ACU-RITE

TURNPWR

User's Manual

Controls

Motion control keys

Key	Function
GO	**GO** key (e.g. run a program).
STOP	**STOP** key (dual function: press once to pause, press twice to stop a program).

Data entry keys

Key	Function
CANCEL	**CANCEL** key cancels operation, i.e. form.
CLEAR	**CLEAR** key clears selections, i.e. values in a field, a program step.
USE	**USE** key completes operation, i.e. data entered in a form.
ENTER	**ENTER** key completes selection, i.e. values entered in a field.

Function keys

Key	Function
ABS INCR	**ABS/INC** key toggles between Absolute or Incremental positioning.
DRO PGM	**DRO/PGM** key toggles between the DRO display or Program mode display.
VIEW	**VIEW** key opens a form to configure graphics parameters; i.e. 2D or 3D.
INFO	**INFO** key opens on-screen manual.
MM	**MM** key toggles between INCH or MM mode.
SETUP	**SETUP** key opens configuration menu.

Numeric keys

Key	Function
0	**Zero** key.
1	**One** key.
2	**Two** key.
3	**Three** key.
4	**Four** key.
5	**Five** key.
6	**Six** key.
7	**Seven** key.
8	**Eight** key.
9	**Nine** key.
•	**Decimal** key.
+/–	**Plus/Minus** key.

Axis keys

Key	Function
	Axis keys open the datum, or preset form.

Soft keys

Key	Function
	Soft Keys perform the function directly above them.

Move tool and navigation keys

Key	Function
◄	**Left Arrow** key will move the tool or display cursor depending on the function selected.
►	**Right Arrow** key will move the tool or display cursor depending on the function selected.
▲	**Up Arrow** key will move the tool or display cursor depending on the function selected.
▼	**Down Arrow** key will move the tool or display cursor depending on the function selected.

Calculator function keys

Key	Function
CALC	**CALC** key opens the calculator.
+	**Plus** key.
−	**Minus** key.
×	**Multiply** key.
÷	**Divide** key.

Turning function keys

Key	Function
TOOL	**TOOL** key opens the **Set Tool** data input dialogue.
THREAD	**THREAD** key opens the **Thread Cycles** popup menu.
RECESS	**RECESS** key opens the **Recess Cycles** popup menu.
TURN	**TURN** key opens the **Turning Cycles** popup menu.
POS	**POS** key opens the **Position / Drill** data input dialogue.
LINE	**LINE** key opens the **Line** data input dialogue.
ARC	**ARC** key opens the **Arc** data input dialogue.
BLEND	**BLEND** key opens the **Blend \ Chamfer** data input dialogue.
	Blank key opens the user defined turning function data input dialogue.

Potentiometers for override

Feed rate override   **Spindle speed override**

⌁ % FEED   ↻ % SPINDLE

Peripherals supported:

- USB memory devices; e.g. a memory stick
- Networking, USB pointing devices; e.g. a mouse, USB keyboard

Contents

Contents

1

Fundamentals

1.1   About this manual

Safety precautions

Comply with all safety precautions indicated in this document and in your machine tool builder's documentation!

Precautionary statements warn of hazards in handling software and devices and provide information on their prevention. They are classified by hazard severity and divided into the following groups:

⚠**DANGER**

Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **will result in death or severe injury.**

⚠**WARNING**

Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury**.

⚠**CAUTION**

Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in things other than personal injury, such as property damage**.

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

- ■ Signal word indicating the hazard severity
- ■ Type and source of hazard
- ■ Consequences of ignoring the hazard, e.g.: "There is danger of collision during subsequent machining operations"
- ■ Escape – Hazard prevention measures

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.

In these instructions, you will find the following informational notes:

ℹ	The information symbol indicates a **tip**. A tip provides additional or supplementary information.

⚙	This symbol prompts you to follow the safety precautions of your machine tool builder. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.

📖	The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine tool builder or other supplier.

Have you found any errors or would you like to suggest changes?

We are continuously striving to improve our documentation for you. Please help us by sending your suggestions to the following e-mail address:

acurite@heidenhain.com

1.2 Control model

This manual describes functions and features provided by the control as of the following NC software number.

Control model	NC software number
ACU-RITE® TURNPWR	751006-01

The machine tool builder may not allow some of the functions described in this manual, therefore they may not be among the features provided by the control on your machine tool.

The machine tool builder representative can assist you with becoming familiar with the features of the machine.

Many machine manufacturers, as well as HEIDENHAIN Corporation, offer programming courses for the control. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other users.

> **Installation Instructions:**
> ID: 1266125-xx or 1302521-xx
> If you need one of these Installation Manuals, please contact HEIDENHAIN.

> **Installation Manual Kit Style for AC Motors:**
> ID: 1247975-xx
> If you need this Installation Manual, please contact HEIDENHAIN.

> **Technical Manual:**
> ID: 1034383-xx
> If you need this Technical Manual, please contact HEIDENHAIN.

Intended place of operation

The control is intended for use primarily in industrially-zoned areas. Refer to the respective installation manual for additional information.

New functions of software

751006-01

Changed functions of software

751006-01

Access code

ⓘ	The parameter access code is 8891

The access code must be entered before the installation setup parameters can be accessed or changed.

SETUP	▶	Press the **SETUP** key to enter the Job Setup dialogue
Install Setup	▶	Press the **Install Setup** soft key, and enter the access code

---

NOTICE

To prevent setup parameters from being changed:

▶ Remove this page from the manual after initial system setup
▶ Retain this information in a safe place for future use

2

Introduction

2.1   TURNPWR

The ACU-RITE® TURNPWR control is a workshop-oriented turning control that enables you to program conventional machining operations right at the machine in an easy-to-use conversational programming language. It is designed for turning machine tools, with 2 axes and controlled spindle.

The control was developed to satisfy the wants and needs of lathe machinists where manual and automated operation are both useful and needed. The control will enable you to maximize your throughput by significantly reducing set-up time, scrap, and other non-productive operations, thereby increasing your efficiency, productivity and profitability.

The control has many powerful features that will improve your productivity. The screen layout is clearly arranged in such a way that the functions are easy to access, fast and user friendly.

The control is a closed-loop system with positioning feedback provided by compatible feedback devices. When fitted with the (optional) ACU-RITE® precision glass scales (1um/0.00005" resolution), the control also includes Position-Trac™, an advanced, unique feature that enables you to easily, quickly and accurately re-establish work piece zero after shutting down, or power loss.

Conversational and G-code format

The ACU-RITE® conversational programming format is a user-friendly method of writing part programs, however g-code (ISO) programming can also be used. Basic editing of g-code programs is also possible. Preview graphics in the editor illustrate the individual machining steps for programming the contour as well as the corresponding tool path generated. A production drawing does not need to be dimensioned for NC programming, the control can be programmed using the dimensions directly from the production drawing. The programming format is the same as used in the popular MILLPWR^{G2} product line.

Powering up

▶ Turn on the power switch located on the door of the servo cabinet

▶ Turn the power switch on **1** (to the I position) on the console which is located on the back of the control

▶ Follow the builder's instructions to turn on the machine

▷ The start up screen with 3 soft keys will be displayed:
- ▪ **Shutdown**
- ▪ **Find Home**
- ▪ **Cancel**

▶ Press the **Find Home** or **Cancel** soft key

▷ The default DRO screen will be displayed
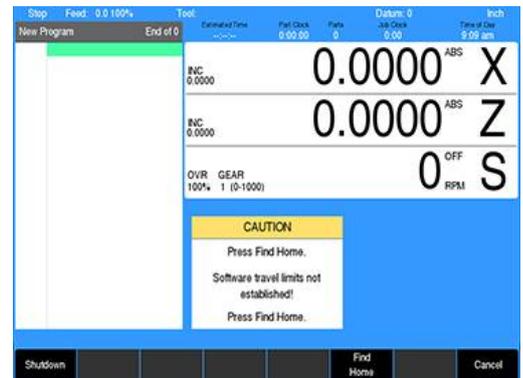
> ℹ️ It is strongly recommended that the control performs the **Find Home** feature at start up, prior to any other action taken.

▷ If a program was loaded when the control was shut down, that same program will be reloaded when the unit is powered up again

> ℹ️ If the control did not perform the **Find Home** feature at start up:
> ▶ Press the **Datum** soft key from the default DRO screen to display the **Home** soft key
> ▶ Press the **Home** soft key
> ▷ The **Find Home** soft key will be displayed

E-STOP and shutdown

The E-STOP is used for emergency program shut down by turning off the servo motors.

NOTICE
On some machines the E-STOP does not shut down the spindle motor. On such machines the spindle motor must always be manually stopped using the spindle switch to stop the motor, and the cutting tool.

▶ When the E-STOP button is pressed, the servo motors are stopped. On some machines it does not shut down the spindle motor.

▶ When the **STOP** key is pressed once, the servo motors pause, but are still active. All axes are locked, and can not be moved.

▶ The program can now either continue by pressing the **GO** key, or stopped by pressing the **STOP** key a second time.

Shutting down the system is done by using the **Shutdown** soft key.

▶ Press the **Shutdown** soft key to power down the control, so that the main switch can be turned off

▶ Wait for the indication that it is OK to turn power off

▶ Press the **Restart** soft key to reboot the control, shutting down and subsequently starting back up

▶ Press the **Cancel** soft key to cancel and exit the shut down procedure

NOTICE
Always shutdown the control before turning power off to the machine.
Refer to the builder's instructions for additional information on turning power off.

Emergency stop (E-STOP)

▶ Press E-STOP to take all axes servos offline

> This ends all machine movement, and allows the operator to manually move the tool out of the way

To reset the E-STOP

▶ Turn the rotary switch clockwise in the direction of the arrows

> The switch pops outward and is reset

Resetting E-STOP does not reactivate the servos.

Resetting the Servos

For safety reasons, the machine powers up with the servo motors disengaged. While the servos are disengaged the axes cannot move under servo power. When hand wheels are connected to the ball screws of the machine, the axes can be positioned manually if necessary.

To reset the servos:

If a limit switch disengaged the servos:

▶ Manually reposition the machine inside its normal range of travel

If a miscount occurs:

▶ Press the **Find Home** soft key to reset the servos and return all axes to their home position

Find home

You should find home before a program is run, or immediately after startup. During start up, the **Find Home** soft key is provided on the start up screen soft key area.

If the find home step is not performed at initial start up, it can be initiated at any time during operation.

▶ Press the **Datum** soft key

▶ Press the **Home** soft key

▶ Press the **Find Home** soft key

> One axis at a time, each axis (X and then Z) will automatically move the required distance to find the reference marks and establish the home position.

The sequence for finding home will depend on the machine. Some machines may use reference switches, ACU-RITE encoders with Position-Trac™ (distance-encrypted reference marks), or absolute encoders; or a combination of these.
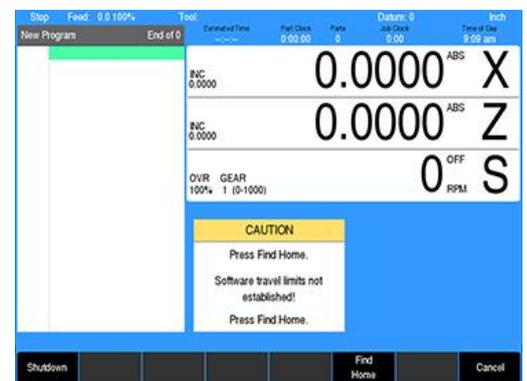
> ℹ️ Finding home applies to the X and Z axes.

After home has been found, the tool's position (relative to the most recent datum set) will be displayed.

NOTICE

Not finding home before moving the machine will risk exceeding the machine's software travel limits, and possible damage to the machine, and the system.

> ℹ️ Programs will not be allowed to run if the homing process does not complete successfully.

> ℹ️ The **Find Home** soft key is not available if there is an error, and the front panel LED indicator is flashing. The error must be corrected, and then cleared from the message log. Then homing is allowed. For additional information, see "Message log", Page 179 for information on opening the message log and clearing errors.

Writing programs

Overview

The control allows many features to be used without having to write a program. For operations that repeat, or complex machining it is best to write a program. Before writing a program, determine the work-holding device and the location of Part Zero (the point to which all movement is referenced). Since absolute positions are defined from Part Zero, try to select a location that directly corresponds to dimensions provided on the part print, such as the right face of the work. Then you can develop a program.

The following is a general approach to programming:

▶ First, select the unit of measurement (Inch/MM) using the **MM** key.

> This will place the DRO in the required unit of measure, and all dialogues will use the selection. If the selection is changed after data has been entered, the control will convert the data to the new unit of measure.

▶ The first step in a program selects the tool that is to be used. The **Tool** dialogue provides fields for defining the tool attributes, as well as for the tool position. This is a tool change position, a location away from the work area where the axes can return for safe tool changing.

The remaining steps in the program describe the required moves, single cycles, and tool changes to complete the machining.

The last step in the program should return the axes to the Tool change position before the program ends.

▶ After writing a program, verify it. Run it to troubleshoot for errors. Verify that all programmed moves are safe, and accurate to the part print dimensions.

▶ Setup the work piece into the intended holding device

▶ First use the simulation graphics to verify that the programmed operations and results match the intended part description and dimensions

▶ Next use either **Graphics Only**, **Dry Run**, **Single Step**, or a combination of these **Run Options** to verify that the actual program and the setting of tool offsets are correct. Make any necessary corrections. Once verified, the program can be run in full Auto Mode.

▶ When the finished program is ready for production, back it up on a USB memory device.

> ℹ️ If there is an interruption to the power supply, the program is not lost. The program is periodically saved. Verify that the most recent steps (prior to the power failure) are in the program. The fixture zero location is also remembered.
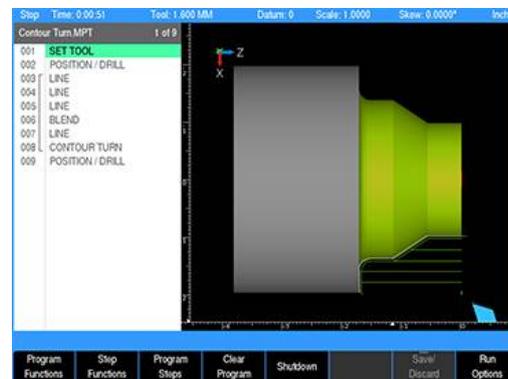
2.2   Console

The control console has a 12.1-inch Flat-Panel Color Screen Display. The following list of items are located on the front panel.

For mapping information of the start up screen, see "DRO display screen", Page 34.

For a full description of the console keypad layout, see "Console keypad", Page 33. For full descriptions of the individual keys, see "Controls", Page 2.

The following features are located on the front panel of the console:

1   Color flat panel screen display

2   Emergency "E" Stop

3   Soft keys

4   Power On indicator light / Error indicator light

5   Potentiometer for feed rate override

6   Potentiometer for spindle speed override

7   Go, Pause/Stop, Navigation keys, and Move Tool keys

8   Axis keys

9   Console keypad with calculator

10   USB connection
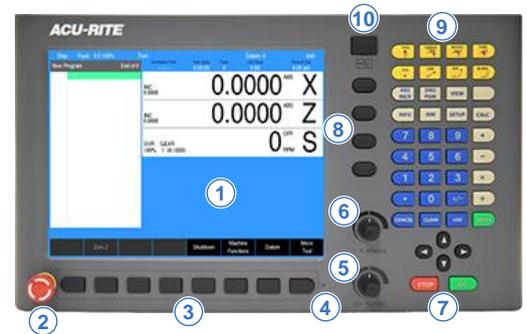
For a full description of the calculator keypad, see "Calculator", Page 31 .

Rear panel

1   USB

2   Not used at this time

3   Servos

4   E-stop relay

5   Pendant (Remote start/stop)

6   Ethernet

7   Power switch

8   Power connector

9   Earth (ground) terminal

10   Handwheel

11   Auxiliary Machine Interface (AMI)

12   Encoders
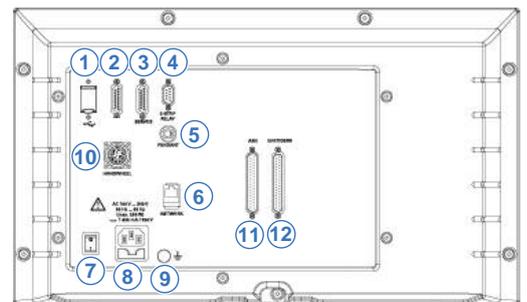
Screen navigation

The display layout changes between **DRO** mode and **PGM** (Program) mode by pressing the **DRO/PGM** key. The following illustrates the differences between the two screen modes.

DRO mode display

In general, the display changes as different functions are activated. Soft keys in the lower display area change per the function selected. Soft keys perform their associated function by pressing the key directly below it. Basic procedures and features remain the same regardless of which mode is selected.

For a complete description of the display areas see "DRO screen", Page 49.

1 Status Bar display for Servo Motor Status, Feed rate, Tool, Datum, X-Axis Mode (Diameter/Radius), Units (Inch/MM), Estimated Time, Part Clock, Parts (run), Job Clock, and Time of Day.

For additional information, see "Status bar display", Page 50.

2 Axes Display (current position)

3 Operator Intervention Message line (OIM)

4 Soft keys display area

5 Program steps

PGM mode display

When PGM mode is selected, the display changes from DRO mode to display program functions and graphics. Soft keys change to programming functions.

For additional information, see "Status bar display", Page 50.

1 Status Bar display for Servo Motor Status, Estimated Time, Tool, Datum, X-Axis Mode (Diameter/Radius), Units (Inch/MM).

For additional information, see "Status bar display", Page 50.

2 Display window of graphical simulation

3 Operator Intervention Message (OIM)

4 Soft keys display area

5 Program name

6 Program steps

Dialogues and drop down menus

This manual provides complete information where specific examples of actions are being explained.

As a general overview of the dialogues provided by the control, several fields are likely to be provided for input. To navigate a dialogue, use the **Arrow** keys to select the desired field. After entering the data into a field, press either the **ENTER** key, or the **Up Arrow** or **Down Arrow** keys to move to the next field. Exiting a field with one of these key methods will retain the entered data in that field.

When the required data has been entered, press the **USE** key to accept the data entered for use in that specific function.

Most dialogues contain Drop Down menus. Specific machining operations may require additional information. The choices available are provided in a drop down menu. In most cases, they are also provided as soft keys. The choices can be selected from either location.
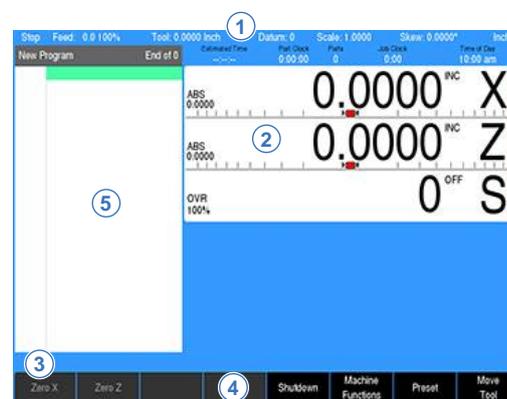
Operator prompts

For actions that require immediate input or acknowledgment to continue the operation an operator prompt will be displayed by the control in the message bar. The required data can be entered with the numerical keypad, and in some instances using the alphanumeric on screen keyboard.

Cursor

The control uses a highlighted cursor to mark a field for selection or editing. In some instances, the cursor will default to a field without highlighting the field. Use the **Arrow** keys to move the cursor. The **Up Arrow** and **Down Arrow** move the cursor through the fields available. The **Right Arrow** will open a field that contains more choices, or subfolders in the folder tree window. The **Left Arrow** will close the menu, or subfolders. The cursor will also change from a highlighted bar to a text cursor when a field selected is having data entered.

General operating guidelines

General operating guidelines for the control:

- Additional operation soft keys are located in the soft key area along with task selection and dialogue. The soft keys change in relation to the task being performed
- Selection of a particular soft key that requires additional information may open a popup menu **1**
- A soft key may open a dialogue menu that requires input necessary to continue with the operation selected
- The status bar in the top of the display is constant, and reflects only what has been selected for the current program
- Use the Context Sensitive Help (**INFO** key) feature when assistance is desired. This is an intuitive feature that aids the user by going directly to the section in the manual in relation to the feature, or key that has been selected to obtain assistance with.
- A feature that is not available will have it's soft key disabled
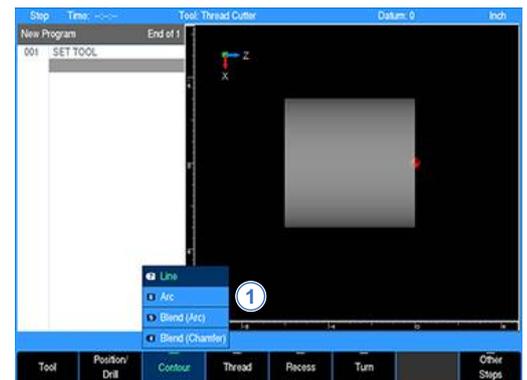
> ℹ️ Soft keys that have a line at the top center indicate that a popup menu is available. Additional choices are available when the key is selected.

Operating modes

The control has two main operating modes: **DRO** and **PGM** (Program). These are accessed from the **DRO/PGM** key. This is a toggle key.

DRO mode

DRO mode shows the current position of each axis relative to current datum. In DRO mode, manual machining, and Single Cycles can be performed. Tool selection, units, along with most setup features can be accessed.
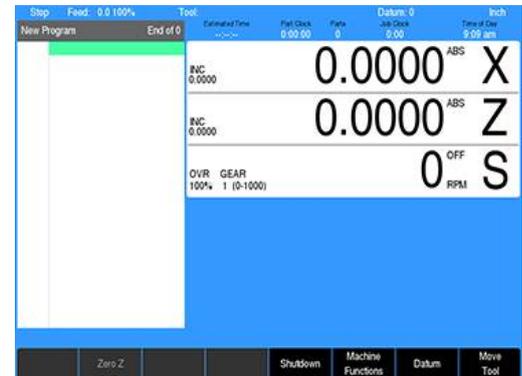
PGM mode

Program mode PGM, displays the list of program steps and part-view graphics. New programs can be created as well as editing existing programs. Programs, new or existing, can be saved, deleted, or copied using program functions.

> ℹ️ A USB pointing device e.g. mouse, trackball, etc. may also be used. If a pointing device is being used, the action of clicking on a soft key button is the same as pressing the corresponding soft key. A USB keyboard can also be used for data entry and cursor control.

Popup menus

Within the soft keys, additional features may be available in program mode. A soft key that has a small line at the bottom center indicates that additional soft keys are available for that feature.

As an example, pressing the **Program Steps** soft key (in PGM mode) opens another set of soft keys available for this function. Then you will notice a small line at the top center on most of the soft keys. This indicates a popup menu will open when that key is pressed.

- Soft keys for PGM provide access to edit existing programs, or create a new program. Sub menus provide dialogue for machining operations such as shoulder turning, recess turning, thread cutting, etc.

Pressing the required operation step soft key will open a popup menu to further define the machining operation required.

A selection for a popup can occur in one of two ways:

▶ Use the shortcut number to the left of the feature; for example, pressing **7** on the numeric keypad will select the option at the top of the list

or

▶ Use the **Up Arrow** and **Down Arrow** keys to highlight the feature to be used, then press **ENTER**.

When the type of machining operation has been selected, the corresponding dialogue opens so that the required data can be entered.

A dialogue in DRO mode will retain the previous data entered making it possible to re-run the previous operation without having to re-enter all the data.

- At anytime when entering data into a dialogue the calculator can be accessed by pressing the **CALC** key.
- Soft keys for **CALC** provide access to additional math functions such as trig functions.

ℹ When a dialogue is activated, it is not possible to change to another dialogue. The current dialogue must be exited by pressing **USE** to save the data in the dialogue or pressing **CANCEL** to discard the data.
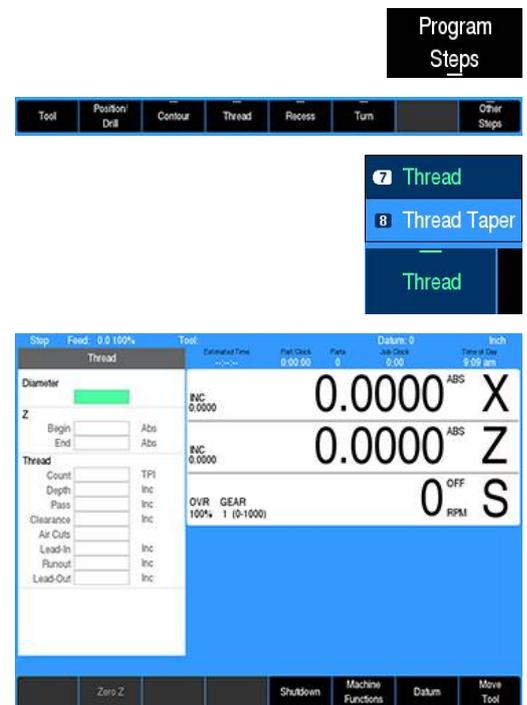
Keyboard

An on screen QWERTY keyboard will automatically popup when you enter a field that requires text information input.

- The keyboard becomes visible (popup) when text, and numerical information is required for an action (e.g. saving a program, or entering text for a comment step).
- Use the **Arrow** keys for navigation of the cursor to go to the desired character and press **ENTER** to select the character.
- When the information has been entered using the keyboard, press the **Save** soft key. The keyboard will disappear from the screen.
- When using a USB keyboard, the on screen keyboard can be disabled by pressing the **Keyboard** soft key. The on screen keyboard will remain disabled until it is enabled by pressing the **Keyboard** soft key again. When the console goes through a power cycle, the keyboard will retain its setting prior to the power cycle.

Special characters

The keyboard also provides a Special Characters display which is accessed by the **(#*\!...** key.

- **Arrow** over to the **(#*\!...** key to highlight the key, and press the the **ENTER** key.

Navigational soft keys

The following navigational soft keys are always available on the keyboard.

⬆	▶ The **Up Arrow** key switches the keyboard from upper to lower case letter mode, or from lower to upper case letter mode depending on the current keyboard mode
Abc...	▶ The **Abc...** key switches the keyboard from the special character key mode to the alpha numeric key mode
(#*\!...	▶ The **(#*\!...** key switches the keyboard from the alpha numeric key mode to the special character key mode.

Editing keys

The following keys are available for editing text on the keyboard.

📋	▶ The **Copy** key will copy the highlighted text in the text field.
📋	▶ The **Paste** key will paste to the text field text that was previously highlighted, and copied.
∕∕	▶ The **Clear** key will remove all text in the text field.
⌫	▶ The **Backspace** key will delete text in the text field one character at a time.
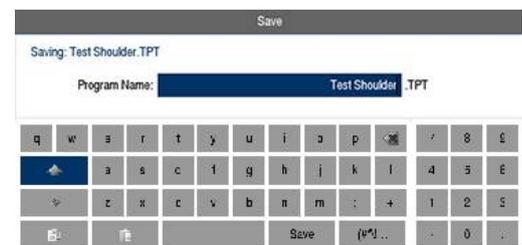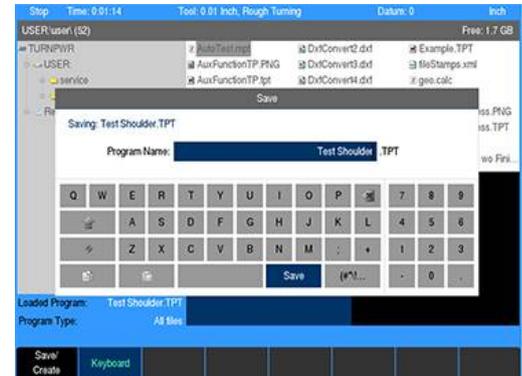
Calculator

Numeric keypad

The numeric keypad on the front of the console resembles a standard calculator with keys for numbers **0** through **9**, four math function symbols (**+**, **-**, **x**, and **÷**), a decimal point, and a positive/negative sign (**+/-**).

The calculator is accessible from nearly any screen or field.

CALC
▶ Press the **CALC** key to access the calculator
> Math function soft keys are displayed in the soft key area
▶ Calculations can be entered directly into a highlighted field

---

🛈 Using the calculator in an entry field in a dialogue requires the operator to press the **ENTER** key to run the calculation. To move to the next field using the **ENTER** key requires the operator to press **ENTER** a second time.

🛈 Anytime data is entered into an entry field in a dialogue and the value is changed using the **+/-** key, the operator is required to press the **ENTER** key to use the value change. Moving to the next field using the **ENTER** key requires the operator to press **ENTER** a second time.

Context sensitive help

The control uses an intuitive method to aid the user when assistance is required. When assistance is needed with a feature, the User's Manual can be displayed directly at the point which describes the feature.

To use this help, in this example, the console is in **PGM** mode, and a Contouring cycle is being programed. It is desired to see descriptions of the cycle parameters, and the cycle itself. The following steps describe the actions needed to take.
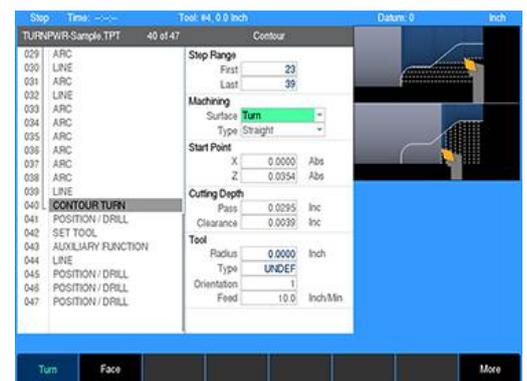
Using context sensitive help

INFO

▶ Press the **INFO** key to open the on screen User's Manual

ⓘ If the Help window stays at the cover page instead of jumping to the pertinent location after loading:

▶ Exit the help window

▶ Press the **INFO** key again

■ This will display a new window containing the section in the User's Manual, opened directly to the description of the desired feature. This is the active window when Context Sensitive Help is activated.

■ To activate the Contents View on the left side of the display, press the **Content View** soft key. This allows the user to see all available topics from the User's Manual. The **Forward** and **Back** soft keys will navigate the users previous history. The **Page up** and **Page down** soft keys will select the previous and next topic in the manual contents (shown in the Contents View on the left hand side). These soft keys allow the user to select other sections of the User's Manual while in Context Sensitive Help. Using these keys will navigate the User's Manual even if the Contents View is hidden.

BACK

▶ The **Back** soft key navigates one page back per key press to a previous page that was viewed from the history. History is not cleared when the Help Screen is exited.

FORWARD

▶ The **Forward** soft key navigates one page forward per key press from the viewed pages in history.

PAGE

▶ The **Page up** soft key navigates one topic up in the contents window per key press.

PAGE

▶ The **Page down** soft key navigates one topic down in the contents window per key press.

Content View

▶ The **Content View** soft key is a toggle key to show/hide the contents view on the left hand side of the display.

Change Window

▶ The **Change Window** soft key is a toggle key to move input focus between the content/tree view on the left side, and the current page view on the right side.

Exit

▶ To close, press the **Exit** soft key.
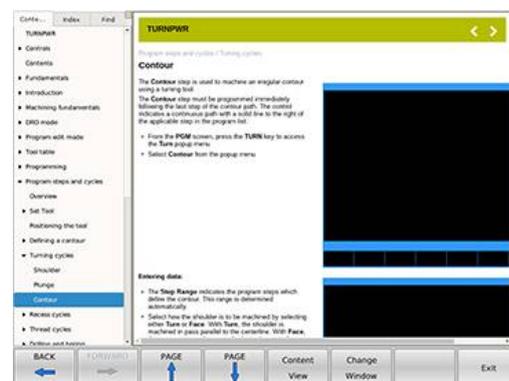
Console keypad

The following keys are located on the console keypad. There is also a quick reference guide located at the beginning of this manual, see "Controls", Page 2.

1 Axis keys, use to select the required axis.

2 Turning function keys. Selection of appropriate key provides direct access to the turning function dialogue.

3 Function keys, select appropriate key for quick access to the function and data entry.

4 **CALC** key, opens the fully functional calculator.

5 Numeric keys, use to enter numeric data.

6 **+/-** key, use to toggle key for "Plus/Minus" data entry.

7 **CANCEL** key, use to cancel current action.

8 **CLEAR** key, use to clear selections such as values entered in a field.

9 **USE** key, use to complete an operation, i.e. data values entered in a dialogue.

