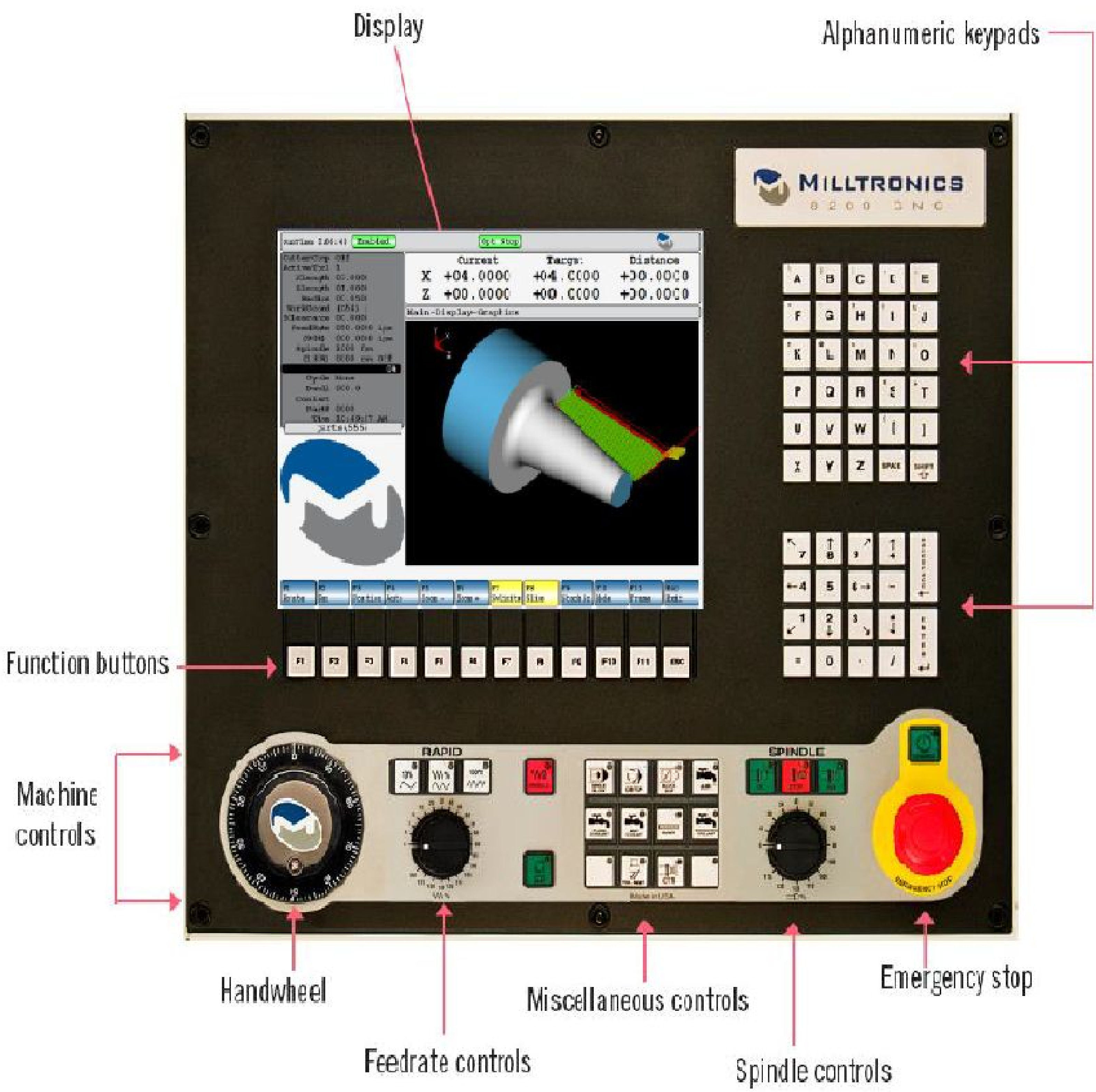
8200 T SERIES CNC TURNING

CONTROL

Operation Manual

Version 2.2



8200 T SERIES CNC TURNING CONTROL

Operation Manual

Version 2.2

Preface

This manual describes the operation of Milltronics 8000 Series CNC controls. The 8000 T Series has three controllable axes; X, Z, and C.

The programming portion of this manual is divided into two sections: conversational programming and text programming. The conversational programming section explains the various menus, screen entries, and the general flow from one screen to another. Text programming commands are described in the M and G code sections and should be referenced there.

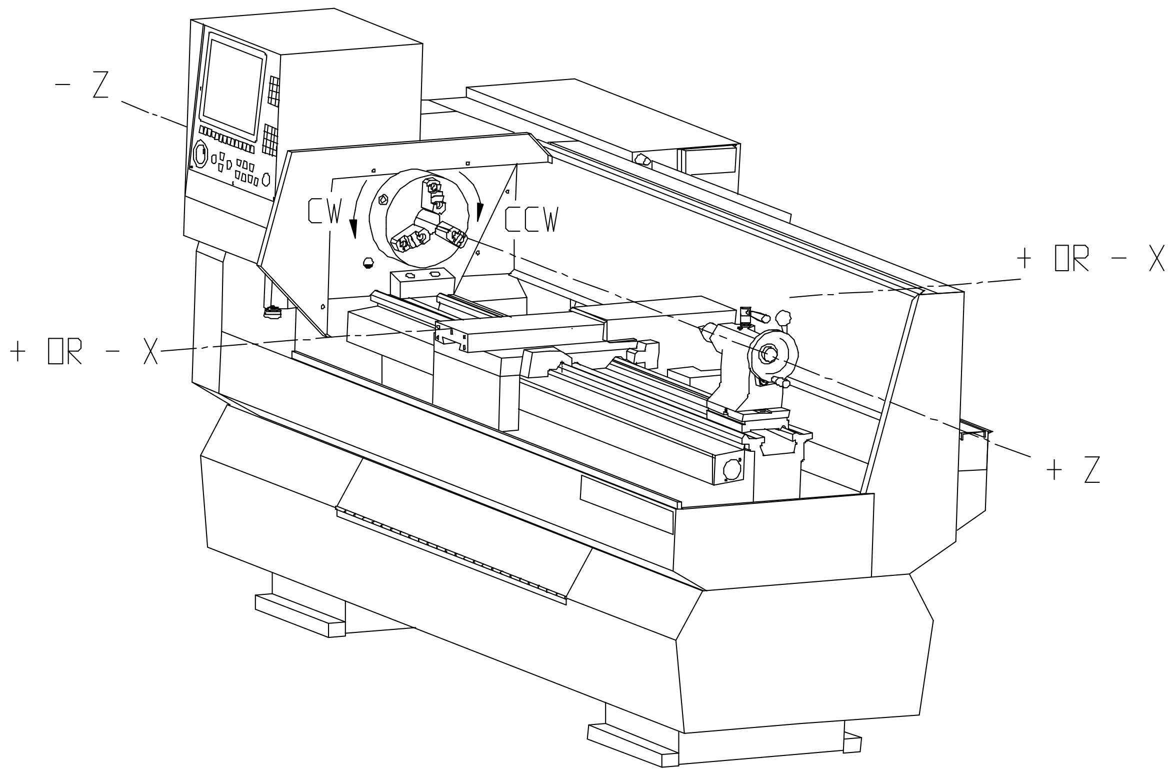
Although every effort has been made to create a complete and accurate manual, some omissions and errors may have occurred.

1

Section 1 Contents

Axis Directions

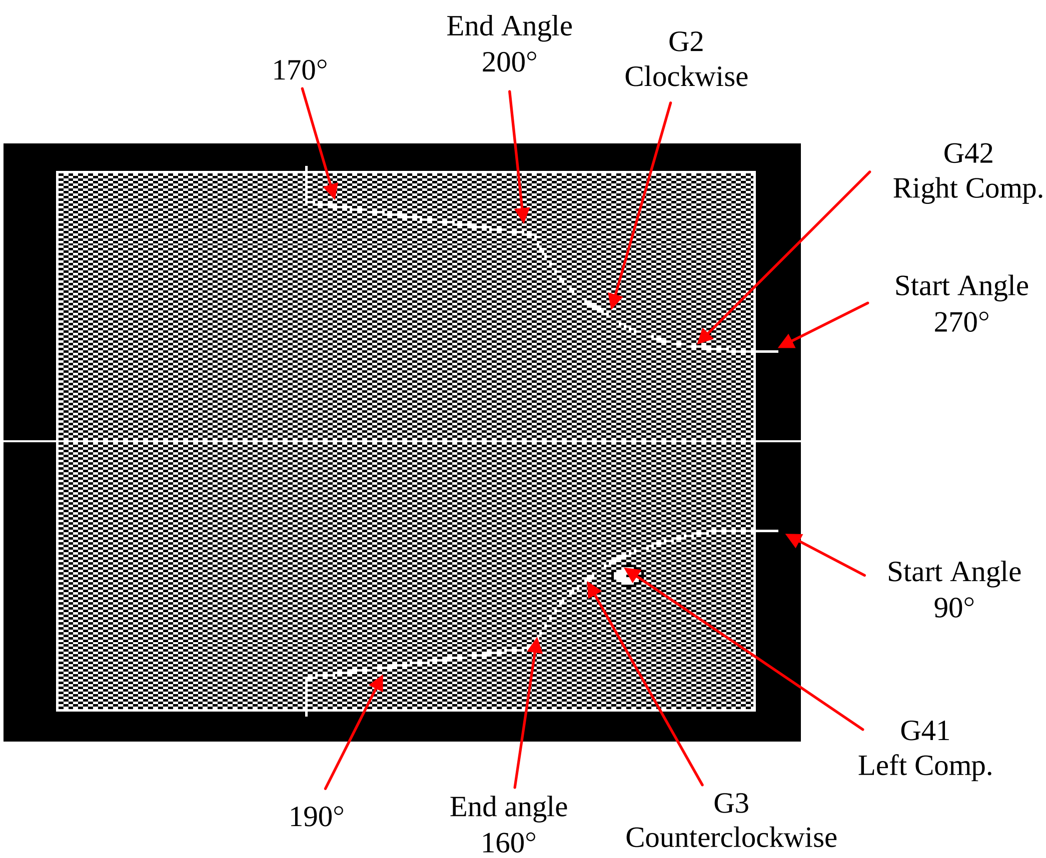
All directions are referenced with respect to the tool. The following illustrates the X and Z directions.



The control can be setup so the X+ is either towards that operator or away from the operator. Machines that have a tool holder on the front side of the spindle are normally configured with X+ being towards the operator and machines that have a tool holder on the back side of the spindle are normally configured with X- being towards the operator. The parameter used to define this orientation is the POWER parameter "Front Turret". When set to "Yes" the tool is shown coming from the bottom on the graphics screen. It also relates the tool types in the tool table as coming from the bottom. A G13 can be used to specify back side programming or a G14 for front side programming.

When programming from the front verses from the back there are 3 differences:

1) G2/G3 or Clockwise/Counterclockwise arcs
2) G41/G42 or Left/Right cutter comp
3) The polar lines and arcs angles are different

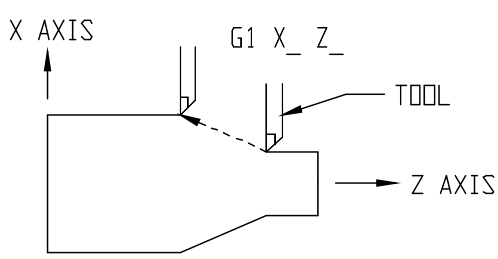


All the examples in the manual are written using back side programming.

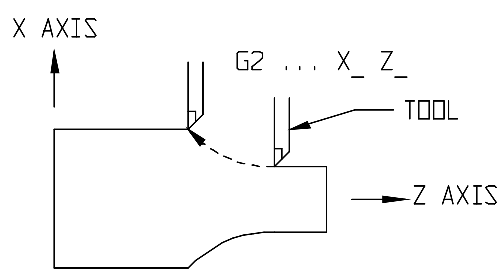
Definition of a Program

A group of commands given to the CNC for operating the machine is called a program. By specifying commands the tool is moved along a straight line or arc, and machine functions such as coolant on/off, tool change or spindle on/off are performed.

The function of moving the tool along straight lines and arcs is called interpolation.



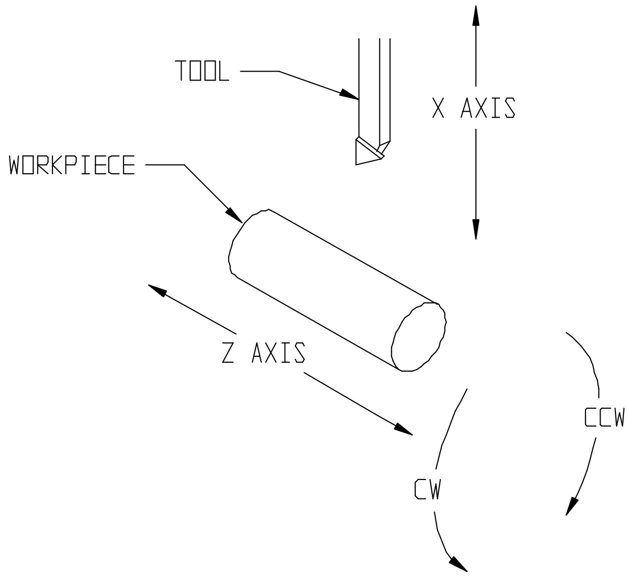
linear interpolation



circular interpolation

Coordinate Systems

When the commanded position to be reached by the tool is executed, the CNC moves the tool to that position via one of the interpolation modes, circular or linear. The position is given as a coordinate value in a rectangular Cartesian coordinate system.

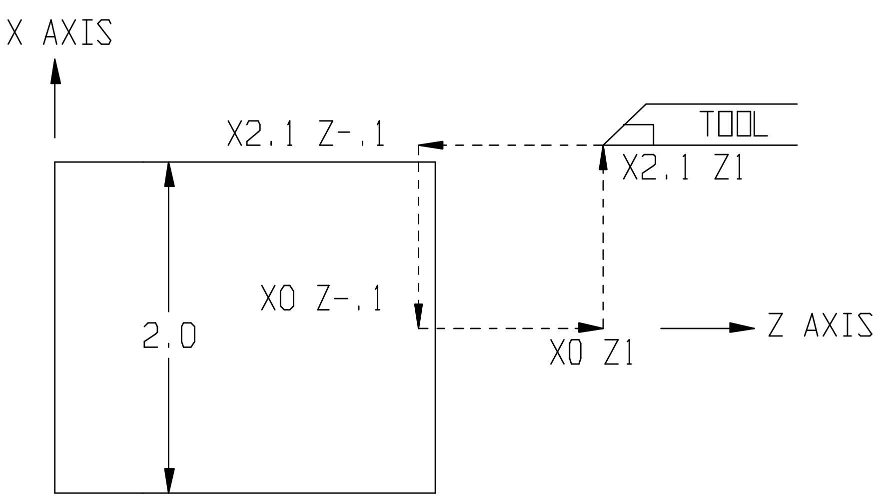


Coordinates

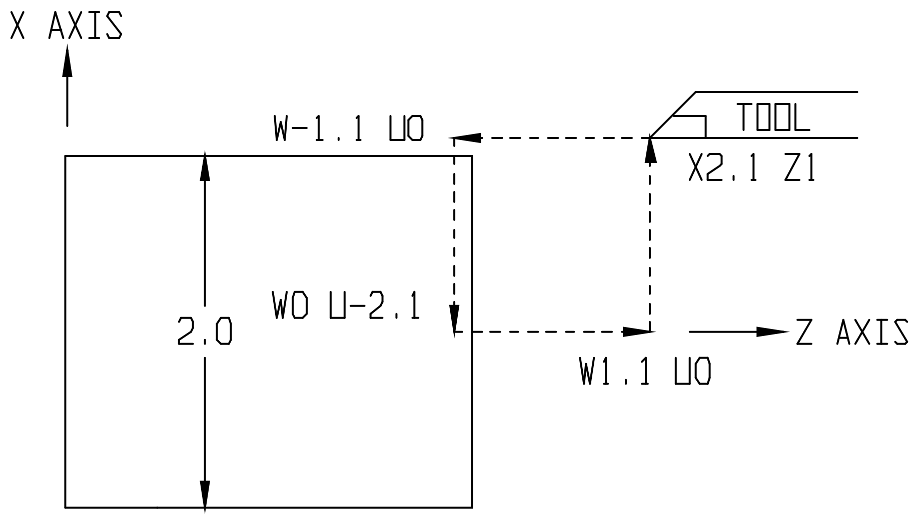
The position to be reached by the tool is commanded with a coordinate value referenced to the above coordinate system. The coordinate value consists of one component for axis X and Z.

Coordinate Values

Coordinate values may be given in either absolute or incremental mode. In absolute mode the tool moves to a point the programmed distance from the zero point of the coordinate system.

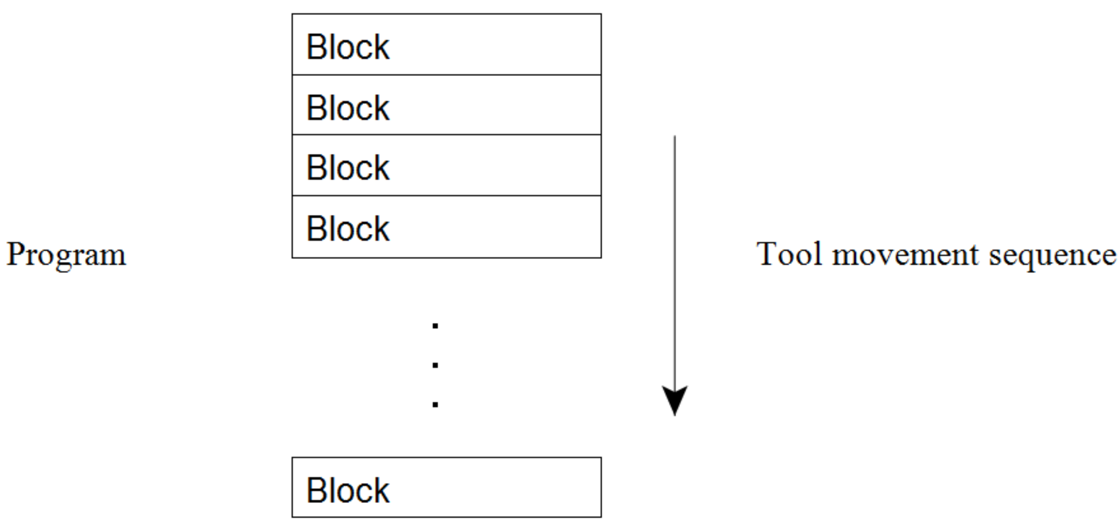


In incremental mode the tool moves to a point the programmed distance from the current tool position.



Program Structure

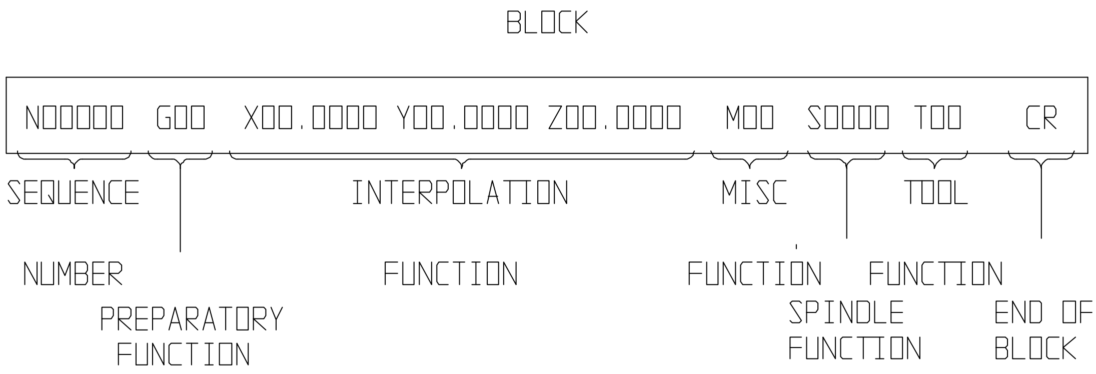
By definition, a program is a group of commands given to the CNC for operating a machine. By specifying commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In a program, specify the commands in the sequence of actual tool movements.



A group of commands at each step of the sequence is called the **block**. The program consists of a group of blocks for a series of machine moves. An optional number for definition of each move is called the **block number.**

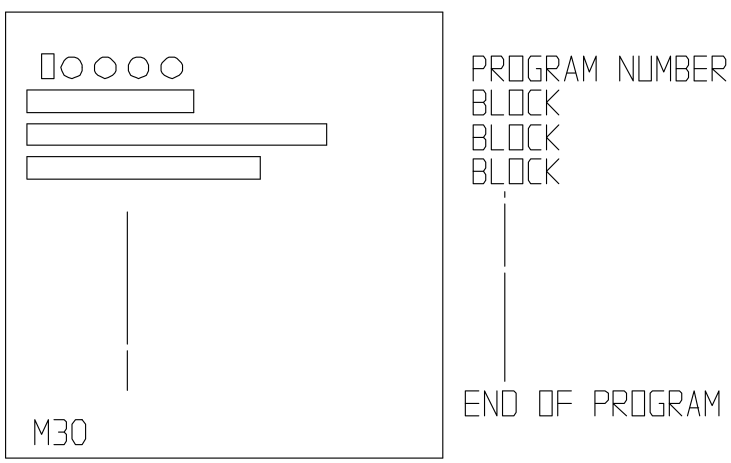
The block and the program have the following configurations.

Block



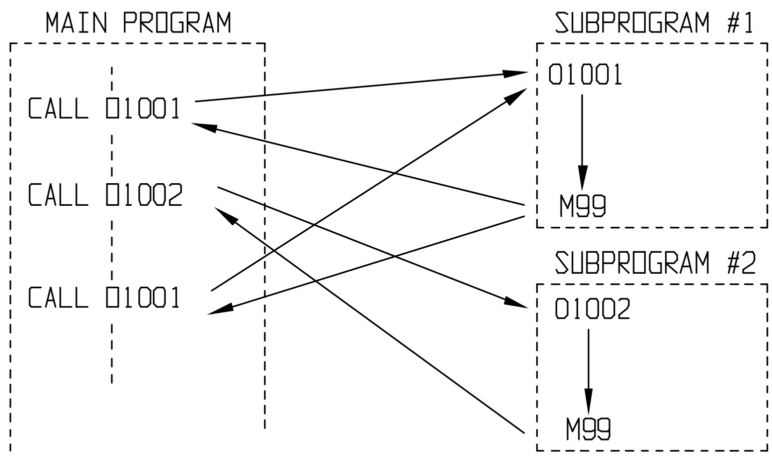
Each block begins with an optional number and ends with a <CR> carriage return.

Program



Normally a program number is specified at the beginning of a program, and a program end code (M99, M02, M30) is specified at the end of the program. Neither is required; however, it may be advantageous to omit the program end code from programs that are used as subprograms. An end program code is assumed when the end of the main program is encountered.

Main Program, Subprogram, and Subroutines



When it is necessary to machine the same pattern at many places on a part, a program for the pattern should be created. This is called a **subprogram**. When an “M98” or “CALL” (subprogram call) appears in the main program, the commands of the subprogram are performed before execution of the next block of the main program.
Subprograms can be used to build part libraries of commonly used patterns and can reside anywhere in memory.

Command Format Ranges

The basic address and command value ranges are listed in the table below. Note these figures give the maximum numerical limit for the control. These limits will always be greater than or equal to the physical limits of the machine. The machine limits are set via parameters in the machine setup section of the control.

Command Format Ranges

FUNCTIONS	COMMAND LETTER	INCH INPUT	METRIC INPUT
Subprogram # and Program #	O	1 - 9999	1 - 9999
Sequence #	N	1 - 99999999	1 - 99999999
Preparatory function	G	0 - 999	0 - 999
Dimension * words	XYZUVWQ ABCIJKRP	0 - 999.9	0 - 999.9
Dwell	P	.1 - 999.99	.01 - 9999.99
Feedrates *	F	.0001 - 999.9999	.01 - 99999.99
Spindle speed *	S	1 - 9999	1 - 9999
Tools	T	0 - 99	0 - 99
Misc. function	M	0 - 999	0 - 999
Repeat or loop	L	0 - 999	0 - 999

*These functions have selectable decimal positions. There may be any number of leading or trailing places as long as the total number of digits fits in the field.

Command Formats for Axes: M and G Codes

Axis commands can be programmed in a calculator format. No leading or trailing zeros are necessary. Whole numbers may be programmed without the decimal point. A decimal point may be used with mm, inches, or second values. The location of the decimal point is as follows.

Z15.0	Z15 millimeters or Z15 inches	(same as Z15)
F10.0	10 mm/min., or 10 inch/min. (in G98)	(same as F10)
G04 P1	Dwell for one second	(same as G4 P1)

The following addresses can be used with a decimal point: X, Y, Z, U, V, W, A, B, C, I, J, K, R, F, P, Q, AA, AB, XC, YC, ZC, E, H, L, N, O, S, and T.

Axis Min/Max Values for Standard Systems

	Least Increment	Maximum Value
Metric	0.001 mm	99999.999 mm
English	0.0001 inch	99999.9999 inch
Degrees	0.001 deg	99999.999 deg

Axis positions are stored in floating point; therefore, digit commands greater than 8 will be accepted.

Data Entry Fields

When editing or entering parameter values (or any other numeric value on the control), you can use the built in calculator.

Example: Instead of entering .3750 you may enter 3/8 Instead of entering 1.3750, you may enter 1 + 3/8.
If you want to modify the current value, you may use "." as the current value.
If the current value is .358 and you want to add .002, type .+.002 (instead of entering .360).

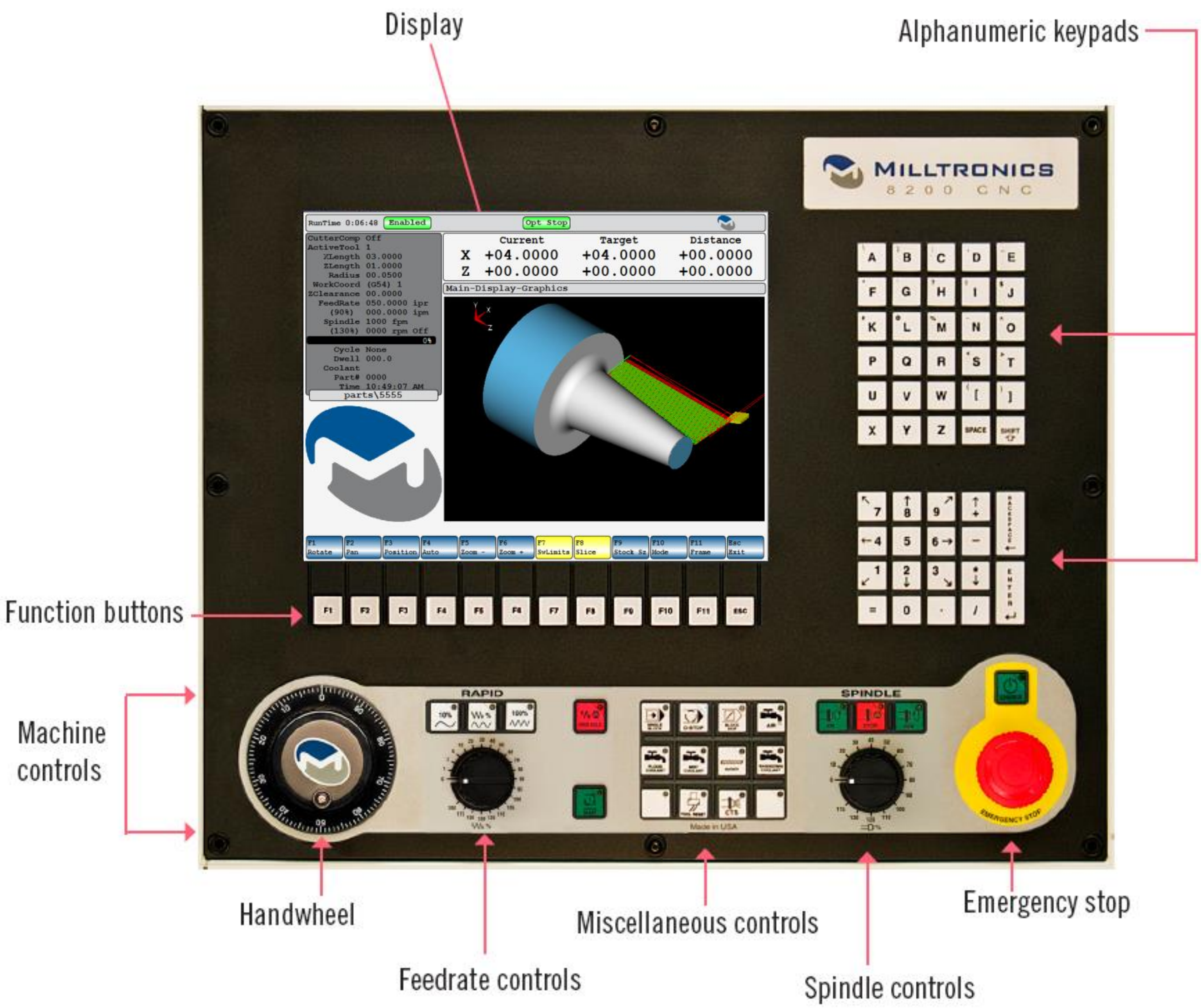
2

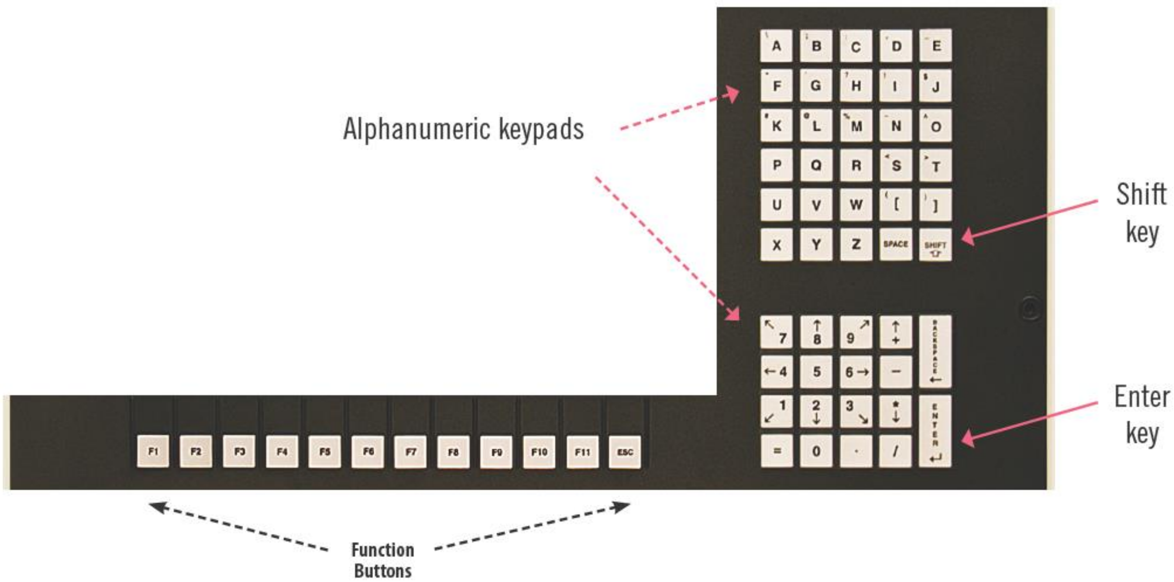
Section 2 Contents

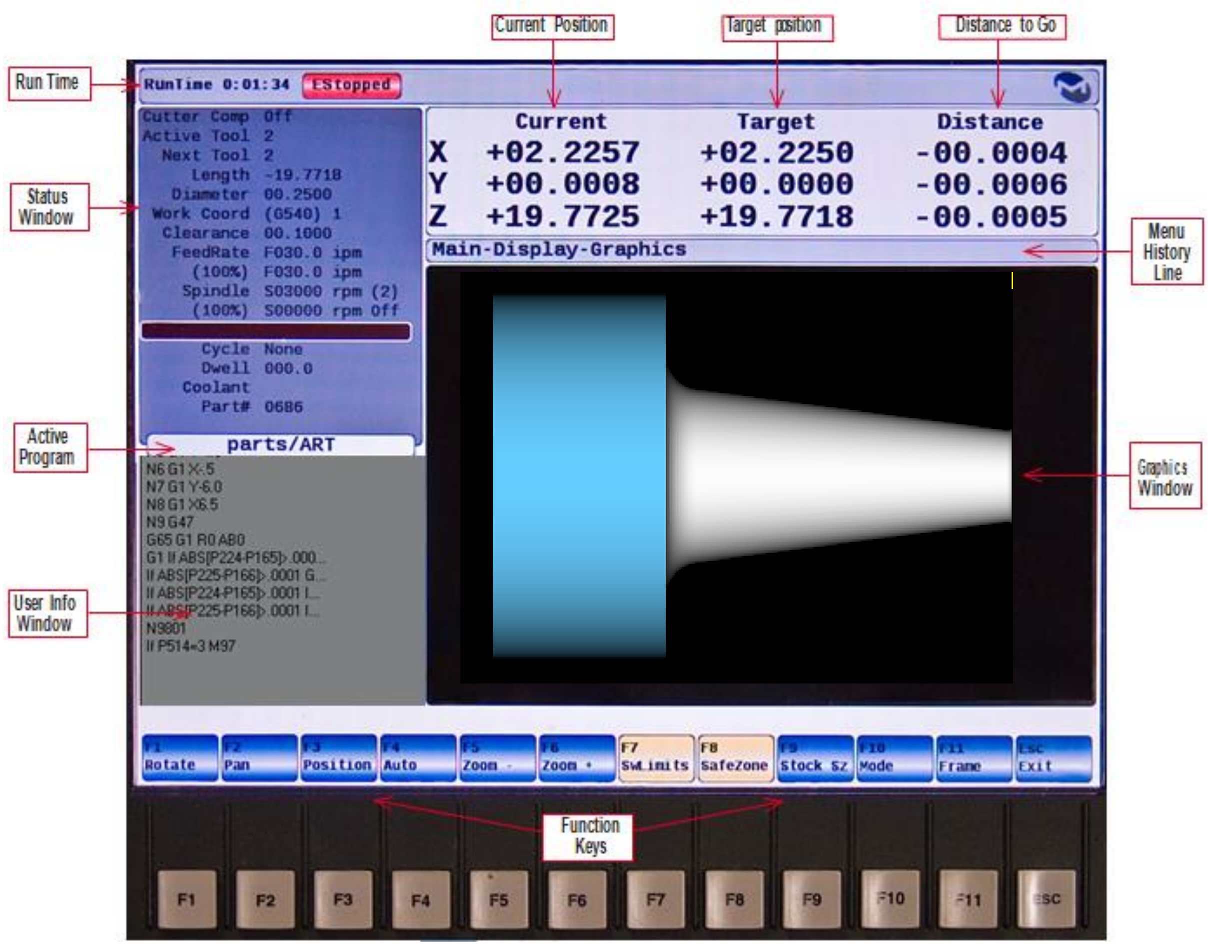
In this Section

This section of the manual describes the various knobs, buttons, and controls of the 8000 T Series CNC Front Panel including the functions of the various sections of the Display.

The 8000 T Series CNC Front Panel




Alphanumeric keypads
Shift key
Enter key
Function Buttons


Current Position
Target position
Distance to Go
Run Time
Status Window
Active Program
User Info Window
Menu History Line
Graphics Window
Function Keys

Alphanumeric Keypads

The 8000 Series front panel has two alphanumeric keypads. The keypads are used to enter alphanumeric data for the CNC. The upper keypad is used primarily to enter alpha characters.

To enter one of the shifted characters simply press and release the **Shift** key then the character. After the character has been entered, the control automatically returns to the non-shifted character set. **Shift** also works in the same manner on the lower or numeric keypad. Spaces between commands are optional when data is entered, but the **Enter key** must be pushed to end a line of data or to go to the next function.

Display and Function Buttons

Run Time

When verifying or running a program Run Time shows the time since the program was started.

Active Program

Active Program displays the program being run or verified. In Edit mode it shows the active edit program.

Current Position

* Current Position is the position of the machine relative to the work offset zero.

Target Position

* When running or verifying, Target Position is the next position the machine is going to.

Distance to Go

* Distance is the distance remaining in the current move.

** Note: Column displays can be changed. Set Control Parameter "Enable Modify Column Displays". F6 (Display) will allow options for; Current, Target, Distance, Machine, Measure, Actual, None, and Error.*

Function Buttons

Pressing these buttons performs whatever function is indicated on the corresponding function key.

Function Keys

A blue function key means there are additional menu choices available under that function key. Green and yellow function keys perform an action or designate a state. When green, the action or state is on; when yellow the action or state is off.

Graphics Window

The Graphics Window displays solid model and wireframe graphics.

Menu History Line

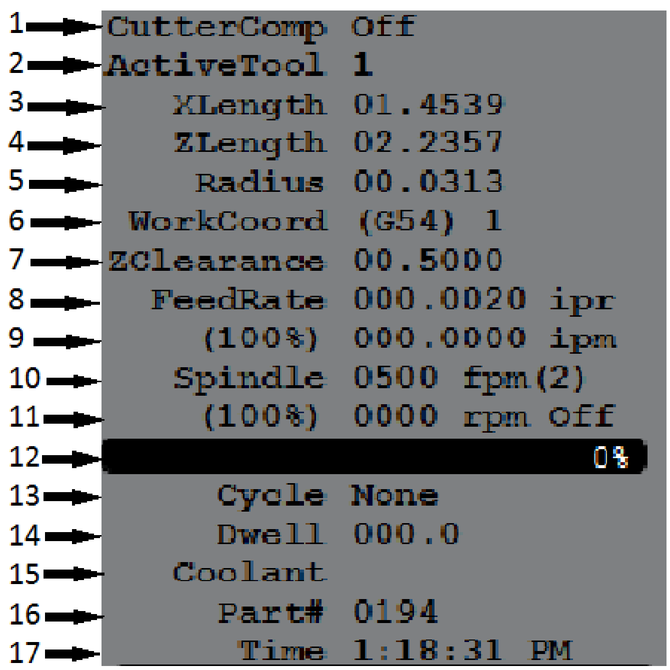
This line displays the path the operator follows to get to the current screen. The last word displayed is the name of the current screen.

User Info Window

Contents of the User Info Window depend on the contents of the Main Graphics Window. For example, program blocks appear in the User Info Window when a program is running. Help pictures appear in the User Info Window during conversational programming.

Status Window

The Status Window displays information on the state of the control. A more detailed description of the status window is given below.



Status Window details

1 **Cutter Comp:** Tool Diameter Compensation (Left, Right or Off)

2 **Active Tool:** The active tool number

3 **XLength:** The active X length .

4 **ZLength:** The active Z length.

5 **Radius:** The active tool radius.

6 **Work Coord:** The work offset currently in effect.

7 **ZClearance:** Clearance or R-Plane in the Z axis.

8 **FeedRate:** The programmed feedrate. For Inch mode, inches per minute (ipm) or inches per revolution (ipr). For Metric mode, millimeters per minute (mmpm) or millimeters per revolution (mmpr).

9 **(100%):** The position of the feedrate override knob and the actual feedrate. For Inch Mode, inches per minute (ipm). For Metric mode, millimeters per minute (mmpm).

10 **Spindle:** The programmed spindle speed, for Inch mode, feet per minute (fpm). For Metric mode meters per minute (mpm). Or in revolutions per minute (rpm). And for multi-geared spindle, the spindle gear.

11 **(100%):** The position of the spindle override knob and the actual spindle rpm. This line also displays whether the spindle is off or running CW or CCW.

12 **The Spindle Load Meter:** Becomes longer as the spindle load increases. A percentage of the load is also displayed.

13 **Cycle:** If there is a canned cycle or autoroutine active it is displayed on this line.

14 **Dwell:** When a dwell is executing the dwell time counts down to zero.

15 **Coolant:** Off, Mist, Flood, etc.

16 **Part#:** The Parts Counter increments each time a program ends normally. It does not increment if a program is aborted or if there is an error in the program. It does not increment in Dry Run, Verify or MDI. It does increment in Run. The parameter number used for the counter is P699. You can zero, increment or decrement the counter using:

Shift-F1 (zero/reset)

Shift-F2 (decrement) (it will not decrement below zero)

Shift-F3 (increment)

Press and hold the Shift button while pressing the F button.

17 **Time of day**

Machine Controls

The lower section of the front panel is dedicated to machine controls. Details of these controls are shown below.

Handwheel and Feedrate Controls

Located on the far left of the panel is the electronic handwheel, which when turned in the handwheel mode will cause the selected axis to move. The handwheel can also be used to scroll through programs and menus.

Lathes with dual electronic handwheels typically do not have a handwheel located on the front panel as shown.

Next to the handwheel are the feedrate controls. Turning the feedrate override will modify the current machine feedrate by the indicated percentage.

Pressing the **FEED HOLD** button will cause axis motion to stop. To restart axis motion, press the **CYCLE START** button. The **CYCLE START** button needs to be pressed any time a machine command is to be executed. **CYCLE START** will blink when it needs to be pressed.

Above the Feedrate override are three buttons for controlling the behavior of the machine when it is in rapid (G0) mode. The default setting is when the middle **RAPID** button is lit. With the middle button lit, the feedrate override knob overrides both rapid and feed (G1) moves. If the override knob is turned all the way counterclockwise to zero, the machine stops.

If the leftmost (10%) **RAPID** button is pushed so it is active (and lit), then all rapid moves are executed at 10% of max rapid. In this case, the feedrate override knob changes G1 feedrates but has no effect on rapid moves, which are fixed at 10%.

If the rightmost (100%) **RAPID** button is pushed so it is active (and lit), then all rapid moves are executed at max rapid. As in the previous case, the feedrate override knob changes G1 feedrates but has no effect on rapid moves, which remain fixed at full rapid.

Miscellaneous Control Buttons

The block control buttons (**SINGLE BLOCK, O-STOP, BLOCK SKIP**) allow the operator to interrupt the running of a program. These buttons were soft keys in previous versions of the controller.

Block, Opt-Stop, and Blk-skip functions

When **Single Block** is active, the program stops at the end of each block. Each time **Cycle Start** is pressed, one more block is run.

When **O-Stop (optional stop)** is active, the program stops at each M01 command. When **Cycle Start** is pressed, the program continues.

Block Skip (/)

A line of program can be skipped or ignored by the control. Inserting a / (slash) at the beginning of a line and enabling the Block skip will cause the control to skip that line. In the example below with Block skip disabled, the machine will move to the first, second, and third points. When Block skip is enabled, the machine will move to the first then third points. Block two is skipped.

```
N1      X0 Z0
/N2     X2 Z2
N3      X4 Z0
```

The coolant buttons **(AIR, MIST** and **FLOOD**) work identically to **CW** and **CCW**. When they are lit, the function is active. The **AUGER** and **WASHDOWN COOLANT** buttons allow the operator to manually remove chips (safety doors must be closed).

The **TOOL RESET** button is only active during an M6 command. This button is a safety interlock, which prevents the spindle from starting during a manual tool change. The button will start flashing during a tool change and will need to be pressed after the tool change is completed before program operation can be resumed.

Spindle Controls

The next section of the panel deals with the spindle. The spindle override switch will modify the current spindle rpm by the selected percentage. The spindle **CCW, CW** and **STOP** buttons will override the current control commands giving the operator full manual override capabilities. The active state of the spindle will be represented by the illuminated button. In order to start the spindle on lathes using the CW and CCW buttons, the Enable button must be pressed simultaneously with the CW or CCW button.

Enable button

Emergency Stop button

Emergency Stop Button

The **EMERGENCY STOP** button, when pushed, will stop all machine actions instantly.

Once **EMERGENCY STOP** is pressed, the **ENABLE** button will flash indicating that it must be pushed before any machine motion can be performed. The control is always in an Emergency Stop state after power-on.

External Keyboard Operations

Connect an external keyboard to the external keyboard jack of the 8200 Controller. The following key strokes simulate the buttons to operate the FastCam Simulator on a desktop computer.

Note: The Feedrate and Spindle Override ALT keys only work on FastCAMs, and not on the CNC control.

Machine Controls

ALT +

F6 = Cycle Start

F8 = Tool Reset

Feed rate over ride ALT+	Spindle over ride ALT+
A = 0%	1 = 0%
S = 10%	2 = 10%
D = 20%	3 = 20%
F = 30%	4 = 30%
G = 40%	5 = 40%
H = 50%	6 = 50%
J = 60%	7 = 60%
K = 70%	8 = 70%
Z = 80%	Q = 80%
X = 90%	W = 90%
C = 100%	E = 100%
V = 110%	R = 110%
B = 120%	T = 120%
N = 130%	Y = 130%
M = 140%	U = 175%

Connect To
External Keyboard Jack
on Controller

NOTE: A USB keyboard can be connected to the USB connector.

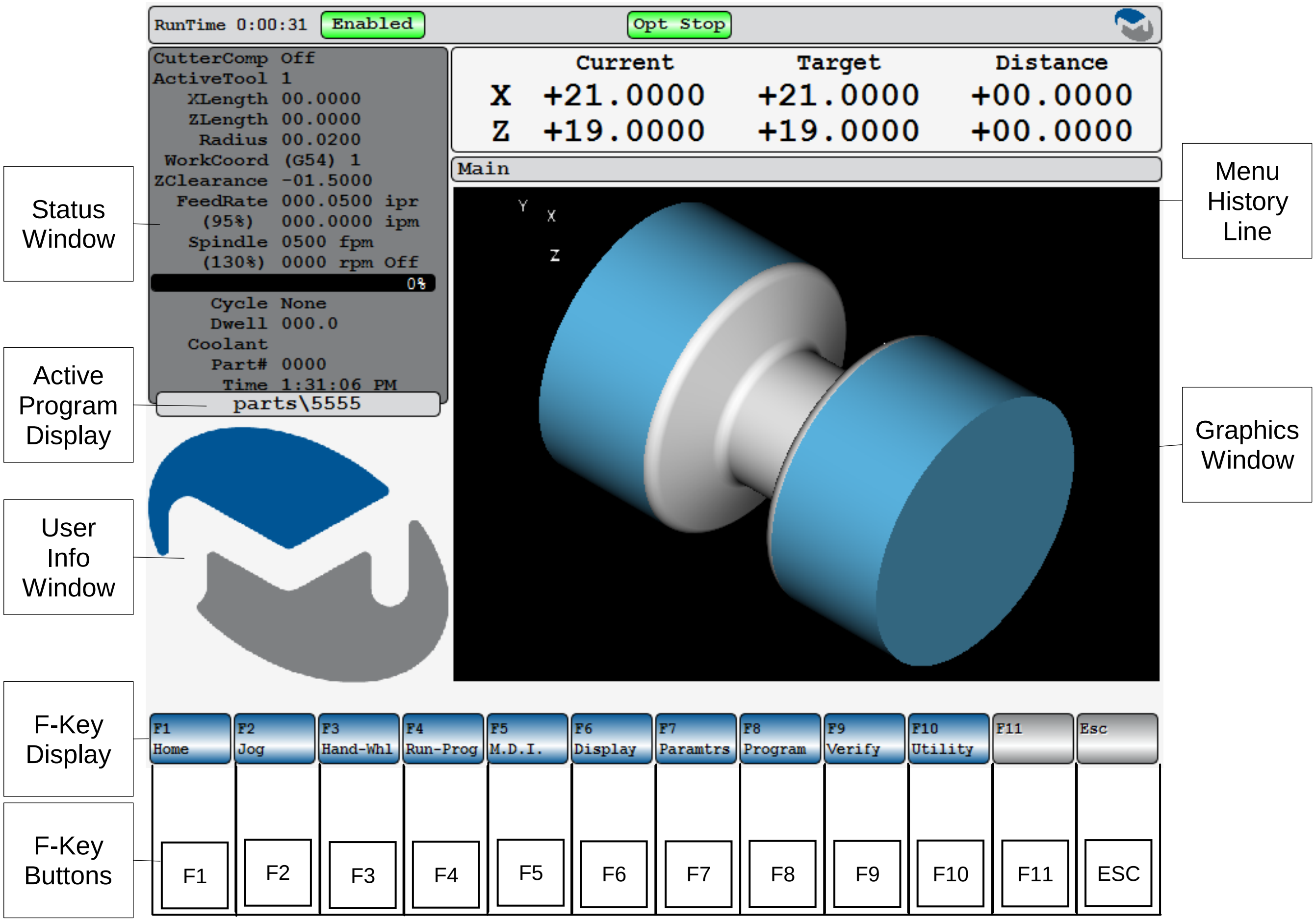
3

Section 3 Contents

In This Section

This section of the manual describes the various menus, screen entries, and the general flow from one screen to another. There will be some duplication of these descriptions in other sections of the manual. The first set of menu screens shown are those which can be accessed without entering any new information. These screens are shown with the top of the screen cropped for simplicity.

Navigation is accomplished by pressing the function buttons corresponding to the different function keys. For simplicity however, the manual will simply refer to the function key.



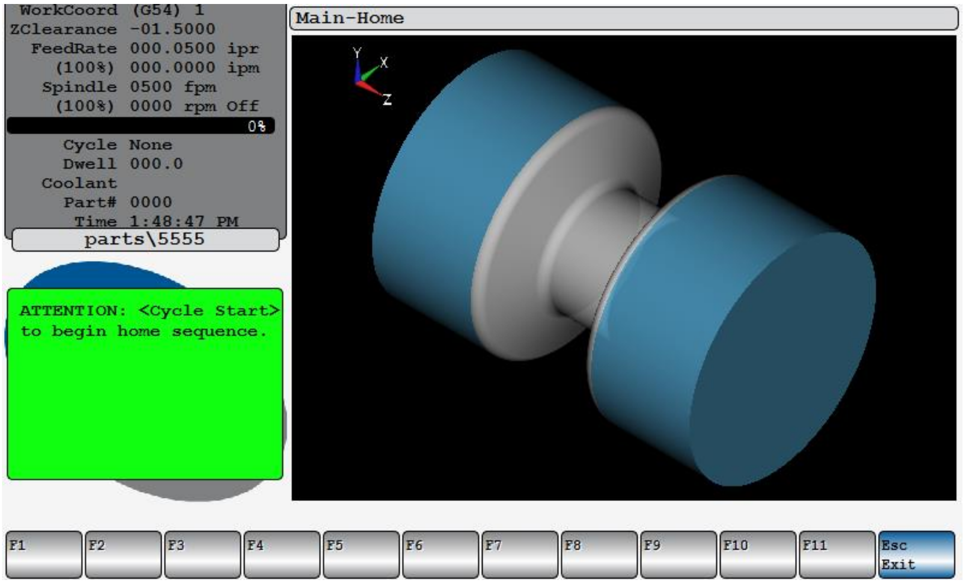
The Main Menu

The main screen (shown above) is the starting point for accessing all of the other screens. It is the screen that appears when the controller is first powered up.

After verifying a part program, the graphics display window in the MAIN screen shows the finished part.

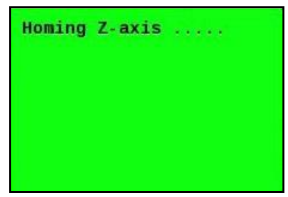
F1 (Home)

The Main-Home screen appears when the (F1) Home function key is pressed on the Main screen.



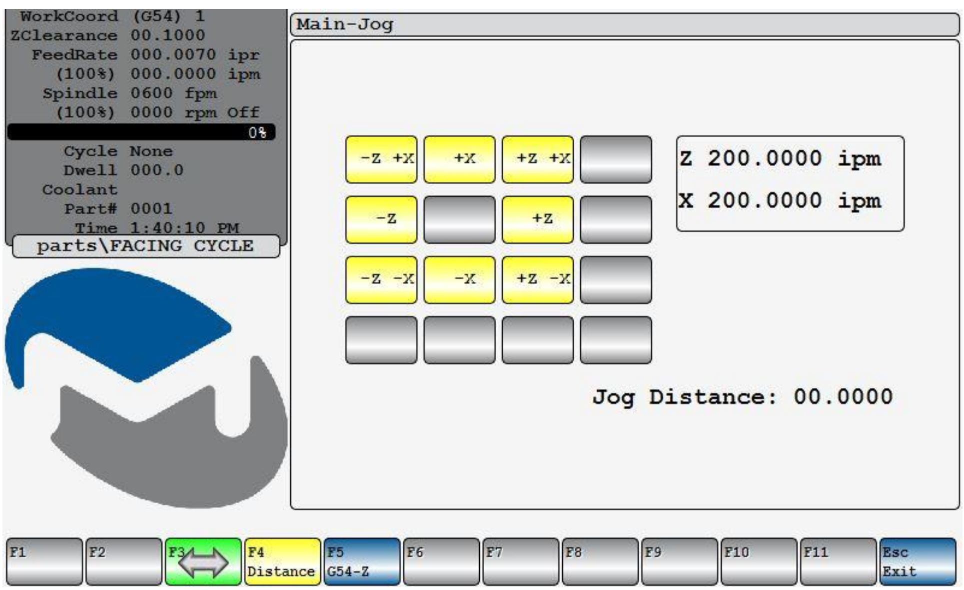
When Cycle Start is pressed, the machine moves the saddle and cross-slide to the machine home position.

During the homing process the User Info Window displays progress. See the figure at right.



F2 (Jog)

The Main-Jog screen appears when the (F2) Jog function key is pressed on the Main screen.

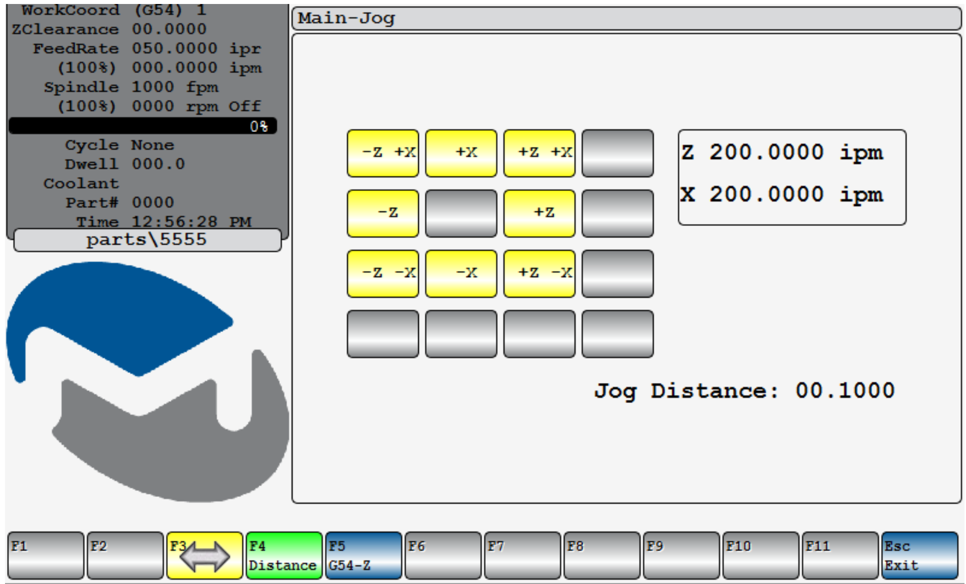


The machine must be homed prior to jogging. JOG is used to move the machine around in a manual mode to pick up zeros and align parts. The function keys across the bottom of the screen select the desired jogging mode. The F3 key is the default, selecting the continuous jog mode.

The feedrate for continuous jogging is set in Axis parameters. The jog feedrate may be adjusted with the feedrate override knob. In continuous jog, the selected axis continues moving until the user releases the axis key or encounters a software limit. The keyboard diagram displays the direction in which the axes move when the corresponding key on the numeric keypad is pressed.

F2 (Jog) – F4 (Distance)

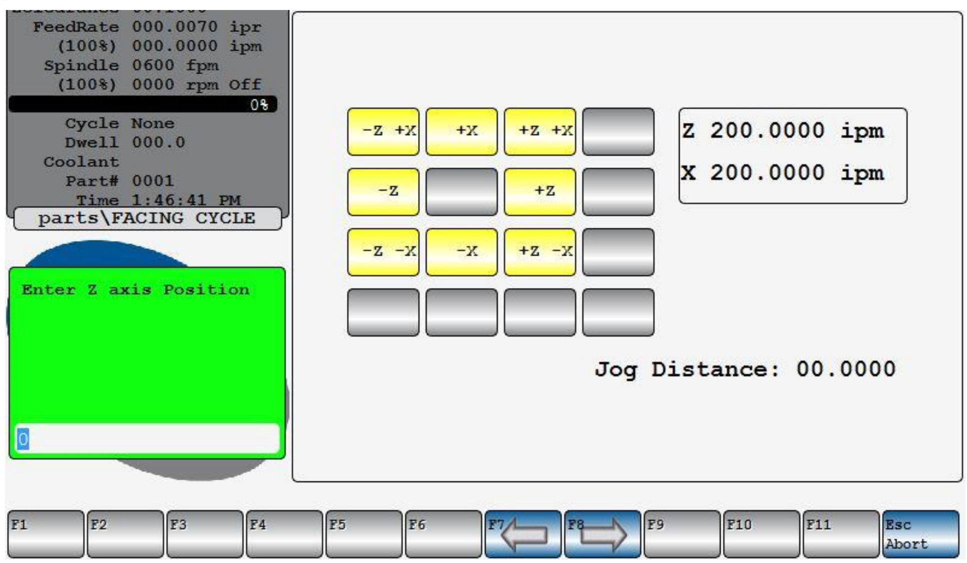
The main-Jog-Distance screen appears when the F4 (Distance) function key is pressed on the Main-Jog screen.



In incremental (or Distance) jog, the axis moves the selected increment and then stops each time the user presses and releases an axis key. After the operator presses the F4 (Distance) key, the control prompts the user to enter the desired amount of increment.

F2 (Jog) – F5 (G54-Z)

The Main-Jog-G54Z screen appears when the F5 (G54-Z) function key is pressed on the Main-Jog screen.



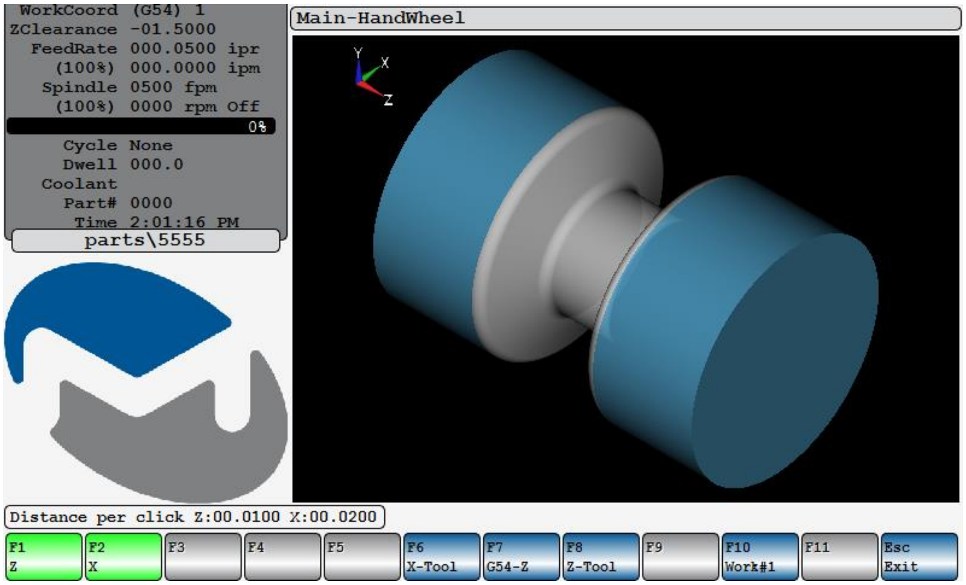
The F5 key sets G54 at the current machine position or at an offset from the current position.

Pressing the F5 key brings up a dialog box asking where the machine is relative to G54Z.

Pressing F5 can be thought of as “call this position ##.####”.

F3 (Handwheel)

The Main-Handwheel screen appears when the F3 (Handwheel) function key is pressed on the Main screen. The machine must be homed prior to handwheeling.



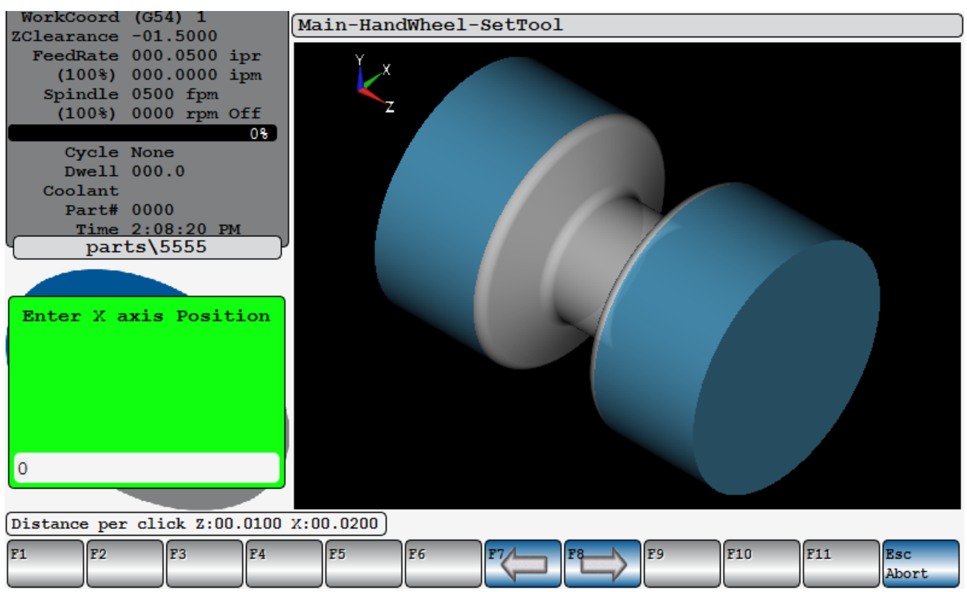
Handwheel mode is used to move the machine around using the electronic handwheel (or dual handwheels if equipped). Its main use is for setting tool length offsets, setting work offsets, and aligning parts. It is also used to step through run cycles in HDW-RUN mode.

F1 & F2 are used to select which axis moves when the handwheel is turned. The feedrate override switch will determine the distance per handwheel click. Dual handwheels if equipped, are both active whenever in handwheel mode.

Note: The distance per click is shown at the left, just above the F-keys.

F3 (Handwheel) – F6 (X-Tool)

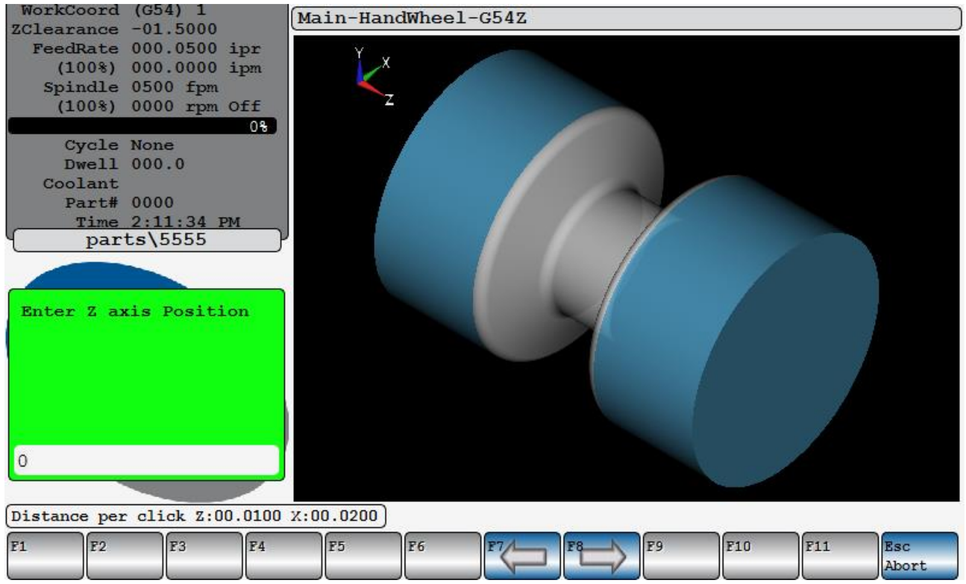
The Main-Handwheel-X-Tool screen appears when the F6 (X-Tool) function key is pressed on the Main-Handwheel screen.



F6 (X-Tool) is used to set a X tool length offset into the tool table. When F6 (X-Tool) is pressed, the CNC prompts for a X position.

F3 (Handwheel) - F7 (G54-Z)

The main-Handwheel-G54Z screen appears when the F7 (G54-Z) function key is pressed on the Main-Handwheel screen.



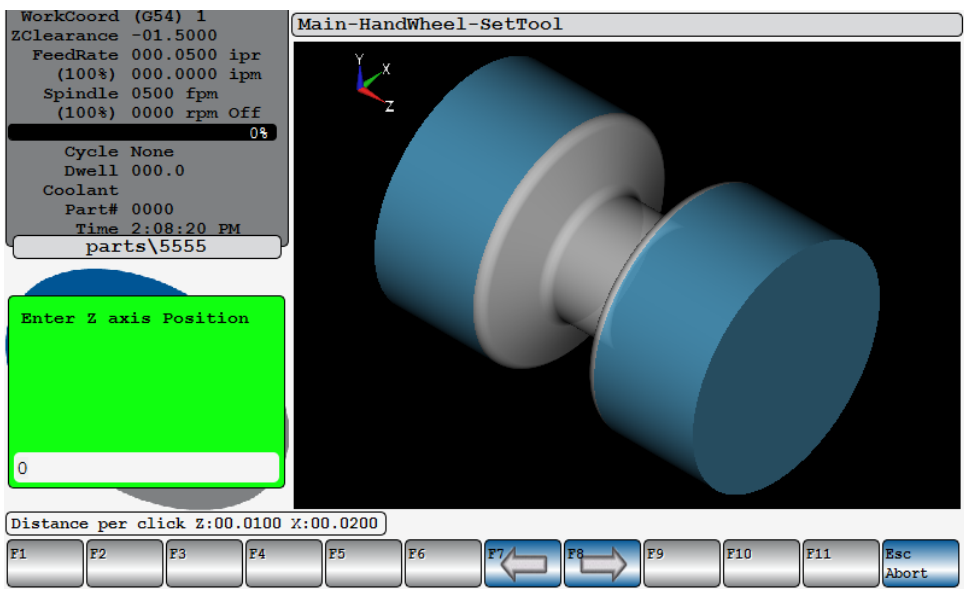
The F7 key sets G54 at the current machine position or at an offset from the current position.

Pressing the F7 key brings up a dialog box asking where the machine is relative to G54Z.

Pressing F7 can be thought of as “call this position ##.####”.

F3 (Handwheel) – F6 (ZTool)

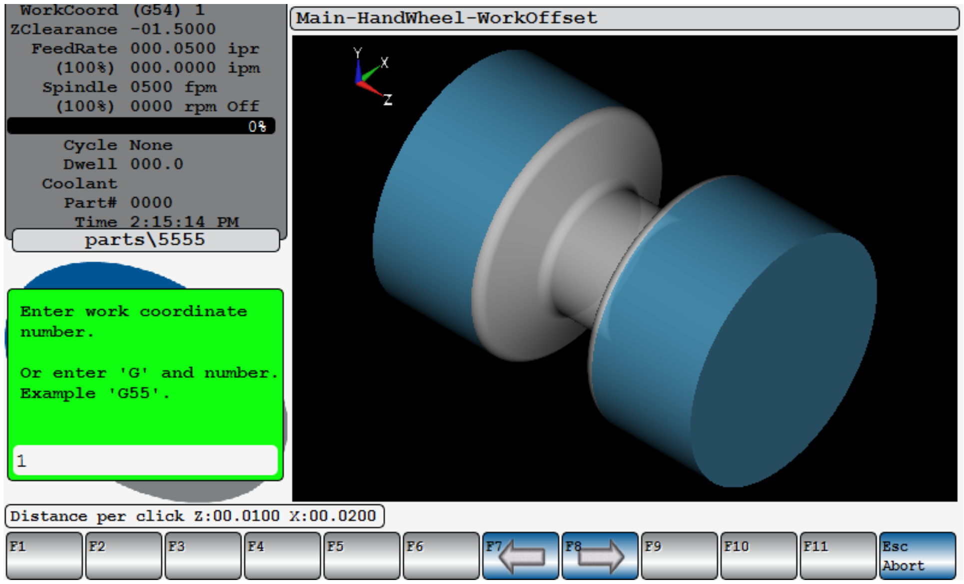
The Main-Handwheel-ZTool screen appears when the F6 (ZTool) function key is pressed on the Main-Handwheel screen.



F6 (ZTool) is used to set a Z tool length offset into the tool table. When F6 (ZTool) is pressed, the CNC prompts for a Z position.

F3 (Handwheel) - F10 (Work#)

The Main-Handwheel WorkOffset screen appears when the F10 (Work#) function key is pressed on the Main-Handwheel screen.



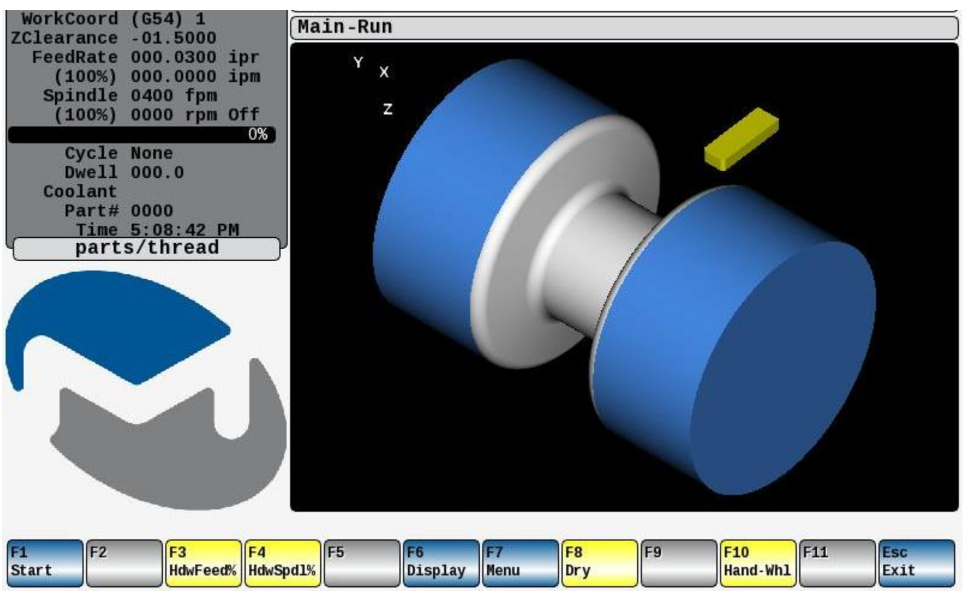
The current work coordinate number is shown on the F10 soft key. When the F10 key is pressed the CNC prompts for a work coordinate number.

Running a Program

Below is listed the general procedure for running the active program. The steps are listed as function keys assuming that the operator begins on the main screen.

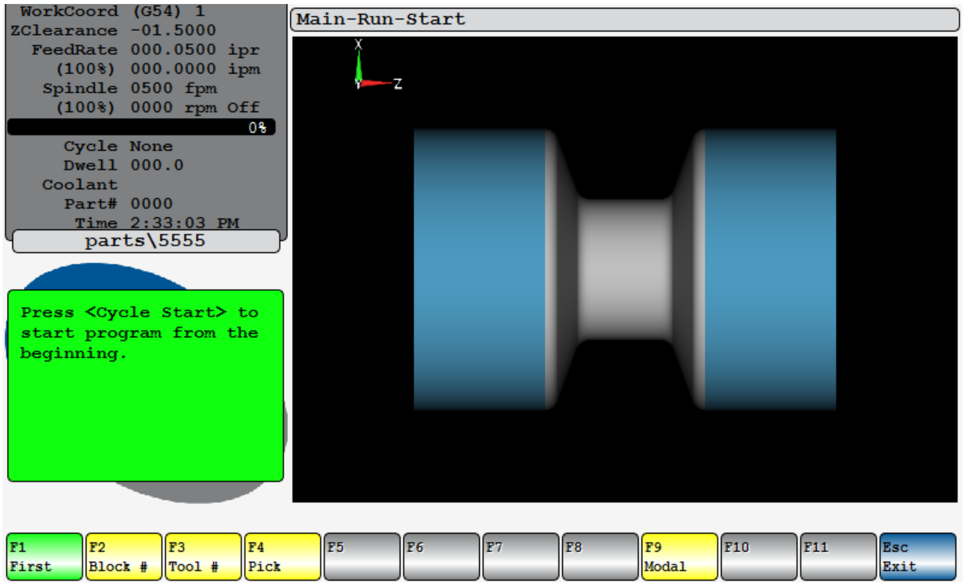
F4 (Run-Prog)

(The machine must be homed prior to running a program)
When from the Main screen the F4 (Run) key is pressed, the following screen appears:



F4 (Run-Prog) → F1 (Start)

Preassing Cycle Start will run the program from the beginning.



Run Start Options

F4 (Run-Prog) → F1 (Start) → F1 (First)

F1 (First) is automatically selected when this screen is displayed. To run the active program from the beginning (from the first block), press Cycle Start.

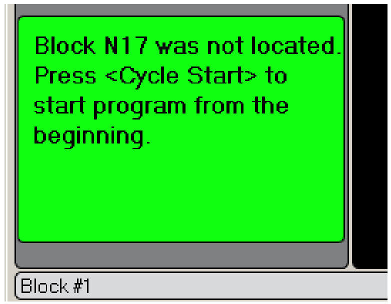
F4 (Run-Prog) → F1 (Start) → F2 (Block #)

If F2 (Block #) is pushed, the control requests the block number to start from. When Cycle Start is pressed, the active program will start running from the selected block number.

F4 (Run-Prog) → F1 (Start) → F3 (Tool #)

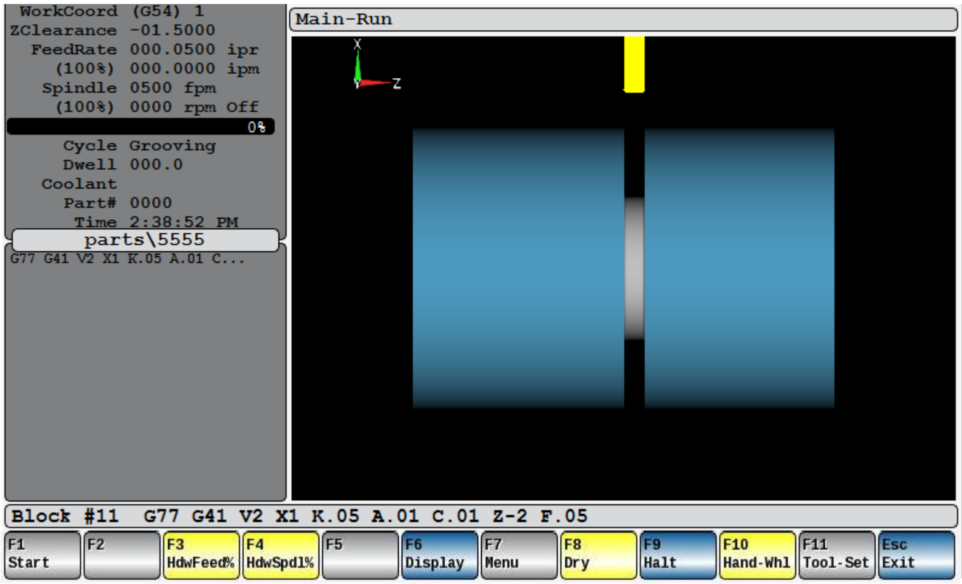
If F3 (Tool #) is depressed, the control requests the tool number to start from. When Cycle Start is pressed, the active program starts running at the desired tool.

Note: If the block number or tool number requested is not found in the active program, the following window will appear.



F4 (Run-Prog) → F1 (Start) → F4 (Pick) (option)

When F4 (Pick) is selected, a program viewer is displayed on the screen making it possible to select a specific line of the program to do a Modal Restart from. Note that the Misc Parameter "allow modal restart" must be set to "true". See appendix more detailed operation notes for the Modal Restart capability.



This screen is the basic run screen. The space just above the F-keys shows the block number and the block currently being executed. The F9 (Halt) key is similar to Feedhold in that when it is pushed the machine will stop. However, unlike Feedhold, F9 (Halt) also can exit the Run mode and allow a new program to be started.

F4 (Run) → F3 (HdwFeed%)

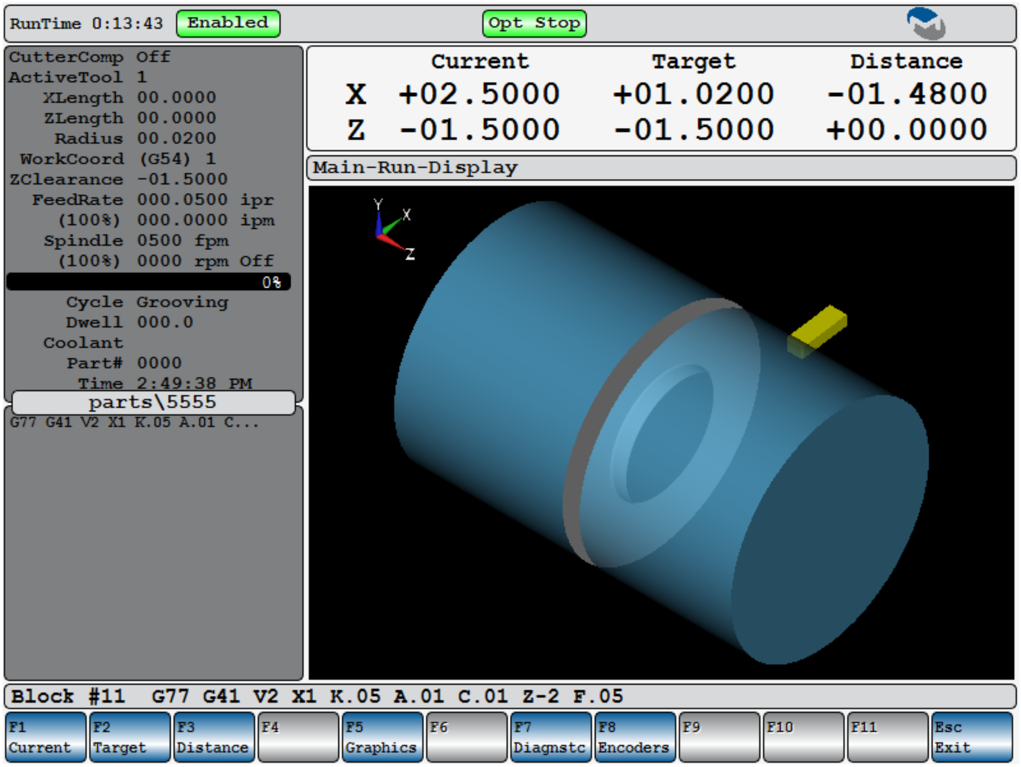
Allows incrementing up to 200%, or decrementing the feedrate override using the handwheel. Each click of the hand-wheel represents 1%.

F4 (Run) → F4 (HdwSpindle%)

Allows incrementing up to 200%, or decrementing the spindle override using the handwheel. Each click of the hand-wheel represents 1%.

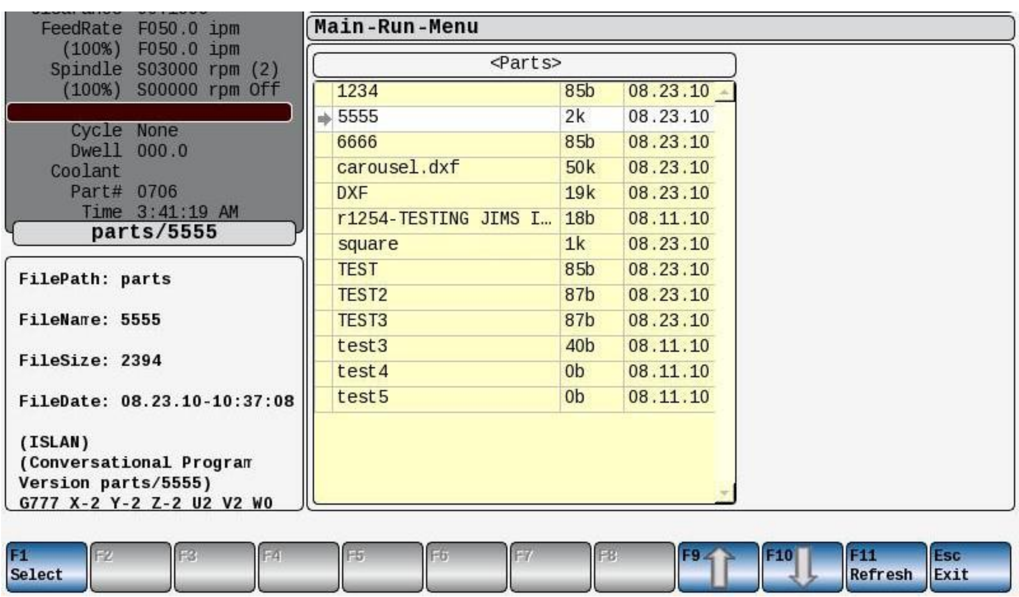
F4 (Run-Prog) → F6 (Display)

The F6 (Display) key can be accessed from a number of screens. The following screen is shown as though the F6 (Display) was entered from the Verify screen. All the display functions and screens are identical, independent of the entry point. See Page 3-13 for a complete list of F6 Display options.



F4 (Run-Prog) → F7 (Menu)

When the F7 (Menu) key is selected from the Run or Verify a window lists all available programs.



To make active one of the programs listed in the window, use the F9 and F10 arrow keys to move the cursor to the desired program and press F1 Select or Enter on the front panel. The Menu function can be called from other screens but works the same way from all. When called from the Run or Verify screen, the selected program becomes the active program. The active program for Run and Verify is always displayed in the space just above the user info window and below the status box.

When called from an edit screen, the selected program becomes the current program being edited. Subdirectories, if any, appear at the top of the list of files and are recognizable by (FOLDER) in the third column instead of a date. For files, summary information is provided in the main file list window while file details are provided in the user info window at left. To navigate to program sources other than the Parts directory, press F9 (Up) repeatedly until the highlight moves into the uppermost window. Position it as needed, e.g. <USB>, then press F1 (Select) or Enter on the front panel to drill down into it.

F4 (Run-Prog) → F8 (Dry)

When F8 (Dry) is active, the active program runs feed moves at the dry run feedrate, even in Feed Per Rev mode (set in Axis parameters). The default Dry Run feedrate is 75 ipm. Switching to or from Dry Run cannot be done if the program is running. The program must first be F9 (Halt)ed, switched to or from F8 (Dry) Run, and then F9 (Resum)ed. Rapid moves remain at rapid speed, and M3s, M4s, M7s, M8s are ignored while in Dry Run mode.

F4 (Run-Prog) → F9 (Halt/Resume)

If a program has been halted, the resume feature of the control becomes active. The F9 (Resume) key will now be displayed on the Run screen. A program can be resumed as long as one of the following functions is not performed: F9 (Verify), F5 (MDI), F1 (Home), or Emergency Stop.

The axes can be jogged or handwheeled away from the work, the spindle may be turned on/off, and F9 (Resume) remains active. As long as the Resume is active, the F9 key on the Run screen will show a Resume function.

When a Resume Cycle Start is selected, the active program will be resumed at the halted point after cycle start is pressed. First, Z will move to the clearance point (R-Plane). Second, X will rapid to the halted point. When X is in position a Cycle Start will be requested. When **Cycle Start** is pressed, the Z axis will feed to its halted point. The program will then start running as if nothing happened.

F4 (Run-Prog) → F10 (Hand-Whl)

When the F10 (Hand-Whl) key is active, program blocks are executed in proportion to the rotation rate of the electronic handwheel and programmed feedrates. Parameters that affect this behavior include:

1. Cranking Minutes/Rev (located in Misc Parameters)
   Multiplying an IPM feed by this factor results in an IPR feed (inches per turn of the handwheel). This should be about 0.0010 for our current system.

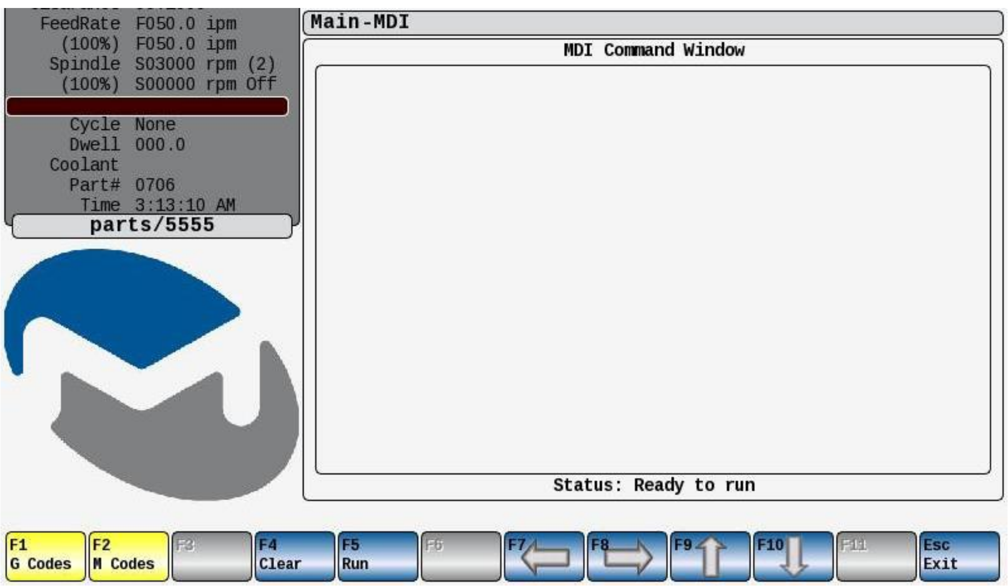
2. Cranking Max IPM (located in Misc Parameters)
   This limits the feedrate while hand-cranking to get reasonable response at slow programmed feedrates. Adjusting the other handwheel parameters can give excess errors on rapids above 100 IPM. It should be about 100 for our current system.
3. Cranking Factor (located in Misc Parameters)
   The multiplier for each handwheel click should be about 100 for our current system.

Other Notes:

Cranking Factor/Cranking Mins Per Rev is proportional to the max feedrate allowed while hand-cranking. If handwheeling and dry running a program, the distance moved per click of the handwheel relates to the Dry Run Feed parameter. When switching from the handwheel mode to automatic mode, the operator is prompted to press cycle start. The direction of the handwheel will not affect the direction of the program execution. If operator is handwheeling a program and a tappingor threading cycle is started, the message *"Tapping or threading cycle will not be in handwheel mode. Press any key to continue"* will appear. The machine taps/threads the hole and returns to handwheel mode.

F5 (MDI)

The F5 (M.D.I.) key on the Main menu selects the MDI (manual data input) function. Through MDI one or several commands may be executed. When MDI is selected the following screen appears.



As commands are typed in they appear in the window. After a line is complete, **Enter** must be pressed to end the block. Several lines of commands may be entered, in essence creating a small program in the MDI window. When command entry is complete, pressing F5 (Run) then **Cycle Start** executes the contents of the MDI window. The Feedhold button, Feedrate Override knob, Spindle Override knob, Single Block button, all work during an MDI sequence. F5 (Stop) terminates any MDI sequence. Pressing F4 (Clear) clears the contents of the MDI window.

F5 (MDI) → F1 (G Codes)

F1 (G Codes) brings up a list of G codes with a short description of each code. Pressing F9 (up arrow) and F10 (down arrow) scrolls the G code list.

00 Linear Rapid	01 Linear Feed
02 CW Arc	03 CCW Arc
04 Dwell	09 One-Shot Exact Stop
10 Set Data Mode	11 Clear Data Mode
12 Clear FLZ	13 Back Programing
14 Front Programing	18 ZX Plane
20 Inch	21 Metric
22 Safe zone check off	23 Safe zone check on
28 Reference Return	29 Return From Ref
30 2nd-4th Ref Return	32 Threading
34 Threading	35 Threading
36 Threading	38 Intial Point return
39 R-plane return	40 CutterComp Off
41 Left CutterComp On	42 Right CutterComp On
45 Z to Clearance	46 X to Clearance
47 Z to ToolChange	48 X to ToolChange
49 Tool Length Cancel	50 Set FLZ, Max RPMs
52 Local Coordinate	53 Machine Coordinates
54 Worksystem 1	55 Worksystem 2
56 Worksystem 3	57 Worksystem 4
58 Worksystem 5	59 Worksystem 6
60 One-Shot Rapid	61 Exact Stop Mode
63 Tapping	64 Cutting mode
65 Move Lockout Block	70 Finish cycle
71 Rough Turning	72 Rough Facing
73 Pattern Repeat	74 Face Grooving
75 Diameter Grooving	76 Threading Cycle
77 Diam/Face Grooving	80 Canned Cycle Cancel
81 Drill Cycle	82 Drill/Dwell
83 Peck/Drill	84 Tap
85 Bore	86 Bore II
89 Bore/Dwell	90 Turning Cycle A
91 Incremental	92 Thread Cutting Cyc.
93 Absolute	94 Facing Cycle B
96 Const Surf Spd	97 Revs per minute
98 Feed per Minute	99 Feed per Revolution
501 Mirror Image Off	511 Mirror Image On
550 Scaling Off	551 Scaling On
568 Rotation On	569 Rotation Off
778 Draw Stock	981 FPR / wait for mark
982 FOV unlock	983 FOV lock at 100%
984 SOV unlock	985 SOV lock at 100%

Note: The text file that displays the legal G codes on the screen is GCODESL.ENG. The control first looks for GCODESL.ENG in /usr/local/bin/ram. If not found, it then searches / usr/local/bin/rom for the file.

F5 (MDI) → F2 (M Codes)

F2 (M Codes) brings up a list of legal M codes with a short description of each code.

00 Program Stop	01 Optional Stop
02 End of Program	03 Spindle On CW
04 Spindle On CCW	05 Spindle Off
06 Tool Change	07 Mist On
08 Flood On	09 Coolant Off
30 Spindle Off, End of Program	90 Graph Off
91 Graph On	93 3D Sweep Off
94 3D Sweep On	95 Tapered Wall
96 Rounded Wall	97 Pocket Clear
98 Call Jump	99 End of Program

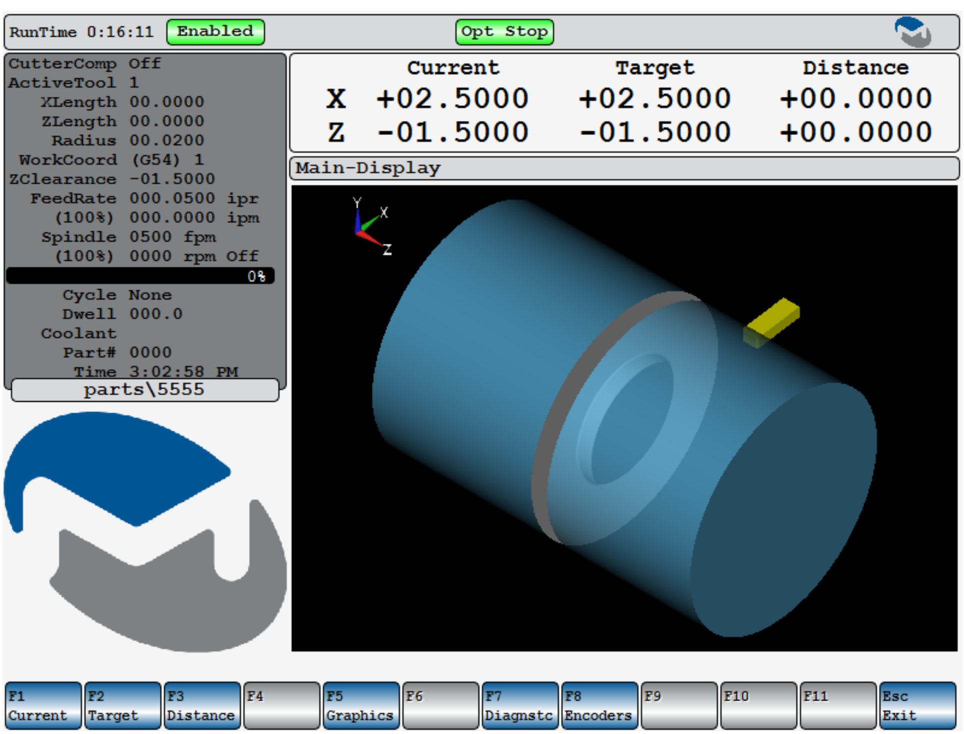
Note: The text file that displays the legal M codes on the screen is MCODESL.ENG. The control first looks for MCODESL.ENG in /usr/local/bin/ram. If not found, it then searches /usr/local/bin/rom for the file. Depending upon the machine options, the legal M codes vary. The MCODESL.ENG in /usr/local/bin/rom is for a basic machine with no options.

F6 (Display)

This menu contains the various display options that are available on the control. Each of these options can also be accessed from the F4 (Run) and F9 (Verify) menus.

F6 (Display) → F1, F2, F3

The F1, F2, and F3 keys are used to select the 3 column displays. *Note: To enable these options control parameter, enable modify column displays must be set.*



The options for each column are:

Current: The position of each axis relative to the active work offsets.

Target: When an axis is moving this indicates where the axis is moving to.

Distance: The difference between the current position and the target position.

Machine: The distance from home (no work offsets)

Measure: Can be used to determine a distance between 2 points, when one of the three options is "measure" the F4 key will be "ZeroMeas". This can be selected to set a zero point for each axis to measure from.

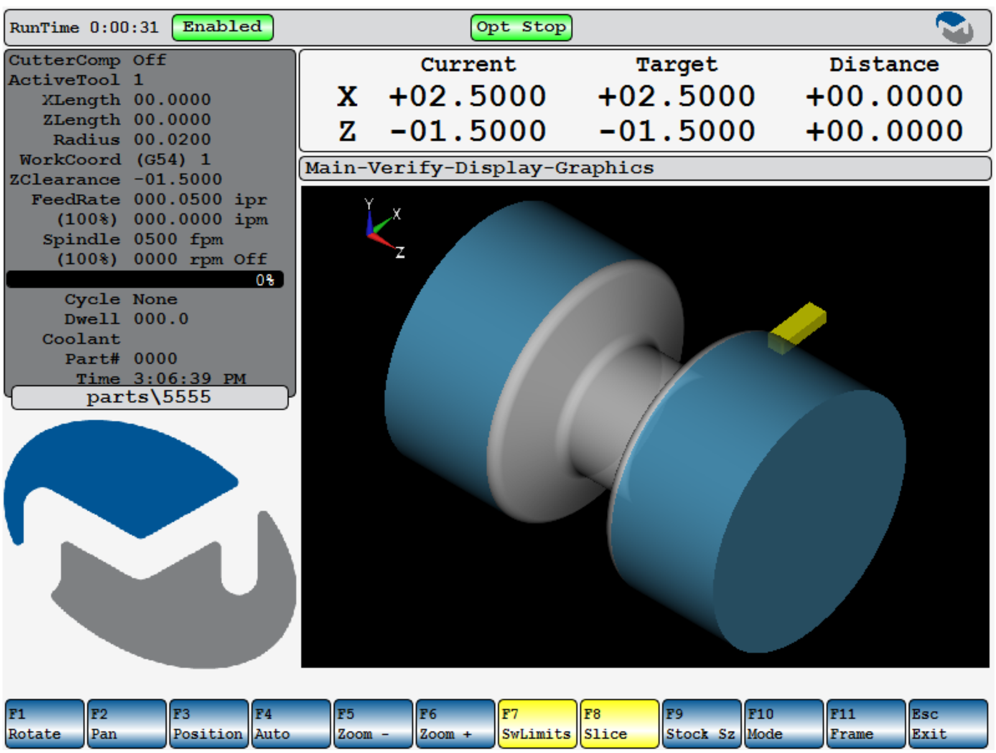
Actual: Actual feedback encoder position.

None: Display is blank

Error: Following error intended for machine setup and troubleshooting. Only an option if setup password has been entered.

F6 (Display) → F5 (Graphics)

If F5 (Graphics) is pressed, the following screen will appear.



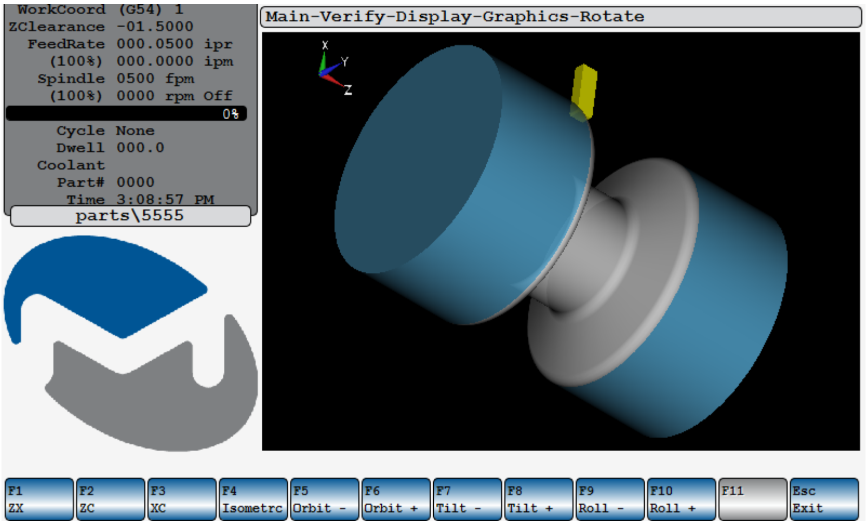
CNC8000 series graphics may be displayed in wireframe or solid model formats, the choice made by toggling F11. There is additional functionality for manipulating the graphic image under F1 (Rotate), F2 (Pan) and F3 (Window). F4 (Auto) resizes the image to fill the display area stock size. Zoom- and Zoom+ behave as expected. F7 and F8 superimpose their respective data when active. F9 is a shortcut key for setting solid model stock size.

The next section will explain how to manipulate the part displayed in the graphics area. All the following functions are accessible through the Graphics screen.

