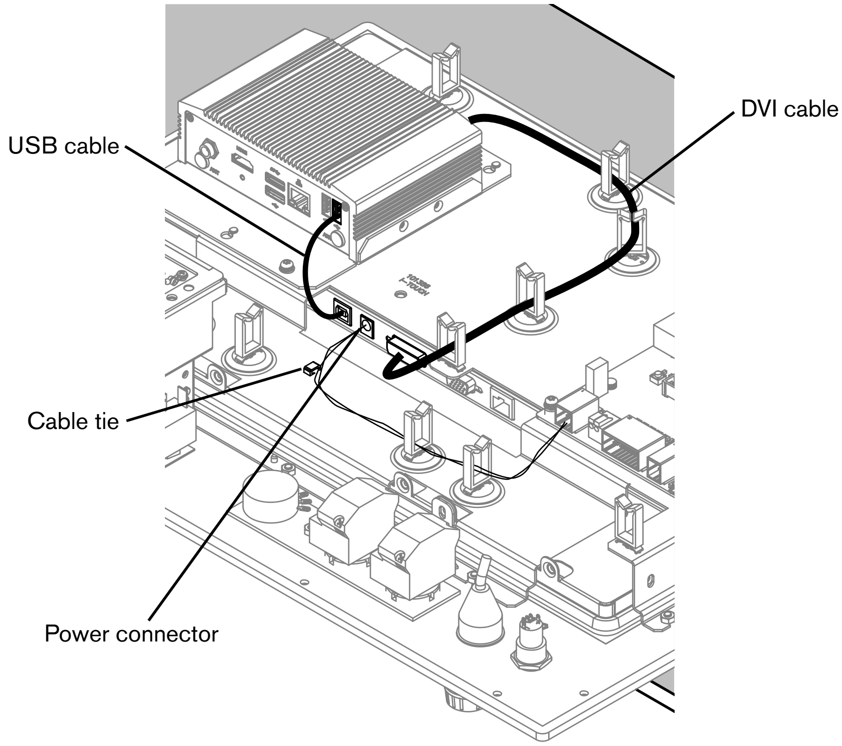
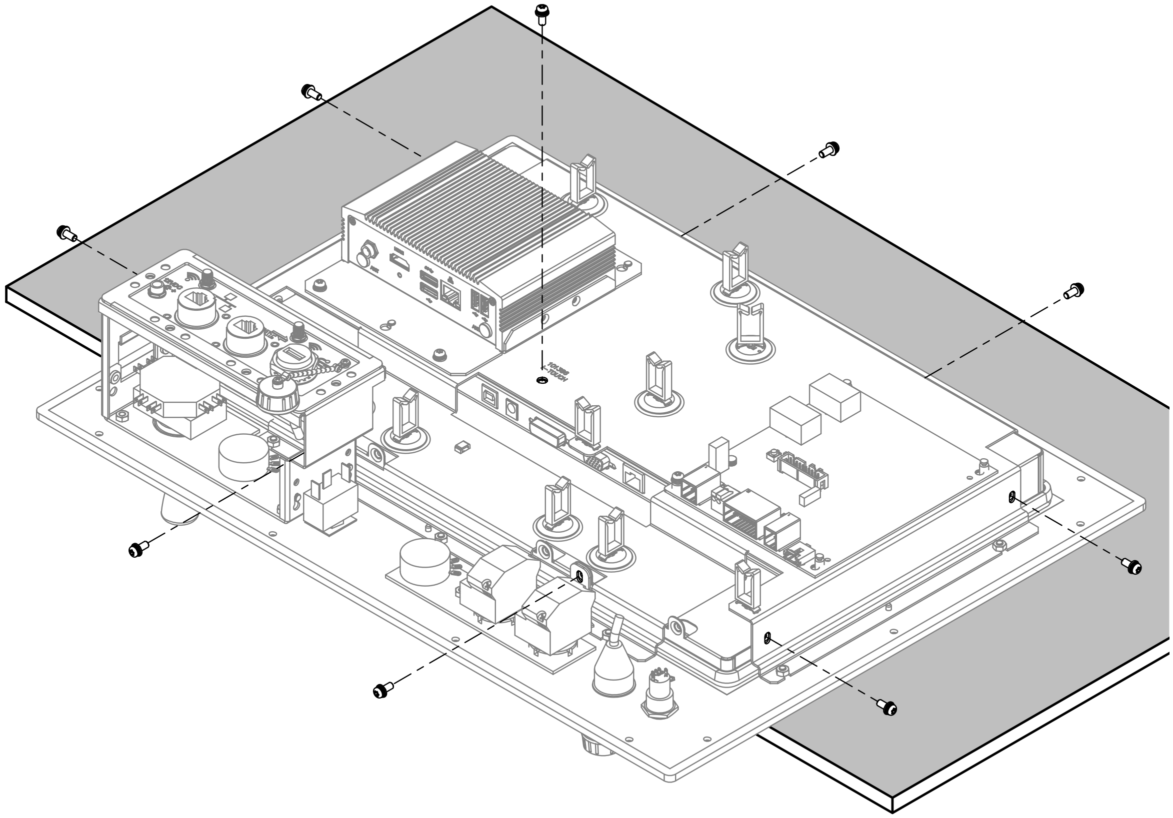


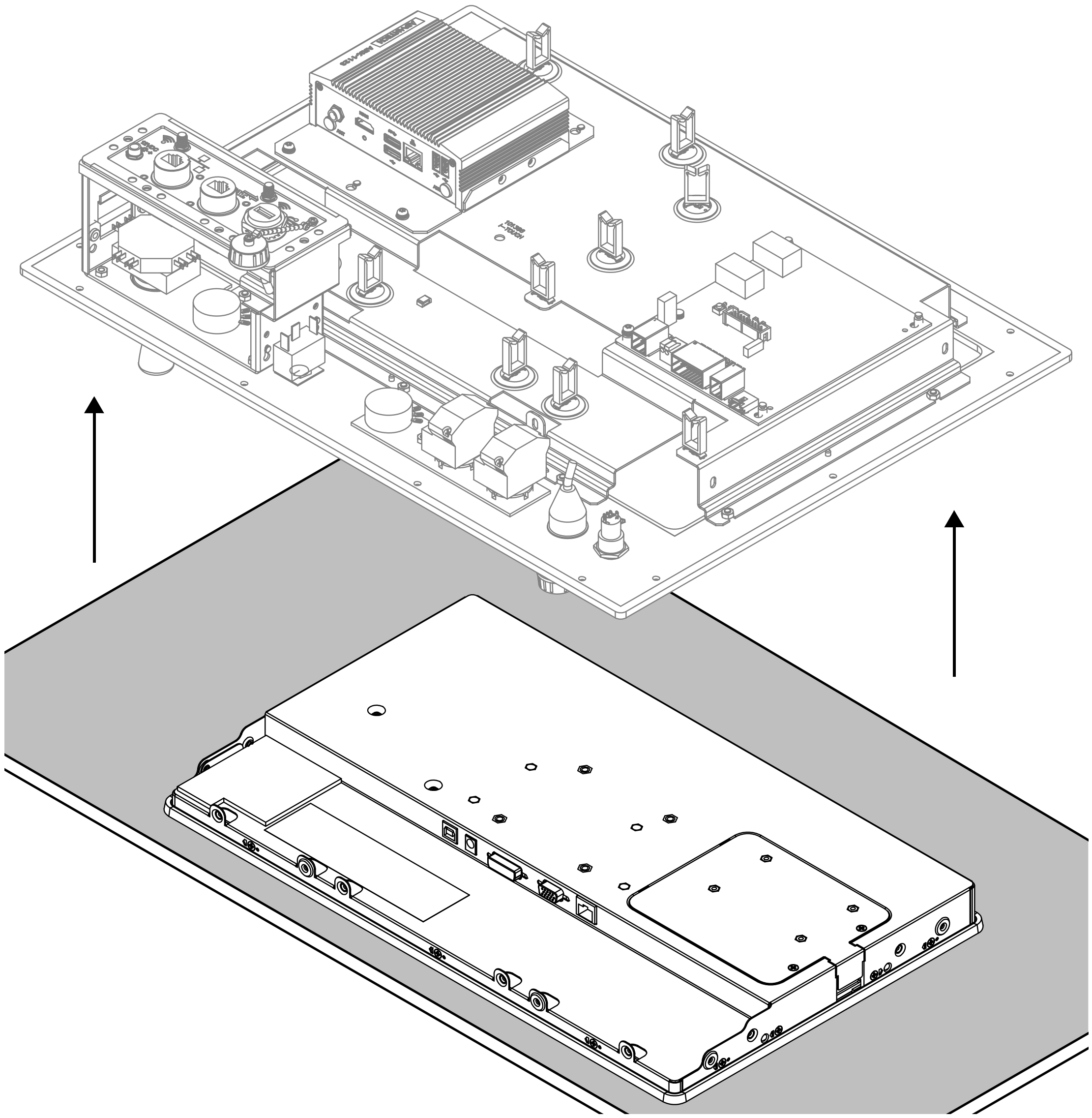
Figure 44 – Remove cables from model number 090185 (469.9 mm/18.5 inch touchscreen)



7. Remove the screws from the rear of the front panel assembly.



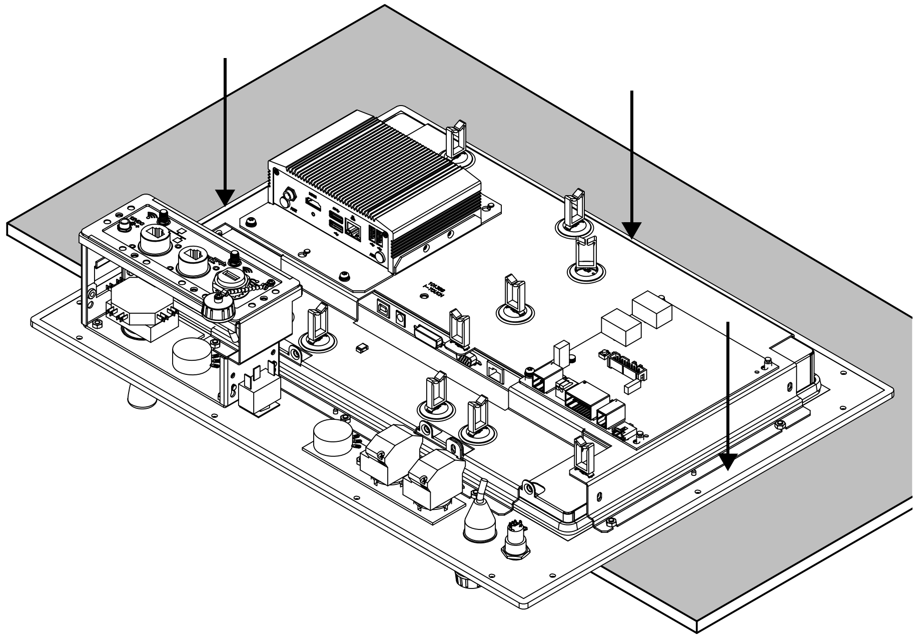
8. Hold down the touchscreen while you lift the front panel up. Do not let the touchscreen fall.



9. Put the new touchscreen on the clean, flat surface.

10. Lower the front panel over the new touchscreen.

11. Firmly push down on the edges of the front panel until it is flush with the surface.



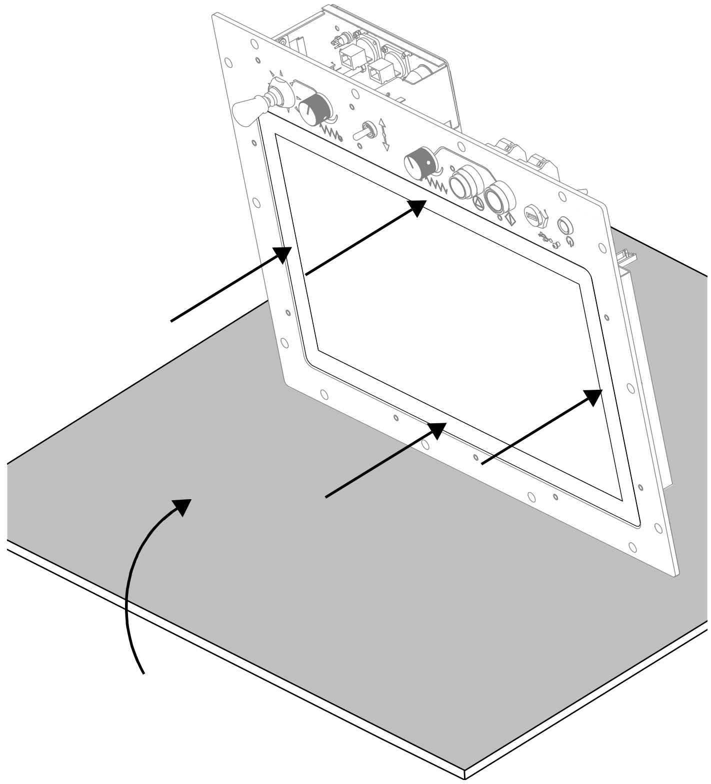
12. Start all of the screws. Do **not** tighten them into the touchscreen.

If necessary, push down on the edge of the front panel near the screw hole to put the screw into the opening.

13. Lift the bottom edge of the CNC until the CNC is on its top edge. See *Figure 45*.

14. Push the edges of the touchscreen into the front panel until the touchscreen has a recess of approximately 1 mm (0.04 inch).

Figure 45



15. Keep the recess at approximately 1 mm (0.04 inch) while you tighten all of the screws to 1.1 N·m (10 in·lb). Turn the CNC onto its other edges to get access to the other screw holes.

16. Install the USB cable and video cable between the touchscreen and the embedded-CNC, and connect the power connector to the touchscreen.

17. Install a cable tie on the yellow/black power connector wires.

18. (Model number 090198 only) Install a cable tie on the USB cable.