10 **ENTER** key, use to activate selections, and entries.

11 **Arrow** keys, use to navigate around the screen, and highlight dialogue fields. Also used to move the tool.

12 **STOP** key, use to pause, or stop a program that is running, or a machine operation.

13 **GO** key, use to start, or resume a program.
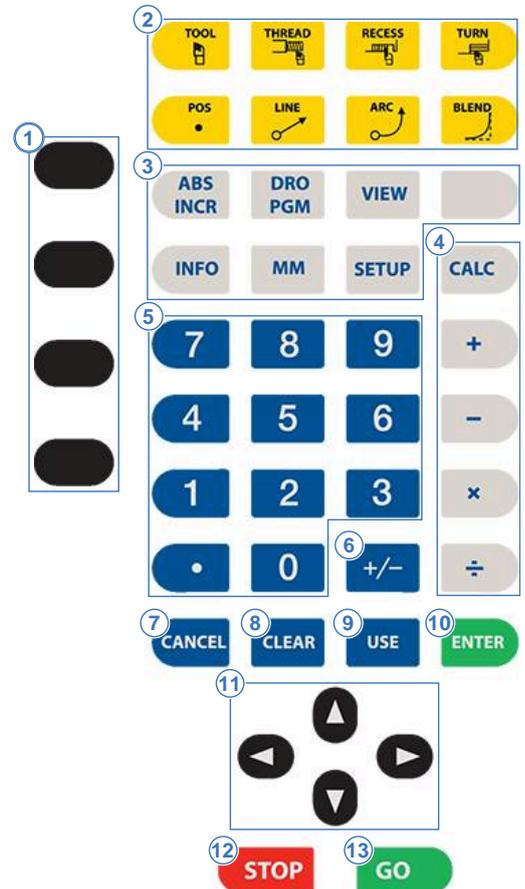
2.3 Operating mode screens

Display navigation

The **DRO** mode is the default start up mode, and **PGM** mode can be selected from the front panel key after start up. A condensed description of these has been provided here on how to navigate, and become familiar with the information that is being provided. For complete descriptions and details, see "DRO screen", Page 49.

DRO display screen

The display screen provides the information required to monitor the machine, it's movement, programming, and more.

The default screen, or home screen, displays two main windows, and soft keys in the soft key display area. When the console is first turned on, the unit defaults to this screen.

For additional information, see "Screen navigation", Page 25.

The DRO screen displays information per the current operation. **DRO** mode allows the use of the turning function keys, or can be used as a DRO in manual machining operations.

The system **Shutdown** soft key is located here as well as in **PGM** mode.
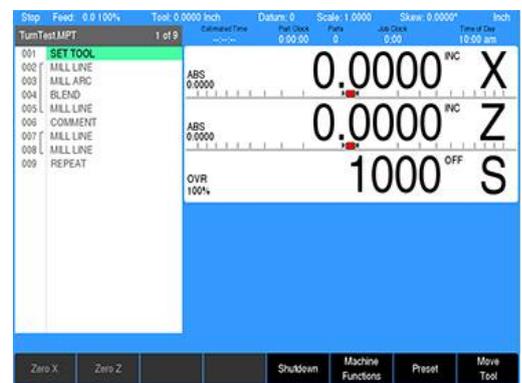
Program display screen

```
DRO
PGM
```

▶ From the **DRO** mode, press the **DRO/PGM** key, and the display changes to program mode showing the currently selected or loaded program.

When programing a job, the display activates a graphic view of the program. While the program is running, the operation that is being performed is graphically shown as well. The **VIEW** key provides features to configure the graphic area.

For additional information, see "Program function screen", Page 61.
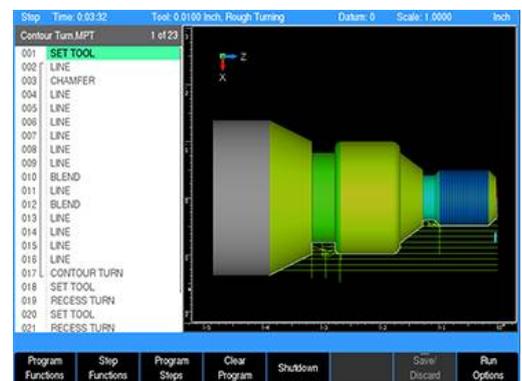
2.4   Accessories

Electronic handwheel and connecting cable

The AH 500 handwheel (ID 1275517-01) facilitates remote operation, moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available.

The handwheel requires Connecting cable (ID 1117852-03) for the connection to the console.

Simulation software for Windows PCs

Simulation software provides a convenient way to write part programs and simulate machine behavior using a Windows computer.

3

Machining
fundamentals

3.1 Machining fundamentals

Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the slide or tool. When a machine axis moves, the corresponding position encoder generates an electrical signal. The control evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the control when they are crossed over. This enables the control to re-establish the assignment of the displayed position to the current machine position. For linear encoders with distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.), and a maximum of 20° for angle encoders.

If incremental position encoders are without reference marks, fixed reference positions have to be traversed after switch-on. The control knows the exact distance between these reference positions and the machine datum (see figure).

With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.
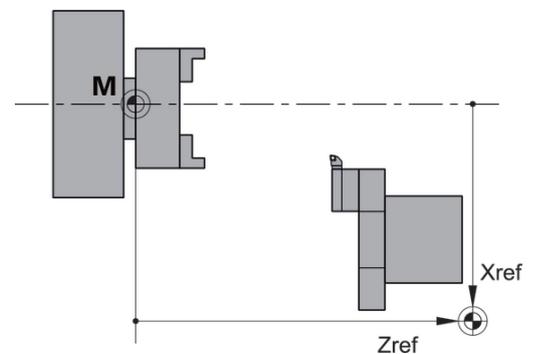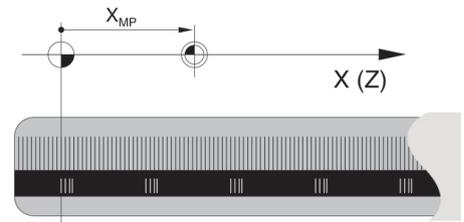
Axis designations

The cross slide is referred to as the **X axis** and the saddle as the **Z axis**.

All X-axis values that are displayed or entered are regarded as either diameters or radius values, depending on user settings.

When programming paths of traverse, remember to:

- Program a **positive value** to depart the workpiece
- Program a **negative value** to approach the workpiece
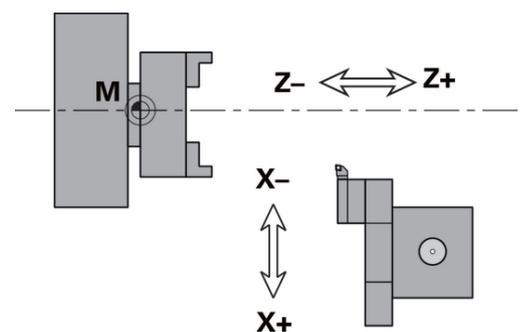
Coordinate system

The coordinates entered for the principal axes X and Z are referenced to the workpiece datum.

The axis designations X and Z describe positions in a two dimensional coordinate system. As you can see from the figure to the center right, the position of the tool tip is clearly defined by its X and Z coordinates.

The control can connect points by linear and circular paths of traverse (interpolations). Workpiece machining is programmed by entering the coordinates for a succession of points and connecting the points by linear or circular paths of traverse.

Like the paths of traverse, you can also describe the complete contour of a workpiece by defining single points through their coordinates and connecting them by linear or circular paths of traverse.

Positions can be programmed to an accuracy of 1 μm (0.001 mm). This is also the accuracy with which they are displayed.
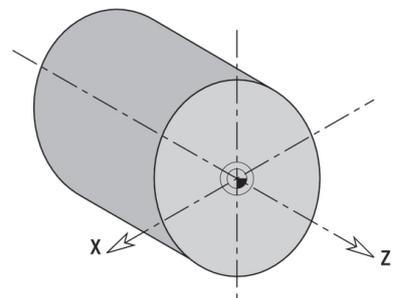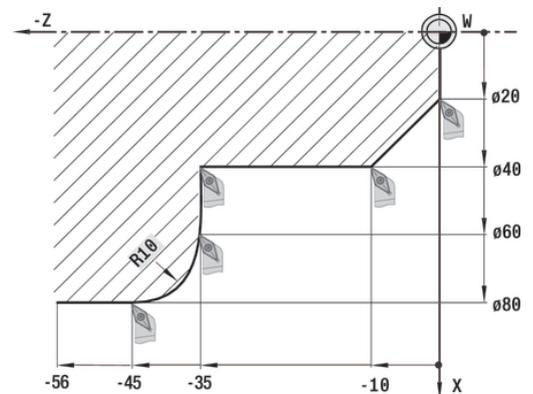
Absolute coordinates

If the coordinates of a position are referenced to the workpiece datum, they are referred to as absolute coordinates. Each position on a workpiece is clearly defined by its absolute coordinates.
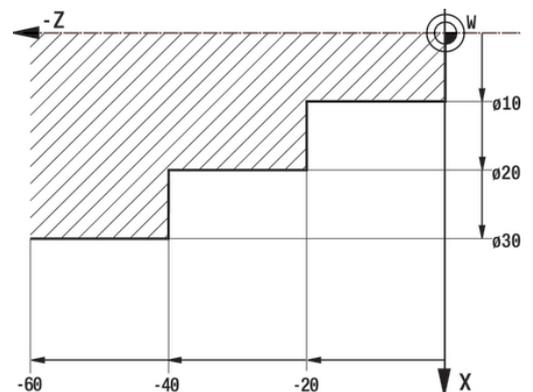
Incremental coordinates

Incremental coordinates are always given with respect to the last programmed position. They specify the distance from the last active position to the subsequent position. Each position on a workpiece is clearly defined by its incremental coordinates.
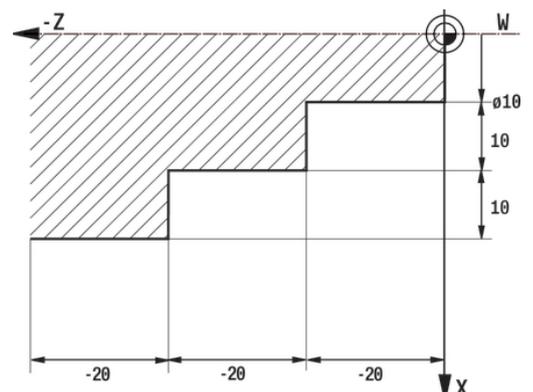
Machine datum

The point of intersection of the X and Z axes is called the **machine datum**. On a lathe, the machine datum is usually the point of intersection of the spindle axis and the spindle surface. The machine datum is designated with the letter **M**.
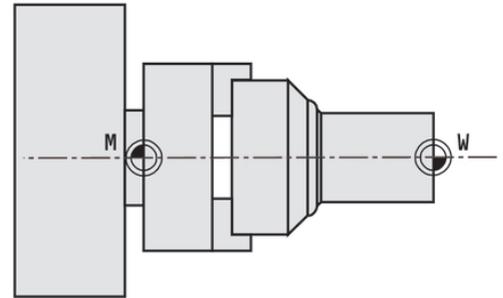
Workpiece datum

To machine a workpiece, it is easier to enter all input data with respect to a zero point located on the workpiece. By programming the zero point used in the workpiece drawing, you can take the dimensions directly from the drawing, without further calculation (dimensioning origin). This point is the workpiece datum. The workpiece datum is designated with the letter **W**.
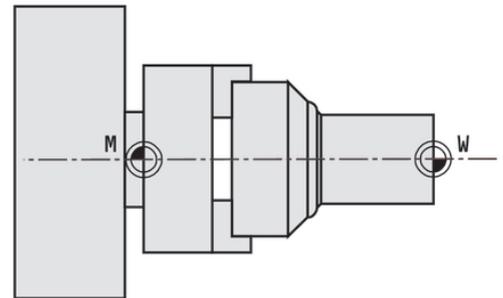
Units of measure

You can program the control either in the **metric** or **inch** system. The units of measurement listed in the table below apply to all inputs and displays.

Dimensions	Metric	Inches
Coordinates	mm	inch
Lengths	mm	inch
Angle	Degrees	Degrees
Spindle speed	rpm	rpm
Cutting speed	m/min	ft/min
Feed per revolution	mm/rev	inch/rev
Feed per minute	mm/min	inch/min
Acceleration	m/s^2	ft/s^2

3.2 Establishing a datum

Overview

The datum is the workpiece zero or absolute zero, and is a point of reference from where the control bases all of the part coordinates.

A datum must be established for every job. The datum location may be indicated on the print, or the operator may establish a datum that allows most of the part dimensions to be entered directly using the least amount of calculations.

For the X axis, the datum is always located at the centerline of the workpiece.

For the Z axis, the datum may be set to any face surface.

Establishing a datum

To establish a datum:

▶ Mount a tool to use for setting the datum. The tool's offset allows the system to determine the distance from the tool tip to the workpiece surface.

> **ⓘ** Besides physically mounting the tool, a **Set Tool** step must be run in order for the system to know what tool dimensions and offsets to use.

▶ From the **DRO** mode, press the **Datum** soft key
▶ In the **Datum Number** field, enter a number for the datum to be defined. Datums 1 through 99 may be set.

This example will establish the datum on the face surface of the part. This will be accomplished by "touching" the face with the mounted tool.

In the **Set Datum** dialogue, the X and Z fields are always active, displaying the machine position relative to the selected datum.

Z axis datum

To set the Z axis datum:

▶ Press the **Move Tool** soft key and use the arrow keys or the handwheel to position the tool so that its tip touches the surface of the face
▶ Exit **Move Tool** to return to the **Set Datum** dialogue
▶ With the cursor in the Z axis field, enter the numeric location of the face surface or press the **Zero Z** soft key
▶ Press the **ENTER** key

> **ⓘ** Do not move the tool away from the part before entering the location or zeroing from the datum dialogue.

▶ Press the **USE** key

3.3   Tool dimensions

The control requires information on the specific tools for a variety
of tasks, such as positioning the axes, calculating the cutting radius
compensation or the proportioning of cuts.

Tool length

All programmed and displayed position values are given with
respect to the distance between the tool tip and workpiece datum.
Since the control only knows the absolute position of the tool
carrier (slide), it needs the dimensions **XL** and **ZL** to calculate and
display the position of the tool tip.
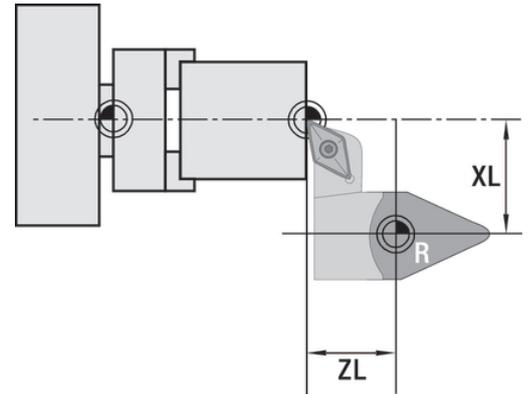
Tool-tip radius compensation (TRC)

The tip of a lathe tool has a certain radius. When machining tapers,
chamfers and radii, this results in inaccuracies which the control
compensates with its cutting radius compensation function.

Programmed paths of traverse are referenced to the theoretical
tool tip S. With non-paraxial contours, this will lead to inaccuracies
during machining.

The TRC function compensates for this error by calculating a new
path of traverse, the **equidistant line**.
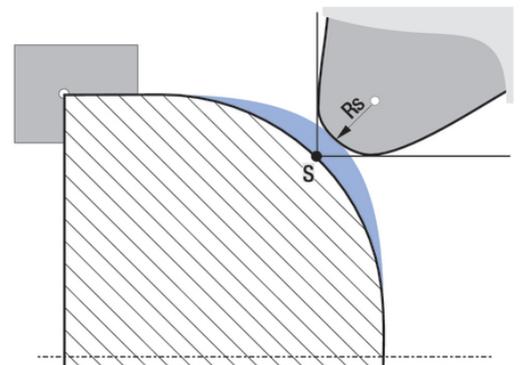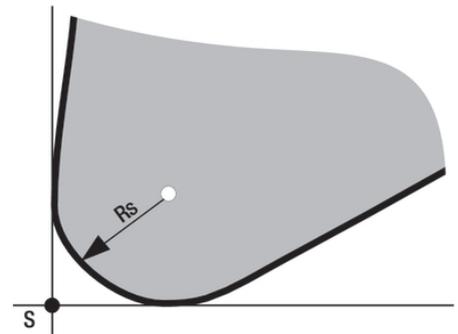
3.4   Manual machine positioning

Move tool

Non cutting mode

You can make or change jog moves when in **DRO** mode with the servos on.

Jog mode	Description
Rapid	Default rapid speed for continuous jogs. Actual speed determined at machine setup, and can be overridden using the potentiometer.
STEP 0.1	Conventional jog increment set at 0.100"
STEP 0.01	Conventional jog increment set at 0.010″
STEP 0.001	Conventional jog increment set at 0.001″

In **mm** units the jog increment is 2000 microns, 200 microns, and 20 respectively.

A finer jog increment of 0.0001" or 2 microns can be enabled via parameter. The default setting is **Off**.

Changing the mode

Jog move modes are performed at the currently active feedrate. The active jog is selected by pressing the appropriate soft key, then pressing the appropriate **Arrow** key to move the tool in the direction required. The **Up Arrow** and **Down Arrow** move the tool in the X axis. The **Left Arrow** and **Right Arrow** move the tool in the Z axis.
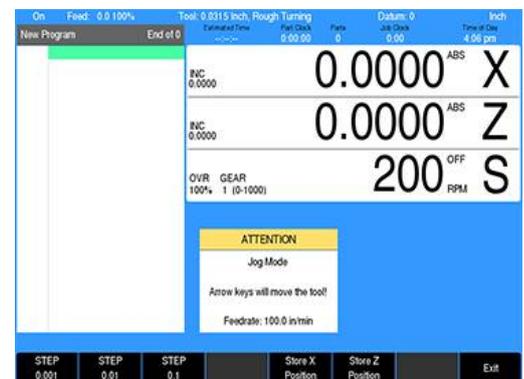
Changing the jogging speed

The active jogging speed defaults to the max feed as determined by the machine builder. The jogging speed may be changed by pressing **CLEAR**. Enter the new speed in the dialogue and press **ENTER**. The new value will remain active until set again or on the next power up. The active jogging speed (at 100% override) is displayed in the Jog Mode **ATTENTION** dialogue.

Incremental moves

In DRO Mode:

▶ Press the **Move Tool** soft key
▶ Select the soft key jog increment required
▶ Press the **Arrow** keys

One jog increment is moved per **Arrow** key press.

Continuous moves

To manually position the machine using continuous movement:

▶ Press the **Move Tool** soft key

▶ Make sure none of the incremental jog keys are highlighted

▶ Press the appropriate **Arrow** key to move the tool in the direction required for the axis move

> The tool will move at the currently active jogging speed. The tool will move until the **Arrow** key is released.

Handwheel operation

When a handwheel is connected and enabled under **Install Setup** it can be activated and used for jogging.

▶ Press the **Move Tool** soft key to activate the handwheel for jogging

▶ Press the **Exit** soft key to deactivate the handwheel and exit **Move Tool**

> ℹ️ In order to use the handwheel, the two permissive buttons must be pressed in (squeezed) and held on the handwheel unit.

The axes can be moved by first selecting the desired **Axis** key, then selecting the desired **Speed** / **Feedrate** key, and then either pressing the **+** or **-** key or moving the wheel on the unit.

The handwheel does offer some functionality outside of **Move Tool** as well. It can be used to issue an Emergency Stop condition, issue an NC Start command (equivalent to console **GO** button), issue an NC Stop command (equivalent to console **STOP** button), and / or issue a **Spindle FWD (O4)** command. All operations, except for Emergency Stop, require that the permissive buttons be pressed and held during the command.

Adjusting the feedrate

The **Feedrate Override** potentiometer can be used to override the currently active feedrate or rapid rate for machine moves. The switch provides a range of 0% to max% as defined in **Job Setup**. Setting the switch to 100% will allow the actual feedrate or rapid rate currently active to be used.

> ℹ️ The machine builder determines the default rapid rate and maximum feedrate at setup. If the control is shut down, the configuration file reloads these default rates at the next power up.
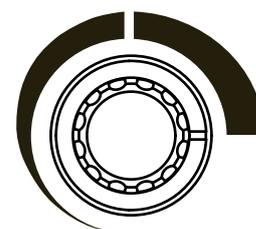
〽 % FEED

Preset moves

The **Preset** feature allows the machine to be used manually like a readout. The preset dialogue is used to set a target location. After entering a target location, the **Incr** display will show the distance between the current position, and the target position.

| DRO PGM | ▶ Press the **DRO/PGM** key on the front panel to place the control in **DRO** mode |

| ABS INCR | ▶ Press the **ABS/INC** key on the front panel to place the control to **Incr** |

▶ Press the **Preset** soft key to open the **Preset** dialogue

▶ Enter the target coordinates for X and Z

▶ Press the **USE** key to set the target and exit the **Preset** dialogue

▶ Press the **CANCEL** key at any time to cancel and exit the **Preset** dialogue

▶ Manually move the tool until the **Incr** axis positions are zero

> When the **Incr** positions are zero, the machine is at the target position

Zero incremental

The **Incr** display may be set to zero at the machines current location.

▶ Press the **DRO/PGM** key on the front panel to place the control in **DRO** mode

▶ Press the **ABS/INC** key on the front panel to place the control to **Incr**

▶ Press the **Zero X** soft key to zero the X axis

▶ Press the **Zero Z** soft key to zero the Z axis

> The **Incr** display will show zero. As the tool is moved, the **Incr** display will show the distance from the current position to the zero location.
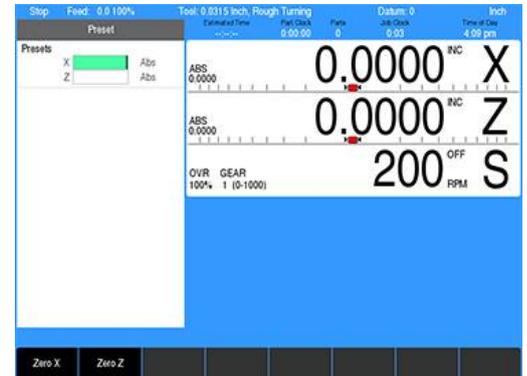
4

DRO mode

4.1 Overview

The control is in either **DRO** mode or **PGM** mode. **DRO** mode allows data input for short one step machining operations. Manual operation, single step operation, and single commands can be entered.

DRO mode also allows the running of programs, see "DRO cycles", Page 57.

The following describes the concepts, and formats used with the control which are being introduced in this chapter.

- DRO screen
- Status bar display
- Move tool
- Turning functions
- Zeroing an axis
- Teach position
- Diameter/Radius mode
- Turning function keys
- DRO turning cycles

4.2 DRO screen

The **DRO** screen displays information per the current operation. The DRO mode allows the use of the turning function hard keys, or can be used as a DRO in manual machining operations.

In addition to the information provided in "DRO mode display", the following table describes what is being viewed on the display in DRO mode.

When the console is first turned on, the soft keys Shut Down, Find Home, and Cancel are displayed.

▶ Press the **Find Home** soft key

> After pressing either the **Find Home**, or **Cancel** soft key, the default **DRO** screen will be displayed
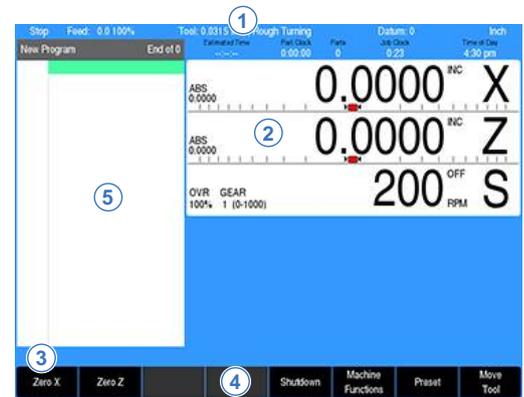
> 🛈 If the control did not perform the **Find Home** feature at start up, press the **Datum** soft key from the default DRO screen to display the **Home** soft key. Press the **Home** soft key then the **Find Home** soft key will be displayed.

1 Status Bar displays the servo motor status Stop/Run/Pause, feedrate override, current Tool (blank if none is selected), Datum, Diameter/Radius mode, unit of measure selected (Inch/MM), Estimated Time, Part Clock, Parts (run), Job Clock, and Time of Day. For additional information, see "Status bar display", Page 50.

2 Display current position of each axis, and relative to the current datum.

3 Operator Intervention Message line (OIM) that provides prompts, and messages to the operator.

4 Soft keys display. The soft keys change per the function that has been selected. Each are activated by the associated hard key that is directly below it. When a soft key appears as being pressed, it is activated. If the key is not pressed in, it is not currently active. The soft key text and background colors change to grey when the feature of the key is not currently available.

5 This window displays program steps if a program is loaded. If no program is loaded, then this window is blank. When a function has been selected (e.g. **Set Tool**), its dialogue would be displayed in this window.

4.3 Status bar display

Status bar display	Description
Program state	This field indicates the program state:**Stop** when program is not running, **Run** when program is running, or **Pause** when program is paused for operator.
Feed	This field indicates the current feedrate of the axes.
Tool	In **DRO** mode, this field indicates the current tool selection. In **PGM** mode, this field indicates the programmed tool for the highlighted step (TPT programs only).
Datum	This field indicates the active datum number.
Rad/Dia	This area shows the "Ø" symbol when X-Axis is in Diameter mode, or nothing when in Radius mode.
Units	This field indicates the active display units. All dimensions are displayed in these units (inch or mm). The current feedrate is displayed in these units (inch or mm per minute).
Estimated time	This field indicates the estimated machining time for the active program. This time assumes the program runs at 100% of the programmed feeds. It does not include the time necessary to perform manual operations such as changing a tool. An unfinished simulation run will result in no time displayed (--:--:--).
Part clock	This field shows the cumulative running time of the current program. The timer is reset when a program is loaded.
Parts	This field shows the number of times the current program was run to create a part.
Job clock	This field shows the total elapsed time for the current job. For additional information, see "Job clock", Page 177.
Time of day	This field shows the current time.

4.4 Move tool

The **Move Tool** soft key when pressed will turn on the servo motors. Pressing it again will turn the servo motors off.

The desired feed rate can be set, or it can be left at 100% for rapid feed rate. For additional infomation, see "Changing the jogging speed", Page 43 and see "Adjusting the feedrate", Page 44.

To move the tool in increments, first select the soft key increment wanted (e.g. 0.001, or 0.01, or 0.1).

The X and Z axes can be moved simultaneously by pressing the desired direction **Arrow** keys at the same time.

> **i** The axis will move one increment per key press when using step increment. The axis moves continuously until released when using continuous jog.
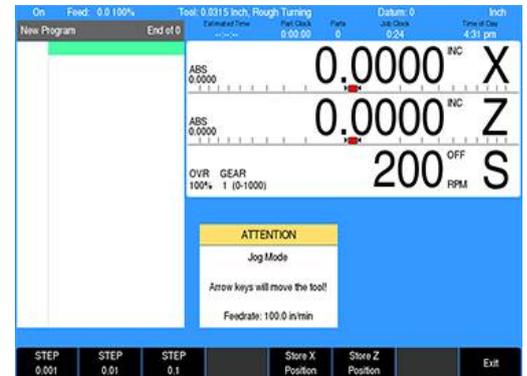
4.5 Turning functions

One Step Turning function may also be referred to as Single Cycle turning functions. Most functions can be used as one time machining operations as well as for programming. The **BLEND** key does not support single operation. Each function will either open its own dialogue or allow selection of choices from a soft key popup. These function keys are described in more detail later in this chapter.

> ℹ️ Single cycle turning functions allow use of internal incremental reference.
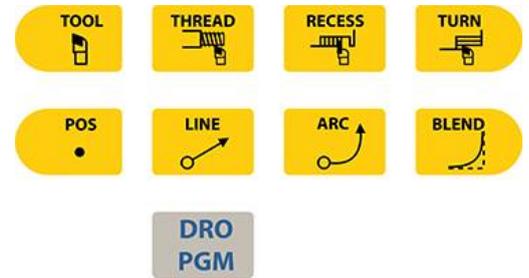
4.6 Zeroing the Z axis

A datum must be set to establish the point from which all absolute dimensions are based.

Pressing the **Zero Z** soft key in **ABS** (absolute) mode will establish this datum position for the Z axis by setting the current axis position as the datum's zero point reference. Press **USE** in the Datum dialogue to save and activate the changes. For **Incr** (incremental) mode, see "Preset moves", Page 45.
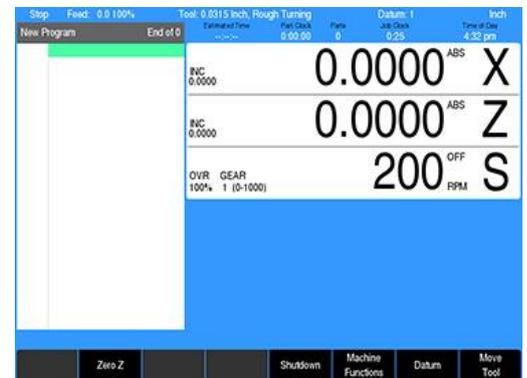
4.7 Teach position

Whenever X or Z coordinates are being entered, the **Teach Position** soft key will appear, enabling the control to be in teach mode and to use the current coordinate(s). The control will base each coordinate on the current absolute position, and enter that position into the field that is highlighted.

NOTICE
While programming, the control can switch between the **DRO** and **PGM** screens without losing the program that is either being created, or being edited.

To teach the control a coordinate while programming a turning function:

▶ Use the **Arrow** keys to highlight the X or Z axis fields
▶ Move your tool to the position you want to teach
▶ Press the **Teach Position** soft key to enter that location then press the **ENTER** key
▶ The numeric keypad and calculator functions can be used to adjust the number
▶ Repeat the steps for each axis and each location that will be used to teach the control
▶ Press **USE** to accept the dialogue or press **CANCEL** to cancel the dialogue
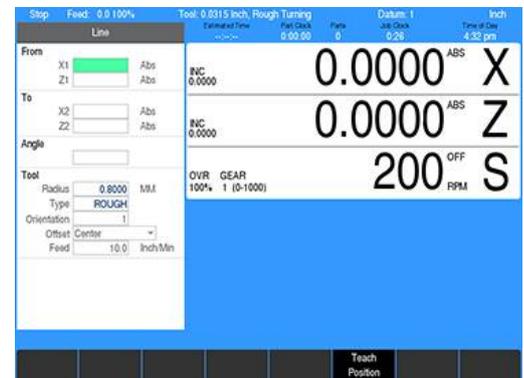
4.8 Diameter/Radius mode

The system can treat the values for the X-Axis as either diameter or radius values. The current mode activated in the system is indicated in the top status bar of the screen, as well as next to the position value of the X-Axis in the DRO position display.

When the active mode is Diameter, the symbol "Ø" will be displayed in these areas. When the active mode is Radius, the symbol is not displayed in these areas. Depending on user preference, or on engineering drawing types, it may be necessary to change the active system mode.

Refer to Section 11 for details on changing the active mode.

4.9 Turning function keys

The following describes in general, the turning function keys.

Most turning functions can be used as one time machining operations. The **BLEND** key does not support single operation.

Key	Function
TOOL	**TOOL** opens the **Set Tool** dialogue where information regarding the tool can be entered.
THREAD	**THREAD** opens the popup menu above the **Thread** soft key. Choose the type of threading that is to be done by selecting it from the popup menu. The required dialogue will open for that choice.
RECESS	**RECESS** opens the **Recess** dialogue where information can be entered for either a Turn or Face recess turning cycle.
TURN	**TURN** opens the popup menu above the **Turn** soft key. Choose the type of turning that is to be done by selecting it from the popup menu. The required dialogue will open for that choice.
POS	**POS** opens the **Position / Drill** dialogue where information can be entered to move the tool to a specified position in the X & Z axes; or to perform a drilling, boring, or tapping operation.
LINE	**LINE** opens the **Line** dialogue where information can be entered to define a line.
ARC	**ARC** opens the **Arc** dialogue where information can be entered to define an arc.
BLEND	**BLEND** opens the **Blend** dialogue while in **PGM** mode where information can be entered to define an arc, blending two surfaces together. Also used for chamfer.
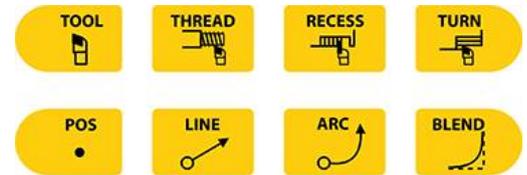
4.10 DRO cycles

The following explains a few of the machining operations that are available. This example is providing an overview to the operator of the control's capabilities.