F6 (Display) → F5 (Graphics) → F1 (Rotate) (In solid graphics mode)

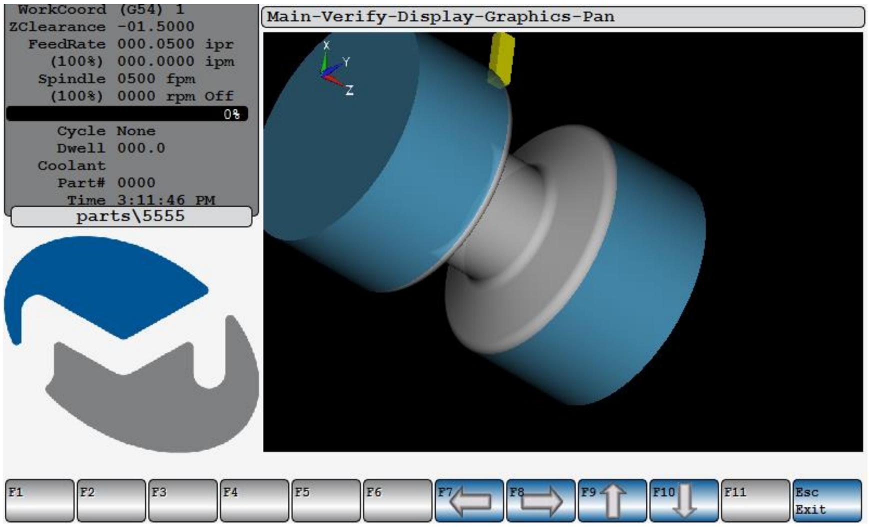
When the display rotate function F1 (Rotate) is selected the following screen is displayed.

The F1 (ZX), F2 (ZC), F3 (XC) and F4 (Isometric) keys give the four standard rotations of a part: ZX top, XC front, XC end, and isometric views. The orientation index in the upper left corner of the screen shows the current part orientation. Pressing the F1 thru F4 keys moves the orientation index to its new position. The F5 thru F10 keys are used to rotate any of the selected axes in 5° increments.



F6 (Display) → F5 (Graphics) → F2 (Pan)

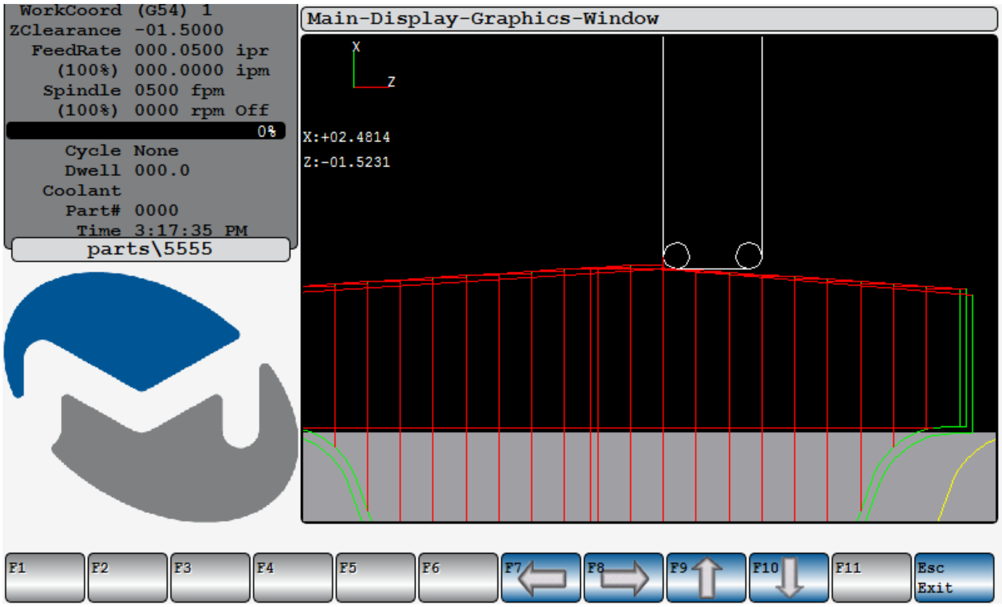
The F2 (Pan) key selects the pan function, which allows the operator to pan around a part. The following display will appear.



Use F7 through F10 to pan the image. An additional **Ente**r key press is required in wire-frame mode to complete the pan maneuver.

F6 (Display) → F5 (Graphics) → F3 (Window) (In wire graphics mode)

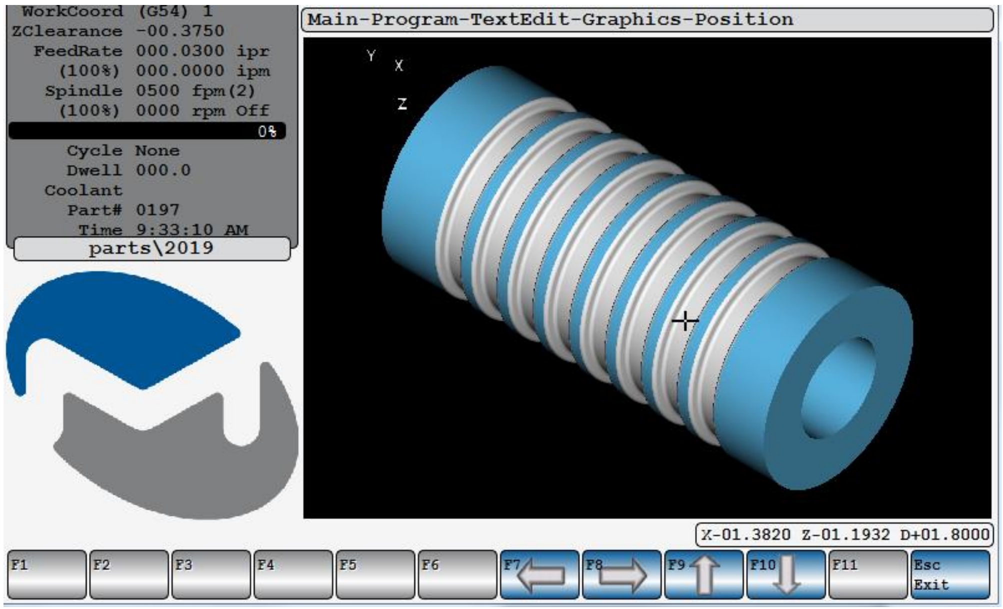
The F3 (Window) key selects the window function, which allows the operator to window in on a particular area of the part. The following display appears when F3 (Window) is selected.



Upon entering F3 (Window) a crosshair cursor appears in midscreen. The cursor can be moved using F7 thru F10 (the same as pan). Move the cursor to one corner of the area to be zoomed. Press **Enter**. Then use F7 through F10 again to move the cursor to the opposite corner of the area to be zoomed and **Enter** again. Window is not available in solid model mode.

F6 (Display) → F5 (Graphics) → F3 (Position) (Solid mode only)

Available only in solid model mode, will put a cross-hair on the part. Moving the cross-hair using the F7-F9 keys will show the dimensions of the part in the lower right of the display.



F6 (Display) → F5 (Graphics) → F4 (Auto)

Will auto scale the part to fit in the graphics window.

F6 (Display) → F5 (Graphics) → F5 (Zoom -)

Zooms out

F6 (Display) → F5 (Graphics) → F5 (Zoom +)

Zooms in

F6 (Display) → F5 (Graphics) → F7 (SwLimits)

The F7 (SwLimits) key draws a box on the screen which corresponds to the axis limits of the machine. This allows viewing of the part in relation to the machine's overtravels. If the part extends beyond this box, it cannot be run on the machine unless some corrective action is taken to change the work offsets. The axis overtravel limits are set from the parameter screens. If the tool is programmed outside this box, an "axis software limit overtravel" error will result.

F6 (Display) → F5 (Graphics) → F9 (Coords) (Wire graphics mode)

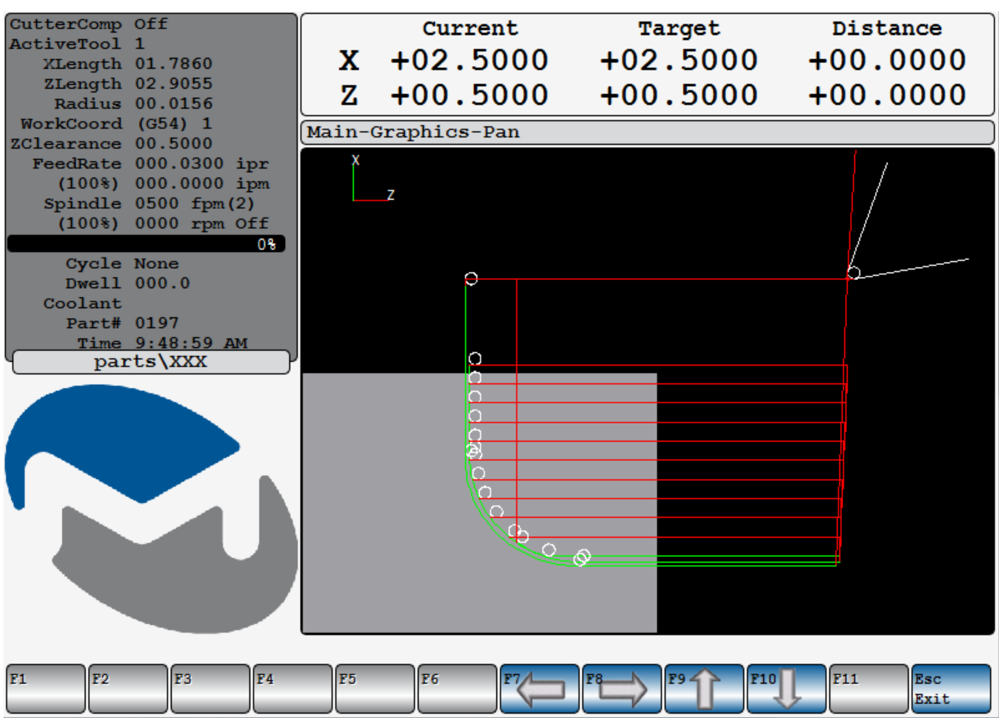
The F9 (Coords) key draws an axis coordinate through X0, Y0, Z0 in wireframe graphics mode. This gives a visual reference to where the zero is on the part.

F6 (Display) → F5 (Graphics) → F9 (Stock) (Solid mode only)

Is a shortcut key for setting the solid model stock size. The user can set the stock type by filling in the appropriate fields.

F6 (Display) → F5 (Graphics) → F10 (ShowTool) (Wire mode only)

A circle based on the tool size will be shown at the end points of each line and arc in the display. When verifying both part and tool path this can be useful for showing cutter compensation.

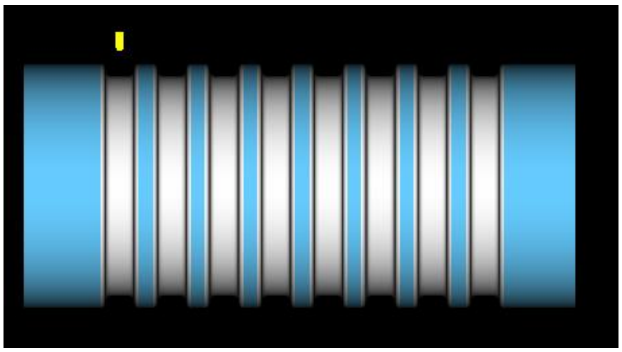
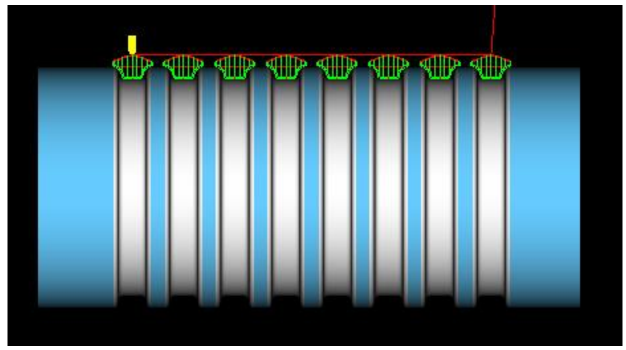
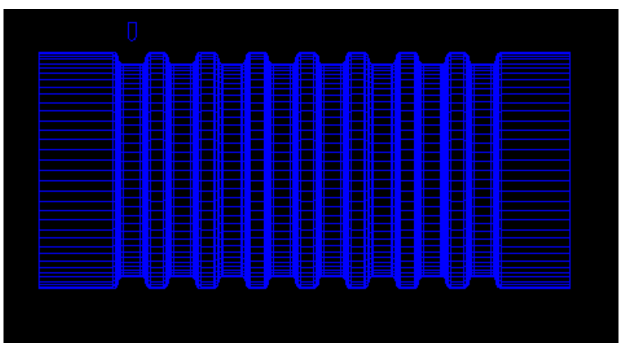
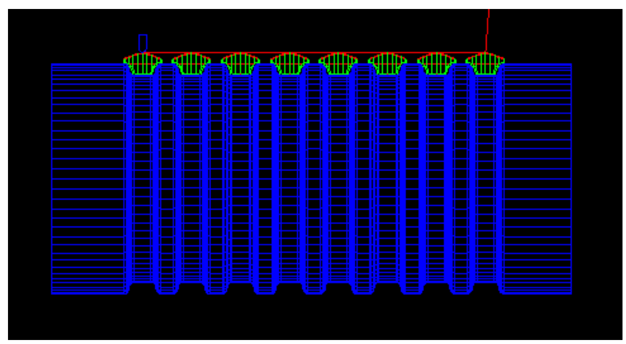


F6 (Display) → F5 (Graphics) → F10 (Mode) (Solid Mode Only)

Toggles between eight display modes (or views). The eight modes are:

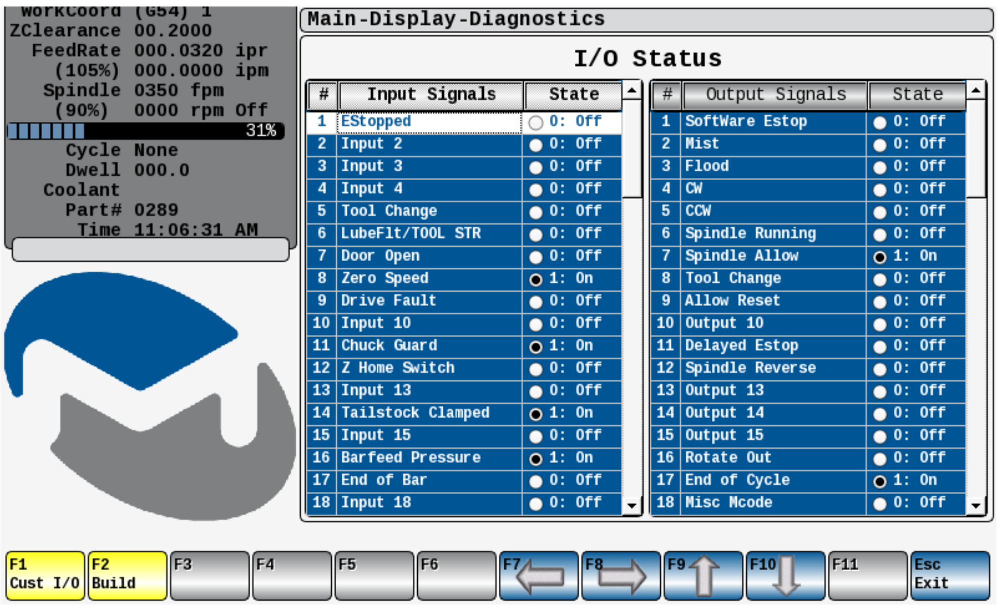
- Solid - opaque - no tool paths
- Wire - opaque - no tool paths
- Solid - transparent - no tool paths
- Wire - transparent - no tool paths
- Solid - opaque - with tool paths
- Wire - opaque - with tool paths
- Solid - transparent - with tool paths
- Wire - transparent - with tool paths

Several of the modes are shown below:

F6 (Display) → F7 (Diagnostic)

Underneath F6 (Display), the F7 (Diagnostic) key is used for machine setup and troubleshooting. The diagnostic screens bring up I/O connected to the CNC. The status of each bit is continuously displayed on the screen. Position of the highlight bar defaults to input 1, the topmost input on the left hand list. Function keys F7 and F8 move the highlight bar left and right between the left hand column of inputs and the right hand column of outputs. A white radio button indicates the input or output is in the OFF state, black indicates it is in the ON state.



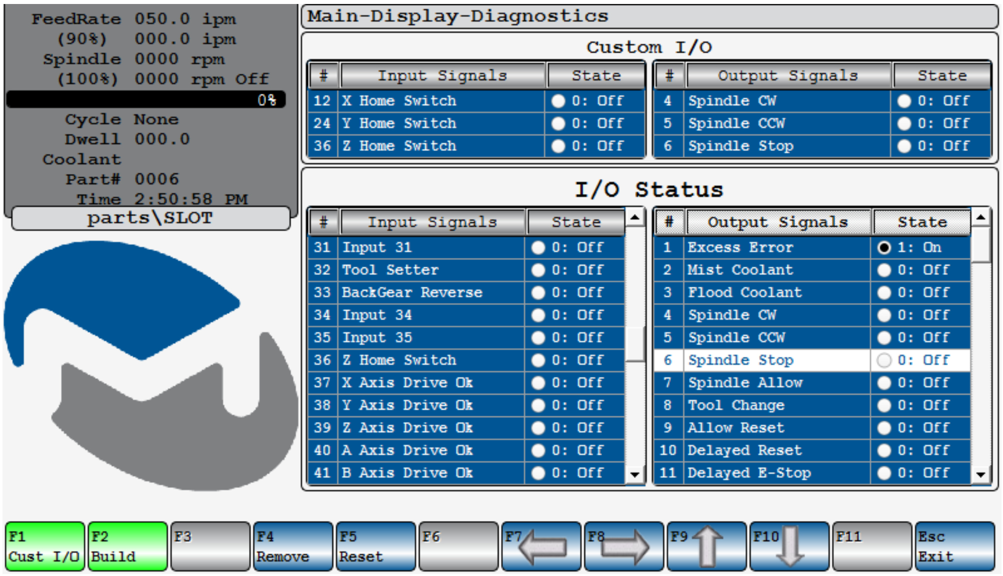
Note: Diagnostic screens differ from machine to machine, depending on machine type and options. The text that shows up on the screen is from the files /usr/local/bin/ram/ INPUT.ENG and /usr/local/bin/ram/OUTPUT.ENG. Default INPUT.ENG and OUTPUT. ENG files for a basic machine with a minimum of options are present in the rom directory. Files for a particular machine with its particular options and I/O complement are in /usr/local/bin/ram. The control first checks /usr/local/bin/ram for INPUT.ENG and OUTPUT.ENG. If it does not find them, it uses the default files.

Diagnostic display is only available if a setup password has been entered.

Creating a Custom I/O Display

A Custom I/O list can be created from the diagnostic display menu. Pressing F2 (Build) allows inputs and outputs to be selected and added to the custom I/O display. When "build" is active, position the cursor to the signal you wish to display, then press F4 (Add), pressing F4 again will "remove" the signal from the custom I/O list. F5 (Reset) clears the custom I/O display. Once out of "build" mode, pressing F1 (Cust I/O) repeatedly causes the custom I/O list to be turned on and off alternately.

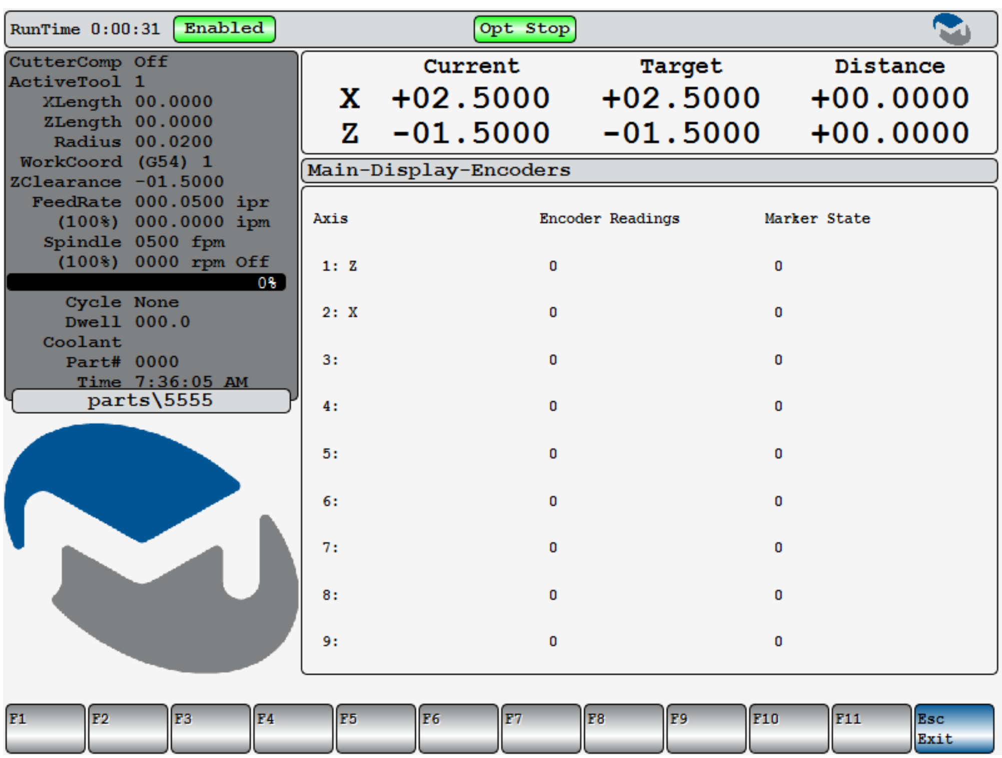
Custom I/O Display



When active, the custom I/O screen remains docked at the top of the main graphics window. It may be used in MDI, for example, to allow a troubleshooter to see the effect on I/O as he executes an MDI command. To suppress the custom I/O display, toggle it off by pressing F1 (Cust I/O) from the F6 (Display) F7 (Diagnstc) menu. This causes the key to turn yellow (inactive).

F6 (Display) → F8 (Encoders)

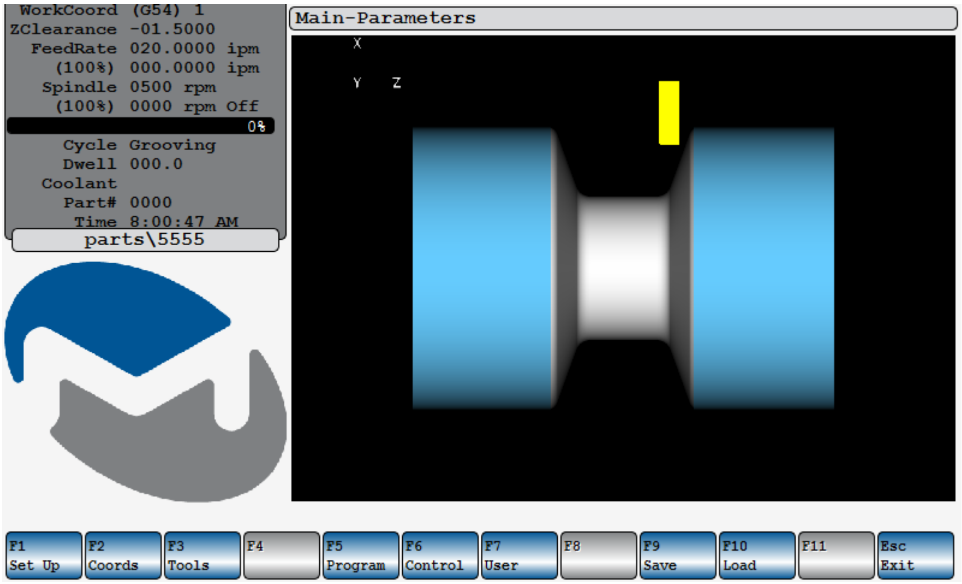
F8 (Encoders) is displayed when the (F8) Encoders function key is pressed on the display screen. "Encoders" is only available if a setup password has been entered. It shows the actual encoder counts and marker state of each encoder.



Encoders display is only available if a setup password has been entered.

F7 (Paramtrs)

The F7 (Paramtrs) key from the main screen brings up the following parameter menu.

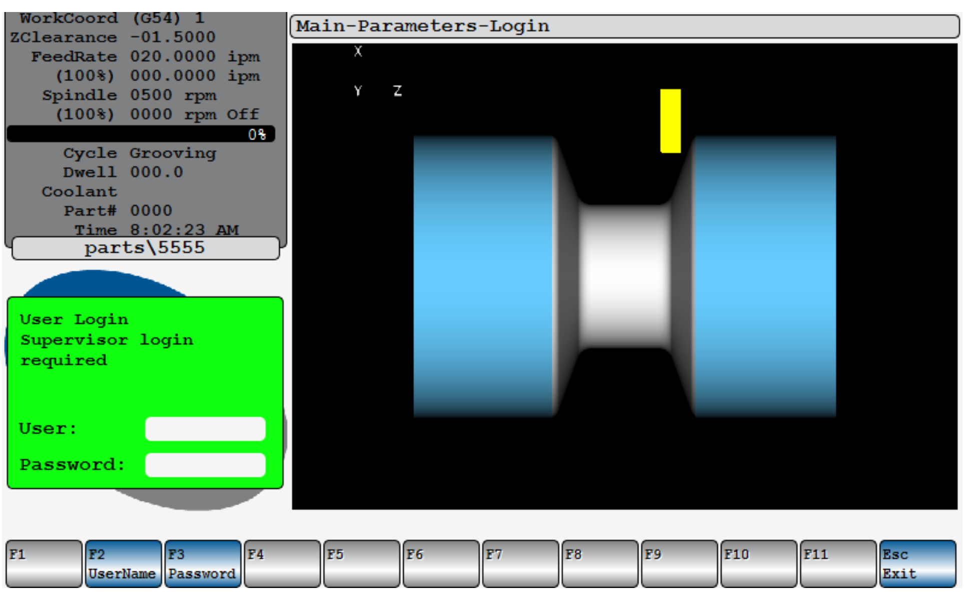


F7 (Paramtrs) → F1 (Set Up)

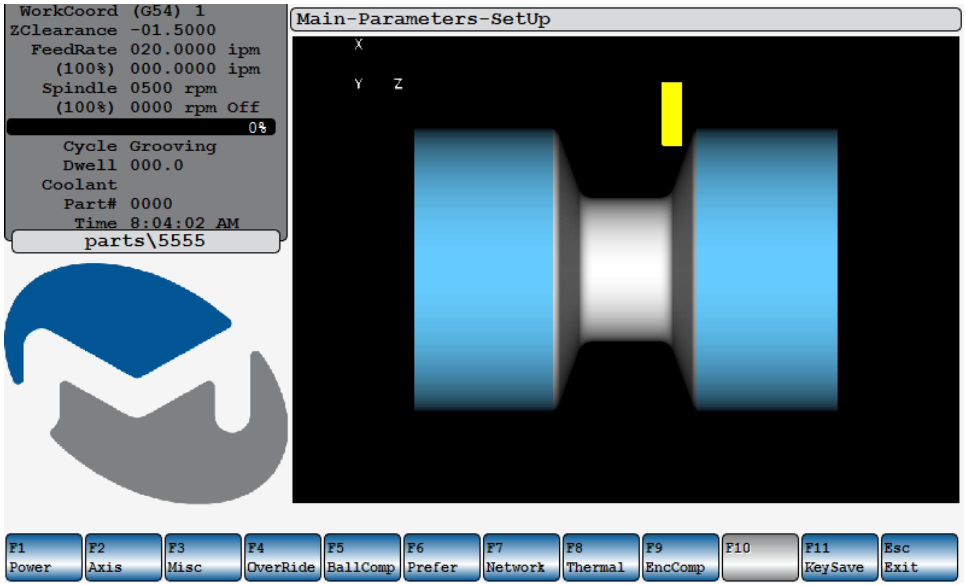
The F1 (Set Up) selection brings up the parameters that make the control unique to a particular machine or application. When F1 (Set Up) is selected the following screen appears.

Note: The parameters in the setup sections are normally set by the machine tool builder. Changing these parameters can affect a large number of machine functions and machine performances and should only be modified by experienced service personnel.

The CNC requires a User and Password to allow the machine setup parameters to be displayed or changed.



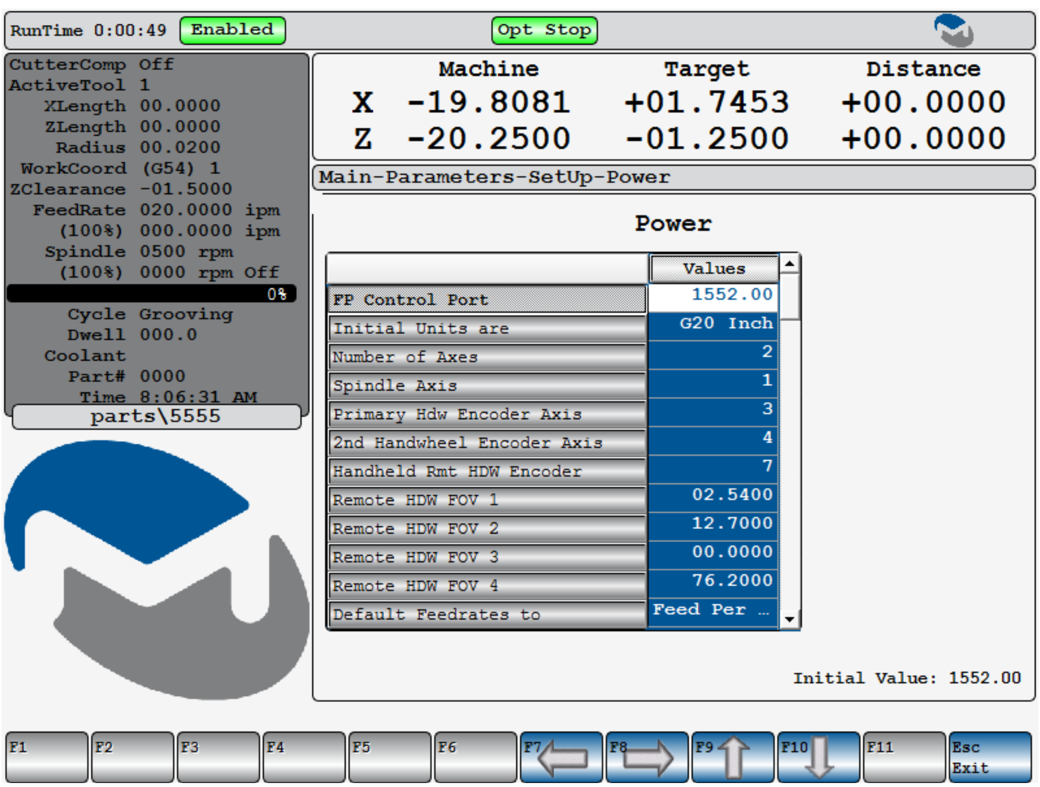
Assuming the correct code has been entered, the setup parameters may be accessed.



F7 (Paramtrs) → F1 (Set Up) → F1 (Power)

Power parameters are parameters that directly relate to the configuration of the machine tool and will normally be set by the machine tool builder. Power parameters are read by the CNC only at power-up.

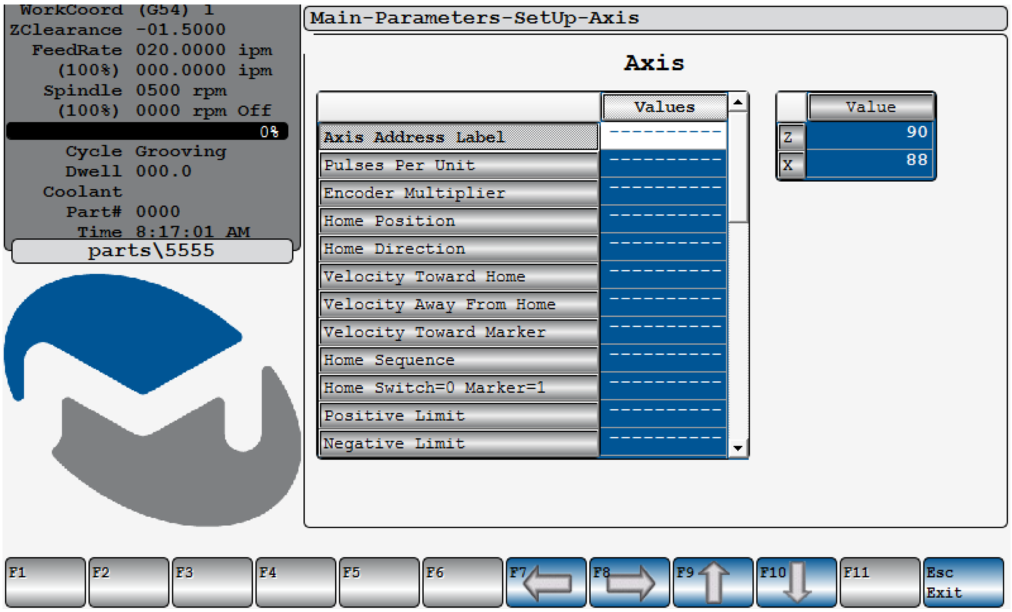
The F1 key brings up the power-up defaults as shown in the following screen.



See the Appendix for a listing of Machine Setup POWER parameters.

F7 (Paramtrs) → F1 (Set Up) → F2 (Axis)

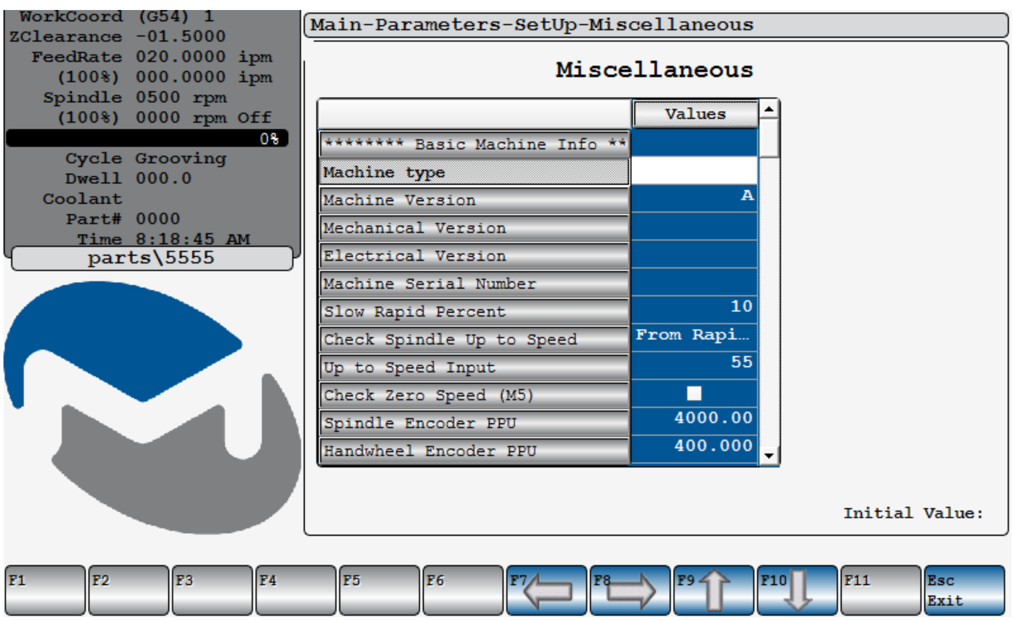
If the F2 (Axis) key is pressed, the following screen will be displayed.



See the Appendix for a listing of Machine Setup AXIS parameters.

F7 (Paramtrs) → F1 (Set Up) → F3 (Misc)

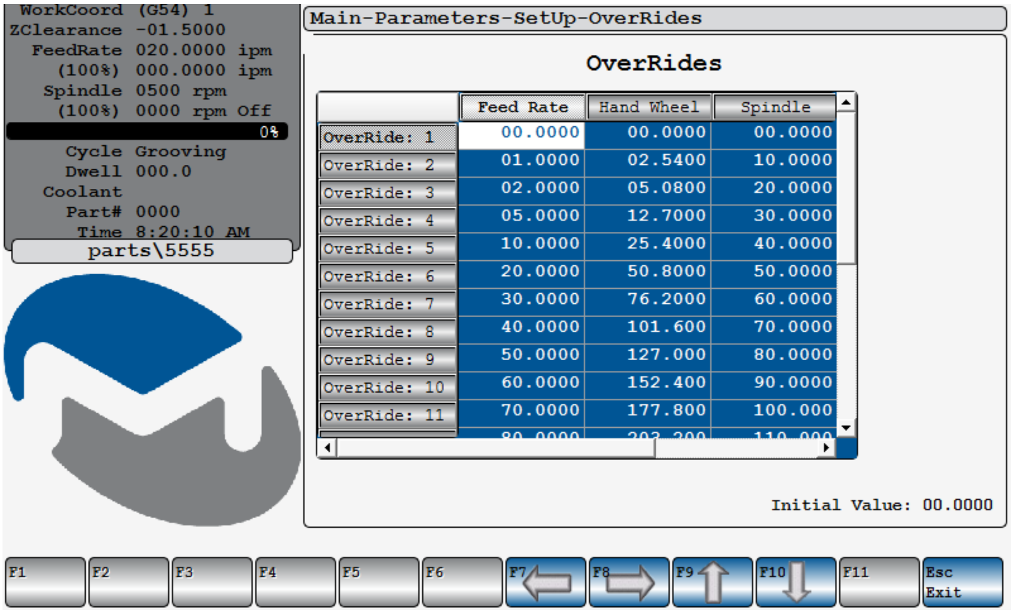
The F3 (Misc) key brings up various miscellaneous setup parameters dealing with the spindle and M codes. When F3 (Misc) is selected, the following screen appears.



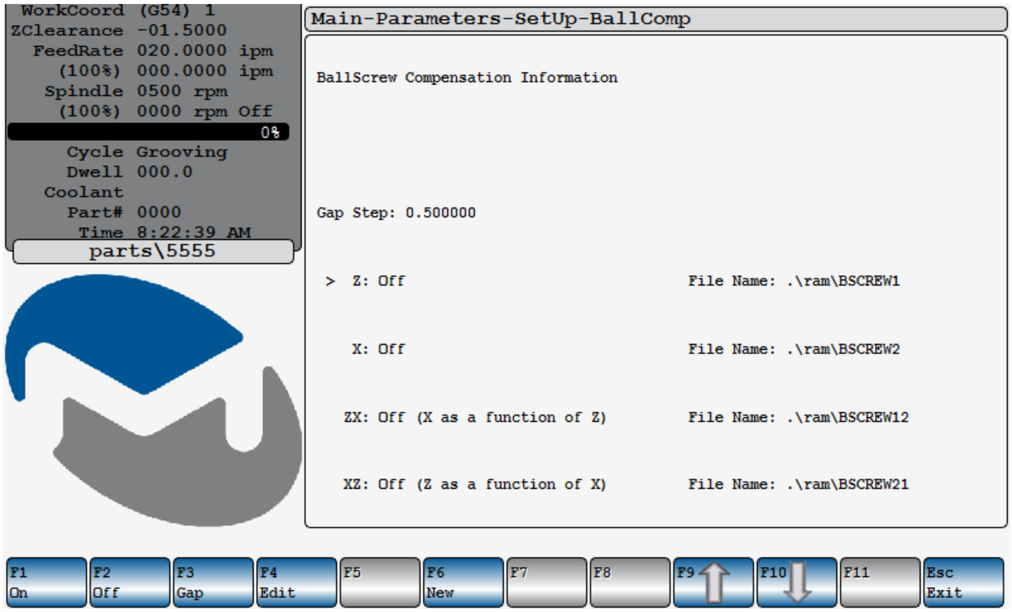
Miscellaneous parameters are edited similarly to Power and Axis parameters see the Appendix for a listing of Machine Setup MISC parameters.

F7 (Paramtrs) → F1 (Set Up) → F4 (OverRide)

The F4 (OverRide) key brings up the feedrate override parameter settings. These settings determine the feedrate percentage assigned to each of the feedrate override rotary switch positions, the increment per click of the handwheel for each of the feedrate override rotary switch positions, and the spindle speed percentage for each of the spindle override rotary switch positions. Without a compelling reason not to do so, the Feed Rate and Spindle percentages should match the values etched on the molded front panel next to the rotary switch positions. The following screenshot shows the override settings page.



F7 (Paramtrs) → F1 (Set Up) → F5 (BallComp)



Ballscrew Compensation Table Creation Help

Use F9 (up arrow) and F10 (down arrow) to select the comp axis
F1 (On) turns ballscrew comp on for given axis.
F2 (Off) turns ballscrew comp off for given axis.
F3 (Gap) changes the spacing in the ballscrew file generated from F6 (New).
F4 (Edit) jumps into editor with ballscrew table.
F6 (New) creates a new, zero ballscrew table.
ESC (Done)

Ballscrew Compensation (“Ball comp”) is turned on after homing. It is not turned on after a Home → Here. Homing will not load a new table if the table has been edited or added. It will only make active a table previously loaded during power-up initialization.

F7 (Paramtrs) → F1 (Set Up) → F6 (Prefer)

The preferences page is for setting passwords for super user access and for supervisor access. The preferences page also is used to set the access level for the machine at power up. The default passwords are always in effect.

F7 (Paramtrs) → F1 (Set Up) → F7 (Network)

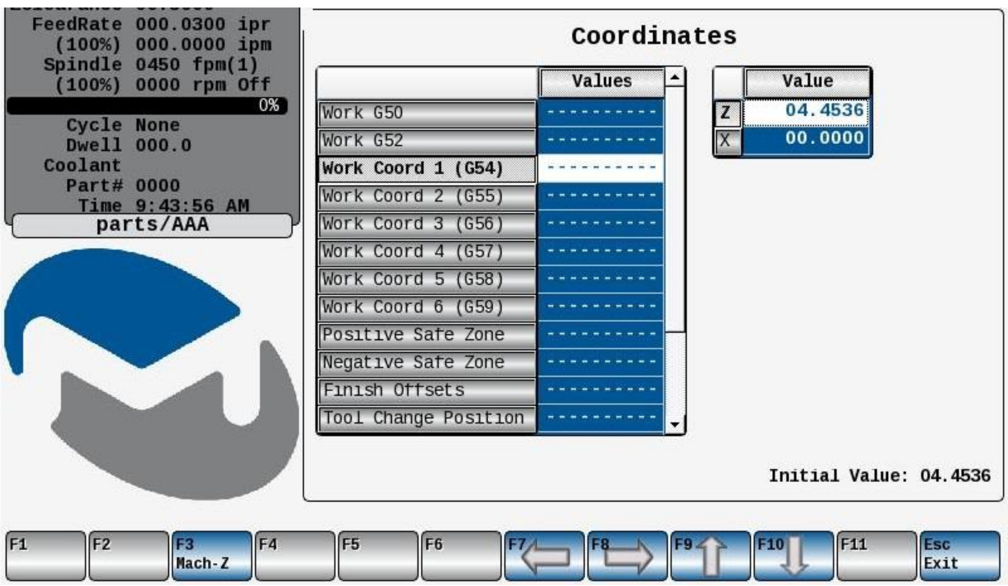
On this page, enter IP addresses, passwords, etc. for network connection

F7 (Paramtrs) → F1 (Set Up) → F8 (Thermal)

On this page, monitor and control axis thermal compensation.

F7 (Paramtrs) → F2 (Coords)

The F2 (Coords) key of the parameter screen brings up the parameters dealing with the various coordinate systems in the control. The highlight defaults to Work G50. To edit the familiar G54 work coordinates, press F10 (down arrow) twice to bring the highlight down to Work Coord 1, then press F8 (right arrow) to bring the highlight into the settings box at right. New values are written when ESCaping to the main menu.



When editing the coordinates, F3 (Mach-Z) will set the coordinate for the Z axis to the current position.

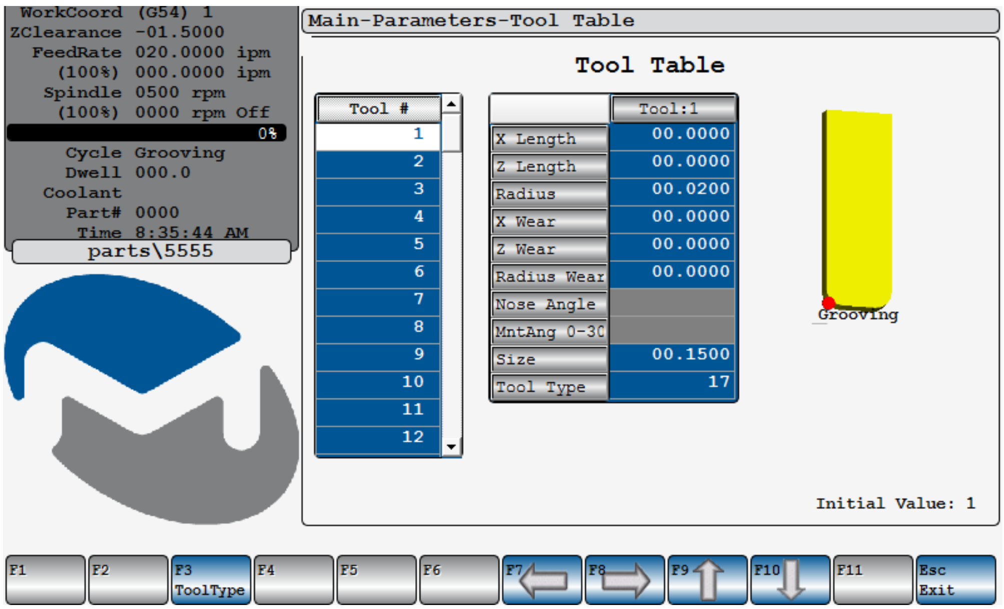
The Coords settings page has capacity for up to 6 work coordinates. In addition to the work coordinate settings, at the bottom of the list there are fields for setting positive and negative safe zones, tool change position, and G28 and G30 reference points.

Positive Safe Zone Z 00.0000 X 00.0000	A position relative to machine zero which, along with the negative safe zone position, describes a cube that the machine cannot enter. If the machine is programmed into this cube an error will be displayed.

Negative Safe Zone Z 00.0000 X 00.0000	G22 turns the safe zone off G23 turns the safe zone on
Tool Change Z + 00.0000 X + 00.0000	Z moves to this location on a G47 (Z to toolchange) command. X moves to this location on a G48 (X to toolchange) command. This position is specified in inches or mm, and is home relative.

F7 (Paramtrs) → F3 (Tools)

The F3 (Tools) key brings up the following screen.



Fields that are in the tool table are selectable in the Power parameters. The table above shows all items in the tool table. It may be desirable not use the wear offsets. Specific tools do not use some of the items. For example grooving tools would not use a nose angle. The fields that are unused are grayed out. Section # page # shows specifics about how to set up tools.

F7 (Paramtrs) → F5 (Program)

These parameters provide access to all the internal parameters the CNC uses to execute a program. Normally these parameters would be used for display purposes only as an aid to program debugging. However, it is possible to read and change these parameters in a parametric program. Great care must be taken when doing this because these parameters are used directly by the CNC to produce the next machine movement or function.

See Appendix for the complete listing of Program Parameters.

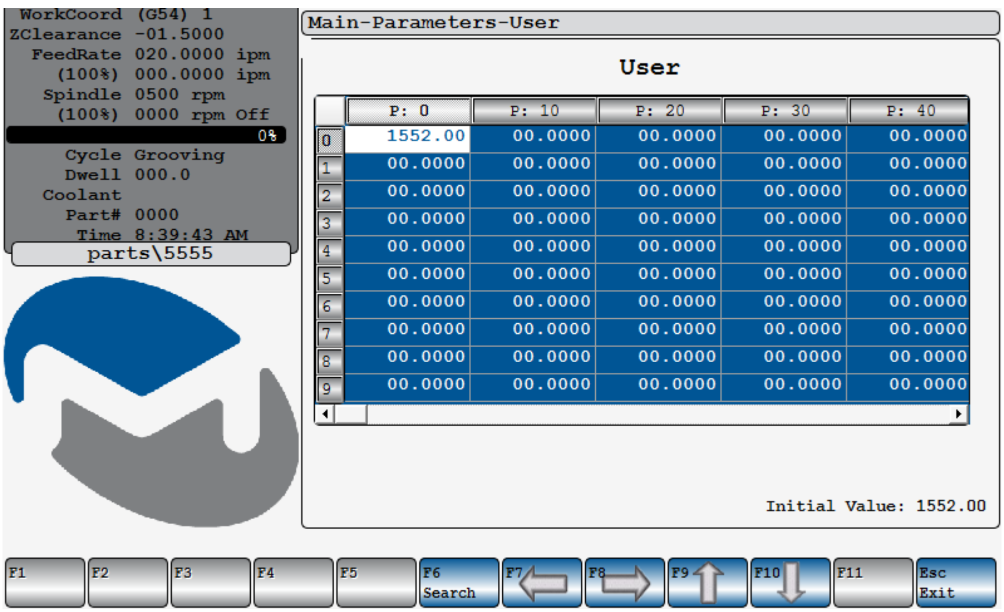
F7 (Paramtrs) → F6 (Control)

The collection of control parameters, in the beginning, is a miscellany of switches and values for control behavior and, at the end, is a listing of variables used to create autoroutine and canned cycles.

See Appendix for the complete listing of Control Parameters.

F7 (Paramtrs) → F7 (User)

This set of 100 parameters is reserved for the parts programmer to use when writing parametric programs. These parameters are undefined and can be edited, displayed, or loaded from this screen. See Section Eight for information on parametric programming.



F7 (Paramtrs) → F9 (Save)

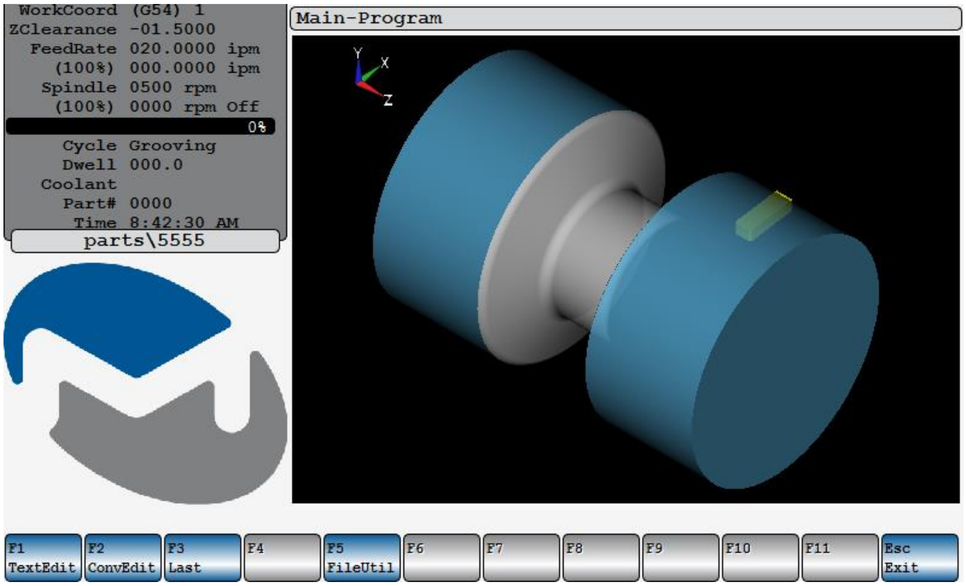
F6 (Save) saves all files in the /ram directory to a USB flash drive. Machine parameters of all types are saved in the /ram directory. Before a major change to control hardware or software, it is either prudent or necessary to save parameters off to a flash drive so the machine condition may be recreated after the hardware and/or software change is complete.

F7 (Paramtrs) → F10 (Load)

Loads files from a USB flash drive to the /ram directory. After parameters have been saved to a USB flash drive and a major change to control hardware or software is complete, then loading parameters from the USB flash drive restores the machine to the state it was in when parameters were saved, including work coordinates, tool offsets, slots table, current tool, active program.

F8 (Program)

There are two modes of program file creation/editing available on 8000 Series controls: text and conversational. Pressing F8 (Program) enables selection of the type of programming. It also allows quick access to the last program edited and access to the file utility screen.

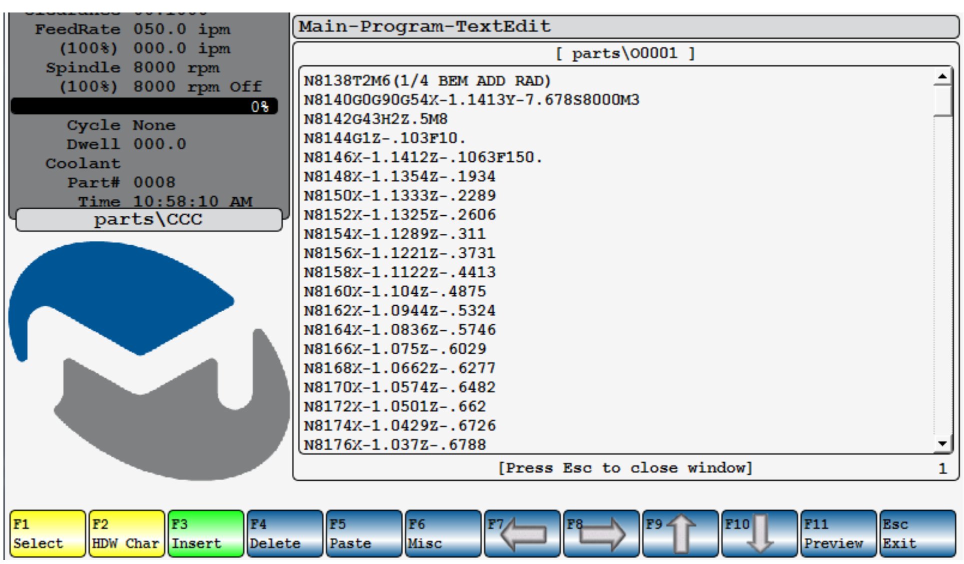


Text and conversational programs are stored in the control in different file formats and have different extensions. All conversational programs have a .CNV extension. A program without a .CNV extension is a text program. So a program with no extension, or one with a .NC extension, or a .txt extension -- all are text programs.

The control post-processes a conversational program into M- and G-codes and creates a new text program without the .CNV extension. A program created as a conversational program exists in two forms. It exists as a conversational program (PARTPROGRAM. CNV) and as a text program (PARTPROGRAM).

F8 (Program) → F1 (TextEdit)

Pressing F1 (TextEdit) brings up a menu of all programs in the <parts> directory. The highlight defaults to the last program edited. Pressing F1 (Select) opens the highlighted file for editing.



F8 (Program) → F1 (TextEdit) → F1 (Select)

Pressing F1 (Select) marks the beginning of a section of text to highlight. Moving the cursor laterally with F7 (Left Arrow) and F8 (Right Arrow) keys extends the highlight within a line. Using the arrow keys F9 (Up Arrow) and F10 (Down Arrow) extends the highlight up and down. Highlighted text has a dark background. The usual array of editing actions (copy, cut, paste) is available. In the Series 8000 CNC, text may be copied and pasted from one program to another.

F8 (Program) → F1 (TextEdit) → F2 (Hdw Char) or F2 (HdwLine) or F2 (HdwPage)

Selects how the cursor moves per click of the handwheel.

F8 (Program) → F1 (TextEdit) → F3 (Insert)

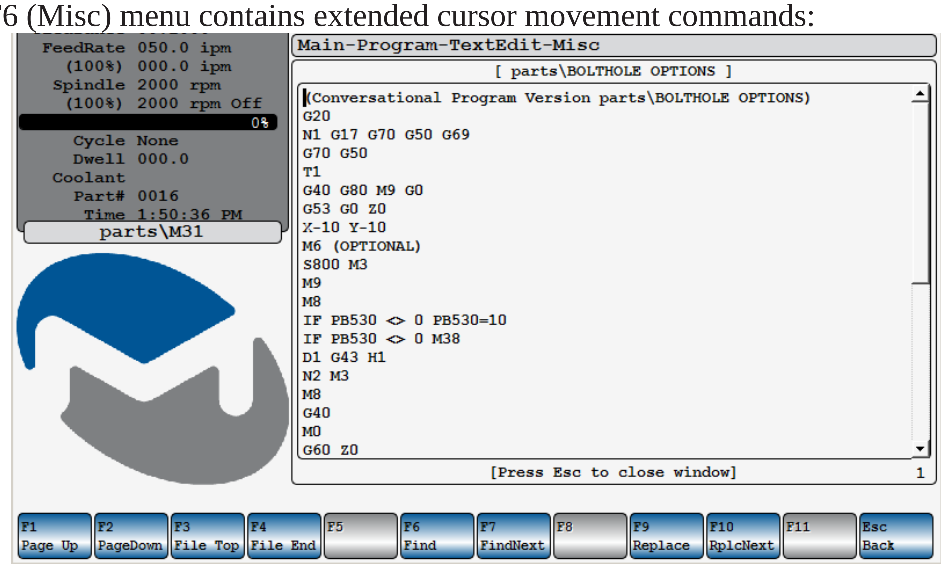
Toggles from insert to overwrite mode.

F8 (Program) → F1 (TextEdit) → F4 (Delete)

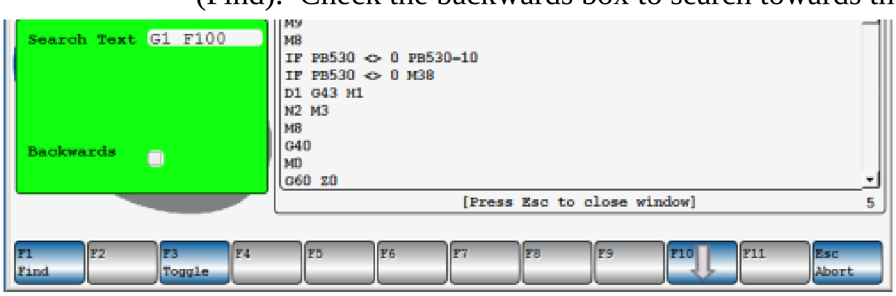
Deletes the character the cursor is on.

F8 (Program) → F1 (TextEdit) → F6 (Misc)

The F6 (Misc) menu contains extended cursor movement commands:

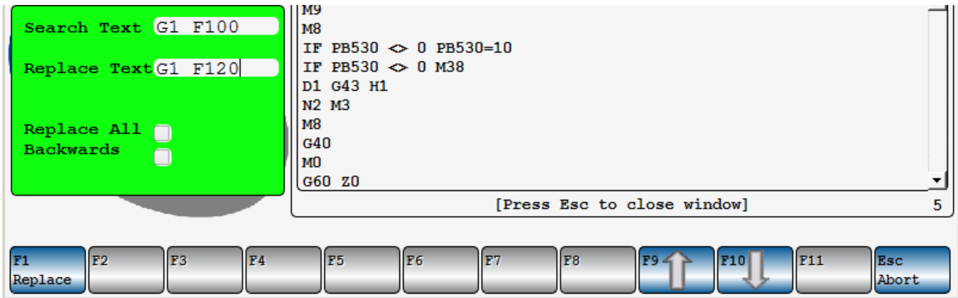


F1 (Page Up)	Moves the cursor up 1 page.
F2 (Page Down)	Moves the cursor down 1 page.
F3 (File Top)	Moves the cursor to the top of the program.
F4 (File End)	Moves the cursor to the bottom of the program.
F6 (Find)	Used to find text in a program. Enter the text to search for and press F1 (Find). Check the backwards box to search towards the top of the program.



F7 (FindNext) Searches for the next occurrence of the same text string.

F9 (Replace) Used to replace text in a program. Enter the text to search for the text to replace it with and press F9 (Replace). Check the replace all box to replace all of the search strings between the curser and the bottom of the file, (or the top of the file if the backwards box is checked).



F10 (RplcNext)

Repeats the previous replace command.

F8 (Program) → F1 (TextEdit) → F11 (Preview)

A file may be verified while it is being edited.
(Preview-within-Edit is not available if another program is currently running or verifying.) When F11 (Preview) is pressed, the program is Fast verified without cutter comp in wireframe or in solid model mode, depending which mode is currently selected. Graphics are auto-scaled. They can be rotated, scaled, zoomed, etc. Pressing ESC returns to the text editor.

Notes: When the program is being Previewed, it will ignore M6s, M0s, M1s, INPUT statements, etc. Preview-within-Edit is designed to make a quick sketch of the program toolpath. A full-featured Verify (similar to Run), if required, may still be performed on the part program at any time by exiting the editor, returning to the main screen, and doing a Verify from there.

F8 (Program) → F1 (TextEdit) → Esc (Exit)

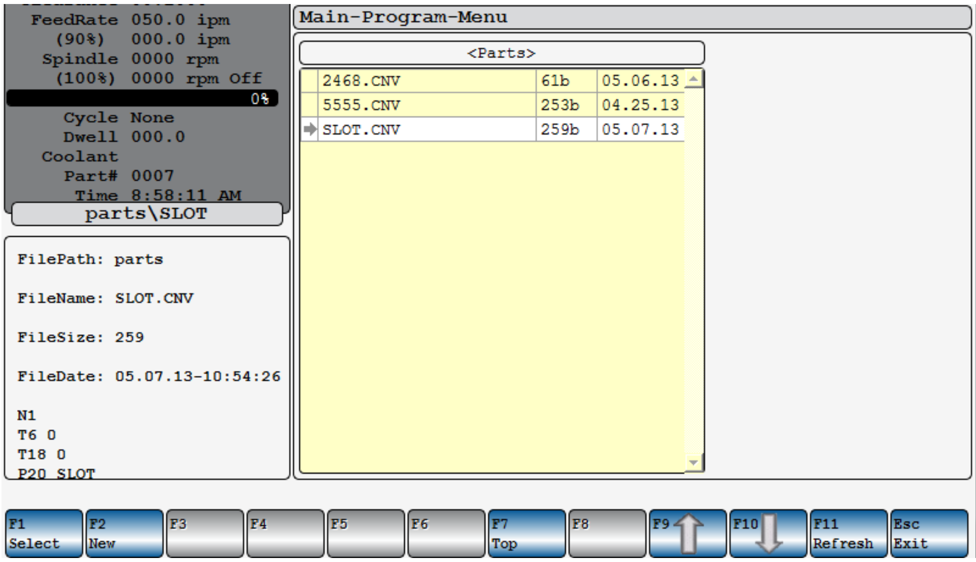
Upon pressing the ESC (Exit) key to leave the editor, the active edit program is checked to determine if it was modified. If it was, a prompt will be displayed in the message window asking if the changes should be accepted and stored. Pressing F1 (Yes) accepts the changes and alters the program file. Pressing F5 (No) aborts the changes and leaves the file unchanged.

F8 (Program) → F1 (TextEdit) → F2 (New)

The F2 (New) key allows entry of a new text program name. After a name is entered, a blank window is available for entry of a new program.

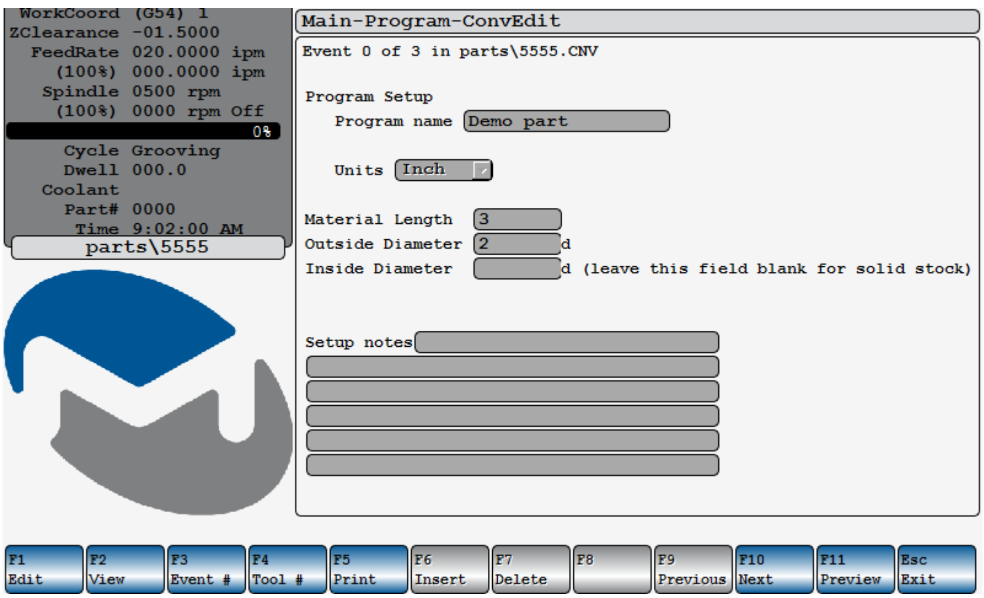
F8 (Program) → F2 (ConvEdit)

Pressing the F2 (ConvEdit) key brings up a menu of conversational programs for editing.



F8 (Program) → F2 (ConvEdit) → F1 (Select)

Pressing the F1 (Select) key selects the highlighted program for editing and brings up the screen below.



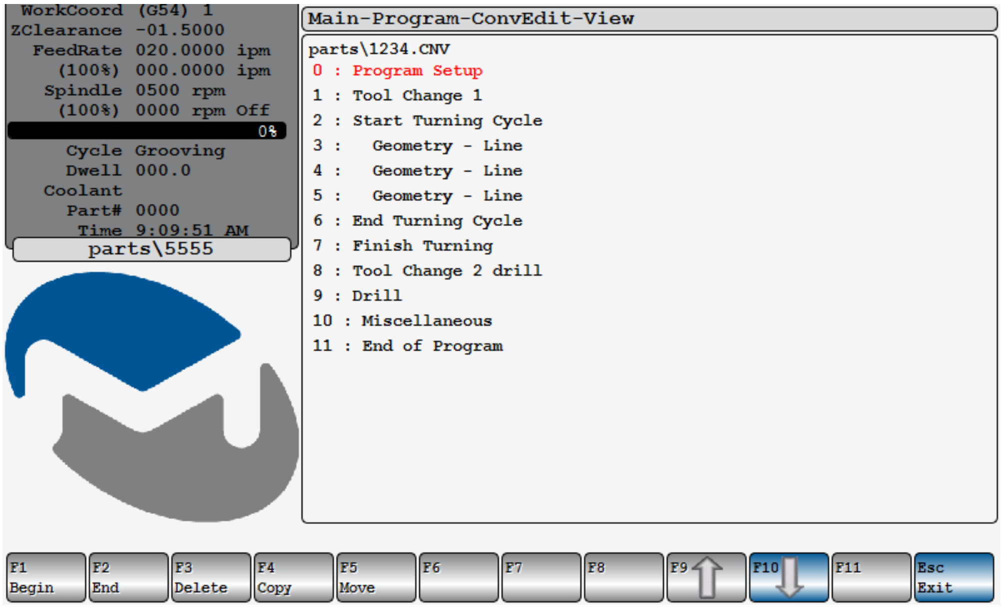
The set of function keys shown above allow movement from event to event. It is possible to step through the program, edit events, and insert or delete events.

F8 (Program) → F2 (ConvEdit) → F1 (Edit)

Pressing the F1 (Edit) key positions the cursor at the first field of the current event. The store/input keys will appear and the event may be edited.

F8 (Program) → F2 (ConvEdit) → F2 (View)

Allows viewing of the entire program and lets the operator position to any of the events in the program. A window similar to the following will be displayed.



F9 (Up Arrow) and F10 (Down Arrow) are used to move from event to event. Pressing Enter displays the event. Events may also be moved in blocks.

F8 (Program) → F2 (ConvEdit) → F2 (View) → F1 (Begin)

Marks the block beginning

F8 (Program) → F2 (ConvEdit) → F2 (View) → F2 (End)

Marks the block end

F8 (Program) → F2 (ConvEdit) → F2 (View) → F3 (Delete)

Deletes the block of events (or the current event, if no blocks are highlighted).

F8 (Program) → F2 (ConvEdit) → F2 (View) → F4 (Copy)

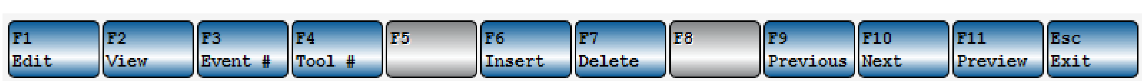
Copies the block of events to the cursor position

F8 (Program) → F2 (ConvEdit) → F2 (View) → F5 (Move)

Moves the block of events to the cursor position

Note: F4 (Copy), and F5 (Move) do not become active until a block of events has been selected. A single event may constitute a block. A single event is selected as a block if F1 (Begin) then F2 (End) are pressed while the block is highlighted.

The program setup screen and end of program event may not be moved, deleted or copied.



F8 (Program) → F2 (ConvEdit) → F3 (Event #)

Allows entry of an event number for which to search. If the event number is not found, the end of program screen will be displayed.

F8 (Program) → F2 (ConvEdit) → F4 (Tool #)

Allows entry to search for a tool number. If tool number is not found the event displayed will not change.

F8 (Program) → F2 (ConvEdit) → F6 (Insert)

INS is used to insert events in a program. The new event(s) will be inserted before the event that is currently displayed. Inserting will continue until the F10 (Exit) soft key is pressed.

F8 (Program) → F2 (ConvEdit) → F7 (Delete)

Will delete the event currently being displayed.

F8 (Program) → F2 (ConvEdit) → F9 (Previous)

Displays the previous event in the program file.

F8 (Program) → F2 (ConvEdit) → F10 (Next)

Displays the next event in the program file.

F8 (Program) → F2 (ConvEdit) →F11 (Preview)

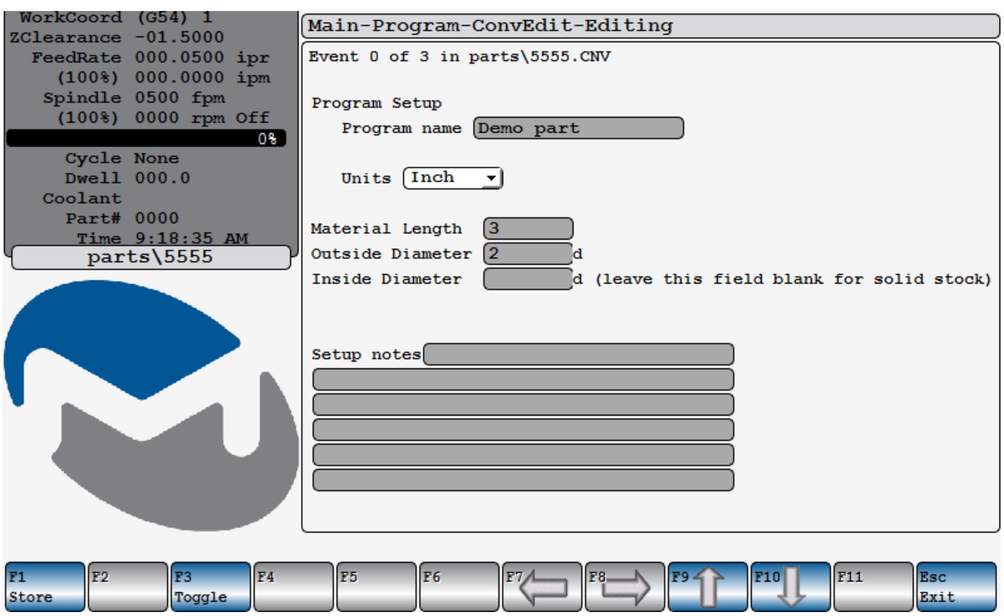
For verifying the program being edited.

(Preview-within-Edit is not available if another program is currently running or verifying.) When F11 (Preview) is pressed, the program is Fast verified without cutter comp in wireframe or solid model mode, depending which mode is currently selected. Graphics are auto-scaled. They can be rotated, scaled, zoomed, etc. Pressing ESC returns to the conversational editor.

Notes: When the program is being previewed, it will ignore M0s, M1s, INPUT statements, etc. Preview-within-Edit is designed to make a quick sketch of the program toolpath. A full-featured Verify (similar to Run), if required, may still be performed on the part program at any time by exiting the editor, returning to the main screen, and doing a Verify from there.

F8 (Program) → F2 (ConvEdit) → Esc (Exit)

Exits the conversational system and automatically creates the executable text program.



The screen shot above is an example of the soft keys that appear after the key sequence F8 (Program) → F2 (ConvEdit) → F1 (Select) → F1 (Edit). The screen and soft keys are no different in the edit mode than they are when the event is first created. <Return> advances the highlight from field to field. Some fields require a numeric value. When a field offers a choice between a finite set of possibilities, the F3 (Toggle) button appears.

Pressing F3 (Toggle) scrolls through the possibilities. Sometimes another field appears as the result of a toggle choice.

To leave a field unchanged, <Enter> or F10 (Down Arrow) past it. With successive <Enter> key presses, the highlight reaches the end of the event and wraps around to the top. Alternately, the F9 (Up Arrow) key may be used to move the highlight up.

F8 (Program) → F2 (ConvEdit) → F1 (Edit) → F1 (Store)

Accepts the entries and adds to the program file. If all required data has not been entered, the F1 (Store) key is grayed out and the field requiring input is red. Each screen stored is called an event.

F8 (Program) → F2 (ConvEdit) → F1 (Edit) → F3 (Toggle)

Pressing this key results in the next toggle value being displayed in the field. The F3 (Toggle) button is entirely absent if the current field is not a toggle field.

F8 (Program) → F2 (ConvEdit) → F1 (Edit) → F5 (Clear)

Used to clear a data field. Absent if the current field is a toggle field.

F8 (Program) → F2 (ConvEdit) → F1 (Edit) → F9 (Up Arrow)

Moves cursor to the previous field.

F8 (Program) → F2 (ConvEdit) → F1 (Edit) → F10 (Down Arrow)

Moves cursor to the next field.

F8 (Program) → F2 (ConvEdit) → F1 (Edit) → F7 (Left Arrow)

Moves cursor to the left. Has no effect in a toggle field.

F8 (Program) → F2 (ConvEdit) → F1 (Edit) → F8 (Right Arrow)

Moves cursor to the right. Has no effect in a toggle field.

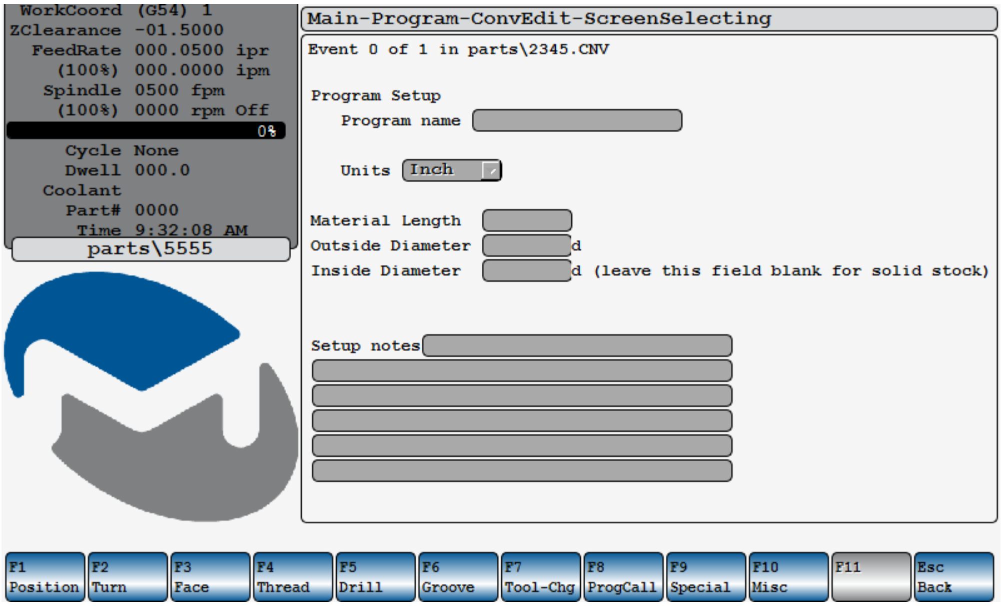
F8 (Program) → F2 (ConvEdit) → F1 (Edit) → Esc (Exit)

Aborts event input. Returns to the menu keys.

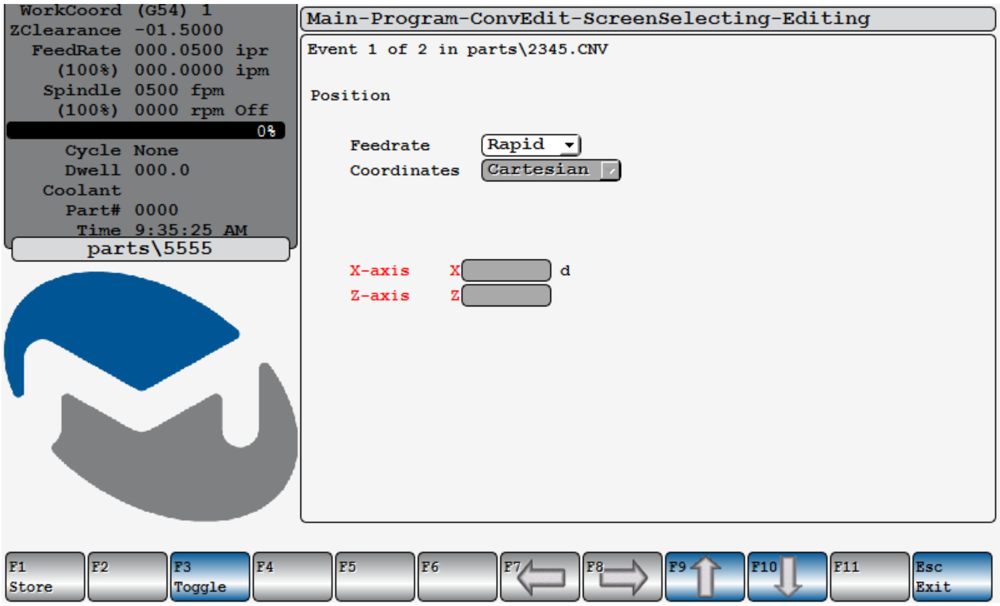
F8 (Program) → F2 (ConvEdit) → F2 (New)

Pressing the F2 (New) key allows entry of a name for a new conversational program. After a name is entered, a new conversational program setup page pops into the main window. Every conversational program begins with a setup page. There is plenty of information that can be entered on the program setup page, but there are no required fields. The setup page can be F1 (Store)'d without entering anything.

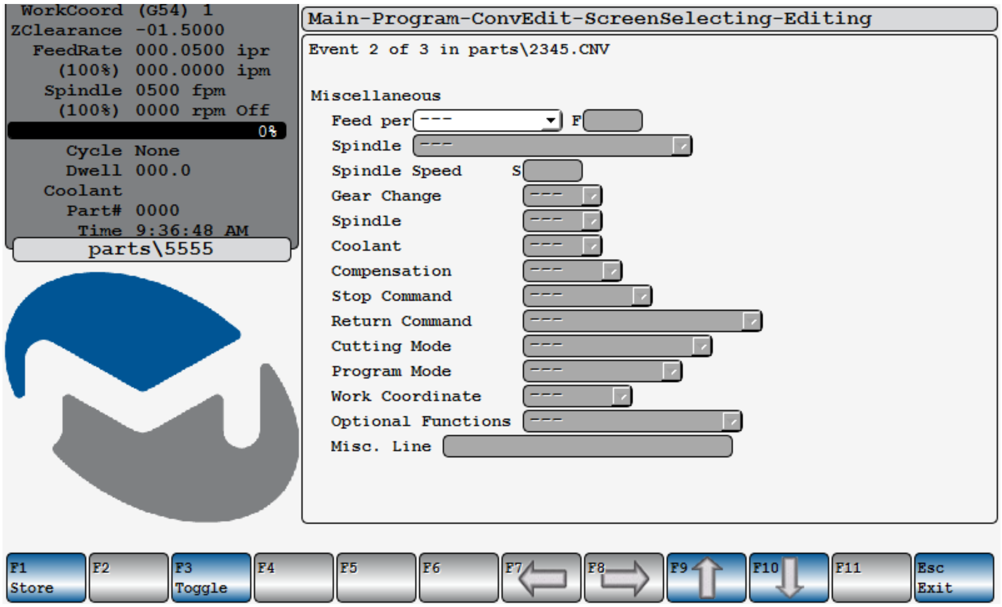
After hitting F1 (Store) to store the program setup page, a main conversational menu appears, listing top level choices for the next program event. These same keys become available if, while editing, the F6 (Insert) key is pressed. The conversational system is offering top level choices for the next program event or event to be inserted.



Pressing a function key will either bring up an input screen (e.g. F1 (Position)) much like the following:



or another screen (e.g. F6 (Misc) like this:



In both of these two examples, the screen that appeared was ready for input, and the expected collection of input/store keys appeared. For some operations, an input screen does not appear until further drilling down into the menus.

F8 (Program) → F3 (Last)

Will open the last program that was edited, If a text program was edited last it will be opened, if a conservational program was edited it will be opened.

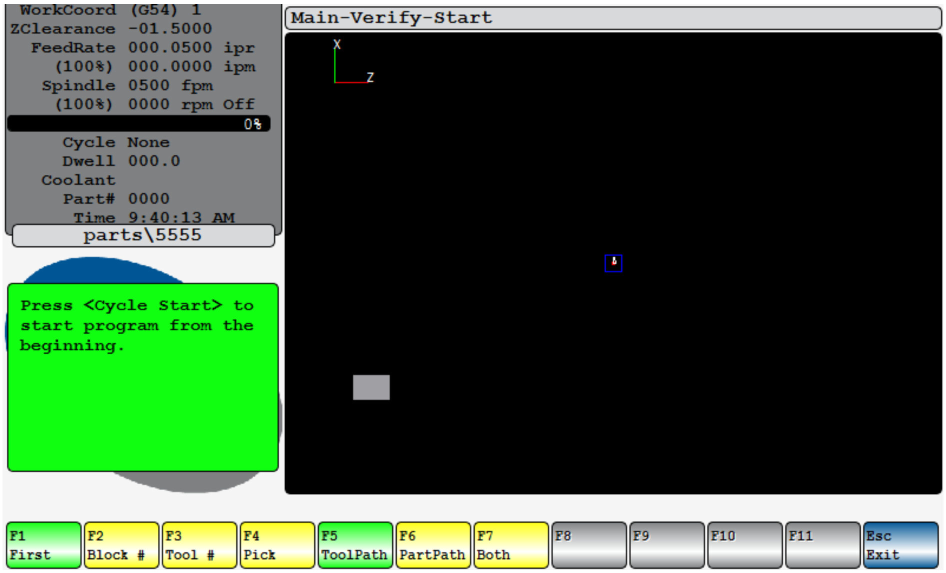
F8 (Program) → F5 (File Util)

Same as shown on page 3-42, F10 (Utilty) → F5 (FileUtil)

F9 (Verify)

The F9 (Verify) function is used to verify part programs.

The run times shown in the upper left hand corner of the display are valid during verify and can be used to estimate machining times. The program that is verified is the active program. To get coordinate information to compare against a print, put the control in block mode and step through the program. The tool will step around the part, and the X Y Z display will read out the coordinate values of each point.



F4 (Verify) F1 (Start) → F1 (First)

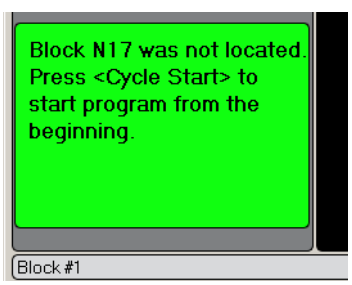
F4 (Verify) F1 (Start) → F2 (Block#)

F4 (Verify) F1 (Start) → F3 (Tool#)

F9 (Verify) F1 (Start) → F4 (Pick) (Option)

F1 (First) is automatically selected when this screen is displayed. To verify the active program from the beginning (from the first block), press Cycle Start. If **F2 (Block #)** is pushed, the control requests the block number to start from. When Cycle Start is pressed, the active program will start verifying from the selected block number. If **F3 (Tool #)** is depressed, the control requests the tool number to start from. When Cycle Start is pressed, the active program starts verifying at the desired tool.

Note: If the block number or tool number requested is not found in the active program, the following window will appear.



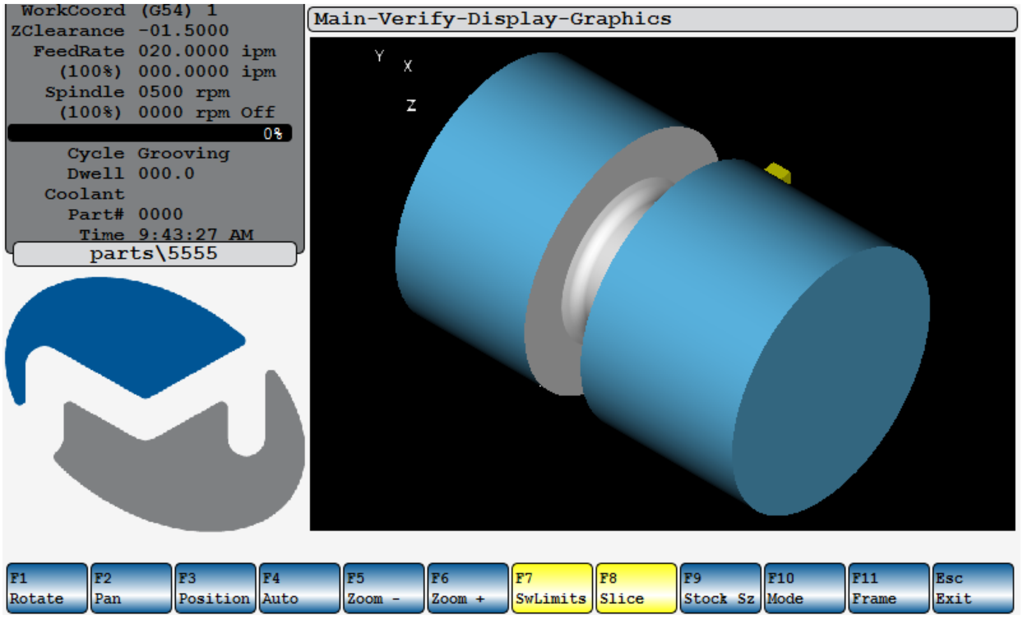
F4 (Pick) option - When F4 (Pick) is selected, a program viewer is displayed on the screen making it possible to select a specific line of the program to do a Modal Restart from. Note that the Misc Parameter "allow modal restart" must be set to "true". See appendix for detailed operation notes for the Modal Restart capability.

F9 (Verify) → F6 (Display)

The F6 (Display) key can be accessed from a number of screens. The following screen is shown as if the F6 (Display) was entered from the F9 (Verify) screen. All the display functions and screens are identical, independent of the entry point. Only the return point differs based on the original entry point. See page 3-13 for a complete list of F6 Display options.

F9 (Verify) → F6 (Display) → F5 (Graphics)

If the F5 (Graphics) key is activated, the following screen appears:



The graphics functions used in the verify mode are the same functions used in the run mode. For a full explanation of these functions, see Page 3-14 on runtime graphics.

F9 (Verify) → F8 (Fast)

For feedrate override positions 100% and greater, F8 (Fast) runs the part approximately 100 times faster than programmed.

When Verify starts running, the following options appear.

F9 (Verify) → F9 (Halt)

F9 (Verify) → F9 (Resume)

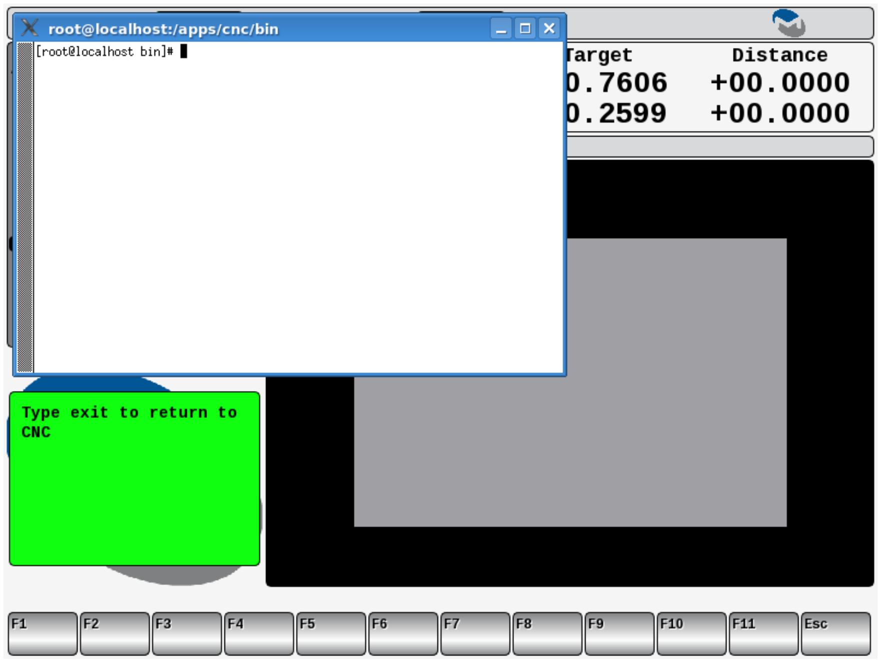
Once a program has been F9 (Halt)ed, the resume feature of the control becomes active. The F9 (Resume) key will now be displayed on the verify screen. A program can be resumed as long as resume is active. If the resume function is selected, the active program will be resumed at the halted point.

Note: *You can switch to run from verifying. This is convenient for long programs as it picks up on all feeds/speeds/tool offsets/cutter comp/etc. Press the F9 (Halt). The F10 key will now show F10 (Run). When the F10 (Run) is pressed, the control will be as if a halt/resume was done while running a program. Do not try this in the middle of automatic tool changes or other I/O related routines.*

F10 (Utility)

When F10 (Utility) is pressed, there are options for F1 (Console), F2 (Calcultr), F5 (FileUtil), F6 (Log), F7 (Panel) F8 (Info), F9 (SysInfo), and Esc (Exit).

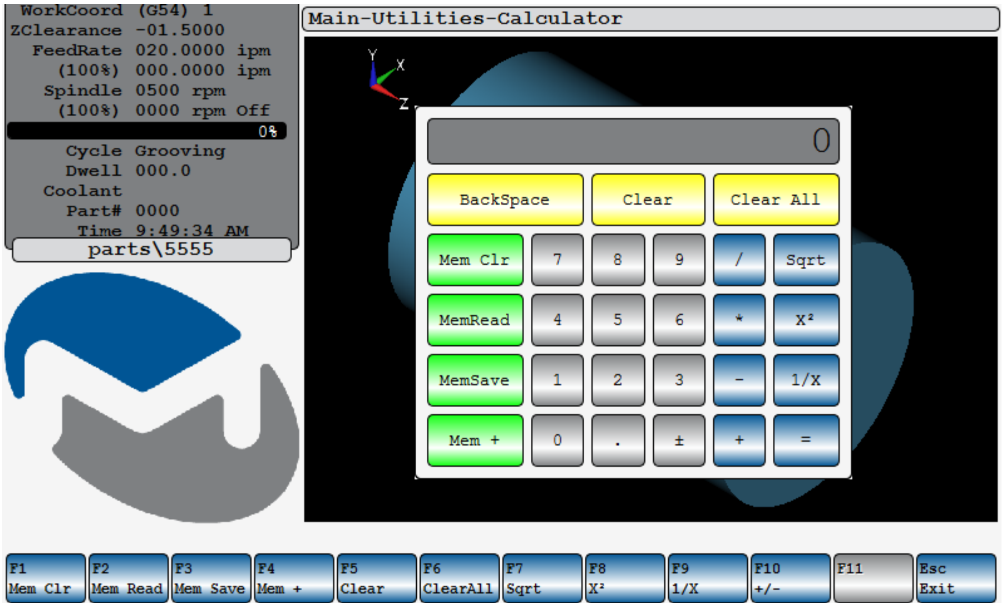
F10 (Utility) → F1 (Console)



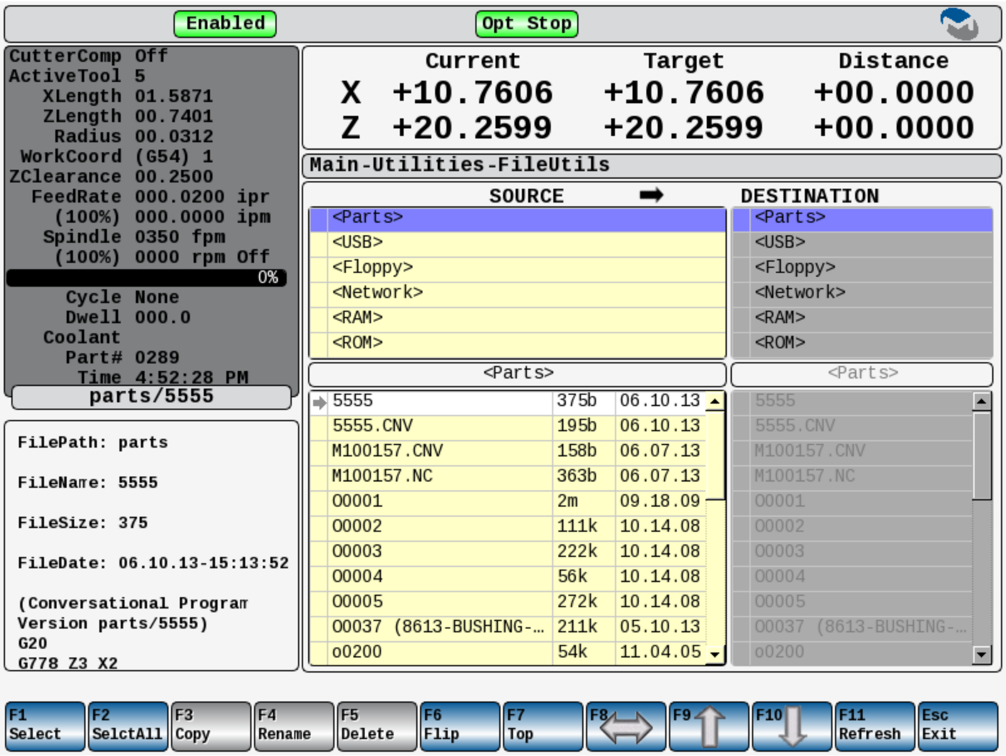
F1 (Console) requires a username and password. It opens a console that allows command lines to be executed. The console is generally used by technicians for setup or trouble shooting. To exit the console type exit.

F10 (Utility) → F2 (Calculatr)

The following screen is displayed:



F10 (Utility) → F5 (FileUtil)



In the CNC8000 files move from left to right. **Always.** The source directory -- where the files come from -- is always on the left and the destination directory -- where the files go -- is always on the right.

To change the source, use the F9 (Up Arrow) and F10 (Down Arrow) keys to move the highlight. Then **press F1 (Select). The selected source directory has a blue background**.

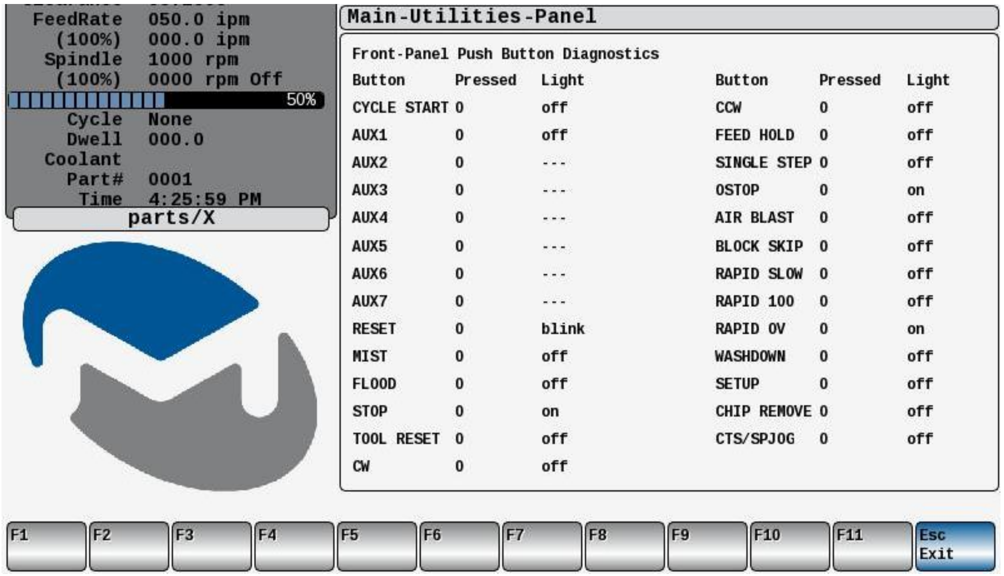
To change the destination, press F8 (Right/Left Arrow) to bring focus to the right hand part of the screen. Use the F9 (Up Arrow) and F10 (Down Arrow) keys to move the highlight. Then **press F1 (Select). The selected destination directory has a blue background.**

To choose a file to move from Source to Destination, press the F8 (Right/Left Arrow) to return focus to the left hand part of the screen. Use the F9 (Up Arrow) and F10 (Down Arrow) keys to move the highlight to the file. Then **press F1 (Select)**. **The selected file has a blue background.**

In the above screenshot, USB is the source directory, Parts is the destination directory, and the highlight is pointing at a file named 5555.cnv. The file 5555.cnv has been selected.

When a file is selected, the F3 (Copy), F4(rename), and F5 (Delete) buttons become active. To copy the file from Source to Destination, press F3 (Copy). Oftentimes, an operator wants to save this tweaked and debugged part program for the next time he runs the part. To accomplish this turnaround, press the F6 (Flip) button, reversing the source and destination directories.

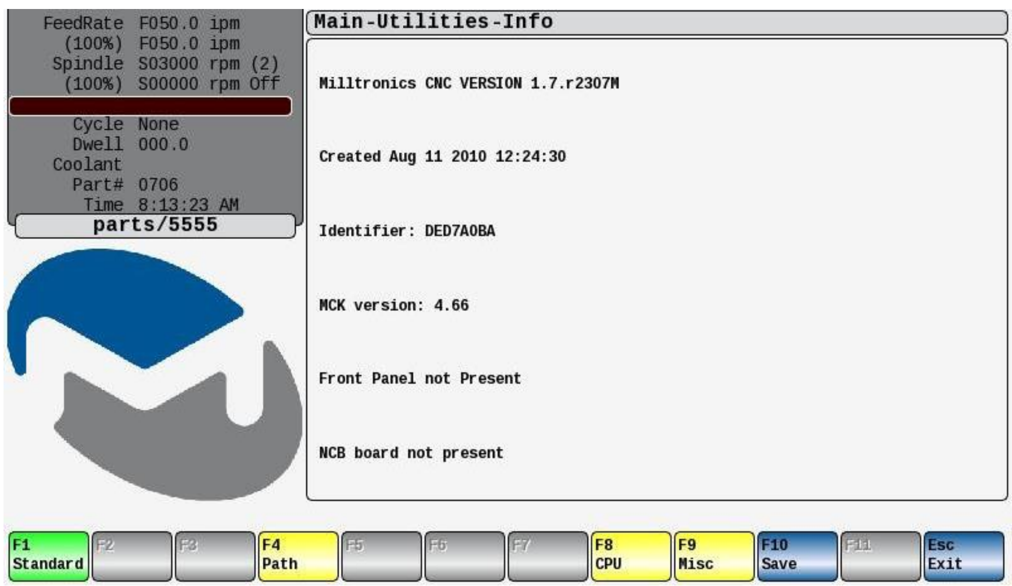
F10 (Utility) → F7 (Panel)



The F7(Panel) is used to check front panel button states. Buttons that are pressed will show a “1” Buttons that are not pressed will show a”0”. Lights will show off, on or blink.