19. Put the cables through the cable clips.

Figure 46 – Install the cables on model number 090198 (495.3 mm/19.5 in. touchscreen)

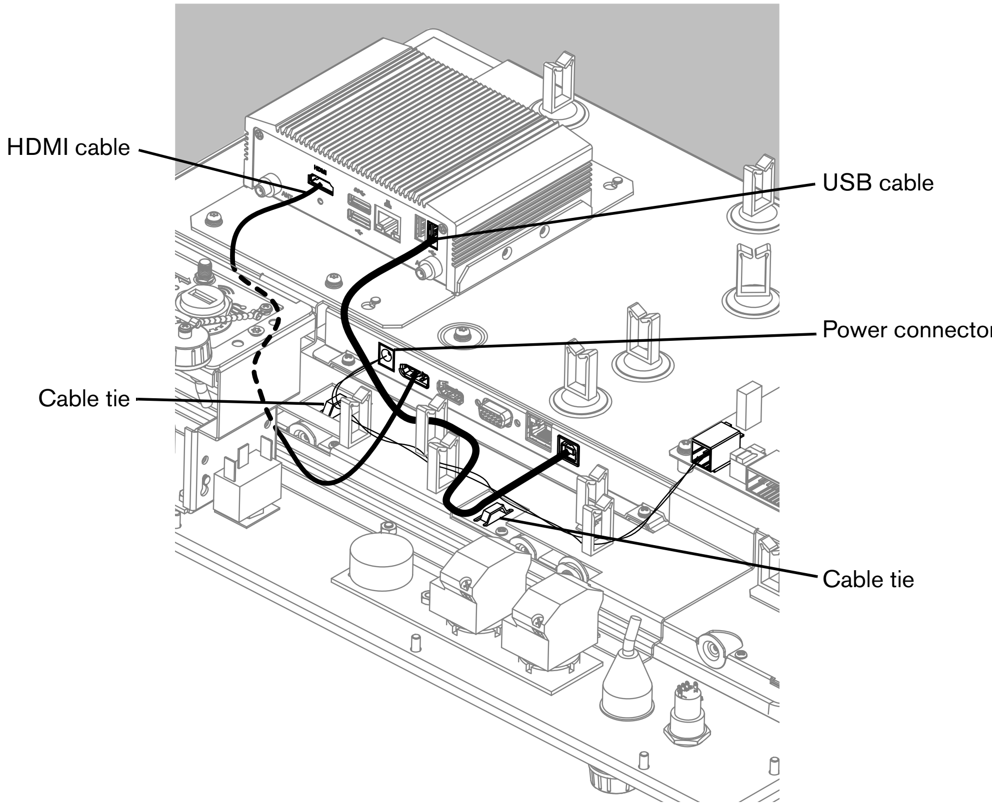
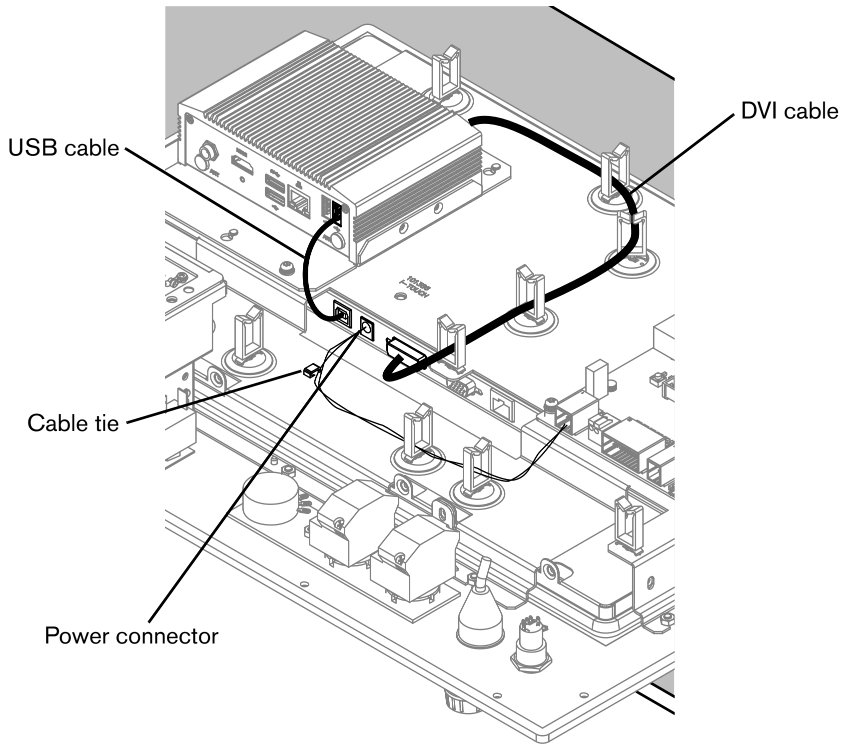


Figure 47 – Install the cables on model number 090185 (469.9 mm/18.5 inch touchscreen)



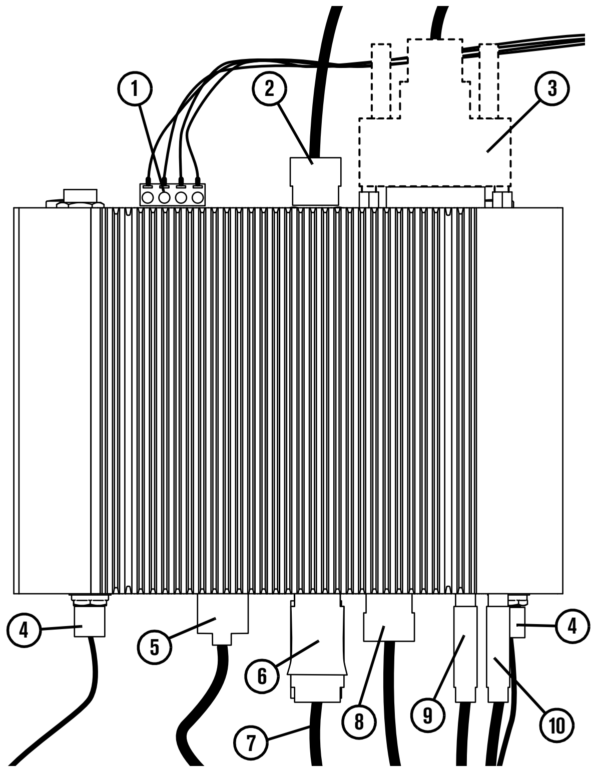
20. Install the rear enclosure on the front panel. See *Install the rear enclosure* on page 283.

Replace the embedded-CNC

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.

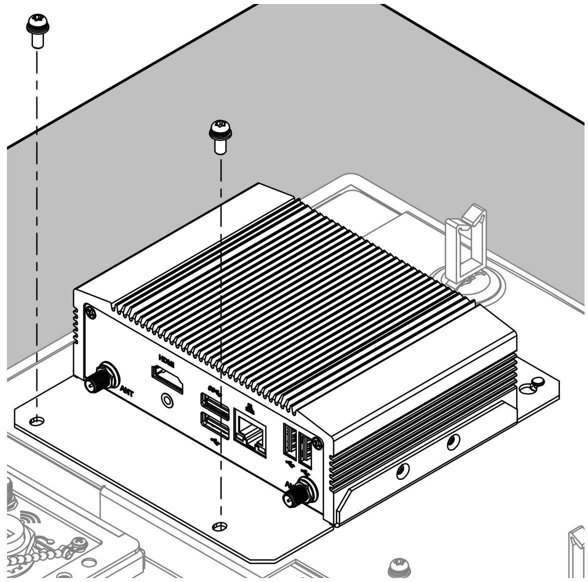
2. Disconnect all of the cables and the HASP software key from the embedded-CNC.

Keep the HASP software key to use in the replacement embedded-CNC.

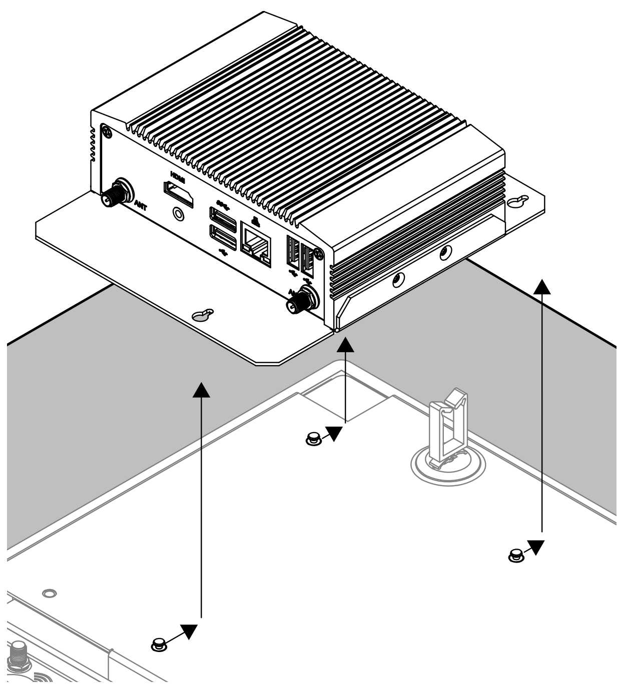


- **1** Power terminal and wiring harness
- **2** EtherCAT cable to PCB or EtherCAT connector
- **3** DVI cable to touchscreen – model number 090185 (469.9 mm/18.5 inch touchscreen) **only**
- **4** Wireless antenna cables to rear I/O panel
- **5** HDMI cable to touchscreen – model number 090198 (495.3 mm/19.5 in. touchscreen) **only**
- **6** HASP software key (top connector)
- **7** USB to rear USB connector (bottom connector)
- **8** Ethernet cable to rear I/O panel
- **9** USB cable to front USB connector
- **10** USB cable to touchscreen

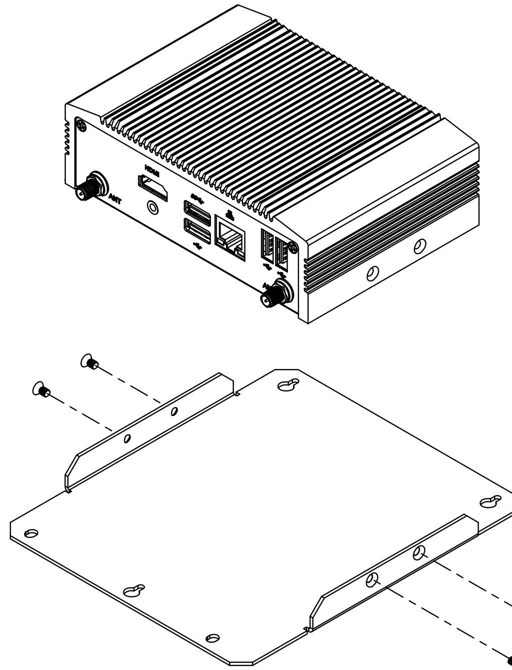
3. Remove the screws from the mounting plate.



4. Remove the mounting plate from the 3 pins.



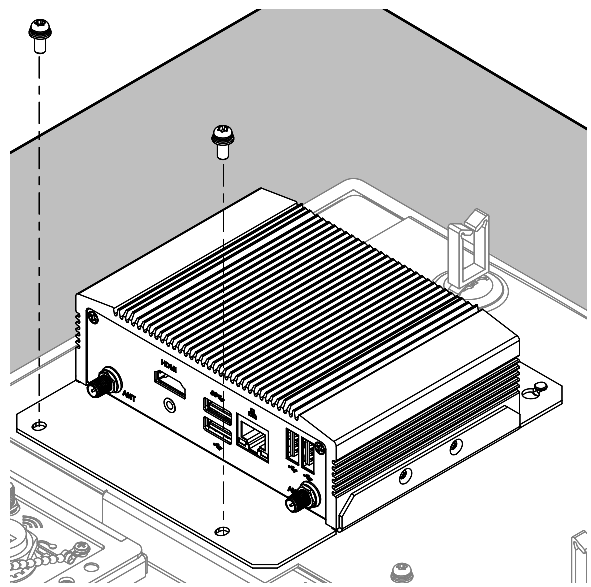
5. Use a #2 Phillips® screwdriver to remove the screws from the sides of the mounting plate.
6. Remove the embedded-CNC from the mounting plate.
7. Use the screws to install the new embedded-CNC onto the mounting plate with the wireless connectors toward the rear I/O panel.



8. Install the mounting plate onto the front panel assembly.

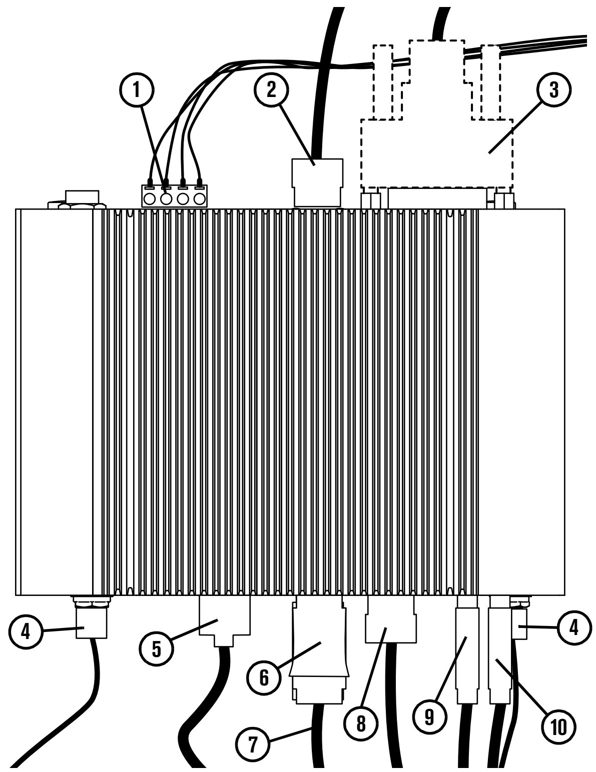
   Make sure to engage all 3 pins.

9. Install the screws on the mounting plate.



10. Connect all of the cables and the HASP to the embedded-CNC.

You must use the same HASP software key that you disconnected in *step 2* on page 295.

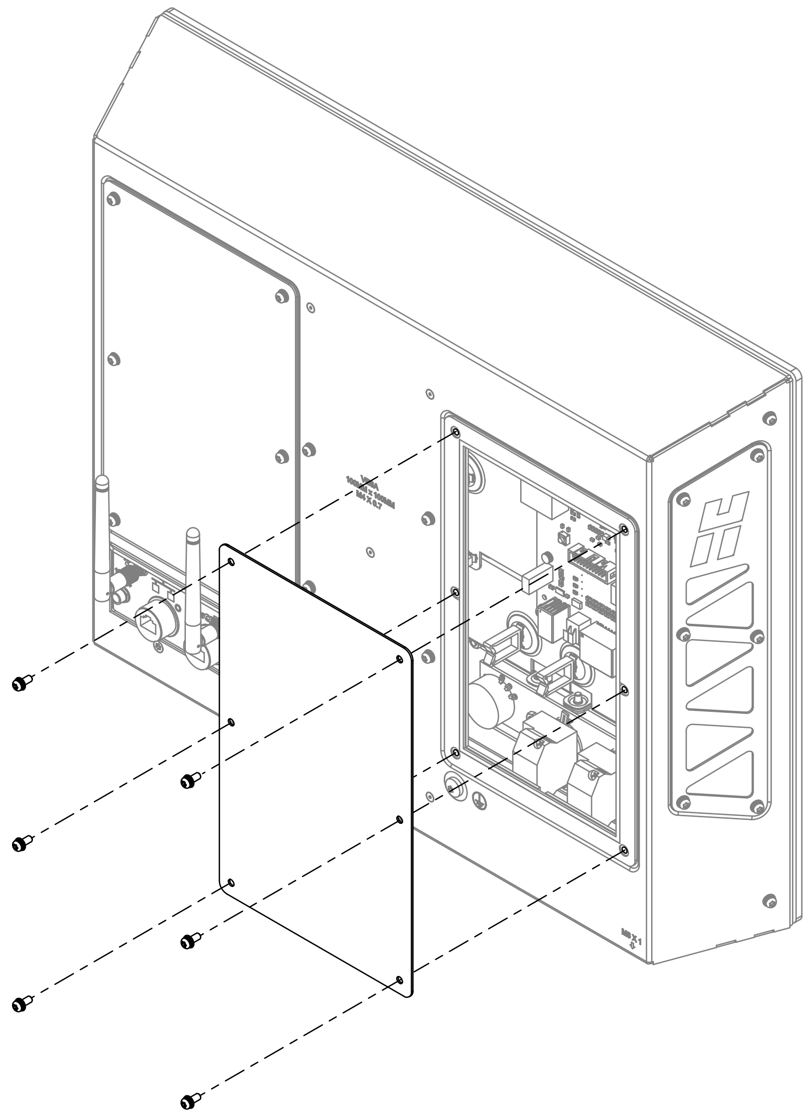


1	Power terminal and wiring harness
2	EtherCAT cable to PCB or EtherCAT connector
3	DVI cable to touchscreen – model number 090185 (469.9 mm/18.5 inch touchscreen) **only**
4	Wireless antenna cables to rear I/O panel
5	HDMI cable to touchscreen – model number 090198 (495.3 mm/19.5 in. touchscreen) **only**
6	HASP software key (top connector)
7	USB to rear USB connector (bottom connector)
8	Ethernet cable to rear I/O panel
9	USB cable to front USB connector
10	USB cable to touchscreen

11. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

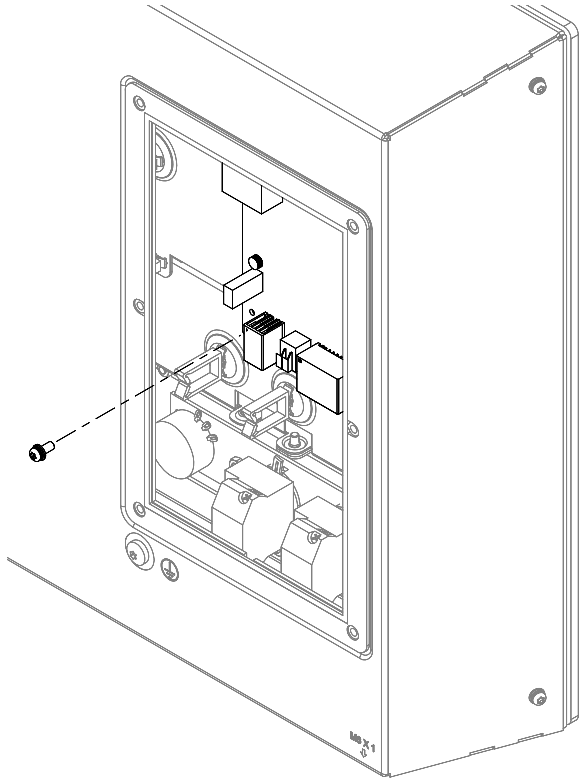
Replace the power distribution or hardware operator console PCB

1. Turn OFF the CNC.
2. Disconnect the power cord from the CNC.
3. Remove the screws from the right rear access panel.
4. Remove the right rear access panel.