The **RECESS** or **TURN** keys as an example access popup menus to further refine the type of turning to be completed. Selecting the type of operation required will open the dialogue for that type of turning operation.

An operation is run by pressing the **GO** key. The **STOP** key can be pressed at anytime to pause or cancel an operation.

The following one step functions are available:

Threading cycles
- Thread
- Thread taper

Recess cycles
- Recess turn
- Recess face

Turning cycles
- Shoulder turn
- Shoulder face
- Plunge turn
- Plunge face

Other cycles
- Set tool
- Line
- Arc

5

Program edit mode

5.1 Programming introduction

Program display mode

> **ℹ** PGM mode cannot be accessed when the Calculator is active in **DRO** mode.
>
> If the Calculator is active while in **DRO** mode, exit the Calculator by pressing the **CALC** key, returning the control to **DRO** mode.

| DRO |
| PGM |

▶ From **DRO** mode, press the **DRO/PGM** key, and the display changes to **PGM** mode

> If a program is loaded, it will be displayed, otherwise a blank **New Program** will be displayed.

The program display provides soft keys for program selection, creating, editing, and running functions. The **Clear Program** soft key will exit out of a current loaded program. The system **Shutdown** soft key is provided on the **PGM** mode display the same as it is on the **DRO** mode display. The **Program Functions** soft key provides access to program management features.

Display area

Programming functions, graphics, and soft keys are provided in the display area. Each soft key is activated by pressing the corresponding hard key located directly below it.

1 Status Bar displays the servo motor status Stop/Run/Pause, Time, loaded Tool (blank if none is selected), Datum, Radius/ Diameter mode, and unit of measure selected (inch/mm).

2 Graphic display window of the current loaded program. If a program is loaded, a graphical simulation is shown.

3 Operator Intervention Message (OIM) provides prompts, and messages to the operator.

4 Soft keys display area. The soft keys change per the function that is selected. They are activated by the associated hard key that is directly below it. When a soft key appears as being pressed in, it is activated and its text is yellow. When a soft key is grayed out, it is not available for the current program function.

5 The name of the current loaded program is displayed above the program steps display.

6 The program display window displays the program steps of the current loaded program.
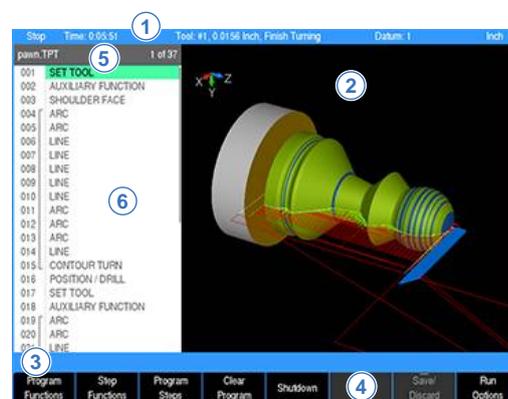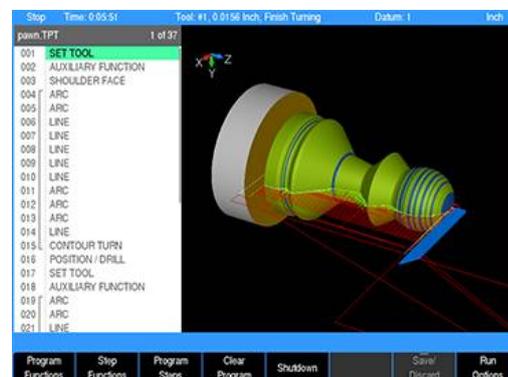
Program function screen

The Program Function screen displays several windows, and soft keys which are defined in the contents of this chapter. The display area window is briefly described in the following table. The **Program Functions** soft key provides access to Network or USB memory devices by pressing the **Folder View** soft key. For additional information see "Folder view", Page 62.

| Program Functions |

▶ Press the **Program Functions** soft key to access programs stored in the Console, on a Network, USB memory devices, or to create a new program

The **Program Functions** screen displays all programs that are stored in a selected folder. The default storage folder for the console is v:\user\. General Information and the programming steps of the highlighted program are displayed in the lower left window **3**, for a quick access review. If a program has ran in its entirety (without errors), graphic preview **2** is also displayed.

1 Available soft keys

2 Program graphic preview

3 Program steps preview

4 Current folder selected (number of programs within selected folder of the active program type)

5 Constant status bar

6 List of all the programs stored in the current location (or folder)

7 Free space available on the current drive

Folder view

The **Folder View** soft key includes additional access to the user. The Folders Tree **9** is displayed which allows the user to access different folders, and narrow a search to selected program types.

Access to Network or USB memory devices is available in the Folders Tree.

- The **Folder View** soft key stays active until it is pressed again

9   Shows the type of programs being displayed (e.g. TURNPWR, G-Code, DXF, etc.)

10 Displays the current program loaded

11 The Folders Tree can be used when connected to a network, USB memory devices, or folder management
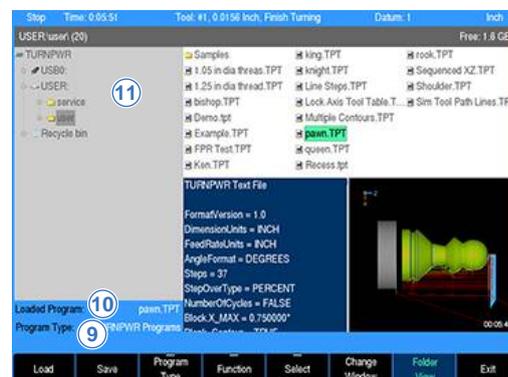
The control can save programs to a USB device, or to network drive (e. g. a PC) as well as to the User folder.

> ℹ️ The internal memory of the control is organized into a folder structure. The default folder is the User folder. Programs can be stored in the User folder. Additional folders can be created to better organize programs. Features in **Program Functions** allow for navigation of all folders, USB memory devices and network folders.

To return to the Program Display Mode:

| Folder View | ▶ | From **Folder View**, press the **Folder View** soft key to exit back to the **Program Functions** display |
| Exit | ▶ | Press the **Exit** soft key to exit back to Program Mode |

Program drawing view

While a program is running (or being created), individual steps being performed are graphically shown.

Pressing the **VIEW** key shows the View Setup dialogue. The **VIEW** key acts as a toggle switch, pressing it again will close the dialogue and return to the program listing. Pressing **CANCEL** or **USE** will also exit the view dialogue.

VIEW	▶	To activate the View Setup dialogue press the **VIEW** key.
		For additional information see "View hard key", Page 68.
USE	▶	To save the changes to the graphics display, close the dialogue and return to the program listing, press the **VIEW** key again, or the **USE** key.
CANCEL	▶	If the **CANCEL** key is pressed, it will close the dialogue and return to the program listing, any settings which do not take immediate effect will revert back to as it was prior to opening the dialogue.
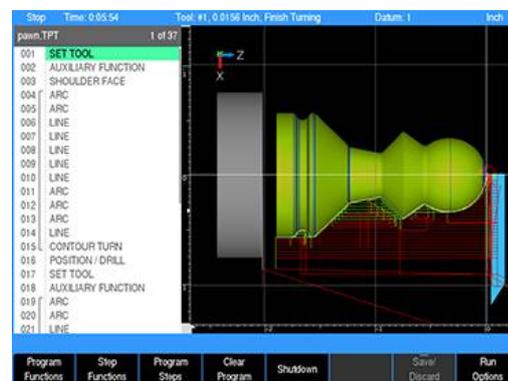
Program screen display

When programming a machining step, the display changes according to the machining step selected. Once a machining step has been selected using the **Program Steps** soft key, the display will appear as shown here. The **Program Steps** display windows are described in the following table.

1 Current program steps

2 Dialogue for a new turning step (e.g. Contour Turn)

3 Dialogue field graphic assistant. Graphic changes per input field illustrating information required

4 Soft keys

5 Program name. If an asterisk appears at the start of the name, it is indicating that the program has not been saved since its last edit.

ℹ️   Soft keys with a bar on the bottom have additional soft keys available when selected.

Soft keys with a bar on the top open a popup menu when selected.

Program mode soft keys

When the **DRO/PGM** key is selected from **DRO** mode, the control screen changes to **PGM** mode, and the program default screen is displayed.

> **i** When the control is set to program mode, types of machining functions specific to the hard key selected become available as soft keys.

Soft key	Function
Program Functions	**Program Functions** opens additional soft keys to **Load**, **Save**, **Clear**, or **Copy** a program along with several other functions. For additional information see "Program functions", Page 65.
Step Functions	**Step Functions** opens additional soft keys for modifying the program steps (e.g. **Reverse Path**, **Delete Steps**, ect.)
Program Steps	**Program Steps** opens additional soft keys for programming, or modifying an existing program by adding additional machining functions
Clear Program	**Clear Program** will unload the current program when the **Yes** soft key is selected. The program is not saved. If any changes were made, they are lost.
Shutdown	**Shutdown** will shut down the control system. Only then can power be turned off
Save/ Discard	**Save/Discard** opens a popup menu which provides the choice to either save the changes made to the program, or discard the changes by returning the program to its original state
Run Options	**Run Options** opens additional soft keys for running a program (e.g. dry run, etc.)

NOTICE

Once **Clear Program** is used the program cannot be recovered.

Program functions

Program functions soft keys

The **Program Functions** soft key provides access to the following soft keys for saving, clearing, or loading an existing program to either edit or run.

Soft key	Function
Load	The **Load** soft key will load the selected program. If a program is currently loaded, a prompt will be displayed asking for confirmation to clear the loaded program. If a program is currently loaded, it must be saved prior to pressing this key.
Save	The **Save** soft key will save the currently loaded program, or the program can be saved under a new program name.
Program Type	The **Program Type** soft key opens a popup Menu providing the choices which type of programs to display in the folder listing; TURNPWR programs, G-code programs, DXF drawings, All programs or All files.
Function	The **Function** soft key opens a popup menu providing the choices to Cut, Copy, Paste, Merge, Rename, or Delete any program that is listed in the folder. It does not need to be the currently loaded program.
Select	The **Select** soft key allows the operator to select one or more programs from the current folder, or select all the programs. Select the Clear option to un-select the current selected program(s). This allows features within the Function soft key to be done on the selected group of programs.
Change Window	The **Change Window** soft key toggles the cursor between the folder list window **1**, folder tree window **2** (if displayed) and program list window **3**
Folder View	The **Folder View** soft key opens the folder tree window to access all folders, and sub folders. Use the Change Window soft key to navigate to the available windows.

Soft key	Function
Exit	Press the **Exit** soft key to return to the Program Function screen
Select	▶ Pressing the **Select** soft key feature opens a popup menu that provides these three choices:

■ The **Select** feature will select one or more programs. Highlight each program to be selected, and choose Select for each program highlighted.

■ The **Select All** feature will select all programs in the folder.

■ The **Clear** feature will un-select one or more programs that have been highlighted. Clear does not delete a program, it will only un-select it.

Having a program, or programs selected, use the **Function** soft key to continue on taking the appropriate action required.

Function	▶ The **Function** soft key provides several choices that can be executed for the action taken after using the **Select** soft key feature:

■ The selected program(s) can be **Cut** from the current folder, then moved to another folder using the **Paste** feature.

■ In the same fashion, using the **Copy** feature, the selected program(s) can be copied to the same folder, or to a different folder using the **Paste** feature.

■ Use the **Merge** feature to paste the contents from the selected program to the currently loaded program at the current cursor location. This feature is only available for TURNPWR (.TPT) programs. Once this function is executed, it will exit the program functions, and return to program view.

■ Use the **Rename** feature to rename the selected program. Only one program at a time can be selected to use this feature.

■ Choosing the **Delete** feature will delete one or more programs selected. The message bar will be displayed showing the first selected program asking for confirmation to delete it. Four soft keys are displayed for deleting options.

   ■ **Yes** will delete the current program displayed in the message bar.

   ■ **Cancel** will cancel out of the current operation. **Cancel** will not restore any prior deletions that had been made.

   ■ **All** will delete all the selected programs by automatically answering yes to confirm deletion of each program.

   ■ **No** will cancel the deletion of the current program in the message bar.

The **Plus** and **Minus** keys on the console keypad are used to facilitate navigation among folders in the console, network and USB memory devices.

■ Pressing the **Minus** key always takes you back to the previous folder location. Pressing the **Plus** key takes you to the next folder in the history. **Plus** is only available if **Minus** has been used.

> ℹ️ The control will save 20 navigation history steps. The 21st history step will replace the 1st history step.

Recycle bin

The **Recycle Bin** stores deleted files and can also be accessed to restore the deleted files to their original location.

Accessing the Recycle Bin:

▶ Press **DRO/PGM** to activate program editing mode

▶ Press the **Program Functions** soft key

▶ Enable **Folder View**

▶ Select the **Recycle Bin** in the Folder Tree window

> ℹ️ The Recycle Bin may take 1-2 seconds to refresh and display its contents.
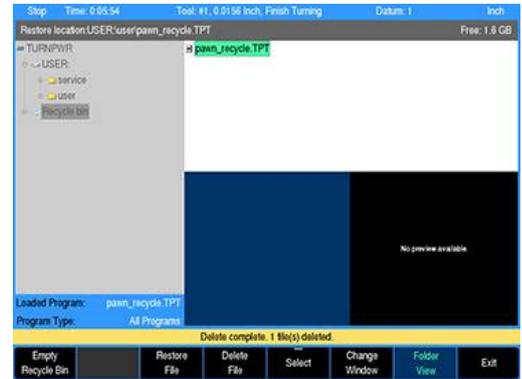
Soft key	Function
Empty Recycle Bin	The **Empty Recycle Bin** soft key will permanently delete all files in the Recycle Bin.
Restore File	The **Restore File** soft key will restore the selected file(s) to the original location. This location will be displayed in the status bar.
Delete File	The **Delete File** soft key will permanently delete the selected file(s) in the Recycle Bin.

> ℹ️ Deleting a folder will move its contents into the **Recycle Bin** as separate files and not as a directory itself.

> ℹ️ Files which were deleted from a removable drive will be restored to USB0: if it is present, otherwise they will be restored to V:\user_Recovered\.

> ℹ️ The **Program Type** filtering will automatically change to **All Files** when entering the **Recycle Bin**, and revert back when exiting **Recycle Bin**.

Enabling/Disabling the Recycle Bin

The **Recycle Bin** can be enabled or disabled through the **Install Setup** soft key. For additional information, see "Installation setup", Page 174.

Size of the Recycle Bin

The **Recycle Bin** can store up to 500 MB of data. If the Recycle Bin reaches its maximum storage capacity prior to or while deleting a file(s), a prompt will provide the option to either permanently delete the file(s) or cancel the delete operation for the remaining files.

View hard key

While in PGM mode or running a program, pressing the **VIEW** key opens the View Setup dialogue. Several adjustment and graphical display options can be set here. To save changes, press the **VIEW** or **USE** key. Pressing the **CANCEL** key rejects the changes made and restores the graphics view to the previous state.

For drop down menus, use the **Right Arrow** key to open, the **Left Arrow** key to close, and the **Up Arrow** or **Down Arrow** key to make a menu selection from the list.

All settings, except Adjust View and Simulation Speed, will remain until changed again. This also applies to system reboots and software updates.

For proper simulation when Show Model is active, make sure that tools are defined in the program. Each unique tool will be assigned a different color which will be used to display the material it removes so that it is clearly visible which tool removed which areas of block material.

The following describes the View Setup dialogue soft keys and drop down menu options available.

View setup

Features	Features can be turned on and off and are graphical representations of the following:
	▪ **Tool** is the solid tool component, which represents the tool as it moves, and changes between the color blue when it is outside of the material and red while it is actively cutting material
	▪ **Rapids** are the tool movements that are set at a rapid feed rate
	▪ **Contour** is the geometry of the programmed contour
	▪ **Markers** are the zero point marker and reference point markers
	▪ **Edges** are the blockform edge lines which outline the material in the X, and Z axes
2D Grid	In 2D views, an overlay grid can be used in conjunction with the edge ruler to aid with visual measurements within the graphic display.
	▪ **Visible** allows the grid to be turned on or off
	▪ **Spacing** adjusts the amount of distance between the grid lines, and is relative to the actual part size
Adjust View	▪ These three buttons allow you to adjust the graphic display.
	▪ Cursor to the appropriate button: **Pan**, **Rotate**, or **Zoom** then press the **ENTER** key to activate the adjustment mode. The button text will shift to the right side of the button to indicate that the adjustment mode
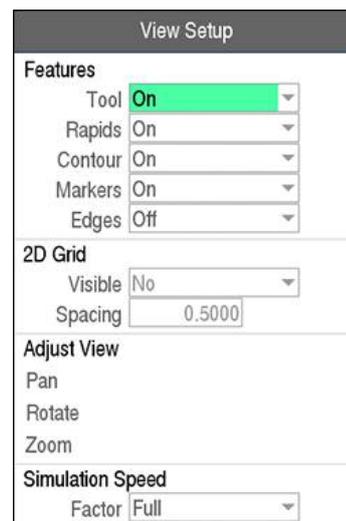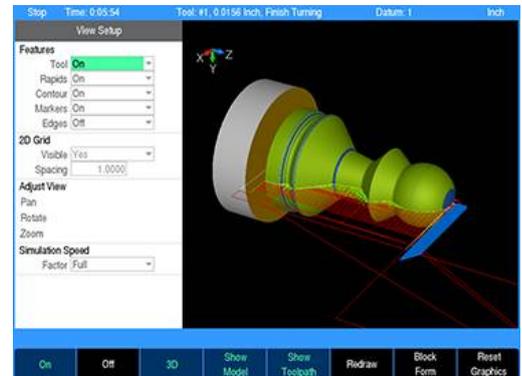
View setup	
	is active. Use the **Arrow** keys to adjust the display.
	▪ Press **ENTER** or **CANCEL** to exit the active adjustment mode
Simulation Speed	The **Simulation Speed** can be changed to any of the following values:
	▪ **Full**: Simulates the program as fast as possible
	▪ **10X**: Simulates the program at ten times the programmed feed rate
	▪ **5X**: Simulates the program at five times the programmed feed rate
	▪ **Actual**: Simulates the program at the programmed feed rate
	▪ **½X**: Simulates the program at half the programmed feed rate

> ℹ Machining a part in PGM mode will show the soft keys to easily toggle between 2D or 3D, toggle the solid model on/off, or toggle the tool path lines on/off, without having to open the View Setup dialogue.

View setup soft keys

The following soft keys are always available while the **View Setup** dialogue is open. Additional soft keys are also made available dynamically depending on which setting is active in the dialogue.

Soft key	Function
2D 3D	The **2D/3D** soft key toggles the graphic display between a two-dimensional and three-dimensional view
Show Model	The **Show Model** soft key displays or hides the solid model rendering in the graphic display
Show Toolpath	The **Show Toolpath** soft key displays or hides the rapid and feed toolpath lines in the graphic display
Redraw	The **Redraw** soft key re-runs the program simulation and update graphic display. If the program is currently mid-simulation, it will be cancelled. The decimal "." key is a shortcut to achieve this same behavior even when the View Setup dialogue is not open.
Block Form	The **Block Form** soft key displays the Block Form dialogue, which shows the minimum and maximum dimensions of the material block form of the part. The values can be overridden in this dialogue to match the actual block of material being used if desired, and can be saved by pressing **USE**.
Reset Graphics	The **Reset Graphics** soft key restores the graphic display adjustments to the default values. The Pan, Zoom, and Rotate adjustments will be reset.
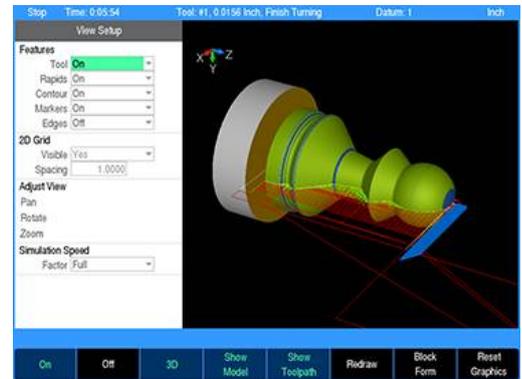
Block form dialogue

While in the **View Setup** dialogue, pressing the **Block Form** soft key opens the **Block Form** dialogue. The calculated minimum and maximum dimensions of the material block form will display. To override these dimensions, new values can be entered and saved by pressing the **USE** hard key.

The **User Values**, **Enabled** field alters the active block form dimensions switching between the manually entered or automatically calculated values.

The following soft keys are available while the **Block Form** dialogue is open.
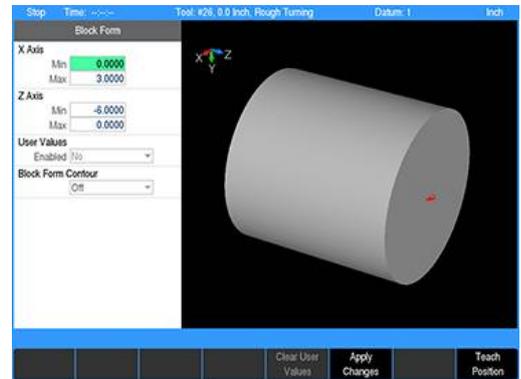
Soft key	Function
Clear User Values	Clears the values entered manually and fills the dimensions with automatically calculated values.
Apply Changes	The graphics view is redrawn to reflect the changes made by the new values.
Teach Position	Inserts the current coordinate in the field.

> ℹ️ For G-code programs: When there is a G120 in the program, changes are effective immediately. The **CANCEL** key does not undo any changes made. Switching the **User Values** setting will comment / uncomment the G120 in the program.

Block form contour

The **Block Form Contour** field allows you to control whether or not the system optimizes the tool path when program steps have areas of overlapping material removal.

With the setting **On**, material removed from the original block form size is tracked. On subsequent steps, the system detects areas of the part that have already been removed by the previous program steps. Only the remaining material will be machined. This reduces machining time by minimizing "air" cuts.

With the setting **Off**, all steps are machined as programmed. This setting may be required in situations where material has already been removed by an operation which the system cannot track.

ℹ	For the tool path optimization to be most effective, the **Block Form** size needs to be set correctly.

ℹ	The contour optimization setting is saved with each individual program and will apply each time the program is loaded and run.

ℹ	The optimization only detects material removed during turning steps (shoulder, recess, etc.) and turning contours (contour and recess contour). Material removed by Drill/Bore steps or Line/Arc paths is not detected.

ℹ	If the program is started mid-path, the system will not avoid the areas machined by the previous program steps.

Step functions soft keys

The **Step Functions** soft key provides access to the following soft keys for assistance with programing steps, and modifying or arranging the programs steps.

Soft key	Function
Explode	The **Explode** soft key will explode a program step e.g. such as a repeat step converting it into individual program steps.
Reverse Step	The **Reverse Step** soft key will reverse the tool path of the selected step so that it will be followed in the reverse order
Reverse Path	The **Reverse Path** soft key will reverse the tool path selected of all the steps so that it will be followed in the reverse order
Shift Steps	The purpose of the **Shift Steps** feature is to transpose a range of steps from one location to another on the actual work piece within a program
Change Steps	The **Change Steps** soft key opens a dialogue that will allow the operator to select a range of steps that can be modified for the Z axis, Offset, and Feed Rate
Delete Steps	The **Delete Steps** soft key opens a dialogue that will allow the operator to select a range of steps, or a step to be deleted.
Copy/Move Steps	The **Copy/Move Steps** soft key opens a dialogue that will allow the operator to select a range of steps to either be copied, or moved else where in the program.
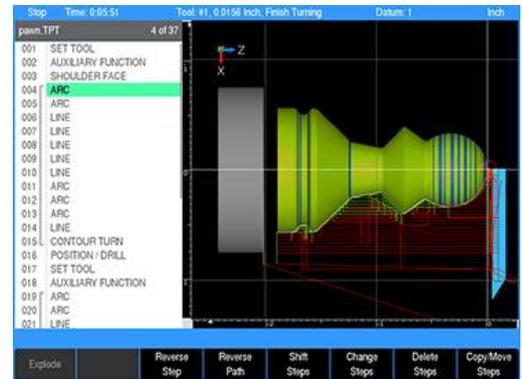
Program steps soft keys

The **Program Steps** soft key accesses the soft keys listed in the table below. Some soft keys provide additional selections on a popup menu, which allows to further define the operation to be programmed.

Soft key	Function
Tool	The **Tool** soft key opens a tool dialogue to enter the size and type of tool to be used for the following step(s).
Position/ Drill	The **Position / Drill** soft key opens a dialogue to define either a Drill, Bore, Position, or Tap operation
Contour	The **Contour** soft key opens a popup menu to further define a contour element, e.g. a Line, Arc, Blend (Arc), or Blend (Chamfer)
Thread	The **Thread** soft key opens a popup menu to further define a threading operation, e.g. a straight thread or a tapered thread
Recess	The **Recess** soft key opens a popup menu to further define the type of recess operation, e.g. either a standard recess cycle or a free-form contour recess cycle
Turn	The **Turn** soft key opens a popup menu to further define the type of turning opera-tion, e.g. a Shoulder, Plunge, or free-form Contour cycle
Other Steps	The **Other Steps** soft key opens a popup menu to further define the type of operation e.g. Repeat, Comment, Auxiliary Function, Dwell, or Reference Point

Clear program soft key

This soft key provides a quick method to clear the current loaded program. It does not provide an option to save before exiting. Its function is meant to quickly clear a program (e.g. once it has been run and is no longer needed), so that a new program can be loaded, or a blank program is available, or return to **DRO** mode without having a program loaded. If the program has been edited the operator is asked to confirm the Clear operation. Save program changes before using Clear. All changes are lost and cannot be recovered.

▶ Press the **Clear Program** soft key to close the program
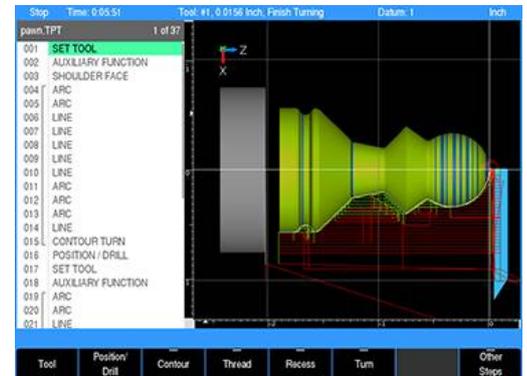
Save/Discard soft key

The **Save/Discard** soft key opens a popup menu which provides the choice to either save the changes made to a program, or discard the changes by returning the program to its original state.

Save/ Discard	▶ Press the **Save/Discard** soft key and select from the popup menu to either save the program, or discard the changes made to the program

Run options soft keys

The **Run Options** soft key allows the program to be run and tested with various options for a program. These options allow the operator to test a program without actually cutting material. The **Run Options** must be set before you start to run the program.

Soft key	Function
Single Step	The **Single Step** soft key will move the machine one step at a time
Dry Run	The **Dry Run** soft key will run the program's tool path at the speed defined in Job Setup. No material should be placed within the tool path for this function. Dry Run is intended to provide a quick verification of the machine movements in the program.
Graphics Only	The **Graphics Only** soft key will run the program's tool path without activating the servo motors. The graphics and DRO display shows the progress of the program without motion or AMI output. **Graphics Only** previews the tool path at programmed speed without the risk of tool or part damage.
Optional Stop	The **Optional Stop** soft key will allow the program to be stopped at different locations to allow for verification of particular program steps selected has the operator. If a program has an optional stop (i.e.: M01 in a G-code program), the system will only stop at this block when **Optional Stop** is enabled. If it is disabled, it will be skipped. The Optional Stop is an M01 in a G-code program.
Exit	The **Exit** soft key will exit the Run Options function and return to the previous display
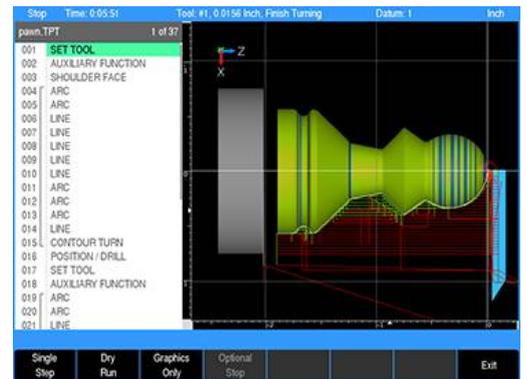
Program saving

When you create programs with the control, they can be saved within the control's User folder, on a USB device, or a network location.

The control will retain a program in memory so it is not lost in the case of a power failure.

Programs can be organized in the control, on a USB device, or a network location; or in personalized folders that the operator can create.

Saving a program

A new blank program requires a minimum of one step added before it can be saved. A newly created set of program steps can be saved to the existing program, or can be saved as a new program by replacing the current name.

To save the current active program:

Save	▶ Press the **Save** soft key
	> The keyboard will popup with the **Save** key highlighted
ENTER	▶ Press the **ENTER** key
	> A prompt will ask to overwrite the existing program
Yes	▶ Press the **Yes** soft key

The keyboard will automatically be displayed with the current program name highlighted in the keyboard display window. For additional information see "Keyboard", Page 30.

The **Save/Discard** key is also available to quickly save a program, or discard any changes made since the last time it was saved.

Save/ Create	▶ Press the **Save/Create** soft key to save the program with its current name
	or
	▶ Enter a new name for the current program to create a new program

---

NOTICE

A name can not be longer than 60 alphanumeric characters. The control displays program names as they were entered. No two programs can have the same name.

NOTICE

The program's graphic preview is updated whenever the program is saved. In order for the new graphic preview to be created correctly, the program must not have any errors. The operator must allow the program graphics to finish drawing the program before pressing any keys.

5.2 Folder management

The control offers several versatile features for loading, saving, deleting, and backing up programs. Programs can be organized by creating folders. Folders may be used to group programs by job, operator, date, customer, or any other method preferred. Folders can be quickly selected, created, make a backup copy, or if necessary, deleted.

Creating folders can only be done in **PGM** mode.

Creating a folder

First decide where to place the folder. It can be placed on V:\User which is the control folder in the console, on a USB device, on a network, or within sub-folders that have already been created. Use the following steps to create any type of folder, top level or sub level folders.

> ℹ When the **Folder View** soft key is activated, it remains active until it is pressed again to deactivate folder view.

To create a folder:

▶ Press the **Program Functions** soft key
▶ Press the **Folder View** soft key
▶ Press the **Change Window** soft key to select the Folder Tree window
▶ Navigate through the folders to place the cursor where the new folder is to be created by using the **Arrow** keys
▶ To expand a folder use the **Right Arrow** key
▶ To collapse an expanded folder, use the **Left Arrow** key
▶ Highlight the folder where the new sub-folder is to be placed
▶ Press the **Create Folder** soft key

Naming a new folder

To name a new folder:

▶ Using the on screen Keyboard, enter the name of the new folder
▶ If the keyboard is not automatically displayed, press the **Keyboard** soft key
▶ Once the new folder's name has been entered, press the **Save/Create** soft key to create the new folder

Deleting a folder

The control will delete folders that contain programs, or files. The control will prompt the operator that there are files within the folder, and ask for confirmation if they are to be deleted.

▶ Press the **Program Functions** soft key

▶ Press the **Folder View** soft key

▶ Press the **Change Window** soft key to select the Folder Tree window

▶ Navigate through the folders, and place the cursor where the folder is to be deleted

▶ Press the **Function** soft key

▶ Select **Delete** from the popup menu

▶ Press the **Yes** soft key to erase the folder, and all files in the folder

▶ or

▶ Press the **No** soft key to cancel

NOTICE
Once a folder is deleted it cannot be recovered, however the files that were within the folder can be retrieved from the Recycle Bin.

