Tormach® RapidTurn™
Operator Manual

IMPORTANT! Read all safety precautions and instructions thoroughly before attempting RapidTurn installation, operation, or maintenance.

Questions or comments?
Please email us: info@tormach.com

UM10393_RapidTurn_Manual_1217A
Document Part Number: 37143

Preface

SAVE THESE INSTRUCTIONS!

This manual contains important safety warnings and operating instructions for the RapidTurn™ which is an accessory to PCNC 1100 and PCNC 770 mills only. Refer to these instructions before attempting installation, operation, or maintenance. Keep these instructions together with your RapidTurn so they are readily accessible. This manual supplements the safety information and instructions contained in the mill's operator manual. The most recent manual updates for the RapidTurn, PCNC 1100, and PCNC 770 are available at: www.tormach.com/tormach_product_manuals

Read Before Operating

Read and follow all warnings, cautions, and operating instructions before operating the RapidTurn. Failure to do so 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 workpieces must be clamped properly. Failure to do so could result in serious injury or death.*
	CAUTION! *Indicates a hazard which, if not avoided, could result in injury or mill damage.*	***CAUTION! Sharp Objects:*** *Be sure to wear gloves when uncrating RapidTurn. Failure to do so could result in serious injury.*
IMPORTANT!	***IMPORTANT!*** *Addresses important practices not related to personal injury.*	***IMPORTANT!*** *Damage to mill may occur if motor weight is supported by motor wires.*
NOTE:	***NOTE:*** *Provides additional information, clarification, reminders, or helpful hints.*	***NOTE:*** *For further information on automatic oiler troubleshooting, refer to operator manual.*

Safety Overview

Any machine tool is potentially dangerous. The automation inherent in a CNC machine presents added risk not present in a manual machine. Tormach machines can deliver sufficient force to break tools, crush bones, and tear flesh.

This manual provides guidance on safety precautions and techniques, but because the specifics of any one workshop (or other local conditions) can vary greatly, Tormach accepts no responsibility for machine performance or any damage or injury caused by its use. It is your responsibility to ensure you understand the implications of what you are doing and comply with any legislation and codes of practice applicable to your city, state, or nation.

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 machine 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 drugs or alcohol.
- Keep work area well lit and deploy additional lighting, if needed.

Operational Safety:

- Understand CNC machines 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 unbalanced workpieces or spindle fixtures.
- Remove all tools (wrenches, chuck keys, etc.) from spindle and machine table before starting operations; loose items can become dangerous projectiles.
- Use adequate work clamping; loose workpieces can become dangerous projectiles.
- Do not extend unsupported bar stock past the left end of the spindle bore.
- Protect your hands. Stop machine spindle and ensure motion has stopped before:

  - Reaching into any part of the machine motion envelope
  - Changing tools, parts, or adjusting the workpiece
  - Changing belt/pulley position
  - Clearing away chips, oil, or coolant; always use a chip scraper or brush
  - Making an adjustment to part, fixture, coolant nozzle, or when taking measurements
  - Removing protective shields or safeguards; never reach around a guard

- Keep work area clear of clutter as machine motion can occur when keys are accidently pressed or objects fall on keyboard, resulting in unexpected motion.
- Position clamping attachments clear of tool path. Be aware of workpiece cutoffs that could be cut free during operations and become dangerous projectiles.
- Always use proper feeds/speeds, as well as depth/width of cut to prevent tool breakage.

- Check for damaged tools/workpieces and cease operations if detected; replace before restarting operations as these can become dangerous projectiles. Never use longer or larger tools than necessary.
- Chips and dust from certain materials (e.g., magnesium) can be flammable. Fine dust from normally non-flammable materials may be flammable or even explosive.
- Chips, dust, and vapors from certain materials can be toxic. Always check the Materials Safety Data Sheet (MSDS) for each material.

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

- OSHA 1910.212 – General Requirements for All Machines
- OSHA 1910.212 – Milling Machines, Point of Operation Safeguarding
- 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
- Safety Requirements for Construction, Care, and Use of Drilling, Milling and Boring Machines (ANSI B11.8-1983). Available from American National Standards Institute, 1430 Broadway, New York, New York 10018
- Concepts and Techniques of Machine Safeguarding (OSHA Publication Number 3067). Available from The Publication Office – OSHA, U.S. Department of Labor, 200 Constitution Avenue, NW, Washington, DC 20210

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.*

The RapidTurn is supplied with 230 VAC power from the host mill's variable frequency drive (VFD). The wiring and electrical components associated with this circuit is capable of delivering lethal electrical shocks. Care should be exercised when working inside the electrical cabinet.

Support

Tormach provides no-cost technical support to our customers through multiple channels. The quickest way to get the answers you need is normally in this order:

- Refer to operator manual
- Reference related technical documentation, operator manuals, and support videos at: http://www.tormach.com/documents
- Email: info@tormach.com
- Phone: 608-849-8381 x2001 (Monday-Friday 8 a.m. to 5 p.m. CST)
- Fax: 209-885-4534

Scope and Intellectual Property

This document is intended to provide sufficient information to allow you to install, setup, and use your Tormach PCNC mill accessory. It assumes that you have appropriate experience and/or access to training for any computer-aided design/manufacturing software to use with this product.

Tormach Inc. is dedicated to continual improvement of its products, so suggestions for enhancements, corrections, and clarifications are welcome.

The right to make copies of this manual is granted solely for the purpose of training courses related to, evaluation of, and/or use of the machine. It is not permitted, under this right, for third parties to charge for copies beyond the cost of printing.

Every effort has been made to make this manual as complete and as accurate as possible but no warranty or fitness is claimed or implied. All information provided is on an *as is* basis. The authors, publisher, and Tormach Inc. shall not have any liability for, or responsibility to, any person or entity for any reason for any loss or damage arising from the information contained in this manual.

Tormach, PCNC 1100 Personal CNC, PCNC 770 Personal CNC, Tormach Tooling System (TTS), RapidTurn, and PathPilot are trademarks or registered trademarks of Tormach Inc. If other trademarks are used in this manual, but not acknowledged, please notify Tormach Inc. so this can be remedied in subsequent editions. Tormach milling machines and accessories are covered by one or more of the following U.S. Patents: 7,386,362, D606,568, D612,406, D621,859 and Patent(s) Pending.

Intended Use Statement

The RapidTurn is intended for use as an accessory to the PCNC 1100 and the PCNC 770. The intended use includes cutting conventional (non-abrasive) materials such as unhardened mild or alloy steels, aluminum, plastics, wood, and similar materials.

Outside Scope of Intended Use

Applications for the equipment or modifications of the equipment outside of the *Intended Use Statement* are supported through consulting engineering and excluded from Tormach's no-cost technical support.

All of the technical information and insight required to support variations from the intended use cannot possibly be foreseen. If the extensive documentation provided is insufficient, Tormach can provide additional information and engineering support on a consulting-engineering basis. If you have your questions well organized, we can normally provide all the information you need in short order. Consulting engineering is done by electrical and mechanical engineers and billed at current hourly rates.

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.

Performance Expectations and Cutting Ability

The following table summarizes the cutting performance envelope of the RapidTurn.

Spindle Speed Range	180-3500 RPM	
Spindle Power Rating	1 hp	
Maximum Feed Rate	**PCNC 770**	**PCNC 1100**
	X: 110 IPM	X: 90 IPM
	Z: 135 IPM	Z: 110 IPM

The RapidTurn is capable of cutting any material that a lathe can at or near their recommended feeds and speeds. Care should be exercised so that programmed cuts do not exceed the maximum available spindle horsepower.

Each RapidTurn ships with a Certificate of Inspection. This report details quality assurance measurements performed at the factory by a Tormach quality assurance team member on each unit prior to shipping. A sample certificate of inspection and more information on quality assurance measurements is available at: http://www.tormach.com/quality_overview.html

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 mill'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

Warranty

Limited Warranty Coverage

Each Tormach (the "Manufacturer") RapidTurn ("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 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.

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. Extended warranties are also available.

Should you have a problem with your machine, please consult your operator manual first. If this does not resolve the problem, contact Tormach through our website at www.tormach.com or call (608) 849-8381.

Table of Contents

1. Overview

The RapidTurn™ is intended for use on either a PCNC 1100 or PCNC 770 mill. Pictured below is a typical RapidTurn set up.

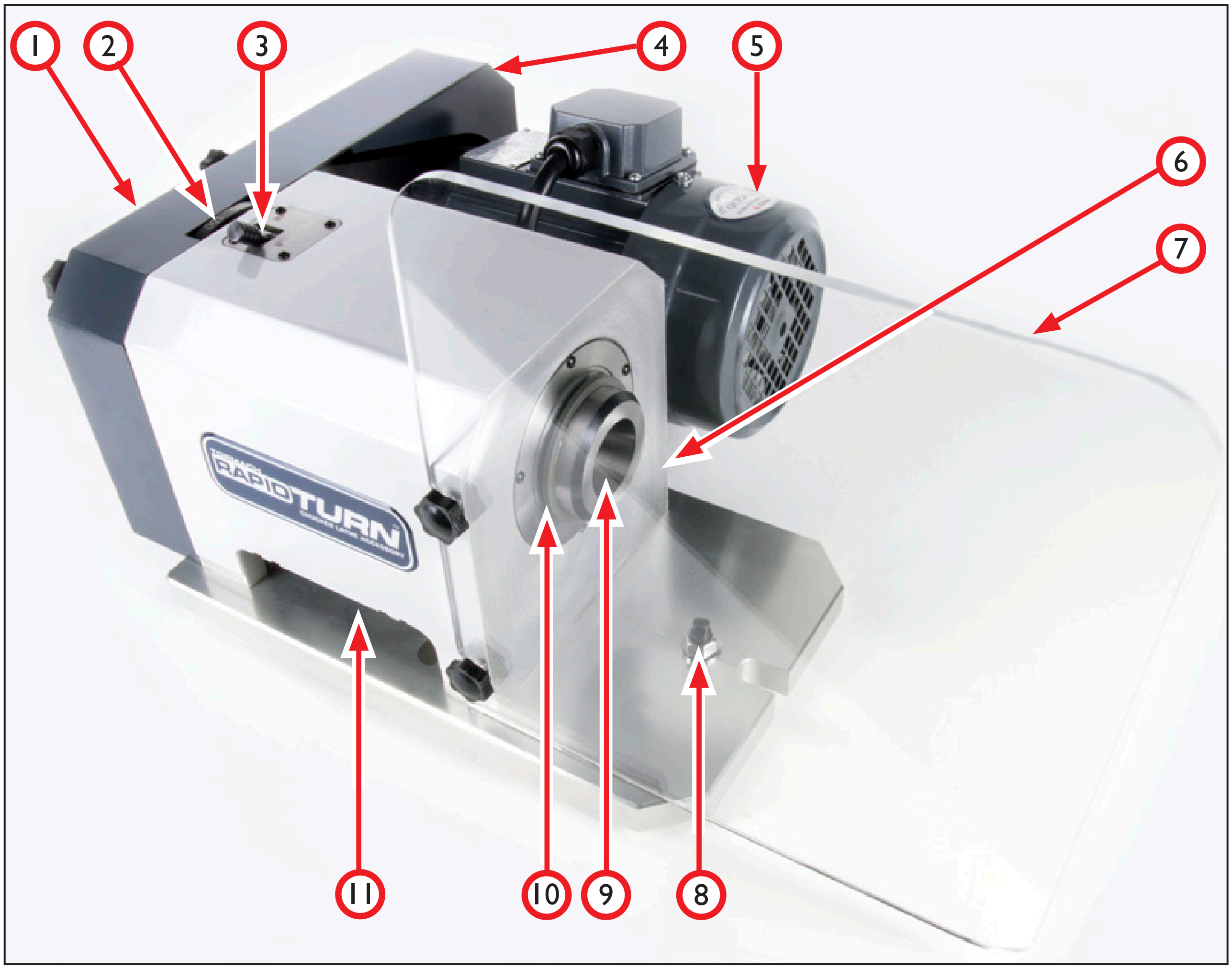


Figure 1.1

Item #	Component
1	Draw Tube (not shown)
2	Index Plate
3	Spindle Lock
4	Belt Guard
5	Spindle Motor
6	Serial Number Plate (not shown)

Item #	Component
7	Chip Guard
8	Eccentric Alignment Pin
9	5C Spindle
10	Spindle Seal
11	Spindle Speed Sensor (not shown)

Overview

1.1 Specifications

Mechanical			
Key Dimensions	Swing-over Base Plate	6" maximum	
	Weight	60 lbs.	
	Footprint	16.25" W x 13.75" L x 9.5" H	
Spindle	Spindle HP	1 hp continuous	
	Spindle Design	Native 5C taper	
	Spindle Speed Range	Belt Driven (two positions)	Low Belt: 180-2000 RPM
			High Belt: 300-3500 RPM
	Tool Holding	Spindle nose clamp with quick change tool post	
Feed Rates	Rapid Speeds	PCNC 1100 (Series 3)	90 X
			110 Z
		PCNC 770 (Series 3)	110 X
			135 Z
Temperature	Operating Range	55° F-100° F (12° C-38° C)	

Electrical	
Connections	Spindle motor connects to mill's VFD using Quick Change Motor Connection Kit
	Spindle index sensor connects to mill’s accessory port

2. Installation

This chapter covers basic installation and alignment of the RapidTurn™ on a PCNC 1100 or PCNC 770 mill, which takes approximately one hour.

2.1 Required Tools

- Tin snips
- 3/4" wrench
- 6 mm, 13 mm, 17 mm, and 19 mm wrench
- 8 mm hex wrench
- Adjustable wrench
- Machinist's stone or similar
- Dead-blow hammer
- Dial indicator (0.0005" graduation) with magnetic base

PN	Description
31947	Basic 3-piece Indicator Set

2.2 Receiving, Partial Uncrating, and Initial Inspection

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

2.2.1 Shipment Arrival

Depending on products and options ordered, the RapidTurn system arrives in one or more shipments:

- RapidTurn (freight)
- Accessory shipment (freight or parcel service, depending on size)

2.2.2 Partial Uncrating

CAUTION! Sharp Objects: *Be sure to wear gloves when uncrating RapidTurn. Failure to do so could result in serious injury.*

1. Using tin snips, remove top of crate from shipping pallet.
2. Locate and retain certificate of inspection/packing list.

2.2.3 Shipment 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.

2.3 Install Quick Change Motor Connection Kit

The RapidTurn's spindle motor is driven by the mill's spindle motor variable frequency drive (VFD). The Quick Change Motor Connection Kit (PN 35178) simplifies switching between the mill and the RapidTurn. Refer to documentation that ships with the kit for instructions on installation and use.

PN	Description
35178	Quick Change Motor Connection Kit

2.4 VFD Programming (PCNC 1100 only)

If using RapidTurn with a PCNC 1100 mill, a VFD Programming Stick Set (PN 37174) is required to correctly program the VFD for operation.

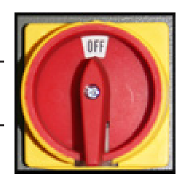
***NOTE:** Using RapidTurn with a PCNC 770 mill does not require reprogramming of the VFD.*

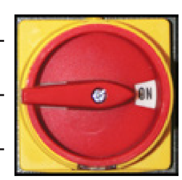
1. Identify VFD Programming Stick Set; two sticks are included:
   - RapidTurn
   - PCNC 1100
2. Power off mill according to *Power Off/On Procedure* detailed below.

***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.*

Power Off/On Procedure

Power Off	1. Push red *E-stop* button 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





IMPORTANT! *Ensure mill is off before inserting RapidTurn VFD Programming Stick.*

3. Insert RapidTurn VFD Programming Stick into VFD with gold contacts on left as shown in **Figure 2.1**.
4. Leave RapidTurn VFD Programming Stick plugged into VFD during RapidTurn operation.

NOTE: *To switch back to mill operation, follow the steps in this section using the included PCNC 1100 VFD Programming Stick.*

Figure 2.1

2.5 Basic Installation Procedure

CAUTION! Collision Hazard: *Any tooling, workpiece, workholding device, fixture, or accessory installed within the machine envelope presents a collision hazard during operation. Do not operate before carefully proving out part programs. Failure to do so could result in machine damage.*

2.5.1 Setup PathPilot Controller

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.*

1. Power on mill according to *Power Off/On Procedure* detailed earlier in this chapter.

PathPilot Interface

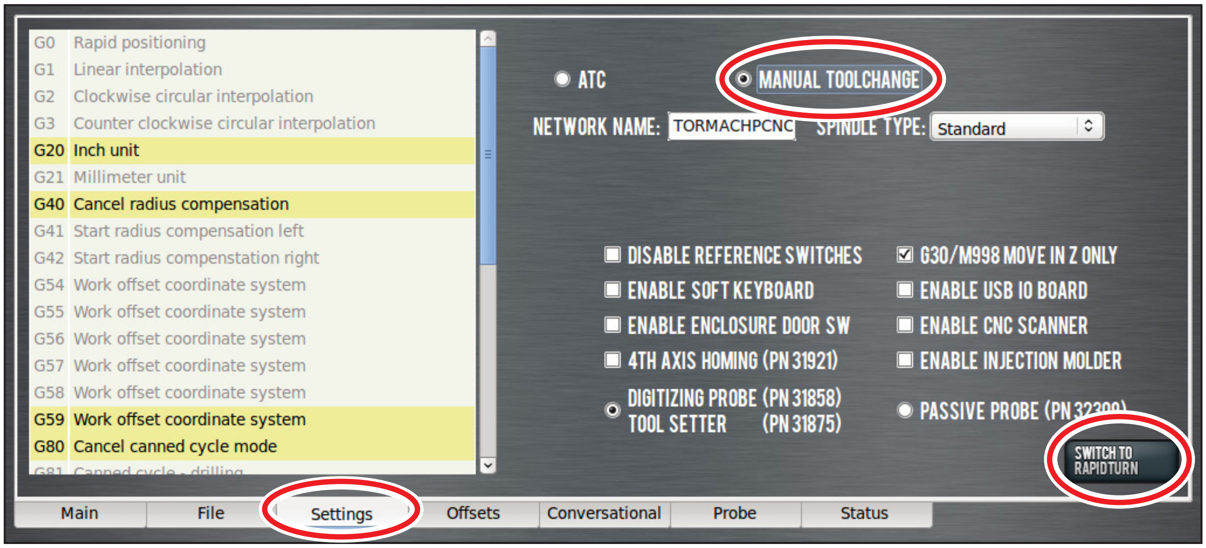


Figure 2.2

2. In the PathPilot® interface, click *Manual Tool Change* on the *Settings* tab (see **Figure 2.2**).

IMPORTANT! *Automatic Tool Changer (ATC) cannot be used with Universal Spindle Arm installed on spindle nose. Do not operate mill equipped with an ATC before removing Universal Spindle Arm. Failure to do so could result in machine damage.*

3. Click *Switch to RapidTurn* (see **Figure 2.2**). This closes mill's PathPilot interface and opens RapidTurn's PathPilot interface. For more information on switching from mill operation to RapidTurn operation, refer to chapter 3, *Operation*.

NOTE: *The PathPilot machine configuration is listed on the bottom of the screen (see **Figure 2.3**). Refer to this before operating to confirm correct configuration is selected.*

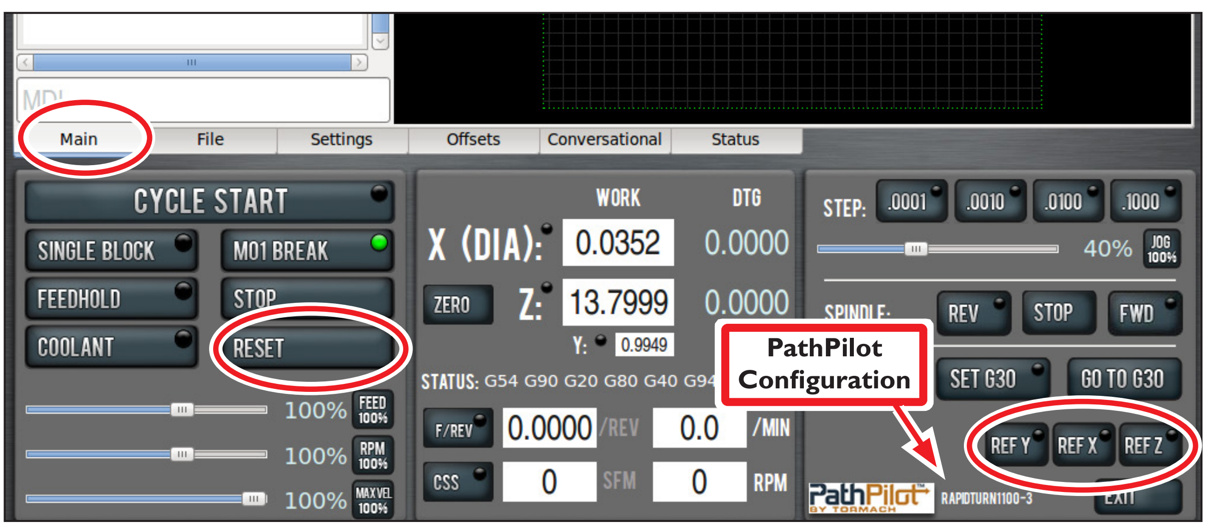


Figure 2.3

WARNING! Motion Direction Hazard: *Lathe axis orientation and motion direction differs from a PCNC mill. Do not operate RapidTurn before becoming familiar with controls. Failure to do so could result in death, serious injury, and/or machine damage.*

4. Ensure machine table is free from obstructions.
5. On *Main* tab, click flashing *Reset* button (see **Figure 2.3**).
6. Reference mill by clicking *REF X*, followed by *REF Y* and *REF Z* (see **Figure 2.3**).

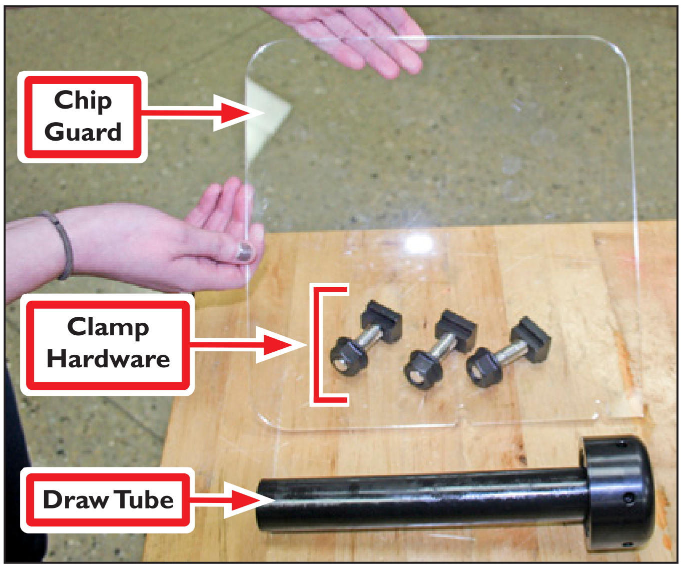


Figure 2.4

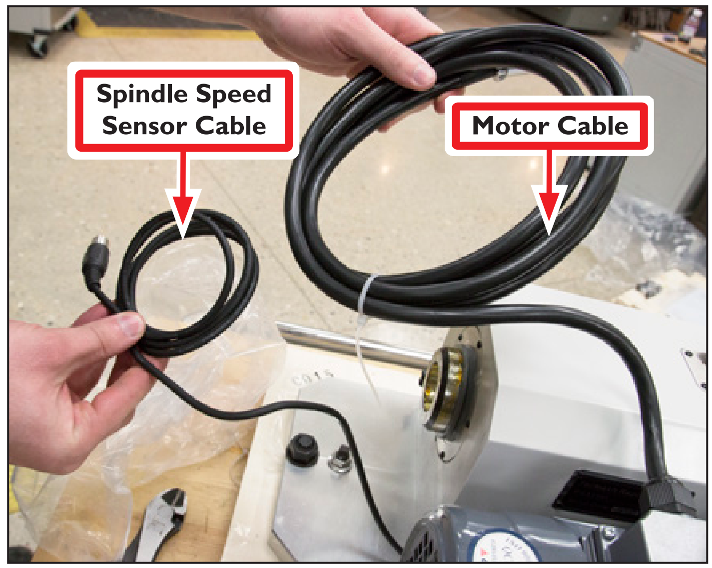


Figure 2.5

2.5.2 Uncrate

This procedure separates the RapidTurn motor from its base plate to facilitate lifting on to the machine table and installation.

1. Remove Chip Guard, Clamp Hardware (three flange nuts, three threaded studs, and three T-nuts), and Draw Tube from pallet (see **Figure 2.4**); set aside.
2. Remove Spindle Speed Sensor Cable and Motor Cable packaging; cut cable ties (see **Figure 2.5**).
3. Remove three thumbscrews securing Belt Guard to RapidTurn (see **Figure 2.6**); remove Belt Guard and set all aside.

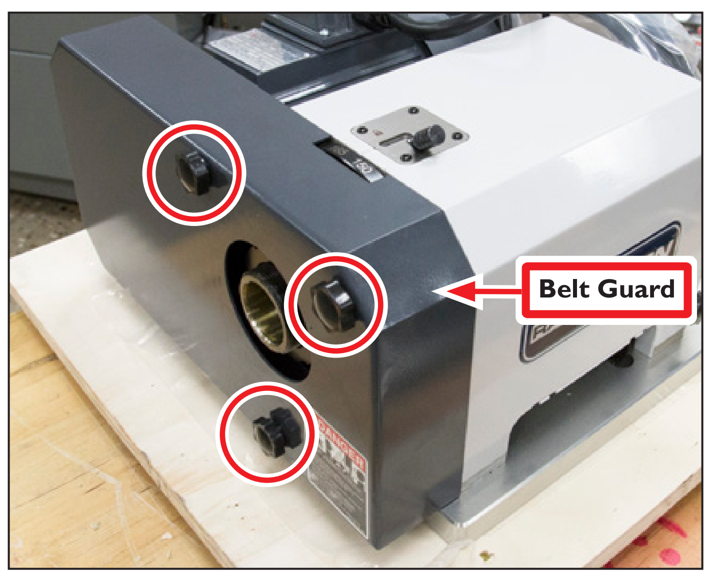


Figure 2.6

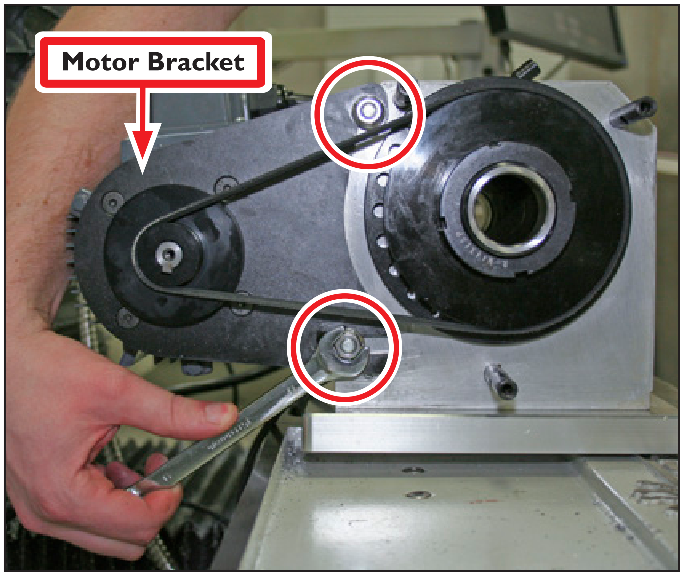


Figure 2.7

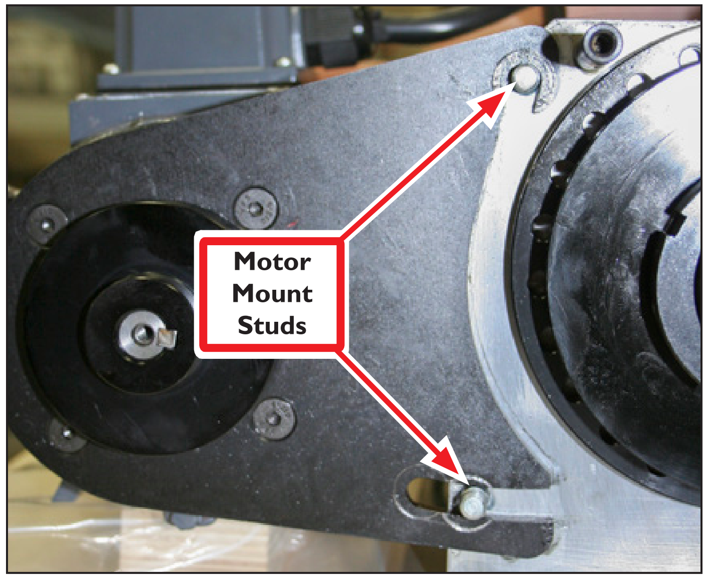


Figure 2.8

4. Using a 13 mm wrench, remove two M8 flange nuts securing Motor Bracket to RapidTurn (see **Figure 2.7**); set aside.
5. Spindle belt slackens when M8 flange nuts are removed; remove belt and set aside.
6. Slide motor sideways off of two Motor Mount Studs (see **Figure 2.8**); set aside.

NOTE: *Motor Bracket hooks around top Motor Mount Stud and slides on to bottom Motor Mount Stud (see* ***Figure 2.8****).*

7. Using an adjustable wrench, loosen and remove three hex head bolts securing RapidTurn Base Plate to shipping pallet (see **Figure 2.9**); discard. Clamp Hardware (see **Figure 2.4**) is used in place of hex head bolts to secure RapidTurn to machine table.

NOTE: *One hex head bolt is on front of Base Plate; two others are underneath RapidTurn headstock (see* ***Figure 2.9****).*

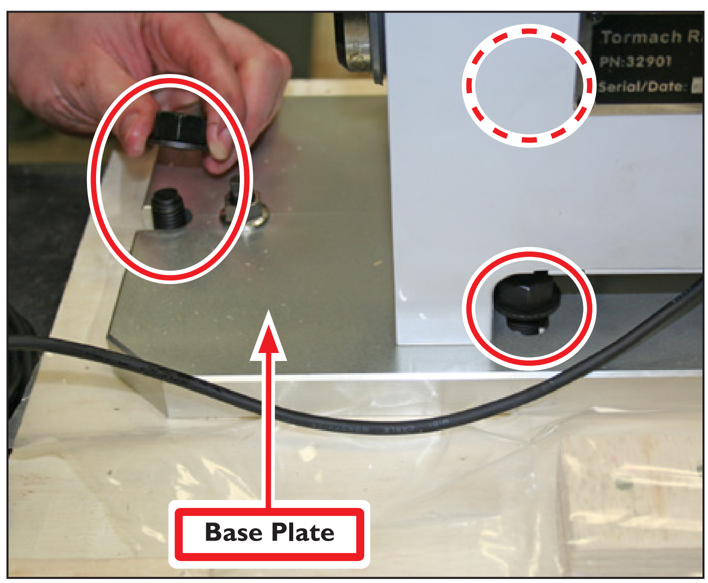


Figure 2.9

8. Lift up RapidTurn base plate to identify Eccentric Alignment Pin and Fixed Alignment Pin (see **Figure 2.10**). The rear (fixed) pin does not move; the forward (eccentric) pin rotates to bring RapidTurn's headstock into alignment with mill's X-axis. For more information, refer to *Align Headstock* section later in this chapter.
9. Tuck Spindle Speed Sensor Cable under headstock prior to RapidTurn placement on mill.

2.5.3 Install Headstock

1. Using machinist's stone or similar, lightly stone surface of machine table to remove burrs.

IMPORTANT! *Presence of burrs may mar or misalign the RapidTurn.*

NOTE: *If intending to leave RapidTurn on machine table for an extended period of time, lightly oil the machine table prior to RapidTurn placement on mill.*

2. Identify Clamp Hardware set aside earlier (see **Figure 2.4**); remove flange nut from each threaded stud and set aside.
3. Jog Z-axis to move machine table as far to right of operator as possible.
4. Slide one threaded stud and T-nut assembly into machine table's center T-slot; position approximately 4" to right of Mill Spindle centerline (see **Figure 2.11**).
5. Slide two threaded stud and T-nut assemblies into T-slots as shown in **Figure 2.11**; position 6" behind front threaded stud and T-nut assembly (see **Figure 2.11**).