F10 (Utility) → F8 (Info)

Pressing F8 (Info) produces the following screen:



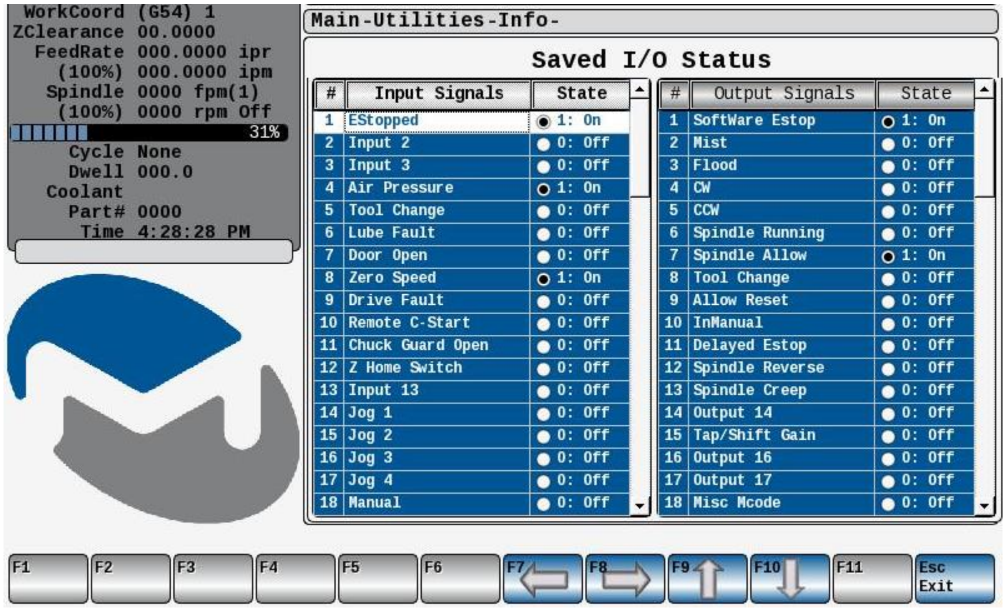
F10 (Utility) → F8 (Info) → F1 (Standard)

The third item on this screen, the unique control identifier, is an 8-character hexadecimal number required for pay timer code generation, true whether the code be a limited-time code or a permanent code. Pressing F9 (Misc) from this screen shows the 8-character hexadecimal servo revision, which is required for generation of a permanent pay timer code.

F10 (Utility) → F8 (Info) → F4 (Path)

Shows the Paths for folders used by the control. Paths include ROM, RAM, PARTS, VIRTUAL, FLOPPY, REMOTE, and USB.

F10 (Utility) → F8 (Info) → F7 (Saved IO)



F7 (Saved IO) displays the I/O states of the machine just prior to e-stopping.
F7 (Saved IO) is used to diagnose the reason for an e-stop.

F10 (Utility) → F8 (Info) → F8 (CPU)

Displays the CPU temperature, the CPU fan speed, and the Voltage.

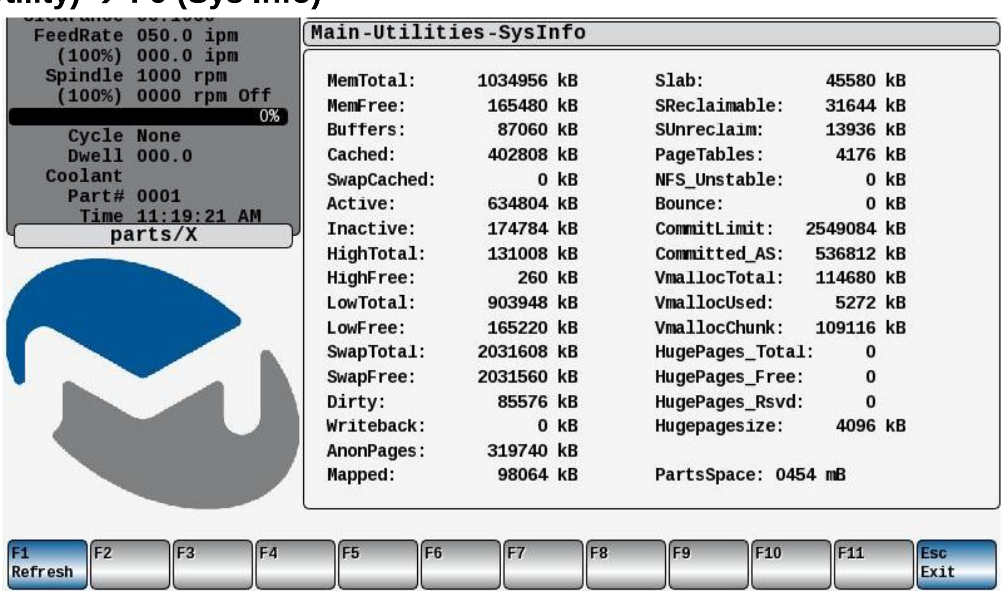
F10 (Utility) → F8 (Info) → F9 (Misc)

Displays information about NCB card, servo revision, and front panel.

F10 (Utility) → F8 (Info) → F10 (Save)

Saves system version to file.

F10 (Utility) → F9 (Sys Info)



F9 (Sys Info) Displays information about the memory and disk space on the motherboard.

4

Section 4 Contents

In This Section

This section describes the tasks that must be completed before running a program on the 8200 CNC to machine a part. The tasks include enabling the machine, homing the machine and loading a program from a USB drive. Also included are instructions for importing a CAD file and converting a 7200 conversational program to an 8200 conversational program.

Enabling the Machine

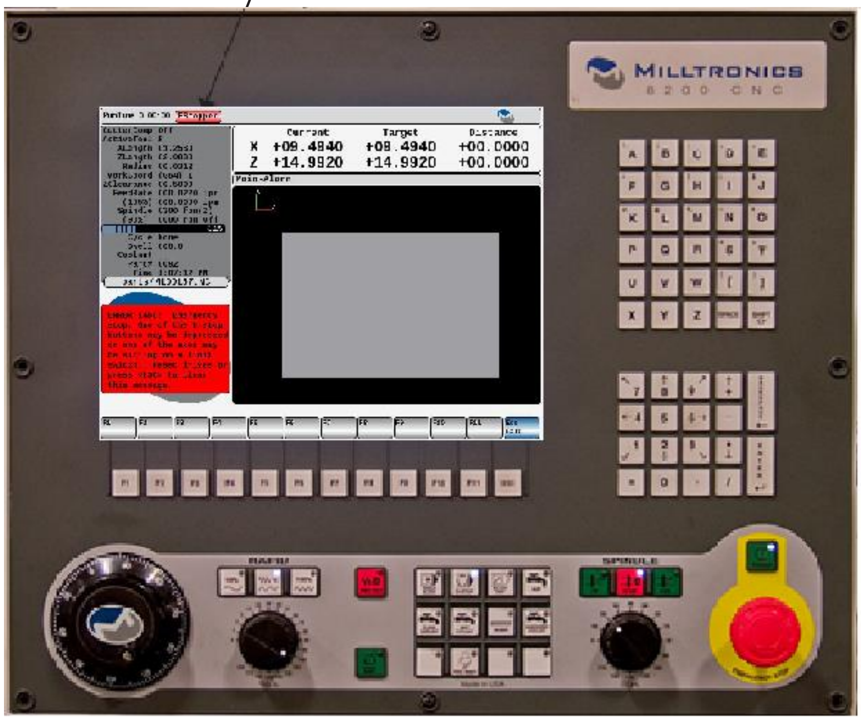
Make sure the red E-STOP mushroom switch on the front panel is pushed in. Rotate the main power switch on the back of the machine to ON. The control accepts defaults at power up automatically. Ignore the various system messages that scroll by. Wait for the main CNC screen.

When the machine is first powered up, it is in the disabled state. The machine's servo motors are not active. The machine state button is red and shows the word "EStopped" (see top figure on next page).

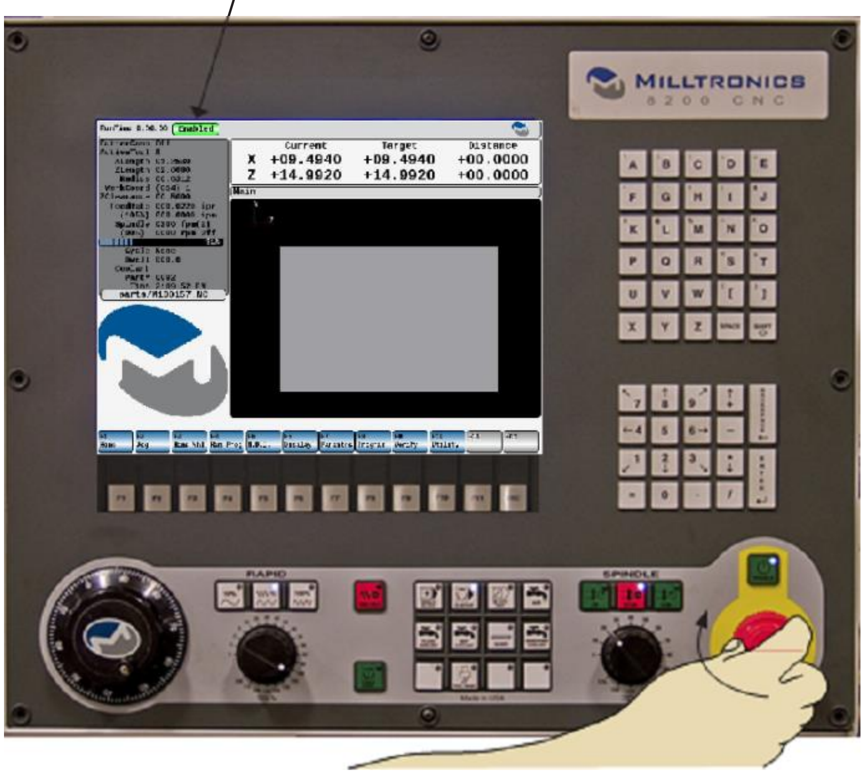
Twist the **EMERGENCY STOP** button clockwise until it releases. Press the **ENABLE** button located above the **EMERGENCY STOP** button.

The machine is now enabled and the machine state button turns green and shows the word "Enabled." (see bottom figure next page).

Machine state button is red



Machine state button is green



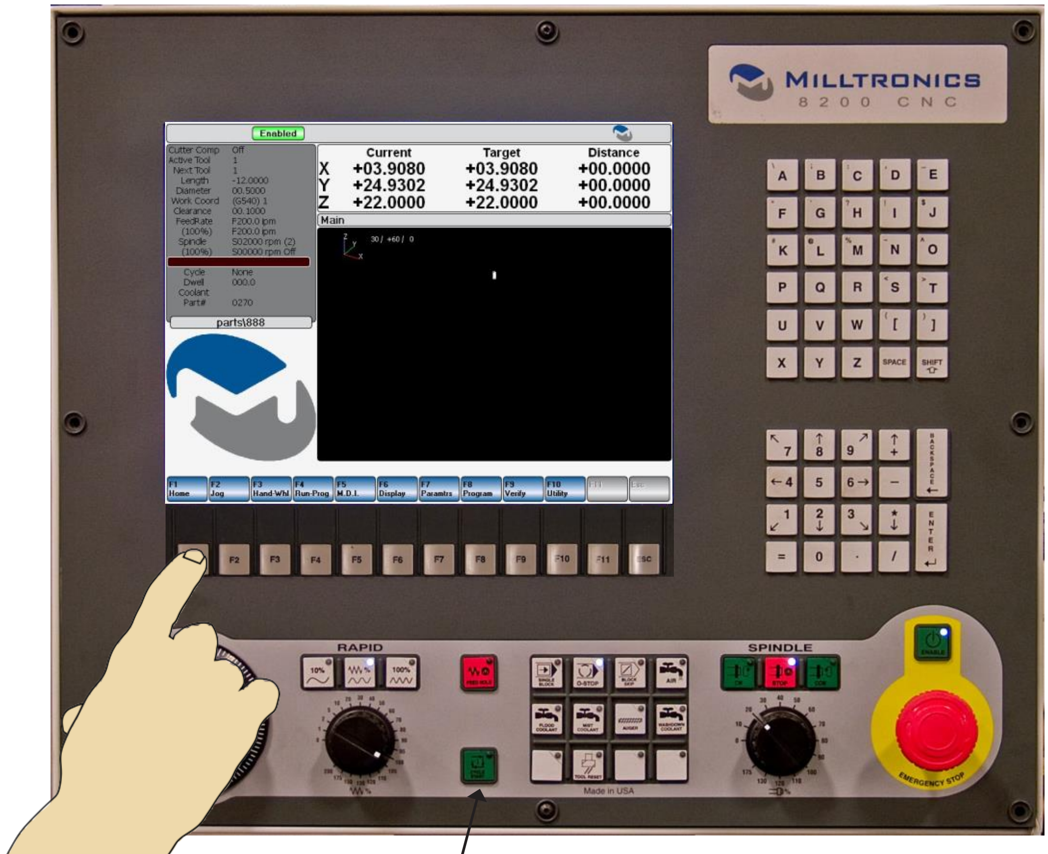
Pay Code

If the machine is an upgrade from a CNC7000, it will require a new pay code. The CNC8000 puts up messages saying operation is disabled and a new pay code is required. The machine control buttons flash continuously and they do not work. All of this behavior is normal. Contact Milltronics and let them know that you need a pay code.

Homing the Machine

After the machine is enabled and the servo motors are active, the next step is to establish "home" and position the machine to machine zero. To accomplish this, press the Home button (F1) and then, when prompted, press **CYCLE START**. Wait until the machine completes its homing cycle (See figure below).

During the homing process, the User Info window indicates progress of the homing of each axis.

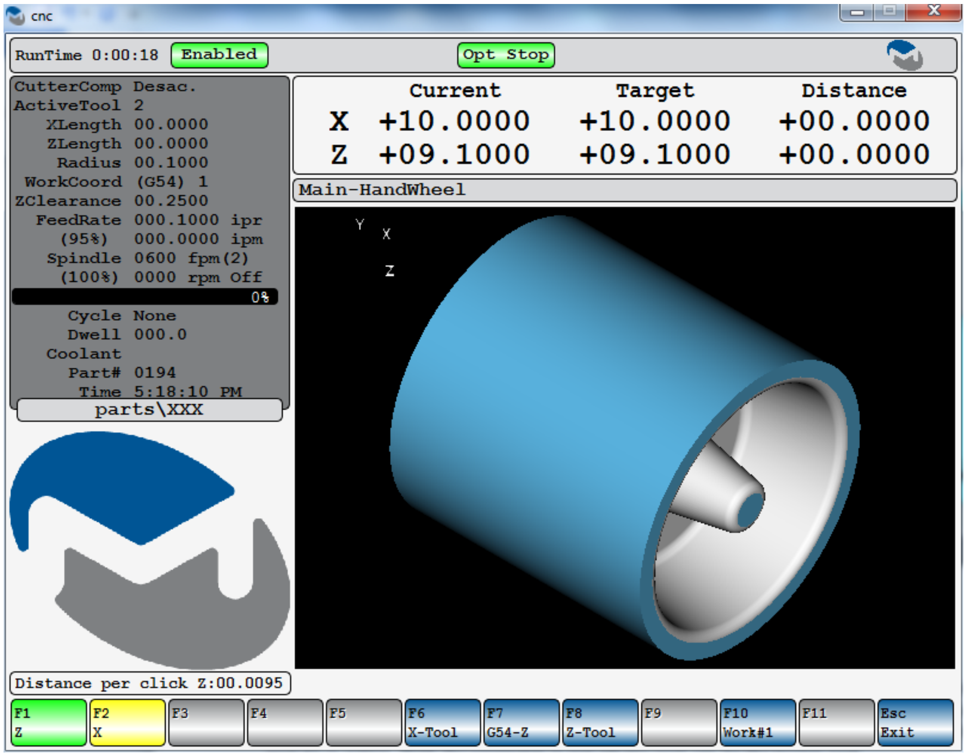


Cycle Start button

User Info window shows progress of homing process

Note: Axis parameters allow setting of home velocities, directions, sequence, etc.

Setting Tool Length and Work Offsets



The following procedure assumes that G54 (Work Offsets #1) offsets are being used. It also assumes that the face of the material is considered Z0. When Misc parameter “G5#-Z on tool #1 only” is set, it designates tool #1 as the master tool and the F7(G54-Z) key is only active when tool #1 is active. With the parameter is not set any tool can be the master tool.

1. Make sure all G50 and G52 offsets are zero. They are zeroed on power up or can be zeroed by using F7 (Paramtrs)-F2 (Coords) and editing the G50 and G52 offsets.
2. Make sure G54 X offset is zero. It can be zeroed by using F7 (Paramtrs)-F2 (Coords) and editing the G54 X offset.
3. Make tool #1 active. Do an F5 (MDI) T0101.
4. Using the handwheel, touch the tip of the tool to the face of the material. Then press F7 (G54-Z). Type in the Z axis position (normally zero).
5. Using the handwheel, make a small turning cut on the diameter of the material. Then press F6 (X-Tool) then enter the diameter of your small cut for the position.

   (For drills, no X tool length should be needed. Commanding a tool to X0 should position the drill to the center line of the spindle)
6. Make the next tool active. Do an MDI T#### M6. (T0202 for tool #2, T0303 for tool #3 . . . etc.)
7. Using the handwheel, touch the tip of the tool to face of the material. Press F8 (Z-Tool) then enter the Z axis position for the tool position (normally zero).
8. Using the handwheel, make a small turning cut on the diameter of the material. Press F6 (X-Tool). Then enter the diameter of the small cut for the position.

(For drills, no X tool length should be needed. Commanding a tool to X0 should position the drill to the center line of the spindle)

9. Go to step #6 (for the next tool).
10. Enter the tool types, tool nose radii, etc. in the tool table by pressing F7 (Paramtrs) -F3 (Tools) and then editing the tool table. F1 (ToolType) toggles the tool type.

When the repositioning the face of the material, you can handwheel tool #1 to the face of the parts and press F7 (G54-Z) and type in the Z axis position (normally zero).

Note 1: The tool type relates to the imaginary tool tip and how the tool offsets are set.

Note 2: If the front turret parameter is set, the graphics of the tool in the tool table reflect the fact that the tool is coming from the front. It does not affect the actual tool type number.

Note 3: When setting the tool length for X or Z axis, the corresponding wear offset is zeroed.

To check the work and tool offset, select F5 (MDI).
For each tool: Type T0101, T0202 . . etc.)
G0 Z.1 (or safe distance)
X0
F5 (Run) [CYCLE START]

Caution: The machine will move at a rapid feedrate. Each tool should position itself to the center of the spindle and 0.1000 or safe distance from the face of the part.

File Names

A. Legal part program file names

Part program file names are not restricted to a particular format. All of the following are legal file names:

XYZ.123
ABC.NC
O0056
O1234.NC
PROGRAM 123.CNC PRG (spaces are allowed)

All of the example program names are upper case because alphabetic input from the front panel is upper case by default. The control is case sensitive about filenames, however, so beware that.

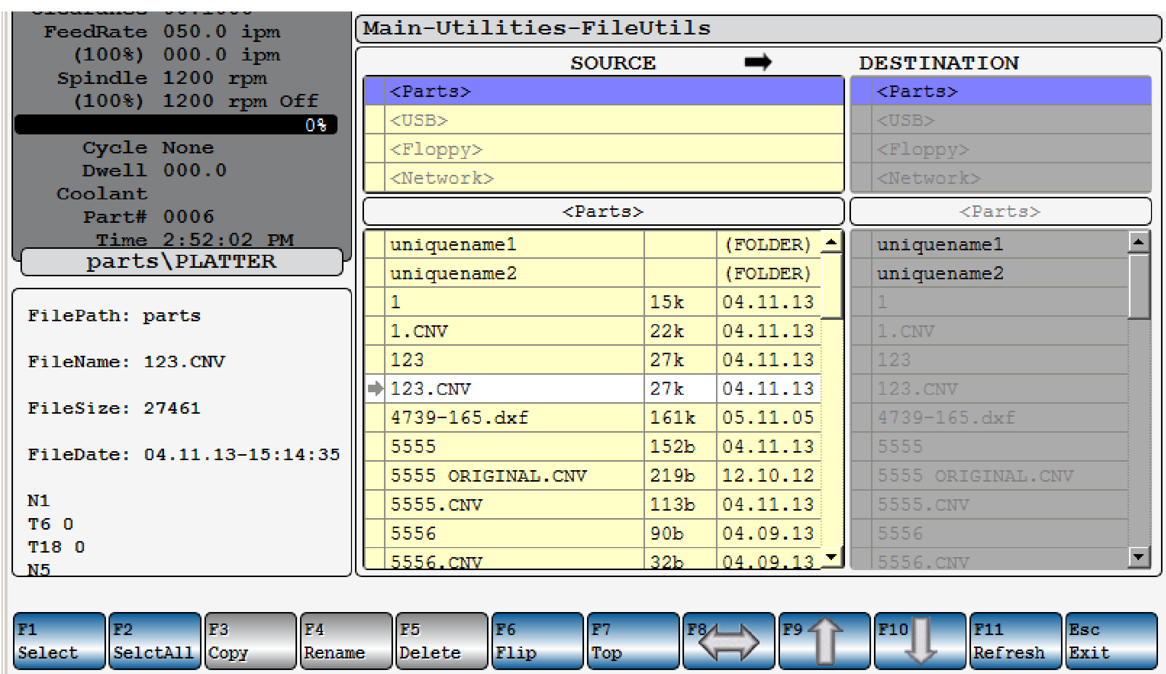
PROG.NC
Prog.nc
prog.nc

Are all unique filenames.

Files with a .CNV extension are considered conversational files. They can be opened as text files, but cannot be used in Run or Verify modes.

B. File Info box

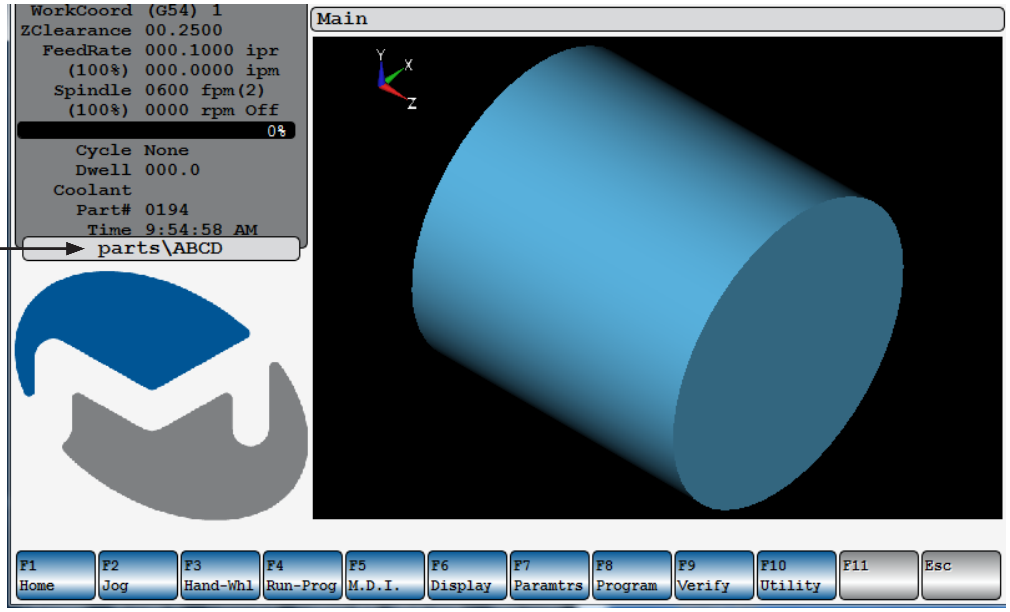
When files are listed in the graphics window, as in Program → Text Edit or Utility → File Utility, detailed information about the file appears in the information window underneath the status window. The file path, file date, and length all are listed in the information window, along with the first three lines of the file.



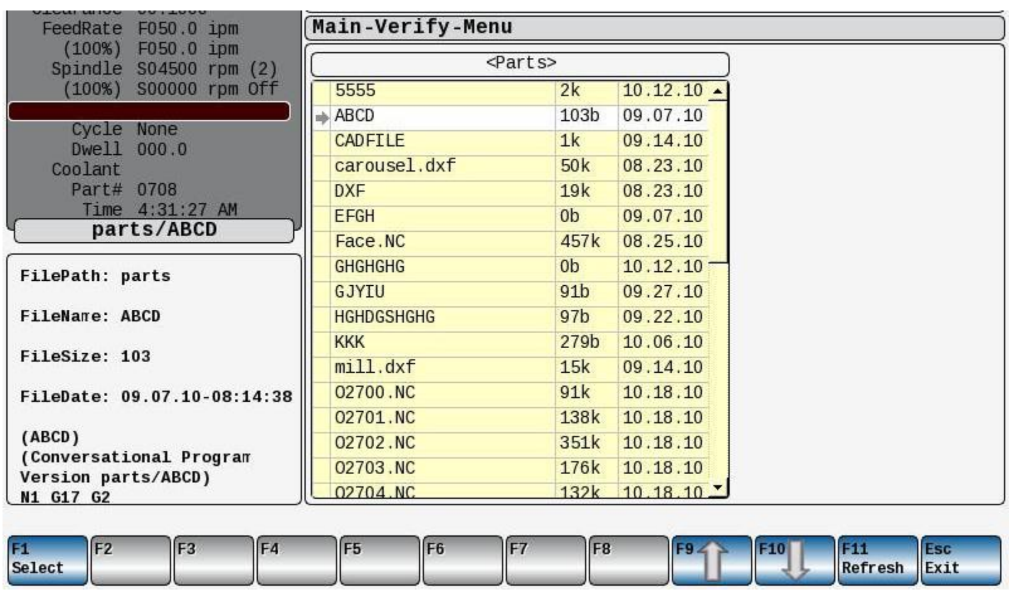
Making a Program Active

In order to run a particular program stored in the Parts directory, you must first make that program active. In the example below, let us assume that you are starting at the screen below and that you wish to run a program called 5555.

Active program



The current active program is “ABCD.” First, press the F9 (Verify) key and on the next screen press the F7 (Menu) key. This action brings up a screen showing all the files.
Note: You could also start with the F4 (Run-Prog) key and repeat the same procedure.



Move the cursor to the file you want (5555) and press the F1 (Select) key. This will make 5555 the active program and it will show up in the active program window.

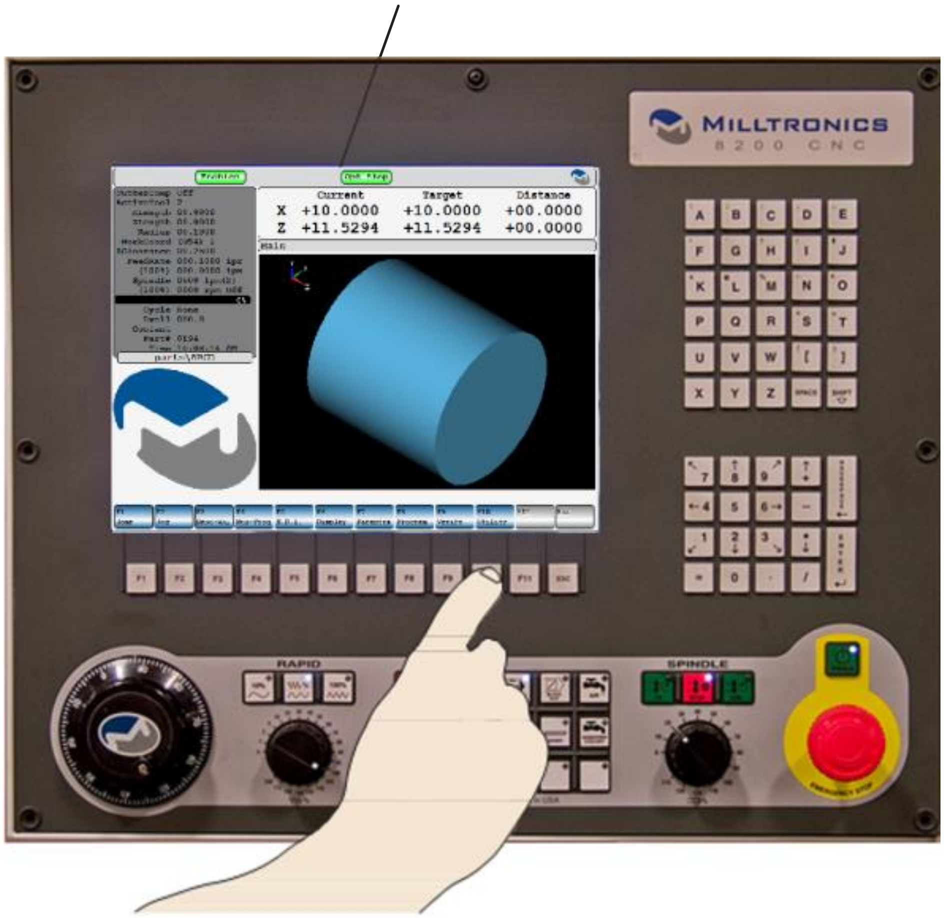
Active program

Loading a Parts Program from a USB Drive

For illustrative purposes, let us suppose that the conversational program you wish to load into the controller is on a USB drive and that you have installed the USB drive in the controller.

With the MAIN SCREEN visible, The first step in loading a part program is to press the UTILITY function key (F10) to bring up the UTILITY screen. (see top figure on next page)

Main Screen

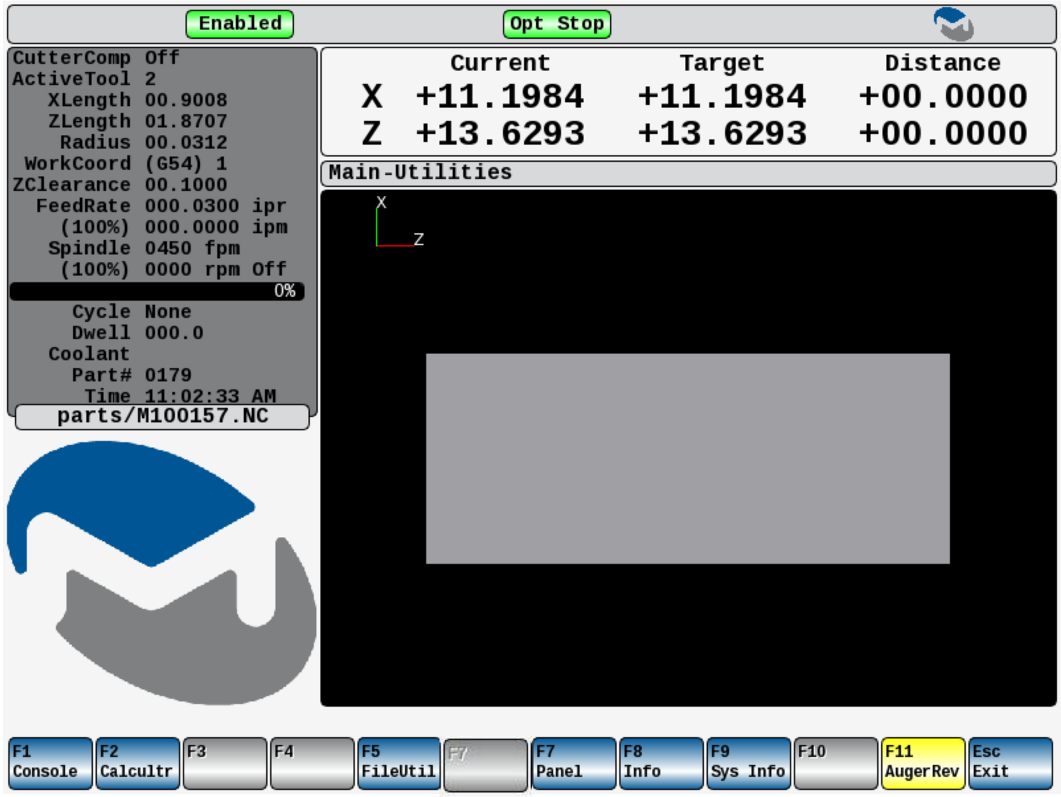


Next, with the UTILITIES MENU visible, press the FILE UTILITIES function key (F5) to bring up the FILE UTILITIES screen. (see bottom figure on next page)

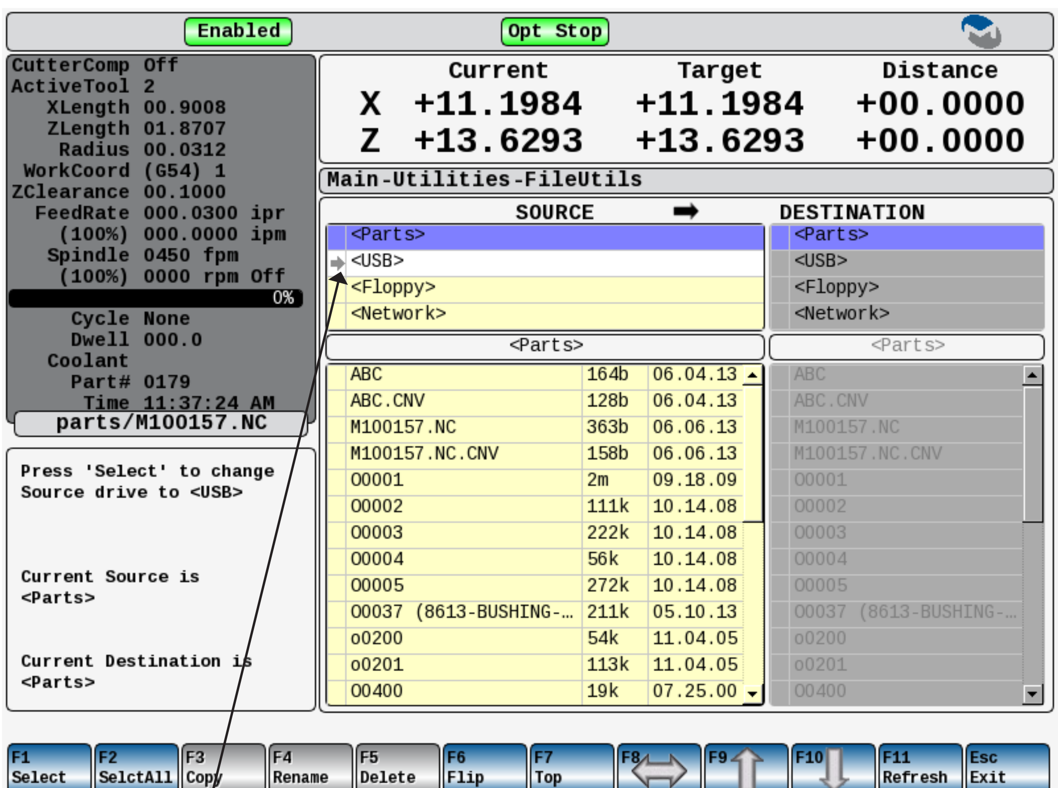
With the FILE UTILITIES SCREEN visible, note that the Main Graphics Window is divided into two halves, SOURCE and DESTINATION. SOURCE is always on the left and DESTINATION is always on the right. Each of these two halves shows the type of device at the top and a list of files on the device on the bottom. The active half is bright and the passive half is dark. The F8 function key can be used to make either SOURCE or DESTINATION the active half.

With the active half the SOURCE, Use the direction arrow keys (F9 and F10) to move the cursor to USB and press the SELECT function key (F1). Notice the prompt in the User Info Window.

UTILITIES MENU



FILE UTILITIES SCREEN



Cursor

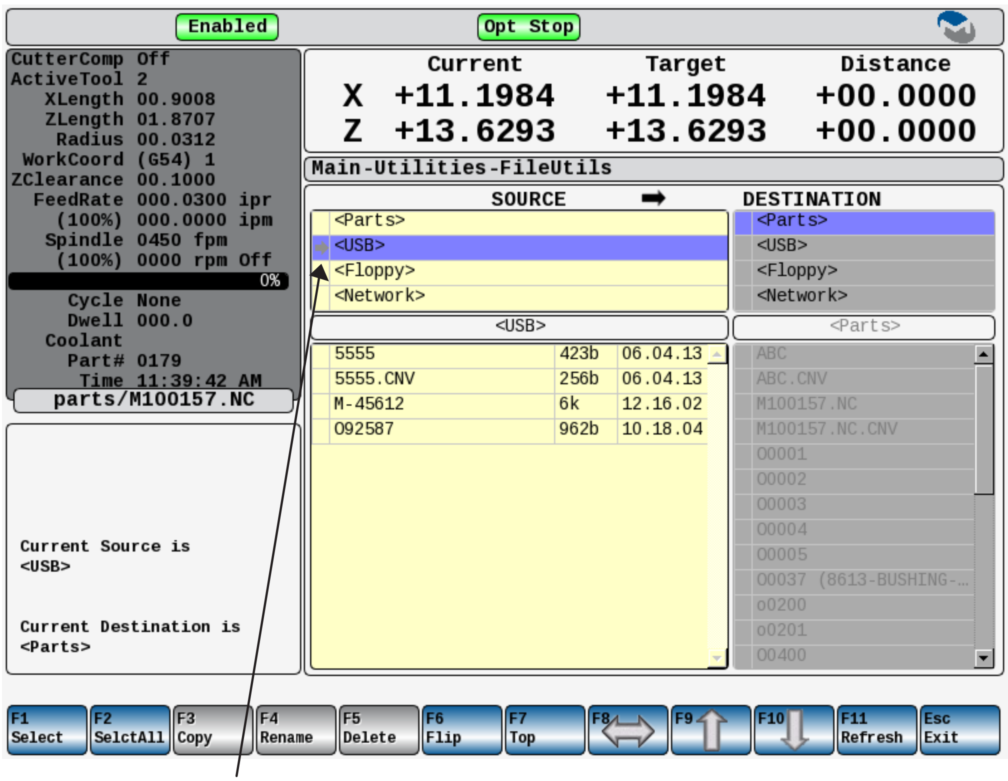
With USB selected as the SOURCE, a list of the files on the USB drive appears at the bottom of the SOURCE part of the window.

Make sure that the DESTINATION half of the window has PARTS selected.

Move the cursor to select the file you want to load (in this example 5555.CNV) and press the SELECT function key.

With the 5555.CNV file selected, press the COPY function key (F3) to copy the file to the Parts DESTINATION.

FILE UTILITIES SCREEN



Cursor

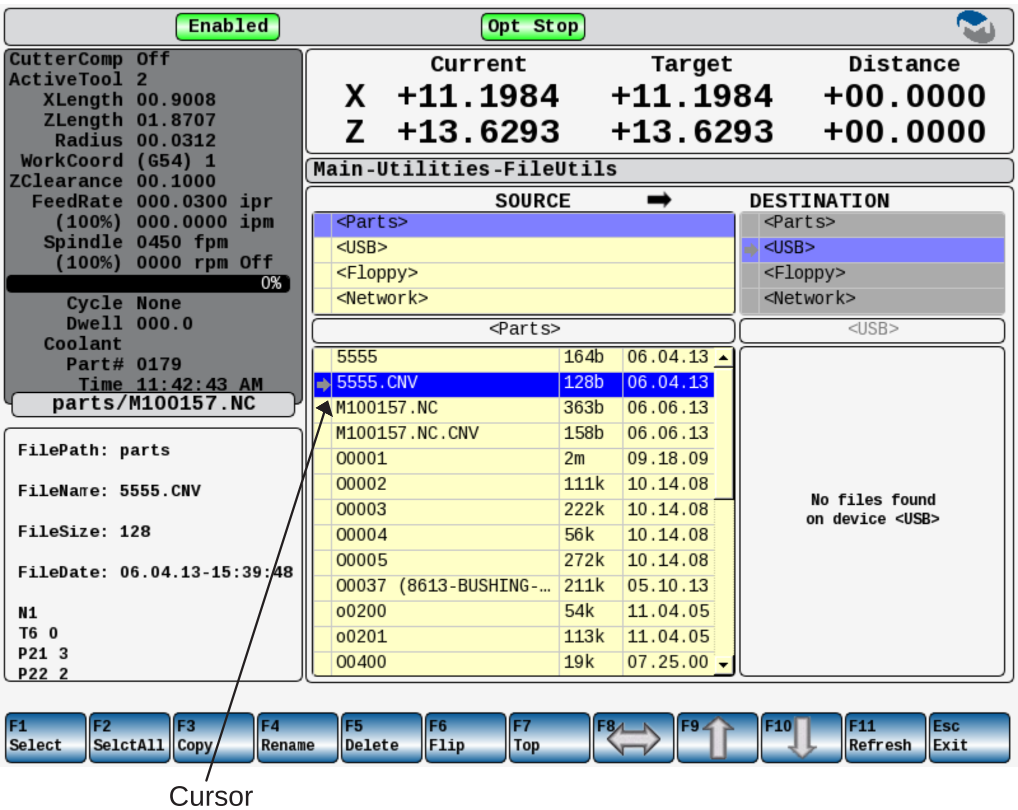
The 5555.CNV file is now in the 8200 controller ready for use. The file will now appear under the list of files in the <PARTS> directory.

Saving a Parts Program to a USB Drive

The procedure for saving a part program in the 8200 controller onto a USB drive is analogous to the previous procedure.

First, mount the USB drive in the controller and get to the FILE UTILITIES SCREEN. Make sure that Parts is selected for the SOURCE, USB is selected for the DESTINATION. Then move the cursor to the file (in this case, 5555.cnv) to be copied to the USB drive.

FILE UTILITIES SCREEN



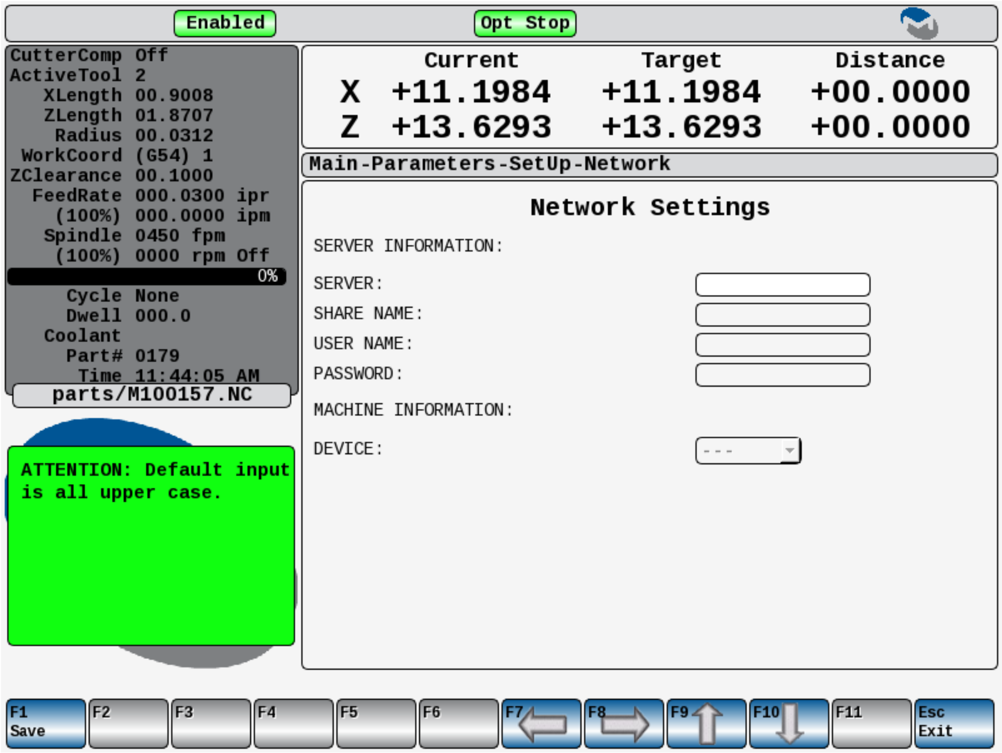
Press the F1 (Select) function key to select the file and press the F3 (Copy) function key to write the file to the USB drive.

The file 5555.cnv is now on the USB drive, ready to be carried somewhere else.

When a file is selected, the F3 (Copy), F4 (Rename), and F5 (Delete) buttons become active. To copy the file from Source to Destination, **press F3 (Copy)**.

Oftentimes, an operator wants to save his tweaked and debugged part program for the next time he runs the part. To accomplish this turnaround on the CNC8000, press the F6 (Flip) button, reversing the source and destination directories.

Network Setup Page



SERVER The IP address of the network server, for example “10.100.40.99” . This information is available from the server itself. On the server, from the Start menu, choose Run. Type “cmd” and <Enter>, which produces a DOS-style command prompt. At the prompt type “ipconfig” and <Enter>. The resulting list includes the IP address of the server.

SHARE NAME Name of the directory on the server where part programs are stored, designated for sharing with network clients (the CNC8000 is a network client). For example “VMPARTPROGRAMS” . If the machine is an upgrade from CNC7000, the share name is often available from the CNC7000 side in c:\autoexec.bat. Open autoexec. bat with a text editor to look at it. In the line that starts “net use ...” the last directory name is the share name.

USER NAME User name of any authorized network user. For example “JVENTURA” . Even if the shop network does not have user names and passwords, the CNC8000 network setup interface still requires a user name. Make something up.

PASSWORD User’s password. For example “TheBody123” . Be careful to enter upper case and lower case letters correctly in passwords because passwords are often case sensitive. By default, the CNC sees letters entered at the front panel as upper case. To get a lower case letter, hold down <Shift> then <Space> then press the letter. Release <Shift> and <Space>, then repeat the sequence for the next lower case letter. Using an external keyboard, to get lower case letters, hold down <Shift> then press the letter. If a shop network does not have user names and passwords, no password entry is required. This field can be left blank.

******** MACHINE INFORMATION ********

DEVICE Toggle choices are “eth0”, “eth1”, and “--”. Try “eth0” .

TYPE Toggle choices are “Static”, “Dynamic”, and “None” . Choosing Dynamic is sufficient unto itself because the machine requests a temporary IP address from the network DHCP server and no further information is needed. Assigning a static IP to the machine is the method used by many shops and requires further information.

MACHINE IP The static IP assigned to the network port. For example, “10.110.100.218” . If the machine is an upgrade from CNC7000, match the setting from the CNC7000 side by looking in the c:\net\protocol.ini file in the line “IPAddressx= ...”
The IP address of the network port may be verified on the CNC8000 side at the Linux console. At the console prompt do “ip addr” and <Enter> to see data associated with each network port.

GATEWAY Ask the network administrator. An example would be “10.110.10.1” . The default gateway is also available from the shop network server. On the server, from the Start menu, choose Run. Type “cmd” and <Enter>, which produces a DOS-style command prompt. At the prompt type “ipconfig” and <Enter>. The resulting list includes the default gateway.

SUBNET MASK Ask the network administrator. An example would be “255.255.0.0”. The subnet mask is also available from the shop network server. On the server, from the Start menu, choose Run. Type “cmd” and <Enter>, which produces a DOS-style command prompt. At the prompt type “ipconfig” and <Enter>. The resulting list includes the subnet mask.

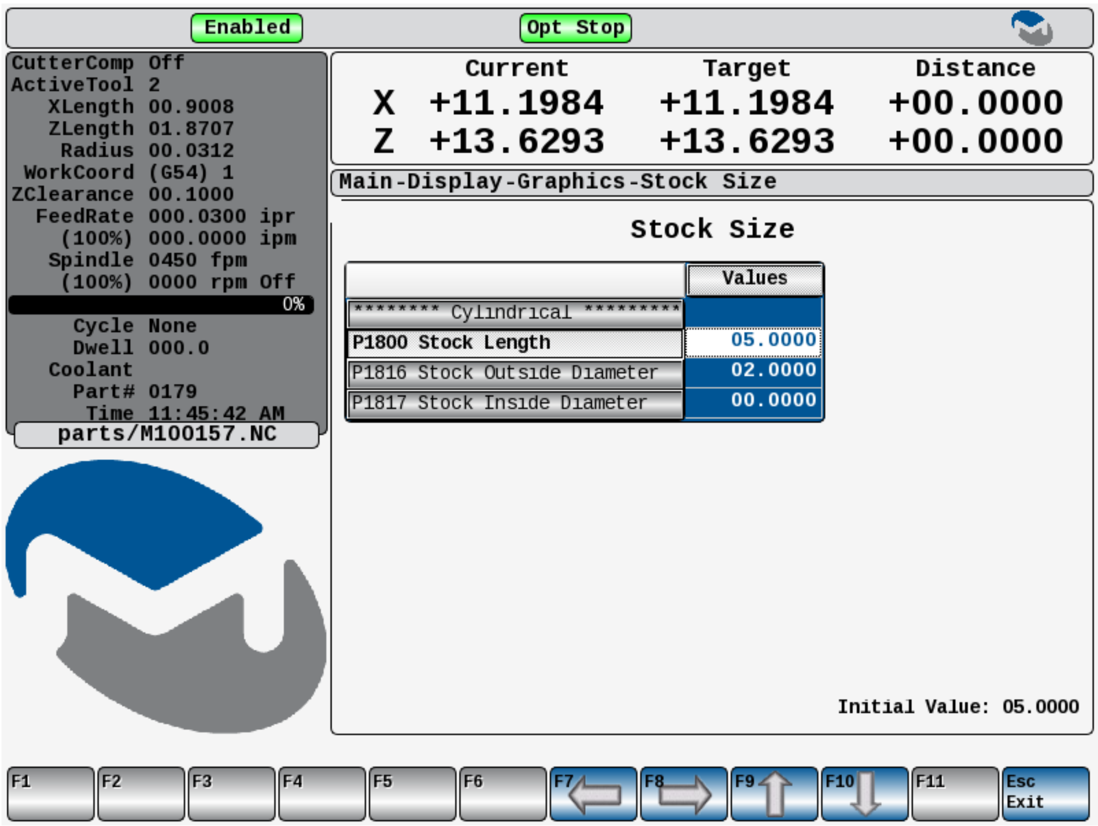
After filling in fields in the network setup page, **press F1 (Save)** to save them. Escape back up to the main screen. Cycle power to allow the CNC8000 to see the network settings.

Solid Model Stock Size (Solid Graphics Mode Only)

Default Stock Size on Stock Size Screen

If a program (whether conversational or text) does not specify stock size, the CNC8000 defaults to the solid model stock size set on the Stock Size screen. The Stock Size screen is available when graphics are in solid model mode.

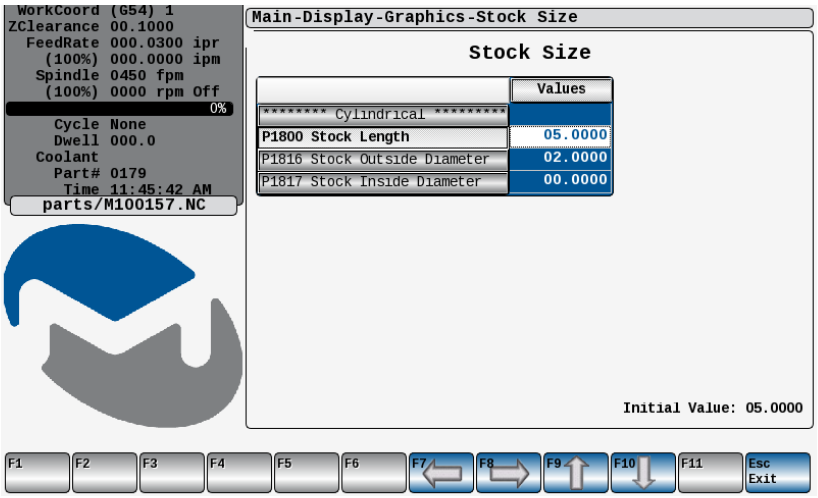
Press F6 (Display) → F5 (Graphics) → F9 (Stock Sz). See picture below.



1. Fill in the appropriate data for the stock.
2. Press Esc

Stock size in a conversational program

The setup page (Event 0) of a conversational program contains three fields for entering the solid model stock size. See picture below



Stock Size in a Text Program

Near the top of the program, insert text as follows:

For Cylindrical Stock:

G778 Z3 X2 I1

Z = Length

X = Diameter

I = Inside Diameter

How to Change the User Image

The control looks first in the /ram folder for a file named user_image.jpg. If it does not find one, it uses the Milltronics logo. To customize the user image, put any .jpg image in the /ram folder, renamed as user_image.jpg. The control automatically scales the image to fit it into the user info window.

If there is already a file named user_image.jpg in the /ram folder, delete the old user_ image.jpg before inserting the new one.

The File Utilities screen may be used to accomplish all these file manipulations. The ram folder does not appear as one of the available directory choices in File Utilities until logged in as supervisor with valid “user” and “password”.

After a new user_image.jpg file has been inserted in the /ram folder, cycle power.

Converting 7200 Conversational Programs to 8200 Conversational Programs

There are two ways to convert 7200 conversational programs to 8200 conversational program.

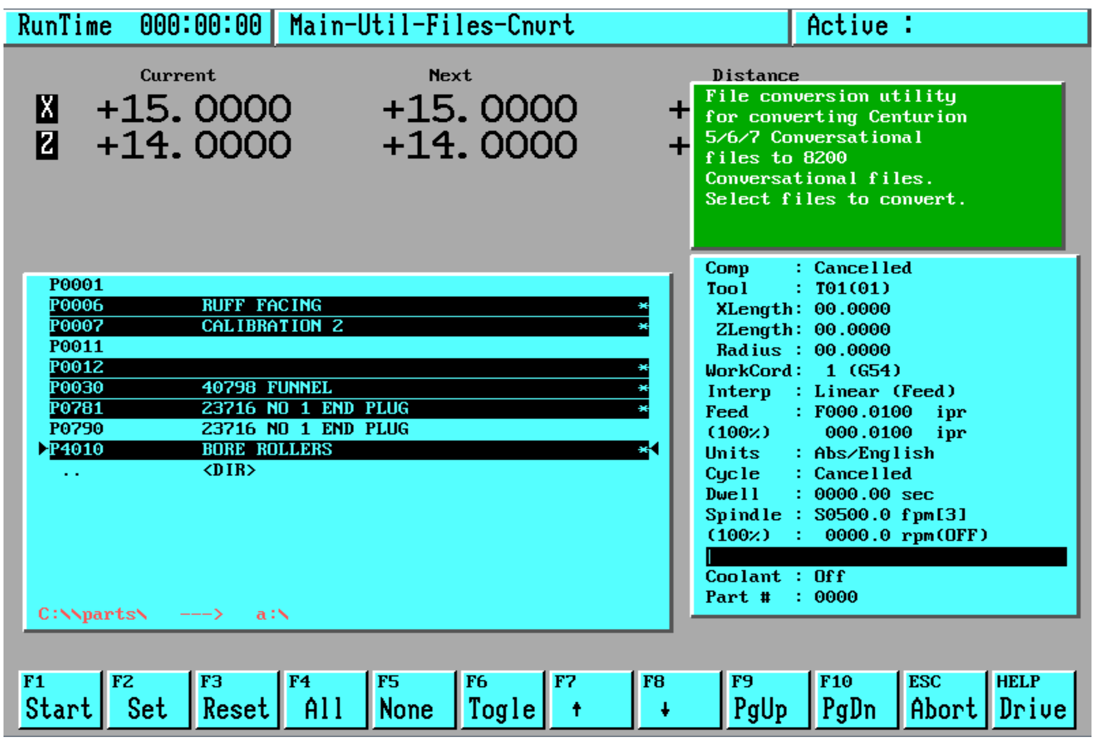
Method One

Use 7200 LatheCam to convert 7200 conversational programs to 8200 conversational programs as follows.

- From the Main menu F10 (Util)→F3(Files)→F7 (Cnvrt).
- Select the files that you want to convert using the F2(Set), key.
- Press the F1(Start) key.

In the illustration below, selected c:\cnc\parts\P#### conversational files will be converted to 8200 a:\P####.CVN conversational files.

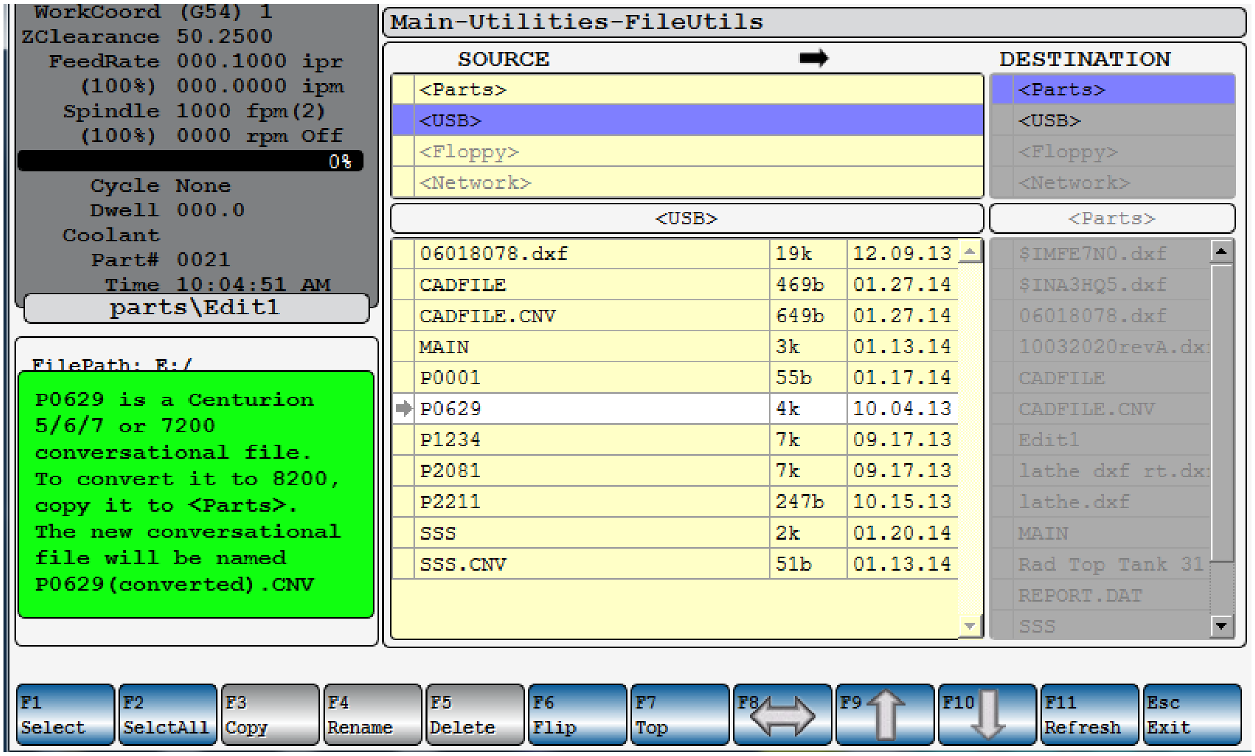
Note: After the files are loaded into the 8200 control you will need to Post each of the files. To Post the file, open it with the 8200 conversational editor and then escape from the editor.



Method Two

Use the 8200 to convert 7200 conversational programs to 8200 conversational programs.

- Load the 7200 P#### conversational files onto a USB flash drive.
- From the Main menu F10(Utility)→F5(FileUtil).
- Select USB as the source and Parts as the destination.
- When the cursor is pointing to a 7200 conversational file it will be indicated with a message as shown below.



- F1(Select) the files that you want converted
- F3(Copy) the files to parts. The new conversational files are named P####(converted) .CNV and P####(converted).

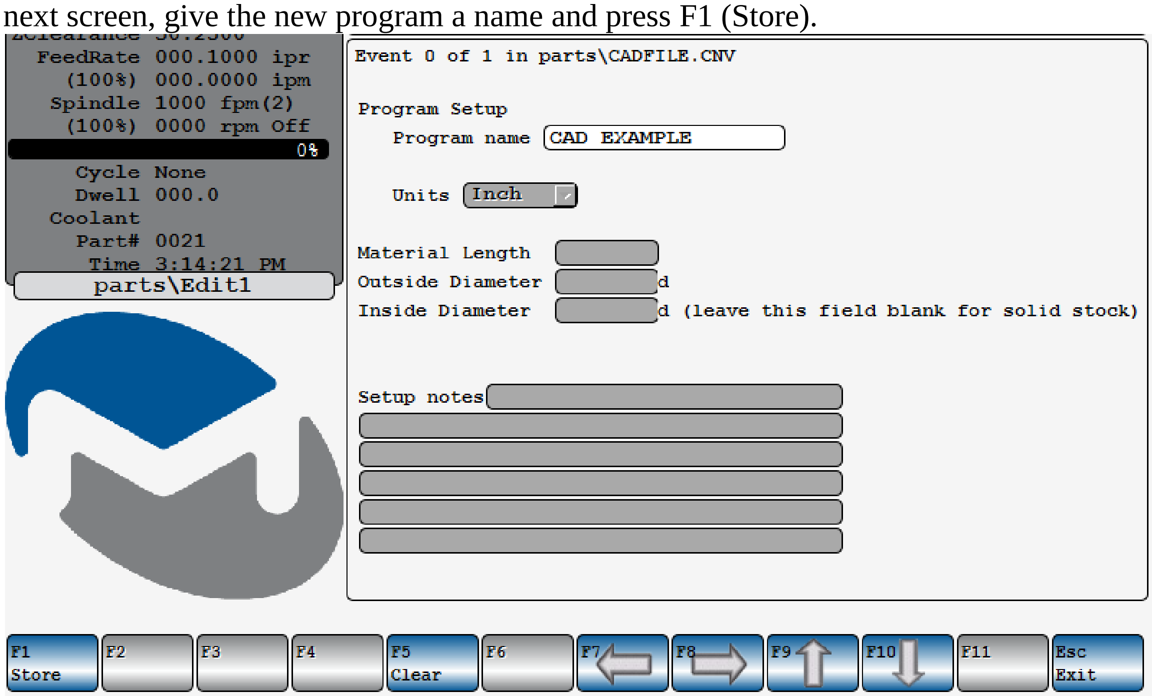
Example: P0629 will be converted to P0629 (converted).CNV and P0629 (converted) in the parts folder.

CAD File Import

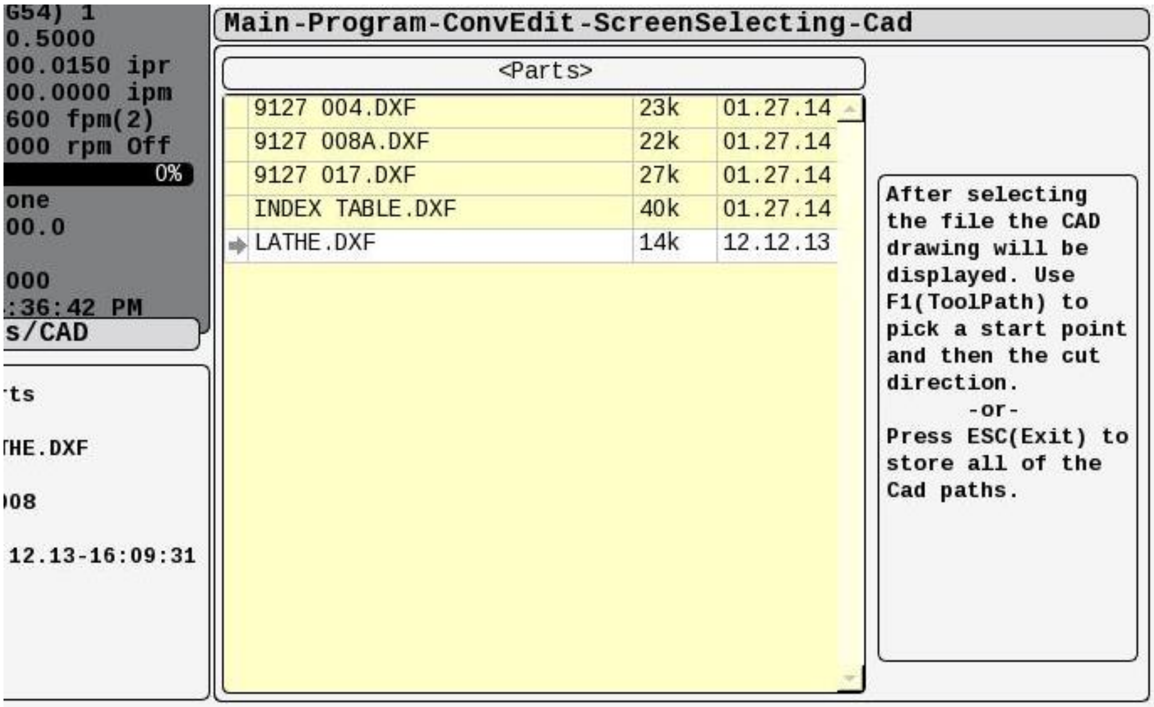
The 8200 Controller has the ability to import CAD files to a conversational program. To begin this process, first load the CAD (.DXF or .IGS) file into the <Parts> directory using the procedure previously mentioned (top half of screens cropped for simplicity).

Starting at the Main screen, press F8 (Program), then F2 (Conv), then F2 (New), to start a new conversational program. Enter a new file name and press Enter.

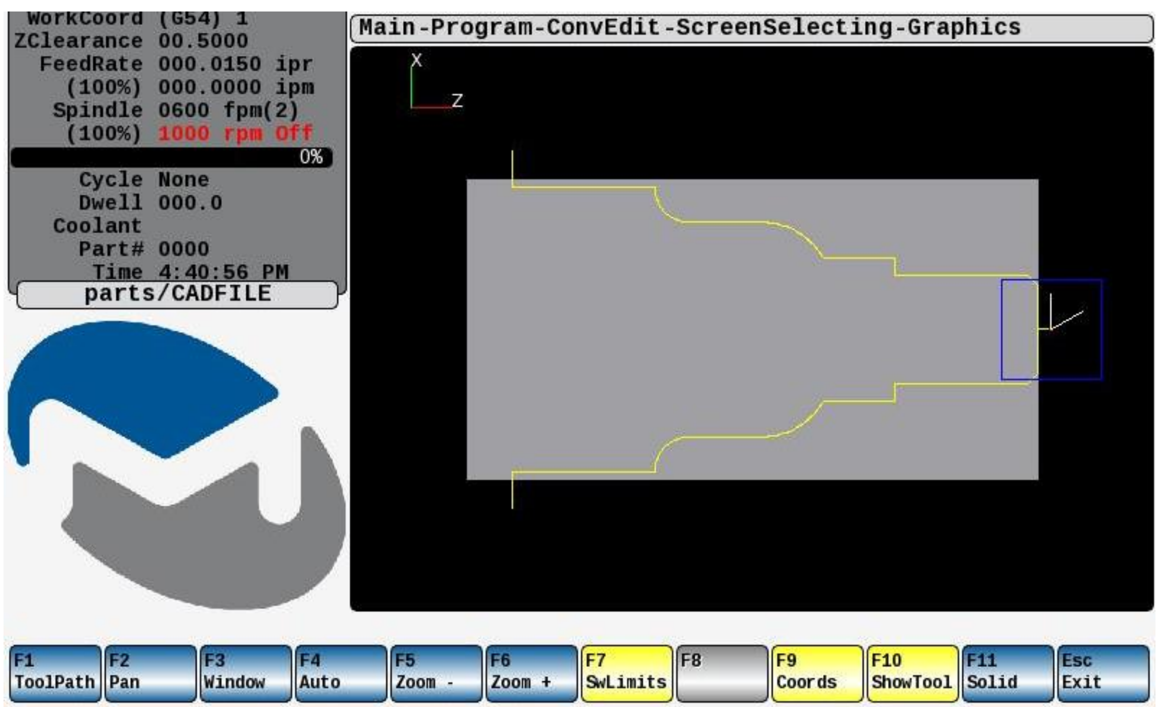
On the next screen, give the new program a name and press F1 (Store).



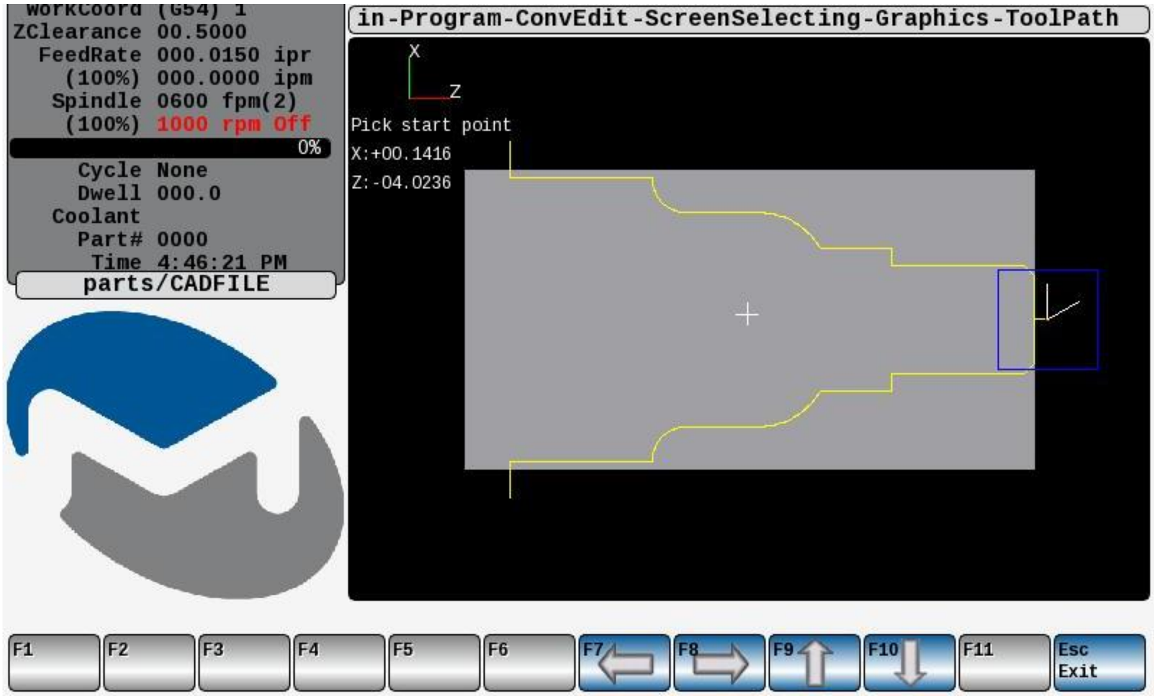
On this screen, press F9 (Special) and on the next screen press F1 (CAD). This brings up the screen shown below.



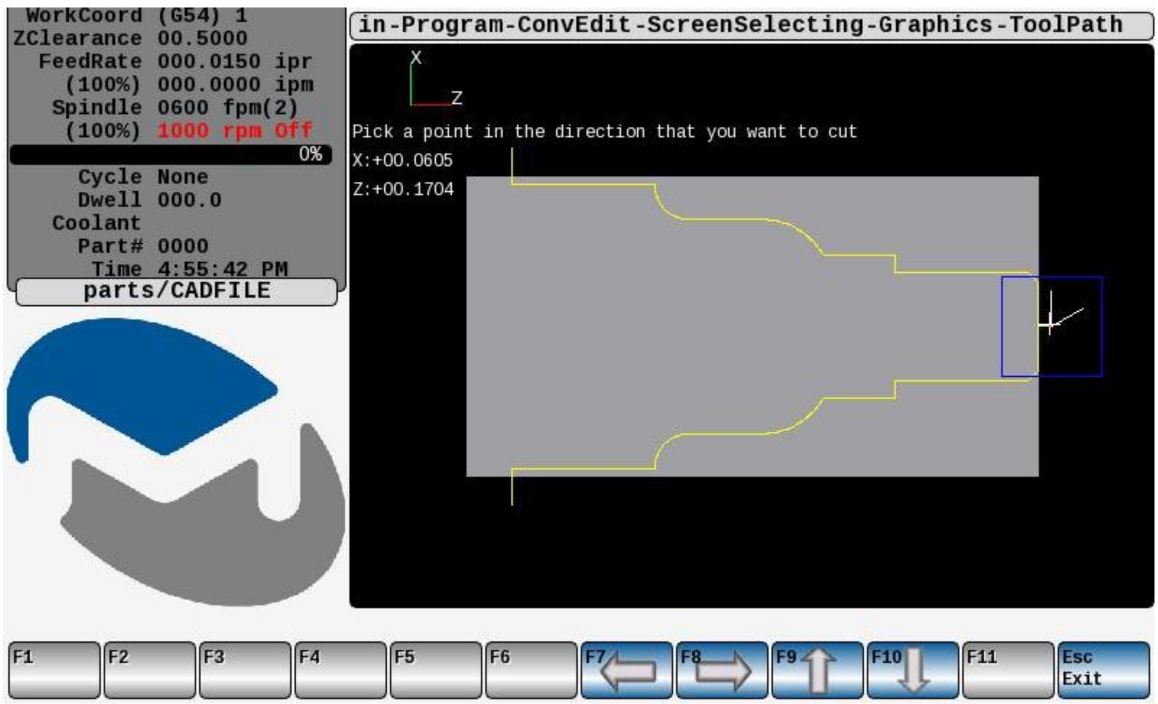
On this screen, select the file you wish to import from the <Parts> directory (in this example, LATHE.DXF) and press F1 (Select). This shows the geometry from the CAD file.



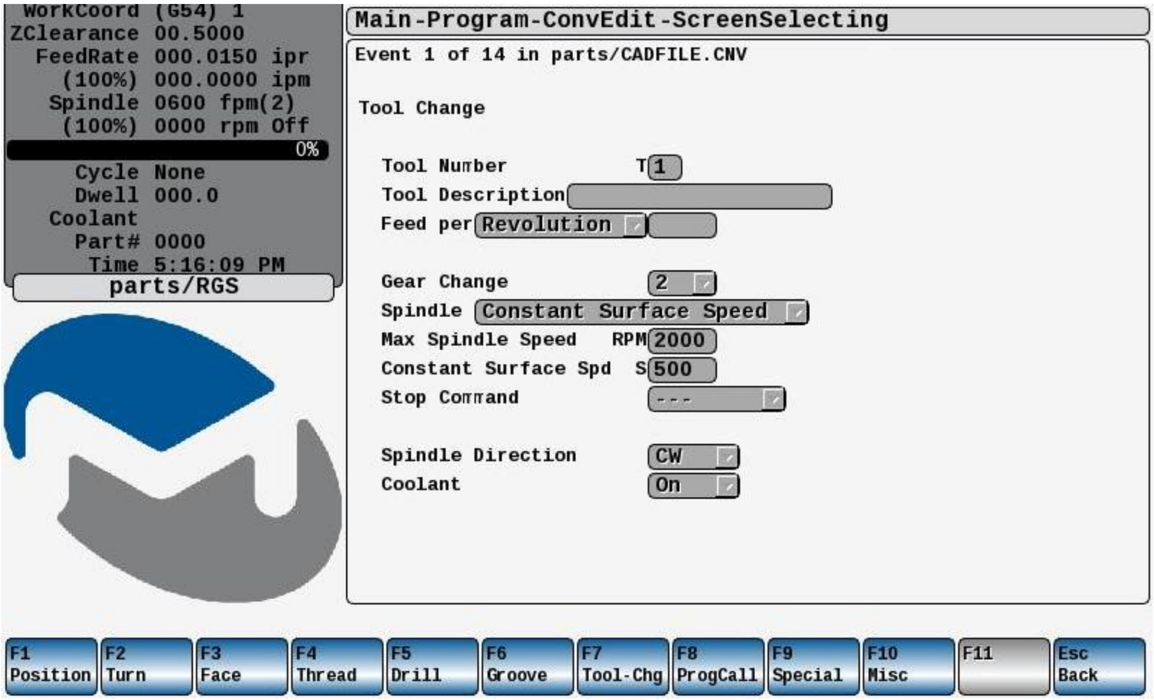
On this screen choose F1 (Tool Path), which brings up the screen below which requests that the operator move the cross hair to the start point.



After choosing a starting point, press Enter and the screen below appears, requesting the operator to pick a point in the direction that you want the cut.



After moving the cursor in the desired direction, press Enter to bring up a screen which displays the last event in the program. The program has created events to cut the geometry in the CAD file. You may now fine tune the program by editing the various steps to insert, proper stock size, tool changes, spindle speeds, feed-rates , start and end turning (or facing) cycles, etc.



Finally, verify the new program to make sure the finished part matches the CAD geometry.

Notes on CAD files:

- You can specify DXF or IGS file types in the Miscellaneous parameters.
- Cad file geometry should be trimmed or extended to avoid breaks in the tool path (The Miscellaneous parameter CAD epsilon can be used to span small gaps in the CAD path).
- CAD geometry should be drawn such that X CAD dimensions are Z part dimensions and Y CAD dimensions are (X) radius values of the part.
- If your tools are on the back side of the material, positive CAD Y dimensions specify the (X+) back side of your part.
- If your tools are on the front side of the material, negative CAD Y dimensions specify the (X+) front side of your part.
- Only one side of the part should be drawn in your CAD file (do not draw both sides)

5

Section 5 Contents

Introduction to Writing a Conversational Program

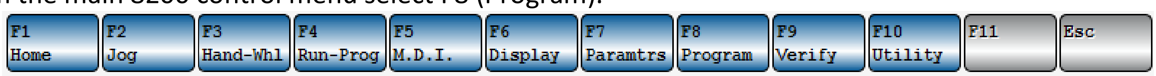
Conversational Programming is a method of programming the CNC wherein the programmer selects the type of task he/she wishes the machine to perform on the part and only has to enter data requested by the program on various menu screens. It is very intuitive for most machine operators and does not require entering text commands.

Each task is called an event and a complete program consists of a number of events, beginning with setting up the stock to be machined and the tools required and ending with retracting the last tool.

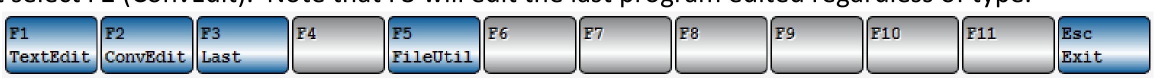
To write a conversational program, the operator must understand the sequence of tasks he wants the machine to perform on the part and then, with a general knowledge of the screen choices offered by the 8200 Controller, navigate to the proper menu screen to program these tasks.

Accessing the Conversational Editor

From the main 8200 control menu select F8 (Program).

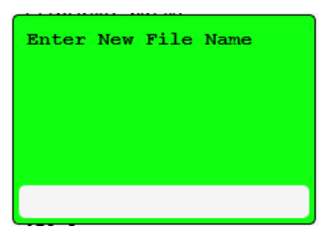


Next select F2 (ConvEdit). Note that F3 will edit the last program edited regardless of type.



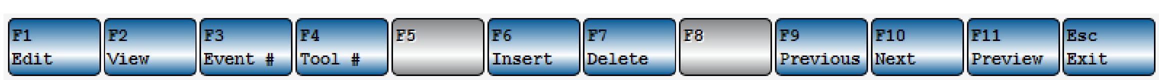
Then select F2 (New), or select a program to edit from the menu shown.

If you selected F2 (New) you will be prompted for a new file name. Type in a name for the file then press enter. You will then arrive in the program setup screen for your new program.



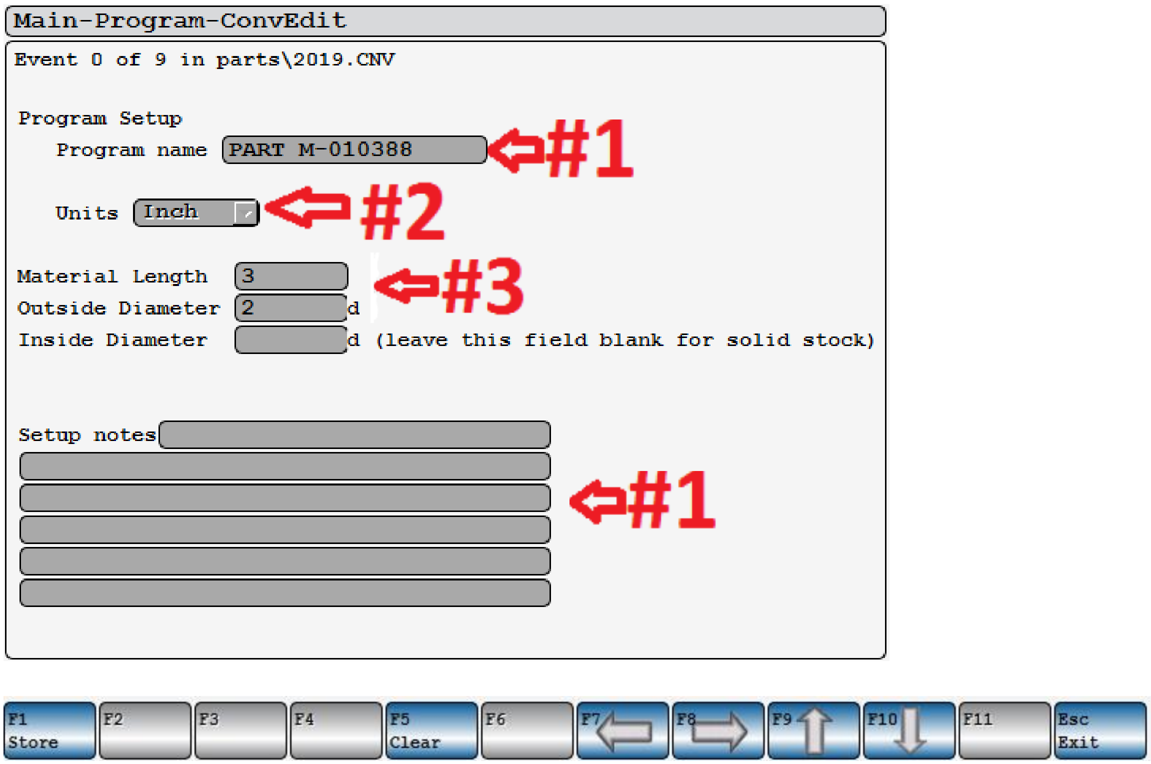
Navigating the Conversational Editor

After a name is specified, The Program Setup screen in the conversational editor will appear. This section describes the various field types and convenience features found within the conversational programming system. The navigation menu is shown below.



F1 (Edit) will allow editing of the event currently displayed

Field Types



There are 3 field types in conversational programs:

#1 Data fields that allow **Alpha and Numeric** data.
The Program name and the Setup Notes (above)

#2 Toggle fields, uses F3 (Toggle) to select
The Units (Inch or Metric)
The Work Coordinate (G54, G55…G59) (above)

#3 Data fields that allow **Numeric data**. F5 (Clear) will clears the data field
Stock sizes (above)
Note 1: Numeric fields allow math statements such as:

- 2+3/8
- 5/8-.004
- .+.003 ← "."uses the current value in the field +.003
- ./2 ← "."uses the current value in the field /2

Note2: Numeric fields allow parameter statements such as:

- P10+.1
- 3+P11
- P1/3
- P1+P2
- P[P1]

F1 (Edit) - F1 (Store)

Accepts the entries and adds to the program file. If all required data has not been entered, the F1 (Store) key is grayed out and the field requiring input is red. Each screen stored is called an event.

F1 (Edit) - F3 (Toggle)

Pressing this key results in the next toggle value being displayed in the field. The F3 (Toggle) button is absent if the current field is not a toggle field.

F1 (Edit) - F5 (Clear)

Used to clear a data field. Absent if the current field is a toggle field.

F1 (Edit) - F7 (Left Arrow)

Moves cursor to the left. Has no effect in a toggle field.

F1 (Edit) - F8 (Right Arrow)

Moves cursor to the right. Has no effect in a toggle field.

F1 (Edit) - F9 (Up Arrow)

Moves cursor to the previous field.

F1 (Edit) - F10 (Down Arrow)

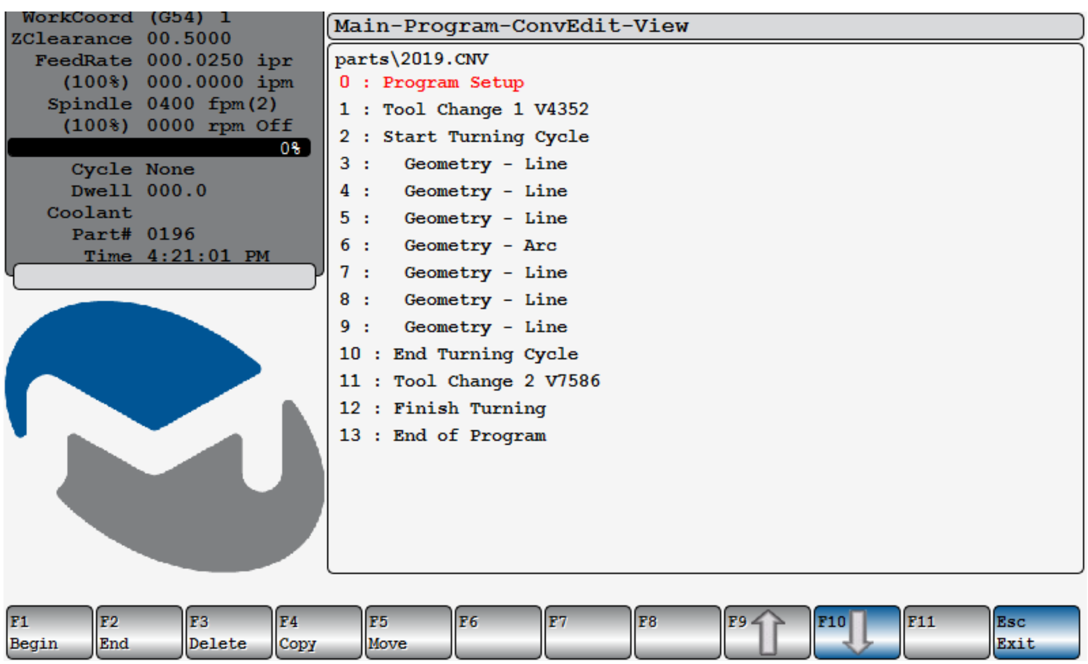
Moves cursor to the next field.

F1 (Edit) – Esc (Exit)

Aborts event input. Returns to the menu keys.

F2 (View)

Allows viewing of the entire program and lets the operator position to any of the events in the program. A window similar to the following will be displayed. F9 (Up Arrow) and F10 (Down Arrow) are used to move from event to event. Pressing Enter displays the event. Events may also be moved in blocks.



F2 (View) - F1 (Begin)
Marks the block beginning.

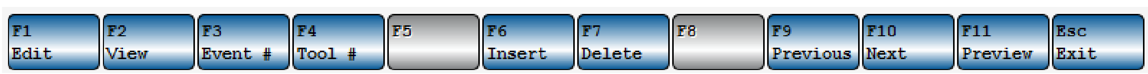
F2 (View) - F2 (End)
Marks the block end.

F2 (View) - F3 (Delete)
Deletes the block of events (or the current event, if no blocks are highlighted).

F2 (View) - F4 (Copy)
Copies the block of events to the cursor position

F2 (View) - F5 (Move)
Moves the block of events to the cursor position

Note: F4 (Copy), and F5 (Move) do not become active until a block of events has been selected. A single event may constitute a block. A single event is selected as a block if F1 (Begin) then F2 (End) are pressed while the block is highlighted.
The program setup and end of program events may not be moved, deleted or copied.



F3 (Event #)
Allows entry of an event number for which to search. If the event number is not found, the end of program screen will be displayed.

F4 (Tool #)
Allows entry to search for a tool number. If tool number is not found the event displayed will not change.

F6 (Insert)
INS is used to insert events in a program. The new event(s) will be inserted before the event that is currently displayed. Inserting will continue until the F10 (Exit) soft key is pressed.

F7 (Delete)
Deletes the event currently being displayed.
The program setup and end of program events may not be moved, deleted or copied.

F9 (Previous)
Displays the previous event in the program file.

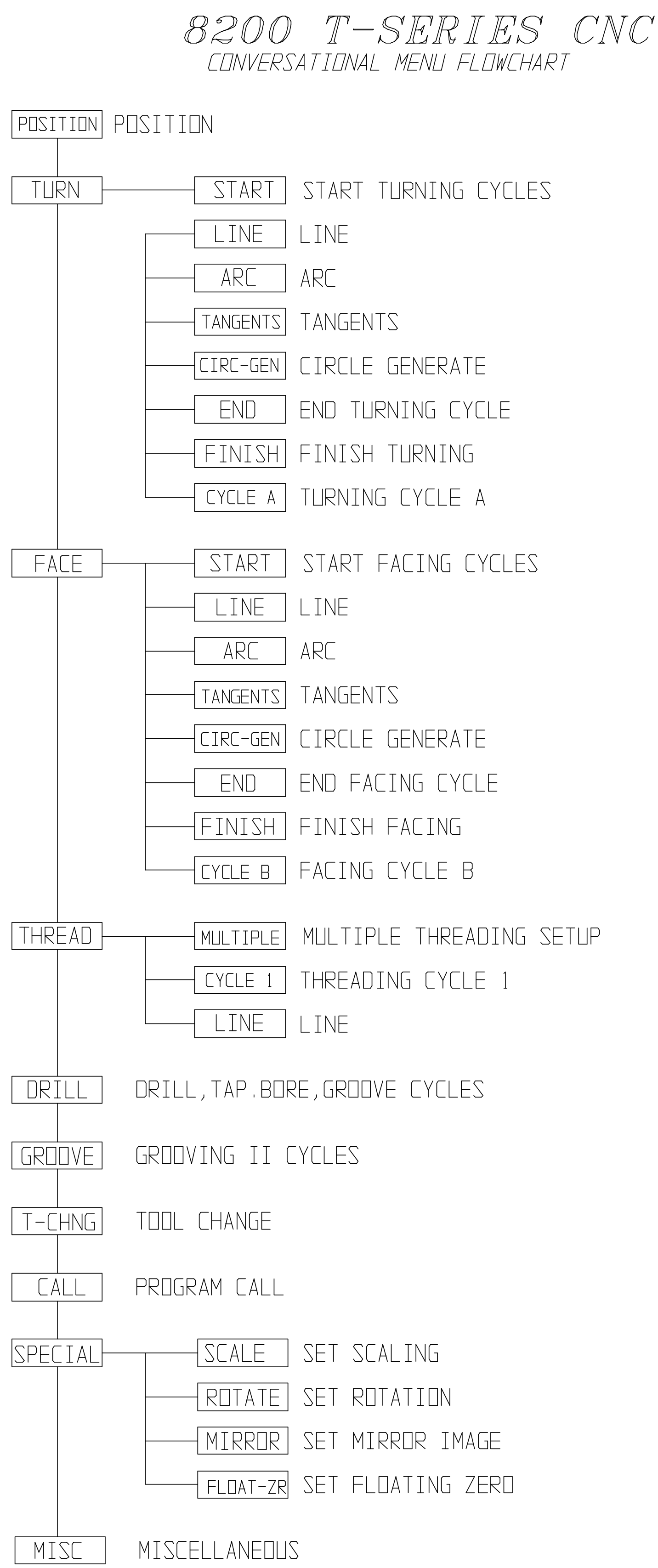
F10 (Next)
Displays the next event in the program file.

F11 (Preview)

For previewing the program being edited.
Preview-within-Edit is not available if another program is currently running or verifying. When F11 (Preview) is pressed, the program is Fast verified without cutter comp (in wireframe) or in solid model mode, depending which mode is currently selected. Graphics are auto-scaled. They can be rotated, scaled, zoomed, etc. Pressing ESC returns to the conversational editor.

Notes: When the program is being previewed, it will ignore T####s, M0s, M1s, INPUT statements, etc. Preview is designed to make a quick sketch of the program toolpath. A full-featured Verify (similar to Run), if required, may be performed on the part program.

Conversational Menu Flowchart



Explanation of Conversational Input Screens

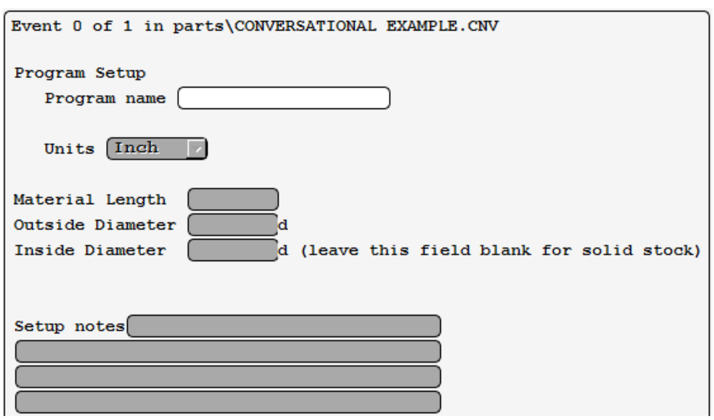
Each conversational program has a text program associated with it. The conversation program has any name, with a .CNV extension. The text file has the same name as the conversational file from which it was generated, but without the .CNV extension. Changes in the conversational program create a new text program from the modified conversational program. The operator can view or change the text program, but his modifications will not be transferred to the conversational program. Reposting the conversational program will overwrite his modified text program.

This section contains diagrams of the conversational input screens and an explanation of each screen. Not all possible combinations of screen inputs are shown; therefore, if additional information concerning any particular screen or field is required, the appropriate section of the manual should be referenced.

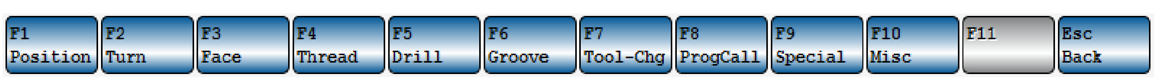
On the CNC display, **fields that are red are required entries**. These fields appear black after the data is entered. The program setup screen appears at the beginning of every program. Where you see Rapid, the indicates that there are other choices that can be selected by pressing F3 (Toggle). For example “Rapid” can be toggled to “Feed”.

Program Setup Event

The conversational screen for program setup is the first event of every conversational program. The program setup screen is where you can enter an optional program name to associate the program with a job name or number, etc. choose inch or metric mode, enter stock size data, and select a work coordinate. Under setup notes it is possible to put any kind of text information helpful to running this particular program.



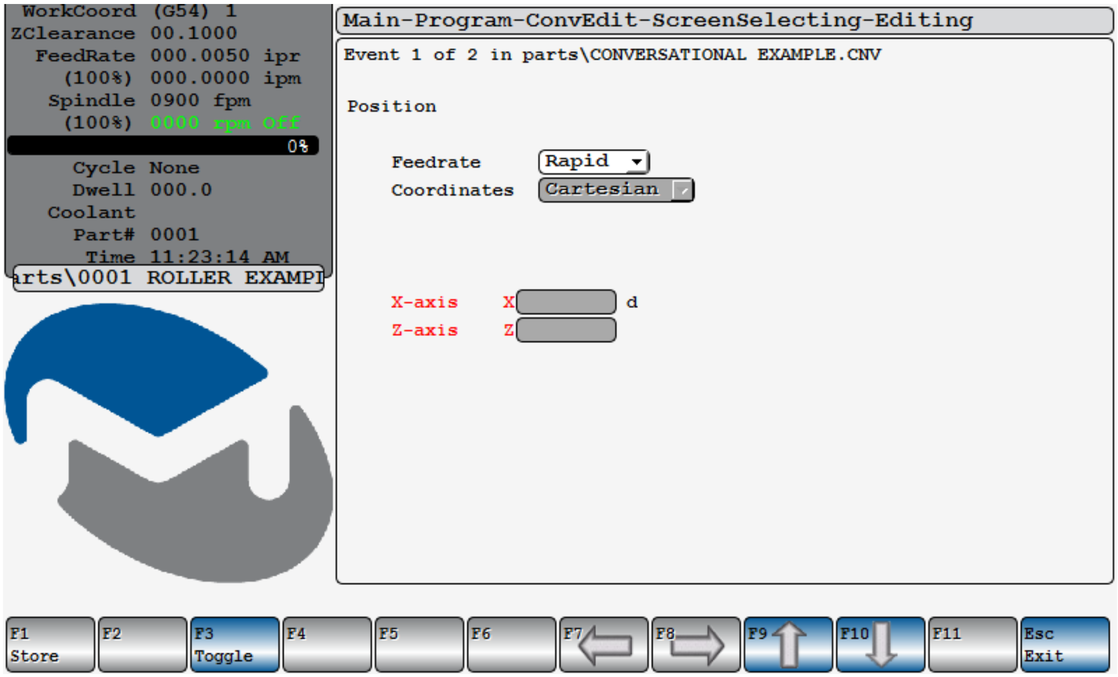
After the setup screen is stored (by pressing F1 (Store)), the main menu for conversational programming is presented as function key choices:



See the conversational menu flowchart diagram on the previous page for a general scheme of the menu choices within the conversational menu system.

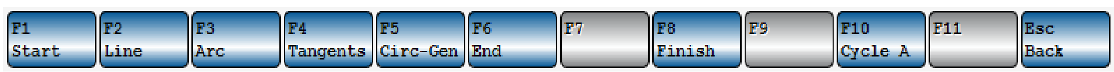
F1 (Position) Position

The position screen normally is used for rapid positioning. Feed moves may be made by toggling the feedrate field and entering a feedrate. A move may be defined in Cartesian coordinates or in polar coordinates.



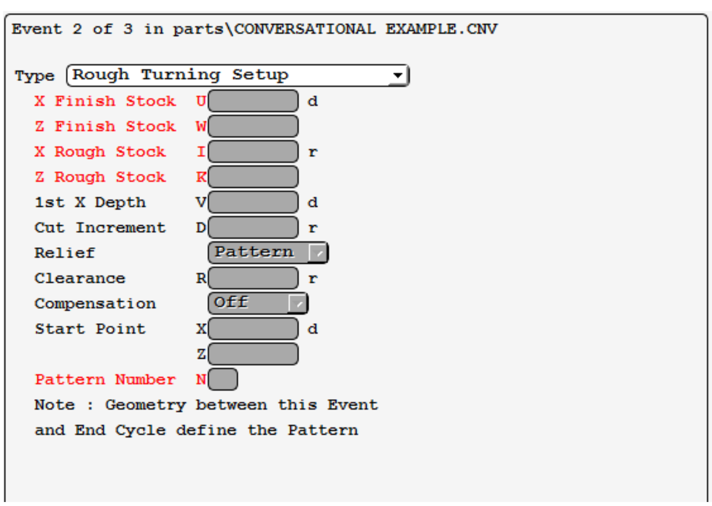
F2 (Turn) Turning Functions

The F2 (Turn) selection brings up the following soft keys.