5.3　Program management

Saving a program

When creating programs, they can be saved within the control's User folder, on a USB device, or to a network location. A graphic view of the program is displayed in the Preview Window if it has been run successfully.

Programs can be organized in the control, on a USB device, in a network location, or in personalized folders that the operator can create.

To save the current program for the first time:

▶ Press the **Program Functions** soft key

▶ Press the **Save/Create** soft key

The keyboard should automatically be displayed with the cursor in the Program Name field. For additional information, see "Keyboard", Page 30.

Naming a program

Before you can save a program, the control requires it to be named.

Enter the program name using the **Arrow** keys to navigate the keyboard. Highlight the key to be used (e.g. letter, or number), then press the enter key to add the selection into the **Program Name** Field. Continue in the same manner until the name is complete.

To add numbers to your program name, press any of the number keys on the numerical keypad, or from the keyboard display.

▶ Press the **Save/Create** soft key

> The control will store the program in the folder that had been previously selected.

A message will alert you if the program was not saved properly, or if the name that you've chosen already exists.

Back up programs regularly to avoid accidental loss (e.g. USB device, or to a network location).

Deleting a program

▶ Press the **Change Window** soft key to select the Folder Contents window

▶ Highlight the desired program

▶ Press the **Function** soft key

▶ Select **Delete** from the popup menu

▶ Press the **Yes** soft key to erase the program

or

▶ Press the **No** soft key to cancel

Loading a program

The **Load** soft key allows you to open programs in the folder listing. The steps below explain how to load a program from the User folder, a USB device, or from a network location.

> **ℹ** Program extensions:
> - TURNPWR programs have a .tpt extension
> - G-code programs have a .G or .NC extension
> - DXF files must have a .DXF extension

▶ If a program is loaded, save the open program, it does not have to be cleared

▶ From the **PGM** screen, press the **Program Functions** soft key

▶ When a program is not located in the current folder, select the location of the program using the **Folder View** soft key, and **Change Window** soft key as needed

▶ Verify that **TURNPWR** programs is selected in the **Program Type:** popup menu. Only **TURNPWR** programs stored in the selected folder will be displayed. For additional information, see "Program type filter", Page 83.

▶ Using the **Arrow** keys for navigation in the display window, highlight the program to load

▶ Press the **Load** soft key, and verify that the desired program is now loaded

When loading a program from a network, or a USB memory device, first select the device from the Folders Tree, then follow the same procedure as would be done from the User folder.

> **ℹ** Use **Folder View** for locating folders, and programs to load.

> **ℹ** It is recommended that all programs be copied to the local User folder for usage.

Backing up a program

Backing up a program is similar to saving a program. A backup program can be saved to another location (e.g. memory device, or on a network), or to the same program location. If it is saved to the same program location, then it can be with the same name plus an indication in the name that it is a backup copy. The **Copy** and **Paste** functions do this automatically, and is further explained below.

In the same manner, this can also apply to backing up a G-code program.

It is also possible to select all programs in a folder and be saved as a backup to the same folder.

Copy and paste programs

▶ Press the **Program Functions** soft key

▶ Press the **Change Window** soft key to select the folder tree window

▶ Repeat to select the Folder Contents window

> **i** While copying, moving, or deleting a large number of files a dialog will appear to display the progress. At this time a cancel key will be available to cancel the operation.

▶ Use the **Arrow** keys to highlight the program to be copied, then press the **Select** soft key

> A popup menu will open where either the highlighted program can be selected, or all programs in the folder can be selected

▶ Highlight **Select**, or **Select All**, then press the **ENTER** key

▶ If the program that was selected is not the correct one, select the Clear feature, and press **ENTER**

> The program is de-selected, and another program can now be selected

▶ Press the **Function** soft key

▶ Select **Copy** from the popup menu

▶ Press the **ENTER** key

▶ Press the **Function** soft key again

▶ Select **Paste** from the popup menu

▶ Press the **ENTER** key

A copy of the selected program has now been copied to the same folder and was named with a suffix of "Copy". Another folder location could have been chosen, before **Paste**, to copy the program to the new location.

If a program is copied multiple times using the **Paste** function, each pasted program copy would then have sequential numbering (e.g. Copy, Copy (2), Copy (3), and so on).

Importing a DXF drawing

Save any running programs. Locate the folder containing the DXF drawing.

To import a DXF file:

▶ If a program is loaded, save the open program, it does not have to be cleared

▶ From the **PGM** screen

▶ Press the **Program Functions** soft key

▶ When a DXF drawing is not located in the current folder, select the location of the DXF drawing using the **Folder View** soft key, and **Change Window** soft key as needed

▶ Verify that **DXF drawings** is selected in the **Program Type:** popup menu. Only DXF drawings stored in the selected folder will be displayed. For additional information, see "Program type filter", Page 83.

▶ Using the **Arrow** keys for navigation in the display window, highlight the DXF drawing to load

▶ Press the **Load** soft key, and verify that the desired program is now loaded

The control will assign default values for any information that's missing from the DXF drawing (e.g. tool offset, feed rate, etc.). The required **Set Tool** steps must be added.

The control will then arrange the steps in a logical order based on common end points, and create a tool path. The program will then appear on the display.

Test the program before machining to ensure that the program steps and tool path do what is expected. The program can be edited and steps rearranged as needed.

5.4 Program mode functions

Program type filter

In **PGM** mode, press the **Program Functions** soft key to display the **Folder View**, and **Program Type** soft keys. Program type aids with locating programs by type, and reduces the number of programs that are displayed.

▶ To select what type of programs to show, press the **Program Type** soft key, and select which program type(s) to display using the **Arrow** keys.

USB access

As soon as the USB memory device is inserted, its contents will be displayed in the Folder Contents window. From here the operator can modify, copy, delete, etc. any of the programs, or folders on the USB device.

In **PGM** mode, the operator can access a USB memory device by inserting the USB device into a USB connection of the control.

▶ From **DRO** mode, press the **DRO/PGM** key to enter **PGM** mode
▶ Press the **Program Functions** soft key
▶ Press the **Folder View** soft key

The display will now show the USB device in the Folder Contents window, and when selected, will show its content in the upper right window.

> **ⓘ** It is recommended to copy programs from the USB device to the user folder and use the program from the control.

> **ⓘ** Prior to removing the USB memory device, check that the activity indicator light on the USB drive is not active or blinking. This will ensure that all commanded disk operations have completed, and that it is now safe to remove the drive.

5.5 Creating programs overview

New part program

The following briefly describes how a new program is created. For additional information, see "Fundamentals for creating a program", Page 98.

▶ Press the **DRO/PGM** key from **DRO** mode to enter **PGM** mode

A new program can be created from an existing program, or create a completely new program.

To create a new program from an existing program:

▶ Press the **Program Functions** soft key

▶ Select an existing program that will be used to create the new program

▶ Press the **Load** soft key

▶ Press the **Program Functions** soft key

▶ Press the **Save** soft key

▶ Press the **Keyboard** soft key to display the keyboard if the keyboard is not automatically displayed

▶ Type in the name of the new program

▶ Select **Save**

> The new program is now loaded, and can be modified as required

To create a new (blank) program:

▶ When a program is currently loaded, press the **Clear Program** soft key in the **PGM** mode display to clear the loaded program from memory, and to create a new program

▶ Add at least one program step

▶ The **Save/Discard** soft key will become enabled

▶ Press the **Save** soft key

▶ Press the **Keyboard** soft key to display the keyboard if the keyboard is not automatically displayed

▶ Type in the name of the new program

▶ Select **Save**
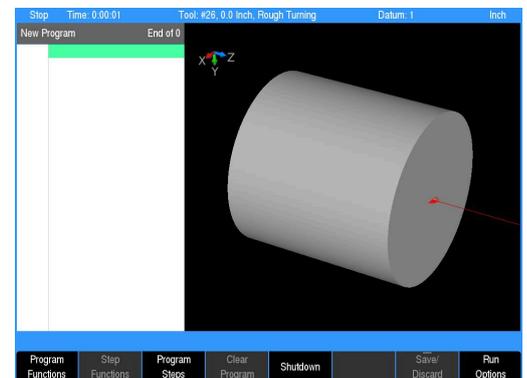
6

Tool table

6.1 Overview

The Tool Table is available to define and store up to 100 tools. Each tool is identified by a number between 1 and 100. The Tool Table is used to store tool cutting radius, length, tool angle, etc., depending on the type of the tool (Rough, Finish, Recess, Button, Thread, Drill, etc.)

Tool Table values are displayed in inches or millimeters based on the current units of measure when the table is opened. Tool data is entered into the table using the Set Tool Dialog.

Tool data can be entered directly in the part program or in the Tool Table. When running a G-code program's tool block or a conversational program's tool step, specified by tool number, the tool data is provided from the tool table. Example: Running a G-code T1 block prompts the operator to mount tool 1. The tool data is retrieved for tool #1 from the tool table and used to adjust the X and Z tool path

> **ℹ** The tool number 0 is automatically defined as zero tool (tool holder empty) with radius = 0, length = 0, width = 0, etc.

6.2   Accessing the tool table

The Tool Table is accessed from the DRO setup screen.

To access the tool table:

▶ Press the **SETUP** key

▶ Press the **Tool Table** soft key.

6.3 Navigating the tool table

To navigate the tool table:

▶ Press the **Up Arrow** or **Down Arrow** keys to navigate the **Tool table**.

▶ Press the **ENTER** key to edit the highlighted tool number

▶ Press the **CANCEL** key to exit the Tool Table and return to previous operating mode

6.4 Soft keys

Soft key	Function
Goto Tool Number	**Goto Tool Number**: Jump to a specific tool number
Page Up	**Page up**: Navigate to the previous page in the **Tool Table**
Page Down	**Page down**: Navigate to the next page in the **Tool Table**
Exit	**Exit**: Exit the **Tool Table**

6.5 Editing the tool table

When the **Tool Table** is opened, a new tool can be added, or an existing tool can be edited or removed.

Entering a new tool

To enter a new tool:
- ▶ Using the **Arrow** keys, highlight the next available blank row
- ▶ Press the **ENTER** key to open the **Set Tool** dialog
- ▶ Select the tool type from the drop down box in the **Type** field
- > The **Set Tool** dialog is updated to show tool information required based on the selected tool type
- ▶ Press the **ENTER** key
- ▶ Using the **Arrow** keys, navigate to the other fields and enter the tool information
- ▶ Press the **USE** key
- > The information is saved to the **Tool Table**

or

- ▶ Press **CANCEL** and the tool information is not saved
- ▶ Continue to add additional tools as necessary
- ▶ When all tools have been added, press the **Exit** soft key

Editing a tool

To edit a tool:
- ▶ Using the **Arrow** keys, highlight the desired tool to be changed
- ▶ Press the **ENTER** key to open the **Set Tool** Dialog
- ▶ Make the necessary changes
- ▶ Press the **USE** key
- > The information is saved to the **Tool Table**
- ▶ Press the **Exit** soft key to close the **Tool Table** and return to previous operating mode

Deleting a tool

To delete a tool:
- ▶ Using the **Arrow** keys, highlight the desired tool to be deleted
- ▶ Press the **CLEAR** key
- > The highlighted tool information is removed from the table

6.6   Tool data

Field	Description
Tool	Number of the tool
Type	The tool type (rough, finish, recess, button, thread, drill, tap, bore)
Orientation	The tool orientation, refer to the help image for the orientation number
Radius	The cutting edge radius (rough, finish, recess, or button tools). Hole diameter (divided by 2) for drill tools.
Length	Length of the tool (drill, tap, recess)
Width	Width of the tool (recess)
X Offset	Offset from the tool to the X workpiece surface
Z Offset	Offset from the tool to the Z workpiece surface
Tool Angle	Angle from the surface of the part to the primary cutting edge of the tool (rough, finish)
Point Angle	Angle from the primary cutting edge to the secondary edge (rough, finish)

6.7   Tool offsets

With repeatable tooling, it is useful to establish the tool offset for each tool. This allows tools to be changed without the need to re-establish the datum.

Calculating tool offsets from the tool table

To calculate tool offsets from the tool table:

▶ From the tool table, cursor to the tool to be set and press the **ENTER** key

▶ Cursor to the **X Offset** field

▶ Press the **Move Tool** soft key

▶ Use the arrow keys or handwheel to move the tip of the tool to the X axis surface

▶ Press the **Store X Position** soft key to store the tool's current X axis location

▶ Use the arrow keys to move the tool away from the part

▶ Press the **Exit** soft key

▶ Measure the diameter of the part and enter the value into the **X Offset** field

▶ Press the **Calculate Offset** soft key

> The offset will be calculated

▶ Cursor to the **Z Offset** field

▶ Press the **Move Tool** soft key

▶ Use the arrow keys or handwheel to move the tip of the tool to the Z axis surface

▶ Press the **Store Z Position** soft key to store the tool's current Z axis location

▶ Use the arrow keys to move the tool away from the part

▶ Press the **Exit** soft key

▶ Enter the location of the Z axis surface into the **Z Offset** field

▶ Press the **Calculate Offset** soft key

> The offset will be calculated

▶ Press the **USE** key

i	Once used in a calculation, the stored position for that axis is cleared. When no stored position is available, the **Calculate Offset** soft key will be disabled.

Calculating tool offset from a program

To calculate the offsets from a program, the tool positions must first be stored.

To store tool positions:

▶ Mount the tool to be set
▶ From the **DRO** mode, press the **Move Tool** soft key
▶ Use the arrow keys or handwheel to move the tip of the tool to the X axis surface
▶ Press the **Store X Position** soft key to store the tool's current X axis location
▶ Use the arrow keys or handwheel to move the tip of the tool to the Z axis surface
▶ Press the **Store Z Position** soft key to store the tool's current Z axis location
▶ Exit **Move Tool**

To calculate tool offsets:

▶ From a program, cursor to the **Set Tool** step
▶ Press the **ENTER** key
▶ Cursor to the **X Offset** field
▶ Measure the diameter of the part and enter the value into the **X Offset** field
▶ Press the **Calculate Offset** soft key
▶ The offset will be calculated
▶ Cursor to the **Z Offset** field
▶ Enter the location of the Z axis surface into the **Z Offset** field
▶ Press the **Calculate Offset** soft key
▶ The offset will be calculated
▶ Press the **USE** key

Setting tool offsets

If a tool's offset values are known or were determined by a tool presetter, the offsets may be entered directly.

To set tool offsets:

▶ Open the **Set Tool** dialogue
▶ From the tool table:
▶ Cursor to the tool to be set and press **ENTER**
▶ From a program:
▶ Cursor to the **Set Tool** step and press **ENTER**
▶ Cursor to the axis offset field
▶ Enter the offset value
▶ Press the **ENTER** key

7

Programming

7.1 Conversational programming

Programming considerations

From and To points

Lines and arcs are defined by their **From** point (the point where they begin) and **To** point (the point where they end).
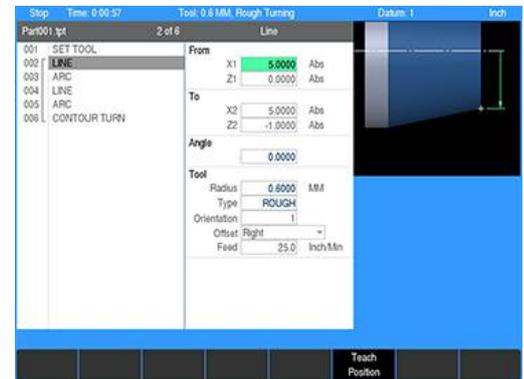
Tool offset

The control will calculate the actual tool path when using left and right offsets. Program the dimensions of the part as identified by the part drawing.

Program a line, arc, etc. using the Tool Offset field to tell the control which side of the line the tool is to be on.

To determine which offset to use: If the tool needs to be on the left side of the line, use a Left offset. If the tool needs to be on the right side of the line, use a Right offset.

When using a Center offset, the programmed dimensions are for the center of the tool radius. For arcs, Inside and Outside offsets are available to make it easier to define the tool offset.
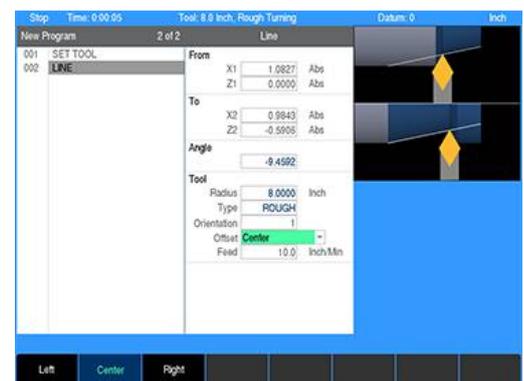
Absolute vs. incremental dimensions

The control allows both absolute and incremental dimensions to be entered. A dimension measured from the point you defined as datum is an absolute dimension. A dimension measured from any other point is an incremental dimension. For additional information, see "Absolute coordinates", Page 39 and see "Incremental coordinates", Page 39.

- The dimension moves of 8, then 8 again are incremental moves
- The dimension moves of 8, then 16 are absolute moves
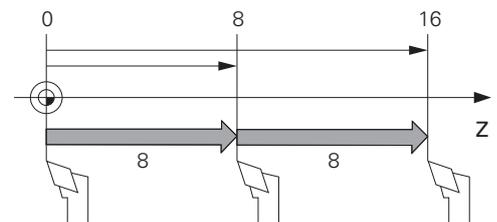
Continuous machining

When you program a continuous contour of lines and/or arcs the contour will cut without stopping if there is no programmed stop. The control will automatically recognize continuous contours as you're programming. There are no special key presses or other functions to learn.

For lines and arcs to be continuous, they must:

- Be consecutive steps in a program
- Be cut with the same tool
- Be cut using the same tool offset
- Share a common "From" or "To" point (one step must end at the point where the next one begins)

If one step follows another, then the control assumes that they are to be connected (see steps 002 through 009). The single line bracket indicates the steps that will be machined with continuity.

The control automatically fills in the From point, Depth, and Tool Offset from the previous step. The To point must be filled in, and then press the **USE** key.

> ℹ️ Different feed rates within each step of a continuous contour are allowed, and can be used.

Restore position

When a program is paused during machining, the **Move Tool** soft key is available to move the tool away from the part. This makes it possible to inspect the part or tool before continuing with the program.

If the tool is moved while the program is paused, it must be restored to the previous location before the program can continue running. The system will automatically restore the tool to this location when the **GO** key is pressed.

If the tool was moved away from the part, soft keys **1** will indicate which axes need to be restored.

If both axes have been moved, the soft keys will indicate that both axes will be restored when the **GO** key is pressed. If the axes need to be restored in a specific order, ensure that only the axis which is safe to move is selected. Pressing the soft key will toggle whether or not that axis will be restored. Only the enabled axis(es) will be restored when the **GO** key is pressed.

The **GO** key will only continue running the program after both axes have been restored.

> ℹ️ If both axis soft keys are pressed, both axes will be restored sequentially in the configured order.

Fundamentals for creating a program

▶ In **DRO** mode press the **DRO/PGM** key to enter program mode

Programs are created by developing a list of steps to be performed. As steps are added to the list, each will immediately be drawn on the screen so that it can be displayed graphically, showing the part machining in progress.

> ℹ️ The control allows a maximum of 9999 program steps.
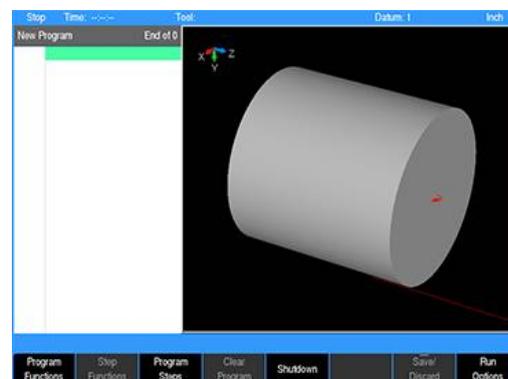
Entering steps

Before any programming steps are entered, program a **Set Tool** step that accurately identifies the tool that is to be used.

The turning function keys are among the eight yellow keys located in the upper right corner of the console. The function you select will appear in the program listing and will enable you to enter the data and add the step into the program.

Additional machining functions are available in **PGM** mode, and are accessed by pressing the **Program Steps** soft key. Program Steps soft keys are also displayed when the turning function keys are pressed in **PGM** mode.

To enter a step:

▶ Press the appropriate function key

Some keys will open a popup menu with the soft key to further define the type of step that is to be performed.

USE
▶ After entering the data for the program step, press the **USE** key to add the step to the program

▶ Press the cancel key at anytime to exit the step dialogue and not add it to the program

Adding and inserting steps

A new program step number automatically appears following the last step entered, and the new step is highlighted.

▶ If the cursor was moved for any reason, use the **Arrow** keys to highlight the new step

▶ Select the function to be added

To insert a step between two existing steps:

▶ Position the cursor to the step below where the new step is to be placed

▶ Press the desired function key and it will be inserted above the highlighted program step

Additional functions are available by pressing the Program Steps soft key. The soft keys change, displaying several machining function soft keys. Each soft key when selected will have a popup menu for further selection to refine the type of operation to be entered.
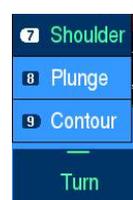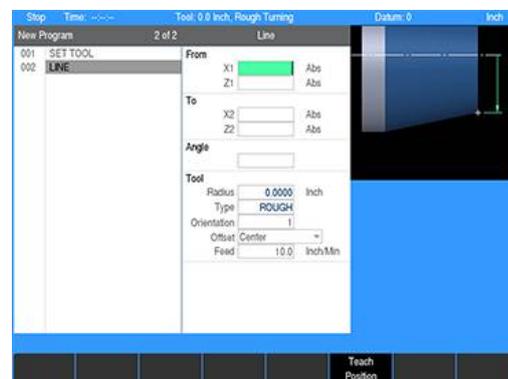
Editing or deleting a step

To edit a program step:

▶ Use the **Arrow** keys to highlight the step to be changed

▶ Press the **USE** or **ENTER** key, or enter a step number using the numeric keypad

＞ A **Goto program line number** popup dialogue is immediately displayed. The total number of lines in the program are shown in the **Line count:** field

▶ Enter the desired line number

▶ Press **ENTER**

▶ Press **ENTER** or **USE** again to edit the program step

▶ After making the appropriate changes, press the **USE** key to accept the changes and place the step back into the program

or

▶ Press **CANCEL** to discard any changes to the step

To delete a step:

▶ Highlight the step to be deleted

▶ Press the **CLEAR** key
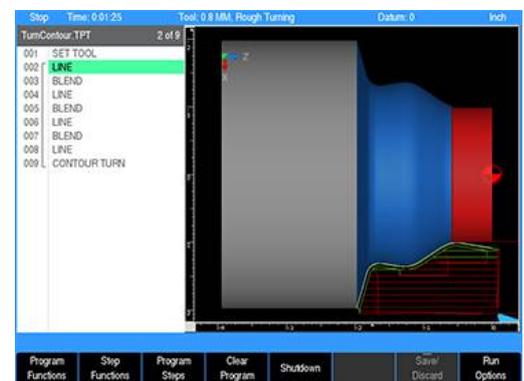
Program errors

When an error is detected in a program, the step with the error is highlighted in the program listing with a red **x** symbol. Edit the program step to correct the error.

Information about the error is added to the message log. For additional information, see "Message log", Page 179.

> **ℹ** After correcting the errors, remember to clear them from the message log. A quick method for clearing all errors in the log is to press the **0** on the numeric keypad.

> **ℹ** If there is more than one error in the program, only the first step with an error will be highlighted. After correcting the first error, the next error will be highlighted.

Edited programs

When a saved program is loaded, the name and extension is displayed.

If the saved program is then edited, the program name will show an asterisk at the end of the name. This tells the operator that the program has changed, but has not been saved.

Saving the program will remove the asterisk.

A program that has been edited, but not saved, will revert back to the last saved version when closed. The program will not display an asterisk the next time it is loaded.

Step form validation

Once a step has been programed in the program dialogue and the use key has been pressed, if further data is required, that field will display a red **x**.

Once a step has been programed in the program dialogue and the **USE** key has been pressed, if the data in any field has been entered incorrectly, that field will display a yellow warning symbol.

> **ℹ** Correct any errors or warnings before proceeding.
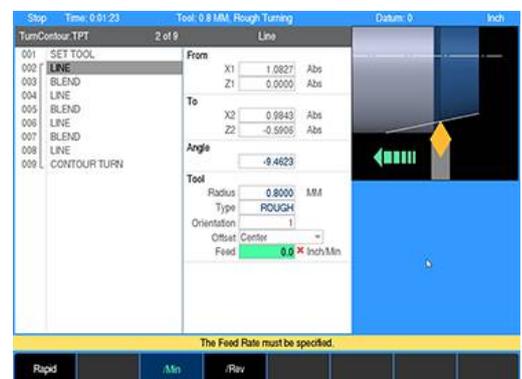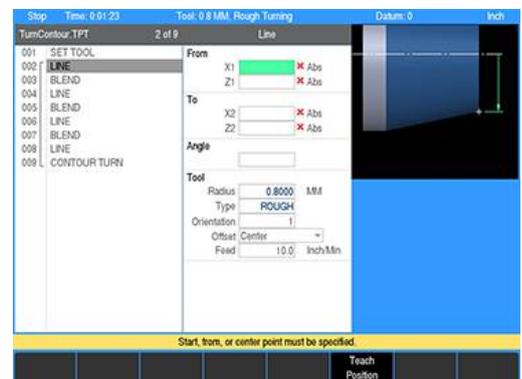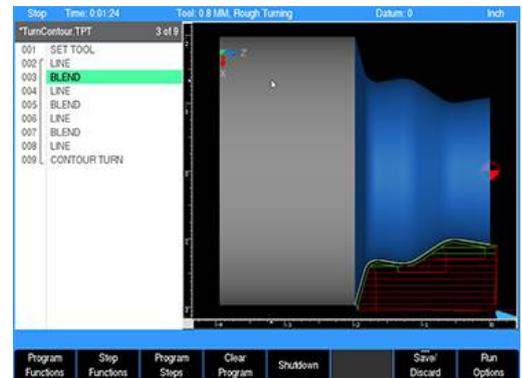
Testing a program

Before machining a part, test the program for things like correct tool path, count direction, feed rate, and sequence of operations. The control provides several run-time choices to assist doing this. Always verify a program when it was loaded, or edited.

▶ From the **PGM** screen, press **Run Options** to display the following soft keys:

Soft key	Function
Single Step	**PGM** runs through the program one step at a time. Some steps will run as a single group such as a Shoulder or Recess
Dry Run	**Dry Run** runs through the program at the defined dry run feedrate instead of programmed feeds. This option is useful to more quickly test the tool path motion. The dry run feedrate is set in the **Job Setup** dialogue.
Graphics Only	The **Graphics Only** soft key will run the program's tool path without activating the servo motors or AMI outputs. Begin from any step in the program. Use the graphics and **DRO** display to observe and verify the tool path.
Optional Stop	The **PGM** soft key will allow the program to be stopped at different locations to allow for verification of particular program steps selected by the operator. If a G-code program has an M01 (optional stop) in it, the system will only stop at this block when **Optional Stop** is enabled. If it is disabled, it will be skipped.

▶ Press any soft key to activate the option
▶ Press it again to deactivate it

NOTICE

Before pressing the **GO** key to begin the **Single Step** or **Dry Run** features, check that the tool will not touch the workpiece when the tool begins to move.

To avoid interference, do one or more of the following:

▪ Remove the tool or workpiece
▪ Re-establish datum away from the part
▪ Select **Graphics Only** run option

Single step

Normally, a continuous contour will be machined without stopping. With **Single Step** activated, the conrol will stop after each step. This enables a check of the tool position of the Z axis relative to the part and ensure that the tool path and other program details are correct. Press the **GO** key to begin, and after each step finishes.

Dry run

With **Dry Run** activated, the control will run the entire program at high speed without stopping. Visually follow the position of the tool relative to the part and ensure that the tool path and other program details are correct. This can be started at the beginning, or midway of a path. The **Dry Run** speed is defined in **Job Setup**. Press the **GO** key to begin.

Graphics only

With **Graphics Only** activated, the control will run the entire program at programmed speed without activating the servo motors or AMI outputs. Follow the position of the tool on the graphics or DRO display (**ABS** position only) to ensure the tool path and other program details are correct. This can be initiated at the beginning or midway through a path. Press the **GO** key to begin.

> **i** The **Dry Run** and **Graphics Only** run options may be combined at **Dry Run** speed without any table motion.

Skipping steps in a program

The option to temporarily skip one or more steps when running a program is available by pressing the **+/-** key. Navigate to the desired step and use the **+/-** key to skip the step without removing it from the program. The program runs as if the step is not present. Press the **+/-** key a second time to restore the step.

▶ Move the cursor to the step to be skipped

▶ Press **+/-** to skip or restore the step. A skipped step is grayed out and an asterisk is placed before the step name.

One or multiple steps may be skipped or restored by using the **Change Steps** dialogue which can be found under the **Step Functions** menu.

▶ From the **PGM** screen, press the **Step Functions** key

▶ Press the **Change Steps** soft key

▶ Enter the first and last step numbers to be changed

▶ Use the **Arrow** keys to navigate to the **Skip Steps** field

▶ Select **Yes** to skip the steps or **No** to restore the steps

▶ Press the **USE** key

> **i** A step marked to be skipped cannot be edited. Restore the step before opening the step dialogue.

Machining your part

Before running a program step, check the Status bar (located along the top of the control's screen) to ensure that the tool identified by the control matches the tool in the spindle. If there's no tool identified, or if it's incorrect, start with a **Set Tool** step that accurately identifies the tool that is to be used.

To move quickly to a step, use the **Up Arrow** or **Down Arrow** keys to highlight the required step, or just enter a step number using the numeric keypad. A **Goto program line number** program line number popup dialogue is immediately displayed. The total number of lines in the program are shown in the **Line Count:** field.

▶ Enter the desired line number

▶ Press **ENTER**

▶ Press **ENTER** or **USE** again to edit the program step

After highlighting the step to begin with, press the **GO** key. An Operator Intervention Message may ask to confirm that the tool is correct. Confirm that the correct tool is being used, then press the **GO** key again to begin machining.
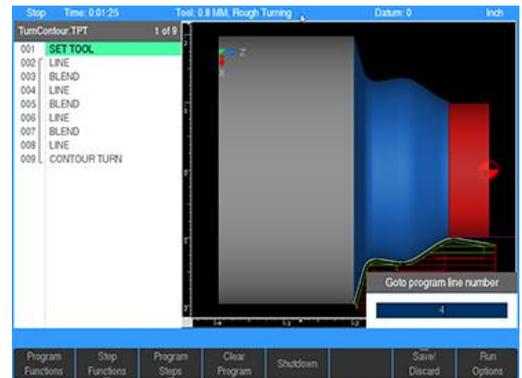
> ℹ️ The remote **STOP**/**GO** switch acts as a pause switch if the table is moving, and as a **GO** switch if the machine is paused or stopped.

The control will automatically pause at points that require action to be taken (e.g., change tools). After each task has been completed, press the **GO** key.

If the **STOP** key is pressed once while cutting, the tool will pause in its cutting path and an Operator Intervention Message will appear. Press **GO** to resume machining, or **STOP** again to end the program.

> ℹ️ If machining is started from the middle of a program, the control will determine which tool should be used, and will prompt the operator to mount the tool if it is not mounted. It will also begin at the beginning of the current path.

Potentiometer for feedrate override

Feedrate Override is controlled with the potentiometer which will change the feed rate by a certain percentage with the knob rotation. The feed rate percentage will be displayed in the status bar at the top of the screen. A feed rate percentage of 100% means that actual feed rates will run at 100% of the programmed feed rates. If the feed rate percentage is 50%, actual feed rates will run at half of the programmed feed rates.

Adjustment of the feed rate with the potentiometer can be done at any time, even while the tool is moving.

Program tracking

The Tracking feature allows you to slowly run the program by using the handwheel to control the feed. The feed is proportional to the speed at which the handwheel is turned.

The Tracking feature only affects feed moves. Rapid moves are not controlled by the handwheel and will run at normal speed.

Tracking is controlled by either the X or Z handwheel. Only forward progress is possible and only by turning the handwheel in the positive (CW) direction.

Tracking is disabled at the end of the program or if Move Tool is activated.