Underside of RapidTurn Base Plate

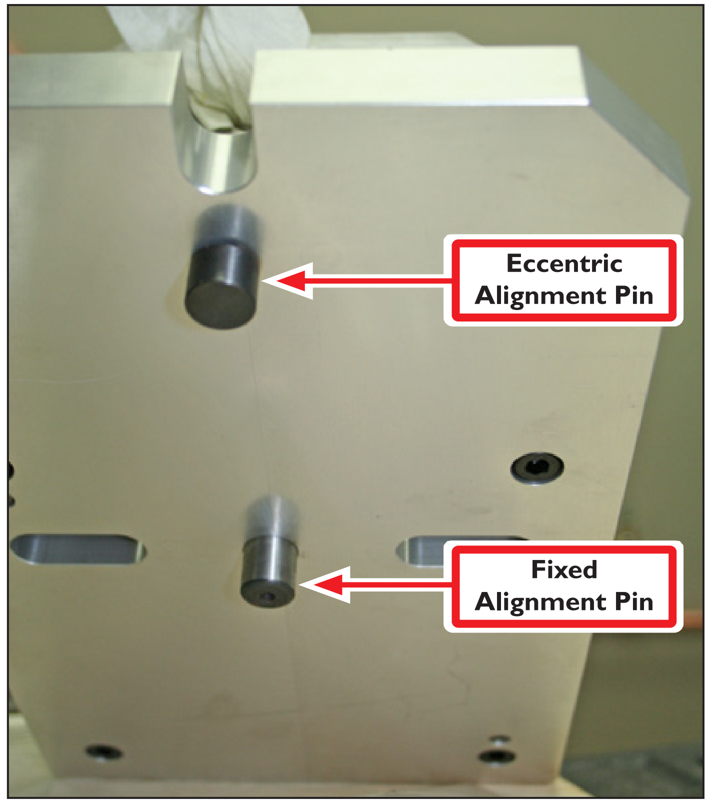


Figure 2.10

Table Moved to Right of Operator

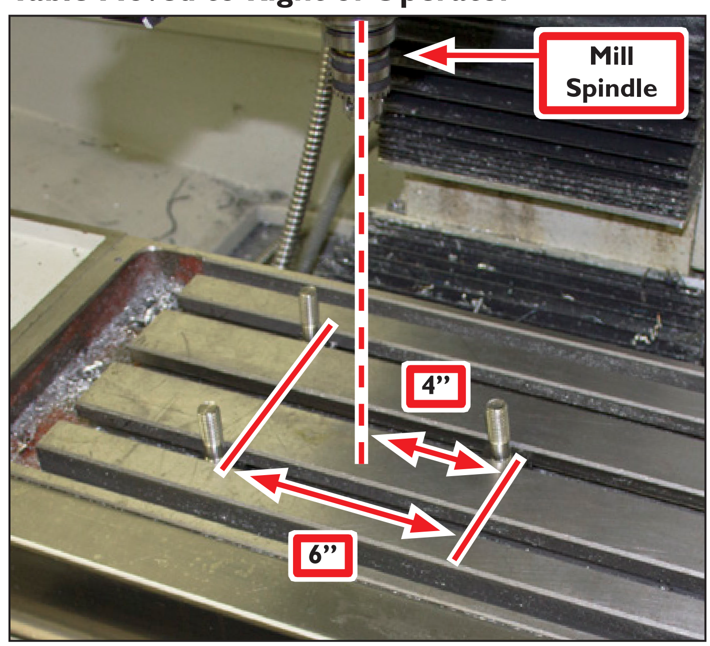


Figure 2.11

IMPORTANT! *Ensure RapidTurn's two alignment pins are positioned to rest in the center T-slot in line with front threaded stud (see **Figure 2.12**). Failure to do so could result in machine damage.*

6. Carefully lift RapidTurn headstock over threaded studs and lower onto machine table (see **Figure 2.12**).
7. Thread three flange nuts set aside earlier onto exposed ends of three threaded studs on machine table until finger-tight.


Figure 2.12

2.5.4 Align Headstock

This procedure aligns the RapidTurn headstock with the X-axis of the mill by adjusting the Eccentric Alignment Pin.

1. Using a 17 mm wrench, loosen the flange nut on Clamp Hardware that secures the Eccentric Alignment Pin to the base plate.
2. Using a 6 mm wrench, adjust Eccentric Alignment Pin until RapidTurn headstock is visually aligned with T-slots on machine table (see **Figure 2.13**).
3. Attach magnetic dial indicator to spindle nose; position indicator tip at edge of Headstock Face furthest away from mill column (see **Figure 2.14**).
4. Slowly jog Y-axis to sweep indicator tip across Headstock Face (see **Figure 2.14**).
5. After reaching opposite side of Headstock Face, make note of reading on dial indicator; divide that figure in half.

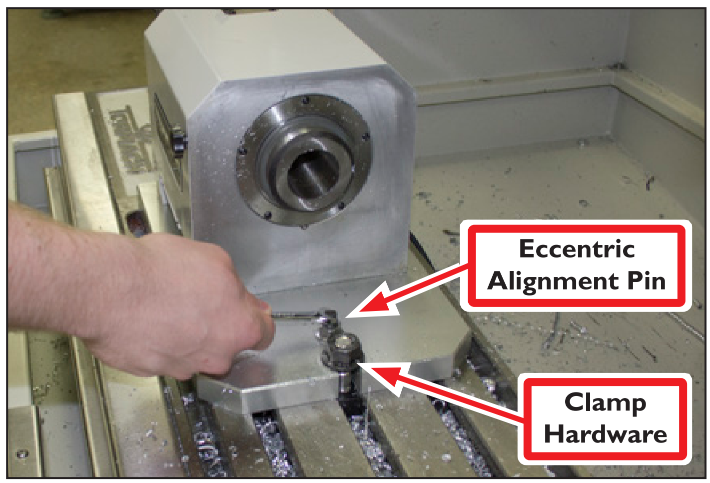


Figure 2.13

Figure 2.14

6. Using a 6 mm wrench, slowly adjust Eccentric Alignment Pin until reading on dial indicator reflects figure determined in step 5. Repeat steps 5-6 until reading on dial indicator does not move.
7. Using a 3/4" wrench, carefully tighten down flange nuts on Clamp Hardware. To ensure headstock has not moved during this process, slowly jog Y-axis to sweep indicator tip across Headstock Face. If necessary, loosen flange nuts slightly and adjust Eccentric Alignment Pin. Repeat this step until reading on dial indicator does not move.
8. Using a 17 mm wrench, carefully tighten down flange nut on Eccentric Alignment Pin. To ensure headstock has not moved during this process, slowly jog Y-axis to sweep indicator tip across Headstock Face. If necessary, loosen flange nut slightly and adjust Eccentric Alignment Pin. Repeat this step until reading on dial indicator does not move.

2.5.5 Reinstall Motor

1. Reattach RapidTurn motor onto two motor mount studs; using an adjustable wrench, attach two M8 flange nuts set aside earlier.

NOTE: *Do not completely tighten flange nuts at this time.*

2. Reinstall spindle belt to bottom pair of pulleys (low speed range); tension spindle belt by lifting motor upward (see **Figure 2.15**) while tightening down two M8 flange nuts (see **Figure 2.15** and **Figure 2.16**).
3. Using three thumbscrews set aside earlier, reinstall Belt Guard to RapidTurn.

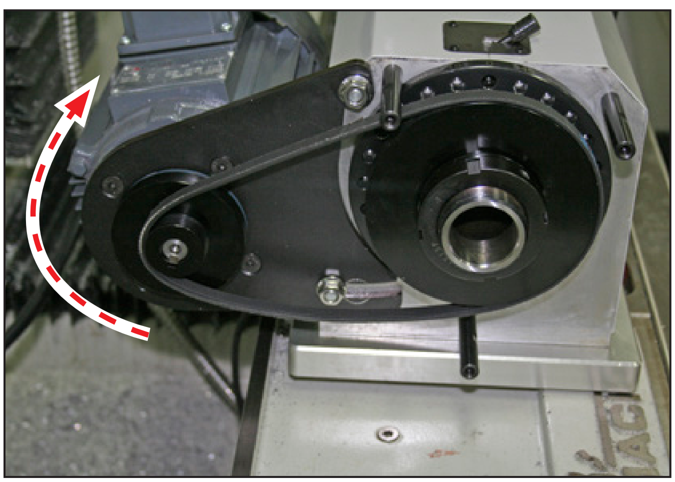


Figure 2.15

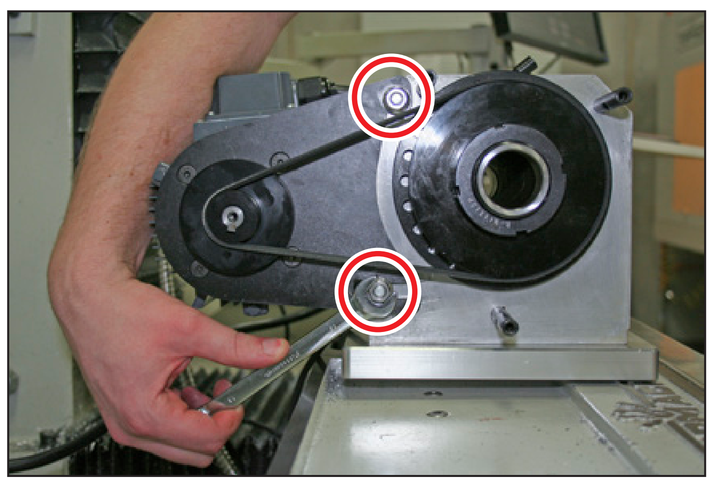


Figure 2.16

2.6 Install and Align Tool Post

Tools for the RapidTurn are secured to the mill's spindle nose using the Universal Spindle Arm Kit (PN 31891), which mounts a 0XA size Reverse Action Quick Change Tool Post (PN 31050). A Tool Post Stabilizer (PN 35690) is used to hold tool post steady. An optional Spindle Alignment Clamp (PN 31892) is available for use with the Universal Spindle Arm to allow quick indexing for repeatable positioning.

PN	Description
31891	Universal Spindle Arm Kit
31050	Reverse Action Quick Change Tool Post
35690	Tool Post Stabilizer
31892	Spindle Alignment Clamp

IMPORTANT! *Ensure ATC is disabled in PathPilot interface before installing tool post. For more information, refer to Basic Installation Procedure section earlier in this chapter.*

In order to correctly orient the tool relative to the workpiece, the tool post must be aligned so that the base of the Universal Spindle Arm is parallel to the RapidTurn's Z-axis and the faces of the tool holders are parallel to the RapidTurn's X-axis (see **Figure 2.17**).

1. Reset and reference mill, ensuring RapidTurn's PathPilot interface is selected (see **Figure 2.3**).
2. After ensuring mill's spindle nose is clean and free of debris, slide Universal Spindle Arm onto mill's spindle nose; position with socket head cap screw to the right of operator and front face visually aligned with machine table T-slots.
3. Using an 8 mm hex wrench, loosely secure one M10 x 30 mm socket head cap screw (included) to Universal Spindle Arm (see **Figure 2.18**); do not tighten down completely.

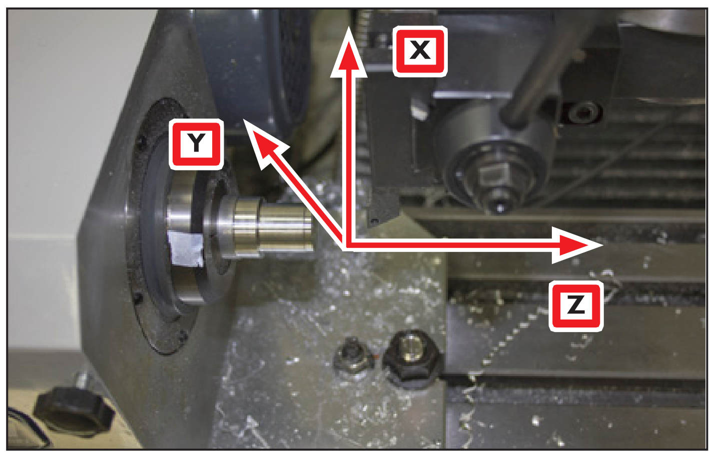


Figure 2.17

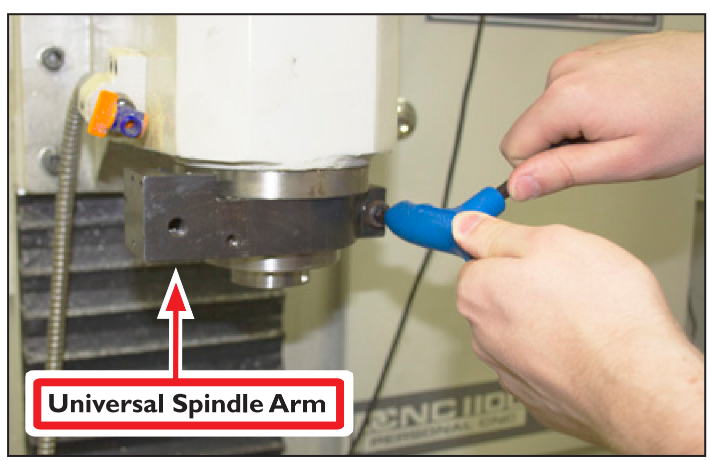


Figure 2.18

4. Attach magnetic dial indicator to machine table; position indicator tip so that it may be swept along front face of Universal Spindle Arm (see **Figure 2.19**).
5. Slowly jog Z-axis to sweep indicator tip back-and-forth across front face of Universal Spindle Arm. Reading on dial indicator should not move.
6. If necessary, use a dead-blow hammer to tap Universal Spindle Arm into alignment (see **Figure 2.20**). Repeat steps 5-6 until reading on dial indicator does not move.
7. Using an 8 mm hex wrench, carefully tighten down Universal Spindle Arm's M10 x 30 mm socket head cap screw.
8. Identify Reverse Action Quick Change Tool Post (PN 31050); screw Tool Post Stud into Universal Spindle Arm and tighten (see **Figure 2.21**).
9. Slide Tool Post over Tool Post Stud and position so that top edge of Tool Post is visually squared up with top edge of Universal Spindle Arm; attach Retaining Nut to Tool Post Stud and tighten until finger-tight (see **Figure 2.21**).

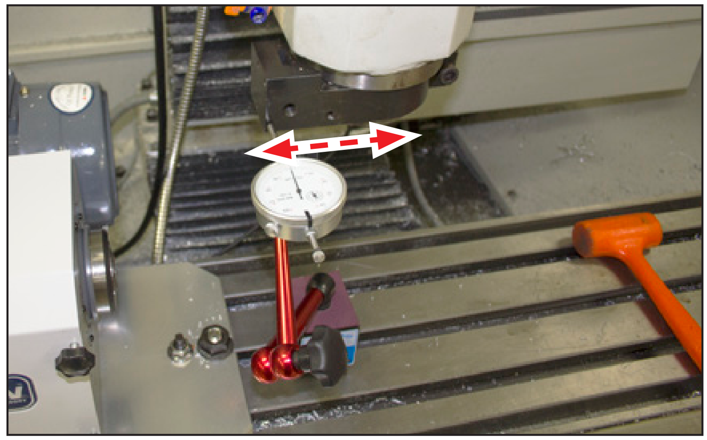


Figure 2.19

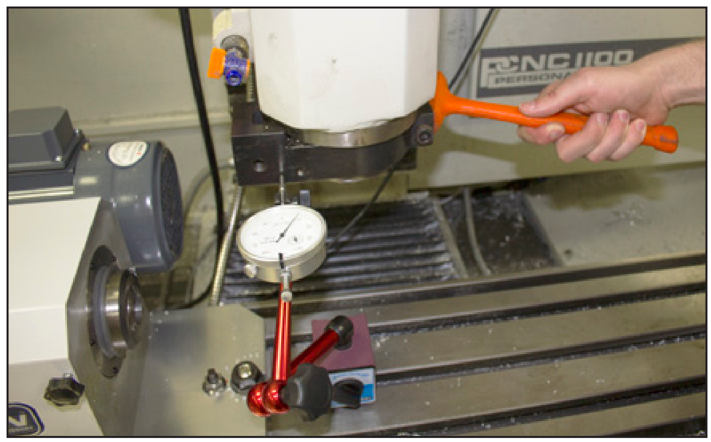


Figure 2.20

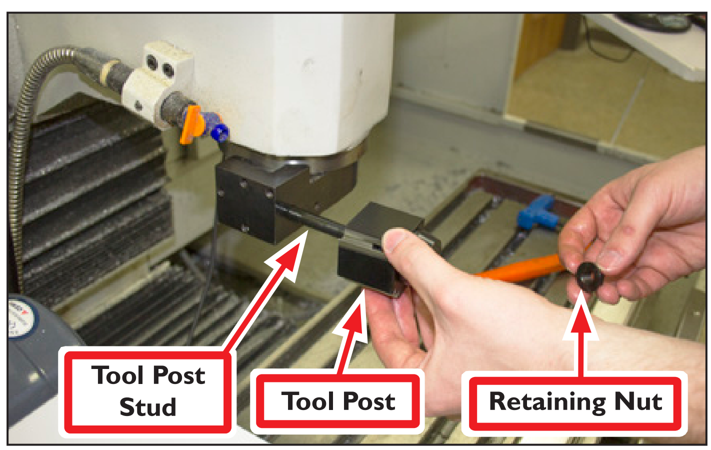


Figure 2.21

10. Screw knob into Tool Post Handle; screw Tool Post Handle into tool post (see **Figure 2.22**) and tighten until finger-tight.
11. Slide Tool Holder on to tool post in direction shown in **Figure 2.22**; tighten Tool Post Handle to secure tool holder in place.
12. Reset and reference mill, ensuring RapidTurn's PathPilot interface is selected (see **Figure 2.3**).
13. Attach magnetic dial indicator to machine table as shown in **Figure 2.23**, positioning indicator tip to be swept along vertical surface of tool holder pocket (see **Figure 2.24**).
14. Slowly jog X-axis to sweep indicator tip back-and-forth across vertical surface of tool holder pocket. Reading on dial indicator should not move.
15. If necessary, use a dead-blow hammer to tap Tool Holder into alignment. Repeat steps 14-15 until reading on dial indicator does not move.
16. Using a 14 mm wrench, carefully tighten down Retaining Nut to Tool Post Stud.

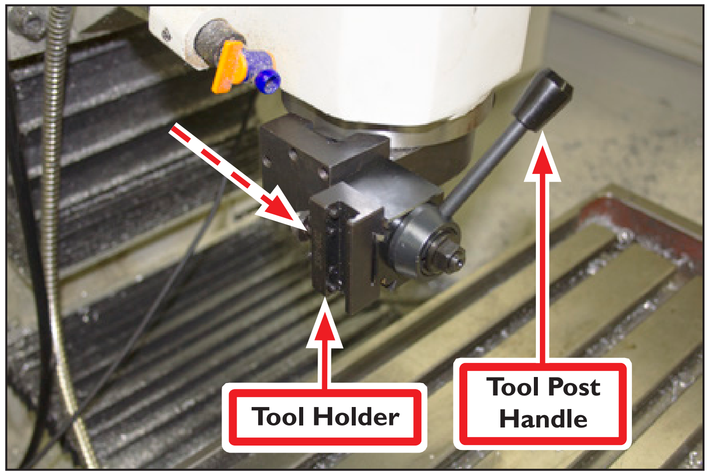


Figure 2.22

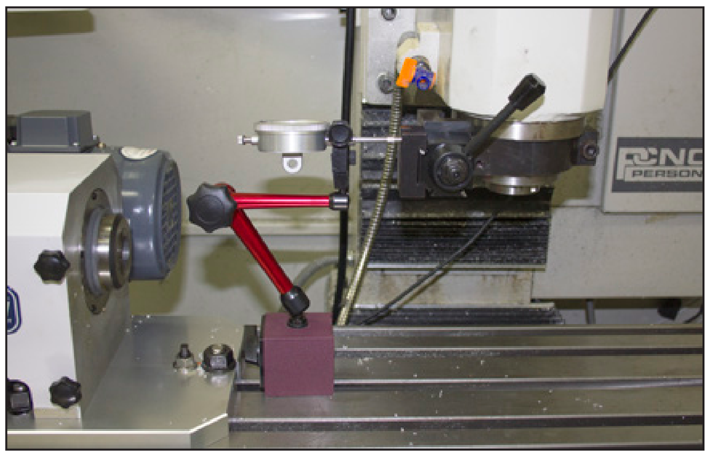


Figure 2.23

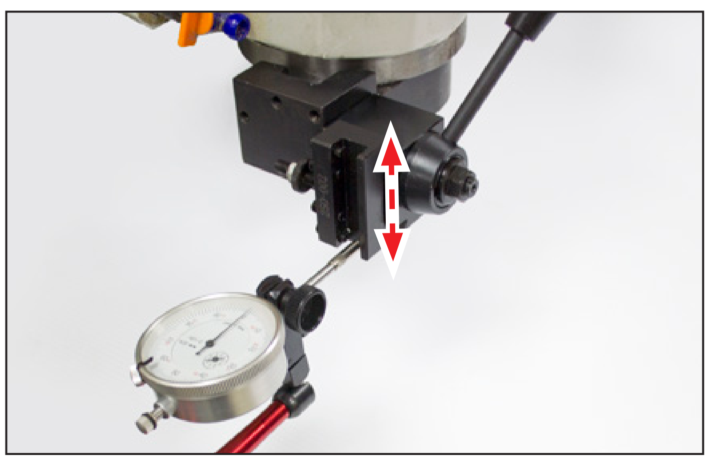


Figure 2.24

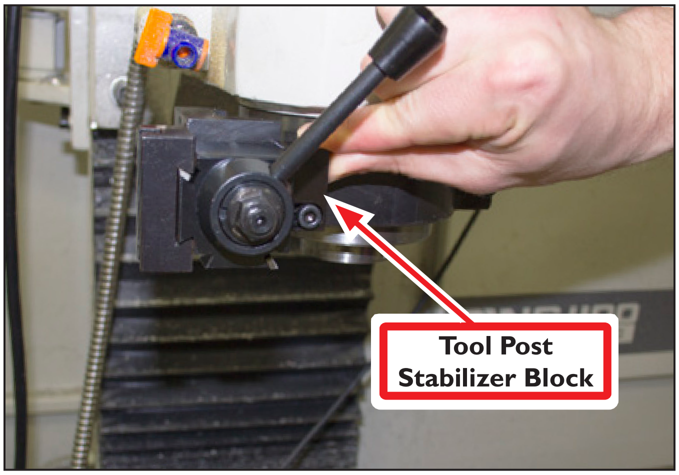


Figure 2.25

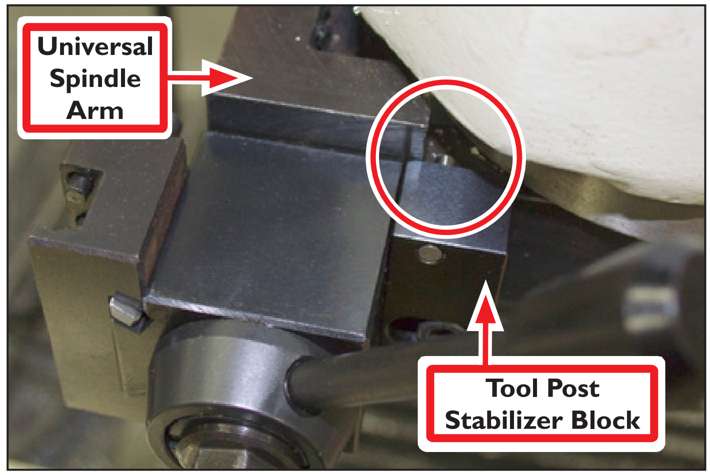


Figure 2.26

17. Identify Tool Post Stabilizer (PN 35690); press Tool Post Stabilizer Block flush against Tool Post on right side of operator (see **Figure 2.25**); secure with one M6 x 22 mm socket head cap screw (included).

IMPORTANT! *Ensure that pin on top of Tool Post Stabilizer Block is firmly seated against upper surface of Universal Spindle Arm (see* ***Figure 2.26****).*

2.7 Cable Connections

1. Power off mill according to the *Power Off/On Procedure* detailed earlier in this chapter.
2. Connect the RapidTurn as follows:
   - Plug Motor Cable (see **Figure 2.27**) into Quick Change Motor Connection Kit's quick disconnect junction box; screw down threaded connector
   - Plug Spindle Speed Sensor Cable (see **Figure 2.27**) into mill's *Accessory* port

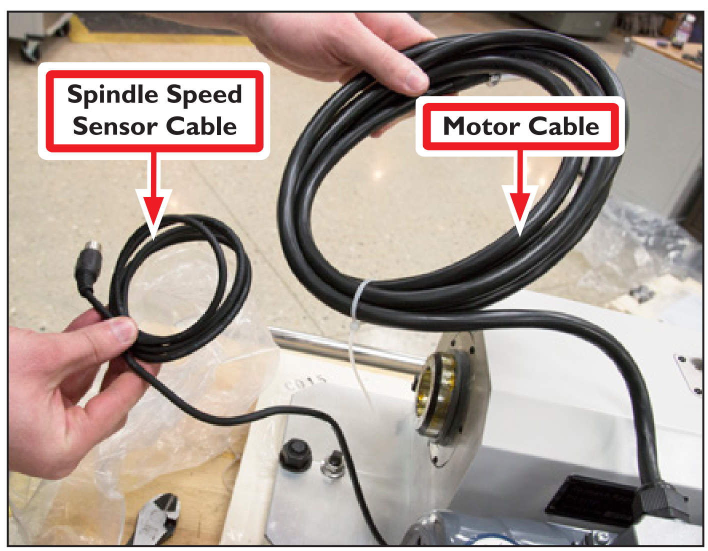


Figure 2.27

2.8 Install Chip Guard

WARNING! Ejection Hazard: *Tooling, fixtures, workpieces, or other loose items could become dangerous projectiles; ensure all components are appropriately secured. Do not operate with chip guard removed. Failure to do so could result in death and/or serious injury.*

Before performing any turning operations, ensure chip guard is securely installed.

1. Loosen two thumbscrews on RapidTurn headstock (see **Figure 2.28**).
2. Orient Chip Guard with two slots in line with two thumbscrews on RapidTurn headstock; slide through groove on base plate as shown in **Figure 2.28**.
3. Tighten down two thumbscrews on RapidTurn headstock.

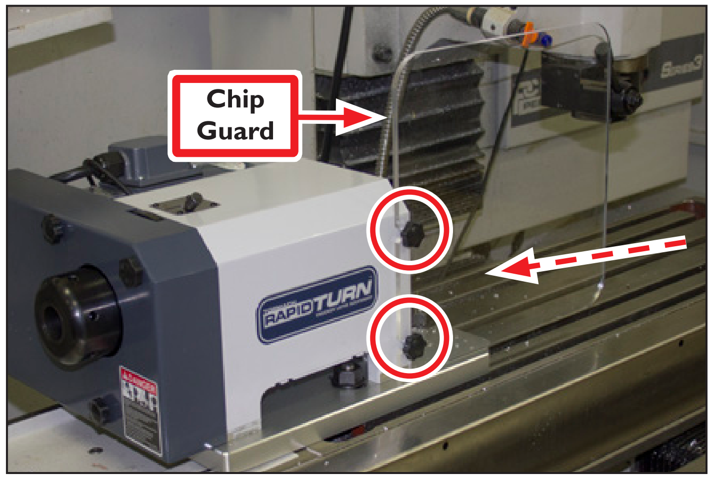


Figure 2.28

3. Operation

This chapter provides an overview of the basic controls of the RapidTurn.

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 RapidTurn Axes

3.1.1 RapidTurn Axes Definition

The RapidTurn™ has two axes of motion used for turning operations, denoted as the X-axis and the Z-axis. The Z-axis is parallel to rotational axis of the spindle along the long axis of the machine table. The X-axis is perpendicular to the Z-axis and nominally parallel to the long dimension of the mill column (see **Figure 3.1**). The -X direction is toward the machine table and +X direction is away from the machine table. The -Z direction is toward the headstock and the +Z direction is opposite the headstock.

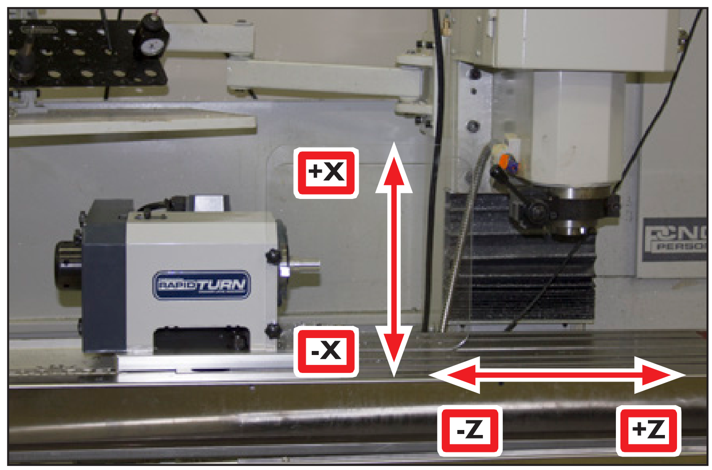


Figure 3.1

3.1.2 RapidTurn Axes Control

WARNING! Motion Direction Hazard: *Lathe axis orientation and motion direction differs from a PCNC mill. Do not operate RapidTurn before becoming familiar with controls. Failure to do so could result in death, serious injury, and/or machine damage.*

The RapidTurn's PathPilot® interface behaves like a full-size CNC lathe. The host mill's X-axis is reassigned as the RapidTurn's Z-axis. Similarly, the host mill's Z-axis is reassigned as the RapidTurn's X-axis:

Axis Reassignment

Mill	RapidTurn
X-axis	Z-axis
Y-axis	Y-axis (positioning only)
Z-axis	X-axis

The reassignment of axes can be disorienting for operators not accustomed to running CNC lathes. Use an abundance of caution and think twice about jog axis selection and anticipated motion direction when positioning the machine using RapidTurn's PathPilot interface.

***CAUTION! Collision Hazard:** Any tooling, workpiece, workholding device, fixture, or accessory installed within the machine envelope presents a collision hazard during operation. Do not operate before carefully proving out part programs. Failure to do so could result in machine damage.*

The RapidTurn can be jogged using the keyboard (see **Figure 3.2**).

Jogging RapidTurn with Keyboard Keys

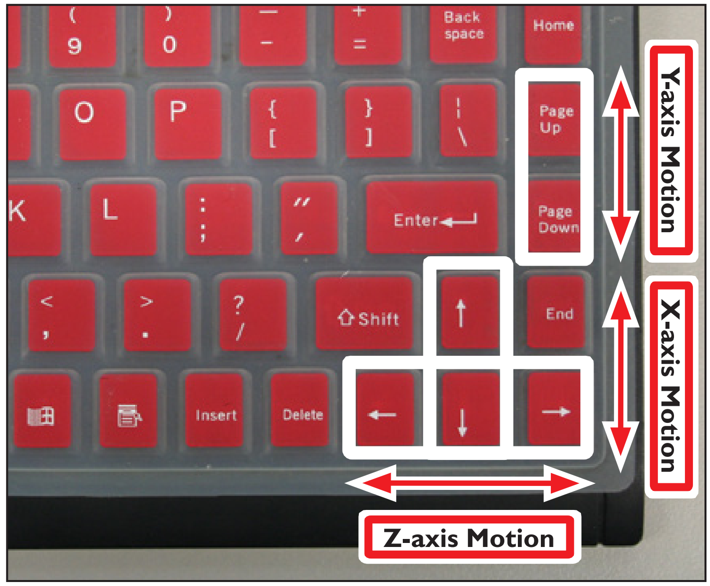


Figure 3.2

***IMPORTANT!** Make note of keyboard function differences between the RapidTurn and the mill, as outlined in the following table. For more information on jogging the mill with the keyboard, refer to chapter 4, Operation, in the mill operator manual.*

Key	Mill	RapidTurn
Left/right arrow	X-axis	Z-axis
Up/down arrow	Y-axis	X-axis
Page up/page down	Z-axis	Y-axis

- Right arrow jogs Z-axis in positive Z direction (headstock moves away from mill head)
- Left arrow jogs Z-axis in negative Z direction (headstock moves toward mill head)
- Up arrow jogs X-axis in positive X direction (mill head moves up)
- Down arrow jogs X-axis in negative X direction (mill head moves down)
- Page up jogs the Y-axis in positive Y direction (table moves toward operator)
- Page down jogs the Y-axis in negative Y direction (table moves away from operator)

***NOTE:** Jogging is not permitted during G-code program execution or MDI moves.*

3.2 Confirm Spindle Direction

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

WARNING! Ejection Hazard: *Tooling, fixtures, workpieces, or other loose items could become dangerous projectiles; ensure all components are appropriately secured. Do not operate with chip guard removed. Failure to do so could result in serious injury or death.*

1. Power on mill following Power Off/On Procedure detailed in chapter 2, *Operation;* ensure Chip Guard is installed (see **Figure 3.3**).
2. To control spindle via PathPilot, flip the operator panel's spindle mode switch to *Auto*.