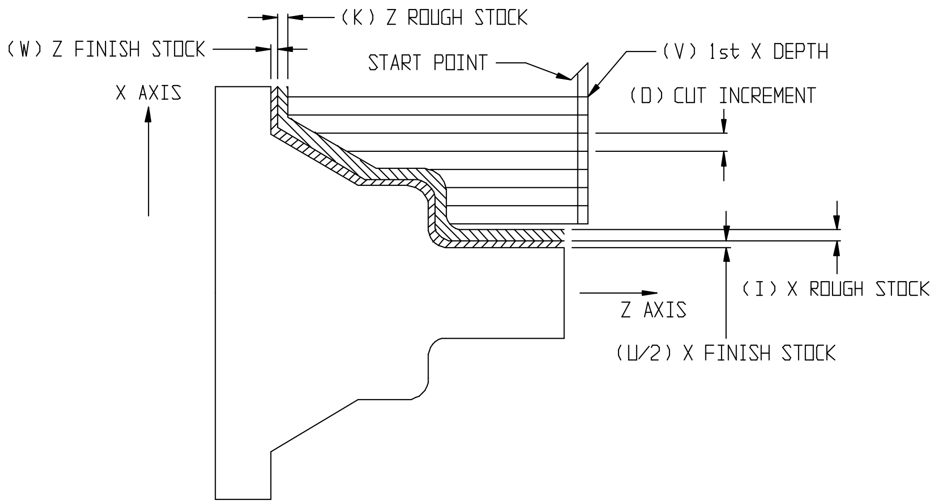
F2 (Turn) → F1 (Start) Turning Setup

Rough Turning Setup selected. The Turning Set-Up is used to define how to remove the material defined in the geometry between this screen and the end of the cycle.



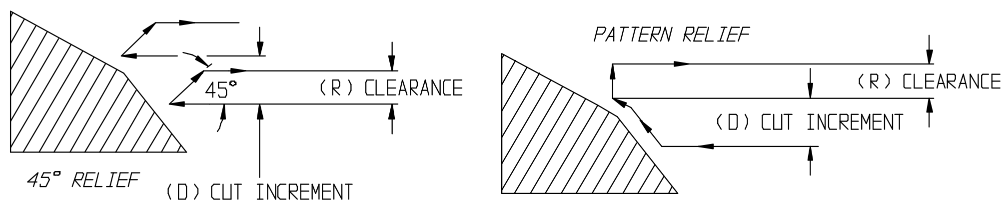
(U) X Finish Stock:	Distance and direction of the finish allowance in the X axis.
(W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis.
(I) X Rough Stock:	Distance and direction of the rough finish allowance in the X axis.
(K) Z Rough Stock:	Distance and direction of the rough-finish allowance in the Z axis.
(V) First X Depth:	The X depth of the 1st cut in the roughing cycle. If this is not entered the 1st cut is at (X start point) + (X finish stock/2) + (X rough stock) - (cut increment).
(D) Cut Increment:	Depth of cuts. Designated without sign. The cutting direction depends on the direction of the 1st move in the pattern.
Relief	The type of relief made at the end of the cutting. PATTERN will cut along the pattern of the part, 45 DEGS. will retract at a 45 degree angle before retracting Z for the next cut. If the relief type is 45 DEGS. no pockets are allowed in the pattern.
(R) Clearance:	Distance of the 45 degree relief or an additional distance to pull away after the pattern relief.
Compensation:	Tool nose radius compensation. OFF, LEFT, or RIGHT
(X,Z) Start Point:	The start point of the roughing cycle. X and Z return to the start point at the end of the cycle.
(N) Pattern Number:	Any number between 0 and 99. The pattern can be called by a finishing cycle later in the program.

Rough cutting is done by first cutting parallel to the Z axis down to the pattern leaving the rough stock. After cutting parallel to the Z axis is completed, a pass is made along the finish pattern leaving the finish stock.

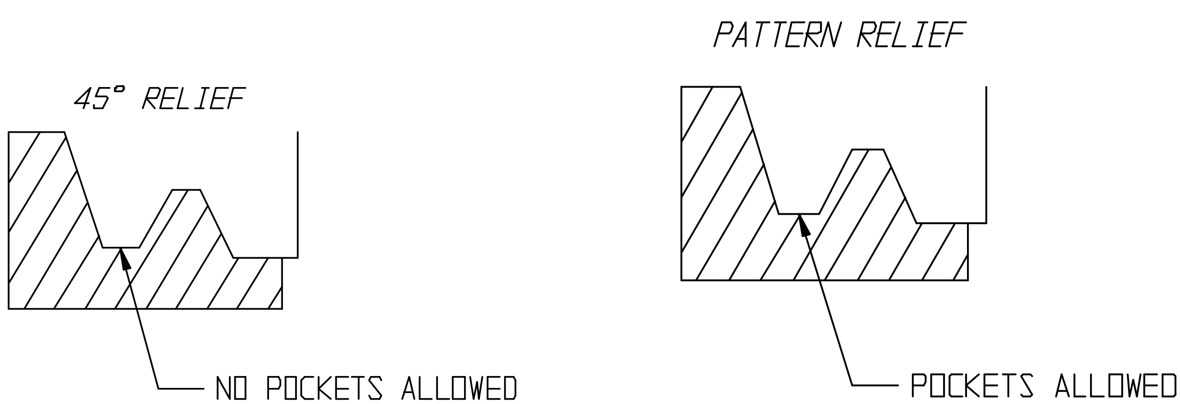


Feed and speeds within the pattern are ignored during the parallel cuts. Feeds and speeds are effective during the pass along the pattern. Rapid moves within the pattern are effective when following the finish pattern.

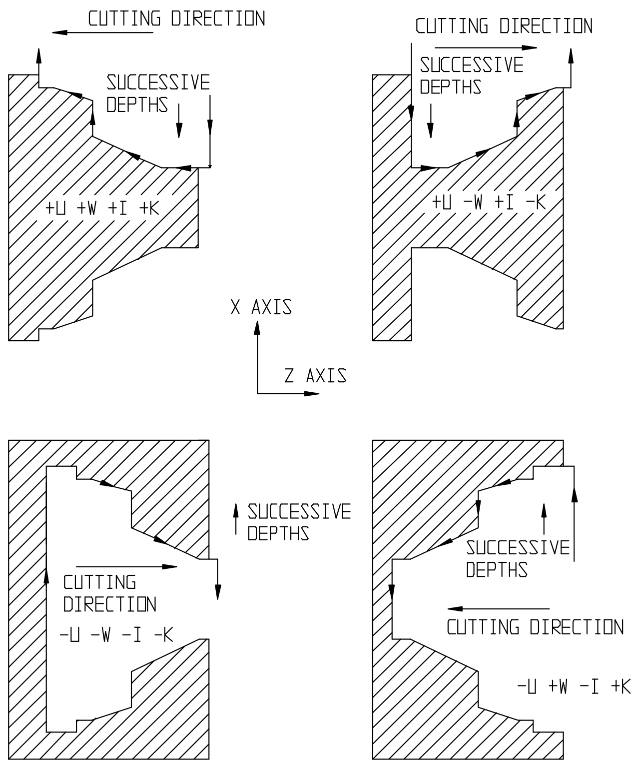
Note: If the X rough stock and the Z rough stock are both zero, the pass made along the finish pattern is skipped.



After rough cutting parallel to the Z axis there are 2 types of relief. 45º relief will rapid away from the pattern at a 45º angle. Pattern relief follows the pattern and then retracts in the X axis. A 0 (zero) clearance is allowed in the pattern relief.

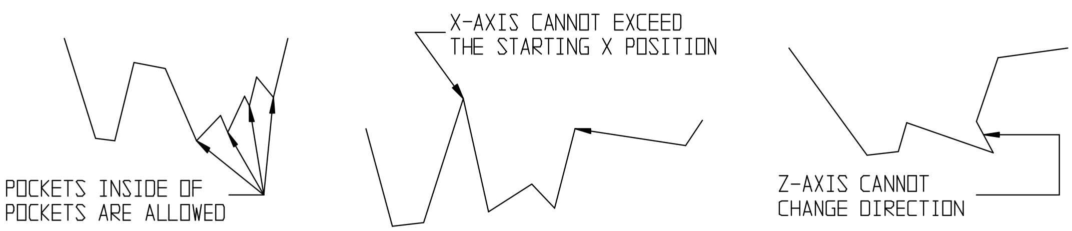


No pockets are allowed when using the 45° relief.



There are four types of patterns made by the rough turning cycle. Rough cutting for each of these four patterns is done parallel to the Z axis. The signs of X and Z rough and finish stocks are shown for each pattern.

When there are pockets, the Z rough and finish stocks are normally 0.If Z rough and finish stocks other than 0 are specified, it may cut beyond the finish pattern.

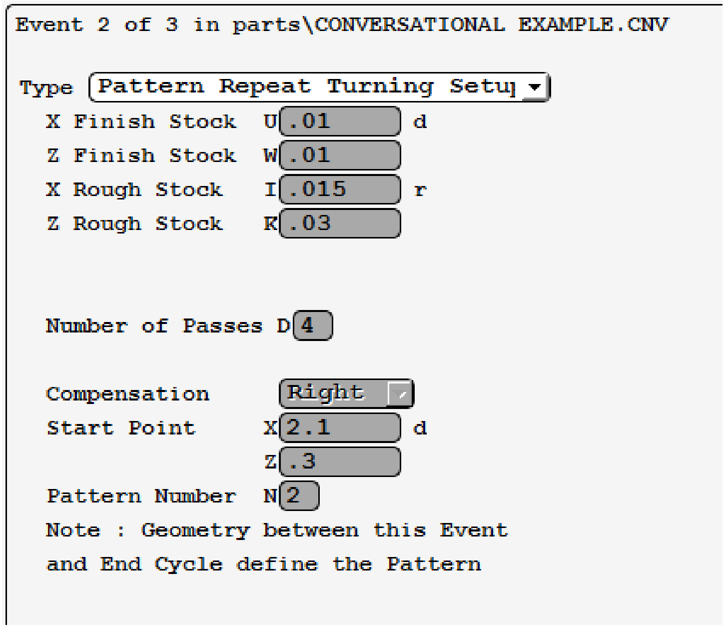


Note 1: *Moves back to the start point from the end of the pattern are rapid moves.*

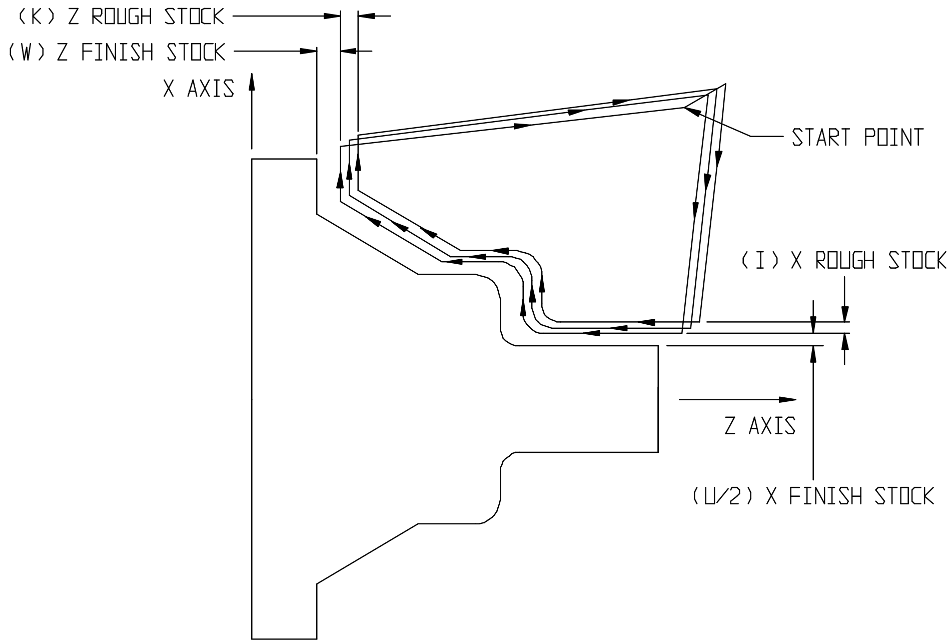
Note 2: *If the X end point in the pattern is not equal to the X start point of the pattern, a shoulder to the X start point will be added.*

F2 (Turn) → F1 (Start) Turning Setup

Pattern Repeat Turning Setup selected. The Pattern Repeat Cycle repeatedly cuts a pattern incrementing the cuts down to the desired size. This cycle is used to efficiently cut a part whose shape is already defined (forged or cast).



(U) X Finish Stock:	Distance and direction of the finish allowance in the X axis.
(W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis.
(I) X Rough Stock:	Distance and direction of the rough allowance in the X axis.
(K) Z Rough Stock:	Distance and direction of the rough allowance in the Z axis.
(D) Number of Passes:	The number of passes to cut the rough stock.
Compensation:	Tool nose radius compensation. OFF, LEFT, or RIGHT
(X,Z) Start Point:	The start point of the roughing cycle. X and Z return to the start point at the end of the cycle.
(N) Pattern Number:	Any number between 0 and 99. The pattern number can be called by a finishing cycle later in the program.



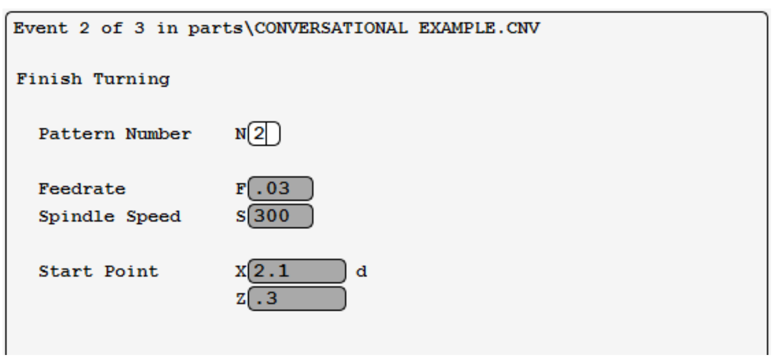
Note 1: *Any feeds or speeds within the pattern are ignored.*

Note 2: *Rapid moves within the pattern will be executed as rapid moves during the pattern repeat cycle.*

Note 3: *Moves back to the start point, from the end of the pattern, are rapid moves.*

F2 (Turn) → F8 (Finish) Finish Turning

After rough cutting with the rough turning cycle or pattern repeat cycle, the finish cycle can be used to remove the finish stock.



Event 2 of 3 in parts\CONVERSATIONAL EXAMPLE.CNV

Finish Turning

Pattern Number N 2

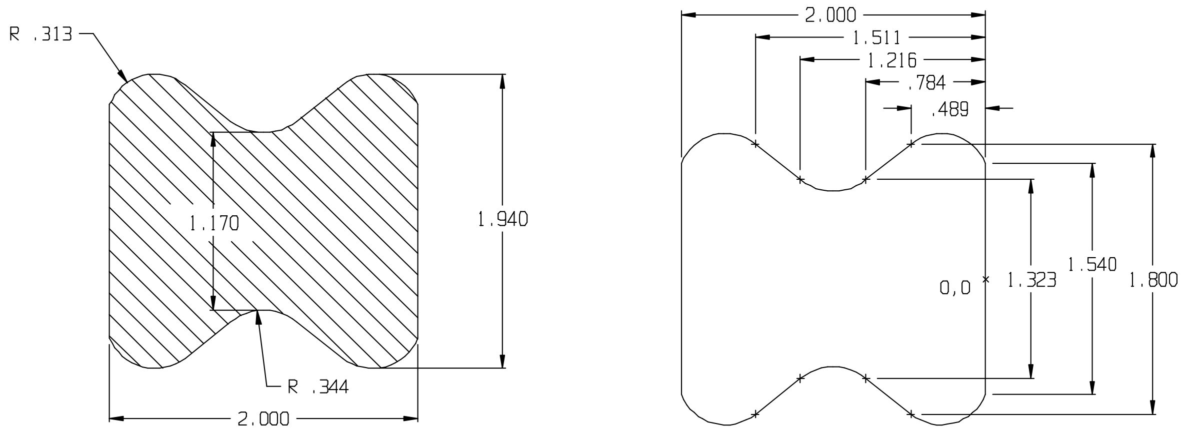
Feedrate F .03
Spindle Speed S 300

Start Point X 2.1 d
Z .3

(N) Pattern Number:	Any pattern defined by a rough turning or pattern repeat cycle.
(F) Feedrate:	The feedrate to cut the finish pattern. Any feedrates within the pattern will be effective.
(S) Spindle Speed:	The spindle speed to cut the finish pattern. Any spindle speed within the pattern will be effective.
(X,Z) Start Point:	The start point of the finish cycle. X and Z return to the start point at the end of the cycle at a rapid feedrate.

Note: The only difference in the code generated between the Finish Turning and Finish Facing Cycle is X retracts to the start point before Z after the cycle has been executed.

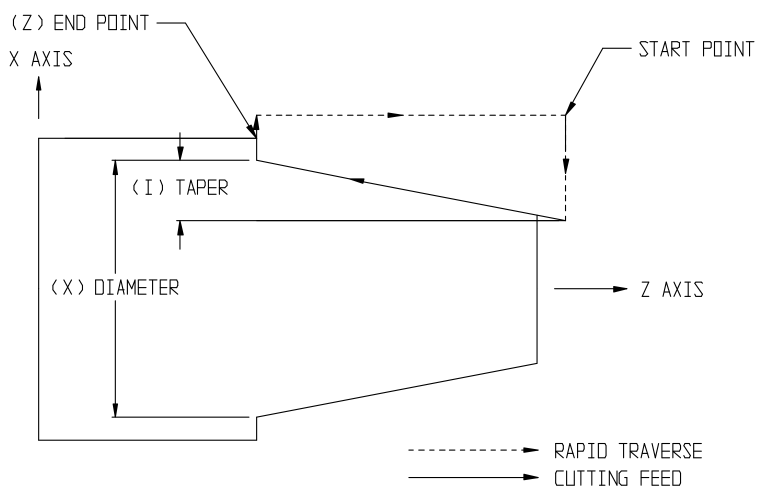
Example Program



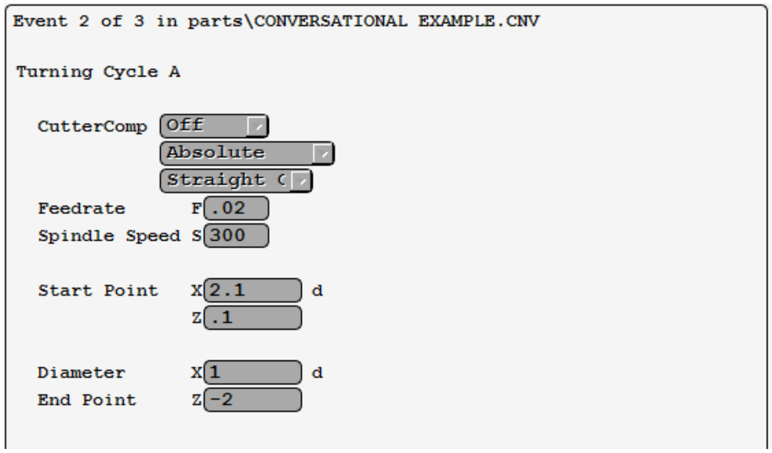
See Section 9 for an example program using a rough turning cycle with a finish cycle for the part shown above:

F2 (Turn) → F10 (Cycle A) Turning Cycle A

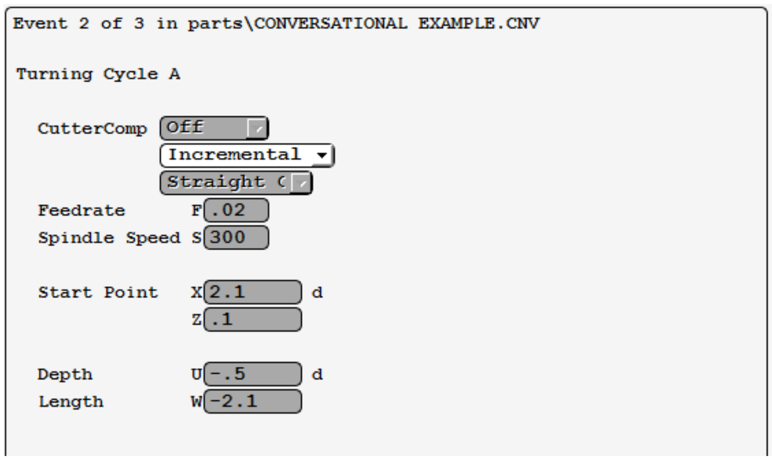
Cycle A is used to make a single turning cut. The cycle rapids to the X cutting depth, makes the cut, retracts X axis and rapids back to the start point. Tapered cuts are allowed.



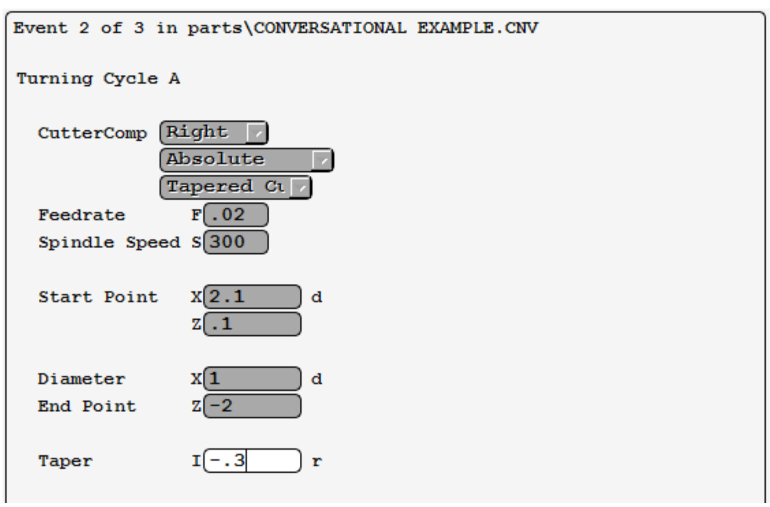
Tapered Cutting Using Cycle A with Absolute Dimensions


Event 2 of 3 in parts\CONVERSATIONAL EXAMPLE.CNV
Turning Cycle A
CutterComp Off
Absolute
Straight C
Feedrate F .02
Spindle Speed S 300
Start Point X 2.1 d
Z .1
Diameter X 1 d
End Point Z -2

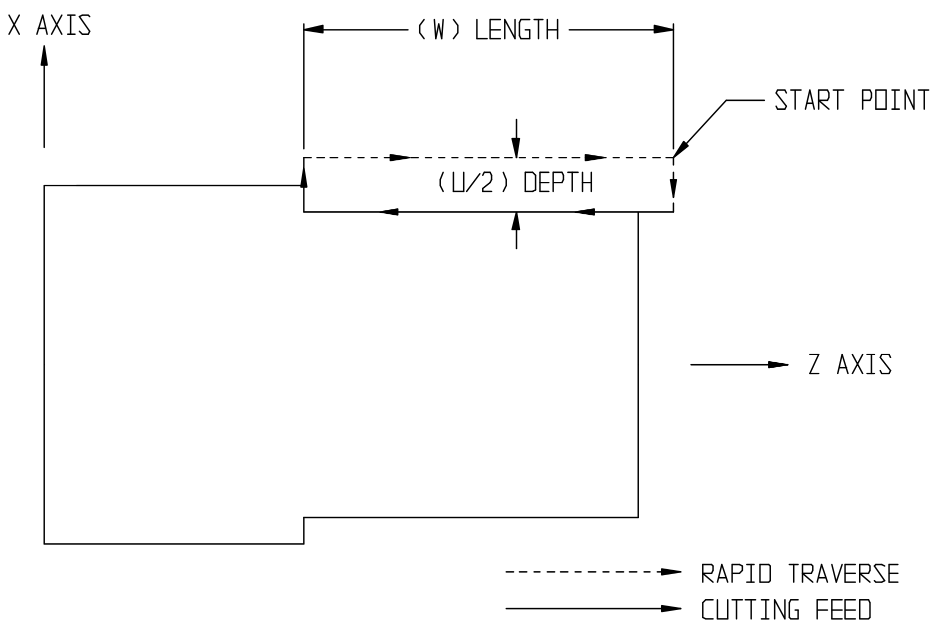
-OR-


Event 2 of 3 in parts\CONVERSATIONAL EXAMPLE.CNV
Turning Cycle A
CutterComp Off
Incremental
Straight C
Feedrate F .02
Spindle Speed S 300
Start Point X 2.1 d
Z .1
Depth U -.5 d
Length W -2.1

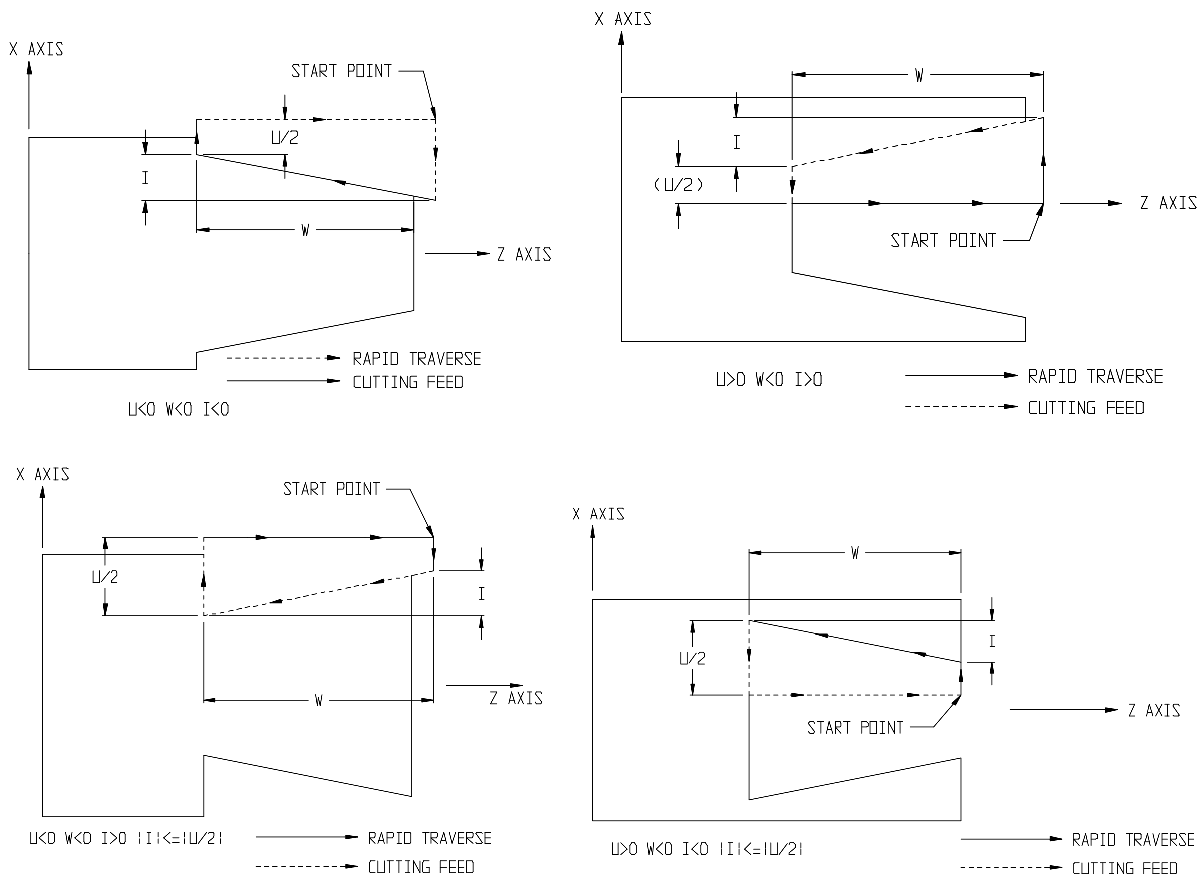
-OR-


Event 2 of 3 in parts\CONVERSATIONAL EXAMPLE.CNV
Turning Cycle A
CutterComp Right
Absolute
Tapered Cu
Feedrate F .02
Spindle Speed S 300
Start Point X 2.1 d
Z .1
Diameter X 1 d
End Point Z -2
Taper I -.3 r

Compensation:	Tool nose radius compensation. OFF, LEFT, or RIGHT.
Absolute or Incremental:	Toggles between absolute dimensions (diameter, end point) and incremental dimensions (depth and length).
(F) Feedrate:	The feedrate to turn the part.
(S) Spindle Speed:	The spindle speed to turn the part.
(X, Z) Start Point:	The start point of Cycle A. X and Z return to the start point at the end of the cycle at a rapid feedrate.
(X) Diameter:	The diameter to turn.
(Z) End Point:	The end point of the cut.
(U) Depth:	The incremental X distance and direction from the start point to the diameter to turn.
(W) Length:	The length and direction of the cut in Z from the start point.
(I) Taper:	The amount and direction of the taper to turn.



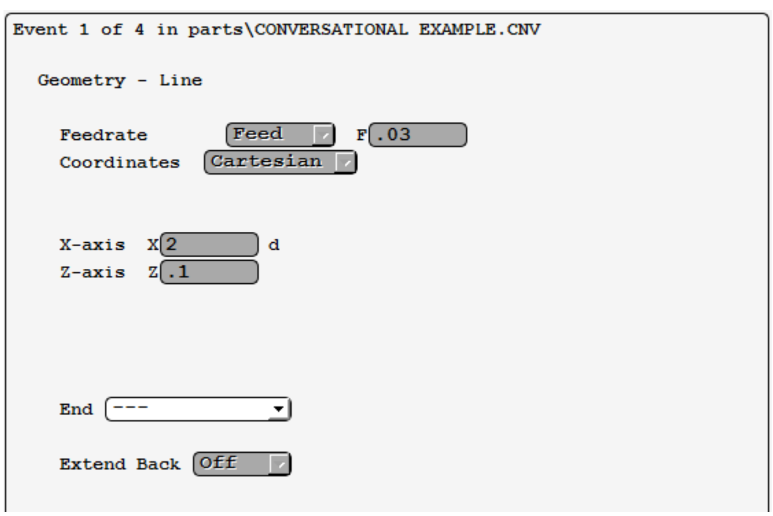
Straight Cutting Using Cycle A with Incremental Dimensions



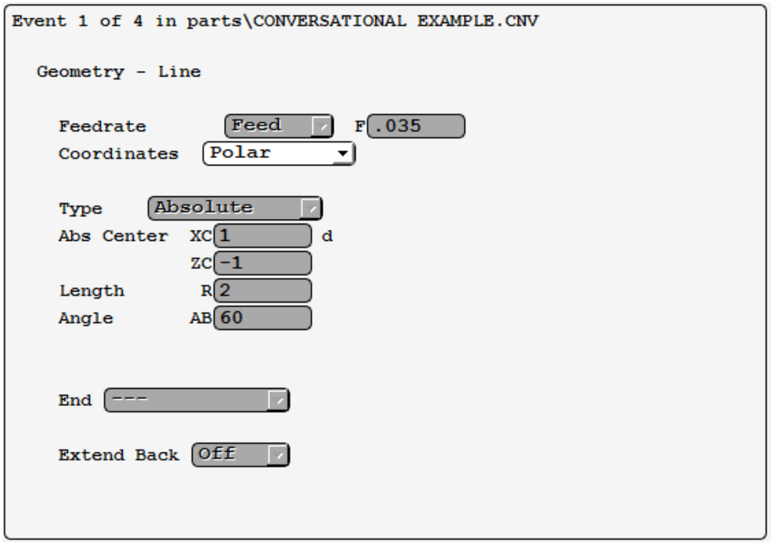
When using Cycle A with incremental dimensions, the signs of the depth, length and taper indicate the directions shown in the above figures.

F2 (Turn) → F2 (Line) Geometry, Line

The Line Screen is used to do linear interpolation in Feed mode. Rapid moves are possible by toggling the feedrate field. For more information on Line see the G code section on G0 or G1, Page 6-4.



Cartesian Line

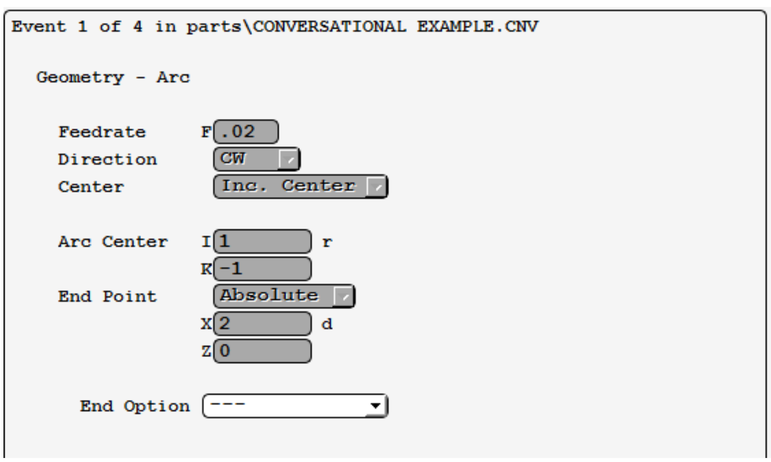


Polar Line

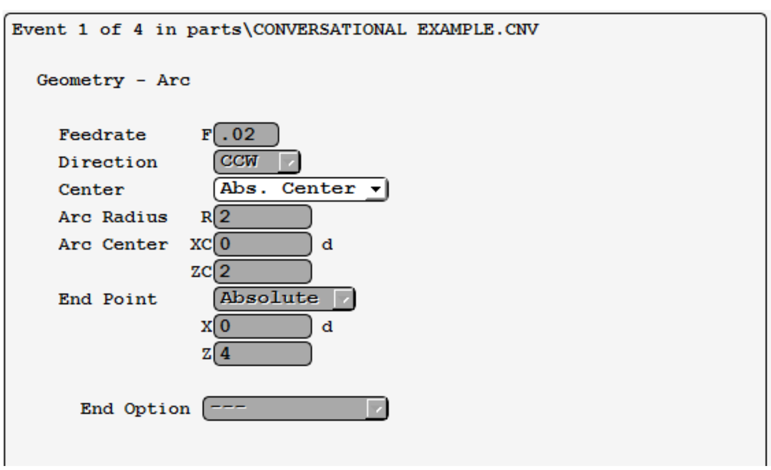
Note 1: *Chamfers are allowed for lines only. They are not allowed to or from arcs.*
Note 2: *For more information on lines see page Section Six page 6-5 on G1.*

F2 (Turn) → F3 (Arc) Geometry, Arc

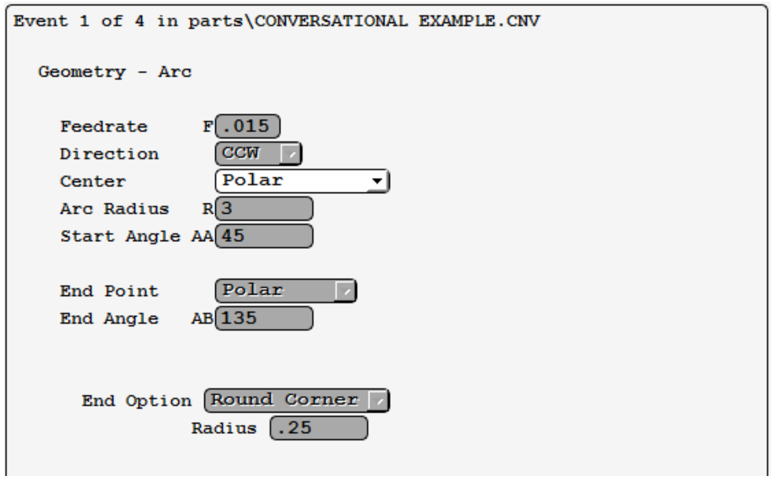
The Arc Screen is used to do circular interpolation in the feed mode. For more information on arcs see the G Code section on G2/G3 in Section Six page 6-7.



Incremental Center Arc



Absolute Center Arc



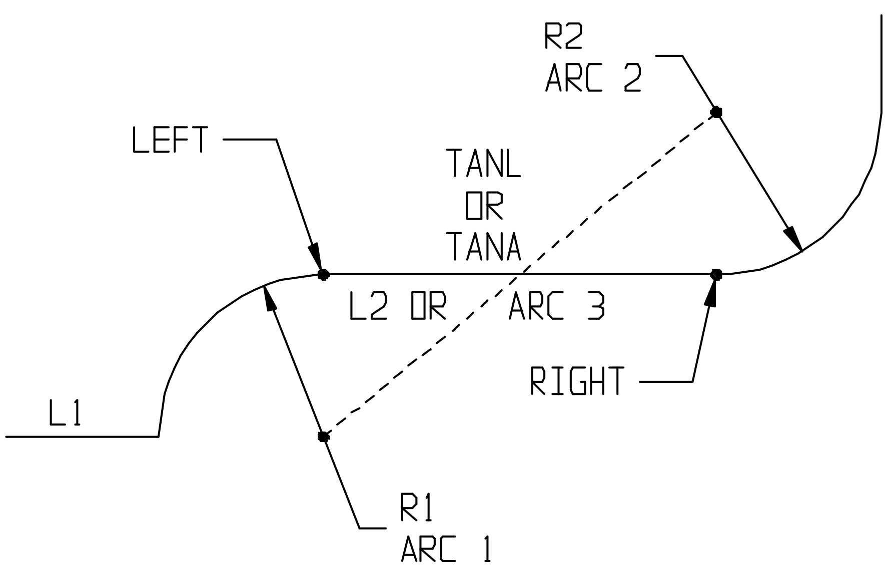
Polar Arc With Round Corner

Note: *For more information see G2/G3 in Section 6, page* 6-7

F2 (Turn) → F4 (Tangents) Geometry, Tangent Line or Arc

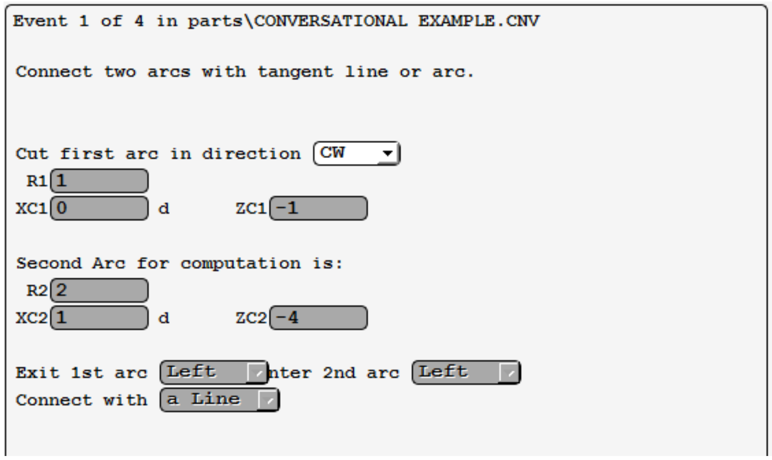
The Tangs (Tangents) screen is used to compute the intersection points necessary for a tangent arc or tangent line between two arcs. When this function is used the first arc and the tangent line or arc will be entered into the program. The second arc information will only be used for calculation purposes. This was done to enable a series of tangent lines or arcs to be programmed consecutively. In most cases, a TANL and TANA command would be followed with an arc command describing the second arc.

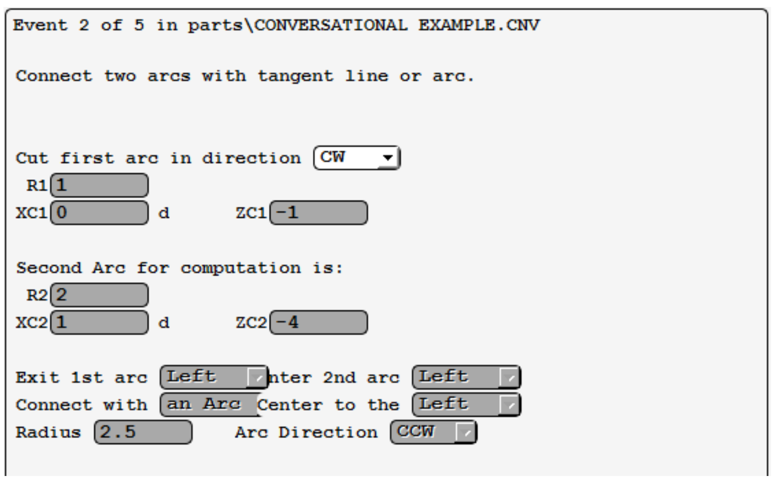
To determine the value of the right or left entries on these screens, draw a line connecting the centers of the two arcs in the direction of tool movement. Then determine if the desired points are to the right or left of this line and enter these values.



The general sequence for the above shape would be as follows:

Event 1	Line L1
Event 2	Tangent line or arc function describing arc R1 and line L2 or arc 3
Event 3	Arc R2





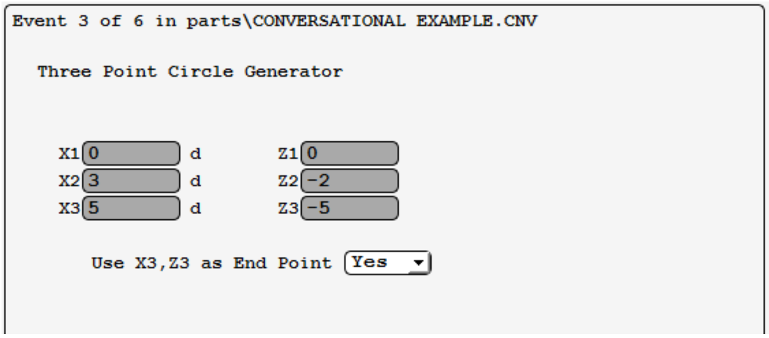
For more information on Tangent see Section Eight, page 8-9.

Note: The tangent function does not post code for the second arc. In most cases the tangent screen will be followed by the arc screen defining this second arc.

A Sample program using a Tangent Arc is located in Section 9, Sample Program 2.

F2 (Turn) → F5 (Circ-Gen) Geometry, Three Point Circle Generate

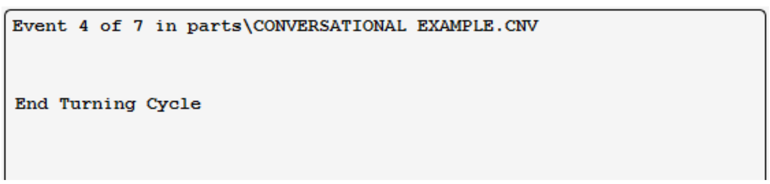
To use the Circle Generator (CGen) function, simply fill in any three points on an arc. These three points will be used to compute the center and radius of the specified arc.



For more information on the circle generate function, see Section Eight page 8-9.

Note: The Circle Generator Screen does not move to the 1st point on the arc before the arc. In most cases the circle generator screen would be preceded by a line move to this position.

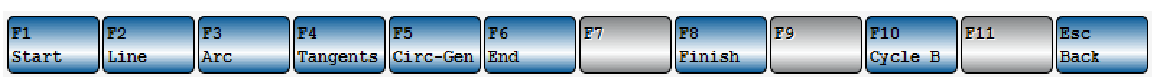
F2 (Turn) → F6 (End) End Turning Cycle



The End Cycle is a marker to denote the end of a pattern.

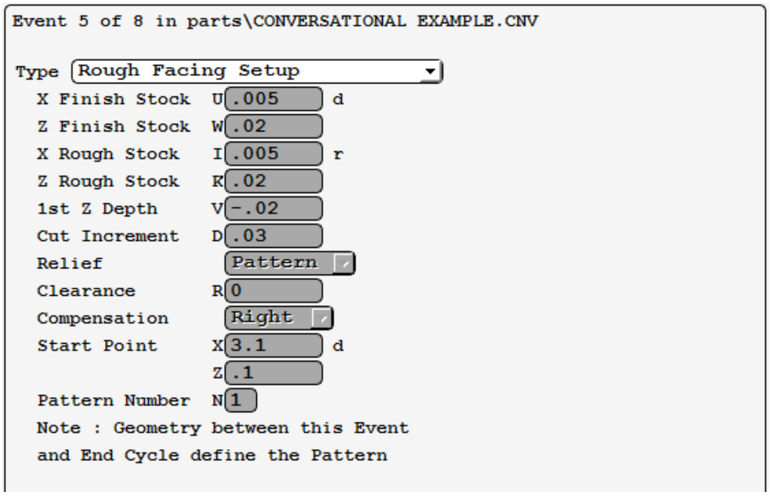
F3 (Face)

The F3 (Face) selection brings up the following soft keys.

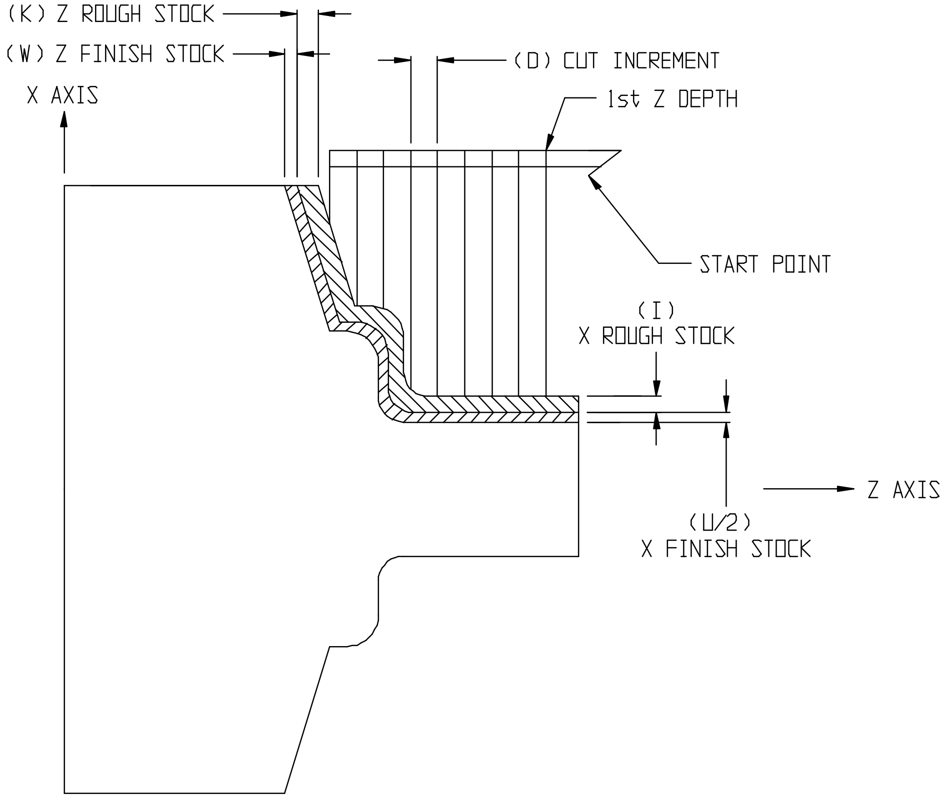


F3 (Face) → F1 (Start) Start Facing Cycle

Rough Facing Setup Selected. The Rough Facing Cycle generates all the roughing tool paths from a specified finish pattern. The finish pattern is specified by the geometry between the rough facing and the end cycle.



(U) X Finish Stock:	Distance and direction of the finish allowance in the X axis.
(W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis.
(I) X Rough Stock:	Distance and direction of the rough allowance in the X axis.
(K) Z Rough Stock:	Distance and direction of the rough allowance in the Z axis.
(V) 1st Z Depth:	The Z depth of the 1st cut in the roughing cycle. If this is not entered the 1st cut is at (Z start point) + (Z finish stock) + (Z rough stock) - (cut increment).
(D) Cut Increment:	Depth of cuts. Designated without sign. The cutting direction depends on the direction of the 1st move in the pattern.
Relief:	The type of relief made at the end of the cutting. PATTERN will cut along the pattern of the part; 45 DEGS. will retract at a 45 degree angle before retracting Z for the next cut. If the relief type is 45 DEGS. no pockets are allowed in the pattern.
(R) Clearance:	Distance of the 45 degree relief or an additional distance to pull away after the pattern relief.
Compensation:	Tool nose radius compensation. OFF, LEFT, or RIGHT.
(X,Z) Start Point:	The start point of the roughing cycle. X and Z return to the start point at the end of the cycle at a rapid feedrate.
(N) Pattern Number:	Any number between 0 and 99. The pattern number can be called by a finishing cycle later in the program.



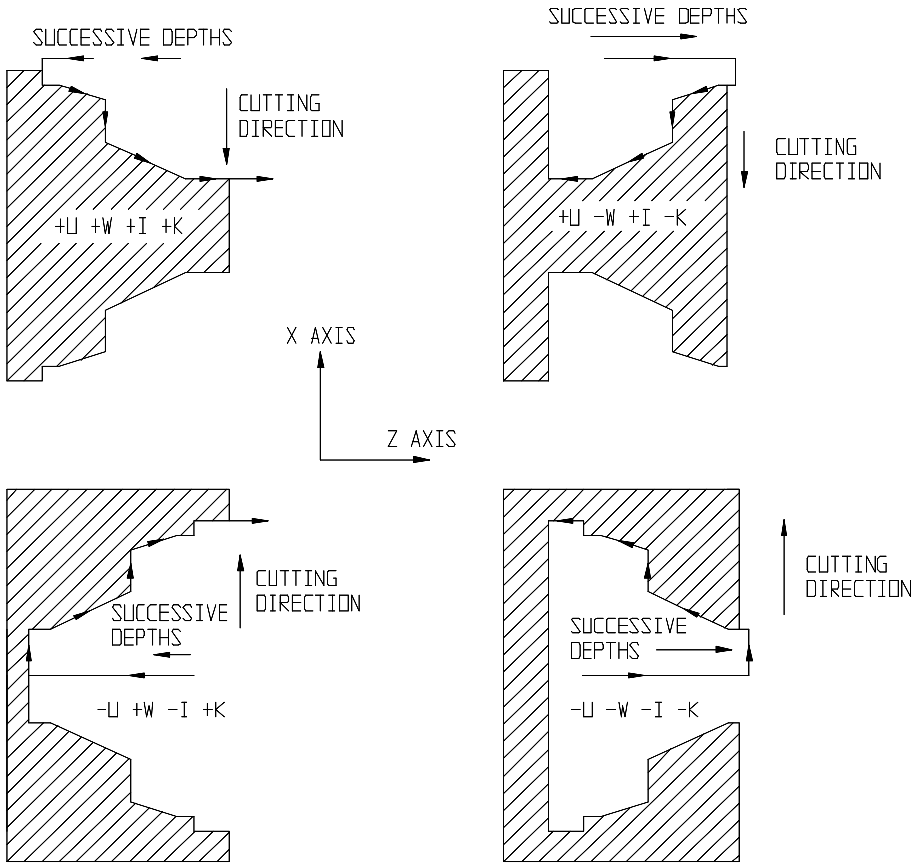
Rough cutting is done by first cutting parallel to the X axis down to the pattern, leaving the rough stock. After the cuts along the X axis are completed, a pass is made along the finish pattern leaving the finish stock.

Feeds and speeds within the pattern are ignored during the parallel cuts. Feeds and speeds are effective during the pass along the pattern.

Rapid moves within the pattern are effective when following the finish pattern.

Note: If the X rough stock and Z rough stock are both zero, the pass made along the finish pattern is skipped.

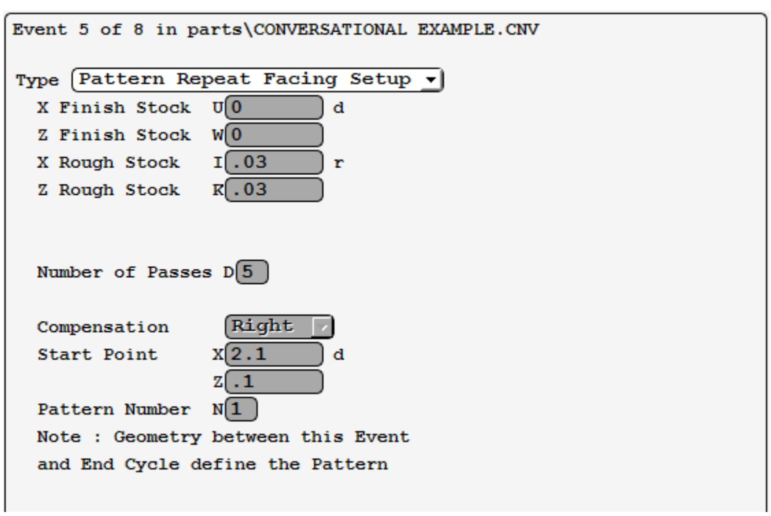
Features concerning pattern/45 degree, pockets, X stock with pockets, and X axis changing direction are the same as the rough turning cycle.



These are four patterns made by the Rough Facing cycle. Rough cutting for each of these patterns is done parallel to the X axis. The signs of U, W, I and K are shown for each pattern.

F3 (Face) → F1 (Start) Start Facing Cycle

Pattern Repeat Facing Setup selected. The Pattern Repeat Cycle repeatedly cuts a pattern incrementing the cuts down to the desired size. This cycle is used to efficiently cut a part whose shape is already defined (forged or cast).



Event 5 of 8 in parts\CONVERSATIONAL EXAMPLE.CNV

Type Pattern Repeat Facing Setup

X Finish Stock U 0 d

Z Finish Stock W 0

X Rough Stock I .03 r

Z Rough Stock K .03

Number of Passes D 5

Compensation Right

Start Point X 2.1 d

Z .1

Pattern Number N 1

Note : Geometry between this Event

and End Cycle define the Pattern

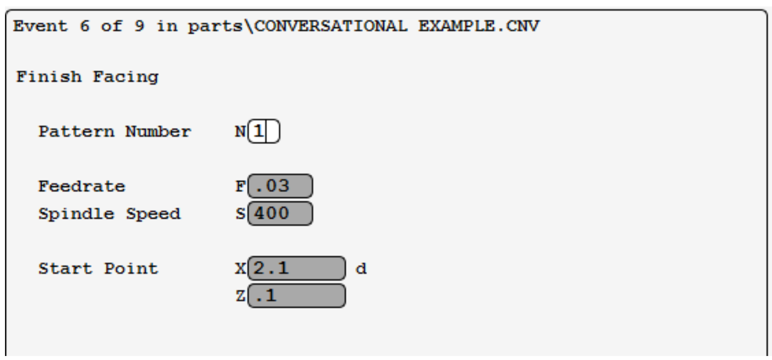
(U) X Finish Stock:	Distance and direction of the finish allowance in the X axis.
(W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis.
(I) X Rough Stock:	Distance and direction of the rough allowance in the X axis.
(K) Z Rough Stock:	Distance and direction of the rough allowance in the Z axis.
(D) Number of Passes:	The number of passes to cut the rough stock.
Compensation:	Tool nose radius compensation. OFF, LEFT, or RIGHT.
(X,Z) Start Point:	The start point of the pattern repeat cycle. X and Z return to the start point at the end of cycle at a rapid feedrate.
(N) Pattern Number:	Any number between 0 and 99. The pattern number can be called by a finishing cycle later in the program.

There are no differences between the turning pattern repeat cycle and the facing pattern repeat cycle.

Code generated from the Facing Pattern Repeat is identical to that of the Turning Pattern Repeat.

F3 (Face) → F8 (Finish) Finish Facing

After rough cutting with the Rough Facing Cycle or Pattern Repeat Cycle, the Finishing Cycle can be used to remove the finish stock.



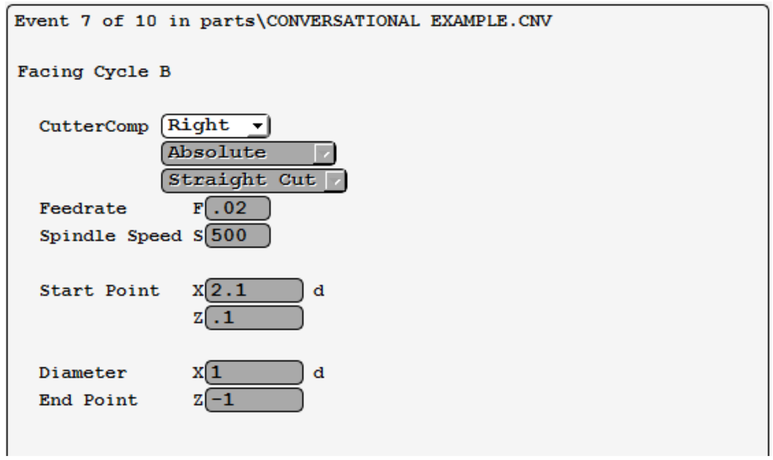
(N) Pattern Number:	Any pattern defined by a rough facing or pattern repeat cycle.
(F) Feedrate:	The feedrate to cut the finish pattern. Any feedrates within the pattern will be effective.
(S) Spindle Speed:	The spindle speed to cut the finish pattern. Any spindle speed within the pattern will be effective.
(X,Z) Start Point:	The start point of the finish cycle. X and Z return to the start point at the end of the cycle at a rapid feedrate.

Note: The only difference between the Finish Facing Cycle and the Finish Turning Cycle is that the Facing Cycle retracts the Z before the X axis.

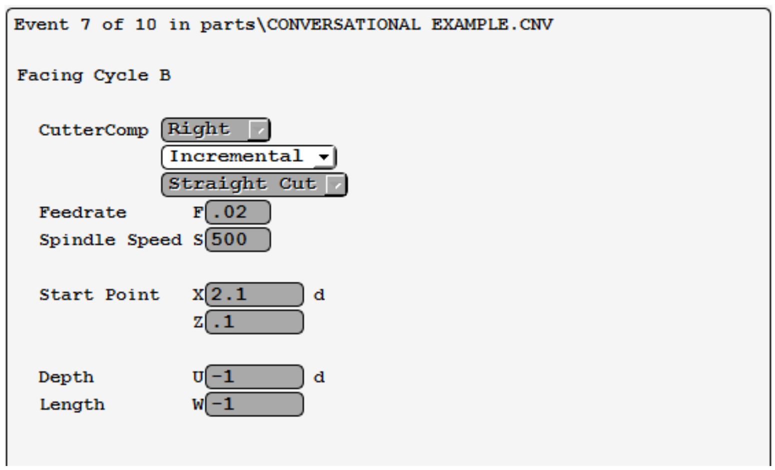
An example program using a Rough Facing Cycle with a Finish Cycle is located in Section 9, Sample Program 3.

F3 (Face) → F10 (Cycle B) Facing Cycle B

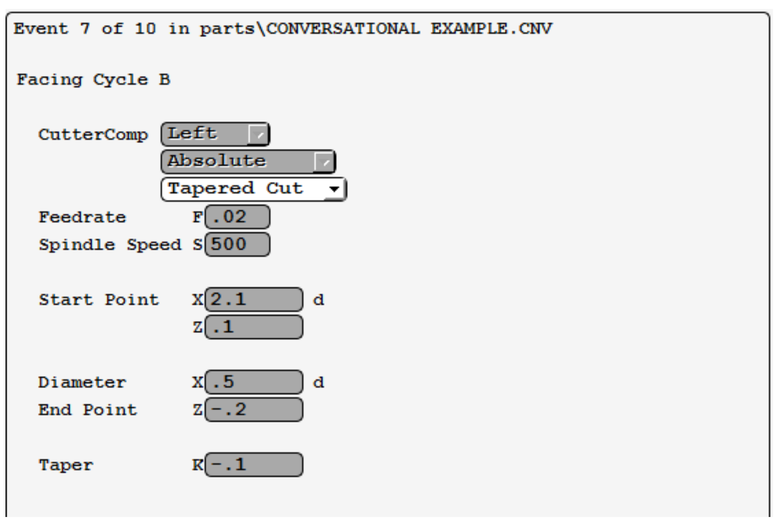
Cycle B is used to make a single facing cut. The cycle rapids to the Z cutting depth, makes the cut, retracts Z axis and rapids back to the start point. Tapered cuts are allowed.



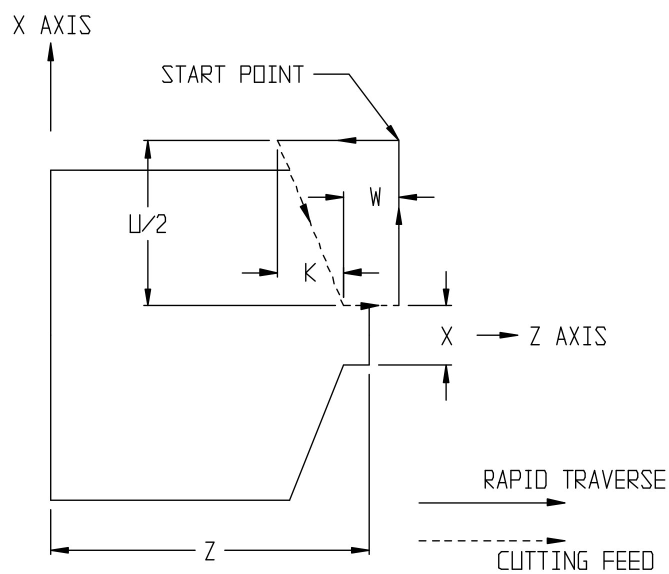
-OR-



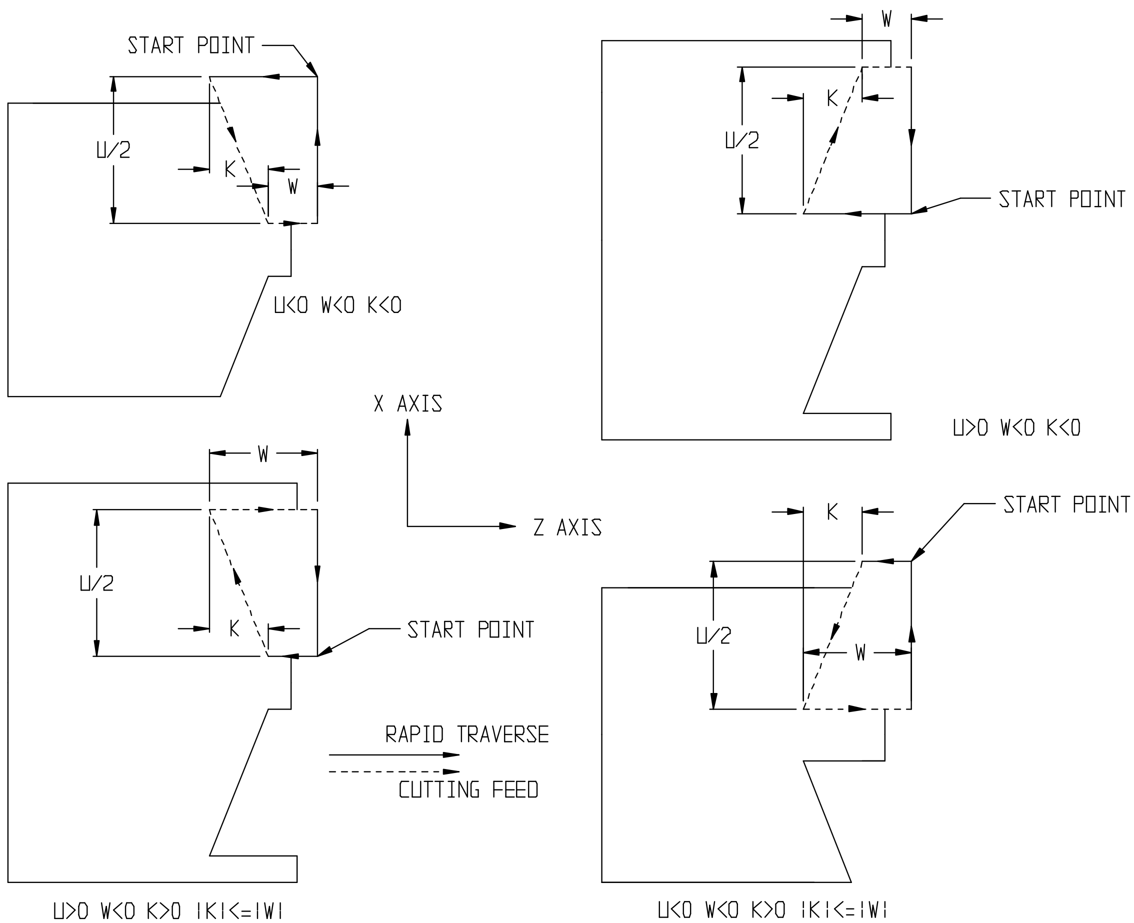
-OR-



Compensation:	Tool nose radius compensation. OFF, LEFT or RIGHT
Absolute or Incremental:	Toggles between absolute dimensions (diameter, end point) and incremental dimensions (depth and length).
(F) Feedrate:	The feedrate to face the part.
(S) Spindle Speed:	The spindle speed to face the part.
(X,Z) Start Point:	The start point of Cycle B. X and Z return to the start point at the end of the cycle at a rapid feedrate.
(X) Diameter:	The diameter to face to.
(X) End Point:	The end point of the cut.
(U) Depth:	The incremental X distance and direction from the start point to the diameter to face to.
(W) Length:	The length and direction of the cut in Z from the start point.
(I) Taper:	The amount and direction of the taper to face.



Tapered Cut For Cycle B Absolute and Incremental Dimensions



When using Cycle B with incremental dimensions, the signs of the depth, length and taper indicate the direction shown in the figures above.

Note: Lines, Arcs, Tangents and 3 Point Circle Generators in Facing are identical to the Lines, Arcs, Tangents and 3 Point Circle Generators in the Turning Section.

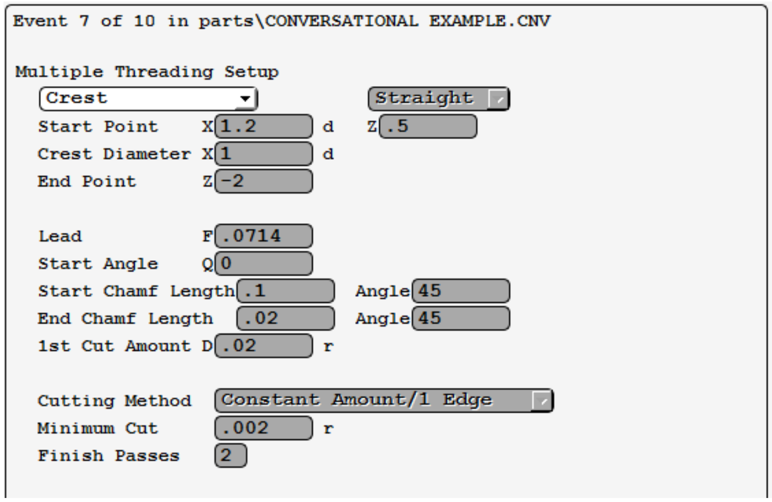
F4 (Thread)

The F4 (Thread) selection brings up the following soft keys for threading:

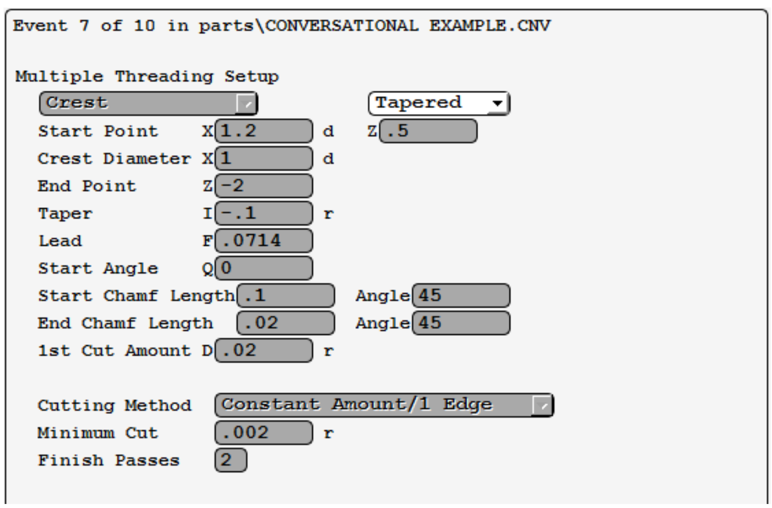
F1 Multiple	F2 Cycle 1	F3	F4	F5 Line	F6	F7	F8	F9	F10	F11	Esc Back

F4 (Thread) → F1 (Multiple) Multiple Threading Setup

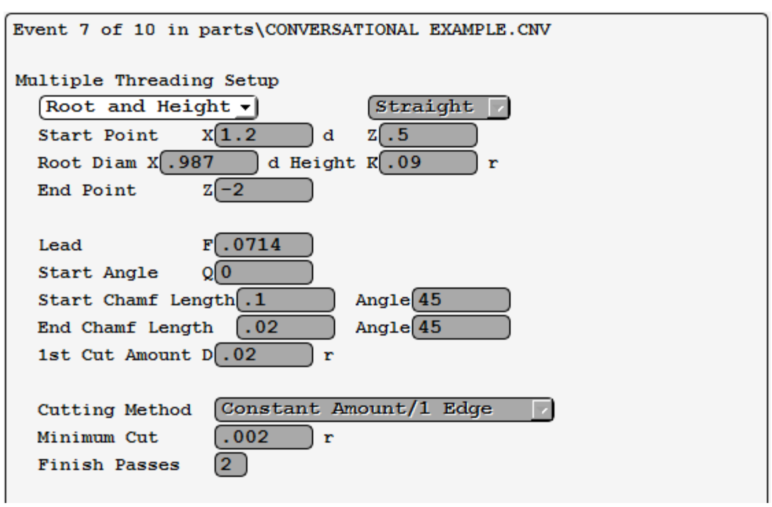
The Multiple Thread Cutting cycle is used to make multiple passes on a thread using one of four different cutting methods.



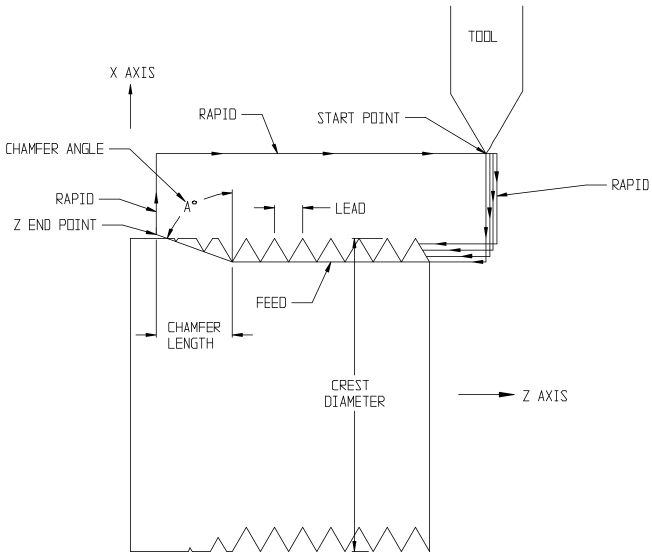
- OR-



- OR-

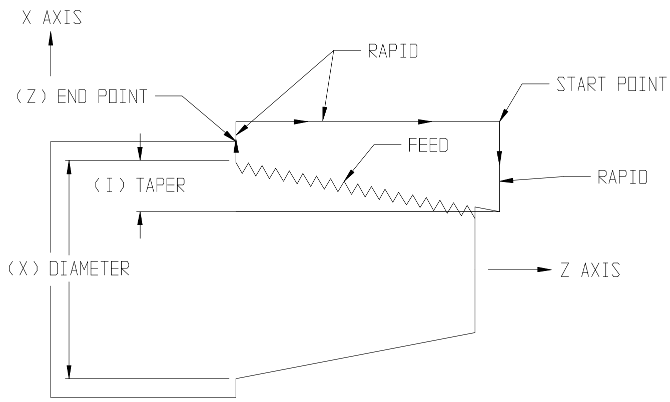


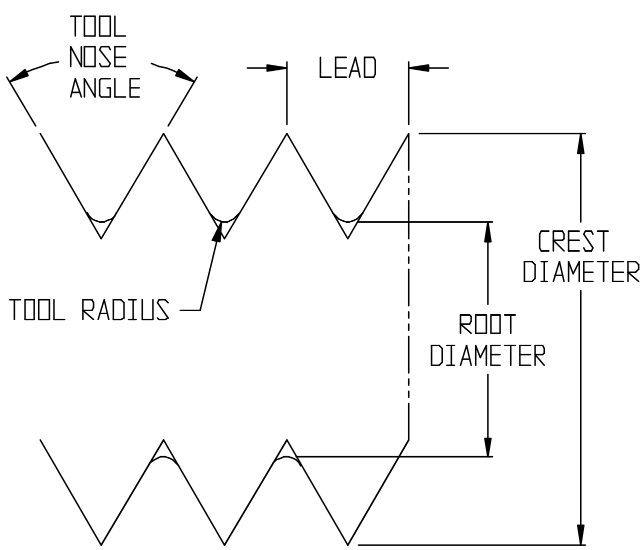
Crest or Root and Height:	If Crest is selected, only the crest diameter is entered and the root and height are calculated from the crest diameter and tool nose angle. The crest selection is useful for sharp V threads. If root and height are selected those two items must be entered. This is used for threads other than sharp V. (See items 4, 4a and 4b.)
Straight or Tapered:	If Straight is selected no taper is entered. If Taper is selected then a taper must be entered.
(X,Z) Start Point:	The start point of threading cycle. X and Z return to the start point after each pass on the thread at a rapid feedrate.
(X(Crest Diameter:	The major diameter for external threads or the minor diameter for internal threads.
(X) Root Diameter:	The minor diameter for external threads or the major diameter for internal threads.
(K) Height:	Height of the thread specified as a radius value.
(Z) End Point:	The end of the thread.
(I) Taper:	The amount and direction of the taper to turn. (This is a radius value.)
(F) Lead:	Lead of the thread.
(Q) Start Angle:	Shift angle of the thread start angle. Used for multiple threads, i.e. shift 180º, 120º etc. 0º thru 360º are allowed.
Chamfer Length:	Length of the chamfer on the end of the thread.
Chamfer Angle:	The angle of the chamfer on the end of the thread. Angles 0º thru 90º will work for all threads, where 0º is straight out and 90º is no chamfer for a straight thread.
(D) First Cut Amount:	Cutting depth of the first cut. (This is a radius value.)
Minimum Cut:	When the cutting amount is less than the minimum cut, the cutting amount is clamped at the finishing allowance. (This is a radius value).
Finish Passes:	The last pass can be repeated a number of times. If the finish pass is 0 or 1 the last pass will be made one time.
Cutting Method:	Constant Amount/1 Edge - Cutting amount constant, single edge cutting.
	Constant Amount/Both Edges - Cutting amount constant, both edges cutting.
	Constant Depth/1 Edge – Cutting amount constant, single edge cutting.
	Constant Depth/Both Edges - Cutting depth constant, both edges cutting.



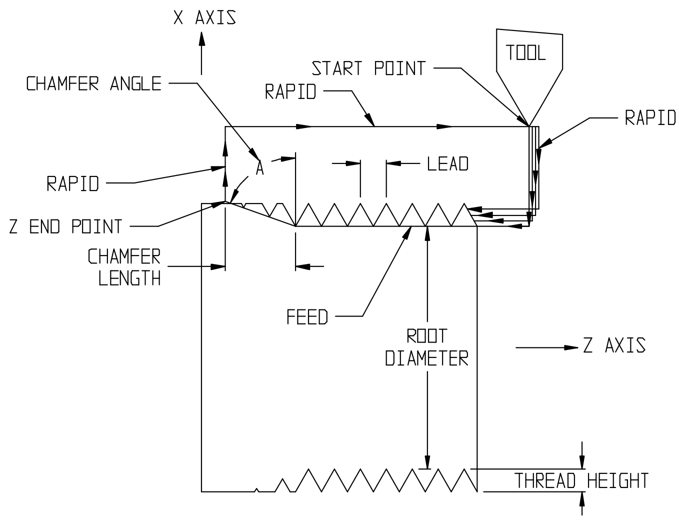
Inside or outside threads are determined by the start point relative to the diameter. The direction of the taper is determined by the sign of I.

The tapers are the same as those shown in the cycle A.



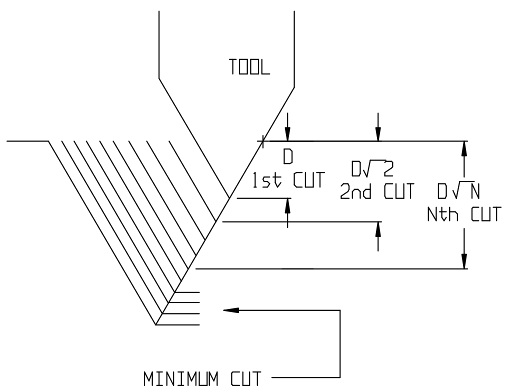


When the crest option is selected, the root diameter is calculated from the crest diameters, the tool nose angle, the tool nose radius and the lead.

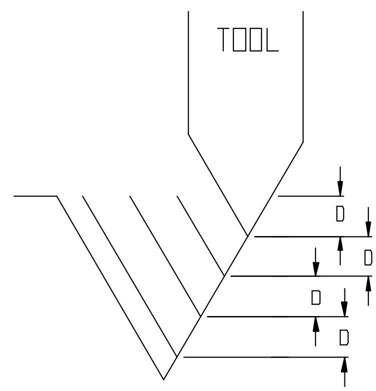


When the root and height option is selected the crest diameter is calculated from the root diameter, the thread height and the lead.

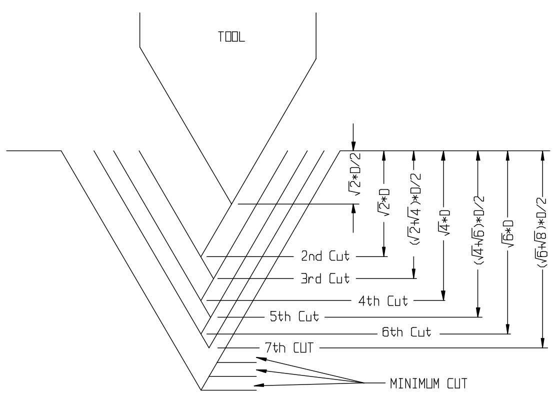
The four cutting methods are shown below:



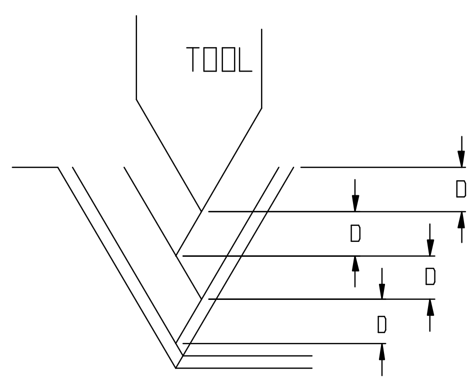
Constant Depth/1 Edge



Constant Amount/1 Edge



Constant Amount/Both Edges

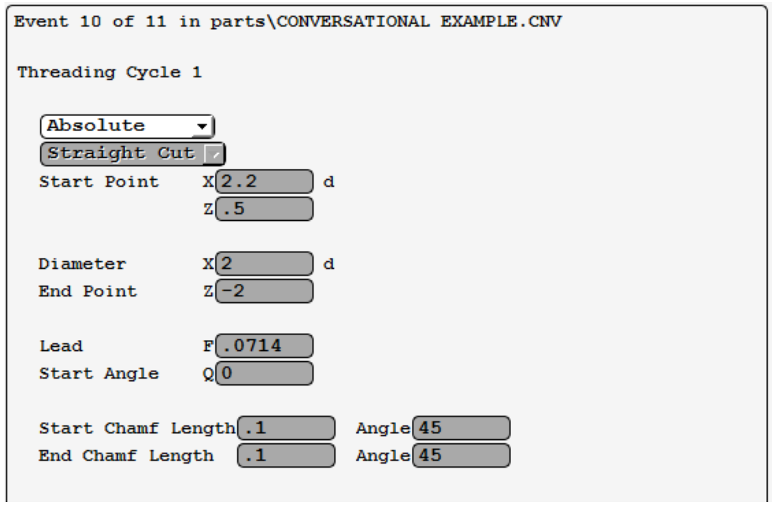


Constant Depth/Both Edges

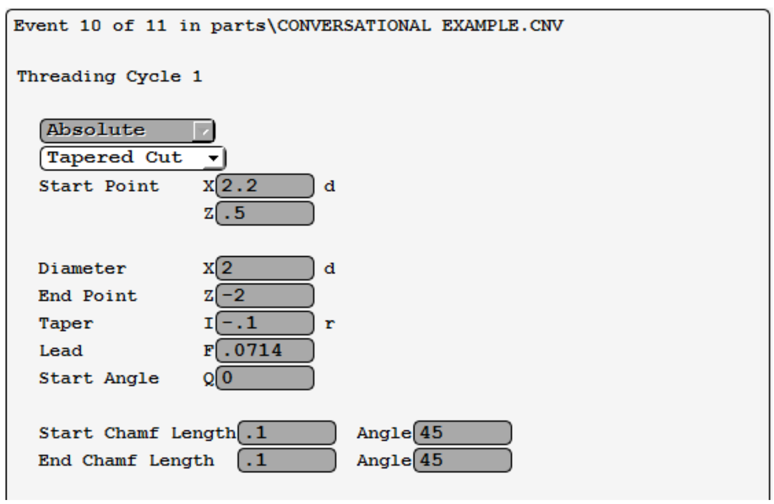
If the cutting depths using constant amount one edge, or constant amount both edges, becomes less than the finish allowance, the cutting depths are clamped at the finish allowance.

F4 (Thread) → F2 (Cycle 1) Threading Cycle 1

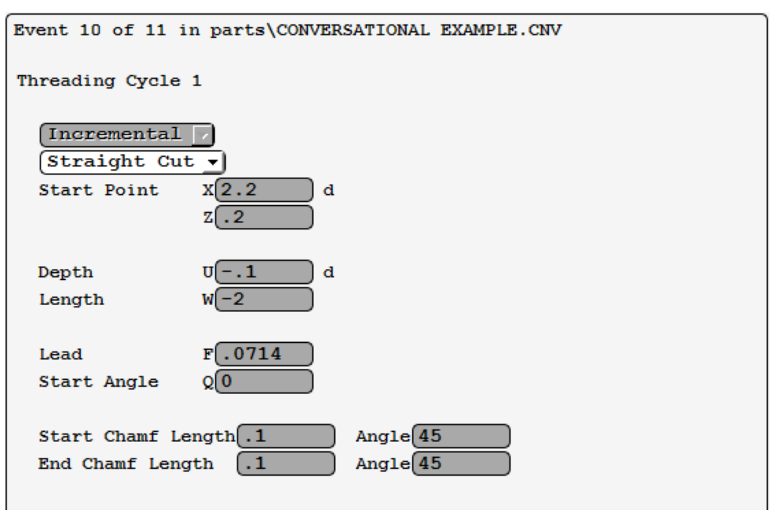
Cycle 1 is used to make a single threading cut. The cycle rapids to the X cutting depth, makes the threading cut, retracts X axis and rapids back to the start point. Tapered threads are allowed. Threads must be cut in inches per revolutions.



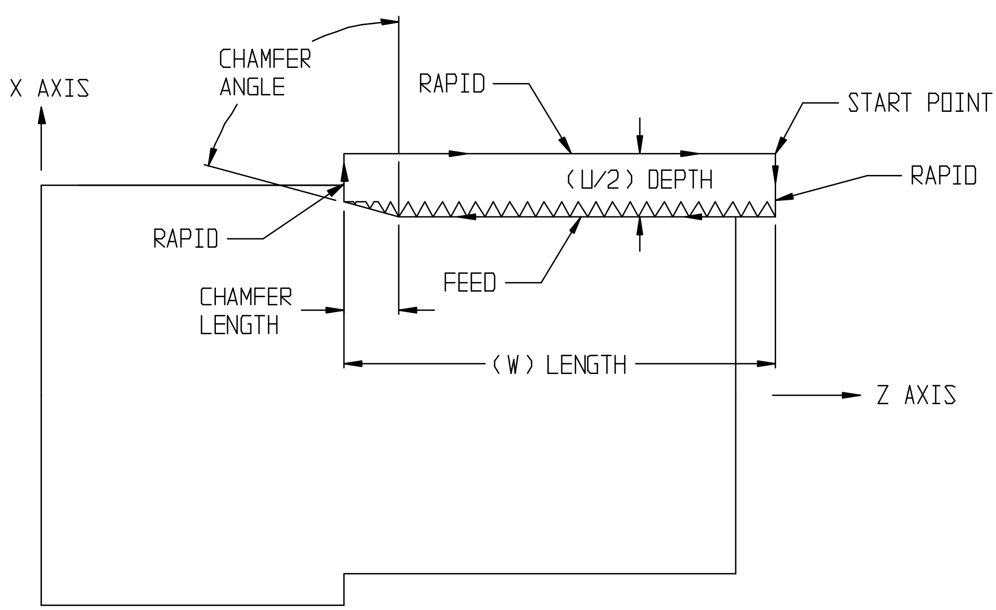
-OR-



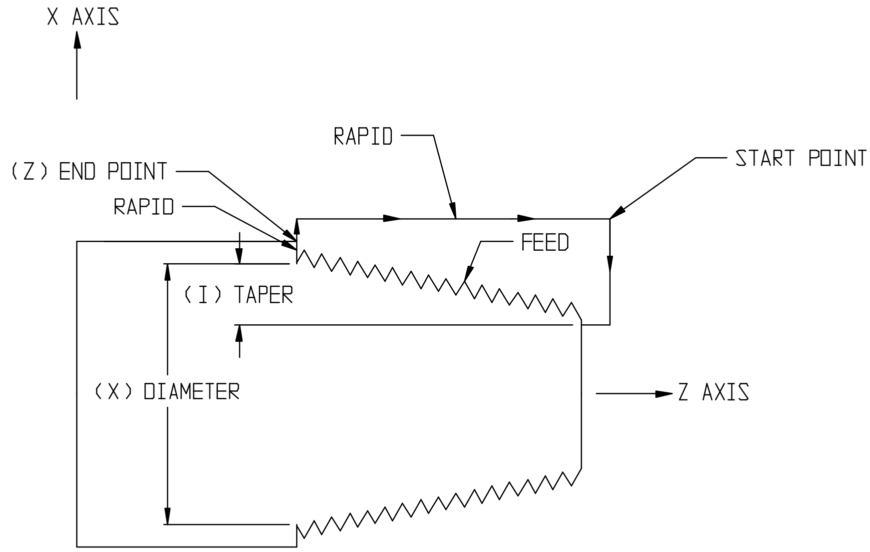
-OR-



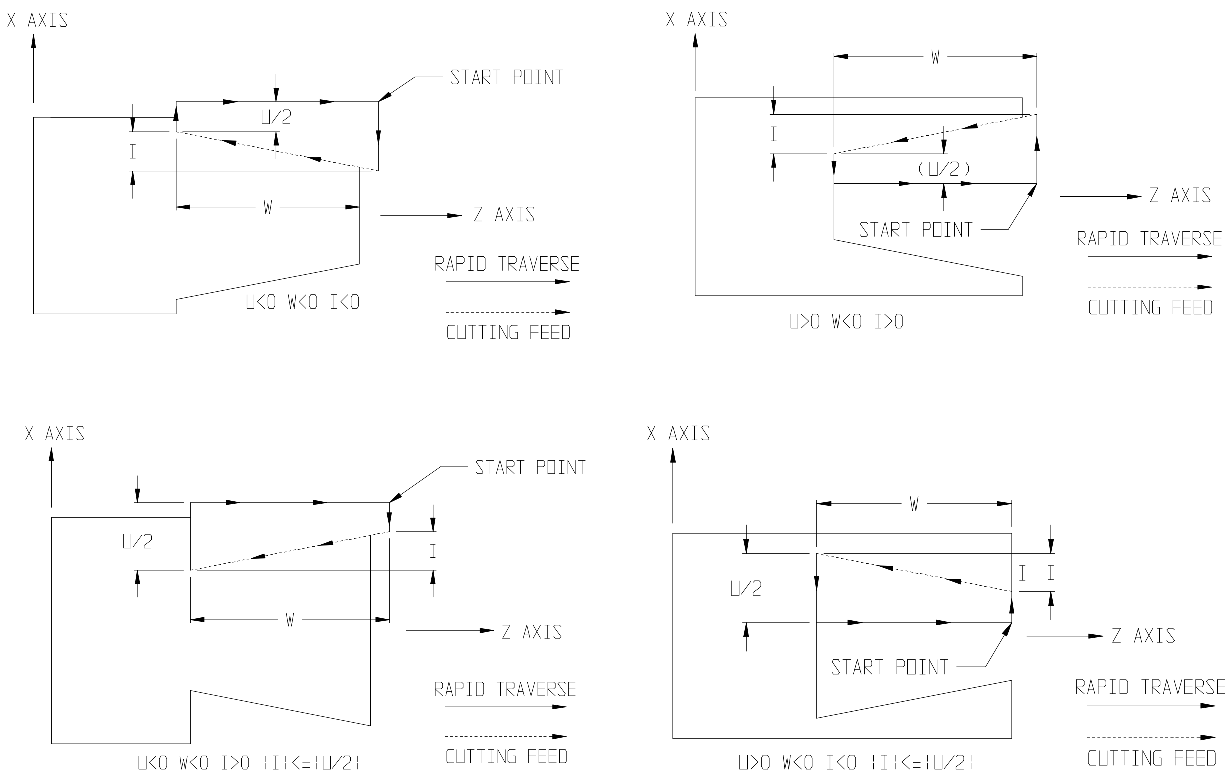
ABSOLUTE or INCREMENTAL:	Toggles between absolute dimensions (diameter, end point) and incremental dimensions (depth and length).
(X,Z) Start Point:	The start point of Cycle 1. X and Z return to the start point at the end of the cycle at a rapid feedrate.
(X) Diameter:	The root diameter to thread.
(U) Depth:	The incremental X distance and direction from the start point to the diameter to thread.
(Z) End Point:	The end point of the thread.
(W) Length:	The length and direction of the thread in Z from the start point.
(I) Taper:	The amount and direction of the taper to thread.
(F) Lead:	Lead of the thread.
(Q) Start Angle:	Shift angle of the thread start angle. Used for multiple threads, i.e. shift 180º, 120º etc. 0º thru 360º are allowed.
Chamfer Length:	Length of the chamfer on the end of the thread.
Chamfer Angle:	The angle of the chamfer on the end of the thread. Angles 0º thru 90º will work for all threads, where 0º is straight out and 90º is no chamfer for a straight thread.



Straight Cutting Using Cycle 1 with Incremental Dimensions



Tapered Cut Using Cycle 1 with Incremental Dimensions

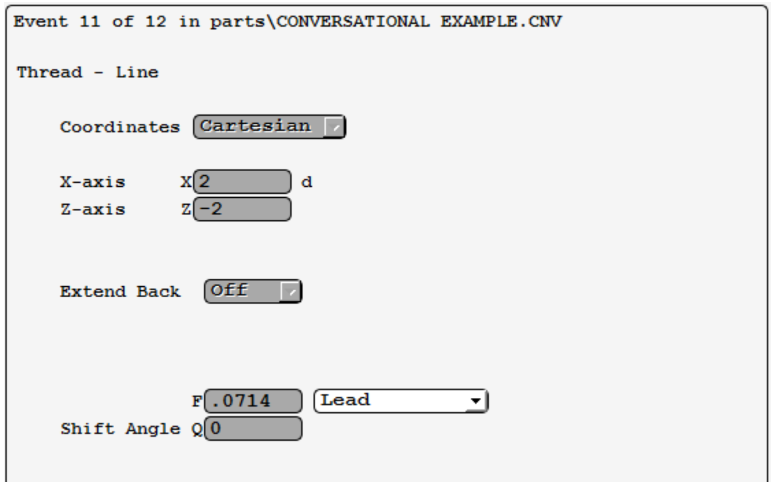


When using Cycle 1 with Incremental Dimensions, the signs of the depth, length and taper indicate the directions shown in the above figures.

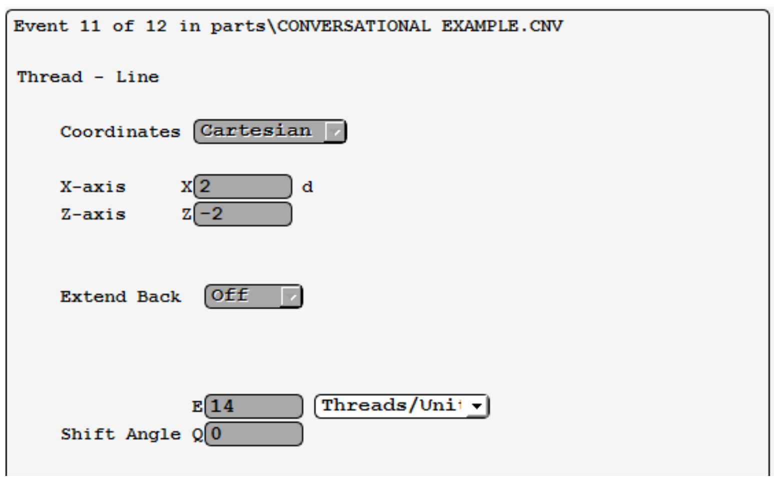
F4 (Thread) → F5 (Line) Thread - Line

Thread Lines are used for a single thread. The start of the thread is the initial position of the machine.

The line defined by the thread line is identical to the lines used in turning or facing with the additional threading information for the line.

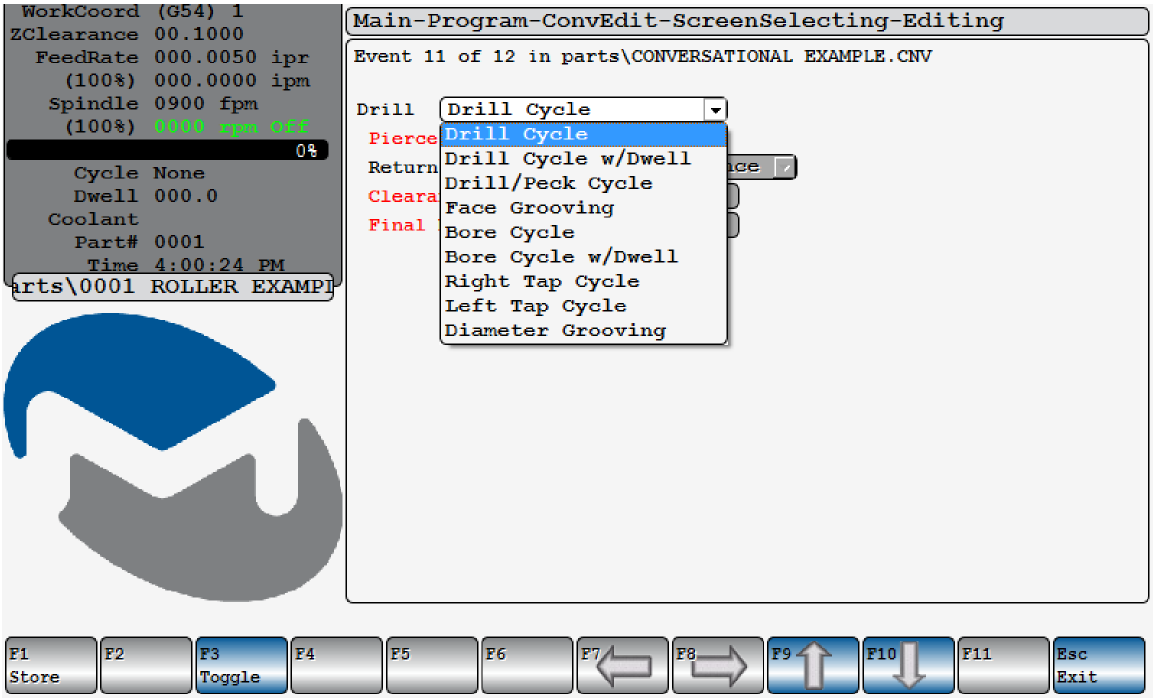


-OR-



F5 (Drill) Drilling, Taping, Boring, Grooving Cycles

F5 (Drill) → Select Operation Type



Pressing F3 (Toggle) will cycle through the various canned cycles that are available. Detailed information about each canned cycle is located in the G-Code section of this manual.

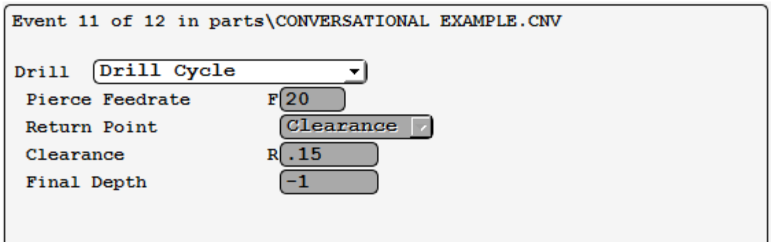
There are a variety of different drilling cycles available in the conversational programming system. The different cycles are available by toggling the first field on the drill screen. A position move typically to the center line of the spindle (X0) should be commanded before the drill cycle. The Drill Cycle drills a hole at the current position.

The position move must be made to the location of hole to be drilled, tapped or start of the groove before the cycle is called. None of the drill cycles are modal; they are one shot cycles. Each cycle is detailed below.

Additional Grooving Cycles are available on the control under F6 (Groove) covered next in this section of the manual.

Drill Cycle –

The drill cycle rapids to the Z clearance plane, feeds to the Z depth, and then rapids to the clearance plane or initial position. The conversational screen appears as follows:



Pierce Feedrate: Z feedrate for drilling the hole.

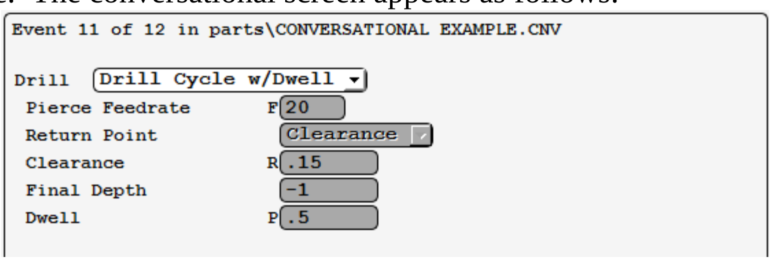
Return point: CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.

Clearance: Z rapids to the clearance at the beginning drill cycle.

Final Z Depth: The depth to drill the hole to.

Drill Cycle W/Dwell –

The Drill Dwell Cycle is identical to the Drill Cycle with the addition of a Dwell at the bottom of the hole. The conversational screen appears as follows:



Pierce Feedrate: Z feedrate for drilling the hole.

Return Point: CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.

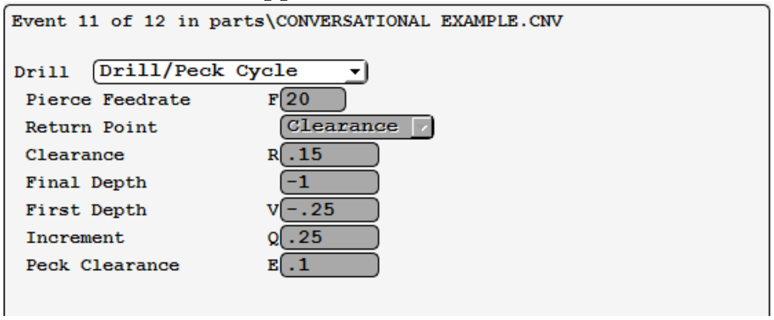
Clearance: Z rapids to the clearance at the beginning of the drill cycle.

