# Tormach® 15L Slant-PRO™

## Operator's Manual



*IMPORTANT! Read all safety precautions and instructions thoroughly before attempting 15L Slant-PRO installation, operation, or maintenance.*

# Preface

## SAVE THESE INSTRUCTIONS!

This manual contains important safety warnings and operating instructions for the 15L Slant-PRO lathe. Refer to these instructions before attempting installation, operation or maintenance. Keep these instructions together with your 15L Slant-PRO lathe, so they are readily accessible. Any updated versions of this manual are available at www.tormach.com/documents

## Safety Precautions

Failure to read and follow these safety warnings and operating instructions may result in voided warranty, property damage, serious injury or death.

| Symbol | Description | Example |
|---|---|---|
| ⚠️ | **WARNING!** *Indicates a hazard which, if not avoided, could result in death or serious injury.* | **WARNING! Ejection Hazard:** *Tools and work pieces must be clamped properly. Failure to do so could result in death and/or serious injury.* |
| ⚠️ | **CAUTION!** *Indicates a hazard which, if not avoided, could result in injury or machine damage.* | **CAUTION!** *Be sure to wear gloves when uncrating lathe. Failure to do so could result in serious injury.* |
| **IMPORTANT!** | *Addresses practices not related to personal injury.* | **IMPORTANT!** *Tighten the drawbar so the collet holds the tool holder securely in position.* |
| **NOTE:** | *Indicates additional information, clarification, or helpful hints.* | **NOTE:** *For further information on automatic oiler troubleshooting, refer to the operator manual.* |

## Machine Safety

Any machine tool is potentially dangerous. Computer-controlled machines are potentially more dangerous than manual machines because, for example, a controller is quite prepared to plunge a turning tool at 60 inches per minute into a D1-4 chuck face spinning at 2500 RPM.

The Tormach lathe can deliver sufficient force to break tools, to crush bones, and to tear flesh. If loose hair, clothing, gloves, or jewelry gets caught by a rotating workpiece the results can be disastrous.

This manual tries to give you guidance on safety precautions and techniques, but because the details of each work area or other local conditions differ, we can accept no responsibility for the performance of the machine or any damage or injury caused by its use. It is your responsibility to ensure that you understand the implications of what you are doing and to comply with any legislation and codes of practice applicable to your country or state.

## Machine Safety

Safe operation of the machine depends on its proper use and the precautions taken by the operator. Read and understand this manual prior to use. Only trained personnel – with a clear and thorough understanding of its operation and safety requirements – should operate this machine.

**General Safety:**

- Wear OSHA-approved safety glasses, safety shoes, and ear protection.
- Remove loose-fitting clothing, neckties, gloves, and jewelry.
- Tie up long hair or secure under a hat.
- Never operate a machine after consuming alcohol or taking medication.
- Keep work area well lit and deploy additional lighting, if needed.

**Operational Safety**

- Understand CNC lathes are automatically controlled and may start at any time.
- Do not leave machine unattended during operation.
- Always power off machine when not in use.
- Never operate with the door or chuck guard open; never bypass safety switches.
- Do not operate without knowing the function of every control key, button, knob or handle. Refer to the operator manual or contact Tormach if any function is not understood.
- Do not extend unsupported bar stock past the left end of the spindle bore.
- Do not operate with an unbalanced workpiece or spindle fixtures.
- Remove all tools (wrenches, chuck keys, etc.) from the spindle and machine surfaces before starting operations; loose items can become dangerous projectiles.
- Use adequate work clamping; loose workpieces can become projectiles.
- Be aware of workpiece cutoffs that could be cut free during operations and become dangerous projectiles.
- Protect your hands. Stop machine spindle and ensure lathe motion has stopped before:

    - Reaching into any part of the machine motion envelope

    - Changing tools or parts

    - Clearing away chips, oil, or coolant; always use a chip scraper or brush

    - Making an adjustment to the workpiece, fixture, or coolant nozzle

    - Taking measurements

- Keep work area clear of clutter as machine motion can occur when keys are accidentally pressed or objects fall on keyboard, resulting in unexpected motion.

- Avoid getting pinched in places where the table, saddle, or spindle head create pinch points while in motion.

- Always use proper feeds, speeds, and depths of cut to prevent tool breakage.

- Chips and dust from certain materials (e.g., magnesium) can be flammable. Fine dust from normally non-flammable materials can be flammable or even explosive.

- Chips and dust from certain materials can be toxic. Vapors from certain overheated materials can be toxic. Always check a Material Safety Data Sheet (MSDS) of suspect materials. Refuse machining work requests of unknown materials.

- If in any doubt, seek guidance from a professionally qualified expert rather than risk injury to yourself or to others.

- Use adequate safeguarding around the operating envelope.

It is the responsibility of the operator/employer to provide and ensure point of operation safeguarding per the following:

- OSHA 1910.212 - General Requirements for All Machines

- ANSI B11.5-1984 (R1994) Lathes

- ANSI B11.22-2002 Safety Requirements for Turning Centers and Automatic Numerically Controlled Turning Machines

- ANSI B11.TR3-2000 Risk Assessment and Risk Reduction - A Guideline to Estimate, Evaluate, and Reduce Risks Associated with Machine Tools.

## Electrical Safety

⚠️ *WARNING! Electrical Shock Hazard: Be sure to power off machine before making any electrical modifications. Failure to do so may result in serious injury or death.*

Never operate the machine tool with the cabinet door open. Never allow a coolant pump to operate with the cabinet door open. Do not allow the coolant system to flow coolant directly at the cabinet door seal or on the operator console controls. Neither the cabinet door seal nor the electrical controls are sealed against liquids.

**Dual Power Input:** The Tormach 15L Slant-PRO has two electrical power inputs. The primary supply is 200 to 250 VAC single phase and is used for all axis and spindle motion. This should be on a dedicated circuit of at least 20A. Either of the 15L Slant-PRO's two electrical power inputs can provide lethal electrical shocks. Both power inputs should be unplugged before working in the electrical cabinet.

**Electrical Service:** Certain service and troubleshooting operations require access to the electrical cabinet while the electrical power is on. Only qualified electrical technicians should perform such operations.

**Retained Electrical Power:** Electronic devices within the electrical cabinet may retain dangerous voltage levels after the power has been removed.

## Warranty

### Limited Warranty Coverage

Each Tormach (the "Manufacturer") 15L Slant-PRO machine ("Machine") and its components ("Components") — except those listed below under limits and exclusions — is warranted against defects in material and workmanship for a period of 12 months from the date of delivery. The foregoing is a limited warranty and it is the only warranty by the manufacturer. Manufacturer disclaims all other warranties, express or implied, including but not limited to all warranties of merchantability and fitness for a particular purpose.

### Repair or Replacement Only

Manufacturer's liability under this agreement shall be limited to repairing or replacing, at the discretion of manufacturer, parts or components. Shipment for items replaced under warranty is free, but the shipment method is at the discretion of Tormach. In general delivery is via UPS ground service for domestic customers or the U.S. Postal Service (USPS) for international customers. If overnight or express delivery is requested, additional fees will apply.

Direct sales and phone support are part of the equation that allows us to provide high value at low cost. You must be comfortable with general electrical and mechanical repair concepts, including appropriate safety procedures, before working on your machine. If you do not have the required skills you need to find someone locally to assist you. We do not have factory technicians to send to your facility.

### Limits and Exclusions of Warranty

Except as provided above, buyer agrees that all warranties express or implied, as to any matter whatsoever, including but not limited to warranties of merchantability and fitness for a particular purpose are excluded. Components subject to wear during normal use and over time such as paint, labels/decals, finish/condition, seals, flex cabling, spindle, etc., are excluded from this warranty.

Tormach-specified maintenance procedures must be adhered to in order to maintain this warranty. This warranty is void if the machine is subjected to mishandling, misuse, neglect, accident, improper installation, improper maintenance, or improper operation or application, or if the machine was improperly repaired or serviced. Warranty of general machine tolerances is void if the machine is disassembled or altered by customer. Without limiting the generality of any of the exclusions or limitations described in other paragraphs, manufacturer's warranty does not include any warranty that the machine or components will meet buyer's production specifications or other requirements or that operation of the machine and/or components is uninterrupted or error-free. Manufacturer assumes no responsibility with respect to the use of the Machine and Components by Buyer, and manufacturer shall not incur any liability to Buyer for any failure in design, production, operation, performance or otherwise of the Machine or components other than repair or replacement of same as set forth in the Limited Warranty above. Manufacturer is not responsible for any damage to parts, machines, business premises or other property of Buyer, or for any other incidental or consequential damages that may be caused by a malfunction of the machine or components.

# Preface

### Limitation of Liability and Damages

Manufacturer is not liable to Buyer, or any customer of buyer for loss of profits, lost data, lost products, loss of revenue, loss of use, cost of down time, business good will, or any other incidental or consequential damage, whether in an action in contract or tort, arising out of or related to the machine or components, other products or services provided by manufacturer or seller, or the failure of parts or products made by using the machine or components, even if manufacturer or seller has been advised of the possibility of such damages.

Manufacturer's liability for damages for any cause whatsoever shall be limited to repair or replacement, at the discretion of manufacturer, of the defective parts, components or machine. Buyer has accepted this restriction on its right to recover incidental or consequential damages as part of its bargain with Seller. Buyer realizes and acknowledges that the price of the equipment would be higher if Seller or Manufacturer were required to be responsible for incidental or consequential damages, or punitive damages. This warranty supersedes any and all other agreements, either oral or in writing, between the parties hereto with respect to the warranties, limitations of liability and/or damages regarding the Machine or Components, and contains all of the covenants and agreements between the parties with respect to such warranties, liability limitations and/or damages. Each party to this warranty acknowledges that no representations, inducements, promises, or agreements, orally or otherwise, have been made by any party, or anyone acting on behalf of any party, which are not embodied herein regarding such warranties, liability limitations and/or damages, and that any agreement, statement, or promise not contained in this warranty shall be not be valid or binding regarding such warranties, liability limitations and damages.

### Transferability

This warranty is transferable from the original end-user to another party if the machine is sold via private sale before the end of the warranty period. Should you have a problem with your machine, please consult your operator's manual first. If this does not resolve the problem, contact Tormach through our Web site at www.tormach.com or call (608) 849-8381.

### Extended Warranty

An extended warranty (PN 34381) is available for purchase at www.tormach.com or by calling (608) 849-8381.

## Intended Use

The Tormach 15L Slant-PRO is intended for use as a general purpose CNC lathe. The intended use includes cutting conventional non-abrasive materials such as mild or alloy steels, aluminum, plastics, and similar materials. Applications for the equipment or modifications of the equipment outside of the Intended Use Statement are supported through consulting engineering, not through our free support policy. There are no limits to the applications that Tormach products can be used for or to the modifications that can be applied to the Tormach machinery. Tormach designs use standard industrial components and incorporate the principles of Open Architecture specifically to allow and promote these variations. With Open Architecture controls, industrial engineers will find Tormach products cost effective to incorporate into larger manufacturing systems. All of the technical information and insight required to support these variations from the intended use cannot possibly be foreseen. If the extensive documentation provided does not supply all the information you need, we can provide additional information and engineering support required for your project on a consulting engineering basis. Consulting engineering is done by electrical and mechanical engineers and billed at current hourly rates.

As you might expect, all warranties for Tormach equipment are voided through modification to the equipment or use outside of the Intended Use. Individuals or companies involved with modifying the equipment or applying the products assume all consequent liability.

## Nomenclature

This manual uses the following typographical nomenclature

| | |
|---|---|
| *Software Control* | Refers to a Software Control e.g., an on-screen button |
| *Hardware Control* | Refers to a button or switch on the lathe's Operator Panel |
| `G-code (e.g., G01X34.8)` | Used to show G-code programs |
| *Key name* (e.g., *Enter*) | Tells you to press the indicated key |
| *Button name (e.g., Stop)* | Tells you to press the indicated button |

# Preface

## Table of Contents

# Preface

# Overview

## 1.  Overview

The 15L Slant-PRO lathe is a general purpose CNC lathe. Pictured below is a typical 15L Slant-PRO lathe set up.
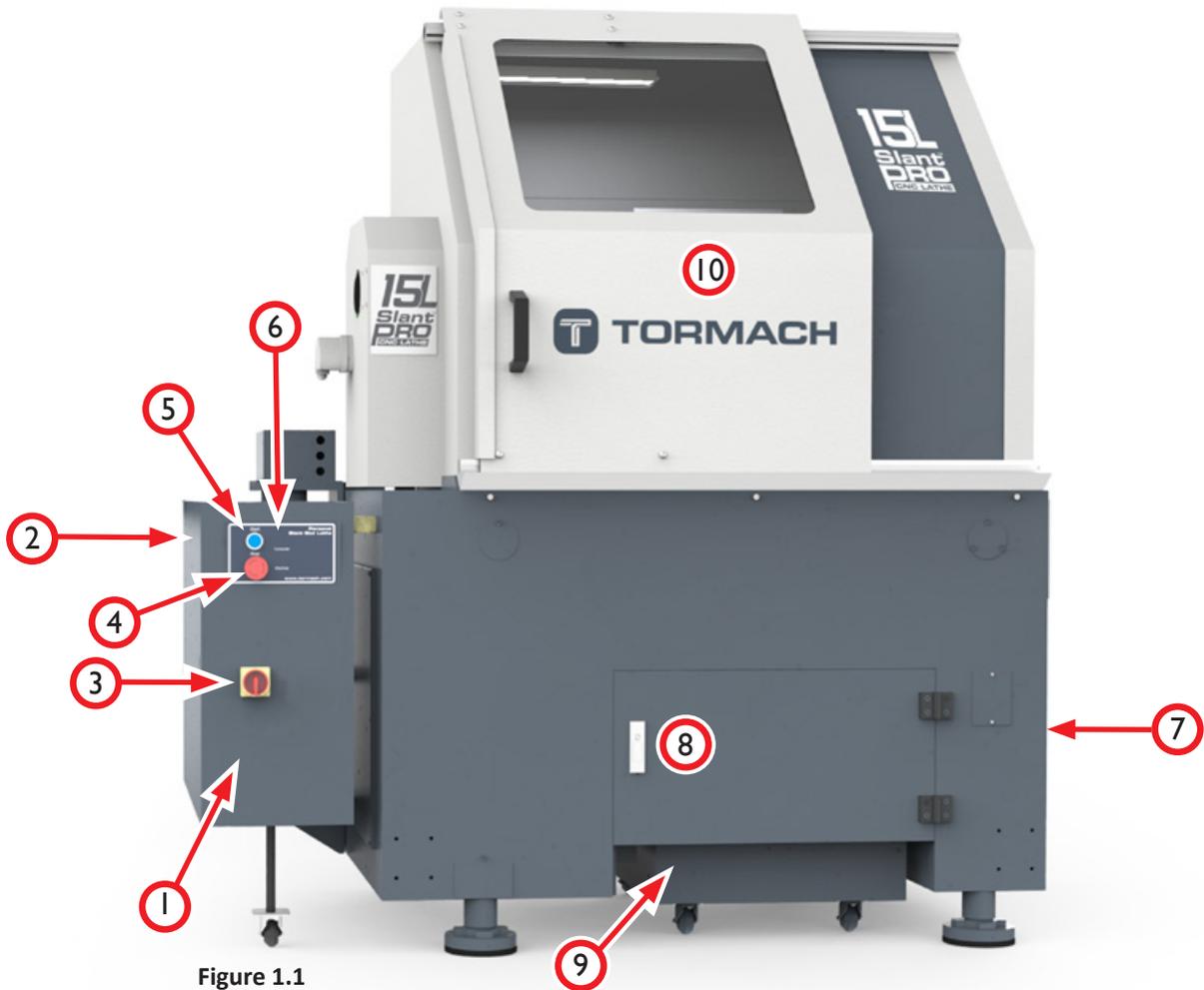


**Figure 1.1**

| Item # | Component Name | Item # | Component Name |
|--------|----------------|--------|----------------|
| 1 | Electrical Cabinet | 6 | Operator Panel |
| 2 | Serial Number Plate | 7 | Controller Cabinet |
| 3 | Main Disconnect | 8 | Coolant Tank Door |
| 4 | Emergency Stop Button | 9 | Coolant Tank |
| 5 | Start Button | 10 | Full Enclosure |

## 1.1 Specifications

| Mechanical | | |
|---|---|---|
| Travels | X-axis | 10" |
| | Z-axis | 12" |
| Feed Rates | Rapid Speeds | 60 IPM |
| Key Dimensions | Table Length | 26" |
| | Table Width | 6" |
| | Tool Holding | Gang tooling plate with three T-slots 0.625" (5/8") Center Slot: 0.00 + 0.004" – Outer Slots: 0.000 + 0.008" |
| | | Manual quick change tool post. |
| | | 8-position turret |
| | Swing Over Bed | 15" |
| | Swing Over Carriage | 6.2" (turret or manual tool post); 1.88" (gang plate) |
| | Footprint | 55" W x 66" H x 29" D |
| Spindle | Speed Range | 250 to 3500 RPM (5C), 180 to 2500 RPM (6"chuck) |
| | Horsepower | 3 hp continuous |
| | Drive System | Cartridge style spindle with double V-belt transmission |
| | Design | D1-4 spindle w/ removable 5C taper insert |
| Temperature | Operating Range | 55°F-100°F (13°C-38°C) |

| Electrical | | |
|---|---|---|
| Power Requirements | Primary (machine tool) | 200 to 250 VAC[1] single phase |
| | Secondary (control/coolant systems) | 115 VAC single phase |
| Recommended Circuit Amperage[2] | Primary | 20 AMP |
| | Secondary | 15 AMP (GFI protected) |

[1] 230 VAC recommended; For voltages under 230 VAC, use of a Buck-Boost Transformer (PN 32554) is recommended.

[2] Dedicated circuits recommended; do not use ground-fault interrupter (GFI) with primary circuit.

# Installation

## 2.  Installation

### 2.1  Lathe Prep and Setup

#### 2.1.1  Location

The area should be well lit, dry, have proper ventilation, provide for unobstructed machine motion and operation, and ensure unrestricted access to all lathe controls and the electrical cabinet.
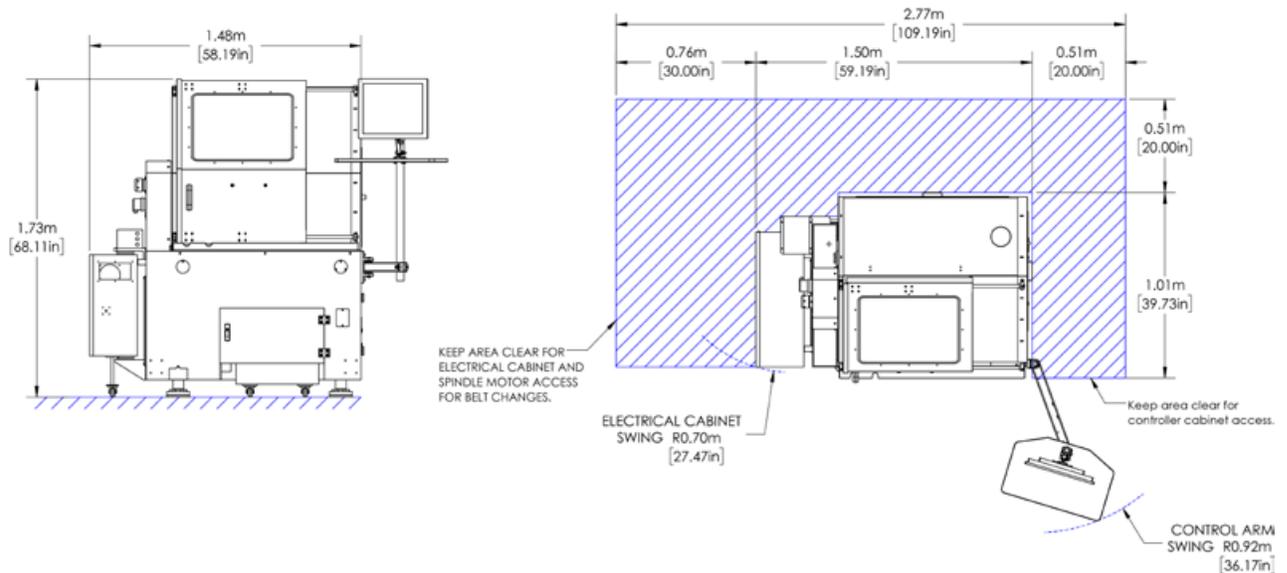


#### 2.1.2  Electrical Requirements

⚠️  **WARNING! Electrical Shock Hazard:** *Be sure to power off machine before making any electrical modifications. Failure to do so could result in serious injury or death.*

- **Primary: 200-250 VAC, 50/60 Hz at 20 Amps**

    **NOTE:** *Continuous current is below 15 Amps, but requires a 20 Amp breaker or slow blow fuse to prevent nuisance trips of the circuit breaker on spindle start.*

    Either use a NEMA L6-20 plug, or hard wire the machine. A certified electrician must make all electrical connections, and it's your responsibility to verify that the electrical installation of the machine meets all local regulations and electrical codes.

- **Secondary: Separate wall outlet of 115 VAC, 50/60 Hz at 15 Amps**

    A separate wall outlet of 115 VAC at 15 Amps is required to act as power source for the controller, monitor, and coolant pump. This connection is made to the IEC power inlet at the rear of the electrical cabinet.

For non-conforming sites:

1. Replace the stock coolant pump with a 230 VAC compatible pump (Hi-Quality 1/8 HP Coolant Pump Upgrade Kit, PN 33216), and change the Voltage Setting Switch on the controller to 230 VAC (see Figure 2.8).

2. Use a step down transformer to convert 230 VAC to 110 VAC (Step-Up/Step-Down Transformer, PN 32009).

### 2.1.2.1   Electrical Noise

Both primary and secondary power should be provided by dedicated circuits. At the minimum, circuits should be isolated from electrically-noisy devices. In particular, high-inductive loads from units like vacuum cleaners or air compressors can be troublesome and/or cause controller malfunction. For non-conforming sites, refer to electrical requirement table earlier in this section.

### 2.1.2.2   Grounding

All power inputs to the lathe must be properly grounded. Check continuity between bare metal on machine frame and true earth ground (water pipe or similar) to ensure proper grounding.

### 2.1.2.3   Ground Fault Interrupter (GFI) Use

Primary power should not be protected by a ground fault interrupter (GFI) or residual current circuit breaker (RCCB), as this interferes with proper operation of the lathe's variable frequency drive (VFD) spindle controller.

A GFI or RCCB must be used to supply power to the secondary 115 VAC power supply. The controller, monitor, and coolant system are not bolted to the machine frame so proper grounding cannot be assumed. The combination of electrical power and water-based coolant systems makes GFI protection for the 115 VAC circuit very important.

## 2.2   Receiving, Uncrating, and Initial Inspection

### 2.2.1   Uncrating

⚠ **WARNING! Transport and Lift Hazard:** *The transport, lifting, and moving of lathe should be done by trained professionals. Failure to do so could result in machine damage, serious injury, or death.*

1. The 15L Slant-PRO is shipped on a standard pallet and can be off-loaded from a truck with a tailgate lift and moved  to the desired install location with a hydraulic pallet jack.

⚠ **CAUTION! Sharp Objects:** *Wear gloves when uncrating the machine. Failure to do so could result in serious injury.*

2. Use strap snips and a small pry bar to open and disassemble shipping crate. Remove top of crate first, followed by four sides.

3. Remove and discard the plastic packaging from the machine.

### 2.2.2 Shipping Damage or Shortages

Once received, inspect and note any shipping damage that may have occurred during transit. Also check received goods against packing list. Any damage claims or shortages must be addressed within 30 days of receipt.

### 2.2.3 Remove Machine from Pallet

⚠️ **WARNING! Transport and Lift Hazard:** *The transport, lifting, and moving of the machine should be done by trained professionals. Failure to do so could result in machine damage, serious injury, or death.*

#### 2.2.3.1 Lathe Lift Kit

The preferred method of lifting the lathe is using the Lathe Lift Kit (PN 33269) to raise the machine with four temporary legs and jack screws.

*NOTE: Before separating the lathe from the pallet, ensure there is enough space on the sides of the lathe to slide the pallet out from underneath it. If walls or immoveable objects are in the way, a different location may be required.*

1. Cut a strip of wood down the entire length of the pallet, on both the front and the back side, to provide clearance for the lifting feet. A skill saw works well for this (see **Figure 2.1**).
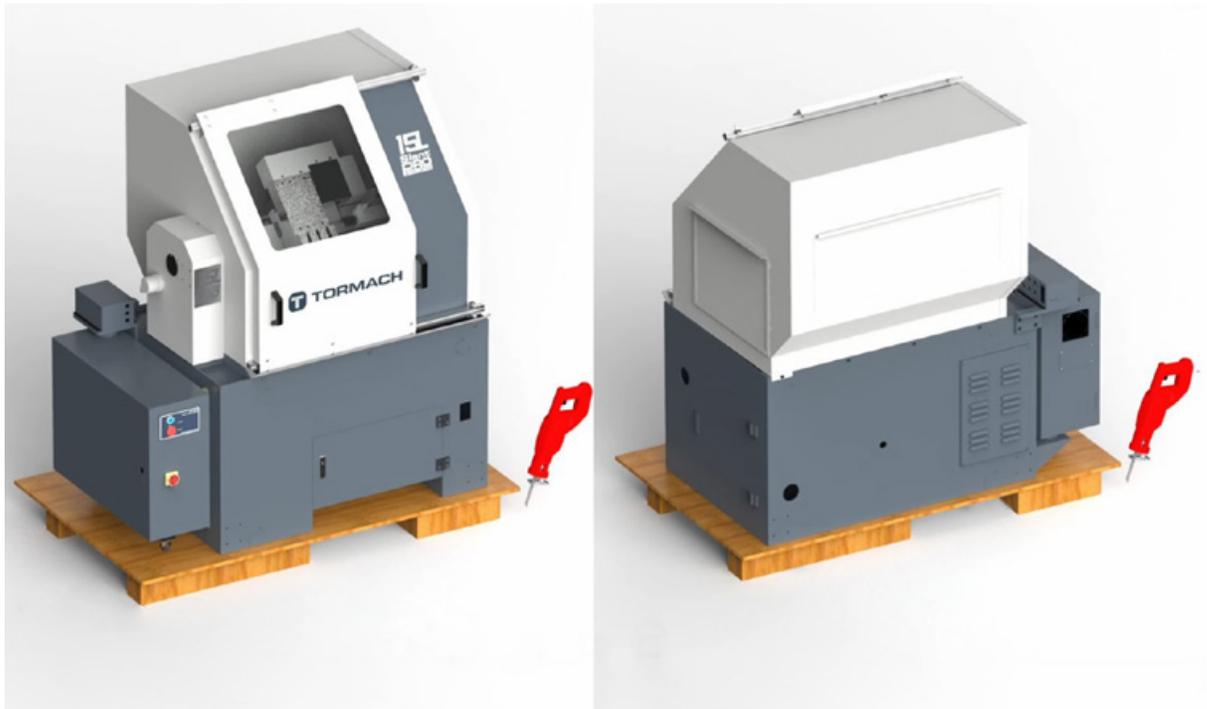


**Figure 2.1**

2. Tape the electrical cabinet door shut to prevent it from swinging open during the lift procedure.

3. Open the coolant tank door on the front of the lathe and temporarily slide it off of the shipping pallet.

*NOTE: Do not remove the pallet out before removing the Coolant Tank. Failure to do so could result in machine damage.*

4. Cut the shipping straps the secure the emergency stop box and power cable to the pallet.

5. Four pallet bolts are used to temporarily attach the pallet to the lathe; identify two right-hand bolts located inside the controller cabinet (see **Figure 2.3**).

6. Loosen and remove nuts and washers from two bolts inside the controller cabinet.

7. Identify two small Access Panels on left side of lathe (see **Figure 2.2**).

8. Using a Phillips screwdriver, remove Access Panels; set aside.

9. Loosen and remove nuts and washers from two bolts inside the Access Panels.

10. Using four M8 x 20 mm Socket Head Cap Screws and four M8 Flat Washers, attach one Lifting Foot to each of the four corner attachment sites (see **Figure 2.4**); securely tighten.



**Figure 2.2**



**Figure 2.3**



**Figure 2.4**

11. Slide a Lifting Pad underneath each of the four installed Lifting Feet; align hole on Lifting Pad with Lifting Foot's bolt hole (see **Figure 2.5**).

12. Attach one M20 x 250 mm Threaded Rod through each Lifting Foot and into its Lifting Pad (see **Figure 2.6**).

13. Using a socket wrench, gradually adjust each Lifting Foot to raise the lathe evenly until lathe is clear of the floor (see **Figure 2.7**).

14. Slide the pallet from underneath the lathe.

15. Depending on the accessories that you're going to install on your lathe, do one of the following:

    a. **Installing a Turret:** Keep the lifting feet on the lathe for the turret installation procedure. When finished installing the turret, go to *Attach Lathe Feet*.

    b. **No Turret:** Go to *Basic Installation Procedure* and follow the instructions to attach the lathe feet onto the machine.



**Figure 2.5**



**Figure 2.6**
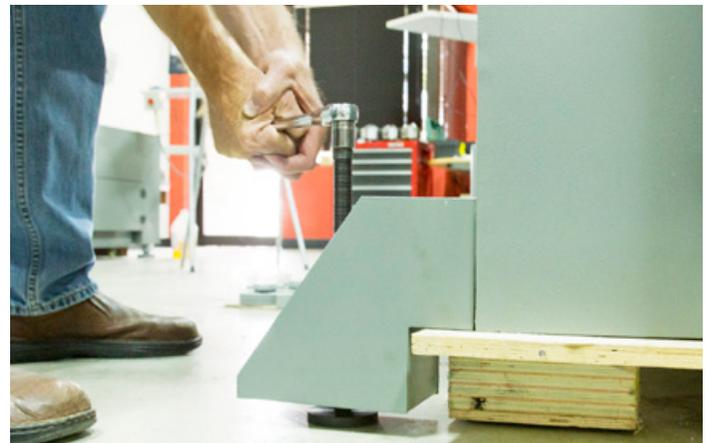


**Figure 2.7**
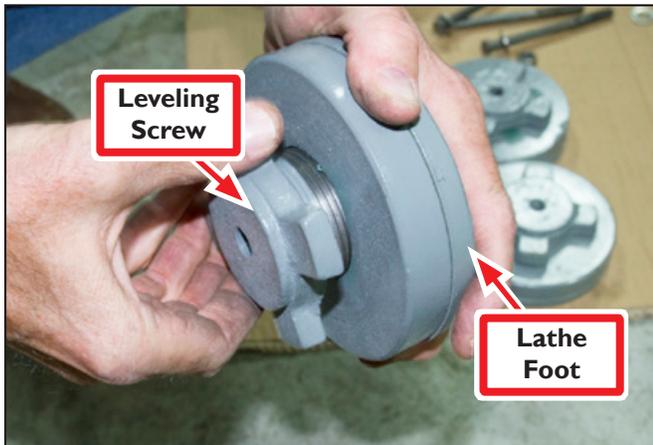
**Figure 2.8**
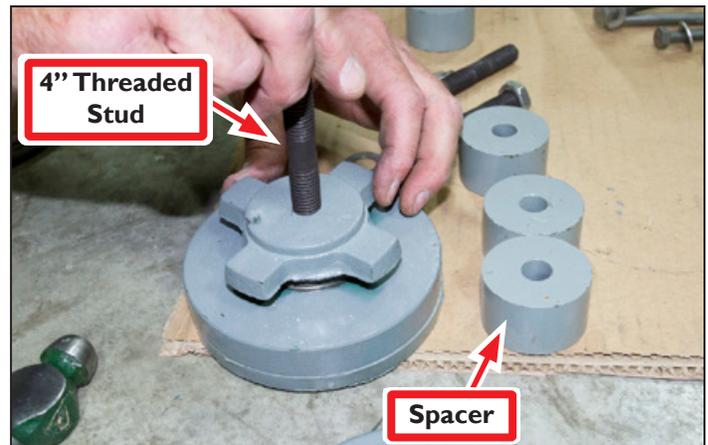


**Figure 2.9**

## 2.3  Basic Installation Procedure

This section covers basic installation of a 15L Slant-PRO lathe.

### 2.3.1  Attach Lathe Feet

1. Unscrew Leveling Screw on each Lathe Foot until close to full height (see **Figure 2.8**); apply a dab of grease on threads.

2. Screw one 4" Threaded Stud into each Lathe Foot (see **Figure 2.9**); slide a Spacer onto each stud.

3. Insert each lathe foot assembly into base of lathe via a lag bolt hole (see **Figure 2.10**). Slip a Washer onto each Threaded Stud and secure feet tightly with nuts.

*NOTE: Lifting Feet are not intended to support the lathe during operation.*

4. Reattach two Access Panels set aside earlier (see **Figure 2.2**).

5. When the lathe is completely supported by four lathe feet, lower four M20 x 250 mm Threaded Rods in stages until the lathe rests on its feet; remove rods.

6. Unscrew and remove four M8 x 20 Socket Head Cap Screws and M8 Washers securing each Lifting Foot to the lathe; remove four Lifting Feet from lathe.
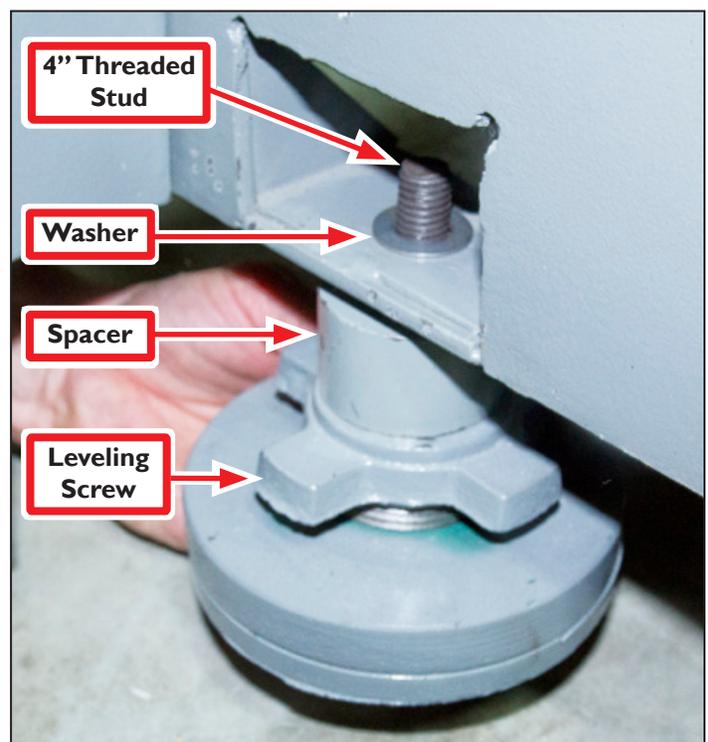


**Figure 2.10**

### 2.3.2 Install the Controller Arm

The Controller Arm mounts to the right-hand side of the lathe stand. One of the two red Emergency Stop buttons mounts to the underside of the keyboard tray and should protrude past the edge of the tray far enough to be easily accessible.

*IMPORTANT! If you set up the controller in a different configuration, you must make sure that the lathe's remote Emergency Stop button is installed so that it is easily accessible to you.*

To install the Controller Arm:

1. Identify the correct mounting position for the controller arm on the machine stand.

2. Make sure that there are no obstructions inside of the machine stand.

3. Use two sets of M8 x 30 mm socket head cap screws, M8 hex nuts, M8 flat washers, and M8 lock washers to attach the mounting clevis to the machine stand (see **Figure 2.11**). Make sure that the cable slots on the square tube arm are facing as shown in **Figure 2.11**.
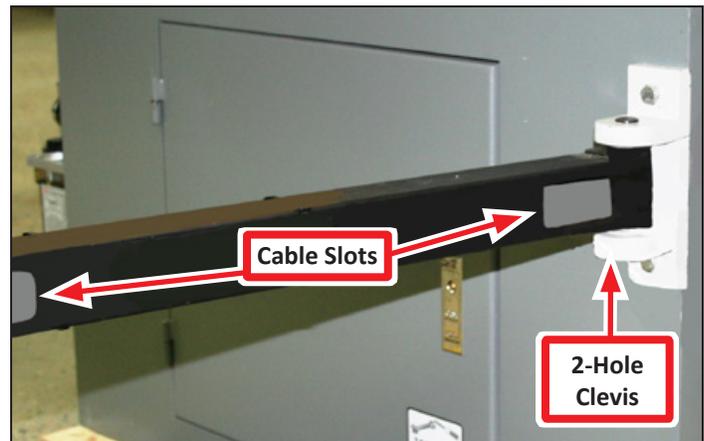


**Figure 2.11**

4. Insert the round monitor post into the square tube arm and tighten the cross bolt (see **Figure 2.12**). Make sure that the holes are along the inside of the round monitor post as shown in **Figure 2.12**.

*NOTE: You can adjust the working height of the post the keyboard table and the monitor are attached.*

5. Use a dead-blow hammer (or similar) to tap the end plug into the square tube arm (see **Figure 2.13**).



**Figure 2.12**

**Figure 2.13**

Figure 2.14
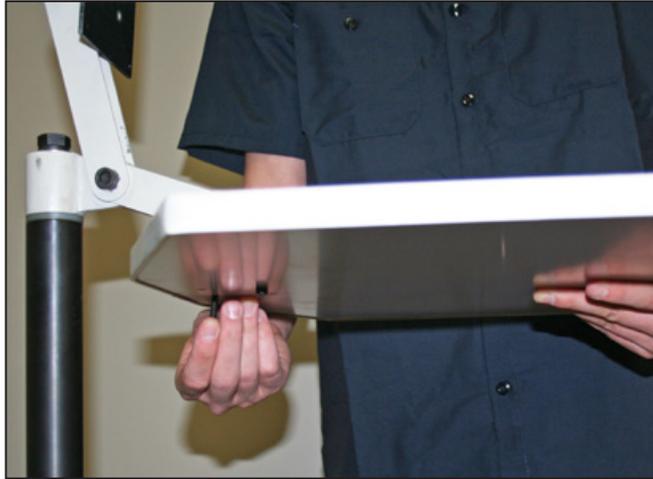


Figure 2.15



Figure 2.16

6. Position the round monitor post to mount the keyboard table as shown in **Figure 2.14**. Tighten the applicable screws to maintain the desired position.

7. Use four M5 x 25 mm socket head screws, four M5 flat washers, and four M5 lock washers to attach the keyboard table to the round monitor post (see **Figure 2.15**).

8. Use four M4 x 12 mm wood screws to attach the Emergency Stop box below the keyboard table (see **Figure 2.16**).

9. Use four M4 x 8 mm flat head screws to attach four wire tie mounts to the round monitor post (see **Figure 2.17**).

10. Use two M4 x 8 mm flat head screws to attach two Emergency Stop cable clips to the bottom of the square tube arm.

11. Use four M4 x 10 mm Phillips round head screws to attach the monitor, and then route the cables through the controller arm.

**NOTE:** *We recommend wrapping all cable connections in electrical tape.*

12. Adjust the monitor height and keyboard angle as needed.
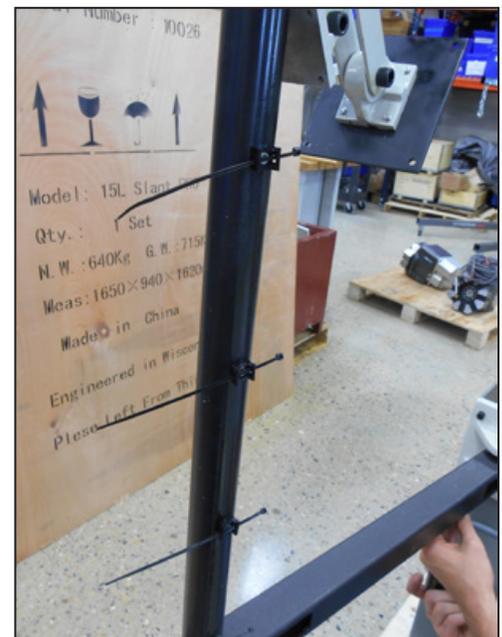


Figure 2.17

# Installation

### 2.3.3 Install the Monitor

The PathPilot controller mount allows you to install the PathPilot controller behind the monitor (which is attached to the Controller Arm).

*Note: If you're using a Touch Screen Kit (PN 35575), you must first remove the stock mounting bracket from the back of the monitor.*



**Figure 2.18**

1. Put the PathPilot controller mount against the monitor mounting plate. Then, put the monitor on the other side of the PathPilot controller mount, and align the holes on the three components.

2. Attach the monitor, PathPilot controller mount, and monitor mounting plate together with four M4 × 12 mm socket head cap screws (provided with the PathPilot controller mount).

3. Adjust the position of the monitor and the keyboard tray with an 8 mm hex wrench and a 16 mm wrench. Once complete, securely tighten the pivot screws.
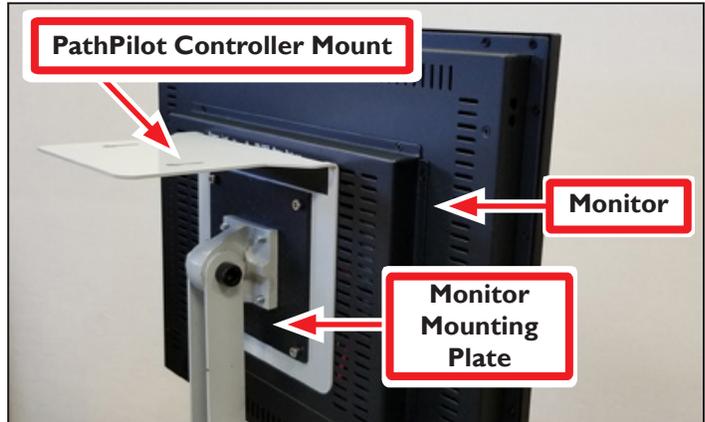
### 2.3.4 Install the PathPilot Controller

The PathPilot controller attaches to the top of the PathPilot controller mount and behind the monitor.

1. Put four standoffs into the controller and tighten them by hand (see **Figure 2.19**).

2. Find the four M4 screws and the VESA plate included with the controller. Then, mount the VESA plate to the PathPilot Controller VESA Mount (PN 50382). Make sure to put it flat side down with the keyholes toward the monitor (see **Figure 2.20**).



**Figure 2.19**



**Figure 2.20**

3. Attach the controller to its mount by sliding the standoffs through the key slots (see **Figure 2.21**).

4. Find the PathPilot Controller for 15L Lathe/PCNC Mills (PN 50848) provided. Then, remove the film from the double-sided tape on the case.

5. Put the case below the PathPilot Controller VESA Mount. Press down firmly to attach it in place.

6. Connect all USB accessories to the controller (or USB hub, if you're using one):



**Figure 2.21**

   • Jog Shuttle (PN 30616)

   • Keyboard

   • Mouse

   • Monitor

7. Connect the monitor's video cable to the PathPilot controller.

8. Route the loose end of the video cable toward the monitor, and then connect it to the monitor.

9. Connect the power supply to the PathPilot controller.

10. Route the loose end of the power cords (from the USB hub and both PathPilot controller units) to a power strip, and then connect them.

11. Connect the DB-25 interface cable into the DB-25 outlet on the side of the electrical cabinet.

12. Route the loose end of the DB-25 interface cable to the PathPilot Controller for 15L Lathe/PCNC Mills case, and then connect it to the mill interface port.

13. Find the Ethernet cable and connect it to the Ethernet outlet on the PathPilot controller for 15L Lathe/PCNC Mills case.

14. Route the loose end of the Ethernet cable toward the PathPilot controller, and then connect it to the PathPilot controller.

15. Secure the wire loom, Ethernet cable, and power supply cables to the Controller Arm with six wire tie mounts and six cable ties.

# Installation

## 2.4 Installation of Add-ons

### 2.4.1 Oiler

There are two oilers available for the 15L Slant-PRO lathe: the Automatic Oiler (PN 31374) and the Manual Oiler (PN 31302).

For information on installation and use of the Automatic Oiler (PN 31374), go to tormach.com/docs.

#### 2.4.1.1 Manual Oiler

1. Remove four bracket screws shown in **Figure 2.22**.

2. Reverse plunger as shown in **Figure 2.23** and reinstall screws.

3. Using two M6 x 15 mm socket head cap screws, mount oiler to pre-drilled holes on side of lathe (see **Figure 2.24**).



**Figure 2.23**



**Figure 2.24**

4. Identify the access panel on the right side of the machine's enclosure. Remove the 16 screws securing the panel to the enclosure, and set the screws and the panel aside.

5. Find the oil line inside the machine's enclosure.
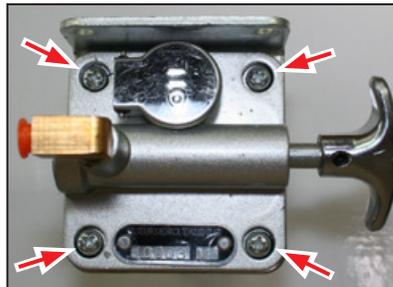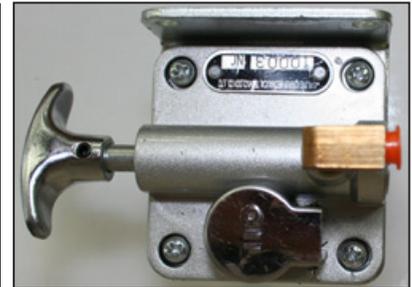


**Figure 2.22**

6.  Route the oil line to the manual oiler:

    a.  Following the X-axis motor conduit toward the lathe bed (see **Figure 2.25**)

    b.  Through the access hole in the lathe bed casting toward the spindle (see **Figure 2.26**)

    c.  Through the opposite access hole in the lathe bed casting toward the back of the lathe (see **Figure 2.27**)

    d.  Toward the manual oiler installed on the stand along the back of the lathe (see **Figure 2.28**)

7.  Connect oil line to manual oiler.

8.  Using four M5 x 10 mm screws, mount Tubing Clips to pre-drilled holes and insert oil line; discard two surplus M5 x 10 mm screws.



**Figure 2.25**

9.  Fill manual oiler reservoir with ISO VG68 Machine Oil (PN 31386) or similar.

    Retract and release plunger until oil is pushed through system. After that, pull plunger:

    •   Each time lathe is powered on

    •   Every four hours of operation

10. Reinstall the access panel to the right side of the machine's enclosure using the 16 screws that you set aside earlier.
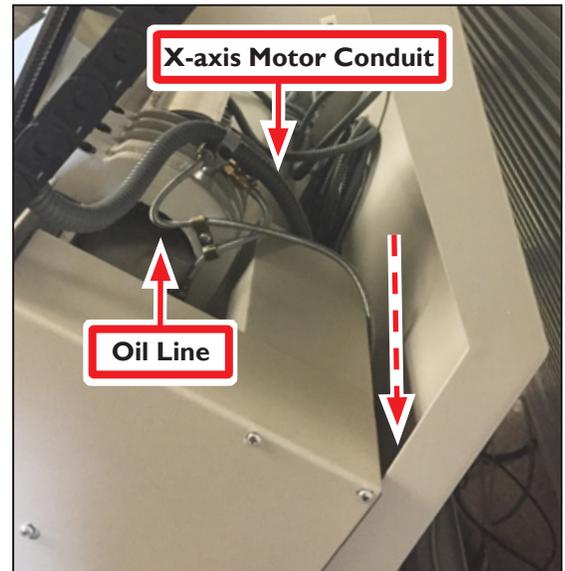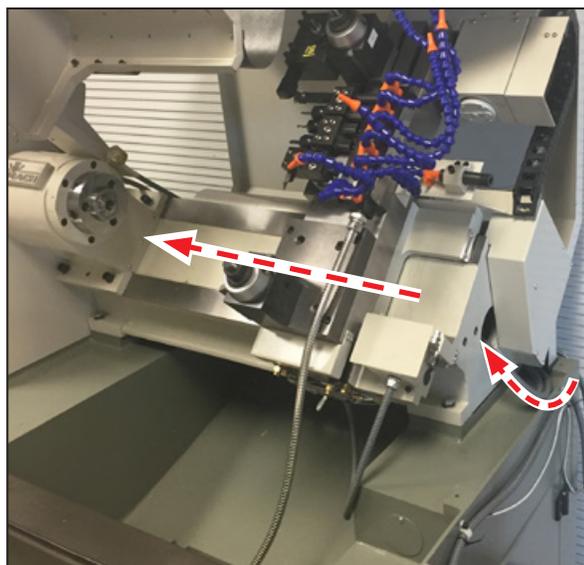


**Figure 2.26**
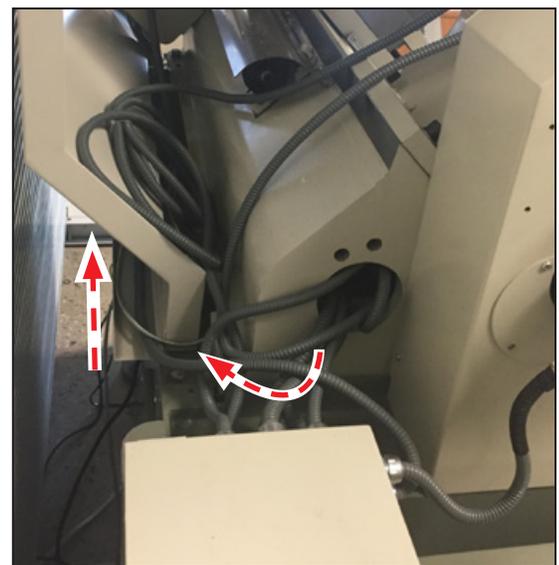


**Figure 2.27**

### 2.4.2 Turret

For information on installation and use of the Turret (PN 33273), go to [tormach.com/docs](tormach.com/docs).

## 2.5 Controller Customization

### Date and Time

To set or edit controller's date and time, type *ADMIN DATE* in the *MDI* line and click *Enter* on keyboard (see **Figure 2.28**). This opens a dialog box to enter or edit date, time, and time zone. Click *Close* when finished.

### Keyboard Language

If you do not have a USA keyboard (QWERTY) layout, type *ADMIN KEYBOARD* in the *MDI* line and click *Enter* on your keyboard (see **Figure 2.29**).

Next, click on the *Layouts* tab to change the layout of the keyboard to a different language (see **Figure 2.30**). Select a layout and click *Close* when finished.



**Figure 2.28**

### Touch Screen (optional)

For information on setup and calibration of the 17" Touch Screen Kit (PN 35575), go to [tormach.com/docs](tormach.com/docs).
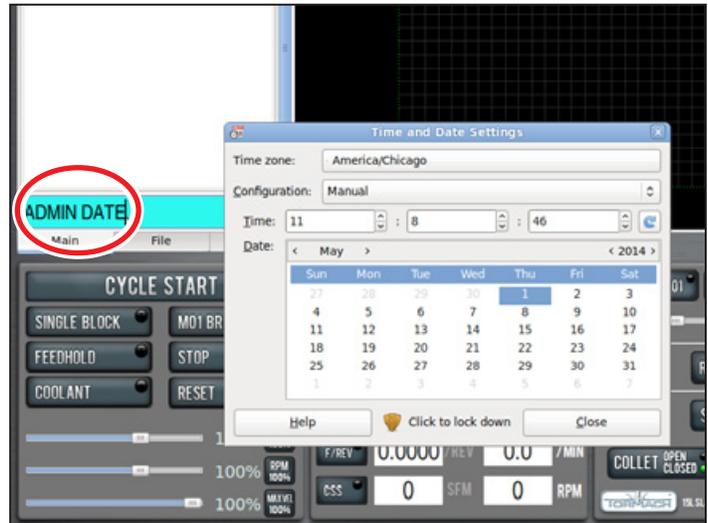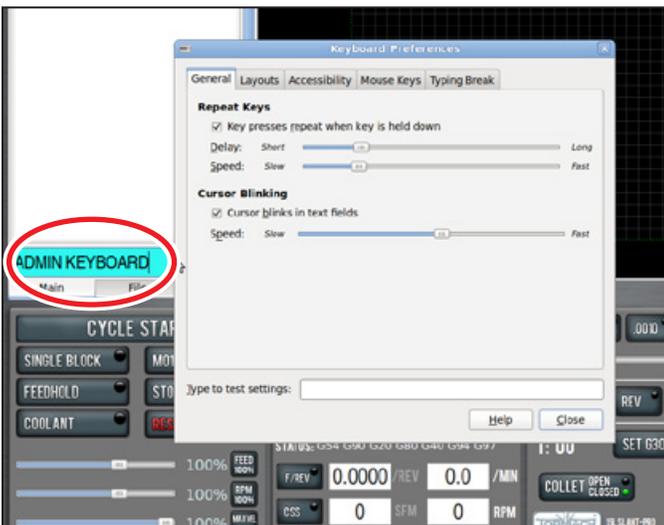


**Figure 2.29**
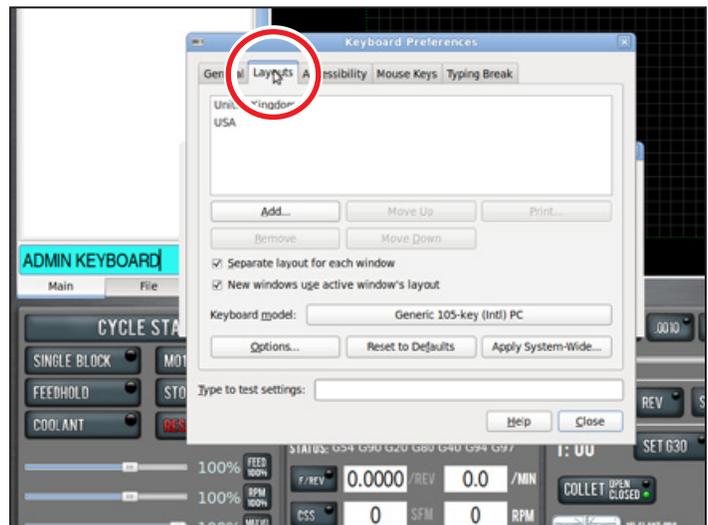


**Figure 2.30**

# 3. Operation

This chapter provides an overview of the basic operation of the lathe.

⚠ **WARNING! Unattended Operation:** *Machine is not designed to operate unattended. Do not leave machine unattended during operation. When machine is not in use, turn the main disconnect off. Failure to do so could result in death, serious injury, and/or machine damage.*

## 3.1 Power Off/Power On Procedure

**IMPORTANT!** *Do not power on motors and drives via the green Start button before powering on the controller that oversees their operation. Ensure the controller is on and the PathPilot® interface is loaded before powering on the lathe. Likewise, make sure to depress the red E-stop before powering off the controller using the Exit button in the PathPilot interface.*

**Power Off/On Procedure**

| | | |
|---|---|---|
| **Power Off** | 1. Push red E-stop in | |
| | 2. Click *Exit* on screen; when prompted click *OK* to power off | |
| | 3. Turn Main Disconnect *Off* (see image at right) | |

| | | |
|---|---|---|
| **Power On** | 1. Turn Main Disconnect *On* (see image at right) | |
| | 2. After software loads, turn red E-stop clockwise to release | |
| | 3. Press green Start button | |
| | 4. Click *Reset* on screen | |

## 3.2 Confirm Spindle Direction

Upon initial lathe setup, confirm the lathe's spindle is rotating in the correct direction.

⚠ **WARNING! Entanglement Hazard:** *Keep hands, hair, jewelry, and clothing away from moving parts. Failure to do so could result in serious injury or death.*

1. Power on the lathe and controller according to the *Power Off/On Procedure* detailed earlier in this chapter; close the chuck guard.

# Operation
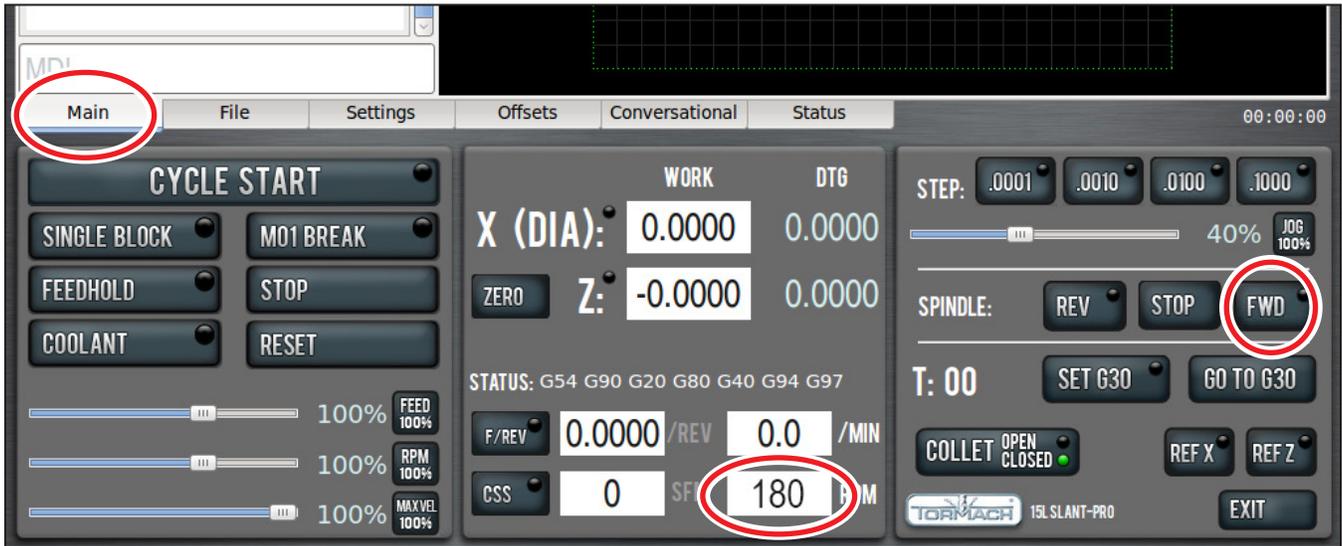


**Figure 3.1**

2.  On the *Main* screen in the PathPilot interface, set chuck speed by entering 180 in the RPM field (see **Figure 3.1**). Then, press the **Enter** key on the keyboard.

3.  Click Spindle *FWD* to start spindle rotation (see **Figure 3.1**).

4.  Observe spindle direction to confirm forward is correct (see **Figure 3.2**); repeat to confirm reverse is correct by clicking Spindle *REV* and observing spindle direction (see **Figure 3.3**). If not rotating as shown, refer to *Spindle Direction* section in chapter 8, *Troubleshooting*.

## Correct Forward Spindle Direction
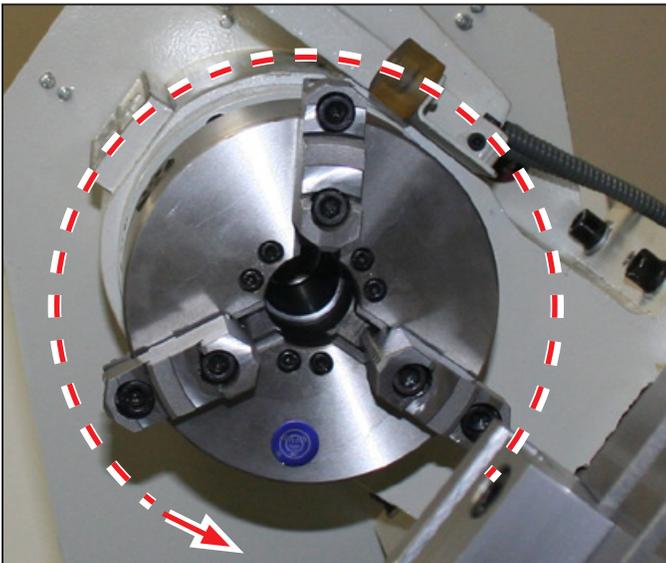


**Figure 3.2**

## Correct Reverse Spindle Direction
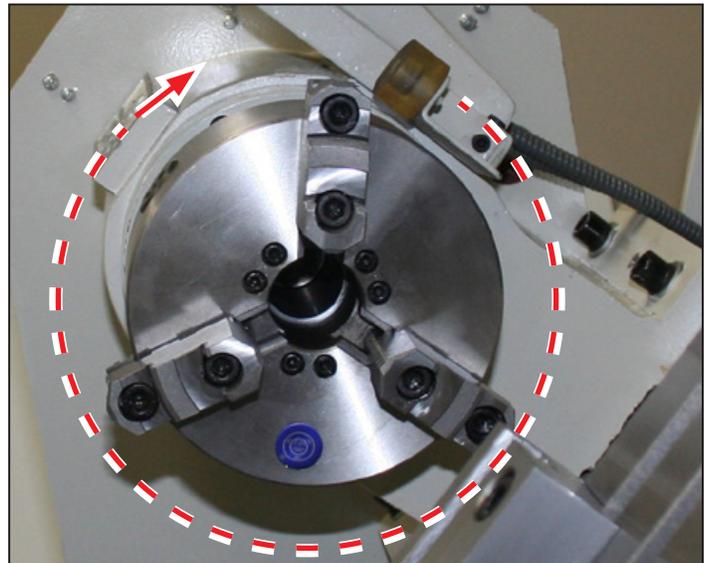


**Figure 3.3**

## 3.3 Lathe Axes Definition

The lathe has two axes of motion, denoted as the X-axis and the Z-axis. The Z-axis is parallel to rotational axis of the spindle. The X-axis is perpendicular to the Z-axis and nominally parallel to the long dimension of the carriage (see **Figure 3.4**). The −X direction is toward the front and +X direction is toward the rear. The −Z direction is toward the headstock and the +Z direction is opposite the headstock.
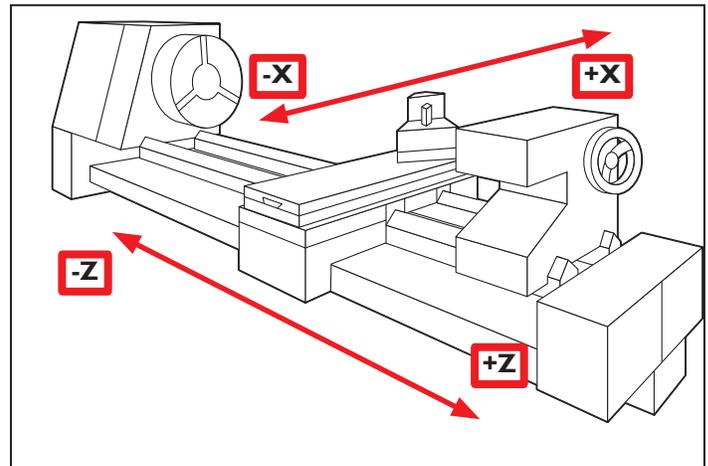


**Figure 3.4**

## 3.4 Workholding

⚠ **WARNING! Unsupported Workpiece Hazard:** *Unsupported bar stock can bend and whip with deadly force. Properly support any bar stock extended out back of spindle bore. Failure to do so could result in serious injury or death.*

The 15L Slant-PRO comes with a D1-4 spindle that includes a 5C taper insert for precision turning of stock less than 1.125" in diameter. Because this spindle has both D1-4 mounting and a 5C insert, you may configure lathe for use with either 6" D1-4 chuck or a 5C collet closer. A different motor pulley is included for use with the 5C configuration that allows you to run the lathe with a higher maximum spindle RPM when using 5C. For example, 3500 RPM for the 5C configuration and 2500 RPM with the 6" chuck configuration.