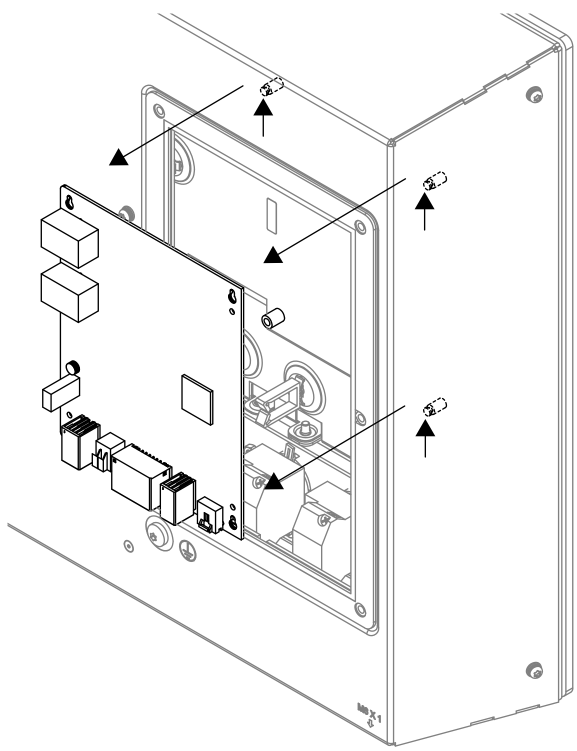


5. Disconnect all of the cables from the PCB.

6. Remove the screw from the PCB.



7. Remove the PCB from the pins.



8. Install the new PCB onto the front panel assembly.

   Make sure to engage all 3 pins.

9. Install the screw.

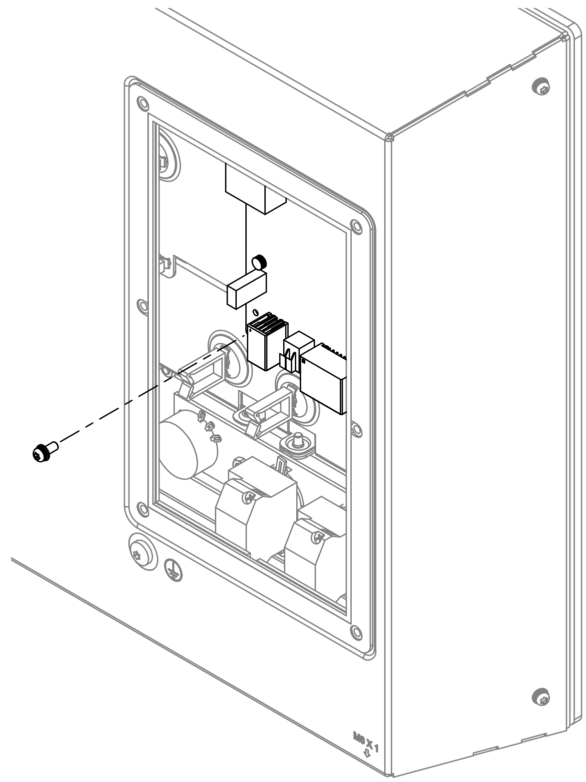
10. Connect the blue EtherCAT cable (from the rear I/O panel) to the OUT connector on the PCB.

11. Connect the yellow EtherCAT cable (from the embedded-CNC) to the IN connector on the PCB.

12. Connect all of the other cables to the PCB.

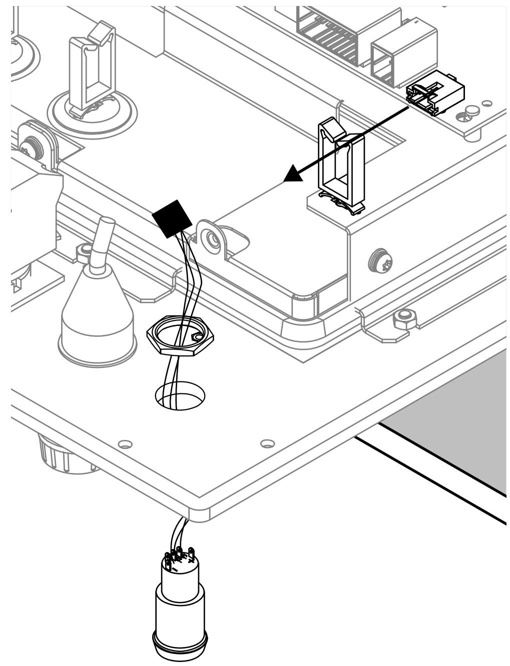
13. Install the right rear access panel.

14. Connect the power cord to the CNC.



Replace the power button

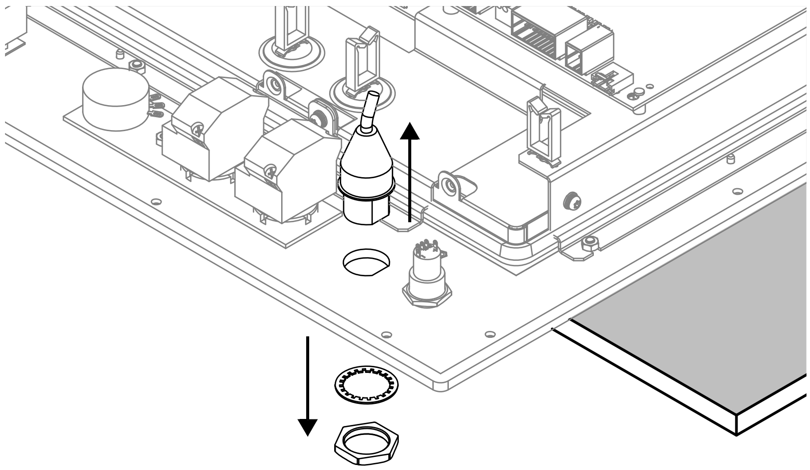
1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
2. Disconnect the power button connector from the PCB.



3. Remove the nut from the power button.
4. Pull the power button and wires through the hole.
5. Install the new power button in the hole.
6. Put the nut over the wires, and tighten it onto the power button to 1.4 N·m (12 in·lb).
7. Connect the power button connector to the PCB.
8. Put the wires through the cable clip.
9. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the front USB connector

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
2. Disconnect the front USB connector cable from the embedded-CNC.
3. Remove the nut and washer from the front of the USB connector.



4. Install the new USB connector in the hole.
5. Install the washer and nut on the front of the USB connector, and tighten the nut to 3.4 N·m (30 in·lb).

6. Connect the USB cable to the embedded-CNC.

7. Put the cable through the cable clips.

Figure 48 – Model number 090198 (495.3 mm/19.5 in. touchscreen)

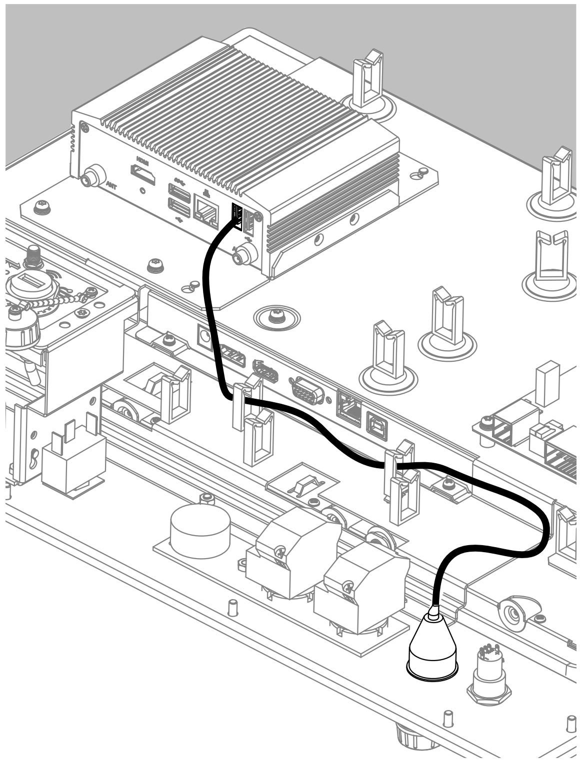
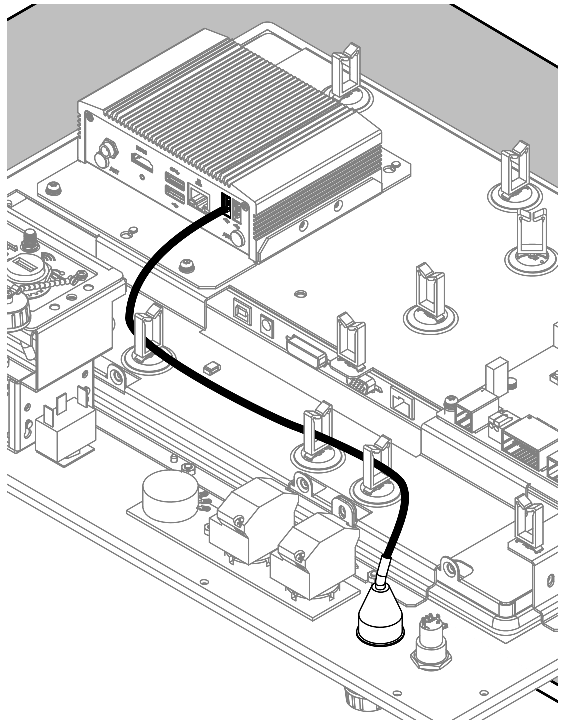


Figure 49 – Model number 090185 (469.9 mm/18.5 inch touchscreen)



8. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the start or stop switch assembly

This procedure shows only the start button. You can follow the same procedure to replace the stop button. Do one button at a time so that you do not interchange the wires. These buttons do not come with a wiring harness.

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
2. Loosen the screws that hold the wire pins in place on the switch block.
3. Remove the wires from the switch block.

Figure 50 – Wires on model number 090198 (495.3 mm/19.5 in. touchscreen)

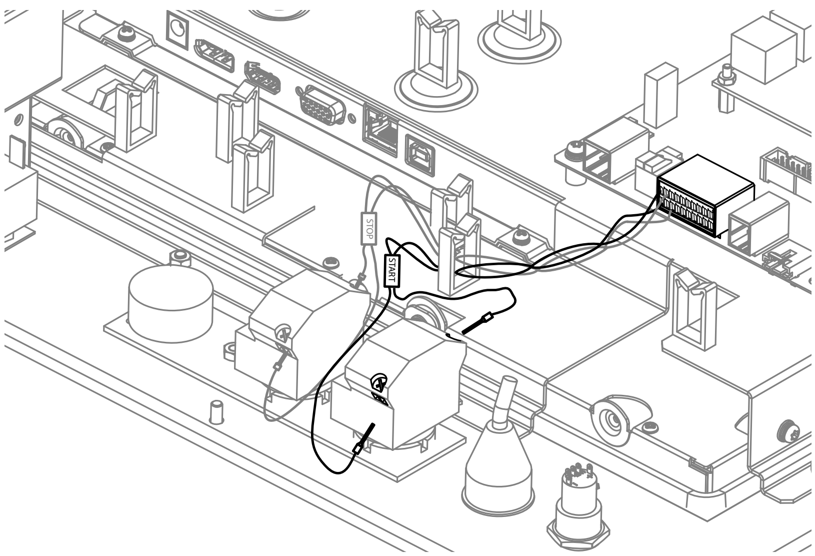
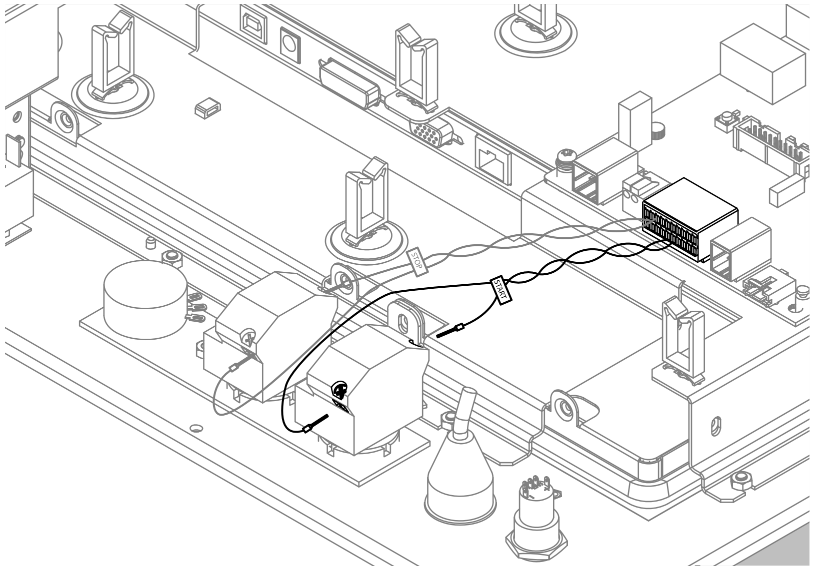
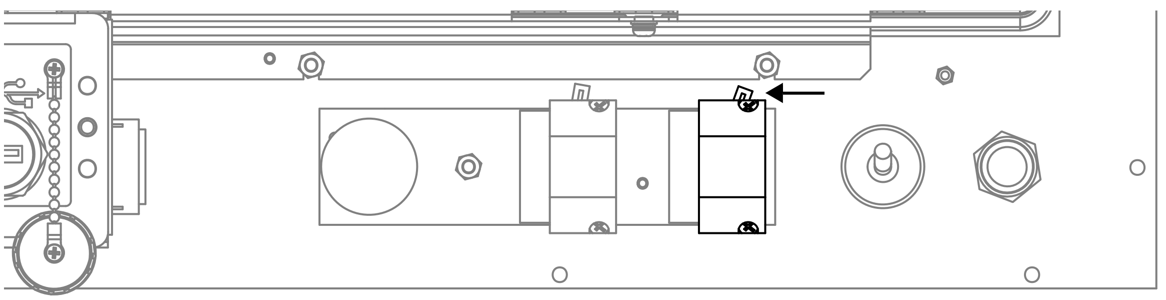


Figure 51 – Wires on model number 090185 (469.9 mm/18.5 inch touchscreen)



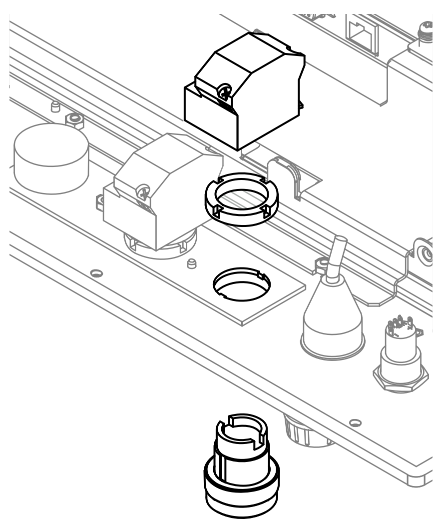
4. Push the latch on the switch block to the left.