Using the Tracking feature:

▶ While the program is running, press **STOP** to pause

▶ Press the **Tracking** soft key

▶ Press **GO** to resume running the program in Tracking mode

▶ Use the handwheel to control forward progress of the program

To disable Tracking mode and return to normal program run:

▶ Press **STOP** to pause the program

▶ Press the **Tracking** soft key to exit Tracking mode

▶ Press **GO** to resume running the program in normal mode.

7.2 G-code programs

G-code editing capabilities

G-code programs have basic editing capabilities. The value of an existing g-code address can be changed.

There are no soft keys available for editing, all navigation is done using the **Arrow** keys. Use **Up Arrow** and **Down Arrow** keys to move the cursor to the line that needs to be edited and press the **ENTER** key to edit the line.

Use the **Left Arrow** and **Right Arrow** keys to move from address value to address value within the line. Enter the new value for an address. The **Up Arrow** and **Down Arrow** keys do not function in this mode.

Only the value of an address when highlighted can be changed. The **CANCEL** key will cancel any edits made. The **ENTER** key and the **Left Arrow** or **Right Arrow** key accepts changes made to an address, and then moves to the next address. The **USE** key accepts the changes to the line and allows use of **Up Arrow** and **Down Arrow** keys.

The values of the following addresses can be edited: D, F, I, J, K, L, M, N, O, P, R, S, T, U, W, X, Y, and Z. For additional information, see "G-code and M-Code definitions", Page 108.

A highlighted line can be removed by pressing the **CLEAR** key.

The control will calculate the block form of the G-code program. The values for the block form can be viewed using the **Block Form** soft key. The block form can be edited and even inserted into the program by using the soft key **Insert G120**. Once inserted in the program (G120), the block form in the program is used. Remove the line containing the G120 to revert back to the calculated block form.
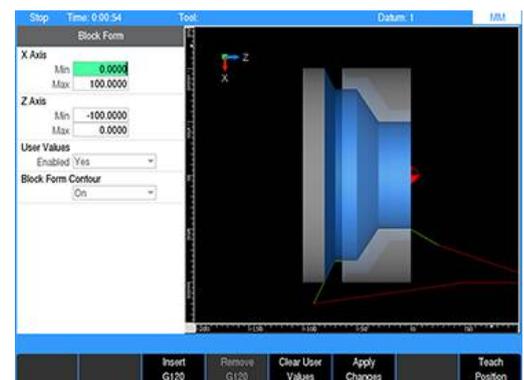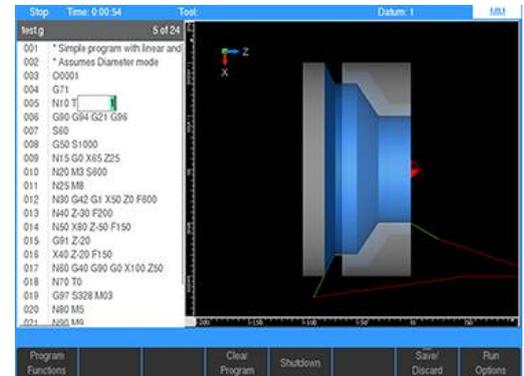
Loading a G-code program

The control has the ability to read and run G-code programs. Basic editing is also possible. It is important to fully test the G-code program before machining a part. A G-code program can be loaded in the same manner as TURNPWR programs. Once loaded, the control will indicate with an **x** the first error regardless of the cursor location. An error message also appears in the message line indicating that the line contains invalid code. The invalid code can be removed by editing the program outside of the control. The **Load** soft key opens programs that have already been saved. Use the following steps to load a G-code program from the User folder, a USB device, or from a network location.

▶ If a program is loaded, save the open program, it does not have to be cleared

▶ When a G-code program is not located in the current folder, select the location of the G-code program using the **Folder View** soft key, and **Change Window** soft key as needed

▶ Verify that **G-code programs** is selected in the **Program Type** popup menu. Only G-code programs stored in the selected folder will be displayed. For additional information, see "Program type filter", Page 83.

▶ Using the **Arrow** keys for navigation in the display window, highlight the G-code program to load

▶ Press the **Load** soft key, and verify that the desired program is now loaded

When loading a program from a network, or a USB memory device, first select the device from the directory tree and follow the same procedure as would be done from the User folder.

Running a G-code program

The control has the ability to read and run G-code programs. It is important to proof the G-code program before attempting to machine a part.

Verify the program in the CAD/CAM system that generated the program. The control will draw the program when Loaded. Finally, dry run the program to verify that the tool path, feeds and speeds are correct.

Using the tool table

Each "T" block refers to the corresponding number in the Tool Table. For example, T1 will cause the control to retrieve the tool length offset from tool 1 of the Tool Table. The control will then offset the spindle by this amount. T2 will cause the control to retrieve the tool length offset from tool 2 of the Tool Table, etc. For addional information, see "Tool table", Page 85.

It is very important not to have any tool length offsets in the Tool Table if the tooling is not repeatable. The user will need to set the Z datum after mounting the new tool. This is done before pressing **GO** to continue running the program.

Failure to maintain the Tool Table can cause unpredictable results. Verifying tool length offsets prior to program execution is strongly recommended.
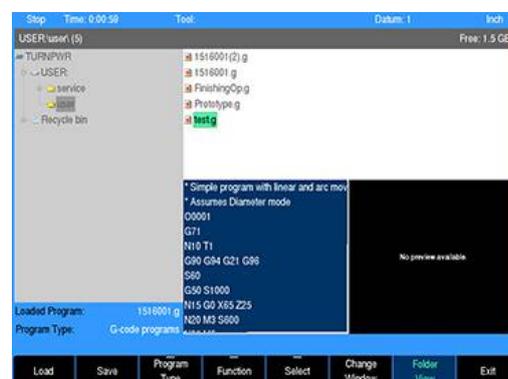
Starting or stopping a G-code program

Always start the program from a place in the program where the feed rate, X, and Z axes position are known, such as a tool step. Alternate starting points can be programmed by placing the proper code in the desired locations.

Pressing the **GO** key will cause the control to begin executing the G-code program. Always verify the program step highlighted is an appropriate starting point.

When a program is running, pressing the **STOP** key or the remote pendant will cause the program and all axis motion to pause. Pressing the remote pendant switch again or the **GO** key will cause the program to resume. Pressing the **STOP** key a second time will halt the program execution.

7.3 G-code and M-Code definitions

G-code

The following is a list of supported, and unsupported G-codes.

† Represents supported G-codes.

G-code	Description	Comment
G0 †	Linear Interpolation (Rapid)	These commands generate tool motion. The motion command applies to current and subsequent blocks containing at least one X or Z coordinate. The default motion command is a linear move at feed (G1).
G1 †	Linear Interpolation (Feed)	
G2 †	Circular Interpolation (CW)	
G3 †	Circular Interpolation (CCW)	
G4 †	Dwell	This command causes the system to pause for the specified period of time. The period of time is determined by the P address (in milliseconds) or X address (in seconds). T Address also specifies the time in seconds.
G20 †	Set Program Units (INCH)	Same functions as G70 and G71. These commands set the unit of measure. The setting applies to current and subsequent blocks. The default is G20 (INCH).
G21 †	Set Program Units (MM)	
G28 †	Return to Home Reference	The control does not have a method for establishing a "home" position. If one or more coordinates are specified in the block, the tool will rapid to that location. Program execution will continue with the next program block.
G30 †		
G40 †	Cancel Cutter Compensation	The control supports automatic cutter compensation. Enable cutter compensation using G41 (left) or G42 (right). Disable compensation using G40 (center).
G41 †	Cutter Compensation (Left)	
G42 †	Cutter Compensation (Right)	
G43	Tool Length Offset (+)	The control does not support tool length offsetting. The offset is retrieved from the control's tool library when a tool change is executed. These commands are ignored.
G44	Tool Length Offset (-)	
G49	Cancel Tool Length	
G50 †	Max spindle speed	Maximum spindle speed in CSS mode. Only active during the active program.
G53 †	Fixture Offset	Activate datum number assumed to have been previously defined in the Set Datum form in DRO mode. G53 O(Datum number)
G54	Work Coordinate System	The control does not support presettable work coordinate systems. G54 through G59 are ignored.
G55		
G56		
G57		
G58		
G59		
G61 †	Set "stop" Path Mode	These commands set the path mode. The setting applies to current and subsequent blocks. The default is G64 (continuous).
G64 †	Set "continuous" Path Mode	
G70 †	Set Program Units (INCH)	Same functions as G20 and G21. These commands set the unit of measure. The setting applies to current and subsequent blocks. The default is G70 (INCH).
G71 †	Set Program Units (MM)	

G-code	Description	Comment
G90 †	Set absolute positioning	
G91 †	Set incremental positioning	
G94 †	Feedrate in units per minute (ipm or mmpm)	
G95 †	Feedrate in units per revolution (ipr or mmpr)	
G96 †	Constant Surface Speed (CSS)	
G97 †	Fixed spindle speed (RPM)	
G120 †	Block Form	The BlockForm command is used to define a window in relation to the part zero. This is used by the Draw function to present a solid model of the raw stock. Block Form can be placed anywhere within the program and must be accompanied by all of the addresses.
		G120 X(xMax) Z(zMax) I(xMin) K(zMin)
G*		All other G codes not listed will generate a run-time error.

M-code

The following is a list of available M-Codes. Be advised that many M-codes are machine dependant, and often machine manufacturers will add, and/or remove some M-Codes.

† Represents supported M-codes.

M-code	Description	Comment
M* †		All other M codes not listed will generate a run-time error.
M0 †	Program Stop	This command pauses the program. Press GO to resume.
M1 †	Optional Program Stop	This command pauses the program if the Optional Stop run option is selected. Press GO to resume.
M2 †	Program End	This command stops the program after completing of the block. The cursor moves to the beginning of the program. The current settings are reset to default values.
M3 †	Spindle On (CW)	If spindle control hardware is present, the spindle is turned on or off automatically. If the hardware is not present, the operator is prompted to turn the spindle on or off.
M4 †	Spindle On (CCW)	
M5 †	Spindle Off	
M6	Tool Change	M6 is not necessary. A tool change occurs when the Tool Selection command is processed.
M7 †	Coolant On (Mist)	If the AMI hardware is present, the coolant is turned on or off automatically.
M8 †	Coolant On (Flood)	
M9 †	Coolant Off	
M30	Program End w/ Pallet Shuttle	The control does not support control of a pallet changer. This code has the same effect as M2.
M48	Enable Speed/Feed Override	It is not possible to disable feed rate override on the control. These commands are ignored.
M49	Disable Speed/Feed Override	
M60	Program Stop w/ Pallet Shuttle	The control does not support control of a pallet changer. This code has the same effect as M0.
F †	Set Feed Rate	The feed rate uses the current program units in effect (ipm or mmpm). The setting applies to current and subsequent blocks. The default is determined from the control's configuration setup.
I †	Arc center coordinate parallel to X axis	These are the arc center coordinates for G2 and G3 arcs. They are assumed to be programmed in incremental from the current tool position.
K †	Arc center coordinate parallel to Z axis	
N †	Line Number	Line numbering is optional and for readability only. The control does not make use of this information.
O †	Program Number	Used at the beginning of a program.
S †	Spindle Speed	The spindle speed is set to the specified speed (rpm). If the spindle is currently off (M5), it will not be turned on unless accompanied by a spindle direction block (M3 or M4). The setting applies to current and subsequent spindle direction blocks.
T †	Tool Selection	This command is used to select the active tool. The tool is specified by tool number. The tool attributes must be specified in the Tool table.

M-code	Description	Comment
		The program pauses at a tool command and prompts the user to complete the tool change. Ensure the tool is clear of the part or is at the appropriate position before pressing GO to resume.
X †	X Axis Coordinate	The coordinates represent the destination for the G0, G1, G2, or G3 command currently in effect. They use the current units (G20/21 or G70/71) and positioning mode (G90 or G91).
Z †	Z Axis Coordinate	

Additional G-code conventions

The following lists some of the expectations and limitations of programs ran in the control.

- Operator comments should be enclosed in parentheses
- An operator comment with "MSG" appearing within the text is considered a message. The text following "MSG" (up to 60 characters) is displayed to the operator at run-time. Program execution pauses until the operator acknowledges the message by pressing **GO**. Format is MSG (Operator message) or MSG ("Operator message").
- Parametric programming (use of variables or algebraic operations) is not supported
- Program delimiters ("%") are ignored. Text following the delimiter is ignored
- White space is ignored between address but not within a numeric value or message
- Tool length offset is read from the tool library. For example: In a G-code file, T1 will use the tool length offset from Tool #1 in the tool library. T2 will use tool length offset from tool library tool #2, etc.

8

Program steps and
cycles

8.1 Overview

Many of the program steps described in this section can be performed as one-step turning functions from the **DRO** screen or be included as steps in a program. Many of the dialogues are similar in the way that the data is entered. Much of the information in this chapter should be familiar after performing the demonstration program."Demonstration program"

Set Tool defines the tool and should appear as the first step in a program. A **Set Tool** step is required anywhere a tool change is required. The control will apply the latest tool setting to the program steps that follow.

⚠ CAUTION
Spindle speeds used in this manual are for example only. A spindle speed is dependent on many factors (i.e. material, if coolant is used, etc.). The speeds listed are not meant to be a recommendation.

8.2 Set Tool

The **Set Tool** step is used to program a tool change.

To set a tool:

▶ From the **PGM** screen, press the **TOOL** key

> The **Set Tool** dialogue opens

In the **Set Tool** dialogue, the tool settings may be entered directly into the fields, the current settings from a tool in the Tool Table may be imported into the dialogue, or a tool in the table may be programmed by its number.
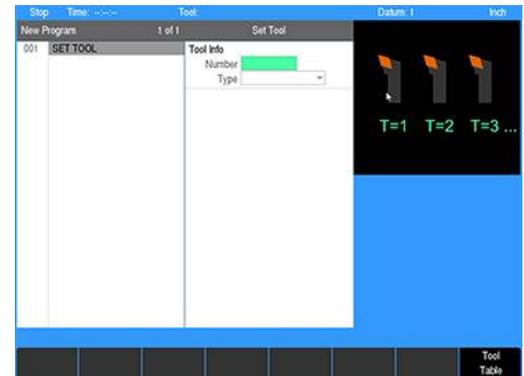
Specifying the tool by its tool table number

If the tool's number is known:

▶ Enter the number in the **Tool Number** field

▶ Press the **ENTER** key

If the tool's number is not known:

▶ Press the **Tool Table** soft key to open the tool table

▶ Highlight the tool to be used

▶ Press the **USE** key

> The tool's number is transferred to the **Set Tool** dialogue

🛈	When a tool is specified by number, only the tool number is saved to the program. The current settings for the specified tool are displayed, but not saved as part of the program step. The tool settings are retrieved from the tool table each time the **Set Tool** step is run.

Importing tool settings from the tool table

To import tool settings:

▶ Highlight any of the **Tool Size** data fields

▶ Press the **Tool Table** soft key to open the table

▶ Highlight the tool to be used

▶ Press the **USE** key

Only the tool's data is transferred to the **Set Tool** dialogue.

Entering the tool data

To enter tool data:

▶ Select the tool **Type** from the drop down menu

> The dialogue will show fields applicable to the **Type** selected

▶ Enter the tool **Size** information

The fields shown depend on the **Type** of tool selected.

▶ Enter the **Offset** to the tool tip in the **X** and **Z** axes

Entering spindle data

To enter spindle data:

▶ Select the **Spindle** direction from the drop-down menu

▶ Enter the **Spindle Speed**

The **Spindle Speed** may be entered as an **RPM** value or, for certain turning tools, a **Surface Speed** value. When a **Surface Speed** is specified, the spindle rpm is adjusted to maintain a constant surface speed at the tool tip.

If surface speed is specified:

▶ Enter the gear selection

When Constant Surface Speed is selected, the RPM is limited to the specified gear's speed range. To limit the maximum spindle speed to a lower RPM, set **Max Speed** in the **Job Setup** dialogue.

Tool position

Tool Position is optional. The current tool will move to the entered coordinates before prompting to change the tool. This allows the current tool to be moved away from the part for changing.

The control moves to the **Tool Position** before prompting the user for a tool change, making it unnecessary to program a separate **Position / Drill** step before the tool change.

NOTICE
Ensure that moving to this location does not interfere with the part or fixtures.

Entering a **Tool Position**:

▶ Highlight **Tool Position** and enter the location where it is best to make a tool change away from the part and fixtures

8.3   Positioning the tool

The **Position** function is used to move the tool to a specific location.

Entering the **Position**:

▶ From the **PGM** screen, press the **POS** key to open the **Position / Drill** dialogue

▶ Select **Position** using the soft keys, or from the drop-down menu

▶ In the **Point** field enter the **X** and/or **Z** axes coordinates

If a value is not entered for one of the coordinates, that axis will remain at its current location.
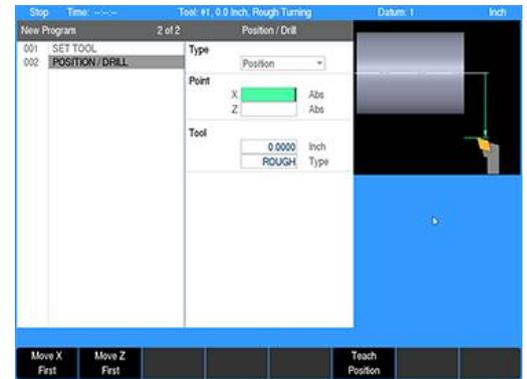
8.4 Defining a contour

A contour is any shape consisting of lines, arcs, and/or blends which form a continuous path. The control will indicate a contour with a solid line to the right of the applicable steps in the program list.

Line

Lines are defined by their **From** point (the point where they begin) and **To** point (the point where they end).

There are two ways you can program a line:

- With four coordinates (**X1**, **Z1**, **X2**, **Z2**)
- With three of the coordinates above (**X1**, **X2**, **Z2**, or **X1**, **Z1**, **X2**, etc.) and an angle (calculated from 9:00 position)

Choose a method based upon the information available from your print.

Entering data:

▶ From the **PGM** screen, press the **LINE** key to access the **Line** dialogue

▶ Or press the **Program Steps** soft key, then press the **Contour** soft key, and select **Line** from the popup menu

▶ Enter the beginning X and/or Z axes coordinates into the **From** field

▶ Enter the ending X and/or Z axes coordinates into the **To** field

▶ If one of the X or Z axes fields above was left blank, enter an angle

▶ Highlight the **Offset** field and press the **Left**, **Center**, or **Right** soft key

▶ Enter the feed rate

The default feed rate is what was entered in the **Job Setup** dialogue

▶ Press the **USE** key

> ℹ If the tool size, **Type**, and **Orientation** listed in the **Tool** fields are incorrect, change the tool settings before running the program.

Arc

An arc can be defined several ways:

- With a **From** point, **To** point, and a **Radius**
- With a **From** point, **To** point, and a **Center** point
- With a **From** point, **To** point, and a **3rd Point** along the arc
- With a **Start Point** to an **End Point** for a sweep angle

Choose a method based upon the information available from the part drawing. While programming, keep in mind that the arc's sweep angle is measured from the X axis.
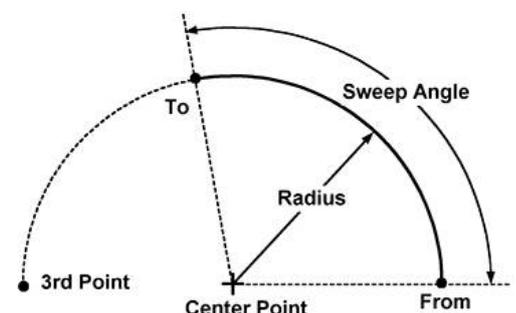
Entering data:

▶ From the **PGM** screen, press the **ARC** key to access the **Arc** dialogue

▶ Or press the **Program Steps** soft key, then press the **Contour** soft key, and select **Arc** from the popup menu

▶ Enter the beginning coordinates for the X axis (**X1**) and Z axis (**Z1**) in the **From** field

▶ Enter the ending coordinates for the X axis (**X2**) and Z axis (**Z2**) in the **To** field

▶ Enter the arc's radius

▶ Select the cutting direction. Press the **CW** soft key for a clockwise direction or the **CCW** soft key for a counter-clockwise direction.

▶ Highlight the **Offset** field

▶ Using the soft keys, select the tool offset: **Left**, **Center**, **Right**, **Inside** , or **Outside**

▶ Enter the feed rate

◼ **Center** field:

▶ Enter the center coordinate's position for the X and Z axes

◼ **3rd Point** field:

▶ Enter your 3rd coordinate's position for the X axis (**X3**) and Z axis (**Z3**)

◼ **Sweep Angle** field:

▶ Enter the **Sweep Angle**

Information that appears in blue has been calculated. If any of these values are already displayed in blue, then the control has enough data for the arc and has calculated the rest.

▶ Press the **USE** key
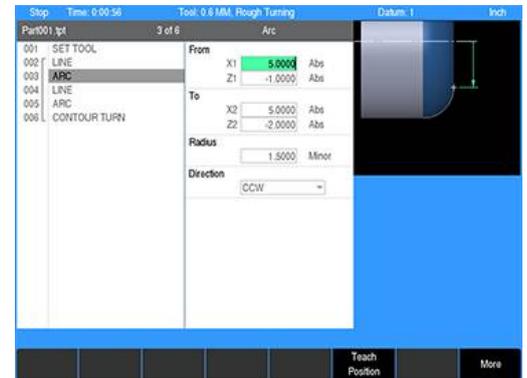
> **ⓘ** If the tool size and type listed in the **Tool** field are incorrect, change the tool settings before running the program.

Blend (Arc)

A **Blend** is an arc that connects two lines, two arcs or a line and an arc. A **Blend** is tangent to the adjacent steps. An inverted **Blend** is perpendicular to the adjacent steps.

The two steps to be blended can, but don't have to, intersect or touch. If they don't come into contact with each other, check that the blend radius is large enough to connect them.

▶ Highlight a step within your program where you want to place a **Blend**

> The **Blend** is added between this step and the previous step in the program

> 🛈 The blend step can be added prior to adding the connecting line in the program step, or between two existing connecting lines. When placed before the connecting line is added, it will not be displayed until the connecting line is placed in the program.

▶ From the **PGM** screen, press the **BLEND** key to access the **Blend** dialogue

▶ Or press the **Program Steps** soft key, then press the **Contour** soft key, and select **Blend (Arc)** from the popup menu

▶ Select the blend **Type**. A **Blend** is tangent to the two steps. A **Blend (Inv)** is perpendicular to the two steps.

> 🛈 Confirm that the blend **Type** is selected in the soft key.

▶ Check that the steps listed in the **From** and **To** fields are the steps to be blended. If they're incorrect, press the **CANCEL** key and highlight the appropriate step.

▶ Enter the blend's **Radius**

▶ Enter the **Feed** rate
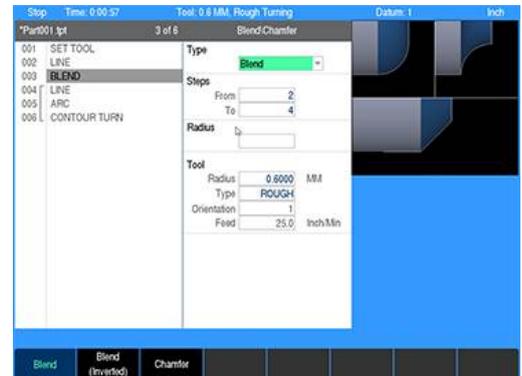
Blend (Chamfer)

A **Chamfer** is a bevel or line that is inserted between two lines to relieve sharp corners on a part. A **Chamfer** can be inserted between two intersecting lines whose steps are adjacent in the program.

▶ From the **PGM** screen, locate the lines where the **Chamfer** is to be inserted between

▶ Highlight the second line

▶ Press the **BLEND** key

▶ Or press the **Program Steps** soft key, then press the **Contour** soft key, and select **Blend (Chamfer)** from the popup menu

▶ Press the **Chamfer** soft key after the dialogue opens

▶ Check that the steps listed in the **From** and **To** fields are the steps to be blended. If they're incorrect, press the **CANCEL** key and highlight the appropriate step.

▶ For **Length1**, enter the distance from the common point to the **From** line

▶ For **Length2**, enter the distance from the common point to the **To** line

▶ Enter the **Feed** rate
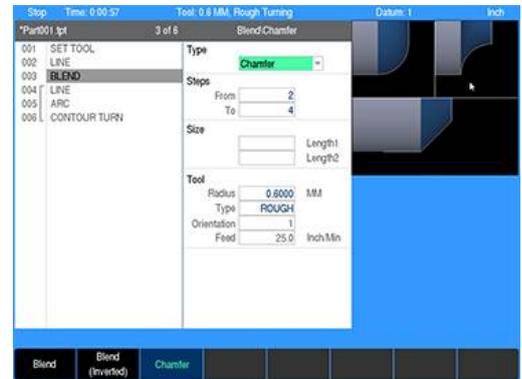
8.5 Turning cycles

Basic turning steps consist of shoulders and contours which are machined using a rough or finish turning tool.

Shoulder

A **Shoulder** step is used to remove material from a corner of the part.

▶ From the **PGM** screen, press the **TURN** key to access the **Turn** popup menu

▶ Select **Shoulder** from the popup menu
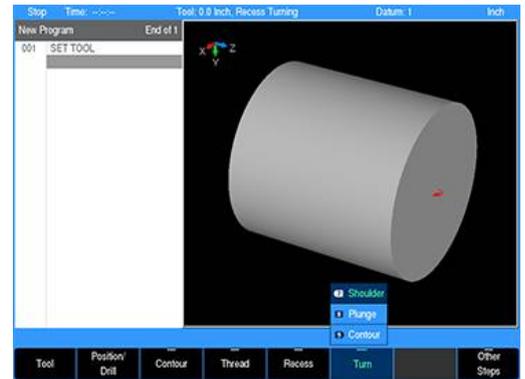
Entering data:

▶ Select how the shoulder is to be machined by selecting either **Turn** or **Face**. With **Turn**, the shoulder is machined in pass parallel to the centerline. With **Face**, the passes are made perpendicular to the centerline.

▶ Enter the **From** point. This is the starting surface (**X1**) and face (**Z1**) of the shoulder.

▶ Enter the **To** point. This is the ending surface (**X2**) and face (**Z2**) of the shoulder. The location of the **From** and **To** points determines the direction in which the shoulder is machined.

▶ Enter the **Pass** depth. This is the maximum depth the tool may cut on each pass.

▶ Enter the **Clearance** distance. This is the safety distance away from the part where the tool will start.

▶ Enter the roughing **Feed** rate in inches/min (ipm) or millimeters/min (mmpm). Enter 0 to skip running the roughing passes.

▶ Enter the angle of the shoulder's turning **Surface**. This surface is parallel to the centerline when the angle is 0.

▶ Enter the angle of the shoulder's **Face** surface. Only positive angles are allowed in this field.

▶ Enter the **Finish** allowance in the X axis. This is the amount of material left after the roughing passes to be removed on the finish pass.

▶ Enter the **Finish** allowance in the Z axis. This is amount to be removed on the finish pass.

▶ Enter the finish **Feed** rate. Enter 0 to skip running the finish pass.

▶ Add a round to the inside corner of the shoulder

▶ To add a chamfer or round at the start of the shoulder, select either **Chamfer** or **Round** for **Start Element**

▶ For **Chamfer**, enter the distance from the starting point to where the chamfer begins. For **Round**, enter the radius

▶ To add a chamfer or round at the end of the shoulder, select either **Chamfer** or **Round** for **End Element**

▶ For **Chamfer**, enter the distance from the ending point to where the chamfer ends. For **Round**, enter the radius.
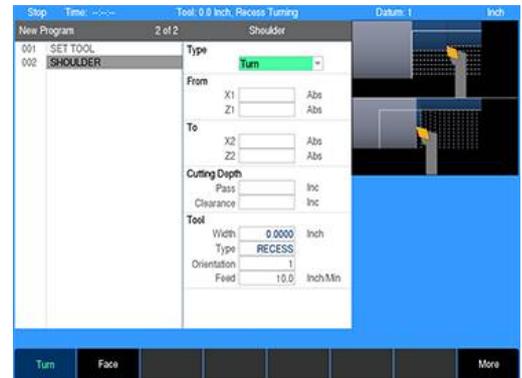
Tool path description:

▶ Machining of the shoulder begins at the corner defined by the **From** point and machines towards the **To** point

▶ The tool initially moves to the **From** point, offset by the **Clearance** distance

▶ The roughing passes are optional. If the roughing feed rate is 0, the rough passes are skipped.

▶ The amount of material machined on each rough pass is determined by the system and will not exceed the specified **Pass** value

▶ If a **Finish** allowance is specified, the programmed amounts are left on the X and Z surfaces to be removed on the finish pass

▶ The tool returns to the **Starting point** after completing the rough passes

▶ The **Finish** pass is optional. If finish feed is 0, the **Finish** pass is skipped.

▶ The tool returns to the **Starting point** after the **Finish** pass completes

> **ℹ** If the approach move interferes with the part or fixturing, it may be necessary to program a **Position** step to position the tool on the proper side of the part or fixture prior to the **Shoulder** step.

Plunge

A **Plunge** step is used to remove material from within the middle of a surface. The side walls may be limited by the cutting angle of the tool.

▶ From the **PGM** screen, press the **TURN** key to access the **Turn** popup menu

▶ Select **Plunge** from the popup menu
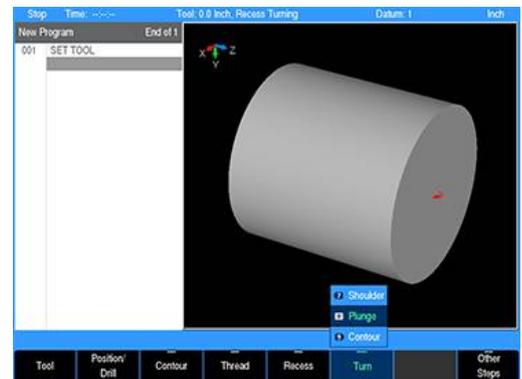
Entering data:

▶ Select which surface the recess is to be machined by selecting either **Turn** or **Face**

▶ Enter the **From** point. This is the starting surface (**X1**) and face (**Z1**) of the shoulder.

▶ Enter the **To** point. This is the ending surface (**X2**) and face (**Z2**) of the shoulder. The location of the **From** and **To** points determines the direction in which the shoulder is machined.

▶ Enter the **Pass** depth. This is the maximum depth the tool may cut on each pass.

▶ Enter the **Clearance** distance. This is the safety distance away from the part where the tool will start and finish.

▶ Enter the roughing **Feed** rate in inches/min (ipm) or millimeters/min (mmpm). Enter 0 to skip running the roughing passes.

▶ Enter the **Finish** allowance in the **X** axis. This is the amount of material left after the roughing passes to be removed on the finish pass.

▶ Enter the **Finish** allowance in the **Z** axis. This is amount to be removed on the finish pass.

▶ Enter the finish **Feed** rate. Enter 0 to skip running the finish pass.

▶ Enter the angle of the **1st Side** of the step. Only positive angles are allowed.

▶ Enter the angle of the **2nd Side** of the step. Only positive angles are allowed.

▶ Add rounds to the inside corners of the step

▶ To add a chamfer or round at the start of the plunge step, select either **Chamfer** or **Round** for the **Start Element**

▶ For chamfers, enter the distance from the starting point to where the chamfer begins. For rounds, enter the radius.

▶ To add a chamfer or round at the end of the plunge step, select either **Chamfer** or **Round** for the **End Element**

▶ For chamfers, enter the distance from the ending point to where the chamfer ends. For rounds, enter the radius.

Tool path description:

▶ Machining of the recess begins at side defined by the **From** point and machines towards the **To** point side

▶ The tool initially moves to the **From** point, offset by the **Clearance** distance

▶ The roughing passes are optional. If the roughing feed rate is 0, the rough passes are skipped.

▶ The amount of material machined on each rough pass is determined by the system and will not exceed the specified **Pass** value

▶ The **Plunge** is machined at the feed rate programmed in the step. When the tool plunges directly into the material, the full cut override is applied to the feed.