### 3.4.1 5C Collet Configuration (Maximum Speed Rating: 3500 RPM)

An optional lever-action 5C Collet Closer (PN 33283) must be purchased to use a 5C collet with a lathe. This lever-style closer can be refitted with an air cylinder for automatic control of closer operation via M-code. For more information and setup and adjustment procedures, refer to documentation that ships with the product.

### 3.4.2 D1-4 Chuck Configuration (Maximum Speed Rating: 2500 RPM)

For larger work pieces, 3-jaw 6" Chucks (PN 33156) and 4-jaw 6" Chucks (PN 33336) are available that fit the D1-4 cam lock spindle nose. Because larger work pieces require lower spindle RPMs and higher torque, the 6" chuck accessories necessitate the use of a smaller motor pulley. This pulley must be installed along with the chuck. This reduces the maximum spindle speed from 3500 RPMs to 2500 RPMs, but greatly increases the low-speed torque available for turning at lower RPMs.

# Operation

## 3.5   Tooling

The three tooling styles available for the Tormach 15L Slant-PRO are detailed below. All three options may be mixed – for example, a gang plate mounted tool could be used in conjunction with tooling mounted in a turret (for more information, refer to chapter 4, *Intro to PathPilot*).

### 3.5.1   Quick Change Tool Post (QCTP)

A manual Quick Change Tool Post Kit (PN 33272) can be mounted to the carriage to allow fast, repeatable manual tool changes (see **Figure 3.5**). The QCTP uses a dovetail and tapered gib to hold tool holders to the post at adjustable, repeatable distances from the lathe bed so that individual tools can be brought up to centerline height of the spindle. Available QCTP tool holders include standard 3/4" OD, ID (boring), and parting tool holders.

### 3.5.2   8-position Turret

The 8-position Turret (PN 33273) allows automatic tool changing of both turning and boring tools as well as through-turret flood coolant. When equipped with a turret (see **Figure 3.6**), the lathe automatically changes tools, numbers 1-8. Gang tooling or quick change tooling may be used in conjunction with a turret on lower portion of carriage.
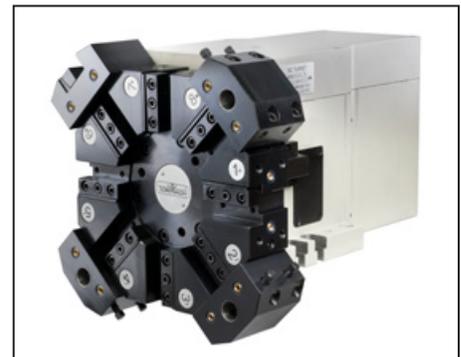
### 3.5.3   Gang Tooling

A gang tooling plate with three T-slots and a riser block can be mounted to the carriage to facilitate mounting tool holders in industry-standard, gang-tool holding blocks (see **Figure 3.7**). Risers are available for both 1/2" and 3/4" centerline heights. This option offers higher precision and faster tool change times than a manual tool post or turret, but requires vigilance to ensure appropriate clearance between tool, workpiece, and chuck/spindle nose. Gang setups are ideal for small diameter, short parts. Additionally, gang blocks can be swapped out as a single unit, dramatically reducing part setup time.
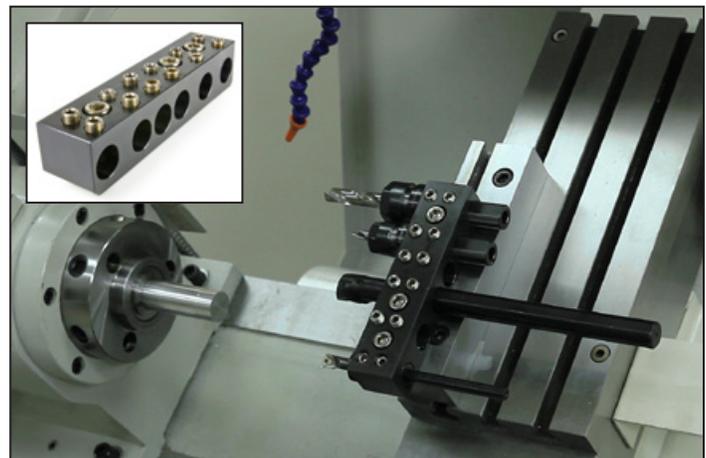

**Figure 3.5**


**Figure 3.6**


**Figure 3.7**

**Right-handed Tool**
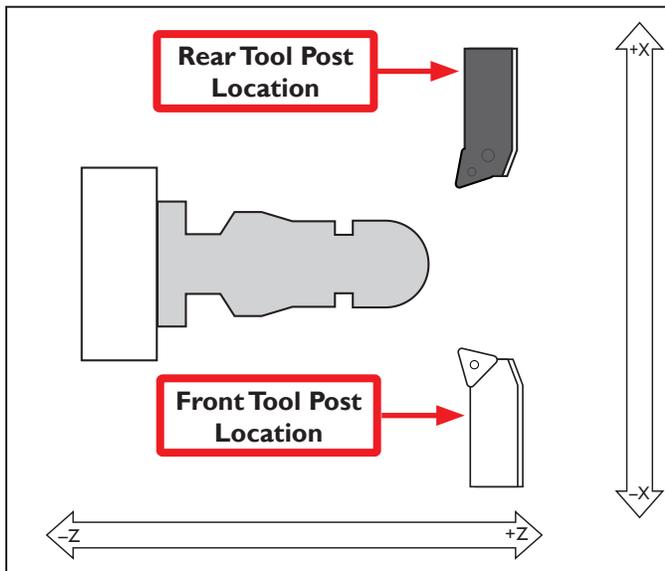
**Left-handed Tool**



Figure 3.8
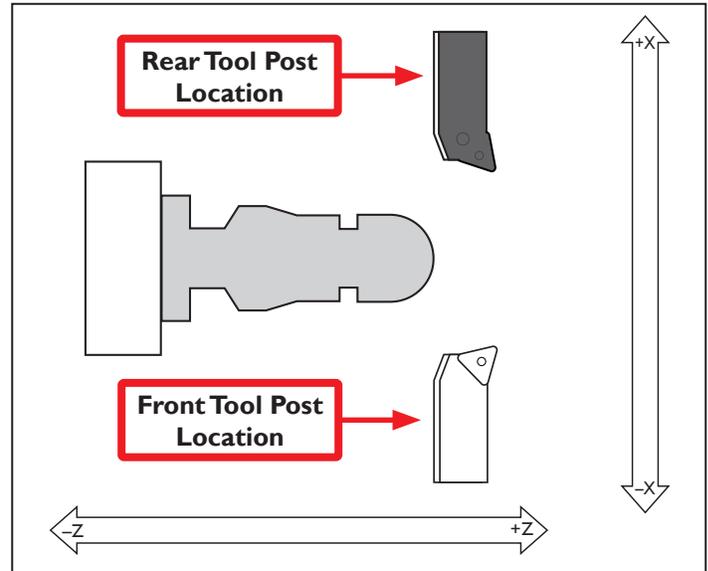


Figure 3.9

## 3.6   Front Tool Post and Rear Tool Post Locations

Tools are mounted on the carriage in either a front tool post or rear tool post position. Front tool post tools address the workpiece by moving in +X direction; rear tool post tools address the workpiece by moving in –X direction (see **Figure 3.8** and **Figure 3.9**).

The three tooling configurations for the 15L Slant-PRO have different tool post locations, detailed in the following table:

| Tooling Configuration | Tool Post Locations |
|---|---|
| Quick change tool post | Front tool post |
| Gang tooling | Front or rear tool post |
| Turret | Rear tool post |

## 3.7   Right-handed and Left-handed Tools

Tools have either a right-handed or left-handed orientation; right- or left-handed tools can be mounted on the carriage in either the front tool post or rear tool post position (see **Figure 3.8** and **Figure 3.9**). The tool can be mounted either with the cutter facing toward the operator or turned with the cutter facing away from the operator, as shown in **Figure 3.8** and **Figure 3.9**. Four cutting configurations are available for both a right- and left-handed tool (see **Figure 3.8** and **Figure 3.9**).

Right- and left-handed tools cut in different directions depending on the spindle rotation direction, detailed in the following table:

| Tool | Spindle Direction | Cutting Direction |
|---|---|---|
| Right-handed | Forward | –Z |
| | Reverse | +Z |
| Left-handed | Forward | +Z |
| | Reverse | –Z |

Right-handed tools are commonly used for general purpose turning. Left-handed tools are sometimes used for back chamfering or grooving.

# 4. Intro to PathPilot

## 4.1 Making Your First Part

This chapter shows you how to make your first part with the 15L Slant-PRO and assumes you have no prior experience running a part program on a CNC (computer numerically controlled) machine. If you have prior CNC experience, following this tutorial offers an introduction to the controls of the machine. After reading through this chapter, read chapters 5 and 6 for details on the PathPilot® operating system. This chapter is only intended to be an introduction to the PathPilot interface and several basic tasks. The first part program uses two tools (a right-hand turning tool and a parting tool) to turn a 1" aluminum work piece down to a 0.750" diameter bushing blank with flange (see **Figure 4.1**). For machine operators with the



**Figure 4.1**

optional 8-position turret, this bushing can be used in the ID tool holders to hold drill bits. The following instructions will give you a quick start introduction to the lathe controls.

### 4.1.1 Reference the Lathe

Follow the power on procedure in chapter 3, *Operation*, to turn the controller and lathe on. After clicking the flashing *Reset* button, you can reference the X- and Z-axes by clicking the Ref X and Ref Z buttons. You may reference the axes in any order, but it is common to reference the Z-axis first to move the tooling as far as possible from any workpiece or chuck to avoid a crash. Both axes must be referenced before



Right-hand Turning Tool (PN 33131) shown in QCTP holder (PN 33123)

Parting Tool (PN 33134) shown in QCTP holder (PN 33123)

**Figure 4.2**

operating the lathe to establish soft limits, protecting the lathe from over travel and to giving meaning to work offset values. After referencing the axes you will notice that the LEDs on the *Ref X* and *Ref Z* buttons turn green, indicating that the machine tool has been referenced. While you can jog the machine before referencing, you cannot run parts until the machine has been referenced. Should one of your home/limit switches fail to work, manually reference the machine as discussed in chapter 8, *Troubleshooting*.

### 4.1.2   Prepare the Workpiece

Using the 1" dia. x 3.5" long aluminium bar included with your lathe (PN 34718), insert the bar into the collet or chuck so that 2" protrudes from the face (see **Figure 4.3**). You will be machining the first 1.5" of this bar, and if you don't leave a bit more sticking out, you risk crashing a tool into the chuck or spindle.
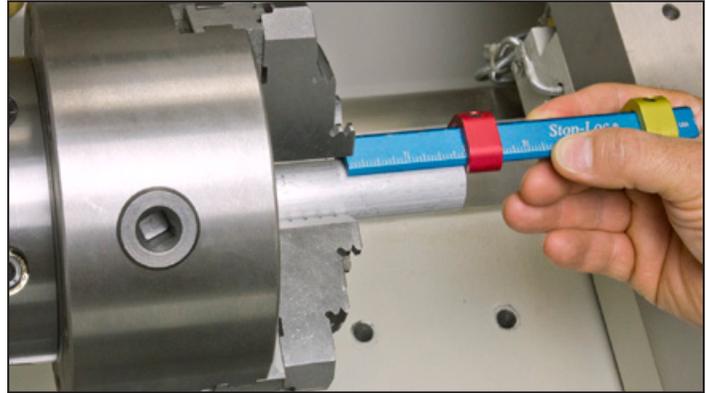


**Figure 4.3**

### 4.1.3   Prepare the Tools

Regardless of whether your lathe is set up for quick change, turret, or gang-mounted tooling (see **Figure 4.4**, **Figure 4.5**, and **Figure 4.6**), you need both a right-handed turning tool (PN 33131 or similar) and a parting tool (PN 33123 or similar) as shown in **Figure 4.2**. If using gang tooling, make sure there is at least 2" between tools. Also, ensure the Z position of the tools is within 1"-2" of each other. For example, a long drill bit may stick out much further in Z than other gang-mounted tools and collide with chuck or spindle when one of the shorter tools is brought in to machine workpiece. If using a turret, insert the turning tool in position 1 and the parting tool in position 2.



**Figure 4.5**
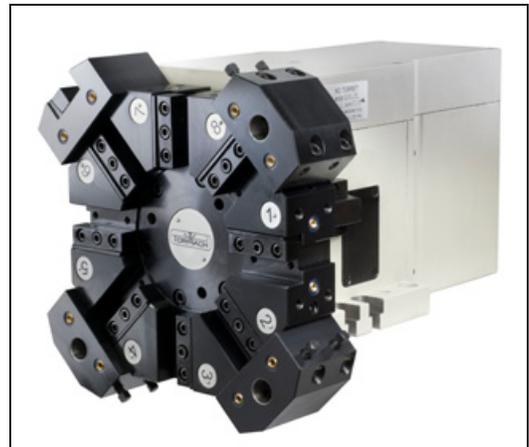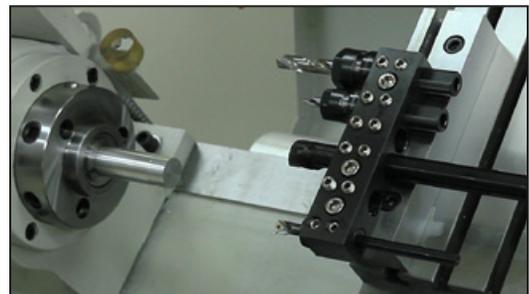


**Figure 4.4**



**Figure 4.6**

### 4.1.4   Understand Machine Position, Work Offsets, and Tool Offsets

Work offsets are a concept that allows the operator to think in terms of X/Z coordinates with respect to the part instead of thinking of them with respect to the machine position.

Tool offsets allow the operator to use tools of different length or, in the case of gang tooling, different X/Z positions on the carriage. Tool offsets are broken down into two components: geometry offsets and wear offsets. Geometry offsets represent the distance from the work offset zero location to the tool's control point. Unlike on a milling machine (where G43 must be called out to apply an offset), tool geometry offsets are automatically applied with the Txx tool change command.

Lastly, and very importantly, is the sign convention for a rear tool post slant bed CNC lathe like the 15L Slant-PRO: Z negative is toward the spindle, and X negative is toward the operator. This becomes important when operators choose the manual tool change option or the gang tooling option.

***IMPORTANT!*** *All tools mounted for use on the operator side of the workpiece are touched off using negative X (diameter) values, and most X words in part programming for these tools will have negative values.*

In many cases, PathPilot warns the operator of a violation of this rule, but it has no way of preventing bad code from running.

### 4.1.5   Set the Length Units

You can program your machine in either inches or in millimeters. The machine uses the defined setting until you program a different command (G20 or G21). The settings are also retained after a power cycle, once the machine is out of reset.

#### 4.1.5.1   Programming in Inches

Depending on your workflow, do one of the following:

- Type `G20` in the MDI line and press *Enter* on your keyboard
- Program `G20` in your G-code program

#### 4.1.5.2   Programming in Millimeters

Depending on your workflow, do one of the following:

- Type `G21` in the MDI line and press *Enter* on your keyboard
- Program `G21` in your G-code program

### 4.1.6   Touch Off the Workpiece to Set the Work Offsets

There are many ways of conceptualizing tool and work offsets, but we use the idea of a true positive tool length to demonstrate this first part program. When using this method we must select a reference surface to use when establishing the work offset zero position. If you select the faces of a quick change tool holder or turret pocket as the reference surface, then the geometry offsets are equal in value to the stick out of the tool, hence the name, true positive tool length.

The actual method differs depending on whether you have a gang, turret, or quick change tooling setup. For the purposes of this instruction, we assume you have the quick change setup and that you have adjusted your quick change tool holders to centerline height. Make sure that there are no tool geometry offsets currently applied by clicking *Reset Tool* on the *Offsets* screen (see **Figure 4.12**), which clears the geometry offsets for the currently loaded tool. If the *Offsets* screen displays a selection of tools (see **Figure 4.11**), this indicates no tool offsets have been applied.

#### 4.1.6.1   Setting the Z Work Offset

1.  Insert an empty toolholder in your quick change tool post and jog the machine until the tool holder touches the end of the workpiece (see **Figure 4.7**). You may use a sheet of paper as a feeler gauge to tell you when you're making contact.

2.  When the tool holder has just touched the end of the work, click the *Zero* button next to the Z digital read out (DRO) to set the current work offset Z position to zero (see **Figure 4.8**).



Figure 4.7
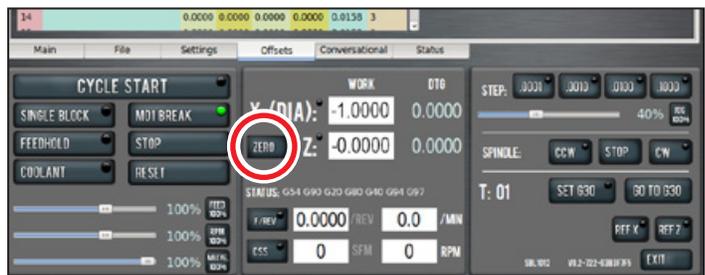


Figure 4.8

### 4.1.6.2   Setting the X Work Offset

1. Jog the tool holder down in X, over in Z, and approach the workpiece in X until the face of the holder just barely contacts the workpiece in X (see **Figure 4.9**).

2. Because this is a one inch diameter workpiece, and all positions on the operator side of the spindle centerline are negative, you need to enter -1.0 in the diameter DRO to touch off at 1" in X. Press *Enter* after typing in the DRO to apply the value (see **Figure 4.10**).
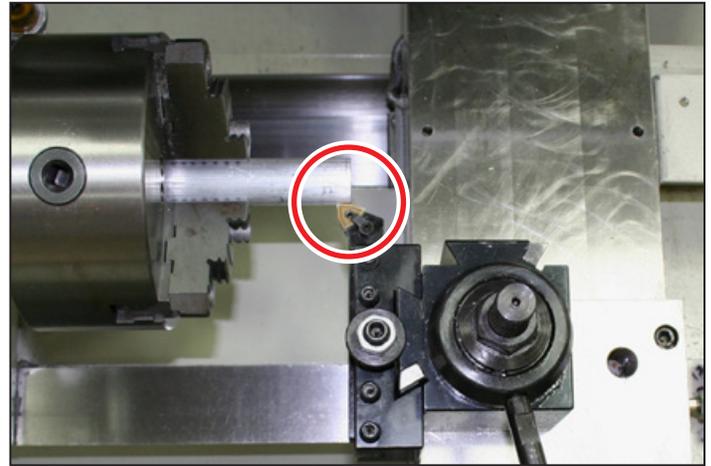


**Figure 4.9**

*NOTE: This technique is used for setting any DRO. Remember to press Enter after any DRO change. If you forget and just click on another DRO, any value you have just entered is discarded. The purpose of this is to avoid accidental changes.*

3. Jog the machine back a safe distance away from the part.



**Figure 4.10**

### 4.1.7   Set the Tool Geometry Offsets

#### 4.1.7.1   Setting Tool 1

Next we measure the tool geometry offsets for tool 1, a right-hand turning tool. If you are using a quick change tool post, manually insert tool 1 (a right-hand turning tool) into the tool post.

⚠  **WARNING! Entanglement Hazard:** *If using the (optional) turret tool changer, typing T01 in the MDI line causes the turret to index to the pocket 1 position. Keep hands, hair, jewelry, and clothing away from moving parts. Failure to do so could result in serious injury or death.*

1. Select tool one in the PathPilot interface. This can be done either by typing *T01* in the MDI line or by clicking the *Offsets* tab and typing *1* in the *Tool* DRO; click *Enter*.

2. Click on the *Tool Touch* tab on the *Offsets* screen and select the front tool post, right-hand turning tool (see **Figure 4.11**).

*NOTE: Select the rear tool post right-hand turning tool for use with a turret; the next touch off screen is shown in Figure 4.12.*

3. Click the *Main* tab and jog the machine (using either the jog shuttle or the keyboard arrow keys) until the tool is near the part (if using the keyboard arrow keys, left and right arrow correspond to -/+ Z, respectively, and up and down arrow keys correspond to +/- X, respectively). To switch from step jogging to continuous mode, click the jog speed slider or the *Jog 100%* button. To switch from continuous mode to step jogging, click any step increment button.

4. Close the chuck guard or enclosure door.
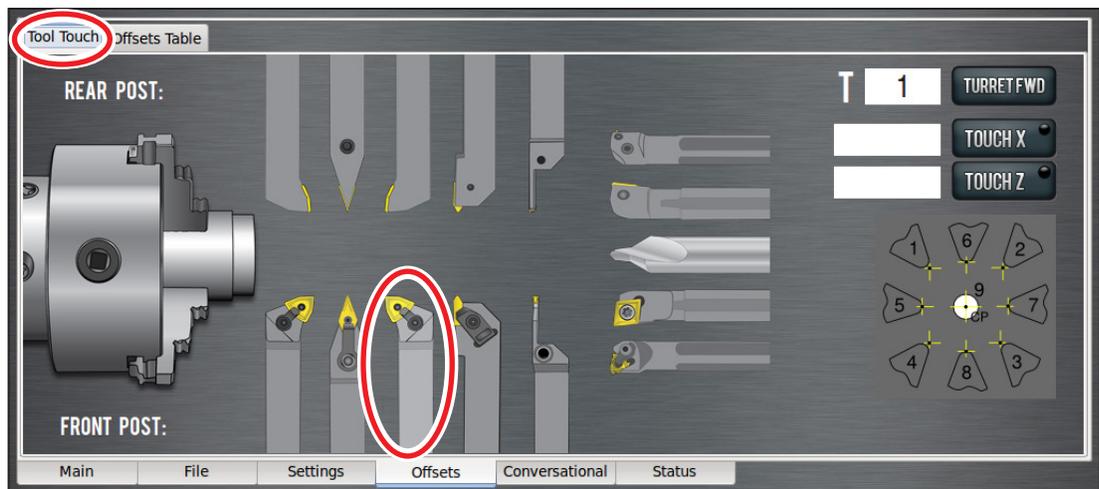


**Figure 4.11**
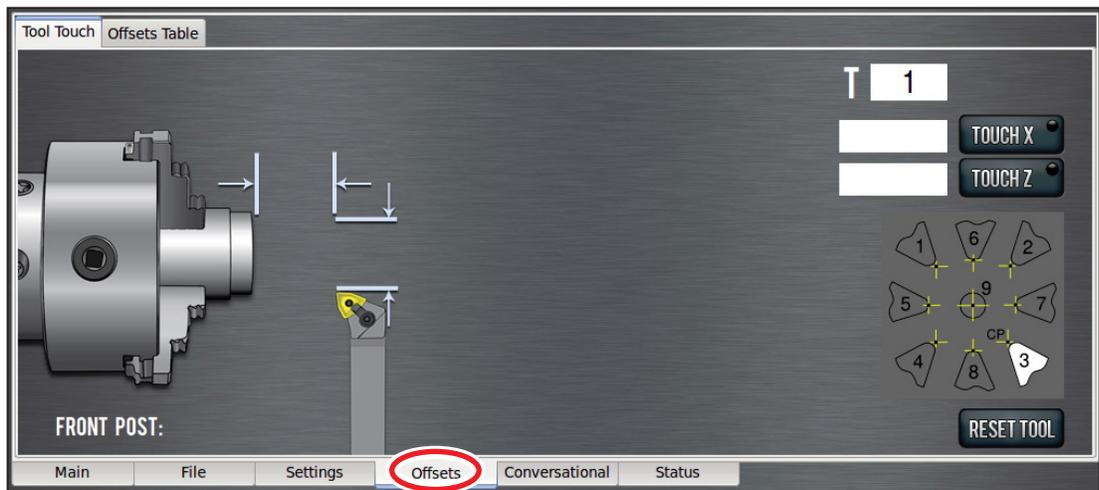


**Figure 4.12**
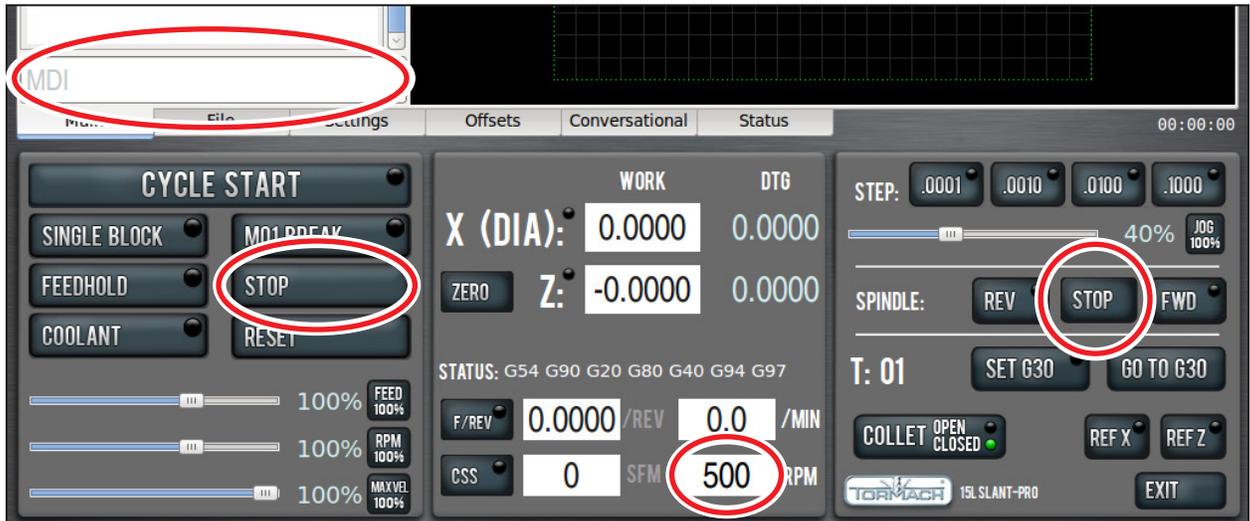
**Figure 4.13**

5. Adjust spindle speed to 500 RPMs – either by entering *500* in the *RPM* DRO and clicking *FWD* or by typing G97 M03 S500 in the MDI line (see **Figure 4.13**).

6. Using the jog controls, take a skim cut off of the diameter of the workpiece. This cut need only be long enough to measure the cut diameter with a micrometer.

7. Jog the tool away from the workpiece in Z.

*IMPORTANT! Do not jog the machine in X.*

8. Stop the spindle by clicking either *Stop* or Spindle *Stop*, or by typing M05 in the MDI line (see **Figure 4.13**).

9. Measure the diameter of the skim-cut workpiece with a micrometer, then enter the negative of that value in the Touch X DRO on the *Offsets* screen *Tool Touch* tab and click the *Touch X* button. If using a turret or a rear-mounted gang tool, enter the positive value. Note that the LED on that button illuminates, indicating that a tool offset has been set for that tool.

10. With the spindle off, jog the machine toward the face of the workpiece in the Z direction. You may touch off Z by using a piece of paper placed between the tool and the workpiece. Move the piece of paper back and forth while jogging the machine slowing in steps of 0.001" towards the face. When the paper binds up, enter the thickness of the paper in the DRO next to the *Touch Z* button (typically 0.003"); click *Touch Z*.

11. Jog away from the workpiece. Look at both DROs: *X (Dia.)* and *Z*. Verify that they look appropriate for the current position of the tool.

*NOTE: The X (Dia.) DRO always displays diameter values, not radius. Therefore, when front tool post tool is 2" away from spindle centerline, X (Dia.) DRO should read 4.0".*
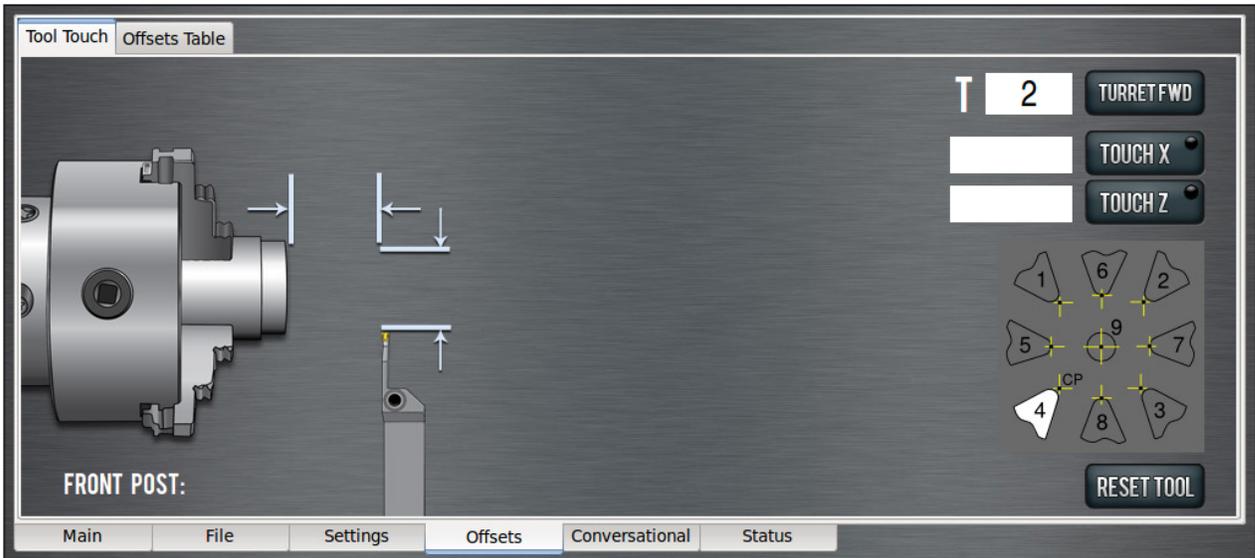
**Figure 4.14**

### 4.1.7.2 Touching Off the Parting Tool

1. Jog far enough away from the part that any tool change to another tool can be completed safely without crashing into the workpiece or chuck. This is especially important for gang tooling and turret operation, as the control completes these tool changes automatically.

2. Set the tool change position by clicking *Set G30* on the *Main* tab. An illuminated LED in the corner of this button indicates that the G30 position has been set.

3. Go to the *Tool Touch* screen by clicking on the *Offsets* tab, and making sure the *Tool Touch* tab is active.

4. Change tools to tool 2 (the parting tool) by typing *2* in the *T* DRO on the *Tool Touch* tab; click *Enter* (see **Figure 4.14**).

5. You should see the tool type selection screen again (see **Figure 4.11**). Select the front tool post parting tool (unless using a rear tool post parting tool, in which case you should select the rear tool post parting tool).

6. If you make a mistake selecting the tool type, click *Reset Tool* to reset the tool type and bring you back to the tool selection screen.

**NOTE:** *The control point for this tool is 4 (see* **Figure 4.14***). This may seem unintuitive at first but when you consider that the length of the part (not the remaining stock in the spindle) is the critical dimension, it becomes clear that a parting tool's control point should be 4. The control point diagram shown in* **Figure 4.14** *illustrates the control point, but does not necessarily match the insert shape.*

7. Turn the spindle on and jog the machine to take a skim cut off of the diameter of the workpiece.

8. Jog away from the workpiece in Z.

**IMPORTANT!** *Do not move the tool in X.*

9. Turn the spindle off and measure the diameter of the workpiece with a micrometer. Enter the negative of this diameter value in the DRO to the left of the *Touch X* button, then click *Touch X*.

10. Touch Z using the same procedure described for tool 1. When the paper binds up, enter the thickness of the paper plus the width of the parting tool insert in the DRO next to the *Touch Z* button (typically 0.003" + 0.120") and click *Touch Z*.

***NOTE:*** *You must enter width of the insert because the control point of a parting tool is the right edge, not the left edge.*

11. To double-check your work, you can go to the *Offsets Table* tab on the *Offsets* screen and look at the geometry offsets for each tool: the values in the X and Z columns represent the stick out of your tool past the faces of the tool holder.

***NOTE:*** *X values are expressed in diameters so the actual stick out is half the number shown in the column, and is negative for front tool post tools (see* **Figure 4.15***).*

Congratulations, you've just successfully measured your tool offsets. If you don't remove these tools from their holders (or pockets, for turret changers) you don't need to go through the tool offset setting process again until you replace an insert.



**Figure 4.15**

### 4.1.8 Write the G-code

Now use the conversational programming capabilities of the PathPilot interface to generate G-code to produce our part. We'll break this down into five steps:

1. Face the part.

2. Turn the 1" stock down to 0.9375" along the entire length of the part (1.5").

3. Turn the first 1.25" of part down to .750".

4.  Chamfer the edge with a 0.050" break.

5.  Part off.

The conversational screens are divided into two sections: parameters common to most operations are displayed on the left and parameters (including part geometry) that are operation-specific are displayed on the right.

The entry fields should be self-explanatory and are covered in detail in following chapters. For now, enter the values shown in each operation. Make note of a few things:

*   The initial X value is -1.000, not +1.000. This is because we're using a front tool post tool, and all diameters on the operator side of the spindle centerline are expressed as negative values. If you have a turret or a rear-mounted gang tool, this would be a positive value.

*   The spindle speed (both roughing and finishing) is expressed as SFM, not RPM. On a CNC lathe it is advantageous to use constant surface speed (which varies the RPM as the diameter changes) to increase part production rates and tool life and maintain a better cut throughout the program.

*   The feed rates (roughing and finishing) are shown in inch/rev, not in inch/min. Like constant surface speed, programming in inch/rev allows for optimal tooling performance.

### 4.1.8.1    Operation 1

1.  To write the code for the first operation, click the *Conversational* tab to bring up the conversational screens. Click the *Face* tab to bring up the facing screen (see **Figure 4.16**) to generate code to face the part back to Z of -0.100".

2.  Enter the values as shown in **Figure 4.16.**

3.  Click *Post to File* on the *Face* tab to save the G-code (see **Figure 4.16**).
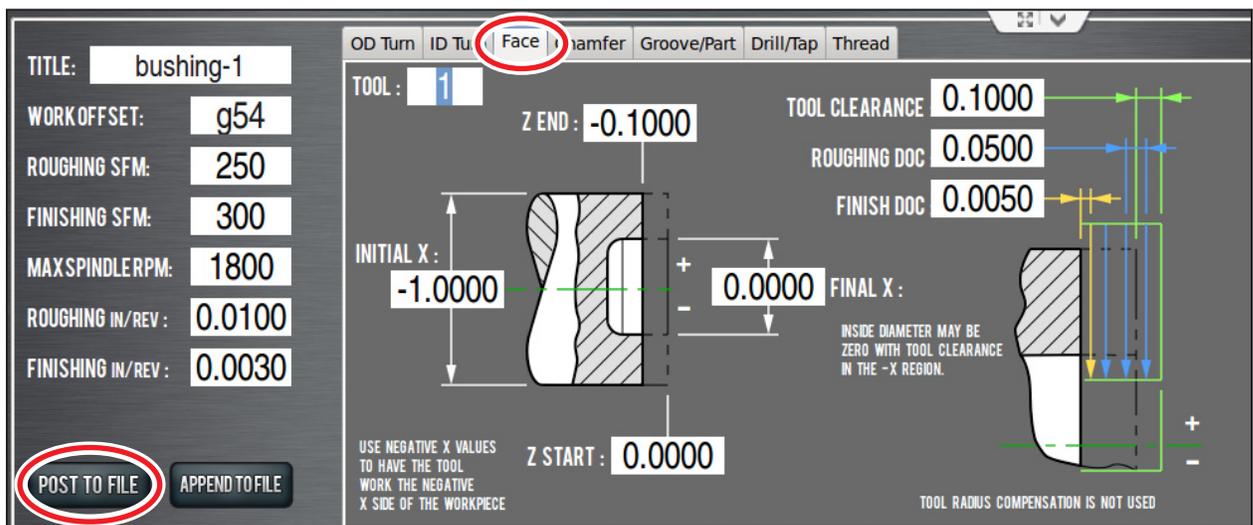


**Figure 4.16**

4. In the dialog window, click *Save* (see **Figure 4.17**)*;* the file is saved and automatically loads into the control and displays the tool path.

To run the program:

1. Click and drag the *MAXVEL slider* (see **Figure 4.18**) to bring it down to zero percent.

2. Click *Cycle Start* (see **Figure 4.18**). If the current tool is not tool 1 and the machine is configured for manual tool changes, the *Cycle Start* LED may blink requesting a tool change. Change the tool in your quick change tool post and confirm by clicking *Cycle Start* again.

**NOTE:** *Turret and gang tool changes happen automatically without operator intervention.*

3. Grab the *MAXVEL slider* again and slowly increase the allowed velocity. Bring the velocity back down to zero when you get close to the part and double check the values in the DROs. If everything looks correct, go ahead and move the *MAXVEL slider* back up to resume running the part program.



**Figure 4.17**
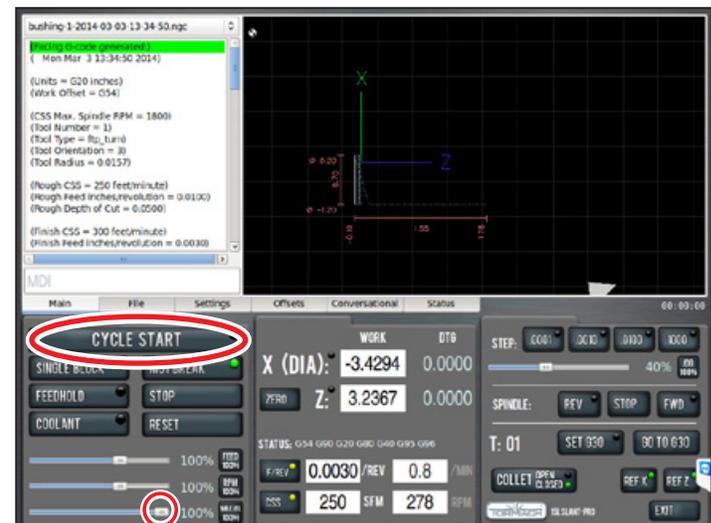


**Figure 4.18**

### 4.1.8.2 Operation 2

1. On the *Conversational* tab, click on the *OD Turn* tab. Enter the values as shown in **Figure 4.19**.

**NOTE:** *If you are using front tool post tooling, all X (diameter) values must be written as negative numbers.*
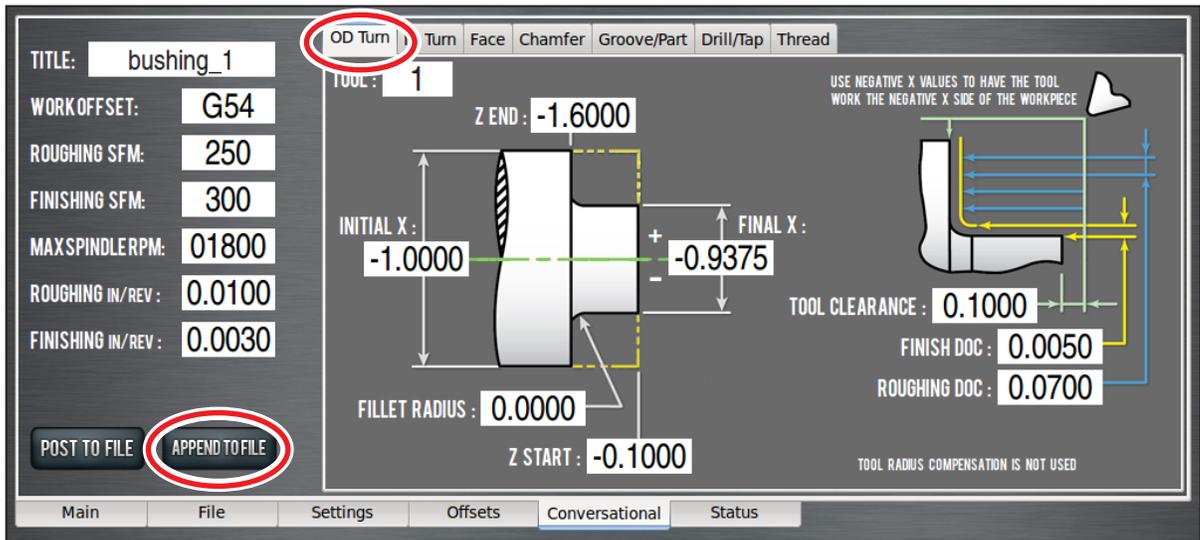


**Figure 4.19**

2. After you have entered the values, click *Append to File* (see **Figure 4.19**).

3. In the dialog window, select the file created when completing Operation 1 to add the OD Turn G-code and confirm *Append to File* (see **Figure 4.20**).

4. The changes to your file are loaded into the control, and you should see a tool path that looks something like **Figure 4.21**.

**IMPORTANT!** *Do not run this code until remainder of operations are written, then proceed to actually making the part.*
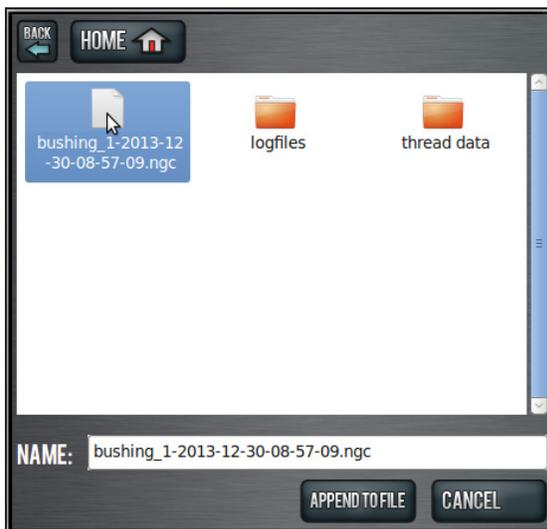


**Figure 4.20**
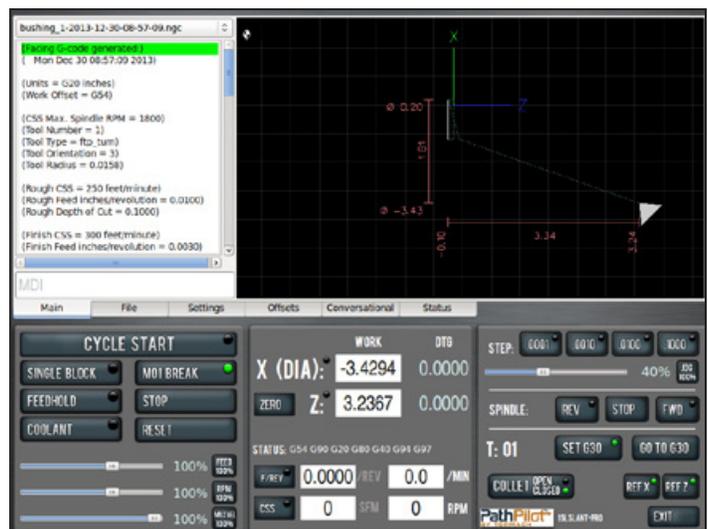


**Figure 4.21**

### 4.1.8.3 Operation 3

1. The third operation uses another OD turn routine to turn the first 1.125" of the part down to 0.750". Enter the values as shown in **Figure 4.22** on the *OD Turn* tab of the *Conversational* screen.
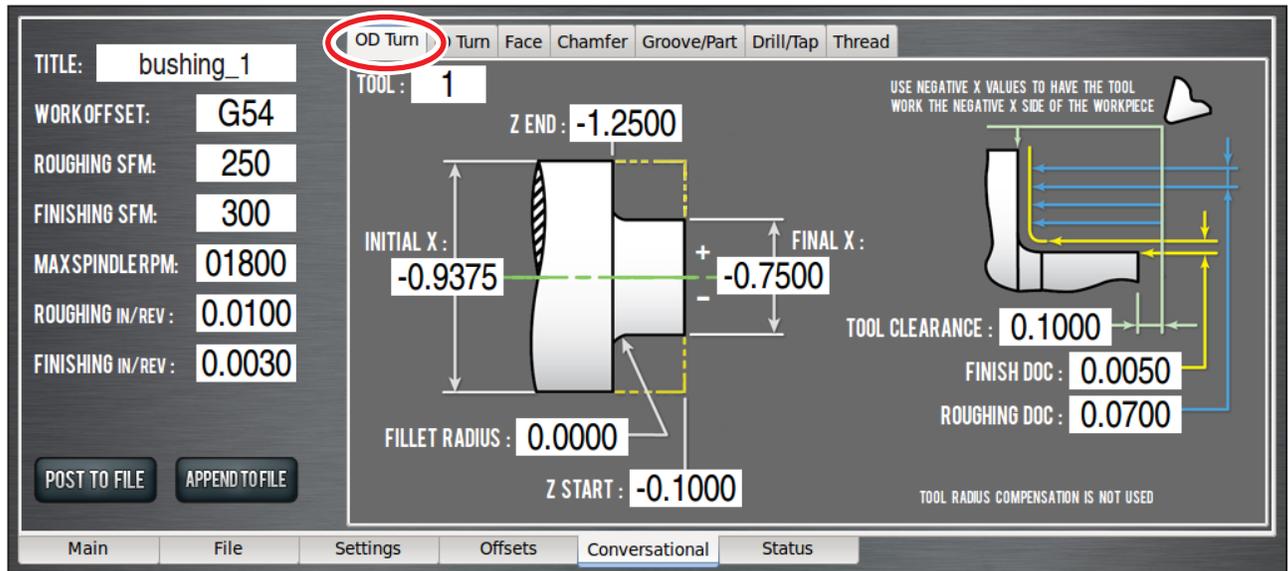


**Figure 4.22**

2. Append this code to the file as you did at the end of the *Operation 2* section earlier in this chapter. The tool path should now resemble **Figure 4.23**.
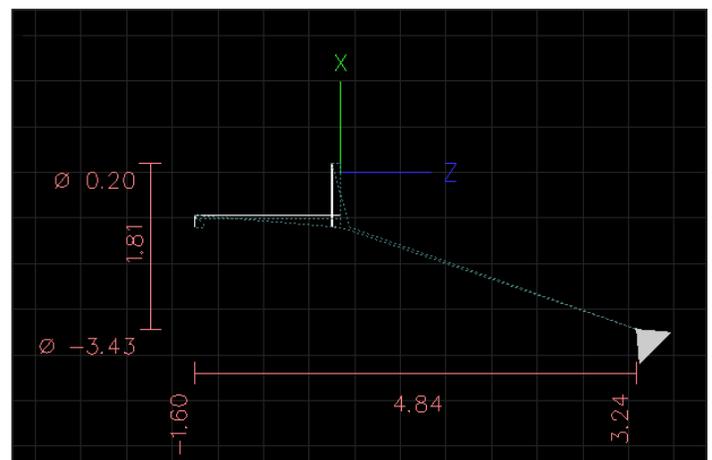


**Figure 4.23**

#### 4.1.8.4 Operation 4

1. Put a small chamfer on the part using the *Chamfer* tab of the *Conversational* screen. Enter the values as shown in **Figure 4.24**.

**NOTE:** *If you are using front tool post tooling, all X (diameter) values must be written as negative numbers.*
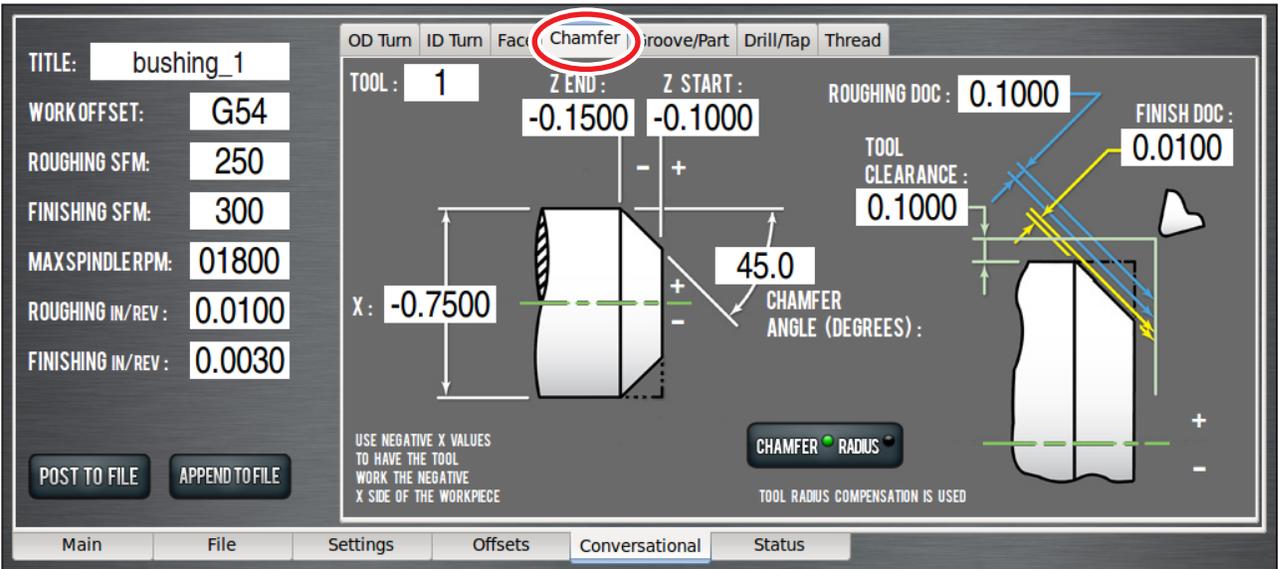


**Figure 4.24**

2. Click *Append to File* on the *Chamfer* tab to add this code to the program. The tool path should now look like **Figure 4.25**.
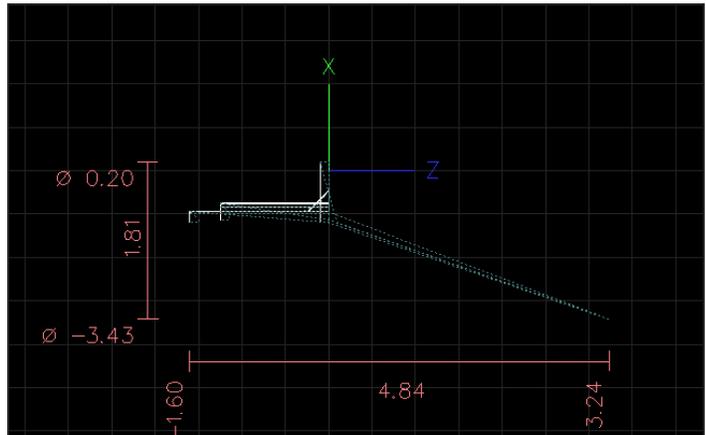


**Figure 4.25**

### 4.1.8.5  Operation 5

The final operation uses tool 2 (a parting tool) to part off the workpiece.

1.  Click the *Groove/Part* tab, then toggle the *Groove/Part* button to illuminate the *Part* LED. On the parting screen, enter the following values as shown in **Figure 4.26**.
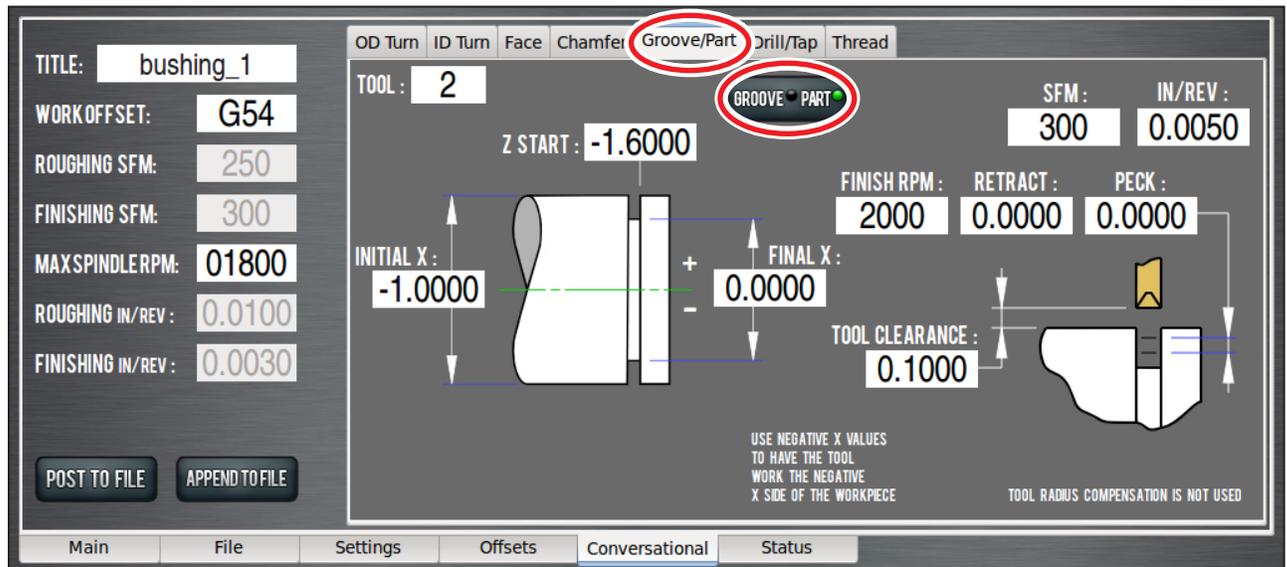


**Figure 4.26**

2.  Make sure to enter tool 2 in the tool DRO. If you try to use tool 1 you get an error message letting you know that a right-hand turning tool is not a valid tool for a parting routine.

3.  Finally, click on *Append to File* and choose the file you generated in the *Operation 1* section earlier in this chapter. Your screen should now look like the screen shown in **Figure 4.27**. The finished workpiece is shown in **Figure 4.28**.
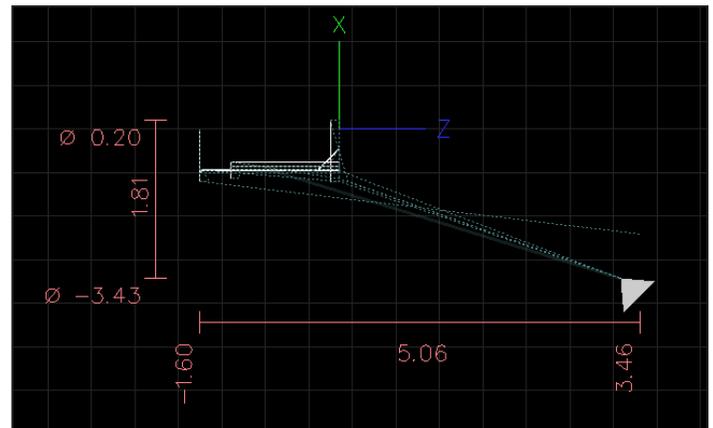


**Figure 4.27**

### 4.1.8.6   Run the Completed G-code Program

**Figure 4.27** shows a tool path representing the first, second, third, fourth and fifth. Run the program using the steps described in the *Operation 1* section earlier in this chapter.

The techniques used to program the first part form the basis of most of the operations performed on the lathe. The big concept to remember is that front tool post tools use negative diameter values. This includes using negative diameters when setting work and tool offsets, as well as when using the conversational screens.
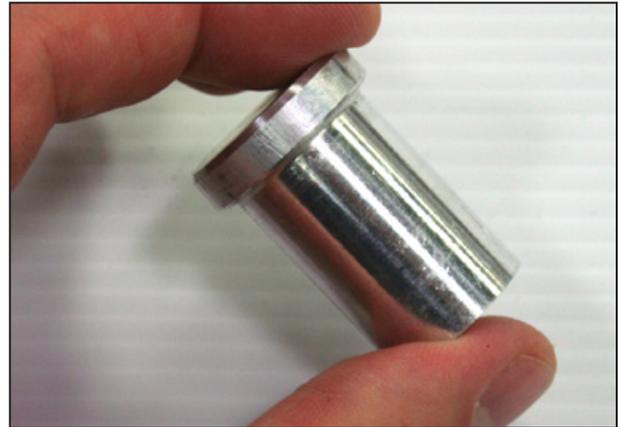


**Figure 4.28**

# 5.  PathPilot Interface

## 5.1  Overall Layout

The PathPilot® interface is divided into two main sections, the *Notebook* and the *Persistent Controls* (see **Figure 5.1**). The *Persistent Controls* make up the bottom half of the screen and include three groups: *Program Control Group*, *Position Status Group*, and *Manual Control Group*. These controls are present regardless of what task the operator is doing. The top half of the screen is a *Notebook*, which includes six tabs (*Main*, *File*, *Settings*, *Offsets*, *Conversational*, and *Status*) that can be used to select different *Notebook* pages, each of which display buttons, digital read outs (DROs), and information pertinent to different functions of the PathPilot interface. For example, the buttons and information on the *File* tab are used for such tasks as transferring a G-code file from a USB drive to controller hard drive, loading a G-code file into memory, or editing a G-code file.
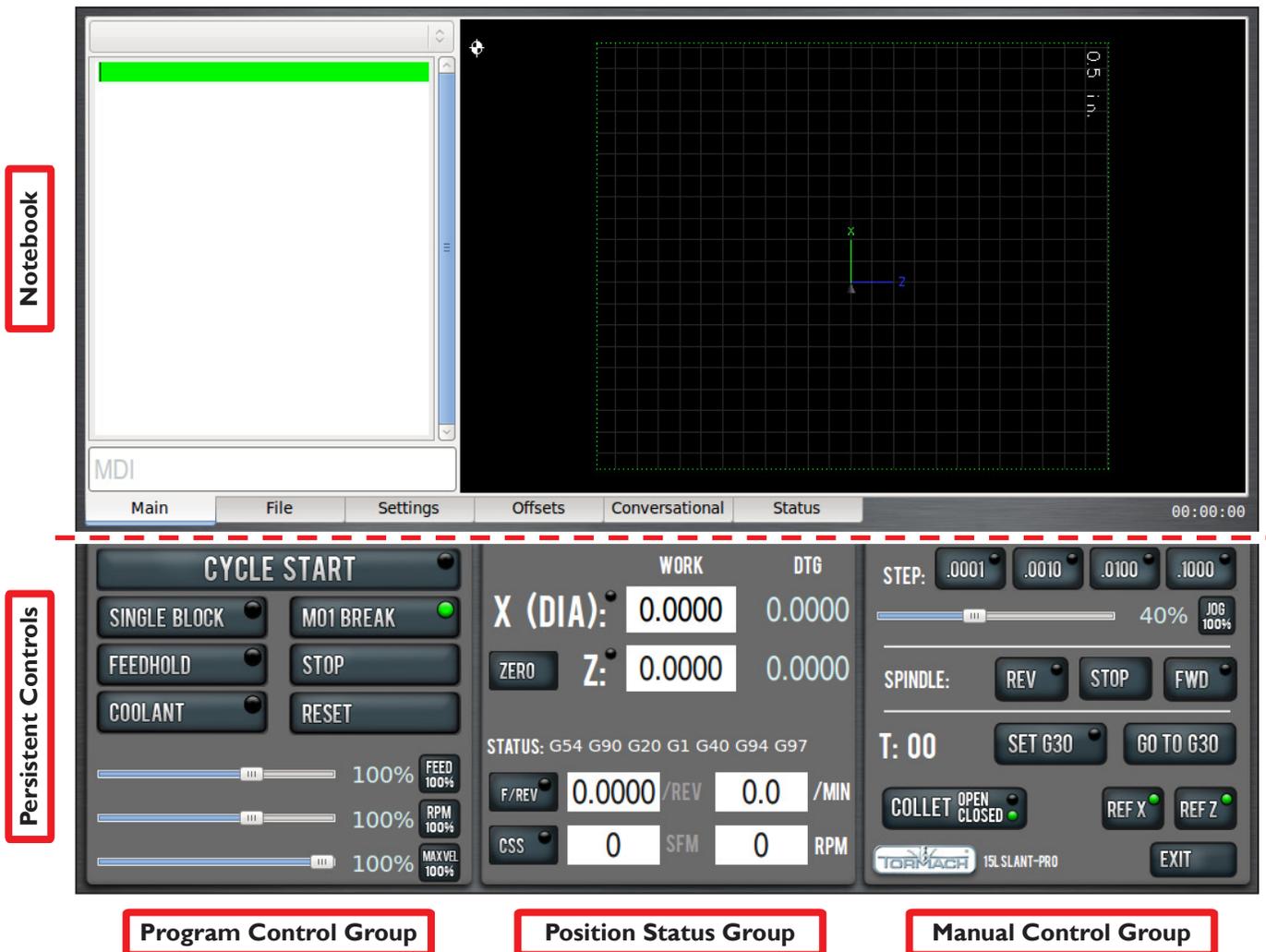


**Figure 5.1**

While the *Notebook* allows you to perform a variety of tasks (loading G-code file, writing G-code with the conversational interface, touching off tools, etc.), the *Persistent Controls* contains the controls used to set up a job and execute G-code. Operators familiar with Tormach milling machines (or most other CNC machines) are familiar with many of the buttons on the bottom half of the screen.