5. Remove the switch block.
6. Remove the plastic nut.
7. Remove the button.
8. Install the new button.

   Make sure that the TOP marking on the button is toward the touchscreen.

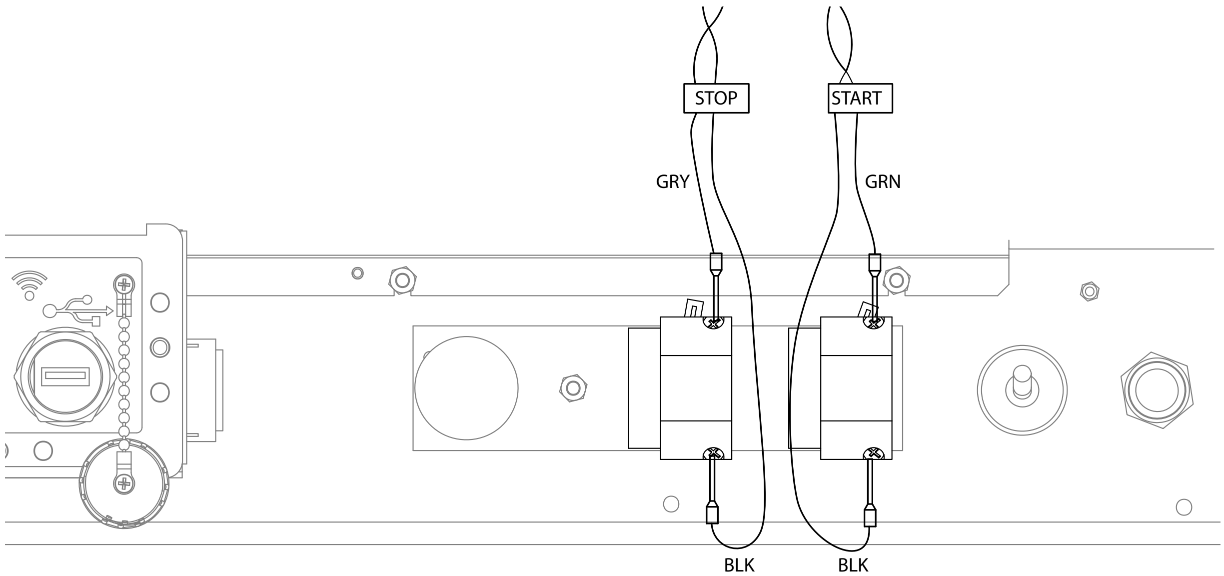
   Make sure that the button locks in position.



9. Install the plastic nut, and tighten to 0.7 N·m (6 in·lb).
10. Install the switch block.

11. Push the latch on the switch block to the right.

12. Install the wire pins into the switch block, and tighten the screws.



Label	Wire color	Signal	J6 pin number
START	Black (BLK)	Start sw com	1A
	Green (GRN)	Start sw	1B
STOP	Black (BLK)	Stop sw com	2A
	Gray (GRY)	Stop sw	2B

13. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the potentiometers and harness

The potentiometers come attached to the wiring harness.

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.

2. Disconnect the connector from J5 on the PCB.

Figure 52 – Model number 090198 (495.3 mm/19.5 in. touchscreen)

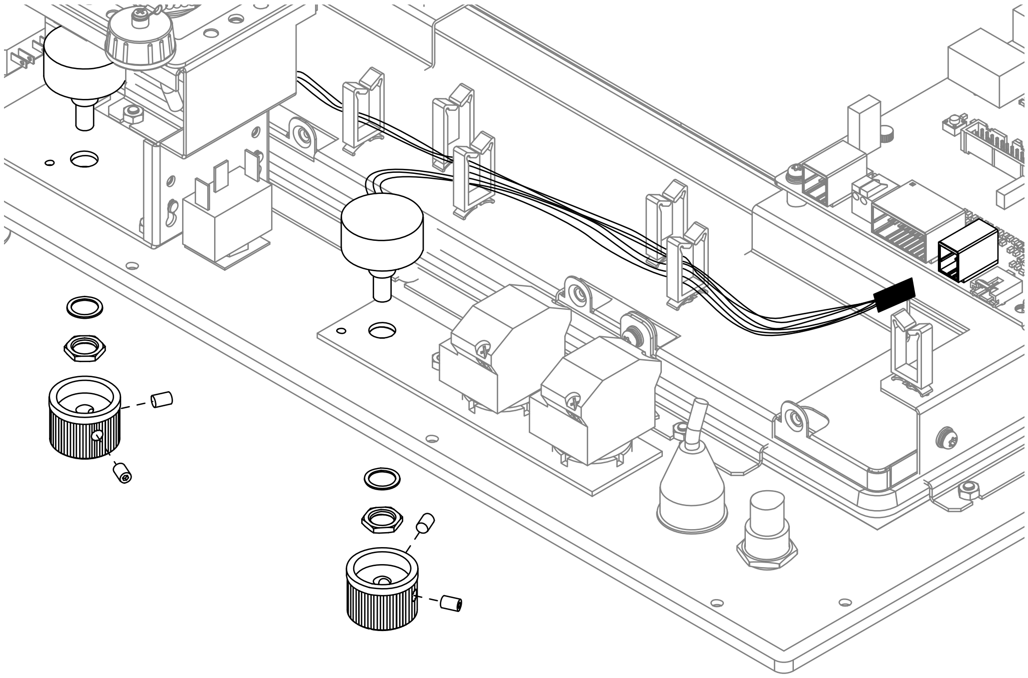
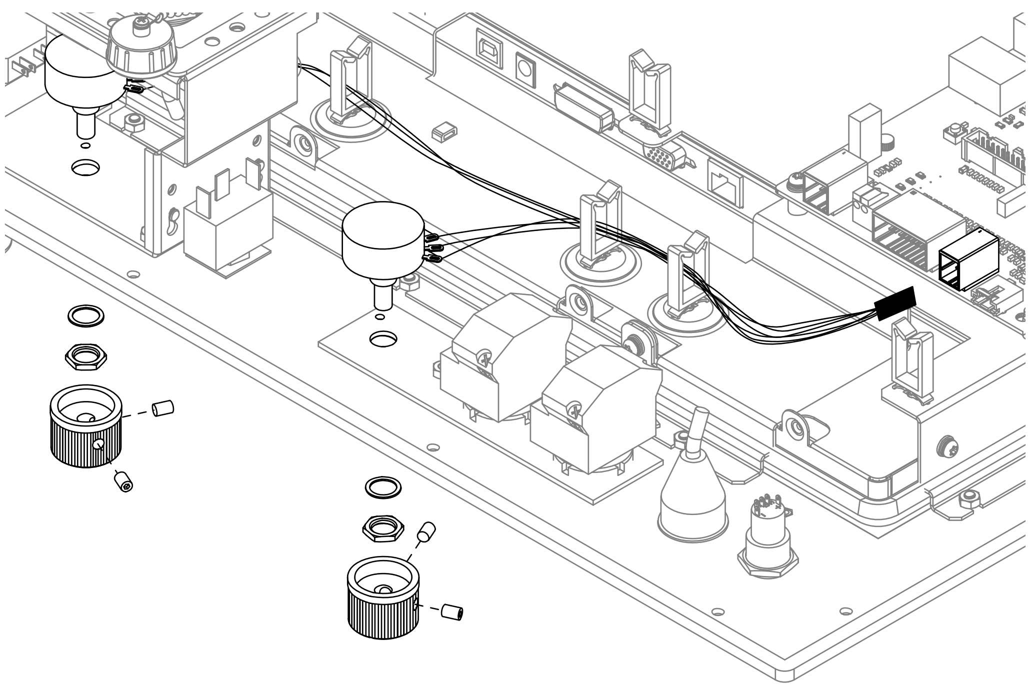


Figure 53 – Model number 090185 (469.9 mm/18.5 inch touchscreen)



3. Use a 5/64-inch Allen® wrench to loosen the set screws one full turn.

4. Remove the knobs.

5. Use a 1/2-inch wrench to remove the nut and lock washer from each potentiometer.

6. Remove the harness and potentiometers.

7. Install the new potentiometers.

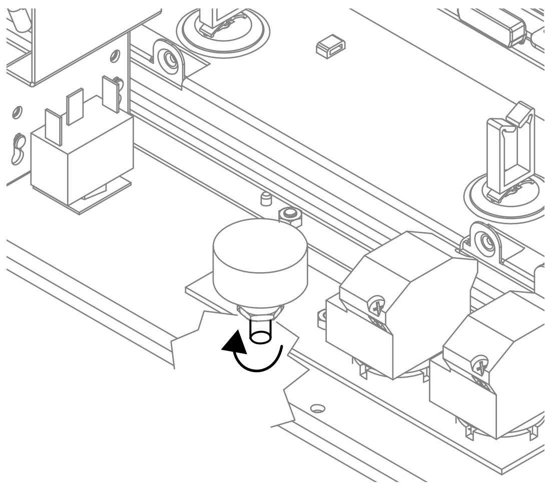
Make sure that the tab on the potentiometer goes into the hole in the front panel.

8. Install the lock washer and nut on each potentiometer, and tighten the nut to 1.4 N·m (12 in·lb).

9. Put the wires in the cable clips.

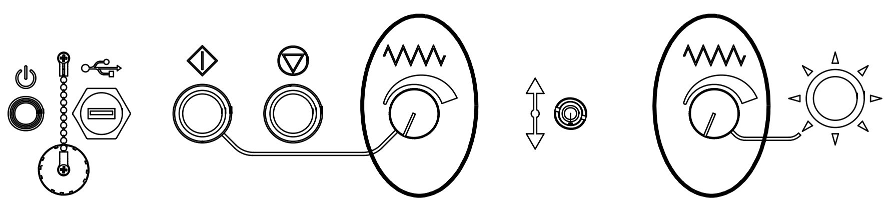
10. Connect J5.

11. Turn each potentiometer shaft clockwise (when viewed from the back of the CNC) until it stops.



12. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

13. Put the knobs onto the potentiometer shafts as shown.



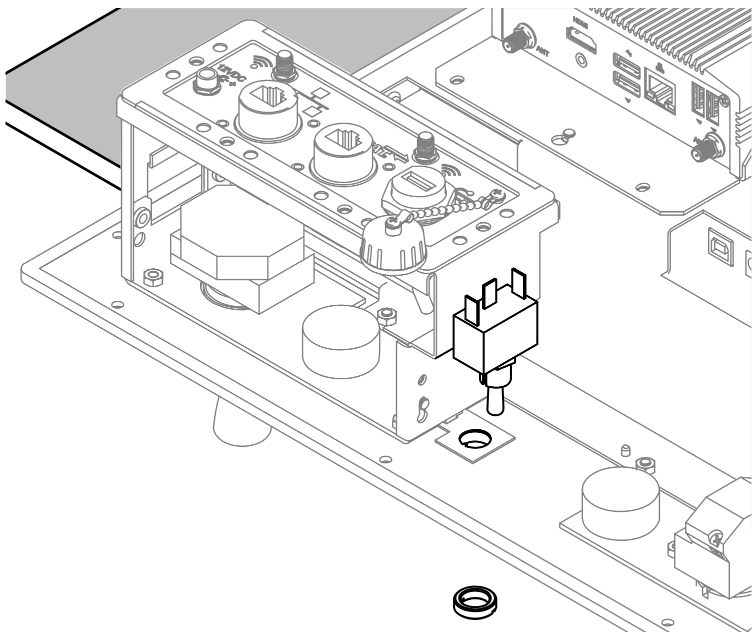
14. Lift the knobs away from the front panel about 1 mm (0.04 inch), and install the set screws.

This prevents the knobs from scratching the paint.

Replace the toggle switch

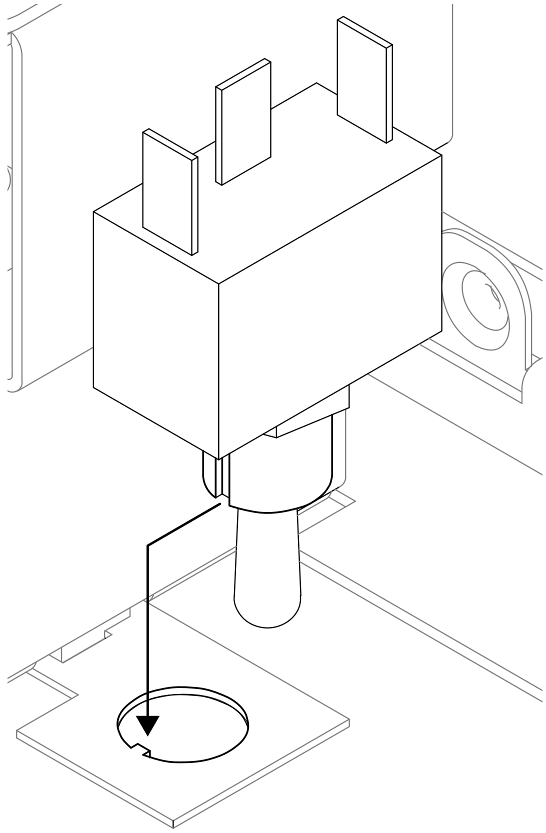
The toggle switch does not come with a wiring harness.

1. Discard the hardware that comes with the switch.
2. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
3. Disconnect the 3 wires from the rear of the toggle switch.
4. Remove the nut from the front of the toggle switch.



5. Remove the toggle switch from the hole.

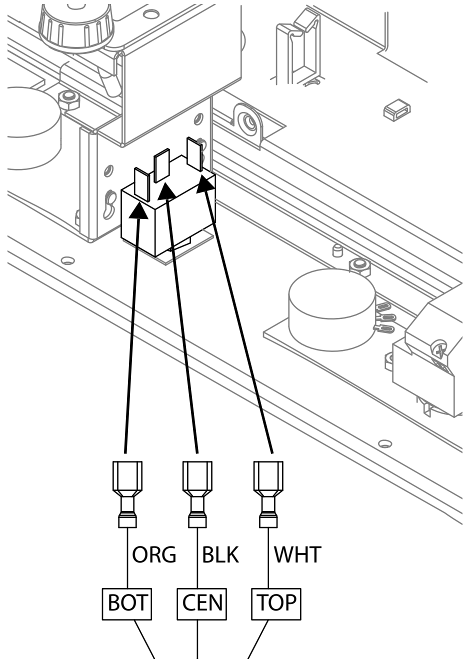
6. Install the new toggle switch.



Make sure that the notch on the toggle switch aligns with the mark in the hole.

7. Install the nut on the front of the toggle switch, and tighten to 1.4 N·m (12 in·lb).

8. Connect the 3 wires to the toggle switch.



Label	Wire color	Signal	J6 pin number
BOT (bottom terminal)	Orange (ORG)	Lift switch up	7B
CEN (center terminal)	Black (BLK)	Lift switch com	7A
TOP (top terminal)	White (WHT)	Lift switch down	8B

9. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the joystick

The joystick does not come with a wiring harness.

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
2. Pull up and then out to remove the rear I/O panel. Tilt it toward the touchscreen. Be careful not to damage any connectors or cables.