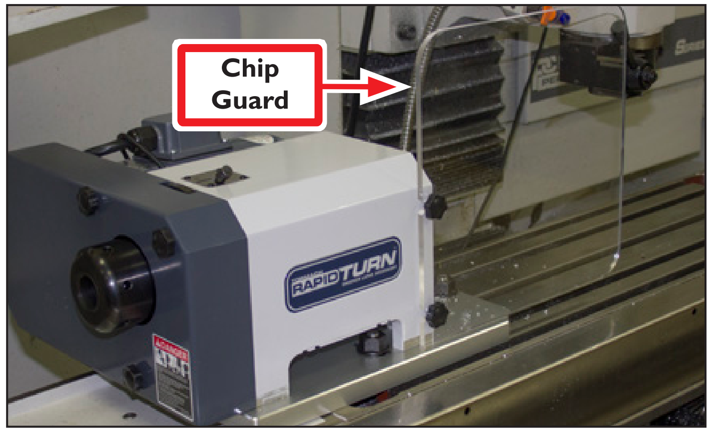


Figure 3.3

WARNING! Overspeed Hazard: *To prevent speeds in excess of workpiece or workholding limitations, ensure spindle speed selected in PathPilot matches actual spindle belt position. Failure to do so could result in serious injury or death.*

3. On *Settings* screen, select correct spindle belt position from *Spindle Pulley Ratio Options*: either *High Speed* or *Low Speed* (see **Figure 3.4**). For more information, refer to *Changing Spindle Speed Range* section later in this chapter.

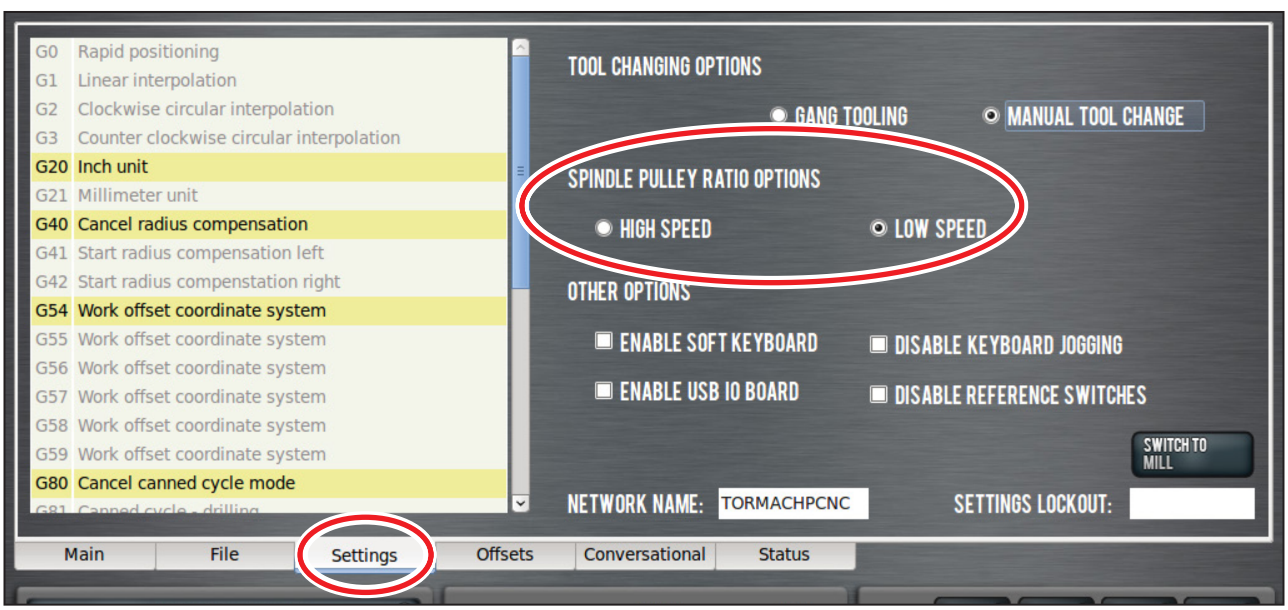


Figure 3.4

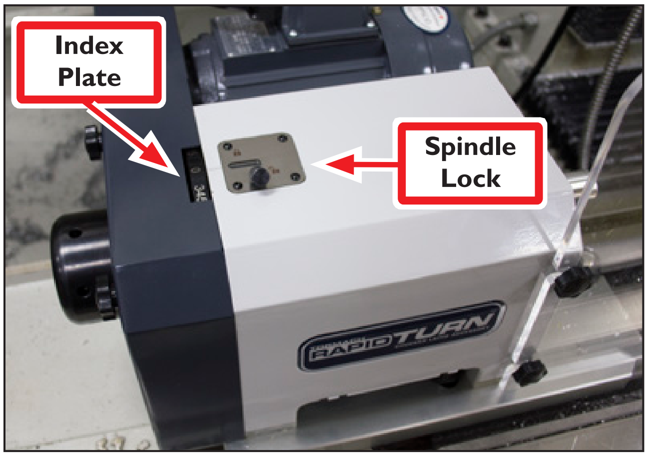


Figure 3.5

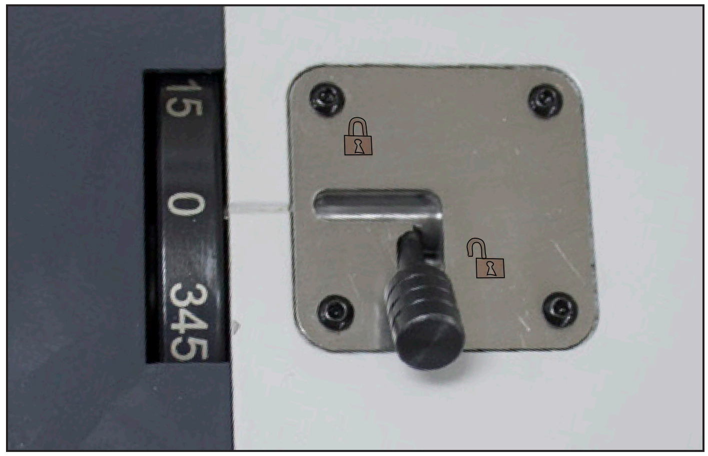


Figure 3.6

4. Ensure Spindle Lock (see **Figure 3.5**) is set to *unlocked* position as shown in **Figure 3.6**. For more information on using the spindle lock, refer to *Workholding* section later in this chapter.
5. On *Main* screen, set chuck speed by entering 180 in *RPM* digital readout (DRO) as shown in **Figure 3.7**.

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

6. Click Spindle *FWD* to start spindle rotation (see **Figure 3.7**).

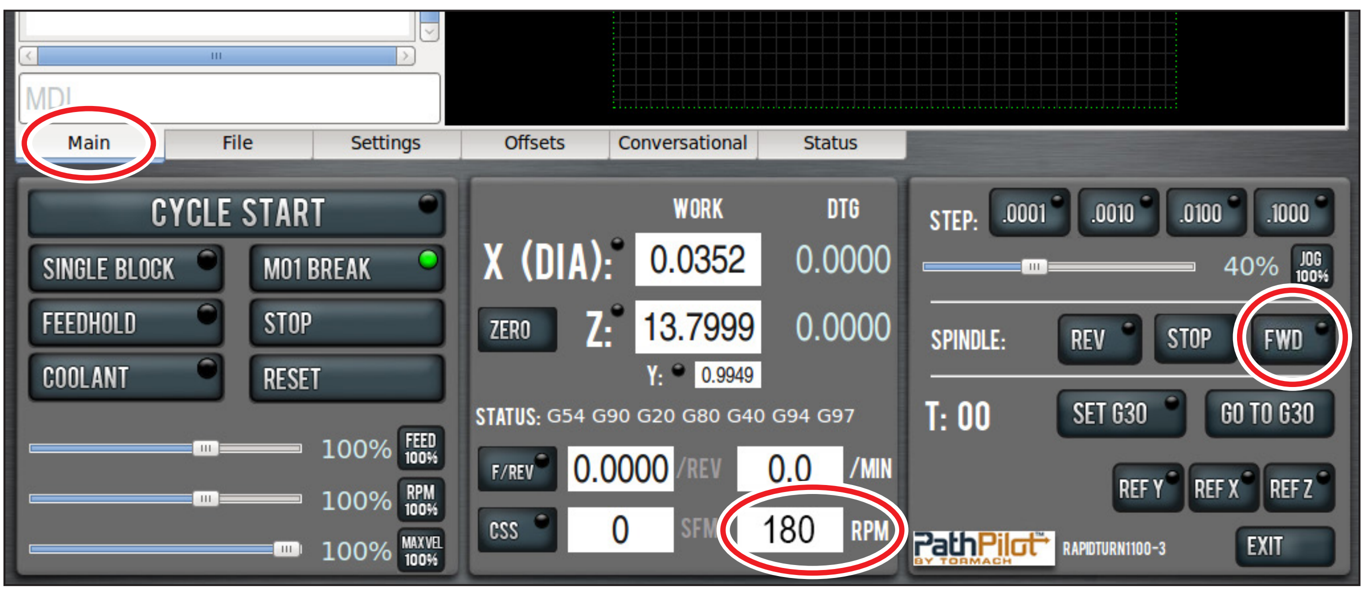


Figure 3.7

Forward (M03) Spindle Direction

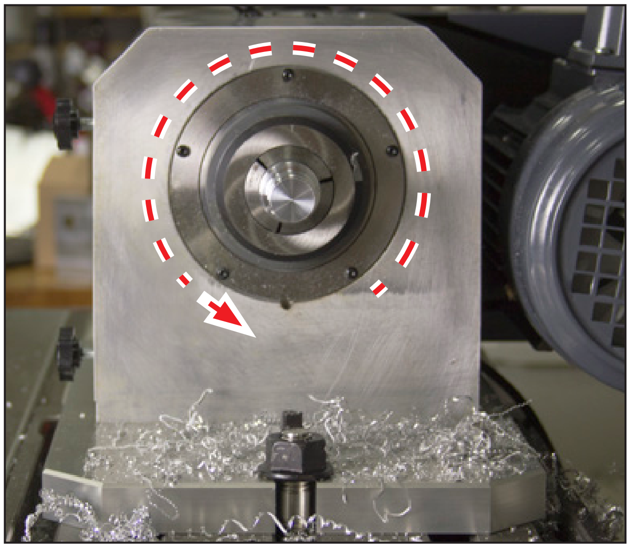


Figure 3.8

Reverse (M04) Spindle Direction

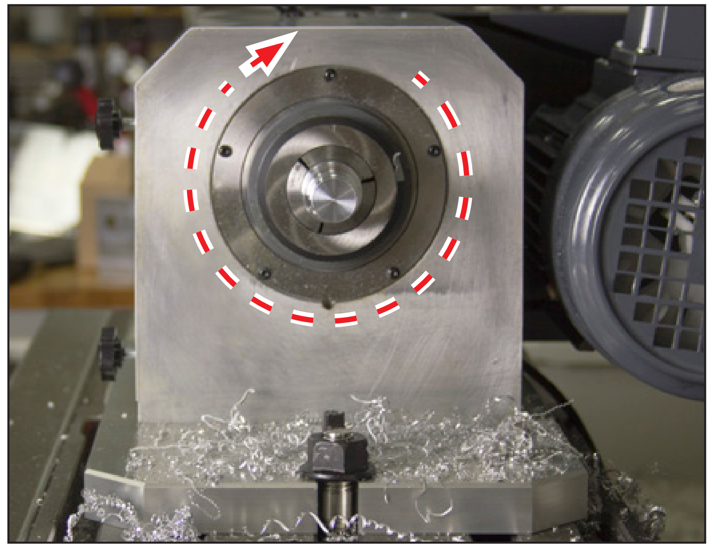


Figure 3.9

7. Observe spindle direction to confirm forward is correct (see **Figure 3.8**); repeat to confirm reverse is correct by clicking Spindle *REV* and observing spindle direction (see **Figure 3.9**). If not rotating as shown, refer to chapter 7, *Troubleshooting*, for more information.

3.3 Changing Spindle Speed Range

Spindle Speed Range

Low	High
180-2000 RPM	300-3500 RPM

The range change is performed by moving the spindle belt from the top pair of pulleys (high speed range) to the lower pair of pulleys (low speed range).

WARNING! Entanglement Hazard: *Automatically controlled devices may start at any time. Be sure to power off machine before making any mechanical modifications. Failure to do so could result in serious injury or death.*

1. Power off mill according to Power Off/On Procedure detailed in chapter 2, *Installation*.
2. Loosen and remove three thumbscrews securing belt guard; remove belt guard and set all aside.

3. Using an adjustable wrench, loosen two M8 flange nuts securing motor bracket to RapidTurn; the belt will slacken and can be moved from one pulley to another (see **Figure 3.10**).
4. Re-tighten two M8 flange nuts on motor bracket; re-install belt guard.

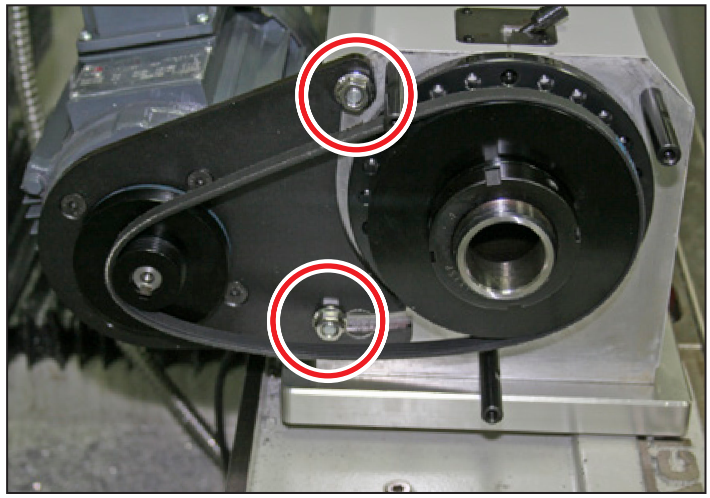


Figure 3.10

3.4 Tooling

3.4.1 Tool Post Locations

Turning tools for the RapidTurn are mounted via a Reverse Action Quick Change Tool Post (PN 31050). In this configuration, most tools will be mounted with the insert facing away from the operator (see **Figure 3.11** and **Figure 3.12**).

For information on mounting tools, refer to chapter 2, *Installation*.

Right-handed Tool

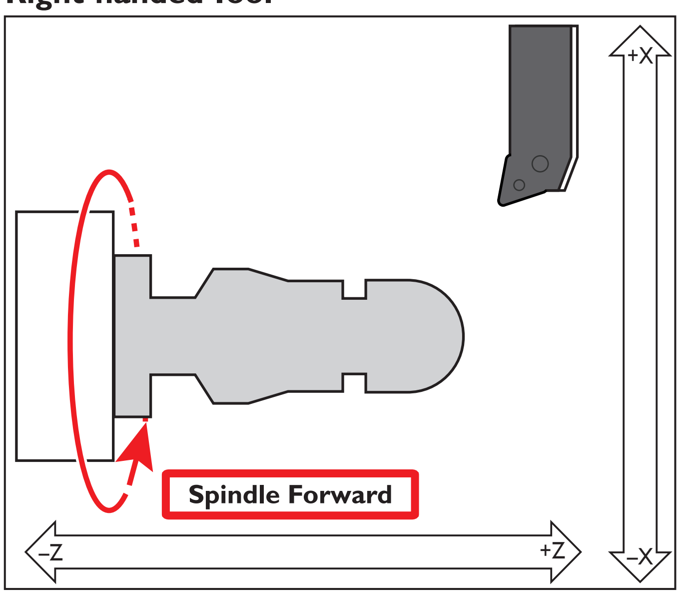


Figure 3.11

Left-handed Tool

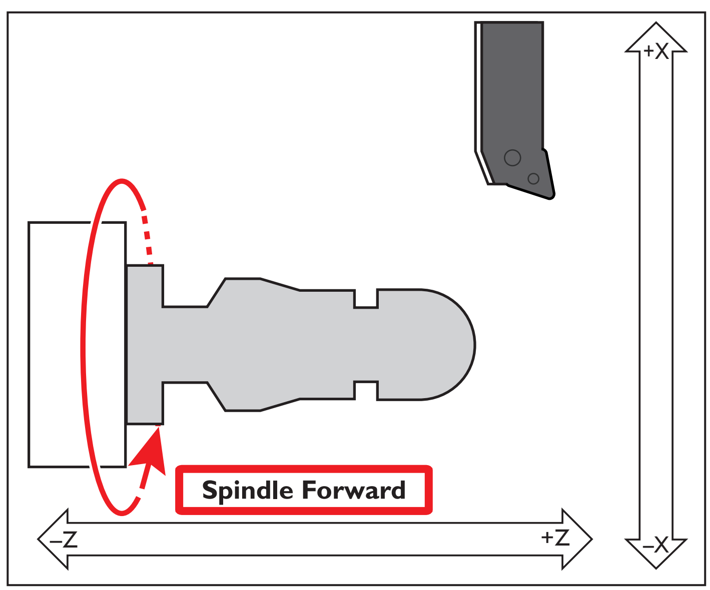


Figure 3.12

3.4.2 Right-handed and Left-handed Tools

Tools have either a right-handed or left-handed orientation as shown in **Figure 3.11** and **Figure 3.12**. 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 3.11** and **Figure 3.12**. Four cutting configurations are available for both a right- and left-handed tool (see **Figure 3.11** and **Figure 3.12**). 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.

3.5 Workholding

WARNING! Extended Workpiece Hazard: *Unsupported stock can bend and whip with deadly force. Do not extend workpiece past end of spindle bore. Failure to do so could result in serious injury or death.*

The RapidTurn's spindle has a native 5C taper and draw tube, which allows for holding a workpiece with either a collet or 5C-compatible chuck. Less rigid workpieces, such as those with a high length to diameter ratio, may require additional support from a tailstock or other support device. For more information, refer to *Tailstock* section later in this chapter.

Install RapidTurn work holder as follows:

1. Ensure contact surfaces of both the spindle's tapered bore and the workholding device's shank are clean and dry.
2. Insert work holder (either collet or chuck) into spindle bore as shown in **Figure 3.13**.

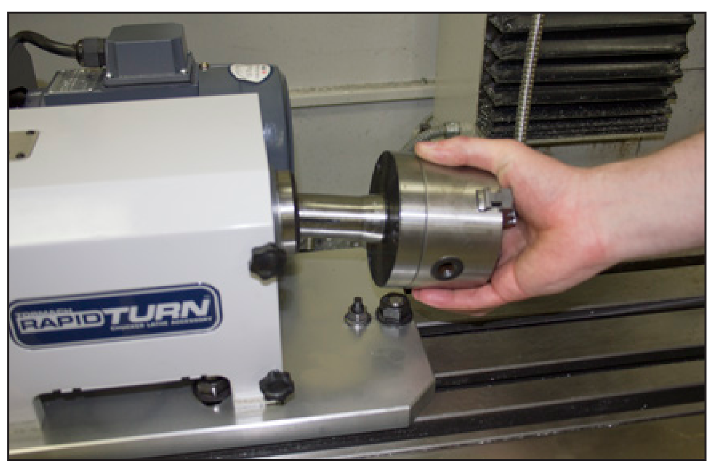


Figure 3.13

3. While supporting work holder, insert draw tube into opposite end of spindle bore (see **Figure 3.14**) and thread onto work holder until tight.
4. Tighten draw tube using Hand Wheel Tightening Pin (PN 37232) to avoid overtightening.

PN	Description
37232	Hand Wheel Tightening Pin

Figure 3.14

IMPORTANT! *Do not overtighten the draw tube. To avoid overtightening, only use the Hand Wheel Tightening Pin (PN 37232) to secure the workholding device to the draw tube. Failure to do so could result in machine damage.*

3.5.1 Collet Configuration (maximum speed rating: 3500 RPM)

For workpieces less than 1.125” in diameter, 5C collets can be used with RapidTurn’s draw tube. Tormach carries a number of 5C collets (round, square, hexagonal, serrated, and machinable).

3.5.2 Chuck Configuration (maximum speed rating: 2000 RPM)

For larger workpieces, Tormach carries a 3-jaw 5C Collet Chuck (PN 32855). This chuck is capable of securing workpieces by clamping on either the inner diameter or the outer diameter of the part. Because larger workpieces require lower spindle speeds and higher torque, the chuck configuration mandates the use of the low speed belt position. The low speed belt position limits the RapidTurn’s maximum speed to 2000 RPM but greatly increases the available torque.

PN	Description
32855	3-jaw 5C Collet Chuck

3.5.3 Tailstock

The RapidTurn is compatible with Medium Tailstock for Rotary Table (PN 30272) and MT-1 Live Center (PN 34104). The tailstock supports less rigid workpieces during the machining process.

PN	Description
30272	Tailstock for Rotary Table (Medium)
34104	MT-1 Live Center

3.5.4 Indexing

The RapidTurn has an Index Plate (with 15 degree increments) and Spindle Lock (see **Figure 3.5**), which allows the operator to perform indexed, 4th axis operations on a PCNC 1100 or PCNC 770 mill without the need to change workholding.

3.6 Regular Maintenance

Scheduled maintenance intervals for the RapidTurn are detailed in the table below.

RapidTurn Maintenance Schedule

Frequency	Completed	Item
Daily		Clean chips off RapidTurn, machine table, and way covers
		Spray exposed, non-painted metal surfaces areas with rust preventative
		Check spindle seal for proper seating and excessive wear
Weekly		Clean chip basket on coolant tank
		Use mild cleaner to clean all exterior surfaces (no solvents)

For information on mill-specific maintenance procedures, refer to chapter 9, *Maintenance*, in the mill operator manual.

4. Intro to PathPilot

4.1 First Part Tutorial

This chapter outlines how to make the first part with RapidTurn™ and assumes the operator has no prior experience running a part program on a CNC (computer numerically controlled) lathe, but has some degree of familiarity with a PathPilot® controlled mill. Even with previous CNC lathe experience, following this tutorial offers an introduction to the controls of the machine. After reading 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.

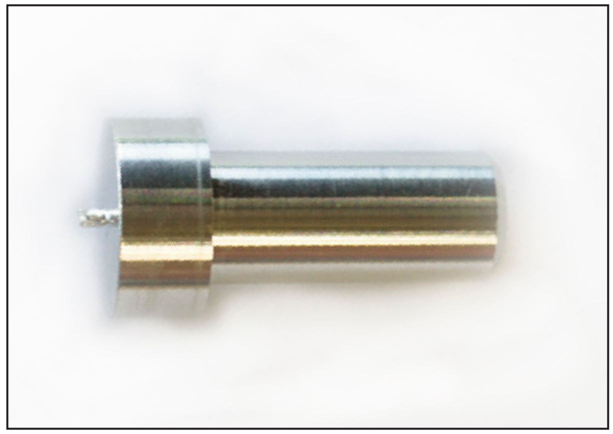


Figure 4.1

The first part program uses a right-handed turning tool and a parting tool to turn a 1” diameter aluminum workpiece down to a 0.500” diameter bushing blank with flange (see **Figure 4.1**). The following instructions will give you a quick start introduction to the lathe controls.

4.1.1 Reference the Machine

Follow the Power Off/On Procedure in chapter 2, *Installation*, to turn the PathPilot controller and machine on. After clicking the flashing *Reset* button, reference the X-, Z-, and Y-axes. The axes may be referenced in any order, but it is customary to reference the X-axis first to move the tooling as far as possible from any workpiece or chuck to avoid a crash. While the machine can be jogged before referencing, parts cannot be run until the machine has been referenced. Referencing establishes soft limits, protects from over travel, and gives meaning to work offset values. After referencing the axes, LEDs on the *REF X, REF Z,* and *REF Y* buttons illuminate. For information on manually referencing the machine, refer to chapter 10, *Troubleshooting*, in the mill operator manual.

4.1.2 Workholding Preparation

Before installing a chuck or 5C collets, ensure the contact surfaces of both the spindle’s tapered bore and the workholding device’s shank are clean and dry.

Install the workholding device by inserting it into the spindle bore and securing with the draw tube (for more information on installation of workholding, refer to chapter 3, *Operation*). Tighten the draw tube using the Hand Wheel Tightening Pin (PN 37232) to avoid overtightening.

IMPORTANT! *Do not overtighten the draw tube. To avoid overtightening, only use the Hand Wheel Tightening Pin (PN 37232) to secure the workholding device to the draw tube. Failure to do so could result in machine damage.*

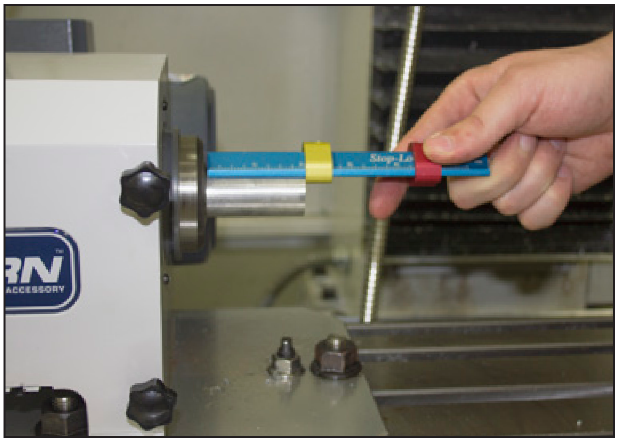


Figure 4.2

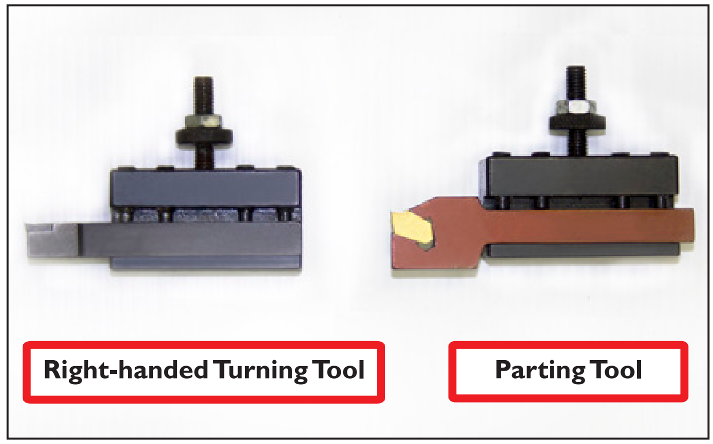


Figure 4.3

4.1.3 Workpiece Preparation

Using a 1” diameter aluminum bar that is approximately 3.5” long, insert the workpiece into the chuck or 5C collet so that 2” protrudes from the jaws of the chuck or face of the collet (see **Figure 4.2**). The first 1.5” will be machined; leaving a bit more sticking out reduces risk of crashing a tool into the chuck or spindle.

4.1.4 Tooling Preparation

Both a right-handed turning tool and a parting tool are required for this first part tutorial (see **Figure 4.3**). Install both tools in quick change tool holders: either turning/facing or turning/boring holders.

PN	Description
35634 or similar	Right-handed Turning Tool
37128 or similar	Parting Tool
30705	0XA Turning and Facing Tool Holder
30706	0XA Turning and Boring Tool Holder

4.1.4.1 Tool Holder Setup

1. Using Right-handed Turning Tool (see **Figure 4.3**), install tool into quick-change tool post.

Front Face of Headstock

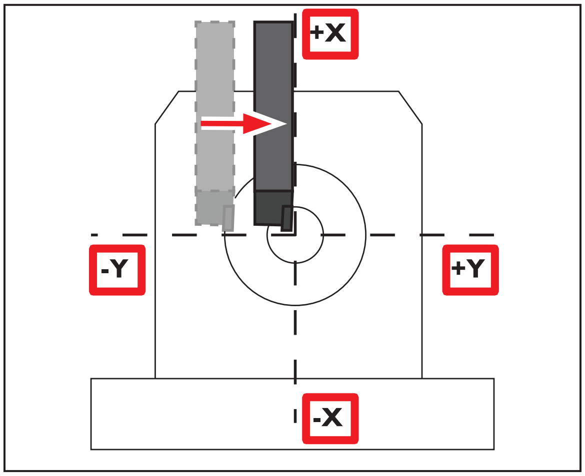


Figure 4.4

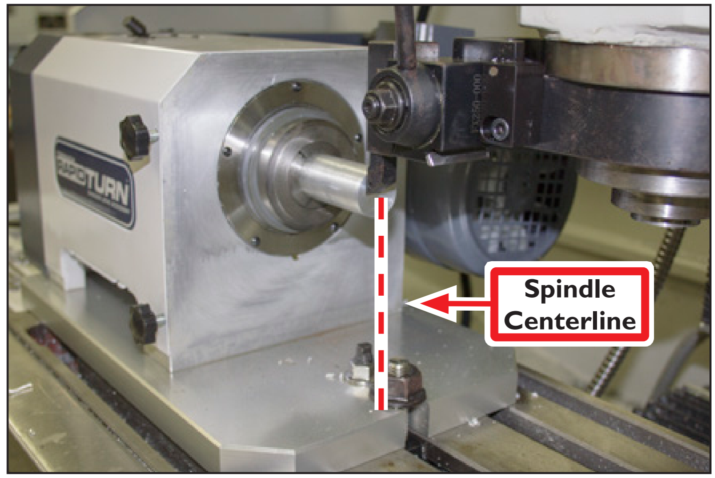


Figure 4.5

2. Jog machine until tool approaches workpiece; visually align tool's cutting tip with Spindle Centerline by jogging Y-axis (see **Figure 4.4** and **Figure 4.5**).
3. Enter 0 in *Y* digital read out (DRO) to zero Y-axis (see **Figure 4.6**). Press *Enter* after typing in the DRO to apply the value.

NOTE: *Press Enter after any DRO changes; clicking on another DRO discards any value just entered. This is intended to avoid accidental changes.*

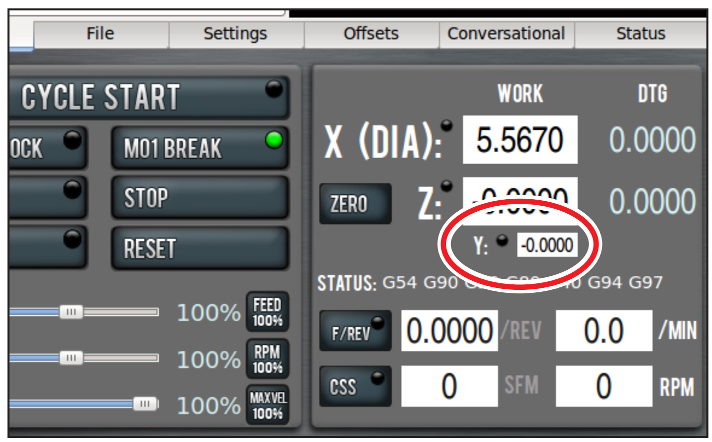


Figure 4.6

The Y-axis is used only for initial positioning of the machine and should not need to be adjusted after this. Use the Knurled Thumbwheel on tool holder to bring the cutting tip of subsequent tools to the same Spindle Centerline (see **Figure 4.7**); adjust until the tool is in alignment, as shown in **Figure 4.4**, and secure using Locking Nut (see **Figure 4.7**).