## 5.2   Persistent Controls

The controls on the lower half of the screen are always present – they don't move or disappear while paging through the notebook. Persistent controls are divided into three logical families: Program Control Group (see **Figure 5.2**), Position Status Group (see **Figure 5.3**), and Manual Control Group (see **Figure 5.6**).



**Figure 5.2**

### 5.2.1   Program Control Group

The buttons, sliders, and DROs on the *Program Control Group* are all functions that relate to tasks the operator might perform while running a G-code program. They may be used at any time while running a program, or before running a program to set modes like *Single Block* or *M01 Break*.
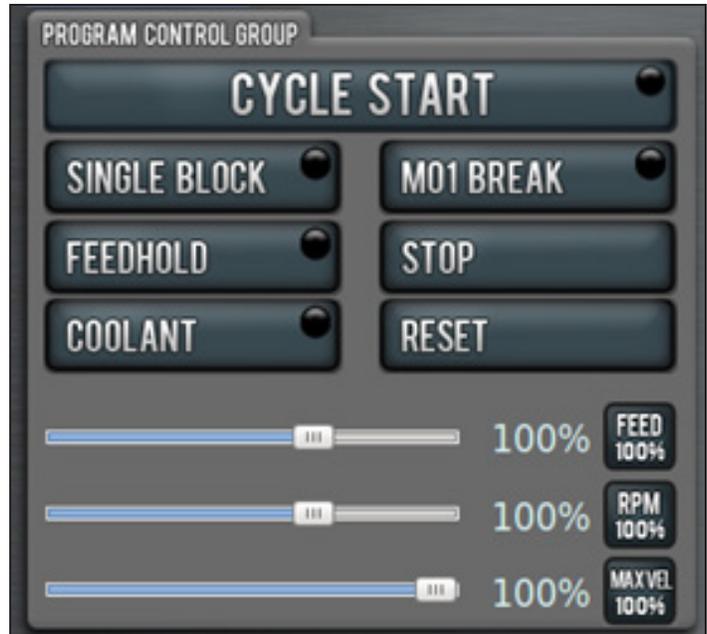
**Cycle Start** – Starts a program. Clicking this button triggers an alarm unless the chuck guard or enclosure door is closed and a valid G-code program is loaded. While running a program, the *Cycle Start* LED illuminates green.

If *Single Block* is active, the lathe executes one line of G-code per *Cycle Start* click. When running a program, if lathe motion is paused due to *Feedhold*, *M01 Break*, *Single Block*, or because the lathe is waiting on a manual tool change, the *Cycle Start* LED flashes until clicked again.

**Single Block** – Turns *Single Block* on (LED illuminated) or off (LED off). When single block mode is active, the machine executes one block of G-code, then pauses and flashes the *Cycle Start* LED, inviting the operator to click *Cycle Start* to execute the next line of G-code. This feature may be turned on or off before running a program or during program execution.

*NOTE: Non-motion lines are ignored by single block mode. This means that the interface will skip comment lines and preparatory (non-motion G-codes) commands.*

**M01 Break** – Turns *M01 Break* on (LED illuminated) or off (LED off). When M01 break is active, and an M01 (optional stop) is programmed in the G-code file, the machine stops when it reaches the M01 line and the *Cycle Start* LED flashes. The lathe continues to execute the program lines after the M01 when *Cycle Start* is clicked. This feature may be turned on or off before running a program or during program execution.

**Feedhold** – Turns *Feedhold* on (LED illuminated). Turning feedhold on pauses machine motion and the *Cycle Start* LED flashes. Turning feedhold on leaves the spindle running if it is already on. To turn feedhold off, click *Cycle Start*. The *Feedhold* button works during program execution or during manual data input (MDI) moves. Feedhold has no effect when the machine is not moving. Also, application of *Feedhold* is delayed if clicked during a G76 threading or G33.1 rigid tapping move until that spindle-synchronized move is complete.

**Stop** – Stops all machine motion, including spindle motion. If clicked while running a program or during an MDI move, the machine stops. If clicked while running a G-code program, it rewinds the G-code program (see *Reset* later in this section). Stop doesn't change the current modal state of the machine (G54, G01, etc.).

**Coolant** – Turns *Coolant* on (LED illuminated) or off (LED off). Clicking this button powers on or off the coolant accessory port on the side of the electrical cabinet. This is the equivalent to M08 or M09 G-code commands. It may be clicked before, after, or during program execution or an MDI move.

**Reset** – Brings the machine out of an E-stop condition, resets G-code modalities, clears alarm messages, and rewinds the G-code program. When the machine is first powered on, or after an emergency stop, the *Reset* LED flashes. When flashing and after the machine has been powered on again, clicking *Reset* starts and verifies communication between the machine and the controller.

Reset may be clicked at any time subsequent to powering on the machine. It has these effects:

- Resets all modal G-codes to their normal state including work offset to G54 default
- Rewinds a G-code program
- Stops a program or MDI move if one is currently in progress
- Clears alarms (for more information, refer to chapter 4, *Intro to PathPilot*)
- Clears the tool path backplot

**Spindle Override** – The *Spindle Override* slider and *100 Percent* button allow overriding of the commanded spindle speed by percentages ranging from 1-150 percent. The *100 Percent* button returns the override to 100 percent of the commanded value (in effect, no override). The spindle must be running for these controls to have a noticeable effect. If overriding the spindle when the spindle is stopped, the speed is overridden the next time the spindle is started. The override doesn't drive the spindle past its maximum speed of 3500 RPM (5C configuration) or 2500 (6" chuck configuration). It does affect the speed of a spindle command limited by a D word.

For instance, G96 S300 D1000 prevents the spindle from going faster than 1000 RPM in constant surface speed mode, but commanding an override of 150 percent causes the spindle to turn at 1500 RPM when the surface speed requirements and the diameter value demand it.

**Feedrate Override** – The *Feedrate Override* slider and *100 Percent* button work similarly to the spindle override controls. They affect the commanded feedrate by a percentage from 1-150 percent. The feedrate override works for both MDI, jogging, and G-code program G01/G02/G03 moves. The override has no effect on G00 (rapid) moves.

**MAXVEL Override** – The *Maxvel Override* slider and *100 Percent* button work similarly to the federate override controls, except that these controls affect both G00 and G01 moves. They clamp the machine velocity to a percentage of the maximum velocity. The *Maxvel* slider can be very useful when running new code for the first time. It can be used to stop the machine by sliding it down to 0 percent and verifying that the Distance To Go and Dia./Z DROs look appropriate before continuing.

### 5.2.2   Position Status Group

The buttons and DROs of the *Position Status Group* pertain to machine position, active G-code modalities, and feed and speed settings (see **Figure 5.3**). These controls may be used at any time before or after running a G-code program or MDI move. They are unavailable for operator input while the machine is moving.

X (Diameter) and Z work offset DROs – next to the labels X (DIA) and Z – are DROs that display the current machine position expressed in the currently active work offset coordinate system (G54, G55, etc.). In the case of the X DRO , all values are expressed in terms of the diameter of the workpiece, not the radius. For instance, if the active tool is 2" away from the spindle centerline, the X DRO reads 4.000.

When the machine is at rest these readouts are also user entry fields. You can change the current work offset position by clicking the mouse in the DRO. It becomes highlighted (see **Figure 5.4**). Type a number, for example 4.0, and press *Enter* on the keyboard. The *Esc* key returns to the original value.
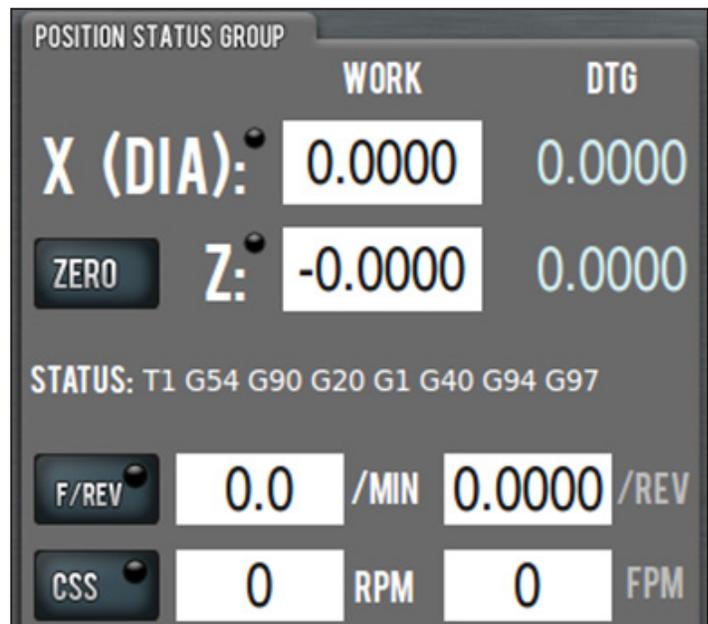


**Figure 5.3**
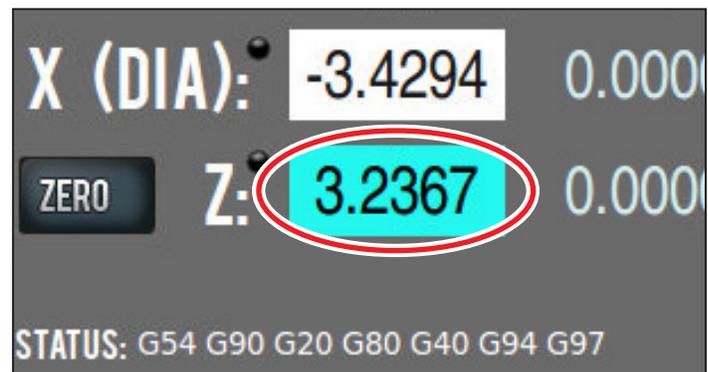


**Figure 5.4**

***IMPORTANT!*** *This action does not move the machine, but does change the work offset setting.*

This technique is used for setting any DRO. Remember to click *Enter* after any DRO change. Clicking on another DRO before clicking *Enter* discards any value just entered. This is designed to avoid accidental changes. For convenience, the *Zero* button to the left of the Z DRO can be used to set the Z work offset to 0.000.

**X (Diameter) and Z Distance-to-Go Labels** – Just to the right of the X and Z DROs are read-only distance-to-go labels in light blue. They display the distance remaining in any single move.

If you feedhold the machine in the middle of a move, or turn the *MAXVEL* or *Feedrate* overrides to zero percent, these labels display the distance left in the commanded move. These can be very helpful in proving out a part.

**Status Line** – The status line displays the currently active G-code modalities and the active tool. A more detailed description of these active G-codes is provided on the *Settings* screen.

**F/REV, F/min DROs** – As described in the G-codes section, the 15L Slant-PRO recognizes two feed rate modes: units per minute (G94) and units per revolution (or REV) of spindle (G95) as shown in **Figure 5.5**. In units per minute feed rate mode, a G-code F word is interpreted to mean the controlled point should move at a certain number of inches or millimeters per minute, depending upon what length units are being used and which axis or axes are moving.
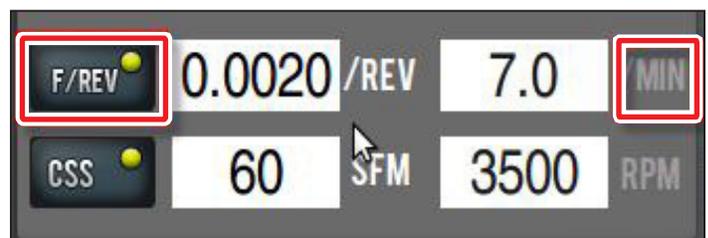


**Figure 5.5**

In units per REV feed rate mode, an F word is interpreted to mean the controlled point should move at a certain number of inches or millimeters per spindle revolution depending upon what length units are being used and which axis or axes are moving.

***NOTE:*** *For almost all machining on a lathe, programming in units per revolution is more desirable.*

These settings are mutually exclusive. In other words, you cannot simultaneously be in both *F/Rev* and *F/Min* modes. The *F/Min* and *F/Rev* DROs display the current F word in both units/revolution and units/minute. The active mode is indicated by both the *F/Rev* LED and the */REV* and */MIN* labels next to the DROs. When *F/REV* is active, its LED illuminates, the *F/REV* label is white, and the *F/MIN* label is gray. You can turn on *F/REV* mode using any of the following methods (see **Figure 5.5**):

- Click in the F/REV DRO, enter a value, and press *Enter* on the keyboard
- Click the *F/REV* button when the button LED is dark
- Type G95 into the MDI field and press *Enter*
- Run a G-code program that contains a G95 command

F/REV mode can be turned off by any of the following:

- Click in the F/MIN DRO, enter a value, and press *Enter* on the keyboard
- Click the *F/REV* button when the button LED is yellow
- Type G94 into the MDI field and press *Enter* on the keyboard
- Run a G-code program that contains a G94 command

**SFM/RPM DROs** - The 15L Slant-PRO recognizes two spindle speed command modes: constant RPM (G97) and constant surface speed (G96). For more information, refer to chapter 6, *Programming*.

In constant RPM mode (G97), the spindle turns at an RPM equal to the S word. For instance, G97 S500 M03 starts the spindle at 500 RPMs. In constant surface speed mode (G96) the spindle RPM varies to maintain a constant speed in terms of linear feet per minute over the workpiece. For example, G96 S50 M03 turns the spindle on, and the linear speed at which the cutting tool's control point travels over the work piece's surface is 50 feet per minute. In constant surface speed mode, the actual spindle RPM depends on the diameter of the work being turned, increasing as diameters get smaller and decreasing as diameters get larger. Because tool life and efficiency depends on surface speed (and not RPMs), constant surface speed mode is the preferred mode of operation of CNC lathes.

### 5.2.3    Manual Control Group

The buttons and slider on the Manual Control Group allow the operator to jog the machine axes and start or stop the spindle (see **Figure 5.6**).

**Jogging Controls** – The 15L Slant-PRO can be jogged with either the jog shuttle or by using the arrow keys on the keyboard. When using the arrow keys:

- *Left arrow* jogs Z-axis in negative Z direction (toward spindle)
- *Right arrow* jogs Z-axis in positive Z direction (away from spindle)
- *Up arrow* jogs X-axis in positive direction (away from operator)
- *Down arrow* jogs X-axis in negative X direction (toward operator)



**Figure 5.6**

Keyboard jogging is only available on the *Main* screen and on the *Tool Touch* tab of the *Offsets* screen. The machine does not jog using the arrow keys when on screens other than *Main* or *Offsets*. Whether using the jog shuttle or keyboard, there are two modes of jogging: step and continuous.

In step mode, the machine jogs in steps, where the step size is controlled by the four buttons to the right of the *Step:* label. Notice than in imperial (G20) units the step sizes range from 0.0001" to 0.1", whereas in metric (G21) mode the step sizes range from .01 mm to 10 mm. The LED in the upper right-hand corner of each of the step buttons indicates which step size is active. When all step LEDs are dark, the machine is in continuous jogging mode.

In continuous mode, the machine jogs at a continuous velocity when clicking and holding the arrow keys. The machine stops when the arrow key is released. The velocity is set using the jog speed slider (to the left of the *Jog 100%* button). To set jogging velocity to the maximum speed, either click and drag the jog speed slider to the far right position or simply click the *Jog 100%* button. To switch from continuous jogging to step jogging, click one of the step size buttons. To switch back to continuous jogging click the slider or *Jog 100%* button. Jogging is not available during program execution or during MDI moves.

### 5.2.3.1 Spindle Controls

The *REV*, *Stop*, and *FWD* buttons can be used to manually control the spindle. *REV* is the equivalent of typing M04 in the MDI field – it starts the spindle in the reverse direction at the RPM specified in the spindle RPM DRO (if in G97 – RPM mode) or at an RPM specified by the SFM DRO and the X (Dia.) value in the X DRO (if in G96 – Constant Surface Speed mode). The *Stop* button stops the spindle, similar to M05 command. *FWD* is the equivalent of typing M03 in the MDI field – it starts the spindle in the forward direction at the requested RPM. These buttons are unavailable when running a G-code program or in the middle of an MDI move. Clicking *REV* or *FWD* triggers an alarm if the chuck guard or enclosure door is open.

### 5.2.3.2 Tool Label, Collet Open/Closed, G30 Controls, Ref Axes Buttons

Below the spindle controls are the tool label (*T:*) and the *Set G30* and *Go To G30* buttons. The tool label displays the current tool, and if wear offsets are applied for that tool displays the wear offset number as well. In **Figure 5.6**, tool 1 is the current tool, but if the wear offsets for tool 1 were also applied, the field would read *T: 0101*. If the machine is equipped with a quick change tool post, and the manual tool change option is selected on *Settings* screen, this label flashes with the requested tool number when T command is encountered while running a G-code program.

The *Set G30* button allows you to jog the machine to a safe tool change position and set this position as the G30 position (for more information, refer to chapter 6, *Programming*). Clicking this button is equivalent to entering G30.1 in the MDI field.

The *Go To G30* button on the *Offsets* tab causes the machine to move to the G30 position in the Z-direction only, and is equivalent to typing G30 Z#5422 in the MDI field.

To set the G30 position, jog the machine to the desired position and click the *Set G30* button. Subsequent uses of the G30 command in G-code or the *Go To G30* button will cause the machine to move to this position.

Ref X and Ref Z buttons reference the axes to their home switch locations. This must be done after power on before you running a part program or using MDI commands. The axes must be referenced individually and not simultaneously. When referenced, the LED on each button turns green.

## 5.3 Keyboard Shortcuts

Several keyboard shortcuts are provided for operator convenience. Below is a list of shortcuts used in the PathPilot interface:

| Spacebar | Feedhold | | Alt+F | Coolant |
|---|---|---|---|---|
| Alt + Enter | Give focus to MDI line | | Alt+E | Edit currently loaded G-code program[1] |
| Alt+R | Cycle Start | | | |

[1]Use the Alt+E command on any PathPilot screen to edit G-code.

## 5.4 Main Tab

The Main tab is active by default when the PathPilot controller first powers on, and contains four controls: recent files, G-code window, MDI line, and tool path display (see **Figure 5.7**).
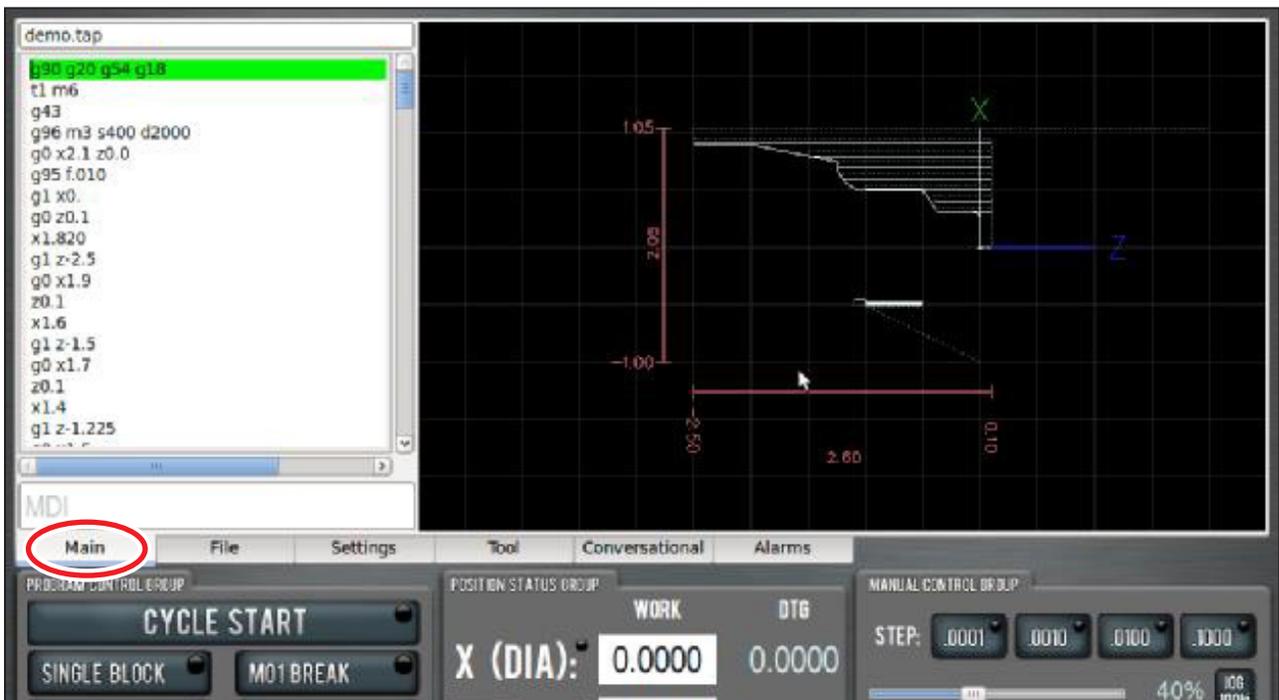


**Figure 5.7**

### 5.4.1 Selecting a Recent G-code Program File

The recent files drop-down menu displays the currently loaded G-code program file (see **Figure 5.8**). Click the drop-down menu to display the last five program files loaded into PathPilot; select the name of the program from the menu to load the G-code.
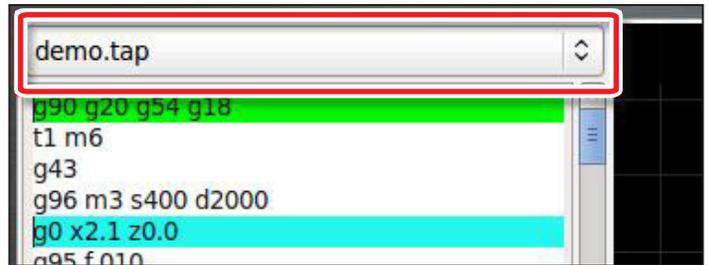


Select *Clear Current Program* in the recent files drop-down menu to close the currently loaded G-code program file.

**Figure 5.8**

### 5.4.2 Working in the G-code Window

The G-code window displays the G-code of the currently loaded program file. Use the scroll bars to view the entire file.

PathPilot highlights certain lines of code of interest. When running a G-code program in single block mode, there may be as many as three lines of G-code highlighted, each with a different color:

- Green line: indicates the start line, which is the first line in the program unless this has been changed with the Set Start Line feature
- Blue line: indicates the line of code that is currently executing
- Brown line: indicates the move that will occur the next time Cycle Start is pressed

The line at which the G-code program is executing – or start line – is highlighted in green. This usually is the first line of the code, but may be changed by placing the mouse pointer over the preferred start line, right clicking, and choosing the *Set Start Line* option.

#### 5.4.2.1 Setting a New Start Line

The start line is the line at which the G-code program begins. By default, this is the first line of code in the G-code program. Right-click the preferred start line of code in the program and select *Set Start Line* to change the start line.

When using the Set Start Line option, the operator is responsible for ensuring that the machine is in the proper state before the code executes.

To set a start line in the middle of a G-code program file, make sure any preparatory moves (like turning the spindle and coolant on) are manually completed before clicking *Cycle Start*.

The machine reads backwards through the beginning of the G-code program file to do things like set the appropriate G5x active work offset, G61/64 setting, and other modal states. It will not turn the spindle or coolant on.

### 5.4.2.2 Expanding the G-code Window

Double-click the G-code window to expand the G-code window and shrink the tool path display. Double-click the G-code window again to return the display to its original size.

## 5.4.3 Manually Entering Commands

When running a G-code program, commands to the machine are read from a file. You can also send G-code commands to the machine directly with the MDI line (see **Figure 5.9**).

**Figure 5.9**

Click the *MDI* field to use the MDI line; the line is highlighted. Type the command, using the *Backspace*, *Delete*, *Left* and *Right* arrow keys to correct typing errors. Press *Enter* to execute the command; press *Esc* to abandon it.

Click the *MDI* field and use the *Up* or *Down* arrow keys to copy a recent MDI command into the MDI line. Up to 100 MDI commands are stored for reuse; these commands are saved between sessions. Command history is available after a power cycle.

*NOTE: When the MDI line is open, all keystrokes are registered as a typed command. Jogging is not possible when you are clicked inside the MDI field.*

### 5.4.3.1 Searching in the Code

MDI has the ability to search the text of a G-code program file for specific numbers, codes, or items of interest like tools, feeds, and speeds.

Type *FIND* followed by the text to be searched in the MDI line (see **Figure 5.10**). Pressing *Enter* finds the next instance of the searched text; pressing *Enter* while holding down the *Shift* key finds the previous instance.

If found, PathPilot scrolls to the line containing the searched text and highlights it in yellow (see **Figure 5.10**). When the search reaches the end of the G-code file, it wraps and starts again from the beginning.

**Figure 5.10**

Change the starting point of the search by clicking on any line in the G-code window.
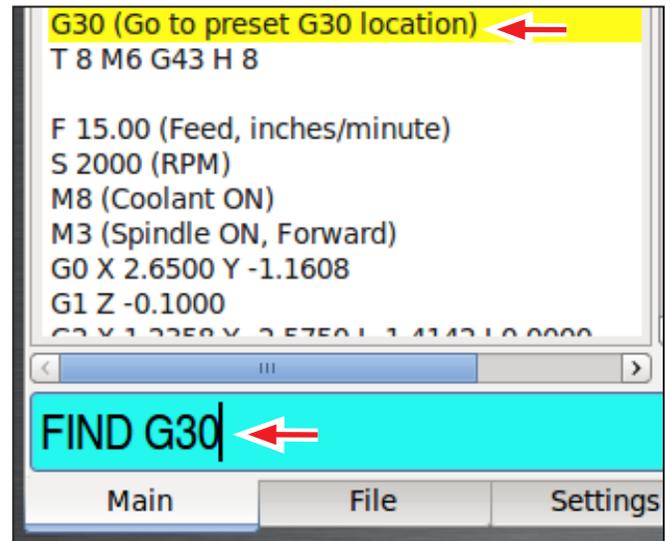
When used in conjunction with the *FIND* command, certain search terms (listed below) initiate a search through the G-code file to find more than just the actual search term:

- *FIND TOOL*: Searches for instances of the actual word *Tool* in the G-code and any *T* G-code command, which calls up a tool (e.g., T12)

- *FIND SPEED*: Searches for instances of the actual word *Speed* in the G-code and any *S* G-code command

- *FIND FEED*: Searches for instances of the actual word *Feed* in the G-code and any *F* G-code command (see **Figure 5.11**)
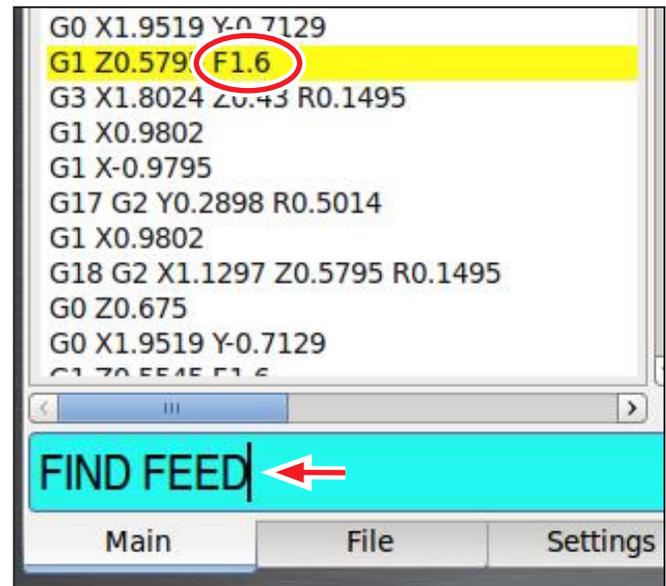


Figure 5.11

*NOTE: Search text ignores case, so the command FIND TOOL will match TOOL, Tool, tool, etc.*

The *FIND* command simplifies searching of a G-code file to verify speed and feed values and tool calls before cutting a part, or to find a specific set start line point in a large G-code file. For more information on using set start line, refer to *G-code Window* earlier in this section.

### 5.4.4    Working in the Tool Path Window

The tool path window displays a graphic representation of the tool path that is executed for the currently loaded G-code file, each with a different color:

- White lines: indicates the preview lines

- Red lines: indicates the tool path as it is cut

- Yellow lines: indicates jogging moves

- Dotted blue lines: indicates the boundary box, which represents the ends of travel of the axes

Double-click the tool path window, or click Reset, to erase the jogging or tool path lines.

#### 5.4.4.1    Changing the View of the Tool Path Window

Grid lines are visible behind the tool path when the view is top, front, or right. Grid lines are not available in ortho. By default, grid lines are drawn at 0.5" intervals when in G20 mode (5 mm intervals when in G21 mode). Right-click anywhere in the tool path display and select a different grid spacing to change the resolution of the grid lines. When a program is loaded, the program extents (furthest points to which the tool will travel while executing the G-code) are displayed to the left and bottom of the tool path.

## 5.5 File Tab

The *File* tab is used to transfer files to and from a USB drive, copy, delete, and rename files and folders (see **Figure 5.12**). The left window shows files and folders on the controller hard drive; the middle window shows files and folders on a removable USB drive.

*NOTE: The controller does not run programs from the USB drive. Programs must first be copied to the controller before they can be loaded into memory and run. For more information, refer to Transferring Files or Folders from a USB Drive later in this section.*

### 5.5.1 Managing Files

Use the *New Folder*, *Rename*, and *Delete* buttons below the respective USB Drive Window and Hard Drive Window for file management (see **Figure 5.12**). To move files into a folder, right-click on the file and select cut or copy from the pop-up menu.



**Figure 5.12**

### 5.5.1.1 Transferring Files or Folders from a USB Drive

1. Insert a USB drive into an open USB port.

2. Navigate to the file to transfer in the USB drive window (see **Figure 5.12**).

*NOTE: Use Back to navigate backwards; use USB to jump to the highest (home) level (see **Figure 5.12**).*

3. In the hard drive window, navigate to the desired location in the PathPilot system to copy the transferred file from the USB drive.

4. Highlight the file or folder to copy in the USB drive window; click *Copy From USB* (see **Figure 5.12**).

5. If the file to be transferred has the same name as an existing file on the controller, you can either overwrite the file, give it a different name, or cancel the file transfer.

6. When copied to the new location, the file displays in the USB drive window.

7. Click *Eject* to disconnect the USB drive from the controller (see **Figure 5.12**).

*NOTE: Ejecting the USB drive this way helps to avoid corrupting data on the USB drive.*

### 5.5.2   Loading G-code

The *Load G-code* function is only available for files stored on the controller (see **Figure 5.12**).

1. Navigate to the desired .nc file in the hard drive window (see **Figure 5.12**); highlight the file and click *Load G-code* (see **Figure 5.12**).

2. Click on the *Main* tab; verify the G-code file name appears in recent files drop-down window. For more information on selecting a recent G-code program file, refer to *Main Tab* section earlier in this chapter.

### 5.5.3   Editing G-code

The *G-code File Preview* window displays the contents of the selected .nc file (see **Figure 5.12**). You can edit G-code in two ways on your PathPilot controller:

• Using a text editor

• Using the *Conversational* tab to edit a file created in PathPilot's conversational programming

*NOTE: For more information on using conversational programming in PathPilot, refer to Conversational Tab section later in this chapter.*

#### 5.5.3.1   Editing G-code with a Text Editor

1. Highlight the file and click the *Edit G-code* (see **Figure 5.12**).

2. A text editor opens the file in a new window for editing the contents of the file. Make the appropriate changes to the file and click *Save*.

3. Click the *X* in the upper right-hand corner of the screen to close the text editor.

4. Click *OK* when asked to reload the file.

#### 5.5.3.2   Editing G-code with Conversational Programming

To edit the G-code using conversational programming, highlight the file and click *Conv. Edit* (see **Figure 5.12**). A job assignment editor opens the file in a new window for editing the contents of the file. The left window shows job assignments of a program; the right window previews the program (see **Figure 5.13**).

Use *Move Up*, *Move Down*, *Duplicate*, or *Remove* to edit the order in which the steps of the program occur.

# PathPilot Interface
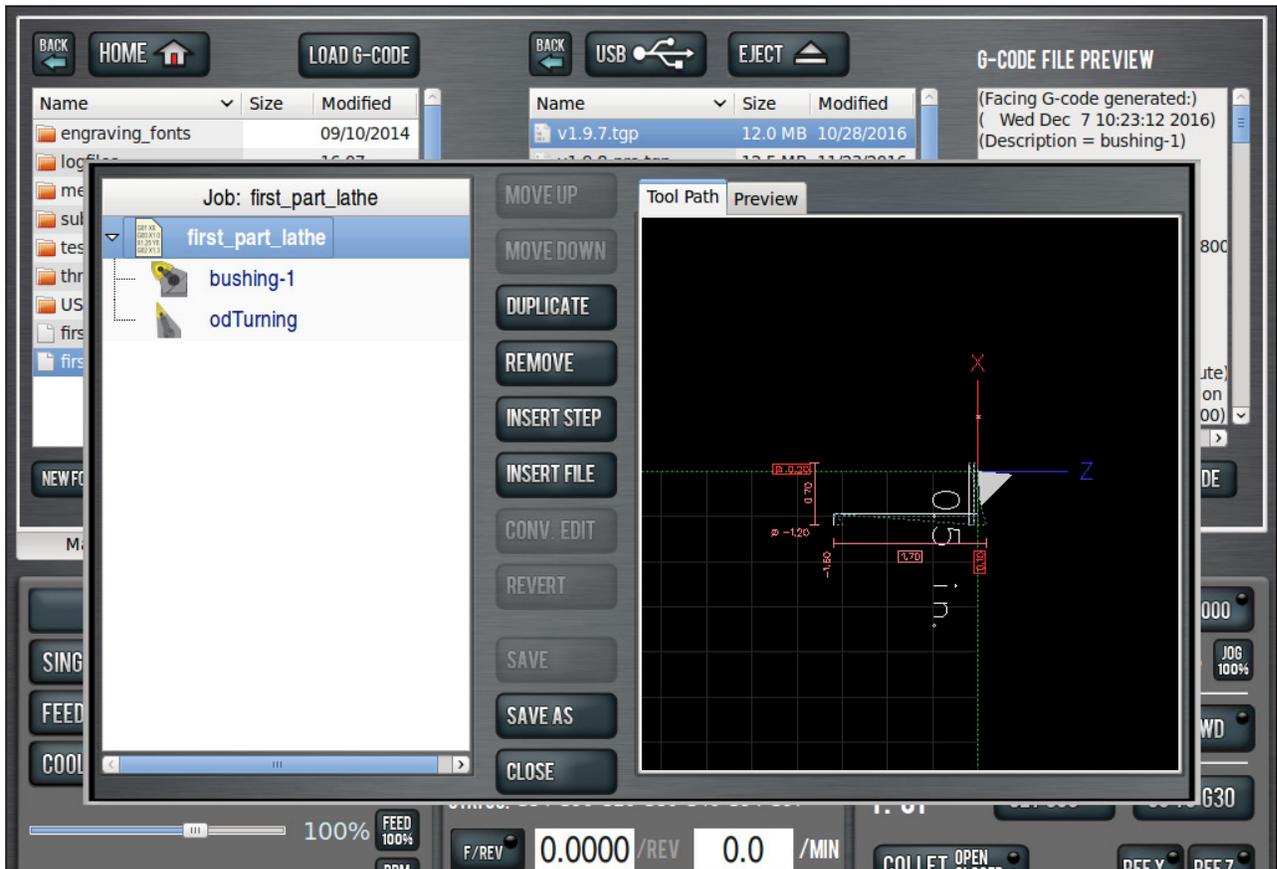


**Figure 5.13**

Click *Insert Step* to create a new job assignment using conversational programming. PathPilot opens the *Conversational* tab; after creating the job assignment, click *Insert*. If necessary, edit the order of the newly created job assignment in the program.

Click *Insert File* to load an already created G-code file into the program – allowable G-code files can be hand-written, generated from a CAM software, or generated from PathPilot's conversational programming. Navigate to the desired .nc file and highlight; click *Open*. If necessary, edit the order of the newly created job assignment in the program.

Highlight a job assignment and click *Conv. Edit* to edit a job assignment created in conversational programming. PathPilot opens the relevant tab of the *Conversational* tab; make the appropriate changes to the job and click *Finish Editing* to update the G-code program file.

Click *Revert* to restore any edited job assignment to its original parameters. *Revert* is only available for individual job assignments created in conversational programming.

Click *Close* and, when prompted, *Close Without Saving* to restore the entire G-code program file to its original parameters. PathPilot exits the conversational edit screen without overriding the original G-code program file.
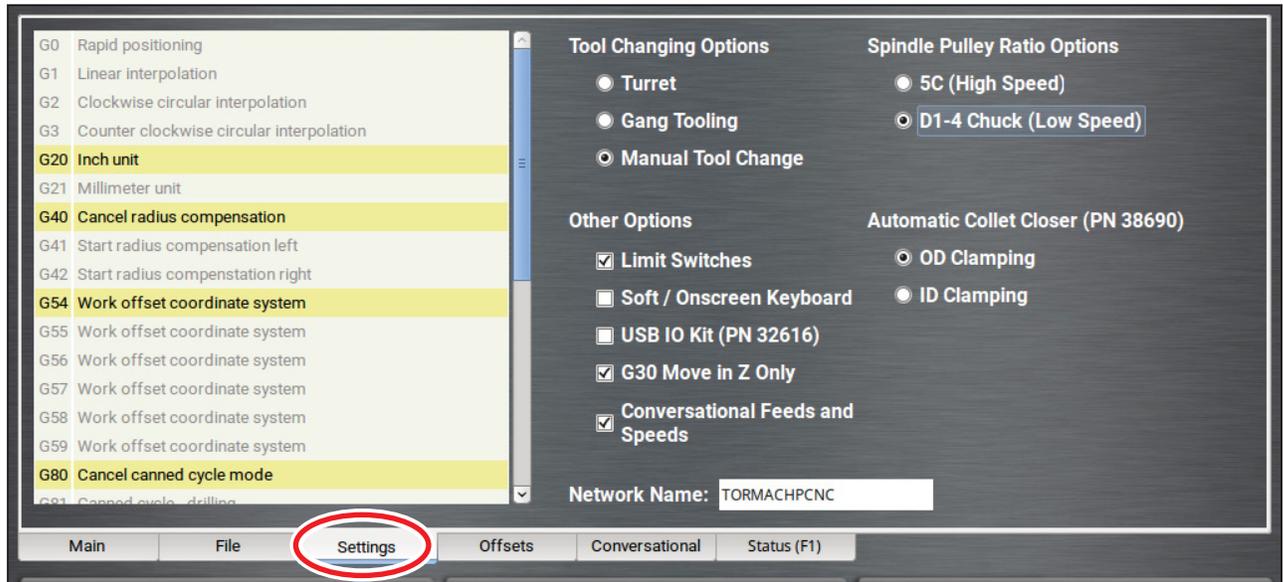
**Figure 5.14**

## 5.6 Settings Tab

The *Settings* tab displays active settings of the PathPilot controller, allowing you to configure PathPilot to suit your machine configuration (see **Figure 5.14**).

The window on the left side of the *Settings* tab displays a list of available G-code modalities. Active G-codes are highlighted in yellow (see **Figure 5.14**).

### 5.6.1 Selecting the Tool Changer Type

Select the tool changer type for specific machine configuration. Machine behavior when it encounters a T command is different depending on which tool changing option is selected (see **Figure 5.14**).

**Turret:** T commands T01 through T08 cause the turret to index (regardless of current machine position) and the offsets for the requested tool are applied. With the turret setup, tool numbers higher than eight are still available, and a T command of, for example, T09 does not cause turret rotation but does apply the offsets for tool 9. It is possible to mount a parting tool or gang of drills to the lower portion of the carriage and assign tool numbers higher than eight to these tools.

The machine does not automatically retract to G28 or G30 position before a tool change. It is the operator's responsibility to program the machine to a safe position before executing a T command. If generating code using the conversational features of the control, a G30 is inserted before each tool change command.

**Gang:** If the gang option is selected, the T command simply applies the offsets for the selected tool. Unlike the manual tool change option, T commands with the gang option selected does not pause the program during the tool change. Offsets are applied nearly instantaneously, and machining resumes automatically during part program execution.

**Manual:** The manual tool change option causes the machine to pause at the T command during G-code program execution. This allows the operator to manually change tools on a quick change tool post (QCTP). After changing tools, clicking the *Cycle Start* button resumes program execution with the new tool offsets applied. When the machine is paused waiting for a manual tool change in the middle of a G-code program, the *Cycle Start* LED blinks and the tool label flashes with the requested tool number.

**Mixed Tooling:** When mixing turret or gang tools with QCTP-mounted tools, there are two choices:

1. If the operator selects the turret or gang setting, the lathe will not automatically pause for manual changes of QCTP tools. The operator must program these manual M01 stops. This is the recommended option.

2. If the operator selects the QCTP option, (s)he will have to click the *Cycle Start* button to confirm both manual and turret/gang changes.

### 5.6.2   Enabling Feeds and Speeds Suggestions in Conversational Programming

Select *Conversational Feeds and Speeds* to enable feeds and speeds suggestions in PathPilot when using conversational programming. For more information on use, refer to *Using Feeds and Speeds Suggestions* later in this chapter.

## 5.7   Offsets Tab

The *Offsets* screen is subdivided into two tabs – *Tool Touch* and *Offsets Table* (see **Figure 5.15**). The *Tool Touch* tab provides the selection of a tool type (i.e., front tool post left hand turning tool), as well as providing controls for setting the geometry offsets for tools (or touching off). The *Offsets Table* tab has an editable table of tool geometry, wear offsets, and a read-only table of work offsets.
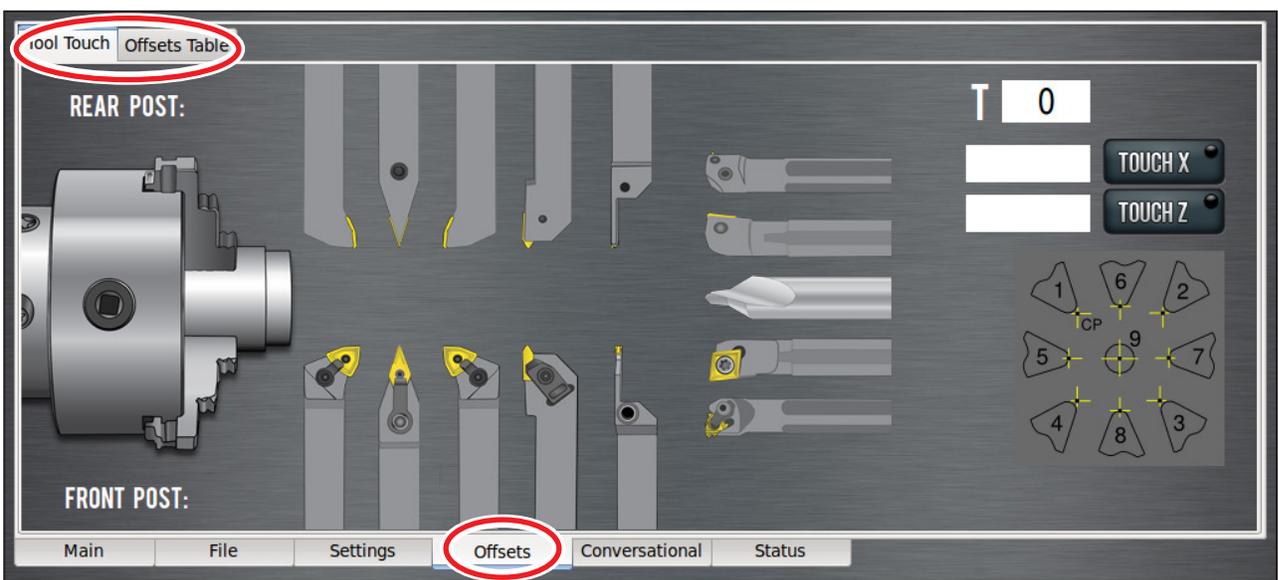


**Figure 5.15**
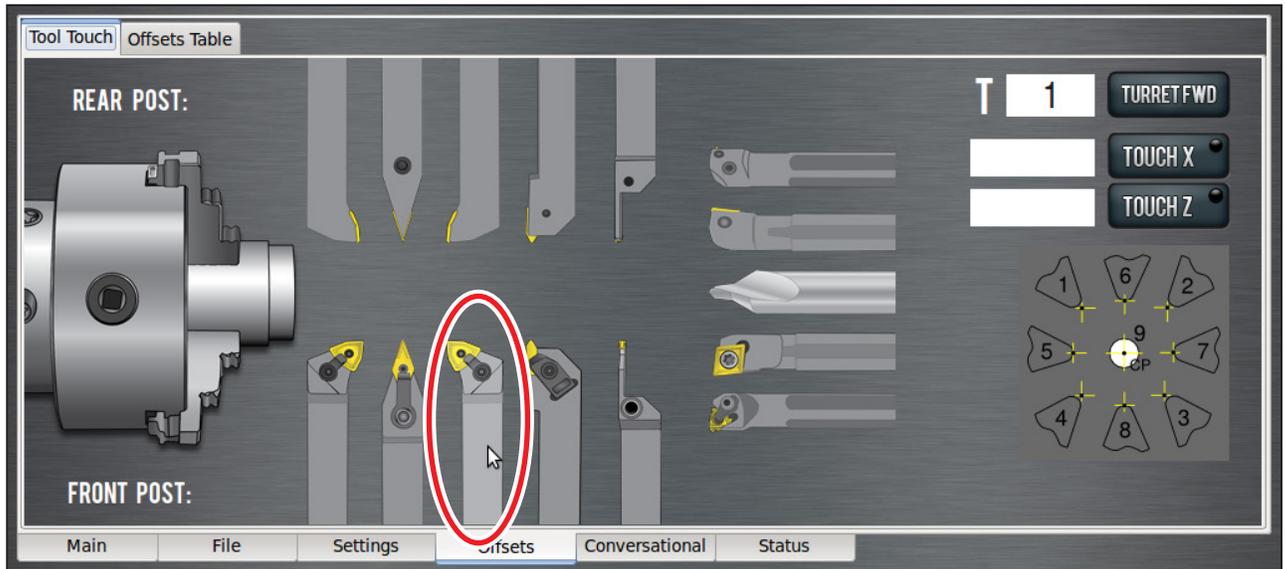
**Figure 5.16**

### 5.7.1 Tool Touch Tab

The *Tool Touch* screen allows the operator to graphically select a tool, then touch off the tool to set the geometry offsets. The tool images act as buttons, and they highlight as they are moused over. The tools along the bottom of the screen are front tool post tools and are used by machines with a quick change tool post setups. The tools along top are rear tool post tools and are used by machines with a turret setup. Gang tooling setups typically use both front and rear tool post tools. For more information on front and rear tool post locations, refer to chapter 3, *Operation*.

**Figure 5.16** shows a right-handed, front tool post turning tool highlighted as mouse hovers over it. Clicking on a tool image does a few things:

1. Sets the tip orientation for the tool, used (along with tip radius) in cutter compensation.

2. Sets the tool type (front tool post or rear tool post), used by the conversational routines to double check the user entry fields in an attempt to try to detect and prevent crashes.

3. Brings up the tool touch off dialogue. **Figure 5.17** shows the tool touch off dialogue for a front tool post right-handed turning tool.

The tip orientation has been automatically set to 3 in orientation graphic (see **Figure 5.17**). This dialogue is intended to make setting the tool geometry offsets (or touching off the tool) easy and intuitive.

#### 5.7.1.1 Touch X (Dia.)

1. Take a skim cut off of the diameter of the workpiece. This cut only needs to be long enough to measure the cut surface with a micrometer. When done, jog the tool away from workpiece in Z, but do not jog machine in X.
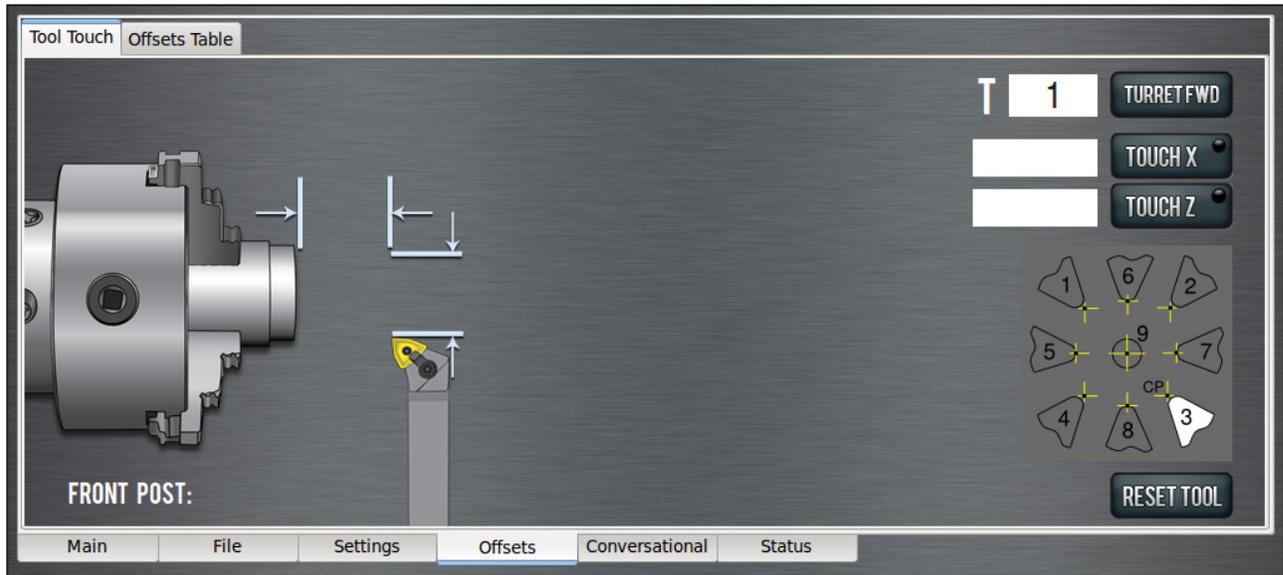
**Figure 5.17**

2. Measure the diameter of the skim-cut workpiece with a micrometer, then enter that value in the *Touch X* DRO. If touching off a front tool post tool, enter the negative of the value measured. When this value has been entered, click the *Touch X* button. Notice that the LED on that button turns green, indicating that a tool offset has been set for that tool. If touching off a front tool post tool using a positive X value, PathPilot accepts the value, but warns that something is potentially incorrect. Front tool post tools should be touched off with negative X values.

### 5.7.1.2   Touch Z

1. Jog the machine toward the part zero (usually the face of the workpiece) in the Z direction. Move the tool so the cutting edge is just touching the surface of the material and define this as Z = 0. For now, use a thin sheet of regular paper.

2. Jog the Z-axis to approximately 1/4" away from part zero on the workpiece. If using keyboard jogging, change the slow jog percent DRO from 10 percent to 2 percent. The jog shuttle allows control of speed depending on how far the ring is turned.

3. With the paper between the tool and the workpiece, carefully jog until the paper just gets trapped by the tool. Do not ram the tool into the material; just touch the paper.

4. Enter the paper thickness in the DRO next to the *Touch Z* button (typically 0.003") and click the *Touch Z* button.

5. One thing to consider when using these methods to measure your tool geometry is that while X of zero never changes (the spindle centerline is always X=0), the Z zero location may change depending on the length of the workpiece that is chucked into the spindle. As long as each tool is measured to a face that has been zeroed, only measure these tools one time or until an insert replacement is needed.

### 5.7.2 Offsets Table Tab

The *Offsets Table* tab of the *Offsets* screen displays an editable table of tool offsets (both geometry offsets and wear offsets) as well as a read-only table of work offsets (see **Figure 5.18**). To alter a value, double-click on the field; click *Enter* on the keyboard to accept the value.

*NOTE: The lathe must be powered on and out of Reset to edit these fields.*

#### 5.7.2.1 Creating Tool Descriptions

PathPilot uses keywords and patterns in the tool description to recognize tooling features. Refer to the section *Using Tool Keywords* later in this chapter for more information.
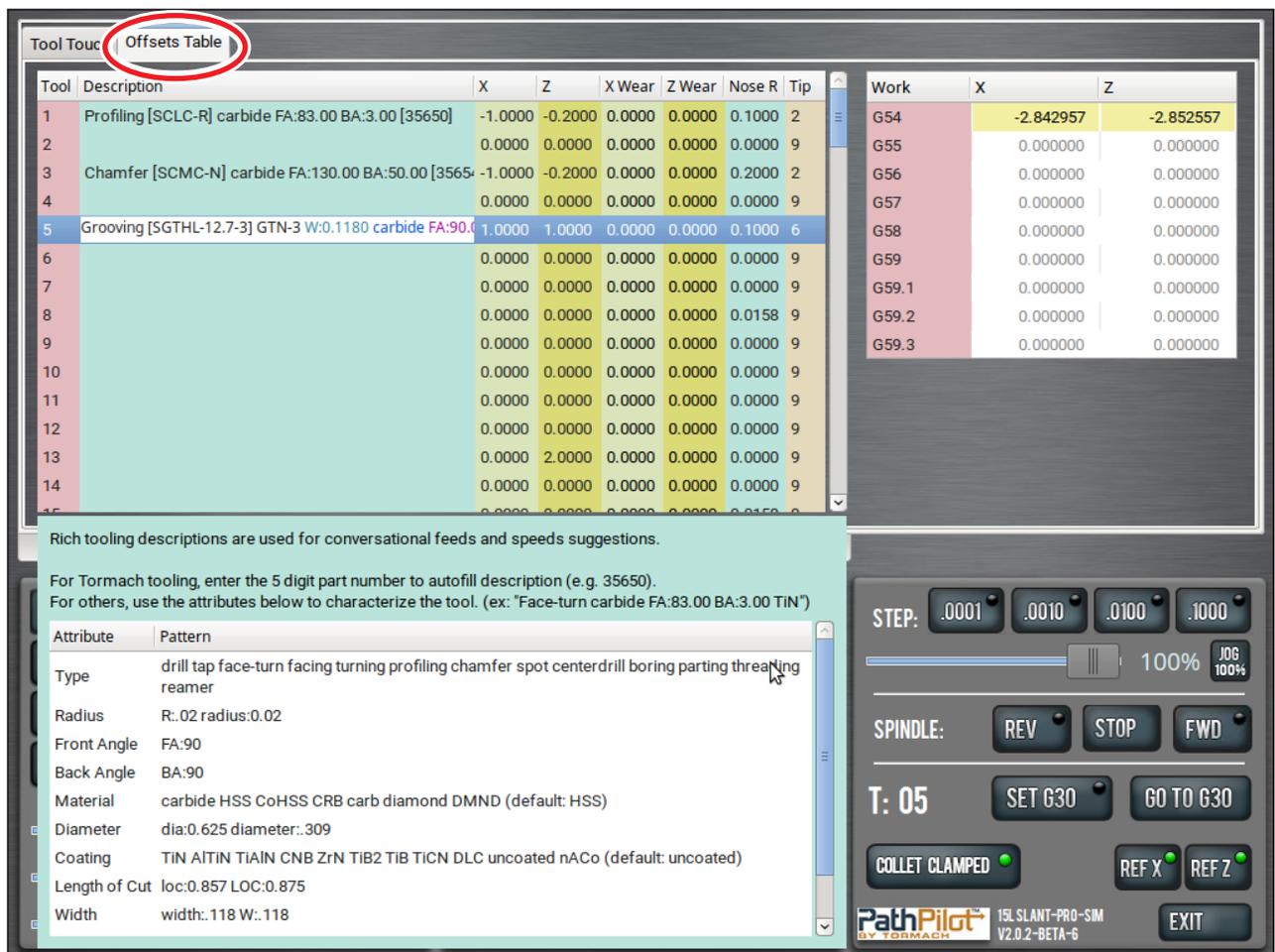


**Figure 5.18**

Example: "MVJN-R12-3B carbide AlTiN CRB FA:87.0 BA:52.0 finishing"

This description provides excellent information for PathPilot to calculate machining information:

- MVJN holder geometry
- Aluminum-titanium nitride coated carbide (AlTiN)
- Carbide
- 87° front angle
- 52° back angle

To get accurate machining information, all tooling must be described with detail: the more detail, the better results. Using a personal description likely won't contain meaningful information for PathPilot.

Example: "Gold colored tool bit from middle drawer"

This description provides virtually no information. PathPilot will default to tool bit features from the setup on the Tool Touch tab.

**To Manually Enter Tool Descriptions**

You can manually enter tool descriptions in the Offsets Table window. Descriptions are not case sensitive. On the lathe, tools are divided into two groups: Radial tools (which include all single-point turning tools), and axial tools (like drills, taps, and reamers).

If a pattern or word in the description is recognized, PathPilot uses syntax highlighting to indicate a valid description.

Refer to *Using Tool Keywords* section later in this chapter for more information.

**Using Tool Keywords**

| Item | Pattern | Example | Notes |
|---|---|---|---|
| type | drill, tap, face-turn, facing, turning, profiling, chamfer, spot, centerdrill, boring, parting, threading, reamer | • Drill<br>• Chamfer<br>• Boring | |
| tool radius | "R" or "radius", followed by a colon, followed by a decimal number | • R:.0158<br>• radius:0.32 | No radius specified is the same as a zero radius. |
| Front angle | "FA", followed by a colon, followed by a decimal number | • FA:90 | |

| Item | Pattern | Example | Notes |
|---|---|---|---|
| Back angle | "BA", followed by a colon, followed by a decimal number | • BA:90 | |
| tool material | carbide, HSS, CoHSS, CRB, carb, diamond, DMND | • HSS<br>• CoHSS<br>• crb | HSS: High-speed steel<br>CoHSS: Cobalt high-speed steel<br>CRB: Carbide<br>No tool material specified is the same as "HSS". |
| tool diameter | "dia" or "diameter", followed by a colon, followed by a decimal number | • Dia:.0341<br>• diameter:.750 | For axial tools (drills taps, reamers, etc.). |
| tool coating | TiN, AlTiN, TiAlN, CNB, ZrN, TiB2, TiB, TiCN, DLC, uncoated, nACo | • TiN<br>• ZrN<br>• TiB2 | No coating specified is the same as "uncoated" |
| length of cut | "loc", followed by a colon, followed by a decimal number | • loc:0.875 | For axial tools (drills, reamers, etc.). If no length of cut is specified, a length will be assumed based on cutter diameter. |
| tool width | "width" or "w", followed by a colon, followed by a decimal number | • W:.118<br>• width:.118 | For grooving and parting tools |
| min bore | "min-bore", followed by a colon, followed by a decimal number | • min-bore:.522 | |
| flutes | A number, followed by "FL" or "FLUTE" | • 4FL<br>• 12FL<br>• 2flute | No flutes specified is the same as 2 flutes |
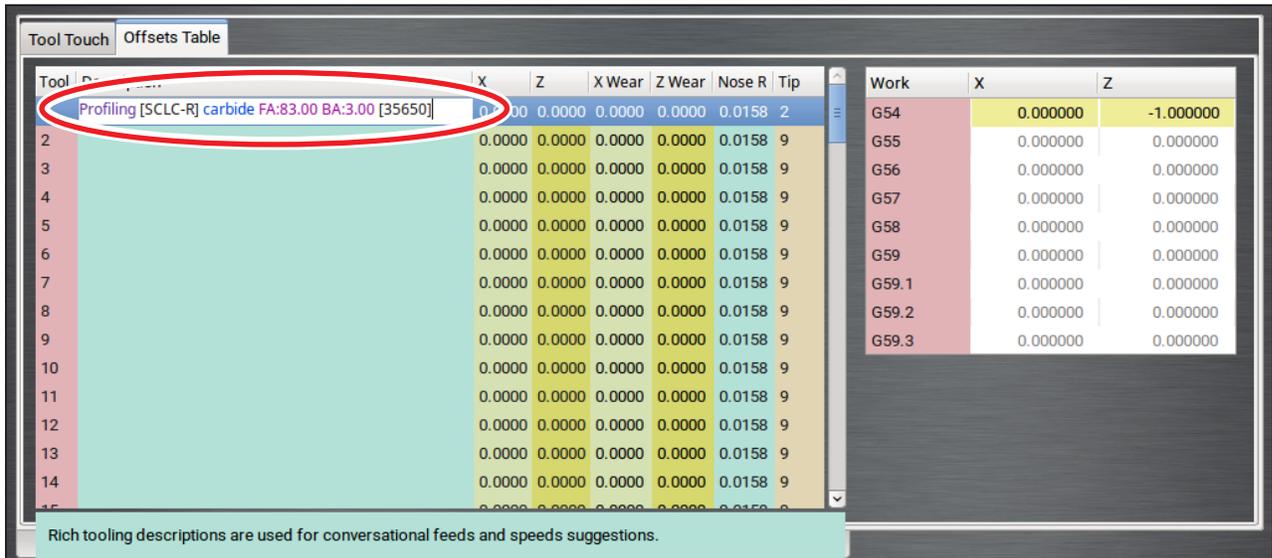
**Figure 5.19**

### To Generate Automatic Tool Descriptions

If you are using a Tormach tool, you can enter the part number to automatically generate tool descriptions in the Offsets Table window.