Final Z Depth: The depth to drill the hole.

Dwell: Time in seconds to dwell at the bottom of the hole.

Drill/Peck Cycle –

The Drill Peck Cycle is useful for drilling deep holes. It drills to successive depths, then rapids out of the hole and back to a peck clearance for each depth. The sign or direction of the Z increment and peck clearance assumes drilling is done from tail stock towards the chuck.
The conversational screen appears as follows:



Pierce Feedrate:	Z Feedrate for drilling the hole.
Return Point:	CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the drill cycle starts.
Clearance:	Z rapids back to the clearance at the beginning of the drill cycle.
Final Z Depth:	The depth to drill the hole to.
First Z Depth:	First peck depth.
Z Increment:	Unsigned amount to drill in each successive peck.
Peck Clearance:	Unsigned distance to rapid to above the previous depth.

Face Grooving Cycle –

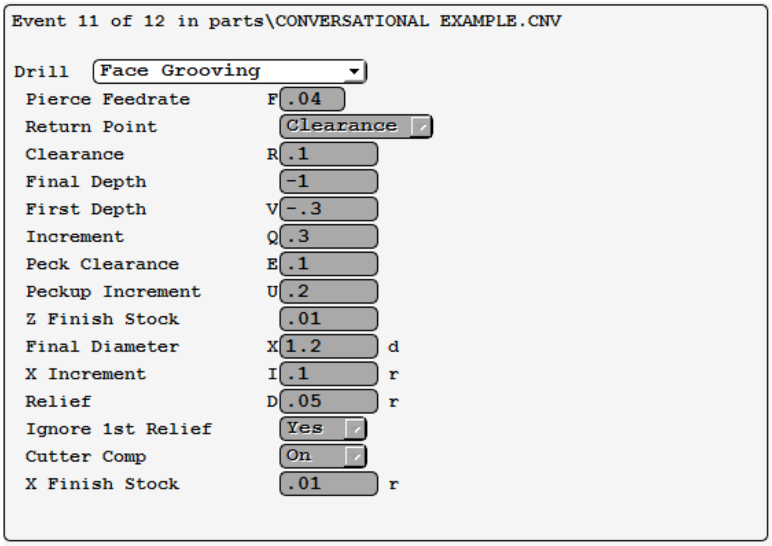
The Face Grooving Cycle can be used as fast peck drilling cycle by leaving the last three fields blank. The fast peck cycle is similar to the peck cycle except the fast cycle does not pull out to the clearance; it pulls out the peck increment. There are four possible patterns going tail stock to chuck or vice versa, and outside to inside or vice versa.

The sign (or direction) of the Z increment, peck clearance and peck up increment are determined by the direction of the final Z depth to the initial Z position.

The sign (or direction) of the X increment and relief are determined by the direction of the final diameter to the initial X position.

If the final diameter and the X increment are blank, the sign of the relief is valid and it determines its direction.

The conversational screen appears as follows:



Pierce Feedrate:	Z feedrate for grooving the slot.
Return Point:	CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the grooving cycle started.
Clearance:	Z rapids to the clearance at the beginning of the grooving cycle.
Final Z Depth:	The depth to slot the groove to.
First Z Depth:	First peck depth.
Z Increment:	Unsigned amount to groove in each successive peck.
Peck Clearance:	Unsigned distance to rapid to above the previous depth.
Peckup Increment:	Unsigned distance to rapid up each peck.
Z Finish Stock:	The unsigned amount of stock to leave on the bottom of the groove before making a final pass. *(See Note 1 below.)*
Final Diameter:	The diameter to cut the groove to. The starting diameter is the X dimension the tool is at when the grooving cycle started.
X Increment:	The unsigned amount to step over along the face. (This is a radius value.)
Relief:	Amount to back X away at the bottom of the groove before retracting X to the Z clearance.
X Finish Stock:	The unsigned amount of stock to leave on each side of the groove before making a final pass. (This is radius value.)
Ignore 1st Relief	Yes will skip the X axis retract to avoid dragging out of the groove on the first pass. No will do the relief on all of the retracts.
Cutter Comp	Yes will adjust the groove size based on the tool radius. The software checks for specific tool types and will give an error if the tool type is not valid. No requires that the dimensions of the groove account for the tool width.

Note 1: If both X and Z finish stocks are zero, then no finish pass is made. If either X or Z stock is non-zero, a finish pass will be made as follows:

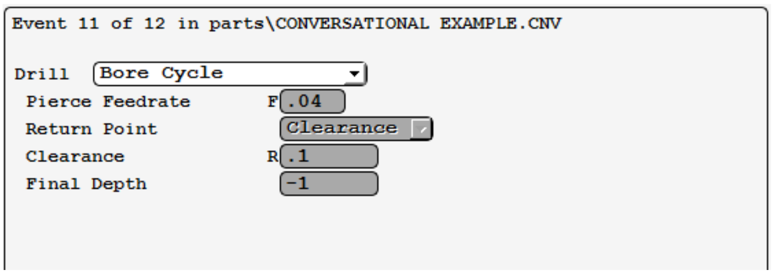
A. *Remove the X finish stock at the final diameter side of the groove to the bottom of the groove.*
B. *Move to the center of the groove along the bottom.*
C. *Retract out of the groove.*
D. *Move to the starting diameter side of the groove.*
E. *Remove the X finish stock at the starting diameter side of the groove to the bottom of the groove.*
F. *Remove the Z stock along the bottom of the groove to the final diameter side.*
G. *Retract out of the groove.*

Note 2: A position move should be made prior to the Face Grooving Cycle so that the X axis is positioned to the starting side of the groove.

Additional Grooving features are available under F6 (Groove)

Bore Cycle –

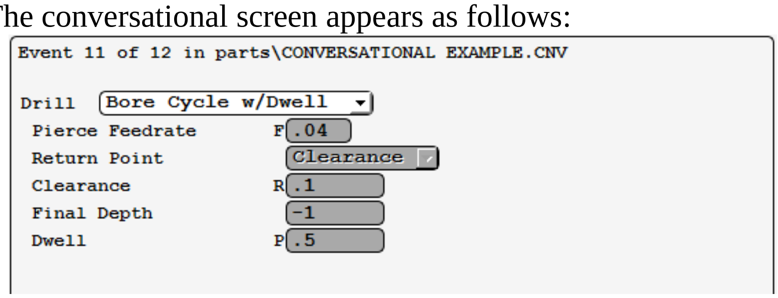
The Bore Cycle Feeds to the Z depth and feeds back to the clearance. The conversational screen appears as follows:



Pierce Feedrate:	Feedrate for drilling the hole.
Return Point:	CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.
Clearance:	Z rapids to the clearance at the beginning of the bore cycle.
Final Z Depth:	The depth to bore the hole to.

Bore Cycle w/Dwell –

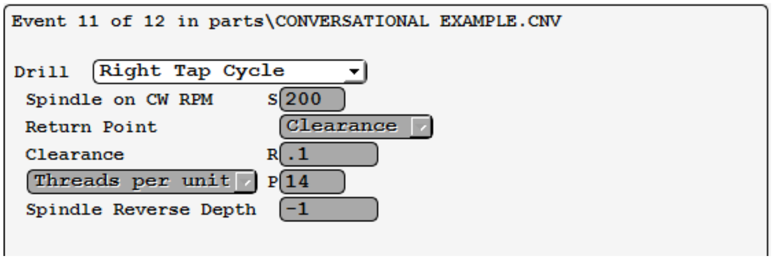
The Bore/Dwell Cycle is the same as the bore cycle, but a dwell is performed at the bottom of the hole. The conversational screen appears as follows:



Pierce Feedrate:	Z feedrate for drilling the hole.
Return Point:	CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the bore/dwell cycle started.
Clearance:	Z rapids to the clearance at the beginning of the bore/dwell cycle.
Final Z Depth:	The hole will be drilled to this depth.
Dwell:	Time in seconds to dwell at the bottom of the hole.

Right Tap Cycle –

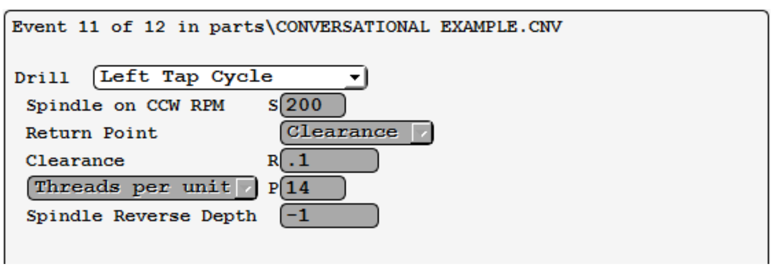
The Right Tap Cycle always runs in FEED/REV and REVS/MINUTE. The spindle is turned on CW at the start of the cycle and reverses direction (CCW) at the spindle reverse depth.
The conversational screen appears as follows:



Spindle on CW RPM:	Revs per min to tap at. *Note: The spindle will always be turned on clockwise even if this field is empty.*
Threads per Unit:	The lead of the tap.
Return Point:	CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the tap cycle started.
Clearance:	Z rapids to the clearance at the beginning of the tap cycle.
Spindle Reverse Depth:	The spindle reverse direction at this depth. It may coast some and cause the tap to go a little deeper.

Left Tap Cycle –

The Left Tap Cycle is similar to the right tap except the left tap cycle turns the spindle on counterclockwise at the start of the cycle and reverses direction (CW) at the spindle reverse depth.
The conversational screen appears as follows:



Spindle on CCW RPM:	Revs per minute to tap at. *Note: The spindle will always be turned on counterclockwise even if this field is empty.*
Threads per Unit:	Threads per inch or threads per millimeter.
Return Point:	CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the Z dimension the tool was at before the tap cycle started.
Clearance:	Z rapids to the clearance at the beginning of the tap cycle.
Spindle Reverse Depth:	The spindle reverse direction at this depth. It may coast some and cause the tap to go a little deeper.

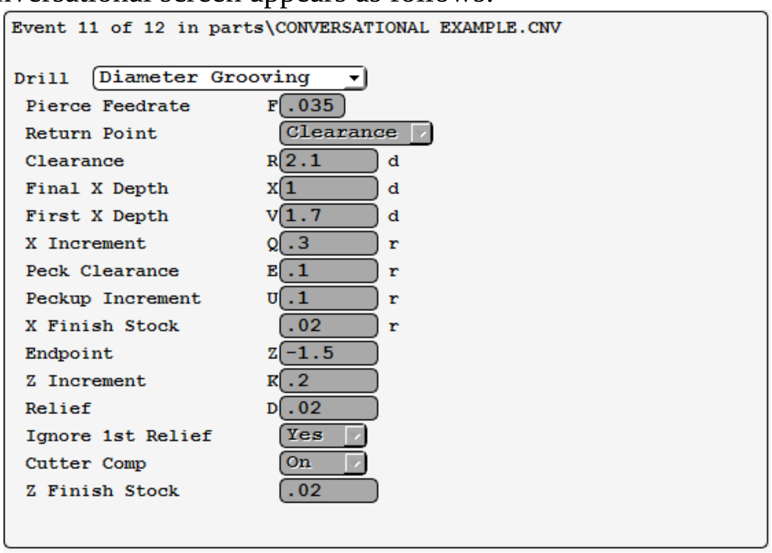
Diameter Grooving Cycle –

The Diameter Grooving Cycle is similar to the face grooving cycle except the fast peck cycle is used on the diameter. If the last three fields are omitted, a single high speed peck cycle is done. There are four possible patterns going tail stock to chuck or vice versa, and outside towards center or vice versa.

The sign (or direction) of the X increment, peck clearance, and peck-up increment are determined by the direction of the final X depth to the initial X diameter.

The sign (or direction) of the Z increment and relief are determined by the Z end point to the initial Z position. The Z end point and Z increment are blank. The sign of the relief determines its direction.

The conversational screen appears as follows:



Pierce Feedrate:	X Feedrate for grooving the slot.
Return Point:	CLEARANCE means the tool will rapid back to the clearance plane at the end of the cycle. INITIAL means the tool will rapid back to the X dimension the tool was at before the grooving cycle started.
Clearance:	X rapids to the clearance at the beginning of the grooving cycle. (This is a diameter value.)
Final X Depth:	The diameter to groove the slot to.
First X Depth:	First peck diameter.
X Increment:	Unsigned diameter to groove in each successive peck. (This is a radius value.)
Peck Clearance:	Unsigned distance to rapid up after each peck. (This is a radius value.)
PeckUp Increment:	Unsigned distance to rapid up after each peck. (This is a radius value.)
X Finish Stock:	The unsigned amount of stock to leave at the bottom of the groove before making a finish pass. (This is a radius value.) *(See Note 1 below.)*
Endpoint:	The Z dimension to cut the groove to. The start of the groove is the Z dimension the tool is at when the grooving cycle started.
Z Increment:	The unsigned amount to step over.

Relief:	Amount to back Z away at the bottom of the groove before retracting to the X clearance.
Ignore first Relief	Yes will skip the X axis retract to avoid dragging out of the groove on the first pass. No will do the relief on all of the retracts.
Cutter Comp	Yes will adjust the groove size based on the tool radius. The software checks for specific tool types and will give an error if the tool type is not valid. No requires that the dimensions of the groove account for the tool width.
Z Finish Stock:	The unsigned amount of stock to leave on each side of the groove before making a finish pass.

Note 1: If both X and Z finish stocks are zero, no finish pass is made. If either X or Z finish stock is zero, a finish pass will be made as follows:

A. The Z finish stock is removed on the ending side of the groove to the final X depth.

B. Move to the center of the groove along the bottom.

C. Retract X out of the groove.

D. Move to the Z start of the groove.

E. Remove the Z finish stock at the starting edge of the groove to the final X depth.

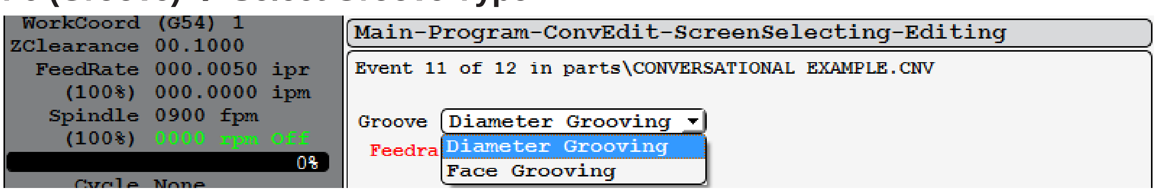
F. Remove the stock along the bottom of the groove to the ending side of the groove.

G. Retract X out the groove.

Note 2: A position move should be made prior to the Diameter Grooving Cycle so that Z is positioned to the start of the groove.

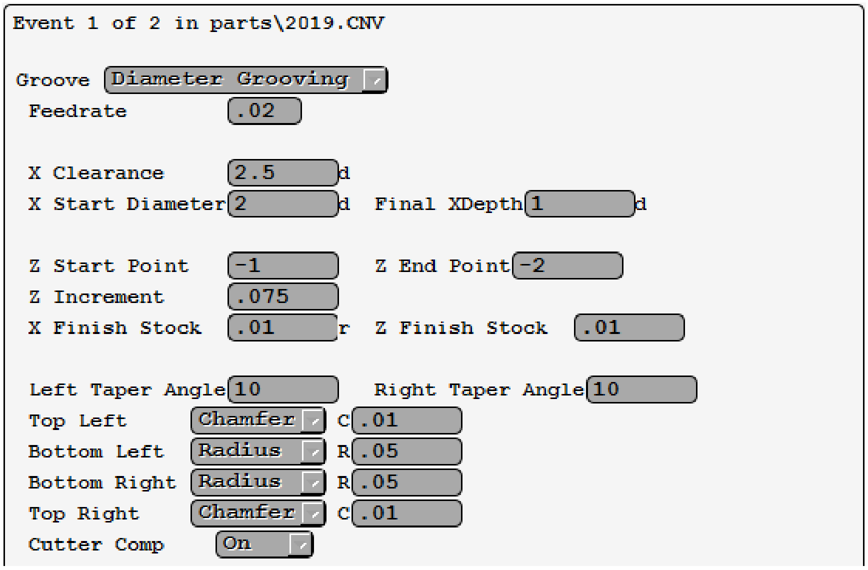
Additional Grooving features are available under F6 (Groove)

F6 (Groove) → Select Groove Type



Pressing F3 (Toggle) will cycle through the grooving cycles that are available. Detailed information about each is located in the G-Code section of this manual under G77.

Diameter Grooving II Using F6 (Groove)


Event 1 of 2 in parts\2019.CNV

Groove: Diameter Grooving
Feedrate: .02

X Clearance: 2.5 d
X Start Diameter: 2 d   Final XDepth: 1 d

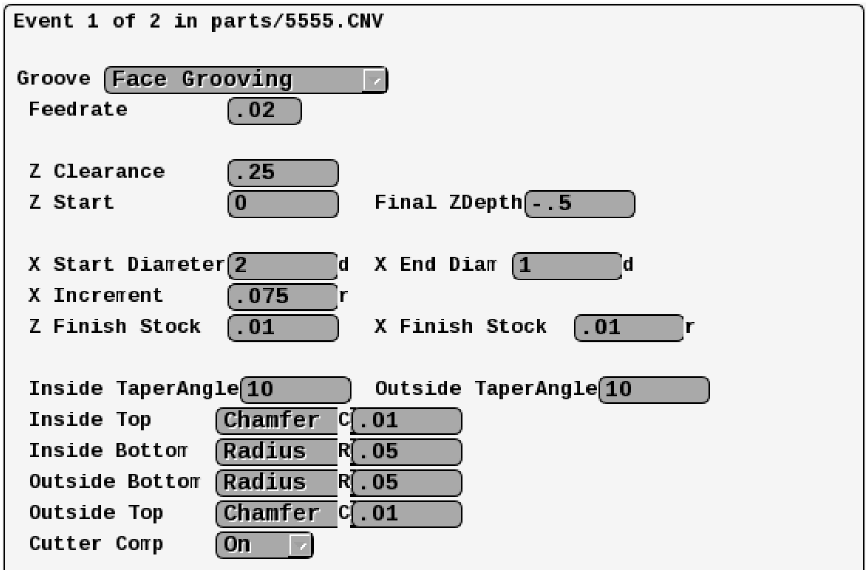
Z Start Point: -1   Z End Point: -2
Z Increment: .075
X Finish Stock: .01 r   Z Finish Stock: .01

Left Taper Angle: 10   Right Taper Angle: 10
Top Left: Chamfer C .01
Bottom Left: Radius R .05
Bottom Right: Radius R .05
Top Right: Chamfer C .01
Cutter Comp: On

The diameter grooving II cycle adds the ability to taper the walls on the groove and round or chamfer any of the 4 corners of the groove.

Groove type:	Diameter groove or Face groove
Feedrate:	The cutting feedrate
X Clearance:	(diameter dimension)
X Start Diameter:	The top of the groove (diameter dimension)
Final XDepth:	The bottom of the groove (diameter dimension)
Z Start Point:	The right side of the groove
Z End Point:	The left side of the groove
Z increment:	Step over in Z
X finish stock:	This material is left and removed on a finish pass (radius dimension)
Z finish stock:	This material is left and removed on a finish pass
Left Taper:	The taper angle on the left side
Right Taper:	The taper angle on the right side
Top left chamfer:	Chamfer amount on the top left.
-or-	
Top left corner round:	Corner round on the top left. (only a chamfer or a radius can be specified)
Bottom left chamfer:	Chamfer amount on the bottom left.
-or-	
Bottom left corner round:	Corner round on the bottom left. (only a chamfer or a radius can be specified)
Bottom right chamfer:	Chamfer amount on the bottom right.
-or-	
Bottom right corner round:	Corner round on the bottom right. (only a chamfer or a radius can be specified)
Top right chamfer:	Chamfer amount on the top right.
-or-	
Top right corner round:	Corner on the top right. (only a chamfer or a radius can be specified)
Cutter comp:	On or off

Face Grooving II Using F6 (Groove)


Event 1 of 2 in parts/5555.CNV

Groove: Face Grooving
Feedrate: .02

Z Clearance: .25
Z Start: 0 | Final ZDepth: -.5

X Start Diameter: 2 d | X End Diam: 1 d
X Increment: .075 r
Z Finish Stock: .01 | X Finish Stock: .01 r

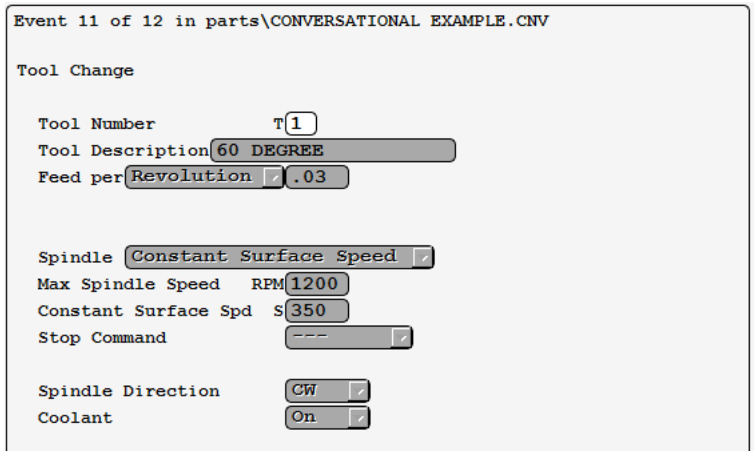
Inside TaperAngle: 10 | Outside TaperAngle: 10
Inside Top: Chamfer C .01
Inside Bottom: Radius R .05
Outside Bottom: Radius R .05
Outside Top: Chamfer C .01
Cutter Comp: On

The face grooving II cycle adds the ability to taper the walls on the groove and round or chamfer any of the 4 corners of the groove.

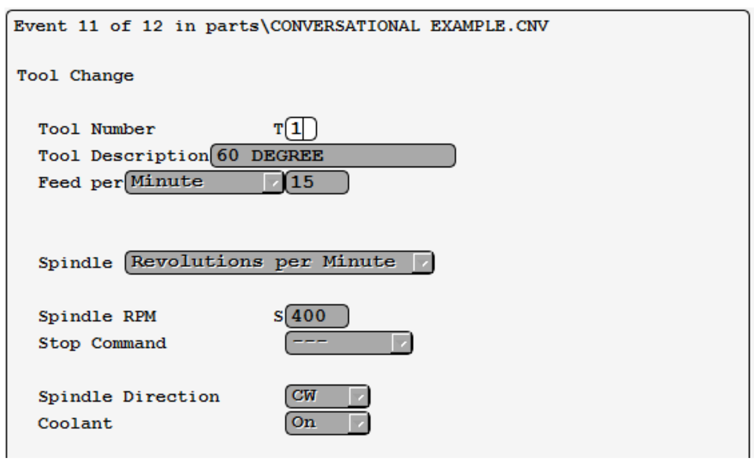
Groove type:	Face groove or Diameter groove
Feedrate:	The cutting feedrate
Z Clearance:	Away from the groove.
Z Start:	top of the groove
Final Z Depth:	Bottom of the groove
X Start Diameter:	The outside diameter
X End Diameter:	The inside diameter
X Increment:	Step over in X (a radius dimension)
Z finish stock:	This material is left and removed on a finish pass
X finish stock:	This material is left and removed on a finish pass (radius dimension)
Inside TaperAngle:	Taper on the smaller diameter
Outside TaperAngle:	Taper on the larger diameter
Inside Top chamfer:	Chamfer amount on the top inside.
-or-	
Inside Top corner round:	Corner round on the top inside. (only a chamfer or a radius can be specified)
Inside Bottom chamfer:	Chamfer amount on the bottom inside.
-or-	
Inside Bottom corner round:	Corner round on the bottom inside. (only a chamfer or a radius can be specified)
Outside Bottom chamfer:	Chamfer amount on the top bottom outside
-or-	
Outside Bottom corner round:	Corner round on the top bottom outside. (only a chamfer or a radius can be specified)
Outside Top chamfer:	Chamfer amount on the top outside.
-or-	
Outside Top corner round:	Corner round on the top outside. (oOnly a chamfer or a radius can be specified)
Cutter comp:	On or Off

F7 (Tool-Chg) → Tool Change

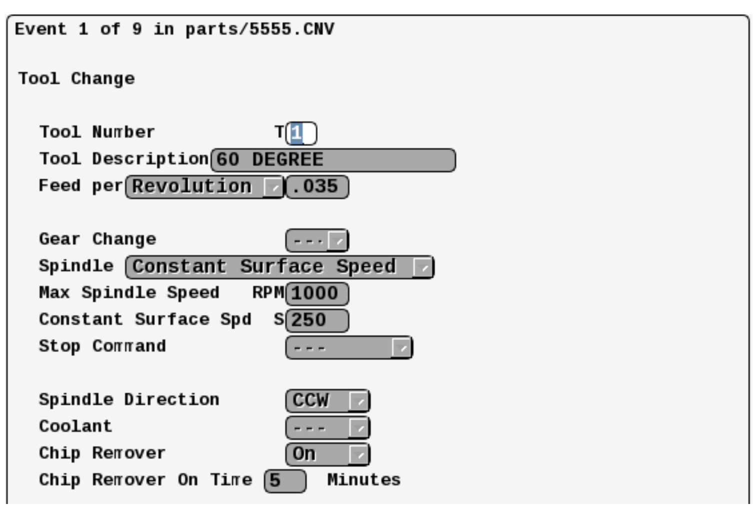
The Tool Change Screen is used for changing tools and tool offsets. The standard tool change rapids X to the tool change position (set by the tool change position parameter), waits for the tool change reset signal, then waits for the cycle start signal.



-OR-



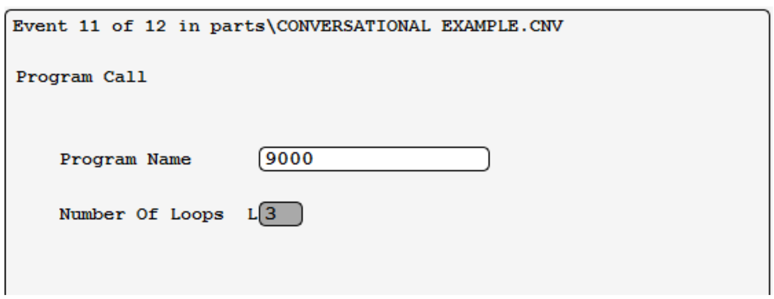
-OR-



Tool Description:	The tool description is used to prompt the operator when the control is waiting for the tool change reset signal.
Feed Per:	Feed/Revolution or Feed/Minute.
Feedrate:	The programmed spindle speed.
Gear Change:	Select Gears 1 through 6.
Spindle:	Constant surface speed or revolution/minute.
Max Spindle Speed:	If constant surface speed is in effect a maximum spindle speed is used to clamp the spindle speed.
Spindle Speed:	The programmed spindle speed.
Stop Command:	Program stop or optional stop.
Spindle Restart:	Spindle restart CW or CCW.
Coolant:	Coolant ON.
Chip Remove:	On or OFF
Chip Remove Time:	# of minutes to run chip conveyor

F8 (ProgCall) Program Call

The Program Call screen is used to transfer program execution to another program for a specified number of loops.

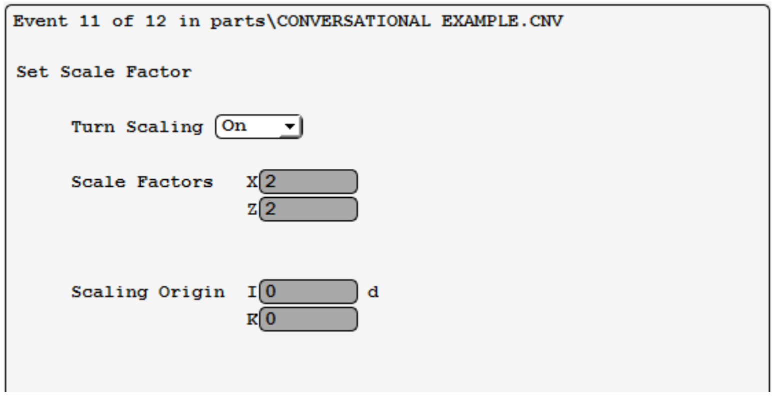


F9 (Special) Special Functions

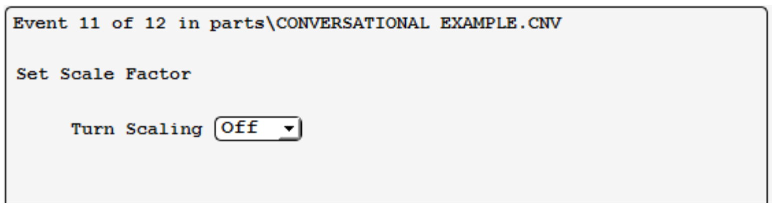
These are screens for setting or adjusting various parameters in the control. The parameters control various functions such as scale factors, rotation angles, mirror image, and floating zeroes.

The special selections are:

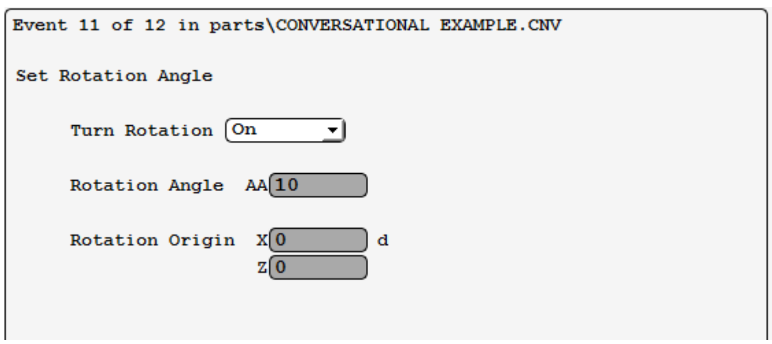
F9 (Special) → F3 (Scale) Set Scale Factor

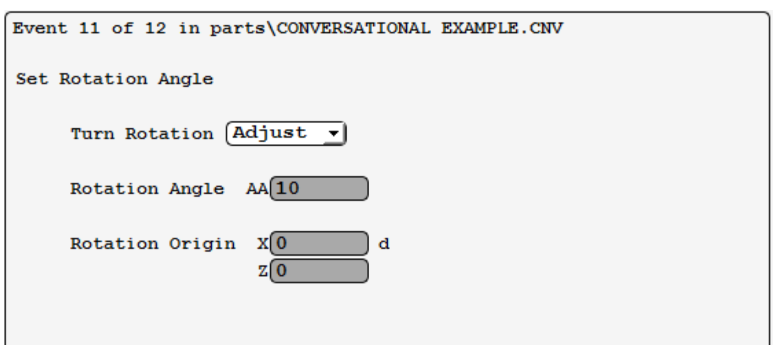


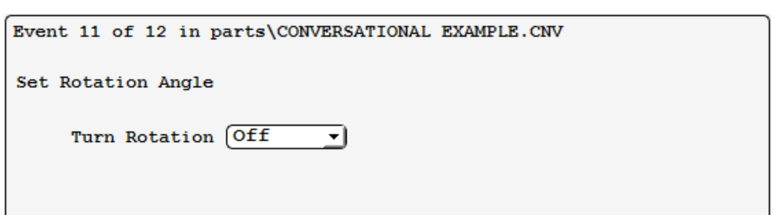
-OR-



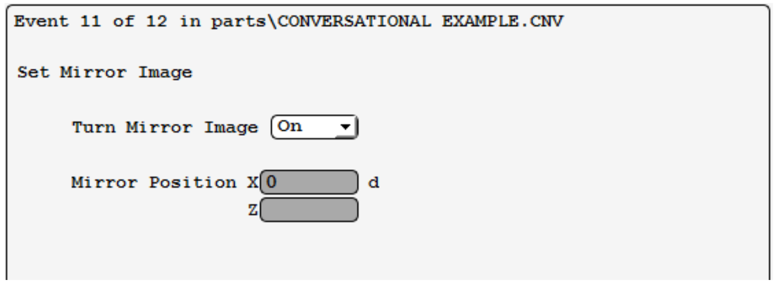
F9 (Special) → F4 (Rotate) Set Rotation Angle



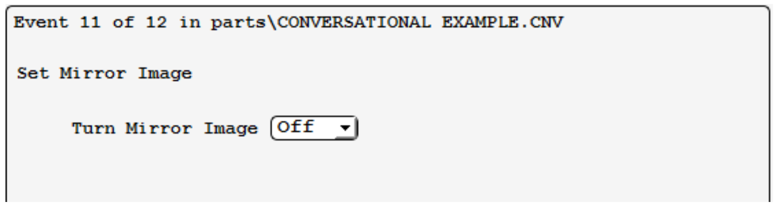




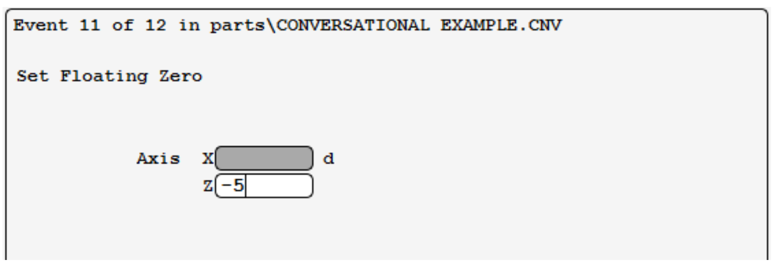
F9 (Special) → F5 (Mirror) Set Mirror Image



-OR-

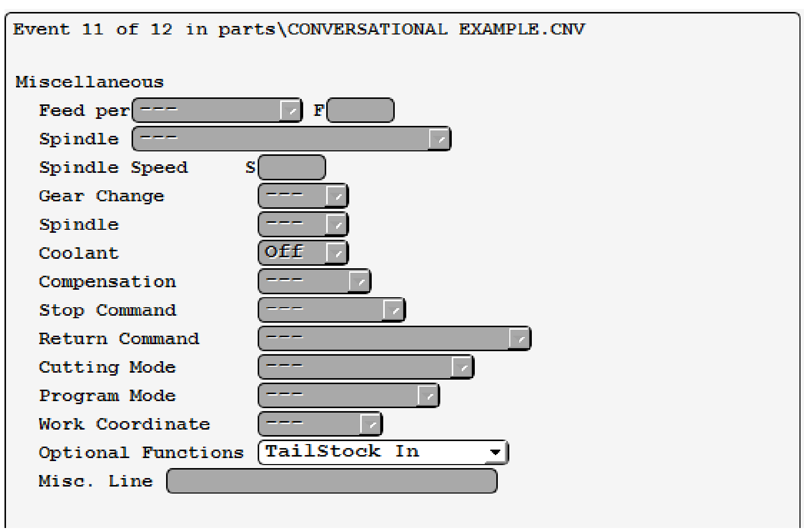


F9 (Special) → F6 (Float-Zr) Set Floating Zero



F10 (Misc) → Miscellaneous

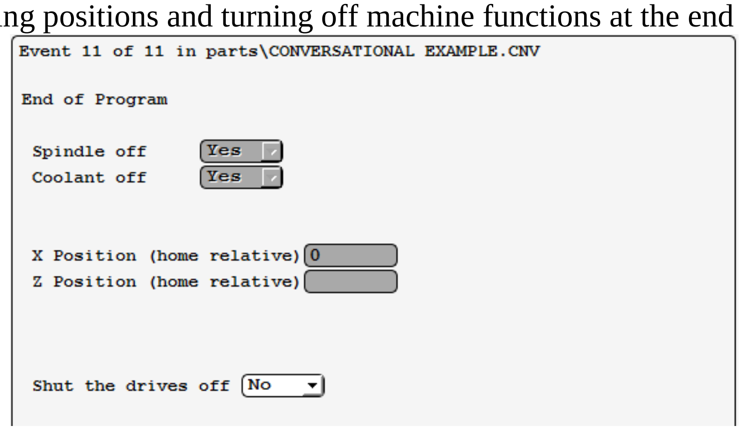
As a program is being created it may be necessary to add certain miscellaneous functions, such as coolant and spindle commands. This is done through the Miscellaneous Screen.



Feed:	Per Rev or Per Minute (posts G99 or G98 respectively).
Feedrate:	In inches/Rev, mm/Rev, Inches/Min or mm/Min (posts F###.####)
Spindle:	Constant surface speed or revolutions per minute (posts G96 or G99 respectively) (if constant surface speed is selected a maximum spindle speed can be entered)
Spindle Speed:	In feet/Min, mm/Min or Revs/Min (posts S###.#)
Gear Change:	Low, Medium or High (posts M41, M42, or M93 respectively)
Spindle:	CW, CCW or OFF (posts M3, M4 or M5 respectively)
Coolant:	ON or OFF (posts M8 or M9 respectively)
Compensation:	LEFT, RIGHT, or OFF (Posts G41, G42 or G40 respectively)
Stop Command:	Program or Optional (posts M0 or M1 respectively)
Return Command:	Z to clearance, X to clearance, Z-X to clearance, X-Z to clearance, Z to home, X to home, Z-X to home or X-Z to home Z to Clearance posts G45 X to Clearance posts G46 Z-X to Clearance posts G45, G46 X-Z to Clearance posts G46, G45 Z to Home posts G47 X to Home posts G48 Z-X to Home posts G47, G48 X-Z to Home posts G48, G47
Cutting Mode:	Point to Point or Continuous (posts G61 or G64 respectively)
Program Mode:	Absolute or Incremental (posts G93 or G91 respectively)
Work Coordinates:	1, 2 . . .6 (posts G54, G55, ...G59 respectively)
Other Functions:	Chuck in, chuck out, bar feed, tail stock in, tail stock out, parts catcher in, parts catcher out, air blast on, air blast off, accessory 1, accessory 2... accessory 5
Miscellaneous Line:	The Miscellaneous Line is used to type in any M Code, G Code or text that is not on the standard list.

End Of Program Event

Conversational programs always end with the End of Program screen. These options allow for part unloading/loading positions and turning off machine functions at the end of the cycle.



6

Section 6 Contents

G-Codes - Definition

These codes are used if the operator is programming the 8000T Series control in the text or MDI mode. They are also generated from conversation programs. It should be noted that most programmers, particularly new ones, use the conversational programming mode. If you are planning to use the text mode of programming, pay close attention to this section explaining these codes. If you are planning to use conversational programming, you can ignore or skim this section and concentrate on the conversational section.

The preparatory function code is a two-digit number preceded by the letter G. Preparatory functions are used to determine the program operating mode and are divided into two types: one-shot and modal. **One-shot** G codes are only in effect during execution of the block in which they are present. **Modal** G codes establish operating modes, which remain in effect until replaced by another mode in the same category. If the G code is less than 10, the zero entry is optional (G2 or G02).

The following table lists G codes accepted by the 8000T Series control system. Each code will have a detailed explanation later in the manual.

G Code Listing

		Active On Power-up	Modal	One Shot
00	Rapid Positioning		X	
01	Linear interpolation	X	X	
02	Circular/helical interpolation CW		X	
03	Circular/helical interpolation CCW		X	
04	Dwell			X
09	Exact stop			X
10	Set data on		X	
11	Set data off	X	X	
12	Clear floating zero			X
13	Back Side Programming	*[1]	X	
14	Front Side Programming	*[1]	X	
20	Inch mode	*[2]	X	
21	Metric mode	*[2]	X	
22	Safe zone check off	*[3]	X	
23	Safe zone check on	*[3]	X	
28-30	Reference point return			X
32	Threading			X
34-36	Threading			X

[1] Back Side or Front Side programming is selectable on power-up via a parameter.
[2] G20 and G21 are selectable on power up via a parameter.
[3] Safe Zone on or off is selectable on power-up via a parameter.

		Active On Power-up	Modal	One Shot
38	Initial Point Return		X	
39	R-Plane Return		X	
40	Cutter compensation cancel	X	X	
41	Cutter compensation left		X	
42	Cutter compensation right		X	
45	Z to Clearance			X
46	X to Clearance			X
47	Z to tool change			X
48	X to tool change			X
49	Cancel H offset		X	
50	Set FLZ, Max RPM’s			X
52	Local coordinate system set		X	
53	Machine coordinate system			X
54	Work coordinate 1 system	X	X	
55-59	Work coordinate 2-6 systems		X	
60	One-Shot Rapid			X
61	Exact stop mode		X	
63	Tapping mode		X	
64	Cutting	X	X	
65	Non-movement / Program call			X
70	Finish Cycle			X
71	Rough Turning			X
72	Rough Facing			X
73	Pattern Repeat			X
74	Face Grooving			X
75	Diameter Grooving			X
76	Threading Cycle			X
77	Grooving II			X
80	Cancel canned cycle	X	X	
81	Drill			X
82	Drill/dwell			X
83	Peck/drill			X
84	Tap			X
85	Bore			X
86	Bore / Spindle Stop			X

		Active On Power-up	Modal	One Shot
89	Bore / Dwell			X
90	Turning Cycle A			X
91	Incremental dimension		X	
92	Single Thread Cutting Cycle			X
93	Absolute	X	X	
94	Facing Cycle B			X
96	Constant Surface Speed	*[4]	X	
97	Revs per Minute	*[4]	X	
98	Feed per Minute	*[5]	X	
99	Feed per Rev.	*[5]	X	
501	Mirror Image OFF	X	X	
511	Mirror Image ON		X	
550	Scaling OFF	X	X	
551	Scaling ON		X	
568	Rotation ON		X	
569	Rotation OFF	X	X	
778	Draw Stock			X
980	Feed per Rev.		X	
981	Feed per Rev. / Wait for Marker		X	
982	FOV Unlock		X	
983	FOV Lock at 100%		X	
984	SOV Unlock		X	
985	SOV Lock at 100%		X	
990	Store parameters			X
991	Read parameters			X
997	Force error			X
998	Beep			X

Note: Unrecognized G codes will cause an error 549 to occur.

[4] Constant Surface Speed or Revs Per Minute is selectable on power-up via a parameter.
[5] Feed Per Minute or Feed Per Rev is selectable on power-up via a parameter.

Positioning (G0) Rapid Traverse (modal)

G0 specifies positioning in rapid traverse mode. There is no need to program rapid traverse rates because the rates are preset by parameters. Rapid traverse rates can be overridden by the feedrate override switch on the machine operator's panel.

G0 moves the tool at a rapid traverse rate to a position in the work coordinate system for both incremental and absolute commands.

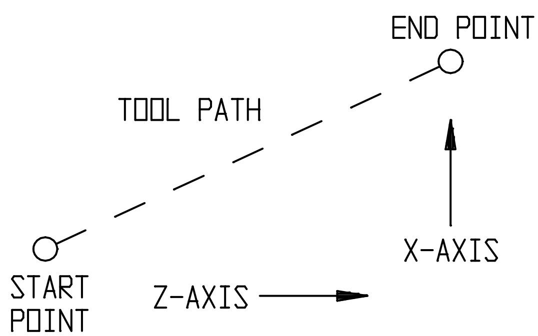
Format: G0 ~-;

where ~- is: a combination of optional axis address (of X or Z) as X-Z...

where ; is: End of block (CR for EIA ASCII code)

This manual uses this notation hereinafter.

The programmed feed remains in the feedrate register and can be activated by canceling the G0 command with a G1 command. The motions of all axes in G0 mode will be interpolated with all axes reaching the end point simultaneously.



Note 1: The rapid traverse rate in the G0 command is set for each axis independently by the machine tool builder. Accordingly, the rapid traverse rate cannot be specified in the address F. In the positioning mode actuated by G0, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position. "In-position" means that the axis position is within a specified range. (This range is determined by the machine tool builder.)

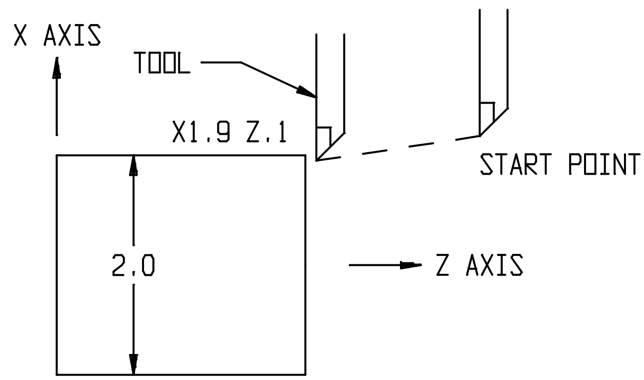
G0 mode automatically accelerates and decelerates in a linear fashion allowing the controlled axis to start and stop smoothly. The rate of accel/decel can be changed by the machine tool builder.

Linear Interpolation (G1)

G1 ~___F___;

This command actuates the linear interpolation mode. The value of ~ defines the distance the tool will travel. The feedrate is set to a cutting feed by the F code and is modal. An example follows:

G1 X1.9 Z.1 F20 ;



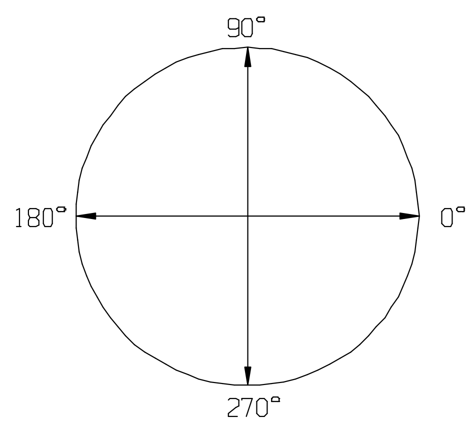
The feedrate specified by the F code is the vector rate along the path, not the rate of each axis.

Note: There are two feed modes: Feed per Minute, and Feed per Rev.

Polar Definition of a Line

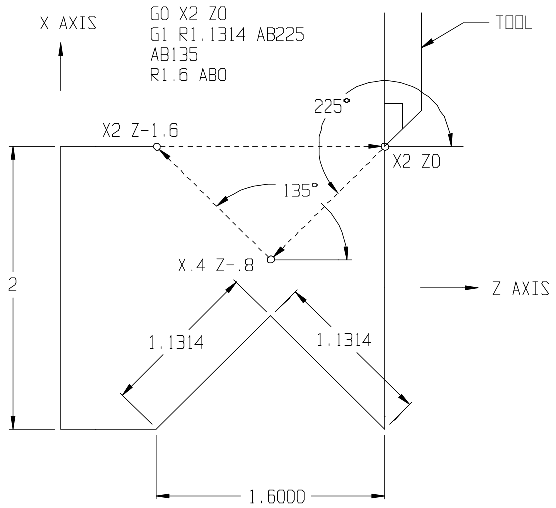
A polar line is specified by a polar radius/length (R), an angle (AB), and a polar center (AA or I, K, or XC, ZC).

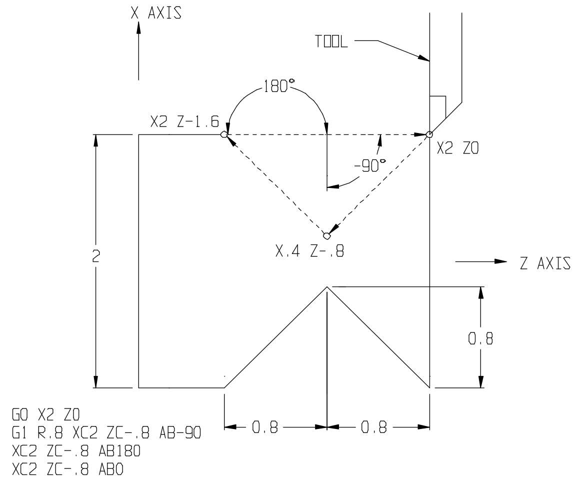
The 3 o'clock position is always 0 degrees. Positive angles result in CCW rotation of the polar radius, while negative angles result in CW rotation of the polar radius. Polar lines can be used when estimating lengths during trig help.

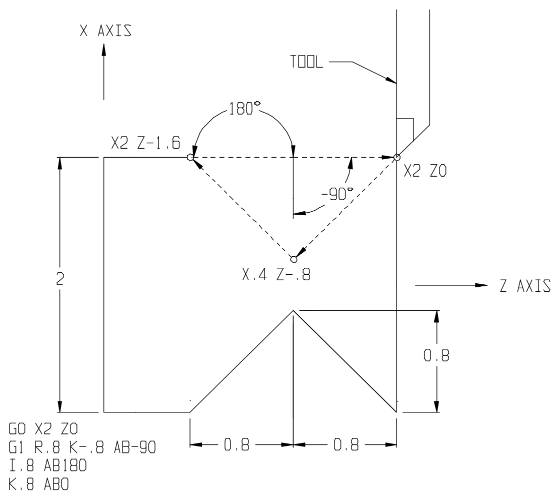


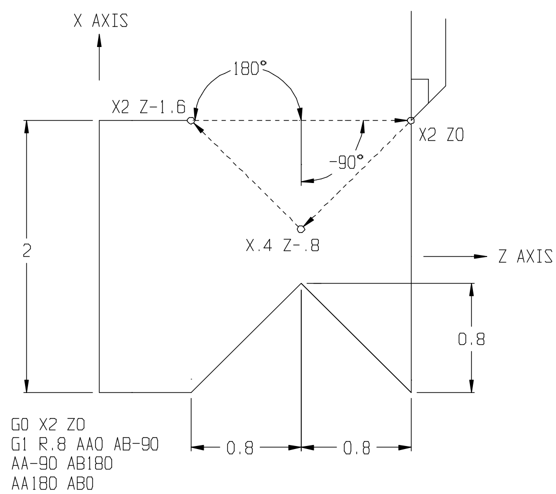
If the polar radius/length (R) or angle (AB) is not specified, then the previous values will be used. If the polar center is not specified then it is taken to be the current machine position.

Examples of Polar Programming:









Cutting Feedrate G1, G2, G3 Mode

The feedrate of linear interpolation (G1) and circular interpolation (G2, G3) are commanded with numbers after the F code.

The F command can appear anywhere in a block and specifies the rate of motion in inches or millimeters per minute, or inches or millimeters per rev.

Tangential Feedrate Control

The cutting feed is controlled so that speed along the path is always the commanded feedrate.

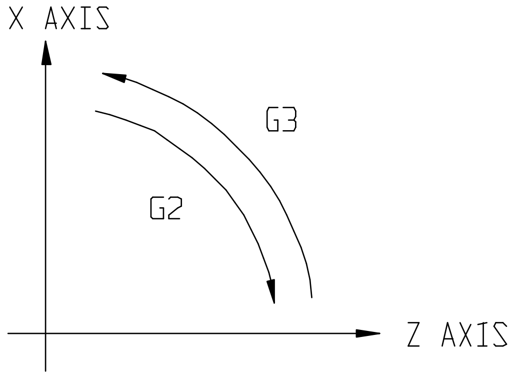
Feedrate Override

The feed can be overridden using this switch on the machine operator's panel by 0 to 200%. Feedrate override cannot be applied to functions in which override is inhibited (e.g. tapping cycle, threading).

Circular Interpolation (G2, G3)

The general command format to move along a circular arc is as follows:

	Data to be given		Command	Meaning
1	Direction of rotation		G2	Clockwise (CW)
			G3	Counter-clockwise (CCW)
2	Radius and absolute center		R, XC and/or ZC	The radius and signed absolute coordinates of the arc center
	Radius and start angle		R and AA	Angle from radius and the center to the start point
	Distance from start point to center		I and/or K	The signed distance from start point to center $R=\sqrt{(I^2+K^2)}$
	Radius Only		R	Arc radius (if no center is specified a center is calculated from the start point and end point) (-R is the longer arc)
3	End point position	G93 mode	X and or Z	End point position in work coordinate system
		G91 mode	X and or Z	Distance from start point to end point
	Radius and end angle		R and AB	Radius and angle from the center to the end point
	Radius and incremental end point		R, W and or U	Radius and distance from the start point to the end point
4	Feedrate		F	Velocity along arc



Clockwise and Counterclockwise Directions
Describing an Arc Center

The end point of an arc is specified by address X and Z, and is expressed as an absolute or incremental value depending on G93 or G91. In incremental the coordinate of the end point is related to the start point of the arc. The incremental arc center can also be defined by I and K for the X and Z axes. The numerical value following I, or K is the distance from the start point to the arc center in X and Z axes. I and K are always incremental values independent of G93 and G91.

The sign of I and K depends on the relationship of the center to the start point as shown below:

Programming with Circular Interpolation

Describing an Arc Using a Radius and No Center

When describing an arc using a radius value there are a number of valid formats. The various command formats are as follows:

G2	X____	Z____	R____	
or				(or –R)
G3	X____	Z____	R____	

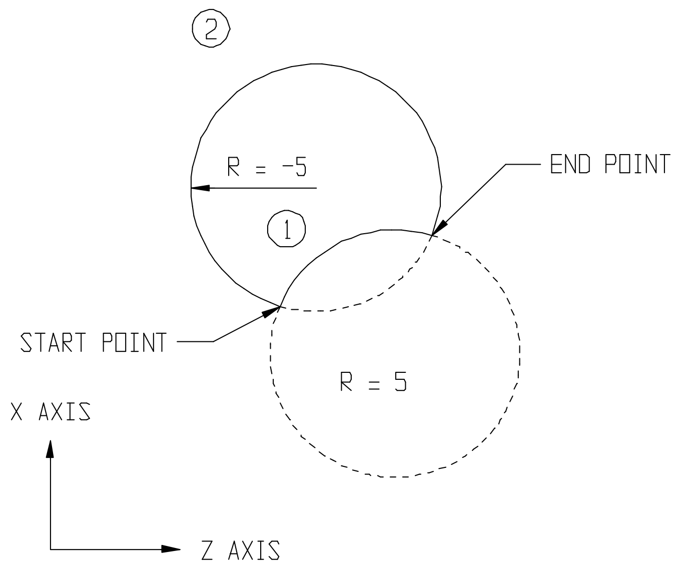
Arc End Points

The radius is always specified as its true value. The end points are incremental or absolute depending on G93 and G91. If a radius is used without a center point there are two types of arcs that can be generated. One is less than 180º, and the other is greater than 180º, as shown in the figure that follows. When the arc exceeds 180º* the radius must be specified as a negative value.

Examples:

For arc 1 (less than 180º)
G2 X4 Z6 R5 F30

For arc 2 (greater than 180º)*
G2 X4 Z6 R-5 F30



Describing an Arc Using Absolute Center and Trig Help

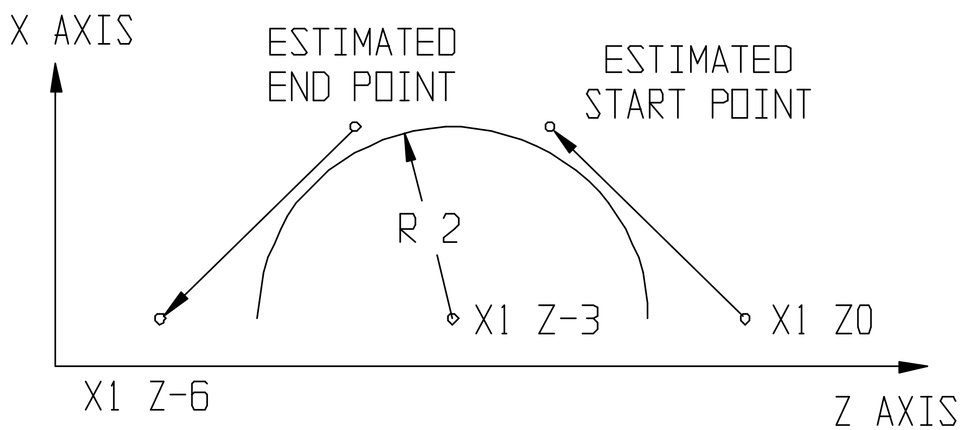
G2	X___	Z___	XC___	ZC___	R___
or					
G3	X___	Z___	XC___	ZC___	R___
	End Point		Center Point		Radius

Trig Help will allow the programmer to estimate both the start and end points of any arc. The control will calculate the true start and end points based on the moves preceding and trailing the arc. Where there are two possible correct answers, the control will choose the point closest to the estimated point. If the slope of the line entering or leaving the arc is such that no intersection occurs, the line will be made tangent to the arc.

* Note: Arcs greater than 180º are rare in lathe applications.

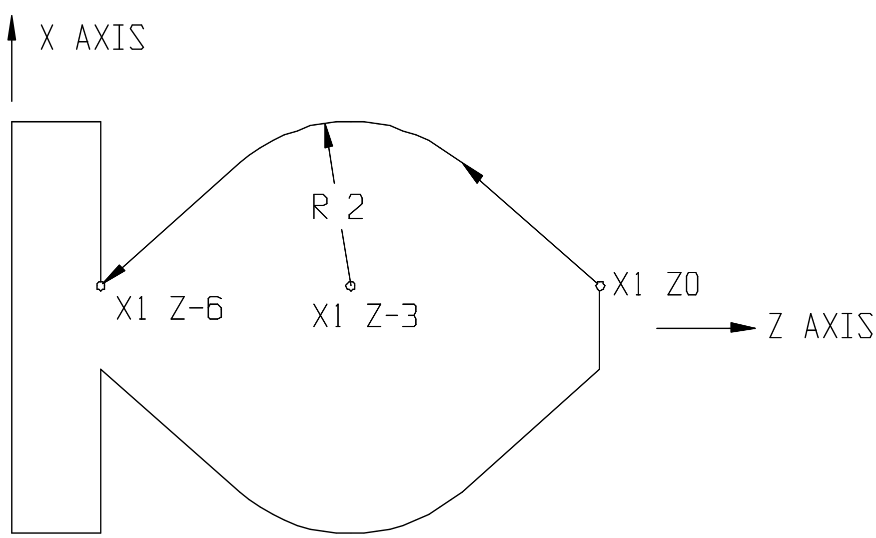
Examples of Trig Help

Program 1



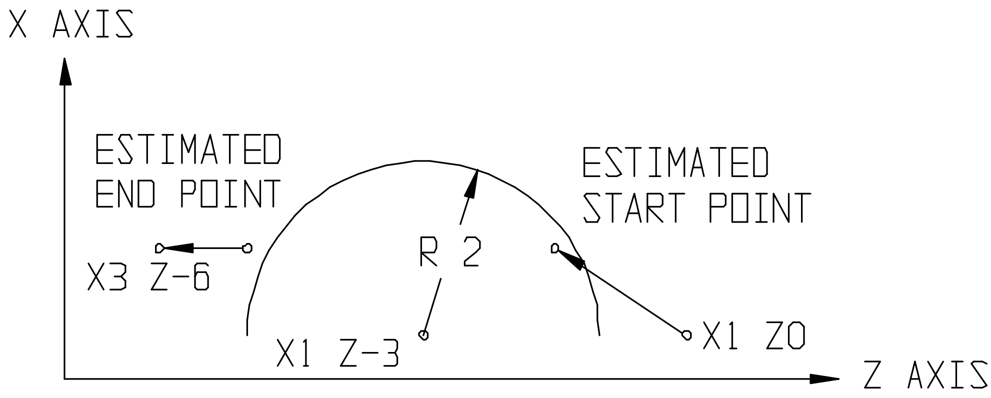
Programmed Path

G1 X1 Z0
X5 Z-2 (estimated start point)
G3 R2 XC1 ZC-3 X5 Z-4 (estimated end point)
G1 X1 Z-6



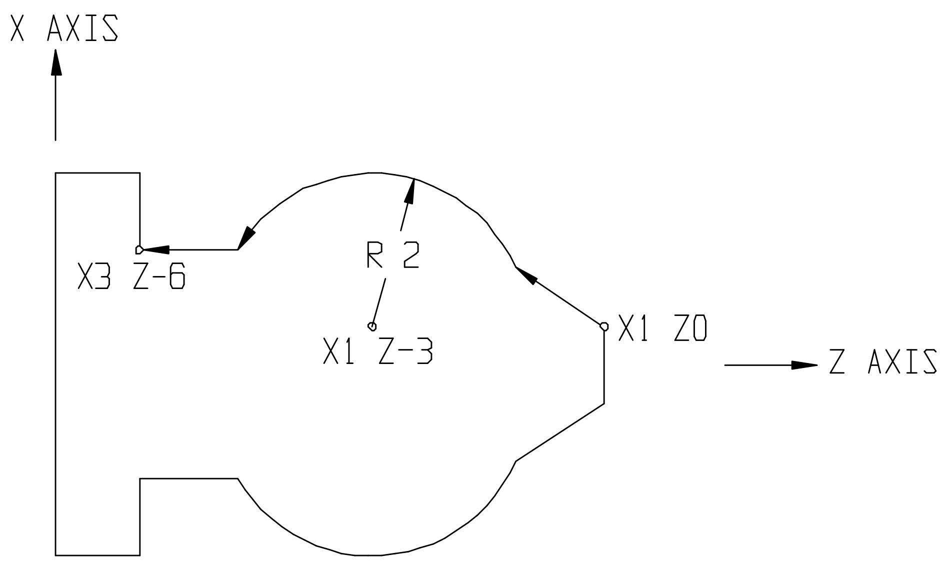
Path Generated by Program 1

Program 2



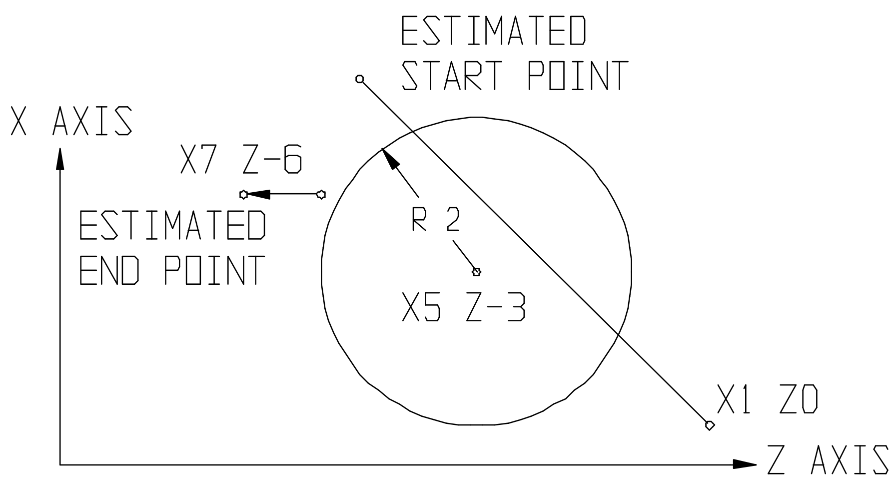
Programmed Path

G1 X1 Z0
X3 Z-1.5 (estimated start point)
G3 R2 XC1 ZC-3 X3 Z-5 (estimated end point)
G1 X3 Z-6



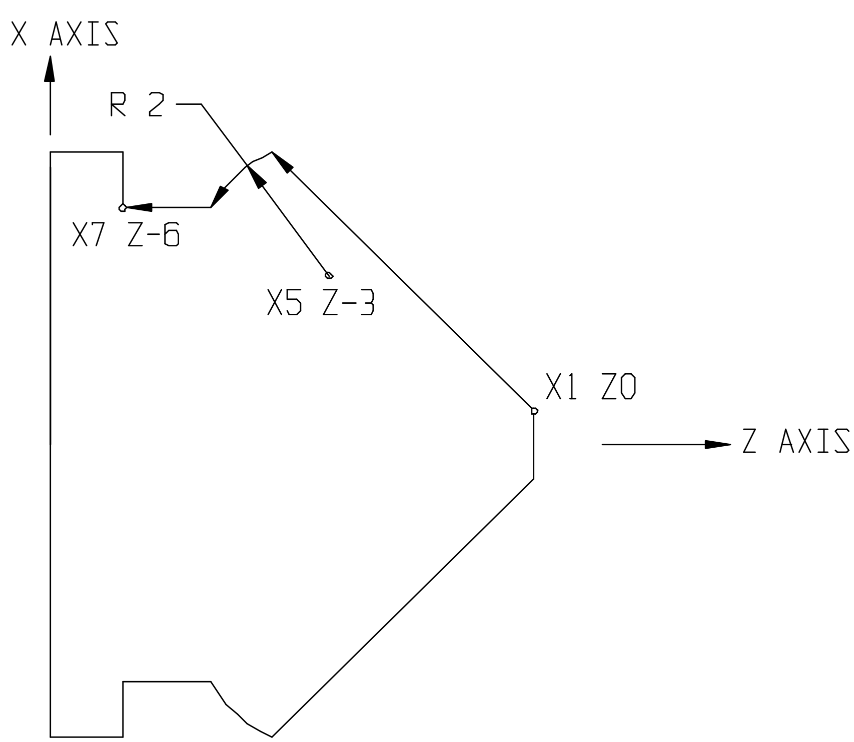
Path Generated by Program 2

Program 3



Programmed Path

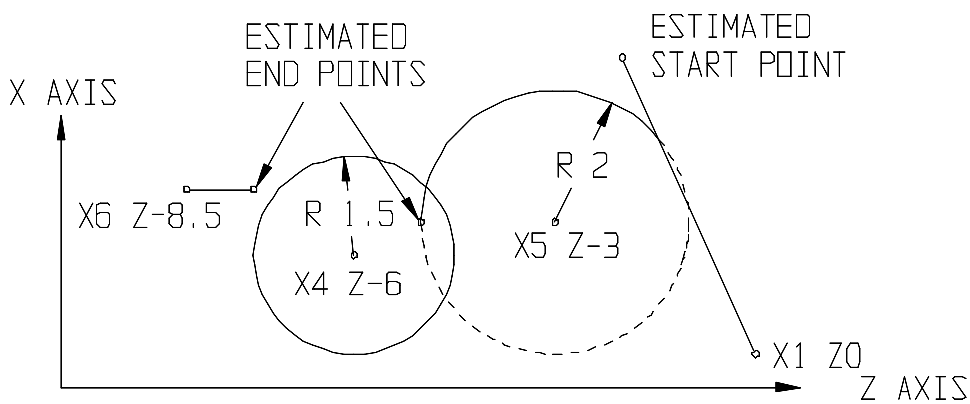
G1 X1 Z0
X10 Z-4.5 (estimated start point)
G3 R2 XC5 ZC-3 X7 Z-5 (estimated end point)
G1 X7 Z-6



Path Generated by Program 3

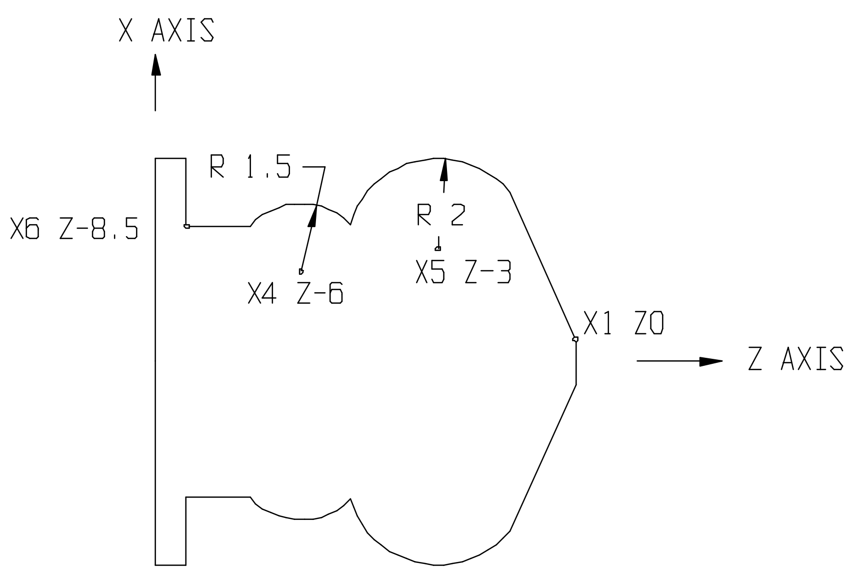
In general, when dealing with lines and arcs, if the line is programmed short of the arc it will be extended to the arc. If the line is programmed past the arc it will be shortened to the arc, and if the line does not intersect the arc it will be made tangent.

Program 4



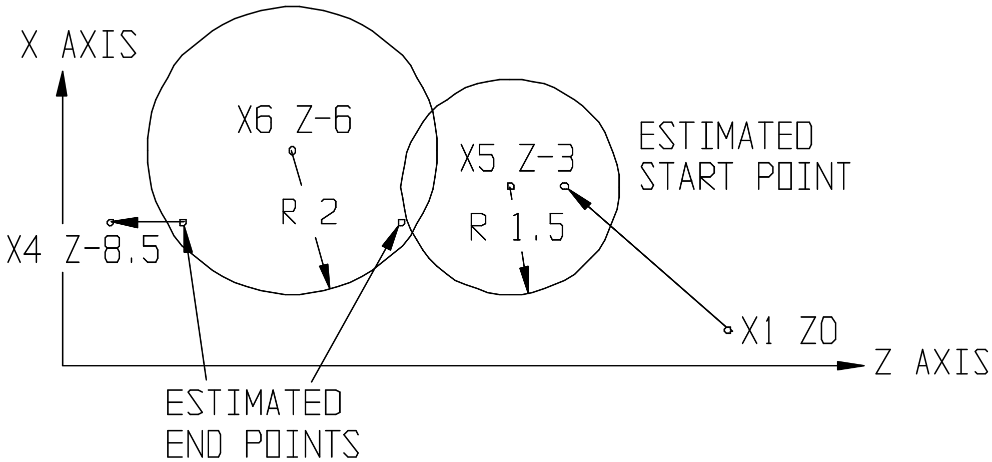
Programmed Path

G1 X1 Z0
X10 Z-2 (estimated start point)
G3 R2 XC5 ZC-3 X5 Z-5 (estimated end point)
G3 R1.5 XC4 ZC-6 X6 Z-7.5
G1 X6 Z-8.5



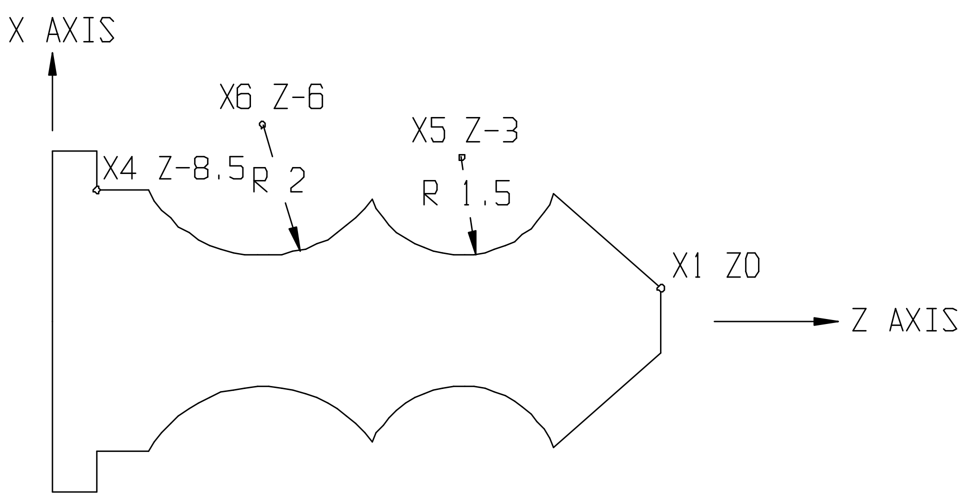
Path Generated By Program 4

Program 5



Programmed Path

G1 X1 Z0
X5 Z-2.25 (Estimated Start Point)
G2 R1.5 XC5 ZC-3 X4 Z-4.5 (Estimated End Point)
G2 R2 XC6 ZC-6 X4 Z-7.5 (Estimated End Point)
G1 X4 Z-8.5



Path Generated by Program 5

<u>Things To Remember When Estimating Points</u>

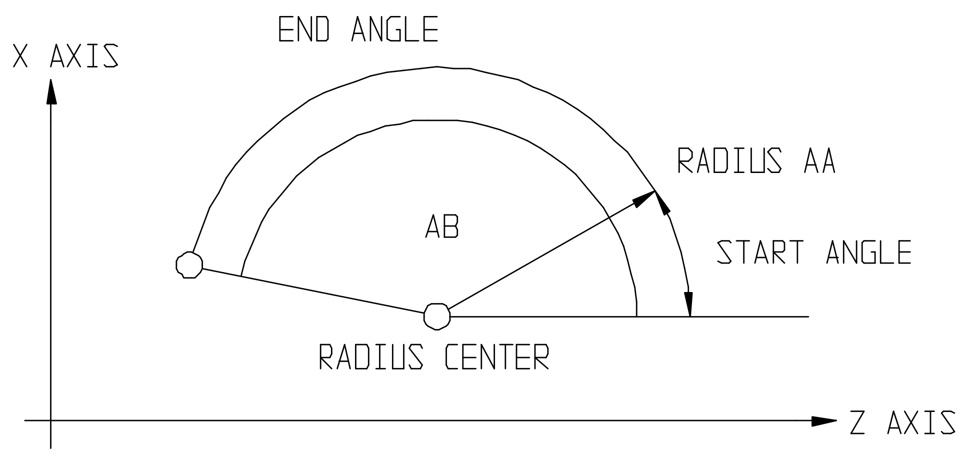
- Estimating can be used with line to arc, arc to arc, and arc to line paths.
- The center and radius of arcs cannot be estimated.
- For line to arc and arc to line, the start and end point estimates <u>must</u> lie on the line; i.e. the <u>slopes</u> of the lines entering or leaving the arc must be correct.
- If a line intersects an arc at two points, the estimated point should be <u>closer</u> to the desired point of intersection.
- If the above conditions are met there is no limit on how far the estimated point is away from the correct point.
- When estimating the intersection of one arc to another arc, the easiest point to pick on an arc end point is at one of the quadrant points (0º, 90º, 180º, or 270º).

Describing an Arc Using Polar Definitions

The polar definitions do not change from absolute to incremental. The center of the arc is always considered the pole and all angles are related to it. The basic polar definition is as follows:

G2
or AA_____ AB_____ R_____
G3 start angle end angle radius

Polar Arc Definitions



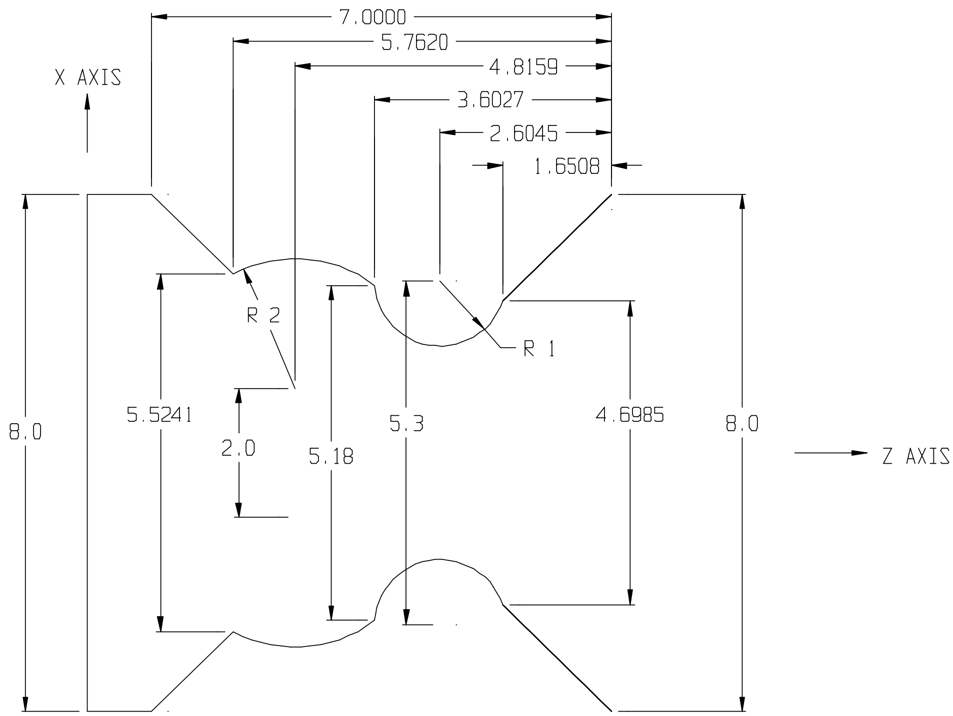
The polar format for arcs can be mixed with the Cartesian formats. The following are legal formats.

G2 X_____ Z_____ AA_____ R_____
end point start angle
G2 AB_____ XC_____ ZC_____ R_____
end angle center point
G2 I_____ K_____ AB_____
center point end angle

Trig Help is only valid in polar when using an arc with valid center point and radius.

Program 6

The following programs will all produce the same part, and which programming method is used is totally optional.



1) Absolute coordinates (Polar No Trig Help)

```
G0 X8 Z0
G1 F100 R2.3345 AB225
G2 R1 AA342.496 AB183.441
G3 R2 AA52.655 AB118.234
G1 R1.7507 AB135
```

2) Absolute coordinates (Polar Trig Help)

```
G0 X8 Z0
G1 F100 R1 AB225
G2 R1 XC5.3 ZC-2.6045 AB180
G3 R2 XC2 ZC-4.8159 AB120
G1 X8 Z-7 BACK C0 W135
```

Note: When using Trig Help you must have a valid arc center and radius.

3) Absolute coordinates (Cartesian No Trig Help)

G0 X8 Z0
G1 X4.6985 Z-1.6508
G2 R1 XC5.3 ZC-2.6045 X5.18 Z-3.6027
(or) (G2 I.30075 K-.9537 X5.18 Z-3.6027)
(or) (G2 R1 X5.18 Z-3.6027)
G3 R2 XC2 ZC-4.8159 X5.5241 Z-5.7621
G1 X8 Z-7

4) Absolute coordinates (Cartesian Trig Help)

G0 X8 Z0
G1 X5 Z-1.5
G2 R1 XC5.3 ZC-2.6045 X5 Z-3
G3 R2 XC2 ZC-4.8159 X4 Z-5
G1 X8 Z-7

5) Incremental coordinates

G0 X8 Z0
G1 U-3.3015 W-1.6508
G2 I.30075 K-.9537 U.4815 W-1.9519
G3 I-1.59 K-1.2132 U.3441 W-2.1593
G1 U2.4759 W-1.238

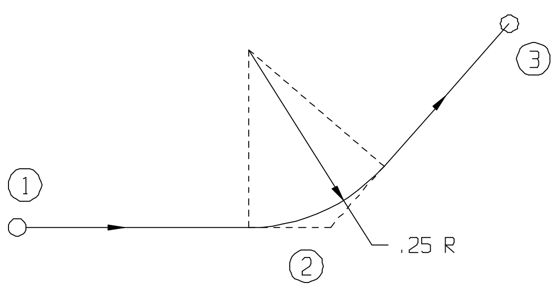
Note: *In Incremental, Trig Help cannot be used as each point is related to the current position.*

Trig Help can be shut off by setting bit 2 of the special flags parameter. This may be desirable for programs generated from some Cad/Cam systems.

Corner Rounding

By adding *,R*___ to the end of blocks commanding linear or circular interpolation, corner rounding can be automatically inserted.

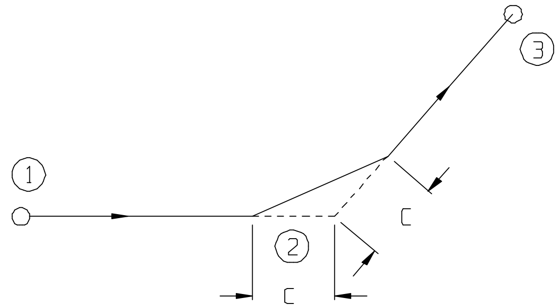
(1) G91 G01 X0 Z0
(2) Z1, R.25
(3) X2 Z2



Angle Chamfering

By adding ,*C*___ to the end of blocks commanding linear interpolation, angle chamfering is automatically inserted.

(1) G91 G01 X0 Z0
(2) Z1, C.25
(3) X2 Z2

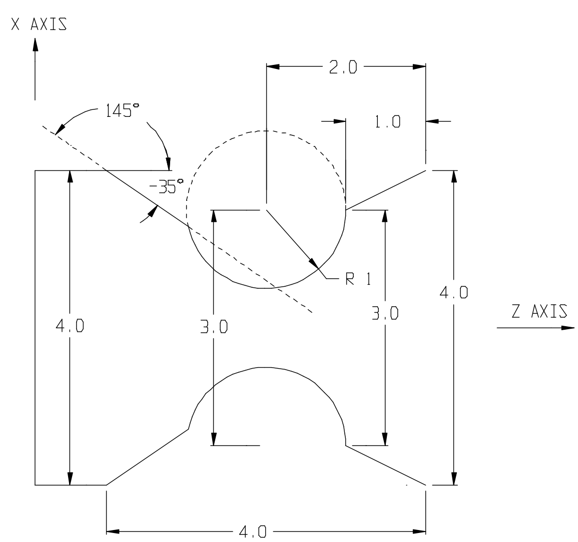


Back Line (Extend Back)

The back line function can be used on any line command. This function reverses the direction of a programmed line. It would normally be used when you know the end point of the line and not its start point. The end point would be programmed and the line would be extended backwards to the start point. When using this function all Trig Help functions are still valid.

Example 1:

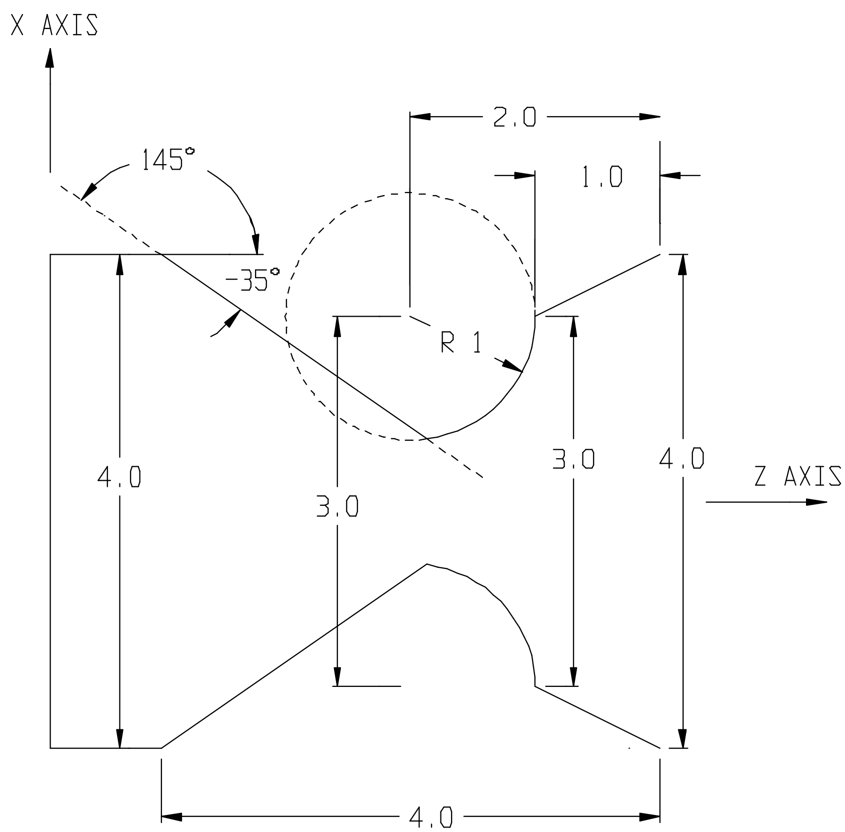
(1) X4 Z0
(2) X3 Z-1
(3) G2 R1 ZC-2 AB270
(4) G1 X4 Z-4 BACK C0 W145



Back	extend line backwards from (4,-4)
C0	use the arc intersection closest to (4,-4)
W145	extend the line from (4,-4) at an angle of 145º

Example 2:

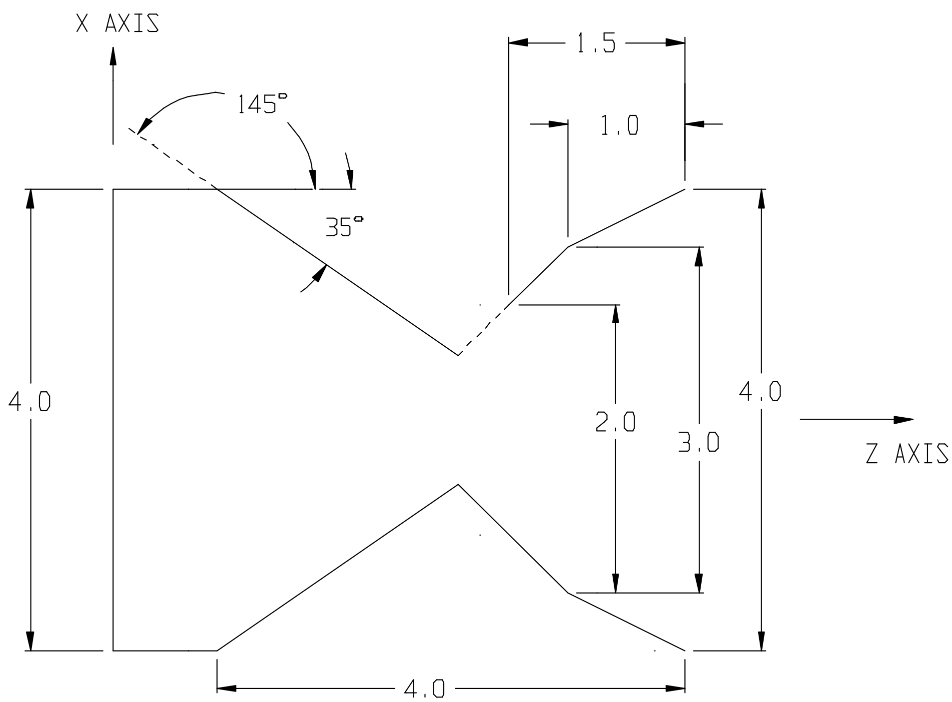
(1) X4 Z0
(2) X3 Z-1
(3) G2 R1 ZC-2 AB270
(4) G1 X4 Z-4 BACK C2 W145



C2 use the arc intersection farthest from (4,-4)

Example 3:

(1) X4 Z0
(2) X3 Z-1
(3) X2 Z-1.5
(4) X4 Z-4 BACK C0 W145
(or C2) W-35



This example used a back line between two lines to program an unknown point.

Other Notes on Circular and Linear Turning

The feedrate in circular and linear is equal to the feedrate specified by the F code. This feedrate is the tangential feedrate along the arc and the vector feed on the linear moves.

Note 1: I0 and K0 can be omitted.

Note 2: If X and Z are omitted, or if the end point of an arc is located at the same position as the start point, and the center is commanded by I and K, an arc of 360º (a complete circle) is assumed.

G2 I _____; (a complete circle)

When only R is used, an arc of 0º is programmed.

G2 R _____; (The cutter does not move.)

Note 3: The error between the specified feedrate and the actual tool feedrate is 2% or less. However, this feedrate is measured along the arc after the cutter compensation is applied.

Note 4: If I, K and R addresses are specified simultaneously, the arc specified by address R takes precedence and the others are ignored.

Note 5: The X Z I K R AA AB commands are retained by the control. If an interpolation block is left incomplete, the missing axis information will be defaulted to the value previously entered.

Dwell command (G4)

The G4 code must be immediately followed by an F###.# instruction. This instruction will then cause the program to stop or dwell for ###.# seconds.

General Format

G4F2.5 will cause the program to dwell for 2.5 seconds.

G4F25 will cause the program to dwell for 25 seconds.

Note: A P or a X can be used in place of an F following the G4 command.

Exact Stop (G9)

Moves commanded in blocks with G9 decelerate at the end point, and in-position check is performed. This function is used when sharp edges are required for work piece corners in cutting feed.

Set Data On/Off (G10, G11)

This function allows all the CNC's configuration, setup, axis and offset table parameters to be loaded via a program rather than through the front panel. (This function is the only way to change parameters 700 and higher from a program.) The format for loading the parameters is as follows:

G10	Set data On
P*** = value	
P*** = value	P*** = parameter number
P*** = value	to be loaded
P*** = value	
G11	Set data Off

When the G11 is performed, the CNC will start using the new parameter settings. Refer to the Appendix for information on parameter assignments.

Clear Floating Zero (G12)

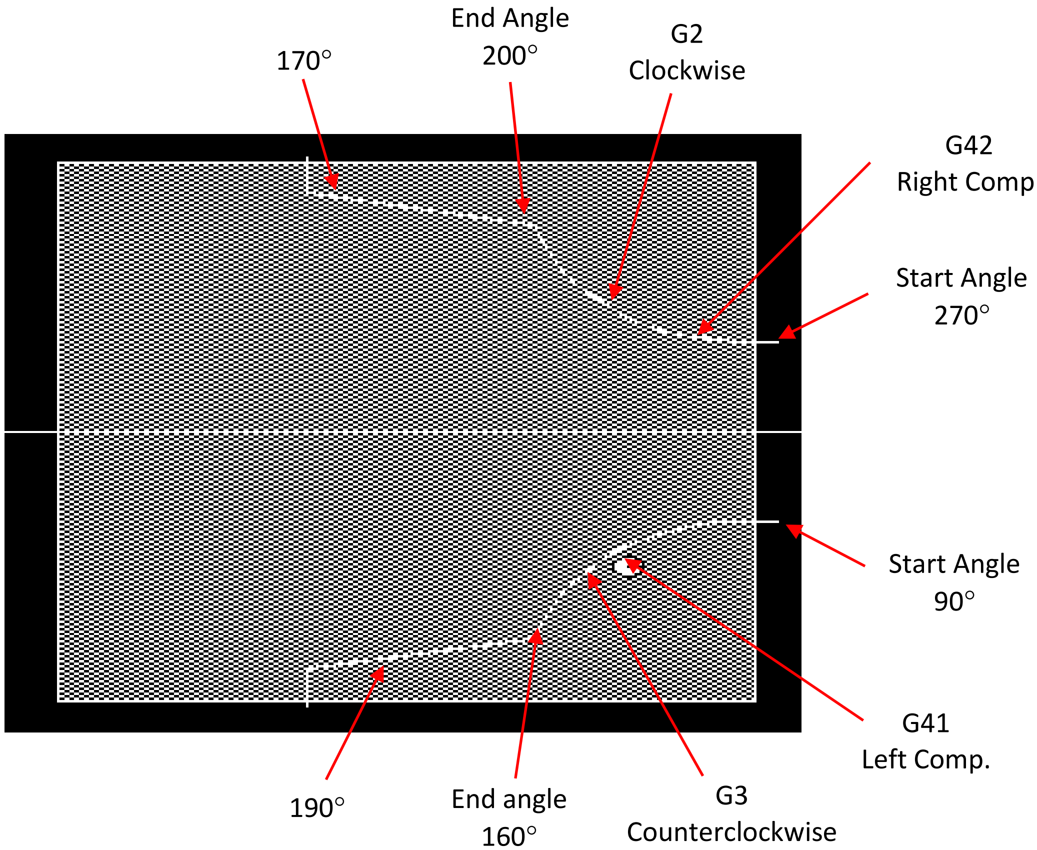
This function will zero out the floating zero's. See G50 in Section Two, page 40 for setting floating zero's.

Back / Front Programming (G13 / G14)

The control can be setup so the X+ is either towards that operator or away from the operator. Machines that have a tool holder on the front side of the spindle are normally configured with X+ being towards the operator and machines that have a tool holder on the back side of the spindle are normally configured with X- being towards the operator. The parameter used to define this orientation is the POWER parameter “Front Turret”. When set to “Yes” the tool is shown coming from the bottom on the graphics screen. It also relates the tool types in the tool table as coming from the bottom. A G13 can be used to specify back side programming or a G14 for front side programming.

When programming from the front verses from the back there are 3 differences:

1) G2/G3 or Clockwise/Counterclockwise arcs
2) G41/G42 or Left/Right cutter comp
3) The polar lines and arcs angles are different



Note that all of the examples in the manual are written using back side programming.

Inch Dimensioning Mode (modal) (G20)

This function will cause the system to go into the inch mode. In this mode the system will accept dimensions in inches. This function can be initiated in a program or in MDI.

Active power up is selectable. See power parameter initial units.

G20 cancels G21.

Metric Dimensioning Mode (modal) (G21)

This function will cause the system to go into the metric mode. In this mode the system will accept dimensions in millimeters (mm). This function can be initiated in MDI or program mode. Feedrate in the metric mode is in millimeters per minute (mmpm), or millimeters per revolution (mmpr).

Notes: The CNC does a conversion from metric to inch, and inch to metric, on all tool offsets. This means that a 1.0 inch offset entered in the inch mode will change to 25.4 mm when the system is switched to metric. The opposite happens when switching from metric to inch mode.

Safe Zone Off/On (G22, G23)

This control is equipped with a programmable safe zone. Any area of the machine's travels can be designated as a safe zone. This is an area of the travels the machine cannot enter. If the machine is programmed into this area when the safe zone check is enabled, an “attempt to move into safe zone” error will be displayed. The safe zone is defined in the coordinate parameters (Refer to Section 10, Appendix on parameters). The check is turned off with a G22 and on with a G23. The normal state of this check is off. Active on power-up is selectable.

Reference Point Return (G28, G29, G30)

These commands allow the machine to be commanded to a fixed point (reference point) by first passing through an intermediate point on the way to the reference point. First a fixed reference point in XZ is entered via the front panel into the reference point parameters. The reference point is relative to machine zero. Once this point is established it will remain unchanged until changed by another front panel command. Each time a G28 or G30 is commanded it will return the machine to the designated reference point. Positioning to the intermediate end reference points are done in rapid traverse. If a G28 or G30 is executed with no axis definitions it has no effect. If only one axis is commanded as an intermediate point, only the commanded axis will move to its intermediate point and reference point. Once an intermediate point is programmed it will be remembered until the next G28 is executed (i.e. for use in a G29).

The command format is as follows:

Reference point set at X-10 Z-0.1

G28	(no movement)
X1 Z-2	(X1 Z-2)
G28 X3	(X3 Z-2 then X-10 relative to machine zero)
X-3 Z-8	
G28 Z-7	(X-3 Z-7 then Z-0.1 relative to machine zero)
G29	(Z to -7)

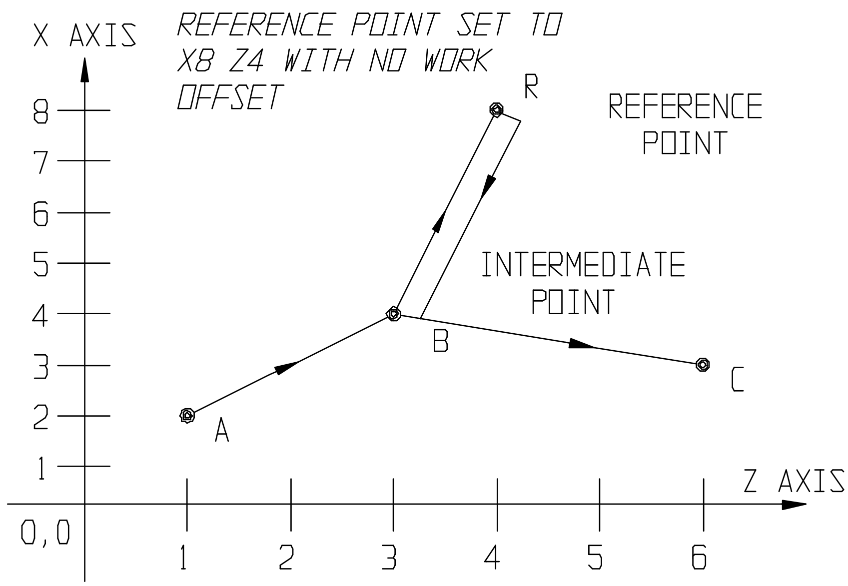
The G29 command is just the converse of a G28. The G29 will return the machine to the programmed point via the last intermediate point stored by a G28 command. The command format is as follows:

G29 X____ Z____
programmed point

In general G28 is used for a tool change or part loading/unloading position. It is normally used to move to the reference point only. Example: G91 G28 Z0 or G28 W0, Z moves to reference point.

When G29 is executed by itself, only the axis that was commanded with the previous G28/G30 goes to the intermediate point.

Example of G28 and G29



X2 Z1	Point A
G28 X4 Z3	Point B then Point R
G29 X3 Z6	Point B then Point C

G30 2nd, 3rd, 4th Reference Point Return

This function works in an identical manner to the G28 reference point return except that a 2nd, 3rd and 4th reference point can be called. The command format is as follows:

G30 P2 X____ Z____
G30 P3 X____ Z____
G30 P4 X____ Z____
reference point / intermediate point

In no P is specified, P2 is assumed. P$\leq$1 and P$\geq$5 are illegal and will result in an error 542 "*G30 illegal return to reference parameter on G30 block*".

Threading (G32, G34-G36)

G32 and G34 through G36 codes on a block with an axis move will force a threading cut to be made. Threading cuts wait for the spindle marker, then move at the programmed feed per revolution feedrate. The axis move can be in the G1 (linear mode) or G2/G3 (circular mode).

Return to Initial Level or to R level (G38/G39)

These two G codes are only used when the control is in one of the Drilling, Tapping or Grooving cycles (G74, G75/G81 thru G89). A G38 or G39 can be executed anywhere in a program or sub-routine. A G38 will cause a canned cycle to return the axis to the same level it was at when the cycle was activated. A G39 will cause a canned cycle to return the axis to the clearance.

```
G0 X0 Z1
G39
G81 Z-1 R.1 F.01 < Returns to Z.1

G0 X0 Z1
G38
G81 Z-1 R.1 F.01 < Returns to Z1
```

Cutter Compensation (G40, G41, G42)

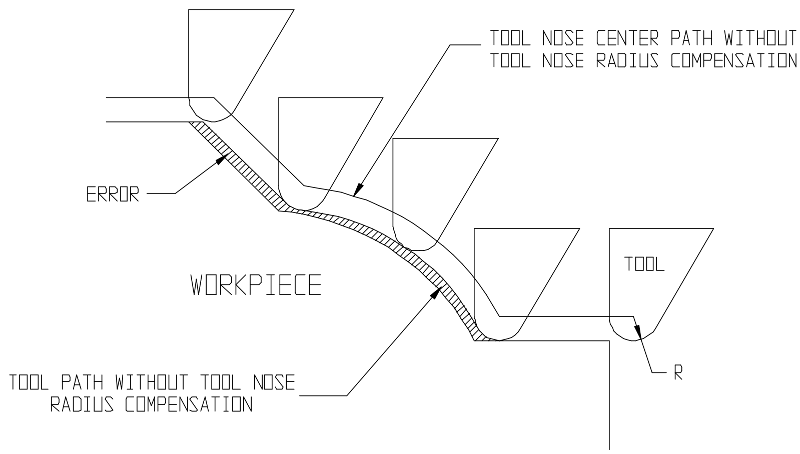
G40	Compensation Off
G41	Left Compensation
G42	Right Compensation

This section will explain how the cutter compensation works and give pointers on how to use it optimally.