Figure 54 – Model number 090198 (495.3 mm/19.5 in. touchscreen)

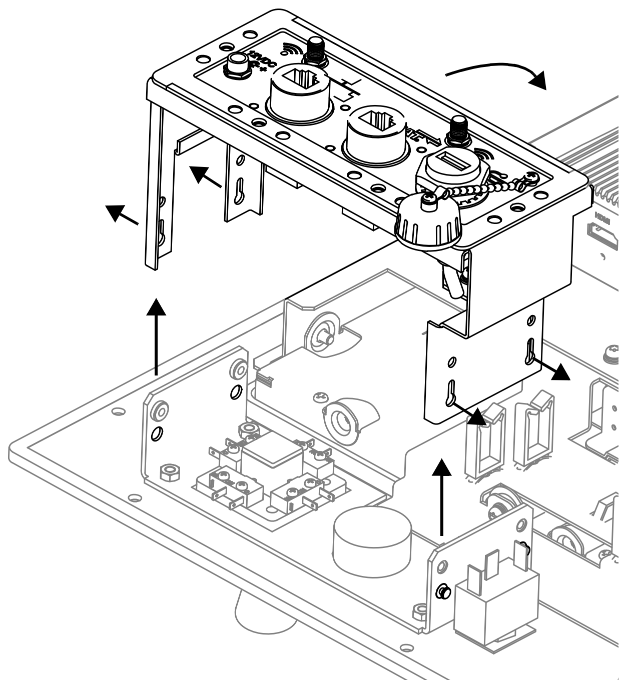
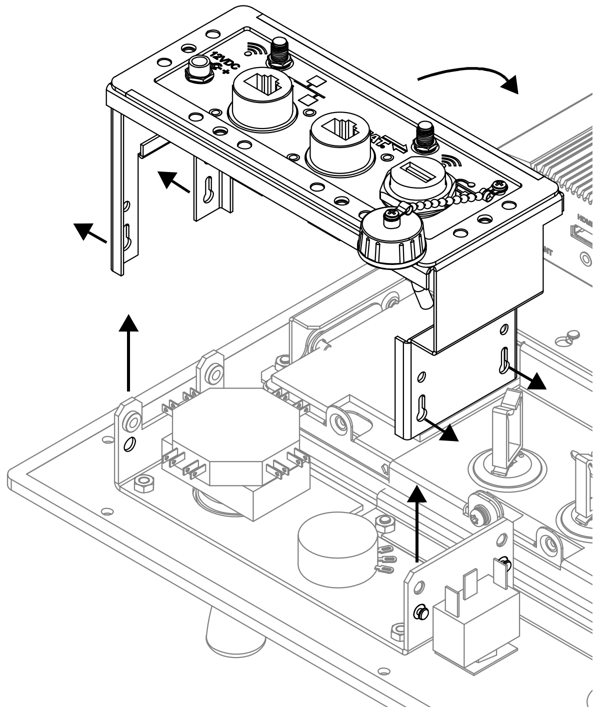


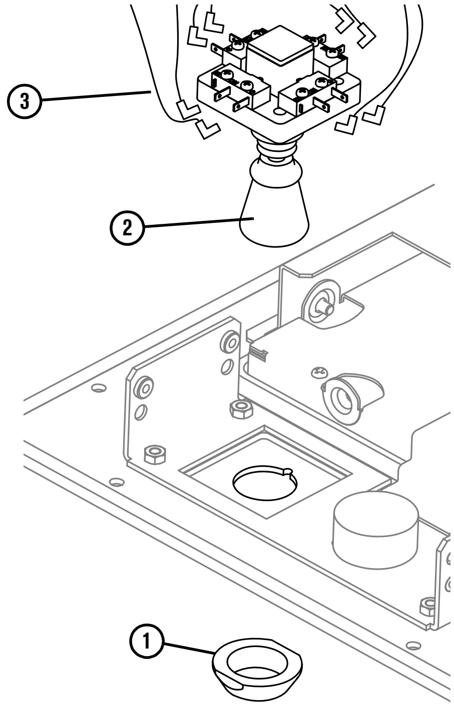
Figure 55 – Model number 090185 (469.9 mm/18.5 inch touchscreen)



3. Remove the joystick. See *Figure 56* on page 316 or *Figure 57* on page 317.

4. Disconnect the connectors from the switch block.

Figure 56 – Remove the joystick from model number 090198 (495.3 mm/19.5 in. touchscreen)

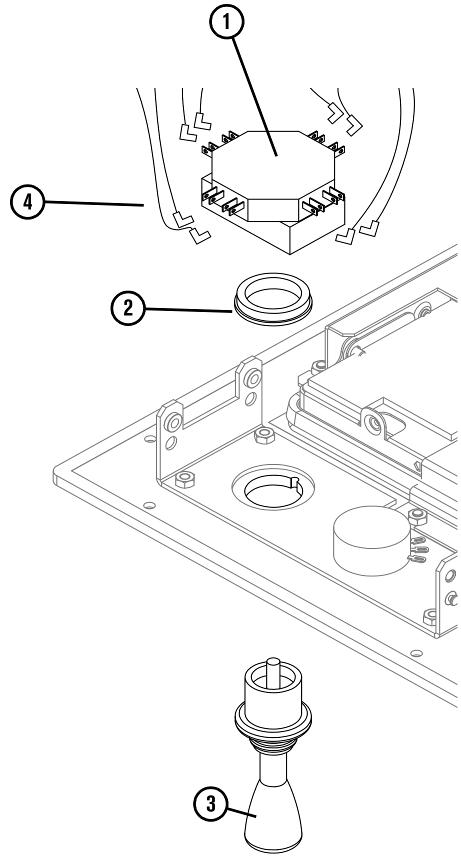


1 Remove the black plastic nut.

2 Remove the joystick and switch block assembly.

3 Disconnect the connectors.

Figure 57 – Remove the joystick from model number 090185 (469.9 mm/18.5 inch touchscreen)



1 Push the latch on the switch block to the right, and remove the switch block.

2 Remove the black plastic nut.

3 Remove the joystick.

4 Disconnect the connectors.

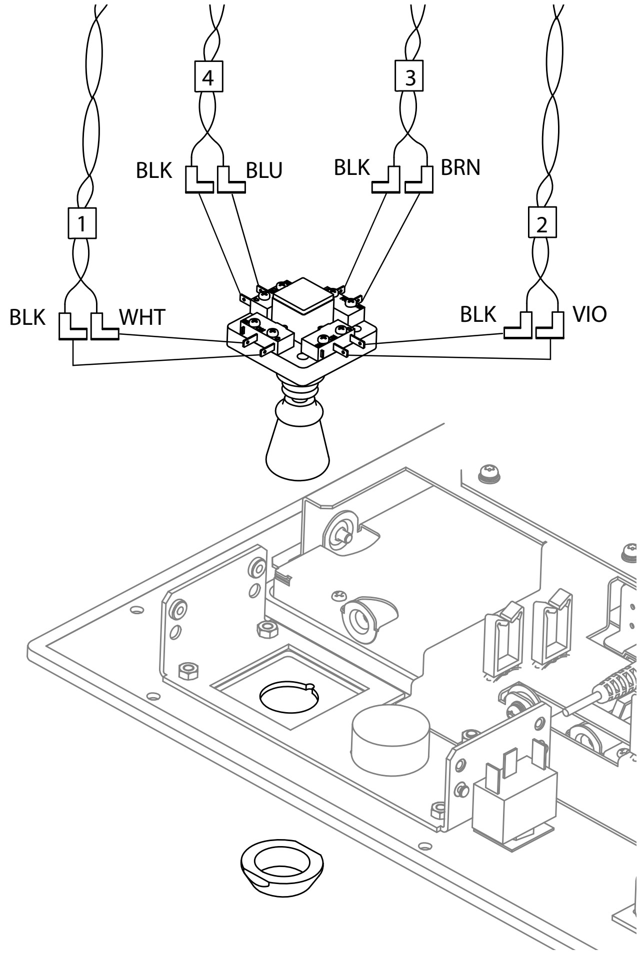
5. Connect the wires to the switch block on the new joystick. See *Figure 58*.

> The joystick does not come with a new wiring harness. Use the same connectors that you disconnected in *step 4* on page 315.

6. Install the new joystick and switch block in the front panel assembly.

7. Install the black plastic nut, and tighten it with your fingers. Do not tighten the nut too much.

Figure 58 – Install the new joystick



For each side, the black wire goes to the pin on the right, and the colored wire goes to the pin on the left.

Wire color	Signal	J6 pin number
Black (BLK)	Joy up com	3A
White (WHT)	Joy up	3B
Black (BLK)	Joy left com	5A
Blue (BLU)	Joy left	5B
Black (BLK)	Joy down com	4A
Brown (BRN)	Joy down	4B
Black (BLK)	Joy right com	6A
Violet (VIO)	Joy right	6B

8. Install the rear I/O panel.

9. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the hardware operator console wiring harness

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
2. Disconnect the connector from J6 on the PCB.
3. Disconnect the wires from the start and stop buttons.
4. Disconnect the wires from the toggle switch.
5. Disconnect the wires from the joystick switch block.

   You may need to remove the joystick from the front panel assembly to access the joystick wires. See *step 2* of *Replace the joystick* on page 314.

Figure 59 – Model number 090198 (495.3 mm/19.5 in. touchscreen)

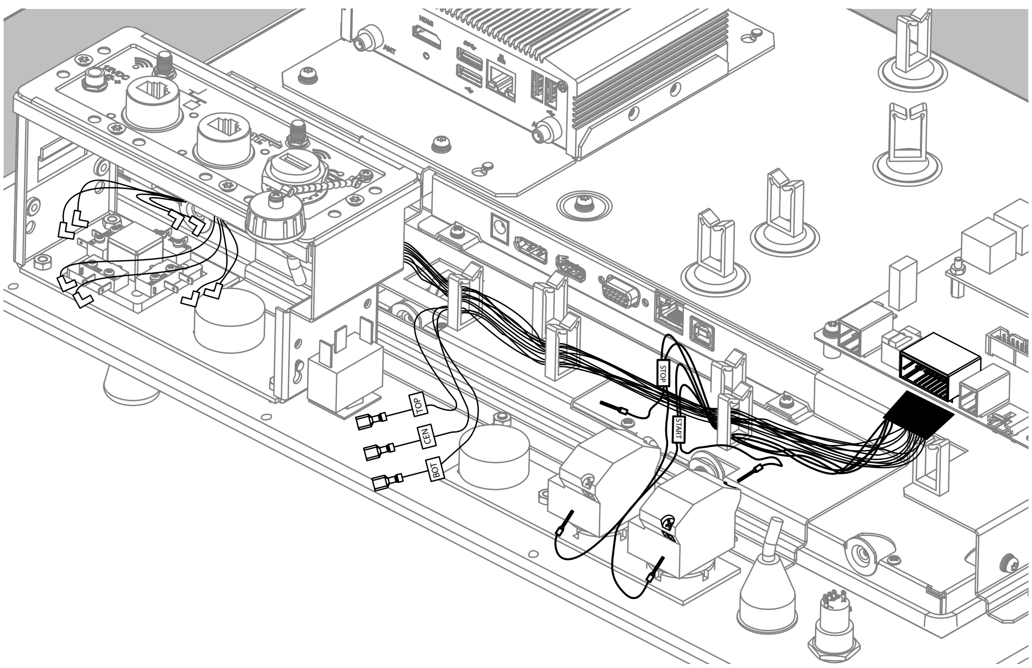
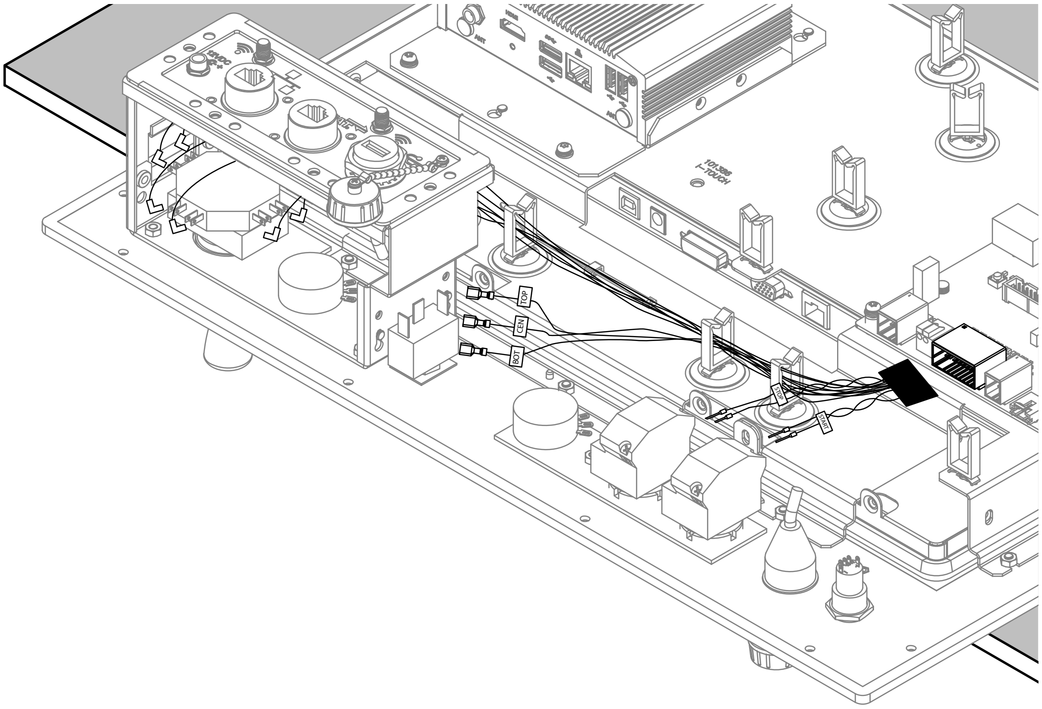


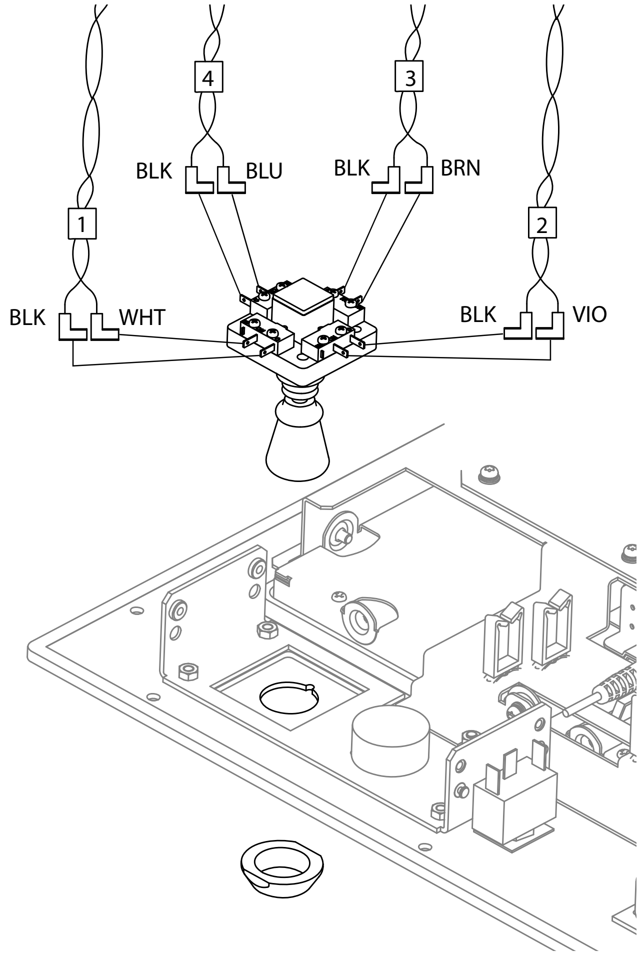
Figure 60 – Model number 090185 (469.9 mm/18.5 inch touchscreen)



6. Remove the wires from the cable clips.

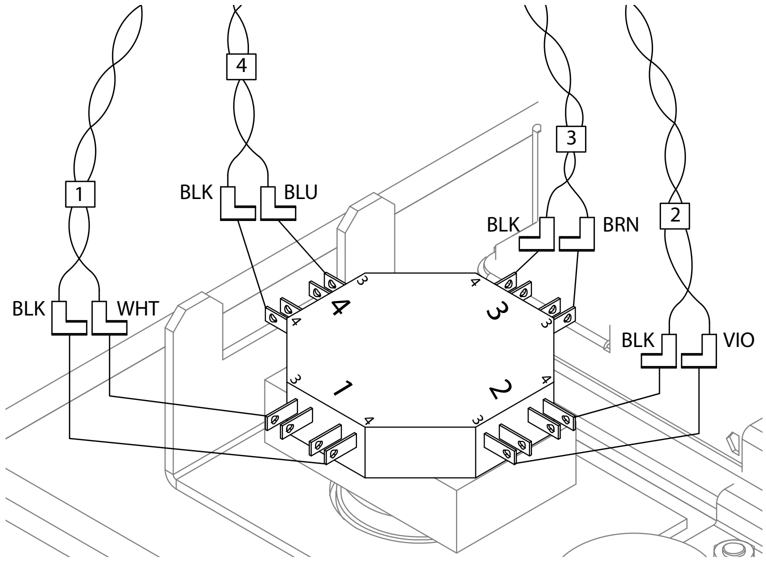
7. Connect the new wires to the joystick switch block.