*NOTE: If you're unsure of the part number, you can search for the tool at tormach.com.*

1. From the PathPilot interface, on the Offsets tab, in the Offsets Table window, select a blank line.

2. Type the part number for the tool.

   Example: "33133"

   A full description for a right-handed profiling tool (PN 33133) displays.

3. You must use the Tool Touch window to zero the tool for the correct X offset and Y offset.

### 5.7.3   Tool Offset Information Backup

Make a periodic backup of the tool offset information and machine settings to store externally should the controller get replaced or need to be restored to factory settings.

Create a tool offset information backup on a PathPilot controller as follows:

1. Insert a USB drive into any open USB slot on the controller.

2. On the *Main* screen, type *ADMIN SETTINGS BACKUP* in the MDI line.

3. In the dialog box, navigate to a location to store the backup .zip file on the USB drive and rename if desired; click Save.

*NOTE: Keep this file somewhere safe and easily accessible.*

Restore tool offset information backup on a PathPilot controller as follows:

1. Transfer the tool offset information and machine settings backup to a USB drive; insert into any open USB slot on the controller.

2. On the *Main* screen, type *ADMIN SETTINGS RESTORE* in the MDI line.

3. In the dialog box, navigate to the backup .zip file on the USB drive; click *Open*. PathPilot exits, restores from the backup file, and then restarts PathPilot.

## 5.8   Conversational Screens

### 5.8.1   Overview

The included G-code generators are intended for making G-code programs for simple parts or parts made up from a collection of simple features (for more information, refer to chapter 4, *Intro to PathPilot*). For complex parts or parts with complex shapes, using CAD/CAM may be better.

*Conversational* screens are divided into two sections: parameters common to most operations on the left and operation-specific parameters (including part geometry) on the right. DRO fields which are not used for a particular generator do not allow editing. Each generator produces G-code with common elements:

- Comments describing the file and parameter values entered
- G-code for setting up lathe
- G30 to move to tool change position
- Tool change with wear offsets applied
- Coolant and spindle on
- Move to workpiece starting point, usually starting somewhere on the tool clearance zone
- Roughing as needed
- Finishing as needed
- Coolant and spindle off
- G30 to park tool
- M30 to end program

During roughing and finishing, the tool clearance zone is often used as a landmark for path direction and feed/rapid rate changes, so the tool clearance value may need to be adjusted to compromise between different requirements, such as clearance from both the stock OD and an uneven face. At the end of finishing, the tool is usually returned to the workpiece starting position.

When appending to an existing file, the M30 in the existing file is removed to allow the appended M30 to end the program. All other G-code for each appended generator remains untouched. The tool always goes to tool change position between routines, even if there is no tool change required.

**Tool Table and Work Offset (User Coordinates)**

The *Work Offset* DRO field is used to select the work offset coordinate system to be used when the program runs.

**NOTE:** *The tool table and work offset need to be set up before the DRO fields are filled in; the data is the same when the G-code program is loaded and run.*

**Adjusted DOCs**

Usually a value for a depth of cut (DOC) is set by considering the tool configuration and workpiece material properties. If this value is used directly across a cutting range, such as for roughing from a stock diameter to a rough OD, it is likely there is a fraction of the depth of cut left for the last cut. Where appropriate, an adjusted value is determined by dividing the range by the entered DOC to find the integer number of cuts that fit within the range. One is added to make sure at least one cut is made, and also that the adjusted value is always less than the requested depth. For example:

(Xend – Xstart) / DOC = n, the positive integer from n = N, DOCadj = (Xend – Xstart) / (N + 1)

Some tool and material conditions may need a minimum depth of cut, but because the adjusted value is less than the requested value, it is wise to check the adjusted value in the resulting G-code file to make sure it meets a minimum if needed. If running the lathe in the conversational programming mode, enter operational parameters for the specific part geometry on the appropriate tab: *OD Turn*, *ID Turn*, *Profile*, *Face*, *Chamfer*, *Groove/Part*, *Drill/Tap*, or *Thread*.

**NOTE:** *If the ENTER key is pressed, the cursor moves to the next text field on the screen. Pressing ENTER rather than clicking through the fields using a mouse reduces the likelihood of inadvertently typing an invalid value and confirms the value is established within that field.*

After entering the appropriate operational and part-parameters, click *Post to File* or *Append to File*, depending on workflow.

**Post to File**

1. The *Post to File* window opens on the screen.

2. If the appropriate file name is not displayed in the *Name:* field, click the correct file; verify it is displayed.

3. Click *Save* or *Cancel*. If *Save* is selected, a message confirming the save is displayed.

**Common DRO Fields**

Common DRO fields – parameters that are most likely to be reused by operations to make a part – are on the left side of the programming screen (see **Figure 5.20**). Some operations do not use a common DRO field, in which case it is grayed out.
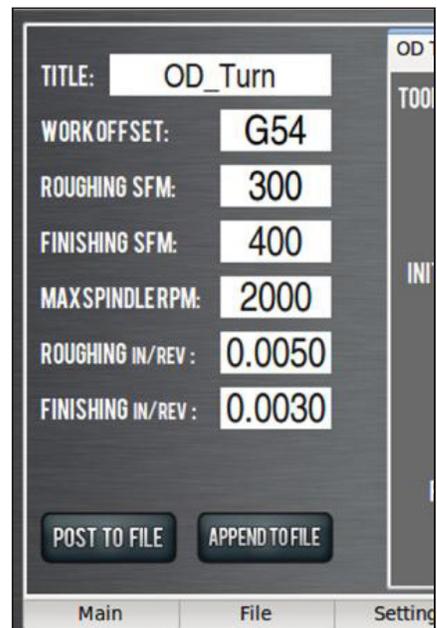


**Figure 5.20**

**Title:** This is an optional entry to allow the operator to prefix the automatic date stamp that appears when *Post to File* is used.

**Work Offset:** Sets which coordinate system to use. The available offsets are:

- Machine coordinates (G53)
- Six work offsets (G54-G59)

**Roughing SFM or MPM:** This sets the constant surface speed value (not RPM) for the roughing section of the G-code. Usually set to optimize the rate of material removal. When this is grayed out, a DRO field should be available to set an RPM rather than the constant surface speed input.

**Finishing SFM or MPM:** This sets the value (or constant surface speed), not RPM for the finishing section of the G-code. This is usually set to optimize surface finish. When this is grayed out, a DRO field should be available to set RPM.

**Max. Spindle RPM:** Constant surface speed varies spindle speed dependent on the tool's X position. As X decreases the RPM increases, but at some point the RPM needs to be limited by the maximum RPM that the workpiece, work holder, spindle, or tooling can handle. The operator needs to consider which feature has the lowest RPM limit and enter this value. When this is grayed out, a DRO field should be available to set a simple spindle RPM.

**Roughing IN/Rev. or MM/Rev:** This sets the value for feed or units per spindle revolution for the roughing section of the G-code. When this is grayed out, a DRO field should be available to set a simple feed per minute.

**Finishing IN/Rev. or MM/Rev:** This sets the value for feed or units per spindle revolution for the finishing section of the G-code. When this is grayed out, a DRO field should be available to set a simple feed per minute.
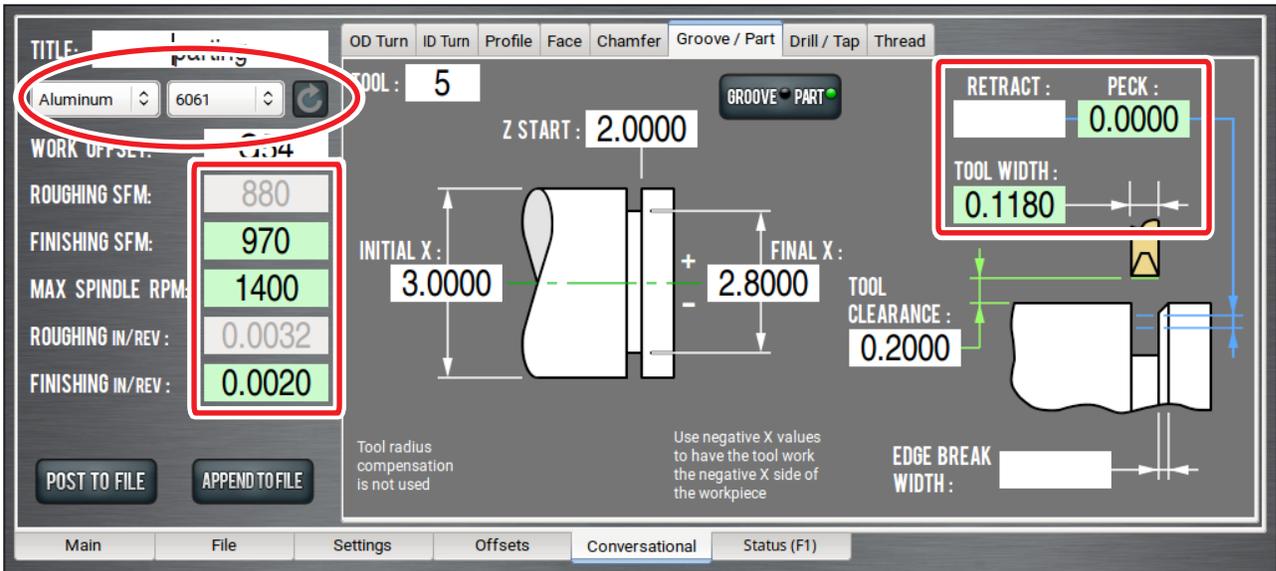
**Figure 5.21**

### 5.8.2 Using Feeds and Speeds Suggestions

You can use PathPilot to automatically calculate feeds and speeds: from the *Conversational* tab, in the Conversational DROs group, select a material, a sub-type, and a tool. Then, PathPilot automatically loads all machining-related DROs (see **Figure 5.21**).

1. From the PathPilot interface, on the *Conversational* tab, locate the *Material* dropdowns in the Conversational DROs group.

2. From the *Material* dropdown, select your material.

   Examples:

   - Aluminum
   - Plastic

3. If required, from the *Sub-type* dropdown, select the material sub-type.

   Examples:

   - -any-
   - 6061

4. In the *Tool* DRO, type the assigned tool number.

5. Click *Refresh* (to the right of the *Sub-type* dropdown).

   The following machining-related DRO fields are calculated:

   - *Roughing SFM*
   - *Finishing SFM*
   - *Max Spindle RPM*
   - *Roughing (in/rev)*

- *Finishing (in/rev)*
- *Finishing DOC*
- *Roughing DOC*
- *Peck (for drilling)*

**NOTE:** *After PathPilot calculates values for the machining-related DROs, the background turns green (see* **Figure 5.21**).

### 5.8.2.1   Adjusting DRO Values

After selecting the material and tool, you can adjust the values in the calculated DROs, like *Feedrate* or *Stepover*. Adjusting the value in one of these DROs does not change the value in the other machining-related DROs.

Once you adjust the value in the DRO, the background switches from green back to white (see **Figure 5.22**). This helps you identify which DROs have suggested values (those with a green background), and which DROs have values you've supplied (white background).
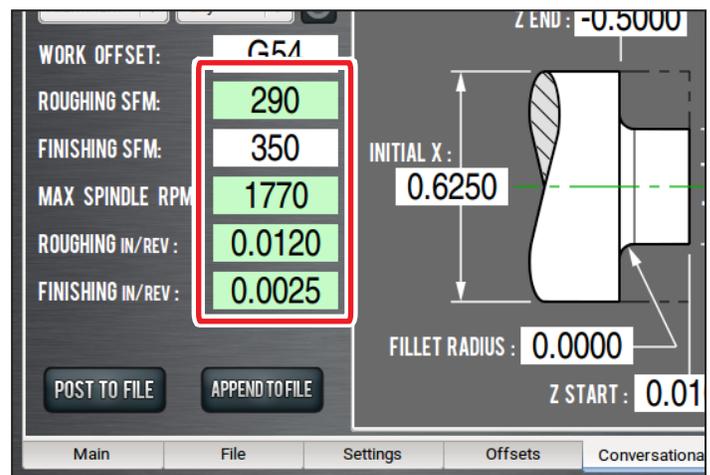


Figure 5.22

### 5.8.2.2   Refreshing DRO Values

The suggested feeds and speeds are no longer valid if:

- You select different material or sub-type values, or if you type a new value in to the *Tool* DRO.

  The suggested feeds and speeds are made by taking into account all of these values. Changing any value requires you to refresh.

- You select a different *Conversational* tab.



Figure 5.23

  The suggested feeds and speeds are made by taking into account the current, specific conversational operation — like Face, or OD Turn. Changing your conversational operation requires you to refresh.
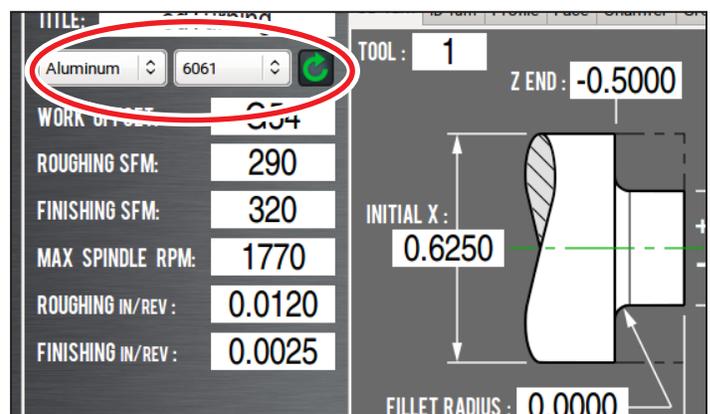
When the feeds and speeds are no longer valid, the *Refresh* button turns green, and the machining-related DRO backgrounds switch from green to white (see **Figure 5.23**).
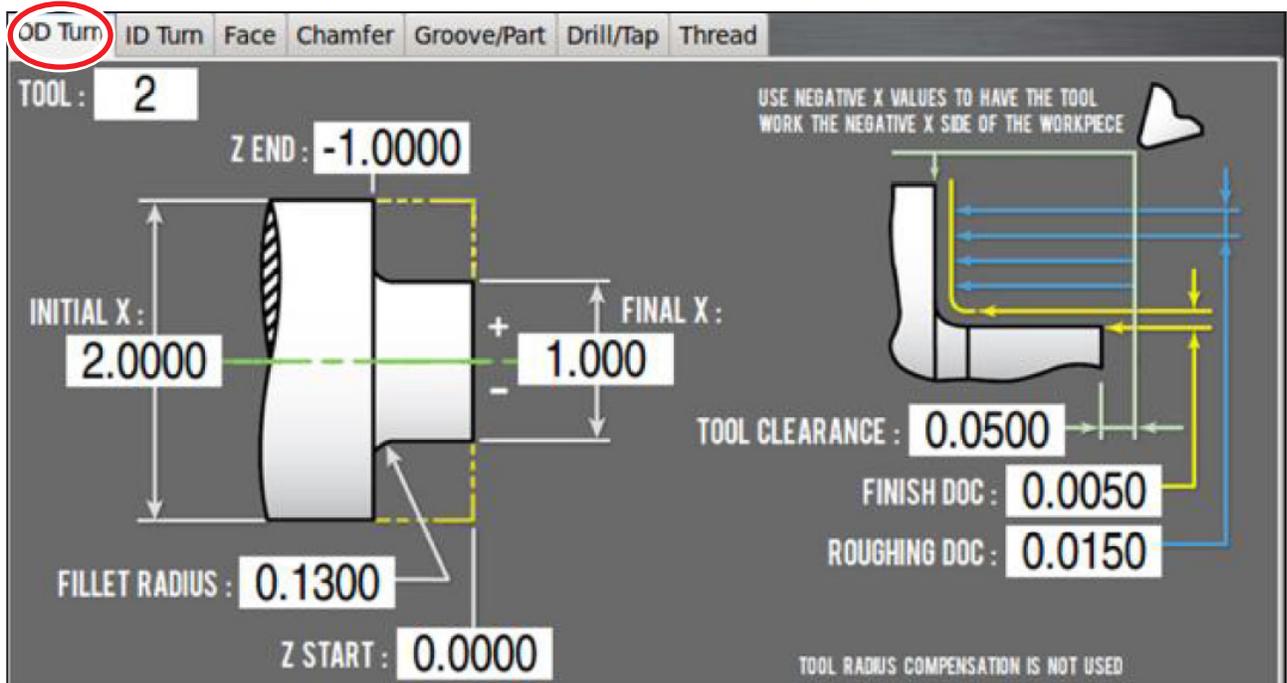
**Figure 5.24**

### 5.8.3   OD Turn Tab

OD Turn creates G-code to rough and finish three features; an outside diameter, a fillet (corner radius), and an adjacent face (see **Figure 5.24** and **Figure 5.25**). The face is always on the headstock end of the diameter being cut. Cutter Radius Compensation (CRC) is not used, so the fillet does not follow a true radius for cutters having a finite tip radius.



**Figure 5.25**

Roughing starts at the *Initial X* diameter and incrementally cuts diameters at an adjusted depth of cut (for more information, refer to section *Adjusted DOCs* earlier in this chapter) using *Roughing DOC* until the start of the finish diameter [FINAL X + ( 2 * FINISH DOC)]. Finishing is done in one pass at the value entered into the *Finish DOC* DRO field.

The finish pass is started at the +Z end of the OD and feeds to the middle of the fillet. The tool is then retracted to the stock diameter, and the face finish pass is cut from the stock diameter to the end of the fillet. Since there is only one finish pass, the *Finish DOC* is not adjusted. The tool is cutting both an OD and face, so valid tools are limited to orientation 2 for a rear tool, and type 3 for a front tool.

The *Tool* DRO field is a command value (meaning that it sets the location of a new feature) that sets the tool number for a tool change at the start of the program.

The *Initial X* DRO field is a reference value (the value should describe an existing feature), usually set to the stock diameter. It is also used with *Tool Clearance* DRO field to locate some of the transitions between Rapid and Feed rate. If the values in the *Initial X* DRO field and the *Final X* DRO field are both positive, the tool will work on the positive X side of the spindle center, the side away from the operator. If they are both negative, the tool works the negative side of the spindle which is the side closer to the operator. A positive and negative value for each produces an error.

The *Final X* DRO field is a command value. It sets the location of the new outside diameter.

The *Z Start* DRO field is a reference value for the stock face location. It is used with *Tool Clearance* DRO field to set the transition between Rapid and Feed rate on some Z moves. The value entered in the *Z Start* DRO field should be larger than the value in the *Z End* DRO field.

The *Z End* DRO field is a command value that sets the face location. The value entered in the *Z End* DRO field should be smaller than the value in the *Z Start* DRO field.

The *Fillet Radius* DRO field is a command value that sets the radius between the new outside diameter and the face. The fillet calculation does not use CRC, so the middle of the fillet is not on the true radius for a tool with a tip radius. Valid values are greater than or equal to zero. A value less than the tip radius is allowed and drives the cutter to the corner. A value larger than the Z range (Z START - Z END) or the X range [(INITIAL X - FINAL X) / 2] are valid, but will have a fillet start or end outside of the stock perimeter and not end at the X and Z locations entered.

The *Tool Clearance* DRO field is a reference value that should match the desired space beyond the stock outside diameter and face needed for rapid movements to clear the workpiece. Since there is one value used for X and Z moves, the value should be set to the greater of the two clearances. Larger values may be safer; smaller values may save time once the operator is familiar with how the program works. A typical move would be a rapid move from G28 to the start of the first cut, which would be at Z START + TOOL CLEARANCE, then the rate would change to a cutting feed rate. The *Tool Clearance* DRO field is also sometimes used as a location for retracting the tool while making cutting passes.

The *Roughing DOC* DRO field is a command value that sets the depth of material being cut. This depth of cut is adjusted (for further information, refer to the *Adjusted DOCs* section) to get the value used in the G-code. Valid values are positive.

The *Finish DOC* DRO field is a command value used to leave enough material after roughing to do one finish pass. Valid values are positive.

# PathPilot Interface
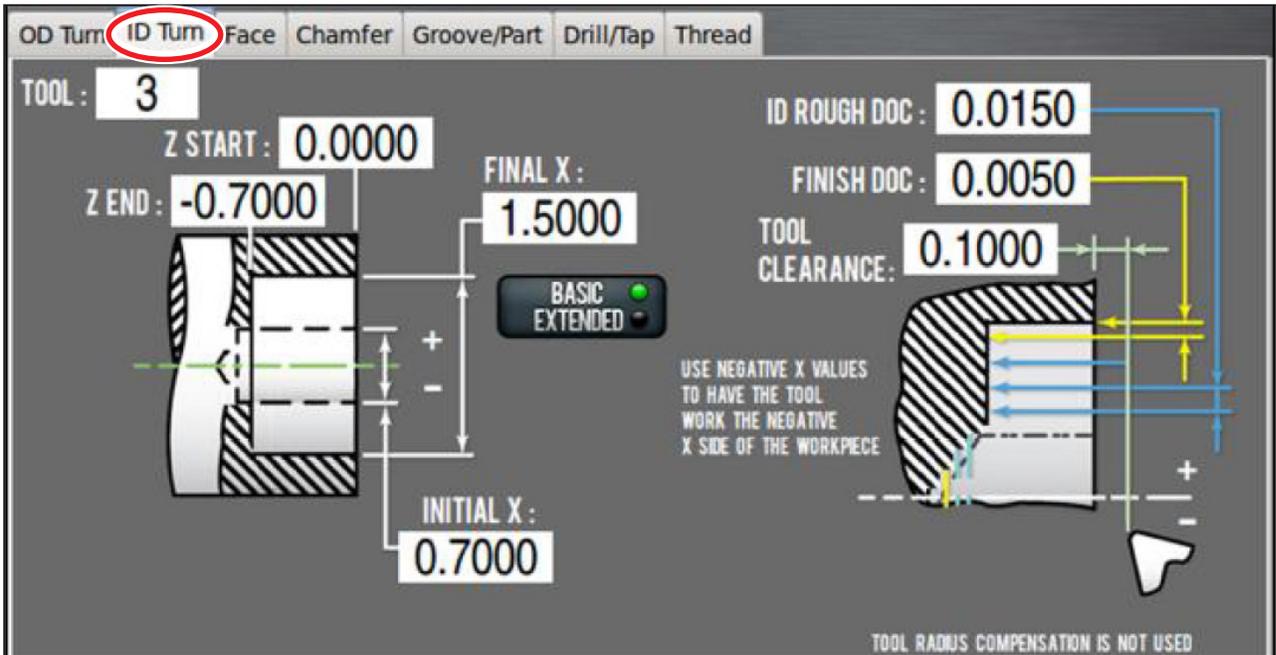


**Figure 5.26**

### 5.8.4 ID Turn Tab

ID Turn creates G-code that cuts an internal diameter. There are two varieties available: Basic and Extended. The *ID Turn* tab initially opens with a screen for *Basic* (see **Figure 5.26**). A button is provided to switch to *Extended* (see **Figure 5.27**).

Both versions use CSS for spindle speed control, and FPR for feed control. The fillet does not use CRC so the fillet will not follow a true radius for tools with a tip radius.

The *Basic* version has one operation which roughs and finishes from an initial pilot hole diameter to a final internal diameter without cutting a face at the bottom of hole. This should be used with through holes or holes that don't need a finished face. Each pass ends at *Z End*.

Roughing starts at the pilot hole diameter, *X Start*, and incrementally cuts diameters with an adjusted DOC (for further information, refer to section *Adjusted DOCs* earlier in this chapter) until the start of the finish diameter (X END - ( 2 * FINISH DOC)). Finishing is done in one pass.

The *Extended* version has three operations: ID roughing, face roughing, and an ID, fillet, and face finish pass. The extended ID roughing passes stop at the bottom of the pilot hole in order to prevent engaging too much of the tool cutting edge. Once the rough ID is cut to the pilot hole bottom, rough facing is started. Depending on the tool geometry, ID roughing and face roughing may need different DOCs so a separate facing DOC DRO field is provided.

For each face pass, the tool tip cuts to the hole center + *Tool Clearance* which requires a rough hole diameter (which was cut in the first operation) that is a little more than twice the tool's X width. Caution is needed to prevent hitting the back of the tool holder on the side of the hole.
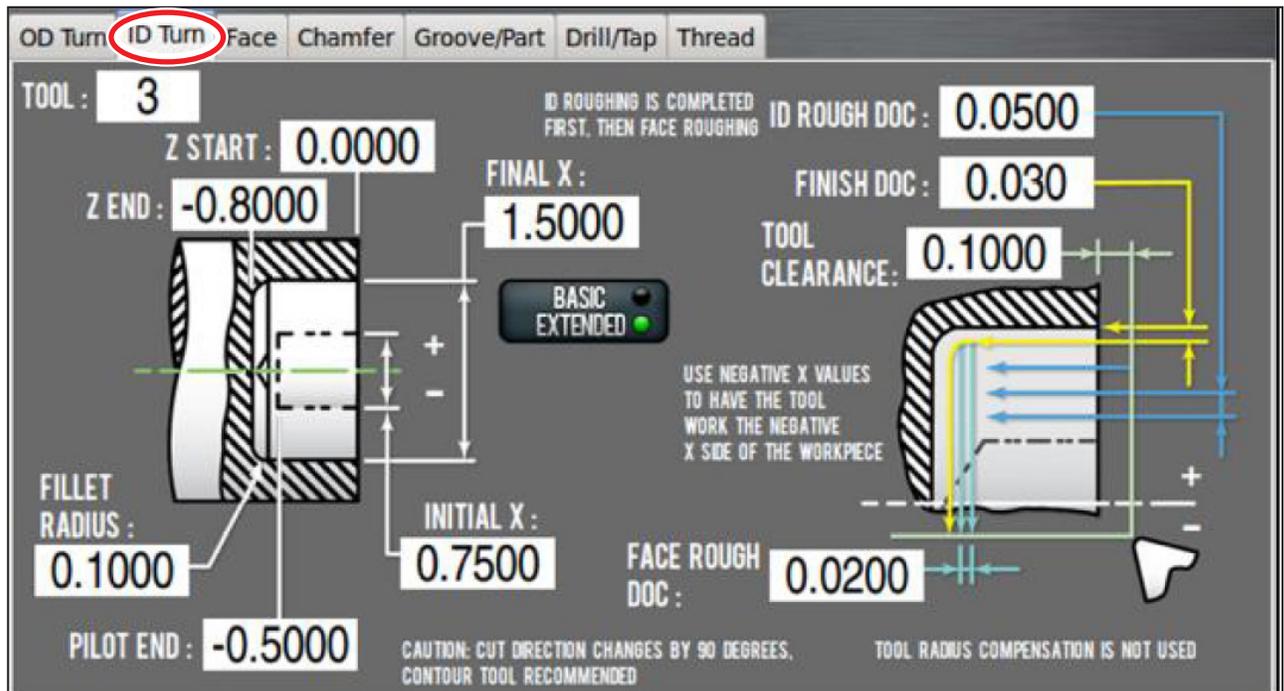
**Figure 5.27**

Valid tool orientations are limited to type 2 for a front tool, and type 3 for a rear tool. Other orientation types will create an error alarm on selecting the *Post To File* button, so it is best to have the proper tool information set in the tool table before starting. The tool path also changes by 90° on the same side of the tool, so a form tool (narrow tip angle) and separate roughing DOCs are needed.

The *Tool* DRO field is a command value that sets the tool number for a tool change at start of the program.

The *Initial X* DRO field  **i**s a reference value (meaning that the value should describe an existing feature). This value should be set to the pilot hole diameter which needs to be large enough to clear the tool holder X width.

The *Final X* DRO field is a command value (meaning that it will set the location of a new feature). It sets the location of the new finished inside diameter. For *Extended*, this value needs to be greater than twice the tool holder's X width plus tool clearance.

The *Z Start* DRO field **i**s a reference value for the location of the stock face. It is used with the *Tool Clearance* DRO field to set the transition between rapid and feed rates on some moves.

The *Z End* DRO field Is a command value for the finished face location.

The *Fillet Radius* DRO field Is a command value that sets the radius between the finished inside diameter and face. The fillet calculation does not use CRC, so the middle of the fillet may not be on the true radius for a tool with a tip radius. Valid values are 0 or positive. Values larger than the Z range (Z START - Z END) or the X range ((INITIAL X - FINAL X) / 2) are valid, but will have a fillet start or end short of the finish locations, which may not be practical.

**ID Turn (Basic) Tool Path**



Figure 5.28

**ID Turn (Extended) Tool Path**



Figure 5.29

The *Tool Clearance* DRO field is a reference value that should match the desired space needed for tool retracting and transitions between rapid and cutting feed rate. Since there is one value used for X and Z moves, the value should be set to the greater of the two clearances. Larger values may be safer, but brings the back of the tool holder closer to the ID wall on the end of facing cuts; smaller values may save time once the operator is familiar with how the program works.

The *ID Rough DOC* DRO field is a command value that sets the depth of material being cut. This depth of cut is adjusted (for further information, refer to section *Adjusted DOCs* earlier in this chapter) to get the value used in the G-code. Valid values are positive.

The *Face Rough DOC* DRO field is a command value that sets the depth of material being cut. This depth of cut is adjusted (for further information, refer to section *Adjusted DOCs* earlier in this chapter) to get the value used in the G-code. Valid values are positive. Note that the reverse or back cutting direction is sensitive to DOC. Form tools with a small angle between cutting edges allows for a larger DOC.

The *Finish DOC* DRO field is a command value that is used to leave enough material after roughing to do one finish pass on the ID, fillet, and face. Valid values are positive.

### 5.8.5   Profile Tab

Use the *Profile* tab to create G-code that roughs and finishes an arbitrary internal or external profile on a part (see **Figure 5.30**). The profile is created from a list of points that describes the part geometry. A profile can have both forward (away from the headstock) and rear (toward the headstock) facing features, and can also start behind the highest Z plane.
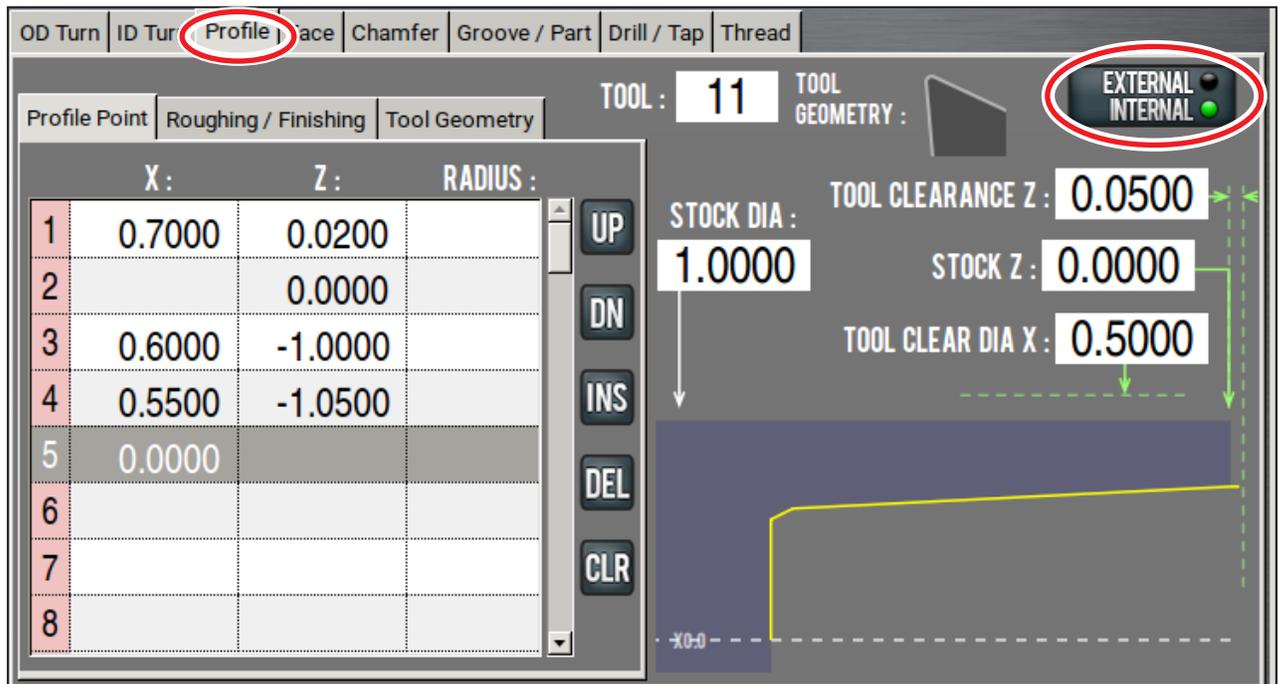
**Figure 5.30**

On the *Profile* tab, you can also specify things like:

- Tool geometry
- Feeds and speeds
- Finish depth
- Number of finish passes
- Roughing depth of cut (roughing DOC)

The *Profile* tab initially opens with a screen for *External*. To switch to *Internal*, click the *External/ Internal* button (see **Figure 5.30**).

To create an internal profile, you must make sure that:

- The *Tool Clear Dia X* DRO field has a value of smaller diameter than first X value on the Profile Point table.
- An internal tool is indicated in the *Tool* DRO field.
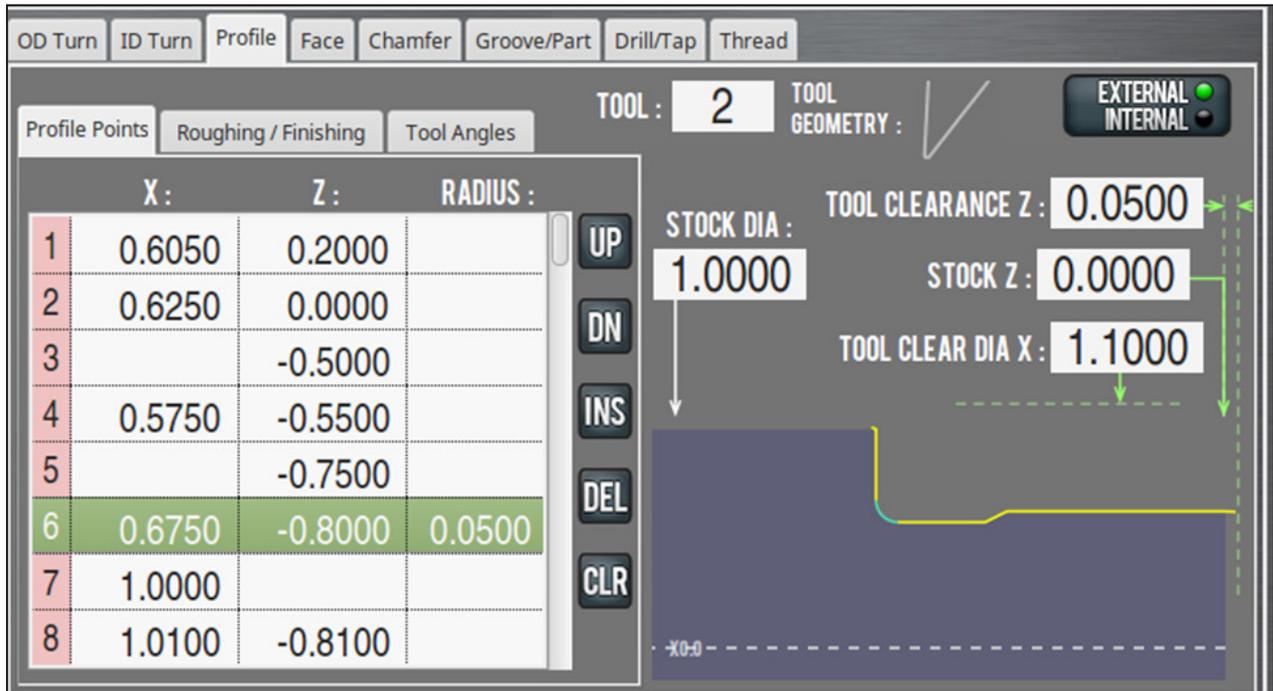
**Figure 5.31**

## Describe the Stock

Use the DRO fields on the right side of the Notebook to describe the stock for which you want to create a profile (see **Figure 5.31**).

1. From the PathPilot interface, on the *Profile* tab, in the *Stock Dia* DRO field, type the diameter of the stock. Then, press **ENTER** on the keyboard.

2. In the *Tool Clearance Z* DRO field, type the Z value for the tool clearance on the Z-axis. Then, press **ENTER** on the keyboard.

   *Tool Clearance Z* is the Z plane the tool goes from rapid to feed.

3. In the *Stock Z* DRO field, type the starting Z value for the profile. Then, press **ENTER** on the keyboard.

4. In the *Tool Clear Dia X* DRO field, type the X value – as a diameter – for the tool clearance on the X-axis.

   • For external profiles: You must make sure the value typed in the *Tool Clear Dia X* DRO field is a larger diameter than the value typed in the *Stock Dia* DRO field.

   • For internal profiles: You must make sure the value typed in the *Tool Clear Dia X* DRO field is a smaller diameter than the first diameter (specified in the *Profile Point* table).

**Identify the Profile Points**

Use the *Profile Point* table to describe the point-to-point values of a profile – from a larger Z value to a smaller Z value.

1. From the PathPilot interface, on the *Profile* tab, click the *Profile Point* tab.

2. In the *Profile Point* table, type the X and Z values for the profile. X values are in diameters terms. The X field is expressed in terms of X+ cutting (X- G-code will properly be generated if the tool selection calls for it).

**NOTE:** *If the value is unchanged from the previous row in the Profile Point table, PathPilot assumes the value is repeated. If you are using the same value, you can leave the cell empty.*
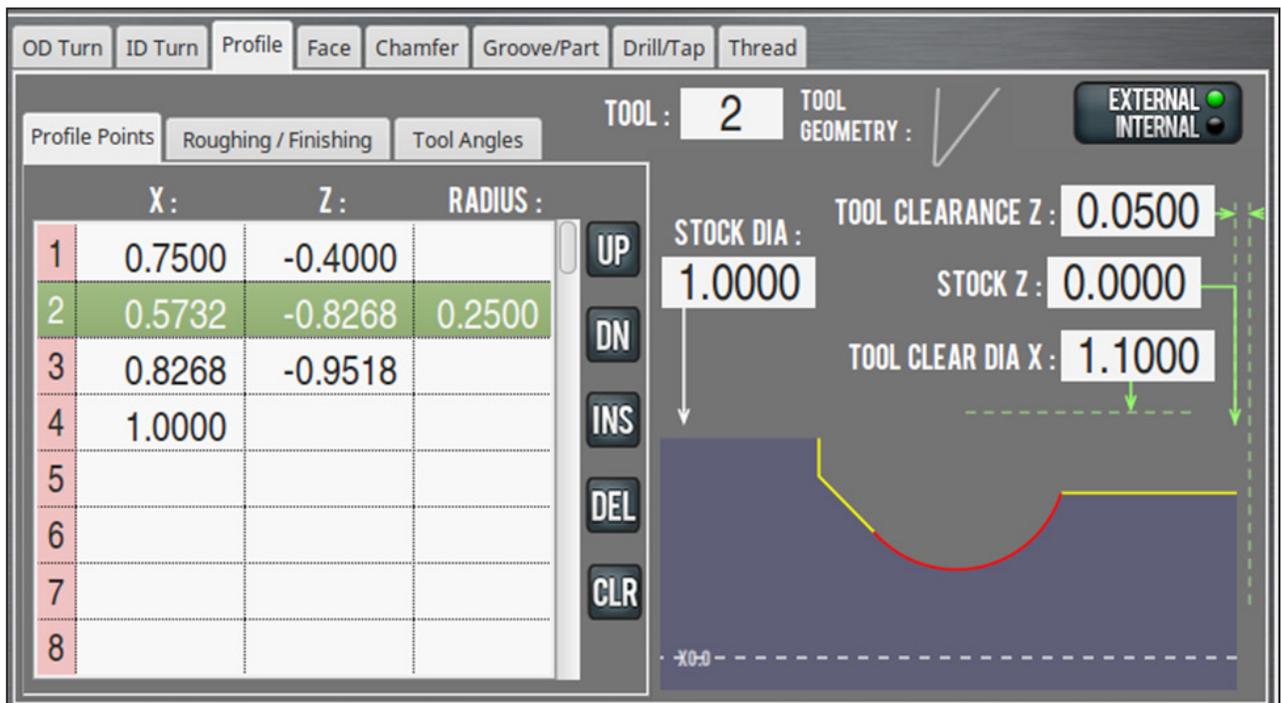


Figure 5.32

**To Create an Arc with the Profile Point Table**

- In the *Profile Point* table, type a Radius value to X and Z end points (see **Figure 5.32**).

  - A positive radius value creates a center point above and/or to the right of the start point.

  - A negative radius value creates a center point below and/or to the left of the start point.

**To Preview the Profile**

- As you work through the *Profile Point* table, PathPilot displays a graphical representation of the profile on your part.

  Click on any segment in the graphic to highlight the corresponding row in the *Profile Point* table.

Alternately, you may click on any row in the *Profile Point* table to highlight the corresponding segment in the graphic.

Point the mouse toward any area in the graphic and use the scroll wheel to zoom in and out to enlarge small features. To quit zooming, either press **ESC** on the keyboard or click on another line in the *Profile Point* table.

**Identify the Tool**

1. In the *Tool* DRO field, type the number of the tool to use for creating the profile.

   *Tool Geometry* displays a graphical representation of the selected tool.

2. On the graphical representation of the profile on your part, make sure there are no red line segments. If there are red line segments, you must specify a new tool or edit the fields in the *Profile Point* table.

A red line segment indicates that the geometry of the selected tool cannot cut the programmed angle without gouging the part profile – typically, when feature entry or exit angles are too steep for the tool geometry to clear (see **Figure 5.32**).

For example, in **Figure 5.32**, the back angle of the tool will not clear the entry to the arc feature.

**Identify the Roughing and Finishing**

Use the *Roughing / Finishing* tab to describe the required roughing and finishing for the part's profile.

*NOTE: You can use **only** roughing or **only** finishing. It is an error if all three of the following DRO fields are empty:* Finishing Passes*,* Roughing DOC*, **and** Finishing DOC.*
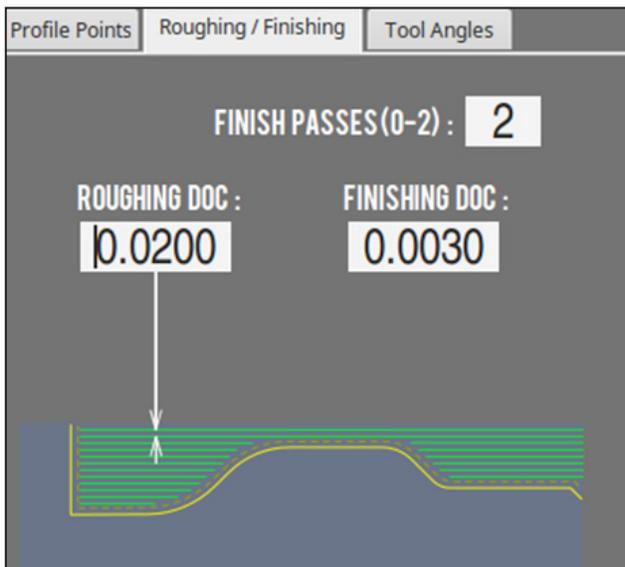


**Figure 5.33**
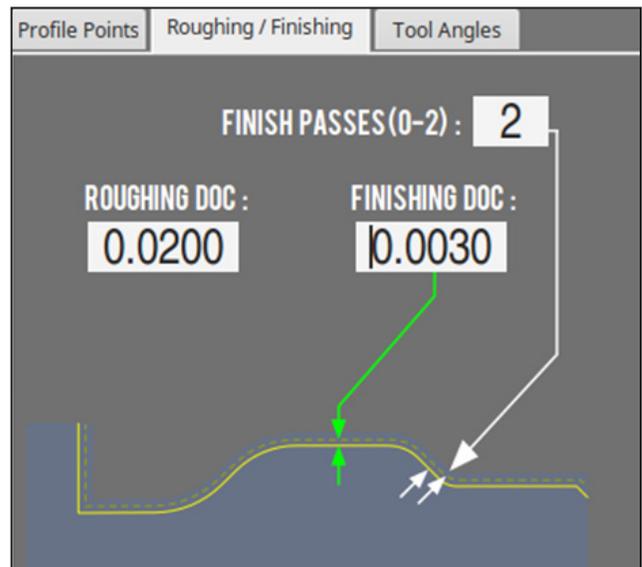


**Figure 5.34**

1. From the PathPilot interface, on the *Profile* tab, click the *Roughing / Finishing* tab.

   PathPilot updates the graphical representation of the profile depending on which DRO field you select:

   - Click the *Roughing DOC* DRO field to display a roughing graphic (see **Figure 5.33**).
   - Click the *Finishing DOC* DRO field or the Finish Passes DRO field to display a finishing graphic (see **Figure 5.34**).

2. In the *Roughing DOC* DRO field, type the depth of cut for each roughing pass. Then, press **ENTER** on the keyboard.

*NOTE: The default value is 0.02 inches.*

3. In the *Finishing DOC* DRO field, type the depth of cut for each finishing pass. Then, press **ENTER** on the keyboard.

*NOTE: The default value is 0.003 inches.*

4. In the *Finishing Passes* DRO field, type the number – from 0-2 – of finishing passes. Then, press **ENTER** on the keyboard.

*NOTE: The default value is 2 (passes).*

A finishing pass is a continuous pass from the start of the profile (away from the headstock) to the end of the profile (toward the headstock).

**Describe the Tool Geometry**

Use the *Tool Geometry* tab to describe the front and rear profiling angles (see **Figure 5.35**). Because there are many tool and tool holder geometries, the Tool Geometry tab allows you to properly describe each tool and avoid part gouging.

To describe the tool geometry, you must first determine the cutting direction: either X+ or X-, based on the tool orientation and an external or internal profile.

For example, if you are cutting an external profile on a part and have a tool orientation of 2, PathPilot creates X+ G-code.

Use the following table to determine the cutting direction:

| Cutting Direction | Internal or External | Tool Orientation |
|---|---|---|
| X+ (typically, while using a turret) | External | 1, 2, 6 |
| | Internal | 3, 4, 8 |
| X- (typically, while using a quick-change tool post) | External | 3, 4, 8 |
| | Internal | 1, 2, 6 |
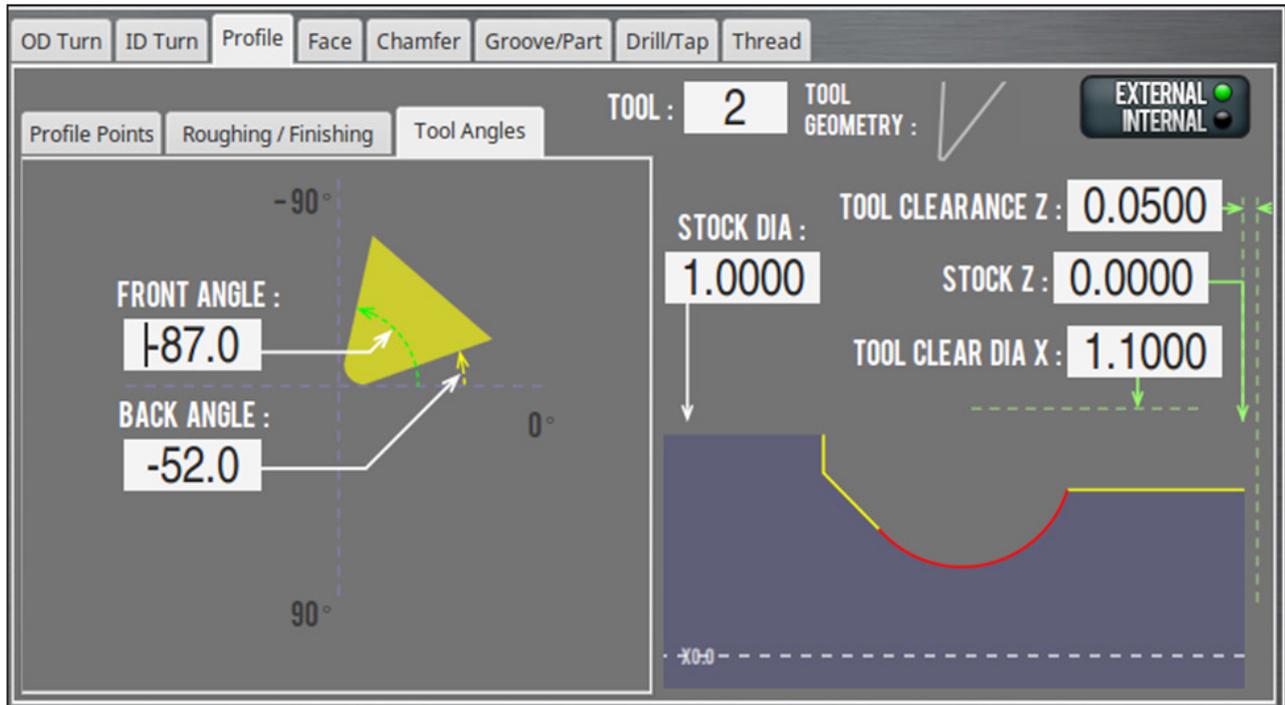
**Figure 5.35**

### To Describe X+ Cutting Tool Geometries

- In the *Front Angle* DRO field and the *Back Angle* DRO field, type the value of the tool geometry expressed as a negative angle in the counterclockwise direction from 0˚.

*NOTE: The Tool Geometry graphic preview (to the right of the Tool DRO field) updates as angles are changed.*

### To Describe X- Cutting Tool Geometries

- In the *Front Angle* DRO field and the *Back Angle* DRO field, type the value of the tool geometry expressed as a negative angle in the clockwise direction from 0˚.

### 5.8.6  Face Tab

Face creates G-code that cuts a face with tool paths from the stock OD to the spindle center, or to an ID. Passes progress in Z toward the headstock (see **Figure 5.36**). For cutting with a rear tool both the value in the *Initial X* DRO field and the value in the *Final X* DRO field need to be positive. For front tool cutting, both need to be negative. CSS is used for spindle speed control. FPR controls feedrate. Rough facing starts at Z Start and incrementally cuts at the DOC until the start of the finish face pass (Z END + FINISH DOC).

The start of each pass is at the Initial X diameter + Tool Clearance and moves in the minus X direction until the Final X diameter – Tool Clearance is reached. If the value in the Final X DRO field is zero, the end of the pass will go beyond the spindle center. Finishing is done in one pass at the value entered into the *Finish DOC* DRO field.
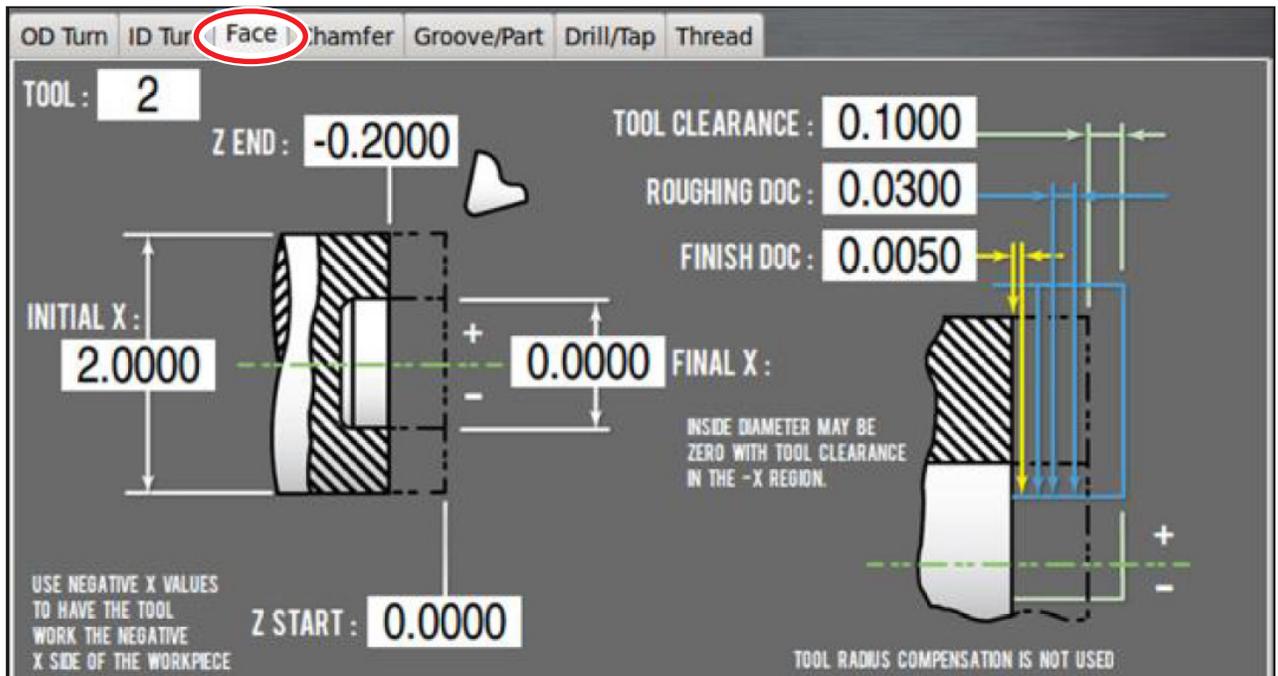
**Figure 5.36**

The *Tool* DRO field is a command value (meaning that it will set the location of a new feature) that sets the tool number for a tool change at the start of the program.

The *Initial X* DRO field is a reference value (meaning that the value should describe an existing feature). This value is usually set to the stock diameter. It is also used with Tool Clearance to locate some of the transitions between rapid and feed rate. If the values in the *Initial X* DRO field and the *Final X* DRO field are both positive, the tool works on the positive X side of spindle center, the side away from operator. If they are both negative, the tool will work the negative side of the spindle which is the side closer to the operator. A positive and negative value for each will produce an error.

## Facing Tool Path



**Figure 5.37**

The *Final X* DRO field is a command value. It sets the location of the face inside diameter. The tool path will go beyond this diameter by the tool clearance. For tools with a tip radius, the control point and face contact point are not the same, so the tool clearance value, if greater than the tool tip radius, can be used to extend the path to the contact point.

The *Z Start* DRO field is a reference value for the location of the stock face. Roughing passes start here. It is also used with Tool Clearance to set the transition between rapid and feed rates on some moves.

The *Z End* DRO field Is a command value for the finished face location.
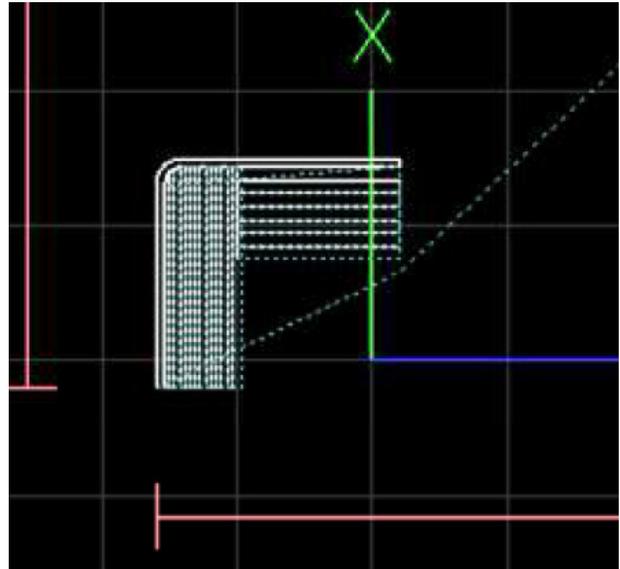
The *Tool Clearance* DRO field is a reference value that should match the desired space needed for tool retracting and transitions between rapid and cutting feed rate. Since there is one value used for X and Z moves, the value should be set to the greater of the two clearances. Larger values may be safer, but brings the back of the tool holder closer to the ID wall on the end of facing cuts. Smaller values may save time once the operator is familiar with how well the program works.

The *Roughing DOC* DRO field is a command value that sets the depth of material being cut. This depth of cut is adjusted (for more information, refer to section *Adjusted DOCs* earlier in this chapter) to get the value used in the G-code. Valid values are positive.

The *Finish DOC* DRO field is a command value that is used to leave enough material after roughing to do one finish pass. Valid values are positive.

### 5.8.7 Chamfer and Radius Tab

Chamfer/radius creates G-code for cutting a chamfer, taper or corner radius (see **Figure 5.38**). Cutter radius compensation (G41, G42) is used, so tools with a nose radius can cut to the correct profile. Radii are limited to 90° arcs that start on the outside diameter (Initial X, Z End). Depending on the sense of the X and Z DRO field entries, the chamfer or radius feature may face away from or toward the spindle, or use back or front mounted tooling.

Spindle speed is controlled by CSS. Tool feed is controlled by FPR. Roughing starts at the corner of X and Z Start in adjusted DOC increments perpendicular to the chamfer angle or incremental arcs for radius (for more information, refer to section *Adjusted DOCs* earlier in this chapter). The last roughing pass leaves enough material for the finish pass; finishing is done with a single pass.

Passes start and end on the perimeter of the tool clearance space, which is set by adding the tool clearance DRO value to the stock OD, X, and the face location, Z Start. Extra caution is advised when using chamfer angles less than 30° or greater than 60°, due to the extra travel involved in traversing the tool clearance space at an angle. The path may take the tool into the chuck, spindle or adjacent workpiece features.

The *Tool* DRO field is a command value (meaning that it sets the value of a new feature) which sets the tool number for a tool change at the start of the program.

The *X* DRO field is a reference value (meaning that the value should describe an existing feature).
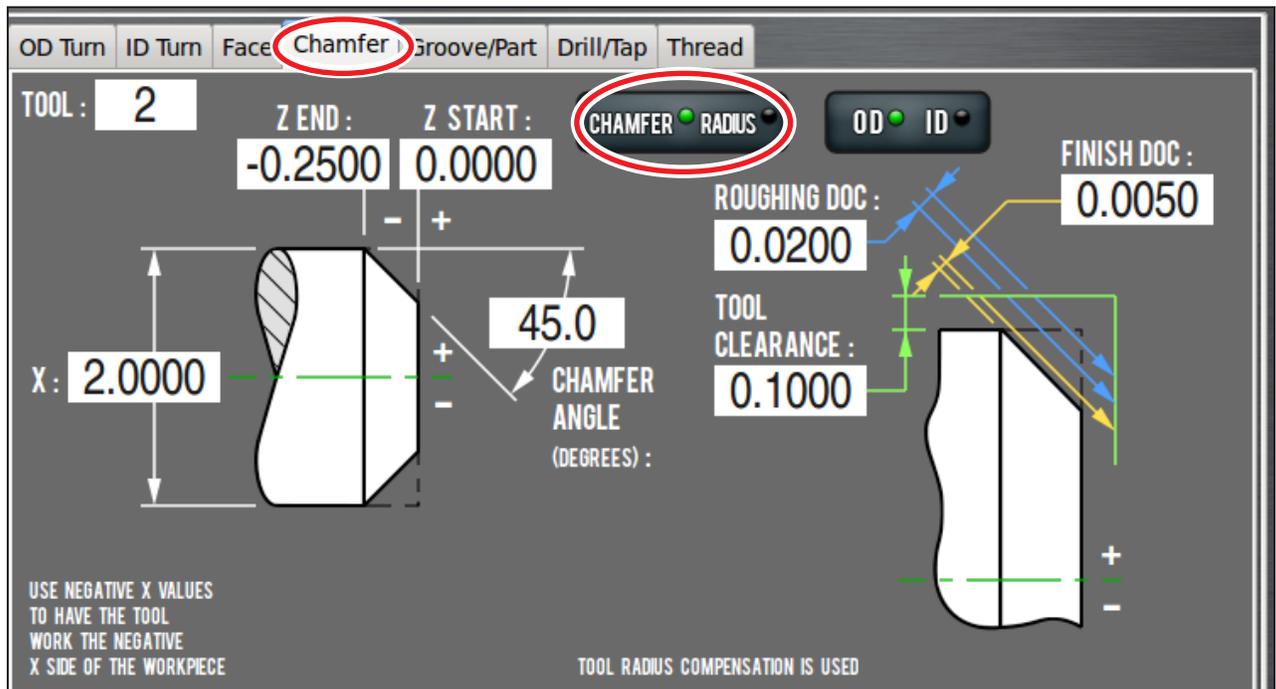
**Figure 5.38**

This value is usually set to the stock diameter. It is also used with Tool Clearance to locate some of the transitions between rapid and feed rates. If Initial X is positive, the tool works on the positive X side of the spindle center, the side away from the operator. If the value in the *Initial X* DRO field is negative, the tool will work the negative side of the spindle which is the side closer to the operator.

The *Z Start* DRO field is a reference value for the stock face or the end of the chamfer or radius. It is used with *Tool Clearance* to set the transition between rapid and feed rates on some Z moves.

The *Z End* DRO field is a command value that sets the location of the start of the chamfer or radius. The Z width of a chamfer or the radius of a corner radius equals Z END – Z START.

The *Chamfer Angle* DRO field is a command value that sets the angle between the workpiece center line and the chamfer. Valid values are between 0° and 90° (non-inclusive).