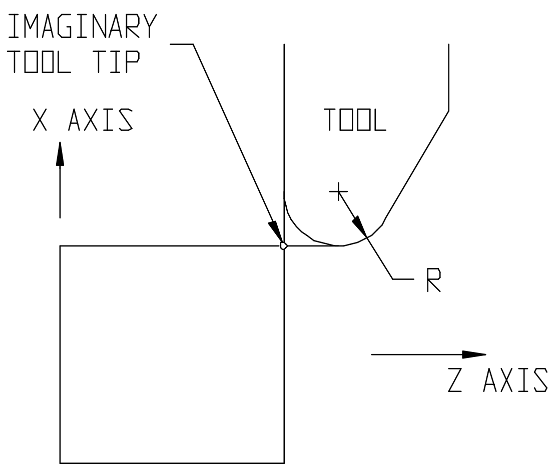
Cutter compensation is the displacement of the tool path, perpendicular to the programmed path, by the amount equal to the cutter radius. The programmed path can be figured by the programmer for a zero tool radius.

If the parts program is written for a zero tool radius, i.e., directly off the print, then by entering the actual tool radius into the system and activating cutter compensation the operator can make the control calculate the displaced path.

It is important to understand the various tool types to understand how cutter compensation works. The various tool types are described on the following pages.

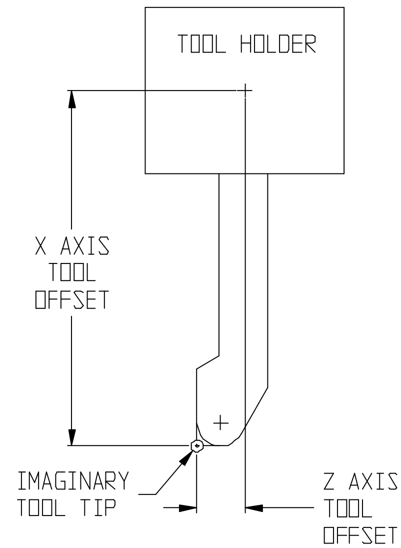


The above illustration shows the path taken by the tool without tool nose radius compensation. The imaginary tool tip follows the programmed path.

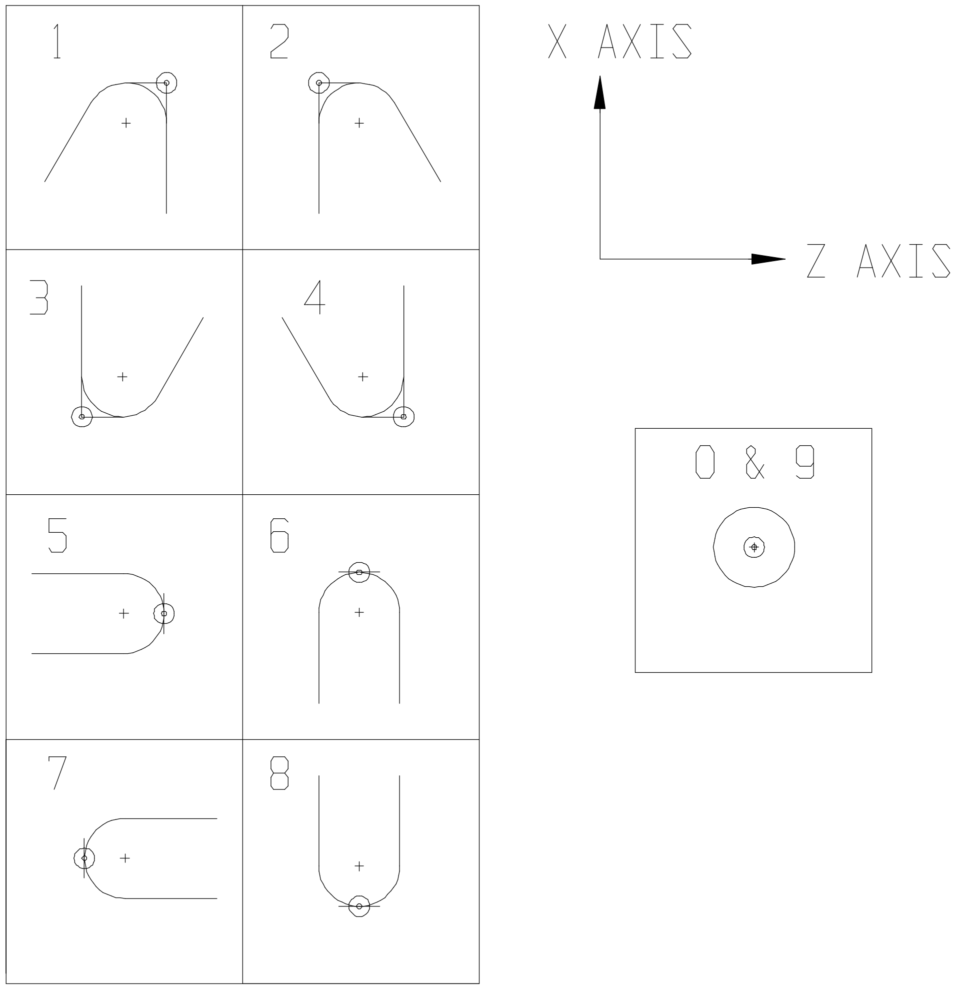


The imaginary tool tip does not actually exist. When position moves are commanded without tool nose compensation, the imaginary tool tip is positioned at the commanded position. The imaginary tool tip is defined by the tool length offsets and tool type specified.

The direction of the imaginary tool tip is defined by the type of tool being used. This direction is defined by the tool type in the tool table.



The 21 different tool types.



Note: The above tool displays are for systems with the front turret parameter set to no.

Below is a partial list of tools used for each type:

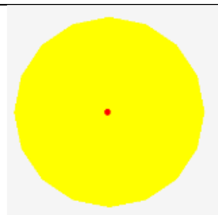
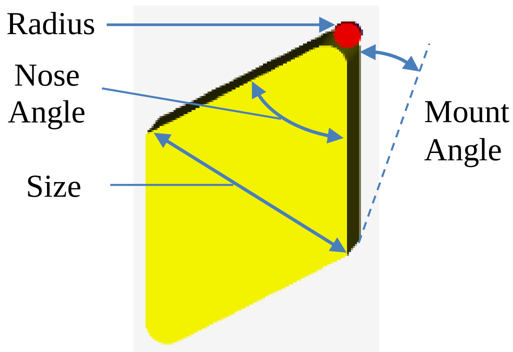
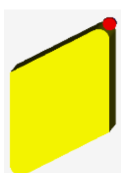
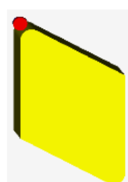
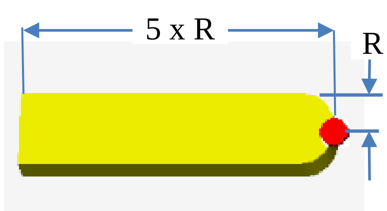
0. Tool Type #0
   *
   *
   *
1. Tool Type #1
   * Inside Turning (chuck toward tail stock)
   *
   *
2. Tool Type #2
   * Inside Turning (tail stock towards chuck)
   *
   *

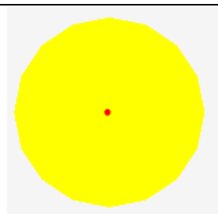
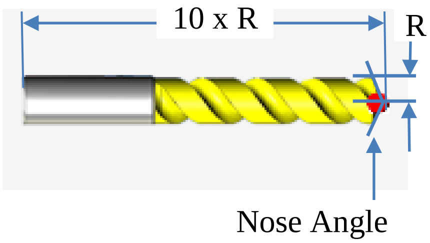
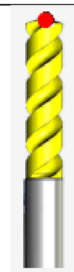
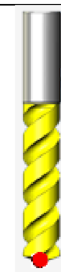
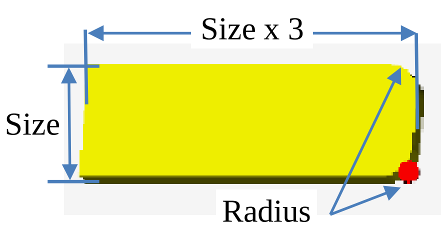
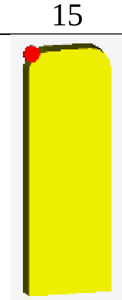
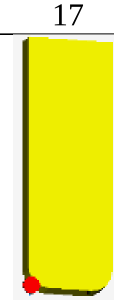
3. Tool Type #3
   * Turning (tail stock towards chuck)
   *
   *
4. Tool Type #4
   * Turning (chuck towards tail stock)
   *
   *
5. Tool Type #5
   *
   *
   *
6. Tool Type #6
   * Inside Turning Using a Diamond Insert
   * Boring Tools
   *
7. Tool Type #7
   * Tools for Face Grooving
   * Facing Tools
   *
8. Tool Type #8
   * Turning Using Diamond Insert
   * Cut-off Tools
   * Tools for Diameter Grooving
9. Tool Type #9
   *
   *
   *

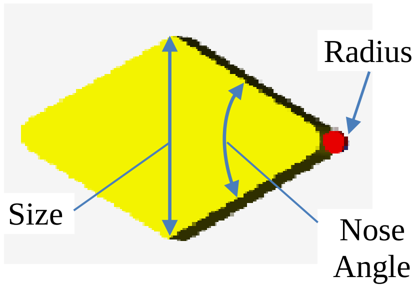
When tools are called using the "T" code, the format used is T####. The first two digits specify the Tool #; the last two digits specify the tool length offsets, tool tip radius and the tool type.

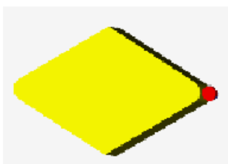
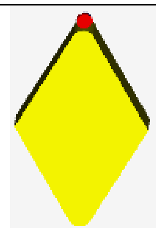
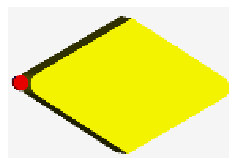
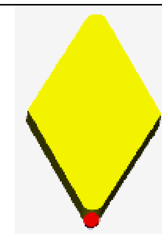
The following additional tool types are available. The types are similar to types 1 thru 8 as far as imaginary tool tip configurations, how tool lengths are set and how cutter comp is addressed. The additional tool types differ in how solid model cuts are shown

Tool Types Defined

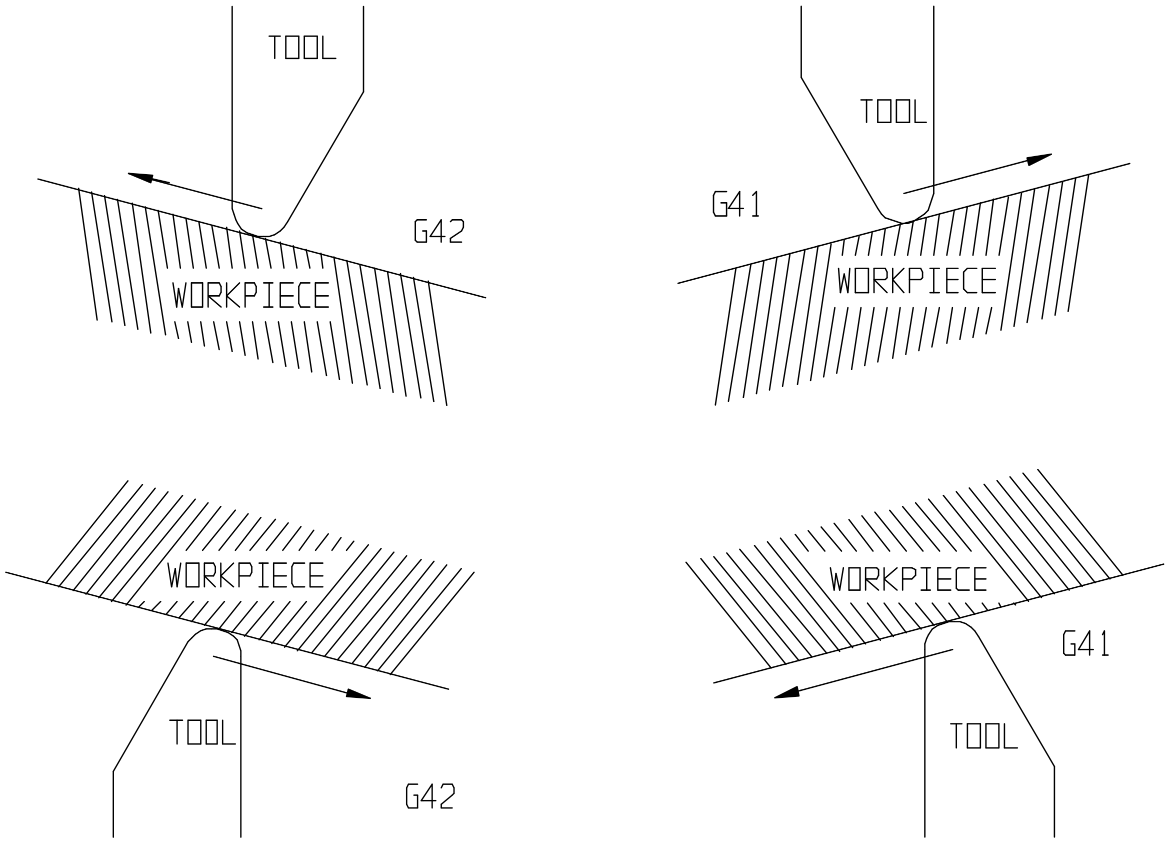
<table>
<tr><th>Tool Type</th><th colspan="4">Description</th></tr>
<tr><td>0 & 9</td><td></td><td colspan="3">Tool Type #0 & 9
* Generic tools</td></tr>
<tr><td rowspan="4">1-4</td><td colspan="2">
</td><td colspan="2">* Tool types 1-4 are turning tools. Variations are shown below.</td></tr>
<tr><td>1</td><td>2</td><td>3</td><td>4</td></tr>
<tr><td></td><td></td><td></td><td></td></tr>
<tr><td>* Inside Turning (chuck toward tail stock)</td><td>* Inside Turning (tail stock towards chuck)</td><td>* Turning (tail stock towards chuck)</td><td>* Turning (chuck towards tail stock)</td></tr>
<tr><td rowspan="4">5-8</td><td colspan="2">
</td><td colspan="2">Tool types 5-8 are multi-use. Variations are shown below.</td></tr>
<tr><td>5</td><td>6</td><td>7</td><td>8</td></tr>
<tr><td></td><td></td><td></td><td></td></tr>
<tr><td>Back side face grooving</td><td>Inside turning, boring.</td><td>Facing, face grooving</td><td>Turning and cut-off, tools.</td></tr>
</table>

<table>
<tr><th>Tool Type</th><th colspan="4">Description</th></tr>
<tr><td>9</td><td colspan="2"></td><td colspan="2">Same as tool 0</td></tr>
<tr><td rowspan="4">10-13</td><td colspan="2"></td><td colspan="2">Tool types 10-13 are drills for various directions as shown below.
Drills do not show circular cuts. For live tooling they show only the hole.</td></tr>
<tr><td>10</td><td>11</td><td>12</td><td>13</td></tr>
<tr><td></td><td></td><td></td><td></td></tr>
<tr><td>Towards tail stock</td><td>Away from operator</td><td>Towards chuck</td><td>Towards operator</td></tr>
<tr><td rowspan="5">14-17</td><td colspan="2"></td><td colspan="2">Tool Types 14-17 are grooving tools used in various directions as described below.</td></tr>
<tr><td>14</td><td>15</td><td>16</td><td>17</td></tr>
<tr><td></td><td></td><td></td><td></td></tr>
<tr><td>Towards tail stock
* Facing cycles</td><td>Internal diameter cycles</td><td>Towards chuck
* Facing cycles</td><td>toward operator
* Diameter cycles</td></tr>
<tr><td colspan="4">* = “Size” is considered only for Groove II cycle</td></tr>
</table>

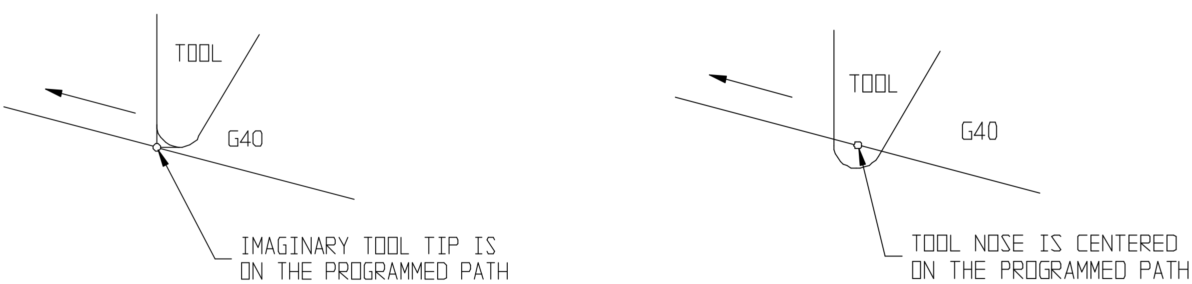
<table>
<tr><th>Tool Type</th><th colspan="4">Description</th></tr>
<tr><td rowspan="4">18-21</td><td colspan="4">

Threading tools in various directions as described below.
Cuts are shown as threads where the lead is the programmed feedrate.</td></tr>
<tr><td>18</td><td>19</td><td>20</td><td>21</td></tr>
<tr><td></td><td></td><td></td><td></td></tr>
<tr><td>Threading tool on face towards chuck</td><td>Internal threading</td><td>Threading tool on face toward tailstock</td><td>Threading tool on diameter towards operator.</td></tr>
</table>

G40 Cancel tool nose compensation
G41 Tool nose radius compensation on, moving on the left side of the programmed path.
G42 Tool nose radius compensation on, moving on the right side of the programmed path.



For Tool Types 1-8: *For Tool Types 0 & 9:*



The block that turns cutter comp on (to G41 or G42) from cutter comp off (G40) is the start-up block. Special care should be taken to assure that the start block positions the tool to make the subsequent cut.

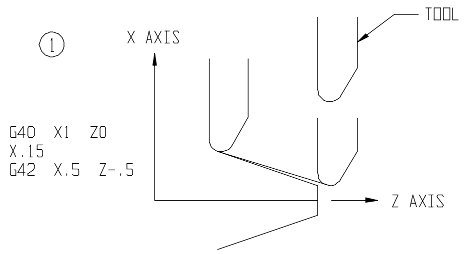


Figure 1 shows the cutter comp is not active until the G42 X.5 Z-.5 block is completed.

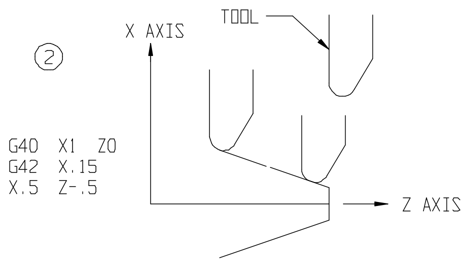


Figure 2 shows a problem on the G42 X.15 block. The desired position at the face of the part is not obtained.

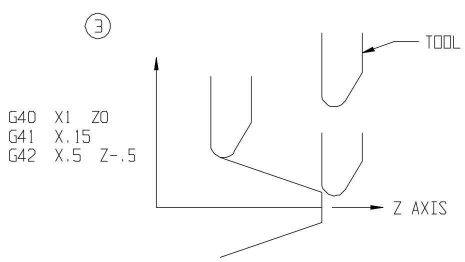
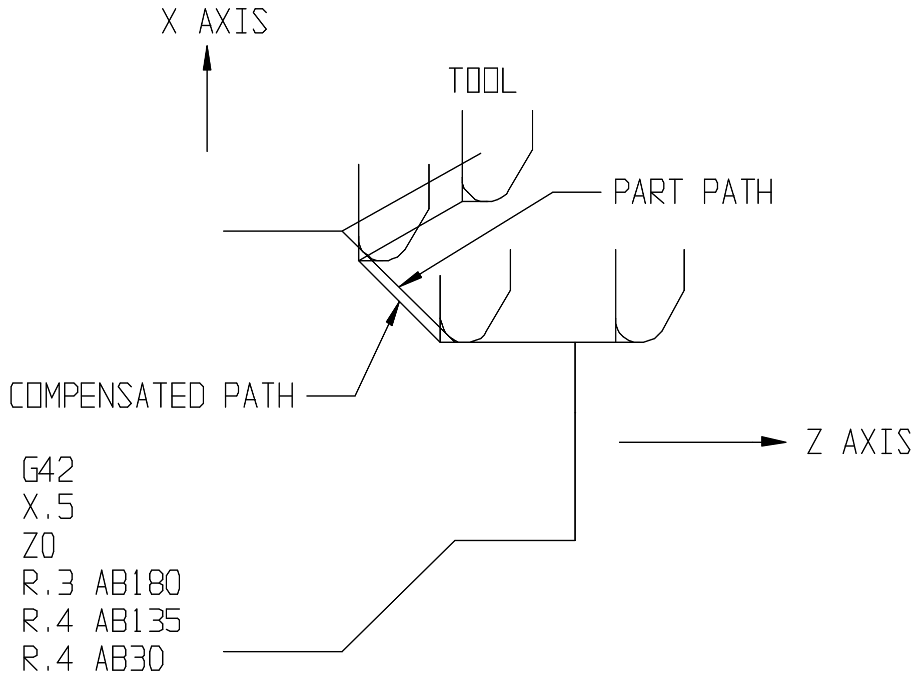
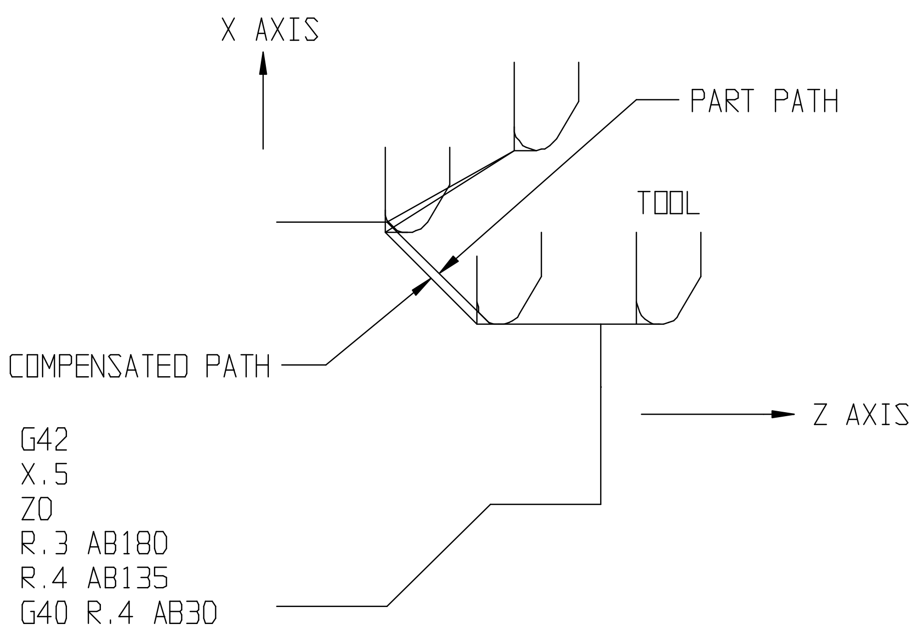


Figure 3 shows an approach that obtains the desired part.

The block that turns tool nose radius compensation off (from G41 or G42 to G40) is called the offset cancel block. The imaginary tool tip will be positioned at the end of the offset cancel block.





In many cases retracting perpendicular to the part will obtain accurate cutting.

Throughout the program the control keeps a record of the previous programmed point, the current programmed point, and the next programmed point along the tool path.

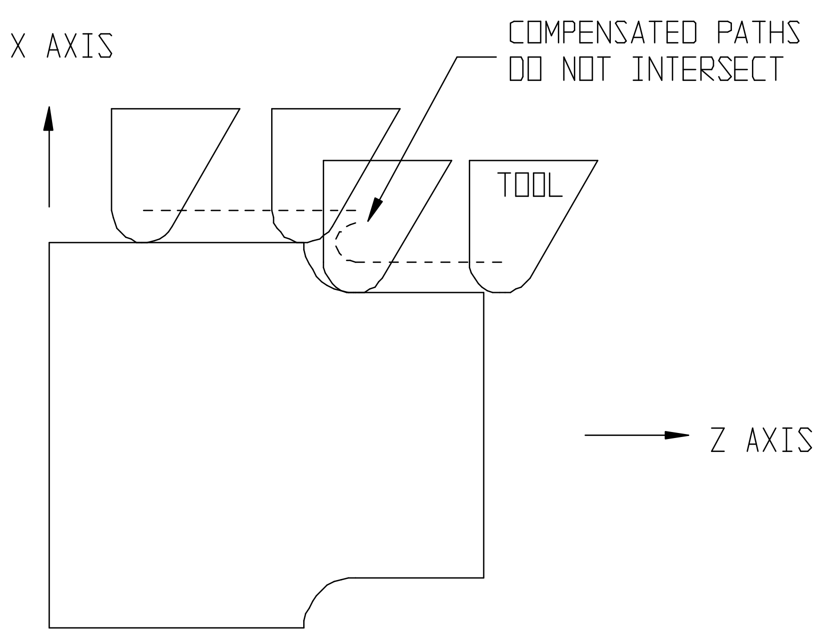
With three points, information on how they are connected, the cutter radius, and whether it is a left or a right compensation, the control can calculate the current compensated point. The control will also employ its Trig Help function discussed earlier to connect lines and arcs during cutter compensation. (See section on Trig Help beginning on page 6-9.)

After each successful calculation of a compensated point the current programmed point becomes the previous programmed point, the next programmed point becomes the current programmed point and a new programmed point is read up to become the next programmed point. This mechanism is repeated over and over again until the end of the program is reached. This sequence should be understood clearly in order to understand many points that will come up later on how the compensation works.

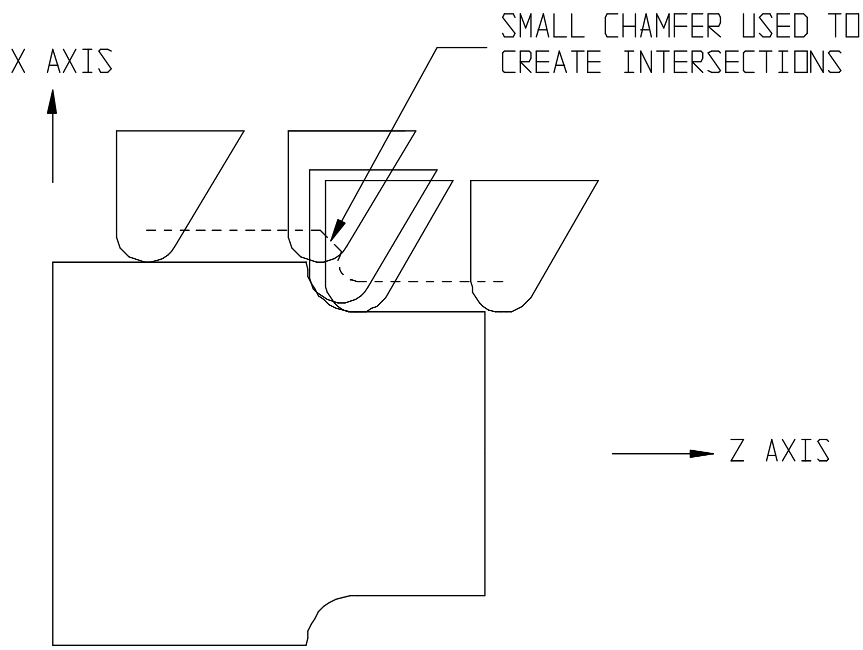
The compensation in this control is truly intersectional. Given these three points, the control calculates the intersection of the compensated path between the previous and the current programmed points and the compensated path between the current and the next programmed points. These paths can be a mixture of straight lines and arcs.

Because of the intersectional nature of the compensation package, there has to be an intersection of all the displaced paths for the system to work. If there is no intersection between two paths the control will give an error.

Explanation of How Displaced Tool Paths
Cannot Have an Intersection



The solution of the above part is to introduce a 00.0001" chamfer at the point between the non-intersecting surfaces.



Non-Movement (G65)

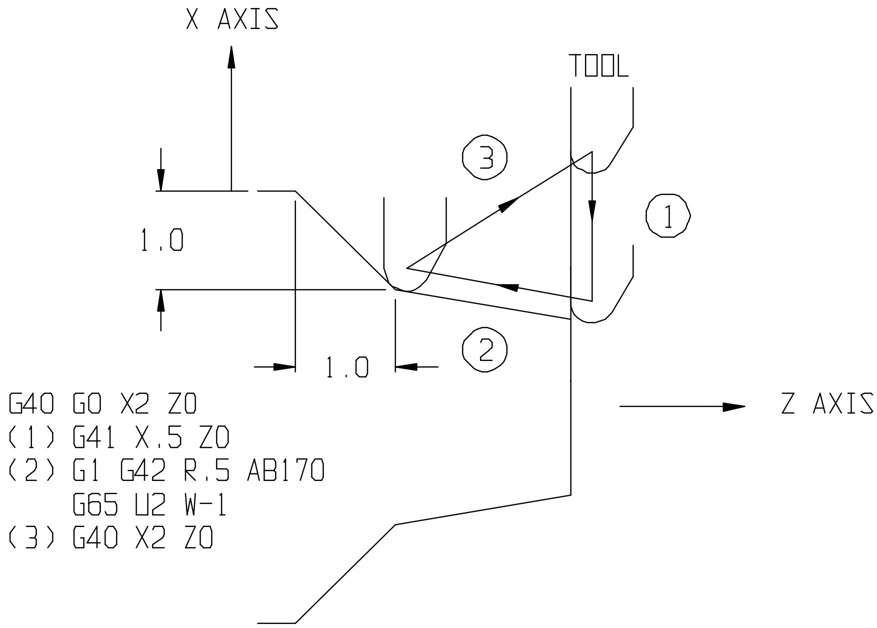
Starting and Ending Cutter Compensation

The G65 code placed on a line with coordinates will cause these coordinates to be used for cutter compensation points but skipped during machine movement.

G65 X___ Z___ Machine will not move to the XZ coordinates

The G65 will allow the programmer to turn cutter compensation on and get the tool to start cutting or retract at a specific point without doing any extra moves.

Lead in or lead outs can be obtained using the G65 non-move block to obtain cuts where other surfaces must be avoided.



G65 can also be used for calling a program. The program to be called is specified by the value of P.

Example: G65 P4371 (calls program #4371)

To pass arguments to the program, other addresses can be added to the block.

Example: G65 P1402 A500 (calls program #1402 and sets parameter #1 to 500 and parameter #16 to 1402)

The addresses refer to the parameters as follows:

Address	Parameter #
A	1
B	2
C	3
.	.
.	.
X	24
Y	25
Z	26

Notes on G65 P####

Note 1: If the program specified by address P does not exist an error will occur.

Note 2: The program called is the rounded value of address P. Example: G65 P12.75 calls program #13

Note 3: If no P is in the G65 block, the block is treated as a non-movement block.

Note 4: Addresses not specified are set to -999.

Notes on Cutter Compensation

Note 1: Turning compensation on can be done both in a block with no axis move, or in a block containing axes moves.

Note 2: There is no restriction on how many successive blocks can have no axis information.

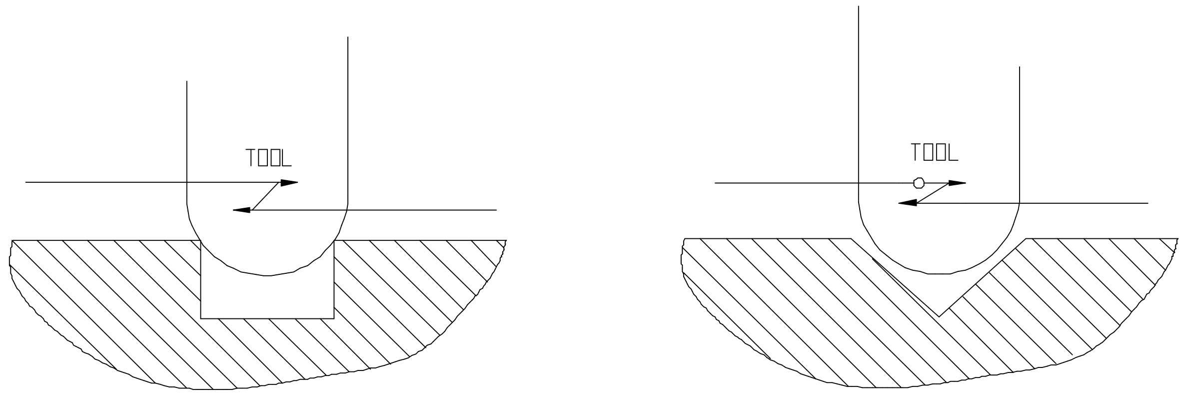
Note 3: Given the correct centers and radii for two intersecting circles, the system automatically checks and corrects the programmed point of intersection, i.e. Trig Help.

Note 4: In general, when using cutter compensation, no feature on the part can be smaller than the tool radius. This includes such things as slots, arcs and vees. If a part contains such features they should be replaced by straight lines, cutter compensation should be turned off, or a smaller tool should be used.

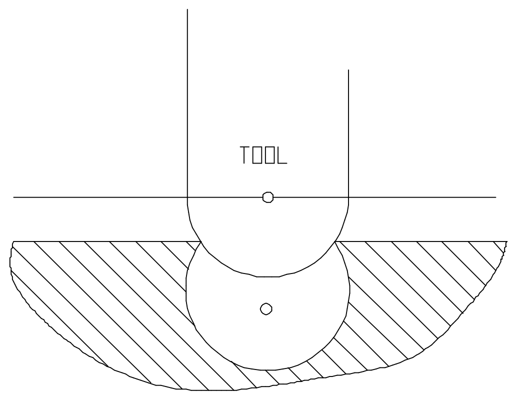
Note 5: Cutter compensation can be shut off by setting bit 4 of the special flag parameter.

Note 6: A G41 with a negative tool tip radius is the same as a G42 with a positive tool radius.

Note 7: A G42 with a negative tool tip radius is the same as a G41 with a positive tool radius.



In the above cases the tool will back up as it tries to place itself tangent to the walls of the slots or V.



This case will give a *"Compensated Line/Arc Do Not Intersect" error.*

Z to Clearance (G45)

The G45 function will retract Z to the clearance position. This position defaults to the last clearance position but may be changed by editing parameter 140 or set in drilling cycles with the "R" parameter.

X to Clearance (G46)

The G46 function will retract X to the clearance position. This position defaults to the last clearance position but may be changed by editing parameter 139 or set in canned cycles.

Z to Tool Change (G47)

The G47 function will retract Z to the tool change position. This position is set by the machine tool builder but may be changed by editing the tool change coordinate parameters.

X to Tool Change (G48)

The G48 function will retract X to the tool change position. This position is set by the machine tool builder but may be changed by editing the tool change coordinate parameters.

Tool Length Offsets, Cancel (G49)

Tool length offsets are zeroed by using the G49 command.

The last two digits of the tool code ("T") are used to load the offsets. T0315 loads tool #3 and offsets (X length, Z length, tool nose radius, tool type, X wear length, Z wear length and tool nose wear radius) from the tool table for tool #15.

* *Offsets for tool 0 are all zero.*

* *Legal tool #'s and offset #'s are 0 thru 99.*

Coordinate Systems

The machine zero is a fixed point on the machine. The machine zero point is normally decided by the machine tool builder and set by a limit switch and encoder marker pulse on each axis.

Work offsets are normally used for Z axis only. Tool offsets are used for offsetting X axis.

The machine zero point is established when the home command is first executed.

Once the machine zero point is established it is not changed by reset, coordinate system call (G54-G59), coordinate system shift (G50) or local coordinate system setting (G52).

Software limits are set from the machine zero point.

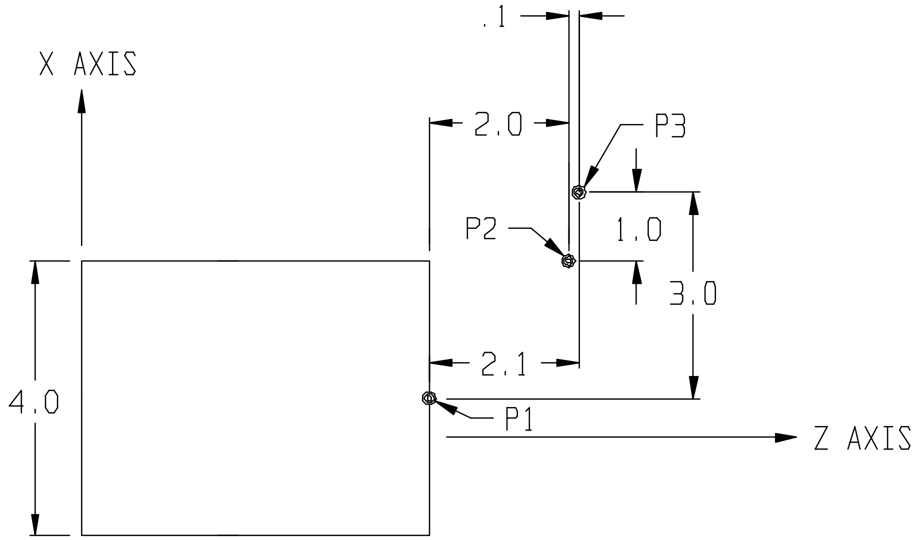
Floating Zero (G50)

G50 is used to establish a work coordinate system. It is also used to set a maximum spindle speed (to clamp the spindle RPM used with constant surface speed).

G50 X___ Z___ creates a work coordinate system with the 0,0 at the specified distance from the tool tip. The X value is a diameter value.

Example 1: Move the tool tip to the front edge of a 3" diameter bar, then command G50, X3, Z0. Then, regardless of the displayed position, a new work coordinate system is established where the center of the part is X0 and the face of the part is Z0.

Example 2:



If the machine is positioned at P2, which is a command of X4 Z2, and then a G50 X0 Z0 is commanded, the next time X2 Z.1 is commanded the machine will position to P3; or if the machine is positioned at P1 and G50 X-4 Z-2 is commanded, the next time X2 Z.1 is commanded the machine will position to P3.

Note: 1. *G50 should not be commanded when cutter compensation is active; the control should always be in G40 mode.*

2. *The distance shifted via a G50 in one work coordinate system will be applied to other work coordinate systems when they are activated via G54 - G59 commands. G50 offsets are zeroed on power up and after homing the machine.*

3. *On power up the G54 coordinate system is active.*

4. *See the section on G96 (constant surface speed) for information on using G50 to set the maximum spindle speed.*

5: *The chuck can be assumed Z0, but all examples in the manual assume the face of the part as Z0.*

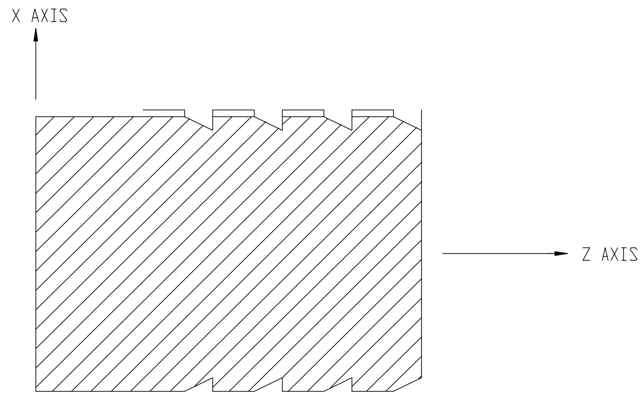
6: *G50's within a program are active during the program. However, when a program ends, the work offset that was active when the program started will be restored.*

7: *G50's within subprograms or subroutines will be active in the subprograms or subroutines only. When a subprogram or subroutine ends (and returns to the calling program) the G50's are restored to the values that were active when the subroutine was called.*

8: *G50 X___ Z___ can be thought of as call this position X___ Z___.*

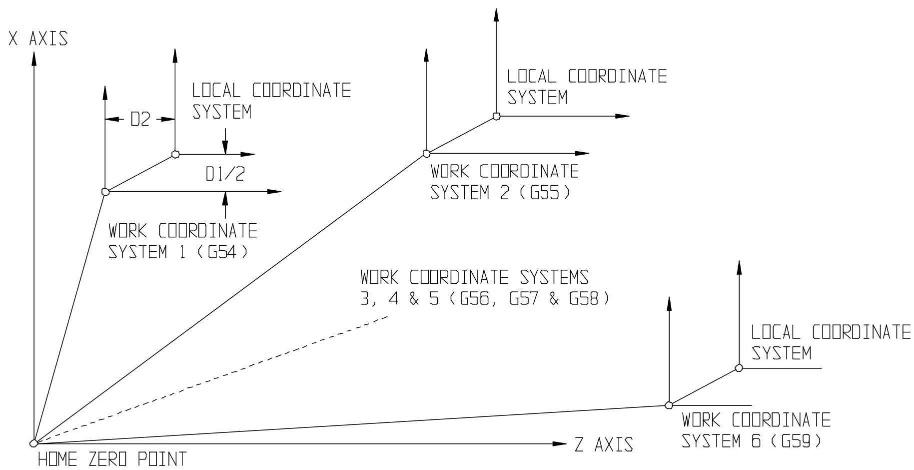
Example:

```
G0 X2.1 Z0
GOSUB 1 L4
M30
N1
G50 Z0
G1 X1.8
X2 Z-.2
G0 X2.1
Z-.5
Return
```



Local Coordinate System (G52)

The G52 command is similar to the G50 command except that it uses the current coordinate system zero as its reference point instead of the current machine position.

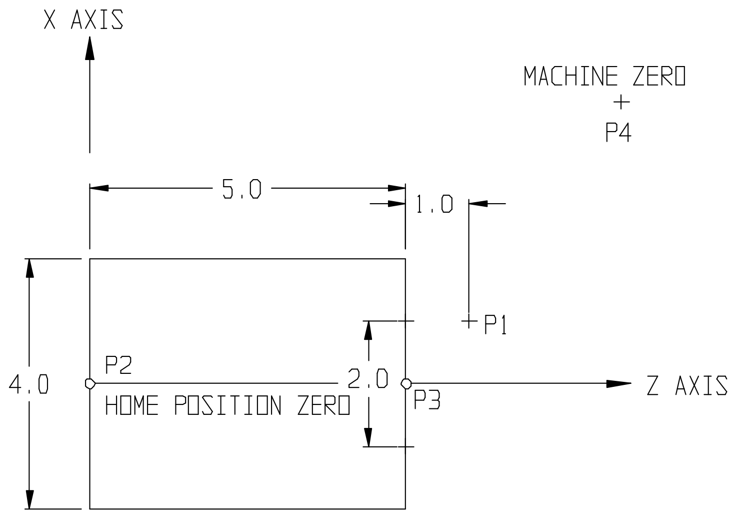


Notes:

1. *A G52 is modal; therefore it will affect all coordinate systems once set.*
2. *G52 X___ Z___ can be thought of as shift the work coordinates by X___ Z___.*
3. *To cancel a G52, enter G52 X0 Z0.*
4. *G52 offsets are not affected by the position of the machine. G50 offsets are affected by the position of the machine.*
5. *G52 offsets are zeroed on power up, after homing, after any G50 command, and at the start of any program.*

Machine Coordinate System (G53)

A G53 code preceding any XZ move will cause those dimensions to be relative to the machine zero point.



When no offsets are active:

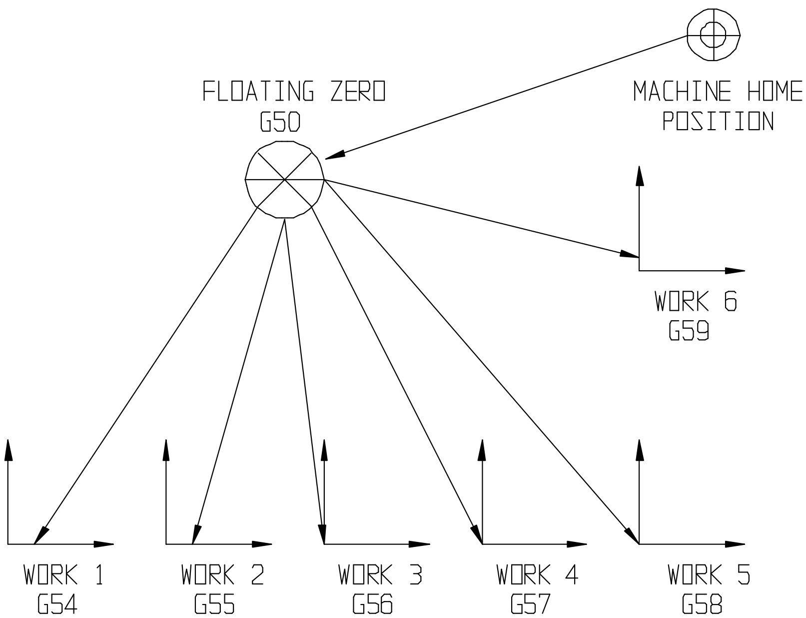
X0 Z0 move to P3
X2 Z1 move to P1
X0 Z-5 move to P2
G53 X0 Z0 move to P4

A coordinate system used to align the work part dimensions to the machine's programs is called a work coordinate system. The work coordinate system is set by either of the following methods:

1. using a G50 command
2. using a G52 command
3. using G54 - G59 commands.

Work Coordinate Systems (G54 - G59)

These work coordinate systems are set via the keyboard. The dimensions of the coordinate zero point are always relative to the G50 Floating Zero point. To set a work coordinate system, select F7 (Paramtrs) on the main menu, then F2 (Coords). The work coordinate menu will come up allowing you to type in the offset coordinates for each coordinate system. The "Home Position" offsets are parameters which shift all coordinate systems relative to the Machine Zero Point.



G55 X1 Z1	moves to X1 Z1 in work offset 2
G59 X1 Z1	moves to X1 Z1 in work offset 6
G54	is always the power on coordinate system.

Note #1: *G54-G59 offsets are not zeroed on power up or after homing.*

Note #2: *See Section Four, page(s) 4-4 for a procedure for setting work and tool length offsets.*

Single direction or one shot rapid positioning (G60)

G60 is a one-shot G code and is used in place of G00.

For accurate positioning without backlash, positioning from one direction is available.

G60 X___ Z___

Notes:

1. *The amount of overrun is preset by the machine tool builder.*
2. *Overrun direction is not affected by mirror imaging.*
3. *If "G00 unidirectional approach" was set by the machine tool builder, the same positioning sequence would happen with each G00 move.*

Exact Stop Mode (modal) (G61)

When G61 is commanded, deceleration is applied to the end point of the cutting block and in-position is performed per block thereafter. G61 is valid until G63 (tapping mode) or G64 (cutting mode) is commanded.

Tapping Mode (modal) (G63)

When G63 is commanded, feedrate override and spindle speed override are ignored (always regarded as 100%), block mode and feedhold become invalid. G63 is valid until G61 (exact stop mode) or G64 (cutting mode) is commanded.

Cutting Mode (modal) (G64)

G64 is the default at the beginning of each program. When G64 is commanded, deceleration based on the angle between blocks at the end point of each block thereafter is performed, and cutting goes on to the next block. This command is valid until G61 (exact stop mode) or G63 (tapping mode) is commanded.

Finish Cycle (G70)

After rough cutting with the G71 rough turning cycle, G72 rough facing cycle or G73 pattern repeat cycle, the finish cycle can be used to remove the finish stock.

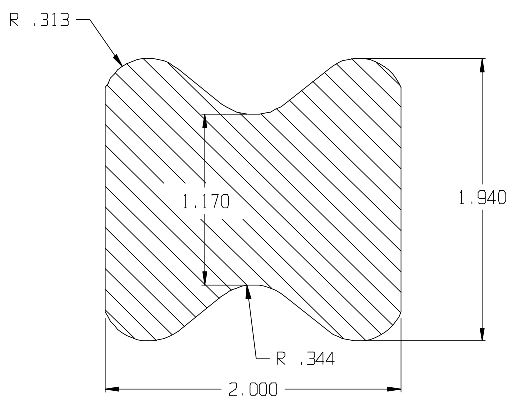
Code used for Finish Cycle:

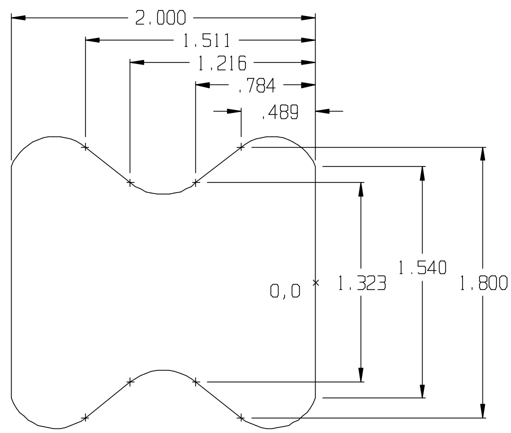
G0 X##.#### Z##.####

G70 P#### Q####

1. (X,Z) Start Point: The start point of the Finish Cycle.
2. (P) Start Block: Designate the starting block of the Finish Cycle.
3. (Q) End Block: Designates the ending block of the Finish Cycle.

Example Program:

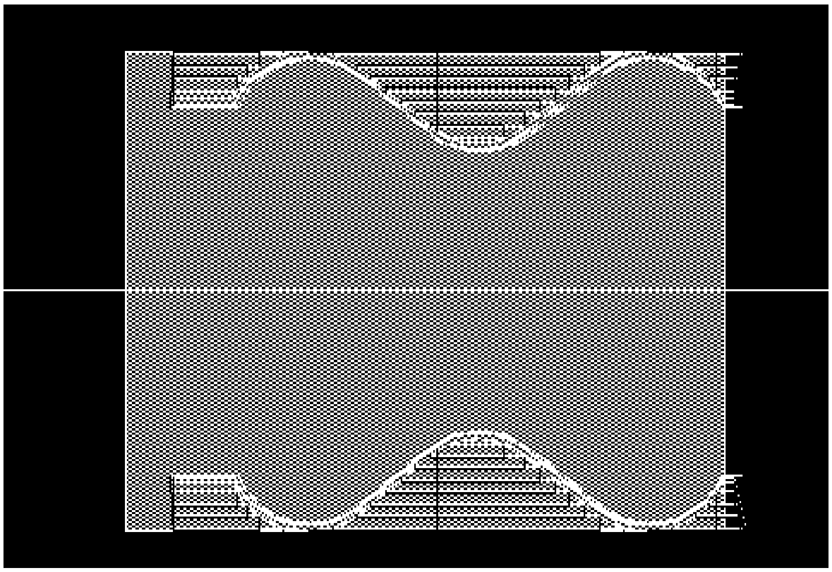




Example program using a rough turning cycle with a finish cycle for the part shown above:

```
T0101
G99 F.01
G50 S2000
G96 S600
M3
M8
G0 X2.1 Z.1
P117=0
G71 P1 Q2 U.015 W0 I.025 K0 R0 V1.975 D.05 (Rough Turning Cycle)
N1  (Start Block)

G42
G0 X1.5394
G1 Z0
G3 R.313 Z-.489 X1.8004       .
G1 X1.3234 Z-.7835            .
G2 R.344 Z-1.2165 X1.3234  (Pattern)
G1 X1.8004 Z-1.511            .
G3 R.313 Z-2 X1.5394          .
G1 R.3 AB180                  .
X2.1
N2  (End Block)
```

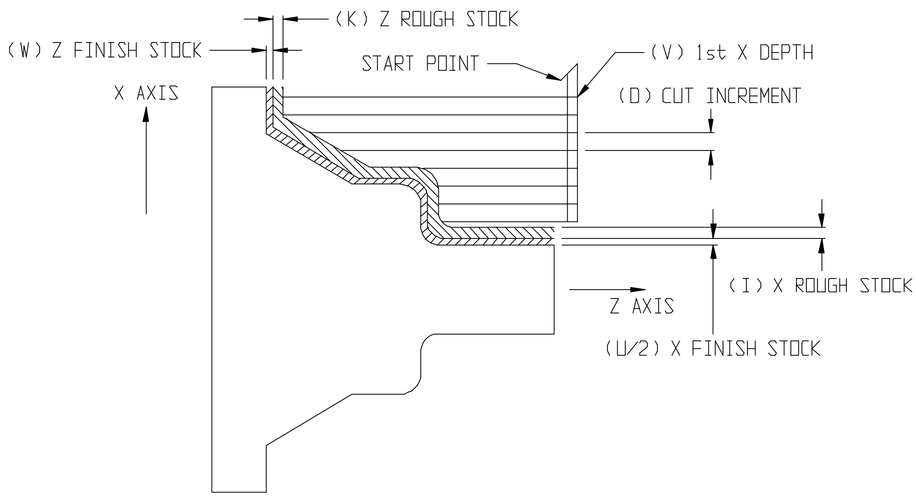
T0202
G99 F.005
G50 S2000
G96 S900
M3
M8
G0 X2.1 Z.1 (Start Point)
G70 P1 Q2 (Finish Cycle)



Rough Turning Cycle (G71)

The rough turning cycle is used to define how to remove the material defined in the geometry between the start block and the end block.

Rough cutting is done by first cutting parallel to the Z axis down to the pattern leaving the rough stock. After cutting parallel to the Z axis is completed, a pass is made along the finish pattern leaving the finish stock.



Code for Rough Turning Cycle:

G0 X##.#### Z##.####
(1) (1)

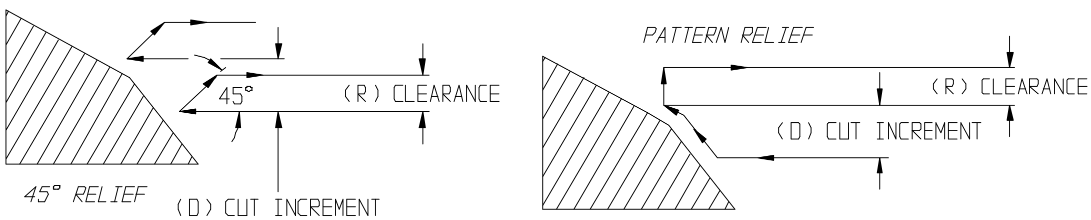
P117=0
(2)
G71 U##.#### W##.### I##.### K##.#### V##.#### D##.## R##.### P#### Q####
(3) (4) (5) (6) (7) (8) (9) (10) (11)

N####
(10)
. Commands between the Start Block and the End Block
. define the shape to rough out.
N####
(11)

1. (X,Z) Start Point:	X and Z return to the start point at the end of the cycle.
2. Relief:	The type of relief made at the end of the cutting. P117=0 (PATTERN) will cut along the pattern of the part, P117=1 (45 DEGS.) will retract at a 45 degree angle before retracting Z for the next cut. If the relief type is 45 DEGS., no pockets are allowed in the pattern.
3. (U) X Finish Stock:	Distance and direction of the finish diameter dimension allowance in the X axis.
4. (W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis.

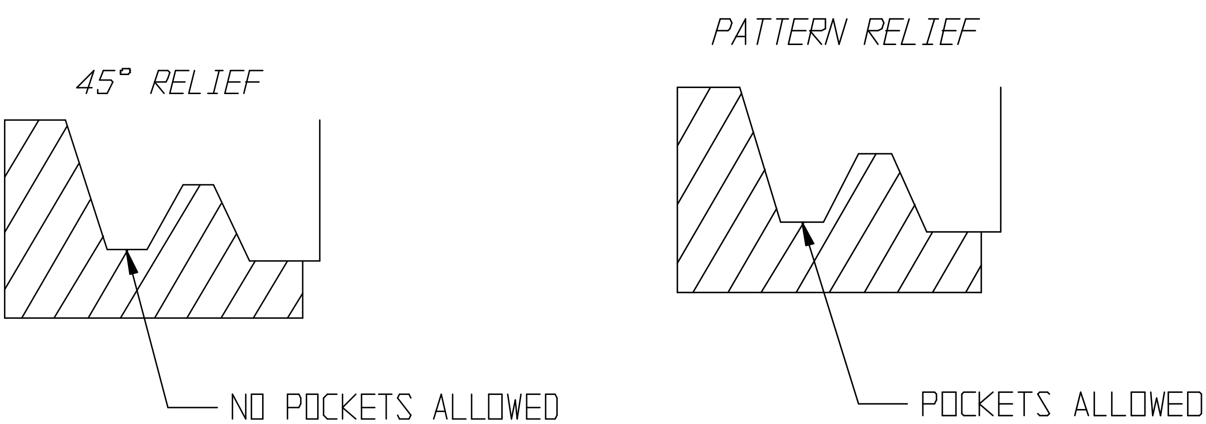
5. (I) X Rough Stock:	Distance and direction of rough finish allowance in the X axis (radius dimension).
6. (K) Z Rough Stock:	Distance and direction of the rough finish allowance in the Z axis.
7. (V) 1st X Depth:	The X depth of the 1st cut in the roughing cycle (diameter dimension). If this is not entered the 1st cut is at (X start point) + (X finish stock/2) + (X rough stock) - (cut increment).
8. (D) Cut Increment:	Depth of cuts designated without sign. The cutting direction depends on the direction of the 1st move in the pattern (radius dimension).
9. (R) Clearance:	Distance of the 45 degree relief or an additional distance to pull away after the pattern relief (radius dimensions).
10. (P) Start Block:	Designates the start of the geometry to rough. (Any number between 1 and 9999.)
11. (Q) End Block:	Designates the end of the geometry to rough. (Any number between 1 and 9999.)

Feed and speeds within the pattern are ignored during the parallel cuts. Feeds and speeds are effective during the pass along the pattern. Rapid moves within the pattern are effective when following the finish pattern.



P117<>0 P117=0

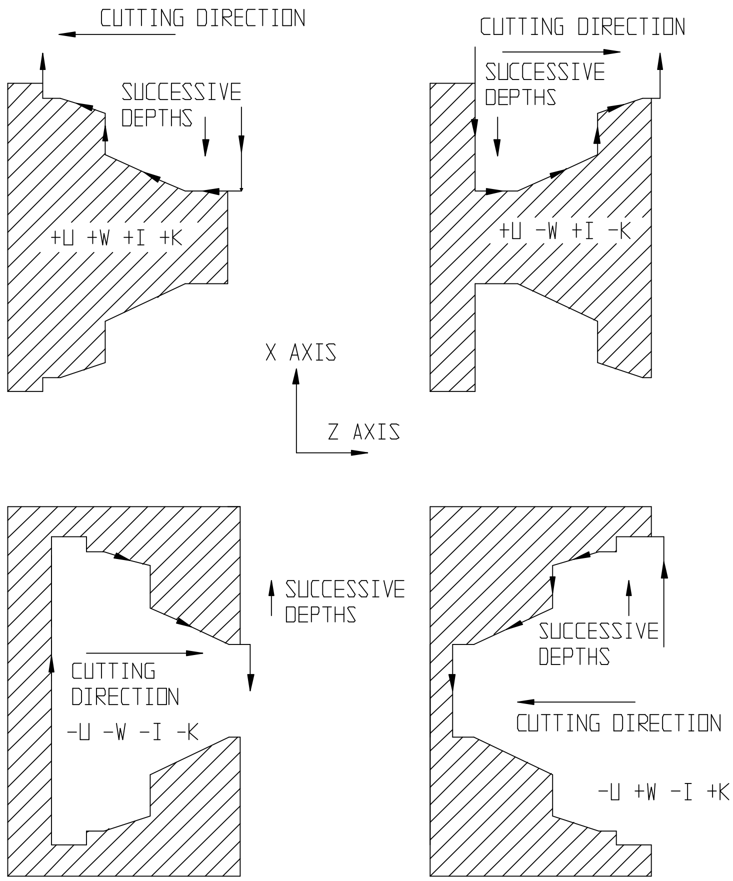
After rough cutting parallel to the Z axis there are two types of relief. 45° relief will rapid away from the pattern at a 45° angle. Pattern relief follows the pattern and then retracts in the X axis. A 0 (zero) clearance is allowed in the pattern relief.



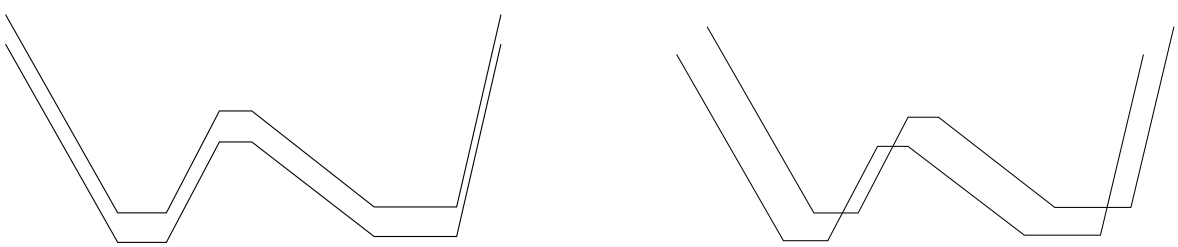
P117<>0 P117=0

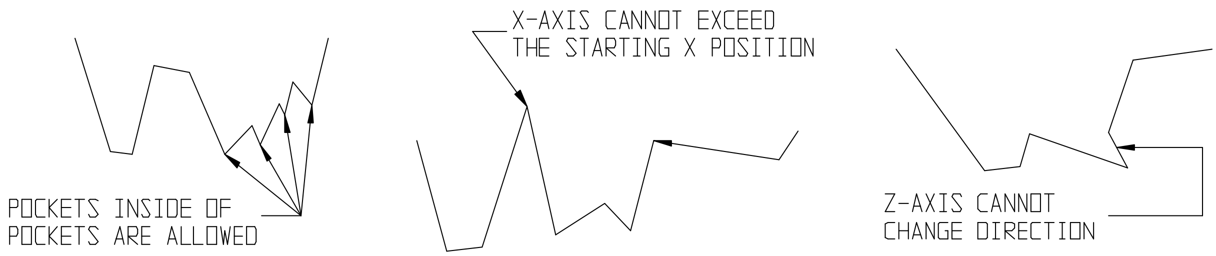
No pockets are allowed when using the 45° relief.

There are four types of patterns made by the rough turning cycle. Rough cutting for each of these four patterns is done parallel to the Z axis. The signs of U, W, I and K are shown for each pattern.



When there are pockets, the Z rough and finish stocks are normally 0. If Z rough and finish stocks other than 0 are specified, it may cut beyond the finish pattern.



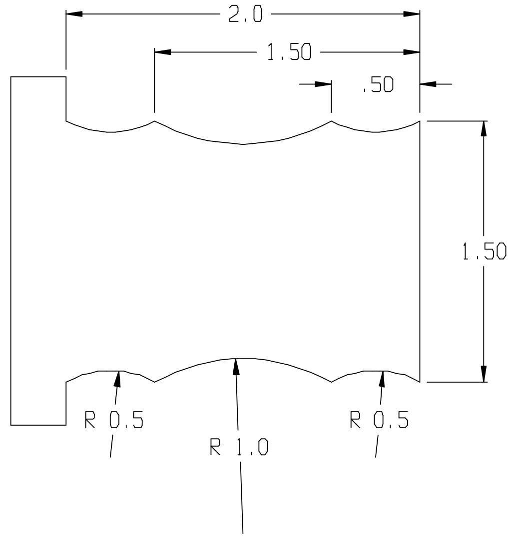


Note 1: *Moves back to the start point from the end of the pattern are rapid moves.*

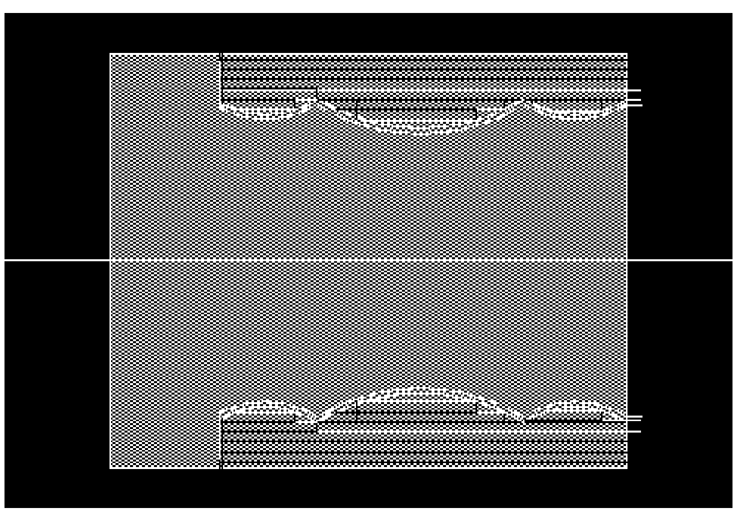
Note 2: *If the X end point in the pattern is not equal to the X start point of the pattern, a shoulder to the X start point will be added.*

Note 3: *If the X rough stock and the Z rough stock are both zero, the pass made along the finish pattern is skipped.*

Example of Rough Turning Program



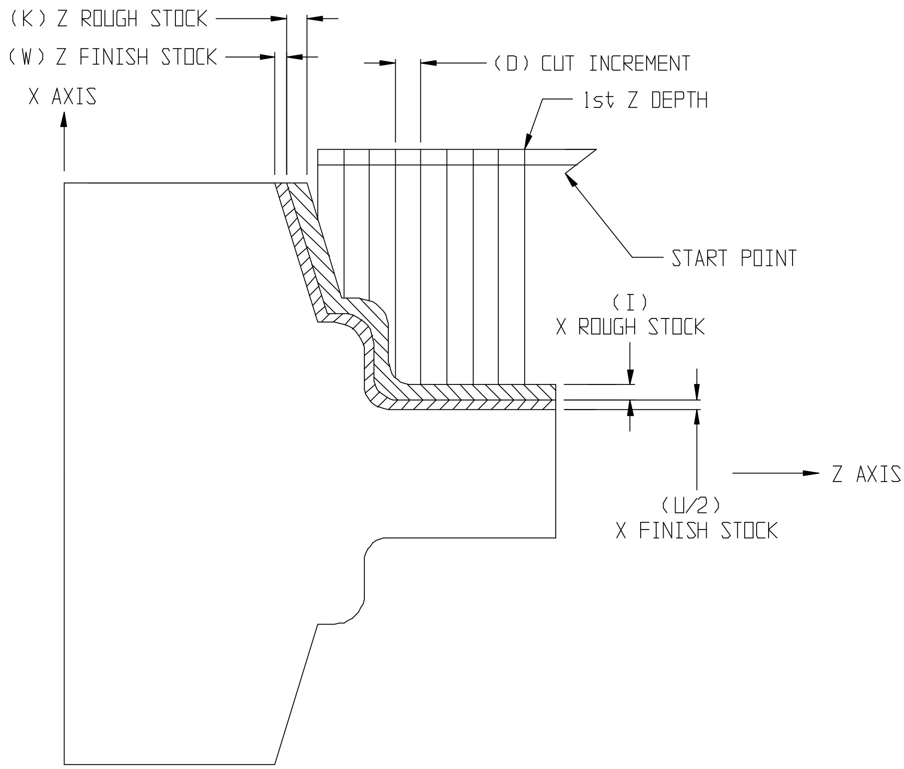
G0 X2.1 Z.1 (Start Point)
P117=0 (Use Pattern Relief)
G71 P1 Q2 U0 I.03 K0 R0 V1.95 D.05 (Roughing Information)
N1 (Start Block)
G42
G0 X1.5
G1 Z0
G2 R.5 Z-.5 (Geometry)
G2 R1 Z-1.5
G2 R.5 Z-2
G1 X2.1
N2 (End Block)



Graphics from Example of Rough Turning Program

Rough Facing Cycle (G72)

The rough facing cycle generates all the roughing tool paths from a specified finish pattern. The finish pattern is specified by the geometry between the start block and the end block.



Rough cutting is done by first cutting parallel to the X axis down to the pattern leaving the rough stock. After the cuts along the X axis are completed, a pass is made along the finish pattern leaving the finish stock.

Feeds and speeds within the pattern are ignored during the parallel cuts. Feeds and speeds are effective during the pass along the pattern.

Rapid moves within the pattern are effective when following the finish pattern.

Features concerning pattern/45 degree relief, pockets, X stock with pockets, and X axis changing direction are the same as the rough turning cycle.

Code for Rough Facing Cycle:

G0 X##.#### Z##.####
1

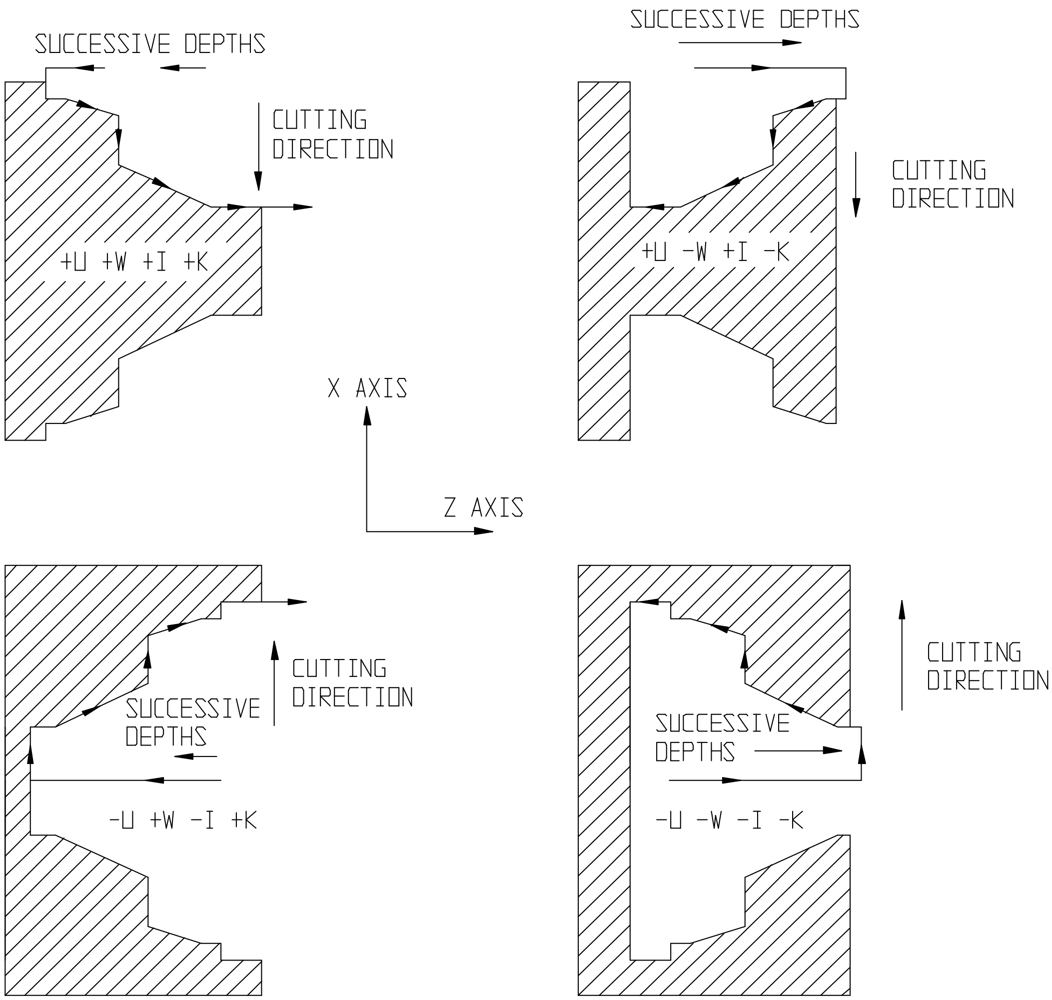
P117 = 0
2

G72 U##.#### W##.#### I##.#### K##.#### V##.#### D##.#### R##.#### P#### Q####
3 4 5 6 7 8 9 10 11

N####
10
.
. Commands between the start block and the end
. block define the shape to rough out, i.e. Rough-Out.
.
N####
11

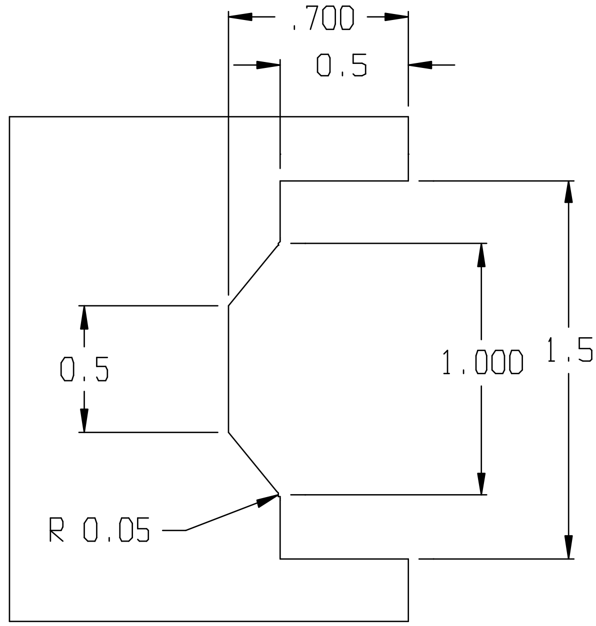
1. (X,Z) Start Point:	The start point of the roughing cycle. X and Z return to the start point at the end of the cycle at a rapid feedrate.
2. Relief (PATTERN or 45 DEGS.):	The type of relief made at the end of the cutting. PATTERN will cut along the pattern of the part; 45 DEGS will retract at a 45 degree angle before retracting X for the next cut. If the relief type is 45 DEGS no pockets are allowed in the pattern.
3. (U) X Finish Stock:	Distance and direction of the finish allowance in the X axis (diameter dimension).
4. (W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis (radius dimension).
5. (I) X Rough Stock:	Distance and direction of the rough allowance in the X axis.
6. (K) Z Rough Stock:	Distance and direction of the rough allowance in the Z axis.
7. (V) 1st Z Depth:	The Z depth of the 1st cut in the roughing cycle. If this is not entered the 1st cut is at (Z start point) + (Z finish stock) + (Z rough stock) - (cut increment).
8. (D) Cut Increment:	Depth of cut designated without sign. The cutting direction depends on the direction of the 1st move in the pattern.
9. (R) Clearance:	Distance of the 45 degree relief or an additional distance to pull away after the pattern relief.
10. (P) Start Block:	Designates the start of the geometry to face. (Any number between 1 and 9999.)
11. (Q) End Block:	Designates the end of the geometry to face. (Any number between 1 and 9999.)

Notes: *1. The start block number should not be the same as the ending block number.*
2. If the X rough stock and Z rough stock are both zero the pass made along the finish pattern is skipped.

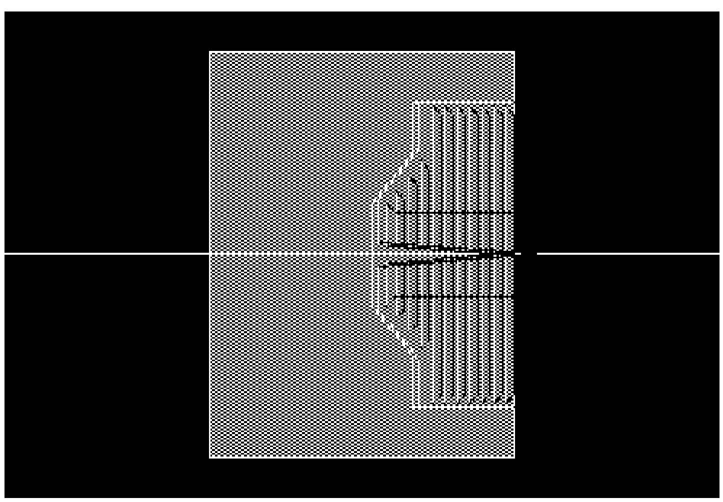


These are four patterns made by the rough facing cycle. Rough cutting for each of these patterns is done parallel to the X axis. The signs of U, W, I and K are shown for each pattern.

Example Rough Facing Cycle Program:



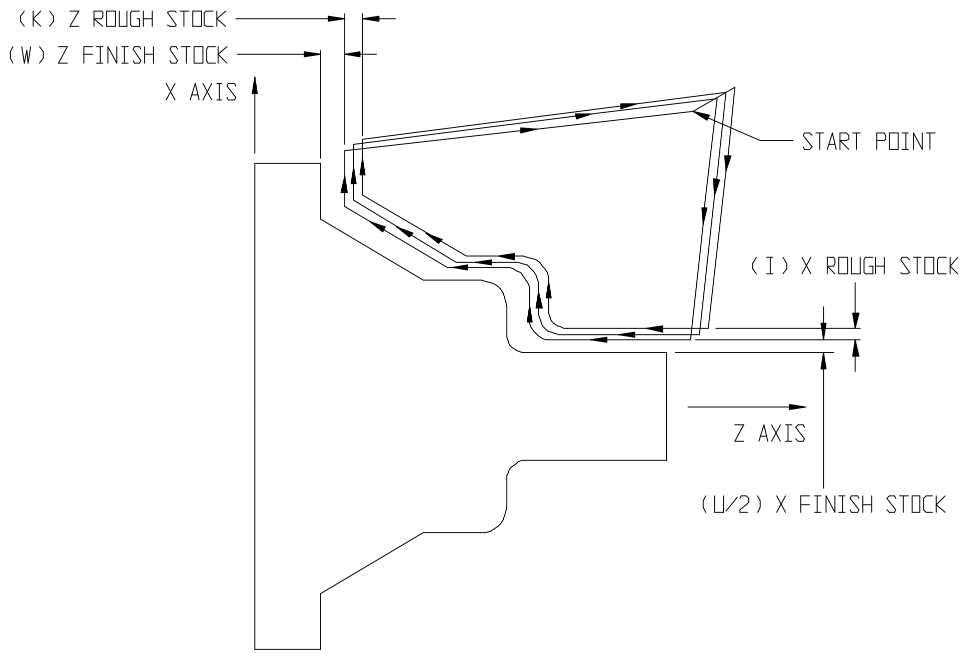
G0 X0 Z.1 (Start Point)
P117=1 (45° Relief)
G72 P1 Q2 U0 W0 I-.02 K.04 R.04 V-.04 D.06 (Roughing Information)
N1 (Start Block)
G42
G0 Z-.7   (Geometry)
G1 X.5
X1 Z-.5, R.05
X1.5
Z0.1
N2 (End Block)



Graphics From Example of Rough Facing Program

Pattern Repeat Cycle (G73)

The pattern repeat cycle repeatedly cuts a pattern incrementing the cuts down to the desired size. This cycle is used to efficiently cut a part whose shape is already defined (forged or cast).



Code for Pattern Repeat Cycle:

```
   G0   X##.####   Z##.####
          1          1

G73 U##.### W##.#### I##.#### K##.#### D## P#### Q####
          2          3          4            5        6       7       8
    N####
      7
     .  Commands between the Start and End Block define
     .   the shape to cut.
    N####
      8
```

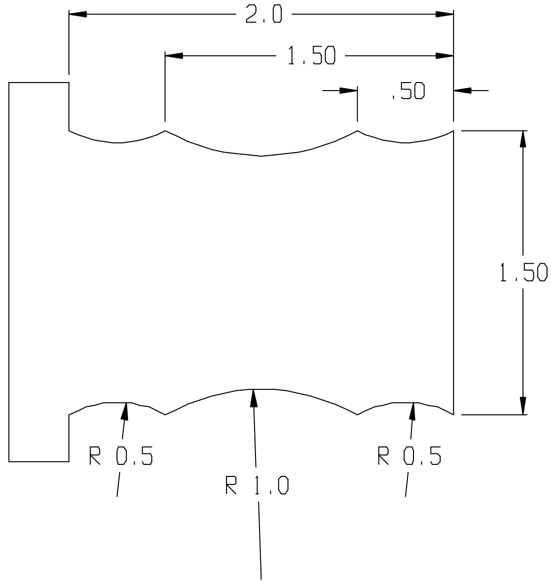
1. (X,Z) Start Point:	The start point of the roughing cycle. X and Z return to the start point at the end of the cycle.
2. (U) X Finish Stock:	Distance and direction of the finish allowance in the X axis.
3. (W) Z Finish Stock:	Distance and direction of the finish allowance in the Z axis.
4. (I) X Rough Stock:	Distance and direction of the rough allowance in the X axis.
5. (K) Z Rough Stock:	Distance and direction of the rough allowance in the Z axis.
6. (D) Number of Passes:	The number of passes to cut the rough stock.
7. (P) Start Block:	Designates the start of the geometry to cut. (Any number between 1 and 9999.)
8. (Q) End Block:	Designates the end of the geometry to cut. (Any number between 1 and 9999.)

Note 1: *Any feeds or speeds within the pattern are ignored.*

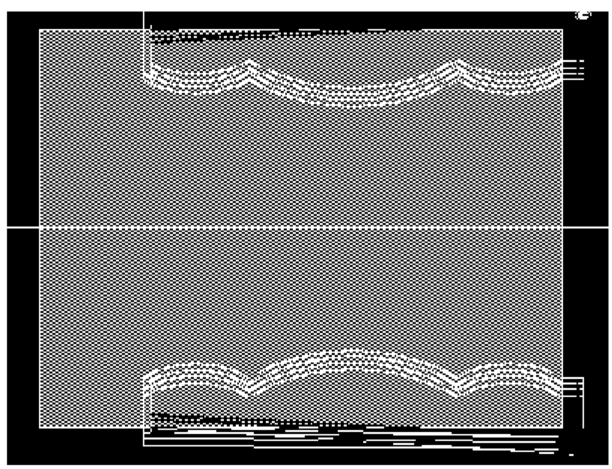
Note 2: *Rapid moves within the pattern will be executed as rapid moves during the pattern repeat cycle.*

Note 3: *Moves back to the start point, from the end of the pattern, are rapid moves.*

Example of Repeat Program:

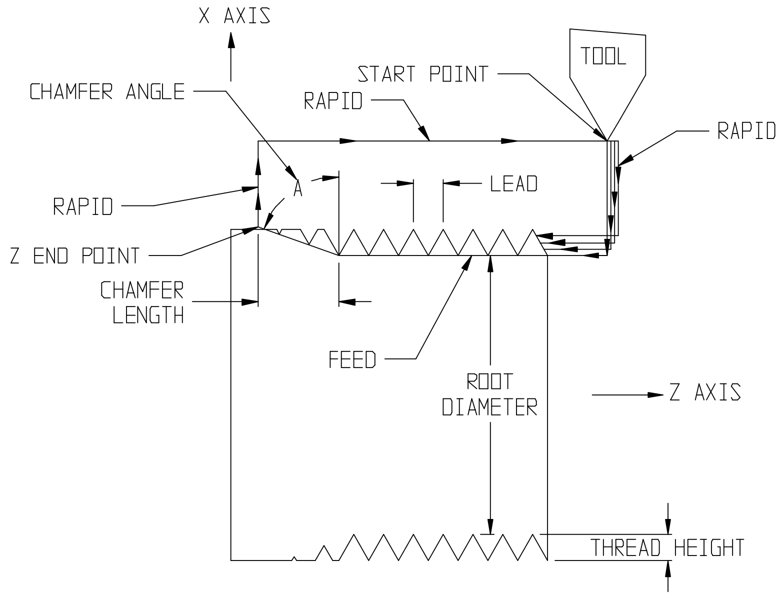


G0 X2.1 Z.1 (Start Point)
G73 P1 Q2 I.095 K0 U0 W0 D4 (Repeat Information)
N1 (Start Block)
G42
G0 X1.5
G1 Z0
G2 R.5 Z-.5
G2 R1 Z-1.5 (Pattern)
G2 R.5 Z-2
G1 X2
N2 (End Block)



Graphics from Example of Repeat Cycle Program

Multiple Thread Cutting Cycle (G76)



The multiple thread cutting cycle is used to make multiple passes on a thread using one of four different cutting methods.

Code for Multiple Thread Cutting Cycle:

P103 = ##.####
1
P104 = ###.###
2
P130 = ##.####
3
P131 = ##
4
G0 X##.#### Z##.####
5 *5*

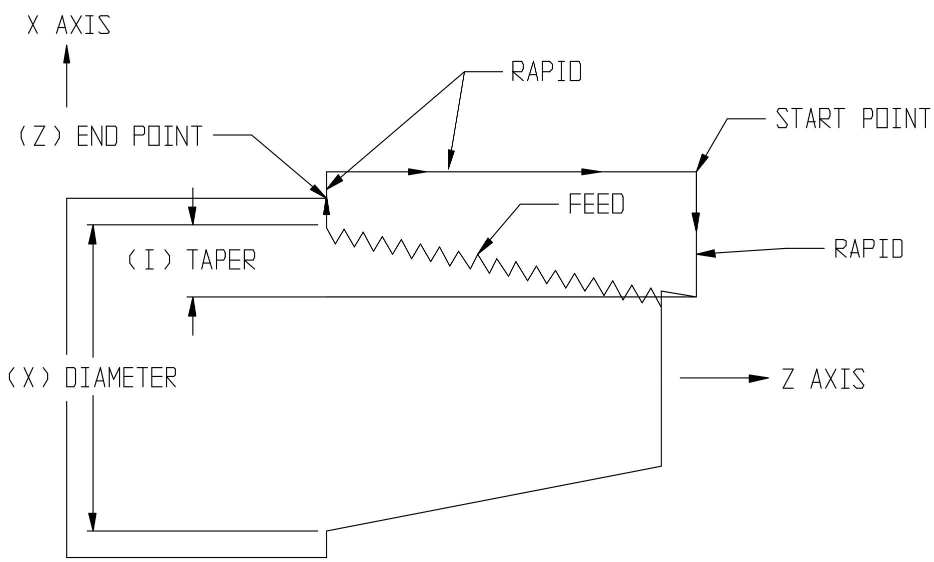
G76 X##.#### Z##.#### I##.#### F.#### Q###.### D##.#### A###.### P# K##.####
6 *7* *8* *9* *10* *11* *12* *13* *14*

1. Chamfer Length:	Length of the chamfer on the end of the thread.
2. Chamfer Angle:	The angle of the chamfer on the end of the thread. Angles 0° thru 90° will work for all threads, where 0° is straight out and 90° is no chamfer for a straight thread.
3. Minimum Cut:	When the cutting amount is less than the minimum cut, the cutting amount is clamped at the minimum cut (radius value).
4. Finish Passes:	The last pass can be repeated a number of times. If finish passes value is 0 or 1, the last pass will be made one time.

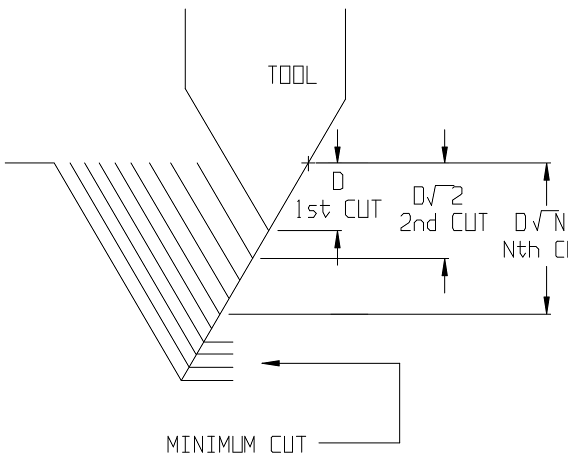
5. (X,Z) Start Point:	The start point of threading cycle. X and Z return to the start point after each pass on the thread at a rapid feedrate.
6. (X) Root Diameter:	The minor diameter for external threads and the major diameter for internal threads.
7. (Z) End Point:	The end of the thread.
8. (I) Taper:	The amount and direction of the taper to thread.
9. (F) Lead:	Lead of the thread.
10. (Q) Shift Angle:	Shift angle of the thread start angle. Used for multiple threads, i.e. shift 180°, 120° etc. 0° thru 360° are allowed.
11. (D) First Cut Amount:	Cutting depth of the first cut (radius value).
12. Tool Nose Angle:	The read angle of the tool nose.
13. (P) Cutting Method:	P1=CONSTANT AMOUNT/1 EDGE P2=CONSTANT AMOUNT/BOTH EDGES P3=CONSTANT DEPTH/ONE EDGE P4=CONSTANT DEPTH/BOTH EDGES
14. (K) Thread Height:	Height of the thread (X direction). K is specified as a radius value.

Inside or outside threads are determined by the start point relative to the root diameter.

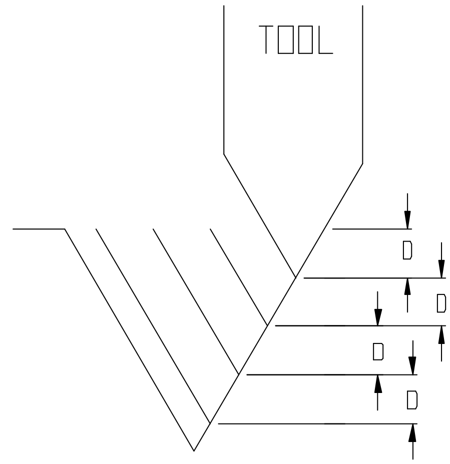
Note 1: It is possible to chamfer the front of the thread by setting P468 to the length and P469 to the angle.



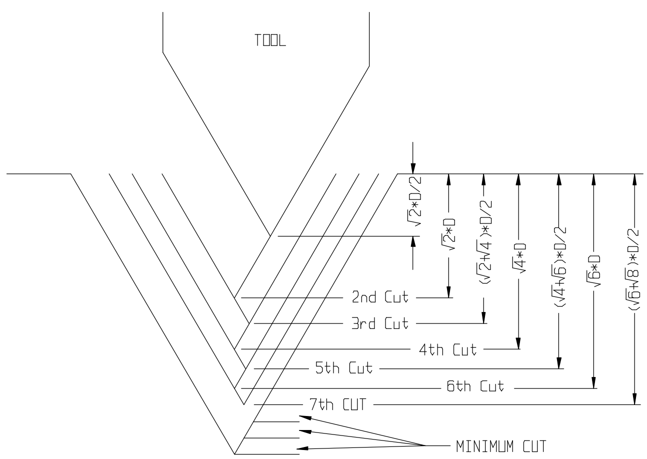
The four cutting methods are shown below:



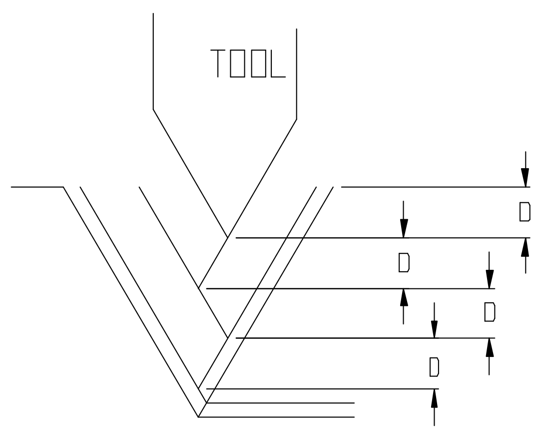
P1: Constant Amount/1 Edge



P3: Constant Depth/1 Edge



P2: Constant Amount/Both Edges

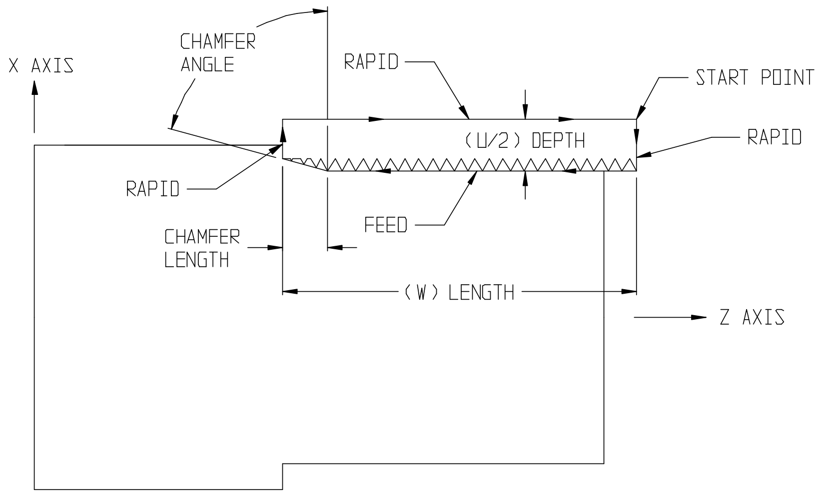


P4: Constant Depth/Both Edges

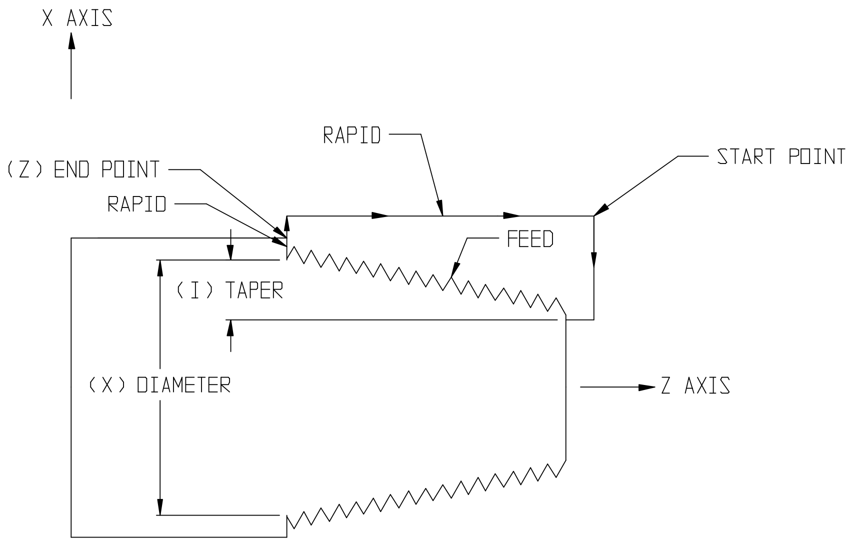
If the cutting depths using constant amount one edge, or constant amount both edges, become less than the minimum cut, the cutting depths are clamped at the minimum cut.

Cycle 1 (G92)

Cycle 1 is used to make a single threading cut. The cycle rapids to the X cutting depth, makes the threading cut, retracts X axis and rapids back to the start point. Tapered threads are allowed. Threads must be cut in feed per revolutions.



Straight Threading Using G92 Cycle 1 With Incremental Dimensions



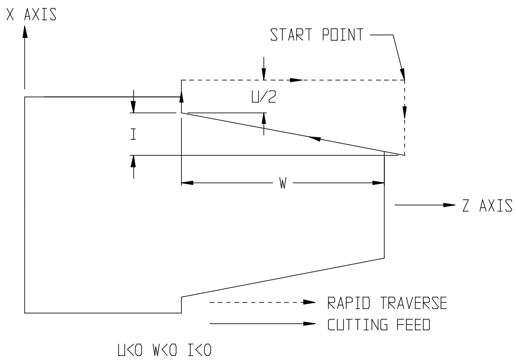
Tapered Threading Using G92 Cycle 1 With Absolute Dimensions

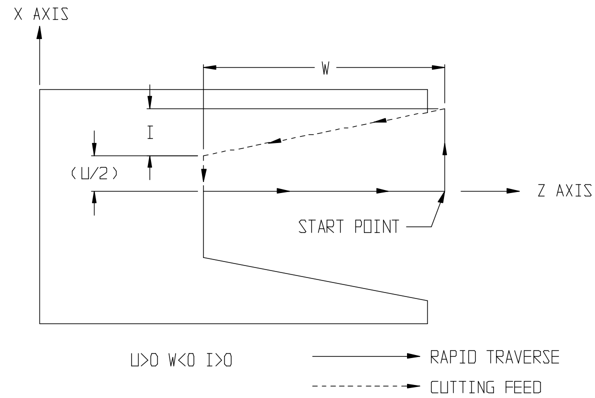
Code for Cycle 1 Thread Cutting:

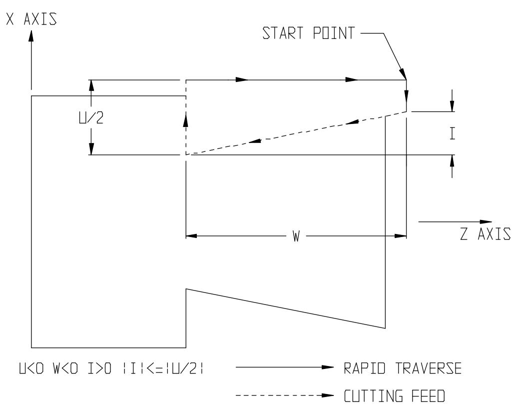
```
G0  X##.####  Z##.####
       1          1

P103 = ##.####
         2

P104 = ###.###
         3

G92 I##.#### X##.#### Z##.#### Q###.### F.####
      4         5         6          7      8
```

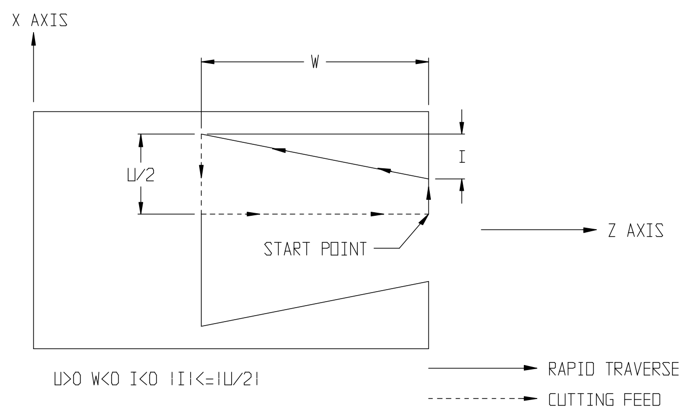
1. (X,Z) Start Point:	The start point of Cycle 1. X and Z return to the start point at the end of the cycle at a rapid feedrate.
2. Chamfer Length:	Length of the chamfer on the end of the thread.
3. Chamfer Angle:	The angle of the chamfer on the end of the thread. Angles 0° thru 90° will work for all threads, where 0° is straight out and 90° is no chamfer for a straight thread.
4. (I) Taper:	The amount and direction of the taper to thread (radius value).
5. (X) Diameter:	The root diameter to thread.
6. (Z) End Point:	The end point of the thread.
5a. (U) Depth:	The incremental X distance and direction from the start point to the diameter to thread. (U can be used in place of X.)
6a. (W) Length:	The length and direction of the thread in Z from the start point. (W can be used in place of Z.)
7. (Q) Start Angle:	Shift angle of the thread start angle. Used for multiple threads, i.e. shift 180°, 120° etc. 0° thru 360° is allowed.
8. (F) Lead:	Lead of the thread.

Note: It is possible to chamfer the front of the thread by setting P468 to the length and P469 to the angle.