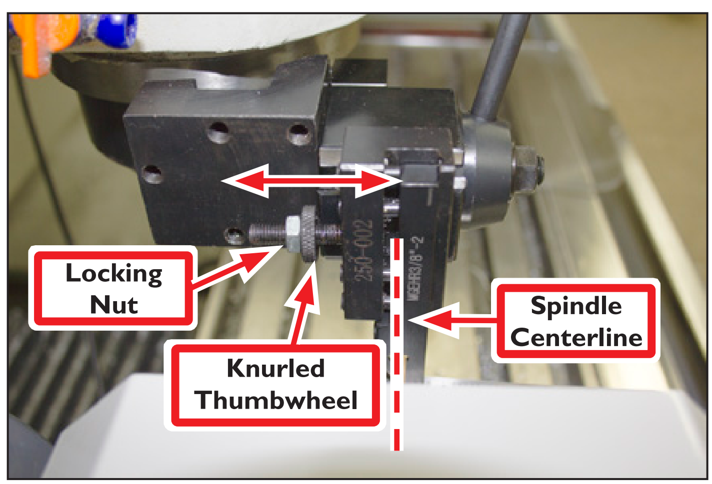


Figure 4.7

4.1.5 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 X/Z positions. 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 CNC lathe like the RapidTurn: Z negative motion moves the tool toward the spindle, and X negative motion moves the tool toward the mill table. For more information on RapidTurn axes definitions, refer to chapter 3, *Operation*.

NOTE: *All tools mounted on the mill's spindle nose are touched off using positive X (diameter) values, and most X words in part programs for these tools will have positive values.*

In many cases, PathPilot generates a warning if this rule is violated, but it has no way of preventing bad code from running.

4.1.6 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.

4.1.6.1 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

4.1.6.2 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

4.1.7 Set Work Offset by Touching Off Workpiece

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 will select a reference surface to use when establishing the work offset zero position. If you select the faces of a quick change tool holder as the reference surface, then the geometry offsets are equal in value to the stick out of the tool (i.e., a true positive tool length).

For the purposes of this tutorial, it is assumed that the quick change tool post is set up (for more information, refer to chapter 2, *Installation*) and the tool holders have been adjusted to centerline height (for more information, refer to *Tool Holder Setup* earlier in this chapter).

Ensure there are no tool geometry offsets currently applied in PathPilot by clicking *Reset Tool* on the *Offsets* screen (see **Figure 4.8**), which clears the geometry offsets for the currently loaded tool. If the *Offsets* screen displays a selection of tools (see **Figure 4.9**), this indicates no tool offsets have been applied.

Set Z Work Offset

1. Insert empty tool holder in quick change tool post.
2. Jog machine until tool holder approaches end of aluminum bar; switch from jog mode to step mode by clicking Step *.0001* button (see **Figure 4.10**).

NOTE: *For more information on jogging controls, refer to chapter 5, PathPilot Interface.*

3. Place a piece of paper between tool holder and end of workpiece to act as a gauge; step machine toward headstock in -Z and feel for paper to bind up (see **Figure 4.11**).
4. Click the *Zero* button next to the *Z* DRO to set the current work offset Z position to zero (see **Figure 4.10**).

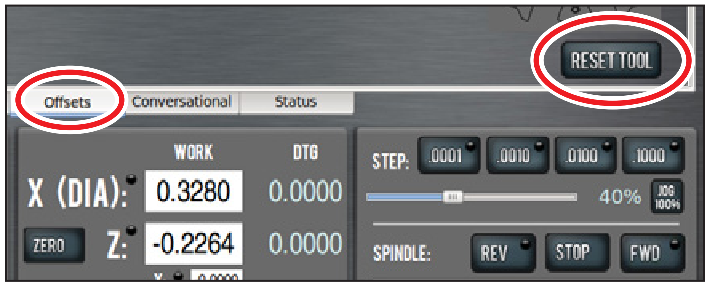


Figure 4.8

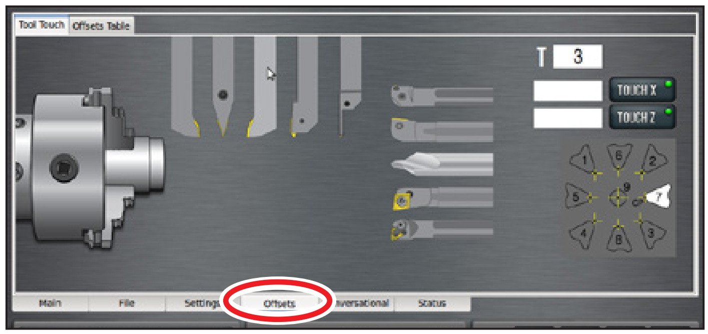


Figure 4.9

Figure 4.10

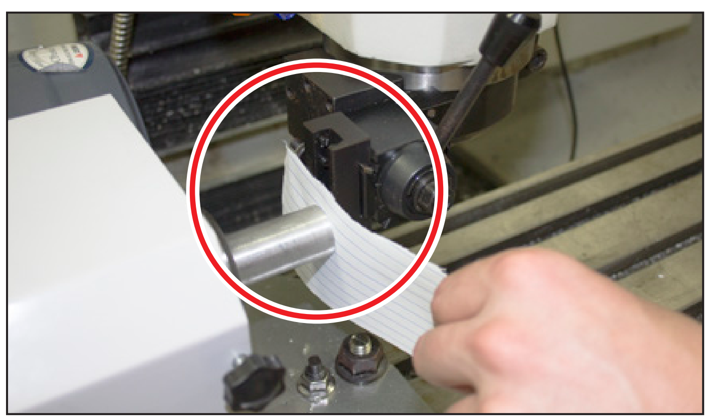


Figure 4.11

Set X Work Offset

1. Jog machine up in X and toward headstock in Z until tool holder approaches top of workpiece; switch from jog mode to step mode by clicking Step *.0001* button (see **Figure 4.10**).
2. Place a piece of paper between tool holder and top of workpiece to act as a gauge; step machine toward headstock in -Z and feel for paper to bind up (see **Figure 4.12**).

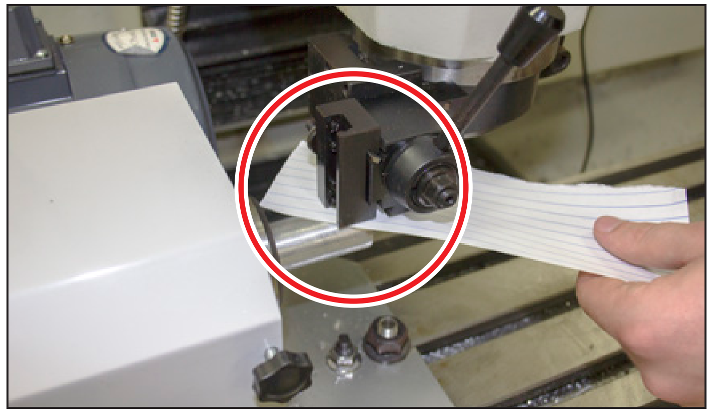


Figure 4.12

3. Enter diameter of the workpiece plus twice the thickness of the paper (typically 1.0 + .006) in the *X (DIA)* DRO to touch off at 1.006" in X (see **Figure 4.10**).

NOTE: *A typical piece of paper is 0.003" thick. This value is doubled to account for the paper feeler gauge because the X (DIA) DRO always displays diameter values, not radius.*

4. Jog machine back a safe distance away from workpiece.

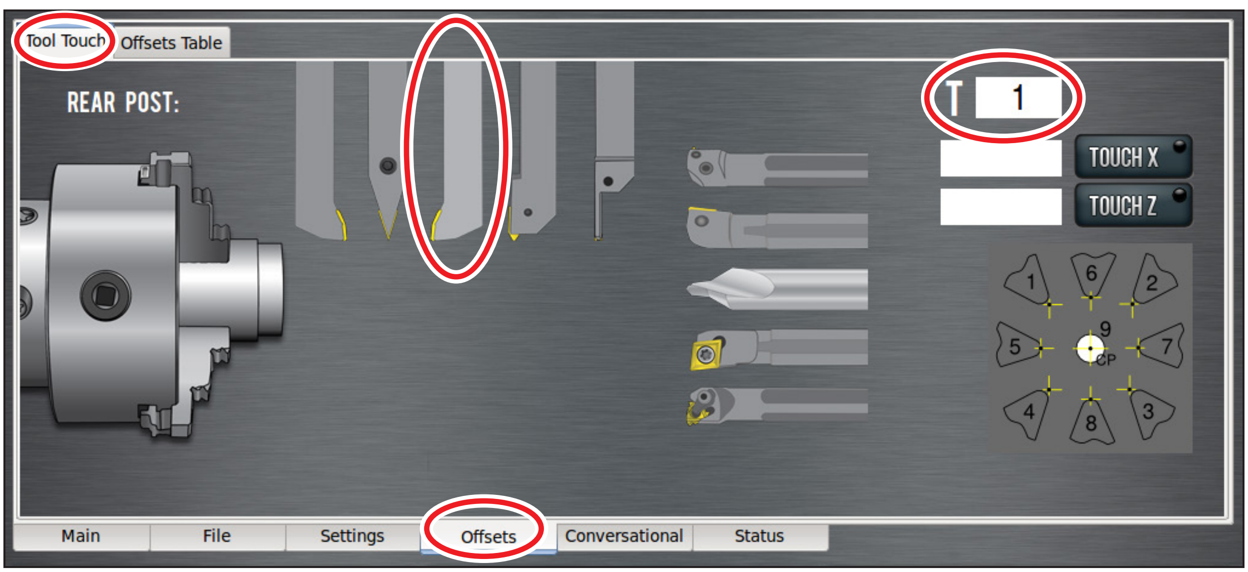


Figure 4.13

4.1.8 Set Tool Geometry Offsets

4.1.8.1 Tool 1

1. Insert a right-handed turning tool into the quick change tool post.
2. Select tool 1 in PathPilot interface – either click *Tool Touch* tab on the *Offsets* screen and type 1 in the tool *T* DRO (see **Figure 4.13**), or type T01 in the MDI line; click *Enter*.

3. On the *Tool Touch* tab on the *Offsets* screen, select right-handed turning tool (see **Figure 4.13**).

NOTE: *If the wrong tool type is selected, click Reset Tool to reset the tool type (see **Figure 4.8**) and return to the tool selection screen (see **Figure 4.13**).*

4. Click the *Main* tab (see **Figure 4.14**).

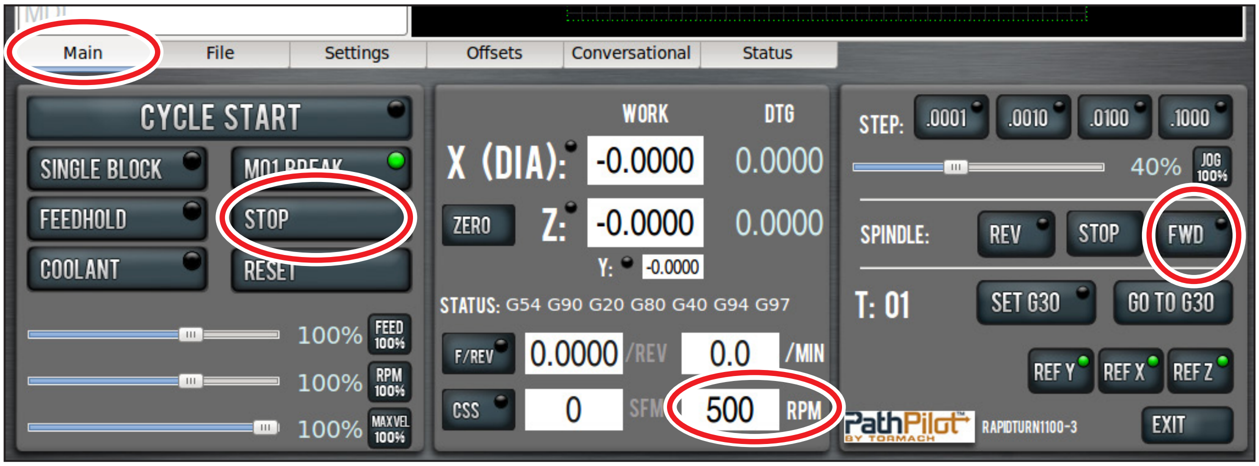


Figure 4.14

5. Jog machine until tool is near workpiece (see **Figure 4.15**).

WARNING! Ejection Hazard: *Tooling, fixtures, workpieces, or other loose items could become dangerous projectiles; ensure all components are appropriately secured. Do not operate with chip guard removed. Failure to do so could result in serious injury or death.*

6. Ensure chip guard is installed.
7. Adjust spindle speed to 500 RPMs – enter 500 in the *RPM* DRO and click *FWD* (see **Figure 4.14**), or type G97 M03 S500 in the MDI line.
8. Using the jogging controls, take a skim cut off of the diameter of workpiece. Ensure cut is long enough to measure the diameter with a micrometer.
9. Jog tool away from workpiece in Z.

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

10. Stop the spindle – click *Stop* (see **Figure 4.14**), or type M05 in the MDI line.
11. Measure diameter of skim cut on workpiece with a micrometer (see **Figure 4.16**); enter value in *Touch X* DRO on the *Offsets* screen's *Tool Touch* tab and click *Touch X* (see **Figure 4.17**). An illuminated LED button indicates that a tool offset has been set for that tool.
12. With spindle stopped, jog machine toward face of the workpiece in Z.
13. Ensure spindle speed is set to 500 RPM; click spindle *FWD*.
14. Jog machine toward workpiece; face the workpiece with the right-handed turning tool.

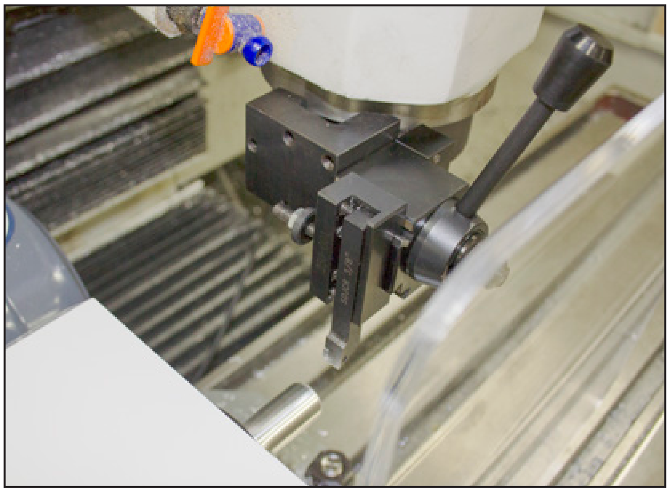


Figure 4.15

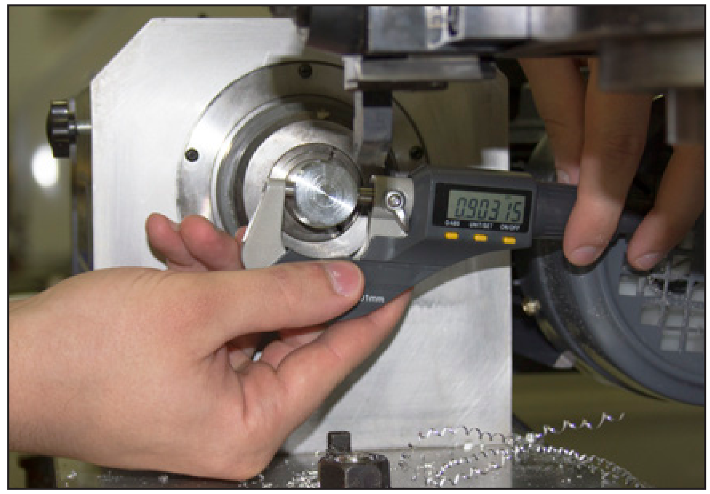


Figure 4.16

15. Stop the spindle; jog machine away from workpiece.

IMPORTANT! *Do not jog machine in Z.*

16. Enter 0 in *Touch Z* DRO; click *Touch Z* (see **Figure 4.17**). Use faced end of workpiece as Z-zero reference surface for all subsequent tools.

NOTE: *If the tool rubs the workpiece or leaves a pip on the end of the workpiece, adjust the knurled thumbwheel and lock nut on the tool holder to better align the tool with the spindle centerline and repeat steps 12-16. For more information, refer to Tool Holder Setup earlier in this chapter.*

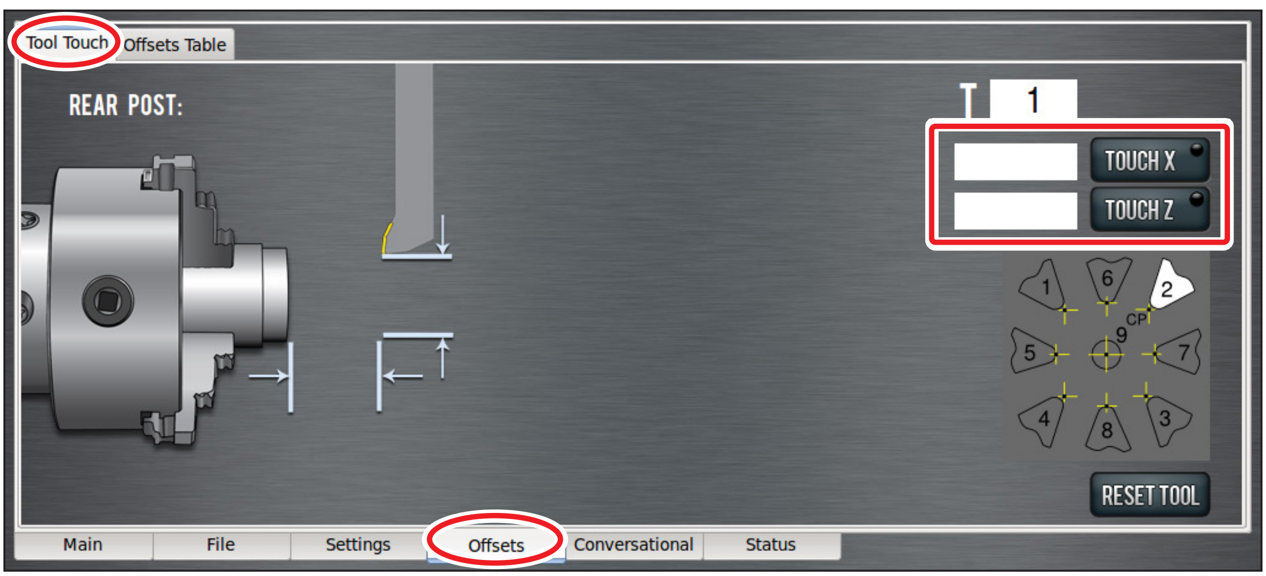


Figure 4.17

17. Verify that both *X (DIA)* DRO and *Z* DRO look appropriate for current position of the tool.

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

4.1.8.2 Tool 2

1. Jog far enough away from part to allow a tool change.
2. On *Main* tab, click *Set G30* button to set the tool change position. An illuminated LED indicates that a G30 position has been set.
3. On the *Tool Touch* tab on the *Offsets* screen, type 2 in the *T* DRO on the *Tool Touch* tab; click *Enter*.
4. Select parting tool (see **Figure 4.18**).

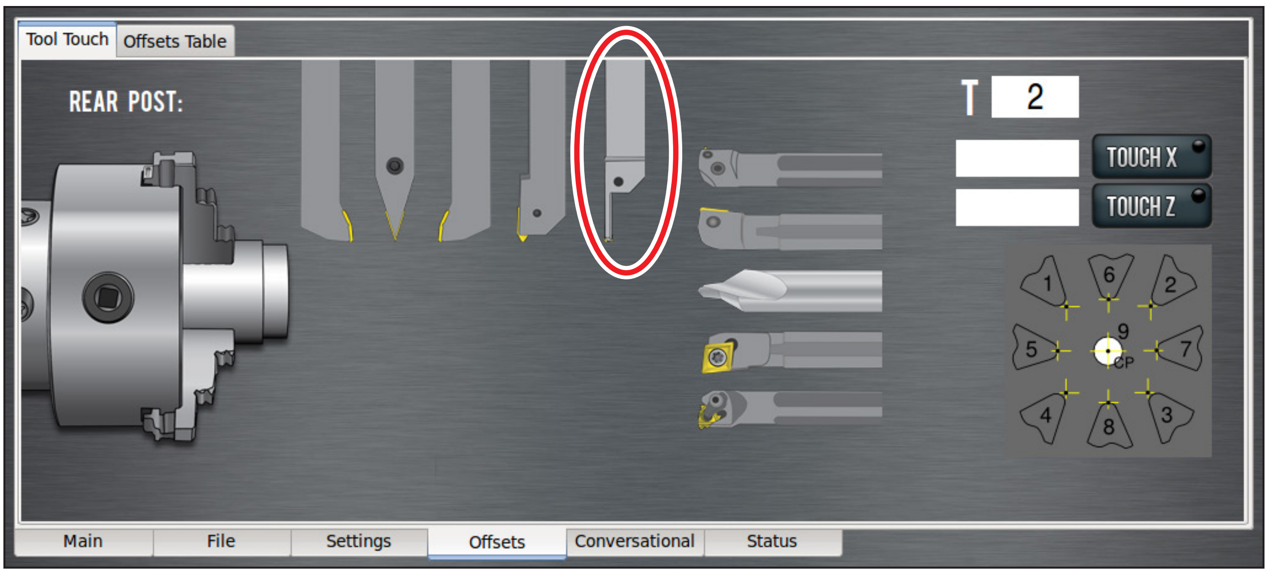


Figure 4.18

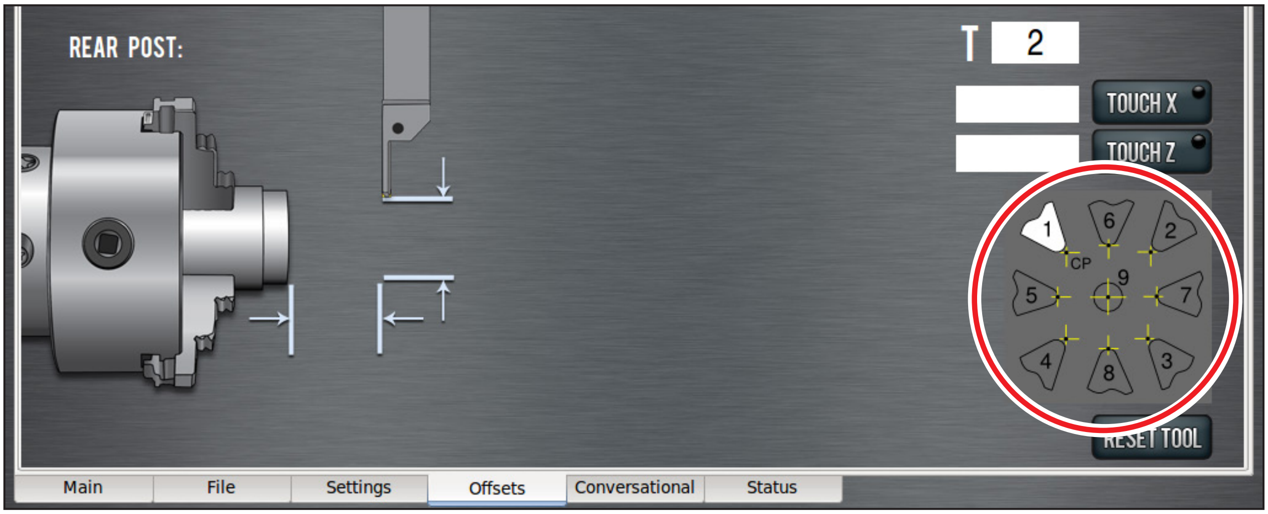


Figure 4.19

NOTE: *The control point diagram (see* ***Figure 4.19****) illustrates the parting tool's control point is 1 – on the right of the tool – as shown in* ***Figure 4.21****. This takes into account the length of the part, not the remaining stock in the spindle (see* ***Figure 4.20*** *and* ***Figure 4.21****). For more information on control points, refer to chapter 6, Programming.*

5. Start the spindle; jog machine to take a skim cut off of the diameter of the workpiece. Ensure cut is long enough to measure the diameter with a micrometer.
6. Jog tool away from workpiece in Z.

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

7. Stop the spindle and measure the diameter of the workpiece with a micrometer. Enter that value in the *Touch X* DRO; click *Touch X*.
8. With the spindle stopped, jog machine toward face of workpiece in Z direction.

Control Point 2

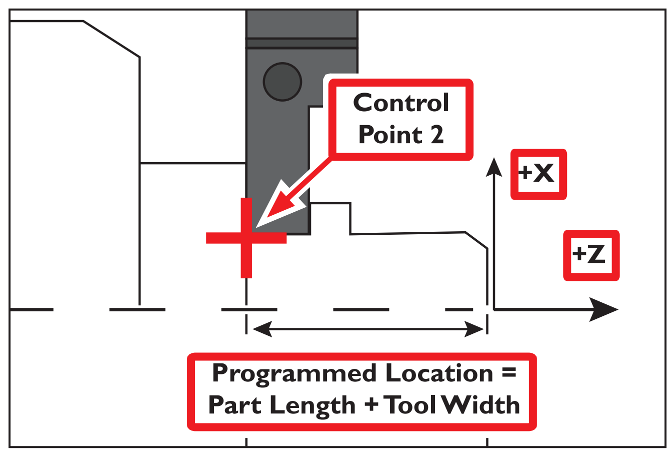


Figure 4.20

Control Point 1

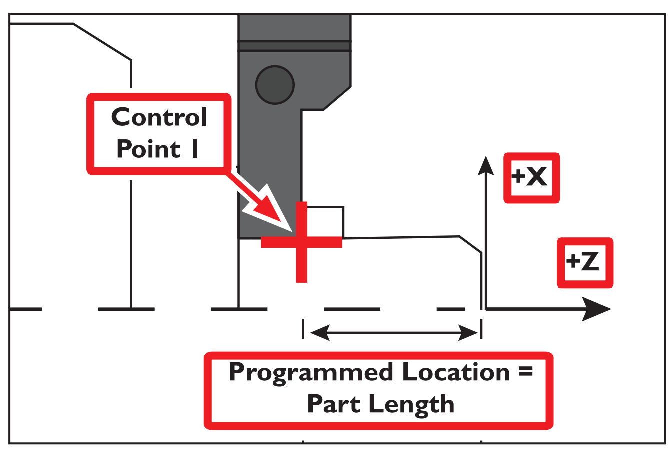


Figure 4.21

9. Place a piece of paper between tool and workpiece to touch off Z. Step machine to bring tool's tip toward faced end of workpiece used to set Z-zero for tool 1. When paper binds up, stop stepping; enter thickness of paper (0.003") plus width of parting tool insert (typically 0.120" for a GTN-3 type insert) in *Touch Z* DRO; click *Touch Z*.

NOTE: *The width of the insert is included when setting tool 2 because the control point of a parting tool is the right edge, not the left edge (see* ***Figure 4.20*** *and* ***Figure 4.21****).*

10. Click the *Offsets Table* tab on the *Offsets* screen to double-check geometry offsets for each tool (see **Figure 4.22**). Values in the X and Z columns represent stick out of tool past faces of tool holder.

Figure 4.22

NOTE: *The values in the X column are expressed in diameters, so the actual stick out is half the number shown in the column.*

11. Verify tool is visually aligned with spindle centerline; adjust knurled thumbwheel and locking nut to better align tool with spindle centerline, if necessary. For more information, refer to *Tool Holder Setup* earlier in this chapter.

Tool offsets do not need to be re-set until an insert is replaced, so long as the tools remain in their tool holders.

4.1.9 Write G-code

Using the conversational programming capabilities of PathPilot, the next step of this tutorial is generating G-code to produce the part. 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. For the purposes of this tutorial, enter in the values shown throughout this section; entry fields should be self-explanatory, but are covered in detail in following chapters.

Make note of a few things:

- The spindle speed (both roughing and finishing) is expressed in surface feet per minute (SFM), not revolutions per minute (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, extend 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.

The first part tutorial is broken down into five operations:

1. Face the part.
2. Turn the 1” stock down to 0.7500” along the entire length of the part (1.5”).
3. Turn the first 1.25” of part down to .5000.”
4. Chamfer the edge with a 0.050” break.
5. Part off.

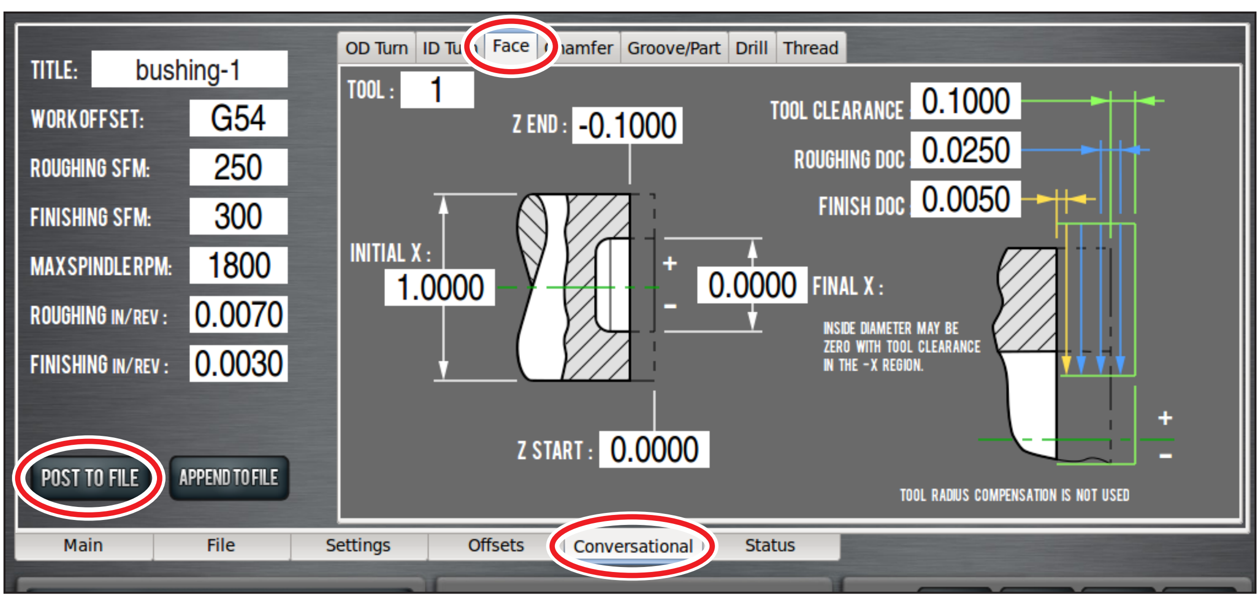


Figure 4.23

4.1.9.1 Operation 1

1. Insert tool 1 into the quick change tool post.
2. Click *Face* tab on the *Conversational* screen (see **Figure 4.23**) and enter values shown in **Figure 4.23** to generate code that faces the part back to Z of -0.100".
3. Click *Post to File* to save G-code (see **Figure 4.23**).
4. In dialog box, click *Save* (see **Figure 4.24**); file is saved and automatically loads into control, displaying tool path (see **Figure 4.25**).
5. Click and drag *MAXVEL Slider* (see **Figure 4.25**) to zero percent.
6. Click *Cycle Start* (see **Figure 4.25**). If current tool is not tool 1 and machine is configured for manual tool changes, *Cycle Start* LED may flash to request tool change. Change tool in quick change tool post; click *Cycle Start* again to confirm.
7. Click and drag *MAXVEL Slider* to slowly increase allowed velocity. Bring velocity back down to zero when tool is close to part; double-check DRO values. If everything looks correct, slowly move *MAXVEL Slider* back up to resume running part program.