The *Tool Clearance* DRO field is a reference value that should match the desired space beyond the stock outside diameter and face that is needed for some movements to clear the workpiece. Since there is one value used for X and Z moves, the value should be set to the greater of the two clearances. Larger values may be safer; smaller values may save time once the operator is familiar with how well the program works. A typical move would be a rapid move from G28 to the start of the first cut, which might be at X + TOOL CLEARANCE, Z START + TOOL CLEARANCE, then the rate would change to a roughing feed rate. *Tool Clearance* is also sometimes used as a location for retracting the tool while making cutting passes.
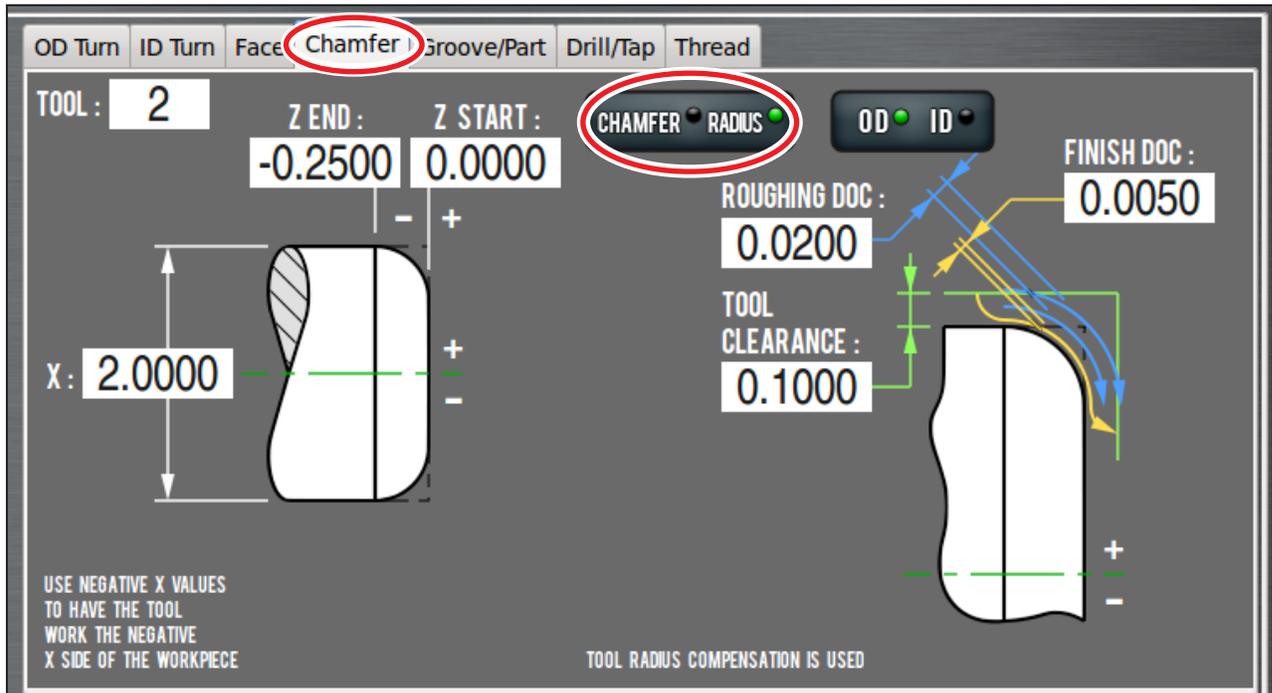
**Figure 5.39**

The *Roughing DOC* DRO field is a command value that sets the depth of cut during roughing. The depth of cut is adjusted (for more information, refer to section *Adjusted DOCs* earlier in this chapter). In this case the roughing range is the distance from the workpiece corner (the intersection of the face and OD) and the closest point on the chamfer or radius minus the finish DOC. Valid values are positive.

The *Finish DOC* DRO field is a command value that is used to leave enough material after roughing to do one finish pass. Valid values are positive.

### Chamfer on Bar End Tool Path



**Figure 5.40**

### Radius on Bar End Tool Path
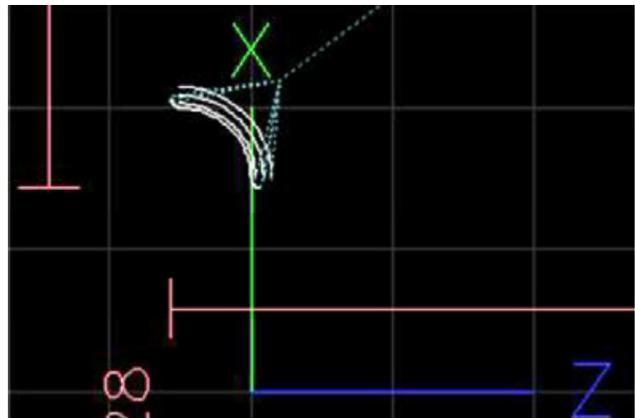


**Figure 5.41**
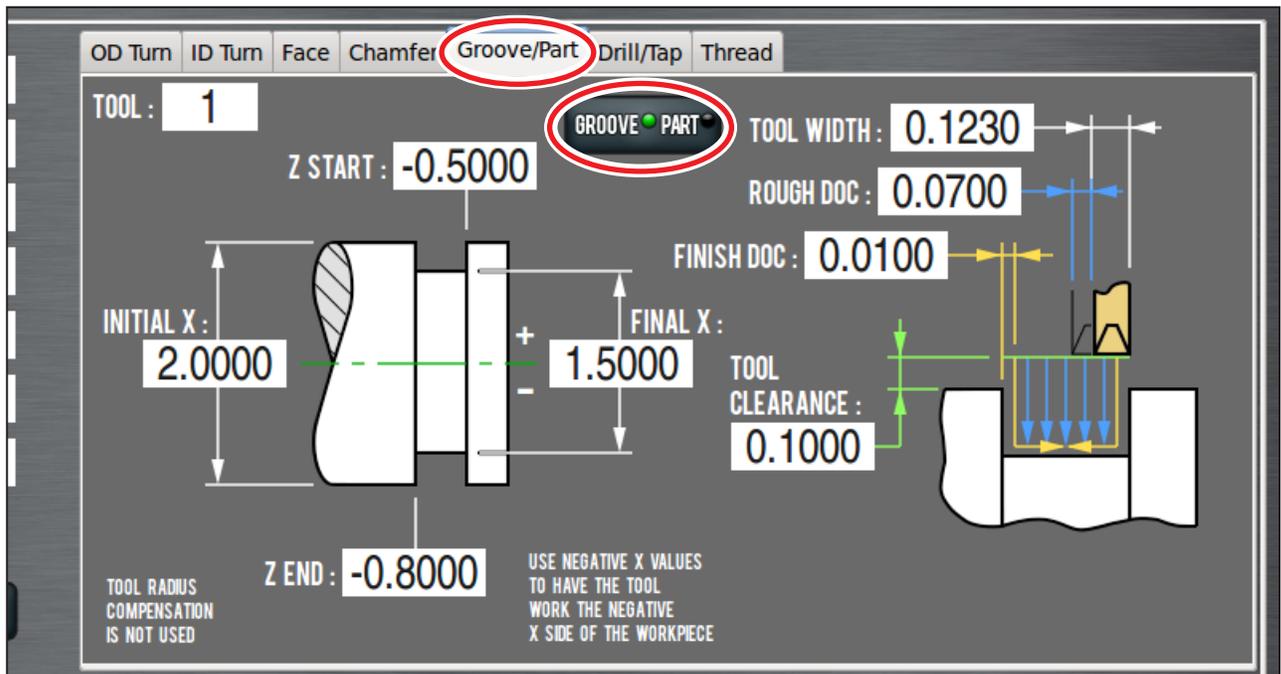
**Figure 5.42**

### 5.8.8 Groove and Part Tab

Groove/Part creates G-code to rough and finish a groove, or to part a workpiece from stock. Initially, the *Groove* screen is presented (see **Figure 5.42**), but can be switched to *Part* with the *Groove/Part* button (see **Figure 5.43**). Grooving paths are based on Z START and Z END values. If the value in the *Z Start* DRO field is greater than the value in the *Z End* DRO field, the tool's control point is set to the +Z side of the tool. If the value in the *Z Start* DRO field is less than the value in the *Z End* DRO field, the control point is set to the -Z side of the tool (see **Figure 5.44**). Groove roughing is done with plunge cuts in the X direction. Each plunge is incremented in the Z direction from Z START – or + FINISH DOC to Z END – or + (TOOL WIDTH + FINISH DOC).

Groove finishing is done with a plunge cut down the Z END side. When the tool reaches the bottom, the tool is moved in the Z direction toward the center of the groove, then retracts. Next, the tool is plunged on the Z START side of the groove, then again is moved in Z toward the groove center and retracted. This requires a grooving tool, which can side cut. Part does one plunge cut at Z START. The tool's control point is on the +Z side of the tool. The plunge cannot be set to go beyond the spindle center (X = 0).

Even though a grooving/parting tool may be considered to have two tips, valid tool orientation is limited to:

- Groove on the – side of Z START, Back Tool = Type 1
- Groove on the + side of Z START, Back Tool = Type 2
- Groove on the + side of Z START, Front Tool = Type 3
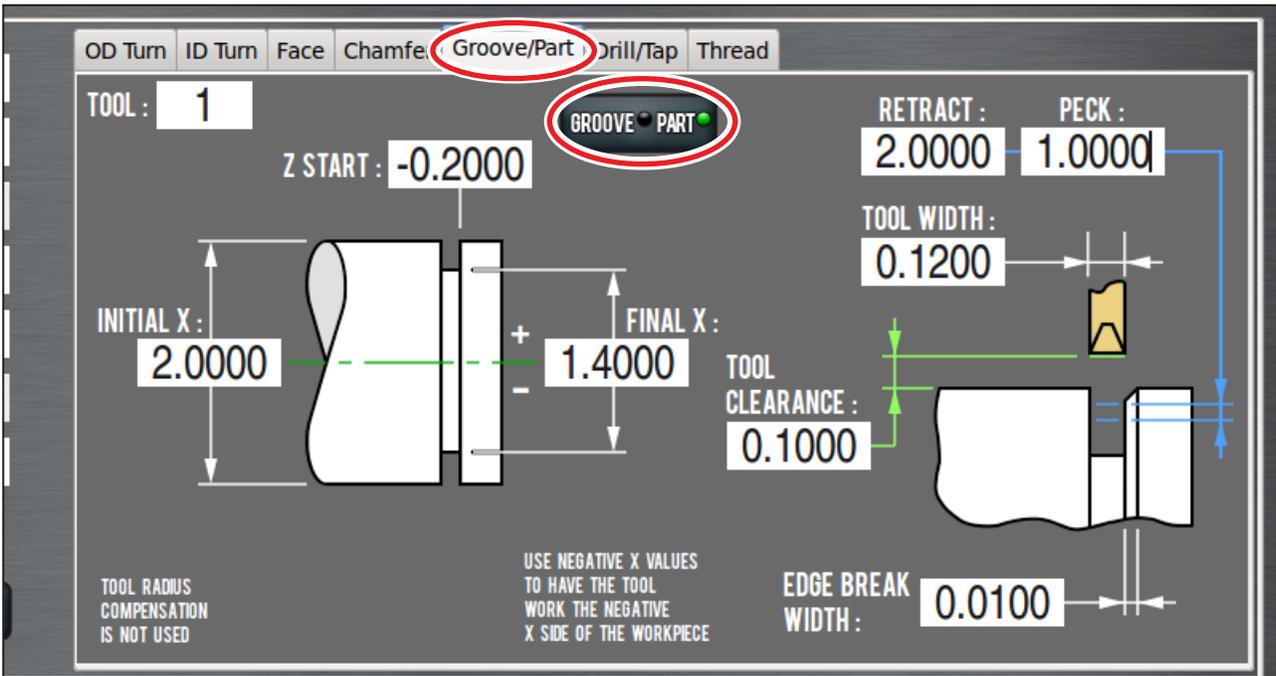
**Figure 5.43**

- Groove on the – side of Z START, Front Tool = Type 4

- Part on the – side of Z START, Back Tool = Type 1

- Part on the – side of Z START, Front Tool = Type 4

CSS is used for spindle speed control. FPR is used for feed rate control. CRC is not used.

If the value in the *Z Start* DRO field is less than the value in the *Z End* DRO field, the control point is set to the –Z side of the tool (see **Figure 5.44**).

The *Tool* DRO field is a command value (sets the location of a new feature) that sets the tool number for a tool change at the start of the program.

The *Initial X* DRO field is a reference value (the value should describe an existing feature). This value is usually set to the stock diameter. It is also used with Tool Clearance to locate some of the transitions between rapid and feed rates. If the value in the *Initial X* DRO field and the value in the *Final X* DRO field are both positive, the tool will work on the positive X side of the spindle center, which is the side away from the operator. If they are both negative, the tool will work the negative side of the spindle which is the side closer to the operator. A positive and negative value for each produces an error.
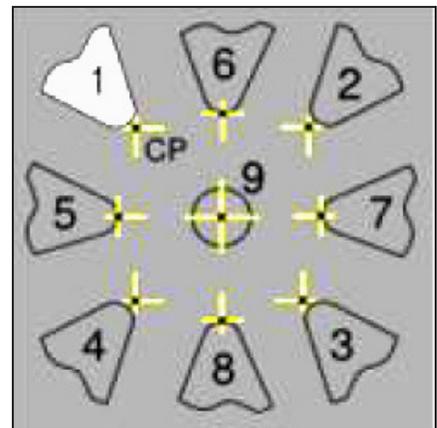


**Figure 5.44**

The *Final X* DRO field is a command value for setting the diameter of the new groove bottom, or the end of parting. If the value in the *Initial X* DRO field and the value in the *Final X* DRO field are both positive, the tool will work on the positive X side of the spindle center, which is the side away from the operator. If they are both negative, the tool will work the negative side of the spindle which is the side closer to the operator. A positive and negative value for each produces an error.

The *Z Start* DRO field is a command value for setting the location of the groove start. For parting, *Z Start* sets the location of the +Z side of the slot.

The *Z End* DRO field is a command value for setting the location of the groove end. This value is not used with part.

The *Tool Width* DRO field is a reference value for setting the groove or parting tool width. This value is added or subtracted based on the tool's control point (see **Figure 5.44**).

The *Rough DOC* DRO field is a command value that sets the depth of material being cut. In this case, for groove, it is the Z offset for each plunge cut. The depth of cut is adjusted. Valid values are positive and normally should be less than the full depth width of the tool tip (usually the distance between tip radii centers). This value is not used with part.

**Groove Tool Path**



The *Finish DOC* DRO field is a command value for the amount of material roughing should leave for the finishing pass (groove sides and bottom). Valid values are positive. This value is not used with part.

The *Tool Clearance* DRO field is a reference value that should match the desired space beyond the stock outside diameter for rapid movements to clear the workpiece. Larger values may be safer; smaller values may save time once the operator is familiar with how well the program works. A typical move would be a rapid move from G28 to the start of the first cut, which would be at *Z Start*, *Initial X + Tool Clearance*, then the rate would change to a cutting feed rate. *Tool Clearance* is also sometimes used as a location for retracting the tool between cutting passes.
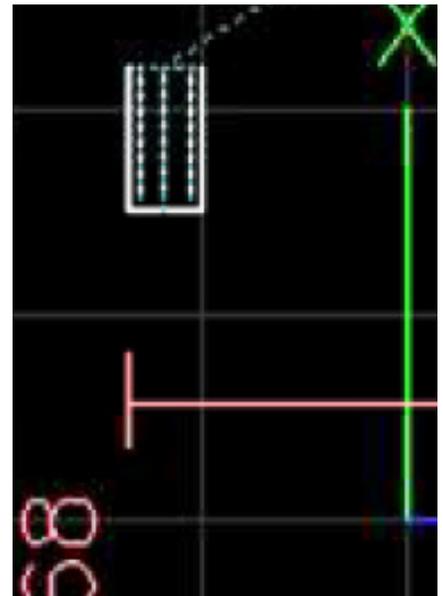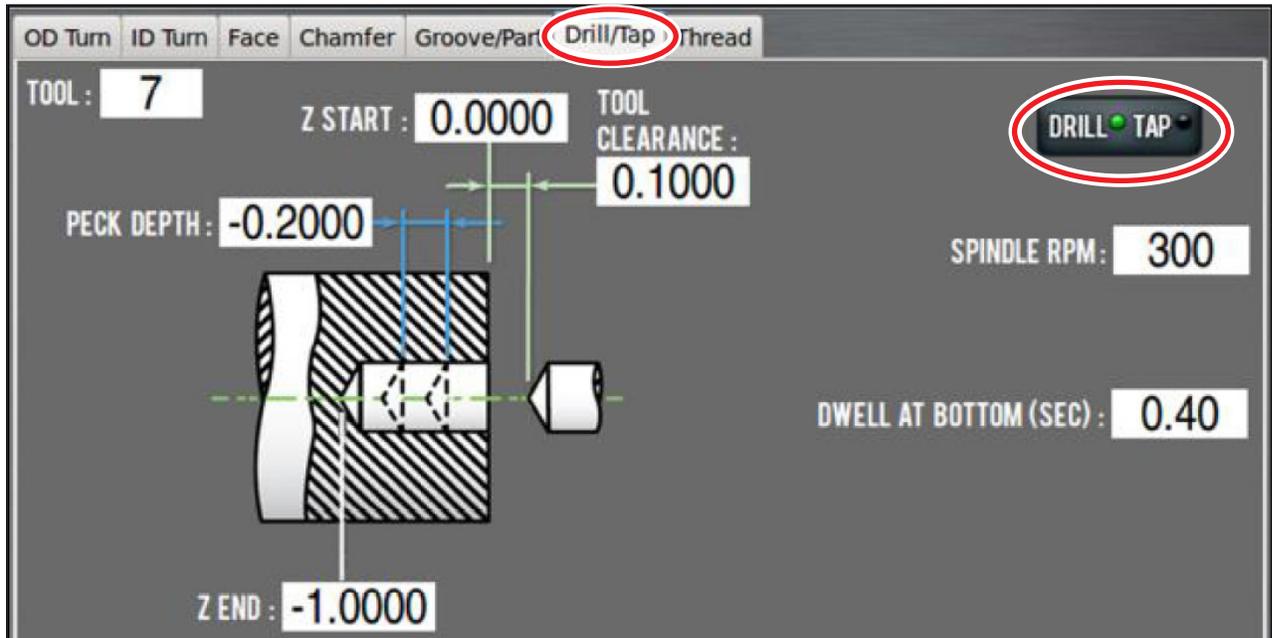
**Figure 5.45**

**Figure 5.46**

### 5.8.9   Drill and Tap Tab

The *Drill/Tap* tab creates G-code to drill a hole or use rigid tapping to thread a hole. Initially, the *Drill* screen is presented, but can be switched to the *Tap* screen using the *Drill/Tap* button (see **Figure 5.46** and **Figure 5.47**). Drill and tap use the tool at X = 0, so an RPM DRO is used instead of CSS from the common DRO section to the left. Otherwise, X = 0 would call for the maximum spindle speed.

For feed rate control, drill uses a millimeter or inch feed per revolution (G95) to feed from Z START + TOOL CLEARANCE until Z END is reached, then rapids back to Z START + TOOL CLEARANCE. Drilling is limited to the -Z direction, toward the spindle. A *Dwell* DRO field is provided to allow a pause for the drill to stay at Z END long enough to cut a full revolution at the bottom of the hole, otherwise the drill is immediately retracted and may leave an irregular bottom. Pecking can be used to help clear chips before they bind in the hole during drilling. The peck motion retracts to Z START + TOOL CLEARANCE on each cycle. Due to motion control limits, the retract to Z START + TOOL CLEARANCE may not retract fully before starting the next drilling feed. Please verify the pecking retract motion meets requirements for your application.

Tap uses electronic gearing (G33.1, Rigid Tapping) to lock the Z-axis and spindle motion together for rigid tapping. Tapping starts with the tap at Z START + TOOL CLEARANCE. Z motion waits until the spindle encoder index is tripped, at this point, the gears are engaged and Z feeds at the rate set by the pitch or threads per unit (TPU) and spindle encoder count. The Z motion follows the spindle motion no matter what the spindle does. For tapping, the spindle is run forward until Z END is reached, the spindle is reversed, which causes it to slow to a stop, then reverse. During this time, the tap continues to follow the spindle motion and continues to make threads until the spindle reverses. These extra threads needs to be considered when setting the *Z End* DRO field. The reverse motion continues until reaching Z START + TOOL CLEARANCE where the gearing is disengaged.
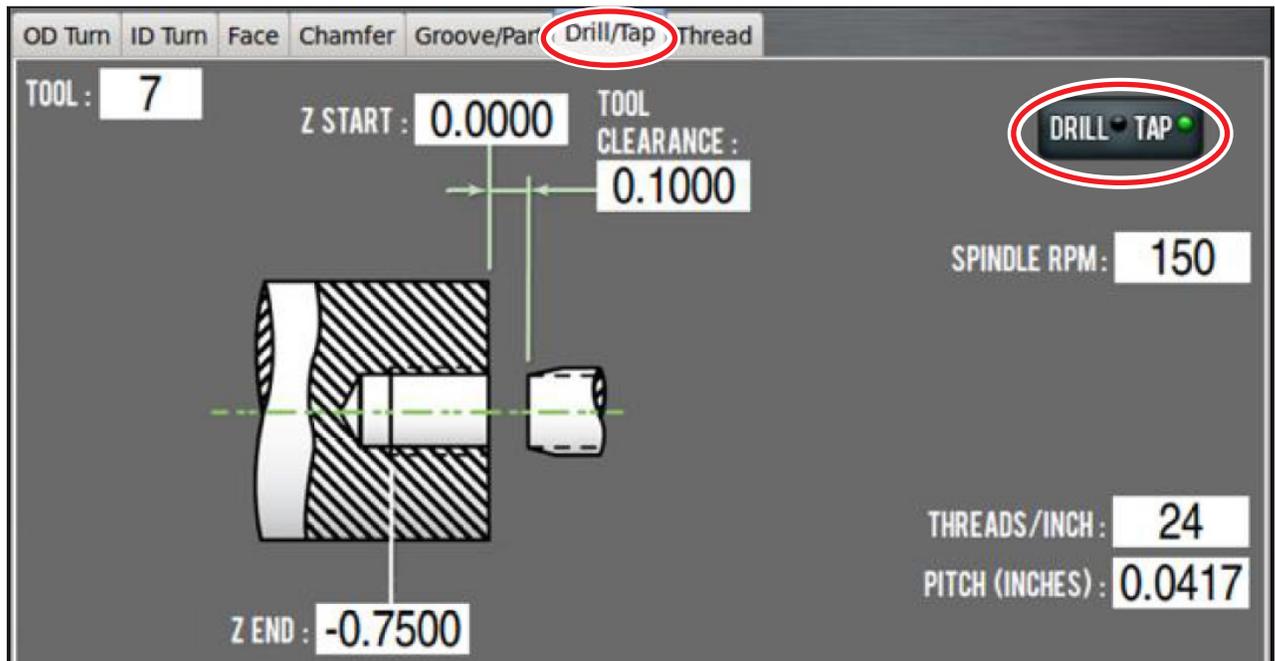
**Figure 5.47**

The *Tool* DRO field is a command value that sets the tool number for a tool change at start of the program.

The *Z Start* DRO field is a reference value for the stock face location. It is used with *Tool Clearance* to set the transition between rapid and the feed for drilling or tapping. An error is produced if the value in the *Z Start* DRO field is not larger than the value in the *Z End* DRO field.

The *Z End* DRO field is a command value.

For drill, *Z End* DRO field sets the location where the drill feed stops and optionally dwells. The hole depth is from *Z Start* to the drill's radial corner, but typically the drill tip is touched-off, so the Z distance between the tip and the drill's radial corner must be considered when selecting a value for the *Z End* DRO field.

For tap, *Z End* is a command value that sets the Z location where the spindle rotation is reversed. The value in the *Z End* DRO field should be smaller than the value in the *Z Start* DRO field and should allow for extra threading while the spindle comes to a stop and reverses.

The *Peck Depth* DRO field is a command value that sets an incremental depth for retracting the drill to clear chips from the hole if needed. If drilling the hole does not need a peck, enter zero. To make each peck depth equal, the *Peck Depth* DRO field value is adjusted to fit an integer number of pecks within the hole depth. The *Peck Depth* DRO field also needs a direction, so should have a negative value.

The *Tool Clearance* DRO field is a reference value that should match the desired space needed for tool retracting and transitions between rapid and cutting feed rate.

The *Spindle RPM* DRO field is a command value for the RPM (G97) desired for drill or tap.

The *Dwell at Bottom* DRO field is a command value used with drill for the length of time that Z motion is paused in order for the drill to finish cutting the hole bottom before retracting. If dwell is needed, a value can be calculated by converting RPM to seconds/revolution or 60/RPM.

The *Threads/Inch, mm* DRO field is a command value that is used with tap to set the Z motion to spindle ratio to match the thread pitch needed. This DRO field is coupled with the *Pitch* DRO field, so entering a value in either DRO field calculates and enters a value in the other.

The *Pitch* DRO field is a command value that is used with tap to set the Z motion to spindle ratio to match the required thread pitch. This DRO field is coupled with the *Threads/Inch, mm* DRO field, so entering a value in either DRO field calculates and enters a value in the other.

### 5.8.10 Thread Tab

Thread can be used to create G-code to single point an external or internal thread on an existing OD (**Figure 5.48**) or ID (**Figure 5.49**). Initially, the screen is in external mode, but can be switched to internal mode with the *External/Internal* button.
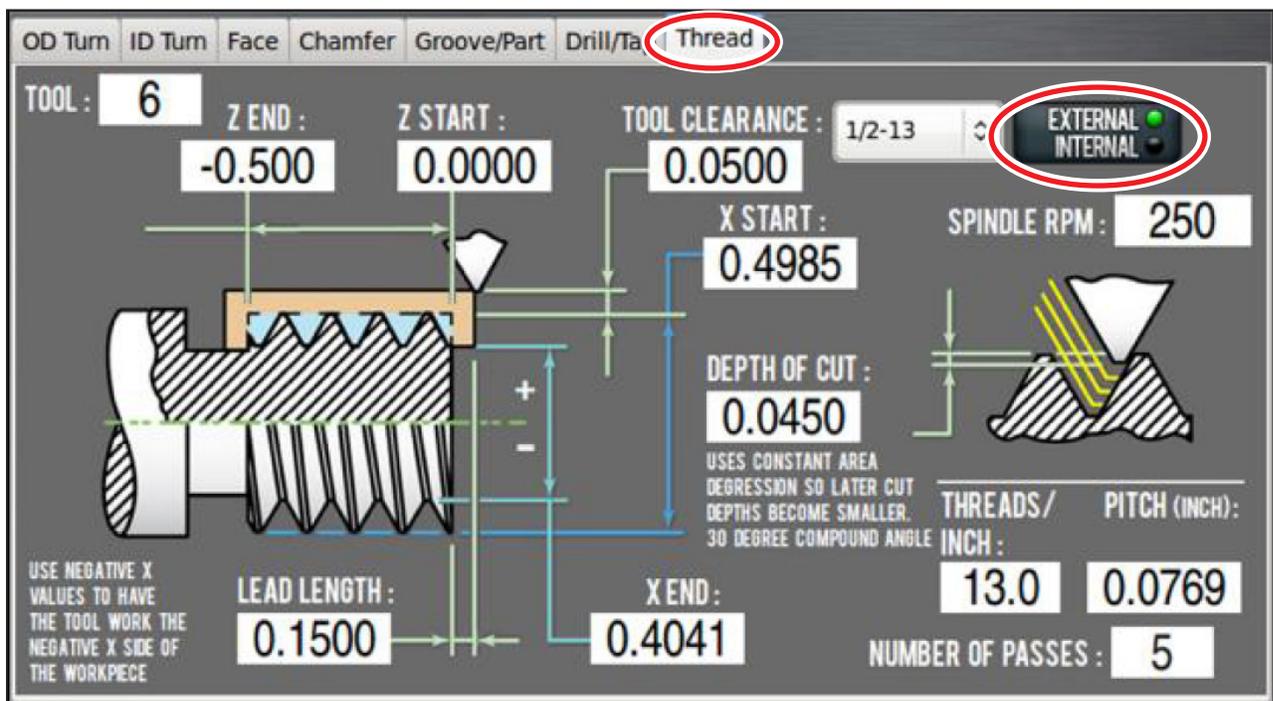


**Figure 5.48**

Thread is based on the G76 Threading Cycle. This canned cycle contains a lead-in, cut, lead out, and return path for each threading pass. Each cycle is incrementally offset in X and Z to account for a 30° software compound angle. The offset is calculated such that the each pass cuts the same amount of material by cross sectional area. The first pass has the most X displacement, and this decreases with each pass.
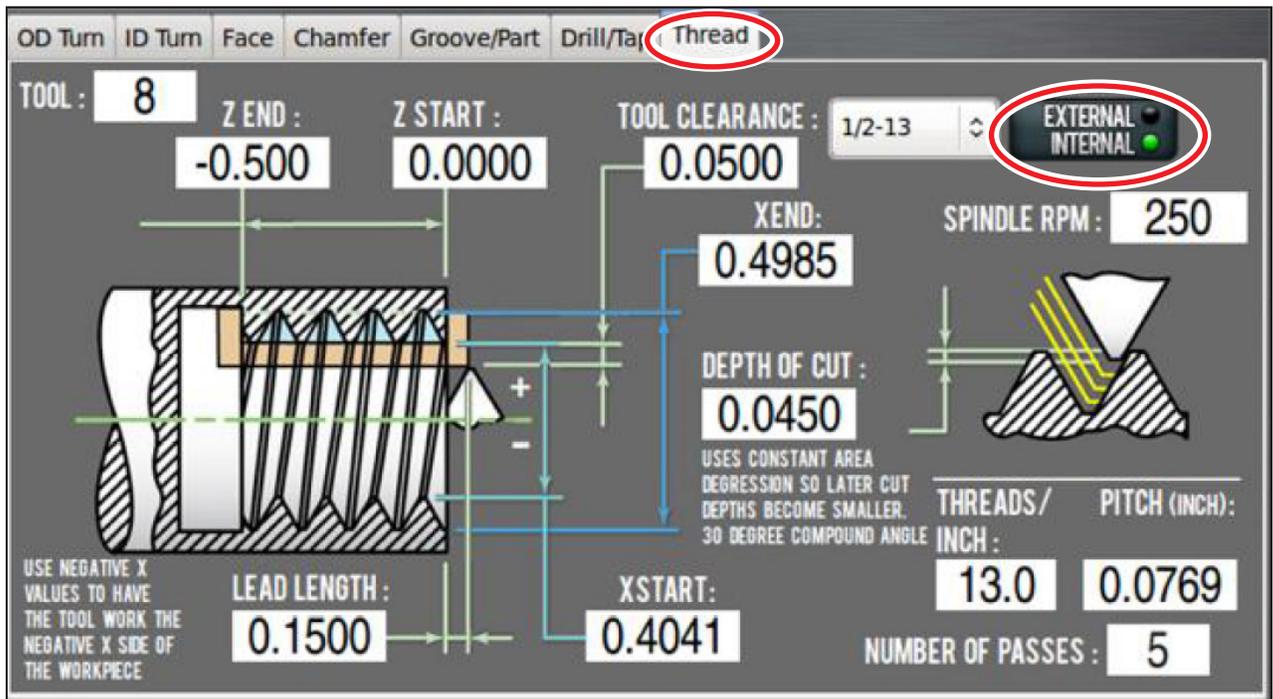
**Figure 5.49**

The *Tool* DRO field is a command value that sets the tool number for a tool change at the start of a program.

The *X Start* DRO field is a reference value (meaning that the value should describe an existing feature). This value is usually set to the existing major diameter for external threads or minor diameter for internal.

The *X End* DRO field is a command value. It sets the location of the new outside diameter.

The *Z Start* DRO field is a reference value for the stock face location. It is used with Tool Clearance to set the transition between rapid and feed rate on some Z moves. The value in the *Z Start* DRO field should be larger than the value in the *Z End* DRO field.

The *Z End* DRO field is a command value that sets the face location. The value in the *Z End* DRO field should be smaller than the value in the *Z Start* DRO field.

The *Tool Clearance* DRO field is a reference value that should match the desired space beyond the stock outside diameter that is needed for rapid movements to clear the workpiece. In thread, this sets the starting X diameter of the thread cycle return path. Larger values may be safer; smaller values may save time once the operator is familiar with how well the program works.

The *Lead Length* DRO field is a reference value that sets the required length during the start of the cutting path that allows the motion to stabilize before cutting material. The start of a cutting pass waits for the spindle encoder index to trip; when it does, the Z motion tries to instantly match the spindle speed, but actually needs time to accelerate and match the spindle encoder count. This value needs to be a compromise between spindle speed, thread pitch, and workpiece clearance.

The *Thread Table* drop-down menu contains values for some common threads. The threads listed follow the current unit setting (inch or millimeter). Once a selection is made, the data from the selected thread is copied to the appropriate DRO fields. This table is stored in user-editable text files found in the *thread_data* subdirectory of the G-code folder on the controller's hard drive; to edit (e.g., to add to or modify the defaults), highlight the file and click *Edit G-code*. For more information on files stored on the controller's hard drive, refer to *File Tab* section earlier in this chapter.

***NOTE:*** *The values entered in these tables assume a full form thread tool. If using a fine point threading tool to cut coarse threads, the root diameter must be modified to account for the smaller tool nose radius of the fine point threading tool.*

The *Spindle RPM* DRO field is a command value that sets the spindle RPM as opposed to the CSS DRO in the common DRO section.

The *Depth of Cut* DRO field is a command value that sets the depth of material being cut. Each pass is incremented based on a calculation of the area of the material being removed. This allows for a constant tool load for each pass. Valid values are positive.

The *Threads/In, mm* DRO field is a command value that sets the number of threads per inch or millimeter. This DRO field is coupled to the *Pitch* DRO field, so whichever value is handy – threads per unit or pitch – may be entered.

The *Pitch* DRO field is a command value that sets the distance per thread. This DRO field is coupled to the *Threads/In, mm* DRO field, so whichever value is handy – threads per unit or pitch – may be entered.

The *Number of Passes* DRO field is coupled to the *Depth of Cut* DRO field. A value may be entered into either DRO field and its compliment updates to reflect the new value. Often, a depth of cut is entered and the number of passes presents a number that might seem excessive. Either DRO field can be adjusted to reach a compromise between the depth of cut and the number of passes.
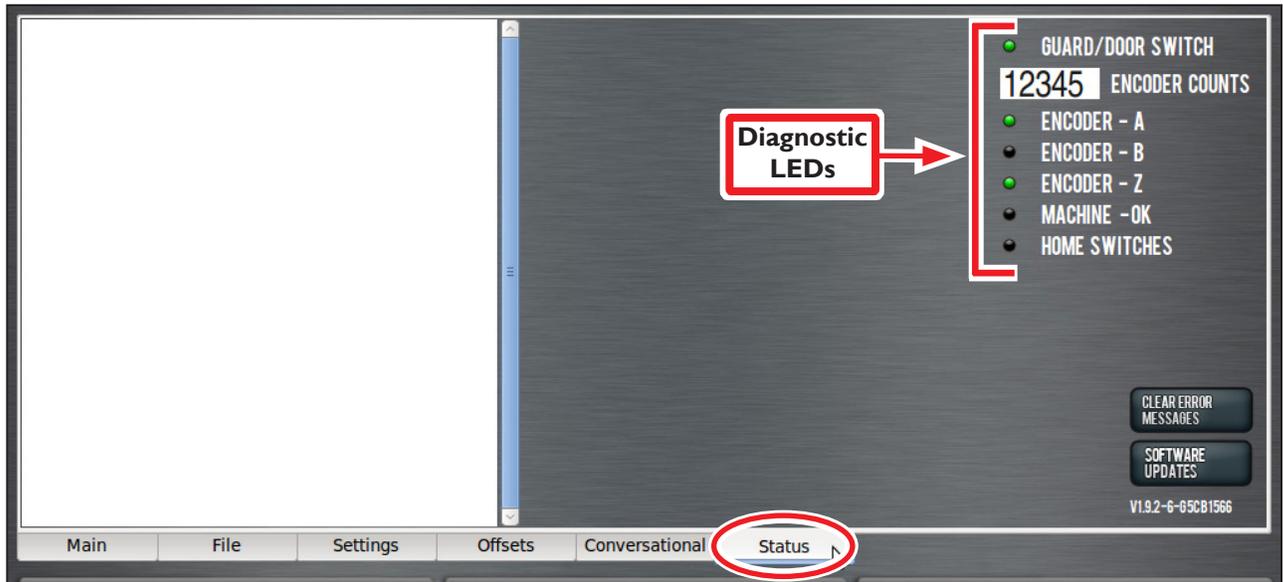
**Figure 5.50**

## 5.9   Status Tab

The *Status* tab displays diagnostic machine information, error messages, and the *Software Updates* button, used to update PathPilot as new releases become available (see **Figure 5.50**).

**Error Messages**

The left hand side of this screen consists of a scrolled text box that displays error messages should any errors occur. For instance, attempting to enter an MDI command that moves the axes before referencing the machine generates an error message (see **Figure 5.51**).
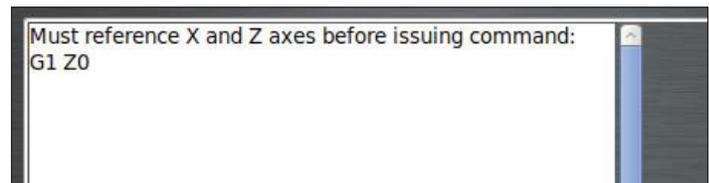


**Figure 5.51**

There are three levels of error messages:

1.  Minor errors: these errors will cause the *Status* tab to change color to yellow type on a red background. Typing an invalid G-code into the MDI field is an example of a minor error.

2.  Major errors: these errors change the color of the *Status* tab, but they also cause PathPilot to switch to the *Status* page, instantly making the operator aware of the error. Errors in conversational code generation are examples of major errors.

3.  E-stop errors: these errors stop all machine motion, G-code program execution, and causes the *Reset* button to flash. The machine must be brought out of reset condition before continuing if this type of error occurs. An example of an E-stop error is a failed turret tool change – if the turret is jammed or otherwise fails to change a tool, the machine puts itself into an E-stop condition.

If the operator encounters a minor error message (the *Status* tab turns red), they can hover over the Status tab with the mouse to a tool tip of the error message (see **Figure 5.52**).
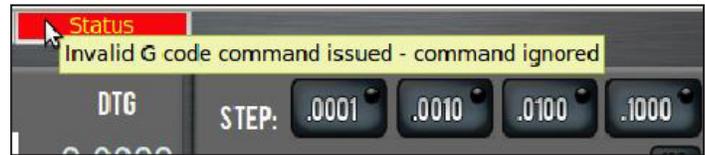


**Figure 5.52**

Alternately, pressing the *F1* key temporarily changes from the current screen to the *Status* screen, allowing the operator to quickly scan the error message. Releasing the *F1* key returns the interface to the previous screen. A history of error messages is displayed in the scrolled text box with the most recent error messages displayed at the top. If desired, the operator may clear the message history using the *Clear Error Messages* button. The entire history is also saved in a log file, found in the *G-code/logfiles* directory.

**Diagnostic LEDs and Counts**

On the upper right-hand side of the *Status* screen are diagnostic LEDs and a DRO displaying encoder counts (see **Figure 5.50**) used for troubleshooting purposes that should not be needed in normal operation. For more information, refer to chapter 7, *Maintenance*, and chapter 8, *Troubleshooting*.
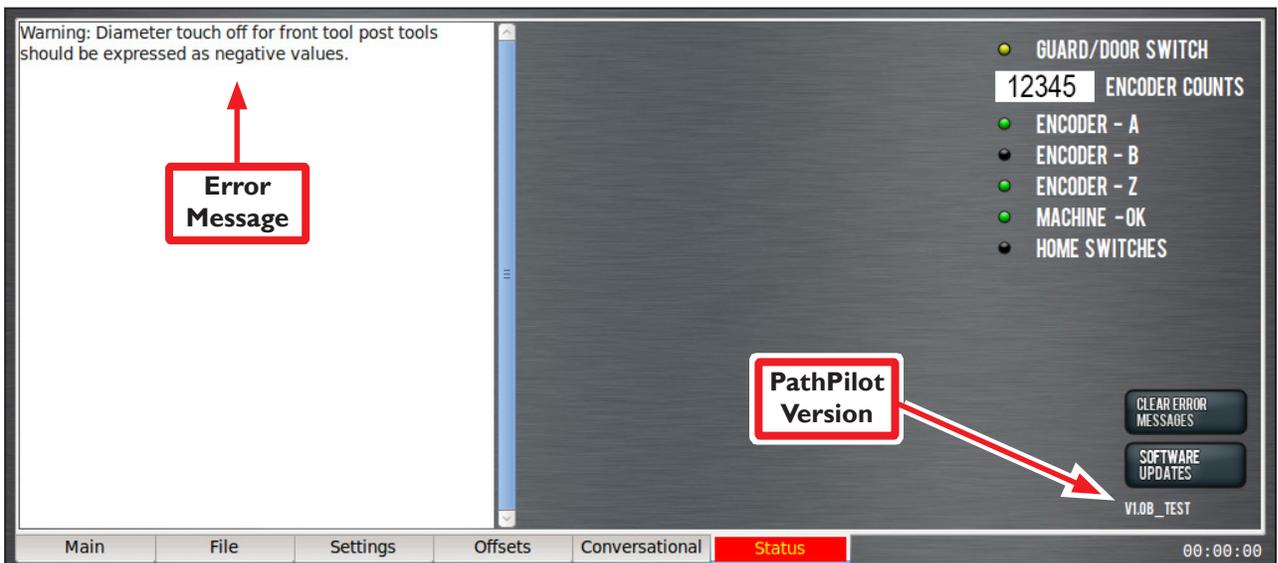


**Figure 5.53**

**Tool Orientation Errors**

If the operator enters information on a *Conversational* page that doesn't match the information in the tool table – for instance entering positive diameters but selecting a front tool post mounted tool – the *Status* screen displays error message in the text box as well as images of the expected tool orientation and the actual tool orientation (see **Figure 5.53**).

**NOTE:** *The current PathPilot version is listed in the lower right corner of the screen (see Figure 5.53).*

# 6.  Programming

This section defines the language (G-codes, etc.) that is understood and interpreted by the PathPilot® operating system, and is is intended for reference purposes. If you want to learn about the principles of the control language so you can write programs by hand from first principles, consult an introductory textbook on G-code programming.

## 6.1  Definitions

The following terms are defined as follows:

**PathPilot Controller**

This is the Tormach motion controller.

**PathPilot Operating System**

This is the PathPilot controller operating system.

**Linear Axes**

The X- and Z-axes are at right angles to each other. The Z-axis lies along the centerline of the spindle; distance values increase moving away from the spindle towards the right-hand side of the machine. The X-axis distance values increase as the tool moves up and away from the operator. The tool cutting position (controlled point) is represented by coordinates on these axes.

**Controlled Point**

The controlled point is the point whose position and rate of motion are controlled to make cuts. In practice, a tool is not a point – the sharper the point of a tool, the weaker it is – so a tip radius is always part of the tip profile. **Figure 6.1** shows a real tool. If this tool is in a front tool post, it can turn to a diameter moving toward the headstock (negative-Z) and face toward the centerline (positive-X).

Many standard shapes of tools are available; PathPilot does not need to know the exact shape of the tool, but does need to know where it is used to cut and how to interpret the controlled point.

There are nine possible orientations for the cutting point, identified by numbers shown in **Figure 6.2**; orientation code 3 is illustrated. If this tool is put in the turret, or a rear position on the gang plate, it is orientation 2.
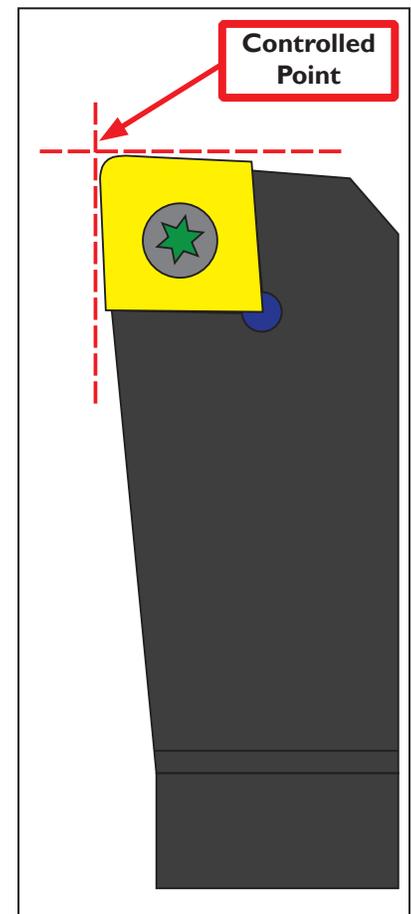


**Figure 6.1**
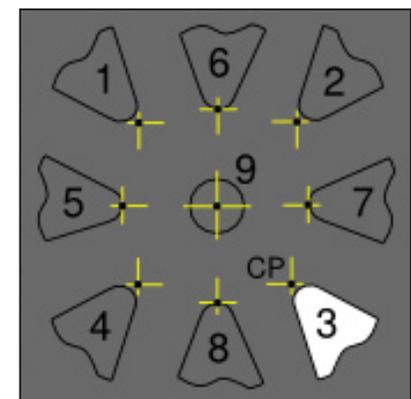


**Figure 6.2**

Tools with orientations 1, 2, 3 and 4 all turn to size and face. Their controlled points are the intersection of tangents to the cutting tip radius so the radius has no effect on the diameter or length of the part made.

If, however, the cut is angled, then the radius means it does not cut the expected part but one a little larger than required. This is represented by the gap between a colored part and the tip of the middle tool in **Figure 6.3**.

To machine a part corresponding to the angled tool path defined in the G-code, it is necessary to use tool-tip radius compensation (i.e., G41 or G42).

Orientations 6 and 8 turn diameters to size and are often used for making profiles. It is difficult to accurately estimate their controlled point in the Z-direction.
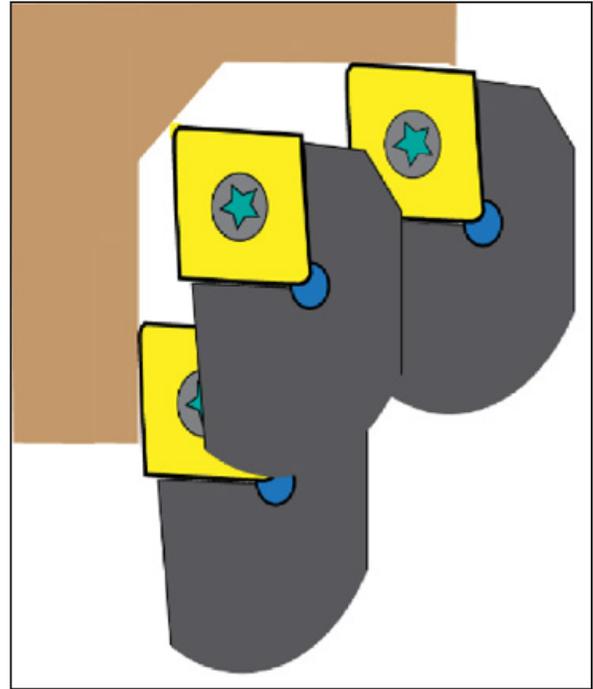
**Figure 6.3**

Orientations 5 and 7 turn on the face of stock and are unusual. It is difficult to accurately estimate their controlled point in the X-direction without a test cut.

Orientation 9 is used when tool-tip compensation is applied in all moves.

Some orientations seem unusual at first glance. For example, a parting or grooving tool in the front tool post has orientation 4. This is because the program defines the part to the right of the tool. The size of the waste left in the chuck is unimportant.

**Coordinated Linear Motion**

To drive a tool along a specified path, a machining system must often coordinate the motion of both. We use the term coordinated linear motion to describe the situation in which each axis moves at constant speed and both axes move from their starting positions to their end positions at the same time. As we only have the X- and Z-axes, this produces motion in a straight line – hence the word linear in the term. In actual motions, it is often not possible to maintain constant speed because acceleration or deceleration is required at the beginning and/or end of the motion. It is feasible, however, to control the axes so that, at all times, each axis has completed the same fraction of its required motion as the other axes. This moves the tool along the same path and we also call this kind of motion coordinated linear motion.

Coordinated linear motion can be performed either at the prevailing feed rate or at rapid traverse rate. If physical limits on axis speed make the desired rate unobtainable, the axes are slowed to maintain the desired path.

**Feed Rate**

The rate at which the controlled point or the axes move is nominally a steady rate which may be set by the operator. In the interpreter, the interpretation of the feed rate is as follows unless inverse time feed rate (G93) mode is being used: the feed rate means the length in units per minute or units per spindle revolution along the programmed linear path.

**Arc Motion**

The axes can be controlled to move in a circular arc in the plane of the axes. While this is occurring, as in coordinated linear motion, the motions can be coordinated so that acceleration and deceleration do not affect the path. The feed rate during arc motion is as described in *Feed Rate* above.

**Coolant**

Although G-code provides for separate flood and mist coolant, either M07 or M08 turn on the 15L Slant-PRO coolant pump and M09 turns it off.

**Dwell**

A machining system may be commanded to dwell (i.e., keep the axes unmoving) for a specific amount of time. The most common use of dwell is to break and clear chips or for a spindle to get up to speed. The units in which you specify dwell are seconds; a decimal value can be used to get less than one second.

**Units**

Units used for distances along the X- and Z-axes may be measured in millimeters or inches. Units for all other quantities involved in machine control cannot be changed. Different quantities use different specific units. Spindle speed is measured in revolutions per minute. Feed rates are expressed in current length units per minute.

***IMPORTANT!*** *Be sure to carefully double check tool and work offsets when changing units (G20, G21) within a part program.*

**Current Position**

The controlled point is always at some location called the current position and PathPilot always knows where that is. The numbers representing the current position are adjusted in the absence of any axis motion if any of several events take place:

- Length units are changed
- Tool length offsets are changed
- Work offsets are changed

***NOTE:*** *These events do not move the tool, instead they change its displayed position in the axis DROs.*

### Work Offsets

Work offsets allow you to jog the machine to an arbitrary location (i.e., the end of a workpiece) and call that location zero. Up to nine different work offsets can be saved in the machine memory, but only one can be active at any given time. The default (used in this example) is G54. More information on work offsets can be found in the part programming reference section of this manual.

### Selected Plane

There is always a selected plane, which for a lathe is the XZ-plane of the machining system.

### Tool Table

Zero or one tool is assigned to each slot in the tool table. The table defines the offset of the controlled point of the tool from the work offset coordinate system's origin, the wear corrections to be made to this tool offset, the tool tip radius, and the orientation of the tool tip.

### Wear Offsets

Wear offsets are values used to fine tune a part program to compensate for things like tool wear, spring back, etc. Wear offsets are applied with the T command as well, by specifying the desired wear offset register with the last two digits of a four-digit T command. For example, T02 applies the geometry offsets for tool 2, whereas T0202 applies both geometry offsets and wear offsets. Note that either command causes the turret to change position to pocket 2 if the machine is equipped with a turret. There is no M6 tool change command for lathes like there is for CNC milling machines.

### Path Control Modes

The lathe may be put into any one of three path control modes:

1. Exact stop mode – the machine stops briefly at the end of each programmed move.

2. Exact path mode – the machine follows the programmed path as exactly as possible, slowing or stopping if necessary at sharp corners of the path.

3. Continuous mode with optional tolerance – sharp corners of the path may be rounded slightly so that the feed rate may be kept up (but by no more than the tolerance, if specified).

For further information, refer to sections *Set Path Control Mode – G61* and *G61.1* and *Set Blended Path Control Mode – G64* later in this chapter.

### Feed and Speed Override Controls

PathPilot has commands which enable (M48) or disable (M49) the feed and speed override slider controls. It is useful to be able to override these for some machining operations. Default settings in the program are set and the operator should not change them.

**Block Delete Control**

PathPilot does not implement the optional omission of blocks of code that are prefixed with the forward slash symbol (/).

**Optional Program Stop Control**

The optional program stop control (M01 BREAK) works as follows. If this button is on and an input line contains an M01-code, program execution is stopped at the end on the commands on that line until the *Cycle Start* button is pushed.

## 6.2   Tool File

PathPilot maintains a tool file for each of the 28 tools which can be used. Each data line of the table contains the data for one tool. This allows the definition of the tool length (X- and Z-axis), tool wear compensation values, tool tip radius, and tip orientation. The data is displayed on the *Offsets Table* tab of the *Offsets* screen of main notebook.

## 6.3   Part-programs Language

**IMPORTANT!** *Do not use a word processor to create or edit G-code files. A word processor leaves unseen codes that cause problems and may prevent a G-code file from working. Use a text editor like Gedit or Notepad++ to create or edit the files.*

### 6.3.1   Overview

The program language is based on lines of code. Each line, also called a block, may include commands to the machining system to do several different things. Lines of code may be collected in a file to make a program.

A typical line of code consists of an optional line number at the beginning followed by one or more words. A word consists of a letter followed by a number (or strictly something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, G01 X3 is a valid line of code with two words. G01 is a command meaning move in a straight line at the programmed feed rate, and X3 provides an argument value (the value of X should be 3 at the end of the move).

Most commands start with either G (general) or M (miscellaneous). The words for these commands are called, respectively, G-codes and M-codes.

The language has two commands (M02 or M30), execution of either of which ends a program. A program may end before the end of a file. Lines of a file that occur after the end of a program are not to be executed in the normal flow, so generally they're parts of subroutines.

### 6.3.2 Parameters

***NOTE:** There are significant differences between controls in the way parameters work. Do not assume that code from another control works in the same way with PathPilot. Tormach advises against writing parametric G-code as this is difficult to debug and very difficult for another operator to understand. Modern CAM virtually eliminates the need for it.*

PathPilot maintains an array of 10,320 numerical parameters. Many of them have specific uses. The parameters that are associated with fixtures are persistent over time. Other parameters are undefined when PathPilot is loaded. The parameters are preserved when the interpreter is reset. Parameters 1-1000 can be used by the code of part-programs.

### 6.3.3 Coordinate Systems

PathPilot has an absolute coordinate system and 9 work offset (fixture) systems.

You can set the offsets of tools by using `G10 L1 P~ X~ Z~`. The P word defines the tool number to be set.

You can set the offsets of the fixture systems using `G10 L2 P~ X~ Y~ Z~ A~ B~ C~`. The P word defines the fixture to be set. The X, Y, Z, etc. words are the coordinates for the origin of the axes in terms of the absolute coordinate system.

You can select one of the first seven work offsets by using G54, G55, G56, G57, G58 or G59. Any of the 254 work offsets can be selected by `G59 P~` (e.g., `G59 P23` would select fixture 23). The absolute coordinate system can be selected by `G59 P0`.

You can offset the current coordinate system using G92 or G92.3. This offset is then applied on top of work offset coordinate systems. This offset may be cancelled with G92.1 or G92.2.

You can make straight moves in the absolute machine coordinate system by using G53 with either G0 or G01.

### 6.3.4 Formatting G-code Lines (Block)

A permissible line of input code consists of the following, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line.

- Block delete character, which is a forward slash (/)
- Optional line number
- Any number of words, parameter settings, and comments
- End of line marker (carriage return or line feed or both)

Any input not explicitly allowed is illegal and causes the interpreter to either signal an error or to ignore the line.

Currently, programs are limited to 999,999 lines of code.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. This makes some strange-looking input legal. For example, the line `g0x +0. 12 34y 7` is equivalent to `g0 x+0.1234 y7`

Blank lines are allowed in the input, but are ignored.

Input is case insensitive, except in comments, therefore any letter outside a comment may be in upper or lower case without changing the meaning of a line.

### 6.3.5   Line Number

A line number is the letter N followed by an integer (with no sign) between 0 and 99,999,999 written without commas. Line numbers may be repeated or used out of order, although normal practice is to avoid such usage. A line number is not required to be used (and this omission is common), but it must be in the proper place if it is used.

### 6.3.6   Subroutine Labels

A subroutine label is the letter O followed by an integer (with no sign) between 0 and 99,999 written with no more than five digits (000009 is not permitted, for example). Subroutine labels may be used in any order but must be unique in a program. Nothing else except a comment should appear on the same line as a subroutine label.

*NOTE: Line numbers are not permitted with an O word in the current release of PathPilot.*

### 6.3.7   Word

A word is a letter other than N or O followed by a real value. Words may begin with any of the letters detailed in *Word Initial Letters* table later in this section**.** The table includes N and O for completeness, even though, as defined above, line numbers are not words. Several letters (I, J, K, L, P and R) may have different meanings in different contexts. A real value is some collection of characters that can be processed to come up with a number. A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression or a unary operation value. Definitions of these follow (for more information, refer to *Word Initial Letters* table). Processing characters to come up with a number is called evaluating. An explicit number evaluates to itself.

### 6.3.8   Number

The following rules are used for (explicit) numbers. In these rules, a digit is a single character between 0 and 9.

- A number consists of:
  - (1) an optional plus or minus sign, followed by
  - (2) zero to many digits, followed, possibly, by
  - (3) one decimal point, followed by
  - (4) zero to many digits, provided that there is at least one digit somewhere in the number.

- There are two kinds of numbers: integers and decimals. An integer does not have a decimal point; a decimal does.

- Numbers may have any number of digits, subject to the limitation on line length. Only about 17 significant figures are retained, however (enough for all known applications).

- A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required. A number written with initial or trailing zeros has the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes by PathPilot are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close to integers; this includes the values of indexes (for parameters and carousel slot numbers, for example), M-codes, and G-codes multiplied by 10.

A decimal number which is supposed to be close to an integer is considered close enough if it is within 0.0001 of an integer.

### 6.3.9 Comments and Messages

Comments can be added to lines of G-code to help clear up the intention of the programmer. Comments can be embedded in a line using parentheses ( ) or for the remainder of a line using a semicolon. The semicolon is not treated as the start of a comment when enclosed in parentheses.

Comments may appear between words, but not between words and their corresponding parameter.

**Example:**

`S100(set speed)F200(feed)` is OK while

`S(speed)100F(feed)` is not.

If there are several comments on a line, only the last comment is interpreted according to these rules. Hence, a normal comment following an active comment in effect disables the active comment.

**Example:**

`(foo) (msg, Start machining cycle1)` shows the message, however

`msg, Start machining cycle1)(foo)` does not.

A comment introduced by a semicolon is, by definition, the last comment on that line and is always interpreted for active comment syntax.

If the comment occurs on a line with `M00` or `M01` and contains a file name with a .jpg or .png extension, PathPilot displays the image in the tool path window when it reaches a programmed M00 or M01 break.

1. Move an image file with a .jpg or .png extension to your PathPilot controller in one of the following locations:

   • In the same folder as the G-code program file

   • In a folder called images within the G-code program file's folder

   • In a folder called images within the home directory

2. Program an `M00` or `M01` break.

3. Using parentheses ( ), embed a comment within the line of G-code.

4. Type the file name of the image within the comment.

**NOTE:** *Ensure the file name has either a .jpg or .png extension.*

**Example**

`M01 (photo_of_my_setup.jpg)`

`(MSG,)` – displays message if MSG appears after the left parenthesis and before any other printing characters. Variants of MSG, which include white space and lower case characters, are allowed. The rest of the characters before the right parenthesis are considered to be a message. Messages are displayed on the *Status* screen.

## 6.3.10 Item Repeats

A line may have any number of G words, but two G words from the same modal group can not appear on the same line.

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line. For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, `#3=15  #3=6`, for example, only the last setting takes effect. It is illogical but not illegal to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one is used. Each of the other comments is read and its format is checked, but ignored thereafter. It is expected that putting more than one comment on a line is very rare.

# Programming

## Word Initial Letters

| Letter | Meaning |
| --- | --- |
| A | A-axis of machine |
| B | B-axis of machine |
| C | C-axis of machine |
| D | Tool radius compensation number |
| F | Feed rate |
| G | General function (see *Modal Groups* table) |
| H | Tool length offset index |
| I | X-axis offset for arcs<br>X offset in G87 canned cycle |
| J | Y-axis offset for arcs<br>Y offset in G87 canned cycle |
| K | Z-axis offset for arcs<br>Z offset in G87 canned cycle |
| L | Number of repetitions in canned cycles/subroutines<br>key used with G10 |
| M | Miscellaneous function (see *Modal Groups* table) |
| N | Line number |
| O | Subroutine label number |
| P | Dwell time in canned cycles<br>dwell time with G04<br>key used with G10<br>tapping depth in M871 – M874 |
| Q | Feed increment in G83 canned cycle<br>repetitions of subroutine call |
| R | Arc radius<br>canned cycle retract level |
| S | Spindle speed |
| T | tool selection |
| U | Synonymous with A |
| V | Synonymous with B |
| W | Synonymous with C |
| X | X-axis of machine |
| Y | Y-axis of machine |
| Z | Z-axis of machine |

### 6.3.11  Item Order

The three types of items whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of items are divided into three groups by type. The first group (the words) may be reordered in any way without changing the meaning of the line.

If the second group (the parameter settings) is reordered, there is no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter takes effect. For example, after the line `#3=15  #3=6` has been interpreted, the value of parameter 3 is 6. If the order is reversed to `#3=6  #3=15` and the line is interpreted, the value of parameter 3 is 15. If the third group (the comments) contains more than one comment and is reordered, only the last comment is used.