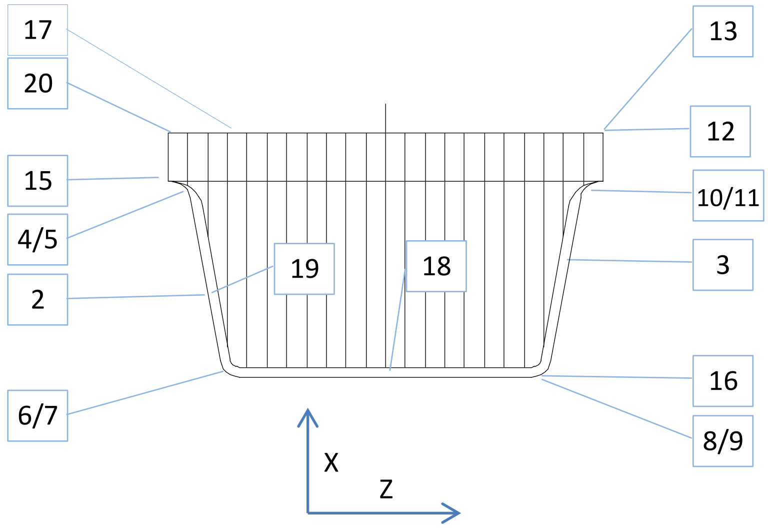
When using G92 Cycle 1 with Incremental Dimensions, the signs of the depth, length and taper indicate the directions shown in the above figures.

Note: The feedrate override is locked at 100% during threading. The spindle override is active but should not be changed during thread cutting.

Diameter/Face Grooving II Cycle (G77)

Diameter/Face Grooving II (G77)
The G77 cycle adds the ability to taper the walls on the groove and round or chamfer any of the 4 corners of the groove.

Diameter Grooving II



P545 = 0
1
P531 = ##.####
2
P532 = ##.####
3
P534 = ##.#### P533 = ##.####
4 5
P536 = ##.#### P535 = ##.####
6 7
P538 = ##.#### P537 = ##.####
8 9
P540 = ##.#### P539 = ##.####
10 11
G65 X##.#### Z##.####
12 13
G77 G4# V##.#### X##.#### K##.#### A##.#### C##.#### Z##.#### F##.####
14 15 16 17 18 19 20 21

1. 0=Diameter groove (P545 specifies 0=diameter or 1=face)
2. Left Taper (the taper angle on the left side)
3. Right Taper (the taper angle on the right side)

4 or 5. Top left chamfer or round corner. Use a Zero value in both unless using one of them (P533 and P534). Unused options must have a value of 0 in the parameter.

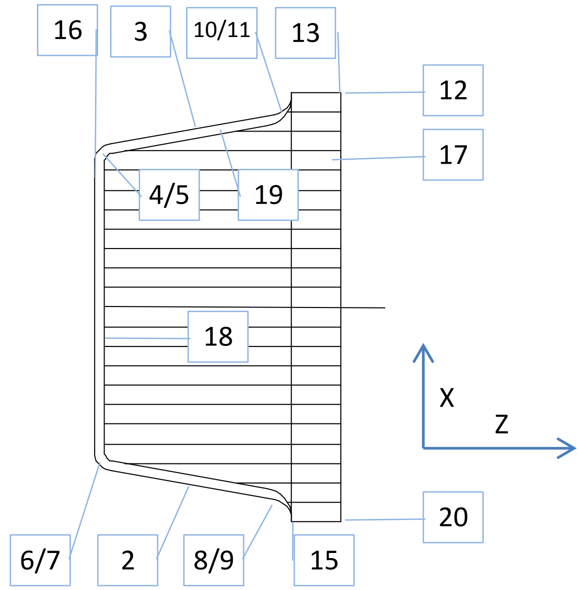
6 or 7. Bottom left chamfer or round corner. Use a Zero value in both unless using one of them (P535 and P536). Unused options must have a value of 0 in the parameter.

8 or 9. Bottom right chamfer or round corner. Use a Zero value in both unless using one of them (P537 and P538). Unused options must have a value of 0 in the parameter.

10 or 11. Top right chamfer or round corner. Use a Zero value in both unless using one of them (P539 and P540). Unused options must have a value of 0 in the parameter.

12. X clearance (diameter dimension).
13. Z start , the right side of the groove.
14. Cutter comp (G41 or G42=on G40=off).
15. X start diameter, the top of the groove (diameter dimension).
16. X bottom diameter, the bottom of the groove (diameter dimension).
17. Z increment, step over in Z.
18. X finish stock, this material is left and removed on a finish pass (radius dimension).
19. Z finish stock, this material is left and removed on a finish pass.
20. Z end dimension.
21. Feedrate, the cutting feedrate

Face Grooving II



P545 = 1
1
P531 = ##.####
2
P532 = ##.####
3
P534 = ##.#### P533 = ##.####
4 5
P535 = ##.#### P536 = ##.####
6 7
P537 = ##.#### P538 = ##.####
8 9
P540 = ##.#### P539 = ##.####
10 11
G65 X##.#### Z##.####
12 13
G77 G4# V##.#### Z##.#### K##.#### C##.#### A##.#### X##.#### F##.####
14 15 16 17 18 19 20 21

1. 1=Face groove
2. Inside Taper, taper on the smaller diameter
3. Outside Taper, taper on the larger diameter

4 or 5. Inside Top chamfer or round corner. Use a Zero value in both unless using one of them (P533 and P534). Unused options must have value of 0 in the parameter.

6 or 7. Inside Bottom chamfer or round corner. Use a Zero value in both unless using one of them (P535 and P536). Unused options must have value of 0 in the parameter.

8 or 9. Outside Bottom chamfer or round corner. Use a Zero value in both unless using one of them (P537 and P538). Unused options must have value of 0 in the parameter.

10 or 11. Outside Top chamfer or round corner. Use a Zero value in both unless using one of them (P539 and P540). Unused options must have value of 0 in the parameter.

12. X start diameter, the outside diameter.
13. Z clearance.
14. Cutter comp (G41 or G42=on G40=off).
15. Z start depth, top of the groove.
16. Z depth, bottom of the groove.
17. X increment, step over in X (a radius dimension).
18. Z finish stock, this material is left and removed on a finish pass.
19. X finish stock this material is left and removed on a finish pass (radius dimension).
20. X end diameter.
21. Feedrate, the cutting feedrate.

Drilling, Tapping and Grooving

These canned cycles simplify the program by using a single block with a G code to specify the machining operations usually specified in several blocks.

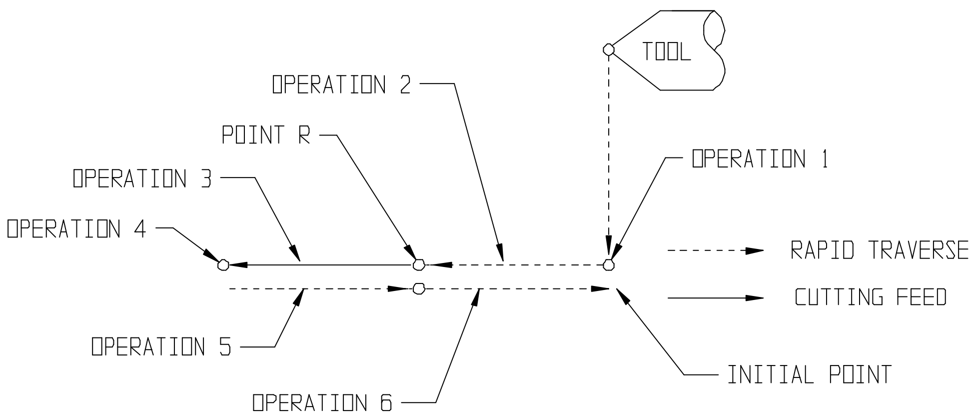
G code	Drilling	Operation at hole bottom	Retraction	3. Application
G74	Intermittent feed	-	Rapid traverse	High speed peck drilling or Face grooving cycle
G75	Intermittent Feed	-	Rapid traverse	Diameter grooving cycle
G81	Feed	-	Rapid traverse	Drilling cycle, Spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, Counter boring cycle
G83	Intermittent feed	-	Rapid	Peck drilling cycle
G84	Feed	Reverse spindle	Feed	Tapping cycle
G85	Feed	-	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

Table 1

Canned Cycles

Generally these canned cycles consist of a sequence of six operations as shown below.

Operation 1: Positioning of axes X and Z
Operation 2: Rapid traverse to point R
Operation 3: Hole machining
Operation 4: Operation at the bottom of a hole
Operation 5: Retraction to point R (G39 Mode)
Operation 6: Rapid traverse up to the initial point (G38 Mode)



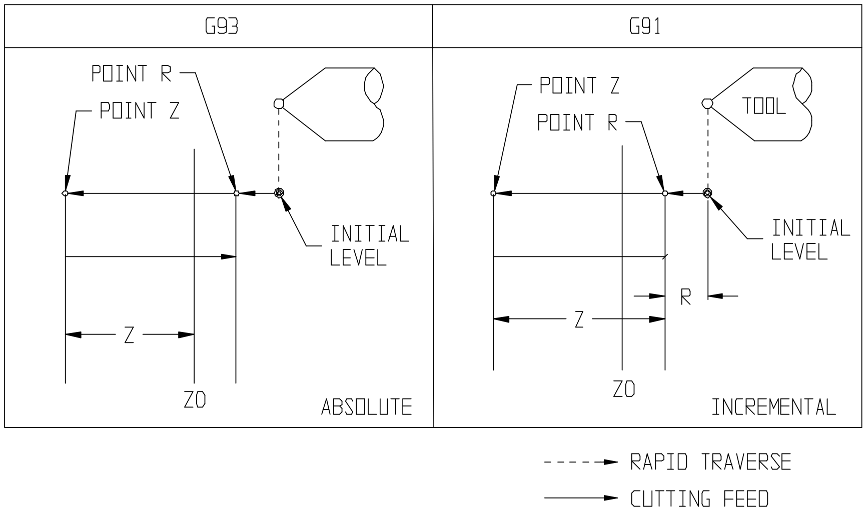
Canned Cycle Operation

Canned cycle operations consist of three basic modes that are specified by particular modal G codes as shown below.

(1) Data format	G93 Absolute G91 Incremental
(2) Return point level	G38 Initial point level G39 R point level
(3) Drilling mode	G74 G75 G81 . See Table 3 on page 67 . . G89

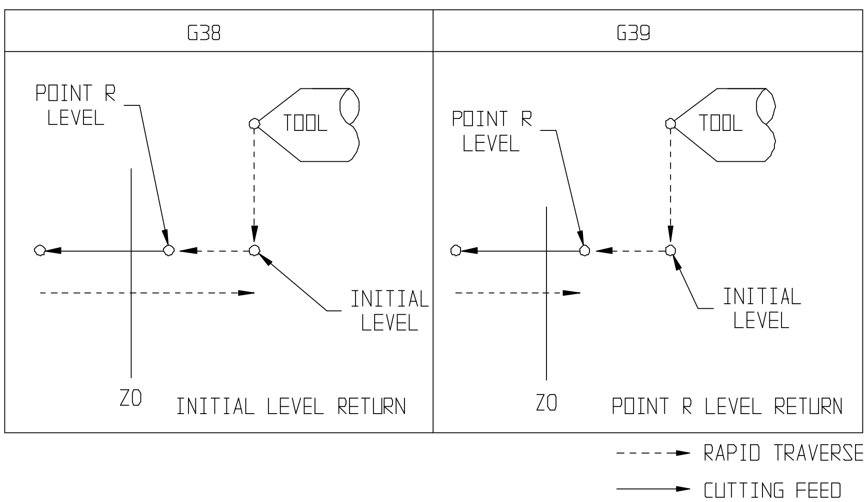
Note: The initial level means the position of the Z axis, or X axis for G75 (diameter grooving) when the canned cycle is turned on.

The figure below shows how to specify data in G93 or G91 mode.



Absolute and Incremental Programming

If the tool is to be returned to point R or to the initial level, it is specified by G38 or G39. (See Figure below)



Initial Level and Point R Level

The drilling data is specified following G74/G75/G81 to G89. Data is stored in the control as modal values and is retained for future use in other cycles.

The machining data in a canned cycle is specified as shown below:

G _____ Z__ R__ V__ Q__ P__ F__ U__ E__ X__ D__ K__ I__

Drilling Mode — Drilling Data

Drilling mode	G ____	See Table 1 on page 6-67.
Drilling data	Z . . .	Specifies the distance from point R to the bottom of the hole with an incremental value, or the position of the hole bottom with an absolute value or the end point for G75 (diameter grooving).
	R . . .	Specifies the distance from the initial level to the point R with an incremental value, the position of point R with an absolute value.
	V . . .	Specifies the first depth in peck and grooving cycles.
	Q . . .	Specifies the increment value for peck and grooving cycles.
	P . . .	Specifies the dwell time at the bottom of the hole. The time is specified in seconds.
	F . . .	Specifies the plunge feedrate.
	U . . .	Peckup increment, unsigned distance to rapid retract in peck/grooving cycles.
	E . . .	Peck Clearance, the unsigned distance to rapid to above the previous depth.
	X . . .	Final diameter of the groove in the face grooving cycle or final depth in the diameter grooving cycle.
	D . . .	Amount of relief to back off before retracting in the grooving cycle.
	K . . .	Z increment, the unsigned amount of step-over in the diameter grooving.
	I . . .	X increment, the unsigned amount of step-over in the face grooving cycle.

The drilling cycles use the following parameters:

P116 Cut Increment
P118 Relief
P124 Diameter
P139 X R-Plane
P140 Z R-Plane
P141 Final Depth
P142 Initial Level
P143 Cut Increment
P144 First Depth
P145 Plunge Feedrate
P146 Peckup Increment
P147 Peck Clearance
P148 Dwell

Once the drilling data has been specified in a canned cycle, it is retained until it is changed.

Canned cycles are cancelled at the beginning of each program.

The position moves must be made to the location of the hole to be drilled, tapped or the start of the groove before the cycle is called. None of the drill cycles are modal; they are one shot cycles.

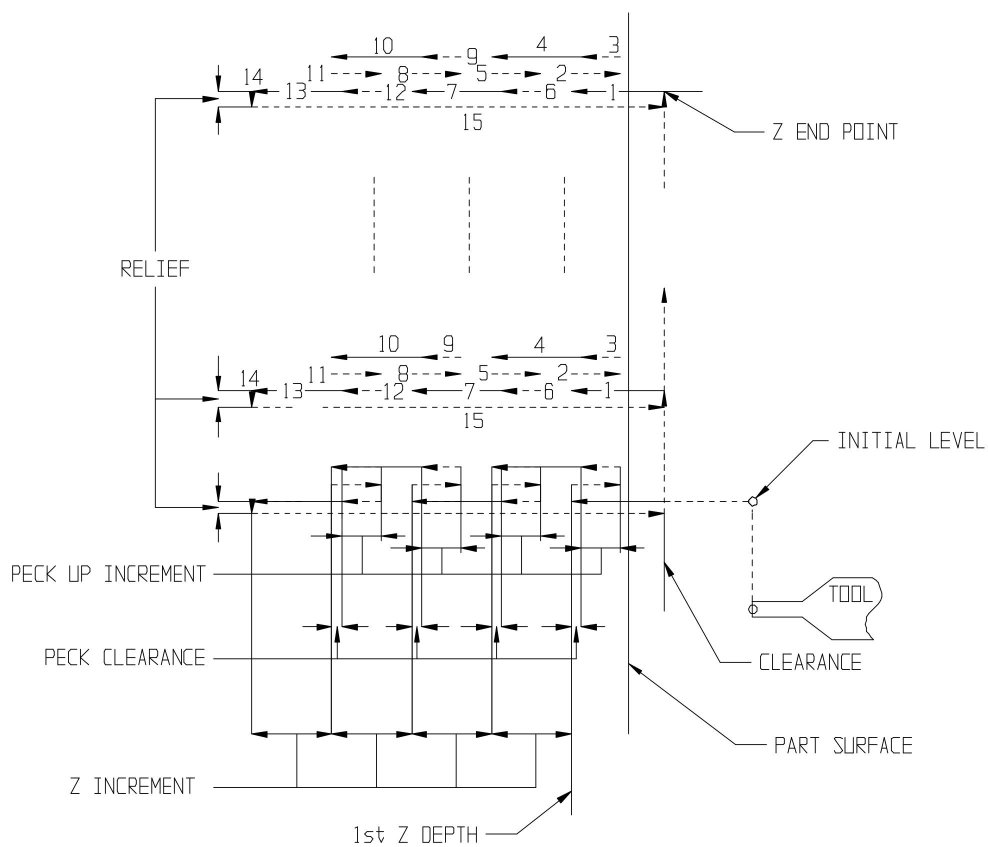
Each cycle will be detailed next.

Face Grooving/High Speed Peck Drilling Cycle (G74)

The face grooving cycle can be used as a chip breaking drilling cycle by omitting the X dimension (X-Final Diameter, I-X Increment and D-Relief) fields out of the G74 blocks. If X (final diameter) and I (X increment) are omitted, the sign of D (relief) determines its direction. The fast peck cycle is similar to peck cycle except the fast cycle does not pull out to the clearance, it pulls out the peck increment. There are four possible patterns going tail stock to chuck or vice versa, and outside to inside and vice versa.

The sign (or direction) of the Z increment, peck clearance and peck up increment are determined by the direction of the final Z depth to the initial Z position.

The sign (or direction) of the X increment and relief are determined by the direction of the final diameter to the initial X position.



CODE USED FOR THE FACE GROOVING CYCLE

G74 F.#### G39 R##.#### Z##.#### V##.#### Q##.#### E##.#### U##.#### X##.#### I##.#### D##.#### A##.#### C##.#### H# G4#
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15

1. (F) Feedrate:	Z feedrate for grooving the slot.
2. Return Point:	G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the grooving cycle started.
3. (R)Clearance:	Z rapids to the clearance at the beginning of the grooving cycle.
4. (Z) Final Z Depth:	The depth to cut the groove to.
5. (V) First Z Depth:	First depth. If the 1st depth is omitted the 1st depth is at the clearance - Z increment.
6. (Q) Z Increment:	Unsigned amount to groove in each successive peck.
7. (E) Peck Clearance:	Unsigned distance to rapid to above the previous depth.
8. (U) Peckup Increment:	Unsigned distance to rapid up each peck.
9. (X) Final Diameter:	The diameter to cut the groove to. The starting diameter is the X dimension the tool is at when the grooving cycle started.
10. (I) X Increment:	The unsigned amount to step over in diameter. (This is a radius value.)
11. (D) Relief:	Amount to back X away at the bottom of the groove before retracting to the Z clearance. (This is a radius value.)
12. (A) X Finish Stock:	The unsigned amount of stock to leave on each side of the groove before making a finish pass. (This is a radius value.)
13. (C) Z Finish Stock:	The unsigned amount of stock to leave on the bottom of the groove before making a finish pass.
14. (H) First Relief	If H=0 then X will back away at the bottom of the groove before retracting Z to the clearance. If H<>0 then X will not retract after the first peck to the bottom of the groove.
15. (G4#) Cutter Comp	G40, G41, or G42 cutter compensation can be used to obtain the desired groove size (only valid for tool type 7 and type 5).

Note 1:If both X and Z finish stock are zero, no finish pass is made. If either X or Z finish stock is non-zero, a finish pass will be made as follows:

1. *Remove the X finish stock at the starting diameter to the bottom of the groove.*
2. *Move to the center of the groove, along the bottom.*
3. *Retract out of the groove.*
4. *Move to the starting diameter.*
5. *Remove the X finish stock at the final diameter to the bottom of the groove.*
6. *Remove the Z stock along the bottom of the groove to the starting diameter.*
7. *Retract out of the groove.*

Note 2:If cutter compensation is not used when using the Face Grooving Cycle, the dimensions of the groove must account for tool width.

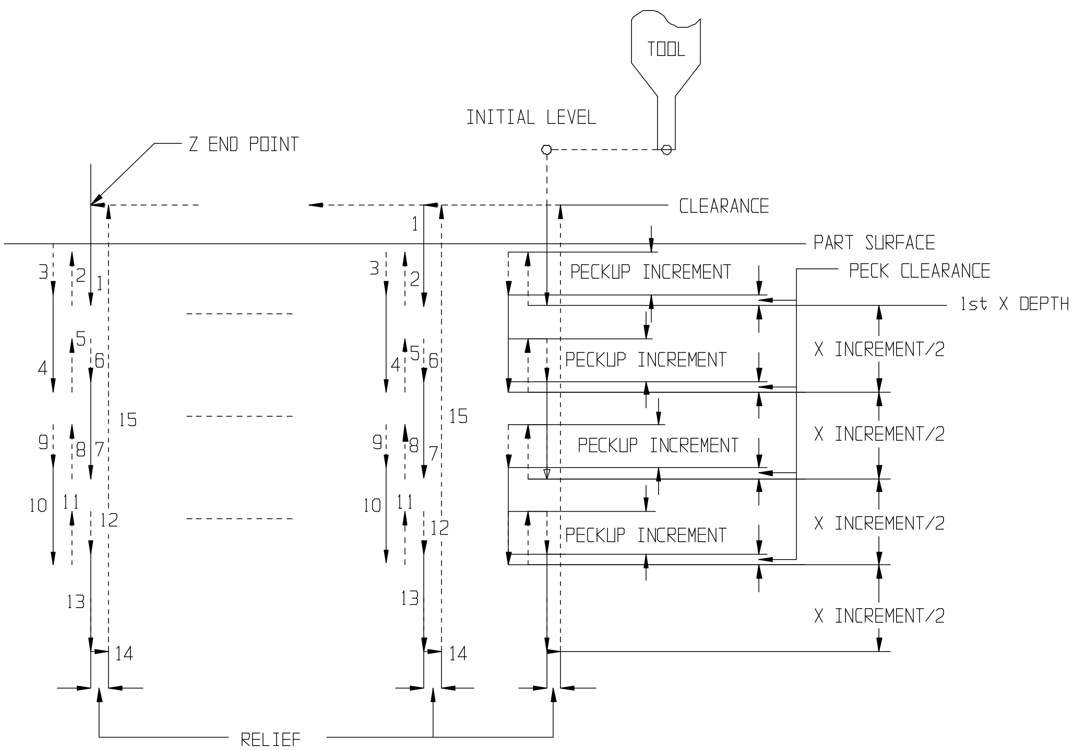
Note 3:A position should be made prior to the Face Grooving Cycle so the X axis is positioned to the start of the groove.

Diameter Grooving (G75)

The diameter grooving cycle is similar to the face grooving cycle except that peck cycle is used on the diameter. If the Z dimensions (Z-End Point, I-Z Increment and D-Relief) are omitted, a single high speed peck cycle is done on the diameter. There are four possible patterns going tail stock to chuck or vice versa and outside towards center or vice versa.

The sign (or direction) of the X increment, peck clearance, and peck-up increment are determined by the direction of the final X depth to the initial X diameter.

The sign (or direction) of the Z increment, and relief are determined by the direction of the Z end point to the initial Z position. If Z end point and Z increment are omitted, the sign of the relief determines its direction.



CODE USED FOR THE DIAMETER GROOVING CYCLE

G75	F.####	G39	R##.####	X##.####	V##.####	Q##.####	E##.####	U##.####	Z##.####	K##.####	D##.####	A##.####	C##.####	H#	G4#
	1	*2*	*3*	*4*	*5*	*6*	*7*	*8*	*9*	*10*	*11*	*12*	*13*	*14*	*15*

1. (F) Feedrate: X feedrate for cutting the groove.

2. Return Point: G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the X dimension the tool was at before the grooving cycle started.

3. (R) Clearance: X rapids to the clearance at the beginning of the grooving cycle. (This is a diameter value.)

4. (X) Final X Depth: The diameter to cut the groove to.

5. (V) First X Depth: First peck diameter. (If the first depth is omitted, the first depth will be the clearance - X increment.)

6. (X) X Increment: Unsigned diameter to groove in each successive peck. (This is a radius value.)

7. (E) Peck Clearance: Unsigned distance to rapid up after each peck. (This is a radius value.)

8. (U) Peckup Clearance: Unsigned distance to rapid up after each peck. (This is a radius value.)

9. (Z) Endpoint: The Z dimension to cut the groove to. The start of the groove is the Z dimension the tool is at when the grooving cycle is started.

10. (K) Z Increment: The unsigned amount to the step over.

11. (D) Relief: Amount to back Z away at the bottom of the groove before retracting X to the X clearance.

12. (A) X Finish Stock: The unsigned amount of stock to leave at the bottom of the groove before making a finish pass. (This is a radius value.)

13. (C) Z Finish Stock: The unsigned amount of stock to leave on each side of the groove before making a finish pass.

16. *14.* (H) First Relief: If H=0 then Z will back away at the bottom of the groove before retracting X to the clearance. If H<>0 then Z will not retract after the first peck to the bottom of the groove.

17. (G4#) Cutter Comp: G40, G41, or G42 cutter compensation can be used to obtain the desired groove size (only valid for tool types 1,2,3,4,6 or 8).

Note 1: If both X and Z finish stock are zero, no finish pass is made. If either X or Z finish stock is non-zero, a finish pass will be made as follows:

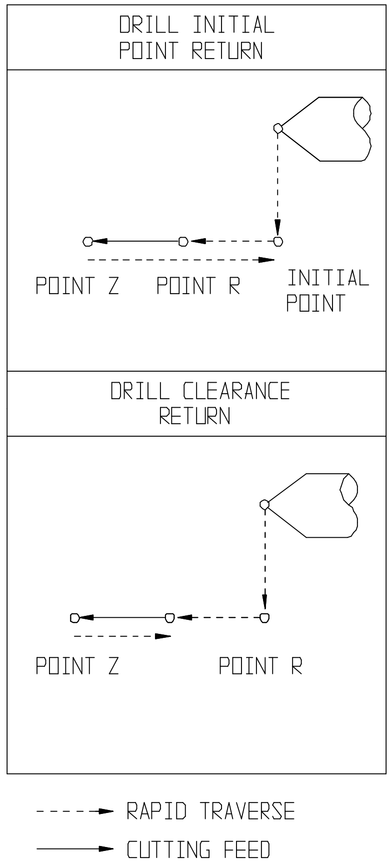
1. *Remove the Z finish stock at the final Z dimension to the bottom of the groove.*
2. *Move to the center of the groove, along the bottom.*
3. *Retract out of the groove.*
4. *Move to the Z starting dimension.*
5. *Remove the Z finish stock at the starting Z dimension to the bottom of the groove.*
6. *Remove the X stock along the bottom of the groove to the final Z dimension.*
7. *Retract out of the groove.*

Note 2: If cutter compensation is not used in the Diameter Grooving Cycle, the dimensions of the groove must account for tool width.

Note 3: A position should be made prior to the Diameter Grooving Cycle so that Z is positioned to the start of the groove.

Drill Cycle (G81)

The drill cycle rapids to the Z clearance plane, feeds to the Z depth and then rapids to the clearance plane or initial position.



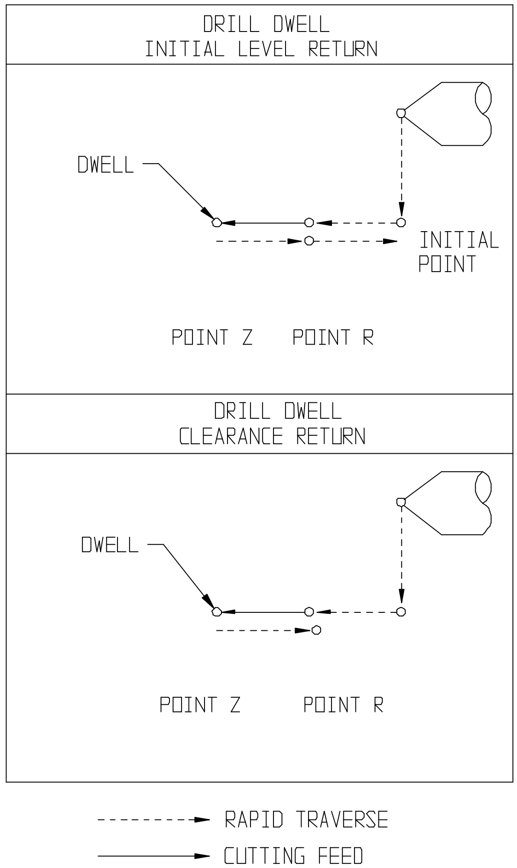
CODE USED FOR THE DRILL CYCLE

G81 F.#### G39 R##.#### Z##.####
1 *2* *3* *4*

1. (F) Feedrate:	Z Feedrate for drilling the hole.
2. Return Point:	G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.
3. (R) Clearance:	Z rapids to the clearance at the beginning of the drill cycle.
4. (Z) Final Z Depth:	The depth to drill the hole to.

Drill Dwell Cycle (G82)

The drill dwell cycle is identical to the drill cycle with the addition of a dwell at the bottom of the hole.



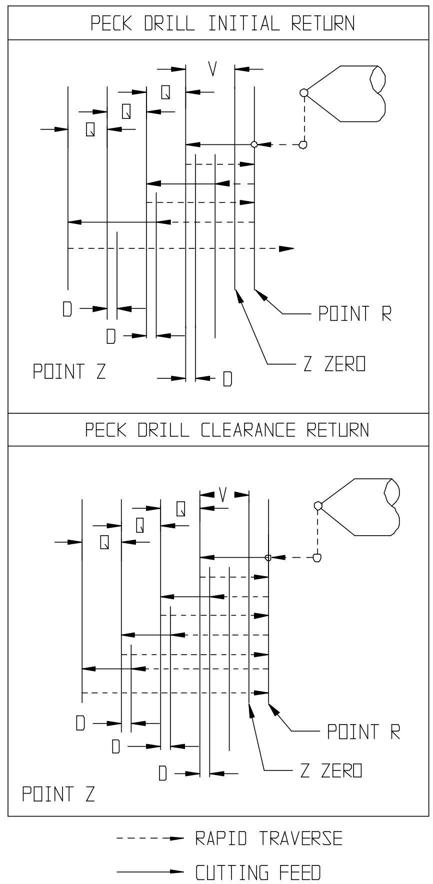
CODE USED FOR THE DRILL/DWELL CYCLE

G82 F.#### G39 R##.#### Z##.#### P###.#
1 *2* *3* *4* *5*

1. (F) Feedrate:	Z feedrate for drilling the hole.
2. Return Point:	G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.
3. (R) Clearance:	Z rapids to the clearance at the beginning of the drill cycle.
4. (Z) Final Z Depth:	The depth to drill the hole at.
5. (P) Dwell:	Time in seconds to dwell at the bottom of the hole.

Drill Peck Cycle (G83)

The drill peck cycle is useful for drilling deep holes. It drills to successive depths, then rapids out of the hole and back to a peck clearance for each depth. The sign or direction of the Z increment and peck clearance assumes drilling is done from tail stock towards the chuck.



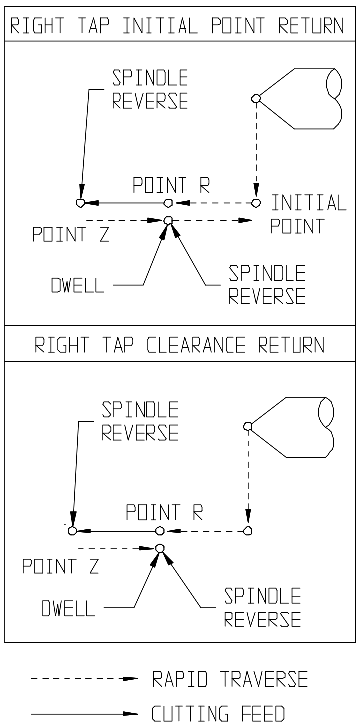
CODE USED FOR THE DRILL PECK

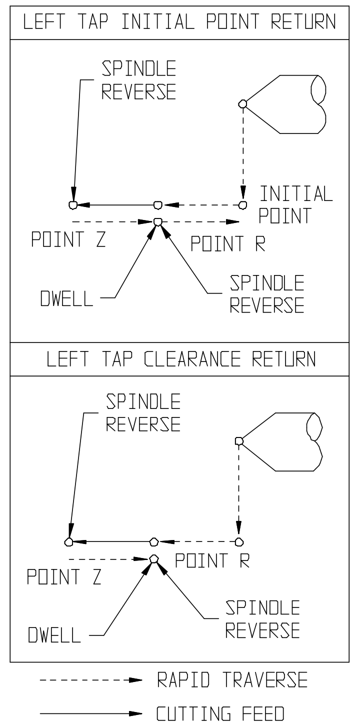
G83 F.#### G39 R##.#### Z##.#### V##.#### Q##.#### E##.####
1 2 3 4 5 6 7

1. (F) Feedrate:	Z feedrate for drilling the hole.
2. Return Point:	G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the drill cycle starts.
3. (R) Clearance:	Z rapids back to the clearance at the beginning of the drill cycle.
4. (Z) Final Z Depth:	The depth to drill the hole to.
5. (V) First Z Depth:	First peck depth. If the first Z depth is omitted the first Z depth is the clearance- the Z increment.
6. (Q) Z Increment:	Unsigned amount to drill in each successive peck.
7. (E) Peck Clearance:	Unsigned distance to rapid to above the previous depth.

Tap Cycle (G84)

G84 specifies a hard tapping cycle. The tap cycle rapids Z to the clearance, feeds Z to the reverse depth (in feed per rev), reverses the spindle, and retracts Z to the clearance (in feed per rev). The tap cycle then reverses the spindle to the original direction. Z axis rapids to the initial level if initial point return is specified.





CODE USED FOR THE TAP CYCLE

```
G97 G99 M3 S####.#
G84 F.#### G39 R##.#### Z##.####
         1        2        3           4
```

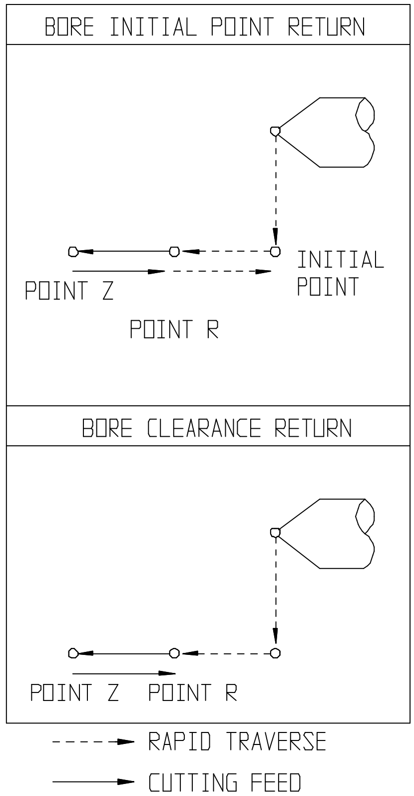
1. (F) Lead:	Distance between threads.
2. Return Point:	G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the tap cycle started.
3. (R) Clearance:	Z rapids to the clearance at the beginning of the tap cycle.
4. (Z) Spindle Reverse Depth:	The spindle reverses direction at this depth. It may coast some and cause the tap to go a little deeper.

Note 1: If the spindle is running clockwise when the G84 is initiated, right tapping is done. If the spindle is running counterclockwise when the G84 is initiated, left tapping is done.

Note 2: The tap cycle will not stop until the end of the tap cycle even if feedhold or block mode are applied. If halt is applied the tap reverses and stops when it is out of the hole.

Bore Cycle (G85)

The bore cycle feeds to the Z depth and feeds back to the clearance.



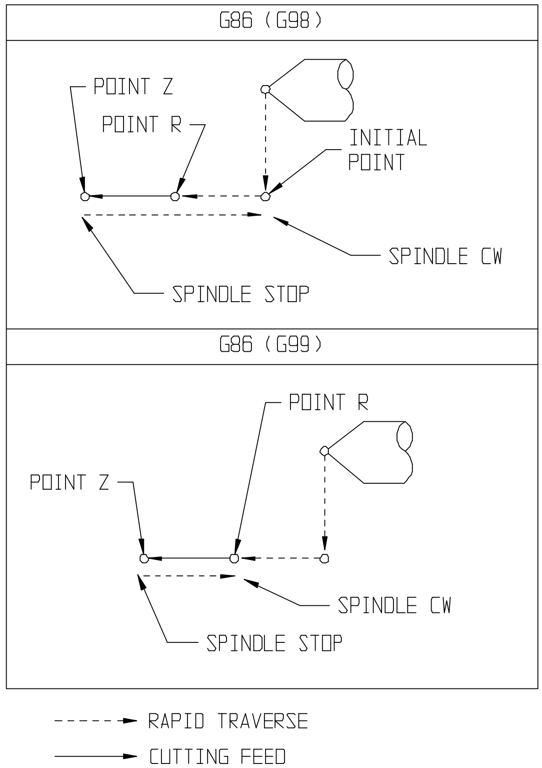
CODE GENERATED FROM THE BORE CYCLE

G85 F.#### G39 R##.#### Z##.####
1 2 3 4

1. (F) Feedrate:	Feedrate for boring the hole.
2. Return Point:	G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.
3. (R) Clearance:	Z rapids to the clearance at the beginning of the bore cycle.
4. (Z) Final Z Depth:	The depth to bore the hole to.

High Speed Boring Cycle (G86)

The high speed boring cycle will first start the spindle clockwise and then rapid Z to the clearance. Then it will feed Z to the Z depth, stop the spindle, and rapid to the initial Z if specified by G38.



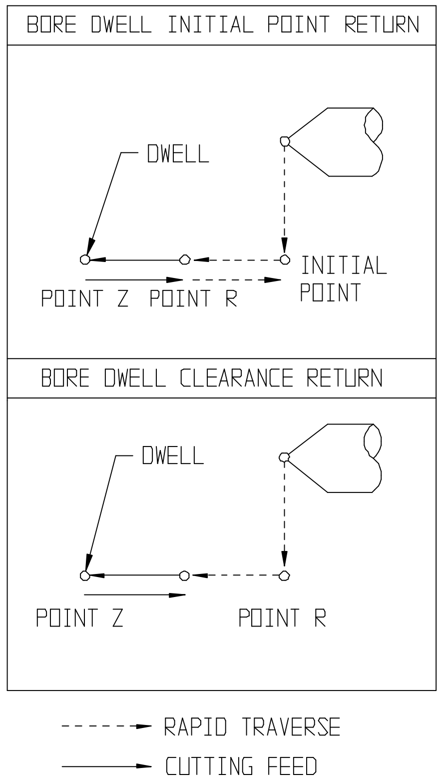
CODE USED FOR THE HIGH SPEED BORING CYCLE

G86 F.#### G39 R##.#### Z##.####
1 *2* *3* *4*

1. (F) Feedrate: Feedrate for boring the hole.
2. Return Point: G38 means the tool will rapid back to the clearance plane at the end of the cycle. G39 means the tool will rapid back to the Z dimension the tool was at before the drill cycle started.
3. (R) Clearance: Z rapids to the clearance at the beginning of the bore cycle.
4. (Z) Final Z Depth: The depth to bore the hole to.

Bore/Dwell Cycle (G89)

The bore/dwell cycle is the same as the bore cycle, but a dwell is performed at the bottom of the hole.



CODE USED FOR THE HIGH SPEED BORING CYCLE

G89 F.#### G39 R##.#### Z##.#### P###.#
1 *2* *3* *4* *5*

1. (F) Feedrate:	Feedrate for boring the hole.
2. Return Point:	G39 means the tool will rapid back to the clearance plane at the end of the cycle. G38 means the tool will rapid back to the Z dimension the tool was at before the bore/dwell cycle started.
3. (R) Clearance:	Z rapids to the clearance at the beginning of the bore/dwell cycle.
4. (Z) Final Z Depth:	The depth to bore to.
5. (D) Dwell:	Time in seconds to dwell at the bottom of the hole.

NOTES ON CANNED CYCLE SPECIFICATIONS:

Note 1: *The spindle must be turned on before a drilling, grooving or tap cycle is specified.*

M3	Spindle CW
.	
.	
.	
G __ __	Correct
.	
.	
.	
M5	Spindle Stop
.	
.	
.	
G __ __	Incorrect (M3 or M4 must be specified before this block.)

Note 2: *Operator Precautions*

a) *Feedhold*

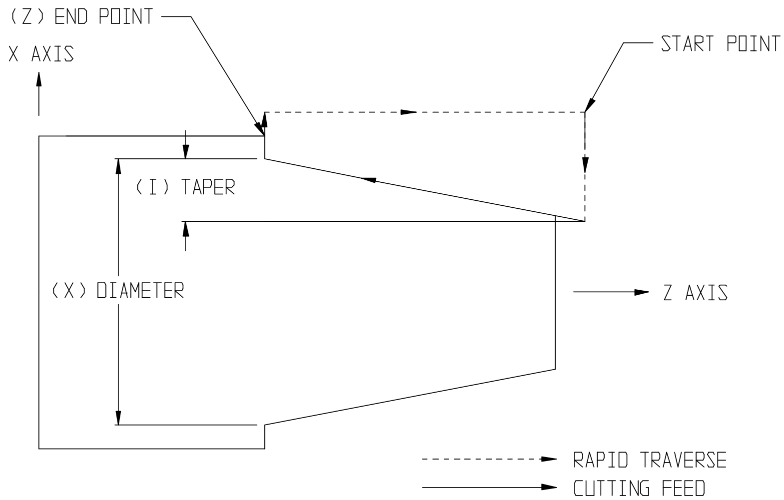
When a feedhold is applied between Operations 3 to 5 in tap cycle G84, the FEEDHOLD lamp immediately lights, but the control continues to operate up to Operation 6 and stops. If a FEEDHOLD is applied during Operation 6, it immediately stops.

b) *Override*

The feedrate override and spindle override is locked at 100% during the operation of canned cycle G84.

Cycle A (G90)

Cycle A is used to make a single turning cut. The cycle rapids to the X cutting depth, makes the cut, retracts X axis and rapids back to the start point. Tapered cuts are allowed.



Tapered Cutting Using Cycle A With Absolute Dimension

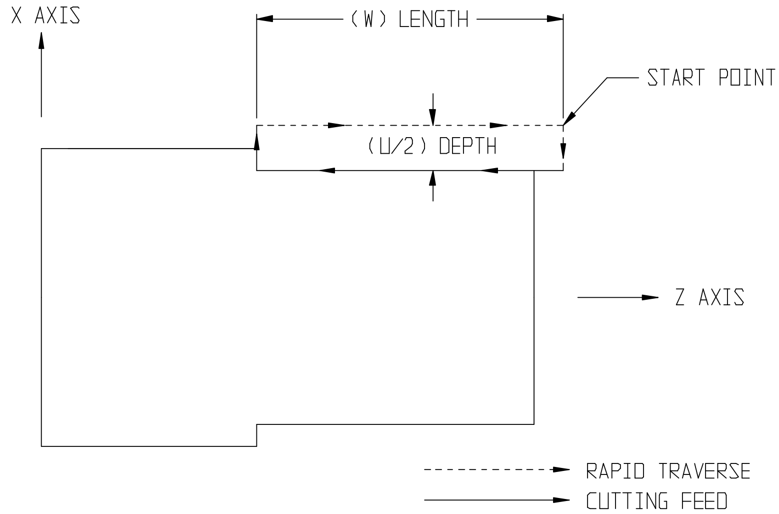
G0 X##.#### Z##.####
1 *1*

G90 I##.#### X##.#### Z##.####
2 *3* *4*

1. (X,Z) Start Point:	X and Z return to the start point at rapid feedrate.
2. (I) Taper:	The amount and direction of the taper.
3. (X) Diameter:	The diameter to turn.
4. (Z) End Point:	The end point of the cut.

4. -or-

Cycle A (G90) With Incremental Dimensions

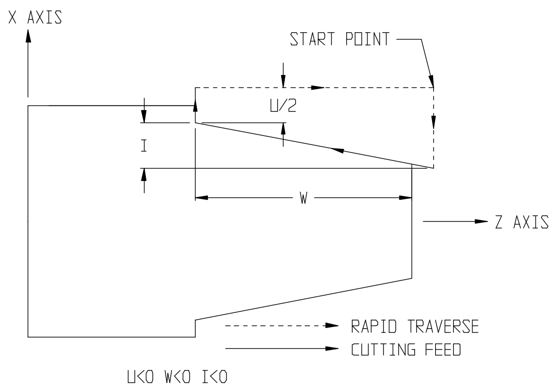


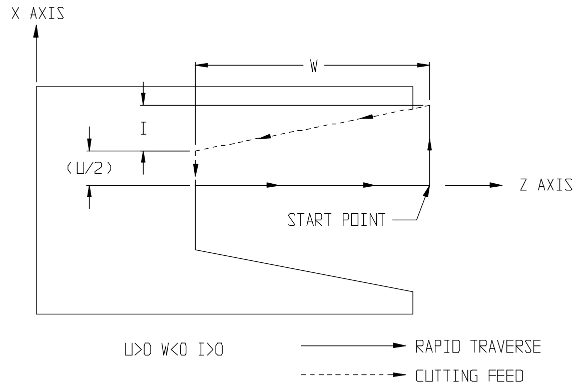
Straight Cutting Using Cycle A With Incremental Dimensions

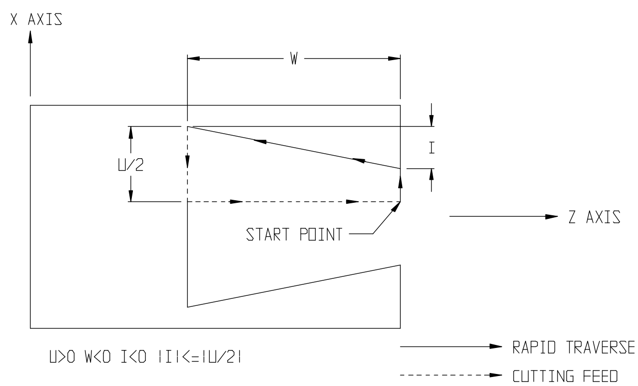
G0 X##.#### Z##.####
1 *1*

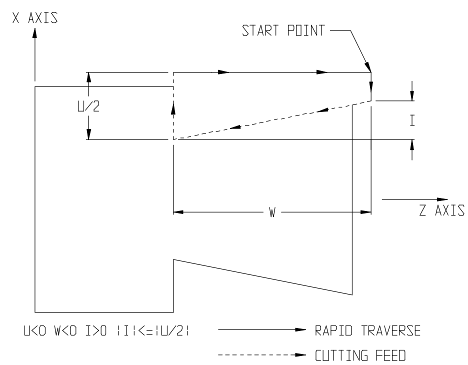
G90 U##.#### W##.####
2 *3*

1. (X, Z) Start Point: X and Z return to the start point at a rapid feedrate.
2. (U) Depth: The incremental X distance and direction from the start point to the diameter to turn. (U is a diameter value.)
3. (W) Length: The length and direction of the cut in Z from the start point.





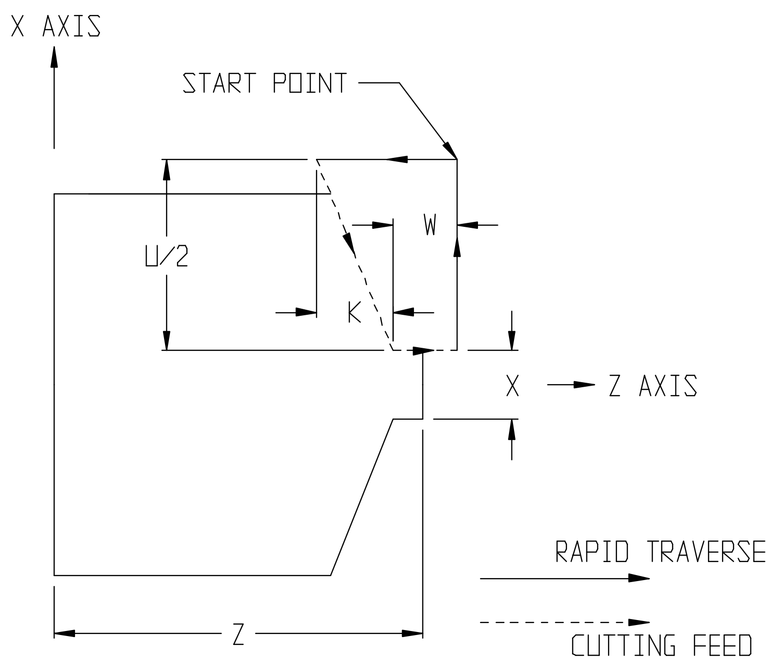




When using Cycle A with incremental dimensions, the signs of the depth, length and taper indicate the directions shown in the above figures.

Cycle B (G94)

Cycle B is used to make a single facing cut. The cycle rapids to the Z cutting depth, makes the cut, retracts Z axis and rapids back to the start point. Tapered cuts are allowed.



Tapered Cut For Cycle B Absolute And Incremental Dimensions

When Using Absolute Dimensions

G0 X##.#### Z##.####
1 *1*

G94 K##.#### X##.#### Z##.####
2 *3* *4*

1. (X,Z) Start Point: X and Z return to the start point at a rapid feedrate.
2. (K) Taper: The amount and direction of the taper.
3. (X) Diameter: The diameter to face to.
4. (Z) End Point: The Z end point.

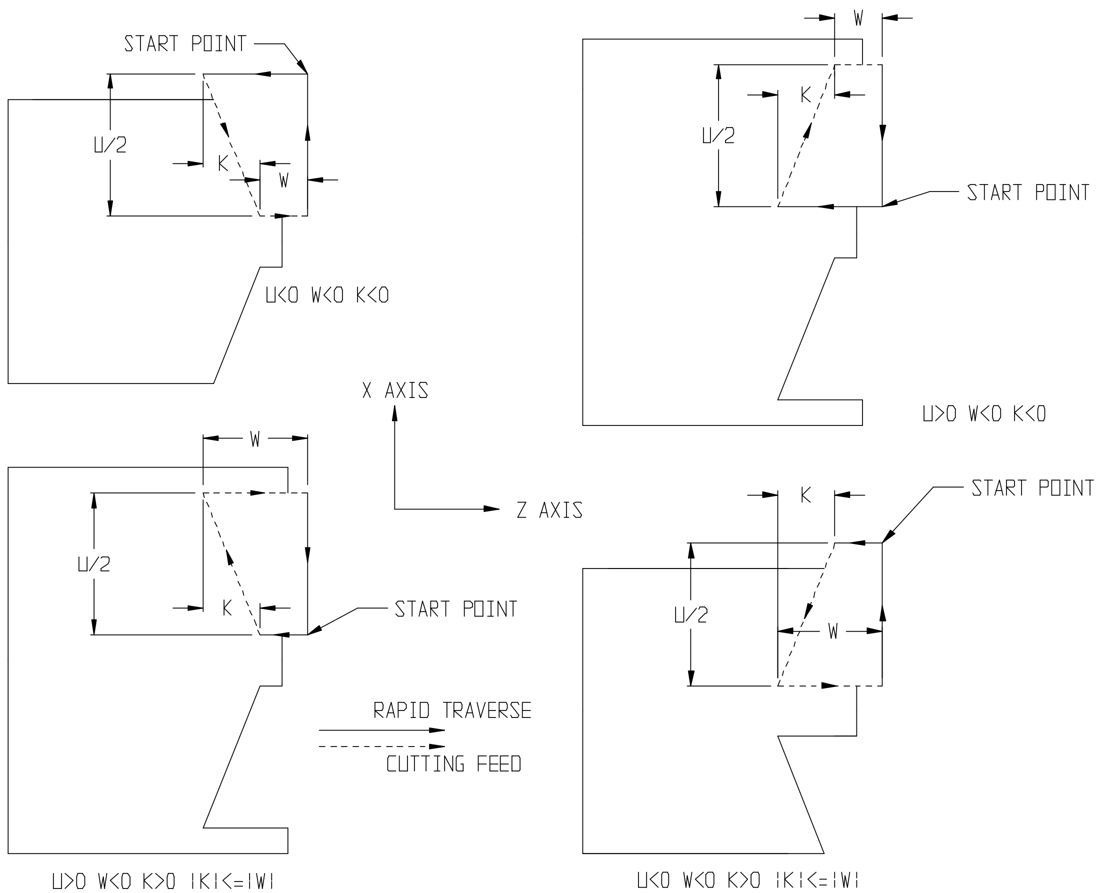
-OR-

When Using Incremental Dimensions

G0 X##.#### Z##.####
1 *1*

G94 K##.#### U##.#### W##.####
2 *3* *4*

1. (X,Z) Start Point:	X and Z return to the start point at a rapid feedrate.
2. (K) Taper:	The amount and direction of the taper.
3. (U) Depth:	The incremental X distance and direction from the start point to the diameter to face to. (Depth is a diameter value.)
4. (W) Length:	The amount and direction of the taper to face.



When using Cycle B with incremental dimensions, the signs of the depth, length and taper indicate the direction shown in the figures above.

Incremental Mode (G91)

This function causes the control to go into the incremental mode. In this mode all dimensions are entered relative to the machine position in the previous block. In the case of MDI, the dimensions are relative to the current machine position. Dimensions in G91 can be either positive or negative. Care should be taken when using G91. Whenever activating tool offsets, or setting "Floating Zeroes" via G50, the control should not be in the G93 (absolute) mode.

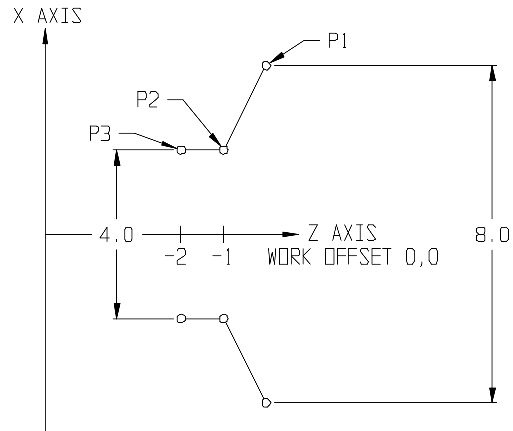
XC, ZC (centers), AA (start angle) and AB (end angle) are always an absolute.

G91 cancels G93.

(P1) G93 X8 Z0
(P2) G91 X-4 Z-1
(P3) Z-1

Incremental positioning can also be commanded using address U for X axis and W for Z axis.

(P1) G93 X8 Z0
(P2) U-4 W-1
(P3) W-1



INCREMENTAL POSITIONING

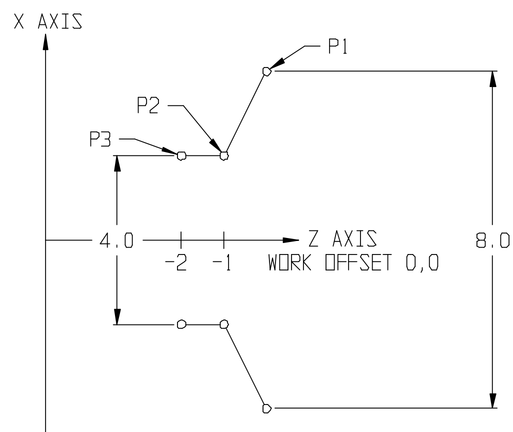
Absolute Mode (G93)

This function causes the control to go into its normal Absolute operating mode. In this mode all dimensions are referenced from a single reference point. This reference can either be the "Home Zero" point which is a fixed point on the machine, or an operator defined work coordinate point.

G93 is active at the beginning of each program. G93 cancels G91.

(P1) G93 X8 Z0
(P2) X4 Z-1
(P3) Z-2

Dimensions relative to work coordinates can be either negative or positive depending on where the operator sets the zero coordinate.



ABSOLUTE POSITIONING

Constant Surface Speed (CSS) (G96)

The surface speed (of the tool relative to the work piece) can be controlled by G96 (constant surface speed). Using this code will cause the spindle to rotate faster as the tool moves closer to the center of the work (X=0) and slower as the tool moves further from the center of the work (larger diameters). G96 S#### specifies the speed in feet/minute (in inch mode) or meters/minute (in metric mode). G97 (revolution per minute) cancels G96.

To calculate the spindle speed at a diameter use:

$$\text{Spindle RPM's} = \frac{(\text{Constant Surface Speed})\,(12)}{(\pi)\,(\text{Diameter})} \quad \textit{(Inches)}$$

OR

$$\text{Spindle RPM's} = \frac{(\text{Constant Surface Speed})\,(1000)}{(\pi)\,(\text{Diameter})} \quad \textit{(Metric)}$$

Notes: 1. In G0 (Rapid Mode) moves, the surface speed is calculated on the target position of the move, not on the actual position during the move.

2: To clamp the spindle speed (as the tool moves toward smaller diameters), use G50 S#### (#### is in RPM's). The maximum spindle speed is only effective in G96 (constant surface mode).

Revolutions Per Minute (RPM) (G97)

G97 S#### commands the spindle to run at #### RPM's (with the spindle speed override at 100%) regardless of the tool position. G96 (constant surface speed) cancels G97.

Notes 1: The spindle override allows percentages of the programmed spindle speed to be applied at percentages of 0%, 10%, 20% thru 130% and 175%.

2: When switching from G97 to G96 or G96 to G97 the previous commanded speed for the new mode (G96 or G97) will be in effect.

Feed Per Minute (G98)

With G98 active the feedrates are commanded by the value of F####.#. Units are specified in inches per minute (in inch mode) or millimeters per minute (in metric mode). G99 (feed per revolution) cancels G98.

Feed per Revolution (G99)

With G99 active the feeds are specified by distance per revolution of the spindle (i.e. the faster the spindle spins, the faster the tool moves). G99 F.#### specifies the distance per revolution in inches per revolution (inch mode) or millimeters per revolution (metric mode).

Note 1: The feedrate override allows percentages of the programmed feedrate to be applied.

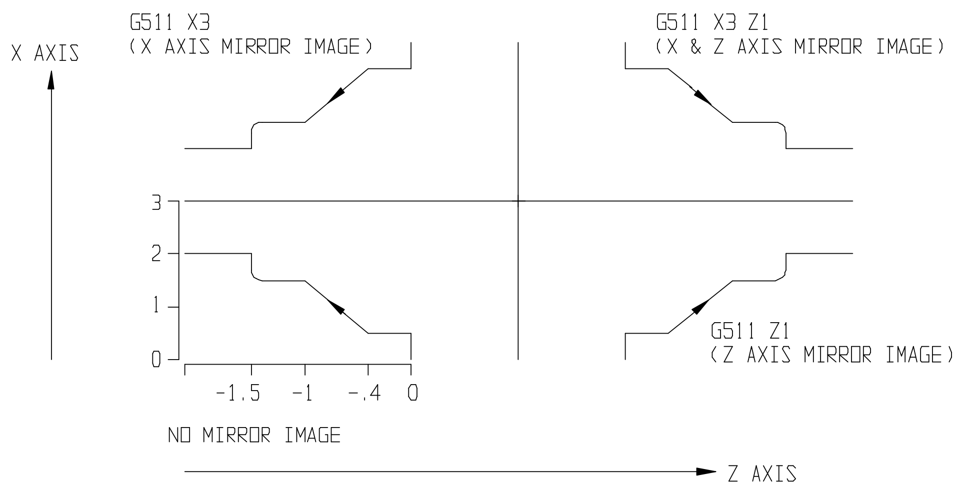
Note 2: When G0 (rapid position) moves are commanded the programmed feedrates are not considered.

Cancel Mirror Image (G501), Set Mirror Image (G511)

The mirror image commands allow mirroring about any centerline. The mirror image centerline is not affected by either scaling or rotation being on or off.

The command is as follows: G511 X____ Z____

X and Z are the axes to mirror and the distance from the current coordinate zero to create the mirror centerline. There must be at least one X or Z after the G511 command.



Original Path G0 X.5 Z0
G1 Z-.4
X1.5 Z-1
Z-1.5,R.1
X2
Z-2

G501 cancels mirror image.

Mirroring an axis is similar to scaling by -1 except that cutter compensation works on mirroring, and scaling by -1 doesn't.

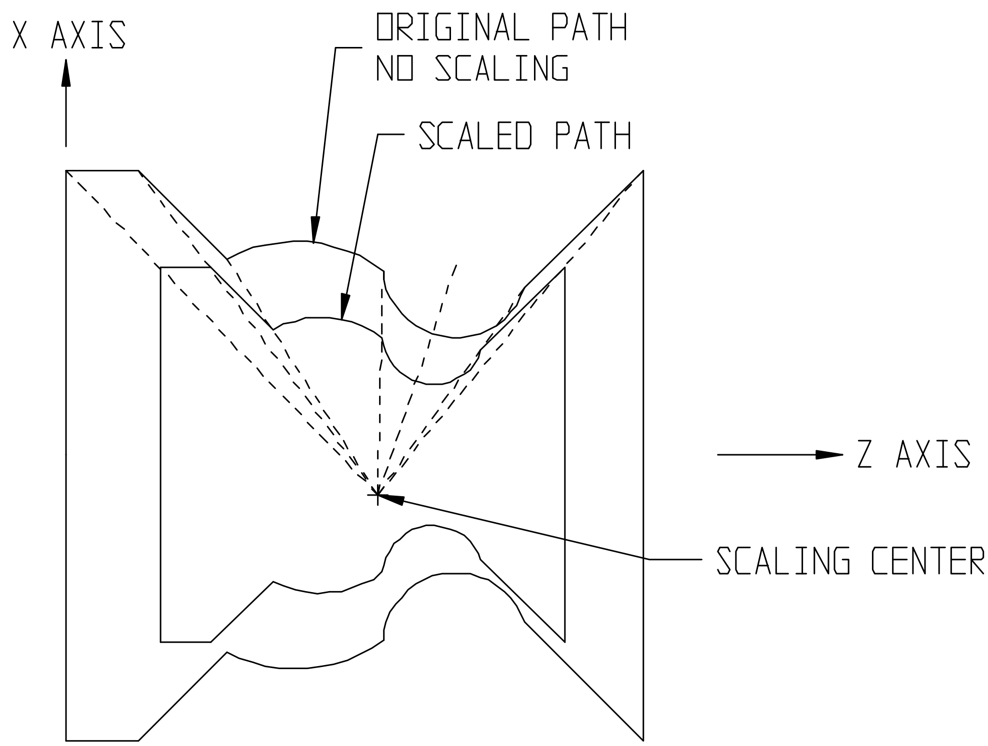
Cancel Scaling (G550), Set Scaling (G551)

Scaling can be commanded at any time during a program by using the G551 command.

Command format: G551 I____ K____ X____ Z____

I and K are the scaling center. If I and K are not specified in the G551 line, the scaling center will default to the last center used. The scaling center is set to 0, at the start of each program and after a G550.

X and Z are the scale factors for each axis. The range of each scale factor is ±999.9999 to ±000.0001. The scale factors, once set, remain in effect until changed or cancelled by a G550. At the start of each program all scale factors are set to 1.



Notes on Scaling

1. Once set, scaling remains in effect until cancelled by a G550.
2. If arcs are being scaled, the Z axis scale factors are used to scale the radius.
3. G28, G29, G30 and G50 are not affected by the scale factors.
4. To scale all axes to the same scale factor you can use G551 P.
5. G550 sets scale factors to 1 and scaling centers to 0.

Coordinate System Rotation (G568 - G569)

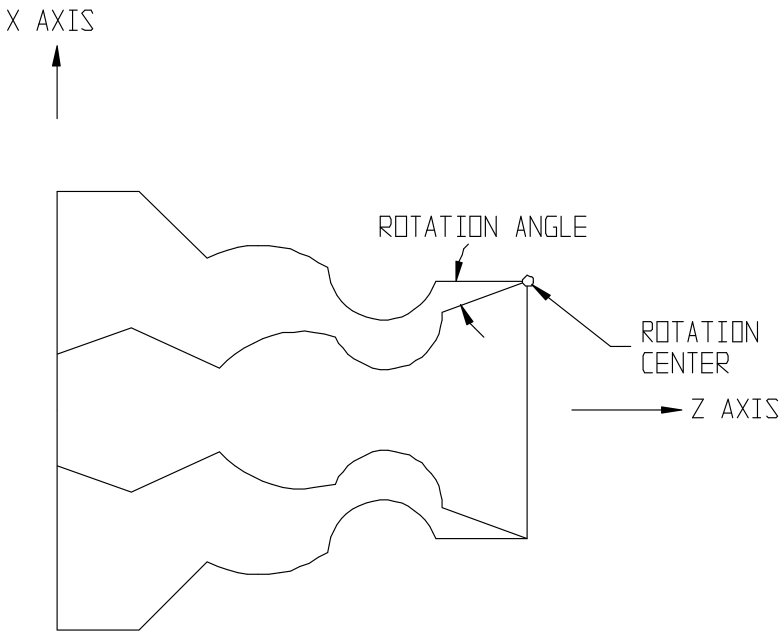
G568 can be used to rotate a programmed shape about a predefined center point. The center of rotation is defined by I and K, and the angle of rotation by AA. The command format is as follows:

G568 AA+_____ I____ K____

AA+ is CCW
AA- is CW

I and K values specify the center of rotation. The center of rotation defaults to the current position. If the IK's are not present in the G568 block the center of rotation will be the current axis position.

G569 zeroes the rotation angle and rotation center.



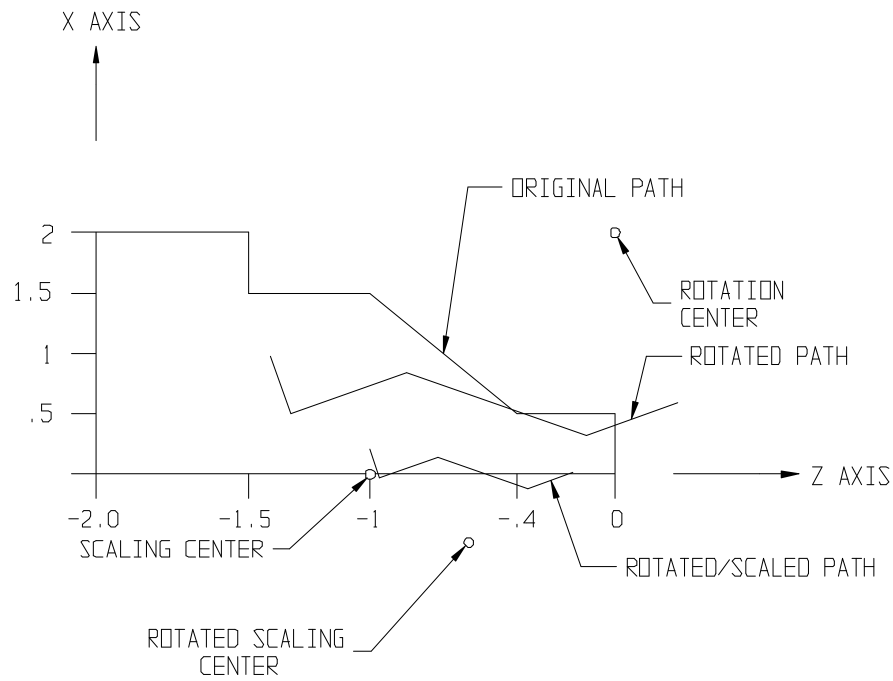
Care needs to be taken when using rotation in conjunction with other functions. Functions such as mirror image, scaling and cutter compensation need to be thought about carefully when used together with rotations. Some of the basic rules are as follows:

1. Cutter compensation should be off (G40) when rotation is called.
2. If scaling is on before rotation, the rotation center will be scaled; if rotation is called before scaling, the rotation center will not be scaled.
3. The order of on and off is first on last off.

   G551 . . . . scaling on
   G568 . . . . rotation on
   G41 . . . . .cutter compensation on
   .
   .
   .
   G40 . . . . .cutter compensation off
   G569 . . . . rotation off
   G550 . . . . scaling off

4. If the rotational center is scaled it will remain scaled until replaced by a new center or cancelled with G569.
5. X and Z can be used instead of I and K for rotation center.
6. R can be used instead of AA for rotation angle.

Part Scaled Then Rotated:
G551 I0 K-1 X.5 Z.5
G568 X2 Z0 AA20
X.5 Z0
Z-.4
X1.5 Z-1
Z-1.5
X2



Part Rotated Then Scaled:
G568 X2 Z0 AA20
G551 I0 K-1 X.5 Z.5
X.5 Z0
Z-.4
X1.5 Z-1
Z-1.5
X2

G776 Clear Stock (solid graph mode only)

Clears all cuts from the stock

G778 Cylinder Stock (solid graph mode only)

Creates cylinder stock
G778 Z3 X2 I1
Z is the length
X is the diameter
I is the inside diameter (No I for a solid cylinder

G980 Set feed per rev mode

The same as G95

G981 Set feed per rev mode and wait for the spindle marker

The same as G95 and G980 but the control will wait for the spindle marker before starting each move.

G982 Unlocks the feedrate override

G983 Locks the feedrate override at 100%

G984 Unlocks the spindle override

G985 Locks the spindle override at 100%

A discussion follows on several specialized and non-standard G codes.

G990/G991 Store Restore Parameters

Pp L1 Qq G990 (store parameters)
Pp L1 Qq G991 (restore parameters)

G990/991 allows parameters to be saved and restored.

Parameters are:

Pp (base parameter number, default 0),
L1 (number of parameters, default 10),
Qq (file identification, default 0).

CAUTION: Have no other commands on the line.

Example:

N0020 (My subroutines)
P0 L10 Q20 G990 (Save P0-P9)

P0 = ... (freely use and modify P0-P9)

P0 L10 Q20 G991 (restore P0-P9)
M99 (return to caller with original P0-P9)

Force Error (G997)

Forces an error code to be displayed
Error code generated is round (parameter #1)

Example:

P1=1407
G997 forces error #1407 -axis excess error to be displayed. It does not excess error, it only displays the error.

Note 1: P1=0 will not produce an error.
Note 2: All valid error codes on the control are between 1 and 1549.

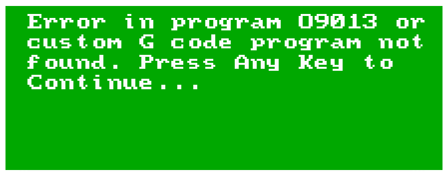
Beep (G998)

G998 will cause the speaker to beep if a speaker is installed.

Custom G Codes

Custom G codes can be created to execute a user-defined cycle such as a nonexistent drilling cycle. To setup the new custom G code, first create a text program (numbered from 9010 to 9019) that defines the new G code. Next, enter the number of the new custom code in the F3 (Power) parameter section after the newly created text program.

The text program for the custom G code can reside in the RAM directory or parts directory. The program in the RAM directory holds precedence over the program in the parts directory. If you call any custom M or G code from within a custom code, it will execute its normal function. If a syntax error exists in a custom code, the following window will appear upon power-up.



Example: Set F3 (Power) parameter custom G code O9014 to 005, and enter the program into RAM/O9014. Each time you execute a G5 it is similar to a call to RAM/O9014.

7

Section 7 Contents

M-Code description

These codes are used if the operator is programming the 8000T Series control in the text or MDI mode. They are also generated from conversational programs. It should be noted that most programmers, particularly new programmers, use the conversational programming mode. If you are planning to use text mode programming, pay close attention to this section explaining these codes. If you are planning to use conversational mode programming, you can ignore or skim over this section and concentrate on the conversational section.

The Miscellaneous Function codes are one or two digit numbers preceded by the letter M. If the code is less than 10, zero entry is optional (M02 or M2). These codes are used to perform a variety of machine and control functions as listed in the following table.

Note: The majority of the M codes that work with I/O are specific to Milltronics machines; however, they are flexible and can be tailored to specific applications. There may be several other M codes not listed here that deal with optional features on specific machines.

M Codes	Function	Executed Before Move	Executed After Move
M0	Program Stop		X
M1	Optional Stop		X
M2	End of Program		X
M30	End of Program / Spindle Off		X
M3	Spindle on CW	X *	
M4	Spindle on CCW	X *	
M5	Spindle Off		X *
M8	Flood Coolant On	X *	
M9	Coolants Off		X *
M31	Emergency Stop	X *	
M32	Wait Channel	X *	
M90	Graphics Off	X *	
M91	Graphics On	X *	
M98	Program Call Statement	X *	
M99	End of Program Statement		X

Caution: *The control will accept more than one M code on a line; however, it is recommended that only one M code per line be programmed. When more than one M code per line exists, the order of execution is somewhat undefined and the program may not run as expected. In general the M codes will execute in numerical order "M0 first M99 last" unless they have been defined to execute after the move statements. (See section on machine setup.)*

* These functions are selectable for either before or after the move.

Program stop (M0)

The execution of the program is halted on the block containing the M0 and prompts the operator to press cycle start. Program execution will be resumed when CYCLE START is pushed. If M0 is on a line with a move command the move will be executed before the stop. If there is a comment on the M0 block it will be displayed to prompt the operator.

Optional stop (M1)

M1 is the same as M0 except it is only executed if the optional stop switch is enabled.

Block skip (/)

A line of program can be skipped or ignored by the control. Inserting a "/" at the beginning of a line and enabling the BLOCK SKIP will cause the control to skip that line. In the example below with BLOCK SKIP disabled, the machine will move to the first, second, and third points. When BLOCK SKIP is enabled, the machine will move to the first then third points. Block two is skipped.

```
N1   X0 Z0
/N2  X2 Z2
N3   X4 Z0
```

End of program (M2, M30, M99)

All of these codes can be used to indicate the end of a program. The only difference is that M2 and M99 will leave the spindle and coolant on while the M30 will turn them off. All of the codes will return to the beginning of the program and start over when CYCLE START is pushed. If the code is executed in a subprogram, execution returns to the calling program.

Spindle on/off (M3, M4, M5)

These codes turn the spindle on CW (M3), CCW (M4) and off (M5). The spindle on commands will be executed before an axis command. M5 will execute after an axis command. The spindle set up parameters determine the exact sequencing each command will use when turning the spindle on and off.

Coolants on/off (M8, M9)

The Coolant On command (M8) is executed before an axis command. The Coolant Off command (M9) will be executed after an axis command.

Miscellaneous M codes (M65/75, M67/77, M68/78, M69/79, M50/60)

The standard M code is controlled by M65 (on) and M75 (off). These optional M codes control the four spare functions. Spare function one is controlled by M67 (on) and M77 (off). Spare function two is controlled by M68 (on) and M78 (off). Spare function three is controlled by M69 (on) and M79 (off). Spare function four is controlled by M50 (on) and M60 (off).

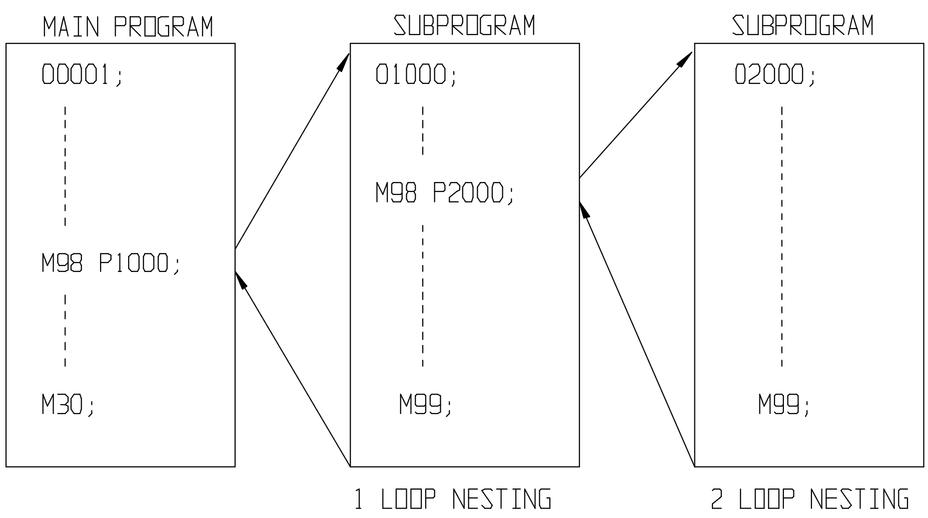
Graphics off/on (M90, M91)

On a few rare occasions the graphics function on the CNC should be turned off to prevent redundant graphics. A general rule of thumb when writing a program with loops is to do an M90, Graphics Off, after the first loop. This prevents redundant lines from building up in the graphics memory. After the loop is finished M91, Graphics On, can be executed to display the next section of the program. If the last command executed was a "Graphics Off" and the program is started over, the "Run" program will always restore the CNC to a "Graphics On" state. The size of the graphics file is limited to a size set by the machine tool builder. If this size is exceeded, no additional graphics will be added to the screen.

Subprogram call (M98)
Subprogram terminate (M99)

The subprogram call command (M98) can be used to execute any program residing in memory from another program. The called program will be executed until the end, and then control will transfer back to the calling program one block after the M98 command.

When the main program calls a subprogram it is regarded as a one loop nest. Thus a two loop nesting can be executed as shown below.



An M98 command when used with an L___ command can call a subprogram repeatedly. An L___ command can specify up to 999 repetitions of a subprogram.

Nesting up to 50 levels is allowed.

M2 or M30 can be used instead of M99. If a subprogram ends without an M2, M30 or M99 it will return to the calling program as if an M2, M30 or M99 was encountered.

Note: "CALL" can be used in place of "M98".

Preparation of subprogram

A subprogram is the same as any other program.

Specifying M99 at the end of a subprogram is optional. If the program was called by an "M98" or "CALL", an M2, M30 or M98 will return. Subprograms are entered into memory the same as normal programs.

Note: M30 also shuts the spindle and coolant off.

Subprogram execution

A subprogram is executed when called by the main program or another subprogram. A subprogram call has the following format:

M98 P#### L###

L### — number of times to repeat the subprogram

P#### — subprogram number

Example: M98 P0002 L5

M98 P2 L5

This command is read like this: call subprogram number 2 five times.

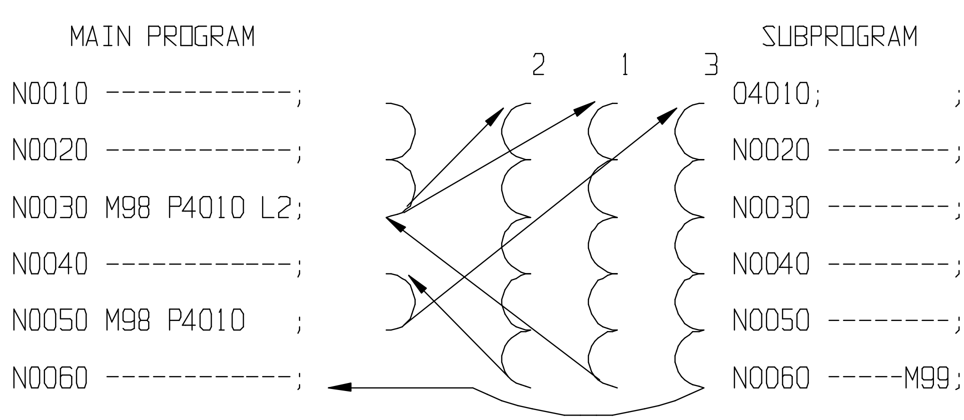
When the loop number is omitted the subprogram is run once.

A subprogram call command and move command can be specified in the same block.

Example: X1 M98 P0200

In this example the subprogram #200 is called after completing movement in the X axis direction.

The execution sequence of a main program, which calls a subprogram is as follows:



When the subprogram is called by another subprogram, it is executed in the same sequence as shown in the above example.

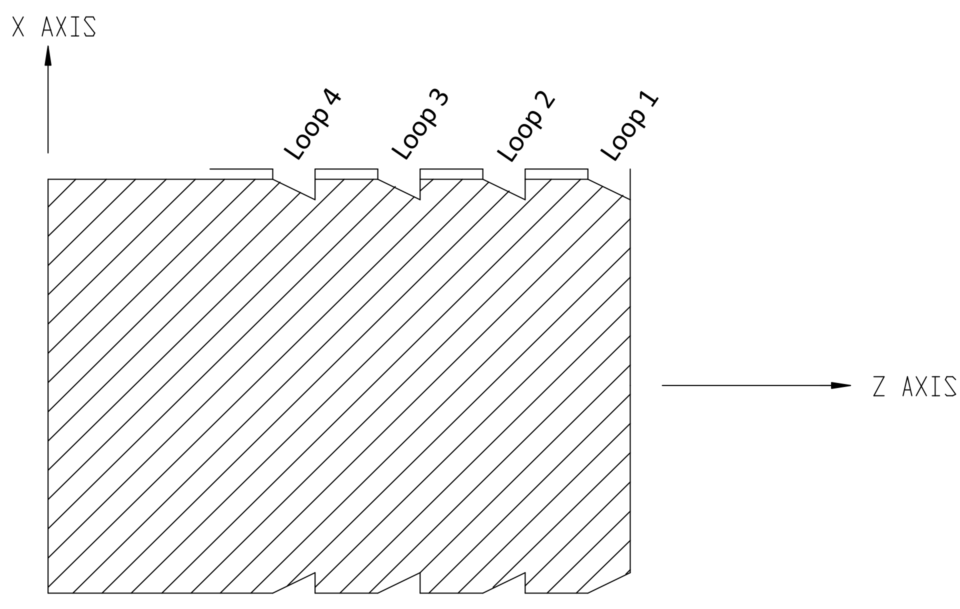
Note 1: *If the subprogram number specified cannot be found, A 603 Error "Program o### Does Not Exist" message is displayed.*

Note 2: *If a subprogram modifies the work offset, (normally G50 on the first block of the subroutine) the work offset will be restored to the same value upon leaving the subprogram as it was when entering the subprogram. This can be useful for cutting the same pattern at different locations.*

Note 3: *When using M98 to Call a program, the called program does not need to be in a separate file.*

Example:

```
G0  X2.1 Z0
M98  P2  L4
M30

O0002
G50  X2.1   Z0
G1  X1.8

X2  Z-.2
G0  X2.1
Z-.5
```



Instead of using M98, a CALL statement can be used.

Example 1: Call O0002 (call program named O0002)
Example 2: Call XYZ.PRG (call program named XYZ.PRG)

Note that "L" for number of loops cannot be used with the CALL statement.

8

Section 8 Contents

Parametric Programming - Definition

Parametric programming is similar to macro programming in that equations can be used to specify axis position rather than decimal numbers. The 8000 Series control does not restrict the use of parametrics to subroutines or macros. They may be used anywhere throughout a program. Parametric expressions may be used to specify M, G, F, and S functions. When a parametric expression is used for an axis position, it will first be evaluated and then cutter compensation will be applied. All the normal cutter compensation rules will apply to the evaluated point. Values generated by equations may be displayed on user parameter screens. Other listed system parameters may be used as input data to parametric equations, but under normal circumstances these parameters should not be changed.

Parametric Reference

A parameter reference is specified by the letter "P" followed by a valid parameter number. When a parameter reference is used for a coordinate position it must be contained in brackets.

Example: X [P10]
Z [-P145]

Parametric Assignment Command

Assigning is the most basic command in the use of parameters. The assignment character is an equal (=) sign.
Assignment statements replace the current value of a variable with a new value.

Example: P1 = 1.234

In this example the value 1.234 is assigned to parameter 1. Therefore both of the following commands would move to the same coordinate position.

X1.234 or X[P1]

Note: If assigning parameters numbers greater than 699, the control needs to be in the data mode (G10).

PB Byte Parameter

Byte parameters generally hold values that are representative of the machine configuration. For example, the spindle range parameter PB50 will hold a value of 1 for low range, 2 for medium, and 3 for high.

Example: PB50=2

Note: Byte parameters are not written in the verify mode.

Arithmetic Operators

The following list shows the available arithmetic operators.

Operator	Operation
+	addition
-	subtraction
*	multiplication
**	exponent
/	division
DIV	integer division
MOD	remainder

Note: The value of A DIV B is the mathematical quotient of A/B with any fractional portion or remainder dropped.

Examples:		
	3/2 = 1.5	3 DIV 2 = 1
	24/5 = 4.8	24 DIV 5 = 4
	72/8 = 9.0	72 DIV 8 = 9
	5.46/2.1 = 2.6	5.46 DIV 2.1 = 2

The MOD returns the remainder obtained by dividing its two numbers.

3 MOD 2 = 1 remainder = 1
24 MOD 5 = 4 remainder = 4
72 MOD 8 = 0 remainder = 0
3.57 MOD 2.1 = 1.47 Fract (3.57 ÷ 2.1 =.7)
(.7 x 2.1 = 1.47)

Relational Operators

The following list shows the available relational operators.

Operator	Operation
EQ or =	equal
NE or <>	not equal
LT or <	less than
GT or >	greater than
LE or <=	less or equal
GE or >=	greater or equal

Function Operators

A function call is specified by the function name (e.g. SIN, ATAN, . . .) followed by the function argument in brackets. When a function is used for a coordinate position it must be contained in brackets.

Examples: X [SIN [45]]
Z [SQRT[9]]

A function returns a value and can be used interchangeably anywhere a decimal value is accepted.

The functions supported are as follows.

Sine (SIN) returns the sine of the argument.

SIN [90] = 1

Cosine (COS) returns the cosine of the argument.

COS [180] = -1

Tangent (TAN) returns the tangent of the argument.

TAN [135] = -1

Arctangent (ATAN) returns the arctangent of the argument. The argument must be specified in fractional form (e.g. ½, 2/1, -5/6, ...).

ATAN [1/-1] = 135

Square root (SQRT) returns the square root of the argument.

SQRT [9] = 3

Absolute value (ABS) returns the absolute value of the argument.

ABS [-15] = 15

Integer (INT) returns the integer part of the argument.

INT [5.5099] = 5

Random (RND) returns an integer random number between 0 and its argument.
RND [25] will return an integer >=0 and <=24

Natural Logarithm (LN) returns the natural logarithm of the argument.

LN [10] = 2.3026

Exponential (EXP) returns the exponential of the argument.

EXP [1] = 2.7183

Rounds (ROUND) rounds a decimal value to an integer value.

Values halfway in-between are rounded up.

ROUND [2.3] = 2
ROUND [7.88] = 8
ROUND [1.5] = 2
ROUND [-1.5] = -2

Values exactly halfway between are rounded to the nearest even number.

ROUND [2.5] = 3
ROUND [3.5] = 4

Mathematic Expressions

Any combination of the previously described expressions are made up of arithmetic functions.

Examples: X[SIN[P123]*COS[P124]]
Z[[P2DIV3]+[P2MOD3]]

Conditional Statements

The 8000 Series control supports two types of conditional statements. These statements are used to transfer control of a program from one point to another based on some condition generated in the program. These statements are the IF-THEN statement and the WHILE-WEND statement.

IF-THEN

The IF-THEN statement is a way of conditionally executing a block if the result of an expression evaluates to true. The expression must contain one of the relational operators which allows the expression to be reduced to either true or false. If the expression is true, the THEN portion of the IF statement is executed. If the expression is false, the next line after the IF-THEN statement is executed.

Example of General Form:

	[any]	(relational)	[any]	
If	[mathematical]	(operator)	[mathematical]	Then any action
	[expression]		[expression]	

```
N20 IF P1 LT P2 THEN GOTO 15
N21
          Or

N20 IF P1 < P2 GOTO 15
N21
```

The above two statements accomplish the same thing. If the statement is true, N15 is executed; if it is false, N21 is executed.

Examples: IF P1*P3/COS[P90] GE TAN[P6] THEN X1

IF P4/P3 LT P6 GOTO 25

IF P1 = P2 THEN P4 = P5 - P6

Multiple IF statements can be used to check for multiple conditions.

Example: IF P36<5 THEN IF P1<>0 THEN M5
Defined, this means if P36 is less than 5 and P1 does not equal 0, shut the spindle off.

Note: The word THEN is optional in all cases.

WHILE-WEND

The second type of conditional statement is the WHILE-WEND statement. A WHILE statement contains an expression that controls the repeated execution of the blocks contained between the WHILE and WEND statements.

The expression controlling the repetition must contain one of the set of relational operators which allows the expression to be reduced to either true or false. The expression is evaluated before the contained blocks are executed. The contained blocks are executed repeatedly as long as the expression is true. If the expression is false at the beginning the blocks are not executed.

```
Example:   N20    WHILE [[P2*P3]/COS[P6]] LT P2
           N21    P6 = P6 + 1
           N22    Y[P2] Z[P3]
           N23    X[P6]
           N24    X1 Y0 Z0
           N25    WEND
           N26    M30
```

In this example lines N20 thru N25 will be repeated until the WHILE expression becomes false. Then line N26 will be executed instead of N21.

Note: Nested WHILE loops are allowed.

Transfer Statements

Transfer statements transfer control from one section of a program to another. They are unconditional transfers in that when the statement is executed, control always transfers. The GOSUB/RETURN, and CALL statements return control to the N+1 block after they are finished, and the GOTO statement transfers control to the specified block without a return.

GOTO Statement

The statement N### defines a label. GOTO's/GOSUB's can branch or transfer control to blocks containing these labels. A GOTO statement transfers progam execution to the block prefixed by the block label referenced in the GOTO statement.

GOTO 30 (The next block executed is block N30.)

Note: If there is more than one N30 program, it will transfer to the first N30.

CALL Statement

A CALL statement transfers control to any program residing in the CNC's memory. Upon completion of the called program or an M2, M99, or M30, control is returned to the main program at the block immediately following the CALL statement.

The CALL format is as follows.

```
CALL    XXXX
        Program
        Name
```

The call statement is similar to an M98.

Example:
X1 Y1
.
.
Call ABC (call to ABC)
.
.
.
M99 (the M99 here is optional) Notes on Call Statement
Note 1: Subprogram call allows nesting 50 levels deep.
Note 2: See page 6-40 for information on using G50 in subprograms.
Note 3: You cannot use the L option to loop when using the CALL STATEMENT.

GOSUB and RETURN

A GOSUB transfers program execution to the block number specified in the GOSUB statement. Execution will continue until a block containing a RETURN statement is encountered. The RETURN will transfer control back to the block immediately following the GOSUB statement. To use a GOSUB statement, the called block number must be part of the same program. Generally the subroutines are at the end of the main program.

The GOSUB format is as follows.

GOSUB	####	L###
	Line #	Loop Count (optional)

If the L is omitted the GOSUB routine will be executed once.

```
N1
N2
N3
N4 GOSUB 100                  Main Program
N5
.
.
.
N90 M30
N100
N101
.
.                             Subroutine
.
N200
N201 RETURN
N202
```

When the GOSUB is executed in N4 the program will jump to N100 and start executing until N201 is reached. At N201 control will transfer to N5 and lines N5 thru N90 will be executed. The M30 will terminate the main program and keep lines 100 thru 202 from being executed again. Note: subroutine nesting is allowed to 50 levels deep.

Computational Functions

1. Tangent Arc TANA
2. Tangent Line TANL
3. Point Circle Generate CGEN

The above three functions can be used anywhere throughout a program to solve various intersection problems. These functions receive input data in parameters P90 thru P96, and they return the answer in parameters P80 through P85. The answers can then be used in line and circle commands to produce the desired results. The format for these three functions follows.

TANA Tangent Arc and TANL Tangent Line

General Format for TANA & TANL

Input Parameters

P90 = Z1 center of arc 1
P91 = X1 center of arc 1
P92 = R1 radius of arc 1
P93 = Z2 center of arc 2
P94 = X2 center of arc 2

Calculated Output Parameters

P80 = Zs Z starting point of tangent arc or line
P81 = Xs X starting point of tangent arc or line
P82 = Ze Z end point of tangent arc or line
P83 = Xe X end point of tangent arc or line
P84 = Zt ZC center of tangent arc (TANA case only)
P85 = Xt XC center of tangent arc (TANA case only)

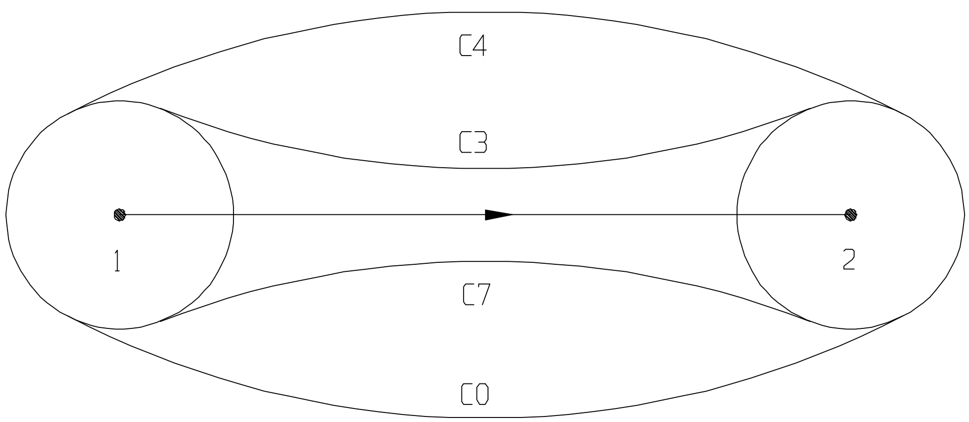
The general form for a tangent arc or line function is as follows.

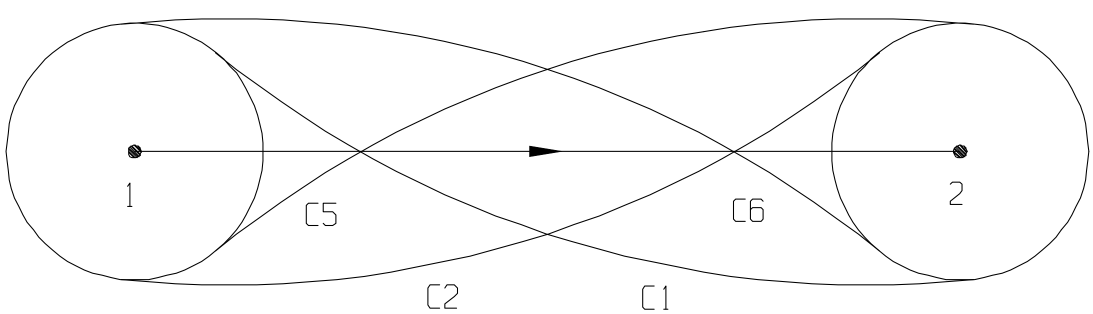
TANA C# or TANL C#

is a number 0 through 7 in the tangent arc case, and 0 through 3 in the tangent line case. This number selects one of the eight possible solutions of the TANA or one of the four solutions of the TANL. The values of C# are defined as the tangent point being to the right or left of a line connecting the centers of the arcs and center of the connecting arc, when facing in the direction of tool movement. See the following diagrams for illustration of tangent solutions.

TANA Cases

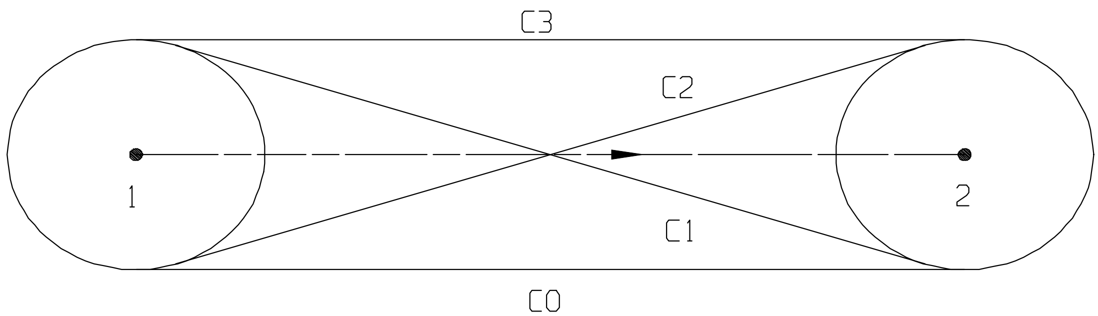
1st	2nd	Center
C0 = Right	Right	Left
C1 = Left	Right	Left
C2 = Right	Left	Left
C3 = Left	Left	Left
C4 = Left	Left	Right
C5 = Right	Left	Right
C6 = Left	Right	Right
C7 = Right	Right	Right





TANL Cases

	1st	2nd
C0=	Right	Right
C1=	Left	Right
C2=	Right	Left
C3=	Left	Left



Sample Program Using TANA or TANL

N1 P90=0 (ZC of arc 1)
N2 P91=0 (XC of arc 1)
N3 P92=1.5 (radius of arc 1)
N4 P93=5 (ZC of arc 2)
N5 P94=4 (XC of arc 2)
N6 P95=2 (radius of arc 2)
N7 P96=5 (radius of tangent arc, not used for tangent line)
N8 TANA C3 or TANL C3 (calculate line or arc points)
N9 G2 R1.5 XC0 ZC0 X[P81] Z[P80] (first arc #1)
N10 G3 R5 XC[P85] ZC[P84] X[P83] Z[P82] (TANA calculated center and end points)
(or)
N10 G1 X[P83] Z[P82] (TANL calculated end point)
N11 G2 R2 XC4 ZC5 X_{end} Z_{end} (second arc #2)
Depends on next line of program…

CGEN Circle Generate

The circle generate function will calculate the center and radius of an arc through any three non-co-linear points. The general format for the CGen function is as follows.

Input Parameters

P90=Z1 P91=X1	coordinates of first point
P92=Z2 P93=X2	coordinates of second point
P94=Z3 P95=X3	coordinates of third point

Output Parameters

P80=ZC P81=XC	center of calculated circle
P82=R	radius of calculated circle

Sample Program Using CGen

N1 P90=0 P91=0 first point
N2 P92=1 P93=1 second point
N3 P94=2 P95=0 third point
N4 CGEN
N5 G1 X0 Z0 position to start of arc
N6 G2 R[P82] XC[P81] ZC[P80] X end____ Z end_____
N7 … Program continues

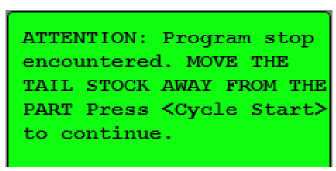
The CGEN function can be used anytime throughout the program to calculate the radius and center of an arc. These calculations can then be used in a normal arc command along with trig help, chamfer, corner round, extend back, and any other function available.

Note: The arc direction is returned in P97 so the block N6 may be substituted with G17 G[P97] R[P82] XC[P81] ZC[P80] X end ______ Yend________

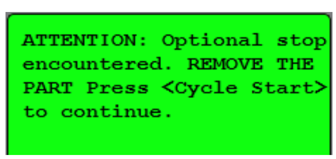
Miscellaneous Commands

Comments are any text enclosed in parentheses and they are ignored by the control. Comments can be anywhere in a program or in a block. When a comment is on a block with an M0 (block stop) or M1 (optional stop), the comment will be displayed in the message window.

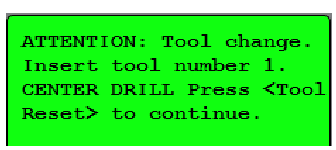
Example: M1 (MOVE TAIL STOCK AWAY FROM THE PART)



Example: M1 (REMOVE THE PART)



Example: T0707 (CENTER DRILL)



Spaces Spaces can be used anywhere within the program. For example, Z1.234 can be written as Z 1 . 23 4 if desired.