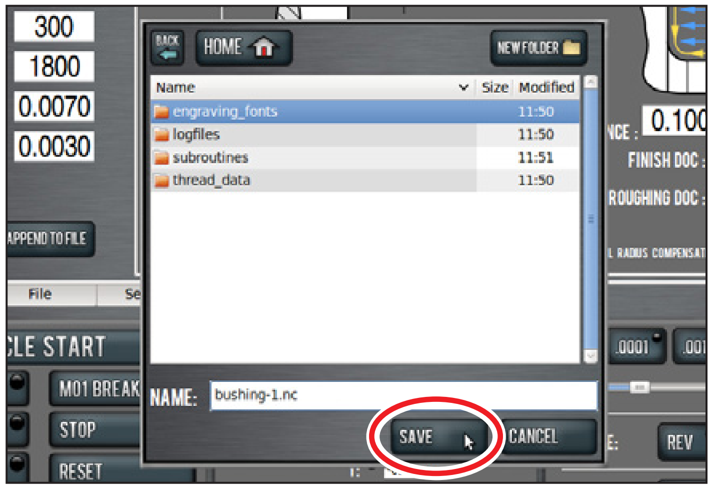


Figure 4.24

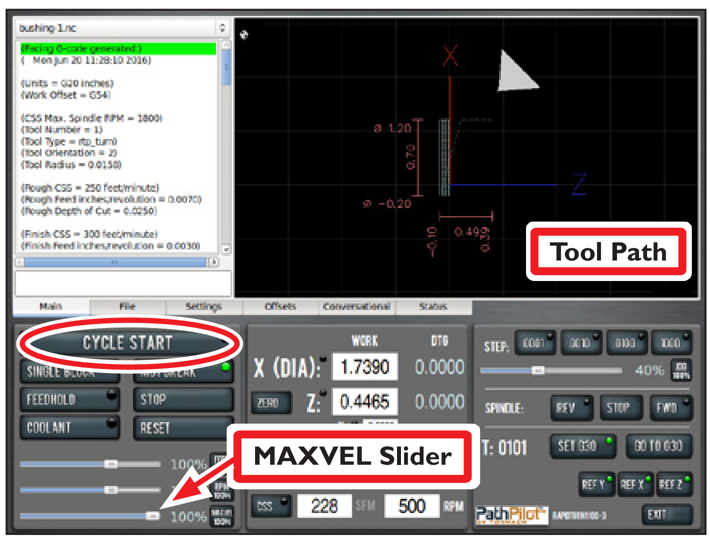


Figure 4.25

4.1.9.2 Operation 2

1. Click on *OD Turn* tab on the *Conversational* screen. Enter values shown in **Figure 4.26**.
2. Click *Append to File* (see **Figure 4.26**).

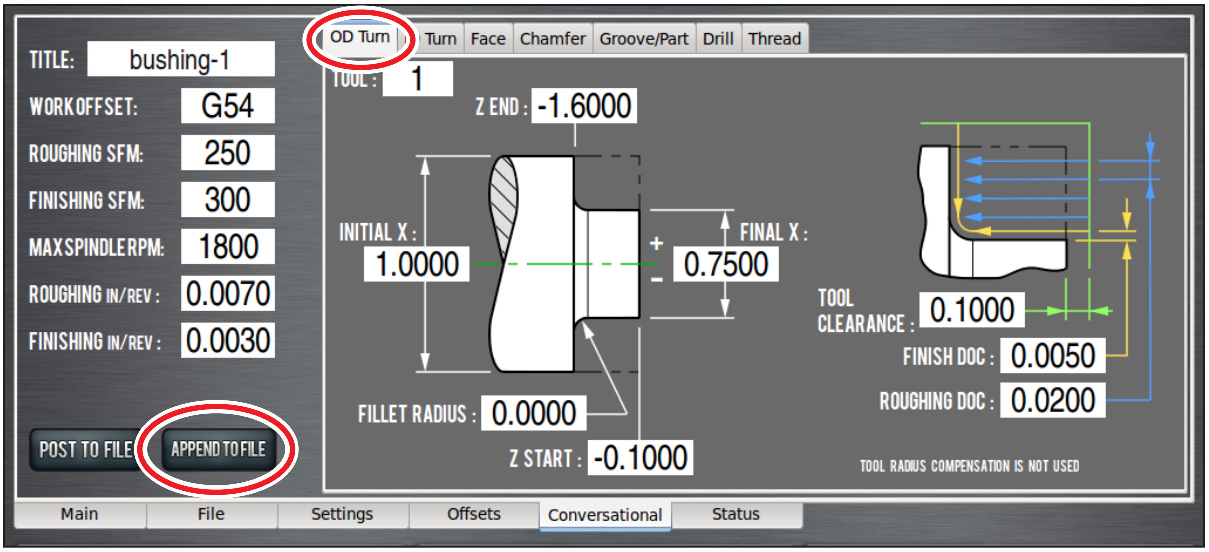


Figure 4.26

3. In dialog box, select file created in Operation 1 (see **Figure 4.27**); this adds OD turn G-code to that program.
4. Confirm Append to File in dialog box (see **Figure 4.27**); this loads changes from Operation 2 to the G-code file into control and generates the tool path (see **Figure 4.28**).

IMPORTANT! *Do not run this code until remainder of operations are written.*

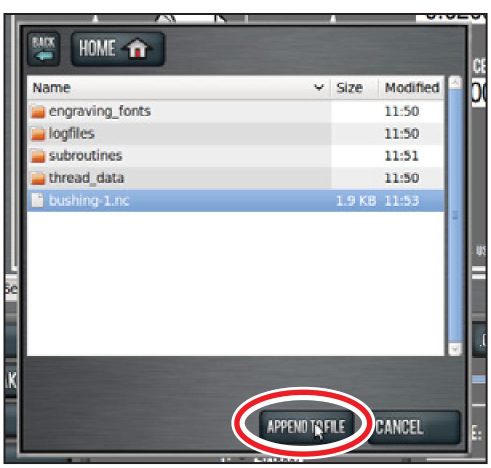


Figure 4.27

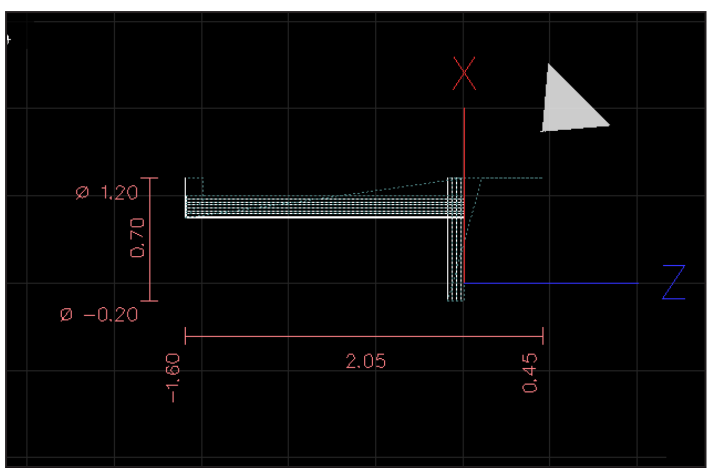


Figure 4.28

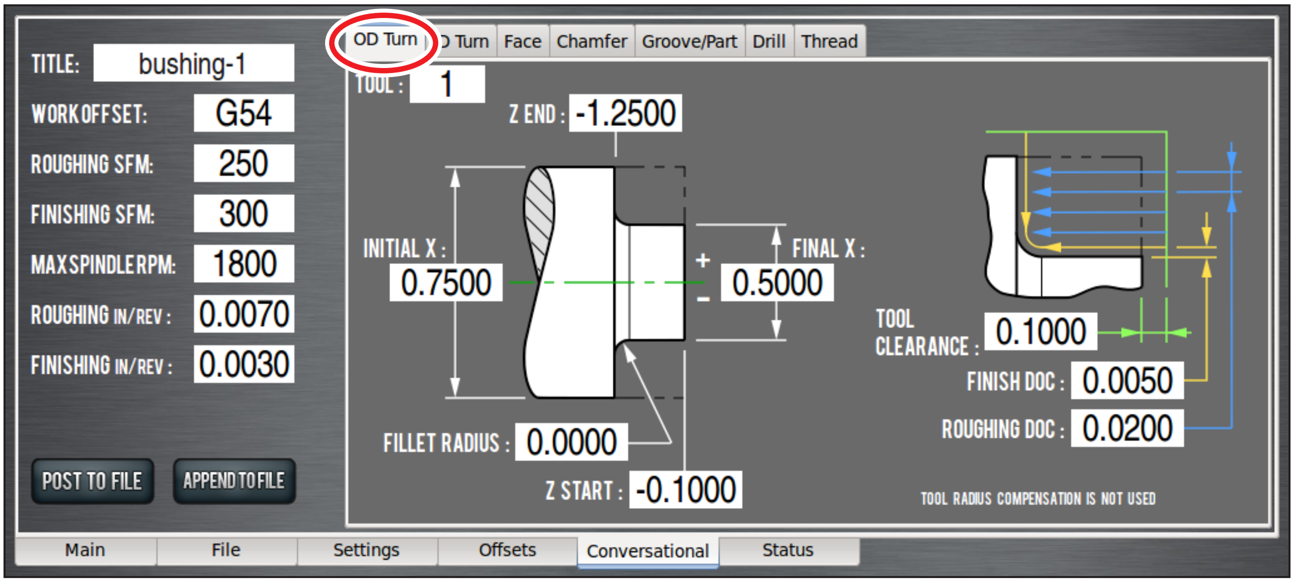


Figure 4.29

4.1.9.3 Operation 3

1. Click on *OD Turn* tab on the *Conversational* screen. Enter values shown in **Figure 4.29** to create another OD turn routine that turns the first 1.125" of the part down to 0.5000".
2. Click *Append to File* to add this code to part program and generate tool path shown in **Figure 4.30**.

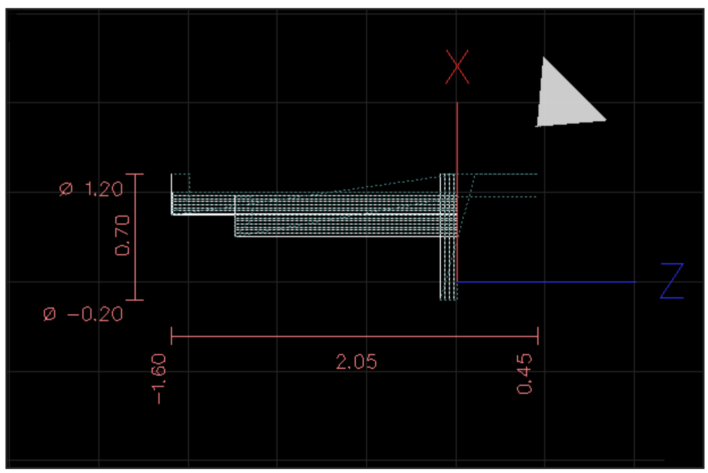


Figure 4.30

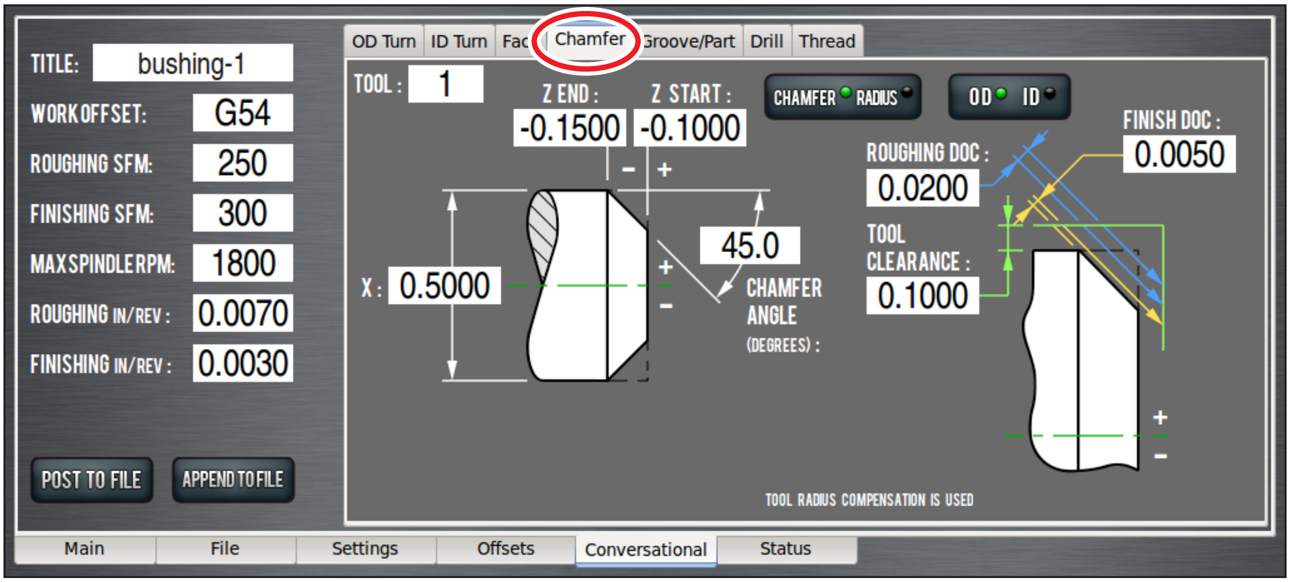


Figure 4.31

4.1.9.4 Operation 4

1. Click on *Chamfer* tab of the *Conversational* screen. Enter values shown in **Figure 4.31** to put a small chamfer on the part.
2. Click *Append to File* to add this code to part program and generate tool path shown in **Figure 4.32**.

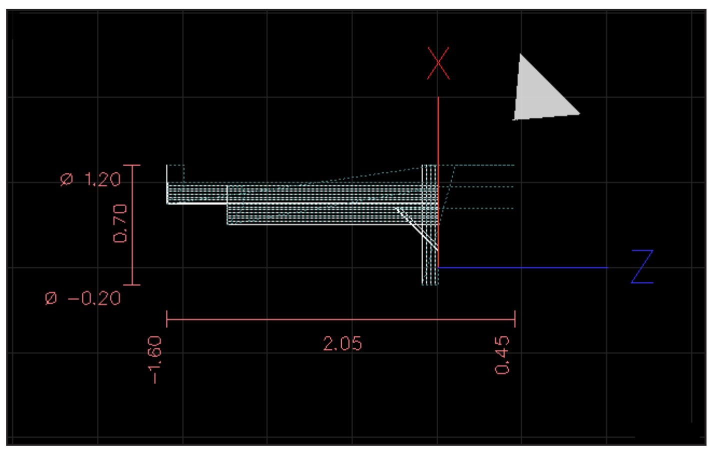


Figure 4.32

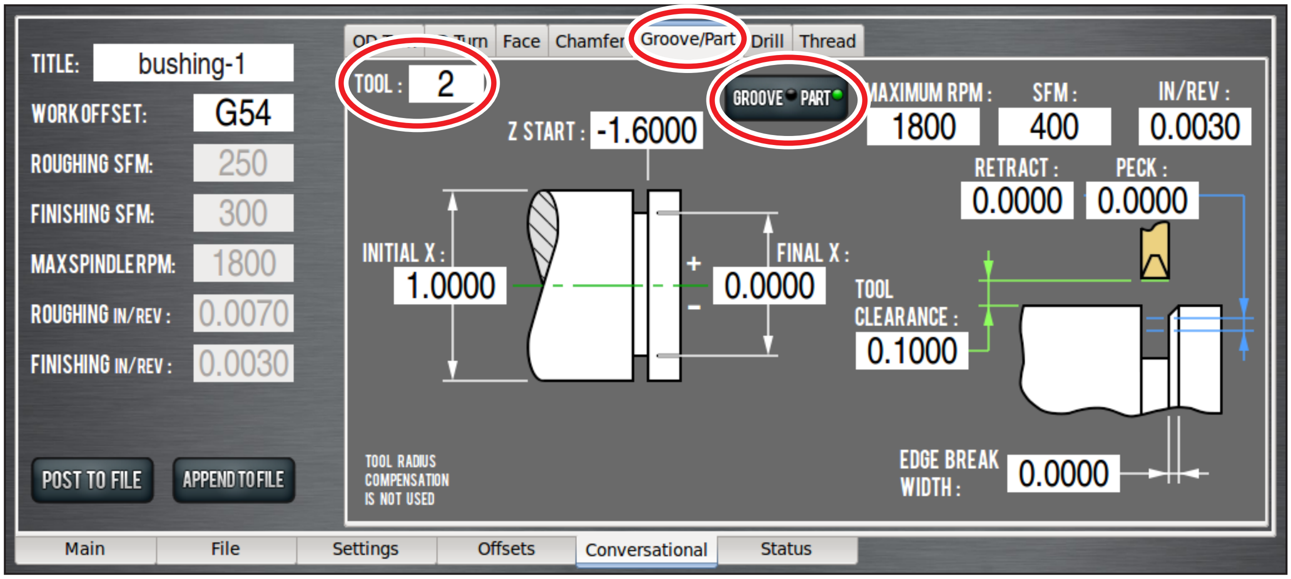


Figure 4.33

4.1.9.5 Operation 5

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

1. Click *Groove/Part* tab on the *Conversational* screen; toggle the *Groove/Part* button to select the *Part* screen. Enter values shown in **Figure 4.33**.

NOTE: *Ensure tool 2 is entered in the Tool DRO (see* ***Figure 4.33****). Using tool 1 generates an error message to alert that a right-hand turning tool is not a valid tool for a parting routine.*

2. Click *Append to File* to add this code to part program and generate tool path shown in **Figure 4.34**.

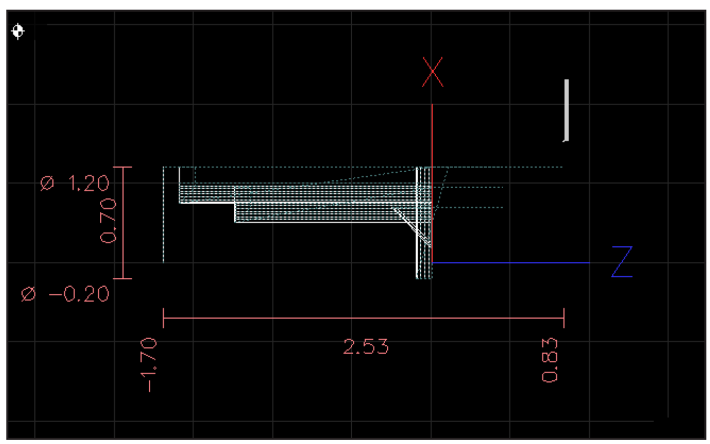


Figure 4.34

4.1.10 Run Completed G-code Program

1. On the *Main* tab, click and drag *MAXVEL Slider* to zero percent (see **Figure 4.35**).
2. Click *Cycle Start* (see **Figure 4.35**); click and drag *MAXVEL Slider* to slowly increase allowed velocity. Bring velocity back down to zero when tool is close to part; double-check DRO values. If everything looks correct, move *MAXVEL Slider* back up to resume running part program.

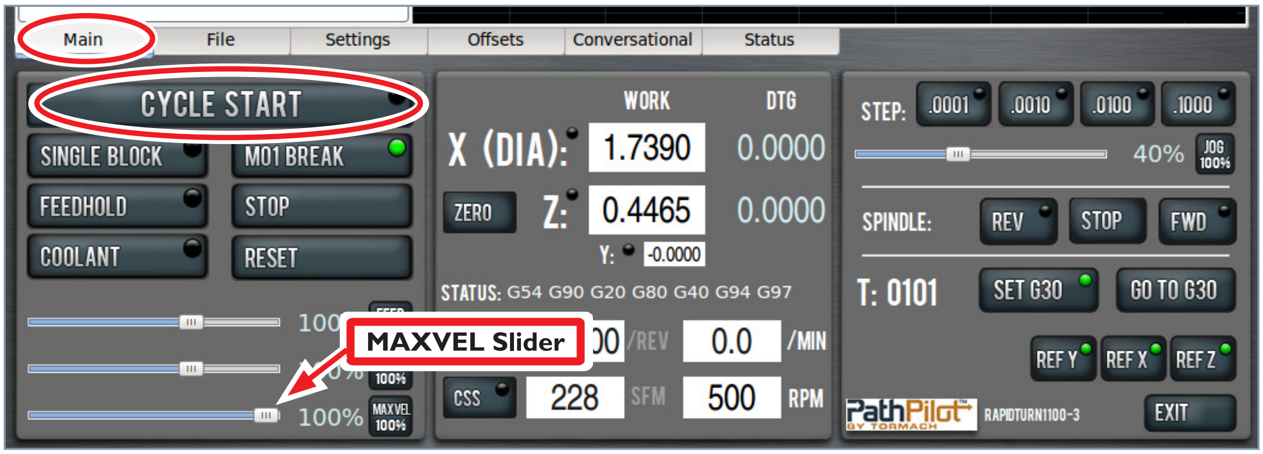


Figure 4.35

The techniques used to program the first part (see **Figure 4.36**) form the basis of most of the operations performed on a CNC lathe.

Finished First Part

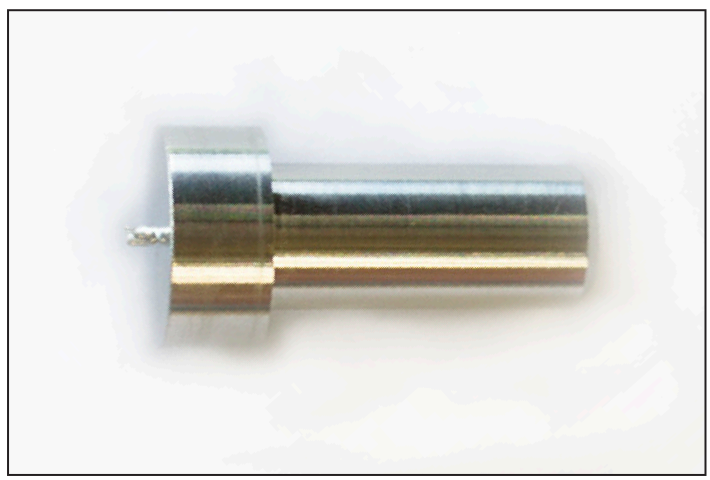


Figure 4.36

5. PathPilot Interface

5.1 Overall Layout

The PathPilot® interface is divided into two main sections: *Notebook* and *Persistent Controls* (see **Figure 5.1**). *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 PathPilot interface. For example, 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.

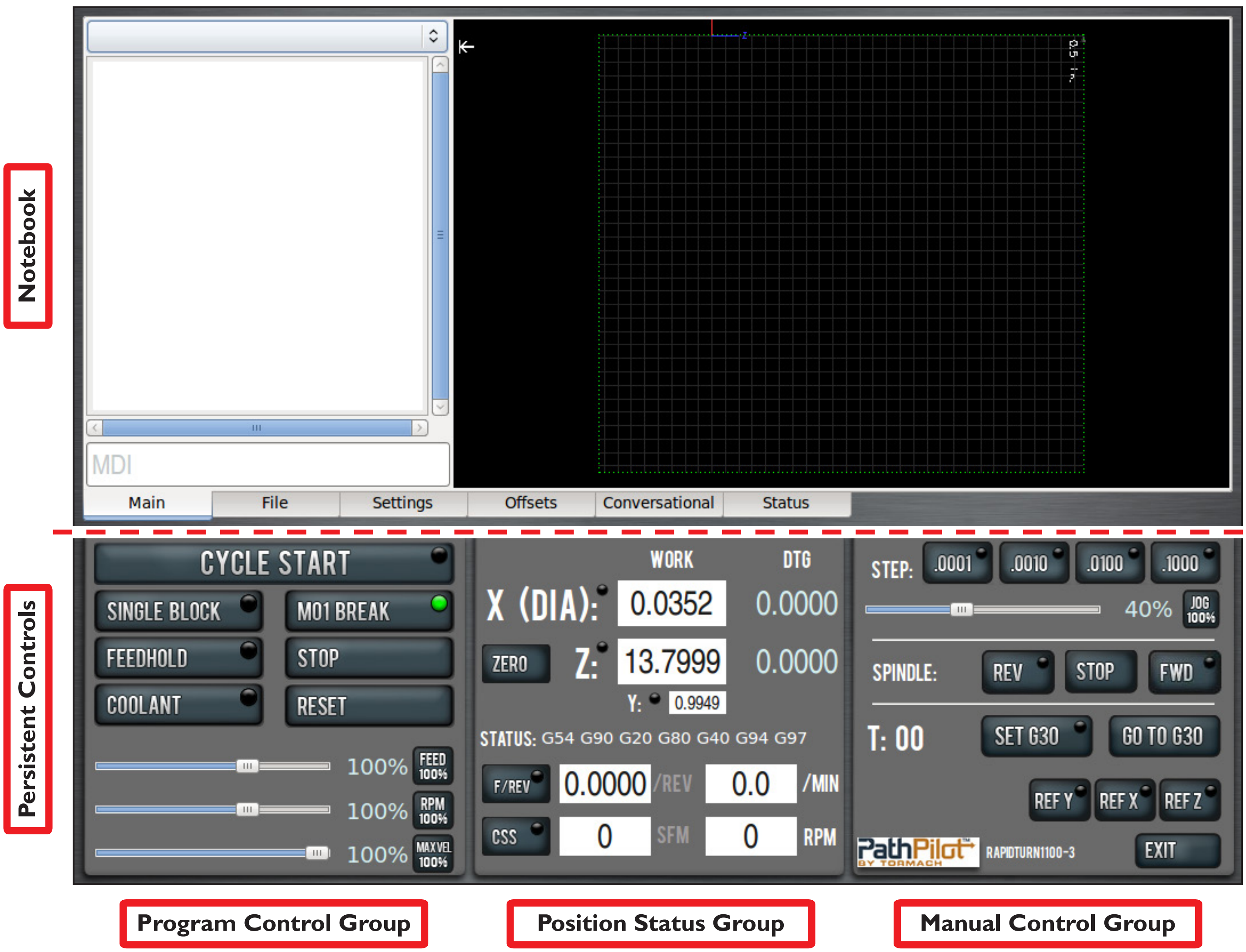


Figure 5.1

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.

5.2 Persistent Controls

The controls on the lower half of the screen are present on each page of 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**).

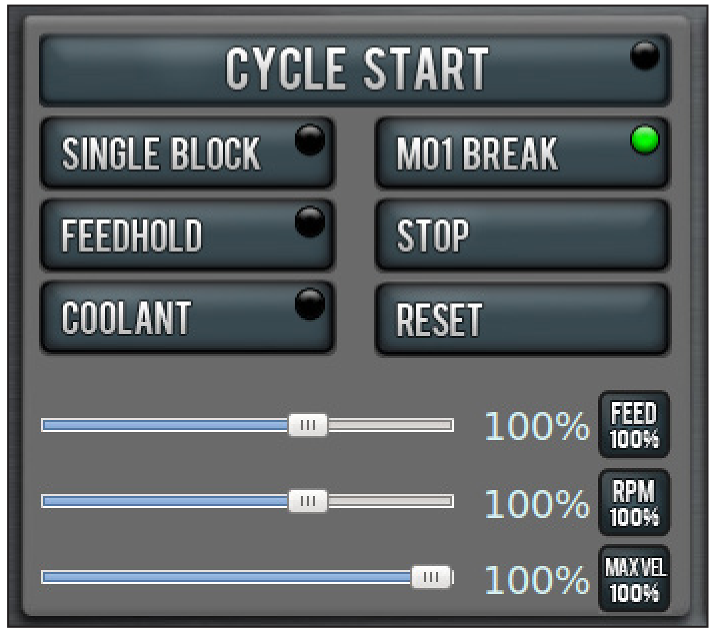


Figure 5.2

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 spindle door is closed and a valid G-code program is loaded. During a program, the *Cycle Start* button LED is illuminated.

If *Single Block* is active, RapidTurn™ executes one line of G-code per *Cycle Start* click. When running a program – if RapidTurn motion is paused due to *Feedhold*, *M01 Break*, *Single Block*, or because the program 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; 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; *Cycle Start* LED flashes. The machine 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 move until the 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 (for more information, see *Reset* later in this section). Clicking the *Stop* button does not 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 outlet on the bottom 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 any time after 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
- 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 does not drive the spindle past its maximum speed of 3500 RPM (5C configuration) or 2000 (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 Override* slider can be useful when running new code for the first time; stop the machine by sliding it down to 0 percent and verifying that the Distance To Go (DTG) and X (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* – 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.

The RapidTurn functions as a two axis lathe on a three axis mill platform. Unlike the X- and Z-axes which work together to execute commanded moves, the Y-axis is only used for aligning the RapidTurn’s spindle centerline to the tooling’s nominal height. In practice, the Y-axis is only used once per setup of the controller after the machine is referenced. After the tools are brought onto the spindle’s centerline, the axis can be zeroed and left alone during operation.

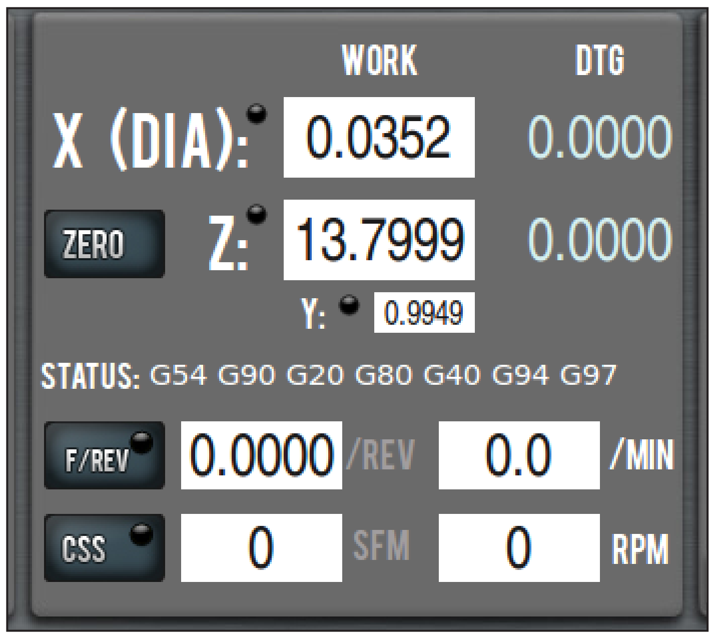


Figure 5.3

When the machine is at rest, these DROs 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 and press *Enter* on the keyboard. The *Esc* key returns to the original value.

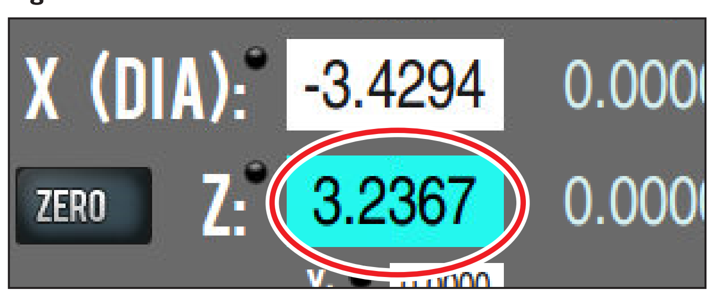


Figure 5.4

***NOTE:** This action does not move the machine, but does change the work offset setting.*

This technique is used for setting any DRO. It is important 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 (Dia)* 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 – The RapidTurn recognizes two feed rate modes: units per minute (G94) and units per revolution (or REV) of spindle (G95) as shown in **Figure 5.5**. For more information on feed rate modes, refer to chapter 6, *Programming*. 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.

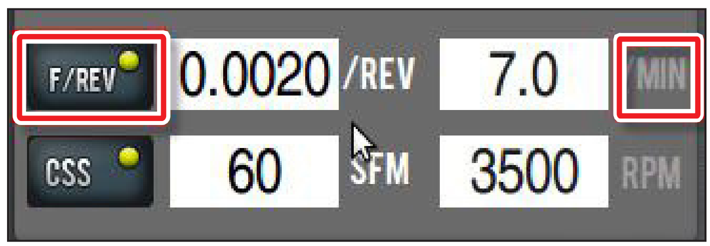


Figure 5.5

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 turning, programming in units per revolution is more desirable.*

These settings are mutually exclusive – 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 is illuminated, the *F/REV* label is white, and the *F/MIN* label is gray. *F/REV* mode can be turned on by any of the following methods:

- 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 off
- 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 methods:

- 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 illuminated
- 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 RapidTurn 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 workpiece'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. Tool life and efficiency depends on surface speed (and not RPMs), so constant surface speed mode is the preferred mode of operation of CNC lathes.

5.2.3 Manual Control Group

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

Figure 5.6

Jogging Controls – The RapidTurn can be jogged with either the jog shuttle or by using the arrow keys on the keyboard.

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*.

For more information on using the keyboard keys to jog the machine, refer to chapter 3, *Operation*.

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 off, 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.

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.

Tool Label – The tool label (*T:*) 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*. This label flashes with the requested tool number while waiting for a tool change when a T command is encountered while running a G-code program.