If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line.

Example:

The line `g40 g01 #3=15 (so there!) #4=-7.0` has five items and means exactly the same thing in any of the 120 possible orders – such as `#4=-7.0 g01 #3=15 g40 (so there!)` – for the five items.

### 6.3.12  Commands and Machine Modes

PathPilot has many commands, called modal commands, that cause a machining system to change from one mode to another. The mode stays active until some other command changes it implicitly or explicitly. For example, if coolant is turned on, it stays on until it is explicitly turned off.

The G-codes for motion are also modal. If a G01 (straight move) command is given on one line, for example, it is executed again on the next line if one or more axis words is available on the line, unless an explicit command is given on that next line using the axis words or canceling motion. In contrast, non-modal codes have effect only on the lines on which they occur. For example, G04 (dwell) is non-modal.

## 6.4  Modal Groups

Modal commands are arranged in sets called modal groups, and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time (i.e., measure in inches vs. measure in millimeters). A machining system may be in many modes at the same time, with one mode from each modal group being in effect. For several modal groups, when a machining system is ready to accept commands, one member of the group must be in effect. There are default settings for these modal groups. When the machining system is turned on or re-initialized, default values are automatically in effect.

Group 1 is a group of G-codes for motion. One of these is always in effect, called the current motion mode. It is an error to put a G-code from group 1 and a G-code from group 0 on the same line if both of them use axis words. If an axis word-using G-code from group 1 is implicitly in effect on a line (by having been activated on an earlier line) and a group 0 G-code that uses axis words appears on the line, the activity of the group 1 G-code is suspended for that line. The axis word-using G-codes from group 0 are G10, G28, G30 and G92. PathPilot displays current mode on the screen.

**Modal Groups**

| Modal Group | Group # | Specific Code |
|---|---|---|
| For G-codes | Group 1 | = {G00, G01, G02, G03, G33, G38.x, G76, G80, G81, G82, G84, G85, G86, G87, G88, G89} motion |
| | Group 2 | = {G17, G18, G19, G17.1, G17.2, G17.3} plane selection |
| | Group 3 | = {G90, G91} distance mode |
| | Group 4 | = {G90.1, G91.1} arc IJK distance mode |
| | Group 5 | = {G93, G94} feed rate mode |
| | Group 6 | = {G20, G21} units |
| | Group 7 | = {G40, G41, G42, G41.1, G42.1} cutter radius compensation |
| | Group 8 | = {G43, G43.1, G49} tool length offset |
| | Group 10 | = {G98, G99} return mode in canned cycles |
| | Group 12 | = {G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3} coordinate system selection |
| | Group 13 | = {G61, G61.1, G64} path control mode |
| | Group 14 | = {G96, G97} spindle speed mode |
| | Group 15 | = {G07, G08} lathe diameter mode |
| For M-codes | Group 4 | = {M0, M1, M2, M30, M60} stopping |
| | Group 7 | = {M3, M4, M5} spindle turning |
| | Group 8 | = {M7, M8, M9} coolant (special case: M7 and M8 may be active at same time) |
| | Group 9 | = {M48, M49} enable/disable feed and speed override controls |
| | Group 10 | = {user defined M100 to M199} |
| For non-modal G-codes | Group 0 | = {G4, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3} |

## 6.5  G-codes

The supported G-codes are shown and described in more detail in this section. The descriptions contain command examples set in `Courier` type font.

**G-code Table**

| G-code | Summary |
| --- | --- |
| G00 | Rapid positioning |
| G01 | Linear interpolation |
| G02 | Clockwise circular interpolation |
| G03 | Counterclockwise circular interpolation |
| G04 | Dwell |
| G07 | Diameter mode |
| G10 L1 | Set Tool Table Entry |
| G10 L10 | Set Tool Table – calculated - Workpiece |
| G10L11 | Set Tool Table – calculated - Fixture |
| G10 L2 | Set work offset origin |
| G10 L20 | Set work offset origin - calculated |
| G17, G18, G19 | Plane selection |
| G20/G21 | Inch/millimeter unit |
| G28 | Return home |
| G28.1 | Reference axes |
| G30 | Return home |
| G33 | Spindle sync. motion (e.g. threading) |
| G33.1 | Rigid tapping |
| G40 | Cancel cutter radius compensation |
| G41/G42 | Start cutter radius compensation left/right |
| G41.1, G42.1 | Dynamic cutter compensation |
| G43 | Apply tool length offset – not used on lathe |
| G49 | Cancel tool length offset |
| G53 | Move in absolute machine coordinate system |
| G54 | Use fixture offset 1 |
| G55 | Use fixture offset 2 |
| G56-58 | Use fixture offset 3, 4, 5 |
| G59 | Use fixture offset 6 / use general fixture number |
| G61/G61.1 | Path control mode |
| G64 | Path control with optional tolerance |
| G76 | Multi-pass threading cycle |
| G80 | Cancel motion mode (including canned cycles) |
| G90, G90.1 | Absolute distance mode |
| G91, G91.1 | Incremental distance mode |
| G92 | Offset coordinates and set parameters |
| G92.x | Cancel G92 etc. |

(continued on next page...)

**G-code Table   (...continued)**

| G-code | Summary |
|---|---|
| G93, G94, G95 | Feed modes |
| G96, G97 | CSS, RPM modes |
| G98 | Initial level return / R-point level after canned cycles |

In the command examples, the tilde symbol (~) stands for a real value. A real value may be:

- An explicit number (4.4, for example)

- An expression ([2+2.4], for example)

- A parameter value (#88, for example)

- A unary function value (acos[0], for example)

In most cases, if axis words (X~ or Z~) are given, they specify a destination point. Axis numbers relate to the currently active coordinate system, unless explicitly described as being in the absolute coordinate system. Where axis words are optional, any omitted axes will have their current value. Any items in the command examples not explicitly described as optional are required. It is an error if a required item is omitted.

In the examples, the values following letters are often given as explicit numbers. Unless stated otherwise, the explicit numbers can be real values. For example, G10 L2 could equally well be written G[2*5] L[1+1]. If the value of parameter 100 were 2, G10 L#100 would also mean the same. Using real values which are not explicit numbers as shown in examples is rarely useful. If L~ is written in a example, the "~" is often referred to as the L number. Similarly the "~" in H~ may be called the H number, and so on for any other letter.

### 6.5.1   Rapid Linear Motion – G00

For rapid linear motion, program G00 X~ Z~ where the axis words are optional, except that at least one must be used. The G00 is optional if the current motion mode is G00. This produces coordinated linear motion to the destination point at the current traverse rate (or slower if the machine does not go that fast). It is expected that cutting won't take place when a G00 command is executing. It is an error if all axis words are omitted.

If cutter radius compensation is active, the motion differs from the above; for more information, refer to section *G41, G42 – Cutter Compensation* later in this chapter. If G53 is programmed on the same line, the motion also differs; for more information, refer to section *G90 – Absolute Coordinates* later in this chapter. Depending on where the tool is located, there are two basic rules to follow for safety: if the Z value represents a cutting move in the positive direction (i.e., out of a hole), the X-axis should be moved first. If the Z value represents a move in the positive direction, the X-axis should be executed last.

### 6.5.2    Linear Motion at Feed Rate – G01

For linear motion at feed rate (for cutting or not), program `G01  X~  Z~`, where the axis words are optional, except that at least one must be used. The G01 is optional if the current motion mode is G01. This produces coordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast). It is an error if all axis words are omitted.

If cutter radius compensation is active, the motion differs from the above; for more information, refer to section *G41, G42 – Cutter Compensation* later in this chapter. If G53 is programmed on the same line, the motion also differs; for more information, refer to section *G90 – Absolute Coordinates* later in this chapter.

### 6.5.3    Arc at Feed Rate – G02, G03

A circular or helical arc is specified using either G02 (clockwise arc) or G03 (counterclockwise arc). The axis of the circle is normal to the XZ plane of the machine coordinate system. The direction is viewing from above the lathe.

If cutter radius compensation is active, the motion differs from the above; for more information, refer to section *G41, G42 – Cutter Compensation* later in this chapter. Two formats are allowed for specifying an arc. We call these the center format and the radius format. In both formats, the G02 or G03 is optional if it is the current motion mode.

#### 6.5.3.1    Radius Format Arc

In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program `G02  X~  Z~  R~` (or use G03 instead of G02). The axis words are optional except that at least one must be used. The R number is the radius. A positive radius indicates that the arc turns through 180° or less, while a negative radius indicates a turn of 180° to 359.999°. The arc center is absolute or relative as set by G90.1 or G91.1 respectively.

It is an error if:

- Both of the axis words are omitted
- No R word is given
- The end point of the arc is the same as the current point

#### 6.5.3.2    Center Format Arc

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the center of the arc. Program G02 X~ Z~ I~ K~ (or use G03 instead of G02). The axis words and arc center words are optional except that at least one of each type must be used.  I and K specify the distance offset of the center of the arc from the start point of the arc in X and Z, respectively.

It is an error if:
- Both of the axis words are omitted
- Both I and K are omitted
- The end point of the arc is the same as the current point

### 6.5.4 Dwell – G04

For a dwell, program `G04 P~`. This keeps the axes unmoving for the period of time in seconds specified by the P number.

It is an error if:

- The P number is negative.

```
G04 P4.2 (To wait 4.2 seconds)
```

### 6.5.5 Lathe Diameter Mode – G07

The standard operating mode where X1.2 turns a part with diameter 2.4.

### 6.5.6 Set Offsets – G10

Offsets are generally set up using the graphical PathPilot interface, but can be set programmatically.

### 6.5.7 Set Tool Table – G10 L1

To define an entry in the tool table, program `G10 L1 P~ X~ Y~ R~ I~ J~ Q~`

- P – tool number
- R – radius of tool
- I – front angle (lathe)
- J – back angle (lathe)
- Q – orientation (lathe)

G10 L1 sets the tool table for the P tool number to the values of the words.

A valid G10 L1 rewrites and reloads the tool table.

**Example:**

```
G10 L1 P2 R0.015 Q3 (setting tool 2 radius to 0.015 and orientation
to 3)
```

It is an error if:

- Cutter Compensation is on
- The P number is unspecified
- The P number is not a valid tool number from the tool table
- The P number is 0

For more information on cutter orientation used by the Q word, see **Figure 6.2**.

### 6.5.8   Set Coordinate System – G10 L2

- To define the origin of a work offset coordinate system, program `G10 L2 P-`

- P = number of coordinate system to use (`G54 = 1, G59.3 = 9`)

**Important Concepts:**

- G10 L2 PN does not change from the current coordinate system to the one specified by P; you have to use G54-59.3 to select a coordinate system.

- If a G92 origin offset was in effect before `G10 L2`, it continues to be in effect afterwards.

- The coordinate system whose origin is set by a `G10` command may be active or inactive at the time the `G10` is executed. If it is currently active, the new coordinates take effect immediately.

It is an error if:

- The P number does not evaluate to an integer in the range 0 to 9

- An axis other than X or Z is programmed

### 6.5.9   Set Tool Table – G10 L10

`G10 L10 P- X~ Z~ R~ I~ J~ Q~`

- P – tool number

- R – radius of tool

- I – front angle

- J – back angle

- Q – orientation

G10 L10 changes the tool table entry for tool P so that if the tool offset is reloaded, with the machine in its current position and with the current G5x and G92 offsets active, the current coordinates for the given axes will become the given values. The axes that are not specified in the G10 L10 command are not changed. This could be useful with a probe move as described in the G38 section.

It is an error if:

- Cutter Compensation is on

- The P number is unspecified

- The P number is not a valid tool number from the tool table

- The P number is 0

### 6.5.10 Set Tool Table – G10 L11

`G10 L11 P~ X~ Z~ R~ I~ J~ Q~`

- P – tool number
- R – radius of tool
- I – front angle (lathe)
- J – back angle (lathe)
- Q – orientation (lathe)

G10 L11 is just like G10 L10, except that instead of setting the entry according to the current offsets, it is set so that the current coordinates would become the given value if the new tool offset is reloaded and the machine is placed in the G59.3 coordinate system without any G92 offset active. This allows the operator to set the G59.3 coordinate system according to a fixed point on the machine, and then use that fixture to measure tools without regard to other currently active offsets.

It is an error if:

- Cutter Compensation is on
- The P number is unspecified
- The P number is not a valid tool number from the tool table
- The P number is 0

### 6.5.11 Set Coordinate System – G10 L20

`G10 L20 P- X~ Z~`

- P – coordinate system (0-9)

G10 L20 is similar to G10 L2 except that instead of setting the offset/entry to the given value, it is set to a calculated value that makes the current coordinates become the given value.

- The P number does not evaluate to an integer in the range 0 to 9
- An axis other than X or Z is programmed

### 6.5.12 Plane Selection – G17, G18, and G19

The lathe should always be operated in G18 (XZ-plane). Do not use G17 or G19.

### 6.5.13 Length Units – G20 and G21

Program `G20` to use inches for length units and program `G21` to use millimeters. It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs and not to use either one anywhere else in the program. It is the responsibility of the operator to be sure all numbers are appropriate for use with the current length units.

### 6.5.14 Return to Pre-defined Position – G28, G28.1

G28 uses the values stored in parameters 5161 and 5163 as the X and Z final point to move to. The parameter values are absolute machine coordinates in the native machine units of inches.

- `G28` makes a rapid traverse move from the current position to the absolute position of the values in parameters.

- `G28 X~ Z~` makes a rapid traverse move to the position specified by axes including any offsets, then makes a rapid traverse move to the absolute position of the values in parameters 5161 and/or 5163. Any axis not specified will not move.

- `G28.1` stores the current absolute position into parameters 5161-5163.

It is an error if:

- Cutter Compensation is turned on

### 6.5.15 Return to Pre-defined Position – G30, G30.1

G30 uses the values stored in parameters 5181 and 5183 as the X and Z final point to move to. The parameter values are absolute machine coordinates in the native machine units of inches.

A Tool-change request, `Tnnww`, also causes a G30 to be executed.

- `G30` makes a rapid traverse move from the current position to the absolute position of the values in parameters.

- `G30 X~ Z~` makes a rapid traverse move to the position specified by axes including any offsets, then makes a rapid traverse move to the absolute position of the values in parameters 5181 and/or 5183. Any axis not specified won't move.

- `G30.1` stores the current absolute position into parameters 5181-5183.

It is an error if:

- Cutter Compensation is turned on

### 6.5.16 Spindle Synchronized Motion – G33

To make a cut – like a thread – where the spindle and the tool motion are synchronized, program `G33 X~ Z~ K~`

- K = distance per revolution

For spindle-synchronized motion in one direction, program `G33 X~ Z~ K~` where K gives the distance moved in X and Z for each revolution of the spindle. For instance, if starting at Z=0, `G33 Z-1 K.0625` produces a 1 inch motion in Z over 16 revolutions of the spindle. This command might be part of a program to produce a 16TPI thread.

Another example in metric, `G33 Z-15 K1.5` produces a movement of 15 mm while the spindle rotates 10 times for a thread of 1.5 mm pitch.

Spindle-synchronized motion waits for the spindle index and spindle at speed signals from the machine so multiple passes line up. G33 moves end at the programmed endpoint. Thus, G33 could be used to cut tapered threads, a face scroll like in a 3-jaw chuck.

The axis words are optional, except that at least one must be used.

**NOTE:** *K follows the drive line described by $X\sim$ $Z\sim$. K is not parallel to the Z axis if X endpoint is used, for example, when cutting tapered threads.*

At the beginning of each G33 pass, PathPilot uses the spindle speed and the machine acceleration limits to calculate how long it takes Z to accelerate after the index pulse, and determines how many degrees the spindle rotates during that time. It then adds that angle to the index position and computes the Z position using the corrected spindle angle. That means that Z reaches the correct position just as it finishes accelerating to the proper speed, and can immediately begin cutting a good thread.

**G33 Example:**

```
G90 (absolute distance mode)
G0 X1 Z0.1 (rapid to position)
S100 M03 (start spindle turning)
G33 Z-2 K0.125 (move Z axis to -2 at a rate to equal 0.125 per revolution)
G0 X1.25 (rapid move tool away from work)
Z0.1 (rapid move to starting Z position)
M2 (end program)
```

For more information, refer to G90, G0, and M2 sections later in this chapter.

It is an error if:

- One axis word is not present
- The spindle is not turning when this command is executed
- The requested linear motion exceeds machine velocity limits due to the spindle speed

### 6.5.17   Rigid Tapping – G33.1

For rigid tapping, program `G33.1 Z~ K~`

- K – distance per revolution

For rigid tapping (spindle synchronized motion with return), program `G33.1  Z~  K~` where K~ gives the distance moved for each revolution of the spindle. A rigid tapping move consists of the following sequence.

⚠️ ***CAUTION! Crash Hazard:*** *If the X coordinate is specified non-zero or the current X coordinate is non-zero (i.e., not tapping a hole in the axis of the workpiece), a crash is likely.*

1. A move to the specified Z coordinate, synchronized with the spindle at the given ratio and starting with a spindle index pulse.

2. When reaching the endpoint, a command to reverse the spindle (i.e., from forward to reverse).

3. Continued synchronized motion beyond the specified end coordinate until the spindle actually stops and reverses.

4. Continued synchronized motion back to the original coordinate.

5. When reaching the original coordinate, a command to reverse the spindle a second time (i.e., from reverse to forward).

6. Continued synchronized motion beyond the original coordinate until the spindle actually stops and reverses.

7. An unsynchronized move back to the original coordinate.

Spindle-synchronized motions wait for spindle index, so multiple passes line up. G33.1 moves end at the original coordinate.

The Z-axis word must be used.

**G33.1 Example:**

```
G90 (set absolute mode)
G0 X0 Z0.100 (rapid move to starting position)
G33.1 Z-0.750 K0.05 (rigid tap a 20 TPI thread 0.750 deep)
M2 (end program)
```

See G90, G0, and M2 sections for more information.

It is an error if:

- The Z-axis word is omitted
- The spindle is not turning when this command is executed
- The requested linear motion exceeds machine velocity limits due to the spindle speed

### 6.5.18    Compensation Off – G40

`G40` – Turn cutter compensation off. If tool compensation was on, the next move must be a linear move and longer than twice the tool tip radius. It is OK to turn compensation off when it is already off.

It is an error if:

- A G2/G3 arc move is programmed next after a G40
- The linear move after turning compensation off is less than twice the tool tip radius

### 6.5.19    Cutter Compensation – G41, G42

`G41 D~` (left of programmed path)

`G42 D~` (right of programmed path)

where D = tool number

The D word is optional; if there is no D word, the radius of the currently loaded tool is used. If no tool is loaded and no D word is given, a radius of 0 is used.

If supplied, the D word is the tool number to use.

To start cutter compensation to the left of the part profile, use G41. G41 starts cutter compensation to the left of the programmed line as viewed looking down on the lathe.

To start cutter compensation to the right of the part profile, use G42. G42 starts cutter compensation to the right of the programmed line as viewed looking down on the lathe.

The lead in move must be at least as long as the tool radius. The lead in move can be a rapid move. User M100-M199 commands are allowed when Cutter Compensation is on.

The behavior of the 15L Slant-PRO lathe when cutter compensation is on is described in the Cutter Compensation section.

It is an error if:

- The D number is not a valid tool number or 0
- Cutter Compensation is commanded to turn on when it is already on

### 6.5.20    Dynamic Cutter Compensation – G41.1, G42.1

`G41.1 D~ L~` (left of programmed path)

`G42.1 D~ L~` (right of programmed path)

where D = twice tip radius, and L = tool orientation

G41.1 and G42.1 function the same as G41 and G42 with the added scope of being able ignore the tool table and to program the tool diameter. The L word defaults to 0 if unspecified.

It is an error if:

- The L number is not in the range from 0 to 9 inclusive
- Cutter compensation is commanded to turn on when it is already on

### 6.5.21 Move in Absolute Coordinates – G53

For linear motion to a point expressed in absolute coordinates, program `G01 G53 X~ Z~` (or similarly with G0 instead of G1), where all the axis words are optional except that at least one must be used. The G0 or G01 is optional if it is in the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This produces coordinated linear motion to the programmed point. If G01 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

- G53 is used without G0 or G01 being active
- G53 is used while cutter radius compensation is on

For more information, refer to section *Coordinate Systems* earlier in this chapter.

### 6.5.22 Select Work Offset Coordinate System – G54 to G59 and G59 P~

G54 – Select coordinate system 1

G55 – Select coordinate system 2

G56 – Select coordinate system 3

G57 – Select coordinate system 4

G58 – Select coordinate system 5

G59 – Select coordinate system 6

G59.1 – Select coordinate system 7

G59.2 – Select coordinate system 8

G59.3 – Select coordinate system 9

It is an error if one of these G-codes is used while cutter radius compensation is on.

The X- and Z-axis work offset values are stored in parameters corresponding to the system in use (i.e., System 1 X=5221, Z=5223; System 2 X=5141, Z=5143; up to System 9 X= 5381, Z = 5383).

For more information, refer to section *Coordinate Systems* earlier in this chapter.

### 6.5.23    Set Path Control Mode – G61, G61.1

Program `G61` to put the machining system into exact path mode, or program `G61.1` for exact stop mode which behaves in an identical way. The tool comes to a stop at corners that are sharp.

### 6.5.24    Set Blended Path Control Mode – G64

Program `G64 P~ Q~` to attempt to maintain the defined feed velocity.

The P~ word specifies a permitted deviation to round corners and so maintain speed. If P is omitted, then the speed is maintained however far from the programmed path the tool cuts.

If Q~ is not present, then it turns on the naive cam detector; when there are a series of linear XZ feed moves at the same feed rate that are less than Q~ away from being collinear, they are collapsed into a single linear move.

It is OK to program for the mode that is already active.

### 6.5.25    Threading Cycle – G76

To cut a thread in multiple passes, program
`G76 P~ Z~ I~ J~ R~ K~ Q~ H~ E~ L`

- Drive Line – A line through the initial X position parallel to the Z

- P~ – The thread pitch in distance per revolution

- Z~ – The final position of threads; at the end of the cycle the tool is at this Z position
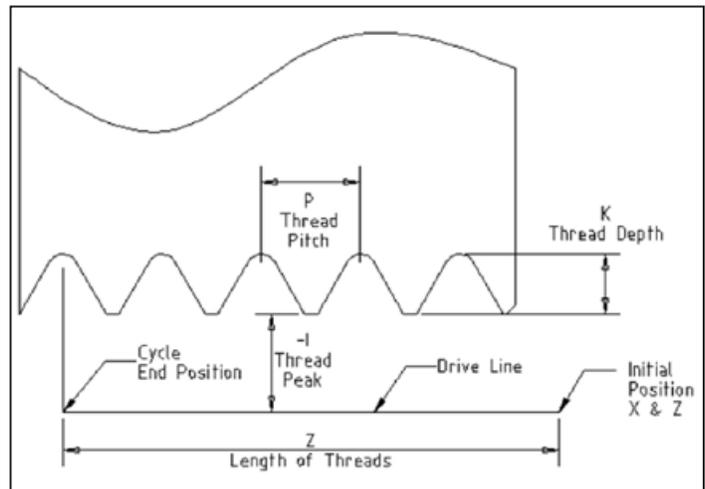


**Figure 6.4**

**NOTE:** *As G07 Lathe Diameter Mode is always in force, the values for I, J and K are diameter measurements.*

- I~ – The thread peak offset from the drive line. Negative I values are external threads; positive I values are internal threads. Generally the material has been turned to this size before the G76 cycle.

- J~ – A positive value specifying the initial cut depth. The first threading cut is J beyond the thread peak position.

- K~ – A positive value specifying the full thread depth. The final threading cut is K beyond the thread peak position.

**NOTE:** *Signs (either negative or positive) of I, J, and K values discussed in this section assume a rear tool post configuration. If using a front tool post configuration (either quick change or gang tooling), reverse the sign of I, J, and K.*

### 6.5.25.1   Optional Settings

- R~ – The depth degression. R1.0 selects constant depth on successive threading passes. R2.0, which is usual, selects constant area. Values between 1.0 and 2.0 select decreasing depth but increasing area. Values above 2.0 select decreasing area. Beware that unnecessarily high degression values causes a large number of passes to be used (degression = a descent by stages or steps).

- Q~ – The compound slide angle is the angle (in degrees) describing to what extent successive passes should be offset along the drive line. This is used to cause one side of the tool to remove more material than the other. A positive Q value causes the leading edge of the tool to cut more heavily. Typical values for threads with a 60° angle are 29, 29.5 or 30.

- H~ – The number of spring passes. Spring passes are additional passes at full thread depth used to allow for any tool or workpiece deflection during the main cuts. If no additional passes are desired (for example on a work-hardening material), program H0.

- E~ – Specifies the distance along the drive line used for the entry/exit taper. The angle of the taper is so the last pass tapers to the thread crest over the distance specified with E.' E0.2' will give a taper for the first/last 0.2 length units along the thread. For a 45° entry/exit taper, program E the same as K.

- L~ – Specifies which ends of the thread get the taper. Program L0 for no taper (the default), L1 for entry taper, L2 for exit taper, or L3 for both entry and exit tapers. Entry tapers pause at the drive line to synchronize with the index pulse then feed in to the beginning of the taper. No entry taper and the tool will rapid to the cut depth, then synchronize and begin the cut.

The tool should be moved to the initial X and Z positions prior to issuing the G76. The X position is the drive line and the Z position is the start of the threads.

The tool pauses briefly for synchronization before each threading pass, so a relief groove is required at the entry unless the beginning of the thread is past the end of the material or if an entry taper is used.

Unless using an exit taper, the exit move (traverse to original X) is not synchronized to the spindle speed. With a slow spindle, the exit move might take only a small fraction of a revolution. If the spindle speed is increased after several passes are complete, subsequent exit moves requires a larger portion of a revolution, resulting in a very heavy cut during the exit move. This can be avoided by providing a relief groove at the exit, or by not changing the spindle speed while threading.

The final position of the tool is at the end of the drive line. A safe Z move is needed with an internal thread to remove the tool from the hole.

It is an error if:
- Other axis words, such as X-, are specified
- The R- degression value is less than 1.0
- All the required words are not specified
- P~, J~, K~ or H~ is negative

- E~ is greater than half the drive line length

The G76 canned cycle is based on the G33 Spindle Synchronized Motion. For more information, refer to section *G33 – Spindle Synchronized Motion* earlier in this chapter.

### 6.5.26 Tapping Cycle – G84

The G84 cycle is intended for tapping. This cycle rotates the spindle clockwise to tap a pre-drilled hole; when the bottom of the hole is reached, the spindle rotates in the reverse direction and exits the hole.

This cycle uses a P word, where P specifies the number of seconds to dwell. The P word is optional – if it is not included, PathPilot calculates a dwell for you (half of a second per 1000 RPM).

Program: `G84 X~ Y~ Z~ R~ P~ F~`

| Word | Definition |
|------|-----------|
| X~ | The position of the hole on the X-axis. |
| Y~ | The position of the hole on the Y-axis. |
| Z~ | The depth, tapping from the R-plane to Z-depth. |
| R~ | The position of the retract plane (R-plane). |
| P~ | The number of seconds to dwell. |
| F~ | The federate. |

- Spindle speed must be commanded before calling a G84 cycle.
- Feedrate override is ignored during a tapping cycle.
- Feedhold is ignored until the return operation is executed.
- After the tapping operation is completed, either a G98 or G99 command controls the return height. G99 returns the tool to the R-plane (see **Figure 6.5**); G98 returns the tool to the initial height (see **Figure 6.6**).
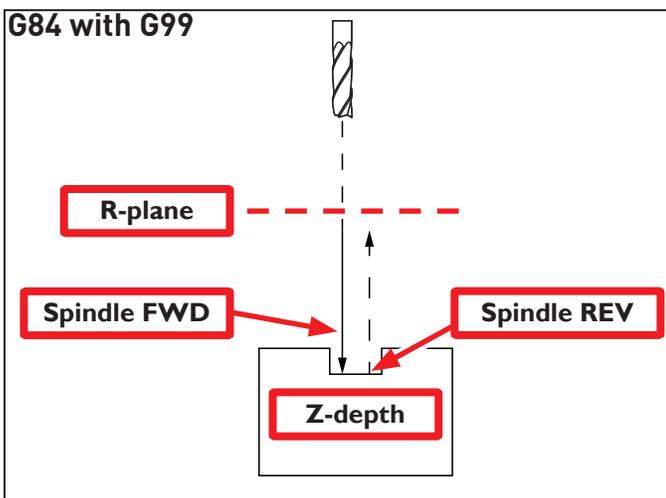


Figure 6.5
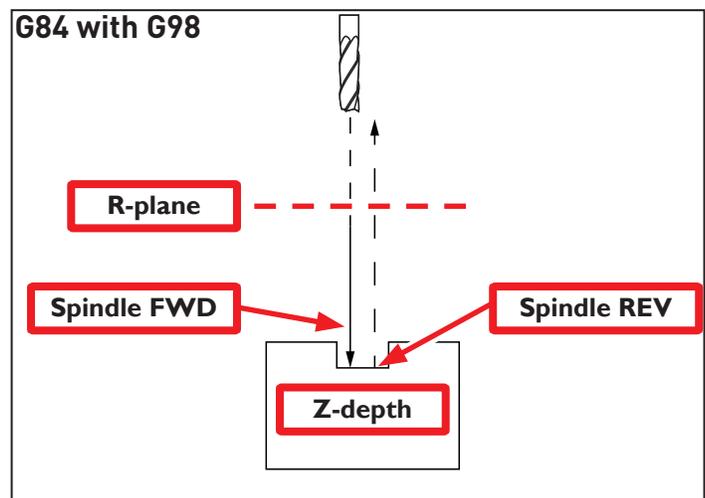


Figure 6.6

**G84 Example:**

```
N40 T51 G43 H51 M6
N45 S400 M3
N50 G54
N55 M8
N65 G0 X0.5 Y-0.75
N70 G43 Z0.6 H51
N80 G0 Z0.2
N85 S400
N90 G98 G84 X0.5 Y-0.75 Z-0.605 R0.2 F20.
N95 X1.0 Y -1.25
N100 G80
N105 G0 Z0.6
```

### 6.5.27 Distance Mode – G90, G91

Interpretation of the PathPilot-code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program `G90`. In absolute distance mode, axis numbers (X, Y, Z, A) usually represent positions in terms of the currently active coordinate system. Any exceptions to that rule are described explicitly in this section describing G-codes.

To go into incremental distance mode, program `G91`. In incremental distance mode, axis numbers (X, Y, Z, A) usually represent increments from the current values of the numbers.

I and J numbers always represent increments, regardless of the distance mode setting. K numbers represent increments.

### 6.5.28 Arc Distance Mode – G90.1, G91.1

`G90.1` – Absolute distance mode for I and K offsets. When G90.1 is in effect, I and K both must be specified with G2/3 for the XZ-plane, or it is an error.

`G91.1` – Incremental distance mode for I and K offsets. G91.1 Returns I and K to default behavior.

### 6.5.29 G92 Offsets – G92, G92.1, G92.2, and G92.3

This is a legacy feature. It is usually more straightforward to rely on work offsets.

To apply the offsets, program `G92 X~ Z~`

`G92.1` – Reset axis offsets to zero and set parameters 5211-5219 to zero

`G92.2` – Reset axis offsets to zero

`G92.3` – Set the axis offset to the values saved in parameters 5211-5219

G92 makes the current point have the coordinates you want (without motion), where the axis words contain the axis numbers you want. The axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

When G92 is executed, the origins of all coordinate systems move. They move such that the value of the current controlled point, in the currently active coordinate system, becomes the specified value. All coordinate system's origins are offset this same distance.

For example, suppose the current point is at X=4 and there is currently no G92 offset active. Then G92 X7 is programmed. This moves all origins -3 in X, which causes the current point to become X=7. This -3 is saved in parameter 5211.

Being in incremental distance mode has no effect on the action of G92.

G92 offsets may be already be in effect when the G92 is called. If this is the case, the offset is replaced with a new offset that makes the current point become the specified value.

It is an error if:

• All axis words are omitted

PathPilot stores the G92 offsets and reuses them on the next run of a program. To prevent this, one can program a G92.1 (to erase them), or program a G92.2 (to stop them being applied – they are still stored).

For more information, refer to *Coordinate System* section earlier in this chapter.

### 6.5.30   Feed Rate Mode – G93, G94, and G95

G93 – Inverse Time Mode. This mode is very unusual on a lathe with only X- and Z-axes. In inverse time feed rate mode, an F word means the move should be completed in [one divided by the F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute.

When the inverse time feed rate mode is active, an F word must appear on every line which has a G1, G2, or G3 motion, and an F word on a line that does not have G1, G2, or G3 is ignored. Being in inverse time feed rate mode does not affect G0 (rapid traverse) motions.

G94 – Units per Minute Mode. In this mode, an F word is interpreted to mean the controlled point should move at a certain number of inches per minute, or millimeters per minute, depending upon what length units are being used.

G95 – Units per Revolution Mode. In this mode, an F word is interpreted to mean the controlled point should move a certain number of inches per revolution of the spindle, depending on what length units are being used. G95 is not suitable for threading; for threading, use G33 or G76.

It is an error if:

- Inverse time feed rate mode is active and a line with G1, G2, or G3 (explicitly or implicitly) does not have an F word

- A new feed rate is not specified after switching to G94 or G95 Canned Cycle Return Level – G98 and G99

### 6.5.31  Spindle Control Mode – G96, G97

`G96 D~ S~` (Constant Surface Speed = CSS)

`G97` (RPM Mode)

D = Maximum spindle RPM

S = Surface speed

`G96 D~ S~` – Selects constant surface speed of S feet per minute (if G20 is in effect) or meters per minute (if G21 is in effect).

`D~` is optional.

When using G96 (the most common mode of lathe operation), X0 in the current coordinate system (including offsets and tool lengths) must be the spindle axis.

`G97` selects RPM mode.

**G96 Example Line:**

`G96 D2500 S250` (set CSS with a max rpm of 2500 and a surface speed of 250)

It is an error if:

- S is not specified with G96
- A feed move is specified in G96 mode while the spindle is not turning

# Programming

## 6.6 Built-in M-codes

M-codes interpreted directly by PathPilot are detailed in the following table.

| M-code | Meaning |
|---|---|
| M00 | Program stop |
| M01 | Optional program stop |
| M02 | Program end |
| M03 | Rotate spindle in the forward direction |
| M04 | Rotate spindle in the reverse direction |
| M05 | Stop spindle rotation |
| M07 or M08 | Coolant on |
| M09 | All coolant off |
| M30 | Program end and rewind |
| M48 | Enable speed and feed override |
| M49 | Disable speed and feed override |
| M98 | Call subroutine |
| M99 | Return from subroutine/repeat |
| M100-M199 | Operator defined M-codes |

### 6.6.1 Program Stop and Program End – M00, M01, M02, and M30

To stop a running program temporarily, regardless of the setting of optional stop switch, program `M00`.

To stop a running program temporarily, but only if the optional stop switch is on, program `M01`.

It is OK to program M00 and M01 in MDI mode, but the effect probably will not be noticeable because normal behavior in MDI mode is to stop after each line of input, anyway.

If a program is stopped by an M00 or M01, clicking the *Cycle Start* button restarts the program at the following line of the G-code program.

To end a program, program `M02` or `M30`. M02 leaves the next line to be executed as the M02 line. M30 rewinds the G-code file. These commands will have the following effects:

- Distance mode is set to absolute (like G90)

- Feed and speed overrides are set to *On* (like M48)

- Cutter compensation is turned off (like G40)

- The spindle is stopped (like M05)

- The current motion mode is set to G01 (like G1)

- Coolant is turned off (like M9)

No more lines of code in the file is executed after the M02 or M30 command is executed. Clicking *Cycle Start* starts the program back at the beginning of the file.

### 6.6.2 Spindle Control – M03, M04, and M05

To start the spindle turning reverse at the currently programmed speed, program `M03`.

To start the spindle turning forward at the currently programmed speed, program `M04`.

The speed is programmed by the S word.

To stop the spindle from turning, program `M05`.

It is OK to use M03 or M04 if the spindle speed is set to zero; if this is done (or if the speed override switch is enabled and set to zero), the spindle will not start turning. If, later, the spindle speed is set above zero (or the override switch is turned up), the spindle starts turning. It is permitted to use M03 or M04 when the spindle is already turning or to use M05 when the spindle is already stopped.

### 6.6.3 Coolant Control – M07, M08, and M09

To turn coolant on, program `M07` or `M08`.

To turn flood coolant on, program `M08`.

To turn all coolant off, program `M09`.

It is always OK to use any of these commands, regardless of what coolant is on or off.

### 6.6.4 Automatic Collet Closer Control – M10 and M11

To open the collet closer, program `M10`.

To close the collet closer, program `M11`.

### 6.6.5 Override Control – M48, M49

To enable the speed and feed override, program `M48`. To disable both overrides, program `M49`. It is OK to enable or disable the switches when they are already enabled or disabled.

### 6.6.6 Feed Override Control – M50

`M50 P1` – Enable the feed rate override control; P1 is optional

`M50 P0` – Disable the feed rate control

While disabled, the feed override slider will have no influence, and the motion is executed at programmed feed rate (unless there is an adaptive feed rate override active).

### 6.6.7 Spindle Speed Override Control – M51

`M51 P1` – Enable the spindle speed override control; P1 is optional

`M51 P0` – Disable the spindle speed override control

While disabled, the spindle speed override has no influence, and the spindle speed has the exact program specified value of the S-word (described in Spindle Speed section).

### 6.6.8 Set Current Tool Number – M61

`M61 Q~` – Change the current tool number while in MDI or Manual mode. One use is when you power on the system with a tool selected but the tool turret is set for a different tool to that indicated. You can set that tool number without doing a tool change operation.

It is an error if:

- Q~ is not 0 or greater.

### 6.6.9 Call Subroutine – M98

To call a subroutine, program `M98 P~ L~` or `M98 ~P ~Q`. The program must contain a letter O line with the number of the P word of the call (for instance O1, O125, O777). This O line is a label which indicates the start of the subroutine. The O line, and the associated G-code, is normally placed at the end of the program with other subroutines following an M2, M30 or M99, so it is not reached directly by the flow of the program.

The L word (or, optionally, the Q word) gives the number of times that the subroutine is to be called before continuing with the line following the M98. If the L (Q) word is omitted, value defaults to 1.

By using parameters, values, or incremental moves, a repeated subroutine can make several roughing cuts around a complex path or cut several identical objects from one piece of material.

Subroutine calls may be nested. That is to say a subroutine may contain a M98 call to another subroutine. As no conditional branching is permitted, it is not meaningful for subroutines to call themselves recursively.

### 6.6.10 Return from Subroutine – M99

To return from a subroutine, program `M99`. Execution continues after the M98 G-code which called the subroutine.

## 6.7 Other Input Codes

### 6.7.1 Feed Rate – F

To set the feed rate, program `F~`.

Depending on the setting of the *Feed Mode* toggle, the rate may be in units-per-minute or units-per-rev of the spindle. The units are those defined by the G20/G21 mode. The feed rate may sometimes be overridden (see *Override Control – M48 and M49* section earlier in this chapter).

### 6.7.2   Spindle Speed – S

To set the speed in revolutions per minute (rpm) of the spindle, program `S~`. The spindle turns at that speed when it has been programmed to start turning. It is OK to program an S word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100 percent, the speed is different from what is programmed. It is OK to program `S0`, but the spindle does turn if that is done.

It is an error if:

- The S number is negative

### 6.7.3   Change Tool – T

To select a tool, program `T~~`

The first number indicates the tool to be selected and the geometry offsets entry for it in the tool table. The second number, which is optional, indicates the line of the tool table giving the wear offsets to be applied.

Example:

`T03` – Selects tool number 3 with geometry offsets from line 3 of the tool table

`T0303` – Selects tool 3 and applies geometry and wear offset values from line 3 of the tool table

`T0309` – Selects tool 3 and wear offsets from line 9 of the tool table; not often used in practice

If the tool turret is installed, numbers T01 through T08 automatically actuates the turret to select the chosen tool (see chapter 5, *PathPilot Interface*, for more information). Numbers greater than T08 are assumed to be gang tools and the program continues after applying the relevant offsets.

If *Manual Tool Change* is selected then the machine stops flashing the display of the required tool number and the LED on the *Cycle Start* button. The program continues on clicking *Cycle Start*.

It is the programmer's responsibility to ensure that the carriage is in a safe place for changing tools, for example by using G30. This allows optimization of motion which can save a lot of time, especially with gang tooling. A pause for manual intervention can always be provided by an M00 or M01 before the tool change.

It is an error if:

- A negative T number is used or a T number larger than 54 is used

# Programming

## 6.8   Repeated Items

A line may have any number of G words, but two G words from the same modal group may not appear on the same line (for more information, refer to section *Modal Groups* earlier in this chapter).

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter. If a parameter setting of the same parameter is repeated on a line, `#3=15  #3=6`, for example, only the last setting takes effect. It is illogical but not illegal to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one is used. Each of the other comments is read and its format is checked, but it is ignored thereafter. Putting more than one comment on a line is very rare.

## 6.9   Order of Execution

The order of items on a line does not determine the order of execution on the commands. This is as defined in the *Order of Execution* table later in this section. The three types of items whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of items are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line which is as defined above. If the second group (the parameter settings) is reordered, there is no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter takes effect.

Example:

After the line `#3=15  #3=6`  has been interpreted, the value of parameter 3 is 6. If the order is reversed to  `#3=6 #3=15`  and the line is interpreted, the value of parameter 3 is 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment is used. If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line.

Example:

The line  `g40 g1 #3=15 (foo) #4=-7.0`  has five items and means exactly the same thing in any of the 120 possible orders, such as `#4=-7.0 g1 #3=15 g40 (foo)`, for the five items.

The order of execution of items on a line is critical to safe and effective machine operation. Items are executed in the order shown in the following table if they occur on the same line. Impose a different order (i.e., turn coolant off before spindle is stopped), by coding the commands on separate blocks.

**Order of Execution**

| Order | Item |
|---|---|
| 1 | Comment (including message) |
| 2 | Set feed rate mode (G93, G94, G95) |
| 3 | Set feed rate (F) |
| 4 | Set spindle speed (S) |
| 5 | Special I/O (M62 to M68) – not currently supported |
| 6 | Change tool (T) |
| 7 | Spindle On/Off (M03, M04, M05) |
| 8 | Save State (M70, M73, Restore State (M72), Invalidate State (M71) |
| 9 | Coolant On/Off (M07, M08, M09) |
| 10 | Enable/disable overrides (M48, M49, M50, M51, M52, M53) |
| 11 | User defined commands (M100 to M199) |
| 12 | Dwell (G4) |
| 13 | Set active plane (G17, G18, G18) – not applicable to lathe |
| 14 | Set length units (G20, G21) |
| 15 | Cutter radius compensation On/Off (G40, G41, G42) |
| 16 | Tool table offset On/Off (G43, G49) – not applicable to lathe |
| 17 | Fixture table select (G54 – G58 and G59 P~) |
| 19 | Set path control mode (G61, G61.1, G64) |
| 19 | Set distance mode (G90, G91) |
| 20 | Set canned cycle return level mode (G98, G99) |
| 21 | Home, change coordinate system data (G10) or set offsets (G92, G94) |
| 22 | Perform motion (G00 to G03, G12, G13, G80 to G89 as modified by G53 |
| 23 | Stop (M00, M01, M02, M30, M60 |

## 6.10   Error Handling

PathPilot sometimes ignores things it does not understand. If a command does not work as expected or does nothing, check that you have typed it correctly. PathPilot does not check for excessively high machining feeds or speeds. Nor does it detect situations where a legal command does something unfortunate, such as machining a fixture.

## 6.11   Parameters and Expressions

***NOTE:*** *You may wish to skip this section on first reading as it describes features of PathPilot not used by code generated by the Conversational screens in PathPilot or by most CAM systems.*

### 6.11.1   Parameters

The RS274/NGC language supports parameters or what in other programming languages would be called variables. There are several types of parameters of different purpose and appearance, each described in the following sections. The only value type supported by parameters is floating-point; there are no string, Boolean or integer types in G-code like in other programming languages. However, logic expressions can be formulated with Boolean operators (AND, OR, XOR, and the comparison operators EQ,NE,GT,GE,LT,LE), and the MOD, ROUND, FUP and FIX operators support integer arithmetic. Parameters differ in syntax, scope, behavior when not yet initialized, mode, persistence and intended use.

**Syntax**

There are three kinds of syntactic appearance:

- Numbered – `#4711`
- Named local – `#<localvalue>`
- Named global – `#<_globalvalue>`

**Scope**

The scope of a parameter is either global or local within a subroutine. Subroutine parameters and local named variables have local scope. Globally-named parameters and numbered parameters starting from number 31 are global in scope.

RS274/NGC uses lexical scoping. In a subroutine, only the local variables defined therein and any global variables are visible. The local variables of a calling procedure are not visible in a called procedure.

**Behavior of Uninitialized Parameters**

1. Uninitialized global parameters and unused subroutine parameters return the value zero when used in an expression.
2. Uninitialized named parameters signal an error when used in an expression.

**Mode**

Most parameters are read/write and may be assigned to within an assignment statement. However, for many predefined parameters this does not make sense, so they are read-only and may appear in expressions, but not on the left-hand side of an assignment statement.

**Persistence**

When PathPilot is powered off, volatile parameters lose their values. All parameters, except numbered parameters in the current persistent range (5163 to 5390), are volatile. Persistent parameters are saved in a disc file and restored to their previous values when PathPilot is powered on again. Volatile numbered parameters are reset to zero.

**Intended Use**

User parameters: numbered parameters in the range 31 to 5000 and named global and local parameters, except predefined parameters, are available for general-purpose storage of floating-point values, like intermediate results, flags, etc., throughout program execution. They are read/write (can be assigned a value).

Subroutine parameters: Used to hold the actual parameters passed to a subroutine.

Numbered parameters: Most of these are used to access offsets of coordinate systems.

System parameters: Used to determine the current running version and are read-only.

**Numbered Parameters**

A numbered parameter is the pound symbol # followed by an integer between 1 and 5399. The parameter is referred to by this integer, and its value is whatever number is stored in the parameter.

A value is stored in a parameter with the = operator; for example:

```
#3 = 15 (set parameter 3 to 15)
```

A parameter setting does not take effect until after all parameter values on the same line are found.

Example:

If parameter 3 has been previously set to 15 and the line `#3=6 G01 X#3` is interpreted, a straight move to a point where X equals 15 occurs and the value of parameter 3 is then set to 6.

The # symbol takes precedence over other operations, so that, for example, `#1+2` means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, `#[1+2]` does mean the value found in parameter 3. The # character may be repeated; for example `##2` means the value of parameter whose index is the (integer) value of parameter 2.

The PathPilot interpreter maintains a number of read-only parameters. Only axes X and Z are relevant to the 15L Slant-PRO so parameters for Y, A, B, C, U, V, W axes are undefined and unused.

1-30 – Subroutine local parameters of call arguments. These parameters are local to the subroutine. For further information, refer to section *O-codes* later in this chapter.

31-5000 – G-code user parameters. These parameters are global in the G-code file.

5061-5070 – Result of G38.2 Probe (X Y Z A B C U V W)

# Programming

5161-5169 – G28 Home for (X Y Z A B C U V W)

5181-5189 – G30 Home for (X Y Z A B C U V W)

5210 – 1 if G92 offsets are active, 0 if not

5211-5219 – G92 offset (X Y Z A B C U V W)

5220 – Current Coordinate System number 1-9 for G54-G59.3

5221-5230 – Coordinate System 1, G54 (X Y Z A B C U V W R); R denotes the XY rotation angle around the Z-axis

5241-5250 – Coordinate System 2, G55 (X Y Z A B C U V W R)

5261-5270 – Coordinate System 3, G56 (X Y Z A B C U V W R)

5281-5290 – Coordinate System 4, G57 (X Y Z A B C U V W R)

5301-5310 – Coordinate System 5, G58 (X Y Z A B C U V W R)

5321-5330 – Coordinate System 6, G59 (X Y Z A B C U V W R)

5341-5350 – Coordinate System 7, G59.1 (X Y Z A B C U V W R)

5361-5370 – Coordinate System 8, G59.2 (X Y Z A B C U V W R)

5381-5390 – Coordinate System 9, G59.3 (X Y Z A B C U V W R)

5399 – Result of M66; check or wait for input

5400 – Current Tool Number

5401-5409 – Tool Offset (X Y Z A B C U V W)

5410 – Current Tool Diameter

5411 – Current Tool Front Angle

5412 – Current Tool Back Angle

5413 – Current Tool Orientation

5420-5428 – Current Position including offsets in current program units (X Y Z A B C U V W)

**Subroutine Parameters**

1-30

Subroutine local parameters of call arguments. These parameters are local to the subroutine.

**Named Parameters**

Named parameters work like numbered parameters but are easier to read and remember. All parameter names are converted to lower case and have spaces and tabs removed. Named parameters must be enclosed with $<$ $>$ marks.

`#<named parameter here>` is a local named parameter. By default, a named parameter is local to the scope in which it is assigned.

You can't access a local parameter outside of its subroutine. This is so two subroutines can use the same parameter names without fear of one subroutine overwriting the values in another.

`#<_global named parameter here>` (i.e., name starting with an underscore) is a global named parameter. They are accessible from within called subroutines and may set values within subroutines that are accessible to the caller. As far as scope is concerned, they act just like regular numeric parameters. They are not made persistent by storage in a file.

**Examples**

Declaration of named global variable

```
#<_endmill_dia> = 0.049
```

Reference to previously declared global variable

```
#<_endmill_rad> = [#<_endmill_dia>/2.0]
```

Mixed literal and named parameters

```
o100 call [0.0] [0.0] [#<_inside_cutout>-#<_endmill_dia>] [#<_Zcut>] [#<_feedrate>]
```

***NOTE:*** *The global parameters _a, _b, _c, . . . _z have been reserved for special use. Two global, read only named parameters are available to check which version is running from G-code.*

`#<_vmajor>` – Major package version. If current version was 2.5.2 would return 2.5.

`#<_vminor>` – Minor package version. If current version was 2.5.2 would return 0.2.

### 6.11.2   Expressions

An expression is a set of characters starting with a left bracket (`[`) and ending with a balancing right bracket (`]`). In between the brackets are numbers, parameter values, mathematical operations, and other expressions. An expression is evaluated to produce a number. The expressions on a line are evaluated when the line is read and before anything on the line is executed.

Example:

```
[1 + acos[0] - [#3 ** [4.0/2]]]
```

**Binary Operators**

Binary operators only appear inside expressions. There are four basic mathematical operations: addition (+), subtraction (−), multiplication (*), and division (/). There are three logical operations: non-exclusive or (`OR`), exclusive or (`XOR`), and logical and (`AND`). The eighth operation is the modulus operation (`MOD`). The ninth operation is the power operation (`**`) of raising the number on the left of the operation to the power on the right. The relational operators are equality (`EQ`), inequality (`NE`), strictly greater than (`GT`), greater than or equal to (`GE`), strictly less than (`LT`), and less than or equal to (`LE`). The binary operations are divided into several groups according to their precedence, detailed in *Binary Operators* table later in this section.

If operations in different precedence groups are strung together (for example in the expression `[2.0 / 3 * 1.5 - 5.5 / 11.0]`), operations in a higher group are to be performed before operations in a lower group. If an expression contains more than one operation from the same group (such as the first / and * in the example), the operation on the left is performed first.

**Binary Operators**

| Operators | Precedence |
|---|---|
| ** | highest |
| * / MOD | |
| + - | |
| EQ NE GT GE LT LE | |
| AND OR XOR | lowest |

Thus, the example is equivalent to: `[ [ [2.0 / 3] * 1.5] - [5.5 / 11.0] ]`, which is equivalent to `[1.0 - 0.5]`, which is `0.5`.

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical `false`, and any non-zero number is equivalent to logical `true`.

**Functions**

A function is either ATAN followed by one expression divided by another expression (for example `ATAN[2]/[1+3]`) or any other function name followed by an expression (for example `SIN[90]`). The available functions are detailed in the Functions table later in this section.

Arguments to unary operations which take angle measures (COS, SIN, and TAN) are in degrees, as are values returned by unary operations which return angle measures (ACOS, ASIN, and ATAN).

The FIX function rounds towards the left (less positive or more negative) on a number line, so, for example, `FIX[2.8] =2 and FIX[-2.8] = -3`.

The FUP operation rounds towards the right (more positive or less negative) on a number line; so, for example, `FUP[2.8] = 3` and `FUP[-2.8] = -2`.

The EXISTS function checks for the existence of a single named parameter. It takes only one named parameter and returns 1 if it exists and 0 if it does not exist. It is an error if you use it with a numbered parameter or an expression.

<div align="center">

**Functions**

| Function name | Function Result |
|---|---|
| ATAN[Y]**/**[X] | Four quadrant inverse tangent |
| ABS[arg] | Absolute value |
| ACOS[arg] | Inverse cosine |
| ASIN[arg] | Inverse sine |
| COS[arg] | Cosine |
| EXP[arg] | *e* raised to the given power |
| FIX[arg] | Round down to integer |
| FUP[arg] | Round up to integer |
| ROUND[arg] | Round to nearest integer |
| LN[arg] | Base-*e* logarithm |
| SIN[arg] | Sine |
| SQRT[arg] | Square Root |
| TAN[arg] | Tangent |
| EXISTS[arg] | Check named Parameter |

</div>

## 6.12   O-codes

O-codes provide for flow control in NC programs. Each block has an associated number, which is the number used after O. Care must be taken to properly match the O-numbers. O-codes use the letter O, not the number zero, as the first character in the number like O100.

### Numbering Example

```
o100 sub
  (notice that the if-endif block uses a different number)
  o110 if [#2 GT 5]
    (some code here)
  o110 endif
  (some more code here)
o100 endsub
```

The behavior is undefined if:

- The same number is used for more than one block
- Other words are used on a line with an O-word
- Comments are used on a line with an O-word

*NOTE: Using the lowercase o makes it easier to distinguish from a 0 that might have been mistyped. For example o100 is easier to determine to be different from O100, which is not a 0.*

### Subroutines

Subroutines extend from a `O~ sub` to an `O~ endsub`. The lines between `O~ sub` and `O# endsub` are not executed until the subroutine is called with `O~ call`.

### Subroutine Example

```
o100 sub
  G53 G0 X0 Y0 Z0 (rapid move to machine home)
o100 endsub
  ...
o100 call (call the subroutine here)
M2
```

For further information, refer to G53 and G0 and M2 sections.

`O~ Return` – Inside a subroutine, `O~ return` can be executed. This immediately returns to the calling G-code, just as though `O~ endsub` was encountered.

**O~ Return Example**

```
o100 sub
o110 if [#2 GT 5] (test if parameter #2 is greater than 5)
o100 return (return to top of subroutine if test is true)

o110 endif
(some code here that only gets executed if parameter #2 is less than 5)
o100 endsub
```

For further information, refer to *Binary Operators* in section *Parameters and Expressions* earlier in this chapter.

`O~ Call` – O~ Call takes up to 30 optional arguments, which are passed to the subroutine as #1, #2 , ..., #N. Parameters from #N+1 to #30 have the same value as in the calling context. On return from the subroutine, the values of parameters #1 through #30 (regardless of the number of arguments) are restored to the values they had before the call. Parameters #1-#30 are local to the subroutine.

Because 1 2 3 is parsed as the number 123, the parameters must be enclosed in square brackets. The following calls a subroutine with three arguments.

**O~ Call Example**

```
o200 call [1] [2] [3]
```

Subroutine bodies may not be nested. They may only be called after they are defined. They may be called from other functions, and may call themselves recursively if it makes sense to do so. The maximum subroutine nesting level is 10.

Subroutines do not have return values, but they may change the value of parameters above #30 and those changes are visible to the calling G-code. Subroutines may also change the value of global named parameters.

**Looping**

The while loop has two structures: `while/endwhile`, and `do/while`. In each case, the loop is exited when the while condition evaluates to false. The difference is when the test condition is done. The `do/while` loop runs the code in the loop then checks the test condition. The `while/endwhile` loop does the test first.

### While Endwhile Example

```
(draw a sawtooth shape)
G0 X1 Y0 (move to start position)
#1 = 1 (assign parameter #1 the value of 0)
F25 (set a feed rate)
o101 while [#1 LT 10]
  G01 X0
  G01 Y[#1/10] X1
  #1 = [#1+1] (increment the test counter)
o101 endwhile
M2 (end program)
```

### Do While Example

```
#1 = 0 (assign parameter #1 the value of 0)
o100 do
 (debug, parameter 1 = #1)
 o110 if [#1 EQ 2]
  #1 = 3 (assign the value of 3 to parameter #1)
  (msg, #1 has been assigned the value of 3)
  o100 continue (skip to start of loop)
  o110 endif
  (some code here)
  #1 = [#1 + 1] (increment the test counter)
o100 while [#1 LT 3]
(msg, Loop Done!)
M2
```

Inside a while loop, `O~ break` immediately exits the loop, and `O~ continue` immediately skips to the next evaluation of the while condition. If it is still true, the loop begins again at the top. If it is false, it exits the loop.

### Conditional

The `if` conditional consists of a group of statements with the same o number that start with `if` and end with `endif`. Optional `elseif` and `else` conditions may be between the starting if and the ending `endif`.

If the `if` conditional evaluates to true then the group of statements following the `if` up to the next conditional line are executed.

If the `if` conditional evaluates to `false` then the `elseif` conditions are evaluated in order until one evaluates to true. If the `elseif` condition is true then the statements following the `elseif` up to the next conditional line are executed. If none of the `if` or `elseif` conditions evaluate to true then the statements following the `else` are executed. When a condition is evaluated to true no more conditions are evaluated in the group.

**If Endif Example**

```
o101 if [#31 EQ 3] (if parameter #31 is equal to 3 set S2000)
  S2000
o101 endif
```

**If ElseIf …Else … EndIf Example**

```
o102 if [#2 GT 5] (if parameter #2 is greater than 5 set F100)
  F100
o102 elseif [#2 LT 2] (else if parameter #2 is less than 2 set F200)
  F200
o102 else (else if parameter #2 is 2 through 5 set F150)
  F150
o102 endif
```

**Repeat**

The `repeat` executes the statements inside of the `repeat/endrepeat` the specified number of times. The example below shows how you might mill a diagonal series of shapes starting at the present position.

**Repeat Example**

```
(Mill 5 diagonal shapes)
G91 (Incremental mode)
o103 repeat [5]
  ... (insert millinG-code here)
  G0 X1 Y1 (diagonal move to next position)
o103 endrepeat
G90 (Absolute mode)
```

**Indirection**

The O-number may be given by a parameter and/or calculation.

**Indirection Example**

```
o[#101+2] call
```

For more information on computing values refer to these items in section *Parameters and Expressions* earlier in this chapter:

- *Parameters*
- *Expressions*
- *Binary Operators*
- *Functions*

**Calling Files**

To call a separate file with a subroutine, name the file the same as your call and include a `sub` and `endsub` in the file. The file must be in the directory `Subroutines`. The file name can include lowercase letters, numbers, dashes, and underscores only. A named subroutine file can contain only a single subroutine definition. The file must end with the extension `.nc`.

**Named File Example**

```
o<myfile> call
```

**Numbered File Example**

```
o123 call
```

In the called file you must include the `oxxx sub` and `endsub` and the file must be a valid file.

**Called File Example**

```
(filename myfile.nc)
o<myfile> sub
  (code here)
o<myfile> endsub
M2
```

***NOTE:*** *File names are lowercase letters only;* `o<MyFile>` *is converted to* `o<myfile>` *by the interpreter.*

# 7.   Maintenance

## 7.1   Lathe Maintenance Schedule

| Frequency | Completed | Item |
|---|---|---|
| **Daily** | | Check coolant level (12 gallon capacity) |
| | | Check oiler level and top off as needed (Auto Oiler 2.1 quarts / Manual Oiler 0.25 quart) |
| | | Clean chips from ways and carriage |
| | | Spray exposed, non-painted metal surfaces areas with WD-40® or similar to prevent rust |
| **Weekly** | | Check/adjust draw tube tension on 5C lever closer (if applicable) |
| | | Clean chip basket on coolant tank |
| | | Check air gauge/regulator for proper PSI (90 PSI minimum) |
| | | Use mild cleaner to clean all exterior surfaces (no solvents) |
| | | Check oil and grease reservoirs; top up as needed |
| **Monthly** | | Dispose of oil in drain bucket |
| | | Remove coolant tank pump and clean out sediment buildup in tank; reinstall |
| | | Inspect electrical cabinet vent(s) for dust build up; if necessary, wipe vents with clean cloth or use compressed air to remove dust if accumulation is excessive |
| **Every Six Months** | | Inspect way covers as needed to ensure proper operation; replace as needed |
| | | Inspect oil system for blockages; clean/replace as needed |
| | | Check lubrication hoses for signs of wear or cracking; replace as needed |
| **Every 12 Months** | | Check limit switches for proper function; replace as needed |
| | | Check door switches for proper function; replace as needed |

## 7.2    Understanding Machine Design

The 15L Slant-PRO lathe has been designed for operator maintenance, but an important prerequisite to effective maintenance is an understanding of some general machine-design concepts. For many owners, a Tormach machine is their first experience with a machine tool and they may be unfamiliar with the ideas presented here. This section should be read and understood before attempting any adjustments to the machine.