▶ If a **Finish** allowance is specified, the programmed amounts are left on the X and Z surfaces to be removed on the finish pass

▶ The tool returns to the **Starting point** after completing the rough passes

▶ The **Finish** pass is optional. If finish **Feed** is 0, the **Finish** pass is skipped.

▶ The tool returns to the **Starting point** after the **Finish** pass completes

---

> **ℹ** If the approach move interferes with the part or fixturing, it may be necessary to program a **Position** step to position the tool on the proper side of the part or fixture prior to the **Plunge** step.
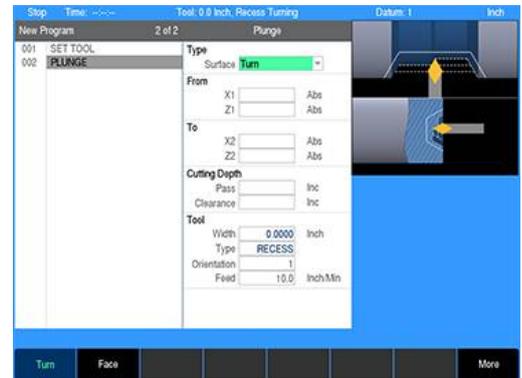

Contour

The **Contour** step is used to machine an irregular contour using a turning tool.

The **Contour** step must be programmed immediately following the last step of the contour path. The control indicates a continuous path with a solid line to the right of the applicable step in the program list.

▶ From the **PGM** screen, press the **TURN** key to access the **Turn** popup menu

▶ Select **Contour** from the popup menu
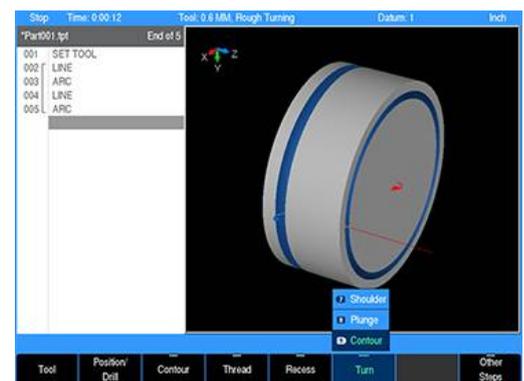
Entering data:

▶ The **Step Range** indicates the program steps which define the contour. This range is determined automatically.

▶ Select how the shoulder is to be machined by selecting either **Turn** or **Face**. With **Turn**, the shoulder is machined in pass parallel to the centerline. With **Face**, the passes are made perpendicular to the centerline.

▶ Select between **Straight**, **Constant** (chip load), and **Equidistant** for tool path type. With facing, only **Straight** tool path type is possible.

▶ Enter the **Start Point**. The tool moves to this point before machining the contour. Program a start point on the proper side of the contour at a location where machining can safely begin.

▶ Enter the **Pass** depth. This is the maximum depth the tool may cut on each pass.

▶ Enter the **Clearance** distance. This is the distance the tool is moved away from the material before moving to the start of next pass.

▶ Enter the roughing **Feed** rate in inches/min (ipm) or millimeters/min (mmpm). Enter 0 to skip running the roughing passes.

▶ Enter the **Finish** allowance in the **X** axis. This is the amount of material left after the roughing passes to be removed on the finish pass.

▶ Enter the **Finish** allowance in the **Z** axis. This is amount to be removed on the finish pass.

▶ Enter the finish **Feed** rate. Enter 0 to skip running the finish pass.

Tool path description:

▶ The tool initially moves to the **Start Point**, offset by the **Clearance** distance

▶ The roughing passes are optional. If the roughing feed rate is 0, the rough passes are skipped.

▶ The amount of material machined on each rough pass is determined by the system and will not exceed the specified **Pass** value

▶ The **Contour** is machined at the feed rate programmed in the step. When the tool plunges directly into the material, the full cut override is applied to the feed.

▶ If a **Finish** allowance is specified, the programmed amounts are left on the X and Z surfaces to be removed on the finish pass.

▶ The tool returns to the **Starting point** after completing the rough passes

▶ The **Finish** pass is optional. If finish **Feed** is 0, the **Finish** pass is skipped.

▶ The direction of the **Finish** pass matches the direction of the programmed contour

▶ The tool returns to the **Starting point** after the **Finish** pass completes

> ℹ️ If the approach move interferes with the part or fixturing, it may be necessary to program a **Position** step to position the tool on the proper side of the part or fixture prior to the **Contour** path.
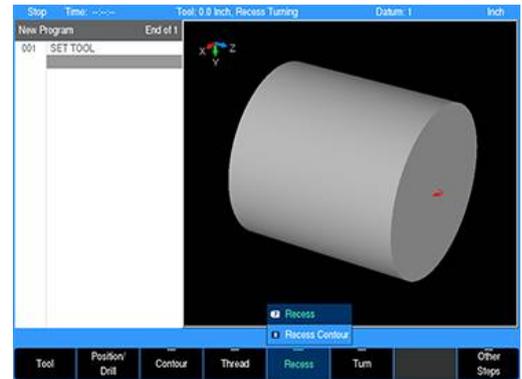
8.6 Recess cycles

Recessing steps are used to machine simple or irregular recesses into the part using a recess tool.

Recess

A **Recess** is used to remove material from within the middle of either surface of the part using a recessing tool.

▶ From the **PGM** screen, press the **RECESS** key to access the **Recess** popup menu

▶ Select **Recess** from the popup menu

Entering data:

▸ Select which surface the recess is to be machined by selecting either **Turn** or **Face**

▸ Select how the recess will be machined

▪ **Parallel**: The recess is machined from the first side towards the second side in a series of passes parallel to the surface

▪ **Perpendicular**: The recess is machined in a series of plunging moves perpendicular to the surface

▸ Enter the **From** point. This is the starting surface (**X1**) and face (**Z1**) of the recess.

▸ Enter the **To** point. This is the ending surface (**X2**) and face (**Z2**) of the recess. The location of the **From** and **To** points determines the direction in which the recess is machined.

▸ Enter the **Pass** depth. This is the maximum depth the tool may cut on each pass.

▸ Enter the **Clearance** distance. This is distance the tool is moved away from the material before moving to the start of next pass.

▸ Enter the roughing **Feed** rate in inches/min (ipm) or millimeters/min (mmpm). Enter 0 to skip running the roughing passes.

▸ Enter the **Finish** allowance in the **X** axis. This is the amount of material left after the roughing passes to be removed on the finish pass.

▸ Enter the **Finish** allowance in the **Z** axis. This is amount to be removed on the finish pass.

▸ Enter the finish **Feed** rate. Enter 0 to skip running the finish pass.

▸ Enter the angle of the **1st Side** of the recess. Only positive angles are allowed.

▸ Enter the angle of the **2nd Side** of the recess. Only positive angles are allowed.

▸ Add rounds to the inside corners of the recess

▸ To add a chamfer or round at the start of the recess, select either **Chamfer** or **Round** for the **Start Element**.

▸ For chamfers, enter the distance from the starting point to where the chamfer begins. For rounds, enter the radius.

▸ To add a chamfer or round at the end of the recess, select either **Chamfer** or **Round** for the **End Element**

▸ For chamfers, enter the distance from the ending point to where the chamfer ends. For rounds, enter the radius.

▸ When machining in the perpendicular direction, enter the stepover percentage or distance.
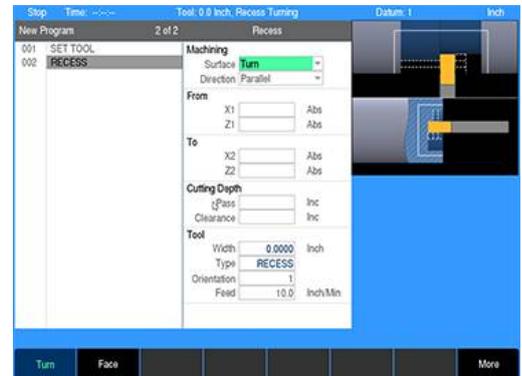
Tool path description:

▶ Machining of the recess begins at side defined by the **From** point and machines towards the **To** point side

▶ The tool initially moves to the center of the recess, offset by the **Clearance** distance

▶ The roughing passes are optional. When the roughing feed rate is 0, the rough passes are skipped.

▶ The depth of each rough pass is determined by the system and will not exceed the specified **Pass** value

▶ The **Recess** is machined at the feed rate programmed in the step. When the tool plunges directly into the material at full width, the full cut override is applied to the feed.

▶ If a **Finish** allowance is specified, the programmed amounts are left on the X and Z surfaces to be removed on the finish pass

▶ The tool returns to the **Start Point** after completing the rough passes

▶ The **Finish** pass is optional. If finish **Feed** is 0, the **Finish** pass is skipped

▶ The tool returns to the **Start Point** after the **Finish** pass completes

> **ℹ** If the approach move interferes with the part or fixturing, it may be necessary to program a **Position** step to position the tool on the proper side of the part or fixture prior to the **Recess** step.

Recess contour

The **Recess Contour** step is used to machine an irregular contour using a recess tool.

The **Recess Contour** step must be programmed immediately following the last step of the contour path. The control indicates a continuous path with a solid line to the right of the applicable step in the program list.

▶ From the **PGM** screen, press the **RECESS** key to access the **Recess** popup menu

▶ Select **Recess Contour** from the popup menu
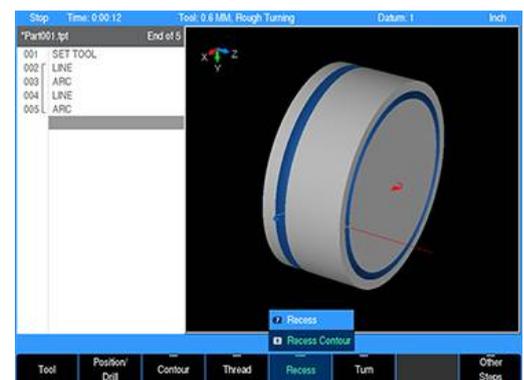
Entering data:

▸ Select which surface the recess is to be machined by selecting either **Turn** or **Face**

▸ Select how the contour will be machined

■ **Parallel**: The contour is machined from the first side towards the second side in a series of passes parallel to the surface

■ **Perpendicular**: The contour is machined in a series of plunging moves perpendicular to the surface

▸ Enter the **Start Point**. The tool moves to this point before machining the contour. Program a start point on the proper side of the contour at a location where machining can safely begin.

▸ Enter the **Pass** depth. This is the maximum depth the tool may cut on each pass.

▸ Enter the **Clearance** distance. This is the distance the tool is moved away from the material before moving to the start of next pass.

▸ Enter the roughing **Feed** rate in inches/min (ipm) or millimeters/min (mmpm). Enter 0 to skip running the roughing passes.

▸ Enter the **Finish** allowance in the **X** axis. This is the amount of material left after the roughing passes to be removed on the finish pass.

▸ Enter the **Finish** allowance in the **Z** axis. This is amount to be removed on the finish pass.

▸ Enter the finish **Feed** rate. Enter 0 to skip running the finish pass.

▸ When machining in the **Perpendicular** direction, enter the **Stepover Percentage** (of tool width) or distance

Tool path description:

▶ The tool initially moves to the **Start Point**, offset by the **Clearance** distance

▶ The roughing passes are optional. If the roughing feed rate is 0, the rough passes are skipped.

▶ The amount of material machined on each rough pass is determined by the system and will not exceed the specified **Pass** value

▶ The contour is machined at the feed rate programmed in the step. When the tool plunges directly into the material at full width, the full cut override is applied to the feed.

▶ If a **Finish** allowance is specified, the programmed amounts are left on the X and Z surfaces to be removed on the finish pass

▶ The tool returns to the **Start Point** after completing the rough passes

▶ The **Finish** pass is optional. If finish **Feed** is 0, the **Finish** pass is skipped.

▶ The direction of the finish pass matches the direction of the programmed contour

▶ The tool returns to the **Start Point** after the finish pass completes

> **ℹ** If the approach move interferes with the part or fixturing, it may be necessary to program a **Position** step to position the tool on the proper side of the part or fixture prior to the **Recess Contour** path.
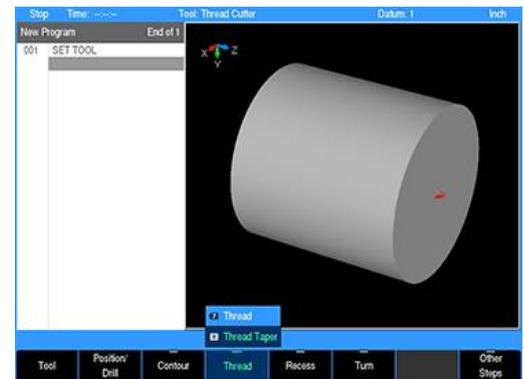
8.7 Thread cycles

Threading steps are used for machining a thread along a straight or tapered surface using a threading tool.

Thread (straight)

Cutting an inside or outside straight thread is possible with the **Thread** step.

▶ From the **PGM** screen, press the **THREAD** key to access the **Thread** popup menu
▶ Select **Thread** from the popup menu

Entering data:

▶ Enter the outside or inside **Diameter** of the thread
▶ Enter the **Z** axis **Begin** and **End** locations of the thread
▶ For English threads, enter the number of threads per inch (TPI). For metric threads, enter the pitch of the thread (mm).
▶ Enter the **Depth** of the thread as measured from the programmed diameter. If a value is not entered, **Depth** will default to the standard value based on the thread pitch. If calculated, the value is shown in blue text.
▶ Enter the maximum depth per machining **Pass**. When multiple passes are needed, the depth of each subsequent pass is reduced to maintain a constant load on the tool.
▶ Enter the **Clearance** distance. This is the distance the tool is moved away from the material before moving to the start of the next pass.
▶ Enter the number of follow-up passes, or **Air Cuts**, made after completing the final machining pass. The valid range is 0 to 10.
▶ Enter the **Lead-In** distance. This is the distance before the start of the thread at which the tool begins Z axis cutting motion. This distance allows the Z axis to accelerate to speed and synchronize with the spindle location before the start of the thread.
▶ Enter the **Runout** distance. This is the distance before the end of the thread at which the tool begins pulling out of the material. Enter 0 to machine to the programmed end of the thread before retracting straight out of the material.
▶ Enter the **Lead-Out** distance. This is the distance the tool continues moving past the programmed end of thread. This distance allows the tool to decelerate without affecting the main section of the thread.
▶ Enter the **Start Angle** of the thread. If cutting multiple threads, each thread should start at the appropriate angle relative to each other.

Tool path description:

▶ The tool orientation determines whether an inside or outside thread is being cut

▶ The tool initially moves to the start of the thread, offset by the **Clearance** distance

▶ Each pass begins away from the material by the specified **Lead-In** distance. This distance allows the tool to accelerate and synchronized with the spindle position.

▶ The amount of material machined on each rough pass is determined by the system and will not exceed the specified **Pass** value. The depth of each cut is reduced as the current depth increases.

▶ The tool begins to pull out of the material before reaching the end of the thread length as specified by the **Runout** distance

▶ Once the end of the thread is reached, the tool continues by the specified **Lead-Out** distance as the tool decelerates

▶ The tool rapids to the safety **Clearance** distance before moving to the start of the next pass

▶ After completing all cutting passes, additional **Air Cuts** (optional) are made as specified

▶ The tool returns to the **Start Point**

> **ℹ** If the approach move interferes with the part or fixturing, it may be necessary to program a **Position** step to position the tool on the proper side of the part or fixture prior to the thread step.

> **ℹ** The program may not be paused when running a thread step. If **STOP** is pressed, the tool will retract away from the part and the program will stop.
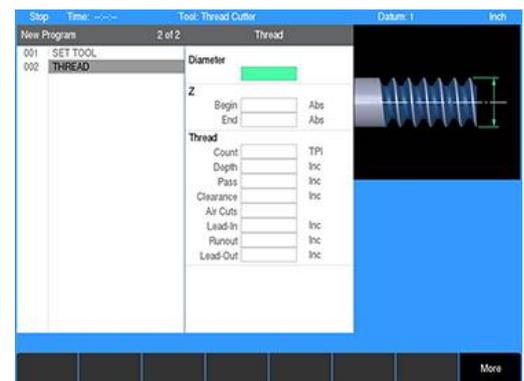
Thread (taper)

Cutting an inside or outside tapered thread is possible with the **Thread Taper** step.

▶ From the **PGM** screen, press the **THREAD** key to access the **Thread** popup menu

▶ Select **Thread Taper** from the popup menu
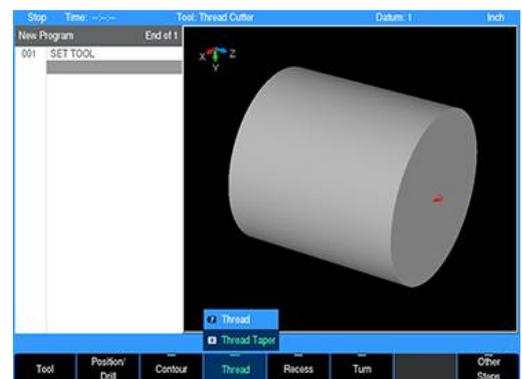
Entering data:

There are two ways to program a taper:

- ▪ With four coordinates (**X1**, **Z1**, **X2**, **Z2**) for the start and end points of the taper
- ▪ With three of the coordinates above (**X1**, **Z1**, **X2**, etc.) and the **Taper Angle**
- ▶ Enter the starting location of the thread in the **From** point
- ▶ Enter the ending location of the thread in the **To** point
- ▶ If specifying the thread by the angle, enter the **Taper Angle**
- ▶ For English threads, enter the number of threads per inch (TPI). For metric threads, enter the pitch of the thread (mm).
- ▶ Enter the **Depth** of the thread as measured from the programmed diameter. If a value is not entered, depth will default to the standard value based on the thread pitch. If calculated, the value is shown in blue text.
- ▶ Enter the maximum depth per machining **Pass**. When multiple passes are needed, the depth of each subsequent pass is reduced to maintain a constant load on the tool.
- ▶ Enter the **Clearance** distance. This is the distance the tool is moved away from the material before moving to the start of next pass.
- ▶ Enter the number of follow-up passes, or **Air Cuts**, made after completing the final machining pass. The valid range is 0 to 10.
- ▶ Enter the **Lead-In** distance. This is the distance before the start of the thread at which the tool begins the Z axis cutting motion. This distance allows the Z axis to accelerate to speed and synchronize with the spindle location before the start of the thread.
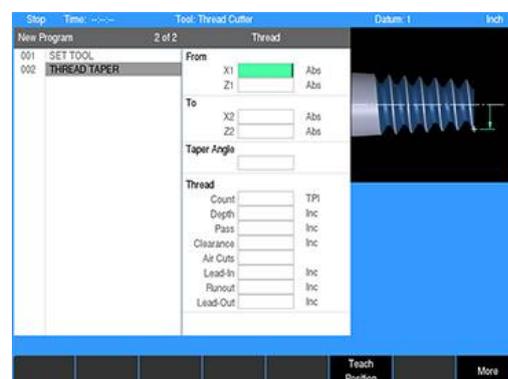- ▶ Enter the **Runout** distance. This is the distance before the end of the thread at which the tool begins pulling out of the material. Enter 0 to machine to the programmed end of the thread before retracting straight out of the material.
- ▶ Enter the **Lead-Out** distance. This is the distance the tool continues past the programmed end of thread. This distance allows the tool to decelerate without affecting the main section of the thread.
- ▶ Enter the **Start Angle** of the thread. If cutting multiple threads, each thread should start at the appropriate angle relative to each other.

Tool path description:

▶ The tool orientation determines whether an inside or outside thread is being cut

▶ The tool initially moves to the start of the thread, offset by the **Clearance** distance

▶ Each pass begins away from the material by the specified **Lead-In** distance. This distance allows the tool to accelerate and synchronized with the spindle position.

▶ The amount of material machined on each rough pass is determined by the system and will not exceed the specified **Pass** value. The depth of each cut is reduced as the current depth increases.

▶ The tool begins to pull out of the material before reaching the end of the thread length as specified by the **Runout** distance

▶ Once out of the material, the tool continues by the specified **Lead-Out** distance as the Z axis decelerates

▶ The tool rapids to the safety clearance distance before moving to the start of the next pass

▶ After completing all cutting passes, additional **Air Cuts** (optional) are made as specified

▶ The tool returns to the starting point

🛈 If the approach move interferes with the part or fixturing, it may be necessary to program a **Position** step to position the tool on the proper side of the part or fixture prior to the thread step.

🛈 The program may not be paused when running a thread step. If **STOP** is pressed, the tool will retract away from the part and the program will stop.

8.8 Drilling and boring

The **Position / Drill** function is used to perform a center drilling operation or boring with a boring bar.

▶ From the **PGM** screen, press the **POS** key to open the **Position / Drill** dialogue

▶ Select the job option to be used: **Drill** or **Bore** using the soft keys, or from the drop down menu

■ **Drill**: Basic drilling cycle used for center drilling

■ **Bore**: Used with a boring bar to turn an inside diameter

Drill

▶ In the **Z** field enter the **Begin** and **End** depths

▶ Enter the distance between each **Peck** (also known as Chip Break)

▶ Enter the Z axis **Feed** rate

▶ If you want the tool to periodically retract out of the part to remove material, enter the **Distance** between full retracts

▶ Enter the length of time (in seconds) the tool should **Dwell** (pause) after it has retracted out of the part

▶ Enter the length of time (in seconds) the tool should **Dwell** at the end depth before the final retract
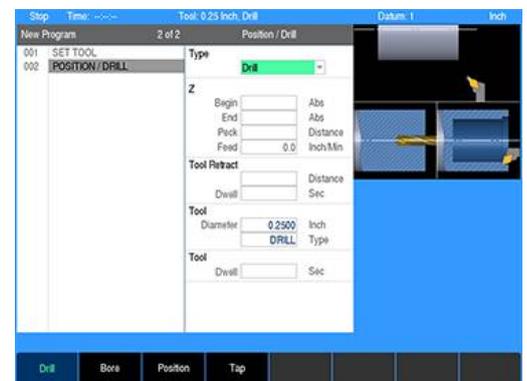
Bore

▶ In the **Z** fields enter the **Begin** and **End** depths

▶ Enter the **Feed** rate

▶ Enter the **Finish** allowance in the **X** axis. This is the amount of material left after the roughing pass to be removed on the finish pass.

▶ Enter the finish **Feed** rate. Enter 0 to skip running the finish pass.
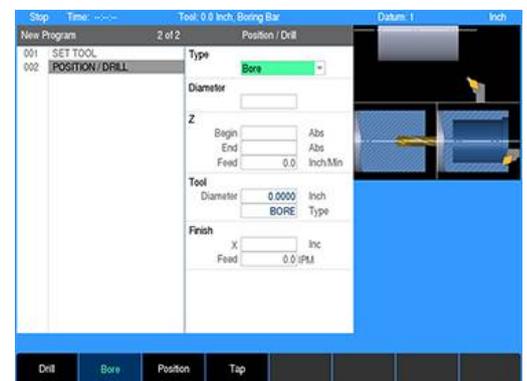
NOTICE

The tool rapids to the **Begin** depth before feeding into the material. Program the **Begin** depth with enough safety clearance.

8.9 Tapping

The **Position / Drill** function is used to cut an inside thread using a tapping tool. The tap cuts the thread while plunging into the part. The spindle direction is reversed, and the tap is retracted out of the part.

NOTICE

Tapping requires a close match between spindle and tool positions. For floating tapping, the spindle response must be within the limits of the floating tool holder. Incorrectly programming a tapping step may result in a damaged part or a broken tool.

NOTICE

The program may not be paused when running a tapping step. Pressing **STOP** could result in tool breakage.

▶ From the **PGM** screen, press the **POS** key to open the **Position / Drill** dialogue

▶ Select **Tap** using the soft key, or from the drop down menu

Entering data:

▶ Enter the **Clearance** distance. This is the safety distance away from starting Z location where the tool will start feeding.

▶ Enter the starting Z location of the hole

▶ Enter the ending Z location of the hole

▶ Enter the **Retract** distance. This is the safety distance away from starting Z location where the tool will stop after retracting from the hole.

▶ Enter the **Feed** rate

▶ Enter the time in seconds for the tap to **Dwell** at final depth. This time needs to be long enough to allow the spindle to stop before reversing direction and retracting out of the part.

Tool path description:

> The tool initially moves to a safe distance away from the hole as specified by the **Clearance** setting

> The tool begins feeding from the clearance location to the end depth

> After reaching the end depth, the spindle stops

> The tool and spindle pause for the length of time specified by the **Dwell** time

> The spindle direction is reversed and the tool feeds out of the part until the clearance location is reached

> If the Z retract distance is greater than the clearance distance, the tool rapids to the Z retract location
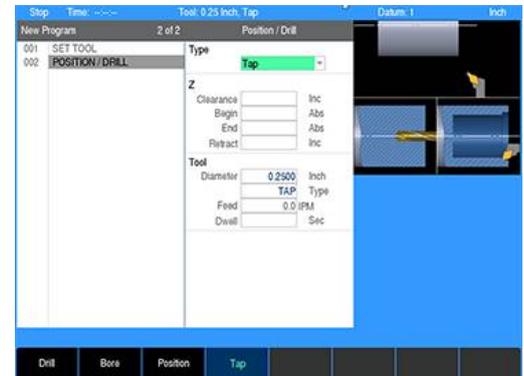
8.10   Other steps

The **Other Steps** soft key provides additional turning functions available with the control.

[Other Steps]

▶ From the **PGM** screen, press the **Program Steps** soft key, then press the **Other Steps** soft key to display the popup menu

Repeat

Using the **Repeat** step you can repeat a section of the program with any combination of X or Z offsets.

To program a repeat:

▶ From the **PGM** screen, press the **Program Steps** soft key

▶ Press the **Other Steps** soft key

▶ Select **Repeat** from the popup menu

▶ Enter the numbers of the **First** step and the **Last** step in the **Step Range** you want to repeat

▶ Enter the **Offset** for the **X** and/or **Z** axis. The offset is the distance between repeats.

▶ Enter the number of times you want to **Repeat** the original steps

> ⓘ   The steps being repeated must precede the **Repeat** step. The number of repeats must be 1 or more.

Repeat header

When running a program at the point of executing a **Repeat**, the header in the program list view will display the line number, step type, and iteration of the step.
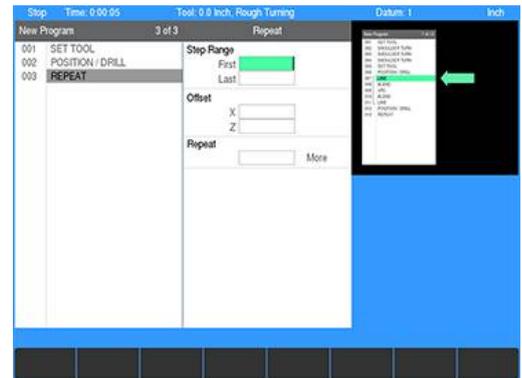
Comment step

With the control, you have the ability to insert messages anywhere within a program. These messages can be displayed during machining (at run-time) or as **Operator Intervention** Messages (OIM). These messages become operational steps within the program and communicate pertinent information like "Rotate Part" or "Activate Coolant".

For comments that don't require an operator intervention, select **No** when asked if you want the comment displayed at run-time, and the control will skip over them during machining. You can always retrieve the message by highlighting the comment step in your program steps list and pressing **ENTER**.

To program a comment step:

▶ From the **PGM** screen, place the cursor below the last step where the comment is needed

▶ Press the **Program Steps** soft key

▶ Press the **Other Steps** soft key

▶ Select **Comment** in the popup menu

▶ Enter your message. You may include up to 60 characters, mixing numbers, letters, spaces and symbols if needed. Press the **Alphanumeric Keyboard** soft key to chose alphabet characters. For additional information, see "Keyboard", Page 30.

▶ Highlight **Display at Run Time**

▶ Press the **Yes** soft key to display the message during machining or the **No** soft key if you don't want the message displayed

▶ Press the **USE** key

Auxiliary functions

▶ In **PGM** mode, press the **Program Steps** soft key

▶ Press the **Other Steps** soft key

▶ Select **Auxiliary Function** from the popup menu

▶ In the **Coolant** field select **Off**, **Mist**, or **Flood**

▶ Highlight the auxiliary relay(s) to be programed. The **Aux** numbers in each field correspond to the relay output on the AMI. Select from the **Off**, **Pulsed**, and **On** soft keys for each. When **Pulsed** is selected, another field allows the duration of the pulse length to be entered in seconds. Enter the required time. The range is 0.1 to 999.9.

Position	Description
On	The AMI output contacts will switch to an energized state
Off	The AMI output contacts will switch to their default states
Pulsed	The AMI output contacts will switch to an energized state for the length of time that is defined
Display User Prompt	The AMI allows the operator to pause the step before or after the relays are turned on or off

▶ If **Pulsed** is selected, then enter the amount of time (in seconds or tenths of seconds) for the relay to remain on. The maximum amount of time that a relay can pulse is 999.9 seconds.

> ⓘ The pulse length must expire before the program continues to run.

▶ Set the **Display User Prompt** for the step to pause before or after by pressing either the **Before Step**, or **After Step** soft key. If a pause is not to be included, press the **No** soft key. These can also be selected from the drop down menu in the field window.

> ⓘ The **Auxiliary Function** relays will not turn off automatically during an **Operator Intervention Message** (OIM).

▶ Set the **Set Continuous** field to **Yes** if the step is to be part of a continuous path. When set to **Yes**, the step does not break path continuity. The step will be run with the tool at the path depth. The **Auxiliary Function** step cannot be used to start or end a continuous path.

▶ Press the **USE** key

> ⓘ The **Auxiliary Function** requires the AMI hardware for these settings to take effect automatically, otherwise they will activate operator prompts that must be responded to during the machining operation.

Dwell

This is where the time is set for the machine to pause its movement, and stay in its current position for a programmed amount of time measured in seconds.

▶ From the **PGM** screen, press the **Program Steps** soft key

▶ Press the **Other Steps** soft key

▶ Select **Dwell** in the popup menu

▶ Enter the **Dwell** time in seconds, tenth of seconds can also be entered

▶ Press the **USE** key

> **ℹ** A **Dwell** of 0 will cause the program to pause until you press **GO**.

Reference point

Reference points are commonly used to identify center points, tangent points, and other part features. They can even be used as the basis for incremental moves.

As you program, note that placing a reference point in a continuous tool path will break the path. Otherwise, reference points do not affect your program's performance in any way in fact, the control will skip over them altogether when you run a program.

To program a reference point:

▶ From the **PGM** screen, press the **Program Steps** soft key

▶ Press the **Other Steps** soft key

▶ Select **Reference Point** in the popup menu

▶ Enter your reference point's position for **X** and **Z**

▶ Press the **USE** key

> The reference point will show graphically at the programmed location
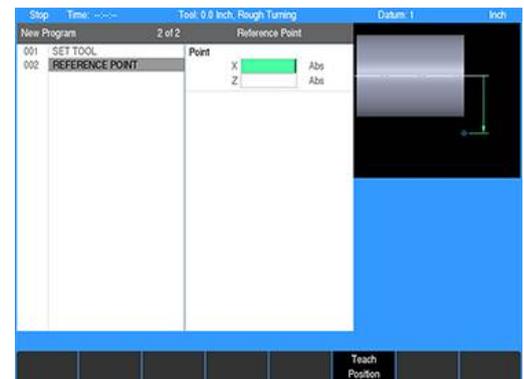
9

Demonstration
program

9.1 Overview

This section will introduce you to the control and familiarize you with how it operates.

The following steps and key stroke sequences will guide you through creating a demonstration program based on information from the part shown.
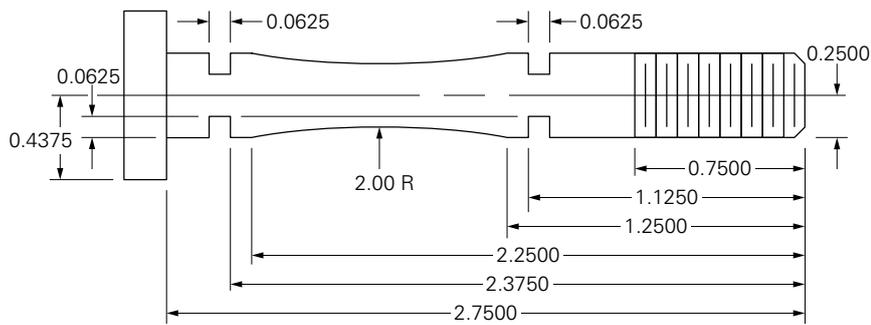
All dimensions are in inches.