G30 Controls – 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 Axes Buttons – The *REF X*, *REF Z*, and *REF Y* buttons move the X-, Z-, and Y-axis to their respective reference switch locations. This must be done after power on and before running a part program or using MDI commands. The axes must be referenced individually and not simultaneously. When referenced, each button’s LED is illuminated.

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 controller is first powered on (see **Figure 5.7**). The *Main* screen contains four controls: recent files/current file display, tool path preview, G-code window, and MDI field.

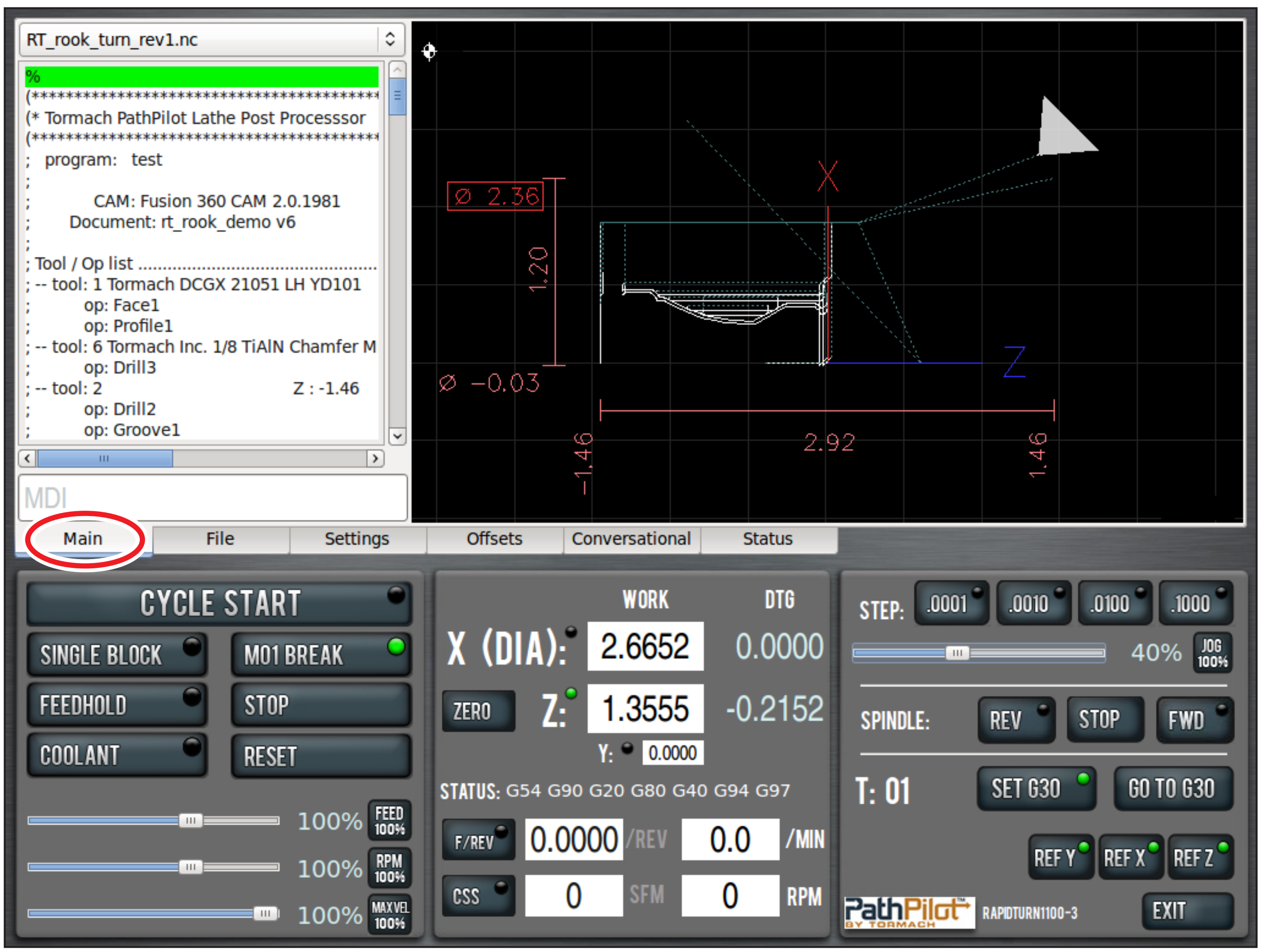


Figure 5.7

Recent Files Drop-down/Current File Display

The top left portion of the *Main* screen has a drop-down menu that displays the currently loaded G-code file (see **Figure 5.8**). Clicking on it displays the names of the last six files that have been loaded into the control. Selecting any one of those files from the drop-down will load that file.

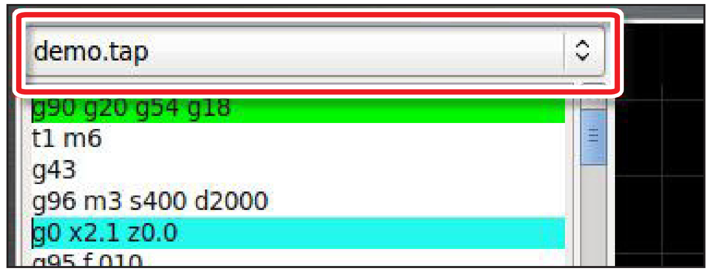


Figure 5.8

G-code Window/Set Start Line

The G-code window displays the G-code in the currently loaded file. Use the scroll bars to scroll down to see the entire file. The line at which the G-code program is executing – the 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. Keep in mind that when using the *Set Start Line* option, the operator must be responsible for ensuring that the machine is in the proper modal state before the code executes. The machine does not read backward through the beginning of the file to do things like turn the spindle on; it starts from the start line as if it was reading in a new G-code file. To start from the middle of a program, make sure that the right G modes (94/5, 96/7) are set and that any preparatory moves (i.e., turning the spindle on) are taken care of manually before clicking *Cycle Start* after selecting *Set Start Line*.

MDI Line

When making parts the commands to the machine – G- and M-codes – are generally read from a file. However, it is often convenient to command the RapidTurn directly. This can be done by typing a command into the MDI line. The command to start the spindle in the forward direction at 500 RPM is `M03 S500` and the command to stop it is `M05`. Click the mouse in the bar marked *MDI*; it will highlight, indicating a command can be typed in.

The *Backspace*, *Delete*, left arrow and right arrow keys are available to help correct any typing errors. Pressing *Enter* executes the command; pressing *Esc* abandons it and closes the MDI field. For example, try starting and stopping the spindle with M03 and M05 G-codes (i.e., `M03 S500`).

The recent commands are available by pressing the up arrow key. Up to 100 commands are saved between sessions, so that the history is available even after a controller power cycle.

NOTE: *All keystrokes go to the MDI when open, so it is not possible to jog the axes using the arrow keys.*

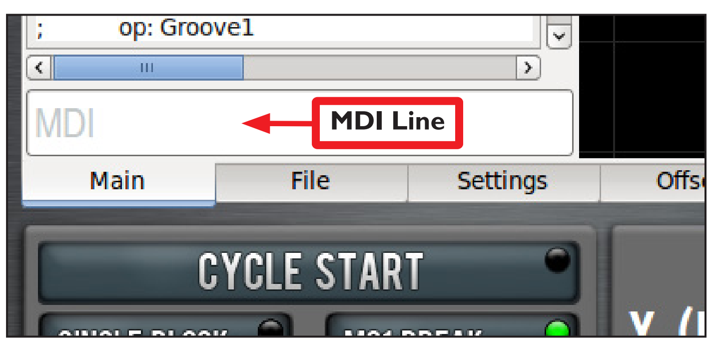


Figure 5.9

MDI has the ability to search text of a G-code 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. Change the starting point of the search by clicking on any line in the G-code window.

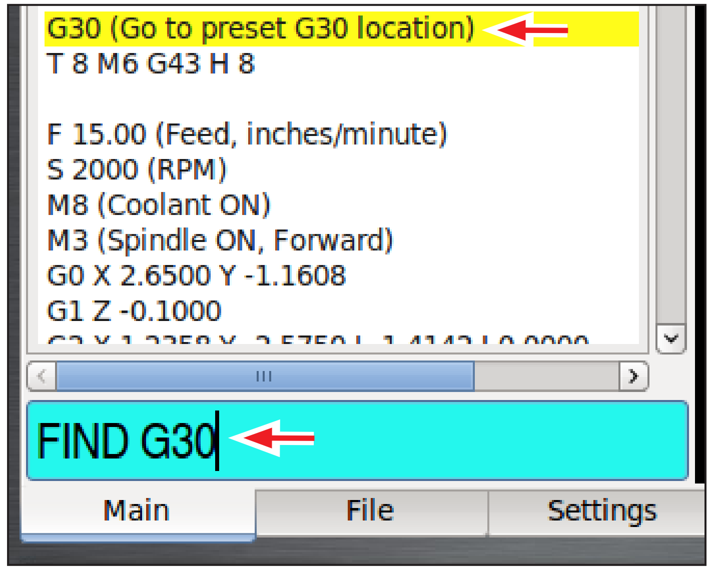


Figure 5.10

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**)

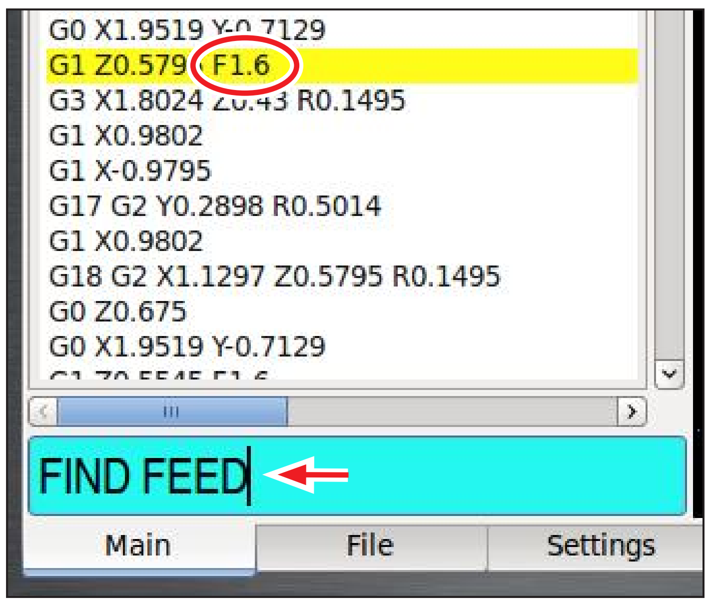


Figure 5.11

***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.

Tool Path Display

The tool path window displays a graphic representation of the tool path that is executed for the currently loaded G-code file. Preview lines are drawn in white, the tool path as it is cut is drawn in red, and jogging moves are drawn in yellow. To erase the yellow jogging and red tool path lines at any time, double-click the display. Grid lines are visible behind the tool path – by default these are drawn at 0.5" intervals (or at 5 mm intervals when in G21 metric mode). To change the resolution of the grid lines, right click anywhere in the tool path display and select the desired grid spacing.

Notice that 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 for transferring files to and from a USB drive, copying, deleting, and renaming files and folders (see **Figure 5.12**). The left window shows files and folders on the hard drive of the controller; 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.*

Figure 5.12

The *Back* and either *Home* or *USB* buttons above the left and center window (see **Figure 5.12**) can be used to navigate through the file structure, which is similar to the file structure on a home computer. The *Copy to USB* and *Copy From USB* arrow buttons between the Hard Drive Window and USB Drive Window are used to move files between USB drive and controller (see **Figure 5.12**).

Move File from USB to Controller

1. To transfer a file from the USB drive to the controller, insert a USB drive into an open USB port.
2. Navigate to the file you want to copy by double clicking on folders in the USB Drive Window (see **Figure 5.12**).

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

3. Navigate to the desired folder/location in the hard drive window and click *Copy From USB* (see **Figure 5.12**).
4. If the file transferring has the same name as an existing file on the controller, either overwrite the file, re-name it, or cancel the file transfer. When copied to the new location, the file displays in the USB Drive Window (see **Figure 5.12**).
5. Click the *Eject* button when you are ready to disconnect the USB drive from the controller (see **Figure 5.10**). Ejecting the USB drive this way helps to avoid corrupting data on the USB drive.

File Management

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**). Files can also be moved into a folder by dragging and dropping the file or right-clicking on the file cutting or copying from the pop-up menu.

Edit G-code

1. The *G-code File Preview* window displays the contents of the selected .nc file (see **Figure 5.12**). To edit the G-code, highlight the file and click the *Edit G-code* button.
2. A text editor will open the file in a new window for editing the contents of the file. Make the appropriate changes to the file and click *Save*.
3. Close the text editor by clicking on the *X* in the upper right-hand window of the screen. When asked to reload the file, click *OK* (see **Figure 5.12**).

Load G-code

The Load G-code function is only available for files stored on the controller.

1. Navigate to the desired .nc file in the Hard Drive Window (see **Figure 5.12**).
2. With the file highlighted, click *Load G-Code* button (see **Figure 5.12**).
3. Click on the *Main* tab to open the *Main* screen; verify the G-code file name appears in the drop-down window located in the upper left-hand corner of the screen. For information on running a program, refer to *Main Tab* section earlier in this chapter.

5.6 Settings Tab

On the left-hand side of the *Settings* tab screen is a window that displays a range of available G-code modalities (see **Figure 5.11**). Active G-codes are highlighted in yellow. The right-hand side of the *Settings* tab screen has checkboxes that allow configuration of PathPilot to suit the machine.

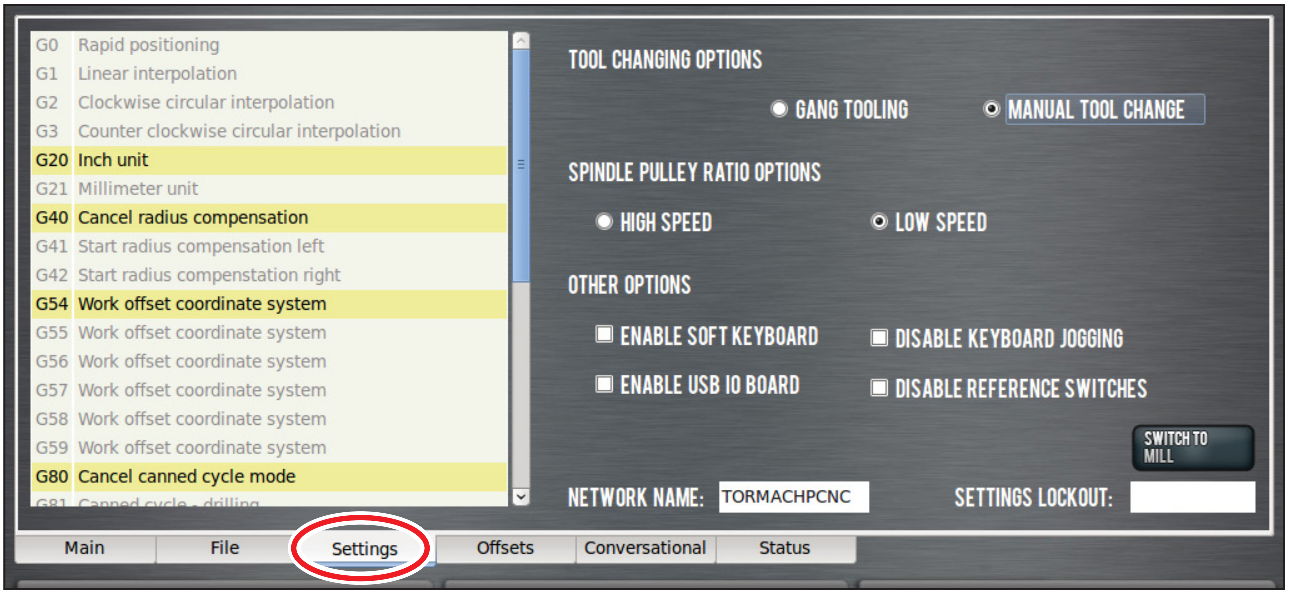


Figure 5.13

5.7 Offsets Tab

The *Offsets* screen is subdivided into two tabs: *Tool Touch* and *Offsets Table* (see **Figure 5.14**). 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.

5.7.1 Tool Touch Tab

The *Tool Touch* tab screen allows the operator to graphically select and touch off a tool to set the geometry offsets. The tool images act as buttons, and they highlight as they are moused over (see **Figure 5.15**).

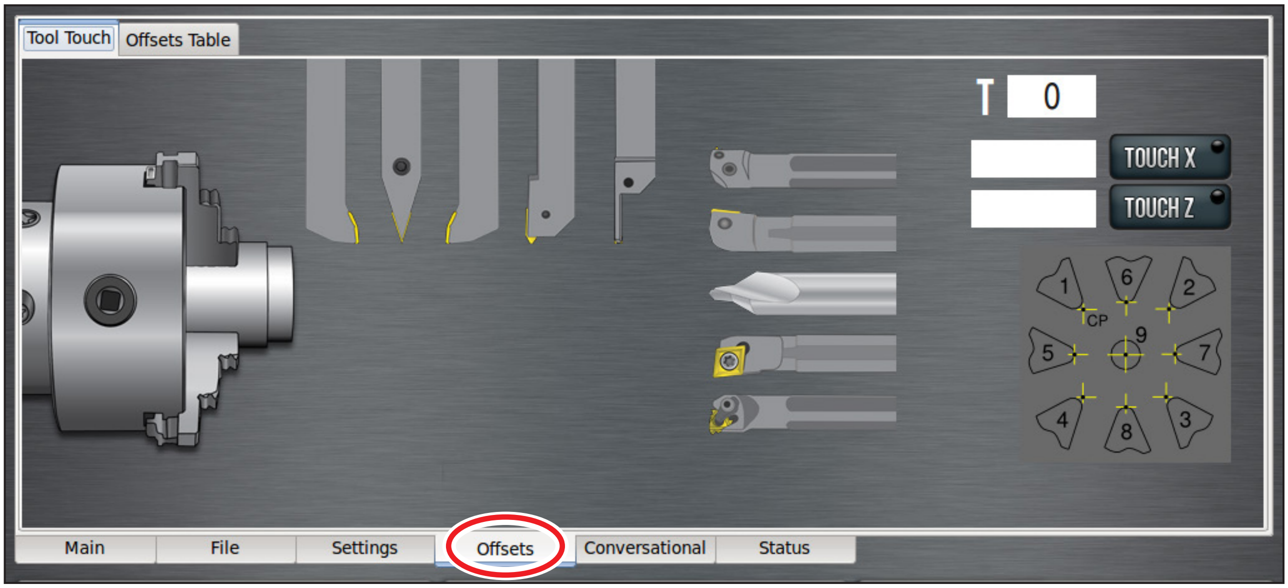


Figure 5.14

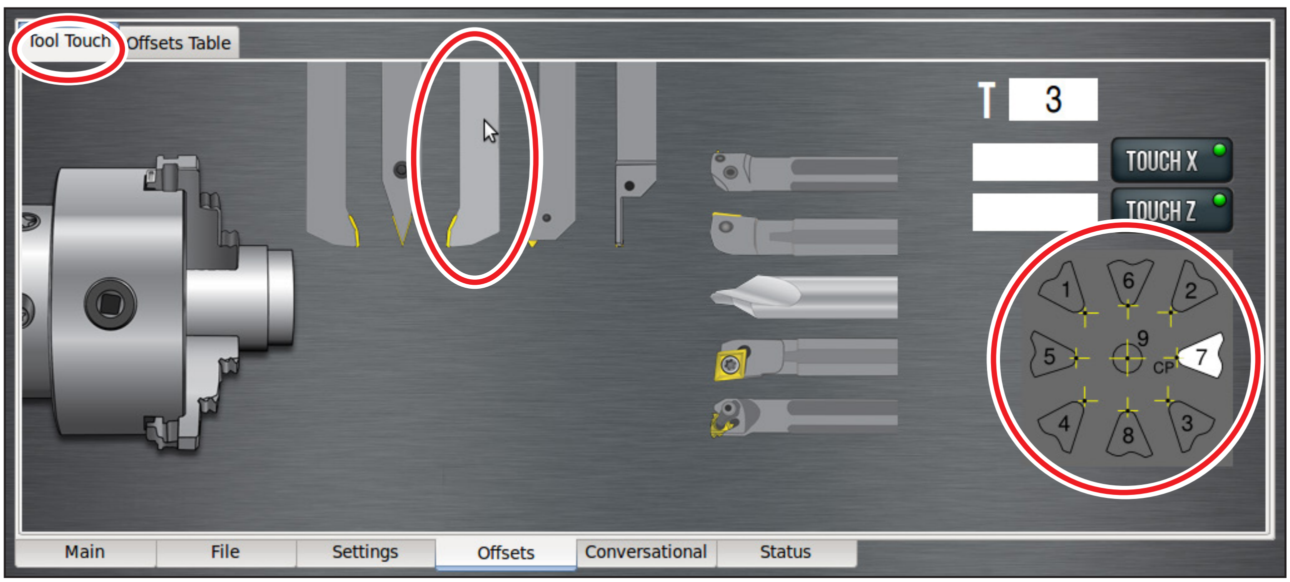


Figure 5.15

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 orientation, used by the conversational routines to double check the user entry fields in an attempt to try to detect and prevent crashes.

NOTE: *Tip orientation has been automatically set to Z, as shown in* ***Figure 5.15****. This dialog is intended to make setting the tool geometry offsets (or touching off the tool) easy and intuitive.*

3. Brings up the tool touch off dialog (see **Figure 5.16**).

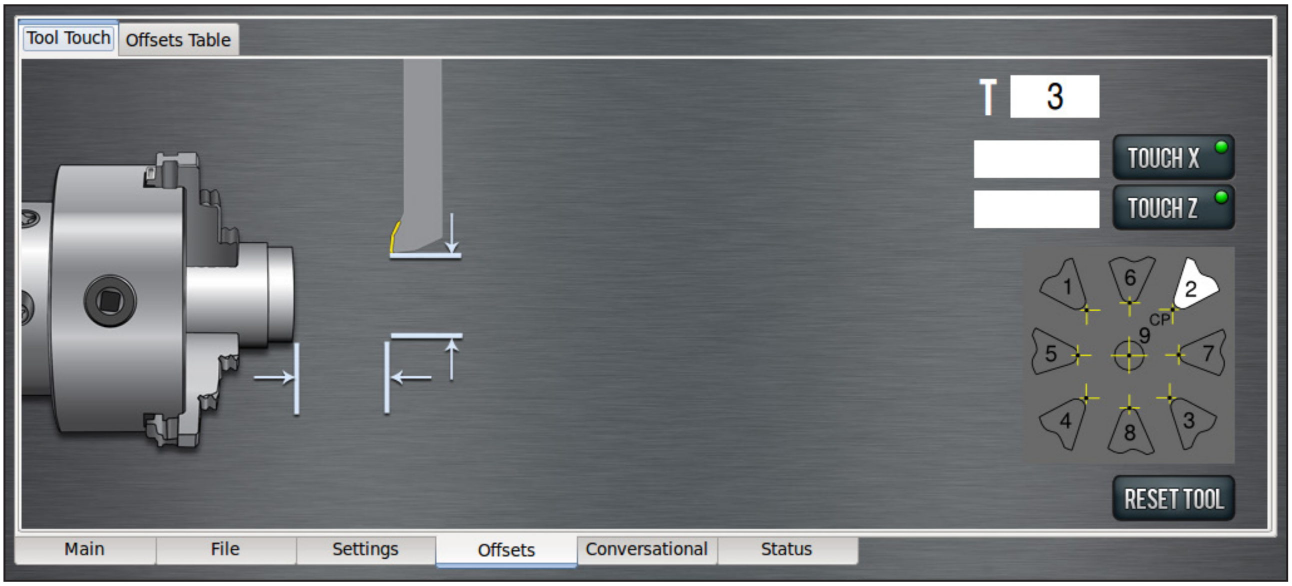


Figure 5.16

5.7.1.1 Touching Off Tool

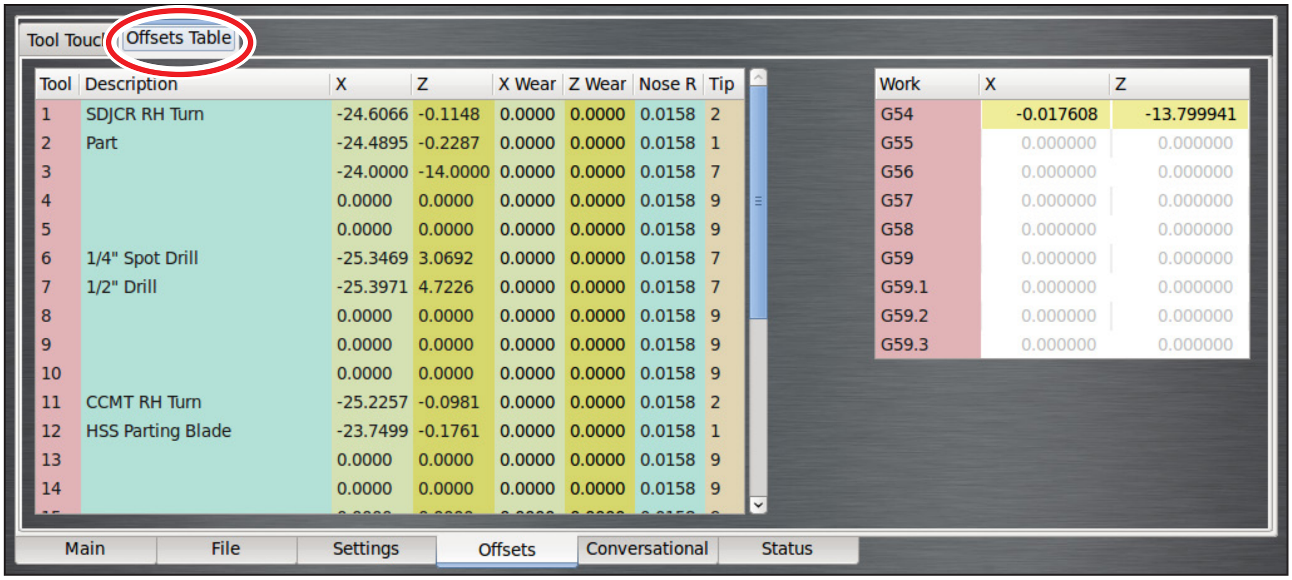
Touch X (Dia.)

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.
2. Measure the diameter of the skim-cut workpiece with a micrometer, then enter that value in the *Touch X* DRO. If touching off an inverted boring or internal threading tool, enter the negative of the value measured. When this value has been entered, click the *Touch X* button. Notice that the *Touch X* button's LED is illuminated, 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. Inverted boring or internal threading tools should be touched off with negative X values.

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.

NOTE: *While X 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 being turned. 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.*



Tool	Description	X	Z	X Wear	Z Wear	Nose R	Tip
1	SDJCR RH Turn	-24.6066	-0.1148	0.0000	0.0000	0.0158	2
2	Part	-24.4895	-0.2287	0.0000	0.0000	0.0158	1
3		-24.0000	-14.0000	0.0000	0.0000	0.0158	7
4		0.0000	0.0000	0.0000	0.0000	0.0158	9
5		0.0000	0.0000	0.0000	0.0000	0.0158	9
6	1/4" Spot Drill	-25.3469	3.0692	0.0000	0.0000	0.0158	7
7	1/2" Drill	-25.3971	4.7226	0.0000	0.0000	0.0158	7
8		0.0000	0.0000	0.0000	0.0000	0.0158	9
9		0.0000	0.0000	0.0000	0.0000	0.0158	9
10		0.0000	0.0000	0.0000	0.0000	0.0158	9
11	CCMT RH Turn	-25.2257	-0.0981	0.0000	0.0000	0.0158	2
12	HSS Parting Blade	-23.7499	-0.1761	0.0000	0.0000	0.0158	1
13		0.0000	0.0000	0.0000	0.0000	0.0158	9
14		0.0000	0.0000	0.0000	0.0000	0.0158	9

Work	X	Z
G54	-0.017608	-13.799941
G55	0.000000	0.000000
G56	0.000000	0.000000
G57	0.000000	0.000000
G58	0.000000	0.000000
G59	0.000000	0.000000
G59.1	0.000000	0.000000
G59.2	0.000000	0.000000
G59.3	0.000000	0.000000

Figure 5.17

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 5.17**).

To alter a value in the tool offsets table, double-click on the field to edit. After entering the desired new value, click *Enter* on the keyboard to accept the value.

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

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 Tab

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.

The *Conversational* screens are divided into two sections: parameters common to most operations are displayed on the left; parameters (including part geometry) that are operation-specific are displayed on the right. DROs which are not used for a particular generator have gray text and 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 machine
- 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, tool clearance zone is often used as a landmark for path direction and feed/rapid rate changes – 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 with the *Append to File* button, 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. This means that 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 selects the work offset coordinate system used when the program runs.

NOTE: *The tool table and work offset need to be set up before the DROs 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 in the conversational programming mode, enter operational parameters for the specific part geometry on the appropriate tab: *OD Turn*, *ID Turn*, *Face*, *Chamfer*, *Groove/Part*, *Drill*, 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* dialog box opens on the screen.
2. If the appropriate file name is not displayed in the *Name* field, click the correct file and verify the name is now displayed.
3. Click *Save* or *Cancel*. If *Save* is selected, a message confirming the save is displayed.

Common DROs

Common DROs are on the left side of the programming screen (see **Figure 5.18**). These cover parameters that are most likely to be reused by operations to make a part. Some operations do not use a common DRO, in which case the DRO is grayed out.

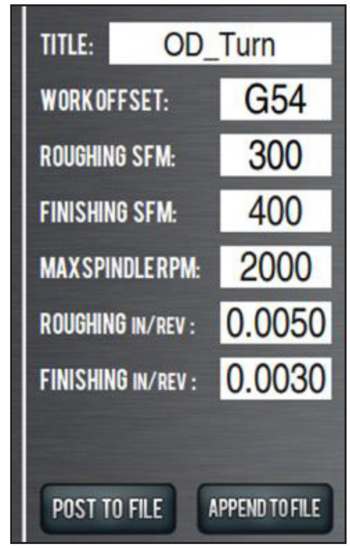


Figure 5.18

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. Available offsets:

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

Roughing SFM or MPM – 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 should be available to set an RPM rather than the constant surface speed input.

Finishing SFM or MPM – 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 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 should be available to set a simple spindle RPM.

Roughing (IN/Rev or MM/Rev) – 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 should be available to set a simple feed per minute.

Finishing (IN/Rev or MM/Rev) – 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 should be available to set a simple feed per minute.

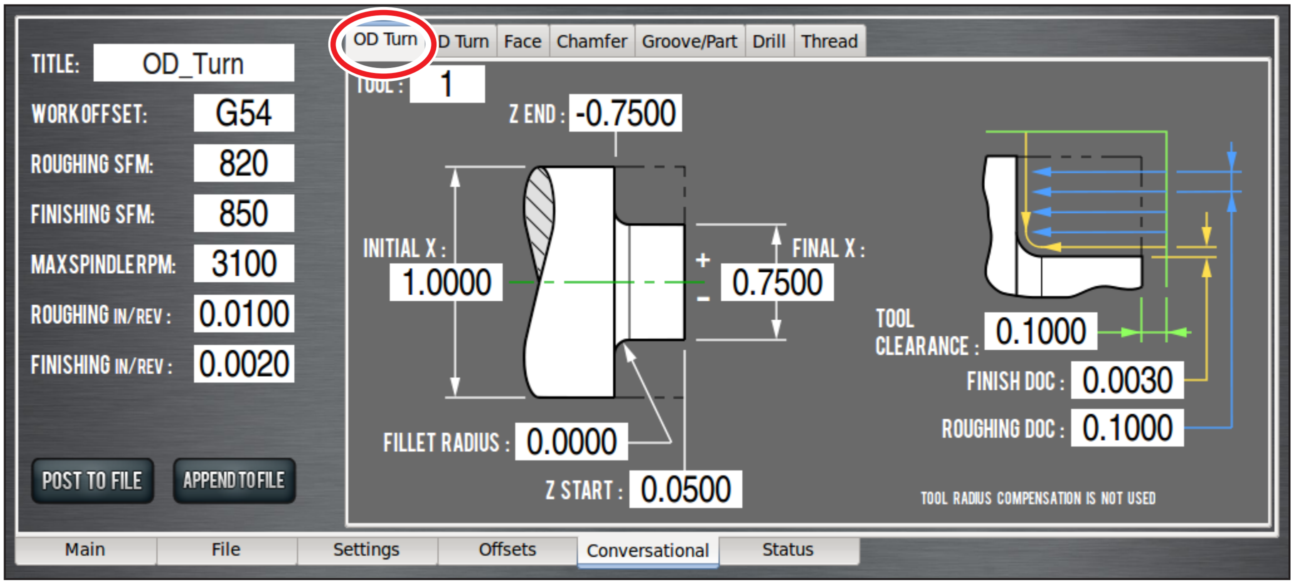


Figure 5.19

5.8.2 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 5.19** and **Figure 5.20**).