Figure 61 – Switch block for joystick with part number 005715



For each side, the black wire goes to the pin on the right, and the colored wire goes to the pin on the left.

Figure 62 – Switch block for joystick with part number 005644

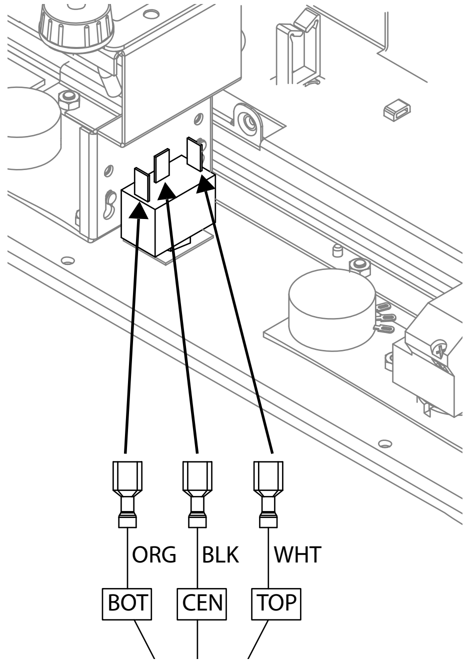


For each side, the black wire goes to pin 4, and the colored wire goes to pin 3.

Figure 63 – Pinouts for 005715 and 005644

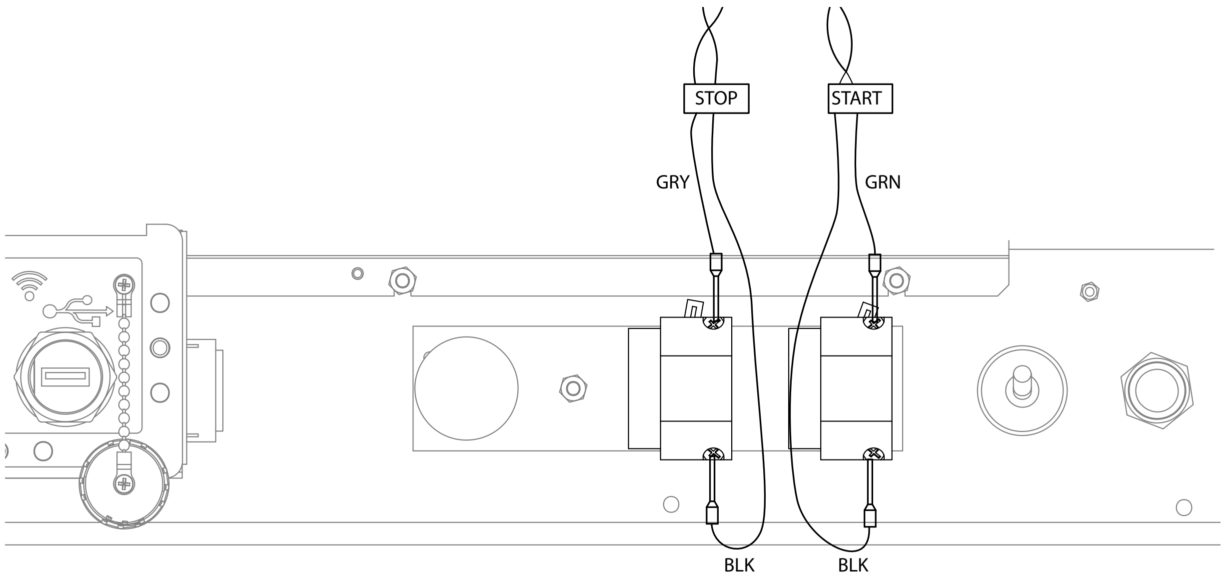
Label (005644 only)	Wire color	Signal	J6 pin number
1	Black (BLK)	Joy up com	3A
	White (WHT)	Joy up	3B
4	Black (BLK)	Joy left com	5A
	Blue (BLU)	Joy left	5B
3	Black (BLK)	Joy down com	4A
	Brown (BRN)	Joy down	4B
2	Black (BLK)	Joy right com	6A
	Violet (VIO)	Joy right	6B

8. Connect the wires to the toggle switch.



Label	Wire color	Signal	J6 pin number
BOT	Orange (ORG)	Lift switch up bottom	7B
CEN	Black (BLK)	Lift switch com center	7A
TOP	White (WHT)	Lift switch down top	8B

9. Connect the wires to the start and stop buttons and tighten the screws.

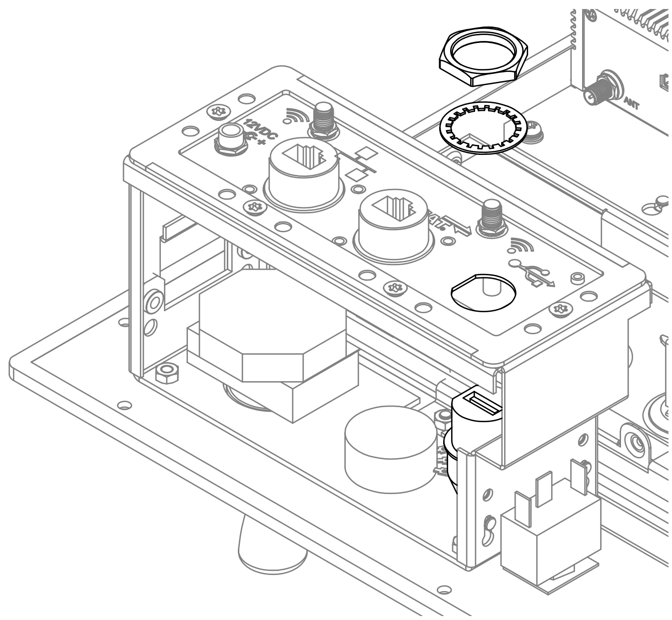


Label	Wire color	Signal	J6 pin number
START	Black (BLK)	Start sw com	1A
	Green (GRN)	Start sw	1B
STOP	Black (BLK)	Stop sw com	2A
	Gray (GRY)	Stop sw	2B

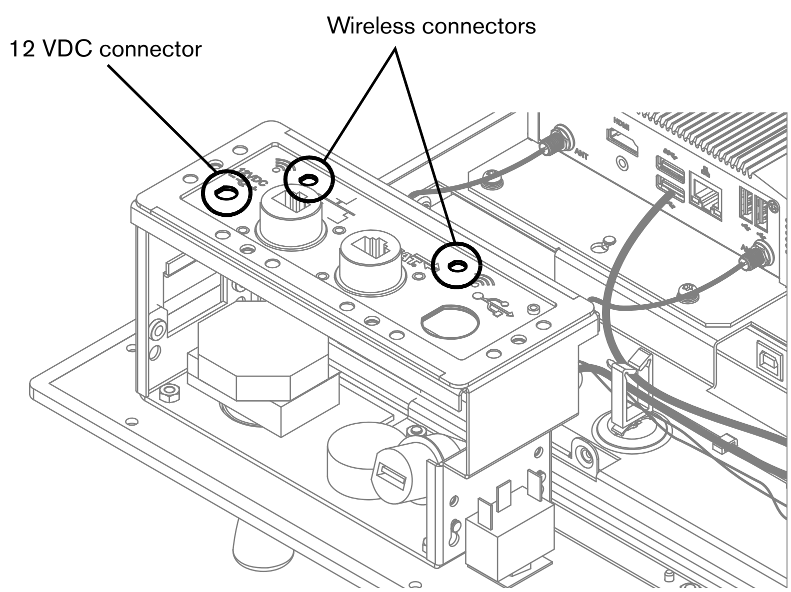
10. Put the wires through the cable clips.
11. Connect the connector to J6 on the PCB.
12. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the rear I/O panel

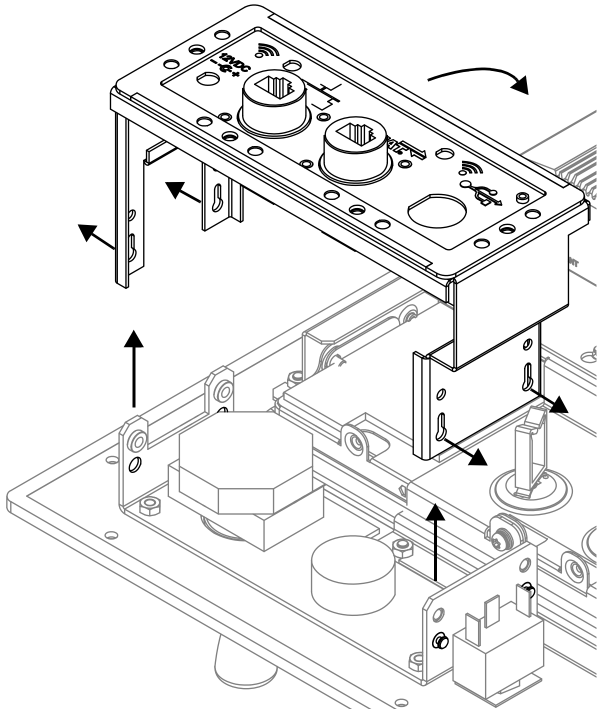
1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
2. Remove the USB cover assembly.
3. Remove the nut and washer from the front of the USB connector.
4. Remove the USB connector from the rear I/O panel.



5. Remove the nut and lock washer from the front of both of the wireless connectors.
6. Remove the wireless connectors from the rear I/O panel.
7. Remove the nut and washer from the front of the 12 VDC connector.
8. Remove the 12 VDC connector from the rear I/O panel.



9. Pull up and then out to remove the rear I/O panel. Point it toward the touchscreen. Be careful not to damage any connectors.



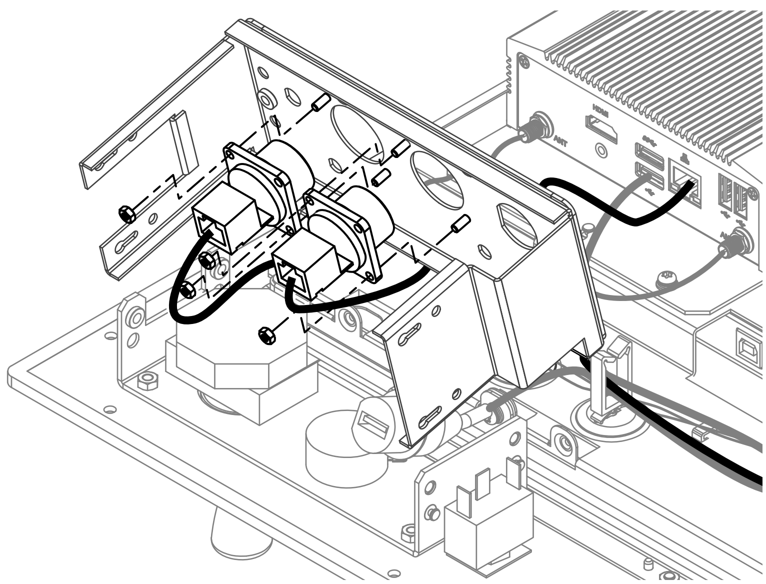
10. Remove the 4 nuts from the Ethernet and EtherCAT connectors.

11. Remove the Ethernet and EtherCAT connectors from the rear I/O panel.

12. In the new rear I/O panel, install the Ethernet and EtherCAT connectors.

13. Install the 4 nuts on the connectors. Tighten them to 0.7 N·m (6 in·lb).

14. Install the new rear I/O panel on the front panel assembly.



15. Install the 12 VDC connector in the rear I/O panel.

16. Install the washer and nut on the front of the 12 VDC connector, and tighten the nut to 1.4 N·m (12 in·lb).

17. Install the wireless connectors in the rear I/O panel.

18. Install the lock washer and nut on the front of both of the wireless connectors, and tighten the nut to 1.1 N·m (10 in·lb).

19. Install the USB connector in the rear I/O panel.

20. Install the washer and nut on the front of the USB connector, and tighten the nuts to 3.4 N·m (30 in·lb).

21. Install the USB cover assembly.

22. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the rear USB connector

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.

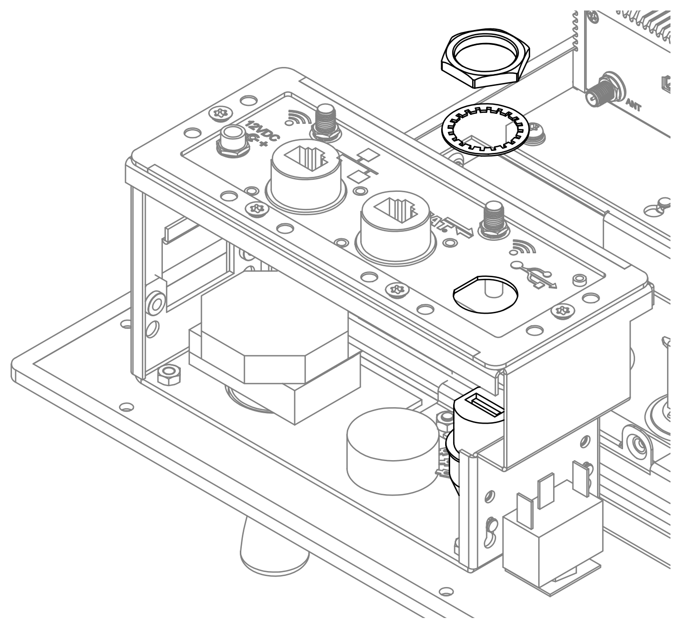
2. Disconnect the USB cable that goes to the rear USB connector from the embedded-CNC.