9.2   Selecting a datum

This part is dimensioned from the right face of the part.

▶ Use the right face and centerline as the datum

The datum will be set before running the program.

9.3 Programming

▶ From the **DRO** mode, begin by pressing the **DRO/PGM** key to enter **PGM** mode

If a program is loaded, save the open program, it does not have to be cleared.

This example is done in Inch mode.

▶ Verify that **Inch** is selected

Selecting a tool

The first step for this program is to choose the tool that will be used to turn the shoulder.

For this example, the tools will be selected from the tool table.

For additional information, see "Tool table", Page 85.

To select a tool:

▶ Press the **TOOL** key to access the **Set Tool** dialogue

▶ Select **Tool Table** in the **Set Tool** dialogue

▶ Select or define the appropriate turning tool (**Rough Turning** or **Finish Turning**)

▶ Press the **USE** key

> The tool number and its dimensions and offsets will be displayed in the **Set Tool** dialogue

▶ Select **Forward** in the **Spindle** direction field

▶ Enter 500 in the **Speed** field

▶ Enter 2.00 in the **X To** field

▶ Enter 3.00 in the **Z To** field

▶ Press the **USE** key

For additional information, see "Tool position", Page 116.

Setting the Block Form

It is helpful to set the **Block Form** to match the size of the stock in the spindle so the graphic will more closely match the actual part.

To set the **Block Form**:

▶ Press the **VIEW** key

▶ Press the **Block Form** soft key

▶ Enter 0.5 in the **X Max** field

▶ Enter -3 in the **Z Min** field

For this example, turn on tool path optimization. The system will use the size of the **Block Form** and monitor material removal to optimize the tool path.

▶ Select **On** in the **Block Form Contour** field
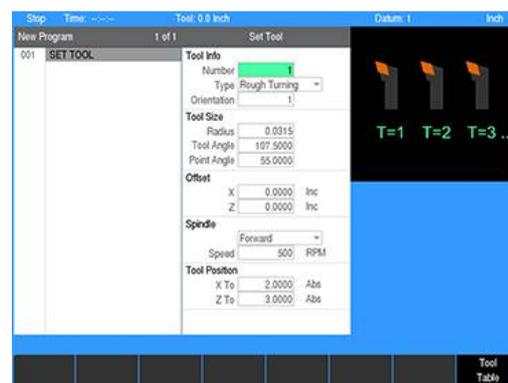
Programming a shoulder

For this example, the program will begin by turning the part to the body diameter using the **Shoulder** step. The tool will machine from right to left working toward the centerline.

▶ Press the **TURN** key

> A pop-up menu will appear for selecting the turning step

▶ Select **Shoulder**

> The **Shoulder** dialogue is displayed

▶ Press the **Turn** soft key to select longitudinal turning

▶ Press the **ENTER** key

Using the numeric keypad to enter the following information:

▶ In the **From:X1** field enter a radius of **0.5** to start away from the part

▶ Press the **ENTER** key

▶ In the **From:Z1** field enter **0**

▶ Press the **ENTER** key

▶ In the **To:X2** field enter the final radius of **0.25** inch

▶ Press **ENTER**

▶ In the **To:Z2** field enter **-2.75** inch

▶ Press **ENTER**

▶ In the **Cutting Depth**:**Pass** field enter **0.1** inch for the maximum depth per pass

▶ In the **Clearance** field, enter **0.1** inch for the distance the tool should retract away from the part between passes

The control has been factory set with a feed rate of 10 inches per minute. To change the feed rate:

▶ Enter the desired feed rate in the **Feed** field

▶ Press the **More** soft key

▶ Select the **Start Element**:**Type** field

▶ Press the **Chamfer** soft key to cut a chamfer on the right end of the part

▶ Enter **0.0625** inch in the **Size** field

▶ Press the **USE** key

Programming a tool for turning a contour

The next feature to be turned is the contour near the center of the part. For this, a tool change is necessary. Use a tool with a neutral orientation (2).

▶ Press the **TOOL** key to access the **Set Tool** dialogue
▶ Press the **Tool Table** soft key to open the Tool Table
▶ Select or define the appropriate tool with an orientation of **2**
▶ Press the **USE** key
> The tool number and its dimensions and offsets will be displayed in the **Set Tool** dialogue
▶ Select **Forward** in the **Spindle** field
▶ Enter **500** in the **Speed** field
▶ Enter **2.00** in the **Tool Position:X** field
▶ Enter **3.00** in the **Tool Position:Z** field
▶ Press the **USE** key

Programming a contour's profile

A contour's profile is defined by one or more connected lines, arcs, blends, or chamfers. For this example, the contour consists of a single arc.

▶ Press the **ARC** key
> The cursor will default to the **From:X1** field in the **ARC** dialogue

Use the numeric keypad to enter the following information:

▶ In the **From:X1** field enter **0.25**
▶ In the **From:Z1** field enter **-1.25**
▶ In the **To:X2** field enter **0.25**
▶ In the **To:Z2** field enter **-2.25**
▶ Enter **2** in the **Radius** field
▶ Select **CCW** in the **Direction** field
▶ Press the **USE** key

Programming a contour step

Although this is a simple profile which could be run as a single arc step, we will program a contour step to show some of its capabilities.

▶ Press the **TURN** key
▶ Select **Contour** from the popup menu
▶ Press the **Turn** soft key for longitudinal machining along the rotary axis
▶ Select the appropriate tool path **Type**
▶ Enter **0.75** for **Start Point:X**
▶ Enter **-1.25** for **Start Point:Z**
▶ Enter **0.1** for the **Cutting Depth**:**Pass**
▶ Enter **0.1** for the **Cutting Depth**:**Pass**
▶ Press the **USE** key
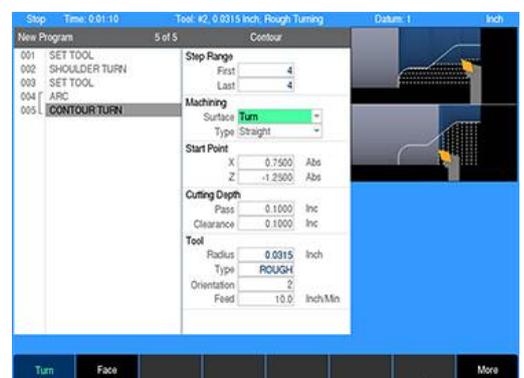
Programming a tool for recessing

Next, the o-ring grooves will be programmed. The grooves are machined using a recess tool and recess steps.

▶ Press the **TOOL** key to access the **Set Tool** dialogue

▶ Press the **Tool Table** soft key to open the Tool Table

▶ Select or define a recess tool with a width less than or equal to the groove width

▶ Press the **USE** key

> The tool number and its dimensions and offsets will be displayed in the **Set Tool** dialogue

▶ Select **Forward** in the **Spindle** field

▶ Enter **500** in the **Speed** field

▶ Enter **2.00** in the **Tool Position:X** field

▶ Enter **3.00** in the **Tool Position:Z** field

▶ Press the **USE** key

Programming a recess

▶ Press the **RECESS** key

▶ Select **Recess** from the popup menu

▶ Select **Turn** in the **Surface** field for a radial recess

▶ Select **Perpendicular** in the **Direction** field for lateral machining

Using the numeric keypad, enter the following information:

▶ Enter **0.25** in the **From:X1** field

▶ Enter **-1.125** in the **From:Z1** field for the groove edge location

▶ Enter **0.25 - 0.0625** in the **To:X2** field based on the groove depth

▶ Enter **-1.125 + 0.0625** in the **To:Z2** field based on the groove width

▶ Enter **0.125** in the **Cutting Depth:Pass** field for the depth per plunge

▶ Enter **0.1** in the **Clearance** field

▶ Press the **USE** key

Programming the second recess

▶ Press the **RECESS** key

▶ Select **Recess** from the popup menu

▶ Select **Turn** in the **Surface** field

▶ Select **Perpendicular** in the **Direction** field

Using the numeric keypad, enter the following information:

▶ Enter **0.25** in the **From:X1** field

▶ Enter **-2.375** in the **From:Z1** field for the groove edge location

▶ Enter **0.25 - 0.0625** in the **To:X2** field based on the groove depth

▶ Enter **-2.375 - 0.0625** in the **To:Z2** field based on the groove width

▶ Enter **0.125** in the **Cutting Depth**:**Pass** field for the depth per plunge

▶ Enter **0.1** in the **Clearance** field

▶ Press the **USE** key

Programming a tool for threading

▶ Press the **TOOL** key to access the **Set Tool** dialogue

▶ Press the **Tool Table** soft key to open the Tool Table

▶ Select or define a **Thread Cutter** tool

▶ Press the **USE** key

> The tool number and its dimensions and offsets will be displayed in the **Set Tool** dialogue

▶ Select **Forward** in the **Spindle** field

▶ Enter **100** in the **Speed** field

▶ Enter **2.00** in the **Tool Position:X** field

▶ Enter **3.00** in the **Tool Position:Z** field

▶ Press the **USE** key

Programming a thread step

▶ Press the **THREAD** key

▶ Select **Thread** from the popup menu

Using the numeric keypad, enter the following information:

▶ Enter **0.5** in the **Diameter** field for the threads outside diameter

▶ Enter **0** in the **Z**:**Begin** field as the location of the start of the thread

▶ Enter **-0.75** in the **Z**:**End** field as the location of the end of the thread

▶ Enter **12** threads per inch in the **Thread**:**Count** field

The thread depth is calculated based on the thread's pitch. This could be changed if necessary.

▶ Enter **0.02** In the **Thread**:**Pass** field. This is the maximum depth per pass. The system will gradually reduce the depth of each subsequent pass but will not exceed the value entered here.

▶ Enter **0.1** In the **Thread**:**Clearance** field. This is the distance the tool is moved away from the thread before moving to the start of the next pass.

▶ Enter **2** in the **Thread**:**Air Cuts** field. This is the number of extra passes made at final depth.

▶ Enter **0.25** in the **Thread**:**Lead-In** field. This is the distance from the start where the tool will begin a pass. This extra distance allows the tool to accelerate and synchronize with the spindle before reaching the start of the thread.

▶ Enter **0.03125** in the **Thread**:**Runout** field. This is the distance from the end of the thread at which the tool starts to retract out of the thread.

▶ Enter **0.05** in the **Thread**:**Lead-Out** field. This is the distance past the end of the thread during which the tool decelerates.
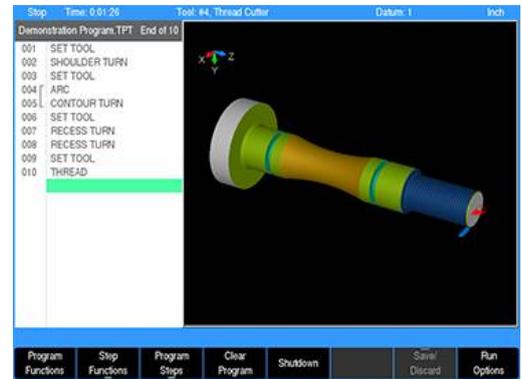
▶ Press the **USE** key
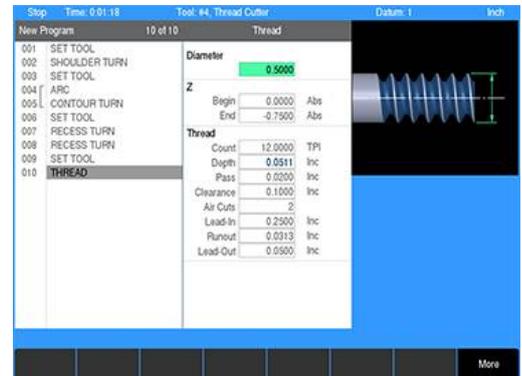
9.4   Saving the program

The program can be saved and given a name.

For additional information, see "Saving a program", Page 79.

▶ Press the **Save/Discard** soft key

▶ Select **Save** from the popup menu

▶ Name the program **Demonstration Program.TPT** using the popup keyboard

▶ Press the **Save/Create** soft key

9.5 Testing the program

A new program should always be tested first.

Run options

Two options are available:

- **Dry Run**
- **Single Step**

Dry Run will move the tool but no material will be cut.

Single Step will run the program one step at a time, then pause until **GO** is pressed to proceed with the next step.

To perform a test run:

▶ Press the **Run Options** soft key

▶ Selected **Dry Run** or **Single Step**

Dry run with tool movement

NOTICE
Before you press the **GO** key to begin the dry run, ensure that the tool will not touch the workpiece. To avoid tool interference, do one or more of the following: ▶ Remove the tool or workpiece ▶ Reestablish the Z datum away from the part

The complete program will be run, and the cutting of the part will be displayed graphically as the program steps are being performed.

The following steps will run the program graphically, and the tool will move.

To perform a **Dry Run**:

▶ Press the **Run Options** soft key

▶ Press the **Dry Run** soft key

▶ Highlight step **001** of the program

▶ Press the **Exit** soft key

▶ Press the **GO** soft key (the tool carriage will move)

9.6   Running the program

The first step in running a new program is to establish the datum.
The X datum is always at the part centerline.

▶ Place the workpiece into the spindle chuck

▶ From the **DRO** screen, ensure that the tool indicated in the DRO
  status bar is mounted in the tool post

▶ Press the **Move Tool** soft key

▶ Position the tool against the face of the part, which the print
  shows as the datum, using the arrow keys and the handwheel

▶ Press the **Exit** soft key to return to the **DRO** screen

▶ Press the **Datum** soft key

▶ Press the **Zero Z** soft key to establish the current position as the
  datum

▶ Press the **USE** key

9.7 Tool changes

Whenever you encounter a **Set Tool** step, the control will display the **DRO** screen and let you know which tool to mount.

▶ Press the **Move Tool** soft key
▶ Use the arrow keys or handwheel to move the tool away from the workpiece
▶ Mount the tool in the tool post
▶ Press the **GO** key to continue the program

9.8   Clearing the program

> ℹ️   If a program is loaded, and saved, it does not have to be cleared to open a new program.

To clear a program:

▶ Press the **DRO/PGM** key to return to **PGM** mode

▶ Press the **Clear Program** soft key

▶ Press the **Yes** soft key to confirm clearing the program

> The program is cleared from the screen, at any time another program can be loaded or a new program written

If for some reason a program cannot be cleared with the **Clear Program** soft key, there is an alternate method.

▶ Cycle the power to the console

▶ While at the **DRO** mode home screen, prior to finding home, press and hold the **CLEAR** key for 5 seconds

▶ Follow the instructions to clear the current program

10

Calculators

10.1 Overview

The built-in calculator is capable of handling everything from simple arithmetic to complex trigonometry, geometry, and RPM calculations.

The numeric keypad resembles a standard calculator with keys for numbers (**0** through **9**), four math function symbols (**+**, **-**, **×**, **÷**), a decimal point, and a positive/negative sign (**+/-**).

The calculator is accessible from most screens and fields.

10.2   Math calculator

The math calculator is used to apply the mathematical functions, **+**, **-**, **×**, and **÷**. It can be accessed at any time using the numerical key pad. These functions are normally accessed while in a field of a dialogue during **PGM** mode.

Using the calculator to insert data

The calculator can be used in the numeric field of dialogues.
Example:

The radius of the part is required in the data input field, and the part drawing provides a diameter dimension.

▶   Enter the diameter dimension into the dialogue field
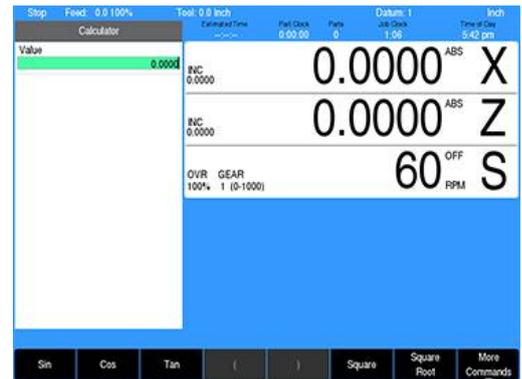
÷	▶ Press the **Divide** key
	> The ÷ symbol will appear in the field showing the calculation step
2	▶ Press the **2** key
	> This will also appear in the field as part of the calculation steps
ENTER	▶ Press the **ENTER** key
	> The result will be entered into the dialogue field

Trig functions

The Trig function calculator can be accessed at any time, during most operations.

To access the calculator's Trig functions:

CALC ▶ Press the **CALC** key

Trig ▶ Press the **Trig** soft key

▶ Press the soft key of the function to be used

Soft key	Description
Sin	**Sin**: Trigonometric Sine function
Cos	**Cos**: Trigonometric Cosine function
Tan	**Tan**: Trigonometric Tangent function
(	Open parenthesis **(**: Used as one of a pair to enclose a sum, or other expression
)	Close parenthesis **)**: Used as one of a pair to enclose a sum, or other expression
Square	**Square**: Number multiplied by itself
Square Root	**Square Root**: Square root of a number
More Cmds	**More Cmds**: (see next soft key display area)
ArcSin	**ArcSin**: Inverse function of Sine
ArcCos	**ArcCos**: Inverse function of Cosine
ArcTan	**ArcTan**: Inverse function of Tangent

NOTICE

When calculating the Sine, Cosine, or Tangent of an angle:
▶ Enter the angle first (in decimal degrees)
▶ Press the function soft key

Clearing an entry field

To clear the entry field:

CLEAR   ▶   Press the **CLEAR** key

Exiting the calculator

To exit the calculator:

CALC   ▶   Press the **CALC** key a second time

or

▶   Press the **CANCEL** key

10.3 Geometry calculator

The geometry calculator is used to construct **Point**, **Line** , and **Arc** entities which can then be transferred to the part program.

The geometry calculator contains a listing of geometry steps and a graphic visualization.
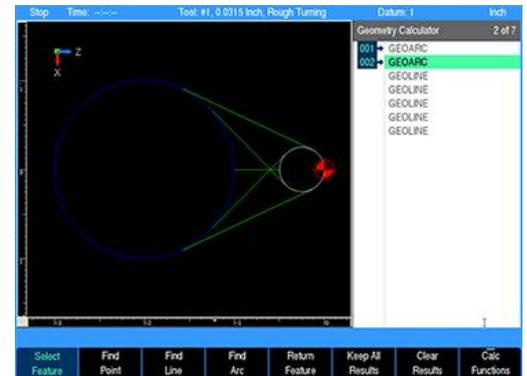
Accessing the geometry calculator

To access the geometry calculator:

CALC

▶ Press the **CALC** key

Geometry

▶ Press the **Geometry** soft key

Exiting the calculator

To exit the calculator:

CALC
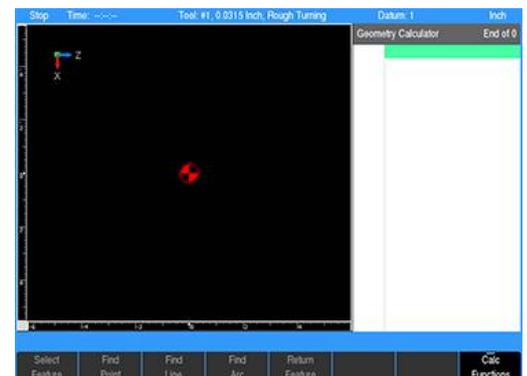
▶ Press the **CALC** key a second time

or

▶ Press the **CANCEL** key

Geometry calculator user interface

Layout

The geometry calculator's layout is similar to the part program in **PGM** mode. To differentiate it from the part program, the geometry step listing is on the right third of the screen and the graphic window is on the left two-thirds of the screen.

Listing

The geometry calculator listing contains all steps:

- Limited to geometry steps: point, line, and arc.
- No continuity indicator. Margin is used to display step number for reference only, and has no incremental references. Use the arrow keys for step selection.

To create a new Geo step:

▶ Press the **POS**, **LINE**, or **ARC** key

To edit an existing Geo step:

▶ Cursor to the step and press **ENTER** or **USE**

To delete an existing Geo step:

▶ Cursor to the step and press **CLEAR**

- Goto step number is supported.

Graphic window

The graphics window provides visual feedback:

- Limited to 2D Line view
- Extents determined by the size of the entered steps and calculated results
- The block size matches the extents plus the normal margin. The datum symbol is displayed at (0,0). A ruler is also shown.
- Highlight features:
  - Focus feature – Blue
  - Selected features – White
  - New features – Green
  - Normal features – Yellow
- A GeoPoint is displayed as a crosshair (×)
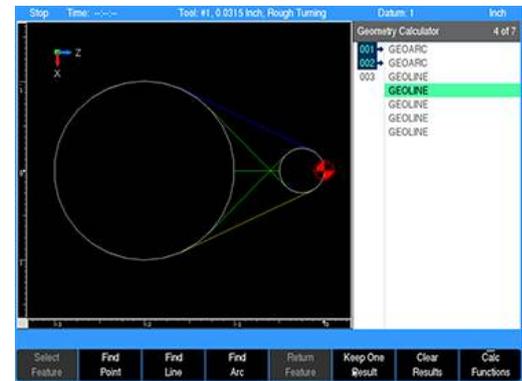
Adding geometry features to the listing

Geometry steps are limited to **GEOPOINT**, **GEOLINE**, and **GEOARC**. A new step is added by pressing the **POS**, **LINE**, or **ARC** key. As with a part program, the new step is added before the currently highlighted geo step.

GeoPoint

- Defined by a single coordinate

GeoLine

- Defined by combinations of From point, To point, and Angle

GeoArc

- Defined by combinations of From point, To point, Center point, 3rd point, Radius, Direction, and Sweep Angle. From and To points are optional (calculated at 3:00 position), otherwise, combinations follow programmed arcs.

Managing geometry results

Since there are generally more solutions possible than desired, all possible solutions are presented (without a step number). To keep the result required, use the **Up Arrow** or **Down Arrow** keys to move the cursor to that feature and press the **Keep One Result** soft key.

When the cursor is on an existing feature the following soft keys are available:

Soft key	Function
Keep All Results	Pressing the **Keep All Results** soft key will save all results found
Clear Results	Pressing the **Clear Results** soft key will clear all results not saved

When the cursor is on a feature that is not saved, the following soft keys are available:

Soft key	Function
Keep One Result	Pressing the **Keep One Result** soft key will save the highlighted result found
Clear Results	Pressing the **Clear Results** soft key will clear all results not saved

After selecting the features to be retained, press the **Clear Results** soft key to remove the remaining unselected results from the listing.

Existing or new features can be viewed (or edited) by moving the cursor to the step and pressing the **ENTER** key.

The results are cleared automatically when the **Find Function** soft key is pressed again.

Returning features to the part program

▶ Press the **Return Feature** soft key to transfer the current geo step to the part program.

The soft key is disabled when it is not possible to transfer the current step to the program.

■ If no program step is open, **Return Feature** adds an equivalent program step at the current location. The system returns to **PGM** view with the new step open.

■ If a program step is open, **Return Feature** (GeoPoint) returns the GeoPoint's coordinate to the current field (x and z)

■ If a program step is open, **Return Feature** (GeoLine or GeoArc) is only possible if the feature matches the step type

■ If the return type does not match the destination step/field, nothing is returned

Geometry calculator functions

▶ Pressing the **Calc Functions** soft key opens a pop-up menu for selecting a function

Clear calculator

■ Removes all Geo steps from the calculator

Loading and saving calculator contents

▶ Press **Save Calculator** to save the current GeoCalc session to internal memory

▶ Press **Load Calculator** to load the previously saved GeoCalc session into the calculator

**Loading the part program into the geometry calculator**

■ The **Load Program** function adds the equivalent geometry steps (**GEOPOINT**, **GEOLINE**, and **GEOARC**) from the program to the calculator listing at the current cursor location

■ Position / Drill steps are converted to GeoPoints

■ Turning steps are converted to profile GeoLines and GeoArcs

■ All other steps (tool, repeat, etc.) are not supported

Program example

In some cases, a part drawing may not provide all of the dimensions needed to program the part. In the following illustration, a coned end with rounded edges is specified. But the coordinates of the line from point **B** to point **C** are unknown. The line is tangent to the rounded tip of the part and tangentially rounds to the diameter of the body at a specified distance from the tip.

The circles **A** and **D** were added to the illustration to show how the line's endpoints will be calculated.
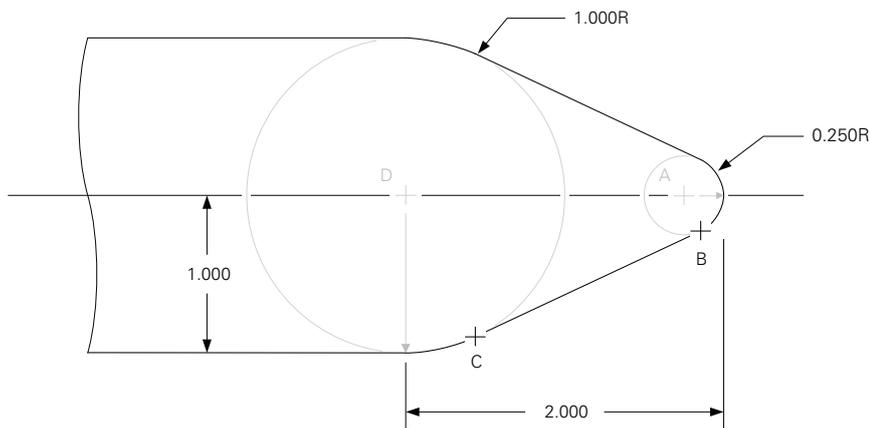
Locating points B and C

First program the circle that forms the rounded tip and the circle tangent to the part's outside diameter.

To locate points B and C:

- Enter an arc centered at **A**
- Enter an arc centered at **D**
- Find the line that is tangent to arcs **A** and **D**
- Calculate the endpoints
- Return the coordinates and features to the part program
  This example is explained fully in the following step by step illustrations.

Enter an arc centered at A

Arc **A** has a radius of 0.25, is centered on the part's centerline, 0.25 from the tip end.

To Enter an arc centered at **A**:

- ▶ Press the **ARC** key and enter the data in the **GEOARC** dialogue
- ▶ Enter **0.25** for the **Radius**
- ▶ Enter **0** in the **Center:X** field
- ▶ Enter **-0.25** for the **Center:Z** field
- ▶ Press the **USE** key to enter the data into the program

Entering the **From** or **To** points is not necessary. The **Center** and **Radius** is enough information to define a circle.

Enter an arc centered at D

Arc **D** has a radius of 1. The X center coordinate is 1 from the surface. The **Z** center coordinate is -2 from the tip end.

To enter an arc centered at **D**:

- ▶ Press the **ARC** key and enter the data in the **GEOARC** dialogue
- ▶ Enter **1** for the **Radius**
- ▶ Enter **0** in the **Center:X** field
- ▶ Enter **-2** for the **Center:Z** field
- ▶ Press the **USE** key to enter the data into the program
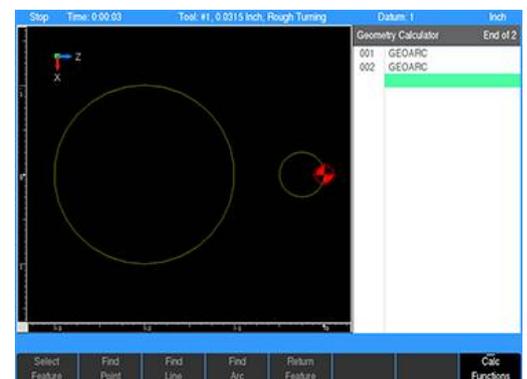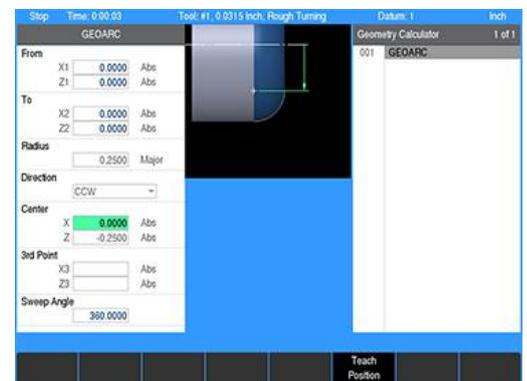- ➤ The **Geometry Calculator** screen now has the two circles displayed.

Finding the line that is tangent to arcs A and D

To find the line that is tangent to the two arcs:

▶ Highlight **001 GEOARC**

▶ Press the **Select Feature** soft key

> The arrow beside the step number indicates that the feature has been selected

▶ Highlight **002 GEOARC**

▶ Press the **Select Feature** soft key

▶ Press the **Find Line** soft key

> The **Geometry Calculator** will display all of the lines that are tangent to the two arcs

▶ Using the arrow keys, scroll through the list of found **GEOLINE**s until the desired one is highlighted in blue

▶ Press the **Keep One Result** soft key

> The **Geometry Calculator** will save the line as item **003**

▶ Press the **Clear Results** soft key to erase the remaining lines
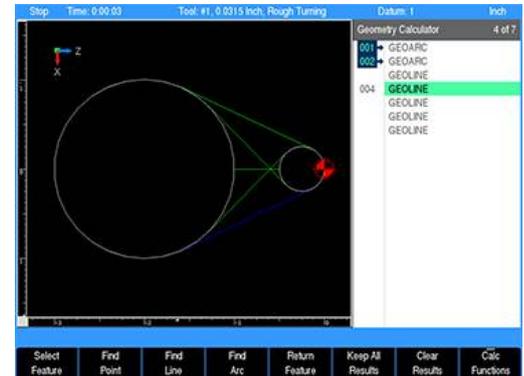
Calculate the endpoints

To find the endpoints of the tangent line:

To program the first arc, the endpoint of the arc is needed. This is point **B**, which is also the start point of line **BC**.

▶ Highlight **001 GEOARC**

▶ Press the **Select Feature** soft key

> The arc will be de-selected and the arrow will disappear

▶ Highlight **002 GEOARC** and de-select it

▶ Using the arrow key, scroll down until the **003 GEOLINE** is highlighted

▶ Press the **Find Point** soft key

▶ Two points will appear, one is the start point and the other is the end point

▶ Press the **Keep All Results** soft key to save both points

▶ Press the **CANCEL** key to return to the program
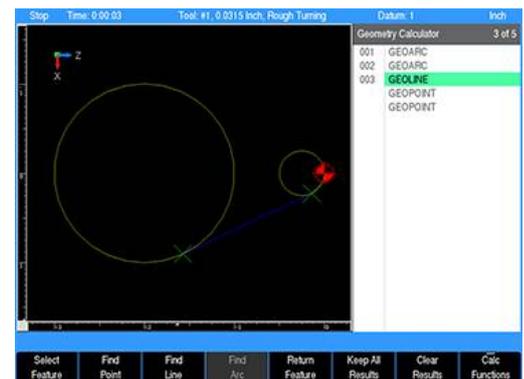
Return the arc coordinate

When you press the **Return Feature** soft key, the **Geometry Calculator** will transfer the feature you have highlighted into your part program.

> ℹ️ Program the tool step prior to proceeding.

To return the arc coodinate:

▶ Press the **ARC** key to begin programming the arc at the tip of the part

▶ Enter **0** in the **From:X1**

▶ Enter **0** in the **From:Z1**

▶ Highlight the **To** field. This is point **B**, one of the points found in the **Geometry Calculator**.

▶ Press the **CALC** key

▶ Press the **Geometry** soft key

▶ Highlight the **GEOPOINT** representing point **B** (step **005**)

▶ Press the **Return Feature** soft key

> The program screen will return, and point **B**'s coordinates will appear in the **To** field

▶ Enter **0.25** for the **Radius**

▶ Select **CW** for the **Direction**

▶ Press the **USE** key to accept the arc into the part program

Returning the line feature

The **Geometry Calculator** will also transfer the feature you have highlighted into your program at the current program if you do not have a step dialogue open.

To return the line feature:

▶ Press the **CALC** key

▶ Press the **Geometry** soft key

▶ Highlight the **GEOLINE** (step **003**)

▶ Press the **Return Feature** soft key

> The program screen will return and automatically add a line step to your program and place the line's information into the dialogue

▶ Press the **USE** key to accept the line into the part program

Finish the program

To finish the program:

▶ Press the **ARC** key to add the second arc to the part program

> The **From** point will default to the same coordinate as the previous line's endpoint. It is not necessary to open the **Geometry Calculator** since the arc's start point was filled in and the end point and radius are known from the print.