Blocks Blocks without any information are allowed.

Print

Print is used to print text to the screen. PRINT can be used to display text, parameter values, times, dates, etc. on the History Line (above the graphics window).

Example: P87=32.45
PRINT [N=#87] shows "N=32.4500"

#n[LT] #n[LT] displays parameter n with L leading digits and T trailing digits.

Example: P100=1.235 P101=2.87656
PRINT [P100=#100[04] P101=#101[33]] shows
"P100=1.2350 P101=002.877"

If the leading and trailing fields are left blank, the default leading and trailing format for the machine setup parameters is used.

When LT=0, exceptions apply.
If LT=0, the ASCII value of the parameter is shown.

Example: P1=72 (ASCII H)
P2=105 (ASCII i)
PRINT [#1[00]#2[00]] shows "Hi"

If LT=-1, the parameter is ignored. The time is shown in format 0hh:mm:ss. If LT=-2, the parameter is ignored. The date is shown in format yy/mm/dd.

Example: PRINT [Time=#1[-1] Date=#1[-2]] shows "Time=012:51:52 Date=97/10/23"

Dprnt (print to file or RS-232 port)

DPRNT outputs text to file or to the RS-232 port.
See Control Parameters DPRNT Data to set the file name to DPRNT to.
If the file name is blank data is written to REPORT.DAT

Example: DPRNT [DATA FOR HEAD CASTINGS]

#n writes the value of a parameter

Example: DPRNT [X#208 Z#210] outputs the current X and Z positions to a file or an RS-232 port.

#n[LT] will output parameter n with L leading and T trailing digits.

Example: P100=1.235 P101=2.87656
DPRNT [P100=#100[04] P101=#101[33]] outputs
"P100=1.2350 P101=002.877"

If the leading and trailing fields are left blank, the default leading and trailing format for the machine setup parameters is used.

If LT=00, the ASCII value of the parameter is shown.

Example: P1=40 (ASCII ( )
P2=72 (ASCII H)
P3=105 (ASCII i) P4=41 (ASCII))
DPRNT [#1[00]#2[00]#3[00]#4[00]] outputs (Hi)

If LT=-1, the parameter is ignored. The time is shown in format 0hh:mm:ss.

If LT=-2, the parameter is ignored. The date is shown in format yy/mm/dd.

Example: DPRNT [Time=#1[-1] Date=#1[-2]] shows "Time=12:51:52 Date=2012/05/23"

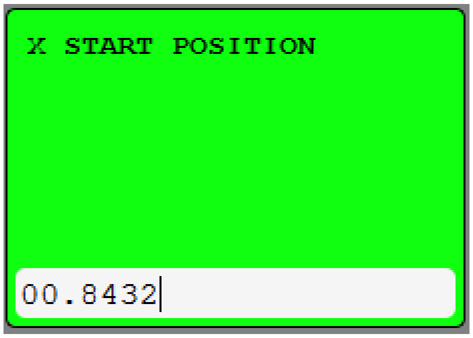
DPRNT can be used to generate report files relating to parts production, parts inspections, digitizing information, etc.

To begin writing to a new file, do DPRNT [REWRITE].

INPUT The INPUT statement is used for data input from the front panel.

Example: INPUT (X START POSITION) P1

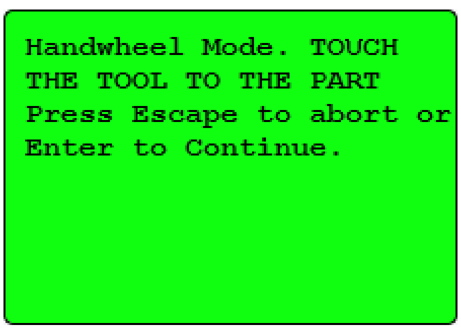
The operator will be prompted to input data.



The operator can use the data displayed by pressing the ENTER key. If ESC is pressed during an input statement, the program will be terminated.

The HDW command can be used to enter the handwheel mode during a program. A comment may be added to prompt the operator during the HDW command. HDW is ignored when verifying a program.

Example: HDW (TOUCH THE TOOL TO THE PART) prompts the operator with the following message.



The operator presses the <enter> key after handwheeling the machine. If <esc> is pressed during handwheeling, the program will be terminated.

To manually set and clear outputs, refer to the following

STO STO sets an output.

Example: STO20 (sets the 20th output)

CLO CLO clears an output.

Example: CLO25 (clears the 25th output)

IPIN

IPIN refers to an input pin. The argument is the input number. IPIN [32] is the 32nd input. The IPIN statements can be used in conditional statement or to assign variables.

Examples:

```
IF IPIN[20] EQ 1 THEN PB50=1

WHILE IPIN[10] NE 0
M62
M63
WEND
P5=P3 + IPIN[41]
```

OPIN

OPIN refers to an output pin. The argument is the output number. OPIN [14] is the 14th output. The OPIN statements can be used in conditional statements or to assign variables.

Parts Inspection and Digitizing Commands

PROBE1 can be used with any axis move to command an interrupted move. The move will be terminated when an input goes low.

Example: PROBE1 G98 F1 X1.3 Z-2
The X and Z move toward 1.3 and -2 respectively until the probe contacts the part.

PROBE2 may be used with any axis move to command an interrupted move. The move will be terminated when an input goes high.

Example: PROBE2 G98 F1 X1.3 Z-2
The X and Z move toward 1.3 and -2 respectively until the probe releases from the part.

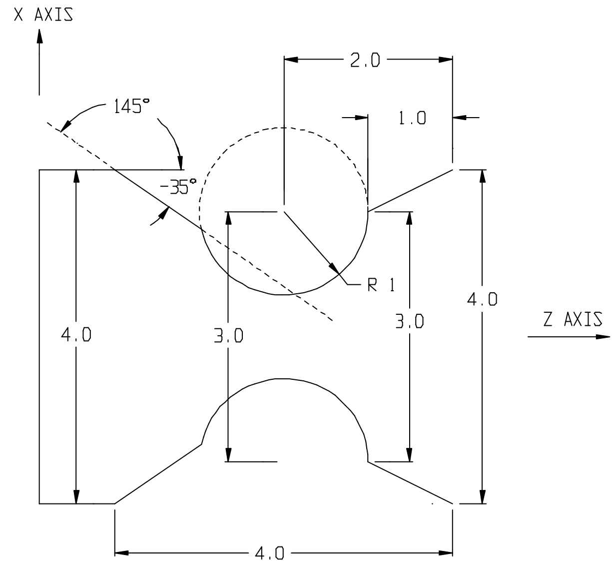
Note: The input used by the probe commands is selectable in the miscellaneous parameters.

Back line

The back line function can be used on any line command. This function reverses the direction of a programmed line. It would normally be used when you know the end point of the line and not its start point. The end point would be programmed and the line would be extended backwards to the start point. When using this function all Trig Help functions remain valid.

Example 1:

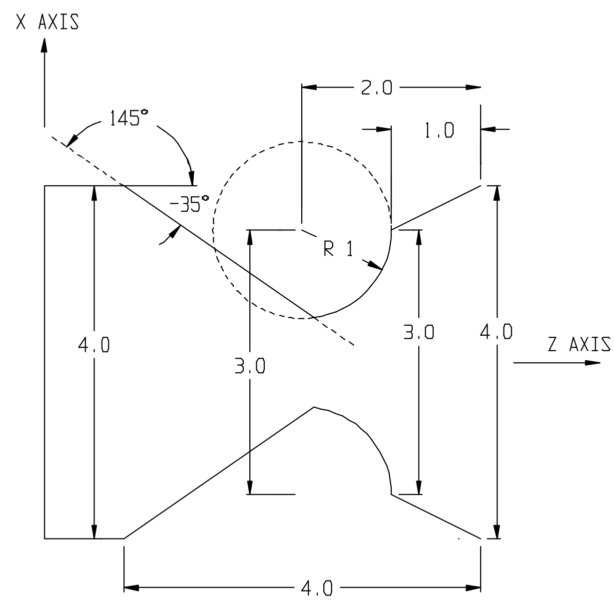
(1) X4 Z0
(2) X3 Z-1
(3) G2 R1 ZC-2 AB270
(4) G1 X4 Z-4 BACK C0 W145



Back	extend line backwards from (4,-4)
C0	use the arc intersection closest to (4,-4)
W145	extend the line from (4,-4) at an angle of 145°

Example 2:

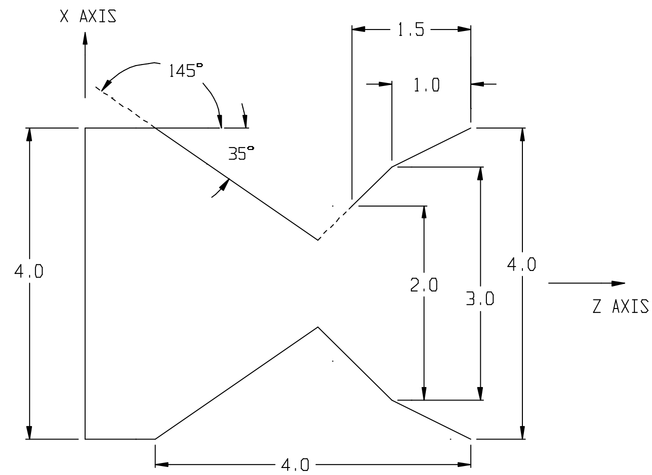
(1) X4 Z0
(2) X3 Z1
(3) G2 R1 ZC-2 AB270
(4) G1 X4 Z-4 BACK C2 W145



C2 use farthest intersection

Example 3:

(1) X4 Z0
(2) X3 Z-1
(3) X2 Z-1.5
(4) X4 Z-4 BACK C0
or C2 W-35



This example used a back line between two lines to program an unknown point.

MOD

MOD is used to shift an axis position. It is generally used for rotary axis to obtain a positive position between 0 and 360 degrees. It can be useful after a rotary axis has made several revolutions in the same direction.

Examples:

G0 C750
MOD C360 (C axis position is now 30°)

G0 C-100
MOD C360 (C axis position is now 240°)

G0 C-500
MOD C360 (C axis position is now 220°)

G0 C437
MOD C20 (C axis position is now 17°)

G0 C33.285
MOD C2.1 (C axis position is now 1.785°)
33.285 / 2.1 = 15.85
15.85 remainder is .85
*.85 * 2.1 = 1.785*

Note: Using the MOD command also shifts the software limits. This is desirable for most rotary axis, but is undesirable for linear axis.

ORIGIN

Origin is another way to shift axis coordinates. It shifts the axis coordinates by the operator amount.

Examples:

G0 X437
ORIGIN X20 (X axis position is now 417)
G0 X33.285
ORIGIN X2.1 (X axis position is now 31.185)
G0 X361
ORIGIN X360 (X axis position is now 1)
G0 X-500
ORIGIN X360 (X axis position is now -860)

Note: The origin command also shifts the software limits.

HOME

To home an axis within a program, use the HOME command.

Example 1: Home Z0 (Homes the Z axis the number “0” doesn’t mean anything)
Example 2: Home X0Z0 (Homes the X and Z axis)

Sample Program Using Some Special Statements

```
(DIGITIZING PROGRAM)
M5
T0101
G00
Z15

INPUT (OUTER DIAMETER) P1 (prompt for data)
INPUT (INNER DIAMETER) P4
INPUT (PROBE INCREMENT) P5
DPRNT [REWRITE]
DPRNT [(WHEEL CUTTING)] (write data to file)
DPRNT [M42] (custom code for cutting program to shift to 2nd gear)
DPRNT [G0 Z2]
DPRNT [X#1] (X to outer diameter)
DPRNT [G99 F.01] (feeds to cut at)
DPRNT [G50 S1200] (speeds to cut at)
DPRNT [G96 S1200]
DPRNT [G1]
DPRNT [M3]

P157=P1 (set length and diameter for graphics)
P156=P142
G0 Z1
X[P1] (move to outer diameter to start digitizing)
G98
P2=P1 (X position)
G1 F20
N1
  PROBE1 Z-6 (probe fast to the part)
  PROBE2 Z1  (probe off the part)
  F.5
  PROBE1 Z-6 (probe slow to the part)

  P3=P209*2 (X diameter = current X position *2)
  DPRNT[G1 X#3 Z#208] (write current X and Z positions to file)

  F5
  P2=P2-P5 (subtract probe increment from current X position)
  X[P2] (move over to next X position)
  PROBE2 Z1 (move off part)

IF P2>P4 GOTO 1 (are we at the inner diameter yet?)

G0R.1 AB0 (retract the probe)
X6.0
Z10.
X17.25
```

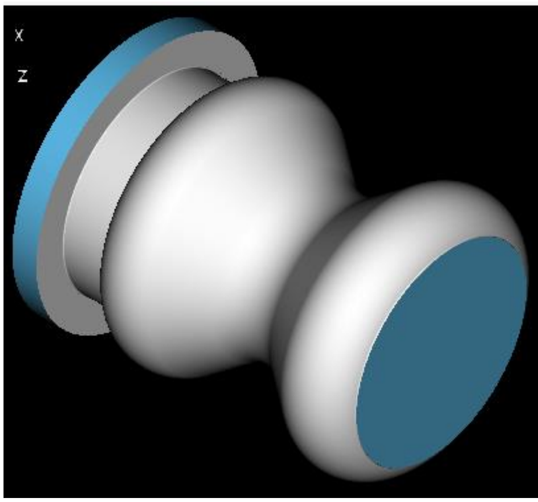
```
DPRNT[G0R.1AB0] (retract for the cutting program)
DPRNT[X6]
DPRNT[Z10.]
DPRNT[M5]
DPRNT[X#1]
```

OUTPUT FROM THE DIGITIZING PROGRAM

```
(WHEEL CUTTING)
M42
G0 Z2
X16.4856
G99 F.01
G50 S1200
G96 S1200
G1
M3
G1 X16.4856Z-0.1214
G1 X16.4656Z-0.0931
G1 X16.4456Z-0.0752
G1 X16.4256Z-0.0614
G1 X16.4056Z-0.0500
G1 X16.3856Z-0.0404
G1 X16.3656Z-0.0318
G1 X16.3456Z-0.0246
   .
   .
   .
G1 X6.1256Z-0.3276
G1 X6.1056Z-0.3258
G1 X6.0856Z-0.3241
G1 X6.0656Z-0.3224
G1 X6.0456Z-0.3206
G1 X6.0256Z-0.3189
G1 X6.0056Z-0.3172
G0R.1AB0
X6
Z10.
M5
X16.4856
```

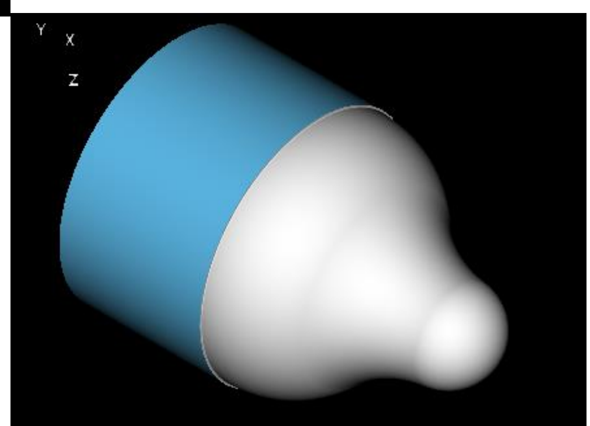
9

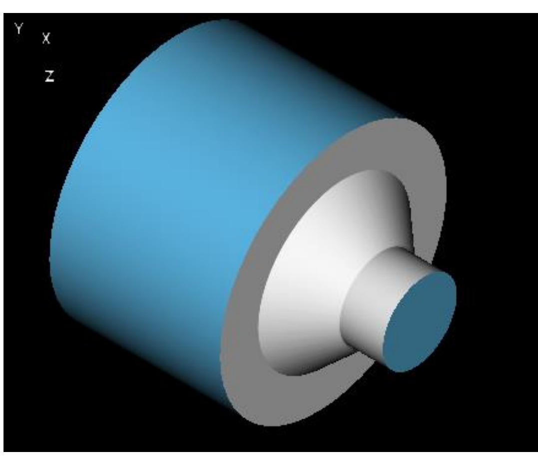
Section 9 Contents



1

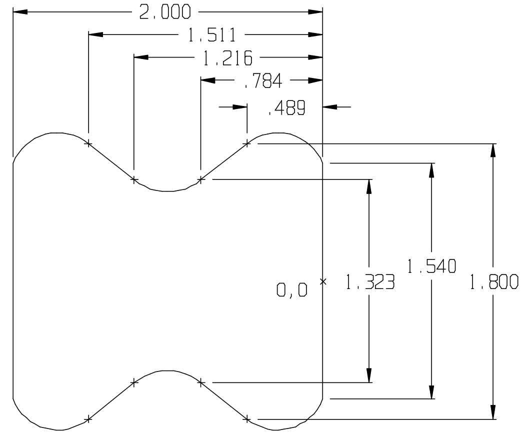
2





3

Sample Program #1



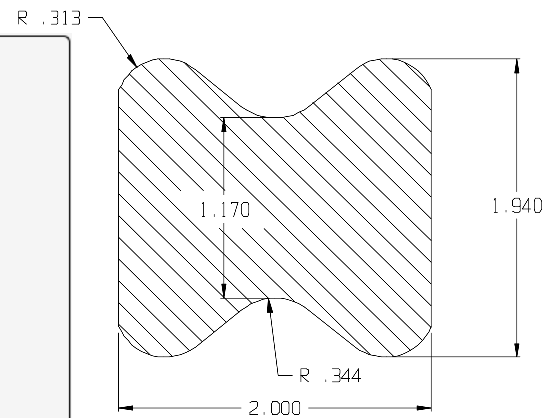
Event 0 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Program Setup
Program name: ROLLER EXAMPLE

Units: Inch

Material Length: 2.5
Outside Diameter: 2 d
Inside Diameter: d (leave this field blank for solid stock)

Setup notes



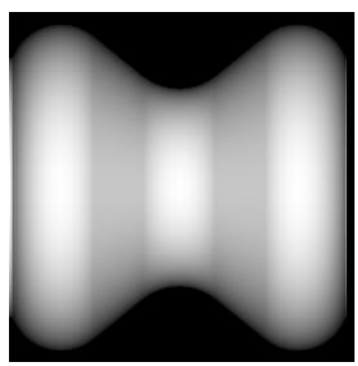
Event 1 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Tool Change

Tool Number T 1
Tool Description ROUGHING TOOL
Feed per Revolution .01

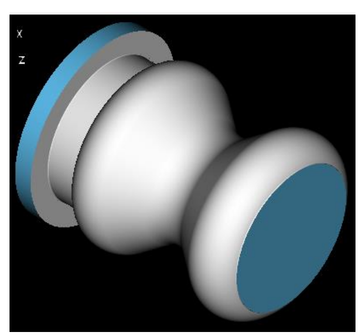
Spindle Constant Surface Speed
Max Spindle Speed RPM 2000
Constant Surface Spd S 600
Stop Command ---

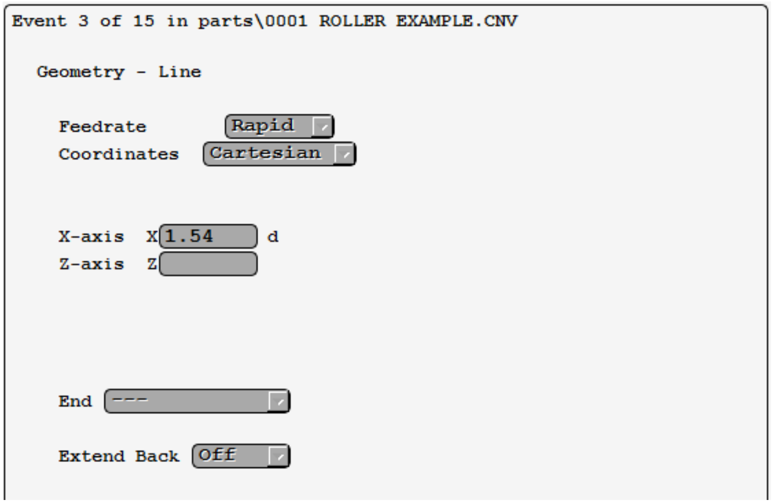
Spindle Direction CW
Coolant On



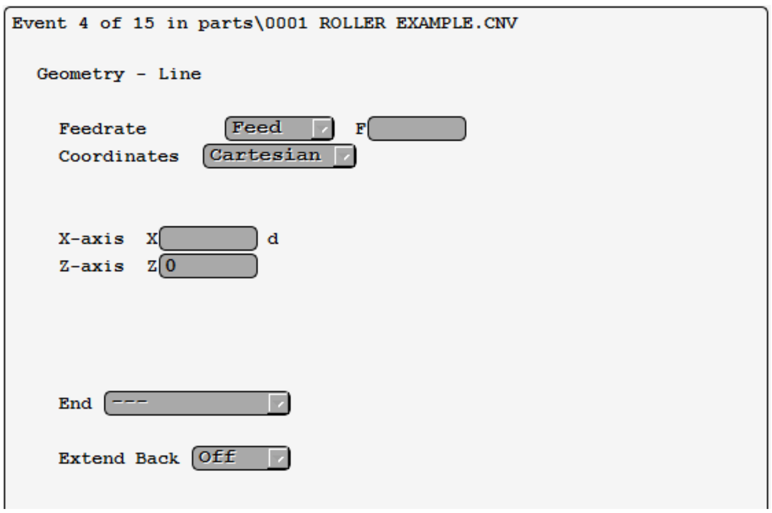
Event 2 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Type Rough Turning Setup
X Finish Stock U .015 d
Z Finish Stock W 0
X Rough Stock I .025 r
Z Rough Stock K 0
1st X Depth V 1.975 d
Cut Increment D .05 r
Relief Pattern
Clearance R .01 r
Compensation Right
Start Point X 2.1 d
Z .1
Pattern Number N 1
Note : Geometry between this Event and End Cycle define the Pattern

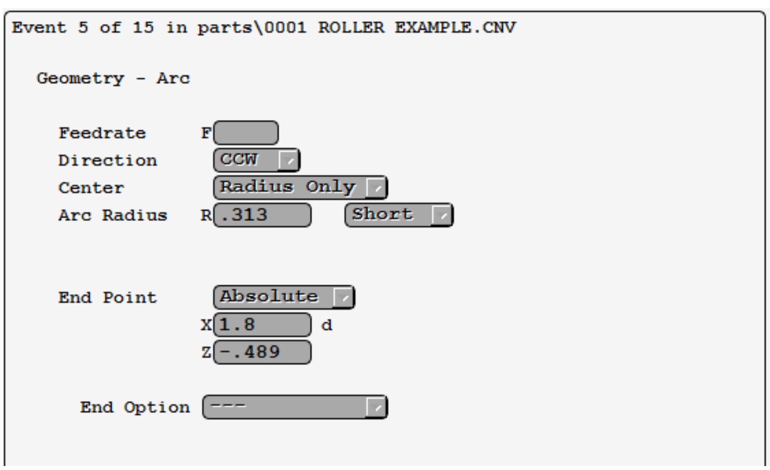



Event 3 of 15 in parts\0001 ROLLER EXAMPLE.CNV
Geometry - Line
Feedrate
Rapid
Coordinates
Cartesian
X-axis
X 1.54
d
Z-axis
Z
End

Extend Back
Off


Event 4 of 15 in parts\0001 ROLLER EXAMPLE.CNV
Geometry - Line
Feedrate
Feed
F
Coordinates
Cartesian
X-axis
X
d
Z-axis
Z 0
End

Extend Back
Off


Event 5 of 15 in parts\0001 ROLLER EXAMPLE.CNV
Geometry - Arc
Feedrate
F
Direction
CCW
Center
Radius Only
Arc Radius
R .313
Short
End Point
Absolute
X 1.8
d
Z -.489
End Option

Event 6 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Geometry - Line

Feedrate Feed F

Coordinates Cartesian

X-axis X 1.323 d

Z-axis Z -.784

End ---

Extend Back Off

Event 7 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Geometry - Arc

Feedrate F

Direction CW

Center Radius Only

Arc Radius R .344 Short

End Point Absolute

X 1.323 d

Z -1.216

End Option ---

Event 8 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Geometry - Line

Feedrate Feed F

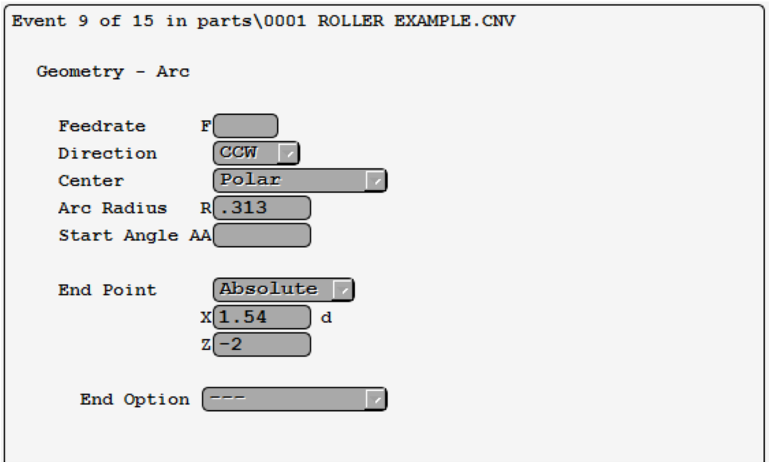
Coordinates Cartesian

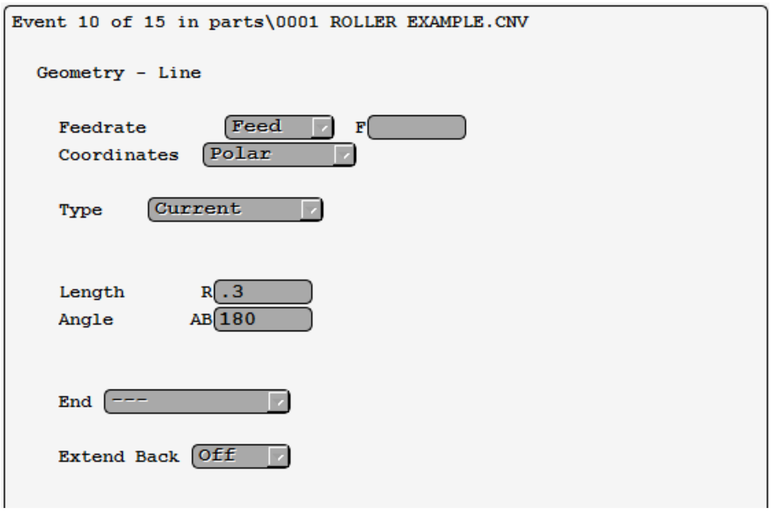
X-axis X 1.8 d

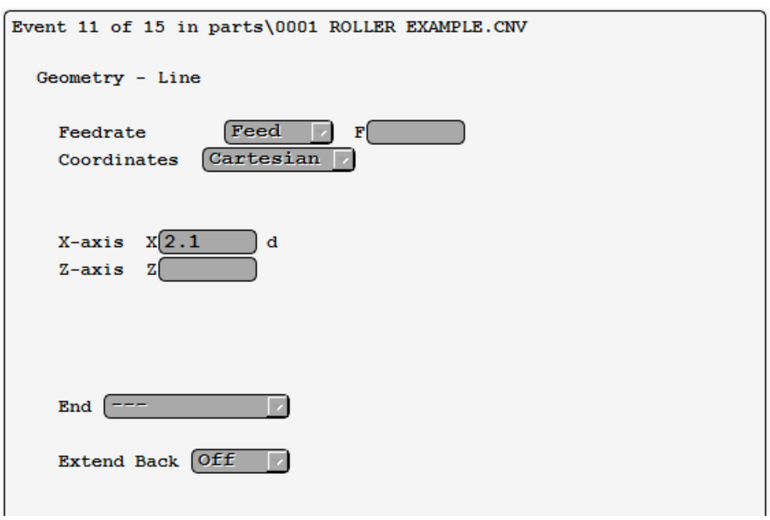
Z-axis Z -1.511

End ---

Extend Back Off


Event 9 of 15 in parts\0001 ROLLER EXAMPLE.CNV
Geometry - Arc
Feedrate F
Direction CCW
Center Polar
Arc Radius R .313
Start Angle AA
End Point Absolute
X 1.54 d
Z -2
End Option ---


Event 10 of 15 in parts\0001 ROLLER EXAMPLE.CNV
Geometry - Line
Feedrate Feed F
Coordinates Polar
Type Current
Length R .3
Angle AB 180
End ---
Extend Back Off


Event 11 of 15 in parts\0001 ROLLER EXAMPLE.CNV
Geometry - Line
Feedrate Feed F
Coordinates Cartesian
X-axis X 2.1 d
Z-axis Z
End ---
Extend Back Off

Event 12 of 15 in parts\0001 ROLLER EXAMPLE.CNV

End Turning Cycle

Event 13 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Tool Change

Tool Number	T 2
Tool Description	
Feed per	Revolution .005
Spindle	Constant Surface Speed
Max Spindle Speed	RPM 2000
Constant Surface Spd	S 500
Stop Command	---
Spindle Direction	CW
Coolant	On

Event 14 of 15 in parts\0001 ROLLER EXAMPLE.CNV

Finish Turning

Pattern Number	N 1
Feedrate	F
Spindle Speed	S
Start Point	X 2.1 d
	Z .1

Event 15 of 15 in parts\0001 ROLLER EXAMPLE.CNV

End of Program

Spindle off	Yes
Coolant off	Yes
X Position (home relative)	
Z Position (home relative)	
Shut the drives off	No

Sample Program #2

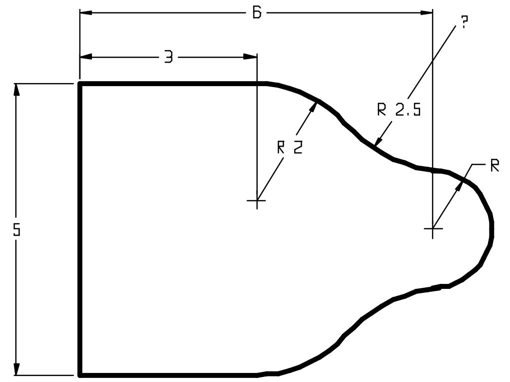
Event 0 of 9 in parts\TURN TANGENT ARC.CNV

Program Setup
Program name: TANGENT ARC

Units: Inch

Material Length: 7
Outside Diameter: 5 d
Inside Diameter: d (leave this field blank for solid stock)

Setup notes



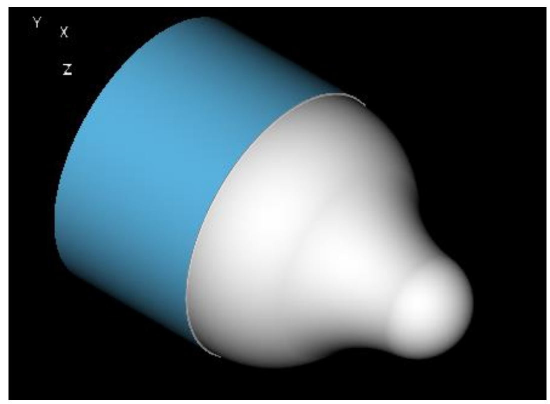
Event 1 of 9 in parts\TURN TANGENT ARC.CNV

Tool Change

Tool Number: T 1
Tool Description
Feed per: Revolution .03

Spindle: Constant Surface Speed
Max Spindle Speed: RPM 3000
Constant Surface Spd: S 500
Stop Command: ---

Spindle Direction: CW
Coolant: ---



Event 2 of 9 in parts\TURN TANGENT ARC.CNV

Type: Rough Turning Setup
X Finish Stock: U 0 d
Z Finish Stock: W 0
X Rough Stock: I .03 r
Z Rough Stock: K .01
1st X Depth: V 4.95 d
Cut Increment: D .04 r
Relief: Pattern
Clearance: R 0 r
Compensation: Right
Start Point: X 5.5 d
Z .5
Pattern Number: N 1
Note : Geometry between this Event
and End Cycle define the Pattern

Event 3 of 9 in parts\TURN TANGENT ARC.CNV

Geometry - Line

Feedrate: Rapid
Coordinates: Cartesian

X-axis X: 0 d
Z-axis Z:

End: ---

Extend Back: Off

Event 4 of 9 in parts\TURN TANGENT ARC.CNV

Geometry - Line

Feedrate: Feed F:
Coordinates: Cartesian

X-axis X: d
Z-axis Z: 0

End: ---

Extend Back: Off

Event 5 of 9 in parts\TURN TANGENT ARC.CNV

Connect two arcs with tangent line or arc.

Cut first arc in direction: CCW
R1: 1
XC1: 0 d ZC1: -1

Second Arc for computation is:
R2: 2
XC2: 1 d ZC2: -4

Exit 1st arc: Right nter 2nd arc: Right
Connect with: an Arc Center to the: Right
Radius: 2.5 Arc Direction: CW

```
Event 6 of 9 in parts\TURN TANGENT ARC.CNV

  Geometry - Arc

    Feedrate       F[    ]
    Direction       [CCW]
    Center          [Abs. Center]
    Arc Radius     R[2]
    Arc Center    XC[1]          d
                  ZC[-4]
    End Point       [Polar]
    End Angle     AB[90]

      End Option  [---]
```

```
Event 7 of 9 in parts\TURN TANGENT ARC.CNV

  Geometry - Line

    Feedrate        [Feed]    F[    ]
    Coordinates    [Cartesian]

    X-axis   X[5.5]      d
    Z-axis   Z[    ]

    End  [---]

    Extend Back [Off]
```

```
Event 8 of 9 in parts\TURN TANGENT ARC.CNV

End Turning Cycle
```

```
Event 9 of 9 in parts\TURN TANGENT ARC.CNV

End of Program

 Spindle off      [Yes]
 Coolant off      [Yes]

 X Position (home relative)[    ]
 Z Position (home relative)[    ]

 Shut the drives off [No]
```

Sample Program #3

Event 0 of 10 in parts\FACING CYCLE.CNV

Program Setup

Program name: FACING CYCLE EXAMP

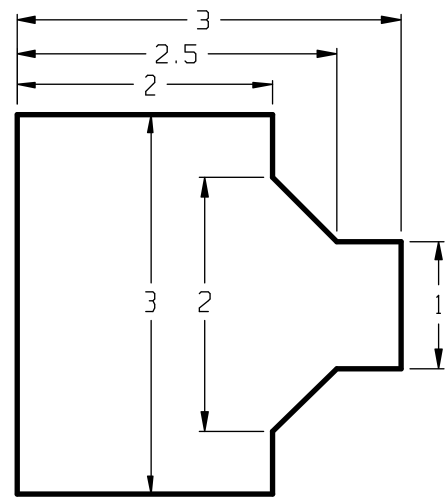
Units: Inch

Material Length: 3

Outside Diameter: 3 d

Inside Diameter: d (leave this field blank for solid stock)

Setup notes



Event 1 of 10 in parts\FACING CYCLE.CNV

Tool Change

Tool Number: T 1

Tool Description: ROUGH FACING

Feed per: Revolution .01

Spindle: Constant Surface Speed

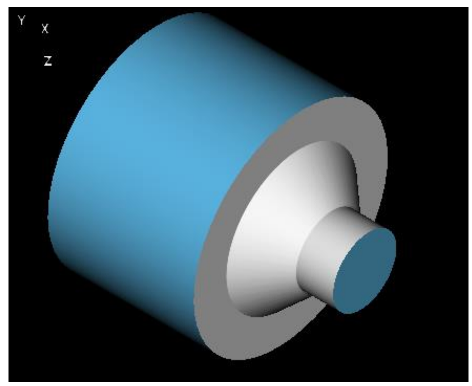
Max Spindle Speed: RPM 2000

Constant Surface Spd: S 500

Stop Command: ---

Spindle Direction: ---

Coolant: ---



Event 2 of 10 in parts\FACING CYCLE.CNV

Type: Rough Facing Setup

X Finish Stock: U .005 d

Z Finish Stock: W .02

X Rough Stock: I .005 r

Z Rough Stock: K .02

1st Z Depth: V -.02

Cut Increment: D .03

Relief: Pattern

Clearance: R 0

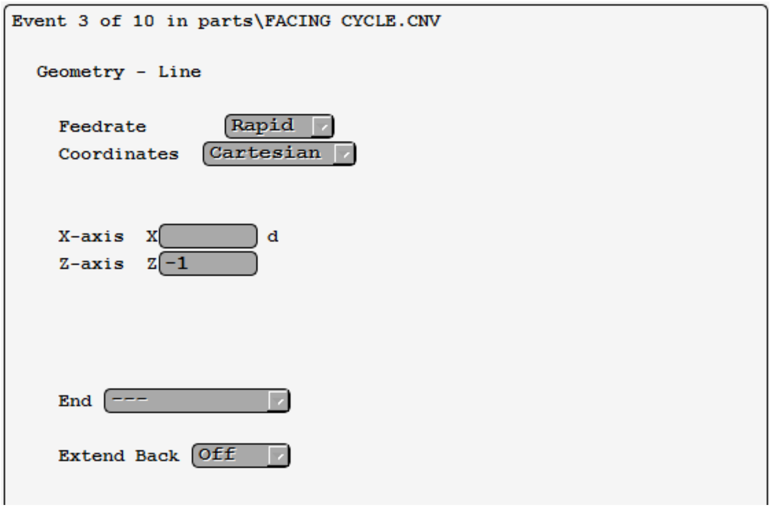
Compensation: Left

Start Point: X 3.1 d

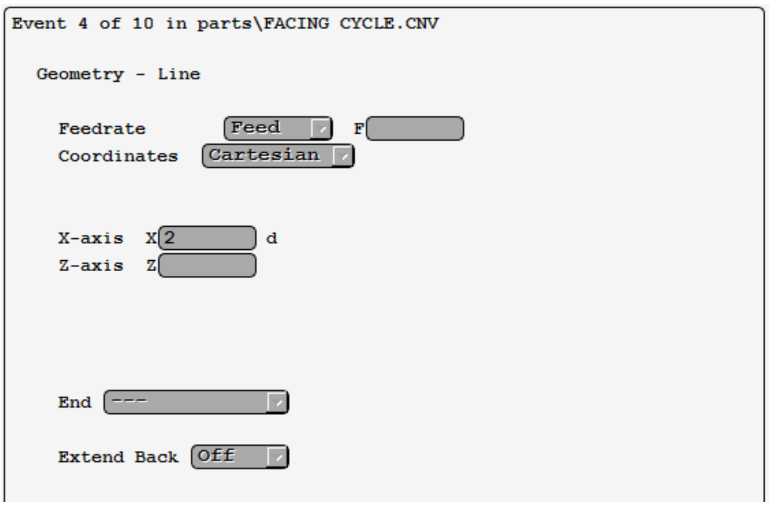
Z .1

Pattern Number: N 1

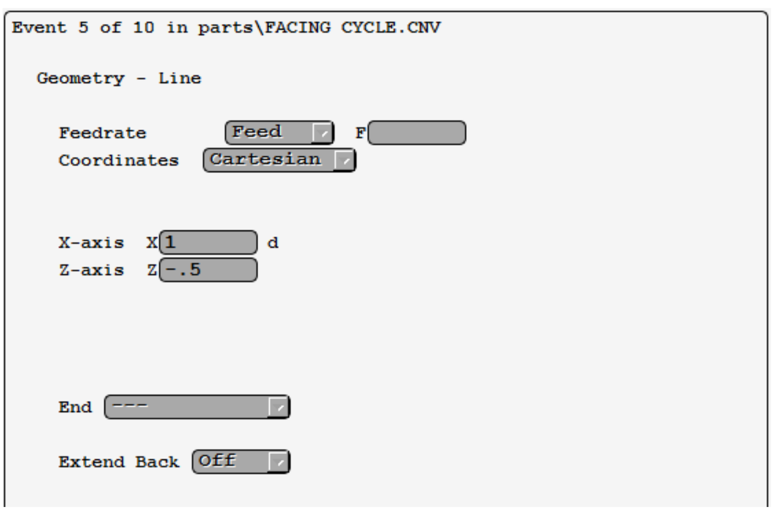
Note : Geometry between this Event and End Cycle define the Pattern


Event 3 of 10 in parts\FACING CYCLE.CNV
Geometry - Line
Feedrate
Rapid
Coordinates
Cartesian
X-axis X
d
Z-axis Z
-1
End

Extend Back
Off


Event 4 of 10 in parts\FACING CYCLE.CNV
Geometry - Line
Feedrate
Feed
F
Coordinates
Cartesian
X-axis X
2
d
Z-axis Z
End

Extend Back
Off


Event 5 of 10 in parts\FACING CYCLE.CNV
Geometry - Line
Feedrate
Feed
F
Coordinates
Cartesian
X-axis X
1
d
Z-axis Z
-.5
End

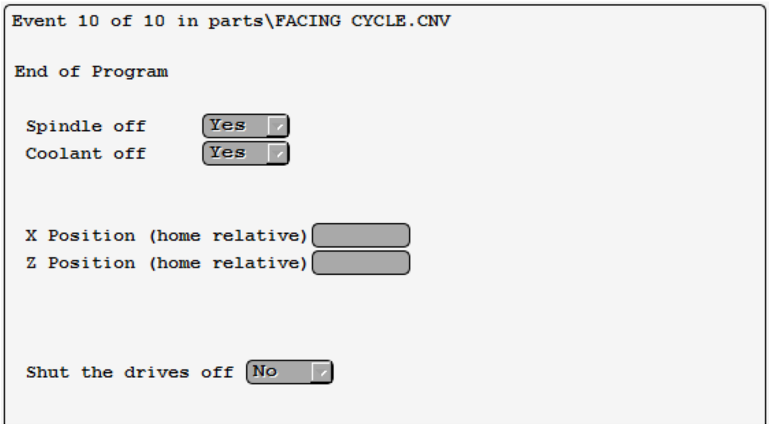
Extend Back
Off

```
Event 6 of 10 in parts\FACING CYCLE.CNV

  Geometry - Line

    Feedrate          Feed          F
    Coordinates     Cartesian

    X-axis   X               d
    Z-axis   Z .1

    End   ---

    Extend Back   Off
```

```
Event 7 of 10 in parts\FACING CYCLE.CNV

End Facing Cycle
```

```
Event 8 of 10 in parts\FACING CYCLE.CNV

Tool Change

  Tool Number               T 2
  Tool Description  FINISH FACING
  Feed per  Revolution      .007

  Spindle   Constant Surface Speed
  Max Spindle Speed      RPM 2000
  Constant Surface Spd     S 600
  Stop Command               ---

  Spindle Direction          CW
  Coolant                    On
```

```
Event 9 of 10 in parts\FACING CYCLE.CNV

Finish Facing

  Pattern Number     N 1

  Feedrate           F
  Spindle Speed      S

  Start Point        X 3.1   d
                     Z .1
```


Event 10 of 10 in parts\FACING CYCLE.CNV

End of Program

Spindle off Yes
Coolant off Yes

X Position (home relative)
Z Position (home relative)

Shut the drives off No

10

Section 10 Contents

Error Messages

200 Division by zero

207 Invalid floating point operation

321 Invalid custom G code

A custom G code has an error or a custom program not found.

322 Invalid custom M code

A custom M code has an error or a custom program not found.

324 Invalid path or program not found for tool change macro.

The tool change macro may have an error in it or can not be found.

399 Software Limit Pre-check

501 Illegal address "#" encountered

The character within quotes " " is not a valid address, such as X, Z, R, G, etc. The block where the error occurred is shown in the block display. Check that block for the invalid address.

502 Undefined canned cycle

503 Return without gosub

504 Coincident points, the start point and end point are the same on an arc without a center

The start point and end point are the same on an arc without a center.

505 Radius too small to span given points

Start and end points are more than "R" distance apart.

507 After compensation, line to arc lacks intersection

509 After compensation, arcs lack intersection

515 Unexpected file size

The control read a file that was not the correct size when using G991.

517 Parameter out of range

Parameter number is less than zero. For parameter numbers greater than 699 you must use data mode (G10, G11).

518 Illegal program statement

Command in program statement is not considered valid.

519 Feedrate out of range

The programmed feedrate is beyond the "maximum feedrate" parameter value in the machine setup parameters. The program feedrate may be negative.

520 Spindle speed out of range

The programmed spindle speed is beyond the "spindle range" parameter in the machine setup parameters. The programmed spindle speed may be negative.

521 Negative arc radius

An attempt was made to generate a negative arc radius.

522 Negative polar radius

A polar radius must be specified as a positive value.

523 Illegal tool number
Valid T numbers are 0 - 99
529 Duplicate address' encountered
The same address was found twice on the same block, such as X0 Z0 X.5.
530 Colinear line to line in round corner
531 Colinear line to arc in round corner
532 Colinear arc to line in round corner
533 Colinear arc to arc in round corner
535 Chamfer length is less than zero
Chamfer length must be a positive number.
536 Can't chamfer and round the same corner
Choose either chamfer or round corner.
537 Can't chamfer to or from arcs
538 Loop counter out of range
The maximum number of loops for a call is 999.
539 Dwell time out of range
Probably a negative number was specified. The maximum dwell time is 999999999 seconds.
540 Illegal dwell time "#" encountered
Try G4 F##.####; specify X, P, or F after G4.
541 No axis moves are allowed on a G45, G46, G47 or G48 block. *Relocate the axis move to another block.*
542 G30 Illegal return to reference parameter on G30 block
Should be P2, P3, or P4 for second, third, and fourth reference point.
543 Illegal (G10) statement
Only parameter assignments are allowed in G10 mode.
544 Too many digits in number
The number of digits used is beyond what the address is expecting.
546 Nested calls or gosubs too deep
Probably a program is calling itself. Nest limit is 50 for program calls.
547 Comment not closed
Always use "( )" (parentheses) in pairs for program comments.
549 Unrecognized G code
G code encountered is not recognized by the control.
550 Bad numeric format
Expecting a numeric value, or a parameter value enclosed within [ ], after an address X, Z, R, etc.
552 Missing "]"
Always use square brackets in pairs.
553 Missing "["

Always use square brackets in pairs.

554 Tangent function overflow

Trying to find the tangent of a number close to 90º

555 Missing “/”

Arctan “ATAN” syntax is P## = ATAN[#/#].

556 Negative SQRT argument

557 Unknown function

560 Illegal relational operator

567 Unresolved call

Program being called does not exist (CALL ####).

568 Unresolved GOTO or GOSUB

N#### does not exist in the program (GOTO ####).

569 The tool is too large to cut inside the arc “Compensated radius is too small”

Eliminate the arc, or use a smaller tool. Or set control parameter “ignore tool too large errors”.

570 The tool is too large to cut inside the arc “1st compensated radius in arc to arc is < 0”

Eliminate the arc, or use a smaller tool. Or set control parameter “ignore tool too large errors”.

571 The tool is too large to cut inside the arc “2nd compensated radius in arc to arc is < 0”

Eliminate the arc, or use a smaller tool. Or set control parameter “ignore tool too large errors”.

579 Compensated arcs do not intersect

580 Invalid floating point operation.

The argument passed to the LN function was zero or negative.

581 Invalid floating point operation.

The Operand passed to the “**” (exponent) function was zero or negative

582 X must be monotonously increasing or decreasing in type 1 of G71 rough cycle.
Pockets are not allowed in the ruff turning cycle with 45º relief.

584 X value exceeds starting X in G71 rough cycle

585 Z must be monotonously increasing or decreasing in type 1 of G72 rough cycle.
Pockets are not allowed in the ruff facing cycle with 45º relief.

587 Z value exceeds starting Z in G72 rough cycle

601 Missing WHILE statement

602 Missing WEND statement

603 Program does not exist

Program being called as a subprogram does not exist. Check to see if the program called is in the parts folder.

606 Program is empty

Text program being run or verified is empty. Try reposting the conversational file.

608 ‘P’ expected in M98 block

1188 Unknown hardware failure

1189 Watchdog Failure

The interface card reported that it dropped the watchdog because no information was received from the ncb card for about 3 heartbeats, about .0015 seconds. CAUSES:

1. *Fiber-optic cable loose.*
2. *Bad ncb card.*
3. *Bad interface card, since it must receive the information from the ncb card.*
4. *A power glitch at the interface card caused the interface card to reset (the interface card comes up with the watchdog off). If this is the case, however, the error may come up as an error 1194.*
5. *It is at least possible communication in one direction would work and not in the other. Especially since each direction has an independent channel, including a separate fiber optic cable. This error would be very rare, especially now that "interface reset" yields a separate error.*

1190 Unexpected control (NCB card) reset

The NCB card reset sometime after power-up, the only time it should reset. CAUSES:

1. *Power glitch at the NCB card.*
2. *Noise on the PC bus reset line, which is tied to the NCB card reset line.*

1192 Control (NCB card) Failure

The NCB card did not respond for a long period of time.

CAUSES: 1. Bad NCB card.

1194 Unexpected interface (card) reset

The interface card reset sometime after power-up, the only time it should reset.

CAUSES: 1. A power glitch at the interface card caused the interface card to reset (the interface card comes up with the watchdog off).

1195 Interface failure

No information was received from the interface card for 10 heartbeats of the control, about .05 seconds. CAUSES:

1. *Fiber-optic cable loose.*
2. *Bad interface card.*
3. *Bad NCB card, since it must receive the information from the interface card.*
4. *Bad fiber-optic cable.*

1395 Chiller fault

1401 Software limit

1407 Excess error

1413 Safe Zone

1437 Pay timer expired

1448 Low air pressure

1449 Drive fault

1450 Emergency stop

1549 Turret power fault

Byte Parameters

16 ....Measurement base for leading and trailing digits.
50 ....Spindle range
60 ....Has a tool setter
62 ....Primary handwheel encoder
64 ....Boot in metric
65 ....Number of axes
66 ....Boot to Feed/Min or Feed/Rev
67 ....Boot to RPM's or CSS (Constant surface speed)
68 ....Spindle axis
77 ....Check spindle up to speed; 0=OFF, 1=ON, 2=From Rapid to Feed
81 ....Special Flags; 2=Shut off trig help, 4=Shut off cutter comp.
85 ....DPRNT to a file or RS-232
86 ....Baud rate
87 ....Parity, data, and stop bits
88 ....Graph look-ahead
95 ....Front Turret
96 ....Power up with block skip on
97 ....Power up with optional stop on
98 ....Slow rapid %
99 ....MDI mode
100 ....Post M-code table
110-119 ....Custom G-code table
120-139 ....Custom M-code table
148 ....Tri-color cycle light
149 ....Servo motor on C-Axis
151 ....Log option
176-183 ....ADC comp axis
185 ....Allow modal restart
186 ....Allow Away Offset
189 ....Tap up double RPM
190 ....Check air pressure
191 ....Scratch
194 ....Data logging path
0 = Parts 1 = Network
204 ....Electronic spindle gears
208 ....Default Trig Help and cutter comp
212 ....Second handwheel Encoder #
214 ....Probe input #
215 ....Cranking factor *(for handwheeling through a program)*
219 ....European code
220 ....Feedhold input
224 ....Live Tool Spindle

227.................Cycle light output

For tri-color EOC light, the other two outputs are hard-coded

229.................AC axis drives

294.................Blocks of lookahead

295.................Reverse spindle in back gear

338.................Look scope blocks

371.................Safe zone ON at power-up

393.................Log axes

397.................Auto Spindle brake

399.................Z Work offset in Handwheel – For Tool 1 Only

400.................Ignore ‘tool radius too large’ errors

401.................X Axis tool wear by Radius or Diameter

402.................CPU Warning Temperature

404.................Front panel type *(molded front panel=2, BandR=3)*

411.................Tool Setting Any Tool, Current Tool

413.................Pre-check Software Limits *(0=Run Only; 1=Always; 2=Never)*

414.................Log auto-stop OFF

415.................Remote HDW Encoder

417.................Log beats skip

420.................Log input

475.................G5# tool set, add work offset or not.

500-599 ....Task global defaults

530.................Auger ON time

532.................Washdown ON time

533.................Washdown OFF Time

535.................Way Lube ON Time (Sec)

536.................Way Lube OFF Time (Min)

537.................AC Drives

A copy of AC Drives = 229 used in NCB file

539.................Clamp on OD

542.................Yaskawa M5 Spindle motor

545.................Front Panel Flags

- bit definition –

FPFLAGS_HOLD_TOOL_CHANGE	*$01;*
FPFLAGS_NOT_OVERRIDE_SWITCH	*$02;*
FPFLAGS_RAPID_OVERRIDE	*$04;*
FPFLAGS_NOT_MANUAL_PANEL	*$10;*

548.................Recycle Pump Time

Used on coolant through spindle (CTS)

549.................Check Up-to-speed2

Copy of PB77 for use in NCB files

550.................Spindle Brake Delay

552.................Up to speed input
555 ....Spindle Rotary Brake2
Copy of PB397 for use in NCB files
558.................Auto Door Opener
559.................Specifies Function of aux function button
562.................Door open timer
564.................Grease Cycle Time
565.................Grease Estop Time
571.................Way lube on time
572.................Way lube off time
*** Tool Table Fields
700.................X Length
701.................Z Length
702.................Radius
703.................X Length Wear
704.................Z Length Wear
705.................Radius Wear
706.................Tool Nose Angle
707.................Tool Mount Angle
708.................Size
714.................Tool Type
720.................Display Column Enable
721.................Column 1 Display
722.................Column 2 Display
723.................Column 3 Display
730.................Full Code Window
Full Code Window (over graphics window) in formRunVerify
801.................Stock Type
816-831 ....Thermo Comp Axis
832-847 ....Thermo Comp Encoder Master
998.................Single block in canned cycles
999.................Lathe Mode zero for mills non zero for lathes

Floating Point Parameters

P00-P100............User Parameters
P101 ....Thread lead
P102 ....Thread shift angle
P103 ....Thread chamfer length
P104 ....Thread chamfer angle
P105 ....Thread type
P106 ....Thread Height
P107 ....First cut amount
P108 ....Tool nose angle
P110 ....Start block
P111 ....End block
P112 ....X finish stock
P113 ....Z finish stock
P114 ....X ruff stock
P115 ....Z ruff stock
P116 ....Cut increment
P117 ....Ruff cut type (Pattern or 45)
P118 ....Relief
P119 ....First cut depth
P121 ....Cut length
P122 ....Cut depth
P123 ....End point
P124 ....Diameter
P125 ....Turning Taper
P126 ....Facing Taper
P130 ....Minimum cut (in G76)
P131 ....Finish passes (in G76)
P139 ....X R-Plane
P140 ....Z R-Plane
P141 ....Final depth
P142 ....Initial Level
P143 ....Cut Increment
P144 ....First Depth
P145 ....Plunge Feedrate
P146 ....Peckup Increment
P147 ....Peck Clearance
P148 ....Dwell 1
P149 ....Dwell 2
P151 ....X initial level
P152 ....X axis drill
P172 ....Mirror position (Z)
P173 ....Mirror position (X)

P180 ....Scale position (Z)
P181 ....Scale position (X)
P188 ....Scale factor (Z)
P189 ....Scale factor (X)
P196 ....Rotate K position
P197 ....Rotate I position
P198 ....Angle of rotation
P199 ....Abs/Inc in G76...G79
P200 ....Previous position (Z)
P201 ....Previous position (X)
P208 ....Current position (Z)
P209 ....Current position (X)
P216 ....Previous machine (Z)
P217 ....Previous machine (X)
P224 ....Current machine (Z)
P225 ....Current machine (X)
P232 ....Work offset (Z)
P233 ....Work offset (X)
P240 ....Tool offset (Z)
P241 ....Tool offset (X)
P248 ....Arc radius
P249 ....Arc K value
P250 ....Arc I value
P251 ....Feed rate
P252 ....Dwell
P253 ....Spindle speed
P260 ....Tool number
P261 ....Tool radius
P266 ....Safe zone status
P267 ....Unit ratio (inch=1 metric = .0394)
P268 ....Pending tool number
P298 ....CSS max spindle rpm
P304 ....Data mode
P306 ....Interpolate mode
P309 ....Cutter compensation mode
P310 ....Active canned cycle
P311 ....Dimension mode
P312 ....Feed unit
P313 ....Spindle unit
P315 ....Linear unit
P316 ....Scale mode
P317 ....Rotate mode
P318 ....Mirror mode
P319 ....Active work coordinate
P323 ....Return plane

P324 ....Tapping mode
P325 ....Custom code active
P326 ....Custom M-code
P327 ....Custom G-code
P328 ....Feed per Rev (wait for marker)
P329 ....Feed rate override lock
P330 ....Spindle override Lock
P335 ....Hard tap fudge factor
P337 ....Tool depth
P338 ....Tool size
P339 ....Tool nose angle
P340 ....Tool mount angle
P345 ....Bore type
P365 ....Feed per minute feed rate
P366 ....Feed per rev feed rate
P367 ....RPM spindle speed
P368 ....CSS spindle speed
P399 ....Spindle jog speed
P400 ....G50 (Z)
P401 ....G50 (X)
P416 ....Temp Z
P417 ....Temp X
P432 ....G52 (Z)
P433 ....G52 (X)
P448 ....Z tool change position
P449 ....X tool change position
P496 ....Graph counter
P517 ....% of feed on ruffing infeeds (G71, G72)
P530 - P545 ....Used in canned cycles
P550 - P556 ....Used for barfeeder
P698 ....Parts Counter Increment
P699 ....Parts Counter
P705-P729 ....Feedrate override % s
P730-P754 ....Handwheel overrides
P755-P780 ....Spindle override % s
P781-P784 ....Remote handwheel overrides
P790 ....Spindle range 1
P791 ....Spindle range 2
P792 ....Spindle range 3
P793 ....Spindle range 4
P794 ....Spindle range 5
P795 ....Spindle range 6
P796 ....Spindle range 7
P797 ....Spindle range 8
P798 ....Max solid graph size

P800 ....Measure position (Z)
P801 ....Measure position (X)
P817 ....Spindle Encoder PPU1
P818 ....Actual Spindle RPM
P840 ....Pulse Delay
P841 ....Max Spindle RPM
P842 ....Tapping Ramp Low Gear
P843 ....Spindle Acceleration Ramp
P844 ....Spindle Deceleration Ramp
P850 ....ADC Sample
P851 ....ADC Scale
P852 ....ADC Value
P853 ....ADC Trigger
P854 ....Tapping Ramp High Gear
P855 ....Spindle Encoder PPU2
P895 ....Cranking Max ipm
P896 ....Handwheel Encoder PPU
P897 ....Cranking Minutes Per Turn
P960 ....Manual mode threading 1st depth
P961 ....Manual mode threading min cut
P962 ....Manual mode threading finish passes
P965 ....SpindleScale
P966 ....BlockTime
P967 ....Small move jerk multiplier
P973 ....Front panel base port
P974 ....Transition radius G187
P975 ....Transition radius G188
P976 ....Transition radius G189
P977 ....Max corner deviation G187
P978 ....Max corner deviation G188
P979 ....Max corner deviation G189
P980 ....Acting transition radius
P981 ....Acting corner max deviation
P982 ....Calculator cutter diameter
P983 ....Calculator width of cut
P984 ....Calculator cutting speed
P985 ....Calculator chip Thickness
P986 ....Calculator speed
P987 ....Calculator feed
P999 ....Max RPM with door open
P1504 ....Turret # of pockets
P1668-P1683 .....Encoder Comp Resolution
P1684-P1699 .....Encoder Comp Factor
P1700-P1707 .....Thermal comp degrees per volt
P1708-P1715 .....Thermal comp inches per volt

P1716-P1723 .....Thermal comp inverse growth per degree
P1724-P1731 .....Thermal comp position sensor
P1732-P1739 .....Thermal comp voltage reference
P1740-P1747 .....Thermal comp position reference
P1748-P1755 .....Thermal comp lower backlash
P1756-P1763 .....Thermal comp inches per degree
P1764-P1771 .....Thermal comp adjust positionA
P1772-P1779 .....Thermal comp adjust voltage
P1780- P1787 ....Thermal comp adjust positionB
P1788-P1795 .....Thermal comp adjust voltageB
P1800 ....Stock length
P1816 ....Stock diameter (OD)
P1817 ....Stock diameter (ID)
P1818 ....Clamped RPM
P1900-P1999 .....Tool table radius
P2100-P2199 .....Tool table X length
P2300-P2399 .....Tool table X length wear
P2500-P2599 .....Tool table radius wear
P2700-P2799 .....Tool table type
P2900-P2999 .....Tool table Z length
P3100-P3199 .....Tool table Z length wear
P3300-P3399 .....Tool table mount angle
P3700-P3799 .....Tool table size
P4500-P4599 .....Tool table nose angle
P4900 ....G50 (Z)
P4901 ....G92 (X)
P4916 ....G52 (Z)
P4917 ....G92 (X)
P4991 ....Spindle brake delay
P5024-P5039......Work Coord G54
P5040-P5055......Work Coord G55
P5056-P5071......Work Coord G56
P5072-P5087......Work Coord G57
P5088-P5103......Work Coord G58
P5104-P5119......Work Coord G59
P6000-P6015 .....Axis address label
P6016-P6031 .....Pulses per unit
P6032-P6047......Home position
P6048-P6063 .....Home direction
P6064-P6079 .....Positive limit
P6080-P6095 .....Negative limit
P6096-P6111 .....Maximum feed
P6112-P6127 .....Dry run feed
P6128-P6143 .....Rapid velocity
P6144-P6159 .....Home sequence

P6160-P6175 .....Velocity toward home
P6176-P6191 .....Velocity away from home
P6192-P6207 .....Velocity toward marker
P6208-P6223 .....Encoder multiplier
P6240-P6255 .....Jog velocity
P6256-P6271 .....In position
P6272-P6287 .....G00 unidirectional
P6288-P6303......G60 unidirectional
P6304-P6319 .....Backlash
P6320-P6335 .....Excess error
P6336-P6351 .....Rotary=0 Linear=1
P6352-P6367 .....English Lead
P6368-P6383 .....English Trail
P6384-P6399 .....Metric Lead
P6400-P6415 .....Metric Trail
P6416-P6431 .....Jog Key Direction
P6432-P6447 .....Home Marker Switch (Home Switch = 0, Marker = 1)
P6448-P6479 .....G28 reference point
P6464-P6479 .....G30 reference point 2
P6480-P6495 .....G30 reference point 3
P6496-P6511 .....G30 reference point 4
P6512-P6527 .....Max Handwheel Error
P6528-P6543 .....Feed Back
P6544-P6559 .....Invert Handwheel
P6560-P6575 .....Gain Proportional
P6576-P6591 .....Gain Velocity
P6592-P6607 .....Gain Handwheel
P6608-P6623 .....Max acceleration/deceleration
P6624-P6639 .....Max jerk
P6640-P6655 .....G187 velocity gain
P6656-P6671 .....G188 velocity gain
P6672-P6687 .....G189 velocity gain
P6688-P6703 .....G187 max jerk
P6704-P6719 .....G188 max jerk
P6720-P6735 .....G189 max jerk
P6736-P6751 .....Acting velocity gain
P6752-P6767 .....Acting max jerk
P6768-P6783 .....Positive safe zone
P6784-P6799 .....Negative safe zone
P6816-P6831 .....Rotary rollover
P6832-P6847 .....Encoder Offset
P6848-P6863 .....Encoder Mark
P6864-P9999 .....Unused

Machine Setup POWER Parameters

Front Panel Type:	Molded or BNR
FP Control Port:	Front Panel port 1552 for CNC8000 systems
Initial Units are:	G20 Inch or G21 Metric to power up in inch or metric.
Number of Axes:	Can be 1 to 5.
Spindle Axis:	Normally set to 1 for Z axis.
Primary HDW Encoder:	The encoder to use for the primary handwheel. 8 for CNC8000 systems.
2nd Handwheel Encoder Axis:	Used for optional manual panel systems
Remote HDW Encoder:	The encoder to use for the remote handwheel.
Remote HDW FOV1 thru FOV4:	Sets the feedrate for the four feedrate switch positions on the remote handwheel.
Default Feedrates to:	Feed per minute or Feed per rev.
Default SpindleSpeeds to:	CSS or RPM
Block Skip On:	If Yes block skip is on at power up.
Optional Stop On:	If Yes optional stop is on at power up.
Safe Zone On:	If Yes safe zone is on at power up. G22 turns the safe zone off. G23 turns the safe zone on.
Foreign Extension:	Refers to the extension for data files. Current valid extensions are listed below. *NOTE: Shift - F8 will toggle between English and any other language.* CHN CHINESE ENG ENGLISH SPN SPANISH GRM GERMAN

******Tool Table Fields******

The following 10 tool characteristics may be displayed in the tool table.

X Length
Z Length
Radius
X Length Wear
Z Length Wear
Radius Wear
Tool Nose Angle
Tool Mount Angle
Size
Tool Type
X Axis Wear Offset By: Diameter or Radius

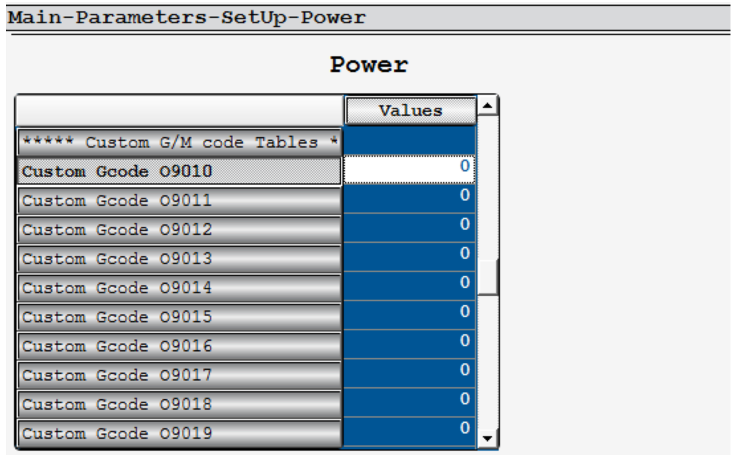
******Tool Changer Information******

ATC tool (pocket count):	Number of Turret pockets.
Tool Change Macro:	Name of the program used for tool change.
Fast Pragati Index:	Stop at each turret station or not.

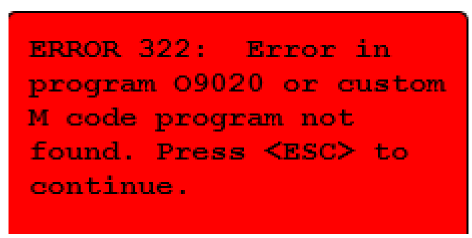
Front Turret: YES for turret on front side.
Clamp on OD: For hydraulic chuck to clamp on OD or ID.
Check Air Pressure: If YES, machine will e-stop if low air is signaled.

Custom G/M Code Tables

- Custom Gcode 09010
- Custom Gcode 09011
- Custom Gcode 09012
- Custom Gcode 09013
- Custom Gcode 09014
- Custom Gcode 09015
- Custom Gcode 09016
- Custom Gcode 09017
- Custom Gcode 09018
- Custom Gcode 09019
- Custom Mcode 09020
- Custom Mcode 09021
- Custom Mcode 09022
- Custom Mcode 09023
- Custom Mcode 09024
- Custom Mcode 09025
- Custom Mcode 09026
- Custom Mcode 09027
- Custom Mcode 09028
- Custom Mcode 09029
- Custom Mcode 09030
- Custom Mcode 09031
- Custom Mcode 09032
- Custom Mcode 09033
- Custom Mcode 09034
- Custom Mcode 09035
- Custom Mcode 09036
- Custom Mcode 09037
- Custom Mcode 09038
- Custom Mcode 09039



When the G or M code is executed the 090## program (in the ram folder) associated will be called. All custom codes are checked for syntax errors prior to running any program (or MDI command). If an error is found (or the 090## file is not found) a message is shown:



Parameter File Version Used to track software during updates.
*******Inch Precision******* Used for displaying inch values other than axis positions.

- Cartesian Leading
- Cartesian Trailing
- Angular Leading
- Angular Trailing
- Spindle Leading
- Spindle Trailing
- Feedrate Leading
- Feedrate Trailing

*******Metric Precision******* Used for displaying metric values other than axis positions.

Cartesian Leading
Cartesian Trailing
Angular Leading
Angular Trailing
Spindle Leading
Spindle Trailing
Feedrate Leading
Feedrate Trailing

Notes on Parameters

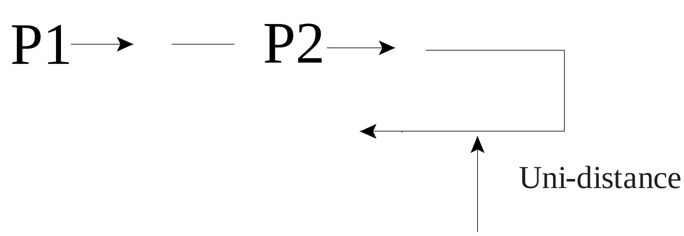
Note 1: On any given parameter screen, parameter values are written when Esc- ing to the main menu. After editing parameters, whether they be Power parameters or any other parameters, remember to Escape to the main menu.

Note 2: For new power parameters to take effect, the machine must be powered down then up again. Power parameters are so named because they are read only on power-up.

Machine Setup AXIS Parameters

Axis Address Label Z 90 X 88	ASCII code for axis assignments
Pulses Per Unit Z 25400.0 X 25400.0	The number of encoder pulses (actual or simulated) per unit of travel; generally depends on drive type and drive setup.
Encoder Multiplier Z 1 X 1	When multiplier is 0, pulses after quadrature are divided by 4; When multiplier is 1, divided by 2; when multiplier is 2, divided by 4. It works backwards. Higher multiplier means fewer encoder pulses.
Home Position Z -21.0000 X -13.9800 Z 00.0000	The dimension assigned to the machine zero or home position. Sometimes used in rotary axis that do not home at zero. X set so drills do not need A X Tool Length to drill on the spindle center line.
Home Direction Z 0 X 0	Defines the direction an axis moves when a home is commanded. CW = 0 CCW = 1
Velocity Toward Home Z 100.0000 X 200.0000	Sets the velocity at which an axis seeks the home limit switch.
Velocity Away From Home Z 10.0000 X 20.0000	Sets the velocity at which an axis moves off the home switch
Velocity Toward Marker Z 01.0000 X 02.0000	Sets the velocity at which an axis searches for the encoder marker pulse.
Home Sequence Z 2 X 1	These numbers determine the order in which the axes seek home: #1 first, #2 next, etc. Axes with the same number home together. 0 will cause that axis not to seek home.
Home Switch=0 Marker=1 Z 0 X 0	A zero setting commands the normal homing algorithm, where the axis finds the switch before finding the marker; with a 1 setting, the axis simply finds the closest marker.
Positive Limit Z 22.0000 X 00.1000	Dimension from machine zero where the positive software limit occurs.

Negative Limit Z -19.000 X -15.000	Dimension from machine zero where the negative software limit occurs.
Maximum Feed Z 200.000 X 400.000	Sets the maximum G01, G02, G03 feedrate in inches per minute or mm per minute. High limit is the value for rapid velocity.
Dry Run Feed Z 75.0000 X 150.0000	Sets the dry run feedrate in inches per minute or mm per minute
Rapid Velocity Z 0500.000 X 1000.000	Set the maximum G00 feedrate in inches per minute or mm per minute
Max Acceleration Z 070.000 X 140.000	The rate velocity increases or decreases in in/sec^2; depends on servo drive/motor sizing relative to the axis load being driven. CNC8000 motion control algorithm manages acceleration to prevent harsh forces.
Jerk Limit Z 2000.00 X 4000.00	The rate acceleration increases or decreases in in/sec^3; a good all purpose value is 2000. High jerk values reduce cycle time; machine motion is more violent. Lower values extend cycle time, machine motion is smoother. These default values are used in the absence of G187, G188, G189 calls.
Jog Velocity Z 200.0000 X 400.0000	Velocity in IPM or MMPM
Jog Key Direction Z 0 X 0	Sets whether the Jog is related to moving the table or moving the tool. A “1” reverses direction.
In Position Z 00.0050 X 00.0100	After any rapid move, the machine will wait until the axis is within this distance from the target destination before starting the next block.
G00 Unidirectional Z 00.0000 X 00.0000	Sets the distance in inches or mm which an axis will go past the destination point in one direction before reversing direction so that the machine will always position from the same direction. Active only in G00 mode.



G60 Unidirectional — Same as G00 unidirectional except only active in a G60 block
Z 00.0000
X 00.0000

Backlash — Backlash compensation in inches or mm. The control compensates for lost motion whenever an axis reversal takes place. Active in all modes. Set when the machine is calibrated.
Z 00.0002
X 00.0002

Excess Error — Sets the max distance in inches or mm the machine lags desired position. The CNC e-stops the system if excess error is exceeded. An important safety parameter.
A zero value turns off excess error checking.
Z 00.5000
X 01.0000

Rotary=0 Linear=1 — Sets whether the cnc treats an axis as rotary or linear.
In rotary, feedrate is interpreted as degrees per minute rather than distance per minute. There is no conversion between inch and metric for a rotary axis.
Z 1
X 1

Rotary Rollover
Off = 0 On = 1 — If Rotary Rollover = 1, then all rotary axis positions are modulus 360, i.e., between 0 and 360. An absolute move command goes the shorter way to a target position and hence never goes more than 180 degrees. An incremental move command goes in the direction specified and hence may go more than 180 degrees.

If Rotary Rollover = 0, then rotary axis position may range from -1,000,000 degrees to +1,000,000 degrees. Typically set to one for C-Axis

Handwheel Normal=0 Invert=1 — A "1" will reverse the direction of the handwheel.
Z 1
X 0

Inch Leading — Sets the number of characters to the left of the decimal point for the inch system, for the specified axis only
Z 2
X 2

Inch Trailing — Sets the number of characters to the right of the decimal point for the inch system, for the specified axis only
Z 4
X 4

Metric Leading — Same as Inch Leading except for the metric case
Z 4
X 4

Metric Trailing — Same as Inch Trailing except for the metric case
Z 3
X 3

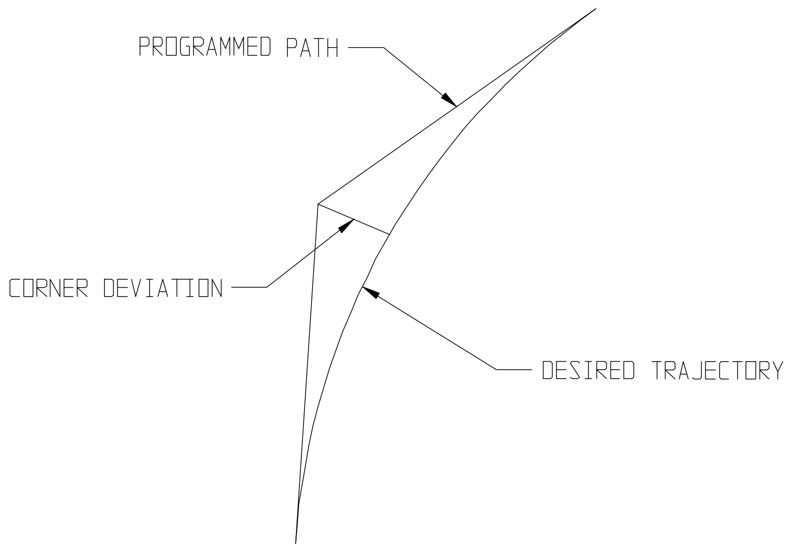
Max Handwheel Error Z 00.4000 X 00.8000	When handwheeling, if following error reaches this value, pulses from the handwheel are ignored. Error is specified in inches or mm. Prevents excess error e-stop if operator gives a rapid spin to the Handwheel. Max Handwheel Error should be set to Excess Error - .100.
Proportional Gain Z 04.0000 X 04.0000	Scaling factor between following error and velocity command signal to the drive. Reasonable values are between 2 and 8.
Velocity Gain Z 20.0000 X 20.0000	Scaling factor between desired velocity and velocity command signal to the drive. Reasonable values are between 10 and 70, these default values used in the absence of G187, G188, or G189. (see below)
Handwheel Gain Z 01.0000 X 01.0000	Same as proportional gain. Smaller values are used to soften handwheel motion. Reasonable values are between 1 and 4.

Machine Setup MISC Parameters

******Basic Machine Info******

Machine Type	Type in ML18, etc.
Machine Version	The current model
Mechanical Version	The current mechanical version
Electrical version	The current electrical version
Machine serial number	The machine serial number
Slow Rapid Percent	Sets the percent of rapid when the leftmost of the three molded front panel Rapid keys is active. Generally set to 10, matching the 10% embossed on the key.
Check Spindle Up to Speed	**“**Off” – The control never waits for a spindle up to speed signal. “On (M3/M4)” - The control will wait for up-to-speed signal from the spindle controller after an M3 or M4 command. “From Rapids to Feeds” – The control will wait for an up-to-speed signal from the spindle controller before continuing to a G1, G2, or G3 block from a G0 block.
Up to Speed Input #	Specifies the input for the up to speed signal.
Spindle Encoder PPU	Pulses per rev of spindle, used for hard tapping option and displaying the RPM.
Handwheel Encoder PPU	Pulses per rev of the handwheel, usually 400.
Spindle Range 1	Max spindle speed for gear 1
Spindle Range 2	Max spindle speed for gear 2
Spindle Range 8	Max spindle speed for gear 8
Spindle Range	What gear the spindle is in
Maximum Spindle Speed	This will clamp the overall spindle speed.
Tap Spindle Ramp Time 1	The tapping deceleration point is adjusted based on spindle RPM, pitch, and spindle ramp time so as not to overshoot the programmed depth. It is the time in seconds to slow the spindle to 0 RPM’s from full RPM’s in gear 1.
Tap Spindle Ramp Time 2	Time in seconds to decelerate the spindle to 0 RPM’s from full RPM’s in any gear other than 1.
Spindle Ramp Time Top	In hard tapping, how long from spindle at full scale in one direction to full scale in the other direction, the reversal the spindle must execute after the tap exits the hole. Setting the parameter too high makes for excessive delay; setting it too low causes the Z axis to thump.
Tap Up Double	When set to Yes, in hard tapping operations, the tap is withdrawn from a hole at twice the spindle RPM (and hence twice the feedrate) compared to going in.
2 Gear Yaskawa M5 Spindle Motor	Yes for M5 spindle drives with 2 gear spindle motors. The spindle does not need to stop to shift.
Hard Tap Fudge Factor	Used to adjust the depth of rigid tap cycle. Higher numbers decrease the amount of overshoot at bottom of the hole. Default value = 70.0.

Spindle Acceleration (Sec)	In seconds, amount of time from zero speed to full speed.
Spindle Deceleration (Sec)	In seconds, amount of time from full speed to zero speed.
Lube Pump Time (sec)	Time for lube pump to stay ON.
Lube Cycle Time (min)	Time for lube pump to stay OFF.
Check Spindle in Gear	If set to YES, control will monitor the "in-gear" prox switches.
Back Gear Reverse	YES only for semi-manual lathes where back gear reverses spindle direction.
Multi-Geared Spindle	YES for spindles that shift (mechanically or electronically)
AC Brushless Axis Motors	Generally Yes.
Servo motor on C-Axis	YES, for live tooling systems with a servo motor on the C-Axis.
FeedHold Input	Input # for the feedhold signal, which is usually the door open switch.
European Code	Yes or No. See notes on European Code at the end of this section.
Spindle JOG Speed	In RPM.
Chip Remove ON (min)	Amount of time to run chip conveyor.
Chip Washdown ON (sec)	Washdown ON time in seconds.
Chip Washdown OFF (sec)	Washdown OFF time in seconds.
Way Lube On Time (sec)	Time in seconds for the waylube pump to run in a cycle
Way Lube Off Time (min)	Time in minutes for the waylube pump to pause in a cycle
Auto Door Open	Enables the door open option.
Auxl Button Function	If set to 1, Aux1 button toggles user M-Code M65/M75 which toggles output 30. If set to 2, Aux1 button toggles auto door. Default is 0, not enabled.
Chiller Fault Input #	The input # for chiller faults. Causes an error #1395 chiller fault message if input is read.
Max Feed w/Door Open	See notes on European code (page 10-26) later in this section.
Probe Input #	Input to use for probe commands.
Cranking Minutes/Rev	How many minutes of normal program execution time correspond to one turn of the handwheel. Multiplying an IPM feed time, this factor results in an IPR feed (inches per turn of the handwheel). This parameter should be approximately 0.0010 for our current systems.
Cranking Factor	Multiplier for each handwheel click is approximately 100 for our current systems. Cranking/Factor/Cranking MinsPerRev is proportional to the max feedrate allowed while hand-cranking.
Cranking Max IPM	This limits the feedrate while hand-cranking to get reasonable response at slow programmed feedrates. Adjusting the other parameters can give excess errors on rapids. This parameter should be approximately 100 for our current systems.
Max Corner Deviation	This number sets a default value for the maximum deviation allowed on a corner. A value of .002 means the control generates a desired path within. .002" of the programmed path.



Tool Setter:	Set if machine has a tool setter.
Tri-Color Cycle Light	Yes if it has the 3 color (Red, Yellow and Green) Cycle Light.
End-of-Cycle Output #	The end of cycle light output #. Zero if it doesn't have an end of cycle light.
	******Software Options******
Allow Modal Restart	Enables the ability to start programs from a specific block in the block. The block is selected using the program editor. It also allows the ability to pick up on modes, before the selected block, such as federates, spindle commands, tool#s, cutter comp, etc.
Look Ahead Blocks	The maximum number of blocks the control is allowed to look ahead. The 8000 Series CNC uses an adaptive lookahead algorithm. It looks no further ahead than it has to. 99.99% of applications require nowhere near the max number of Look Ahead Blocks. Dependent on processor capacity. Setting is 80 on Series 8000 CNC's in early 2010.
Look Scope Blocks	Generally set to zero.
Transition Radius	A first cousin to the earlier parameter Max Corner Deviation, defining how closely the desired path follows the programmed path at corners. A lower value creates a desired path closer to the programmed path. A higher value permits smoothing the corner, reducing cycle time. This default value is used if there is no G187, G188 or G189 command in the program. Usually .020.
Small Move Multiplier	Allows higher speeds in tiny line segments, producing faster and smoother overall performance. A good value is 8.
Max solid graph file size	The high limit for a solid model graphics file. Dependent on processor capacity and graphics chipset. Usually 30,000.
Graph look-ahead	The number of lines and arcs that are graphically displayed ahead of the tool. The range is 1-200 and the default is 15.
Special Flags	Normally set to 0. Bit 2 (#2) will shut off trig help. Bit 3 (#4) will shut off cutter compensation.

Note: Trig Help is cnc background functionality that finds arc endpoints and tangents in conversational programs. The output from some CAD/CAM systems plays poorly with Trig Help. If this interference occurs, usually shown by a huge arc in the toolpath when it should be a small arc, machine performance is corrected by turning Trig Help off.

Setting Misc parameter Special Flags to 2 (turning off Trig Help) does not help the situation, because the setting does not persist. There are certain Prog-Run defaults invoked by the cnc when any program is run. One of them is to enable Trig Help.

A workaround for this circumstance is to insert a line "PB81 = 2" near the top of any CAD/CAM text program. A CAD/CAM post processor can usually be edited to add this line automatically.

Another workaround is to set PB208 = 0 in MDI. If PB208 equals 0, then Trig Help is not turned on as a Prog-Run default and thus the setting in Misc parameters Special Flags does persist. The value in PB208 persists through power cycles, i.e. when PB208 has been set to zero and Special Flags = 2, then Trig Help is turned off and remains off.

Tool Setting	If set to Current Tool when setting tool lengths the control assumes that the active tool # is the tool length that is being set. If set to Any Tool the control will prompt the operator for the tool #.
Tool Setting, Using Work Offsets	If set to Yes, the work offsets are used when setting the tool length. The option is for those that touch tools off on the table.
G5#-Z on Tool #1 Only	Used to specify tool #1 as the master tool.
SingleBlock thru CannedCycles and TC's	If set to Yes, (and in single block) tool changes, drill cycles and canned cycles are executed with 1 cycle start. If set to No each block of the tool changes, drill cycles or canned cycle requires a cycle start.
CPU Warning Temperature	In Centigrade. Used to check for CPU overtemp.
CPU Warning Temperature	If set to a non-zero value and the value is exceeded a message WARNING: CPU is dangerously warm. To avoid abrupt shutdown, shutdown the machine as soon as possible. Press<ESC> to clear this message.

```
WARNING: Cpu is
dangerously warm.  To
avoid abrupt shutdown,
shutdown the machine as
soon as possible
Press <ESC> to clear
this message
```

******Post M codes table******

Post M code #0 Post M code #1 Post M code #2 Post M code #3 Post M code #4 Post M code #5	M codes listed here will be executed after all other operations within the block.

Post M code #6
Post M code #7
Post M code #8
Post M code #9

******Spindle Power Monitoring******

Spindle Power Raw	Value read from ADC, 0 to 255
Spindle Power Scale	Multiplier to obtain Spindle Power Value
Spindle Power Value (Amps)	Calculated from the spindle power raw and scale

******Manual Threading******

1st Depth	First cut amount for manual mode threading.
Minimum Cut	Minimum cut amount for manual mode threading.
Finish Passes	# of finish passes for manual mode threading.

******Thermal Compensation******

SENSOR #1

#1 Comp Axis	The number of the axis that is being compensated.
#1 Position of Sensor	The position (machine relative) the sensor is mounted.
#1 Voltage Reference	Voltage above this value indicates "growth", values below indicate "shrinkage".
#1 Position Reference	The position (machine relative) corresponding to the fixed end of the ballscrew.
#1 Lower Backlash (inch)	The expected amount of voltage change when direction is reversed. This value is due to coupling between the ballscrew and the motor. This value is set by the calibration cycle.
#1 Inches/Volt	For sensors that measure position displacement, the displacement per volt.
#1 Degrees/Volt	For sensors that measure temperature, the temperature change per volt.
#1 Inverse Growth/Degree	For temperature sensors used on a ballscrew, the percentage growth of the ballscrew per degree (or rather the inverse). If using Centigrade, this parameter will be about 1/.000011 for a steel ballscrew.
#1 Inches/Degree	For temperature sensors mounted on the head indicating spindle displacement, the displacement of the spindle per degree.
#1 Adjust Position A (in) #1 Adjust Voltage A #1 Adjust Position B (in) #1 Adjust Voltage B	These four parameters allow an adjustment to the voltage based on the current position of the axis. The amount of correction is calculated by linearly interpolating the values at the two given points. These values are set by the calibration cycle.

(These thermal comp fields are repeated for seven axes)

ENCODER COMPENSATION

Comp Axis	The number of the axis that is being compensated.
Encoder Axis	The encoder number the comp is based off of.

Encoder PPU	The Pulses Per Unit of the comp encoder.
Comp Factor	The amount of comp applied per change in position of the encoder axis.

European Code Parameter and Operation Descriptions

European Code	Limits axis and spindle motion as described below
Feedhold Input	The input for signaling feedhold to the cnc. Depending on machine safety requirements, the door switch or light curtain signal usually goes to the feedhold input.
Setup Button	The feedhold override button, a hold-to-run switch in some European code scenarios.

When European Code is set to Yes, the cnc software limits machine behavior to comply with European Union safety regulations. Below is a description of how the software operates.

If the door open switch is wired to the feedhold input and European Code = No When the door opens, it shuts the spindle off and feedholds the machine. When the door is open, the spindle will not start. When the door is open, the machine will not do an MDI move, will not jog, and will not do a move in run mode; however, it will handwheel.

The machine will do I/O related functions other than turning on the spindle (i.e. arm-in, drawbar, etc.) If the machine is tapping and the doors open, the machine will finish the tap, feedhold, and stop the spindle.

If the doors are opened while the machine is running:

1. The spindle will shut off.
2. The axis will stop moving.
3. The feedhold light will come on.
4. The spindle will not restart.

To continue operation:

1. Close the doors (**Cycle Start** will flash).
2. Push the **CW** button on the front panel (the spindle will restart).
3. Push the **Cycle Start** Button on the front panel (**Cycle Start** will stop flashing and the **Feedhold** light will go out).

If the machine is not running and the spindle is not running, opening the doors should have no effect on the machine. However, the **Feedhold** light will come on.

If the door open switch is wired to the feedhold input and European Code = Yes When the door opens, which is allowed by pressing the **door open** button, the spindle shuts off as well as feedholds the machine. When the door is open, the spindle will not start. When the door is open, the machine will not do an MDI command; it will not jog nor will it start running a program. It will not handwheel. If the machine is tapping and the doors open, the machine will finish the tap and then **feedhold** and stop the spindle.

If the door is open, the machine will allow the tool changer to index only one position at a time using the tool changer utility or the tool setting utility.

If the door is open and **Setup** is held in, the machine will jog up to 60 ipm.

If the door is open and **Setup** is held in, the machine will handwheel up to the 50% rate on the feedrate override switch, which is .1mm per click of the handwheel. (It is difficult to generate speeds greater than 1000 mmpm in this mode). Modifying the distance per click of the handwheel also requires a password available only to the machine builder.

If the doors are opened while the machine is running:

1. The spindle will shut off.
2. The axis will stop moving.
3. The feed-hold light will come on.
4. The spindle will not restart.

If **Setup** is pressed, the machine will move at the clamped feedrate while the button is held in.
To continue operation:

1. Close the doors (the **Cycle Start** Button will flash).
2. Push the **CW** button on the front panel (the spindle will restart).
3. Push the **Cycle Start** Button on the front panel (the **Cycle Start** Button will stop flashing and the **feedhold** light will go out).

PROGRAM Parameters

P200 thru P207	Contains the previous programmed position relative to the current work offsets P200=Z P201=X . . . etc.
P208 thru P215	Contains the current programmed position relative to the current work offsets P208=Z P209=X. . . etc.
P216 thru P223	Contains the previous machine position relative to machine zero P216=Z P217=X. . . . etc.
P224 thru P231	Contains the current machine position relative to machine zero P224=Z P225=X. . . etc.
P232 thru P239	Contains the work coordinate offset relative to the machine zero P232=Z P233=X . . . etc.
P240 thru P247	Contains the active tool lengths; P240=Z P241=X, etc.
P248	Contains the current arc radius
P249	Contains the current arc K value
P250	Contains the current arc I value
P251	Current programmed feedrate
P252	Current dwell time
P253	Current programmed spindle speed
P260	Contains active tool number
P261	Contains active tool radius
P265	Tool Type
P266	Contains safe zone status (0=off, 1=on)
P267	Contains ratio of feedback pulses to program units (1 is inch, .03937 for metric)
P268	Pending tool number
P298	Max spindle speed in CSS mode.
P300 thru 303	Modal 00-Modal 03
P304	Status if control is in data mode or normal programming (0=off, 1=on)
P306	Status of 0=G0, 1=G1, 2=G2, 3=G3 mode
P308	Number of active plane 0=G17 (XY), 1=G18 (ZX), 2=G19 (YZ)
P309	Cutter comp. status 0=G40, 1=G41, 2=G42.
P310	0=None, 1=Drill, 2=Drill Dwell, 3=Peck Drill, 4=Tap, 5=Bore, 6=Bore II, 7=Face Grooving, 8=Dia. Grooving, 9=Bore Dwell, 10=Grooving, 11=Single Turn, 12=Single Face, 13=Single Thread, 14=Thread Cycle, 15=Threading
P311	0 = Absolute, 1= Incremental mode
P312	Feed unit (0 is feed/min, 1 is feed/rev)
P313	Spindle unit (0 is RPM, 1 is constant surface speed)
P315	Linear Unit (0=inch, 1 = metric)
P316	Scaling (0 = off, 1 = on)
P317	Rotation (0 = off, 1 = on)
P318	Mirror image (0 = off, 1 = on)
P319	Current work coordinate number (G54=1, G55=2, G56=3,… G59=6)

P323	0 = Return to R-plane (G39), 1 = initial level (G38)
P324	Tapping mode (0 = off, 1 = on)
P337	Tool depth (for solid graphics)
P338	Tool Size
P339	Tool nose angle
P340	Tool mount angle
P365	Feed per minute feedrate
P366	Feed per rev feedrate
P367	RPM spindle speed
P368	CSS spindle speed
P416 thru P423	The last dimensions that were executed

CONTROL Parameters

Ignore Tool too Large Errors	If set, the errors: #507 After compensation line to arc lacks intersection. #569 The tool is too large to cut inside the arc "Compensated radius is to small" #579 Compensated arcs do not intersect. Are ignored and compensation is done on a line from the start of the arc to the end of the arc.
Pre Check Software Limits	Never: the control never checks the soft limits. The soft limit error only occurs when the axis moves outside the limits. Run Only: checks are not made when verifying. When running, the control will check to see if any axis is going to move outside the limits before the move starts. The graphics will show the move but the machine will not make the move. This can prevent crashes where faster moves go beyond the limits. Always: the control will check to see if any axis is going to move outside the limits when running or verifying. The graphics will show the move, but the machine will not make the move. It may be useful to verify a program before running it.
% of feed on Ruff Infeeds	Reduces feedrate when feeding into material on G71 and G72 cycles.
Enable Modify Column Displays	Enables the function keys to change the axis positions under F6 (Display). Options are: Current, Target, Distance, Machine, Measure, Actual, None or Error.
Full Code Window	Replaces graphics window with program text.
Auto Spindle Brake	Automatically sets and releases brake on C-axis moves.

********* DPRNT DATA *********

The CNC8000 series cnc retains RS232 functionality for data output using DPRNT functionality and for running auxiliary equipment such as programmable coolant nozzles or rotary tables. RS232 for program transfer is not supported in the CNC8000. The parameters in the DPRNT DATA group govern RS232 behavior for supported DPRNT output.

DPRNT to:	A file, or COM 1. The file will be written to the parts directory.
DPRNT File Name	AnyTextString.txt (default filename is REPORT.DAT if blank)
Baud Rate	110 to 57600 baud
Parity; Data and Stop Bits	Many different combinations; E/7/2 is common.

***** Tool Setter Parameters *****

P132 Z+	Calibration data for Z+
P133 X+	Calibration data for X+
P134 Z-	Calibration data for Z-
P135 X-	Calibration data for X-

***** Bar Feeder Parameters *****

P550	Z Bar Feeder Stop Position
P551	X Bar Feeder Stop Position
P552	Z Bar Feeder Machine Position
P553	Z Bar Feeder Eject Position
P554	X Bar Feeder Eject Position
P555	Number of Parts in Bar
P858	Spindle Orient Degree
P101 Thread Lead	Used in the thread cycle
P102 Thread Shift Angle	For multiple lead threads
P103 Thread Chamfer Length	Chamfer length at the end of the thread
P104 Thread Chamfer Angle	Chamfer angle at the end of the thread
P105 Thread Type	Type 1-4 for cutting the thread
P106 Thread Height	K value in the G76 cycle
P107 1st Thread Cut Depth	D value in the G76 cycle
P108 Tool Angle	A value in the G76 cycle
P110 Start Block	P value in G70, G71, G72 and G73 cycles
P111 End Block	Q value in G70, G71, G72 and G73 cycles
P112 X Finish Stock	U value in G71, G72 and G73 cycles
P113 Z Finish Stock	W value in G71, G72 and G73 cycles
P114 X Rough Stock	I value in G71, G72 and G73 cycles
P115 Z Rough Stock	K value in G71, G72 and G73 cycles
P116 Cut Increment	D value in G71, G72 and G73 cycles
P117 Roughing Type	Pattern or 45 degree
P118 Relief	R value in G71 and G72 cycles
P119 First Depth	D value in G76 cycle
P121 Cut Length	Used in G90, G92, and G94 cycles
P122 Cut Depth	Used in G90, G92, and G94 cycles
P123 End Point	Used in G90, G92, and G94 cycles
P124 Diameter	Used in G90, G92, and G94 cycles
P125 Turning Taper	Taper in G90 cycle
P126 Facing Taper	Taper in G94
P130 Threading Minimum Cut	Minimum in the G76 cycle
P131 Threading Finish Passes	# of finish passés in the G76 cycle
P139 X R-Plane	X Clearance
P140 Z R-Plane	Z Clearance
P141 Final depth	For drill cycles
P142 Z Initial Level	For drill cycles

P143 Cut Increment	For drill cycles
P144 First Depth	For drill cycles
P145 Plunge Feedrate	For drill cycles
P146 Peck-up Increment	For drill cycles
P147 Peck Clearance	For drill cycles
P148 Dwell 1	For drill cycles
P149 Dwell 2	For drill cycles
P172 thru P179	Coordinates of the center position for a mirror image command for the enabled axis P172=Z P173=X. . . etc
P180 thru P187	Coordinates of the scaling center for the enabled axis P180=Z P181=X. . . etc
P188 thru P195	Scale factor for each of the enabled axis P188=Z P189=X. . . etc
P196	K position of primary axis center of rotation
P197	I position of secondary axis center of rotation
P198	Angle of rotation
P199	Incremental or absolute on G90, G92 and G94 cycles
P1800 Material Length	For graphics
P1816 Material OD	For graphics
P1817 Material ID	For graphics

Modal Restart (Pick) – Detailed Operation Notes:

To enable this feature, under PARM/MISC, toggle "Allow Modal Restart" to true.

Under RUN or VERIFY/START, there will be new menu options marked PICK and MODAL, as well as the old FIRST/BLOCK#/TOOL#.

Pressing PICK brings up a read-only editor showing the current program being executed. You can then scroll down the line you want to start from, and press PICK (different pick, maybe a rename on one of them is in order).

Modal Restart can be chosen by toggling MODAL on, or a Non-MODAL Restart by toggling MODAL off.

MODAL can also be toggled on and off after doing BLOCK# or TOOL#. Non-Modal Restart means it will start on the chosen line with no further ado. Modal Restart means it will first fast verify up to your chosen line, in the process setting modal codes correctly.

Then, there will be a series of prompts to press cycle start to do the following:

1. Change the tool you should be on that point, e.g. T0202.
2. Set the correct spindle speed (move the value from the verify value to the run value), e.g. S1000.
3. Do the last mcode in the following groups, if any was done in the program:
   M3/M4/M5
   M7/M8/M9

Once this is done the control will do what it does on a resume, go to the "halt" position (in this case the position you would have been at), etc. It will then execute from the chosen line.

If F4 (Pick) is pushed and the program is started from a tool, a block or a line (using pick) it can start the program in the mode (spindle speeds, feed rate, cutter comp, tool #, coolant, etc.) by toggling the F9 (Modal) key.

Cycle Start *starts the program from the beginning.*

If F9 (Modal) is pushed and the program is started from a tool, a block or a line (using pick) it can start the program in the mode (spindle speeds, feed rate, cutter comp, tool #, coolant, etc.) by toggling the F9 (Modal) key.