3. Remove the nut and washer from the front of the USB connector.

4. Remove the USB connector from the rear I/O panel.

5. Install the new USB connector in the rear I/O panel.

6. Install the washer and nut on the front of the USB connector, and tighten the nut to 3.4 N·m (30 in·lb).



7. Connect the USB cable to the embedded-CNC.

Figure 64 – Model number 090198 (495.3 mm/19.5 in. touchscreen)

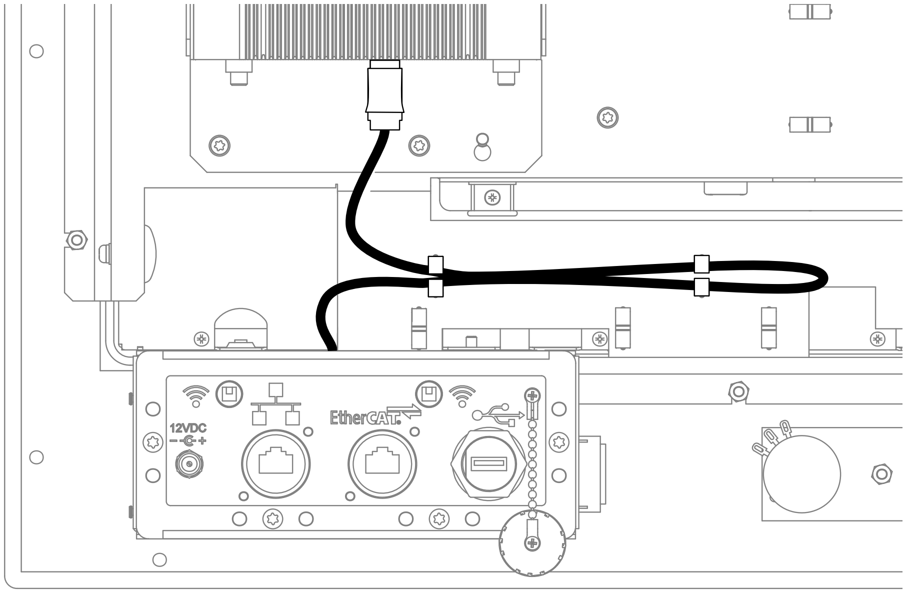
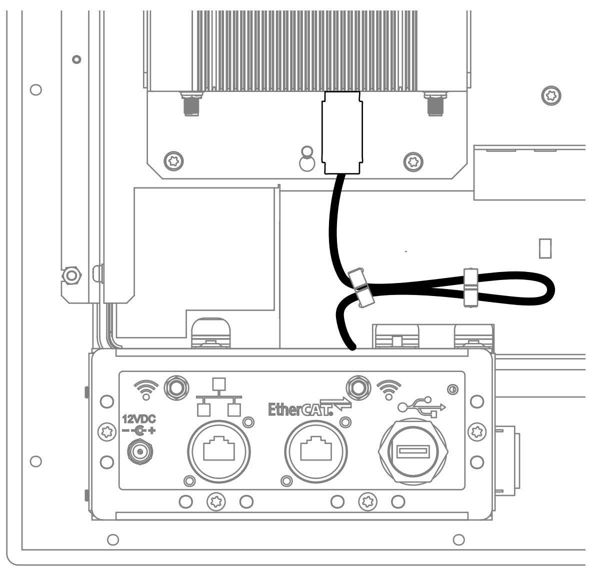


Figure 65 – Model number 090185 (469.9 mm/18.5 inch touchscreen)



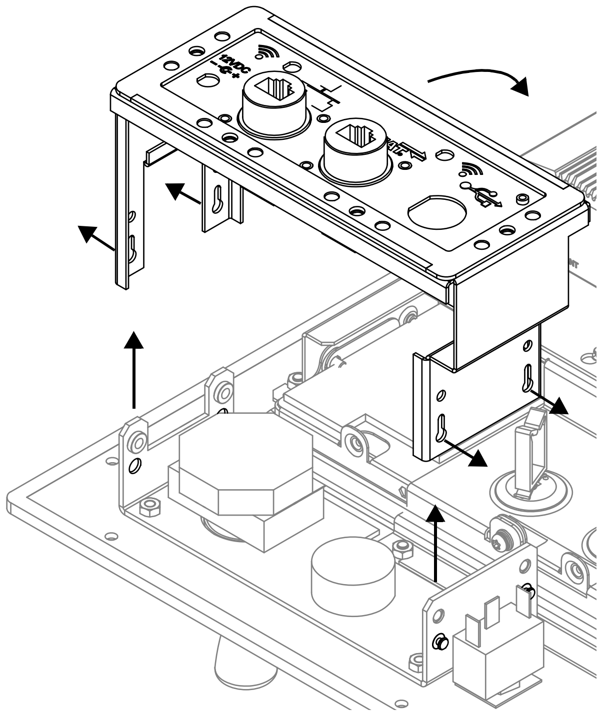
8. Put the cable through the cable clips.

9. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.

Replace the Ethernet or EtherCAT connector

This procedure shows both connectors.

1. Remove the rear enclosure. See *Remove the rear enclosure* on page 280.
2. Pull up and then out to remove the rear I/O panel. Point it toward the touchscreen. Be careful not to damage any connectors.



3. Disconnect the cable from the rear of the connector that you are replacing.

4. Remove the 2 nuts from the rear of the connector.

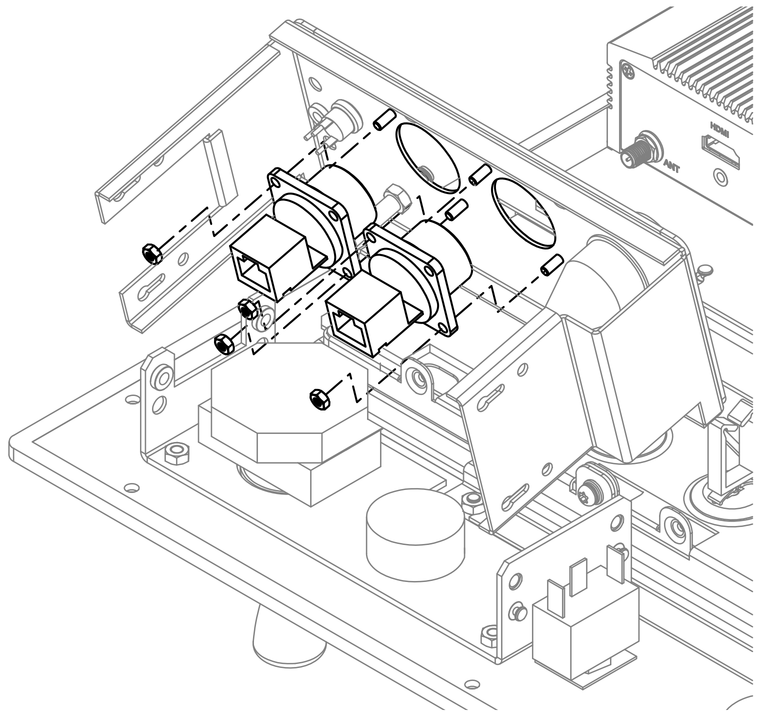
5. Install the new connector.

6. Install the 2 nuts. Tighten them to 0.7 N·m (6 in·lb).

7. Connect the blue cable to the EtherCAT connector from PCB. Connect the red cable to the Ethernet LAN connector from the embedded-CNC.

8. Install the rear I/O panel.

9. Install the rear enclosure on the front panel assembly. See *Install the rear enclosure* on page 283.



16 Parts

For instructions on part replacement, see *Part Replacement* on page 279.

Internal EtherCAT cables

Part number	For model	Color	Length	Quantity
223503	EDGE Connect TC	Blue	0.53 m (21 in.)	1
223504	EDGE Connect TC	Yellow	0.36 m (14 in.)	1
223505	EDGE Connect TC	Red	0.23 m (9 in.)	1

Touchscreen cables

Part number	For model	Cable type	Length	Quantity
223474	EDGE Connect TC	USB	0.30 m (12 in.)	1
223475	EDGE Connect TC – model number 090185 (18.5 inch touchscreen) **only**	Video DVI-D	0.61 m (24 in.)	1
223740	EDGE Connect TC – model number 090198 (19.5 inch touchscreen) **only**	Video HDMI	0.30 m (12 in.)	1

Enclosure and hardware

Part number	For model	Description	Quantity
101378	EDGE Connect TC – model number 090185 (18.5 inch touchscreen) **only**	Rear enclosure	1
101467	EDGE Connect TC – model number 090198 (19.5 inch touchscreen) **only**	Rear enclosure	1
428633	EDGE Connect TC	Kit: Plastic cable clips (13)	
428634	EDGE Connect TC	Kit: Screws	

Front panel and rear I/O panel parts

Part number	For model	Description	Quantity
104827	EDGE Connect/EDGE Connect TC	USB cover with chain assembly	1
223244	EDGE Connect/EDGE Connect TC	USB panel mount connector bulkhead with cable, 0.5 m (19.7 in.)	1
223501	EDGE Connect/EDGE Connect TC	Wireless antenna extension cable, 0.18 m (7 in.)	1
228445	EDGE Connect/EDGE Connect TC	Ethernet connector	1
229736	EDGE Connect/EDGE Connect TC	Power button with wiring harness	1
229762	EDGE Connect/EDGE Connect TC	Rear I/O panel with gasket (does not include connectors)	1
428631	EDGE Connect/EDGE Connect TC	Kit: 12 VDC power supply (1), AC rewireable connector (1), AC cord (1), and plug for 120 V (1)	

Internal parts

Part number	For model	Description	Quantity
229728	EDGE Connect/EDGE Connect TC	12 VDC power input with wiring harness	1
229827	EDGE Connect TC	Operator console PCB	1
229840	EDGE Connect/EDGE Connect TC	Embedded-CNC Note: Disconnect and keep the HASP software key from the defective embedded-CNC to use in the replacement embedded-CNC.	1
428632	EDGE Connect/EDGE Connect TC – model number 090185 (18.5 inch touchscreen) **only**	Kit: 469.9 mm (18.5 inch) touchscreen (1), DVI cable (1), USB cable (1), screws (9)	
428652	EDGE Connect/EDGE Connect TC	HASP software key	1
428765	EDGE Connect TC – model number 090198 (19.5 inch touchscreen) **only**	Kit: 495.3 mm (19.5 inch) touchscreen (1), HDMI cable (1), USB cable (1), screws (9), cable ties (2)	

Operator console parts

Part number	For model	Description	Quantity
005688	EDGE Connect TC	Stop switch assembly (red)	1
005689	EDGE Connect TC	Start switch assembly (green)	1
005715	EDGE Connect TC	Joystick	1
104827	EDGE Connect/EDGE Connect TC	USB cover with chain assembly	1
108861	EDGE Connect TC	Potentiometer knob, 24.8 mm (0.975 inch) diameter	1
223244	EDGE Connect TC	USB panel mount connector bulkhead with cable, 0.5 m (19.7 in.)	1
229727	EDGE Connect TC	Wiring harness for toggle switch, start and stop buttons, and joystick	1
229729	EDGE Connect TC	Potentiometers with wiring harness	2
229736	EDGE Connect TC	Power button with wiring harness	1
428635	EDGE Connect TC	Kit: Toggle switch (1) and nut (1)	

Optional accessories

Wireless antenna extension cables

Part number	For model	Length	Quantity
223250	EDGE Connect	0.30 m (12 in.)	1
223251	EDGE Connect	3.1 m (10 feet)	1

Ethernet and EtherCAT bulkhead connector

Part number	For model	Quantity
208367	EDGE Connect	1

Mounting options

Part number	For model	Description	Quantity
104844	EDGE Connect	Wall mount for EDGE Connect	1
104845	EDGE Connect	DIN rail mount for EDGE Connect	1
428636	EDGE Connect TC	Kit: Rear or flush mount bracket (1) with screws (4) for EDGE Connect TC	

USB with instruction manuals

Part number	For model	Quantity
428628	EDGE Connect/EDGE Connect TC	1

EtherCAT test PCB

Part number	For model	Quantity
428715	EDGE Connect TC	1

* If you want to connect a Powermax® or MAXPRO® to an EDGE Connect CNC, use a discrete connection through the drive I/O or an I/O module on the network.

External EtherCAT cables

All EtherCAT cables used in the cutting system must follow the guidelines in *EtherCAT cable* on page 36 (EDGE Connect) or *EtherCAT cable* on page 43(EDGE Connect TC).

All Hypertherm cables are manufactured to these guidelines and are listed here.

Part number	Length	Quantity
223506	0.3 m (1 feet)	1
223507	0.6 m (2 feet)	1
223508	1.5 m (4.9 feet)	1
223672	2.5 m (8.2 feet)	1
223509	3 m (9.8 feet)	1
223510	6 m (19.7 feet)	1
223511	7.5 m (24.6 feet)	1
223512	10 m (32.8 feet)	1
223513	15 m (49.2 feet)	1
223514	22.5 m (73.8 feet)	1
223515	30 m (98.4 feet)	1
223516	45 m (147.6 feet)	1
223517	60 m (196.9 feet)	1
223714	75 m (246.1 feet)	1

EtherCAT drives and I/O modules

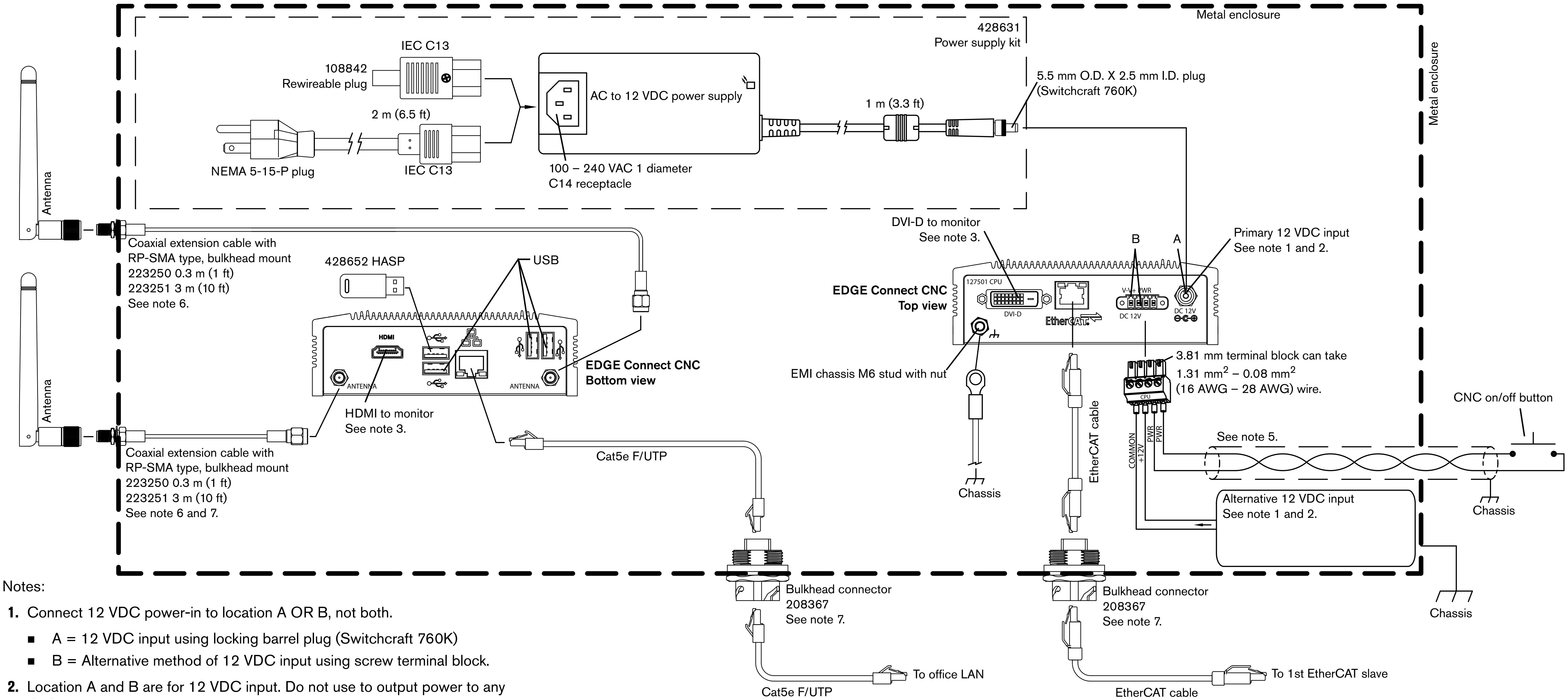
For a list of supported EtherCAT drives and I/O modules, see the *EtherCAT Devices Supported by EDGE Connect CNCs* Application Note (809660). Technical documentation is available at www.hypertherm.com/docs.

If you want to connect a Powermax® or MAXPRO® to an EDGE Connect CNC, use a discrete connection through the drive I/O or an I/O module on the network.

17 Wiring Diagrams

The following diagrams are connection diagrams for the EDGE Connect and EDGE Connect TC.