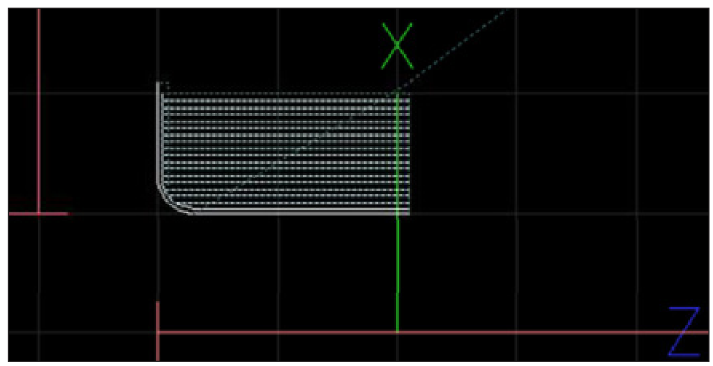


Figure 5.20

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.

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.

The finish pass is started at the +Z (tailstock) 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 type 3 tools.

Tool – 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.

Initial X – Is a reference value (the value should describe an existing feature), 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 INITIAL X and FINAL X 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. For the RapidTurn, the majority of these values will be positive.

Final X – Is a command value. It sets the location of the new outside diameter. For more information, refer to *INITIAL X* earlier in this section.

Z Start – 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. Z START should be larger than Z END.

Z End – Is a command value that sets the face location. Z END should be smaller than Z START.

Fillet Radius – 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.

Tool Clearance – 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. TOOL CLEARANCE is also sometimes used as a location for retracting the tool while making cutting passes.

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

Finish DOC – Is a command value used to leave enough material after roughing to do one finish pass. Valid values are positive.

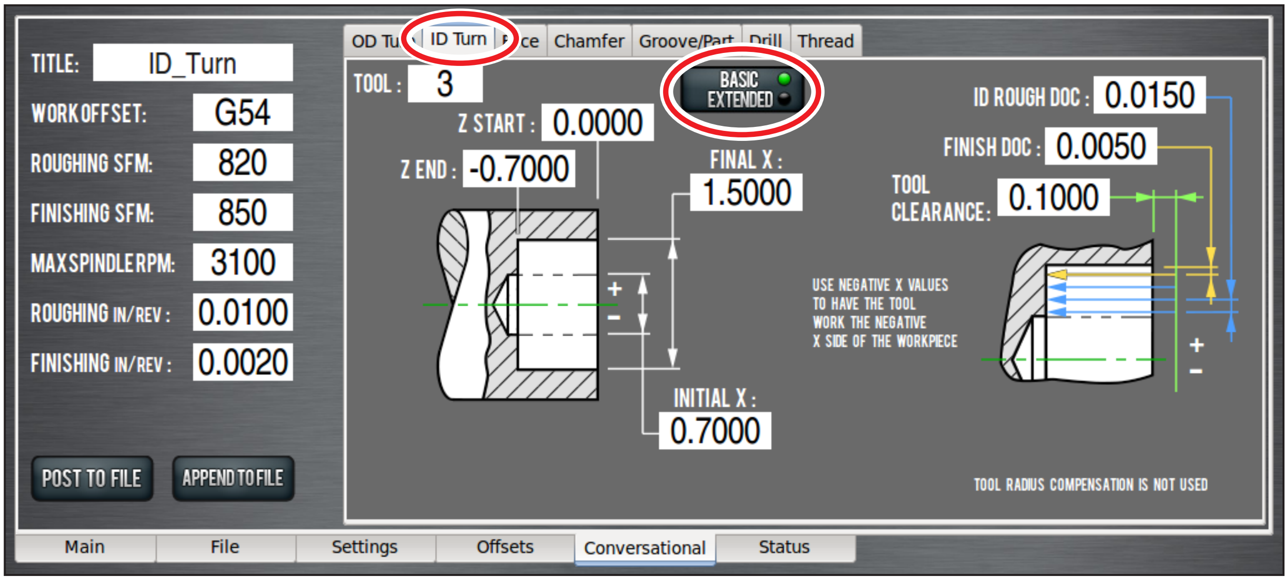


Figure 5.21

5.8.3 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 5.21**). A button is provided to switch to *Extended* (see **Figure 5.22**). 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.

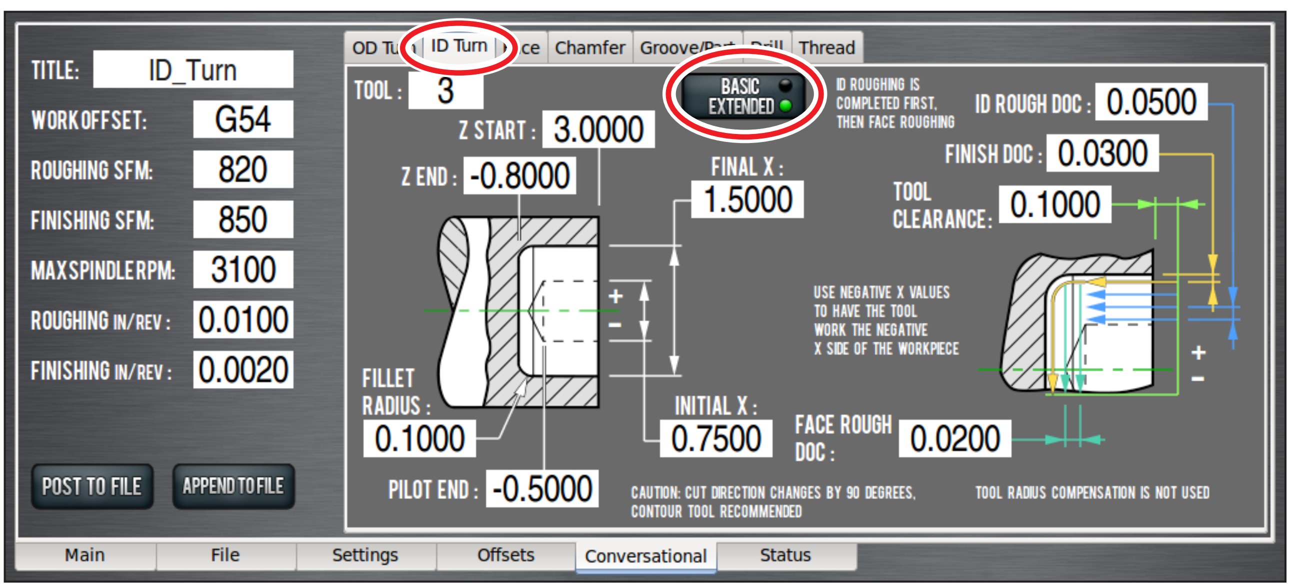


Figure 5.22

ID Turn (Basic) Tool Path

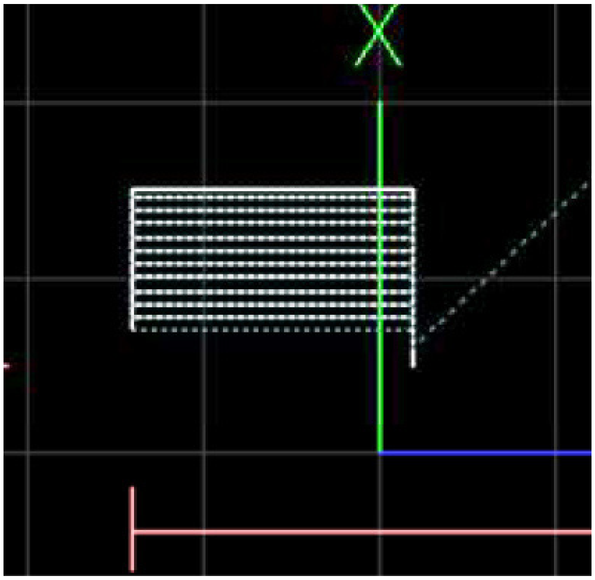


Figure 5.23

ID Turn (Extended) Tool Path

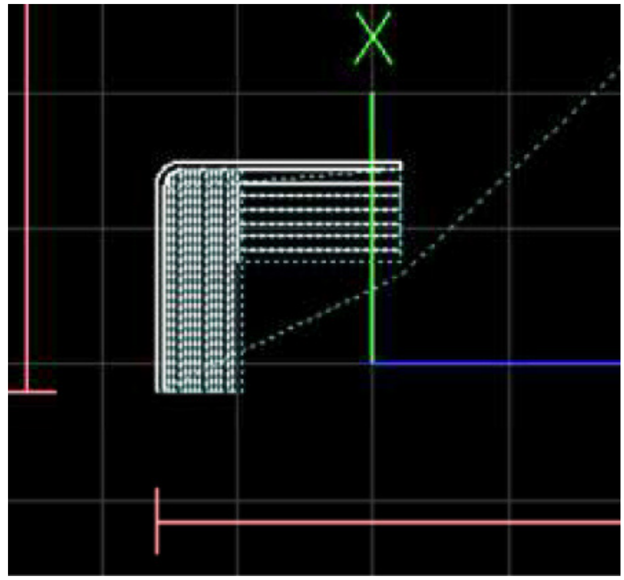


Figure 5.24

Roughing starts at the pilot hole diameter, X START, and incrementally cuts diameters with an adjusted DOC (for more 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 is provided.

For each face pass, the tool tip cuts to the hole center + TOOL CLEARANCE which requires a rough hole diameter (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.

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 when clicking 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.

Tool – Is a command value that sets the tool number for a tool change at start of the program.

Initial X – Is 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.

Final X – 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.

Z Start – Is a reference value for the location of the stock face. It is used with TOOL CLEARANCE to set the transition between rapid and feed rates on some moves.

Z End – Is a command value for the finished face location.

Fillet Radius – 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.

Tool Clearance – 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.

ID Rough DOC – 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.

Face Rough DOC – 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.

Finish DOC – 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.

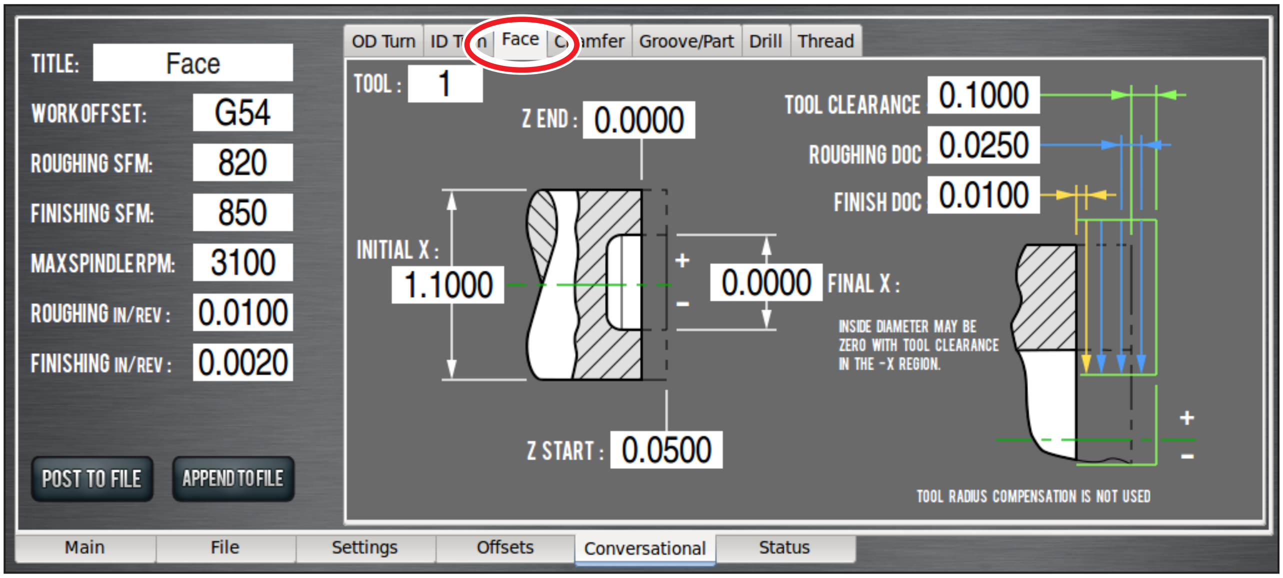


Figure 5.25

5.8.4 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 5.25**). When using the RapidTurn both INITIAL X and FINAL X need to be positive. 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 FINAL X 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.

Tool – 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.

Facing Tool Path

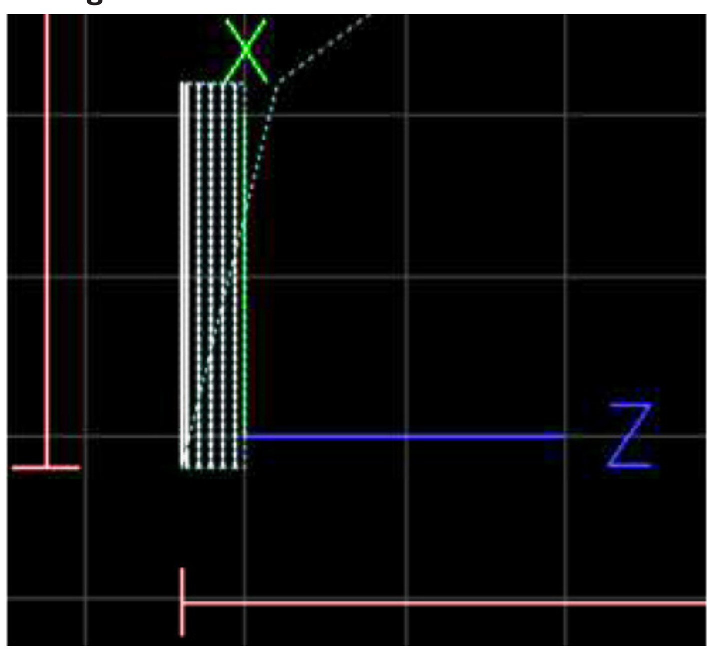


Figure 5.26

Initial X – 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 INITIAL X and FINAL X are both positive, the tool works on the positive X side of spindle center. A positive and negative value for each will produce an error.

Final X – 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.

Z Start – 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.

Z End – Is a command value for the finished face location.

Tool Clearance – 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.

Roughing DOC – 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.

Finish DOC – Is a command value that is used to leave enough material after roughing to do one finish pass. Valid values are positive.

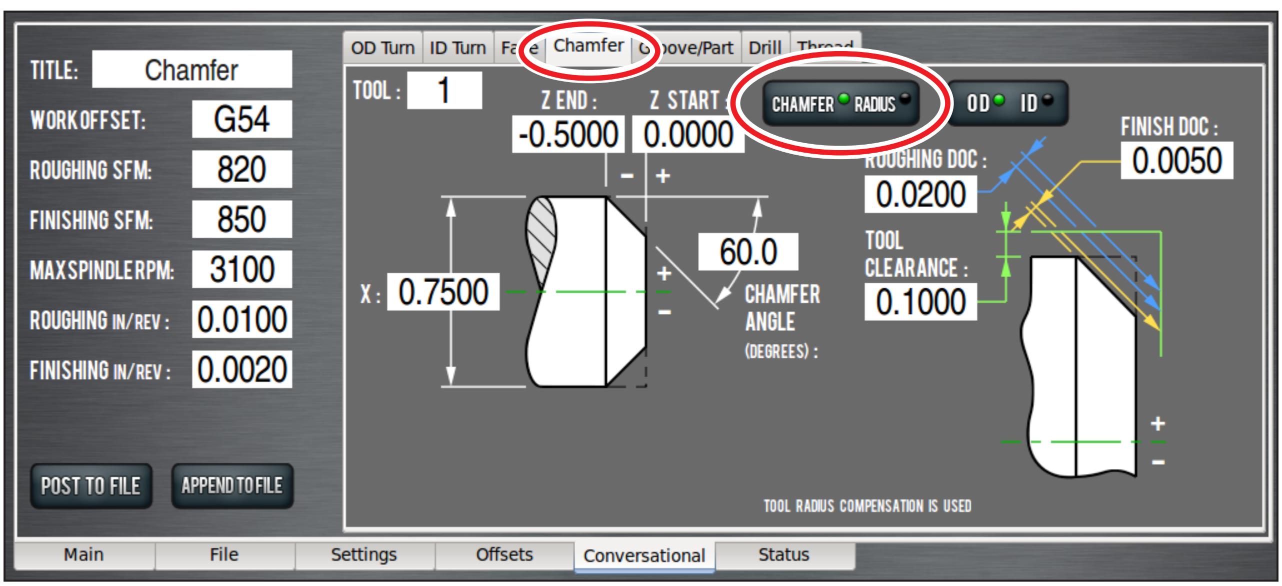


Figure 5.27

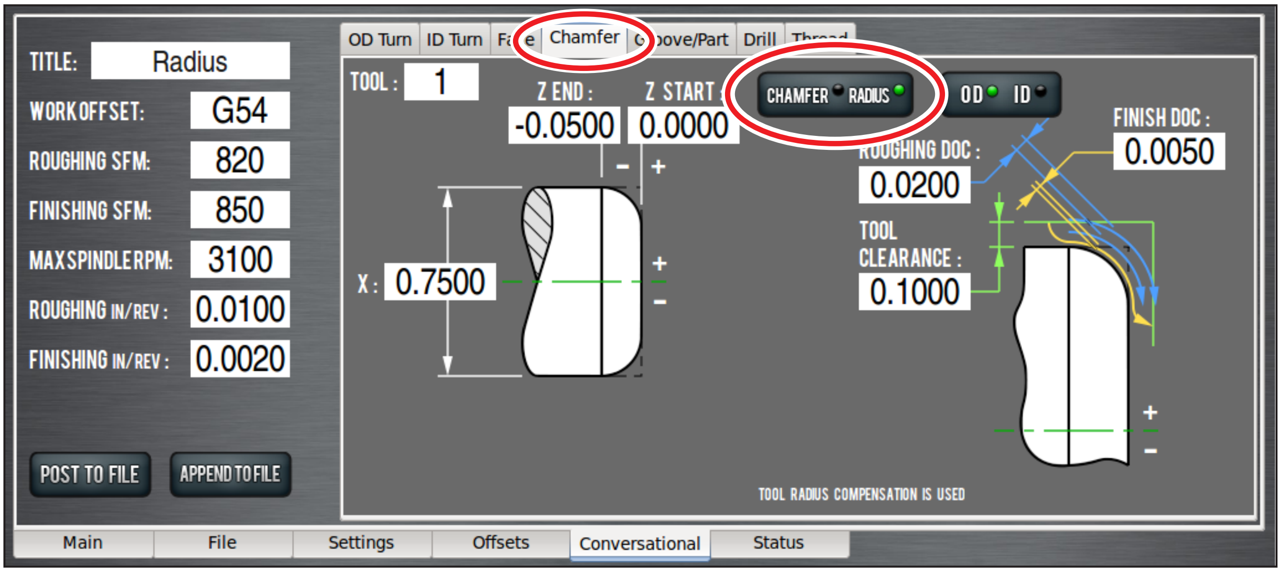


Figure 5.28

5.8.5 Chamfer and Radius Tab

Chamfer/radius creates G-code for cutting a chamfer, taper, or corner radius (see **Figure 5.27**). 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 entries, the chamfer or radius feature may face away from or toward the spindle. Initially, the *Chamfer* screen is presented, but it may be switched to the *Radius* screen (see **Figure 5.28**) by toggling the *Chamfer/Radius* button.

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.

Tool: – 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.

X – 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 rates. If INITIAL X is positive, the tool works on the positive X side of the spindle center.

Chamfer on Bar End Tool Path

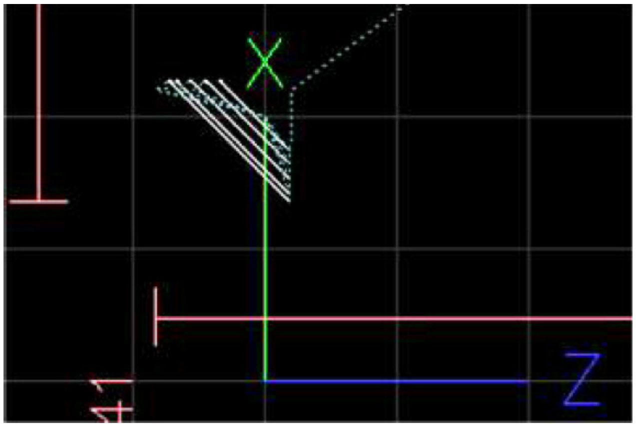


Figure 5.29

Radius on Bar End Tool Path

Figure 5.30

Z Start – 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.

Z End – 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.

Chamfer Angle – 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).

Tool Clearance – 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.

Roughing DOC – 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.

Finish DOC – Is a command value that is used to leave enough material after roughing to do one finish pass. Valid values are positive.

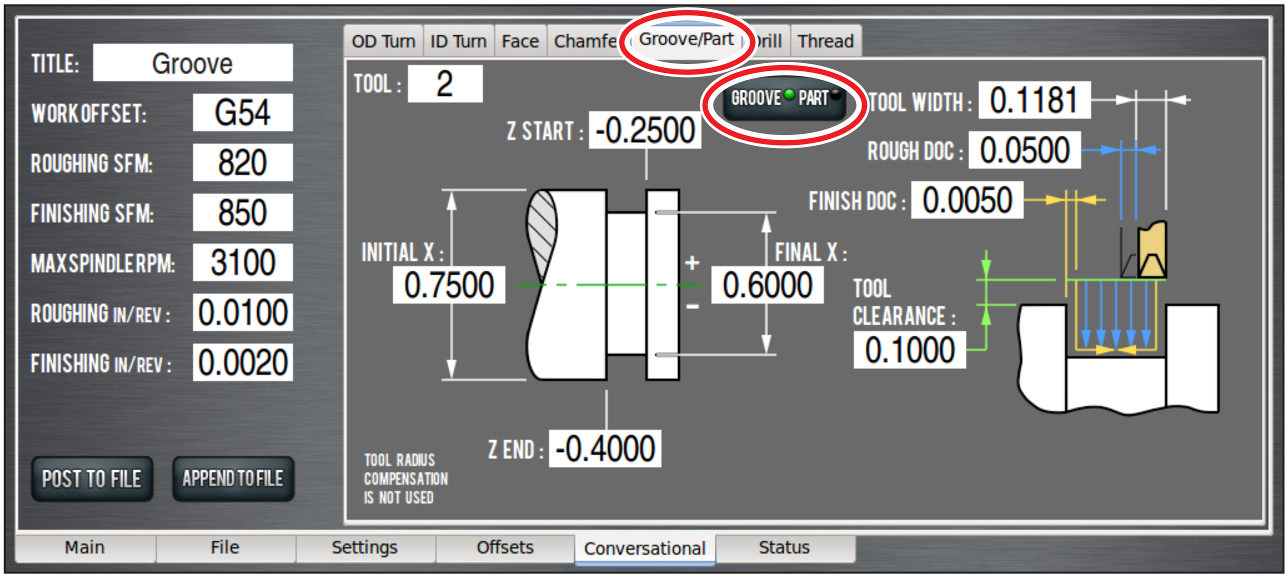


Figure 5.31

5.8.6 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 5.31**), but can be switched to *Part* (see **Figure 5.32**) with the *Groove/Part* button. Grooving paths are based on Z START and Z END values. If Z START is greater than Z END, the tool's control point is set to the +Z side of the tool. If Z START is less than Z END, the control point is set to the -Z side of the tool (see **Figure 5.33**). 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).

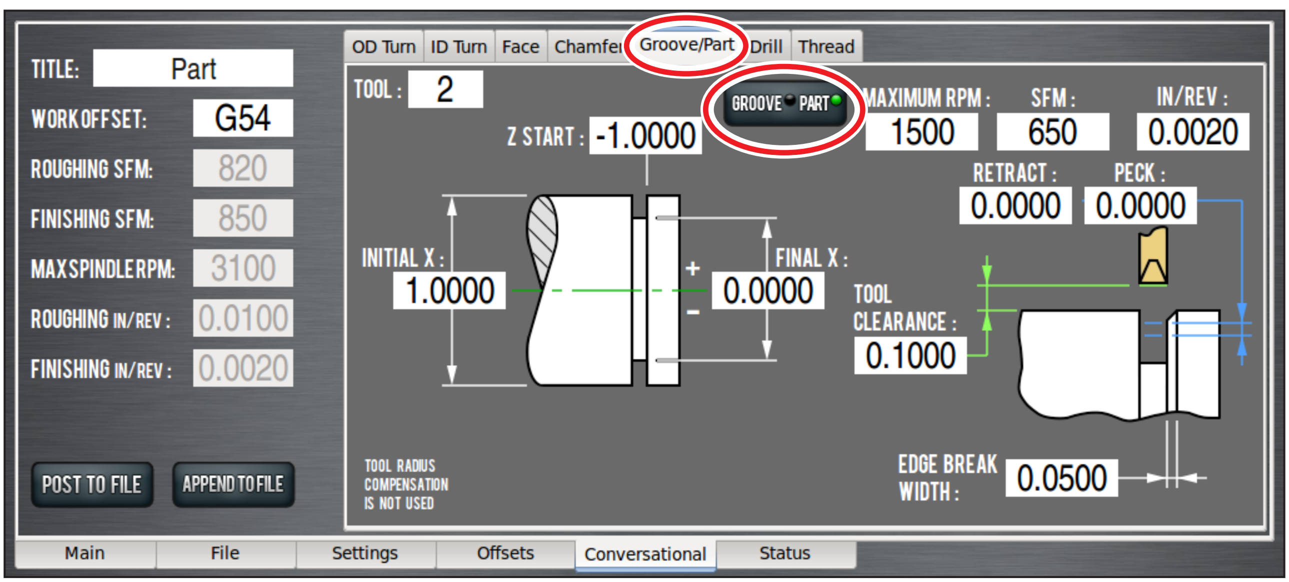


Figure 5.32

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

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

START is less than Z END, the control point is set to the -Z side of the tool (see **Figure 5.33**).

Tool – 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.

Initial X – 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 INITIAL X and FINAL X are both positive, the tool will work on the positive X side of the spindle center. A positive and negative value for each produces an error.

Final X – Is a command value. It sets the diameter of the new groove bottom, or the end of parting. For more information, refer to *INITIAL X* earlier in this section.

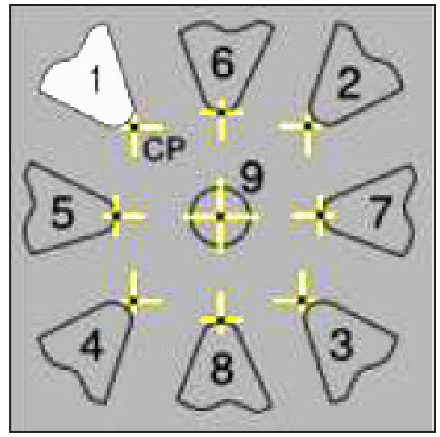


Figure 5.33

Groove Tool Path

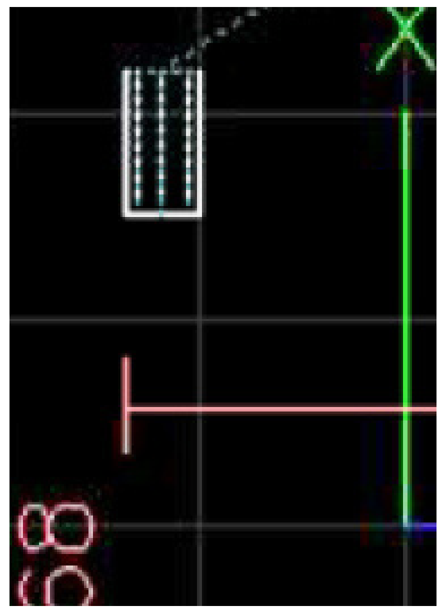


Figure 5.34

Z Start – 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.

Z End – Is a command value for setting the location of the groove end. Value is not used with part.

Tool Width – Is a reference value for the groove or parting tool width in the Z direction. This value is added or subtracted from Z END to locate the grooving tool within the far end of the groove width. This value is not used with part.

Rough DOC – 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.

Finish DOC – 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.

Tool Clearance – 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.

SFM / MPM – Is a command value for controlling spindle speed. This DRO is similar to the CSS DROs provided in the common DRO panel to the left. Since parting commonly has an SFM value different from other operations, a CSS DRO is provided in the parting screen to keep it handy for the *Part* screen. This value is not used with groove.

IN/REV, MM/REV – Is a command value for controlling parting feed rate. This DRO is similar to the feed/rev DROs provided in the common DRO panel (left side of screen). Since parting commonly has a feed/rev value different from other operations, a feed/rev DRO is provided in the parting screen to keep it handy for the *Part* screen. This value is not used with groove.

5.8.7 Drill Tab

The *Drill* tab creates G-code to drill a hole (see **Figure 5.33**). Drill uses 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 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. Verify the pecking retract motion meets requirements for your application.

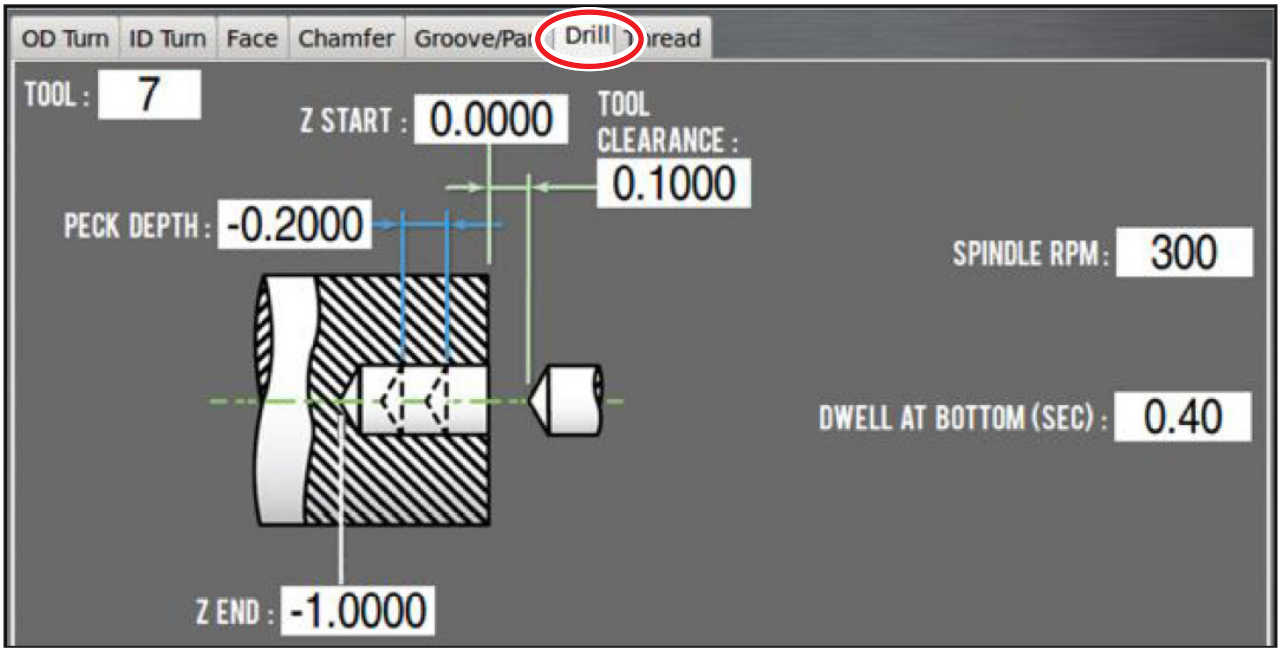


Figure 5.35

Tool – Is a command value that sets the tool number for a tool change at start of the program.