▶ Enter **1** for the **To:X2** point

▶ Enter **-2** for the **To:Z2** point

▶ Enter **1** for the arc **Radius**

▶ Press the **USE** key to accept the arc into the part program
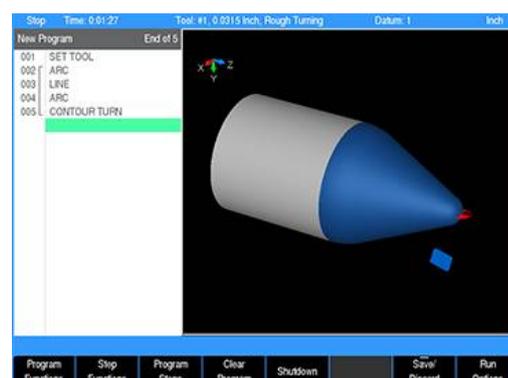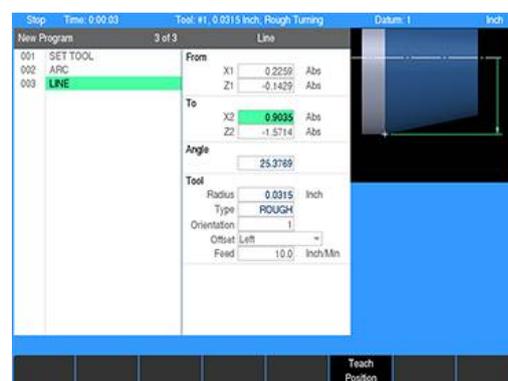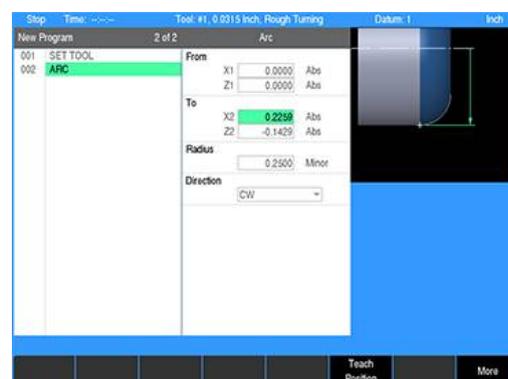
10.4   RPM calculator

The control includes an RPM calculator that performs RPM calculations. The RPM is calculated based on part diameter and surface speed.

Accessing the RPM calculator

To access the RPM calculator:

| CALC | ▶ Press the **CALC** key |

| RPM | ▶ Press the **RPM** soft key |

Exiting the calculator

To exit the calculator:

CALC	▶ Press the **CALC** key a second time
	or
	▶ Press the **CANCEL** key

Calculating the RPM

▶ Press the **Inch** or **MM** soft key
▶ Enter the diameter of the part in the **Diameter** field
▶ Enter the surface speed for the material and cutter type in the **Surface Speed** field
> The control will calculate the **RPM**

> **ⓘ**   If the cursor is in the **RPM** field of a **Set Tool** step before the RPM Calculator is opened, the control can transfer the calculation result directly into the **RPM** field.

To transfer the calculation result directly into the **RPM** field of a **Set Tool** step:

▶ Place the cursor in the **RPM** field of a **Set Tool** step
▶ Open the RPM Calculator and follow the steps above to calculate the **RPM**
▶ Press the **USE** key on the console keypad
> The calculated **RPM** will transfer to the **RPM** field of the **Set Tool** step
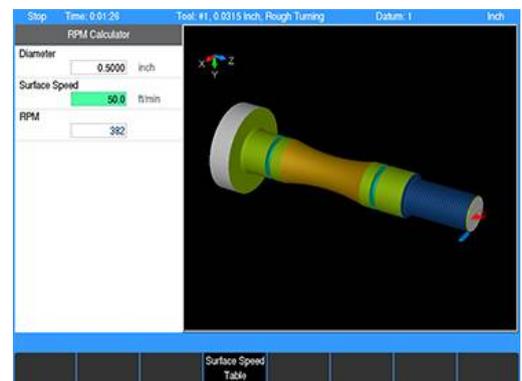
11

Setup

11.1 Overview

The control has a set of parameters that can be setup as default values. Some of these values can be overridden per the program step dialogue, while others remain in effect until changed in the **Job Setup** dialogue.

Job setup

Job Setup parameters can be changed to accommodate specific machining requirements for each job.

To access the **Job Setup** dialogue:

SETUP	▶ Press the **SETUP** key
	> The **Job Setup** dialogue is displayed

Installation setup

Once in the **Job Setup** dialogue the Installation Setup dialogue can be accessed.

To access the Installation Setup dialogue:

Install Setup	▶ Press the **Install Setup** soft key
	> The **Install Setup** dialogue is displayed

Installation setup parameters are established during the initial installation and typically are not changed. The installation setup parameters are protected by a passcode. These parameters are described within the control's Installation Manual.

Tool table

A brief overview of the tool table is provided later in this chapter. In the table, up to 100 tools can be entered and used in any program that would required that tool. For additional information, see "Tool table", Page 85.

To access the tool table:

Tool Table	▶ Press the **Tool Table** soft key
	> The **Tool Table** is displayed

Message log

The message log records all errors, and retains up to the last 128 messages in the control until it is deleted. A message in the log can be highlighted, and an explanation of the message can be displayed by pressing the **Info** soft key.

ℹ	The message log entries may also be cleared from the DRO or PGM screen by pressing the **0** key.

To access the message log:

Message Log	▶ Press the **Message Log** soft key
	> The **Message Log** is displayed

11.2 Job setup

The **Job Setup** dialogue is displayed by default when the **SETUP** key is selected.

In the **Job Setup** dialogue there are parameters that can be set which the control will default to when not overridden by another setting.

- **X Axis**: The mode of the system can be set to either **Radius** (default) or **Diameter** mode. All programming input and position display for the X Axis will adhere to the mode selected here.
- **Feed Rate**:
  - The **Default** setting is the setting that the control will use as the default in a program unless it is defined explicitly by a step that is in the program.
  - The **Dry Run** feed rate specifies the feed rate to be used when running a program in Dry Run mode with no coolant or spindle motion. For additional information see "Dry run", Page 102.
- **Feed Rate Override**:
  - The **Full Cut** field override percentage is applied to a step's programmed feed rate when the tool makes cuts greater than the programmed step over percentage
  - The **Max** field determines the feed potentiometers maximum override percentage. The acceptable range is 150% to 200%. The last click of the dial will allow the **Max** setting. The second to the last click will always be 140%.
- **Spindle**:
  - The **Max CSS RPM** specifies the maximum RPM to allow the spindle to operate at when in Constant Surface Speed mode (equivalent to G50 in ISO programming)
- **Display**:
  - **Stepover**: This defines the step over amount for machining operations as either a percentage of the tool's width or as a distance
  - **Angles**: The setting can either be in **Degrees** or **DMS**
  - **Res. Inch** and **Res. MM**: Select the desired display resolution for the DRO display.
  - **Help Imgs**: The setting can either be set to **On** to always show the field graphic assistant in a step form. Or set to **Automatic** for displaying additional fields when available in place of the graphic assistant.
- **Job Clock - Parts Counter**: For setting the parts counter and stopping or starting the Job Clock. The Job Clock will provide an estimate of time to machine the part and overall Job time, see "Job clock", Page 177.
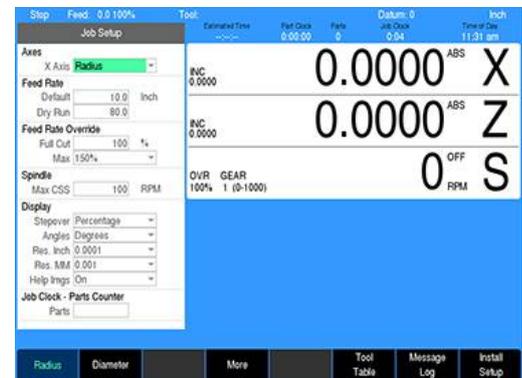
X Axis

This setting defines the active mode of the system for the X Axis, either **Radius** or **Diameter**. All programming input data and position display for the X Axis will follow this mode setting. When set to **Diameter**, the DRO and status bar will indicate this by displaying a Ø symbol.

Feed rate

The parameters set here will be used as the default by the control and will only change if it is modified in any of the turning functions dialogue.

▶ Highlight the **Default** field, and specify the inch per minute

▶ Select the unit of measure, **Inch** or **MM** by pressing the appropriate soft key

▶ Highlight the **Dry Run** field

▶ Enter the inches per minute that is required. The default is 80 IPM.

▶ Highlight the **Full Cut** override field

▶ Enter the override percentage to be applied to the feed rate when making full cuts

▶ Highlight the **Max** override field

▶ Enter the maximum percent override at full potentiometer

Display

▶ Highlight the **Stepover** field, and either select **Percentage** to use a factor of the tool's radius, or select **Distance** to use an exact distance

▶ Highlight the **Angles** field and select **Degrees**, or **DMS**.

▶ Highlight the **Res. Inch** and **Res. MM** fields to select the desired display resolution for the DRO display

▶ Highlight the **Help Imgs** field and set to either**On** to always show the field graphic assistant in a step form or set to **Automatic** for displaying additional fields when available in place of the graphic assistant

Parts counter

The parts counter appears on the status bar when in DRO mode or running a program. Use this feature to track how many times the current program has run and how many parts have been completed.

Here, setting the parts counter and stopping or starting the **Job Clock** are setup. Using setup, the number of parts for the parts counter can be set or reset.

▶ Preset a starting count by entering the value in the **Parts** field

> **ℹ** The parts counter resets to 0 when a new program is loaded.

Job clock

The **Job Clock** begins operation after the control finishes powering up and is displayed on the status bar. The **Job Clock** shows the total elapsed time for the current job. The display begins with minutes, then hours, and if required, days. The **Job Clock** can be paused, resumed, or reset in the parts counter section.

▶ Press the **Pause Clock** soft key to pause the **Job Clock**

▶ Press the **Resume Clock** soft key to resume the **Job Clock** after it has been paused

▶ Press the **Reset Clock** soft key to stop, and reset the **Job Clock** to 0.00

There is no control requirement for these features. They are provided to the operator for job assistance and informational purposes.

11.3 Tool table

The control contains a tool table that can store the tool data for up to 100 tools. For additional information, see "Tool table", Page 85.

> **ℹ** Set up the tool table so that the tools used most often appear first.

The tool table can be accessed a number of different ways. The following is one method to access the tool table:

TOOL
▶ Press the **TOOL** key

Tool Table
▶ Press the **Tool Table** soft key

11.4 Message log

The control includes a built-in message log that will record messages. If an error is detected, the control will automatically record the date, along with a brief description of the error. Only the last 128 errors will be saved.

If an error is present that prevents running of another program the front panel indicator light will flash.

▶ Press the **Info** soft key to display additional information on the highlighted message which will open in its own display window. The additional information includes Cause and suggested corrective Action.

▶ Press the **Detail** soft key to display the detailed information available on the source of the error. This will also open in its own display window.

▶ Press **Log Files** to generate Service Files. For additional information, see "Service files", Page 180.

▶ Press **Change Window** to toggle focus between the list of messages on the top section and the **Info** or **Details** section on the bottom

▶ Press the **Delete** soft key to clear the highlighted message

▪ Pressing the **Delete All** soft key will clear all messages

▪ Pressing the **Delete All** soft key will also exit the Message Log dialogue

▪ Pressing the **Delete** soft key will only exit the dialogue if there was only one error message

▶ To exit the Message Log, press the **Exit** soft key

NOTICE
A quick method for clearing all messages in the log is to press the **0** on the numeric keypad
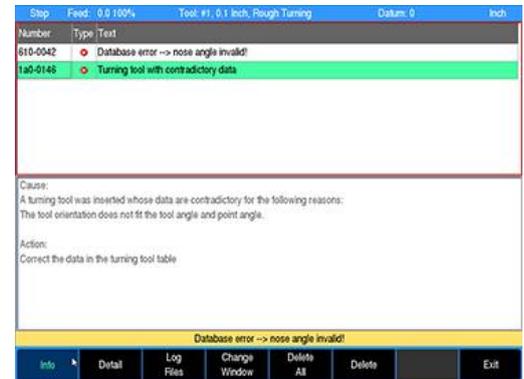
11.5   Service files

Occasionally, it may be necessary to provide HEIDENHAIN Corporation with diagnostic files to document or troubleshoot a problem with the machine or user experience. These are known as Service Files.

- The service files are created by pressing the **Log Files** soft key and then pressing the **Service Files** soft key (accept the default name and press the **Ok** soft key). A dialogue appears as the service files are created. Another dialogue appears once the service files have been successfully created.

- The service files take less than a minute to generate. If the control software terminates unexpectedly, the service files will automatically be generated by the system as well. Once generated (automatically or manually), the resulting ZIP file can be found at V:\service\.

- Automatically generated service files will have the name "nc_auto_service.zip".

- Manually generated service files will have the name "service.zip" by default.

- Multiple service files can be generated. When this occurs, existing ZIP files at V:\service\ will be renamed as service2.zip, service3.zip, etc. No number at the end of the filename indicates the most recently generated file (i.e. service.zip). After that, the lower the number, the more recent the file (i.e. service1.zip was generated more recently than service2.zip).

- Use the cut/copy/paste features in **Program Functions** to move the ZIP to a USB or network drive such that it can be transferred to HEIDENHAIN Corporation upon request.

To keep the maximum amount of free space available on the internal drive, it is advised to delete old service files from the control after they have been safely transferred to an external medium.

12

Machine functions

12.1   Machine functions

The control software can be configured to access the **Machine Functions** menu. This menu includes a set of soft keys that provide manual access, and control of some features specific to the machine.
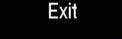
The machine tool builder enables the soft keys when the machine is configured.

| Machine Functions | ▶ Once enabled, the menu can be accessed while at the **DRO** screen, or while a running program is paused, by pressing the **Machine Functions** soft key |

▶ To exit the **Machine Functions** soft keys and return to the main operations soft keys, press the **CANCEL** key

A program run cannot be started, and a running program cannot be continued, until after exiting the **Machine Functions** menu

The Machine Functions Menu has the following soft keys:

Soft key	Description
Fwd (S)	**S_FWD**: Turns the spindle motor On in the Forward direction
Rev (S)	**S_REV**: Turns the spindle motor On in the Reverse direction
Off (S)	**S_OFF**: Turns the spindle motor OFF
Cool Off	**Cool**: Toggles the Coolant Flood circuit ON or OFF
Mist Off	**Mist**: Toggles the Coolant Mist circuit ON or OFF
Custom Cycles	**Custom Cycles**: Available only on Demo Consoles and Windows Simulators by default, as an example for defining/using Custom Cycles in the machine
Auxiliary Functions	**Auxiliary Functions**: Switches the menu to the Auxiliary Functions menu that provides the following soft keys:
Aux Output 1 Off	**Aux1**, **2**, **3**, and **4**: Toggles the Auxiliary Outputs 1 through 4 ON or OFF
Exit	**Exit**: Switches the menu back to the previous set of soft keys

13

Software update

13.1 Updating the system software

If desired as a precautionary method, a backup of the control's internal drive can be made prior to updating the system software, by using the included USB Recovery Drive (ID 1127022-02).

> 📖 For additional information refer to the **USB Recovery Drive User's Manual** (ID 1313217-2x).

Use a blank USB memory stick (1 GB or larger) to update the software. Do not use any memory stick with a smaller storage capacity.

A ZIP file must be obtained which includes the software installation files. The ZIP can be extracted to the root of the empty USB drive. When the control software is started, if a USB drive is present which contains a software update, the control will automatically start a software update.

The required folder structure for the software update on the USB drive is as follows:

<DRIVE>:\install\setup.zip.

▶ Shut down the control software using the normal method of pressing the **Shutdown** soft key

▶ Press the **Restart** soft key to initiate the reboot sequence

▶ Insert the USB memory device containing the folder named "install" into an available USB connection on the console

> After starting the control, the Software Update screen will appear with a confirmation box

▶ If necessary, use the **Up Arrow** or **Down Arrow** keys to select the language (e.g. English) and press **ENTER**

> A confirmation box will appear

▶ If necessary, use the **Left Arrow** or **Right Arrow** keys to select **YES**, then press **ENTER**

> The software update will now be done automatically, taking about 10 minutes

> When the update is complete, it will prompt you to remove the USB drive and restart

▶ Remove the USB drive and press **ENTER** to restart

> After the system has restarted, it will start up normally to the control software

If any issues are encountered, refer to the **Technical Manual** (ID 1034383-2x) or contact your service representative for assistance.

> ℹ️ If the USB memory stick used for installation is not removed, then the prompt to install the software will continue to appear on subsequent power cycles.
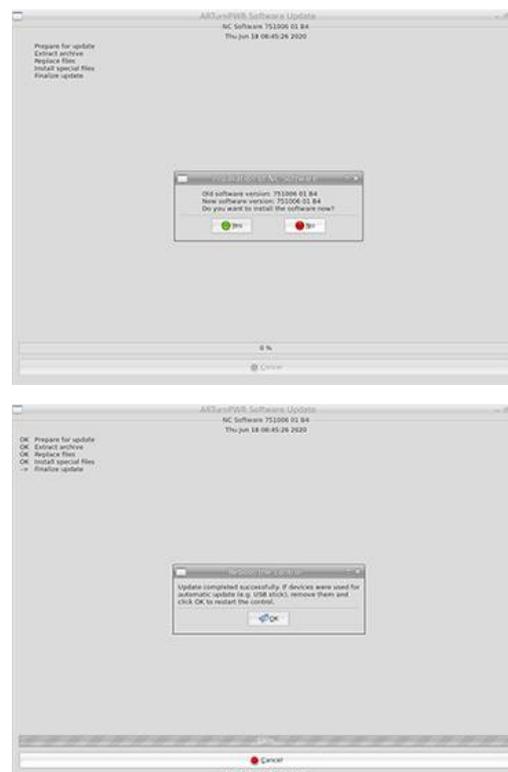
14

Simulators for Windows PCs

14.1 Simulator for Windows PCs

Overview

The control simulation software provides a convenient way to write part programs and simulate machine behavior using a standard Windows computer. Features and functionality are virtually identical to that of the control itself.

System requirements

- Platform: IBM compatible PC
- Operating System: All 32-bit and 64-bit editions of ** Windows 7/8/8.1/10 are supported
- Available Drive Space: 2 GB or higher
- Memory: 2 GB or higher
- Display Resolution: 1024x768 (XGA) or higher required. 1440 x 900 (WXGA+) recommended
- Ports: One available USB connection for protection device
- Peripherals: Keyboard and pointing device.

  **Microsoft, Windows is a registered trademark of Microsoft Corporation in the United States and/or other countries.

On screen keypad

The simulation software includes an on-screen keypad application. This can be used to simulate the hard keys that are normally available on the machine console. Pressing the buttons on the keypad application will activate the same operation that would occur when pressing the corresponding buttons on the machine console.

The **Feed** slider allows a range of 1% - 150%. The **Speed** slider allows a range of 30% to 130%.

The keypad application will automatically start and shutdown when the control software is started and shutdown. If necessary, the keypad application can be manually shutdown by clicking the red "x" button in the top right corner of the keypad window. Likewise, if necessary, the keypad application can be manually started using the **Launch** button on the Keypad tab of the Control Panel window.

Keyboard shortcuts

The keys on the keypad can be simulated using the key combinations described below. These can be used when the on-screen keypad is closed or when the PC keyboard is connected to a console.

Turning function keys

Key	Shortcut
TOOL	Alt + T (Tool)
THREAD	Alt + R (Thread)
RECESS	Alt + C (Recess)
TURN	Alt + H (Turn)
POS	Alt + P (Position)
LINE	Alt + L (Line)
ARC	Alt + A (Arc)
BLEND	Alt + B (Blend)
	Alt + U (User ID)

Function keys

Key	Shortcut
ABS INCR	Alt + I (Absolute/Incremental)
DRO PGM	Alt + D (DRO/Program mode)
VIEW	Alt + V (Display View)
INFO	Alt + N (On screen User Manual)
MM	Alt + F (Inch/MM)
SETUP	Alt + Enter (Configuration menu)

Calculator function keys

Key	Shortcut
CALC	Alt + K (Calculator)
+	+ (Plus)
−	- (Minus)
×	* (Multiply)
÷	/ (Divide)

Data entry keys

Key	Shortcut
CANCEL	Esc (Cancel)
CLEAR	Del (Clear)
USE	Alt + Enter (Use)
ENTER	Enter (Enter)

Motion control keys

Key	Shortcut
GO	Alt + S (Go)
STOP	Alt + J (Press once: Pause / Press Twice: Stop)

Soft keys

| Soft key	Shortcut
1	F1
2	F2
3	F3
4	F4
5	F5
6	F6
7	F7
8	F8
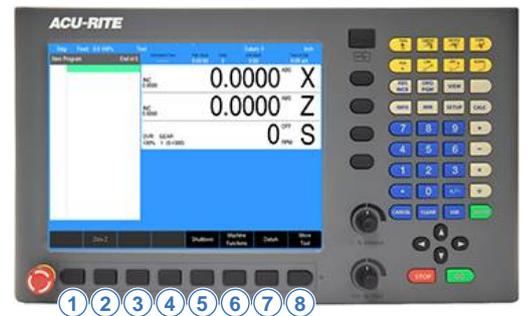
Numeric and arrow keys

The **Arrow** keys on a standard PC keyboard will simulate the console **Arrow** keys. When number lock is turned off, the **Arrow** keys on the number keys (2, 4, 8, 6) will simulate the console **Arrow** keys.

The number keys will simulate the console number keys (using number lock). F12 will toggle the **+/-** key.

14.2 Virtual machine simulator

Simulation software

The simulation software provides a convenient way to write part programs and simulate machine behavior using a standard Windows computer. Features and functionality are virtually identical to that of the control itself.
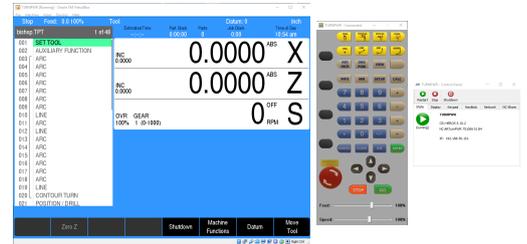
Installation

Installing the Virtual Machine (VM) Simulator is a two step process. First, the VM software is installed and then the control software is installed. The VM is installed by launching the setup program and following the installer prompts. On some systems, you may be required to restart the computer after installation before the application will function properly.

Following the initial VM installation, the first startup of the control software will automatically update the control software within the VM. The installation of the control software in the VM is similar to installation of the software on the actual control. Follow prompts during the installation and reboot as needed. Once the software is fully installed and running, verify that the software version matches the software version running on your machine console.

If the versions do not match see "Updating", Page 192.

Operation

The simulator software can be started by using either the shortcut created on your desktop during installation or from the Start menu. Once the software is running, it is operated in the same manner as the machine itself. Similarly, it should be shut down using the same sequence of selecting **Shutdown** and confirming **Shutdown** at the prompt. Alternatively, a shutdown can be done using the **ShutDown** button in the Control Panel window. For additional information, see "Control panel", Page 193.

Programs created in the simulation software can be copied to the CNC, and (with the correct setup) used on your machine.

Updating

Control software

The Simulator software itself typically does not need to be updated. Only the control application is updated when new releases are available. The same setup.zip used to update the machine software is used to update the simulated software. Although your created part programs will remain intact, it is always recommended to make a backup of your programs and/or configurations before updating the software.

Make sure the control software is not running. To update the software, click the **Install** folder shortcut available from the Windows **Start** menu, under **All Apps** and then under **ACU-RITE**. This will open a folder location in a new Windows Explorer window. Extract the **setup.zip** and **setup.ini** files obtained for the software update directly to this folder. Keep the window open. Start the control software which will detect that an update is available for installation. Follow the prompts to install the software. When the software update is complete, and prompts to remove any removable drives used for the update, proceed to delete the **setup.zip** and **setup.ini** files from the **Install** folder. If this is not done, the control will prompt to update the software on every subsequent startup until the files are manually deleted.

Virtual maching simulator

NOTICE
The part programs will be lost during this process. It is critical to backup these programs prior to uninstalling the existing Simulator.

When the Simulator software requires an update, a new version will be made available (for example, v2.0.0). In order to install a new Simulator version, the existing version must first be fully uninstalled (see "Uninstalling", Page 192), and then the new version installed and setup as an initial installation (see "Installation", Page 191). The control part programs will be lost during this process, so it is critical to backup these programs prior to uninstalling the existing simulator.

Uninstalling

Before uninstalling the control software, reboot the Windows PC in order to ensure a successful removal. To remove the control software from a Windows PC, either use the Windows **Control Panel** (**Add/Remove Programs**, or **Programs and Features**), or the shortcut **Uninstall TURNPWR** from the Windows **Start** menu (in **All Programs** and under **ACU-RITE**). When the uninstaller opens, make sure that all checkboxes are checked so that all components are fully removed from the system.
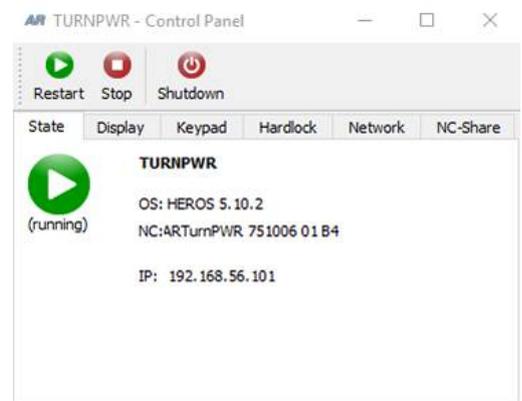
Control panel

The simulator includes a helper application that resides in the Windows taskbar, typically located in the bottom right corner of the desktop screen. Clicking on the **AR** icon will open the **Control Panel** window.

The **Control Panel** provides information about the control software, the Virtual Machine itself, and also allows for some user-configurable settings and control operations. There are multiple tabs in the application window:

State

The **State** tab is the main tab of the **Control Panel** which shows the current status of the VM and the control software. The icon will indicate the state of the software; for example, running or stopped to reflect what is used in the software. The text will indicate the HEROS OS version, the control software version, and the IP address.

Display

The **Display** tab of the **Control Panel** provides optional display settings. The display resolution default is **1024x768** and must not be changed. The display size default is **Normal** but can be changed to increase or decrease the size of the display by selecting **Scale** allowing for manual screen size adjustment, or **Full** which will scale the display to fill the screen.
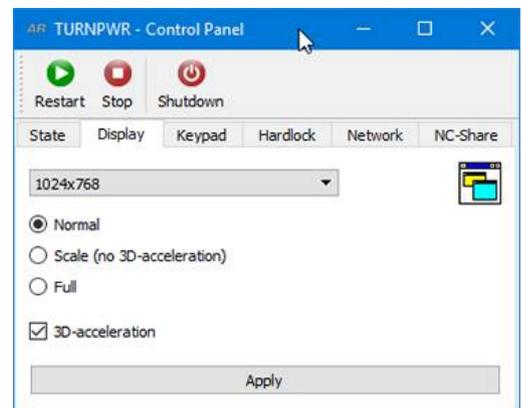
Keypad

The **Keypad** tab of the **Control Panel** provides a button to **Launch** the on-screen keypad application manually if needed.
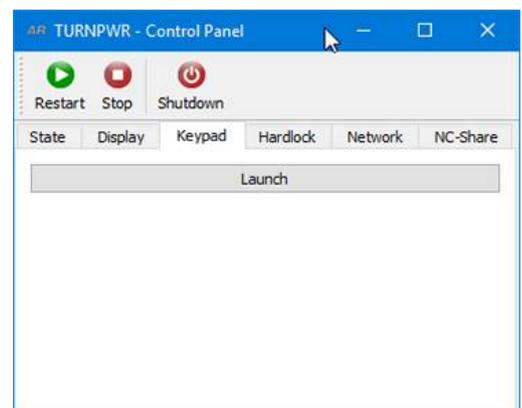
Hardlock

The **Hardlock** tab of the **Control Panel** allows the user to configure the control to use a networked USB Protection Module (683049-01, or 683050-01) instead of a single-user local-only key (683048-01). To use a Network Protection Module, check the **Use license server** box and enter the IP address or hostname of the network **Server** which hosts the Network USB Protection Module. Click **Apply** to save the changes. Follow the prompts to re-start the application as needed.
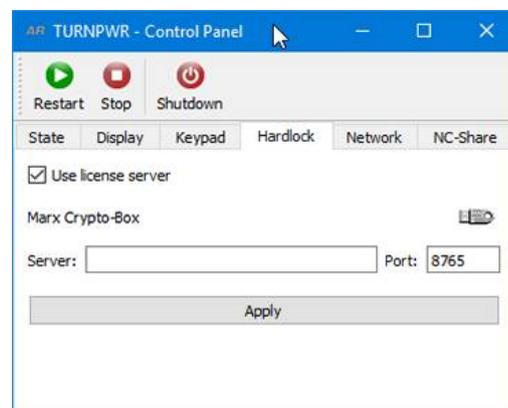
Network

The **Network** tab of the **Control Panel** allows the user to manually assign a unique Hostname to the Virtual Machine, instead of the automatic assignment used by default. To manually set the Hostname, uncheck the **Auto Hostname** box and enter the IP-Address or Hostname and click **Apply** to save.
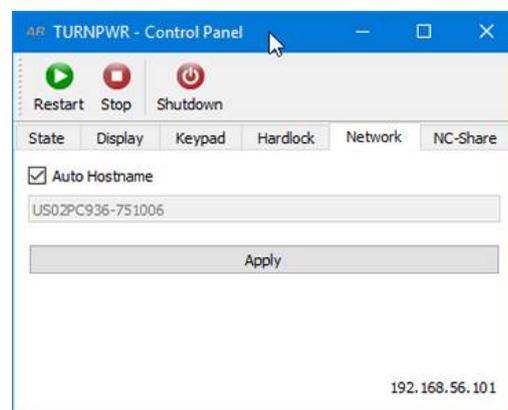
NC-Share

The **NC-Share** tab of the **Control Panel** allows the user to setup a shared folder between the Windows host PC and the control software. This allows easier transfer of NC program files, service files, etc. between the control software and the Windows PC. To set up a shared folder, select an available (unused) drive letter to map the shared folder on the Windows PC. Check the **Connect NC-Share at startup** checkbox if the share should be automatically mounted each time the control software is started. Click **Connect** to save the changes and mount the shared folder. Now you can access the control folders using Windows Explorer and the assigned drive letter.
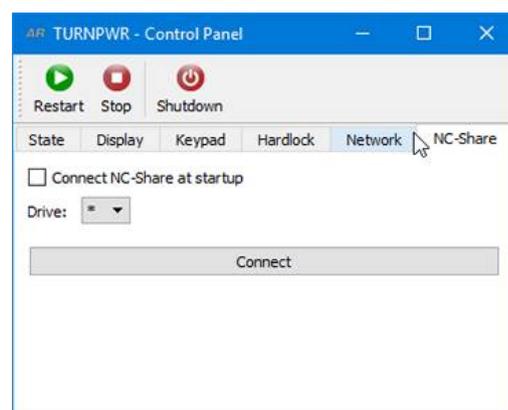
Control buttons

▶ The **Restart** button will restart the control application.

▶ The **Stop** button will conduct a shutdown of the control software and keypad, but will keep the **Control Panel** application open

▶ The Shutdown button will conduct a shutdown of the control software and keypad, and close the **Control Panel** application

HEIDENHAIN CORPORATION

333 East State Parkway
Schaumburg, IL 60173-5337 USA
☎   +1 (847) 490-1191
🖷   +1 (847) 490-3931
E-Mail: info@heidenhain.com

www.heidenhain.com

HEIDENHAIN CORPORATION

333 East State Parkway
Schaumburg, IL 60173-5337 USA
☎   +1 (847) 490-1191
🖷   +1 (847) 490-3931
E-Mail: info@heidenhain.com

www.heidenhain.com