### 7.2.1    Machine Stiffness

In the context of the human experience, the iron and steel of your lathe seems exceptionally stiff and essentially immovable. Nevertheless, any physicist or material scientist will tell you that iron is a Hookean material, meaning that it always obeys Hooke's law and distorts linearly in proportion to applied force. In simple terms, it acts like a spring. A small force makes a small distortion, and a large force makes for a large distortion.

A good dial indicator can show motion of a distance 0.0002". This sort of distance is outside the scope of normal human experience. In the context of a dial indicator, the iron and steel of your lathe no longer seems rigid, and the motion of the frame under force becomes quite apparent. You should know that this flexibility is natural. You would see more flex on an 800 lb. machine and less on an 8000 lb. machine. One of the keys to creating accurate parts with your lathe is developing an awareness of machine flex. The flexure of the machine cutting tool and workpiece are the principle reasons that a light finishing cut yields a more accurate part.

### 7.2.2    Backlash, Friction, and Lost Motion

All machines show backlash. Conventional backlash is the result of open spaces between moving parts. For example most ball bearings have clearance between the balls and races which results in backlash. Pre-loading bearings is the technique of putting opposing pairs of bearings in compression. This technique eliminates the open space, but apparent backlash, more correctly called lost motion, still exists. There is always some force required for motion. Since the bearings are compressible (steel is a spring), there is some motion required for compression before enough force is delivered and motion is obtained. This is lost motion and has the same effect as conventional backlash.

When performing maintenance on the machine you should be aware that if there is backlash of more than 0.003", there is some conventional backlash and something needs to be adjusted or replaced. Backlash of 0.002" or less is probably lost motion and not conventional backlash.

There are compromises and tradeoffs made when adjusting the machine. Tightening the gibs on the X-axis reduces backlash in Z, but it also increases the force required for motion in X, thus increasing lost motion in X. Increasing the preload on a bearing pair reduces lost motion, but it shortens the life of the bearing.

### 7.2.3 Factors Combine

Machine setup and tuning is done under no-load conditions. The accuracy of machined work is going to be worse because tool flex, workpiece flex, and other factors combined with the basic machine accuracy. Even thermal expansion adds to the mix. A 30" ball screw changes 0.0025" in length with a 10°F change in temperature.

### 7.2.4 Adjusting Geometry

Over time, a machine may need adjustment. All of the iron castings of your machine undergo thermal stress reliving. Major elements, such as the column and base, also undergo a vibratory stress relief process between rough machining and final machining. Nevertheless, it is possible that any iron casting can change shape a small amount, whether due to changes in the crystalline structure over time or changes induced through tool crashes and abuse.

Before making any adjustment, think carefully about the need for the adjustment and implications associated with it. Adjustments can be made and are sometimes necessary. Proceed slowly and think carefully, assessing the need for any adjustment and the best method for making that adjustment.

### 7.2.5 Achieving Accuracy in Machining

The key to achieving maximum accuracy is understanding and controlling the magnitude and direction of forces. Maximum accuracy is achieved when the forces are minimized, as occurs in a finishing cut. Maximum repeatability is achieved when the forces are repeatable, both in magnitude and direction.

Machining is a mix of science, skill, and art. The caveat in stating accuracy and repeatability is that these factors depend on the techniques used by the machinist. A skilled machinist can often deliver accuracy that exceeds the accuracy specified by the machine builder, while an inexperienced machinist may have difficulty delivering the expected accuracy.

### 7.2.6 Rust Prevention

Exposed iron and steel surfaces will rust if proper care is not taken to protect them. The following recommendations will slow or reduce the onset of surface rust.

- If possible, install the lathe in a temperature- and humidity-controlled environment.
- Always use a flood coolant recommended for machining; never use water or a coolant that does not contain rust-inhibitors. When using a water-based coolant, always mix the coolant concentrate to the dilution ratio recommended by the coolant manufacturer.
- If the lathe is not used for more than 72 hours, apply a light mist of water repellent oil (WD-40 or similar) to the exposed bare metal surfaces.
- Trapped areas are susceptible to rust. Apply way oil or machine oil directly to the table surface under the trapped area when you mount a gang plate or T-slot plate on the lathe.
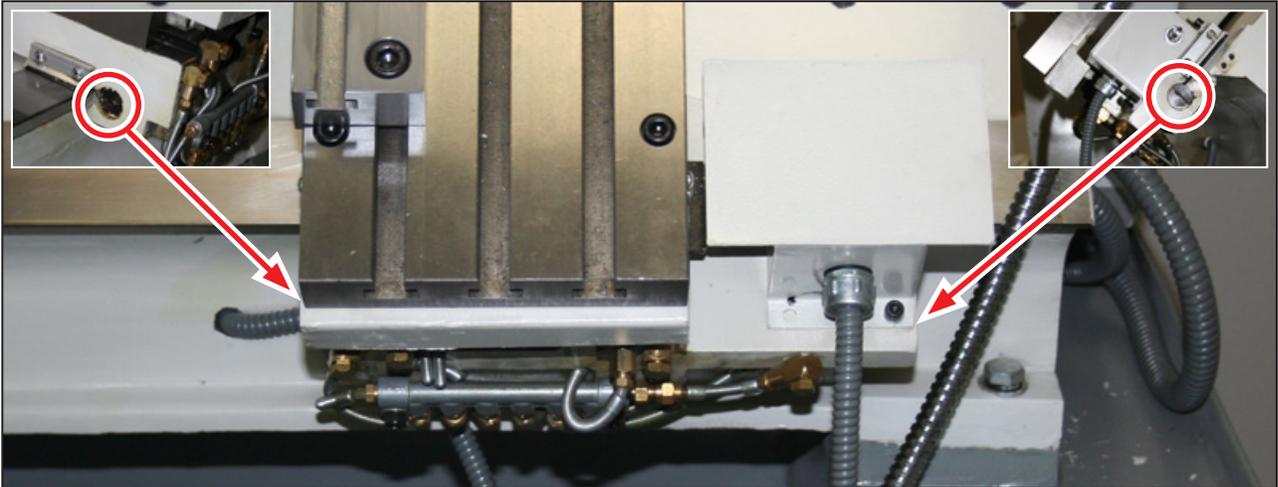
**Figure 7.1**

## 7.3 Gibs, Dovetail Slideways, and Lubrication

The X and Z slideways have a bonded layer of skived plastic compound, a composition of acetyl and PTFE. The material is commonly known under the trade names of Turcite® or Rulon®. This is state of the art technology for oil lubricated slideways and superior to plain ground surfaces or hardened and chromed surfaces. We don't have data on how long the material lasts on the 15L Slant-PRO, but we do know that there have been no reports of appreciable wear – even on machines that are reported to have seen more than 5000 hours of operation. If you use the automatic oiler, the slideways are not normally maintenance items.

The lubrication system should be filled with Tormach Machine Oil (PN 31386). Other options include Perkins Perlube WL-68, Tonna 68 (Shell), Vactra No. 2 (Mobil), Way-lube 68 (Sunoco), WayLube 68 (Texaco), Febis 68 (Esso) or equivalent oil.

A shot of lubrication should be given for each four hours of operation as well as each time the machine is powered on. The oil is delivered to 10 points throughout the machine. This includes the eight sliding surfaces (four on each axis) and two ball nuts. These are some of the most critical (and costly) mechanical parts of the machine. Always make certain that the lubrication oil is clean. Any dirt or foreign material suspended in the oil is going to be delivered directly to these parts and can dramatically shorten the effective life of the machine. Be sure to clean off the cover and surrounding area before refilling the oil reservoir. The strainer at the top of the reservoir is only a screen – it is not a filter.

Extreme axis positions can expose the oil distribution channels that are cut into the way support saddle surfaces. If the pump is used in those positions the hydraulic force of the oil does not apply it throughout the machine as intended. Instead, the oil simply squirts out at the point where the oil channel is exposed. After a long period of inactivity or in cold conditions, the oil system may become clogged. For more information, refer to Tormach service bulletin SB0031 – *Flushing the Lubrication System*.

The slideways are hand scraped as part of the manufacturing process. This means that the Z-axis saddle is fitted to the bed. They are scraped in as a set and neither the saddle nor the base can be had as a replacement component. The slideways have tapered gib plates (also hand scraped), where the position of the gib plate controls the tightness and friction in a slide way. For example, in **Figure 7.1**, the Z-axis gib plate is part of the saddle and is held in position on both front and back via the gib screws.

### 7.3.1   Gib Adjustment Procedure

The adjustment procedure is similar for both axes. The Z-axis is described in detail. The gib plate is tapered from right to left. When the gib plate is moved in the Z negative direction (to the left of the operator), the Z-axis saddle the slide way tightens; when it moves in the Z positive direction, the slideway loosens. With the machine is powered on, move the Z-axis to near the middle of its travel and locate the gib plate screw on either end of the Z saddle (**Figure 7.1**).

It is difficult to assess the correct clearance for the gib as a very small force on its end can, via the shallow taper, apply a large force to the dovetail. Too tight a gib results in zero gap and very high friction since there is no longer space for an oil film.

The easiest method is by a form of dead reckoning:

1. Ensure the left (loosening) gib screw is unscrewed by several turns to allow for clearance for the gib to be adjusted to the left.

2. Tighten the right (tightening) gib screw in, thus pushing the gib left until it is tight yet not hitting the bottom screw. Wiggle the carriage while you are tightening this screw to get the gib reasonably tight – but not so tight as to strip the screw head or flex the castings.

3. Loosen the right screw four full turns.

4. Tighten the left screw, thus pushing the gib back up against the upper screw and creating a positive amount of gib clearance. The gib should be held in place with upper and lower screw tight on the gib.

**IMPORTANT!** *Ensure both adjustment screws are tight after any gib adjustment. Failure to do so will result in gib movement.*

A more precise method of adjusting the gibs is to use lost motion, apparent as backlash on a dial indicator, to arrive at the correct setting:

1. Loosen the right screw eight turns and tighten the left screw eight turns so that the gib clearance is loose.

2. Use a dial indicator to measure the axis backlash. With a very loose gib, the measured backlash is entirely attributable to the backlash in the belt reduction drive, the angular contact bearings, and the ball nut. On a newer machine, the value should be under 0.0025" on either axis.

3. Tighten the gib by one turn (by loosening the lower screw first, then tightening the upper screw). Measure the backlash again.

4. Repeat this procedure until the measured backlash increases – this is the point at which the gib setting has started to induce lost motion in the axis.

5. Back the adjustment off to the point just before you saw the increased backlash. That is the ideal setting for the axis (see **Figure 7.2**).

*NOTE: For the X-axis, the gib tightens with the gib screw on the bottom (operator side). No covers need to be removed.*
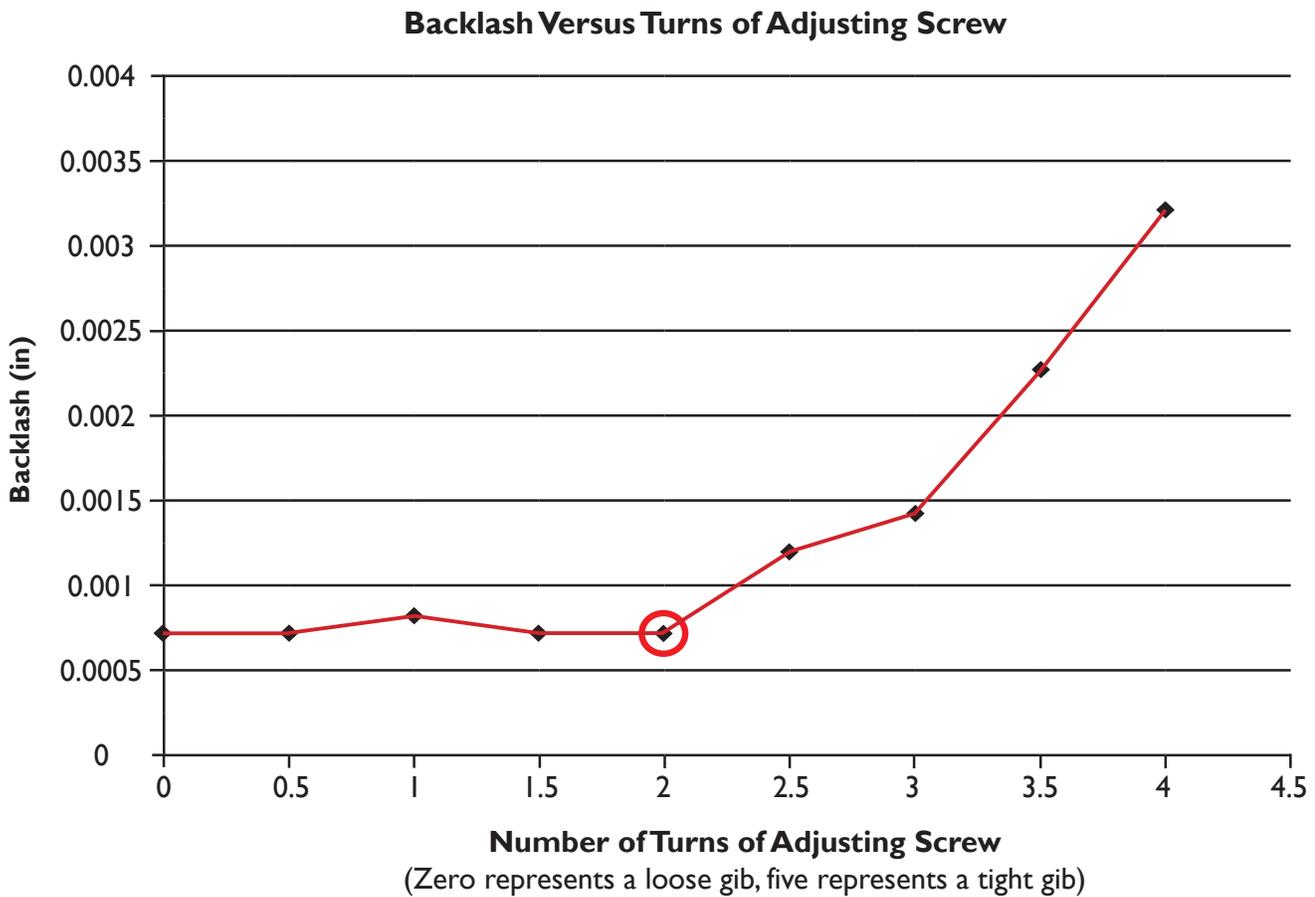


**Figure 7.2**

# 8.  Troubleshooting

## 8.1  Troubleshooting Basics

The 15L Slant-PRO is an integration of mechanical components, electrical components, a controller, and software. Taken as a whole, the 15L is a complex machine; however, the machine comprises several subsystems, each of which is much easier to understand than is the machine in its entirety.

| 1 | Divide and conquer | Work on one problem at a time; focus on related issues only |
|---|---|---|
| 2 | Know the machine | Be familiar with how machine functions under normal circumstances |
| 3 | Environmental changes | Determine if any changes occurred in machine environment including:<br>• The lathe was moved<br>• A fuse blew or breaker tripped causing electronics to malfunction<br>• Water/moisture entered machine area<br>• Major temperature change in unheated facility<br>• Unauthorized lathe use occurred |
| 4 | Work smart | Draft a list of tests to perform by level of difficulty; try simplest, most likely tests first to determine causes of problem; record results |
| 5 | One step at a time | Take time to complete one test before starting another, thereby not making the problem worse by conducting random troubleshooting. |

This chapter builds on these key principles by providing a description of each of the subsystems. This serves to help the operator understand how the machine will work, and provides an overview of the components that are involved in the subsystem. We also list some guidelines for equipment and procedures used in troubleshooting.

As in most electromechanical machines, it is frequently easier to see problems in the mechanical systems than the electrical and controller systems. With this in mind, most of the details in the troubleshooting section will address non-mechanical areas.

## 8.2  Tips and Tools for Troubleshooting (equipment and procedures)

### 8.2.1  Safety

During troubleshooting, the operator is exposed to more hazards than during normal operation: electrical tests may need to be done on live circuits; guards may have to be removed; a safety switch may have to be overriden in order to make an observation. Take things slow and be extra cautious.

**IMPORTANT!** *For complete safety information, refer to the Safety Overview starting on page 2.*

### 8.2.2 Tips on Controller Diagnosis

The PathPilot® operating system provides a status screen that is useful for troubleshooting. If calling Tormach Technical Support, make sure you know what version of PathPilot is running and any pertinent information from the *Status* screen and the error line.

To email the log file to Tormach Technical Support, use the *File* tab to transfer the relevant information to a USB flash drive.

1. Insert a USB flash drive into any open USB slot on the controller.

2. On the *File* screen, click *Home* in the upper right-hand corner of the screen (see **Figure 8.1**).

3. Locate and click the *logfile* folder (see **Figure 8.1**); inside this folder, highlight the *pathpilotlog.txt* file.

4. Click *Copy to USB* (see **Figure 8.1**).

5. After the file transfers, remove the USB flash drive from controller and email the file to Technical Support.



**Figure 8.1**

### 8.2.3 Troubleshooting Tools

The following are some basic tools to have on hand for troubleshooting:

- Good lighting such as a trouble light, headlamp, or flashlight
- A digital multimeter that can test for AC volts up to 300 V, DC volts up to 100 V, and resistance from 0 to 1M ohms (Ω)
- Assorted screwdrivers: #2 and #3 Phillips, 1/8" and 3/16" flat blade
- Assorted hex keys in metric sizes
- A wire stripper
- Measuring tools like tape measure, micrometer, or dial indicator (optional)

### 8.2.4 Using Digital Multimeter for Electrical Tests

Almost all digital multimeters have the capability of measuring AC volts, DC volts, and resistance. These are the important functions used when troubleshooting a Tormach machine tool. Two test leads are required to measure these three functions. While many meters have more than two receptacles in which to plug leads, for our purposes the leads will always remain in the same two receptacles. One receptacle is almost universally labeled COM (for common) and the black lead is to be plugged into this receptacle. The other lead, most often red in color, is plugged into the receptacle labeled VΩ (volts and ohms).

### 8.2.4.1 Measuring DC Voltage

Select the DC voltage scale on the meter. The scale may be labeled DCV. DC voltage has polarity, so if the common lead on the meter is not placed on the common signal, the voltage will read as a negative number. This does not harm the meter, but could be confusing in certain cases. We strive to define the common terminal when asking you to take a measurement.

### 8.2.4.2 Measuring AC Voltage

Select the AC voltage scale on the meter. The scale may be labeled ACV. There is no polarity to AC voltage; either lead can be placed on either location to measure.

### 8.2.4.3 Measuring Resistance

Ohms are the units in which resistance is measured. Select the resistance scale on the meter, most often labeled Ω. There is no polarity to resistance so either lead can be placed on either location we are having you measure; however, resistance measurements are always taken with power off.

*NOTE: When making resistance measurements on motors and other devices with low resistance, always take a tare or zero reading on the meter before doing the resistance measurement on the motor or device. A tare reading is a reading using only the instrument and its test leads. In the case of using a digital multimeter to measure resistance, touch the probes together to get the tare reading. Subtract the tare reading from the meter reading when measuring the resistance of the motor to get a true value.*

## 8.2.5 Contacting Technical Support

Always try to resolve any issue by first reviewing this manual's troubleshooting section. If this effort does not produce results – and you do need to contact Tormach Technical Support – please review the list below prior to calling Tormach Technical Support. This will make the problem resolution process go smoother.

1. Record machine serial number and have it available.

2. We use a case management database system to track problems. Be sure to inform Technical Support staff if you experienced the problem before, as this allows us to review the history of the machine. If possible, please have your case management number ready.

3. If the machine was purchased from someone else, please have that person's name available. This way we can reference your specific machine in our case management database system and identify any problems the previous owner experienced.

4. Analyze what might have changed since the machine last worked correctly and have that information available.

5. Review the troubleshooting-related subsystem you are having problems with to gain a better understanding of that subsystem.

6. Make sure you can repeat the problem(s). Do this several times and record the results to determine if you can repeat the problem(s) exactly.

7. Record any pertinent information from the diagnostic screen of the PathPilot interface and have it available.

8. Try to define the problem as clearly and concisely as possible by writing it down. Often times by doing this you may help pinpoint the problem or find the solution to the problem yourself.

9. Consider sending Technical Support an email first, as this may help define the problem better. Communicating via email has the added benefit of documenting the issue, thus eliminating the need to interpret any hastily transcribed notes made during the phone conversation later on. Even if you do need to speak with Technical Support, sending an email first can help the conversation go smoother.

**Tormach Technical Support**

| | |
|---|---|
| **Email contact** (preferred) | info@tormach.com |
| **Phone contact** (8:00 – 5:00 CST) | 608-849-8381 |

## 8.3   Frequently Found Problems

The sections to follow detail several frequently found problems with all electromechanical machinery. It is not that the problems are frequent, but among the problems that have occurred, these are more frequent than others.

### 8.3.1   Loose Wires

Try as we might, it seems that on occasion we find a poor wire connection. This can be the wire in a screw clamp terminal where the clamp is loose or a problem with a crimp spade or ring connector where the connector is tight in the screw clamp terminal but the wire is loose in the crimp connector. This is most frequently found during the initial startup of the machine. The vibration that occurs during travel tends to loosen connections. Use the two finger pull test: grasp the wire close to its termination point between your thumb and index finger, and gently but firmly tug each wire. If the wire comes loose, re-terminate it before moving on to other wires.

### 8.3.2   Wire Hairs

Sometimes with stranded wire we find that a stray strand, or wire hair, from the stripped end of the wire may be sticking out and touching another wire or the machine frame; this can cause short circuits.

### 8.3.3   Poor Cable Connections

There are a number of cables on a Tormach 15L Slant-PRO lathe. Some are flat cables connecting the control board to the axis drives (or other devices) and some cables connect with the controller. An improperly seated cable can allow some functions to work and cause others not to. The ribbon cables are plugged into connectors with locking ears; ensure that the cables are correctly seated and that the ears are in the locked position.

The parallel port cable connection between the machine tool and the PCI card in the back of the controller should be firmly attached with the locking screws. Loose parallel port cables can cause machine communication issues that are hard to diagnose.

### 8.3.4  Controller Restart Required

It is a good practice to power cycle the controller on a daily basis. Controllers left on for weeks at a time can become problematic.

## 8.4  Review of Subsystems

### 8.4.1  Power Distribution Subsystem

#### 8.4.1.1  Overview

Operator-supplied electrical power (both the primary 230 VAC and secondary 115 VAC) is run through the main disconnect on the machine. This switch controlls all power to the machine and the controller.

#### 8.4.1.2  Details of the Power Distribution Subsystem

The power distribution subsystem portion of the electrical schematic is highlighted in **Figure 8.2**. For a more detailed view, refer to the lathe's electrical schematic in the back of this manual.



**Figure 8.2**

The related power distribution component locations in the electrical cabinet are highlighted in **Figure 8.3**.



**Figure 8.3**

The 15L Slant-PRO lathe is powered by a user provided nominal 230/1/60 (230 V single phase, 60 HZ) 20 Amp, 3-wire electrical circuit in North America and a 220/1/50 20 Amp, three wire electrical circuit in most other parts of the world. Tormach allows an input voltage range of 200 VAC to 250 VAC. The two current-carrying conductors of this supply are connected to the main disconnect located on the electrical cabinet. When the main disconnect is in the *Off* position, no power is supplied to the machine or to the accessories (controller, monitor, coolant pump) plugged into the outlets on the back panel of the electrical cabinet.

In addition to the main 230 VAC electrical supply, the lathe requires a 115 VAC auxiliary supply for the controller, monitor, and coolant pump. The power source for this circuit can be 110/1/50, 115/1/60 or 220/1/50 single phase power with one leg of the two current carrying conductors grounded. Normally, a 115 VAC circuit would be supplied in North America. A third wire ground must also be provided. The ungrounded leg of this supply is connected to a third pole on the main disconnect described earlier in this section. Running this circuit in this manner allows the coolant pump outlet to be controlled through PathPilot. The circuit also provides power to two outlets which can be used for the controller and monitor. Turning the main disconnect to the *Off* position powers off the coolant and controller.

### 8.4.2   Control Power Subsystem

#### 8.4.2.1   Overview

Control power enables running of the machine. When control power is on, the lathe control PCB and the axis motor drives in the cabinet are powered on. The spindle drive will not power on with the control power circuit – it only powers on at the first PathPilot spindle command. Turn control power on by pressing the green Start button; turn control power off by turning the red E-stop clockwise. The Machine LED illuminates when control power is on.

#### 8.4.2.2   Details of the Control Power Subsystem

The control power subsystem portion of the electrical schematic is highlighted in **Figure 8.4**. For a more detailed view, refer to the lathe's electrical schematic in the back of this manual.



**Figure 8.4**

The related control power distribution component locations in the electrical cabinet are highlighted in **Figure 8.5**.



Figure 8.5

The control power circuit is energized when the machine is in a ready-to-run state and de-energized when the red E-stop is in the depressed position. The circuit does not energize until the red E-stop button is and the green Start button is momentarily depressed, energizing the latching C1 contactor.

⚠ **WARNING! Hazardous Voltage:** *When control power is off, some components in the electrical cabinet are still energized, including: the main fuses FU1 and FU2; the control power transformer (XFM1); fuse FU3; and wires 102 and 103 on the stop and start push buttons. Additionally, the filter is powered by L11 and L21 and the filter passes power on wires L23 and L24 to contacts on the contactors C1 and C2. Be sure to remove power from the machine tool by switching the Main Disconnect to the off position. Failure to do so could result in serious injury or death.*

Should the main power to the machine be lost when the control power circuit is on, the control power circuit turns off and stays off until power is restored and the green Start button is pressed. This prevents the machine from restarting without operator action after any power outage, natural or man-made. Transformer XFM1 reduces the incoming voltage from a nominal 230 (or 220) VAC to a nominal 115 (or 110) VAC. 115V control circuits are industry standard in North America (this allows the use of readily available push buttons, contactors, and other control circuit components).

If the wall breaker consistently trips whenever the spindle is powered on, verify that the main machine power is not coming through a ground fault interruptor (GFI, or RCCB in Europe), as the filters in the spindle drive can allow minor leakage current to ground that, while considered safe, may also be sufficient to trip a GFI circuit breaker.

## 8.4.3   Controller Communication Subsystem

### 8.4.3.1   Overview

PathPilot operating system is the control software for the 15L Slant-PRO lathe. It allows for manual jogging of X- and Z-axes, spindle speed, and coolant pump through the PathPilot interface. It also allows the machine to run automatically with operator-supplied programs. PathPilot communicates with the lathe control PCB to provide machine control. The Controller LED on the operator panel must be illuminated before the computer can control the machine.

### 8.4.3.2   Details of the Controller Communication Subsystem

When the machine tool is first powered on following the power off/on procedure detailed in chapter 3, *Operation*, the Machine OK LED on the operator panel illuminates. PathPilot loads automatically when the controller is powered on (the main disconnect turned to the *On* position). When the software is loaded and the flashing *Reset* button is pressed, the LED on the operator panel illuminates.

The controller communicates with the machine tool via a PCI card with a DB-25 connector. This connector is identical to the parallel port connector, and it would be easy to mistakenly plug the cable into the wrong connector. If PathPilot will not communicate with the machine tool, verify that you have plugged the cable into the correct port on the back of the controller. PathPilot provides a *Status* screen with diagnostic LEDs which can be useful for troubleshooting.

## 8.4.4   Axis Drive Subsystem

### 8.4.4.1   Overview

The axis motors (otherwise known as stepper motors) are used to move the X- and Z-axes. The motors are powered by electronic driver modules (also referred to as axis drivers) which receive control signals from the lathe control board. The electronic driver modules get power from the DC bus board. Once the machine has been referenced, it is limited in travel by PathPilot soft limits.

### 8.4.4.2  Details of the Axis Drive Subsystem

The axis drive subsystem portion of the electrical schematic is highlighted in **Figure 8.6**. For a more detailed view, refer to the lathe's electrical schematic in the back of this manual.
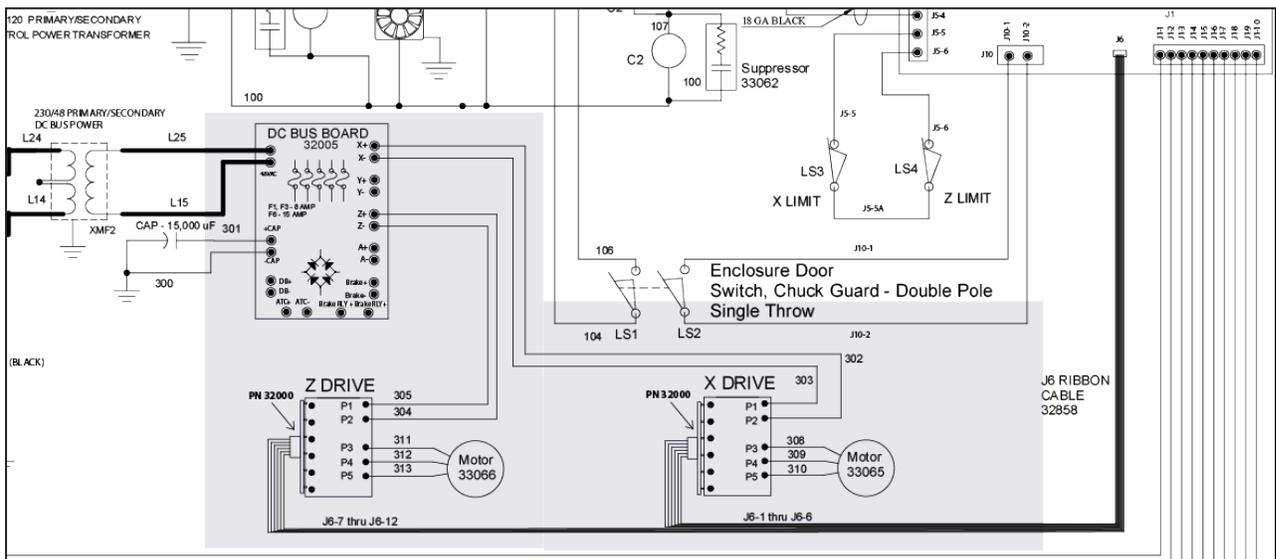


**Figure 8.6**

Motion for the X- and Z-axes is provided by stepper motors. Each motor is powered by an electronic driver module. The driver module receives nominal 65 VDC power from the DC bus board and receives control signals from the lathe control board, which processes and formats information sent by the controller. The axes are driven by stepper motors that have no encoder feedback. It is possible that an axis can be commanded to move but will not move as far as it is commanded to. Excessive friction or load in the mechanical system causes position loss. If losing steps, it is possible to hear a cogging noise from the motor. This problem is most likely to appear when attempting to drill large holes with the lathe: a 3/4" hole in mild steel without a pilot drill will almost certainly require more force than the Z-axis motor can provide. It is our recommendation to use a pilot hole with any drill diameter over 1/2".

The related axis drive distribution component locations in the electrical cabinet are highlighted in **Figure 8.7**.
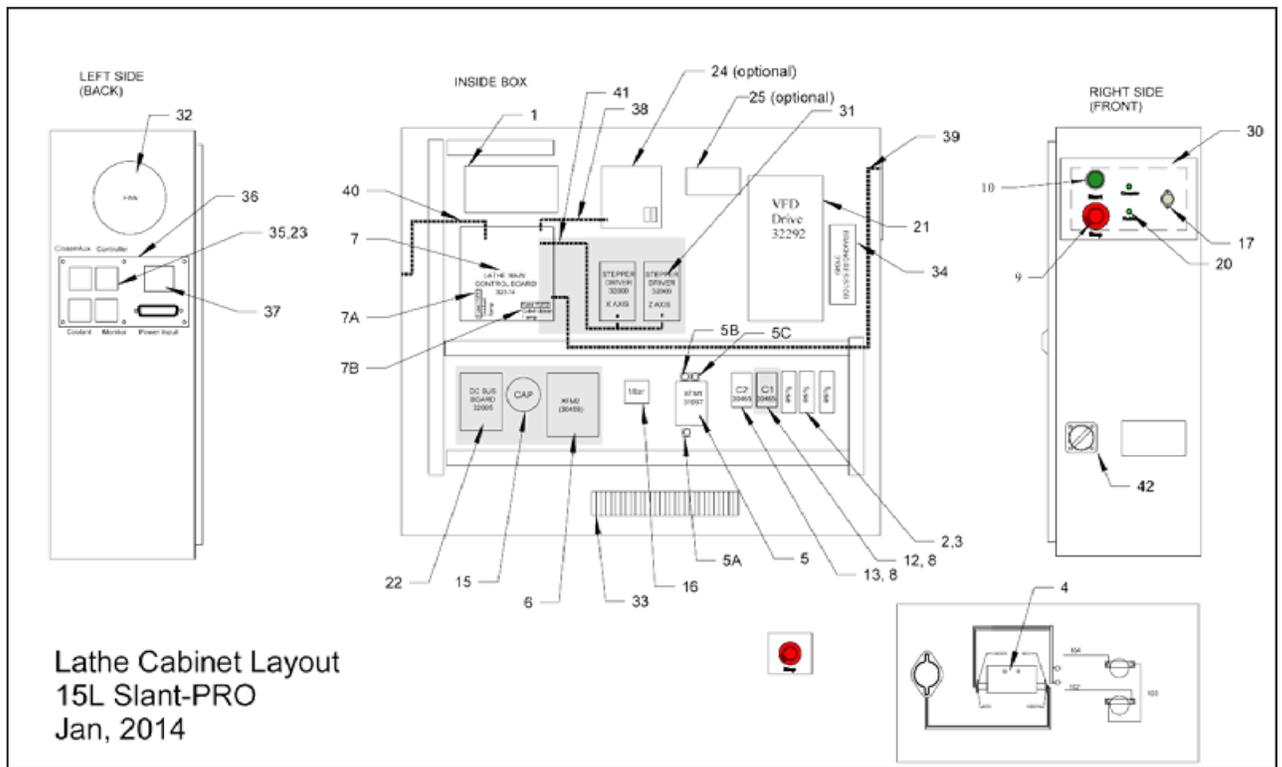


**Figure 8.7**

### 8.4.5    Spindle Drive Subsystem

#### 8.4.5.1    Overview

The spindle drive subsystem portion of the electrical schematic is highlighted in **Figure 8.8**. For a more detailed view, refer to the lathe's electrical schematic in the back of this manual.
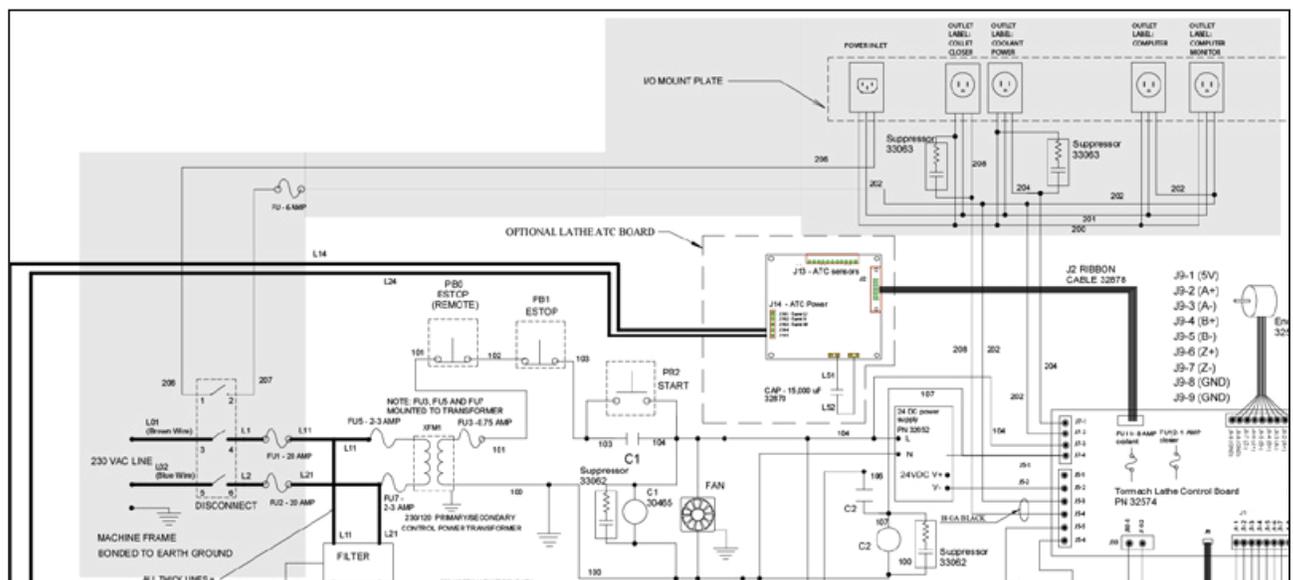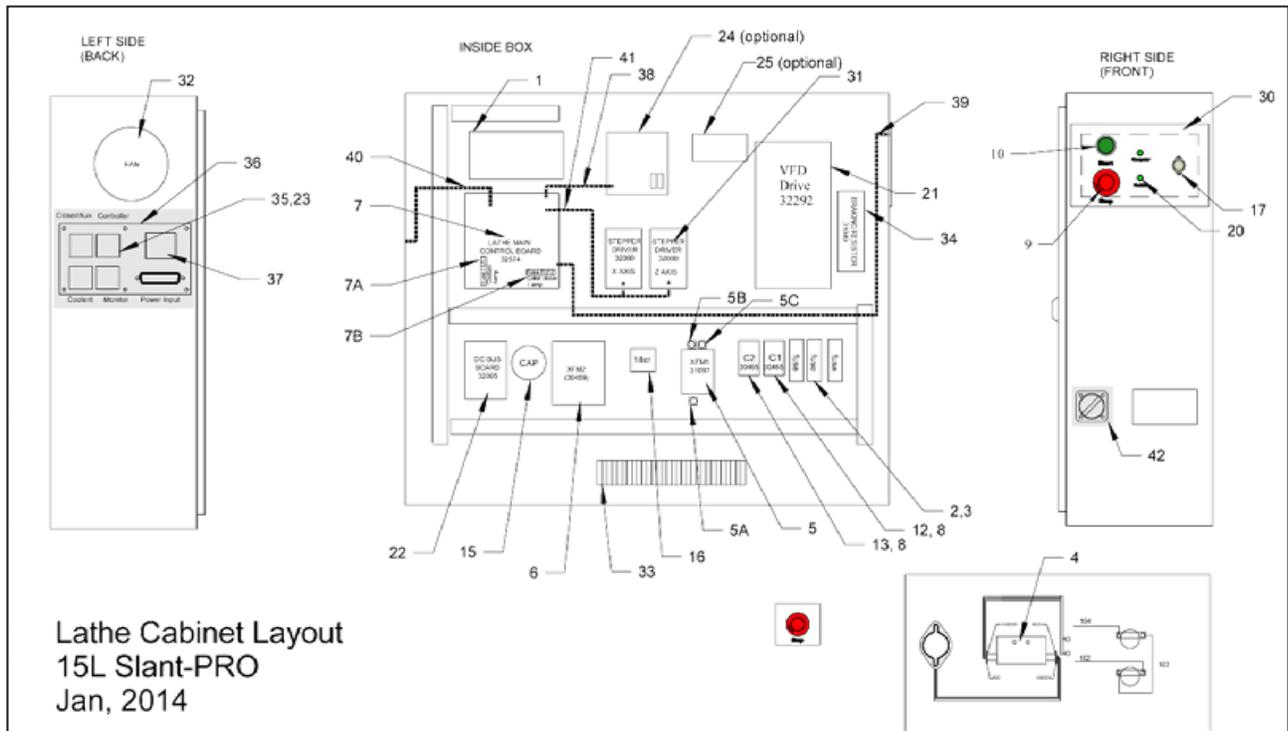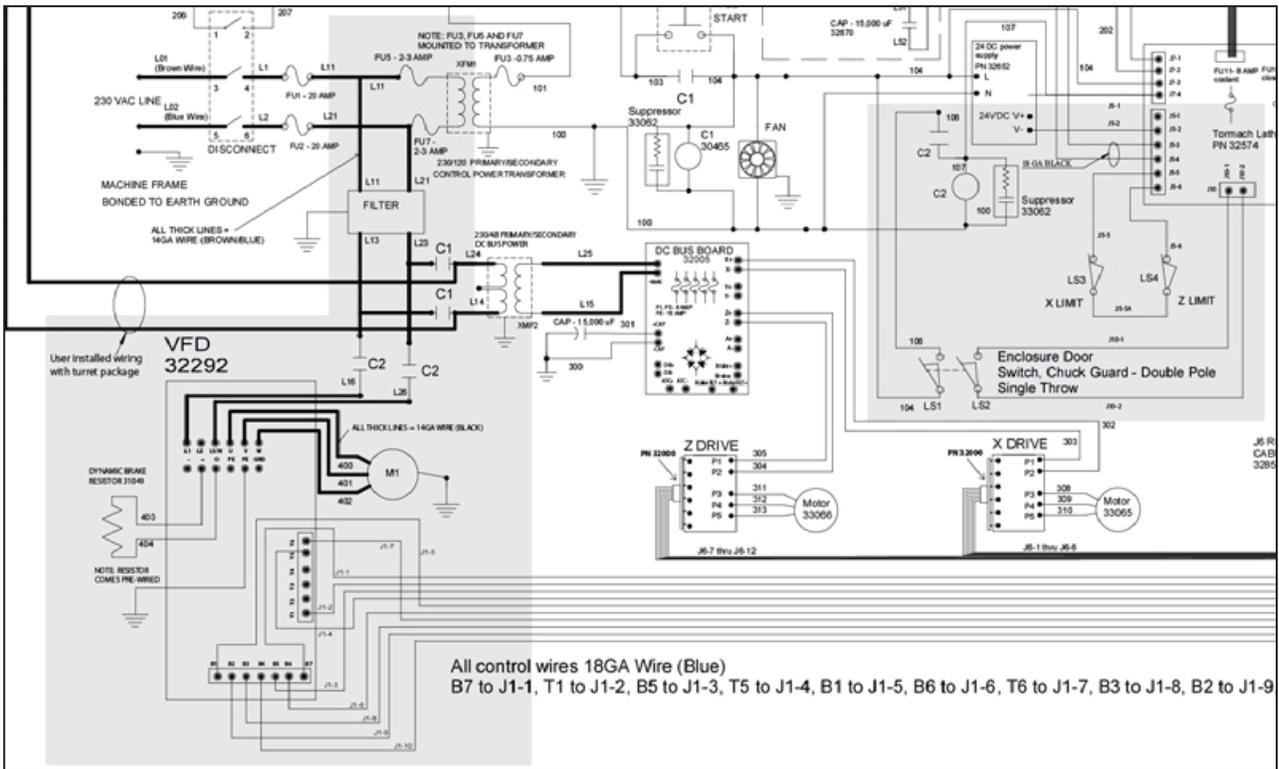


**Figure 8.8**

#### 8.4.5.2    Details of the Spindle Drive Subsystem

The spindle is powered by a three horsepower AC motor whose speed is controlled by a variable frequency drive (VFD). The spindle is in a ready-to-run condition when the control power is on and the chuck guard or enclosure door is closed.

The drive itself is powered by contactor C2, which allows power to pass when the drive is commanded to run by the controller. Power is removed from this drive whenever the chuck guard or enclosure door switch is opened, or when the E-stop removes power from the control power circuit.

*NOTE: After powering off, the VFD only powers back on when first commanded to run. The VFD will not power on by pushing the green Start button on the operator panel. Once the VFD is powered on, it will stay on until one of the above conditions occurs.*

The lathe control board provides a contact closure between wires 106 and 107 to cause power to be applied to the drive. It also provides a run command, a direction command, and a frequency command in the range of 0-10 kHz to wires J1-1 and J1-2 (COM) proportional to desired speed.

The related spindle drive distribution component locations (in machine cabinet) are highlighted in **Figure 8.9**.
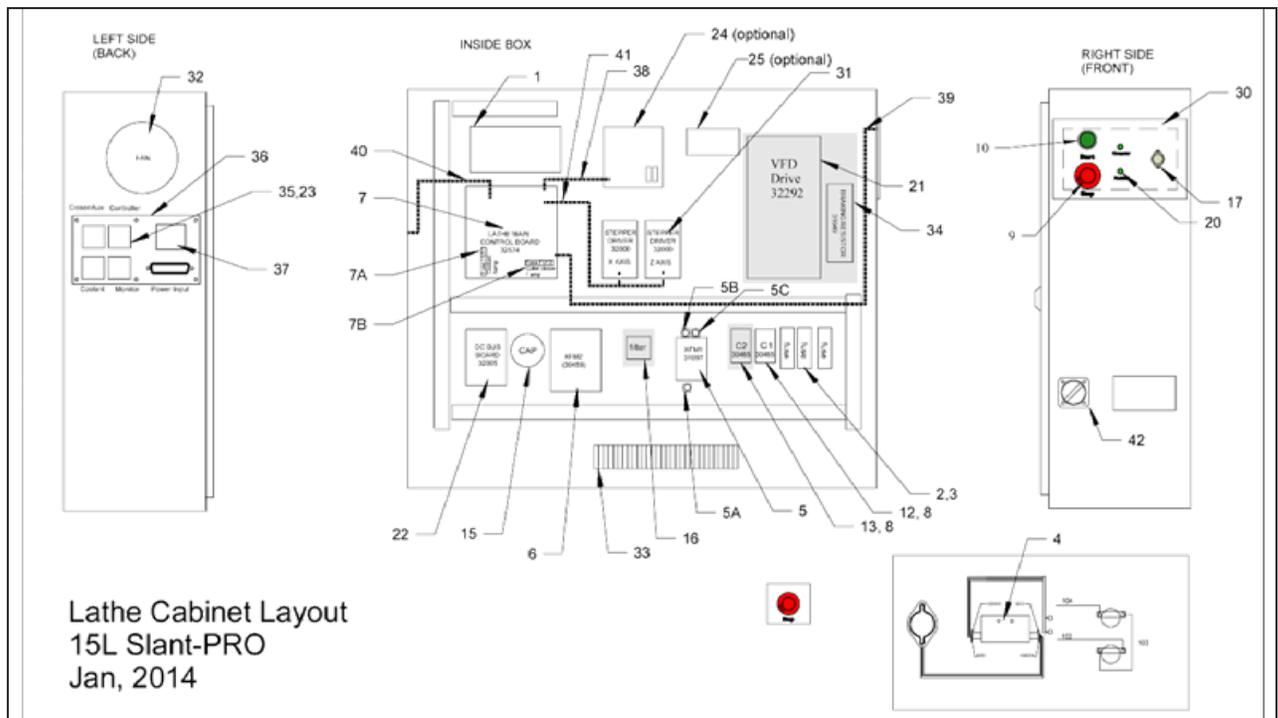


**Figure 8.9**

The display on the VFD provides valuable information for troubleshooting. The display on the VFD provides diagnostics which include:

- Motor speed, presented in motor RPMs
- Load percentage (load is proportional to the motor torque output)
- Status (ready, or rd, and inhibit, or ih) which occur when there is no connection between terminals B2 and B4 on the drive
- Fault information (tr, or trip) and a code for the fault

The drive can also provide parameter information. This is information from the Tormach program of the drive and is usually not important from an operator standpoint. If the drive is displaying this sort of information, which always has two-digit numbers displayed on the left side of the display, the operator must take action to allow the drive to display the diagnostic information.

If the right side digits are flashing, momentarily press and release the *M* button just below the display to cause the left side digits to flash. With the left side digits flashing, press and hold the *M* button for three seconds; the display changes to display diagnostics. If speed is displayed and it is desired to display load, or vice versa, press and hold the *M* button for three seconds.

**NOTE:** *Do not attempt to change any parameter values in the VFD. There are no operator-settable parameters available.*

### 8.4.5.3   Spindle Direction

To prevent unwanted motion and potential damage to lathe, check to ensure spindle rotates in the correct direction. For more information, refer to chapter 3, *Operation*.
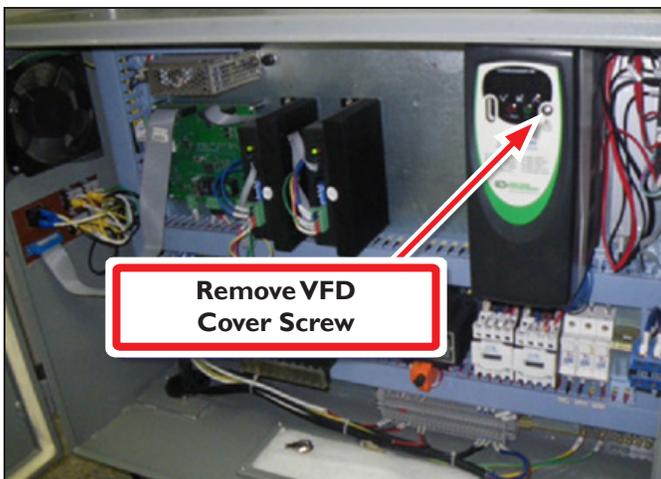


Figure 8.10



Figure 8.11

**Reversing Spindle Direction**

⚠ **WARNING! Electrical Shock Hazard:** *Be sure to power off lathe before making any electrical modifications. Failure to do so may result in personal injury or death.*

1. Power off the lathe following the power off/on procedure detailed in chapter 3, *Operation*.

2. Locate the VFD in the electrical cabinet (see **Figure 8.10**). Using a Phillips screwdriver, remove the VFD cover.

3. Identify motor wires #401 and #402, located at the bottom of the VFD (see **Figure 8.11**). Loosen the set screws above wires #401 and #402 and swap the wire connections (see **Figure 8.11**). When complete, retighten the set screws.

4. Reinstall the VFD cover and close the electrical cabinet door.

5. Power on the lathe following the power off/on procedure detailed in chapter 3, *Operation*.

6. On the *Main* screen in the PathPilot interface, click Spindle *FWD* to start spindle rotation.

7. Confirm the spindle is rotating in the correct forward direction (see **Figure 8.12**).

### 8.4.6   Home Switches

The X- and Z-axes each have one home switch which actuates at the end of travel in each direction (see **Figure 8.13**). These limit switches are used to stop an axis near its extreme travel position during a reference procedure. After referencing, PathPilot uses the referenced position as a soft limit, preventing motion beyond this position.

Sensors such as limit switches are usually the biggest source of problems on a machine. By necessity, they need to be mounted out on the machine where they are detecting events such as end of travel. This makes them vulnerable to damage. They can get fouled by coolant, chips, or physical contact. Sticking and wire damage are also a common problems.

Both X and Z switches are wired normally closed. Therefore, a broken wire or bad connection results in the lathe control card detecting a limit switch is actuated. These switches are also wired in series, so if one switch is bad or a wire becomes disconnected the control board detects that both have been actuated. The *Status* screen shows the status of the switches.

**Correct Forward Spindle Direction**
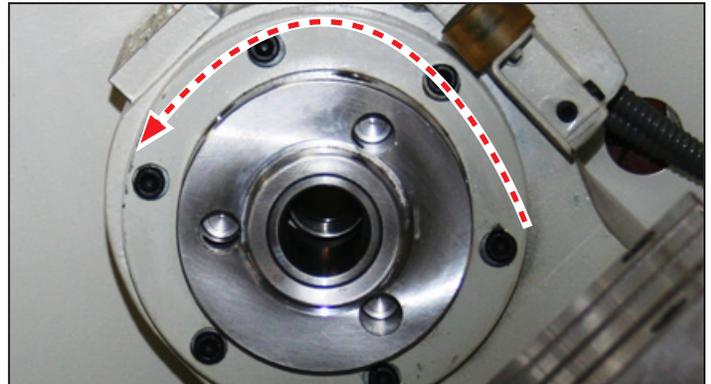


**Figure 8.12**



**Figure 8.13**

## 8.5   Troubleshooting Subsystems

Use the following flowchart to decide which section to troubleshoot:
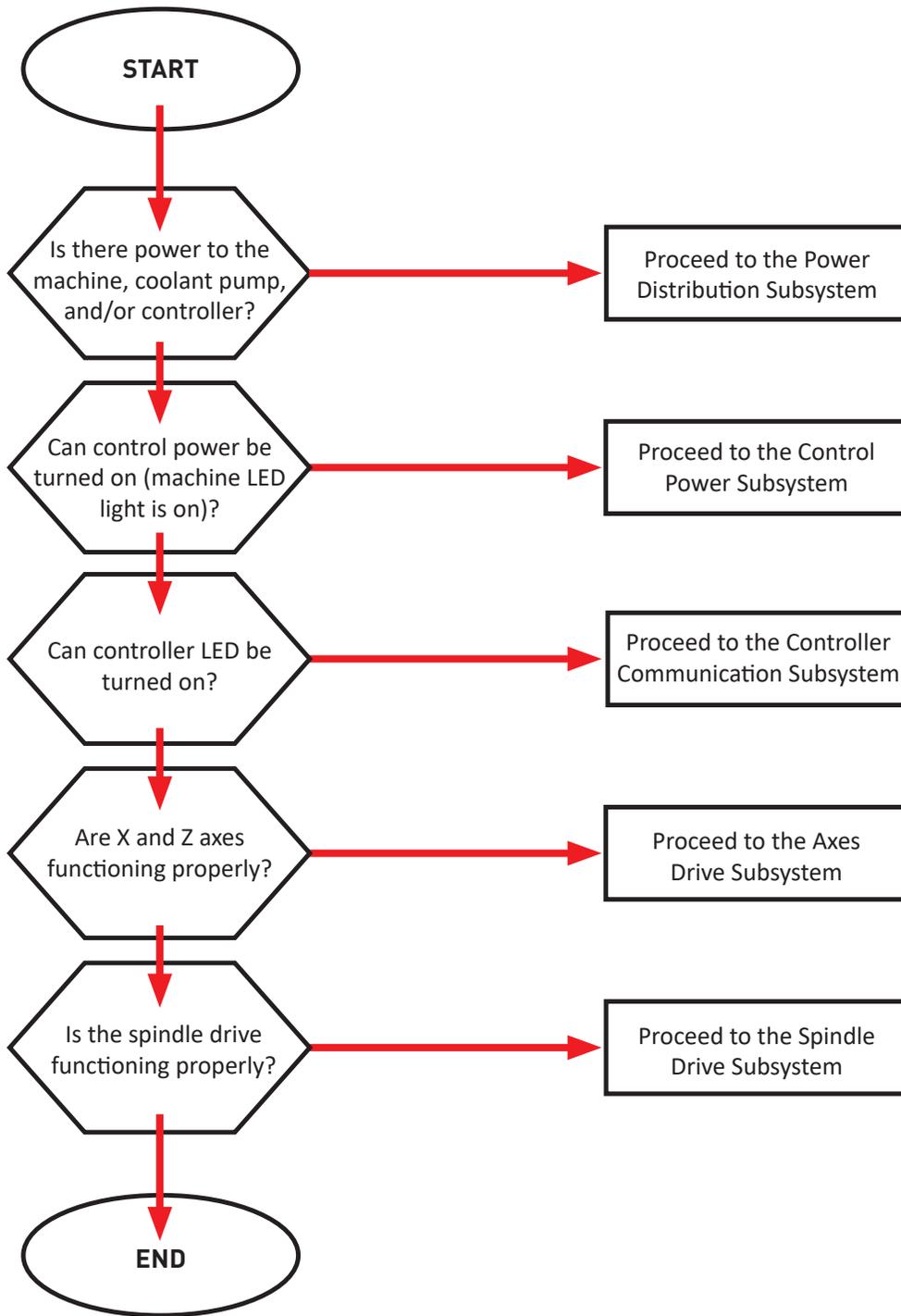
### 8.5.1   Power Distribution Subsystem

**Problem: GFI or circuit breaker for controller/coolant pump trips when main disconnect is turned to the On position**

| | |
|---|---|
| **Possible Causes** | Bad breaker or GFI; test by plugging another device (such as a drill) into the outlet. If the other device works, the outlet is OK. |
| | Loose wires in the machine cabinet; power off the lathe following the power off/on procedure detailed in chapter 3, *Operation*, and check for loose wires. Using the two finger tug test, gently pull on all wires in the power distribution circuit. |

**Problem: Controller will not turn on when main disconnect is turned to the On position**

| | |
|---|---|
| **Possible Causes** | Controller power cord has come unplugged. |
| | Auxiliary power cord from 115 VAC source to electrical cabinet has come unplugged. |
| | Controller BIOS setting is not set to After Power Failure – Power On. |

**Problem: Control power circuit will not turn on when green Start button is pressed**

| | |
|---|---|
| **Possible Causes** | One or more E-stop buttons have been depressed. Ensure that both the E-stop buttons are twisted out into the released positions. |
| | The 230 VAC supply to machine is unplugged or has no power. Check at 230 VAC outlet to make sure that lathe has power. |
| | Main disconnect has been left in the *Off* position. |
| | Fuse FU1 or FU2 blown. Check for 230 VAC across wires L1 and L2, then across wires L11 and L21. If you have voltage across L1/L2 but not across L11/L21, the fuses have blown. A red LED on the fuse holder should also indicate a blown fuse. |
| | Fuse FU3, FU5, or FU7 blown. These fuses are mounted to the machine tool control transformer. Remove the fuses and measure the resistance across them. If the fuse measures open circuit (high resistance) it is blown and should be replaced. |
| | Transformer XFM1 bad. Measure voltage at the input, and ensure that fuses FU3, FU5, and FU7 are good. If you have voltage at the input, but you do not see 115 VAC at the output (wires 100/102) the transformer is bad. |

# Troubleshooting

### 8.5.2 Controller Communications Subsystem

<table>
<tr><td colspan="2" align="center"><b>Problem: Controller communication can not be established<br>(Controller LED on operator panel will not turn on)</b></td></tr>
<tr><td rowspan="4"><b>Possible Causes</b></td><td>Controller is on and PathPilot is loaded, but <i>Reset</i> button is flashing. Controller/machine communications are established when the flashing <i>Reset</i> button is clicked. Only after this button is clicked does the Controller LED on the operator panel illuminate.</td></tr>
<tr><td>DB-25 cable between controller and lathe is unplugged, or plugged into the wrong port on the controller. Verify that both ends of the cable are securely connected, and that the controller end is plugged into the PCI-slot DB-25 connector, not into the controller's native parallel port.</td></tr>
<tr><td>Machine is not powered on. The machine tool must be powered on before the controller communication can be established.</td></tr>
<tr><td>J4 ribbon cable between DB-25 bulkhead and machine control board is unplugged or loose. Check connections and re-seat cable.</td></tr>
</table>

### 8.5.3 Axis Drivers Subsystem

<table>
<tr><td colspan="2" align="center"><b>Problem: None of the axes move when commanded</b></td></tr>
<tr><td rowspan="4"><b>Possible Causes</b></td><td>Software not commanding the move or controller problem. Jog the axis and observe the machine coordinate display. If the displayed position changes but the machine does not move there is a controller communication problem. Try restarting the controller.</td></tr>
<tr><td>Control signals not reaching the electronic driver modules. This could be a problem with the 25 pin D- connector cable from controller to machine, a problem with the ribbon cable from the DB-25 bulkhead connector to the machine control board (J4), or a problem with the ribbon cable from the machine control board to the axes drives. Check and re-seat all cables</td></tr>
<tr><td>Drivers have faulted. Check the LEDs on the X and Z driver modules. If the LED is red, the driver has faulted; power cycle the lathe following the power off/on procedure detailed in chapter 3, <i>Operation</i>. This can happen if the controller is started after the lathe is powered on.</td></tr>
<tr><td>No power to drivers. Check that the LEDs on the axes drivers are illuminated. If there are no green LEDs on the drivers, check the power from the DC bus to the drivers. For more information on troubleshooting the DC bus, refer to in the <i>Power Distribution Subsystem</i> section earlier in this chapter.</td></tr>
</table>

**Problem: One axis either will not move or will move in only one direction, but other axes operate properly**

| | |
|---|---|
| **Possible Causes** | Driver has no power or has faulted. Check the driver for the axis that will not move to make sure that the green LED is illuminated. If there is no LED, check power to the driver. If there is a red LED, power cycle the lathe following the power off/on procedure detailed in chapter 3, *Operation* – the driver has faulted. |
| | Control signal to driver is missing. Check ribbon cable connection between driver and machine control board. Try swapping the ribbon cables between the two drivers. If the axis that failed to move previously now moves, there is a bad control signal. |
| | Stepper motor wire has come loose. The motors on the 15L Slant-PRO are three-phase stepper motors. They will not function properly with one phase disconnected. Visually inspect the connections. If connections appear OK, then use a multimeter to Ohm out the wires (check U to V, V to W, U to W). Power off the lathe following the power off/on procedure detailed in chapter 3, *Operation*, and unplug the motor from the driver before taking these measurements. All measurements should read 0.5–1.0 Ohms. |

**Problem: Can not reference axes**
**(Error message: Cannot reference axis while home switch is active)**

| | |
|---|---|
| **Possible Causes** | Broken switch or disconnected wire to home switches. The limit switches are normally closed, meaning that a broken wire and an actuated switch look identical to the control software. Because the X and Z switches are wired in series, if there is a broken wire both switches will show as being actuated on diagnostic LEDs on the *Status* screen. Visually inspect the wiring to the switches inside the switch box as well as at the connection to the machine control board. Use an Ohm meter to check the function of both switches, and to trace back a possible break in a wire. |
| | Bad machine control board. Make a small jumper wire and insert it between the J5-5 and J5-6 terminals on the control board. If the LED on the *Status* screen doesn't change state when this jumper is inserted, the control board or the DB-25 connection between the control board and the controller is defective. |

**NOTE:** *There is a temporary work around for a malfunctioning home switch circuit. On the Settings screen in the PathPilot interface, check the Disable Home Switches check box. The Ref X and Ref Z buttons now set the machine home position to the machine position at the time the button was pressed.*

**Problem: Steps are lost on axis travel**

| | |
|---|---|
| **Possible Causes** | Improper use of tool offset or work offset (G54-59). By far the most common cause of a perceived loss of position or lost steps is operator error. You can quickly check to see if the machine has truly lost its position by jogging to the end of travel (near where the home switch engages). The machine has soft limits that are set to 0.010" from the point at which the home switch triggers. If you are able to jog passed the trigger point of the home switch, or you can not jog to within 10 thousandths of the home switch, the machine has lost position. If you can jog right up to the soft limit, look at the DRO and record the position read out on the DRO. Then reference that axis again – the new read out should be 0.010" less than the value you wrote down. If these numbers match (within the repeatability of the home switch, +/- 0.001"), the machine has not lost position. |
| | Motor pulley or ball screw pulley loose or cracked. Visually inspect drive pulleys for wear or slippage. |
| | Workpiece not firmly held in chuck or 5C collet. If the apparent position loss occurs in Z only, it may actually be that the workpiece is being pushed back into the spindle by the cutting forces. This is especially possible when using 5C collets. A 5C lever action closer can come out of adjustment over time, or the stock being used can vary in diameter such that holding power is reduced. |
| | Obstruction or excessive friction (gibs not adjusted properly or poor lubrication). Binding in the system is usually easy to diagnose because you can hear the motor stalling. Mechanical issues most often result in losing a large number of steps. Typical mechanical issues include an increase in friction due to lack of oil at the way surfaces or ball screw and/or improperly adjusted gibs. For more information, refer to chapter 7, *Maintenance*. |
| | High load in the mechanical system. This can frequently be the cause of position loss on the Z axis during drilling cycles, which can require thousands of pounds of thrust for larger diameter drills. Keep your drill sizes to less than 1/2", and then step up in successive larger drill sizes or use a boring bar to make larger diameter holes. |

### 8.5.4 Spindle Drive Subsystem

**Problem: Spindle drive will not power on**

| | |
|---|---|
| **Possible Causes** | First check to make sure that the drive has power after a spindle speed command is given. If the spindle drive is powered on but the spindle isn't running, refer to *Spindle VFD Fault Conditions* section later in this chapter. |
| | Chuck guard or enclosure door is not closed. The chuck guard or enclosure door must be fully closed before the spindle powers on. |
| | Zero RPM is commanded in the spindle RPM DRO. The spindle will not start if given a zero RPM command. |
| | RPM override slider is at 0 percent. If the RPM overrides are set to zero percent, the spindle gets a zero speed command and will not start. Verify that the RPM slider is at an appropriate level. |
| | The machine is in constant surface speed mode, but the surface speed command and current X position of the lathe is such that the commanded RPM is less than the minimum. |
| | Chuck guard or enclosure door switches are not functioning properly. There are two switches on the machine guard: one conducts 5VDC and communicates the state of the guarding to PathPilot; the other conducts 115 VAC and allows the contactor that powers the VFD to energize. Both switches must function properly for the spindle to turn on.<br>• A failure of the 5VDC switch should be obvious, as PathPilot tells you that the door must be closed before you can start the spindle. If you get these error messages even though the door is closed, examine the switch for loose wires, make sure the J10 connector on the control board is connected, and Ohm out the switch.<br>• A failure of the 115 VAC switch is less obvious. PathPilot thinks the door is closed and tries to start the spindle, but the spindle drive will not receive power. If you can press the spindle *FWD* button in PathPilot, and you do not receive an error message, but the spindle does not turn on, you may have a fault in the 115 VAC switch circuit. Examine the switch for broken or disconnected wires, as well as the J7-3 and J7-4 connections on the machine control board. If nothing is found, power off the lathe following the power off/on procedure detailed in chapter 3, *Operation*, and Ohm out the switch. |

# Troubleshooting

### Problem: Spindle drive powers on, but spindle will not turn on

| | |
|---|---|
| **Possible Causes** | Speed signal to VFD is not present – drive readout displays *spd 00*. Check wires J1-1 and J1-2 and ensure they are properly seated at both control board and at VFD inputs. |
| | Enable signal is missing at drive – drive readout displays lh. Check wires J1-9 and J1-10 at both the control board and at the VFD inputs. |
| | VFD is tripped – the display shows if the VFD has tripped, along with a trip code. Record the information from the display should the trip be happening frequently. VFD trips can be cleared by powering off the drive, which can be done by opening the chuck guard or enclosure door, or by power cycling the lathe following the power off/on procedure detailed in chapter 3, *Operation*. |

### Run and Direction Commands to Drive

| Command From Card | Monitoring Points One Probe on each | | Voltage Measured | |
|---|---|---|---|---|
| | Common Wire Number | Wire Number | Voltage when Control Board Command *is* on | Voltage when Control Board Command is *not* on |
| Run | J1-2 | J1-3 | 20-28 VDC | 0 VDC |
| Reverse | J1-2 | J1-6 | 0 VDC | 20-28 VDC |

### Spindle VFD Fault Conditions

| Trip Code | Condition | Likely Cause |
|---|---|---|
| UU | DC Bus Under voltage | This happens when VFD is powered off. |
| OU | DC Bus Over voltage | Braking resistor failed open or wiring connection open between VFD and resistor. Resistance to measure 75 Ohms. |
| OI.AC | VFD output instantaneous over current | Phase to phase or phase to ground short on output of VFD to motor. |
| OI.br | Braking resistor instantaneous over current | Full or partial braking resistor, or short in wiring between VFD and resistor. Resistance should measure 75 Ohms. |
| It.br | $I^2t$ (power) on braking resistor | Frequent and severe deceleration cycles trigger excessive braking resistor energy, or AC supply voltage too high. |
| It.AC | $I^2t$ (power) on VFD output current (used to protect motor) | Spindle motor is over worked. Consider cooling down motor by running spindle with no load at half speed for 10 minutes. |

(continued on next page...)