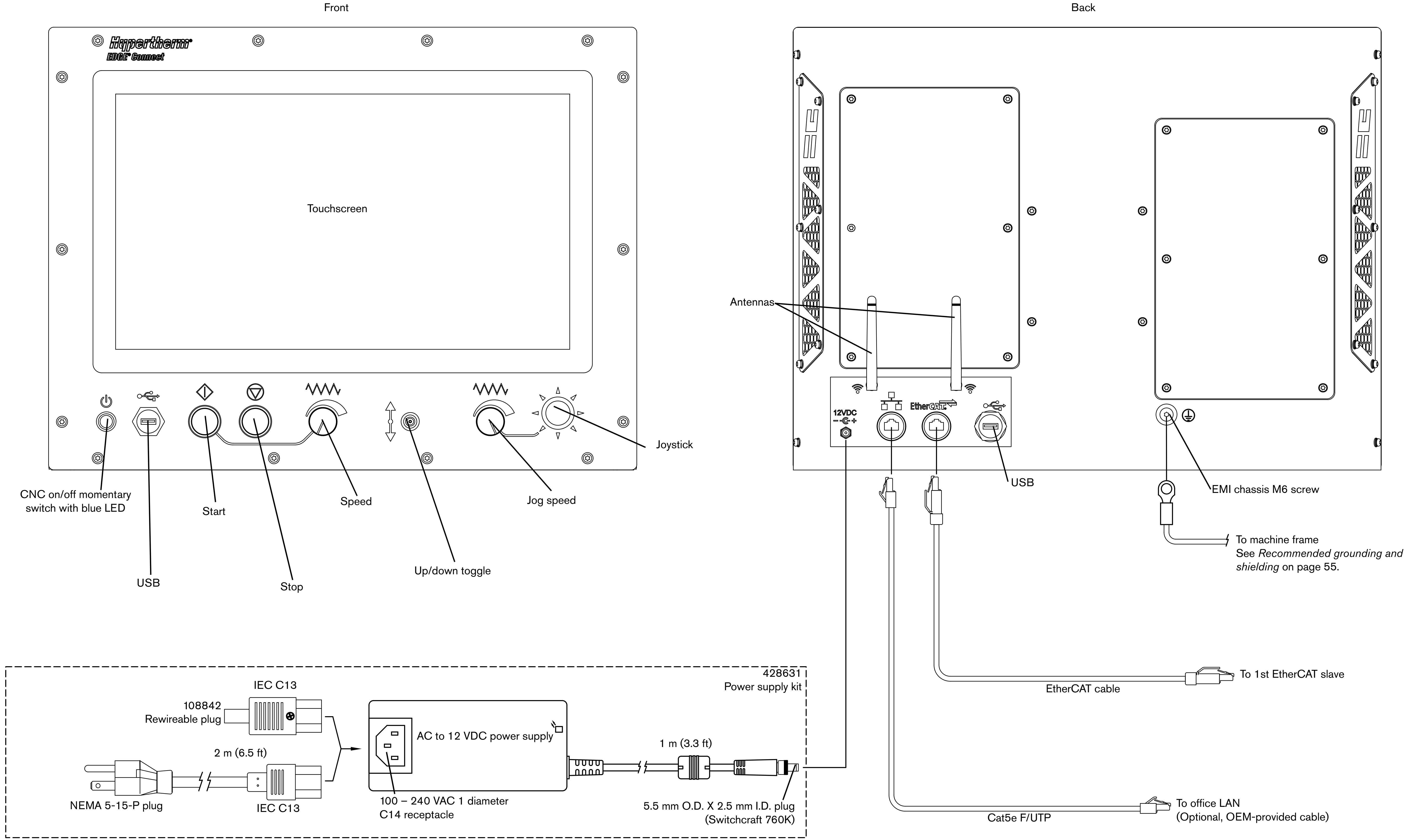
EDGE Connect CNC connection diagram (Sheet 1 of 1)



Notes:

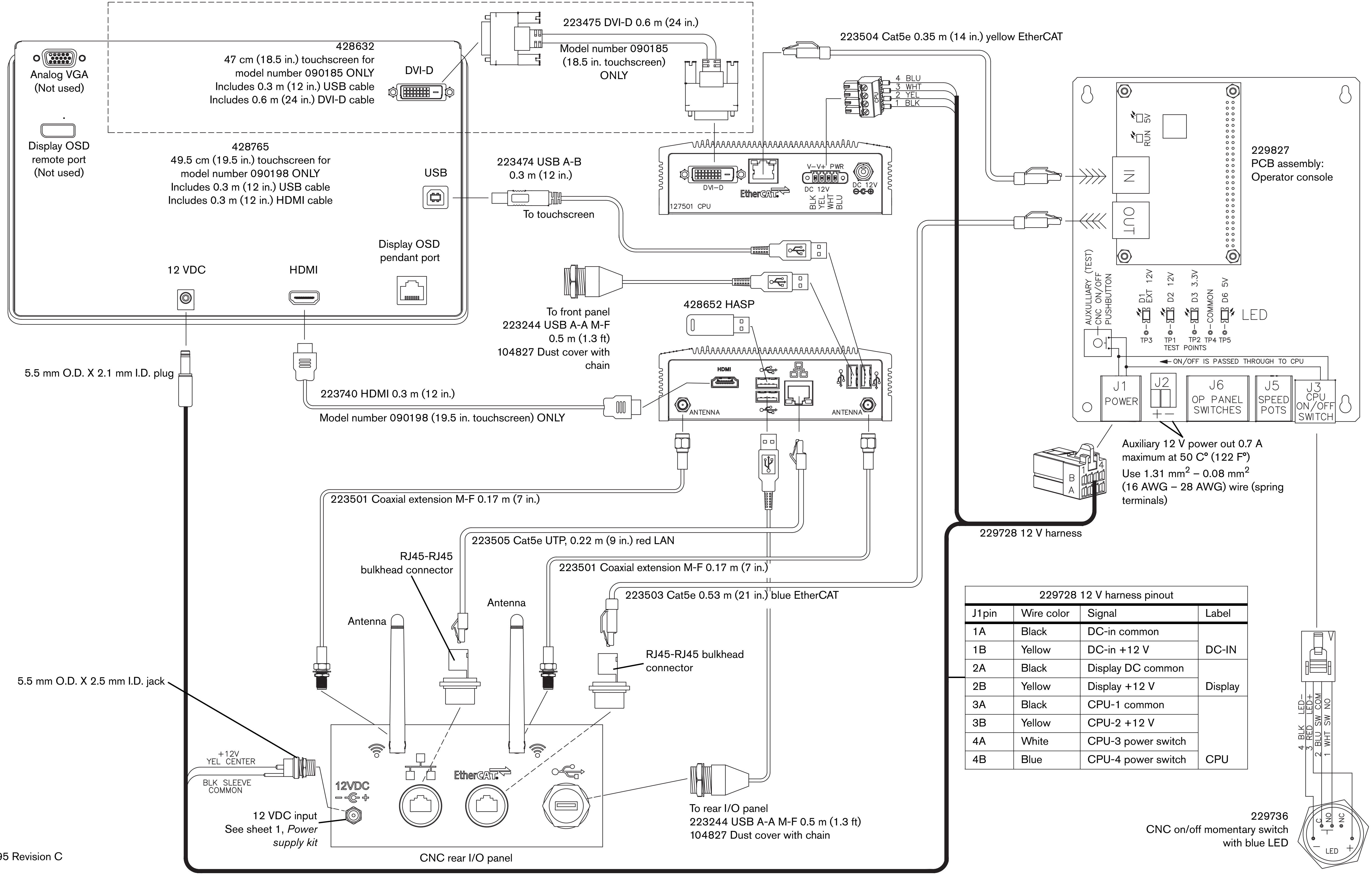
1. Connect 12 VDC power-in to location A OR B, not both.
   - A = 12 VDC input using locking barrel plug (Switchcraft 760K)
   - B = Alternative method of 12 VDC input using screw terminal block.
2. Location A and B are for 12 VDC input. Do not use to output power to any devices.
3. Connect either DVI-D or HDMI type monitor.
4. Monitor requirements: 1366 X 768 resolution, if touchscreen, at least 47 cm (18.5 inch) diagonal. See *Monitor* on page 32.
5. On/off button and cable, OEM provided. Requirements:
   - SPST, momentary, gold plated contacts
   - Use a minimum 0.33 mm^2 (22 AWG) shielded twisted pair cable. 5 m (16 ft) maximum length.
   - Terminate the cable shield to chassis at the switch end.
6. Mount the wireless antennas on the exterior of the overall metal enclosure.
7. Make sure that the bulkhead connectors electrically bond to the metal enclosure.
8. Shorter cable lengths are better to reduce signal attenuation. Test in place to make sure that the cables are compatible and robust. Maximum lengths:
   - USB cable: 5 m (16 ft)
   - DVI-D cable: 5 m (16 ft)
   - HDMI cable: 5 m (16 ft)
   - Antenna coaxial extension cables: 3 m (10 ft)

013394 Revision A



013395 Revision C

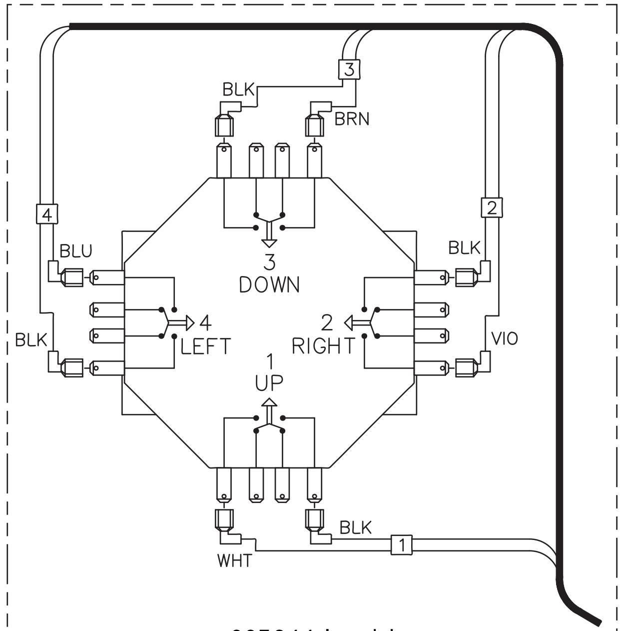
EDGE Connect TC CNC connection diagram (Sheet 2 of 3)



229728 12 V harness pinout

J1pin	Wire color	Signal	Label
1A	Black	DC-in common	
1B	Yellow	DC-in +12 V	DC-IN
2A	Black	Display DC common	
2B	Yellow	Display +12 V	Display
3A	Black	CPU-1 common	
3B	Yellow	CPU-2 +12 V	
4A	White	CPU-3 power switch	
4B	Blue	CPU-4 power switch	CPU

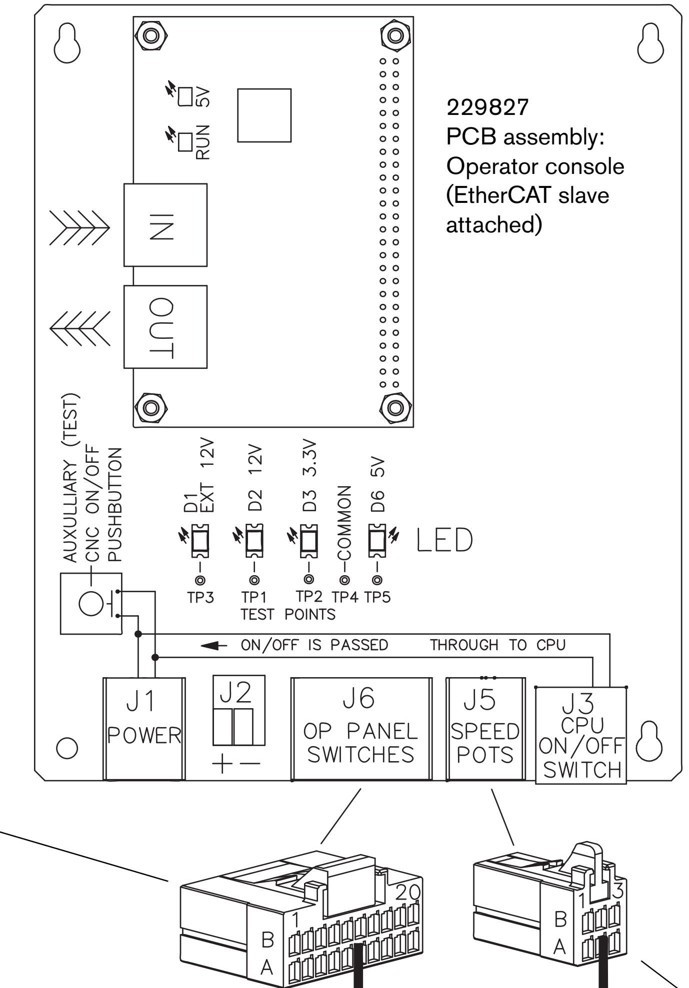
013395 Revision C



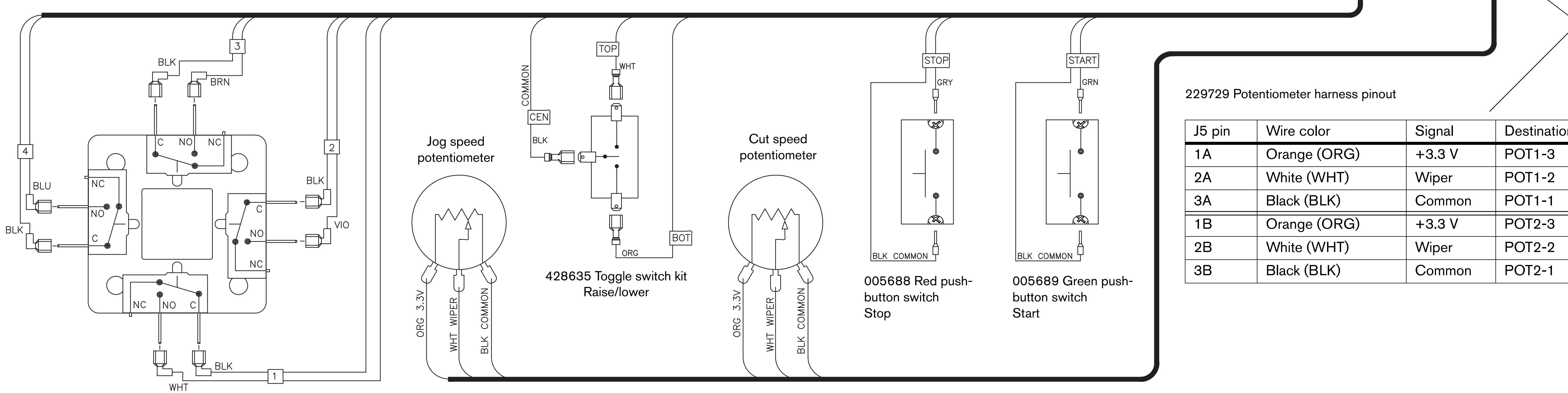
005644 Joystick
Original equipment in model number 090185 (18.5 in. touchscreen) ONLY
Replaced with joystick 005715

229727 I/O harness pinout

J6 pin	Wire color	Signal	Label
1A	Black (BLK)	Start sw com	START
1B	Green (GRN)	Start sw	
2A	Black (BLK)	Stop sw com	STOP
2B	Gray (GRY)	Stop sw	
3A	Black (BLK)	Joy up com	1
3B	White (WHT)	Joy up	
4A	Black (BLK)	Joy down com	3
4B	Brown (BRN)	Joy down	
5A	Black (BLK)	Joy left com	4
5B	Blue (BLU)	Joy left	
6A	Black (BLK)	Joy right com	2
6B	Violet (VIO)	Joy right	
7A	Black (BLK)	Lift switch com	CEN
7B	Orange (ORG)	Lift switch up	BOT
8B	White (WHT)	Lift switch down	TOP



229727 Switch I/O harness



005715 Joystick

428635 Toggle switch kit
Raise/lower

005688 Red push-button switch
Stop

005689 Green push-button switch
Start

229729 Harness (includes speed potentiometers)

229729 Potentiometer harness pinout

J5 pin	Wire color	Signal	Destination
1A	Orange (ORG)	+3.3 V	POT1-3
2A	White (WHT)	Wiper	POT1-2
3A	Black (BLK)	Common	POT1-1
1B	Orange (ORG)	+3.3 V	POT2-3
2B	White (WHT)	Wiper	POT2-2
3B	Black (BLK)	Common	POT2-1

013395 Revision C