Z Start – 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. An error is produced if Z START is not larger than Z END.

Z End – Is a command value. For drill, Z END 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 Z END.

Peck Depth – 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 value is adjusted to fit an integer number of pecks within the hole depth. PECK DEPTH also needs a direction, so should have a negative value.

Tool Clearance – Is a reference value that should match the desired space needed for tool retracting and transitions between rapid and cutting feed rate.

Spindle RPM – Is a command value for the RPM (G97) desired for drilling.

Dwell at Bottom – 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.

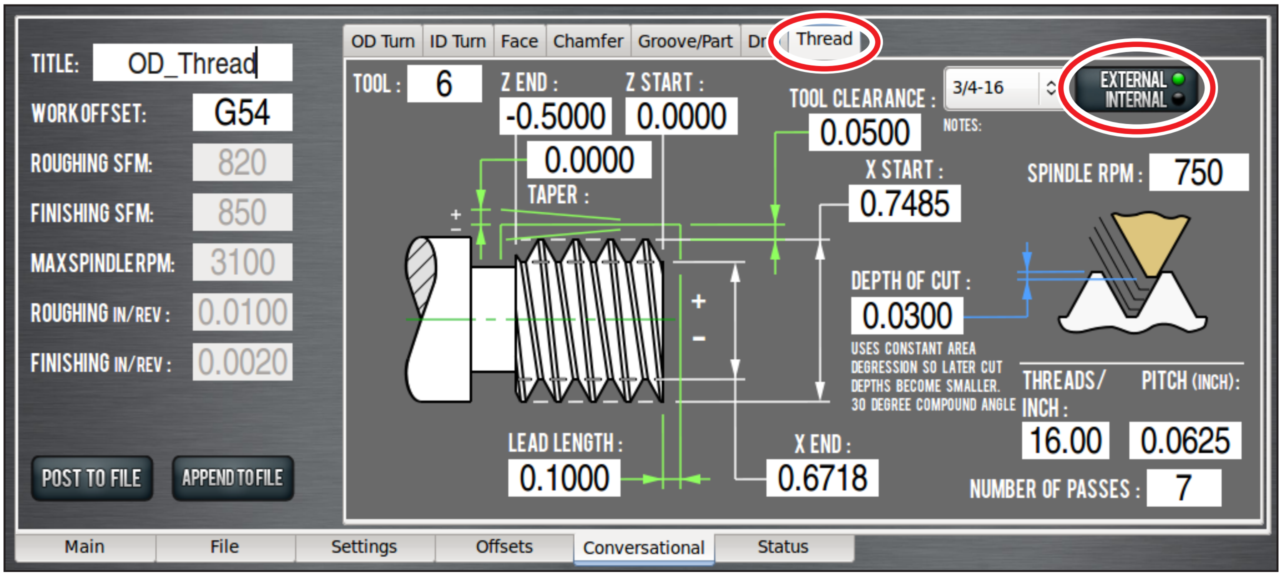


Figure 5.36

5.8.8 Thread Tab

The *Thread* tab creates G-code to single point an external or internal thread on an existing OD (**Figure 5.36**) or ID (**Figure 5.37**). Initially, the screen is in external mode, but can be switched to internal mode with the *External/Internal* toggle button.

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.

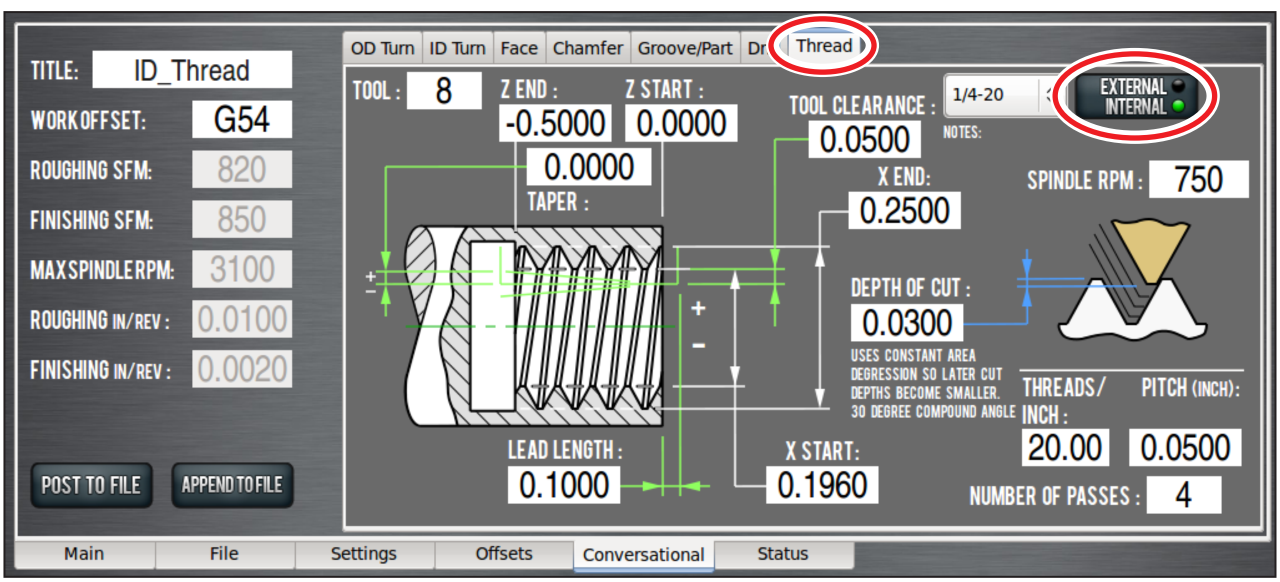


Figure 5.37

Tool – A command value that sets the tool number for a tool change at the start of a program.

X Start – 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 threads.

X End – Is a command value. It sets the location of the new outside diameter. For more information, refer to *INITIAL X* section earlier in this chapter.

Z Start – 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. Z START should be larger than Z END.

Z End – A command value that sets the face location. Z END should be smaller than Z START.

Tool Clearance – 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.

Lead Length – 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 index sensor 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 motion. This value needs to be a compromise between spindle speed, thread pitch, and workpiece clearance.

Thread Table – 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 DROs. 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.*

Spindle RPM – Is a command value that sets the spindle RPM as opposed to the CSS DRO in the common DRO section.

Depth of Cut – 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.

Threads/IN, MM – Is a command value that sets the number of threads per inch or millimeter. This DRO is coupled to PITCH, so whichever value is handy – threads per unit or pitch – may be entered.

Pitch – Is a command value that sets the distance per thread. This DRO is coupled to THREADS/IN, MM, so whichever value is handy (threads per unit or pitch) may be entered.

Number of Passes – This DRO is coupled to DEPTH OF CUT. A value may be entered into either DRO 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 can be adjusted to reach a compromise between the depth of cut and the number of passes.

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 5.38**).

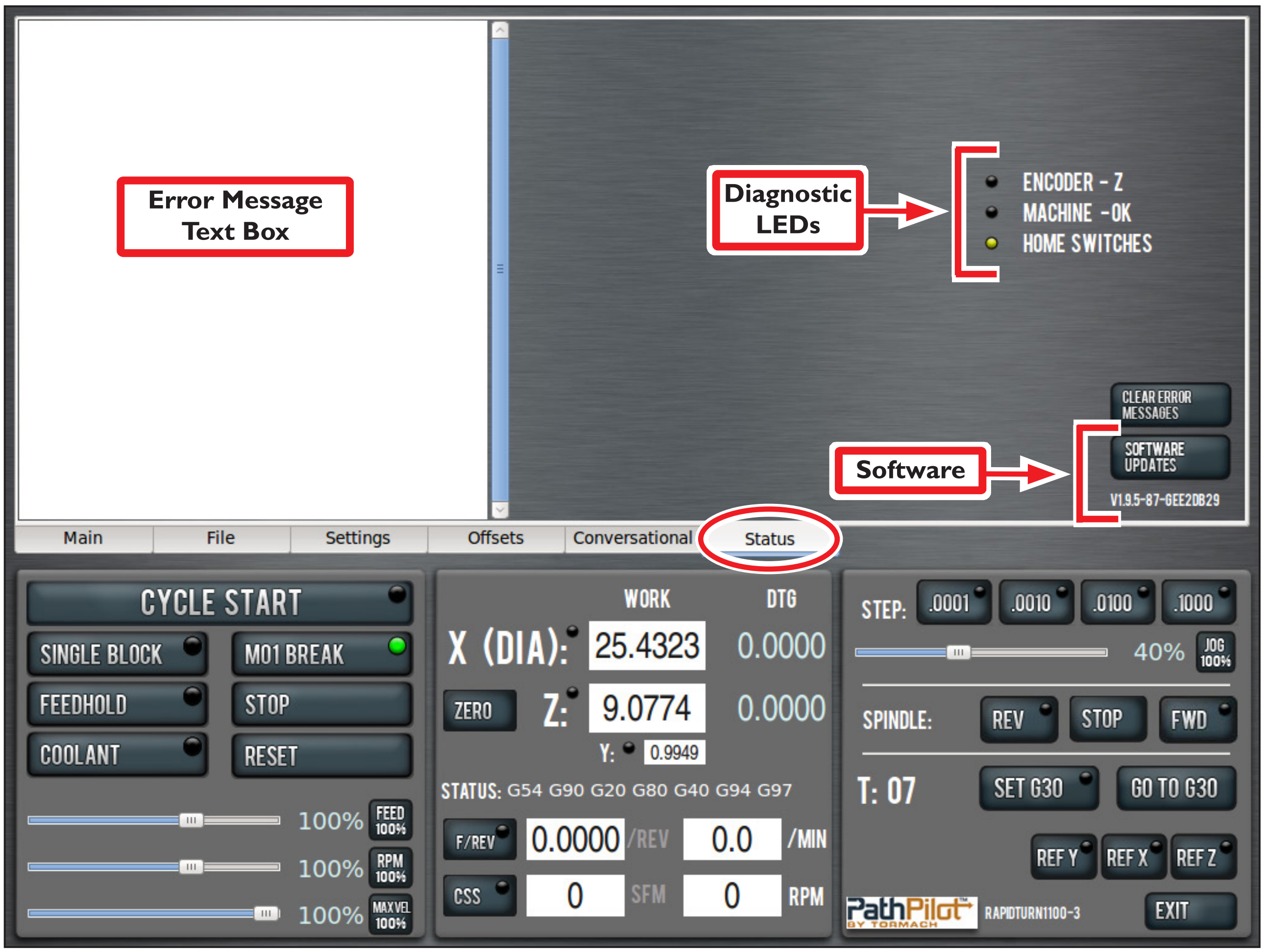


Figure 5.38

Error Messages – The left hand side of this screen consists of a scrolled text box that displays any error messages that occur. For instance, attempting to enter an MDI command that moves the axes before referencing the machine generates an error message (see **Figure 5.39**).

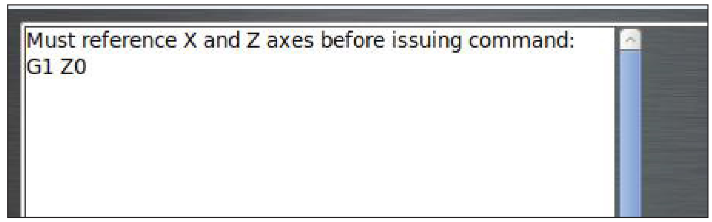


Figure 5.39

There are three levels of error messages:

- 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.
- 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.
- 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.

***NOTE:** When the Status tab turns red (indicating a minor error), hover over the Status tab with the mouse to view the error message as shown in **Figure 5.40**.*

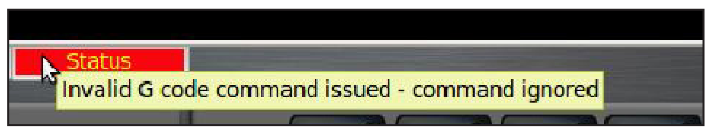


Figure 5.40

Pressing the *F1* key temporarily switches from the current screen to the *Status* screen, allowing the operator to quickly scan the error message. Releasing the *F1* key returns PathPilot to the previous screen. A history of error messages is displayed in the scrolled Error Message Text Box (see **Figure 5.39**) 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 (see **Figure 5.39**) used for troubleshooting purposes that should not be needed in normal operation.

Tool Orientation Errors – If information is entered on a *Conversational* screen that does not match information in the tool table – for instance, positive diameters are entered but an inverted boring bar is selected – the *Status* screen displays error message in the text box as well as images of the expected tool orientation and the actual tool orientation.

***NOTE:** The current version of PathPilot is listed in the lower right corner of the screen (see **Figure 5.38**). For software update instructions and revision information, refer to Tormach service bulletin SB0046: Release Notes for PathPilot.*

6. Programming

This section defines the language (G-codes, etc.) that is understood and interpreted by the PathPilot® operating system, and 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

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 toward the right-hand side of the machine. The X-axis distance values decrease as the tool moves up and the mill's head rises. 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 an illustration of 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. Due to the orientation of the RapidTurn's tool post, it is orientation 2.

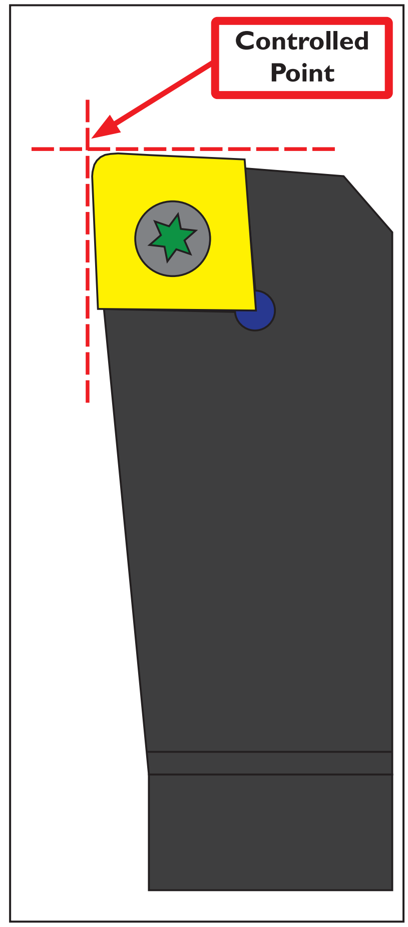


Figure 6.1

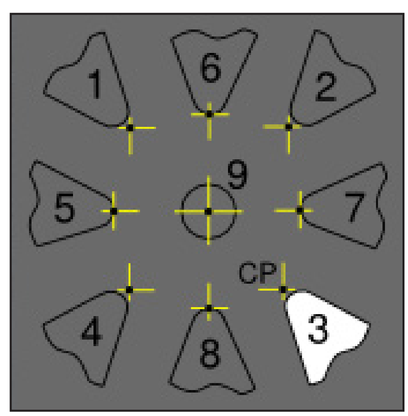


Figure 6.2

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.

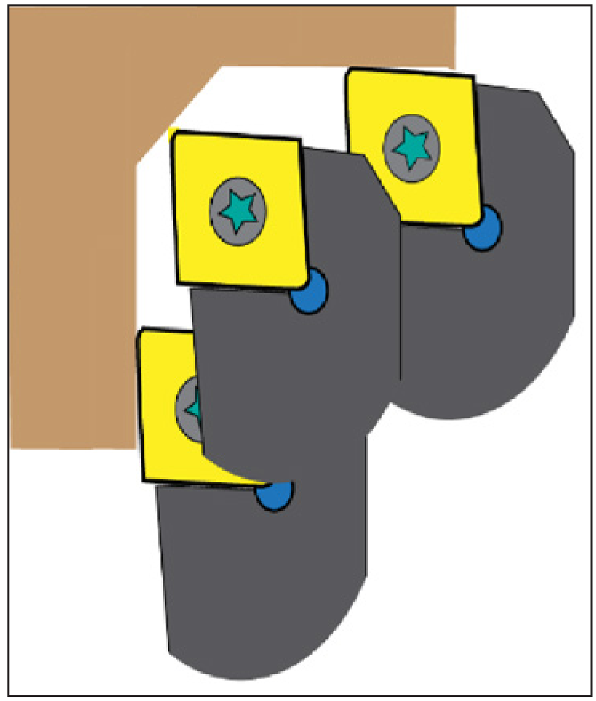


Figure 6.3

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 mill's 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 the 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 are 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.

6.4 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.

6.4.1 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.

6.4.2 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.*

6.4.3 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.

6.4.4 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.

6.4.5 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.

6.4.6 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.

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 X offset in G87 canned cycle
J	Y-axis offset for arcs Y offset in G87 canned cycle
K	Z-axis offset for arcs Z offset in G87 canned cycle
L	Number of repetitions in canned cycles/subroutines key used with G10
M	Miscellaneous function (see *Modal Groups* table)
N	Line number
O	Subroutine label number
P	Dwell time in canned cycles dwell time with G04 key used with G10 tapping depth in M871-M874
Q	Feed increment in G83 canned cycle repetitions of subroutine call
R	Arc radius 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

6.4.7 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.

6.4.8 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.

6.5 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, G73, 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-M199}
For non-modal G-codes	Group 0	= {G4, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}

6.6 G-codes

G-codes of the PathPilot input language are detailed in the *G-code Table* later in this section, and are described in more detail in this section. The descriptions contain command prototypes, set in `Courier` type. In the command prototypes, the tilde symbol (~) stands for a real value. As described in detail elsewhere, 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 prototypes not explicitly described as optional are required. It is an error if a required item is omitted.

In the prototypes, 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 prototype, 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.

6.6.1 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 *Cutter Compensation – G41 and G42* later in this chapter. If G53 is programmed on the same line, the motion also differs; for more information, refer to section *G53 – Move in 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.

6.6.2 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 *Cutter Compensation – G41 and G42* later in this chapter. If G53 is programmed on the same line, the motion also differs; for more information, refer to section *G53 – Move in Absolute Coordinates* later in this chapter.

6.6.3 Arc at Feed Rate – G02 and 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 *Cutter Compensation – G41 and G42* 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.

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	G19 only for lathe
G20/G21	Inch/millimeter unit
G28	Return home
G28.1	Reference axes
G30	Return home
G33	Spindle sync. motion (i.e., threading)

(continued on next page...)

G-code Table (...continued)

G-code	Summary
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
G73	Canned cycle - peck drilling
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.
G93, G94, G95	Feed modes
G96, G97	CSS, RPM modes
G98	Initial level return/R-point level after canned cycles

6.6.3.1 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

6.6.4 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)
```

6.6.5 Lathe Diameter Mode – G07

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

6.6.6 Set Offsets – G10

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

6.6.7 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**.

6.6.8 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

6.6.9 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

6.6.10 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

6.6.11 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

6.6.12 Plane Selection – G17, G18, and G19

The machine should always be operated in the XZ-plane. Do not use G17 or G19.

6.6.13 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.

6.6.14 Return to Pre-defined Position – G28 and 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

6.6.15 Return to Pre-defined Position – G30 and G30.1

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

- `G30` makes a rapid traverse move from the current position to the absolute position of the value in parameter 5183.
- `G30 Z~` makes a rapid traverse move to the Z position (including any offsets), then makes a rapid traverse move to the absolute position of the value in parameter 5183.
- `G30.1` stores the current absolute position into parameter 5183.

It is an error if:

- Cutter Compensation is turned on

6.6.16 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~ Z~`*. 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.

6.6.17 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

6.6.18 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

6.6.19 Cutter Compensation – G41 and 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 RapidTurn 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

6.6.20 Dynamic Cutter Compensation – G41.1 and 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

6.6.21 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.

6.6.22 Select Work Offset Coordinate System – G54-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 *Coordinate Systems* section earlier in this chapter.

6.6.23 Set Path Control Mode – G61 and 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.

6.6.24 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.

6.6.25 Canned Cycle – High Speed Peck Drill – G73

The G73 cycle is intended for deep drilling with chip breaking. The retracts in this cycle break the chip but do not totally retract the drill from the hole. It is suitable for tools with long flutes which clear the broken chips from the hole. This cycle takes a Q number which represents a delta increment along the Z-axis. Program `G73 X~ Z~ R~ L~ Q~`

R – Retract position along the Z-axis

Q – Delta increment along the Z-axis

L – Repeat

Preliminary motion: If the current Z position is to the left of the R position, the Z-axis is traversed to the R position. The X coordinate must be zero.

Move the Z-axis only at the current feed rate toward the spindle by delta or to the Z position, whichever is less deep.

Rapid up retract.

Repeat above move-Z and rapid steps until the Z position is reached.

Retract the Z-axis at traverse rate to R.

It is an error if:

- The Q number is negative or zero
- The R number is not specified

6.6.26 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

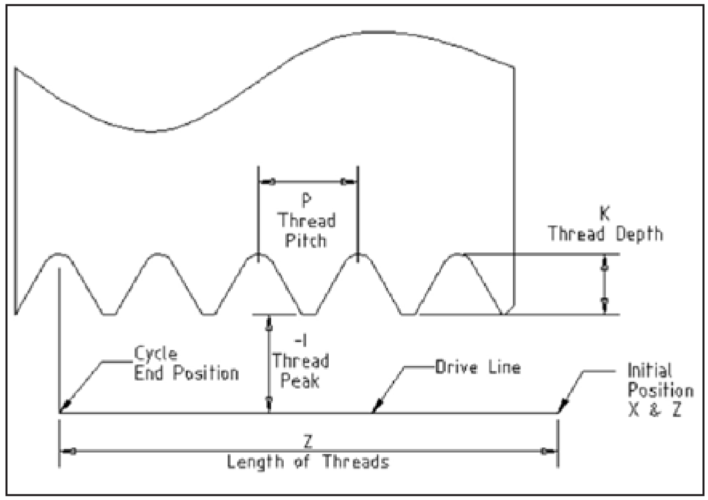


Figure 6.4

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, and 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; 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: *The signs of I, J, and K values assume a standard quick change tool post configuration. If reversing standard tooling, reverse the sign of I, J, and K.*

6.6.26.1 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 *Spindle Synchronized Motion – G33* section earlier in this chapter.

6.6.27 Distance Mode – G90 and G91

Interpretation of 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.

6.6.28 Arc Distance Mode – G90.1 and 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.

6.6.29 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 section *Coordinate System* earlier in this chapter.

6.6.30 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

6.6.31 Spindle Control Mode – G96 and 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.

6.6.32 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

6.7 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	User defined M-codes

6.7.1 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.

6.7.2 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.

6.7.3 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.

6.7.4 Override Control – M48 and 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.

6.7.5 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).

6.7.6 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 (for more information, refer to *Spindle Speed* section).

6.7.7 Set Current Tool Number – M61

`M61 Q~` – Change the current tool number while in MDI or manual mode.

It is an error if:

- Q~ is not 0 or greater.

6.7.8 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.

6.7.9 Return from Subroutine – M99

To return from a subroutine, program `M99`. Execution continues after the M98 G-code which called the subroutine.

6.8 Other Input Codes

6.8.1 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).

6.8.2 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

6.8.3 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

While waiting for a tool change, the machine stops and flashes both the display of the required tool number and the LED on the *Cycle Start* button. The program continues by 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. 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

6.9 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.

6.10 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

6.11 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.

6.12 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 the PathPilot interface or by most CAM systems.*

6.12.1 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 the controller 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 the controller 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 RapidTurn 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)

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.

6.12.2 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 shown in **Figure 6.12**.

Arguments to unary operations which take angle measures (COS, SIN, and TAN) are in degrees. Values returned by unary operations which return angle measures (ACOS, ASIN, and ATAN) are also in degrees.

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.

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

6.13 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.

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. Troubleshooting

Common issues and possible solutions specific to the RapidTurn™ are detailed in this chapter. For information on general techniques and mill-specific troubleshooting procedures, refer to chapter 10, *Troubleshooting*, in the mill operator manual.

Table 1
RapidTurn Spindle Does Not Turn

Possible Cause	Probability	Action to Identify Cause of Problem	Discussion
Spindle motor not connected	High	Check RapidTurn motor cable is connected to Quick Change Motor Connection Kit's junction box	—
Spindle control switch on mill's operator panel set to *Manual*	Medium	Check switch; set to *Auto* if necessary	—

Table 2
RapidTurn Spindle Turns in Reverse

Possible Cause	Probability	Action to Identify Cause of Problem	Discussion
Tooling installed in reverse direction	Medium	Check tool's installed direction	Inserts face away from operator when installed on the reverse-action quick change tool post
Wiring reversed	Low	Check direction of RapidTurn spindle (refer to chapter 3, *Operation*, for more information). Switch to mill's PathPilot® interface and check direction of mill spindle.	If wires were reversed when motor connection kit was installed, one or both spindles may rotate in wrong direction. If RapidTurn and mill spindles are both reversed, swap wires U and V from VFD in junction box of Quick Change Motor Connection Kit. If only RapidTurn spindle is reversed, swap wires U and V inside junction box of RapidTurn spindle motor. If only mill spindle is reversed, swap wires U and V inside junction box of mill spindle motor.

Table 3
RapidTurn Spindle Turns at Incorrect Speed

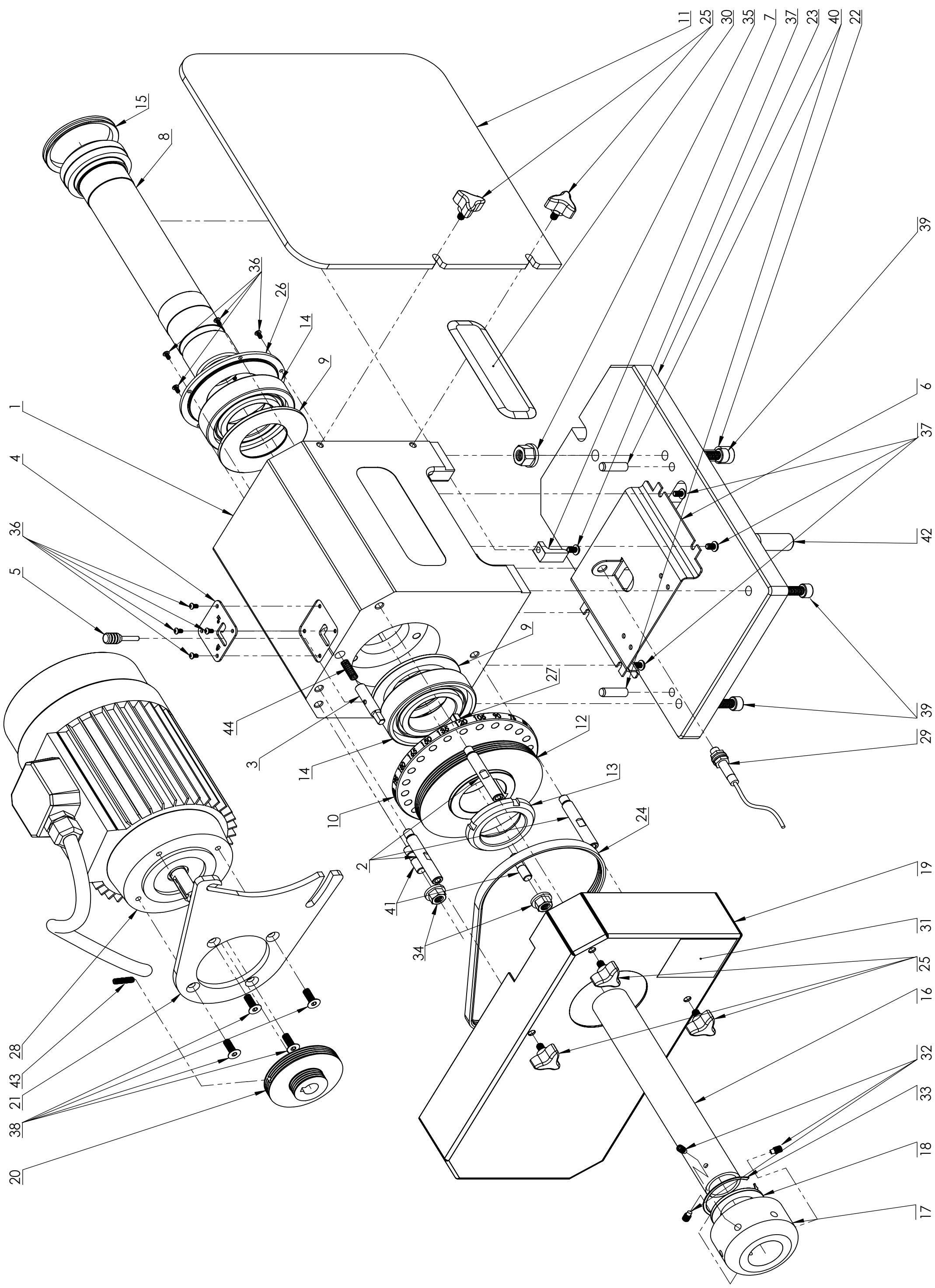
Possible Cause	Probability	Action to Identify Cause of Problem	Discussion
Incorrect belt position for PathPilot configuration	High	Check high/low spindle speed selection on *Settings* tab in PathPilot interface	—
Incorrect VFD program (PCNC 1100 only)	Medium	Verify RapidTurn VFD programming stick is inserted into VFD's programming slot	—

Table 4
RapidTurn Pauses During Operation and Will Not Feed

Possible Cause	Probability	Action to Identify Cause of Problem	Discussion
Spindle speed sensor cable not connected	High	Check spindle speed sensor cable is connected to mill's *Accessory* port	Verify spindle speed sensor function by connecting sensor, completing a full rotation of RapidTurn's spindle, and observing *Encoder Z* LED on *Status* tab in PathPilot interface. LED should illuminate once per spindle rotation.
Spindle speed signal not getting to PathPilot	Low	Inspect connections and wiring	Verify spindle speed sensor function by connecting sensor, completing a full rotation of RapidTurn's spindle, and observing *Encoder Z* LED on *Status* tab in PathPilot interface. LED should illuminate once per spindle rotation.

8. Diagrams and Parts Lists

8.1 RapidTurn™ Assembly (exploded view)



RapidTurn Assembly Parts List

ID	PN	Description	ID	PN	Description
1	32902	Headstock	23	32930	Base Plate
2	32903	Stand Off	24	32931	Poly-Vee Belt[1]
3	32904	Index Pin	25	32932	3-wing Knob
4	32905	Spindle Lock Plate	26	33268	Front Bearing Cover
5	32906	Spindle Lock Lever	27	33304	Square Key M6 x 32 mm
6	32908	Sensor Bracket	28	31431	RapidTurn Spindle Motor
7	32909	Spindle Speed Sensor Flag	29	37261	Spindle Speed Sensor Assembly
8	32910	RapidTurn Spindle	30	34927	RapidTurn Decal
9	32912	Spindle Spacer Inner	31	35932	RapidTurn Safety Sign
10	32913	Degree Wheel	32	37183	M6 x 10 mm Dog Point Set Screw
11	32914	Chip Guard	33	37184	Retaining Ring
12	32915	Spindle Pulley	34	37185	Flange Nut M10
13	32916	Bearing Nut	35	37186	Flange Nut M8
14	32917	Angular Contact Bearing	36	37187	M3 x 6 mm Button Head Cap Screw
15	32918	V-Ring Seal	37	37188	M5 x 8 mm Button Head Cap Screw
16	32919	Draw Tube	38	37189	M6 x 18 mm Flat Head Cap Screw
17	32920	Hand Wheel	39	37190	M8 x 22 mm Socket Head Cap Screw
18	32922	Thrust Bearing	40	37191	Dowel Pin M8 x 28 mm
19	32923	Belt Guard	41	37192	Threaded Stud M8 x 30 mm
20	32925	Motor Pulley	42	37193	Dowel Pin 5/8" x 1-1/2"
21	32927	Motor Bracket	43	37194	M5 x 20 mm Cup Point Set Screw
22	32928	Eccentric Pin	44	37195	Spindle Lock Pin Compression Spring

[1]Gates® 230J Micro-V®

1071 Uniek Drive
Waunakee, WI 53597 USA
Ph: 608.849.8381 / Fax: 209.885.4534
Email: info@tormach.com

UM10393_RapidTurn_Manual_1217A