**Spindle VFD Fault Conditions  (...continued)**

| | | |
|---|---|---|
| O.ht1 | VFD thermal trip – VFD stops to cool down power electronics and prevent failure. | Spindle motor is over worked. Stop running spindle but leave VFD power on and allow power electronics to cool down. |
| O.ht2 | Heat sink temperature is too high because VFD is working too hard and needs to stop to cool power electronics down and prevent failure. Cabinet may also be too hot. | Spindle motor is over worked or room temperature is too high. Stop running spindle but leave VFD power on; let power electronics cool down. Check to see if fan on VFD is running and check filters on cabinet reduce room temperature. |
| Hf.29 | Cooling fan not cooling | Failed drive. |

# Diagrams and Parts Lists

## 9.   Diagrams and Parts Lists

### 9.1   Base Assembly (exploded view)



BASE
EXPLODED VIEW
2-10-2014
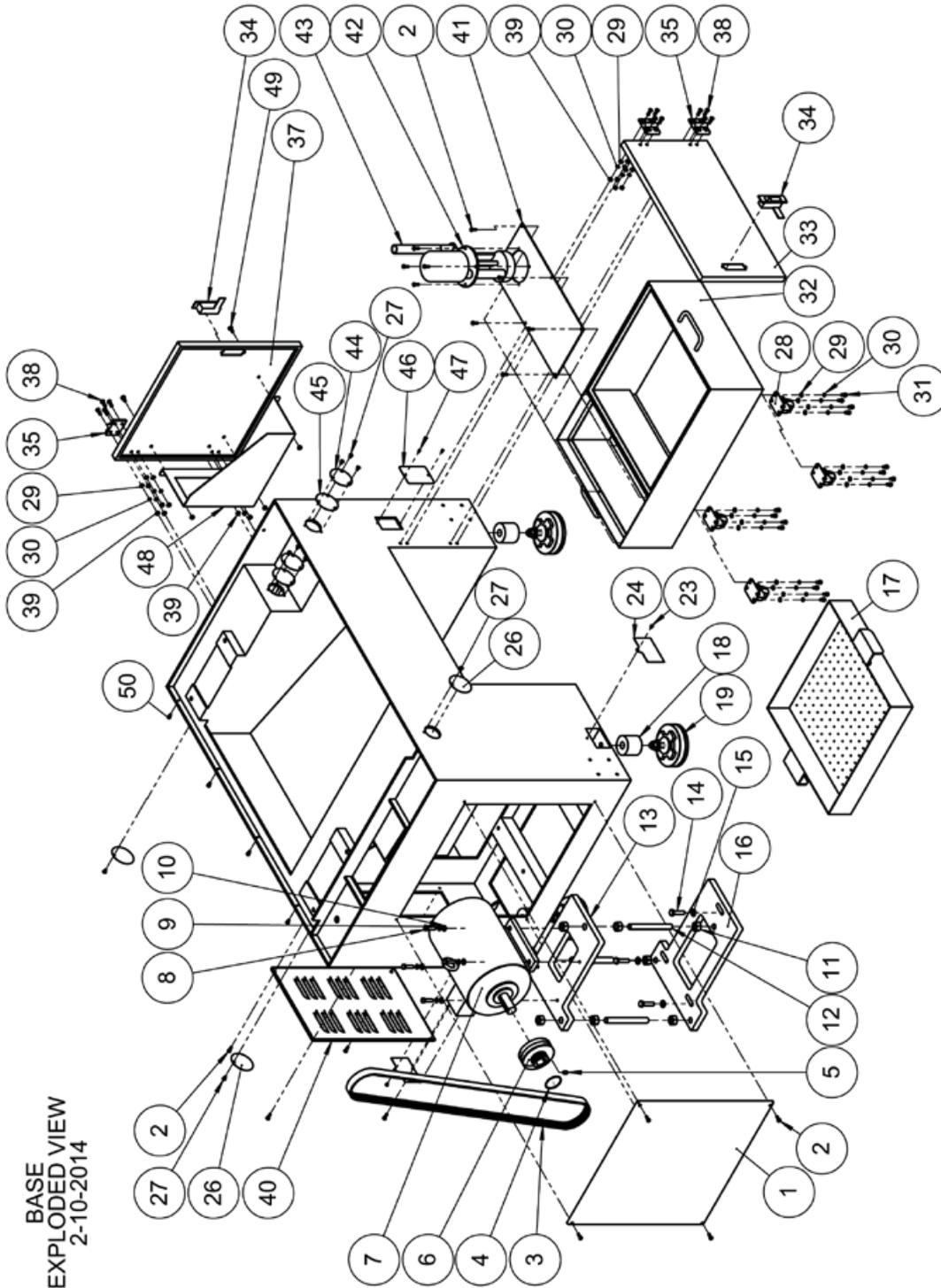
**Base Assembly Parts List**

| ID | PN | Description | ID | PN | Description |
|----|----|-------------|----|----|-------------|
| 1 | 34327 | End Panel | 28 | 34348 | Wheel |
| 2 | 34328 | Screw, M5 x 6 | 29 | 34349 | Washer, Flat M6 |
| 3 | 33226 | Drive Belt[1] | 30 | 34350 | Washer, Lock M6 |
| 4 | 34330 | Ring | 31 | 34351 | Screw, M6 x 10 |
| 5 | 34331 | Set Screw M8 x 14, point | 32 | 34352 | Coolant Tank |
| 6a | 33208 | Drive Pulley 80 mm | 33 | 34353 | Door, Front |
| 6b | 33229 | Drive Pulley 110 mm | 34 | 34354 | Door Latch |
| 7 | 33230 | Spindle Motor | 35 | 34355 | Door Hinge |
| 8 | 34332 | Bolt, M8 x 30 | 36 | 34356 | Base |
| 9 | 34333 | Washer Lock, M8 | 37 | 34357 | Side Door |
| 10 | 34334 | Washer Flat, M8 | 38 | 34358 | Screw, M6 x 20 |
| 11 | 34335 | Nut, M16 | 39 | 34359 | Nut, M6 |
| 12 | 34336 | Motor Mount Stud | 40 | 34360 | Rear Cover |
| 13 | 34337 | Motor Mount Plate | 41 | 34361 | Pump Mount Plate |
| 14 | 34338 | Bolt, M10 x 40 | 42 | 32746 | Coolant Pump |
| 15 | 34339 | Washer, M10 | 43 | 34362 | Coolant Hose |
| 16 | 34340 | Motor Adjustment Plate | 44 | 34363 | Cover Plate |
| 17 | 34341 | Swarf Tray | 45 | 34364 | Cover Plate |
| 18 | 34342 | Foot Spacer | 46 | 34365 | Outlet Cover Plate |
| 19 | 34343 | Foot Pad, 15L Lathe Stand, M14 × 2 - 80 mm Threaded Stud | 47 | 34366 | Screw, M3 x 8 |
| 23 | 34344 | Screw, M4 x 6 | 48 | 34367 | Controller Tray |
| 24 | 34345 | Cover Plate | 49 | 34368 | Screw, M6 x 14 |
| 26 | 34346 | Cover Plate | 50 | 34369 | Screw, M5 x 10 |
| 27 | 34347 | Screw, M5 x 8 | | | |

[1]Gates # 2/11M1950JB

## 9.2 Lathe Bed Z-axis Assembly (exploded view)



LATHE BED / Z-AXIS
EXPLODED VIEW
2-10-2014
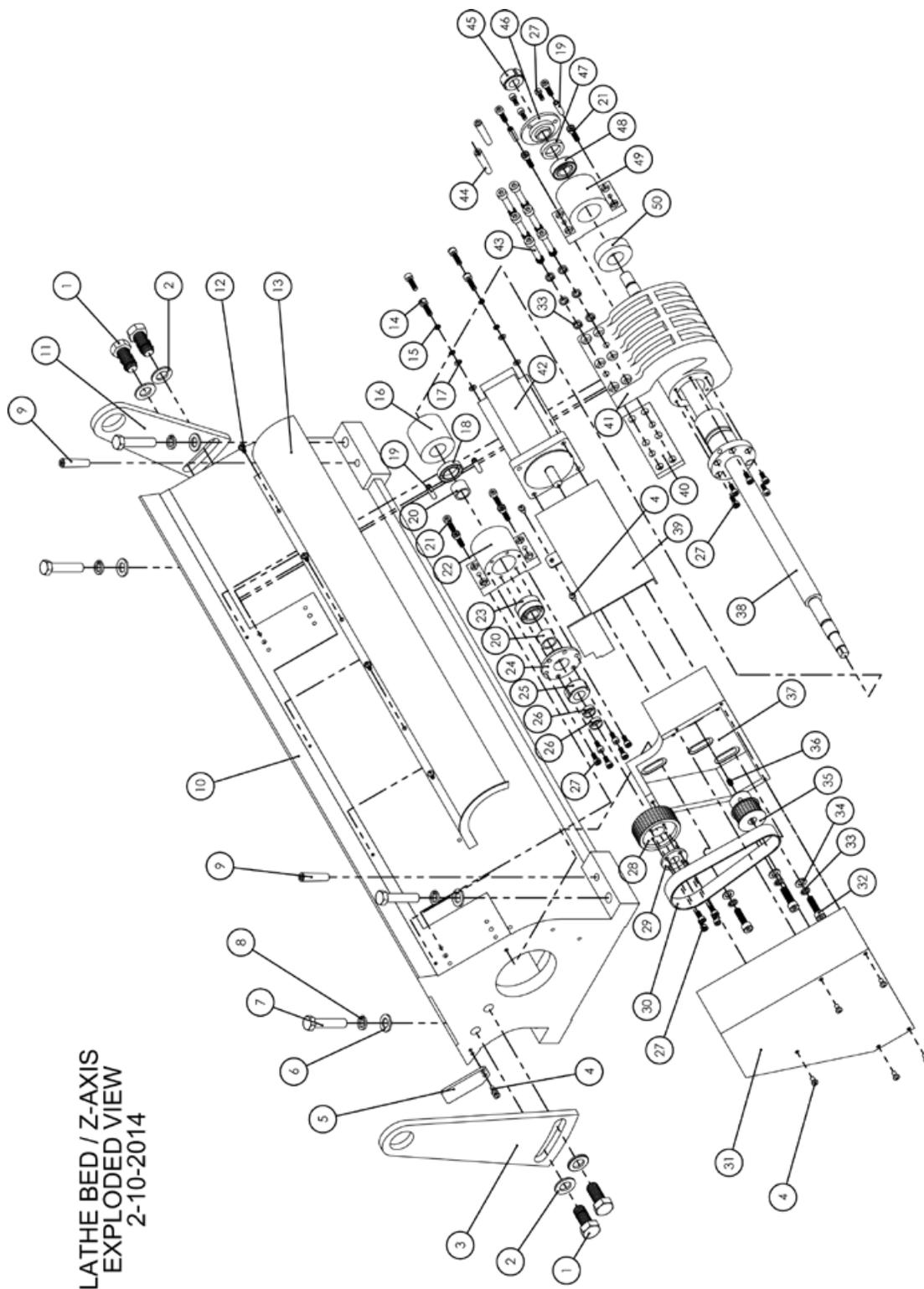
**Lathe Bed Z-axis Parts List**

| ID | PN | Description | ID | PN | Description |
|----|-----|-------------|----|-----|-------------|
| 1 | 34155 | Bolt, M16 x 35 | 26 | 34180 | Spacer, Z Pulley |
| 2 | 34156 | Washer, Lock M16 | 27 | 34181 | Screw, M5 x 12 |
| 3 | 34157 | Lifting Hanger RH | 28 | 34182 | Pulley, Z Ball Screw |
| 4 | 34158 | Screw, M8 x 8 | 29 | 34183 | Pulley Mount Flange Z Screw |
| 5 | 34159 | Z Limit Switch Stop | 30 | 34184 | Belt, Z-axis Drive |
| 6 | 34160 | Washer, Flat M12 | 31 | 34185 | Belt Cover, Z-axis |
| 7 | 34161 | Bolt, M12 x 55 | 32 | 34186 | Screw, M8 x 30 |
| 8 | 34162 | Washer, Lock M12 | 33 | 34187 | Washer, Lock M8 |
| 9 | 34163 | Pin, M10 x 45 | 34 | 34188 | Washer, Flat M8 |
| 10 | 34164 | Bed Casting | 35 | 34189 | Pulley, Z Motor |
| 11 | 34165 | Lifting Hanger LH | 36 | 34190 | Set Screw, M6 x 10 |
| 12 | 34166 | Screw, M5 x 10 | 37 | 34191 | Z Motor Mount |
| 13 | 34167 | Ballscrew Cover, Z-axis | 38 | 33224 | Ball Screw, Z-axis Lathe |
| 14 | 34168 | Screw, M5 x 20 | 39 | 34192 | Motor Cover |
| 15 | 34169 | Washer, Lock M5 | 40 | 34193 | Mount Pad, Z Carrier |
| 16 | 34170 | Bumper, Z-axis RH | 41 | 34194 | Carrier, Z Ballscrew |
| 17 | 34171 | Washer, Flat M5 | 42 | 33066 | Z-axis Stepper Motor |
| 18 | 34172 | Seal, Z-axis Ballscrew | 43 | 34195 | Screw, M8 x 50 |
| 19 | 34173 | Pin, M6 x 30 | 44 | 34196 | Pin, M8 x 65 |
| 20 | 34174 | Spacer, Z Screw bearing | 45 | 34197 | Spanner Nut, M15 x 1 |
| 21 | 34175 | Screw, M6 x 25 | 46 | 34198 | Bearing Cap |
| 22 | 34176 | Bearing Mount Block RH | 47 | 34199 | Felt Washer |
| 23 | 34177 | Bearing, Z Ballscrew Mount RH[1] | 48 | 34200 | Bearing, Z Ballscrew Mount LH[2] |
| 24 | 34178 | Bearing Cap | 49 | 34201 | Bearing Mount Block LH |
| 25 | 34179 | Spanner Nut, M17 x 1 | 50 | 34202 | Bumper, Z-axis LH |

[1]ABEC Identification: 7003B/DB/P5 (1 unit includes a bearing pair)

[2]ABEC Identification: 6003-22

## 9.3　Lathe Cross Slide Assembly (exploded view)

CROSS SLIDE
EXPLODED VIEW
2-10-2014
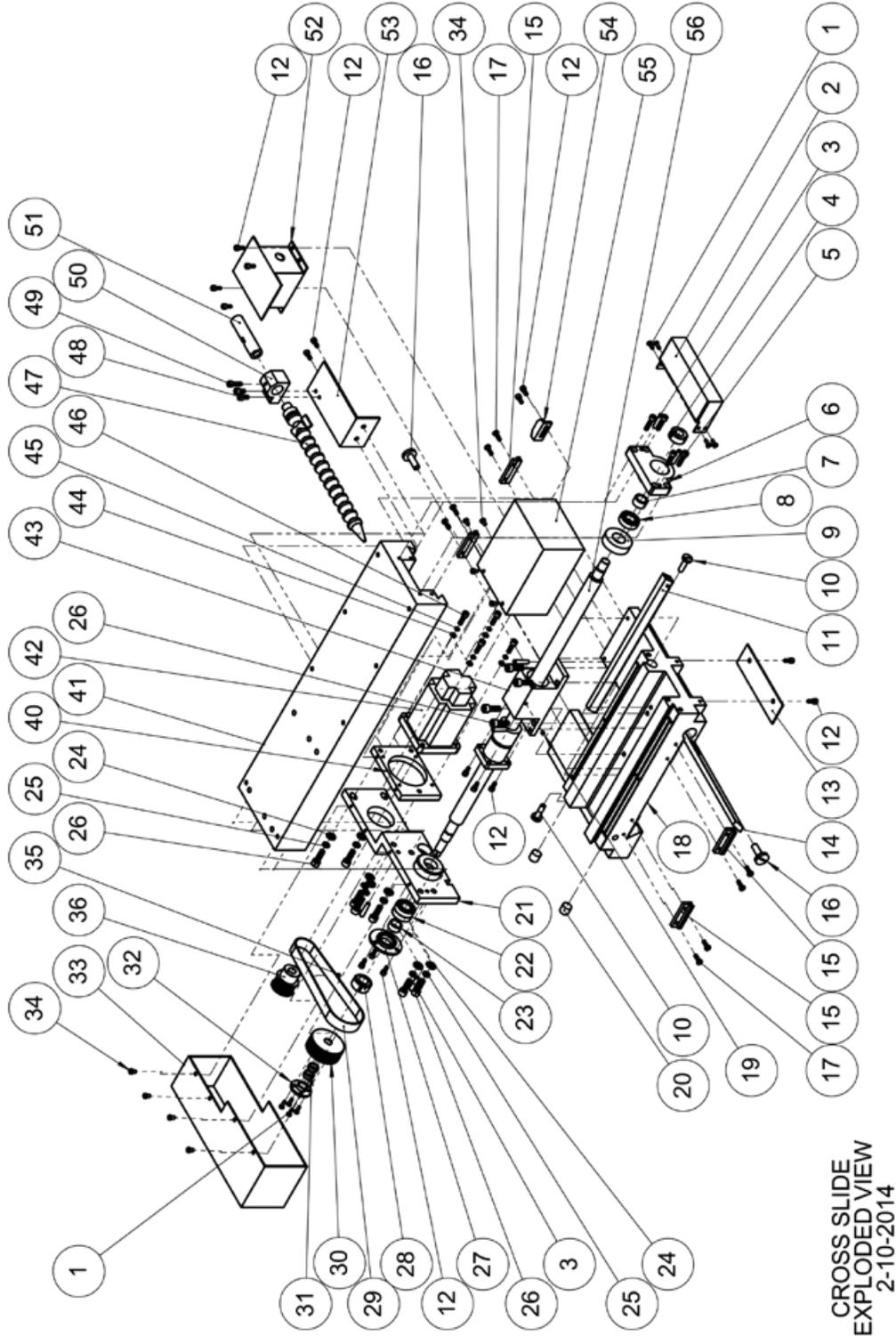
**Lathe Cross Slide Parts List**

| ID | PN | Description | ID | PN | Description |
|---|---|---|---|---|---|
| 1 | 34203 | Screw, M4 x 12 | 28 | 34179 | Spanner Nut, M17 x 1 |
| 2 | 34204 | Cover Tray, X Lower | 29 | 33228 | Belt, X-axis |
| 3 | 34205 | Pin, M6 x 30 | 30 | 34232 | Pulley, X-axis Ballscrew |
| 4 | 34197 | Spanner Nut, M15 | 31 | 34233 | Locking Sleeve |
| 5 | 34207 | Screw, M6 x 25 | 32 | 34234 | Pulley Mount Flange X Screw |
| 6 | 34208 | Mount Plate, X Ballscrew Lower | 33 | 34235 | Belt Cover, X-axis |
| 7 | 34209 | Spacer | 34 | 34236 | Screw, M5 x 8 |
| 8 | 34210 | Bearing, X-axis Idler[1] | 35 | 34237 | Set Screw, M6 x 10 |
| 9 | 34211 | Bumper, X-axis Lower | 36 | 34238 | Pulley, X-axis motor |
| 10 | 34212 | Gib Screw, X-axis | 40 | 34242 | Motor Mount Plate, X-axis |
| 11 | 34213 | Gib, X-axis | 41 | 34243 | Cross Slide |
| 12 | 34214 | Screw, M5 x 12 | 42 | 35452 | Stepper Motor, X-axis Lathe |
| 13 | 34215 | Plate | 43 | 34245 | Carrier Block, X Ballnut |
| 14 | 34216 | Gib, Z-axis | 44 | 34246 | Washer, Flat M5 |
| 15 | 34217 | Way Wiper | 45 | 34247 | Washer, Lock M5 |
| 16 | 34218 | Gib Screw, Z-axis | 46 | 34248 | Screw, M5 x 20 |
| 17 | 34219 | Screw, M5 x 12 | 47 | 32833 | Coolant Nozzle (Blue Orange) |
| 18 | 34220 | Carriage Saddle | 48 | 34244 | Screw, M5 x 12 |
| 19 | 34221 | Set Screw, M5 x 5 Flat Tip | 49 | 34249 | Screw, M6 x 20 |
| 20 | 34222 | Bumper, X-axis Upper | 50 | 31105 | Coolant Coupling Mount |
| 21 | 34223 | Mount, X-axis | 51 | 32832 | Coolant Coupling |
| 22 | 34224 | Bearing, X-axis Upper[2] | 52 | 34239 | Limit Switch Cover |
| 23 | 34225 | Spacer, Bearing | 53 | 34240 | Coolant Mount Bracket |
| 24 | 34226 | Washer, Flat M8 | 54 | 34241 | Limit Switch Ramp, X-axis |
| 25 | 34227 | Washer, Lock M8 | 55 | 34231 | Motor Cover |
| 26 | 34228 | Screw, M8 x 25 | 56 | 33223 | Ballscrew, X-axis |
| 27 | 34229 | Bearing Cap | | | |

[1]ABEC Identification: 6003-2Z

[2]ABEC Identification: 7003B/DB/P5 (1 unit includes a bearing pair)

## 9.4 Lathe Headstock Assembly (exploded view)
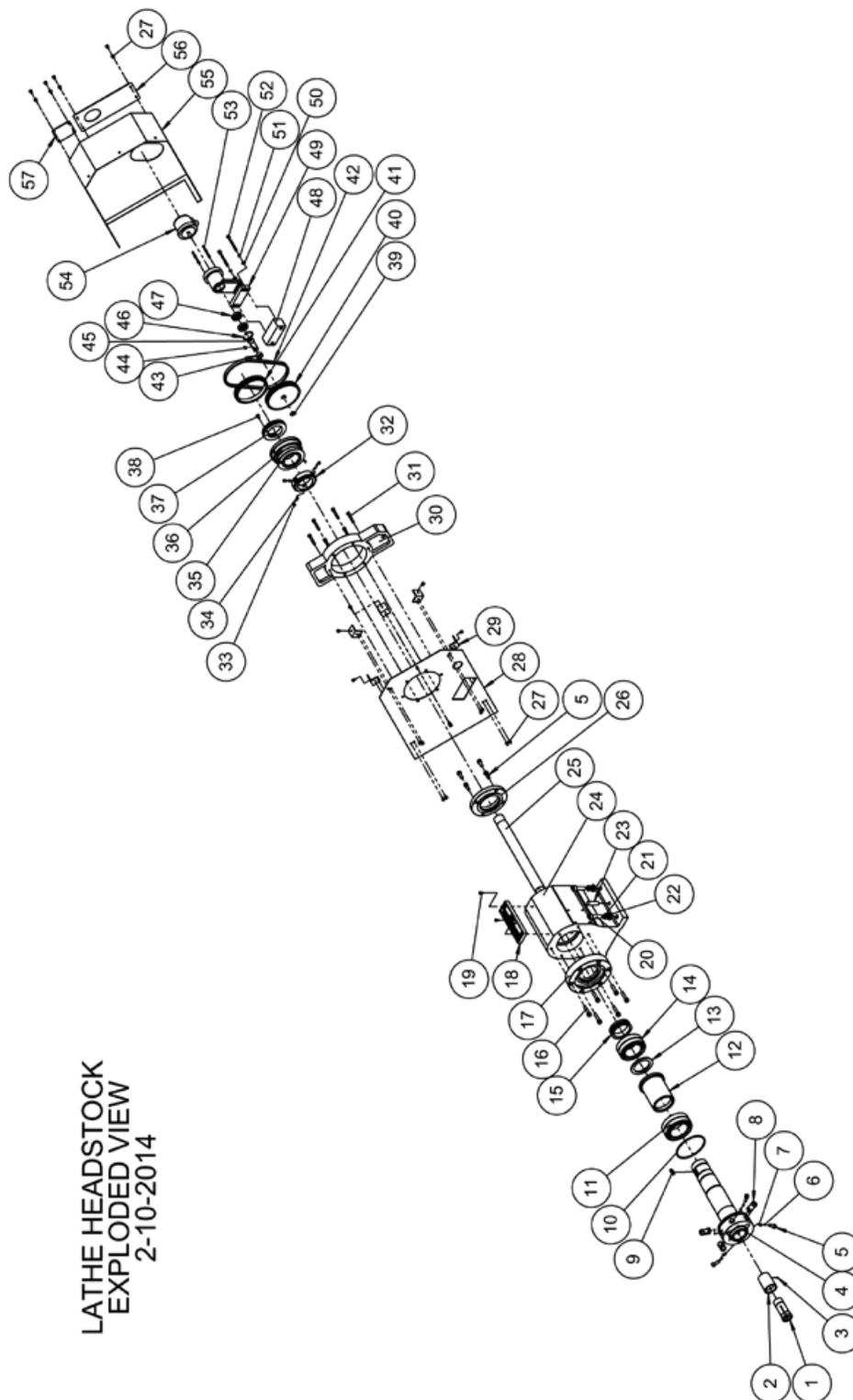


LATHE HEADSTOCK
EXPLODED VIEW
2-10-2014

## Lathe Headstock Parts List

| ID | PN | Description | ID | PN | Description |
|----|-----|-------------|----|-----|-------------|
| 1 | — | 5C Collet | 30 | 34278 | Closer Mount Bracket |
| 2 | 33325 | 5C Insert | 31 | 34279 | Screw, M6 x 40 |
| 3 | 34251 | 5C Insert Pin | 32 | 34280 | Spanner Nut |
| 4 | 34252 | Spindle, Lathe | – | 34306 | Spanner Nut Kit[4] |
| 5 | 34253 | Screw, M8 x 20 | 33 | 34281 | Set Screw, M8 x 12 |
| 6 | 34254 | Spring, Cam Lock | 34 | 34282 | Brass Pad |
| 7 | 34255 | Cam Lock Pin | 35 | 34283 | Set Screw, M5 x 8 Cone Tip |
| 8 | 34256 | Cam Lock | 36 | 34284 | Spindle Pulley |
| – | 34250 | D1 Rebuild Kit[1] | 37 | 34285 | Spindle Lock Nut |
| 9 | 34257 | Key, M8 x 25 | 38 | 34286 | Screw, M6 x 12 |
| 10 | 34258 | O-ring Seal, Spindle Nose | 39 | 34287 | Retaining Clip |
| 11 | 34259 | Bearing, Spindle Nose[2] | 40 | 34288 | Encoder Pulley |
| 12 | 34260 | Spacer Sleeve, Spindle Bearing | 41 | 34289 | Encoder Ring Pulley |
| 13 | 34261 | Spacer Ring, Spindle | 42 | 33227 | Belt, Encoder |
| 14 | 34262 | Bearing, Spindle Tail[3] | 43 | 34291 | Spacer, Encoder Bearing |
| 15 | 34263 | Seal Sleeve | 44 | 34292 | Key, M5 x 10 |
| 16 | 34264 | Screw, M8 x 35 | 45 | 34293 | Encoder Shaft |
| 17 | 34265 | Nose Cap, Headstock | 46 | 34294 | Retaining Ring |
| 18 | 34266 | Logo Casting | 47 | 34295 | Bearing, Encoder |
| 19 | 34267 | Screw, M5 x 20 | 48 | 34296 | Mount Spacer, Encoder |
| 20 | 34268 | Screw, M12 x 40 | 49 | 34297 | Mount, Encoder |
| 21 | 34269 | Washer, Lock M12 | 50 | 34298 | Washer, Flat M6 |
| 22 | 34270 | Washer, Flat M12 | 51 | 34299 | Washer, Lock M6 |
| 23 | 34271 | Pin, M10 x 30 | 52 | 34300 | Screw, M6 x 65 |
| 24 | 34272 | Head Stock Casting | 53 | 34301 | Screw, M4 x 20 |
| 25 | 34273 | Do not use | 54 | 32560 | Encoder |
| 26 | 34274 | Tail Cap, Head Stock | 55 | 34303 | Belt Guard Cover |
| 27 | 34275 | Screw, M5 x 8 | 56 | 34304 | Guard Hole Cover |
| 28 | 34276 | Belt Cover Panel | 57 | 34305 | Guard Hole Cover |
| 29 | 34277 | Cover Bracket | | | |

[1]D1 Cam Lock Rebuild Kit (includes items #5, 6, 7, 8 = qty 3 each)

[2]ABEC Identification: 7013AC/DB (1 unit includes a bearing pair)

[3]ABEC Identification: 7012AC/DB (1 unit includes a bearing pair)

[4]Spanner Nut Kit (includes items #32, 33, 34)

# Diagrams and Parts Lists

## 9.5 Lathe Cabinet Hardware (exploded view)



CABINET
EXPLODED VIEW
2-10-2014
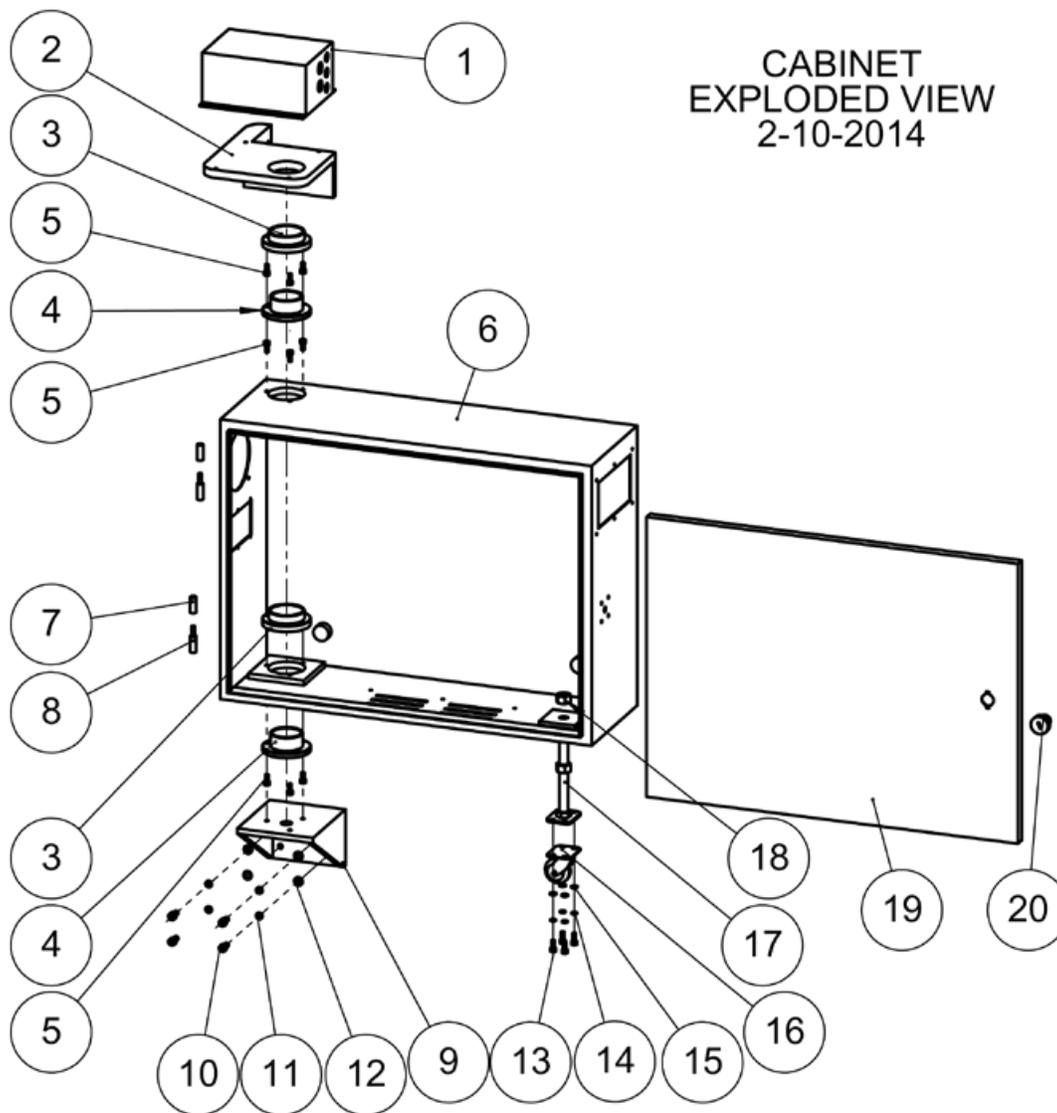
**Lathe Cabinet Hardware Parts List**

| ID | PN | Description |
|----|-------|-------------|
| 1 | 34135 | Junction Box |
| 2 | 34136 | Top Hinge Mount |
| 3 | 34137 | Upper Hinge Collar |
| 4 | 34138 | Lower Hinge Collar |
| 5 | 34139 | Screw, M5 x 12 |
| 6 | 34140 | Cabinet |
| 7 | 34141 | Hinge Body (welded) |
| 8 | 34142 | Hinge Pin (welded) |
| 9 | 34143 | Bottom Hinge Mount |
| 10 | 34144 | Screw, M8 x 16 |
| 11 | 34145 | Washer, Lock M8 |
| 12 | 34146 | Washer, Flat M8 |
| 13 | 34147 | Screw, M6 x 16 |
| 14 | 34148 | Washer, Lock M6 |
| 15 | 34149 | Washer, Flat M6 |
| 16 | 34150 | Wheel |
| 17 | 34151 | Wheel Point |
| 18 | 34152 | Nut, M16 |
| 19 | 34153 | Cabinet Door |
| 20 | 30467 | Door Latch |

## 9.6 Electrical Cabinet
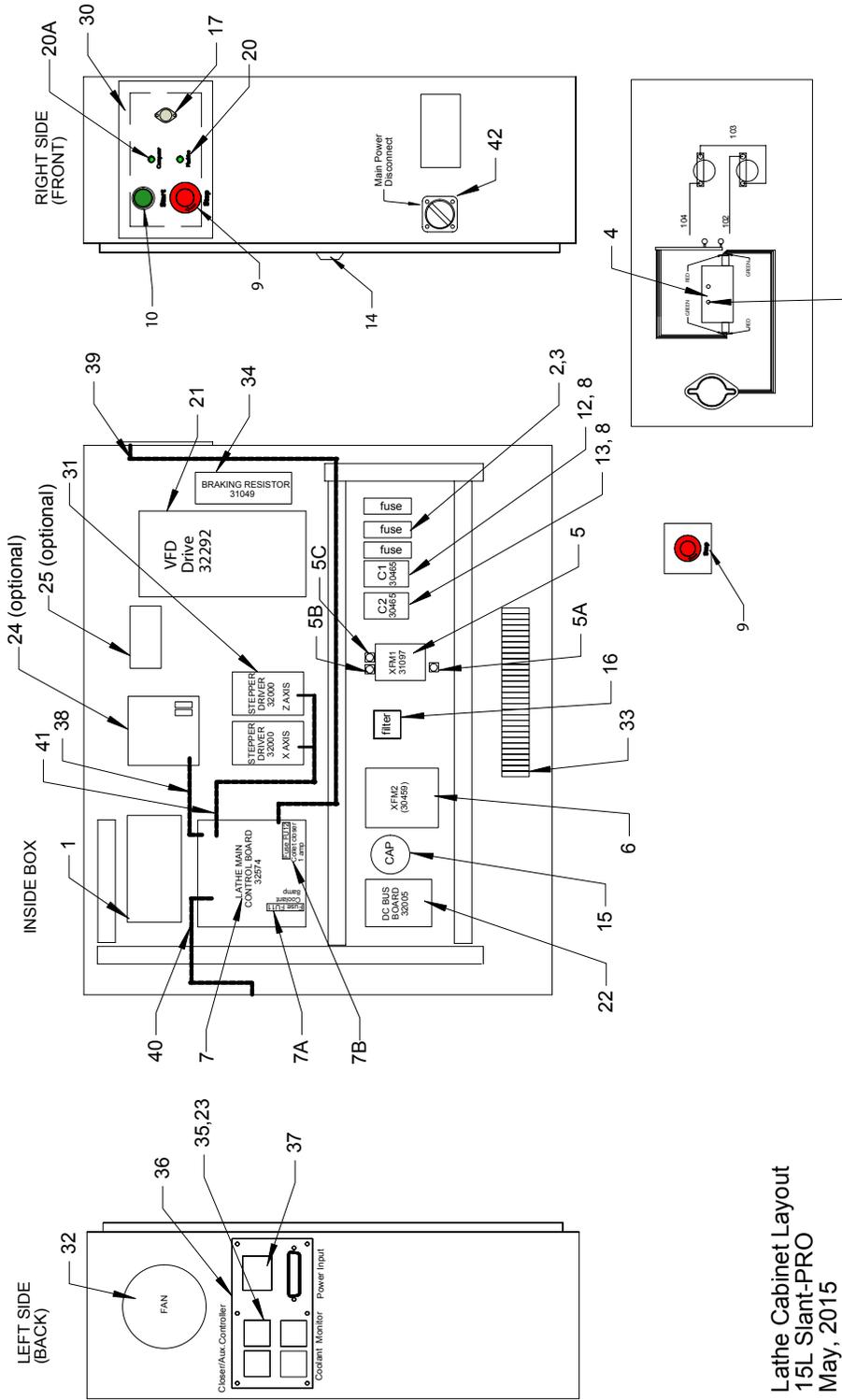


Lathe Cabinet Layout
15L Slant-PRO
May, 2015

## Electrical Cabinet Parts List

| ID | PN | Description | ID | PN | Description | ID | PN | Description |
|----|-----|-------------|----|-----|-------------|----|-----|-------------|
| 1 | 32652 | DC Power Supply 24 Volt | 17 | 31039 | Accessory/Probe Port | 41 | 32858 | J6 Ribbon Cable (axis drivers) |
| 2 | 31657 | Fuse FU1, FU2[1] 20amp | 20 | 31040 | LED Assembly | 42 | 30454 | Disconnect Switch |
| 2A | 30510 | Double Fuse Block | 20A | 31043 | LED Mounting Clip | – | 30627 | Flex Conduit 16 mm OD |
| 3 | 30456 | Fuse FU6[2] 6amp | 20B | 31044 | LED Retaining Ring | – | 30628 | Connector for 16 mm Flex |
| 3A | 30511 | Single Fuse Block | 21 | 32292 | VFD Motor Driver[9] | – | 30722 | Flex Conduit 12 mm OD |
| 4 | 32861 | DIN/LED Circuit Board | 22 | 32005 | DC Bus Board | – | 30723 | Connector for 12 mm Flex |
| 5 | 31097 | Transformer XFM1[3] | 22A | 31655 | DC Bus Fuse F1,3[10] 8amp | – | 33931 | Flex Conduit 18 mm OD |
| 5A | 31119 | Fuse FU3[4] .75amp | 22B | 32404 | DC Bus Fuse F6[11] 15amp | – | 33932 | Connector for 18 mm Flex |
| 5B | 31120 | Fuse FU5[5] 3amp | 23 | 33063 | Noise Suppressor | – | 33371 | Tormach Logo – cast |
| 5C | 31120 | Fuse FU7[5] 3amp | 24 | 32785 | Turret Control Board (optional) | – | | |
| 5D | 31213 | Fuse Holder for FU7 | 25 | 32870 | Turret Capacitor (optional) | – | 34071 | Label, Machine Safety |
| 6 | 30459 | Transformer XFM2[6] | 30 | 33370 | Operator Console Assembly | – | 30224 | Label, Dual Power Supply |
| 7 | 32574 | Lathe Control Board | 31 | 32000 | Stepper Driver | – | 30225 | Label, Retain Voltage Safety |
| 7A | 31655 | Control Board Fuse FU11[7] 8amp | 32 | 30626 | Fan, 115 VAC | – | 30461 | Door Switch (2 per lathe) |
| 7B | 33369 | Control Board Fuse FU12[8] 1amp | 33A | 34127 | Terminal Block | – | 30461 | X and Z Lathe Limit Switch |
| 8 | 33062 | Noise Suppressor | 33B | 34128 | Terminal Block Jumper | – | 32560 | Spindle Encoder |
| 9 | 30462 | E-stop PB0, PB1 | 34 | 31049 | Braking Resistor | | | |
| 10 | 30463 | Push Button PB2 | 35 | 30177 | AC Power Outlet | | | |
| 12 | 30466 | Relay Contactor C1 | 36 | 32869 | I/O Mount Plate | | | |
| 13 | 30466 | Relay Contactor C2 | 37 | 30258 | AC Power Inlet | | | |
| 14 | 30467 | Cabinet Latch and Key | 38 | 32878 | J2 Cable (turret – optional) | | | |
| 15 | 30468 | Capacitor | 39 | 32859 | J3 Cable (console) | | | |
| 16 | 32350 | Filter | 40 | 30684 | J4 Cable (controller) | | | |

[1]Metric size is 10 x 38 mm, 20A. Use Bussmann KTK-20, Littlefuse KLK-20, or Ferraz ATM-20
[2]Metric size is 10 x 38 mm, 6A. Use Bussmann KTK-6, Littlefuse KLK-6, or Ferraz ATM-6
[3]Control Transformer is 230/115-100VA
[4]Metric size is 5 x 20 mm .75A. Use Bussmann GMD-750mA, Littlefuse 239.75P
[5]Metric size is 5 x 20 mm, 3A. Use Bussmann GMD-3A, Littlefuse 239003.P, or Ferraz GSC -3A
[6]Axis Power Transformer is 230/48-500VA
[7]Metric Size is 5 x 20 mm, 8A. Use Bussmann GMD-8A, Littlefuse 239008.P, or Ferraz GSC -8A
[8]Metric size is 5 x 20 mm, 1A. Use Bussmann GMD-1A, Littlefuse 239001.P, or Ferraz GSC-1A
[9]Pre-programmed with current version of PathPilot®
[10]Fuse for DC Bus Board, F1, 3. Use Bussmann GMD-8A, Littlefuse 239008.P, or Ferraz GSC-8A
[11]Fuse for DC Bus Board, F6. Use Bussmann GMD-15A, Littlefuse 239015.P, or Ferraz GSC-15A
[12]This 10 mm used for the X limit switch, 12 mm used for the Y limit switch, all axis motors and spindle motor use 16 m

## 9.7 Connections



Lathe Control Connections
15L Slant-PRO
March 2018
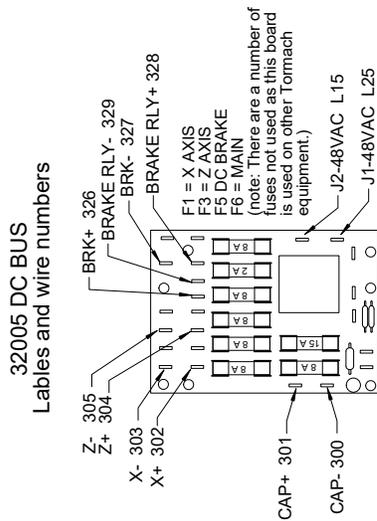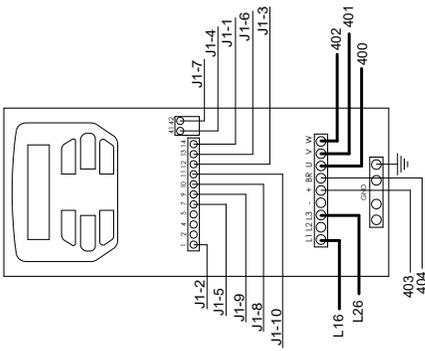
STEPPER MOTORS
X and Z Axis are Three Phase

Three Phase Schematic

| Connector | Function |
|-----------|----------|
| P1 | PWR GND |
| P2 | VDC |
| P3 | U |
| P4 | V |
| P5 | W |

SWITCH SETTINGS FOR DRIVERS

Z AXIS DRIVER

X AXIS DRIVER

VFD WIRING DETAIL
35715

| WIRE COLORS/GAUGE | | |
|---|---|---|
| WIRE NUMBER | COLOR | SIZE (AWG) |
| 100 | WHITE | 14/16 |
| 101-199 | RED | 16 |
| 200 | WHITE | 16 |
| 201 | GREEN/YELLOW | 16 |
| 202-299* | BLACK | 16 |
| 300-399 | MULTI COLOR | 16 |
| 400-499 | BLACK | 14 |
| J1-1 - J1-10 | BLUE | 18 |
| L00-L99 | BROWN/BLUE | 14 |

*See page 1 for detail, 2 wires connecting to the main control board, 202 and 208 are black, 18 ga.

| STEPPER MOTORS TO STEPPER DRIVER | | | |
|---|---|---|---|
| MOTOR LEAD COLOR | WIRE NUMBER | DRIVER TERMINAL | DRIVER NAME |
| RED | 308 | P3 | X DRIVE |
| GRN | 309 | P4 | X DRIVE |
| YEL | 310 | P5 | X DRIVE |
| RED | 311 | P6 | Z DRIVE |
| GRN | 312 | P7 | Z DRIVE |
| YEL | 313 | P8 | Z DRIVE |

32005 DC BUS
Lables and wire numbers

F1 = X AXIS
F3 = Z AXIS
F5 DC BRAKE
F6 = MAIN
(note: There are a number of fuses not used as this board is used on other Tormach equipment.)

BRK+ 326
BRAKE RLY- 329
BRK- 327
BRAKE RLY+ 328

J2-48VAC L15
J1-48VAC L25

Z- 305
Z+ 304
X- 303
X+ 302
CAP+ 301
CAP- 300

PN 32574
TORMACH LATHE
CONTROL BOARD

Console

J1, J2, J3, J4, J5, J6, J7, J9, J10

## 9.8   Operator Panel

33370 Operator Panel assy
(includes buttons, LED's and
DIN connector)

31040 LED Assembly
(31043 LED Mounting Clip
31044 LED Retaining Ring)

31039 Accessory/Probe
Port Assembly

31040 LED Assembly
(31043 LED Mounting Clip
31044 LED Retaining Ring)

Computer

Machine

Start

Stop

30463 Push Button PB2

30462 Estop Button PB1

Lathe User Control Panel Detail
15L Slant-PRO
May, 2015

Stop

Estop Button PB0

103

104

102

RED

GREEN

GREEN

RED

102

101

## 9.9   Ribbon Cable and Miscellaneous
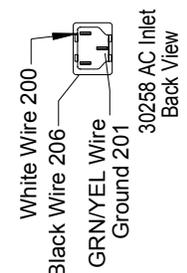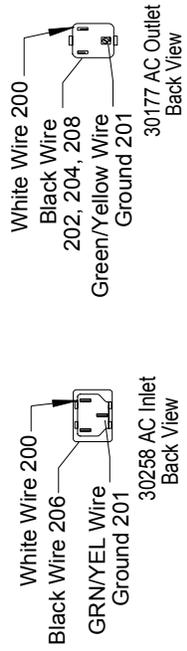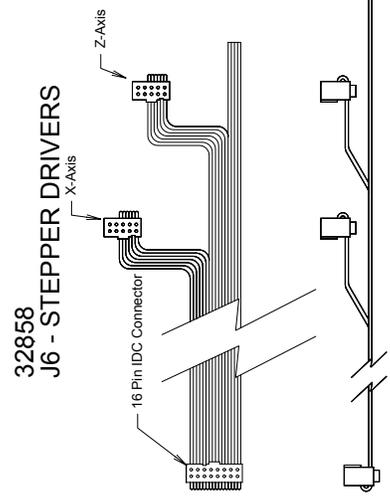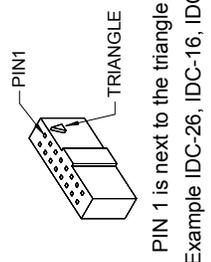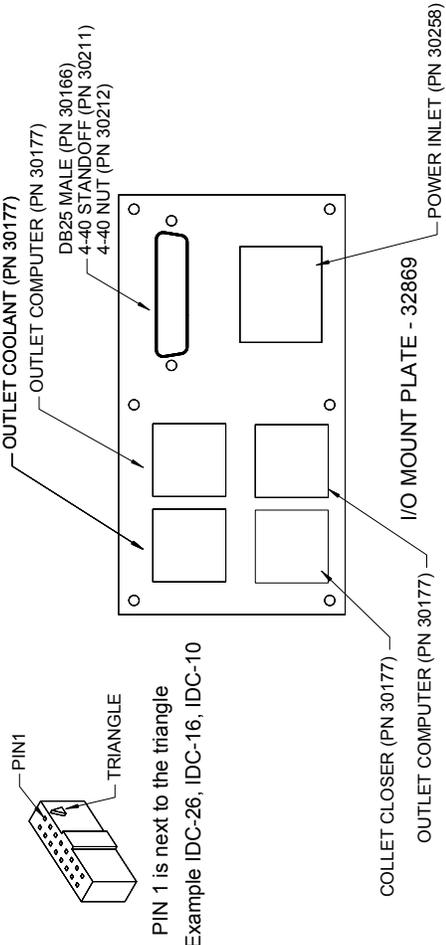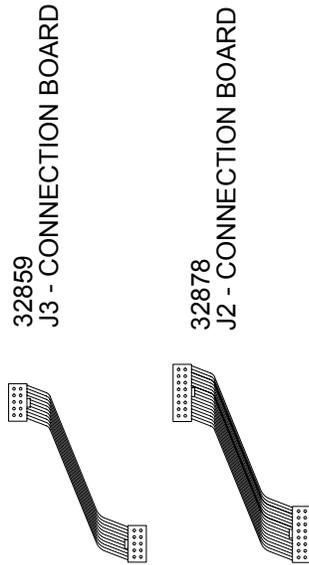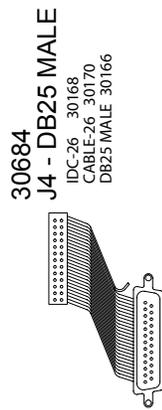
## Tools and Related Items

| PN | Description |
|---|---|
| 34388 | Lathe Controller |
| 30409 | Manual Oiler |
| 31374 | Automatic Oiler |
| 31386 | Machine Way Oil |
| 30485 | Spanner Wrench 25-28 mm DIN 1810 form A standard. Used for setting preload on ballscrew mount bearings – two wrenches are needed. |
| 32470 | Spanner Wrench 78-85 mm DIN 1810 form A standard. Used for setting preload on spindle bearings – two wrenches are needed. |
| 34397 | Tormach 15L Slant-PRO Lathe Operator Manual. Replace old manual with the latest version, spiral bound. Manual PDF can also be downloaded from www.tormach.com |
| 30572 | Touch-up Paint – dark gray 2.5 oz matching the lower section of the stand. Original paint is oil-based; touch-up is latex. |
| 30571 | Touch-up Paint – light gray 2.5 oz matching the PCNC 1100 and the upper section of the stand. Original paint is oil-based; touch-up is latex. Cannot ship paint during colder weather. *NOTE: Valspar Tractor and Implement paint #5339-13 Ford Gray is also a close match.* |
| 30624 | DIN Connector – 5 pin. This is a plug that will fit in the accessory port. |
| 31366 | Coolant Refractometer |

# Diagrams and Parts Lists

## 9.10 Lubrication System



Z ballnut — P3

Z dovetail - upper — P4

Z slideway - upper — P5

Upper manifold (34707)

P2

P1

Pump (31302 or 31374)

Lower manifold (34708)

P12

Z slideway - lower

P6 — X slideway left side

P7 — X ballnut

P8 — X dovetail - left

P9 — X gib

P10 — X slideway right side

P11 — Z gib

**Compression Fittings**

Elbow M8 (35461)

Elbow M6 (31311)

Nut (31310)

Ferrule (31309)

Manifold orifice (31370)

Gib orifice (31408)

Nut and ferrule bundle (31409)

4 mm nylon tubing (31304)

Protective metal sheath (31306)

# ELECTRICAL SCHEMATIC
# 15L Slant-PRO Lathe
## March 2018



Ribbon cable connects J4 to DB25 connector on cabinet. External connection is made to computer

POWER INLET
OUTLET LABEL: COLLET CLOSER
OUTLET LABEL: COOLANT POWER
LABEL: COMPUTER OUTLET
LABEL: MONITOR OUTLET
DB25 MALE

I/O MOUNT PLATE

Suppressor 33063
Suppressor 33063

206
202
208
204
202
202
201
200

FU6 - 6 AMP

L14
L24

OPTIONAL LATHE ATC BOARD

J13 - ATC sensors
J14 - ATC Power

J2 RIBBON CABLE 32878

J9-1 (5V)
J9-2 (A+)
J9-3 (A-)
J9-4 (B+)
J9-5 (B-)
J9-6 (Z+)
J9-7 (Z-)
J9-8 (GND)
J9-9 (GND)

Encoder 32560

J4 RIBBON CABLE 30684

PB0 E-STOP (REMOTE)
PB1 E-STOP

L51
CAP - 15,000 uF 32870
L52

24 DC power supply PN 32652

208
202
204
202

LED2 COMPUTER
LED1 MACHINE
LED ASSY P/N 31040

206
207

PB2 START

101 102 103

FU11- 8 AMP coolant
FU12- 1 AMP closer

32861

L01 (Brown Wire)
L1
L11
FU5 - 2-3 AMP
XFM1
FU3 -0.75 AMP

230 VAC LINE

L02 (Blue Wire)
L2
L21
FU1 - 20 AMP
L11
101

NOTE: FU3, FU5 AND FU7 MOUNTED TO TRANSFORMER

103 104

J7-1
J7-2
J7-3
J7-4

J5-1
J5-2
J5-3
J5-4
J5-5
J5-6

J10-1
J10-2

J1-1
J1-2
J1-3
J1-4
J1-5
J1-6
J1-7
J1-8
J1-9
J1-10

J12
J11 DIN CONNECTOR ASSEMBLY 31039

DISCONNECT
FU2 - 20 AMP
FU7 - 2-3 AMP

230/120 PRIMARY/SECONDARY CONTROL POWER TRANSFORMER

100

C1
Suppressor 33062

C1 30465

FAN

107
106
24VDC V+
V-
C2
107
C2
18 GA BLACK
Suppressor 33062

100

Tormach Lathe Control Board PN 32574

J1

J6

MACHINE FRAME BONDED TO EARTH GROUND

ALL THICK LINES = 14GA WIRE (BROWN/BLUE)

FILTER

L11
L21
L13
L23
C1
L24
C1
L14
L16
L26
C2
C2

J5-5
J5-6
LS3 LS4
X LIMIT
Z LIMIT
J5-5A

106
Enclosure Door Switch, Chuck Guard
104 LS1 LS2

J104
J10-2

326
327
302
303

J6 RIBBON CABLE

230/48 PRIMARY/SECONDARY DC BUS POWER

L25
L15
XMF2
CAP - 15,000u
301
300

DC BUS BOARD 32005

48VAC
F1, F3 - 8 AMP
F6 - 15 AMP

X+
X-
Y+
Y-
Z+
Z-
+CAP
DB+
A+
A-
B+
B-
Brake
Brake
DB  ATC+  ATC- Brake RLY+ Brake RLY-

329 328
C1

Operator installed wiring with Turret Package

VFD 35715

Z DRIVE PN 32000
P1 305
P2 304
P3 311
P4 312
P5 313
Motor 33066

X DRIVE PN 32000
P1 303
P2
P3 308
P4 309
P5 310
Motor 35452

Brake Wires

J6-7 thru J6-12
J6-1 thru J6-6

1 2 4 5 7 9 10 11 12 13 14
J1-1
J1-2
J1-3
J1-4
J1-5
J1-6
J1-7
J1-8
J1-9
J1-10

All control wires 18GA Wire (Blue)

| | |
|---|---|
| 14 to **J1-1** | 13 to **J1-6** |
| 1 to **J1-2** | 42 to **J1-7** |
| 12 to **J1-3** | 10 to **J1-8** |
| 41 to **J1-4** | 9 to **J1-9** |
| 7 to **J1-5** | 11 to **J1-10** |

L1 L2 L3 - + BR U V W

DYNAMIC BRAKE RESISTOR 38354

GND

400
401
402
M1

ALL THICK LINES = 14GA WIRE (BLACK)

403
404

NOTE: RESISTOR COMES PRE WIRED